

Altair HyperMesh 2021

Solvers Tutorials

Updated: 01/11/2021

Intellectual Property Rights Notice

Copyrights, Trademarks, Trade Secrets, Patents & Third Party Software Licenses

Altair Engineering Inc. Copyright © 1986-2020. All Rights Reserved.

Copyrights in the below are held by Altair Engineering, Inc., except where otherwise explicitly stated. This Intellectual Property Rights Notice is exemplary, not exhaustive.

> **Note:** Pre-release versions of Altair software are provided 'as is', without warranty of any kind. Usage of pre-release versions is strictly limited to non-production purposes.

Altair HyperWorks™ - The Platform for Innovation™

Altair AcuConsole™ ©2006-2020

Altair AcuSolve™ ©1997-2020

Altair Activate® ©1989-2020 (formerly solidThinking Activate)

Altair Compose® ©2007-2020 (formerly solidThinking Compose)

Altair ConnectMe™ ©2014-2020

Altair EDEM ©2005-2020 DEM Solutions Ltd, ©2019-2020 Altair Engineering Inc.

Altair ElectroFlo™ ©1992-2020

Altair Embed® ©1989-2020 (formerly solidThinking Embed)

- **Altair Embed SE** ©1989-2020 (formerly solidThinking Embed SE)
- **Altair Embed/Digital Power Designer** ©2012-2020
- **Altair Embed Viewer** ©1996-2020

Altair ESAComp™ ©1992-2020

Altair Feko™ ©1999-2014 Altair Development S.A. (Pty) Ltd., ©2014-2020 Altair Engineering Inc.

Altair Flux™ ©1983-2020

Altair FluxMotor™ ©2017-2020

Altair HyperCrash™ ©2001-2020

Altair HyperGraph™ ©1995-2020

Altair HyperLife™ ©1990-2020

Altair HyperMesh™ ©1990-2020

Altair HyperStudy™ ©1999-2020

Altair HyperView™ ©1999-2020

Altair HyperXtrude™ ©1999-2020

Altair Inspire™ ©2009-2020 including Altair Inspire Motion, Altair Inspire Structures, and Altair Inspire Print3D

Altair Inspire Cast ©2011-2020 (formerly Click2Cast)

Altair Inspire ElectroFlo ©1992-2020

Altair Inspire Extrude Metal ©1996-2020 (formerly Click2Extrude-Metal)

Altair Inspire Extrude Polymer ©1996-2020 (formerly Click2Extrude-Polymer)

Altair Inspire Form ©1998-2020 (formerly Click2Form)

Altair Inspire Friction Stir Welding ©1996-2020

Altair Inspire Mold ©2009-2020

Altair Inspire Play ©2009-2020

Altair Inspire PolyFoam ©2009-2020

Altair Inspire Render ©1993-2016 Solid Iris Technologies Software Development One PLLC, ©2016-2020 Altair Engineering Inc (formerly Thea Studio)

Altair Inspire Resin Transfer Molding ©1990-2020

Altair Inspire Studio ©1993-2020 (formerly 'Evolve')

Altair Manufacturing Solver™ ©2011-2020

Altair Material Data Center ©2019-2020

Altair MotionSolve™ ©2002-2020

Altair MotionView™ ©1993-2020

Altair Multiscale Designer™ ©2011-2020

Altair nanoFluidX™ ©2013-2018 Fluidyna GmbH, ©2018-2020 Altair Engineering Inc.

Altair newFASANT ©2010-2020

Altair OptiStruct™ ©1996-2020

Altair PollEx ©2003-2020

Altair Radioss™ ©1986-2020

Altair Seam™ ©1985-2019 Cambridge Collaborative, Inc., ©2019-2020 Altair Engineering Inc.

Altair SimLab™ ©2004-2020

Altair SimSolid™ ©2015-2020

Altair ultraFluidX™ ©2010-2018 Fluidyna GmbH, ©2018-2020 Altair Engineering Inc.

Altair Virtual Wind Tunnel™ ©2012-2020

Altair WinProp™ ©2000-2020

Altair WRAP ©1998-2020 WRAP International AB, ©2020 Altair Engineering AB

Altair Packaged Solution Offerings (PSOs)

Altair Automated Reporting Director™ ©2008-2020

Altair GeoMechanics Director™ ©2011-2020

Altair Impact Simulation Director™ ©2010-2020

Altair Model Mesher Director™ ©2010-2020

Altair NVH Director™ ©2010-2020

Altair Squeak and Rattle Director™ ©2012-2020

Altair Virtual Gauge Director™ ©2012-2020

Altair Weight Analytics™ ©2013-2020

Altair Weld Certification Director™ ©2014-2020

Altair Multi-Disciplinary Optimization Director™ ©2012-2020

Altair PBSWorks™ - Accelerating Innovation in the Cloud™

Altair PBS Professional® ©1994-2020

Altair Control™ ©2008-2020; (formerly **PBS Control**)

Altair Access™ ©2008-2020; (formerly **PBS Access**)

Altair Accelerator™ ©1995-2020; (formerly **NetworkComputer**)

Altair Accelerator™ Plus©1995-2020; (formerly **WorkloadXelerator**)

Altair FlowTracer™ ©1995-2020; (formerly **FlowTracer**)

Altair Allocator™ ©1995-2020; (formerly **LicenseAllocator**)

Altair Monitor™ ©1995-2020; (formerly **LicenseMonitor**)

Altair Hero™ ©1995-2020; (formerly **HERO**)

Altair Software Asset Optimization (SAO) ©2007-2020

Note:

Compute Manager™ ©2012-2017 is now part of **Altair Access**

Display Manager™ ©2013-2017 is now part of **Altair Access**

PBS Application Services™ ©2008-2017 is now part of **Altair Access**

PBS Analytics™ ©2008-2017 is now part of **Altair Control**

PBS Desktop™ ©2008-2012 is now part of **Altair Access**, specifically **Altair Access desktop**, which also has **Altair Access web** and **Altair Access mobile**

e-Compute™ ©2000-2010 was replaced by **"Compute Manager"** which is now **Altair Access**

Altair KnowledgeWorks™

Altair Knowledge Studio® ©1994-2020 Angoss Software Corporation, ©2020 Altair Engineering Inc.

Altair Knowledge Studio for Apache Spark ©1994-2020 Angoss Software Corporation, ©2020 Altair Engineering Inc.

Altair Knowledge Seeker™ ©1994-2020 Angoss Software Corporation, ©2020 Altair Engineering Inc.

Altair Knowledge Hub™ ©2017-2020 Datawatch Corporation, ©2020 Altair Engineering Inc.

Altair Monarch™ ©1996-2020 Datawatch Corporation, ©2020 Altair Engineering Inc.

Altair Monarch Server ©1996-2020 Datawatch Corporation, ©2020 Altair Engineering Inc.

Altair Panopticon™ ©2004-2020 Datawatch Corporation, ©2020 Altair Engineering Inc.

Altair SmartWorks™

Altair SmartCore™ ©2011-2020 Altair Engineering Inc.

Altair SmartEdge™ ©2011-2020 Altair Engineering Inc.

Altair SmartSight™ ©2011-2020 Altair Engineering Inc.

Altair One™ ©1994-2020

Altair intellectual property rights are protected under U.S. and international laws and treaties. Additionally, Altair software may be protected by patents or other intellectual property rights. All other marks are the property of their respective owners.

ALTAIR ENGINEERING INC. Proprietary and Confidential. Contains Trade Secret Information.

Not for use or disclosure outside of Altair and its licensed clients. Information contained in Altair software shall not be decompiled, disassembled, “unlocked”, reverse translated, reverse engineered, or publicly displayed or publicly performed in any manner. Usage of the software is only as explicitly permitted in the end user software license agreement. Copyright notice does not imply publication.

Third party software licenses

AcuConsole contains material licensed from Intelligent Light (www.ilight.com) and used by permission.

Software Security Measures:

Altair Engineering Inc. and its subsidiaries and affiliates reserve the right to embed software security mechanisms in the Software for the purpose of detecting the installation and/or use of illegal copies of the Software. The Software may collect and transmit non-proprietary data about those illegal copies. Data collected will not include any customer data created by or used in connection with the Software and will not be provided to any third party, except as may be required by law or legal process or to enforce our rights with respect to the use of any illegal copies of the Software. By using the Software, each user consents to such detection and collection of data, as well as its transmission and use if an illegal copy of the Software is detected. No steps may be taken to avoid or detect the purpose of any such security mechanisms.

Technical Support

Altair provides comprehensive software support via web FAQs, tutorials, training classes, telephone, and e-mail.

Altair One Customer Portal

Altair One (https://altairone.com/) is Altair's customer portal giving you access to product downloads, a Knowledge Base, and customer support. We strongly recommend that all users create an Altair One account and use it as their primary means of requesting technical support.

Once your customer portal account is set up, you can directly get to your support page via this link: www.altair.com/customer-support/

Altair Training Classes

Altair's in-person, online, and self-paced trainings provide hands-on introduction to our products, focusing on overall functionality. Trainings are conducted at our corporate and regional offices or at your facility.

For more information please visit: https://learn.altair.com/

If you are interested in training at your facility, please contact your account manager for more details. If you do not know who your account manager is, please contact your local support office and they will connect you with your account manager.

Telephone and E-mail

If you are unable to contact Altair support via the customer portal, you may reach out to technical support via phone or e-mail. Use the following table as a reference to locate the support office for your region.

When contacting Altair support, please specify the product and version number you are using along with a detailed description of the problem. It is beneficial for the support engineer to know what type of workstation, operating system, RAM, and graphics board you have, so please include that in your communication.

Location	Telephone	E-mail
Australia	+61 649 413 7981	anzsupport@altair.com
Brazil	+55 113 884 0414	br_support@altair.com
Canada	+1 416 447 6463	support@altairengineering.ca
China	+86 400 619 6186	support@altair.com.cn
France	+33 141 33 0992	francesupport@altair.com
Germany	+49 703 162 0822	hwsupport@altair.de
Greece	+30 231 047 3311	eesupport@altair.com

Location	Telephone	E-mail
India	+91 806 629 4500 +1 800 425 0234 (toll free)	support@india.altair.com
Israel		israelsupport@altair.com
Italy	+39 800 905 595	support@altairengineering.it
Japan	+81 36 225 5830	support@altairjp.co.jp
Malaysia	+60 32 742 7890	aseansupport@altair.com
Mexico	+52 555 658 6808	mx-support@altair.com
New Zealand	+64 9 413 7981	anzsupport@altair.com
South Africa	+27 21 831 1500	support@altair.co.za
South Korea	+82 704 050 9200	support@altair.co.kr
Spain	+34 910 810 080	support-spain@altair.com
Sweden	+46 46 460 2828	support@altair.se
United Kingdom	+44 192 646 8600	support@uk.altair.com
United States	+1 248 614 2425	hwsupport@altair.com

If your company is being serviced by an Altair partner, you can find that information on our web site at https://www.altair.com/PartnerSearch/.

See www.altair.com for complete information on Altair, our team, and our products.

Contents

Intellectual Property Rights Notice ii
Technical Support vi

1 Abaqus 16

1.1 HM-4300: Set Up an Abaqus Analysis in HyperMesh 17
1.1.1 Load the User Profile and Model 17
1.1.2 Define the Material Properties 18
1.1.3 Define the *SOLID SECTION Properties 19
1.1.4 Assign the Property 21
1.1.5 View the Solid Elements 22
1.1.6 Define the *SPRING Properties 23
1.1.7 Create a Component Collector 24
1.1.8 Reset the View 25
1.1.9 Create the SPRING1 Element 25
1.1.10 Assign a Property to Selected Elements Overview 27
1.1.11 Create a Property 27
1.1.12 Assign a Property to Individual Elements 28
1.2 HM-4310: Define Abaqus Contacts for 2D Models 29
1.2.1 Load the User Profile and Model 29
1.2.2 Start the Contact Browser 29
1.2.3 Define Surfaces for 2D Solid Elements 31
1.2.4 Define Surfaces by Set 34
1.2.5 Define the Surface Interaction Property 36
1.2.6 Define the Contact Pair 39
1.3 HM-4320: Define Abaqus Contacts for 3D Models 43
1.3.1 Load the User Profile and Model 43
1.3.2 Start the Contact Browser 44
1.3.3 Define Surfaces for Solid Elements 46
1.3.4 Create the Box2-Top Surface 47
1.3.5 Create the Cylinder-Top Surface 49
1.3.6 Define Surfaces for Shell Elements 50
1.3.7 Create the Box1-Bot Surface 52
1.3.8 Create the Box2-Bot Surface 54
1.3.9 Define the Surface Interaction Property 56
1.3.10 Define the Contact Pairs 58
1.3.11 Create the Top-Cylinder-Box2 Contact Pair 60
1.3.12 Create the Bot-Cylinder-Box1 Contact Pair 60
1.3.13 Create the Bot-Cylinder-Box2 Contact Pair 60
1.4 HM-4330: Define *STEP using Abaqus Step Manager 61
1.4.1 Load the User Profile and Model 61
1.4.2 Define a *STEP Card 61
1.4.3 Define Load and Boundary Conditions 63

1.4.4 Create Constraints from the Constraints Panel 64
1.4.5 Create Forces 66
1.4.6 Create Forces from the Forces Panel 66
1.4.7 Define Pressure Loads 68
1.4.8 Request ODB File Outputs 72
1.4.9 Request Result File Outputs 74
1.4.10 Export the .inp File 77
1.4.11 Save the .hm File 77
1.4.12 Open HyperView from the Application Menu 78
1.5 HM-4340: Pre-Processing for Bracket and Cradle Analysis 82
1.5.1 Load the User Profile and Model 82
1.5.2 Understand the Relationship between Abaqus and HyperMesh Entities 83
1.5.3 Review the Card Image 84
1.5.4 View the Property Card 85
1.5.5 View the Material Collector Card 86
1.5.6 Create the Material STEEL 87
1.5.7 Assign the Material to the Component 88
1.5.8 Understand HyperMesh Entity Configurations and Types 89
1.5.9 Create a New Component Collector 90
1.5.10 Create a Reference Node 91
1.5.11 Create a Rigid Link 92
1.5.12 Move the *KINEMATIC COUPLING Entities 94
1.5.13 Use Step Manager for Initial Conditions 95
1.5.14 Define *BOUNDARY 96
1.5.15 Define History Data 98
1.5.16 Define the Abaqus Step 98
1.5.17 Specify Output Requests 101

2 AcuSolve 103

2.1 User Interface Introduction 104
2.1.1 ACU-T: 1000 HyperWorks UI Introduction 104
2.2 Turbulence 143
2.2.1 ACU-T: 2000 Turbulent Flow in a Mixing Elbow 143
2.2.2 ACU-T: 2300 Atmospheric Boundary Layer Problem – Flow Over Building 152
2.3 Heat Transfer 167
2.3.1 ACU-T: 3000 Enclosed Hot Cylinder: Natural Convection 167
2.3.2 ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow 186
2.3.3 ACU-T: 3101 Transient Conjugate Heat Transfer in a Mixing Elbow 197
2.3.4 ACU-T: 3110 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping using acuOptiStruct 209
2.3.5 ACU-T: 3200 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model 234
2.3.6 ACU-T: 3201 Solar Radiation and Thermal Shell Tutorial 255
2.3.7 ACU-T: 3202 Heat Transfer Between Concentric Spheres – P1 Radiation Model 276

2.3.8 ACU-T: 3203 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model 289
2.3.9 ACU-T: 3204 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model 301
2.3.10 ACU-T: 3300 Modeling of a Heat Exchanger Component 324
2.3.11 ACU-T: 3310 Single Phase Nucleate Boiling 334
2.3.12 ACU-T: 3311 Multiphase Nucleate Boiling Using the Algebraic Eulerian Model 350
2.4 Multiphase 364
2.4.1 ACU-T: 4000 Transient Dam Break Simulation 364
2.4.2 ACU-T: 4001 Water Filling in a Tank 375
2.4.3 ACU-T: 4002 Sloshing of Water in a Tank 386
2.4.4 ACU-T: 4100 Multiphase Flow using Algebraic Eulerian Model 398
2.4.5 ACU-T: 4200 Humidity – Pipe Junction 413
2.4.6 ACU-T: 4201 Condensation & Evaporation - Air Box 426
2.5 Moving Body 449
2.5.1 ACU-T: 5000 Centrifugal Air Blower with Moving Reference Frame (Steady) 449
2.5.2 ACU-T: 5001 Blower - Transient (Sliding Mesh) 459
2.5.3 ACU-T: 5100 Modeling of a Fan Component: Axial Fan 473
2.5.4 ACU-T: 5201 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve 484
2.6 Porous Medium 507
2.6.1 ACU-T: 6010 Flow Through Porous Medium 507
2.7 Optimization 519
2.7.1 ACU-T: 7000 Parametric Optimization with AcuSolve 519
2.7.2 ACU-T: 7001 Shape Optimization using HyperMorph 540
2.7.3 ACU-T: 7010 Shape Optimization using HyperStudy 561

3 ANSYS 583

3.1 HM-4400: Explore the ANSYS Interface 584
3.1.1 Tools Overview 584
3.1.2 Load the ANSYS User Profile 588
3.1.3 Import a Model 589
3.1.4 Export the Model 590
3.2 HM-4410: Set Up a Model in ANSYS 591
3.2.1 Exercise 1: Define Elements, Real Constants, Materials, Properties, and Components 591
3.2.2 Exercise 2: Introduction to ANSYS Load Steps 607
3.3 HM-4420: Define ANSYS Contacts for Models in HyperMesh 624
3.3.1 Load the ANSYS User Profile 624
3.3.2 Retrieve the Model File 624
3.3.3 Open the Contact Browser 625
3.3.4 Create Target (Secondary) Surface 626
3.3.5 Create Contact (Master) Surface 628
3.3.6 Create a Contact Pair 629
3.3.7 Review the Contact Pair 633

3.3.8 Add /SOLU, ANTYPE, and SOLVE in the Control Cards 634
3.4 HM-4430: Define ANSYS Contacts for Models in HyperMesh 636
3.4.1 Load the ANSYS User Profile 636
3.4.2 Retrieve the HyperMesh Model File 636
3.4.3 Create a Contact Pair 637
3.4.4 Create and Attach a Contact Surface 638
3.4.5 Create and Attach a Target Surface 642
3.4.6 Create and Attach a Property and Material to the Contact Pair 647
3.4.7 Review the Contact Pair 647
3.4.8 Add /SOLU and SOLVE in the Control Cards 648
3.5 HM-4450: Introduction to HyperBeam 650
3.5.1 Load the ANSYS User Profile 651
3.5.2 Retrieve the Model File 651
3.5.3 Create Collectors and Attach Element Types and Materials 652
3.5.4 Create a Beam Element 655
3.5.5 Create Coupled DOF - Rigid Elements 658
3.5.6 Create and Attach a Standard Circle Property 661
3.5.7 Create an Arbitrary Property 661
3.5.8 Update the Component Collector with the Beam Section 664
3.5.9 Save your Work 665
3.5.10 Export the Deck to ANSYS *.cdb Format 665
3.6 HM-4460: Composite 667
3.6.1 Load the ANSYS User Profile 667
3.6.2 Load the Model 667
3.6.3 Mesh all of the Surfaces at Once 669
3.6.4 Load the Ply Information from FiberSim 671
3.6.5 Review and Edit the Element Normals 674
3.6.6 Realize Ply Geometry Shape 676
3.6.7 Add an Element Type 678
3.6.8 Update the Component with Element Type 679
3.6.9 Ply Visualization 680
3.6.10 Laminate Realize the Ply Based Model 682
3.6.11 Export the Deck 683

4 Feko 684

4.1 HM-5000: Importing and Editing a Feko Mesh 685
4.1.1 Exercise 1: Importing and Editing a Feko Mesh 685
4.2 HM-5010: Assigning EM Material Properties to a Mesh forFeko 686
4.2.1 Exercise 1: Assigning EM Material Properties to a Mesh for Feko 686

5 LS-DYNA 687

5.1 HM-4605: Define LS-DYNA Model and Load Data, Controls, and Output 688
5.1.1 Exercise 1: Define Model Data for the Head and A-Pillar Impact Analysis 688

5.1.2 Exercise 2: Define Boundary Conditions and Loads for the Head and A-Pillar Impact Analysis 694
5.1.3 Exercise 3: Define Termination and Output for the Head and A-Pillar Impact Analysis 707
5.2 HM-4610: Use Curves, Beams, Rigid Bodies Joints, and Loads in LS-DYNA 711
5.2.1 Exercise 1: Define Model Data for the Seat Impact Analysis 711
5.2.2 Exercise 2: Define Boundary Conditions and Loads for the Seat Impact Analysis 738
5.3 HM-4615: Model Import, Airbags, Export Displayed, and Contacts with LS-DYNA 744
5.3.1 LS-DYNA Overview 744
5.3.2 Load the LS-DYNA User Profile 747
5.3.3 Import the LS-DYNA Model 747
5.3.4 Define *AIRBAG_WANG_NEFSKE 747
5.3.5 Define a Contact Between the Airbag and the Head 754
5.3.6 Define a Contact Between the Plate and the Airbag 757
5.3.7 Review the Created Solver Entities 763
5.3.8 Export the Model 767
5.3.9 Submit the Input File 767
5.3.10 View the Results 768
5.4 HM-4620: Create Rigid Walls, Model Data, Constraints, Cross Sections, and Output with LS-DYNA 769
5.4.1 Overview of *PART_INERTIA, *CONSTRAINED_EXTRA_NODES, *DATABASE_CROSS_SECTION_(Option), and *RIGIDWALL 769
5.4.2 Load the LS-DYNA User Profile 770
5.4.3 Import the LS-DYNA Model 770
5.4.4 Define *PART_INERTIA 770
5.4.5 Create a *DEFINE_BOX that Contains Non-Barrier Nodes 771
5.4.6 Create Initial Velocity 773
5.4.7 View the Nodes in the Node Entity Set 775
5.4.8 Create *CONSTRAINED_EXTRA_NODES_SET 777
5.4.9 Define Nodes in the Constrain Vehicle 778
5.4.10 View Extra Nodes of the Vehicle Mass Rigid Body 779
5.4.11 Create an Entity Set 779
5.4.12 Create a *CONTACT_AUTOMATIC_GENERAL Contact 780
5.4.13 Define Secondary Surface 781
5.4.14 Create an Entity Set 782
5.4.15 Define a Section 783
5.4.16 Define the Location and Size of the Section Plane 784
5.4.17 Specify the Parts Secondary to the Cross Section 788
5.4.18 View the Entities Secondary to the Rigid Wall 788
5.4.19 Create a *DEFINE_BOX that Contains Barrier and Bumper Nodes 788
5.4.20 Create a *RIGIDWALL_PLANAR_FINITE 790
5.4.21 Define the Location and Size of the Rigid Wall 790
5.4.22 Specify Nodes as Secondary to the Rigid Wall 792
5.4.23 Specify Nodes to be Output 792
5.4.24 Export the Model 794
5.4.25 Submit the Input File 794

5.4.26 View the results in HyperView....795
5.5 HM-4625: Assemblies Using DYNA....796
5.5.1 Exercise 1: Weld Between Geometry Surfaces and Shell Elements....796
5.5.2 Exercise 2: Weld Using a Master Connectors File and Duplicate and Reflect Connectors....805
5.5.3 Exercise 3: Create Connectors from Existing Welds to Create New Welds of a Different Type....818
5.5.4 Exercise 4: Swap Welded Part....823
5.5.5 Exercise 5: Troubleshoot Failure of Connectors to Realize....828
5.6 HM-4630: Interfacing for Arbitrary - Lagrangian - Eulerian Capability Using DYNA....834
5.6.1 Tools Overview....835
5.6.2 Load the LS-DYNA Profile....835
5.6.3 Retrieve and View the Model File....835
5.6.4 Update the *SECTION_SOLID Card....836
5.6.5 Assign *EOS_LINEAR_POLYNOMIAL....838
5.6.6 Define *EOS_LINEAR_POLYNOMIAL....840
5.6.7 Define *INITIAL_VOID_PART....840
5.6.8 Define *ALE_REFERENCE_SYSTEM_NODE....841
5.6.9 Define *ALE_REFERENCE_SYSTEM_GROUP....841
5.6.10 Review the *CONTROL_ALE Card....842
5.6.11 Export the Model....842
5.6.12 Submit the LS-DYNA Input File....842
5.6.13 View the Results in HyperView....842
5.6.14 Save Your Work....842
5.7 HM-4635: Create Joints and Check Minimum Time Step....843
5.7.1 Exercise 1: Create Joints....843
5.7.2 Exercise 2: Check the Minimum Time Step....853

6 MADYMO....856

6.1 HM-4900: Create a MADYMO Model....857
6.1.1 Load the MADYMO User Profile....857
6.1.2 Retrieve and View the Model File....860
6.1.3 Create Coordinate Systems....860
6.1.4 Create the Femur Body....861
6.1.5 Create the Tibia Body....861
6.1.6 Create the Ball Body....862
6.1.7 Create the Ground Body....862
6.1.8 Create Geometry....863
6.1.9 Creating Joints....865
6.1.10 Define Systems of Bodies....866
6.1.11 Define the Initial Conditions and Position the Leg....868
6.1.12 Apply Gavity and Create Vector Data....868
6.1.13 Apply the Gravity Vector to the Dummy....869
6.1.14 Create Ball Ellipsoid Characteristics....870
6.1.15 Define Contacts....872
6.1.16 Export and Run the Model....874

6.2 HM-4910: Set up a MADYMO Occupant Safety Analysis in HyperMesh 875
6.2.1 Load the MADYMO User Profile 875
6.2.2 Retrieve and View the Model File 875
6.2.3 Import the Steering Column File 876
6.2.4 Position the Steering Column 876
6.2.5 Import File 877
6.2.6 Position the Dummy H-Point 877
6.2.7 Position the Dummy Limbs 877
6.2.8 Set the Steering Wheel Tilt 878
6.2.9 Define the MADYMO/SYSTEM/FE MODEL Hierarchy 879
6.2.10 Apply a Pulse Function 879
6.2.11 Apply Gravity 880
6.2.12 Define Contacts 880
6.2.13 Add Seatbelts 881
6.2.14 Equivalence Duplicate Nodes 883
6.2.15 Export and Run the Model 884

7 Nastran 885

7.1 HM-4200: Set Up Nastran Static Analysis in HyperMesh 886
7.1.1 Load the Nastran User Profile 886
7.1.2 Retrieve and View the Model File 886
7.1.3 Create Material Collectors and Components 887
7.1.4 Mesh the Geometry 891
7.1.5 Apply Boundary Conditions to the Model 892
7.1.6 Create a Nastran Subcase 898
7.1.7 Write the Nastran Input Deck 901
7.1.8 View the Results 901

8 PAM-CRASH 2G 904

8.1 HM-4700: Using the PAM-CRASH 2G Interface in HyperMesh 905
8.1.1 Load the PAM-CRASH User Profile 905
8.1.2 Retrieve and View the Model File 906
8.1.3 Create Control Cards 907
8.1.4 Assign Element Types 910
8.1.5 Define a Material and /PART Cards 910
8.1.6 Define HyperMesh Groups 912
8.1.7 Define a Rigid Wall 917
8.1.8 Define Attributes in the Card Previewer 921
8.1.9 Create Boundary Conditions 922
8.1.10 Create Time Histories 924
8.1.11 Create a Function 924
8.1.12 Create a Sensor Card 925
8.1.13 Export a Data Deck 926

9 Permas 927

9.1 HM-4800: Setup a Permas Analysis 928
9.1.1 Load the Permas User Profile 928
9.1.2 Retrieve and View the Model File 928
9.1.3 Create a Permas $COMPONENT and $SYSTEM 929
9.1.4 Create a Component Collector 929
9.1.5 Organize the Elements 930
9.1.6 Create Material and Elemental Properties 935
9.1.7 Create Beam Elements 941
9.1.8 Create a Spring Element 947
9.1.9 Create Multi-Point Constraints 952
9.1.10 Set Definition 956
9.1.11 Create Surfaces from Solid Elements 958
9.1.12 Define the Contact Partners 959
9.1.13 Create the Boundary Conditions 963
9.1.14 Assign the Boundary Conditions to a Load Step 967
9.1.15 Define the Load Collector and Contact Property 967
9.1.16 Define the $NLLOAD Cards 970
9.1.17 Define the $SITUATION Card 971
9.1.18 Export the Deck 971
9.1.19 Acknowledgements 971

10 Samcef 972

10.1 HM-4850: Introduction to Samcef 973
10.1.1 Load the Samcef User Profile 973
10.1.2 Retrieve and View the Model File 973
10.1.3 Check the Hypothesis 974
10.1.4 Create a Material 975
10.1.5 Create and Assign Properties 976
10.1.6 Create Load Collectors 979
10.1.7 Create Constraints 980
10.1.8 Create Forces 981
10.1.9 Create a Samcef Database 983
10.1.10 Export the File 983

Index 984

Abaqus

1

Abaqus tutorials.

This chapter covers the following:

- 1.1 HM-4300: Set Up an Abaqus Analysis in HyperMesh (p. 17)
- 1.2 HM-4310: Define Abaqus Contacts for 2D Models (p. 29)
- 1.3 HM-4320: Define Abaqus Contacts for 3D Models (p. 43)
- 1.4 HM-4330: Define *STEP using Abaqus Step Manager (p. 61)
- 1.5 HM-4340: Pre-Processing for Bracket and Cradle Analysis (p. 82)

1.1 HM-4300: Set Up an Abaqus Analysis in HyperMesh

In this tutorial you will setup an Abaqus analysis in HyperMesh.

You will learn how to:

- Load the Abaqus user profile and a model
- Define a material and properties, and assign them to a component
- View a *SOLID SECTION for solid elements
- Define *SPRING properties and create a component collector for them
- Create a *SPRING1 element
- Assign a property to selected elements

This exercise uses the `abaqus3_0tutorial.hm` file, which can be found in `<hm.zip>/interfaces/abaqus/`. Copy the file(s) from this directory to your working directory.

1.1.1 Load the User Profile and Model

In this step you will load the Abaqus user profile and model file.

When you load a user profile, HyperMesh opens the applicable utility menus, removes the unused panels, disables unneeded entities in the **Find**, **Mask**, **Card** and **Reorder** panels, and makes specific adaptations related to the Abaqus solver.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Abaqus, Standard 3D**.
3. Open the model file by clicking **File** > **Open** > **Model** from the menu bar, or by clicking  on the Standard toolbar.
4. In the **Open Model** dialog, open the file, `abaqus3_0tutorial.hm`.
   The model appears in the graphics area.

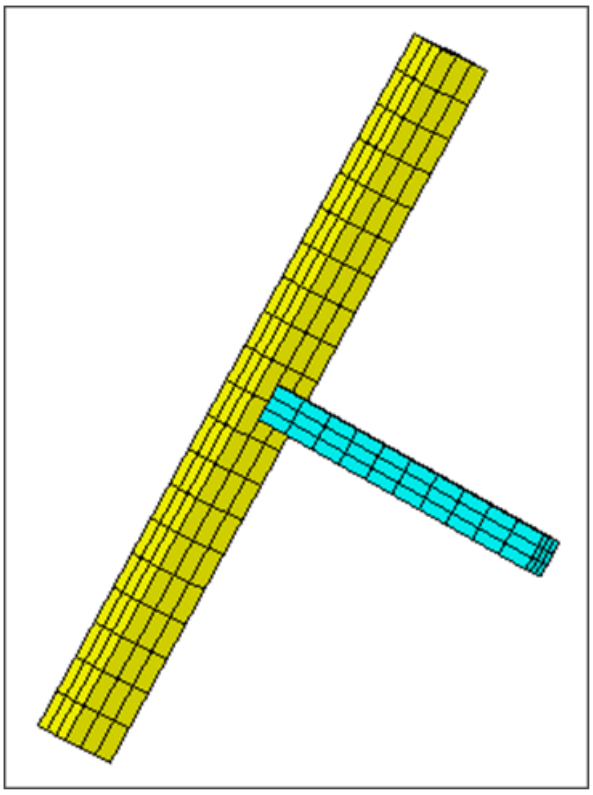


Figure 1:

1.1.2 Define the Material Properties

In this step you will define the material properties.

HyperMesh supports many different material models for Abaqus. In this step, you will create the basic *ELASTIC material model with no temperature variation. You will then assign the material to the property, which is assigned to a component collector.

1. In the Model Browser, right-click and select **Create** > **Material** from the context menu. HyperMesh creates and opens a material in the Entity Editor.

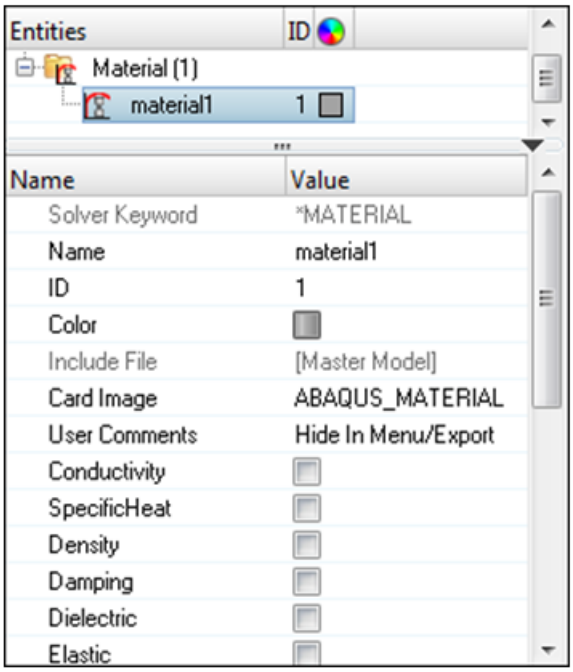


Figure 2:

2. For Name, type `STEEL`.
3. Optional: For ID, type a new ID for the material.
4. Optional: Click the **Color** icon and select a color to display the material.
5. Select the Elastic checkbox.
   HyperMesh displays additional parameters. By default, the selected type is ISOTROPIC and the **ELASTIC INFO**field value is 1.
6. In the E field, type `2.1E5`.
7. In the NU field, type `0.3`.
8. Open the Solver Browser by clicking **View** > **Browsers** > **HyperMesh** > **Solver** from the menu bar.
9. In the Solver Browser, review the material you just created.

1.1.3 Define the *SOLID SECTION Properties

In this step you will define the *SOLID SECTION properties.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu.
   HyperMesh creates and opens a property in the Entity Editor.

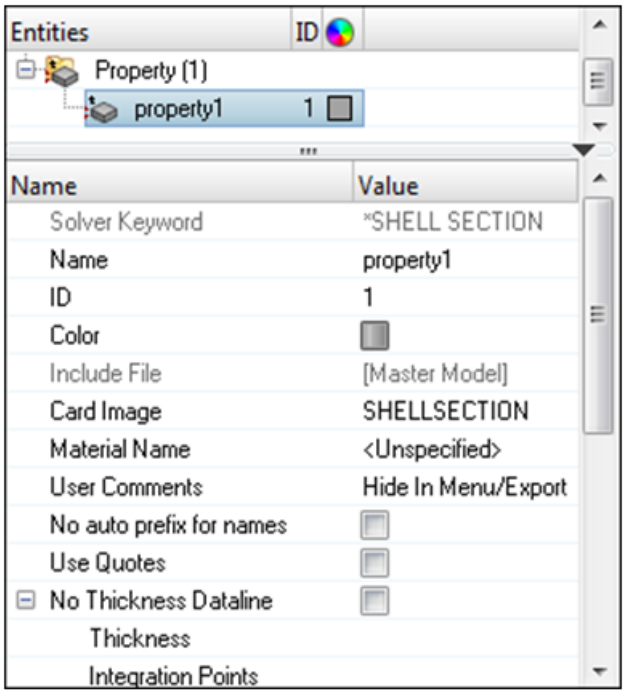


Figure 3:

2. For Name, type `Solid_Prop`.
3. Click the **Color** icon and select a color to display the property.
4. For Card Image, select **SOLIDSECTION**.

> **Note:** This ensures that sections pertaining only to solid elements are available as card image options.

5. In the HyperMesh dialog, click **Yes** to proceed.
6. For Material Name, click **Unspecified >> Material**.

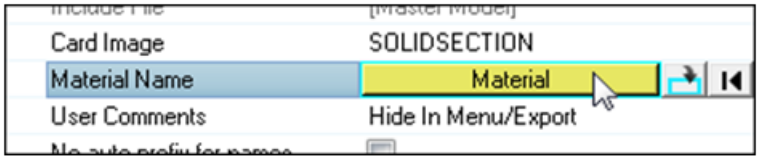


Figure 4:

7. In the **Select Material** dialog, select **STEEL** and then click **OK**.
HyperMesh assigns the material STEEL to the property Solid_Prop.

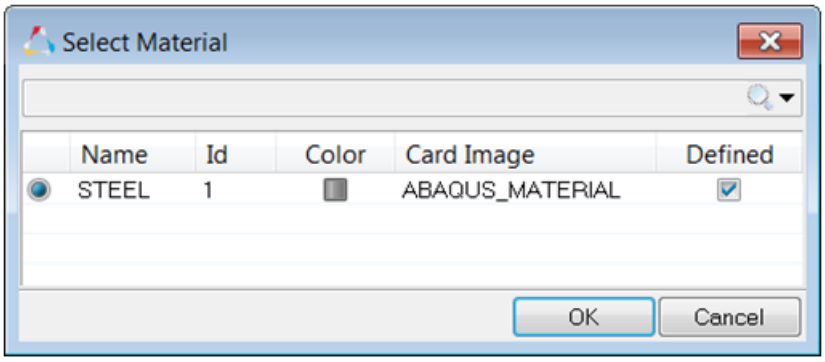


Figure 5:

1.1.4 Assign the Property

In this step you will assign the property to the component.

When a material is assigned to a property, when a property is assigned to a component, the material is automatically assigned as well.

1. In the Model Browser, **Component** folder, select **BEAM** and **INDENTOR**.

> **Note:** You can select multiple components by pressing **Ctrl** while selecting components.

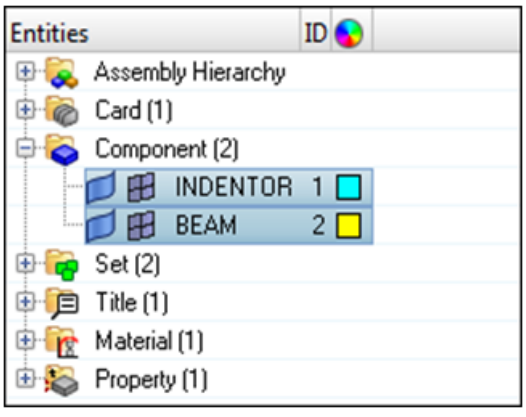


Figure 6:

2. Right-click on the highlighted components and select **Assign** from the context menu.
3. In the **Assign to Component(s)** dialog, select **Solid_Prop** from the **Property** list.

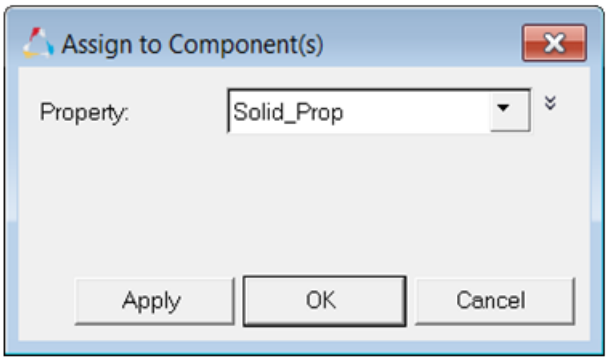


Figure 7:

4. Click **OK**.
   HyperMesh assigns the property Solid_Prop to the components BEAM and INDENTOR.

1.1.5 View the Solid Elements

In this step you will view the *SOLID SECTION for solid elements.

HyperMesh supports sectional properties for all elements from the property collector.

In the Model Browser, **Property** folder, select **Solid_Prop**.
HyperMesh opens the Entity Editor and displays the property's corresponding data. The Card Image shows the keyword related to this property.

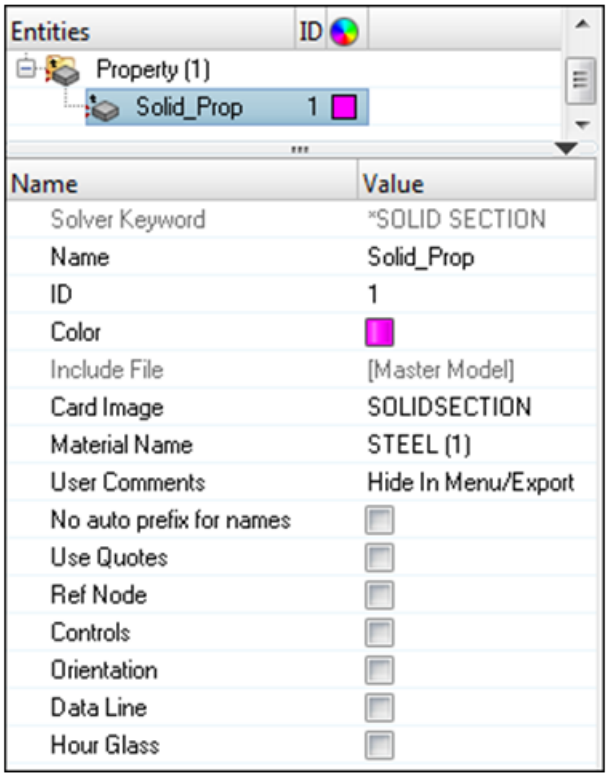


Figure 8:

1.1.6 Define the *SPRING Properties

In this step you will define the *SPRING properties.

In Abaqus contact problems, it is common to use grounded springs to provide stability to the solution in the first loading step.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu. HyperMesh creates and opens a property in the Entity Editor.
2. For **Name**, type `Spring_Prop`.
3. Click the **Color** icon and select a color to display the property.
4. For **Card Image**, select **SPRING**.
5. In the HyperMesh dialog, click **Yes** to proceed.
6. For **dof1**, type `3`.

   Note: The **dof2** parameter in the ***SPRING** card is ignored by Abaqus for SPRING1 elements.

7. In the **Stiffness** field, type `1.0E-5`.

1.1.7 Create a Component Collector

In this step you will create a component collector for the *SPRING property.

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu. HyperMesh creates and opens a component in the Entity Editor.

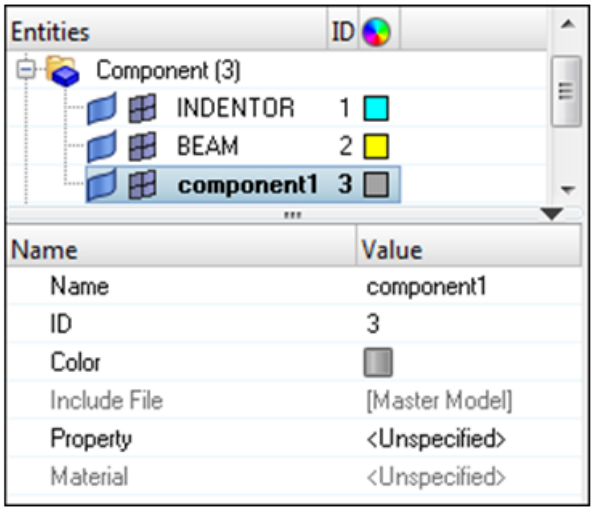


Figure 9:

2. For Name, type `GROUNDED`.
3. Click the **Color** icon and select a color to display the component.
4. For Property, click **Unspecified >> Property**.

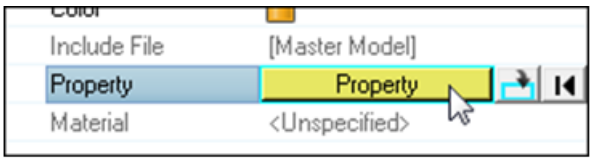


Figure 10:

5. In the **Select Property** dialog, select **Spring_Prop** and then click **OK**. HyperMesh assigns the property **Spring_Prop** to the component **GROUNDED**.

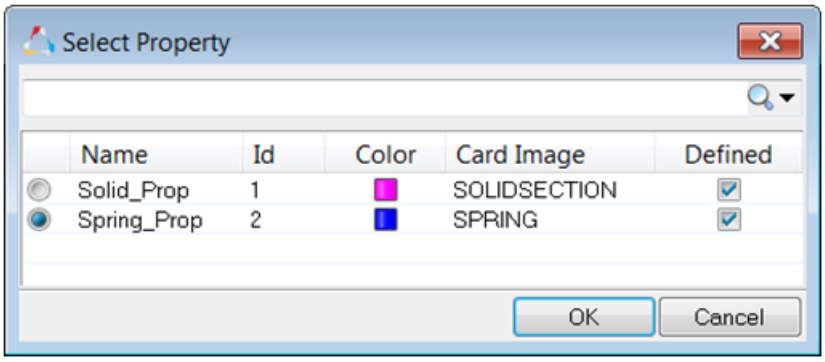


Figure 11:

1.1.8 Reset the View

In this step you will reset the view for further processing.

On the Standard Views toolbar, click .

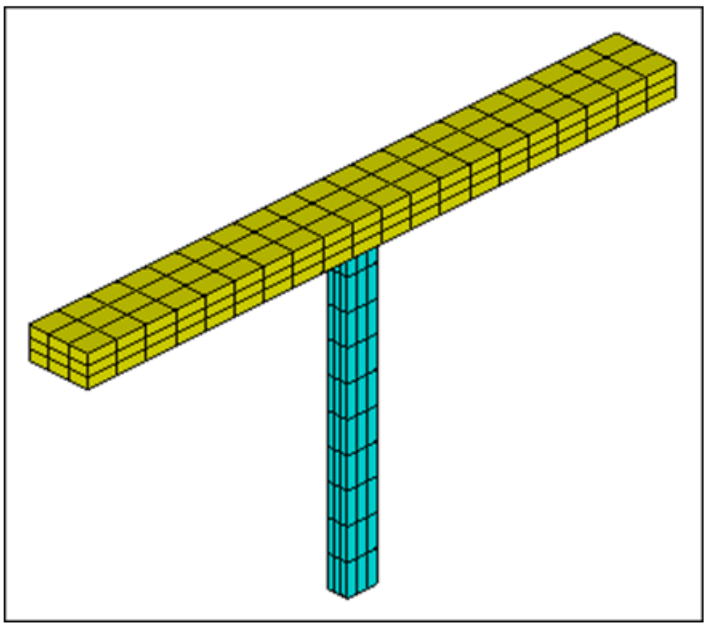


Figure 12:

1.1.9 Create the SPRING1 Element

In this step you will create the SPRING1 element.

1. Open the Element Type panel by clicking **Mesh** > **Assign** > **Element Type** from the menu bar.
2. Go to the **1D** subpanel.

3. Click **mass=**, then select **SPRING1**.

   Note: In HyperMesh, grounded elements are created and stored as mass elements since they only have one node in the element connectivity.

4. Exit the panel by clicking **return**.
5. In the Model Browser, **Component** folder, right-click on **GROUNDED** and select **Make Current** from the context menu.
6. Open the Masses panel by clicking **Mesh** > **Create** > **Masses** from the menu bar.
7. Click **nodes >> by id**.
8. In the id= field, type `451t460b3` and then press **Enter**.
   HyperMesh selects all nodes from 451 to 460 in increments of 3.

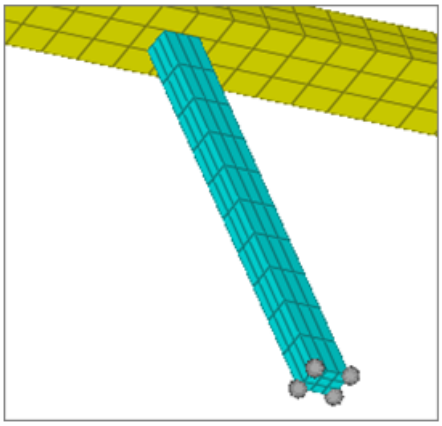


Figure 13:

9. Click **create**. HyperMesh creates SPRING1 elements.

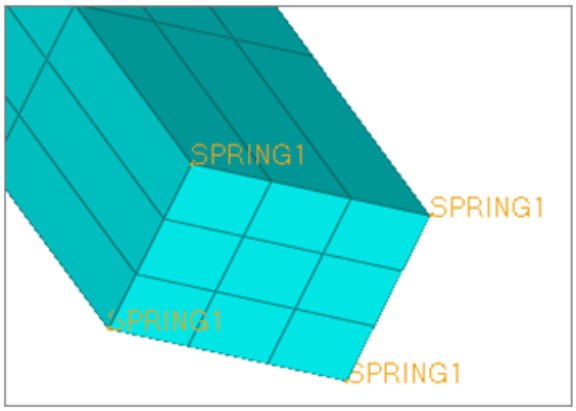


Figure 14:

10. Click **return**.

1.1.10 Assign a Property to Selected Elements Overview

In most cases a property, for example, SHELL SECTION, SOLID SECTION, is assigned to a component collector. The elements belonging to the component collector assume the assigned property.

It is also possible in HyperMesh 11.0 to assign a property to individual elements without having to organize these elements in a separate component collector. A property can be assigned directly to selected elements. In this case, HyperMesh automatically creates an Abaqus element set (ELSET) for the selected elements and assigns the property to the ELSET.

The name of the ELSET follows the naming convention `HMprop_propertyname`, where `propertyname` is the name of the property collector that is assigned to the elements. In the example below, a property collector called `property1` is created and assigned to elements 1. This information is translated in the Abaqus `*.inp` file as:

```
**  Template:  ABAQUS/STANDARD 3D
**
*NODE
        1, 2.5          , 0.0         , 2.5
        2, 2.5          , 0.0         , -2.5
        3, -2.5         , 0.0         , -2.5
        4, -2.5         , 0.0         , 2.5
**HWCOLOR COMP     1     11
*ELEMENT,TYPE=S4R,ELSET=auto1
        1,          1,          2,          3,          4
*ELSET, ELSET=HMprop_property1
        1
**HM set by property     11         22
*SHELL SECTION, ELSET=HMprop_property1, MATERIAL=
```

The `HMprop_ prefix` can be suppressed in the property's card image by selecting the **No_auto_prefix_for_names** checkbox.

The comment `**HM_set_by_property` 11 is written by HyperMesh to the `*.inp` file so that the element property definition and assignment are read properly when the `*.inp` file is imported in HyperMesh. The number 11 refers to the color chosen for the property and 22 as the property ID.

1.1.11 Create a Property

In this step you will create a property.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu. HyperMesh creates and opens a property in the Entity Editor.
2. For **Name**, type `ElemPrp`.
3. Click the **Color** icon and select a color to display the property.
4. For **Card Image**, select **SOLIDSECTION**.

   Note: This ensures that sections pertaining only to solid elements are available as card image options.

5. In the HyperMesh dialog, click **Yes** to proceed.

1.1.12 Assign a Property to Individual Elements

In this step you will assign a property to individual elements.

1. In the Model Browser, **Property** folder, right-click on **ElemPrp** and select **Assign** from the context menu.
2. On the Standard Views toolbar, click .
3. In the graphics area, select the leftmost and rightmost layers of solid elements belonging to the horizontal BEAM component.

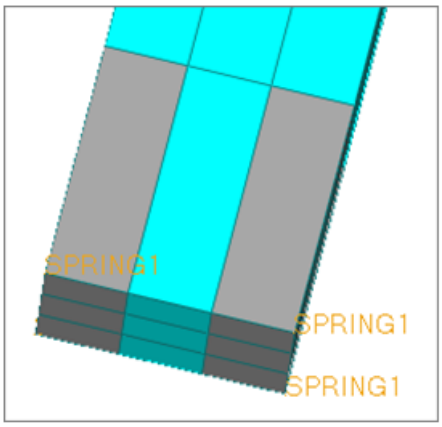


Figure 15:

4. Click **proceed**. HyperMesh assigns the property.
5. In the **Mixed Property Warning** dialog, click **OK**.
6. On the Visualization toolbar, select **By Prop** from the Element Color Mode list.
HyperMesh colors the elements by their property assignment.

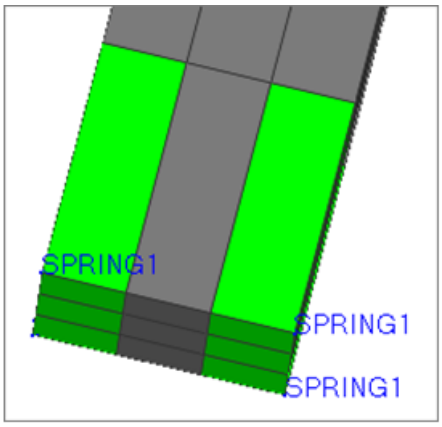


Figure 16:

When a property is assigned to a component collector, HyperMesh writes out the comment *HM_comp_by_property to distinguish the component property assignment from that of individual element.

1.2 HM-4310: Define Abaqus Contacts for 2D Models

In this tutorial you will define Abaqus contacts for 2D models.

You will learn how to:

- Load the Abaqus user profile and model
- Start Abaqus Contact Manager
- Define surfaces for 2D solid elements
- Define surfaces by set
- Define surface interaction property
- Define contact pair

This exercise uses the `abaqus_contactManager_2D_tutorial.hm` file, which can be found in `<hm.zip>/interfaces/abaqus/`. Copy the file(s) from this directory to your working directory.

1.2.1 Load the User Profile and Model

In this step you will load the Abaqus user profile and model.

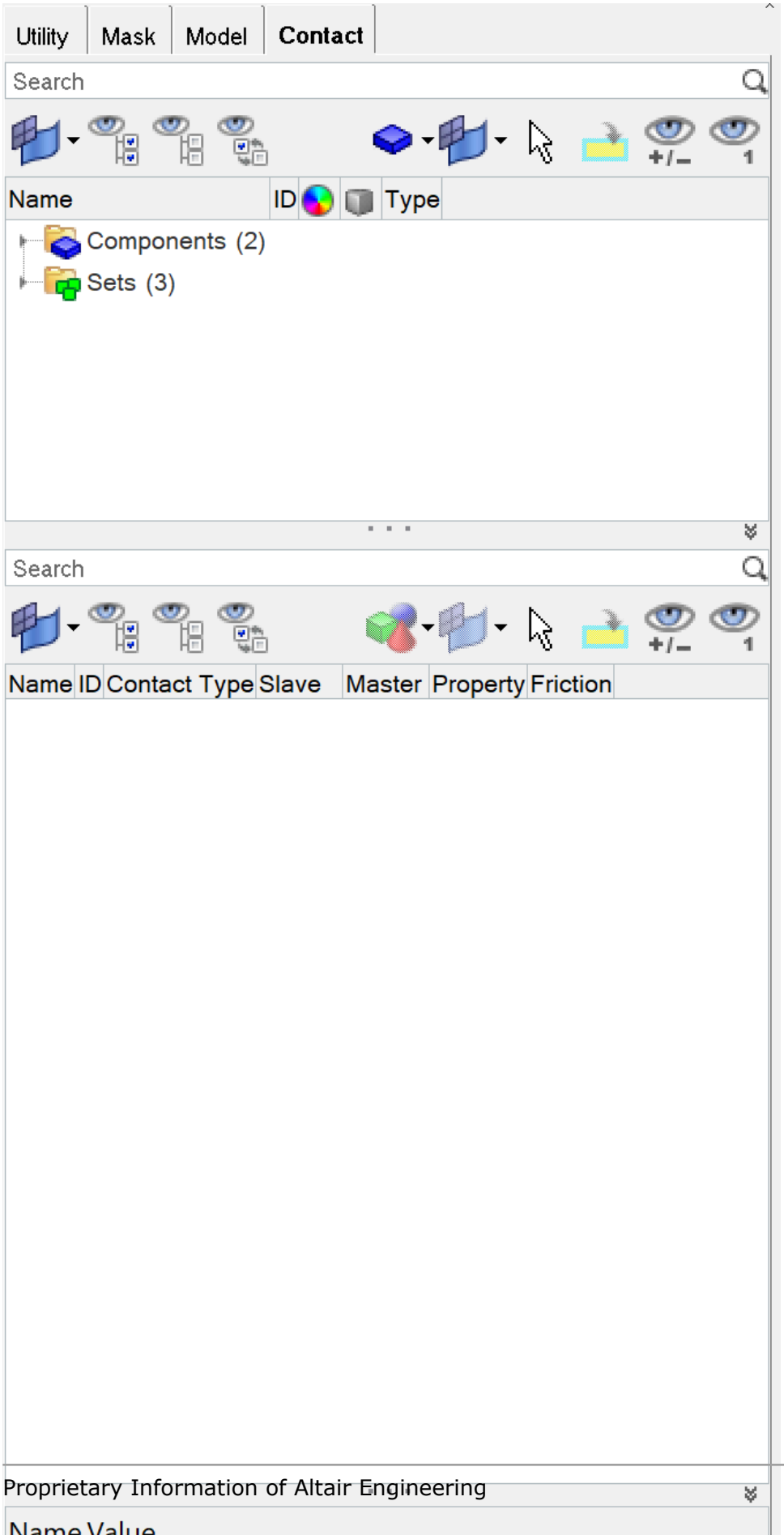
A set of standard user profiles is included in the HyperMesh installation. While the user profiles change the appearance of some panels, they do not affect the internal behavior of each function.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to**Abaqus, Standard 2D**.
3. Open a model file by clicking **File** > **Open** > **Model** from the menu bar, or by clicking  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `abaqus_contactManager_2D_tutorial.hm` file.
   The model appears in the graphics area.

1.2.2 Start the Contact Browser

In this step you will start the contact browser.

From the menu bar, click **Tools** > **Contact Browser**. The **Abaqus Contact Browser** opens.


Utility
Mask
Model
Contact
Search
Name
ID
Type
Components (2)
Sets (3)
Search
Name
ID
Contact Type
Slave
Master
Property
Friction
Name
Value

1.2.3 Define Surfaces for 2D Solid Elements

In this step you will define surfaces for 2D solid elements.

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual element IDs or sets with corresponding face identifiers.

1. In the Abaqus Contact Manager, click the **Surface** tab.
2. Click **New**.
   The **Create New Surface** dialog opens.
3. In the Name field, type `frame-top`.
4. Set Type to **Element based**.
5. Click the box next to **Color** and select the desired color.
6. Click **Create**.
   The **Element Based Surface** dialog for defining elements and corresponding faces for the surface opens.
7. Click the **Define** tab.

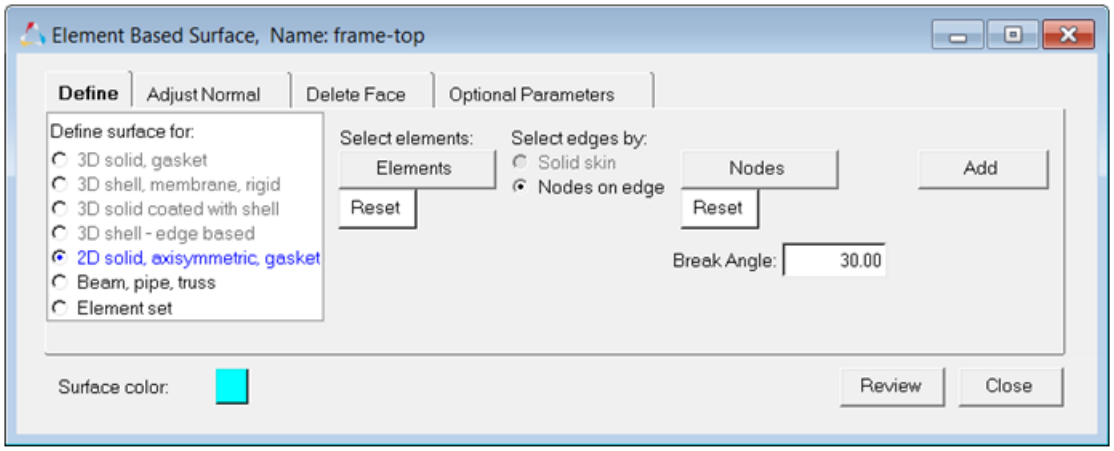


Figure 18:

8. Set Define surface for to **2D solid, axisymmetric, gasket**.
9. Under Select Elements, click **Elements**.
10. In the panel area, click **elems >> by collector**.
11. Select **frame**.
12. Click **select**.
    The elements in the frame component highlight.

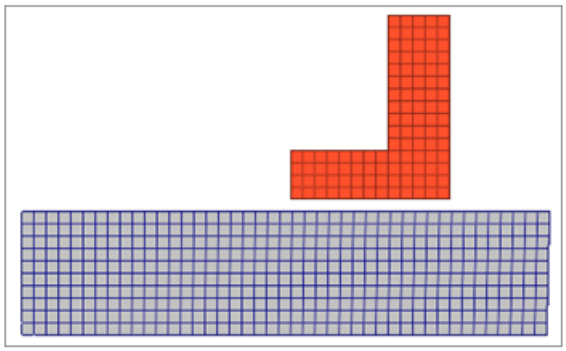


Figure 19:

13. Click **proceed**.
14. In the **Element Based Surface** dialog, set Select edges by to **Nodes on edge**.
15. Click **Nodes**.
16. In the panel area, select two nodes from the top of a selected solid element as shown in the image below.

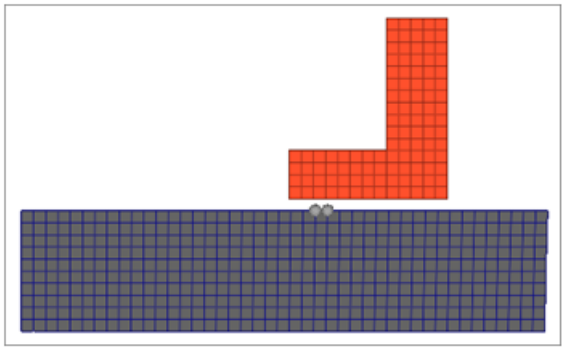


Figure 20:

17. Click **proceed**.
18. In the **Element Based Surface** dialog, type `30.00` in the Break Angle field.
19. Click **Add**.
    All of the edges of the selected solids that fall within the break angle of the edge defined by the two nodes are found.

    These edges are added to the current surface, and special contact surface elements are created to show the normal direction of those elements. In the current example, they show towards the component slider, which is correct.

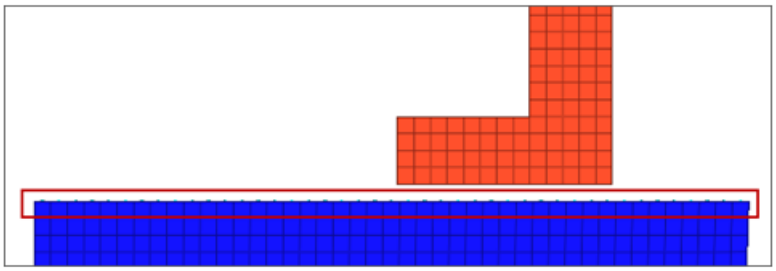


Figure 21:

20. Click the **Adjust Normal** tab to change the normal direction. The element normal should be adjusted towards the mating surface. The element normal should be adjusted towards the mating surface.

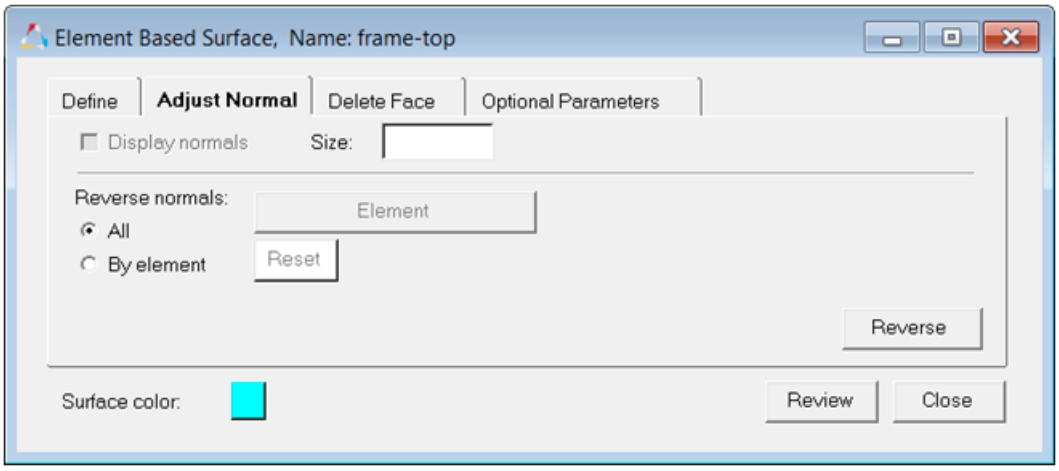


Figure 22:

21. Click the **Delete Face** tab to delete the edges that you do not want.

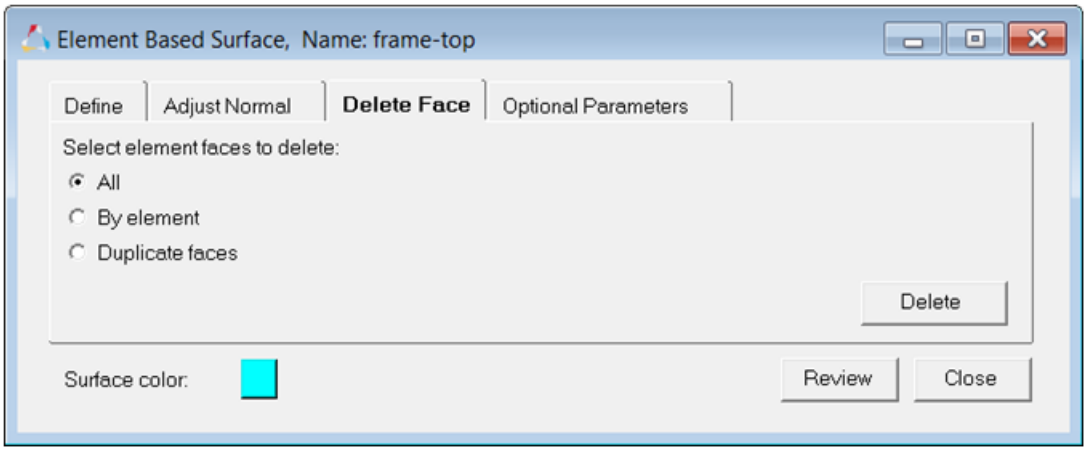


Figure 23:

22. When satisfied with the surface definition, click **Close** to return to the Abaqus Contact Manager.

1.2.4 Define Surfaces by Set

In this step you will create surfaces by defining a set and corresponding face identifiers.

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual element IDs or sets with corresponding face identifiers.

1. In the **Abaqus Contact Manager**, **Surface** tab, click **Display None** to turn off the display of all surfaces.
2. Click **New**.
   The **Create New Surface** dialog opens.
3. In the Name field, type `slider-bot`.
4. Set Type to **Element based**.
5. Click the box next to **Color** to select the desired color.
6. Click **Create**.
   The **Element Based Surface** dialog opens, and you can begin defining elements and corresponding faces for the surface.
7. Click the **Define** tab.
8. Set Define surface for to **Element set**.
9. Set Element set to **slider-bot**.
10. Click **Review Set**.
    All elements in the set highlight.

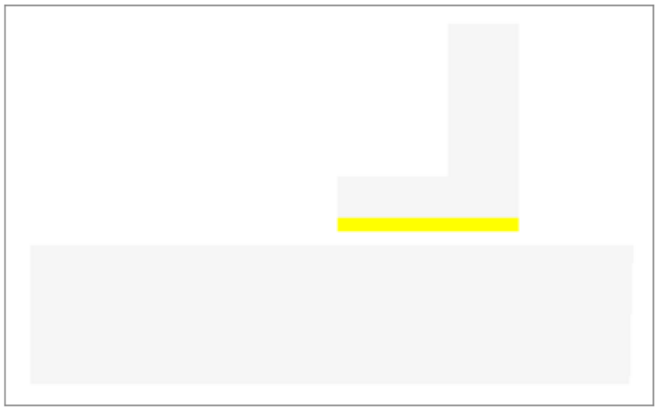


Figure 24:

11. Right-click **Review Set** to reset the highlighting.
12. Click the right arrow to move the slider-bot element set name into the table.
13. In the Face column, select **NoLabel**.
14. Click **Update**.
    A new dataline for ELSET slider-bot has been added to the ***SURFACE** card. By selecting **NoLabel** the face identifier has been left blank.

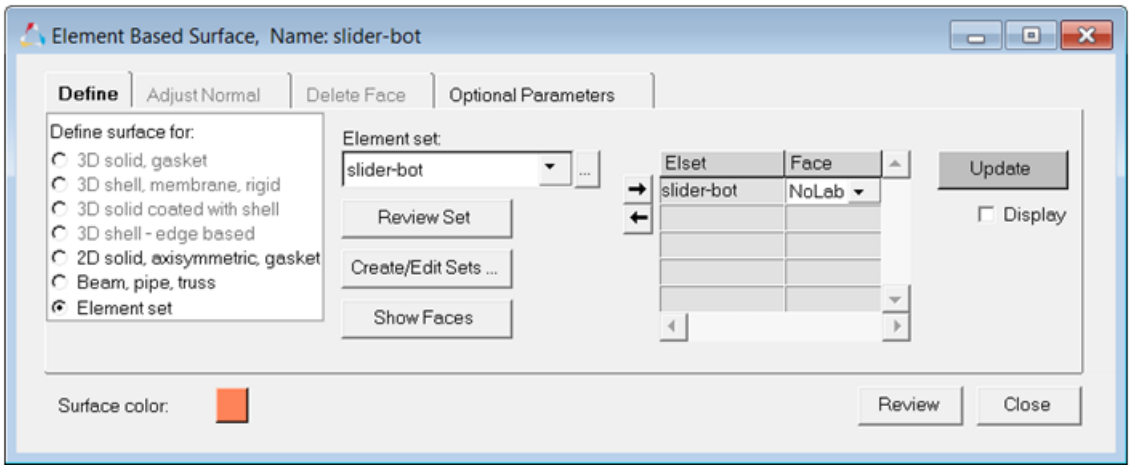


Figure 25:

15. Click **Review** to review the contents of the *SURFACE card. Right-click on **Review** to reset the highlighting.

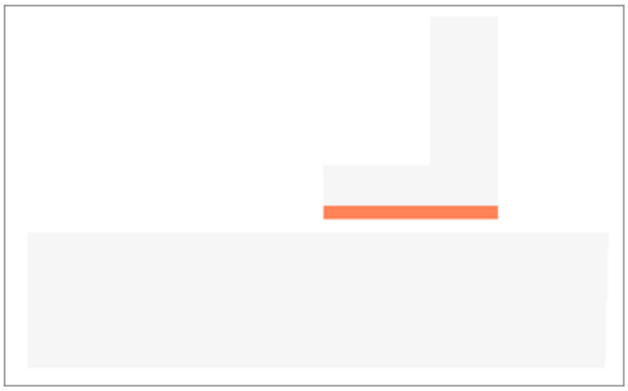


Figure 26:

16. Click the **Optional Parameters** tab.

17. Select the **Trimming of open free surface** checkbox and select **YES**.

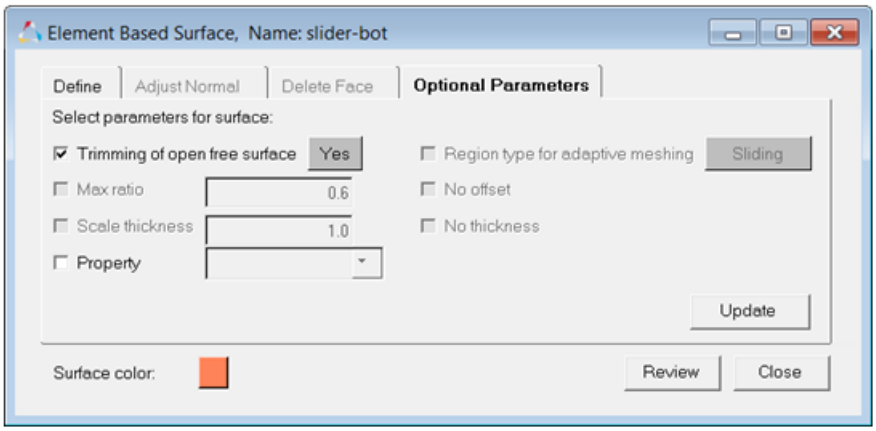


Figure 27:

18. Click **Close** to return to the Abaqus Contact Manager. In the Surface tab, the Display option for the slider-bot surface is disabled.
    At this point, you have created all the surfaces required.
19. Click **Display All** to display all surfaces.

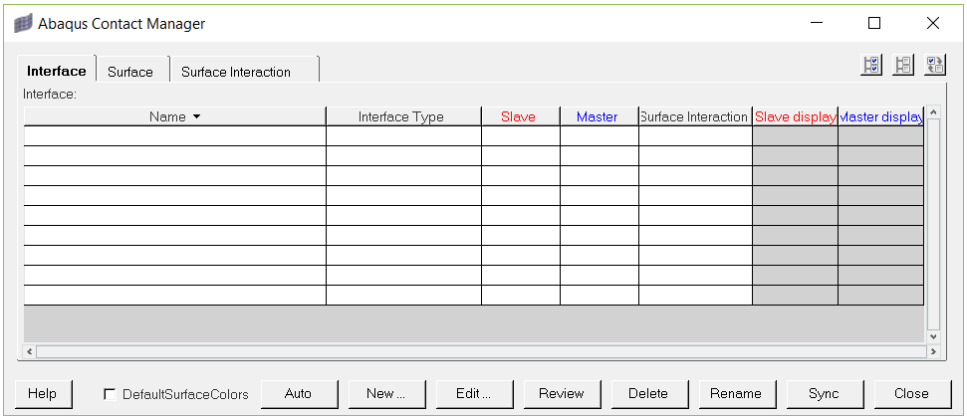


Figure 28:

20. Review surfaces by selecting them from the table and clicking **Review**. The selected surface will be highlighted with white and show up through the solid mesh when using performance graphics. If the surface is defined with sets (display option disabled), the underlying elements are highlighted. Right-clicking on **Review** will clear the highlighting.

1.2.5 Define the Surface Interaction Property

In this step you will define the *SURFACE INTERACTION card with the corresponding *FRICTION card.

1. In the **Abaqus Contact Manager**, click the **Surface Interaction** tab.
2. Click **New**.
   The **Create New Surface Interaction** dialog opens.

3. In the Name field, type `friction1`.
4. Click **Create**.
   The **Surface Interaction** dialog opens.
5. Click the **Define** tab.
6. Set **Select mechanical interaction properties** to **Friction**.



Figure 29:

7. Click the **Friction** tab.
8. Set Friction type to **Default**.
9. In the second pane, select **Direct**.

Note: Selecting this option means that the exponential decay and Anisotropic parameters will not be written to the input file.

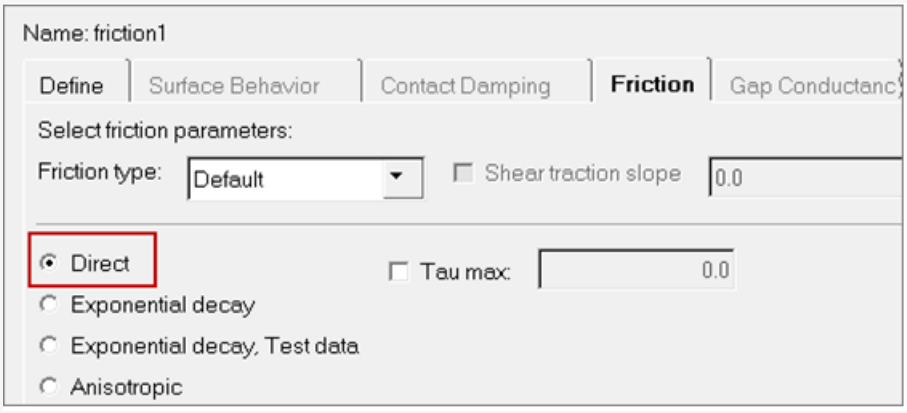


Figure 30:

10. In the No of data lines field, type `1`.
    A single row displays in the Direct table.
11. In the Friction Coeff column, click the first cell and type `0.05`.

For Direct and Anisotropic tables:

- Change the number of columns in the table by specifying a value in the No of Dependencies field; change the number of rows in the table by specifying a value in the No of data lines field.
- Enter values in the table by clicking a cell to make it active and then typing in values. The table works like a regular spreadsheet.
- Read comma-delimited data from a text file by clicking **Read From a File**. In the File Browser, select a file and click **Open** to export the comma-delimited data. The row number will be set to the number of data lines found in the file.
- Access copy, cut, and paste options by right-click in the table. Comma-separated data can be copied/cut into or pasted from clipboard with these options. Relevant hot keys (for example, Ctrl-C, Ctrl-X and Ctrl-V in Windows) will also work.
- Activate cells by left-clicking in a cell. Clicking into an already active cell moves the insertion cursor to the character nearest the mouse.
- Highlight cells by left-clicking while moving the mouse over a cell.
- Move the active cell using the left, right, up, and down arrows.
- Extend the selection in a specific direction using Shift-<arrow>.
- Move the insertion cursor within a cell using Ctrl-left arrow and Ctrl–right arrow.
- Selects all cells using Ctrl-slash.
- Delete the character before the insertion cursor in the active cell using Backspace. If multiple cells are selected, Backspace deletes all selected cells.
- Remove the character after the insertion cursor in the active cell using Delete. If multiple cells are selected, Delete removes all selected cells.
- Move the insertion cursor to the beginning of the active cell using Ctrl-A. Move the insertion cursor to the end of the active cell using Ctrl-E.
- Decrease and increase the width of the column with the active cell in it using Ctrl-minus (-) and Ctrl-equal (=).
- Interactively resize a row or column by left-clicking or right-clicking on a border while moving the mouse.

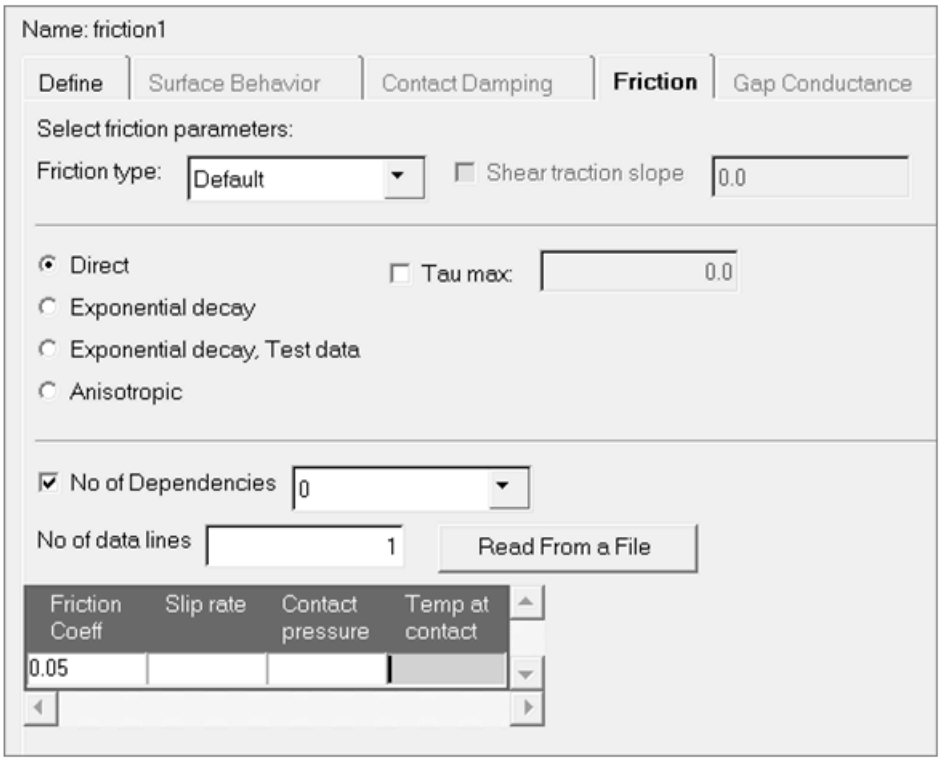


Figure 31:

12. Click **OK** to go back to the Abaqus Contact Manager.

1.2.6 Define the Contact Pair

In this step you will define the *CONTACT PAIR card with corresponding surfaces and surface interactions.

1. In the **Abaqus Contact Manager**, click the **Interface** tab.
2. Click **New**.
   The **Create New Interface** dialog opens.
3. In the Name field, type `slider-frame`.
4. Set Type to **Contact pair**.
5. Click **Create**.
   The **Contact Pair** dialog opens.
6. Click the **Define** tab.
7. Set Surfaces to **slider-bot**.

> **Note:** Clicking **New** opens the **Create New Surface** dialog, from which a new surface can be created. When you are done creating and defining the surface, the **Contact Pair** dialog will return with the new surface selected as the secondary surface.

To ensure that you are dealing with the right surface, click **Review**. The selected surface is highlighted in red, while the rest of the model is grayed out. Right-click on **Review** to clear the highlighting.

8. Click **Slave>>** to move slider-bot into the table as the secondary surface.
9. Repeat steps 7 and 8, selecting **frame-top** and clicking **Master>>** to move it into the table as the master surface.

Note: To more clearly see the surfaces available for selection, click . This opens an enhanced browser where you can easily search for the appropriate item. You can also click **Filter** to filter the items displayed.

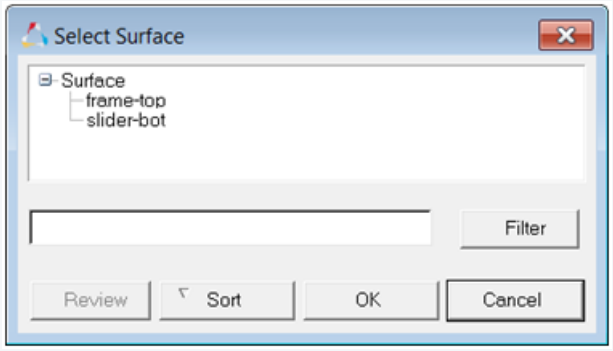


Figure 32:

10. Set Interaction to **friction1**, which is the interaction property for the current contact pair.

Note: To more clearly see the surfaces available for selection, click . This opens an enhanced browser where you can easily search for the appropriate item. You can also click **Filter** to filter the items displayed.

11. Click the **Parameter** tab.
12. Select the **Small sliding** checkbox.
13. Click **OK** to go back to the Abaqus Contact Manager.

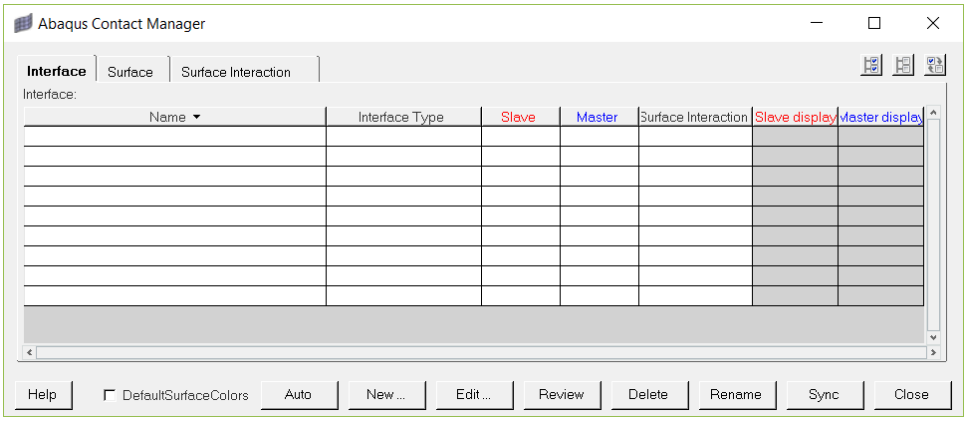


Figure 33:

14. At this point, you have created the contact pairs required. Review any contact pair by selecting it from the table and clicking **Review**. The main surface is displayed in blue, the secondary surface is displayed in red, and the rest of the model is shown in grey. If a surface is defined with sets (display option disabled), the underlying elements are highlighted. Right-clicking on **Review** will clear the highlighting.

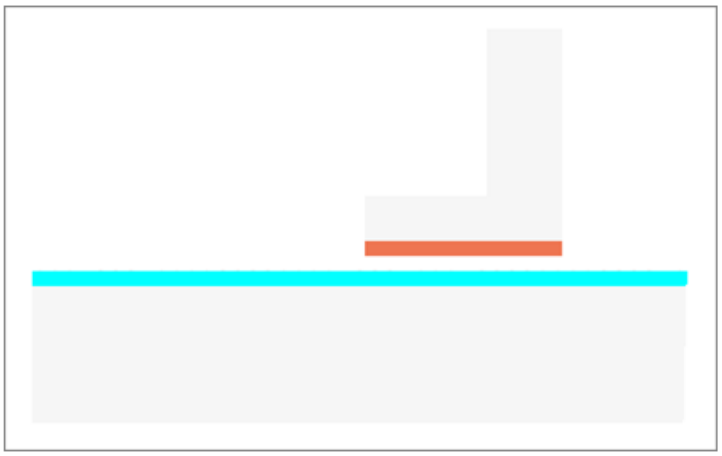


Figure 34:

15. Click **Close** to close the Abaqus Contact Manager.

General comments:

- Click **Edit** to open the dialog for editing the selected interface, surface, or surface interaction.
- Click **Delete** to remove the selected interfaces, surfaces, or surface interactions. Multiple selections can be removed from the Interface table at once.
- Click **Sync** to update the **Contact Manager** with the current HyperMesh database. If you create, update, or delete any components, groups, properties, or entity sets from HyperMesh panels while the Contact Manager is open, click **Sync** to update the Contact Manager.
- If you minimize the Contact Manager dialog or if it goes behind HyperMesh, click **Tools** > **Contact Manager** to restore it.

- Bubble help exists for important buttons. Place the mouse on the buttons for a few moments to view it.
- Double-click on interface, surface, and surface interaction names in the table to open the corresponding edit dialog. Right-click on these names to display a pull down menu with options.
- Left-click or right-click on a table border while moving the mouse can resize columns in a table.
- Shift and Ctrl keys can be used while left-clicking to select multiple items in a table (useful for deleting multiple items).

1.3 HM-4320: Define Abaqus Contacts for 3D Models

In this tutorial you will define Abaqus contacts for 3D models.

You will learn how to:

- Load the Abaqus user profile and model
- Start the Abaqus Contact Manager
- Define surfaces for solid elements
- Define surfaces for shell elements
- Define surfaces by set
- Define surface interaction property
- Define contact pair

This exercise uses the `contactManager_3D_tutorial.hm` file, which can be found in `<hm.zip>/interfaces/abaqus/`. Copy the file(s) from this directory to your working directory.

1.3.1 Load the User Profile and Model

In this step you will load the user profile and model.

A set of standard user profiles is included in the HyperMesh installation. While the user profiles change the appearance of some panels, they do not affect the internal behavior of each function.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Abaqus, Standard 3D**.
3. Open a model file by clicking **File** > **Open** > **Model** from the menu bar, or by clicking on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `abaqus_contactManager_3D_tutorial.hm` file.
   The model appears in the graphics area.

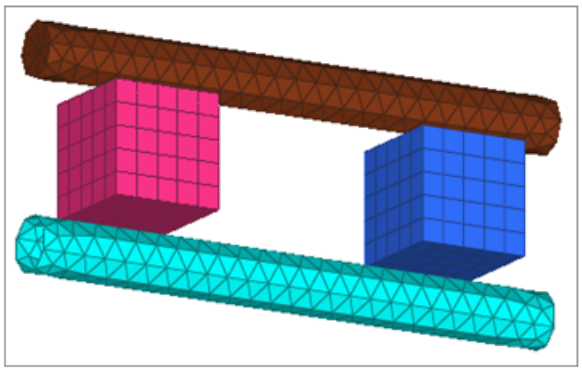
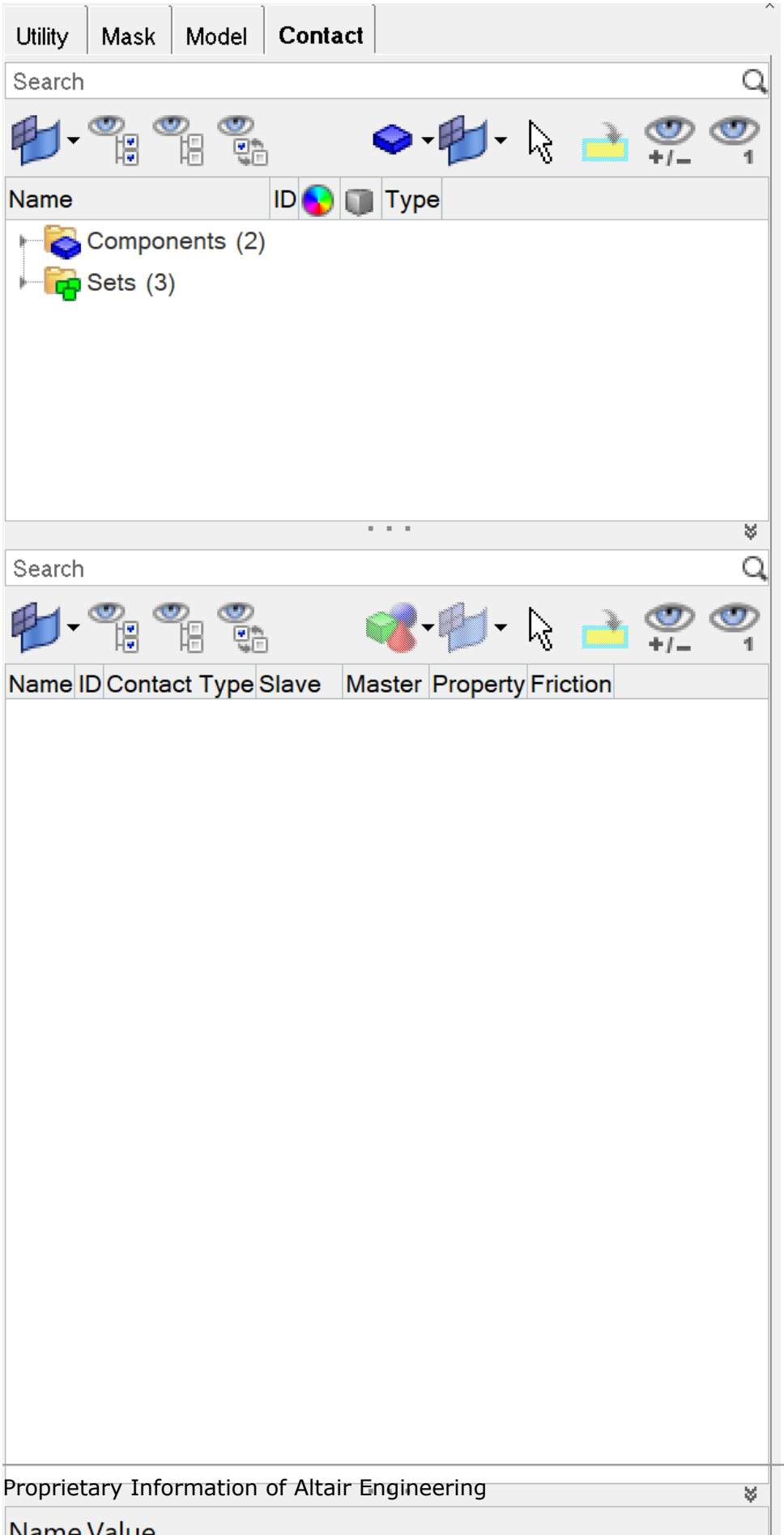


Figure 35:

1.3.2 Start the Contact Browser

In this step you will start the contact browser.

From the menu bar, click **Tools** > **Contact Browser**. The **Abaqus Contact Browser** opens.


Utility
Mask
Model
Contact
Search
Name
ID
Type
Components (2)
Sets (3)
Search
Name
ID
Contact Type
Slave
Master
Property
Friction
Name
Value

1.3.3 Define Surfaces for Solid Elements

In this step you will define surfaces for solid elements.

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual element IDs or sets with corresponding face identifiers. In this exercise, you will create surfaces by defining individual element IDs and corresponding faces.

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **New**.
   The **Create New Surface** dialog opens.
3. In the Name field, type `box1-top`.
4. Set Type to **Element based**.
5. Click the box next to **Color** and select a color.
6. Click **Create**.
   The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
7. In the Model Browser, **Component** folder, right-click on **BOX_1** and select **Isolate** from the context menu.
8. On the Standard Views toolbar, click (**XY Top Plane View**).
9. In the **Element Based Surface** dialog, click the **Define** tab.
10. Set the Define surface for to **3D solid, gasket**.
11. Click **Elements**.
12. In the panel area, click **elems >> by collector**.
13. Select the component, **BOX_1**.
14. Click **select**.
    The elements in the BOX_1 component highlight.

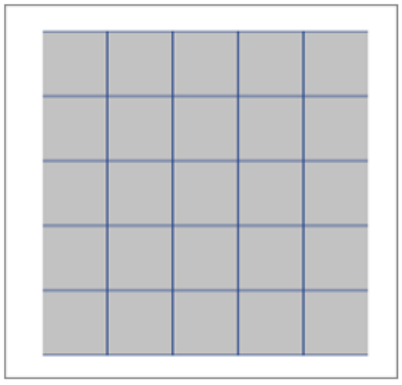


Figure 37:

15. Click **proceed** to return to the **Element Based Surface** dialog.
16. Set **Select faces by** to **Solid skin**.
17. Select a color form the **Solid skin color** button.
18. Click **Faces**.

HyperMesh creates a temporary skin of the selected elements.

19. Select an element from the top of the solid skin.
20. In the panel area, click **elems >> by face**.
    All faces at the top of the solid skin highlight.
21. Rotate the model to verify all desired faces are selected.
22. Optional: Deselect any element by right-clicking or add more if you like.
23. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.
24. Click **Add** to add these faces to the current surface.
    HyperMesh creates special face elements (rectangles with a dot in the middle) for display.

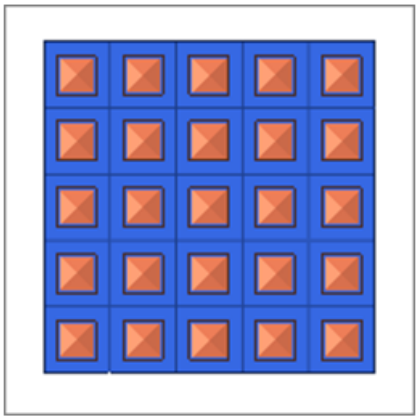


Figure 38:

25. Optional: Reject the recently added faces by clicking **Reject**. You can also delete faces from the Delete page.
26. When satisfied with the surface definition, click **Close** to return to the Abaqus Contact Manager.

1.3.4 Create the Box2-Top Surface

In this step you will create the box2-top surface.

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to turn off the display of all surfaces.
3. Click **New**.
   The **Create New Surface** dialog opens.
4. In the **Name** field, type `box2-top`.
5. Set **Type** to **Element based**.
6. Click the box next to **Color** and select a color.
7. Click **Create**.
   The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.

8. In the Model Browser, **Component** folder, right-click on **BOX_2** and select **Isolate** from the context menu.
9. On the **Standard Views** toolbar, click (**XY Top Plane View**).
10. In the **Element Based Surface** dialog, click the **Define** tab.
11. Set **Define surface for** to **3D solid, gasket**.
12. Click **Elements**.
13. In the panel area, click **elems >> by collector**.
14. Select the component, **BOX_2**.
15. Click **select**.
    The elements in **BOX_2** highlight.
16. Click **proceed** to return to the **Element Based Surface** dialog.
17. Set **Select faces by** to **Nodes on face**.
18. Click **Nodes**.
19. Select two corner nodes (or three nodes) from the top of the selected solids as shown below.

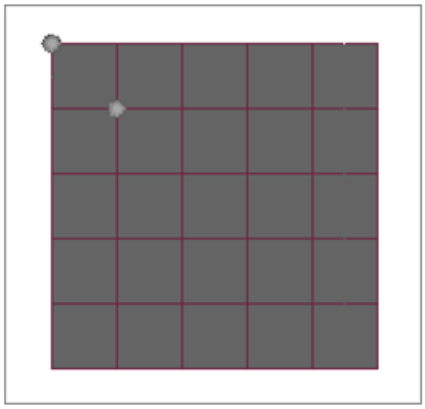


Figure 39:

20. In the panel area, click **proceed** to return to the **Element Based Surface** dialog.
21. In the **Break Angle** field, type `30.00`.
22. Click **Add** to find all faces of the selected solids that fall within the break angle of the face defined by nodes.
    HyperMesh adds these faces to the current surface and creates special face elements (rectangles with a dot in the middle) for display.

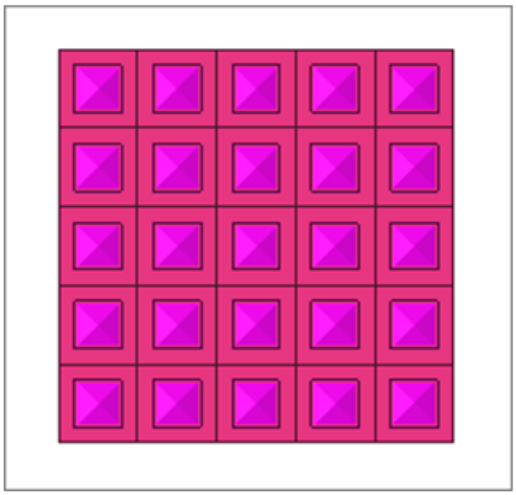


Figure 40:

23. Optional: Reject the recently added faces by clicking **Reject**. You can also delete faces from the **Delete** page.
24. When satisfied with the surface definition, click **Close** to return to the Abaqus Contact Manager.

1.3.5 Create the Cylinder-Top Surface

In this step you will create the cylinder-top surface.

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to turn off the display of all surfaces.
3. Click **New**.
   The **Create New Surface** dialog opens.
4. In the **Name** field, type `cylinder-top`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**.
   The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. In the Model Browser, **Component** folder, right-click on **TOP_CYLINDER** and select **Isolate** from the context menu.
9. In the **Element Based Surface** dialog, click the **Define** tab.
10. Set **Define surface for** to **3D solid, gasket**.
11. Click **Elements**.
12. In the panel area, click **elems >> by collector**.
13. Select the component, **TOP_CYLINDER**.
14. Click **select**.
    Elements in the **TOP_CYLINDER** component highlight.

15. Click **proceed** to return to the **Element Based Surface** dialog.
16. Set **Select faces by** to **Solid skin**.
17. Select a color from the **Solid skin color** button.
18. Click **Faces**.
    HyperMesh creates a temporary skin of the selected elements.
19. Select an element from the solid skin.
20. In the panel area, click **elems >> by face**.
    The faces around the solid skin highlight.
21. Rotate the model to verify all desired faces are selected.
22. Optional: Deselect any element by right-clicking, or add more if you like.
23. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.
24. Click **Add** to add these faces to the current surface.
    HyperMesh creates special face elements (rectangles with a dot at the middle) for display.

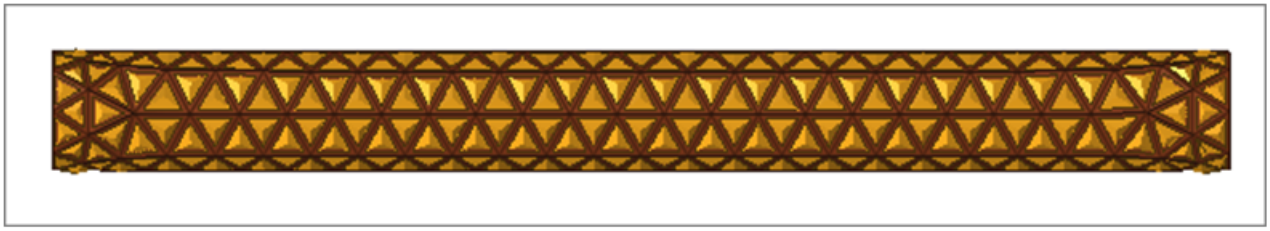


Figure 41:

25. Optional: Reject the recently added faces by clicking **Reject**. You can also delete faces from the **Delete** page.
26. When satisfied with the surface definition, click **Close** to return to the Abaqus Contact Manager.

1.3.6 Define Surfaces for Shell Elements

In this step you will create surfaces by defining individual element IDs and corresponding normals to define the SPOS/SNEG faces.

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual shell element IDs or sets with corresponding SPOS/SNEG face identifiers.

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to turn off the display of all surfaces.
3. Click **New**.
   The **Create New Surface** dialog opens.
4. In the **Name** field, type `cylinder-bot`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**.

The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.

8. Click the **Define** tab.
9. Set **Define surface for** to **3D shell, membrane, rigid**.
10. In the Model Browser, **Component** folder, right-click on **BOT_CYLINDER** and select **Isolate** from the context menu.
11. In the **Element Based Surface** dialog, click **Elements**.
12. In the panel area, click **elems >> by collector**.
13. Select the component, **BOT_CYLINDER**.
14. Click **select**.
    The elements in **BOT_CYLINDER** highlight.
15. Click **proceed**.
16. The normals of the selected elements will be displayed at this point. If the normals are too big, click (**YZ Front Plane View**) on the **Standard Views** toolbar.
17. Check the **Reverse** option.
18. Click **Add** to add these faces to the current surface.
    HyperMesh creates special face elements (rectangles with a dot at the middle) for display.

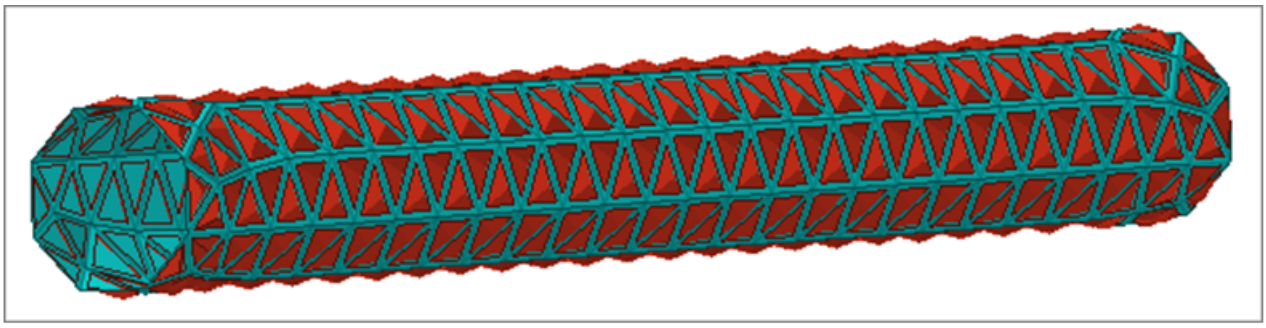


Figure 42:

19. Optional: Reject the recently added faces by clicking **Reject**. You can also delete faces from the **Delete** page.
20. Click the **Adjust Normal** tab.
21. Click **Display normals**. Notice that all normals are pointing outwards.

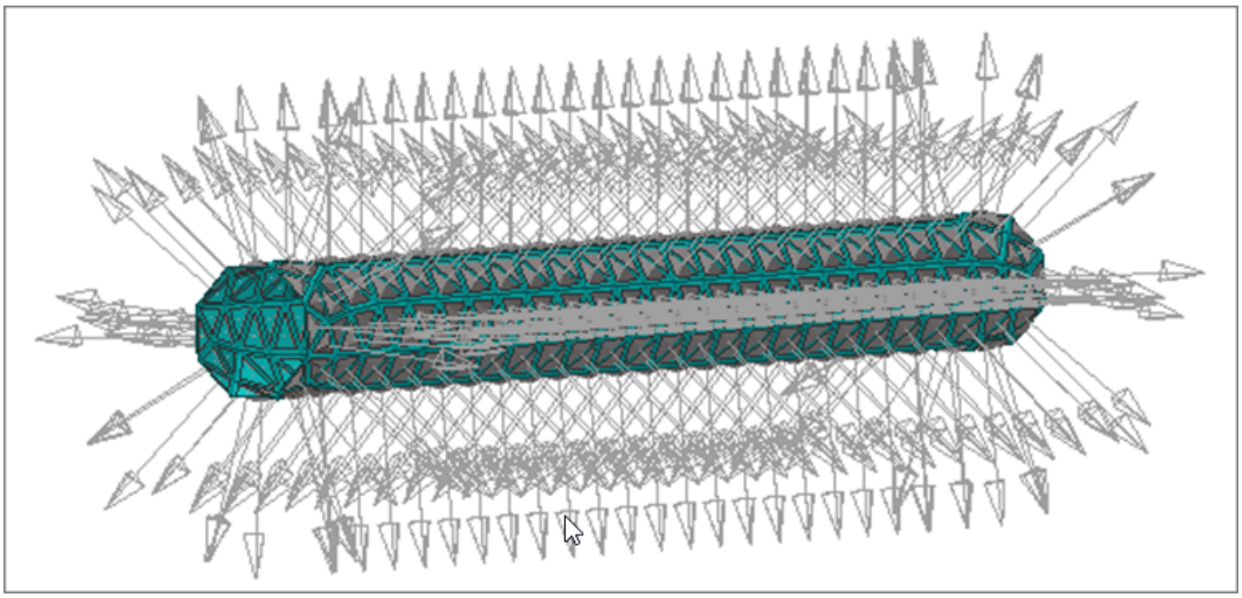


Figure 43:

22. When satisfied with the surface definition, click **Close** to return to the Abaqus Contact Manager.

1.3.7 Create the Box1-Bot Surface

In this step you will create surfaces by defining a set and corresponding face identifiers.

In HyperMesh, you can define the *SURFACE, TYPE=ELEMENT card by using individual element IDs or sets with corresponding face identifiers. HyperMesh allows only one set in a surface. It also does not support a combination of sets and individual elements in the same *SURFACE data line.

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to turn off the display of all surfaces.
3. Click **New**.
   The **Create New Surface** dialog opens.
4. In the **Name** field, type `box1-bot`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**.
   The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. Click the **Define** tab.
9. Set **Define surface for** to **Element set**.
10. In the Model Browser, **Component** folder, right-click on **BOX_1** and select **Isolate** from the context menu.
11. On the **Standard Views** toolbar, click (**YX Bottom Plane View**).

12. In the **Element Based Surface** dialog, set **Element set** to **box1-bot**.
13. Click **Review Set** to highlight all elements in the selected set.
14. Right-click on **Review Set** to reset the highlighting.
15. Click **Show Faces**.
    HyperMesh creates a temporary skin of the selected element set.
16. Select an element from the bottom of the solid skin.
17. In the panel area, click **elems >> by face**.
    The faces on the bottom of the solid skin highlight.
18. Optional: Deselect any element by right-clicking, or add more if you like.
19. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.

    At this point, the face identifier tags (color coded) of the selected faces are displayed. In performance graphics, the solid mesh sometimes blocks these tags. You might have to rotate the model a little to make these tags visible or switch to Standard graphics.
20. Click the right arrow key to move the **box1-bot** set into the table.
21. In the table, click on the **Face** pull-down menu and select **S3**.
22. Select the **Display** checkbox and then click **Update**.
    HyperMesh adds the selected set and face identifier to the current surface. In addition it creates a special display for the surface.

    By default, HyperMesh does not create a display for surfaces defined with sets. However, if you select the **Display** checkbox before clicking **Update**, it will create a special display using contact surface elements. The special display does not have any link to the set in the HyperMesh database. Therefore, if you edit the set later on, the display will not reflect them automatically. In this case, you need to come to this page, select the **Display** checkbox and click **Update** again.

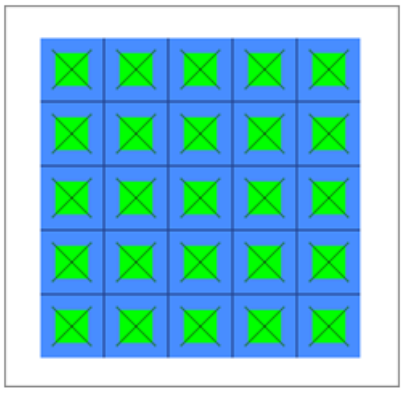


Figure 44:

23. Click **Close** to return to the Abaqus Contact Manager.

1.3.8 Create the Box2-Bot Surface

In this step you will create the box2-bot surface.

1. In the **Abaqus Contact Manager**, click the **Surface** tab.
2. Click **Display None** to turn off the display of all surfaces.
3. Click **New**.
   The **Create New Surface** dialog opens.
4. In the **Name** field, type `box2-bot`.
5. Set **Type** to **Element based**.
6. Click the **Color** button and select a color.
7. Click **Create**.
   The **Element Based Surface** dialog opens, from which you can define elements and corresponding faces for the surface.
8. Click the **Define** tab.
9. Set **Define surface for** to **Element set**.
10. In the Model Browser, **Component** folder, right-click on **BOX_2** and select **Isolate** from the context menu.
11. On the **Standard Views** toolbar, click (**YX Bottom Plane View**).
12. In the **Element Based Surface** dialog, click **Create/Edit Sets**.
13. In the panel area, type `box2-bot` in the **name** field.
14. Click **elems >> by collector**.
15. Select the component, **BOX_2**.
16. Click **select**.
17. Click **create**.

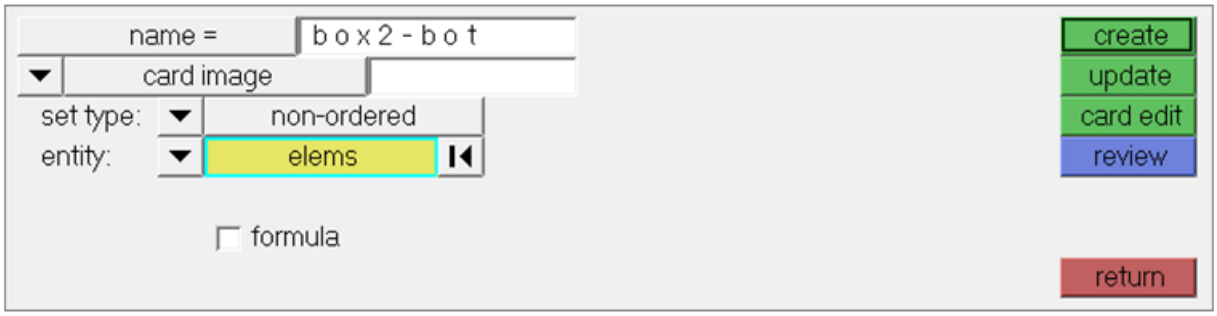


Figure 45:

18. When you are done creating/editing the set, click **return**.
19. Set **Element set** to **box2-bot**.
20. Click **Review Set** to highlight the elements in the selected set.
21. Right-click on **Review Set** to reset the highlighting.
22. Click **Show Faces**.
    HyperMesh creates a temporary skin of the selected element set.
23. Select an element from the bottom of the solid skin.

24. In the panel area, click **elems >> by face**.
    All faces on the bottom of the solid skin highlight.
25. Optional: Deselect any element by right-clicking or add more if you like.
26. When you are satisfied with the element faces selected, click **proceed** to return to the **Element Based Surface** dialog.

    At this point, the face identifier tags (color coded) of the selected faces display. In performance graphics, the solid mesh sometimes blocks these tags. You might have to rotate the model a little to make these tags visible or switch to standard graphics.
27. Click the right arrow to move the **box2-bot** set into the table.
28. In the table, click on the **Face** pull-down menu and select **S3**.

    > **Note:** Because all face identifier tags for the bottom side of the **box2-bot** set are **S3**, you can use the **S3** identifier for this set.

29. Clear the **Display** checkbox if it is selected, then click **Update** to add the selected set and face identifier to the current surface.

    > **Note:** By default, HyperMesh does not create any display for surfaces defined with sets.

30. Click **Close** to return to the Abaqus Contact Manager.

    > **Note:** In the **Surface** table the **Display** option for the **box2-bot** surface is disabled.

    At this point, you have created all required surfaces.
31. Click **Display All** to display all surfaces.
32. In the Model Browser, right-click on the **Components** folder and select **Show** from the context menu.
    All components display.
33. On the Standard View toolbar, click (**Isometric**).

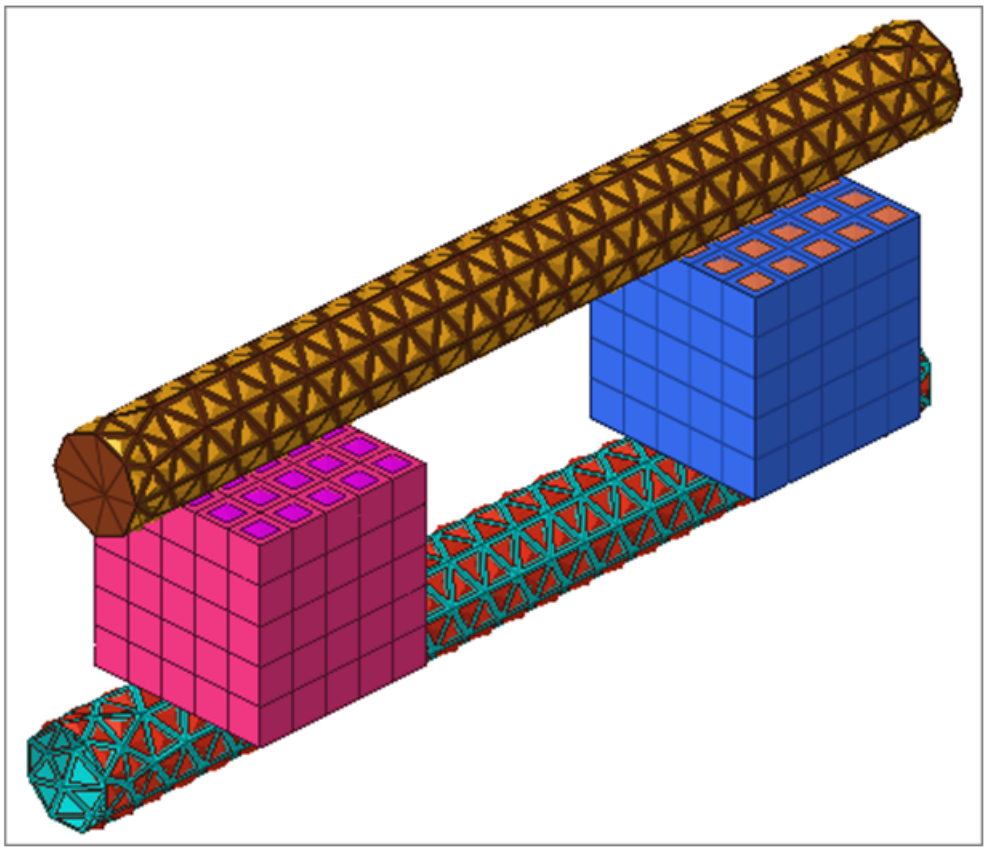


Figure 46:

34. Review a surface by selecting it from the table and clicking **Review**. Selected surfaces will be highlighted in red, while the rest of the model displays in gray. If the surface is defined with sets (display option disabled), the underlying elements are highlighted. Right-click on **Review** to clear the highlighting.

1.3.9 Define the Surface Interaction Property

In this step you will define the *SURFACE INTERACTION card with a corresponding *FRICTION card.

1. In the **Abaqus Contact Manager**, click the **Surface Interaction** tab.
2. Click **New**.
   The **Create New Surface Interaction** dialog opens.
3. In the Name field, type `friction1`.
4. Click **Create**.
   The **Surface Interaction** dialog opens.
5. Click the **Define** tab.
6. Under Select mechanical interaction properties, select **Friction**.
7. Click the **Friction** tab.
8. Set Friction type to **Default**.

9. In the second pane, select **Direct**.

> **Note:** Selecting this option means that the exponential decay and Anisotropic parameters will not be written to the input file.

10. In the No of data lines field, type `1`.
A single row appears in the Direct table.

11. In the Friction Coeff column, click the first cell and type `0.05`.

For Direct and Anisotropic tables:

- Change the number of columns in the table by specifying a value in the No of Dependencies field; change the number of rows in the table by specifying a value in the No of data lines field.
- Enter values in the table by clicking a cell to make it active and then typing in values. The table works like a regular spreadsheet.
- Read comma-delimited data from a text file by clicking **Read From a File**. In the File Browser, select a file and click **Open** to export the comma-delimited data. The row number will be set to the number of data lines found in the file.
- Access copy, cut, and paste options by right-click in the table. Comma-separated data can be copied/cut into or pasted from clipboard with these options. Relevant hot keys (for example, Ctrl-C, Ctrl-X and Ctrl-V in Windows) will also work.
- Activate cells by left-clicking in a cell. Clicking into an already active cell moves the insertion cursor to the character nearest the mouse.
- Highlight cells by left-clicking while moving the mouse over a cell.
- Move the active cell using the left, right, up, and down arrows.
- Extend the selection in a specific direction using Shift-<arrow>.
- Move the insertion cursor within a cell using Ctrl-left arrow and Ctrl –right arrow.
- Selects all cells using Ctrl -slash.
- Delete the character before the insertion cursor in the active cell using Backspace. If multiple cells are selected, Backspace deletes all selected cells.
- Remove the character after the insertion cursor in the active cell using Delete. If multiple cells are selected, Delete removes all selected cells.
- Move the insertion cursor to the beginning of the active cell using Ctrl-A. Move the insertion cursor to the end of the active cell using Ctrl-E.
- Decrease and increase the width of the column with the active cell in it using Ctrl-minus (-) and Ctrl-equal (=).
- Interactively resize a row or column by left-clicking or right-clicking on a border while moving the mouse.

12. Click **OK** to return to the Abaqus Contact Manager.

1.3.10 Define the Contact Pairs

In this step you will define the *CONTACT PAIR card with corresponding surfaces and surface interaction.

1. In the **Abaqus Contact Manager**, click the **Interface** tab.
2. Click **New**.
   The **Create New Interface** dialog opens.
3. In the **Name** field, type `top_cylinder_box1`.
4. Set **Type** to **Contact pair**.
5. Click **Create**.
   The **Contact Pair** dialog opens.
6. Click the **Define** tab.
7. Set **Surface** to **box1-top**.
8. Click **Slave >>** to identify box1-top as the secondary surface and move it into the table.
9. Click **Review**.
   The selected surface highlights red. If the surface is defined with sets (display option disabled), the underlying elements highlight. Right-click on **Review** to clear the highlighting.

   Clicking **New** opens the **Create New Surface** dialog, from which you can create a new surface. When you are done creating and defining the surface, the **Contact Pair** dialog returns with the new surface selected as the secondary surface.

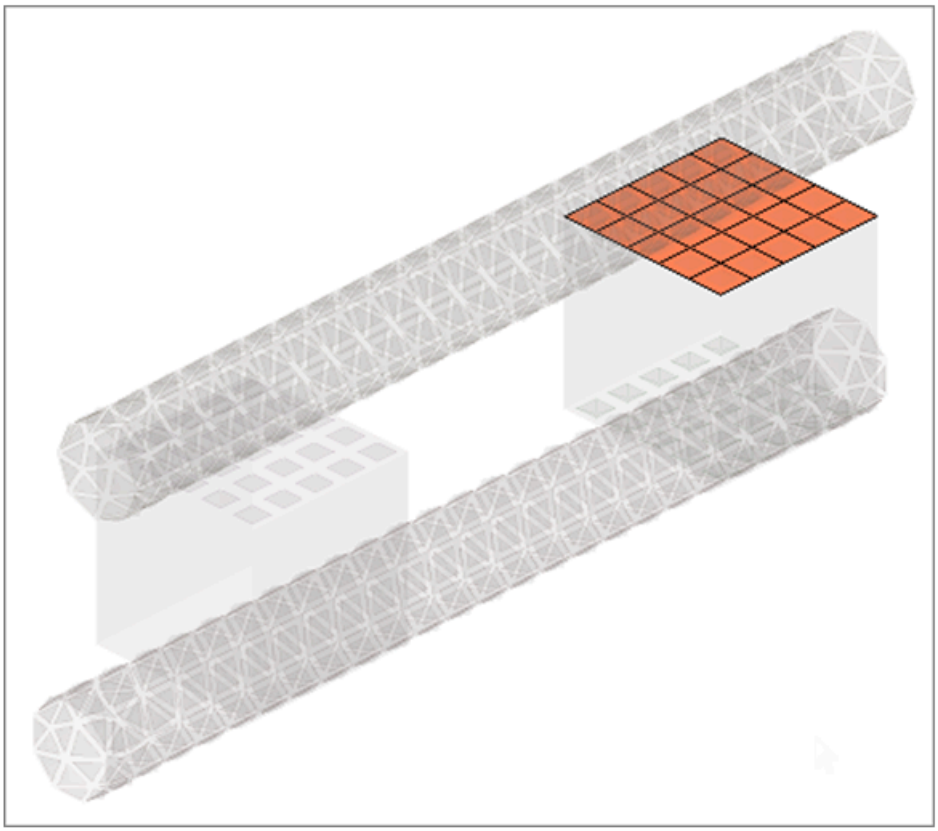


Figure 47:

10. Repeat Steps 10.7 and 10.8, selecting **cylinder-top** and clicking **Master>>** to identify it as the master surface.

> **Note:** To more clearly see the surfaces available for selection, click . This opens an **enhanced** browser where you can easily search for the appropriate item. You can also click **Filter** to filter the items displayed.

11. Set **Interaction** to **friction1**.

> **Note:** To more clearly see the interactions available for selection, click . This opens an **enhanced** browser where you can easily search for the appropriate item. You can also click **Filter** to filter the items displayed.

12. Click the **Parameter** tab.
13. Select the **Small sliding** checkbox.
14. Click **OK** to return to the Abaqus Contact Manager.

1.3.11 Create the Top-Cylinder-Box2 Contact Pair

In this step you will create the top-cylinder-box2 contact pair.

Follow steps 10.1 through 10.14 to define the **top-cylinder-box2** contact pair **withbox2-top** as secondary surface, **cylinder-top** as main and **friction1** as the surface interaction.

1.3.12 Create the Bot-Cylinder-Box1 Contact Pair

In this step you will create the bot-cylinder-box1 contact pair.

Follow steps 10.1 through 10.14 to define the **bot-cylinder-box1** contact pair **withbox1-bot** as secondary surface, **cylinder-bot** as main and **friction1** as the surface interaction.

1.3.13 Create the Bot-Cylinder-Box2 Contact Pair

In this step you will create the bot-cylinder-box2 contact pair.

Follow steps 10.1 through 10.14 to define the **bot-cylinder-box2** contact pair with **box2-bot** as secondary surface, **cylinder-bot** as main and **friction1** as the surface interaction.

At this point, you have created all of the contact pairs required. Review any contact pair by selecting it from the table and clicking **Review**. Both the main and secondary surface highlight in red while the rest of the model is grey. If a surface is defined with sets (display option disabled), the underlying elements highlight. Right-click on **Review** to clear the highlighting.

Click **Close** to close the **Abaqus Contact Manager**.

General comments:

- Click **Edit** to open the dialog for editing the selected interface, surface, or surface interaction.
- Click **Delete** to remove the selected interfaces, surfaces, or surface interactions. Multiple selections can be removed from the **Interface** table at once.
- Click **Sync** to update the **Contact Manager** with the current HyperMesh database. If you create, update, or delete any components, groups, properties, or entity sets from HyperMesh panels while the **Contact Manager** is open, click **Sync** to update the **Contact Manager**.
- If you minimize the **Contact Manager** dialog or if it goes behind HyperMesh, click **Tools** > **Contact Manager** to restore it.
- Bubble help exists for important buttons. Place the mouse on the buttons for a few moments to view it.
- Double-click on interface, surface, and surface interaction names in the table to open the corresponding edit dialog. Right-click on these names to display a pull down menu with options.
- Left-click or right-click on a table border while moving the mouse can resize columns in a table.
- Shift and Ctrl keys can be used while left-clicking to select multiple items in a table (useful for deleting multiple items).

1.4 HM-4330: Define *STEP using Abaqus Step Manager

In this tutorial you will define the *STEP using the Abaqus Step Manager.

You will learn how to:

- Load the Abaqus user profile
- Retrieve the HyperMesh model file
- Define the *STEP card and specify *STATIC as an analysis procedure
- Define loads (*CLOAD) and boundary conditions (*BOUNDARY)
- Define pressure loads (*DLOAD) with an element set
- Define output requests
- Export the database to an Abaqus input file

This exercise uses the `abaqus_StepManager_tutorial.hm` file, which can be found in `<hm.zip>/interfaces/abaqus/`. Copy the file(s) from this directory to your working directory.

1.4.1 Load the User Profile and Model

In this step you will load the Abaqus user profile and model file.

A set of standard user profiles is included in the HyperMesh installation. User profiles change the appearance of a panel, however they do not affect the internal behavior of each function.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Abaqus, Standard 3D.**
3. Open a model file by clicking **File** > **Open** > **Model** from the menu bar, or by clicking  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `abaqus_StepManager_tutorial.hm` file.

> **Note:** The `abaqus_StepManager_tutorial.hm` file contains pre-defined model data. Use this file in the following steps to define the history data portion of this model.

1.4.2 Define a *STEP Card

In this step you will create a *STEP card with the *STATIC analysis procedure.

1. From the menu bar, click **Tools** > **Loadsteps Browser**.
   The **Step Manager** opens.
2. Click **New**.
   The **Create New Step** dialog opens.
3. In the **Name** field, type `step1`.

4. Click **Create**.
   A step labeled **step1** opens the **Load Step** dialog.
5. In the first pane, select **Title**.
   The Step Heading option with a disabled field is displayed.



Figure 48:

6. Select the **Step heading** checkbox and type `100kN` in the text field.



Figure 49:

7. Click **Update** to store the heading information in **step1**.
8. In the first pane, select **Parameter**.
9. Select the **Name** and **Perturbation** checkboxes.

Note: Notice that **Name** is already set to **step1**.



Figure 50:

10. Click **Update**.
11. In the first pane, select **Analysis procedure**.
12. Set **Analysis type** to **static**.

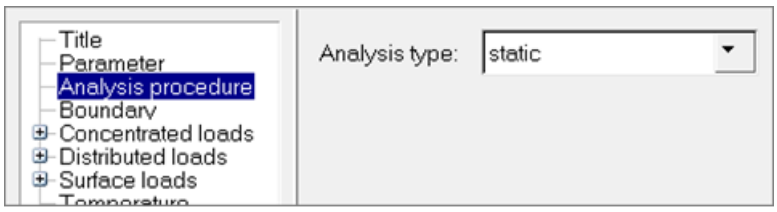


Figure 51:

13. Click **Update**.
14. Click the **Dataline** tab.
15. Select the **Optional dataline** checkbox to add an additional dataline.
16. Add individual data, such as Initial increment, by selecting the appropriate checkbox and entering a value.

Note: When a checkbox is disabled, a space will be added in the ASCII file, and the Abaqus solver will use the default value.



Figure 52:

17. Click **Update**.

1.4.3 Define Load and Boundary Conditions

In this step you will create constraints.

You will add the *CLOAD and *BOUNDARY keywords to the current load collector by defining loads and boundary conditions.

1. In the first pane, select **Boundary**.
2. Click **New**.
   The **Create Load Collector** dialog opens.
3. In the **Name** field, type `loads_and_constraints`.
4. Click **Create**.

5. Optional: In the **Load collector** table, **Display** column, click the color icon to select a color for the load collector.
6. Verify that the **Status** checkbox for **loads_and_constraints** is selected.

   Note: By selecting this checkbox, you are adding this load collector into the loadstep.

7. Click the **loads_and_constraints** load collector.
   New tabs display.
8. From the **Define** tab, verify that **Type** is set to **default(disp)**.

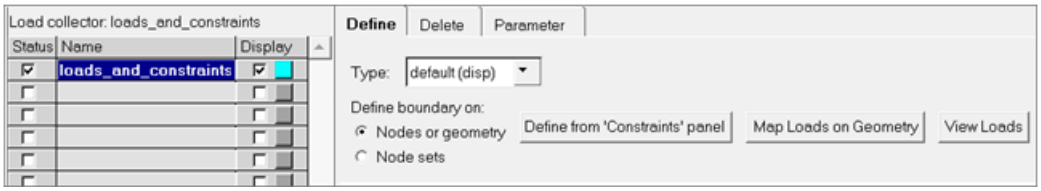


Figure 53:

9. Click **Define from 'Constraints' panel**.
   The **Constraints** panel opens, from which you will create constraints.

1.4.4 Create Constraints from the Constraints Panel

In this step you will create constraints from the Constraints panel.

1. On the Standard Views toolbar, click (**XZ Right Plane View**).
2. In the Constraints panel, click **nodes >> by window**.
3. With the exception of nodes at the ends of the cradle, draw a rectangle around all of the displayed nodes.

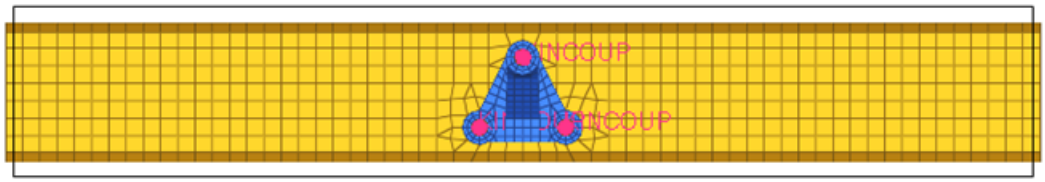


Figure 54:

4. Select the **exterior** checkbox.
5. Click **select entities**.
   HyperMesh selects all nodes outside the window your drew.

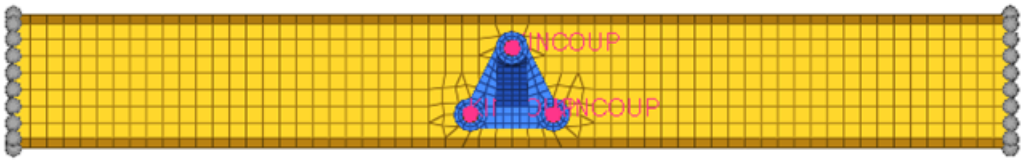


Figure 55:

6. Verify that all six dof checkboxes are selected.

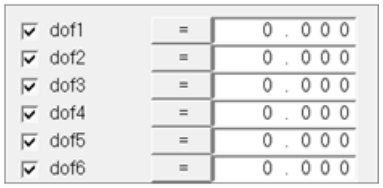


Figure 56:

7. Click **create**.
   HyperMesh creates constraints at the selected nodes.

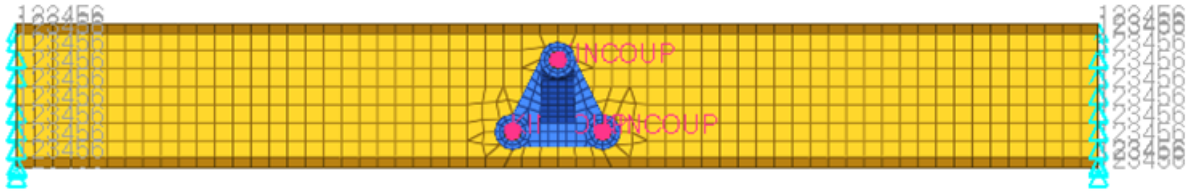


Figure 57:

8. Click **return** to go back to the **Load Step** dialog.
9. At the bottom of the **Load Step** dialog, look at the Load type line.
   Bc (short for BOUNDARY) appears on this line, which indicates step1 is a load type created in the load_and_constraints load collector. The corresponding load type in the first pane is also highlighted.

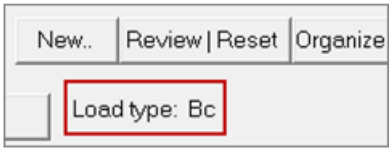


Figure 58:

1.4.5 Create Forces

In this step you will create forces (*CLOAD).

1. In the first pane of the **Load Step** dialog, expand **Concentrated loads**, and select CLOAD-FORCE. New tabs display.

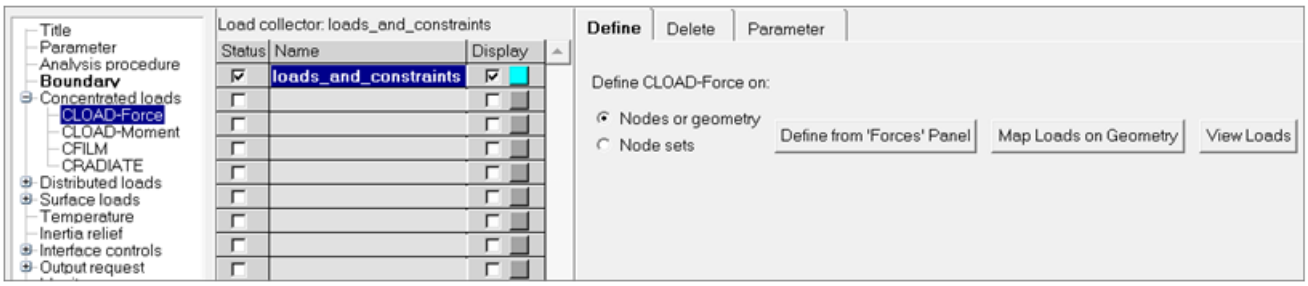


Figure 59:

2. From the Define tab, click **Define from 'Forces' Panel**.
The Forces panel opens from which you will create forces.

1.4.6 Create Forces from the Forces Panel

In this step you will create forces from the Forces panel.

1. Select the central node on the top side of the bracket arm.

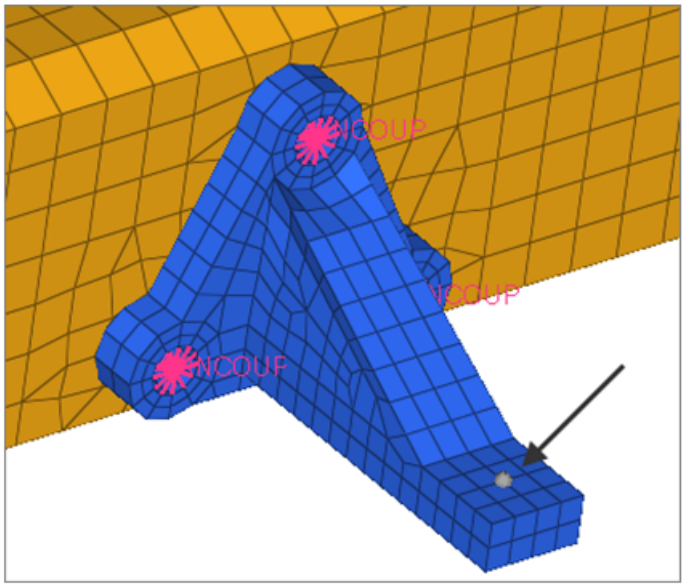


Figure 60:

2. In the **Forces** panel, **magnitude=** field, type `-100`.
3. Set the orientation selector to **z-axis**.

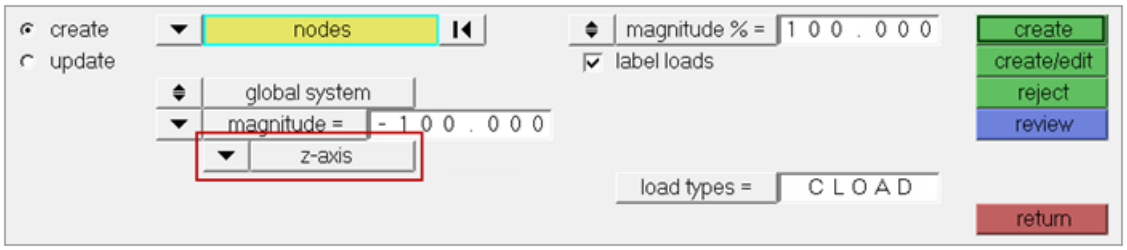


Figure 61:

4. Click **create**.

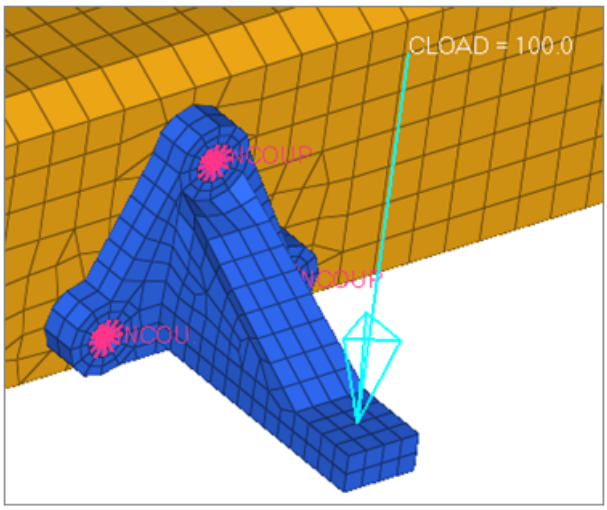


Figure 62:

5. Click **return** to go back to the **Load Step** dialog.
6. At the bottom of the **Load Step** dialog, notice the **Load type** now reads **Cload-f**, which indicates CLOAD-force as another load type created in the loads_and_constraints load collector. The corresponding load type in the first pane is highlighted.

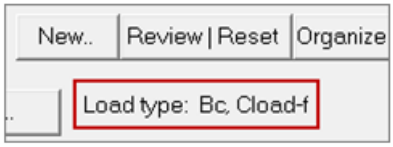


Figure 63:

7. Click **Review/Reset**.
   The constraints and forces that belong to the loads_and_constraints load collector highlight.

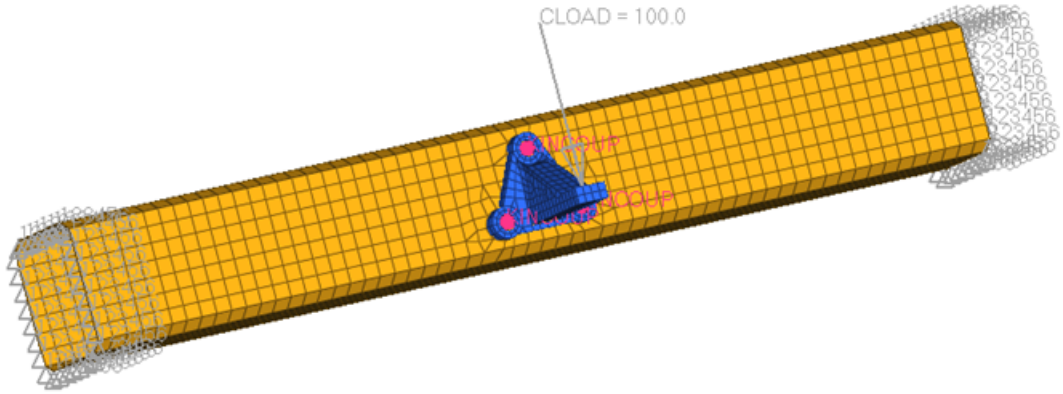


Figure 64:

8. Revert the highlighted constraints and forces to the load collector color by right-clicking on **Review**.

1.4.7 Define Pressure Loads

In this step you will create a *DLOAD pressure load and add it to the current load collector with an element set.

1. In the first pane of the **Load Step** dialog, expand **Distributed loads**, and select **DLOAD**.
   A new set of tabs displays.
2. From the **Define** tab, set **Define DLOAD on** to **Element sets**.
   The elements sets table displays.
3. Rotate the model to the view as shown in the following image.

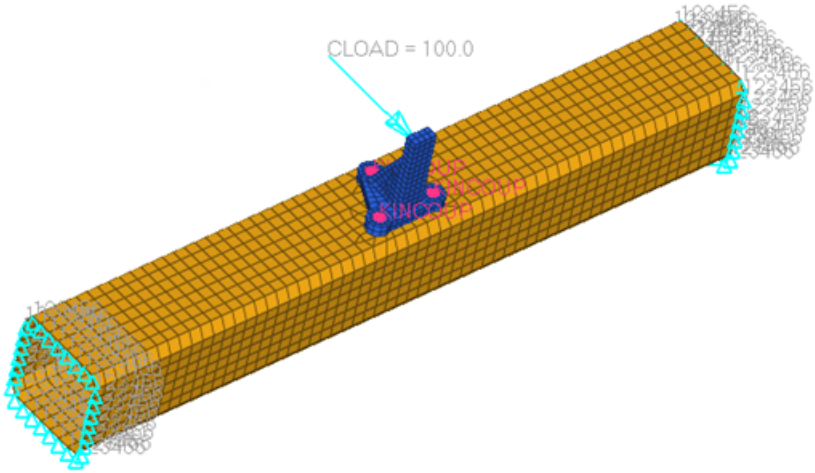


Figure 65:

4. From the **Load Step** dialog, set **Type** to **default (Pressure)**.
5. Set **Element sets** to **pressure_set**.

Note: To view the entire list of element sets, click . Use Fileter and Sort to narrow your search.

6. Click the right arrow to add the selected set to the element sets table.

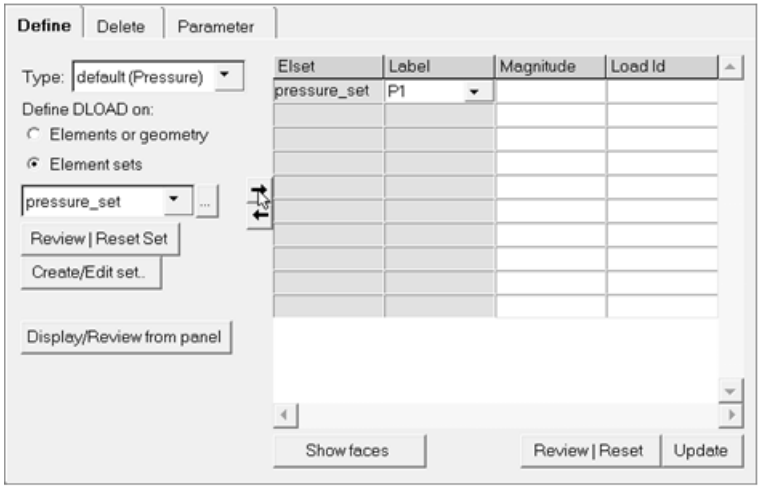


Figure 66:

7. Under **Element sets**, click **Review | Reset Set**. The element set highlights.

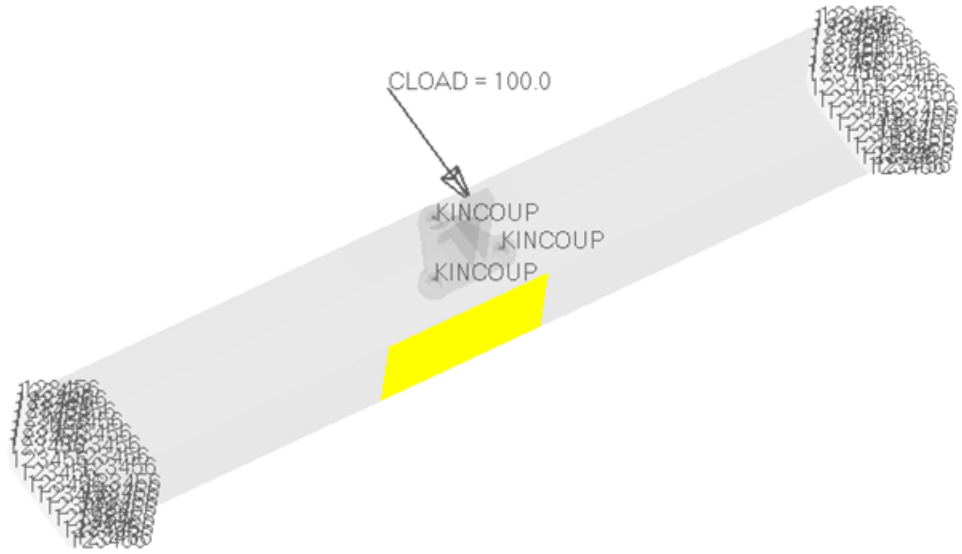


Figure 67:

8. Revert the load collector back to its original color by right-clicking on **Review | Reset Set**.
9. In the element sets table, **Label** column, select **P** for the newly added **pressure_set**.

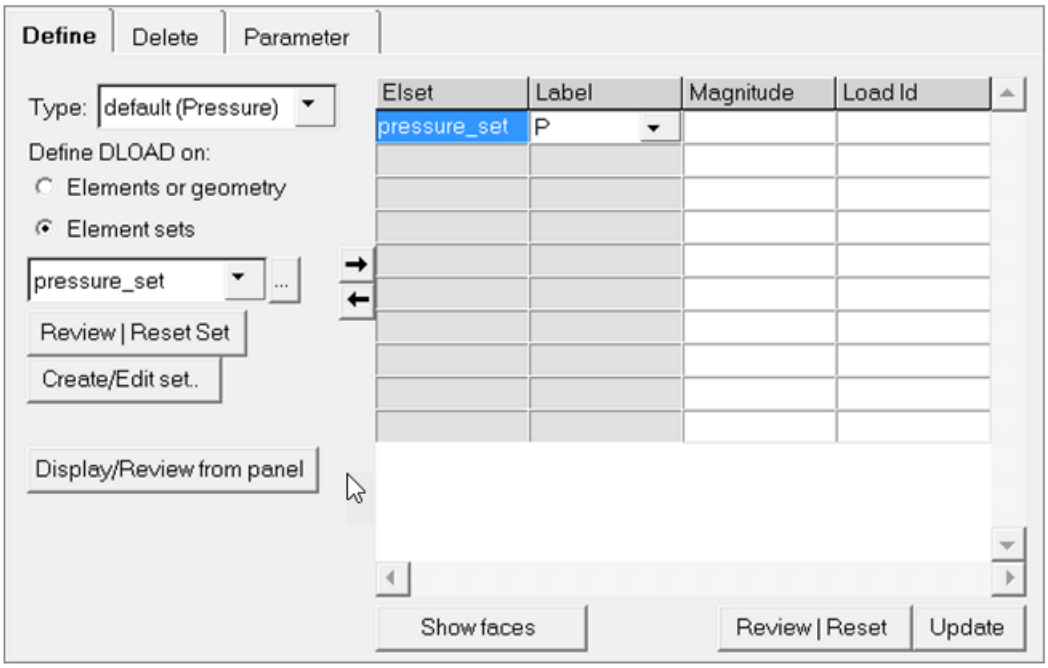


Figure 68:

10. Because the **pressure_set** contains shell elements, the direction of normal to the elements must be known to determine the sign of the magnitude. Find the direction of the normal by selecting the **pressure_set** element from the table and clicking **Show faces**.

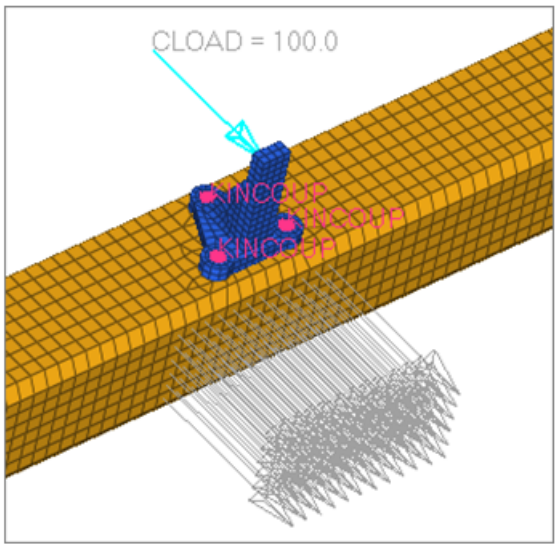


Figure 69:

11. Clear the display by right-clicking on **Show faces**.

12. In the element sets table, **Magnitude** column, type `-10` for **pressure_set**.

> **Note:** The negative magnitude means pressure load in the opposite direction of the underlying shell element normals.

13. Click **Update**.
    The HyperMesh database updates. The **Load type** line, at the bottom of the dialog, now displays Dload, which indicates DLOAD as another load type created in the loads_and_constraints load collector. The corresponding load type is the first pane is also highlighted.

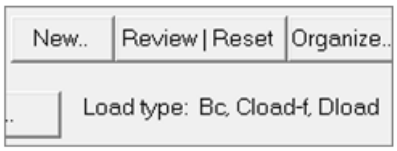


Figure 70:

14. In the element sets table, **Elset** column, click **pressure_set**.
15. Click **Review | Reset Set** to review the loads.

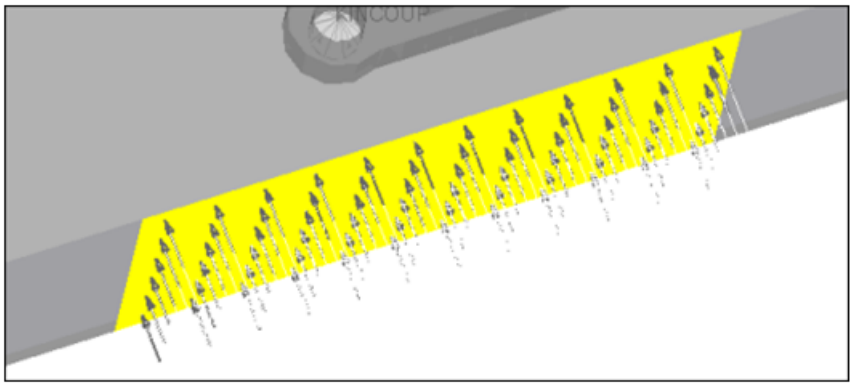


Figure 71:

16. Revert back to the standard view by right-clicking on **Review | Reset Set**.

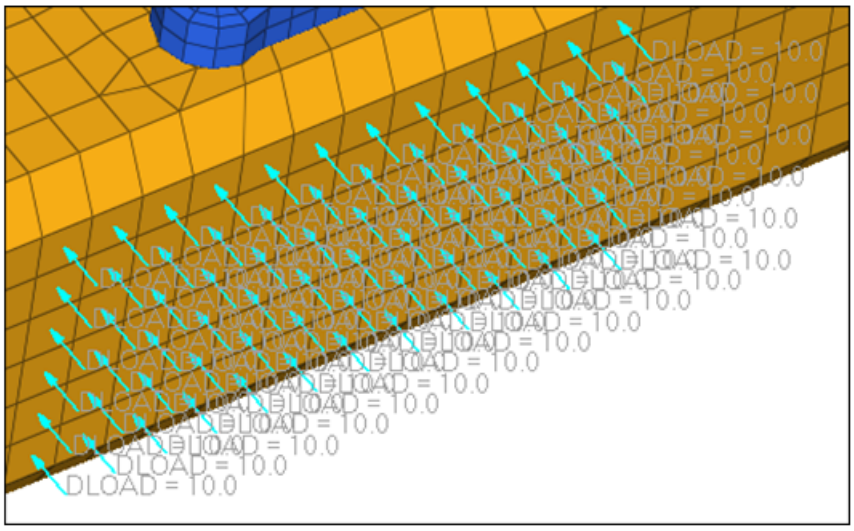


Figure 72:

In this exercise, you constrained and applied distributed loads on the model using HyperMesh panels. The loads (*DLOAD) information is automatically stored in **step1**. Next, you will specify the output requests for this step.

1.4.8 Request ODB File Outputs

In this step you will specify several output requests for **step1**.

1. In the first pane of the **Load Step** dialog, expand **Output request** and click **ODB file**.

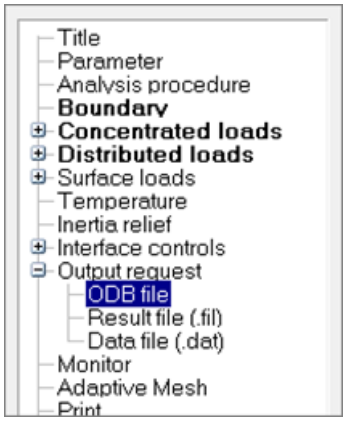


Figure 73:

2. Click **New**.
3. In the **Create Output block** dialog, **Name** field, type `step1`.
4. Click **Create**.
5. In the **Output block** table, click **step1**.
   New tabs appear.

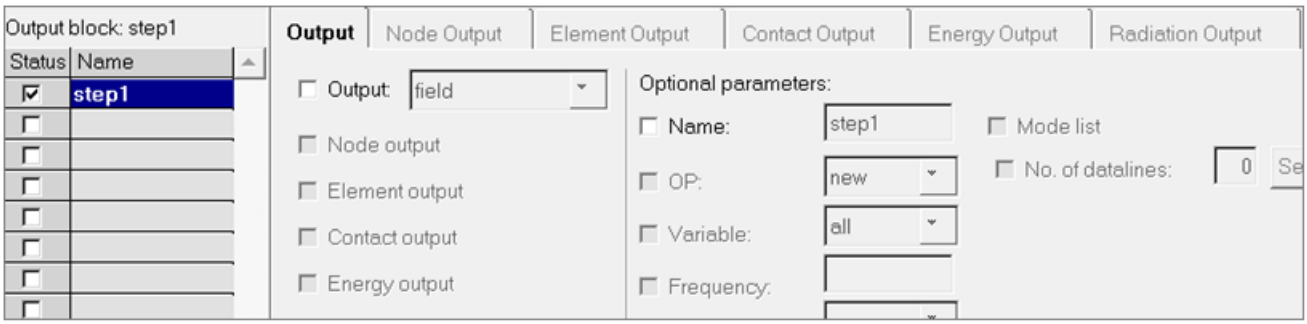


Figure 74:

6. In the **Output** tab, select the **Output** checkbox. Leave **Output** set to **field**.
7. Select the **Node output** and **Element output** checkboxes.
   the **Node Output** and **Element Output** tabs become active.

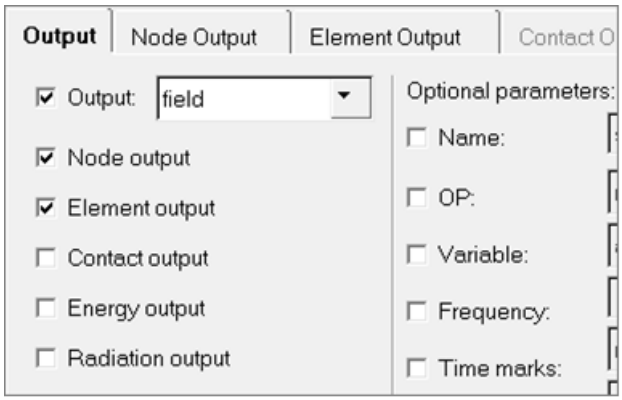


Figure 75:

8. Click the **Node Output** tab.
9. Expand **Displacement** and select **U**.
   The **Data lines** table now displays "U", which allows you to request displacement results `obd` in the file.

Note: You can manually type output request into the **Data lines** table, including unsupported requests. They will be written just as they are entered in the table.

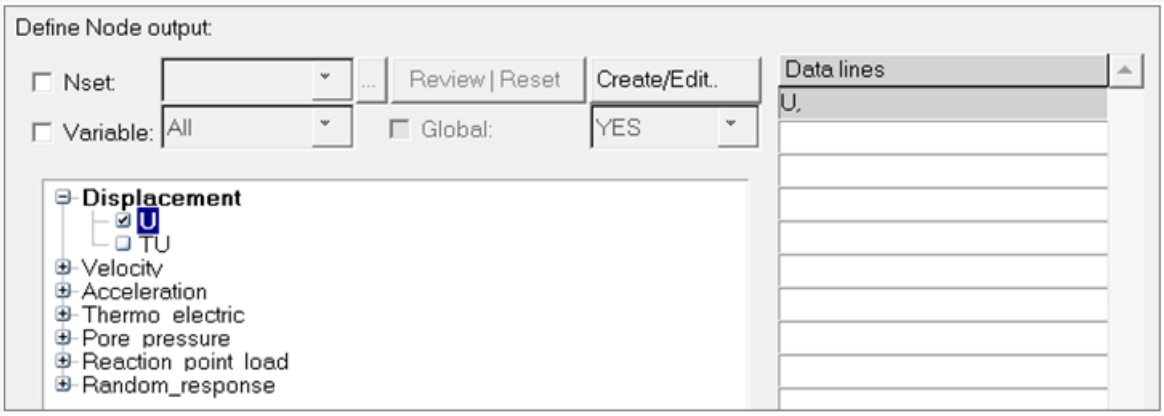


Figure 76:

10. Click **Update**.
11. Click the **Element Output** tab.
12. Select the **Position** checkbox and set it to **Nodes**.
13. Expand **Stress** and select **S**.
    The **Data lines** table now displays "S" which allows you to request stress results in the `.obd` file.

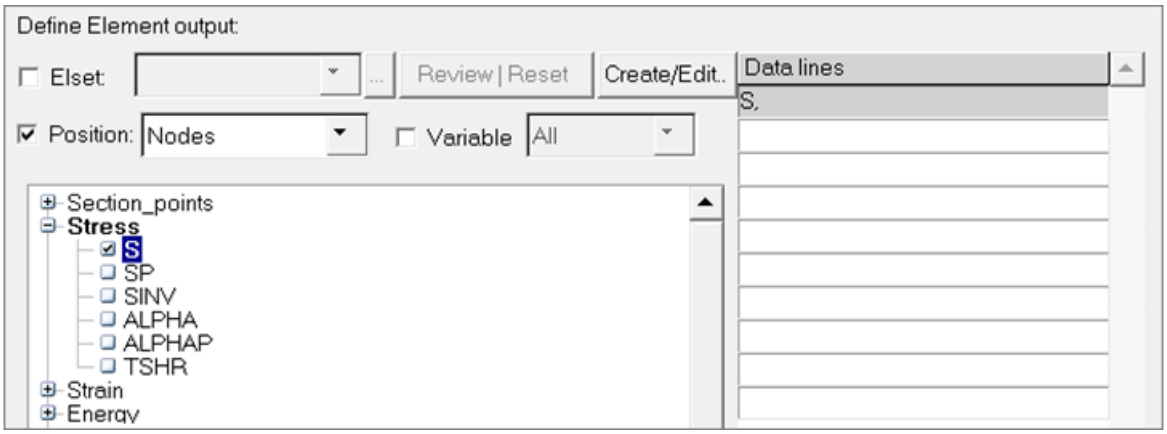


Figure 77:

14. Click **Update**.

1.4.9 Request Result File Outputs

In this step you will request results file (`.fil`) outputs.

1. In the first pane of the **Load Step** dialog, expand **Output request** and click **Result file (.fil)**.

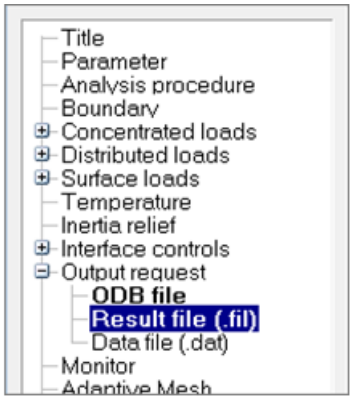


Figure 78:

2. In the Define tab, select the **Node file** and **Element file** checkboxes.
   The Node File and Element File tabs become active.

   Figure 79:



3. Click the **Node File** tab.
4. Expand **Displacement** and select "**U**."
   The Data lines table displays "U," which allows you to request displacement result in the `.fil` file.
5. Click **Update**.
6. Click the **Element File** tab.
7. Select the **Position** checkbox and set it to **averaged at nodes**.
8. Expand **Stress** and select **S**.
   The Data Line table displays "S," which allows you to request str4ess results in the `.fil` file.



Figure 80:

9. Click **Update**.
10. Under the Output block table, click **Review | Reset**.
    The **Review output block** dialog opens and displays the output requests you made.

Note: This is the format used in the Abaqus input file (`.inp`).

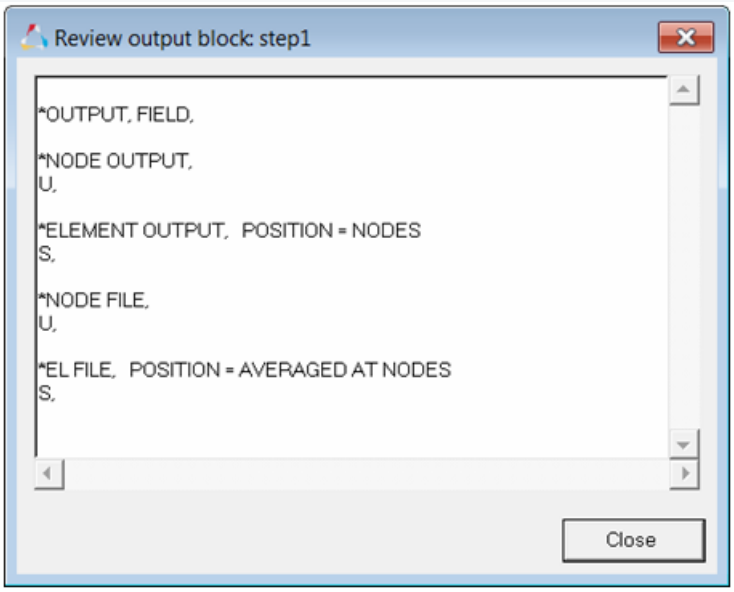


Figure 81:

11. Click **Close** to exit the **Review output block** dialog.
12. In the first pane of the **Load Step** dialog, click **Unsupported cards**.
13. Optional: Select the **Unsupported cards** checkbox to add any unsupported card.
14. Click **Close** to exit the **Load Step** dialog and return to the Step Manager.
    The **Step Manager** dialog displays all information you defined for **step1**.

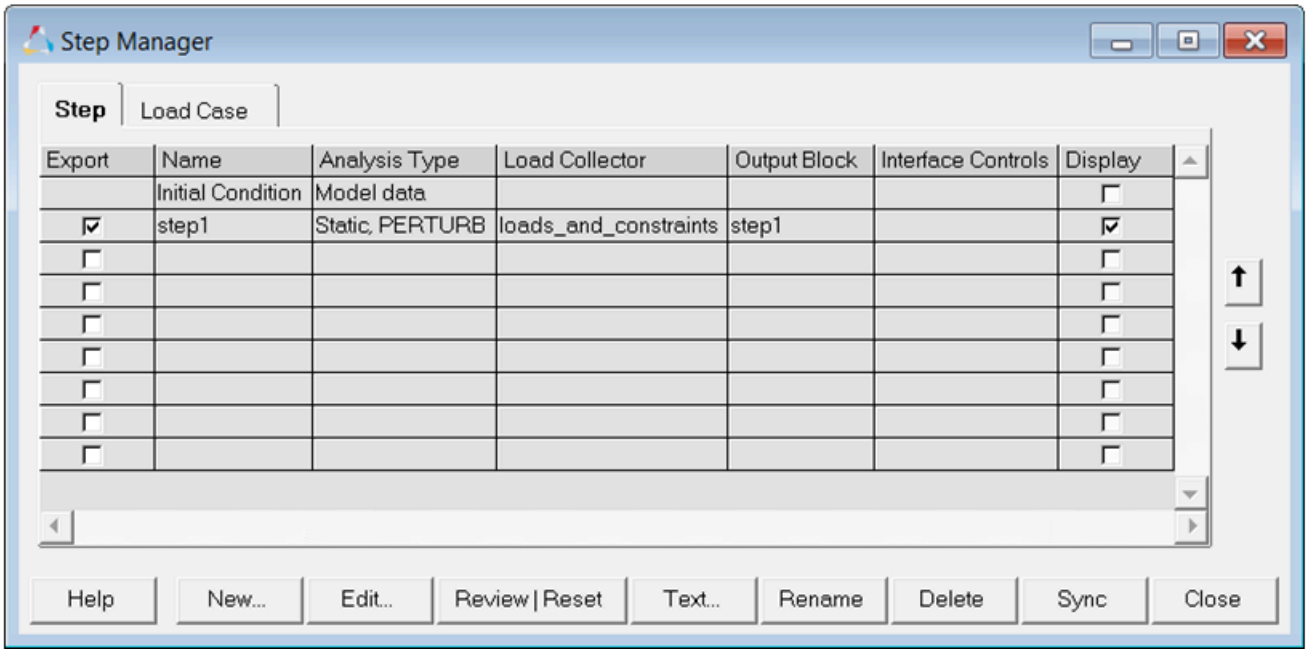


Figure 82:

15. Click **Close** to exit the **Step Manager** dialog.

1.4.10 Export the `.inp` File

In this step you will export the `.inp` file.

The data currently stored in the database must be output to an Abaqus `.inp` file for use with the Abaqus solver. The `.inp` file can then be used to perform the analysis using Abaqus outside of HyperMesh.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
2. In the File: field, type `job1.inp`.
3. To the left of Export Options, click .
4. Set **Export** to **all**.
5. Click **Export**.

1.4.11 Save the `.hm` File

In this step you will save the `.hm` file.

1. From the menu bar, click **File** > **Save as** > **Model**.
2. In the **Save Model As** dialog, type `job1.hm` as the file name.
3. Click **Save**.

   Notes:

- After you quit HyperMesh, you can run the Abaqus solver using the `job1.inp` file that was written from HyperMesh.
- At your site, you can use the Abaqus license to run this model.
- If the batch mode option is being used, then enter the name of the `.inp` file exported in the previous step as the input file.
- After you have successfully completed the analysis, the result file will be available in your working directory with the name <jobname.odb>.
- Use HvTrans to translate the Abaqus solver result file to an H3D file.

1.4.12 Open HyperView from the Application Menu

In this step you will open HyperView from the application menu.

1. On the Client Selector toolbar, select HyperView. HyperView opens.

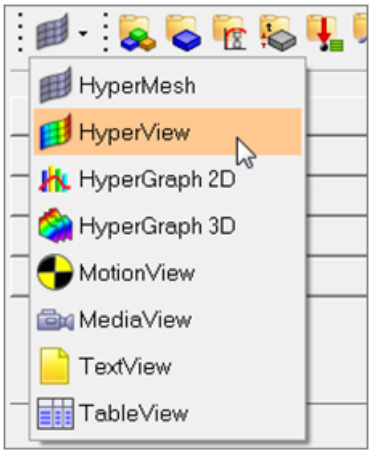


Figure 83:

2. In the panel area, load the model and results files.

Note: Load `*.h3d` files for both the model and result files.

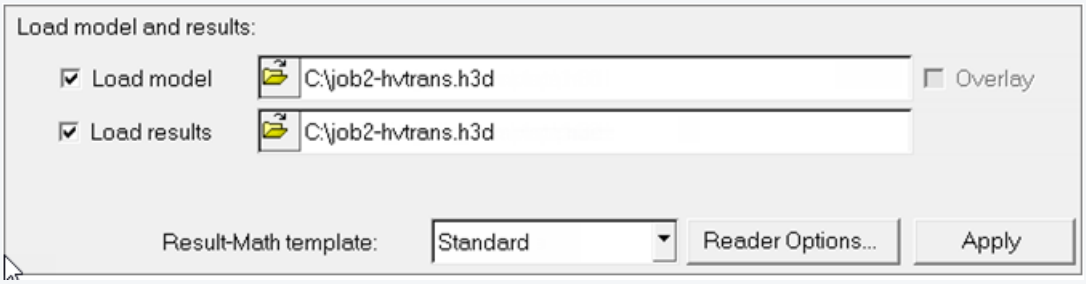


Figure 84:

3. Click **Apply**.

4. On the Results toolbar, click to open the Contour panel.
5. Review displacement (v) results by setting the **Result type** to **Displacement (v)**.

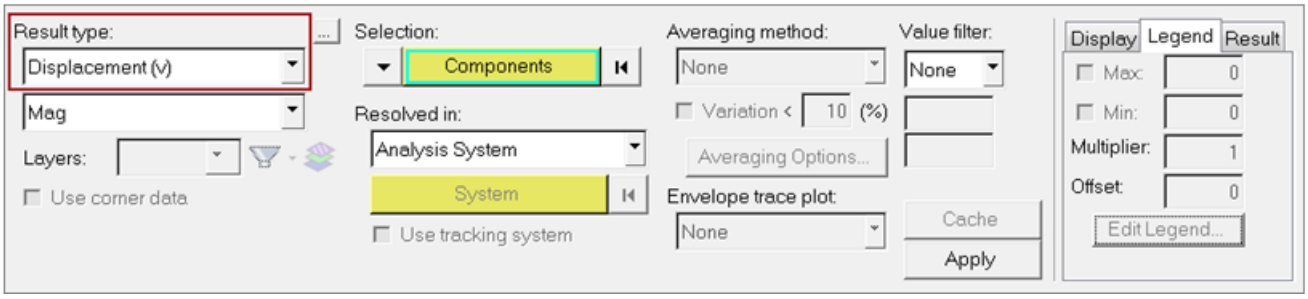


Figure 85:

6. Click **Apply**.

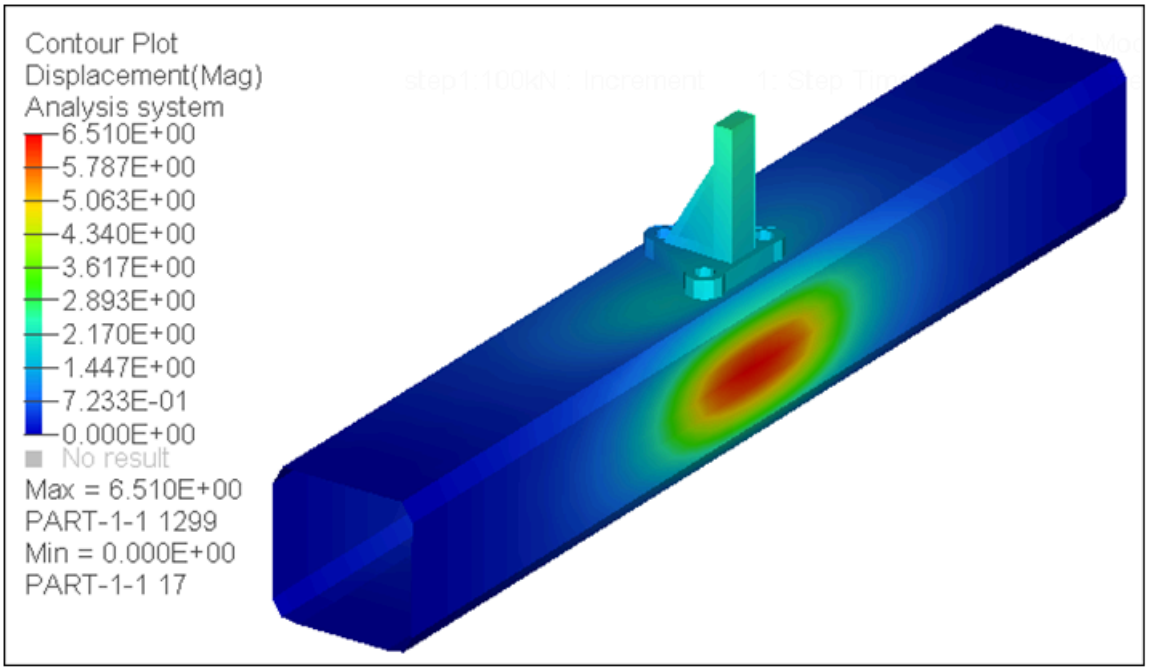


Figure 86:

7. In the Results Browser, review steps and increments.

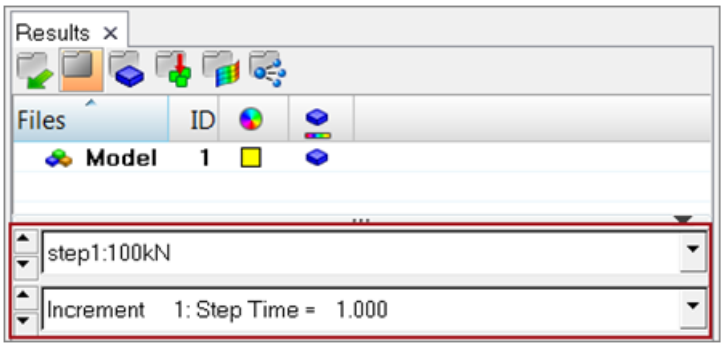


Figure 87:

8. On the Animation toolbar, set the animation mode to **linear**.

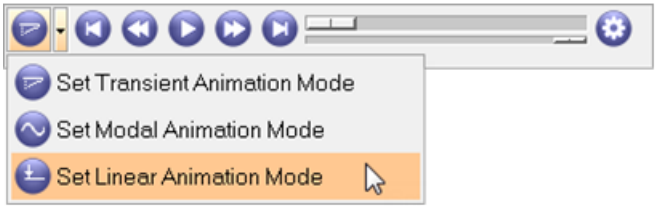


Figure 88:

9. Review the animation by clicking .

10. Review UR-Rotational displacement (v) results by setting the Result type to **UR-Rotational displacement (v)** in the Contour panel.

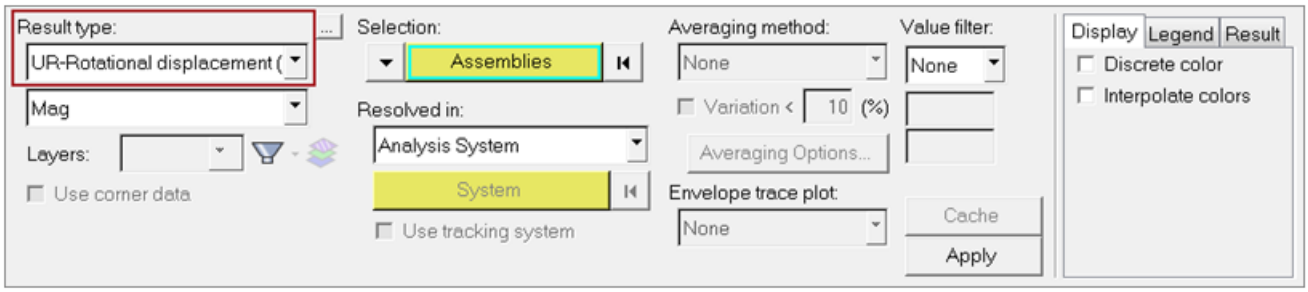


Figure 89:

11. Click **Apply**.

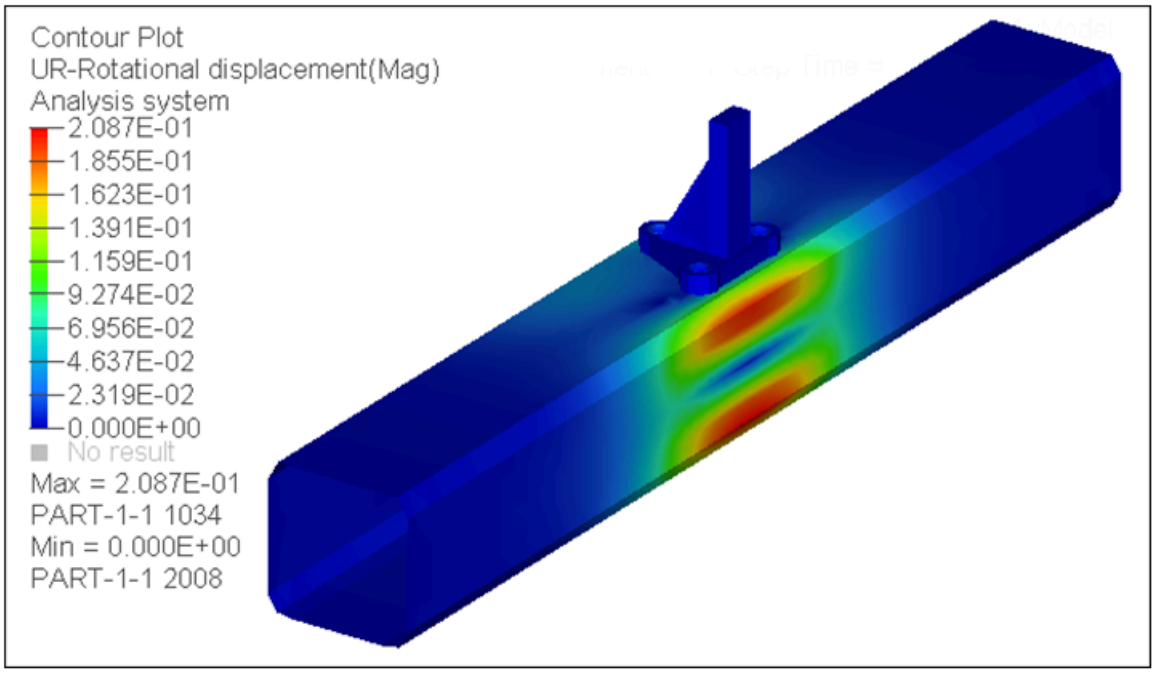

Contour Plot
UR-Rotational displacement(Mag)
Analysis system
2.087E-01
1.855E-01
1.623E-01
1.391E-01
1.159E-01
9.274E-02
6.956E-02
4.637E-02
2.319E-02
0.000E+00
No result
Max = 2.087E-01
PART-1-1 1034
Min = 0.000E+00
PART-1-1 2008

Figure 90:

1.5 HM-4340: Pre-Processing for Bracket and Cradle Analysis

In this tutorial, you will learn how to set up an Abaqus input file in HyperMesh for obtaining the linear static response of a cradle and bracket assembly subjected to a 100 kN load on the bracket, with the cradle's ends fully constrained.

You will learn how to:

- View images of keywords and data lines in HyperMesh as they appear in the Abaqus input file
- Create and edit Abaqus materials and section properties
- Select Abaqus entity types for HyperMesh element and load configurations
- Create loads and boundary conditions for model data (*KINEMATIC COUPLING and *BOUNDARY)
- Create an Abaqus step containing title, analysis procedure, parameters, *CLOAD on bracket, and output requests
- Export a model to an Abaqus formatted input file

Note: The units millimeters and kilonewtons (mm, kN) are used in this tutorial.

This exercise uses the `bracket_cradle.hm` file, which can be found in `<hm.zip>/interfaces/abaqus/`. Copy the file(s) from this directory to your working directory.

1.5.1 Load the User Profile and Model

In this step you will load the Abaqus user profile and model.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Abaqus, Standard3D**.
3. Open a model file by clicking **File** > **Open** > **Model** from the menu bar, or by clicking  on the **Standard** toolbar.
4. In the **Open Model** dialog, open the `bracket_cradle.hm` file.

   The `bracket_cradle.hm` file contains the following Abaqus model data:

   - ELSET bracket modeled with penta (C3D6) and hexa (C3D8) elements
   - ELSET cradle modeled with tria (S3) and quad (S4) elements
   - Two *KINEMATIC COUPLING entities at the bracket's bottom bolt holes
   - Material named aluminum
   - *SOLID SECTION property for ELSET bracket with the aluminum material associated to it

1.5.2 Understand the Relationship between Abaqus and HyperMesh Entities

You can use HyperMesh card images to view images of keywords and data lines for defined Abaqus entities as interpreted by the loaded template. The keywords and data lines appear in the Abaqus input file as you see them in the card images. Additionally, for some card images, you can define and edit various parameters and data items for the corresponding Abaqus keyword.

Review and edit card images by clicking on the **Collectors** toolbar. You can also review and edit the card image for many entities from the panel in which they are created. Most of the card images are also accessible from the Model Browser by right-clicking on the entity and selecting **Card Edit** from the context menu, or simply left-clicking on the entity displays it in the Entity Editor.

*ELEMENT with Sectional Property

The Abaqus keyword *ELEMENT, TYPE = <type>, ELSET = <name>, is defined by Abaqus elements collected in a HyperMesh component collector. One *ELEMENT keyword is written to the Abaqus input file for each element type in the component. The name of the ELSET is the name of the component.

If a property is assigned to a component, the ELSET name of that sectional property will be the name of the component collector. If properties are directly assigned to elements, HyperMesh will write an additional ELSET with the name of the property collector it is assigned to. The sectional property card will then point to this ELSET. Assignment of individual elements to a property takes precedence over assigning a component to a property. Individual element assignment is only recommended when the ratio of elements per component is very small. This is the situation explained in this tutorial.

The material referenced in the sectional property is defined by a HyperMesh material collector associated with the property collector. The diagram below shows how elements and its associated properties are organized in HyperMesh.

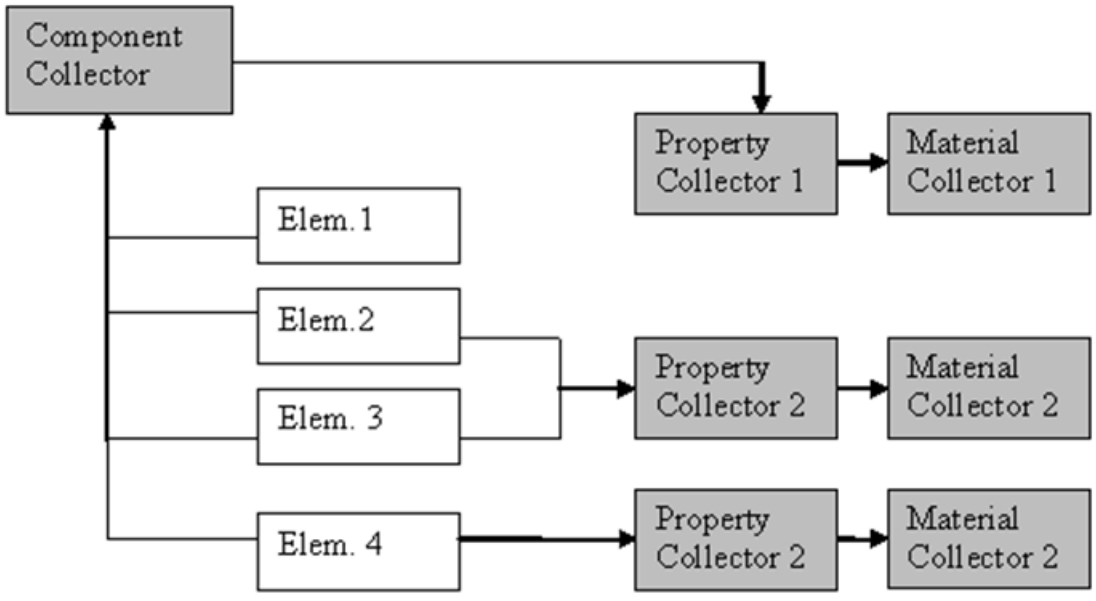


Figure 91:

Abaqus Materials in HyperMesh

An Abaqus *MATERIAL is a HyperMesh material collector with a card image.

There are four card images for the HyperMesh Abaqus templates: ABAQUS_MATERIAL, GASKET_MATERIAL (for the Standard templates), CONNECTOR_BEHAVIOR and GENERIC_MATERIAL. There are two ways to create a material collector and associate it to a component.

Method One: Drop-Down Menu

- From the menu bar, create a material collector with a card image and edit it to define material data. By selecting the material while creating a property, it will automatically be assigned to the sectional property. As pointed out in the paragraph above, you can assign the property containing the sectional properties to a component or to individual elements. Assign properties to existing components or to individual elements, from the menu bar, by clicking **Collectors** > **Assign** > **Component Properties** or clicking **Properties** > **Assign**.

Method Two: Model Browser

- In the Model Browser, while in either the model or material view, right-click and select **Create** to create a new material collector with the appropriate card image. If you select **Create/Edit**, the card image displays so you can set up all necessary parameters and keywords. Properties can also be created in this manner. You can directly assign the material to the new property by selecting the material in the Material name field.

Within the card image of each material card, it is possible to add as many data lines as you want for a material (such as *PLASTIC with yield stress, plastic strain, and temperature data). Manually enter the data in HyperMesh or import an Abaqus formatted input file on top of the model in HyperMesh to create the data lines.

1.5.3 Review the Card Image

In this step you will review the card image of one of the elements.

You will use the card editor panel to review elements, sectional properties, and materials as they appear in the Abaqus input file. Review and edit card images by clicking on an entity in the Model Browser and displaying it in the Entity Editor.

1. Open the Card Edit panel by clicking **Mesh** > **Card Edit** > **Elements** from the menu bar.
2. Verify that the entity selector is set to **elems**.
3. Select an element from the bracket (blue elements).
4. In the panel area, click **edit**.
   The card image for the element opens, and displays the element type (either C3D6 or C3D8) and the ELSET name as bracket.

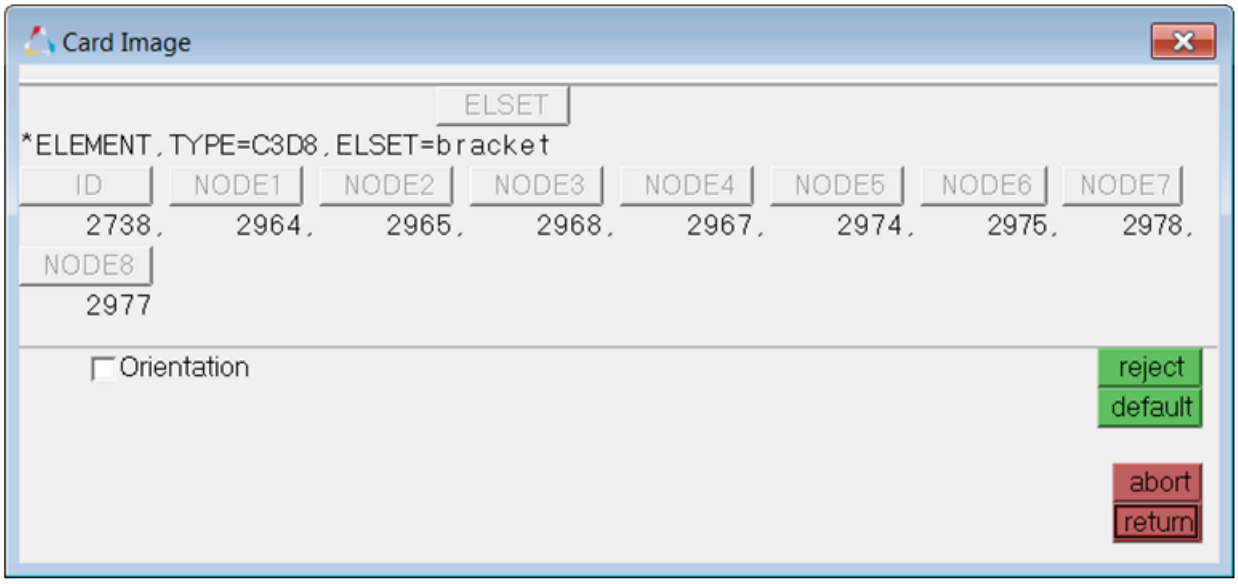


Figure 92:

5. Click **return** to close the card image.

1.5.4 View the Property Card

In this step you will use the Entity Editor to view the sectional properties associated with the elements in the model's components.

In the Model Browser, **Property** folder, click **bracket**.
The Entity Editor opens, and displays the bracket's ***SOLID SECTION** property and material assignment, which is aluminum.

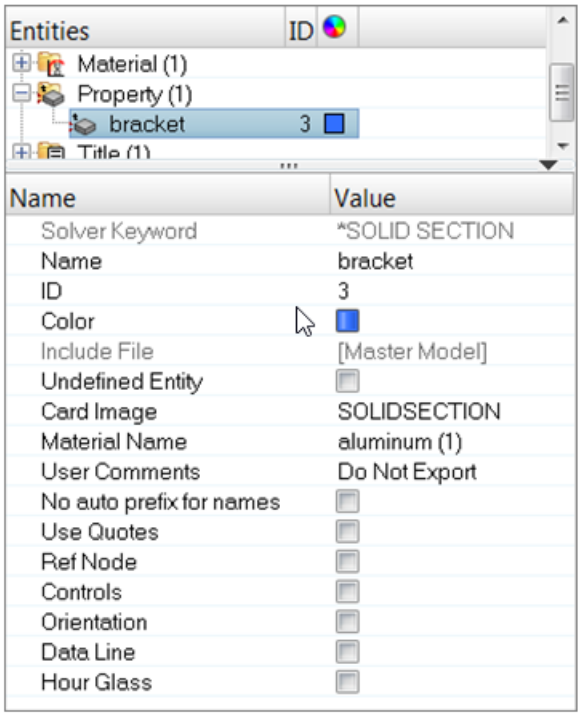


Figure 93:

1.5.5 View the Material Collector Card

In this step you will use the Entity Editor to review the defined material, aluminum.

In the Model Browser, **Material** folder, click **aluminum**.
Then Entity Editor opens and displays the material's card data.

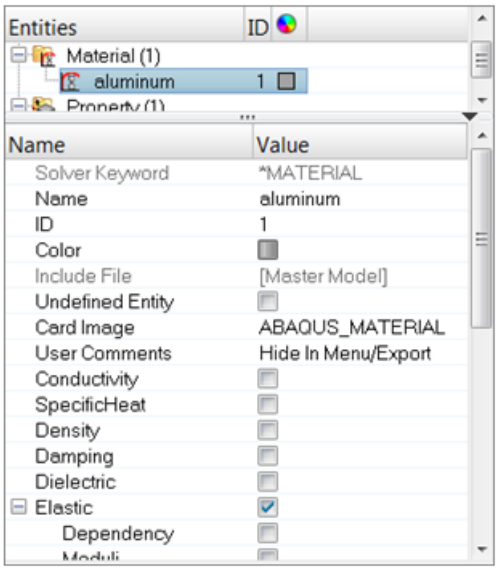


Figure 94:

1.5.6 Create the Material STEEL

In this step you will create a *MATERIAL with *ELASTIC to define as steel for the ELSET cradle. When you define the sectional property for the cradle component in the next section, the material will already be referenced in the property.

1. In the Model Browser, right-click and select **Create** > **Material** from the context menu.
   HyperMesh creates and opens a material in the Entity Editor.

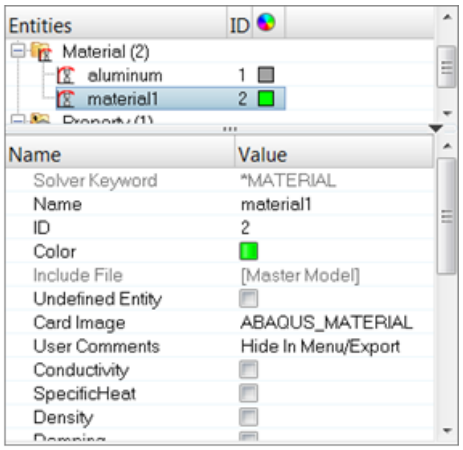


Figure 95:

2. For Name, type `steel`.
3. Select the **Elastic** checkbox.
   This creates *ELASTIC.

4. In the E field, type `200`.
5. For Poisson's ratio NU, type `0.3`.

1.5.7 Assign the Material to the Component

In this step you will assign the material, Steel, to the component and define *SHELLSECTION for the cradle property.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu. HyperMesh creates and opens a property in the Entity Editor.

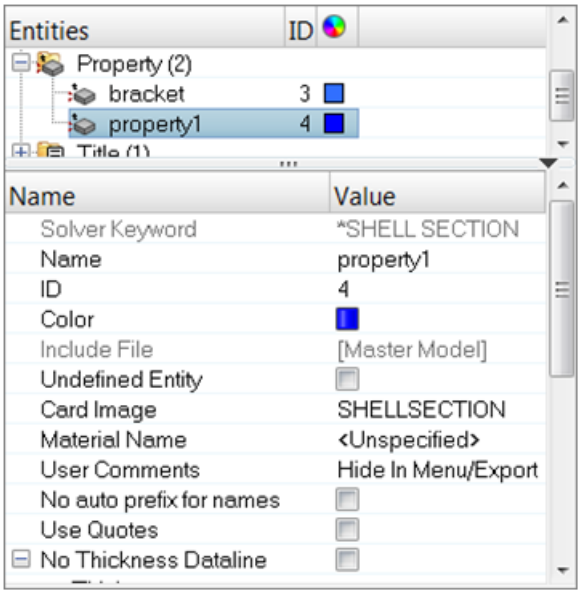


Figure 96:

2. For Name, type `cradle`.
3. For Material Name, click **Unspecified >> Material**.

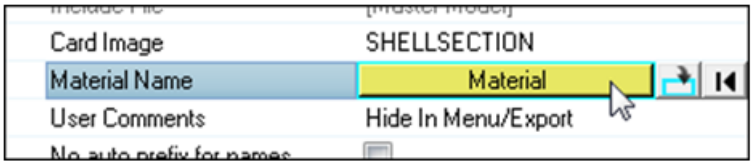


Figure 97:

4. In the **Select Material** dialog, select **steel** and click **OK**.

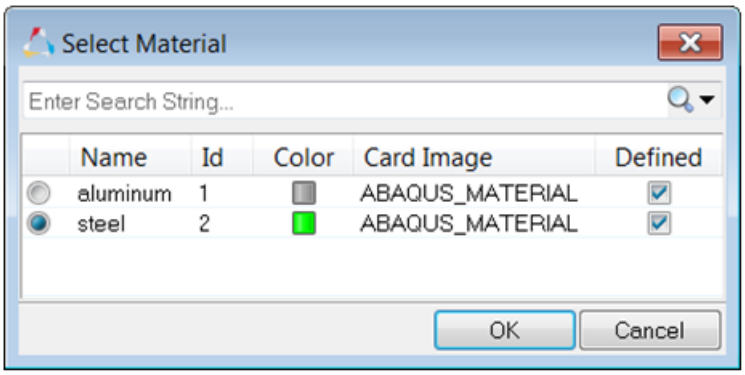


Figure 98:

5. For Thickness, type `2.5`.
6. In the Model Browser, **Component** folder, right-click on **cradle** and select **Assign** from the context menu.
7. In the **Assign to Component(s)** dialog, select **cradle** from the **Property** list.

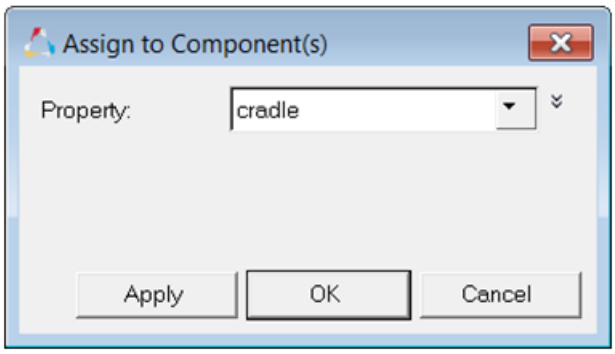


Figure 99:

8. Click **OK**.

1.5.8 Understand HyperMesh Entity Configurations and Types

HyperMesh elements and load entities have two identifiers: configuration and type.

The entity configuration is a HyperMesh core feature while the entity type is defined by the template. For example, HyperMesh element configurations include rigid, spring, quad4, and hex8. Possible quad4 configuration types in the Standard3D template include S4, S4R, S4R5, among others. Similarly, HyperMesh load configurations include constraints, force, pressure, and temperature. In the HyperMesh Abaqus templates, pressure configuration types include DLOAD, DFLUX, FILM, DECHARGE and Radiate.

Most of the HyperMesh element and load configurations have their own panels. From the 1D, 2D, and 3D pages, use the Elem Types panel. Load types can be chosen directly in the related panels. You can also use the load or element types panels to change the type of load.

Kinematic Constraints in HyperMesh

With the exception of *EQUATION, Abaqus kinematic constraints, such as *KINEMATIC COUPLING and *MPC (BEAM, TIE, LINK, PIN), are rigid (1D) elements in HyperMesh. From the **1D** page, use the **Rigids** panel to create them. Organize them into HyperMesh component collectors. No sectional property or material is needed for these entities. Hence, either organize them into their own component or into a component containing different Abaqus entities.

In the next few steps, you will create the Abaqus constraint *KINEMATIC COUPLING to simulate a bolt connecting the bracket's top bolt hole to the cradle. The model already contains two *KINEMATIC COUPLING entities, one at each of the bracket's bottom bolt holes. They are organized into the bracket component.

You will start by creating a new component in which you will organize all the *KINEMATIC COUPLING entities. This is not necessary, but is done to organize the data and demonstrate the selection of entities by configuration. This component will contain the *KINEMATIC COUPLING to be created. You will need to select any existing material to avoid creating one that is not needed.

Be sure to use the Elem Types panel to set the type for rigids to KINCOUP. This allows all elements created from the Rigids panel to be of the type *KINEMATIC COUPLING.

Finally, you will create the *KINEMATIC COUPLING using the Rigids panel.

1.5.9 Create a New Component Collector

In this step you will create a new component collector and set the element type rigid to KINCOUP.

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu. HyperMesh creates and opens a component in the Entity Editor.

   Note: This new component is now the current component. HyperMesh will automatically organize any new elements/geometry into this component.

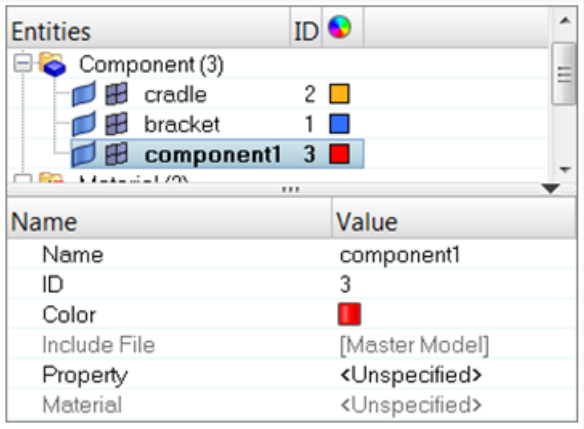
   

   Figure 100:

2. For Name, type `connection`.
3. Click the **Color** icon and select a color for the component.

4. Open the Element Type panel by clicking **Mesh** > **Assign** > **Element Type** from the menu bar.
5. Go to the **1D** subpanel.
6. Click **rigid=** and select **KINCOUP**.
7. Click **update**.
8. Click **return**.

1.5.10 Create a Reference Node

In this step you will use the Distance panel to create a node at the center of the bolt hole to be the *KINEMATIC COUPLING reference node.

1. Zoom into the top bolt hole as indicated in the following image.
2. Open the Distance panel by pressing **F4**.
3. Go to the **three nodes** subpanel.
4. Select the three nodes on the top side of the bolt hole for **N1**, **N2**, and **N3** as indicated in the image below.

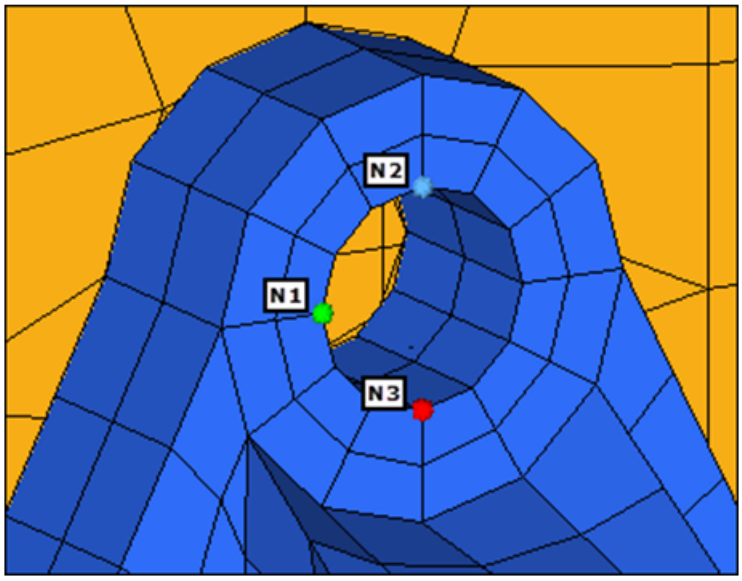


Figure 101:

5. Click **circle center**.
   HyperMesh creates a node at the center of the selected nodes.
6. Click **return**.

1.5.11 Create a Rigid Link

In this step you will use the Rigids panel to create a spider-like rigid link for the *KINEMATIC COUPLING.

1. Open the Rigids panel by clicking **Mesh** > **Create** > **1D Elements** > **Rigids** from the menu bar.
2. Go to the **create** subpanel.
3. Constrain the *KINEMATIC COUPLING reference node in all six directions by selecting all of the dof (degree of freedom) checkboxes.
4. Set the dependent selector to **multiple nodes**.
5. Using the independent selector, select the center node that you created in Step 8.5.
6. Using the dependent selector, select all nodes on the top side of the bolt hole as indicated in the following image.

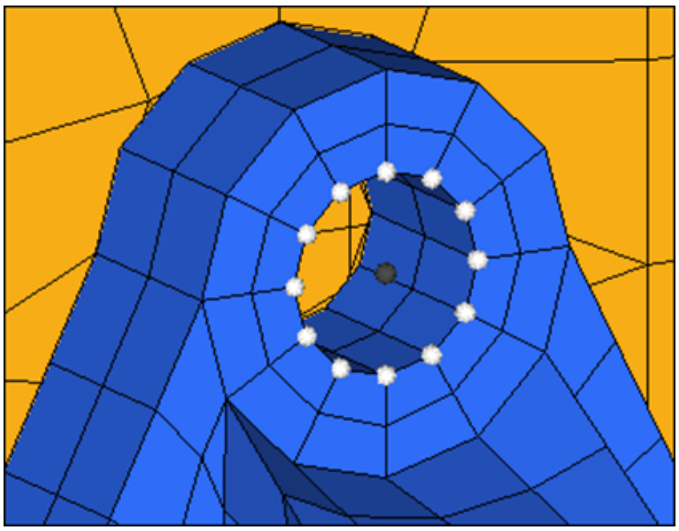


Figure 102:

7. In the Model Browser, Component folder, right-click on **bracket** and select **Hide** from the context menu.
   HyperMesh hides the elements of the bracket component.
8. Using the dependent selector, select the nodes around the hole in the cradle as indicated in the following image.

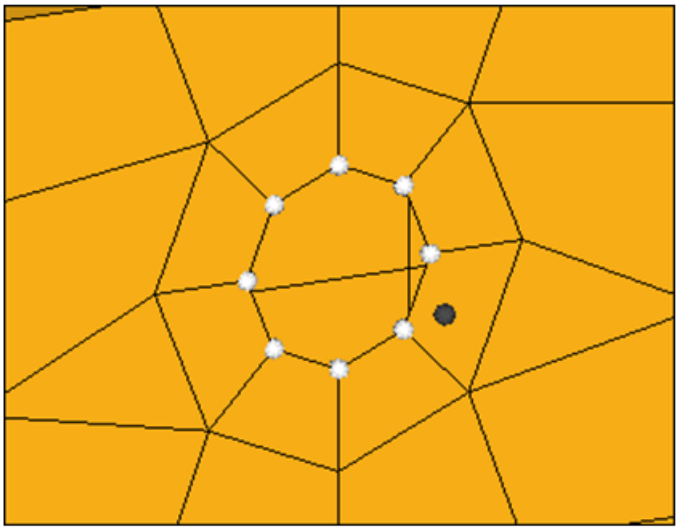


Figure 103:

9. Click **create**.
   HyperMesh creates the *KINEMATIC_COUPLING.
10. In the Model Browser, Component folder, right-click on **bracket** and select **Show** from the context menu.
    HyperMesh displays the elements from the bracket component.

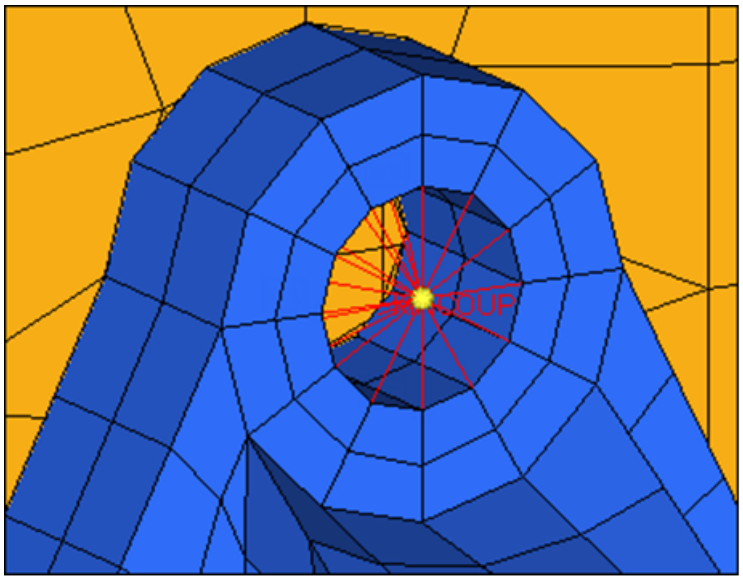


Figure 104:

11. Click **return**.
    The *KINEMATIC COUPLING is now created and organized into the connection component.

1.5.12 Move the *KINEMATIC COUPLING Entities

In this step you will move all of the *KINEMATIC COUPLING entities into the connection component using the Organize panel.

1. Open the Organize panel by clicking **Mesh** > **Organize** > **Elements** > **To Component** from the menu bar.
2. Click **elems >> by config**.
3. Click **config=** and select **rigidlink**.
4. Click **type=** and select **KINCOUP**.
5. Switch the **displayed/all** toggle to **all**.
6. Click **select entities**.
   HyperMesh selects all rigid links in the model.

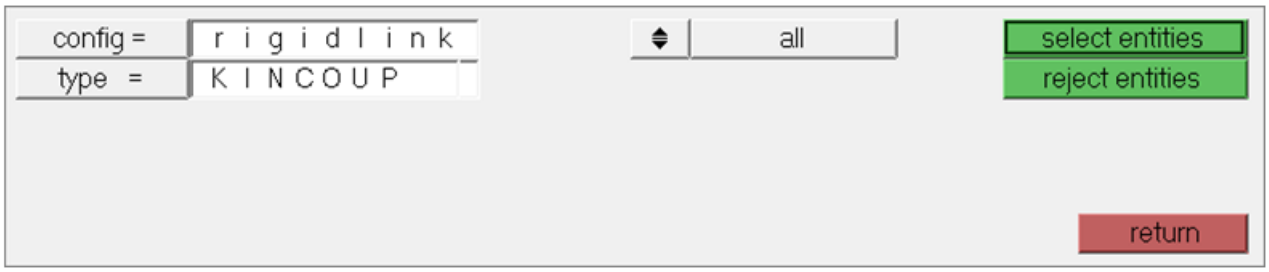


Figure 105:

7. Click **dest component=** and select **connection**.
8. Click **move**.
   HyperMesh moves all rigid links to the connection component.

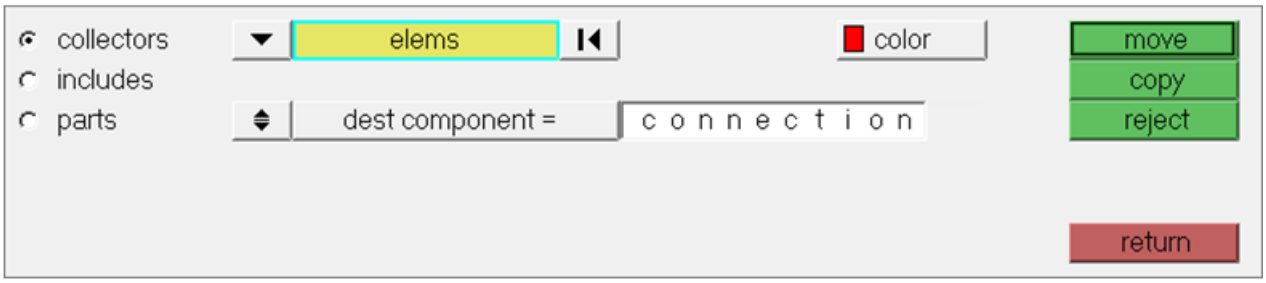


Figure 106:

9. Click **return**.
   All *KINEMATIC COUPLING entities are now organized into the connection component. Model data definition is complete.

1.5.13 Use Step Manager for Initial Conditions

In HyperMesh, you can use the **Step Manager** tool to create, edit, review, re-order, and delete Abaqus initial conditions and steps.

You can open the Step Manager by clicking **Tools** > **Loadsteps Browser** > from the menu bar.

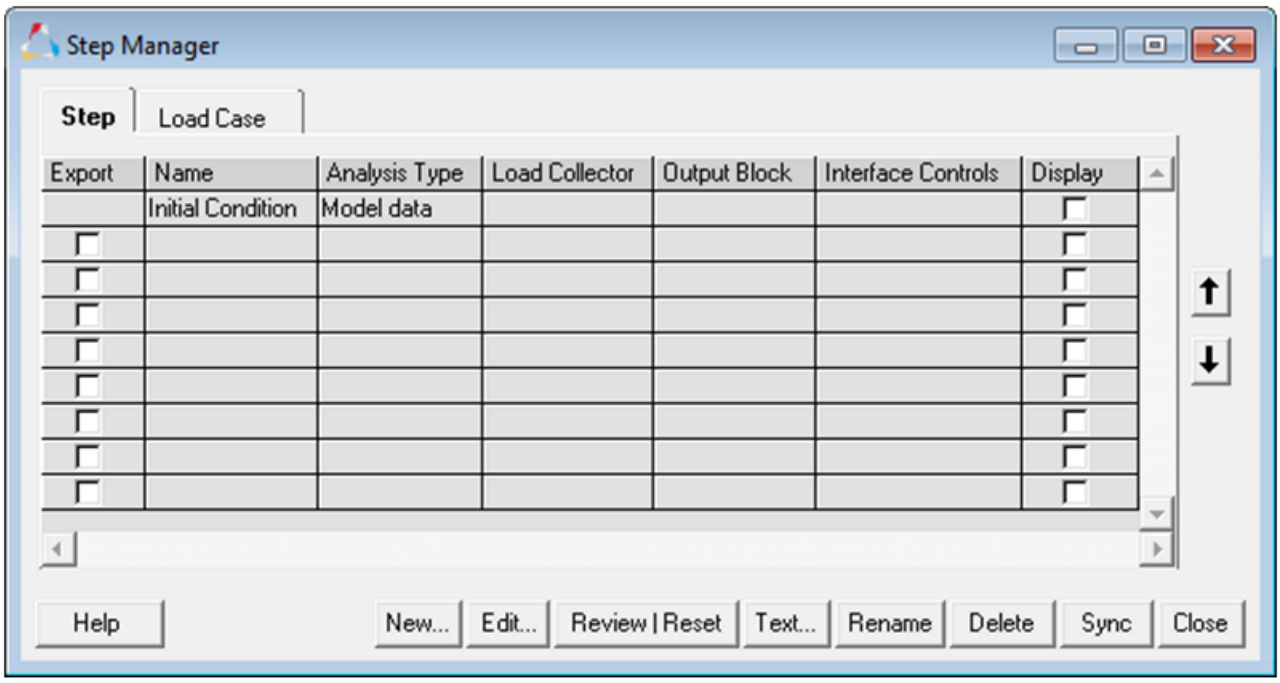


Figure 107:

The Step Manager has a default step named Initial Condition. This step is used to create boundary conditions and loads (initial conditions) in the model data portion of the Abaqus input file.

When you edit the Initial Condition step, the **Load Step: Initial Condition** dialog opens. You can only select valid boundary conditions and load types for model data in this dialog.

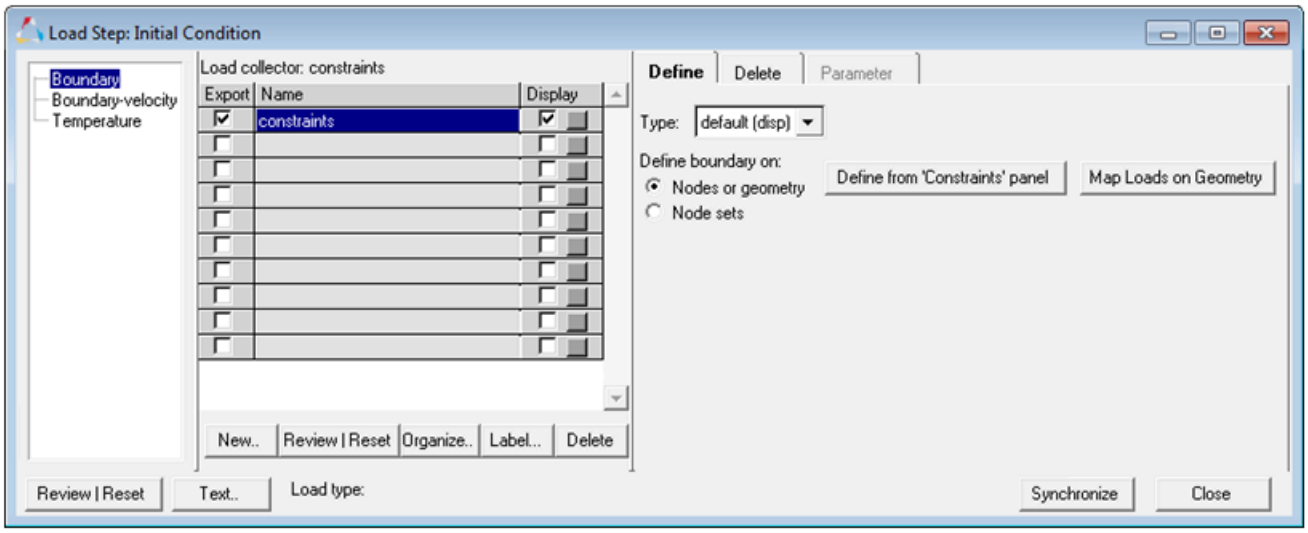


Figure 108:

Use this dialog from left to right by:

- Selecting a load step type from the list (left column)
- Creating a load collector (center area)
- Creating the loads using the menu area (right side)

1.5.14 Define *BOUNDARY

In this step you will create constraints at the cradle's ends using the **Step Manager**.

You will define the constraints as model data, not history data, by editing the Step Manager's default step named Initial Condition. For this step, you will create a load collector and then create the constraints.

1. Open the Load Step Manager by clicking **Tools** > **Loadsteps Browser** from the menu bar.
2. In the Step tab, click **Initial Condition**.
3. Click **Edit** to modify the step.
4. In the **Load Step: Initial Condition** dialog, select **Boundary** from the first pane.

Note: This specifies the type of initial condition you want to create.



Figure 109:

5. Click **New** to create a new load collector.
6. In the **Create Load Collector** dialog, **Name** field, type `constraints`.

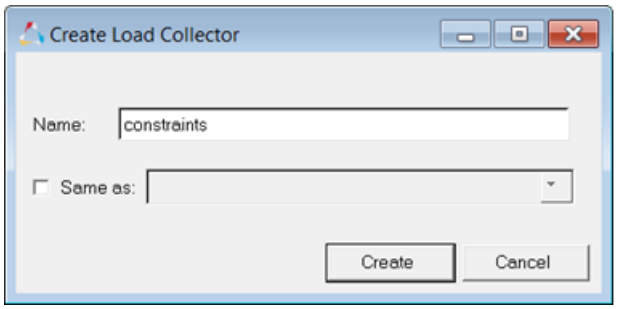


Figure 110:

7. Click **Create**.
8. In the Load Collector table, select the **Display** checkbox for the **constraints** load collector.
9. Optional: Next to the Display checkbox, click the **Color** box and select a color for the load collector.
10. In the Load Collector table, click **constraints**.
    The collector becomes active, and new tabs appear on the right side of the dialog.

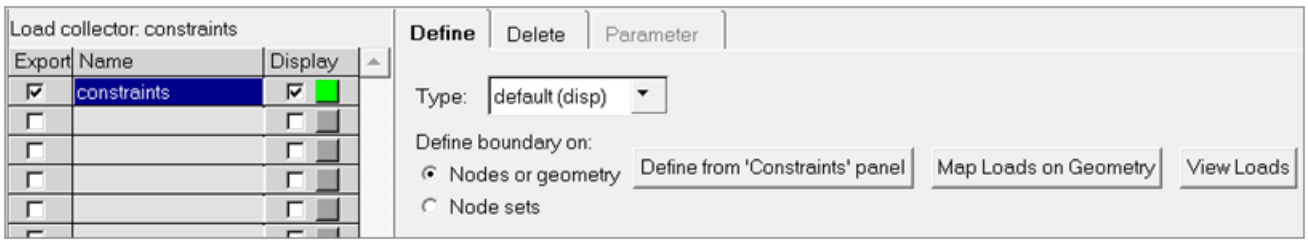


Figure 111:

11. In the Define tab, set **Type** to **default (disp)**.
12. Click **Define from 'Constraints' panel**.
    The panel opens.
13. From the Standard Views toolbar, click .
14. In the Constraints panel, click **nodes >> by window**.
15. With the exception of the nodes at the ends of the cradle, draw a rectangle around all of the displayed nodes to select them as shown below.

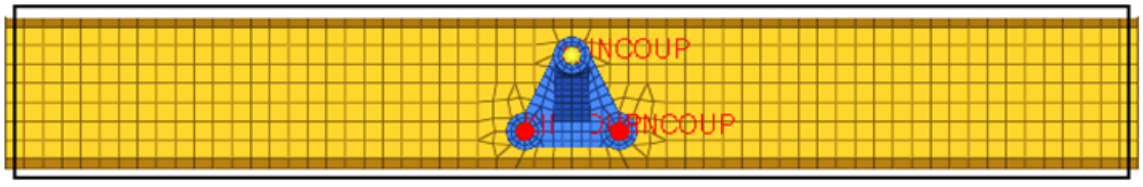


Figure 112:

16. Select the **exterior** checkbox then click **select entities**.
    HyperMesh selects all nodes outside the window you drew.

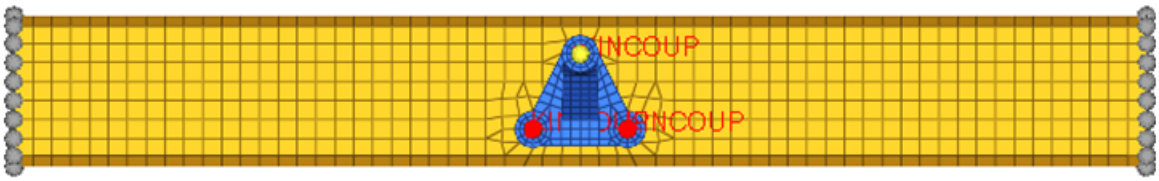


Figure 113:

17. Constrain the selected nodes in all six directions by selecting all **dof** (degree of freedom) checkboxes.
18. Click **create**.
    HyperMesh creates the constraints.

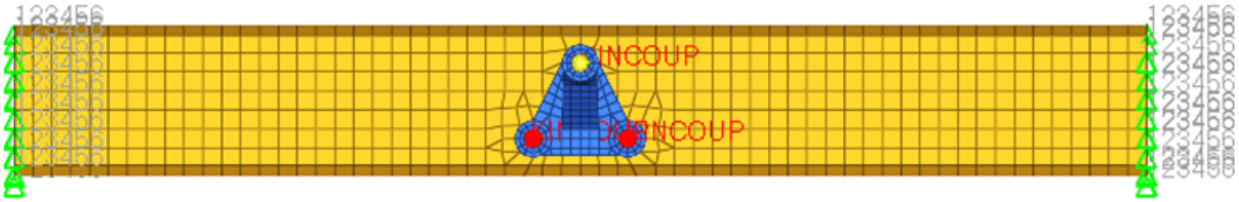


Figure 114:

19. Click **return** to go back to the Step Manager.
20. Exit the Initial Condition step and return to the **Step Manager** by clicking **close**.

1.5.15 Define History Data

The history data portion of the Abaqus input file defines the sequence of events for the simulation.

The loading history is divided into a series of steps. Each step contains the type of simulation, loads, constraints, output requests, and contacts (for Abaqus Explicit). The Abaqus *STEP option marks the start of a step, while the *END STEP option marks the end.

In the Step Manager you can create, review, edit, delete, and re-order the Abaqus steps. In the Step Manager, loads are organized into load collectors and output requests are organized into HyperMesh output blocks.

1.5.16 Define the Abaqus Step

For this analysis, you will be looking at the linear static response of the cradle and bracket assembly to a 100 kN load applied on the bracket, with the cradle's ends fully constrained.

This is a single event, therefore only one Abaqus step is needed. In this step you will use the **Step Manager** to define the step's title, heading, parameters, and the analysis procedure, and then apply a concentrated force (*CLOAD) on the bracket's arm.

1. In the **Step Manager**, **Step** tab, click **New**.
2. In the **Create New Step** dialog, **Name** field, type `step1`.
3. Click **create**.
   HyperMesh creates a new step.
4. In the first pane, click **Title**.
   Additional options display, from which you can define the step's title.
5. Select the **Step heading** checkbox and type `100kN load`.
6. Click **Update**.
7. In the first pane, click **Parameter**.
8. Write the step's name to the Abaqus input file by selecting the **Name** checkbox.
9. Set the analysis for small-scale, linear deformations by selecting the **Perturbation** checkbox.
10. Click **Update**.
11. In the first pane, click **Analysis procedure**.
12. Set **Analysis type** to **static**.
13. Click **Update**.
14. Click **Text** to review what has been defined so far for the step.

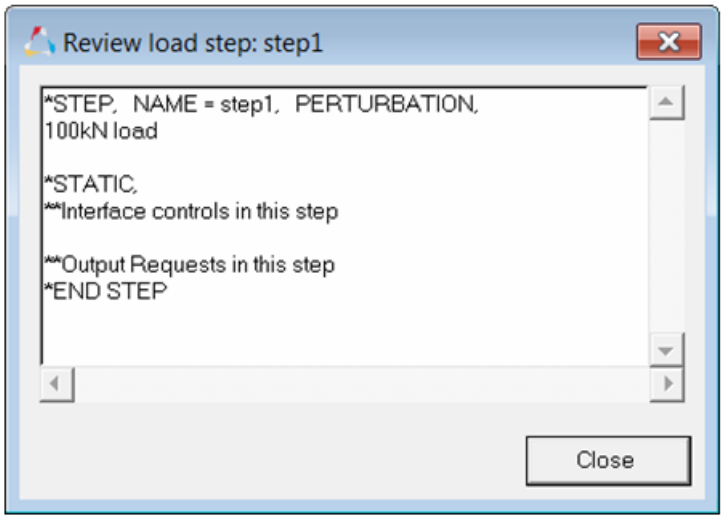


Figure 115:

15. When you are finished reviewing the contents of the window, click **Close**.
16. In the first pane, expand **Concentrated loads** and click **CLOAD-Force**.
17. Click **New** to create a new load collector.
18. In the **Create Load Collector** dialog, **Name** field, type `force`.
19. Click **Create**.
20. Optional: Next to the **Display** checkbox, click the **Color** box and select a color for the load collector.
21. In the **Load Collector** table, click **force**.
    The collector becomes active, and new tabs appear on the right side of the dialog for the selected load type (CLOAD-Force).

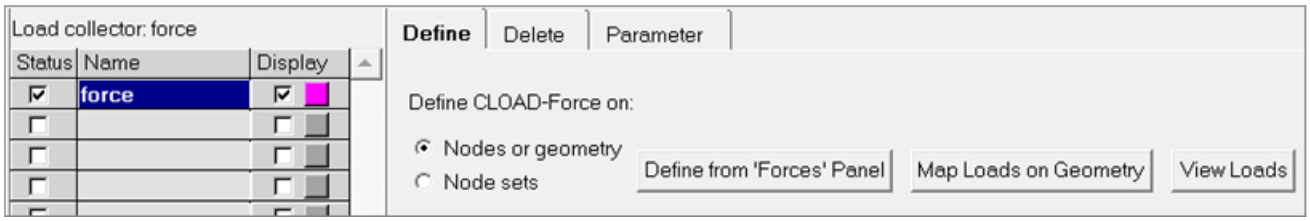


Figure 116:

22. In the **Define** tab, click **Define from 'Forces' Panel**.
HyperMesh opens the **Forces** panel, from which you can create a CLOAD.

23. Select the central node on the top side of the bracket's arm as indicated in the following image.

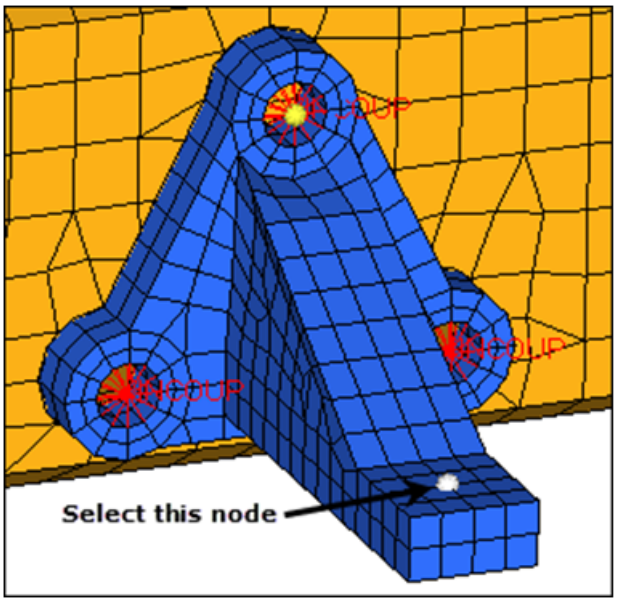


Figure 117:

24. In the magnitude= field, type `-100`.

25. Leave the system selector set to **global system**.

26. Set the orientation selector to **z-axis** for the force's direction vector.

27. Click **create**.
HyperMesh creates the force.

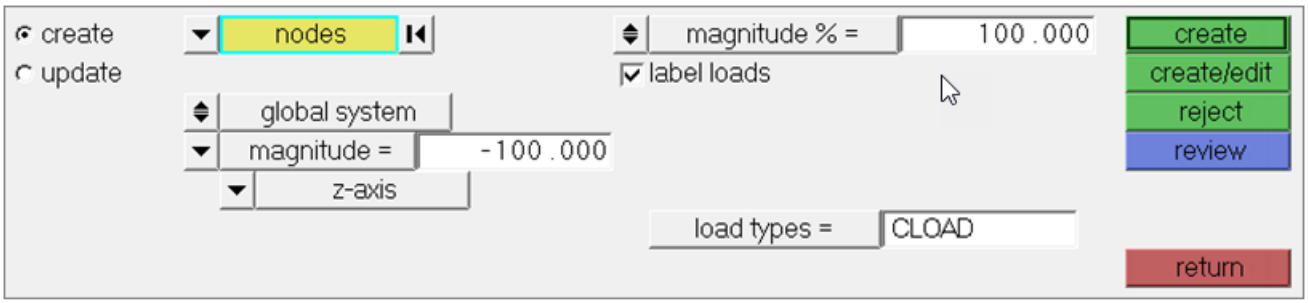


Figure 118:

28. Click **return** to go back to the Step Manager.

1.5.17 Specify Output Requests

In this step you will use the Step Manager to specify displacement and stress results to output to the `.odb` and `.fil` results files for step1. You will then export the model to an `.inp` file.

1. In the first pane of the **Load Step** dialog, expand **Output request** and click **ODB file**.
2. Click **New** to create a new output block.
3. In the **Create Output block** dialog, **Name** field, type `step1_output`.
4. Click **Create**.
5. In the Output block table, click **step1_output**.
6. From the Output tab, select the **Output** checkbox.
7. Leave Output set to **field**.
8. Select the **Node output** and **Element output** checkboxes.

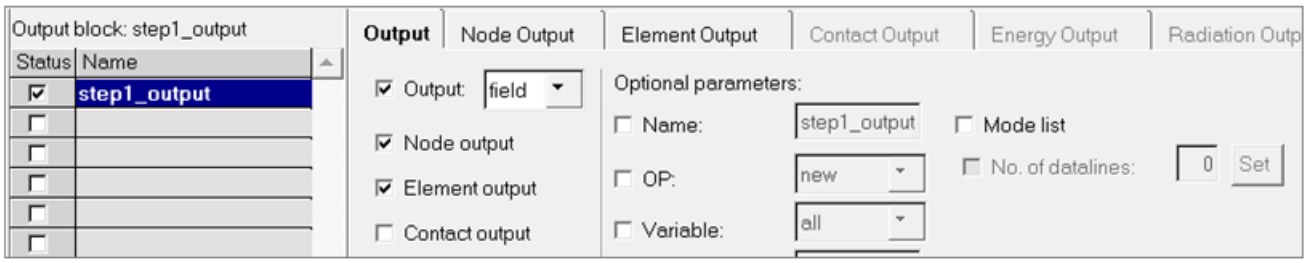


Figure 119:

9. Click the **Node Output** tab. From this tab you can specify the nodal displacement output for the `.odb` file.
10. From the list of output options, expand **Displacement** and select the **U** checkbox.

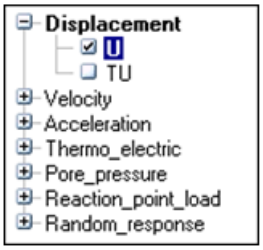


Figure 120:

11. Click **Update**.
12. Click the **Element Output** tab. From this tab you can specify elemental stress output for the `.odb` file.
13. From the list of output options, expand **Stress** and select the **S** checkbox.

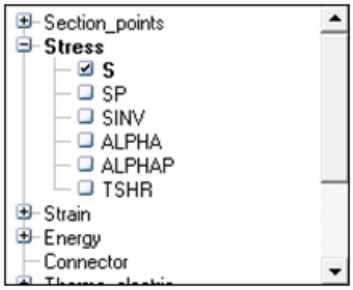


Figure 121:

14. Click **Update**.
15. In the first pane of the **Load Step** dialog, expand **Output request** and select **Result file (.fil)**.

Note: The output block, step1_output, is still highlighted (active) in the Output block table.

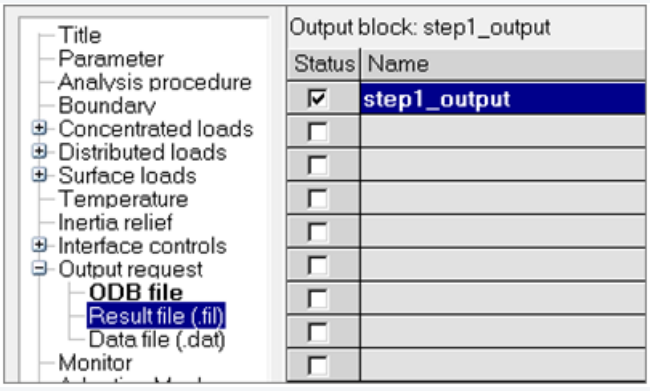


Figure 122:

AcuSolve

2

AcuSolve tutorials.

This chapter covers the following:

- 2.1 User Interface Introduction (p. 104)
- 2.2 Turbulence (p. 143)
- 2.3 Heat Transfer (p. 167)
- 2.4 Multiphase (p. 364)
- 2.5 Moving Body (p. 449)
- 2.6 Porous Medium (p. 507)
- 2.7 Optimization (p. 519)

2.1 User Interface Introduction

2.1.1 ACU-T: 1000 HyperWorks UI Introduction

This tutorial provides the instructions for setting up a Computational Fluid Dynamics (CFD) simulation making use of the HyperWorks package. HyperWorks is a comprehensive suite of various Computer-Aided Engineering (CAE) products, each specialized in a certain aspect of the CAE process. These include HyperMesh as a generic, powerful geometric modeling and pre-processing tool, and HyperView as a post-processing and visualization tool. Bridging these two applications is a complete range of solvers for a gamut of engineering applications. Among these solvers is AcuSolve, which is Altair's offering for fluid flow and thermal analysis simulations.

HyperMesh's inbuilt geometric modeling and finite element meshing capabilities will allow you to create the geometry for your problem and generate excellent quality meshes in a single tool. Meshes generated in HyperMesh can be exported in the format that AcuSolve will recognize. Moreover, HyperMesh's integration with AcuSolve also allows you to complete the pre-processing steps in HyperMesh itself, including the problem setup. Once you have completed setting up your simulation in HyperMesh, you can directly generate the AcuSolve input files. You can also choose to directly launch AcuSolve from within HyperMesh. This integration is expected to be especially beneficial for you if you happen to be a traditional user of HyperMesh for your modeling and meshing requirements.

The HyperWorks package has a powerful tool for post-processing and visualizing the results of your CFD simulations, called HyperView. HyperView enables you to visualize data interactively as well as capture and standardize your post-processing activities using process automation features. HyperView combines advanced animation and XY plotting features with window synching to enhance results visualization. HyperView also saves 3D animation results in Altair's compact H3D format so you can visualize and share CAE results within a 3D web environment using HyperView Player. HyperView has a rich feature set that you might find beneficial to your post-processing activities and are useful to explore. HyperView has inbuilt direct-reading capabilities for AcuSolve results and does not require any conversion steps.

In this tutorial, you will learn how to use HyperMesh for importing a geometric model and generating a mesh. You will then set up and launch the simulation from within HyperMesh. Following that, you will learn how to use HyperView for post-processing AcuSolve results.

In this tutorial you will do the following:

- Analyze the problem
- Start HyperMesh and create a model database
- Import the geometry for the simulation
- Generate and organize the mesh using the Mesh Controls Browser
- Set general problem parameters
- Set solution strategy parameters
- Set the appropriate boundary conditions
- Run AcuSolve
- Monitor the solution with AcuProbe

- Post-process with HyperView

Prerequisites

To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve. This tutorial introduces you to HyperMesh and HyperView so no prior experience is expected.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T1000_manifold.x_t` from `HyperMesh_tutorial_inputs.zip`.

The file `ACU-T1000_manifold.x_t` stores the geometry information for the fluid portion of the model for this problem in Parasolid ASCII format.

The color of objects shown in the modeling window in this tutorial and those displayed on your screen may differ. The default color scheme in HyperMesh is "random," in which colors are randomly assigned to groups as they are created. In addition, this tutorial was developed on Windows. If you are running this tutorial on a different operating system, you may notice a slight difference between the images displayed on your screen and the images shown in the tutorial.

Analyze the Problem

An important step in any CFD simulation is to examine the engineering problem at hand and determine the important parameters that need to be provided to AcuSolve. Parameters can be based on geometrical elements, such as inlets, outlets, or walls, and on flow conditions, such as fluid properties, velocity, or whether the flow should be modeled as turbulent or as laminar.

The system being simulated here is a manifold pipe, analogous to an inlet manifold in an engine. An inlet manifold distributes the incoming flow to multiple outputs. As can be seen in the image below, the pipe has a single inlet and multiple outlets, thus distributing a fraction of the flow among each outlet. Ideally in an inlet manifold used in an engine, the manifold design is such that it ensures near-equal distribution of flow among all the outlets. However, the geometry being used here is purely a demonstration case and not an optimized manifold geometry.

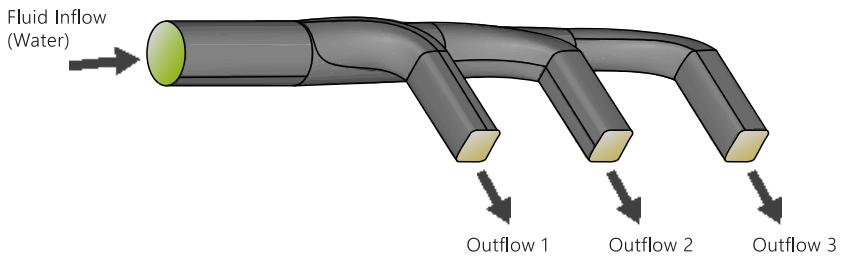


Figure 123: Schematic of the Problem

Introduction to HyperMesh

HyperMesh is a generic tool offering a combination of geometric modeling and pre-processing capabilities.

HyperMesh supports a number of commonly used solvers used in simulating various engineering applications, providing direct interfaces to most of them. This offers you flexibility to use HyperMesh as a single tool for most, if not all, of your modeling and pre-processing activities.

AcuSolve is among the solvers that are closely integrated with HyperMesh. In order to simplify the interfaces associated with each solver, HyperMesh uses user profiles to automatically manage the templates for a given solver. Each user profile has an associated pre-defined set of menus, options and toolbars visible. User profiles ensure that the solver setup is in accordance with the options and requirements of the solver associated with the profile in which it is generated. It is advised that you make sure you are using the correct user profile when setting up a model. Also, it is recommended that the active user profile is not to be changed while the current HyperMesh database is populated.

In this tutorial, you will be working in a user profile associated with AcuSolve. Once you begin the tutorial, you will change the active user profile to the AcuSolve user profile. HyperMesh remembers the last active user profile when it is restarted. If the last HyperMesh user on your machine was working in the AcuSolve user profile when you launch HyperMesh, it will start with the AcuSolve user profile.

A HyperMesh session loaded with the AcuSolve user profile looks like the image below.

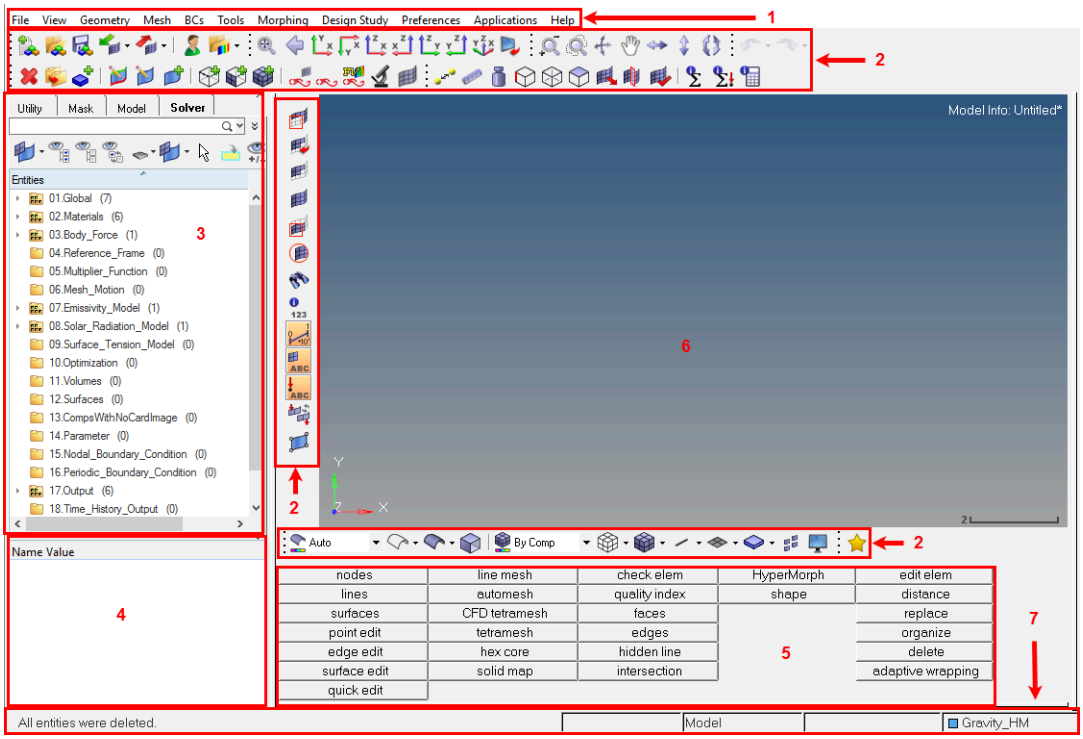


Figure 124: HyperMesh Interface with Active AcuSolve User Profile

1. Menu bar: Located at the top of the window, just under the title bar. Like the pull-down menus in many applications, these menus drop-down a list of options when clicked.

2. Toolbars: Located around the modeling window. These have icons that provide quick access to commonly-used functions, such as changing display options. They can be dragged and placed as per the user preference.

   Below are some of the commonly used toolbars.



Figure 125: Standard Toolbar

Provides the options for creating, opening or saving the database, import/export options and changing user profiles.



Figure 126: Checks Toolbar

On the Checks toolbar, you can access various checks and calculations tools that are commonly used in the model building process.



Figure 127: CFD Toolbar

The CFD toolbar has options for creating, deleting and organizing entities, accessing meshing panels and launching AcuConsole or AcuSolve.

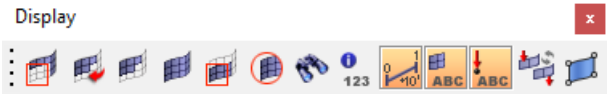


Figure 128: Display Toolbar

On the Display toolbar, you can control what entities HyperMesh displays, primarily by masking entities to hide or display. This toolbar is usually located along the left edge of the modeling window.



Figure 129: Visualization Toolbar

Options available on the Visualization toolbar control how HyperMesh visualizes entities in the modeling window.

3. Tab area: The two areas marked 3 and 4 in Figure 124 make up the tab area. The tab area is so named because various specialized tools display on tabs in this area of the interface. One

of these tabs is the Model tab, which you will be using most frequently. The Model tab will also be the tab active by default when you start a HyperMesh session. The top half of the tab area, marked 3, is the browser area. Depending on the selected tab, you will be able to see the various options or entities which belong to the active HyperMesh database. For example, when the Model tab is selected, the Model Browser will display the entities present in the model, each of which carry some information about the model. This information may be related to the geometrical components that make up the model, the material information, the load information, and so on. The model structure is viewed as a flat, listed tree structure within the browser.

4. Entity Editor: The bottom half, marked 4 in Figure 124, is the Entity Editor. In the Entity Editor, you will be able to view and edit the information associated with the different entities available in the browser. Clicking on an entity in the browser area will display the entity related information in this area.
5. Main Menu: The main menu displays the available functions. You access these functions by clicking on the button corresponding to the function you want to use. Clicking on the button will open the panel associated with the function in the menu area.
6. Modeling Window: The modeling window is the display area for your model. You can interact with the model in three-dimensional space in real time. In addition to viewing the model, entities can be selected interactively from the modeling window.
7. Status bar: The status bar is located at the bottom of the screen. The four fields on the right side of the status bar display the current include file, current part, current component collector and current load collector. As you work in HyperMesh, any warning or error messages also display in the status bar on the left side.

Introduction to HyperView

HyperView is a generic post-processing and visualization environment for finite element analysis (FEA), CFD, multi-body system simulation, digital video and engineering data.

HyperView offers direct-reading capabilities for AcuSolve generated results. AcuSolve results can be directly opened in HyperView. HyperView also has process automation features, which can enable you to expedite and standardize your post-processing activities.

The image below shows the HyperView interface when it is started.

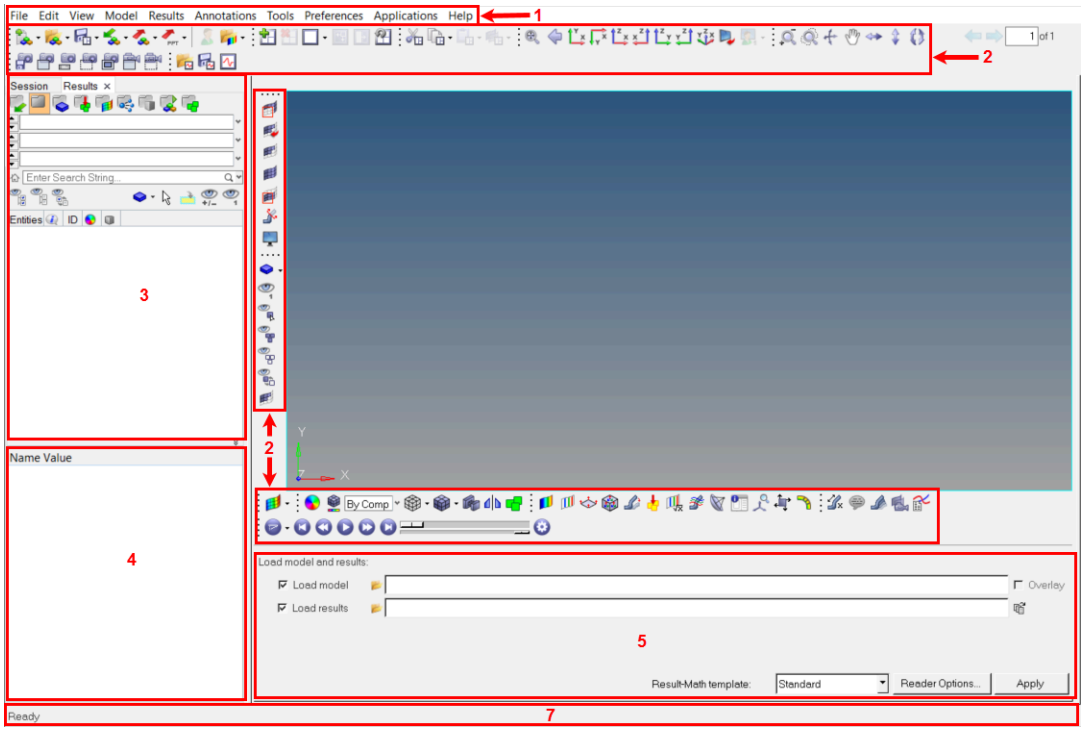


Figure 130: HyperView

1. Menu bar: Located at the top of the window, just under the title bar. Like the pull-down menus in many applications, these menus drop-down a list of options when clicked.
2. Toolbars: Located around the modeling window. These have icons that provide quick access to commonly-used functions, such as changing display options. They can be dragged and placed as per the user preference. Below are some of the commonly used toolbars.



Figure 131: Standard Toolbar

Provides the options for creating or opening a model, saving an HyperView session and import/export options.



Figure 132: Results Toolbar

On the Results toolbar you can access various options related to displaying the results, for example, contours, vectors and streamlines.

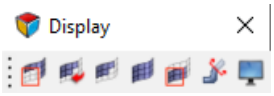


Figure 133: Display Toolbar

The Display toolbar provides you with quick access to the Mask panel, Section Cut panel and Display Controls.



Figure 134: Visibility Controls Toolbar

The Visibility Controls toolbar provides you quick access to the visibility controls of the entities in the Results Browser.

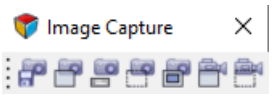


Figure 135: Image Capture Toolbar

The Image Capture toolbar provides you quick access to the image and video capturing capabilities.

3. Tab area: The two areas marked 3 and 4 in Figure 130 make up the tab area. The tab area is so named because various specialized tools display on tabs in this area of the interface. In HyperView, one of these tabs is the Results tab, which you will be using most frequently. Results tab will also be the tab active by default when you start an HyperView session. The top half of the tab area, marked 3, is the browser area. Depending on the selected tab, here you will be able to see the various options or entities which are part of the active HyperView model, in a listed tree structure similar to HyperMesh.
4. Entity Editor: The bottom half, marked 4, is the Entity Editor. In the Entity Editor, you will be able to see and edit the information associated with the different entities available in the browser. Clicking on an entity in the browser area will display the entity related information in the Entity Editor.
5. Panel area: The panel area displays the function panel associated with the active function selection. You can access these functions by clicking on the icon on a toolbar corresponding to the function you want to use. Clicking on the icon will open the panel associated with the function in the panel area. When you launch HyperView, you will see the Load Model panel in this region.

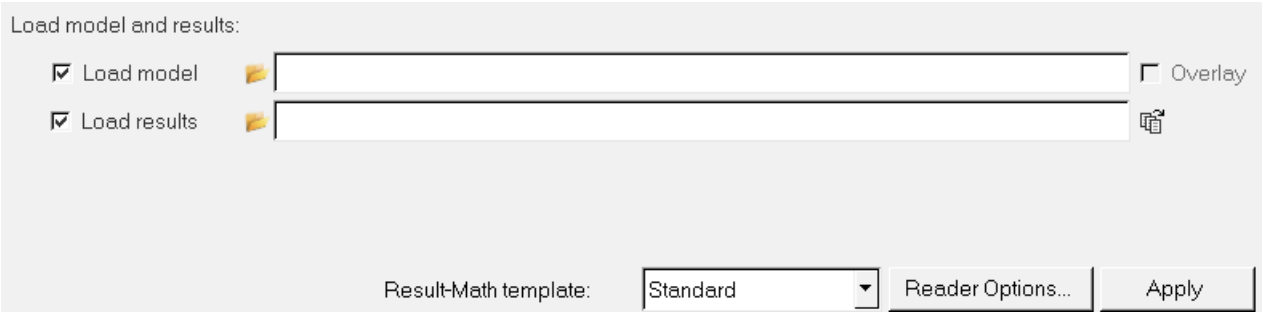


Figure 136:

6. Modeling window: The modeling window is the display area for your model. You can interact with the model in three-dimensional space in real time. In addition to viewing the model, entities can be selected interactively from the modeling window.
7. Status bar: The status bar is located at the bottom of the screen. As you work in HyperView, any warning or error messages also display in the status bar, on the left side.

Define the Simulation Parameters and Import the Geometry

Start HyperMesh and Create a Model Database

In this tutorial, you will begin by creating a model database in HyperMesh, loading the geometry and generating and organizing the mesh. Next you will set up the problem parameters, component parameters, and boundary conditions and then launch AcuSolve to solve for the number of time steps specified. Finally, you will visualize some characteristics of the results using HyperView.

In the next steps you will start HyperMesh and create the database for storage of the simulation settings.

1. Start HyperMesh from the Windows Start menu by clicking **Start** > **Altair <version>** > **HyperMesh**.
   A **User Profiles** dialog opens.

   Note: If it does not show up for you, click **Preferences** on the menu bar and select **User Profiles**.
2. Select **HyperMesh** from the Application drop-down menu.
3. Select **AcuSolve** from the list of applications.
4. Click **OK**.

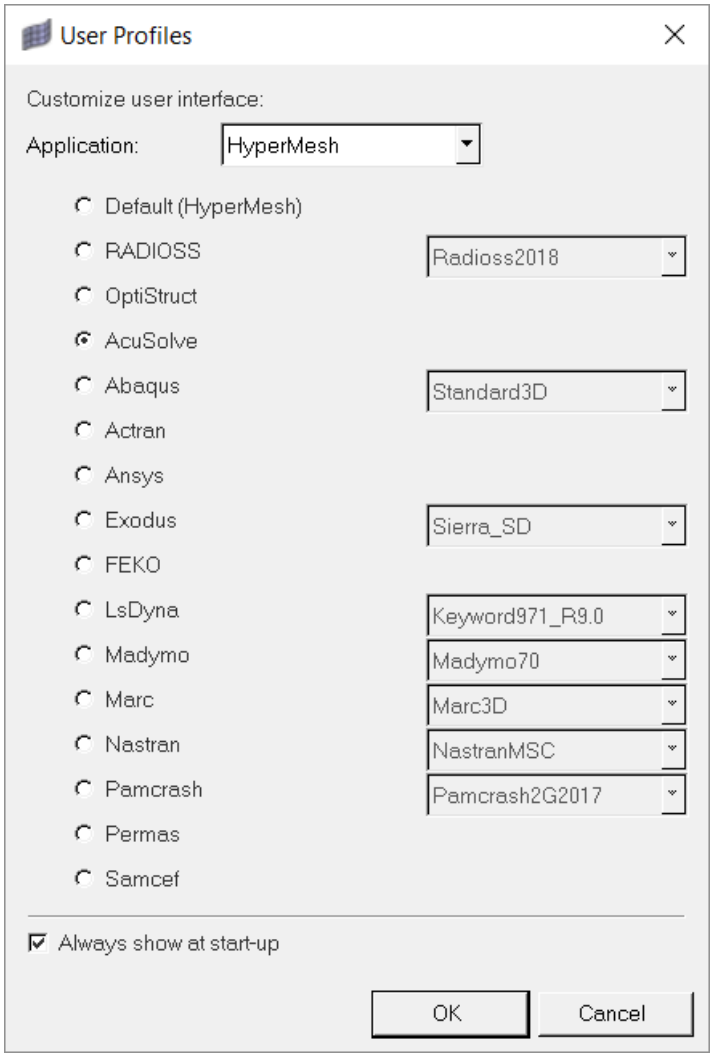


Figure 137:

Traditional HyperMesh users will be able to tell the difference between the default HyperMesh profile and the CFD (AcuSolve) profile. There will be an additional CFD toolbar visible. Also, the Model Browser will be populated with some entities relevant to a CFD simulation setup.

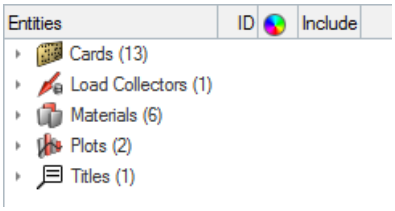


Figure 138:

5. Click **File** > **Save** to open the **Save Model** dialog.
   Save the model database frequently as you proceed through the tutorial steps.
6. Browse to the location that you would like to use as your working directory.

This directory is where all files related to the simulation will be stored. When you are setting up the problem, there will be a file with extension `hm` in this directory, which corresponds to an HyperMesh model database. Once the mesh and solution are generated, additional files and directories will be added by HyperMesh and HyperMesh.

7. Create a new directory in this location. Name it `ACU1000_HyperWorks` and navigate into this directory.
8. Enter `ACU1000_HyperWorks` as the file name for the database, or choose any name of your preference.

   Note: In order for other applications to be able to read the files written by HyperMesh, the database path and name should not include spaces.

9. Click **Save** to create the database.

Import the Geometry

You will import the geometry in the next part of this tutorial. You will need to know the location of `ACU-T1000_manifold.x_t` in order to complete these steps. This file contains information about the geometry in Parasolid ASCII format.

1. Click **File** > **Import** > **Geometry**.

   Tip: Alternatively, click the arrow next to the Import Solver Deck icon on the standard toolbar and select **Import Geometry**.

2. Select **Parasolid** as the File type.

   Note: In general, if you are not sure about the geometry file type, leave the File type option as Auto Detect.

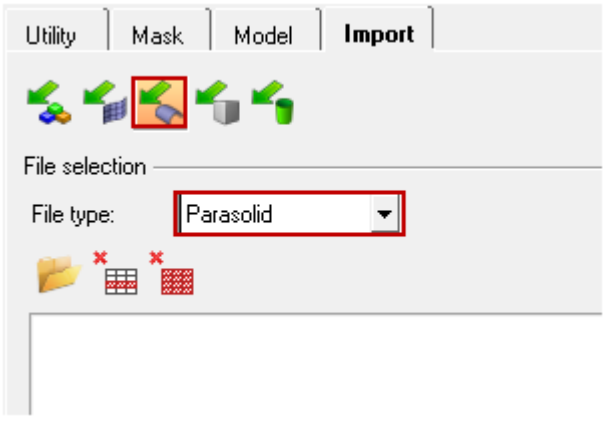


Figure 139:

3. Click .

Note: If you see anything in the list of import files, clear the list before this step by clicking .

4. In the **Select Parasolid file** dialog, select `ACU-T1000_manifold.x_t` and click **Open**.
5. Make sure that the selected file is in the list of import files and click **Import**.
6. Click **Close**.
7. Click on the Visualization toolbar to display the surfaces.

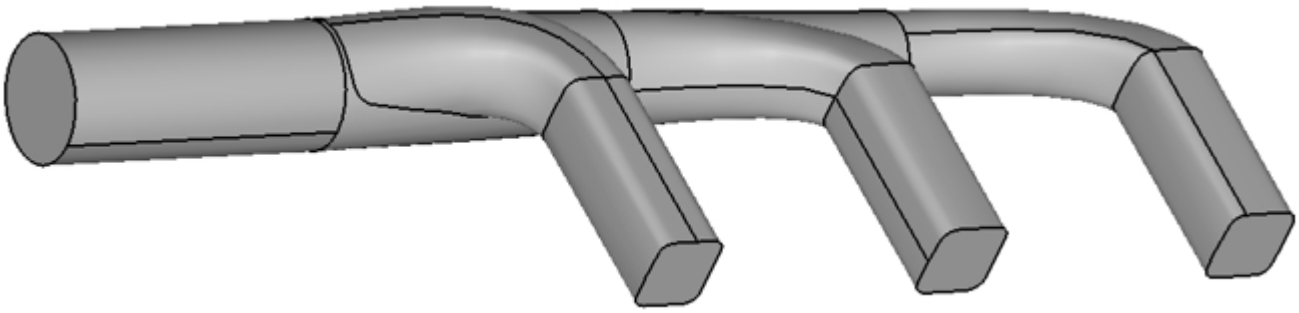


Figure 140:

Tip: Use the following controls for visualizing the model:

1. Ctrl + Left-Click: Rotate the model
2. Ctrl + Scroll: Zoom in/out
3. Ctrl + Right Click: Pan the model

Define Mesh Controls and Generate the Mesh

In the following steps you will set up the mesh controls and generate the mesh for the model. You will be introduced to the Mesh Controls Browser for this purpose.

Select **Mesh** > **Mesh Controls** from the menu bar.
The Mesh Controls Browser opens.

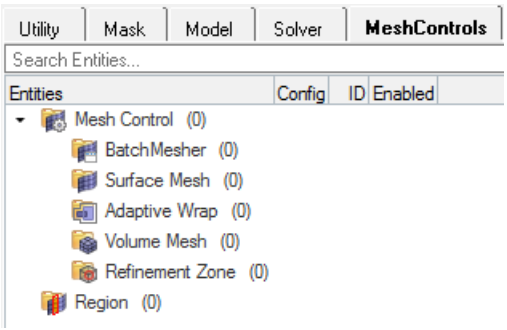


Figure 141:

The Mesh Controls Browser lets you access all of the different meshing technologies in the single browser. As you can see in the image above there are options to generate the surface mesh, volume mesh, refinement zones, and so on. Within these options there are associated model, local, feature, and refinement controls available. The model controls apply to the entire model. The local controls apply to a specific entity in the model, such as surfaces and elements.

You will start by creating a surface mesh control followed by a volume mesh control with active boundary layers. You will then add a volume mesh local control for the surfaces that do not require a boundary layer.

Set up the Surface Mesh Controls and Generate Surface Mesh

1. Right-click on **Surface Mesh** in the Mesh Controls Browser. From the context menu that appears, select **Create** > **Model** > **Size and Bias** > **Surfaces**.
2. Optional: In the Entity Editor, set the entity name to `Surface_Mesh_Control`.
3. Set the Element Size to `0.003`.
4. Set the Element Type to **Trias**.
5. Under the Entity Selection group, click in the field next to Entities then click the **Surfaces** collector.



Figure 142:

The surface entity selector menu opens in the menu area.

6. In the menu area, click the **surfs** collector and select **all**.
7. Click **proceed**.
8. Expand the **Advanced** group and verify the following settings:
   a) Destination Component: Original
   b) Mesh Connectivity: Keep

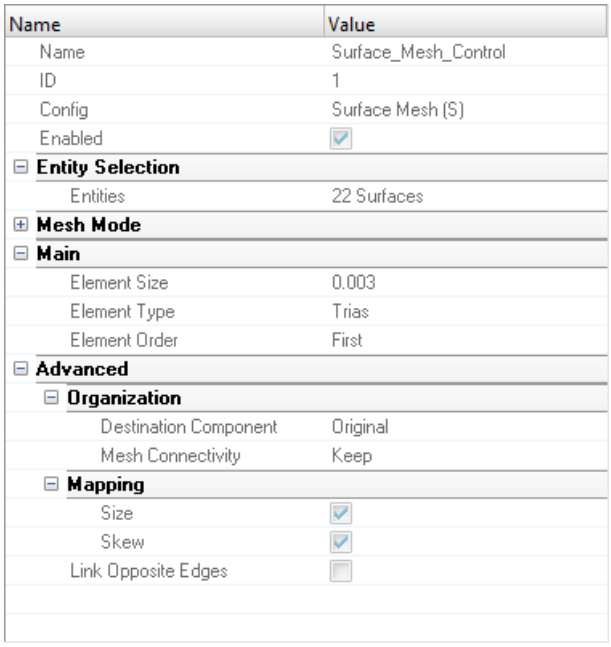


Figure 143:

9. In the Mesh Controls Browser, right-click on **Surface Mesh** and select **Mesh**.
Surface mesh is generated on the model.

Organize the Surfaces Elements

In this step, you will create component collectors for the surface elements and move the surface mesh elements on the Inlet and Outlet surfaces into the respective components. Organizing the surface mesh elements will help you in specify boundary conditions at the surfaces. Use the following figure as reference:

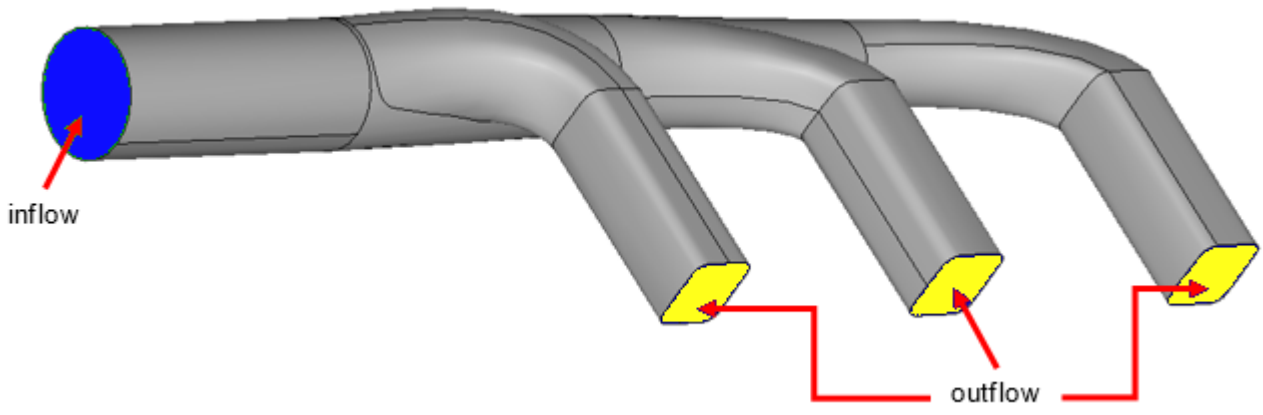


Figure 144:

1. From the menu bar, select **BCs** > **Components** > **CFD**.

The **Create CFD Components** dialog opens.

2. In the dialog, click the Check none icon then activate the **Inflow** and **Outflow** fields.
3. Click **Create** then **Close**.

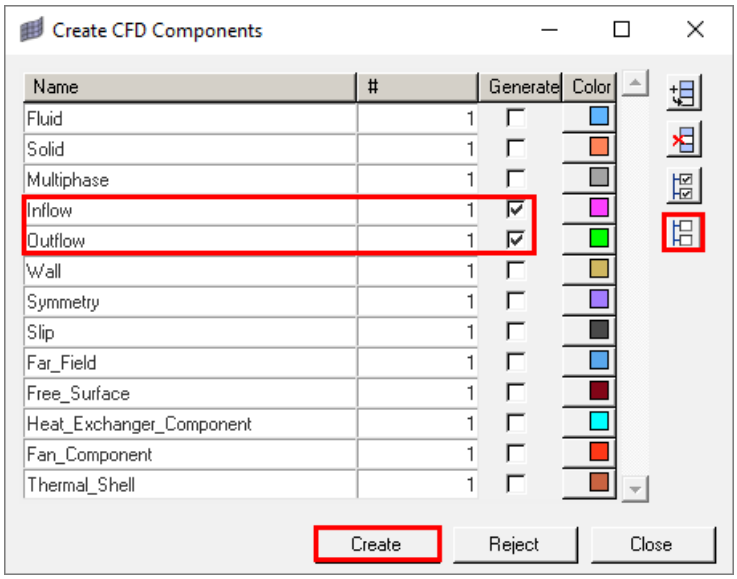


Figure 145:

4. Close the dialog and go to the Model Browser and expand the list of components. Right-click on **Part 1** and select **Rename**.
5. Type `Wall` as the new component name and press Enter.
6. Open the Organize panel by doing one of the following:
   a) Click **organize** in the panel area.

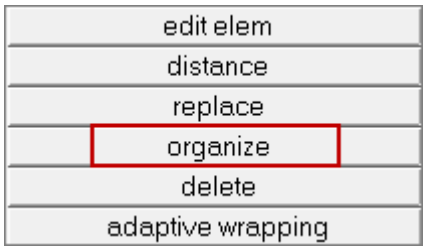


Figure 146:

   b) Click **BCs** > **Organize** from the menu bar.
   c) Click on the CFD toolbar.
7. In the modeling window, zoom in on the inlet surface region and select any mesh element on the inlet surface.

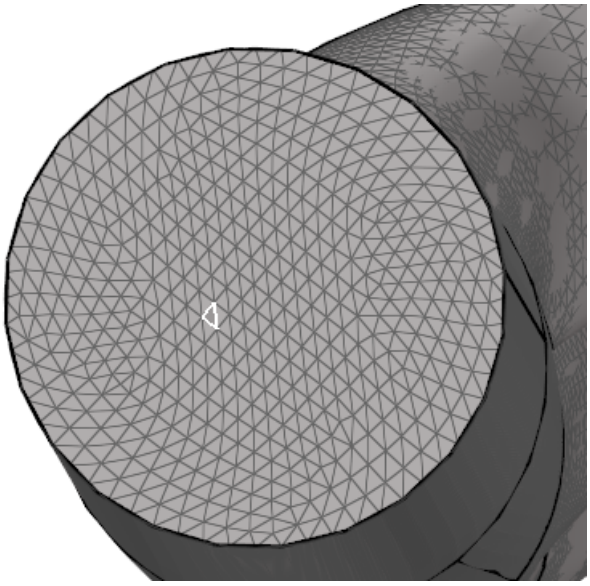


Figure 147:

8. In the panel area, click the **elems** collector and select the **by face** option.
   All the elements on the inlet surface are selected in the modeling window.
9. Click **dest component =** and select **Inflow**.
10. Click **move**.
    All the inlet surface mesh elements are colored in the Inflow component color.
11. Similarly, select a mesh element on each of the Outflow surfaces then click on the **elems** collector and select the **by face** option. Verify that all the surface elements on the three outlet surfaces are now highlighted then set the **dest component =** to **Outflow** and click **move**.
    The model should now look similar to the figure below.

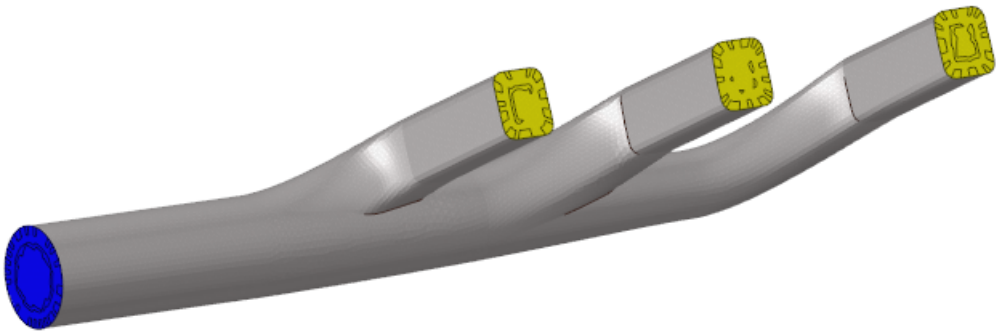


Figure 148:

12. Click **return** to exit the panel.

Set up the Volume Mesh Controls

1. Go to the **MeshControls** tab and right-click on **Volume Mesh**. From the context menu that appears, select **Create** > **Model** > **BL + Tetra**.

   Selecting BL + Tetra will show options for the boundary layer specification in addition to tetra volume meshing options.
2. In the Entity Editor, set the entity name to `Volume_Mesh_Control`.
3. Under the Entity Selection group, click in the value field next to Entities then click the **Components** collector.

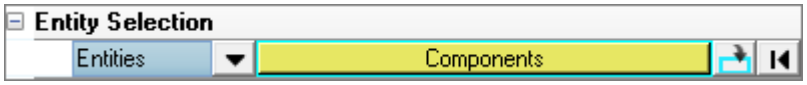


Figure 149:

The **Select Components** dialog opens.

4. Select all three components in the dialog and click **OK**.

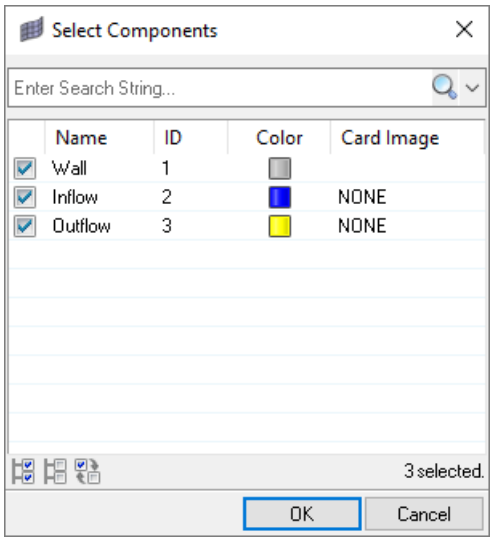


Figure 150:

You can click the icon in the dialog to quickly select all of the components.

5. Click **OK** to close the dialog.
6. Expand the **Boundary Layer** group and set the boundary layer parameters as follows:
   a) Change the Method to **Advanced**
   b) Set First Layer Thickness to `0.0005`
   c) Select **Acceleration** as the BL Growth Rate Method.
   d) Set Initial Growth Rate to `1.3`
   e) Set the Number of Layers to `5`
   f) Change Hexa Transition Mode to **All Prism**

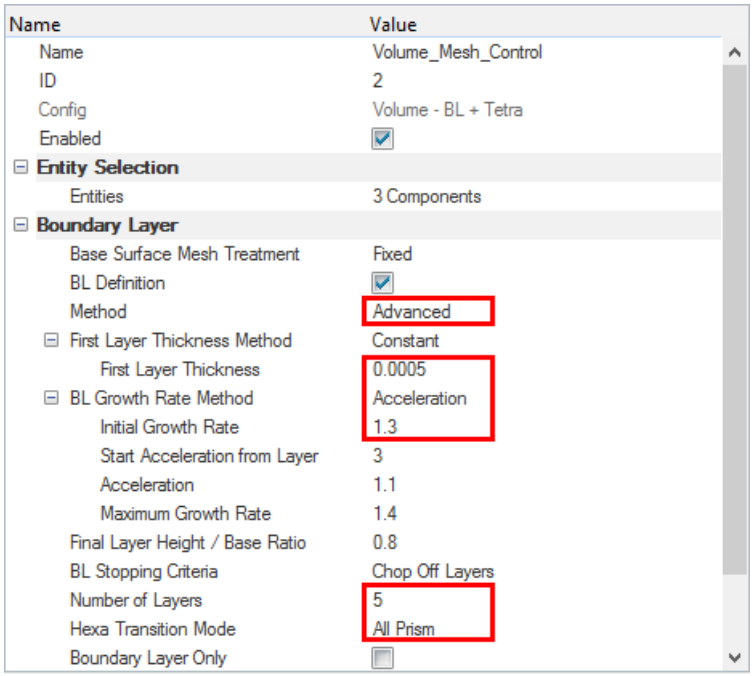


Figure 151:

When generating boundary layer meshes in HyperMesh, it is recommended to use All Prism as the boundary layer meshing mode for superior element quality. The prism elements can later be split into tetrahedral elements, which is the recommended element type for HyperMesh.

This completes the boundary layer mesh control. You will now add a local control for surfaces that do not require a boundary layer.

7. Right-click on **Volume Mesh** in the Mesh Controls Browser. From the context menu that appears, select **Create** > **Local** > **No BL**.
8. Optional: In the Entity Editor, set the entity name to`no_BL_control`.
9. Under the Entity Selection group, click in the value field next to Entities then click the **Components** collector.

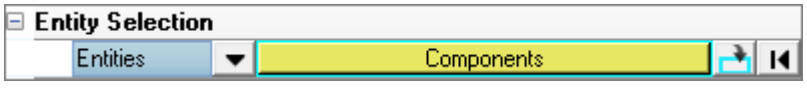


Figure 152:

The **Select Components** dialog opens.

10. Select **Inflow** and **Outflow** from the list and click **OK**.

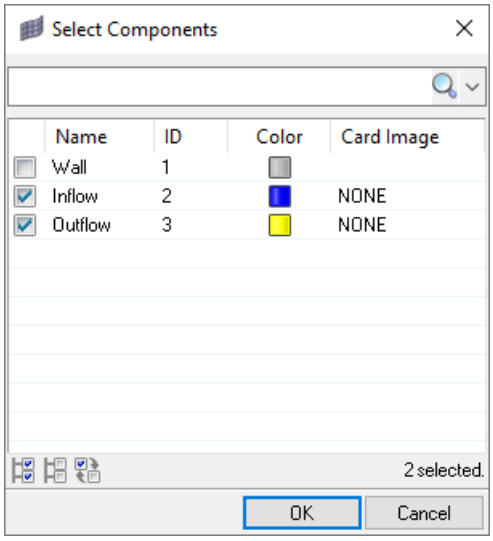


Figure 153:

11. Expand the **Boundary Layer** group and set Base Surface Mesh Treatment to **Float**.

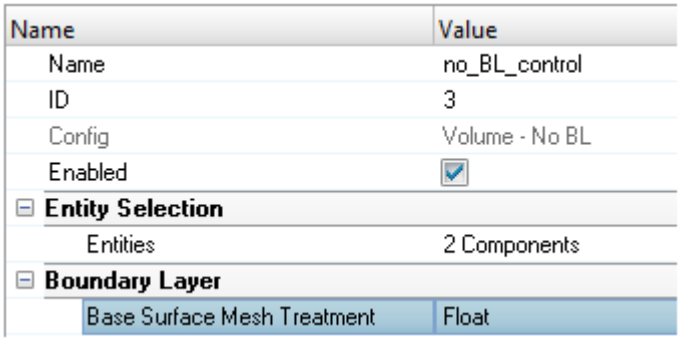


Figure 154:

12. Finally, add a volume selector control to put the boundary layer and the core tetra mesh in the same component.

a) Right-click on **Volume Mesh** in the Mesh Controls Browser. From the context menu that appears, select **Create** > **Volume Selector**.

b) In the Entity Editor, activate the check box for **BL and Tetras in One Component**.

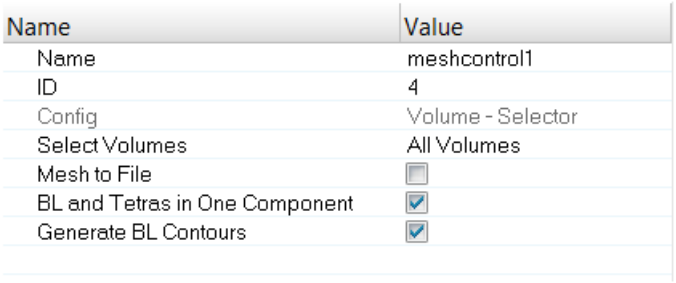


Figure 155:

Generate the Volume Mesh

In the previous steps, you created some model and local mesh controls. Your Mesh Controls Browser should look like the figure below.

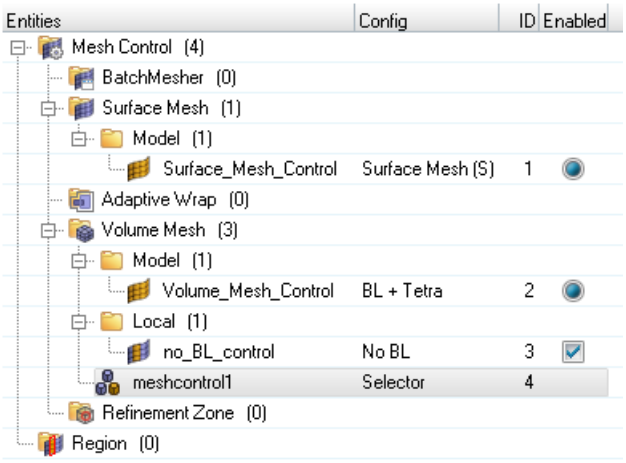


Figure 156:

When you set up the mesh controls, at least one active model control should be present before you generate the mesh. You can create multiple model controls, but only one model control can be active at a time. Surface and volume mesh however have different mesh controls.

Local controls are optional. You can create multiple local mesh controls, however only the ones which are selected at the time of mesh generation will be applied.

1. In the Mesh Controls Browser, make sure that both the Model and Local volume mesh controls are enabled.
2. Right-click on **Volume Mesh** and click **Mesh**.

   The generated volume mesh is placed in a single collector called CFD_tetcore001 under the list of components. This collector will be visible in the Model Browser. Once the meshing is complete, observe the mesh using the visualization controls.

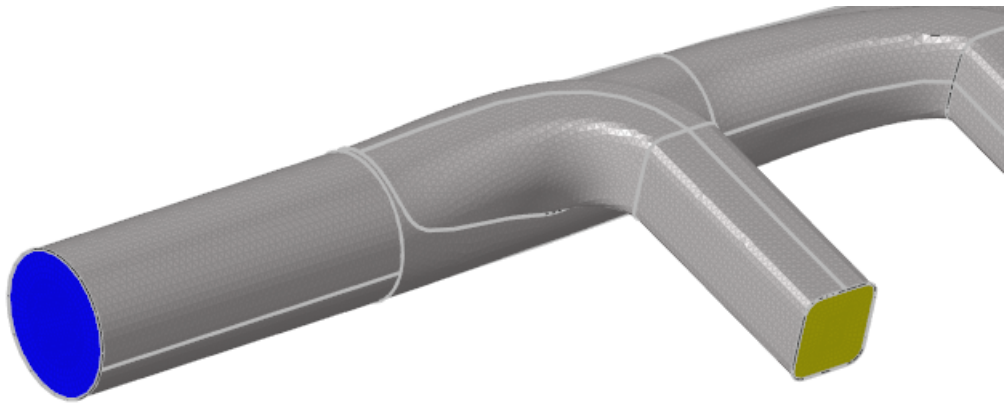


Figure 157:

You can turn off the surface display to view the mesh more clearly. On the Visualization toolbar, click the icon to display the geometry as wire frame. This will turn off the surface display. To turn on the surface display, click the icon. Zoom in to observe the boundary layer generated.

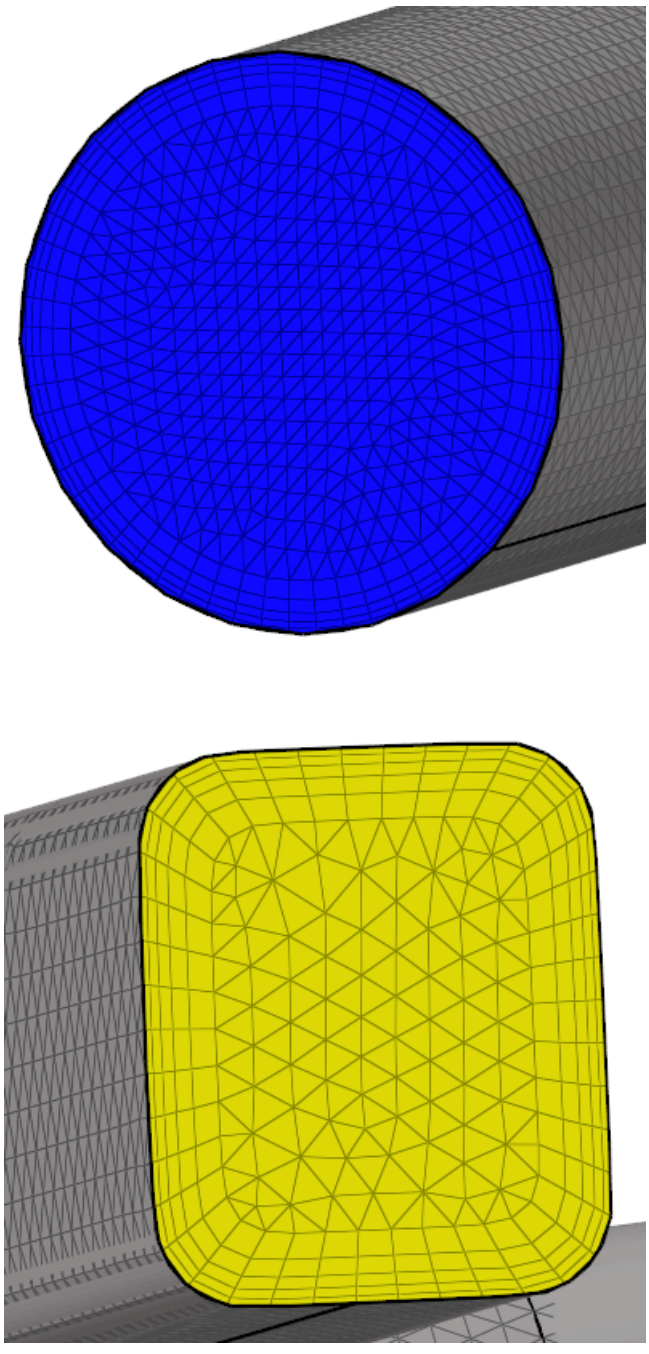


Figure 158:

3. In the Model Browser rename the collector CFD_tetcore001 to `Fluid`. In the Entity Editor, change the Type to **FLUID.**

   This collector will hold all of the 3D volume elements.

The mesh generated has prismatic elements in the boundary layer. These elements will be split to create tetrahedral elements.

4. Click **Mesh** > **Edit** > **Elements** > **Split Elements**.
   The Split Elements panel opens.
5. Select the **solid elements** sub-panel.
6. Click the **elems** collector and select **all**.
7. Change the split pattern to **split into tetras**.
   This is the optimized mode for splitting elements into tetras for a CFD simulation.
8. Click **split**.
   Observe the mesh after the splitting process is complete.

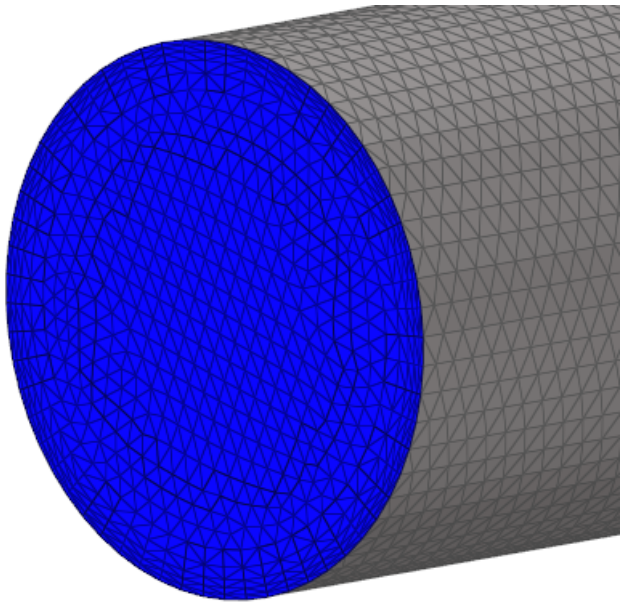


Figure 159:

9. Click **return** to exit the panel.

Set up Simulation Parameters for AcuSolve

The next step after creating the mesh is to set up the simulation parameters. You will use the Solver Browser for this purpose. The Solver Browser provides a solver perspective view of the model structure in flat, listed tree structure.

Set General Simulation Parameters

In next steps you will set parameters that apply globally to the simulation.

1. Click **View** > **Solver Browser** to open the Solver Browser.

The Solver Browser lists every entity mapped to the active solver profile within the session and places those entities into their respective entity group folders.

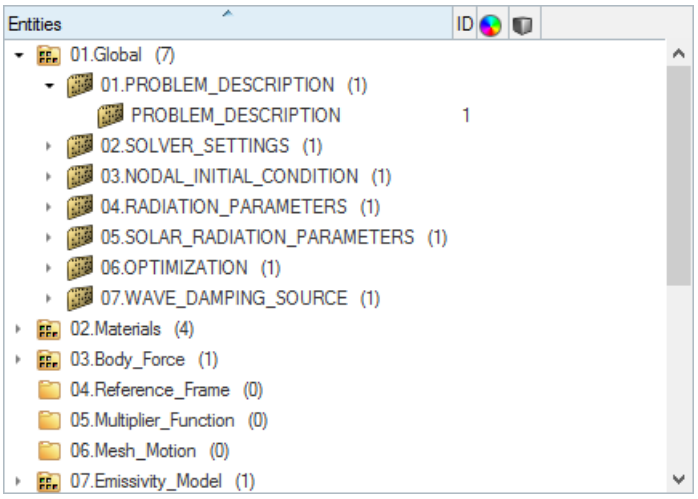


Figure 160:

2. Expand **01.Global** and then expand **01.PROBLEM_DESCRIPTION**.
3. Click **PROBLEM_DESCRIPTION** to open the Entity Editor.
4. Type `Manifold` for the Title.
5. Change the Turbulence model from Laminar to **Spalart Allmaras**.

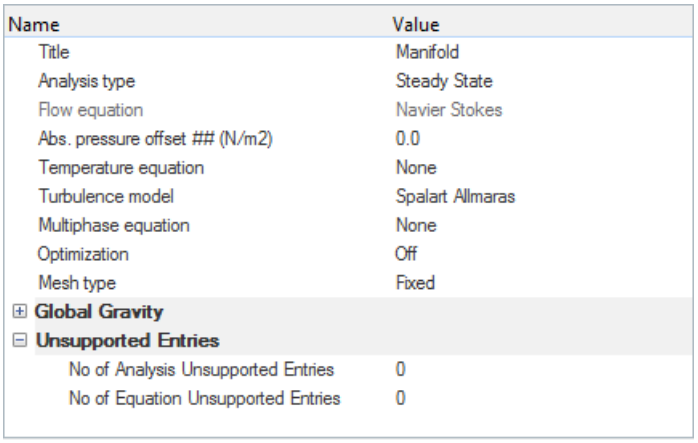


Figure 161:

Specify the Solver Settings

1. In the Solver Browser, expand the **02.SOLVER_SETTINGS** group then click **SOLVER_SETTINGS** to open it in the Entity Editor.
2. Verify that the Convergence tolerance is set to `0.001`.
3. Change the Relaxation factor to `0.4`.

4. Check that Flow and Turbulence are set to **On**.

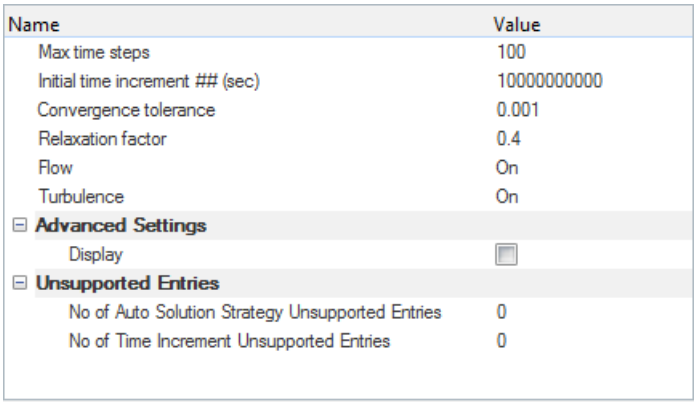


Figure 162:

Set Nodal Initial Conditions

1. In the Solver Browser, expand the **03.Nodal_Initial_Condition** group then click **Nodal_Initial_Condition** to open it in the Entity Editor.
2. Set the X velocity to `2` m/sec.
3. Set the Eddy viscosity to `1e-05` m^2/sec.

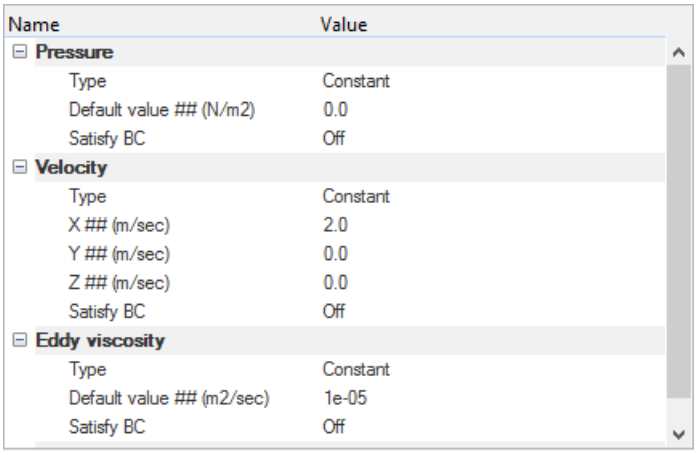


Figure 163:

Apply Volume Parameters

Volume groups are containers used for storing information about a volume region. This information includes solution and meshing parameters applied to the volume and the geometric regions that these settings are applied to.

There is one volume collector in this model, fluid. In the next steps you will set the material properties for it.

1. In the Solver Browser, expand **11.Volumes** then expand the **FLUID** volume group and click **Fluid**.
   The fluid collector entity opens in the Entity Editor.
2. Click in the value field for Material (it is Unspecified by default).
3. Click **Material**.
   The **Select Material** dialog opens.
4. Select **Water_HM** and click **OK**.

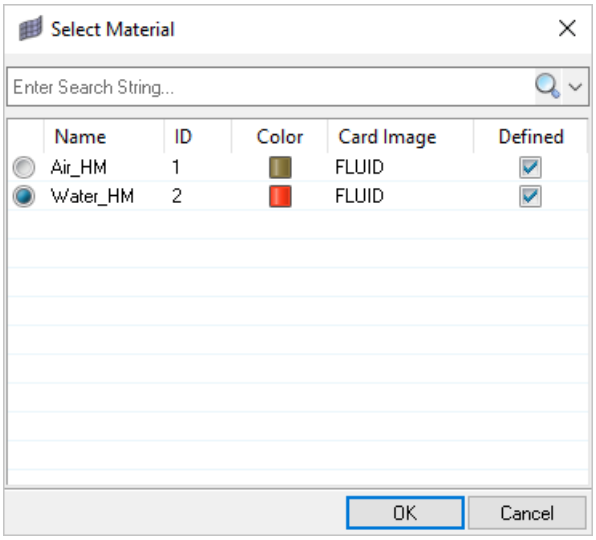


Figure 164:

Apply Surface Parameters

Surface groups are containers used for storing information about a surface, including solution and meshing parameters, and the corresponding surface in the geometry that the parameters will apply to.

1. In the Solver Browser, expand **12.Surfaces** then expand the **WALL** surface group. Click **Wall** to open it in the Entity Editor. Verify that the Type is set to **WALL.**

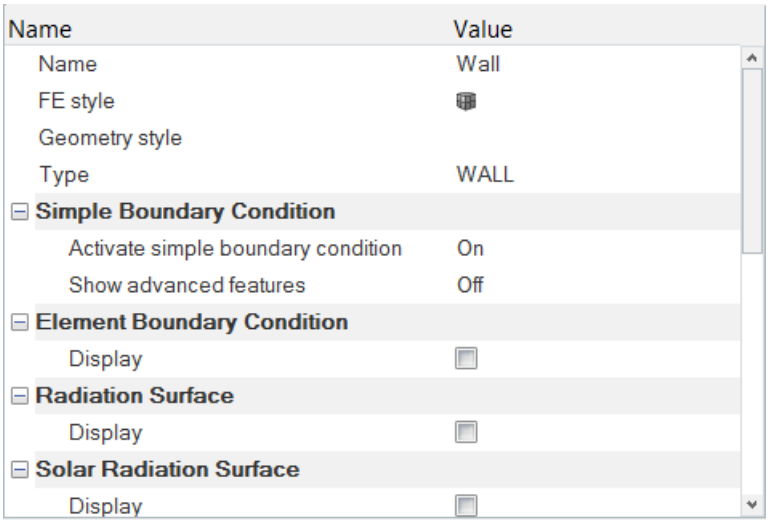


Figure 165:

2. Expand the **OUTFLOW** surface group then click **Outflow** to open it in the Entity Editor. Verify that the Type is set to **OUTFLOW**.

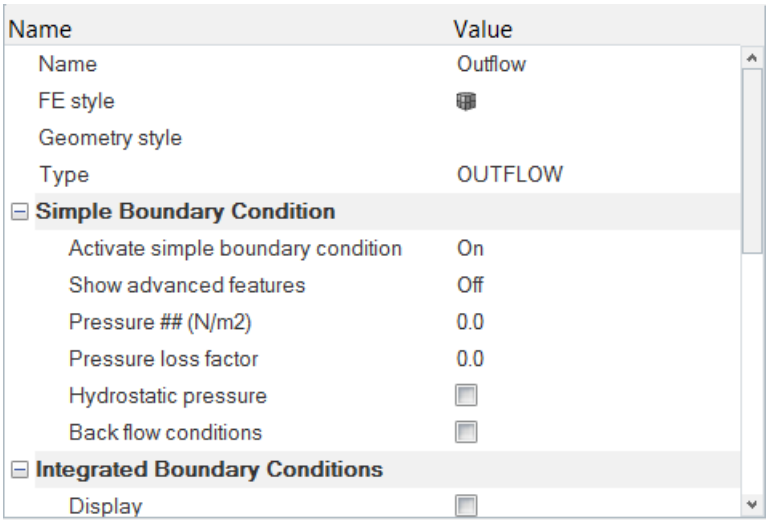


Figure 166:

3. Expand the **INFLOW** surface group then click **Inflow** to open it in the Entity Editor. Verify that the Type is set to **INFLOW**. Set Inflow type to **Average velocity**. Set the Average velocity to 2 m/sec.

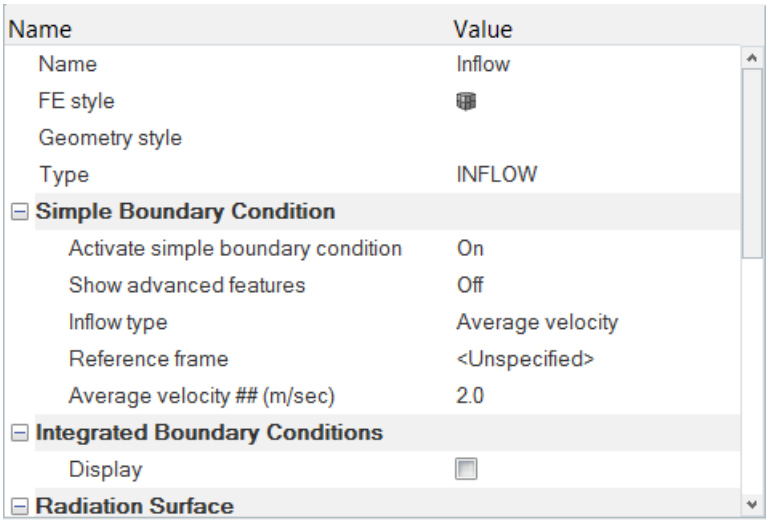


Figure 167:

4. Save the model.

Compute the Solution and Review the Results

Run AcuSolve

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

1. In the Model Browser, ensure that the visibility of the mesh for all collectors to be exported to AcuSolve is activated. In this case, Fluid, Wall, Inflow and Outflow should be activated.

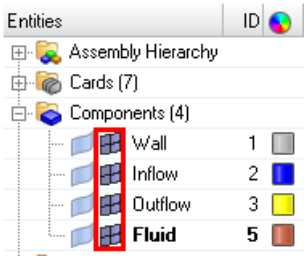


Figure 168:

The display of the mesh icon beside the component name indicates that the visibility of mesh for that component is on. The display of the mesh of a component can be turned on/off by clicking on that icon.

2. Click on the ACU toolbar.

The **Solver job Launcher** dialog opens.

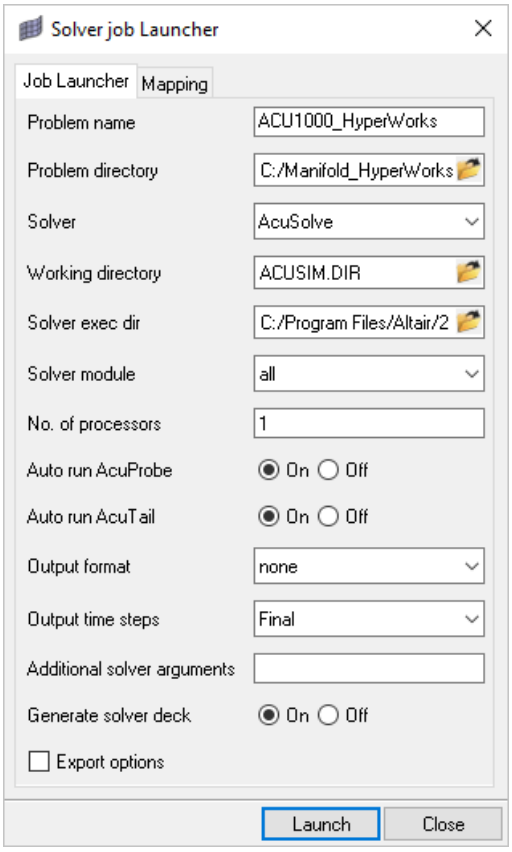


Figure 169:

For this case, the default settings will be used. You may choose to change the number of processors to allow AcuSolve to run using more processors (4 or 8), if available. HyperMesh will generate the required solver input files and launch AcuSolve. AcuSolve will calculate the steady state solution for this problem.

3. Verify that Auto run AcuProbe is **On**.

   This will open an **AcuProbe** dialog which will let you monitor the solution progress.

4. Click **Launch** to start the solution process.
   As the solution progresses, an **AcuTail** and an **AcuProbe** dialog will open. Solution progress is reported in the **AcuTail** dialog. An **AcuSolve Control** dialog will also open from which you can control the solution process. In this dialog you have options to stop the solution or generate the output files at the end of the current time step.

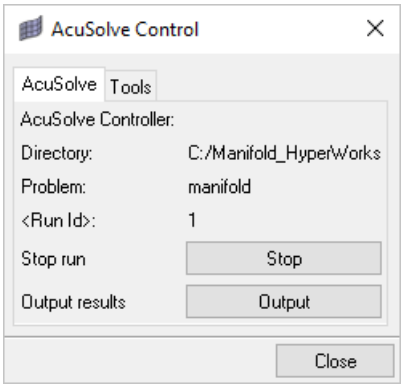


Figure 170:

A summary of the run printed in the **AcuTail** dialog indicates that AcuSolve has finished running the solution.

```
acuSolve: ---------- End Time Step ----------
acuSolve: Input      CPU/Elapse time= 1.734000e+000 4.766000e+000 Sec
acuSolve: Output     CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Exchange   CPU/Elapse time= 3.270000e-001 1.450000e-001 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =         25         25
acuSolve:   Elem.Form CPU/Elapse time= 1.541420e+002 1.674610e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 2.587020e+002 2.787000e+002 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:   No of res/lhs formations =         25         25
acuSolve:   Elem.Form CPU/Elapse time= 9.845200e+001 1.060840e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 2.868700e+001 3.140400e+001 Sec
acuSolve: Total CPU/Elapse time      = 7.128430e+002 7.770980e+002 Sec
acuSolve: Total Memory Usage         = 3.675352e+002 Mbytes
acuRun: ************************************.
```

Figure 171:

Monitor the Solution with AcuProbe

AcuProbe can be used to monitor various variables over solution time.

1. In the AcuProbe dialog, expand **Residual Ratio**.
2. Right-click on **Final** and select **Plot All**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

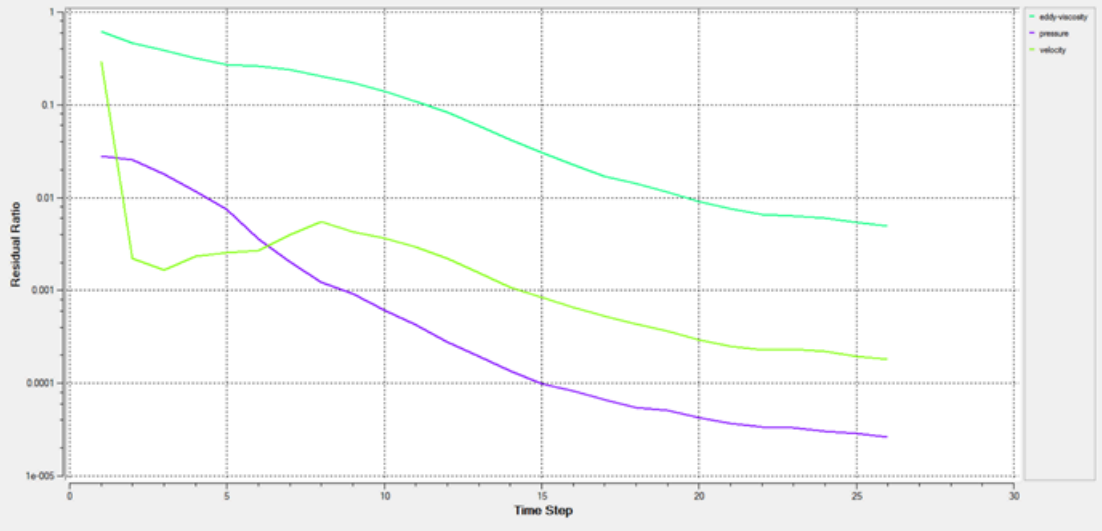


Figure 172:

The plot above shows the residuals of the equations as the solution progresses through each time step. You can see the residuals dropping smoothly. Once the pressure and velocity residual ratios reach a value less than the specified convergence tolerance (0.001), the solution is considered to be converged. By default, the eddy viscosity convergence tolerance is set to a magnitude of one order higher than the specified convergence tolerance (0.01).

3. You can also save the plots as an image.
   a) From the AcuProbe dialog, click **File** > **Save**.
   b) Enter a name for the image and click **Save**.
4. The time series data of the variables can also be exported as a text file for further post-processing.
   a) Right-click on the variable that you want to export and click **Export**.
   b) Enter a File name and choose .txt for the Save as type.
   c) Click **Save**.

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click  next to Load model.

3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `ACU1000_HyperWorks.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.
   The model is colored by geometry after loading.

Apply Pressure Contours on the Boundary Surfaces

1. Click on the Results toolbar to open the Contour panel.
2. In the Contour panel, select **Pressure (s)** as the Result type.
3. Click **Apply**.

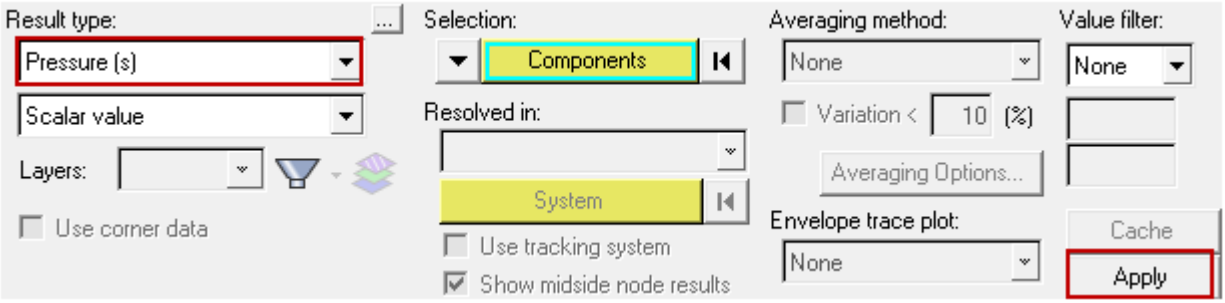


Figure 173:

4. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 174:

5. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.
   The pressure contour should be displayed as shown in the figure below.

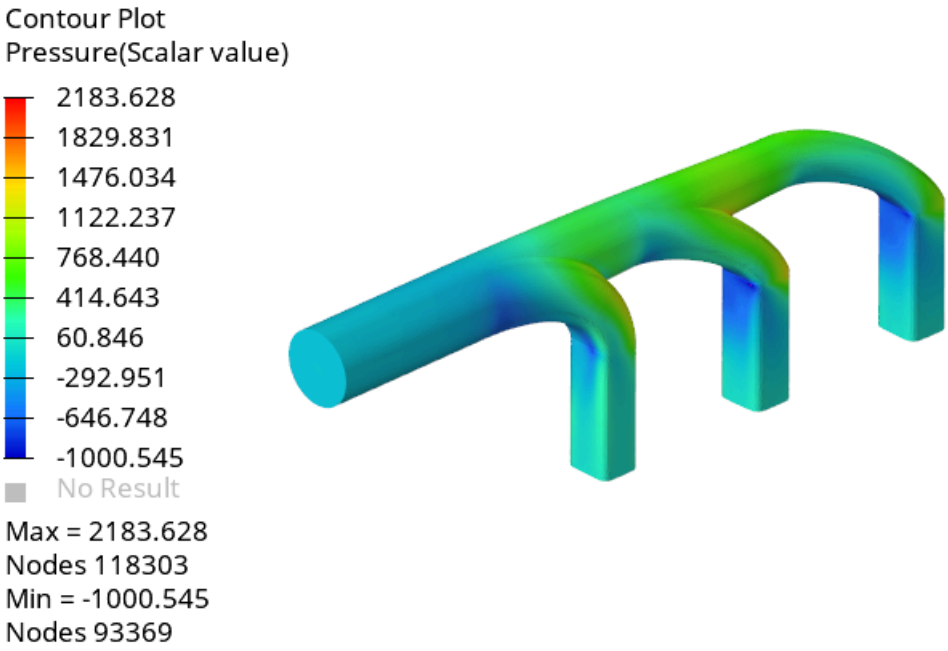


Figure 175:

Save Plots as Image Files

1. On the Image Capture toolbar toggle the / icons so that it shows the icon to save to file.
2. Click the icon on the Image Capture toolbar.
3. Provide a name for the image in the dialog and click **Save**.
   If you want to use the image in a presentation you can copy them to the clipboard by toggling the Save Image to File/Clipboard icon to instead of . Then paste the image in your presentation.

Create Pressure and Velocity Contours on a Cut Plane

1. To create a new cut plane, right-click in the Results Browser and select **Create** > **Section Cut** > **Planar** from the context menu.
   A new entity, Section 1, is created in the Results Browser.
2. Right-click **Section 1** and select **Edit** from the context menu.
3. In the Section Cut panel verify that Define plane is set to **Y Axis**.
4. Under the Display options, activate the **Cross section** check box.
5. Verify that the Clip elements check box is activated.
6. For the Base coordinates, enter a value of `-0.015` for the Y-coordinate and press **Enter**.

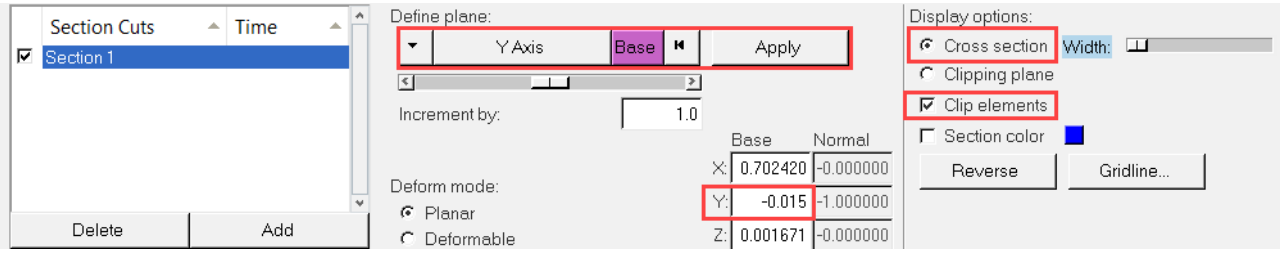


Figure 176:

7. Click on **Gridline** in the panel area.
8. In the dialog, uncheck the **Show** option under Gridline then click **OK**.

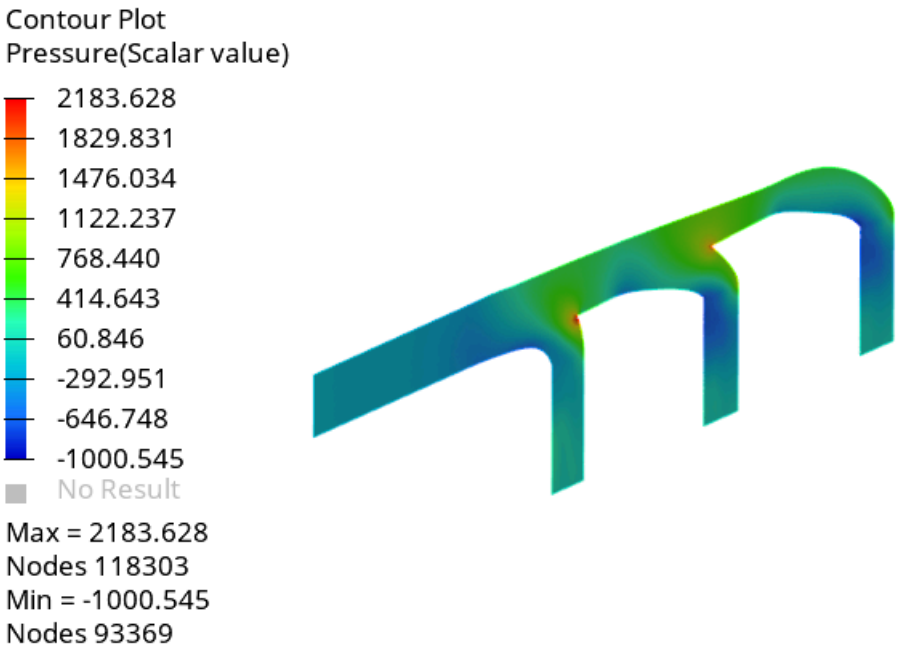


Figure 177:

9. Click on the Results toolbar to open the Contour panel.
10. In the Contour panel select **Velocity** as the Result type.
11. Click **Apply**.

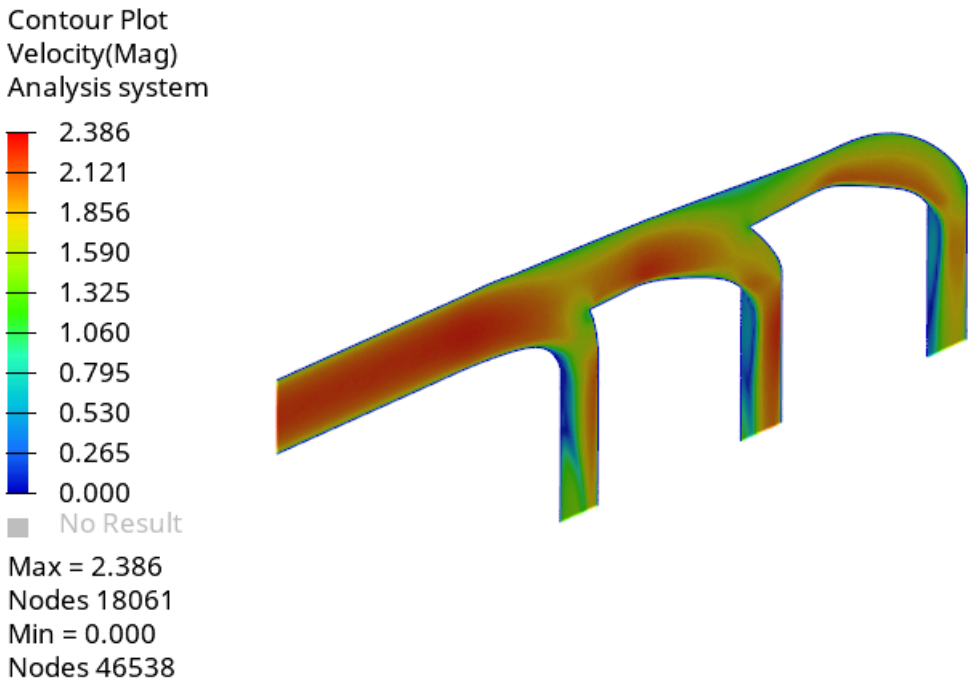


Figure 178:

Create a Clipping Plane

The section cut plane can be used as a clipping plane as well. In this step you will create a clipping plane.

1. Right-click **Section 1** under Section Cuts in the Results Browser and select **Edit** from the context menu.
2. In the Section Cut panel change the selection under Display options from Cross section to **Clipping plane**.

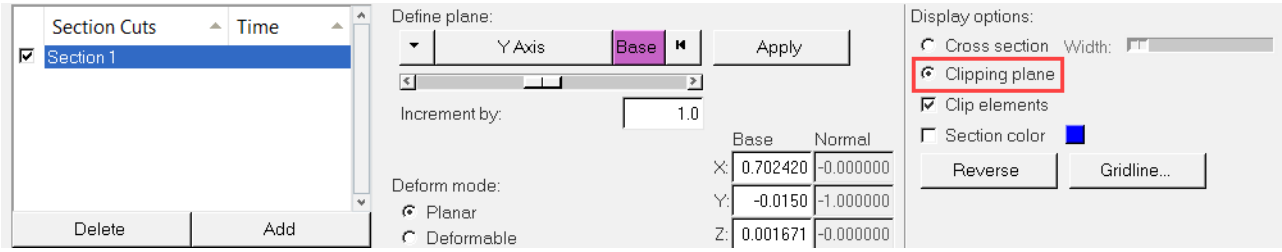


Figure 179:

3. Click **Reverse** to toggle the clipping direction to your choosing.

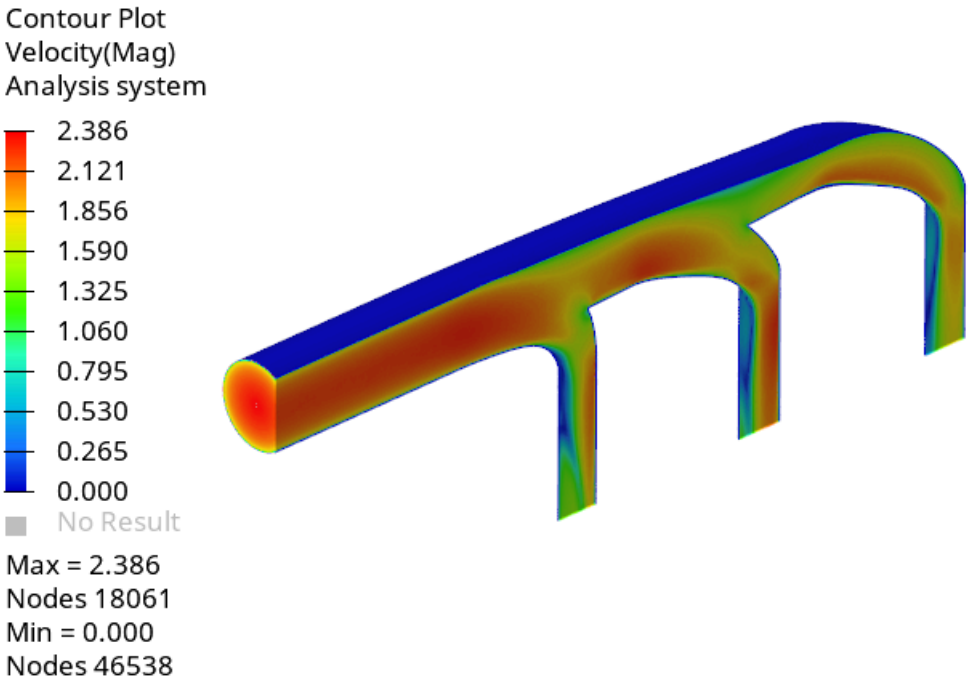


Figure 180:

Create Velocity Vectors

1. In the Section Cut panel under Display options, set the selection back to **Cross section**.
2. Click the  icon on the Results toolbar.
3. On the Vector panel, make sure that the Result type is set to **Velocity (v)**.
4. Set the Selection mode to **Sections** using from the drop-down menu.
5. Click the **Sections** collector to open the **Extended Entity Selection** dialog.
6. Click **Displayed**.



Figure 181:

7. Select the **Z+X** Resultant option.

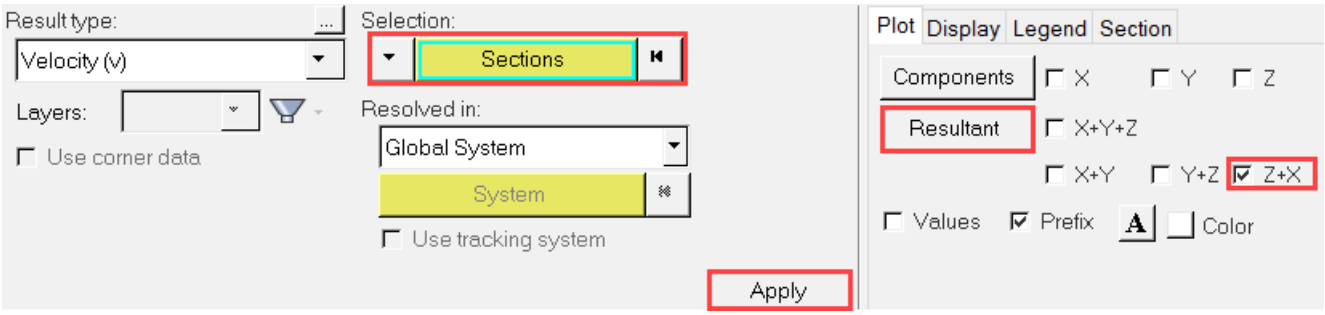


Figure 182:

8. Click **Apply**.
9. Click the **Display** tab and set the options as shown in the figure below.

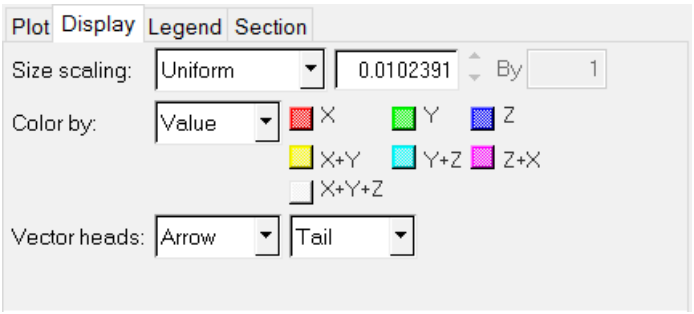


Figure 183:

10. Click the **Section** tab and activate the **Projected** and **Evenly distributed** check boxes.
11. Set the Number of rows and columns to `20` and `50` respectively then click **Apply**.

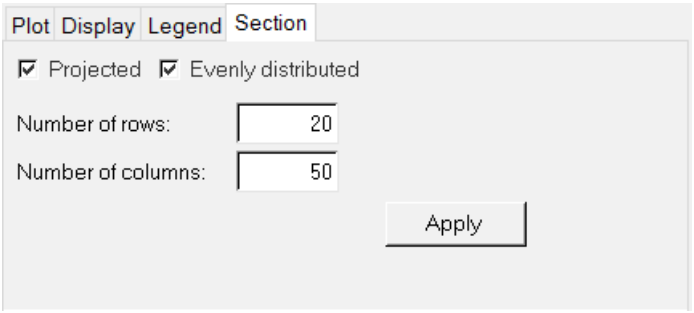


Figure 184:

The vector plot should be displayed as shown in the figure below.

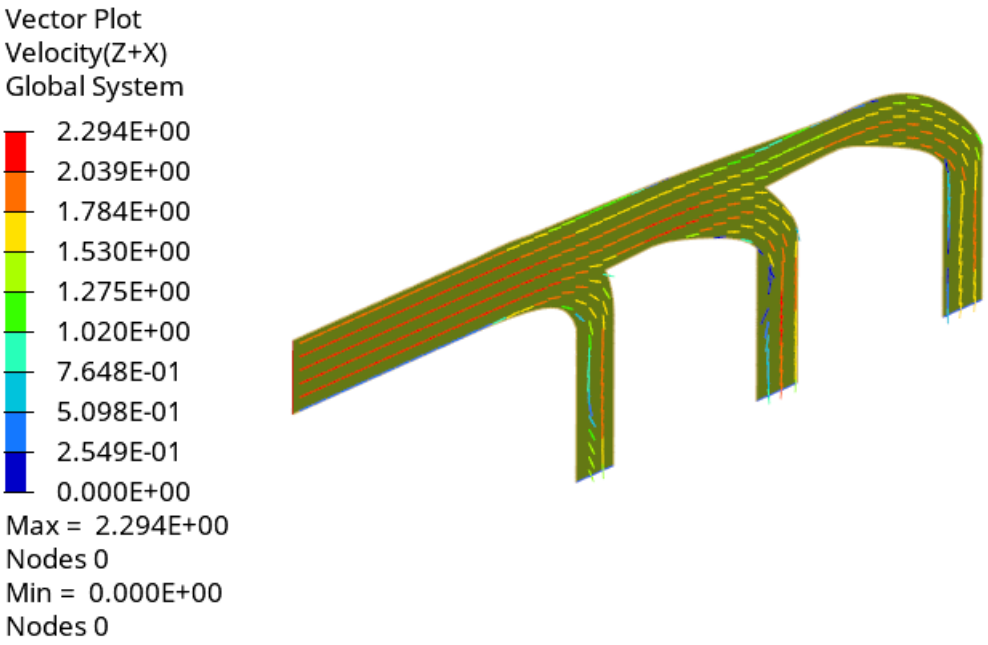


Figure 185:

Display Streamlines

1. In the Results Browser, expand the **Section Cuts** folder.
2. Click the icon next to Section 1 to turn off its display.
3. In the Results Browser, turn off the display for all components except Inflow and Outflow.
4. Click the icon on the Results toolbar to open the Streamlines panel.
5. Click **Add** to add a new set of streamlines.
6. Set the Rake type to **Line**, if not already selected.
7. Click the ... icon.
   The **Reference point** dialog opens.
8. Enter the reference points as shown in the figure below.

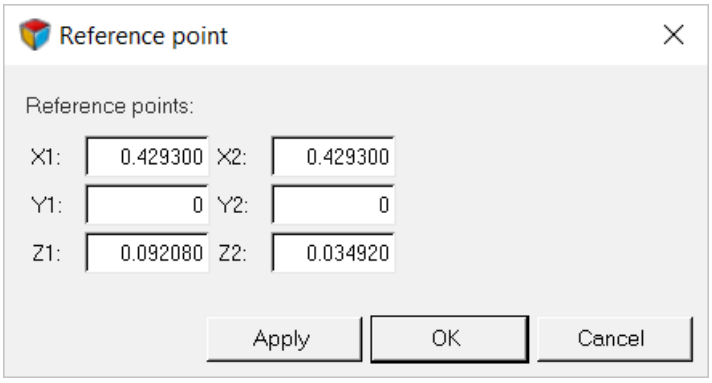


Figure 186:

9. In the panel area, set the Integration mode to **Downstream** and the Number of seeds to `20`.

10. Make sure that the Source is set to **Velocity**.

11. Click **Create Streamlines**.

12. Enter the Streamline Size as `3` and press Enter .

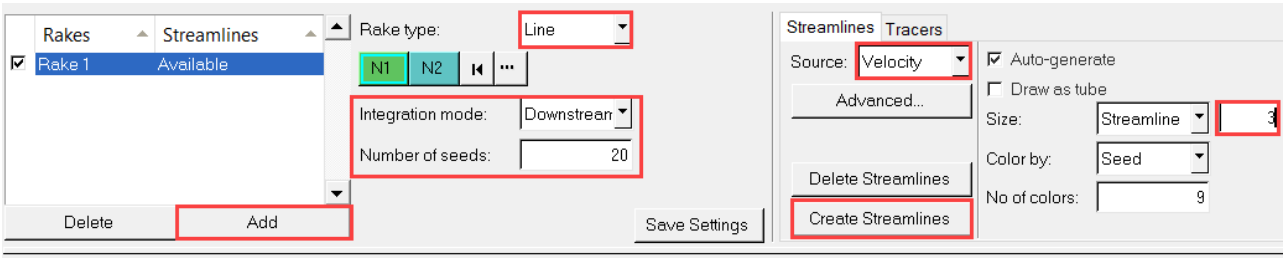


Figure 187:

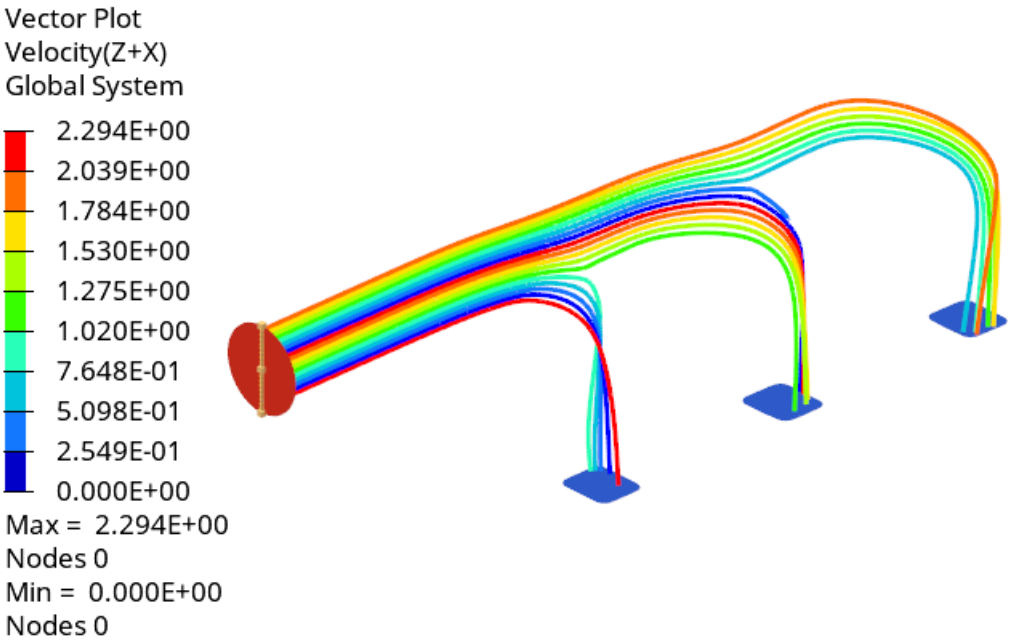


Figure 188:

Summary

In this tutorial, you worked through a basic workflow to carry out a CFD simulation and post-processed the results using HyperWorks products, namely HyperMesh and HyperView. You started by importing and meshing the model in HyperMesh. You also set up the model and launchedAcuSolve directly from within HyperMesh. Upon completion of the solution by AcuSolve, you used HyperView to post-process the results. You learned how to create contours on the boundary surfaces and the section cuts, velocity vectors, and streamlines.

2.2 Turbulence

2.2.1 ACU-T: 2000 Turbulent Flow in a Mixing Elbow

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T2000_MixingElbow.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 1. This is a typical industrial example for mixing in a pipe by injecting high-velocity fluid from a small inlet into relatively low-velocity fluid in the main pipe. It consists of a 90° mixing elbow with water entering through two inlets with different velocities. The geometry is symmetric about the XY midplane of the pipe, as shown in the figure

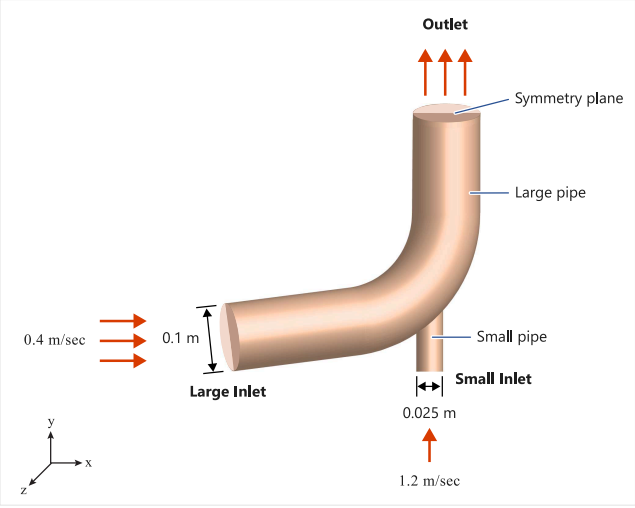


Figure 189: Schematic of Mixing Elbow

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon  located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T2000_MixingElbow.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `MixingElbow_Turbulent` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `MixingElbow` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, set the Turbulence Model to **Spalart Allmaras**.

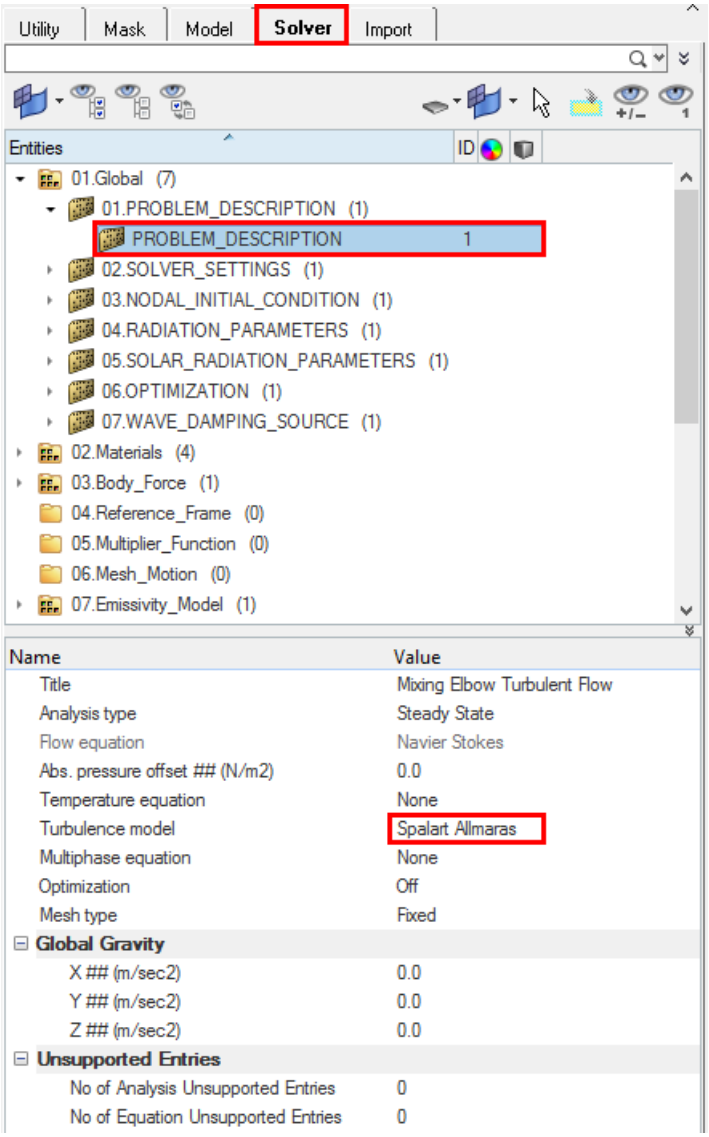


Figure 190:

Set Up Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Large_Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow type to **Average velocity**.
   c) Set the Average velocity to `0.4` m/s.

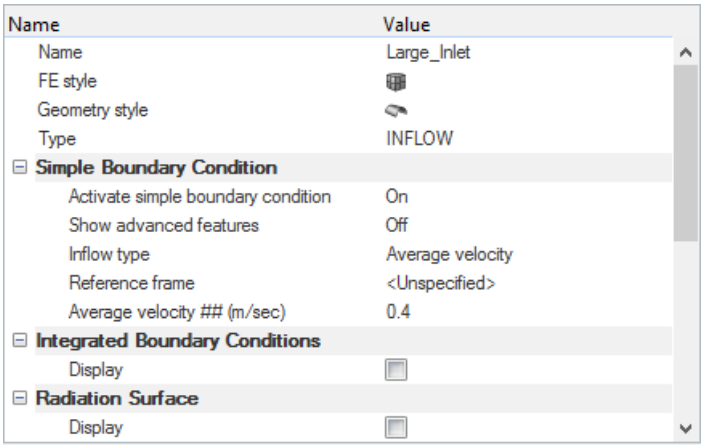


Figure 191:

3. Click **Small_Inlet**. In the Entity Editor,
   a) Change the type to **INFLOW**.
   b) Set the Inflow type to **Average velocity**.
   c) Set the Average velocity to `1.2` m/s.
4. Click **Outlet**. In the Entity Editor, change the Type to **OUTFLOW**.

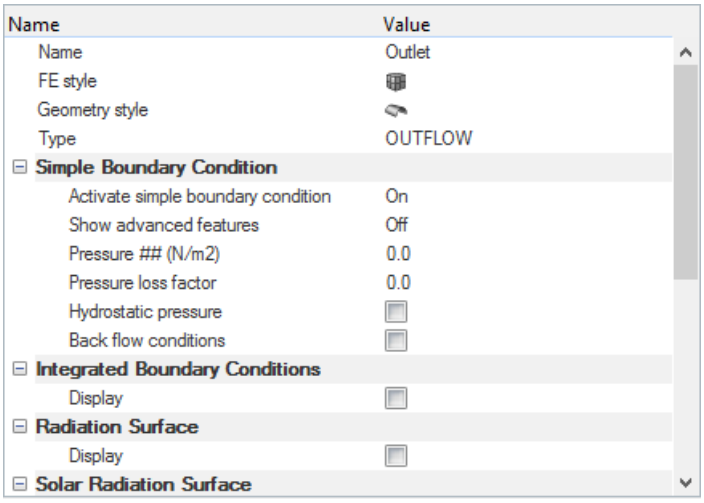


Figure 192:

5. Click **Symmetry**. In the Entity Editor, change the Type to **SYMMETRY**.

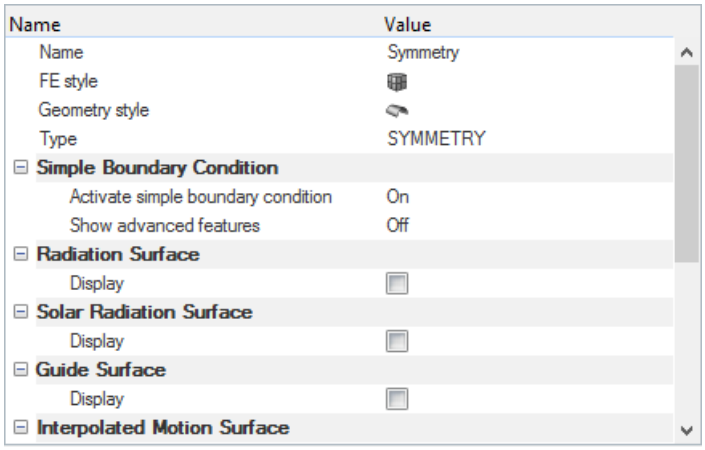


Figure 193:

6. Click **Wall**. In the Entity Editor, verify that the Type is set to **WALL**.

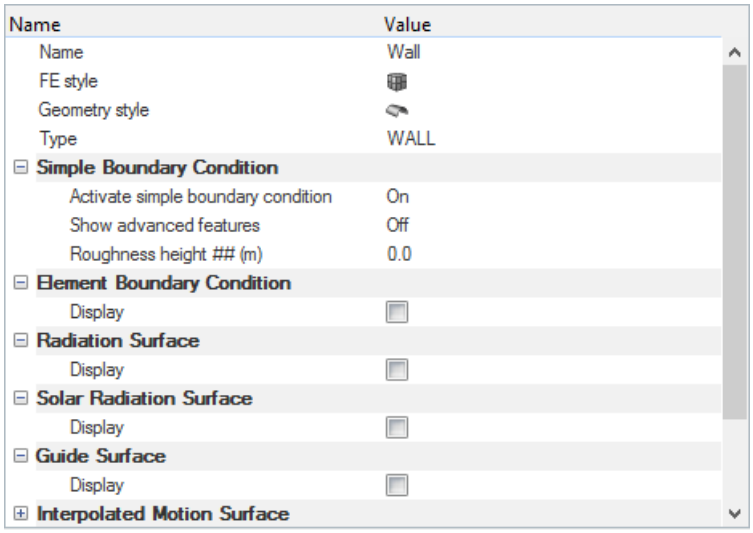


Figure 194:

7. Click **Fluid**. In the Entity Editor,

a) Change the Type to **FLUID**.

b) Select **Water_HM** as the Material.

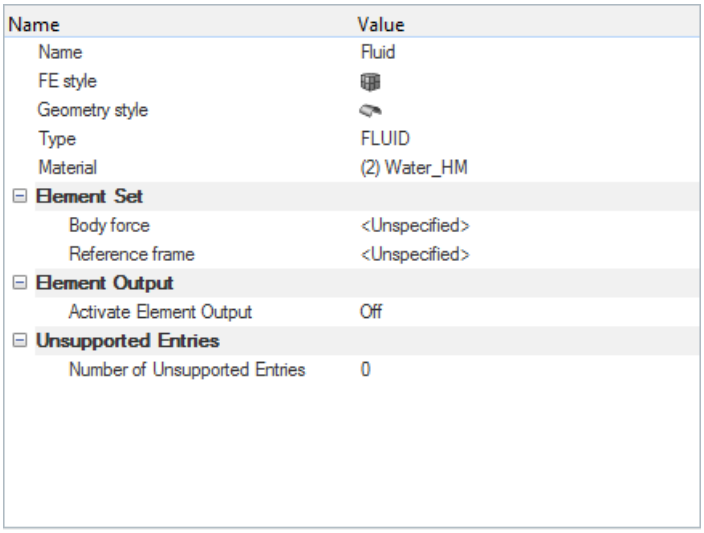


Figure 195:

8. Save the model.

Compute the Solution

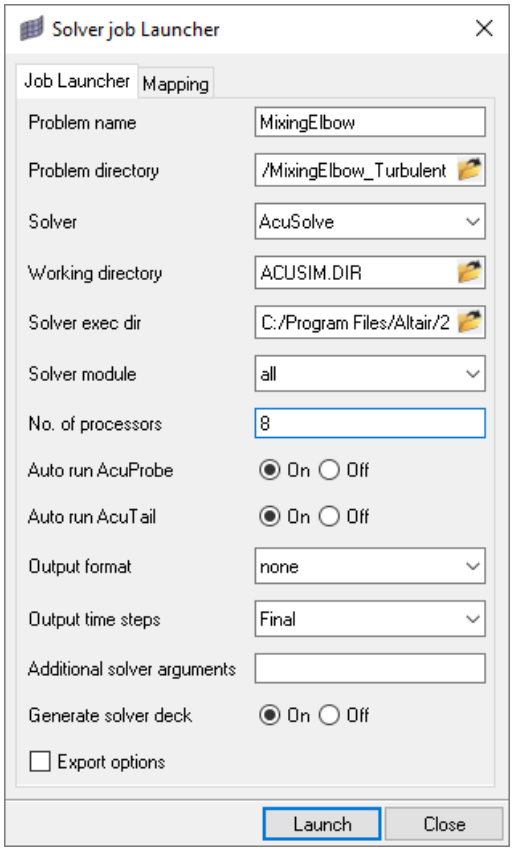
In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.



Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `MixingElbow.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create Contour Plots of Pressure and Velocity

In this step, you will create pressure and velocity contour plots on the symmetry plane.

1. In the Results Browser, expand the list of **Components**.
2. Click the Isolate Shown icon then click the **Symmetry** component to turn off the display of all components in the graphics window except the Symmetry component.

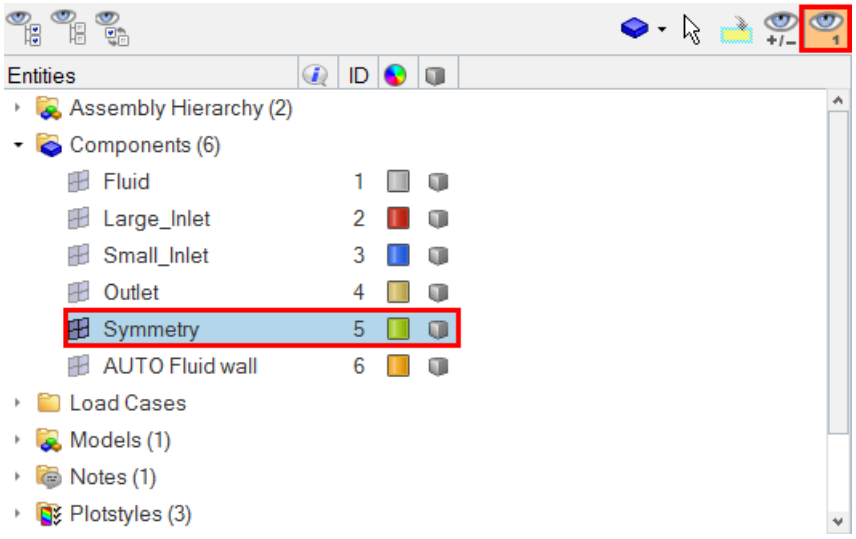


Figure 196:

3. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
4. Click on the Results toolbar to open the Contour panel.
5. Under result type, verify that **Velocity(v)** and **Mag** are selected.
6. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
7. Click **Apply**.
8. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 197:

9. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.

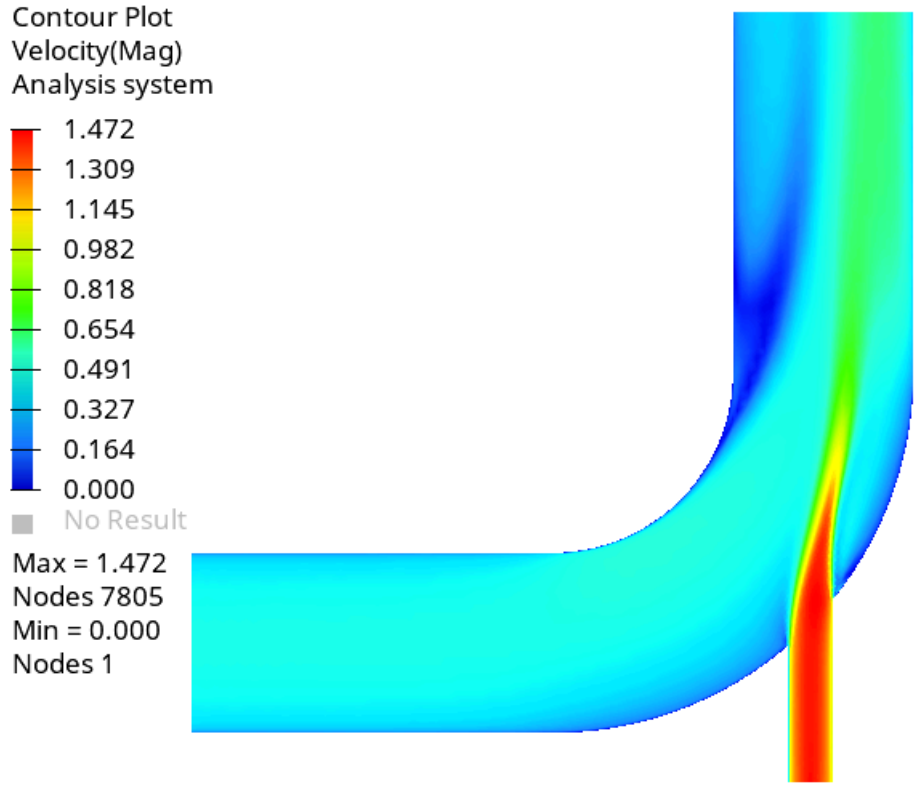


Figure 198:

10. Change the result type to **Pressure(s)** then click **Apply** to view the pressure contour on the symmetry plane.

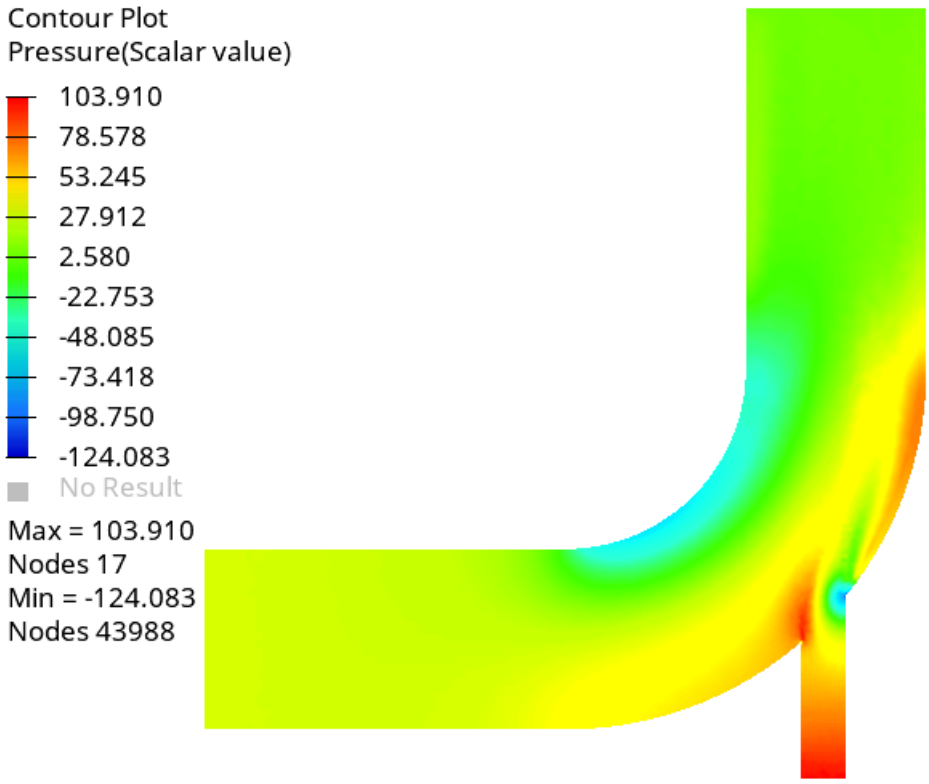


Figure 199:

Summary

In this tutorial, you worked through a basic workflow to set up a CFD model, carry out a CFD simulation, and post-process the results using HyperWorks products, namely AcuSolve, HyperMesh, and HyperView. You started by importing the model in HyperMesh. Then, you defined the simulation parameters and launched AcuSolve directly from within HyperMesh. Upon completion of the solution by AcuSolve, you used HyperView to post-process the results and create contour plots.

2.2.2 ACU-T: 2300 Atmospheric Boundary Layer Problem – Flow Over Building

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files`

`\tutorials\AcuSolve` to a local directory. Extract `ACU-T2300_Building.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 200. As an example, this problem shows the capability of Atmospheric Boundary Layer modelling in AcuSolve.

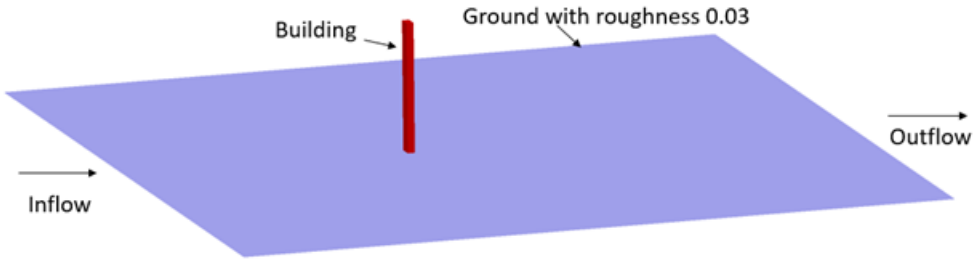


Figure 200:

In this tutorial, you will simulate the air flow over a building with a ground roughness of 0.03. In this case, **User Defined Atmospheric Roughness Type** is considered.

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.
   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.
   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T2300_Building.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Building_ABL` and navigate into this directory.
   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Building` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.

2. In the Entity Editor, enter `Building` as the Title.
3. Verify that the Analysis Type is set to **Steady State**.
4. Set the Turbulence Model to **Spalart Allmaras**.

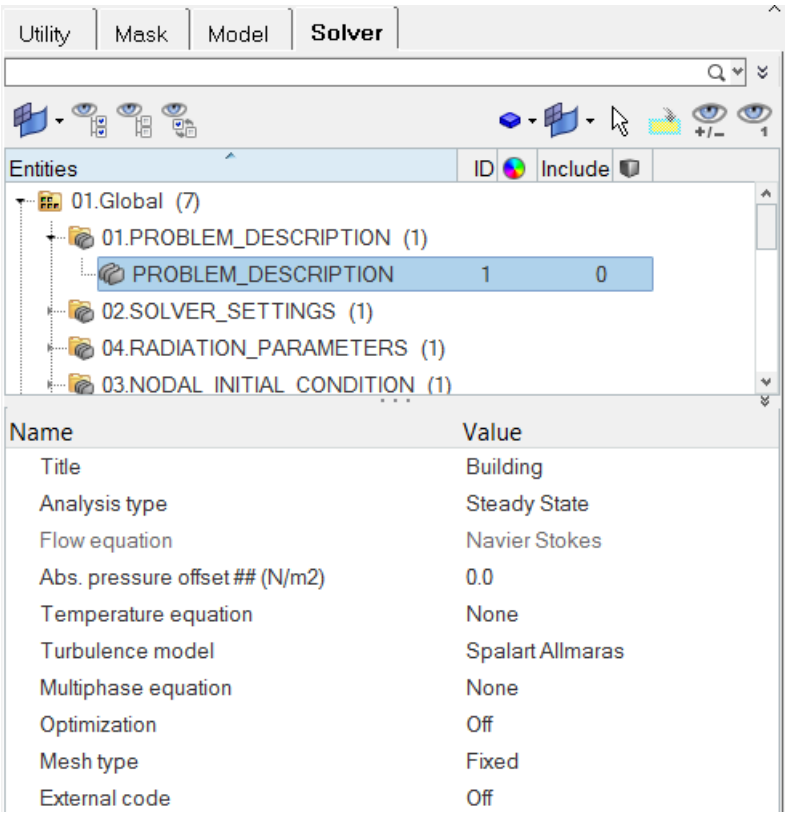


Figure 201:

5. In the Solver Browser, expand the **02.SOLVER_SETTINGS** group then click **SOLVER_SETTINGS** to open it in the Entity Editor.
6. Set Convergence Tolerance `0.001`.
7. Set the Relaxation Factor to `0.4`.

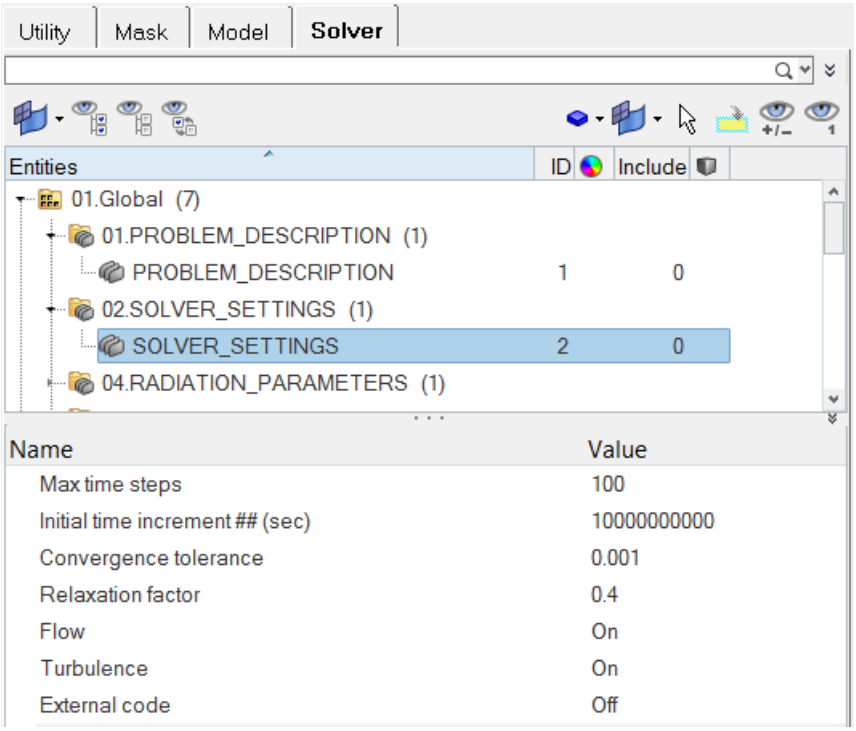


Figure 202:

Set Up Boundary Conditions and Material Model Parameters

In this step, you will define the Boundary Conditions (BCs) for the problem and assign material properties to the fluid volume.

Set Up Material Model Parameters

1. In the Solver Browser, expand **02.Materials** > **FLUID** then click on **Air_HM**.
2. Set the Material Type to **FLUID** if it's not already set. Leave the remaining default values as is.

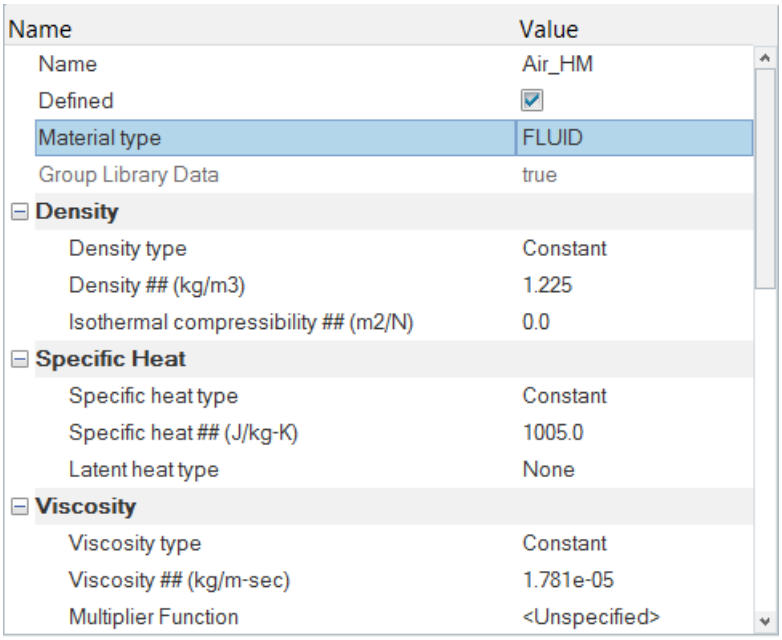


Figure 203:

Set Up Fluid Volume Material

1. In the Solver Browser, expand **11.Volumes** > **FLUID** then click on **fluid**.
2. Set the Type to **FLUID**.
3. Set the Material to **Air_HM**.

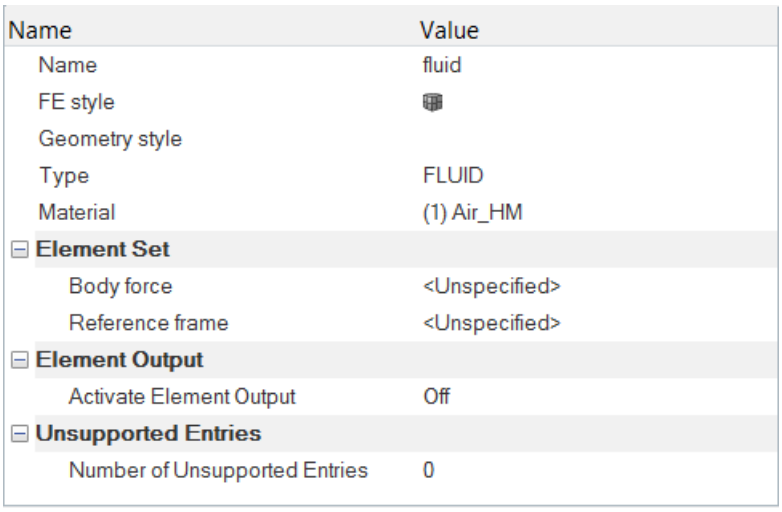


Figure 204:

Set Up Boundary Conditions

1. In the Solver Browser, expand **12.Surfaces** > **INFLOW** then click on **Inflow**. In the Entity Editor,
   a) Set the Type to **INFLOW**.
   b) Set the Inflow Type to **Atmospheric**.
   c) Set the Atmospheric Roughness Type to **User value**.
   d) Set the Atmospheric Ground Roughness to `0.001`.
   e) Set the Atmospheric Reference Vel Type to **Friction velocity**.
   f) Set the Atmospheric Friction Velocity to `0.106`.
   g) For **Atmospheric Ground Origin**, set the coordinates to `(0, 0, 0)`.
   h) For **Atmospheric Ground Normal Direction**, set the coordinates to `(0, 0, 1)`.
   i) For **Atmospheric Flow Direction**, set the coordinates to `(1, 0, 0)`.

Name	Value
Name	Inflow
FE style	
Geometry style	
Type	INFLOW
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Inflow type	Atmospheric
Atmospheric roughness type	User value
Atmospheric ground rougness	0.001
Atmospheric reference vel type	Friction velocity
Atmospheric friction velocity	0.106
Atmospheric Ground Origin	
x coordinate ## (m)	0.0
y coordinate ## (m)	0.0
z coordinate ## (m)	0.0
Atmospheric Ground Normal Direction	
x coordinate ## (m)	0.0
y coordinate ## (m)	0.0
z coordinate ## (m)	1.0
Atmospheric Flow Direction	
x coordinate ## (m)	1.0
y coordinate ## (m)	0.0
z coordinate ## (m)	0.0
Integrated Boundary Conditions	

Figure 205:

2. Expand **OUTFLOW** then click on **Outflow**. In the Entity Editor, change the Type to **OUTFLOW**.

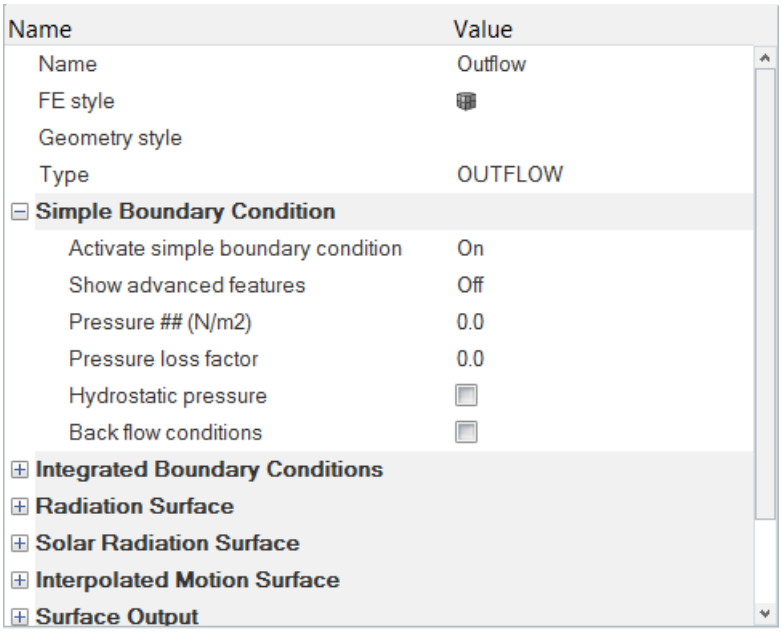


Figure 206:

3. Expand **WALL** then click on **Building**. In the Entity Editor, change the Type to **WALL**.

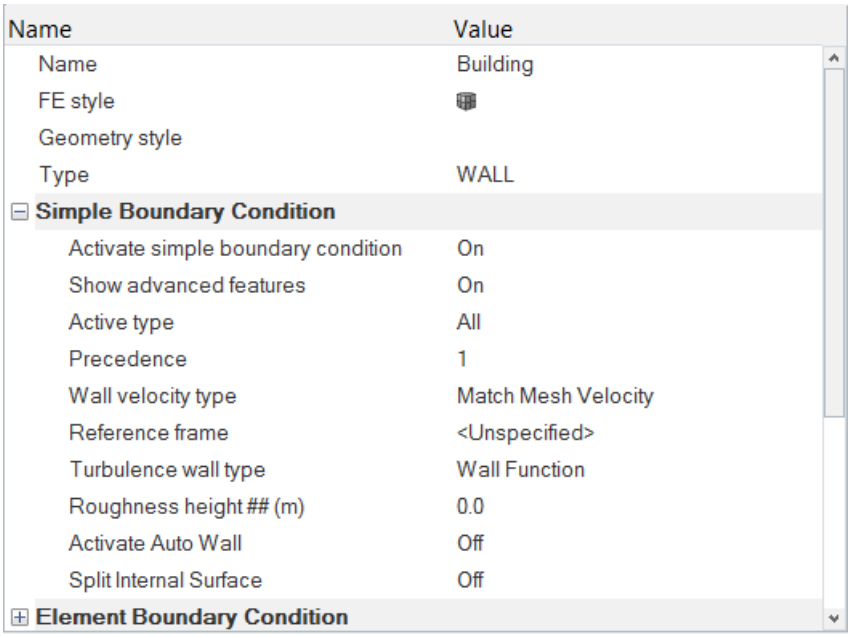


Figure 207:

4. Under **WALL**, click on **Wall**. In the Entity Editor,
   a) Change the Type to **WALL**.
   b) Set the Roughness height to `0.03`.

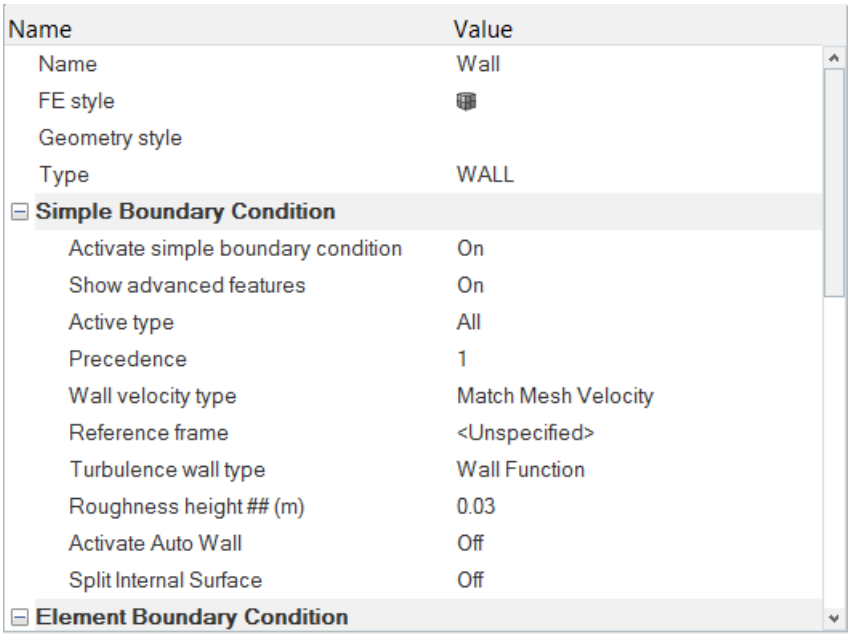


Figure 208:

5. Expand **SLIP** then click on **Slip**. In the Entity Editor, change the Type to **SLIP**.

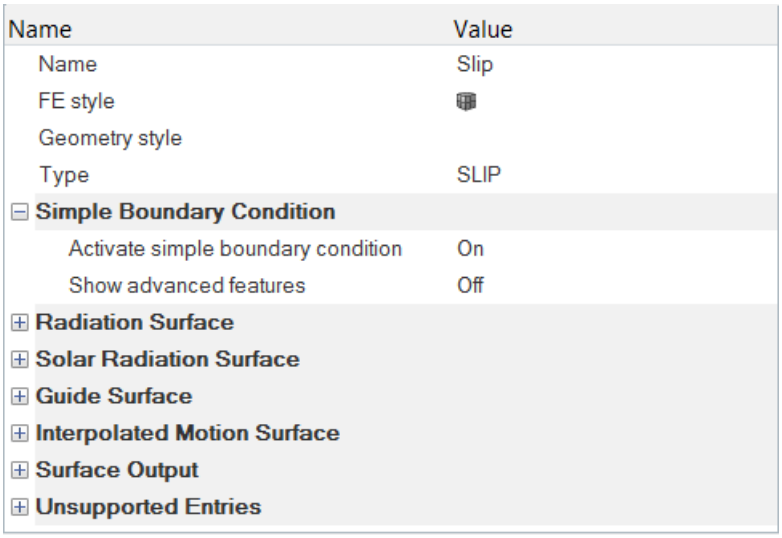


Figure 209:

Set Up Nodal Initial Conditions

1. In the Solver Browser, expand **01.GLOBAL** > **03.NODAL_INITIAL_CONDITIONS** then click on **NODAL_INITIAL_CONDITION**.
2. Verify that the Pressure Default value is `0.0`.
3. Verify that the Velocity in all direction is `0.0`.

4. Change the Eddy viscosity to `0.0001`

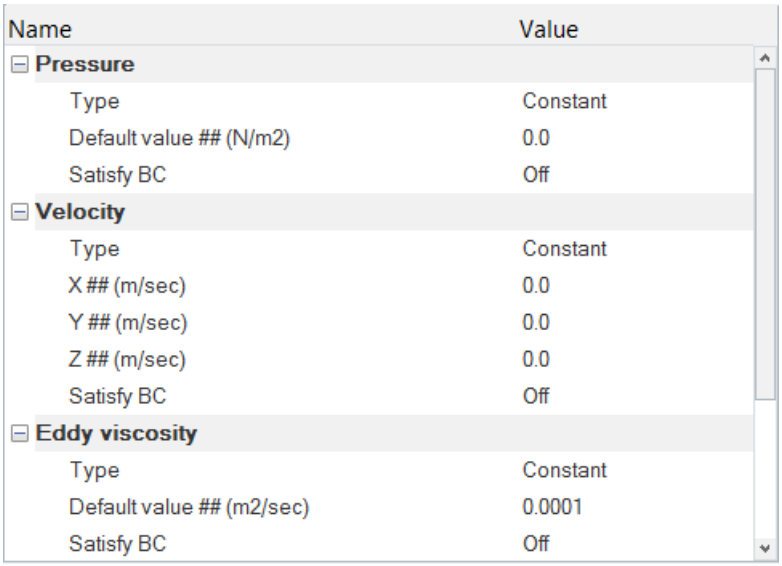


Figure 210:

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

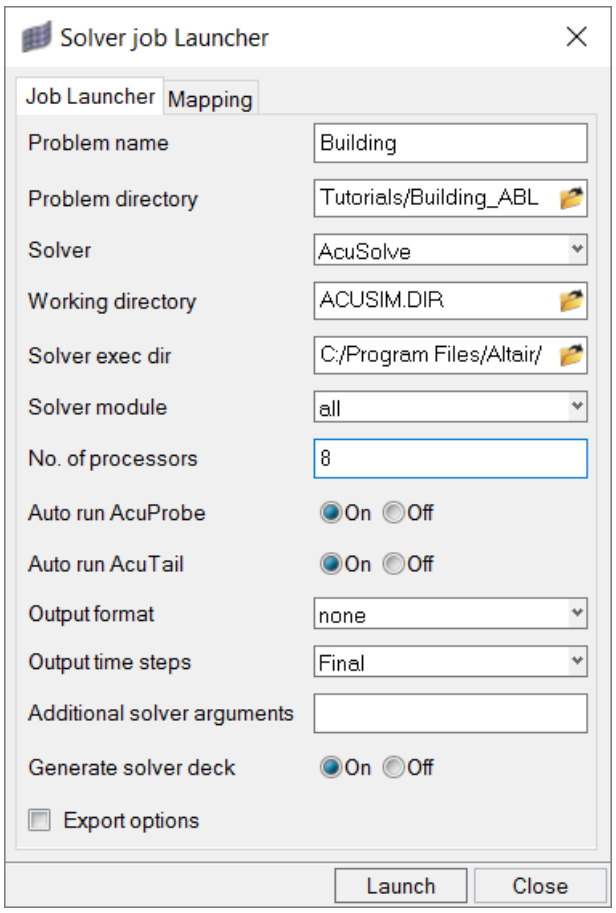


Figure 211:

Post-Process the Results with AcuFieldView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Click on the AcuFieldView icon to launch the **AcuFieldView** dialog.

Load Model and Results

1. In the **AcuFieldView** dialog, click next to File.
2. Navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Building.1.Log`
3. For Solver exec dir, click , then navigate to the AcuSolve installation directory `(<AcuSolve_installation_directory>hwcfdsolvers/acusolve/win64/)` and select the **bin** folder if it is not selected by default.
4. Click **Launch**.

All the surfaces, with mesh, are displayed in AcuFieldView.

5. Click **Viewer Options**.

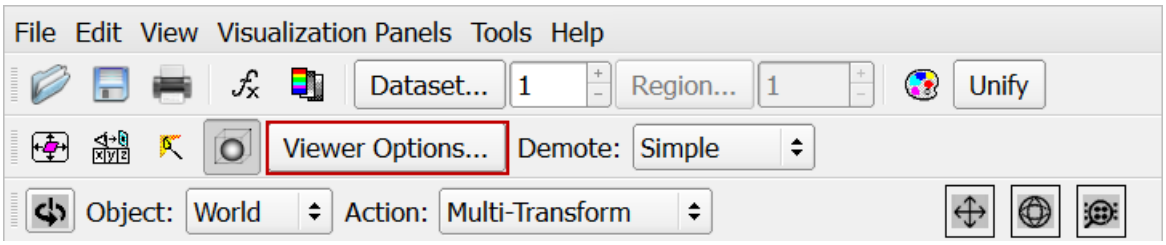


Figure 212:

6. In the **Viewer Options** dialog:
   a) Deselect **Perspective** to turn off the perspective view.
   b) Click **Axis Markers** to disable the axis markers.
   c) Click **Close**.
7. On the toolbar, click the Colormap icon .
8. In the **Scalar Colormap Specification** dialog, click **Background** and select White.
9. Close the **Scalar Colormap Specification** dialog.
10. Click the **Toggle Outline** icon on the toolbar to turn off the outline display.

Coordinate the Surface for Showing Velocity Magnitude on the Y Plane

1. From **Boundary Surface** dialog, Surafce tab, disable the **Visibility** option for the active boundary surfaces.
2. Click to open the **Coordinate Surface** dialog.
3. Click **Create** to create a new Coordinate Surface.
4. Set the Coord Plane to **Y**.
5. Change the Coloring to **Scalar**.
6. Set the Display Type to **Smooth**.
7. In the Scalar Function list, select **velocity_magnitude** as the scalar function to be displayed.
8. In the Colormap tab, change Scalar Coloring to **Local**.
9. In the Legend tab, check the **Show Legend** checkbox to display the velocity magnitude on the coordinate plane.
10. From the Defined Views, select viewing direction as **-Y**.

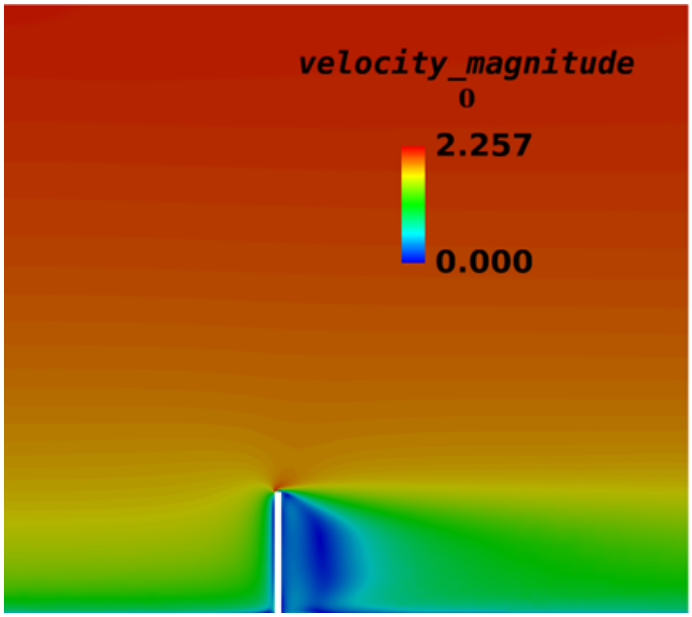


Figure 213:

Coordinate the Surface for Showing Velocity Vectors on the Y Plane

1. From the Surface tab, disable the **Visibility** option for Surface ID 1.
2. Click **Create** to create a new Coordinate Surface.
3. Set the Coord Plane to **Y**.
4. Change the Coloring to **Scalar**.
5. In the Scalar Function list, select **velocity_magnitude** as the scalar function to be displayed.
6. Set the Display Type to **Vectors**.
7. Next to Vectors, click **Options**.
8. Set the VECTOR HEAD type to **2D**.
9. Activate **Head Scaling** and set it to `0.2`.
10. Set the Length Scale to `1`.
11. Activate the **Skip** option and set the value to `75%`.

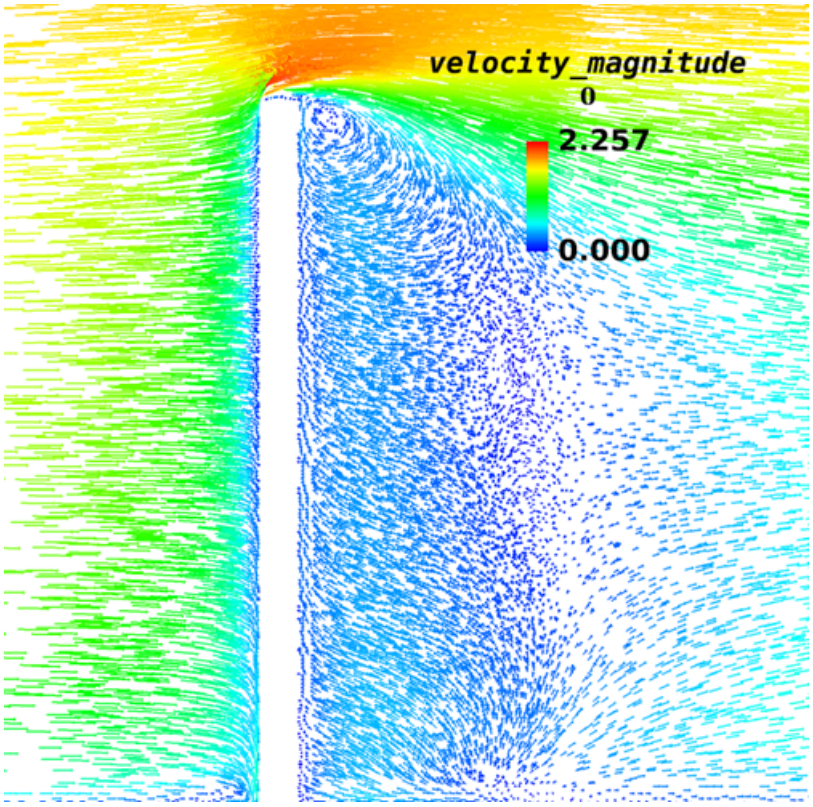


Figure 214:

Coordinate the Surface for Showing Velocity Magnitude on the Z Plane

1. Delete the Coordinate Surfaces with Surface Ids 1 and 2.
2. Click **Create** to create a new Coordinate Surface.
3. Set the Coord Plane to **Z**.
4. Change the Coloring to **Scalar**.
5. Set the Display Type to **Smooth**.
6. In the Scalar Function list, select **velocity_magnitude** as the scalar function to be displayed.
7. In the Colormap tab, change Scalar Coloring to **Local**.
8. In the Legend tab, check the **Show Legend** checkbox to display the velocity magnitude on the coordinate plane.
9. From the Defined Views, select viewing direction as **+Z**.

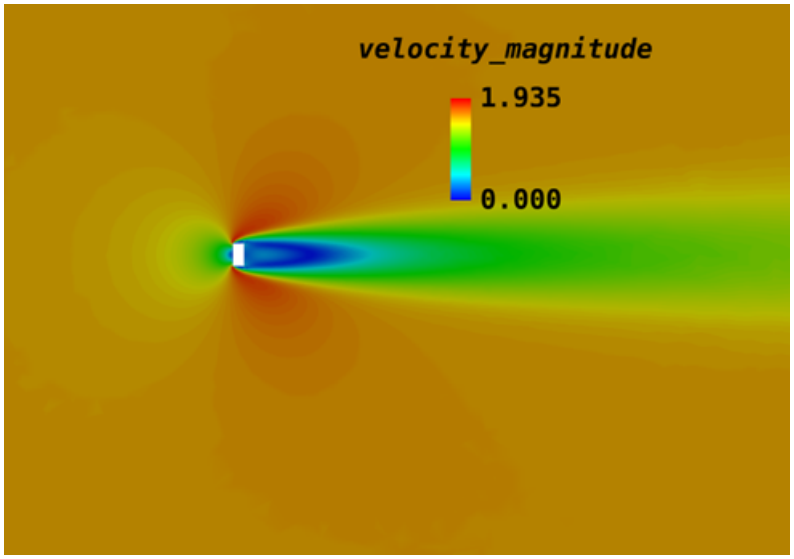


Figure 215:

Coordinate the Surface for Showing Velocity Vectors on the Z Plane

1. From the Surface tab, disable the **Visibility** option for Surface ID 1.
2. Click **Create** to create a new Coordinate Surface.
3. Set the Coord Plane to **Z**.
4. Change the Coloring to **Scalar**.
5. In the Scalar Function list, select **velocity_magnitude** as the scalar function to be displayed.
6. Set the Display Type to **Vectors**.
7. Next to Vectors, click **Options**.
8. Set the VECTOR HEAD type to **2D**.
9. Activate **Head Scaling** and set it to `0.2`.
10. Set the Length Scale to `1`.
11. Activate the **Skip** option and set the value to `75%`.

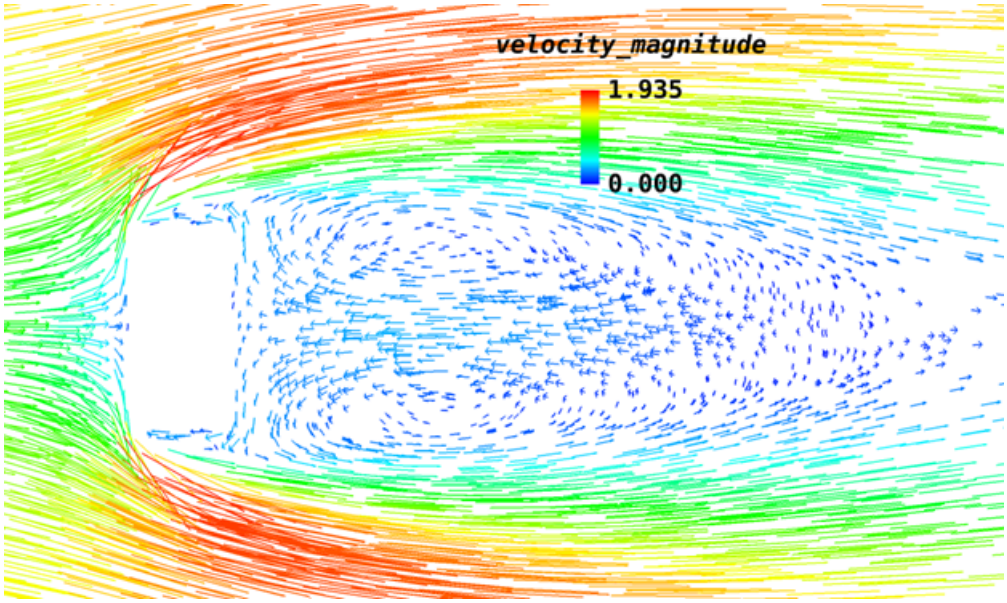


Figure 216:

Summary

In this tutorial, you worked through a basic workflow to set up a CFD model, carry out a CFD simulation, then post-process the results using HyperWorks products, namely AcuSolve, HyperMesh and AcuFieldView. You started by importing the model in HyperMesh. Then, you defined the simulation parameters and launched AcuSolve directly from within HyperMesh. Upon completion of solution by AcuSolve, you used AcuFieldView to post-process the results and create contour plots.

2.3 Heat Transfer

2.3.1 ACU-T: 3000 Enclosed Hot Cylinder: Natural Convection

Prerequisites

This tutorial provides instructions for setting up, solving and viewing results of a CFD simulation of air flow inside the annulus of a concentric cylinder with a heat source due to natural convection. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3000_HotCylinder.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

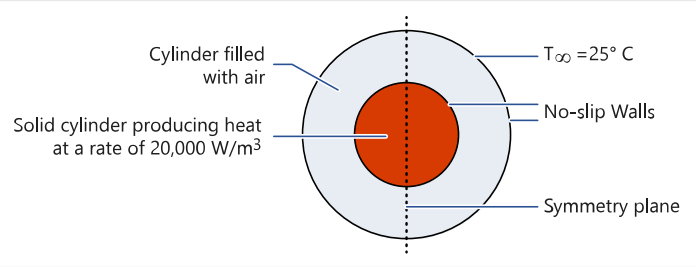


Figure 217:

The schematic of the problem to be solved is shown in the figure above. The inner cylinder is a solid with internal heat generation and the annular space between the inner and outer cylinders is a fluid volume with air as the fluid. The air in contact with the surface of the inner cylinder is heated up and rises to the upper parts of the annulus due to buoyancy effects and displaces the cold fluid at the top. At the same time, the film of the fluid which was in contact with the hot surface is replaced by the surrounding cold fluid. This process is repeated continuously until a steady state is achieved.

Both cylinders are assumed to be infinitely long and are modeled using half symmetry and periodicity. The cylinders are infinite in z-direction and hence periodicity will be applied along this direction.

Introduction to Theory

Natural Convection

Convection is a heat transfer mechanism where the transfer of heat energy happens through the motion of matter. Since the definition of convection involves motion of matter a fluid state is usually present in convection. Usually this type of heat transfer takes place between a hot or a cold surface and a fluid. The film of fluid in contact with the surface absorbs heat from or transfers heat to the surface and is then replaced by a new film. This movement of fluid may either be governed by an external source, such as a fan or pump, or due to internal changes in the fluid properties. When no external sources are responsible for the fluid motion the heat transfer mechanism at work is called the Natural Convection. The driving force for motion of the fluid in a natural convection is density changes in the fluid due to temperature gradients induced in the fluid by heat transfer.

The natural convection mechanism works similarly as described above, whilst discussion of the problem. The fluid which is in contact with the surface absorbs or transfers heat from the surface and becomes hotter or colder than the surrounding fluid. Driven by buoyancy forces due to difference in densities caused by the temperature gradient, the fluid is displaced upwards or downwards. Surrounding fluid fills in the void created by the displaced fluid, which then undergoes the same process again. This gives rise to a convection current which drives the hot fluid to the top and cold fluid to the bottom of the convection cell. Buoyancy effects are driven by gravity, therefore natural convection requires presence of a gravitational force to work. It must be noted, however, that gravity is not the driving force behind the fluid movement. Presence of gravity only enables displacement of the fluid due to the density changes caused by temperature gradients.

Mathematical determination of the onset of natural convection is done through a dimensionless number called the Rayleigh number (Ra). The Rayleigh number is defined as:

$$Ra_x = \frac{g\beta}{\alpha\nu} \quad (T_s - T_\infty) \quad x^3$$

where:

- x is the characteristic length (m)
- Ra_x is the Rayleigh number for characteristic length x
- g is acceleration due to gravity (m/s^2)
- T_s is the surface temperature (K)
- T_∞ is the quiescent temperature (fluid temperature far from the surface of the object) (K)
- ν is the kinematic viscosity (m^2/s)
- α is the thermal diffusivity (m^2/s)
- β is the thermal expansion coefficient (equals to $1/T$ for ideal gases where is absolute temperature).

The fluid properties ν, α and β are evaluated at the film temperature, T_f, which is defined as:

$$T_f = \frac{T_s + T_\infty}{2}$$

When the Rayleigh number is below a critical value for the fluid heat transfer is primarily in the form of conduction. When it exceeds this critical value the dominant heat transfer mechanism is convection.

Boussinesq Density Model

The Boussinesq density model is an approximation method applied to buoyancy driven flows, such as natural convection flows. In the Boussinesq approximation, the density variation terms are neglected everywhere except when multiplied by acceleration due to gravity, g. The basis of this approximation is that since temperature changes are small, the resultant changes in density are small as well and thus can be neglected. However, when multiplied by g, the resultant term gives rise to forces which no longer are negligible. The Boussinesq approximation is:

$$\rho = \rho_0 \left(1 - \beta \Delta T\right)$$

where

- ρ is the instantaneous density at temperature T (kg/m^3)
- ρ_0 is the density at reference temperature T_o (kg/m^3)
- ΔT is change in temperature $T - T_o$ (K)

As stated in the approximation, the Boussinesq density model is only applicable when density variations are small. A general guideline is to check for the condition $\left(\beta \Delta T \ll 1\right)$ to be true. This indirectly puts a limitation on this model to be used to only for cases where expected temperature differences within the fluid are not large.

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3000_HotCylinder.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `NaturalConvection` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `NaturalConvection` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

In this step, you will set the simulation parameters that apply globally to the simulation.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, verify that the Analysis type is set to **Steady State**.
3. Set the Temperature equation to **Advective Diffusive**.

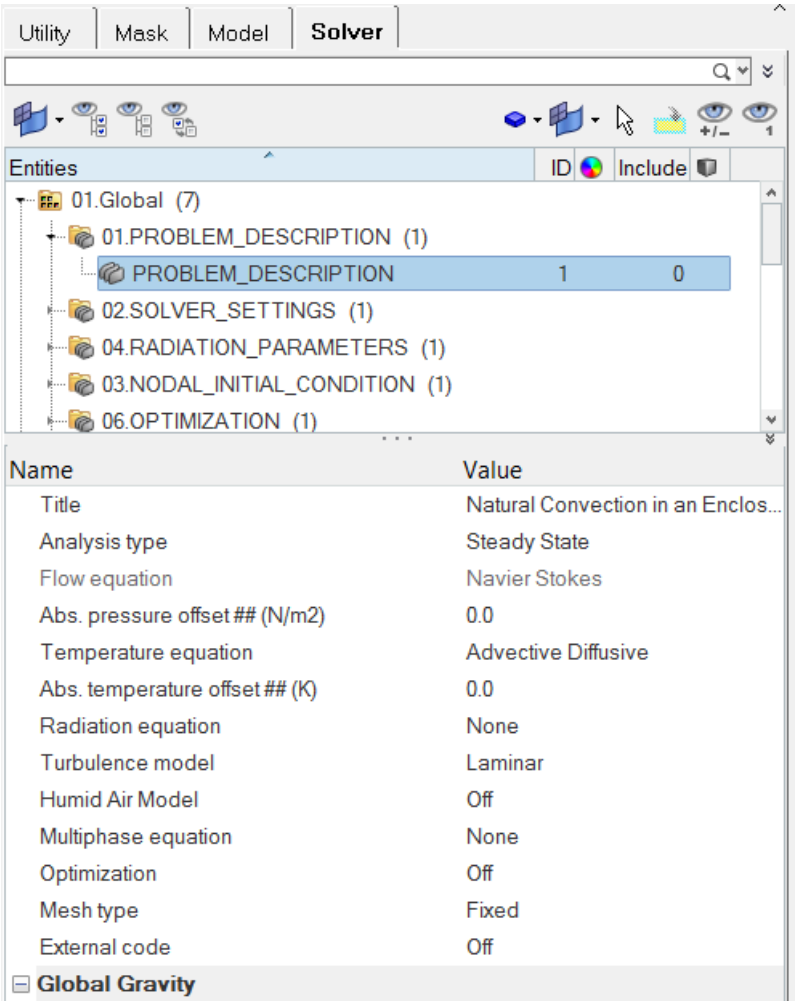


Figure 218:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, change the Convergence tolerance to `0.0001`.
3. Verify that the Flow and Temperature fields are turned **On**.
4. Turn On **Temperature flow**.

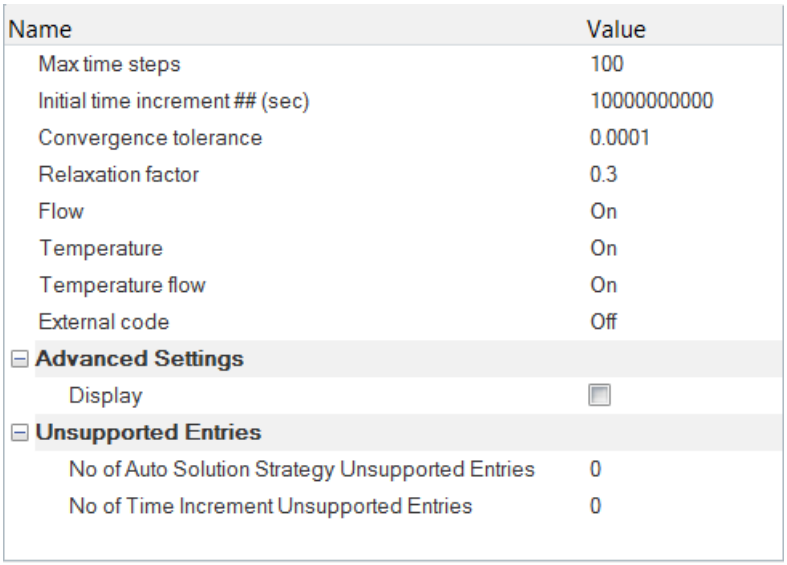


Figure 219:

Create Time History Output Points

The Time History Output command enables you to extract the nodal solution at any point within the domain.

1. In the Solver Browser, right-click on **18.Time_History_Output** and select **Create**.
2. In the Entity Editor, rename the output as `Monitor Points`.
3. Change the Type to **Coordinates**.
4. Set the No of Coordinate Points to `2`.
   A new data entry field is created.
5. Click  to open the **No of Coordinates** dialog. In the dialog, enter the values as shown in the figure below then close the dialog.

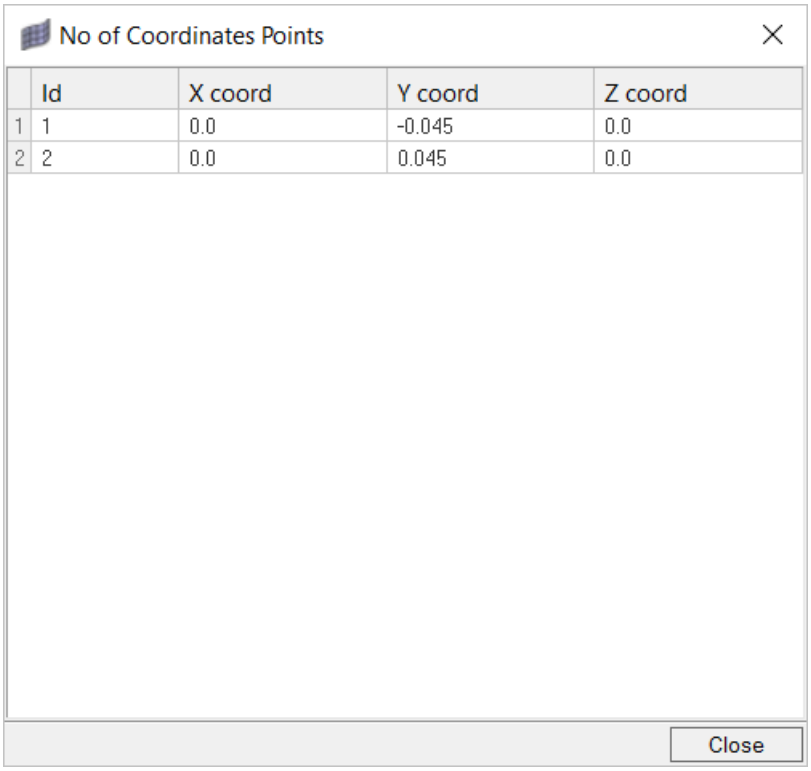


Figure 220:

6. Set the Output frequency to `1`.
   By setting this, the surface output is written for each time step at the specified coordinate points.

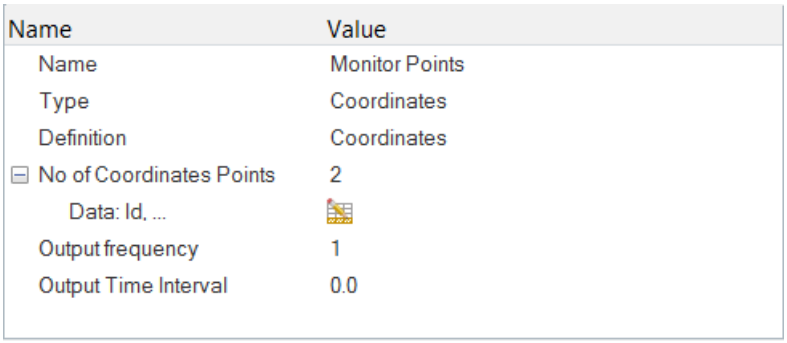


Figure 221:

Set the Body Force and Material Model Parameters

Modify the Material Model Parameters of Air

1. In the Solver Browser, expand **02.Materials** > **FLUID** then click on **Air_HM**.
2. In the Entity Editor, change the Density type to **Boussinesq**.

3. Verify that the Expansivity and Reference temperature values are set to **0.00347222222222** K^{-1} and **288** K respectively.

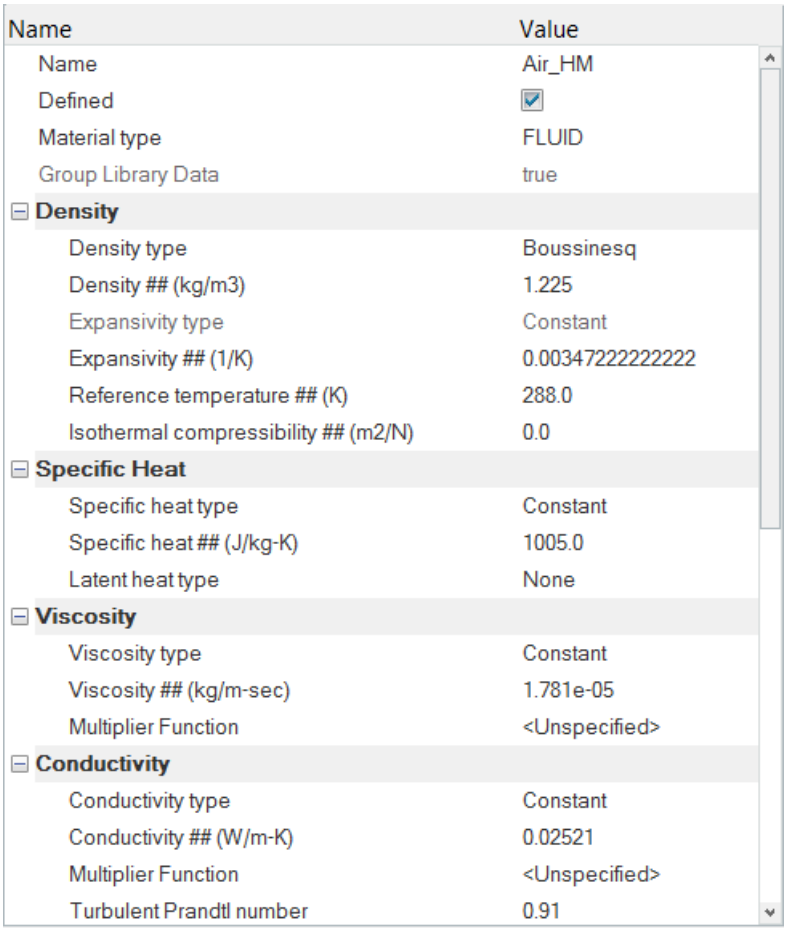


Figure 222:

Create a Material Model for the Solid

1. In the Solver Browser, right-click on **02.Materials** and select **Material(Solid)**.
2. In the Entity Editor, rename the material as `Stainless Steel`.
3. Set the Density value to `8000` kg/m^3.
4. Set the Specific heat value to `500` J/kg-K.
5. Set the Conductivity value to `16.2` W/m-K.

Name	Value
Name	Stainless Steel
Defined	☑
Material type	SOLID
Group Library Data	true
Density	
Density type	Constant
Density ## (kg/m3)	8000.0
Isothermal compressibility ## (m2/N)	0.0
Specific Heat	
Specific heat type	Constant
Specific heat ## (J/kg-K)	500.0
Latent heat type	None
Conductivity	
Conductivity type	Constant
Conductivity ## (W/m-K)	16.2
Multiplier Function	<Unspecified>
Turbulent Prandtl number	0.91
Radiation Properties	
Unsupported Entries	
Library Data	

Figure 223:

Define the Gravitational Body Force

1. In the Solver Browser, expand **03.Body_Force** > **BODY_FORCE** then click on **Gravity_HM**.
2. In the Entity Editor, change the value of Y Gravity to `-9.81` ,/sec^2.
3. Change the Z Gravity to `0`.

Name	Value
Name	Gravity_HM
Medium	Fluid
Gravity	
Gravity type	Constant
X ## (m/sec2)	0.0
Y ## (m/sec2)	-9.81
Z ## (m/sec2)	0.0
Gravity multiplier function	<Unspecified>
Heat Source	
Unsupported Entries	

Figure 224:

Define the Volumetric Heat Source

1. In the Solver Browser, right-click on **03.Body_Force** and select **Create**.
2. In the Entity Editor, rename the body force as `Heat Source`.
3. Change the Medium to **Solid**.
4. Change the Heat source unit type to **Per unit volume**.
5. Set the Volumetric heat source value to `20000` W/m^3.

Name	Value
Name	Heat Source
Medium	Solid
Heat Source	
Heat source unit type	Per unit volume
Heat source type	Constant
Volumetric heat source ## (W/m3)	20000.0
Heat source multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

Figure 225:

6. Save the model.

Specify the Boundary Conditions and Initial Conditions

Specify the Material Properties and Surface Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Solid**. In the Entity Editor,
   a) Change the Type to **SOLID**.
   b) Select **Stainless Steel** as the Material.
   c) Set the Body force to **Heat Source**.

Name	Value
Name	Solid
FE style	
Geometry style	
Type	SOLID
Material	(7) Stainless Steel
Element Set	
Body force	(3) Heat Source
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 226:

3. Click **Fluid**. In the Entity Editor,

a) Change the Type to **FLUID**.

b) Select **Air_HM** as the Material.

c) Set the Body force to **Gravity_HM**.

Name	Value
Name	Fluid
FE style	
Geometry style	
Type	FLUID
Material	(1) Air_HM
Element Set	
Body force	(1) Gravity_HM
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 227:

4. Click **solid_pos_z**. In the Entity Editor, turn **Off** the simple boundary condition.

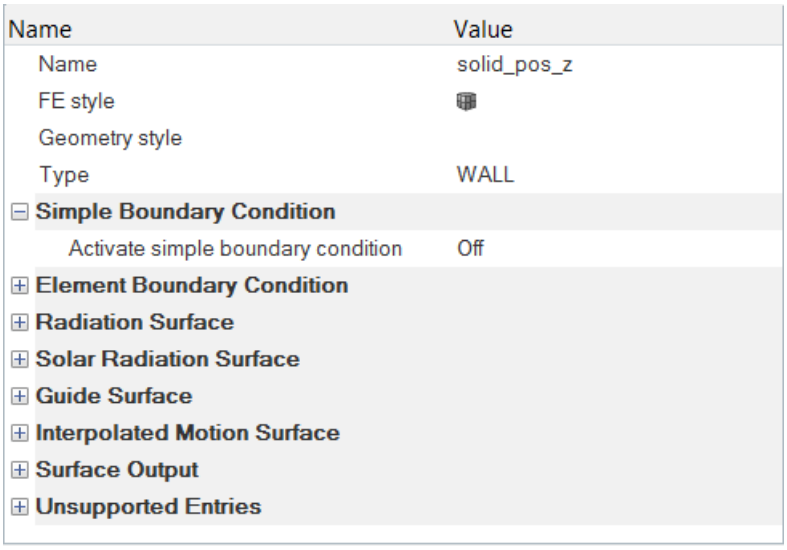


Figure 228:

5. Similarly, turn off the simple boundary condition for the **solid_neg_z**, **fluid_pos_z** and **fluid_neg_z** components.

   These surfaces are the periodic surfaces along the length of the domain and should be modeled as internal surfaces. Hence, the simple boundary condition can be turned off.
6. Click **symmetry**. In the Entity Editor, change the Type to **SYMMETRY**.

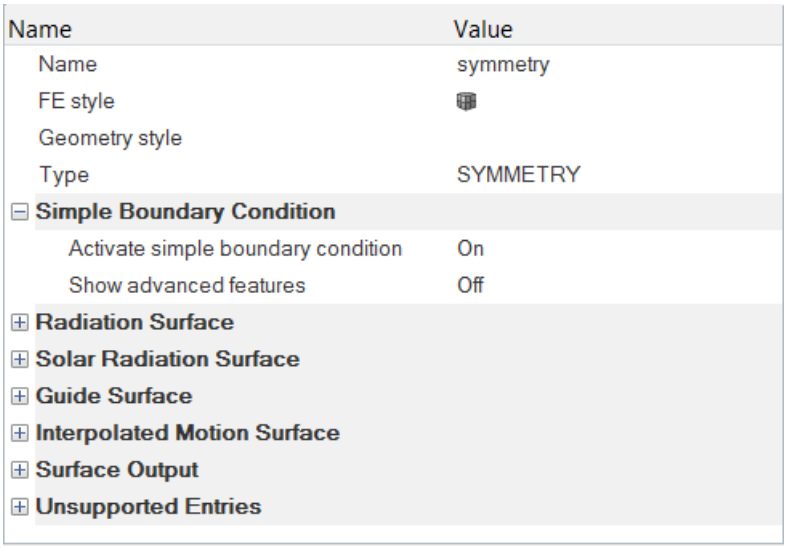


Figure 229:

7. Click **inner_wall**. In the Entity Editor, verify that the Type is set to **WALL**.

Name	Value
Name	inner_wall
FE style	
Geometry style	
Type	WALL
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Temperature BC type	Flux
Heat flux ## (W/m2)	0.0
Convective heat flux coefficient ## (J/m2-sec-K)	0.0
Convective heat flux reference temperature ## (K)	0.0
Element Boundary Condition	
Radiation Surface	
Solar Radiation Surface	
Guide Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 230:

8. Click **outer_wall**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Change the Temperature BC type to **Value**.

c) Set the Temperature to `298` K.

Name	Value
Name	outer_wall
FE style	
Geometry style	
Type	WALL
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Temperature BC type	Value
Temperature ## (K)	298.0
Element Boundary Condition	
Radiation Surface	
Solar Radiation Surface	
Guide Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 231:

Specify the Periodic Boundary Conditions

Since you are modeling only a portion of the infinitely long domain, you need to define the periodicity of the solution by specifying periodic boundary conditions. The solution is then constrained by periodicity. The periodic boundary condition links the corresponding pairs of nodes on the two surfaces on which the periodicity is defined.

1. In the Solver Browser, right-click on **16.Periodic_Boundary_Condition** and select **Create**.
2. In the Entity Editor, rename the boundary condition as `periodicity_fluid`.
3. Verify that the Definition is set to **Periodic components**.
4. In the Source component field, click the **Component** entity collector and select **fluid_pos_z** from the list of components.
5. Similarly, set the Target component to **fluid_neg_z**.

Name	Value
Name	periodicity_fluid
Definition	Periodic components
Precedence	1
Source component	(7) fluid_pos_z
Target component	(8) fluid_neg_z
Periodic BC type	Periodic
Active type	Always
Unsupported Entries	
Number of Unsupported Entries	0

Figure 232:

6. Repeat steps 1 through 5 and create another PBC named `periodicity_solid` with the Source and Target components as **solid_pos_z** and **solid_neg_z**, respectively.

Specify the Reference Pressure

Since this model set up doesn't have an inlet and outlet, we need to specify the reference pressure manually. This can be done by specifying a pressure nodal boundary condition at a node in the fluid region.

1. In the Solver Browser, right-click on **15.Nodal_Boundary_Condition** and select **Create**.
2. In the Entity Editor, rename the boundary condition as `Fixed Pressure node`.
3. Change the Definition to **Nodes**.
4. Set the Number of Nodes to `1`.
5. Change the Boundary condition variable to **Pressure**.
6. Verify that the Type is set to **Zero**.
7. Click the **Node** entity collector then pick a node in the fluid region from the modeling window.

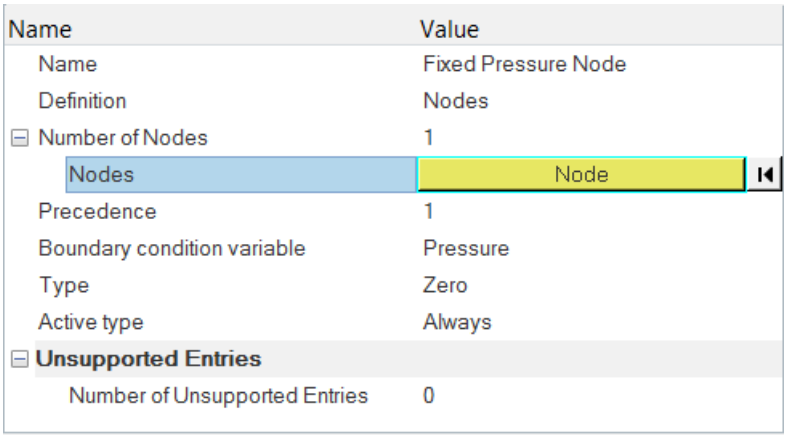


Figure 233:

Specify the Nodal Initial Conditions

1. In the Solver Browser, expand **01.Global** then click on **NODAL_INITIAL_CONDITION**.
2. In the Entity Editor, set the Default value for Temperature to `353.15` K.

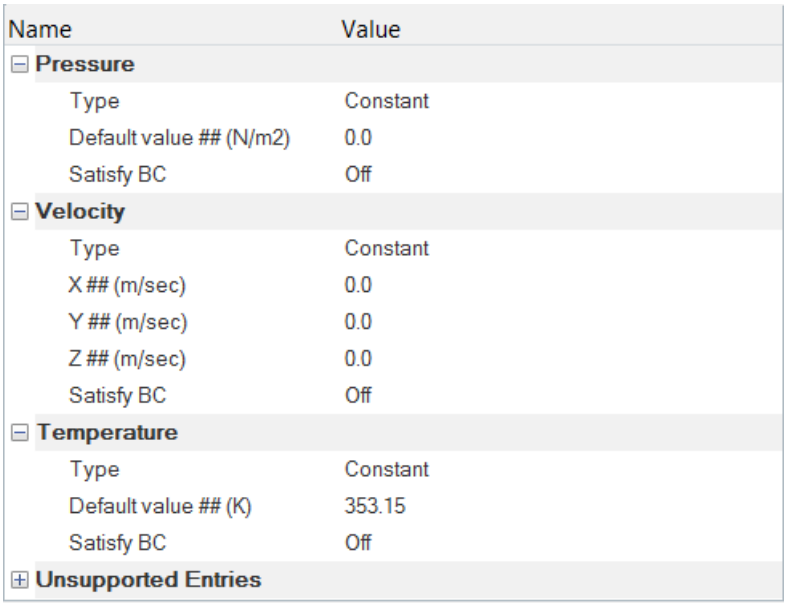


Figure 234:

3. Save the model.

Compute the Solution

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Make sure that the Auto run AcuTail and AcuProbe options are turned **On**.
6. Leave the remaining options as default and click **Launch** to start the solution process.

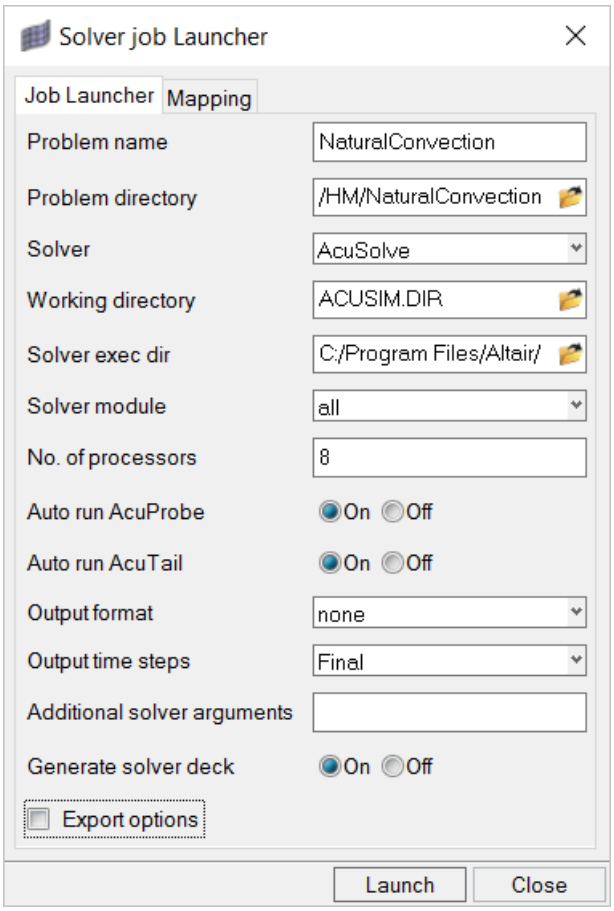


Figure 235:

Post-Process the Results

As the solution progresses, the AcuProbe and AcuTail windows are launched automatically. The progress of the solution can be monitored using these applications.

Post-Process with AcuProbe

1. Once the solution has converged, from the AcuProbe Data Tree, expand **Time History** > **Monitor Points** > **node 1**.
2. Right-click on **temperature** and select **Plot**.
3. Similarly, expand **node 2**, right-click on **temperature**, and select **Plot**.
   The temperature plot at both the locations should be displayed in the plot window as shown below.

   Note: You might need to click on the toolbar in order to properly display the plot.

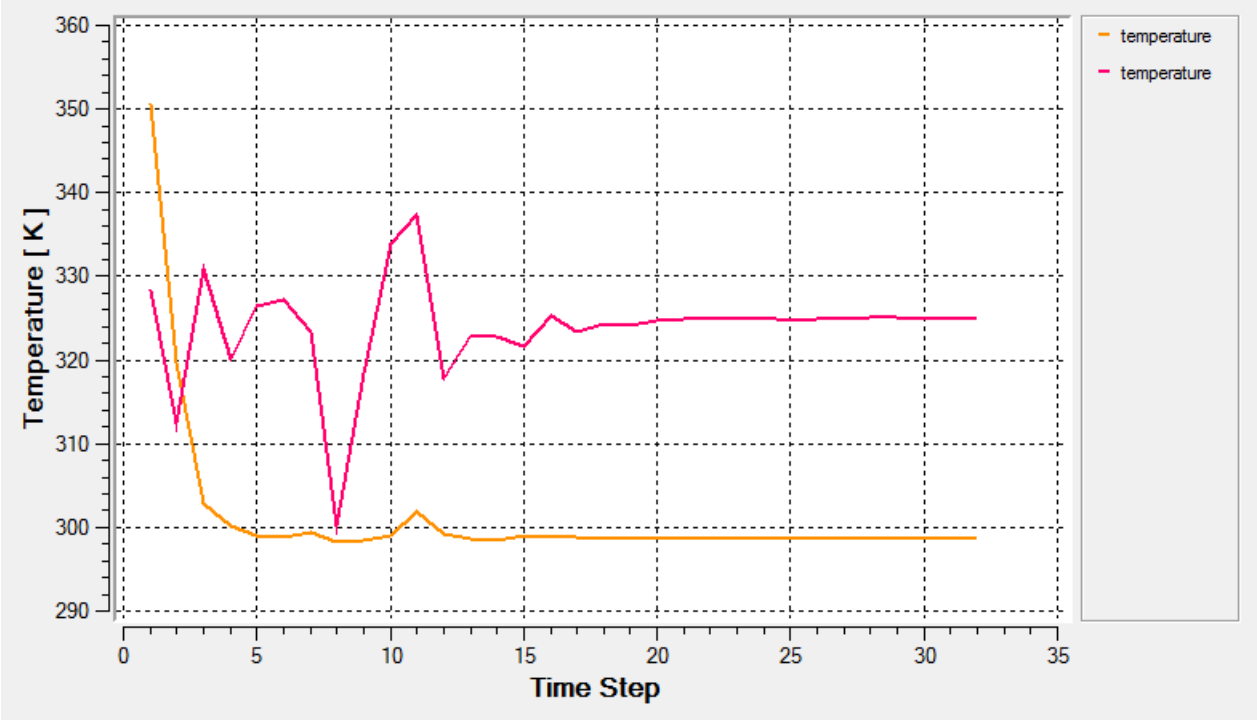


Figure 236:

Close the AcuTail and AcuProbe windows. In the HyperMesh window, close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.
   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `NaturalConvection.1.Log`.

4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.
   The model is colored by geometry after loading.

Create a Temperature Distribution Contour

1. Click on the Results toolbar to open the Contour panel.
2. In the panel area, change the Result type to **Temperature**.
3. Click **Apply** to display the temperature distribution contour.
4. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
5. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 237:

6. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.
   Verify that the contour plot looks like the figure shown below.

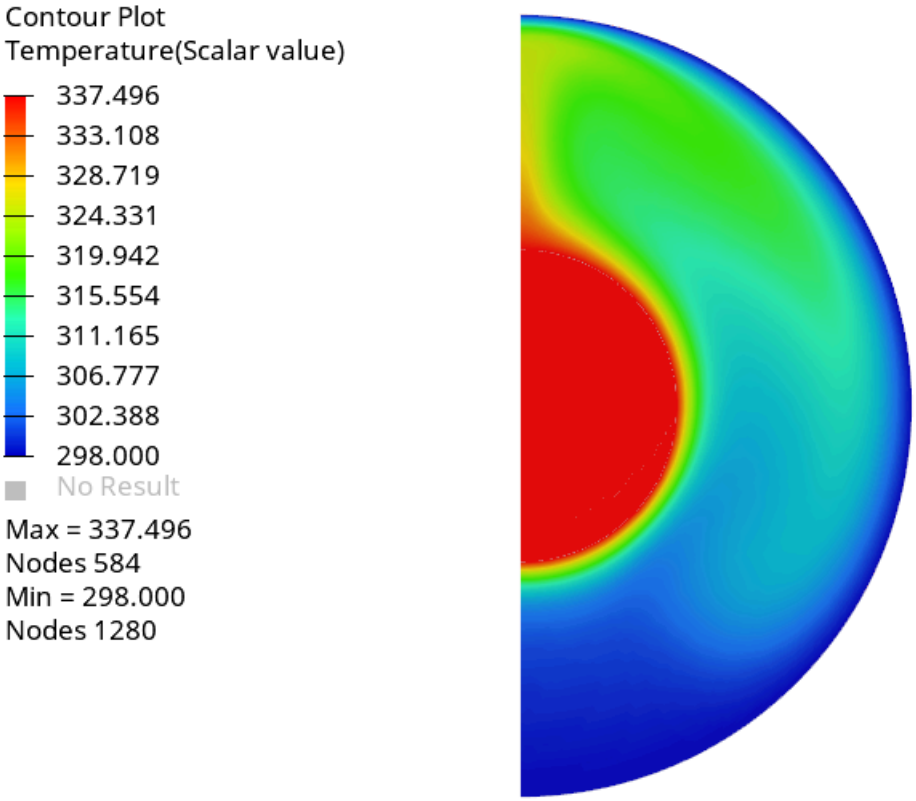


Figure 238:

Create a Velocity Vector Plot

1. Click on the Results toolbar to open the Vector panel.
2. In the panel area, verify that the Result type is set to **Velocity(v)** and **X+Y+Z** is the only check box activated.
3. Activate the **Overlap result display** check box.

   This will imprint the vector plot on top of the temperature contour plot created in the previous step.
4. Click **Apply** in the panel area.

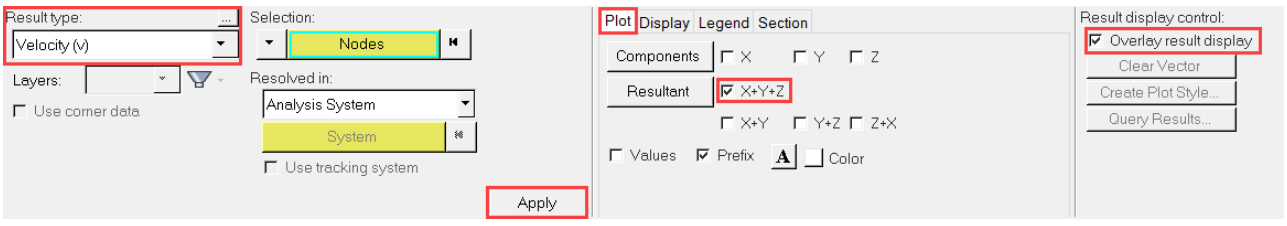


Figure 239:

5. Go to the **Display** tab.

6. Change the Color by option to **Direction**.
7. Click the color button beside **X+Y_Z** and set it to black (or any color of your preference).
8. Set the Vector heads options to **Arrow** and **Tip**.

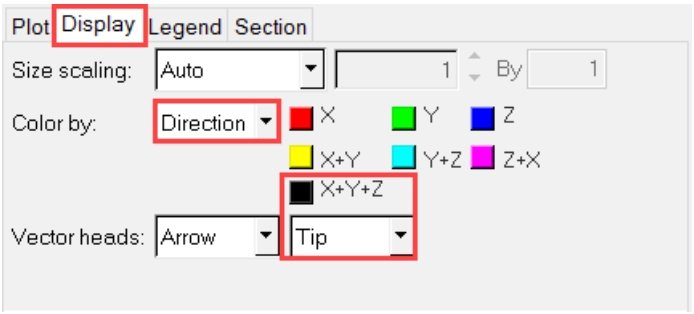


Figure 240:

The Vector plot should be modified accordingly as you make the changes mentioned above. Verify that the vector plot looks like the one shown in the figure below. The plot is zoomed in for a clear display of the direction of air flow.

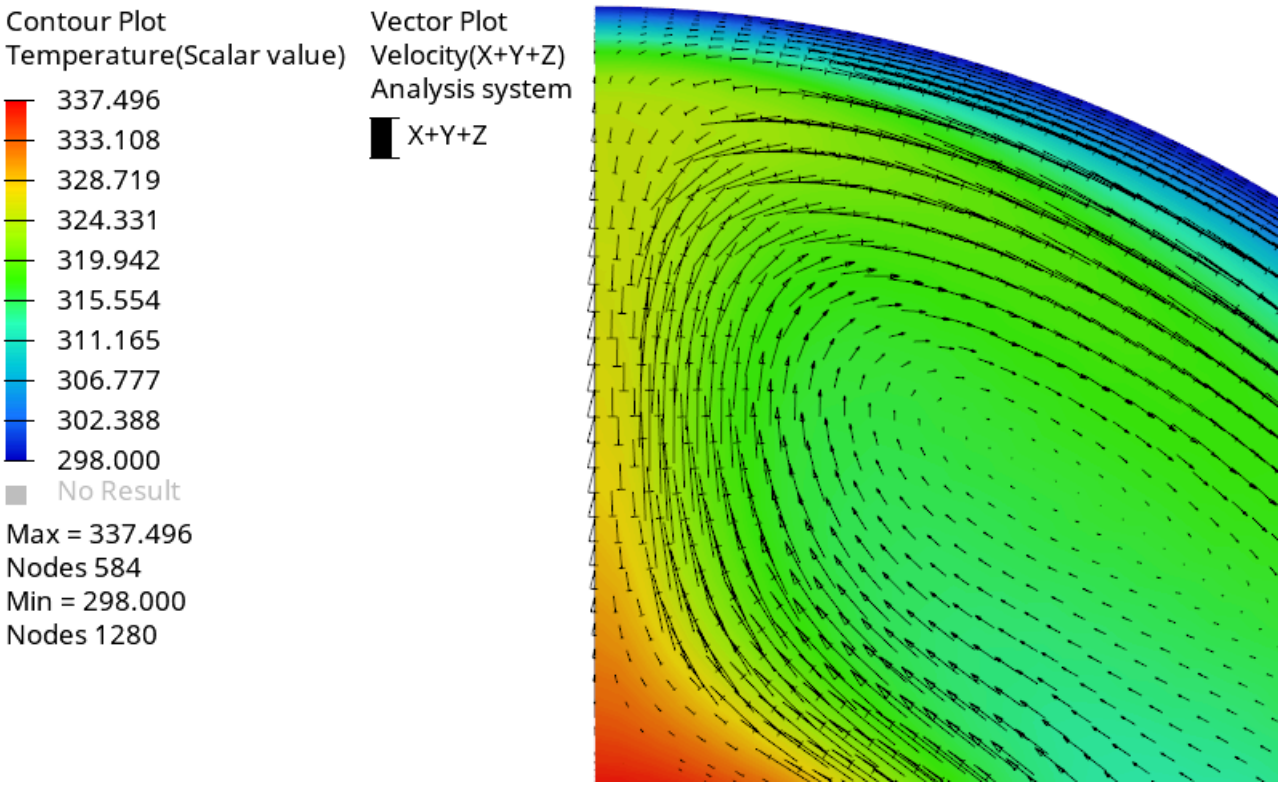


Figure 241:

As mentioned in the problem description, the air which is in contact with the solid surface gets heated up and rises to the top while being replaced by the colder air and the cycle gets repeated until steady state is reached.

Summary

In this tutorial, you successfully learned how to set up and solve a natural convection problem using AcuSolve in HyperMesh. You started by importing the HyperMesh model database and then set up the simulation parameters and boundary conditions. Once the solution was computed, you post-processed the results using HyperView and created a contour plot of temperature distribution across the domain.

2.3.2 ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. Although it is not necessary, it is recommended that you complete ACU-T: 2000 Turbulent Flow in a Mixing Elbow prior to running this simulation. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3100_MixingElbowHeatTransfer.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 1. It consists of a mixing elbow made of stainless steel with water entering through two inlets with different velocities and at different temperatures. The geometry is symmetric about the XY midplane of the pipe, as shown in the figure.

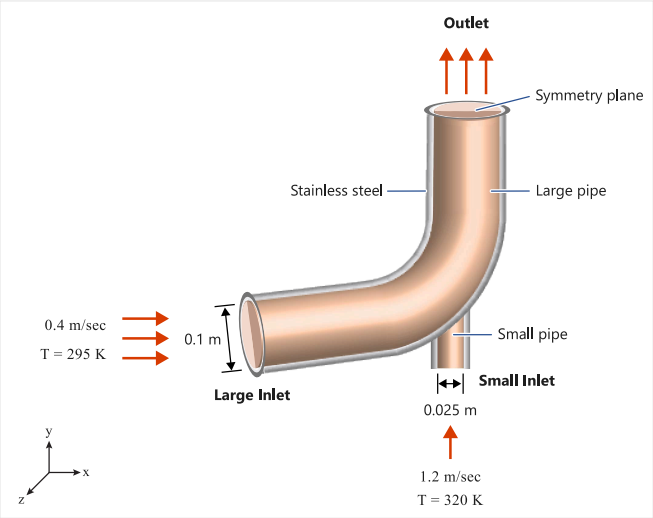


Figure 242: Schematic of Mixing Elbow with Stainless-steel Walls

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3100_MixingElbowHeatTransfer.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `MixingElbow_HeatTransfer` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `MixingElbow_HeatTransfer` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, set the Turbulence equation to **Advective Diffusive**.

3. Set the Turbulence Model to **Spalart Allmaras**.

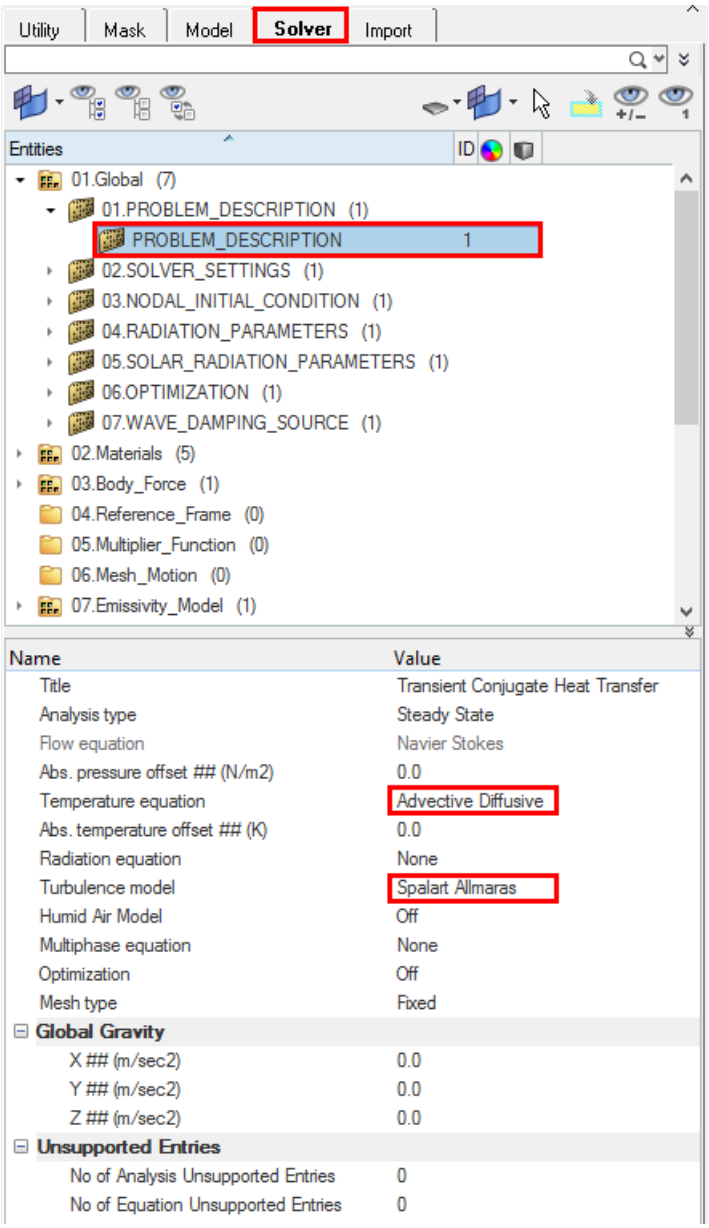


Figure 243:

Set Up Boundary Conditions and Material Model Parameters

In this step, you will start by creating a new material, then you will define the surface boundary conditions for the problem and assign material properties to the fluid and solid volumes.

Create a New Material Model

1. In the Solver Browser, expand the **02.Materials** tree.

2. Right-click on **SOLID** and select **Create**.
3. In the Entity Editor, change the name to `Steel`.
4. Set the Density to `8030` kg/m^3.
5. Set the Specific heat to `500` J/kg-K.
6. Set the Conductivity to `16.2` W/m-k.

Name	Value
Name	Steel
Defined	☑
Material type	SOLID
Density	
Density type	Constant
Density ## (kg/m3)	8030.0
Isothermal compressibility ## (m2/N)	0.0
Specific Heat	
Specific heat type	Constant
Specific heat ## (J/kg-K)	500.0
Latent heat type	None
Conductivity	
Conductivity type	Constant
Conductivity ## (W/m-K)	16.2
Multiplier Function	<Unspecified>
Turbulent Prandtl number	0.91

Figure 244:

Set Up Boundary Conditions

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Large_Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow type to **Average velocity**.
   c) Set the Average velocity to `0.4` m/s.
   d) Set the Temperature to `295.0` K.

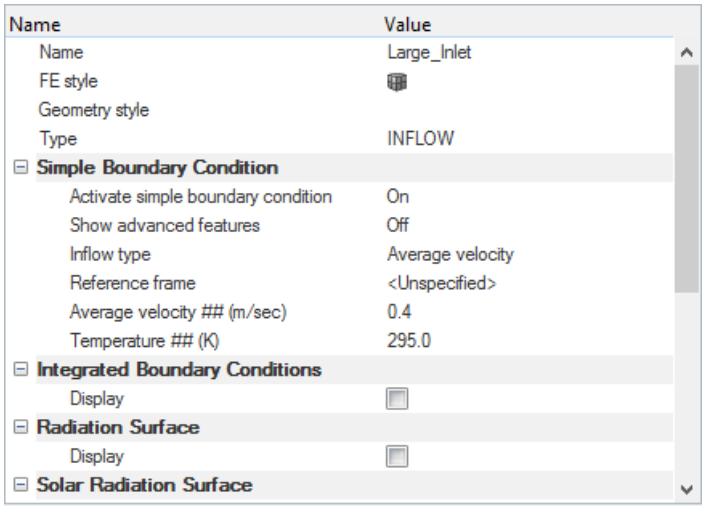


Figure 245:

3. Similarly, click **Small_Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow type to **Average velocity**.
   c) Set the Average velocity to `1.2` m/s.
   d) Set the Temperature to `320.0` K.
4. Click **Outflow**. In the Entity Editor, change the Type to **OUTFLOW**.

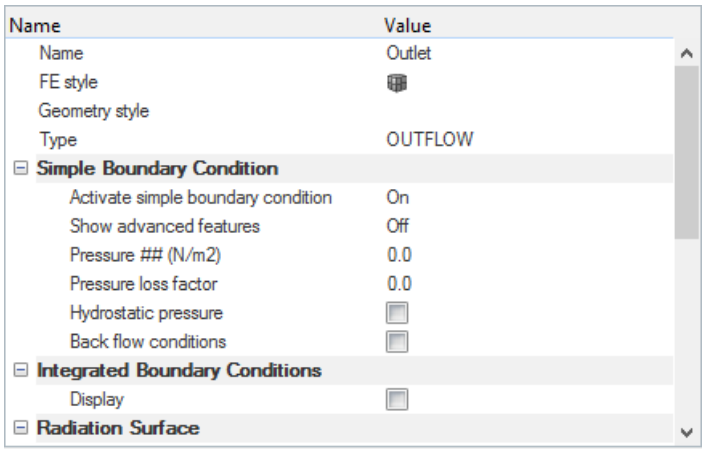


Figure 246:

5. Click **Symmetry**. In the Entity Editor, change the Type to **SYMMETRY**.

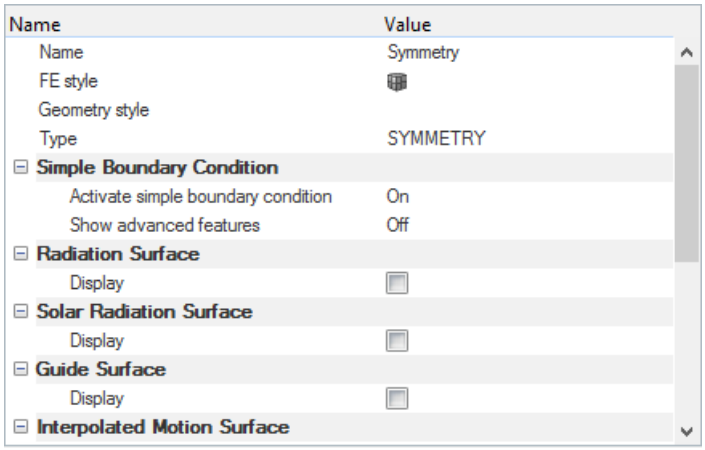


Figure 247:

6. Similarly, click **Pipe_Symmetry** and change the Type to **SYMMETRY**.
7. Click **Pipe_OuterWalls**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Set the Convective heat flux coefficient to 100 J/m^2-sec-K.
   c) Enter `302.594` K for the Convective heat flux reference temperature.

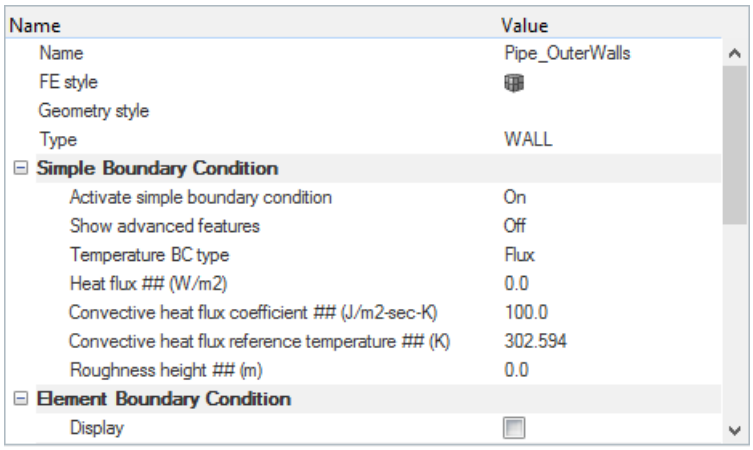


Figure 248:

8. Click **Pipe_InnerWalls**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Turn Show advanced features **On**.
   c) Verify that the Activate Auto Walls option is set to On.

   This surface is an interface between the fluid and solid regions. Auto_Wall is an advanced feature in AcuSolve that automatically detects and splits the internal surface without the user having to do it manually. Hence in this case, the Pipe_InnerWalls surface set will be split into two, one on the fluid region and the other on the solid region.

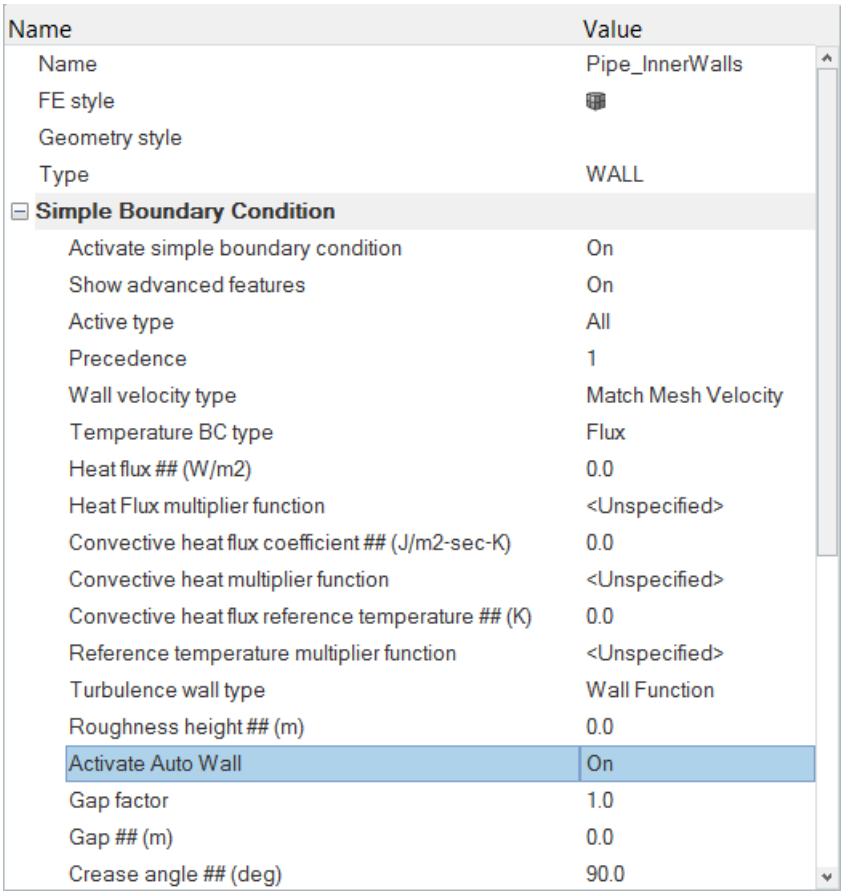


Figure 249:

9. Click **Pipe_Ends**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Change the Temperature BC type to **None**.

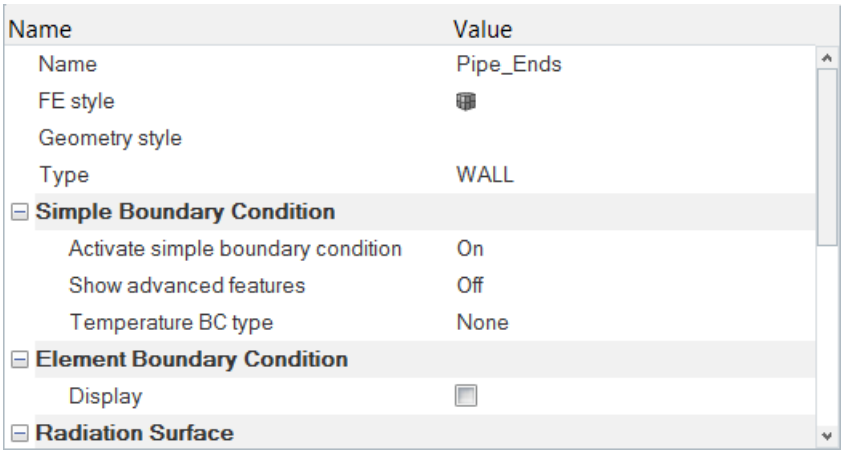


Figure 250:

10. Click **Fluid**. In the Entity Editor,

a) Change the Type to **FLUID**.
b) Select **Water_HM** as the Material.

Name	Value
Name	Fluid
FE style	
Geometry style	
Type	FLUID
Material	(2) Water_HM
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 251:

11. Click **Solid**. In the Entity Editor, and
a) Change the Type to **SOLID**.
b) Select **Steel** as the Material.

Name	Value
Name	Solid
FE style	
Geometry style	
Type	SOLID
Material	(5) Steel
Element Set	
Body force	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 252:

12. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

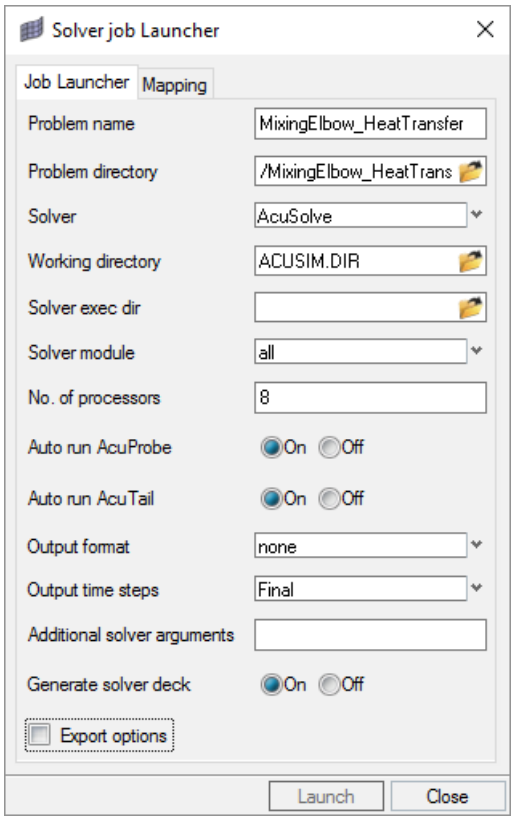


Figure 253:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.

2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `MixingElbow_HeatTransfer.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.
   The model is colored by geometry after loading.

Create Contours for Temperature Distribution

In this step, you will display temperature contours on the symmetry plane and the outlet surface.

1. In the Results Browser, expand the list of **Components**.
2. Click the Isolate Shown icon then hold Ctrl and select the **Symmetry** and **Pipe_Symmetry** components to turn off the display of all components in the graphics window except the Symmetry and Pipe_Symmetry components.

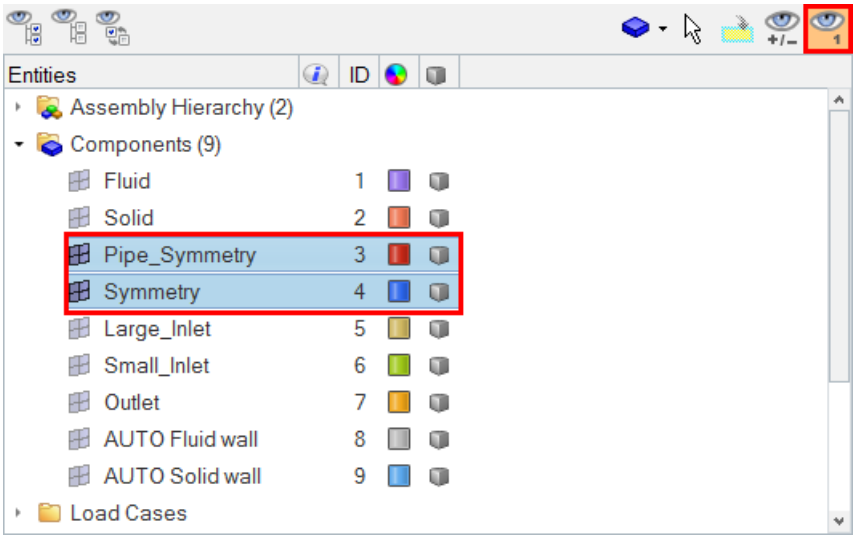


Figure 254:

3. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
4. Click on the Results toolbar to open the Contour panel.
5. Under result type, select **Temperature(s)**.
6. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
7. Click **Apply**.
8. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 255:

9. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.

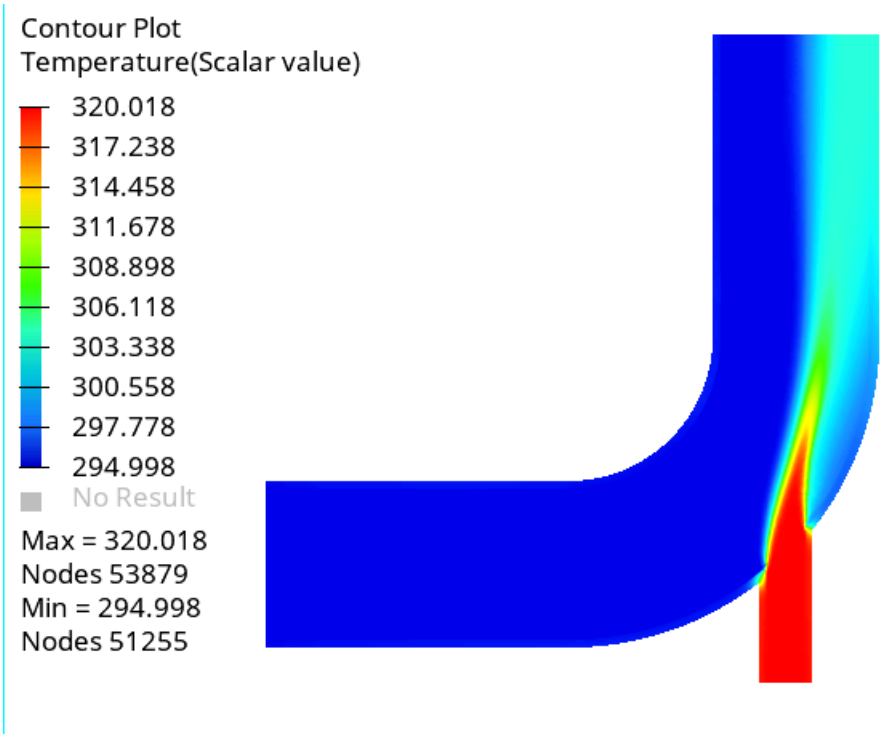


Figure 256:

Next, you will display temperature contours on the outlet surface.

10. Turn off the display for all components except Outlet.

11. Click on the Standard Views toolbar.

12. In the panel area, click on the **Components** entity selector and select **Displayed**.

13. In the panel area, click **Apply**.
The contour plot on the outlet surface is displayed.

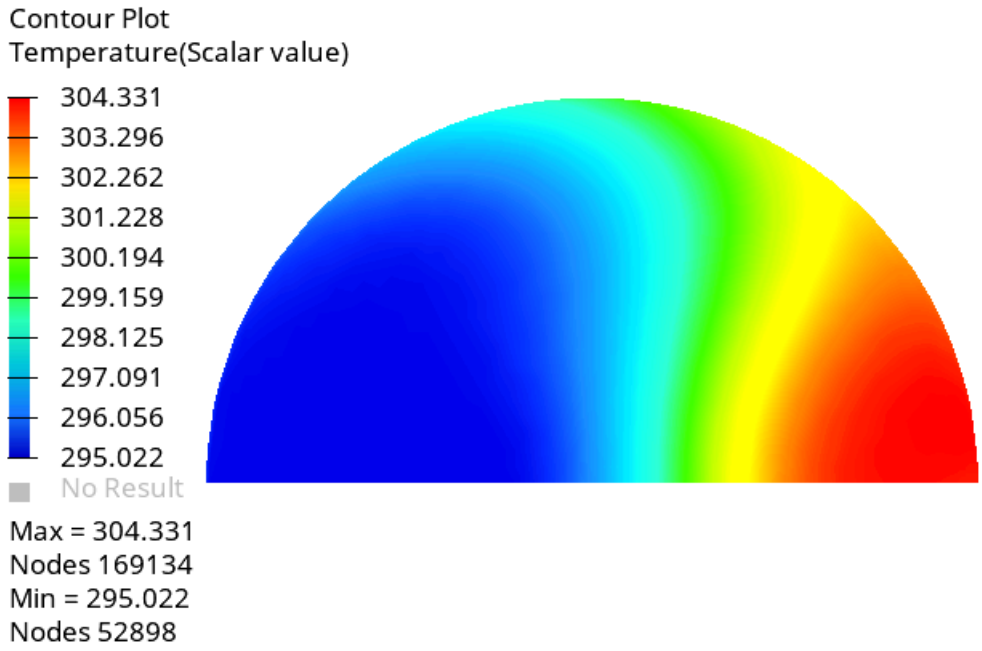


Figure 257:

Summary

In this tutorial, you learned how to set up a conjugate heat transfer CFD simulation using HyperMesh and how to create a new material model. You launched AcuSolve directly from HyperMesh to compute the solution and then post-processed the results using HyperView.

2.3.3 ACU-T: 3101 Transient Conjugate Heat Transfer in a Mixing Elbow

Prerequisites

This tutorial provides you instructions for running a transient simulation of a 3D turbulent flow with conjugate heat transfer in a mixing elbow. You should have already run through the ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow tutorial and have a basic understanding of HyperMesh, AcuSolve and HyperView. The HyperWorks introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, gives a basic introduction to HyperWorks and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3101_MixingElbowTransient.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

This problem is divided into two components, a steady state solution and a transient solution. The schematic of the steady state component is shown below.

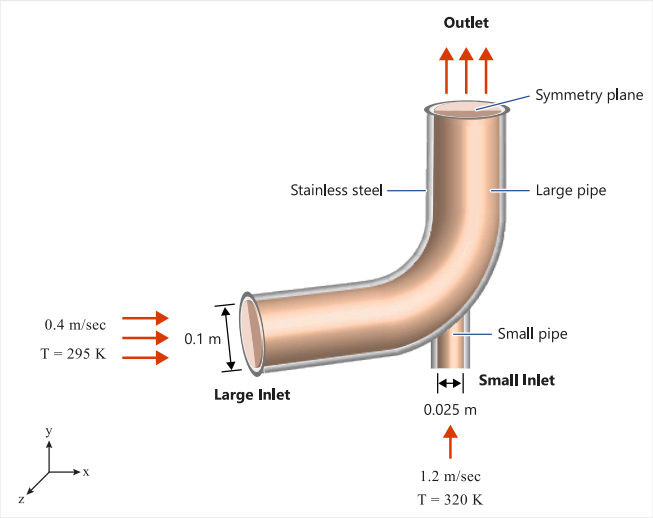


Figure 258:

The diameter of the large inlet is 0.1 m, the inlet velocity (v) is 0.4 m/s and the temperature (T) of the fluid entering the large inlet is 295 K. The diameter of the small inlet is .025 m, the velocity is 1.2 m/s, and the temperature of the fluid entering the small inlet is 320 K. The pipe wall has a thickness of 0.005 m. The fluid in this problem is water and the pipe walls are made of stainless steel with a density of 8030 kg/m3, a conductivity of 16.2 W/m-K, and a specific heat of 500 J/kg-K.

The model file for the steady state part of the problem is provided as the input file. Once the steady state solution is computed, it is projected on to the mesh and used as the initial state for the transient simulation. The starting point for the transient portion of the problem is shown schematically in the figure below.

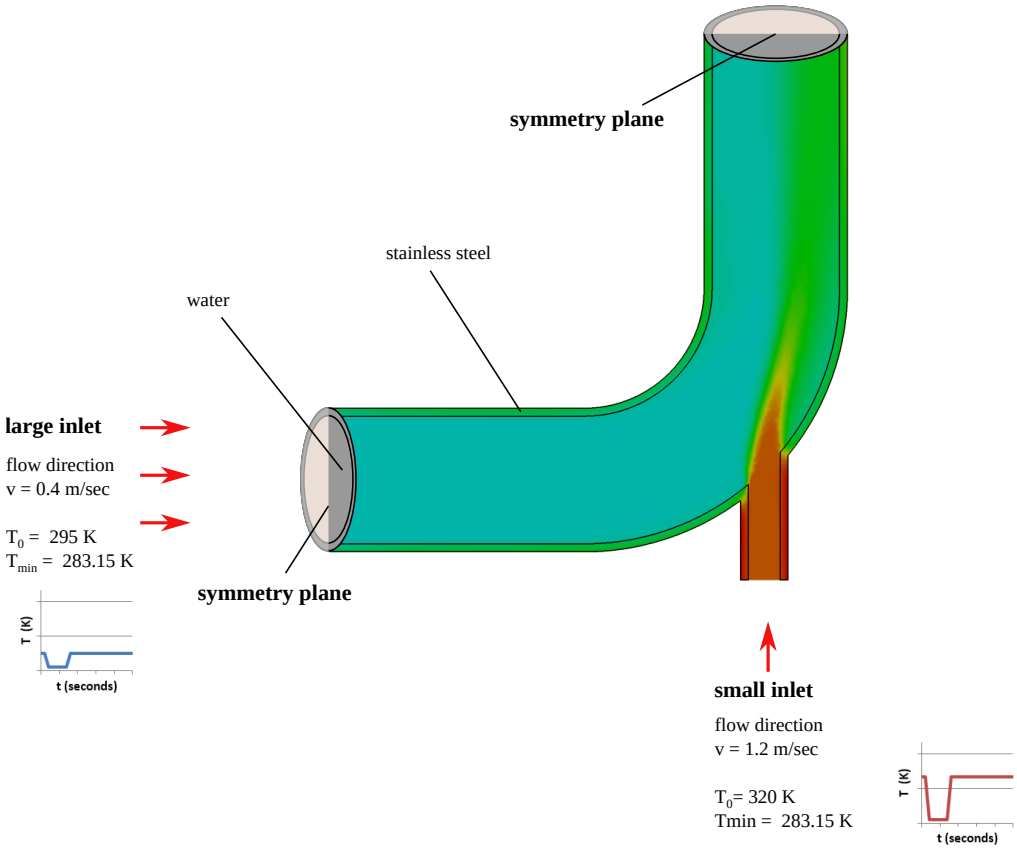


Figure 259:

At 0.2s into the simulation, a cold slug of water is injected at both the inlets and the temperature is ramped down to 283.15 K starting from 0.2 s to 0.4 s. Then it is maintained constant at 283.15 K for 1 sec and then ramped up to initial states from 1.4s to 1.6s. Given a flow path of 0.6356 m, the transit time for the slug is approximately 1.6s. Therefore, our simulation time should be at least 3.2 s to factor in the duration of the slug and transit time. The total simulation time will be 4.5s to allow time for the thermal conditions to return to a steady state.

The temperature change at the large inlet is from 295 K to 283.15 K. At the small inlet, the temperature changes from 320 K to 283.15 K. The ratio of the cold slug temperature to the initial temperature of the large inlet flow is 0.9598. The ratio of the cold slug temperature to the initial temperature of the small inlet flow is 0.8848. These values will be used in creating multiplier functions to model the transient temperatures at the inlets.

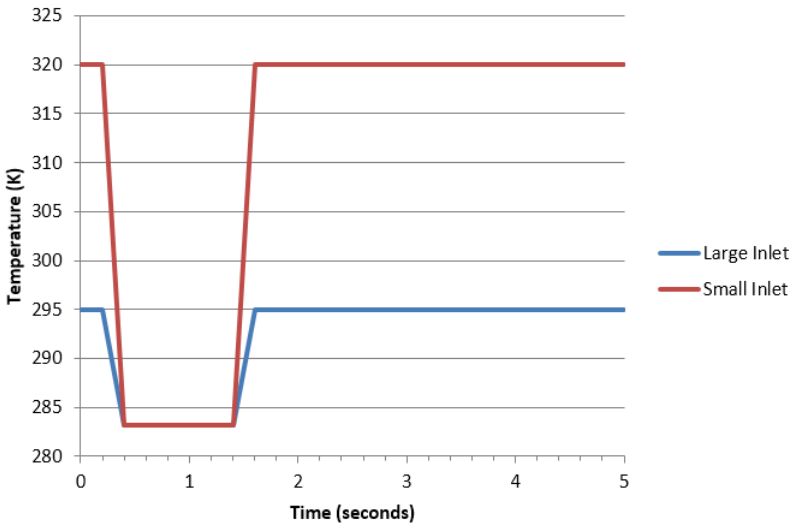


Figure 260:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3101_MixingElbowTransient.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `ConjugateHeatTransfer_Transient` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `ConjugateHeatTransfer_Transient` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Run the Steady State Simulation

In this step, you will run the steady state simulation with the model file provided and then create the nodal initial condition files needed for the transient simulation. Make sure that the visibility of the mesh for all the components is on.

1. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
2. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
3. Once the solution is converged, close the AcuProbe and AcuTail windows. In addition, close the **Solver job Launcher** and **AcuSolve Control** tabs.
4. Start AcuSolve Command Prompt from the Windows Start menu by clicking **Start** > **Altair <version>** > **AcuSolve Cmd Prompt** .
5. In the Command Prompt, change the directory to the working directory by using the `cd` command.
6. Type the command `acuProj -crd HYPERMESH.DIR\ConjugateHeatTransfer_Transient.crd -run 1` and press Enter.
7. Verify that the working directory is now populated with the new files.

   ```
   ConjugateHeatTransfer_Transient.pres.nic
   ConjugateHeatTransfer_Transient.vel.nic
   ConjugateHeatTransfer_Transient.temp.nic
   ConjugateHeatTransfer_Transient.eddy.nic
   ```
8. Move the `.nic` files in the working directory into the HYPERMESH.DIR folder.

Set the Transient Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Change the Analysis type to **Transient** in the Entity Editor.

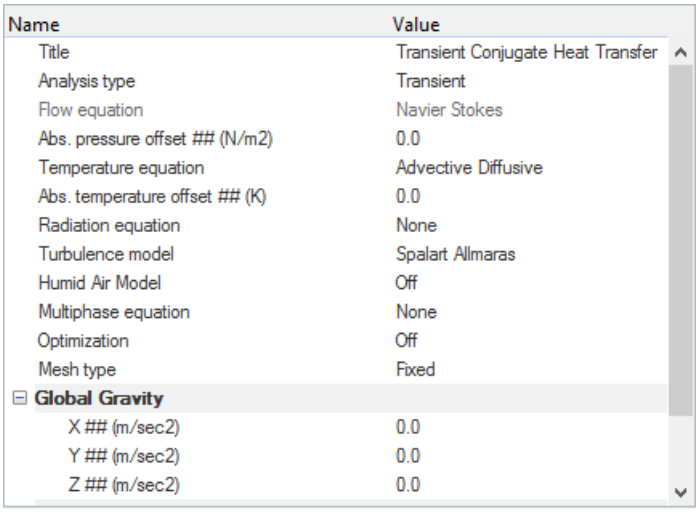


Figure 261:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, set the Max time steps to `0`.
3. Set the Final time to `4.5` sec.
4. Set the Initial time increment to `0.053` sec.
5. Verify that the Convergence tolerance is set to `0.001`.
6. Set the Min stagger iterations to `2` and Max stagger iterations to `5`.
7. Change the Relaxation factor to `0.0`.
8. Turn **Off** the Flow and Turbulence fields.

Name	Value
Max time steps	0
Final time ## (sec)	4.5
Initial time increment ## (sec)	0.053
Auto time increment	☐
Multiplier Function	<Unspecified>
Convergence tolerance	0.001
Min stagger iterations	2
Max stagger iterations	5
Relaxation factor	0.0
Flow	Off
Temperature	On
Turbulence	Off
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 262:

Set the Nodal Output Frequency

1. In the Solver Browser, expand **17.Output** then click **NODAL_OUTPUT**.
2. In the Entity Editor, set the Time step frequency to `3` and the Time frequency to `0`.
3. Turn **On** the Output initial condition field.

Name	Value
Time step frequency	3
Time frequency ## (sec)	0.0
Output initial condition	☑
Continuous output	☐
Number of saved states	0
Unsupported Entries	
Number of Unsupported Entries	0

Figure 263:

4. Save the model.

Specify the Transient Inflow Boundary Conditions and Nodal Initial Conditions

In this step, you will start by creating Multiplier Functions and then specify the transient boundary conditions for both the inlets. Then you will specify the Nodal Initial Conditions for the flow and thermal fields.

Create Multiplier Functions

First, you will create curves for the scaling function to be used for the Multiplier Function type.

1. Right-click on empty space in the Model Browser and select **Create** > **Curve**.
   The **Curve editor** dialog opens.
2. Click **New** and enter **Large_Inlet** as the name in the panel area.
3. Click **Proceed**.
4. In the **Curve editor** dialog, enter the following values for the curve array.

X	Y
0.0	1.0
0.2	1.0
0.4	0.9598
1.4	0.9598
1.6	1.0
4.5	1.0

Figure 264:

5. Click **Update**.
6. Click again on **New** and enter `Small_Inlet` as the name of the second curve, then click **Proceed**.
7. In the **Curve editor** dialog, click on **Small_Inlet** in the top-left corner and enter the following values for the array.

   Make sure that the Current curves is showing Small_Inlet.

X	Y
0.0	1.0
0.2	1.0
0.4	0.8848
1.4	0.8848
1.6	1.0
4.5	1.0

Figure 265:

8. Click **Update**.
   Both the curves should be displayed, as shown in the figure below.

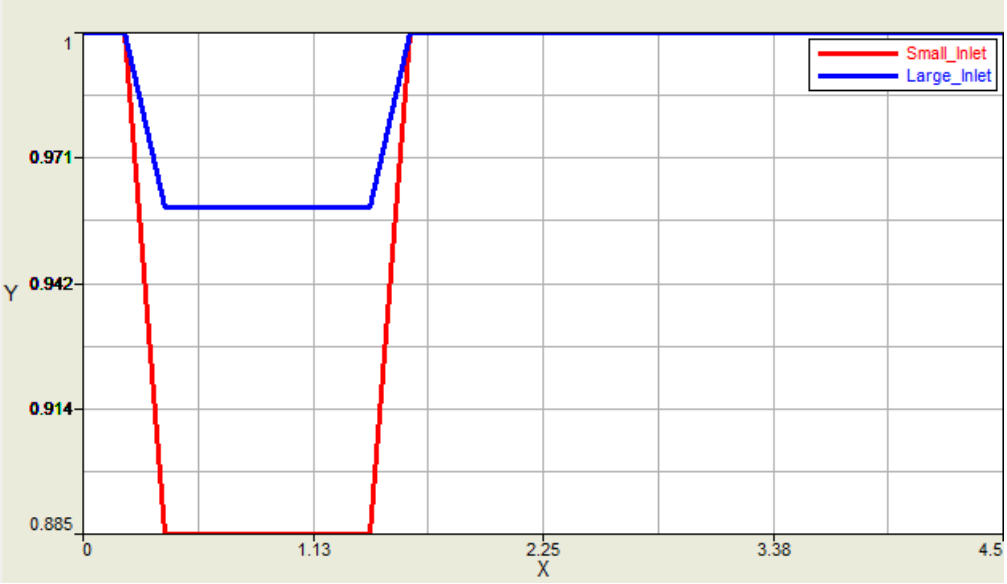


Figure 266:

> **Note:** The default color for the curves is grey, which can be changed using the Color option on the bottom left corner of the **Curve editor** dialog

Next, you will create the Multiplier Functions for both the Inlets.

9. In the Solver Browser, right-click on **05.Multiplier_Function** and select **Create**.
10. Set the Name of the function to `Large_Inlet`.
11. Select **Piecewise Linear** for the Multiplier Function Type.
12. Select the Curve **Large_Inlet**.

Name	Value
Solver Keyword	MULTIPLIER_FUNCTION
Name	Large_Inlet
Type	Multiplier_Function
Multiplier Function Type	Piecewise Linear
Curves	(1) Large_Inlet
Curve fit variable	Time
Evaluation Type	Per Time Step
Unsupported Entries	
Number of Unsupported Entries	0

Figure 267:

13. Repeat the previous four steps to create a Multiplier Function named `Small_Inlet` with **Small_Inlet** as the Curve.

Specify the Nodal Initial Conditions

In this step, first you will specify the initial values of Pressure, Velocity, Temperature and Eddy Viscosity at all nodes and then Transient BCs for both the inlets.

1. In the Solver Browser, expand **01.Global** then click **03.NODAL_INITIAL_CONDITION**.
2. In the Entity Editor, under the Pressure tab, change the Type to **Nodal Values** and Select nodes by to **NIC file**.
3. Click on the select file icon in the value field of NIC file, browse to your working directory, and select the **ConjugateHeatTransfer_Transient.pres.nic** file.

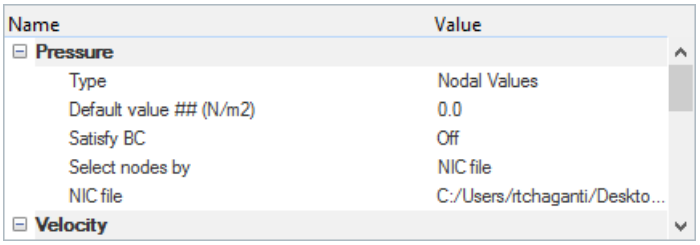


Figure 268:

4. Repeat the steps 2 and 3 for the Velocity, Temperature, and Eddy Viscosity fields and select the **ConjugateHeatTransfer_Transient.vel.nic**, **ConjugateHeatTransfer_Transient.temp.nic**, and **ConjugateHeatTransfer_Transient.eddy.nic** files respectively.

Specify the Transient Inlet Boundary Conditions

1. In the Solver Browser, expand **12.Surfaces** > **INFLOW**.
2. Click **Large_Inlet**. In the Entity Editor, under the Simple Boundary Condition tab,
   a) Turn **On** the Show advanced features field.
   b) Click on the entity collector in the Value field of the Temperature multiplier function and select **Large_Inlet**. Click **OK** to close the dialog.

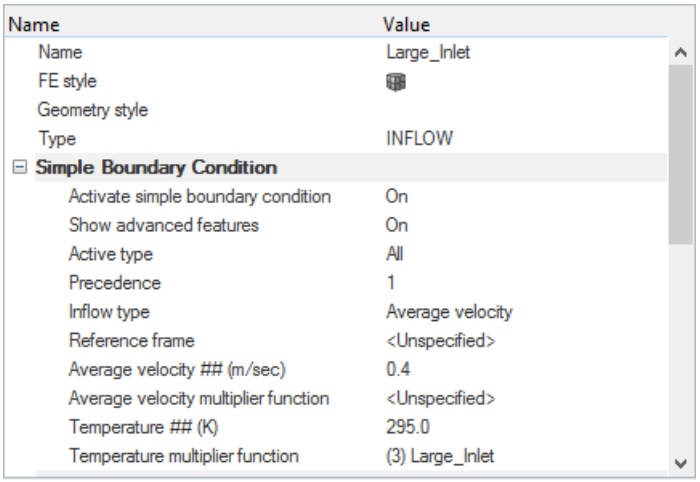


Figure 269:

3. Similarly, click on **Small_Inlet** component and turn **On** the Show advanced features field. Select **Small_Inlet** as the Temperature multiplier function.
4. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Set the Output time steps to **All** if it's not already set.
5. Leave the remaining options as default and click **Launch** to start the solution process.

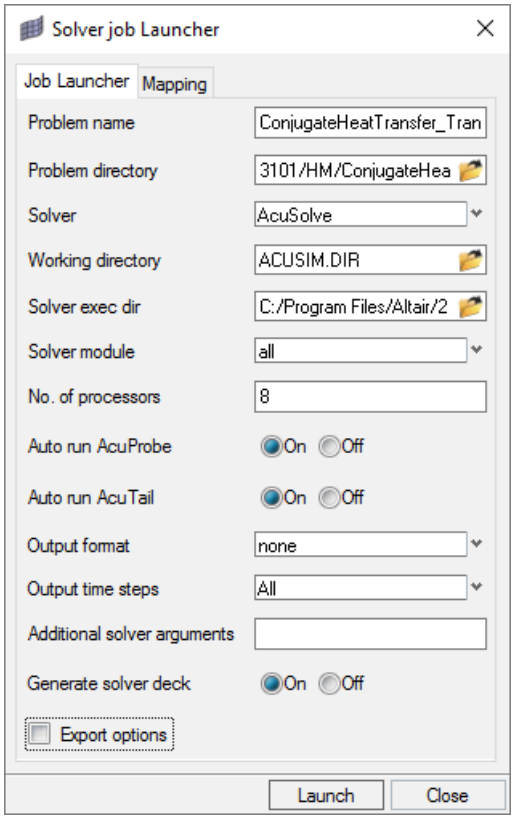


Figure 270:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `ConjugateHeatTransfer_Transient.2.log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create a Temperature Distribution Animation

In this step, you will create an animation of the temperature distribution with time on the Symmetry surface.

1. In the Results Browser, expand the list of **Components**.
2. Click the Isolate Shown icon then hold Ctrl and click the **Symmetry** and **Pipe_Symmetry** components to turn off the display of all components in the graphics window except the Symmetry and Pipe_Symmetry.
3. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
4. Click on the Results toolbar to open the Contour panel.
5. Select **Temperature (s)** as the Result type.
6. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
7. Click **Apply** to display the Temperature contour on the Symmetry plane at the first-time step.
8. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 271:

9. Click the **Legend** tab then click **Edit Legend**.
10. In the **Edit Legend** dialog, change the Type to **Dynamic Scale** and the Numeric format to **Fixed** then click **OK**.
11. On the Animation toolbar, click the Animation Controls icon .
12. Drag the Max frame Rate slider to 1 fps.
13. Click the Start/Pause Animation icon to play the animation in the graphics area.

Save the Animation

1. In the menu area, select **Preferences** > **Export Settings** > **AVI**.
2. In the **Export Settings AVI** dialog, set the Frame rate to 1 fps and click **OK**.
3. On the ImageCapture toolbar, make sure that the Save Image to File option is On.



4. Click the Capture Graphics Area Video icon .

The **Save Graphics Area Video As** dialog opens.

5. Navigate to the location where you want to save the file, enter a name of your choice, and click **Save**.

Summary

In this tutorial, you learned how to set up and run a transient conjugate heat transfer simulation using HyperMesh and AcuSolve. You learned how to specify Nodal Initial Conditions and how to create multiplier functions for setting up the transient boundary conditions. Finally, you used HyperView to create and save an animation of the results of the transient simulation.

2.3.4 ACU-T: 3110 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping using acuOptiStruct

Prerequisites

This tutorial introduces you to setting up and solving a steady state conjugate heat transfer problem using HyperMesh and then using acuOptiStruct to generate an OptiStruct solver deck to perform a thermal stress analysis. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3110_acuOptiStruct.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 272. It consists of an exhaust manifold with four inlets and one outlet. The inlets have flanges with holes for steel bolts to attach the manifold. The body of the manifold is made up of stainless steel.

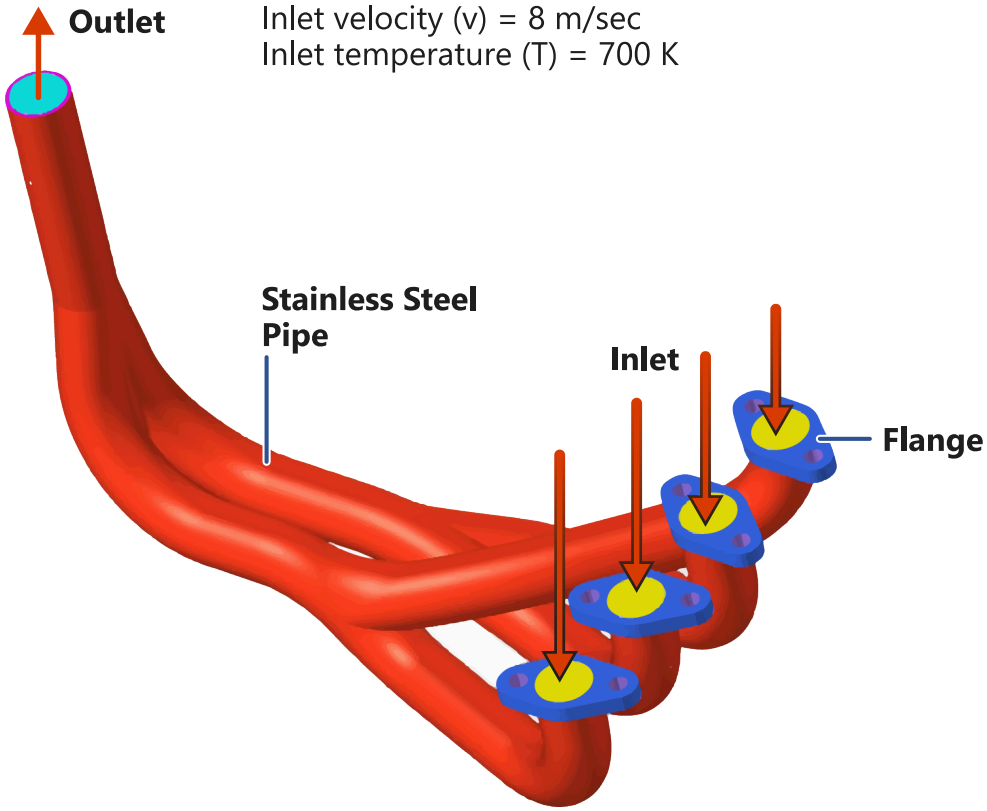


Figure 272: Schematic of Exhaust Manifold

The diameter of the inlets is 0.036 m; the inlet velocity (v) is 8.0 m/s; and the temperature (T) of the fluid entering the inlets is 700 K. The diameter of the outlet is 0.036 m. The pipe wall has a thickness of 0.003 m and the flanges have a thickness of 0.01 m.

The combustion mixture enters the inlets and heat is transferred through conduction inside the manifold. The heat transfer causes deformations and stress in the manifold body which can be simulated using OptiStruct.

The fluid in this problem is air, which has the following material properties:

Density (ρ)	1.225 kg/m^3
Viscosity (μ)	1.781 * 10^{-5} kg/m-s
Specific Heat (C_p)	1005 J/kg-K
Conductivity (k)	0.0251 W/m-K

The exhaust manifold is designed as Steel which has following material properties:

Density (ρ)	800 kg/m^3
Specific Heat (C_p)	500 J/kg-K
Conductivity (k)	16.2 W/m-K

For the HyperMesh simulation, the variation in material properties of air with temperature is ignored.

The HyperMesh simulation will be set up to model steady state heat transfer to determine the temperature and pressure distribution on the walls of the manifold.

The nodal surface output needs to be activated for all the surfaces in order to create the OptiStruct input deck from the acuOptiStruct command.

The temperature distribution and forces on the wetted surfaces are used by OptiStruct to calculate the deformations and stress in the solid body.

The OptiStruct input deck is generated through a utility acuOptiStruct which can be used for a one-way coupled simulation. The following input commands are of importance for this simulation:

`-solids`	Input name for the solid body/bodies where conduction heat transfer would take place.
`-den`	Density values for the solid body/bodies.
`-spcsurfs`	List of surfaces where boundary condition constraints need to be specified.
`-spcsurfsdof`	List of degrees of freedom for the surfaces.
`-spcsurfsdofvals`	List of degrees of freedom values for the surfaces which is zero by default.
`-type`	Stress analysis type for the OptiStruct solver.

For this simulation, the constrained surfaces are the flange bolts and Outlet end of the manifold. These surfaces will be constrained in all six degrees of freedom (Translation and Rotation). The default value of zero is used.

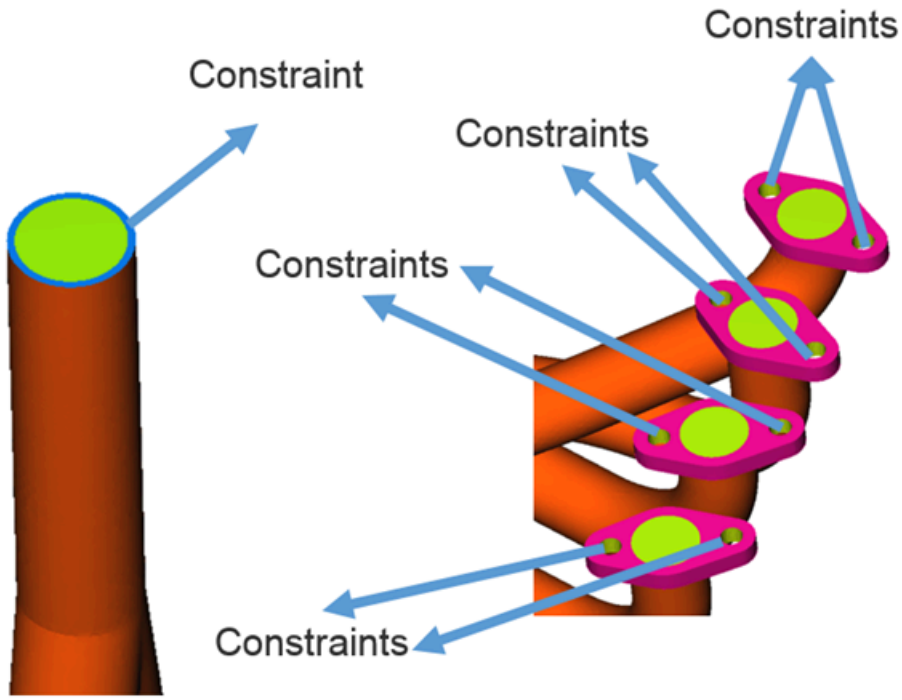


Figure 273:

The stress analysis type is selected as steady linear where the deformations are in the elastic range; that is, the stresses, σ, are assumed to be linear functions of the strains, ε, Hooke's law can be used to calculate the stresses.

Open the HyperMesh Model Database

1. Start HyperMesh Desktop and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3110_acuOptiStruct.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Manifold_TFSI` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Manifold_TFSI` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, verify that the Analysis type is set to **Steady State**.
3. Set the Temperature equation to **Advective Diffusive**.
4. Set the Turbulence model to **Spalart Allmaras**.

Name	Value
Title	Exhaust Manifold - TFSI using AcuOptiStruct
Analysis type	Steady State
Flow equation	Navier Stokes
Abs. pressure offset ## (N/m2)	0.0
Temperature equation	Advective Diffusive
Abs. temperature offset ## (K)	0.0
Radiation equation	None
Turbulence model	Spalart Allmaras
Humid Air Model	Off
Multiphase equation	None
Optimization	Off
Mesh type	Fixed
Nucleate boiling single phase	Off
External code	Off
Global Gravity	
X ## (m/sec2)	0.0
Y ## (m/sec2)	0.0
Z ## (m/sec2)	0.0
Unsupported Entries	
No of Analysis Unsupported Entries	0
No of Equation Unsupported Entries	0

Figure 274:

Assign Material Properties and Boundary Conditions

Create a New Material Model

1. In the Solver Browser, expand the **02.Materials** tree.
2. Right-click on **SOLID** and select **Create**.
3. In the Entity Editor, change the name to `Steel`.
4. Set the Density to `8000` kg/m^3.
5. Set the Specific heat to `500` J/kg-K.
6. Set the Conductivity to `16.2` W/m-k.

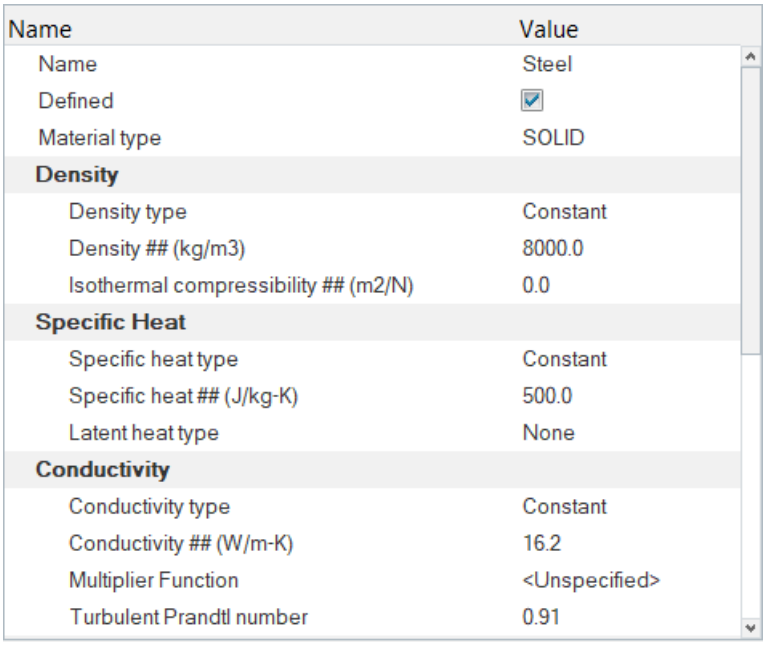


Figure 275:

Assign Material Properties and Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Set the Material to **Air_HM**.

Name	Value
Name	Fluid
Type	FLUID
⊞ Material	(1) Air_HM
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 276:

3. Click **Solid**. In the Entity Editor,

a) Change the Type to **SOLID**.

b) Set the Material to **Steel**.

Name	Value
Name	Solid
Type	SOLID
⊞ Material	(7) Steel
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 277:

4. Click **Inlets**. In the Entity Editor,

a) Change the Type to **INFLOW**.

b) Set the Inflow type to **Velocity**.

c) Verify that the Inflow velocity type is set to **Normal**.

d) Set the Normal velocity to `8.0` m/sec.

e) Set the Temperature to `700` K.

f) Set the Turbulence input type to **Viscosity Ratio**.

A new field for assigning materials appears above the Simple Boundary Condition tab.

g) Set the Material to **Air_HM**.

h) Set the Turbulence viscosity ratio to `40`.

i) Under the Surface Output tab, set the Nodal time step frequency to `100`.

Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will write the nodal surface output at the last time step of the simulation or the 100th time step, whichever is earlier.

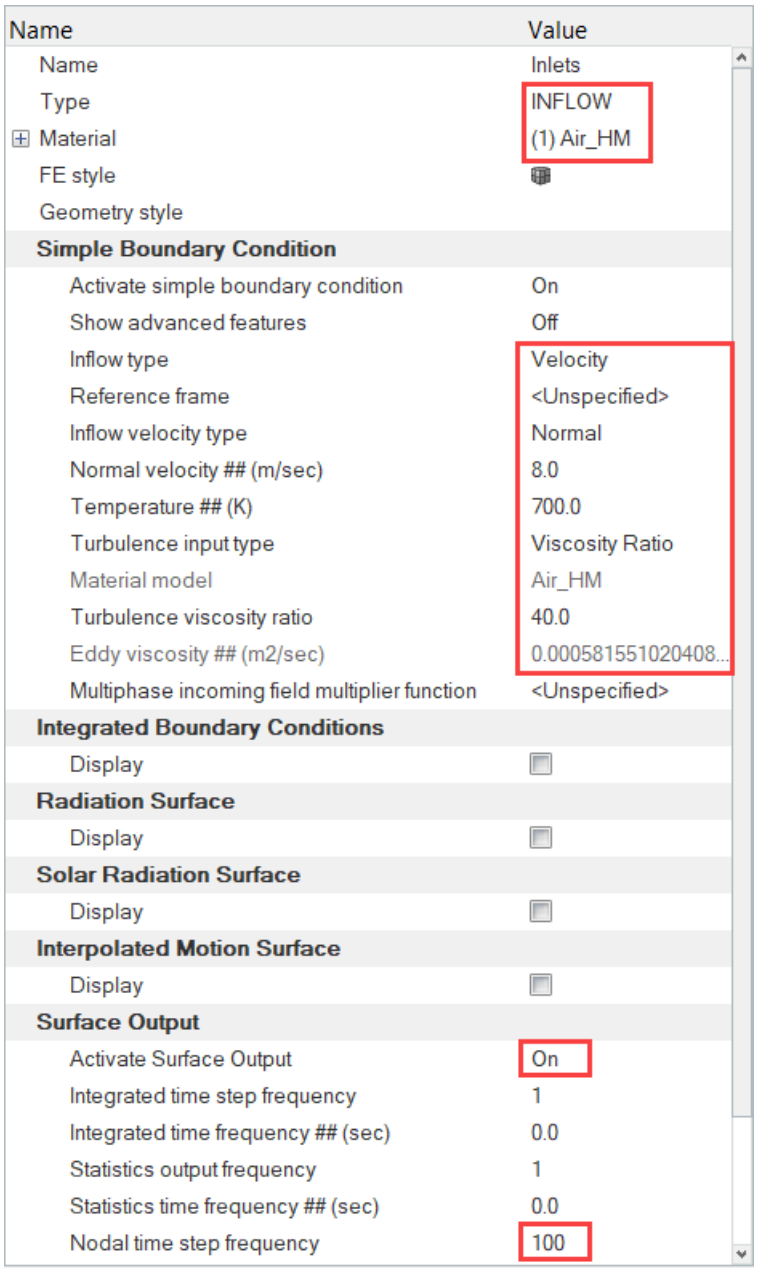


Figure 278:

5. Click **Outlet**. In the Entity Editor,

a) Change the Type to **OUTFLOW**.

b) Under the Surface Output tab, set the Nodal time step frequency to `100`.

Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will write the nodal surface output at the last time step of the simulation or the 100^{th} time step, whichever is earlier.

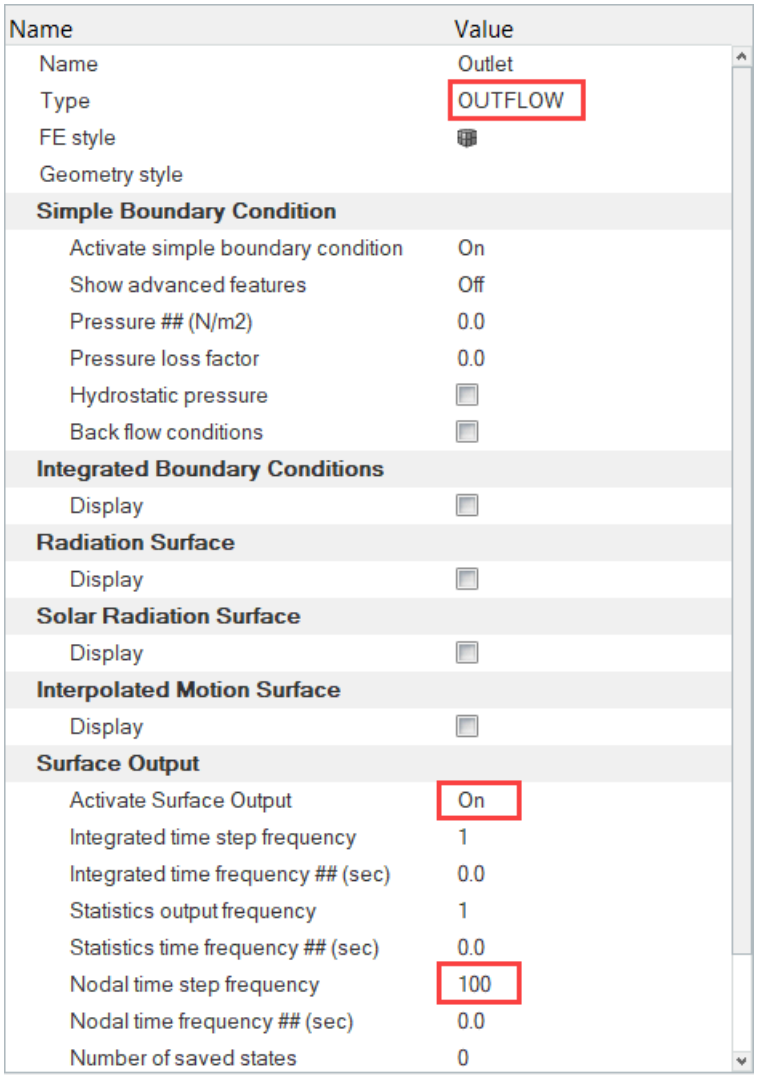


Figure 279:

6. Click **Outlet_End**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Set the Convective heat flux coefficient to `100` J/m^2-sec-K.

c) Set the Convective heat flux reference temperature to `303` K.

d) Under the Surface Output tab, set the Nodal time step frequency to `100`.

Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will

write the nodal surface output at the last time step of the simulation or the 100th time step, whichever is earlier.

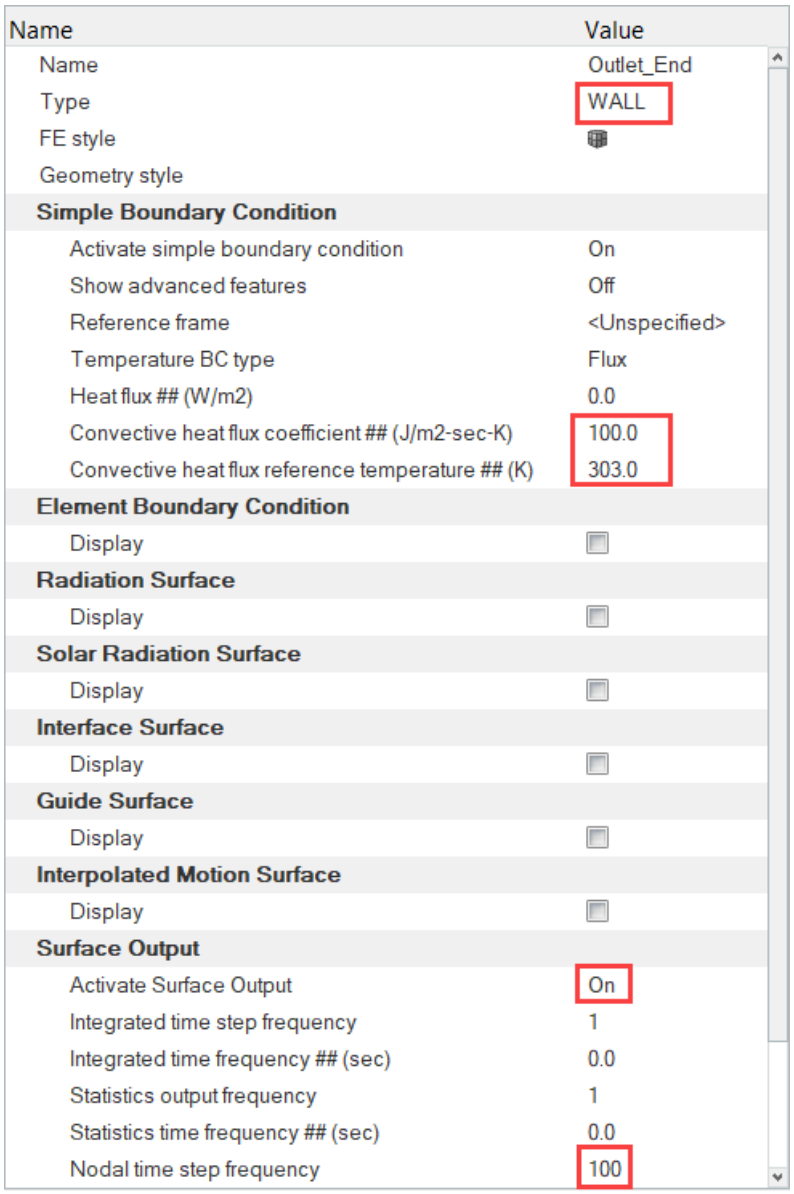


Figure 280:

7. Click **Outer_Wall**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Set the Convective heat flux coefficient to `100` J/m^2-sec-K.
   c) Set the Convective heat flux reference temperature to `303` K.
   d) Under the Surface Output tab, set the Nodal time step frequency to `100`.

      Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will write the nodal surface output at the last time step of the simulation or the 100th time step, whichever is earlier.

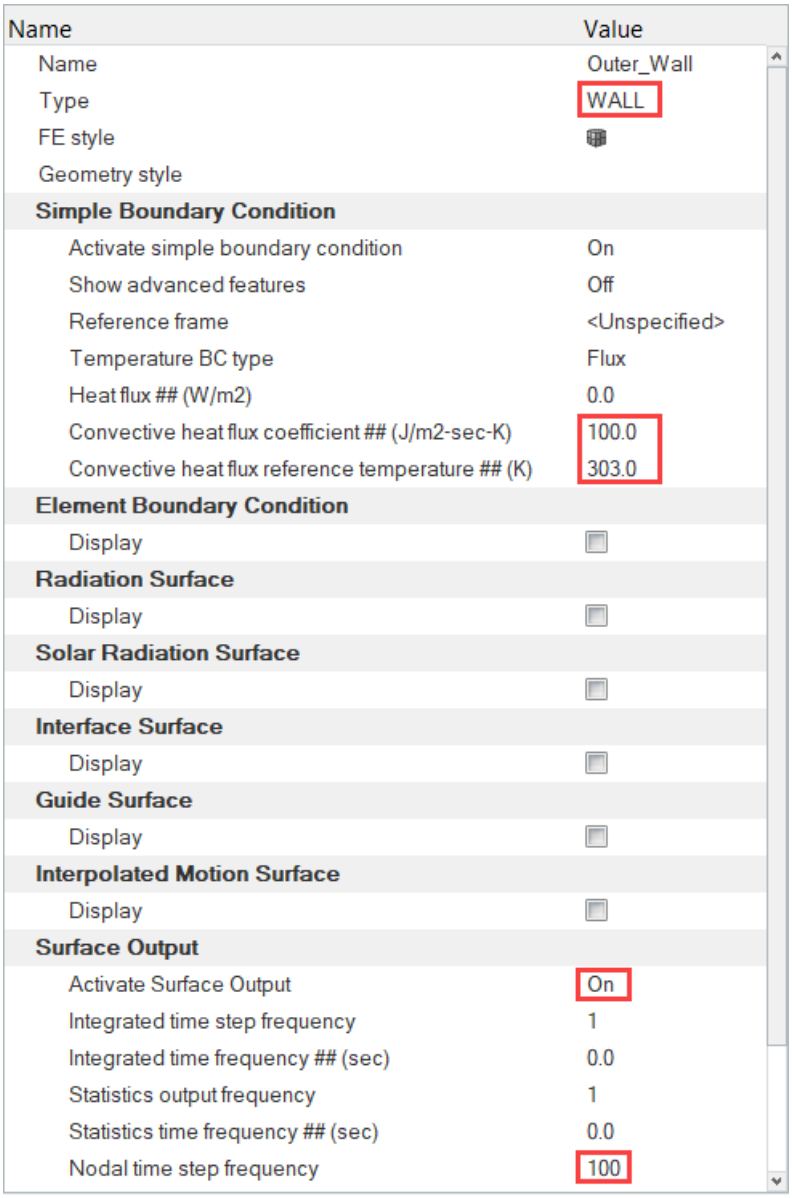


Figure 281:

8. Click **Inner_Wall_Fluid**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Under the Surface Output tab, set the Nodal time step frequency to `100`.

      Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will write the nodal surface output at the last time step of the simulation or the 100^{th} time step, whichever is earlier.

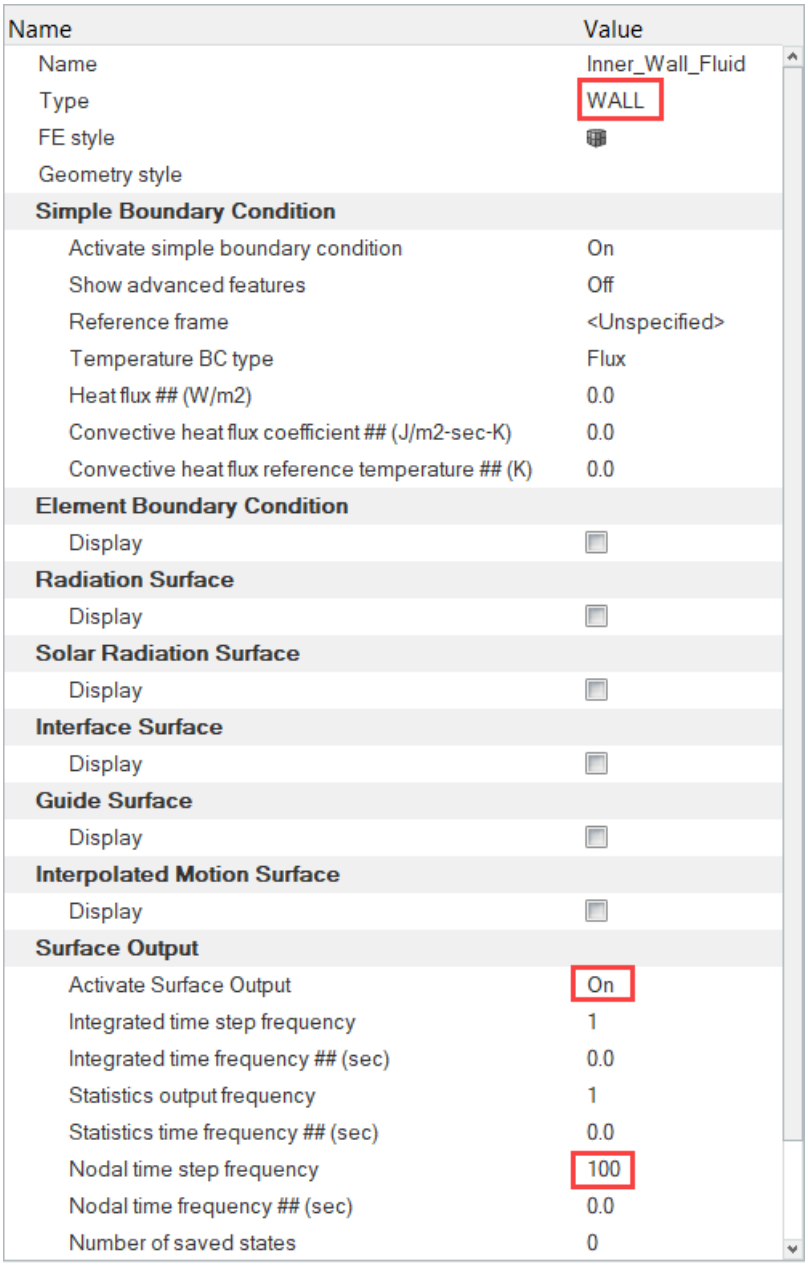


Figure 282:

9. Click **Inner_Wall_Solid**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Under the Surface Output tab, set the Nodal time step frequency to `100`.

Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will write the nodal surface output at the last time step of the simulation or the 100^{th} time step, whichever is earlier.

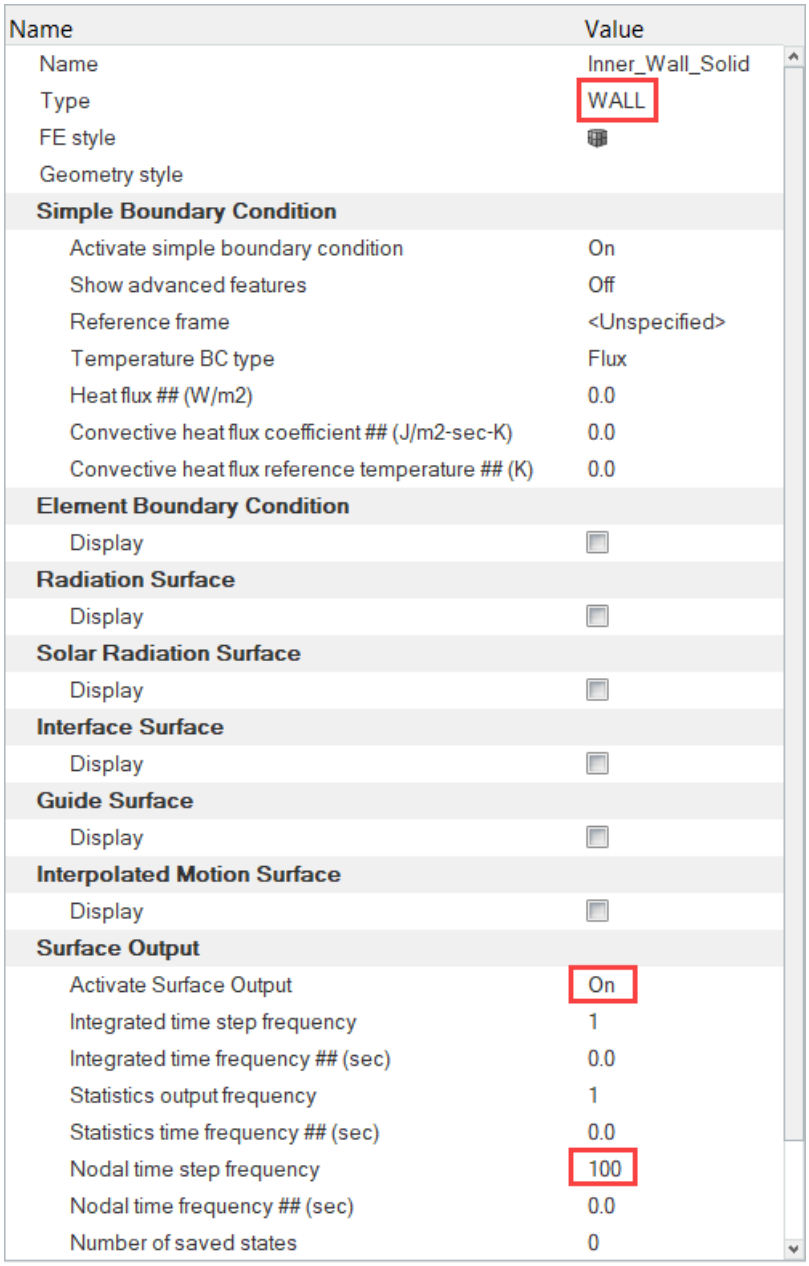


Figure 283:

10. Click **Flange**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Set the Convective heat flux coefficient to `100` J/m^2-sec-K.

c) Set the Convective heat flux reference temperature to `303` K.

d) Under the Surface Output tab, set the Nodal time step frequency to `100`.

Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will write the nodal surface output at the last time step of the simulation or the 100th time step, whichever is earlier.

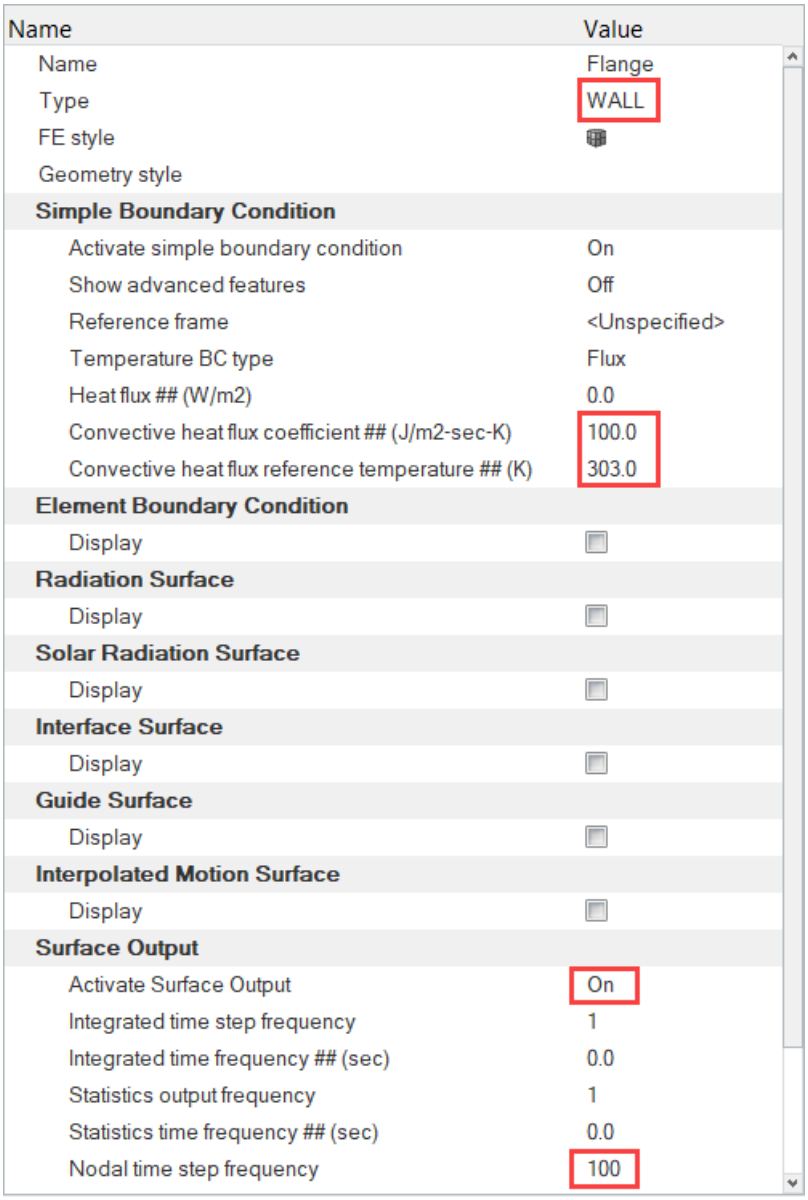


Figure 284:

11. Click **Flange_Bolts**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Set the Convective heat flux coefficient to `100` J/m^2-sec-K.

c) Set the Convective heat flux reference temperature to `303` K.

d) Under the Surface Output tab, set the Nodal time step frequency to `100`.

Since this is a steady state simulation, acuOptiStruct needs the nodal surface output for each surface at the final time step. By setting the Nodal time step frequency to 100, AcuSolve will write the nodal surface output at the last time step of the simulation or the 100th time step, whichever is earlier.

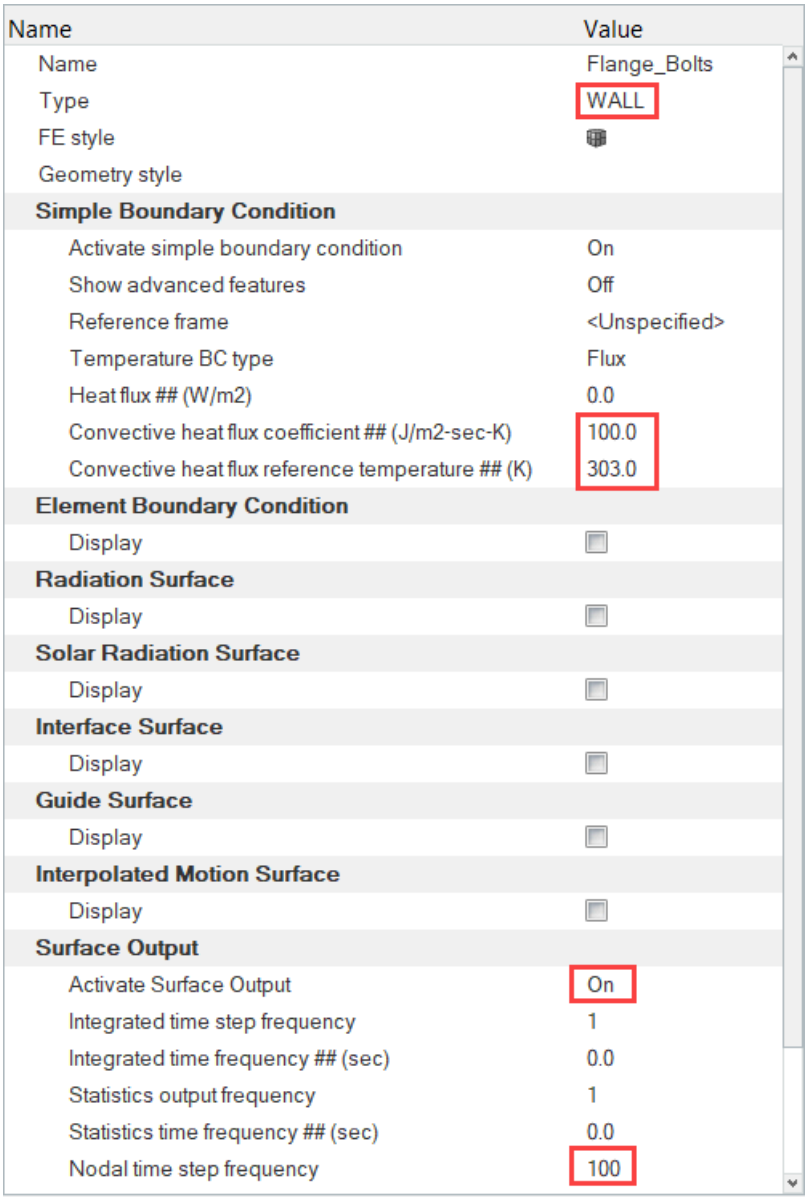


Figure 285:

12. On the menu bar, go to **BCs** > **Components** > **Auto Wall** > **Deactivate**. Make sure that the Auto Wall is deactivated for all the wall surfaces.

To create an OptiStruct solver deck, acuOptiStruct needs a Simple Boundary Condition and Surface Output defined on all the surfaces, including the fluid and solid side of the interfaces. Hence, auto wall should not be used for any wall surfaces while using the acuOptiStruct utility.

13. In the Solver Browser, click the **Inner_Wall_Fluid** component. In the Entity Editor. activate the Interface Surface **Display** option and turn **On** the interface surface.

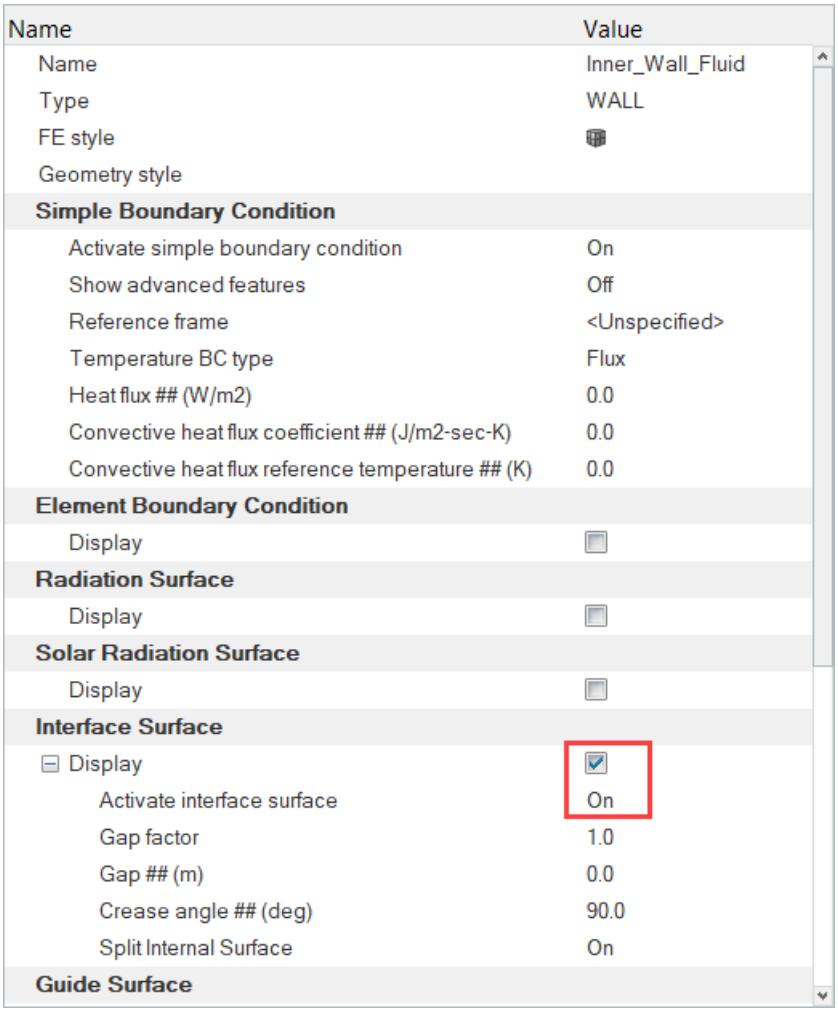


Figure 286:

14. Similarly, click the **Inner_Wall_Solid** component, activate the Interface Surface **Display** option, and turn **On** the interface surface.

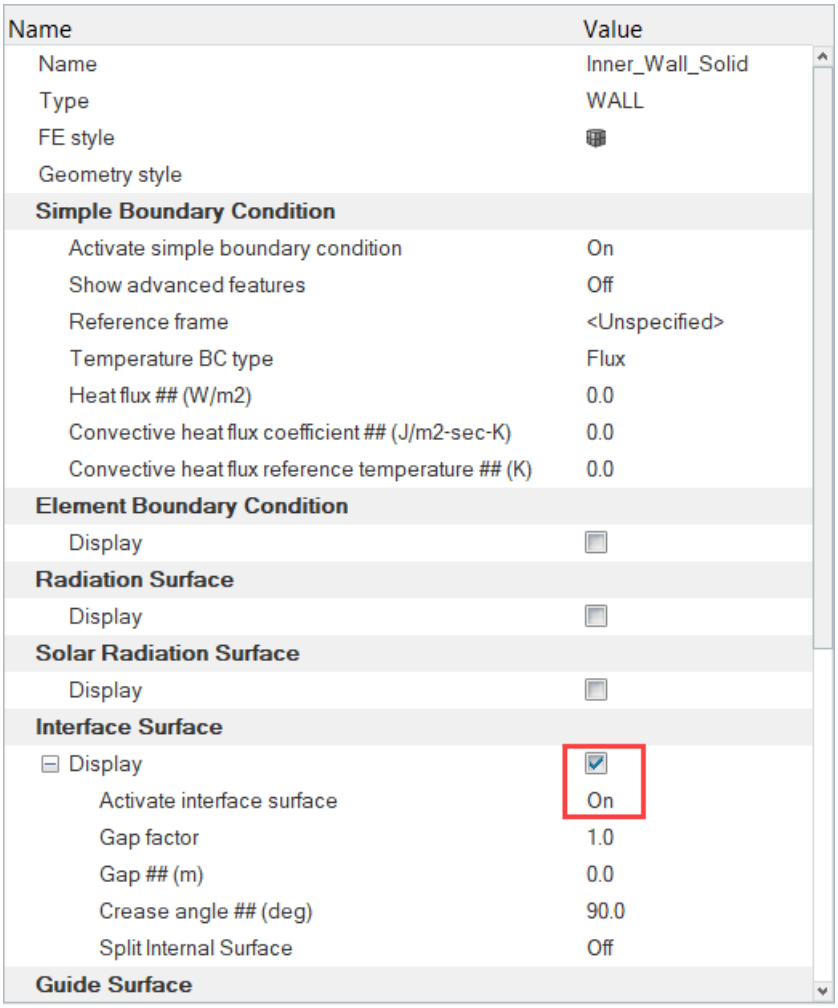


Figure 287:

15. Save the model.

Define the Nodal Output Frequency

1. In the Solver Browser, expand **17.Output** and then click on **NODAL_OUTPUT**.

2. In the Entity Editor, verify that the Time step frequency is set to `1000`.

This means the nodal output will be stored at the last time step, or the time step at which the solution converges, whichever is earlier.

> **Note:** For transient simulations, the nodal output frequency should be set to 1 for both global nodal output and the individual nodal surface output.

3. Save the model.

Compute the Solution

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.

3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.

4. Activate the **Export options** checkbox then turn off the **Always two layers for interfaces** checkbox.

   Since you split the interface between the fluid and domain already, this option need not be activated.

5. Leave the remaining options as default and click **Launch** to start the solution process.

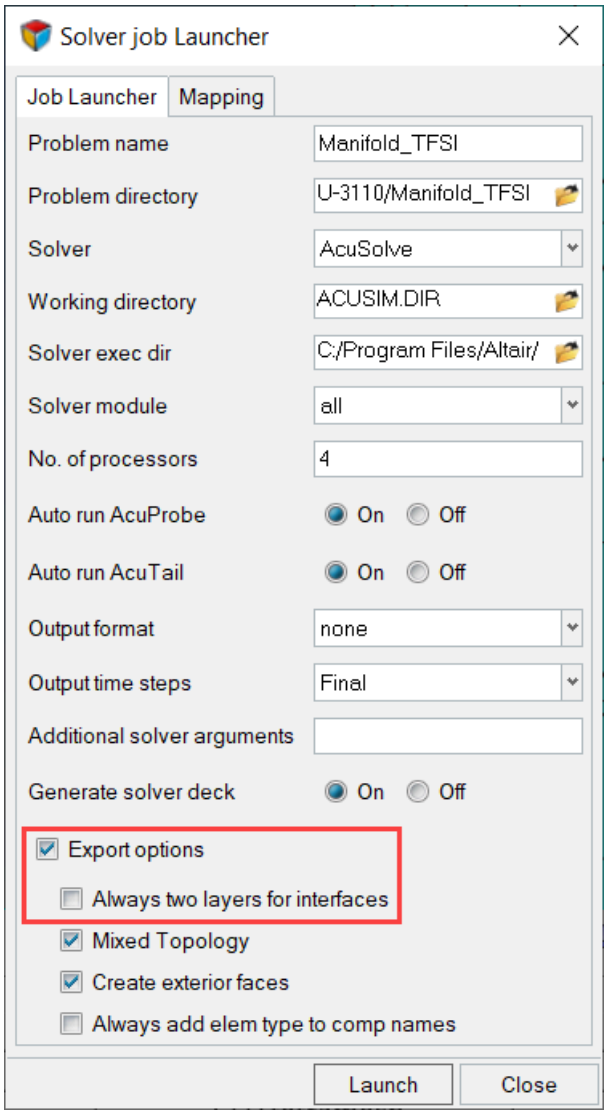


Figure 288:

Once you hit the Launch button, the AcuTail and AcuProbe windows are launched automatically. A summary of the run in the AcuTail window indicates that the solver run is complete. Once the run is compete, you can close the AcuTail and AcuProbe windows.

```
acuSolve: ---------- End Time Step ----------
acuSolve: Input       CPU/Elapse time= 3.780000e+000 9.590000e+000 Sec
acuSolve: Output      CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Exchange    CPU/Elapse time= 2.599000e+000 1.763000e+000 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =         22         22
acuSolve:   Elem.Form CPU/Elapse time= 8.995500e+001 5.029900e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 6.058760e+002 3.106060e+002 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:   No of res/lhs formations =         22         22
acuSolve:   Elem.Form CPU/Elapse time= 4.705800e+001 2.388300e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 3.903000e+001 1.984000e+001 Sec
acuSolve: Temperature stagger "temperature":
acuSolve:   No of res/lhs formations =         22         22
acuSolve:   Elem.Form CPU/Elapse time= 4.719000e+001 2.375000e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 2.376110e+002 1.214330e+002 Sec
acuSolve: Pressure proj. elapse  time= 1.439810e+002 Sec
acuSolve: Linear solver  elapse  time= 2.855720e+002 Sec
acuSolve: Total CPU/Elapse time      = 1.244311e+003 6.538740e+002 Sec
acuSolve: Total Memory Usage         = 4.924766e+002 Mbytes
acuRun: -*-*-*-
```

This command will generate an OptiStruct input deck from the temperature filed of the flow solution by using the specified constraint surfaces and their degrees of freedom. The analysis type is set to steady linear.

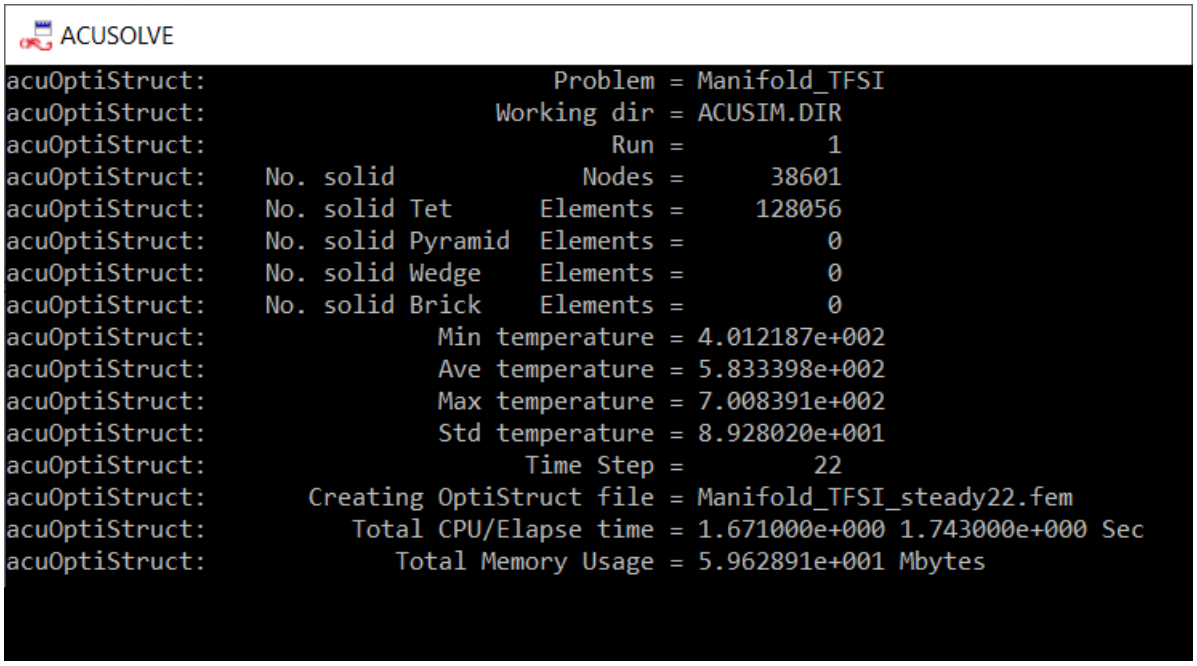


Figure 290:

4. Start OptiStruct from the Windows Start menu by clicking **Start** > **Altair <version>** > **OptiStruct**.
5. Click the next to Input File(s).
6. Browse to the location that you use as your working directory.
7. Select the `.fem` file.

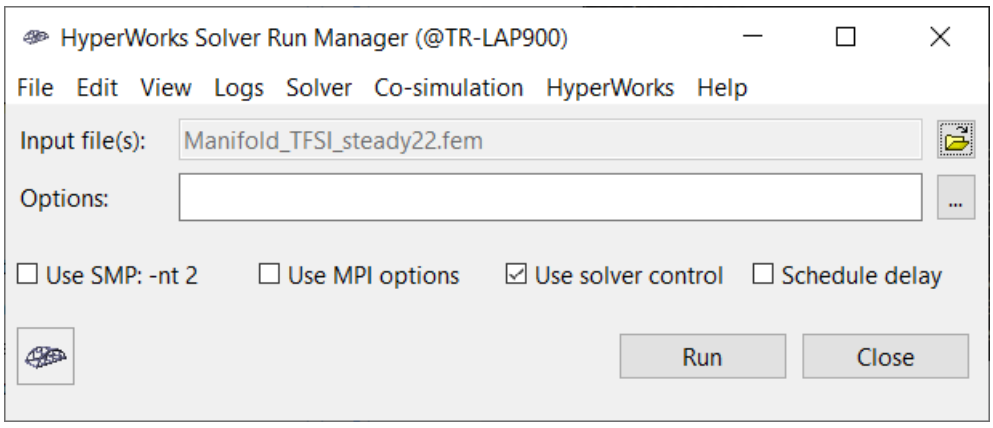


Figure 291:

8. Click **Run** to run the case.

Once the run is complete, the HyperWorks Solver View will show "OptiStruct Job Completed" in the Run summary window.

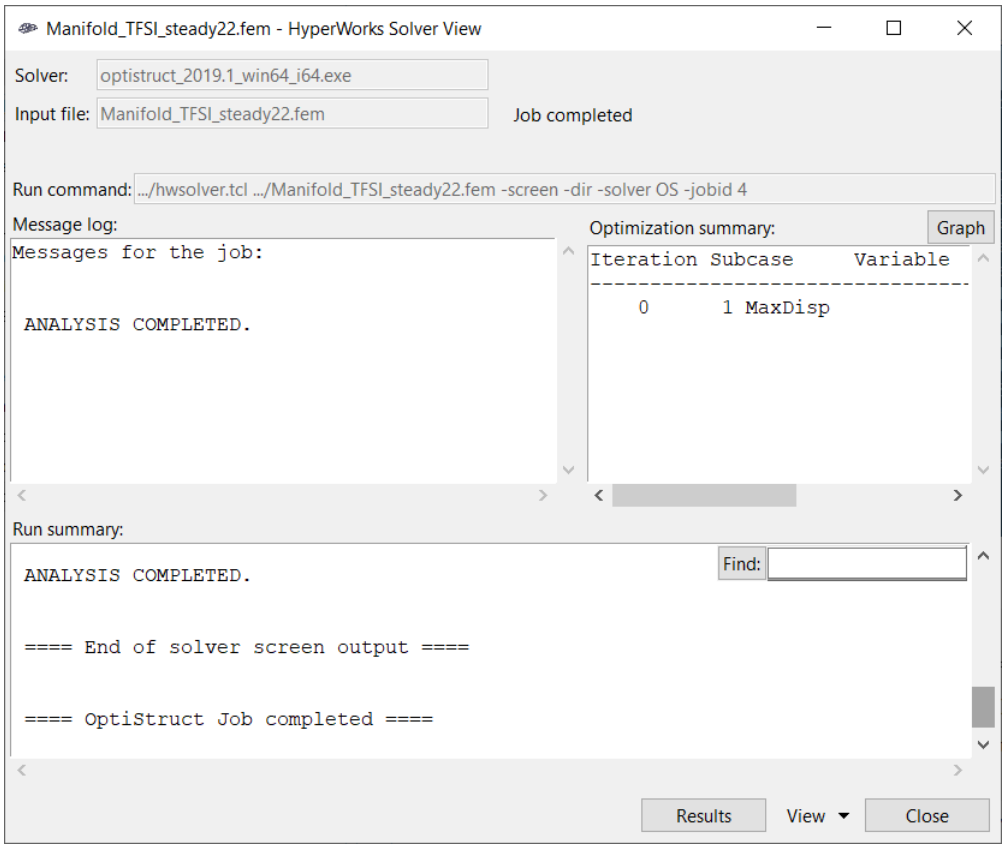


Figure 292:

Post-Process the Results with HyperView

Once the OptiStruct run is complete, close the HyperWorks Solver View dialog. In the HyperMesh Desktop window, close the AcuSolve Control and Solver job Launcher dialogs. In the next few steps, you will plot the contour plot of temperature and pressure on the fluid domain and the displacement and stress contours on the solid domain.

Switch to the HyperView Interface and Load the AcuSolve Model and Results

1. In the HyperMesh Desktop window, click the **ClientSelector** drop-down in the bottom-left corner of the graphics window.

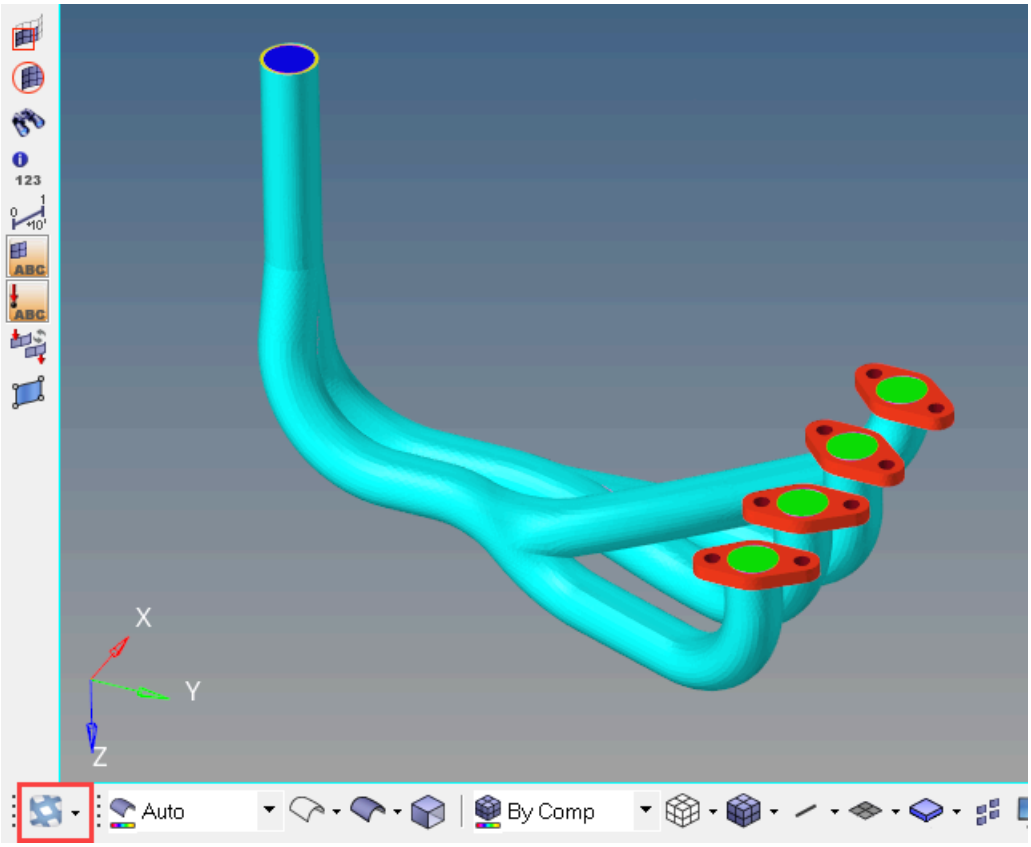


Figure 293:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.
   The interface is changed to HyperView.

   Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Manifold_TFSI.1.Log`.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create a Contour Plot of Temperature and Pressure

1. Click on the Results toolbar to open the Contour panel.
2. In the panel area, set the Result type to **Temperature (s)**.
3. In the panel area, click **Apply** to plot the temperature contours.

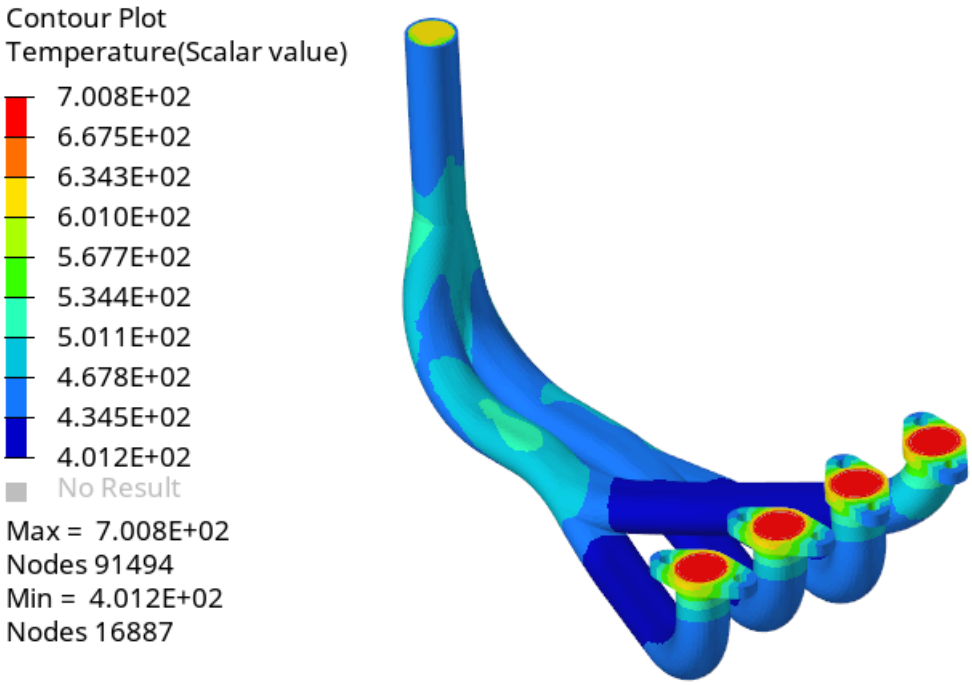


Figure 294:

4. In the Results Browser, expand the list of **Components**.
5. Click the Isolate Shown icon then click the **Inner_Wall_Fluid** component to turn off the display of all components in the graphics window except the Inner Wall component.
6. In the panel area, change the Result type to **Pressure (s)**.
7. Click **Apply** to plot the pressure contours.

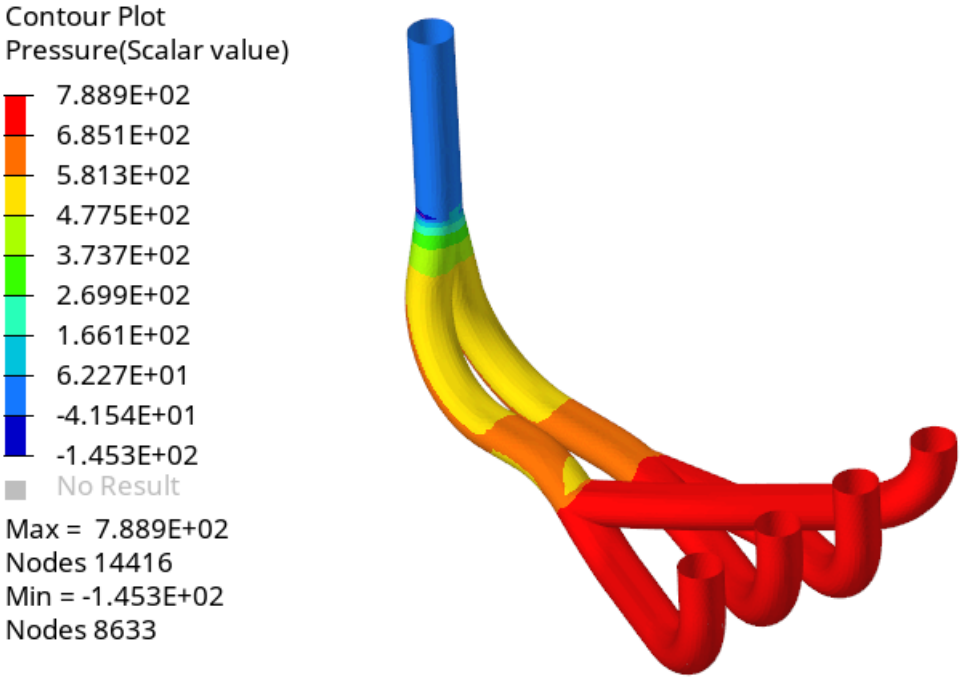


Figure 295:

Load the OptiStruct Results

1. Click on the drop-down beside the Page Window Layout icon on the **PageControls** toolbar.

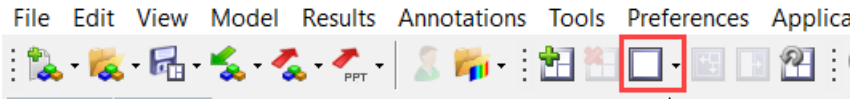


Figure 296:

2. In the drop-down menu, select the vertical 2-window layout icon.

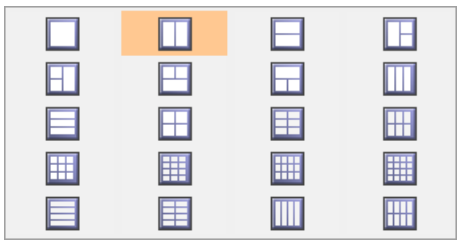


Figure 297:

3. Click in the newly created graphics window then click the **Load Results** icon.
4. In the Load model and results panel, click next to Load model.

5. In the **Load Model File** dialog, navigate to your working directory and select **Manifold_TFSI_steady23.h3d**.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results from the OptiStruct results file.

   Observe that this model contains only the solid domain since only the solid is included while generating the OptiStruct solver deck.

Create a Contour Plot of Displacement and Element Stresses

1. Click on the Results toolbar to open the Contour panel.
2. In the panel area, set the Result type to **Displacement (v)**.
3. Click **Apply** to plot the displacement magnitude contours.

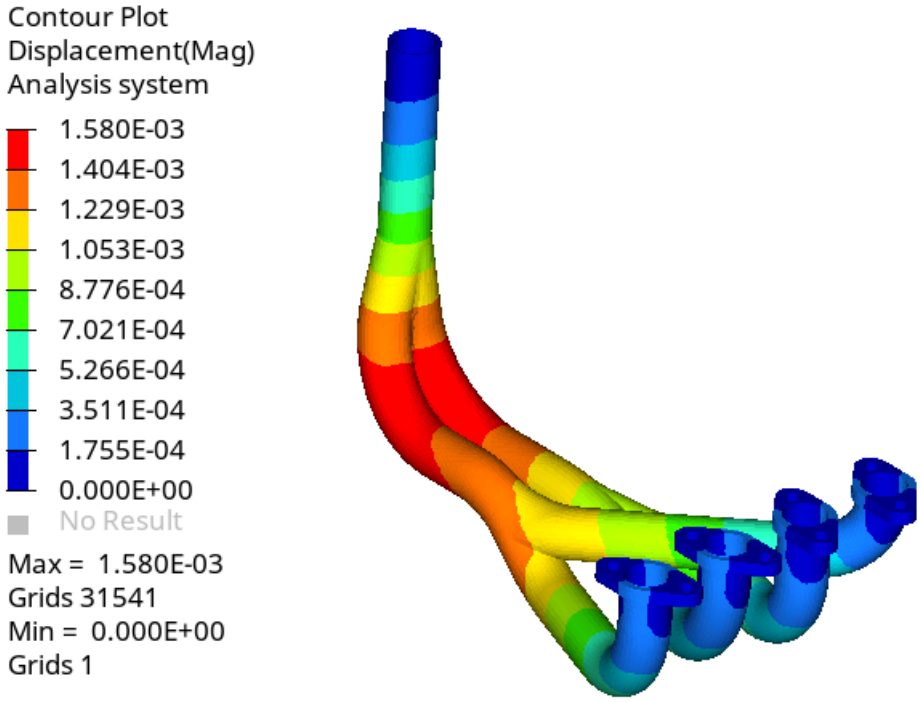


Figure 298:

4. In the panel area, change the Result type to **Element Stresses (2D & 3D) (t)** and select **vonMises** from the drop-down below.
5. Click **Apply**.

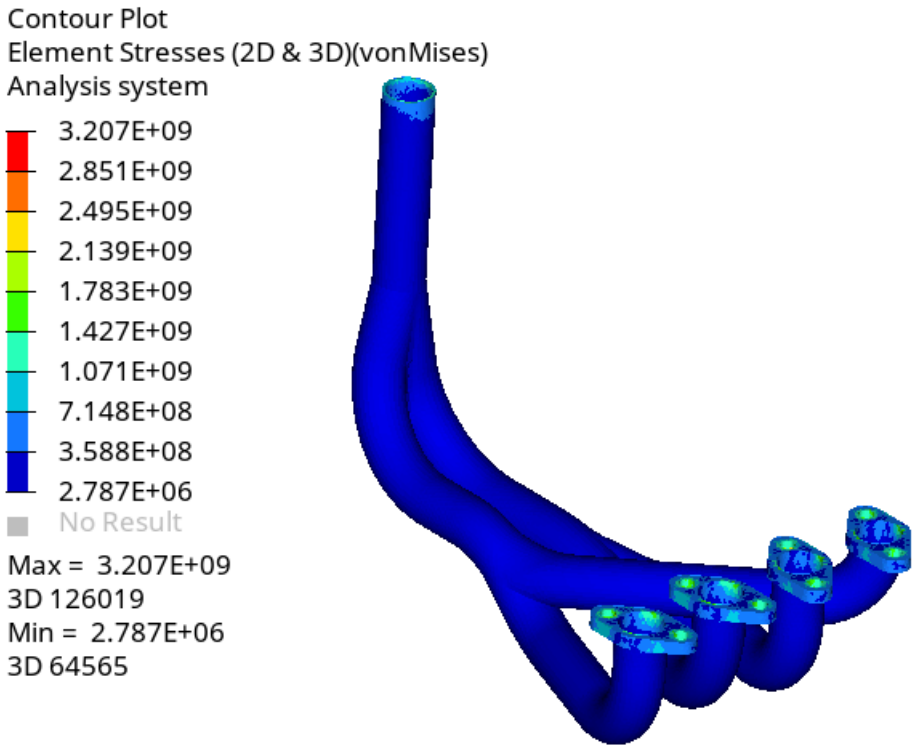


Figure 299:

Summary

In this tutorial you learned how to set up a conjugate heat transfer problem using HyperMesh and solve it using AcuSolve. Once you computed the solution, you used acuOptiStruct to generate the input deck for OptiStruct. Once the solution for the structural analysis was computed, you post-processed the results using HyperView and created contour plots of Temperature, Pressure, Displacement and Stress.

2.3.5 ACU-T: 3200 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model

Prerequisites

This tutorial introduces you to setting up a radiation heat transfer problem using the Enclosure radiation model in HyperMesh and solving using AcuSolve. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files`

`\tutorials\AcuSolve` to a local directory. Extract `ACU-T3200_HeadlampEnclosure.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be solved is shown schematically in Figure 300 and Figure 301. It consists of a simple headlamp with a housing, lens, and a bulb. The inner cavity of the bulb is filled with air and the wattage of the bulb is 1W, which is modeled as a volumetric heat source. The Boussinesq density model is used for the air to consider the natural convection effects in the fluid volume. The heat generated in the bulb is transferred by three means: conduction from the bulb to the housing, natural convection in the air volume, and radiation from the bulb surface to other surfaces. The external reference temperature is 300 K for the outer surfaces of the headlamp. You will use the Enclosure radiation model to simulate the surface to surface radiation.

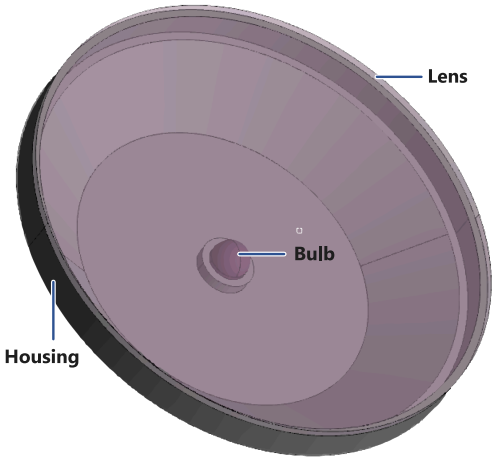


Figure 300:

The enclosure radiation methodology in AcuSolve involves a two-step process: view factor computation and heat flux addition. View factor is the proportion of radiation incident one surface due to another surface. The view factors are computed, and the radiative heat fluxes are added to the energy equation during the solver run. These radiative heat fluxes are computed based on the view factors using the Stefan-Boltzmann law. The enclosure radiation model is supported only on fluid mediums.

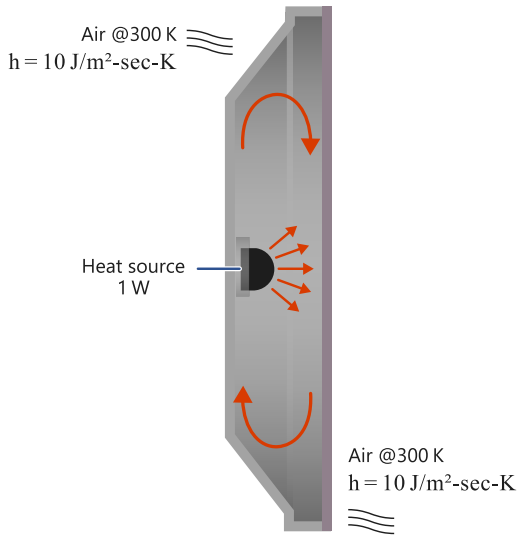


Figure 301:

Open the HyperMesh Model Database

1. Start HyperMesh Desktop and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3200_HeadlampEnclosure.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Headlamp_Enclosure` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Headlamp_Enclosure` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, verify that the Analysis type is set to **Steady State**.
3. Set the Temperature equation to **Advective Diffusive**.

4. Set the Radiation equation to **Enclosure**.
5. Set the Turbulence model to **Laminar** (if not set already).

Name	Value
Title	Radiation in a Headlamp using enclosure model
Analysis type	Steady State
Flow equation	Navier Stokes
Abs. pressure offset ## (N/m2)	0.0
Temperature equation	Advective Diffusive
Abs. temperature offset ## (K)	0.0
Radiation equation	Enclosure
Turbulence model	Laminar
Humid Air Model	Off
Multiphase equation	None
Optimization	Off
Mesh type	Fixed
Nucleate boiling single phase	Off
External code	Off
Global Gravity	
X ## (m/sec2)	0.0
Y ## (m/sec2)	0.0
Z ## (m/sec2)	0.0
Unsupported Entries	
No of Analysis Unsupported Entries	0
No of Equation Unsupported Entries	0

Figure 302:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, turn **On** Temperature flow.
3. Leave the remaining options unchanged.

Name	Value
Max time steps	100
Initial time increment ## (sec)	10000000000
Convergence tolerance	0.001
Relaxation factor	0.3
Flow	On
Temperature	On
Temperature flow	On
Radiation	On
Number of Kyrlov Vector	10
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 303:

Define the Material Models and the Body Force

Define the Material Models

1. In the Solver Browser, expand **02.Materials** > **Fluid** then click on **Air_HM**.
2. In the Entity Editor, change the Density type to **Boussinesq**.
3. In the Solver Browser, under 02.Materials, right-click on **SOLID** and select **Create**.
4. In the Entity Editor, name it `Arnite`.
5. Set the Density to `1670` kg/m^3.
6. Set the Specific heat to `2050` J/kg-K.
7. Set the Conductivity to `1.65` W/m-K.
8. Repeat steps 3-7 and create two more solid material models, `Plastic` and `LED`, with the following material properties:
   1. Plastic:
      a. Density - `1270` kg/m^3
      b. Specific heat - `1900` J/kg-K
      c. Conductivity to `0.22` W/m-K
   2. LED:
      a. Density - `5500` kg/m^3
      b. Specific heat - `0.3` J/kg-K
      c. Conductivity to `5.0` W/m-K

9. Save the database.

Define the Body Force

1. In the Solver Browser, expand **03.Body_Force** > **BODY_FORCE** then click on **Gravity_HM**.
2. In the Entity Editor, set the Y-Gravity to `-9.81` m/sec^2 and change to Z-Gravity to `0`.

Name	Value
Name	Gravity_HM
Type	BODY_FORCE
Medium	Fluid
Gravity	
Gravity type	Constant
X ## (m/sec2)	0.0
Y ## (m/sec2)	-9.81
Z ## (m/sec2)	0.0
Gravity multiplier function	<Unspecified>
Heat Source	
Heat source unit type	Per unit mass
Heat source type	Constant
Mass heat source ## (W/kg)	0.0
Heat source multiplier function	<Unspecified>
Unsupported Entries	

Figure 304:

Define the Heat Source

1. In the Solver Browser, right-click on **03.Body_Force** and select **Create**.
2. In the Entity Editor, name it `LED Heat Source`.
3. Change the Medium to **Solid**.
4. Set the Heat source unit type to **Per unit volume**.
5. Set the Heat Source type to **Constant** and set the Volumetric heat source to `2049180` W/m^3.

Name	Value
Name	LED Heat Source
Type	BODY_FORCE
Medium	Solid
Heat Source	
Heat source unit type	Per unit volume
Heat source type	Constant
Volumetric heat source ## (W/m3)	2049180.0
Heat source multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

Figure 305:

Set the Boundary Conditions

Create the Emissivity Model

1. In the Solver Browser, right-click on **07.Emissivity_Model** and select **Create**.
2. In the Entity Editor, name it `Walls`.
3. Set the Emissivity to `0.7`.

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Air**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Set the Material to **Air_HM**.
   c) Set the Body force to **Gravity_HM**.

Name	Value
Name	Air
Type	FLUID
Material	(1) Air_HM
FE style	
Geometry style	
Element Set	
Body force	(1) Gravity_HM
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 306:

3. Click **Housing**. In the Entity Editor,
   a) Change the Type to **SOLID**.
   b) Set the Material to **Plastic**.

Name	Value
Name	Housing
Type	SOLID
Material	(8) Plastic
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 307:

4. Click **Bulb**. In the Entity Editor,
   a) Change the Type to **SOLID**.
   b) Set the Material to **LED**.
   c) Set the Body force to **LED Heat Source**.

Name	Value
Name	Bulb
Type	SOLID
Material	(9) LED
FE style	
Geometry style	
Element Set	
Body force	(2) LED Heat Source
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 308:

5. Click **Lens**. In the Entity Editor,
   a) Change the Type to **SOLID**.
   b) Set the Material to **Arnite**.

Name	Value
Name	Lens
Type	SOLID
Material	(7) Arnite
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 309:

6. Click **Lens-inner**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Under the Radiation Surface tab, activate the **Display** checkbox, set the Activate radiation surface field to **On**, set the Type to **Wall**, and set the Emissivity model to **Walls**.

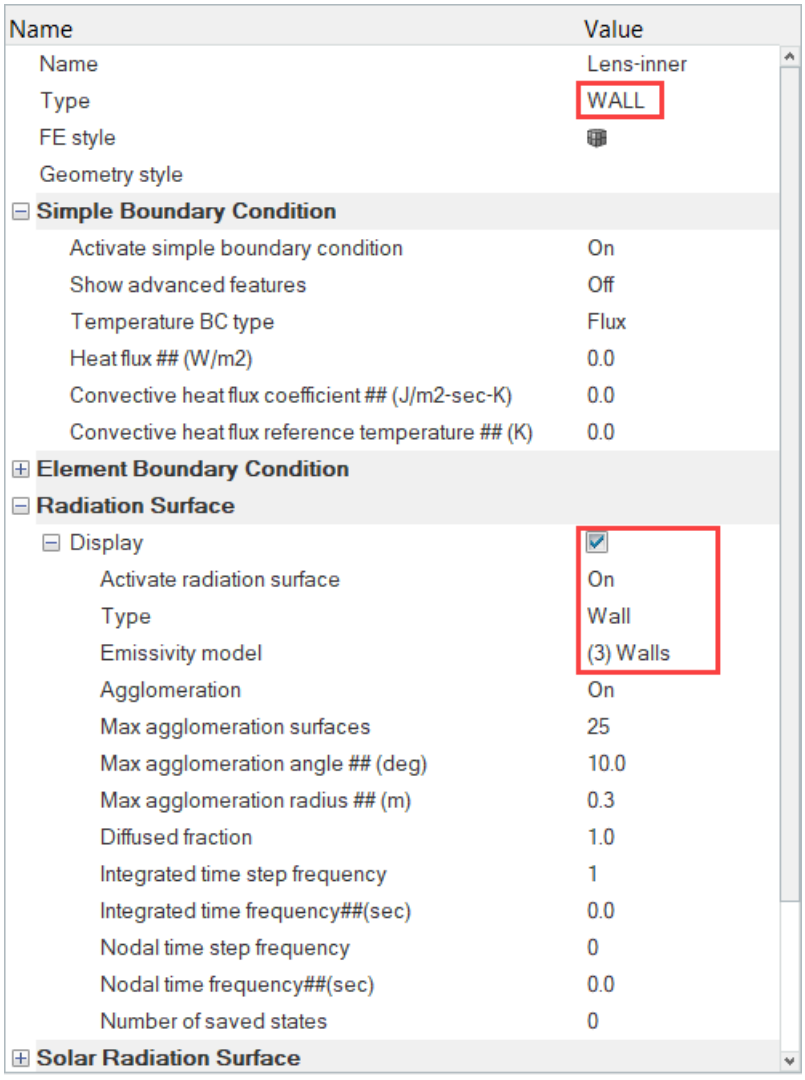


Figure 310:

7. Click **Lens-outer**. In the Entity Editor,

a) Verify that the Type is set to **WALL** and the Temperature BC type is set to **Flux**.

b) Set the Convective heat flux coefficient to `10` J/m^2-sec-K.

c) Set the Convective heat flux reference temperature to `300` K.

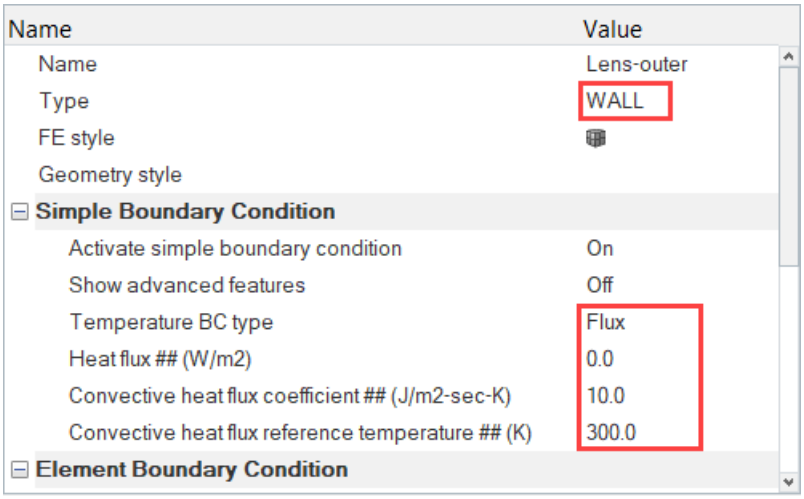


Figure 311:

8. Click **Housing-inner**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Under the Radiation Surface tab, activate the **Display** checkbox, set the Activate radiation surface field to **On**, set the Type to **Wall**, and set the Emissivity model to **Walls**.

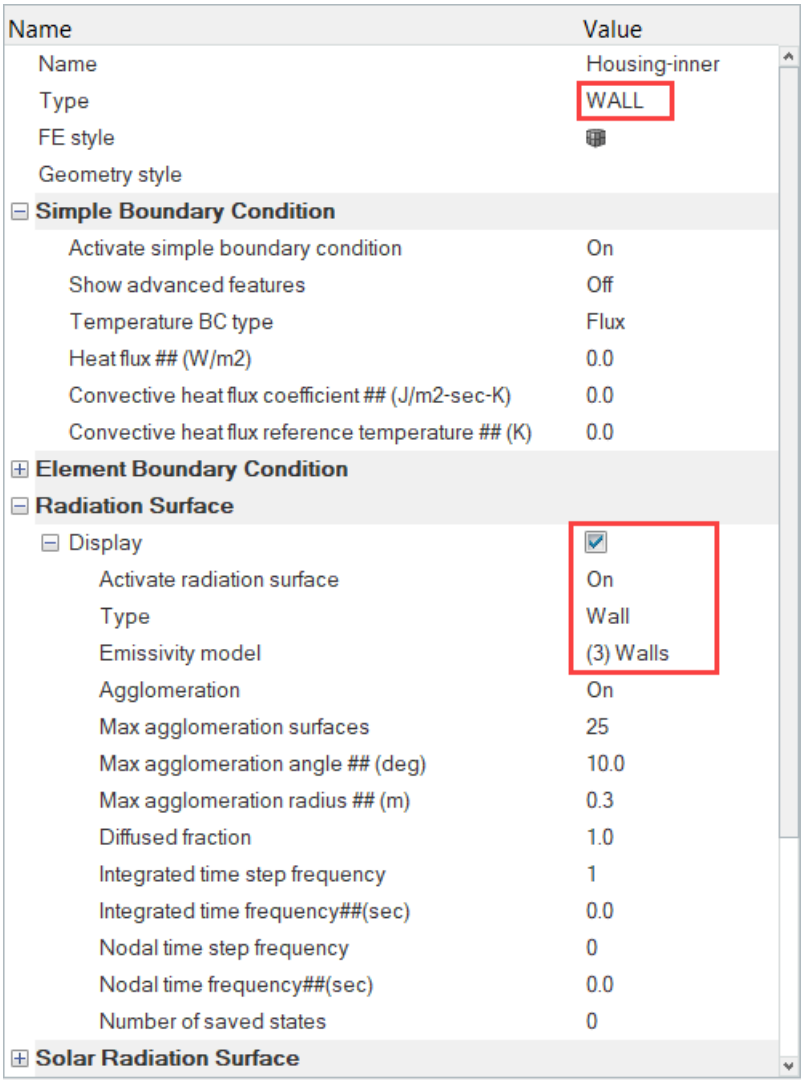


Figure 312:

9. Click **Housing-outer**. In the Entity Editor,

a) Verify that the Type is set to **WALL** and the Temperature BC type is set to **Flux**.

b) Set the Convective heat flux coefficient to `10` J/m^2-sec-K.

c) Set the Convective heat flux reference temperature to `300` K.

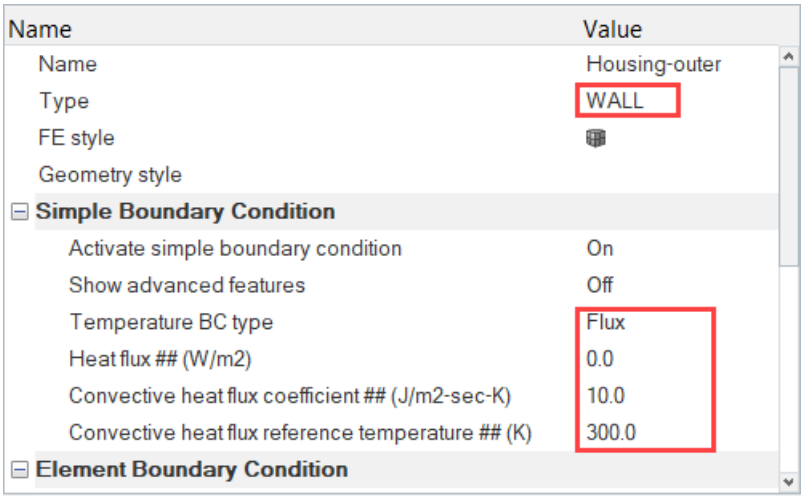


Figure 313:

10. Click **Bulb-walls**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Under the Radiation Surface tab, activate the **Display** checkbox, set the Activate radiation surface field to **On**, set the Type to **Wall**, and set the Emissivity model to **Walls**.

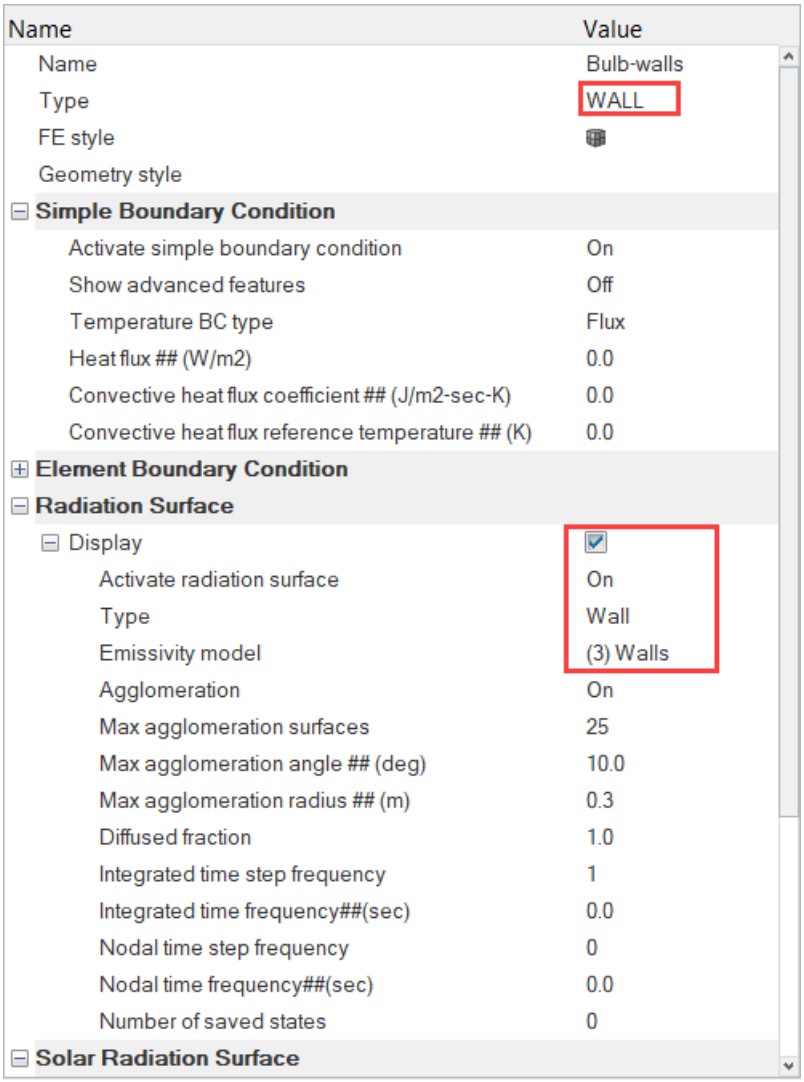


Figure 314:

11. Save the model.

Compute the Solution

Run AcuSolve

In this step, you will launch AcuSolve to compute a solution for this case.

1. Turn on the visibility of all mesh components.
For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.
The **Solver job Launcher** dialog opens.

3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Leave the remaining options as default and click **Launch** to start the solution process.

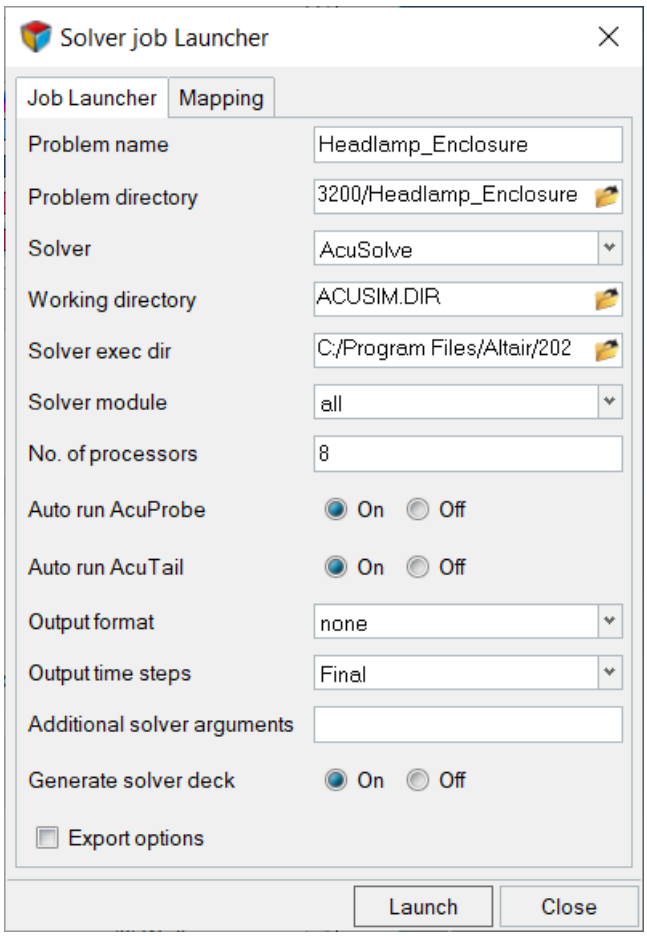


Figure 315:

Monitor the Solution with AcuProbe

While AcuSolve is running, you can monitor the progress of the solution using AcuProbe and plot the values of residual ratios, solution ratios, and variables like temperature, heat flux, etc. Once the solver run has started, the AcuTail and AcuProbe windows should be launched automatically.

1. In the AcuProbe window, under the Data Tree, expand **Residual Ratio**.
2. Right-click on **All** and select **Plot All**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

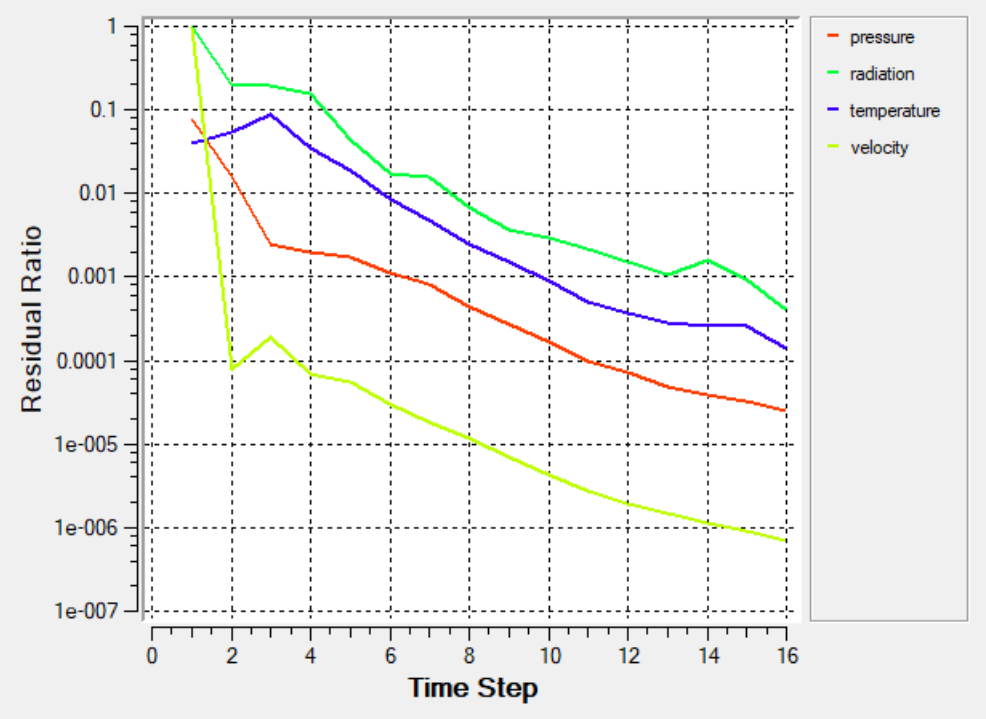


Figure 316:

Once the Solution has converged, close the AcuTail and AcuProbe windows.

Post-Process the Results with HyperView

In this step, you will visualize the results using HyperView. While doing so, you will create contour plots of temperature and velocity magnitude on a section cut and also create a plot of velocity vectors. Once the solver run is complete, close the AcuProbe and AcuTail windows. In the HyperMesh Desktop window, close the AcuSolve Control tab and save the model.

Switch to the HyperView Interface and Load the AcuSolve Model and Results

1. In the HyperMesh Desktop window, click the **ClientSelector** drop-down in the bottom-left corner of the graphics window.

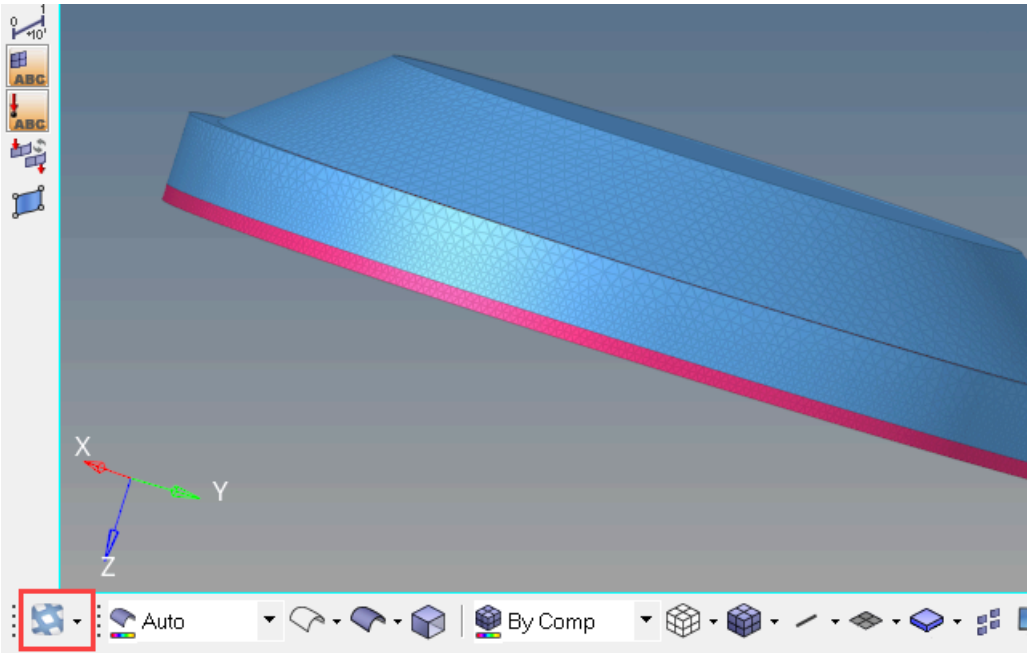


Figure 317:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.
   The interface is changed to HyperView.

   Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Headlamp_Enclosure.1.Log`.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create a Contour Plot of Temperature

1. In the Results Browser, expand the list of **Components** then click the **Isolate Shown** icon .
2. Hold Ctrl and select the **AUTO Housing wall** and **AUTO Bulb wall** components to turn off the display of all the components except the walls of the bulb and housing components.

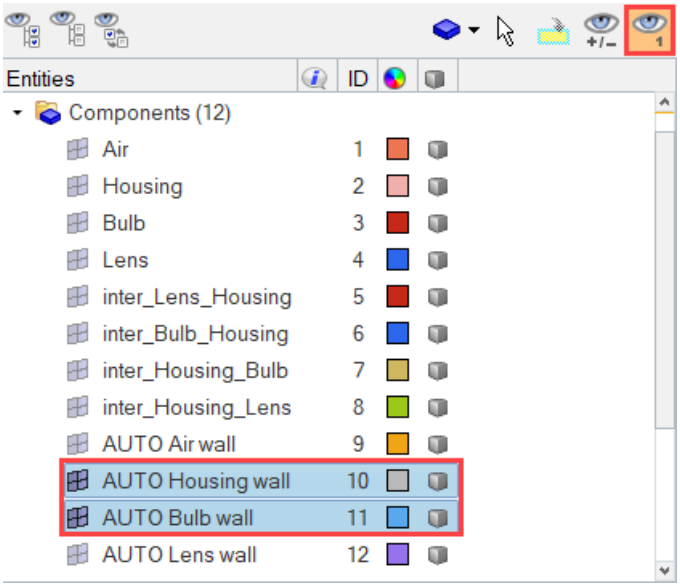


Figure 318:

3. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
4. Click on the Results toolbar to open the Contour panel.
5. In the panel area, set the Result type to **Temperature (s)**.
6. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
7. Click **Apply**.
8. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 319:

9. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.

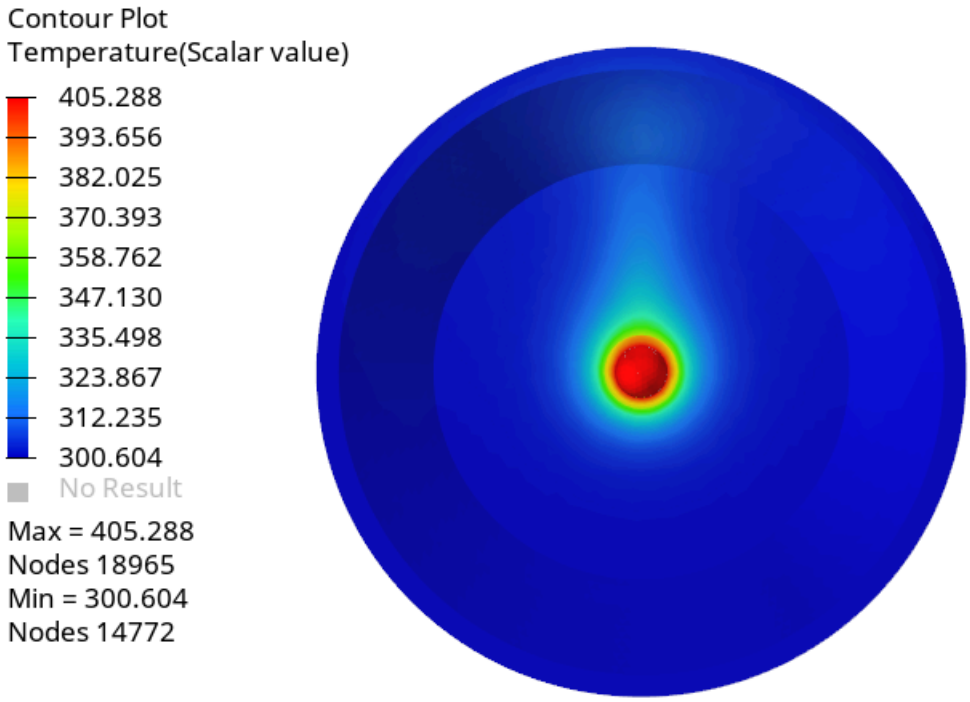


Figure 320:

Display Temperature Contours and Velocity Vectors on a Section Cut

In this step, you will create a section cut on the mid-z plane and then display the temperature and velocity vectors on that cross section.

1. In the Results Browser, turn on the display of all the components.
2. Click the **Section cut** icon icon on the HV-Display toolbar.
3. In the panel area, click **Add** to create a new section cut named Section 1.
4. In the Define plane section, set the axis to **Z Axis** then click **Apply**.
5. Set the Z base coordinate to `0.0005` then press **Enter**.
6. Change the Display options from Clipping plane to **Cross section**.

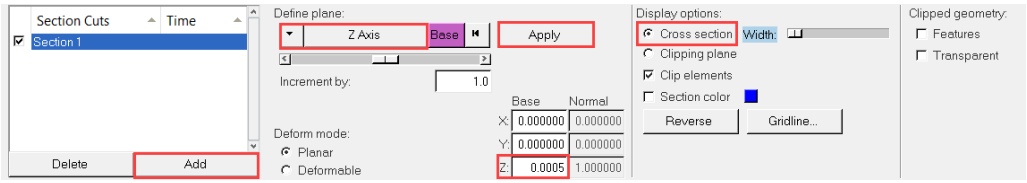


Figure 321:

7. Click **Gridline**. In the **Gridline Options** dialog, deactivate the **Show** check box under Grid line then click **OK**.

8. Click the **Vector** icon on the Results toolbar to open the Vector panel.
9. In the panel area, set the Result type to **Velocity (v)**.
10. Click the **Selection** drop-down and select **Sections** from the list of options.

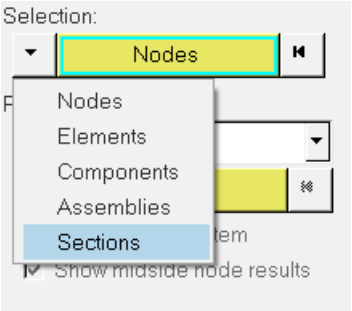


Figure 322:

11. Click the **Sections** entity selector then select **All**.
12. In the panel area, activate the **Overlay result display** check box (if not set already).
13. Click **Apply**.
14. Under the **Plot** tab, verify that only the **X+Y+Z Resultant** option is selected.
15. Go to the **Display** tab, set the Size scaling option to **Uniform**, and enter a value of `0.0015` in the size field.
16. Set the Color by option to **Direction** and set the X+Y+Z color to **White**.

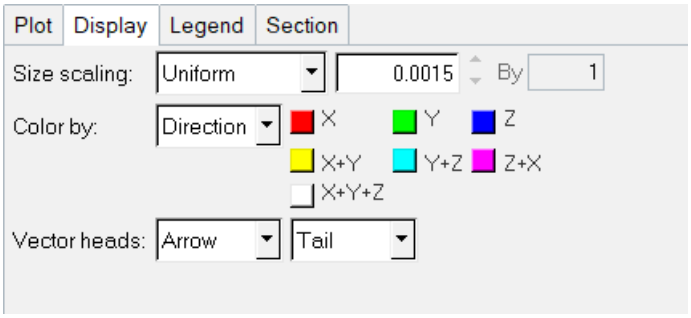


Figure 323:

17. Go to the **Section** tab, activate the **Projected** check box, then click **Apply**.
18. Click on the Results toolbar to open the Contour panel.
19. In the panel area, set the Result type to **Temperature (s)**.
20. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
21. In the panel area, under the **Result** tab, activate the **Overlay result display** check box (if not set already).

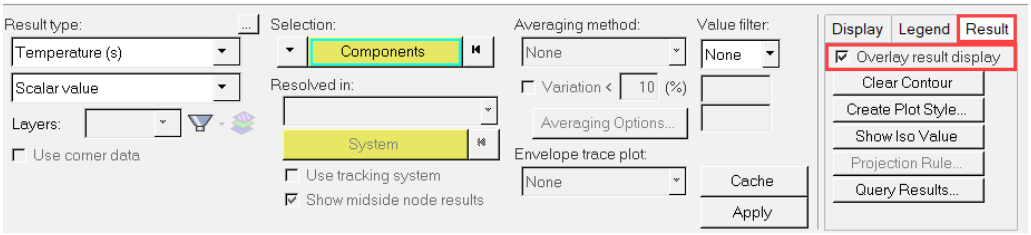


Figure 324:

22. Click **Apply** to create the contour plot of temperature on the section cut along with velocity vectors.

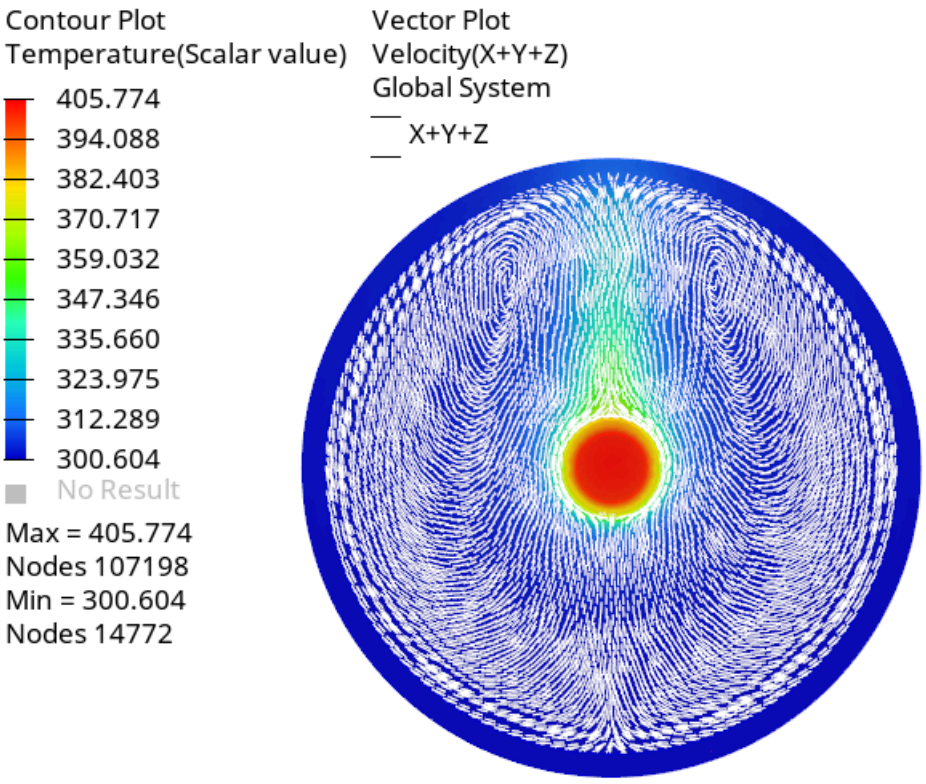


Figure 325:

Zoom in to the contour plot to observe the natural convection phenomenon inside the headlamp.

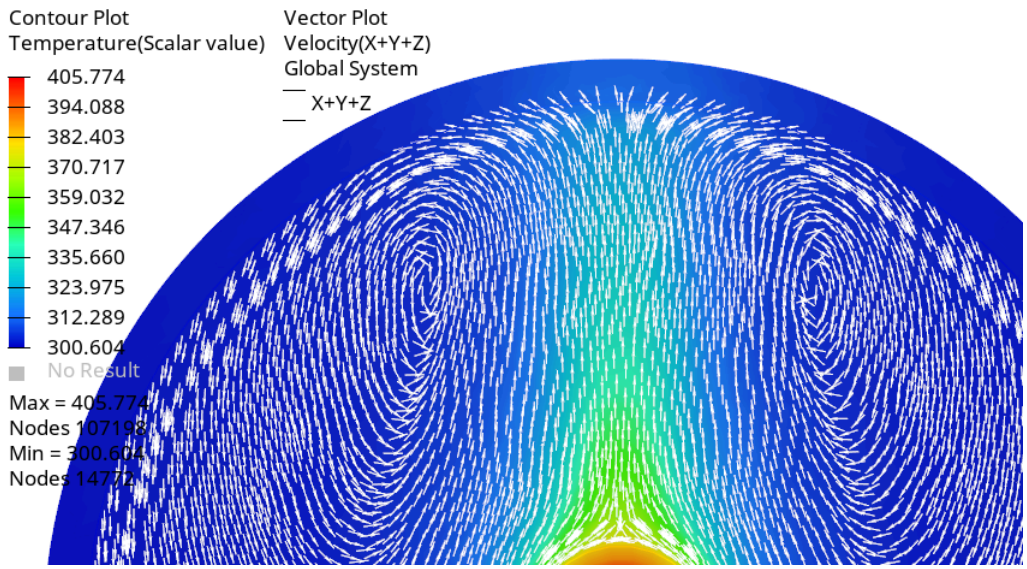


Figure 326:

Summary

In this tutorial, you learned how to set up and solve a radiation heat transfer problem in a headlamp using the enclosure radiation model in AcuSolve. You started by importing the HyperMesh database with the mesh and basic model organization and then set up the simulation parameters and boundary conditions. Once the solution was computed, you processed the results using HyperView, where you created contour plots of temperature and velocity vectors in the fluid domain.

2.3.6 ACU-T: 3201 Solar Radiation and Thermal Shell Tutorial

Prerequisites

This tutorial introduces you to setting up a CFD simulation involving solar radiation and thermal shells using AcuSolve and HyperMesh. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3201_SolarRadiation.hm` and `SolarLoad.dat` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed is shown schematically in Figure 327. The model consists of an atrium with a couch and chairs in the center. Air flows into the atrium through the inlet vent and exits through the outlet. The front portion of the atrium consists of glass walls supported by an aluminum frame. This aluminum frame will be modeled as a thermal shell; hence, this tutorial introduces you to the process of setting up a transient solar radiation simulation and thermal shells in HyperMesh.

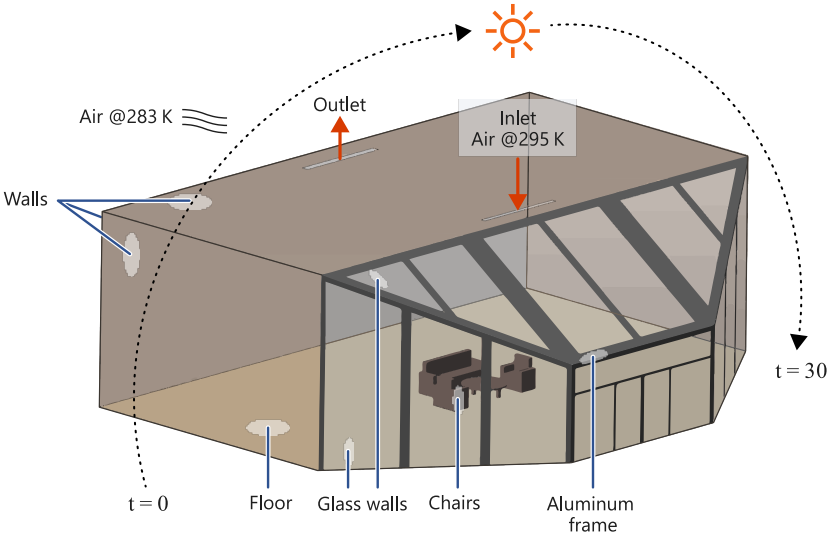


Figure 327:

Solar Radiation Parameters

AcuSolve uses an ideal gray surface solar radiation model to calculate the solar heat flux. The fluxes are computed using a ray trace algorithm and five optical properties of the surface, specular transmissivity (τ_s), diffuse transmissivity (τ_d), specular reflectivity (ρ_s), diffuse reflectivity (ρ_d) and absorptivity (α).

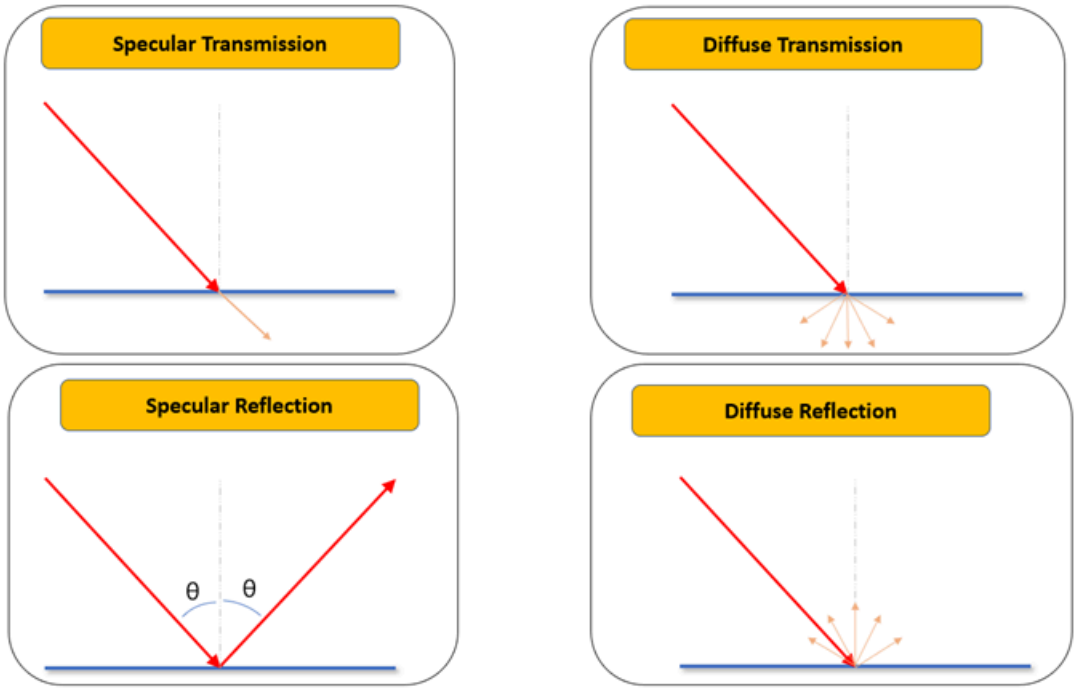


Figure 328:

A specular transmission occurs when a photon passes straight through a surface with no change of direction. In a diffuse transmission the photon penetrates the surface, but its outgoing energy is uniformly distributed in solid angle over the hemisphere, weighted by projected surface area. For a specular reflection, the angle of reflection is equal to the angle of incidence. Diffuse reflections are similar to diffuse transmissions, except the hemisphere over which the outgoing energy is distributed is on the same side of the surface as the incident photon. Finally, the photon may be absorbed by the surface. These five interactions are associated with five surface properties that together must obey the following constraint:

$$\tau_s(\theta)+\tau_d(\theta)+\rho_s(\theta)+\rho_d(\theta)+\alpha(\theta)=1 \quad (1)$$

Where,

τ_s	Specular transmissivity
τ_d	Diffuse transmissivity
ρ_s	Specular reflectivity
ρ_d	Diffuse reflectivity
α	Absorptivity
θ	Angle of incidence

For the solar radiative heat fluxes to be computed, a solar radiation surface needs to be defined on that given surface.

In this tutorial, the solar flux loading is given in the form of a data file which was generated using the `acuSflux` script available in AcuSolve. The script can be used to generate a data file with a four-column array of solar flux vector data values. The piecewise linear type is used in this tutorial to emulate the pattern of sunrise to sunset over the atrium.

For example, to generate the solar load data file for a location with known geological coordinates, enter the following command in the AcuSolve Command Prompt: `acuSflux -time "dec-3-2019 11:00:00" -tinc 1800 -nts 25 -lat 42.6064 -lon -83.1498 -ndir "1,0,0" -udir "0,0,1"`

Here,

time	The start time in GMT (ex: "dec-3-2019 21:00:00")
tinc	The time increment in seconds
nts	Number of discrete time steps
lat	Latitude coordinates of the location in degrees North (ex: 45.112 or -37.56 (equal to 37.56 S))
lon	Longitude coordinates of the location in degrees East (ex: 86.26 or -54.84 (equal to 54.84 W))
ndir	The north direction unit vector in model coordinates (should be enclosed in double quotes) (ex: "0,1,0")
udir	The upward direction unit vector in model coordinates (should be enclosed in double quotes) (ex: "0,1,0")

Thermal Shell Modeling

The thermal shell in AcuSolve is a feature that creates zero physical thickness volumetric shell elements from surface elements. This is useful when the thickness of the component is too small to be modeled as a solid medium. The thermal shell can have multiple layers, each with different thicknesses and material models. A schematic of the thermal shell is shown below.

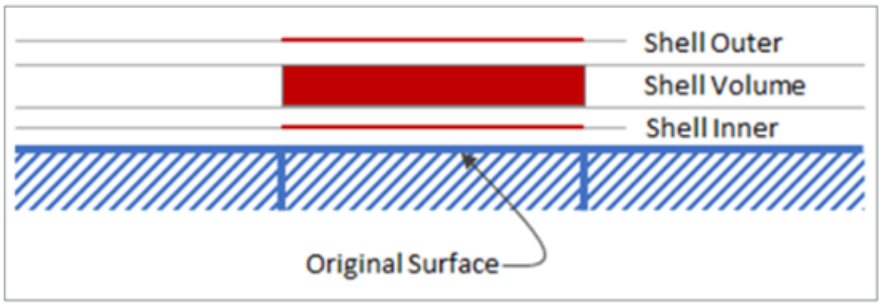


Figure 329:

When defining a thermal shell on a surface, two sets of boundary conditions are needed. One for the Primary Wall surface i.e. Shell Inner and one for the Shell Outer Wall surface. In this tutorial, a solar radiation surface will be defined on the outer shell surface so that it receives solar heat flux, whereas the inner shell surface will be modeled as a default wall.

Open the HyperMesh Model Database

1. Start HyperMesh Desktop and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon  located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3201_SolarRadiation.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Atrium_Solar` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Atrium_Solar` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set Up the Simulation Parameters

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, change the Analysis type to **Transient**.
3. Set the Temperature equation to **Advective Diffusive**.
4. Set the Turbulence model to **Laminar** (if not set already).

Name	Value
Title	Solar Radiation Tutorial
Analysis type	Transient
Flow equation	Navier Stokes
Abs. pressure offset ## (N/m2)	0.0
Temperature equation	Advective Diffusive
Abs. temperature offset ## (K)	0.0
Radiation equation	None
Turbulence model	Laminar
Humid Air Model	Off
Multiphase equation	None
Optimization	Off
Mesh type	Fixed
Nucleate boiling single phase	Off
External code	Off
Global Gravity	
X ## (m/sec2)	0.0
Y ## (m/sec2)	0.0
Z ## (m/sec2)	0.0
Unsupported Entries	

Figure 330:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, set the Max time steps to `0`.
3. Set the Final time to `30` sec.
4. Set the Initial time increment to `1` sec.
5. Set the Max stagger iterations to `3`.
6. Change the Relaxation factor to `0`.
7. Verify that the Flow and Temperature fields are turned **On**.

Name	Value
Max time steps	0
Final time ## (sec)	30.0
Initial time increment ## (sec)	1.0
Auto time increment	☐
Multiplier Function	<Unspecified>
Convergence tolerance	0.0
Min stagger iterations	0
Max stagger iterations	3
Relaxation factor	0.0
Flow	On
Temperature	On
Temperature flow	Off
Number of Kyrlov Vector	10
⊞ **Advanced Settings**	
⊞ **Unsupported Entries**	

Figure 331:

Set the Solar Radiation Parameters

1. In the Solver Browser, click **05.SOLAR_RADIATION_PARAMETERS** under **01.Global**.
2. In the Entity Editor, change the Curve fit options to **Dat file.**
3. Click on the open file icon beside the Dat file field, browse to the location where you saved `SolarLoad.dat`, and open it.
4. Leave the remaining options as default.

Name	Value
Solar flux type	Piecewise_Linear
Curve fit options	Dat file
Dat file	e Tutorials/ACU-3201/Atrium_Solar/SolarLoad.dat
Curve fit variable	Time
Function type	None
Computation type	Ray_Trace
Num rays	1000000
Num diffuse sub rays	4
Max ray reflections	10
Min ray energy	0.0
Redistribution factor	1.0
Smoothing	On
⊞ **Unsupported Entries**	

Figure 332:

5. Save the model.

Assign Material Properties and Boundary Conditions

Create the Solar Radiation Model

1. In the Solver Browser, right click on **08.Solar_Radiation_Model** and select **Create**.
2. In the Entity Editor, name it `Glass`.
3. Set the Specular Transmissivity to `0.8`.
4. Set the Diffusive Transmissivity to `0.08`.
5. Set the Specular reflectivity to `0.01`.
6. Set the Diffusive reflectivity to `0.01`.

Name	Value
Solver Keyword:	SOLAR_RADIATION_MODEL
Name:	Glass
Depth:	-1
Type:	Solar_Radiation_Model
Specular transmissivity type:	Constant
Specular Transmissivity:	0.8
Diffusive transmissivity type:	Constant
Diffusive Transmissivity:	0.08
Specular reflective type:	Constant
Specular reflectivity:	0.01
Diffusive reflective type:	Constant
Diffusive reflectivity:	0.01
Unsupported Entries	
Number of Unsupported Entries:	0

Figure 333:

Assign Material Properties and Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Set the Material to **Air_HM**.

Name	Value
Name	Fluid
Type	FLUID
Material	(1) Air_HM
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 334:

3. Click **Walls**. In the Entity Editor,

a) Verify that the Type is set to **WALL** and the Temperature BC type is set to **Flux**.

b) Set the Convective heat flux coefficient to `20.0` J/m^2-sec-K.

c) Set the Convective heat flux reference temperature to `283.15` K.

Name	Value
Name	Walls
Type	WALL
FE style	
Geometry style	
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Temperature BC type	Flux
Heat flux ## (W/m2)	0.0
Convective heat flux coefficient ## (J/m2-sec-K)	20.0
Convective heat flux reference temperature ## (K)	283.15

Figure 335:

d) Under the Solar Radiation Surface tab, turn on the **Display** field. Turn **On** the Activate solar radiation surface option. Set the Type to **Outward**. Set the Solar radiation model to **Solar Radiation Black Body**.

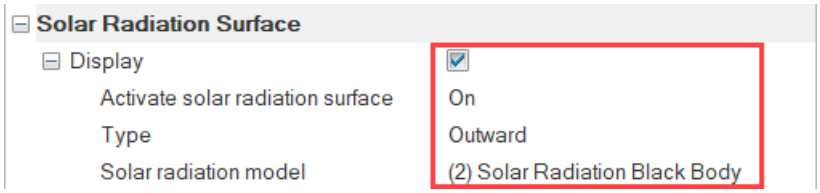


Figure 336:

e) Under the Surface Output tab, turn **On** Surface Output and set the Nodal time step frequency to `1`.

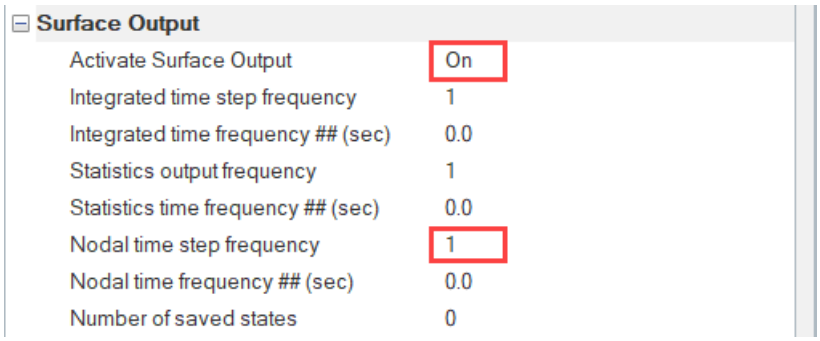


Figure 337:

4. Click **Inlet**. In the Entity Editor,

a) Change the Type to **INFLOW**.

b) Set the Inflow velocity type to **Cartesian**.

c) Set the X, Y and Z-velocity to `0`, `22.78` and `-39.45` m/sec respectively.

d) Set the Temperature of the incoming fluid to `295.35` K.

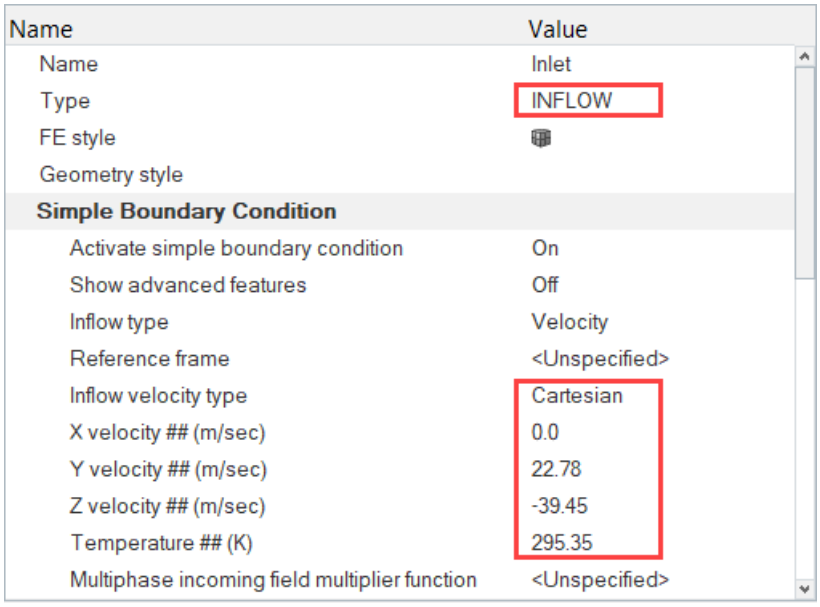


Figure 338:

e) Under the Solar Radiation Surface tab, turn on the **Display** field. Turn **On** the Activate solar radiation surface option. Set the Type to **Outward**. Set the Solar radiation model to **Solar Radiation Black Body**.

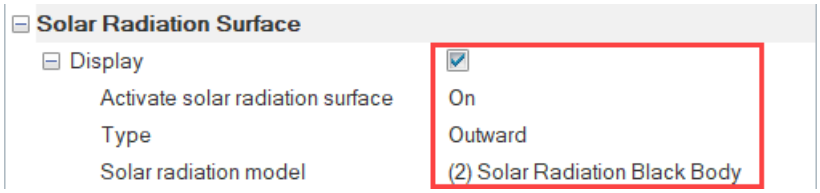


Figure 339:

5. Click **Outlet**. In the Entity Editor, change the Type to **OUTFLOW**.

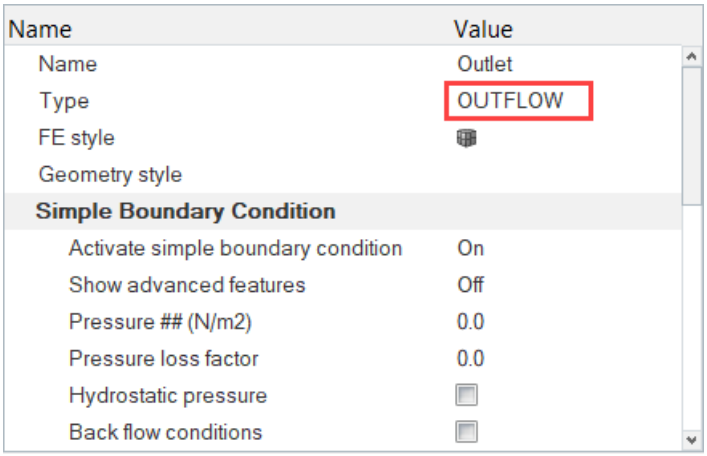


Figure 340:

6. Click **Glass**. In the Entity Editor,

a) Verify that the Type is set to **WALL** and the Temperature BC type is set to **Flux**.

b) Set the Convective heat flux coefficient to `20.0` J/m^2-sec-K.

c) Set the Convective heat flux reference temperature to `283.15` K.

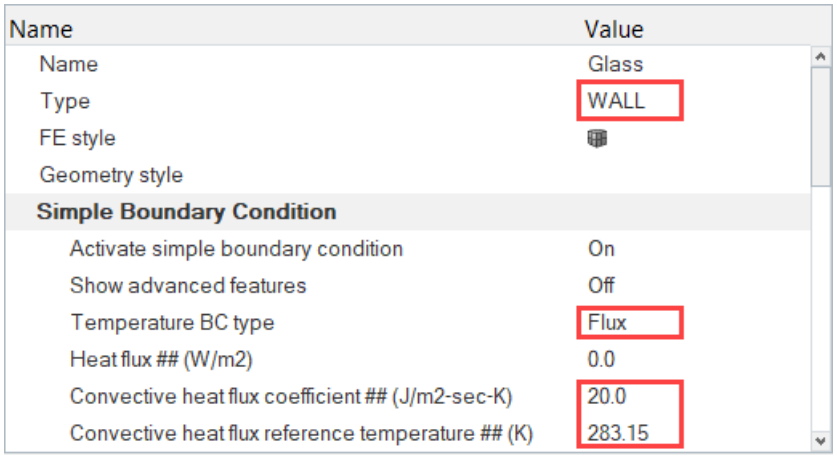


Figure 341:

d) Under the Solar Radiation Surface tab, turn on the **Display** field. Turn **On** the Activate solar radiation surface option. Set the Type to **Both**. Set the Solar radiation model to **Glass**.



Figure 342:

e) Under the Surface Output tab, turn **On** Surface Output and set the Nodal time step frequency to `1`.

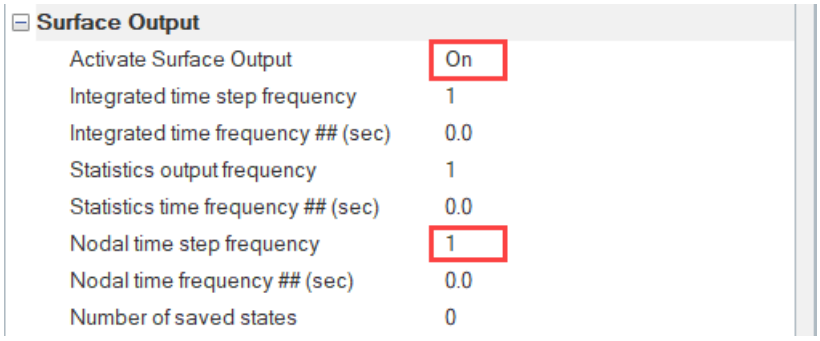


Figure 343:

7. Click **Floor**. In the Entity Editor,

a) Verify that the Type is set to **WALL** and the Temperature BC type is set to **Flux**.

b) Under the Solar Radiation Surface tab, turn on the **Display** field. Turn **On** the Activate solar radiation surface option. Set the Type to **Inward**. Set the Solar radiation model to **Solar Radiation Black Body**.

c) Under the Surface Output tab, turn **On** Surface Output and set the Nodal time step frequency to `1`.

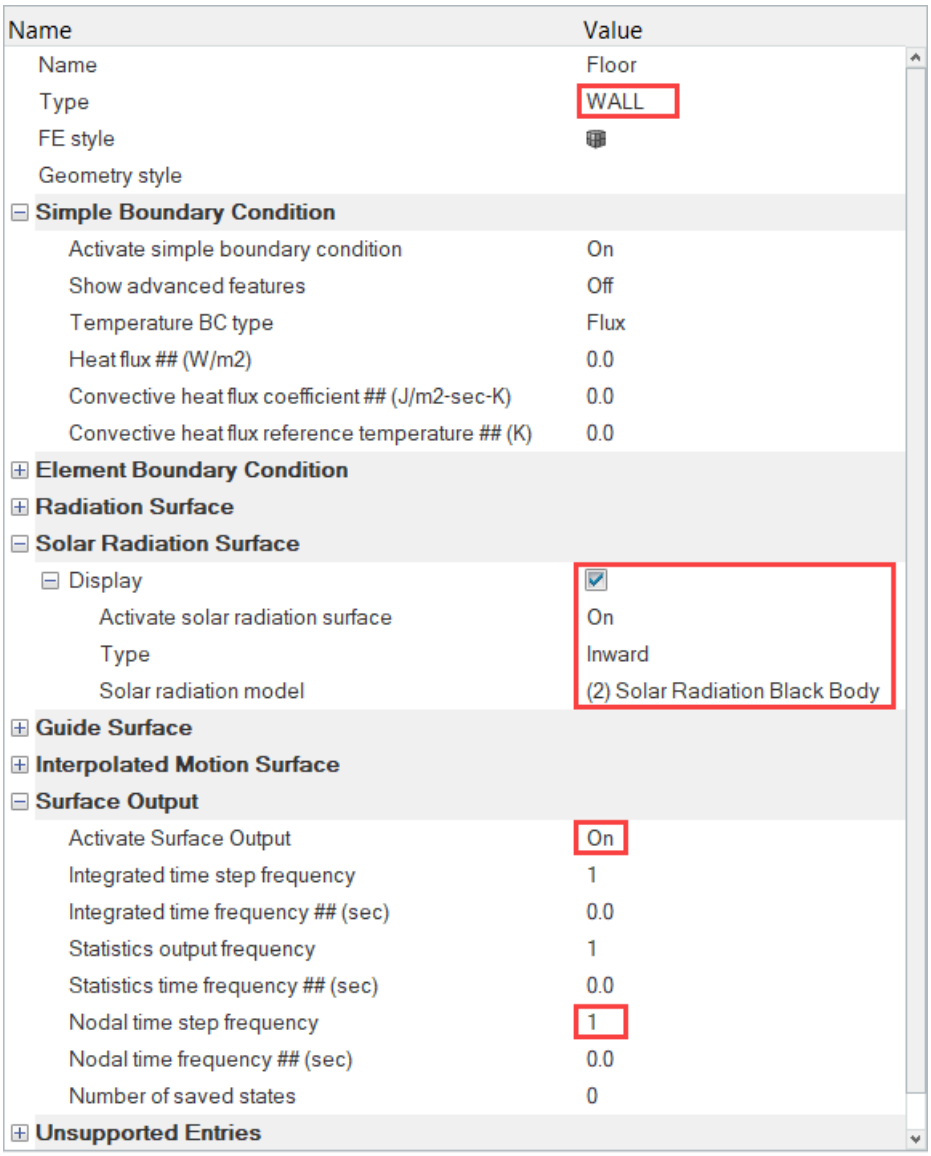


Figure 344:

8. Click **Glass_walls**. In the Entity Editor,

a) Change the Type to **THERMAL_SHELL**.

b) Set the Number of shell layers to `1`.

c) Set the Shell thickness to `0.025` m.

d) Set the Shell Material to **Aluminum_HM**.

e) For the Primary Wall boundary condition, leave the default values unchanged.

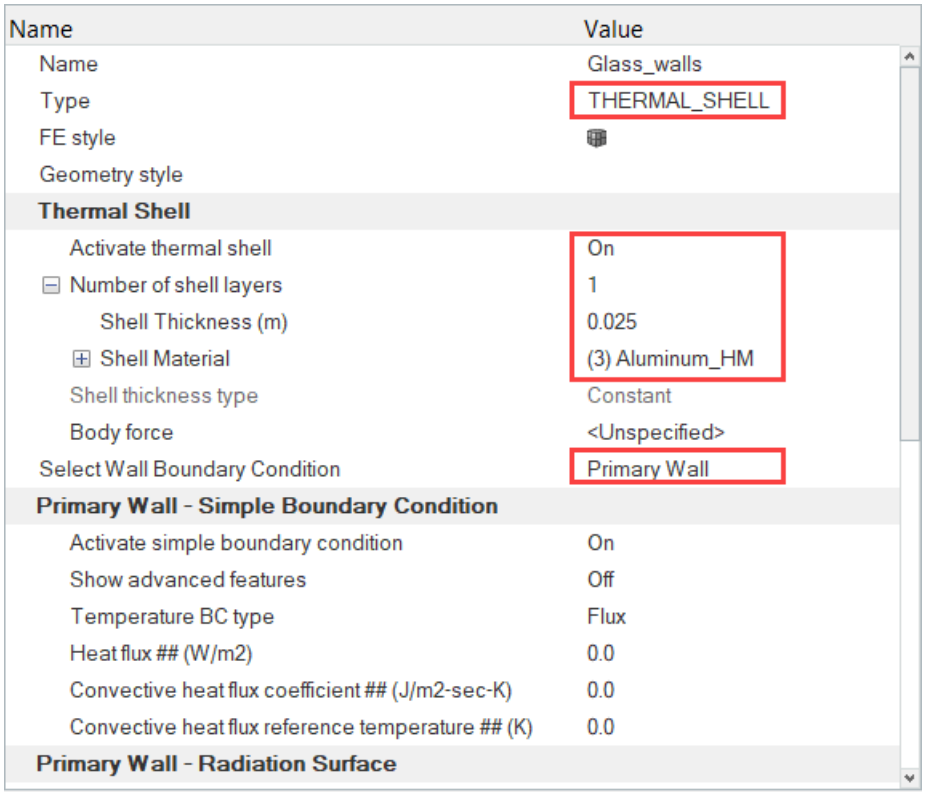


Figure 345:

f) Change the Select Wall Boundary Condition value from Primary Wall to **Shell Outer Wall**.

g) Set the Convective heat flux coefficient to `20` J/m^2-sec-K and the Convective heat flux reference temperature to `283.15` K.

h) Under the Solar Radiation Surface tab, turn on the **Display** field. Turn **On** the Activate solar radiation surface option. Set the Type to **Outward**. Set the Solar radiation model to **Solar Radiation Black Body**.

i) Expand the Surface Output tab, turn **On** Surface Output, and set the Nodal time step frequency to `1`.

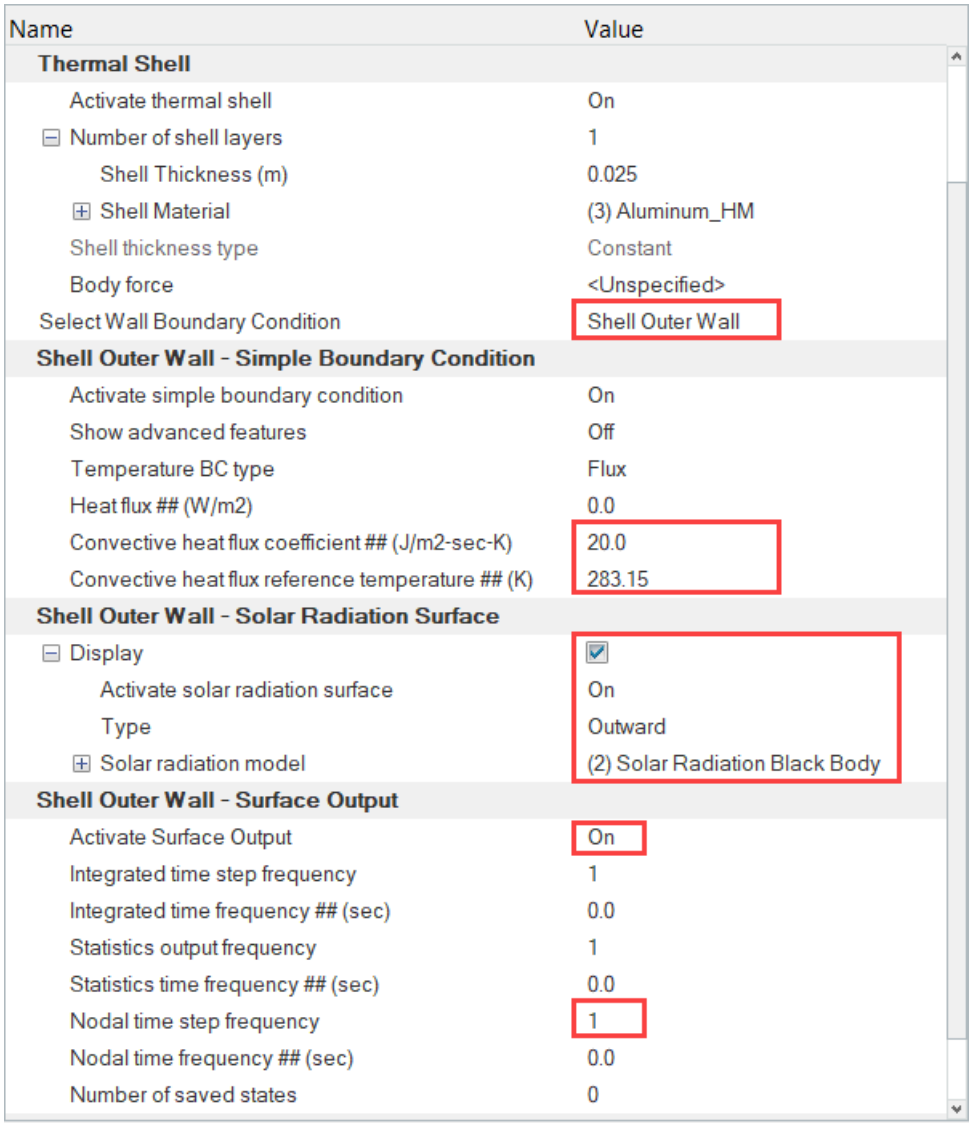


Figure 346:

9. Click **Chair**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Under the Solar Radiation Surface tab, turn on the **Display** field. Turn **On** the Activate solar radiation surface option. Set the Type to **Default**. Set the Solar radiation model to **Solar Radiation Black Body**.

c) Under the Surface Output tab, turn **On** Surface Output and set the Nodal time step frequency to `1`.

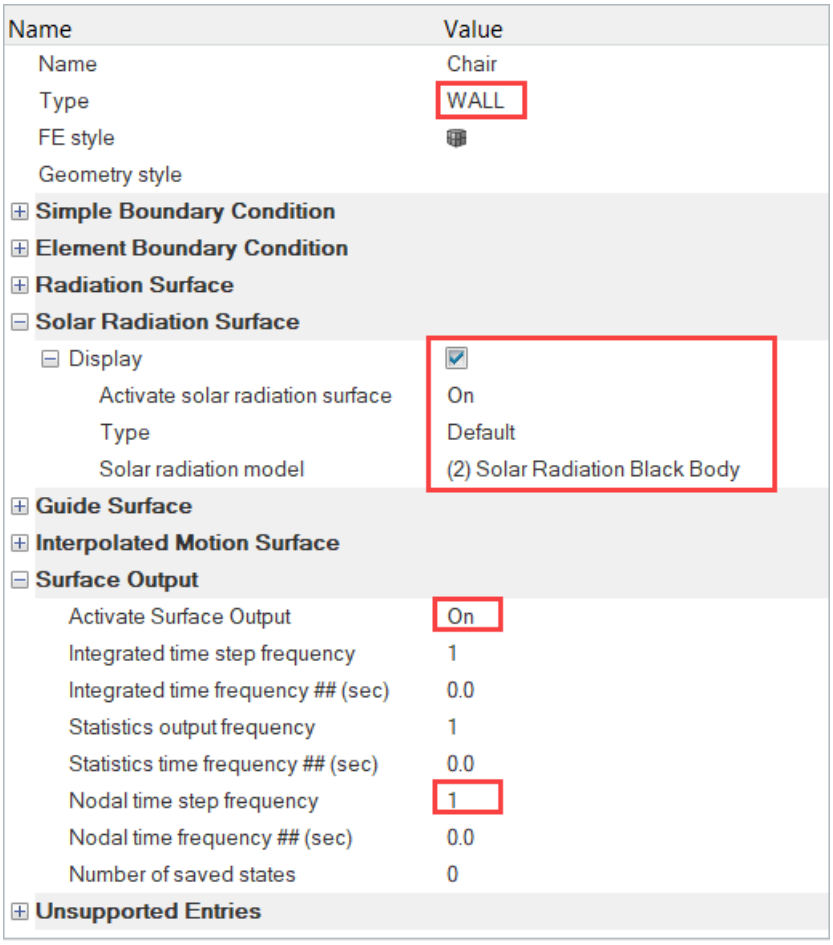


Figure 347:

10. Save the model.

Set Nodal Initial Conditions

1. In the Solver Browser, click **03.NODAL_INITIAL_CONDITION** under **01.Global**.
2. In the Entity Editor, change the Default value of Temperature to `288.15` K.

Define Nodal Output Frequency

1. In the Solver Browser, expand **17.Output** then click **NODAL_OUTPUT**.
2. In the Entity Editor, set the Time step frequency to `1`.
3. Activate the **Output initial condition** checkbox.
4. Save the model.

Compute the Solution

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Leave the remaining options as default and click **Launch** to start the solution process.

Post-Process the Results using HyperView

In this step, you will create an animation of solar heat flux and temperature over run time. Once the solver run is complete, close the AcuProbe and AcuTail windows. In the HyperMesh Desktop window, close the AcuSolve Control tab and save the model.

Switch to the HyperView Interface and Load the AcuSolve Model and Results

1. In the HyperMesh Desktop window, click the **ClientSelector** drop-down in the bottom-left corner of the graphics window.

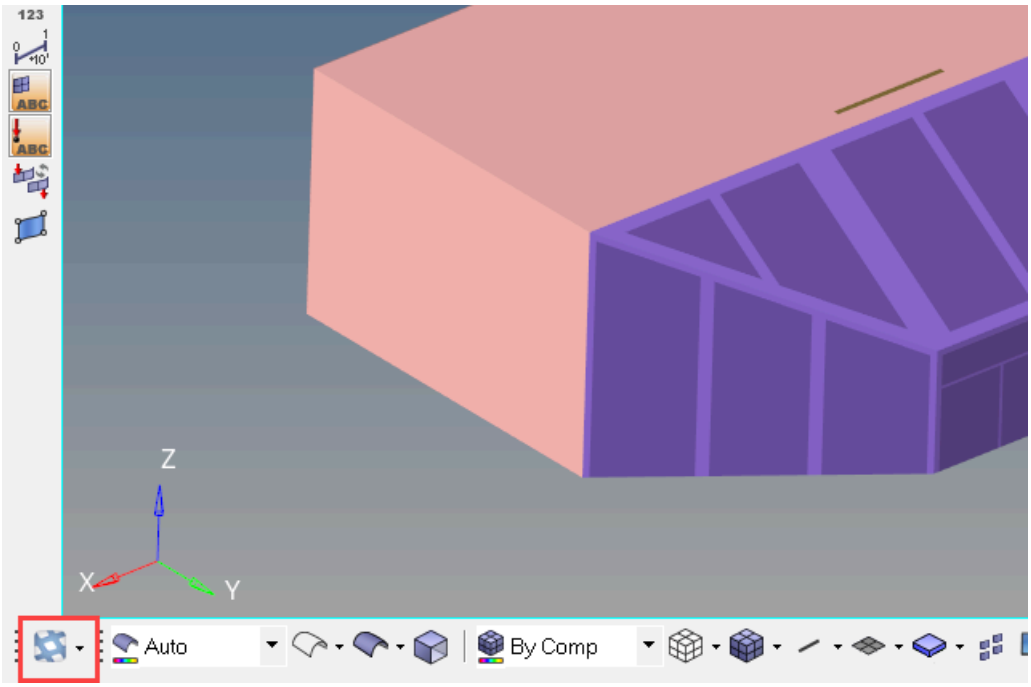


Figure 348:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.

The interface is changed to HyperView.

Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.

4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Atrium_Solar.1.Log`.
6. Click **Open**.
7. In the panel area, click **Reader Options**.
8. In the **Reader Options** dialog, set the Reader to **AcuSolve Result Reader** and the Extended nodal output option to **Yes** then click **OK**.

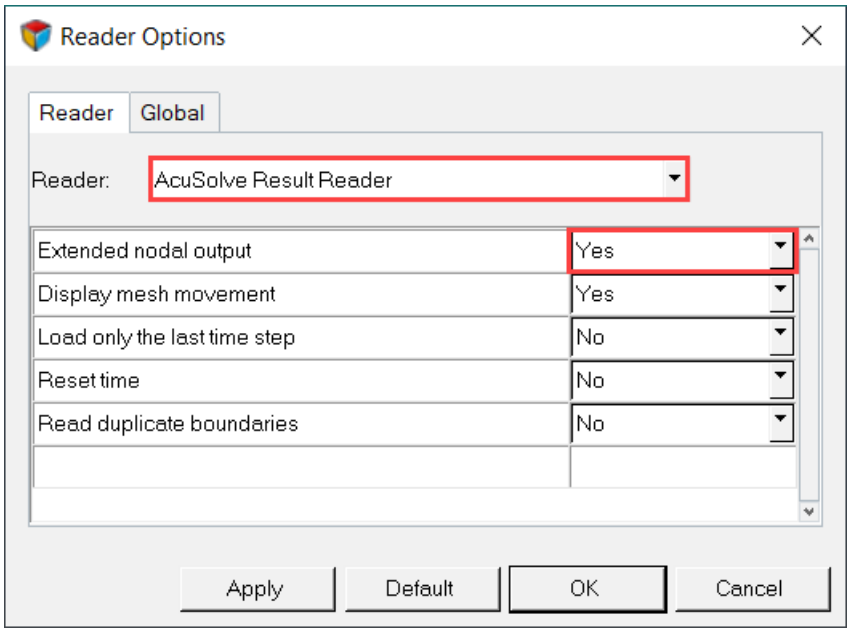


Figure 349:

9. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create an Animation of Temperature Contour

In this step, you will start by creating an expression for plotting the temperature values in Fahrenheit units. Then, you will create an animation of the magnitude of temperature on the floor, glass_walls and the man surface.

1. From the menu bar, go to **Results** > **Create** > **Derived Results**.
2. In the **Expression Builder** dialog, enter `Temperature_fahrenheit` as the Label (name) of the expression.
3. In the Expression text box, enter the following expression: `1.8*(`

4. Set the Table option to **Temperature** and the Resource to **model**. Then, click **Insert** to add the temperature variable in the expression.
5. Complete the expression by entering the remaining portion of the formula as shown in the figure below.

   Here the term 'R1.S16' corresponds to the Temperature (scalar) variable in Kelvin. Variables can be inserted in the expression by selecting the required variable under Table option and then clicking Insert. The actual ID for the scalar variable might be different for your simulation.

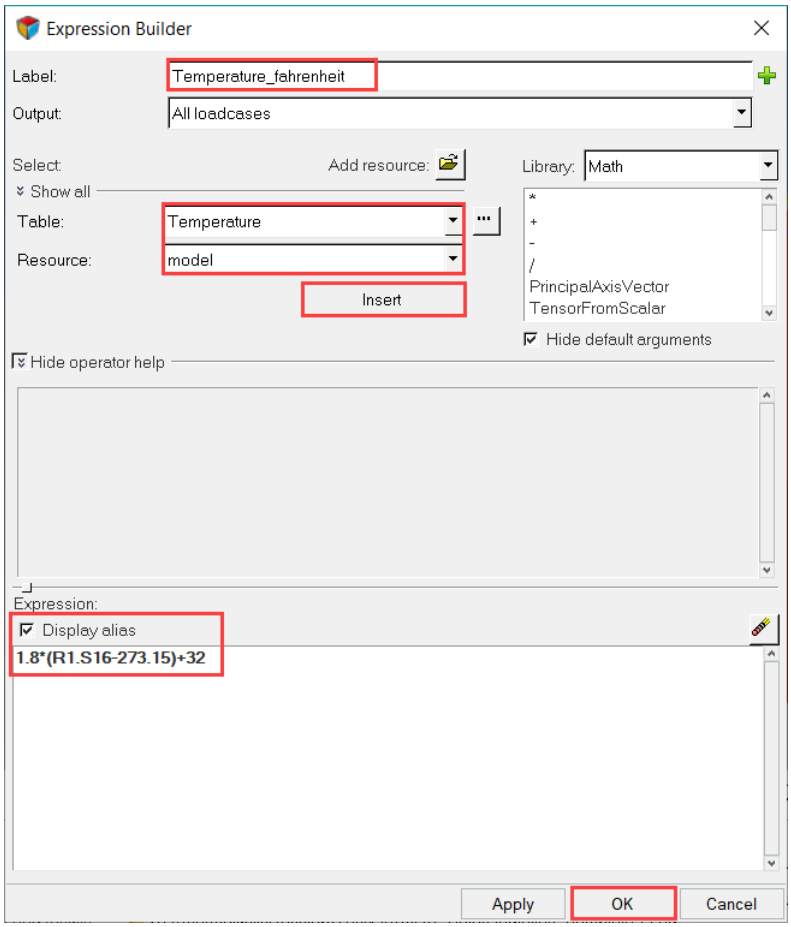


Figure 350:

6. Click **OK** to apply the changes and close the dialog.
7. In the Results Browser, expand the list of **Components**. Turn off the display of all the components except **Floor**, **Glass_walls Shell Outer**, and **Chair** surfaces.

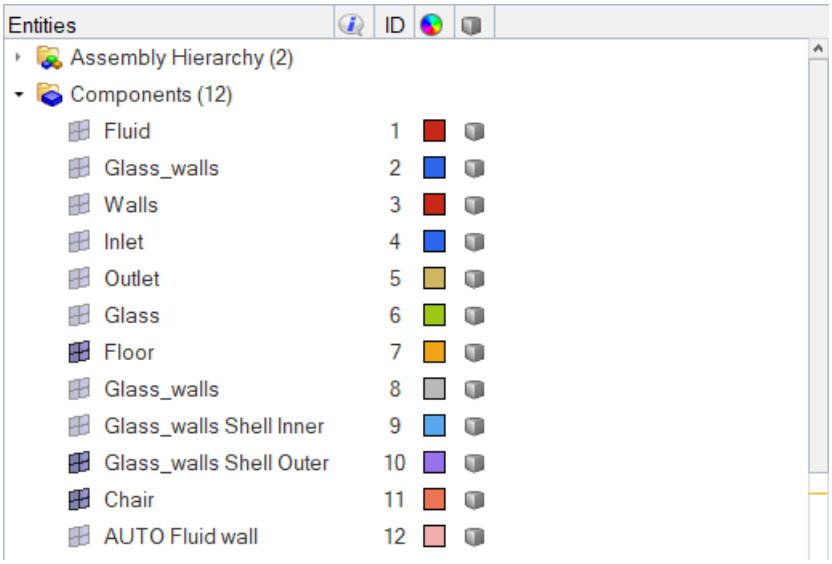


Figure 351:

8. In the graphics window, rotate the model so that you have a better view of the glass walls and the chair.
9. Click on the Results toolbar to open the Contour panel.
10. In the panel area, set the Result type to **Temperature_fahrenheit (s)**.
11. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
12. Click **Apply**.
13. In the panel area, under the Display tab, turn off the Discrete color option.

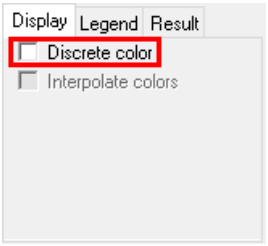


Figure 352:

14. Go to the Legend tab then click **Edit Legend**.
15. In the **Edit Legend** dialog, change the Type to **Dynamic scale** and the Numeric format to **Fixed** then click **OK**.
16. Click on the Animation toolbar to play the temperature animation.
17. Click the **Animation Controls** icon . In the panel area, set the Max Frame Rate to 5 Frames/Sec by dragging the slider.

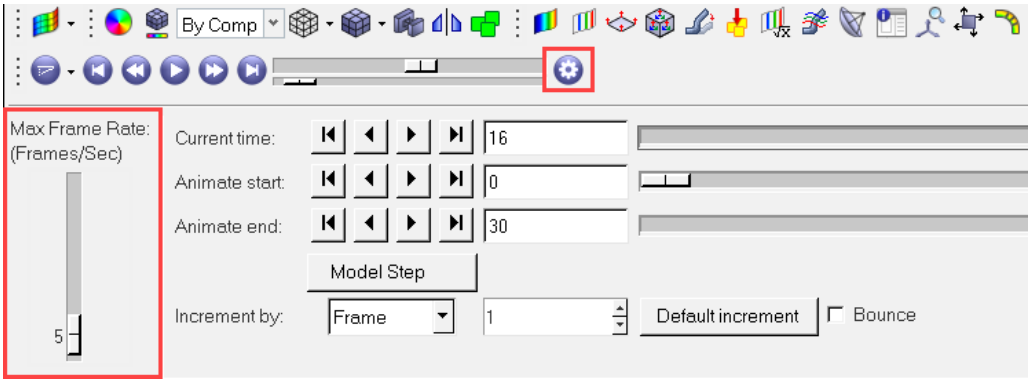


Figure 353:

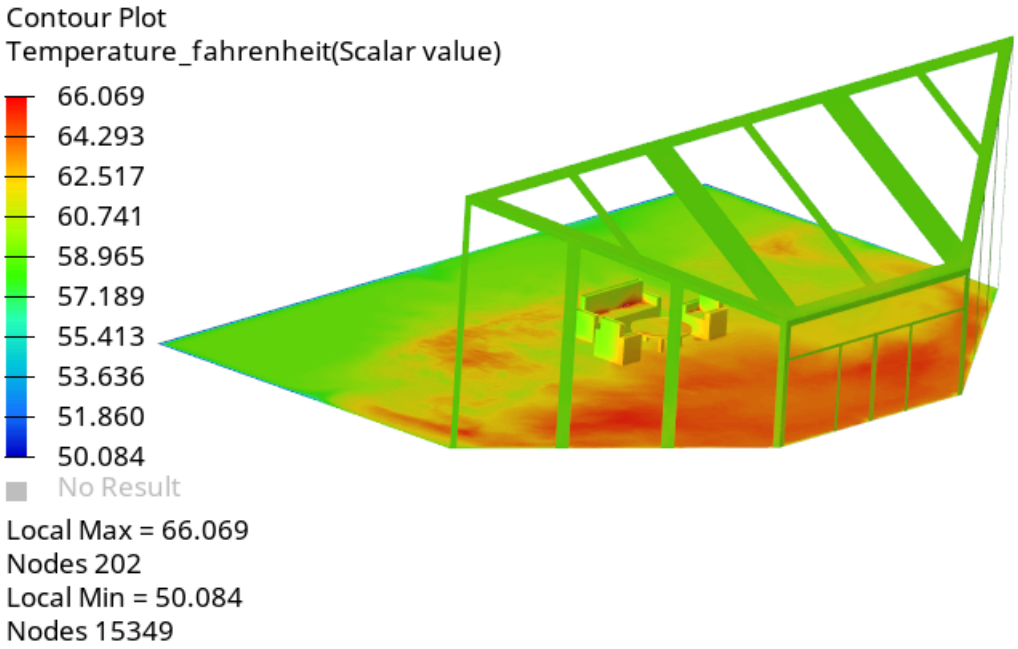


Figure 354:

Create an Animation of Solar Heat Flux

1. Click on the Results toolbar to open the Contour panel.
2. In the panel area, set the Result type to **Solar_heat_flux (s)**.
3. Click **Apply**.

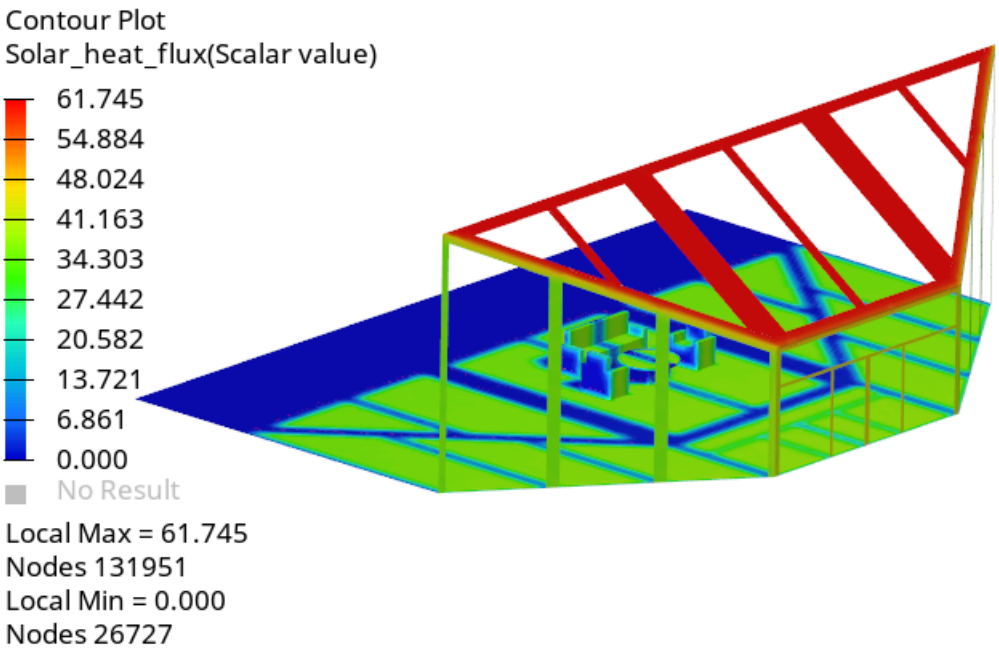


Figure 355:

4. On the ImageCapture toolbar, click on the **Capture Graphics Area Video** icon .
5. In the **Save Graphics Area Video As** dialog, browse to the directory where you want to save the animation and give a name to the video, ex: `solar heat flux animation`, then click **Save**.

Summary

In this tutorial, you learned how to set up and solve a CFD analysis involving solar radiation and thermal shells. You started by importing a HyperMesh model database and set up the simulation parameters and boundary conditions. Once you computed the solution, you post-processed the results using HyperView. Also, you learned how to create expressions in HyperView and build plots using derived results.

2.3.7 ACU-T: 3202 Heat Transfer Between Concentric Spheres – P1 Radiation Model

Prerequisites

This tutorial provides instructions for setting, solving and viewing results for a steady state simulation of radiation heat transfer between concentric spheres using the P-1 Radiation model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic knowledge of HyperMesh, AcuSolve, and HyperView.To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3202_P1Rad.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 356. In this problem, a P1 radiation model is used to simulate the heat transfer due to radiation between concentric spheres. The inside surface of the inner and the outside surface of the outer sphere are both held at constant temperature while the gap between them radiates the heat from one sphere to the other.

The problem consists of a fluid region with arbitrary material properties between two concentric spheres with surfaces held at fixed temperature, as shown in the following figure, which is not drawn to scale. The radius of the outer sphere is 0.04 m and the radius of the inner sphere is 0.01 m. The inner surface of the inner sphere is defined to have a constant wall temperature at 300.0 K (26.85 ºC). The outer surface of the outer sphere is defined to have a constant wall temperature at 1300.0 K (1026.85 ºC). The fluid within the spheres is defined as a non-conducting material, allowing heat to transfer via radiation only.

The problem is solved as a steady state case to allow the heat transfer in the solid and fluid regions to reach an equilibrium.

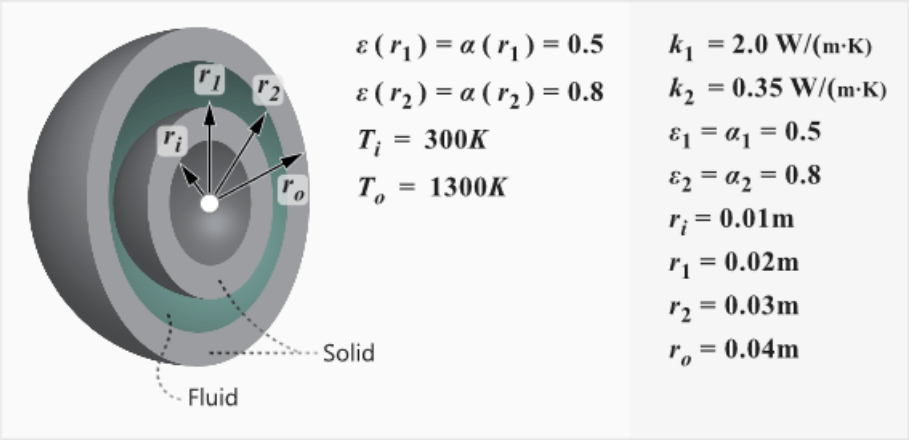


Figure 356:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.

2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3202_P1Rad.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `P1-Radiation_Sphere` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `P1-Radiation_Sphere` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, set the Turbulence equation to **Advective Diffusive**.
3. Set the Radiation equation to **P-1**.

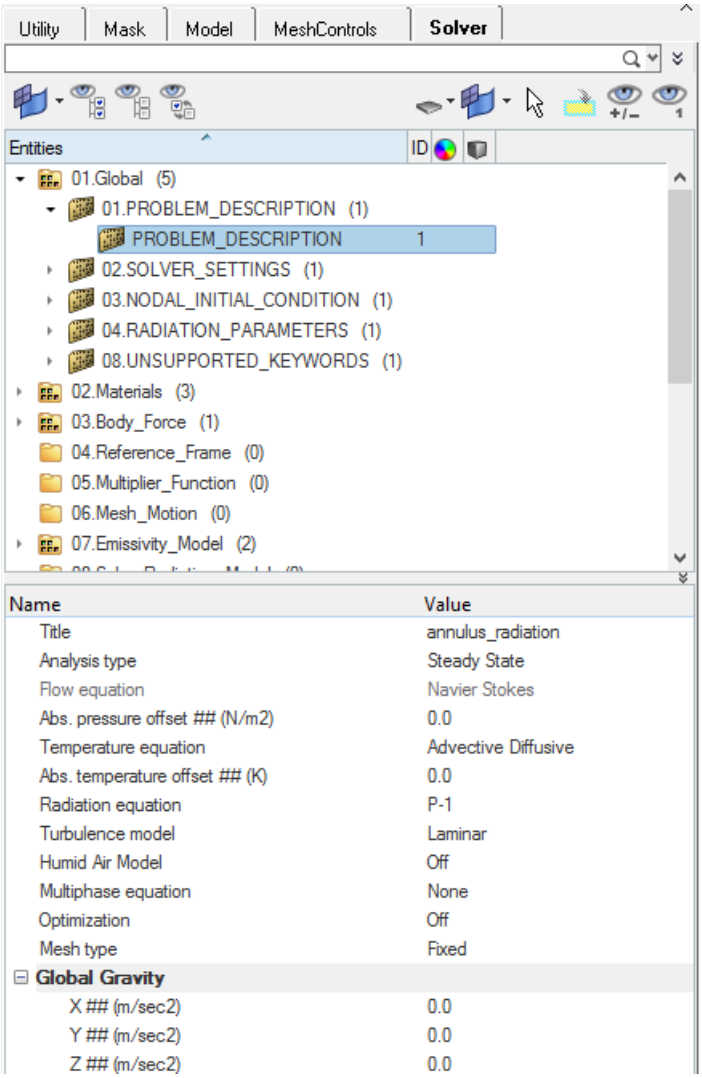


Figure 357:

4. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
5. In the Entity Editor, set the Convergence tolerance to `1e-05`.
6. Verify that the Relaxation Factor is set to `0.3`.
7. Turn off Flow and verify that the Temperature and Radiation fields are turned **On**.

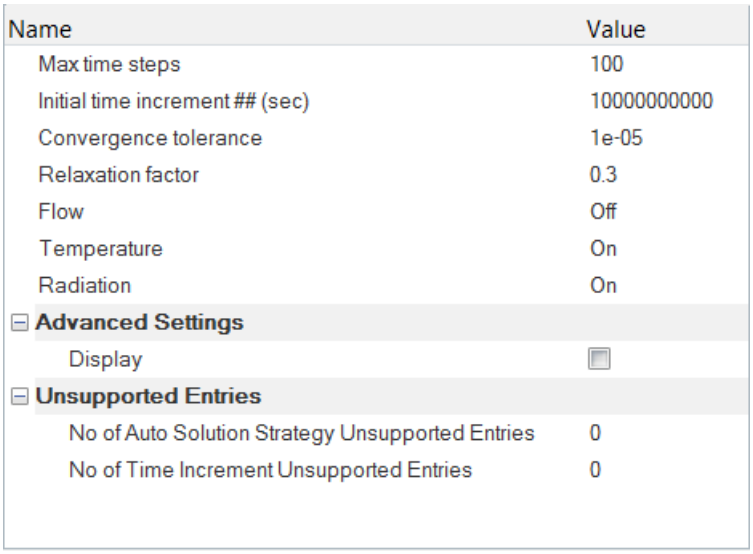


Figure 358:

Set Up Radiation Parameters and Boundary Conditions

In this step, you will define the radiation parameters i.e. emissivity models, surface boundary conditions for the problem, and assign material properties to the fluid and solid regions.

Set Up Material Model Parameters

1. In the Solver Browser, right-click on **02.Materials** and select **Material(Fluid)**.
2. Name the material `Radiating`.
3. In the Entity Editor, set the Density to `1000` kg/m^3.
4. Set the Specific heat to `10000` J/kg-K.
5. Set the Conductivity to `1e-6` W/m-K.
   This is done to prioritize heat transfer only through radiation.
6. Under Radiation Properties, turn on **Allow Participating Media Radiation**.
7. Set the Absorption coefficient to `0.0001`.

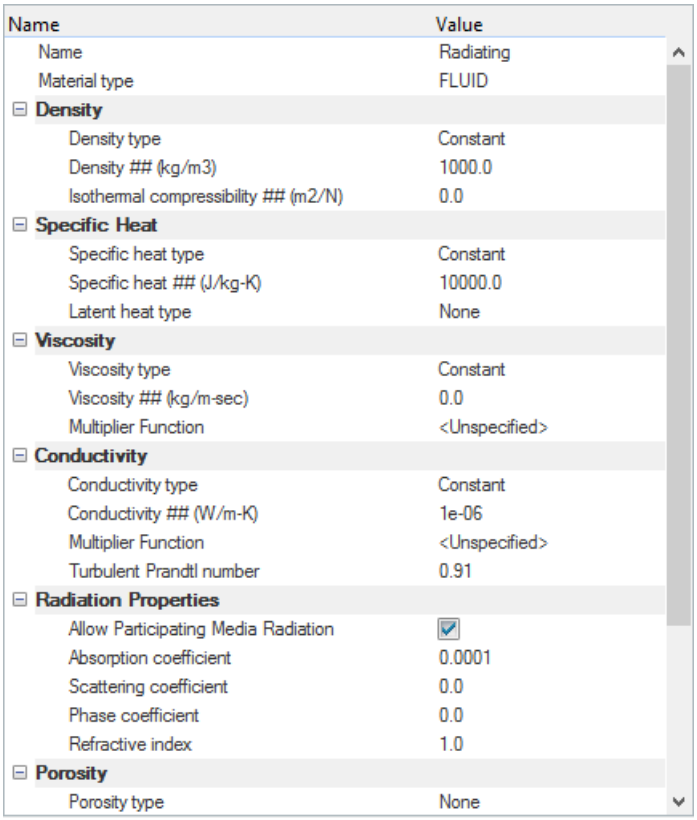


Figure 359:

8. In the Solver Browser, right-click on **02.Materials** and select **Material(Solid)**.
9. Name the material `Inner`.
10. In the Entity Editor, set the Density to `1000` kg/m^3.
11. Set the Specific heat to `10000` J/kg-K.
12. Set the Conductivity to `2` W/m-K.
13. In the Solver Browser, right-click on **Inner** and select **Duplicate**. Rename it `Outer`.
14. In the Entity Editor, change the Conductivity to `0.35` W/m-K.
15. Verify that the **Allow Participating Media Radiation** is turned off for both Inner and Outer. This is done because only the fluid medium participates in heat transfer through radiation.

Set Up Emissivity Model Parameters

1. In the Solver Browser, right-click on **07.Emissivity_Model** collector and select **Create**.
2. Name the emissivity model `Inner`.
3. In the Entity Editor, set the Emissivity to `0.5`.

Name	Value
Solver Keyword	EMISSIVITY_MODEL
Name	Inner
Type	Emissivity_Model
Emissivity Model Type	Constant
Emissivity	0.5
Multiplier Function	<Unspecified>
Unsupported Entries	
Number of Unsupported Entries	0

Figure 360:

4. Repeat the above steps and create another emissivity model named `Outer` with an Emissivity of `0.8`.

Set Up Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Radiating**. In the Entity Editor, change the Type to **FLUID** and set **Radiating** as the Material.

Name	Value
Name	Radiating
FE style	
Geometry style	
Type	FLUID
Material	(1) Radiating
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 361:

3. Click **Inner**. In the Entity Editor, change the Type to **SOLID** and set **Inner** as the Material.

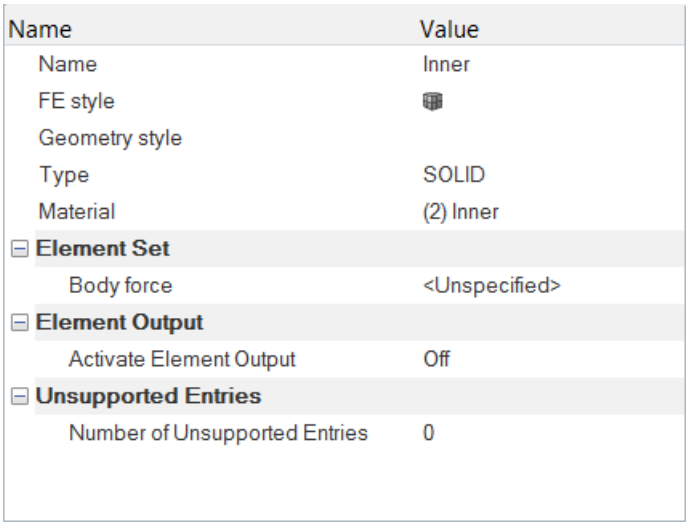


Figure 362:

4. Click **Outer**. In the Entity Editor, change the Type to **SOLID** and set **Outer** as the Material.

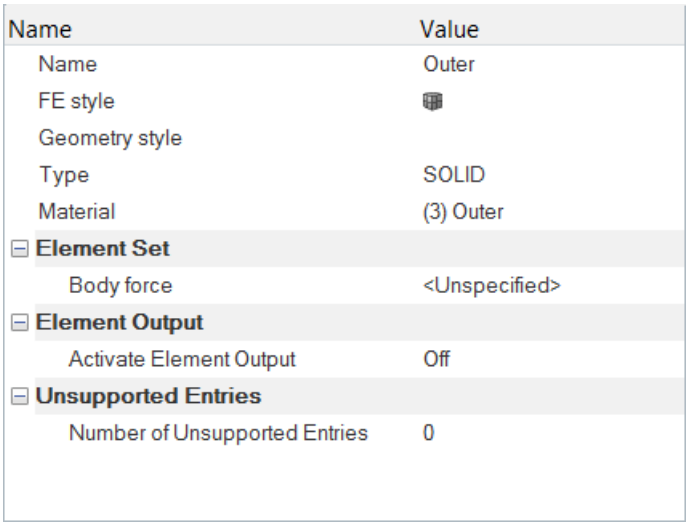


Figure 363:

5. Click **Inner_Inner_ri**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Change the Temperature BC type to **Value**.
   c) Set the Temperature to `300` K.

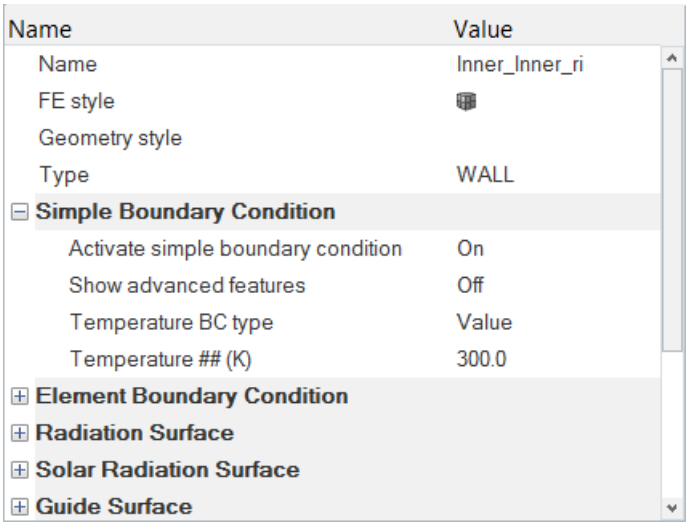


Figure 364:

6. Click **Outer_Outer_ro**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Change the Temperature BC type to **Value**.
   c) Set the Temperature to `1300` K.

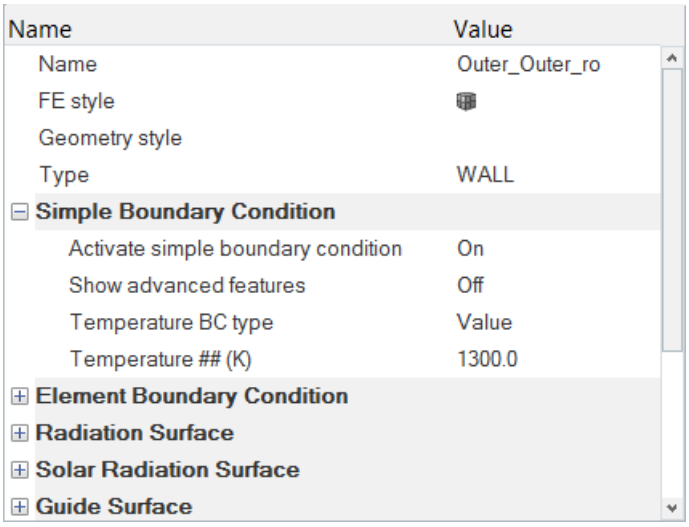


Figure 365:

7. Click **Inner_Radiating_r1**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Change the Temperature BC type to **Flux**.
   c) Under the Radiation Surface tab, activate the **Display** check box and turn **On** the Activate radiation surface field.
   d) Verify that the Type is set to **WALL** and select **Inner** as the Emissivity model.

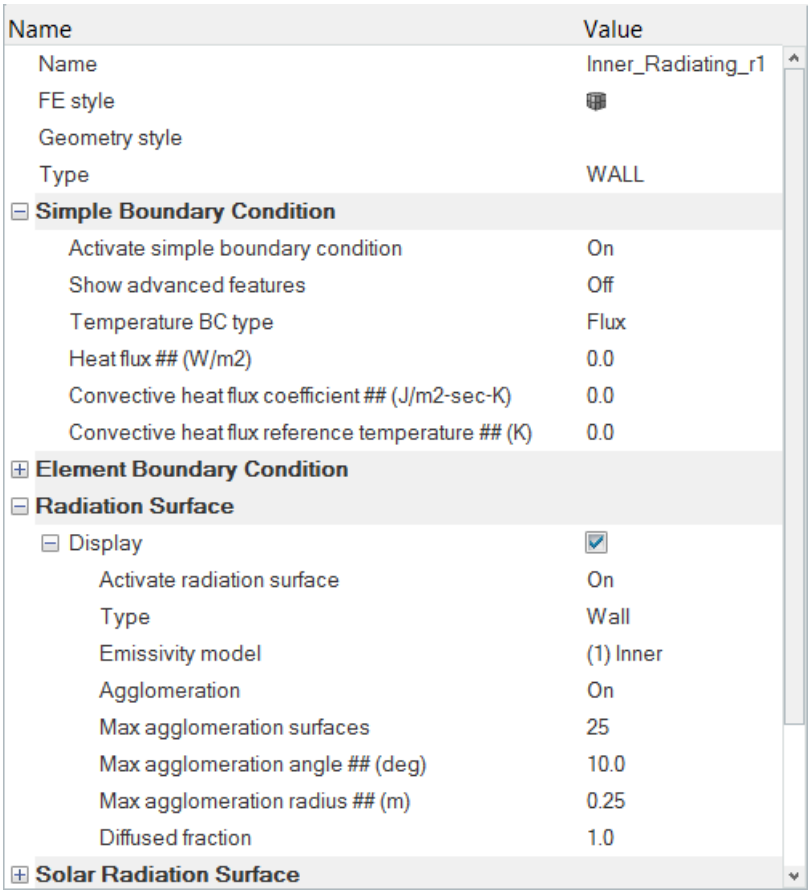


Figure 366:

8. Click **Outer_Radiating_r2**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Change the Temperature BC type to **Flux**.

c) Under the Radiation Surface tab, activate the **Display** check box and turn **On** the Activate radiation surface field.

d) Verify that the Type is set to **WALL** and select **Outer** as the Emissivity model.

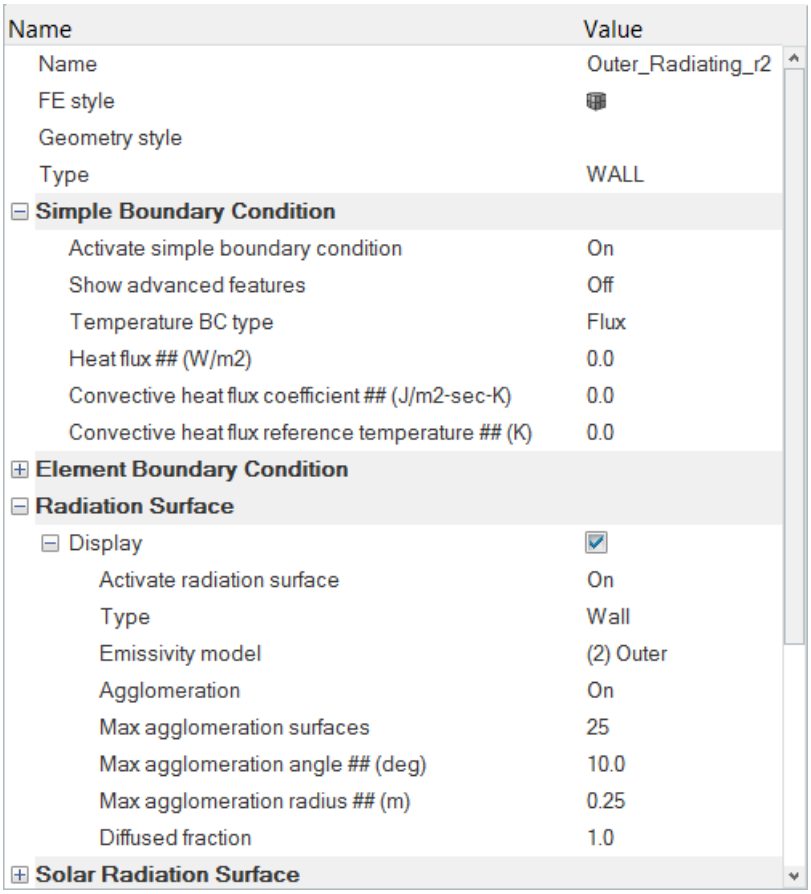


Figure 367:

9. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

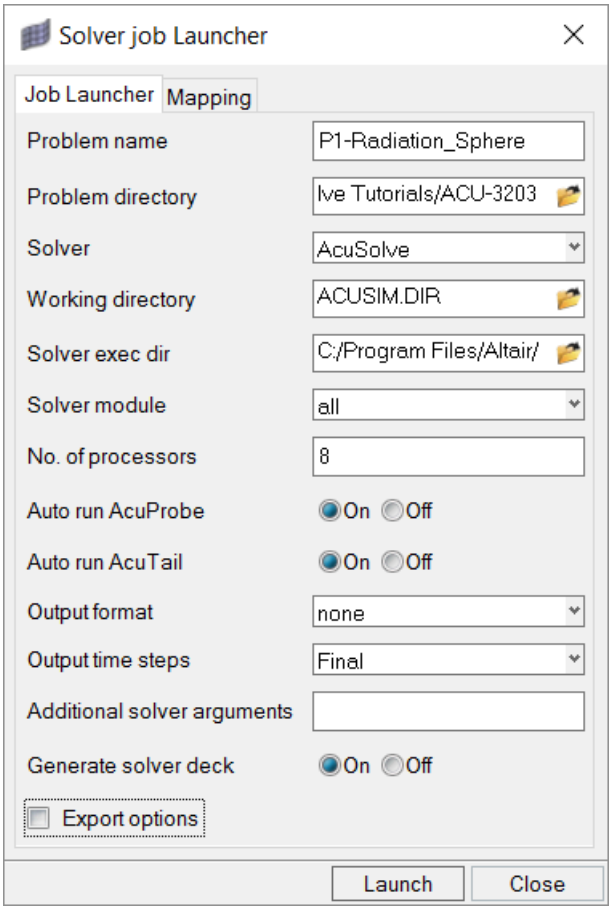


Figure 368:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `P1-Radiation_Sphere.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

The model is colored by geometry after loading.

Create Temperature Contours

In this step, you will create a contour plot of temperature distribution across the domain.

1. Click on the Results toolbar to open the Contour panel.
2. Under Result type, select **Temperature(s)**.
   The drop-down below should be automatically set to Scalar value.
3. Click **Apply**.
4. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 369:

5. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.
6. Right-click on empty space in the Results Browser and select **Create** > **Section Cut** > **Planar** to create a planar section cut.
   A new entity named Section 1 is created under the Section Cuts tree.
7. Click beside Section 1 to turn off the grid display in the graphics window.
8. Orient the display to the xz-plane by clicking on the Standard Views toolbar.
9. Verify that the contour plot looks like the figure below.

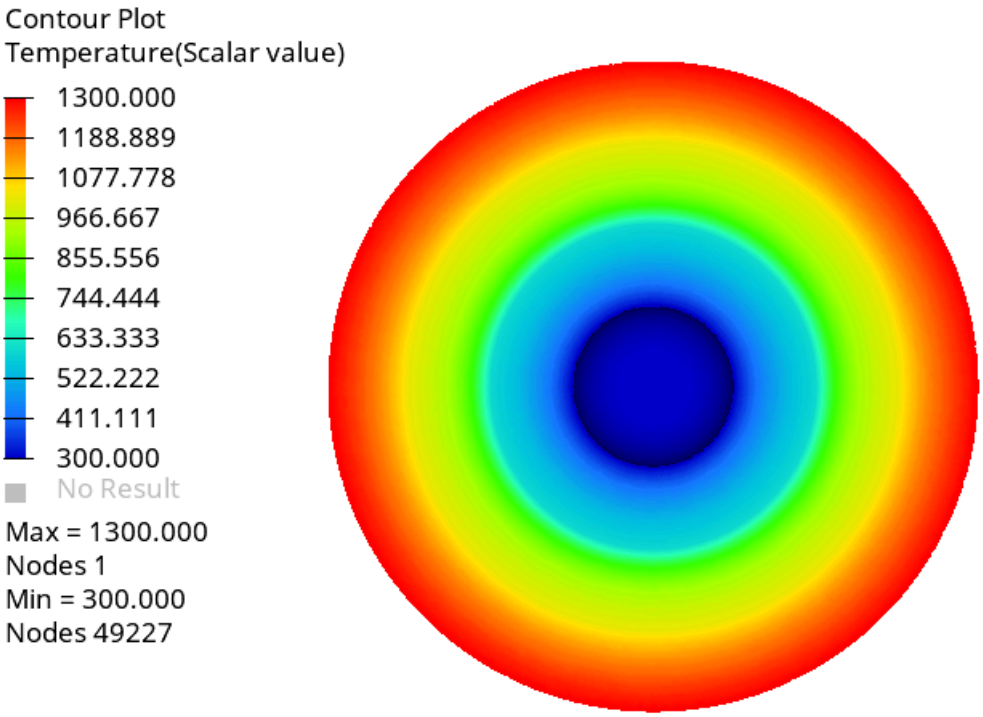


Figure 370:

Summary

In this tutorial, you worked through a workflow to set-up a P1-Radiation model, carry out a radiation heat transfer simulation, and post-process the results using HyperWorks products, namely AcuSolve, HyperMesh, and HyperView. You started by importing the model in Altair HyperMesh. Then you defined the simulation parameters and launched AcuSolve directly from within HyperMesh. Upon completion of the solution by AcuSolve, you used HyperView to post-process the results and create contour plots.

2.3.8 ACU-T: 3203 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model

Prerequisites

This tutorial provides instructions for setting, solving and viewing results for a steady state simulation of radiation heat transfer between concentric spheres using the Discrete Ordinates Radiation model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic knowledge of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3203_DiscreteOrdinate.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 371. In this problem, a DO radiation model is used to simulate the heat transfer due to radiation between concentric spheres. The inside surface of the inner and the outside surface of the outer sphere are both held at constant temperature while the gap between them radiates the heat from one sphere to the other.

The problem consists of a fluid region with arbitrary material properties between two concentric spheres with surfaces held at fixed temperature, as shown in the following figure, which is not drawn to scale. The radius of the outer sphere is 0.04 m and the radius of the inner sphere is 0.01 m. The inner surface of the inner sphere is defined to have a constant wall temperature at 300.0 K (26.85 ºC). The outer surface of the outer sphere is defined to have a constant wall temperature at 1300.0 K (1026.85 ºC). The fluid within the spheres is defined as a non-conducting material, allowing heat to transfer via radiation only.

The problem is solved as a steady state case to allow the heat transfer in the solid and fluid regions to reach an equilibrium.

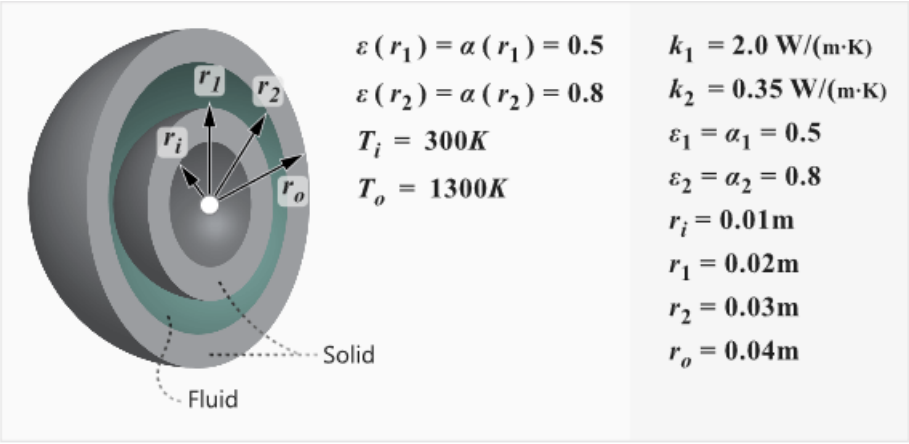


Figure 371:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.

2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3203_DiscreteOrdinate.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `DO-Radiation_Sphere` and navigate into this directory.
   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `DO-Radiation_Sphere` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, set the Temperature equation to **Advective Diffusive**.
3. Set the Radiation equation to **Discrete Ordinate**.
4. Verify that the Radiation quadrature is set to **S4**.

Name	Value
Title:	Radiation Heat Transfer Between Concentric Spheres
Analysis type:	Steady State
Flow equation:	Navier Stokes
Abs. pressure offs...	0.0
Temperature equ...	Advective Diffusive
Abs. temperature ...	0.0
Radiation equation:	Discrete Ordinate
Radiation quadrat...	S4
Turbulence model:	Laminar
Humid Air Model:	Off
Multiphase equati...	None
Optimization:	Off
Mesh type:	Fixed
Nucleate boiling s...	Off
External code:	Off
Global Gravity	
X ## (m/sec2):	0.0
Y ## (m/sec2):	0.0
Z ## (m/sec2):	0.0
Unsupported Entries	
No of Analysis...	0
No of Equation...	0

Figure 372:

5. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.

6. In the Entity Editor, set the Convergence tolerance to `0.001`.
7. Verify that the Relaxation Factor is set to `0.3`.
8. Turn off Flow and verify that the Temperature and Radiation fields are turned **On**.

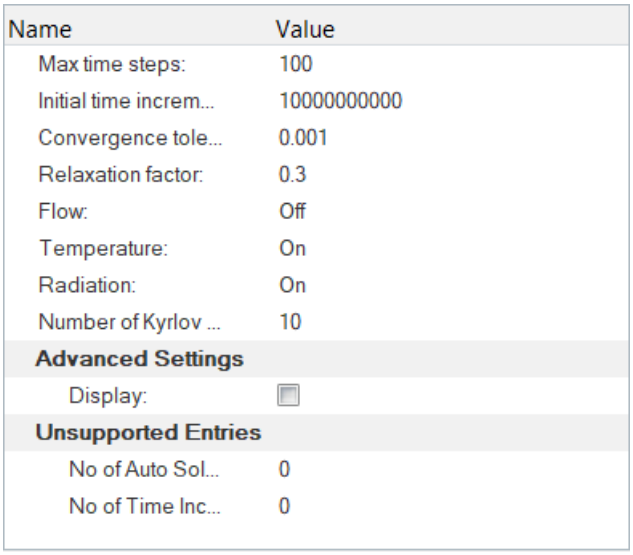


Figure 373:

Set Up Radiation Parameters and Boundary Conditions

In this step, you will define the radiation parameters i.e. emissivity models, surface boundary conditions for the problem, and assign material properties to the fluid and solid regions.

Set Up Material Model Parameters

1. In the Solver Browser, right-click on **02.Materials** and select **Material(Fluid)**.
2. Name the material `Radiating`.
3. In the Entity Editor, set the Density to `1000` kg/m^3.
4. Set the Specific heat to `10000` J/kg-K.
5. Set the Conductivity to `1e-6` W/m-K.
   This is done to prioritize heat transfer only through radiation.
6. Under Radiation Properties, turn on **Allow Participating Media Radiation**.
7. Set the Absorption coefficient to `0.001`.

Name	Value
Name:	Radiating
Defined:	☑
Material type:	FLUID
Density	
Density type:	Constant
Density ## (kg/m3):	1000.0
Isothermal compressibility ## (m2/N):	0.0
Specific Heat	
Specific heat type:	Constant
Specific heat ## (J/kg-K):	10000.0
Latent heat type:	None
Viscosity	
Viscosity type:	Constant
Viscosity ## (kg/m-sec):	1e-06
Multiplier Function:	<Unspecified>
Conductivity	
Conductivity type:	Constant
Conductivity ## (W/m-K):	1e-06
Multiplier Function:	<Unspecified>
Turbulent Prandtl number:	0.91
Radiation Properties	
Allow Participating Media Radiation:	☑
Absorption coefficient:	0.001
Scattering coefficient:	0.0
Phase coefficient:	0.0
Refractive index:	1.0
Porosity	
Porosity type:	None

Figure 374:

8. In the Solver Browser, right-click on **02.Materials** and select **Material(Solid)**.
9. Name the material `Inner`.
10. In the Entity Editor, set the Density to `1000` kg/m^3.
11. Set the Specific heat to `10000` J/kg-K.
12. Set the Conductivity to `2` W/m-K.
13. In the Solver Browser, right-click on **Inner** and select **Duplicate**. Rename it `Outer`.
14. In the Entity Editor, change the Conductivity to `0.35` W/m-K.
15. Verify that the **Allow Participating Media Radiation** is turned off for both Inner and Outer. This is done because only the fluid medium participates in heat transfer through radiation.

Set Up Emissivity Model Parameters

1. In the Solver Browser, right-click on **07.Emissivity_Model** collector and select **Create**.
2. Name the emissivity model `Inner`.
3. In the Entity Editor, set the Emissivity to `0.5`.

Name	Value
Solver Keyword	EMISSIVITY_MODEL
Name	Inner
Type	Emissivity_Model
Emissivity Model Type	Constant
Emissivity	0.5
Multiplier Function	<Unspecified>
Unsupported Entries	
Number of Unsupported Entries	0

Figure 375:

4. Repeat the above steps and create another emissivity model named `Outer` with an Emissivity of `0.8`.

Set Up Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Radiating**. In the Entity Editor, change the Type to **FLUID** and set **Radiating** as the Material.

Name	Value
Name	Radiating
FE style	
Geometry style	
Type	FLUID
Material	(1) Radiating
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 376:

3. Click **Inner**. In the Entity Editor, change the Type to **SOLID** and set **Inner** as the Material.

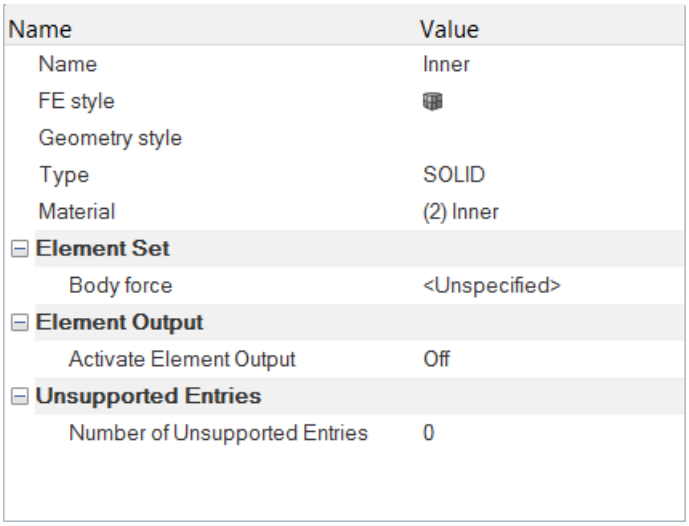


Figure 377:

4. Click **Outer**. In the Entity Editor, change the Type to **SOLID** and set **Outer** as the Material.

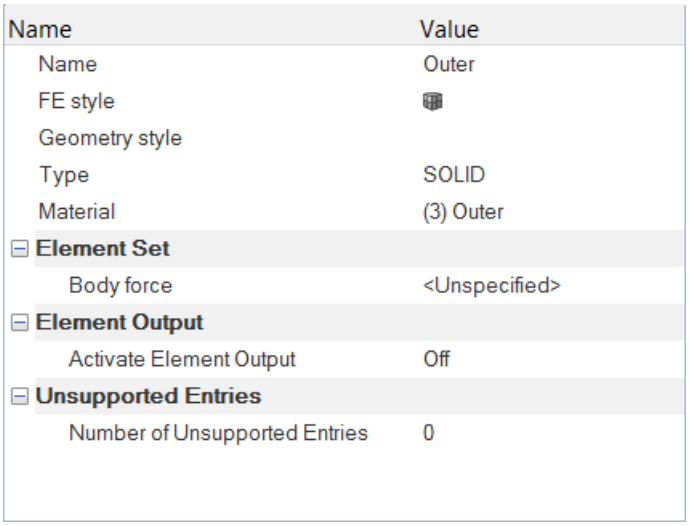


Figure 378:

5. Click **Inner_Inner_ri**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Change the Temperature BC type to **Value**.
   c) Set the Temperature to `300` K.

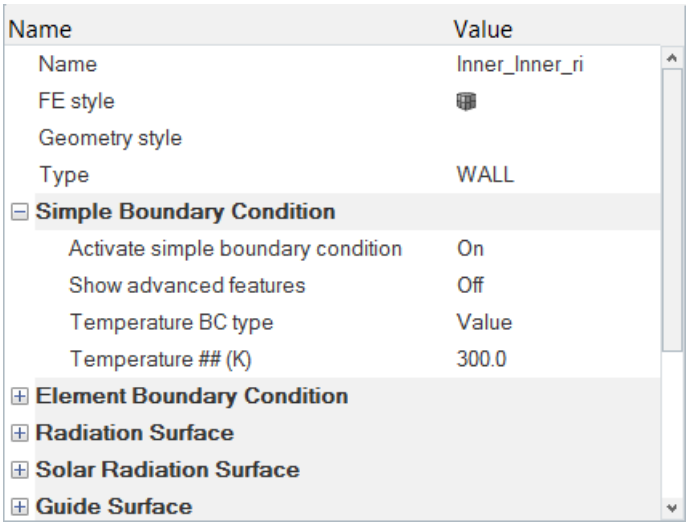


Figure 379:

6. Click **Outer_Outer_ro**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Change the Temperature BC type to **Value**.
   c) Set the Temperature to `1300` K.

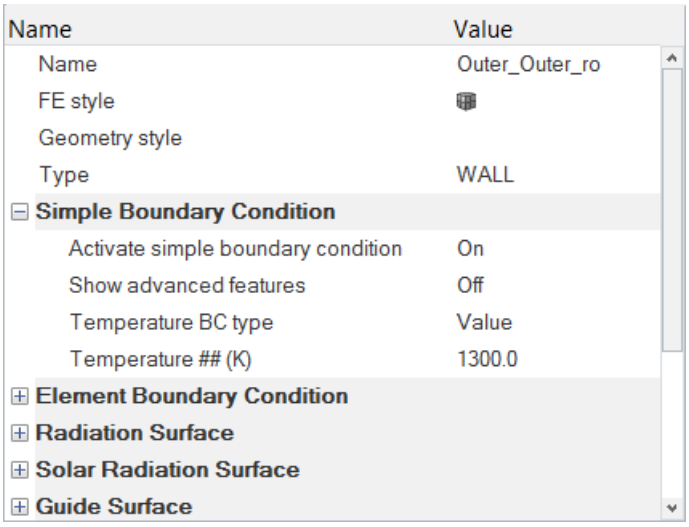


Figure 380:

7. Click **Inner_Radiating_r1**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Change the Temperature BC type to **Flux**.
   c) Under the Radiation Surface tab, activate the **Display** check box and turn **On** the Activate radiation surface field.
   d) Verify that the Type is set to **WALL** and select **Inner** as the Emissivity model.

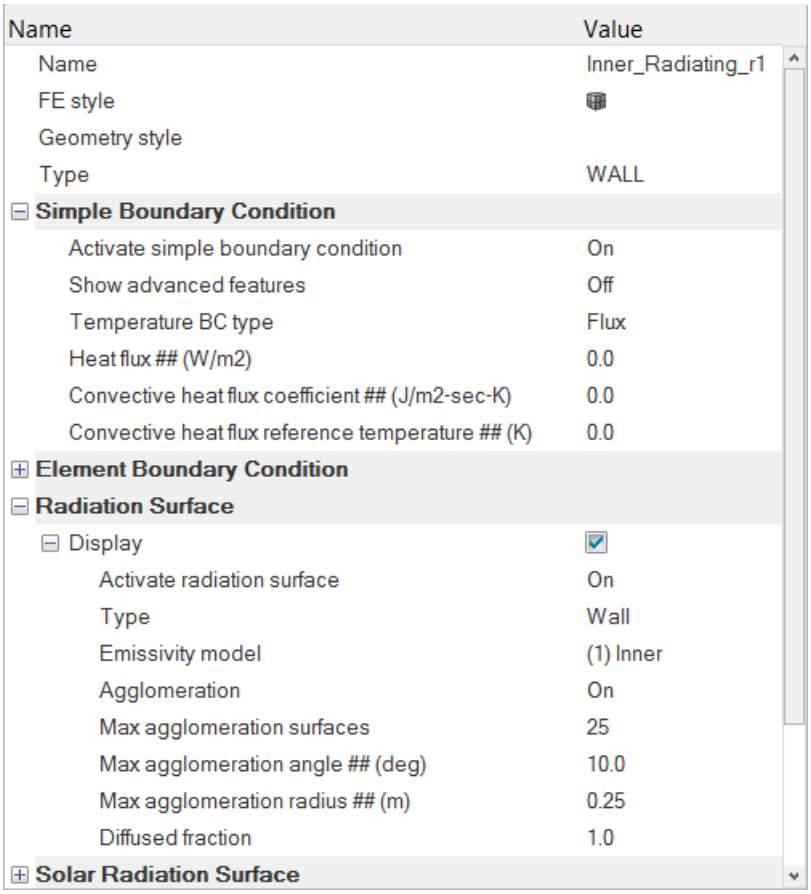


Figure 381:

8. Click **Outer_Radiating_r2**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Change the Temperature BC type to **Flux**.

c) Under the Radiation Surface tab, activate the **Display** check box and turn **On** the Activate radiation surface field.

d) Verify that the Type is set to **WALL** and select **Outer** as the Emissivity model.

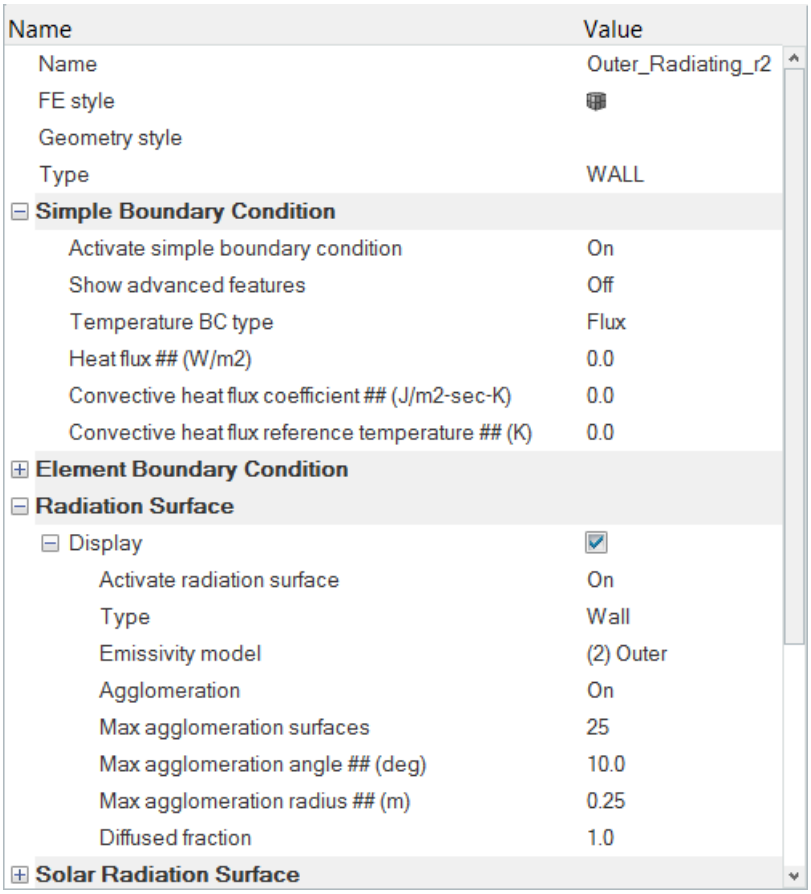


Figure 382:

9. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

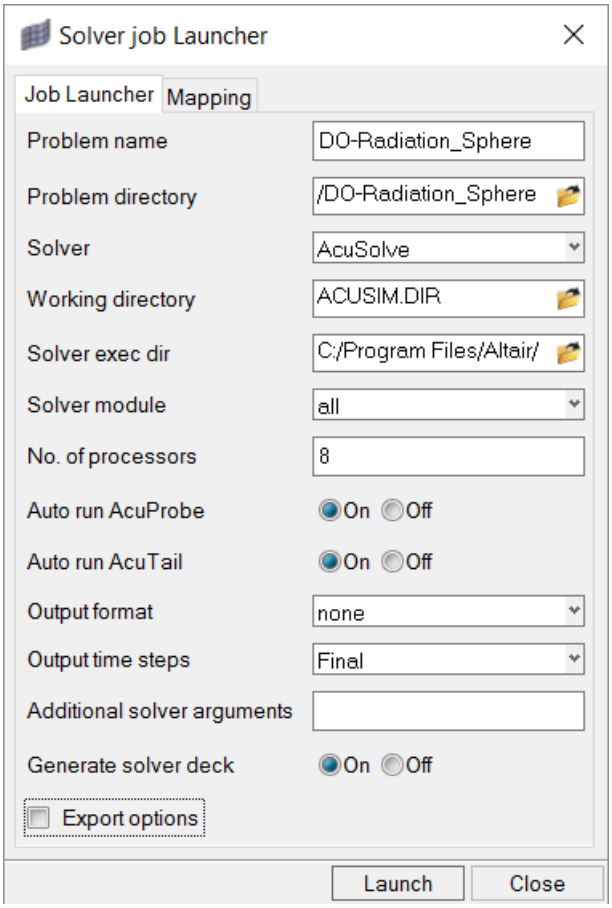


Figure 383:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `DO-Radiation_Sphere.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

The model is colored by geometry after loading.

Create Temperature Contours

In this step, you will create a contour plot of temperature distribution across the domain.

1. Click on the Results toolbar to open the Contour panel.
2. Under Result type, select **Temperature(s)**.
   The drop-down below should be automatically set to Scalar value.
3. Click **Apply**.
4. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 384:

5. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.
6. Right-click on empty space in the Results Browser and select **Create** > **Section Cut** > **Planar** to create a planar section cut.
   A new entity named Section 1 is created under the Section Cuts tree.
7. Click beside Section 1 to turn off the grid display in the graphics window.
8. Orient the display to the xz-plane by clicking on the Standard Views toolbar.
9. Verify that the contour plot looks like the figure below.

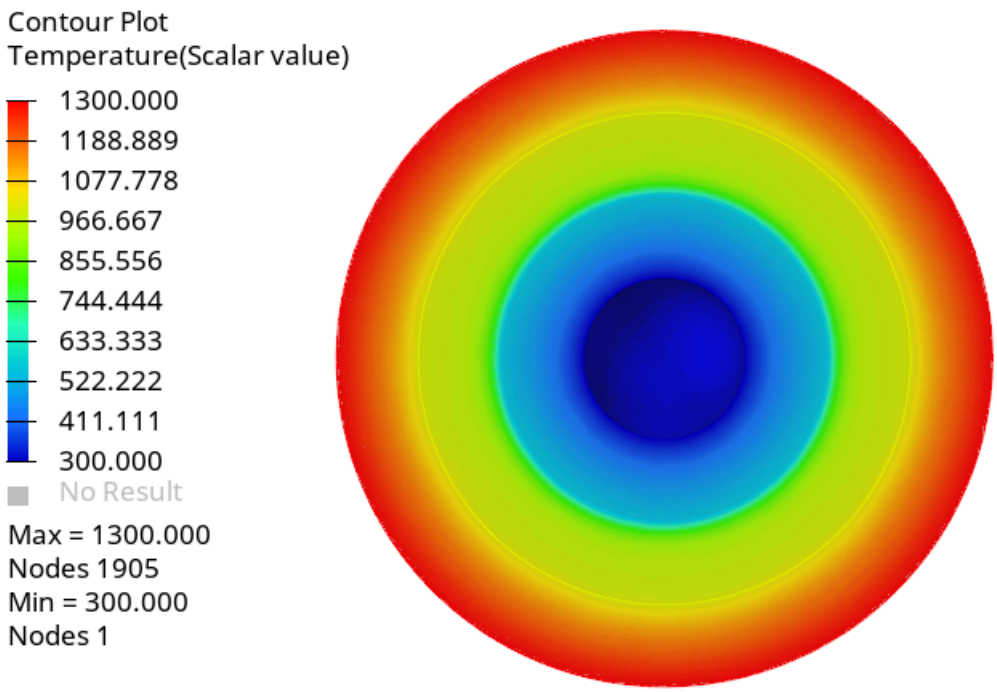


Figure 385:

Summary

In this tutorial, you worked through a workflow to set-up a DO-Radiation model, carry out a radiation heat transfer simulation, and post-process the results using HyperWorks products, namely AcuSolve, HyperMesh, and HyperView. You started by importing the model in Altair HyperMesh. Then you defined the simulation parameters and launched AcuSolve directly from within HyperMesh. Upon completion of the solution by AcuSolve, you used HyperView to post-process the results and create contour plots.

2.3.9 ACU-T: 3204 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model

Prerequisites

This tutorial introduces you to setting up a radiation heat transfer problem using the Discrete Ordinate radiation model in HyperMesh and solving using AcuSolve. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3204_HeadlampDO.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be solved is shown schematically in Figure 386 and Figure 387. It consists of a simple headlamp with a housing, lens, and a bulb. The inner cavity of the bulb is filled with air and the wattage of the bulb is 1W, which is modeled as a volumetric heat source. The Boussinesq density model is used for the air to consider the natural convection effects in the fluid volume. The heat generated in the bulb is transferred by three means: conduction from the bulb to the housing, natural convection in the air volume, and radiation from the bulb to the air and lens volume. The outer surfaces of the headlamp are assumed to be at a constant temperature of 300 K.

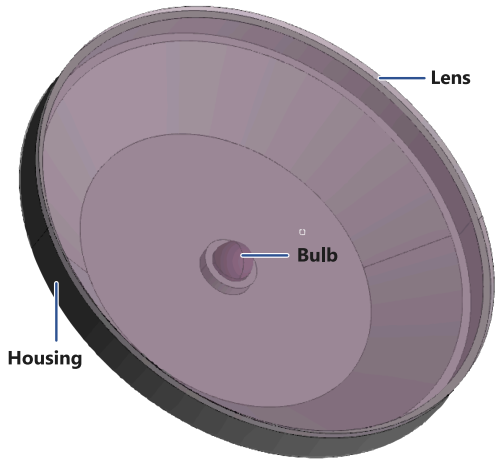


Figure 386:

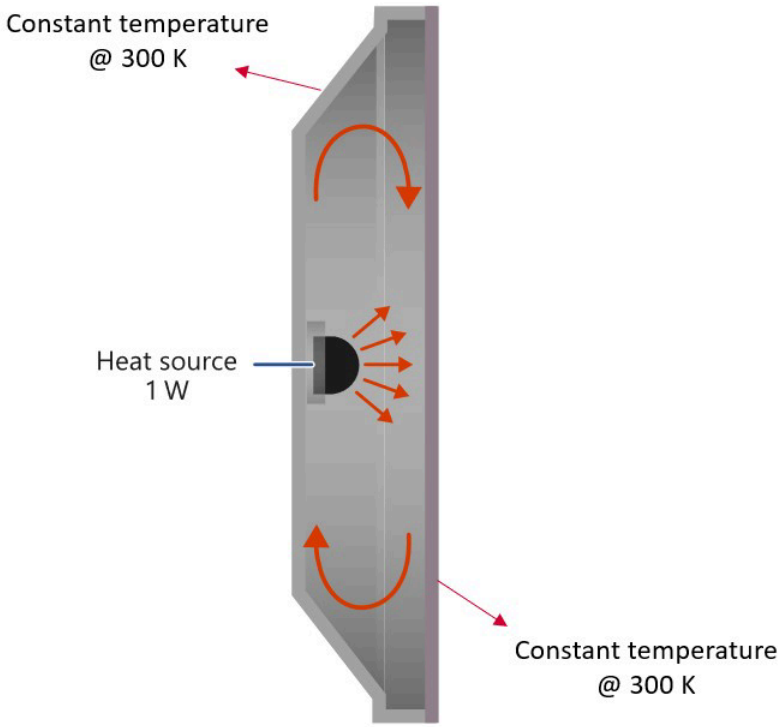


Figure 387:

The air and the lens are modeled as participating media for radiation heat transfer. For the purpose of this tutorial, the absorption coefficient of air is considered to be zero. The following radiation material properties are used for the air and the lens volumes.

Table 1:

	Absorption Coefficient	**Refractive Index**
Air	0	1.0
Lens	900	1.57

For simulations involving participating media, the following radiation surface types must be defined at the boundaries of each participating medium.

Table 2:

Surface Type	**Radiation Surface Type**
Participating medium - Participating medium Interface	Radiation Interface - Internal
Participating medium - (Non-Participating) medium Interface	Wall

Surface Type	Radiation Surface Type
External boundaries of Participating medium	Radiation Interface - External or Wall

The discrete ordinate radiation model implementation in AcuSolve allows for modeling specular, diffuse, and partially specular interfaces. This can be done by specifying the appropriate value of diffused fraction while defining the radiation surfaces. For the diffused fraction, a value of 1 means the surface is purely diffuse, a value of 0 means it is purely specular. Any value between 0 and 1 means the surface is partially specular. In this tutorial, since both the air and lens are modeled as participating media, the lens-air interface and the lens-outer surfaces will be modeled as internal and external radiation interface respectively. The internal interface will be defined as a specular interface whereas the external interface will be defined as a diffuse interface.

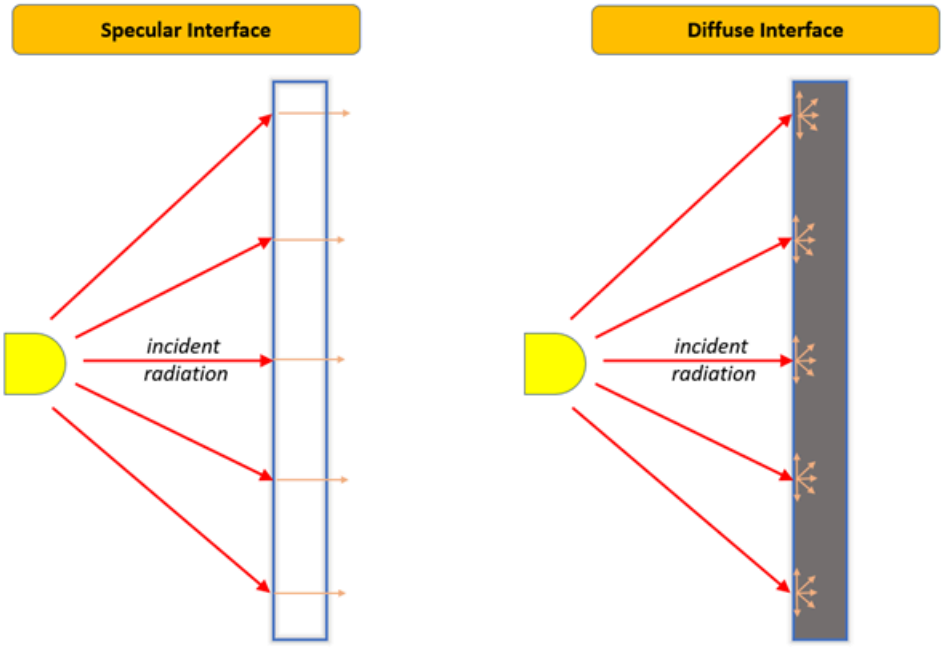


Figure 388:

Open the HyperMesh Model Database

1. Start HyperMesh Desktop and load the AcuSolve user profile.
   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.
   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3204_HeadlampDO.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.

5. Create a new directory named `Headlamp_DO` and navigate into this directory.
   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Headlamp_DO` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, verify that the Analysis type is set to **Steady State**.
3. Set the Temperature equation to **Advective Diffusive**.
4. Set the Radiation equation to **Discrete Ordinate**.
5. Verify that the Radiation quadrature is set to **S4**.
6. Set the Turbulence model to **Laminar** (if not set already).

Name	Value
Title	Headlamp Radiation using Discrete Ordinate Model
Analysis type	Steady State
Flow equation	Navier Stokes
Abs. pressure offset ## (N/m2)	0.0
Temperature equation	Advective Diffusive
Abs. temperature offset ## (K)	0.0
Radiation equation	Discrete Ordinate
Radiation quadrature	S4
Turbulence model	Laminar
Humid Air Model	Off
Multiphase equation	None
Optimization	Off
Mesh type	Fixed
Nucleate boiling single phase	Off
External code	Off
Global Gravity	
X ## (m/sec2)	0.0
Y ## (m/sec2)	0.0
Z ## (m/sec2)	0.0
Unsupported Entries	
No of Analysis Unsupported Entries	0
No of Equation Unsupported Entries	0

Figure 389:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, change the Relaxation factor to `0.4`.
3. Turn **On** Temperature flow.
4. Leave the remaining options unchanged.

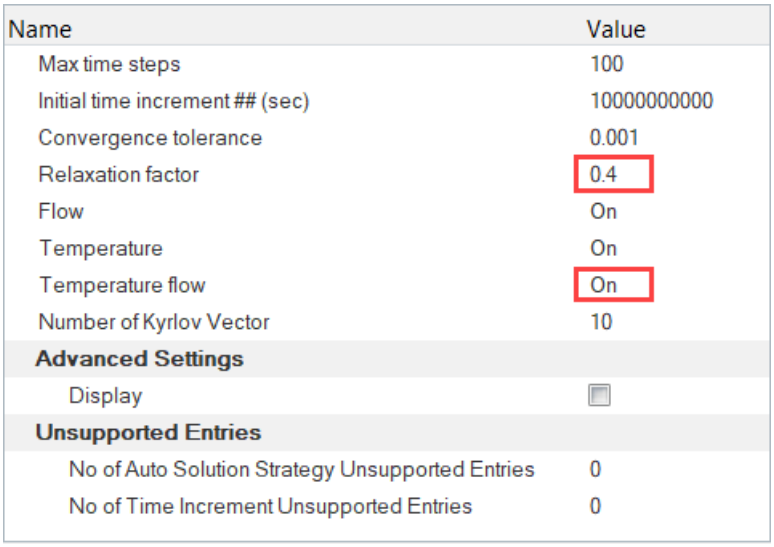


Figure 390:

Define the Material Models and the Body Force

Define the Material Models

1. In the Solver Browser, expand **02.Materials** > **Fluid** then click on **Air_HM**.
2. In the Entity Editor, change the Density type to **Boussinesq**.
3. Under Radiation Properties, activate the **Allow Participating Media Radiation** option.
4. Set the Absorption coefficient to `0` and Refractive index to `1`.

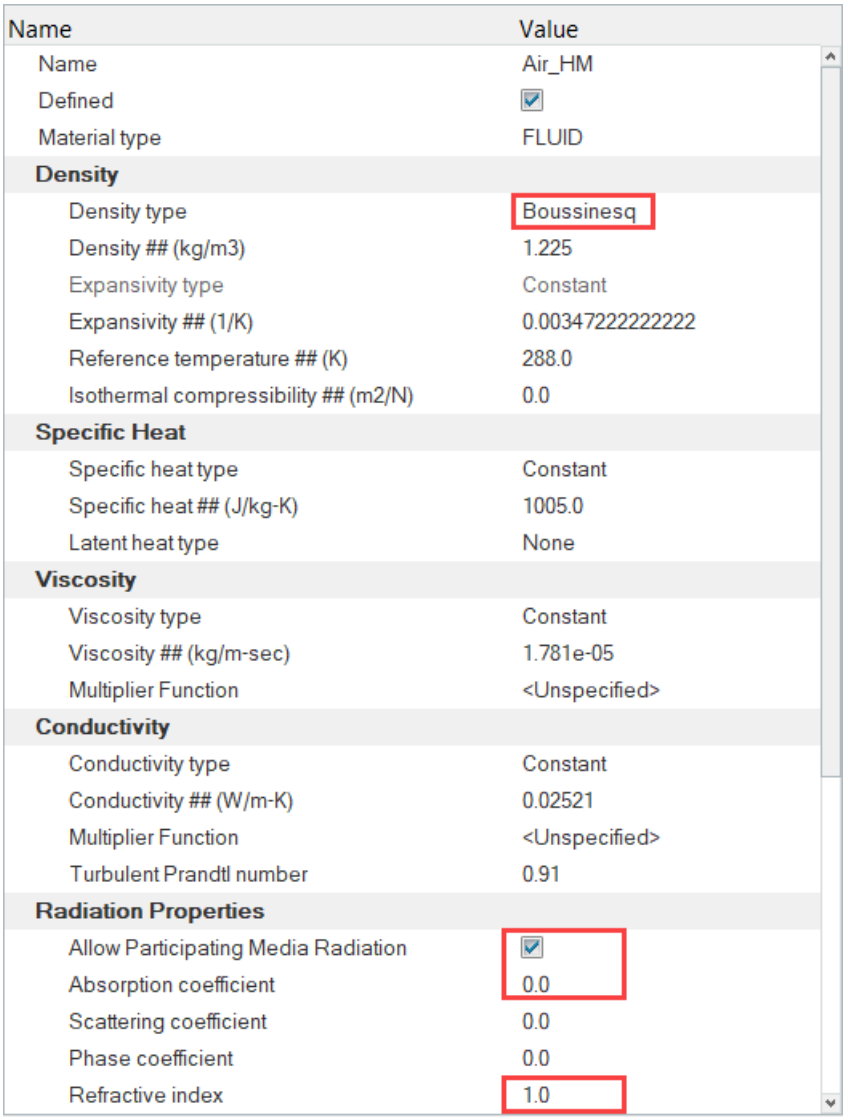


Figure 391:

5. In the Solver Browser, under 02.Materials, right-click on **SOLID** and select **Create**.
6. In the Entity Editor, name it `Arnite`.
7. Set the Density to `1670` kg/m^3.
8. Set the Specific heat to `2050` J/kg-K.
9. Set the Conductivity to `1.65` W/m-K.
10. Under Radiation Properties, activate the **Allow Participating Media Radiation** option.
11. Set the Absorption coefficient to `900` and the Refractive index to `1.57`.

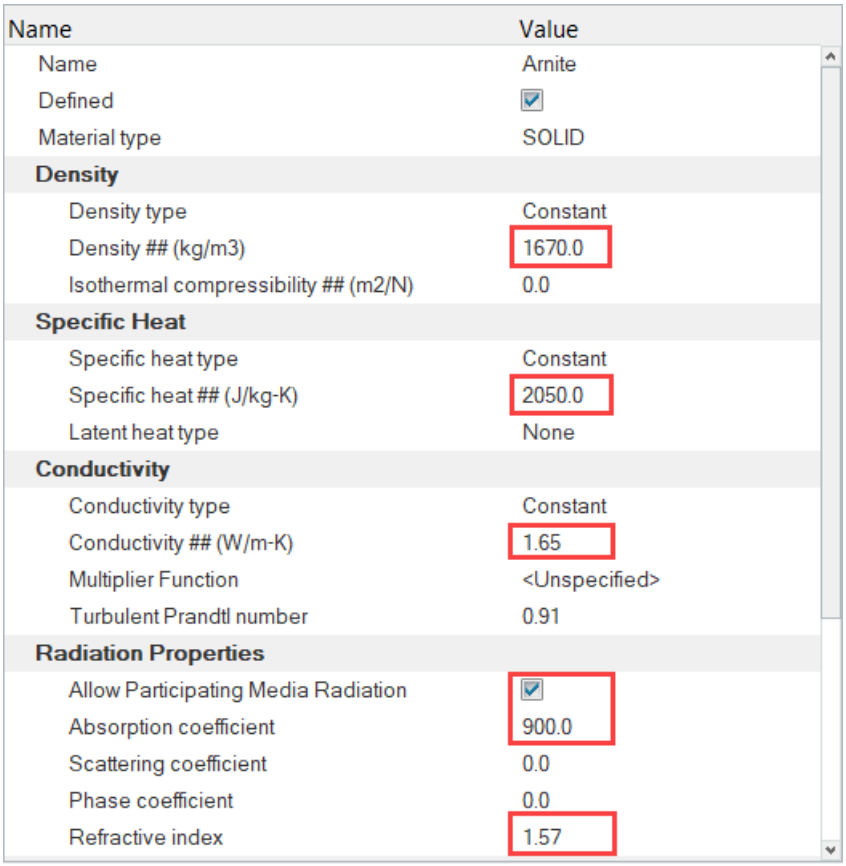


Figure 392:

12. Repeat steps 5-9 and create two more solid material models, `Plastic` and `LED`, with the following material properties:

Radiation properties do not need to be defined for these two materials.

1. Plastic:
   - **a.** Density - `1270` kg/m^3
   - **b.** Specific heat - `1900` J/kg-K
   - **c.** Conductivity to `0.22` W/m-K
2. LED:
   - **a.** Density - `5500` kg/m^3
   - **b.** Specific heat - `0.3` J/kg-K
   - **c.** Conductivity to `5.0` W/m-K

13. Save the database.

Define the Body Force

1. In the Solver Browser, expand **03.Body_Force** > **BODY_FORCE** then click on **Gravity_HM**.
2. In the Entity Editor, set the Y-Gravity to `-9.81` m/sec^2 and change to Z-Gravity to `0`.

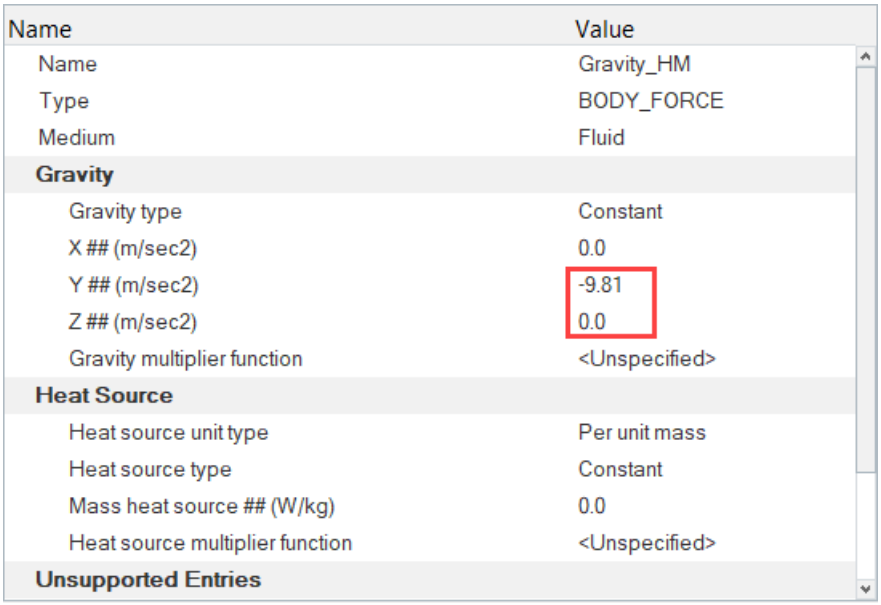


Figure 393:

Define the Heat Source

1. In the Solver Browser, right-click on **03.Body_Force** and select **Create**.
2. In the Entity Editor, name it `LED Heat Source`.
3. Change the Medium to **Solid**.
4. Set the Heat source unit type to **Per unit volume**.
5. Set the Heat Source type to **Constant** and set the Volumetric heat source to `2049180` W/m^3.

Name	Value
Name	LED Heat Source
Type	BODY_FORCE
Medium	Solid
Heat Source	
Heat source unit type	Per unit volume
Heat source type	Constant
Volumetric heat source ## (W/m3)	2049180.0
Heat source multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

Figure 394:

Set the Boundary Conditions

Create the Emissivity Model

1. In the Solver Browser, right-click on **07.Emissivity_Model** and select **Create**.
2. In the Entity Editor, name it `Inner`.
3. Set the Emissivity to `0.05`.

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Air**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Set the Material to **Air_HM**.
   c) Set the Body force to **Gravity_HM**.

Name	Value
Name	Air
Type	FLUID
Material	(1) Air_HM
FE style	
Geometry style	
Element Set	
Body force	(1) Gravity_HM
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 395:

3. Click **Housing**. In the Entity Editor,
   a) Change the Type to **SOLID**.
   b) Set the Material to **Plastic**.

Name	Value
Name	Housing
Type	SOLID
Material	(8) Plastic
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 396:

4. Click **Bulb**. In the Entity Editor,
   a) Change the Type to **SOLID**.
   b) Set the Material to **LED**.
   c) Set the Body force to **LED Heat Source**.

Name	Value
Name	Bulb
Type	SOLID
Material	(9) LED
FE style	
Geometry style	
Element Set	
Body force	(2) LED Heat Source
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 397:

5. Click **Lens**. In the Entity Editor,

a) Change the Type to **SOLID**.

b) Set the Material to **Arnite**.

Name	Value
Name	Lens
Type	SOLID
Material	(7) Arnite
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 398:

6. Click **Lens-inner**. In the Entity Editor, verify that the Type is set to **WALL**. Under the Radiation Surface tab,

a) Activate the **Display** checkbox and set the Active radiation surface field to **On**.

b) Set the Type to **Radiation Interface** and the Radiation interface type to **Internal**.

c) Set the Diffused fraction to `0`.

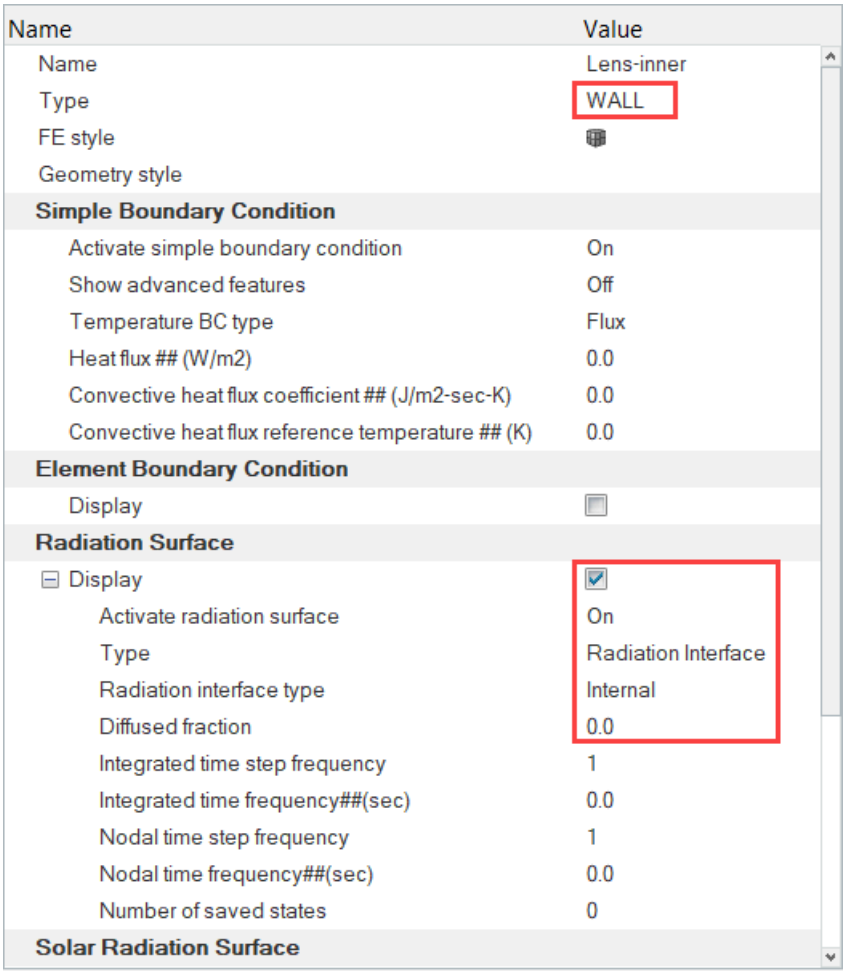


Figure 399:

7. Click **Lens-outer**. In the Entity Editor, verify that the Type is set to **WALL**. Set the Temperature BC type to **Value** and set the Temperature to `300` K. Under the Radiation Surface tab,

a) Activate the **Display** checkbox and set the Active radiation surface field to **On**.

b) Set the Type to **Radiation Interface** and the Radiation interface type to **External**.

c) Set the External emissivity model to **Black Body**.

d) Set the External refractive index to `1` and the External temperature to `300` K.

e) Set the Diffused fraction to `1`.

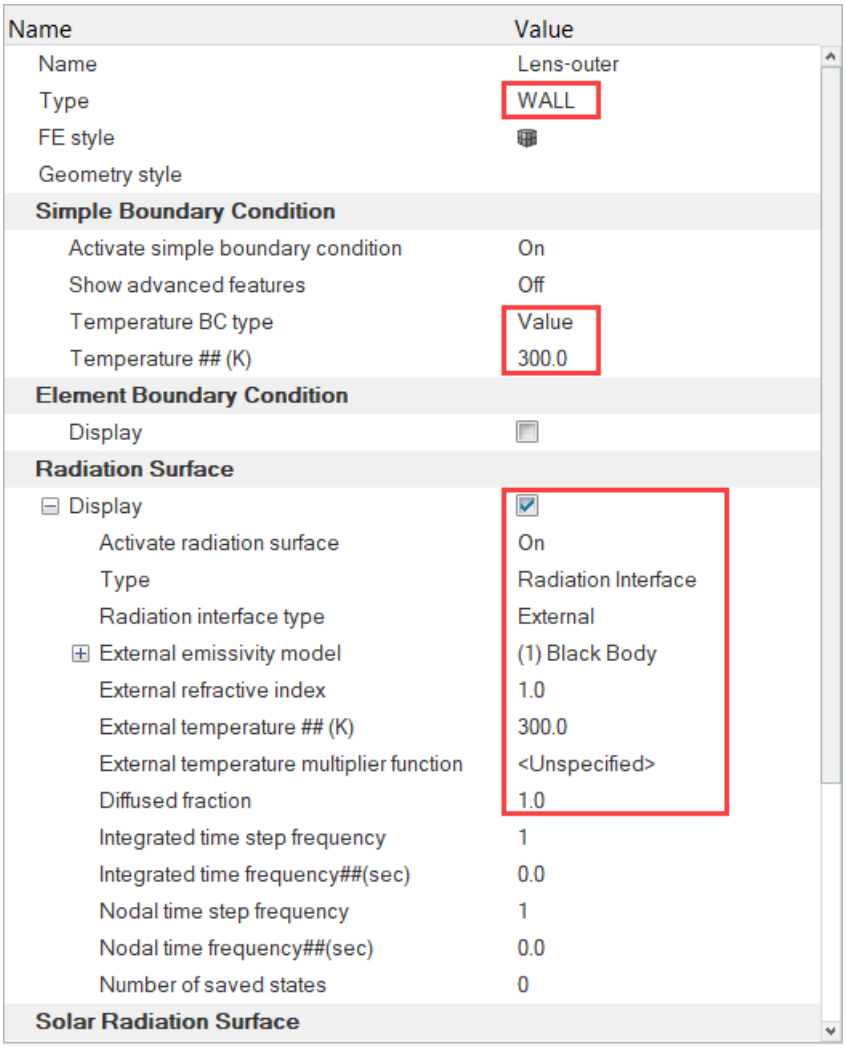


Figure 400:

8. Click **Housing-Lens-interface**. In the Entity Editor, verify that the Type is set to **WALL**. Under the Radiation Surface tab,

a) Activate the **Display** checkbox and set the Active radiation surface field to **On**.

b) Set the Type to **Wall** and the Emissivity model to **Black Body**.

c) Set the Diffused fraction to `1`.

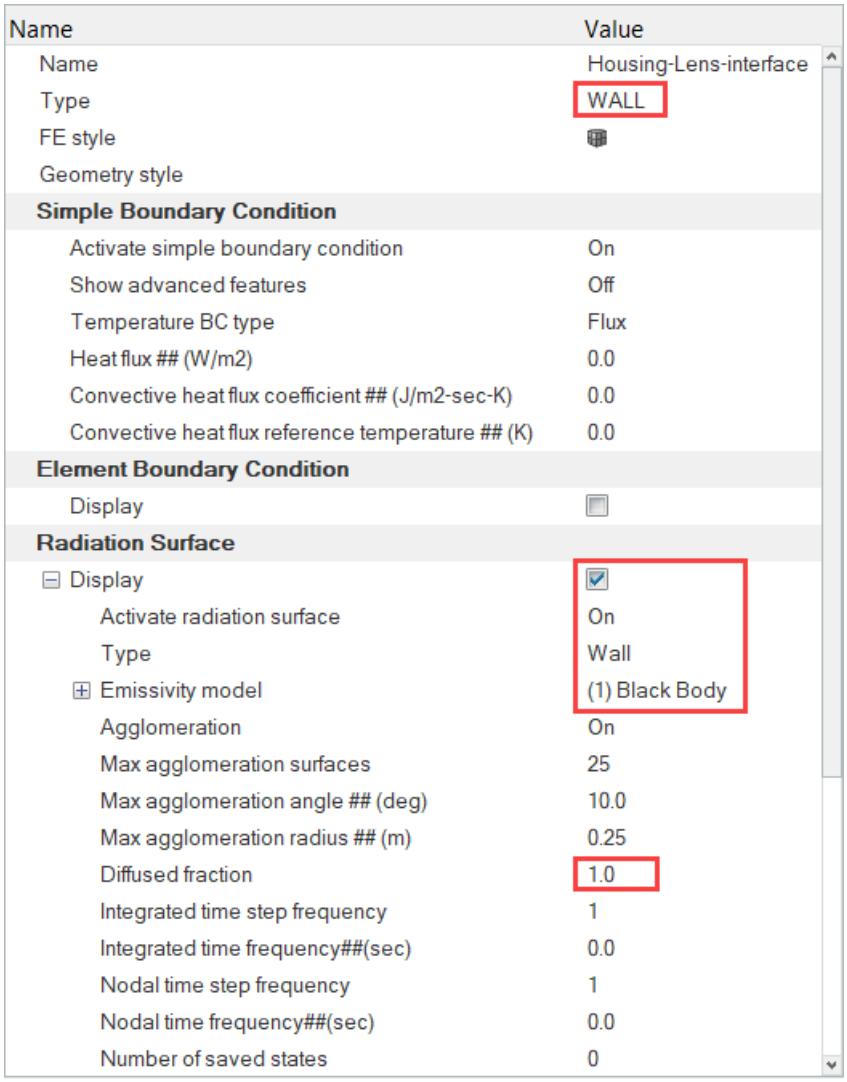


Figure 401:

9. Click **Housing-inner**. In the Entity Editor, verify that the Type is set to **WALL**. Under the Radiation Surface tab,

a) Activate the **Display** checkbox and set the Active radiation surface field to **On**.

b) Set the Type to **Wall** and the Emissivity model to **Inner**.

c) Set the Diffused fraction to `1`.

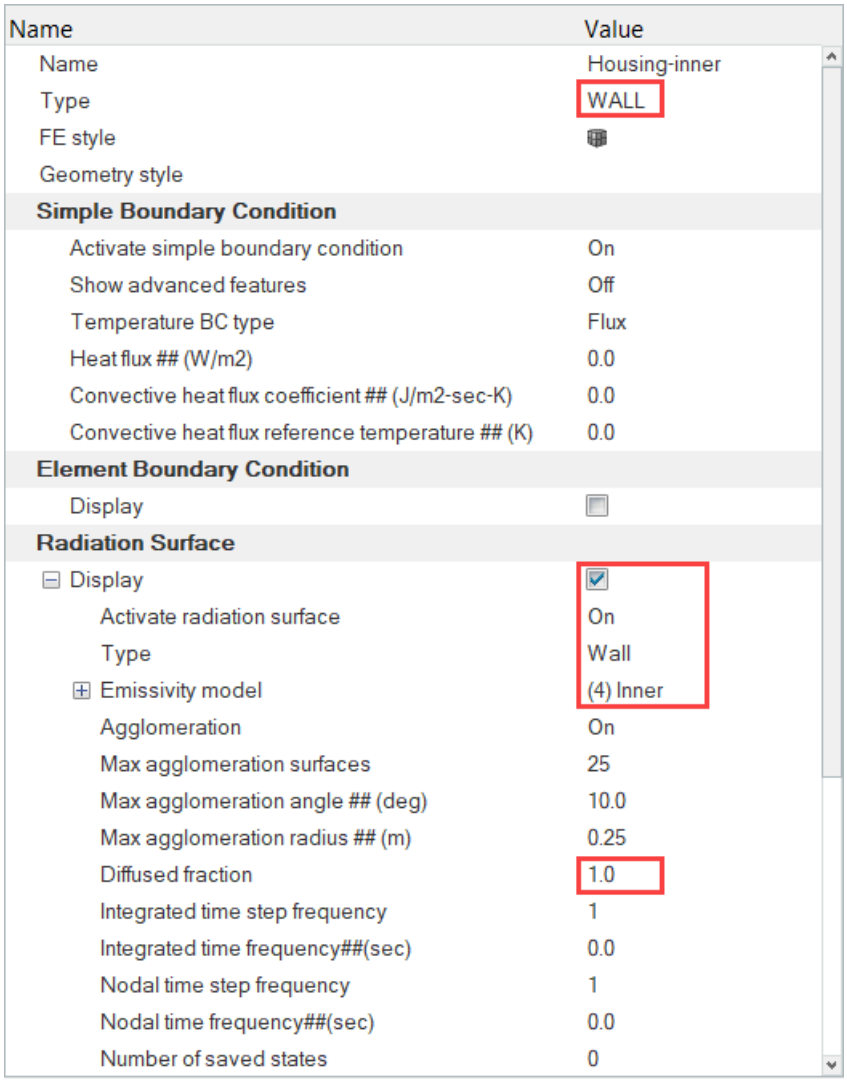


Figure 402:

10. Click **Housing-outer**. In the Entity Editor,

a) Verify that the Type is set to **WALL**.

b) Set the Temperature BC type to **Value**.

c) Set the Temperature to `300` K.

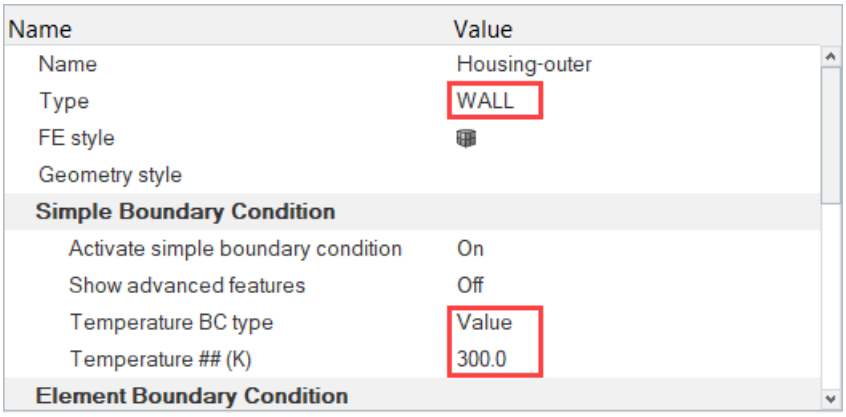


Figure 403:

11. Click **Bulb_walls**. In the Entity Editor, verify that the Type is set to **WALL**. Under the Radiation Surface tab,

a) Activate the **Display** checkbox and set the Active radiation surface field to **On**.

b) Set the Type to **Wall** and the Emissivity model to **Black Body**.

c) Set the Diffused fraction to `1`.

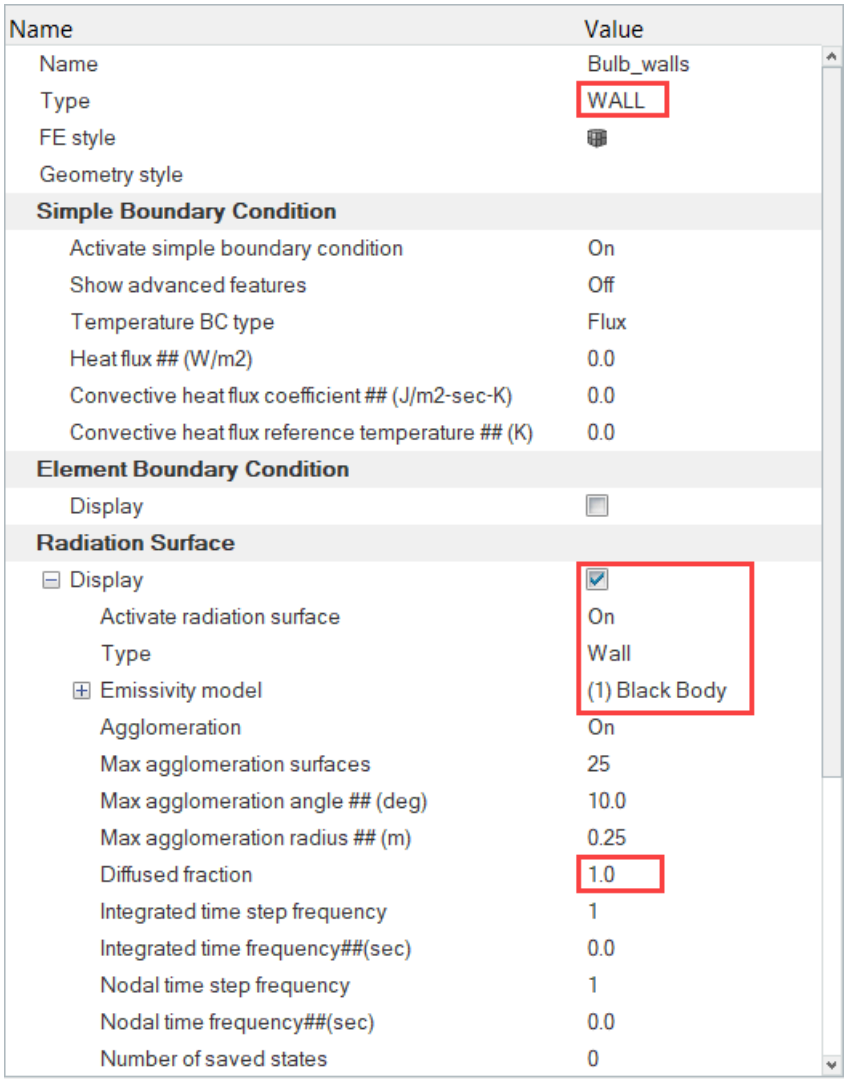


Figure 404:

12. Save the model.

Compute the Solution

Run AcuSolve

In this step, you will launch AcuSolve to compute a solution for this case.

1. Turn on the visibility of all mesh components.
For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.
The **Solver job Launcher** dialog opens.

3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Leave the remaining options as default and click **Launch** to start the solution process.

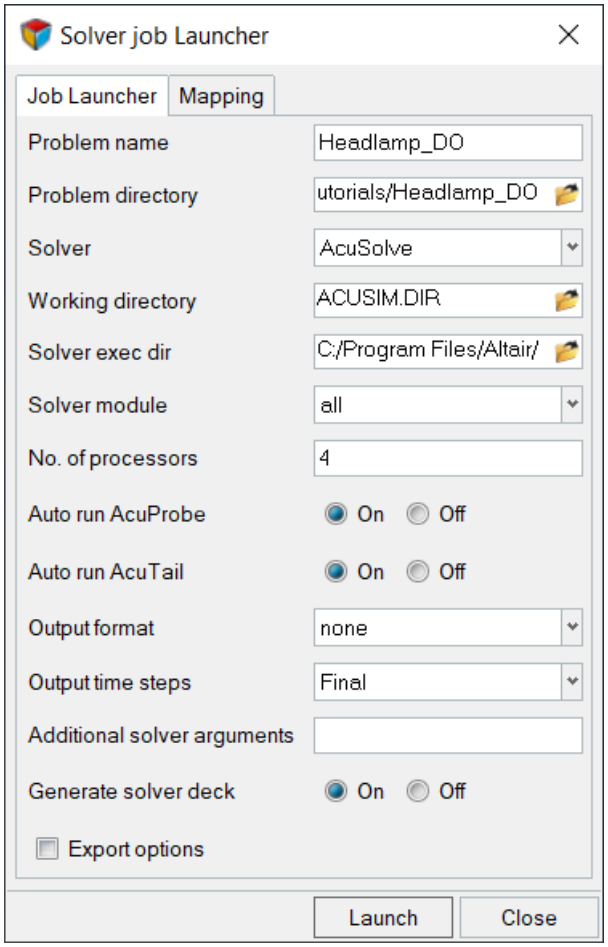


Figure 405:

Monitor the Solution with AcuProbe

While AcuSolve is running, you can monitor the progress of the solution using AcuProbe and plot the values of residual ratios, solution ratios, and variables like temperature, heat flux, etc. Once the solver run has started, the AcuTail and AcuProbe windows should be launched automatically.

1. In the AcuProbe window, under the Data Tree, expand **Residual Ratio**.
2. Right-click on **All** and select **Plot All**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

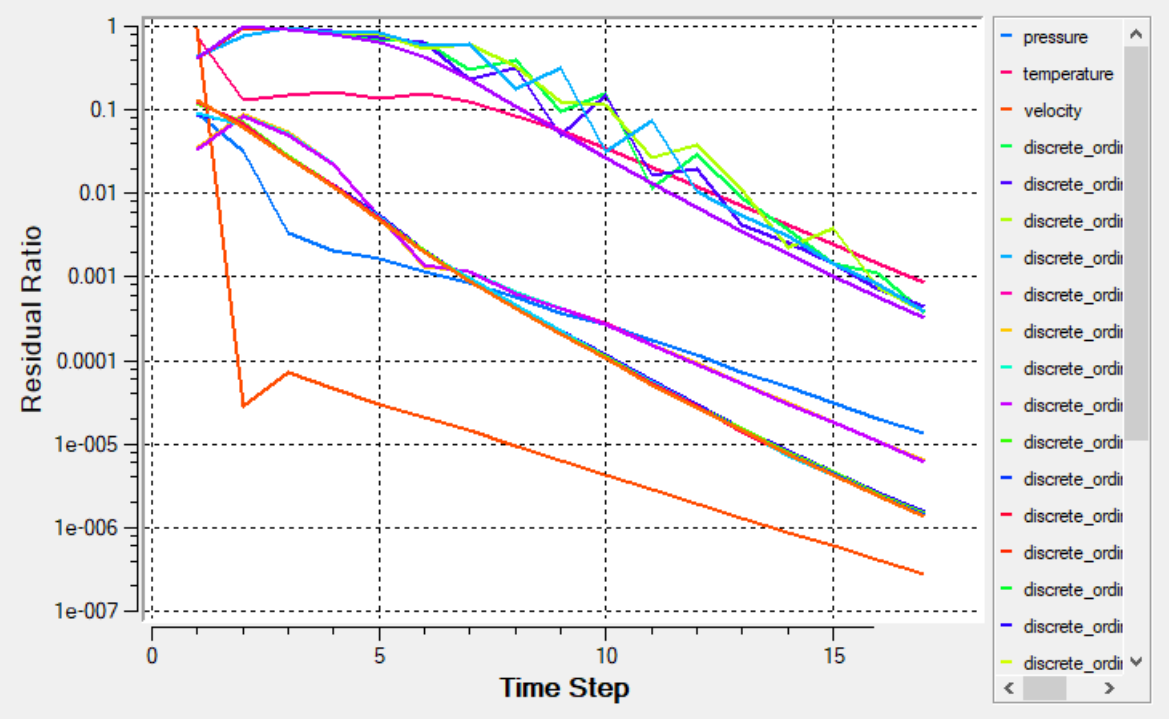


Figure 406:

Once the Solution has converged, close the AcuTail and AcuProbe windows.

Post-Process the Results with HyperView

In this step, you will visualize the results using HyperView. While doing so, you will create contour plots of temperature and incident radiation on a section cut. Once the solver run is complete, close the AcuProbe and AcuTail windows. In the HyperMesh Desktop window, close the AcuSolve Control tab and save the model.

Switch to the HyperView Interface and Load the AcuSolve Model and Results

1. In the HyperMesh Desktop window, click the **ClientSelector** drop-down in the bottom-left corner of the graphics window.

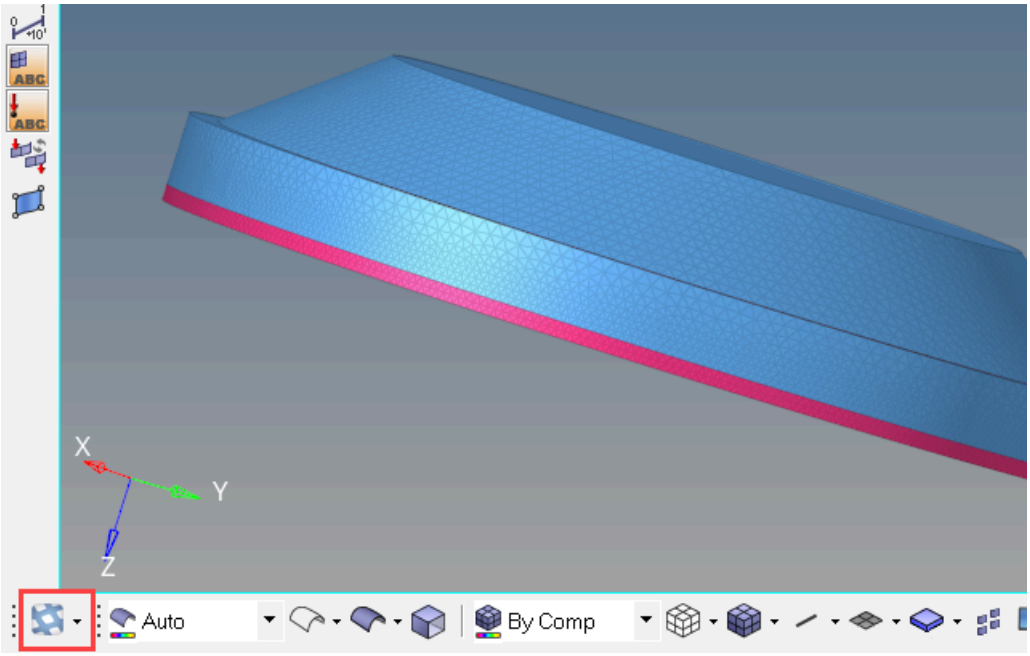


Figure 407:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.
   The interface is changed to HyperView.

   Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Headlamp_DO.1.Log`.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results.
   The model is colored by geometry after loading.

Create Temperature and Incident Radiation Contours on a Section Cut

1. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
2. On the 3DViewControls toolbar, right-click on multiple times until the orientation of the model is as shown in the figure below.



Figure 408:

3. Click the **Section cut** icon on the HV-Display toolbar.
4. In the panel area, click **Add** to create a new section cut named Section 1.
5. In the Define plane section, set the axis to **X Axis** then click **Apply**.
6. Change the Display options from Clipping plane to **Cross section**.

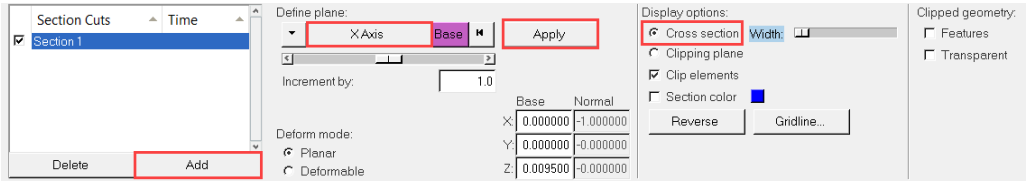


Figure 409:

7. Click **Gridline**. In the **Gridline Options** dialog, deactivate the **Show** check box under Grid line then click **OK**.
8. Click on the Results toolbar to open the Contour panel.
9. In the panel area, set the Result type to **Temperature (s)**.
10. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **All**.
11. Click **Apply**.

12. In the panel area, under the Display tab, turn off the Discrete color option.

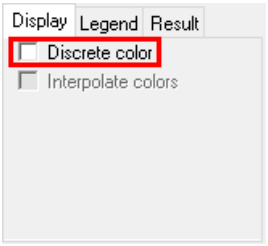


Figure 410:

13. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric precision to 2 then click **OK**.

The contour plot should look like the one shown in the figure below.

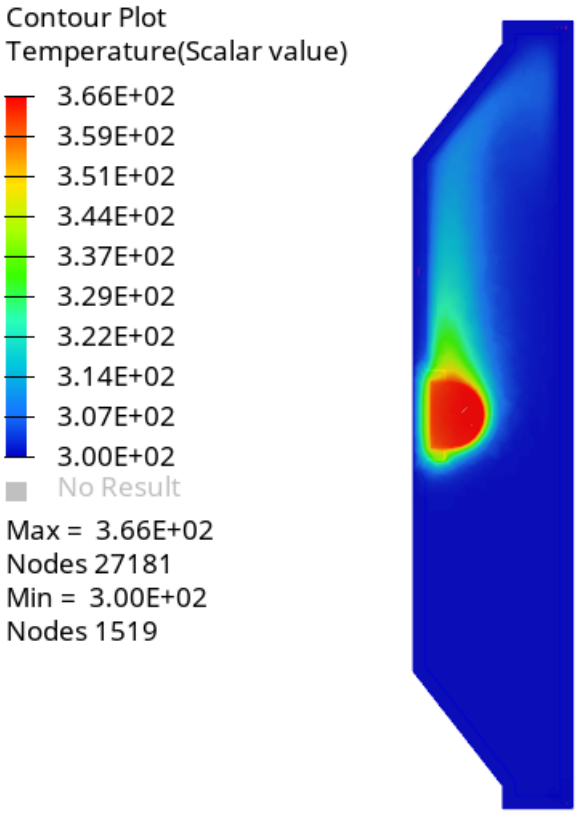


Figure 411:

14. In the panel area, change the Result type to **Incident_radiation** then click **Apply**.

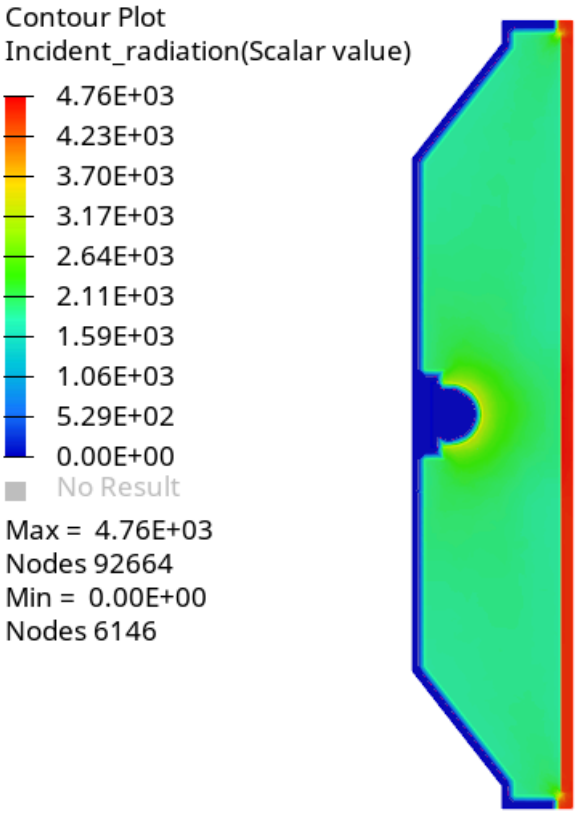


Figure 412:

As seen in the figure above, the rays travel from the bulb through the air and lens before getting radiated to the atmosphere.

Summary

In this tutorial, you learned how to set up and solve a radiation heat transfer problem in a headlamp using the discrete ordinate radiation model in AcuSolve. You started by importing the HyperMesh database with the mesh and basic model organization, and then set up the simulation parameters and boundary conditions. Once the solution was computed, you processed the results using HyperView, where you created contour plots of temperature and incident radiation.

2.3.10 ACU-T: 3300 Modeling of a Heat Exchanger Component

Prerequisites

This tutorial provides instructions for running a steady-state simulation of a flow inside a pipe with an interior heat exchanger placed at the middle of the pipe. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have

a basic understanding of HyperMesh and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3300_HeatExchanger.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in the figure below. It consists of a cylindrical pipe channel with an interior heat exchanger component volume with thickness 't' and radius 'r'. The heat exchanger component parameters are assigned to the HEX_Inlet surface component. Basically, the heat exchanger model is applied to a surface and the temperature rises across that surface to model the effect of the heat exchanger. Air enters the pipe at a velocity of 0.1 m/sec and flows through the heat exchanger volume and then exits through the outlet.

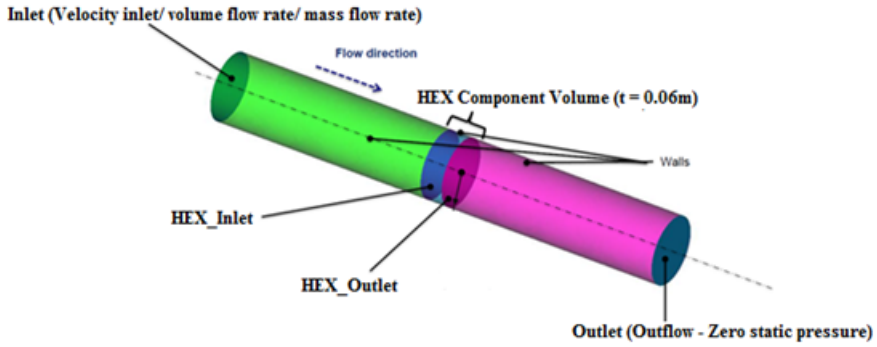


Figure 413:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.

2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3300_HeatExchanger.hm` and click **Open**.

4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.

5. Create a new directory named `HeatExchanger` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.

6. Enter `HeatExchanger` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

In this step, you will set the simulation parameters that apply globally to the simulation.

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, verify that the Analysis type is set to Steady State.
3. Set the Temperature equation to **Advective Diffusive**.
4. Change the Turbulence model to **Spalart Allmaras**.

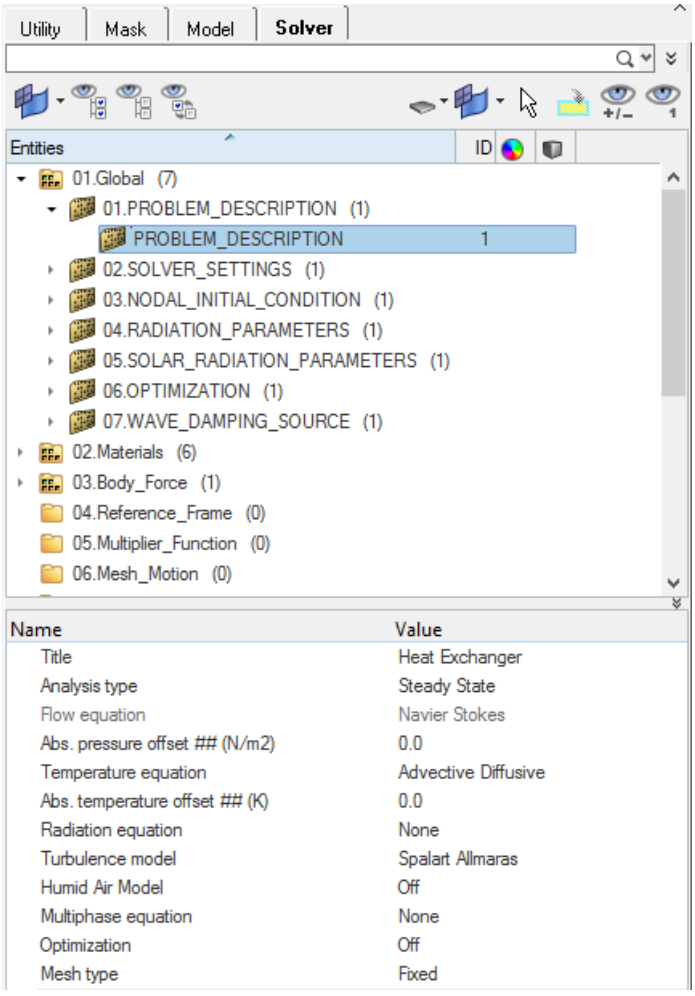


Figure 414:

Set Up Boundary Conditions and Assign Material Model Parameters

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Select **Air_HM** as the Material.

Name	Value
Name	Fluid
FE style	
Geometry style	
Type	FLUID
Material	(1) Air_HM
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 415:

3. Similarly, click **HEX** and change the Type to **FLUID** and select **Air_HM** as the Material in the Entity Editor.
4. Click **Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Change the Inflow velocity type to **Cartesian** and set the X velocity to `0.1` m/sec.
   c) Set the Temperature to `273` K.
   d) Change the Turbulence input type to **Viscosity Ratio**.
   e) For the Turbulent viscosity ratio, enter a value of `40`.

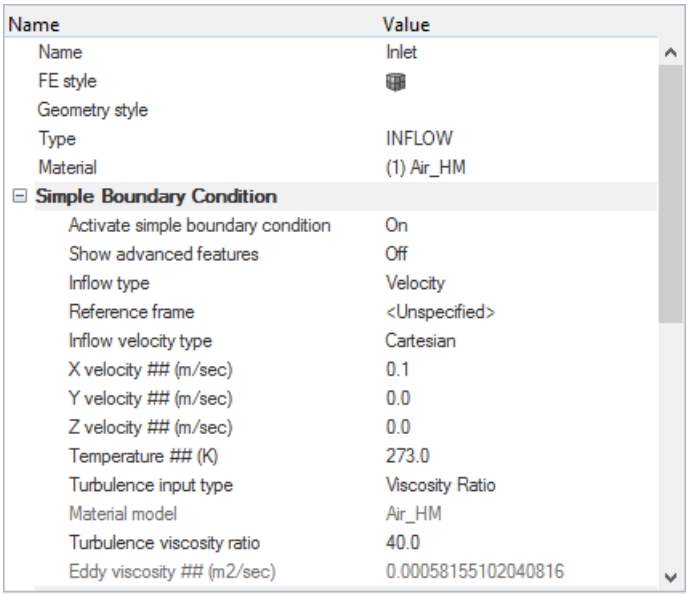


Figure 416:

5. Click **Outlet**. In the Entity Editor, change the Type to **OUTFLOW**.

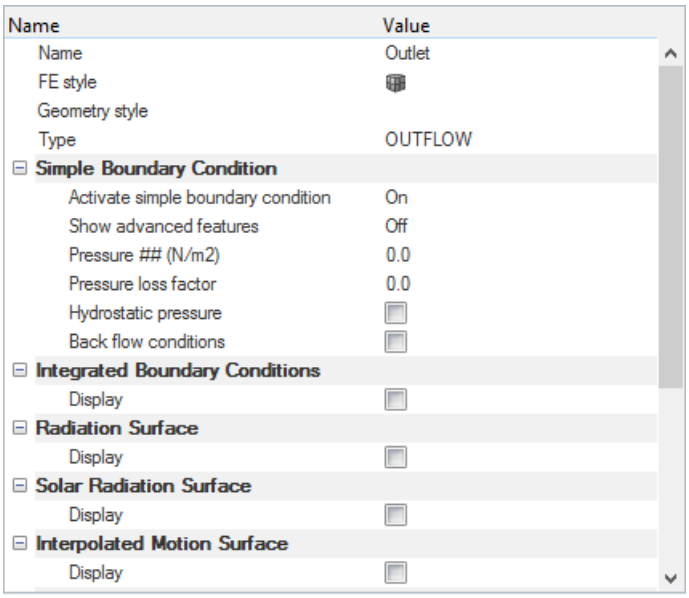


Figure 417:

6. Click **HEX_Inlet**. In the Entity Editor,

a) Change the Type to **HEAT_EXCHANGER_COMPONENT**.

b) Verify that the Heat exchanger type is set to **Constant Coolant Heat Reject**.

c) Set the Coolant Heat Reject to `200` W.

d) Set the Coolant flow rate to `0.0006309` m^3/sec.

e) Set the Heat exchanger thickness to `0.06` m.

f) Verify that the Upstream distance is set to `0`.

g) Change the Friction type to **Kays London**.

h) Change the Core Friction Constant to `20`.

i) Change the Core Friction Exponent to `-0.75`.

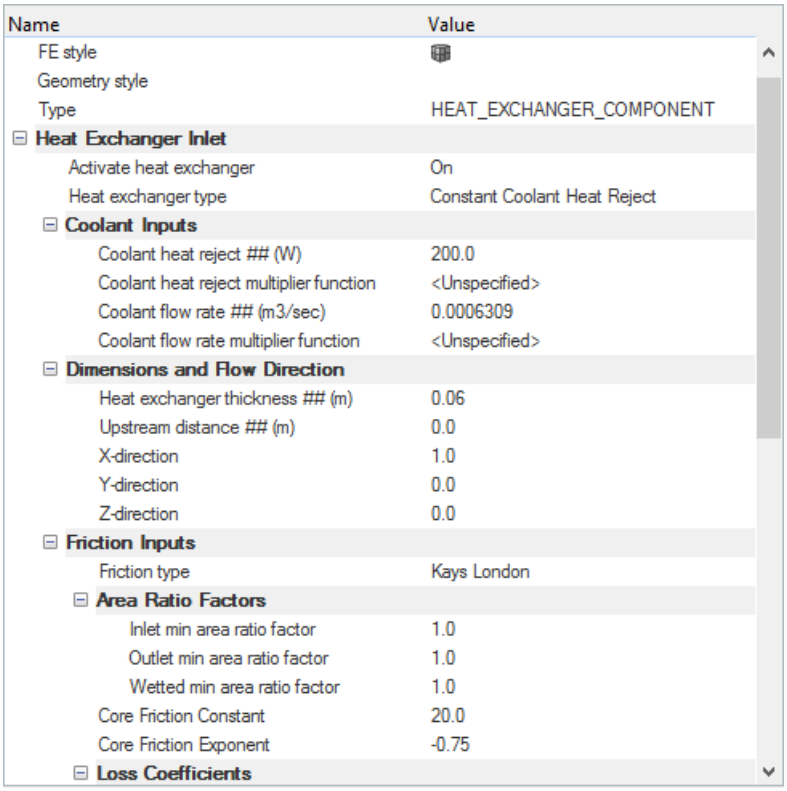


Figure 418:

7. Click **Walls**. In the Entity Editor, verify that the Type is set to **WALL**.

The surface mesh elements on the external wall surfaces and interfaces can be grouped into one single collector. Auto_Wall, which is an advanced feature in AcuSolve, re-groups them into surface sets based on the element set they belong to and whether they are internal or external surfaces. This process is done internally without the user having to do it manually.

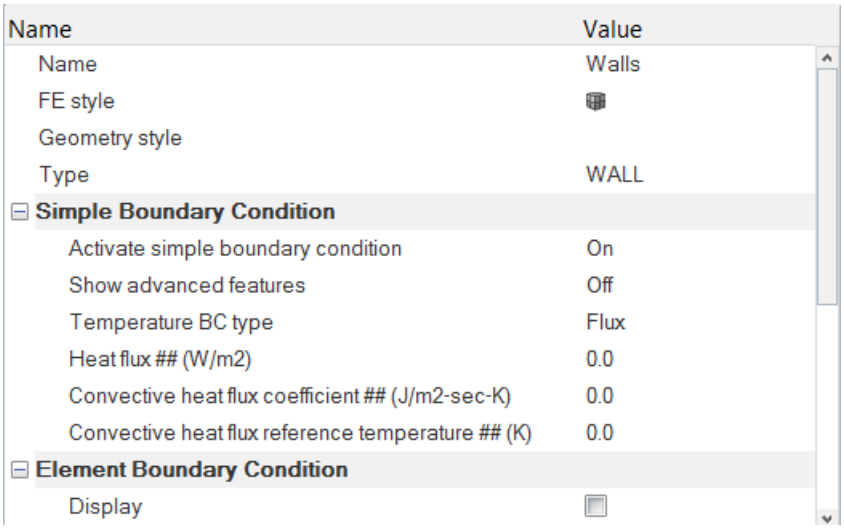


Figure 419:

8. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

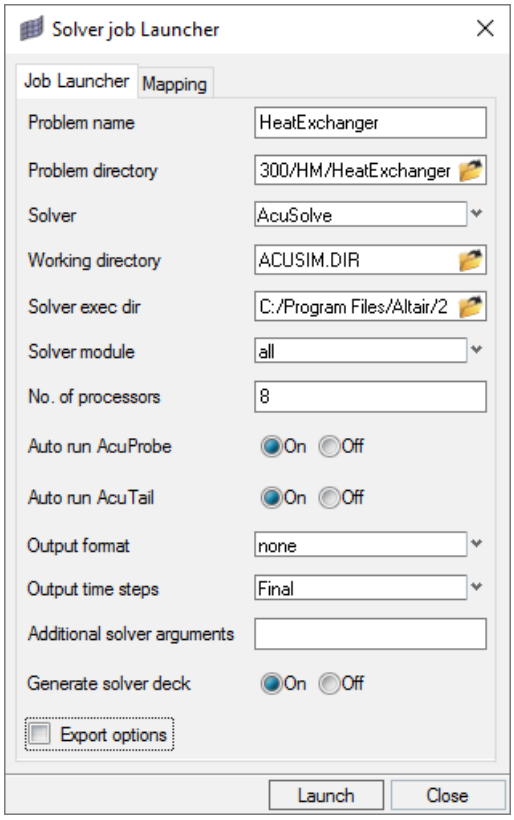


Figure 420:

Post-Process with AcuProbe

As the solution progresses, the AcuTail and AcuProbe windows are launched automatically. The surface output and residual ratios can be monitored using AcuProbe.

1. In the AcuProbe window, under the Data Tree, expand **Residual Ratio**, right-click on **Final**, and select **Plot All**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

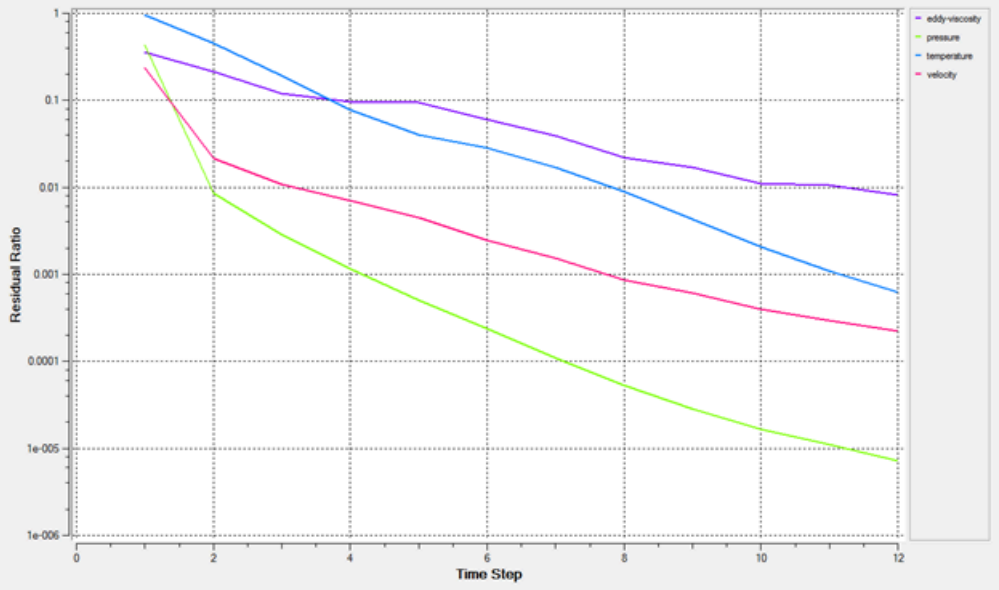


Figure 421:

2. Once the solution is converged, right-click on **Final** under Residual Ratio and select **Plot None**.
3. Click on the toolbar.

   A **User Function** dialog opens.
4. Enter `dT` as the Name.
5. Type `In =` in the Function field.
6. Expand **Heat Exchanger** > **HEX_Inlet**. Right-click on **air_temperature** and select **Copy Name**. Paste the value in the function after In =.
7. Type `Out =` on a new line in the Function field.
8. Expand **Heat Exchanger** > **HEX_Inlet**. Right-click on **coolant_temperature** and select **Copy Name**. Paste the value in the function after Out =.
9. On the next line, type `value = Out - In`.

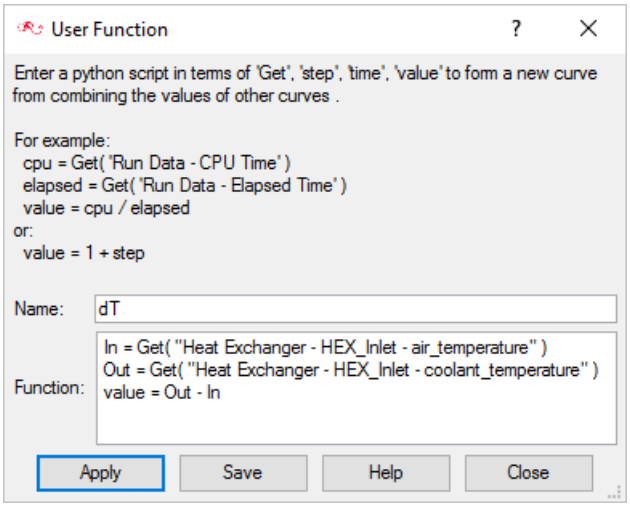


Figure 422:

> **Note:** The word "value" is case sensitive and should always be in lowercase characters. If it starts with a capital letter, it will give you an error window.

10. Click **Apply**.

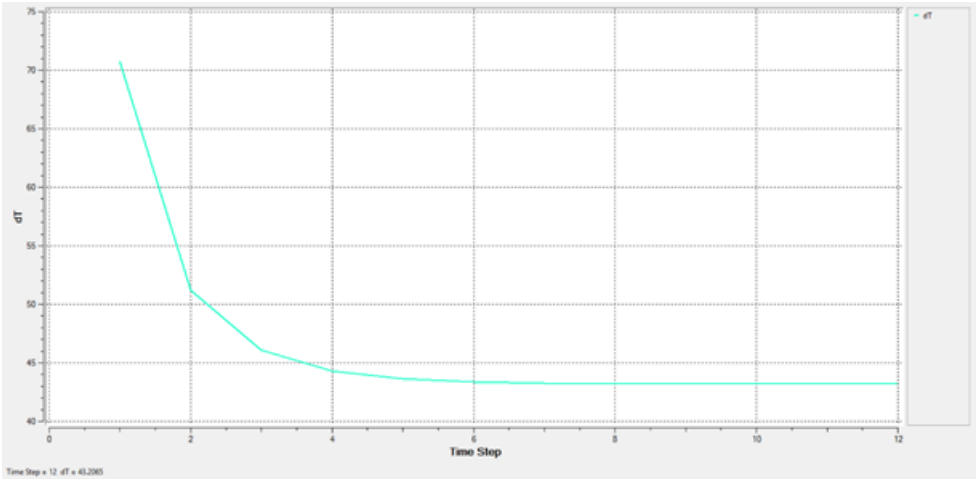


Figure 423:

You can zoom into the plot by clicking and then defining an area at the end of the curve. As shown in the figure below, for the given problem, the temperature rise is 43.21 K.

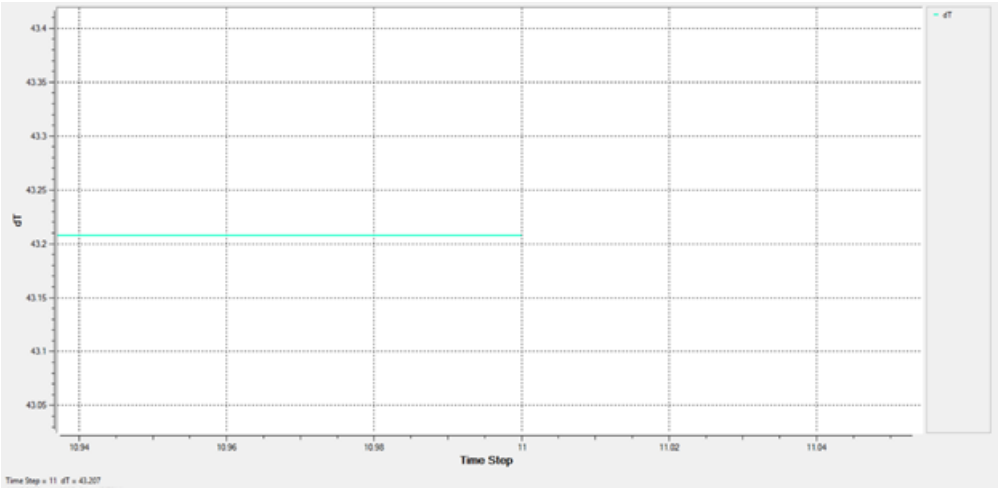


Figure 424:

Summary

In this tutorial, you successfully learned how to set up and solve a simulation involving a Heat Exchanger component. You imported the meshed geometry and then assigned the boundary conditions and material properties for all the regions. Once the solution was computed, you defined a user function in AcuProbe in order to create a plot of the temperature rise across the heat exchanger volume.

2.3.11 ACU-T: 3310 Single Phase Nucleate Boiling

Prerequisites

This tutorial introduces you to setting up and solving a single phase nucleate boiling problem using HyperMesh. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3310_NB1.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 425. It is based on the popular wall heat transfer model for sub-cooled boiling (Steiner Model). It consists of a channel with a

heated wall at the bottom. The temperature of the wall is selected to onset the nucleate boiling at the heated wall.

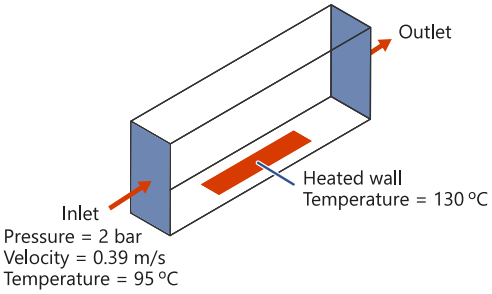


Figure 425: Schematic of Channel

The dimensions of the inlet are 0.03 x 0.04 m; the inlet velocity (v) is 0.39 m/s and the temperature (T) of the fluid entering the inlets is 368.15 K (95 C).

The preheated air enters the inlets and heat is transferred to the fluid from the walls. The heat causes sub-cooled boiling to occur in the region close to the wall and leads to formation of bubbles at nucleation sites.

The heat transfer in this regime is basically dominated by two effects, the macro convection due to the motion of the bulk liquid and the latent heat transport associated with the evaporation of the liquid micro-layer between the bubble and the heated wall.

The fluid in this problem is water, which has temperature dependent material properties: density, viscosity, enthalpy and conductivity. There are also surface tension and vapor phase models specified for this material.

Water vapor which also has temperature dependent material properties is specified as the vapor phase model.

The AcuSolve simulation will be set up to model steady state heat transfer to determine the temperature and heat flux on the heated walls of the manifold.

Open the HyperMesh Model Database

1. Start HyperMesh Desktop and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.

2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.

3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3310_NB1.hm` and click **Open**.

4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `NB1` and navigate into this directory.
   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `NB1_Steiner` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the Simulation Parameters

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, verify that the Analysis type is set to **Steady State**.
3. Set the Abs. pressure offset to `200000` N/m^2.
4. Set the Temperature equation to **Advective Diffusive**.
5. Change the Turbulence model to **Spalart Allmaras**.
6. Set Nucleate boiling single phase to **On**.

Name	Value
Title:	
Analysis type:	Steady State
Flow equation:	Navier Stokes
Abs. pressure offs...	200000.0
Temperature equ...	Advective Diffusive
Abs. temperature ...	0.0
Radiation equation:	None
Turbulence model:	Spalart Allmaras
Humid Air Model:	Off
Multiphase equati...	None
Optimization:	Off
Mesh type:	Fixed
Nucleate boiling s...	On
External code:	Off
Global Gravity	
X ## (m/sec2):	0.0
Y ## (m/sec2):	0.0
Z ## (m/sec2):	0.0

Figure 426:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, set the Relaxation factor to `0.4`.
3. Verify that Flow, Temperature, and Turbulence are set to **On**.

Set the Nodal Initial Conditions

1. In the Solver Browser, click **03.NODAL_INITIAL_CONDITION** under **01.Global**.
2. Set a constant X-Velocity value of `0.39`.
3. Set a constant Temperature value of `368.15`.
4. Set a constant Eddy viscosity value of `0.0001`.

Name	Value
Pressure	
Type	Constant
Default value ## (N/m2)	0.0
Satisfy BC	Off
Velocity	
Type	Constant
X ## (m/sec)	0.39
Y ## (m/sec)	0.0
Z ## (m/sec)	0.0
Satisfy BC	Off
Temperature	
Type	Constant
Default value ## (K)	368.15
Satisfy BC	Off
Eddy viscosity	
Type	Constant
Default value ## (m2/sec)	0.0001
Satisfy BC	Off
Unsupported Entries	

Figure 427:

Assign Material Properties and Boundary Conditions

Create Curves/Plots for Material Properties

1. In the Model Browser, right-click and select **Create** > **Curve**.
2. In the **Curve Editor**, click **New**.
3. In the panel area, enter `Density_Water` for the name then click **proceed**.
4. Enter the X and Y values shown below.

X	Y
353.15	971.83
358.15	968.66
363.15	965.35
368.15	961.93
373.15	958.4
378.15	954.74
383.15	950.98
388.15	947.1
393.15	943.11

Figure 428:

5. Create another curve, name it `Viscosity_Water`, and enter the values shown below.

X	Y
353.15	0.00035437
358.15	0.00033337
363.15	0.00031443
368.15	0.00029731
373.15	0.00028177
378.15	0.00026762
383.15	0.00025471
388.15	0.0002429
393.15	0.00023162

Figure 429:

6. Create a curve named `Enthalpy_Water` and enter the following values

X	Y
353.15	1481993.975
358.15	1504409.075
363.15	1527045.75
368.15	1549874.685
373.15	1572976.51
378.15	1596360.225
383.15	1620034.83
388.15	1644009.325
393.15	1668459.33

Figure 430:

7. Create a curve named `Conductivity_Water` and enter the following values.

X	Y
353.15	0.67007
358.15	0.67288
363.15	0.67532
368.15	0.6774
373.15	0.67915
378.15	0.68058
383.15	0.68172
388.15	0.68259

Figure 431:

8. Create a curve named `Density_Vapor` and enter the following values.

X	Y
393.63	1.1291
398.15	1.1138
403.15	1.0985
408.15	1.0836
413.15	1.0692
418.15	1.0553
423.15	1.0418
428.15	1.0287
433.15	1.016

Figure 432:

9. Create a curve named `Viscosity_Vapor` and enter the following values.

X	Y
393.63	1.2963e-5
398.15	1.315e-5
403.15	1.3345e-5
408.15	1.3541e-5
413.15	1.3737e-5
418.15	1.3935e-5
423.15	1.4133e-5
428.15	1.4331e-5
433.15	1.4561e-5

Figure 433:

10. Create a curve named `Enthalpy_Vapor` and enter the following values.

X	Y
393.63	857404.866
398.15	855305.83
403.15	856411.545
408.15	859359.825
413.15	863442.185
418.15	868413.92
423.15	874058.64
428.15	880276.4
433.15	887004.57

Figure 434:

11. Create a curve named `Conductivity_Vapor` and enter the following values.

X	Y
393.63	0.027493
398.15	0.027795
403.15	0.028123
408.15	0.028463
413.15	0.028813
418.15	0.029173
423.15	0.029544
428.15	0.029923
433.15	0.030311

Figure 435:

12. Close the **Curve Editor**.

13. In the Model Browser, right-click and select **Create** > **Plot**.

14. In the Entity Editor, set the name to `Water-srfTns` and set the type to **Surface_Tension_Model**.

15. Set the value of the surface tension to `0.01`.

Name	Value
Solver Keyword	SURFACE_TENSION_MODEL
Name	Water-srfTns
Type	Surface_Tension_Model
Surface Tension Model Type	Constant
Surface tension	0.01
Surface tension multiplier function	<Unspecified>
Curve fit variable	Temperature
Unsupported Entries	
Number of Unsupported Entries	0

Figure 436:

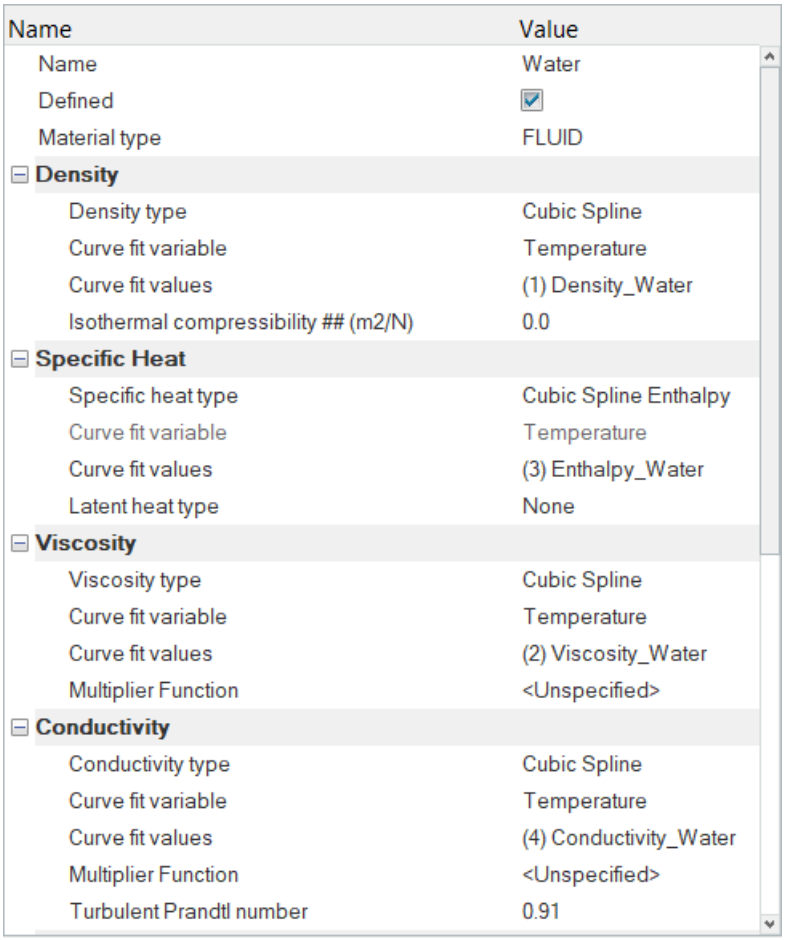
Define Material Properties

1. In the Solver Browser, right-click on the **02.Materials** folder and select **Material(Fluid)** to create a new fluid material. Name it `Vapor`.
2. In the Entity Editor, change the Density type to **Cubic Spline**, set the Curve fit variable to **Temperature**, and select the **Density_Vapor** curve from the list for Curve fit values.
3. Change the Specific Heat type to **Cubic Spline Enthalpy**, set the Curve fit variable to **Temperature**, and select the **Enthalpy_Vapor** curve from the list for Curve fit values.
4. Set the Latent heat type to **Constant**, the Latent heat to `2256000` J/kg, and the Latent heat temperature to `393.45` K.
5. Similarly for Viscosity and Conductivity, set the cubic spline values to **Viscosity_Vapor** and **Conductivity_Vapor**, respectively.

Name	Value
Name	Vapor
Defined	☑
Material type	FLUID
Density	
Density type	Cubic Spline
Curve fit variable	Temperature
Curve fit values	(5) Density_Vapor
Isothermal compressibility ## (m2/N)	0.0
Specific Heat	
Specific heat type	Cubic Spline Enthalpy
Curve fit variable	Temperature
Curve fit values	(7) Enthalpy_Vapor
Latent heat type	Constant
Latent heat ## (J/kg)	2256000.0
Latent heat temperature ## (K)	393.45
Temperature interval ## (K)	0.0
Viscosity	
Viscosity type	Cubic Spline
Curve fit variable	Temperature
Curve fit values	(6) Viscosity_Vapor
Multiplier Function	<Unspecified>
Conductivity	
Conductivity type	Cubic Spline
Curve fit variable	Temperature
Curve fit values	(8) Conductivity_Vapor
Multiplier Function	<Unspecified>
Turbulent Prandtl number	0.91

Figure 437:

6. In the Solver Browser, right-click on **FLUID** and select **Create** to create a new fluid material. Name it `Water`.
7. In the Entity Editor, change the Density type to **Cubic Spline**, set the Curve fit variable to **Temperature**, and select the **Density_Water** curve from the list for Curve fit values.
8. Similarly for Specific Heat, Viscosity, and Conductivity, set the cubic spline values to **Enthalpy_Water Viscosity_Water** and **Conductivity_Water**, respectively.



Name	Value
Name	Water
Defined	☑
Material type	FLUID
Density	
Density type	Cubic Spline
Curve fit variable	Temperature
Curve fit values	(1) Density_Water
Isothermal compressibility ## (m2/N)	0.0
Specific Heat	
Specific heat type	Cubic Spline Enthalpy
Curve fit variable	Temperature
Curve fit values	(3) Enthalpy_Water
Latent heat type	None
Viscosity	
Viscosity type	Cubic Spline
Curve fit variable	Temperature
Curve fit values	(2) Viscosity_Water
Multiplier Function	<Unspecified>
Conductivity	
Conductivity type	Cubic Spline
Curve fit variable	Temperature
Curve fit values	(4) Conductivity_Water
Multiplier Function	<Unspecified>
Turbulent Prandtl number	0.91

Figure 438:

9. Under the vapor Phase heading, set the vapor phase model to **Vapor** by selecting it from the Material list.
10. Under the Surface Tension heading, set the Surface tension model to **Water-srfTns** by selecting it from the Plot list.



vapor Phase	
vapor phase model	(1) Vapor
Surface Tension	
Surface tension model	(1) Water-srfTns

Figure 439:

Assign Material Properties and Boundary Conditions

1. In the Solver Browser, expand **11.Volumes** > **FLUID**.
2. Click **FluidVolume**. In the Entity Editor,
   a) Verify that the Type is set to **FLUID**.

b) Select **Water** as the Material.

Name	Value
Name	FluidVolume
Type	FLUID
Material	(2) Water
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	

Figure 440:

3. In the Solver Browser, expand **12.Surfaces** > **WALL**.
4. Click **Inflow**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow type to **Average Velocity**.
   c) Set the Average velocity to `0.39` m/sec.
   d) Set the Temperature to `368.15` K.

Name	Value
Name	Inflow
Type	INFLOW
FE style	
Geometry style	
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Inflow type	Average velocity
Reference frame	<Unspecified>
Average velocity ## (m/sec)	0.39
Temperature ## (K)	368.15
Multiphase incoming field multiplier function	<Unspecified>
Integrated Boundary Conditions	

Figure 441:

5. Click **Outflow**. In the Entity Editor,
   a) Change the Type to **OUTFLOW**.
   b) Verify that the Pressure value is set to `0` N/m^2.
   c) Turn on the **Backflow conditions** and set the Temperature backflow type and Eddy viscosity backflow type to **Mass Flux Average**.

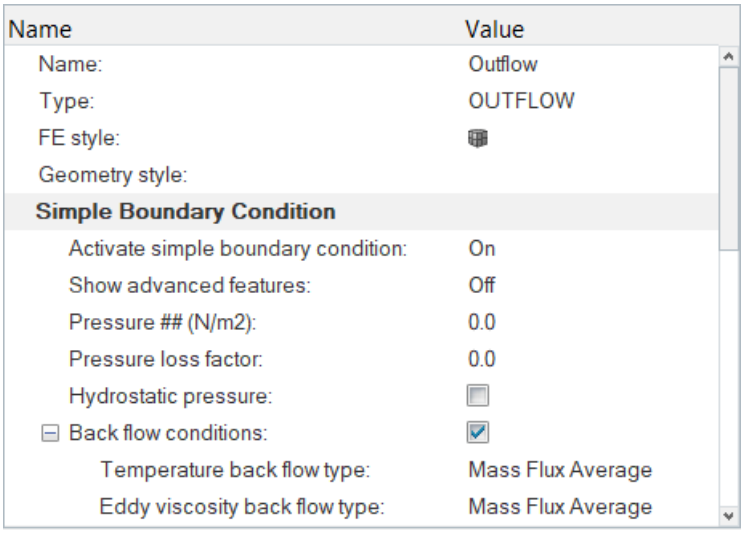


Figure 442:

6. Click **HeatedWall**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Set the Temperature BC Type to **Value**.
   c) Set the Temperature to `403.15` K.

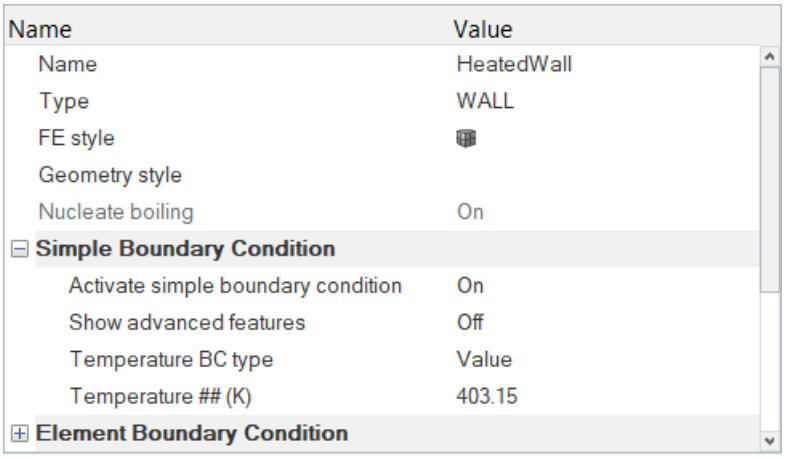


Figure 443:

7. Click **Side_MaxY**. In the Entity Editor, verify that the Type is set to **WALL**.
8. Repeat the above step for the **Side_Miny**, **Top**, and **Bottom** components.

Compute the Solution

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

The **Solver job Launcher** dialog opens.

3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Leave the remaining options as default and click **Launch** to start the solution process.

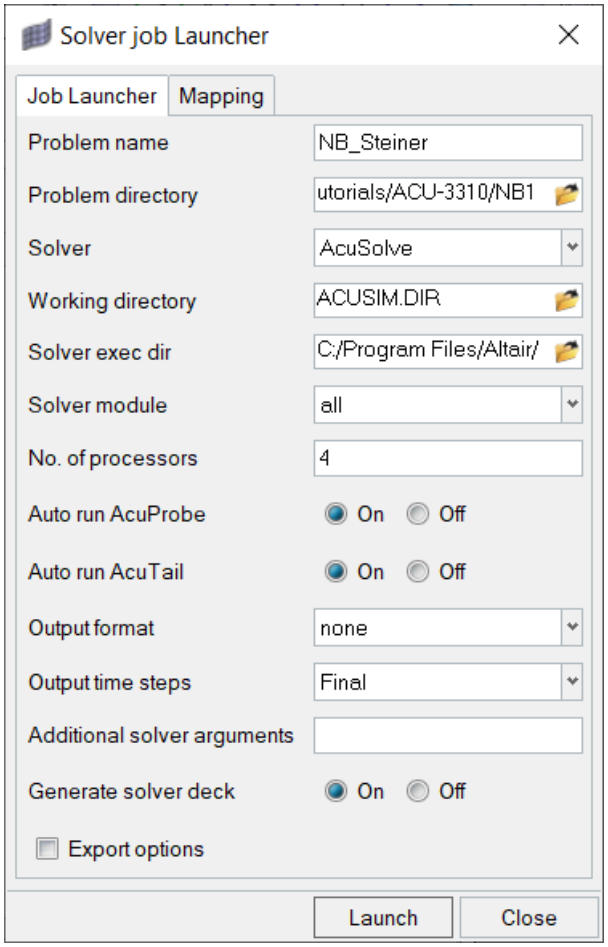


Figure 444:

Once you hit the Launch button, the AcuTail and AcuProbe windows are launched automatically. A summary of the run in the AcuTail window indicates that the solver run is complete.

```
acuSolve: ---------- End Time Step ----------
acuSolve: Input       CPU/Elapse time= 6.265000e+000 1.120300e+001 Sec
acuSolve: Output      CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Exchange    CPU/Elapse time= 3.901000e+000 4.264000e+000 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =         23         23
acuSolve:   Elem.Form CPU/Elapse time= 2.750650e+002 1.412580e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 9.495130e+002 4.933680e+002 Sec
acuSolve: Turbulence stagger "turbulence":
acuSolve:   No of res/lhs formations =         23         23
acuSolve:   Elem.Form CPU/Elapse time= 1.744510e+002 8.996100e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 7.601600e+001 3.968000e+001 Sec
acuSolve: Temperature stagger "temperature":
acuSolve:   No of res/lhs formations =         23         23
acuSolve:   Elem.Form CPU/Elapse time= 1.754230e+002 8.947400e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 4.482980e+002 2.354360e+002 Sec
acuSolve: Pressure proj. elapse  time= 1.333340e+002 Sec
acuSolve: Linear solver  elapse  time= 5.795630e+002 Sec
acuSolve: Total CPU/Elapse time      = 2.538406e+003 1.330290e+003 Sec
acuSolve: Total Memory Usage         = 1.004754e+003 Mbytes
acuRun: -*-*-*-
```

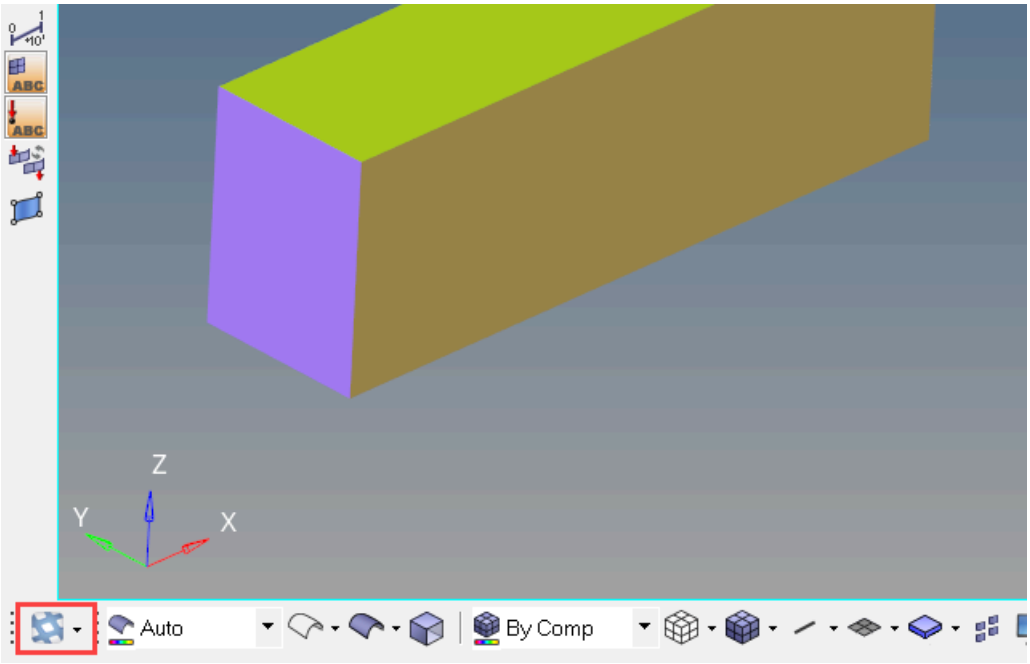


Figure 446:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.
   The interface is changed to HyperView.

   Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `NB1_Steiner.1.Log`.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create Contours for Temperature Distribution

In this step, you will display temperature contours on the HeatedWall and the Bottom surfaces.

1. In the Results Browser, expand the list of **Components**.
2. Click the **Isolate Shown** icon, hold Ctrl, then select the **HeatedWall** and **Bottom** components to turn off the display of all components except those that are required.

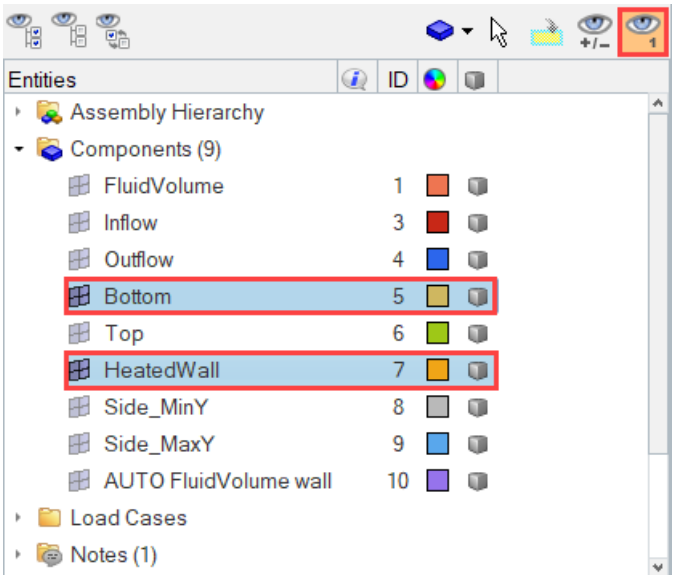


Figure 447:

3. Click on the Results toolbar to open the Contour panel.
4. In the panel area, change the Result type to **Temperature (s)**.
5. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
6. Click **Apply** to plot the temperature contours.
7. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 448:

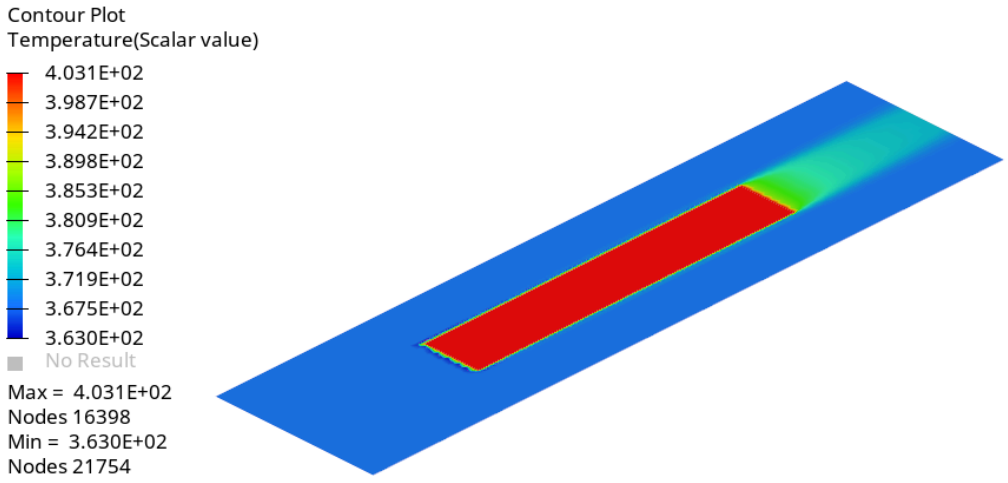


Figure 449:

Summary

In this tutorial, you learned how to set up a single phase nucleate boiling problem using HyperMesh and solve it using AcuSolve. Once you computed the solution, you post-processed the results using HyperView and created temperature contour plots.

2.3.12 ACU-T: 3311 Multiphase Nucleate Boiling Using the Algebraic Eulerian Model

Prerequisites

This tutorial provides instructions for running a transient simulation of a two-phase Nucleate Boiling in a pipe using the Algebraic Eulerian model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T3311_Steiner.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 450. As an example, the Steiner problem is attached here to show the capability of the Multiphase Nucleate Boiling modeling in

AcuSolve. The Algebraic Eulerian (AE) model with phase change is used to simulate the heat transfer and momentum exchange between a carrier field and a dispersed field.

In this problem, water is considered a carrier field material and vapor is considered a dispersed field material. Water at 2 bar pressure and 95℃ temperature enters the inlet at an average velocity of 0.39 m/sec and passes through the heated wall which is maintained at 130 ℃.

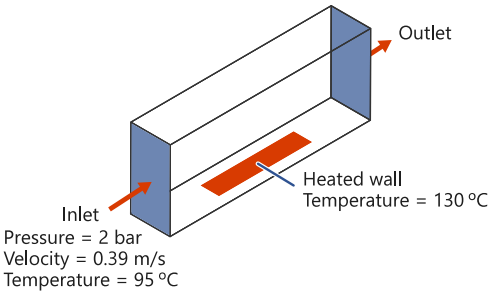


Figure 450: Schematic of Channel

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T3311_Steiner.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Multiphase_NB` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Steiner` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, set the Analysis type to **Transient**.
3. Set Abs. pressure offset to `200000`.
4. Set the Turbulence model to **Spalart Allmaras**.
5. Set the Temperature equation to **Advective Diffusive**.
6. Set Multiphase equation to **Algebraic Eulerian**.
7. Set the Global Gravity in the Z direction to `-9.8`.

Name	Value
Title	Steiner Model
Analysis type	Transient
Flow equation	Navier Stokes
Abs. pressure offset ## (N/m2)	200000.0
Temperature equation	Advective Diffusive
Abs. temperature offset ## (K)	0.0
Radiation equation	None
Turbulence model	Spalart Allmaras
Humid Air Model	Off
Multiphase equation	Algebraic Eulerian
Optimization	Off
Mesh type	Fixed
Nucleate boiling single phase	Off
External code	Off
Global Gravity	
X ## (m/sec2)	0.0
Y ## (m/sec2)	0.0
Z ## (m/sec2)	-9.8
Unsupported Entries	

Figure 451:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, set the Max time steps to `40`.
3. Set the Initial time increment to `0.01`.
4. Set the Min stagger iterations to `2`.
5. Set the Max stagger iterations to `4`.
6. Set the Relaxation factor to `0`.
7. Verify that the Flow, Turbulence, and Field settings are turned **On**.

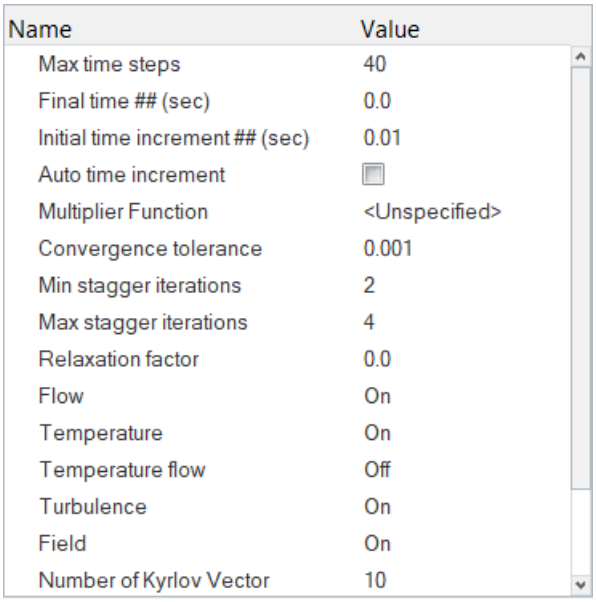


Figure 452:

Define the Nodal Outputs

1. In the Solver Browser, expand **17.Ouput** then click **NODAL_OUTPUT**.
2. In the Entity Editor, set the Time step frequency to `10`.
3. Toggle on the **Output initial condition** field.

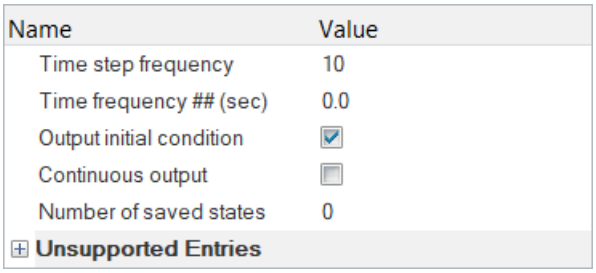


Figure 453:

Set Up Material Model Parameters

In this step, you will start by creating a new material named Vapor. Then, you will set up the Multiphase material model.

1. In the Solver Browser, expand **02.Materials**.
2. Right-click on **FLUID** and select **Create** to create a new fluid material. Name it `Vapor`.
3. In the Entity Editor, set the Density value to `1.128`.
4. Set the Specific heat value to `2174.59`.

5. Change the Latent heat type to **Constant**.
6. Set the Latent heat value to `2256000`.
7. Set the Latent heat temperature value to `393.45`.
8. Set the Viscosity value to `1.29e-05`.
9. Set the Conductivity value to `0.02749`.

Name	Value
Name	Vapor
Defined	☑
Material type	FLUID
Density	
Density type	Constant
Density ## (kg/m3)	1.128
Isothermal compressibility ## (m2/N)	0.0
Specific Heat	
Specific heat type	Constant
Specific heat ## (J/kg-K)	2174.59
Latent heat type	Constant
Latent heat ## (J/kg)	2256000.0
Latent heat temperature ## (K)	393.45
Temperature interval ## (K)	0.0
Viscosity	
Viscosity type	Constant
Viscosity ## (kg/m-sec)	1.29e-05
Multiplier Function	<Unspecified>
Conductivity	
Conductivity type	Constant
Conductivity ## (W/m-K)	0.02749
Multiplier Function	<Unspecified>
Turbulent Prandtl number	0.91

Figure 454:

10. In the Solver Browser, under FLUID, click **Water_HM** and change the material properties.
    a) Change the Density to `956.36`.
    b) Change the Specific heat to `4204.8`.
    c) Change the Viscosity to `0.000314`.
    d) Change the Conductivity to `0.675`.
11. In the Solver Browser, under 02.Materials, right-click on **MULTIPHASE** and select **Create** to create a new multiphase material. Name it `VapourWater-Eulerian_HM`.
12. In the Entity Editor, verify that the Field Interaction Type is set to **Algebraic Eulerian**.
13. Set **Water_HM** as the Carrier Material if it's not already set.
    Since Water is the carrier field for this analysis, leave this option as is.
14. Verify that the Number of Dispersed Field is set to `1`.

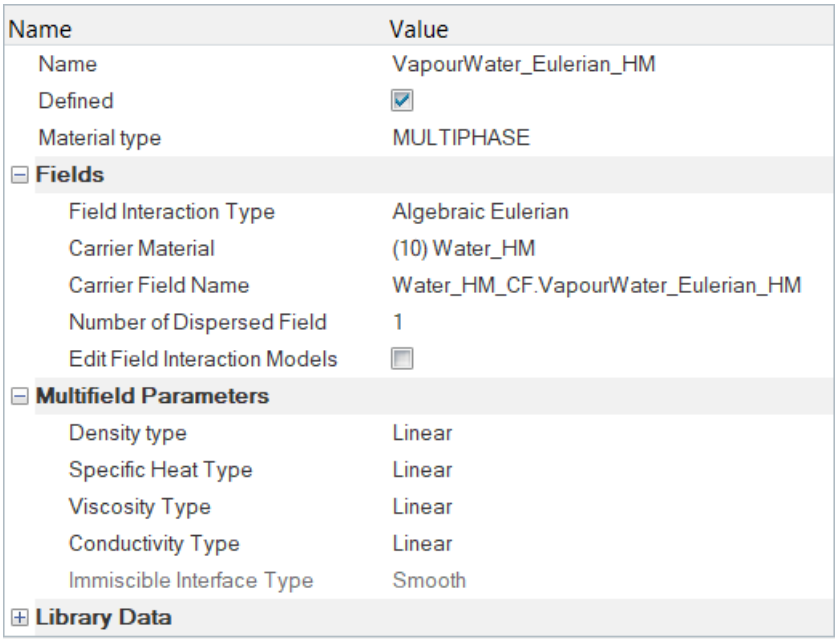


Figure 455:

15. Activate the check box beside **Edit Field Interaction Models**.
    The **Field Interaction Model** dialog opens.
16. In the dialog, set the Dispersed Field 1 Material to **Vapor**.
17. Activate the **Enable Phase change** check box.
18. Set Phase change type to **Wall boiling** if not set.
19. Set Dispersed field diameter type to **Boiling Kurul exponential**.

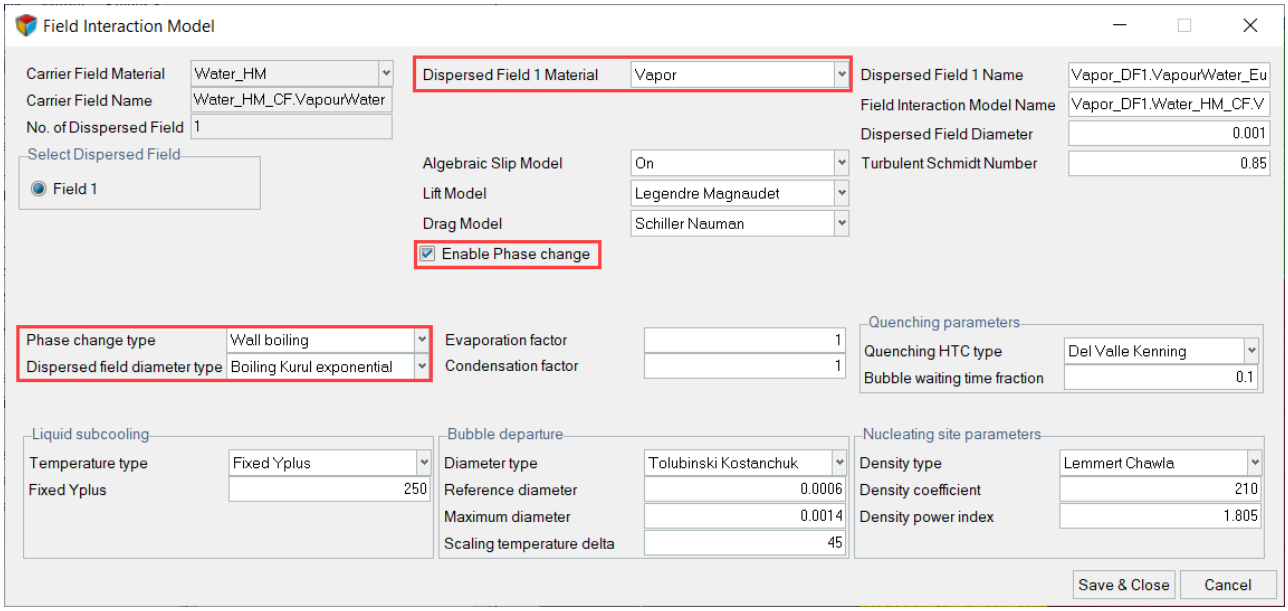


Figure 456:

20. Click **Save & Close**.

Set Up Boundary Conditions and Nodal Initial Conditions

Set Up Boundary Conditions and Nodal Initial Conditions

In this step, you will assign the material properties to the multiphase fluid volume and then assign surface boundary conditions.

1. In the Solver Browser, expand **11.Volumes** > **MULTIPHASE**.
2. Click **FluidVolume**. In the Entity Editor,
   a) Change the Type to **MULITPHASE** if not set.
   b) Set **VaperWater_Eulerian_HM** as the Material.

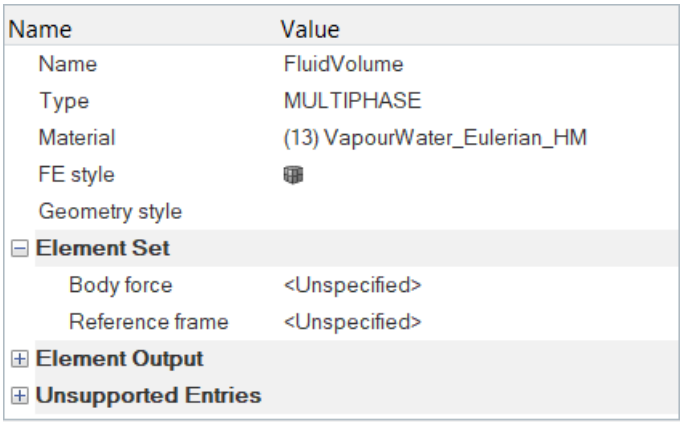


Figure 457:

3. In the Solver Browser, expand **12.Surfaces** > **INFLOW**.
4. Click **Inflow**. In the Entity Editor,
   a) Change the Type to **INFLOW** if not set.
   b) Set the Inflow type to **Average velocity**.
   c) Set the Average velocity to `0.39` m/sec.
   d) Set the Temperature to `368.15`.

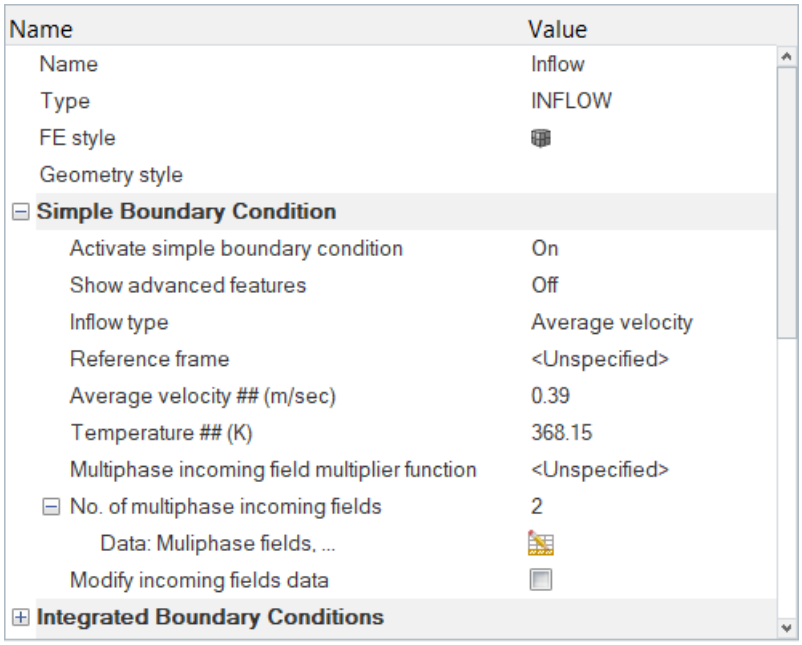


Figure 458:

e) Set the No. of multiphase incoming fields to `2` and press **Enter**.

f) In the dialog, select **Water_CF.VaporWater_Eulerian_HM** as the first Incoming Field and set its Volume Fraction to `1`.

g) Similarly, select **Vapor_DF1. VaporWater_Eulerian_HM** as the second Incoming Field and set its Volume Fraction to `0`.

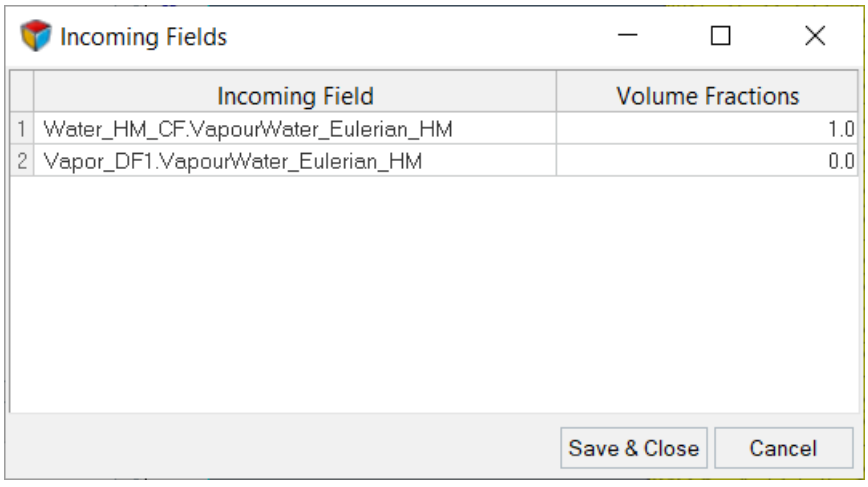


Figure 459:

h) Click **Save and Close**.

5. Similarly, expand **OUTFLOW** then click the **Outflow** component. In the Entity Editor, change the Type to **OUTFLOW** if not set.

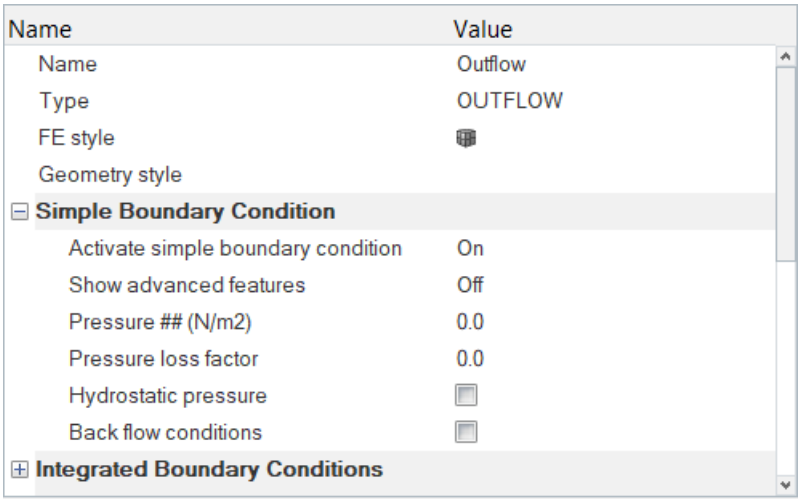


Figure 460:

6. Similarly, expand **WALL** then click the **Side_MaxY** component. In the Entity Editor, verify that the Type is set to **WALL**.

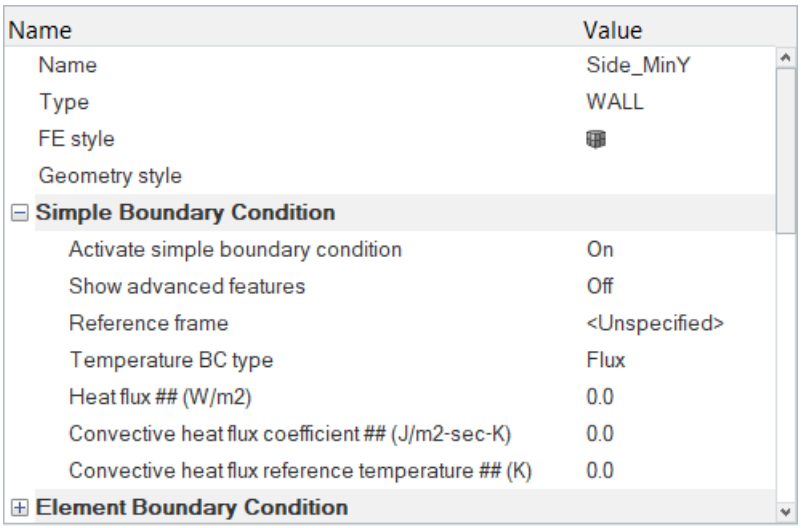


Figure 461:

7. Similarly, click the **Side_MaxY**, **Top**, and **Bottom** components and verify that the Type is set to **WALL**.
8. Click **HeatedWall**. In the Entity Editor,
   a) Verify that the Type is set to **WALL**.
   b) Set Temperature BC type to **Value**.
   c) Set the Temperature value to `403.15`.

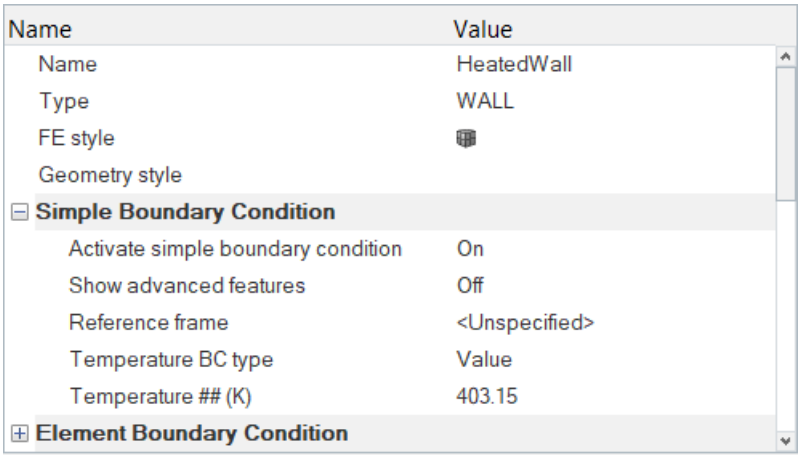


Figure 462:

9. Save the model.

Set the Nodal Initial Conditions

1. In the Solver Browser, expand **01.Global** then click **03.NODAL_INTIAL_CONDITION**.
2. In the Entity Editor, set the x-Velocity value to `0.39`.
3. Set the default value of Temperature to `368.15`.
4. In the Field tab, set the No. of Fields to `2` then press **Enter**.
5. In the **Fields** dialog, set the values shown in the figure below.

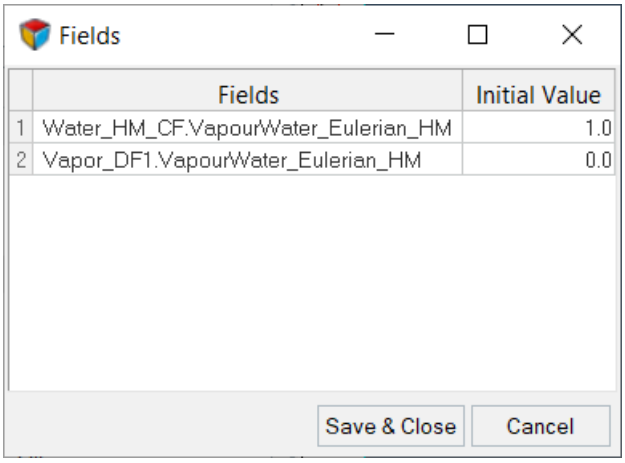


Figure 463:

6. Click **Save & Close**.
7. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Verify that the Output time steps is set to **All**.
5. Leave the remaining options as default and click **Launch** to start the solution process.

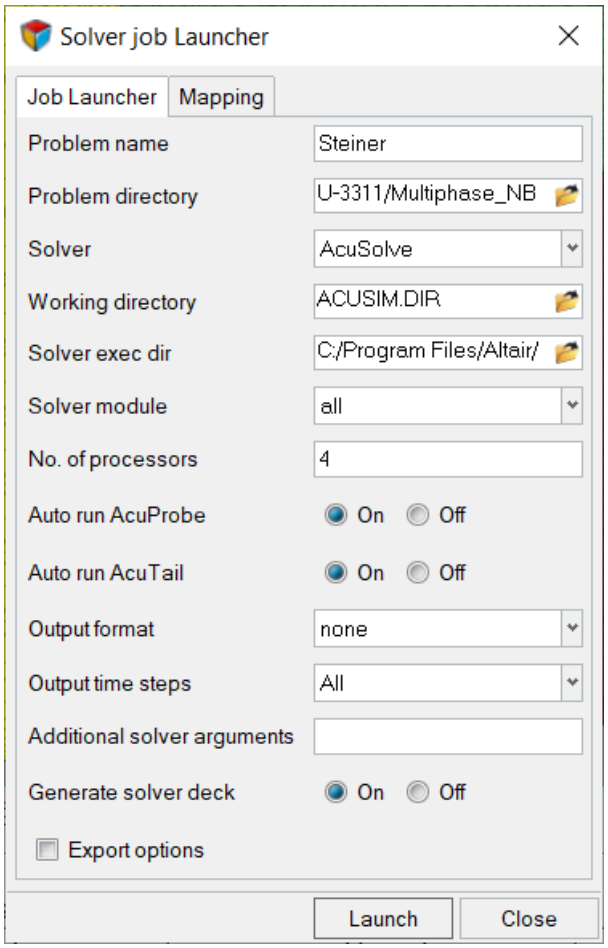


Figure 464:

Post-Process the Results with HyperView

Once the AcuSolve run is complete, close the HyperWorks Solver View dialog. In the HyperMesh Desktop window, close the AcuSolve Control and Solver job Launcher dialogs. In the next few steps, you will plot a contour of the vapor volume fraction.

Switch to the HyperView Interface and Load the AcuSolve Model and Results

1. In the HyperMesh Desktop window, click the **ClientSelector** drop-down in the bottom-left corner of the graphics window.

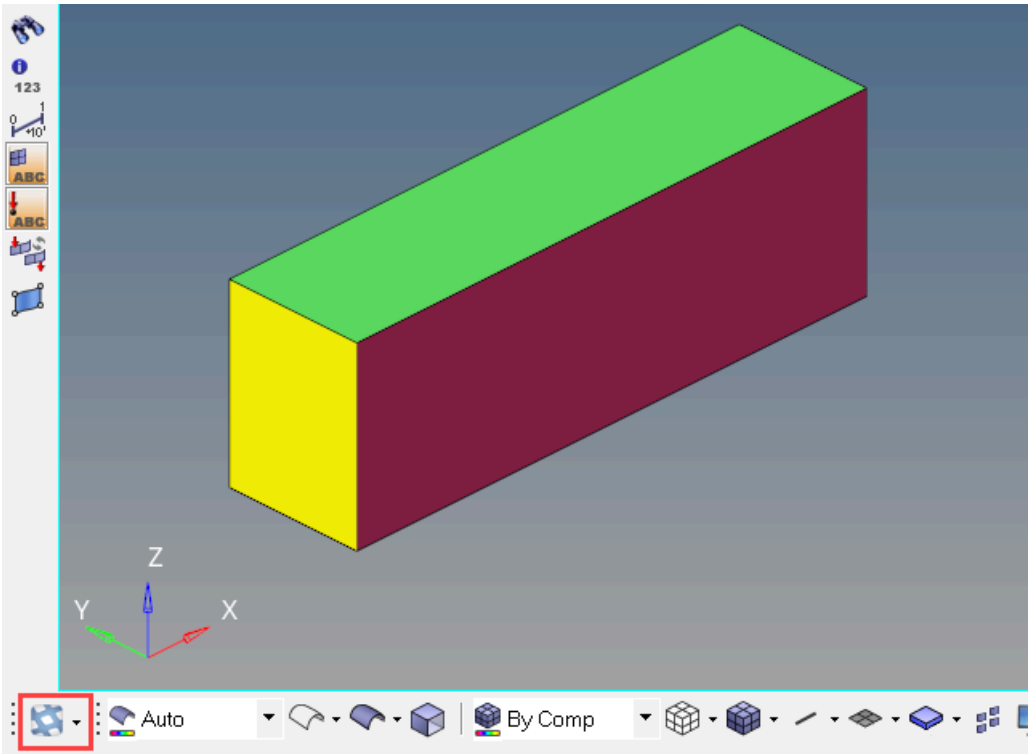


Figure 465:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.
   The interface is changed to HyperView.

   Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Steiner.1.Log`.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results.

The model is colored by geometry after loading.

Create Contours for the Volume Fraction of Vapor

In this step, you will display volume fraction contours of vapor.

1. Since this is a transient case, you need to plot the results at the last timestep. To do this, click the Time drop-down menu in the Results Browser and select the last option in the list.

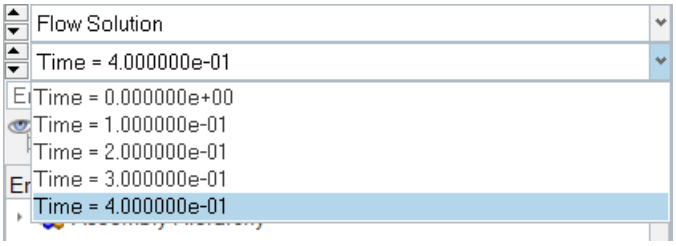


Figure 466:

2. In the Results Browser, expand the list of **Components**.
3. Click the **Isolate Shown** icon , hold Ctrl, then select the **HeatedWall** and **Bottom** components to turn off the display of all components except those that are required.

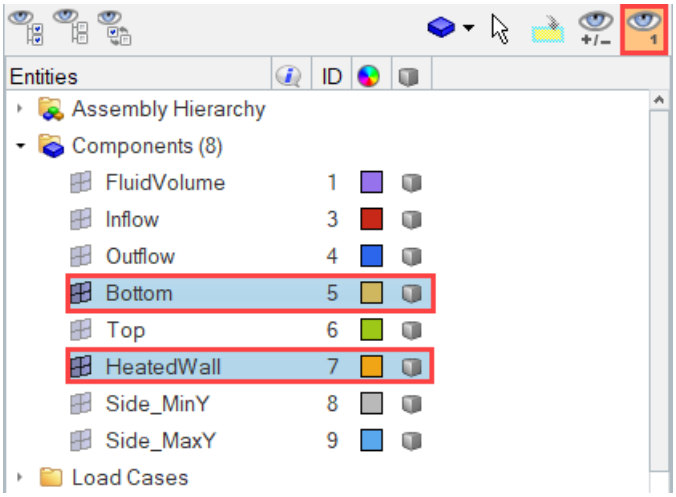


Figure 467:

4. Click on the Results toolbar to open the Contour panel.
5. In the panel area, change the Result type to **Volume_fraction-1-Vapor_DF1.VaporWater_Eulerian_HM(s)**.
6. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
7. Click **Apply** to plot the vapor volume fraction contours.
8. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 468:

9. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.

The contour plot should look similar to the figure below.

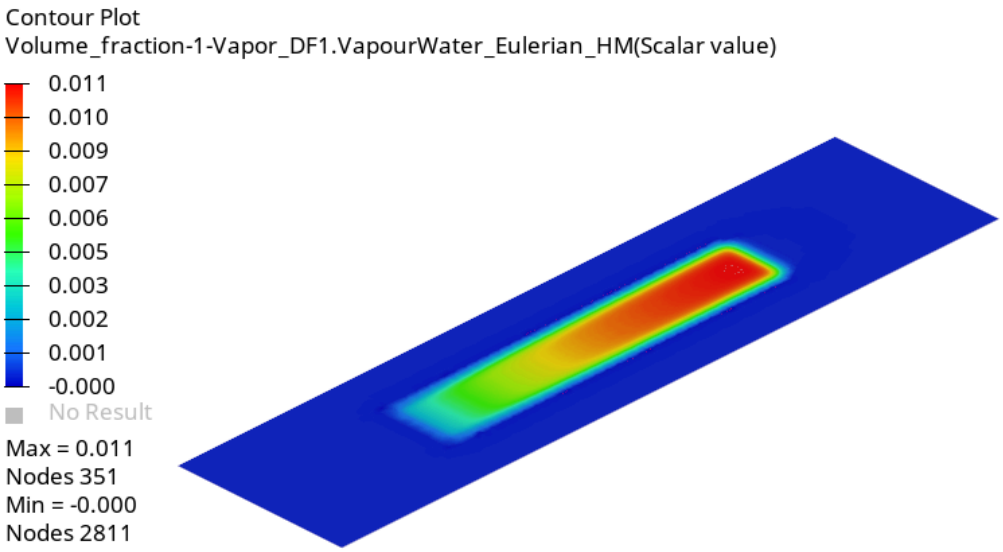


Figure 469:

Summary

In this tutorial, you worked through a basic workflow to set-up and solve a transient two-phase Nucleate Boiling flow problem using the Algebraic Eulerian multiphase model. You started by importing the model in HyperMesh. Then, you defined the simulation parameters and launched AcuSolve directly from within HyperMesh. Upon completion of solution by AcuSolve, you used HyperView to post-process the results and create a contour plot of the volume fraction.

2.4 Multiphase

2.4.1 ACU-T: 4000 Transient Dam Break Simulation

Prerequisites

This tutorial provides instructions for setting, solving and viewing results for a transient dam break simulation using the Level Set method. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T4000_DamBreak.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in the figure below. It consists of a square water column held in place by the reservoir walls. At time t=0, the walls are removed and the water column is now free to flow out. The simulation can be used to visualize and study the surge patterns as the column of water rushes out, as in a dam wall break.

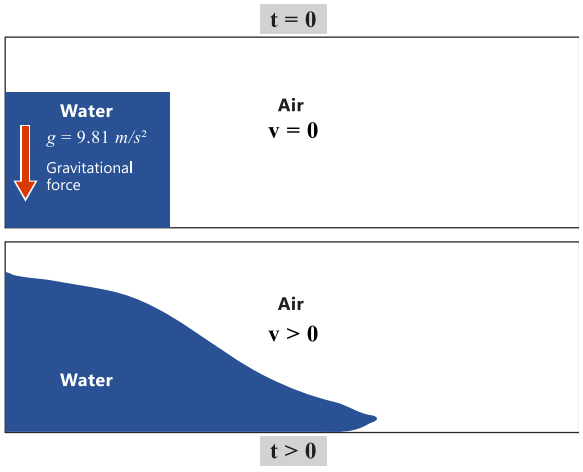


Figure 470:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon  located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4000_DamBreak.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `DamBreak` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `DamBreak` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the Transient Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Change the Analysis type to **Transient**.
3. Set the Multiphase equation to **Level Set**.

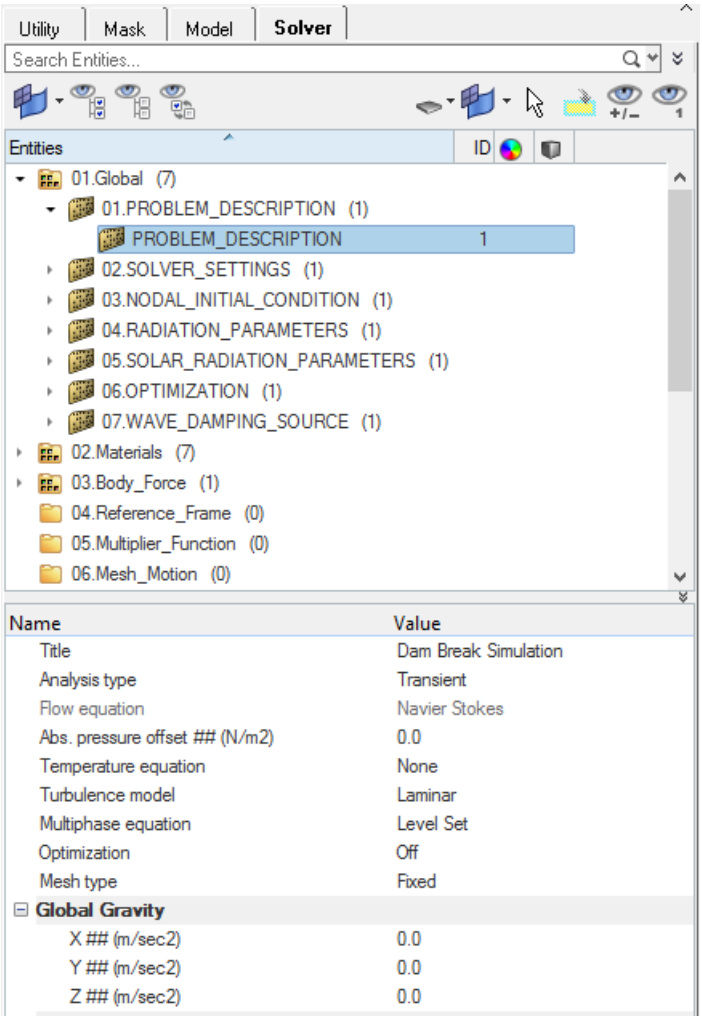


Figure 471:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. Set the Max time steps to `0` and press Enter.
3. Set the Final time to `1.0` sec.
4. Set the Initial time increment to `0.002498` sec.
5. Change the Max stagger iterations to `5` and press Enter.
6. Change the Relaxation factor to `0.0`.
7. Check that the Flow and Field options are turned On.

Name	Value
Max time steps	0
Final time ## (sec)	1.0
Initial time increment ## (sec)	0.002498
Auto time increment	☐
Multiplier Function	<Unspecified>
Convergence tolerance	0.001
Min stagger iterations	0
Max stagger iterations	5
Relaxation factor	0.0
Flow	On
Field	On
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 472:

Define the Nodal Outputs

1. In the Solver Browser, expand **17.Output** then click **NODAL_OUTPUT**.
2. In the Entity Editor, set the Time step frequency to `1` and toggle on the Output initial condition field.

Name	Value
Time step frequency	1
Time frequency ## (sec)	0.0
Output initial condition	☑
Continuous output	☐
Number of saved states	0
Unsupported Entries	
Number of Unsupported Entries	0

Figure 473:

Create a Multiphase Model and Set the Body Force

Create a Multiphase Material Model

1. In the Solver Browser, expand **02.Materials** then right-click on **MULTIPHASE** and select **Create**.
2. Enter `Air-Water` as the name of the model.
3. In the Entity Editor, verify that the Field Interaction Type is set to **Level Set**.
4. Select **Air_HM** as the Field 1 Material.
5. Enter `Air` as the Field 1 name.

6. Select **Water_HM** as the Field 2 Material.
7. Enter `Water` as the Field 2 name.

Name	Value
Name	Air-Water
Material type	MULTIPHASE
Fields	
Field Interaction Type	Level Set
Field 1 Material	(1) Air_HM
Field 1 Name	Air
Field 2 Material	(2) Water_HM
Field 2 Name	Water
Field Interaction Model Name	Air_HM.Water_HM.Air-Water
Field Interaction Parameters	
Surface Tension Model	<Unspecified>
Interface Thickness Type	Automatic
Turbulent Schmidt Number	0.85
Multifield Parameters	
Density type	Linear
Specific Heat Type	Linear
Viscosity Type	Linear
Conductivity Type	Linear
Immiscible Interface Type	Smooth

Figure 474:

Set the Body Force

1. In the Solver Browser, expand **03.Body_Force** then expand **BODY_FORCE** and click **Gravity_HM**.
2. In the Entity Editor, set the Y gravity to `-9.81` m/sec^2 and Z gravity to `0.0`.

Name	Value
Name	Gravity_HM
Medium	Fluid
Gravity	
Gravity type	Constant
X ## (m/sec2)	0.0
Y ## (m/sec2)	-9.81
Z ## (m/sec2)	0.0
Gravity multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

Figure 475:

Set the Nodal Initial Conditions and Boundary Conditions

In this step, you will start by creating a node-set for assigning nodal initial conditions for volume fraction. All the nodes in the set are assigned a volume fraction of 1 for water, thereby creating a water column.

Create a Node Set

1. Go to the Model Browser, right-click on empty space in the browser area, and select **Create** > **Set**.
2. In the Entity Editor, rename the set to `Water_Column`.

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **z_pos**. In the Entity Editor, change the Type to **SYMMETRY**.

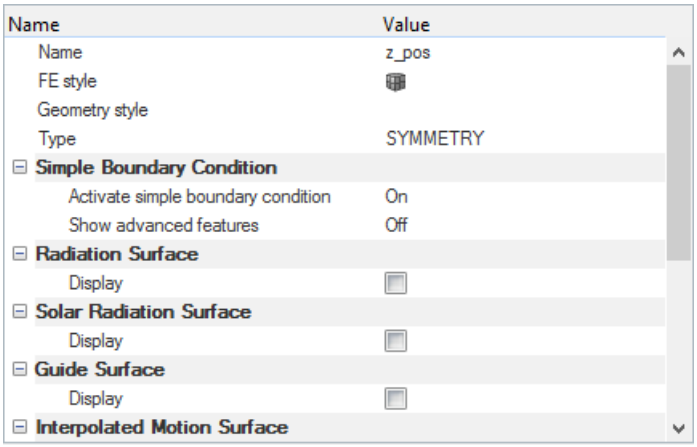


Figure 476:

3. Similarly, change the z_neg component Type to **SYMMETRY**.
4. Click **sides**. In the Entity Editor, verify that the Type is set to **WALL**.

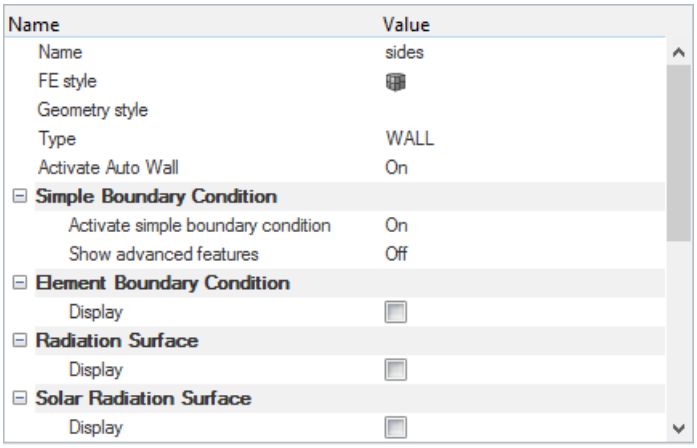


Figure 477:

5. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **MULTIPHASE**.
   b) Select **Air-Water** as the Material.
   c) Select **Gravity_HM** as the Body Force.

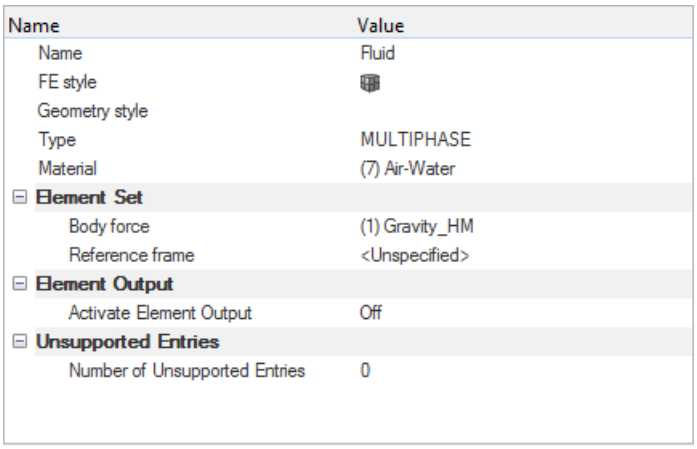


Figure 478:

Assign Reference Pressure

1. In the Solver Browser, right-click on **15.Nodal_Boundary_Condition** and select **Create**.
2. Set the Name to `Fixed Pressure Node` and change the Definition to **Nodes**.
3. Set the Number of Nodes to `1`.
4. Click on the **Node** collector. Then, in the graphics window, click on a node in the Fluid volume.

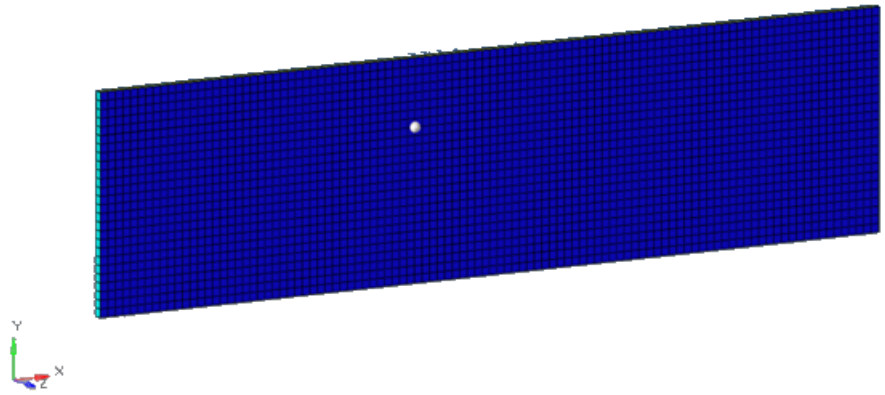


Figure 479:

5. Click **proceed** in the panel area.
6. Change the Boundary condition variable to **Pressure** and leave the remaining settings as default.

Name	Value
Name	Fixed Pressure Node
Definition	Nodes
Number of Nodes	1
Nodes	Node
Precedence	1
Boundary condition variable	Pressure
Type	Zero
Active type	Always
Unsupported Entries	
Number of Unsupported Entries	0

Figure 480:

Set the Nodal Initial Conditions

1. In the Solver Browser, expand **01.Global** then click **03.NODAL_INITIAL_CONDITION**.
2. In the Entity Editor, under the Field tab, set the Type to **Nodal Values**.
3. Set the Multiphase Field to **Water**.
4. Set the Select nodes by option to **Node Set**.
5. With the Node Set Count set to `1`, select **Water_Column** as the Node Set.
6. Set the Initial Volume Fraction to `1.0`.

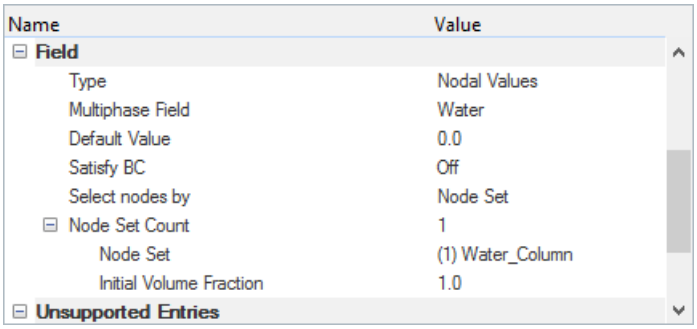


Figure 481:

Assign Nodes to the Node Set

1. Go to the Model Browser, right-click on empty space in the browser area, and select **Create** > **Block**.
2. In the Entity Editor, rename the block to `Water_Column`.
3. Set the Min and Max coordinates to `(0, 0, 0)` and `(0.05715, 0.05715, 0.002748)`, respectively.
4. In the Model Browser, click **Sets** > **Water_Column**. In the Entity Editor, click on the **Nodes** collector in the Entity IDs field.
5. In the panel area, click the **nodes** collector and select the **by block** option.
6. Toggle on the **Water_Column** block then click **select**.
   All the nodes in the Water column block are highlighted in the graphics area.

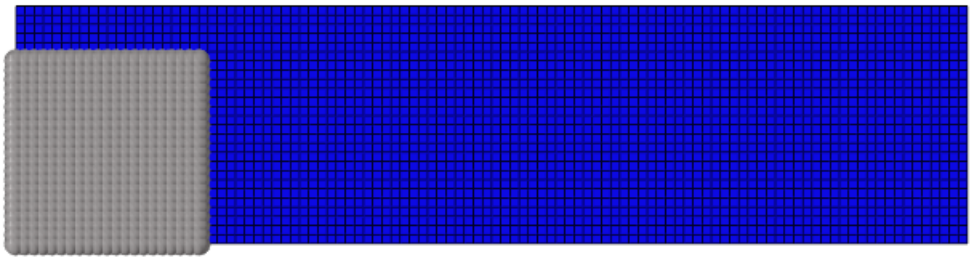


Figure 482:

7. Click **proceed**.
8. Optional: Turn off the display of the Water_Column block.
9. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to All or Final. Since this is a Transient analysis, set it to **All**.
5. Leave the remaining options as default and click **Launch** to start the solution process.

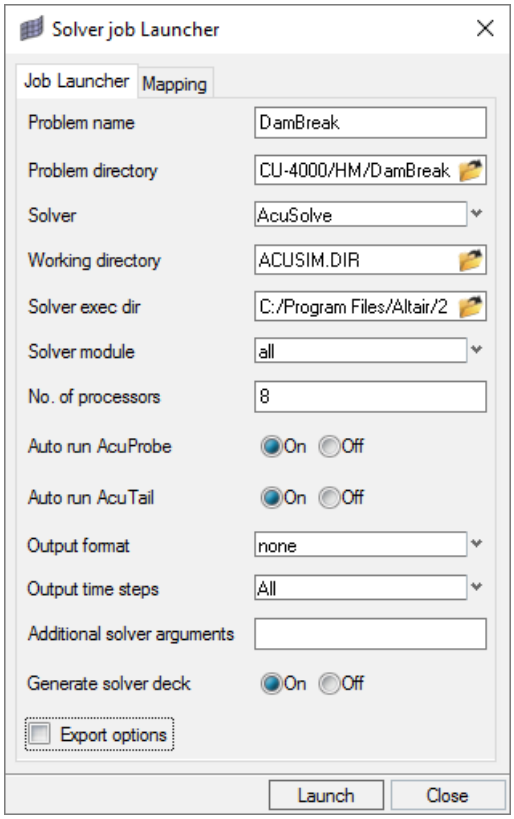


Figure 483:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.

2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `DamBreak.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create the Water Flow Animation

In this step, you will create an animation of the water flow as it surges once the walls restricting the water column are removed.

1. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
2. Click on the Results toolbar to open the Contour panel.
3. Select **Volume_fraction-2-Water (s)** as the Result type.
4. Click **Apply** to display the volume fraction contour at the first time step.
5. Click the **Legend** tab then click **Edit Legend**.
6. In the **Edit Legend** dialog, change the Number of levels to `2` and the Numeric format to **Fixed**.

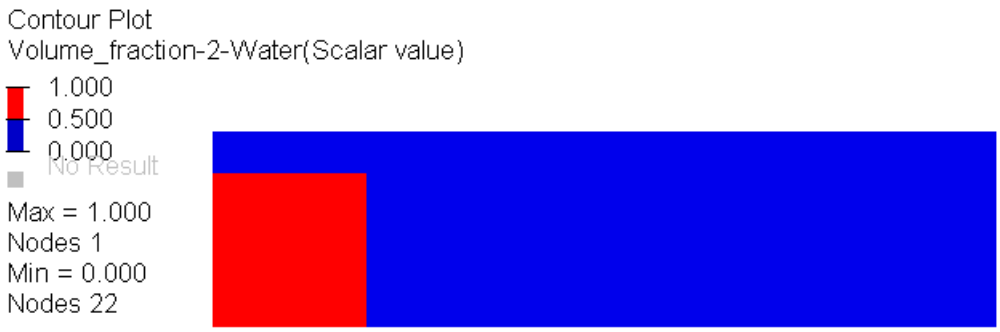


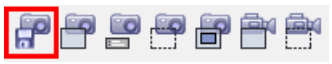
Figure 484:

7. On the Animation toolbar, click the Animation Controls icon .
8. Drag the Max frame Rate slider to `50` fps.
9. Click the Start/Pause Animation icon to play the animation in the graphics area.

Save the Animation

1. In the menu area, select **Preferences** > **Export Settings** > **AVI**.

2. In the **Export Settings AVI** dialog, set the Frame rate to `50` fps and click **OK**.
3. On the ImageCapture toolbar, make sure that the Save Image to File option is On.



4. Click the Capture Graphics Area Video icon  .
   The **Save Graphics Area Video As** dialog opens.
5. Navigate to the location where you want to save the file, enter a name of your choice, and click **Save**.

Summary

In this tutorial, you successfully learned how to set up and solve a multiphase flow problem using HyperMesh and AcuSolve. You also learned how to create a multiphase model using the Level Set method. Once the solution was computed, you post-processed the results in HyperView where you generated an animation of the water flow as it surged once the dam walls were removed.

2.4.2 ACU-T: 4001 Water Filling in a Tank

Prerequisites

This tutorial provides instructions for running a transient simulation of a two-phase flow in a square tank using the Level Set model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T4001_FillingTank.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be solved is shown schematically in the figure below. It consists of a half-filled water tank at time t=0. Water is injected through the Inlet at t=0 and as the water fills in through the inlet, the water-air interface can be visualized in a transient simulation.

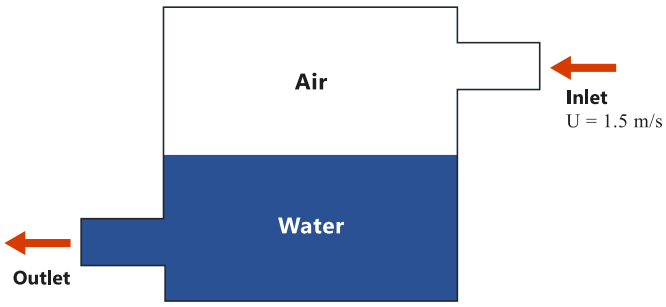


Figure 485:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4001_FillingTank.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `FillingTank` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `FillingTank` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Change the Analysis type to **Transient**.
3. Set the Multiphase equation to **Level Set**.

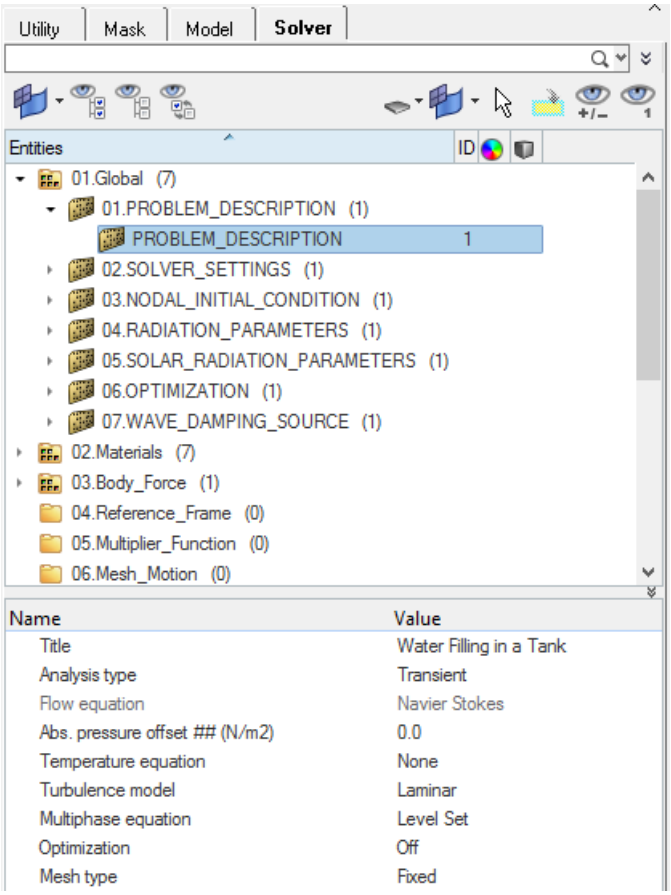


Figure 486:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. Set the Max time steps to `0` and press Enter.
3. Set the Final time to `3.0` sec.
4. Set the Initial time increment to `0.01` sec.
5. Change the Max stagger iterations to `4` and press Enter.
6. Change the Relaxation factor to `0.0`.
7. Check that the Flow and Field options are turned On.

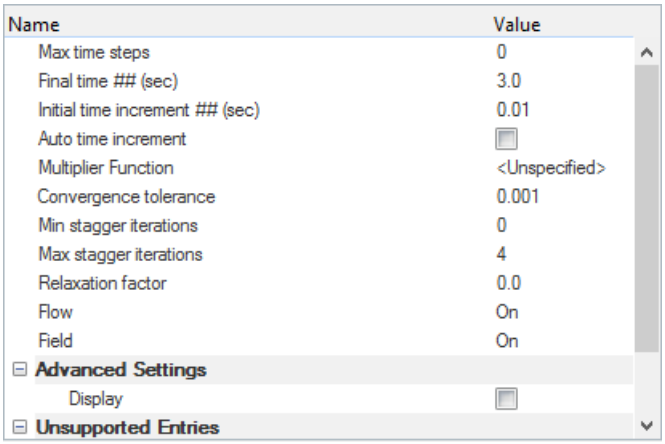


Figure 487:

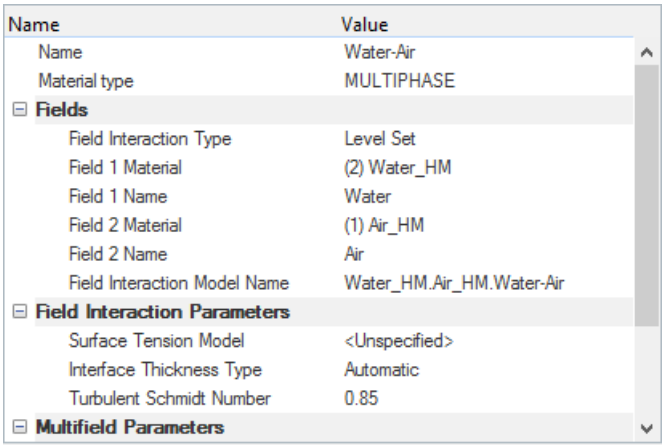
Define the Nodal Outputs

1. In the Solver Browser, expand **17.Output** and click **NODAL_OUTPUT**.
2. Set the Time step frequency to `1`.
3. Toggle **On** the Output initial condition field.
4. Save the model.

Create a Multiphase Model and Set the Body Force

Create a Multiphase Material Model

1. In the Solver Browser, expand **02.Materials** then right-click on **MULTIPHASE** and select **Create**.
2. Enter `Water-Air` as the name of the model.
3. In the Entity Editor, verify that the Field Interaction Type is set to **Level Set**.
4. Select **Water_HM** as the Field 1 Material.
5. Enter `Water` as the Field 1 name.
6. Select **Air_HM** as the Field 2 Material.
7. Enter `Air` as the Field 2 name.



Name	Value
Name	Water-Air
Material type	MULTIPHASE
Fields	
Field Interaction Type	Level Set
Field 1 Material	(2) Water_HM
Field 1 Name	Water
Field 2 Material	(1) Air_HM
Field 2 Name	Air
Field Interaction Model Name	Water_HM.Air_HM.Water-Air
Field Interaction Parameters	
Surface Tension Model	<Unspecified>
Interface Thickness Type	Automatic
Turbulent Schmidt Number	0.85
Multifield Parameters	

Figure 488:

Set the Body Force

1. In the Solver Browser, expand **03.Body_Force** then expand **BODY_FORCE** and click **Gravity_HM**.
2. In the Entity Editor, set the Y gravity to `-9.81` m/sec^2 and Z gravity to `0.0`.

Name	Value
Name	Gravity_HM
Medium	Fluid
Gravity	
Gravity type	Constant
X ## (m/sec2)	0.0
Y ## (m/sec2)	-9.81
Z ## (m/sec2)	0.0
Gravity multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

Figure 489:

Set the Boundary Conditions and Nodal Initial Conditions

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.

2. Click **Fluid**. In the Entity Editor,

a) Change the Type to **MULTIPHASE**

b) Select **Water-Air** as the Material.

c) Select **Gravity_HM** as the Body force.

Name	Value
Name	Fluid
FE style	
Geometry style	
Type	MULTIPHASE
Material	(7) Air-Water
Element Set	
Body force	(1) Gravity_HM
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 490:

3. Click **wall**. In the Entity Editor, verify that the Type is set to **WALL**.

Name	Value
Name	wall
FE style	
Geometry style	
Type	WALL
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Element Boundary Condition	
Display	
Radiation Surface	
Display	
Solar Radiation Surface	
Display	
Guide Surface	
Display	

Figure 491:

4. Click **inlet**. In the Entity Editor,

a) Change the Type to **INFLOW**.

b) Set the Normal Velocity to `1.5` m/s.

c) Select **Water** as the Multiphase incoming field.

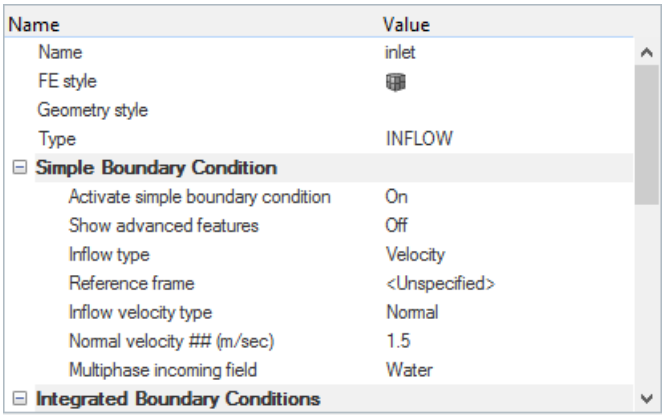


Figure 492:

5. Click **outlet**. In the Entity Editor,

a) Change the Type to **OUTFLOW**.

b) Toggle **On** the Hydrostatic pressure field.

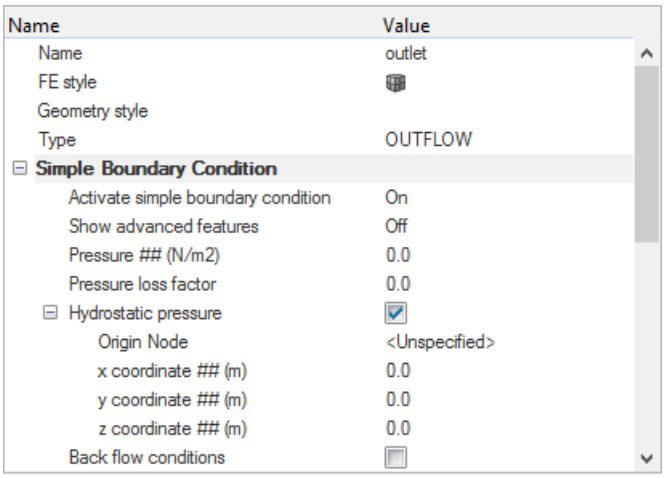


Figure 493:

6. Click **z_pos**. In the Entity Editor, change the Type to **SLIP**.

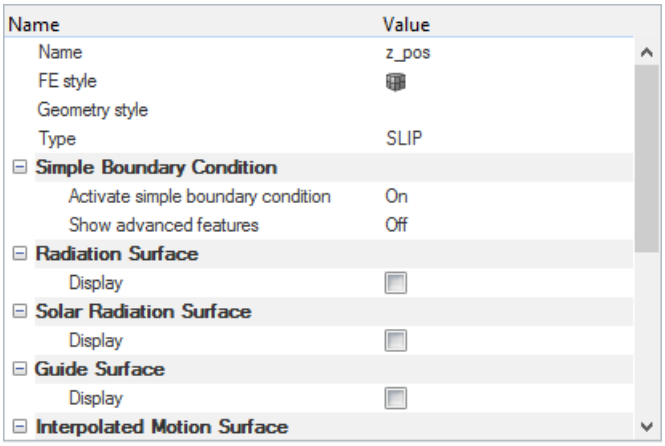


Figure 494:

7. Similarly, change the Type to **Slip** for z_neg.

Create a Node Set

1. Go to the Model Browser, right-click on empty space in the browser area, and select **Create** > **Set**.
2. In the Entity Editor, rename the set to `Water_Column`.

Set the Nodal Initial Conditions

1. Go to the Solver Browser, expand **01.Global**, then click **03.NODAL_INITIAL_CONDITION**.
2. In the Entity Editor, under the Field tab, set the Type to **Nodal Values**.
3. Set the Multiphase Field to **Water**.
4. Set the Select nodes by option to **Node Set**.
5. With the Node Set Count set to `1`, select **Water_Column** as the Node Set.
6. Set the Initial Volume Fraction to `1.0`.

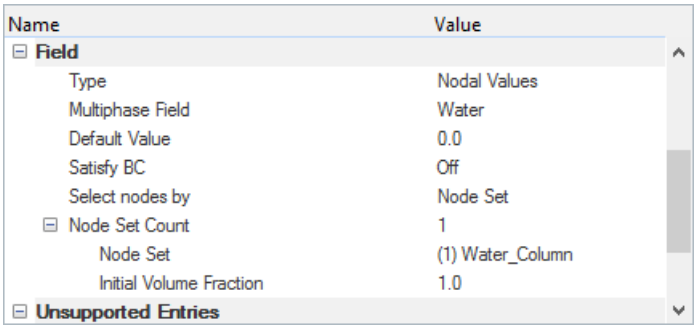


Figure 495:

7. Save the model.

Assign Nodes to the Node Set

1. Go to the Model Browser, right-click on empty space in the browser area, and select **Create** > **Block**.
2. In the Entity Editor, rename the block to `Water_Column`.
3. Set the Min and Max coordinates to `(-0.9, -0.5, -0.005)` and `(0.5, 0, 0.005)`, respectively.
4. In the Model Browser, click **Sets** > **Water_Column**. In the Entity Editor, click on the **Nodes** collector in the Entity IDs field.
5. In the panel area, click the **nodes** collector and select the **by block** option.
6. Toggle on the **Water_Column** block then click **select**.
   All the nodes in the Water column block are highlighted in the graphics area.

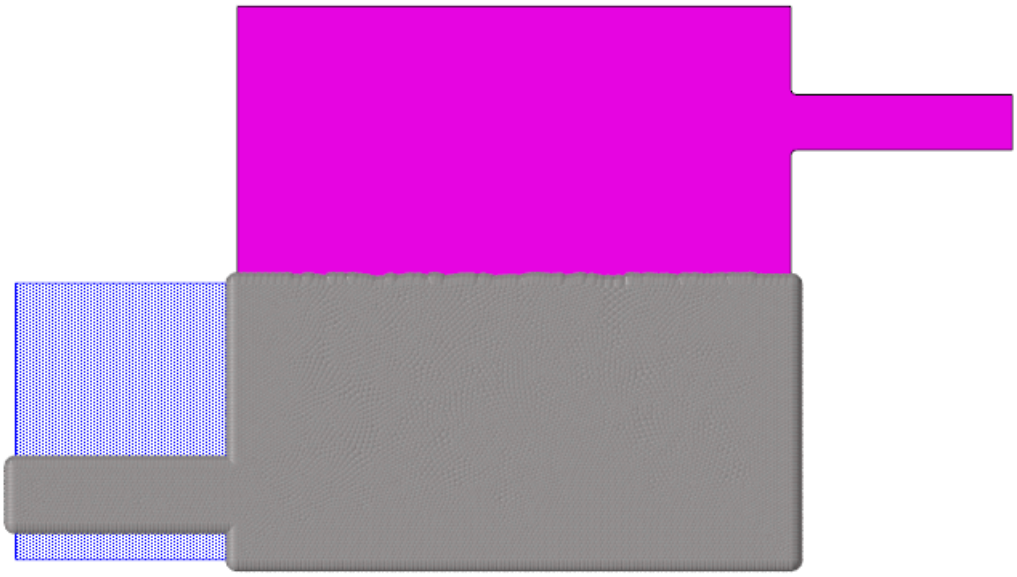


Figure 496:

7. Click **proceed**.
8. Optional: Turn off the display of the Water_Column block.
9. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to All or Final. Since this is a Transient analysis, set it to **All**.
5. Leave the remaining options as default and click **Launch** to start the solution process.

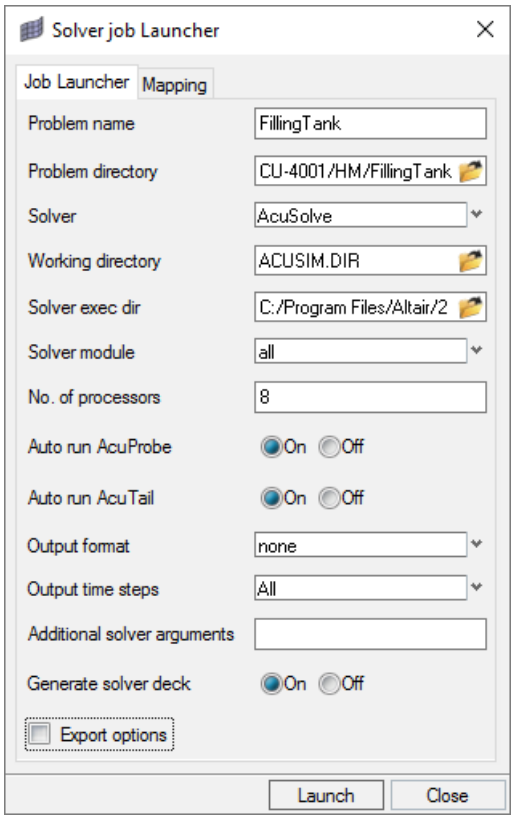


Figure 497:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.

3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `FillingTank.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create the Water Flow Animation

In this step, you will create an animation of the water flow as it fills in through the inlet.

1. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
2. Click on the Results toolbar to open the Contour panel.
3. Select **Volume_fraction-2-Water (s)** as the Result type.
4. Click **Apply** to display the volume fraction contour at the first time step.
5. Click the **Legend** tab then click **Edit Legend**.
6. In the **Edit Legend** dialog, change the Number of levels to `2` and the Numeric format to **Fixed** then click **OK**.

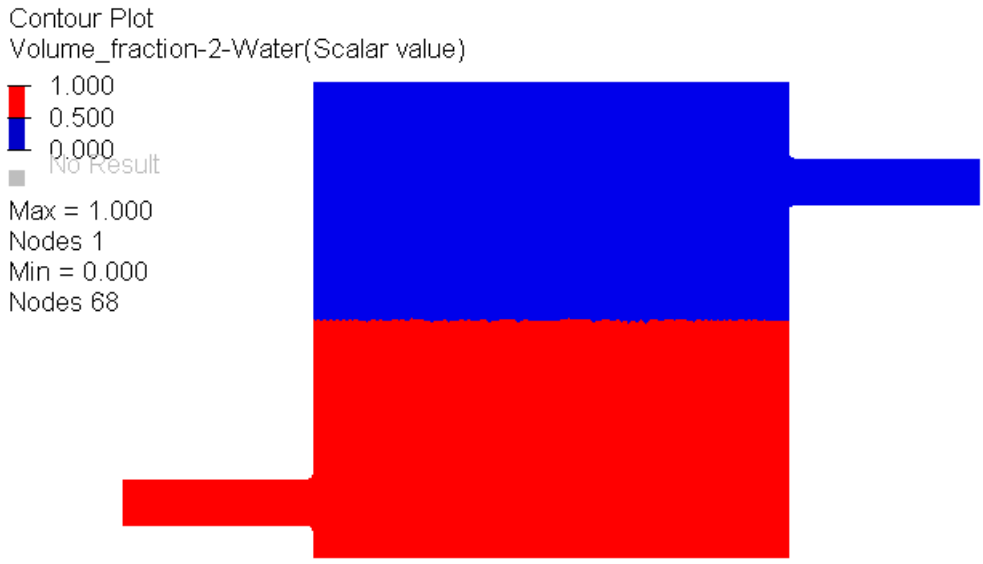


Figure 498:

7. On the Animation toolbar, click the Animation Controls icon .
8. Drag the Max frame Rate slider to `50` fps.
9. Click the Start/Pause Animation icon to play the animation in the graphics area.

Save the Animation

1. In the menu area, select **Preferences** > **Export Settings** > **AVI**.
2. In the **Export Settings AVI** dialog, set the Frame rate to `50` fps and click **OK**.
3. On the ImageCapture toolbar, make sure that the Save Image to File option is On.



4. Click the Capture Graphics Area Video icon .
   The **Save Graphics Area Video As** dialog opens.
5. Navigate to the location where you want to save the file, enter a name of your choice, and click **Save**.

Summary

In this tutorial, you successfully learned how to set up and solve a transient multiphase flow problem involving water filling a tank using HyperMesh and AcuSolve. You also learned how to create a multiphase model using the Level Set method. Once the solution was computed, you post-processed the results in HyperView where you generated an animation of the water flow as it filled the tank.

2.4.3 ACU-T: 4002 Sloshing of Water in a Tank

Prerequisites

This tutorial provides instructions for running a transient simulation of a two-phase flow in a rectangular tank using the level set model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T4002_SloshingTank.hm` and `gravity.c` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be solved is shown schematically in the figure below. It consists of a partially filled water tank and from time t=0, water inside the tank is subjected to a sinusoidal varying body force along x-direction and constant gravity along y-direction.

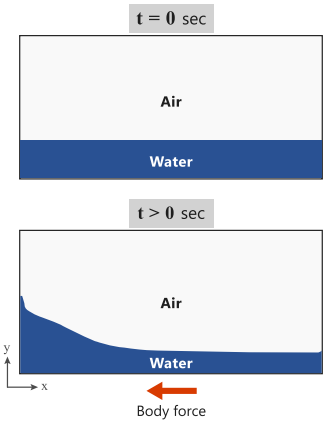


Figure 499:

The body force in the x-direction is given by the expression:

$$A\omega^2 \sin(\omega t + \varphi)$$

Where

- A = Amplitude of oscillation = -0.06 m
- ω = Frequency of oscillation = $\frac{2\pi}{T}$ = 3.6 rad/sec
- T = Time period of oscillation = 1.74 sec
- φ = Phase difference = 0

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4002_SloshingTank.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `TankSloshing` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `TankSloshing` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Change the Analysis type to **Transient**.
3. Set the Multiphase equation to **Level Set**.

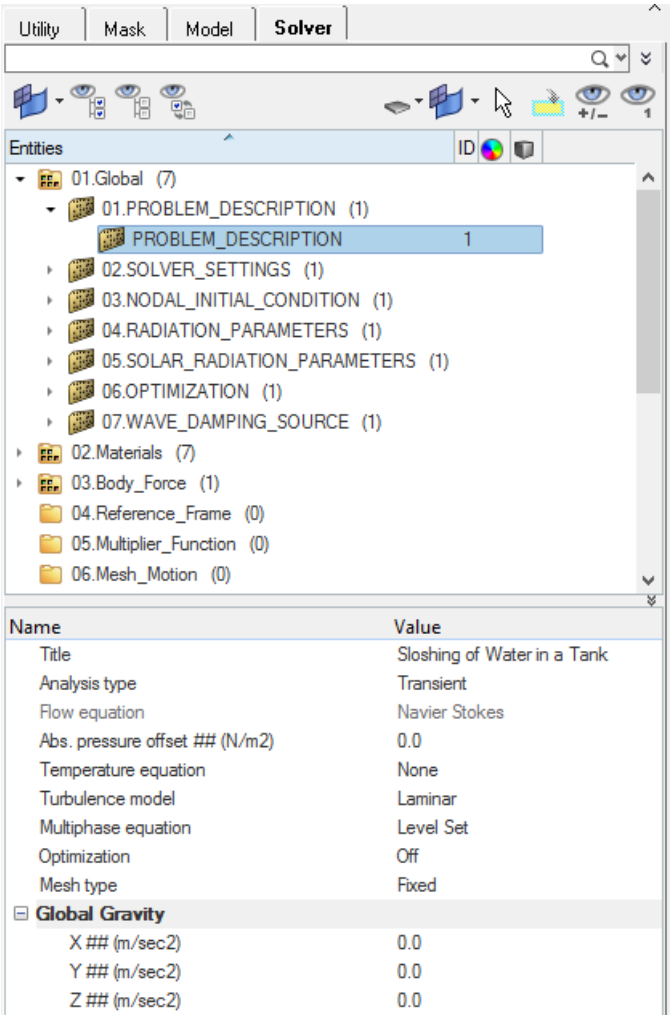


Figure 500:

4. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
5. Set the Max time steps to `0` and press Enter.
6. Set the Final time to `4.0` sec.
7. Set the Initial time increment to `0.006` sec.
8. Change the Min stagger iterations to `2` and press Enter.
9. Change the Max stagger iterations to `6` and press Enter.
10. Change the Relaxation factor to `0.0`.

11. Check that the Flow and Field options are turned On.

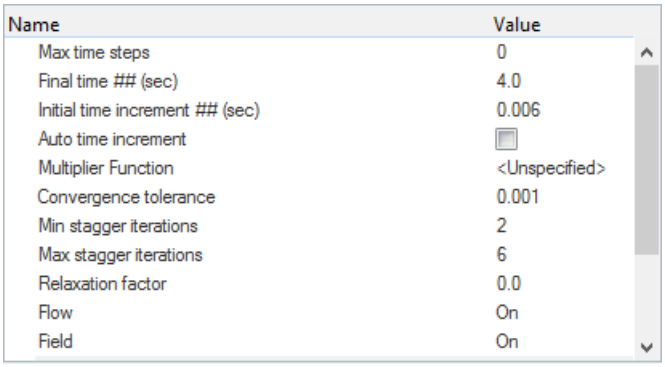


Figure 501:

Define the Nodal Outputs

1. In the Solver Browser, expand **17.Output** and click **NODAL_OUTPUT**.
2. Set the Time step frequency to `10`.
3. Toggle **On** the Output initial condition field.

Create a Multiphase Model and Set the Body Force

Create a Multiphase Material Model

1. In the Solver Browser, expand **02.Materials** then right-click on **MULTIPHASE** and select **Create**.
2. Enter `Air-Water` as the name of the model.
3. In the Entity Editor, under the Fields tab, verify that the Field Interaction Type is set to **Level Set**.
4. Select **Air_HM** as the Field 1 Material.
5. Enter `Air` as the Field 1 name.
6. Select **Water_HM** as the Field 2 Material.
7. Enter `Water` as the Field 2 name.

Name	Value
Name	Air-Water
Material type	MULTIPHASE
Fields	
Field Interaction Type	Level Set
Field 1 Material	(1) Air_HM
Field 1 Name	Air
Field 2 Material	(2) Water_HM
Field 2 Name	Water
Field Interaction Model Name	Air_HM.Water_HM.Air-Water
Field Interaction Parameters	
Surface Tension Model	<Unspecified>
Interface Thickness Type	Automatic
Turbulent Schmidt Number	0.85
Multifield Parameters	
Density type	Linear
Specific Heat Type	Linear
Viscosity Type	Linear
Conductivity Type	Linear
Immiscible Interface Type	Smooth

Figure 502:

Set the Body Force

1. In the Solver Browser, expand **03.Body_Force** then expand **BODY_FORCE** and click **Gravity_HM**.
2. In the Entity Editor, under the Gravity tab, change the Gravity type to **User Function**.
3. Set the User function name to `usrGravity`.
4. Set the Number of user values to `3`.

Name	Value
Name	Gravity_HM
Medium	Fluid
Gravity	
Gravity type	User Function
User function name	usrGravity
User values input type	Table
Number of user values	3
Data: User values	
User strings input type	Table
Number of user strings	0
Gravity multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

Figure 503:

5. Click the table icon in the Data: User values field.

The **Number of user values** dialog opens.

6. Enter `(-0.06, 1.74, 0)` as the three values then click **Close**.

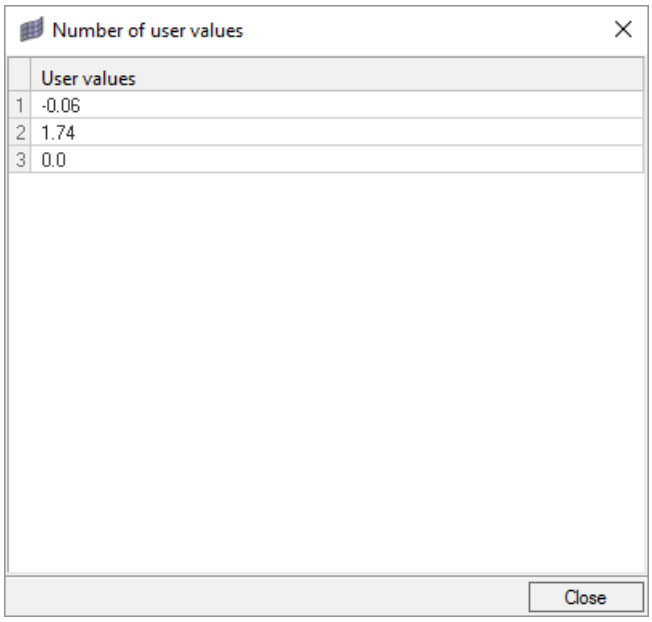


Figure 504:

7. A UDF in the form of C language (`gravity.c`) is provided with the tutorial. This program should be compiled using the following steps:

   For Windows:

   1. Start AcuSolve Command Prompt from the Windows Start menu by clicking **Start** > **All Programs** > **Altair HyperWorks <version>** > **AcuSolve Cmd Prompt** .
   2. Change the directory to the present working directory using the `cd` command.
   3. Enter the command `acuMakeDll -src gravity.c` and press Enter.

   For Linux:

   1. In the terminal, change the directory to the present working directory using the `cd` command.
   2. Enter the command `acuMakeLib -src gravity.c` and press Enter.

   This command creates a set of files necessary for the use of the UDF.

Set the Boundary Conditions and Nodal Initial Conditions

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.

2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **MULTIPHASE**
   b) Select **Air-Water** as the Material.
   c) Select **Gravity_HM** as the Body force.

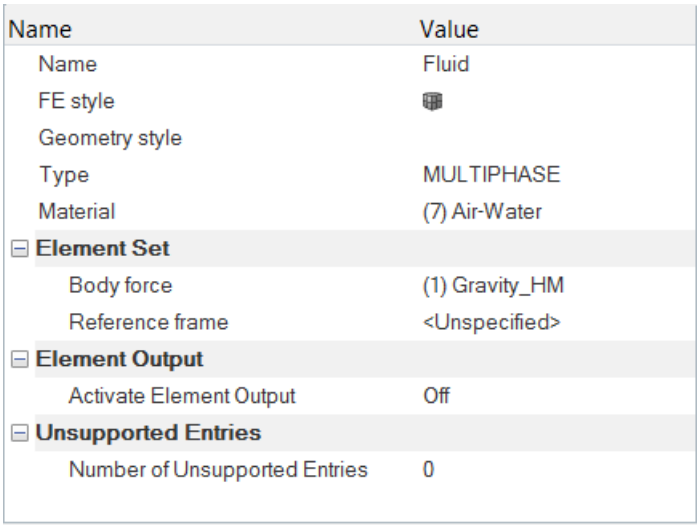


Figure 505:

3. Click **Wall**. In the Entity Editor, verify that the Type is set to **WALL**.

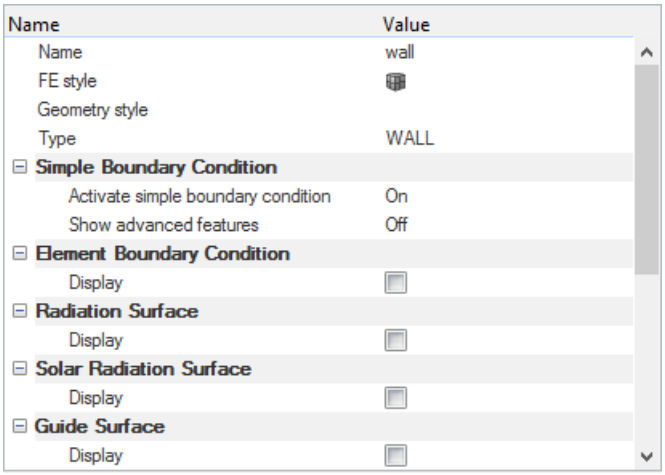


Figure 506:

4. Change the **Front** and **Rear** component Types to **SLIP**.

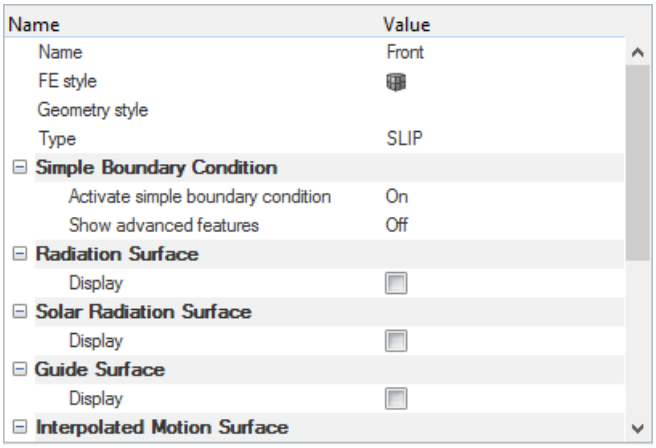


Figure 507:

Create a Node Set

1. Go to the Model Browser, right-click on empty space in the browser area, and select **Create** > **Set**.
2. In the Entity Editor, rename the set to `Water_Column`.

Set the Nodal Initial Conditions

1. Go to the Solver Browser, expand **01.Global**, then click **03.NODAL_INITIAL_CONDITION**.
2. In the Entity Editor, under the Field tab, set the Type to **Nodal Values**.
3. Set the Multiphase Field to **Water**.
4. Set the Select nodes by option to **Node Set**.
5. With the Node Set Count set to `1`, select **Water_Column** as the Node Set.
6. Set the Initial Volume Fraction to `1.0`.

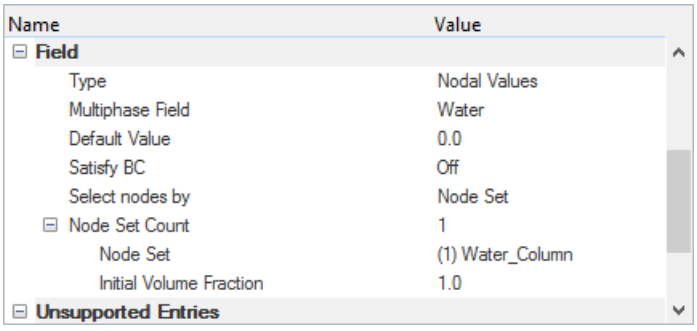


Figure 508:

7. Save the model.

Assign Nodes to the Node Set

1. Go to the Model Browser, right-click on empty space in the browser area, and select **Create** > **Block**.
2. In the Entity Editor, rename the block to `Water_Column`.
3. Set the Min and Max coordinates to `(0, 0, 0)` and `(1.2, 0.12, 0.1)`, respectively.
4. In the Model Browser, click **Sets** > **Water_Column**. In the Entity Editor, click on the **Nodes** collector in the Entity IDs field.
5. In the panel area, click the **nodes** collector and select the **by block** option.
6. Toggle on the **Water_Column** block then click **select**.
   All the nodes in the Water column block are highlighted in the graphics area.

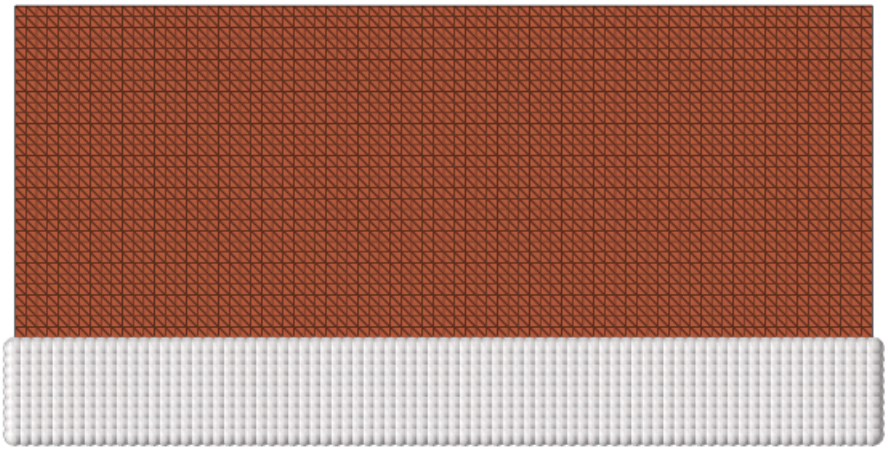


Figure 509:

7. Click **proceed**.
8. Optional: Turn off the display of the Water_Column block.
9. Save the model.

Assign the Reference Pressure

1. In the Solver Browser, right-click on **15.Nodal_Boundary_Condition** and select **Create**.
2. Change the name to `Fixed Pressure Node`.
3. Change the Definition to **Nodes**.
4. Set the Number of Nodes to `1`.
5. Click the **Node** collector and select a node on the Fluid volume in the graphics window.
6. Click **proceed** in the panel area.
7. Change the Boundary condition variable to **Pressure**.

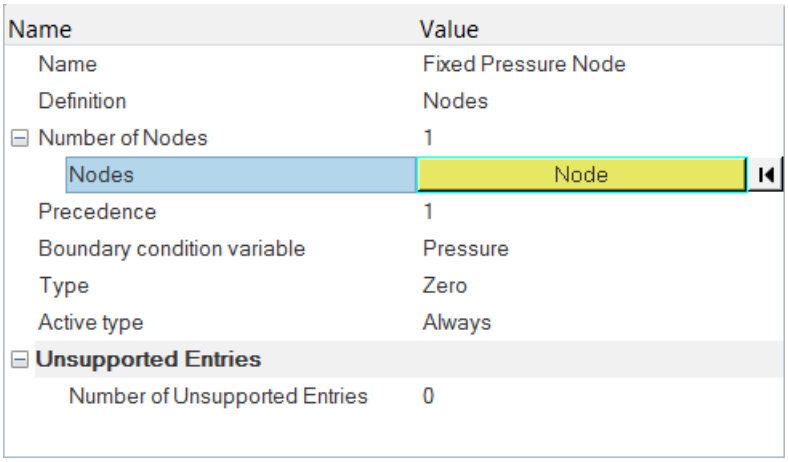


Figure 510:

8. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to All or Final. Since this is a Transient analysis, set it to **All**.
5. Leave the remaining options as default and click **Launch** to start the solution process.

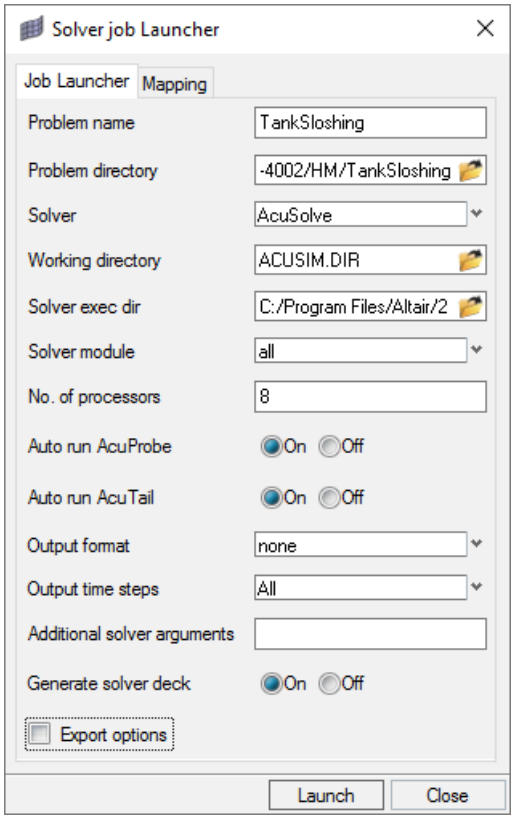


Figure 511:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `TankSloshing.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create the Water Flow Animation

In this step, you will create an animation of the water flow as it fills in through the inlet.

1. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
2. Click on the Results toolbar to open the Contour panel.
3. Select **Volume_fraction-2-Water (s)** as the Result type.
4. Click **Apply** to display the volume fraction contour at the first time step.
5. Click the **Legend** tab then click **Edit Legend**.
6. In the **Edit Legend** dialog, change the Number of levels to 2 and the Numeric format to **Fixed**.
7. On the Animation toolbar, click the Animation Controls icon .
8. Drag the Max Frame Rate slider to 5 fps.
9. Click the Start/Pause Animation icon to play the animation in the graphics area.



Figure 512:

Save the Animation

1. In the menu area, select **Preferences** > **Export Settings** > **AVI**.
2. In the **Export Settings AVI** dialog, set the Frame rate to 5 fps and click **OK**.
3. On the ImageCapture toolbar, make sure that the Save Image to File option is On.



4. Click the Capture Graphics Area Video icon .
   The **Save Graphics Area Video As** dialog opens.

5. Navigate to the location where you want to save the file, enter a name of your choice, and click **Save**.

Summary

In this tutorial, you successfully learned how to set up and solve a transient multiphase flow problem involving water sloshing in a tank using HyperMesh and AcuSolve. You also learned how to create a multiphase model using the Level Set method and specify the body force using a user-defined function and then compile the UDF. Once the solution was computed, you post-processed the results in HyperView where you generated an animation of the water sloshing in the tank.

2.4.4 ACU-T: 4100 Multiphase Flow using Algebraic Eulerian Model

Prerequisites

This tutorial provides instructions for running a transient simulation of a two-phase flow in a pipe using the Algebraic Eulerian model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperWorks, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T4100_Disperse.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 1. As an example, an LPipe problem is attached here to show the capability of the Disperse modeling in AcuSolve. The Algebraic Eulerian (AE) model is used to simulate the momentum exchange between a carrier field and a dispersed field. When simulating multiphase flows using the AE model, the carrier field has to be a fluid and the dispersed field can be of any medium.

In this problem, Water is considered a Carrier field material and Air is considered as Dispersed field material. Fluid enters the Inlet at an Average Velocity of 1 m/sec and the Water and Air volume fractions at the inlet are 96% and 4% respectively.

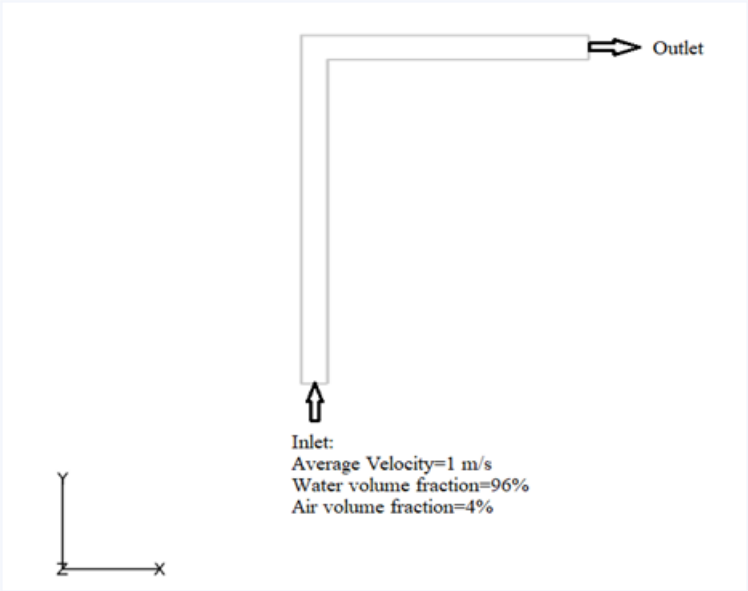


Figure 513:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4100_Disperse.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Lpipe_Disperse` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `LPipe` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.

2. In the Entity Editor, set the Analysis type to **Transient**.
3. Set the Turbulence Model to **Spalart Allmaras**.
4. Set Multiphase equation to **Algebraic Equation**.

Name	Value
Title	Multiphase Flow using Algebraic Eulerian Model
Analysis type	Transient
Flow equation	Navier Stokes
Abs. pressure offset ## (N/m2)	0.0
Temperature equation	None
Turbulence model	Spalart Allmaras
Multiphase equation	Algebraic Eulerian
Optimization	Off
Mesh type	Fixed
Global Gravity	
X ## (m/sec2)	0.0
Y ## (m/sec2)	0.0
Z ## (m/sec2)	0.0
Unsupported Entries	
No of Analysis Unsupported Entries	0
No of Equation Unsupported Entries	0

Figure 514:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, set the Max time steps to `150`.
3. Set the Initial time increment to `0.01` sec.
4. Set the Min stagger iterations to `2`.
5. Set the Max stagger iterations to `4`.
6. Change the Relaxation factor to `0`.
7. Verify that Flow, Turbulence, and Field are turned **On**.

Name	Value
Max time steps	150
Final time ## (sec)	0.0
Initial time increment ## (sec)	0.01
Auto time increment	☐
Multiplier Function	<Unspecified>
Convergence tolerance	0.001
Min stagger iterations	2
Max stagger iterations	4
Relaxation factor	0.0
Flow	On
Turbulence	On
Field	On
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 515:

Define the Nodal Outputs

1. In the Solver Browser, expand **17.Output** and click on **NODAL_OUTPUT**.
2. In the Entity Editor, set Time step frequency to `10`.
3. Toggle **On** the Output initial condition field.

Name	Value
Time step frequency	10
Time frequency ## (sec)	0.0
Output initial condition	☑
Continuous output	☐
Number of saved states	0
Unsupported Entries	
Number of Unsupported Entries	0

Figure 516:

Set Up Material Model Parameters and Body Force

In this step, you will start by setting up the Multiphase material model and body force parameters. Then, you will assign the surface boundary conditions and material properties to the fluid volume.

Set Up Material Model Parameters

1. In the Solver Browser, expand **02.Materials** > **MULTIPHASE** and click on **AirWater Eulerian HM**.
2. In the Entity Editor, verify that the Field Interaction Type is set to **Algebraic Eulerian**.
3. By default, **Water_HM** is set as the Carrier Material. Since water is the carrier field for this analysis, leave this option as is.
4. Verify that the Number of Dispersed Field is set to `1`.

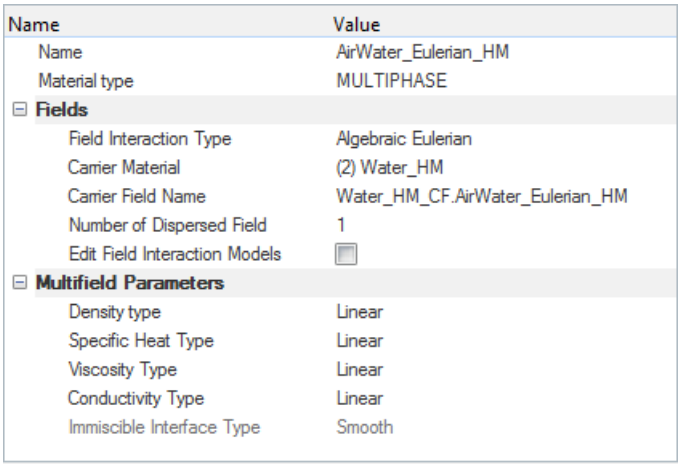


Figure 517:

5. Activate the check box beside **Edit Field Interaction Models**.
   The **Field Interaction Model** dialog opens.
6. In the dialog, set the Dispersed Field 1 Material to **Air_HM**, if not set already.

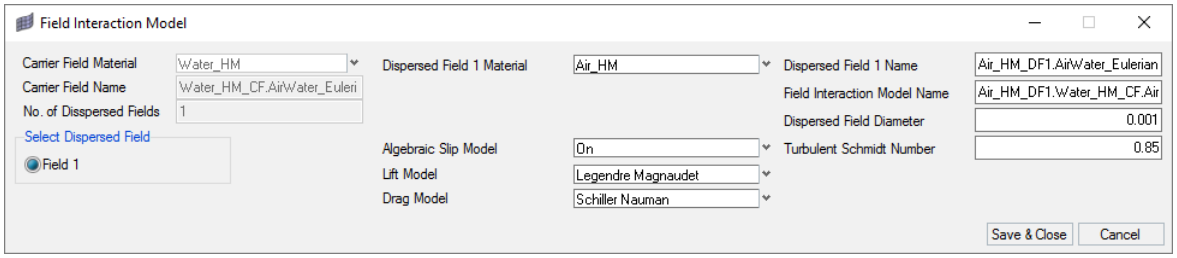


Figure 518:

7. Click **Save and Close**.

Set Up the Body Force

1. In the Solver Browser, expand **03.Body_Force** then expand **BODY_FORCE** and click on **Gravity_HM**.
2. In the Entity Editor, set the Z gravity to `0.0` and change the Y gravity to `-9.81` m/sec^2.

Name	Value
Name	Gravity_HM
Medium	Fluid
Gravity	
Gravity type	Constant
X ## (m/sec2)	0.0
Y ## (m/sec2)	-9.81
Z ## (m/sec2)	0.0
Gravity multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

Figure 519:

Set Up Boundary Conditions and Nodal Initial Conditions

Assign Boundary Conditions and Material Properties

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **MULTIPHASE**.
   b) Select **AirWater_Eulerian_HM** as the Material.
   c) Set Body force to **Gravity_HM**.

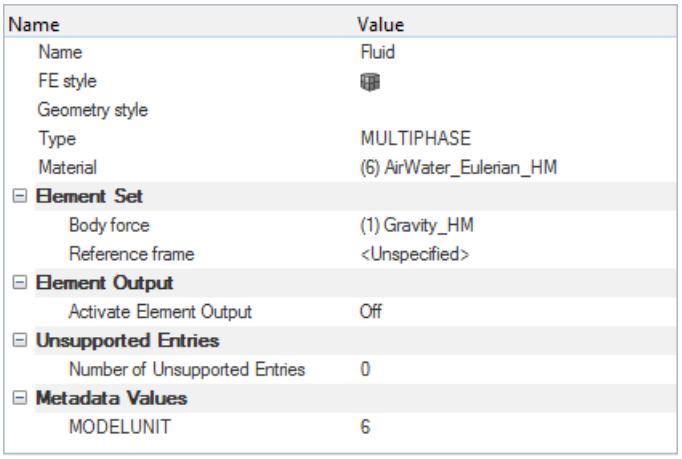


Figure 520:

3. Click **Inflow**. In the Entity Editor,

a) Change the Type to **INFLOW**.

b) Set the Inflow type to **Average velocity**.

c) Set the Average velocity to `1.0` m/s.

d) Set No. of multiphase incoming fields to `2.0` and press Enter on the keyboard.

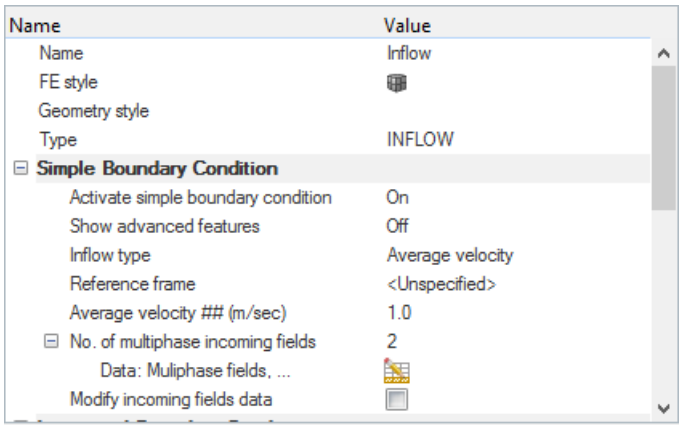


Figure 521:

The **Incoming Fields** dialog opens.

e) Select **Air_HM_DF1.AirWater_Eulerian_Hm** as the first Incoming Field and set Volume Fractions to `0.04`.

f) Select **Water_HM_CF.AirWater_Eulerian_Hm** as the second Incoming Field and set Volume Fraction to `0.96`.

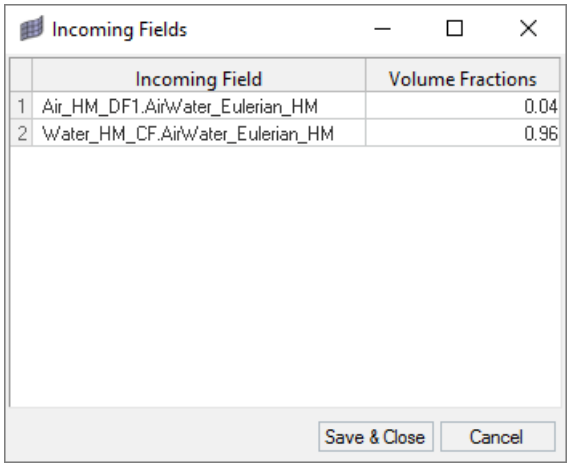


Figure 522:

g) Click **Save & Close**.

4. Click **Outflow**. In the Entity Editor, change the Type to **OUTFLOW**.

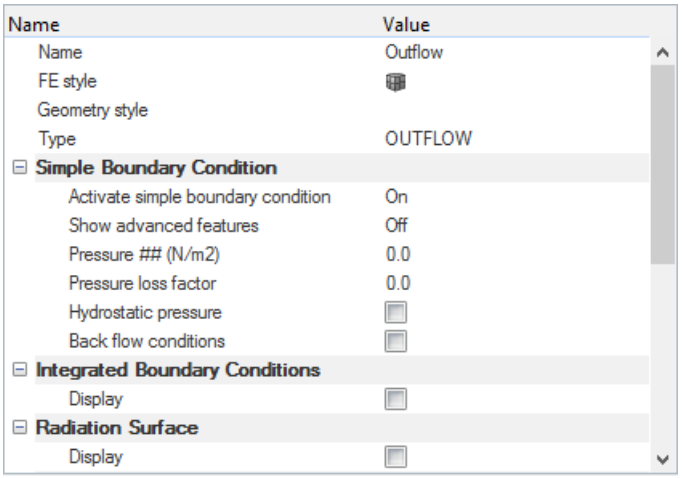


Figure 523:

5. Click **Wall**. In the Entity Editor, verify that the Type is set to **WALL**.

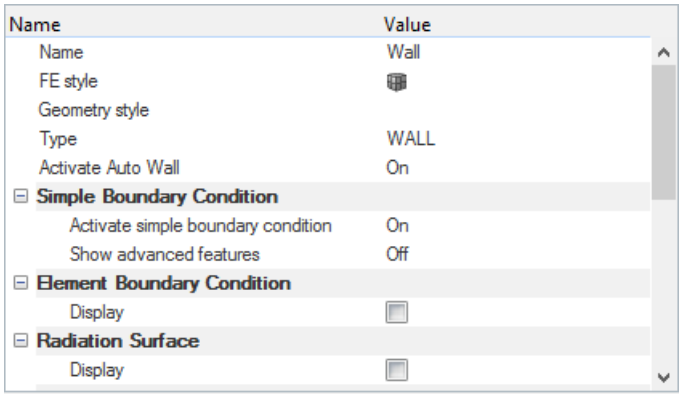


Figure 524:

6. Click **Slip_1**. In the Entity Editor, change the Type to **SLIP**.

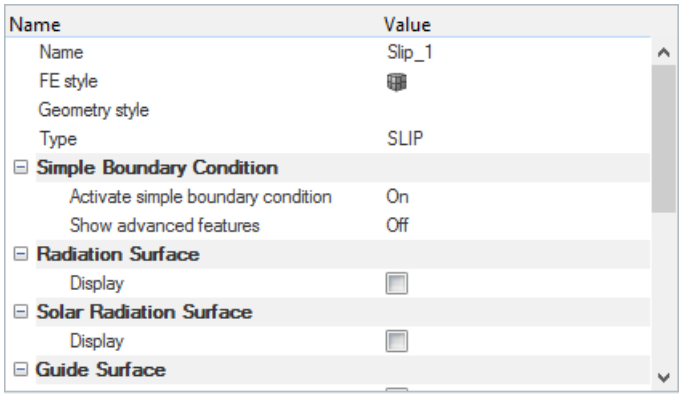


Figure 525:

7. Similarly, change the Type to **SLIP** for the Slip_2 component.
8. Save the model.

Set the Nodal Initial Conditions

1. In the Solver Browser, expand the **01.Global** tree then expand **03.NODAL_INITIAL_CONDITION** and click on **NODAL_INITIAL_CONDITION**.
2. In the Entity Editor, verify that the default value for Eddy viscosity is set to `0.0001` m^2/sec.
3. Under the Field tab, set the No. of Fields to `2` and press Enter.

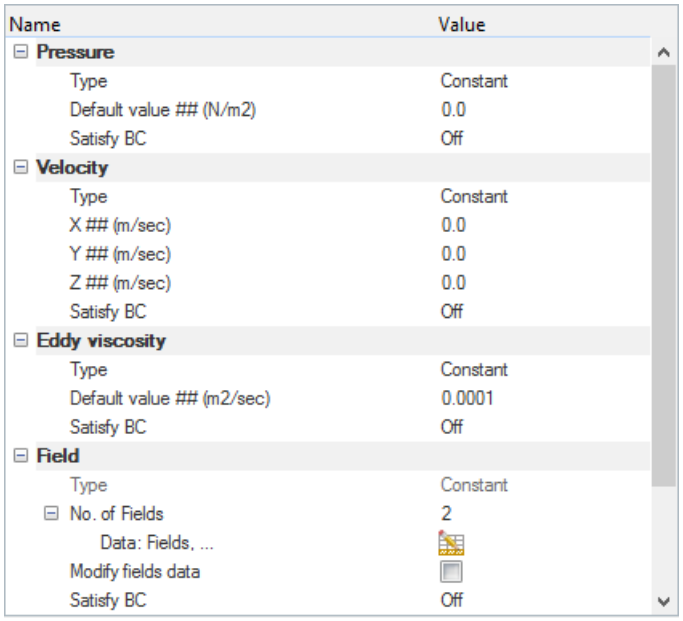


Figure 526:

The **Fields** dialog opens.

4. Set the fields as shown in the figure below.

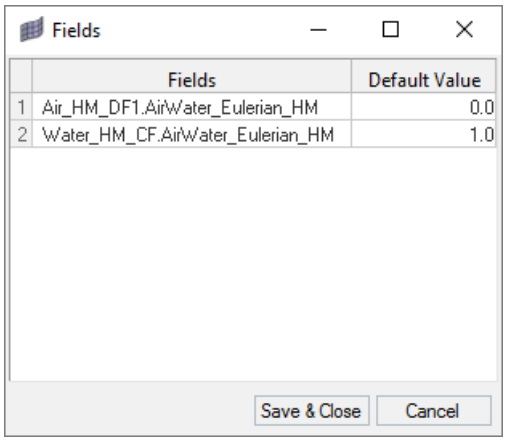


Figure 527:

5. Click **Save & Close**.
6. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to All or Final. Since this is a Transient analysis, set it to **All**.
5. Leave the remaining options as default and click **Launch** to start the solution process.

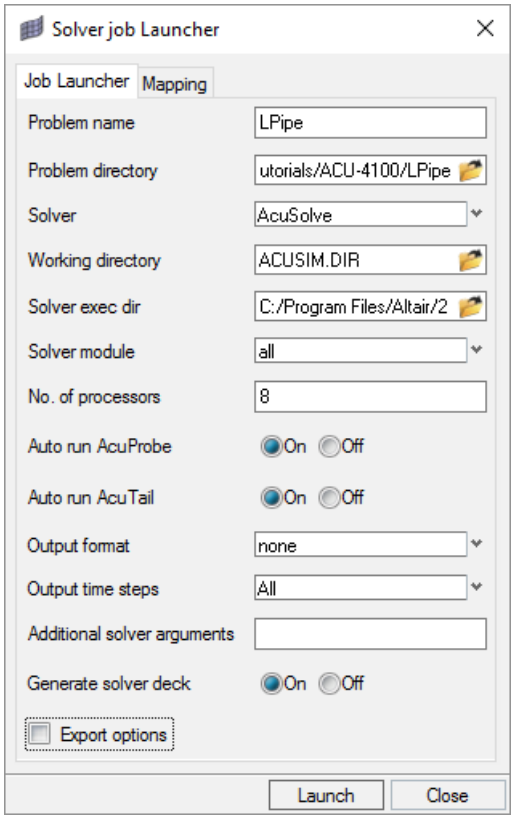


Figure 528:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.
   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.

2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `LPipe.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.
   The model is colored by geometry after loading.

Create Contours for Volume Fraction of Water

In this step, you will create a contour plot of the volume fraction of water at the end of the simulation.

1. In the Results Browser, set the Time to `1.5` sec.

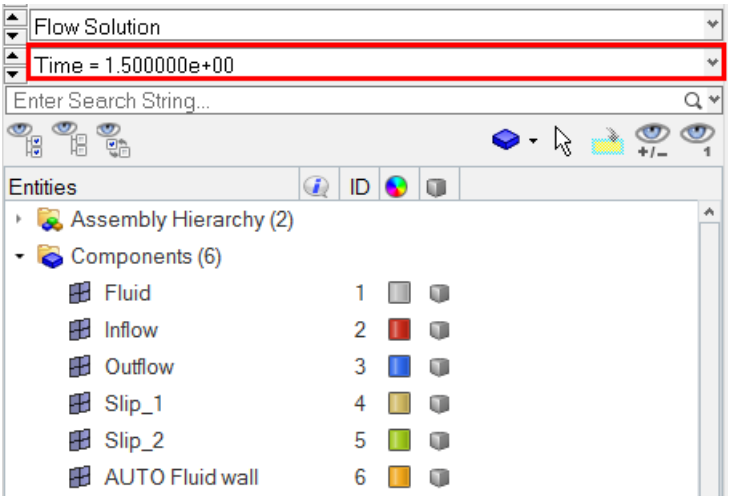


Figure 529:

2. In the Results Browser, expand the list of Components.
3. Click the Isolate shown icon and then click on the **Fluid** component to turn off the display of all components except the Fluid component.

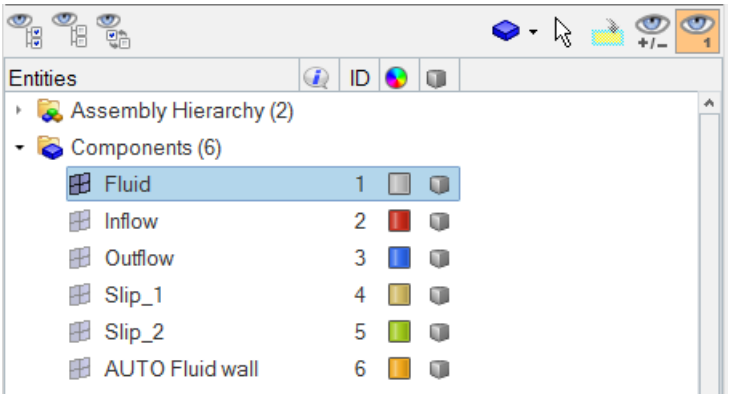


Figure 530:

4. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
5. Click on the Results toolbar to open the Contour panel.
6. From the top drop-down menu under the Result type, select **Volume_fraction-2-Water_HM_CF.AirWater_Eulerian(s)**.
7. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
8. In the panel area, click **Apply**.
9. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 531:

10. Click the **Legend** tab then click **Edit Legend**.
11. In the **Edit Legend** dialog, change the Numeric format to **Fixed**. In the Values section, click on the Minimum value in the legend and set it to `0.80`. Similarly, set the Maximum value to `1.0`.

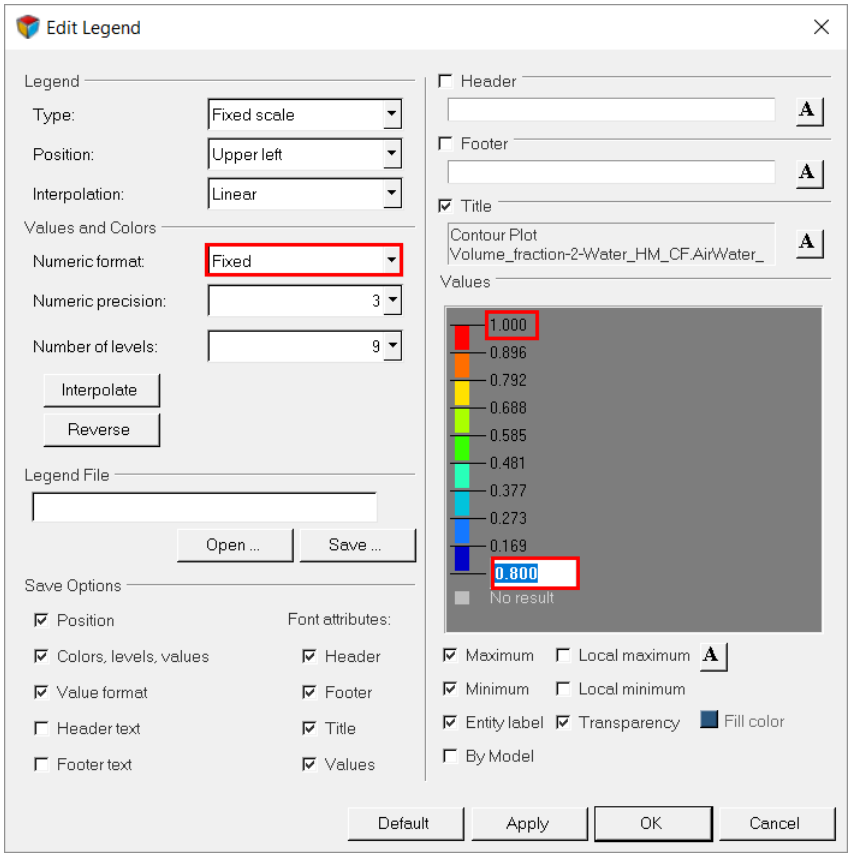


Figure 532:

12. Click **OK** to close the dialog.

13. Verify that the contour plot looks like the figure below at frame 16.

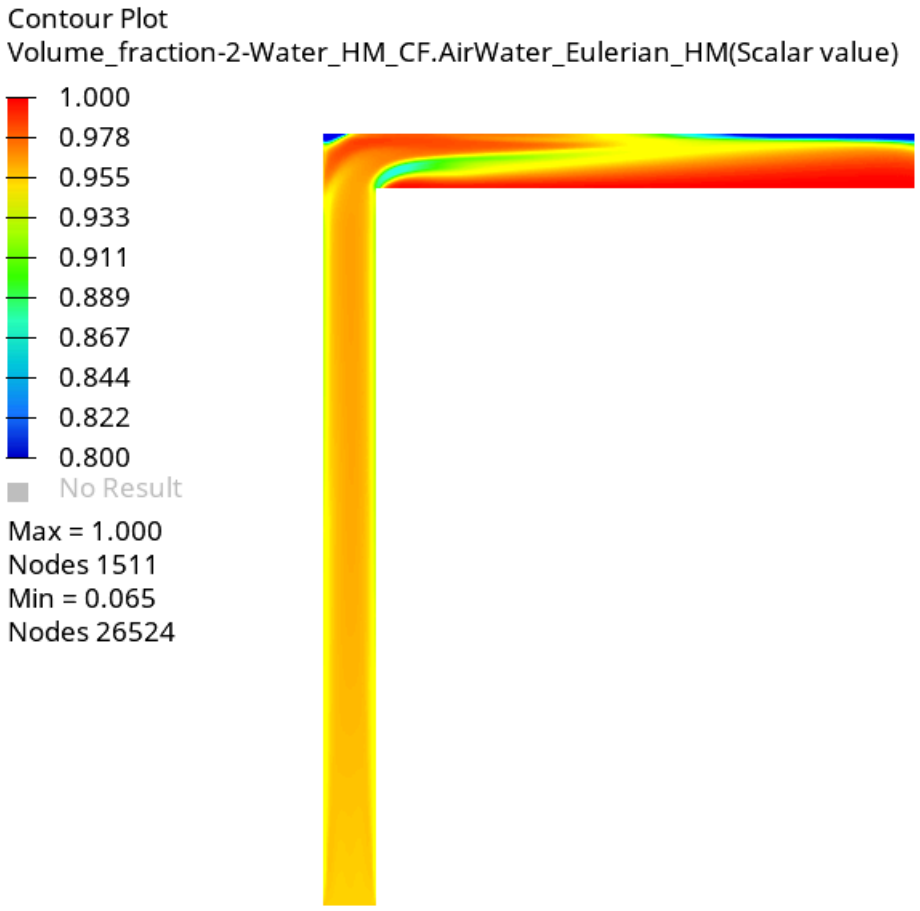


Figure 533:

Summary

In this tutorial, you worked through a basic workflow to set-up and solve a transient multiphase flow problem using the Algebraic Eulerian multiphase model using HyperWorks products, namely HyperMesh and AcuSolve. You started by importing the model in HyperMesh. Then, you defined the simulation parameters and launched AcuSolve directly from within HyperMesh. Upon completion of the solution by AcuSolve, you used HyperView to post-process the results and created a contour plot of the volume fraction.

2.4.5 ACU-T: 4200 Humidity – Pipe Junction

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T4200_Humidity.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 534. As an example, a pipe junction problem is attached here to show the capability of the Humidity modelling in AcuSolve. In this problem, there are two inlets with different flow, thermal, and humidity conditions. As the flow proceeds downstream of the pipe, two pipes merge into a single pipe to create a single outlet and a distinct profile of temperature and humidity is attained. The geometry is symmetric about the XZ midplane of the pipe, as shown in the figure.

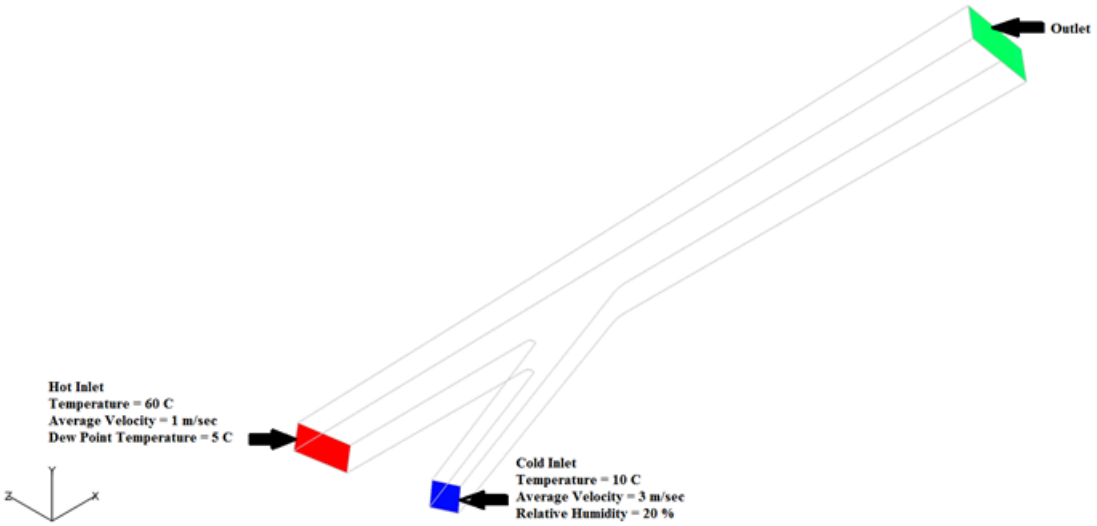


Figure 534:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.

2. Click the Open Model icon  located on the standard toolbar.
   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4200_Humidity.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `PipeJunction_Humidity` and navigate into this directory.
   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Humidity` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Optional: In the Entity Editor, change the Title to `Humidity Modeling`.
3. Set Abs. pressure offset to `101325` N/m^2.
4. Change the Temperature equation to **Advective Diffusive**.
5. Set the Turbulence model to **Spalart Allmaras**.
6. Switch Humid Air Model to **On**.
   This will automatically change the Multiphase equation to Advective Diffusive.

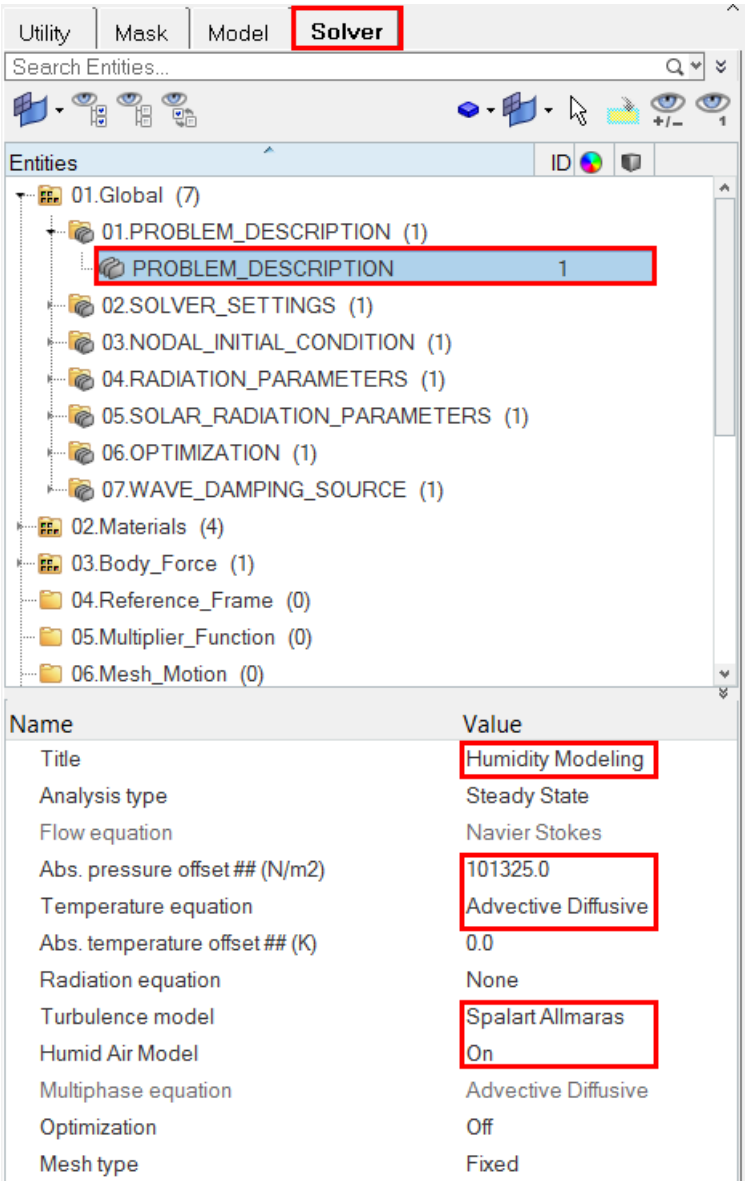


Figure 535:

7. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
8. Set Initial time increment to `1` sec.
9. Set the Relaxation factor to `0`.

Name	Value
Max time steps	100
Initial time increment ## (sec)	1.0
Convergence tolerance	0.001
Relaxation factor	0.0
Flow	On
Temperature	On
Temperature flow	Off
Turbulence	On
Field	On
External code	Off
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 536:

Set Up Body Force Parameters

In this step, you will define the body force.

1. In the Solver Browser, expand the **03.Body_Force** tree then select **BODY_FORCE** > **Gravity_HM**.
2. Set Gravity in the Y direction to `-9.81` m/sec^2 and change the Z direction to `0`.

Name	Value
Name	Gravity_HM
Medium	Fluid
Gravity	
Gravity type	Constant
X ## (m/sec2)	0.0
Y ## (m/sec2)	-9.81
Z ## (m/sec2)	0.0
Gravity multiplier function	<Unspecified>
Heat Source	
Heat source unit type	Per unit mass
Heat source type	Constant
Mass heat source ## (W/kg)	0.0
Heat source multiplier function	<Unspecified>
Unsupported Entries	
No of Body Force Unsupported Entries	0
No of Gravity Unsupported Entries	0
No of Mass Heat Source Unsupported Entries	0
No of Volume Heat Source Unsupported Entries	0

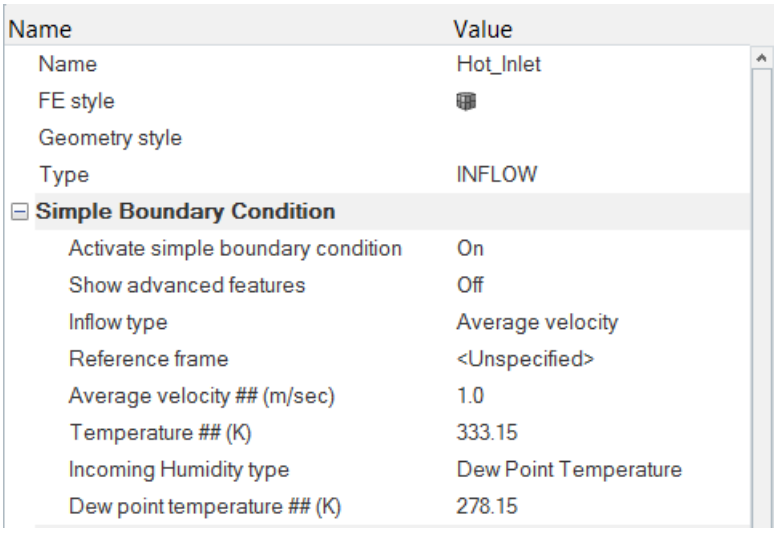
Figure 537:

Set Up Boundary Conditions and Nodal Initial Conditions

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Hot_Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow type to **Average velocity**.
   c) Set the Average velocity to `1` m/s.
   d) Set the Temperature to `333.15` K.
   e) Set Incoming Humidity type to **Dew Point Temperature** and set the value to `278.15` K.



Name	Value
Name	Hot_Inlet
FE style	
Geometry style	
Type	INFLOW
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Inflow type	Average velocity
Reference frame	<Unspecified>
Average velocity ## (m/sec)	1.0
Temperature ## (K)	333.15
Incoming Humidity type	Dew Point Temperature
Dew point temperature ## (K)	278.15

Figure 538:

3. Click **Cold_Inlet**. In the Entity Editor,

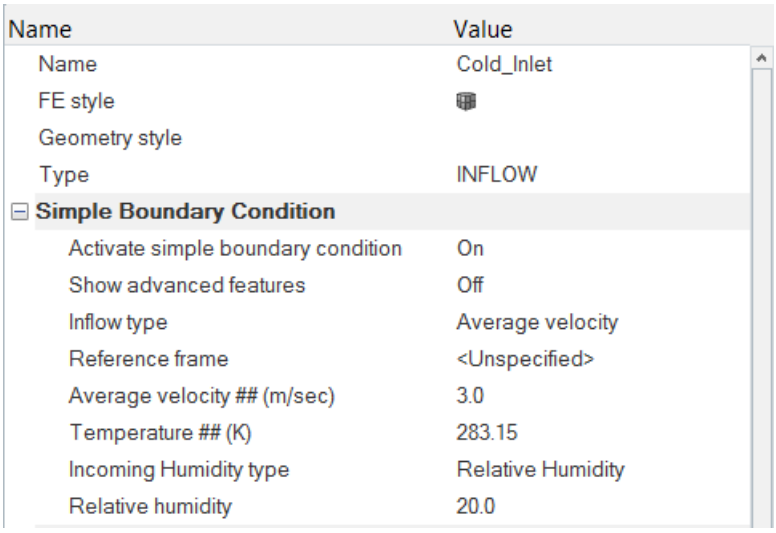
a) Change the Type to **INFLOW**.

b) Set the Inflow type to **Average velocity**.

c) Set the Average velocity to `3` m/s.

d) Set the Temperature to `283.15` K.

e) Set Incoming Humidity type to **Relative Humidity** and set the value to `20`.



Name	Value
Name	Cold_Inlet
FE style	
Geometry style	
Type	INFLOW
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Inflow type	Average velocity
Reference frame	<Unspecified>
Average velocity ## (m/sec)	3.0
Temperature ## (K)	283.15
Incoming Humidity type	Relative Humidity
Relative humidity	20.0

Figure 539:

4. Click **Outlet**. In the Entity Editor, change the type to **OUTFLOW**.

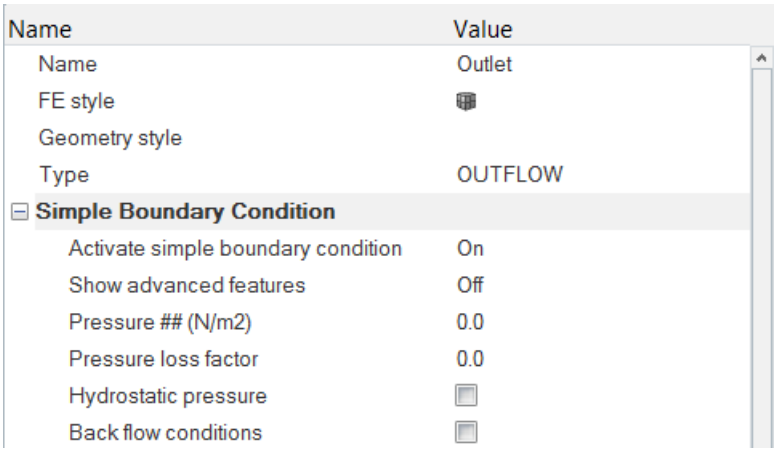


Figure 540:

5. Click **plusY**. In the Entity Editor, change the type to **SLIP**.

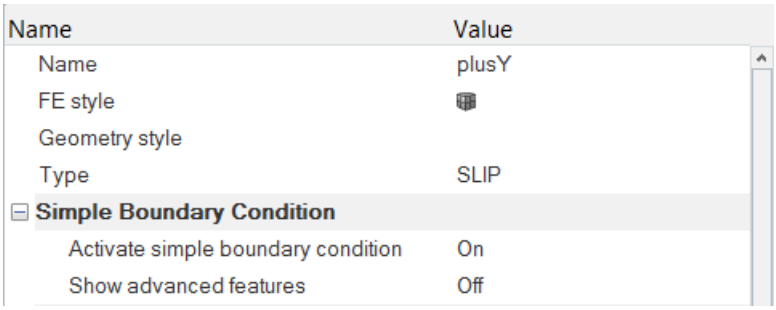


Figure 541:

6. Similarly click **minusY** and change the Type to **SLIP**.
7. Click **Walls**. In the Entity Editor, verify that the Type is set to **WALL**.

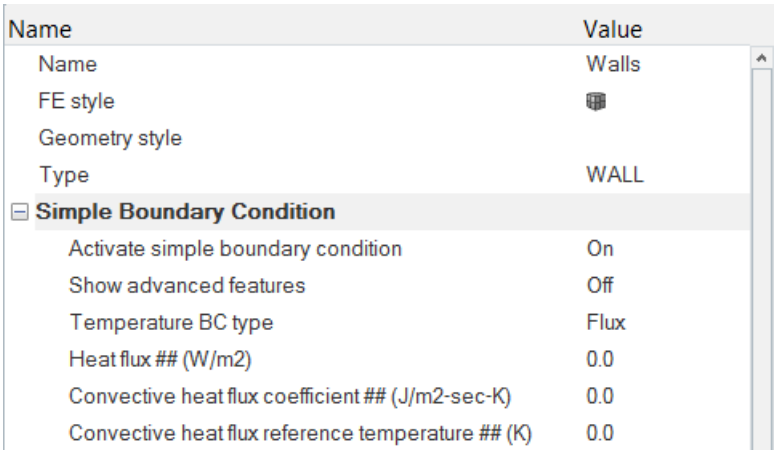


Figure 542:

8. Click **Fluid**. In the Entity Editor,

a) Change the Type to **MULTIPHASE**.
b) Select **HumidAir_HM** as the Material.
c) Set Body force to **Gravity_HM**.

Name	Value
Name	Fluid
FE style	
Geometry style	
Type	MULTIPHASE
Material	(4) HumidAir_HM
Element Set	
Body force	(1) Gravity_HM
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 543:

9. Save the model.

Set the Nodal Initial Conditions

1. Go to the Solver Browser, expand the **01.Global** tree then select **03.NODAL_INTIAL_CONDITION** > **NODAL_INTIAL_CONDITION**.
2. Under the Velocity tab, set X Velocity to `1` m/sec.
3. Change the Default value of Temperature to `333.15` K.
4. Change the Default value of Eddy viscosity to `0` m^2/sec.
5. Change the Default value of Relative Humidity to `20`.

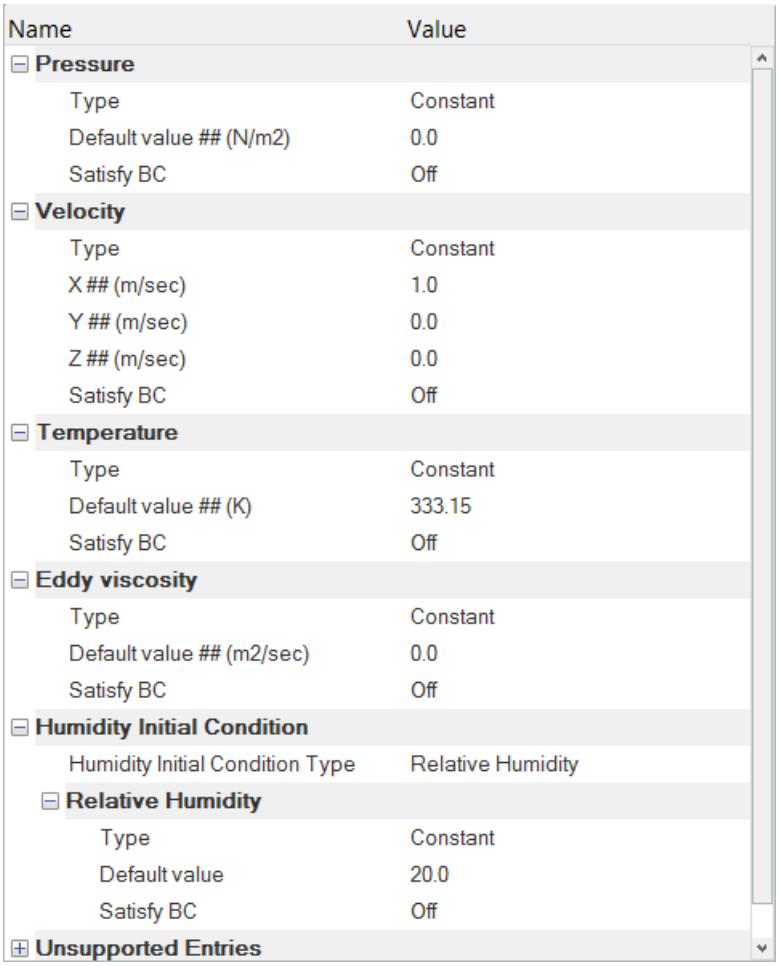


Figure 544:

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Leave the remaining options as default and click **Launch** to start the solution process.

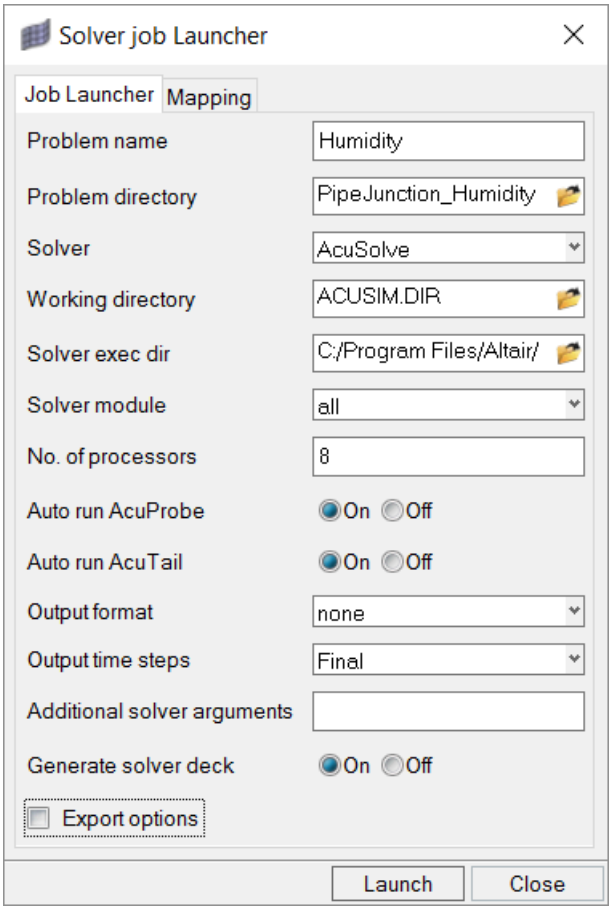


Figure 545:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `PipeJunction.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

The model is colored by geometry after loading.

Create Contour Plots

In this step, you will create contour plots of temperature, relative humidity, mass fraction humidity and velocity magnitude.

1. In the Results Browser, expand the list of Components.
2. Click the Isolate shown icon and then click on the **minusY** component to turn off the display of all components except the minusY component.

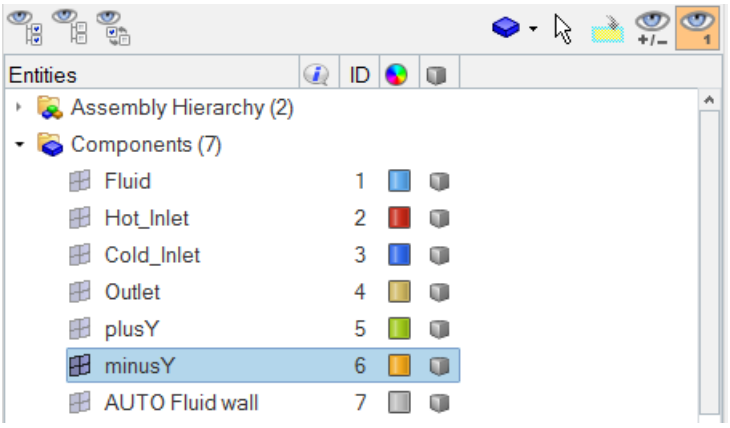


Figure 546:

3. Orient the display to the xz-plane by clicking on the Standard Views toolbar.
4. Click on the Results toolbar to open the Contour panel.
5. Select **Temperature (s)** as the Result type.
6. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
7. Click **Apply**.
8. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 547:

9. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.
10. Verify that the contour looks like the figure below.

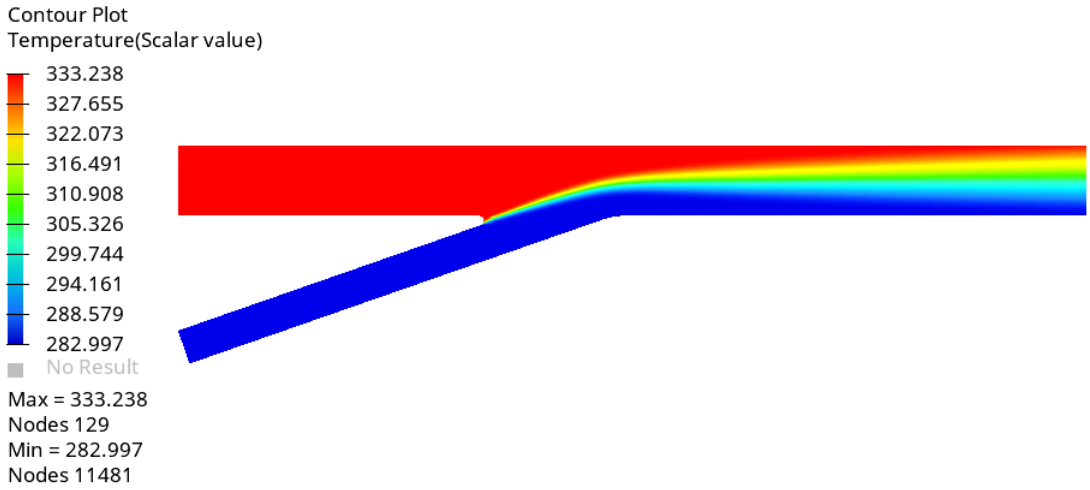


Figure 548:

11. Change the Result type to **Relative_humidity (v)** then click **Apply** to view the relative humidity contour on the minus-Y plane.

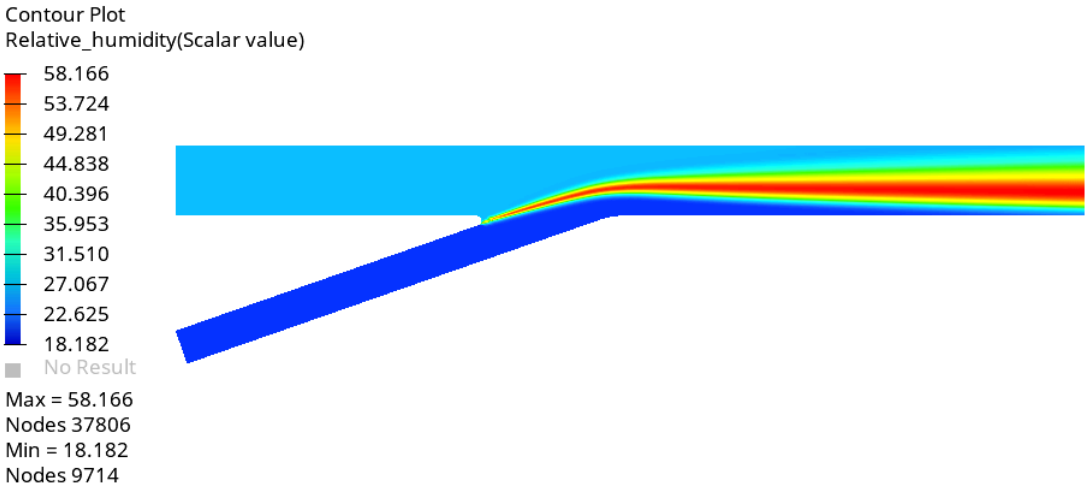


Figure 549:

12. Change the Result type to **Mass_fraction-1-Humidity_HM.HumidAir(s)** then click **Apply**. Use the range 0.001306 to 0.0406.

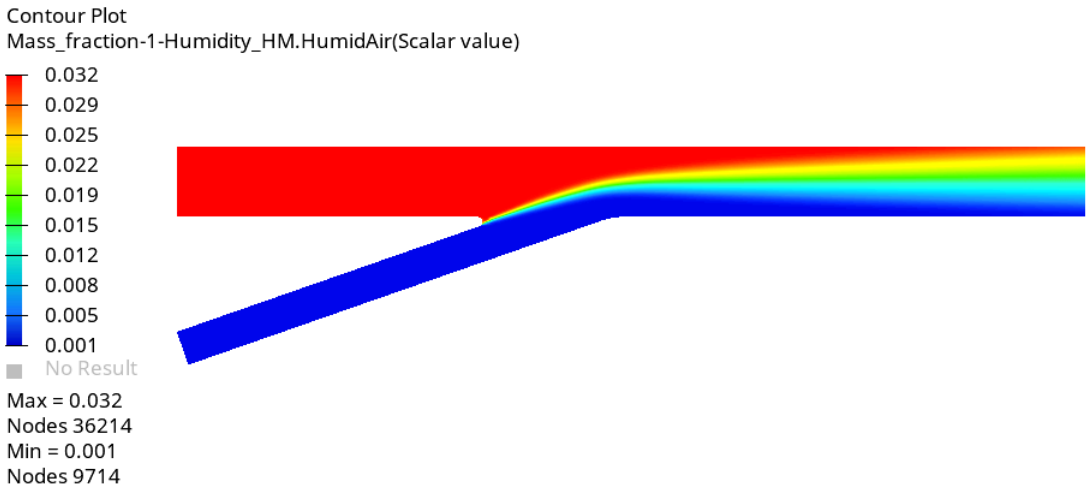


Figure 550:

13. Change the Result type to **Velocity (v)** then click **Apply**.

Use the range 0 to 3.753.

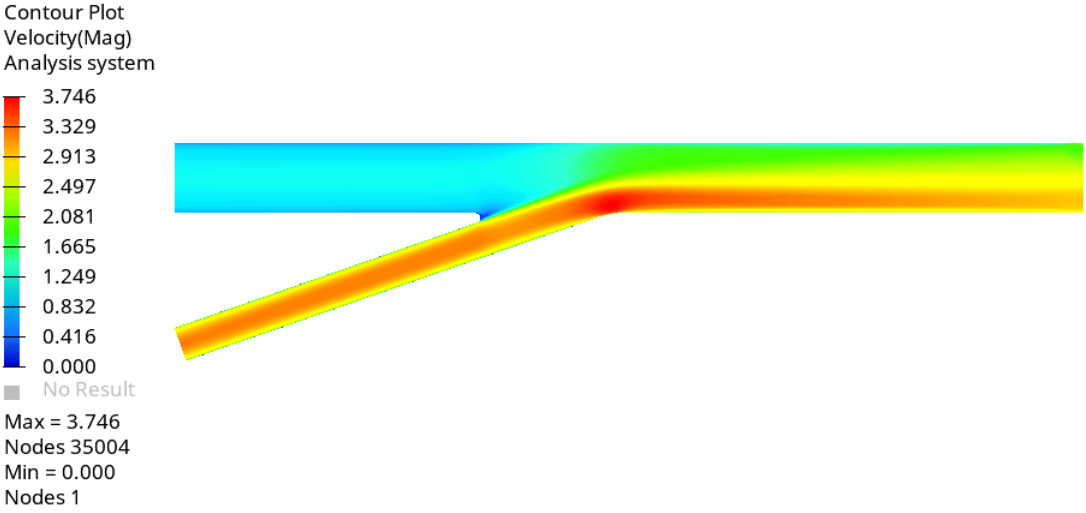


Figure 551:

Summary

In this tutorial, you worked through a basic workflow to set-up a CFD model, carried out a CFD simulation, and then post-processed the results using HyperWorks products, namely AcuSolve, HyperMesh and HyperView. You started by importing the model in HyperMesh. Then, you defined the simulation parameters and launched AcuSolve directly from within HyperMesh. Upon completion of the solution by AcuSolve, you used HyperView to post-process the results and create contour plots.

2.4.6 ACU-T: 4201 Condensation & Evaporation - Air Box

Prerequisites

This tutorial provides instructions for running a transient simulation of an enclosed air-box using the humidity model. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T4201_Air_Box.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 552. As an example, an enclosed air-box problem is attached here to show the capability of the humidity modelling (condensation and evaporation) using AcuSolve. The main goal is to demonstrate and quantify accumulation and loss of water vapor on the bottom wall surface due to temperature change in an air domain at 70% relative humidity.

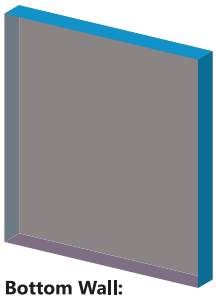


Figure 552:

In this particular domain, we assume that air is at a certain level of humidity. Condensation, evaporation, relative humidity, and temperature are all associated with dew point temperature. When the surface temperature drops below the dew point temperature, that is when condensation should start accumulating, and then accumulate up to a point where it reaches 100%. And again on the surface as the temperature rises and attains dew point again, it starts evaporating. This whole definition of both condensation and evaporation is explained in the below attached figure.

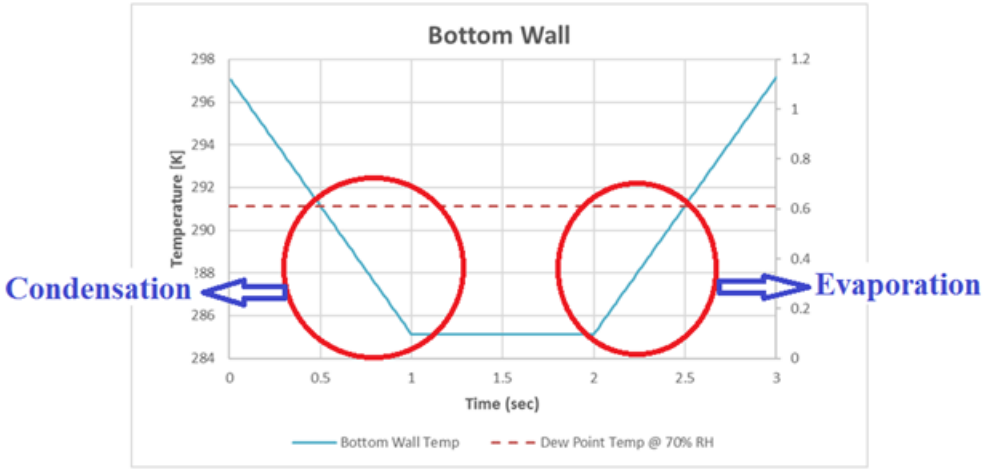


Figure 553:

From the above plot, we can see that the Air volume initial temperature is set to 297.15 K. The Bottom Wall temperature drops to 285.13 K over 1 sec, maintains 285.13 K for 1 sec, and then rises back to 297.15 K over 1 sec. The dew point temperature of the air at 70% RH is 291.14 K and is reached at 0.5 and 2.5 sec. On the whole we can see that both condensation and evaporation occurs when the dew point temperature is reached, as explained above.

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T4201_Air_Box.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Air_Box_Humidity` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Air_Box` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.

2. In the Entity Editor, enter `Air Box` as the Title.
3. Set the Analysis type to **Transient**.
4. Set Abs. pressure offset to `101325`.
5. Change the Temperature equation to **Advective Diffusive**.
6. Set the Turbulence Model to **Spalart Allmaras**.
7. Switch the Humid Air Model to **On**.
   The will automatically change the multiphase equation to **Advective Diffusive**.
8. Set the Global Gravity parameters as shown below.

Name	Value
Title	Air Box
Analysis type	Transient
Flow equation	Navier Stokes
Abs. pressure offset ## (N/m2)	101325.0
Temperature equation	Advective Diffusive
Abs. temperature offset ## (K)	0.0
Radiation equation	None
Turbulence model	Spalart Allmaras
Humid Air Model	On
Multiphase equation	Advective Diffusive
Optimization	Off
Mesh type	Fixed
Nucleate boiling single phase	Off
External code	Off
Global Gravity	
X ## (m/sec2)	0.0
Y ## (m/sec2)	-9.81
Z ## (m/sec2)	0.0

Figure 554:

9. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
10. In the Entity Editor, set Max time steps to `100`.
11. Set the Initial time increment to `0.1` and change other parameters as shown in the figure below.

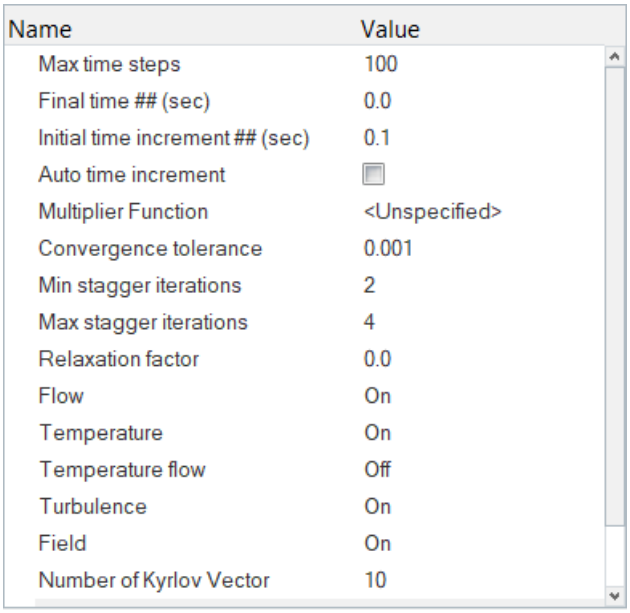


Figure 555:

Set Up Nodal Initial Conditions

In this step, you will define the Nodal Initial Condition (NICs) for the problem for both Temperature and Relative Humidity.

1. In the Solver Browser, click **03.NODAL_INTIAL_CONDITION** under **01.Global**.
2. Change the Default value of Temperature to `297.15` K.
3. Change the Default value of Eddy viscosity to `0.0001` m^2/sec.
4. Change the Default value of Relative Humidity to `70`.

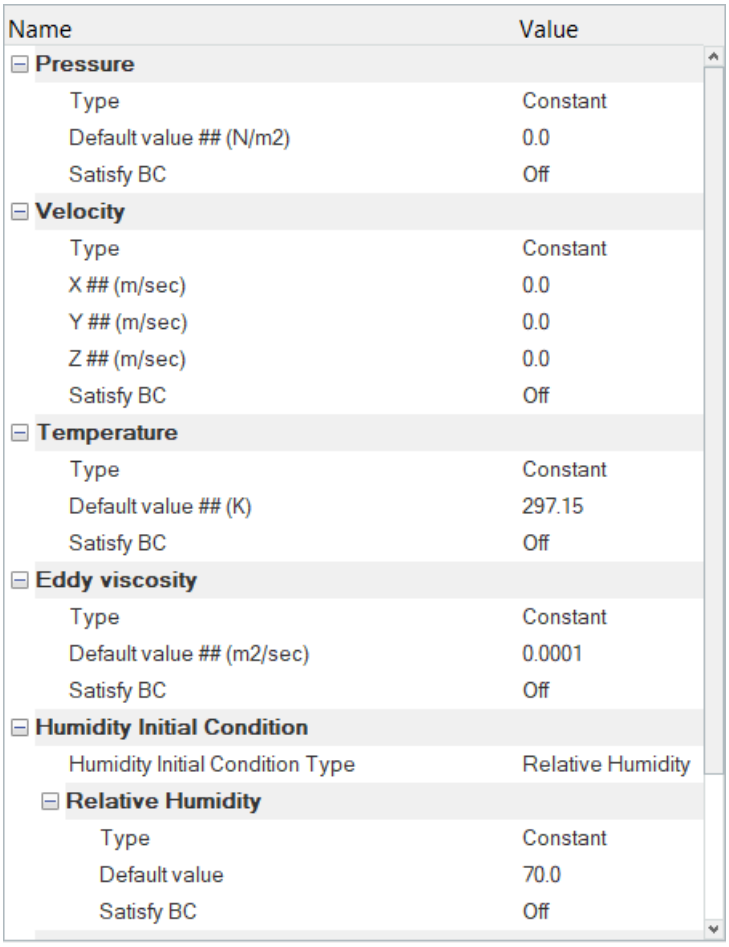


Figure 556:

Set Up Material Model Parameters and Body Force

In this step, you will define the material properties for the problem and assign body force to the fluid volume.

Set Material Model Parameters

1. In the Solver Browser, expand **02.Materials** > **FLUID** then click **Air_HM**.
2. In the Entity Editor, set the Material Type to **FLUID** if not already set.
3. Set the Density type to **Boussinesq**.
4. Enter the values for Density as shown in the figure below and leave the remaining material parameters unchanged.

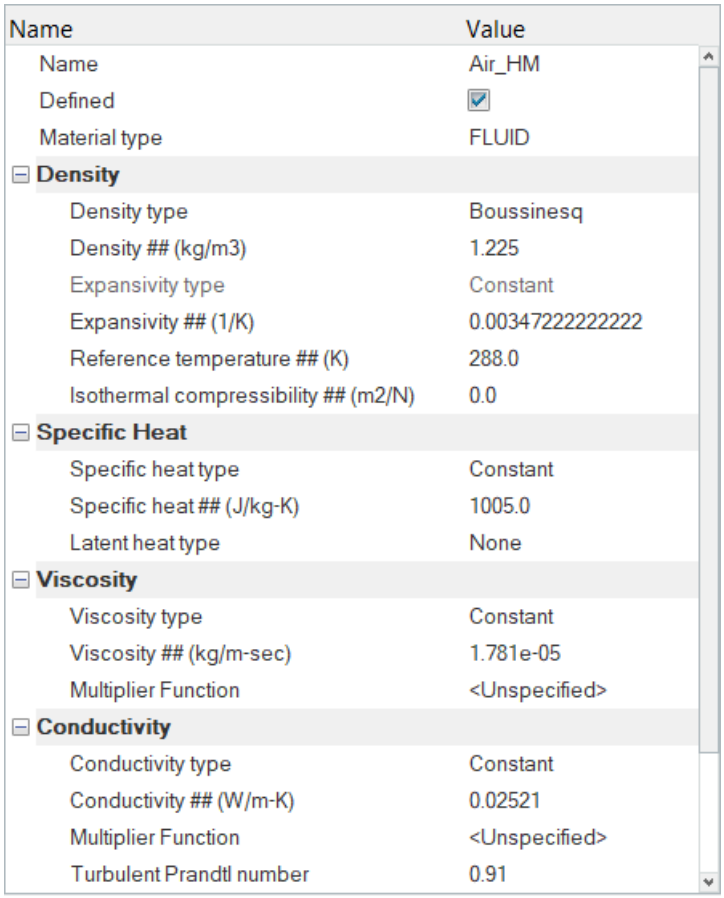


Figure 557:

5. In the Solver Browser, expand **MULTIPHASE** under Materials then click **HumidAir_HM**.
6. Verify that the Material type is set to **MULTIPHASE** and the Filed Interaction Type is set to **Advective Diffusive**.

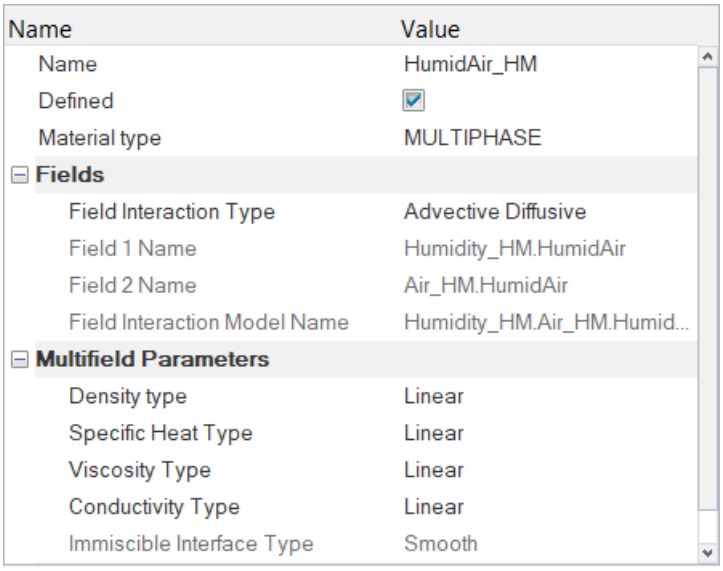


Figure 558:

Set Up Body Force

1. In the Solver Browser, expand **03.Body_Force** > **BODY_FORCE** then click **Gravity_HM**.
2. Set Gravity in the Z direction to `-9.81`.

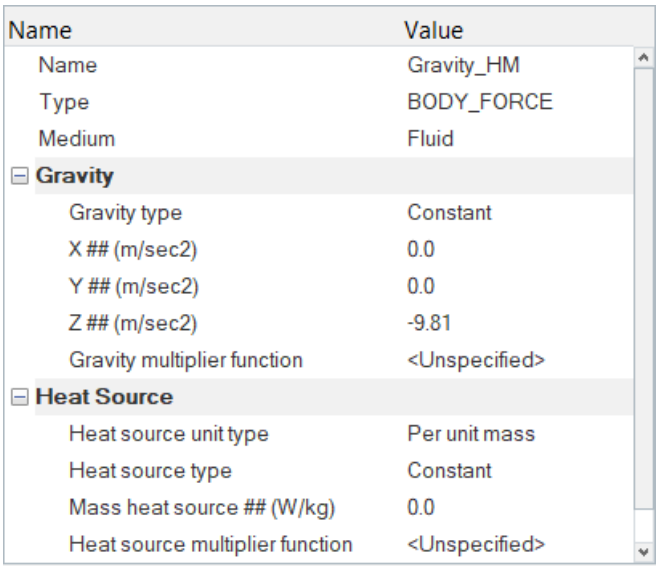


Figure 559:

Create a Multiplier Function

1. In the Solver Browser, right-click on **05.Multiplier_Function** and select **Create**.

A new function named MULTIPLIER_FUNCTION.1 is created.

2. Rename the function as `Temp_MF`.
3. In the Entity Editor, chance the Multiplier Function Type to **Piecewise Linear**.
4. Right-click on the **Curves** field and select **Create**.
5. In the embedded Entity Editor, change the name to `curve_Temp_MF`.

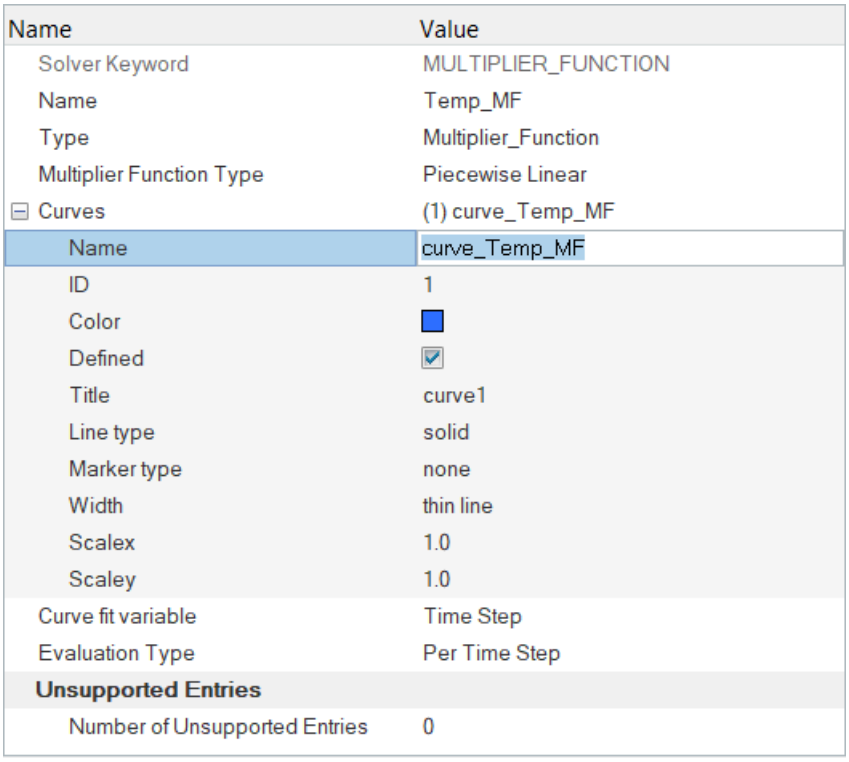


Figure 560:

6. Again, right-click on the **Curves** field and select **Plot Curves**.
   A **Curve editor** window opens.
7. Enter the data as shown in the figure below then click **Update** in the bottom-left of the window.

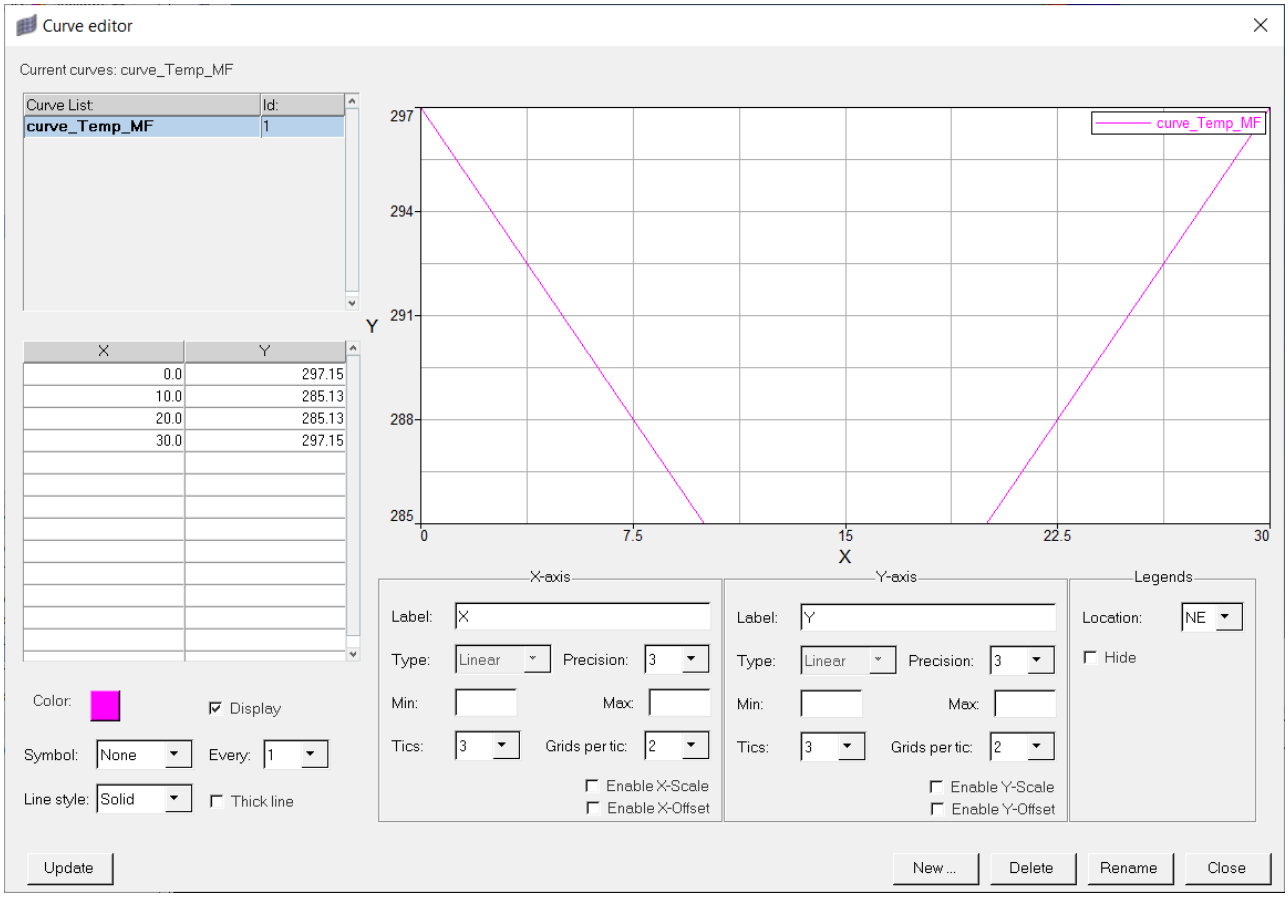


Figure 561:

8. Click **Close** to close the window.

Set Up Boundary Conditions

1. Go to the Model Browser and expand the list of **Components**.
2. Click **Fluid**. In the Entity Editor,
   a) Set the Type to **MULTIPHASE**.
   b) Set the Material to **HumidAir_HM**.
   c) Under Element Output, set Active Element Output to **On** and verify that the other parameters are as shown below.

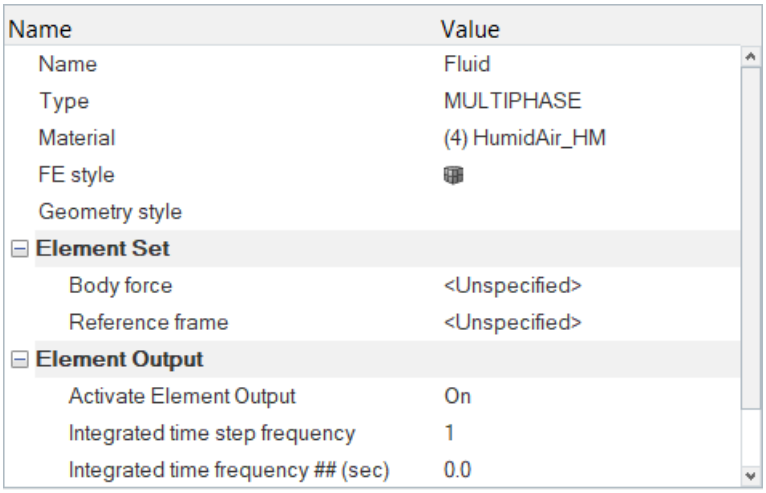


Figure 562:

3. Click **front**. In the Entity Editor, change the Type to **SLIP**.

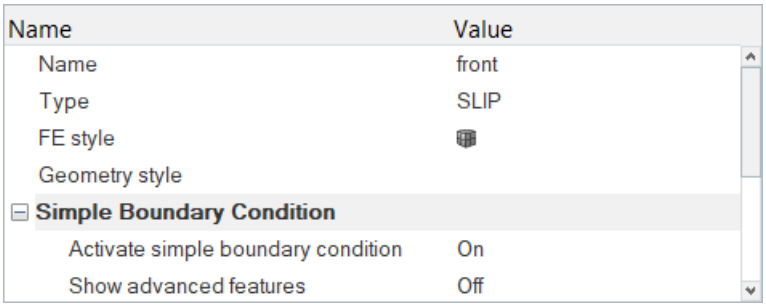


Figure 563:

4. Similarly, click **back** and change the Type to **SLIP**.
5. Click **left**. In the Entity Editor,
   a) Change the Type to **WALL**.
   b) Change the Temperature BC type to **Flux**.

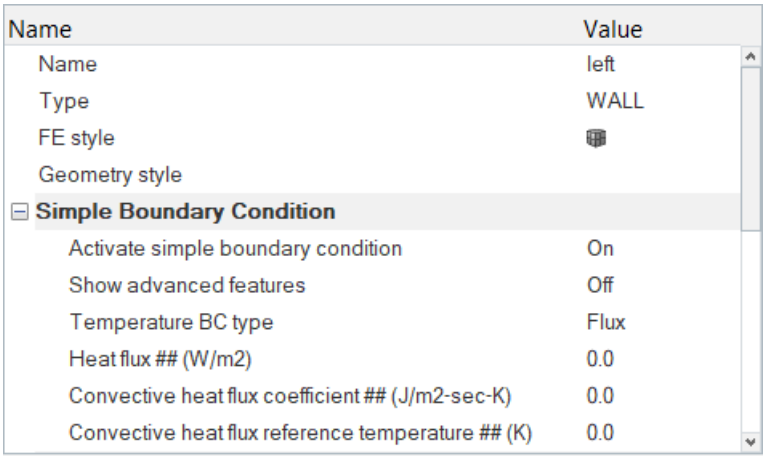


Figure 564:

6. Similarly, click **right** and **top**, change their types to **WALL**, and set the Temperature BC type to **Flux**.
7. Click **bottom**. In the Entity Editor,
   a) Change the Type to **WALL**.
   b) Set Show advanced features to **On**.
   c) Change the Temperature BC type to **Value**.
   d) Set the Temperature value to `1`.
   e) Set the Temperature multiplier function to **Temp_MF**.
   f) Under Surface Output, ensure that Activate Surface Output is **On**.

   Note: Turning this options on allows you to calculate the surface integral output of a particular surface using post-processing programs like AcuProbe and AcuTrans.

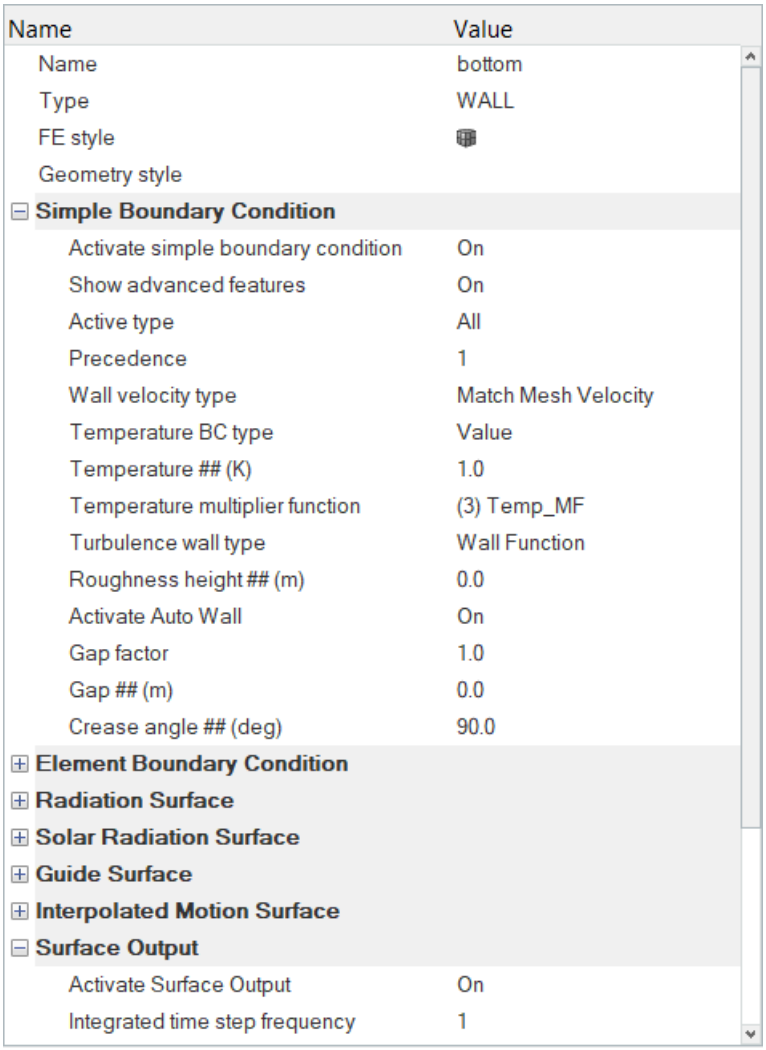


Figure 565:

8. Save the model.

Set Up Nodal Output Variables

1. In the Solver Browser, expand **17.Output** then click **NODAL_OUTPUT**.
2. In the Entity Editor, set the Time step frequency to `5`.
3. Check the box for **Output initial condition**.

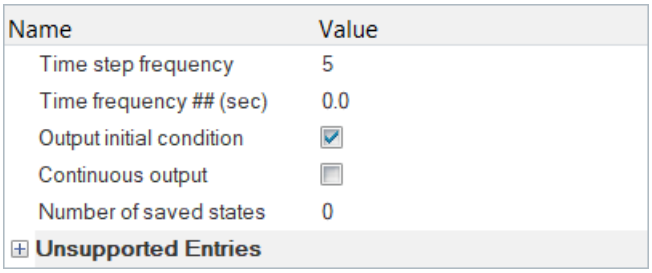


Figure 566:

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Verify that Auto run AcuProbe is **On**.
   This will open an AcuProbe dialog which will let you monitor the solution progress.
5. Verify that the Output time steps is set to **All**.
6. Leave the remaining options as default and click **Launch** to start the solution process.

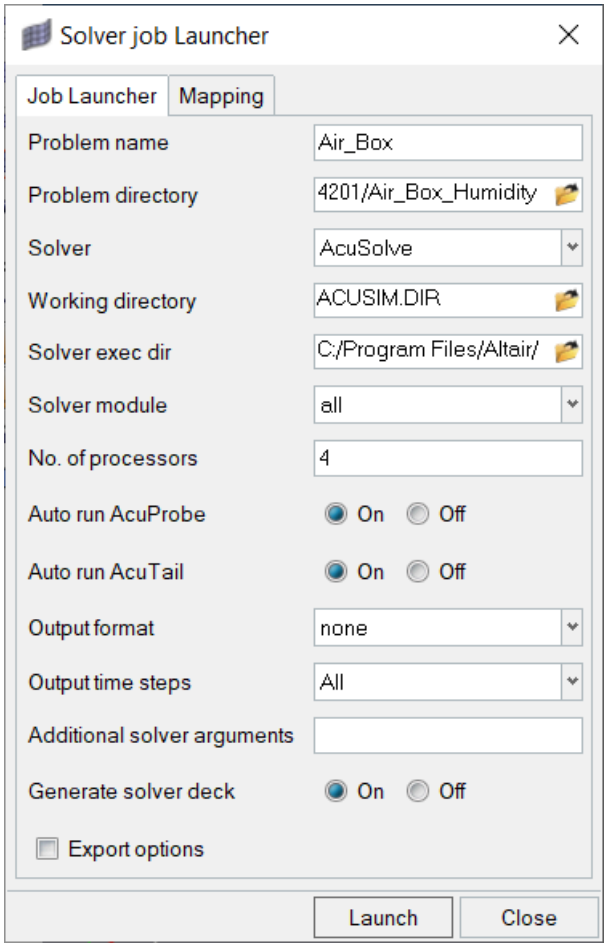


Figure 567:

As the solution progresses, an AcuTail and an AcuProbe dialog will open. Solution progress is reported in the AcuTail dialog. An AcuSolve Control dialog will also open from which you can control the solution process. In this dialog you have options to stop the solution or generate the output files at the end of the current time step.

A summary of the run printed in the AcuTail dialog indicates that AcuSolve has finished running the solution.

Monitor the Solution with AcuProbe

AcuProbe can be used to monitor various variables over solution time.

1. In the AcuProbe dialog, expand **Residual Ratio**.
2. Right-click on **Final** and select **Plot All**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

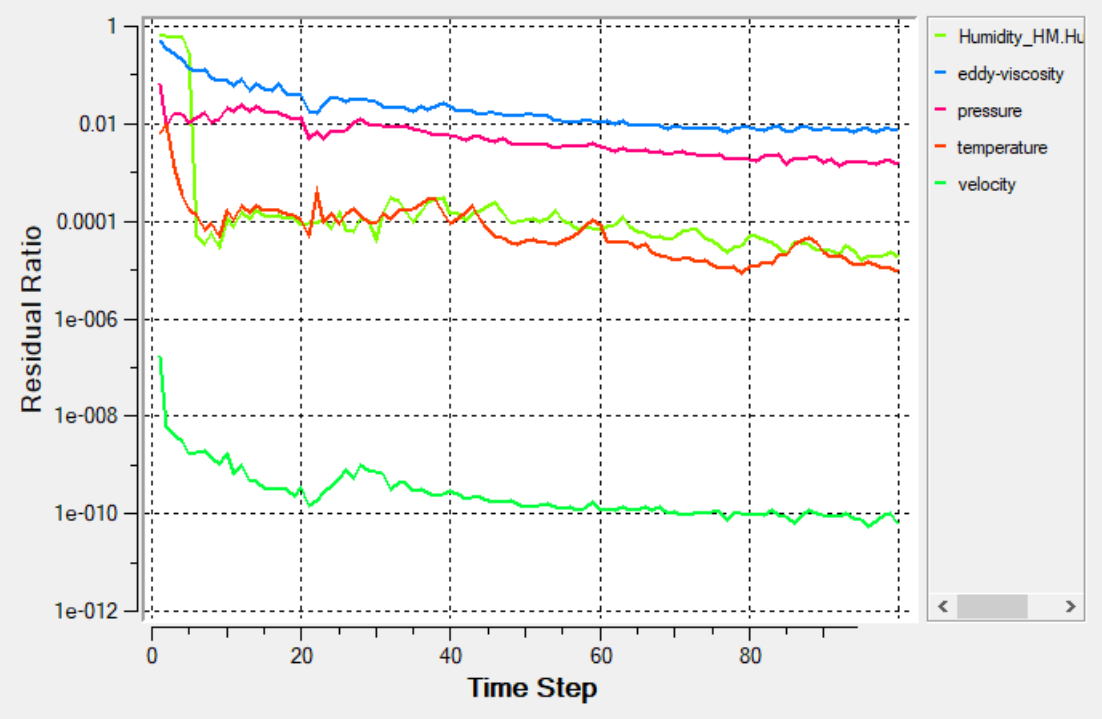


Figure 568:

3. Right click on **Final** and select **Plot None**.
4. Expand **Surface Output** > **bottom** > **Temperature**.
5. Right-click on **temperature** and select **Plot**.
6. Next, expand **Relative Humidity** then right-click on **relative_humidity** and select **Plot**.

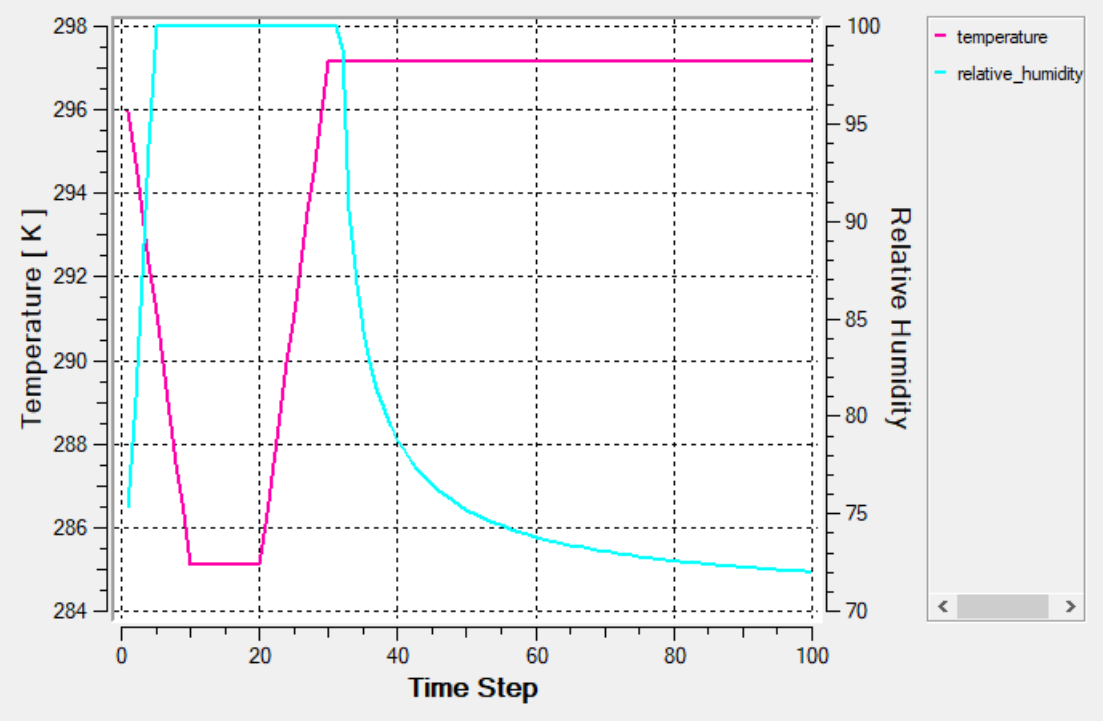


Figure 569:

7. Similarly, plot other variables like **temperature** vs **dewpoint_temperature**, as shown in below figure.

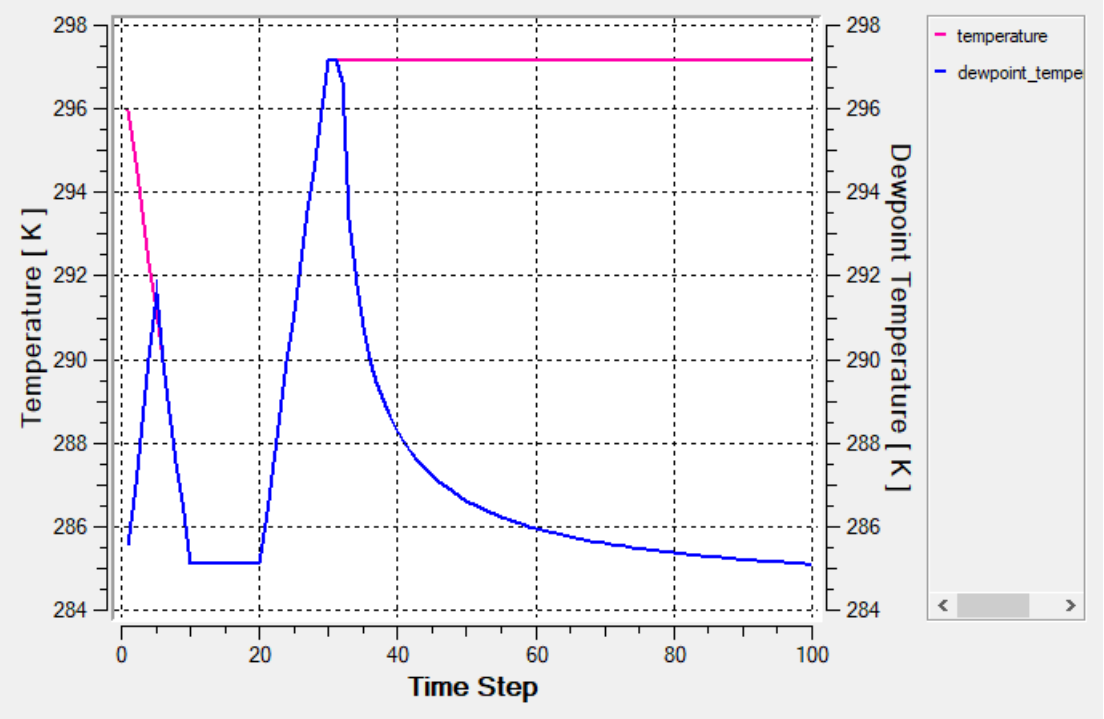


Figure 570:

8. Now, plot **temperature** vs **humidity_film_thickness**, as shown in below figure.

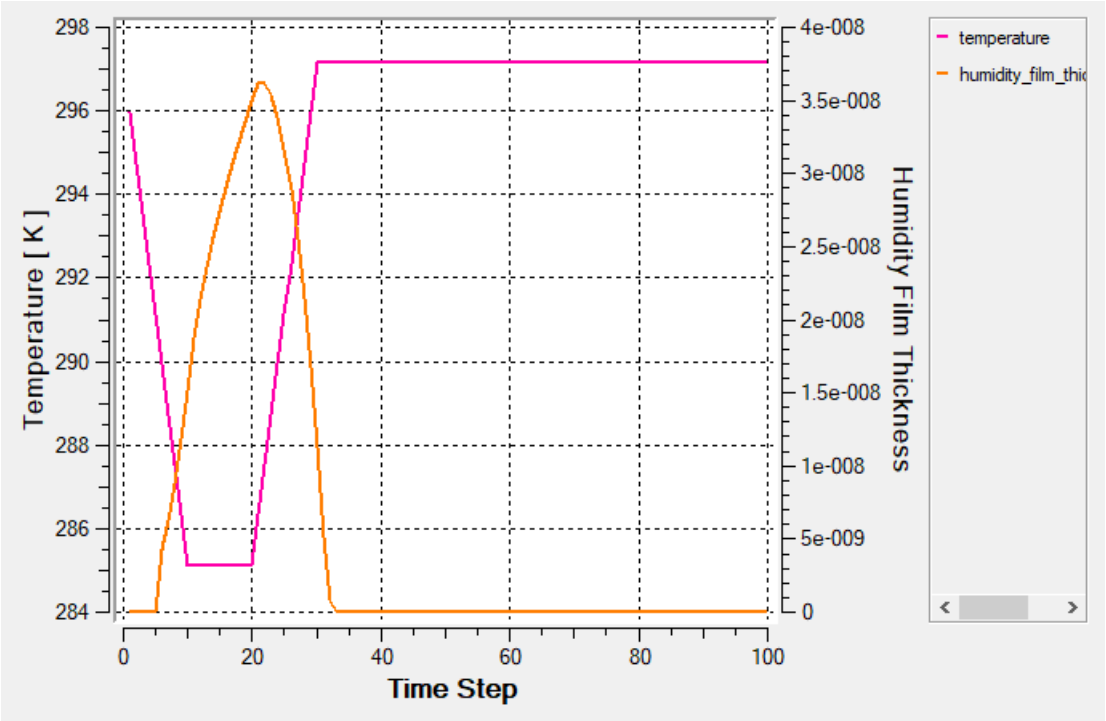


Figure 571:

9. Finally, plot **relative_humidity** vs **humidity_film_thickness**, as shown in below figure.

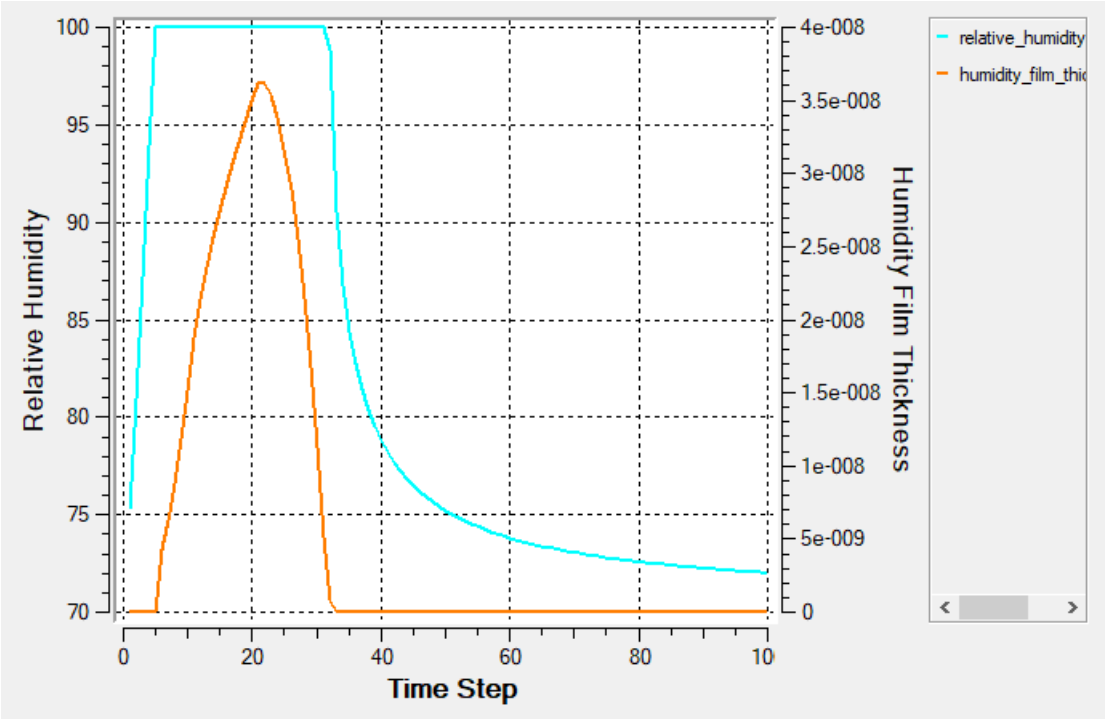


Figure 572:

Post-Process the Results with HyperView

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Air_Box.1.Log`.
4. Click **Open**.
5. In the panel area, click **Reader Options**.
6. In the dialog, set the Reader to **AcuSolve Result Reader** and Extended nodal output to **No**.

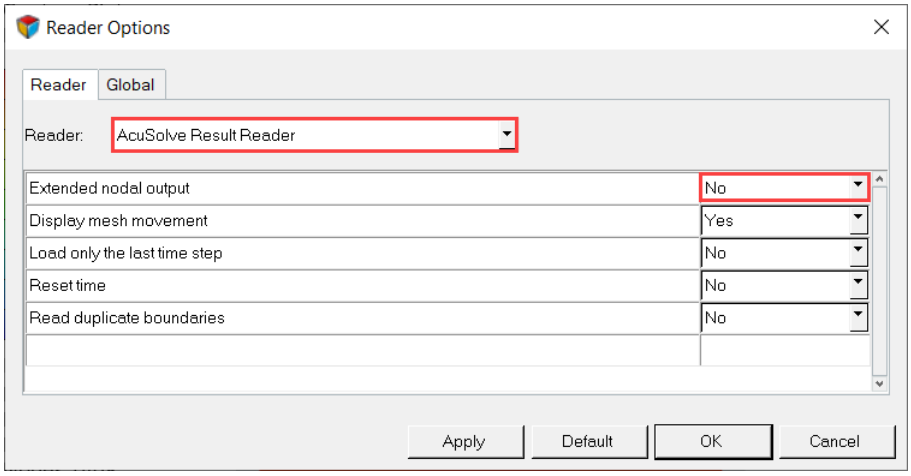


Figure 573:

7. Click **Apply** then close the dialog.
8. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create Contours on a Cut Plane

In this step, you will create temperature, relative humidity, and dew point temperature contours on a cut plane.

1. Since this is a transient case, you need to plot the results at the last timestep. To do this, click the Time drop-down menu in the Results Browser and select the last option in the list.

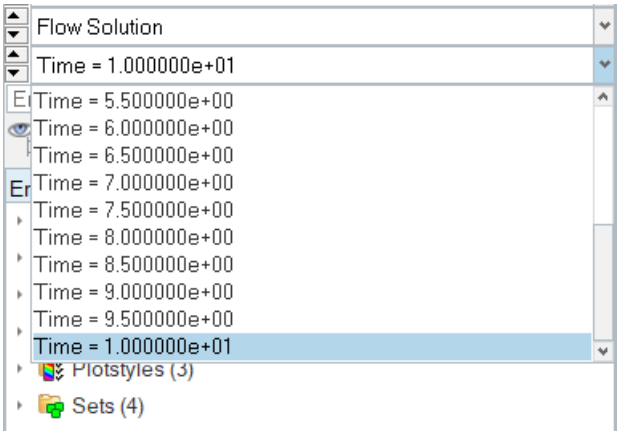


Figure 574:

Here we have the last time as 10 sec.

2. To create a new cut plane, right-click in the Results Browser and select **Create** > **Section Cut** > **Planar** from the context menu.
   A new entity, Section 1, is created under the Section Cuts heading.
3. Right-click on **Section 1** and select **Edit** from the context menu.
4. In the Section cut panel, change Define plane to the **Z Axis**.
5. Under Display options, activate the **Cross section** option.
6. Verify that the **Clip elements** option is active.
7. Click **Apply**.
8. Move the Define plane slider bar (located under the Z Axis button) to choose a desired position for the section cut plane.

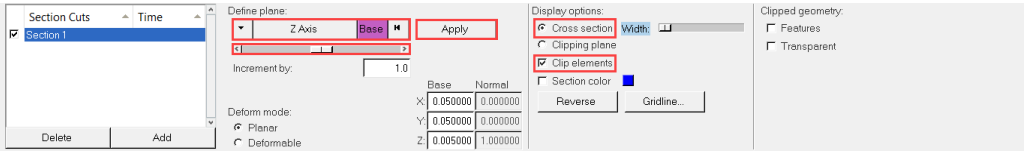


Figure 575:

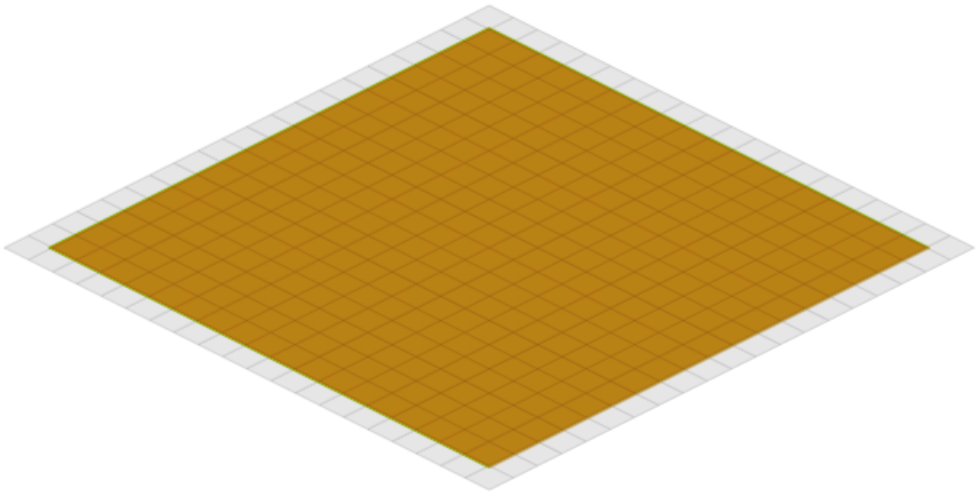


Figure 576:

9. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
10. Click on the Results toolbar to open the Contour panel.
11. In the panel area, set the Result type to **Temperature (s)**.
12. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
13. Click **Apply**.
14. Under the Display tab, turn off the Discrete color option.



Figure 577:

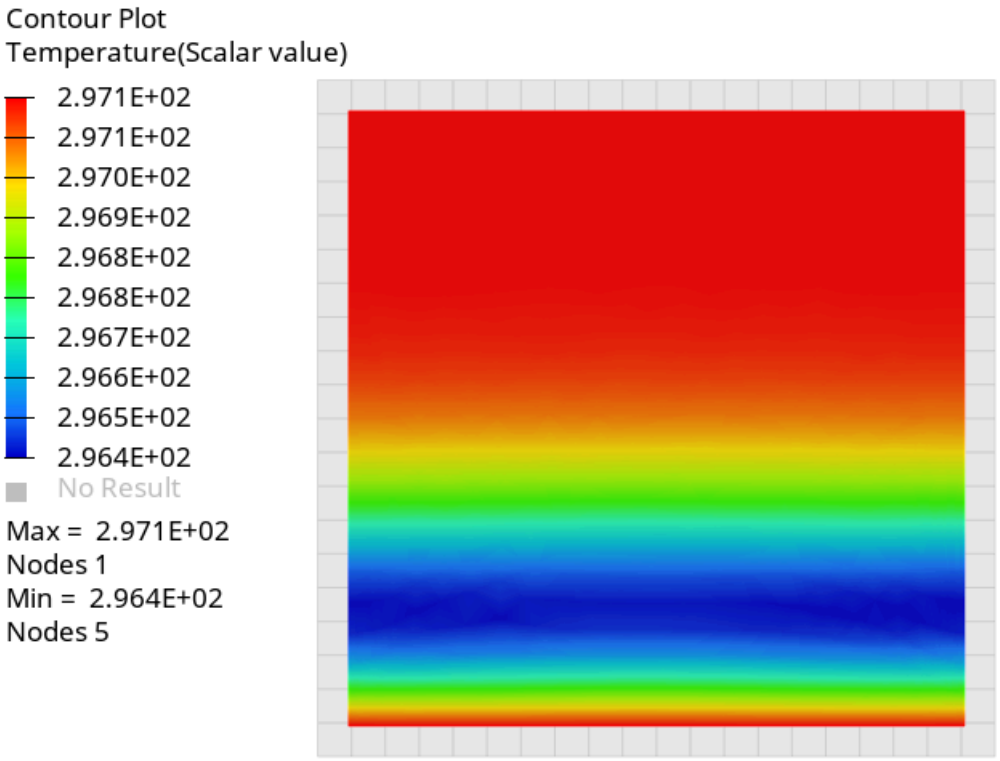


Figure 578:

15. Change the Result type to **Relative_humidity (s)** to view the relative humidity contour of the XY plane.

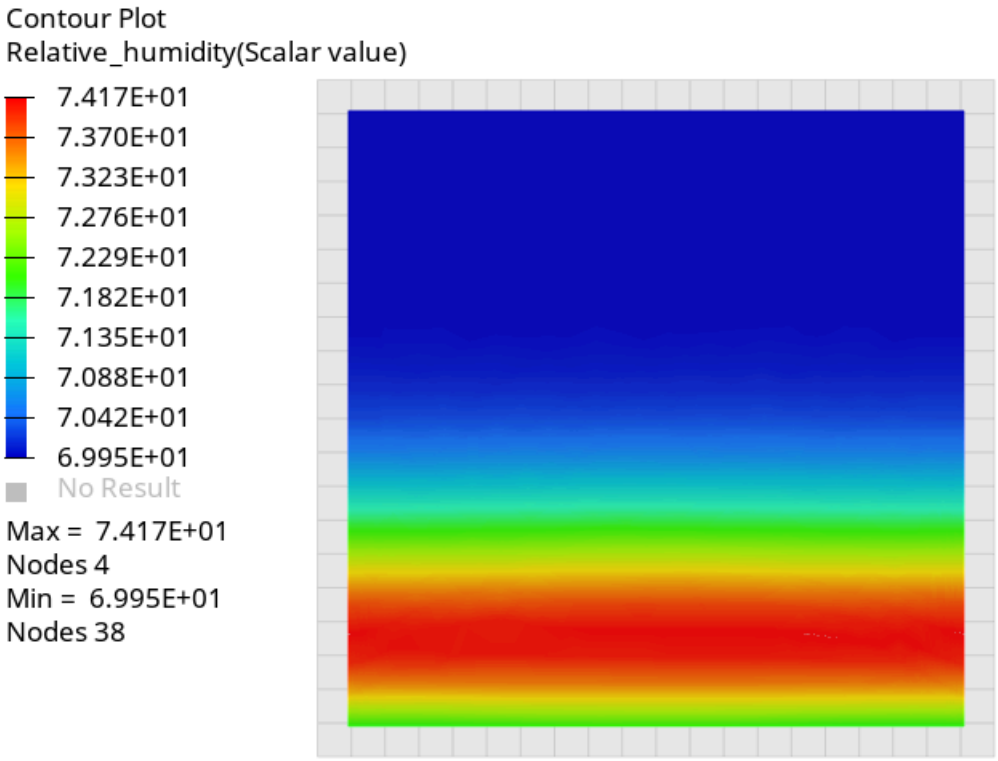


Figure 579:

16. Change the Result type to **Dew_Point_Temperature (s)** to view the dew point temperature contour of the XY plane.

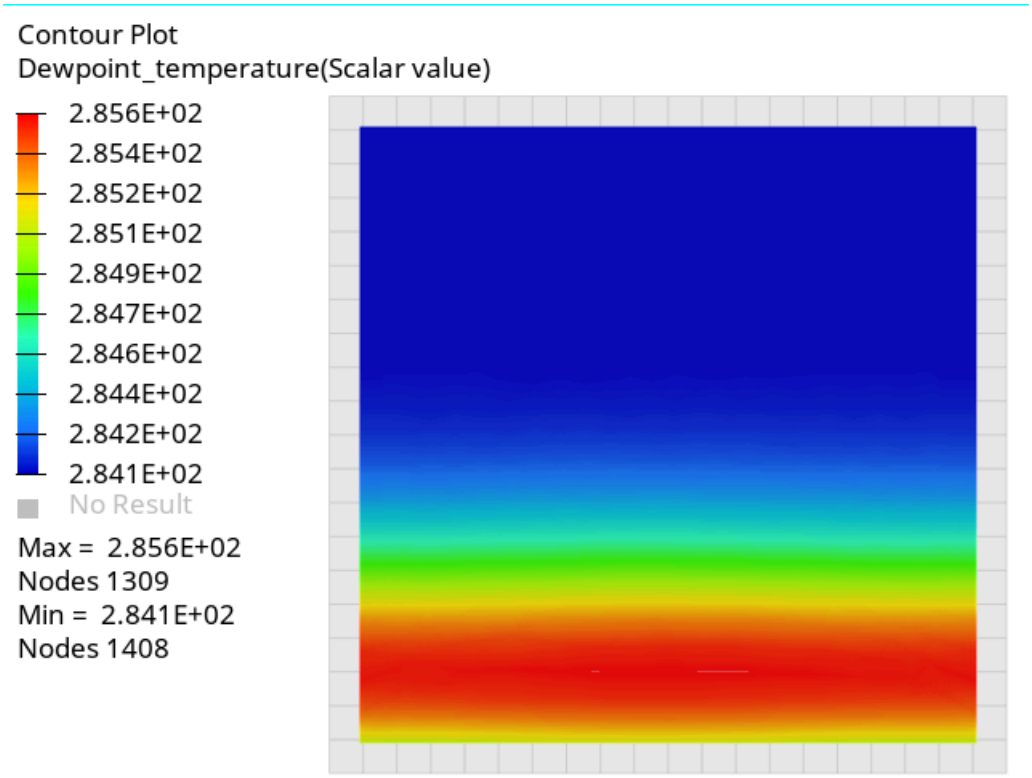


Figure 580:

17. Save the plots as an image file.

a) On the Image Capture toolbar, toggle the **Save Image File/Clipboard** icon (/) so that it shows .

b) Click the **Capture Graphics Area** icon .

c) Provide a name for the image in the dialog then click **Save**.

> **Note:** If you wish to use the images in a presentation, you can copy them to the clipboard by toggling the **Save Image to File/Clipboard** icon to instead of . Then, paste the image in your presentation.

Summary

In this tutorial, you worked through a basic workflow to set up and solve a transient multiphase flow problem using HyperMesh and AcuSolve. You also learned how to use the humidity model to quantify accumulation and loss of water vapor on the bottom wall surface due to temperature change. Once the solution was computed, you post-processed the results in HyperView where you created contour plots.

2.5 Moving Body

2.5.1 ACU-T: 5000 Centrifugal Air Blower with Moving Reference Frame (Steady)

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T5000_BlowerSteady.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 1 and Figure 2. It consists of a centrifugal blower with a wheel of forward curved blades, and a housing with inlet and outlet ducts. The fluid through the inlet plane enters the hub of the blade wheel, radially accelerates due to centrifugal force as it flows over the blades, and then exits the blower housing through the outlet plane. Because they're relatively cheaper and simpler than axial fans, centrifugal blowers have been widely used in HVAC (heating, ventilating, and air conditioning) systems of buildings.

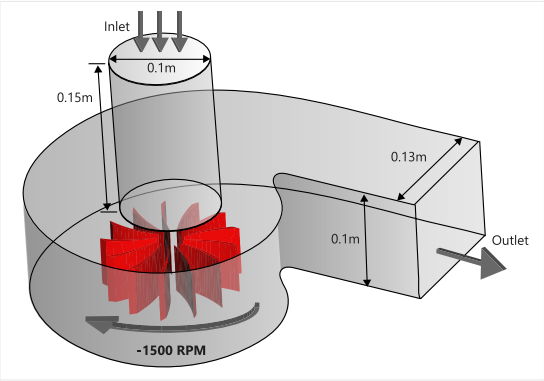


Figure 581: Schematic of Centrifugal Blower

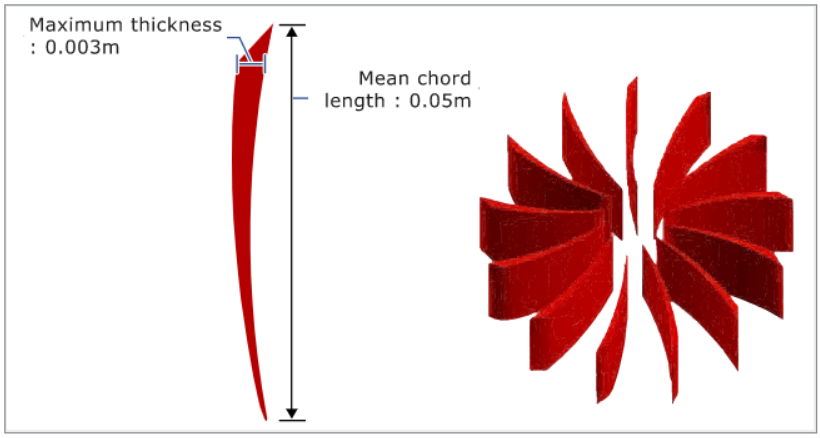


Figure 582: Schematic of Fan Blades

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5000_BlowerSteady.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `CentrifugalBlower` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Blower_Steady` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

In this step, you will set the simulation parameters that apply globally to the simulation.

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Change the Turbulence model to **Spalart Allmaras**.

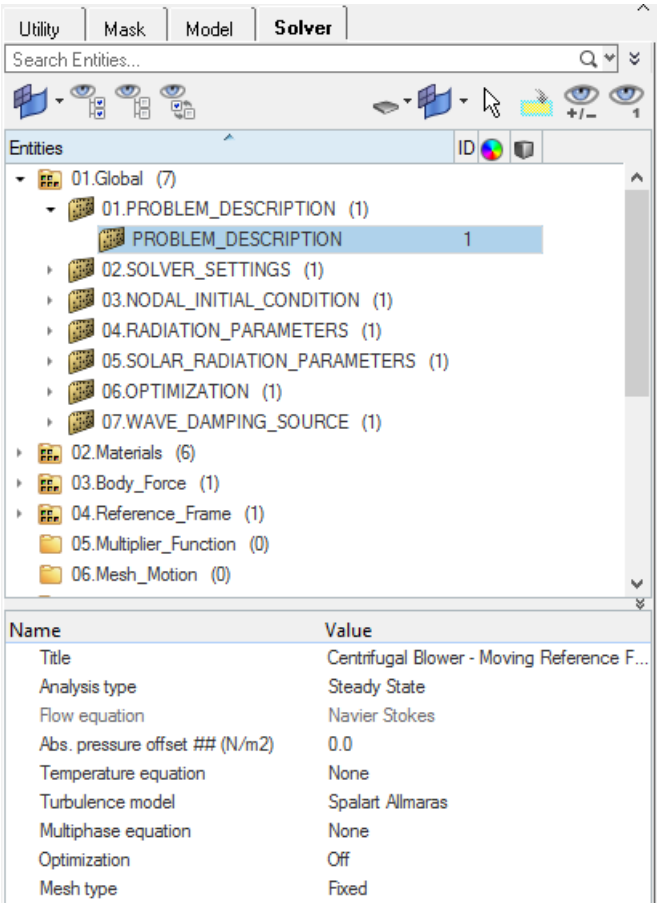


Figure 583:

Create a Moving Reference Frame

In this step, you will create a rotating reference frame for the fluid in the impeller region so that the elements in those regions are solved in the given rotating reference frame and rotational body forces are added to that volume set.

1. In the Solver Browser, right-click on **04.REFERENCE_FRAME** and select **Create**.
2. In the Entity Editor, rename the reference frame as `Impeller_RF`.
3. Set the Rotation center to `(0, 0, 0.05)`.
4. Set the Angular velocity-Z to `-157.08` rad/sec.

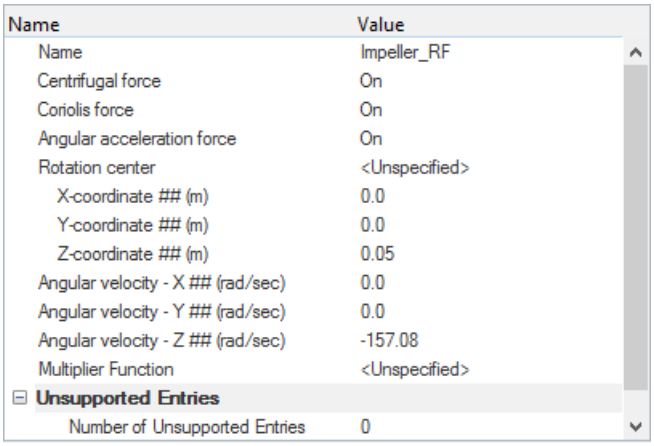


Figure 584:

Set Up Boundary Conditions and Material Model Parameters

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow type to **Stagnation Pressure**.
   c) Set the Turbulence input type to **Viscosity Ratio**.
   d) Set the Material to **Air_HM**.
   e) Set the Turbulence viscosity ratio to `10.0`.

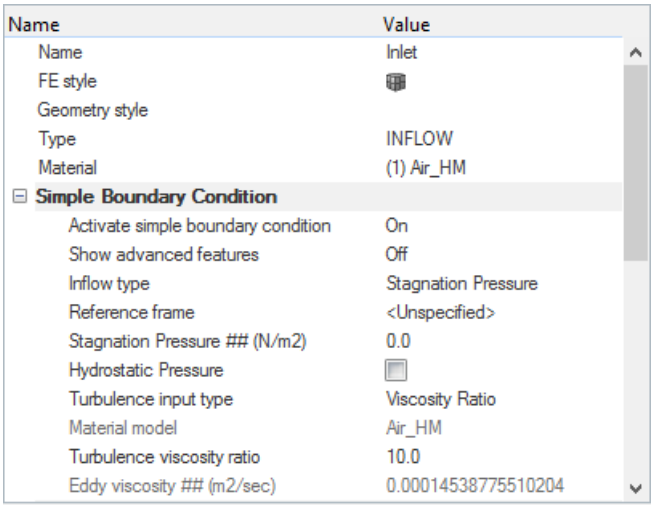


Figure 585:

3. Click **Outlet**. In the Entity Editor, change the Type to **OUTFLOW**.

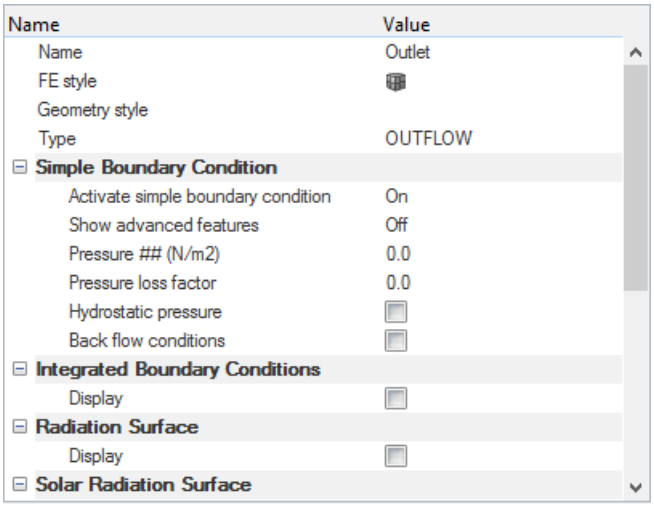


Figure 586:

4. Click **Walls**. In the Entity Editor, verify that the Type is set to **WALL**.

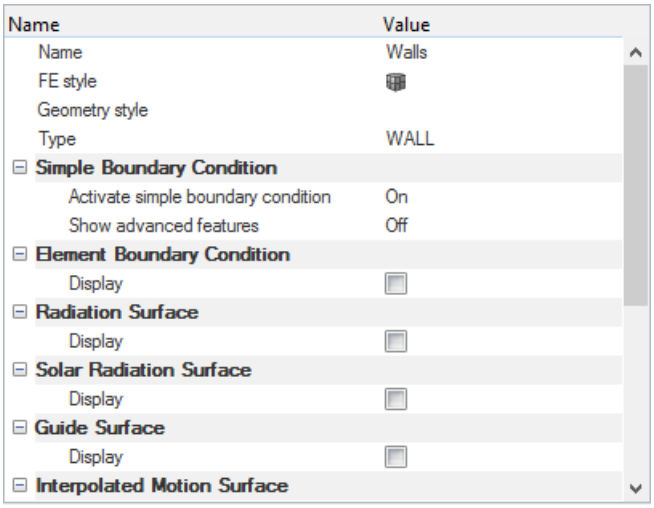


Figure 587:

All the surface elements that make up the outer wall of the blower, the fan blades and the interface between the impeller and main fluid can be grouped into one surface set. Auto_Wall, which is an advanced feature in AcuSolve, re-groups these elements into external and internal walls and applies appropriate wall and interface conditions. In this case, the surface elements on the fan blades are grouped together (AUTO Fluid_Impeller wall) and the reference frame assigned to the impeller fluid will be inherited. The surface elements on the interface will be grouped into (AUTO Fluid_Impeller internal) and the elements on the outer casing will be grouped into (AUTO Fluid_Main wall). This entire process of grouping is done internally without you having to do it manually

5. Click **Fluid_Main**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Select **Air_HM** as the Material.

Name	Value
Name	Fluid_Main
FE style	
Geometry style	
Type	FLUID
Material	(1) Air_HM
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 588:

6. Click **Fluid_Impeller**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Select **Air_HM** as the Material.
   c) Select **Impeller_RF** as the Reference frame.

Name	Value
Name	Fluid_Impeller
FE style	
Geometry style	
Type	FLUID
Material	(1) Air_HM
Element Set	
Body force	<Unspecified>
Reference frame	(1) Impeller_RF
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 589:

7. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

The **Solver job Launcher** dialog opens.

3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

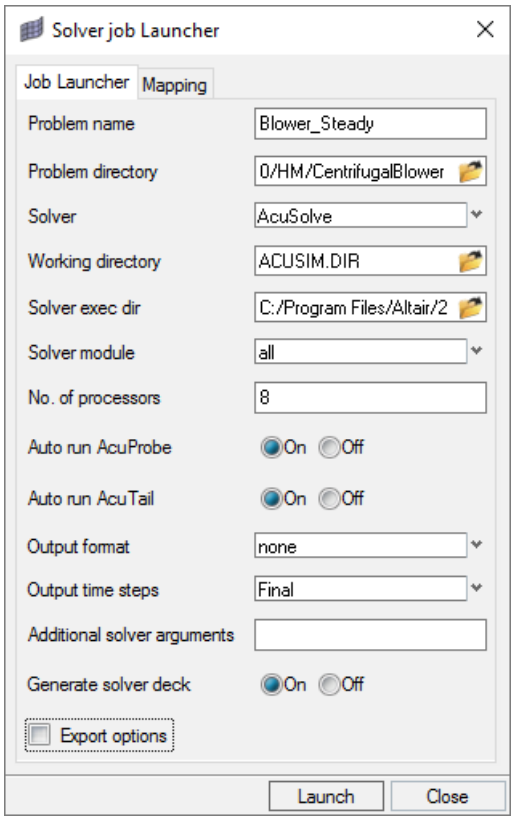


Figure 590:

Post-Process the Results

Create a Pressure-Rise Plot in AcuProbe

As the solution progresses, the AcuTail and AcuProbe windows are launched automatically. In this step, you will create a User Defined Function (UDF) and generate a plot of the pressure rise between the inlet and outlet.

1. Once the solution has converged, click the User Function icon in the AcuProbe window.
   A **User Function** dialog opens.
2. Enter `Pres_Rise` as the function name.
3. Type `P_Outlet =` in the Function field.

4. Expand **Surface Output** > **Outlet** > **Pressure**. Right-click on **pressure** and select **Copy Name**. Paste the value in the Function field after P_Outlet =.
5. On the next line, type `P_Inlet =` and repeat the above step for inlet pressure.
6. On the next line, type `value = P_Outlet - P_Inlet`.

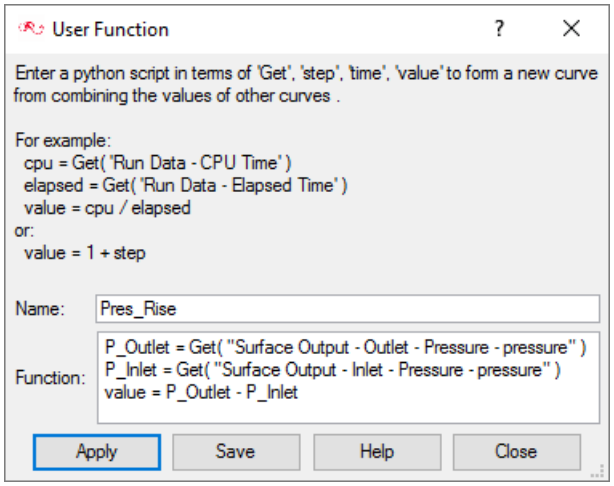


Figure 591:

> **Note:** The word "value" is case sensitive and should always be in lowercase characters. If it starts with a capital letter, it will give you an error window.

7. Click **Apply** to display the plot.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

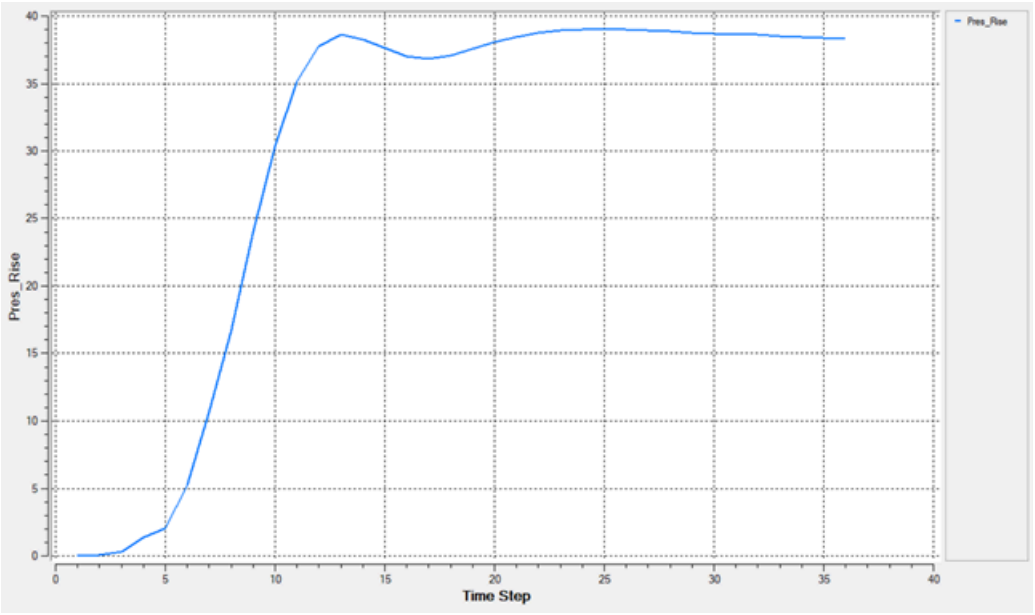


Figure 592:

In the next steps you will use HyperView to create a pressure contour on a section on the z-plane. Close the AcuProbe and AcuTail windows. In the HyperMesh window, close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.
   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Blower_Steady.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.
   The model is colored by geometry after loading.

Create a Pressure Contour on a Section Cut Plane

1. Right-click on empty space in the Results Browser and select **Create** > **Section Cut** > **Planar**.
   A new entity named Section 1 is created
2. Right-click on **Section 1** and select **Edit**.
3. In the Define plane section in the panel area, select **Z-axis** and click **Apply**.
4. Enter `(0,0,0.05)` for the Base values and press Enter.
5. Change the Display options from Clipping plane to **Cross section**.

6. Click **Gridline**. In the dialog, uncheck the **Show** option then click **OK**.
7. Click on the Results toolbar to open the Contour panel.
8. Select **Pressure(s)** as the Result type.
9. Click the **Components** collector and select **All**.
10. Click **Apply**.
11. In the panel area, under the Display tab, turn off the Discrete color option.

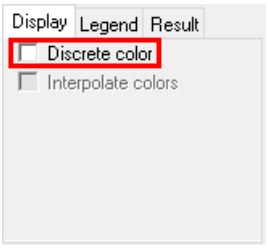


Figure 593:

12. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Numeric format to **Fixed** then click **OK**.
13. Adjust the orientation in the graphics window for a better view of the results and verify that the contour plot looks like the figure below.

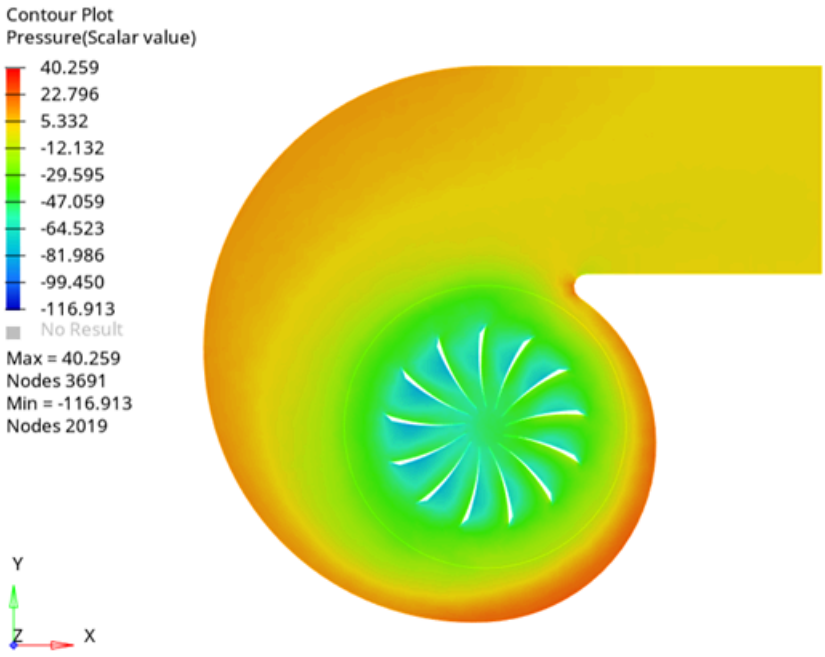


Figure 594:

Summary

In this tutorial, you successfully learned how to set up a steady state simulation involving a rotating reference frame in a centrifugal blower. You started by importing the mesh and then once the case was set up, you generated a solution using AcuSolve. Then, you computed the pressure rise using AcuProbe and created a contour plot for pressure on a cut plane using HyperView.

2.5.2 ACU-T: 5001 Blower - Transient (Sliding Mesh)

Prerequisites

This tutorial provides the instructions for setting up, solving, and viewing results for a transient simulation of a centrifugal air blower utilizing the sliding mesh approach. In order to run this tutorial, you should have already run through ACU-T: 5000 Centrifugal Air Blower with Moving Reference Frame (Steady) and kept the solution in your working directory. It is assumed that you have some familiarity with HyperMesh, HyperMesh, and HyperView.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T5001_BlowerTransient.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be discussed in this tutorial is shown schematically in Figure 595. The simulation is divided into two components, steady state and transient. The steady state solution will be computed first and then projected onto the mesh and used as the initial state for the transient simulation. Please refer to the tutorial ACU-T: 5000 Centrifugal Air Blower with Moving Reference Frame (Steady) to learn the step by step procedure to obtain the steady state solution for this problem. Once the transient solution is computed, you will post-process the results using HyperView.

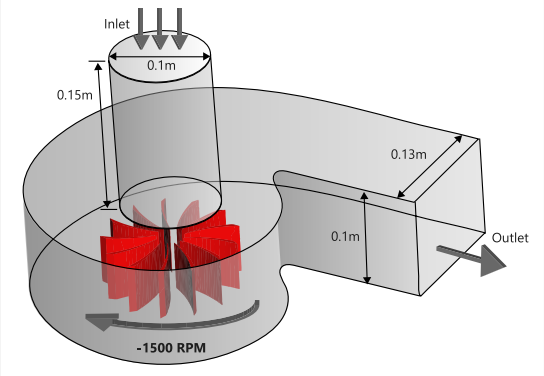


Figure 595:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5001_BlowerTransient.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `BlowerTransient` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `BlowerTransient` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Run the Steady State Simulation

In this step, you will run the steady state simulation with the model file provided and then create the nodal initial condition files needed for the transient simulation.

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Leave the remaining options as default and click **Launch** to start the solution process.
5. Once the solution has converged, close the AcuTail and AcuProbe windows. Also, close the Solver job launcher and the AcuSolve Control tab.
6. Start AcuSolve Command Prompt from the Windows Start menu by clicking **Start** > **Altair <version>** > **AcuSolve Cmd Prompt** .
7. In the Command Prompt, change the directory to the working directory using the `cd` command.
8. Type the command `acuProj -crd HYPERMESH.DIR\Blower_Transient.crd -run 1` and press **Enter**.
9. Verify that the working directory is populated with the new files `Blower_Transient.pres.nic`, `Blower_Transient.vel.nic`, and `Blower_Transient.eddy.nic`
10. Move these files into the HYPERMESH.DIR directory.

Set the Transient Simulation Parameters

In this step, you will set the simulation parameters that apply globally to the transient simulation.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Change the Analysis type to **Transient**.
3. Set the Turbulence model to **Spalart Allmaras**.
4. Set the Mesh type to **Fully Specified**.

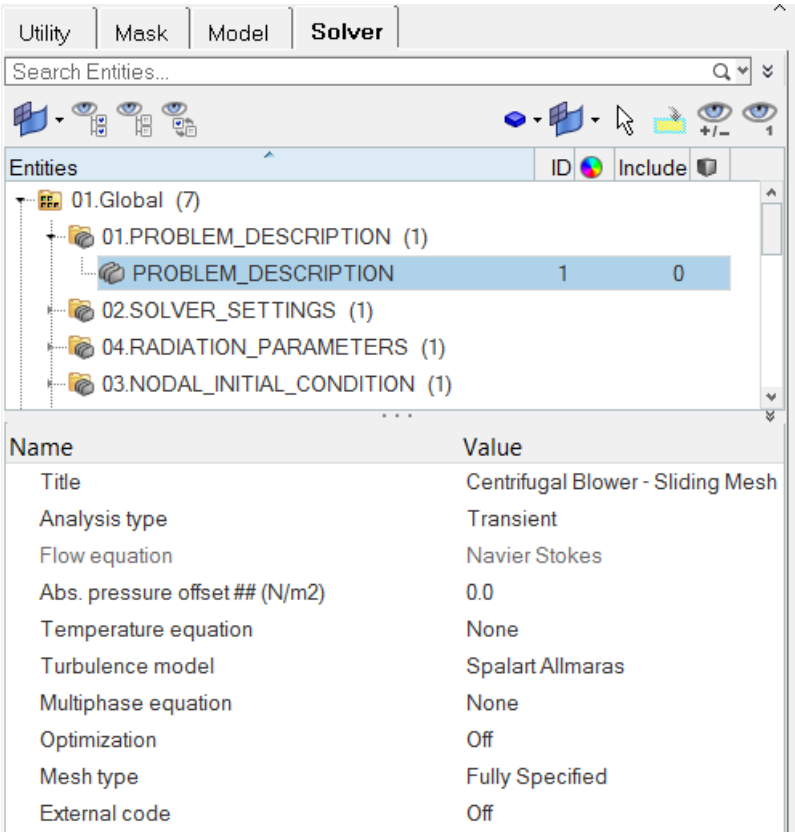


Figure 596:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. Set the Max time steps to `0` and press **Enter**.
3. Set the Final time to `0.12`.
4. Set the Initial time increment to `0.00111`
5. Change the Max stagger iterations to `2`.
6. Change the Relaxation factor to `0`.

Name	Value
Max time steps	0
Final time ## (sec)	0.12
Initial time increment ## (sec)	0.00111
Auto time increment	☐
Multiplier Function	<Unspecified>
Convergence tolerance	0.001
Min stagger iterations	0
Max stagger iterations	2
Relaxation factor	0.0
Flow	On
Turbulence	On
External code	Off
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 597:

Set the Nodal Output Frequency

1. In the Solver Browser, expand **17.Output** then click on **NODAL_OUTPUT**.
2. Set the Time step frequency to `1`.
3. Turn **On** the Output initial condition field.

Name	Value
Time step frequency	1
Time frequency ## (sec)	0.0
Output initial condition	☑
Continuous output	☐
Number of saved states	0
Unsupported Entries	
Number of Unsupported Entries	0

Figure 598:

4. Save the model.

Define Mesh Motion and Set Up Boundary Conditions

Create Mesh Motion

1. In Solver Browser, right-click on **06.Mesh_Motion** and select **Create**.
2. Rename the mesh motion as `Impeller_Motion` in the Entity Editor.
3. Set the Type to **Rotation**.
4. Enter `(0, 0, 0.05)` as the coordinates of the Rotation center.
5. Set the Angular velocity - Z to `-157.08` rad/sec.

Name	Value
Name	Impeller_motion
Type	Rotation
Rotation center	<Unspecified>
X-coordinate ## (m)	0.0
Y-coordinate ## (m)	0.0
Z-coordinate ## (m)	0.05
Angular velocity - X ## (rad/sec)	0.0
Angular velocity - Y ## (rad/sec)	0.0
Angular velocity - Z ## (rad/sec)	-157.08
Rotation variable	Run Time
Unsupported Entries	
Number of Unsupported Entries	0

Figure 599:

Modify the Boundary Conditions

1. In the Solver Browser, expand **11.Volumes** > **FLUID**.
2. Click **Fluid_Impeller.** In the Entity Editor,
   a) Set the Reference frame to **Unspecified**.
   b) Set the Mesh motion to **Impeller_Motion**.

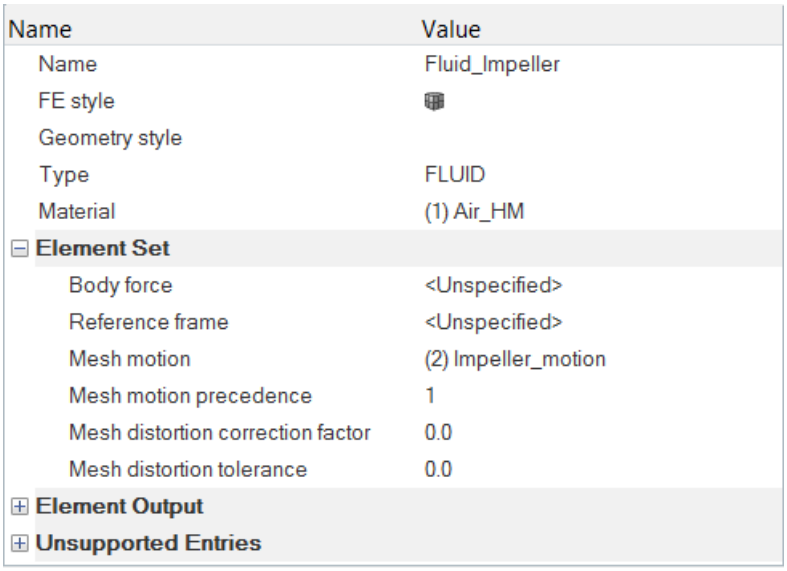


Figure 600:

Specify the Nodal Initial Conditions

1. In the Solver Browser, expand **01.Global** then click on **03.NODAL_INITIAL_CONDITION**.
2. In the Entity Editor, under the Pressure tab,
   a) Set the Type to **Nodal Values**.
   b) Change Select nodes by to **NIC file**.
   c) Click the open file icon in the NIC file field and browse to the HYPERMESH.DIR folder located in your working directory. Select the file **Blower_Transient.pres.nic**.

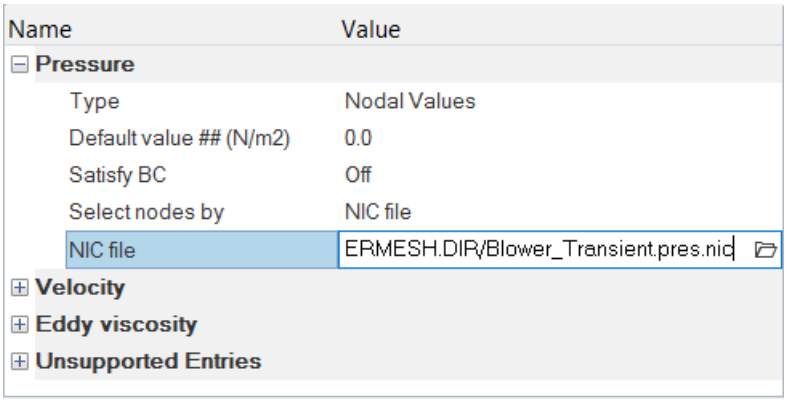


Figure 601:

3. Repeat the above step for the Velocity and Eddy viscosity fields. Select the **Blower_Transient.vel.nic** and **Blower_Transient.eddy.nic** files, respectively.
4. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.

2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.

3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.

4. Observe that the Output time steps is set to **All**. This means that AcuSolve will write solution at all the time steps specified in the Nodal Output Frequency.

5. Leave the remaining options as default and click **Launch** to start the solution process.

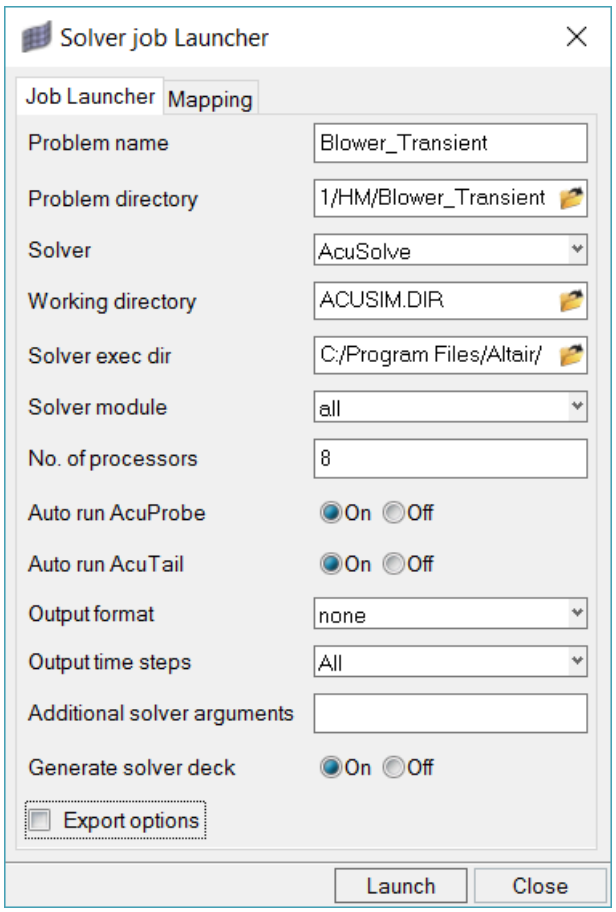


Figure 602:

Once the solution is launched, the AcuSolve Control tab opens. Also, the AcuTail and AcuProbe windows are launched automatically in which the solution progress can be monitored.

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `BlowerTransient.2.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create a Pressure Animation

In this step, you will start by creating a pressure contour on a cut plane on the z-axis. Then, you will create an animation of the pressure contour.

1. Click on the Results toolbar to open the Contour panel.
2. In the panel area, set the Result type to **Pressure (s)**.
3. Click **Apply** to display the velocity contour.
4. In the panel area, under the Display tab, turn off the Discrete color option.

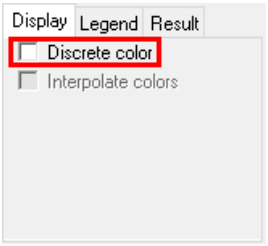


Figure 603:

5. Click the **Legend** tab then click **Edit Legend**. In the dialog, change the Type to **Dynamic scale** and the Numeric format to **Fixed** then click **OK**.

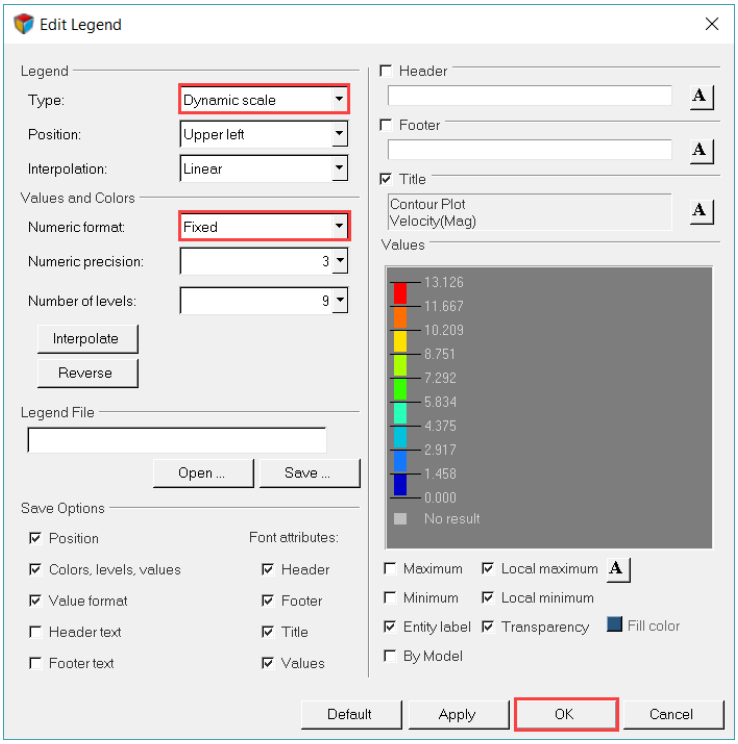


Figure 604:

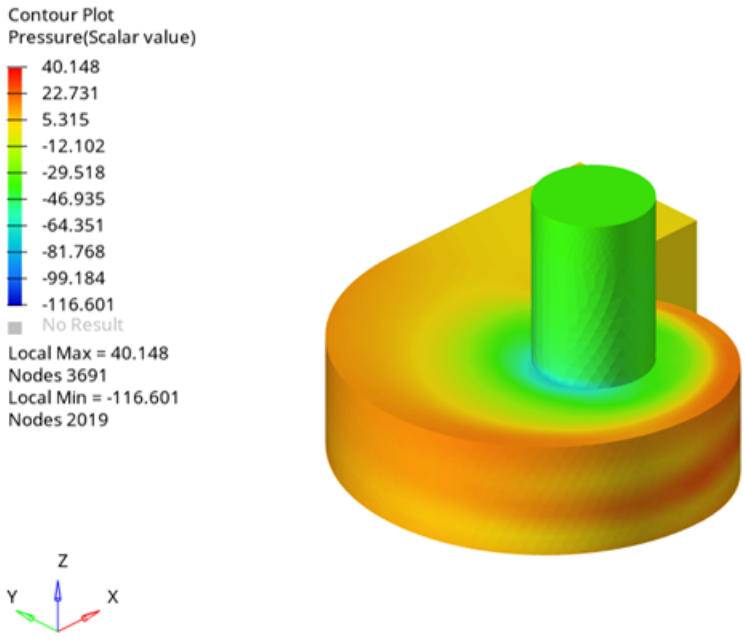


Figure 605:

6. Click on the Display toolbar to open the Section Cut panel.
7. In the panel area, click **Add**.
8. Under the Define plane section, change the plane to **Z-Axis** then click **Apply**.

9. Set the Z-coordinate of the Base to `0.05` then press **Enter**

10. Under the Display options section, activate the **Cross section** option.

11. Click **Girdline...**. In the dialog, turn off the **Show Grid Line** option then click **OK**.

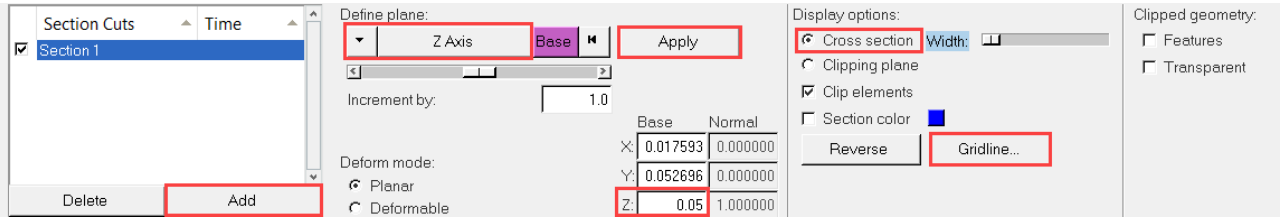


Figure 606:

12. Orient the display to the xy-plane by clicking on the Standard Views toolbar.

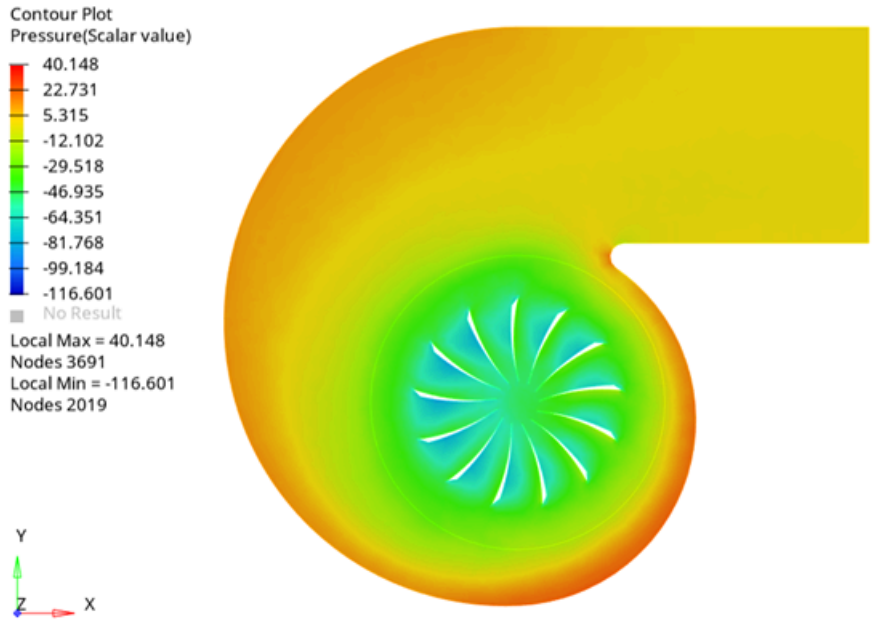


Figure 607:

13. From the Animation toolbar, click the **Animation controls** icon.



Figure 608:

14. In the panel area, set the Max Frame Rate to `15` Frames/Sec by dragging the slider.

15. Click on the **Start/Pause Animation** icon to play the velocity magnitude animation.



Figure 609:

Create Streamlines

1. Click on the Display toolbar to open the Section Cut panel.
2. In the panel area, toggle off the **Section 1** check box.
   This will turn off the display of the section cut, showing the complete model.
3. In the Results Browser, expand the list of **Components** and turn off all components except the **Inlet**, **Outlet**, **AUTO Fluid_Main wall** and **AUTO Fluid_Impeller wall**.

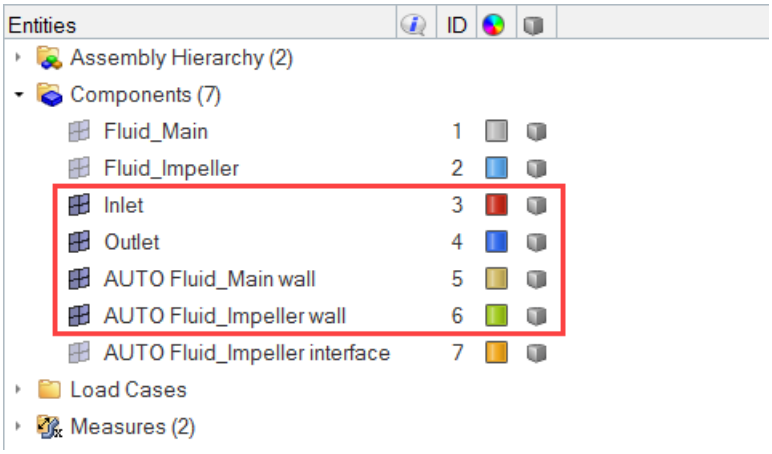


Figure 610:

4. Change the element display mode to **Transparent Elements and Feature Lines**.

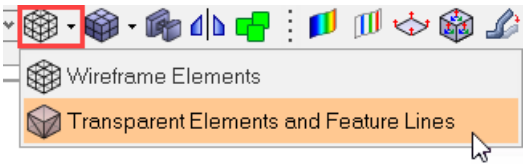


Figure 611:

5. Rotate the model so that you have a clear view of all the components that are being displayed.
6. Go to the Contour panel, change the Result type to **Velocity (v)**, and set the drop-down menu below it to **Mag**.
7. Click **Apply** to plot the velocity magnitude.

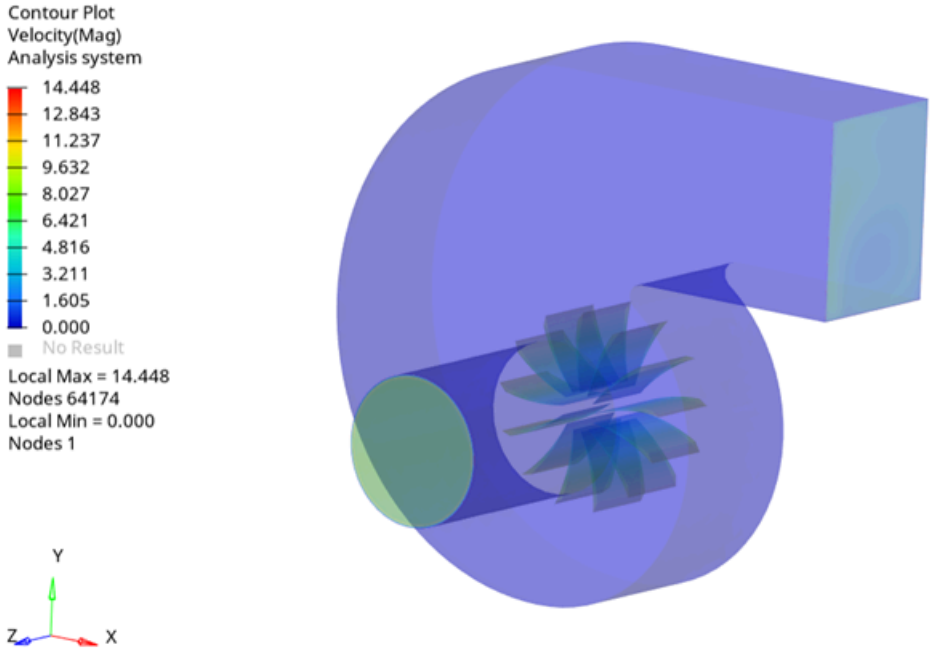


Figure 612:

8. Click on the Results toolbar to open the Streamlines panel.
9. In the panel area, click **Add**, then change the Rake type to **Area**.
10. Click the **Component** entity collector then select **By ID**.
11. In the **Select by ID** dialog, enter `6` in the value field of Component ID then click **OK**.
12. In the panel area, change the Integration mode to **Both**.
13. Set the Number of seeds to `50`.
14. Toggle on the **Draw as tube** checkbox then click **Create Streamlines** to generate the streamlines.



Figure 613:

This will generate streamlines originating from the Inlet surface with seeds that are distributed over the surface area of the fan blades and end at the outlet.

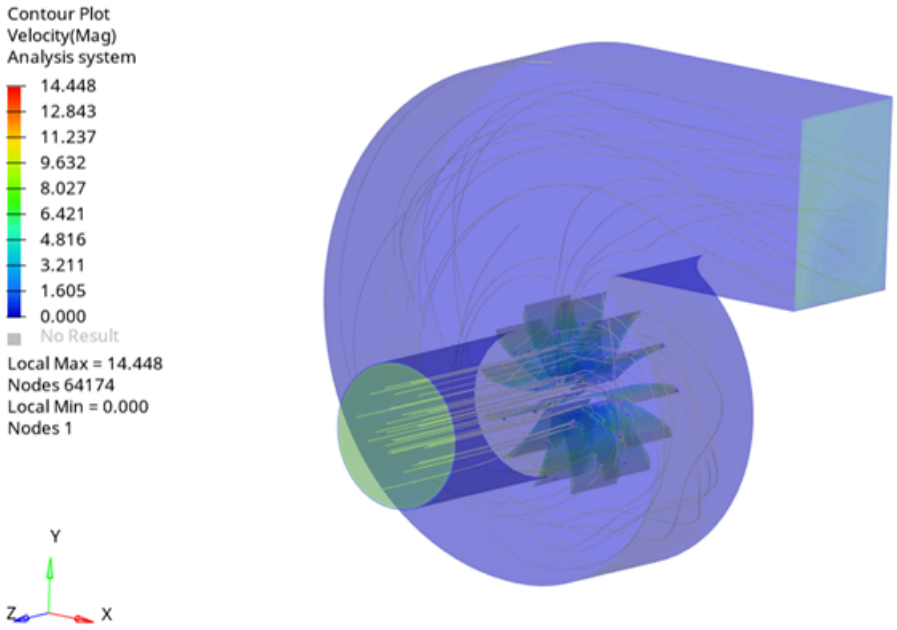


Figure 614:

15. In the panel area, click the **Tracers** tab.

16. Change the Pule to **Endless**.

17. Leave the other default values unchanged then click **Create** to generate the tracers.



Figure 615:

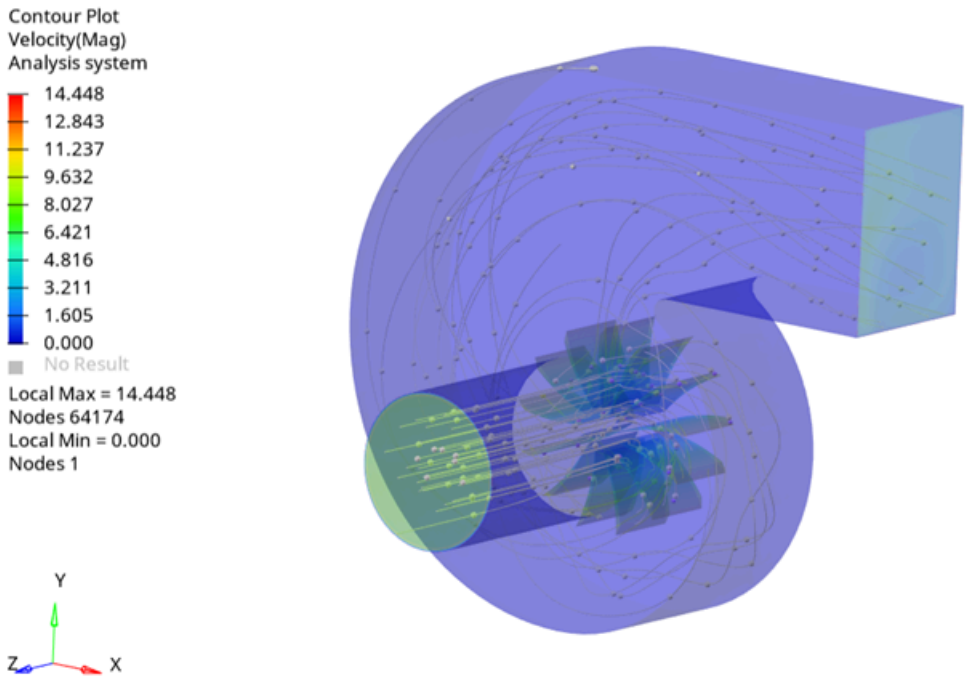


Figure 616:

Summary

In this tutorial, you successfully learned how to set up and solve a transient simulation of a centrifugal air blower using the sliding mesh approach. You started by importing the meshed geometry and then ran the steady state simulation. Then, you used the steady state result as the starting point for the transient simulation using the AcuProj tool to specify the nodal initial conditions and then solved the transient simulation using the sliding mesh approach instead of the moving reference frame approach. Once the transient solution was computed, you launched HyperView and created an animation of pressure contours and streamlines.

2.5.3 ACU-T: 5100 Modeling of a Fan Component: Axial Fan

Prerequisites

This simulation provides instructions for running a steady state simulation of flow inside a pipe with an interior fan placed at the middle of the pipe. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of AcuSolve and HyperMesh. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials`

\AcuSolve to a local directory. Extract ACU-T5100_AxialFanComponent.hm and AxialCoefficient.txt from HyperMesh_tutorial_inputs.zip.

Since the HyperMesh database (.hm file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be solved in this tutorial is shown schematically in the figure below. It consists of an interior fan which rotates at a speed of 377 rad/sec (~3600 RPM) and has a thickness of 0.06 m and a tip radius of 0.11 m. The volumetric flow rate at the inlet is 0.146 m^3/sec (~525.35 m^3/hr). The problem is simulated as a steady state run and the pressure rise across the fan region is computed.

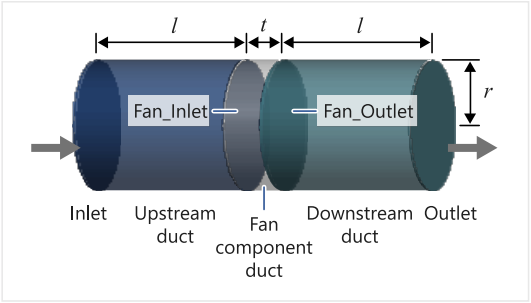


Figure 617:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file ACU-T5100_AxialFanComponent.hm and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named AxialFan and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter AxialFan as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, change the Turbulence model to **Spalart Allmaras**.

3. Ensure that the Analysis type is Steady State.

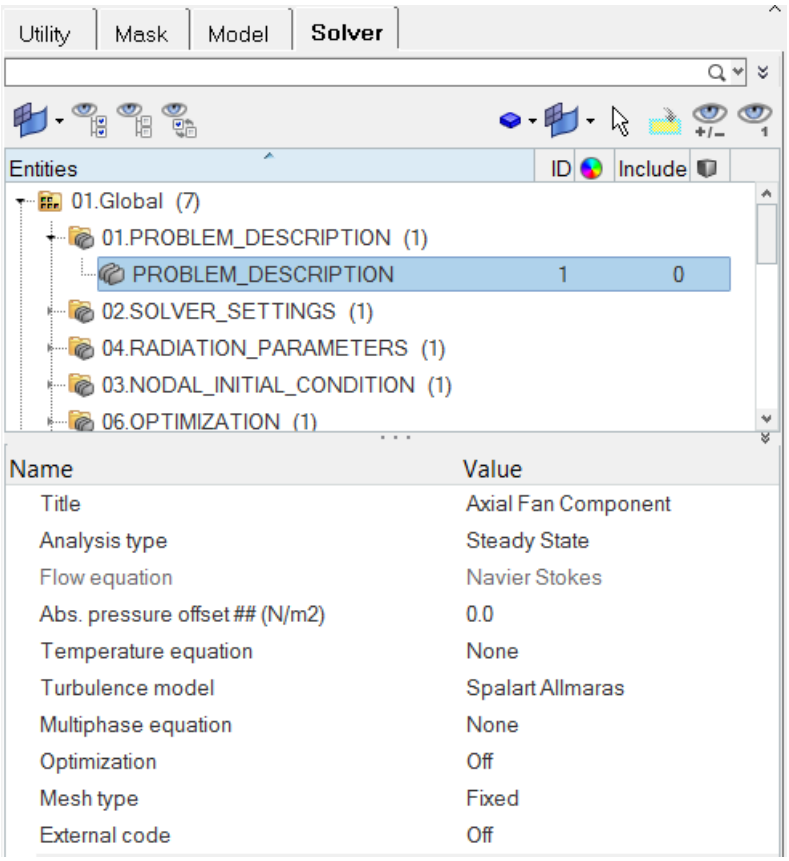


Figure 618:

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow type to **Flow rate**.
   c) Set the Flow rate `0.146` m^3/sec.

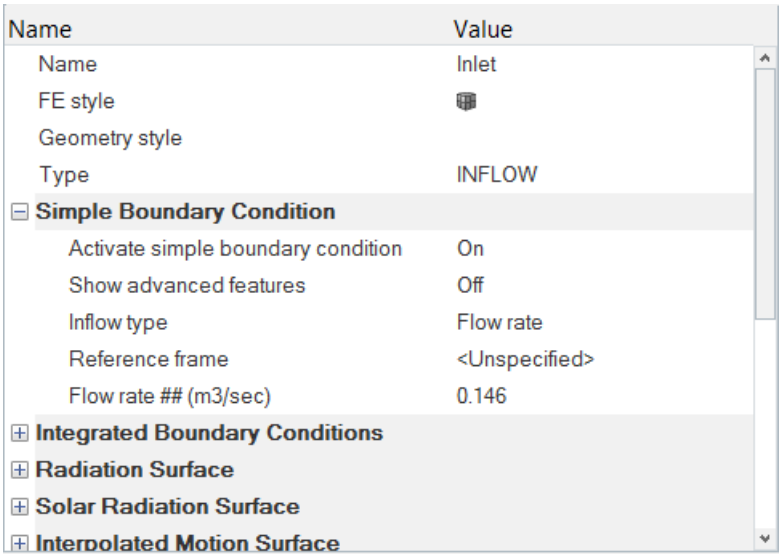


Figure 619:

3. Click **Fan_Inlet**. In the Entity Editor,

a) Change the Type to **FAN_COMPONENT**.

b) Set the Rotational speed to `377` rad/sec.

c) Set the Tip radius to `0.11` m.

d) Set the Fan thickness to `0.06` m.

e) Change the Axial coefficient type to **Piecewise Bilinear**.

f) For the Axial coefficient curvefit file, click the open file icon and browse to the location where you saved `AxialCoefficient.txt` and select it. Click **Open**.

g) Verify that the Radial coefficient and Tangential coefficient are set to `0`.

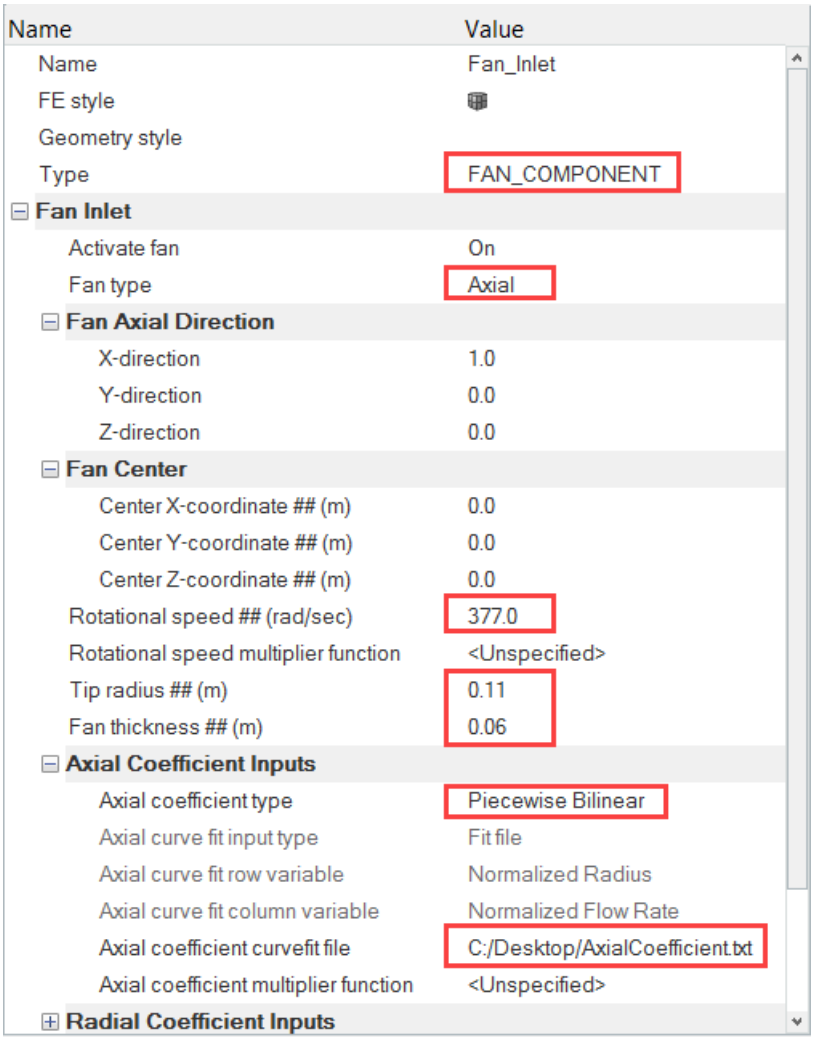


Figure 620:

4. Click **Outlet**. In the Entity Editor, change the Type to **OUTFLOW**.

Name	Value
Name	Outlet
FE style	
Geometry style	
Type	OUTFLOW
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Pressure ## (N/m2)	0.0
Pressure loss factor	0.0
Hydrostatic pressure	
Back flow conditions	
Integrated Boundary Conditions	
Radiation Surface	
Solar Radiation Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 621:

5. Click **Walls**. In the Entity Editor, verify that the Type is set to **WALL**.

Name	Value
Name	Walls
FE style	
Geometry style	
Type	WALL
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Element Boundary Condition	
Radiation Surface	
Solar Radiation Surface	
Guide Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 622:

When component type is assigned as Wall, all the elements in the surface set are automatically re-grouped into surface sets based on the parent volume they belong to and also if they are internal or external. Auto_Wall is an advanced feature in AcuSolve which takes care of this process internally, without you having to do it manually and hence reducing the number of steps in the workflow.

6. Click **Downstream_Duct**. In the Entity Editor,

a) Change the Type to **FLUID**.

b) Select **Air_HM** as the Material.

Name	Value
Name	Downstream_Duct
FE style	
Geometry style	
Type	FLUID
Material	(1) Air_HM
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 623:

7. Similarly, set the Fan and Upstream_Duct component Type to **FLUID** and select **Air_HM** as the Material for both components.
8. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

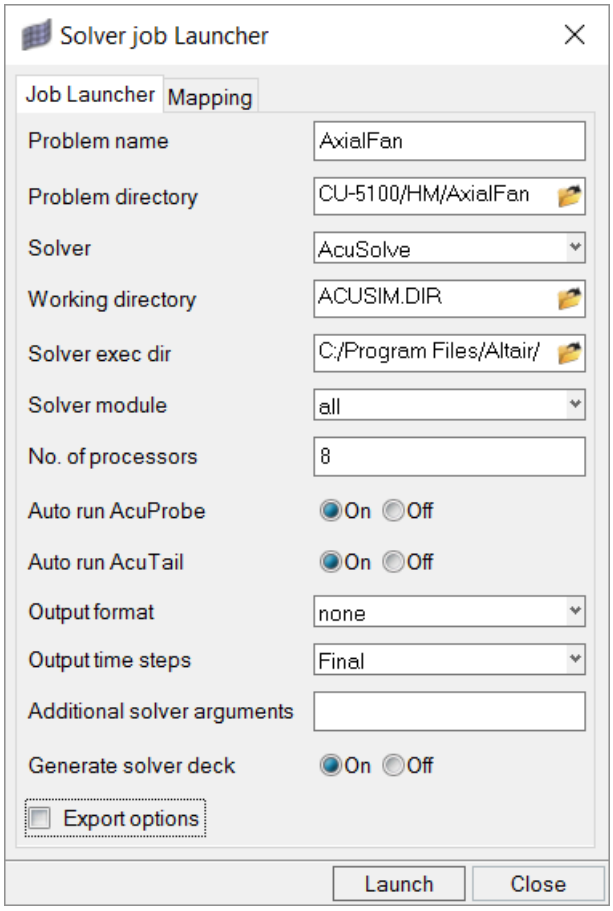


Figure 624:

Post-Process with AcuProbe

As the solution progresses, the AcuProbe window is launched automatically. AcuProbe can be used to monitor various variables over solution time.

1. In the AcuProbe Data Tree, expand **Residual Ratio**.
2. Right-click on **Final** and select **Plot All**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

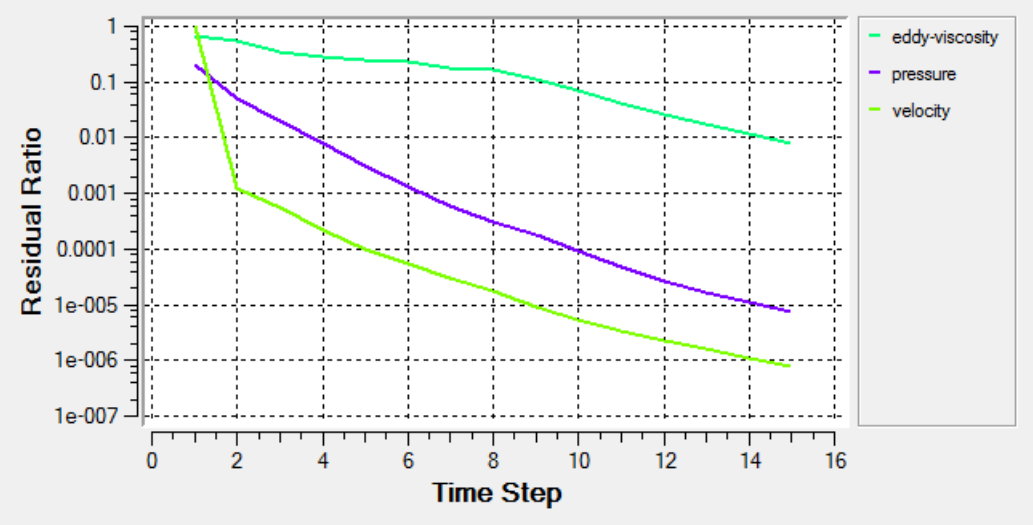


Figure 625:

3. Once the solution is converged, right-click again on **Final** and select **Plot None**.
4. Click the User Function icon from the toolbar.
5. In the dialog, enter the Name as `dP`.
6. In the Data Tree, expand **Surface Output** > **Fan_Inlet** > **Pressure**
7. Right-click on **pressure** and select **Copy name**.
8. In the Function field of the **User Function** dialog, type `Fan_In =` then paste the name you just copied.

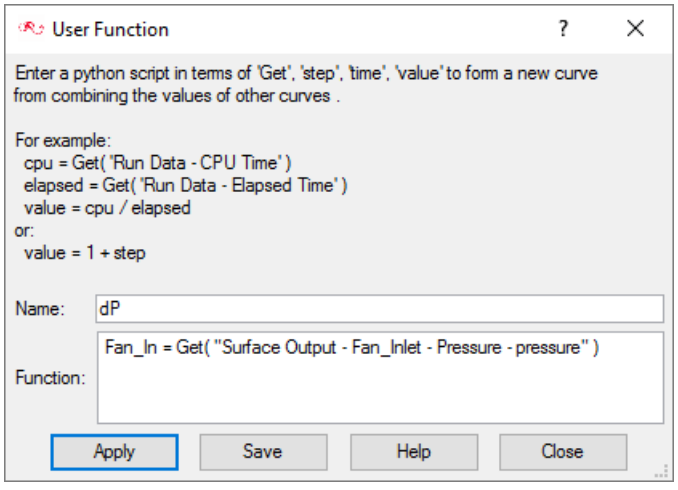


Figure 626:

9. Type `Fan_Out =` on a new line.
10. In the Data Tree, expand **Surface Output** > **AUTO Fan internal** > **Pressure**.
11. Right-click on **pressure** and select **Copy name**.

Here, the surface output (AUTO Fan internal) is the Auto_Wall generated surface output corresponding to the internal surface at the outlet of the fan component volume.

12. Paste the name in the Function field after Fan_Out =.

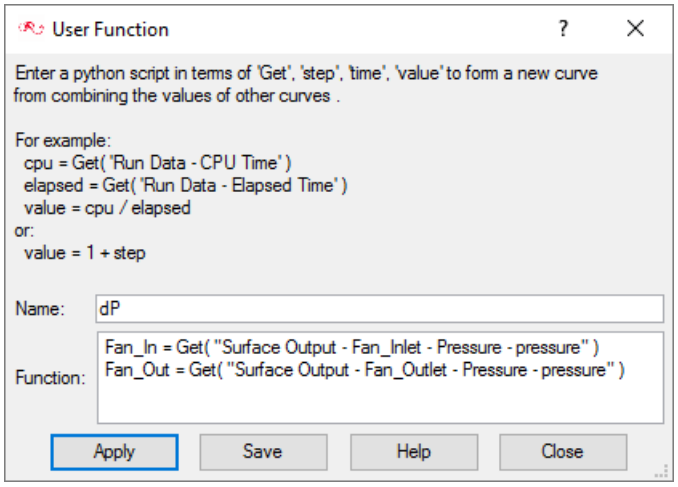


Figure 627:

13. On a new line, type `value = Fan_Out - Fan_In`.

> **Note:** The word "value" is case sensitive and should always be in lower case. If you use a capital letter, an error window appears.

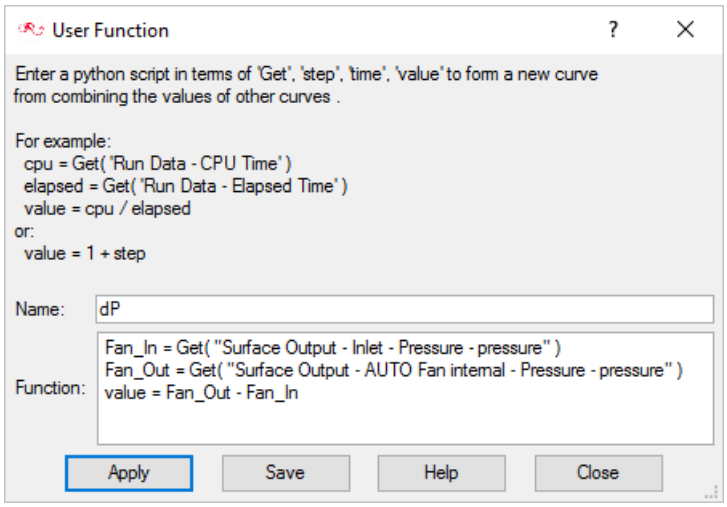


Figure 628:

14. Click **Apply**.

15. In the Data Tree, expand **User function** then right-click on **dP** and select **Plot**.

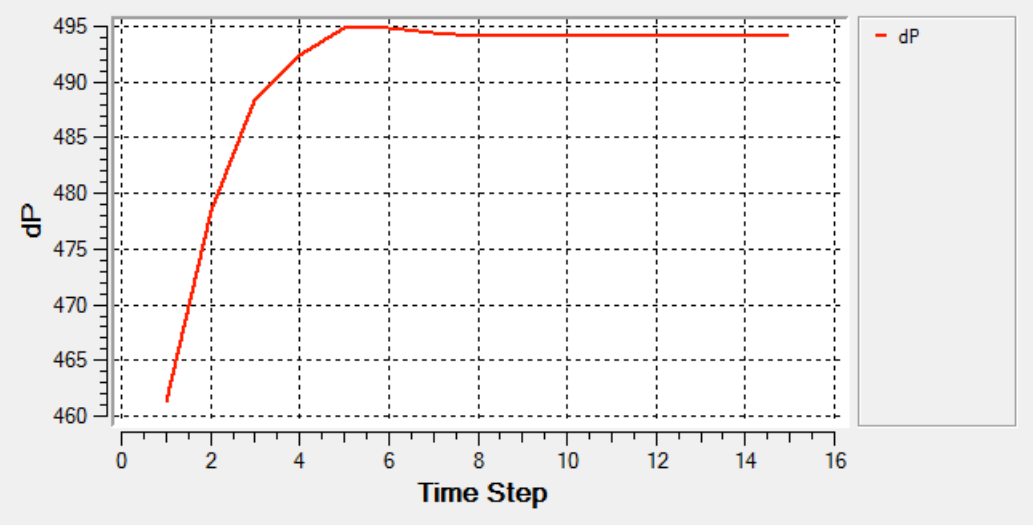


Figure 629:

You can zoom into the plot by clicking then selecting an area at the end of the curve. As shown in the figure below, for the given flow rate of 525.35 m^3/hr (0.146 m^3/sec), the pressure rise is 494.182 Pa.

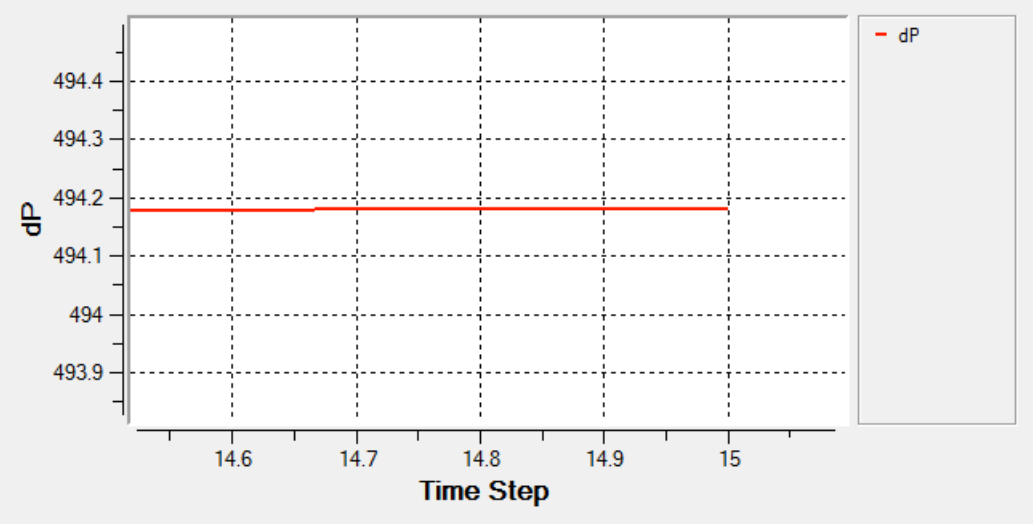


Figure 630:

Summary

In this tutorial you successfully learned how to set up and solve a simulation involving a fan component. You imported the meshed geometry and then assigned the material properties and boundary conditions to all the regions. Once the solution was computed, you defined a user function to create a plot of the pressure rise across the fan component volume.

2.5.4 ACU-T: 5201 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve

Prerequisites

This tutorial introduces you to the workflow for setting up an AcuSolve-MotionSolve co-simulation using HyperMesh Desktop. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T5201_CheckValveCoupled.hm` and `Valve_model.xml` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in Figure 631. It consists of a cylindrical pipe containing water that flows past a check valve with a shutter attached to a virtual spring (not included in the geometry). The inlet pressure varies over time and the movement of the shutter will be determined as a function of the balance of the fluid forces against the reactive force of the spring. The problem is rotationally periodic at 30° increments about the longitudinal axis, and it is assumed that the resulting flow is also rotationally periodic, allowing for modeling with the use of a wedge-shaped section. For this tutorial, a 30° section of the geometry is modeled, as shown in the figure. Modeling a portion of an rotationally periodic geometry leads to reduced computation time while still providing an accurate solution.

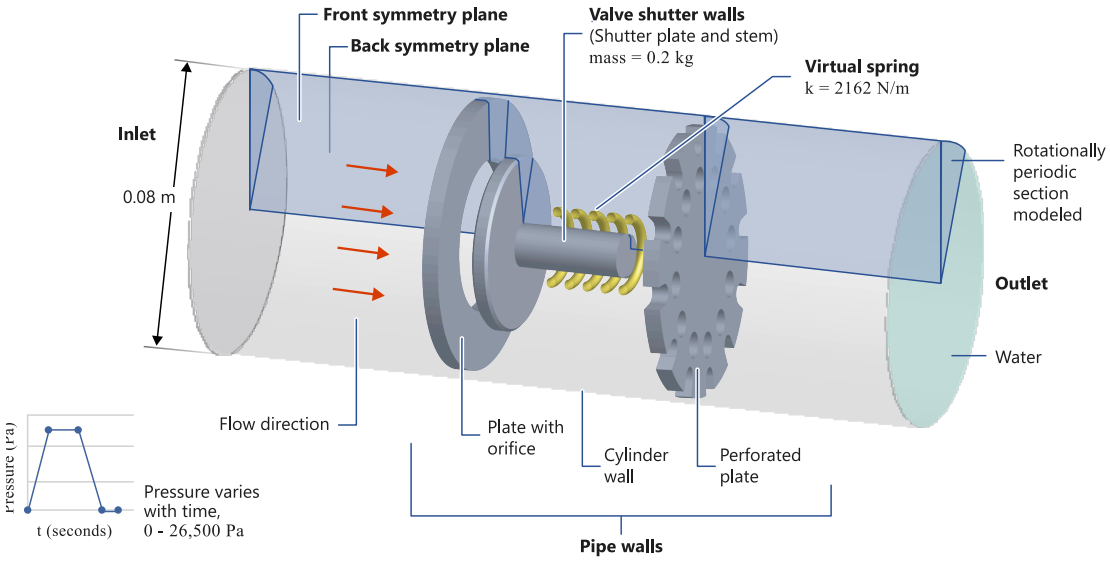


Figure 631: Schematic of Check Valve with Spring-Loaded Shutter

The pipe has an inlet diameter of 0.08 m and is 0.4 m long. The check-valve assembly is 0.085 m downstream of the inlet. It consists of a plate 0.005 m thick with a centered orifice 0.044 m in diameter and a shutter with an initial position 0.005 m from the opening, simulating a nearly closed condition. The shutter plate is 0.05 m in diameter and 0.005 m thick. The shutter plate is attached to a stem 0.03 m long and 0.01 m in diameter. The mass of the shutter and stem is 0.2 kg and its motion is affected by a virtual spring with a stiffness of 2162 N/m. The motion of the valve shutter is limited by a stop mounted on a perforated plate downstream of the shutter.

Modeling the geometry as a 30° section requires that the fluid model is set up to be consistent with the rigid-body model. Since only $^{1}/_{12}$ of the rigid body is modeled, the forces computed by AcuSolve that act on the valve shutter represent $^{1}/_{12}$ of the actual force on the device. The rigid-body-dynamics model was set up in MotionSolve with scaled settings of mass and spring stiffness to account for the fact that you are only modeling a small section of the full geometry. Additional information regarding the setup of this problem in MotionSolve is provided in the MotionSolve documentation.

The fluid in this problem is water, which has a density (ρ) of 1000 kg/m^3 and a molecular viscosity (μ) of 1 X 10^{-3}.

At the start of the simulation the flow field is stationary. Flow is driven by the pressure at the inlet, which varies over time as a piecewise linear function shown in Figure 632. As the pressure at the inlet rises, the flow will accelerate as the valve opens. The turbulence viscosity ratio is assumed to be 10.

The initial inlet pressure is 0 Pa. At 0.002 s the pressure begins to ramp up and reaches 29,000 Pa at 0.05 s. The pressure is held at 29,000 Pa and begins to ramp back down starting at 0.2 s, reaching the initial pressure at 0.25 s where it remains for the rest of the simulation.

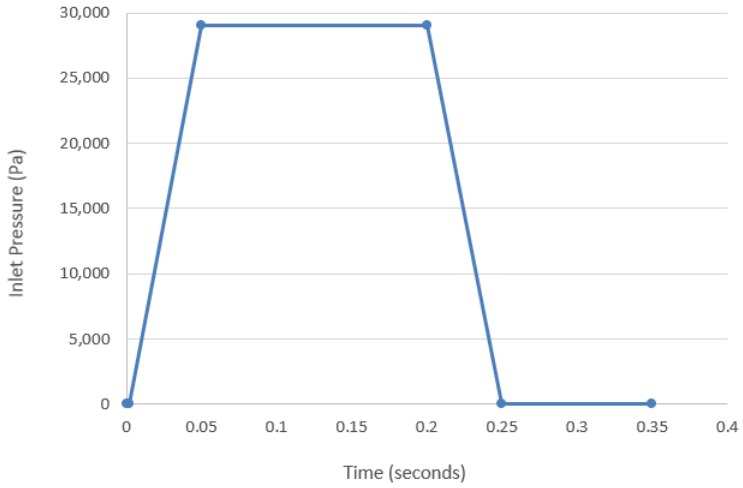


Figure 632: Transient pressure at the inlet

Prior simulations of this geometry indicate that the average velocity at the inlet reaches a maximum of 0.98 m/s. At this velocity, the Reynolds number for the flow is 78,400. When the Reynolds number is above 4,000 it is generally accepted that flow should be modeled as turbulent. Mesh motion will be modeled using arbitrary mesh movement (arbitrary Lagrangian-Eulerian mesh motion).

For this case, the transient behavior of interest occurs in the time it takes for the pressure to ramp up and ramp back down, which is given by the transient pressure profile. To allow time for the spring to recover additional time will be simulated. For this tutorial 0.1 s is added after the pressure drops back to initial conditions for a total duration of 0.35 s.

Another critical decision in a transient simulation is choosing the time increment. The time increment is the change in time during a given time step of the simulation. It is important to choose a time increment that is short enough to capture the changes in flow properties of interest, but does not require unnecessary computation time. The change in inlet pressure from initial conditions to maximum occurs over 0.048 s. A time increment of 0.002 s would allow for excellent resolution of the transient changes without requiring excessive computational time.

Open the HyperMesh Model Database

1. Start HyperMesh Desktop and load the AcuSolve user profile.
   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.
   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T5201_CheckValveCoupled.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.

5. Create a new directory named `CheckValve_Coupled` and navigate into this directory.
   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Valve_Coupled` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.
8. Move the `Valve_model.xml` file (provided along with the input file) to the problem directory.

Set the General Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, change the Analysis type to **Transient**.
3. Set the Turbulence model to **Spalart Allmaras**.
4. Set the Mesh type to **Arbitrary Mesh Movement (ALE)**.
5. Turn External Code **On**.

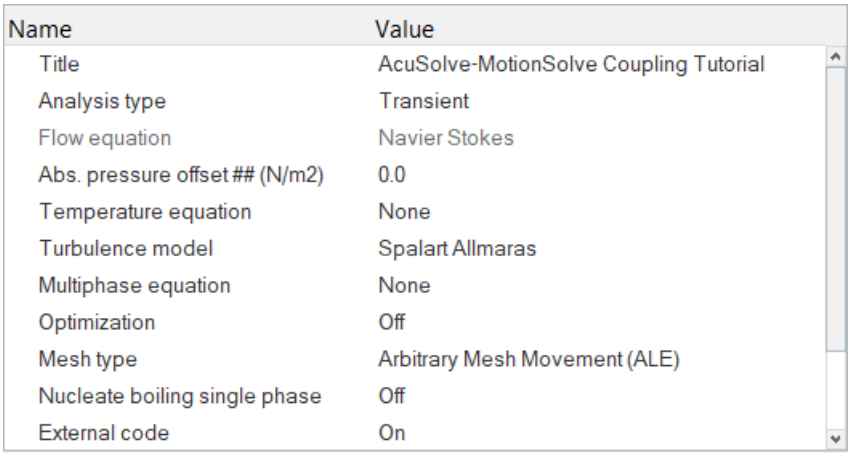


Figure 633:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor, set the Max time steps to `0`.
3. Set the Final time to `0.35` sec.
4. Set the Initial time increment to `0.002` sec.
5. Verify that the Convergence tolerance is set to `0.001`.
6. Set the Max stagger iterations to `3`.
7. Change the Relaxation factor to `0`.
8. Turn External code **On** and verify that the Flow, Turbulence and Mesh options are turned on.

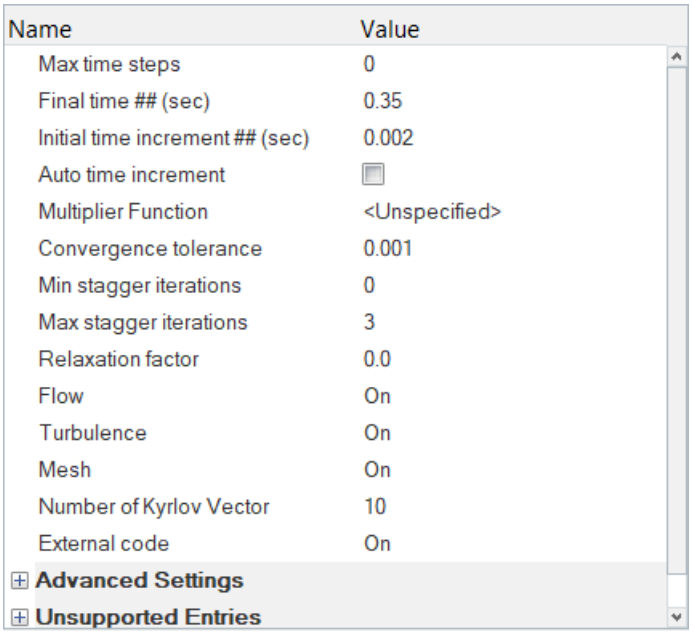


Figure 634:

Set External Code Parameters for Communication with MotionSolve

In this simulation, MotionSolve is being used to calculate the motion of the valve in response to the fluid loads calculated by AcuSolve. In order to achieve this, AcuSolve and MotionSolve will communicate through a socket connection. This communication approach enables the two codes to be run on separate machines and even different operating systems.

1. In the Solver Browser, under 01.Global, click **08.EXTERNAL_CODE**.
2. In the Entity Editor, verify that the Communication option is set to **Socket**.
3. Verify that the **Socket initiate** check box is not activated.

   This will ensure that the external code (MotionSolve) will open the socket and AcuSolve will "listen" for a connection on the indicated socket port.
4. Verify that the Socket host is set to **localhost**.

   This indicates that AcuSolve and MotionSolve are running on the same machine. If MotionSolve was running on a different machine, the host name or IP address would need to be supplied here.
5. Enter `48000` as the Socket port.

   This is the default port used for communication between AcuSolve and MotionSolve.

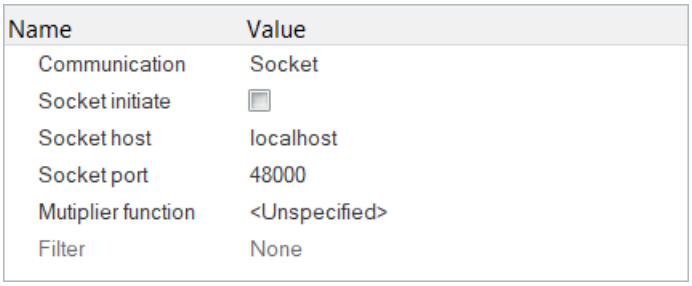


Figure 635:

Define the Nodal Output Frequency

1. In the Solver Browser, expand **17.Output** then click **NODAL_OUTPUT**.
2. In the Entity Editor, set the Time step frequency to `1`.

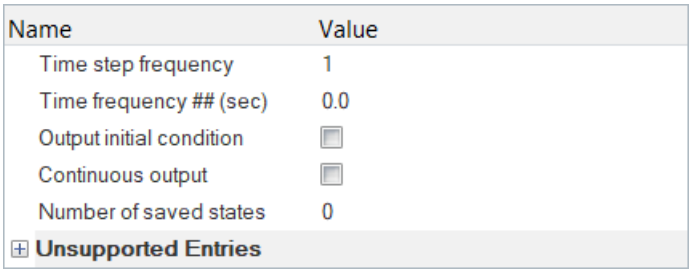


Figure 636:

3. Save the model.

Set the Boundary Conditions

Create a Multiplier Function for Inlet Pressure

In this step, you will create a multiplier function for the pressure at the inlet, which will then be applied at the inlet later in the tutorial. The inlet pressure starts at 0 Pa, ramps up to 29,000 Pa, is then held steady briefly, and then ramps back to 0 Pa.

1. Go to the Model Browser. Right-click anywhere in empty space and select **Create** > **Curve** from the context menu.
2. In the **Curve editor** dialog, click **New**.
3. In the panel area, enter `inlet_pressure` as the Name and then click on **proceed**.
4. In the **Curve editor** dialog, enter the following X,Y values.

X	Y
0.0	0.0
0.002	0.0
0.05	29000.0
0.2	29000.0
0.25	0.0
0.35	0.0

Figure 637:

5. Click **Update** to plot the curve on the graph in the dialog.
6. Click **Close**.
7. Go to the Solver Browser, right-click on **05.Multiplier_Function** and select **Create**.
8. Name the multiplier function `inlet_pressure`.
9. In the Entity Editor, change the Multiplier Function Type to **Piecewise Linear**.
10. For Curves, select **inlet_pressure**.
11. Verify that the Curve fit variable is set to **Time** and the Evaluation Type is set to **Per Time Step**.

Name	Value
Solver Keyword	MULTIPLIER_FUNCTION
Name	inlet_pressure
Type	Multiplier_Function
Multiplier Function Type	Piecewise Linear
Curves	(1) inlet_pressure
Curve fit variable	Time
Evaluation Type	Per Time Step
⊞ **Unsupported Entries**	

Figure 638:

Set the Boundary Conditions

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **FLUID**.

b) Set the Material to **Water_HM**.

c) Leave the remaining default options.

Name	Value
Name	Fluid
Type	FLUID
Material	(2) Water_HM
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Mesh motion	<Unspecified>
Mesh distortion correction factor	0.0
Mesh distortion tolerance	0.0
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 639:

3. Click **Inflow**. In the Entity Editor,

a) Change the Type to **INFLOW**.

b) Turn **On** Show advanced features.

c) Change the Inflow type to **Stagnation Pressure**.

d) Set the Stagnation Pressure to `1` N/m^2.

e) Set the Stagnation Pressure multiplier function to **inlet_pressure**.

f) Set the Turbulence input type to **Viscosity Ratio**.

g) Set the Material to **Water_HM** in the top portion of the Entity Editor.

h) Set the Turbulence viscosity ratio to `10`.

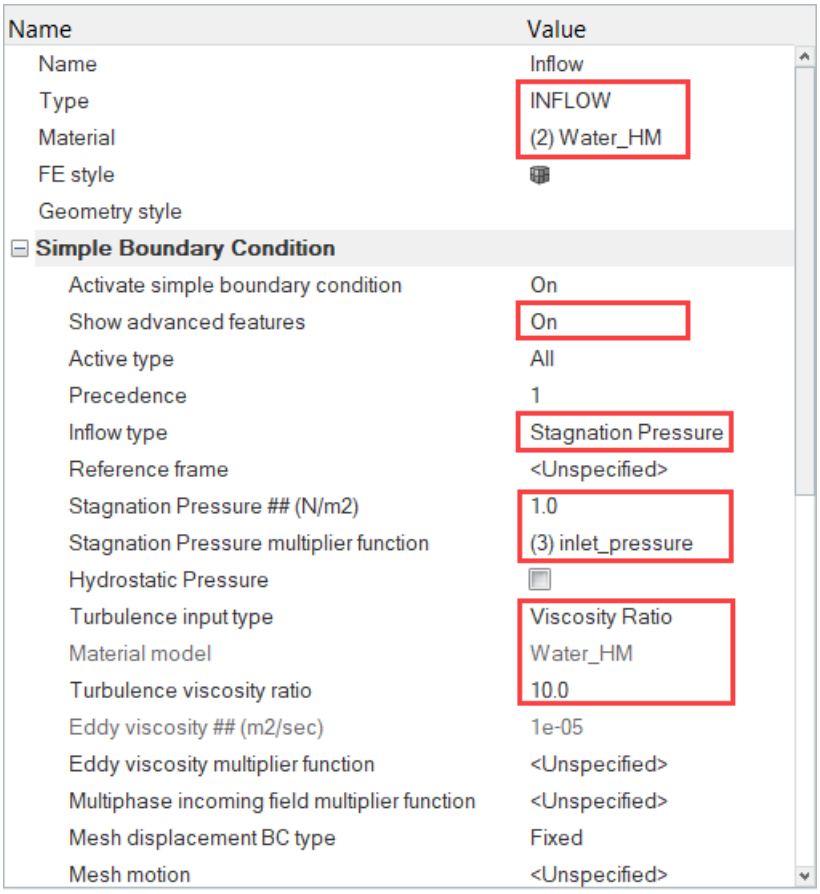


Figure 640:

4. Click **Outflow**. In the Entity Editor, change the Type to **OUTFLOW**.

Name	Value
Name	Outflow
Type	OUTFLOW
FE style	
Geometry style	
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Pressure ## (N/m2)	0.0
Pressure loss factor	0.0
Hydrostatic pressure	
Back flow conditions	
Mesh displacement BC type	Fixed
Mesh motion	<Unspecified>
Integrated Boundary Conditions	
Radiation Surface	
Solar Radiation Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 641:

5. Click **Valve_wall**. In the Entity Editor,
   a) Change the Type to **EXTERNAL_CODE_SURFACE**.
   b) Set the Coupling type to **Rigid body**.
   c) Verify that the Coupling direction is set to **Both**.
   d) Enter `Model-Shutter Body` as the name of the Rigid body.

      The name entered here needs to match the body name in the prepared MotionSolve model, `Valve_model.xml`, to ensure that information about forces and motion for this surface match up between AcuSolve and MotionSolve.

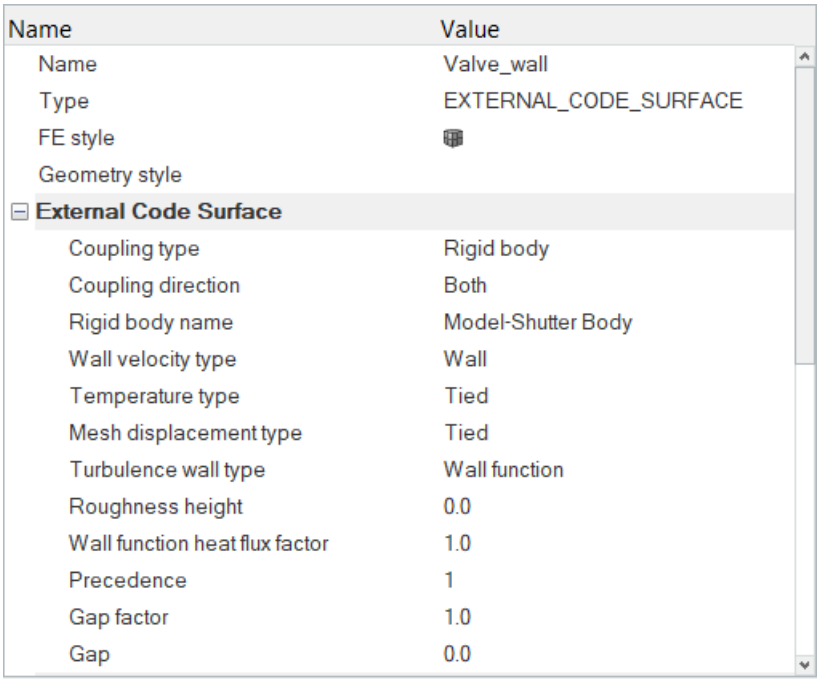


Figure 642:

6. Click **Outflow**. In the Entity Editor, verify that the Type is set to **WALL**.

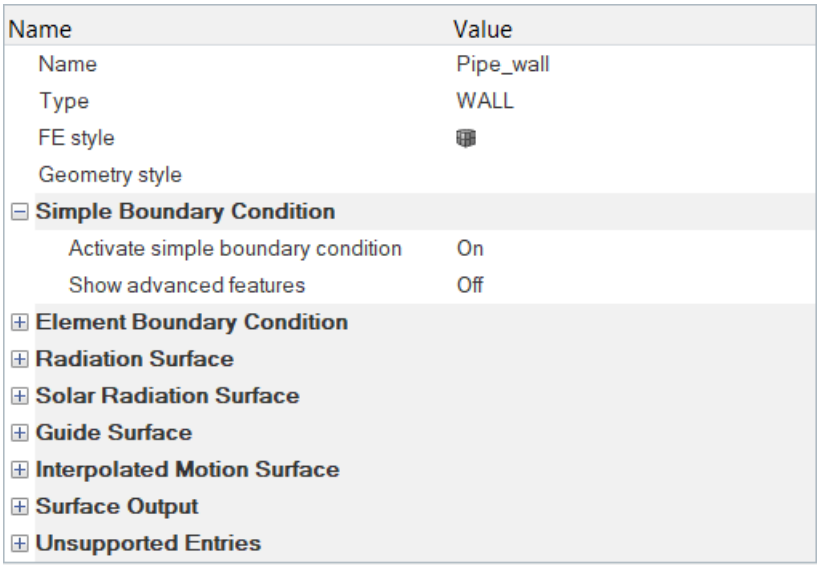


Figure 643:

7. Click **Front_symmetry**. In the Entity Editor,

a) Change the Type to **SYMMETRY**.

b) Change the Mesh displacement BC type to **Slip**.

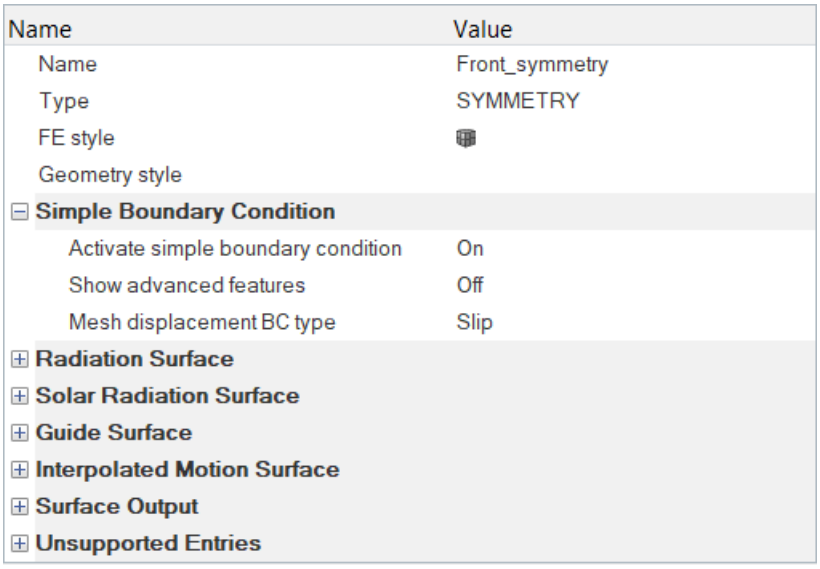


Figure 644:

8. Similarly, click **Back_symmetry**, set the Type to **SYMMETRY**, and the Mesh displacement BC type to **Slip**.
9. Save the model.

Compute the Solution

This tutorial uses a coupled solution between AcuSolve for the flow field and MotionSolve for the rigid-body-dynamics. Solving the problem for this tutorial involves two steps:

1. Start AcuSolve
2. Start MotionSolve

The next sets of steps provide instructions for these two tasks.

Run AcuSolve

In this step, you will launch AcuSolve to compute a solution for this case.

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Set the Number of processors to `4`.
4. Leave the remaining options as default and click **Launch** to start the solution process.

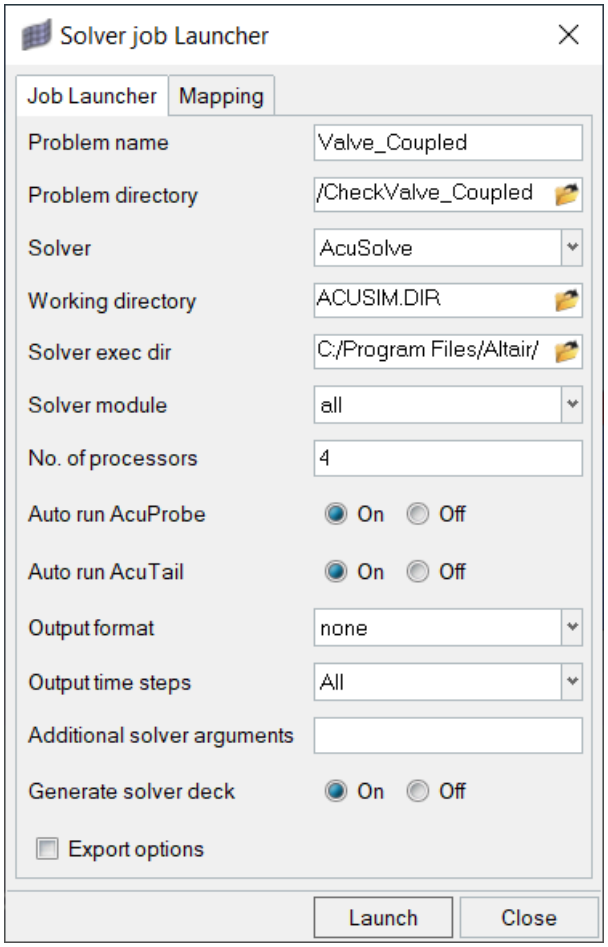


Figure 645:

Run MotionSolve

In the next steps you will start MotionSolve and provide settings for communication with HyperMesh.

1. Run **Start** > **All Programs** > **Altair HyperWorks <version>** > **MotionSolve** to open the **HyperWorks Solver Run Manager**.
2. Click beside the Input file(s) field, browse to the location where you saved `Valve_model.xml`, and open it.
3. Click ... to open the **Available Options** dialog.
4. Enable the **-as_cosim** option to indicate coupling between MotionSolve and HyperMesh.

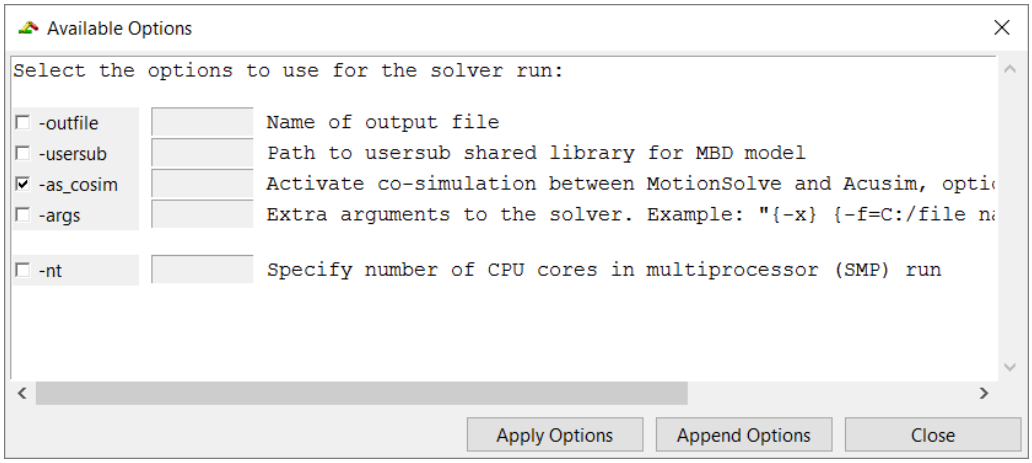


Figure 646:

5. In the **-as_cosim** dialog that appears, click **None**.
6. In the **Available Options** options dialog, click **Apply Options** and close the dialog.
7. Click **Run** to start MotionSolve.

   As the solution progresses, a HyperWorks Solver View window will open. Solution progress is reported in this window. The HyperMesh AcuTail window will also update as the solution progresses.

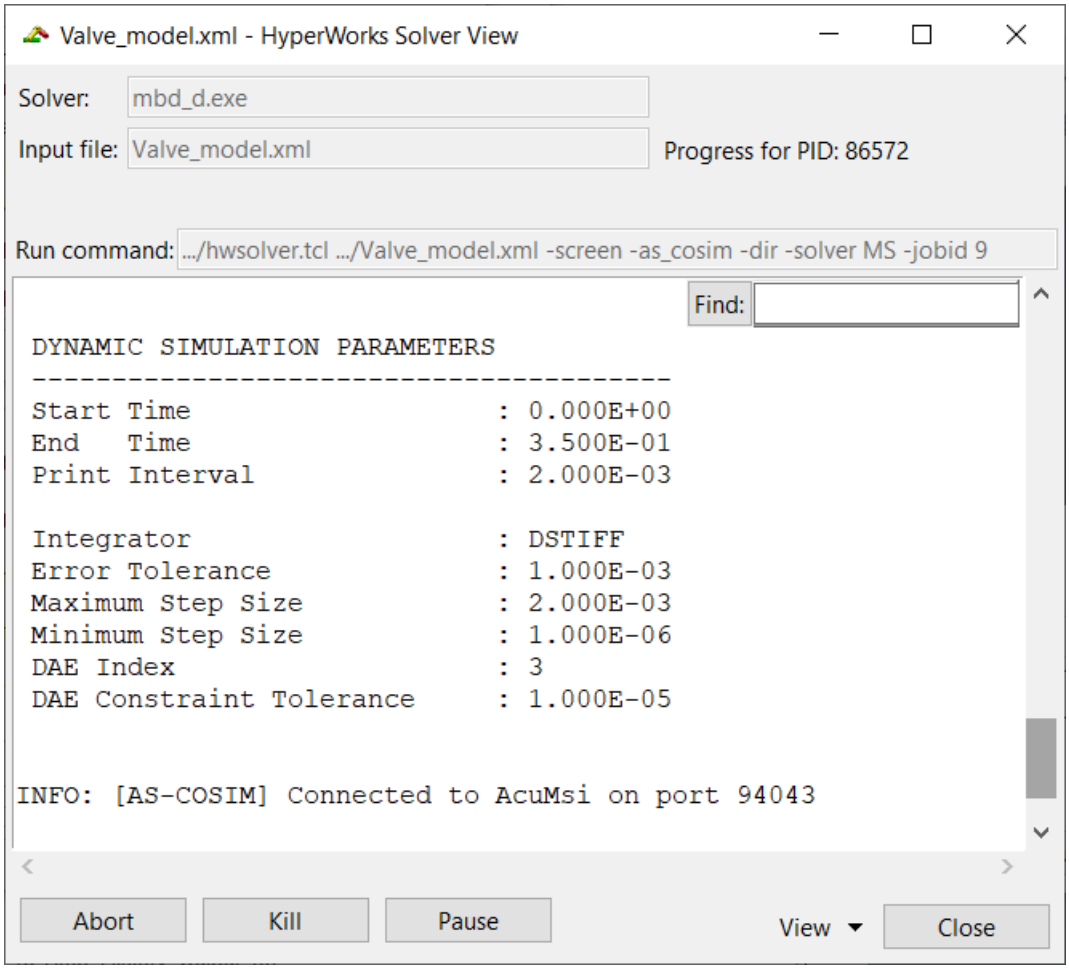


Figure 647:

As the solution progresses, you can monitor the progress with AcuProbe, as detailed in the next steps.

Monitor the Solution with AcuProbe

While AcuSolve is running, you can monitor flow characteristics such as inlet pressure, displacement of the valve, and velocity of the valve, using AcuProbe.

Once the MotionSolve run launched, the AcuProbe window will be launched automatically after the first time step.

1. In the AcuProbe Data Tree, expand **Surface Output** > **Inflow** > **Pressure**.
2. Right-click on **pressure** and select **Plot**.
3. Similarly, expand **Valve_wall** > **Geometry** under Surface Output.
4. Right-click on **mesh_x_displacement** and select **Plot**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

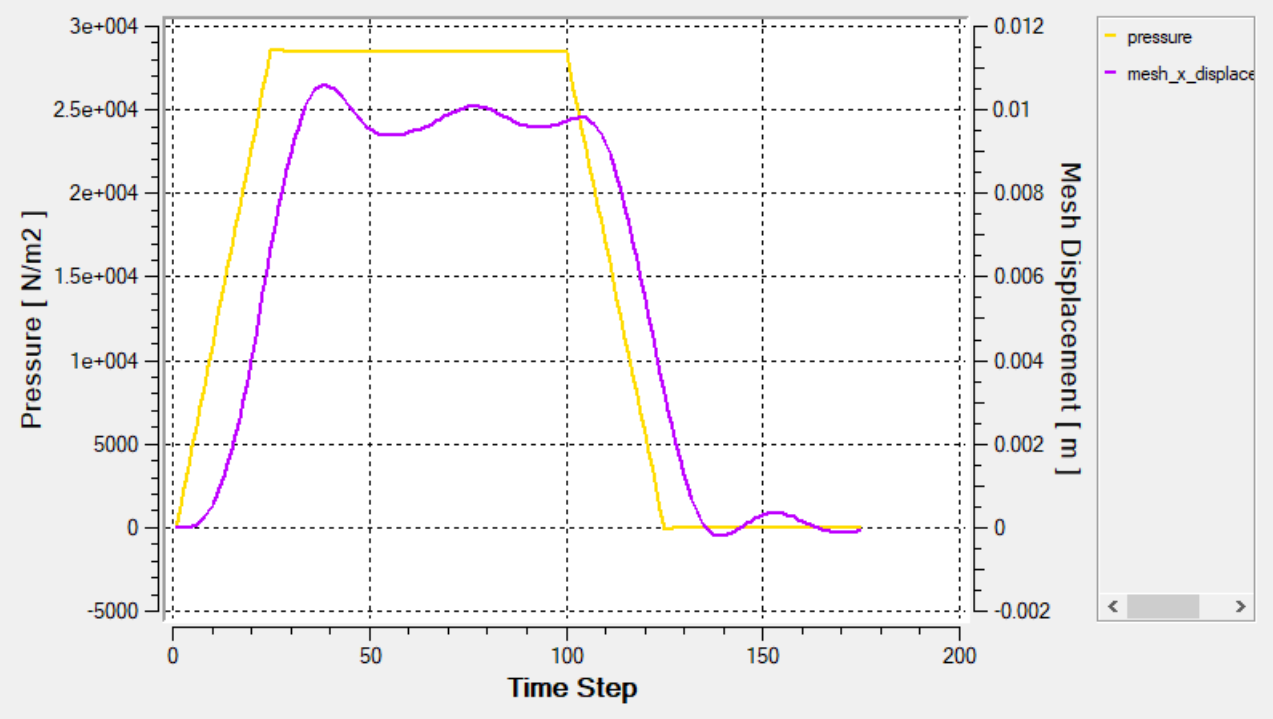


Figure 648:

In the next steps you will create a plot of the x-velocity of the valve walls.

5. Turn off the plot of pressure at the inlet by right-clicking on **Inflow** and selecting **Plot None**.
6. Expand **Valve_wall** > **Momentum**, right-click on **x_velocity**, and select **Plot**.

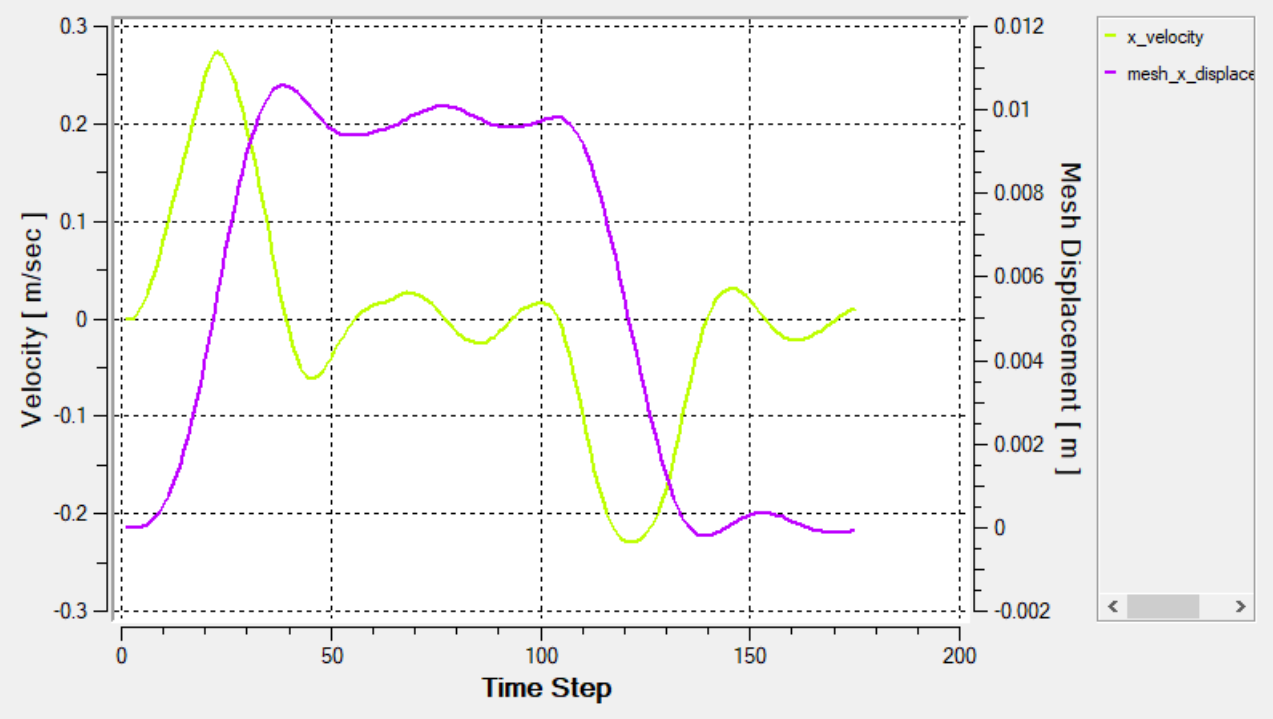


Figure 649:

Post-Process the Results with HyperView

In this step, you will create an animation of the valve motion as the water flows across the valve. Once the solver run is complete, close the AcuProbe and AcuTail windows. In the HyperMesh Desktop window, close the AcuSolve Control tab and save the model.

Switch to the HyperView Interface and Load the AcuSolve Model and Results

1. In the HyperMesh Desktop window, click the **ClientSelector** drop-down in the bottom-left corner of the graphics window.

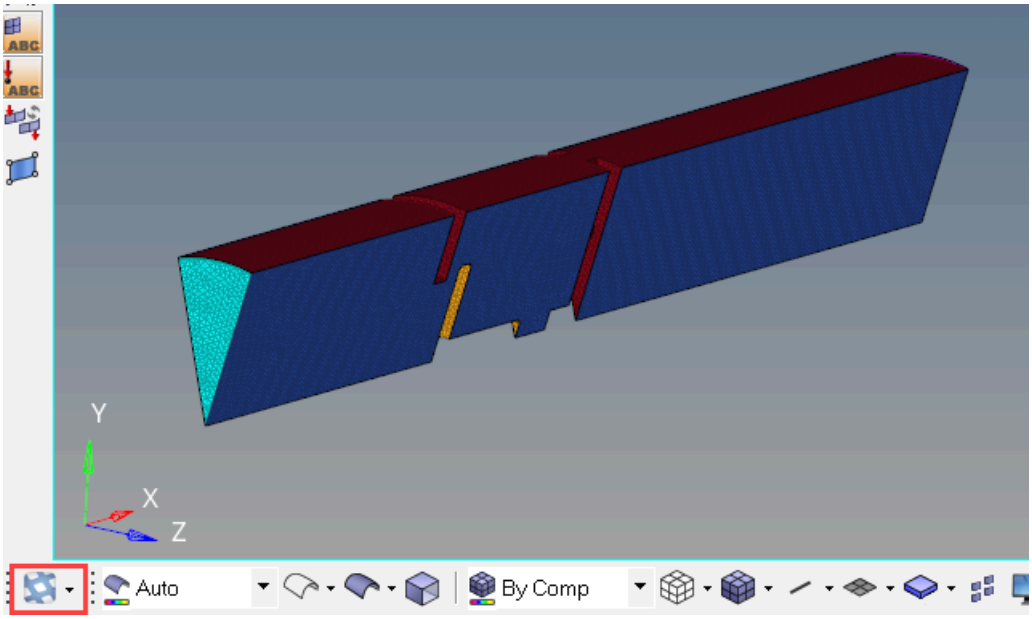


Figure 650:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.
   The interface is changed to HyperView.

   Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `Valve_Coupled.1.Log`.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create an Animation of Velocity Magnitude

In the next step, you will create an animation of the velocity magnitude on the symmetry plane and then save it.

1. In the Results Browser, expand the list of **Components** then click the **Isolate Shown** icon .
2. Click the **Front_symmetry** component to turn off the visibility of all the components except the front symmetry surface.

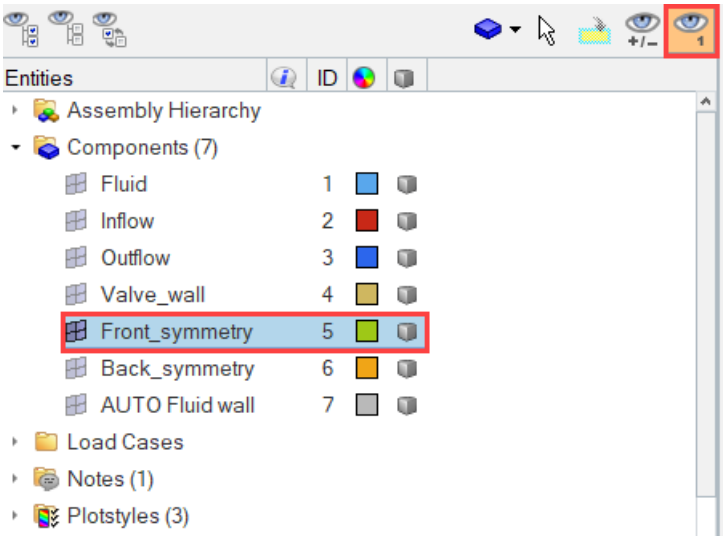


Figure 651:

3. Orient the display to the xy-plane by clicking on the Standard Views toolbar.
4. Click on the Results toolbar to open the Contour panel.
5. In the panel area, change the Result type to **Velocity (v)** and verify that the drop-down below it is set to **Mag** (magnitude).
6. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
7. Click **Apply**.
8. In the panel area, under the Display tab, turn off the Discrete color option.



Figure 652:

9. Go to the **Legend** tab and then click **Edit Legend**.
10. In the **Edit Legend** dialog, change the Type to **Dynamic scale** and the Numeric format to **Fixed** then click **OK**.

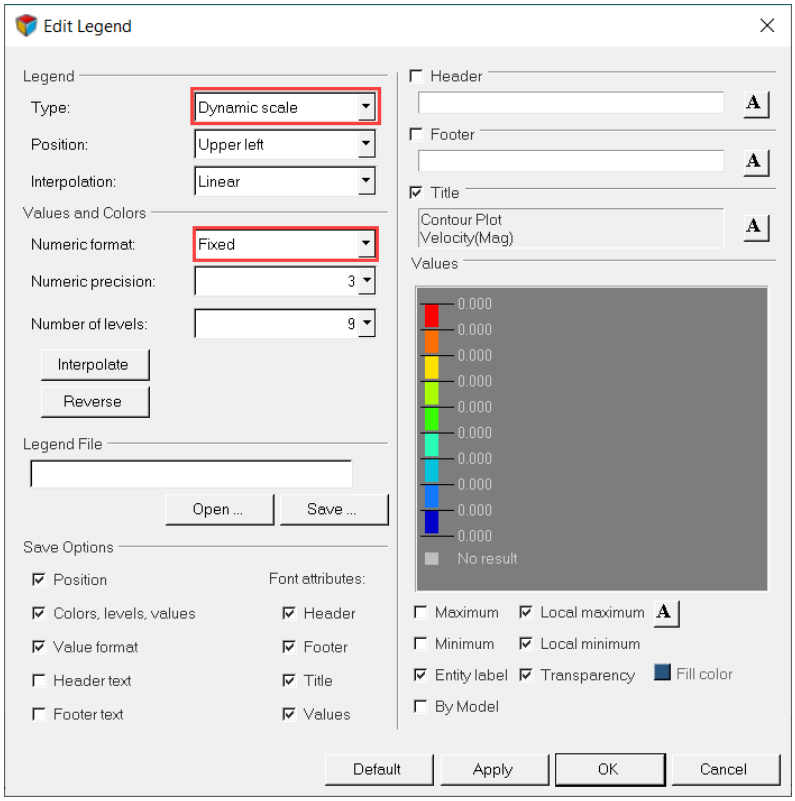


Figure 653:

11. Click on the Animation toolbar to play the velocity magnitude animation on the front symmetry plane.

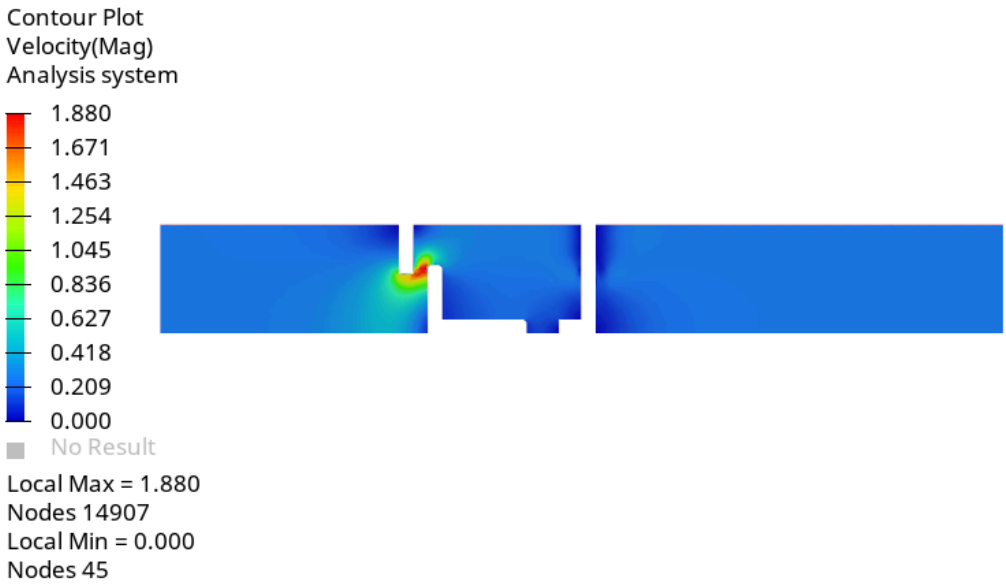


Figure 654:

12. On the ImageCapture toolbar, click on the **Capture Graphics Area Video** icon .
13. In the **Save Graphics Area Video As** dialog, browse to the directory where you want to save the animation, give a name to the video (ex: velocity mag), then click **Save**.

Display Pressure and Velocity Contours on a Section Cut

In the next step, you will create a section cut on the mid-z plane and then display the pressure contours and velocity vectors on that cross section.

1. In the Results Browser, turn off the display of all the Components except the Fluid component.

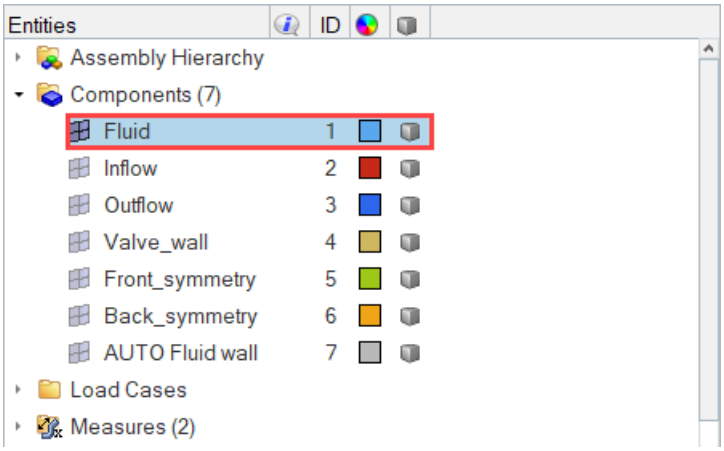


Figure 655:

2. Click on the Results toolbar to open the Contour panel.
3. In the panel area, change the Result type to **Pressure (s)**.
4. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
5. Click **Apply** to create the contour plot of pressure.
6. In the panel area, under the **Result** tab, activate the **Overlay result display** check box (if not already selected).

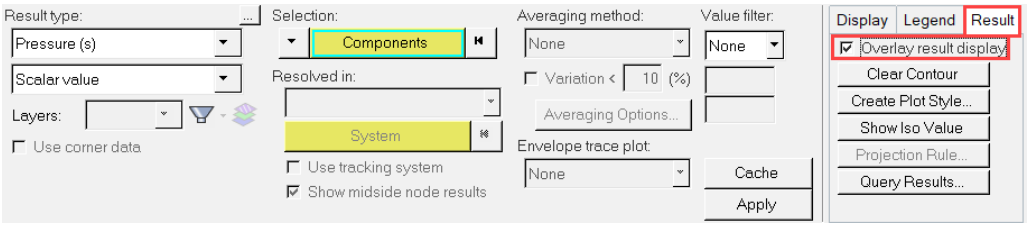


Figure 656:

7. Click the **Section cut** icon on the HV-Display toolbar.
8. In the panel area, click **Add** to create a new section cut named Section 1.
9. In the Define plane section, set the axis to **Z Axis** then click **Apply**.

10. Set the Z Base coordinate to `0` and press **Enter**.
11. Change the Display options from Clipping plane to **Cross section**.
12. Click **Gridline**. In the **Gridline Options** dialog, deactivate the **Show** check box under Grid line then click **OK**.

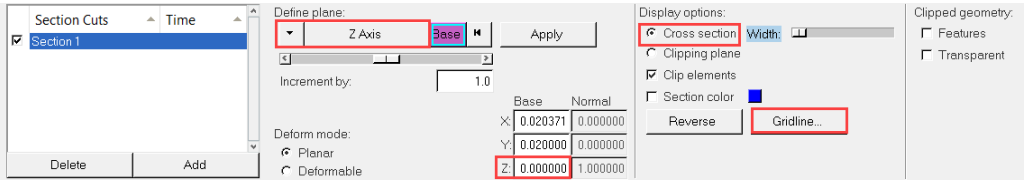


Figure 657:

13. Click on the Results toolbar to open the Vector panel.
14. In the panel area, set the Result type to **Velocity (v)**.
15. Click the **Selection** drop-down and select **Sections** from the list of options.

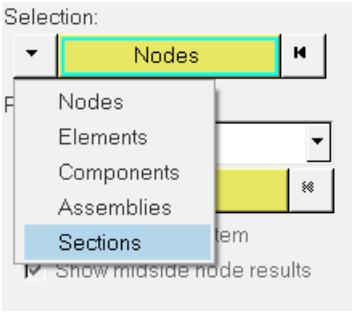


Figure 658:

16. Click the **Sections** entity selector then select **All**.
17. In the panel area, activate the **Overlay result display** check box (if not set already).
18. Click **Apply**.
19. Under the **Plot** tab, verify that only the **X+Y+Z Resultant** option is selected.
20. Go to the **Display** tab, set the Size scaling option to **Uniform**, and enter a value of `0.002` in the size field.
21. Set the Color by option to **Direction** and set the X+Y+Z color to **White**.

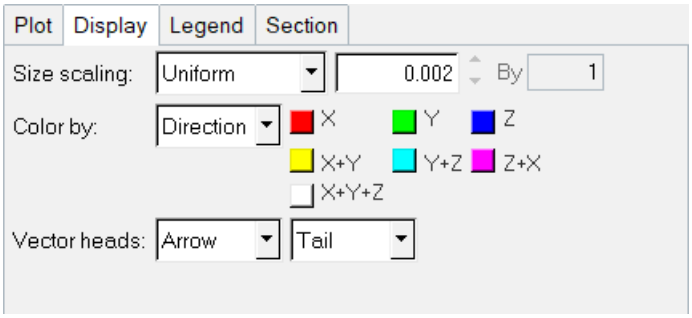


Figure 659:

22. Go to the **Section** tab, activate the **Projected** check box, then click **Apply**. The vector plot should look like the one shown in the figure below.

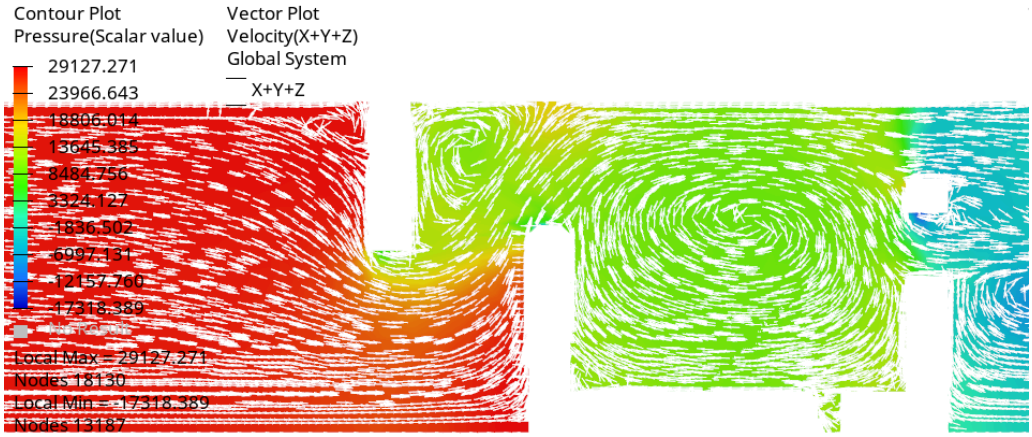


Figure 660:

The result at 0.156 sec is shown.

Summary

In this tutorial, you learned the basic workflow to set up a co-simulation using AcuSolve and MotionSolve. The tutorial introduced you to the steps involved in setting up external code communication between AcuSolve and MotionSolve using HyperMesh Desktop and then running the simulation and post-processing the results using HyperView. You also learned how to create a vector plot on an existing contour plot on a cut plane.

2.6 Porous Medium

2.6.1 ACU-T: 6010 Flow Through Porous Medium

Prerequisites

This tutorial provides the instructions for setting up, solving and viewing results for simulation of flow through a porous medium. Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of AcuSolve, HyperView, and HyperMesh. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T6010_PorousMedia.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The problem to be addressed in this tutorial is shown schematically in the figure below. It consists of a cylindrical channel with a porous medium in the flow section. As the flow passes through this section, a pressure drop is observed. In this simulation, an inlet velocity will be assigned to the flow and pressure drop across the porous medium will be calculated. The length of the porous section is 0.06 m and the fluid is an imaginary air-like fluid with a density of 1 kg/m3 and a molecular viscosity of 0.001 kg/m-s. The inlet velocity of the flow is 0.2 m/s.

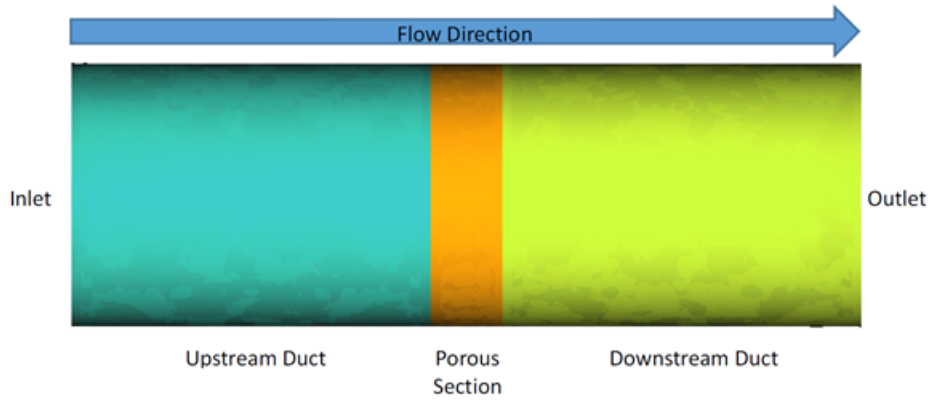


Figure 661:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon  located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T6010_PorousMedia.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `PorousMedia` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `PorousMedia` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the General Simulation Parameters

In this step, you will set the simulation parameters that apply globally to the simulation.

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, verify that the Analysis type is set to **Steady State** and the Turbulence model is set to **Laminar**.

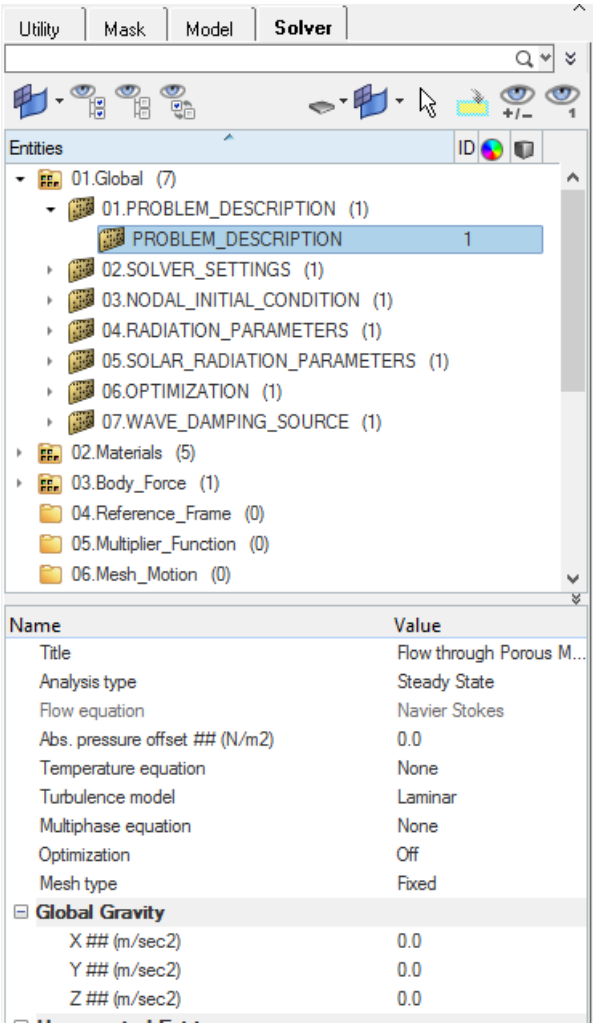


Figure 662:

Set Up Boundary Conditions and Material Model Parameters

In this step, you will start by modifying the material properties of Air and then create a material model for the porous medium. Then, you will assign the surface boundary conditions and material properties for all the fluid volumes.

Create the Material Model

1. In the Solver Browser, expand **02.Materials** > **FLUID** then click on **Air_HM**.
2. In the Entity Editor, change the Density to `1` kg/m^3 and the Viscosity to `0.001` kg/m-sec.

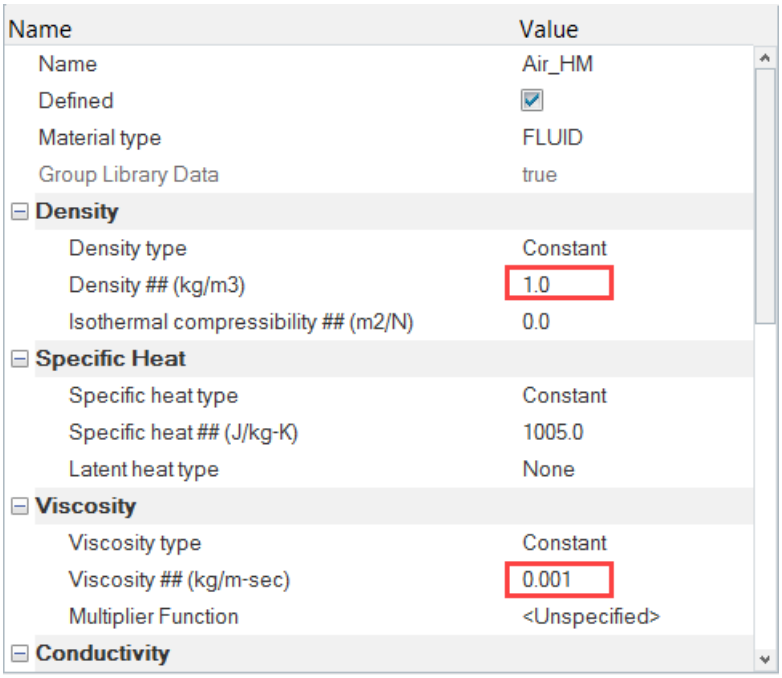


Figure 663:

3. Right-click on **Air_HM** in the Solver Browser and select **Duplicate**.
4. Name the new material `Porous`.
5. In the Entity Editor, scroll down to the Porosity tab and set the Porosity type to **Constant**.
6. Enter the values `1`, `0.001`, and `0.001` for the Direction 1 permeability, Direction 2 permeability and Direction 3 permeability, respectively.
7. Set the value of Darcy coefficient to `4166.67`.
8. Set the Forchheimer coefficient to `8.33`.

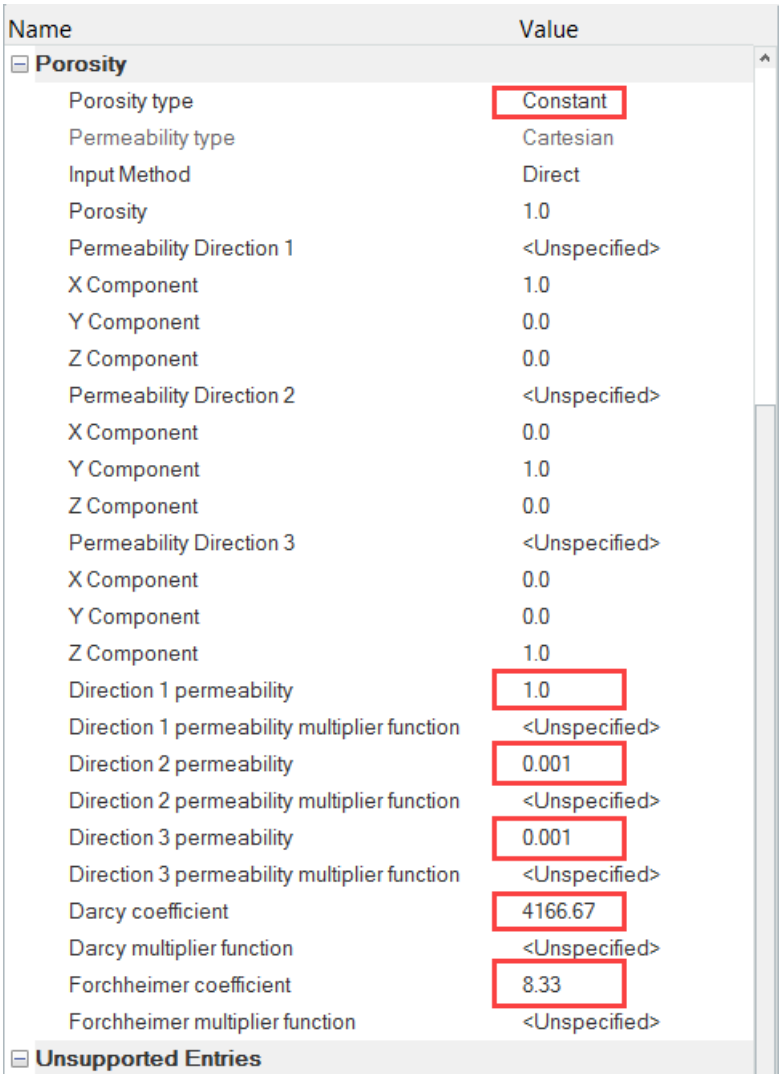


Figure 664:

9. Save the model.

Assign Boundary Conditions and Material Properties

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Inlet**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Set the Inflow velocity type to **Cartesian**.
   c) Set the X velocity to `0.2` m/sec

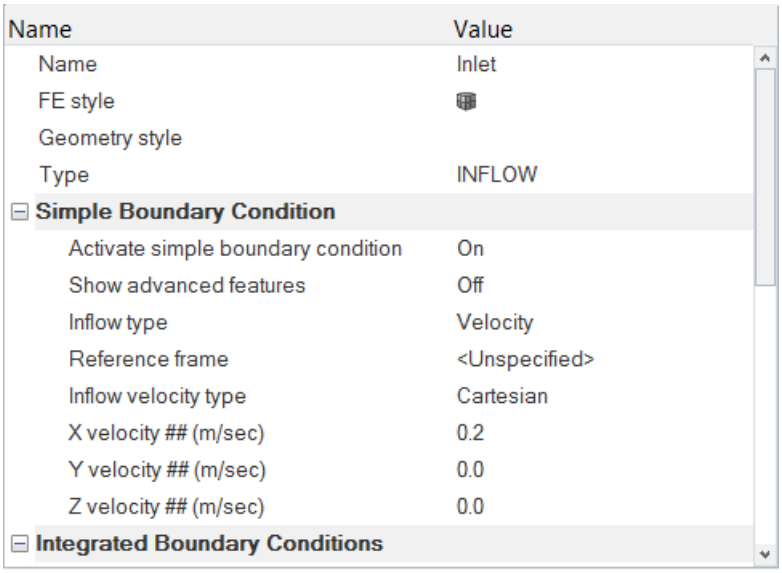


Figure 665:

3. Click **Outlet**. In the Entity Editor, change the Type to **OUTFLOW**.

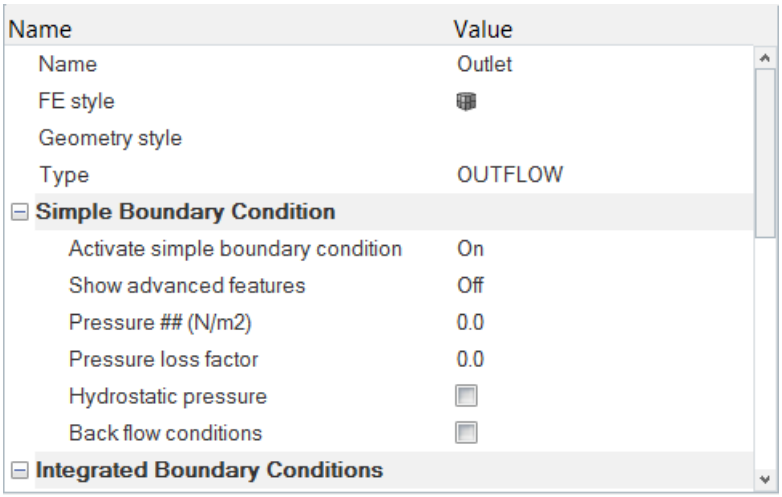


Figure 666:

4. Click **Walls**. In the Entity Editor, verify that the Type is set to **WALL**.

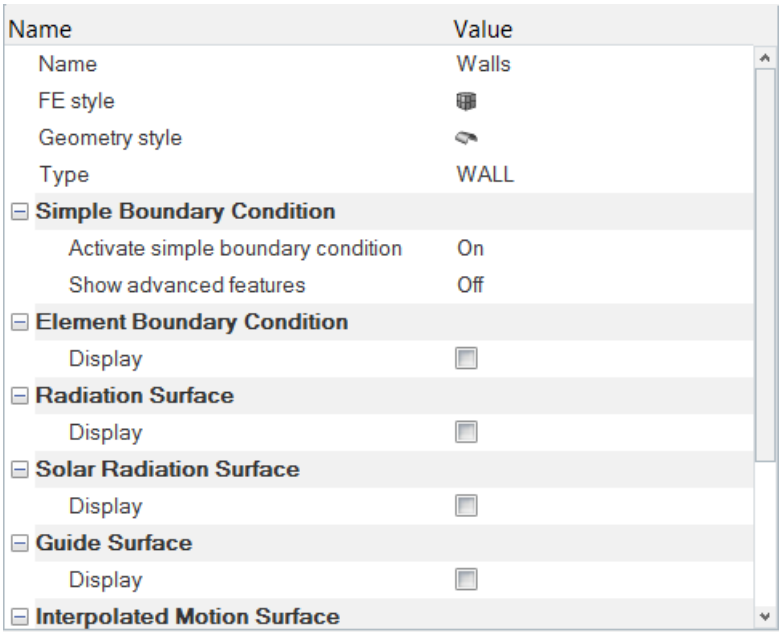


Figure 667:

All the internal surfaces, such as the inlet and outlet of the porous section and the external walls of pipe surface, can be grouped into one single surface set. **Auto_Wall**, which is an advanced feature in AcuSolve, re-groups these elements into internal and external surfaces for each volume and writes the surface output accordingly. This process is done internally, thereby reducing the number of steps in the workflow.

5. Click **Fluid_Upstream**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Select **Air_HM** as the Material.

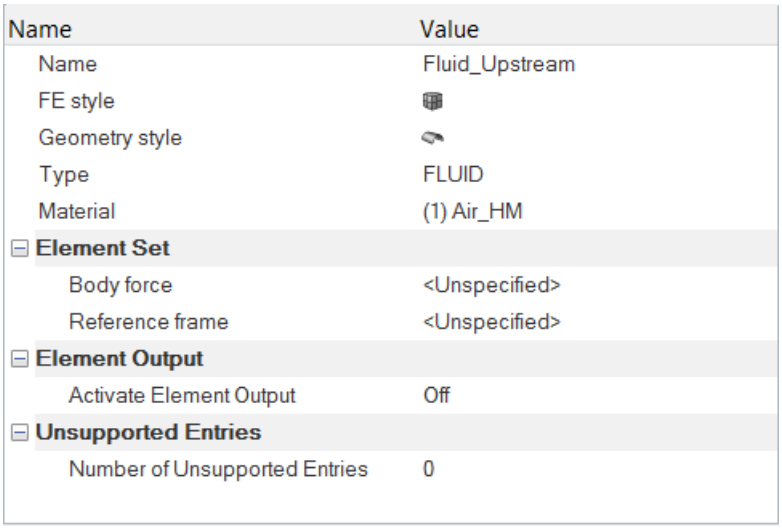


Figure 668:

6. Click **Fluid_Porous**. In the Entity Editor,

a) Change the Type to **FLUID**.
b) Select **Porous** as the Material.

Name	Value
Name	Fluid_Porous
FE style	
Geometry style	
Type	FLUID
Material	(5) Porous
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 669:

7. Click **Fluid_Downstream**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Select **Air_Hm** as the Material.

Name	Value
Name	Fluid_Downstream
FE style	
Geometry style	
Type	FLUID
Material	(1) Air_HM
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 670:

8. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

1. Turn on the visibility of all mesh components.

   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.

   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. The Output time steps can be set to **All** or **Final**. Since this is a steady state analysis, the Final time step output is sufficient.
5. Leave the remaining options as default and click **Launch** to start the solution process.

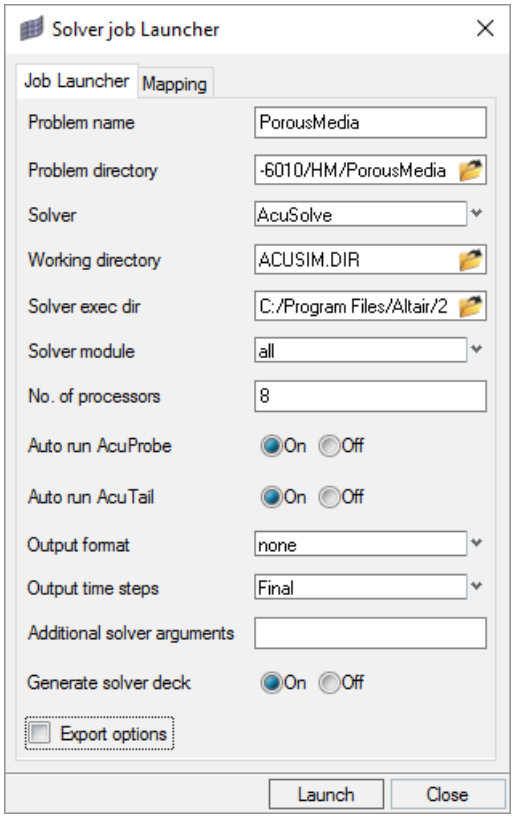


Figure 671:

Post-Process the Results

As the solution progresses, AcuProbe is launched automatically. AcuProbe can be used to monitor different variables over solution time. In this step, you will plot the residual ratio values and then compute the pressure drop across the porous section.

1. In the AcuProbe window, from the Data Tree on the left, expand **Residual Ratio**.
2. Right-click on **Final** and select **Plot All**.

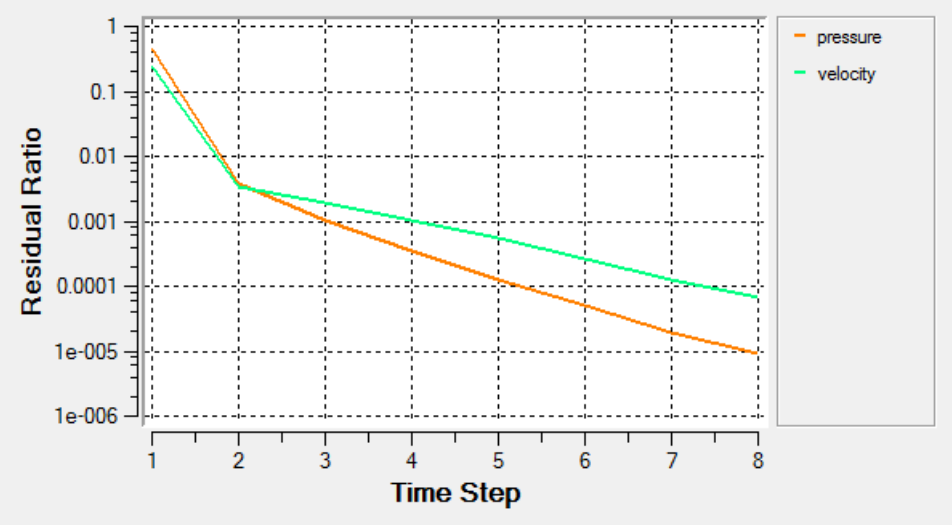


Figure 672:

3. Once the solution is converged, right-click again on **Final** and select **Plot None**.
4. Click the User Function icon from the toolbar.
   The **User Function** dialog opens.
5. In the dialog, enter `dP` as the function name.
6. In the Function field, type `P_In =`.
7. In the Data Tree, expand **Surface Output** > **AUTO Fluid_Porous internal** > **Pressure**.
8. Right-click on **pressure** and select **Copy name**. Paste the value in the Function window after P_In =.

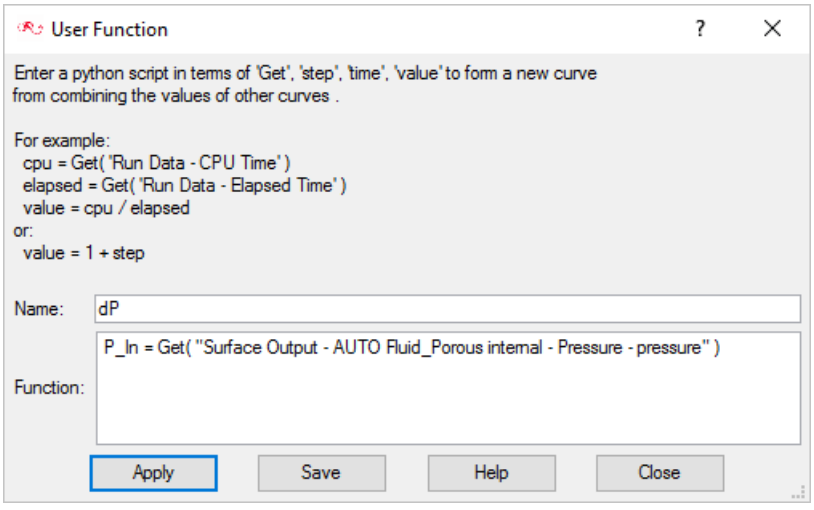


Figure 673:

9. On the next line, type `P_Out =` and repeat the above step for pressure on the **AUTO Fluid_Downstream internal** surface.
10. On the next line, type `value = P_In - P_Out`.

Note: The word "value" is case sensitive and should always be in lower case. If you use a capital letter, an error window appears.

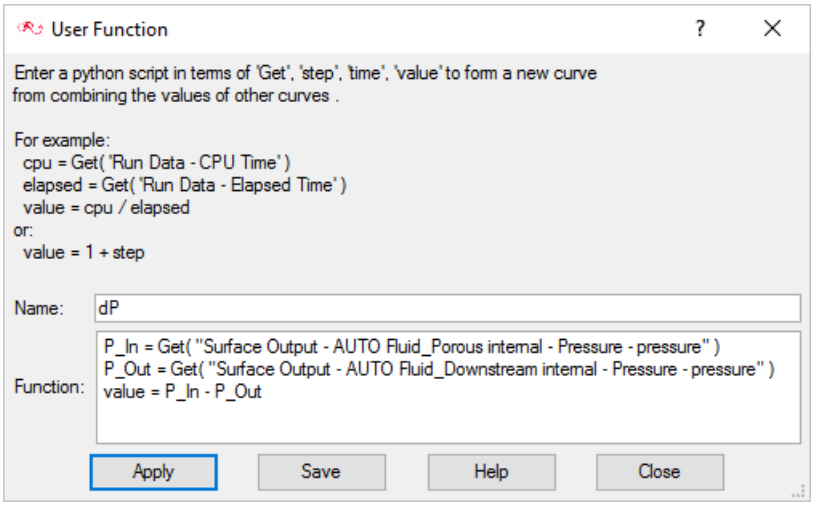


Figure 674:

11. Click **Apply** to display the plot.

Note: You might need to click on the toolbar in order to properly display the plot.

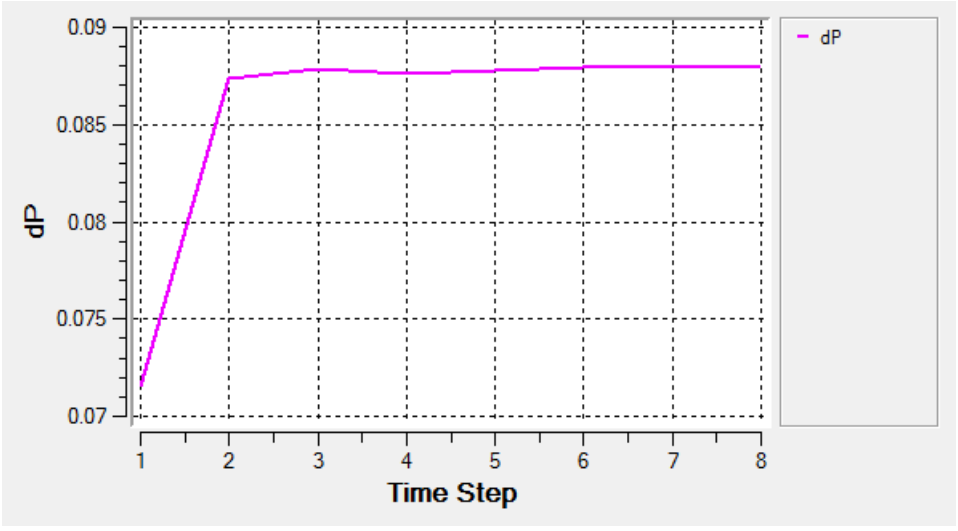


Figure 675:

Summary

In this tutorial, you learned how to set up and solve a problem with porous medium. You started by importing the HyperMesh database and then creating a material model for the porous section. Then, you assigned the boundary conditions and material properties and solved by launching AcuSolve directly from HyperMesh. Finally, you created a pressure drop plot using the user function tool in AcuProbe and calculated the drop in pressure between the inlet and outlet surfaces of the porous section.

2.7 Optimization

2.7.1 ACU-T: 7000 Parametric Optimization with AcuSolve

This tutorial provides instructions for setting up, solving, and viewing results for a parametric optimization problem in AcuSolve. The parametric studies enable you to specify one or more conditions that may vary, such as a sweep over a range of inlet mass flow boundary conditions. The range and frequency of the parameter controlling the mass flow is selected either by simply specifying the desired values or having the program compute suitable values automatically. Several parameters can be used in parametric studies, for instance, controlling the inlet mass flow, the impeller rotational speed, and blade angles of a fan problem. The model used for this tutorial consists of a sphere immersed in a fluid flow. The surface of the sphere is kept hot from an internal heat source. The fluid flow removes the heat from the sphere surface, thus causing the temperature of the fluid to rise. The simulation is run with various values of temperature at the fluid inlet to determine the conditions which result in the outlet temperature closest to a desired value.

The basic steps in a CFD simulation have been discussed in the earlier tutorials. The following additional capabilities of AcuSolve are introduced in this tutorial:

- Setting up a parametric optimization problem in AcuSolve
- Post-processing a parametric optimization problem

In this tutorial, you will:

- Analyze the problem
- Import the HyperMesh model database
- Set general simulation parameters and solver settings
- Set material model parameters and nodal initial conditions
- Set up the optimization study parameters
- Assign surface boundary conditions and material properties
- Run HyperMesh
- Monitor the solution with AcuProbe

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh and AcuSolve. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T7000_ParametricOptimization.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Analyze the Problem

Figure 1 shows the schematic of the problem that will be solved in this tutorial. The fluid enters the channel from the left side, and flows around the sphere immersed in the channel. The sphere surface is maintained at a constant temperature of 500 K. The fluid flow rate is set to be constant. The fluid inlet temperature is varied over a range of values which are a combination of specified values and those chosen by the program on basis of a set of parameters. The fluid model used for this problem is a hypothetical fluid material.

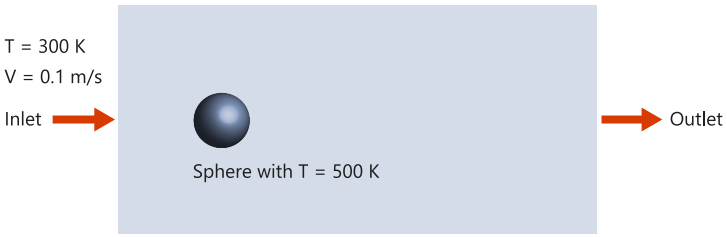


Figure 676:

The inlet temperature here is what is known as the "design variable" for the problem. A target value for outlet temperature is specified, and the desired inlet temperature is the value at which the outlet temperature is closest to the target outlet temperature. The difference between the target outlet temperature and the actual outlet temperature is what is known as the "response variable" for the problem. Since the objective of the problem is to minimize the difference between the outlet temperature and the target outlet temperature, this problem is a "minimization" problem.

Optimization

Optimization, in simple terms, is the process of selecting a best input from a set of available alternatives. Almost all engineering problems today are aimed at optimizing something. Airfoil designers are always trying to come up with new airfoil designs with better lift and low drag. Heat exchanger component designers want to maximize the surface heat flux while also minimizing the pressure drop through the exchanger.

HyperMesh offers you two options to setting up an optimization study: design optimization and parametric studies. Design optimization enables you to optimize an objective function subject to certain constraints and satisfaction of flow equations.

Parametric studies enables you to specify one or more conditions that may vary, such as a sweep over a range of inlet mass flow boundary conditions. In parametric studies, these control parameters vary according to a specified pattern. For example, the range and frequency of the parameter controlling the mass flow is selected either by simply specifying the desired values or having the program computing suitable values automatically. Several parameters can be used in parametric studies, for instance, controlling the inlet mass flow, the impeller rotational speed, and blade angles of a fan problem. Perhaps you want to evaluate the pressure rise and torque of the fan; these outputs are called "response variables". The parameters controlling the run conditions are called "variables" or "design variables". The nomenclature is similar to the optimization problems.

Both design optimization and parametric studies manage analyses where,

1. the geometry is allowed to change (geometric design and shape optimization) and/or

2. internal parameters, such as the inlet flow rate or viscosity, are allowed to change (design of experiments and quantified optimization).

A minimization problem where a local minimum is sought can be stated as:

Minimize, objective(U,Y)
subject to constraints(U,Y)

Here, U is the flow solution variables and Y is a vector of control parameters, called Design Variables (DVs), that determines the shape of an object or a flow parameter. The objective(s) and constraint(s) are specified by you. A maximization problem is the same as above, but Minimize is replaced with Maximize.

Optimization Solution Sequence

The design optimization may be considered as a sequence of cases, where each case first runs the optimizer and updates the design variables and then solves the flow equations for a number of time steps until convergence. Sample data is gathered at the end of each time step.

The optimizer solution consists of:

1. Constructing the response surface from the set of samples.
2. Running the optimizer on the response surface.
3. Updating the design variables.

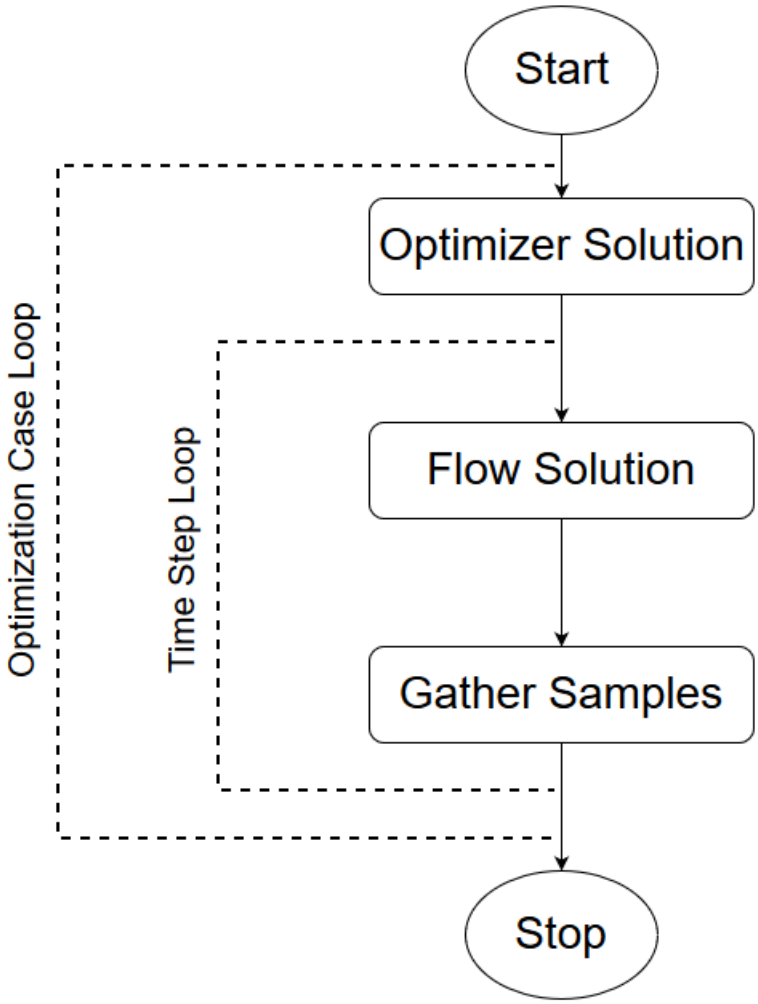


Figure 677:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T7000_ParametricOptimization.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `ParametricOptimization` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.

6. Enter `ParametricOptimization` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the Global Simulation Parameters

Set the Analysis Parameters

In next steps, you will set parameters that apply globally to the simulation.

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. In the Entity Editor, change the Temperature equation to **Advective Diffusive**.
3. Set Optimization to **On**.

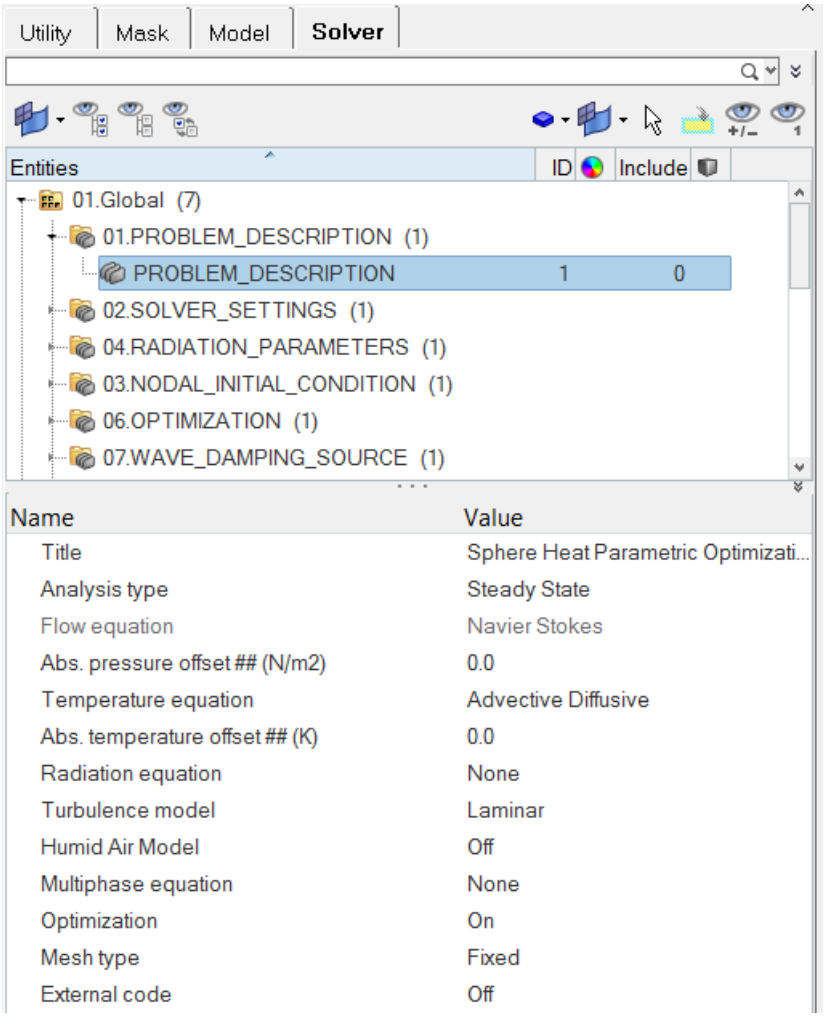


Figure 678:

Specify the Solver Settings

1. In the Solver Browser, expand the **02.SOLVER_SETTINGS** group then click **SOLVER_SETTINGS** to open it in the Entity Editor.
2. Set the Min optimization cases to `10`.
3. Set the Max optimization cases to `50`.
4. Set the Optimization convergence tolerance to `1e-05`.

   The above three parameters are new parameters specific to an optimization setup in HyperMesh. The minimum and maximum optimization cases are the minimum and maximum number of unique input cases that will be solved. The optimization convergence tolerance checks for the change in results on a case-by-case basis. The optimization stops when either the optimization convergence tolerance is achieved, or the specified number of maximum optimization cases is reached.
5. Set Max time steps to `10`.
6. Set the Convergence tolerance to `0.0001`.

   The Min and Max time steps are the minimum and maximum number of time steps solved for each case. The solver moves on to the next case when either the convergence tolerance is achieved, or the specified number of maximum time steps is reached.
7. Set the Relaxation factor to `0.0`.
8. Check that the Flow, Temperature, and Design Optimization flags are set to On and the Temperature flow flag is set to Off.

Name	Value
Min optimization cases	10
Max optimization cases	50
Optimization convergence tolerance	1e-05
Max time steps	10
Initial time increment ## (sec)	10000000000
Convergence tolerance	0.0001
Relaxation factor	0.0
Flow	On
Temperature	On
Temperature flow	Off
Design Optimization	On
External code	Off
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 679:

Set Material Model Parameters

In the next steps, you will create a new custom material and assign relevant material properties to it.

1. In the Solver Browser, expand **02.Materials** the right-click on **FLUID** and select **Create**.
2. Rename the material as `simple_fluid` in the Entity Editor.
3. Set the Density value to `1.0` kg/m^3.
4. Set the Viscosity value to `0.1` kg/m-sec.
5. Set the Conductivity value to `1.0` W/m-K.

Name	Value
Name	simple_fluid
Defined	☑
Material type	FLUID
Group Library Data	true
Density	
Density type	Constant
Density ## (kg/m3)	1.0
Isothermal compressibility ## (m2/N)	0.0
Specific Heat	
Specific heat type	Constant
Specific heat ## (J/kg-K)	1.0
Latent heat type	None
Viscosity	
Viscosity type	Constant
Viscosity ## (kg/m-sec)	0.1
Multiplier Function	<Unspecified>
Conductivity	
Conductivity type	Constant
Conductivity ## (W/m-K)	1.0
Multiplier Function	<Unspecified>
Turbulent Prandtl number	0.91
Radiation Properties	

Figure 680:

6. Save the model.

Set Up the Optimization Parameters

Define the Design Variable

1. In the Solver Browser, right-click on **10.Optimization** and select **Optimization** > **DESIGN_VARIABLES** from the context menu.
2. Rename the variable as `temperature_control` in the Entity Editor.
3. Set the Initial Value to `1.0`.

4. Set the Lower Bound to `0.5`.
5. Set the Upper Bound to `1.5`.
6. Set the Max Update Factor to `0.05`.

   The lower and upper bound values define the range of values between which the design variable can be set. The initial value is the value of the design variable that will be used for the first case, or the initial condition. The maximum update factor controls the maximum change allowed in the design variable value from one case to the next.
7. Set the Number of supervised training points to `3`.
8. Click inside the value field for the newly created **Data: Supervised Training** row.
   The **Number of supervised training points** dialog opens. The number of available rows is equal to the number of points specified in the previous step.
9. Enter `1.0` in row 1, `0.9` in row 2, and `0.8` in row 3 then click **Close**.

   The Supervised Training parameter is used to enforce HyperMesh to use the listed design variable values for as many cases as there are values in the list. After the list is exhausted, optimization control takes over. It is advisable to provide at least three values for Supervised Training as it helps the solver develop a pattern, streamlining the process of selecting the values for design variable for succeeding runs.

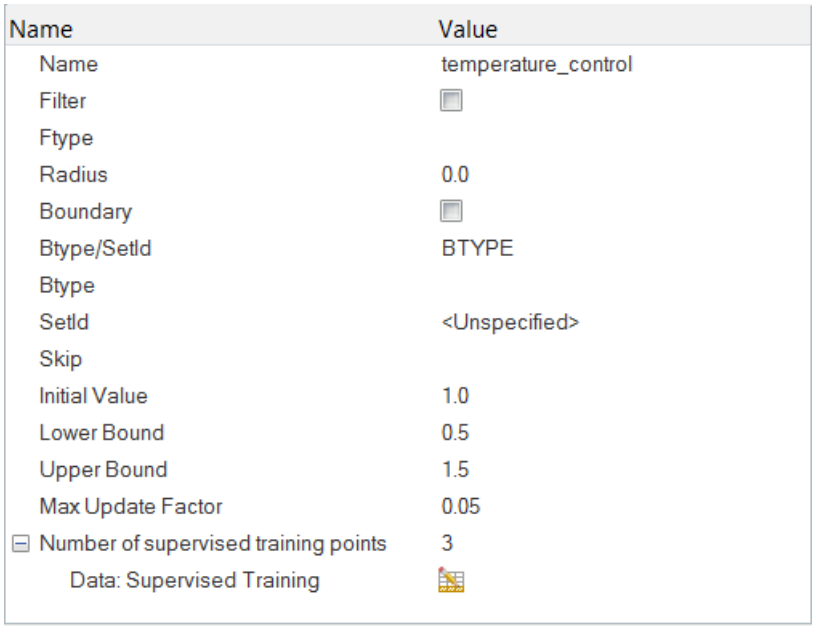


Figure 681:

Create a Multiplier Function for the Design Variable

In HyperMesh, multiplier functions are used to scale values of certain parameters at runtime. For example, viscosity of a fluid may vary with temperature. Or inflow rate or temperature may vary over time.

Multiplier functions can also be used to scale the values of the design variables between cases. In quantity optimization, the design variables are linked with the desired quantities in HyperMesh through multiplier functions.

1. In the Solver Browser, right-click on **05.Multiplier_Function** and select **Create** from the context menu.
2. Rename the function as `inflow_temperature_control` in the Entity Editor.
3. Change the Multiplier Function Type to **Design Variable**.
4. Click in the value field next to Design Variable, click the **Designvar** collector, select **temperature_control** from the appearing dialog, then click **OK**.

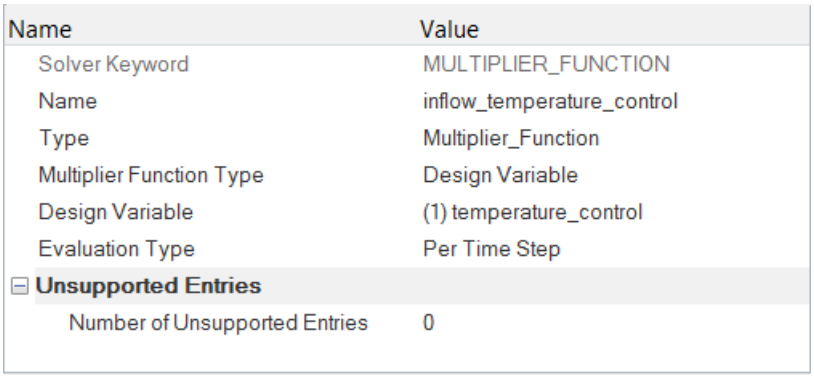


Figure 682:

Define Response Variables

Response variables are the building blocks for deriving objective functions and constraints. Examples include: drag, lift, force in an arbitrary direction, total volume, pressure drop, total pressure, uniformity index on temperature, max speed, etc. In other words, response variables are functions of the flow solution (via output mechanism), the design variables, and/or other response variables.

The most basic response variables are functions of the flow solution, extracted from the SURFACE_OUTPUT and ELEMENT_OUTPUT. More complex response variables can be derived from these basic response variables using expressions. An example of this is demonstrated in the following steps.

1. In the Solver Browser, right-click on **10.Optimization** and select **Optimization** > **RESPONSE_VARIABLES** from the context menu.
2. Rename the variable as `temperature` in the Entity Editor.
3. Repeat the above steps to create three new response variable entries and rename them to:
   a) `target_temperature`
   b) `temperature-target_temperature`
   c) `(temperature-target_temperature)^2`
4. Click on **temperature**. In the entity editor,
   a) Change the Response type to **Surface Output**.

b) Click in the value field next to Surface Elements, click the **Component** collector, select **Outflow** from the appearing dialog, and click **OK**.

c) Set the Output variable to **Temperature**.

This response variable will extract the surface integrated value of the temperature variable from the outflow surface.

Name	Value
Name	temperature
Response type	Surface Output
Surface elements	(3) Outflow
Output Variable	Temperature

Figure 683:

5. Click on **target_temperature**. In the Entity Editor,
   a) Change the Response type to **Constant**.
   b) Set the Value to `325.0`.

   This response variable is set to a constant value or 325 K.
6. Click on **termperature-target_temperature**. In the Entity Editor,
   a) Change the Response type to **Subtraction**.
   b) Click on the value field for the Data: Response variables row.
   c) In the dialog that opens, click inside row 1 then click on the **Optiresponse** collector.
   d) In the **Select Optiresponse** dialog, select **temperature** from the list and click **OK**.
   e) Similarly, select **target_temperature** for row 2.
   f) Click **Close** to exit the dialog.

   This response variable will extract the difference between the two response variables, temperature and target_temperature.
7. Click on **(termperature-target_temperature)^2**. In the Entity Editor,
   a) Change the Response type to **Multiplication**.
   b) Click on the value field for the Data: Response variables row.
   c) In the dialog that opens, click inside row 1 then click on the **Optiresponse**.
   d) In the **Select Optiresponse** dialog, select **temperature-target_temperature** from the list and click **OK**.
   e) Similarly, select **temperature-target_temperature** for row 2.
   f) Click **Close** to exit the dialog.

   This response variable will return the squared value of the response variable, temperature-target_temperature.

Define the Objectives

It was mentioned earlier that the response variables are the building blocks for deriving objective functions and constraints. In this section, you will define a new objective and link an existing response variable to that objective entity.

The objective of an optimization problem is usually minimization or maximization of a response variable. In this problem, the objective is to minimize the difference between the actual and the target outflow temperature. In the next few steps, you will create and define the objective accordingly.

1. In the Solver Browser, expand **Optimization** > **OBJECTIVES** then click on **objective1**.
2. Rename the objective as `find_inflow_temperature` in the Entity Editor.
3. Check that the Objective Type is set to Minimize.
4. Set the Response variable to **(temperature-target_temperature)^2** by clicking in the value field next to Response variable then selecting the correct response variable from the **Select Optiresponse** dialog.

   Note: If there are any constraints in your problem, you can define them similarly under the Constraints group.

Set Up Optimization Controls

In the Optimization command, you will link all the defined objectives and constraints to define the optimization problem. There are no constraints defined for this problem, but if there are any, they should be defined first under the Constraints entry in the Data Tree.

1. In the Solver Browser, expand **01.Global** then click on **06.OPTIMIZATION**.
2. In the Entity Editor, verify that the Objective is set to **find_inflow_temperature**.
3. If there are any constraints in your problem, you can set the value for the Number of Constraints field accordingly and then define the constraints.
4. Verify that the Optimizer convergence tolerance is set to `1e-4`.

Assign Boundary Conditions and Material Properties

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Select **simple_fluid** as the Material.

Name	Value
Name	Fluid
FE style	
Geometry style	
Type	FLUID
Material	(7) simple_fluid
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 684:

3. Click **Inflow**. In the Entity Editor,

a) Change the Type to **INFLOW**.

b) Turn **On** the **Show advanced features** field.

c) Set the Normal velocity to `0.1` m/sec.

d) Set the Temperature to `300` K.

e) Set the Temperature multiplier function to **inflow_temperature_control**.

Name	Value
Name	Inflow
FE style	
Geometry style	
Type	INFLOW
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	On
Active type	All
Precedence	1
Inflow type	Velocity
Reference frame	<Unspecified>
Inflow velocity type	Normal
Normal velocity ## (m/sec)	0.1
Normal velocity multiplier function	<Unspecified>
Temperature ## (K)	300.0
Temperature multiplier function	(3) inflow_temperature_control
Integrated Boundary Conditions	

Figure 685:

4. Click **Outflow**. In the Entity Editor, change the Type to **OUTFLOW**.

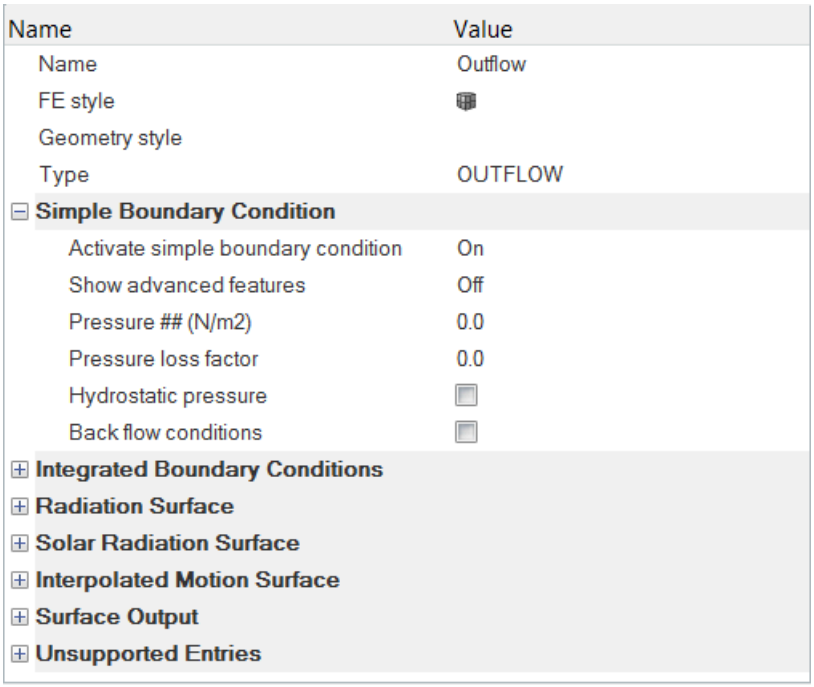


Figure 686:

5. Click **Outer_Walls**. In the Entity Editor, change the Type to **SLIP**.

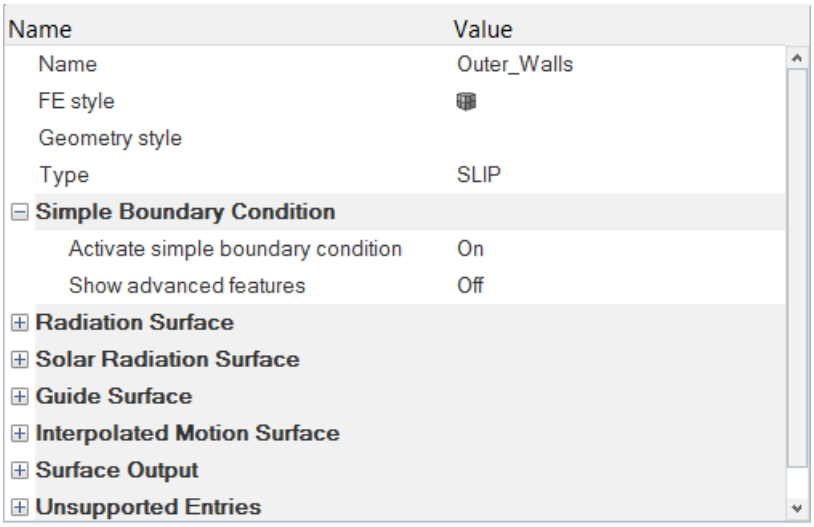


Figure 687:

6. Click **Sphere_Wall**. In the Entity Editor,
   a) Change the Type to **WALL**.
   b) Change the Temperature BC type to **Value**.
   c) Set the Temperature to `500` K.

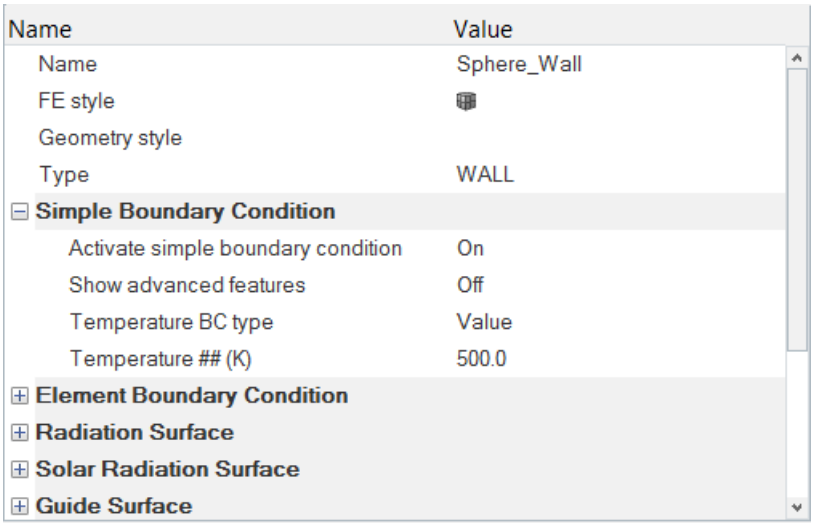


Figure 688:

7. Save the model.

Compute the Solution and Review the Results

Run HyperMesh

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

1. In the Model Browser, ensure that the visibility of the mesh for all collectors to be exported for AcuSolve are activated. In this case, display for all the collectors – Fluid, Inflow, Outflow, Sphere_Wall, and Outer_Walls – should be activated.

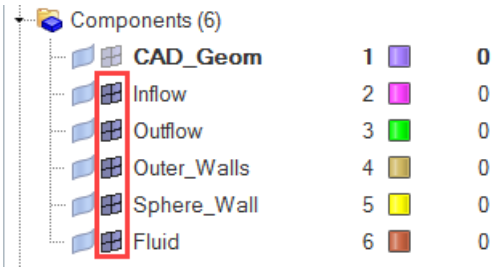


Figure 689:

2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.

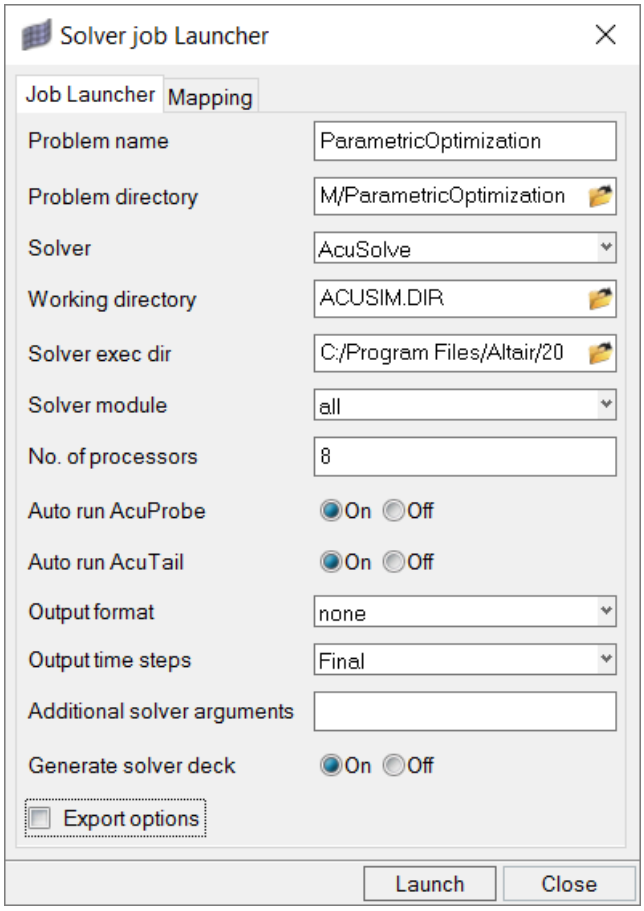


Figure 690:

For this case, the default settings will be used. You may choose to change the number of processors to allow AcuSolve to run using more processors (4 or 8), if available. HyperMesh will generate the required solver input files and launch AcuSolve. AcuSolve will calculate the steady state solution for this problem.

3. Verify that Auto run AcuProbe is **On**.

   This will open an **AcuProbe** dialog which will let you monitor the solution progress.

4. Click **Launch** to start the solution process.
   As the solution progresses, an **AcuTail** and an **AcuProbe** dialog will open. Solution progress is reported in the **AcuTail** dialog. An **AcuSolve Control** dialog will also open from which you can control the solution process. In this dialog you have options to stop the solution or generate the output files at the end of the current time step.

   You'll notice HyperMesh prints the case number before beginning the solution for the case. The Case-Step number is reset to 1 when HyperMesh moves to a new case while Time-Step proceeds sequentially.

```
acuSolve:   Step CPU/Elapse time     = 3.582800e+001 1.118500e+001 Sec
acuSolve: ---------------------- End Case       ts    1  -   10 ------
acuSolve: ---------------------- Begin Case Cycle ---------------------
acuSolve: Case=        2 ; Case-Step=    1
acuSolve: Time-Step=   11 ; timeInc= 1.000000e+010 ; time= 1.000000e+011
```

Figure 691:

A summary of the run printed in the **AcuTail** dialog indicates that AcuSolve has finished running the solution.

```
acuSolve: ----------------------- End Time Step -----------------------
acuSolve: Input       CPU/Elapse time= 5.640000e+000 7.971000e+000 Sec
acuSolve: Output      CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Exchange    CPU/Elapse time= 2.228170e+002 9.061400e+001 Sec
acuSolve: Intf.search CPU/Elapse time= 0.000000e+000 0.000000e+000 Sec
acuSolve: Design.opt  CPU/Elapse time= 1.425650e+002 5.560200e+001 Sec
acuSolve: Flow stagger "flow":
acuSolve:   No of res/lhs formations =         138         138
acuSolve:   Elem.Form CPU/Elapse time= 6.263440e+002 1.871350e+002 Sec
acuSolve:   Solution  CPU/Elapse time= 1.574656e+003 5.068220e+002 Sec
acuSolve: Temperature stagger "temperature":
acuSolve:   No of res/lhs formations =         138         138
acuSolve:   Elem.Form CPU/Elapse time= 2.994020e+002 8.075600e+001 Sec
acuSolve:   Solution  CPU/Elapse time= 2.149890e+002 6.788500e+001 Sec
acuSolve: Total CPU/Elapse time      = 3.864515e+003 1.239437e+003 Sec
acuSolve: Total Memory Usage         = 2.635039e+002 Mbytes
acuRun: -*-*-*-
```

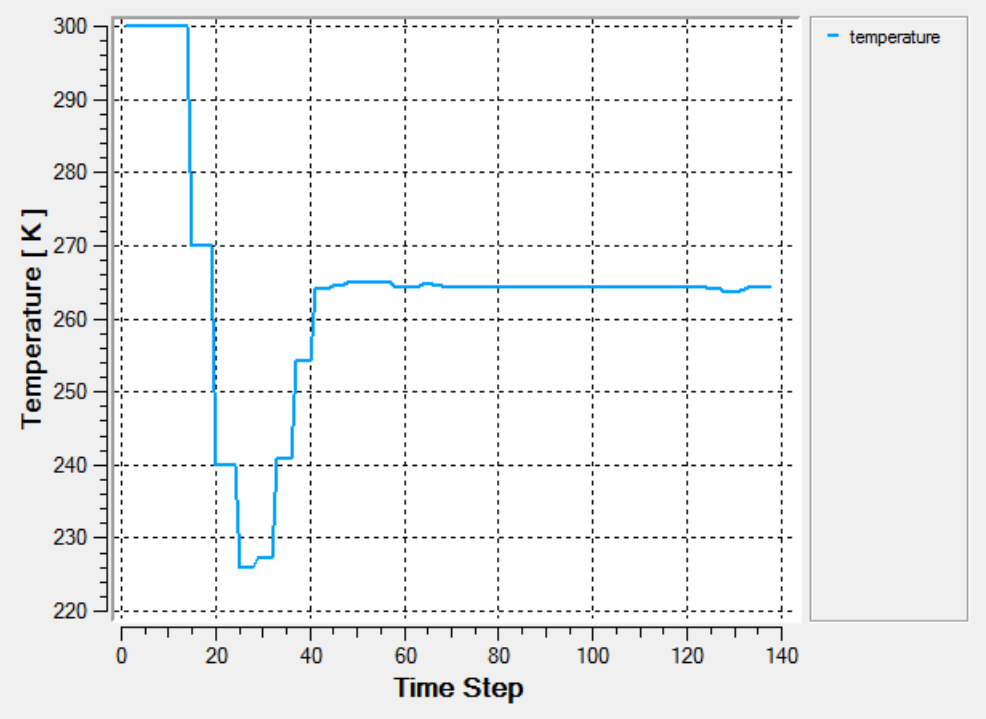


Figure 693:

This plot shows the surface temperature of the inflow surface as the solution progresses.

In the setup, the inflow temperature is a design variable. At the inflow surface, the temperature varies from case to case in accordance with the values provided to the solver by the optimization program. The first three values of inflow temperature correspond to the values provided in the supervised learning parameter, after which the values are provided by the optimizer. The converged value of the inflow temperature is approximately 264.15 K. This is the value at which the outflow temperature is closest to its target value.

3. Right-click on **temperature** and select **Don't Plot** to turn off the plot display.
4. Expand **Surface Output** > **Outflow** > **Temperature**.
5. Right-click on **temperature** and select **Plot**.



Figure 694:

This plot shows the surface temperature of the outflow surface as the solution progresses.

At the outflow surface, the temperature varies in the beginning of the simulation and ultimately converges to a value close to the target temperature value, i.e. 325 K.

6. You can also save the plots as an image.
   a) From the AcuProbe dialog, click **File** > **Save**.
   b) Enter a name for the image and click **Save**.
7. The time series data of the variables can also be exported as a text file for further post-processing.
   a) Right-click on the variable that you want to export and click **Export**.
   b) Enter a File name and choose .txt for the Save as type.
   c) Click **Save**.

AcuGetDv and AcuGetRsp

AcuSolve provides two post-processing utilities specific to optimization problems, AcuGetDv and AcuGetRsp. AcuGetDv provides the values of the design variables for all the cases included in the solution. AcuGetRsp provides the values of the response variables for all the cases included in the solution.

1. Start AcuSolve Command Prompt from the Windows Start menu by clicking **Start** > **Altair <version>** > **AcuSolve Cmd Prompt** .
2. Change the directory to the present working directory using the `cd` command.

3. Enter the following command at the prompt:

```
acuGetDv
```

AcuSolve will print the values of design variables for each case.

```
C:\ParametricOptimization>acuGetDv
  1             1
  2             1
  3           0.9
  4           0.8
  5       0.75294
  6      0.757646
  7      0.802842
  8      0.847135
  9      0.879897
 10       0.88134
 11      0.883108
 12      0.882932
 13      0.882932
 14      0.882932
 15      0.880817
 16       0.88095
 17      0.881987
 18      0.881883
 19      0.880283
 20      0.880269
 21       0.88027
 22      0.881143
 23      0.881664
 24      0.881612
 25      0.881339
 26      0.881352
 27      0.881351
 28      0.881351
 29      0.880632
 30      0.880627
 31      0.880627
 32      0.880644
 33      0.880642
 34      0.880642
 35      0.880601
 36      0.880597
 37      0.880563
 38      0.880566
 39      0.880566
 40      0.880227
 41      0.880261
 42      0.881518
 43      0.881392
 44      0.880361
 45      0.880134
 46      0.880156
 47      0.880647
 48      0.880643
 49      0.880492
 50      0.880507
```

Figure 695:

4. To print the values of response variables, use the command:

```
acuGetRsp
```

Note: The order of columns in which the design variables and the response variables are printed by these commands is the order in which they appear in the INP file.

```
C:\ParametricOptimization>acuGetRsp
1 351.232856876 325.0 26.2328568763 688.162779891
2 351.32283547 325.0 26.32283547 692.891667181
3 329.187159989 325.0 4.18715998923 17.5323087754
4 307.039391061 325.0 -17.960608939 322.58347346
5 296.633085644 325.0 -28.3669143564 804.681830104
6 297.709388449 325.0 -27.2906115505 744.777478802
7 307.771752488 325.0 -17.2282475119 296.812512332
8 317.640189725 325.0 -7.35981027513 54.1668072859
9 324.941272756 325.0 -0.058727244023 0.00344888919054
10 325.274939766 325.0 0.274939766095 0.0755918749805
11 325.679281581 325.0 0.679281581413 0.461423466847
12 325.649823596 325.0 0.64982359568 0.422270705502
13 325.65576569 325.0 0.655765690363 0.430028640658
14 325.661132324 325.0 0.661132324339 0.437095950286
15 325.198347173 325.0 0.198347172997 0.0393416010359
16 325.232122579 325.0 0.232122579143 0.0538808917481
17 325.468290877 325.0 0.468290877427 0.219296345882
18 325.44869358 325.0 0.448693580489 0.201325929172
19 325.098706158 325.0 0.0987061581401 0.00974290565478
20 325.098191008 325.0 0.0981910075785 0.00964147396927
21 325.100957962 325.0 0.10095796179 0.0101925100489
22 325.298101047 325.0 0.298101047062 0.0888642342595
23 325.416824849 325.0 0.416824849487 0.17374295515
24 325.406967946 325.0 0.406967946125 0.165622909173
25 325.348912718 325.0 0.34891271796 0.121740084754
26 325.353086208 325.0 0.353086208203 0.124669870423
27 325.354066046 325.0 0.354066045959 0.125362764901
28 325.355219511 325.0 0.355219510982 0.126180900982
29 325.19696524 325.0 0.196965239879 0.0387953057207
30 325.196796737 325.0 0.196796737216 0.0387289557787
31 325.197639415 325.0 0.197639414934 0.0390613383356
32 325.202206424 325.0 0.202206423856 0.0408874378485
33 325.202532726 325.0 0.202532725789 0.0410195050156
34 325.203178884 325.0 0.20317888359 0.0412816587369
35 325.194545735 325.0 0.194545734809 0.0378480429324
36 325.194251879 325.0 0.194251879338 0.0377337926264
37 325.187126299 325.0 0.18712629922 0.0350162518596
38 325.188340563 325.0 0.188340562609 0.0354721675239
39 325.188756011 325.0 0.188756010862 0.0356288316366
40 325.113947444 325.0 0.113947444429 0.012984020092
41 325.12181318 325.0 0.121813180352 0.0148384509074
42 325.40156579 325.0 0.401565789621 0.161255083394
43 325.37397383 325.0 0.373973830207 0.139856425679
44 325.145353632 325.0 0.145353631625 0.0211276782265
45 325.095183888 325.0 0.0951838882505 0.00905997258249
46 325.100402907 325.0 0.100402907302 0.0100807437946
47 325.209593947 325.0 0.209593947471 0.0439296228166
48 325.208714335 325.0 0.208714335437 0.0435616738168
49 325.175608366 325.0 0.175608365789 0.0308382981352
50 325.179031033 325.0 0.179031032752 0.0320521106882
```

Figure 696:

Summary

In this AcuSolve tutorial, you successfully set up and solved a parametric optimization problem with AcuSolve. You started the tutorial by creating a database in HyperMesh, importing and meshing the geometry, and setting up the simulation parameters. You defined the design variables, the response variables, and set up the objectives of the problem using the response variables. Once the case was setup, the solution was generated with AcuSolve. AcuProbe was used to visualize the variation of design variables and response variables as the solution progressed through the different cases provided to AcuSolve by the optimization program. You also used the utilities provided in AcuSolve to get the design variables and response variables information for all the cases.

2.7.2 ACU-T: 7001 Shape Optimization using HyperMorph

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T7001_ShapeOptimization.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

Optimization, in simple terms, is the process of selecting a best input from a set of available alternatives. AcuSolve offers you two options to setting up an optimization study: design optimization and parametric studies. Design optimization enables you to optimize an objective function subject to certain constraints and satisfaction of flow equations. The design optimization may be considered as a sequence of cases, where each case first runs the optimizer and updates the design variables and then solves the flow equations for a number of time steps until convergence. Sample data is gathered at the end of each time step.

The optimizer solution consists of:

1. Constructing the response surface from the set of samples.
2. Running the optimizer on the response surface.
3. Updating the design variables.

The geometry for this problem consists of a simple pipe channel with perfectly circular cross-section as the base shape. Water enters the Inlet at the rate of 0.0003 kg/s and the outlet is a standard pressure outlet at zero relative pressure. Walls of the channel are no-slip walls.

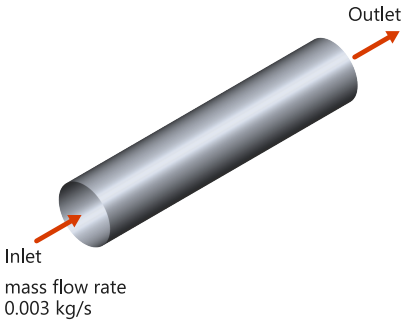


Figure 697:

Open the HyperMesh Model Database

1. Start HyperMesh and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T7001_ShapeOptimization.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `ShapeOptimization` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `ShapeOptimization` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Set the Global Simulation Parameters

Set the Analysis Parameters

1. Go to the Solver Browser, expand **01.Global**, then click **PROBLEM_DESCRIPTION**.
2. Set Optimization to **On**.
3. Set the Mesh type to **Fully Specified**.

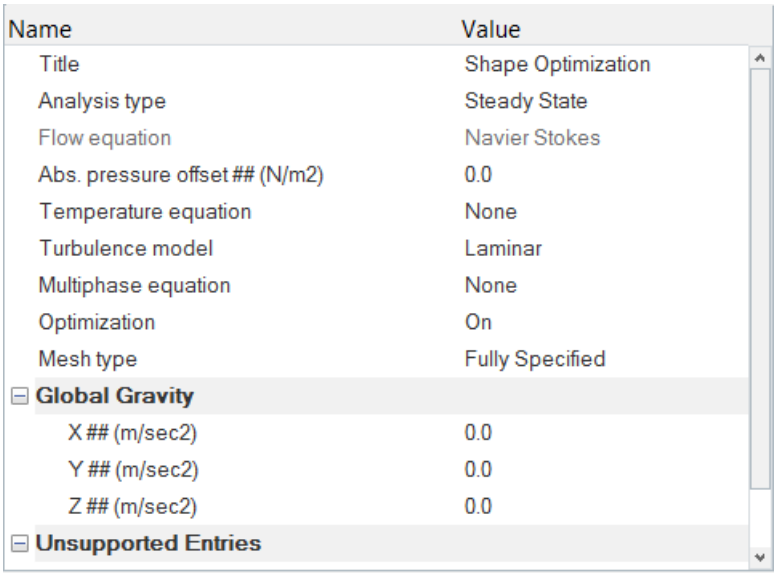


Figure 698:

Specify the Solver Settings

1. In the Solver Browser, click **02.SOLVER_SETTINGS** under **01.Global**.
2. In the Entity Editor , set the Min optimization cases to `10`.
3. Set the Max optimization cases to `200`.
4. Set the Optimization convergence tolerance to `1e-005`.
5. Set Max time steps to `100`.
6. Set the Convergence tolerance to `0.0001`.
7. Set the Relaxation factor to `0.0`.
8. Check that the Flow and Design Optimization flags are set to On.

Name	Value
Min optimization cases	10
Max optimization cases	200
Optimization convergence tolerance	1e-05
Max time steps	100
Initial time increment ## (sec)	10000000000
Convergence tolerance	0.001
Relaxation factor	0.0
Flow	On
Design Optimization	On
Advanced Settings	
Display	☐
Unsupported Entries	
No of Auto Solution Strategy Unsupported Entries	0
No of Time Increment Unsupported Entries	0

Figure 699:

Set the Nodal Output Frequency

1. In the Solver Browser, expand **17.Output** and then click **NODAL_OUTPUT**.
2. Turn **On** the Output initial condition field.
   This will instruct the solver to write the initial state of the problem as the first output file.
3. Save the model.

Create Mesh Motion and Set the Boundary Conditions and Material Model Parameters

Create Mesh Motion

An optimization study can be performed in AcuSolve using either volume morph shapes or surface morph shapes. If volume morph shapes are used for simulation, you do not need to define mesh motion, as the volume nodes will move using the input from morph shapes. If surface morph shapes are used for simulation, interpolated mesh motion is needed to define the motion of volume nodes in the model. For this tutorial, surface morph shapes with interpolated mesh motion is used.

1. In the Solver Browser, right-click on **06.Mesh_Motion** and click **Create**.
2. Name the mesh motion `Volume_MMI`.
3. Set the Type to **Interpolated_Motion** and the Number of Interpolated surfs to `3`.

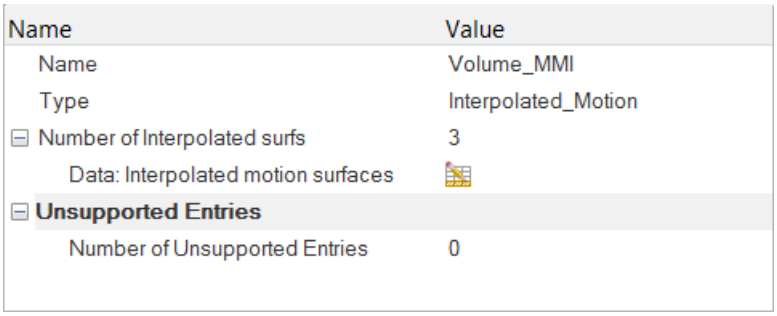


Figure 700:

4. Click the data entry icon .
   The **Number of Interpolates surfs** dialog opens.
5. Select **Wall**, **Inflow**, and **Outflow**.

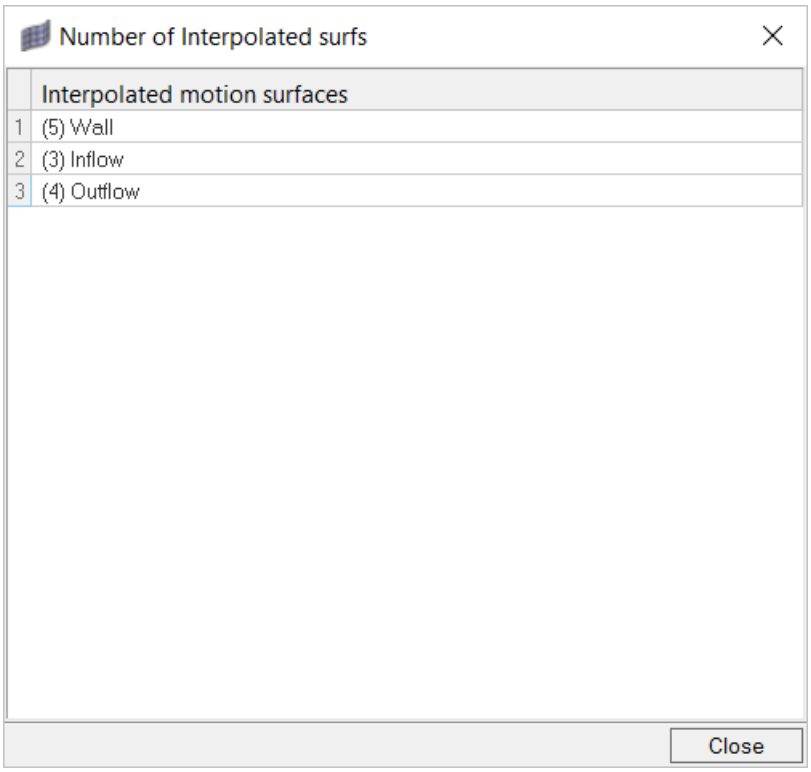


Figure 701:

Specify the Boundary Conditions and Material Models Parameters

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Inflow**. In the Entity Editor,

a) Change the Type to **INFLOW**.
b) Set the Inflow type to **Mass flux**.
c) Set the Mass flux to `0.0003` kg/sec
d) Under the Interpolated Motion Surface section, turn on **Display** and **Activate the interpolated motion surface**.

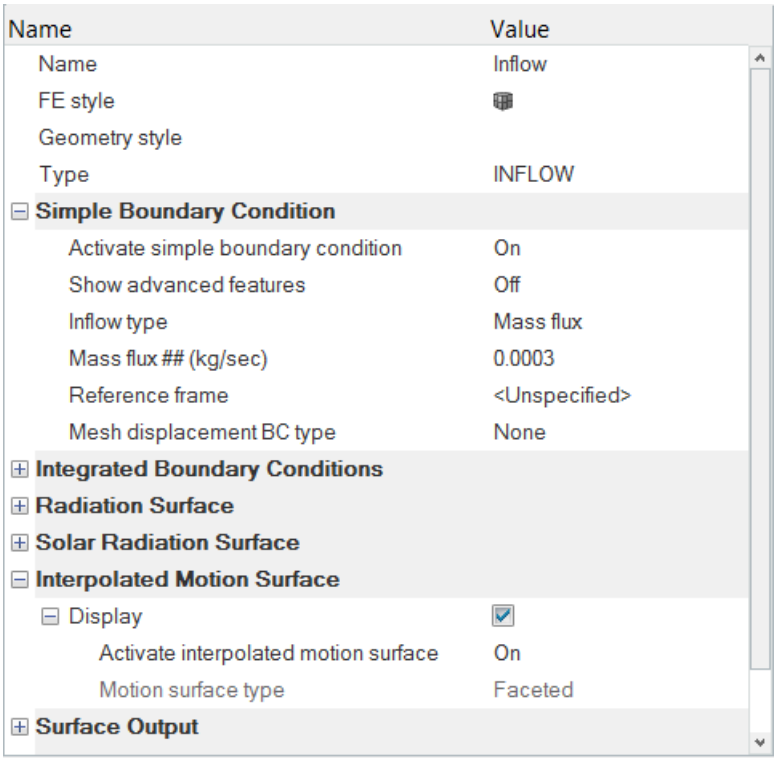


Figure 702:

3. Click **Outflow**. In the Entity Editor,
a) Change the Type to **OUTFLOW**.
b) Under the Interpolated Motion Surface section, turn on **Display** and **Activate the interpolated motion surface**.

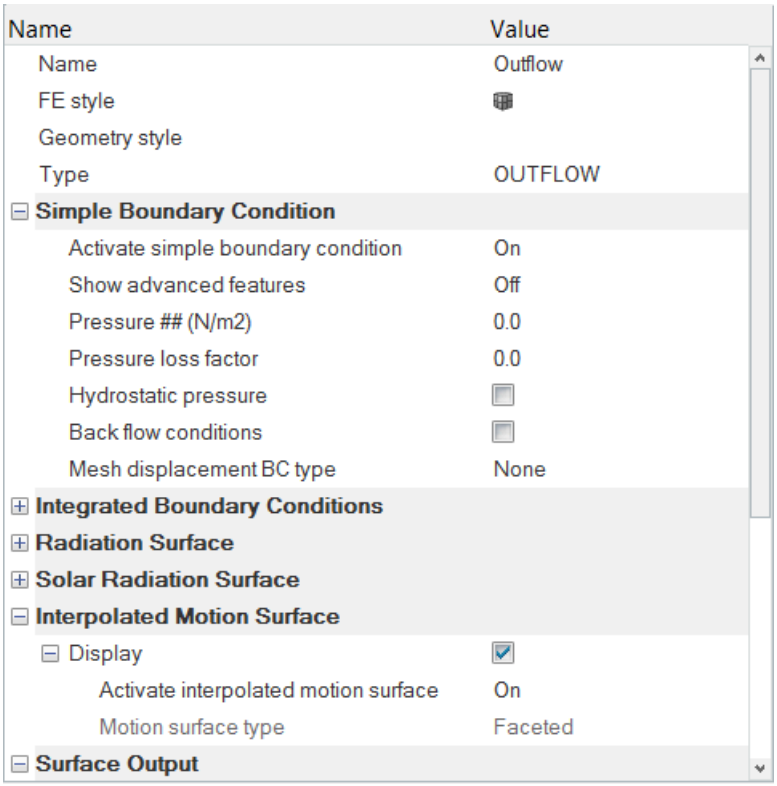


Figure 703:

4. Click **Wall**. In the Entity Editor,
   a) Verify that the Type ia set to **Wall**.
   b) Under the Interpolated Motion Surface section, turn on **Display** and **Activate the interpolated motion surface**.

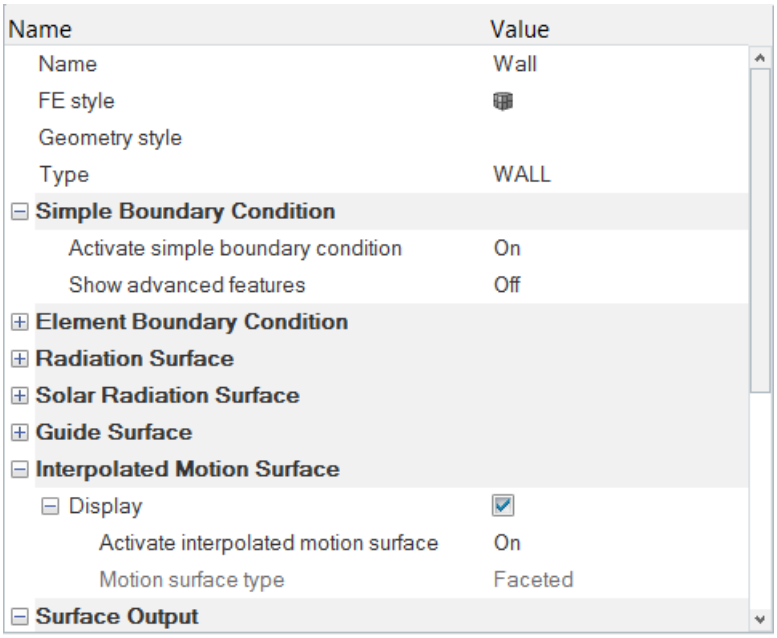


Figure 704:

5. Click **Fluid**. In the Entity Editor,

a) Change the Type to **FLUID**.

b) Select **Water_HM** as the Material.

c) Set the Mesh motion to **Volume_MMI**.

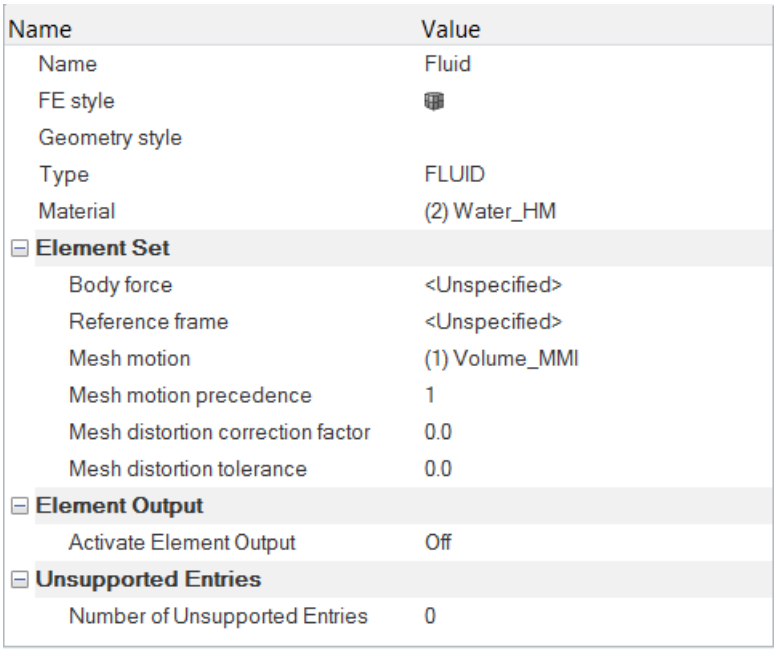


Figure 705:

6. Save the model.

Set Up Optimization Parameters

Generate and Export Morph Shapes

HyperMorph is used to parameterize the shape of the design. You will create morph shapes by moving the surface nodes. The volume nodes will be taken care of using the interpolated mesh motion feature of HyperMesh.

1. In the panel area, click **HyperMorph** then select **morph volumes**.
2. In the morphing sub-panel, change mvols:active to **mvols:skin only**.

   By changing this option to "skin only", you only morph the surfaces and surface morph shapes are generated. If "mvols" is left to default, volume morph shapes are generated where all the nodes in the model are deformed (not just surfaces) during the morphing process.

   > **Important:** Do not click the solve button.

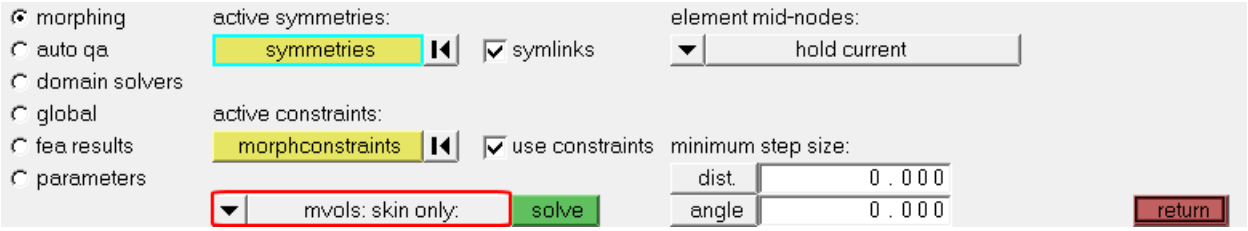


Figure 706:

3. Click on **return**, then from the main HyperMorph menu panel, select **morph volumes**.
4. In the create sub-panel, click the **elems** collector and select **all**.
5. Click **create**.
   A new morph volume is created.

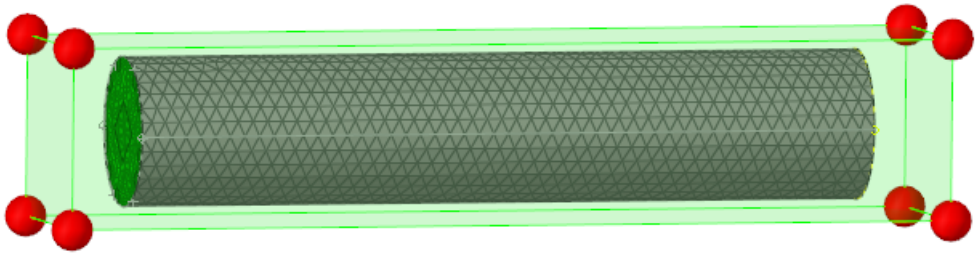


Figure 707:

6. Click **return** to exit to the main HyperMorph menu then select **morph** from the panel area.
7. Select the **move handles** sub-panel if it's not already selected. In the sub-panel, click the second arrow and select **scale**. Leave the x scale at 1.0 and set the y scale and z scale to `1.5`.

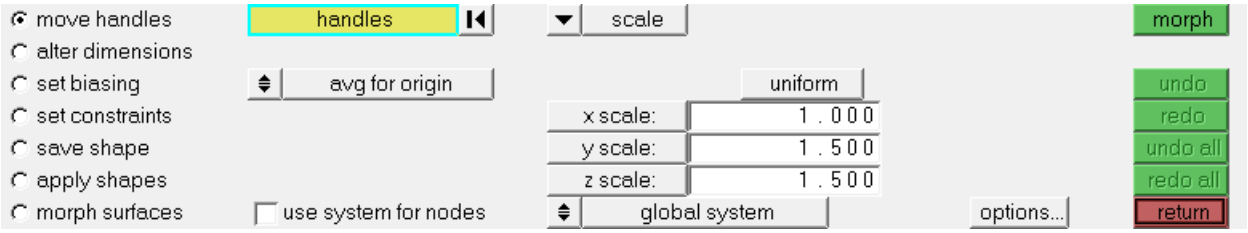


Figure 708:

8. Activate the **handles** collector by clicking on it.
9. In the modeling window, select the four edge handles at the pipe outflow, highlighted in grey in the figure below.

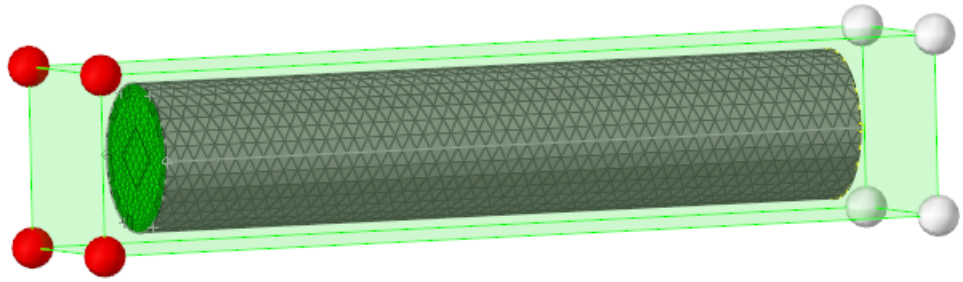


Figure 709:

10. Click **morph**.
    The grid is morphed.

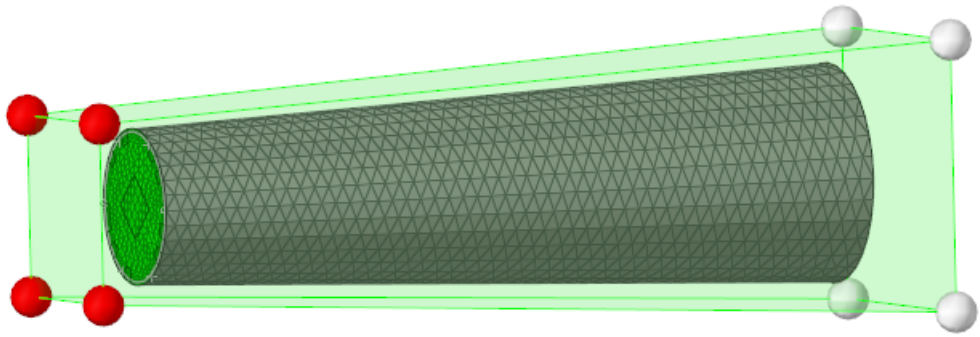


Figure 710:

11. Select the **save shape** sub-panel. In this panel,
    a) Set the name field to `outflow_expand`.
    b) In the second row, set the selector to **as node perturbations**.
    c) Check that the coordinate system is set to global and click **save**.
    d) Select **No** when asked to "Save perturbations for nodes at global and morph volume handles?"

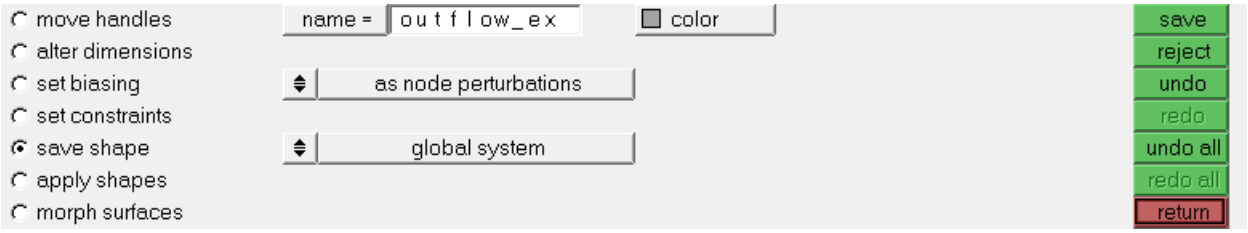


Figure 711:

> **Note:** When you click save, a new entity folder, Shapes, will be created in the Model Browser. The shape outflow_expand will be created inside this folder in the Model Browser. You can turn off the display of the shape nodal perturbations by right-clicking on **outflow_expand** and selecting **Hide**. To show the shape again, right-click on **outflow_expand** and select **Show**. It is recommended to hide the shape display at this point before proceeding to next steps.

12. From the right side of the panel, select **undo all**.
13. Click **move handles** to return to the move handles sub-panel. In the sub-panel, reset the y scale to `1.0` and leave the z scale at 1.5.
14. Activate the **handles** collector then select all eight handles in the modeling window.
15. Click **morph**.
    The grid is morphed.

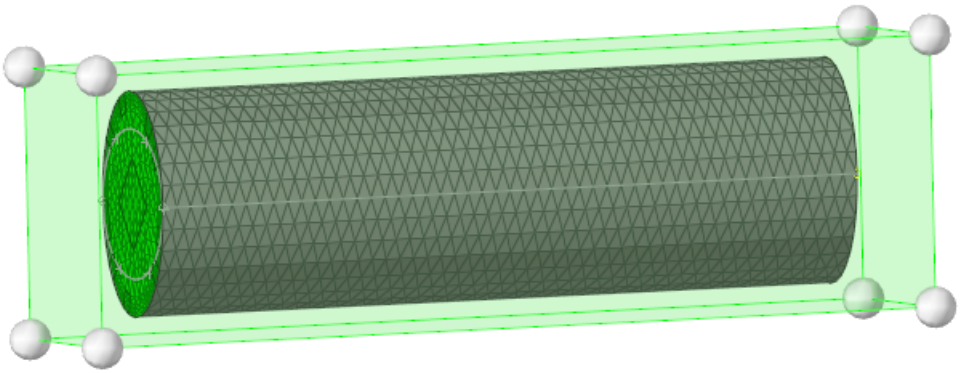


Figure 712:

16. Select the **save shape** sub-panel again. In the panel,
    a) Set the name field to `length_z`.
    b) In the second row, set the selector to **as node perturbations**.
    c) Check that the coordinate system is set to global and click **save**.
    d) Select **No** when asked to "Save perturbations for nodes at global and morph volume handles?"
17. Click **undo all** then click **return** to exit to the main HyperMorph menu.
18. Hide the shape display in the Model Browser

19. Click **morph volumes** to enter the morph volumes panel again then select the **split/combine** sub-panel.

20. In the second row, set the split type to **No. of splits (# of splits)** and enter 3 for the number of splits.

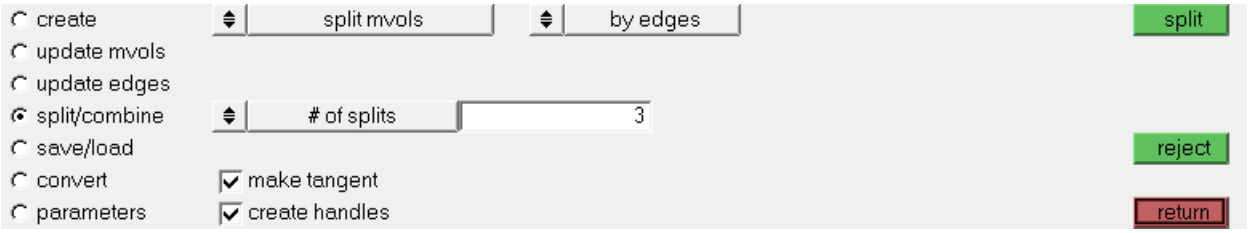


Figure 713:

21. In the modeling window, select the edge of the morph volume marked by green crosses in the figure below.

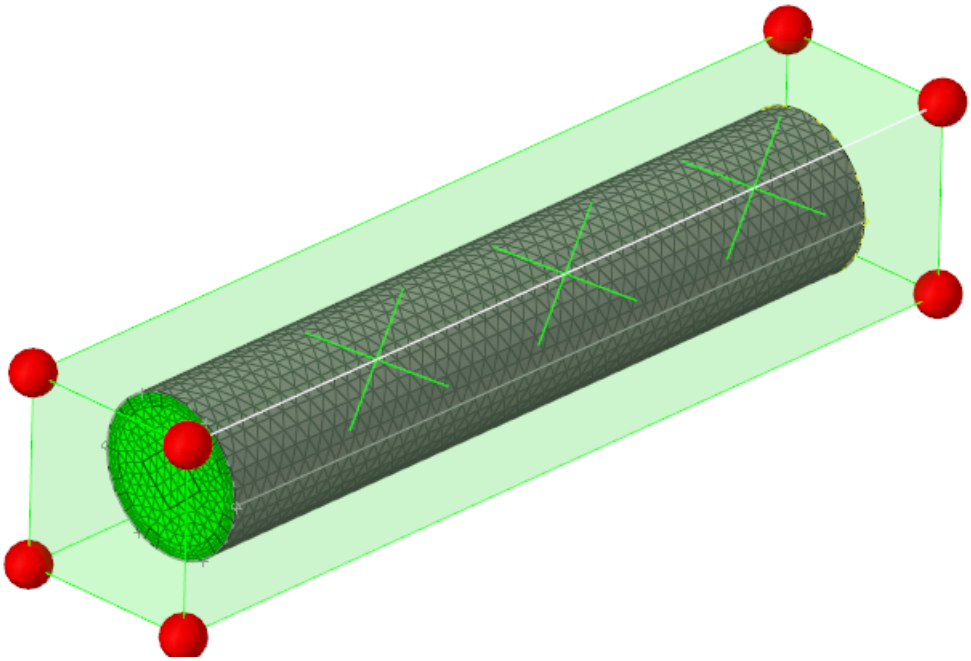


Figure 714:

22. Click **split**.
The morph volume is split at the locations of the crosses.

23. Click **update edges** to open the corresponding sub-panel. Click the first arrow and select **update ends**. Then, click the second arrow and select **mvols**. Finally, click the third arrow and select **main-secondary**.

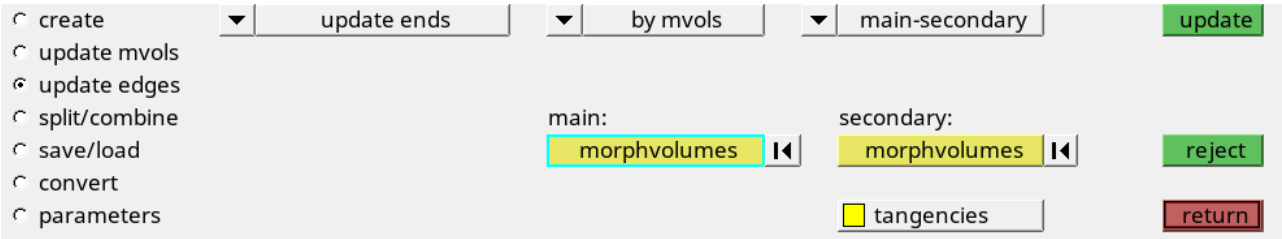


Figure 715:

Note: This option allows you to link any two edges together with a main-secondary relationship between two morph volumes. In this kind of relationship, the secondary edge is forced to follow the curvature of the main edge at the joining end of the two edges.

24. Activate the **main morphvolmes** collector and select the outer two morph volumes shown in the figure below. Then, activate the **secondary morphvolumes** collector and select the inner two morph volumes. After selecting the volumes in the order mentioned, click **update**.

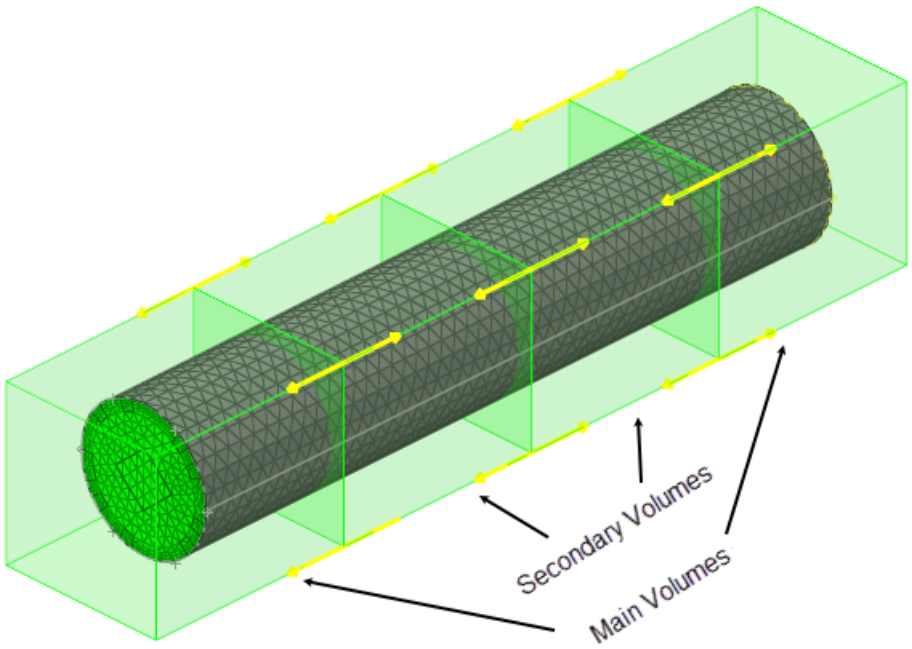


Figure 716:

The edges in the volume should resemble the figure shown below.

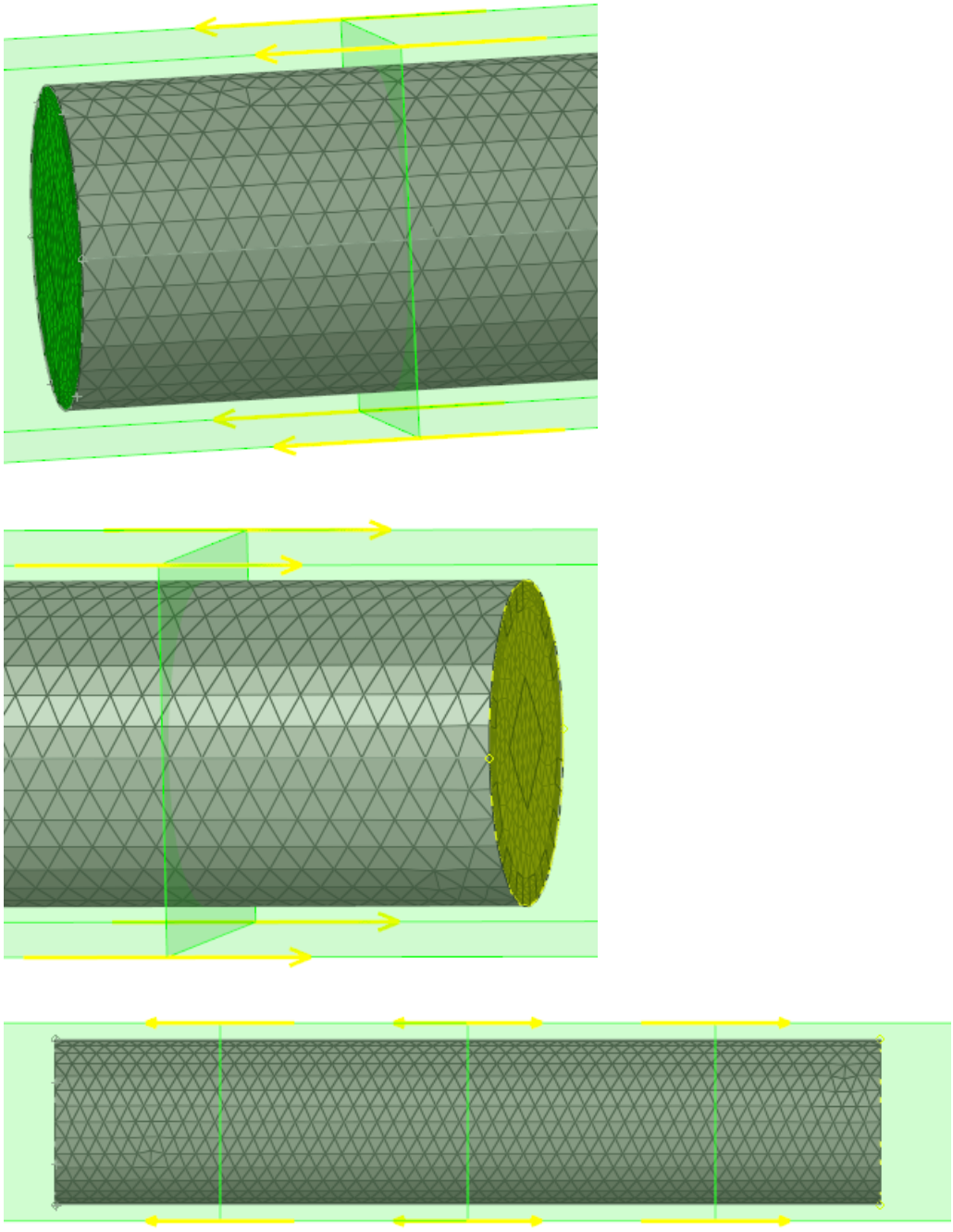


Figure 717:

25. Click **return** to go back to the HyperMorph panel.

26. Select **morph** from the panel area then select the **move handles** sub-panel. In the sub-panel, click the second arrow and select **scale** if it's not already selected. Set the z scale to `1.0` and the y scale to `1.5`.

27. Activate the **handles** collector by clicking on it then select the four middle handles in the modeling window.

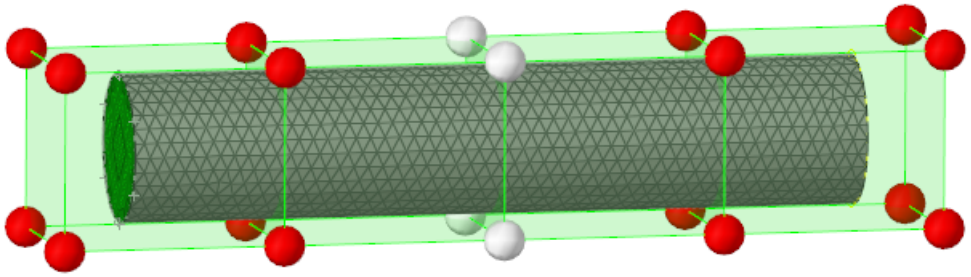


Figure 718:

28. Click **morph**.
The grid is morphed.

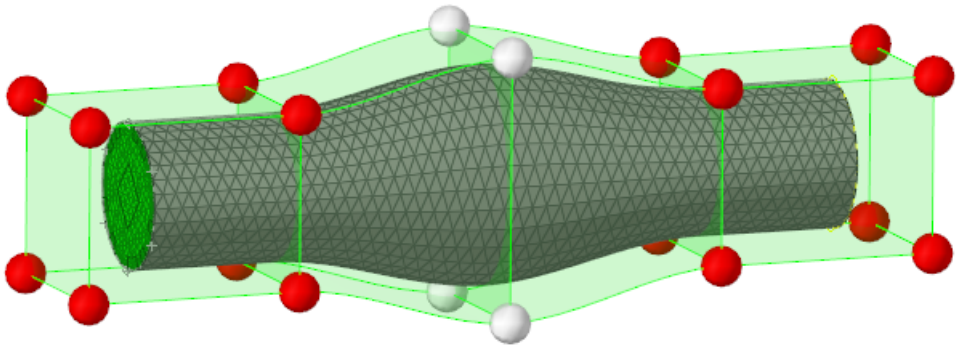


Figure 719:

29. Select the **save shape** sub-panel. In the panel,
   a) Set the name field to `center_y`.
   b) In the second row, set the selector to **as node perturbations**.
   c) Check that the coordinate system is set to global and click **save**.
   d) Select **No** when asked to "Save perturbations for nodes at global and morph volume handles?"

30. Click **undo all** and hide the shape display in the Model Browser.

31. Exit the HyperMorph panel.

Define the Design Variable

1. Right-click on **10.Optimization** in the Solver Browser and select **Optimization** > **DESIGN_VARIABLES** from the context menu.
2. Rename the variable to `outflow_expand` and press Enter.
3. In the Entity Editor, set the Initial Value to `0.7`.
4. Set the Lower Bound to `0.2`.

5. Set the Upper Bound to `1.5`.
6. Set the Max Update Factor to `0.02`.
7. Follow the above steps to create two new design variables, `length_z` and `center_y`. Use identical parameters as above to define these two new design variables.

Define the Response Variable

1. Right-click on **10.Optimization** in the Solver Browser and select **Optimization** > **RESPONSE_VARIABLES** from the context menu.
2. Rename the variable to `inflow_pressure` and press Enter
3. In the Entity Editor, change the Response type to **Surface Output**.
4. Change the Response type to **Surface Output**.
5. Set the Surface elements component to **Inflow**.
6. Change the Output variable to **Pressure**.

   This response variable will extract the surface integrated value of the pressure variable from the inflow surface.

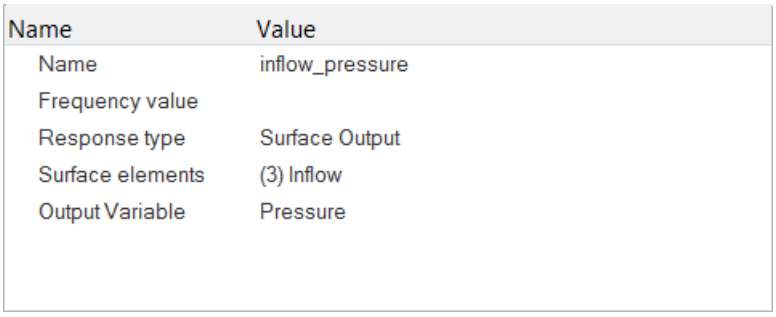


Figure 720:

Define the Objective

1. Right-click on **Optimization** in the Solver Browser and select **Optimization** > **OBJECTIVES** from the context menu.
2. Rename the objective to `maximize_inflow_pressure` and press Enter
3. In the Entity Editor, set the Objective Type is set to **Maximize**.
4. Select **inflow_pressure** as the Response variable.

Set Up the Nodal Shapes

1. Right-click on **Optimization** in the Solver Browser and select **Optimization** > **NODAL_SHAPES** from the context menu.
2. Rename the nodal shape to `pipe_shapes` and press Enter
3. Set the Number of shapes to `3`.

4. Click inside the value field in the newly created Data: Shape row.
   A **Number of shapes** dialog opens. You will notice that the number of available rows is equal to the number of points specific in the previous step.
5. In row 1, click the Shape field then click the **Shape** collector and select **outflow_expand** from the **Select Shape** dialog.
6. Click in the DV field in row 1 then click the **Designvar** collector and select the design variable **outflow_expand** from the **Select Designvar** dialog.
7. Similarly, select **length_z** as the shape and design variable for row 2 and **center_y** as the shape and design variable for row 3.

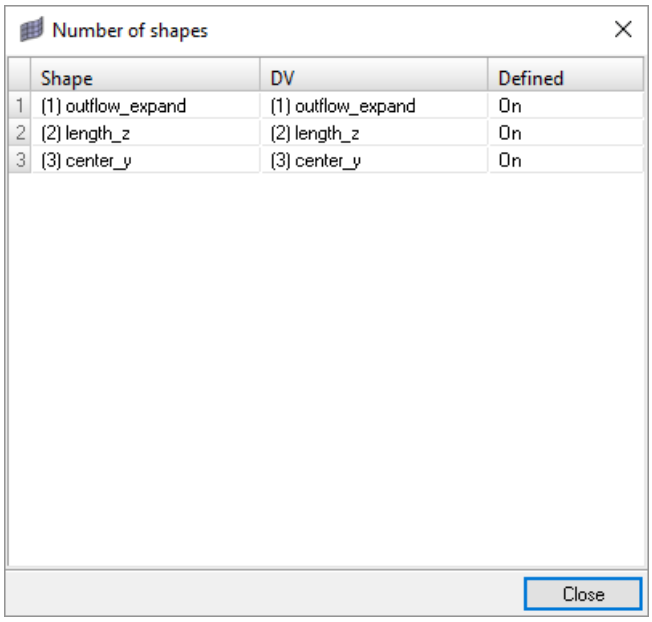


Figure 721:

8. Click **Close** to exit the dialog.

Set Up Optimization Controls

1. In the Entity Editor, expand **01.Global** > **06.Optimization**.
2. In the Entity Editor, select **maximize_inflow_pressure** as the Objective.
3. Verify that the Optimizer convergence tolerance is set to `1e-4`.
4. Save the model.

Compute the Solution

In this step, you will launch AcuSolve directly from HyperMesh and compute the solution.

Run AcuSolve

1. Turn on the visibility of all mesh components.
   For the analysis to run, the mesh for all active components must be visible.
2. Click on the ACU toolbar.
   The **Solver job Launcher** dialog opens.
3. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
4. Leave the remaining options as default and click **Launch** to start the solution process.

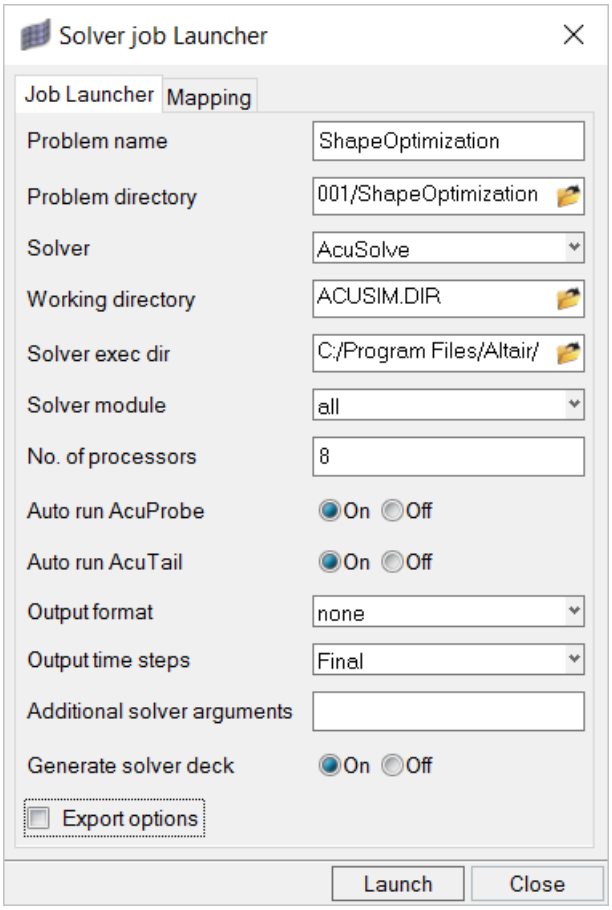


Figure 722:

Monitor the Solution with AcuProbe

AcuProbe can be used to monitor various variables over solution time. An AcuProbe window will be launched by the **Solver Job Launcher** if the Auto run AcuProbe flag is set to On.

1. In the Data Tree on the left, expand **Surface Output** > **Inflow** > **Pressure**.
2. Right-click on **pressure** and select **Plot**.

> **Note:** You might need to click on the toolbar in order to properly display the plot.

3. Select **Edit** > **Toggle X-Axis-Case** to change the x-axis from Time Step to Case ID.

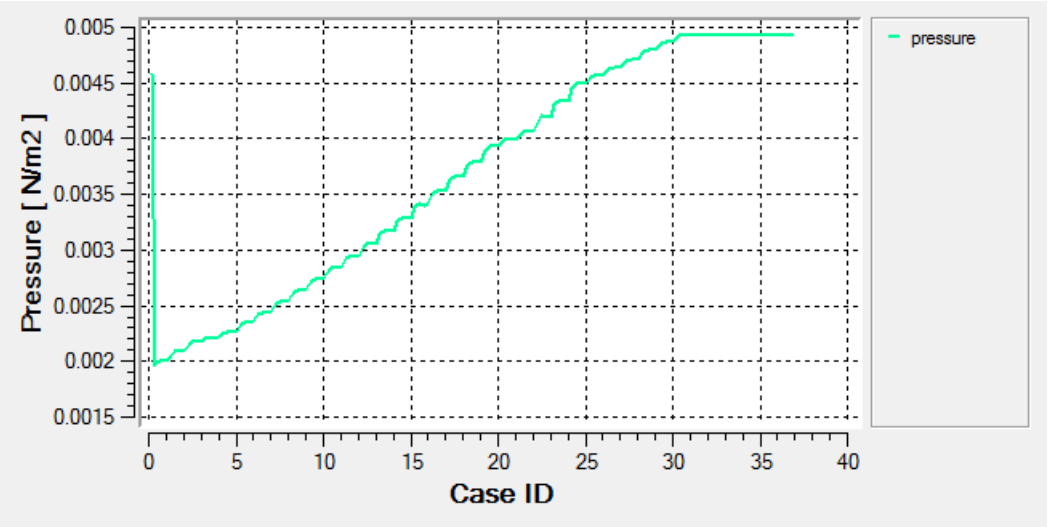


Figure 723:

AcuGetDv and AcuGetRsp

AcuSolve provides two post-processing utilities specific to optimization problems, AcuGetDv and AcuGetRsp. AcuGetDv provides the values of the design variables for all the cases included in the solution. AcuGetRsp provides the values of the response variables for all the cases included in the solution.

1. Start AcuSolve Command Prompt from the Windows Start menu by clicking **Start** > **Altair <version>** > **AcuSolve Cmd Prompt** .
2. Change the directory to the present working directory using the `cd` command.
3. Enter the following command at the prompt:

   ```
   acuGetDv
   ```

   AcuSolve will print the values of design variables for each case.

```
C:\AcuSolve_Tutorials\Shape_Optimization>acuGetDv
   1           0.7           0.7           0.7
   2         0.726         0.726         0.674
   3        0.7234        0.7234        0.6766
   4      0.698655       0.70137       0.70157
   5      0.698953      0.703238      0.703246
   6      0.699356      0.703051      0.703078
   7      0.699316      0.679549      0.699002
                              ≀
 119      0.201199      0.203565           0.2
 120           0.2      0.201168           0.2
 121           0.2           0.2           0.2
 122           0.2           0.2           0.2
 123           0.2           0.2           0.2
 124           0.2           0.2           0.2
 125           0.2           0.2           0.2
 126           0.2           0.2           0.2
```

Figure 724:

4. To print the values of response variables, use the command:

   `acuGetRsp`

   > **Note:** The order of columns in which the design variables and the response variables are printed by these commands is the order in which they appear in the INP file.

```
C:\AcuSolve_Tutorials\Shape_Optimization>acuGetRsp
1 0.00201736191626
2 0.00195648425489
3 0.00196241222916
4 0.00201776097532
5 0.00201469714215
6 0.00201435244455
7 0.00204301598197
8 0.00210890777212
9 0.00217662861536
10 0.00225424091755
        ≀
121 0.00495656634026
122 0.00495661799598
123 0.00495657739144
124 0.00495659037308
125 0.00495659550948
126 0.00495659846459
```

Figure 725:

Post-Process the Results with HyperView

Once the solution has converged, close the AcuProbe and AcuTail windows. Go to the HyperMesh window and close the AcuSolve Control tab.

Open HyperView and Load the Model and Results

1. In the HyperMesh main menu area, click **Applications** > **HyperView**.

   Once the HyperView window is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
2. In the Load model and results panel, click next to Load model.
3. In the **Load Model File** dialog, navigate to your working directory and select the AcuSolve `.Log` file for the solution run that you want to post-process. In this example, the file to be selected is `ShapeOptimization.1.Log`.
4. Click **Open**.
5. Click **Apply** in the panel area to load the model and results.

   The model is colored by geometry after loading.

Create the Pressure Variation Animation

1. Click on the Results toolbar to open the Contour panel.
2. Select **Pressure** as the Result type.
3. Click the **Components** entity selector. In the **Extended Entity Selection** dialog, select **Displayed**.
4. Click **Apply** to display the pressure contour.
5. In the panel area, under the Display tab, turn off the Discrete color option.

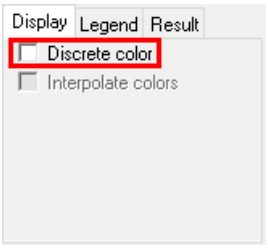


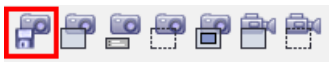
Figure 726:

6. Click the **Legend** tab then click **Edit Legend**.
7. In the **Edit Legend** dialog, change the Type to **Dynamic scale**, the Numeric format to **Fixed**, and the Numeric precision to `6`. Click **OK** to close the dialog.
8. On the Animation toolbar, click the Animation Controls icon .
9. Drag the Max frame Rate slider to `10` fps.
10. Click the Start/Pause Animation icon to play the animation in the graphics area.

Save the Animation

1. In the menu area, select **Preferences** > **Export Settings** > **AVI**.
2. In the **Export Settings AVI** dialog, set the Frame rate to `10` fps.

3. Set the JPEG quality to `99` and click **OK**.
4. On the ImageCapture toolbar, make sure that the Save Image to File option is On.



5. Click the Capture Graphics Area Video icon .
   The **Save Graphics Area Video As** dialog opens.
6. Navigate to the location where you want to save the file, enter a name of your choice, and click **Save**.

Summary

In this tutorial, you learned how to set up and solve a shape optimization problem with AcuSolve using HyperMorph. You started by importing the model database and then created mesh morphs. Then, you defined the design variables, the response variables, and set up the objectives of the problem using the response variables. Once the solution was computed using AcuSolve, you used the AcuSolve Command Prompt to get the design variables and response variables used for the optimization cases. Finally, you used HyperView to visualize how the shape of the pipe changed with the optimization steps.

2.7.3 ACU-T: 7010 Shape Optimization using HyperStudy

Prerequisites

Prior to starting this tutorial, you should have already run through the introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, and have a basic understanding of HyperMesh, AcuSolve, and HyperView. To run this simulation, you will need access to a licensed version of HyperMesh and AcuSolve.

Prior to running through this tutorial, copy `HyperMesh_tutorial_inputs.zip` from `<Altair_installation_directory>\hwcfdsolvers\acusolve\win64\model_files\tutorials\AcuSolve` to a local directory. Extract `ACU-T7010_HyperStudy.hm` from `HyperMesh_tutorial_inputs.zip`.

Since the HyperMesh database (`.hm` file) contains meshed geometry, this tutorial does not include steps related to geometry import and mesh generation.

Problem Description

The geometry for this problem consists of a simple pipe channel with perfectly circular cross-section as the base shape. Water enters the Inlet at the rate of 0.0003 kg/s and the outlet is a standard pressure outlet at zero relative pressure. Walls of the channel are no-slip walls.

The objective is to generate the pressure drop between the inlet and outlet with the effect of changes in pipe shape. After performing the baseline simulation, a DOE study will be executed to analyze the effect of changes in pipe shape on the pressure drop between inlet and outlet. This is just to illustrate one of the many types of studies that can be done using AcuSolve with HyperMesh and HyperStudy.

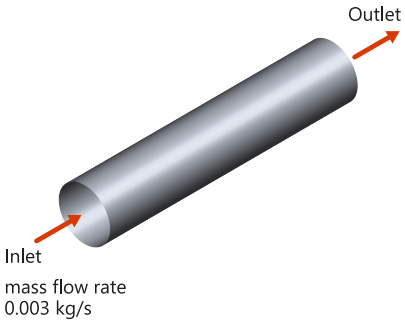


Figure 727:

Open the HyperMesh Model Database

1. Start HyperMesh Desktop and load the AcuSolve user profile.

   Refer to the HM introductory tutorial, ACU-T: 1000 HyperWorks UI Introduction, to learn how to select AcuSolve from **User Profiles**.
2. Click the Open Model icon located on the standard toolbar.

   The **Open Model** dialog opens.
3. Browse to the directory where you saved the model file. Select the HyperMesh file `ACU-T7010_HyperStudy.hm` and click **Open**.
4. Click **File** > **Save As**.
   The **Save Model As** dialog opens.
5. Create a new directory named `Pipe_HyperStudy` and navigate into this directory.

   This will be the working directory and all the files related to the simulation will be stored in this location.
6. Enter `Pipe_HyperStudy` as the file name for the database, or choose any name of your preference.
7. Click **Save** to create the database.

Specify the Boundary Conditions

In this simulation, you will use the standard simulation parameters and solver settings.

By default, all components are assigned to the wall boundary condition. In this step, you will change them to the appropriate boundary conditions and assign material properties to the fluid volumes.

1. In the Solver Browser, expand **12.Surfaces** > **WALL**.
2. Click **Fluid**. In the Entity Editor,
   a) Change the Type to **FLUID**.
   b) Set the Material to **Air_HM**.

Name	Value
Name	Fluid
Type	FLUID
Material	(1) Air_HM
FE style	
Geometry style	
Element Set	
Body force	<Unspecified>
Reference frame	<Unspecified>
Element Output	
Activate Element Output	Off
Unsupported Entries	
Number of Unsupported Entries	0

Figure 728:

3. Click **Inflow**. In the Entity Editor,
   a) Change the Type to **INFLOW**.
   b) Change the Inflow type to **Mass flux**.
   c) Set the Mass flux value to `0.0003` kg/sec.

Name	Value
Name	Inflow
Type	INFLOW
FE style	
Geometry style	
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Inflow type	Mass flux
Mass flux ## (kg/sec)	0.0003
Reference frame	<Unspecified>
Multiphase incoming field multiplier function	<Unspecified>
Integrated Boundary Conditions	
Radiation Surface	
Solar Radiation Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 729:

4. Click **Outflow**. In the Entity Editor, change the Type to **OUTLFOW** and leave the remaining options as default.

Name	Value
Name	Outflow
Type	OUTFLOW
FE style	
Geometry style	
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Pressure ## (N/m2)	0.0
Pressure loss factor	0.0
Hydrostatic pressure	
Back flow conditions	
Integrated Boundary Conditions	
Radiation Surface	
Solar Radiation Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 730:

5. Click **Wall**. In the Entity Editor, verify that the Type is set to **WALL**.

Name	Value
Name	Wall
Type	WALL
FE style	
Geometry style	
Simple Boundary Condition	
Activate simple boundary condition	On
Show advanced features	Off
Element Boundary Condition	
Radiation Surface	
Solar Radiation Surface	
Guide Surface	
Interpolated Motion Surface	
Surface Output	
Unsupported Entries	

Figure 731:

6. Save the model.

Set the Optimization Parameters

Generate and Export Morph Shapes

1. In the panel area, click **HyperMorph** then select **morph volumes**.
2. In the create sub-panel, click the **elems** collector and select **all**.
3. Click **create**.
   A new morph volume is created.

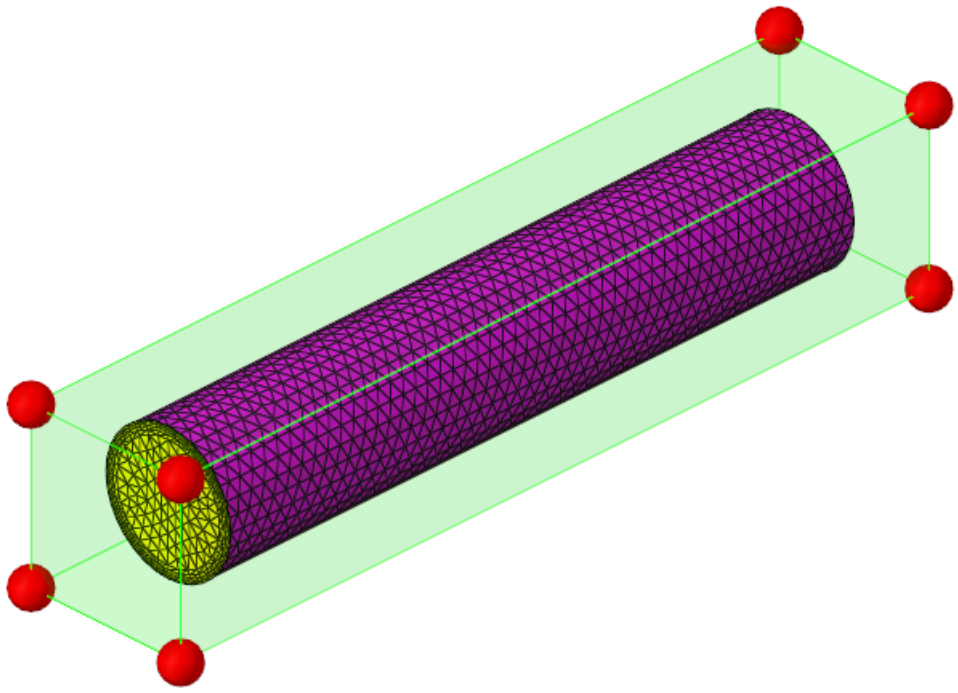


Figure 732:

4. Go to the **split/combine** sub-panel. In the panel area, toggle the option to **# of splits** and set it to 3.



Figure 733:

5. In the graphics area, select the edge shown in the figure below.

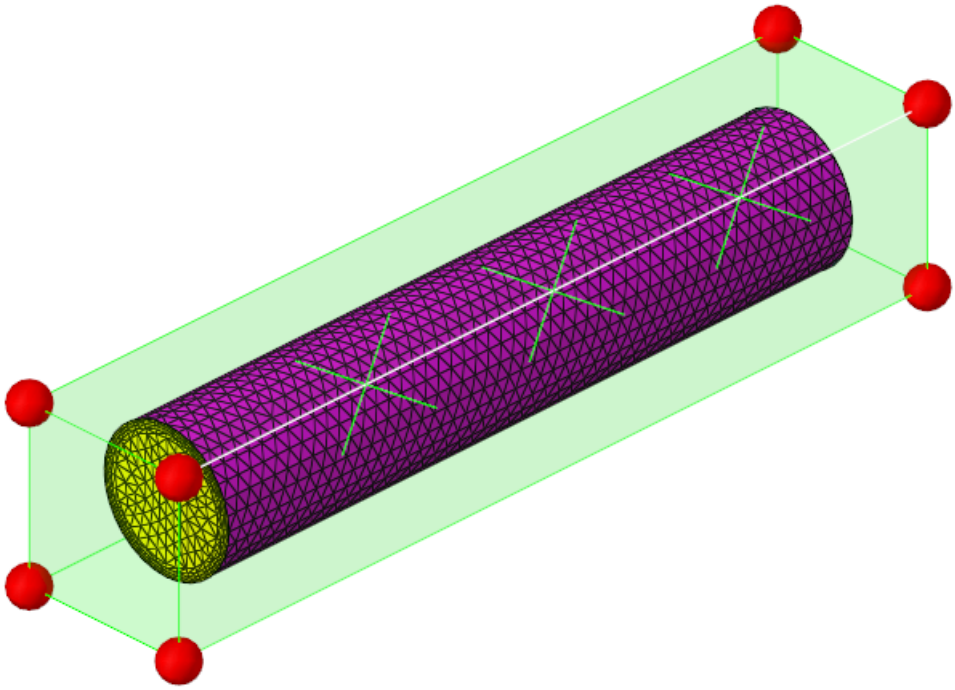


Figure 734:

6. In the panel area, click **split**.
The morph volume is split into smaller volumes.

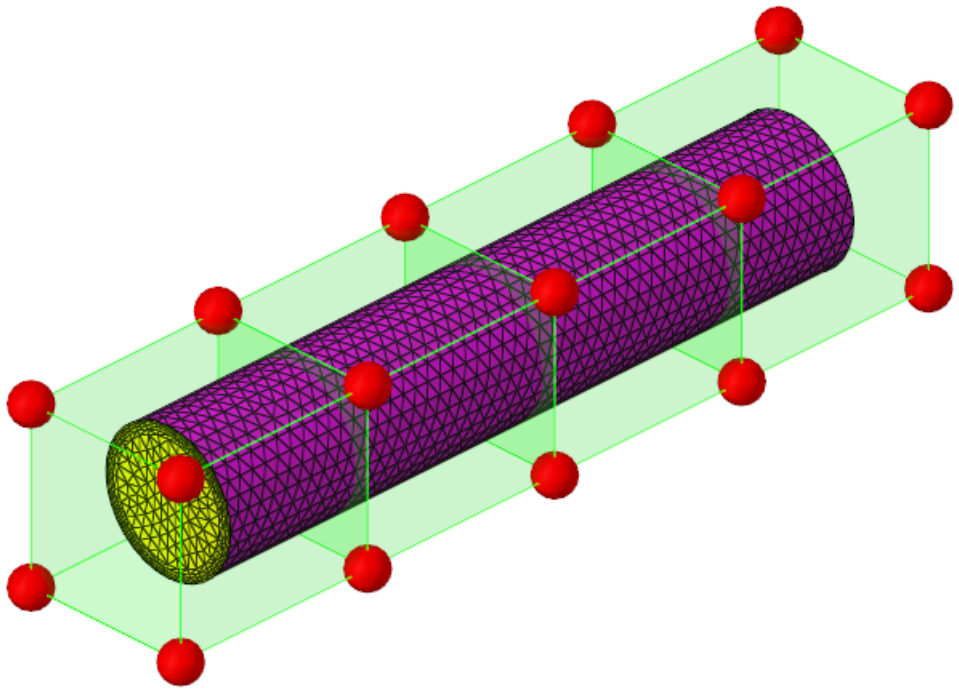


Figure 735:

7. In the panel area, go to the **update edges** sub-panel. Click the first arrow and select **update ends**. Then, click the second arrow and select **by mvols**. Finally, click the third arrow and select the **main-secondary** option.

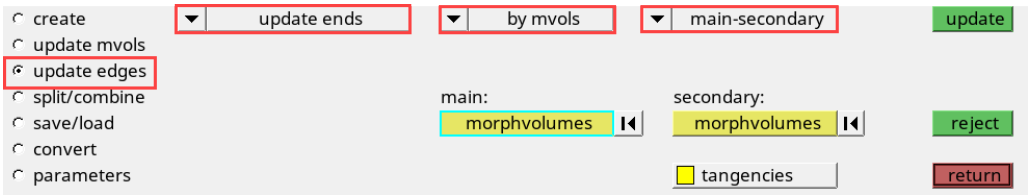


Figure 736:

8. Activate the **main morphvolmes** collector and select the outer two morph volumes shown in the figure below. Then, activate the **secondary morphvolumes** collector and select the inner two morph volumes.

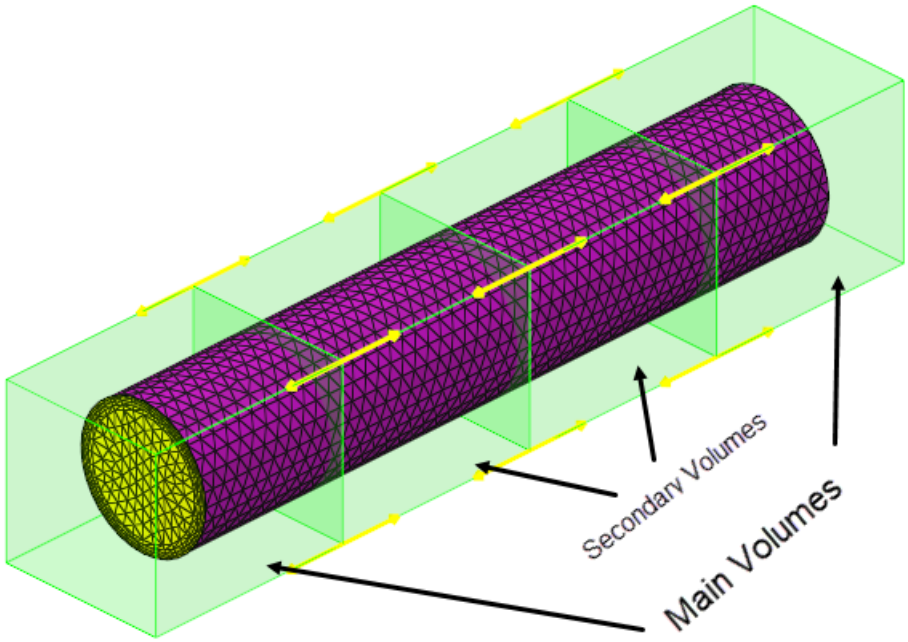


Figure 737:

9. After selecting the volumes in the order mentioned, click **update**.
The edges in the volume should resemble the figure shown below.

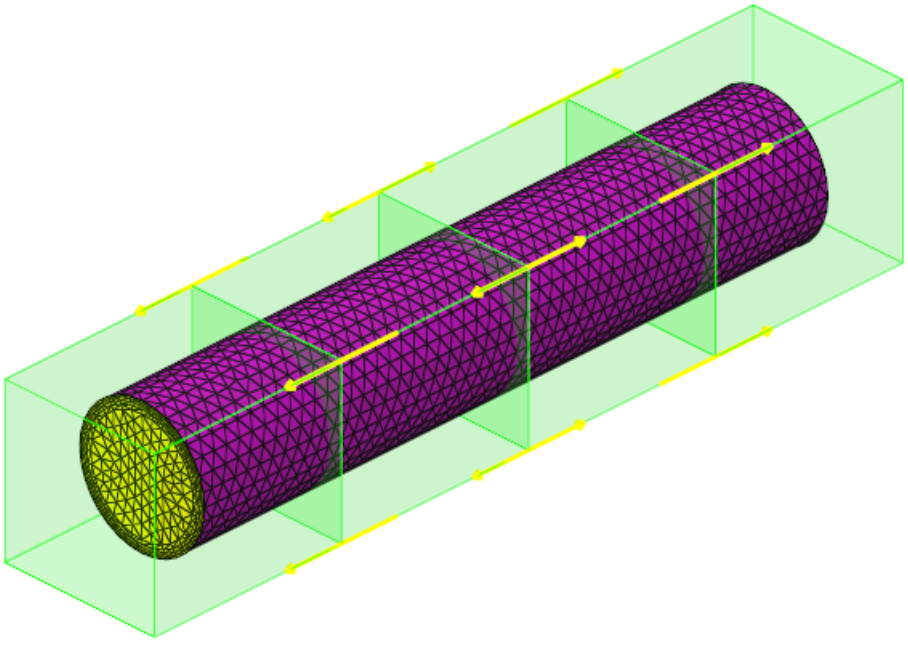


Figure 738:

10. Click **return** to go the HyperMorph panel. Select **morph** in the panel area then select the **move handles** sub-panel. In this panel,

a) Click the second arrow and change it from interactive to **translate**.

b) Below that, click the arrow and select **along vector**.

c) Under that, set the orientation selector to **y-axis**.

d) In the dist= field, enter `0.01`.

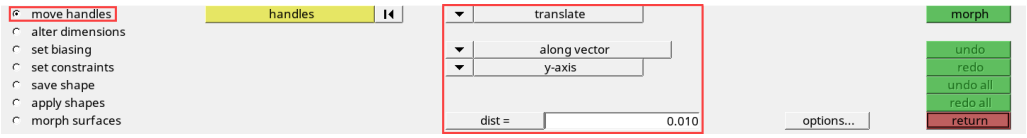


Figure 739:

11. Activate the **handles** collector and select the four middle handles as shown in the figure below.

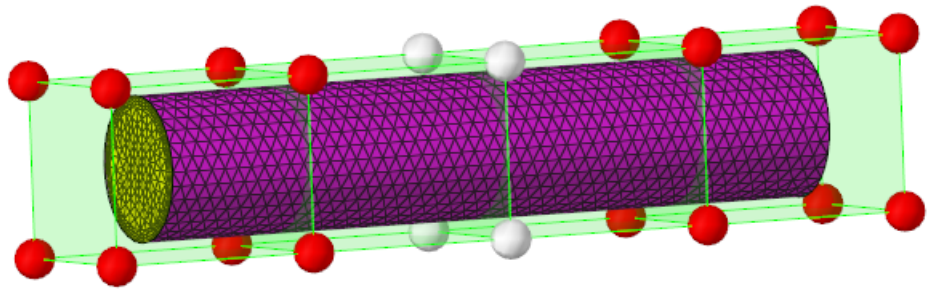


Figure 740:

12. In the panel area, click **morph**.
The grid is morphed.

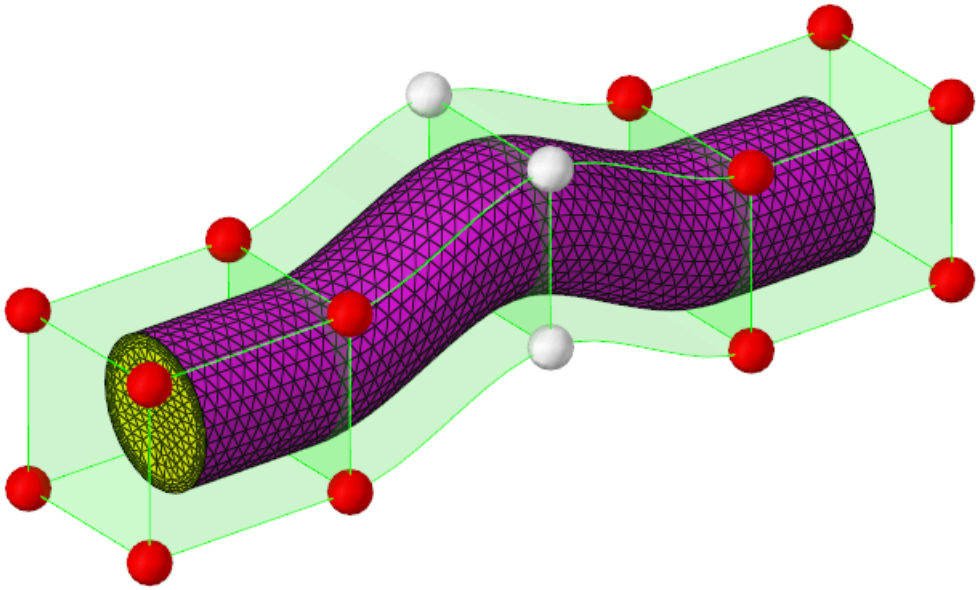


Figure 741:

13. Go to the **save shape** sub-panel. In this panel,
    a) Set the name field to `shape_1`.
    b) In the second row, set the selector to **as node perturbations**.
    c) Click **save**.
    d) Select **Yes** when asked to save perturbations for nodes at global and morph volume handles.

    A new folder named Shapes is created in the Model Browser. The display of the saved shape can be shown or hidden by right-clicking on the shape_1 and selecting the appropriate option.
14. Click **undo all** then click **return** to exit the panel.
15. Go to the **Utility Menu**.

    If you do not see the utility menu, click **View** on the menu bar and select **Browsers** > **HyperMesh** > **Utility** from the drop-down.
16. In the utility menu, click **Disp** then click **Clear Temp Nodes** to remove any temporary nodes in the model.

Define the Design Variable

1. Click **Design Study** on the menu bar and select **Define DV** from the drop-down.
2. In the panel area, in the **desvar** sub-panel, enter `bend` as the name in the desvar= field.
3. Click on the **shape=** field and select **shape_1**.
4. Click **create** to create a design variable named 'bend'.

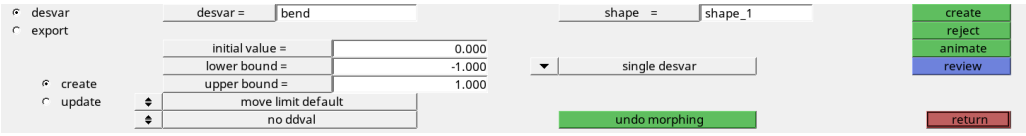


Figure 742:

5. Click **return** to exit the panel.
6. Save the model.

Launch the DOE Study Using HyperStudy

Start the Nominal Run

1. Click on the CFD toolbar.

   The **HyperStudy Job Launcher** dialog.
2. Optional: For a faster solution time, set the number of processors to a higher number (4 or 8) based on availability.
3. Verify that the Solver is set to AcuSolve.
4. In the Define Responses table, select the following to identify what the pressure change will be at inflow due to shape changes.
   a) Set Responses to **Pressure**.
   b) Set Components to **Inflow**.
5. Verify that both **Launch HyperStudy without nominal run** and **Export options** are left unchecked.
6. Click **Launch**.

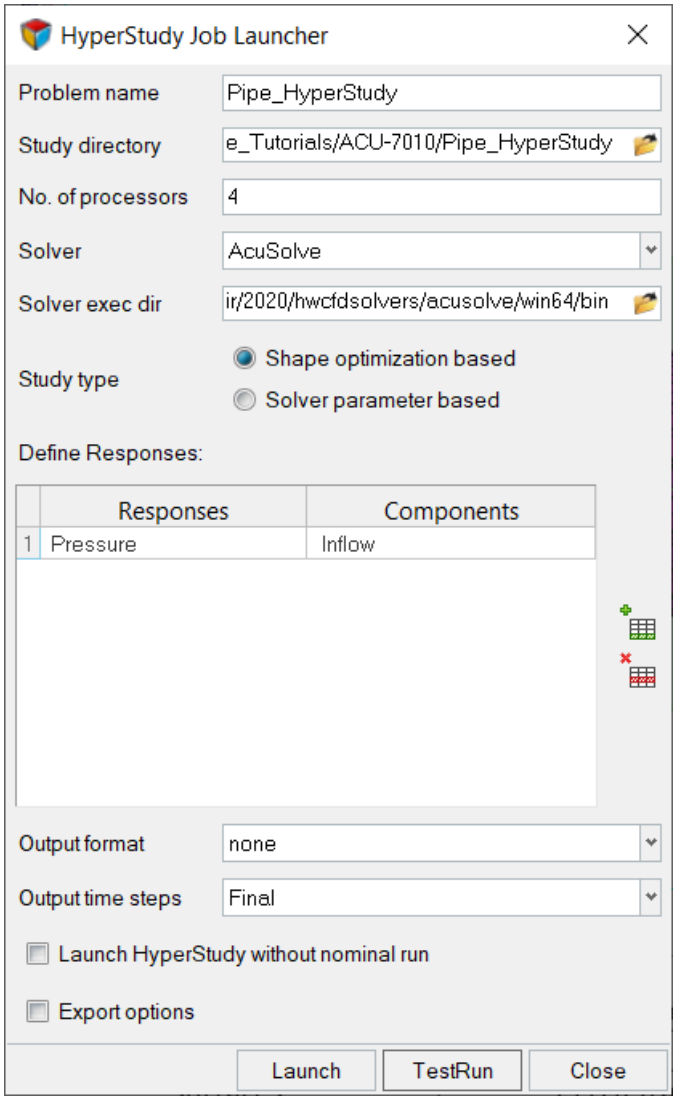


Figure 743:

7. Click **Yes** in the prompt that appears.

A nominal run is launched and the AcuTail and AcuProbe windows are launched automatically. Once the nominal run is completed, the HyperStudy window is launched.

In the HyperStudy window, check step by step by clicking on each setup from the Explorer menu to ensure that everything was properly defined.

Run the DOE Study

1. In the Explorer window, right-click on **Study 1** and select **Add**.

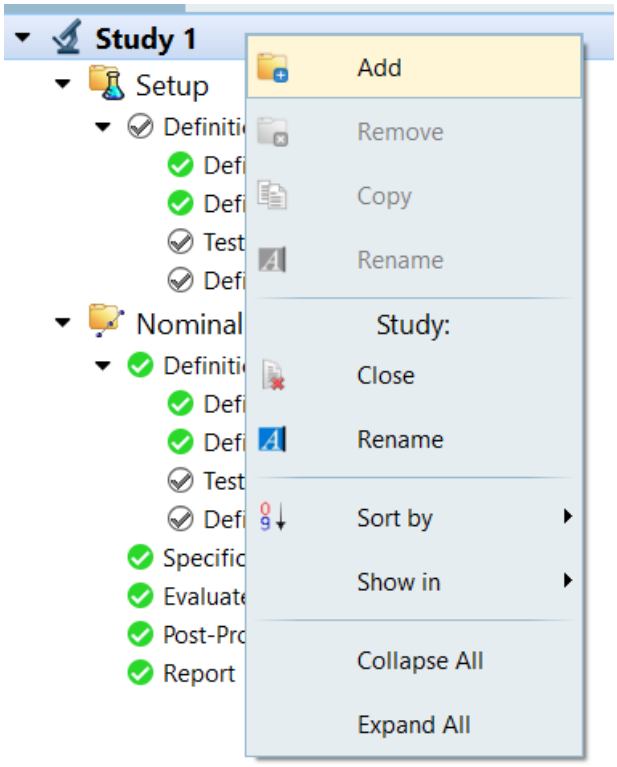
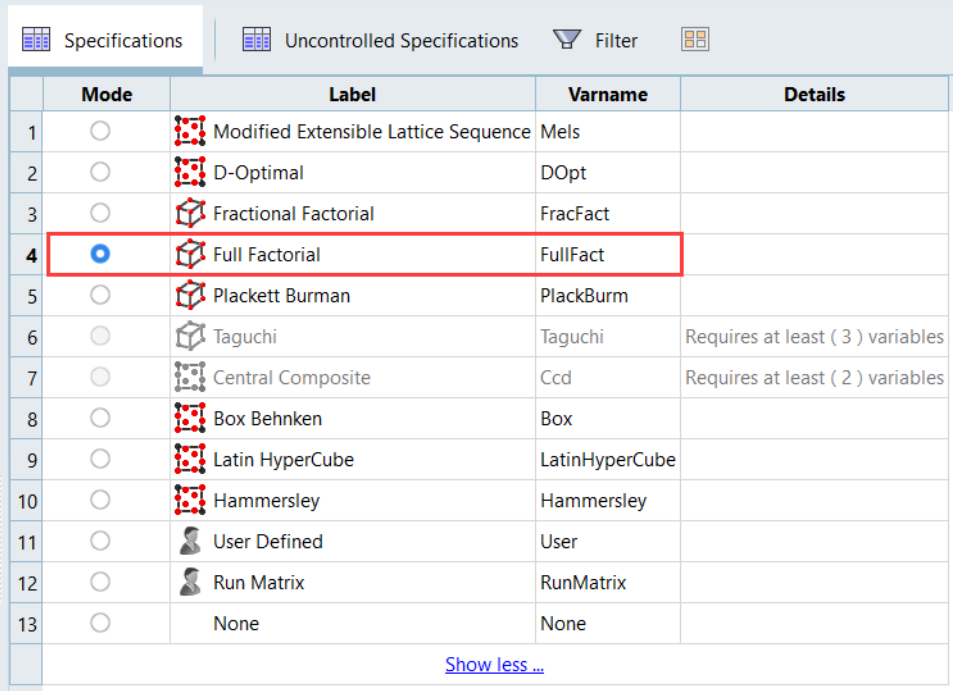


Figure 744:

2. In the **Add – Altair HyperStudy** dialog, select **DOE** then click **OK**.
   A new study folder named DOE 1 is created in the Explorer.
3. In the Explorer, click **Specifications** under DOE 1.
4. In the Work Area, set the Mode to **Full Factorial** by expanding the **Show more...** option.



	Mode	Label	Varname	Details
1	○	Modified Extensible Lattice Sequence	Mels	
2	○	D-Optimal	DOpt	
3	○	Fractional Factorial	FracFact	
4	◉	Full Factorial	FullFact	
5	○	Plackett Burman	PlackBurm	
6	○	Taguchi	Taguchi	Requires at least (3) variables
7	○	Central Composite	Ccd	Requires at least (2) variables
8	○	Box Behnken	Box	
9	○	Latin HyperCube	LatinHyperCube	
10	○	Hammersley	Hammersley	
11	○	User Defined	User	
12	○	Run Matrix	RunMatrix	
13	○	None	None	

Show less ...

Figure 745:

5. Click the **Levels** tab, set the number of levels to `5`, then click **Apply**.



Figure 746:

6. In the Explorer, click **Evaluate** under DOE 1.
   The table used to run the study appears, showing all of the runs (1 - 5) to be executed.

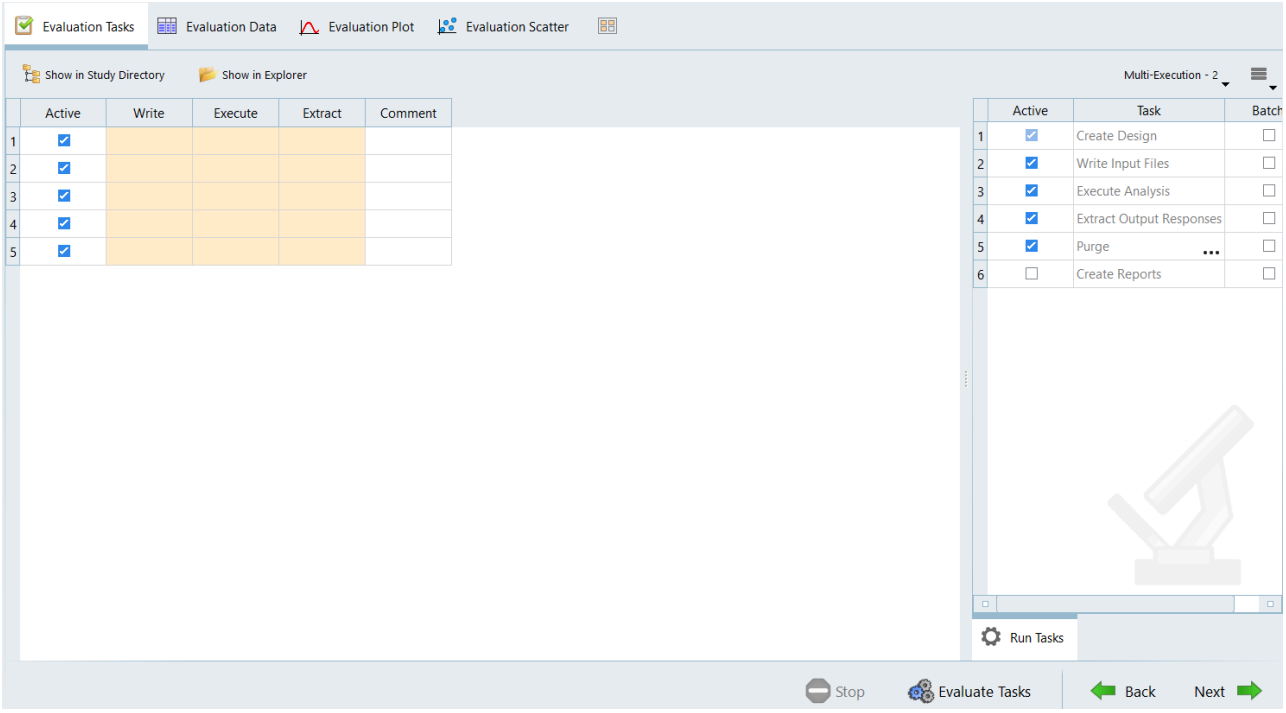


Figure 747:

7. Click **Evaluate Tasks** at the bottom of the Work Area.
   As the runs progress, the status of each run will be updated in the table in the Work Area.
8. Once all the runs are complete, click the **Evaluation Data** tab.
   The table displays the list of values used for the design variable and also the corresponding value of the response variable (inflow pressure).

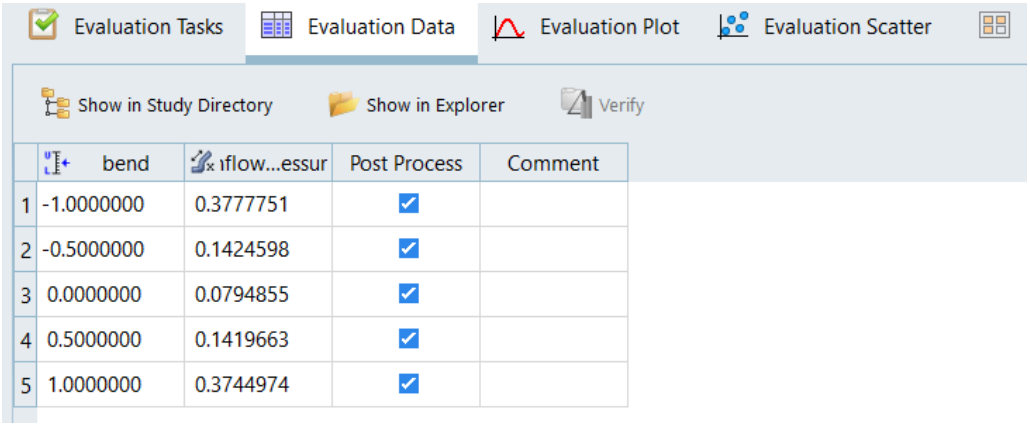


Figure 748:

9. In order to view the above table as a plot, click the **Evaluation Plot** tab.
10. Plot the values of the response variable (inflow pressure) by selecting it in the Channel selector.

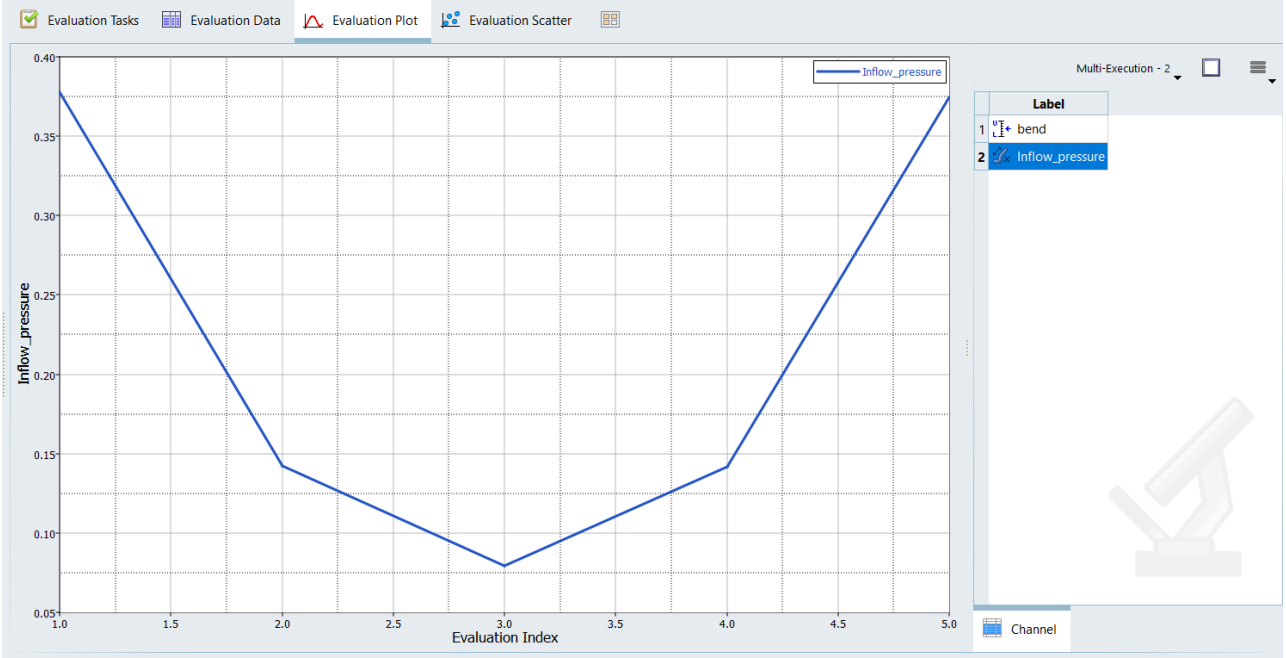


Figure 749:

You can also plot the list of design variables by switching the label to bend.

Post-Process the Results with HyperView

Switch to the HyperView Interface and Load the AcuSolve Model and Results

1. In the HyperMesh Desktop window, click the **ClientSelector** drop-down in the bottom-left corner of the graphics window.

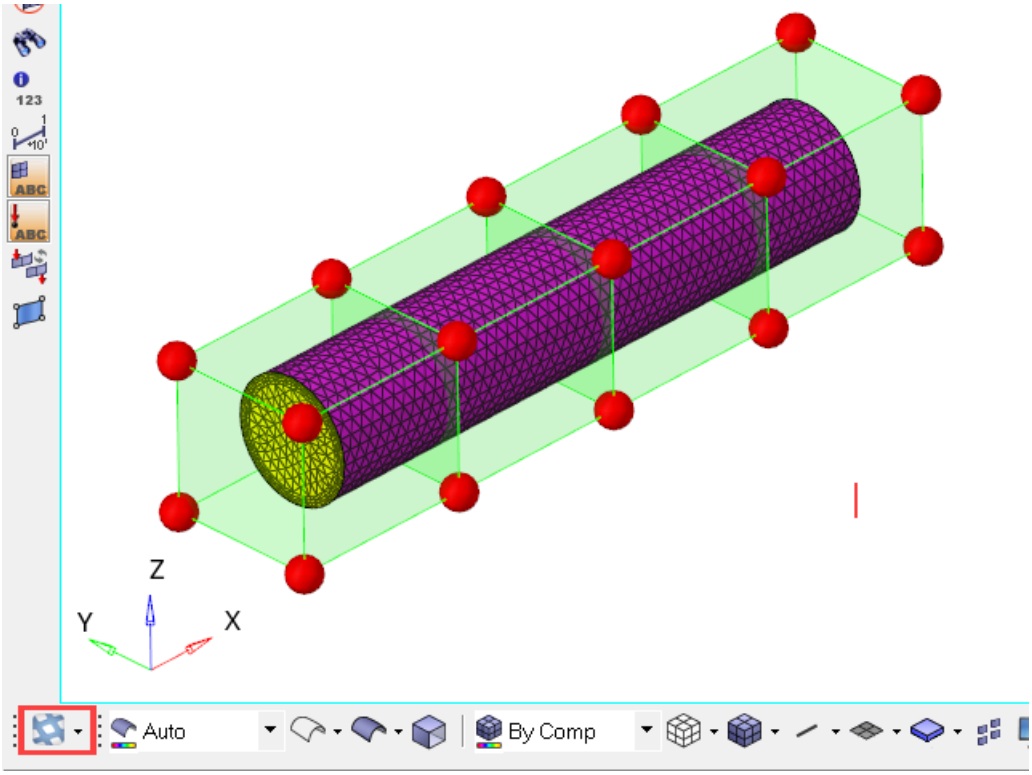


Figure 750:

2. Select **HyperView** from the list.
3. In the pop-up dialog that appears, click **Yes**.
   The interface is changed to HyperView.

   Once HyperView is loaded, the Load model and results panel should be open by default. If you do not see the panel, click **File** > **Open** > **Model**.
4. In the Load model and results panel, click next to Load model.
5. In the **Load Model File** dialog, navigate to your working directory and then navigate into the following directory …\approaches\doe_1\run_00001\m_1\. Select the `Pipe_HyperStudy.1.Log` file.
6. Click **Open**.
7. Click **Apply** in the panel area to load the model and results of the first run of the DOE study.

   The model is colored by geometry after loading.

Create a Pressure Contour on a Cut Plane

In this step, you will create a contour plot of pressure on a section cut on the z-axis.

1. Click on the Results toolbar to open the Contour panel.
2. In the panel area, change the Result type to **Pressure (s)**.
3. Click **Apply** to create a pressure contour plot.

4. Click the **Section cut** icon on the HV-Display toolbar.
5. In the panel area, click **Add** to create a new section cut named Section 1.
6. In the Define plane section, set the axis to **Z Axis** then click **Apply**.
7. Set the Z Base coordinate to `0` and press **Enter**.
8. Click **Gridline**. In the **Gridline Options** dialog, deactivate the **Show** check box under Grid line then click **OK**.

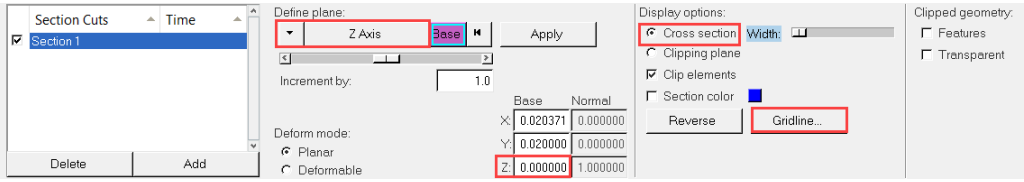


Figure 751:

9. Orient the display to the xy-plane by clicking on the Standard Views toolbar.

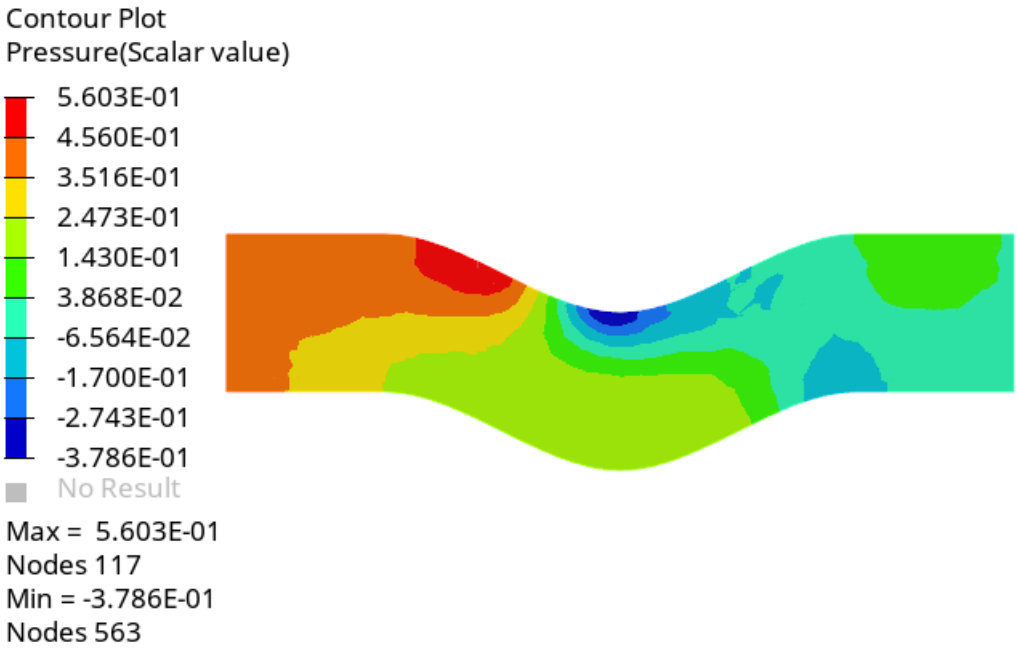


Figure 752: Run 1

10. Repeat the above steps to create a contour plot of pressure for each individual run using the respective log files.

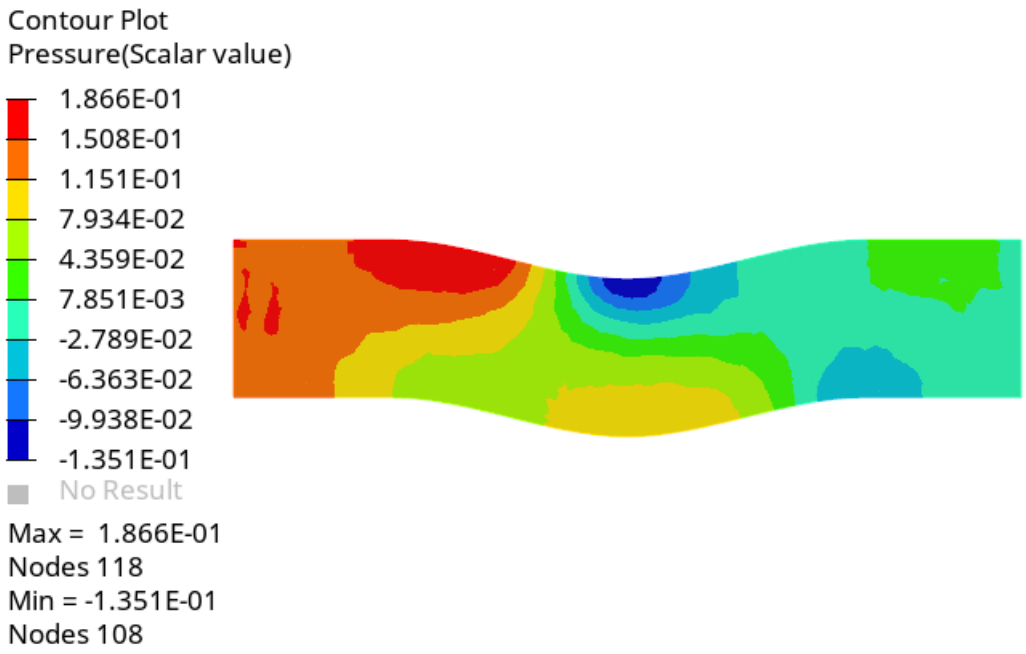


Figure 753: Run 2

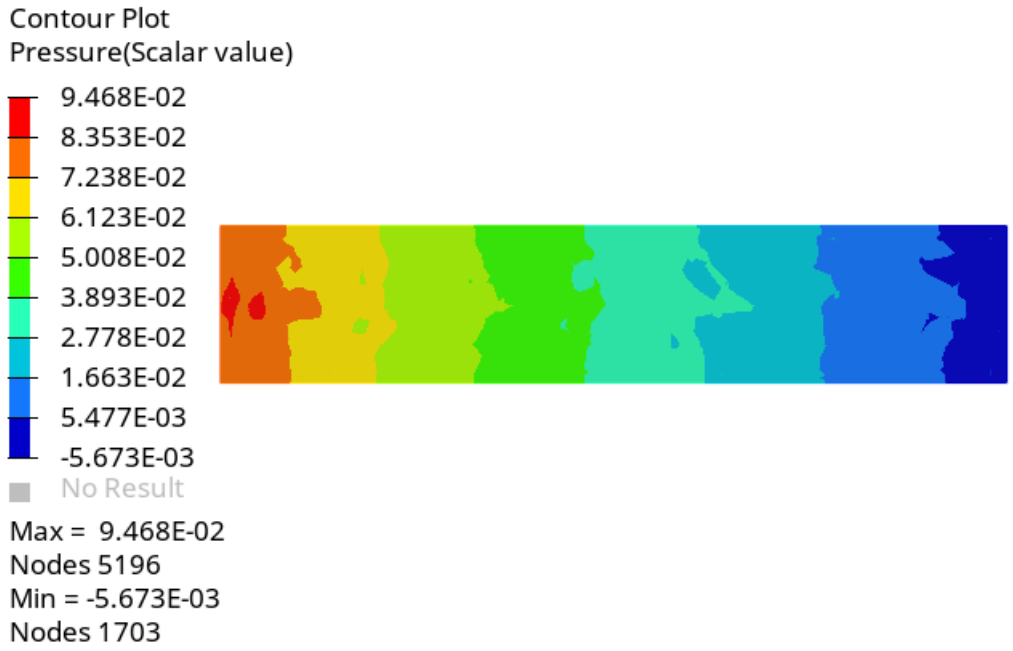


Figure 754: Run 3

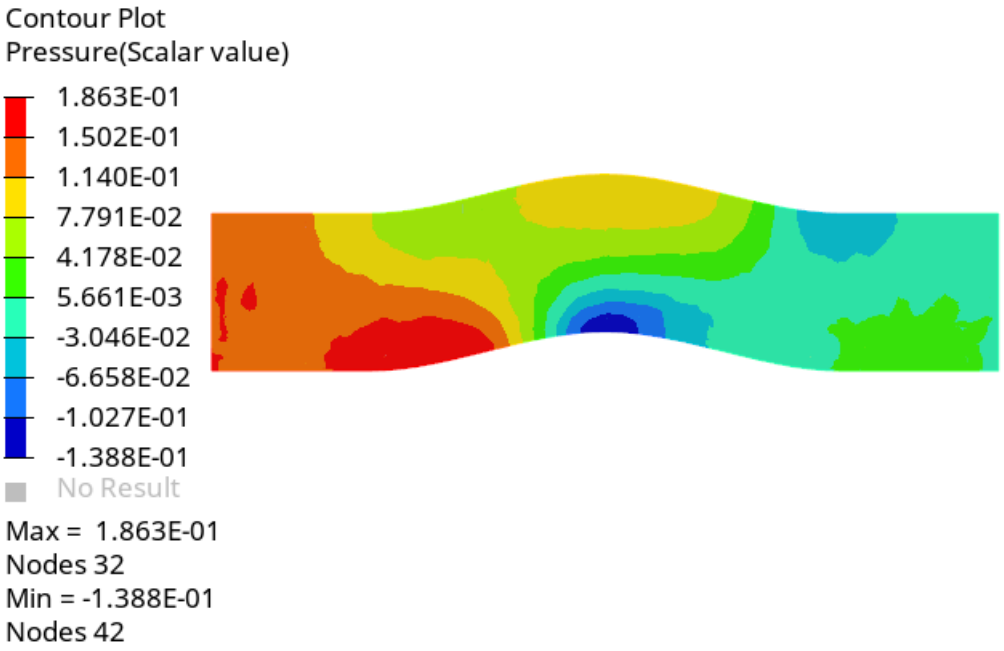


Figure 755: Run 4

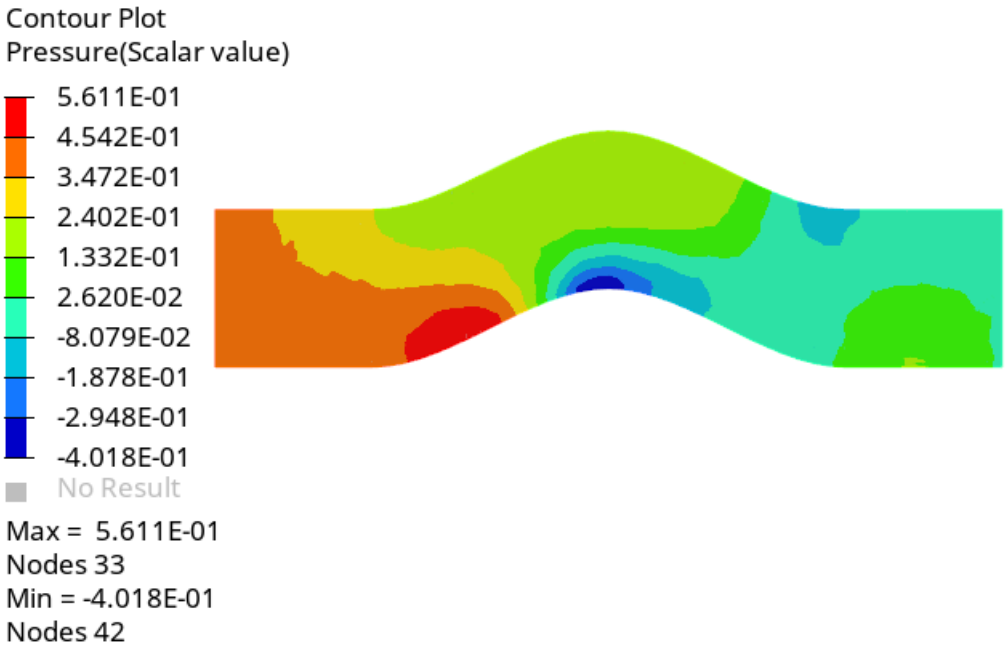


Figure 756: Run 5

Summary

This tutorial introduced you to running a DOE study using Altair products, namely HyperMesh, AcuSolve, HyperStudy and HyperView. You started by importing a HyperMesh database and then set up the simulation parameters for AcuSolve and created a morph shape using HyperMorph. Next, you set up the

design variable and linked the morph shape to it. Then, you proceeded to set up the DOE study. Once the results were obtained, you processed the results using HyperView.

ANSYS

3

ANSYS tutorials.

This chapter covers the following:

- 3.1 HM-4400: Explore the ANSYS Interface (p. 584)
- 3.2 HM-4410: Set Up a Model in ANSYS (p. 591)
- 3.3 HM-4420: Define ANSYS Contacts for Models in HyperMesh (p. 624)
- 3.4 HM-4430: Define ANSYS Contacts for Models in HyperMesh (p. 636)
- 3.5 HM-4450: Introduction to HyperBeam (p. 650)
- 3.6 HM-4460: Composite (p. 667)

3.1 HM-4400: Explore the ANSYS Interface

In this tutorial you will learn about HyperMesh's ANSYS FE input reader, template, macro menu, and user profile.

Before you begin, copy the `tuning_fork_ansys.cdb` model file from `<hm.zip>/interfaces/` to your working directory.

3.1.1 Tools Overview

In this section you will learn about ANSYS FE-input reader, ANSYS Template, ANSYS Utility menu, ANSYS Interface Online Help, ANSYS User Profile, Model Import, and Model Export.

ANSYS FE Input Reader

ANSYS Solver is one of the leading software products in the computer aided engineering (CAE) field. This software is used to simulate designs in general engineering and aerospace industries. ANSYS has the capability to solve structural, thermal, and fluid models. Different analysis like static structural, dynamic, modal, harmonic, buckling, spectrum, steady state and transient thermal, and radiation can be solved. Both linear and non-linear analysis are supported by ANSYS.

HyperWorks has recognized ANSYS as one of the major Solvers available in the current CAE field. HyperWorks has extended its support to ANSYS Solver by recognizing its element types, solution methods, and file formats. You get benefits from using both HyperWorks tools and ANSYS Solver. During pre-processing of the model, you can define ANSYS elements and apply loads and boundary conditions that can be read by ANSYS. Once the analysis is done in ANSYS, both deck and result files can be read back in HyperMesh and HyperView. You can post-process the results of the analysis solved by ANSYS.

ANSYS Solver writes analysis model data in the form of a script file. It also writes the model in binary file format. HyperMesh and HyperView can read the model data written in script format. These files are saved with file extension .cdb. Newer versions of ANSYS use BLOCK format to write their model data in script file. HyperMesh and HyperView can transfer the ANSYS model data into HyperMesh.

ANSYS Solver writes model data in BLOCK format. Older versions of ANSYS (ANSYS 5.4 or earlier) uses APDL format.HyperMesh can read both types.

ANSYS Utility menu

To open the ANSYS Utility menu, click **View** > **Browsers** > **HyperMesh** > **Utility** from the menu bar. The ANSYS Utility menu contains the following macros and tools.

Component Manager

Create, edit, and view HyperMesh components. This tool is available across all Solvers, but it has been customized for ANSYS. From the Component Manager, you can list the element types, thickness for shell elements, property type, material name, number of elements, and nodes in the component for each component.
To update element attributes, click **Table** > **Editable** from the menu bar.

To create new components, click **Action** > **New** from the menu bar. If components were created incorrectly, the Remarks column will inform. Incorrect components will be displayed in red.

Material

Create, edit, and view material collectors in your model.
When started, you can create both MP and MPDATA material sets, and edit the name and ID of the material set, as well as material values.

Section

Create beam and shell sections.
When started, the **Section** dialog opens and lists all of the existing beam or shell sections with subtypes. Click **New** to create new sections. You can use HyperBeam to create sections and then attach them to beamsection card images. Click **Edit** to edit sections by their name, ID and type.

Real Sets

Create and update ANSYS real properties.
When started, a table that lists the real sets in the model opens, and you can review the values in each property set. Click **New** to create new real sets, or go to the card images of each real set. Click **Edit** to update by ID, element type or values.

ET Type

Create and update ANSYS ET Types.
When started, a table listing ET Types in the model opens and you can review the values in each ET Type. Click **New** to create new ET Types or to go to card images of each ET Type. Click **Edit** to update by ID, element type or values.

Surface Manager

Generates thermal, radiosity, and structural surface elements. The **Ansys Surface Manager** dialog has simple steps to create surface elements over base geometry elements.

Convert to Special 2nd Order Macro

Convert fully second order elements to special second order elements.
For more information on this marco, refer to the ANSYS Convert to Special 2nd Order Macro help.

Update Pre 8.0 HM Model

Updates old `.hm` files, prior to 70-SA2-043 update, to `.hm 8.0` files.
Once you load an old `.hm` file, click **Update Pre 8.0 HM Model**. If you do not update old `.hm` files, interface data such as ANSYS element types, properties, and sections will be lost. For complete transfer of data, it is advised that you complete the model in all respects in older versions before transferring to the new interface.

Convert Legacy Documents

Converts old contact pairs to new contact pairs.
Generally, automatically converts legacy files. However if there are discontHyperMeshinuities in the model, you will need to repair them before running this macro.

Import Tab

Access the Import tab by clicking **File** > **Import** > **Session, Model, Solver Deck, Geometry,** or **Connectors** from the menu bar, or by clicking  from the Standard toolbar.

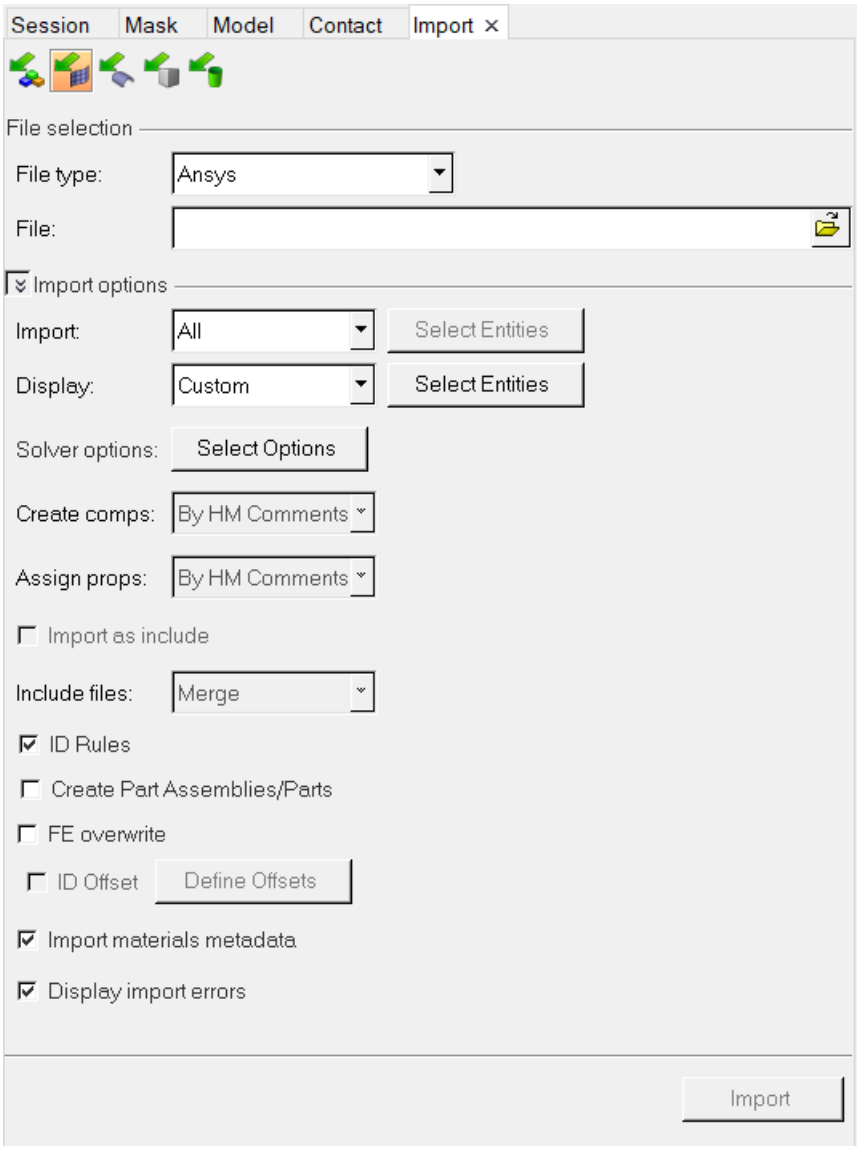


Figure 757:

The Import tab contains the following options:

- Import type: Use these icons to select the type of model you will be importing. For this exercise, click to import a FE model.
- File type: If you are in the ANSYS user profile, HyperMesh will automatically select ANSYS as the file type.
- File: From this location, navigate to your working directory and select the file you would like to import.
- Export: Choose to export All, Displayed, or Custom entities.
- Solver Options: Opens the **Ansys Options** dialog, from which you can select the **Combine loads in loadstep** check box. If selected, this option will combine point loads applied at the same location within a load step.

- Comments: Select the **HyperMesh** checkbox to export all HyperMesh comments; select the **Connector** checkbox to include connectors during export.
- Include Files: In ANSYS, this option only allows you to merge include files.
- Prompt to save invalid elements: Select this check box to save invalid elements.
- Prompt before overwrite: Select this check box to warn you before HyperMesh overwrites existing information.

Export Tab

Access the Export tab by clicking **File** > **Export** > **Model, Solver Deck, Geometry, Connectors** or **Curves** from the menu bar, or clicking  from the Standard toolbar.

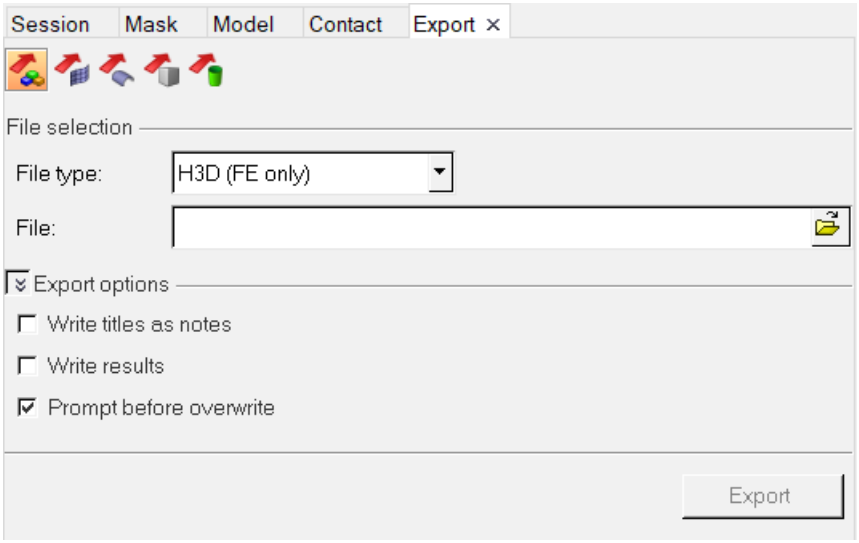


Figure 758:

The Export tab contains the following options:

- Export type: Use these icons to select the type of model you will be exporting. For this exercise, click to export a FE model.
- File type: If you are in the ANSYS user profile, HyperMesh will automatically select ANSYS as the file type to export a `.cdb` file.
- Template: If you are in the ANSYS user profile, HyperMesh will automatically select ANSYS as the default template.
- File: From this location, navigate to your working directory and select the name of the file you would like to export.
- Export: Choose to export All, Displayed, or Custom entities.
- Solver Options: Opens the **Ansys Options** dialog, from which you can select the Combine loads in loadstep check box. If selected, this option will combine point loads applied at the same location within a load step.
- Comments: Select the **HyperMesh** checkbox to export all HyperMesh comments; select the **Connector** checkbox to include connectors during export.
- Include Files: In ANSYS, this option only allows you to merge include files.

- Prompt to save invalid elements: Select this check box to save invalid elements.
- Prompt before overwrite: Select this check box to warn you before HyperMesh overwrites existing information.

3.1.2 Load the ANSYS User Profile

In this step, you will load the ANSYS user profile in HyperMesh.

Setting the user profile to ANSYS saves you time.

1. Start HyperMesh Desktop.
2. Select a user profile by completing one of the following options:
   - From the menu bar click **Preferences** > **User Profiles**
   - On the Standard toolbar click  .
3. In the **User Profile** dialog, select **Ansys** as indicated in Figure 759.

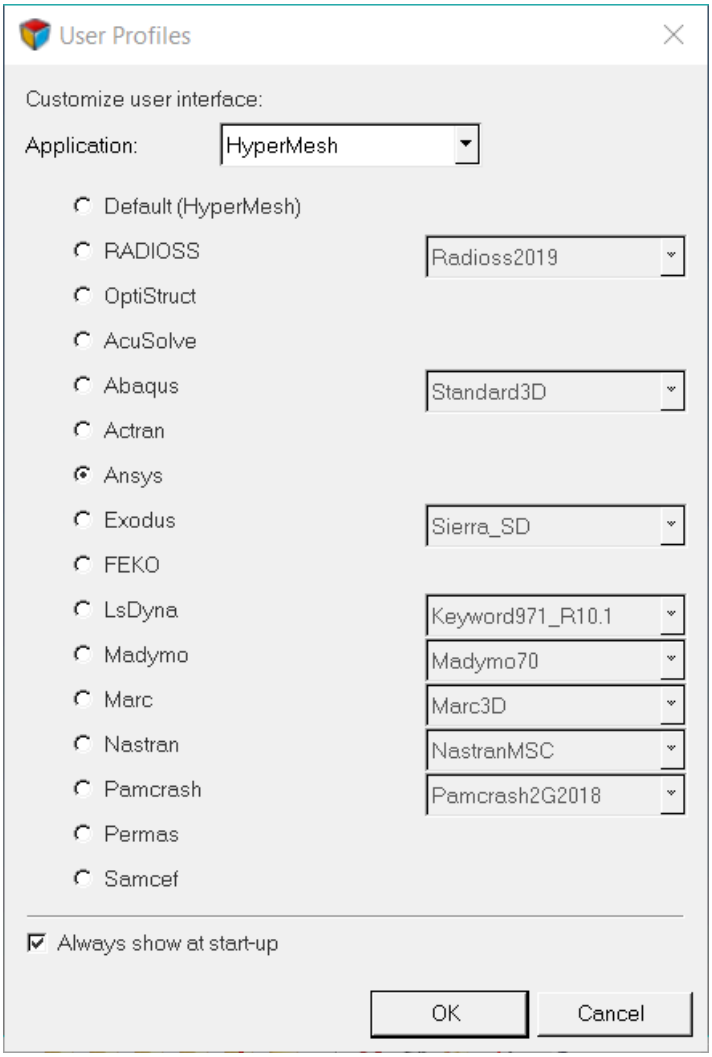


Figure 759:

4. Click **OK**.

HyperMesh sets the defaults for ANSYS.

3.1.3 Import a Model

In this step, you will import an ANSYS deck written by the ANSYS Solver.

HyperMesh can import ANSYS model data in script format. The same procedure can be followed to read a HyperMesh-written deck.

1. From the menu bar, click **File** > **Import** > **Solver Deck**.
2. Verify that (Import Solver Deck) is selected.
3. Set File Type to **Ansys**.

> **Note:** If you are in the ANSYS user profile, HyperMesh automatically sets File type to Ansys.

4. In the File field, click .
5. In the **Select Ansys file** dialog, open the `tuning_fork_ansys.cdb` file.
6. Click **Import**.

The model appears in the graphics area.

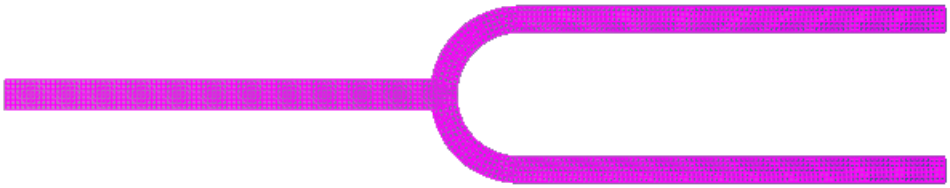


Figure 760:

3.1.4 Export the Model

In this step, you will export a ANSYS model from HyperMesh.

HyperMesh can export ANSYS model data in script format.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
2. Verify that is selected.
3. Set File type to **Ansys**.

> **Note:** If you are in the ANSYS user profile, HyperMesh automatically sets the File type to Ansys.

4. In the File field, click .
5. In the **Select Ansys file** dialog, navigate to the location you would like to export your file to and then save the file as `4400_export`.
6. Click **Export**.

3.2 HM-4410: Set Up a Model in ANSYS

In this tutorial you will learn how to create and define elements, real constants, materials, properties, components, and loadsteps.

Before you begin, copy the `chapter2_1.hm` and `chapter2_2.hm` files from `<hm.zip>/interfaces/ansys/` to your working directory.

3.2.1 Exercise 1: Define Elements, Real Constants, Materials, Properties, and Components

In this exercise you will learn about the ANSYS FE input reader, template, macro menu, and user profile which set the foundation for using ANSYS with HyperMesh.

Load the ANSYS User Profile

In this step, you will load the ANSYS user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, select **Ansys**.

Retrieve the Model File

In this step, you will retrieve and open the model file in HyperMesh.

1. Open a model file by completing one of the following options:
   - From the menu bar click **File** > **Open** > **Model**.
   - On the Standard toolbar click  .
2. In the **Open Model** dialog, open the `chapter2_1.hm` file.
3. Optional: If your model's elements and mesh lines are not shaded, click  on the Visualization toolbar.

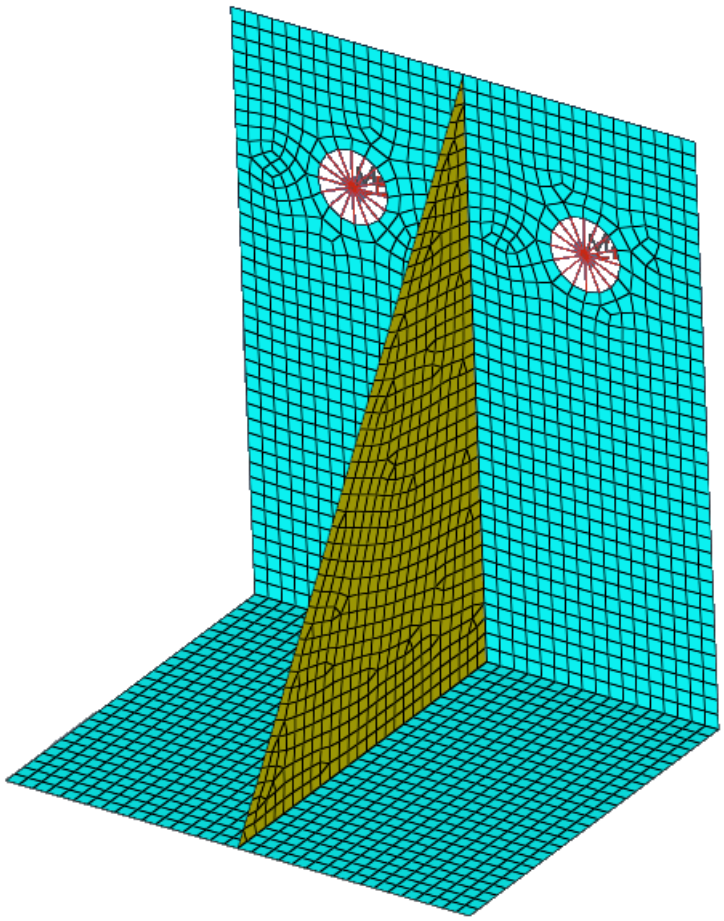


Figure 761:

Add an Element Type

In this step, you will add an element type in HyperMesh.

1. In the Model Browser, right-click and select **Create** > **Sensor** from the context menu. HyperMesh creates and opens a sensor (Et Type) in the Entity Editor.

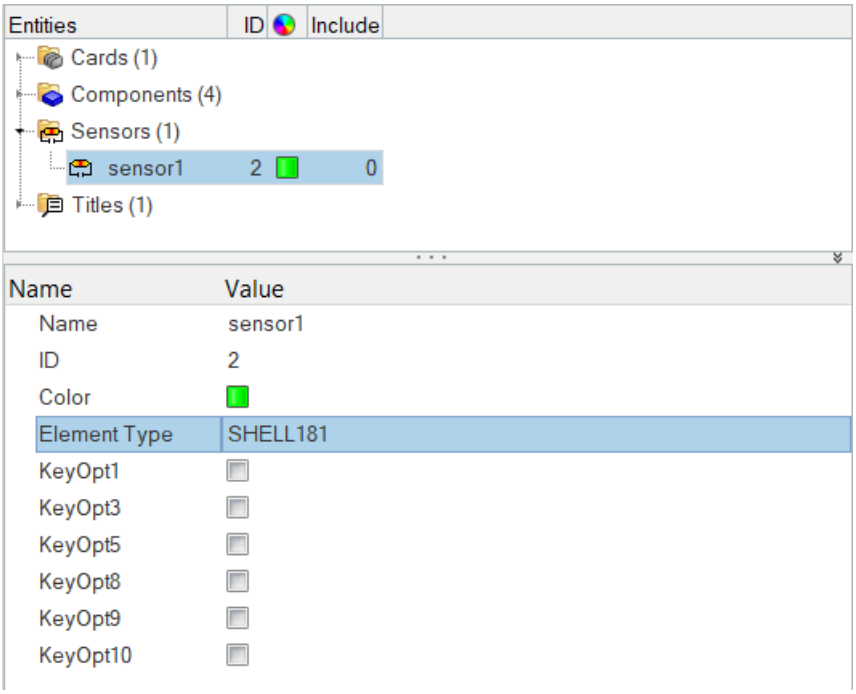


Figure 762:

2. For Name, enter a new name for the Et Type.
3. Optional: For ID, enter a new ID for the Et Type.

> **Note:** By default, HyperMesh sets the ID to 1. If you create a new Et Type, HyperMesh will set the ID to n+1.

4. For Element Type, keep **SHELL181**.

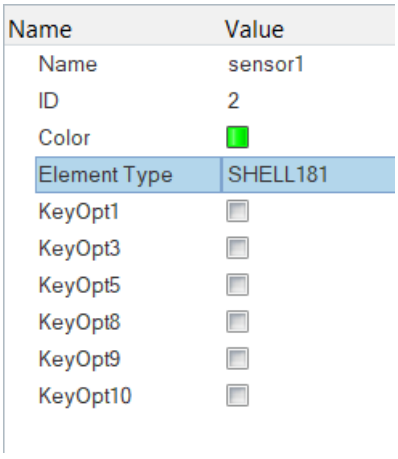


Figure 763:

SHELL181
: Suitable for analyzing thin to moderately-thick shell structures. It is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z direction; rotations

about the x, y, and z axes (if the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.
Well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for nonlinear analysis. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.
May be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory.

Note: By default, HyperMesh set the Element Type to SHELL181. The elements in this model are of type SHELL181, therefore you do not need to change the element type for this tutorial.

5. Set the element stiffness (KeyOpt1), integration (KeyOpt3), layer data storage (KeyOpt8), thickness (KeyOpt9), and/or initial stress (KeyOpt10) options by selecting their corresponding checkboxes in the Value column as indicated in Figure 764.

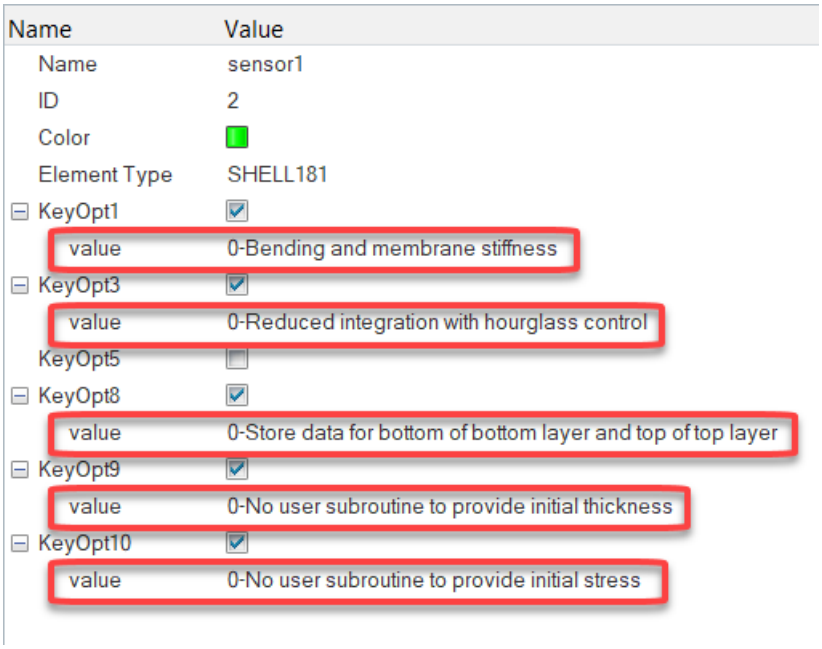


Figure 764:

A value appears below each KeyOpt you selected.

6. For each KeyOpt you selected, keep the default value.
7. Optional: If necessary, from the menu bar, click **View** > **Browsers** > **HyperMesh** > **Solver** to open the Solver Browser.
8. In the Solver Browser, review the ET Type you created.

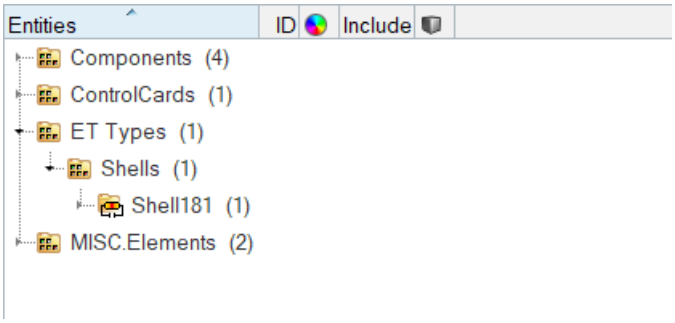


Figure 765:

Define Material Properties

In this step, you will define the properties of a material.

1. In the Model Browser, right-click and select **Create** > **Material** from the context menu. HyperMesh creates and opens a material in the Entity Editor.

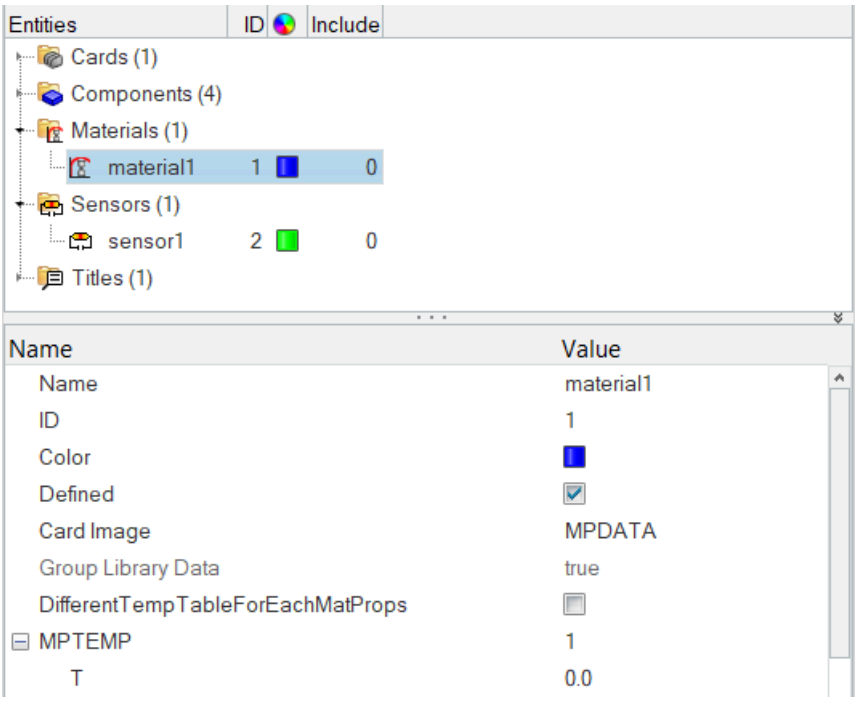


Figure 766:

2. For Name, enter `Steel`.
3. Optional: For ID, enter a new ID.

> **Note:** By default, HyperMesh sets the ID to 1. If you create a new material, HyperMesh will set the ID to n+1.

4. Click the **Color** icon, and select a color.
5. Set card image to **MATERIAL**.

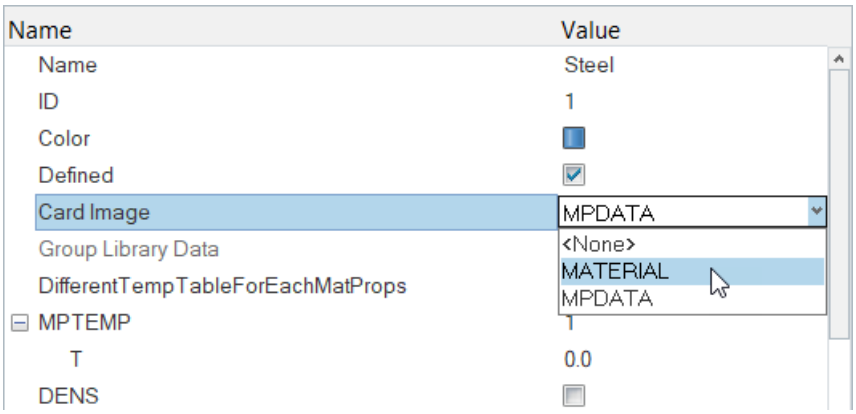


Figure 767:

6. Select the **EX** (Elastic moduli) checkbox.
7. For MP_EX_LEN (Number of Elastic moduli to input), enter `1`.

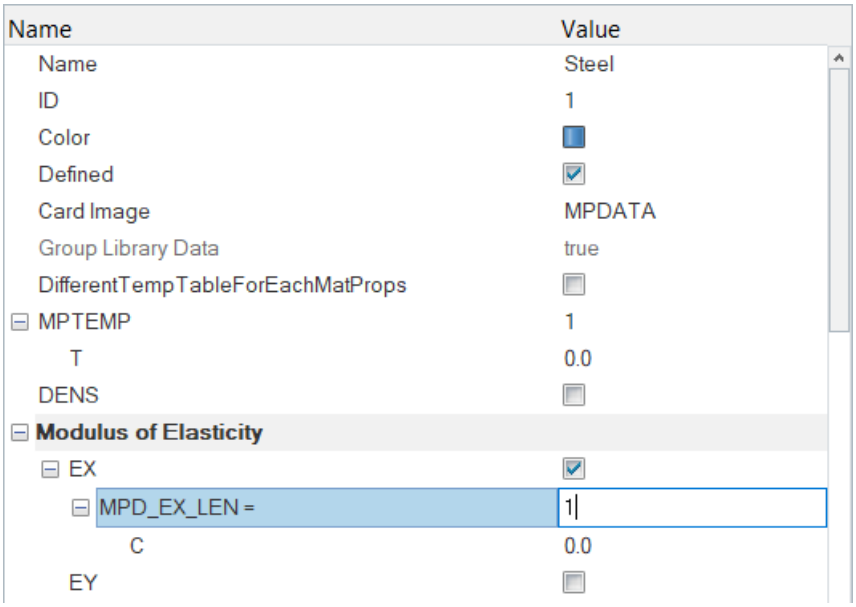


Figure 768:

8. For C, enter `2.1e5`.

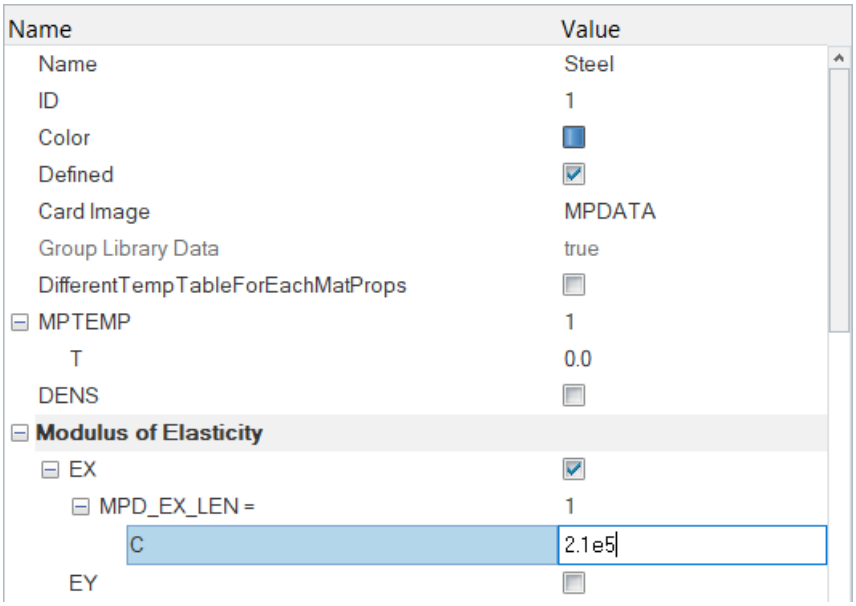


Figure 769:

9. Select the **NUXY** (Minor Poisson's ratio) checkbox.
10. For MP_NUXY_LEN (Number of Minor Poisson's ratio to input), enter `1`.
11. For C, enter `0.3`.
12. In the **MP_NUXY_LEN=** dialog, enter `0.3`.
13. Go to the Solver Browser and review the material you created.

Create the Section Card for the Shell Elements in the Model

In this step, you will create a section card for the model shell elements.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu. HyperMesh creates and opens a property in the Entity Editor.

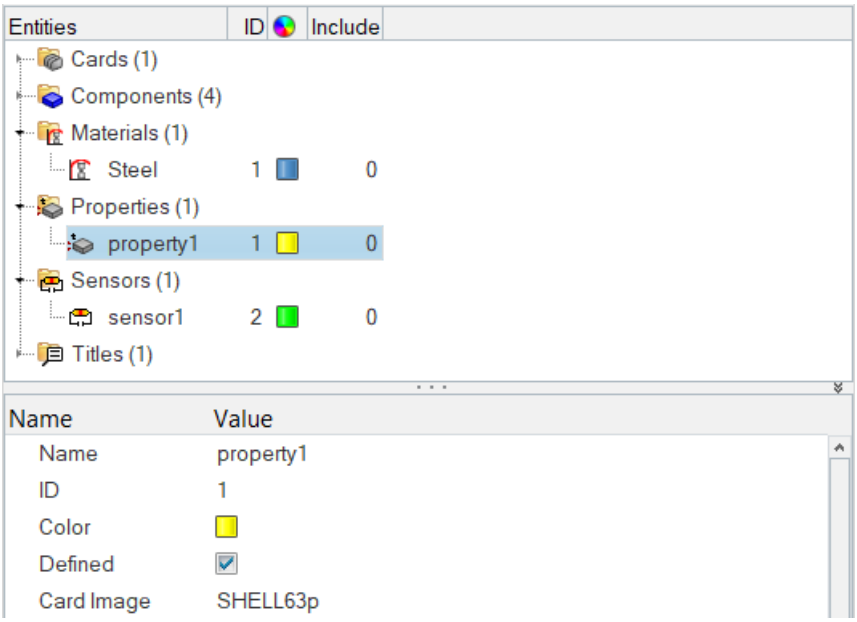


Figure 770:

2. For Name, enter `SECT1`.
3. Optional: For ID, enter a new ID.

   Note: By default, HyperMesh sets the ID to 1. If you create another new beam section collector, HyperMesh will set the ID to n+1.

   Ansys sections are supported under property, with a separate ID pool. Sections are created by setting the card image to SECTYPE. All cards created with the SECTYPE card image are organized under one ID pool. Properties created with a card image other than SECTYPE are organized under another ID pool. If you want to use the same IDs in these two pools, enable the allow duplicate IDs option from the menu bar by selecting **Preferences** > **Meshing Options**. In earlier versions of HyperMesh (14.0 or before), Ansys sections were supported under beam section collectors.
4. Click the **Color** icon, and select a new color.
5. Set card image to **SECTYPE**.
6. Set TYPE to **SHELL**.
7. Under SECDATA, for PLIES, enter `1`.

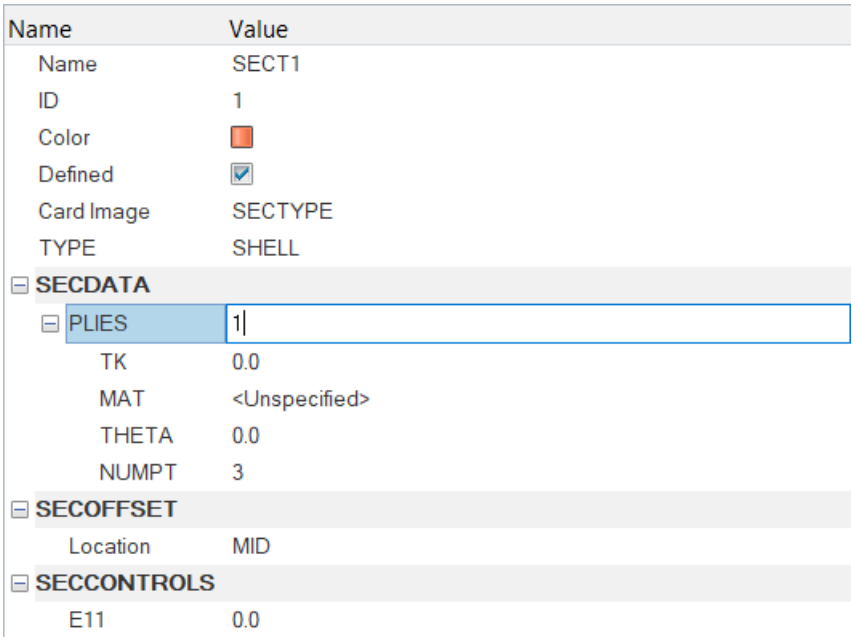


Figure 771:

8. For TK (ply thickness), enter `10`.
9. For MAT, click **Unspecified** > **Material**.

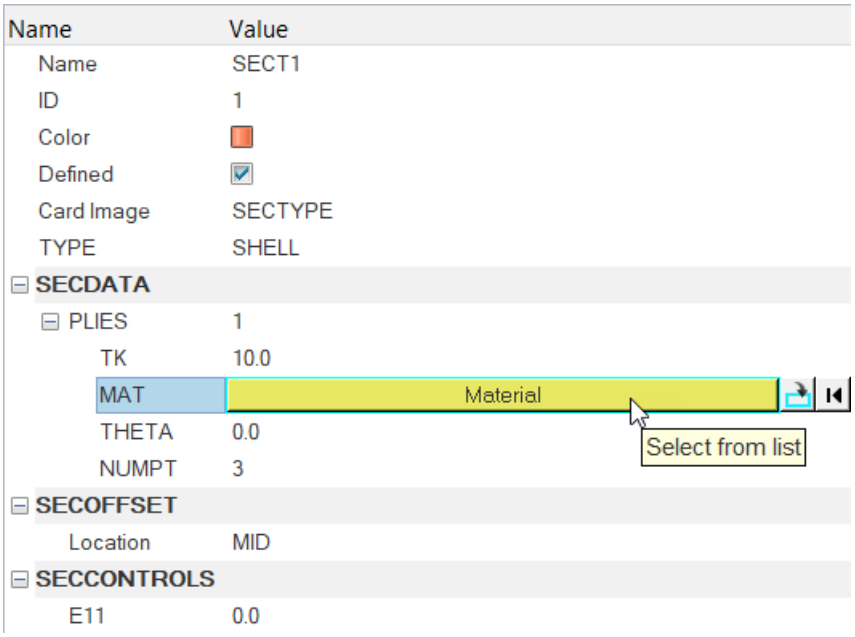


Figure 772:

10. In the **Select Material** dialog, select **Steel**, and then click **OK**.

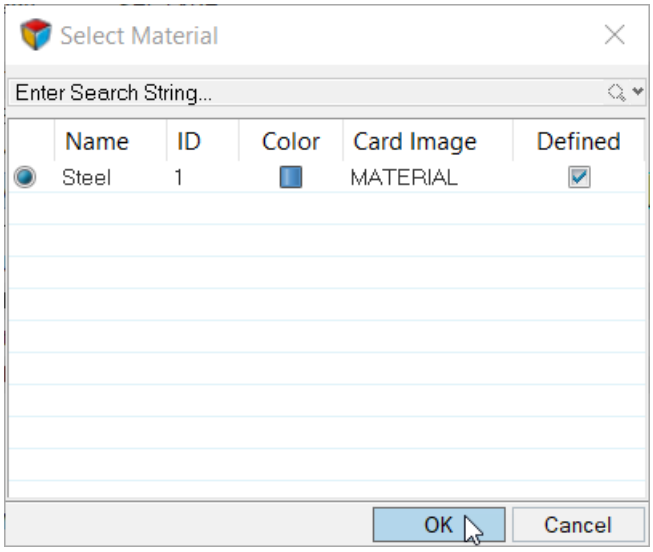


Figure 773:

11. For THETA (ply angle), keep the default value **0.0**.

12. For NUMPT (Integration points through ply thickness), enter `3.0`.

13. In the Property folder of the Model Browser, right-click on **SECT1** and select **Duplicate** from the context menu.
HyperMesh creates a new property using the same card data as SECT1, except a different Name and ID are specified.

14. For Name, enter `SECT2`.

15. Click the **Color** icon, and select a new color.

16. Change the TK value from 10 to `5`.

17. Leave MAT, THETA, and NUMPT unchanged.

18. Go to the Solver Browser and review the two sections you created.

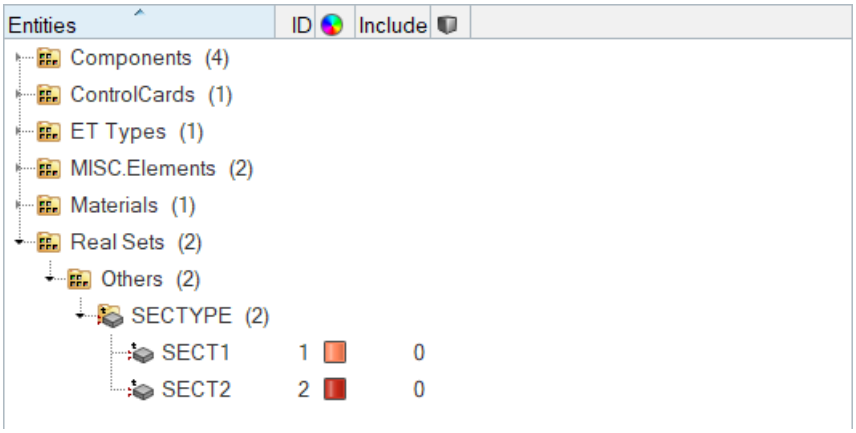


Figure 774:

Update Each Component

In this step, you will update each component with the respective element type, property, material, and section information.

1. In the Component folder of the Model Browser, click **Base**.
   The Entity Editor opens and displays the component's corresponding data.

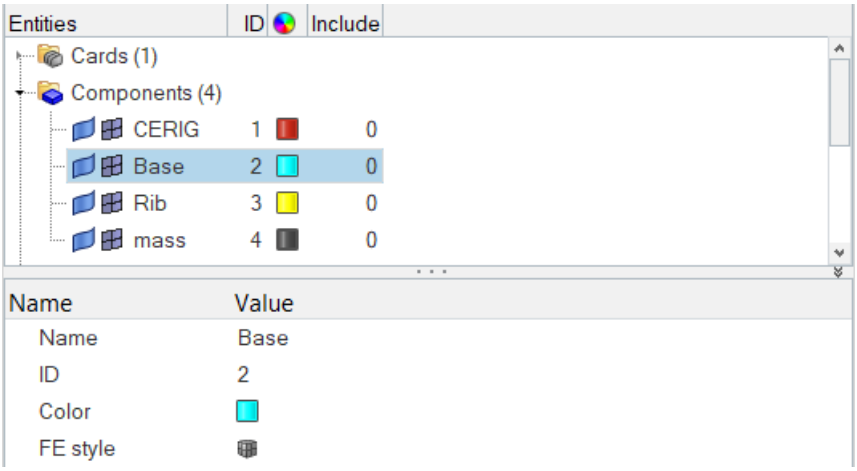


Figure 775:

2. For Type, click **Unspecified** > **Sensor**.

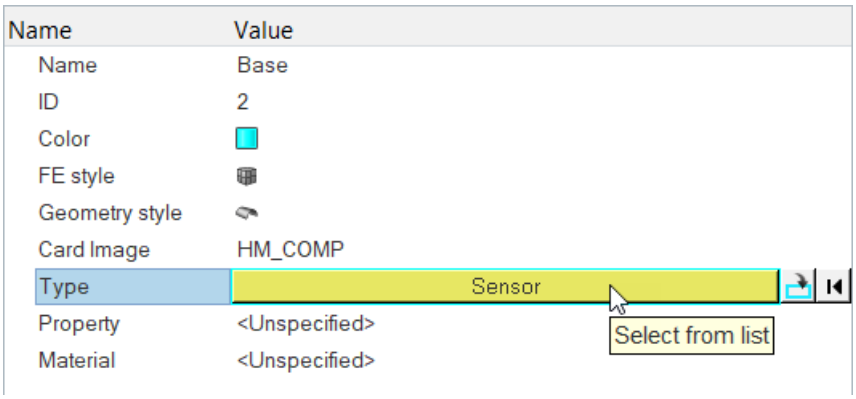


Figure 776:

3. In the **Selector Sensor** dialog, select **sensor1** (SHELL181) and then click **OK**.

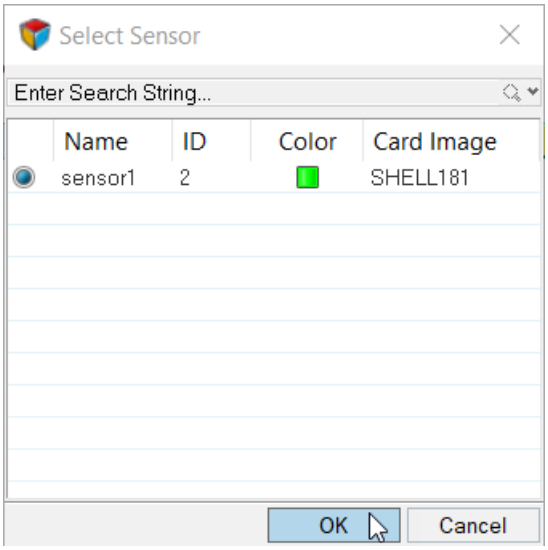


Figure 777:

4. For Property, click **Unspecified** > **Property**.
5. In the **Select Property**dialog, select **SECT1** and then click **OK**.

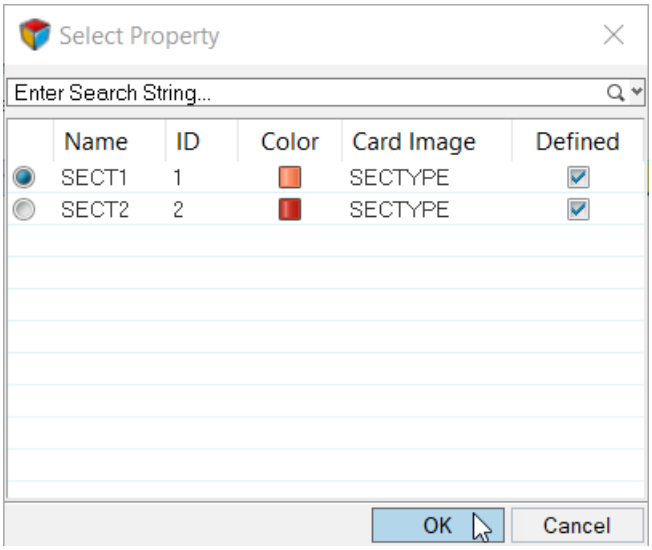


Figure 778:

> **Note:** You do not have to assign a Property or Material to this component, because this information is already defined in SECT1.

6. In the Component folder of the Model Browser, click **Rib**.
   The Entity Editor opens and displays the component's corresponding data.
7. For Type, click **Unspecified** > **Sensor**.
8. In the **Selector Sensor** dialog, select **sensor1** (SHELL181) and then click **OK**.
9. For Property, click **Unspecified** > **Property**.

10. In the **Select Property** dialog, select **SECT2** and then click **OK**.

> **Note:** You do not have to assign a Property or Material to this component, because this information is already defined in SECT2.

11. In the Component folder of the Model Browser, click **mass**.

The Entity Editor opens and displays the component's corresponding data.

> **Note:** The mass component does not currently have a type, property, or material attached to it.

12. Create and attach the **MASS21** element type to the mass component.

a) Right-click on **Type** and select **Create** from thecontext menu.

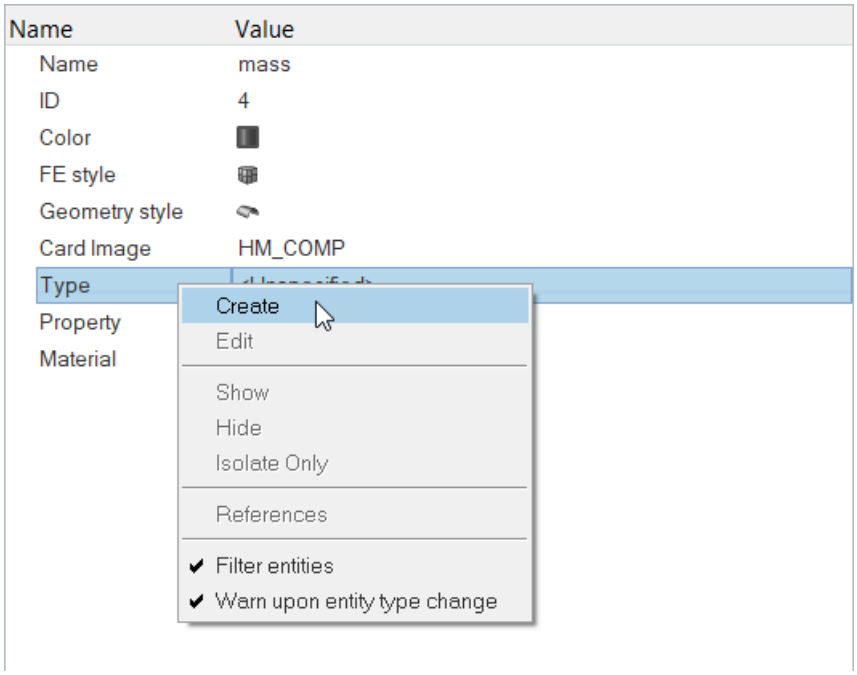


Figure 779:

The **Create Sensors** dialog opens.

b) Set Element Type to **MASS21**.

MASS21

A point element that can have up to six degrees of freedom: translations in the nodal x, y, and z directions; rotations about the nodal x, y, and z axes. A different mass and rotary inertia may be assigned to each coordinate direction.
Defined by a single node, concentrate mass components (Force*Time²/Length) in the element coordinate directions and rotary inertias (Force*Length* Time²) about the element coordinate axes. The element coordinate system may be initially parallel to the global Cartesian coordinate system or to the nodal coordinate system (KEYOPT(2)). The coordinate system rotates with the nodal coordinate rotations during a large deflection analysis. Options are available to exclude the rotary inertia effects and to

reduce the element to a 2-D capability (KEYOPT(3)). If the element requires only the mass input, it is assumed to act in all appropriate coordinate directions.

c) Set KeyOpt3 to **3-D mass without rotary inertia**.

d) Click **Close**.

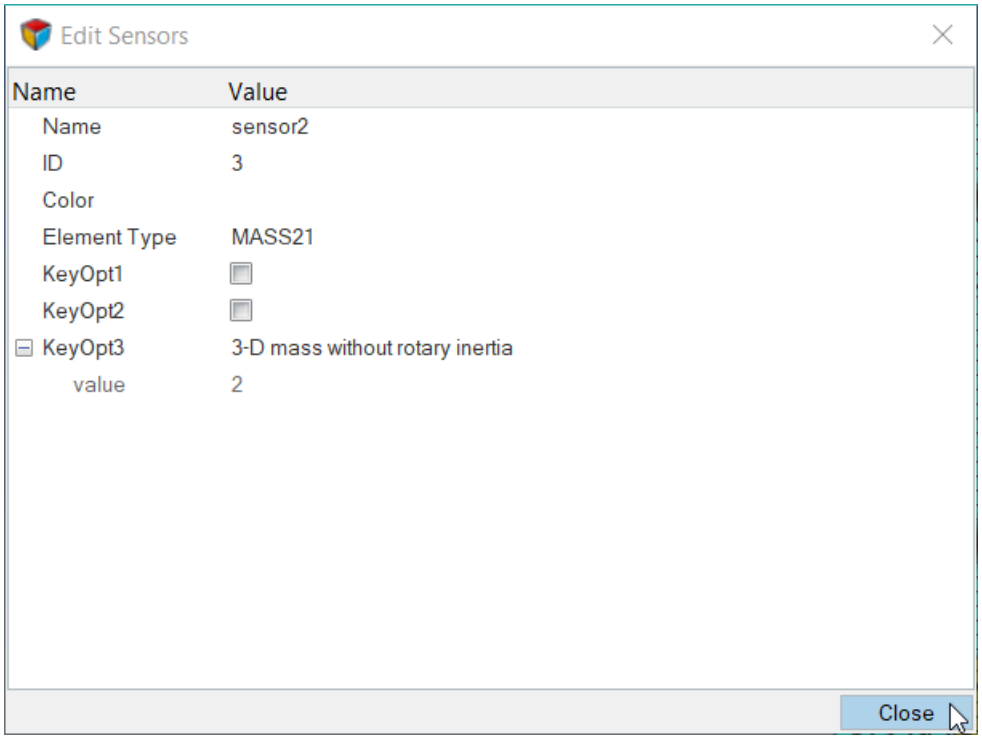


Figure 780:

HyperMesh creates and attaches the new sensor to the mass component.

13. Create a property card that associates a small mass to the mass elements.

a) Right-click on **Property** and select **Create** from the context menu.

In the **Create Properties** dialog, the card image is automatically set to MASS21p because the element type attached to the mass component is MASS21.

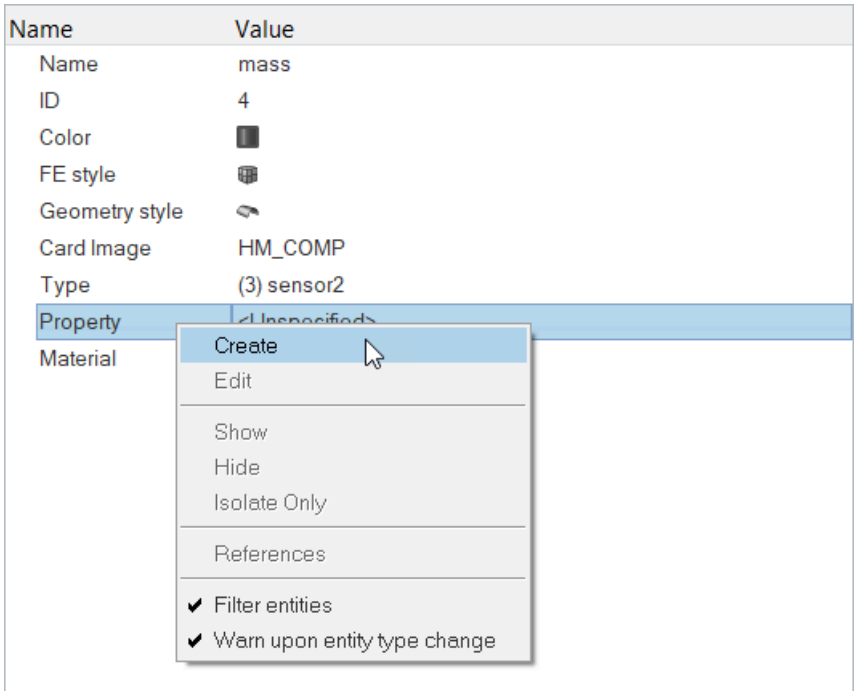


Figure 781:

b) Set KeyOpt3 to **3-D mass without rotary inertia**.
c) Under Real Constants, enter `0.001` for MASS.
d) Click **Close**.

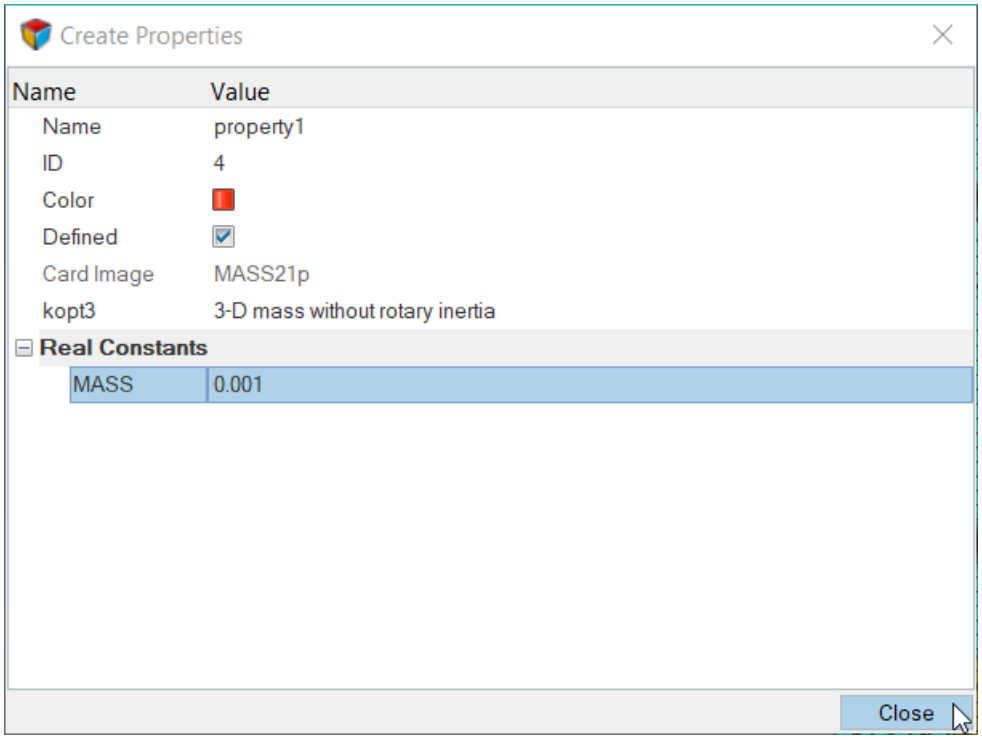


Figure 782:

HyperMesh creates and attaches the new property to the mass component.

14. For Material, click **Unspecified** > **Material**.

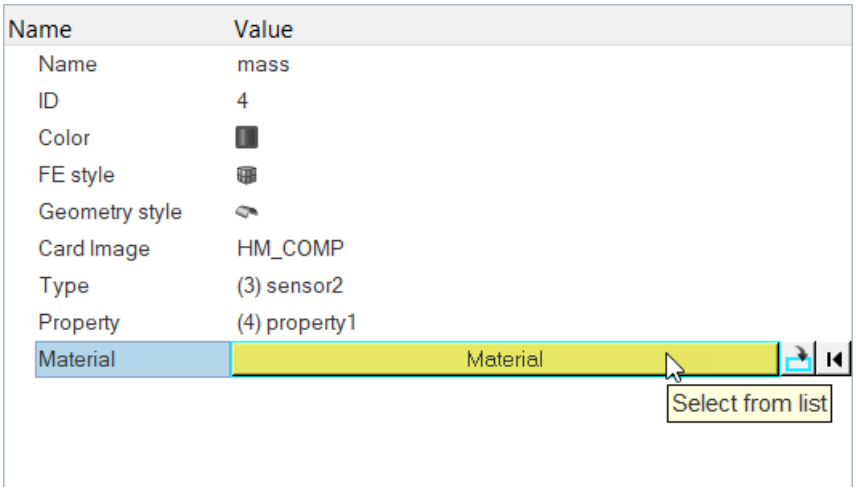


Figure 783:

15. In the **Select Material** dialog, select **Steel** and then click **OK**.

The CERIG component contains ANSYS rigid elements. These elements define the rigid region and do not require an element type, property, or material. Therefore you do not have to assign a card to the CERIG component.

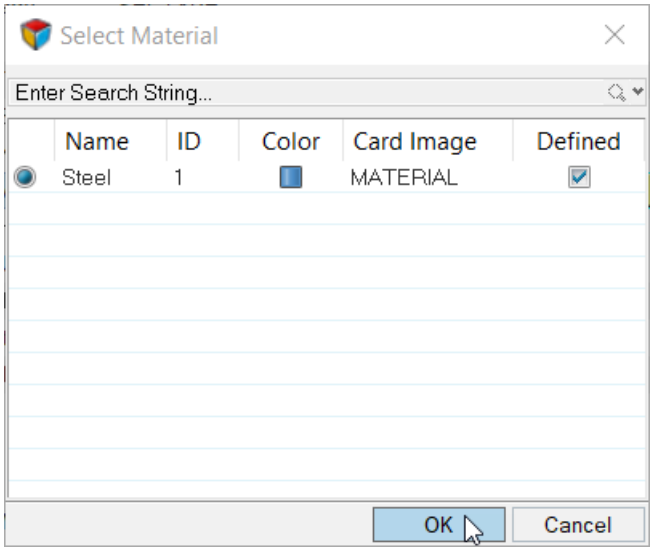


Figure 784:

Save Your Model

In this step, you will save the model to your working directory.

1. From the menu bar, click **File** > **Save As** > **Model**.
2. In the **Save Model As** dialog, navigate to your working directory and save the file.

3.2.2 Exercise 2: Introduction to ANSYS Load Steps

In this exercise you will: create constraint load collectors; apply the constraints to the model; apply the force on mass elements with force1, force2, and force3 load collectors; create multiple load steps; add / SOLU and LSSOLVE in control cards; and export the deck to ANSYS `*cdb` format.

This exercise introduces the concept of ANSYS load steps in HyperMesh. In HyperMesh, you need to have each load or constraints in a separate load collector (load cols). With the help of these load collectors, you can create multiple load steps depending on the requirement. The combination of loads with constraints, form a load step. If you have created load steps in your model, the exported `*.cdb` file will have all of the load step information. This `*.cdb` file when imported into ANSYS, automatically creates the `*.so` files in the working directory which can be used later if needed.

Load the ANSYS User Profile

In this step, you will load the ANSYS user profile in HyperMesh.

Only complete this step if you did not complete Exercise 1: Define Elements, Real Constants, Materials, Properties, and Components.

1. Start HyperMesh.

2. In the **User Profile** dialog, set the user profile to **Ansys**.

Retrieve the HyperMesh Model File

In this step, you will retrieve and open the model file.

Only complete this step if you did not complete Exercise 1: Define Elements, Real Constants, Materials, Properties, and Components.

1. From the menu bar, click **File** > **Open** > **Model**.
2. In the **Open Model** dialog, open the `chapter2_2.hm` file.
3. Optional: If you model's elements and mesh lines are not shaded, click  on the Visualization toolbar.

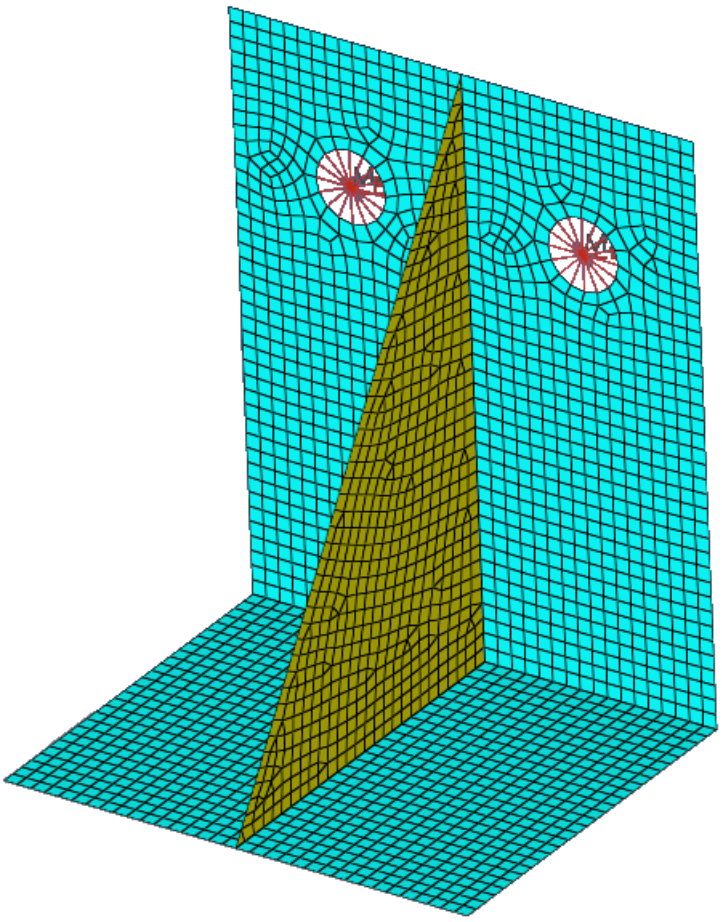


Figure 785:

Create a Constraints Load Collector

In this step you will learn how to create load collectors within HyperMesh.

1. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.
   HyperMesh creates and opens a load collector in the Entity Editor.
2. For Name, enter `constraints`.
3. Click the **Color** icon, and select a new color for the load collector.
4. Repeat steps 1 through 3 to create three more load collectors named `force1`, `force2`, and `force3`.

Four load collectors display in the Model Browser as indicated in Figure 786.

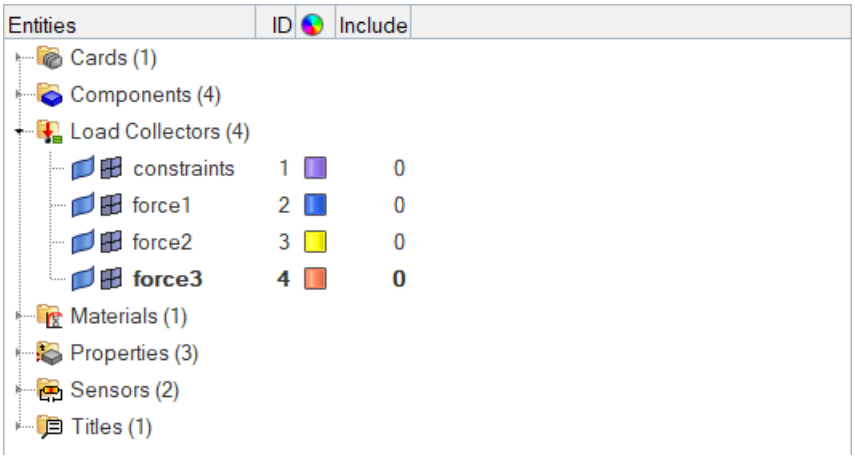


Figure 786:

Apply the Constraints to the Model

In this step, you will apply the constraints you created to the model.

1. In the Load Collectors folder of the Model Browser, right-click on **constraints** and select **Make Current** from the context menu.

   > **Note:** When new loads are created, HyperMesh will place them in this collector.

2. From the menu bar, click **BCs** > **Create** > **Constraints**.
   The Constraints panel opens.
3. Select the **dof** (degree of freedom) checkboxes as indicated in Figure 787.



Figure 787:

4. Click **nodes** > **by path**.

5. Select a start node and an end node on the left side of the model as indicated in Figure 788.

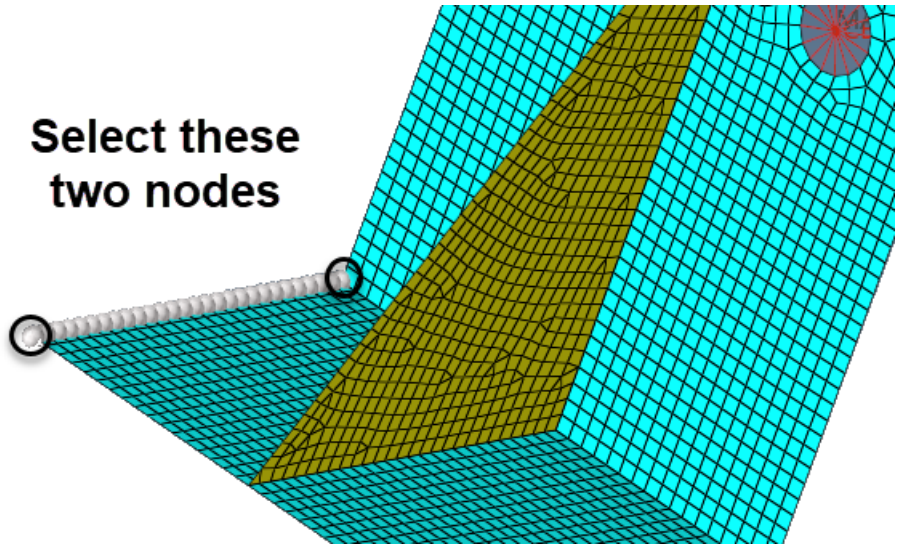


Figure 788:

6. Click **Create**.

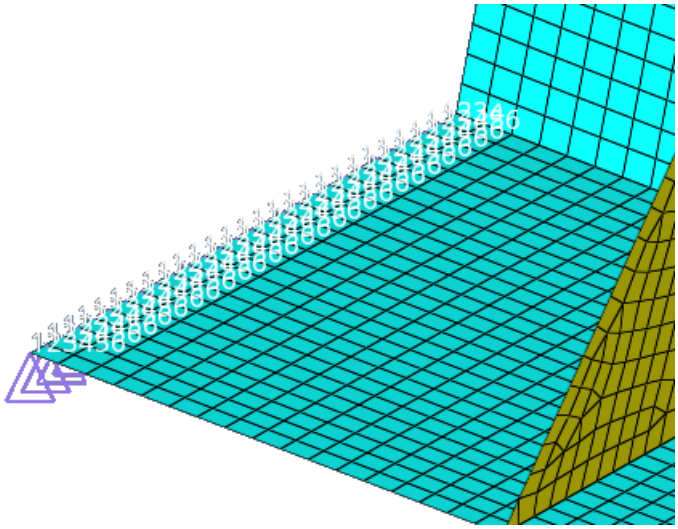


Figure 789:

7. Repeat steps 4 and 5 to select a start node and an end node on the right side of the model as indicated in Figure 790.

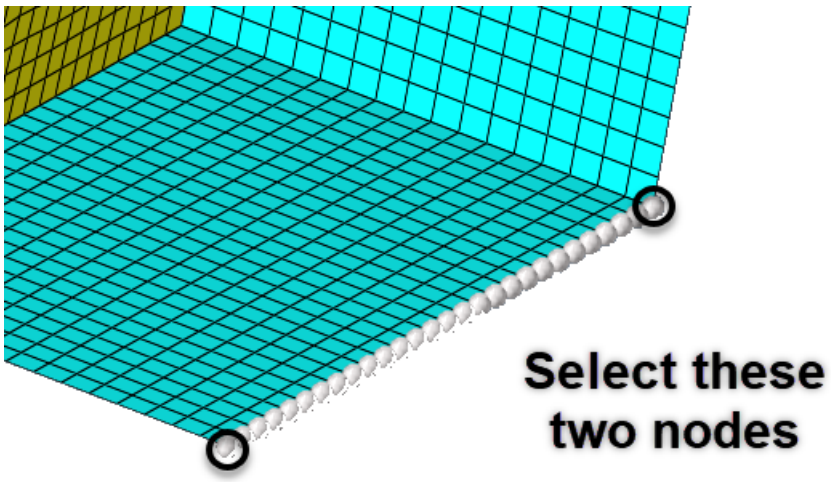


Figure 790:

8. Click **Create**.

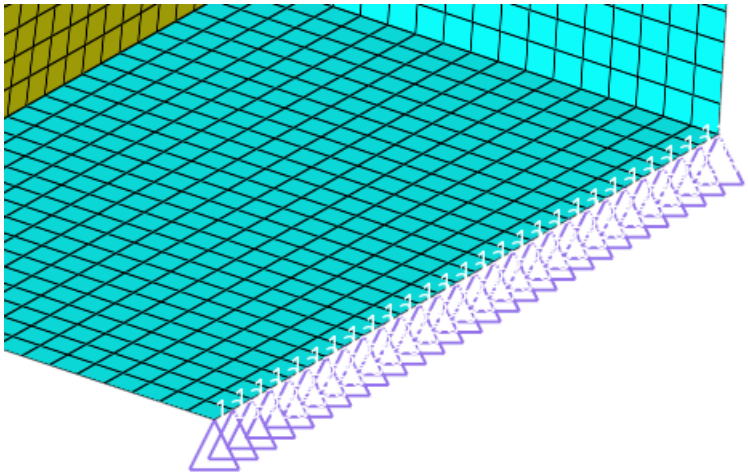


Figure 791:

9. Click **return** to exit the Constraints panel.

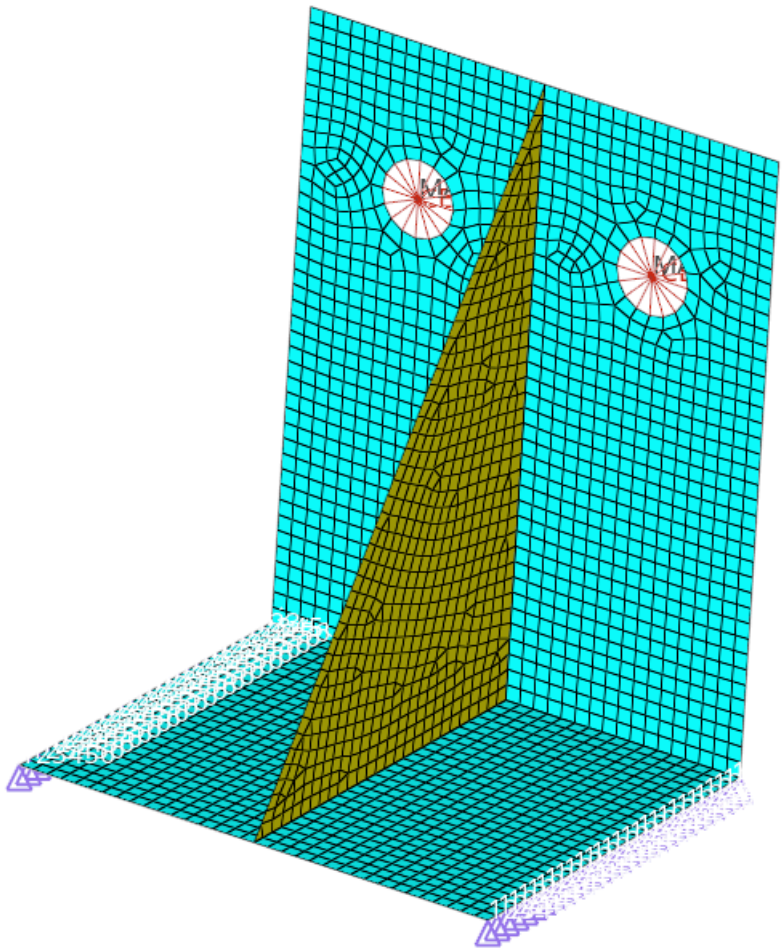


Figure 792:

Apply the Force on Mass Elements with the Force1 Load Collector

In this step, you will apply the force on mass elements with the force1 load collector.

1. In the Load Collector folder of the Model Browser, right-click on **force1** and select **Make Current** from the context menu.
2. From the menu bar, click **BCs** > **Create** > **Forces**.
   The Forces panel opens.
3. Verify the entity selector is set to **nodes**.
4. Select the two nodes in the center of the two bolt holes as indicated in Figure 793.

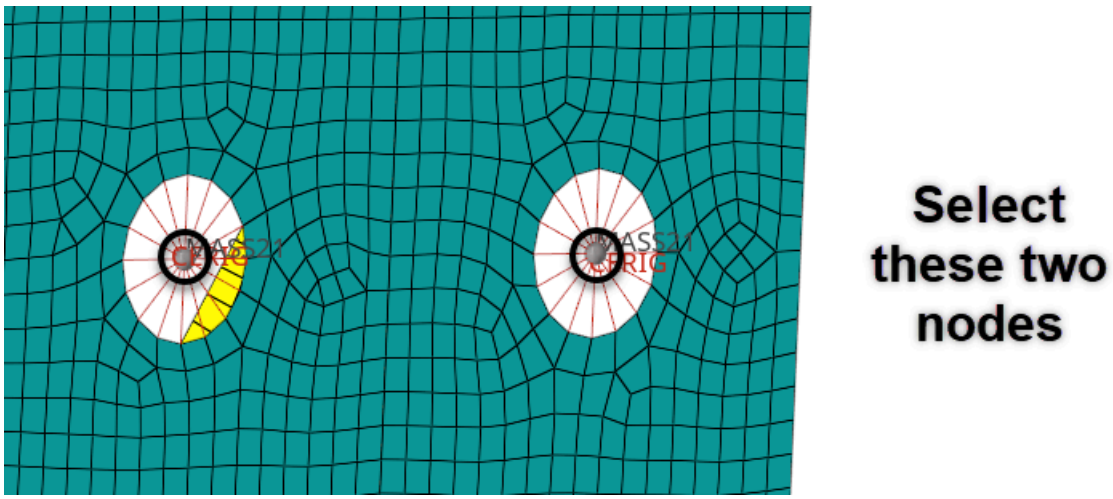


Figure 793:

5. In the magnitude= field, enter `500`.
6. Set the orientation selector to **z-axis** for the direction of application of the force.
7. In the uniform size= field, enter `20`.
8. Click **Create**.

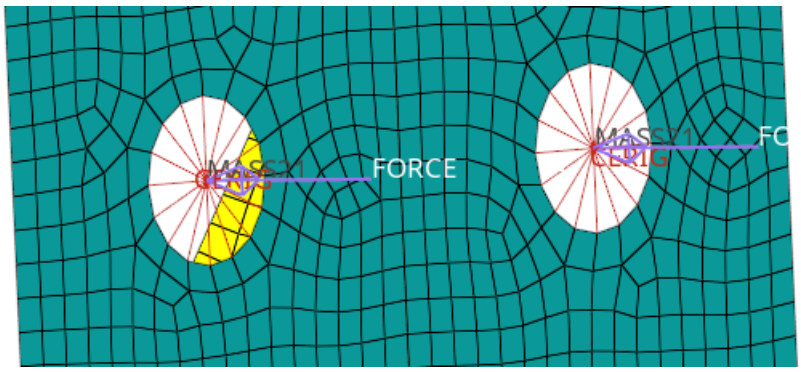


Figure 794:

9. Click **return** to exit the Forces panel.

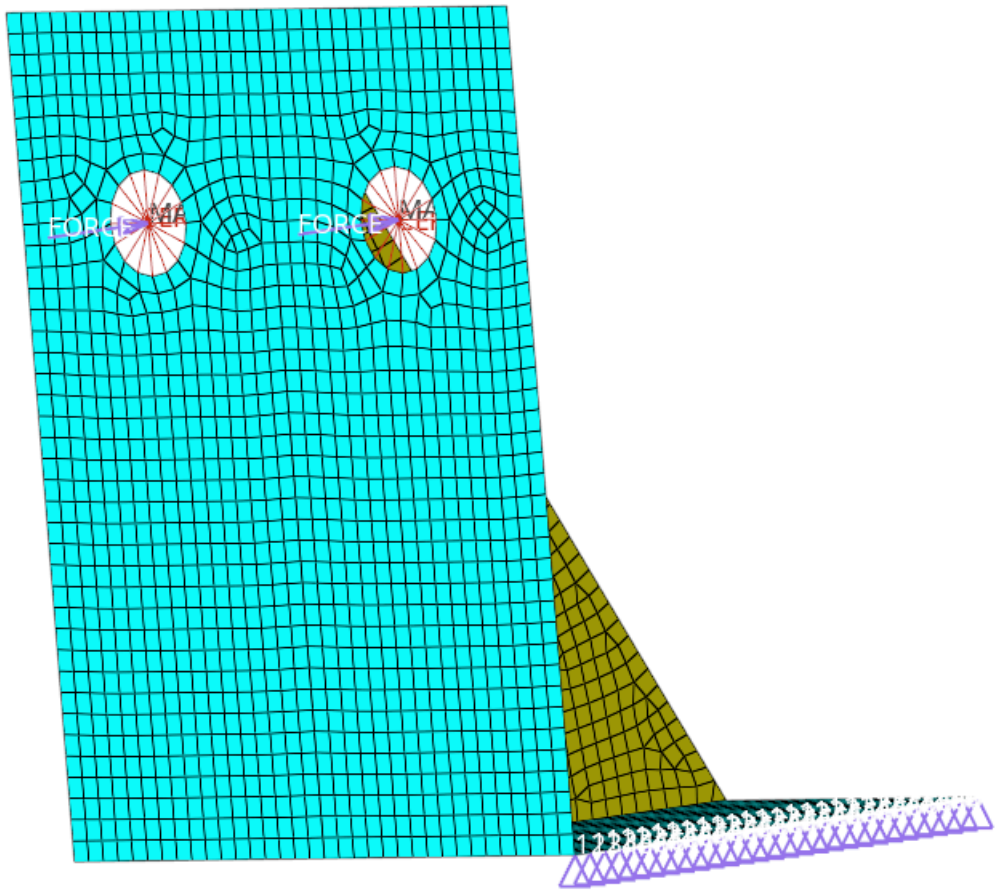


Figure 795:

Apply the Force on Mass Elements with the Force2 Load Collector

In this step, you will apply the force on mass elements with the force2 load collector.

1. In the Load Collector folder of the Model Browser, right-click on **force2** and select **Make Current** from the context menu.
2. For better visualization, press **F5** to open the Mask panel.
3. Set the entity selector to **loads**.
4. Select the two forces you created in step 8 of Apply the Force on Mass Elements with the Force1 Load Collector.
5. Click **mask**.
6. Click **return**.
7. From the menu bar, click **BCs** > **Create** > **Forces**.
   The Forces panel opens.
8. Verify the entity selector is set to **nodes**.

9. On the Standard Views toolbar, click .

10. Select the left side node in the center of the bolt hole as indicated in Figure 796.

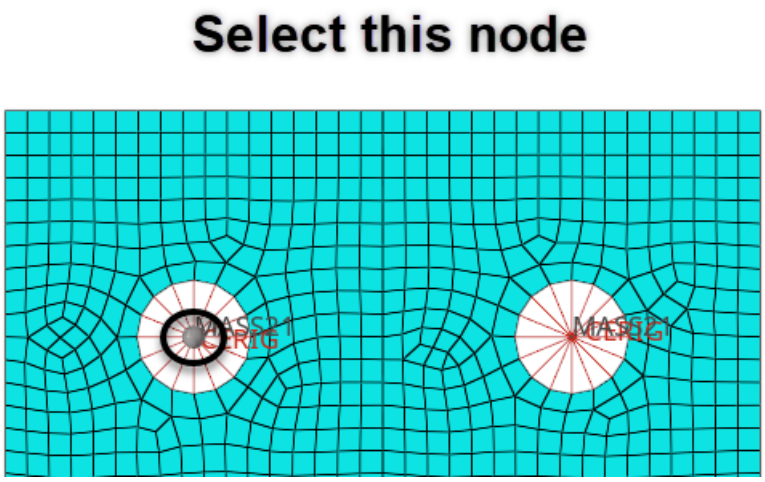


Figure 796:

11. In the magnitude= field, enter `500`.

12. Set the orientation selector to **z-axis** for the direction of application of the force.

13. Click **create**.

14. Select the right side node in the center of the bolt hole as indicated in Figure 797.

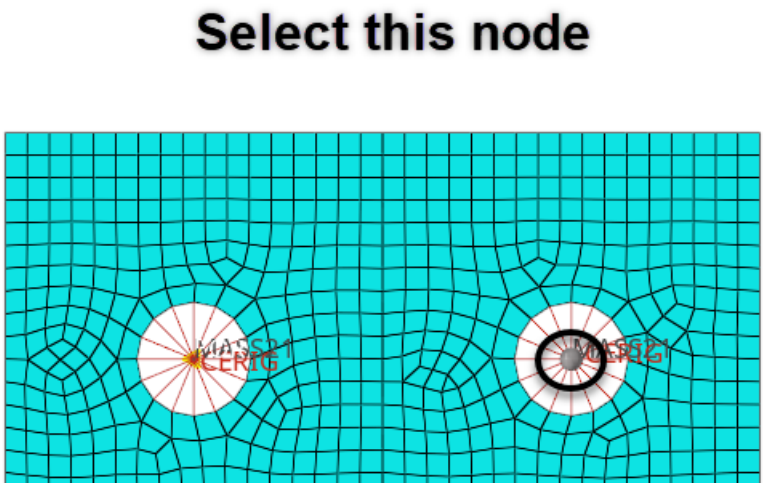


Figure 797:

15. In the magnitude= field, enter `-500`.

16. Set the orientation selector to **z-axis** for the direction of application of the force.

17. Click **create**.

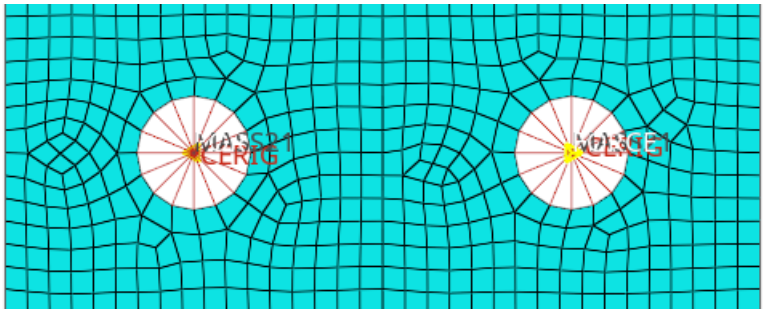


Figure 798:

18. Click **return** to exit the Forces panel.

Apply the Force on Mass Elements with the Force3 Load Collector

In this step, you will apply the force on mass elements with the force3 load collector.

1. In the Load Collector folder of the Model Browser, right-click on **force3** and select **Make Current** from the context menu.
2. Open the Mask panel.
3. Verify the entity selector is set to **loads**.
4. Select the two forces you created in steps 13 and 17 of Apply the Force on Mass Elements with the Force2 Load Collector.
5. Click **mask**.
6. Click **return**.
7. From the menu bar, click **BCs** > **Create** > **Forces**.
   The Forces panel opens.
8. Verify the entity selector is set to **nodes**.
9. Select the two nodes in the center of the two bolt holes as indicated in Figure 799.

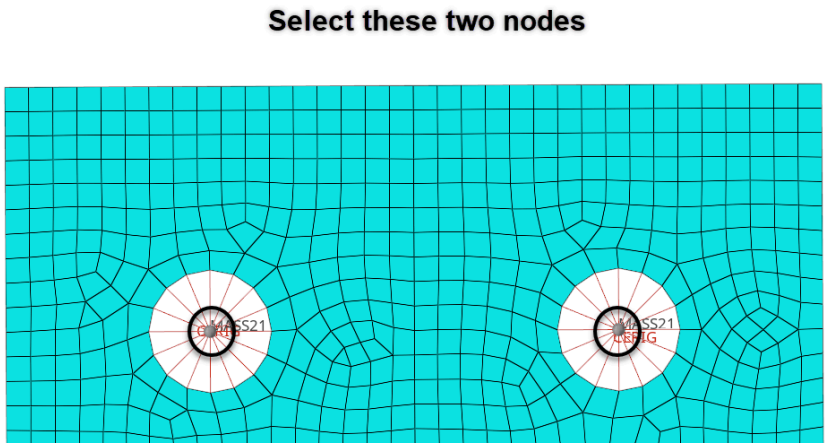


Figure 799:

10. In the magnitude= field, enter `-500`.

11. Set the orientation selector to **z-axis** for the direction of application of the force.

12. Click **create**.

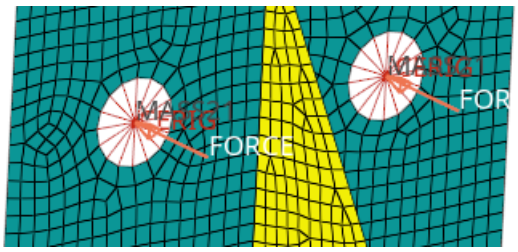


Figure 800:

13. Click **return** to exit the Forces panel.

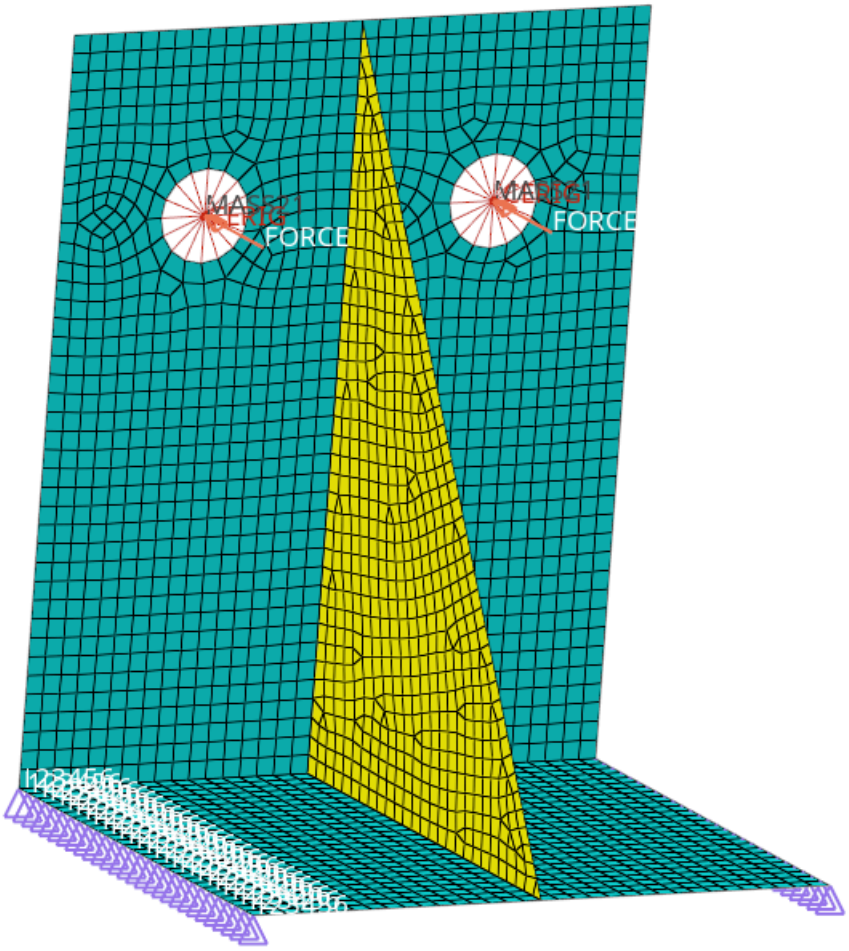


Figure 801:

Create Multiple Load Steps

In this step, you will create multiple load steps.

1. In the Model Browser, right-click and select **Create** > **Load Step** from the context menu.

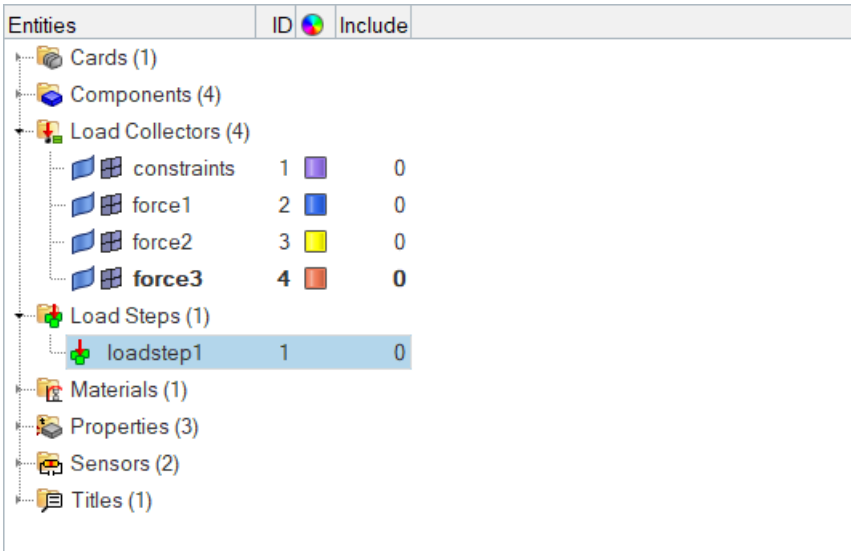


Figure 802:

HyperMesh creates and opens a load step in the Entity Editor.

2. For Name, enter `Step1`.
3. For Loadcol IDs, click **0 Loadcols** > **Loadcols**.

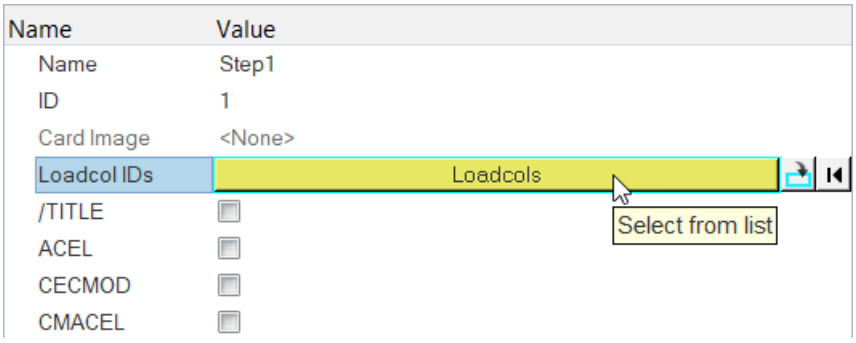


Figure 803:

4. In the **Select Loadcols** dialog, select **constraints** and **force1**.

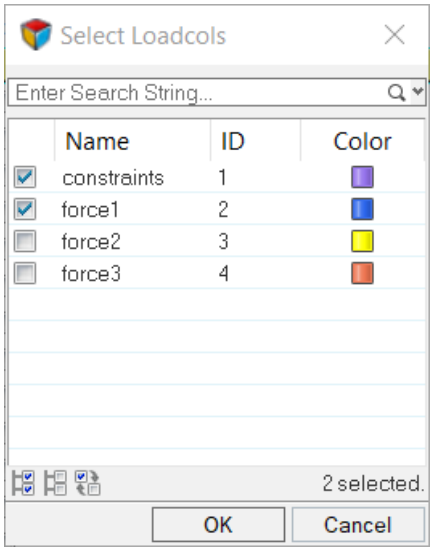


Figure 804:

5. Click **OK**.
6. Repeat steps 1 through 5 to create a second load step named `Step2` with the load collectors **constraints** and **force2**.
7. Repeat steps 1 through 5 to create a third load step named `Step3` with the load collectors **constraints** and **force3**.
8. In the Model Browser, review the Load Collectors and Load Steps you created.

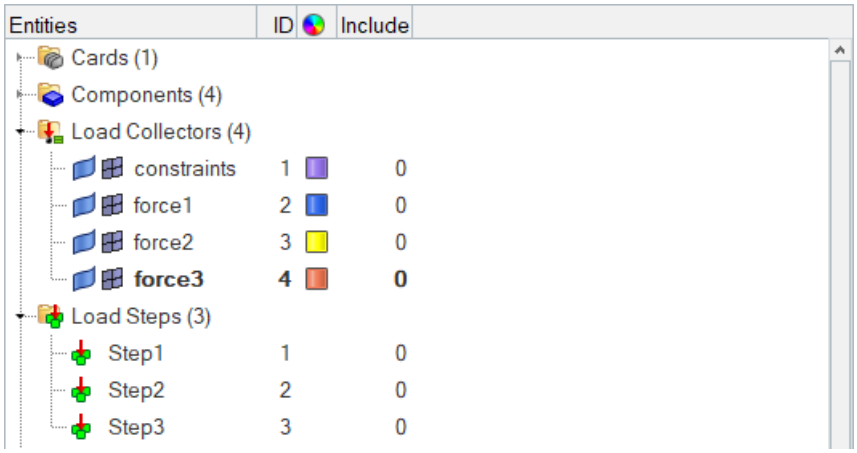


Figure 805:

9. Optional: If necessary, click **View** > **Browsers** > **HyperMesh** > **Solver** from the menu bar to open the Solver Browser.
10. In the Solver Browser, review the Load Collectors and Load Steps you created.

Add /SOLU, ANTYPE, and LSSOLVE in the Control Cards

In this step, you will add the following Control Cards.

1. From the menu bar, click **Setup** > **Create** > **Control Cards**.
   The Control Cards panel opens.
2. In the card image, click **/SOLU** to exit the PREP7 preprocessor and enter the SOLU preprocessor.

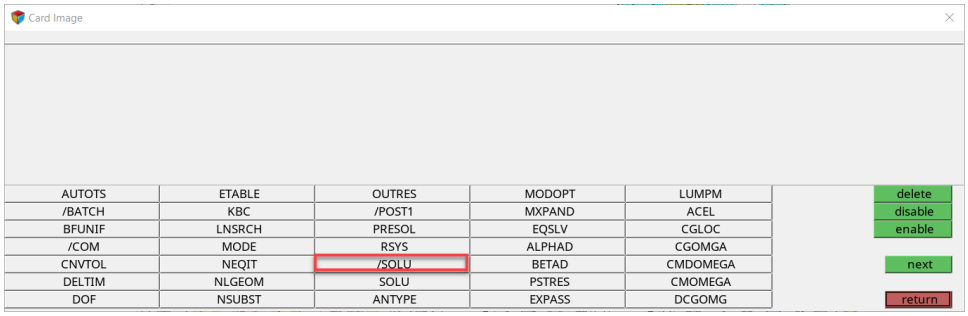


Figure 806:

3. Click **return**.

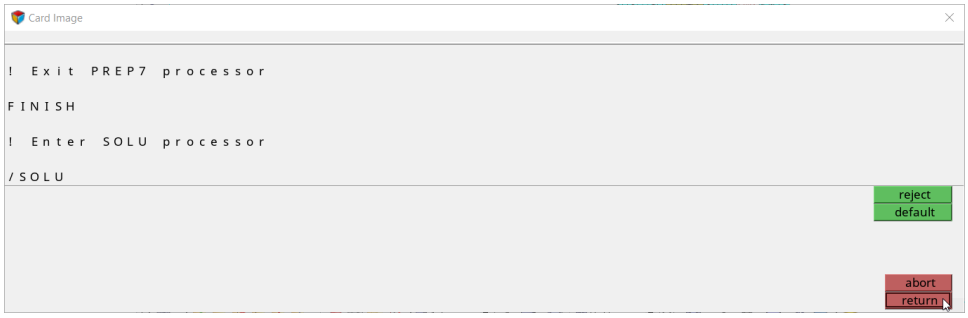


Figure 807:

4. Because you are solving the model for static analysis, click **ANTYPE**.

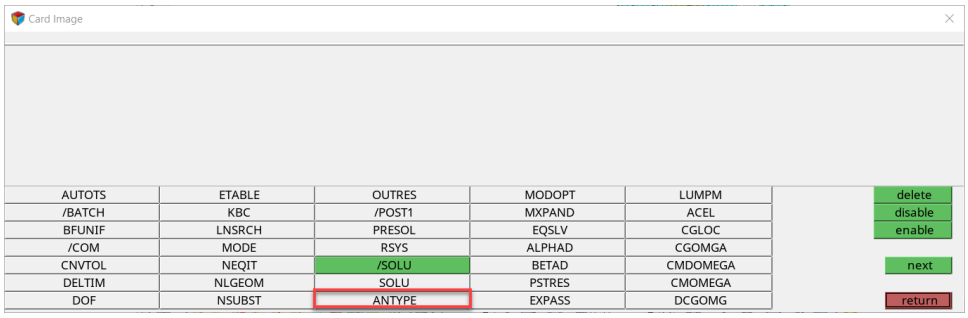


Figure 808:

5. Set type to **STATIC** and status to **NEW**.

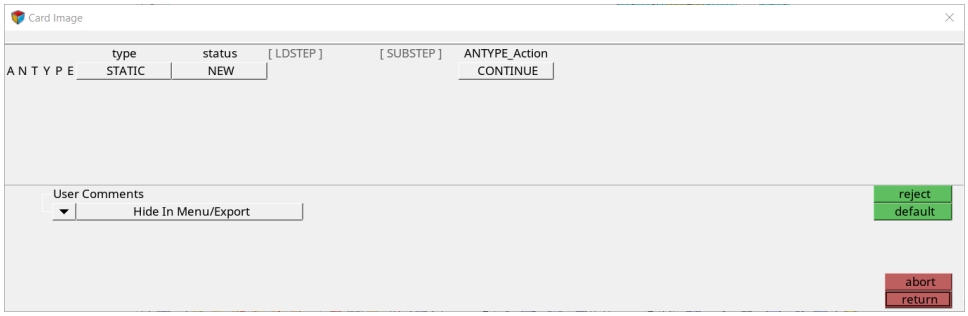


Figure 809:

6. Click **return**.
7. Click **LSSOLVE**.

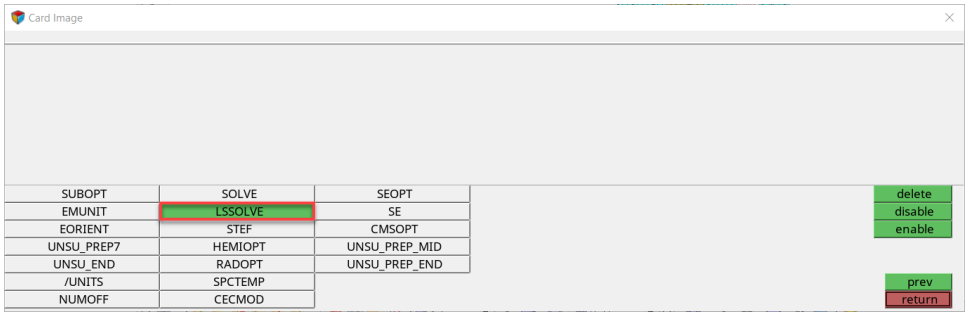


Figure 810:

> **Tip:** If you do not see the LSSOLVE Control Card, click **next**.

8. Enter `1` in the LSMIN field as indicated in Figure 811.
   The minimum number of load steps is set.
9. Enter `3` in the LSMAX field as indicated in Figure 811.
   The maximum number of load steps is set.
10. Enter `1` in the LSINC field as indicated in Figure 811.
    The load step increment is set.



Figure 811:

11. Click **return** to exit the card image.

12. Click **return** to exit the Control Cards panel.

Export the Deck

In this step, you will export the deck to ANSYS *.cdb format.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
   The Export tab opens.
2. Set File type to **Ansys**.

   > **Note:** If you are in the ANSYS user profile, HyperMesh automatically sets the File type to Ansys and loads ANSYS as the default Template.

3. In the File field, navigate to your working directory and save the file as `4410_export.cdb`.
4. Click **Export**.

3.3 HM-4420: Define ANSYS Contacts for Models in HyperMesh

In this tutorial you will learn how to set up edge to edge contacts, set up contacts manually, and create edge to edge contacts between circular and rectangular parts of a model.

Before you begin, it is recommended that you complete HM-1000: Getting Started with HyperMesh and Exploring the ANSYS Interface.

Also, copy the `hm-ansys_contact_manager_2-d_tutorial.hm` from `<hm.zip>/interfaces/ansys/` to your working directory.

In HyperMesh, you can create contact pairs manually or with the Autocontact tool. The edge of the circular body acts as a Target (secondary) surface, and the top edge of the rectangular part acts as a contact (main) surface.

> **Restriction:** This tutorial is only applicable for HyperMesh 14.0.130 and on.

3.3.1 Load the ANSYS User Profile

In this step, you will load the ANSYS user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Ansys**.

3.3.2 Retrieve the Model File

In this step, you will open a model file in HyperMesh.

1. From the menu bar, click **File** > **Open** > **Model**.
2. In the **Open Model** dialog, open the `hm-ansys_contact_manager_2-d_tutorial.hm` file.
   The model loads in the graphics area.
3. Optional: If the load collector is displayed, click in the Model Browser next to the load collector to turn off the display of the element.

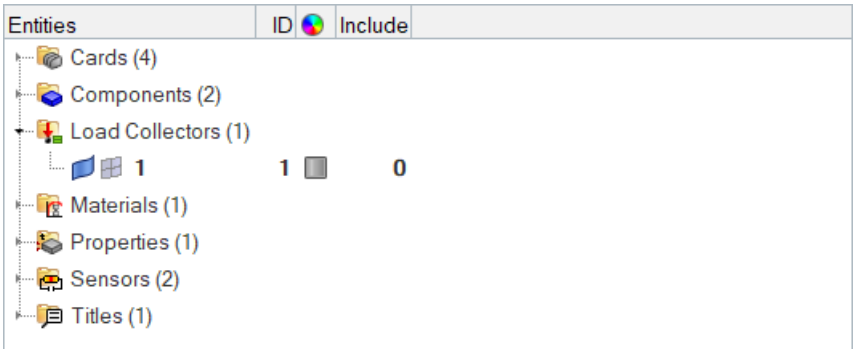


Figure 812:

4. In the graphics area, click once to make it the current window for keyboard commands.
5. Fit the model to the graphics area by pressing **F**.

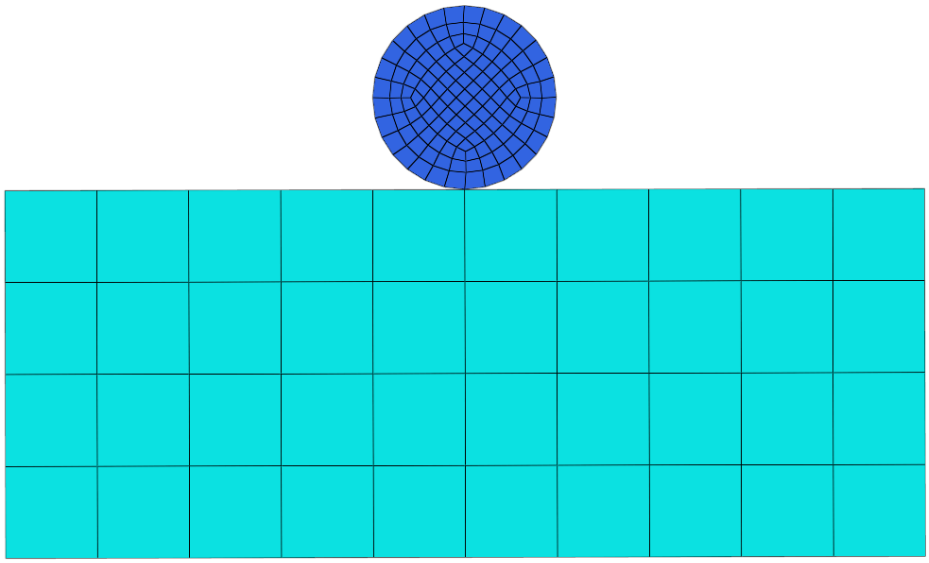


Figure 813:

3.3.3 Open the Contact Browser

In this step, you will open the Contact Browser in HyperMesh.

From the menu bar, click **View** > **Browsers** > **HyperMesh** > **Contact**.
The Contact Browser opens.

3.3.4 Create Target (Secondary) Surface

In this step, you will create a target surface in HyperMesh.

1. In the first pane of the Contact Browser, right-click and select **Create** > **Contact Surfaces** from the .context menu

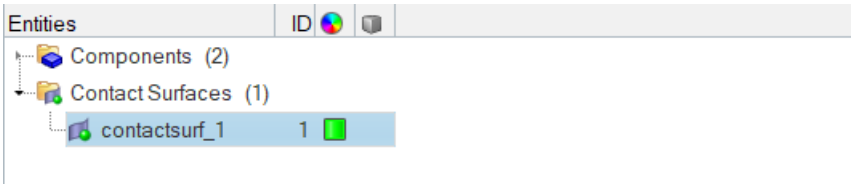


Figure 814:

HyperMesh creates and opens a contact surface in the Entity Editor.

2. In the Entity Editor, enter a name and ID, and select a color for the contact surface.
3. For Elements, click **0 Elements** > **Elements**.

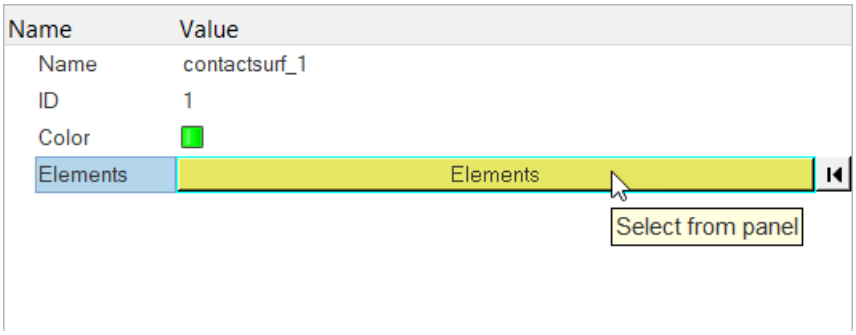


Figure 815:

4. In the panel area, set the first selector to **add shell edges**.
5. Check the **reverse normals** checkbox.
6. For face angle, enter `30.0`.
7. Set the entity selector to **elems**, then select the free edges indicated in Figure 816.

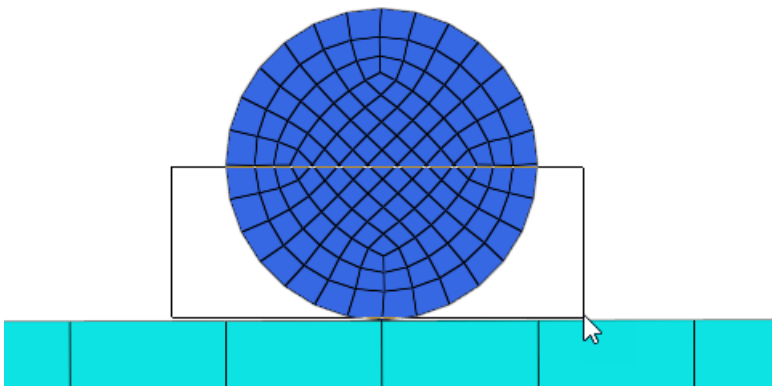


Figure 816:

> **Tip:** Quickly select elements with window selection by pressing **Shift** while clicking and dragging your mouse.

8. Use the nodes selector to select two nodes along the free edges you selected in step 7.

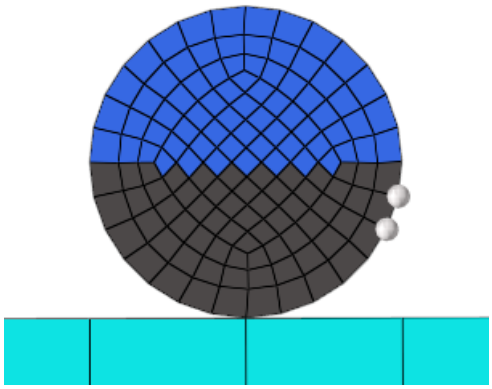


Figure 817:

9. Click **Add**.
All of the edges of the selected elements are added to the surface.

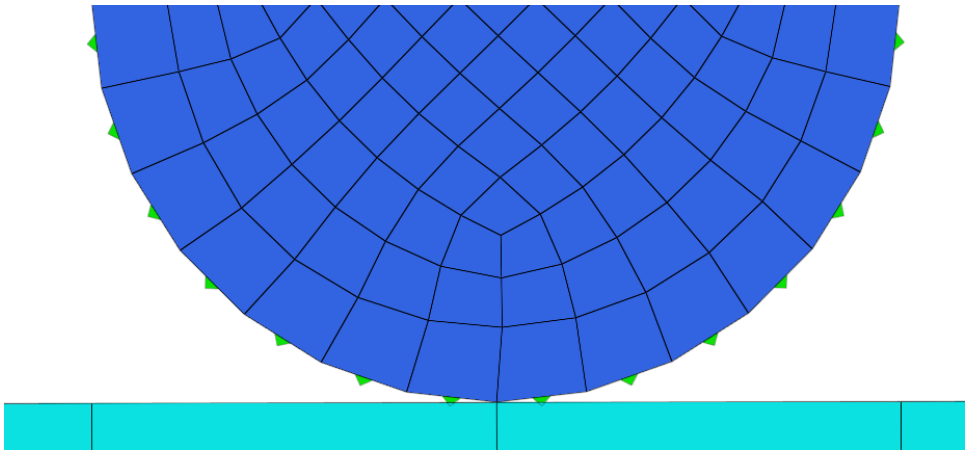


Figure 818:

10. Click **return** to exit the panel.

11. Review the contact surface.

a) In the Contact Browser, right-click on **contactsurf_1** and select **Review** from the context menu.

b) Isolate the contact surface for a better visual display.

The surface created here will be considered the target (secondary) surface during analysis.

3.3.5 Create Contact (Master) Surface

In this step, you will create a contact surface in HyperMesh.

1. In the first pane of the Contact Browser, right-click and select **Create** > **Contact Surfaces** from the context menu.
   HyperMesh creates and opens a contact surface in the Entity Editor.
2. In the Entity Editor, enter a name and ID, and select a color for the contact surface.
3. For Elements, click **0 Elements** > **Elements**.
4. In the panel area, set the first selector to **add shell edges**.
5. For **face angle**, enter `30.0`.
6. Check the **reverse normals** checkbox.
7. Set the entity selector to **free edges**, then select the top free edges of the rectangular box as indicated in Figure 819.

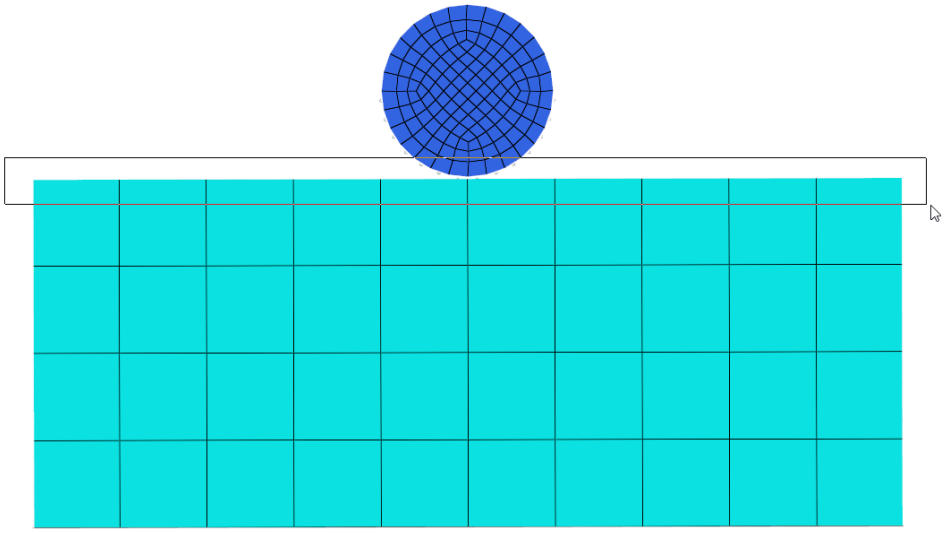


Figure 819:

8. Click **add**.

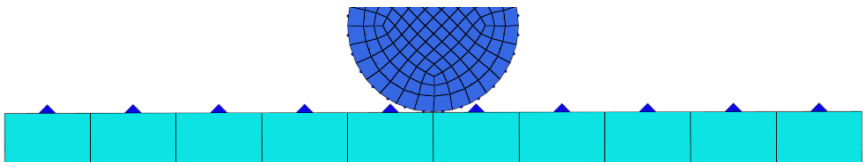


Figure 820:

   Surfaces are created on the top edge of the rectangular box.
9. Click **return** to exit the panel.

3.3.6 Create a Contact Pair

In this step, you will create a contact pair.

1. In the second pane of the Contact Browser, right-click and select **Create** > **Contact Pair** from the context menu.
   HyperMesh creates and opens a contact pair in the Entity Editor.

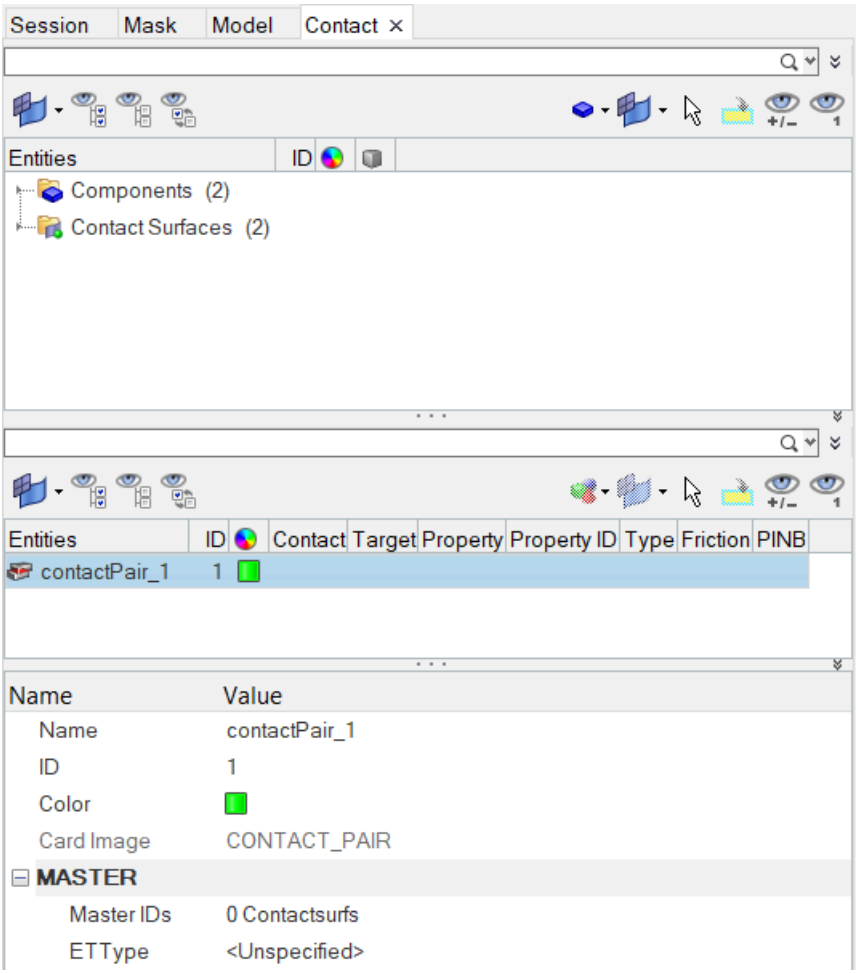


Figure 821:

2. In the Entity Editor, enter a name and ID, and select a color for the contact pair.
3. Attach the contact (master) surface.
   a) For Master IDs, click **0 Contactsurfs** > **Contactsurfs**.

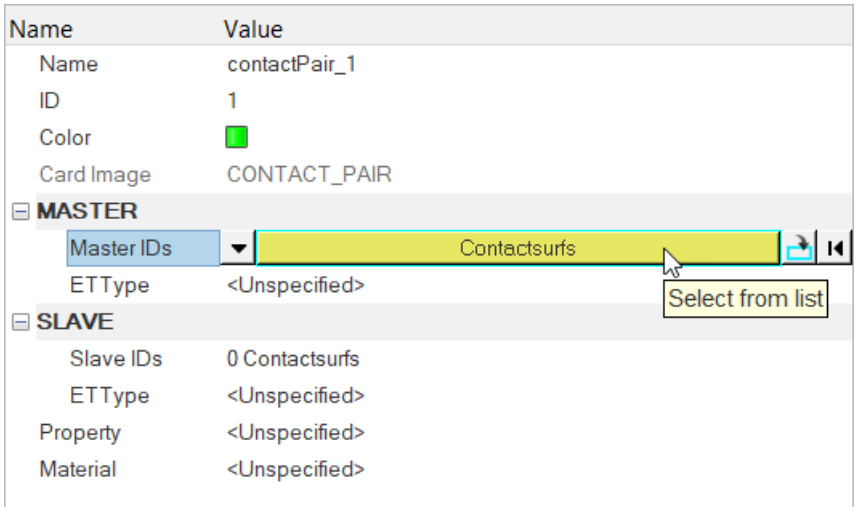


Figure 822:

b) In the **Select Contactsurfs** dialog, select **contactsurf_2** and click **OK**.

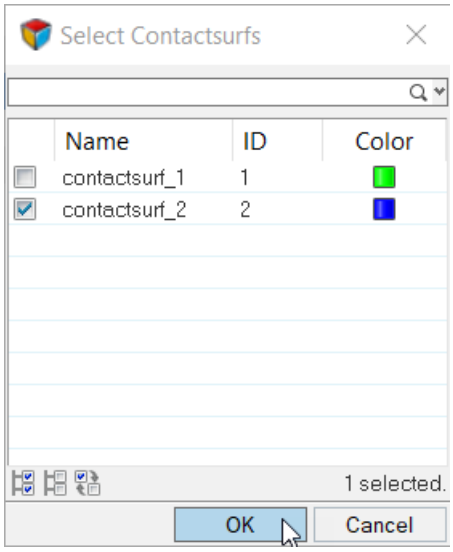


Figure 823:

c) Under MASTER, right-click on **ETType**, and select **Create** from the context menu.

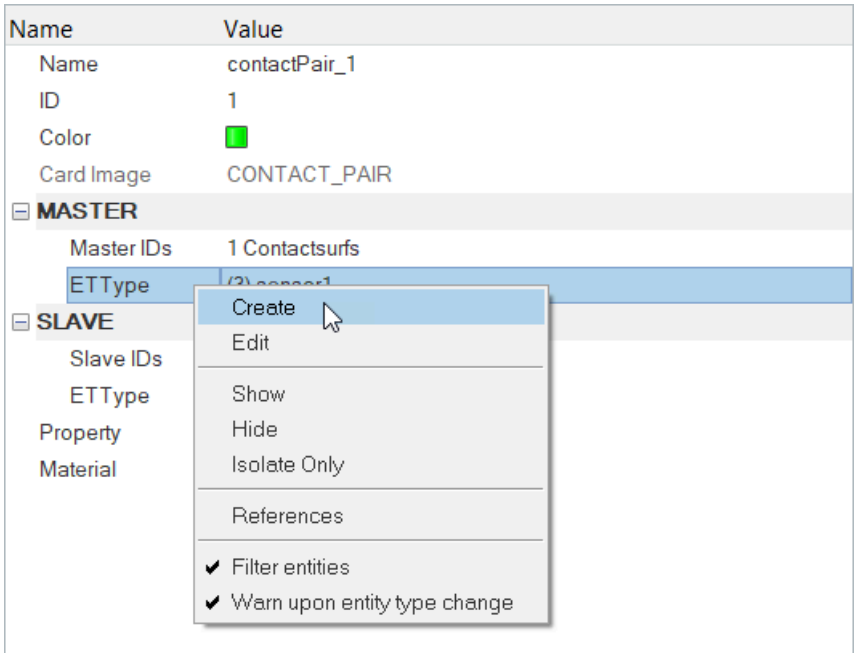


Figure 824:

d) In the **Create Sensors** dialog, define the sensor.

- Enable **KeyOpt12**.
- Select **0-Standard** from the list.

By default, the Element Type is set to CONTA171. Keyopts determine the behavior of contacts during analysis. KeyOpt12 defines the type of contact.

e) Click **Close**.

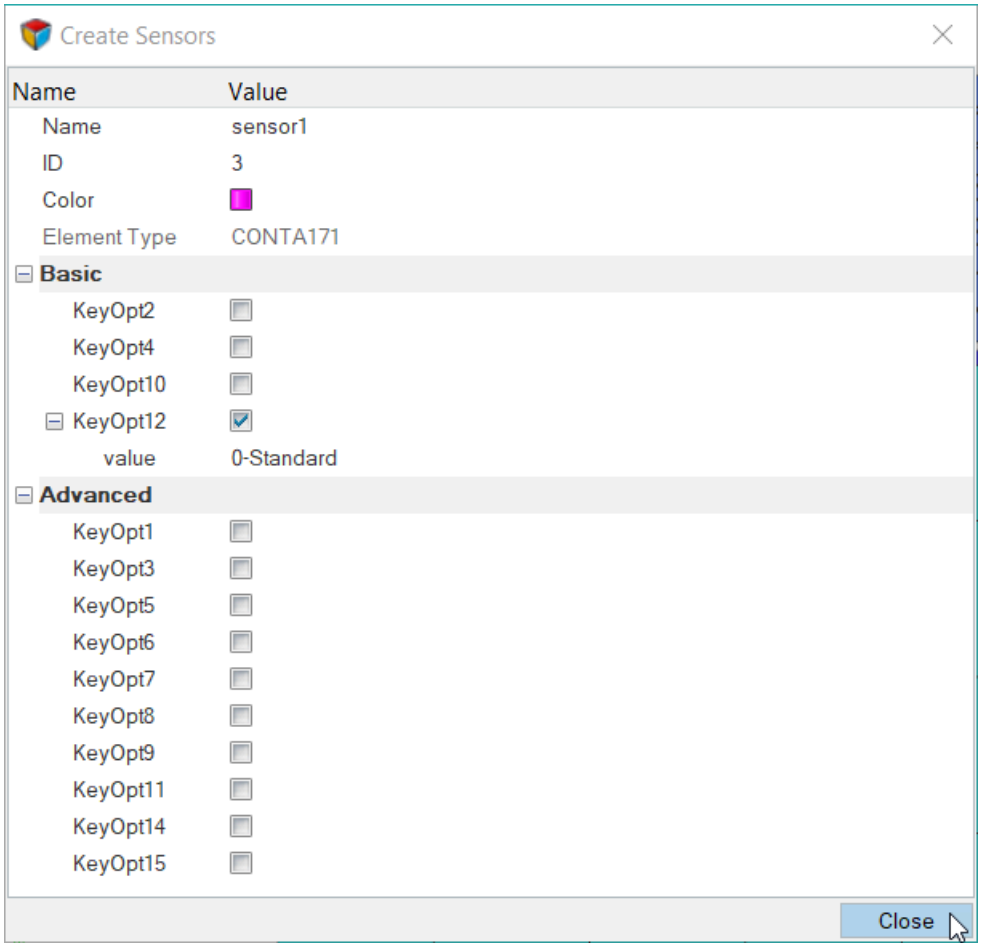


Figure 825:

4. Attach the target (secondary) surface.
   a) For Secondary IDs, click **0 Contactsurfs** > **Contactsurfs**.
   b) In the **Select Contactsurfs** dialog, select **contactsurf_1** and click **OK**.

      In this contact pair, contactsurf_1 acts as the target (secondary) surface. This surface will be exported with the element type Targe169.
   c) Under SECONDARY, right-click on **ETType** and select **Create** from the context menu.
   d) Define the sensor in the **Create Sensors** dialog.
      - Enable **KeyOpt4**.
      - Select **111-ROTZ, UY, UX** from the list.

      By default, the Element Type is set to Targe169. The target (secondary) surface will be exported with the elements type TARGE169.
   e) Click **Close**.
5. Create and attach the contact property.

   A contact property for a pair is always required, as the solver recognizes the pair only by the property ID associated with the main and secondary surfaces. Same property IDs are shared by

the main and secondary surfaces. If a property ID is missing for a contact surface, then it will not be considered as part of the contact pair and it will not recognized.

a) In the Entity Editor, right-click on **Property** and select **Create** from the context menu.

b) Define the property in the **Create Properties** dialog.

- For FKN, enter `1.0`.
- For FTOLN, enter `0.1`.
- For TAUMAX, enter `1e20`.
- For FKOP, enter `1.0`.
- For FKT, enter `1.0`.
- For FHTG, enter `1.0`.
- For RDVF, enter `1.0`.
- For FWGT, enter `0.5`.
- For FACT, enter `1.0`.

c) Click **Close**.

6. Create and attach the contact material.

The properties MU (coefficient of friction) and EMIS (Thermal Emissivity) are available to define. If you are creating a structural contact, select **MU**; if you are creating a thermal contact, select **EMIS**. In this exercise you will be creating a structural contact.

a) In the Entity Editor, right-click on **Material** and select **Create** from the context menu.

b) Define the material in the **Create Materials** dialog.

- Select the **MU** (coefficient of friction) checkbox.
- For MP_MU_LEN, enter `1`.
- For C, enter `0.45`.

c) Click **Close**.

3.3.7 Review the Contact Pair

In this step, you will review the contact pair you created in step Create a Contact Pair.

1. In the second pane of the Contact Browser, select the contact pair created in Create a Contact Pair.
The contact pair entity details display in the Entity Editor.

2. Right-click on the contact pair and select **Isolate Attached** > **Elements** to review the contact area.

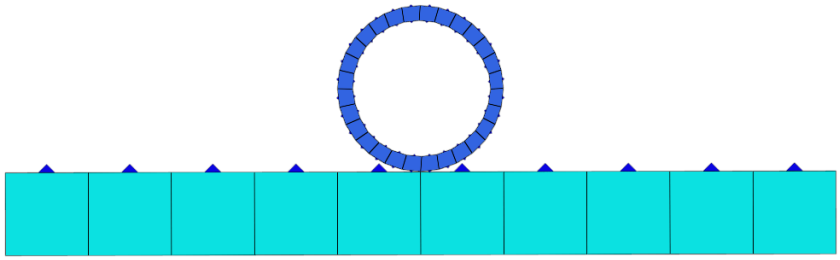


Figure 826:

3.3.8 Add /SOLU, ANTYPE, and SOLVE in the Control Cards

In this step, you will add the following Control Cards.

Restriction: Only complete this step if your model needs to be solved in ANSYS.

1. Click **Setup** > **Create** > **Control Cards** from the menu bar.
   The Control Cards panel opens.
2. To exit the PREP7 preprocessor and enter the SOLU preprocessor click **/SOLU**.

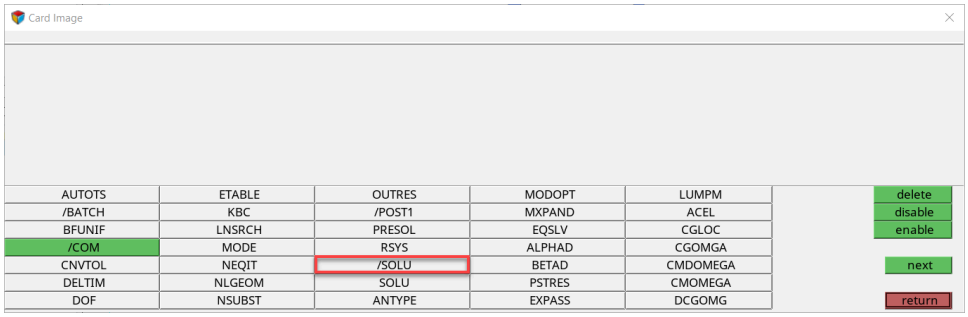


Figure 827:

3. Click **return**.



Figure 828:

4. Since you are solving the model for static analysis, click **ANTYPE**.

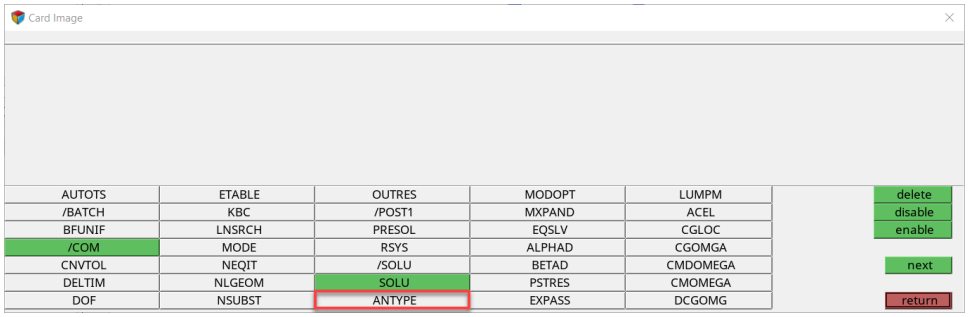


Figure 829:

5. Set type to **STATIC** and status to **NEW**.



Figure 830:

6. Click **return**.
7. Click **SOLVE**.

> **Tip:** If you do not see the SOLVE Control Card, click **next**.

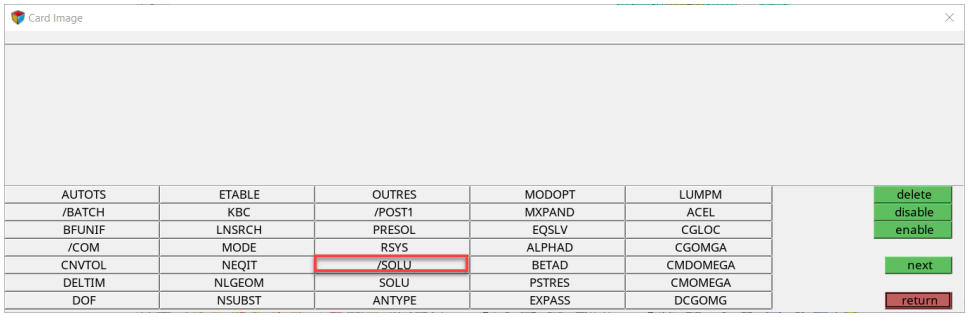


Figure 831:

8. Click **return** to exit the card image.
9. Click **return** to exit the Control Cards panel.

3.4 HM-4430: Define ANSYS Contacts for Models in HyperMesh

In this tutorial you will learn how to set up surface to surface contacts and contacts manually.

Before you begin, it is recommended that you complete HM-1000: Getting Started with HyperMesh and Exploring the ANSYS Interface.

Also, copy `hm-ansys_contact_manager_3-d_tutorial.hm` from `<hm.zip>/interfaces/ansys/` to your working directory.

In HyperMesh, you can create contact pairs manually or with the Autocontact tool.

Restriction: This tutorial is only applicable for HyperMesh 14.0.130 and on.

3.4.1 Load the ANSYS User Profile

In this step you will load the ANSYS user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Ansys**.

3.4.2 Retrieve the HyperMesh Model File

In this step, you will open a model file in HyperMesh.

1. From the menu bar, click **File** > **Open** > **Model**.
2. In the **Open Model** dialog, open the `hm-ansys_contact_manager_3-d_tutorial.hm` file.
   A model loads in the graphics area.
3. If the load collector is displayed, in the Model Browser, click next to the load collector to turn off the display of its elements.

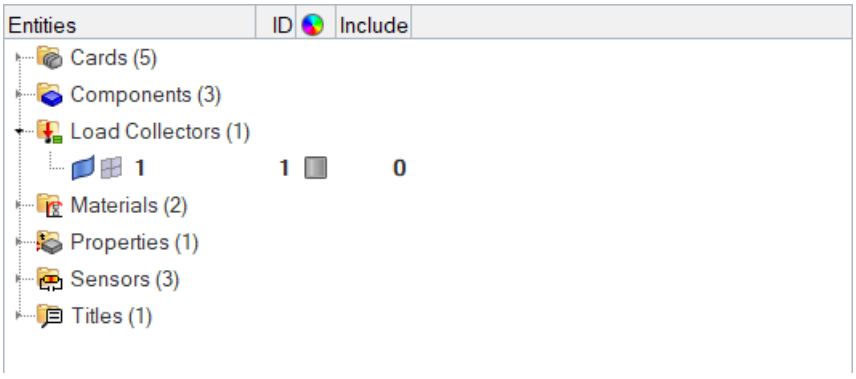


Figure 832:

4. In the graphics area, click once to make it the current window for keyboard commands.

5. Fit the model to the graphics area by pressing **F**.

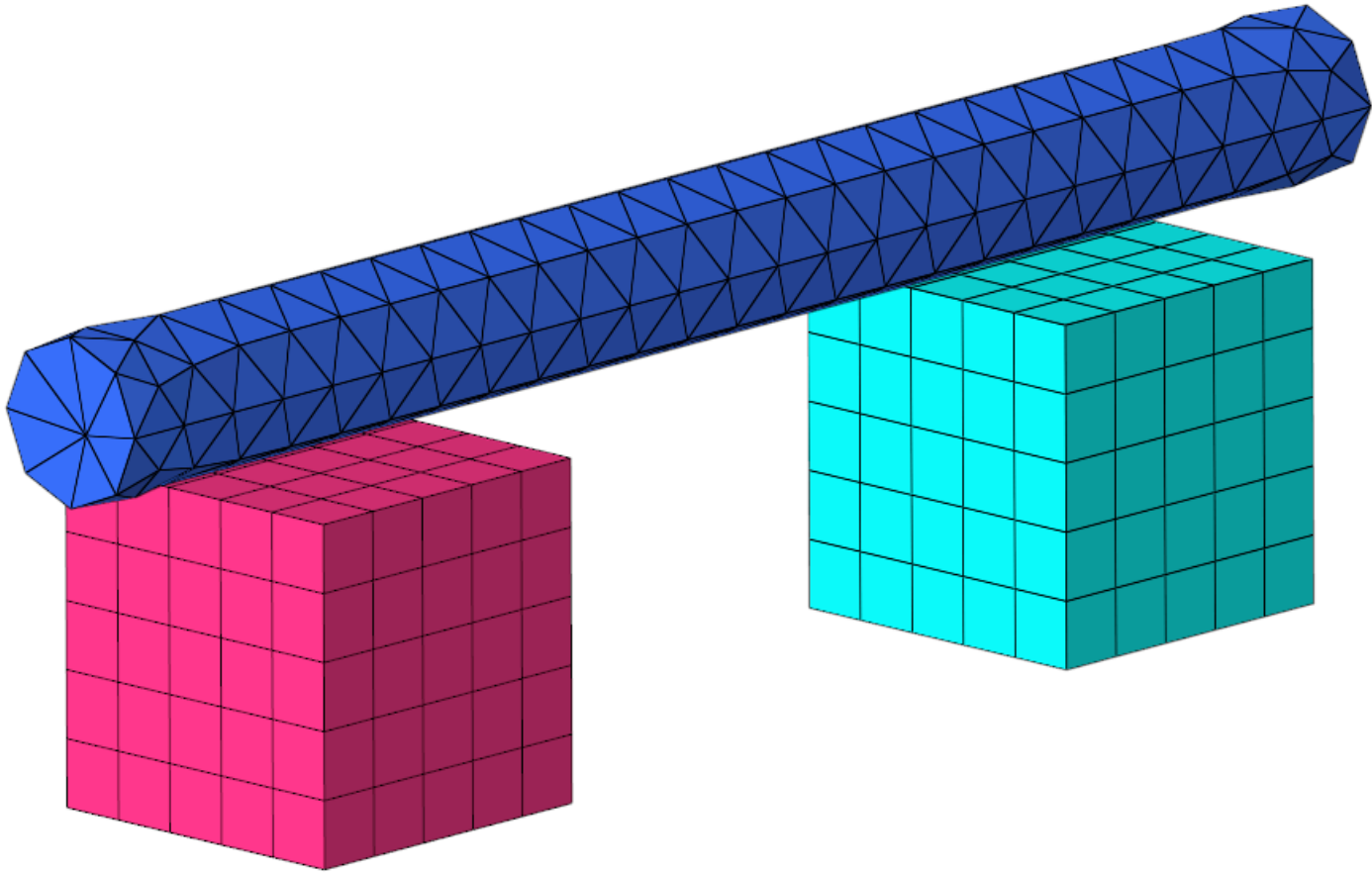


Figure 833:

3.4.3 Create a Contact Pair

In this step you will create a contact pair in HyperMesh.

1. Optional: If necessary, click **View** > **Browsers** > **HyperMesh** > **Contact** to open the Contact Browser.
2. In the second pane of the Contact Browser, right-click and select **Create** > **Contact Pair** from the context menu.
   HyperMesh creates and opens a contact pair in the Entity Editor as indicated in Figure 834.

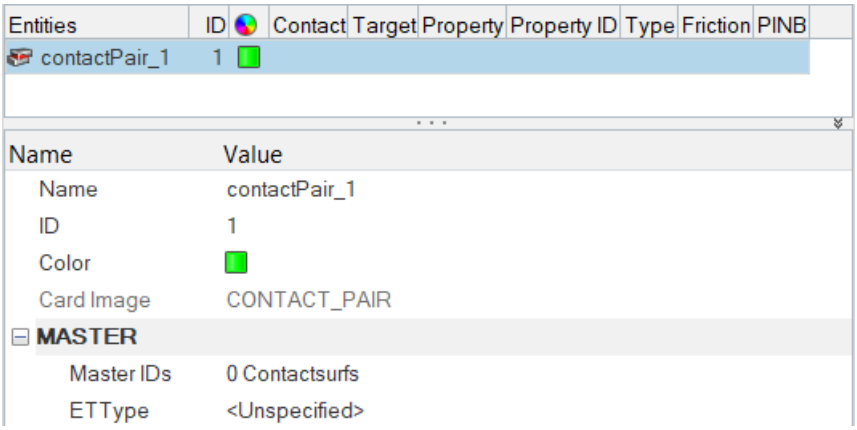


Figure 834:

3.4.4 Create and Attach a Contact Surface

In this step you, will create and attach a contact (master) surface in HyperMesh.

1. Mask the BOX_SOLID45 and CYLINDER_SOLID45 components in the first pane of the Contact Browser.
   a) Right-click on the **BOX_SOLID45** component.
   b) Select **Hide** from the context menu.
   c) Right-click on the **CYLINDER_SOLID45** component.
   d) Select **Hide** from the context menu.

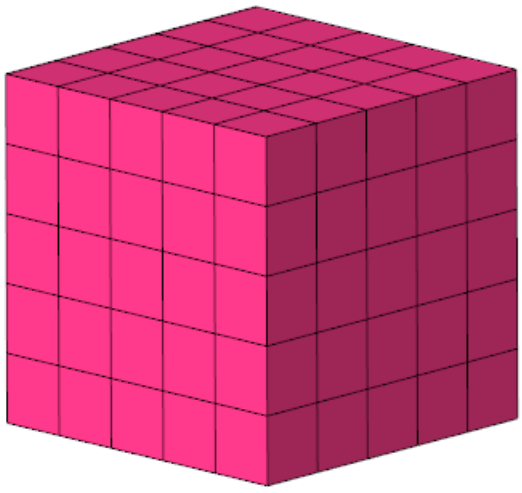


Figure 835:

2. In the second pane of the Contact Browser, select **contactPair_1**.
3. Create and attach a contact (master) surface.
   a) In the Entity Editor, right-click on **Master IDs** and select **Create Contact Surface using Elements** from the context menu.

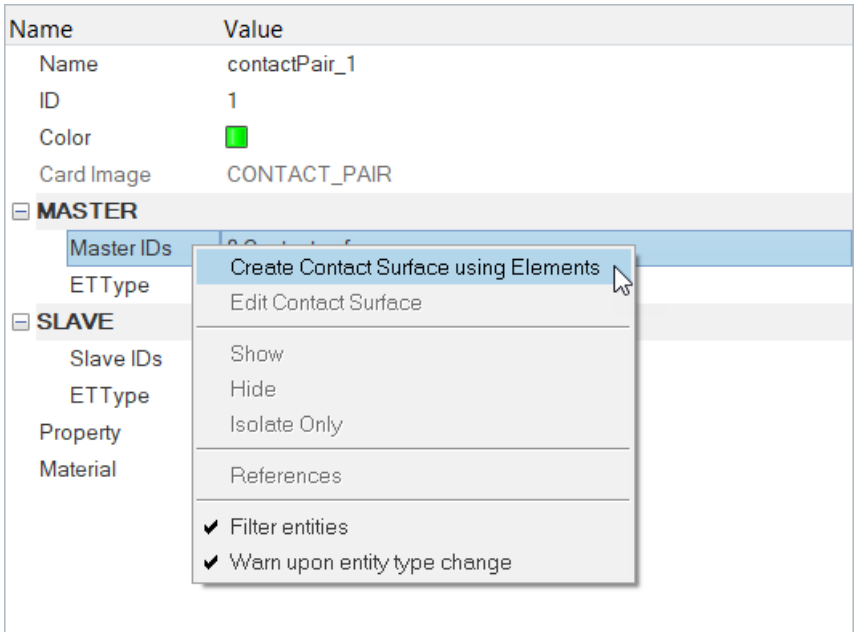


Figure 836:

   b) In the panel area, set the first switch to **add solid faces**.
   c) Set the advanced selection switch to **faces**.

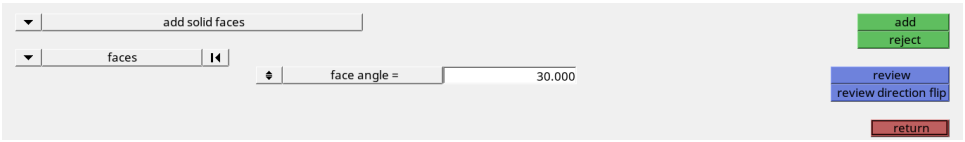


Figure 837:

   d) Select the top faces of the cube as indicated in Figure 838.

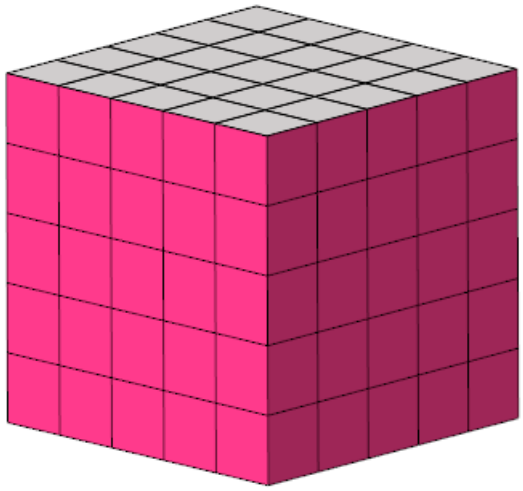


Figure 838:

e) Click **add**.

f) Observe the newly created contact surfaces.

By default, when contact surfaces are created over solid element faces, normals point outwards.

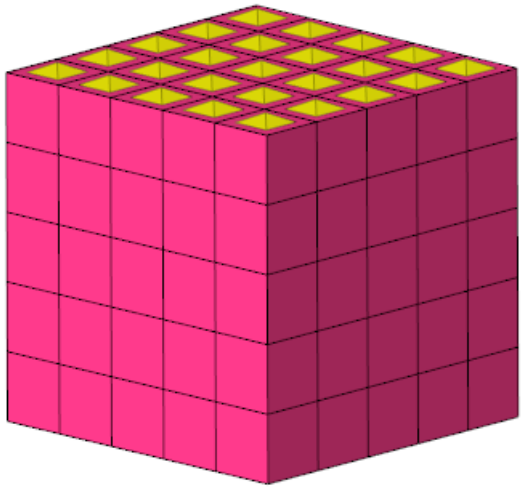


Figure 839:

g) In the panel area, click **return**.

4. Create and attach an Et Type to the contact surface.

a) In theEntity Editor, under MASTER, right-click on **ETType** and select **Create** from the context menu.

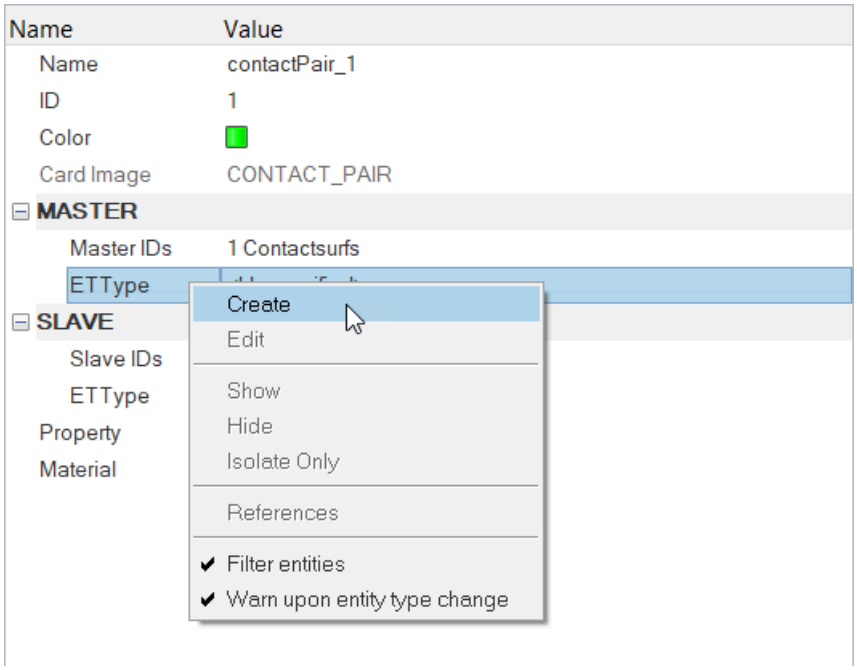


Figure 840:

b) In the **Create Sensors** dialog, define the sensor.

> **Note:** By default, the Element Type is set to CONTA174.

- Enable **KeyOpt12**.
- Select **0-Standard** from the list.

c) Close the dialog.

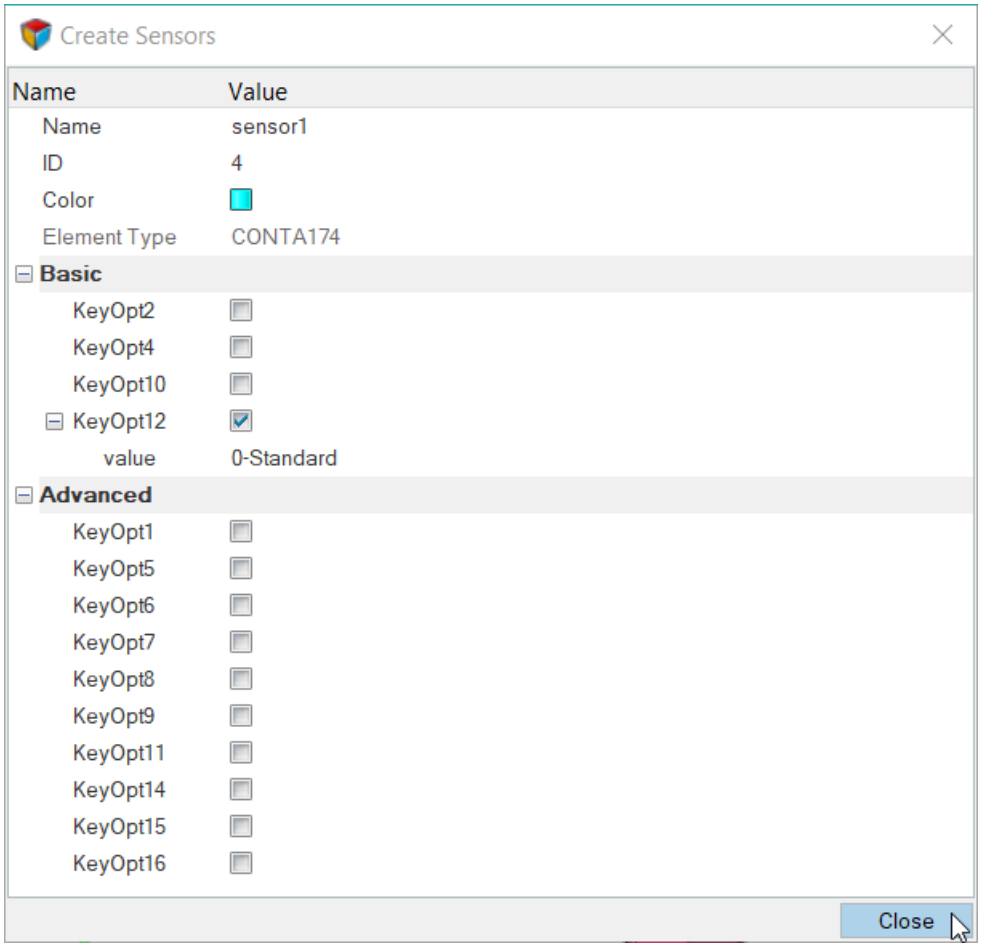


Figure 841:

3.4.5 Create and Attach a Target Surface

In this step, you will create and attach a target surface in HyperMesh.

The bottom half of the CYLINDER_SOLID45 component will be used as the target surface.

1. Unmask the CYLINDER_SOLID45 component.
   a) Right-click on **CYLINDER_SOLID45** in the first pane of the Contact Browser.
   b) Select **Show Attached** > **Contacts** from the context menu.
2. Mask the BOX_SOLID95 component.
   a) Right-click on **BOX_SOLID95** in the second pane of the Contact Browser.
   b) Select **Hide** from the context menu.
3. Mask the contactPair_1 contact surfaces.
   a) Right-click on **contactPair_1** in the second pane of the Contact Browser.
   b) Select **Hide** from the context menu.

4. On the Standard Views toolbar, click  to orient the model to the XY top plane view.

5. Create and attach a target (secondary) surface.

a) In the Entity Editor, right-click on **Secondary IDs** and select **Create Contact Surface using Elements** from the context menu.

b) In the panel area, set the first switch to **add solid faces**.

c) Set the advanced selection switch to **elems**.

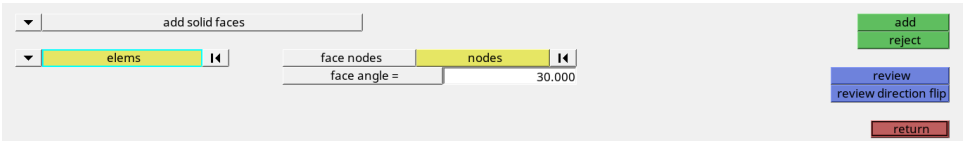


Figure 842:

d) Select the elements indicated in Figure 843.

> **Tip:** Quickly select elements with window selection by pressing **Shift** while clicking and dragging your mouse.

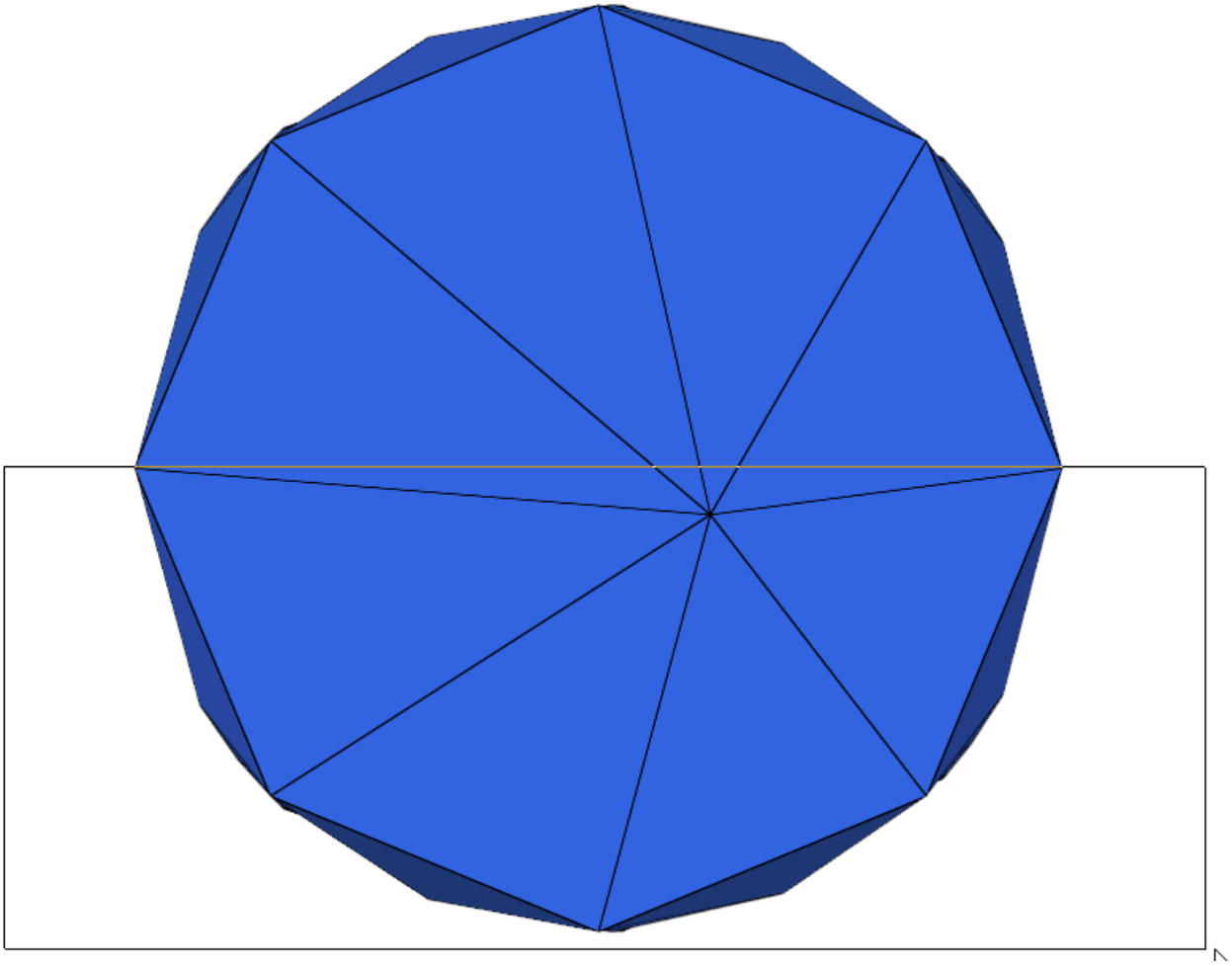


Figure 843:

e) Using the nodes selector, select face nodes on the selected elements as indicated in Figure 844.

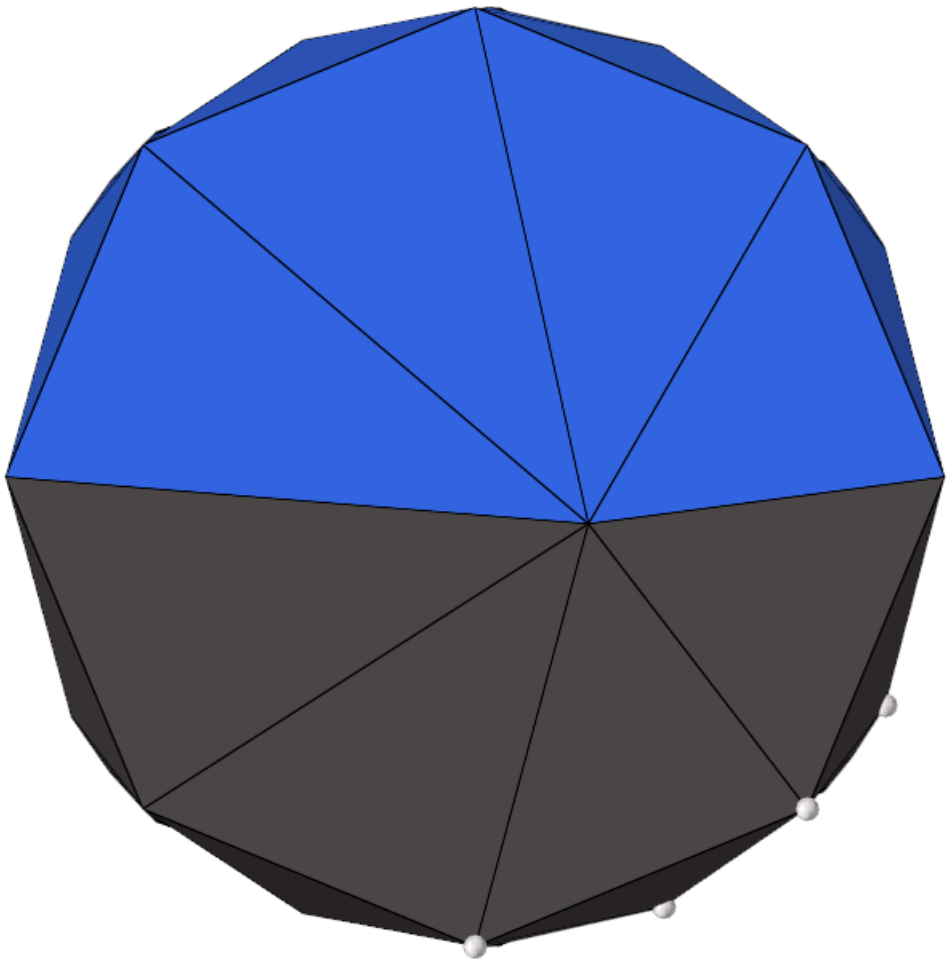


Figure 844:

f) Click **add**.
Surfaces are created on selected elements.

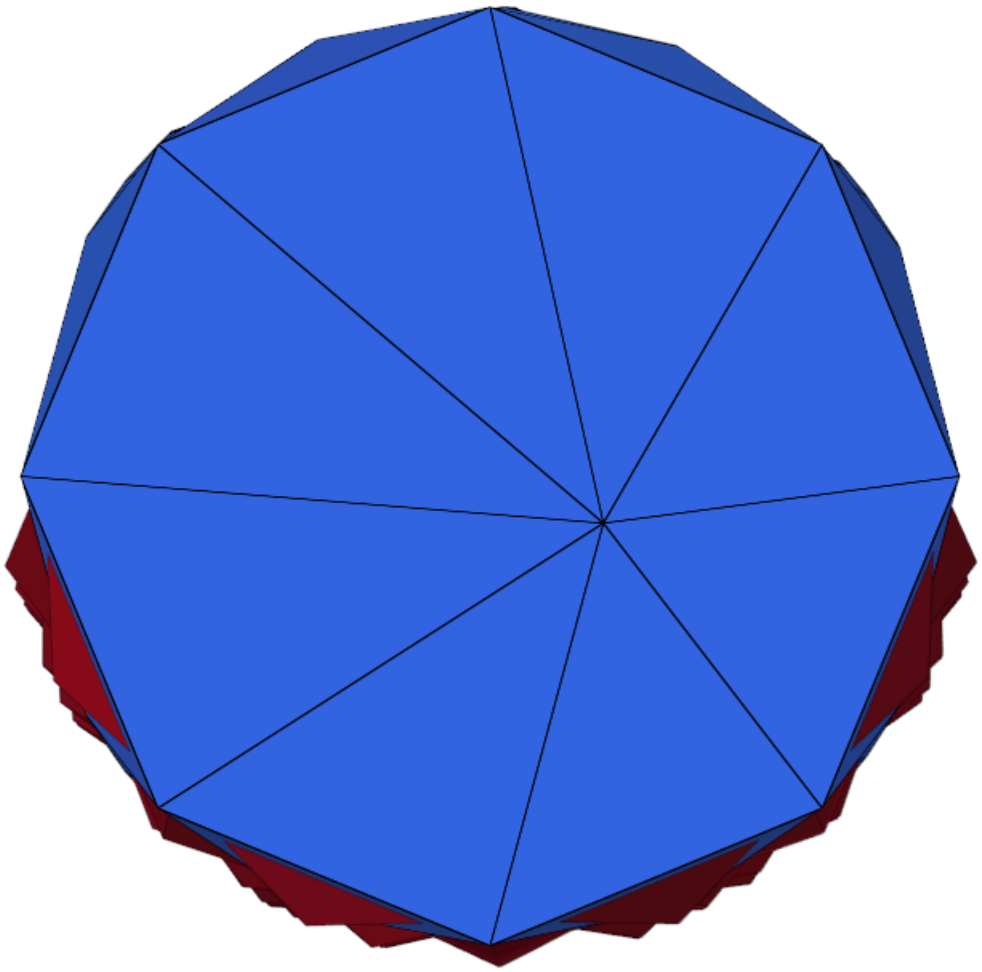


Figure 845:

g) In the panel area, click **return**.

6. Create and attach an Et Type to the contact surface.

a) In the **Entity Editor**, under SECONDARY, right-click on **ETType** and select **Create** from the context menu.

b) In the **Create Sensors** dialog, define the sensor.

> **Note:** The Element Type is set to TARGE169 by default.

- Enable **KeyOpt4**.
- Select **0-All** from the list.

c) Close the dialog.

3.4.6 Create and Attach a Property and Material to the Contact Pair

In this step, you will create and attach a property and material to a contact pair.

1. Create and attach a contact property.
   a) In the **Entity Editor**, right-click on **Property** and select **Create** from the context menu.
   b) In the **Create Properties** dialog, define the property.
   - For FKN, enter `1.0`.
   - For FTOLN, enter `0.1`.
   - For TAUMAX, enter `1e20`.
   - For FKOP, enter `1.0`.
   - For FKT, enter `1.0`.
   - For FHTG, enter `1.0`.
   - For RDVF, enter `1.0`.
   - For FWGT, enter `0.5`.
   - For FACT, enter `1.0`.
2. Click **Close**.
3. Create and attach a contact material.
   a) In the Entity Editor, right-click on **Material** and select **Create** from the context menu.
   The **Create Materials** dialog opens.
   b) In the **Create Materials** dialog, define the material.
   - Set card image to **MPDATA**.
   - Select the **MU** (coefficient of friction) checkbox.
   - For MP_MU_LEN, enter `1`.
   - For C, enter `-0.45`.
   c) Click **Close**.

3.4.7 Review the Contact Pair

In this step, you will review the contact pair you created.

1. In the first pane of the Contact Browser, unmask all of the components and contact surfaces.
2. In the second pane of the Contact Browser, right-click on **contactPair_1** and select **Isolate Only** from the context menu.
   Only the main and secondary surfaces display.



Figure 846:

3. Check the normals of the surfaces.

> **Notice:** Notice how the surfaces are pointing out. If the normals are facing each other then it is a valid direction. In this tutorial, the normals should be pointing at each other.

4. Turn the display of all entities back on when you are done reviewing.
5. Right-click on **contactPair_1** and select **Review** from the context menu.
Contact regions display.

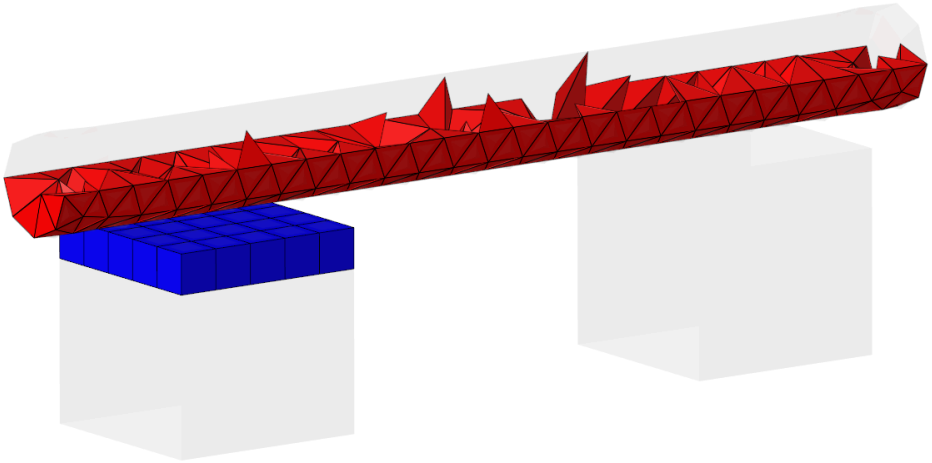


Figure 847:

6. Right-click on **contactPair_1** and select **Reset Review** from the context menu.

3.4.8 Add /SOLU and SOLVE in the Control Cards

In this step, you will add the following Control Cards.

> **Restriction:** Only complete this step if your model needs to be solved in ANSYS.

1. Click **Setup** > **Create** > **Control Cards** from the menu bar.

The Control Cards panel opens.

2. Click **/SOLU** to exit the PREP7 preprocessor and enter the SOLU preprocessor.

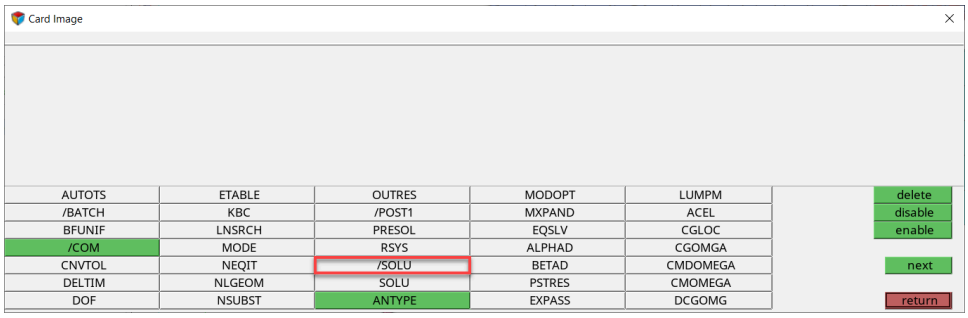


Figure 848:

3. Click **return**.

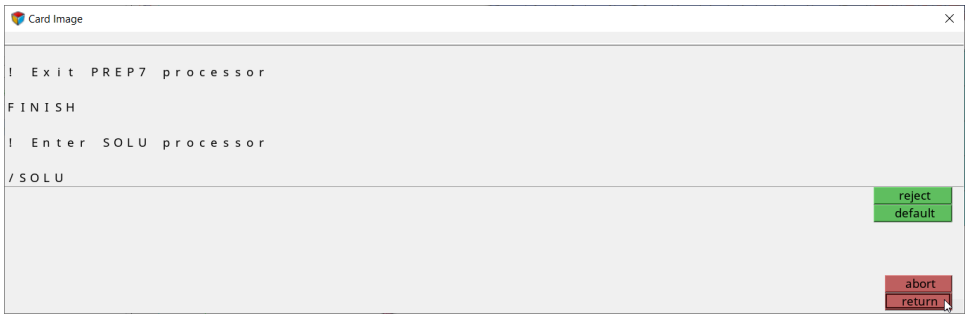


Figure 849:

4. Click **SOLVE**.

> **Tip:** If you do not see the SOLVE Control Card, click **next**.

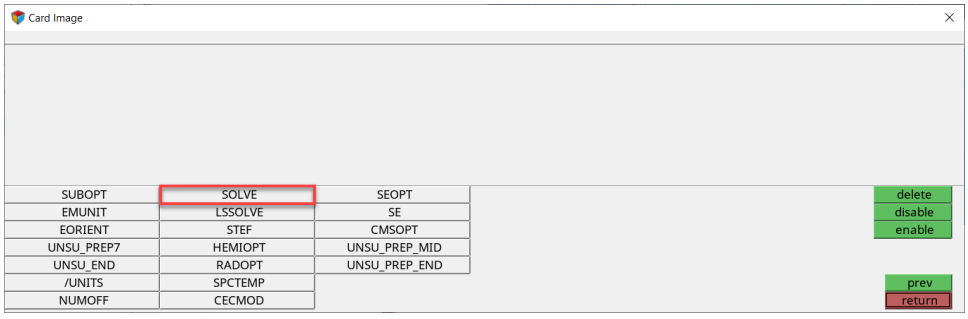


Figure 850:

5. Click **return** to exit the card image.
6. Click **return** to exit the Control Cards panel.

3.5 HM-4450: Introduction to HyperBeam

In this tutorial, you will learn how to couple degrees of freedom and to assign a standard beam section using the Section and HyperBeam panels.

Before you begin, copy `chapter3.hm` from `<hm.zip>/interfaces/ansys/` to your working directory.

Figure 851 and Figure 852 are pictorial representations of the original model. The model can be simplified in such a way that you can extract represent the entire model with a set of shell elements representing the plate part of the model and two lines indicating the beam.

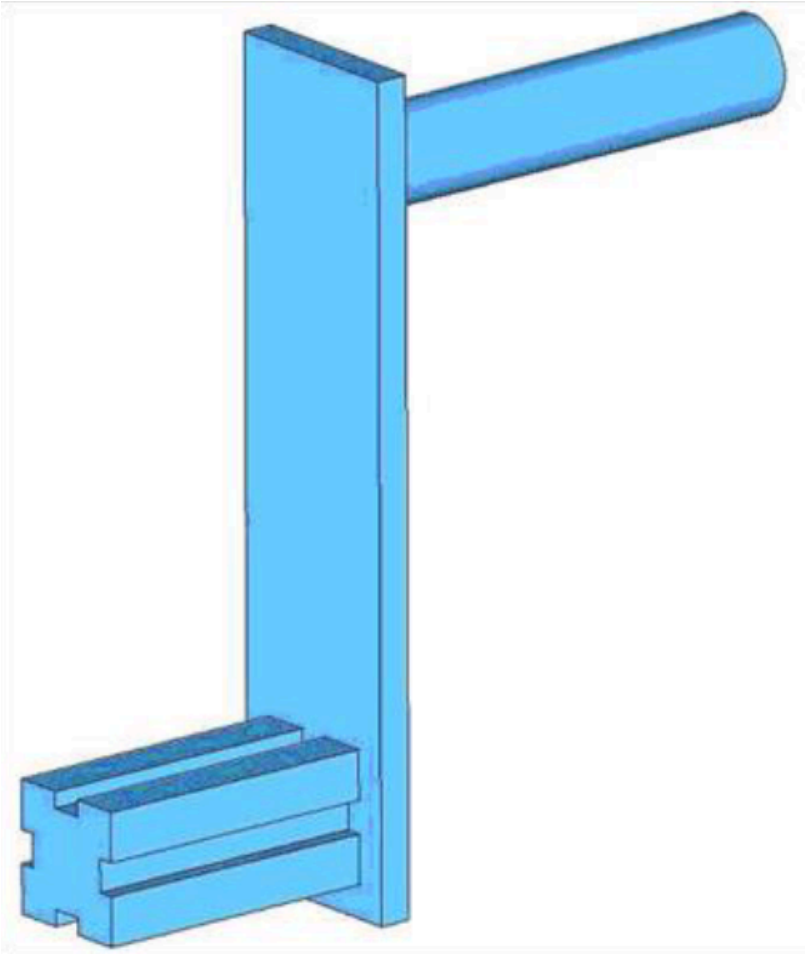


Figure 851:

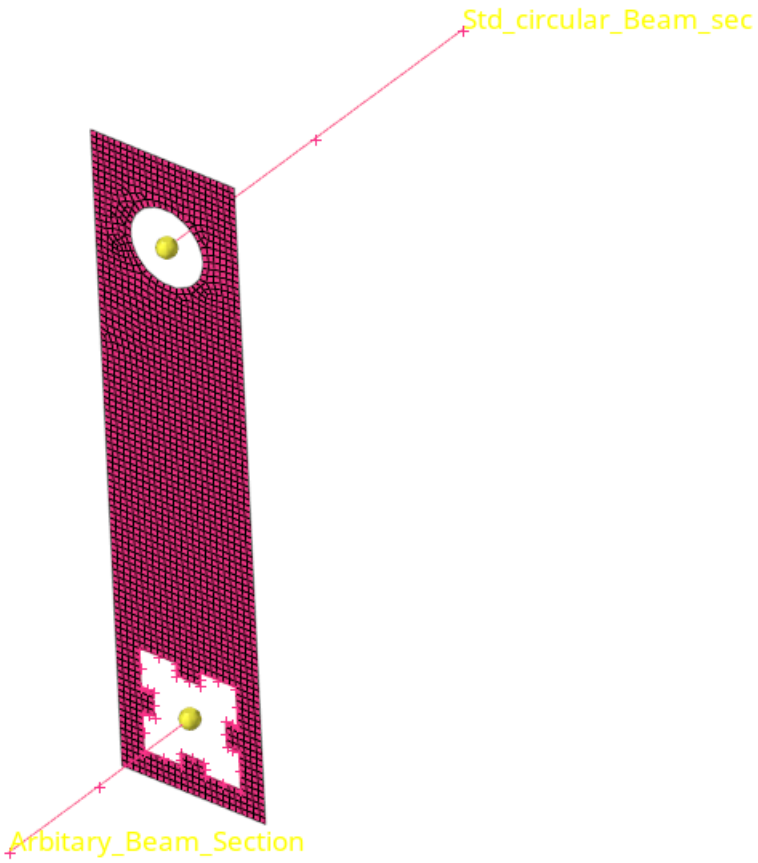


Figure 852:

3.5.1 Load the ANSYS User Profile

In this step, you will load the ANSYS user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Ansys**.

3.5.2 Retrieve the Model File

In this step, you will retrieve and open the model file in HyperMesh.

1. From the menu bar, click **File** > **Open** > **Model**.
2. In the **Open Model** dialog, open the `chapter3.hm` file.

   This model contains a plate collector which includes shell elements. The plate component is updated with the necessary element type, real constant, and material properties.

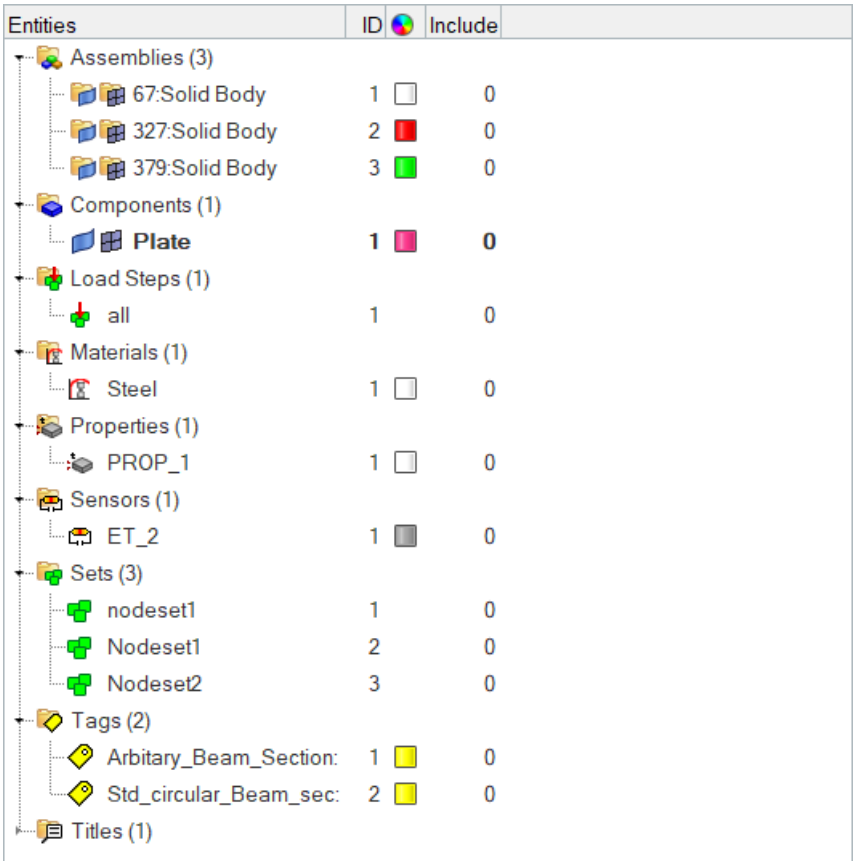


Figure 853:

The model displays in the graphics area.

3. Optional: If your model's elements and mesh lines are not shaded, click  on the Visualization toolbar.

3.5.3 Create Collectors and Attach Element Types and Materials

In this step, you will create collectors and attach element types and materials to the collectors.

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu.

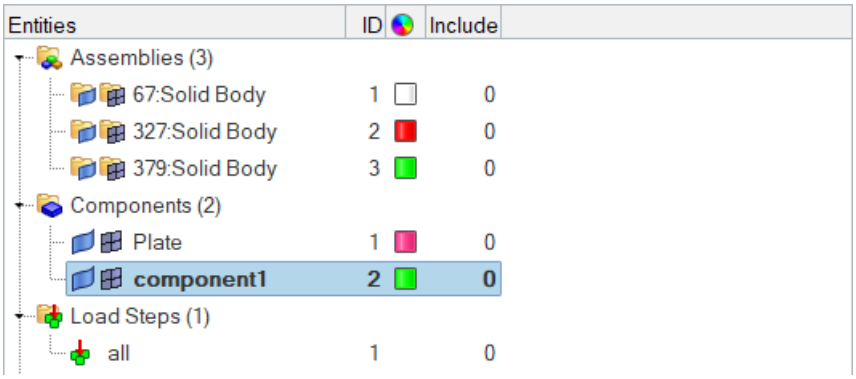


Figure 854:

HyperMesh creates and opens a component in the Entity Editor.

2. For Name, enter `beam_std`.
3. Click the **Color** icon, and select a color.
4. Right-click on **Type** and select **Create** from the context menu.

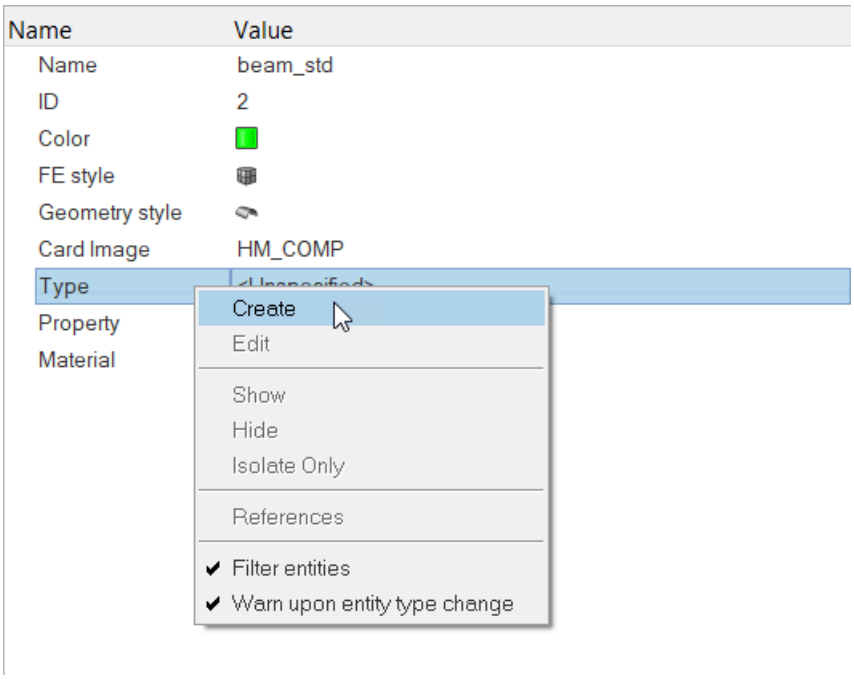


Figure 855:

The **Create Sensors** dialog opens.

5. Set Element Type to **BEAM188**.

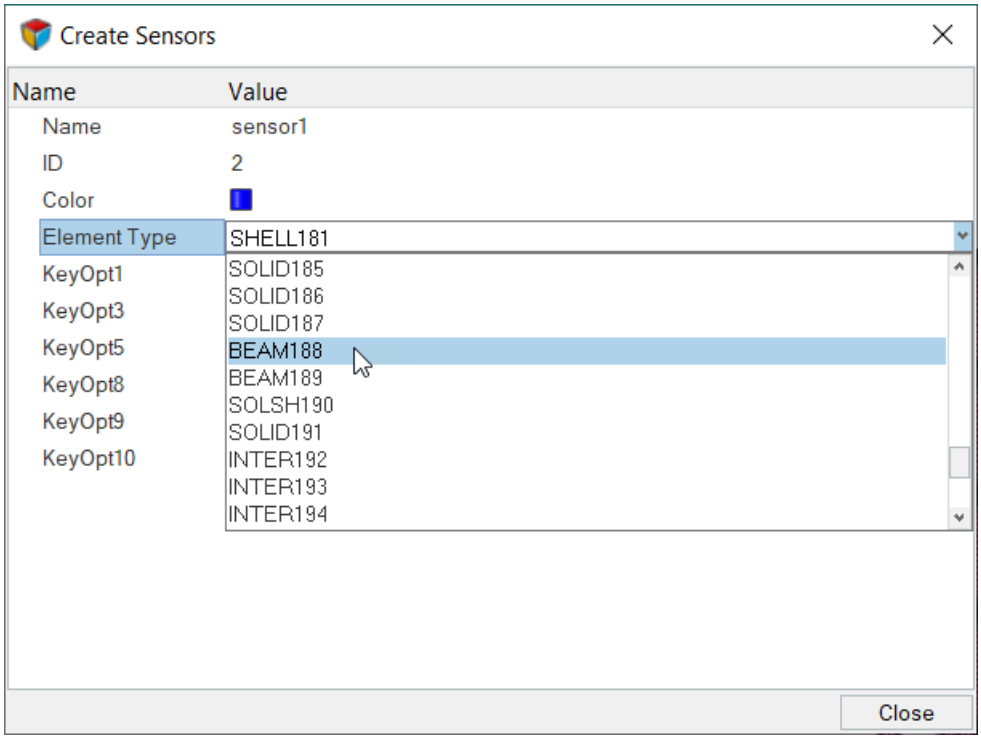


Figure 856:

6. Click **Close**.
7. For Material, click **Unspecified** > **Material**.
8. In the **Select Material** dialog, select **Steel** and then click **OK**.

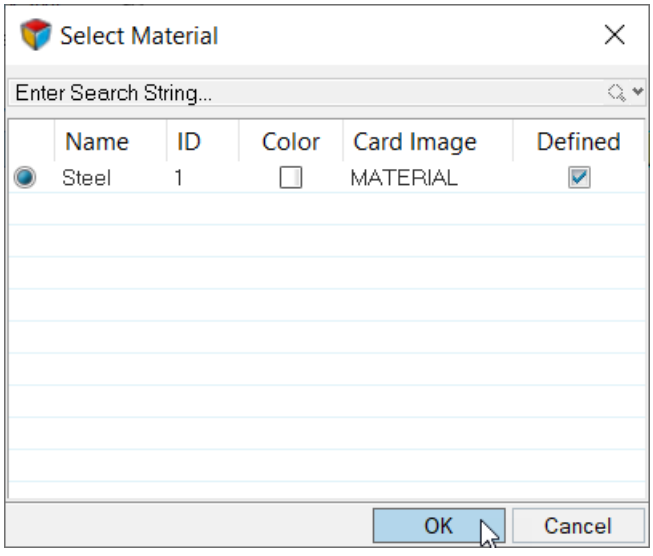


Figure 857:

9. In the Model Browser, right-click and select **Create** > **Component** from the context menu. HyperMesh creates and opens a component in the Entity Editor.

10. For Name, enter `beam_asec`.

11. Click the **Color** icon, and select a new color.

12. For Type, click **Unspecified** > **Sensor**.

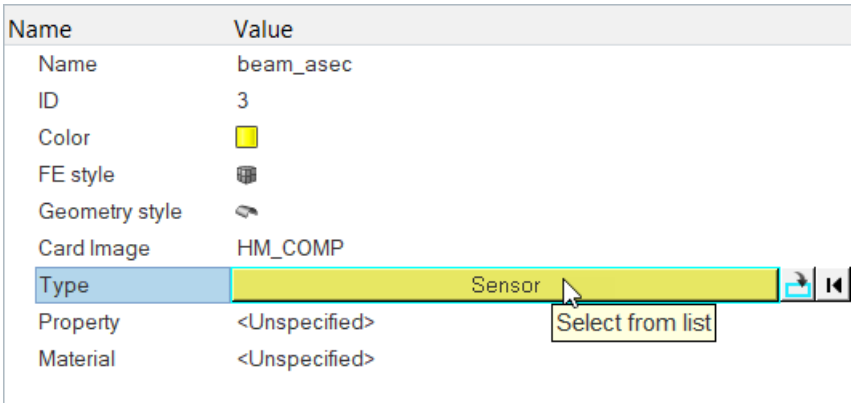


Figure 858:

13. In the **Select Sensor** dialog, select **sensor1** (BEAM188) and then click **OK**.

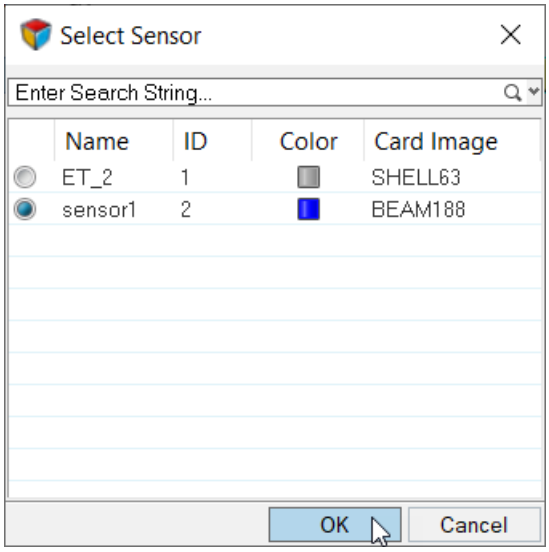


Figure 859:

14. For Material, click **Unspecified** > **Material**.

15. In the **Select Material** dialog, select **Steel** and then click **OK**.

3.5.4 Create a Beam Element

In this step, you will create a Beam element in HyperMesh.

In the Model Browser, notice that the current component is beam_asec.

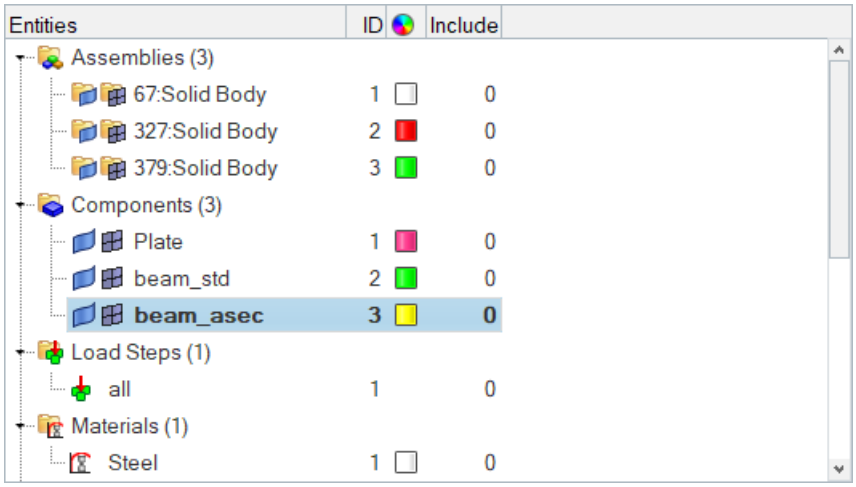


Figure 860:

1. From the menu bar, click **Mesh** > **Create** > **1D Elements** > **Bars**.
   The Bars panel opens.
2. Edit settings within the bar2 subpanel.
   a) Open the **bar2** subpanel.
   b) Set orientation to **plane**.
   c) Set Beam Y Axis to **parallel to XY**.

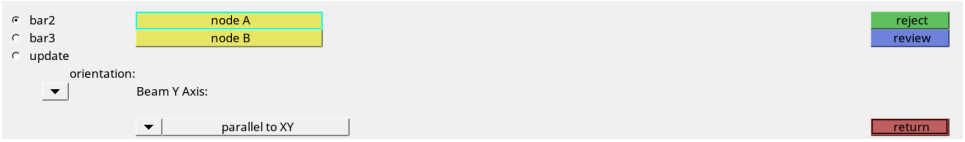


Figure 861:

3. Use the node A and node B selectors to select the two nodes that form the ends of the Arbitary_Beam_Section line as indicated in Figure 862.

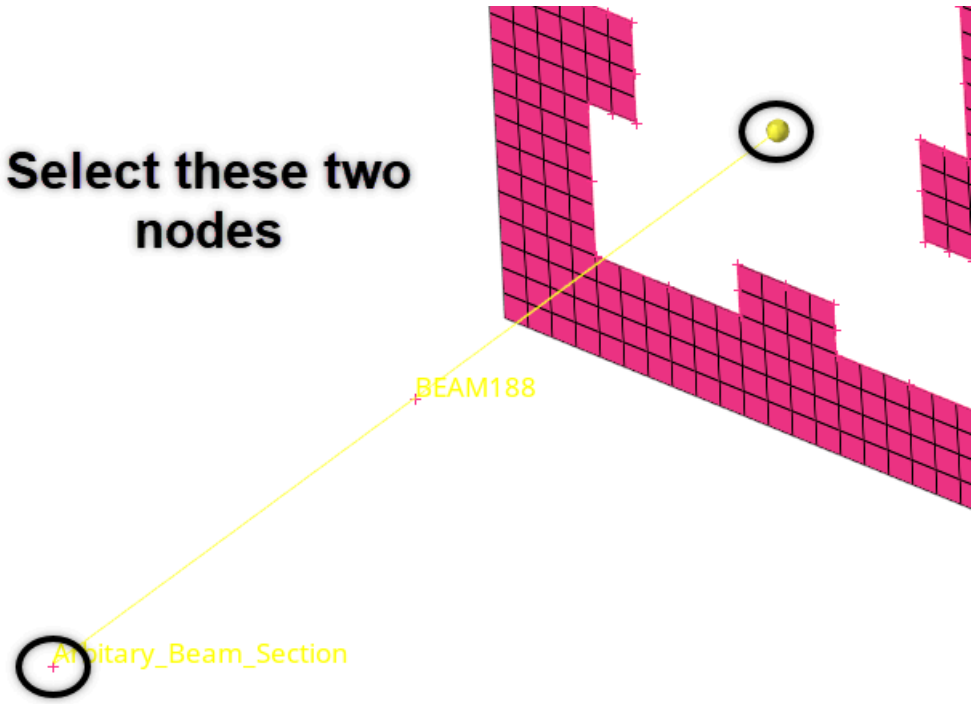


Figure 862:

> **Note:** Arbitary_Beam_Section is shown as a tag in the graphics area.

HyperMesh creates a BEAM188 element.

4. In the Component folder of the Model Browser, right-click on **beam_std** and select **Make Current** from the context menu.
5. Use the node A and node B selectors to select the two nodes that form the ends of the Std_circular_Beam_sec line as indicated in Figure 863.

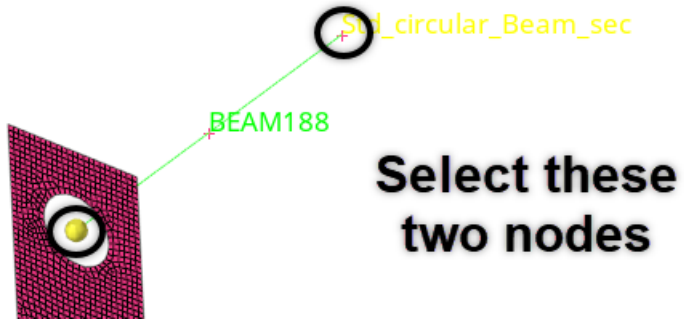


Figure 863:

HyperMesh creates a BEAM188 element.

6. Click **return** to exit the bar2 subpanel.

3.5.5 Create Coupled DOF - Rigid Elements

In this step, you will create coupled DOF and rigid elements.

1. From the menu bar, click **Mesh** > **Create** > **1D Elements** > **Rigids**.
   The Rigids panel opens.
2. Set the dependent selector to **multiple nodes**.
3. Select each of the **dof** checkboxes as indicated in Figure 864.

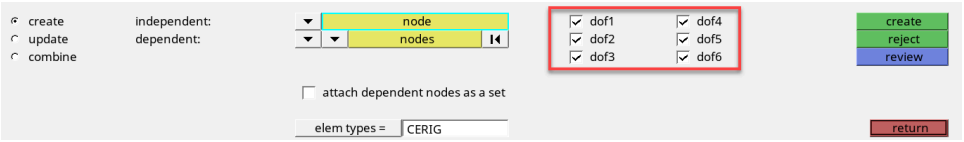


Figure 864:

4. Set elem types to **CERIG**.
5. In the Component folder of the Model Browser, right-click on **beam_asec** and select **Make Current** from the context menu.
6. Activate the independent selector set to **node**.
7. On Arbitary_Beam_Section, select the independent node at the end of the beam element as indicated in Figure 865.

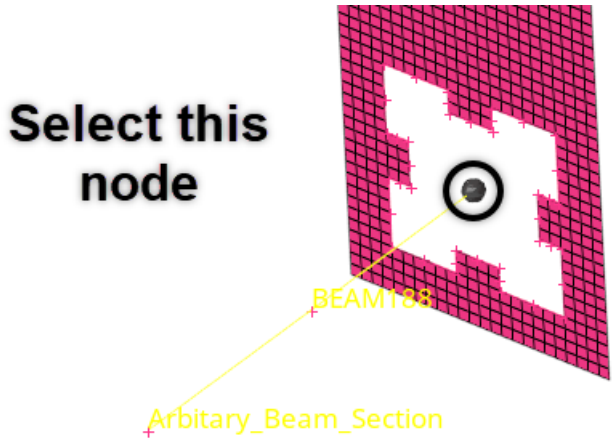


Figure 865:

8. Click **dependent: nodess** > **by path**.
9. On the plate component, select the dependent nodes indicated in Figure 866.

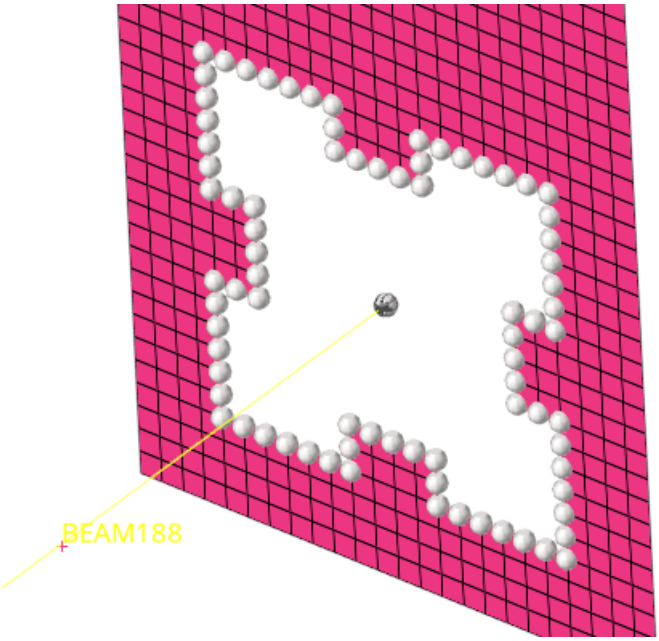


Figure 866:

10. Click **create**.

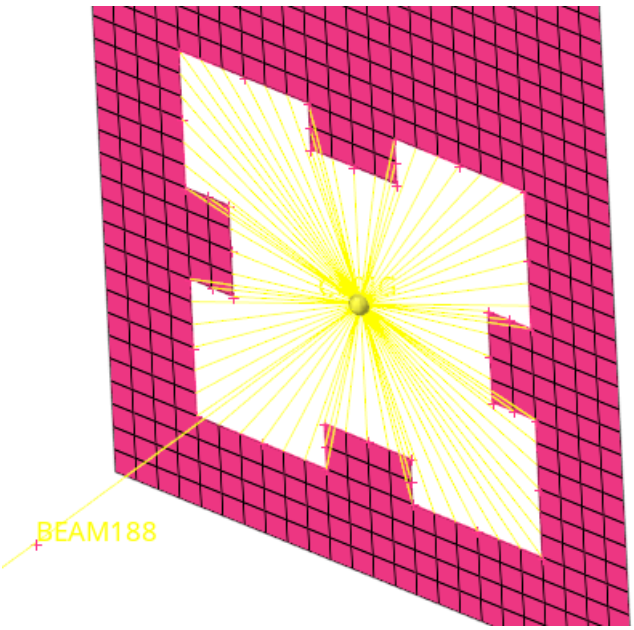


Figure 867:

11. In the Component folder of the Model Browser, right-click on **beam_std** and select **Make Current** from the context menu.

12. Activate the independent selector set to **node**.

13. On Std_circular_Beam_sec, select the independent node at the end of the beam element as indicated in Figure 868.

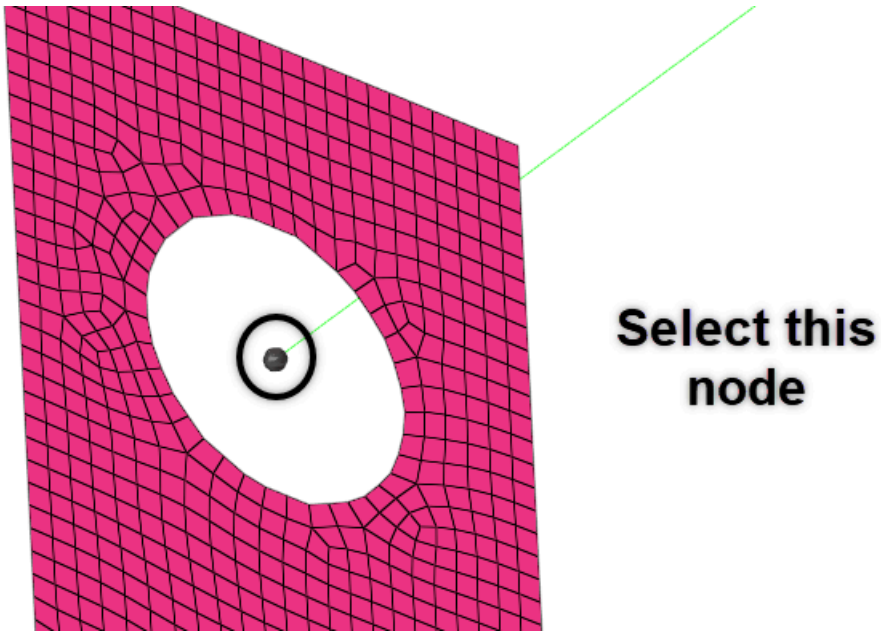


Figure 868:

14. Click **dependent: nodes** > **by path**.

15. On the plate component, select the dependent nodes indicated in Figure 869.

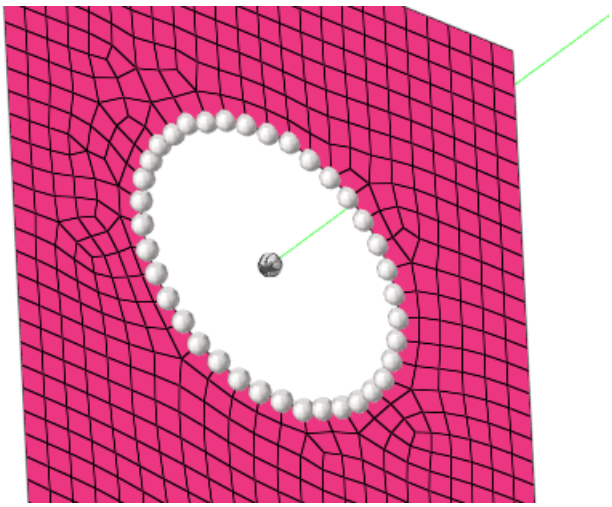


Figure 869:

16. Click **create**.

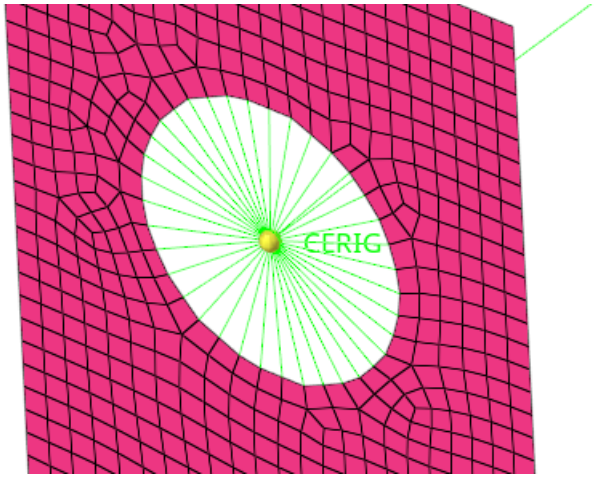


Figure 870:

17. Click **return** to exit the Rigids panel.

3.5.6 Create and Attach a Standard Circle Property

In this step, you will create and then attach a standard circle property to a component in HyperMesh.

1. In the Component folder of the Model Browser, click **beam_std**.
   The Entity Editor opens and displays the component's corresponding data.
2. Right-click on **property** and select **Create** from the context menu.
   The **Create Properties** dialog opens.
3. For Name, enter `Circular_Sec`.
4. Set SUBTYPE to **CSOLID**.
5. Under SECDATA, enter `15` for R.
6. Click **Close**.
   HyperMesh assigns the property Circular_Sec to the beam_std component.

3.5.7 Create an Arbitrary Property

In this step, you will create an arbitrary property in HyperMesh.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu.
   HyperMesh creates and opens a property in the Entity Editor.
2. For Name, enter `arbi_sec`.
3. Set card image to **SECTYPE**.
4. In the Component folder of the Model Browser, click next to beam_std, beam_asec, and Plate to turn off the display of their elements.

Note: You should only see geometric entities in the graphics area.

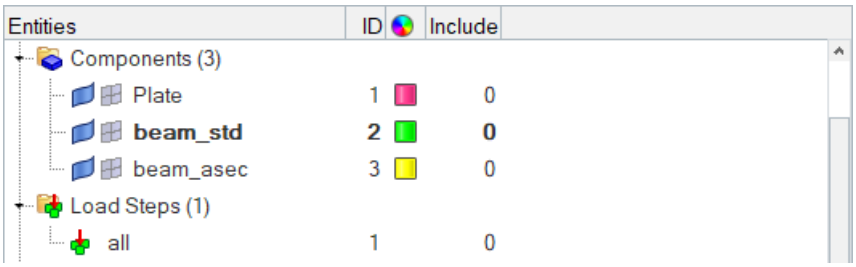


Figure 871:

5. From the menu bar, click **Properties** > **HyperBeam**.
   The HyperBeam panel opens.
6. Open the **solid section** subpanel.
7. Set the entity selector to **lines**.
8. Select all of the lines (displayed in pink as indicated in Figure 872) that form the Arbitrary_Beam_Section.

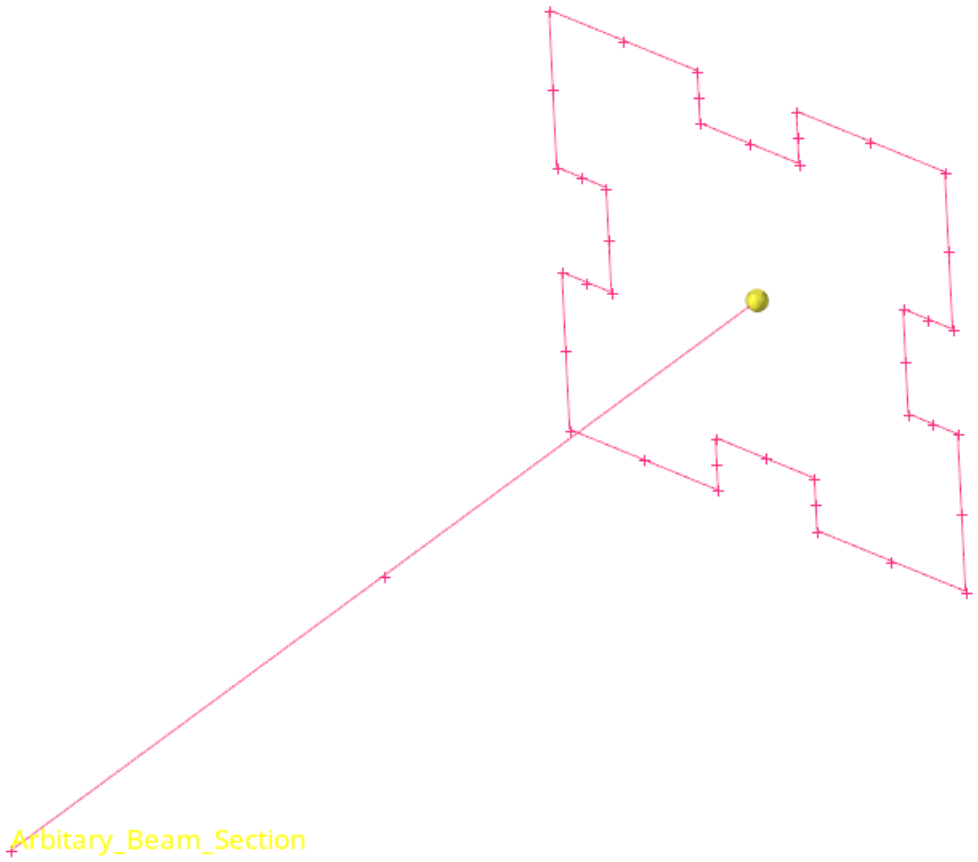


Figure 872:

9. Set section base node to **centroid**.

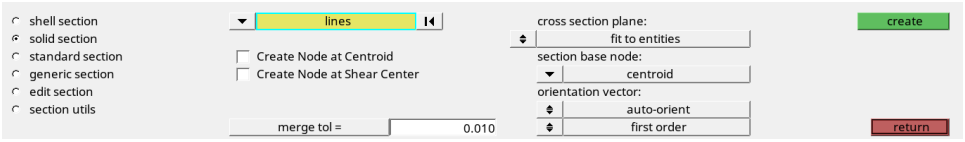


Figure 873:

10. Click **create**.
The **HyperBeam Data Model** dialog opens, meshes the area enclosed by the selected lines with quadrilateral elements, and calculates the properties using these elements.

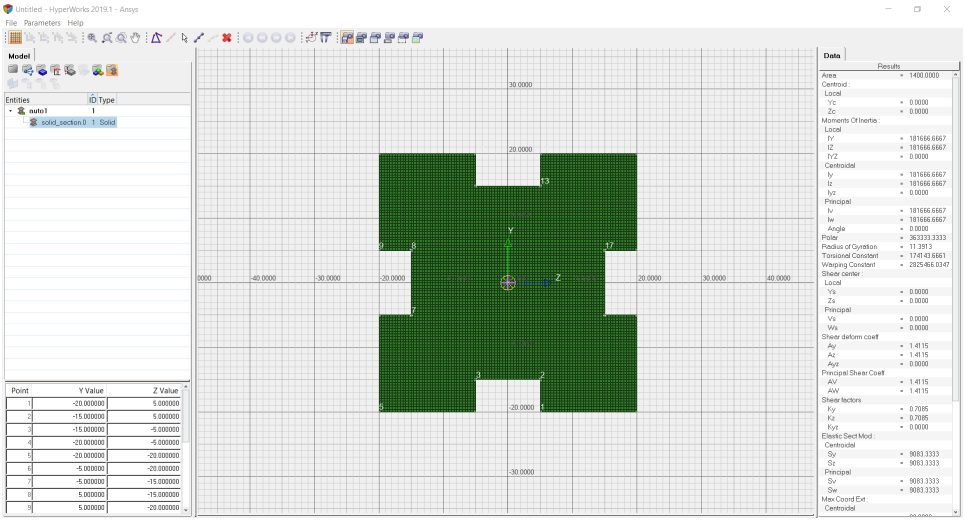


Figure 874:

11. In the Model Browser, select  .
12. In the Properties folder of the Model Browser, click **arbi_sec**.
    The Entity Editor opens and displays the beam section's corresponding data.
13. For Hyper beam section [OPTIONAL], click **Unspecified** > **Beamsection**.

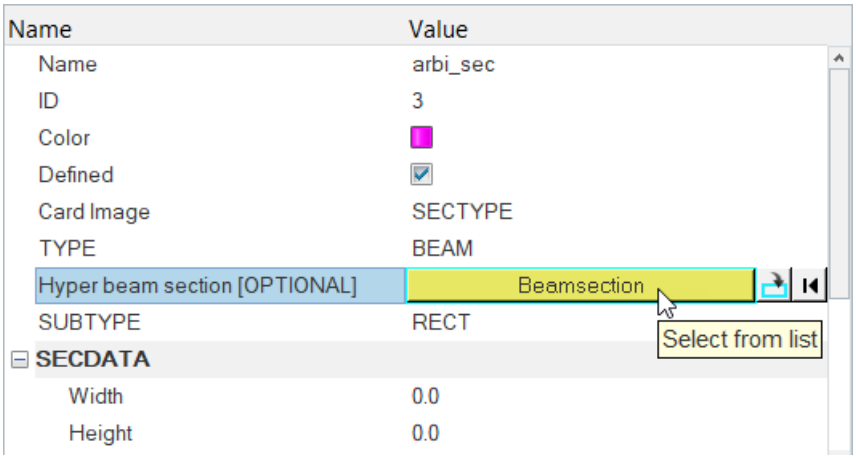


Figure 875:

14. In the **Select Beamsection** dialog, select **solid_section.0** and then click **OK**.

3.5.8 Update the Component Collector with the Beam Section

In this step, you will update the component collector with the beam section.

1. In the Component folder of the Model Browser, click **beam_asec**.
   The Entity Editor opens and displays the component's corresponding data.

2. For Property, click **Unspecified** > **Property**.
3. In the **Select Property** dialog, select **arbi_sec** and then click **OK**.
   HyperMesh attaches this property to the component.
4. In the Model Browser, click  to display all of the entities in the graphics area.
   All of the entities are displayed in the graphics area as indicated in Figure 876.

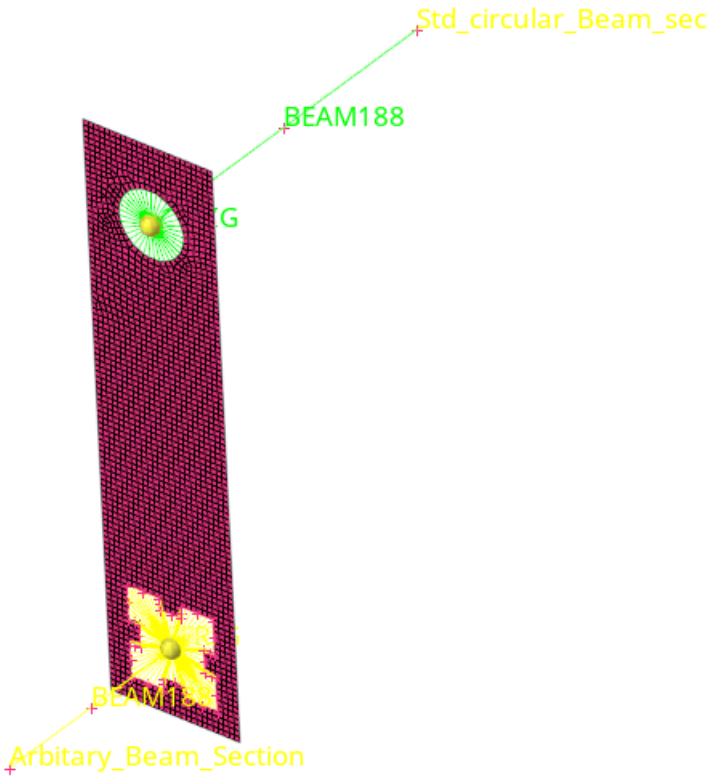


Figure 876:

3.5.9 Save your Work

In this step, you will save your work as a model in HyperMesh.

1. From the menu bar, click **File** > **Save As** > **Model**.
2. In the **Save Model As** dialog, navigate to your working directory and save your file.

3.5.10 Export the Deck to ANSYS *.cdb Format

In this step, you will export your model within HyperMesh.

1. Click **File** > **Export** > **Solver Deck** from the menu bar.

The Export tab opens.

2. Set File type to **Ansys**.

   Note: If you are in the ANSYS user profile, HyperMesh automatically sets the File type to Ansys and loads ANSYS as the default Template.

3. In the File field, navigate to your working directory and save the file as `4450_export.cdb`.
4. Click **Export**.

3.6 HM-4460: Composite

In this tutorial, you will: mesh all of the surfaces at once; define the dummy properties and assign them to the mesh; define an orientation for the component; use the Ply Realization and distribution table option; laminate realize; create and edit a distribution table; and use the Ply thickness visualization representation option.

Before you begin, copy the `pillar_w_ncf.CATPart` model file from `<hm.zip>/interfaces/ansys/` to your working directory.

3.6.1 Load the ANSYS User Profile

In this step, you will load the ANSYS user profile in HyperMesh.

A set of standard user profiles is included in the HyperMesh installation. They include: OptiStruct, Radioss, Abaqus, Actran, ANSYS, LS-DYNA, MADYMO, Nastran, PAM-CRASH, Permas, and CFD. When a user profile is loaded, applicable utility menus are loaded, unused panels are removed, unneeded entities are disabled in the Find, Mask, Card and Reorder panels and specific adaptations related to the Ansys solver are made.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Ansys**.

3.6.2 Load the Model

In this step, you will load the model file in HyperMesh

1. From the menu bar, click **File** > **Import** > **Geometry**.
2. In the Import tab, set File type to **CATIA**.
3. Click  .
4. In the **Select CATIA file** dialog, open the `pillar_w_ncf.CATPart` file.
5. Click **Import**.

   Note: You will import the Ply and Composite data later in this tutorial.

   HyperMesh imports geometry data only.

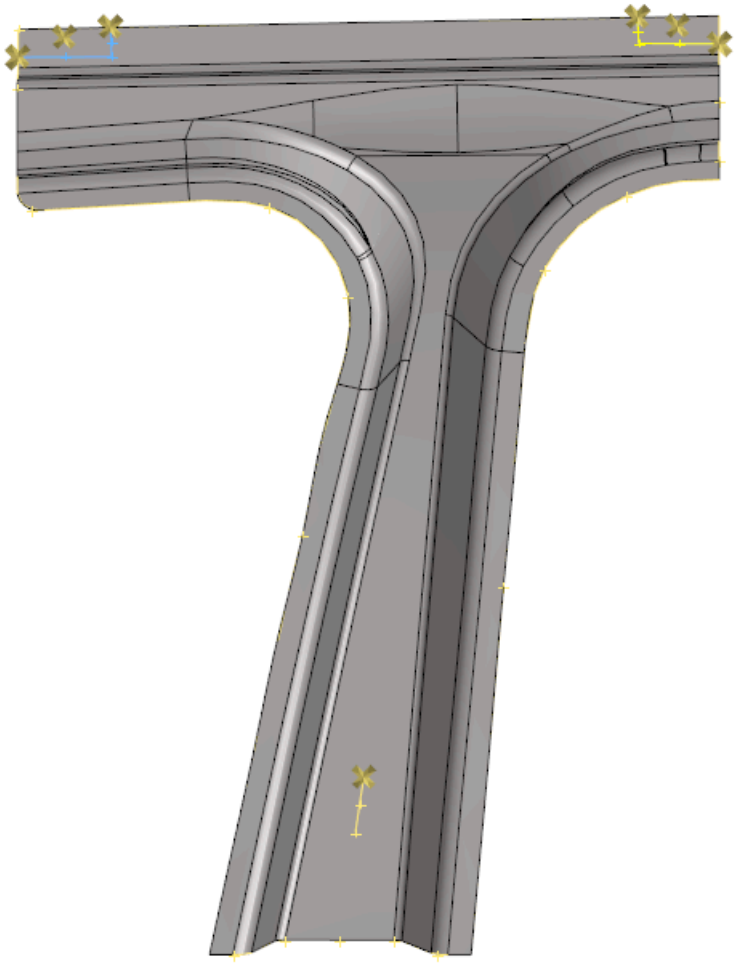


Figure 877:

6. In the Model Browser, review the model contents.

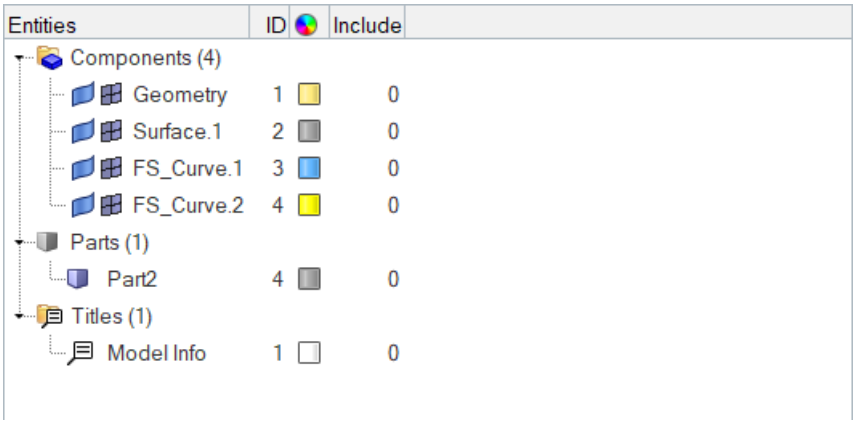


Figure 878:

3.6.3 Mesh all of the Surfaces at Once

In this step, you will mesh all of the model surfaces simultaneously while specifying element sizes and element types.

1. Open the Automesh panel by completing one of the following options:
   - From the menu bar, click **Mesh** > **Create** > **2D AutoMesh**.
   - From the main menu, open the **2D** page and click **automesh**.
   - Press **F12**.
2. Open the **size and bias** subpanel.
3. Set the entity selector to **surfs**.
4. Click **surfs** > **displayed**.
5. In the element size= field, enter `5`.
6. Set mesh type to **mixed**.
7. Set the mesh mode to **interactive**.
8. Set the elements to surf comp/elements to current comp toggle to **elems to current comp**.

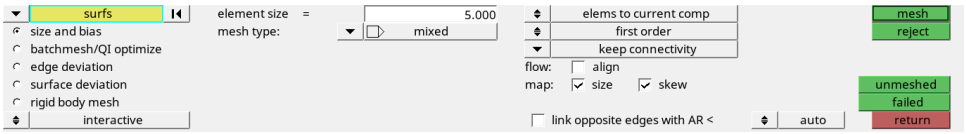


Figure 879:

9. Click **mesh**.

> **Note:** You should now be in the density subpanel of the meshing module. There is node seeding and a number on each surface edge. The number indicates the number of elements that were created along the edge.

The meshing module opens.

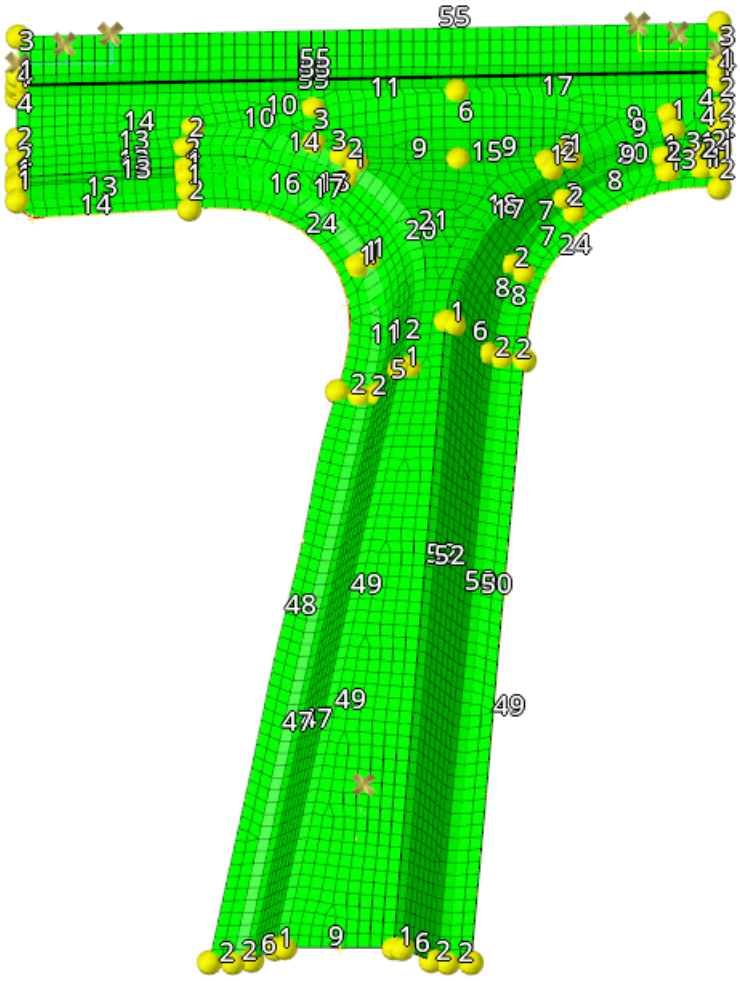


Figure 880:

10. Click **return**.
The mesh is accepted.

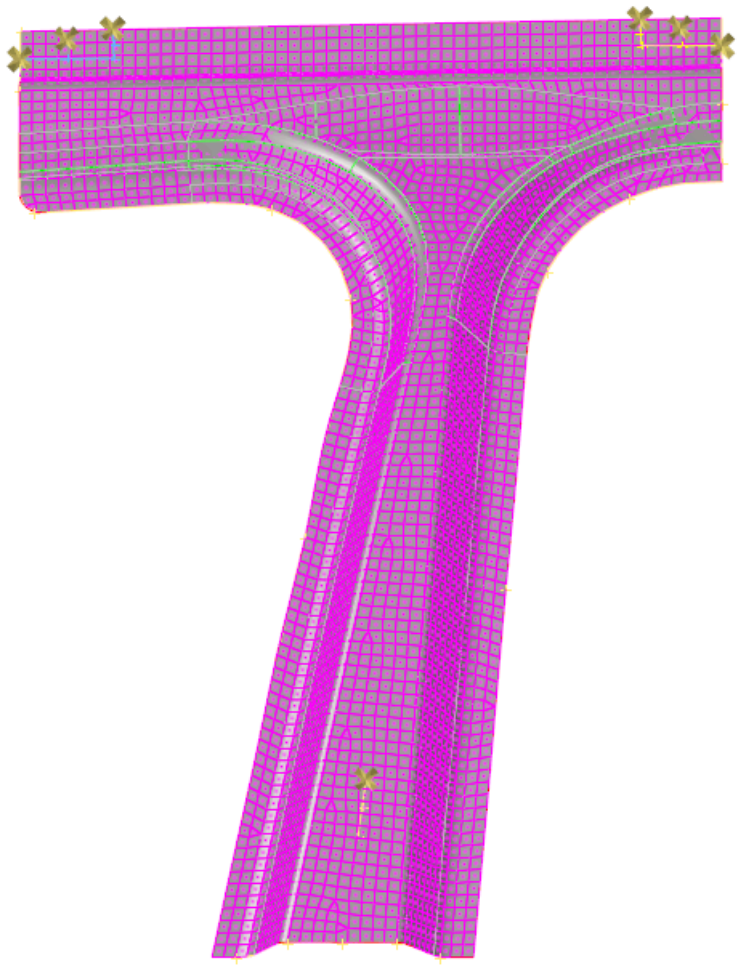


Figure 881:

11. From the menu bar, click **File** > **Save As** > **Model**.
12. In the **Save Model As** dialog, navigate to your working directory and save the HyperMesh database with the name `pillar_w_ncf_FINAL.hm`.

3.6.4 Load the Ply Information from FiberSim

In this step, you will load the ply information from FiberSim in HyperMesh.

1. From the menu bar, click **File** > **Import** > **Geometry**.
2. In the Import tab, set File type to **FiberSim**.
3. Click .
4. In the **Select FiberSim file** dialog, open the `pillar.h5` file.
5. Click **Import**.

HyperMesh imports and populates the database with laminate data (ply book and ply stacking data), composite material information, each ply data (triangular elements spanning a single ply), and a coordinate system.

6. In the Model Browser, review the model's contents.

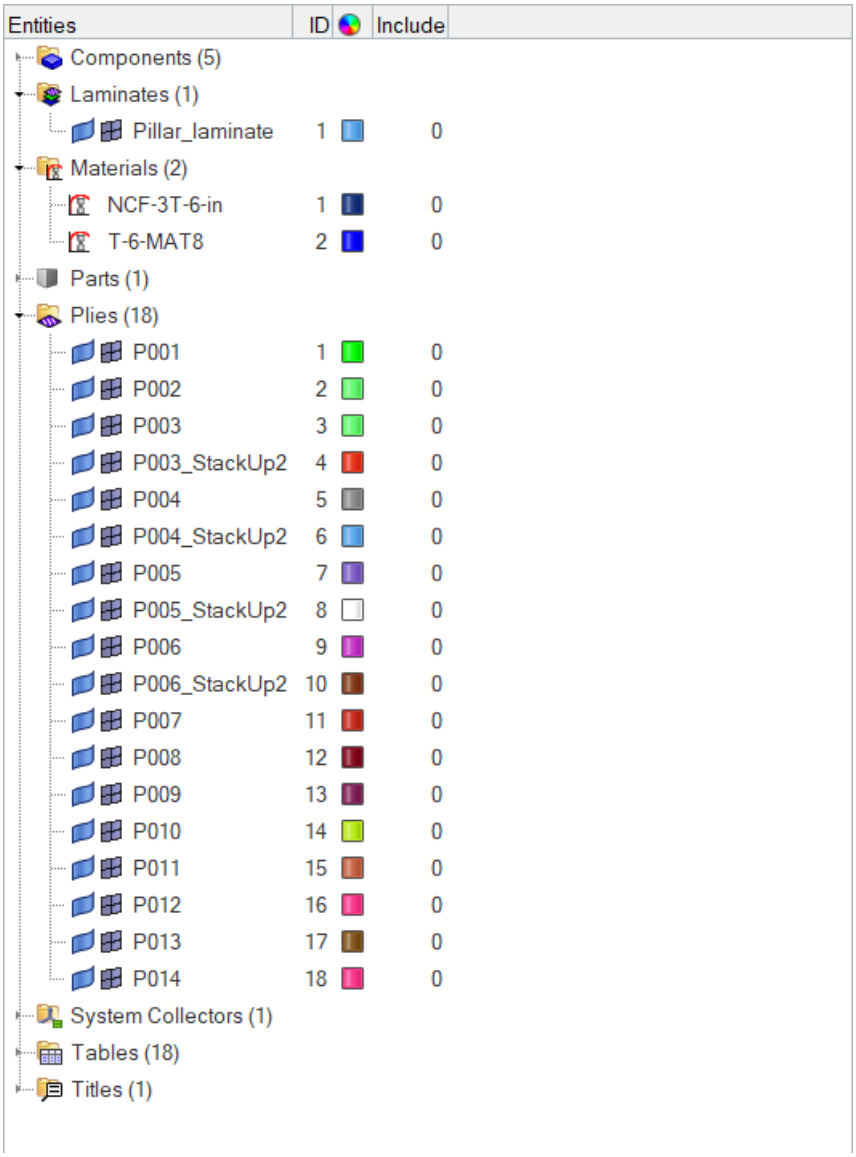


Figure 882:

7. Click  on the Visualization toolbar.
   Elements and feature lines become transparent.
8. In the panel area, click **comps**.
9. Select the **Geometry** and **auto1** components.
10. Click **select**.
11. Use the transparency slider, as seen in Figure 883, to review the system collector imported by the FiberSim model.

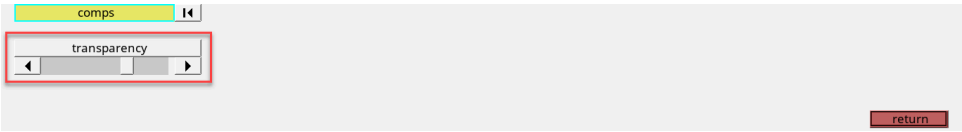


Figure 883:

12. Click **return**.

13. On the Visualization toolbar, click  to shade the elements and mesh lines.

14. On the Visualization toolbar, click  to shade the geometry and surface edges.

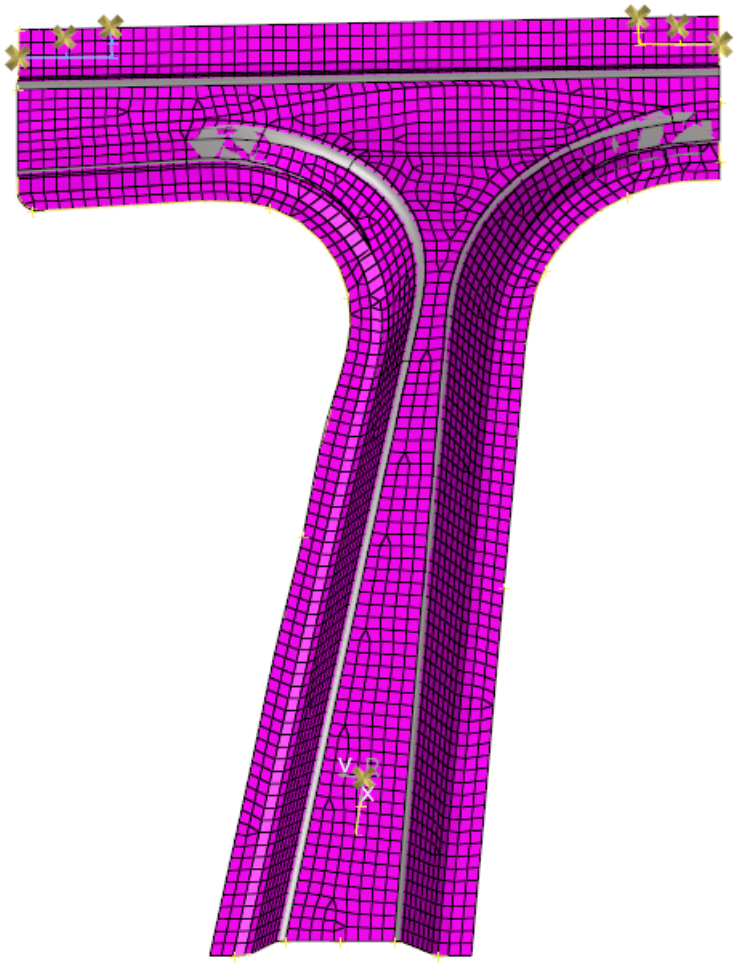


Figure 884:

15. In the Model Browser, turn off the display of geometry for all of the components.

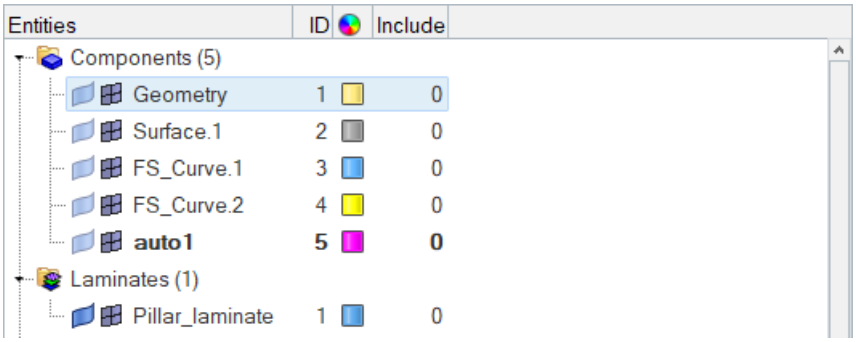


Figure 885:

3.6.5 Review and Edit the Element Normals

In this step, you will review and edit element normals in a model in HyperMesh.

1. From the menu bar, click **Mesh** > **Check** > **Elements** > **Normals**.

   Note: Element normals need to be changed to match the Z direction (red color).

   The Normals panel opens.
2. Set the first switch to **elems** as seen in Figure 886.



Figure 886:

3. Click **elems** > **displayed**.
4. Set the vector display/color display toggle to **color display**.
5. Click **display**.

   Note: The red side of the elements is the positive normal direction Z, while the blue side is the negative normal direction.

   HyperMesh displays, on each side of the part, the element normals using the colors red and blue.

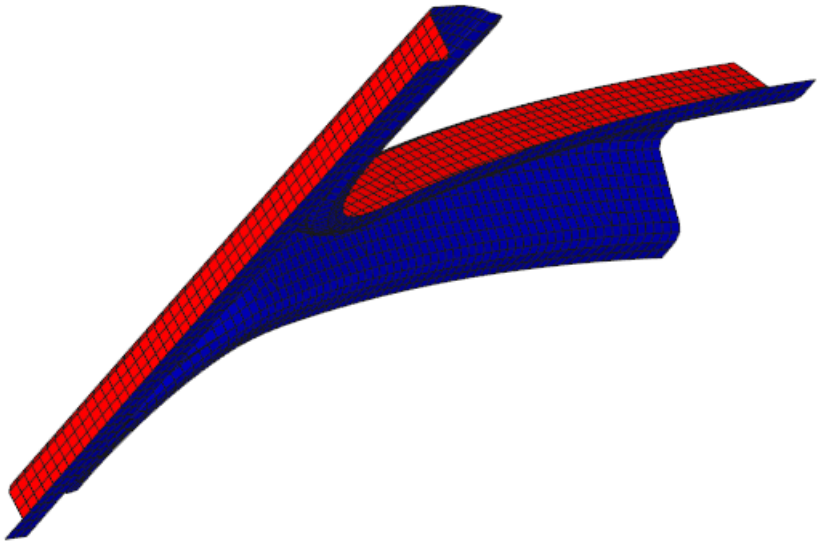


Figure 887:

6. Set orientation to **elem**.
7. Select one element that has the right direction.
   This element will be used as the model for the other elements.
8. Click **adjust**.
   All of the elements are set in the same normal direction.
9. Optional: If the blue color is in the Z direction, click **elems** > **displayed** and then click **reverse**.
   All of the elements are set in the right normal direction (red).

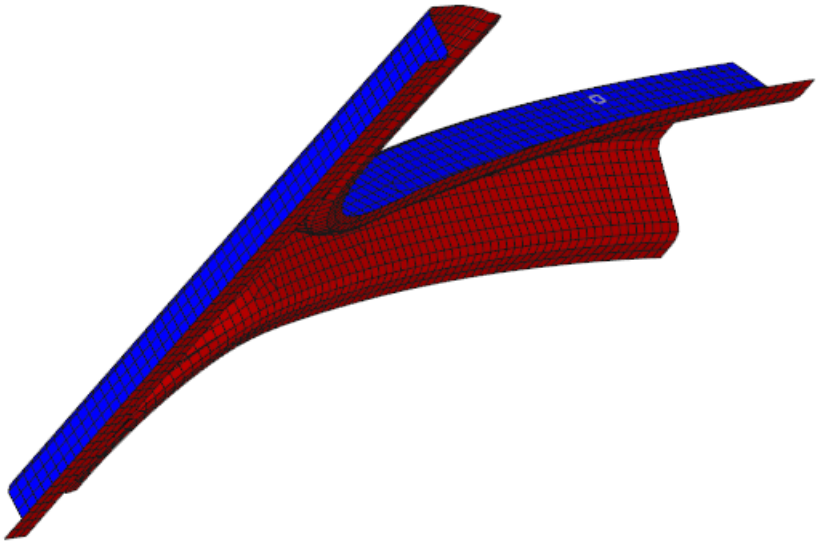


Figure 888:

10. Click **return** to exit the panel.

3.6.6 Realize Ply Geometry Shape

In this step, you will realize the ply geometry shape.

1. In the Model Browser, right-click on the **Plies** folder and select **Realize** from the context menu.
2. In the **Ply Realization** dialog, click **Component**.
3. In the panel area, click **comps** as seen in Figure 889.

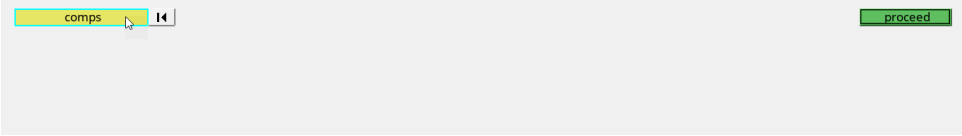


Figure 889:

4. Select the listed components as seen in Figure 890.

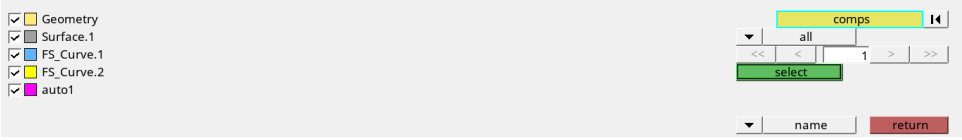


Figure 890:

5. Click **select**.

6. Click **proceed**.
7. Set Projection options to **FiberSim drape map by proximity method**.
8. Set Search Criterion to **Element centroid**.

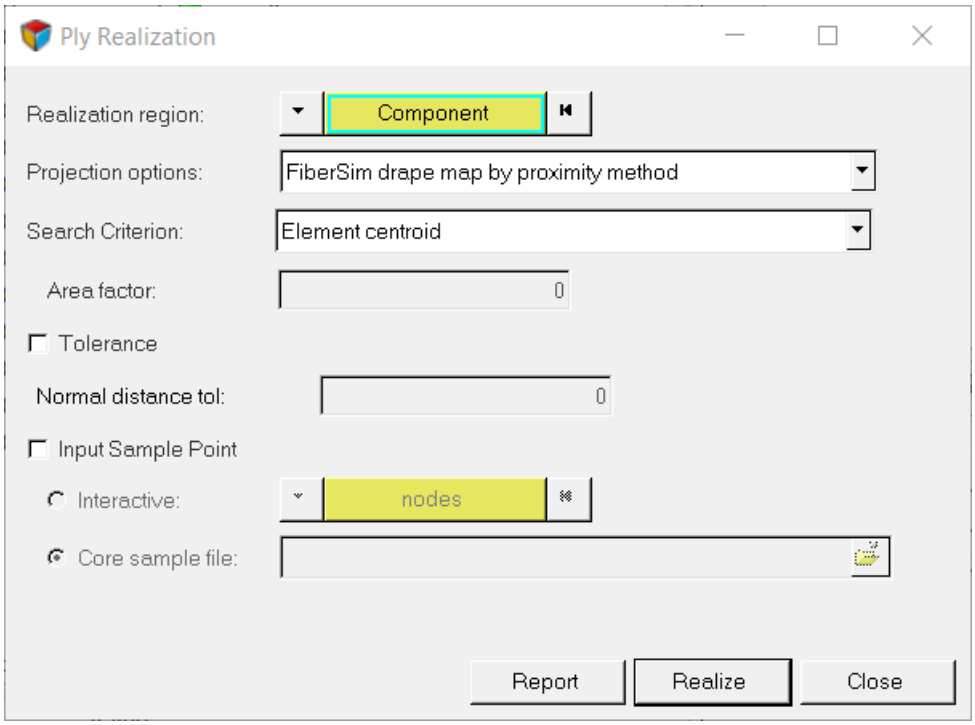


Figure 891:

9. Click **Realize**.

   This process takes each FiberSim Ply data and finds the FE elements which are bounded by the ply boundaries, and transfers the ply directions, draping data, and ply orientation into FE elements. This process also converts geometry plies into FE plies. At the end of realization, HyperMesh creates sets containing FE elements for each ply.

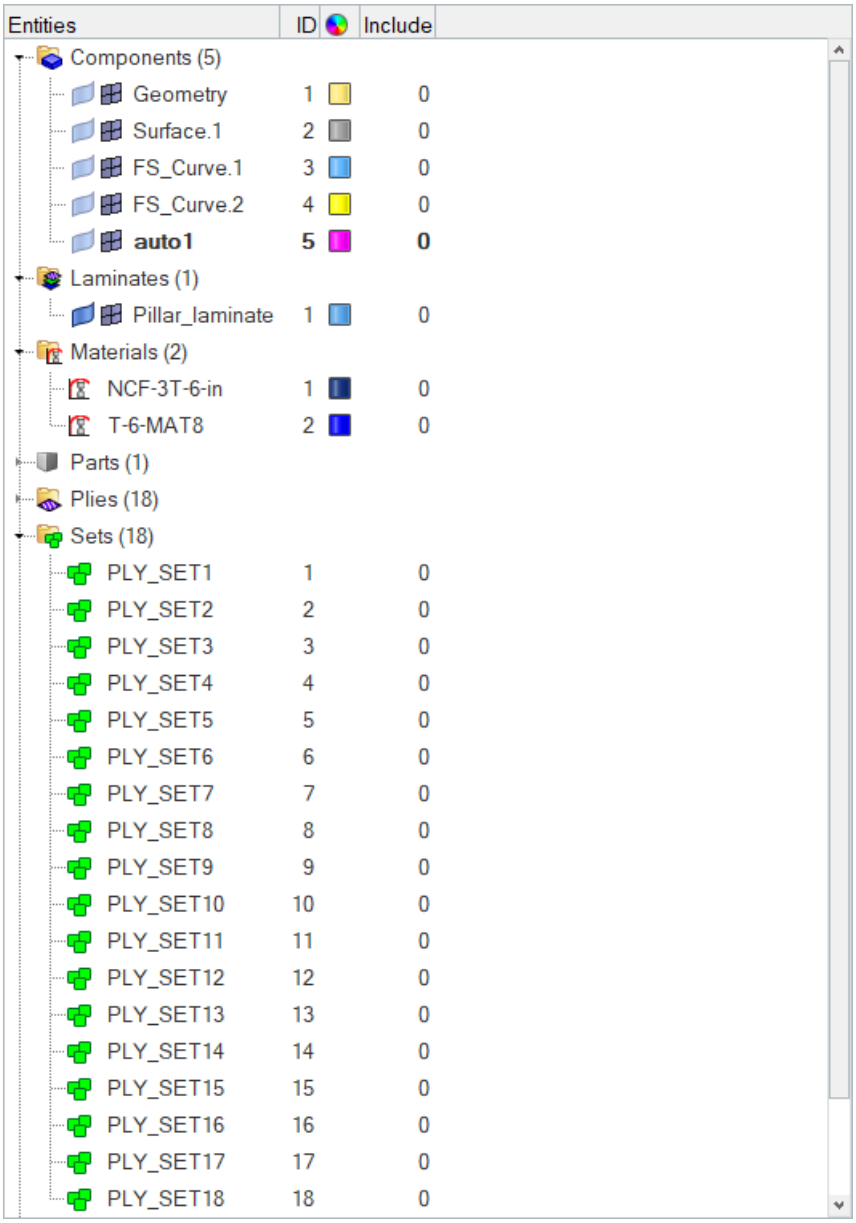


Figure 892:

3.6.7 Add an Element Type

In this step, you will add an element typer.

1. In the Model Browser, right-click and select **Create** > **Sensor** from the context menu. HyperMesh creates and opens a sensor (Et Type) in the Entity Editor.
2. In the Entity Editor, enter a name and ID, and select a color.
3. By default, Element Type is set to SHELL181.

SHELL181

Suitable for analyzing thin to moderately-thick shell structures. It is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.
Well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in a nonlinear analysis. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.
May be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory.

4. To simulate the element stiffness, set the stress stiffening option, extra displacement shapes, extra stress output, pressure loading, mass matrix, stress stiffness matrix, define the element coordinate system and specify the data storage using the respective keyopts, click Create/Edit.
5. Optional: If necessary, click **View** > **Browsers** > **HyperMesh** > **Solver** from the menu bar to open the Solver Browser.
6. In the Solver Browser, review the new element type.



Figure 893:

3.6.8 Update the Component with Element Type

In this step, you will update the componenet with the element type.

All elements in the model are in the Geometry component. In order for all elements in the Geometry component to be exported with the element type SHELL181, you must attach the element type defined in Add an Element Type to the Geometry component. The desired element type is SHELL181, which is why you were instructed to create SHELL181 elements in Add an Element Type. In this step, you will attach the sensor entity, with SHELL181 elements attached, to the Geometry component.

1. In the Component folder of the Model Browser, select **Geometry**.
   The Entity Editor opens and displays the component's corresponding data.
2. In the Entity Editor, set card image to **HM_COMP**.

3. For Type, click **Unspecified** > **Sensor**.
4. In the **Select Sensor** dialog, select **sensor1** and click **OK**.

3.6.9 Ply Visualization

In this step you will verify FE plies thickness and orientation in HyperMesh.

1. On the Visualization toolbar, set the element color mode to By Prop (visualize elements by property).
2. On the Visualization toolbar, set the layer representation mode to (Composite Layers with Fiber Direction).
3. In the Model Browser, **Hide** and **Show** each Ply.

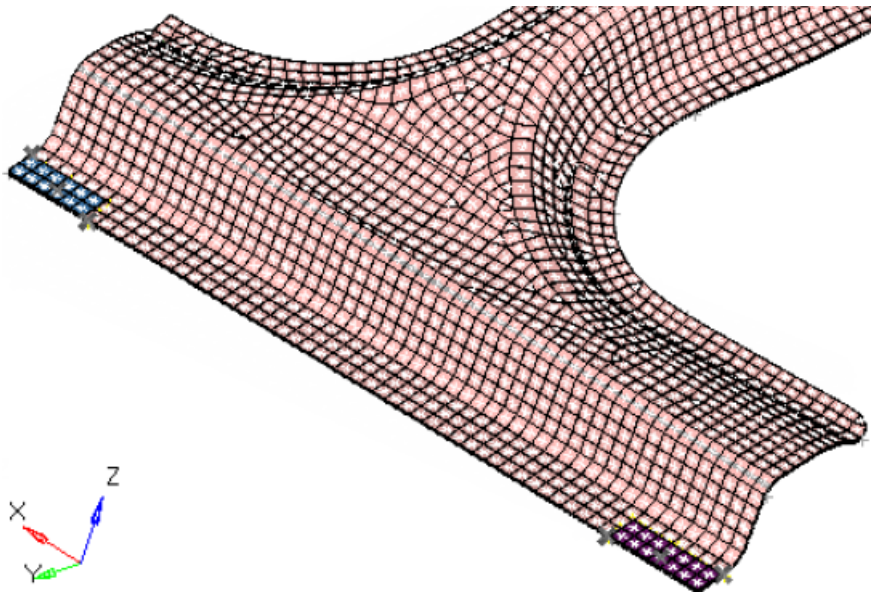


Figure 894:

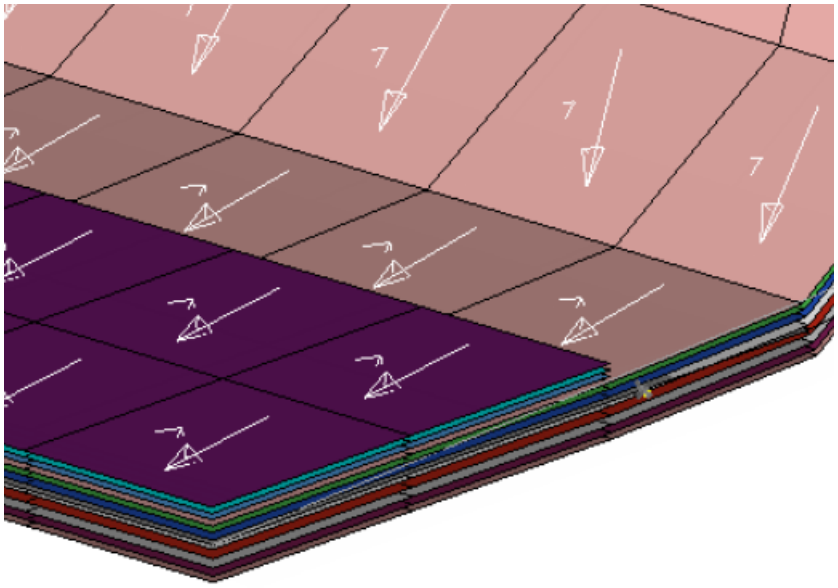


Figure 895:

4. On the Visualization toolbar, set the element representation mode to (2D Detailed Element Representation).

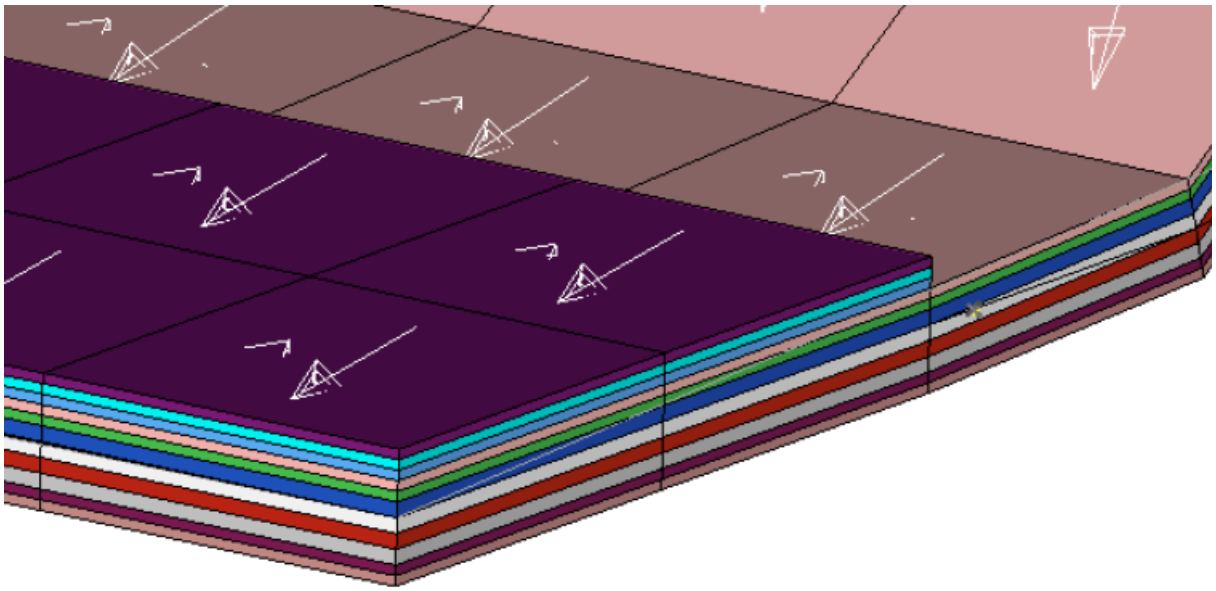


Figure 896:

3.6.10 Laminate Realize the Ply Based Model

In this step, you will laminate realize the ply based model.

1. In the Model Browser, right-click on the **Laminate folder** and select **Realize** from the context menu.
2. In the **Laminate Realize** dialog, accept the default settings and click **Realize**.

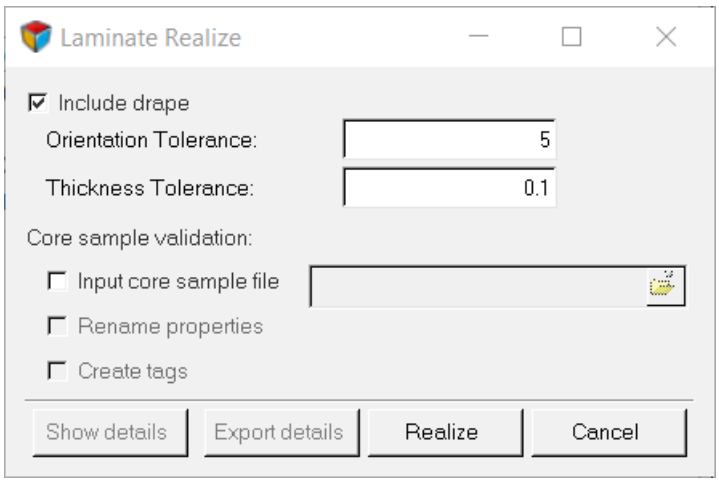


Figure 897:

HyperMesh creates a property for each stack, and assigns it to a component.

3. On the Visualization toolbar, set the element color mode to By Comp.

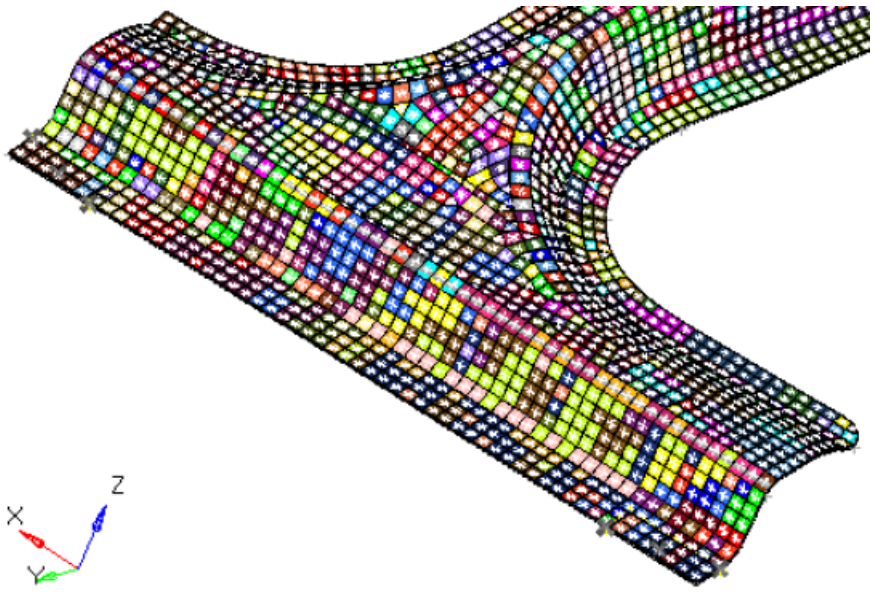


Figure 898:

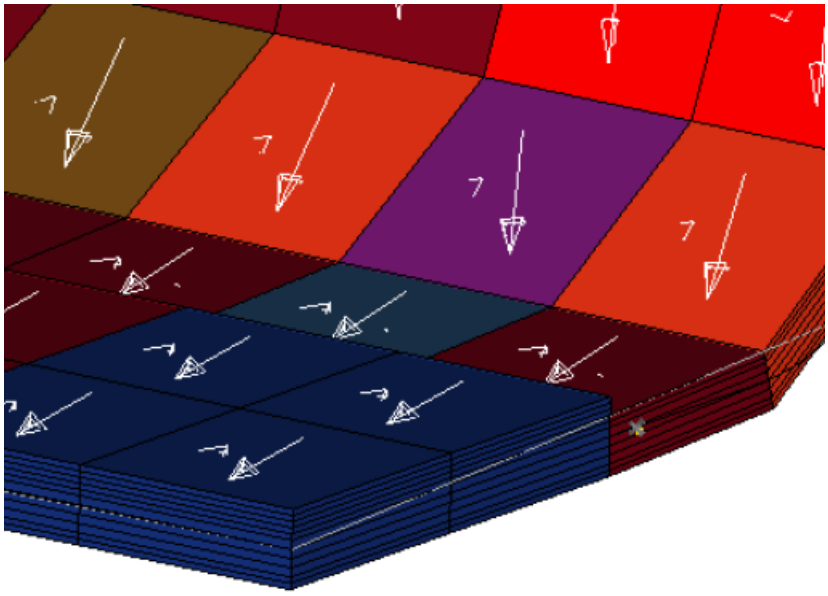


Figure 899:

4. In the Sections folder of the Model Browser, locate the shell sections.
5. Click on a section.
   The corresponding details for the section display in the Entity Editor.
6. In the Entity Editor, under PLIES, next to Data: TK, click .
7. In the **PLIES** dialog, review the number of plies, thickness, orientation, and material data.
8. Click **Close**.

3.6.11 Export the Deck

In this step, you will export the deck to ANSYS *.cdb format.

1. Click **File** > **Export** > **Solver Deck** from the menu bar.
   The Export tab opens.
2. Set File type to **Ansys**.
   If you are in the ANSYS user profile, HyperMesh automatically sets the File type to Ansys and loads ANSYS as the default Template.
3. In the File field, navigate to your working directory and save the file as `pillar_w_ncf-FINALhm.cdb`.
4. Click **Export**.

Feko

4

Feko tutorials.

This chapter covers the following:

- 4.1 HM-5000: Importing and Editing a Feko Mesh (p. 685)
- 4.2 HM-5010: Assigning EM Material Properties to a Mesh forFeko (p. 686)

4.1 HM-5000: Importing and Editing a Feko Mesh

In this tutorial you will import and edit a Feko mesh.

4.1.1 Exercise 1: Importing and Editing a Feko Mesh

In this step you will import and edit the Feko user profile.

1. In]HyperMesh, click **Preferences** > **User Profiles** and select the Feko profile.
2. Click **Import Solver Deck** ( ).
3. Click **Open Model** ( ) and browse to select the .fhm file.
   The .fhm file containing the Feko mesh elements (along with an automatically generated `INC` file containing the material data) can be exported from CADFEKO.
4. Click **Import**.
   Each of the mesh elements associated with each CADFEKO Label (representing a Wire, Face or Region) will import as a separate Component. CADFEKO media definitions will import as Materials. Properties that associate each material with a component will be generated.
5. Modify and edit the mesh elements, materials, and properties as required.
6. Click **Export Solver Deck** ( ).
   A new `INC` file is created automatically.
7. Click **Import** and import the .fhm file into CADFEKO (or EDITFEKO).
   If an `INC` file with the same name is available, it is imported automatically.

4.2 HM-5010: Assigning EM Material Properties to a Mesh forFeko

In this tutorial you will assign EM material properties to a mesh for Feko

4.2.1 Exercise 1: Assigning EM Material Properties to a Mesh for Feko

In this exercise you will assign EM material properties to a mesh.

1. In]HyperMesh, click **Preferences** > **User Profiles** and select the Feko profile.
2. Create or sort the mesh elements into components. Each component should only have on type of mesh element.
3. Create the required materials. You can create anisotropic dielectric media (with optional magnetic properties) and metallic media.
4. Create the required properties. A property maps materials onto an element type. You can create:
   - Triangle, which requires "Front medium," "Back medium," and an optional "Face medium" with thickness value required when the face is set to be a metallic material.
   - Segment, with required materials for "Core medium" and "Surrounding medium." A "Radius" value must be supplied.
   - Tetrahedron, which requires only a "Volume medium" material.
5. Assign the properties to the components to map the material settings to the elements in each component.
6. Export the mesh as an FHM file. An `INC` file with the same file name is created automatically.
7. Import the FHM file into CADFEKO (or EDITFEKO). If an `INC` file with the same file name is available, it is imported automatically.

LS-DYNA

5

LS-DYNA tutorials.

This chapter covers the following:

- 5.1 HM-4605: Define LS-DYNA Model and Load Data, Controls, and Output (p. 688)
- 5.2 HM-4610: Use Curves, Beams, Rigid Bodies Joints, and Loads in LS-DYNA (p. 711)
- 5.3 HM-4615: Model Import, Airbags, Export Displayed, and Contacts with LS-DYNA (p. 744)
- 5.4 HM-4620: Create Rigid Walls, Model Data, Constraints, Cross Sections, and Output with LS-DYNA (p. 769)
- 5.5 HM-4625: Assemblies Using DYNA (p. 796)
- 5.6 HM-4630: Interfacing for Arbitrary - Lagrangian - Eulerian Capability Using DYNA (p. 834)
- 5.7 HM-4635: Create Joints and Check Minimum Time Step (p. 843)

5.1 HM-4605: Define LS-DYNA Model and Load Data, Controls, and Output

In this tutorial, you will: view LS-DYNA keywords in HyperMesh; understand part, material, and section creation and element organization; create sets and velocities; understand the relation of LS-DYNA entity type to HyperMesh element and load configurations; create nodal single point constraints and contacts with set segment ID; define output and termination; and export models to LS-DYNA formatted input files.

Before you begin, copy `head_start.hm`, `head_2.hm` and `head_3.hm` from `<hm.zip>/interfaces/lsdyna/` to your working directory.

The following tools/utilities set the foundation for setting up an LS-DYNA input deck with HyperMesh:

- LS-DYNA FE input translator
- FE output template
- LS-DYNA Utility menu
- LS-DYNA User Profile

5.1.1 Exercise 1: Define Model Data for the Head and A-Pillar Impact Analysis

In this exercise, you will become familiar with defining LS-DYNA materials, sections, and parts in HyperMesh. Also, you will set up model data for a LS-DYNA analysis of a hybrid III dummy head impacting on A-pillar.

The head and A-pillar are depicted in Figure 900.

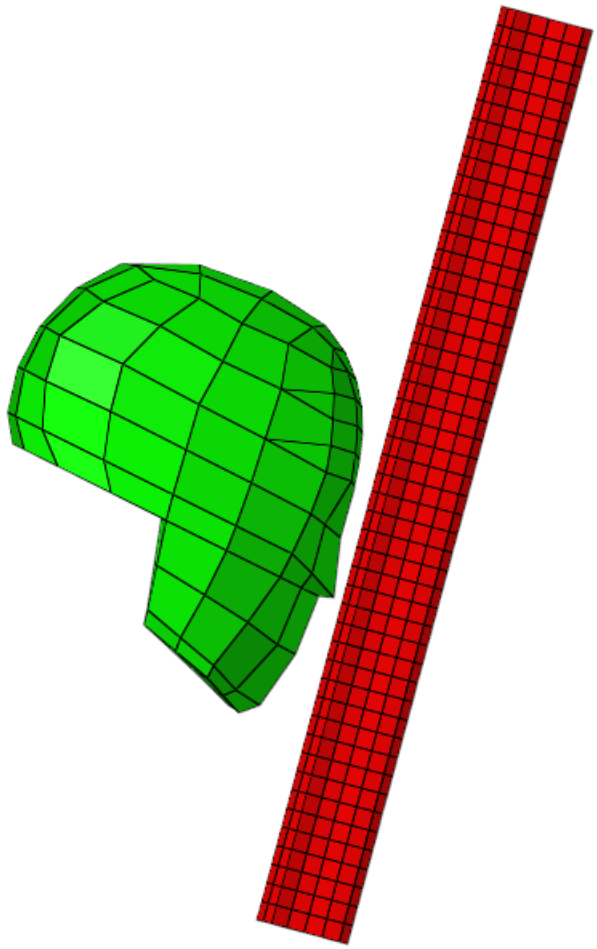


Figure 900:

Load the LS-DYNA User Profile

In this step, you will load the LS-DYNA user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Retrieve the HyperMesh File

In this step, you will open the HyperMesh model file.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
2. In the **Open Model** dialog, open the `head_start.hm` file.

The model appears in the graphics area.

Define the *MAT_ELASTIC Material

In this step, you will define a new material for the A-pillar and head.

1. In the Solver Browser, right-click and select **Create** > ***MAT** > **MAT(1-50)** > **1 - *MAT_ELASTIC** from the context menu.

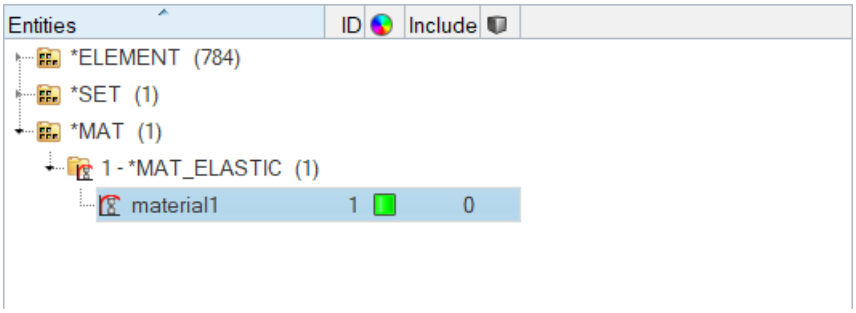


Figure 901:

HyperMesh creates and opens a new material in the Entity Editor.

2. For Name, enter `ELASTIC`.
3. For Rho (Density), enter `1.2E-6`.
4. For E (Young's modulus), enter `210`.
5. For PR (Poisson's ratio), enter `0.26`.

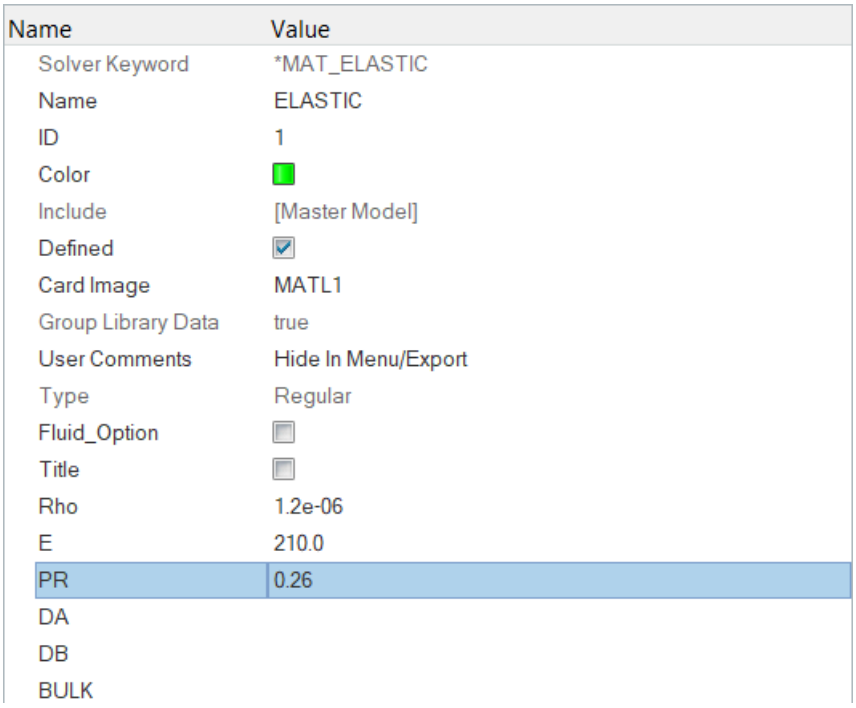


Figure 902:

Define *SECTION_SHELL

In this step, you will define a property with a 3.5 mm thickness for the A-pillar.

1. In the Solver Browser, right-click and select **Create** > ***SECTION** > ***SECTION_SHELL** from the context menu.
   HyperMesh creates and opens a new property in the Entity Editor.
2. For Name, enter `section3.5`.
3. For card image, select **SectShll**.
4. Expand NonUniformThickness, and enter `3.5` for T1.

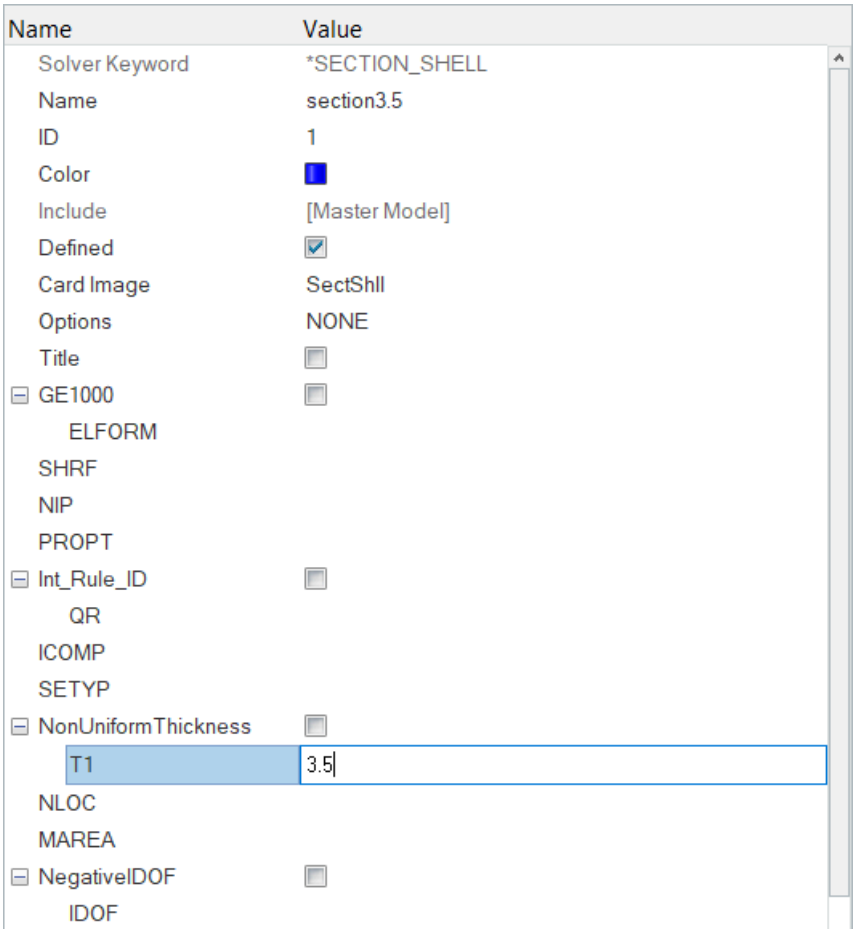


Figure 903:

Define *SECTION_SOLID for the Head

In this step, you will define *SECTION_SOLID for the head of the model.

1. In the Solver Browser, right-click and select **Create** > ***SECTION** > ***SECTION_SOLID** from the context menu.
   HyperMesh creates and opens a new property in the Entity Editor.
2. For Name, enter `solid`.

Define *PART for the A-pillar

In this step, you will define the *PART for the A-pillar in the model.

MAT_ELASTIC is the material collector named "ELASTIC." *SECTION_SHELL is the property collector named "section3.5."

1. In the Component folder of the Model Browser, click **pillar**.

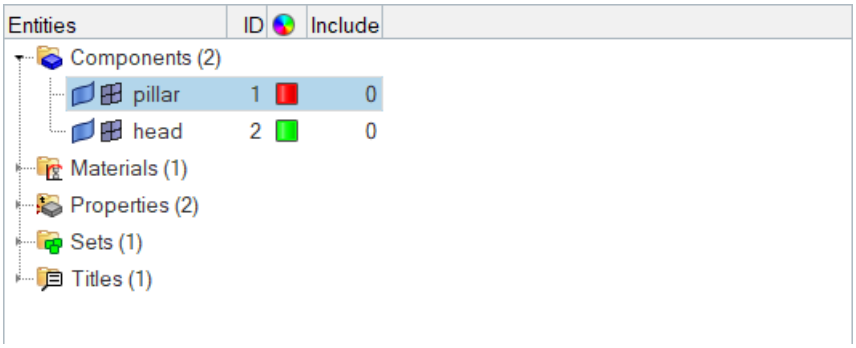


Figure 904:

The Entity Editor opens, and displays the component's corresponding data.

2. Set card image to **Part**.
3. For Material, click **Unspecified** > **Material**.

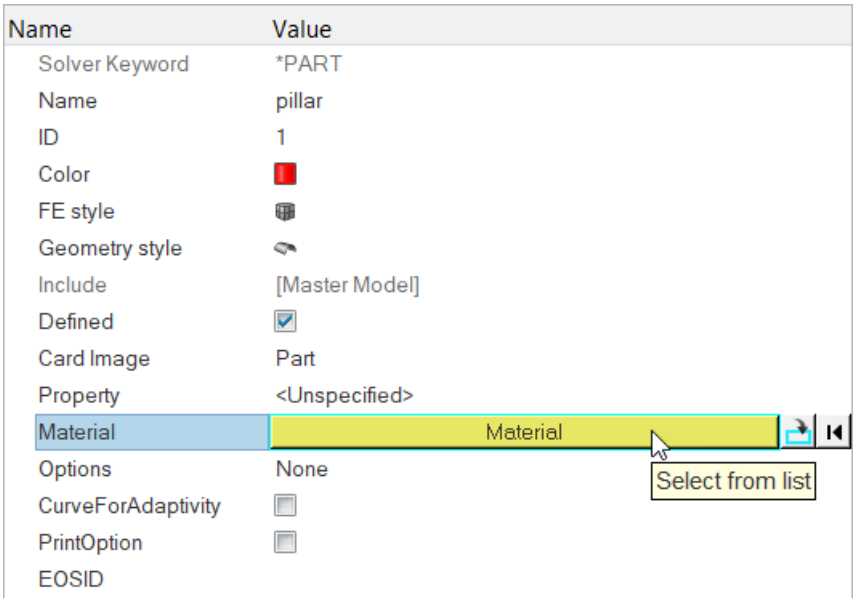


Figure 905:

4. In the **Select Material** dialog, select **ELASTIC** and then click **OK**.

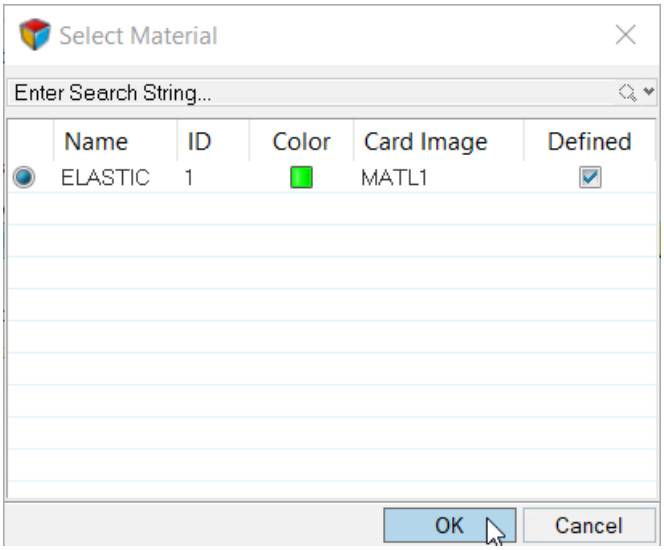


Figure 906:

HyperMesh assigns the material Elastic to the component pillar.

5. For Property, click **Unspecified** > **Property**.
6. In the **Select Property** dialog, select **section3.5** and then click **OK**.
   HyperMesh assigns the property section3.5 to the pillar component.

Define *PART for the Head

In this step, you will define the *PART for the head of the model.

*MAT_ELASTIC is the material collector named "ELASTIC." *SECTION_SOLID is the property collector named "solid."

1. In the Component folder of the Model Browser, click **head**.

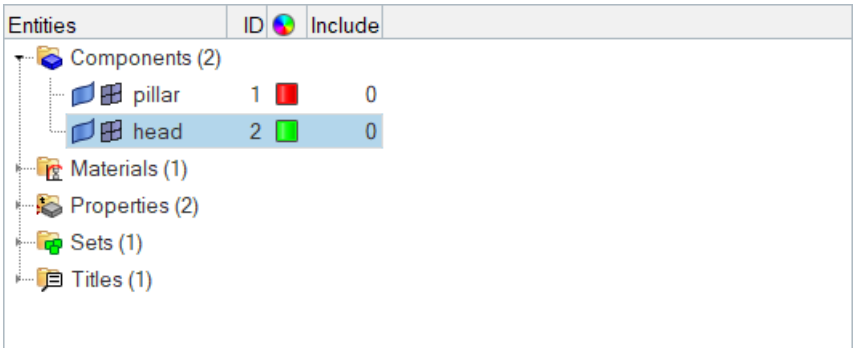


Figure 907:

The Entity Editor opens, and displays the component's corresponding data.

2. Set card image to **Part**.
3. Set Material to **ELASTIC**.

4. Set Property to **solid**.

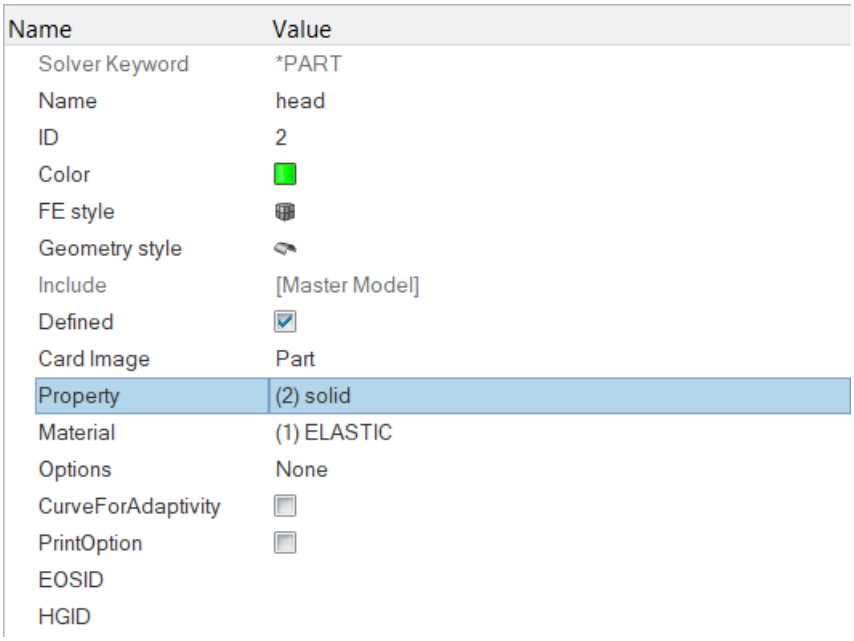


Figure 908:

Save Your Work

In this step, you will save your work.

Save your work as a HyperMesh file named `head_2.hm`.

5.1.2 Exercise 2: Define Boundary Conditions and Loads for the Head and A-Pillar Impact Analysis

In this exercise, you will define LS-DYNA boundary conditions, loads, and contacts in HyperMesh. Also, you will set up the boundary conditions and load data for a LS-DYNA analysis of a hybrid III dummy head impacting an A-pillar.

The head and A-pillar are depicted in Figure 964.

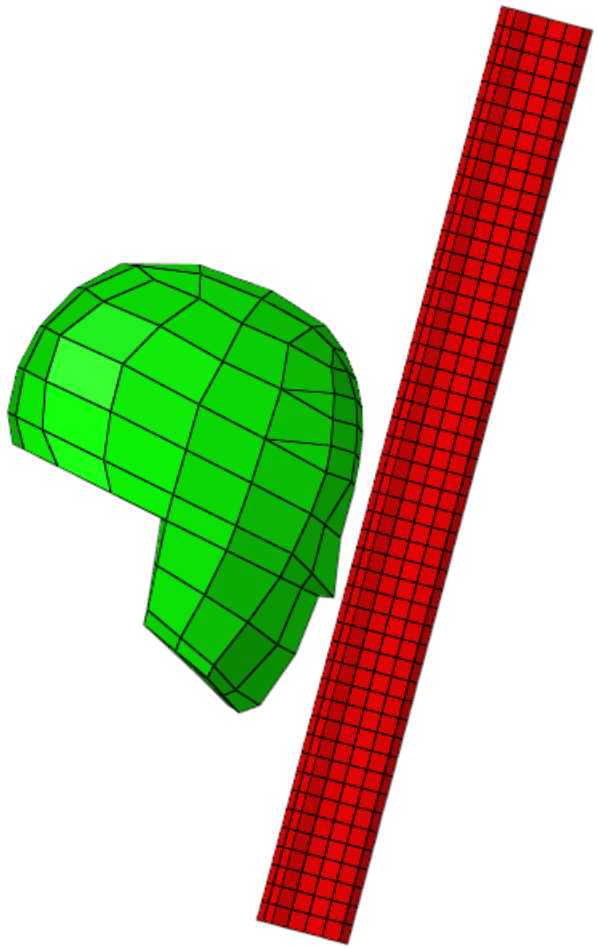


Figure 909:

Load the LS-DYNA User Profile

In this step, you will load the LS-DYNA user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Retrieve the HyperMesh File

In this step, you will open the HyperMesh model file.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
2. In the **Open Model** dialog, open the `head_2.hm` file.

The model appears in the graphics area.

3. Observe the model using various visual options available in HyperMesh, such as rotation and zooming.

Create a Node Set

In this step, you will create a node set, *SET_NODE_LIST, which will contain all of the nodes in the head component of the model.

1. In the Solver Browser, right-click and select **Create** > ***SET** > ***SET_NODE** > ***SET_NODE_LIST** from the context menu.

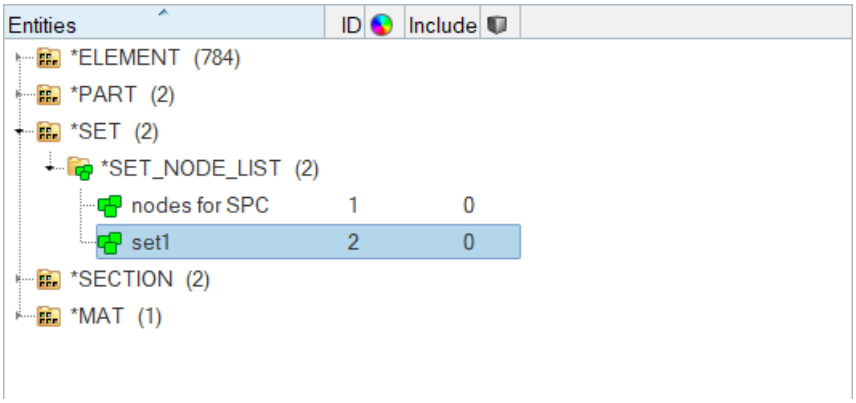


Figure 910:

HyperMesh creates and opens a new set in the Entity Editor.

2. For Name, enter `Vel_Nodes`.
3. For Entity IDs, click **0 Nodes** > **Nodes**.

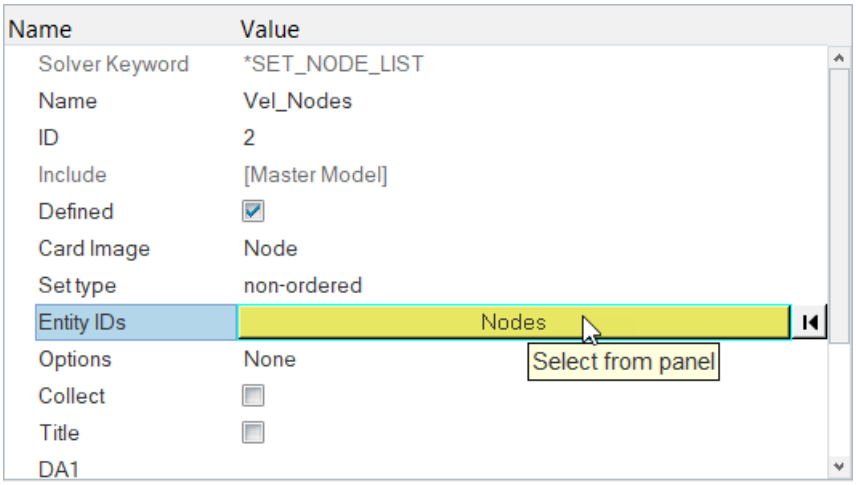


Figure 911:

4. In the panel area, click **nodes** > **by collector**.
5. Select the **head** component checkbox as seen in Figure 912.

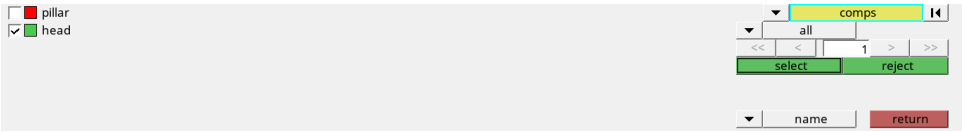


Figure 912:

6. Click **select**.
7. Click **proceed**.

Define the Velocity

In this step, you will define the velocity for the model.

1. In the Solver Browser, right-click and select **Create** > ***INITIAL** > ***INITIAL_VELOCITY** from the context menu.

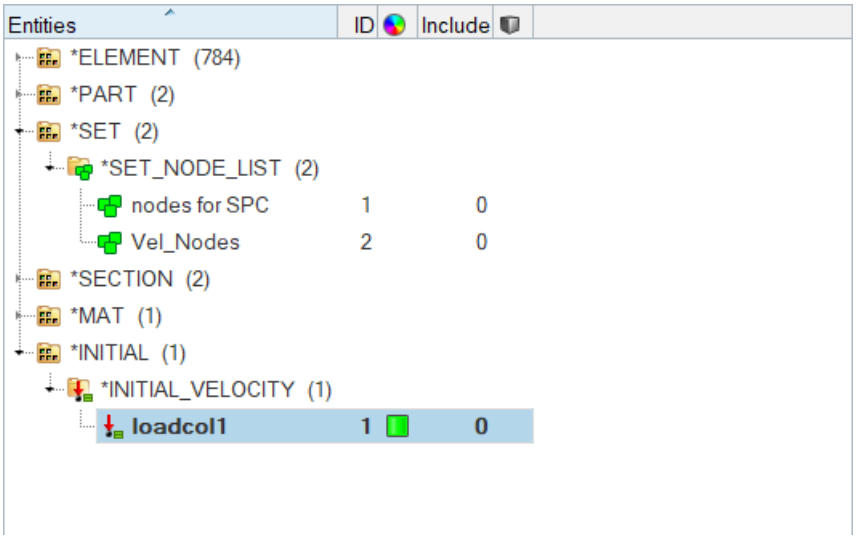


Figure 913:

HyperMesh creates and opens a new load collector in the Entity Editor.

2. For Name, enter `init_vel`.
3. Click `NSID` (Node Set ID), and then select **Set**.
4. In the **Select Set** dialog, select **Vel_Nodes** and then click **OK**.
5. For VX (x-component of mass center of velocity), enter `5`.

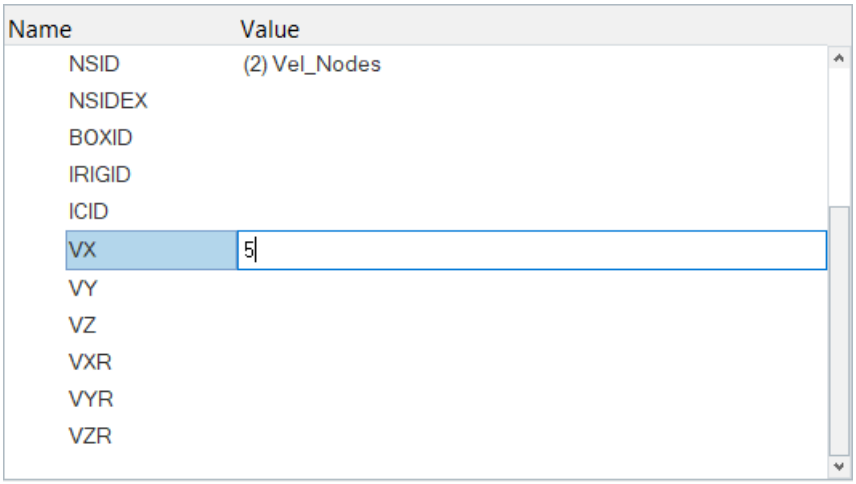


Figure 914:

Create a Load Collector

In this step, you will create a load collector for the constraints to be created.

1. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.
   HyperMesh creates and opens a new load collector in the Entity Editor.
2. For Name, enter `SPC`.
3. Optional: Click the **Color** icon, and select a color to display the load collector.
4. Set card image to **<None>**.

Create Constraints on the Pillar's End Nodes

In this step, you will create constraints on the ends of the pillar component of the model.

1. In the Solver Browser, right-click and select **Create** > ***BOUNDARY** > ***BOUNDARY_SPC_NODE** from the context menu.
   The Constraints panel opens.
2. In the panel area, set the entity selector to **nodes**.
3. Click **nodes** > **by sets**.
4. Select the **nodes for SPC** checkbox as seen in Figure 915.

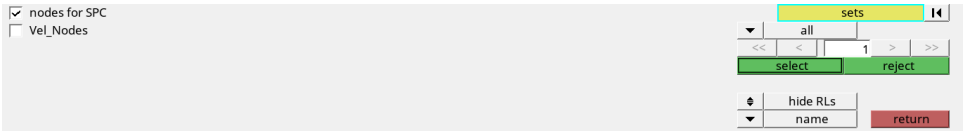


Figure 915:

5. Click **select**.
   HyperMesh selects the nodes on both ends of the pillar.

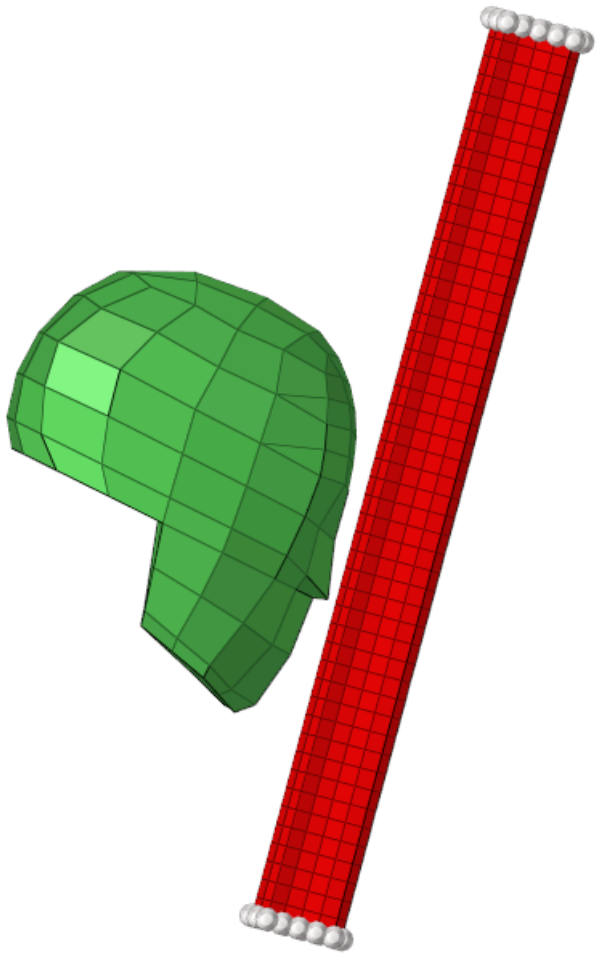


Figure 916:

6. Verify all six **dof** (degree of freedom) checkboxes are selected.
7. Verify the load type is set to **BoundSPC**.
8. Select the **label constraints** checkbox to display each constraint's label in the graphics area.
9. Click **create**.
   Constraints are creaated on the pillar's end nodes.

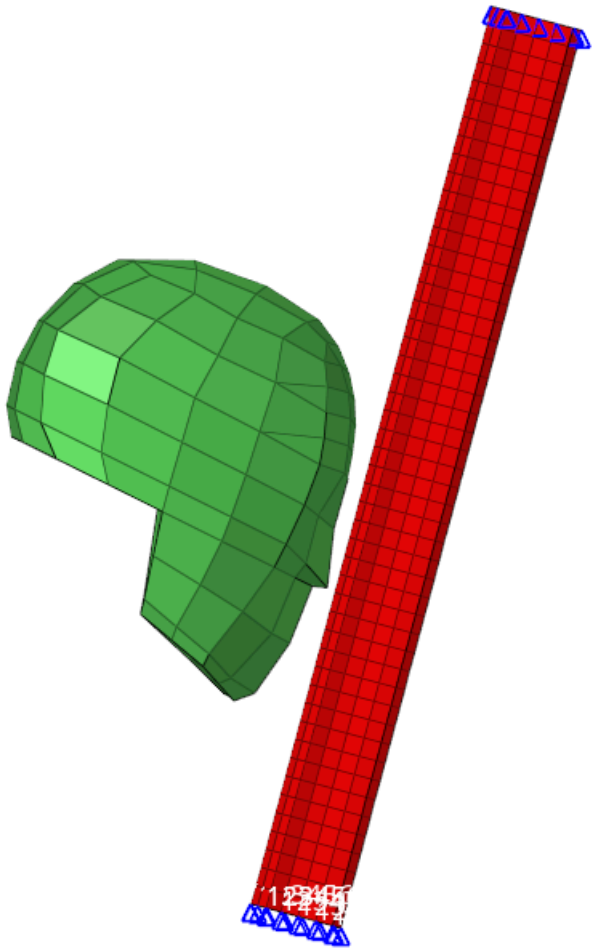


Figure 917:

10. Click **return** to close the panel.

Define a *SET_SEGMENT for the Secondary Entities

In this step, you will create a new contactsurf on the pillar element of the model.

1. In the Solver Browser, right-click and select **Create** > ***SET** > ***SET_SEGMENT** > ***SET_SEGMENT** from the context menu.
   HyperMesh creates and opens a new contactsurf in the Entity Editor.
2. For Name, enter `pillar_secondary`.
3. Optional: Click the **Color** icon, and select a color to display the contactsurf.
4. For Elements, click **0 Elements** > **Elements**.
5. In the panel area, set the second switch to **elems** as seen in Figure 918.

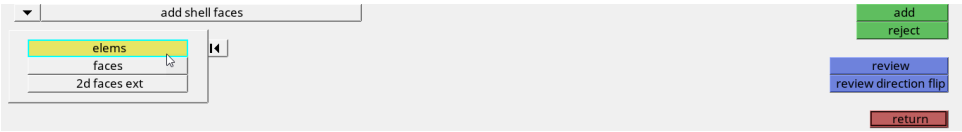


Figure 918:

6. Click **elems** > **by collector**.
7. Select the **pillar** checkbox as seen in Figure 919.

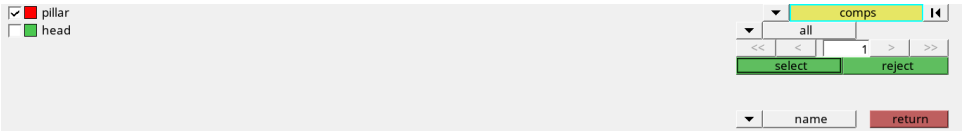


Figure 919:

8. Click **select**.
9. Click **add**.
10. Review the contactsurf to make sure that the pyramids are pointing out of the pillar as seen in Figure 920.

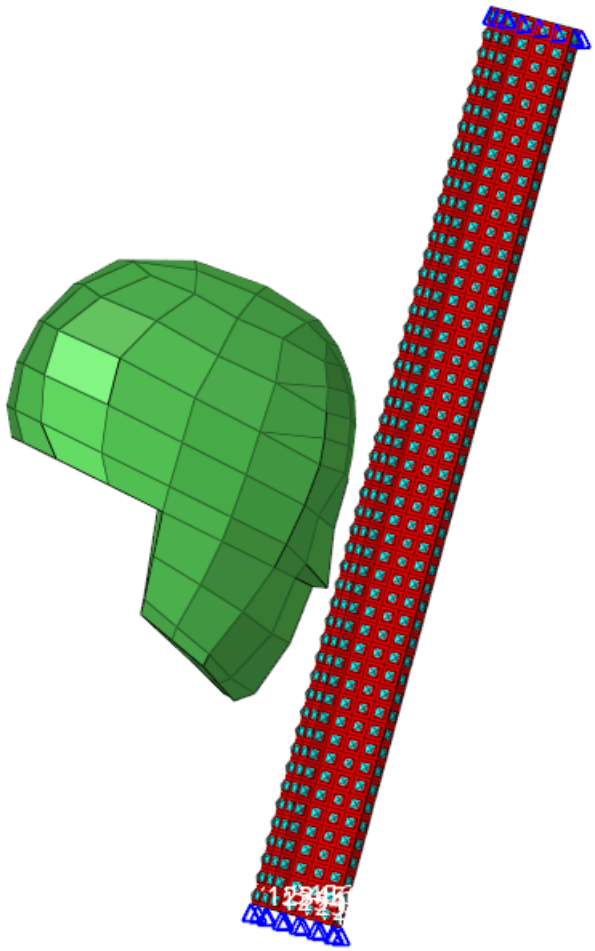


Figure 920:

11. Click **return** to close the panel.

Define a *SET_SEGMENT for the Main Entities

In this step, you will create a contactsurf on the head component of the model.

1. In the Solver Browser, right-click and select **Create** > ***SET** > ***SET_SEGMENT** > ***SET_SEGMENT** from the context menu.
   HyperMesh creates and opens a new contactsurf in the Entity Editor.
2. For Name, enter `head_master`.
3. Optional: Click the **Color** icon, and select a color to display the contactsurf.
4. For Elements, click **0 Elements** > **Elements**.
5. In the panel area, set the first switch to **add solid faces**.
6. Set the second switch to **elems**.
7. Click **elems** > **by collector**.

8. Select the **head** checkbox as seen in Figure 921.

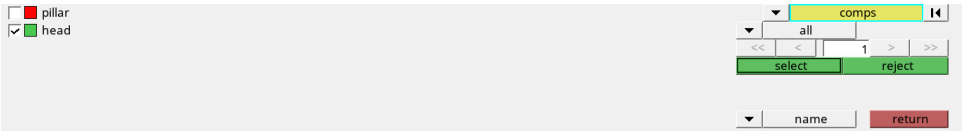


Figure 921:

9. Click **select**.
10. Use the nodes selector to select three nodes that belong to the same face of a solid element.
11. In the face angle field, enter `30`.
12. Click **add**.
13. Review the contactsurf to make sure that its pyramids are pointing out of the head as seen in Figure 922.

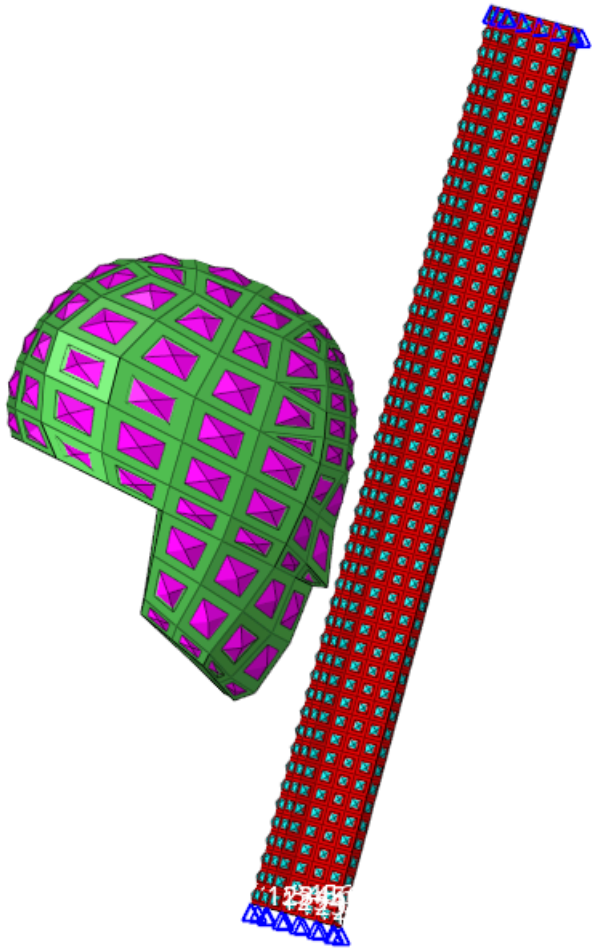


Figure 922:

14. Click **return** to close the panel.

Create a HyperMesh Group

In this step, you will create a HyperMesh group with the SurfaceToSurface card image.

1. In the Solver Browser, right-click and select **Create** > ***CONTACT** > ***CONTACT_SURFACE_TO_SURFACE** > ***CONTACT_SURFACE_TO_SURFACE** from the context menu.
   HyperMesh creates and opens a new group in the Entity Editor.
2. For Name, enter `contact`.

Add the Slave and Master Contactsurfs to the HyperMesh Group

In this step, you will add the slave and master contactsurfs to the HyperMesh group.

In this step, the Entity Editor should be open for the contact group.

1. In the Entity Editor, click **MSID** and set the entity selector to **Contactsurfs**.

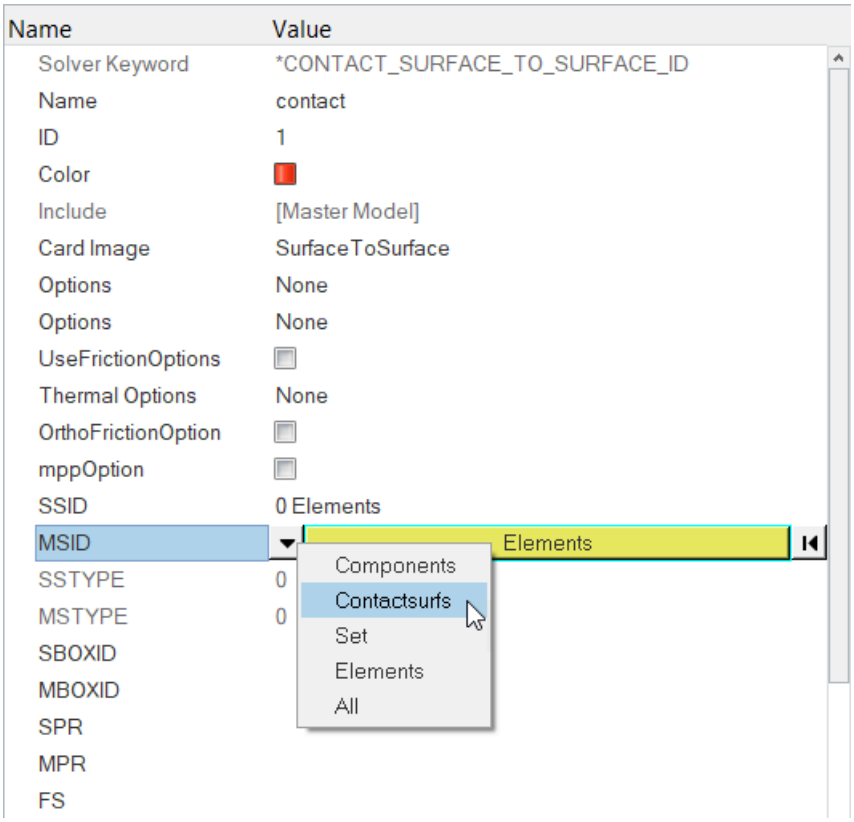


Figure 923:

2. Click **Contactsurfs**.
3. In the **Select Contactsurfs** dialog, select **head_master** and then click **OK**.
4. Click **SSID**, and set the entity selector to **Contactsurfs**.
5. Click **Contactsurfs**.
6. In the **Select Contactsurfs** dialog, select **pillar_slave** and then click **OK**.

Edit the Group card image

In this step, you will define the automatic option for the group in HyperMesh.

In this step, the Entity Editor should still be open for the contact group.

In the Entity Editor, for the first Options parameter, select **Automatic**.

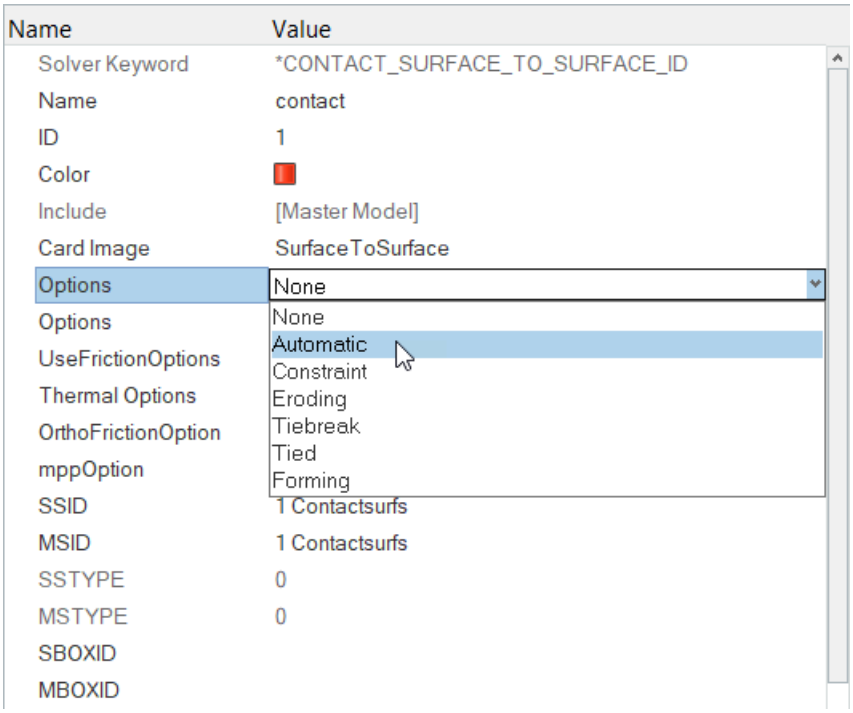


Figure 924:

Review the Group Main and Secondary Surfaces

In this step, you will review the main and secondary surfaces for the group.

1. Click **interfaces** from the Analysis page as seen in Figure 925.

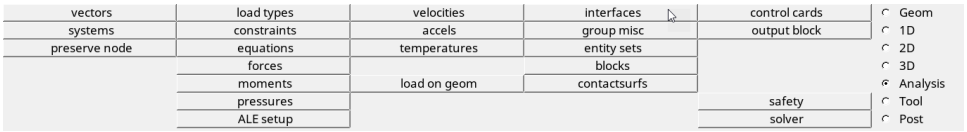


Figure 925:

The Interfaces panel opens.

2. Open the **add** subpanel.
3. Click **name=**, and select **contact** as seen in Figure 926.

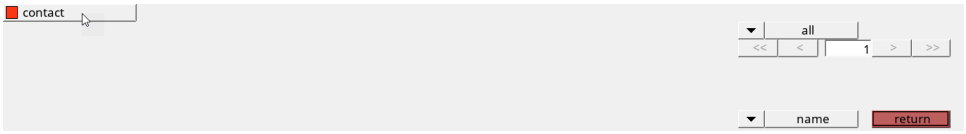


Figure 926:

4. Click **review**.
   HyperMesh temporarily displays the main and secondary entities in blue and red, respectively.

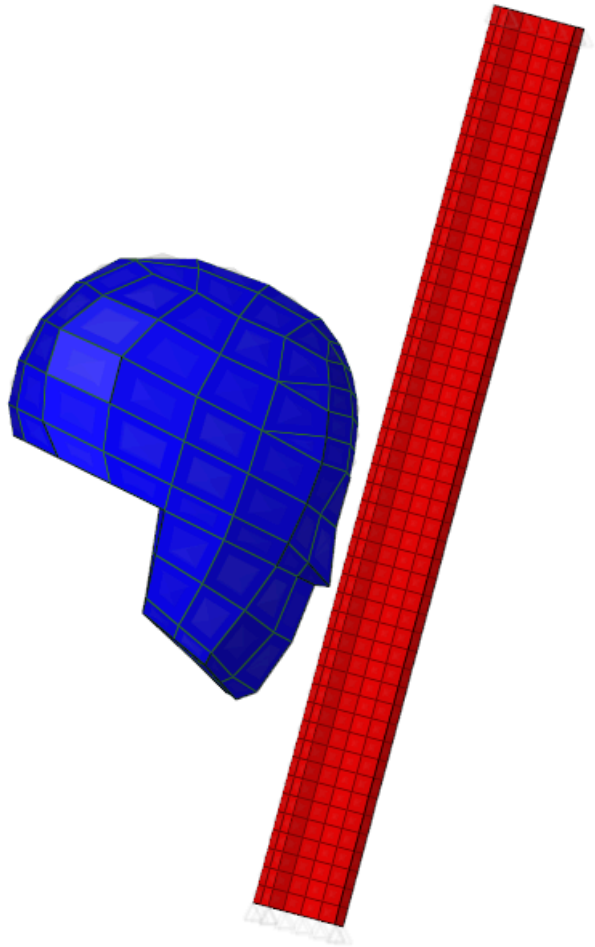


Figure 927:

5. Click **return** to close the panel.

Save Your Work

In this step, you will save your work in HyperMesh.

Save your work as a HyperMesh file named `head_3.hm`.

5.1.3 Exercise 3: Define Termination and Output for the Head and A-Pillar Impact Analysis

In this tutorial, you will become familiar with defining LS-DYNA control data and output requests in HyperMesh. Also, you will define the termination and output for a LS-DYNA analysis of a hybrid III dummy head impacting an A-pillar.

The head and A-pillar are depicted in Figure 928.

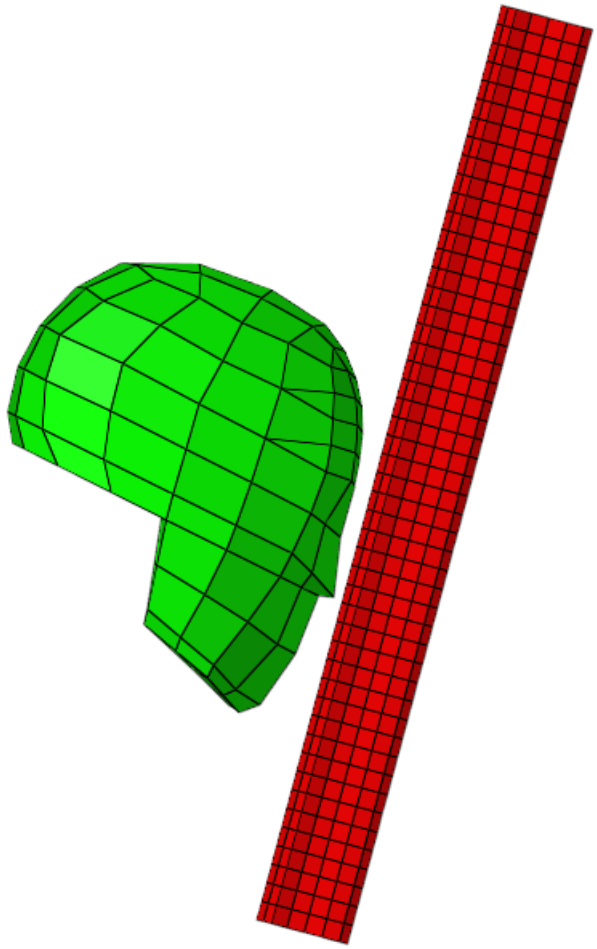


Figure 928:

Load the LS-DYNA User Profile

In this step, you will load the LS-DYNA user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Retrieve the HyperMesh File

In this step, you will open the HyperMesh model file.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
2. In the **Open Model** dialog, open the `head_3.hm` file.

The model appears in the graphics area.

Specify the Analysis Stop Time

In this step, you will specify the time you want LS-DYNA to stop the analysis with *CONTROL_TERMINATION.

1. In the Solver Browser, right-click and select ***Create** > ***CONTROL** > ***CONTROL_TERMINATION** from the context menu.

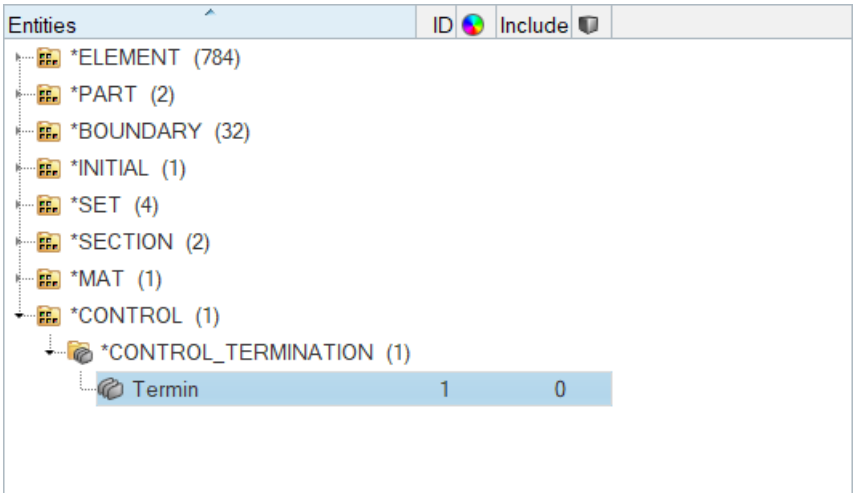


Figure 929:

HyperMesh creates and opens a new control in the Entity Editor.

2. For ENDTIM (termination time), enter `2.5`.

Specify the Output of D3plot Files

In this step, you will specifiy the output of D3plot files with *DATABASE_BINARY_D3PLOT.

1. In the Solver Browser, right-click and select **Create** > ***DATABASE** > ***DATABASE_BINARY_D3PLOT** from the context menu.
   HyperMesh creates and opens a new database in the Entity Editor.
2. For DT (time interval between outputs), enter `0.1`.

Specify ASCII Output

For this step, you will apecify ASCII output with *DATABASE_(Option) cards.

1. In the Solver Browser, right-click and select **Create** > ***DATABASE** > ***DATABASE_OPTION** from the context menu.
   HyperMesh creates and opens a new database in the Entity Editor.
2. Select the **GLSTAT** checkbox.
3. For DT, enter `0.1`.

   Note: This specifies the output of global data at every 0.1 ms.
4. Select the **MATSUM** checkbox.
5. For DT, enter `0.1`.

   Note: This specifies the output of material energies every 0.1 ms.
6. Select the **SPCFORC** checkbox.
7. For DT, enter `0.1`.

   Note: This specifies the output of SPC reaction forces every 0.1 ms.

Export the Model

In this step, you will export the model as an LS-DYNA keyword file.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
2. In the Export - Solver Deck tab, set File type to **LsDyna**.
3. From the Template list, select the appropriate template.
4. In the File field, navigate to your working directory and save the file as `head_complete.key`.
5. Click **Export**.

Submit the LS-DYNA Input File

In this step, you will submit the LS-DYNA input file to LS-DYNA 970 for analysis.

1. From your desktop Start Menu, open the LS-DYNA Manager program.
2. From the solvers menu, select **Start LS-DYNA analysis**.
3. Load the `head_complete.key` file.
4. Click **OK** to start the analysis.

Post-Process the LS-DYNA Results Using HyperView

In this step, you will save your work.

Save your work to a HyperMesh file.

5.2 HM-4610: Use Curves, Beams, Rigid Bodies Joints, and Loads in LS-DYNA

In this tutorial you will: create XY curves to define non-linear materials, constrained nodal rigid bodies, and joints; define beam elements with HyperBeam, *DEFORMABLE_TO_RIGID, *LOAD_BODY, *BOUNDARY_PRESCRIBED_MOTION_NODE; and use the HyperMesh Component Table tool to review the model's data.

Before you begin, copy the `seat_start.hm` and `seat_2.hm` from `<hm.zip>/interfaces/lsdyna/` to your working directory.

5.2.1 Exercise 1: Define Model Data for the Seat Impact Analysis

In this exercise, you will define and review model data for a LS-DYNA analysis of a vehicle seat impacting a rigid block. The seat and block model is shown in Figure 930.

This exercise will help you become familiar with defining an LS-DYNA model data in HyperMesh. The model used in this exercise is depicted in Figure 930.

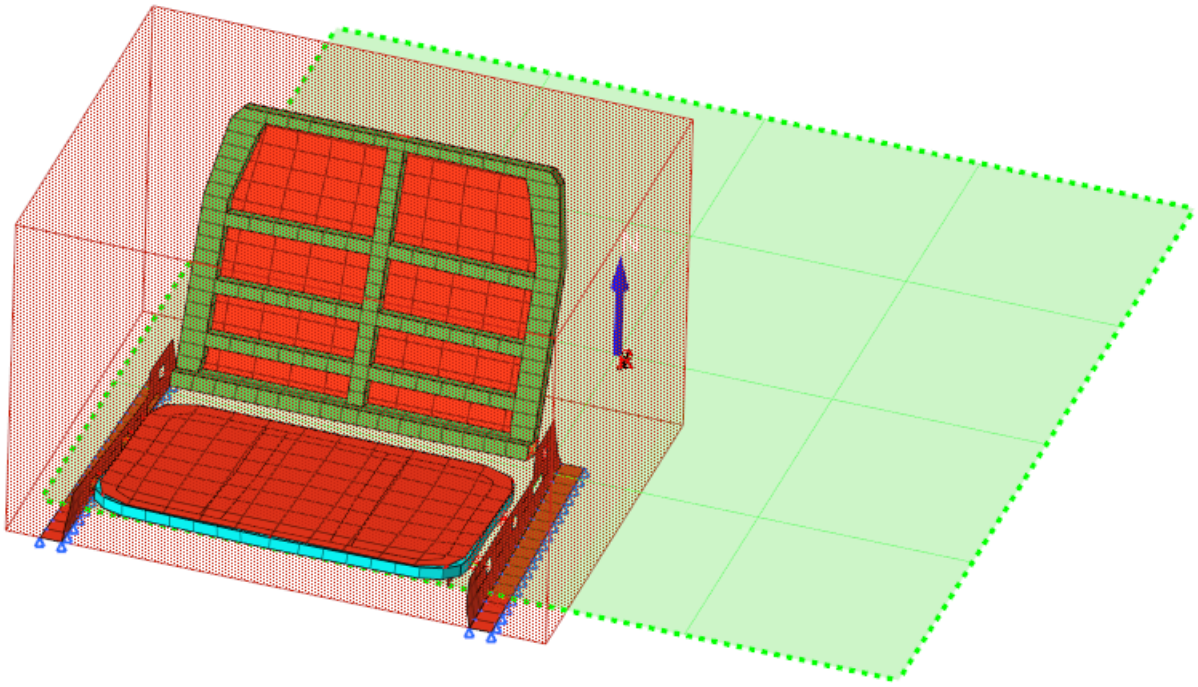


Figure 930:

Load the LS-DYNA User Profile

In this step, you will load the LS-DYNA user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Retrieve the HyperMesh File

In this step, you will open the model file in HyperMesh.

1. Open the model file by completing one of the following options.
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
2. In the **Open Model** dialog, open the `seat_start.hm` file.
   The model appears in the graphics area.
3. Observe the model using various visual options available in HyperMesh, such as rotation and zooming.

Create an XY Plot

In this step, you will create an XY plot.

1. Open the Plots panel by clicking **XYPlots** > **Create** > **Plots** from the menu bar.
2. In the plot= field, enter `seat_mat`.
3. Set plot type to **standard** as seen in Figure 931.



Figure 931:

4. Leave the like = field empty.
   When an existing plot is selected, the new plot adopts its attributes.
5. Click **create plot**.
6. Click **return**.

Create Two Stress-Strain Curves

In this step, you will input data from a file to create two stress-strain curves.

1. Open the Read Curves panel by clicking **XYPlots** > **Create** > **Curves** > **Read Curves** from the menu bar.
2. Leave the plot = field set to `seat_mat`.

3. Click **browse**.
4. In the **Open** dialog, open the `seat_mat_data.txt` file.
5. Click **input**.
   HyperMesh creates two curves, and names them 0.001 and 0.004 as seen in Figure 932.

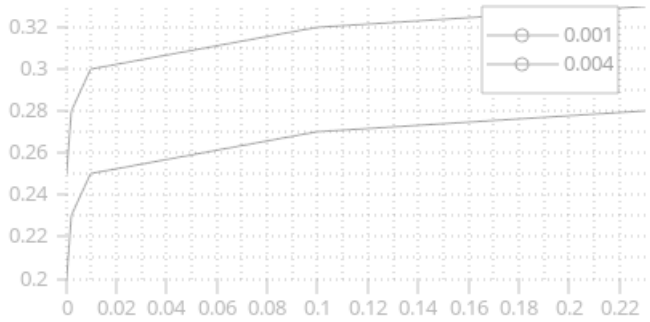


Figure 932:

6. Click **return**.

Create a Dummy XY Curve

In this step, you will create a dummy XY curve to be used to create a *DEFINE_TABLE.

1. Open the Edit Curves panel by clicking **XYPlots** > **Edit** > **Curves** from the menu bar.
2. Open the **create** subpanel.
3. Click **plot =**, and select **seat_mat**.
4. Select **math**.
5. In the x = field, enter `{0.0, 0.2}`.
6. In the y = field, enter `{0.4, 0.4}`.



Figure 933:

7. Click **create**.
   HyperMesh creates a curve in the seat_mat plot, and names it curve3 as seen in Figure 934.

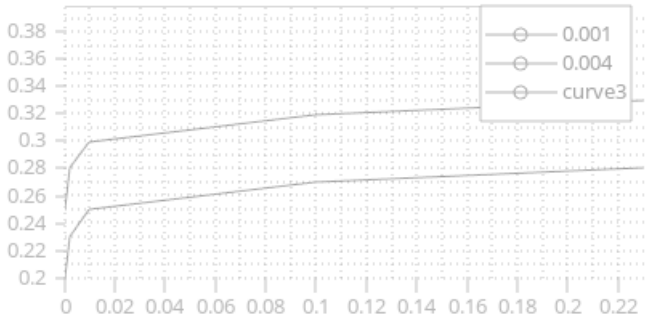


Figure 934:

8. Click **return**.

Create a *DEFINE_TABLE

In this step, you will create a *DEFINE_TABLE from the dummy curve created in Create a Dummy XY Curve.

1. In the Model Browser, Curve folder, click **curve3**.

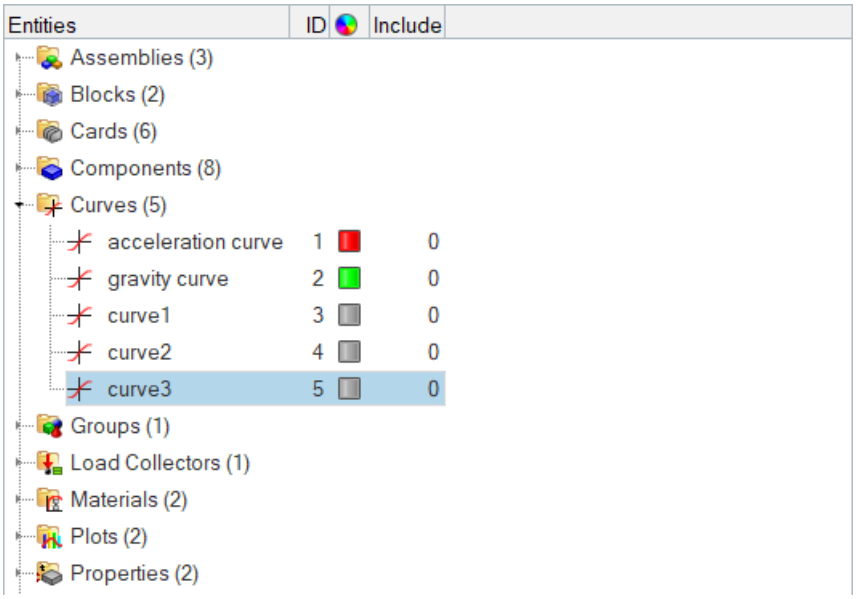


Figure 935:

The Entity Editor opens, and displays the curve's corresponding data.

2. Select the **DEFINE_TABLE** checkbox.
3. For ArrayCount, enter `2`.

> **Note:** This is the number of strain rate values to be specified.

4. In the Data: VALUE row, click .

5. In the **ArrayCount** dialog, enter `0.001` in the strain rate VALUE(1) field and `0.004` in the strain rate VALUE(2) field.

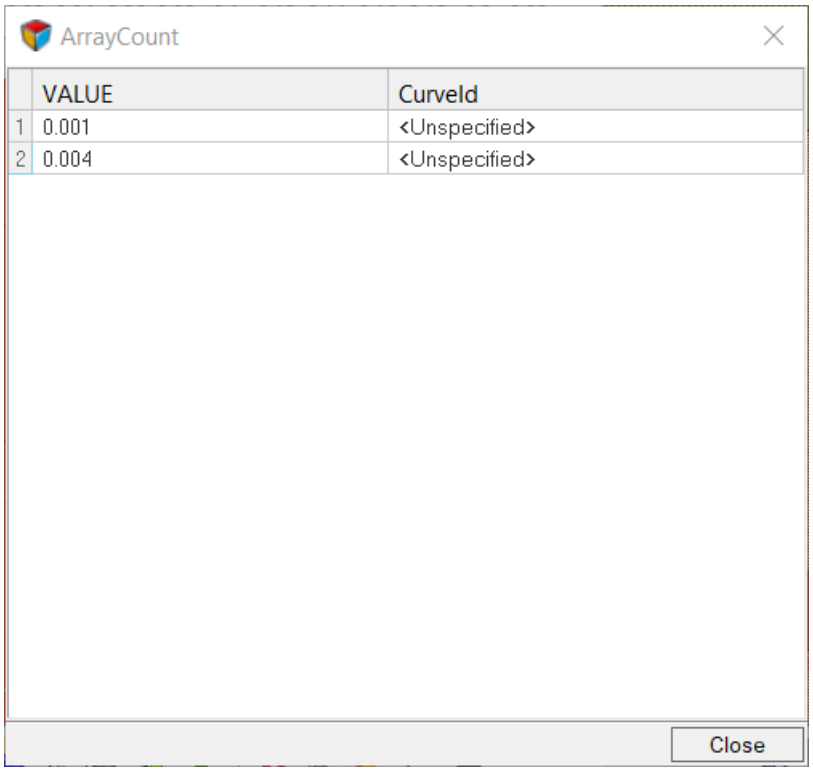


Figure 936:

6. In the CurveId(1) field, click **Unspecified** > **Curve**.

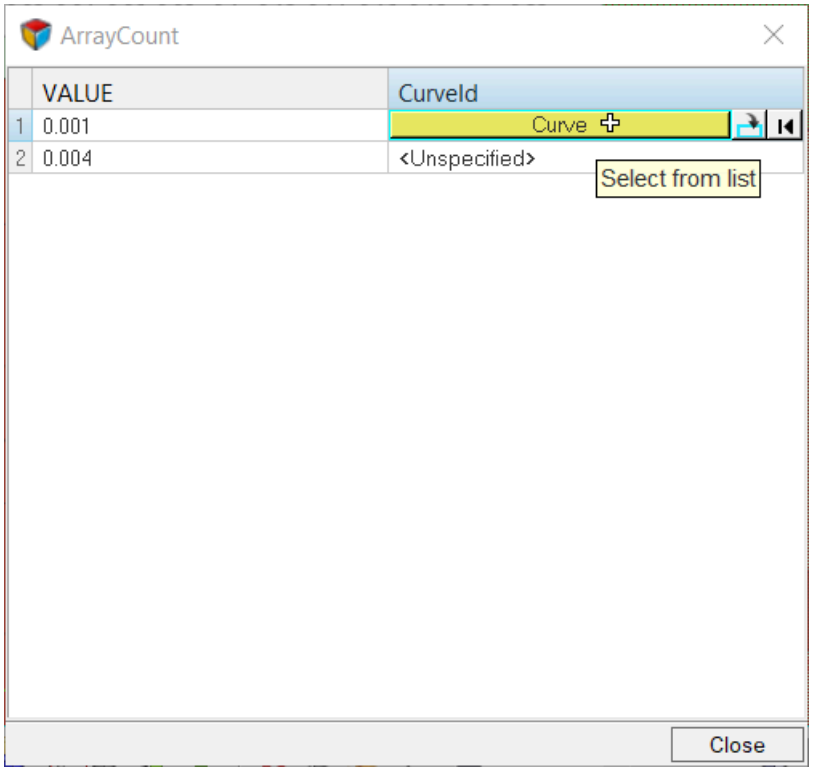


Figure 937:

7. In the **Select Curve** dialog, select **curve1** and then click **OK**.

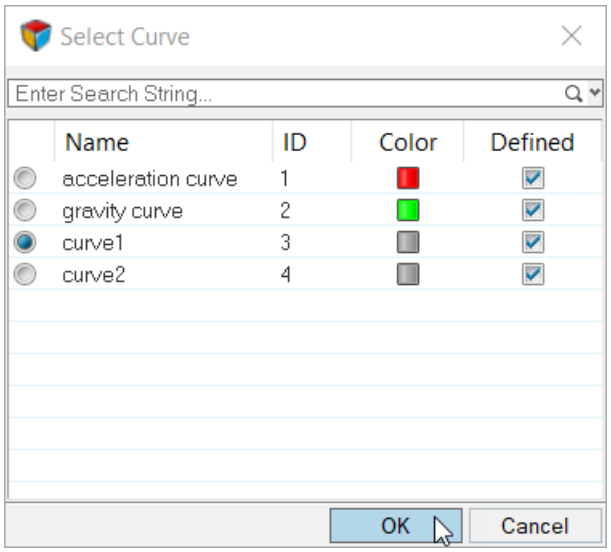


Figure 938:

8. In the CurveId(2) field, click **Unspecified** > **Curve**.
9. In the **Select Curve** dialog, select **curve2** and then click **OK**.
10. Click **Close**.

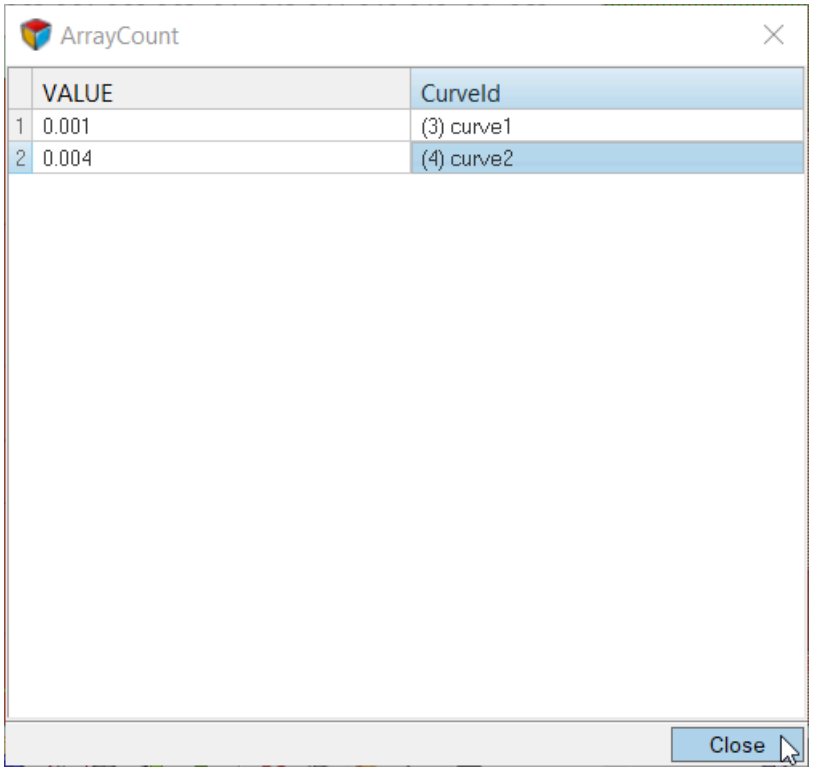


Figure 939:

Create a Non-Linear Material

In this step, you will create a non-linear material (*MAT_PIECEWISE_LINEAR_PLASTICITY).

1. Open the Solver Browser by clicking **View** > **Browsers** > **HyperMesh** > **Solver** from the menu bar.
2. In the Solver Browser, right-click and select **Create** > ***MAT** > **MAT (1-50)** > **24-*MAT_PIECEWISE_LINEAR_PLASTICITY** from the context menu.

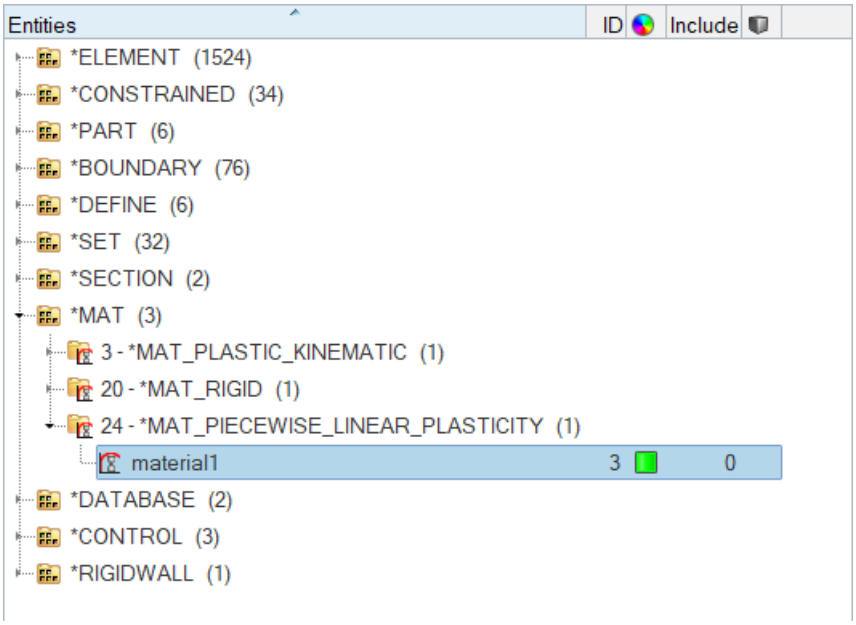


Figure 940:

HyperMesh creates and opens a new material in the Entity Editor.

3. For Name, enter `steel`.
4. For Rho (Mass density), enter `7.8 E-6`.
5. For E (Young modulus), enter `200`.
6. For PR (Poisson ratio), enter `0.3`.
7. For SIGY (Yield stress), enter `0.25`.
8. Click **LCSS** and then click **curve**.

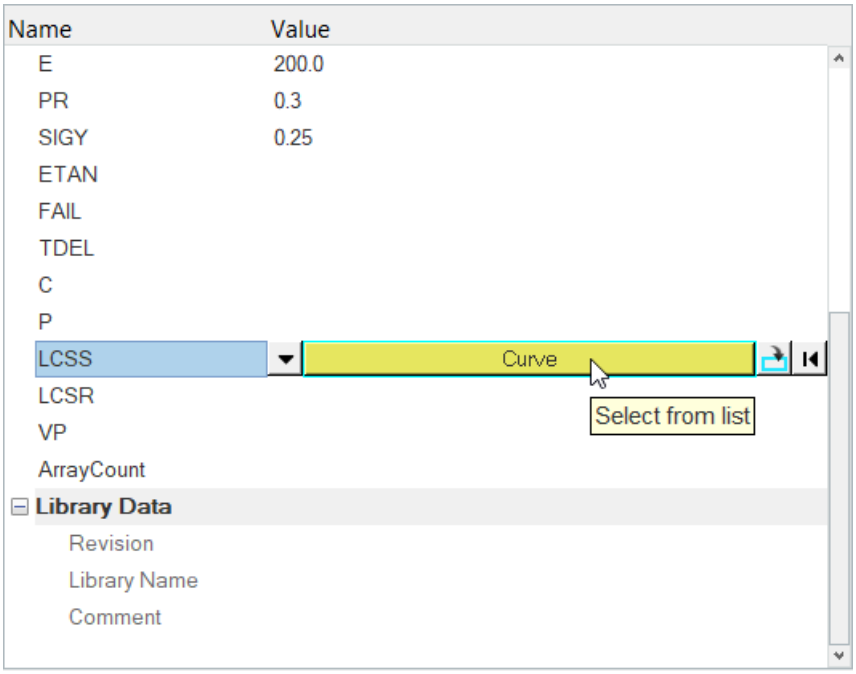


Figure 941:

9. In the **Select Curve** dialog, select **curve3** and then click **OK**.

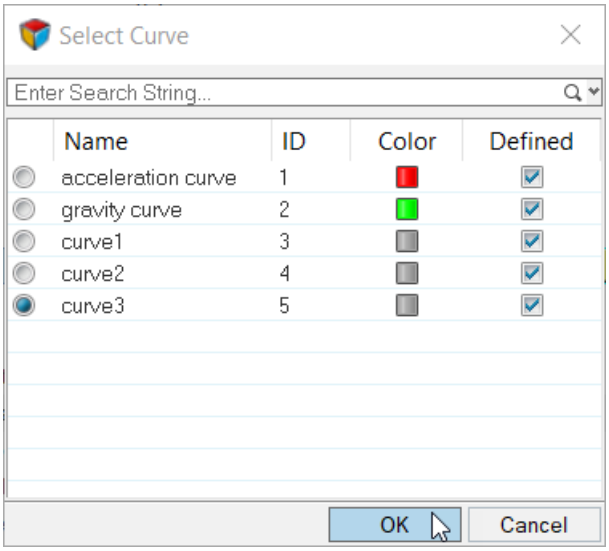


Figure 942:

Update the Components with the New Material

In this step, you will update the base_frame and back_frame components with the non-linear material created in Create a Non-Linear Material.

1. Enable the Quick Access Tool by pressing **Ctrl** > **F**.

2. In the Quck Access Tool, enter `Utility Component Table` and press **Enter**.
   The **Components and Properties** dialog opens.
3. In the **Components and Properties** dialog, click **Table** > **Editable** from the menu bar as seen in Figure 943.

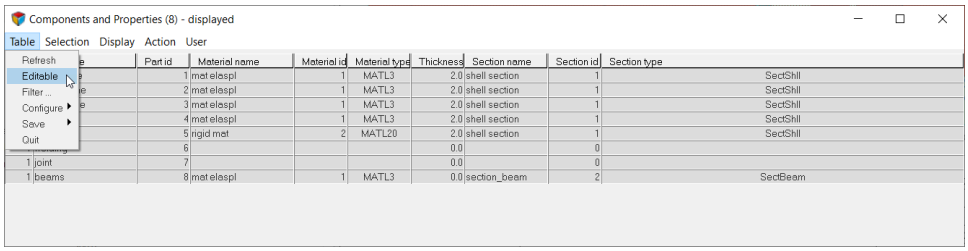


Figure 943:

4. Assign the material steel to the base_frame and back_frame components.
   a) Select the **base_frame** component.
   b) Set Assign Values to **Material name**.
   c) Set HM-Mats to **steel**.
   d) Click **Set**.
   e) In the **Confirm** dialog, click **Yes**.
   f) Repeat steps 4.a through 4.e for the back_frame component.

   The Material name for base_frame and back_frame is set to steel as seen in Figure 944.

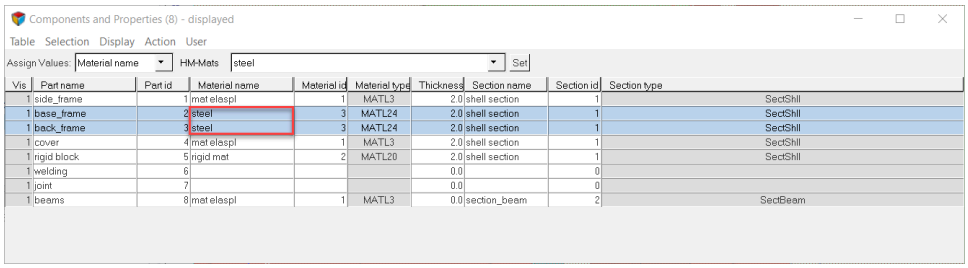


Figure 944:

5. From the menu bar, click **Table** > **Quit**.

Create a Beam Element

In this step, you will create a beam element, *ELEMENT_BEAM, to complete the seat's back_frame connection to the side_frame on the left side.

1. Restore a pre-defined view.
   a) In the Model Browser, Views folder, right-click on **Beam_view** and select **Show** from the context menu.
2. Set the current component to beams.

a) In the Model Browser, Component folder, right-click on **beams** and select **Make Current** from the context menu.
HyperMesh sets the beam component as the current collector.

3. Create the beam element.

a) Open the Bars panel by clicking **Mesh** > **Create** > **1D Elements** > **Bars** from the menu bar.

b) Under orientation, select **node**.

> **Note:** You will select a direction node later to define the beam's section orientation.

c) Using the node A selector, select the center node of the left nodal rigid body as seen in Figure 945.

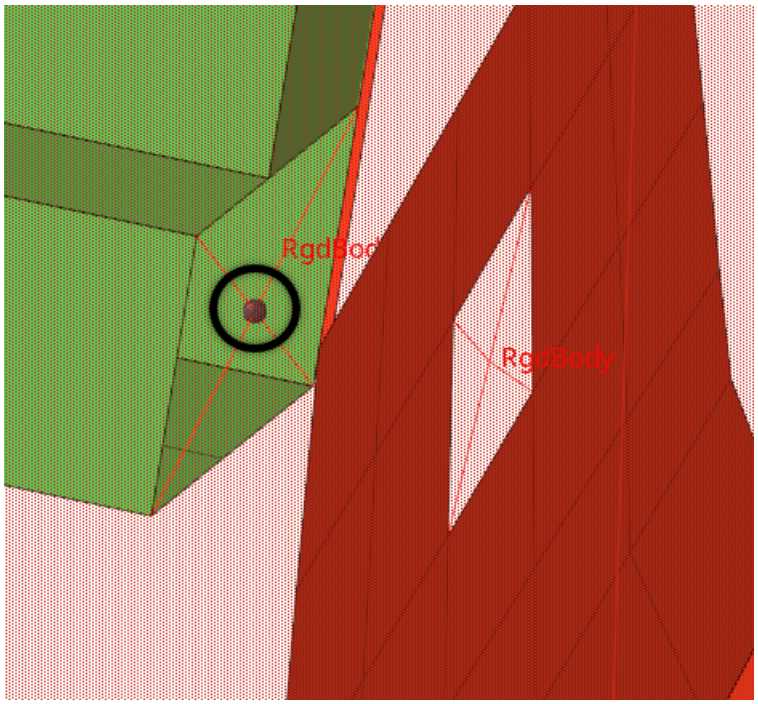


Figure 945:

d) Using the node B selector, select the center node of the right nodal rigid body as seen in Figure 946.

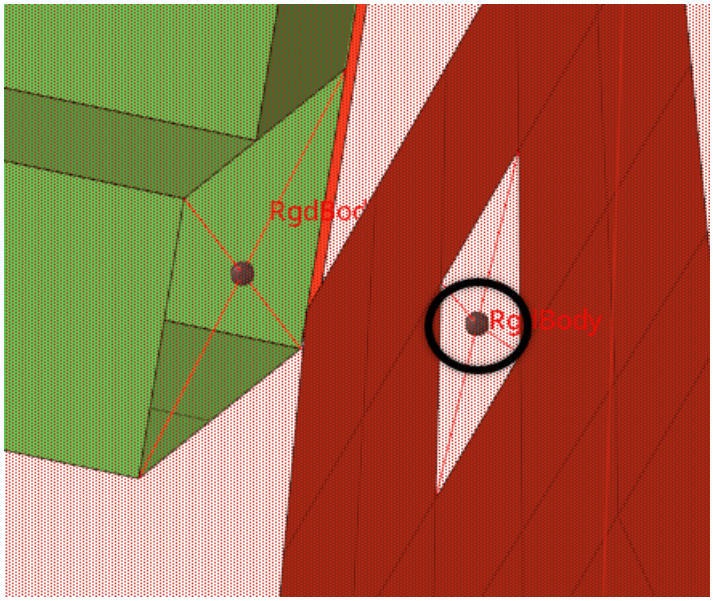


Figure 946:

e) Using the direction node selector, select any non-center node on one of the nodal rigid bodies.
HyperMesh creates the beam as seen in Figure 947.

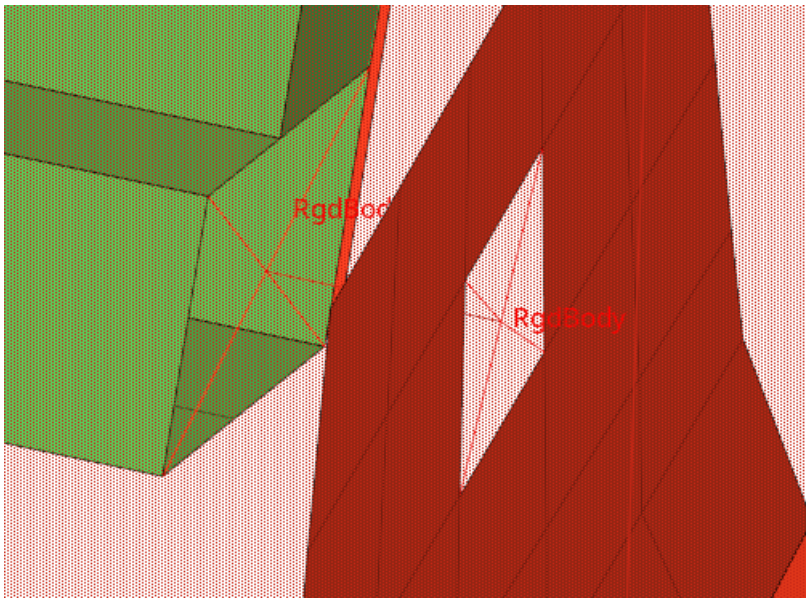


Figure 947:

f) Click **return**.

Display Node IDs

In this step, you will display node IDs.

1. Click on the Display toolbar.
   The Numbers panel opens.
2. Set the entity selector to **nodes**.
3. Click **nodes** > **by id**.
4. In the id= field, enter `425-427, 431`.
5. Press **Enter**.
6. Select the **display** checkbox.
7. Click **on**.
   HyperMesh displays the IDs as seen in Figure 948.

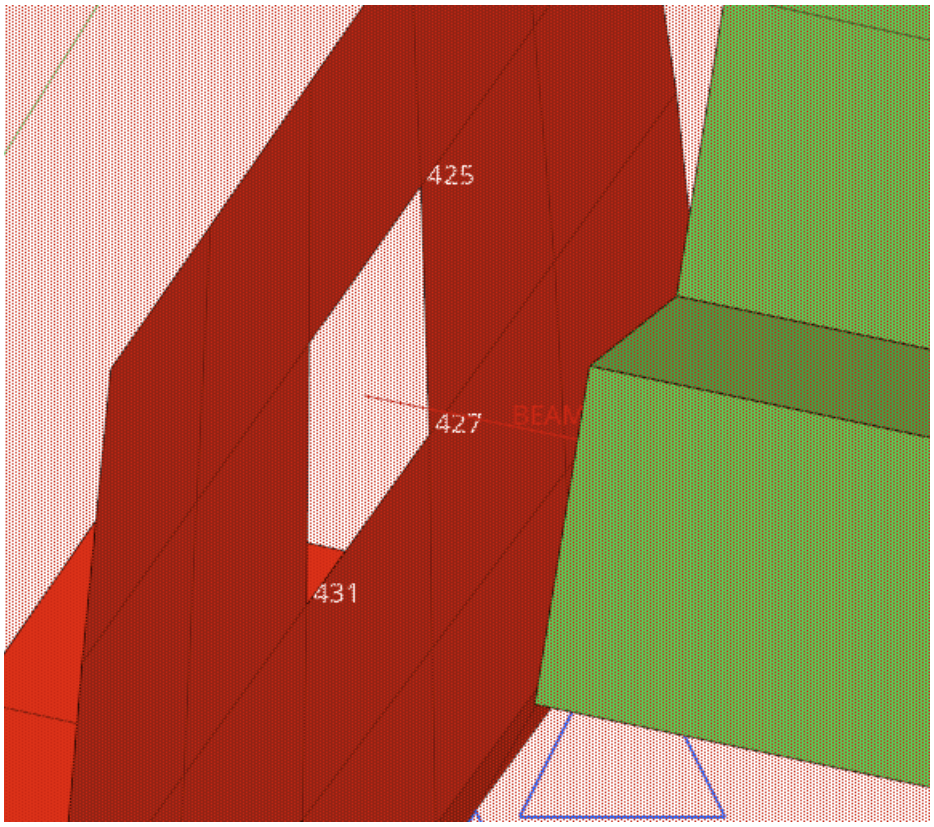


Figure 948:

8. Click **return**.

Set the Current Component to Welding

In this step, you will set the current component to welding.

In the Model Browser, Components folder, right-click on **welding** and select **Make Current** from the context menu.

HyperMesh sets the welding component as the current collector.

Select the RgdBody Type for the Rigid Configuration

In this step, you will select the rgdbody type for theHyperMesh rigid configuration.

1. Click **Mesh** > **Assign** > **Element Type** from the menu bar.
   The Element Type panel opens.
2. Select the elements to update.
   a) Click **rigid =**, and then select **RgdBody**.
3. Click **update**.
4. Click **return**.

Create the Nodal Rigid Body

In this step, you will create the nodal rigid body (*CONSTRAINED_NODAL_RIGID_BODY).

1. In the Solver Browser, right-click and select **Create** > ***CONSTRAINED** > ***CONSTRAINED_NODAL_RIGID_BODY** > ***CONSTRAINED_NODAL_RIGID_BODY** from the context menu.
2. In the Rigids panel, set the nodes 2-n selector to **multiple nodes**.
3. Using the node1 selector, select the beam's free end as seen in Figure 949.

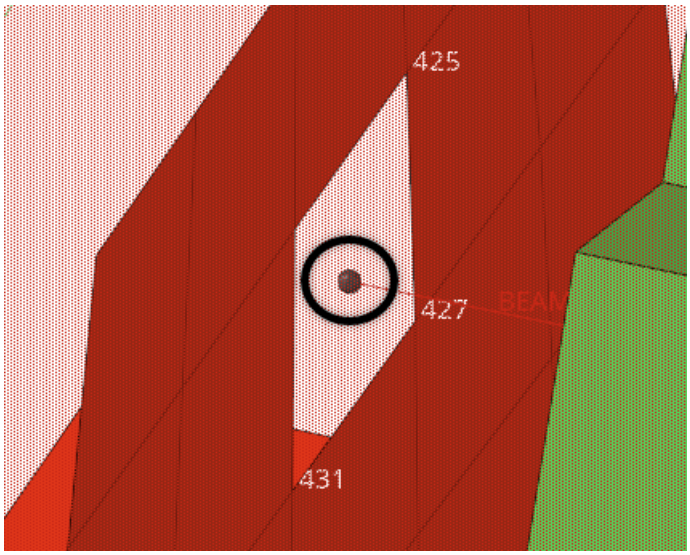


Figure 949:

4. Click **nodes 2-n: nodes** > **by id**.
5. In the id= field, enter `425`, `426`, `427` and `431`.
6. Press **Enter**.

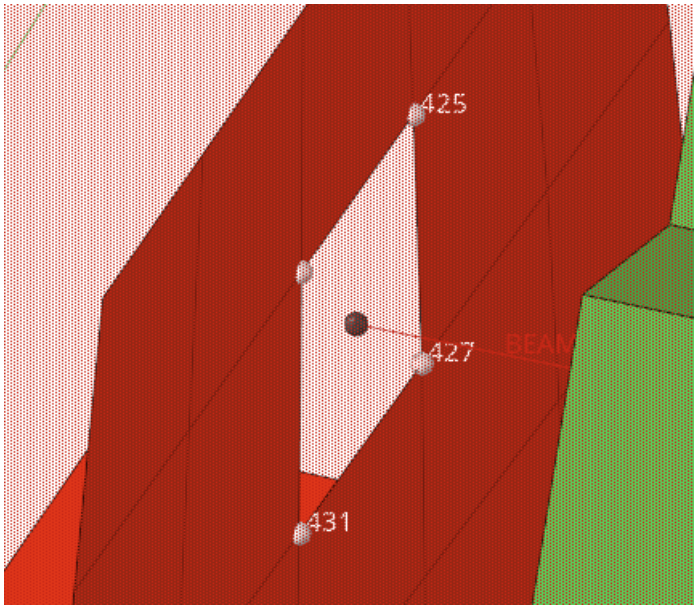


Figure 950:

7. Unselect the **attach nodes as set** checkbox.
8. Click **create**.
   HyperMesh creates the nodal rigid body as seen in Figure 951.

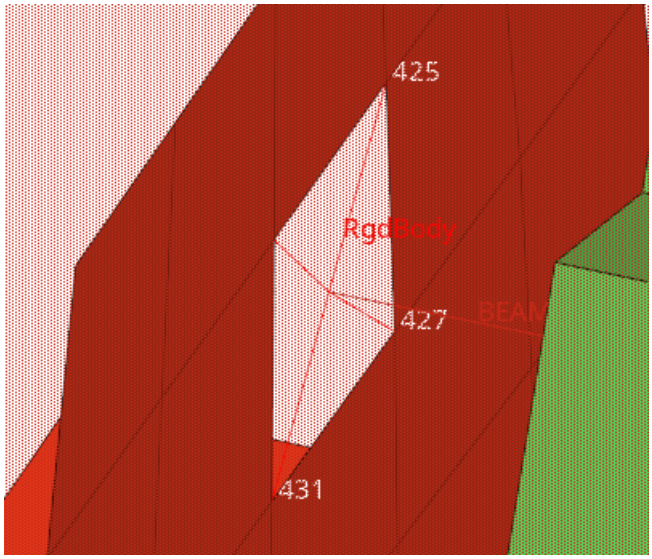


Figure 951:

9. Click **return**.
   HyperMesh does not create *CONSTRAINED_JOINT_STIFFNESS; it is not needed for this joint to work.

Display Node IDs

In this step, you will display node IDs.

1. On the Visualization toolbar, click  to display the model's elements as wireframe elements skin only.
2. Open the Numbers panel.
3. Set the entity selector to **nodes**.
4. Click **nodes** > **by id**.
5. In the id= field, enter `1635, 1636`.
6. Press **Enter**.
7. Select the **display** checkbox.
8. Click **on**.
   HyperMesh displays the IDs as seen in Figure 952.

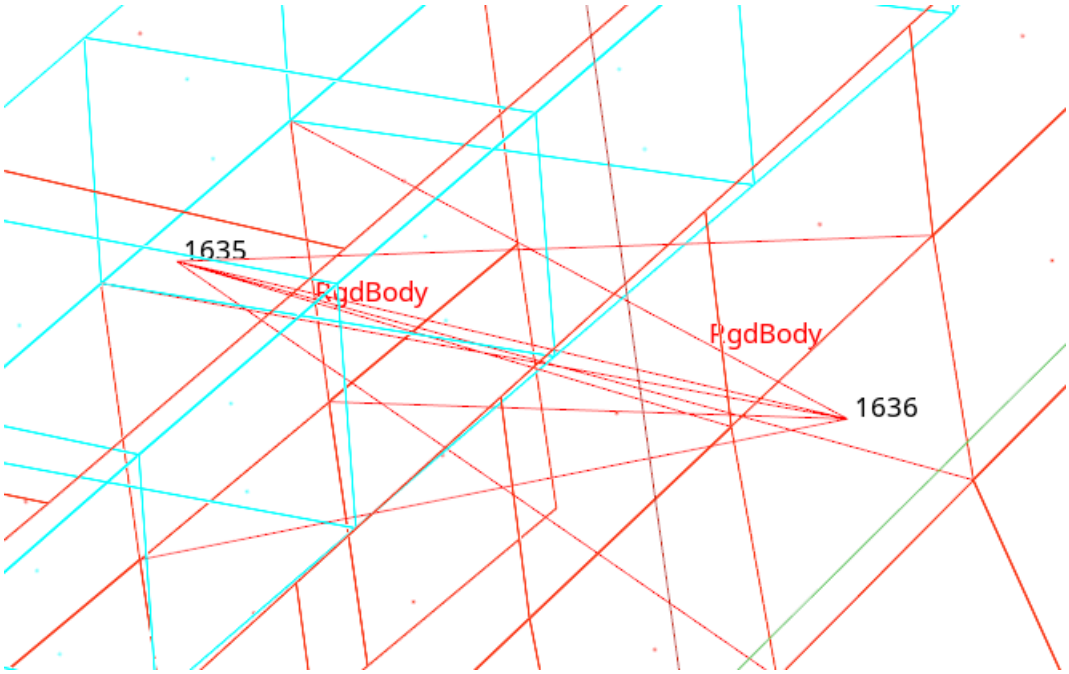


Figure 952:

9. Click **return**.

Activate Coincident Picking

In this step, you will activate coincident picking from the graphics panel.

1. Click **Preferences** > **Graphics** from the menu bar.
   The Graphics panel opens.
2. Select the **coincident picking** checkbox.
3. Click **return**.

Set the Current Component to Joint

In this step, you will set the joint component as the current collector.

In the Model Browser, Components folder, right-click on **joint** and select **Make Current** from the context menu.
HyperMesh sets the joint component as the current collector.

Create a Revolute Joint

In this step, you will create a joint between two nodal rigid bodies (*CONSTRAINED_JOINT_REVOLUTE).

Rigid bodies must share a common edge along which to define a joint. This edge, however, must not have nodes merged together. Two rigid bodies will rotate relative to each other along the axis defined by the common edge.

1. In the Solver Browser, right-click and select **Create** > ***CONSTRAINED** > ***CONSTRAINED_JOINT_REVOLUTE** > ***CONSTRAINED_JOINT_REVOLUTE** from the context menu.
2. In the Joints panel, set joint type to **revolute**.
3. Using the node 1 selector, click node **1635**.
   The coincident picking mechanism displays the nodes 1635 and 1633 as seen in Figure 953.

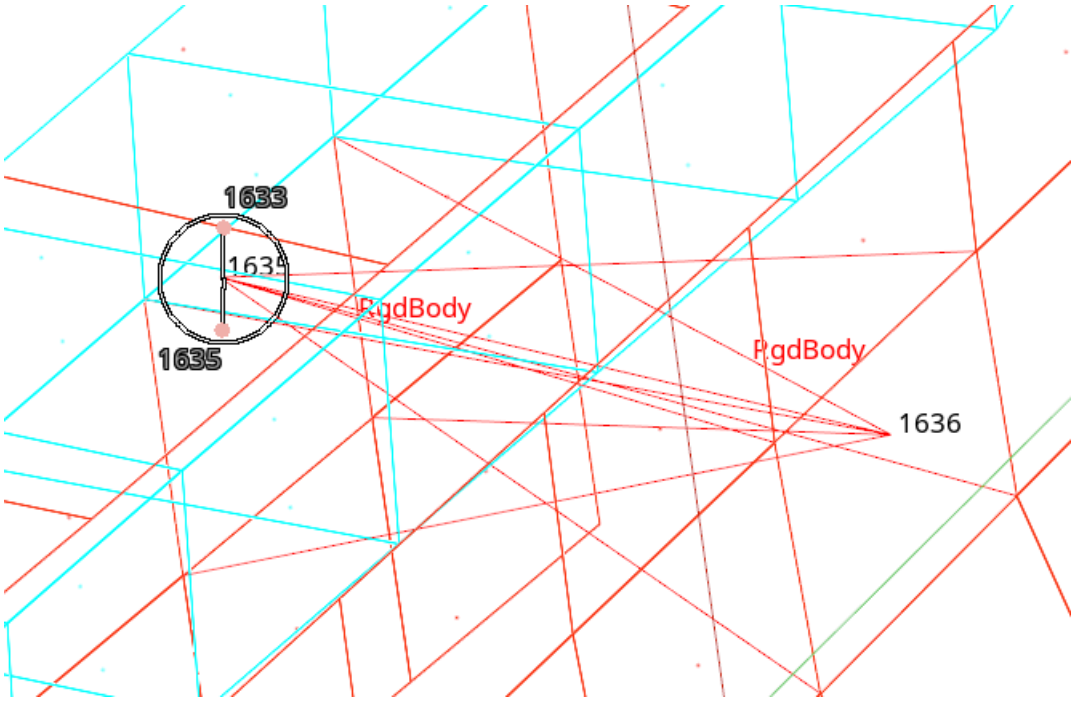


Figure 953:

4. From the coincident picking mechanism, click node **1635**.
   HyperMesh selects node 1635 for node 1 in rigid body A.
5. Using the node 2 selector, select the node created in step 4.
   The coincident picking mechanism displays two nodes: 1635 and 1633.

6. From the coincident picking mechanism, click node **1633**.
   HyperMesh selects node 1633 for node 2 in rigid body B.
7. Using the node 3 selector, click node **1636**.
   The coincident picking mechanism displays two nodes: 1636 and 1634.
8. From the coincident picking mechanism, click node **1636**.
   HyperMesh selects node 1636 for node 3 in rigid body A.
9. Using the node 4 selector, select the node created in step 8.
   The coincident picking mechanism displays two nodes: 1636 and 1634.
10. From the coincident picking mechanism, click node **1634**.
    HyperMesh selects node 1634 for node 4 in rigid body B.
11. Click **create**.
    HyperMesh creates the joint as seen in Figure 954.

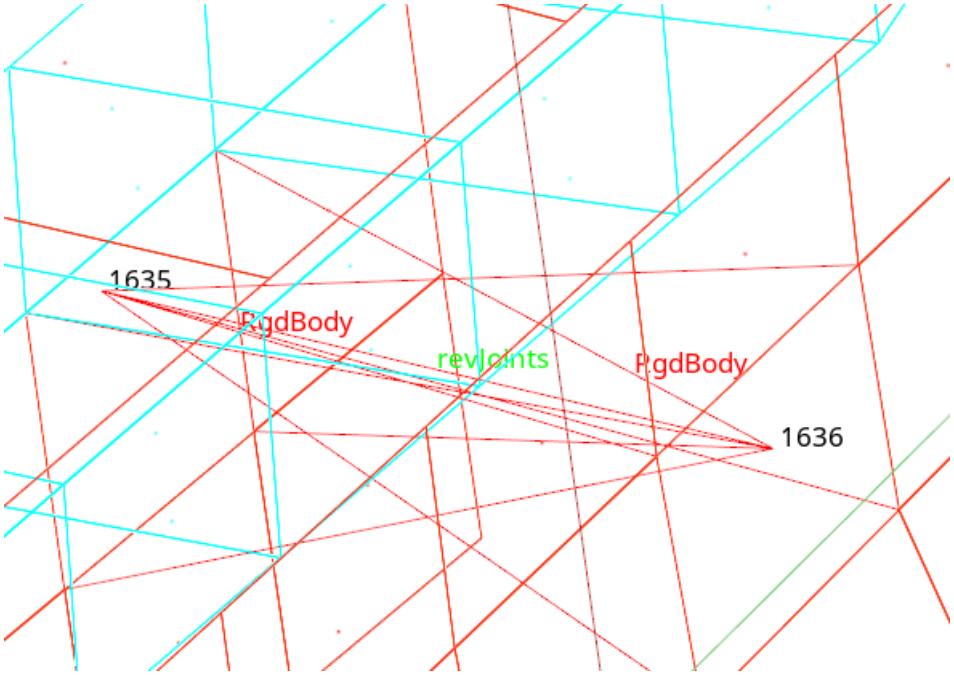


Figure 954:

12. Click **return**.

Define *DEFORMABLE_TO_RIGID

In this step, you will define *DEFORMABLE_TO_RIGID to set up the moving seat as rigid until the time of impact with the block, to reduce computation time.

1. Create an entity set that contains the base_frame, back_frame, and cover components.
   a) In the Solver Browser, right-click and select **Create** > ***SET** > ***SET_PART** > ***SET_PART_LIST** from the context menu.

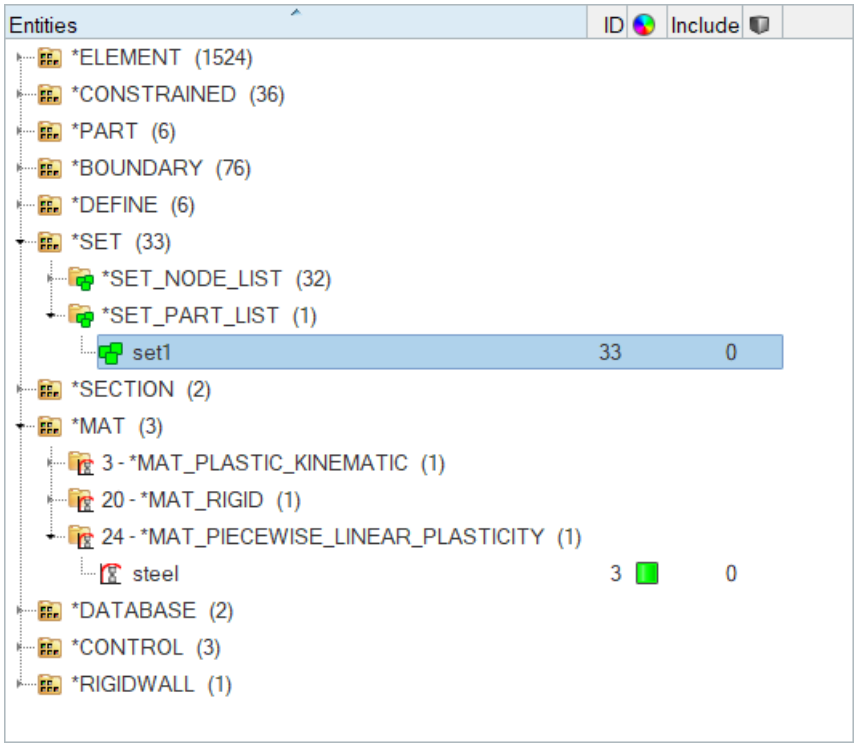


Figure 955:

HyperMesh creates and opens a new set in the Entity Editor.

b) For Name, enter `set_part_seat`.

c) For Entity IDs, click **0 Components** > **Components**.

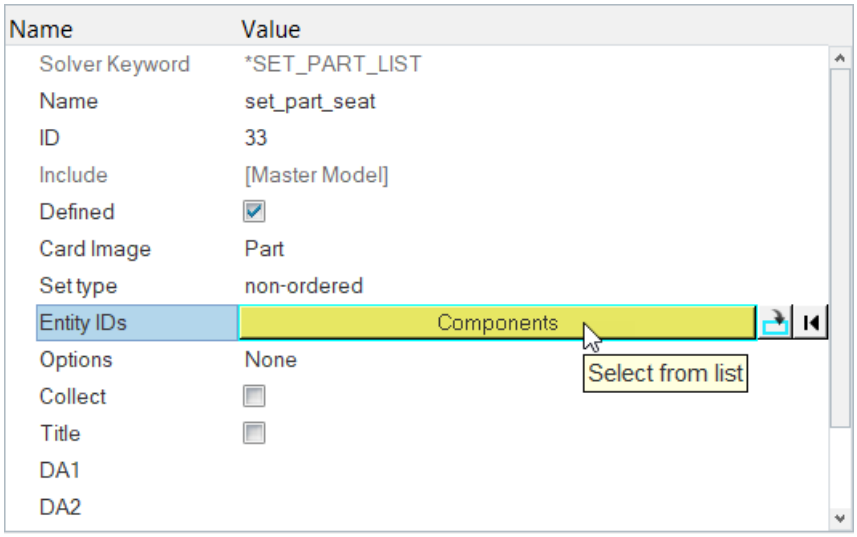


Figure 956:

The **Select Components** dialog opens.

d) In the **Select Components** dialog, select **base_frame**, **back_frame**, and **cover** and then click **OK**.

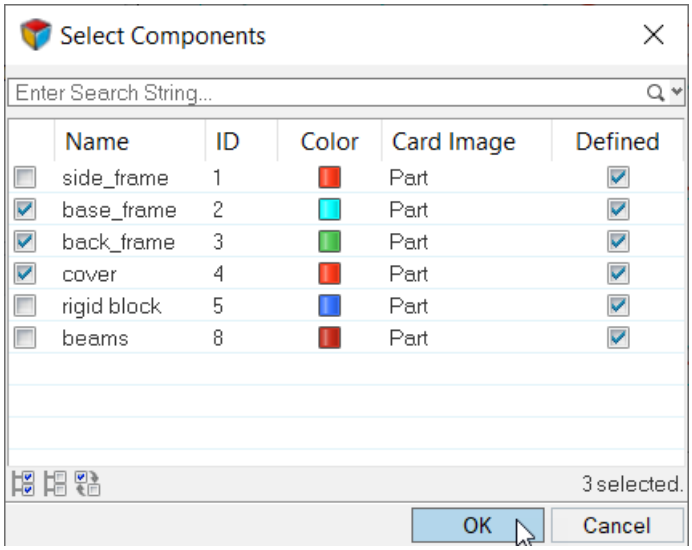


Figure 957:

2. Define *DEFORMABLE_TO_RIGID to switch the deformable seat to rigid at the beginning of the analysis.

a) In the Solver Browser, right-click and select **Create** > ***DEFORMABLE_TO_RIGID** > ***DEFORMABLE_TO_RIGID** from the context menu.

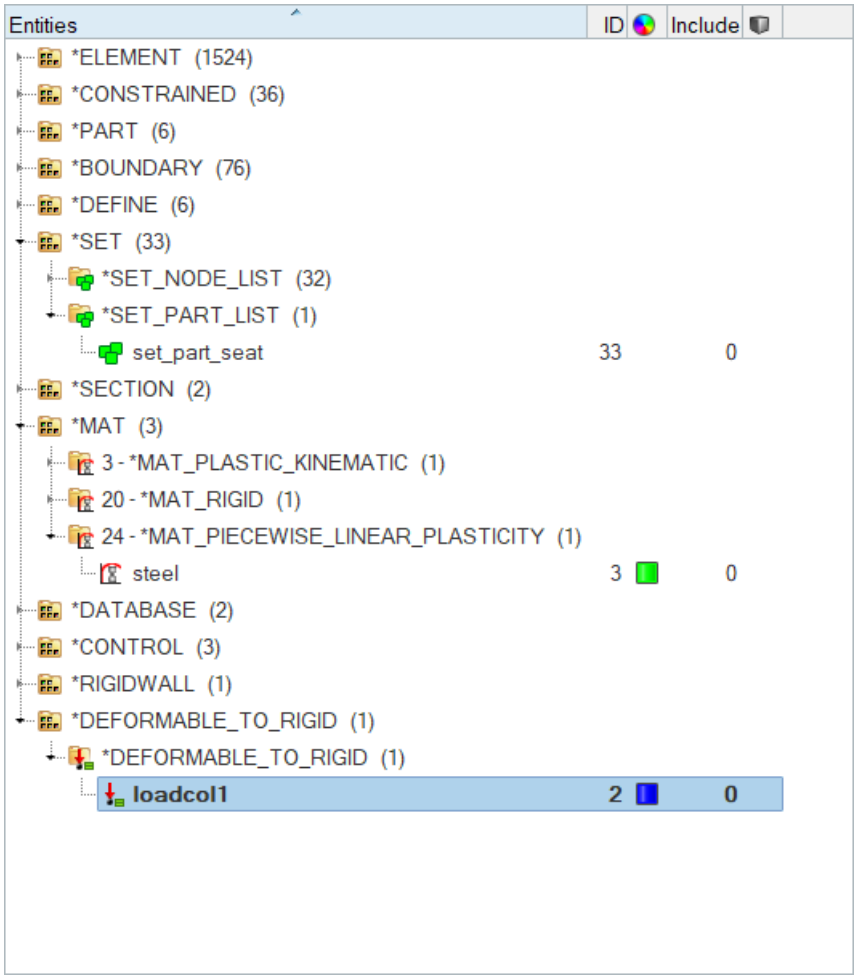


Figure 958:

HyperMesh creates and opens a new load collector in the Entity Editor.

b) For Name, enter `dtor`.

c) For ArrayCount, select **1**.

d) For PSID, click **Unspecified** > **Set**.

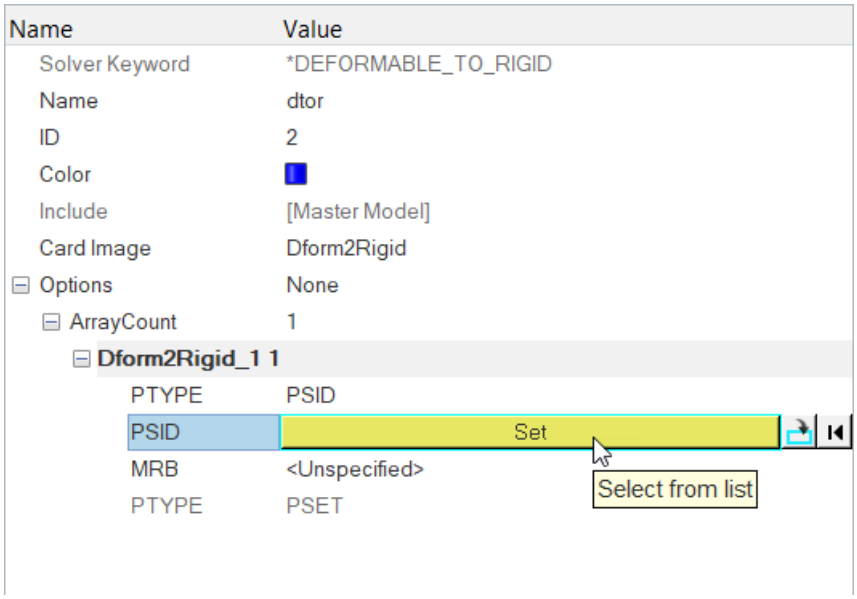


Figure 959:

The **Select Set** dialog opens.

e) In the **Select Set** dialog, select **set_part_seat** and then click **OK**.

f) For MRB, click **Unspecified** > **Component**.

g) In the **Select Component** dialog, select **rigid block** and then click **OK**.

3. Define *DEFORMABLE_TO_RIGID_AUTOMATIC to switch the rigid seat to deformable when contact between the seat and block is detected.

a) In the Solver Browser, right-click and select **Create** > ***DEFORMABLE_TO_RIGID** > ***DEFORMABLE_TO_RIGID_AUTOMATIC** from the context menu.

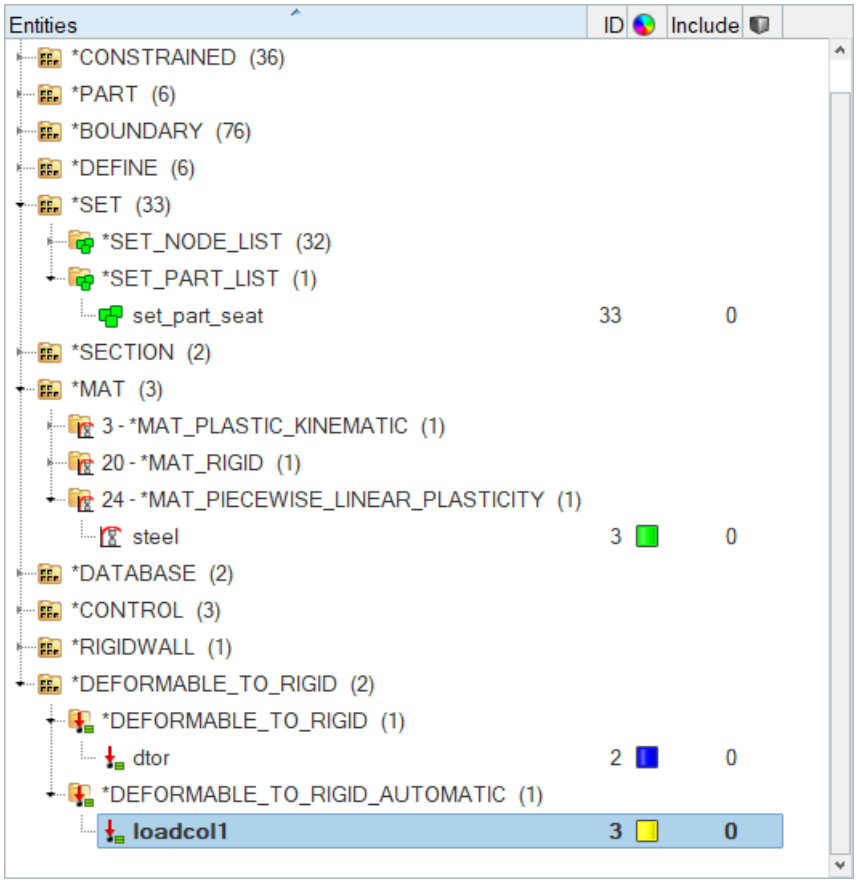


Figure 960:

HyperMesh creates and opens a new load collector in the Entity Editor.

b) For Name, enter `dtor_automatic`.

c) For SWSET (set number of this automatic switch set), enter `1`.

d) Set CODE (activation switch code) to **0: Switch takes place at time 1**.

e) For TIME1, enter `175`.

> **Note:** The switch will not take place before this time.

f) For R2D, select **1**.

> **Note:** On export, the number of rigid parts to be switched to deformable is written to the R2D field (card 2, field 6). This number is based on the number of parts in the entity set you select next.

g) Click **PSID** > **Set**.

> **Note:** PSIDR2D is the part ID of the part which is switched to a rigid material.



Figure 961:

The **Select Set** dialog opens.

h) In the **Select Set** dialog, select **set_part_seat** and then click **OK**.

Review Component Data

In this section, you will review the model's component data using the Model Browser, Solver Browser, or Component Table tool.

Method 1: Use the Model Browser

In this section, you will review the model's component data using the Model Browser.

1. Display only parts of the model with a particular material and section.

   a) In the Model Browser, click .

b) Right-click on **steel** and select **Isolate** from the context menu.

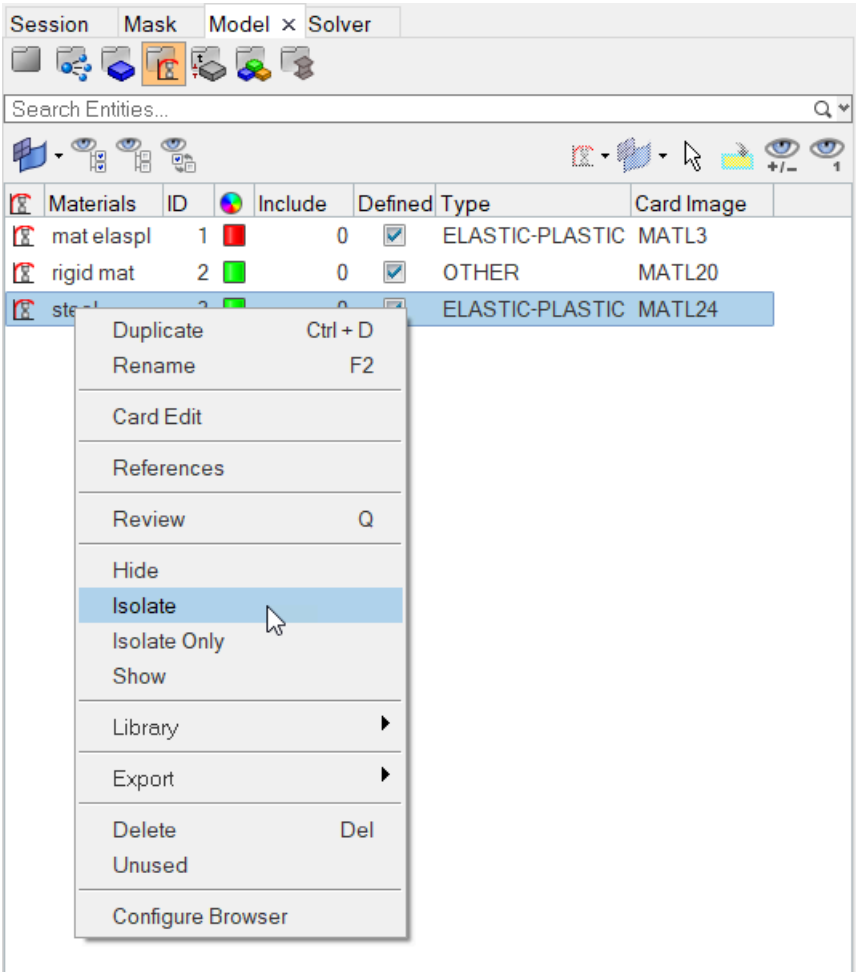


Figure 962:

HyperMesh only displays the components that have the selected material assigned.

c) Click , select a material, and scroll through the material using the up and down arrow keys in the Model Browser to review the materials.

The corresponding parts are automatically isolated in the view.

d) In the Model Browser click .

e) Right-click on **shell section** and select **isolate** from the context menu.

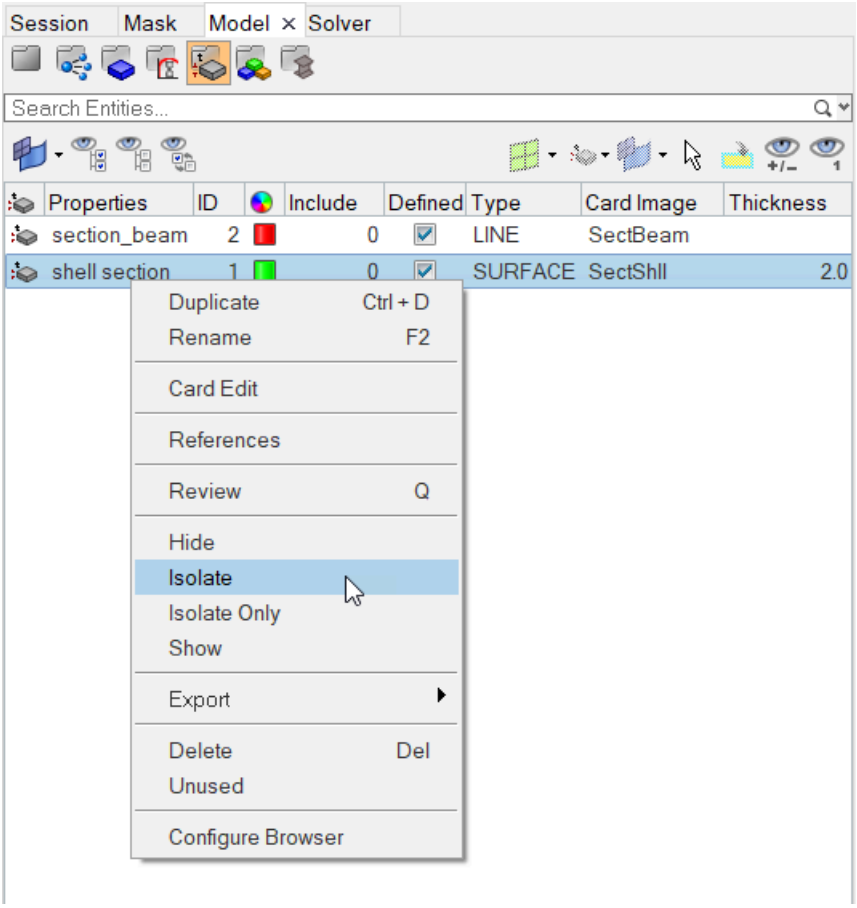


Figure 963:

HyperMesh only displays the components that have the selected section assigned.

2. Display all components.
   a) In the Model Browser, click .
3. Rename a part.
   a) Right-click on the part you would like to rename, and then select **rename** from the context menu.
   b) In the editable field, enter a new name for the entity.
   The part's new name changes in the Solver Browser and the Model Browser.
4. Renumber a part ID.
   a) In the Model Browser, click on a part's ID field.
   The ID field becomes editable.
   b) Enter a number that does not conflict with the existing part IDs, and then press **Enter**.

Method 2: Use the Solver Browser

In this section, you will review the model's component data using the Solver Browser.

1. Display only parts with a particular material.

a) In the Model Browser, Materials folder, right-click on **Steel** and select **Isolate** from the context menu.

b) In the Solver Browser, *SECTION folder, select components based on properties.

2. Display all components.
   a) In the Solver Browser, click the ***MAT** folder.
3. Rename a part.
   a) In the Solver Browser, select the part you would like to rename.
   The Entity Editor opens, and displays the part's corresponding data.
   b) Select **Name**, and enter a new name for the part.
   The part's new name changes in the Solver Browser and the Model Browser.
4. Renumber a part ID.
   a) In the Solver Browser, select the part you would like to change the ID of.
   The Entity Editor opens, and displays the part's corresponding data.
   b) For ID, enter a new ID for the part.
   The part's new ID changes in the Solver Browser and Model Browser.

Method 3: Use the Component Table

In this section, you will review the model's component data using the component table.

1. Display only parts with a particular material.
   a) Enable the Quick Access Tool by clicking **Ctrl** > **F**.
   b) In the Quick Access Tool, enter `Utility Component Table` and press **Enter**.
   The **Components and Properties** dialog opens.
   c) In the **Components and Properties** dialog, click **Display** > **By Material** from the menu bar.
   d) In the panel area, click **mats**.
   e) Select the **steel** checkbox.
   f) Click **Select**.
   g) Click **proceed**.
   The Component Table only displays the components with the material steel assigned. All other components are turned off.
2. Display all components.
   a) From the menu bar, click **Display** > **All**.
   The table displays all of the components in the model.
3. Rename a part.
   a) From the menu bar, click **Table** > **Editable**.

   > **Note:** You can edit any of the columns that have a white background. For example, you can edit Part name, Part id and Thickness.

   The table becomes editable.
   b) Click any Part name field.

The field becomes editable.

c) In the **Confirm** dialog, click **Yes**.
The part's new name changes in the Solver Browser and the Model Browser.

4. Renumber a part ID.
   a) From the menu bar, click **Table** > **Editable**.
   The table becomes editable.
   b) Click any Part Id field.
   The field becomes editable.
   c) Enter a new ID that does not conflict with any existing part IDs.
   d) In the **Confirm** dialog, click **Yes**.
   The part's new ID changes in the Solver Browser and the Model Browser.

Review the Model Data Using the Solver Browser

In this step, you will review the model data using the Solver Browser.

The created solver entities are listed in the Solver Browser, within their corresponding folders.

1. In the Solver Browser, *DEFORMABLE_TO_RIGID folder, right-click on **dtor** and select **Isolate Only** from the context menu.
2. Highlight the entities that are referenced by this keyword by right-clicking on **dtor** and selecting **Review** from the context menu.
3. Right-click on the ***BOUNDARY** folder and then select **Show** from the context menu.

5.2.2 Exercise 2: Define Boundary Conditions and Loads for the Seat Impact Analysis

In this exercise, you will define boundary conditions and load data for an LS-DYNA analysis of a vehicle seat impacting a rigid block.

This exercise will help you become familiar with defining LS-DYNA boundary conditions and loads using HyperMesh. The seat and block model are shown in Figure 964.

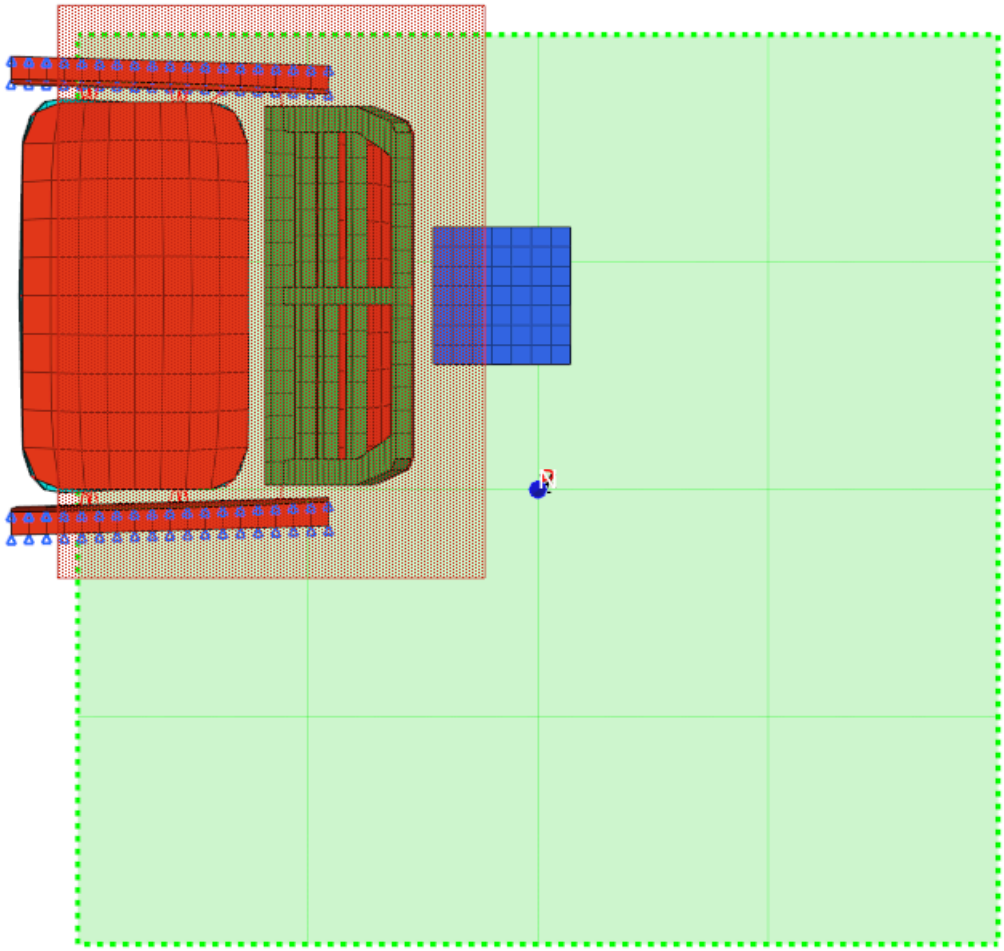


Figure 964:

Load the LS-DYNA User Profile

In this step, you will load the LS-DYNA user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Retrieve the HyperMesh File

In this step, you will open the model file in HyperMesh.

1. Open the model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
2. In the **Open Model** dialog, open the `seat_2.hm` file.

The model appears in the graphics area.

3. Observe the model using various visual options available in HyperMesh, such as rotation and zooming.

Define Gravity Acting in the Negative Z-Direction

In this step, you will define gravity acting in the negative z-direction with *LOAD_BODY_Z.

1. In the Solver Browser, right-click and select **Create** > ***LOAD** > ***LOAD_BODY_Z** from the context menu.

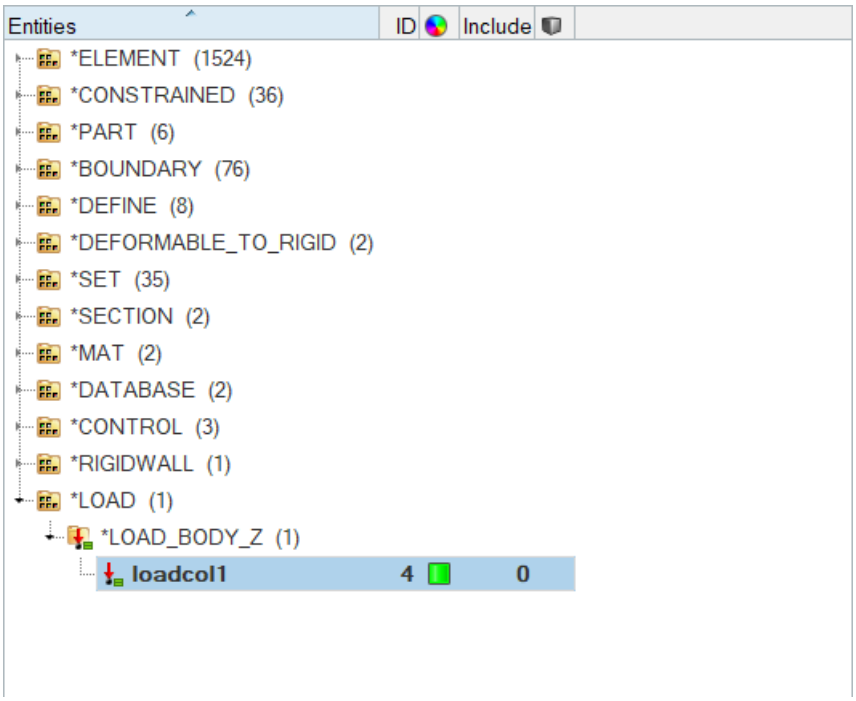


Figure 965:

HyperMesh creates and opens a new load collector in the Entity Editor.

2. For Name, enter `gravity`.
3. Click **LCID**, and then click **curve**.

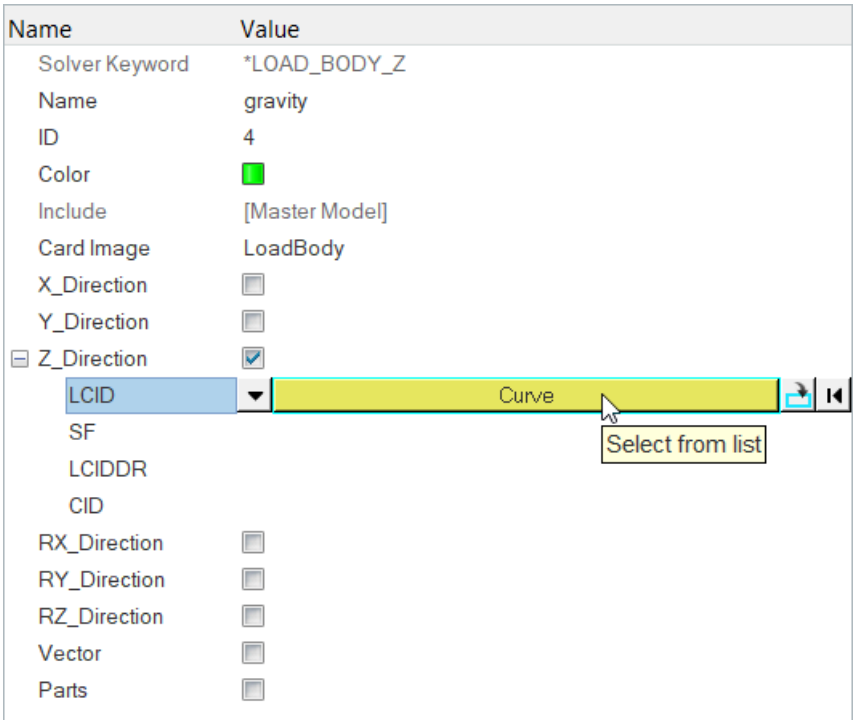


Figure 966:

The **Select Curve** dialog opens.

4. In the **Select Curve** dialog, select **gravity curve** and then click **OK**.

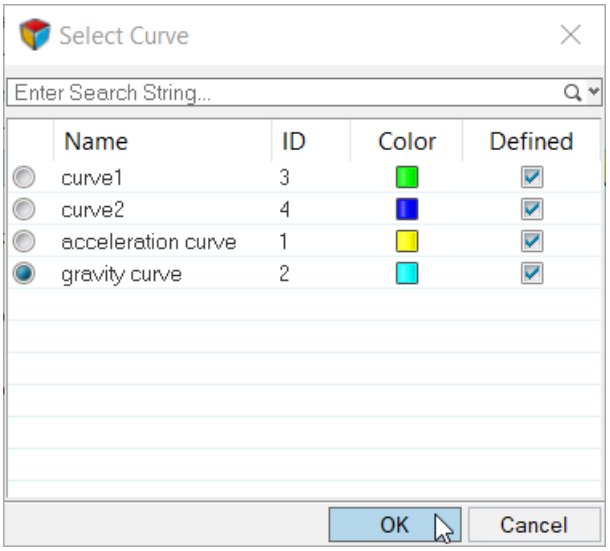


Figure 967:

5. For SF (scale factor for acceleration in z-direction), enter `0.001`.

Define the Seat Acceleration

In this step, you will define the seat acceleration with *BOUNDARY_PRESCRIBED_MOTION_NODE.

1. Create a load collector.
   a) In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.

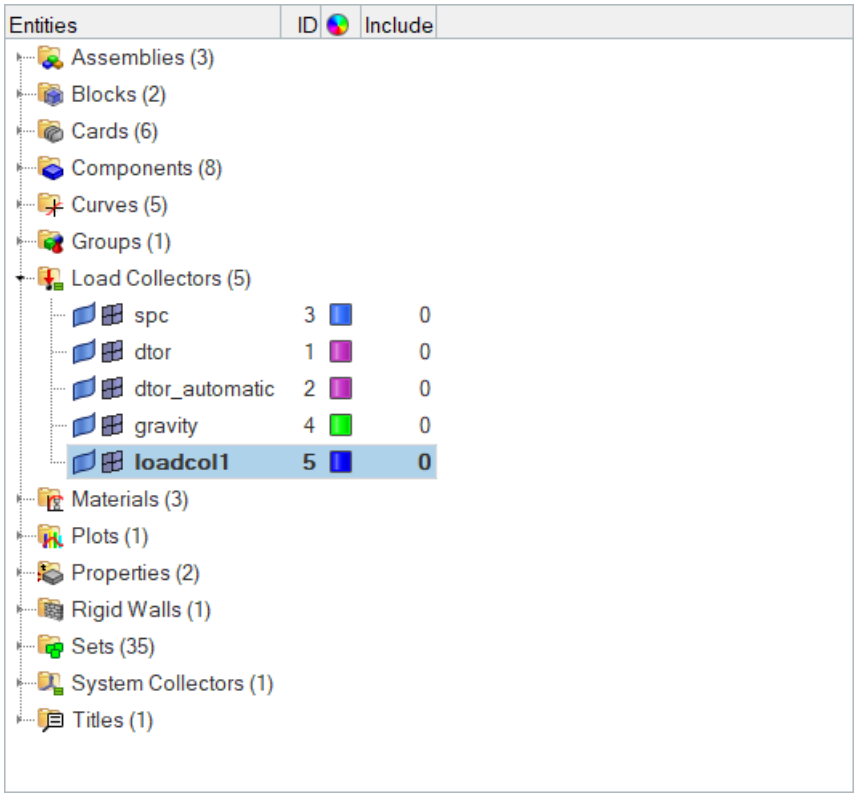


Figure 968:

   HyperMesh creates and opens a new load collector in the Entity Editor.
   b) For Name, enter `accel`.
   c) Set card image to **<None>**.
   d) Optional: Click the **Color** icon, and select a color for the load collector.
2. Create acceleration loads on nodes.
   a) Click **BCs** > **Create** > **Accelerations** from the menu bar.
   The Accelerations panel opens.
   b) Click **load types =**, and select **PrcrbAcc_S**.
   c) Click **sets**.
   d) Select the **accel_nodes** checkbox.
   e) Click **select**.
   f) Click the **magnitude=** switch, and select **curve, vector**.
   g) In the magnitude= field, enter `0.001`.

> **Note:** This is the scale factor for the pre-defined curve to be specified in the next step for the acceleration loads. It will define the seat's acceleration as a function of time.

h) Set the orientation selector to **x-axis**.

> **Note:** This is the x-translational degree of freedom.

i) Double-click **curve**.

j) Select the **acceleration curve** curve.

k) In the magnitude% = field, enter `1.0E+7`.

> **Note:** This is the scale factor for the graphical representation of the acceleration loads. It does not affect the actual acceleration value.

l) Click **create**.
HyperMesh creates the acceleration loads.

m) Click **return**.

Export the Model

In this step, you will export the model to an LS-DYNA 971 formatted input file.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
2. In the Export - Solver Deck tab, set File type to **Ls-Dyna**.
3. In the File field, navigate to your working directory and save the file as `seat_complete.key`.
4. Click **Export**.

Submit the Input File

In this step, you will submit the LS-DYNA Input File to LS-DYNA 971.

1. From the Start Menu on your desktop, open the LS-DYNA Manager program.
2. From the solvers menu, select **Start LS-DYNA analysis**.
3. Load the `seat_complete.key` file.
4. Click **OK** to start the analysis.

View the Results in HyperView

In this step, you will view the results in HyperView.

Save your work as a HyperMesh file.

5.3 HM-4615: Model Import, Airbags, Export Displayed, and Contacts with LS-DYNA

In this tutorial you will define the following: *AIRBAG_WANG_NEFSKE for the airbag mesh geometry; initial velocity of 3 mm/ms in the negative x-direction for the head with *INITIAL_VELOCITY_GENERATION; a contact between the airbag and head with *ICONTACT_AUTOMATIC_SURFACE_TO_SURFACE; *CONTACT_AIRBAG_SINGLE_SURFACE for the airbag; and a contact between the plate and the airbag with *CONTACT_NODES_TO_SURFACE.

Before you begin, copy `airbag_start.key` from `<hm.zip>/interfaces/lsdyna/` to your working directory.

This tutorial will help you learn how to define LS-DYNA airbags, loads, and contacts using HyperMesh. The head and airbag are depicted in Figure 969.

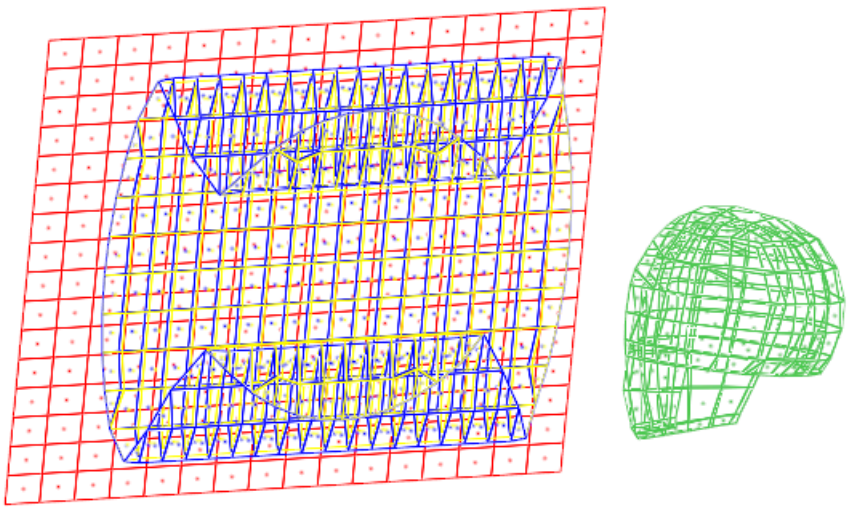


Figure 969:

5.3.1 LS-DYNA Overview

In this section, you will learn how to import a LS-DYNA model, export displayed, and create and review contacts.

Import a LS-DYNA Model

Warning and Error Messages

When importing a LS-DYNA model, HyperMesh warning and error messages will be written to a file named `dynkey.msg` or `dynaseq.msg`, depending on the FE input translator being used. This file is created in the same folder from which HyperMesh was started.

Unsupported Cards

On import, the LS-DYNA cards not supported by HyperMesh are written to the unsupp_cards panel. Access this panel by clicking **Setup** > **Create** > **Control Cards** from the menu bar. Unsupported cards will be exported with the remaining model.

Care should be taken if an unsupported card points to an entity in HyperMesh. An example of this is when an unsupported material references a *PART. HyperMesh stores unsupported cards as text and does not consider pointers.

LSTC Dummy Files

You can read LSTC Hybrid III dummy files into HyperMesh by first converting the tree file to FTSS/ARUP tree file format.

Include Files

HyperMesh supports *INCLUDE. When Include files are imported into HyperMesh, the IDs of non-existing entities are maintained and will not be used for new entities. Use the Include files import option to specify whether to merge, preserve, or skip Include files on import. Access this option by clicking **File** > **Import** > **Solver Deck** from the menu bar.

Support of Include files:

- The LS-DYNA keywords *INCLUDE, *INCLUDE_STAMPED_PART, *INCLUDE_STAMPED_PART_SET, *INCLUDE_TRANSFORM, and *INCLUDE_COMPENSATION_OPTION are mapped to Include files.
- Switch to different types of Include files using the context menu Include File option in the Model Browser, with the exception of *INCLUDE_TRANSFORM, *INCLUDE_STAMPED_PART, and *INCLUDE_STAMPED_PART_SET.
- Manage INCLUDE_TRANSFORM using the Transformation Manager.
- During import, if the same Include file is referenced more than once using the *INCLUDE_TRANSFORM, then it will be imported, but appended with .#, where # = 1…n is shown in the Model Browser. These will not be exported unless you clear the Instance checkbox.
- During import, *INCLUDE_STAMPED_PART and *INCLUDE_STAMPED_PART_SET are imported as "read only" by default to preserve the associativity.
- During import, an Include file can be read when *INCLUDE comes after *INCLUDE_TRANSFORM.
- When importing Include files, use Solver options in the Import browser to choose the type of Include file to import. This option enables you to set the Include file type to one of the following: INCLUDE,

INCLUDE_STAMPED_PART, INCLUDE_STAMPED_PART_SET, or INCLUDE_COMPENSATION_options. This is the only option which enanables you to attach any file of type INCLUDE_STAMPED_PART or INCLUDE_STAMPED_PART_SET.

Export Displayed

From the Export - Solver Deck tab, select the Export Displayed option to export only displayed nodes and elements. Only model data associated to the displayed nodes and elements are exported. This model data includes materials and their associated curves, properties, portions of contacts, and output requests.

Create and Review Contacts

Table 3 describes how all secondary and main set types are created and specified in contacts.

Table 3:

Secondary and Main Set Type	LS-DYNA card	Panel Used to Create Card	Equivalent type in Interfaces panel, add subpanel
EQ. 0: set segment id	*SET_SEGMENT	set_segment (contactsurfs) or …	csurfs
		Interfaces, add subpanel	entity
EQ. 1: shell element	*SET_SHELL_Op	Entity Sets or…	sets
set id	n	Interfaces, add subpanel	subpanel entity
EQ. 2: part set id	EQ. 2: part set id	Entity Sets or… Interfaces, add subpanel	split comps
EQ. 3: part id	*PART	Collectors	comps
* EQ. 4: node set id	*SET_NODE_Opt	Entity Sets or…	sets
		Interfaces, add subpanel	entity
* EQ. 5: include all		Interfaces, add subpanel	all
* EQ. 6: part set id for exempted parts	*SET_PART_LIST	Interfaces, add subpanel and then card image subpanel	sets

Add Subpanel	While the Interfaces panel, add subpanel has several main and secondary entity types to choose from in order to specify the LS-DYNA main or secondary set for a *CONTACT, only the valid main and secondary types are selectable for the particular contact you are creating. When the main or secondary type is set to comps and only one component is selected, the LS-DYNA type is 3, part ID, and *PART is created. When multiple components are selected, the LS-DYNA type is 2, part set ID, and *SET_PART_LIST is created. When the main or secondary type is set to sets, only those sets valid for the particular contact you are creating are selectable. For example, for *CONTACT_NODES_TO_SURFACE, only a list of node sets is available for secondary; you will not see a list of other set types, like element or part sets.
Review Contacts	Review contacts by clicking review in the Interfaces panel, add subpanel.

5.3.2 Load the LS-DYNA User Profile

In this step, you will load the LS-DYNA user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

5.3.3 Import the LS-DYNA Model

In this step, you will import the LS-DYNA model file into HyperMesh.

1. From the menu bar, click **File** > **Import** > **Solver Deck**.
   The Import - Solver Deck tab opens.
2. In the File field, open the `airbag_start.key` file.
3. Click **Import**.
   The **Import Process Messages** dialog opens.
4. Select **Close** to close the **Import Process Messages** dialog.

5.3.4 Define *AIRBAG_WANG_NEFSKE

In this step, you will define *AIRBAG_WANG_NEFSKE for the airbag mesh geometry.

1. Create a set of parts, *SET_PART_LIST, containing the AirbagFront and AirbagRear components.
   a) In the Model Browser, right-click and select **Create** > **Set** from the context menu.

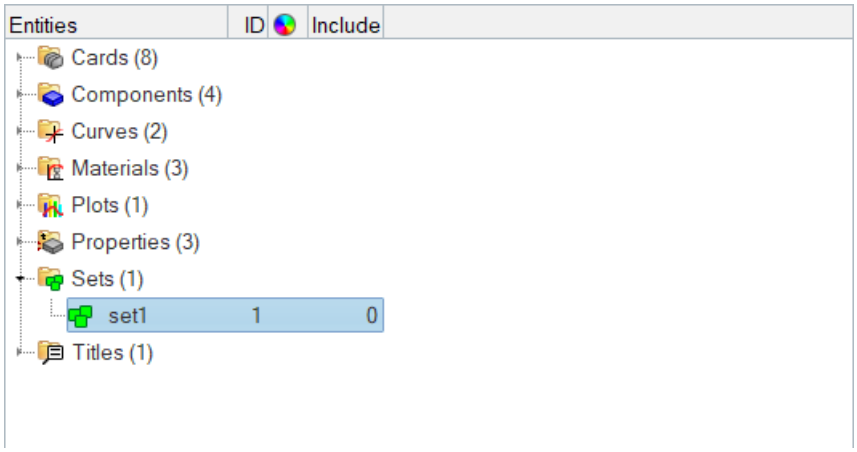


Figure 970:

A new set opens in the Entity Editor.

b) For Name, enter `airbag_set`.

c) Set card image to **Part**.

d) For Entity IDs, click **0 Components** > **Components**.

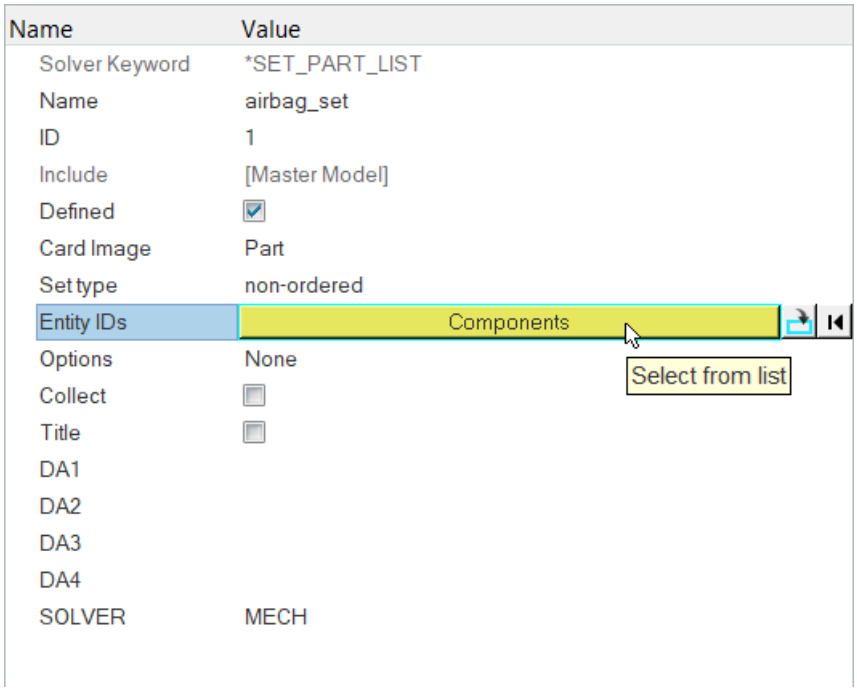


Figure 971:

e) In the **Select Components** dialog, select **AirbagFront** and **AirbagRear** and then click **OK**.

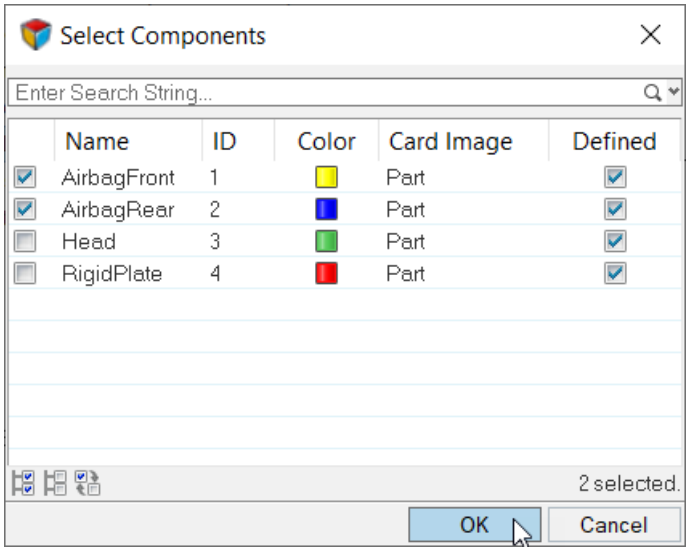


Figure 972:

2. Define the airbag (*AIRBAG_WANG_NEFSKE).

a) Open the Solver Browser by clicking **View** > **Browsers** > **HyperMesh** > **Solver** from the menu bar.

b) In the Solver Browser, right-click and select **Create** > ***AIRBAG** > ***AIRBAG_WANG_NEFSKE** from the context menu.

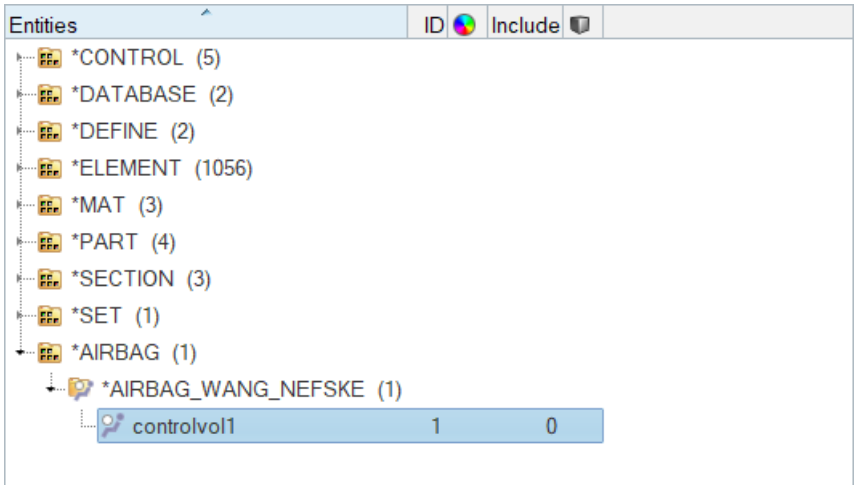


Figure 973:

A new control volume opens in the Entity Editor.

c) For Name, enter `airbag`.

d) Click **SID**.
The entity selector becomes active.

e) Set the entity selector to **Sets** as seen in Figure 974.

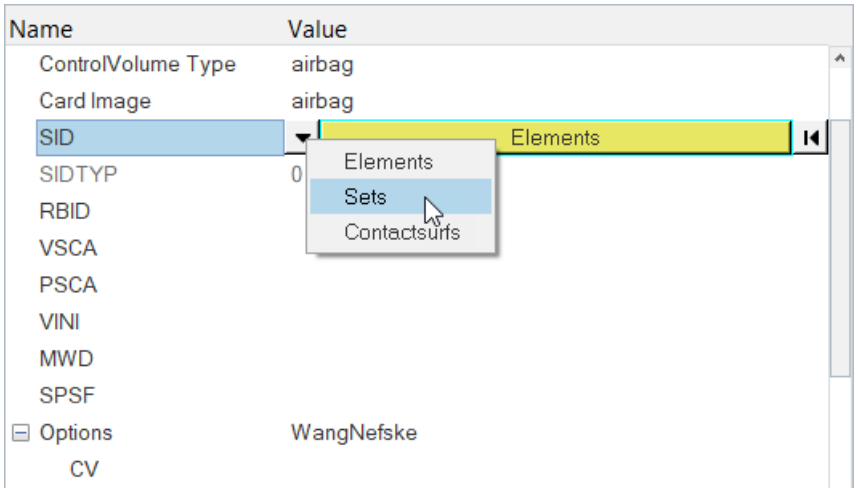


Figure 974:

f) Click **Set**.

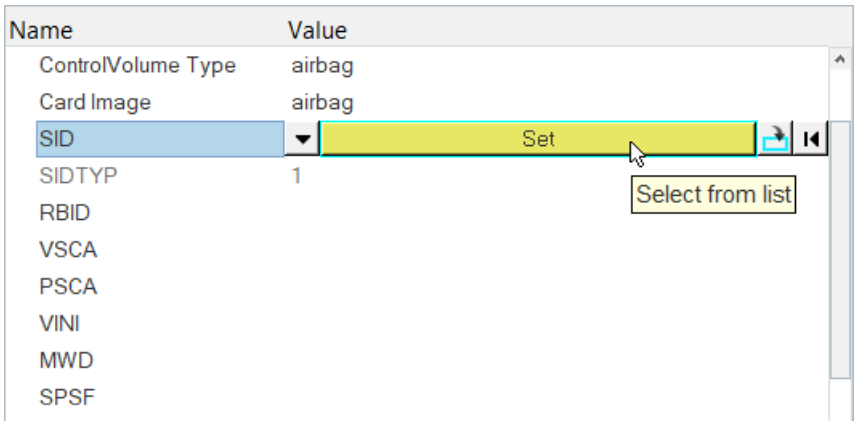


Figure 975:

g) In the **Select Set** dialog, select **airbag_set** and then click **OK**.

Note: The parts in this set define the airbag's geometry.

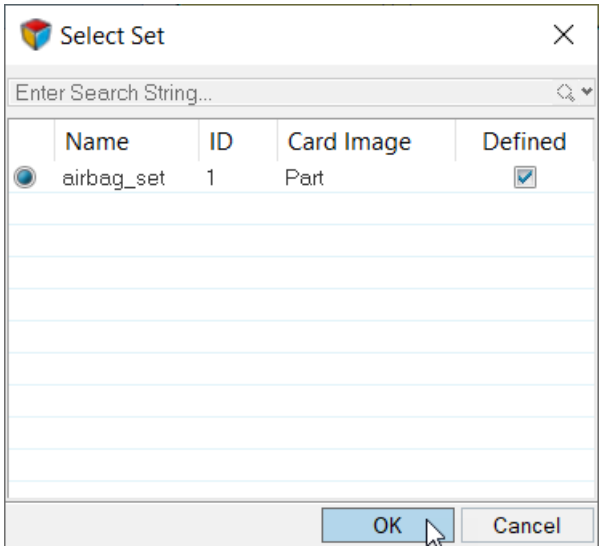


Figure 976:

h) For CV (Heat capacity at constant volume), enter `1023.0`.
i) For CP (Heat capacity at constant pressure), enter `1320.0`.
j) For T (Temperature of input gas), enter `780.0`.
k) Click **LCMT (Load curve specifying input mass flow rate)** > **Curve**.
l) In the **Select Curve** dialog, select **airbag LCMT curve** and then click **OK**.
m) For C23 (Vent orifice coefficient), enter `1.0`.
n) Click **LCA23 (Load curve defining vent orifice area as a function of pressure)** > **Curve**.
o) In the **Select Curve** dialog, select **airbag LCA23 curve** and then click **OK**.
p) For CP23 (Orifice coefficient for leakage), enter `1.0`.
q) For PE (Ambient pressure), enter `1.0E-4`.
r) For RO (Ambient density), enter `1.0E-9`.
s) For GC (Gravitational conversion constant), enter `1.0`.

3. Define an initial velocity of 3 mm/ms in the negative xdirection for the head with *INITIAL_VELOCITY_GENERATION.

a) In the Model Browser, right-click and select **Create** > **Load sCollector** from the context menu.

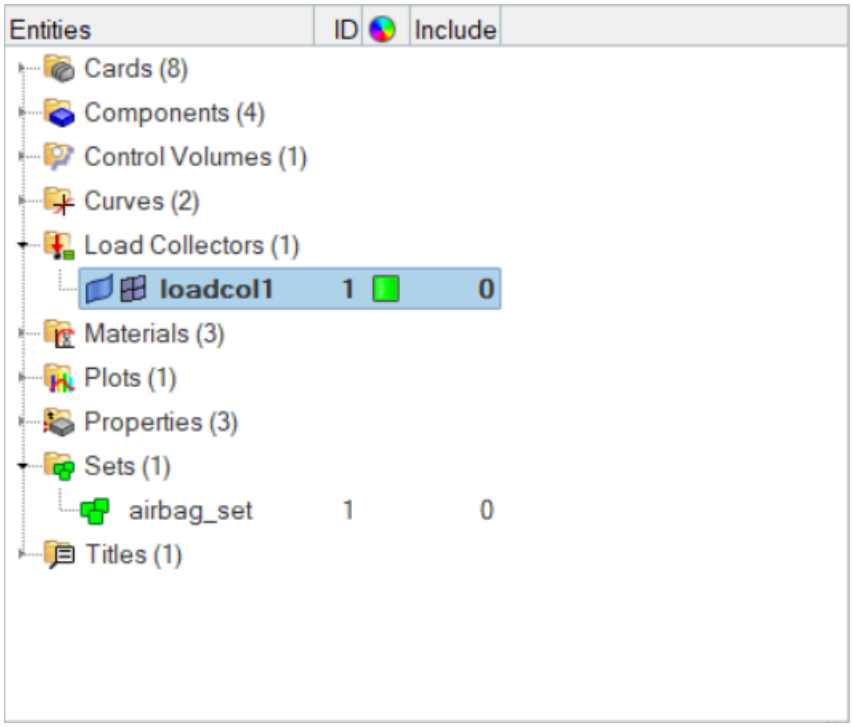


Figure 977:

A new load collector opens in the Entity Editor.

b) For Name, enter `velocity`.

c) Set card image to **InitialVel**.

d) Under Options, right-click on **NSID** and select **Create** from the context menu.

The **Create Sets** dialog opens.

e) For Entity IDs, click **0 Nodes** > **Nodes**.

> **Note:** The **Create Sets** dialog will remain open for steps 3.f through 3.i. If necessary, move the **Create Sets** dialog to access the panel area.

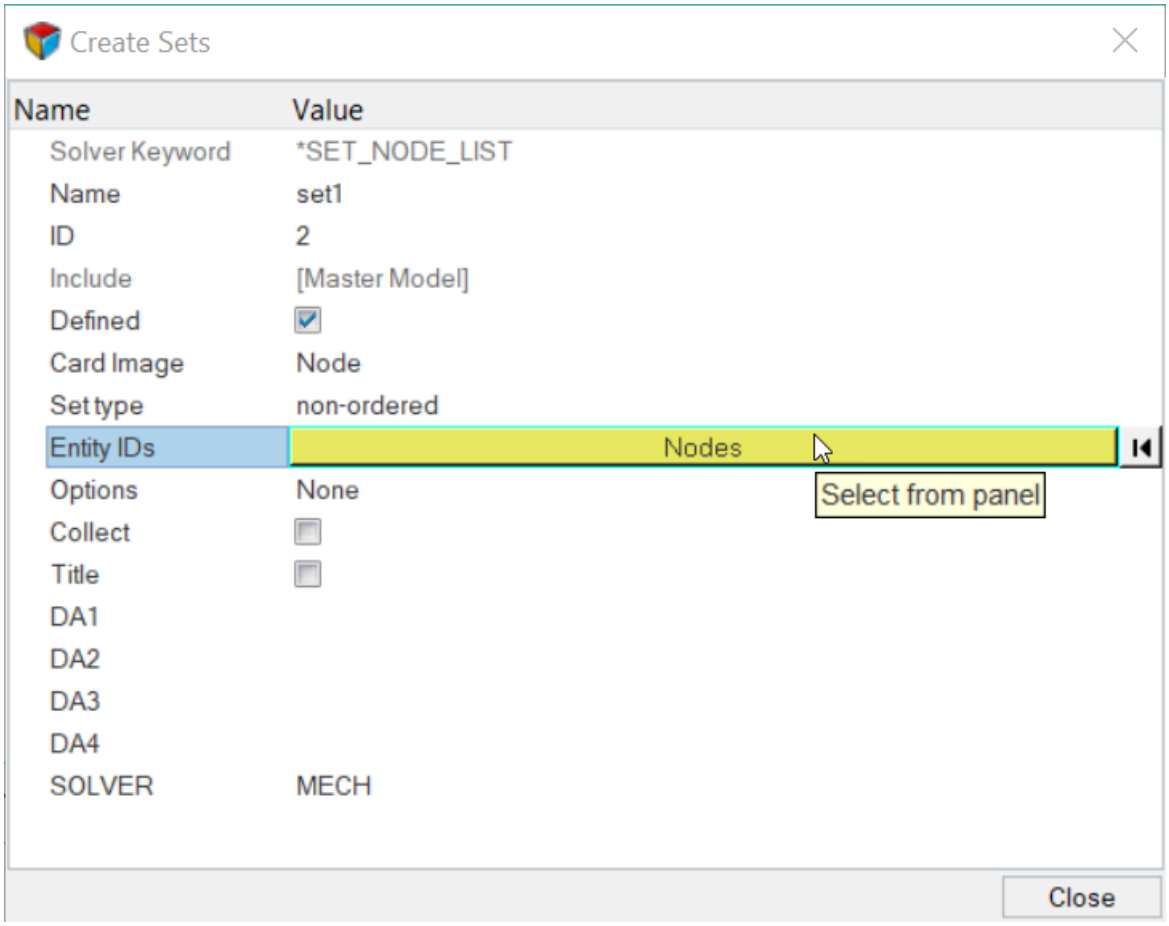


Figure 978:

f) In the panel area, click **nodes** > **by Collector**.

g) Select the **Head** checkbox as seen in Figure 979.

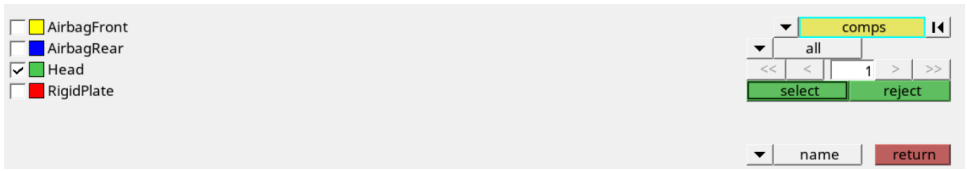


Figure 979:

h) Click **select**.

i) Click **proceed**.

j) In the **Create Sets** dialog, click **Close**.

k) In the Solver Browser, *INITIAL folder, *INITIAL_VELOCITY subfolder, right-click on **velocity** and select **Review** from the context menu.
HyperMesh highlights the load collector and greys out all of the other entities as seen in Figure 980.



Figure 980:

l) Right-click on **velocity** and select **Review** from the context menu.
The entities return to their original display color.

5.3.5 Define a Contact Between the Airbag and the Head

In this section, you will define a contact between the airbag and the head with *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE.

1. Create a HyperMesh group with the card image SurfaceToSurface.

a) In the Solver Browser, right-click and select **Create** > ***CONTACT** > ***CONTACT_AUTOMATIC_SURFACE_TO_SURFACE** > ***CONTACT_AUTOMATIC_SURFACE_TO_SURFACE** from the context menu.

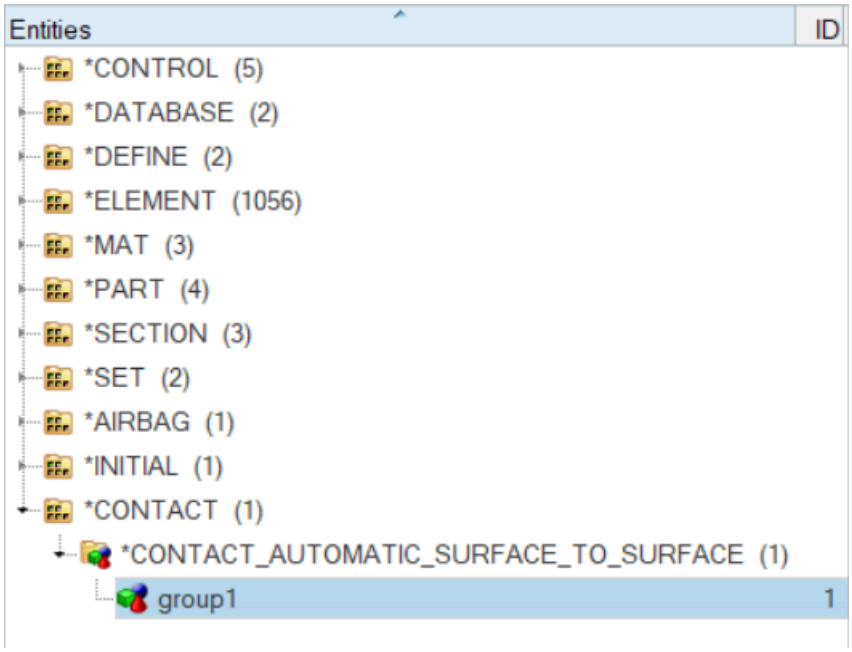


Figure 981:

A new group opens in the Entity Editor.

b) For Name, enter `Airbag_Head`.

2. Specify the head to be the main surface with a surface type of 3, part ID.

> **Tip:** In this step, the Entity Editor should still be open for the Airbag_Head group.

a) Click **MSID**.

b) Set the entity selector to **Components**.

c) Click **Components**.

d) In the **Select Components** dialog, select **Head** and then click **OK**.

3. Specify the airbag to be the secondary surface with a surface type of 2, part set ID.

> **Tip:** In this step, the Entity Editor should be open for the Airbag_Head group.

a) Click **SSID**.

b) Set the entity selector to **Set**.

c) Click **Set**.

d) In the **Select Set** dialog, select **airbag_set** and then click **OK**.

> **Note:** This set contains the components, AirbagFront and AirbagRear.

4. View the main and secondary entities.

> **Tip:** In this step, the Airbag_Head group should be selected in the Solver Browser.

a) In the Solver Browser, right-click on **Airbag_Head** and select **Review** from the context menu.
The main and secondary entities temporarily display in blue and red. The other entities temporarily display grey.

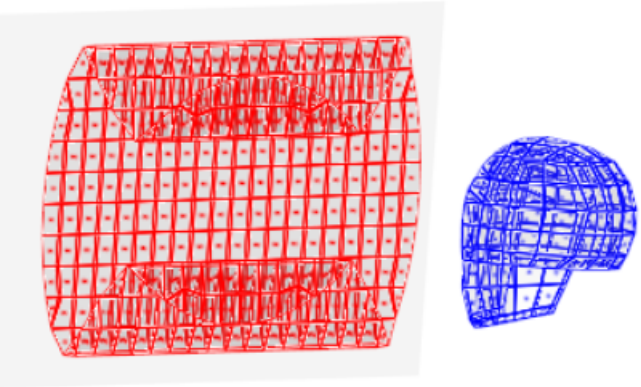


Figure 982:

b) Right-click on **Airbag_Head** and select **Review** from the context menu.
The entities return to their original display color.

5. Define *CONTACT_AIRBAG_SINGLE_SURFACE for the airbag.

a) In the Solver Browser, right-click and select **Create** > ***CONTACT** > ***CONTACT_AIRBAG_SINGLE_SURFACE** from the context menu.

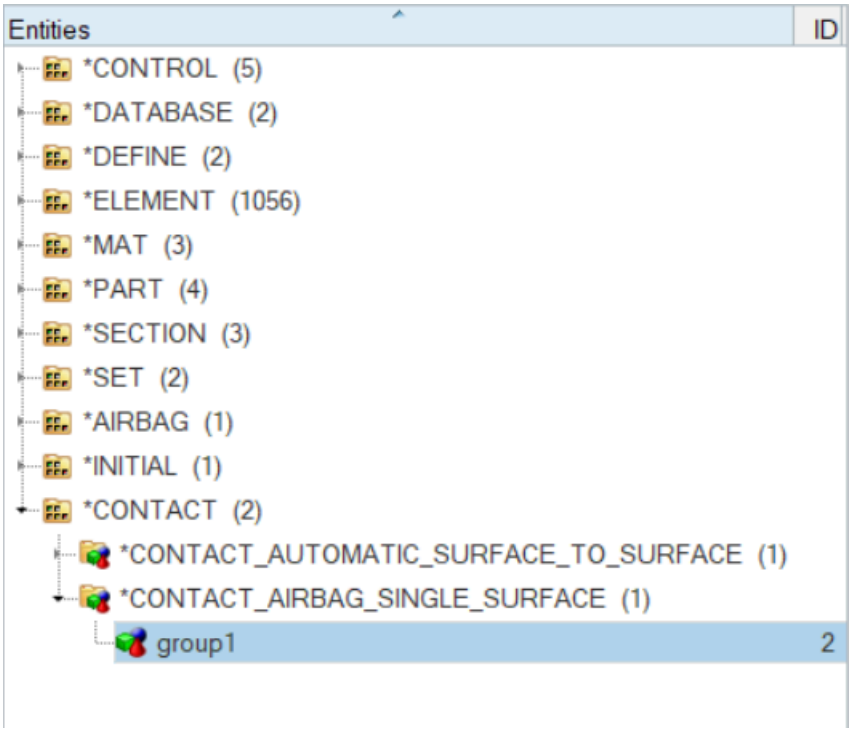


Figure 983:

A new group opens in the Entity Editor.

b) For Name, enter `airbag`.

6. Define the airbag to be a secondary surface with a secondary set type of 2, part set ID.

> **Tip:** In this step, the Entity Editor should be open for the airbag group.

a) Click **SSID**.

b) Set the entity selector to **Set**.

c) Click **Set**.

d) In the **Select Set** dialog, select **airbag_set** and then click **OK**.

7. View the secondary entities.

> **Tip:** In this step, the airbag group should be selected in the Solver Browser.

a) In the Solver Browser, right-click on **airbag** and select **Review** from the context menu.
The main and secondary entities temporarily display white and red. The other entities temporarily display grey.

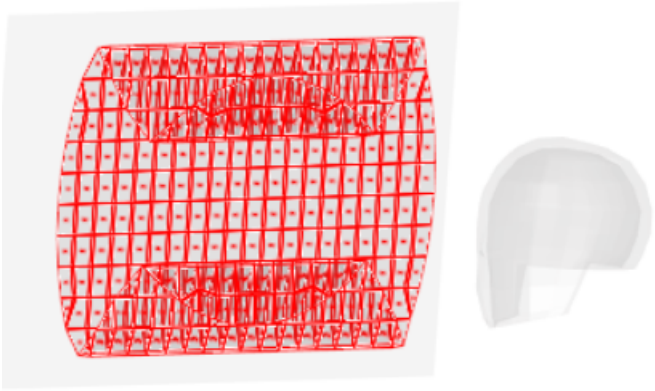


Figure 984:

b) Right-click on **airbag** and select **Review** from the context menu.
The entities return to their original display color.

5.3.6 Define a Contact Between the Plate and the Airbag

In this step, you will define a contact between the plate and the airbag with *CONTACT_NODES_TO_SURFACE.

1. Define the AirbagRear component to be the master surface with a master type of 0, set segment ID.

a) In the Solver Browser, right-click and select **Create** > ***SET** > ***SET_SEGMENT** > ***SET_SEGMENT** from the context menu.

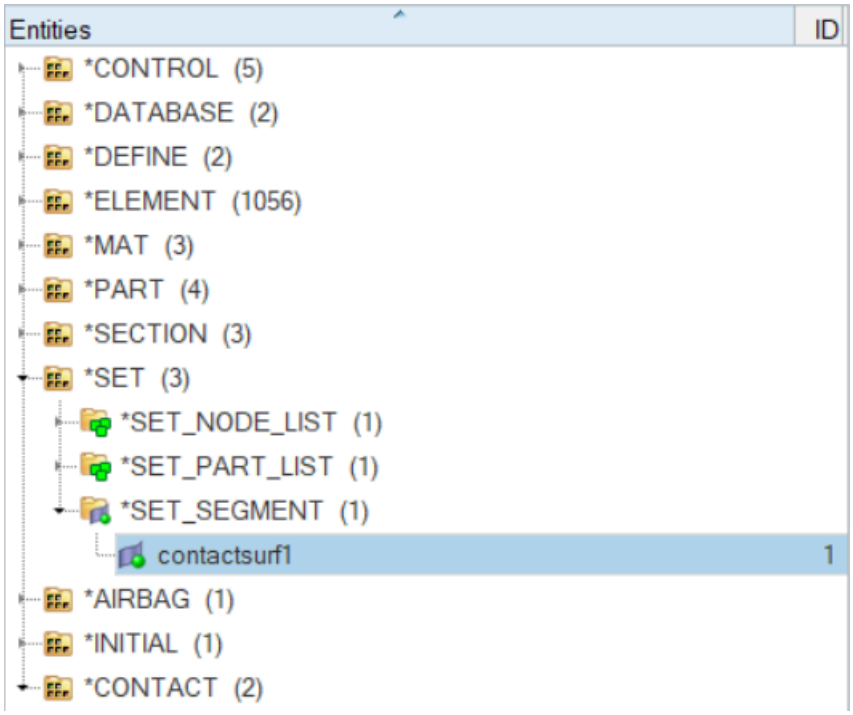


Figure 985:

A new contactsurf opens in the Entity Editor.

b) For Name, enter `AirbagRear_master`.

c) Optional: Click the **Color** icon and select a color for the contactsurf.

d) For Elements, click **0 Elements** > **Elements**.

e) In the panel area, set the second switch to **elems** as seen in Figure 986.

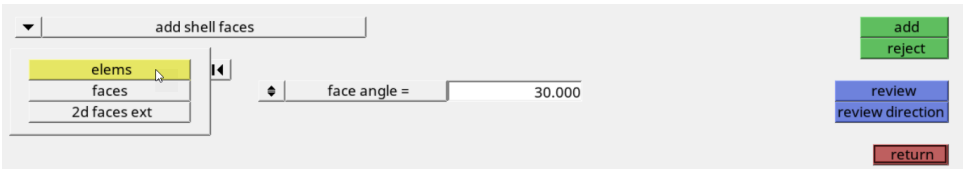


Figure 986:

f) Click **elems** > **by collector**.

g) Select the **AirbagRear** checkbox.

h) Click **select**.

i) Click **add**.

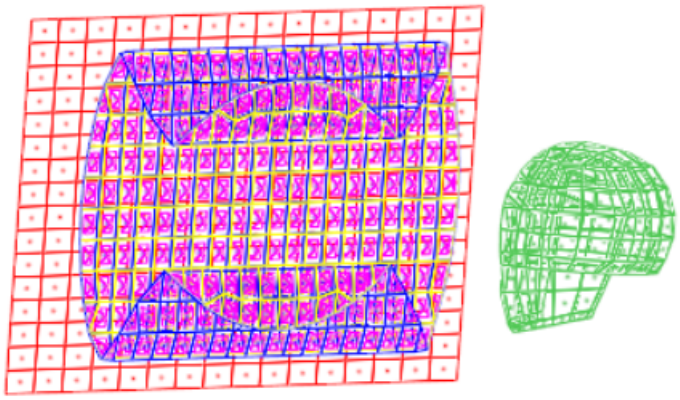


Figure 987:

2. Reverse the contactsurf's pyramids so they point out of the airbag.

a) In the panel area, set the first switch to **adjust contact direction** as seen in Figure 988.

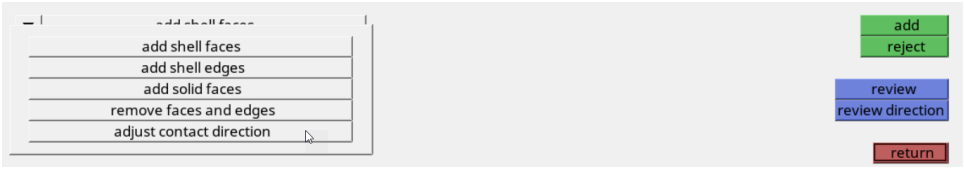


Figure 988:

b) Select the **all elements** checkbox.

c) Click **reverse**.

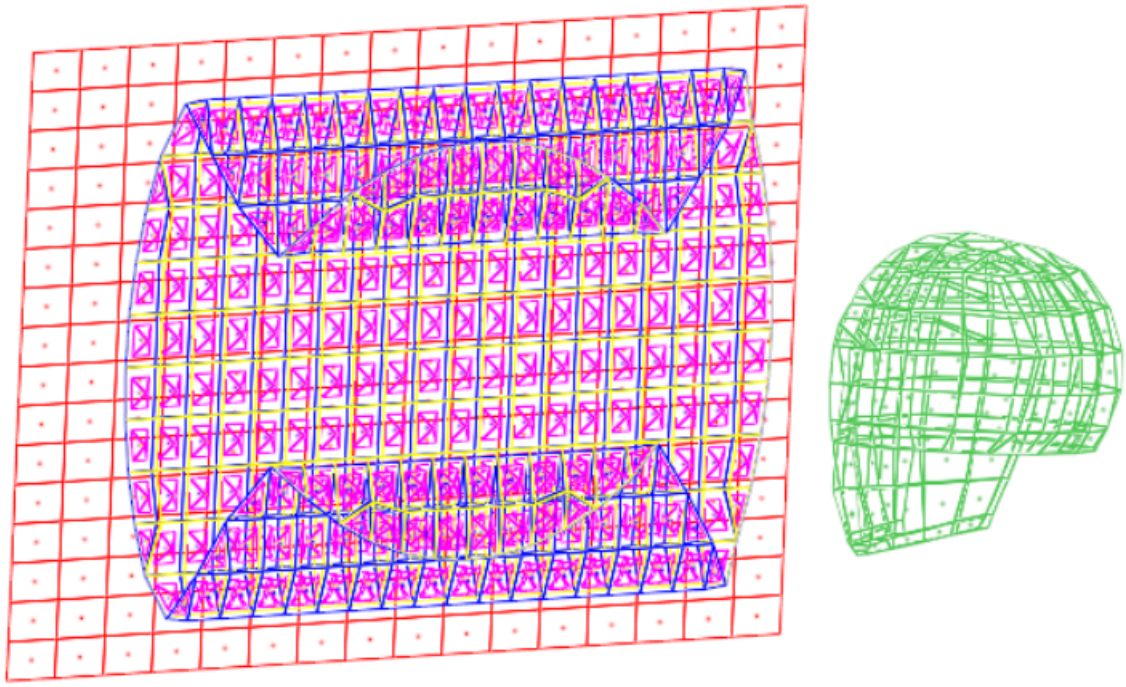


Figure 989:

d) Click **return**.

3. Create a *CONTACT_NODES_TO_SURFACE card.

a) In the Solver Browser, right-click and select **Create** > ***CONTACT** > ***CONTACT_NODES_TO_SURFACE** > ***CONTACT_NODES_TO_SURFACE** from the context menu.

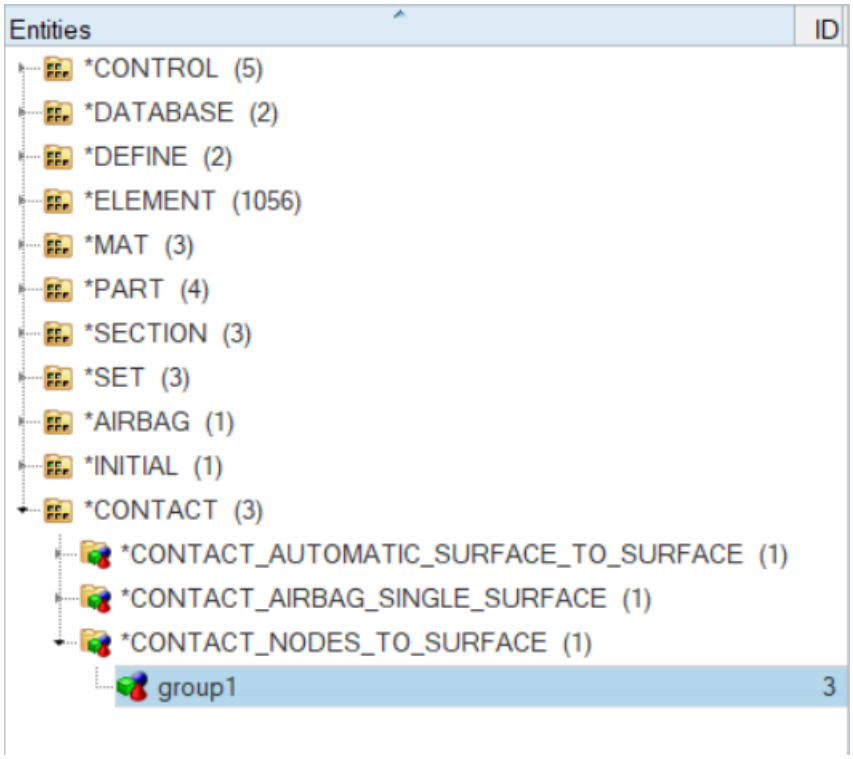


Figure 990:

A new group opens in the Entity Editor.

b) For Name, enter `Airbag_Plate`.

4. Specify the AirbagRear_master contactsurf for the contact's master surface.

> **Tip:** In this step, the Entity Editor should be open for the Airbag_Plate group.

a) Click **MSID**.

b) Set the entity selector to **Contactsurfs**.

c) Click **Contactsurfs**.

d) In the **Select Contactsurfs** dialog, select **AirbagRear_master** and then click **OK**.

5. Define the plate to be the contact's slave surface with a slave type of 4, node set ID.

a) For SSID, click **0 Nodes** > **Nodes**.

b) In the panel area, set the switch to **nodes**.

c) Click **nodes** > **by collector**.

d) Select the **RigidPlate** component.

e) Click **select** as seen in Figure 991.

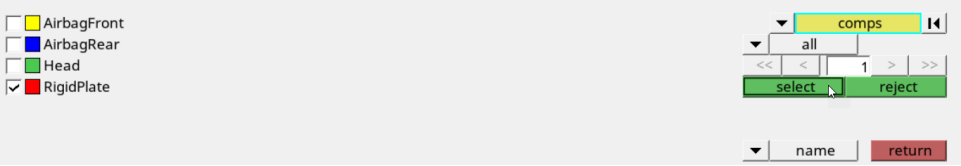


Figure 991:

f) Click **add**.
HyperMesh adds the slave selection to the group Airbag_Plate as seen in Figure 992.

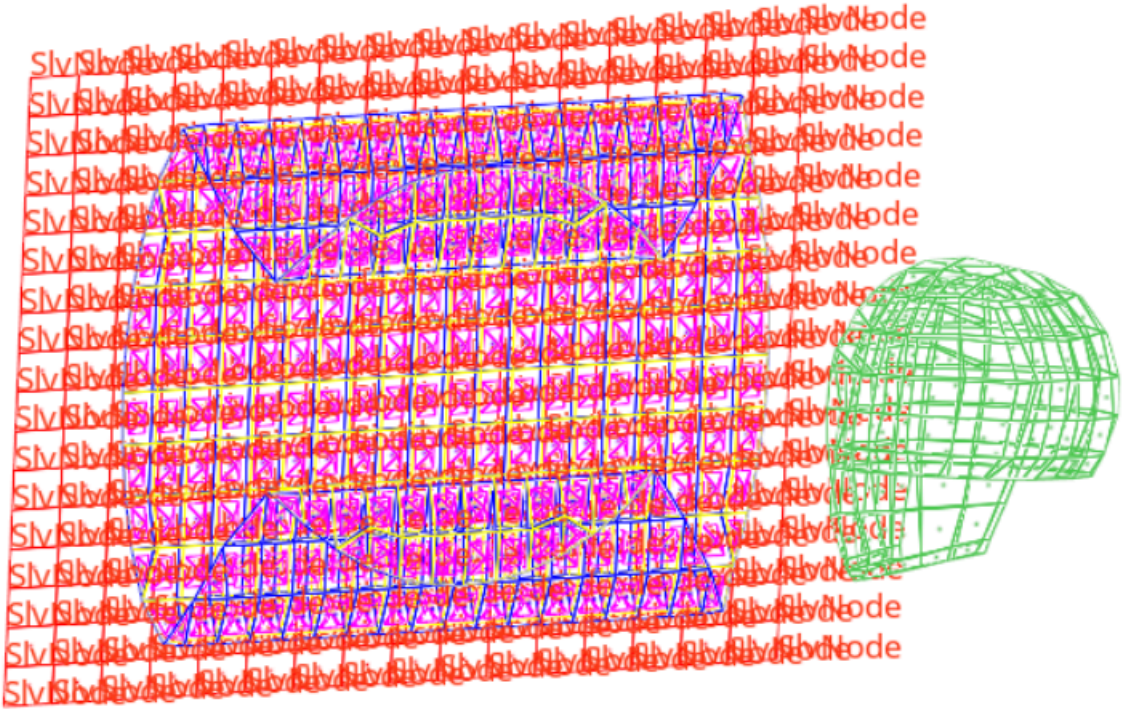


Figure 992:

g) Click **return**.

6. View the master and slave entities.

a) In the Solver Browser, *CONTACT folder, *CONTACT NODES TO SURFACE subfolder, right-click on **Airbag_Plate** and select **Review** from the context menu.
The master and slave entities temporarily display blue and red. The other entities temporarily display grey.

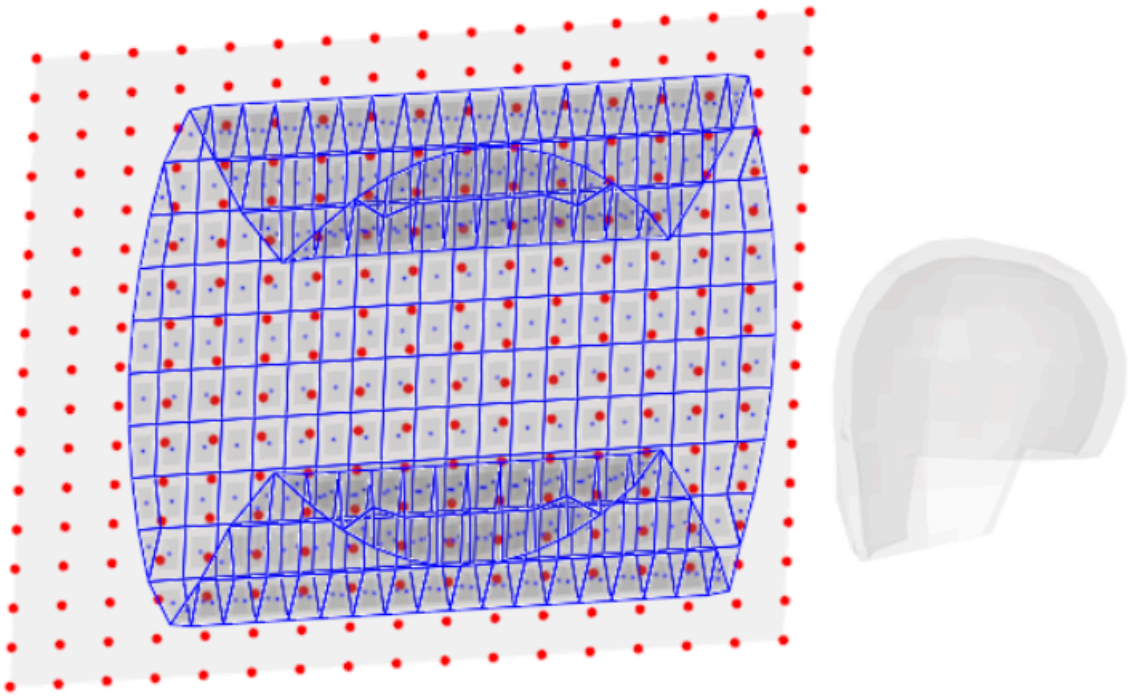


Figure 993:

b) Right-click on **Airbag_Plate** and select **Review** from the context menu.
The entities return to their regular display color.

5.3.7 Review the Created Solver Entities

In this step, you will review the created solver entities using the Solver Browser.

1. In the Solver Browser, *CONTACT folder, *CONTACT_AIRBAG_SINGLE_SURFACE subfolder, right-click on **airbag** and select **Review** from the context menu.

   Note: Only slave (red) entities are shown because there are no master entities for this type of contact.

   The master and slave entities temporarily display blue and red. The other entities temporarily display grey.

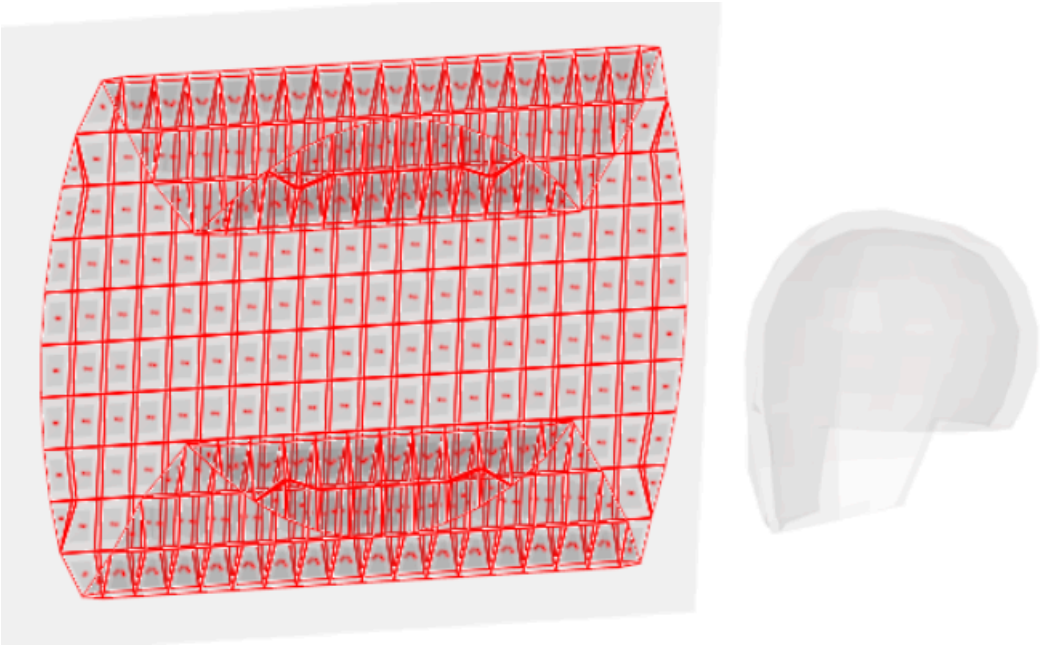


Figure 994:

2. Right-click on **airbag** and select **Review** from the context menu.
   The entities return to their regular display color.
3. In the Solver Browser, *CONTACT folder, *CONTACT NODES TO SURFACE subfolder, right-click on **Airbag_Plate** and select **Isolate Only** from the context menu.

   > **Tip:** If master and slave entities are not visible, make sure the Show/Isolate/IsolateOnly/Attached checkbox is selected in the Options tab of the **Browser Configuration** dialog. Access the **Browser Configuration** dialog by right-clicking in the **Model Browser** and selecting **Configure Browser** from the context menu.

   Only the elements/components that are implicated in this contact display as seen in Figure 995.

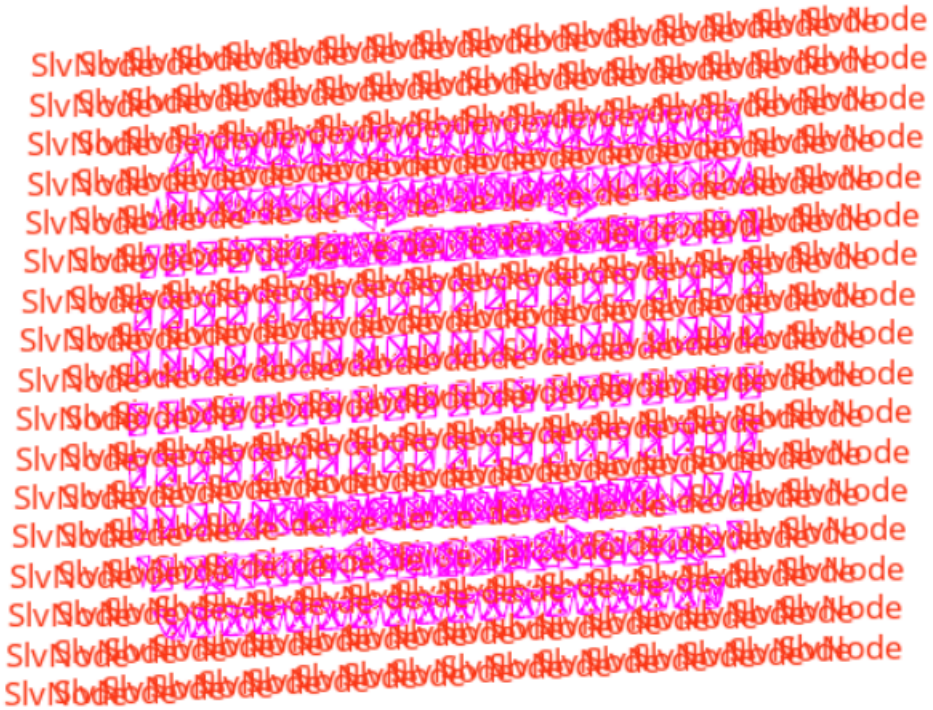


Figure 995:

4. In the Solver Browser, *CONTACT folder, *CONTACT_AIRBAG_SINGLE_SURFACE subfolder, right-click on **airbag** and select **Show** from the context menu.
The entire airbag displays as seen in Figure 996.

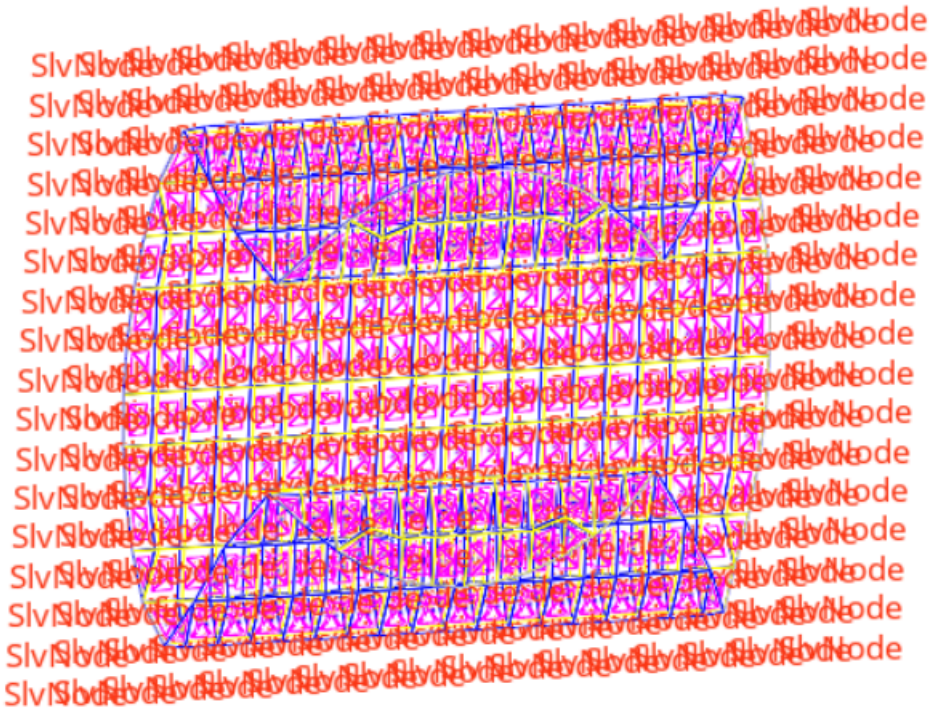


Figure 996:

5. In the *INITIAL folder, *INITIAL_VELOCITY subfolder, right-click on **velocity** and select **Review** from the context menu.
The nodes on which velocity was applied displays as seen in Figure 997.

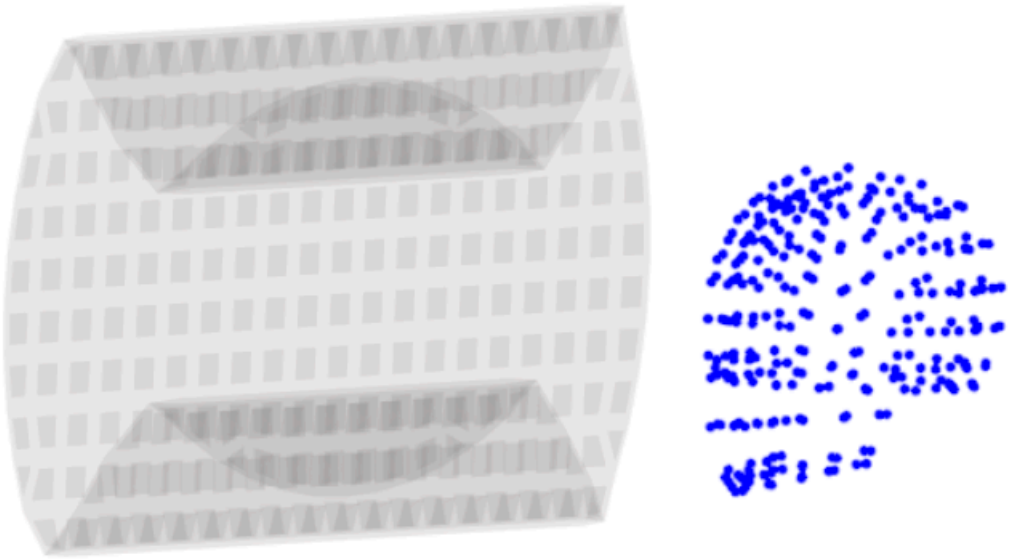


Figure 997:

6. Right-click on **velocity** and select **Review** from the context menu.
   The entities return to their regular display color.

5.3.8 Export the Model

In this step, you will export the model to an LS-DYNA 971 formatted input file.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
   The Export - Solver Deck tab opens.
2. Set Template to **Keyword971**.
3. In the File field, navigate to your working directory and save the file as `airbag_complete.key`.
4. Next to Export options, click .
5. Set Export to **All**.
6. Click **Export**.

5.3.9 Submit the Input File

In this step, you will submit the LS-DYNA input file to LS-DYNA 970.

1. From the Start Menu, open the LS-DYNA Manager program.
2. From the solvers menu, select **Start LS-DYNA analysis**.
3. Load the `airbag_complete.key` file.
4. Start the analysis by clicking **OK**.

5.3.10 View the Results

In this tutorial, you will view the results in HyperView.

Save your work to a HyperMesh file.

5.4 HM-4620: Create Rigid Walls, Model Data, Constraints, Cross Sections, and Output with LS-DYNA

In this tutorial you will: create *PART_INERTIA for the vehicle mass component, velocity on all nodes except barrier nodes with *DEFINE_BOX and *INITIAL_VELOCITY, a contact between the crash boxes, the bumper, and the barrier with *CONTACT_AUTOMATIC_GENERAL, and a stationary rigid wall; use *DATABASE_HISTORY_NODE to specify nodes to be output; use *DATABASE_CROSS_SECTION_PLANE to specify the output of resultant forces; and make nodes a part of the mass rigid body with *CONSTRAINED_EXTRA_NODES.

Before you begin, copy `bumper_start.key` from `<hm.zip>/interfaces/lsdyna/` to your working directory.

5.4.1 Overview of *PART_INERTIA, *CONSTRAINED_EXTRA_NODES, *DATABASE_CROSS_SECTION_(Option), and *RIGIDWALL

In this section, you will learn about *PART_INERTIA, *CONSTRAINED_EXTRA_NODES, *DATABASE_CROSS_SECTION_(Option), and *RIGIDWALL.

*PART_INERTIA

The INERTIA option enables inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies only. When importing a LS-DYNA model into HyperMesh, the *PART_INERTIA IRCS parameter value is changed from 0 to 1. The inertia components are changed from global to local axis.

This allows inertia components to be automatically updated when *PART_INERTIA elements are translated or rotated. When selecting *PART_INERTIA elements to translate or rotate, select elements by comp. This selection method ensures the inertia properties are automatically updated.

*CONSTRAINED_EXTRA_NODES

This card defines extra nodes to be part of a rigid body. In HyperMesh, it is created from the Solver Browser or Model Browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

*DATABASE_CROSS_SECTION_(Option)

*DATABASE_CROSS_SECTION_(Option) defines a cross section for resultant forces written to the ASCII SECFORC file. The options are PLANE and SET.

For the PLANE option, a cutting plane must be defined. For best results, the plane should cleanly pass through the middle of the elements, distributing them equally on either side.

The SET option requires the equivalent of the automatically generated input via the cutting plane to be identified manually and defined in sets. All nodes in the cross-section and their related elements contributing to the cross-sectional force resultants should be defined in sets.

*DATABASE_CROSS_SECTION_SET and *DATABASE_CROSS_SECTION_PLANE are created from the Solver Browser or Model Browser, Create Cards menu (access from the Tools pulldown menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

***RIGIDWALL**

A *RIGIDWALL provides a method for treating contact between a rigid surface and nodal points of a deformable body.

In HyperMesh, *RIGIDWALL keyword cards are created from the Solver Browser or Model Browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

5.4.2 Load the LS-DYNA User Profile

In this step, you will load the LS-DYNA user profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

5.4.3 Import the LS-DYNA Model

In this step, you will import the LS-DYNA model file into HyperMesh.

1. From the menu bar, click **File** > **Import** > **Solver Deck**.
   The Import - Solver Deck tab opens.
2. In the File field, navigate to `bumper_start.key`.
3. Click **Import**.

5.4.4 Define *PART_INERTIA

In this step, you will define *PART_INERTIA for the vehicle mass component to partially take into account the inertia properties and mass of the missing parts.

1. In the Model Browser, Components folder, click **vehicle mass**.

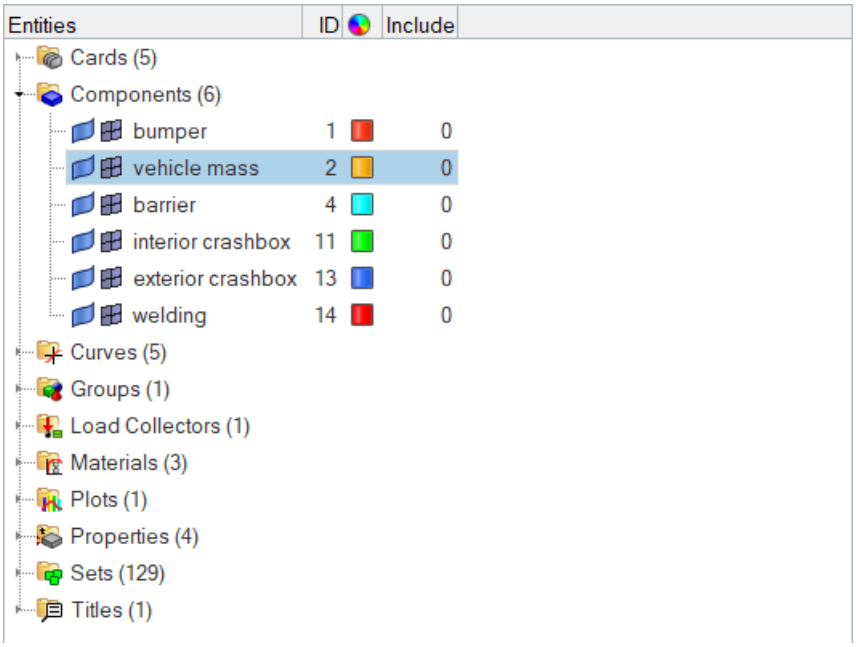


Figure 998:

The Entity Editor opens, and displays the component's card data.

2. In the Entity Editor, Edit the component's card data.
   a) Set Options to **Inertia**.
   b) For XC (X coordinate of center of mass), enter `700`.
   c) For YC (Y coordinate of center of mass), enter `0.0`.
   d) For ZC (Y coordinate of center of mass), enter `170`.
   e) For TM (translational mass), enter `800`.
   f) For IXX (XX component of target inertia), enter `1.5E+07`.
   g) For IXY (XY component of target inertia), enter `-5.0E+03`.
   h) For IXZ (XZ component of target inertia), enter `-8.0E+06`.
   i) For IYY (YY component of target inertia), enter `5.0E+07`.
   j) For IYZ (YZ component of target inertia), enter `-900`.
   k) For IZZ (ZZ component of target inertia), enter `6.0E+07`.
   l) For VTX (Initial translational velocity of rigid body in x direction), enter `-10`.

5.4.5 Create a *DEFINE_BOX that Contains Non-Barrier Nodes

In this step, you will create a *DEFINE_BOX in HyperMesh that contains all nodes except barrier nodes.

1. Open the Solver Browser by clicking **View** > **Browsers** > **HyperMesh** > **Solver** from the menu bar.

2. In the Solver Browser, right-click and select **Create** > ***DEFINE** > ***DEFINE_BOX** from the context menu.

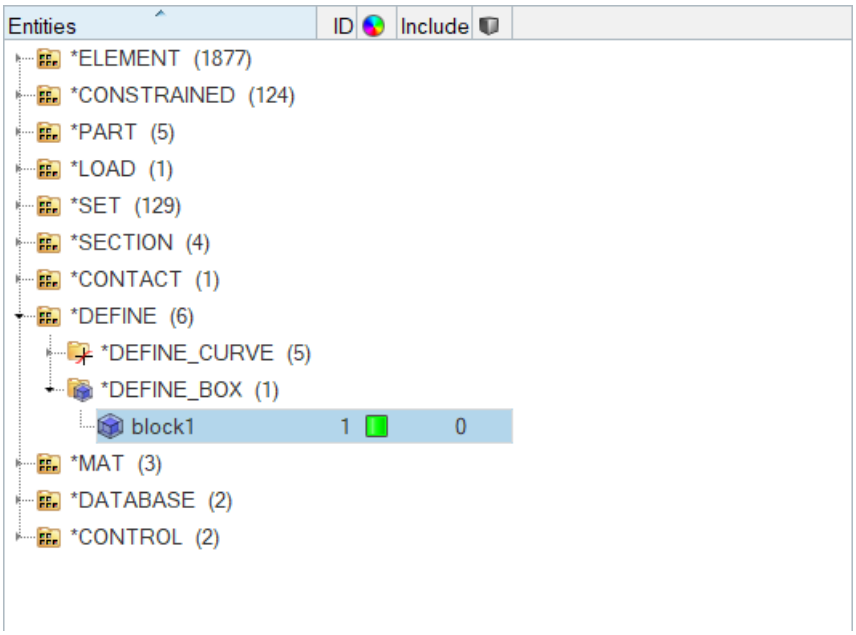


Figure 999:

A new block opens in the Entity Editor.

3. In the Entity Editor, define the block.
   a) For Name, enter `box velocity`.
   b) Optional: Click the **Color** icon, and select a color for the block.
   c) For Xmin Ymin Zmin, enter `-530`, `-800`, `0` as seen in Figure 1000.

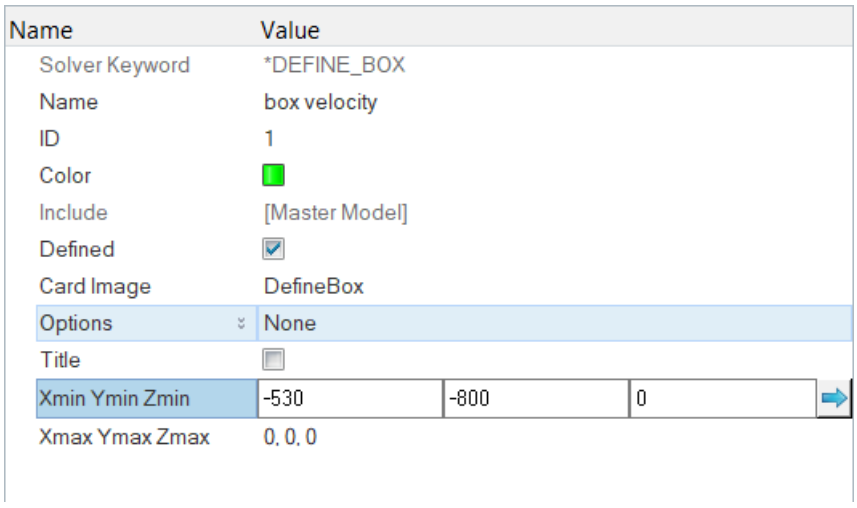


Figure 1000:

   d) For Xmax Ymax Zmax, enter `200`, `800`, `300` as seen in Figure 1001.

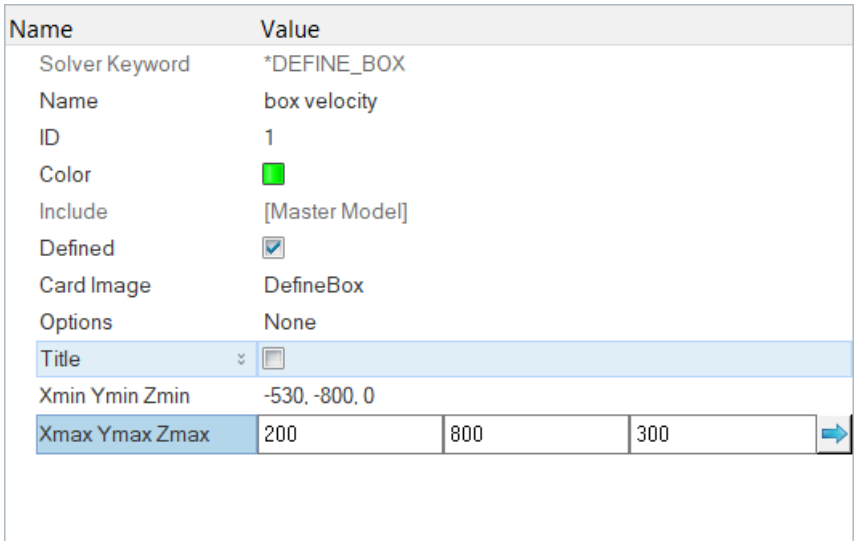


Figure 1001:

5.4.6 Create Initial Velocity

In this tutorial, you will create initial velocity on all nodes except barrier nodes.

A velocity boundary condition can also be created on a set of nodes from the Solver Browser or Model Browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

1. In the Solver Browser, right-click and select **Create** > ***INITIAL** > ***INITIAL_VELOCITY** from the context menu.

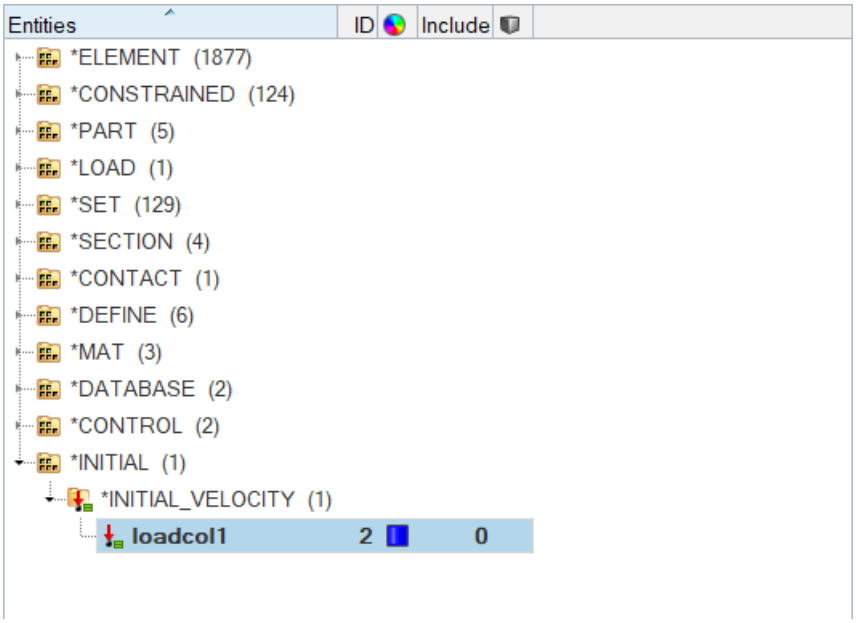


Figure 1002:

A new load collector opens in the Entity Editor.

2. In the Entity Editor, define the load collector.

a) For Name, enter `velocity`.

b) For VX (Initial velocity in the global X direction), enter `-10`.

c) Click **BOXID**, and then click **Block**.

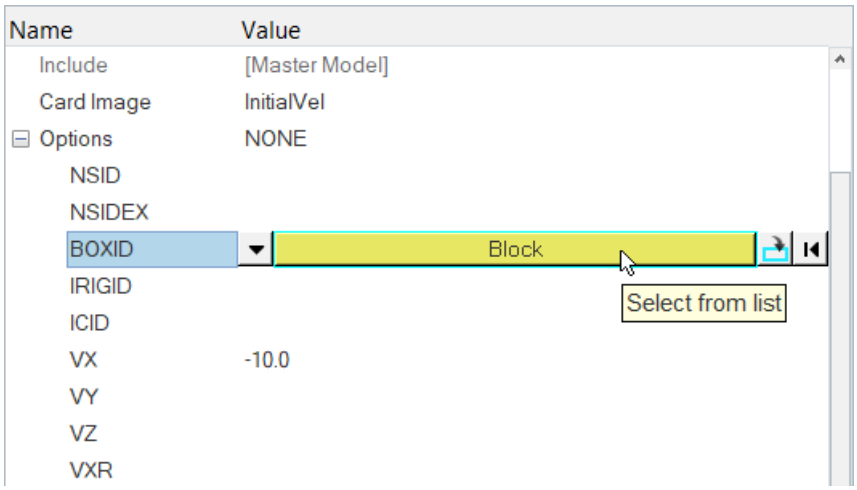


Figure 1003:

d) In the **Select Block** dialog, select **box velocity** and then click **OK**.

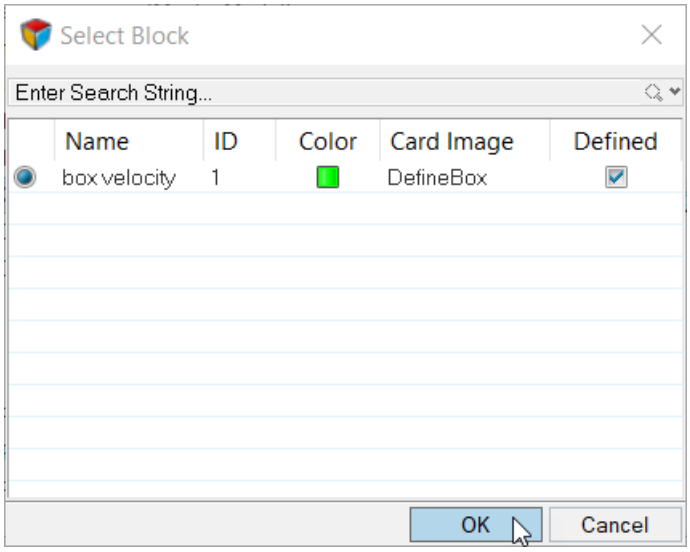


Figure 1004:

5.4.7 View the Nodes in the Node Entity Set

In this step, you will view the closest nodes which are in the pre-defined node entity set (*SET_NODES_LIST) named Constrain Vehicle using two methods.

1. Method 1
   a) In the Solver Browser, SET folder, SET_NODE_LIST folder, right-click on **Constrain Vehicle** and select **Review** from the context menu.
   The set's nodes highlight as seen in Figure 1005.



Figure 1005:

   b) Right-click on **Constrain Vehicle** and select **Review** from the context menu.
   The entities return to their original display color as seen in Figure 1006.

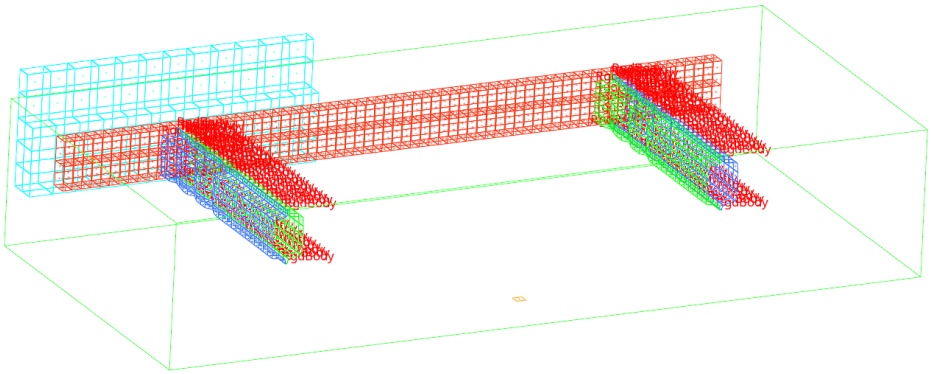


Figure 1006:

2. Method 2

a) From the menu bar, click **Tools** > **Edit** > **Sets**.

b) In the Entity Sets panel, click **review**.

c) Set the display RLs/hide RLs toggle to **hide RLs** as seen in Figure 1007.

> **Note:** This option filters all nodal rigid body sets from the list.

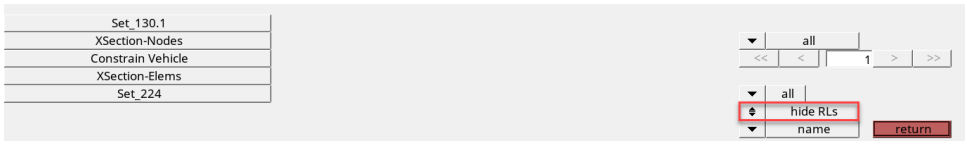


Figure 1007:

d) Select the **Constrain Vehicle** set.
The set's nodes highlight as seen in Figure 1008.

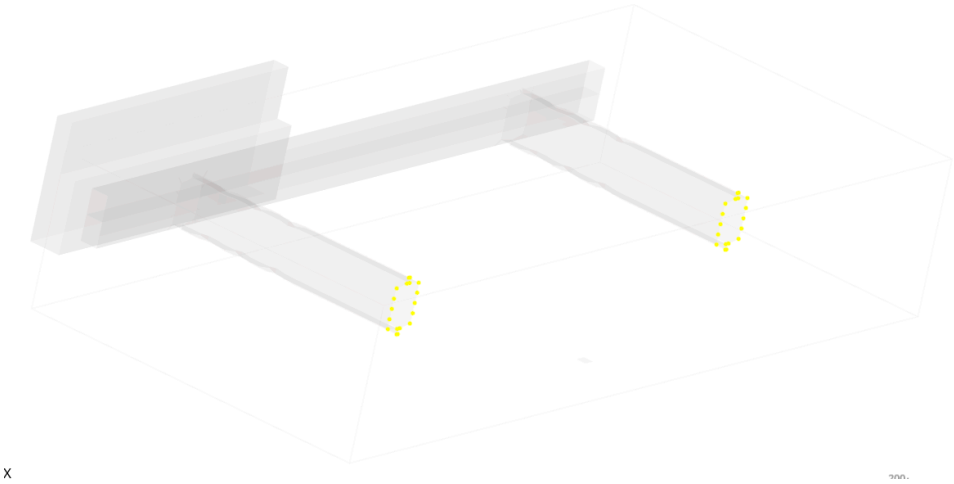


Figure 1008:

e) Close the panel by clicking **return**.

5.4.8 Create *CONSTRAINED_EXTRA_NODES_SET

In this step, you will create a *CONSTRAINED_EXTRA_NODES_SET.

1. In the Solver Browser, right-click and select **Create** > ***CONSTRAINED** > ***CONSTRAINED_EXTRA_NODES_SET** from the context menu.

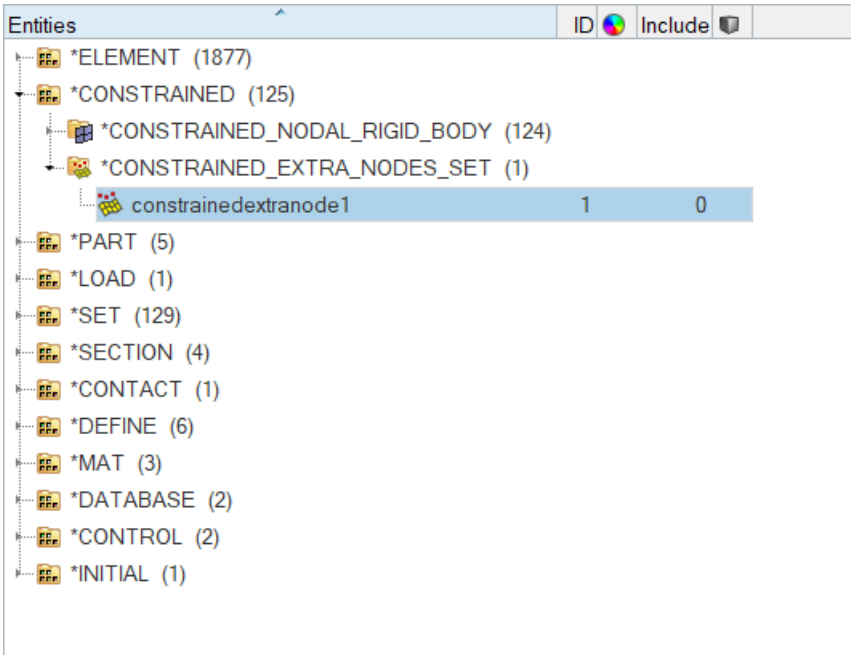


Figure 1009:

A new constrained extra node opens in the Entity Editor.

2. In the Entity Editor, define the constrained extra node.
   a) For Name, enter `ExtraNodes`.
   b) For PID, click **Unspecified** > **Component**.

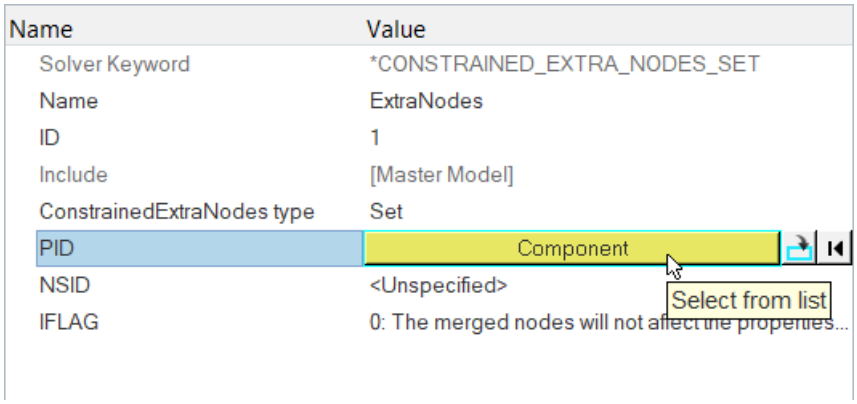


Figure 1010:

c) In the **Select Component** dialog, select **vehicle mass** and then click **OK**.

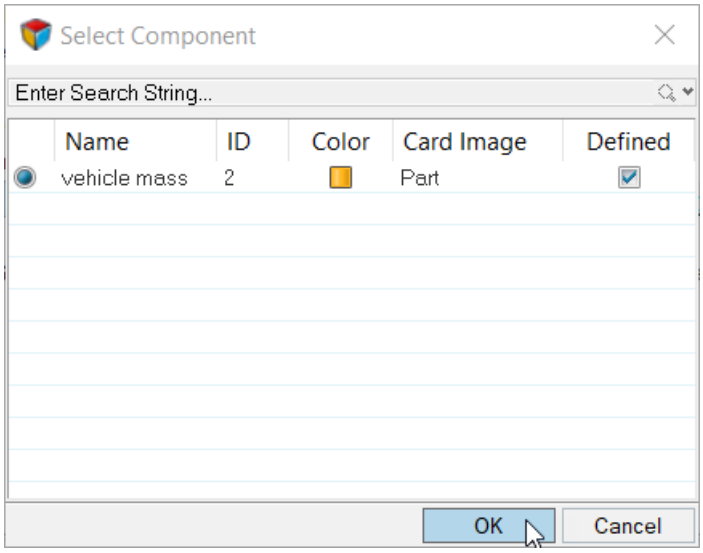


Figure 1011:

5.4.9 Define Nodes in the Constrain Vehicle

In this step, you will define the nodes in the Constrain Vehicle set to be a part of the vehicle mass rigid body.

In this step, the Entity Editor should be open for the ExtraNodes constrained extra node.

1. For NSID, click **Unspecified** > **Set**.
2. In the **Select Set** dialog, select **Constrain Vehicle** and then click **OK**.

5.4.10 View Extra Nodes of the Vehicle Mass Rigid Body

In this step, you will view the extra nodes that are a part of the vehicle mass rigid body.

1. Review ExtraNodes by completing one of the following options:
   - In the Solver Browser right-click on **ExtraNodes** and select **Review** (press Q) from the context menu.
   - In the Model Browser right-click on **ExtraNodes** and select **Review** (press Q) from the context menu.

   The extra nodes temporarily display red, the PID (vehicle mass) displays blue, and all of the other entities temporarily display grey as seen in Figure 1012.

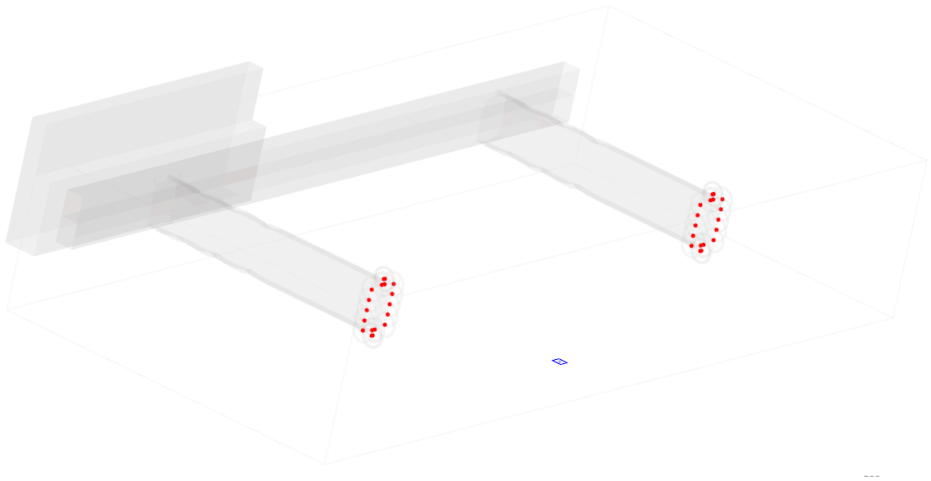


Figure 1012:

2. Return all of the entities to their original display color by right-clicking on **ExtraNodes** and selecting **Review** from the context menu.

5.4.11 Create an Entity Set

In this step, you will create an entity set, *SET_PART_LIST, for the vehicle mass component.

All other components not in this set will be included in the contact.

1. In the Solver Browser, right-click and select **Create** > ***SET** > ***SET_PART** > ***SET_PART_LIST** from the context menu.

   Tip: You can also create a *SET_PART_LIST from the Model Browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

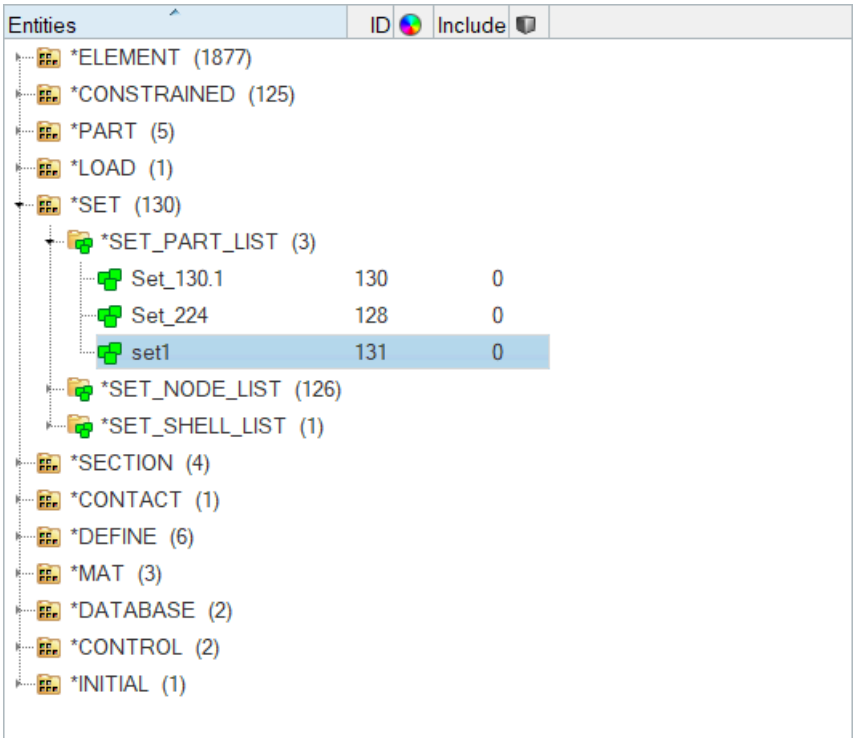


Figure 1013:

A new set opens in the Entity Editor.

2. In the Entity Editor, define the set.
   a) For name, enter `Exempt Parts`.
   b) For Entity IDs, click **0 Components** > **Components**.
   c) In the **Select Components** dialog, select **vehicle mass** and then click **OK**.

5.4.12 Create a *CONTACT_AUTOMATIC_GENERAL Contact

In this step, you will create a *CONTACT_AUTOMATIC_GENERAL contact.

1. In the Solver Browser, right-click and select **Create** > ***CONTACT** > ***CONTACT_AUTOMATIC_GENERAL** from the context menu.

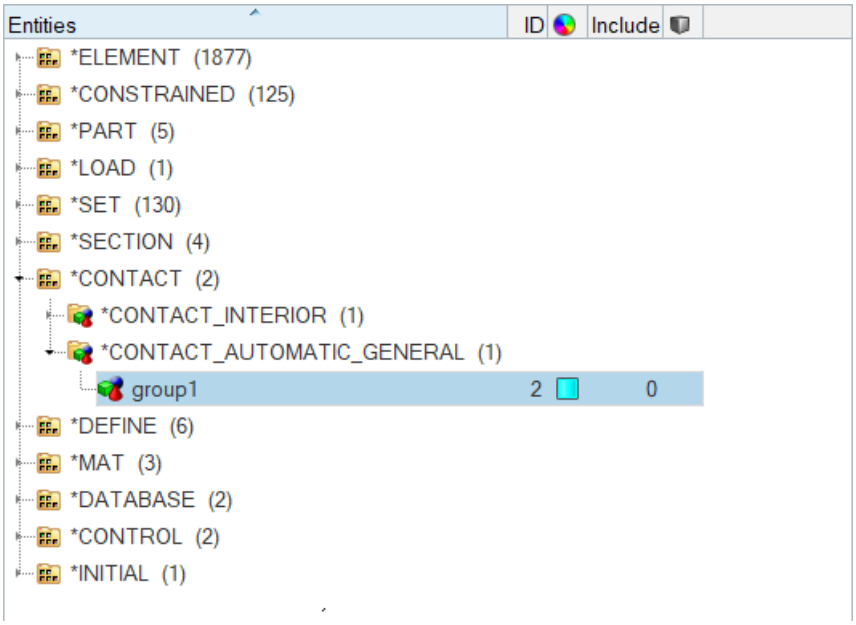


Figure 1014:

A new group opens in the Entity Editor.

2. For name, enter `impact`.

5.4.13 Define Secondary Surface

In this step, you will define the secondary surface with secondary set type 6, part set ID for exempted parts.

In this step the Entity Editor should be open for the impact group.

1. Click **SSID**.
2. Set the entity selector to **Set**.
3. Click **Set**.
4. In the **Select Set** dialog, select **Exempt Parts** and then click **OK**.
5. Select the **ExemptSlvPartSet** checkbox.
   The SSTYPE (secondary surface type) value changes from 2 (part set ID) to 6 (part set ID for exempted parts) as seen in Figure 1015.

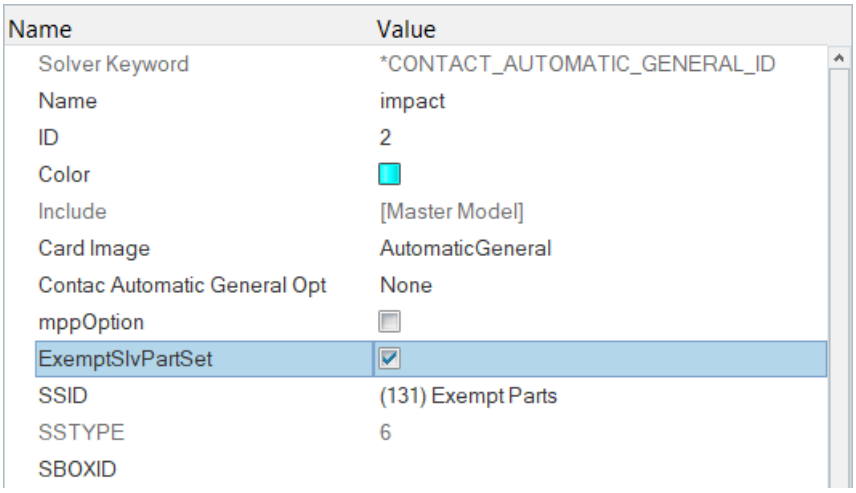


Figure 1015:

5.4.14 Create an Entity Set

In this step, you will create an entity set, *SET_PART_LIST, to specify the elements that will contribute to the cross-sectional force results.

1. In the Solver Browser, right-click and select **Create** > ***SET** > ***SET_PART** > ***SET_PART_LIST** from the context menu.

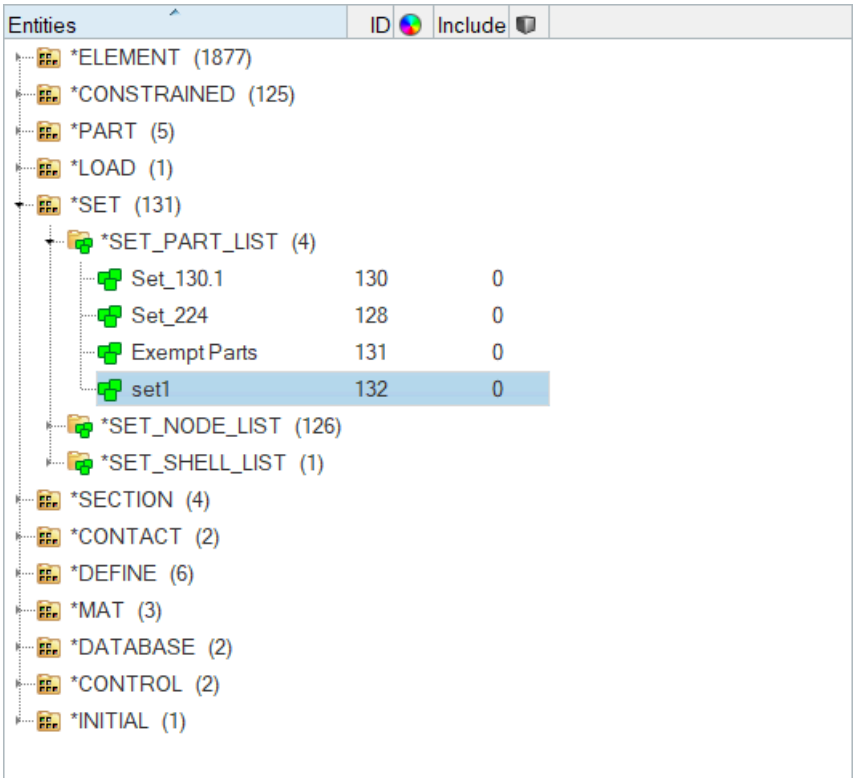


Figure 1016:

A new group opens in the Entity Editor.

2. In the Entity Editor, define the set.
   a) For Name, enter `CrossSectionPlane-Parts`.
   b) For Entity IDs, click **0 Components** > **Components**.
   c) In the **Select Components** dialog, select **interior crashbox** and **exterior crashbox**.
   d) Click **OK**.

5.4.15 Define a Section

In this step, you will define a section by creating a *DATABASE_CROSS_SECTION_PLANE.

1. In the Solver Browser, right-click and select **Create** > ***DATABASE** > ***DATABASE_CROSS_SECTION_PLANE** from the context menu.

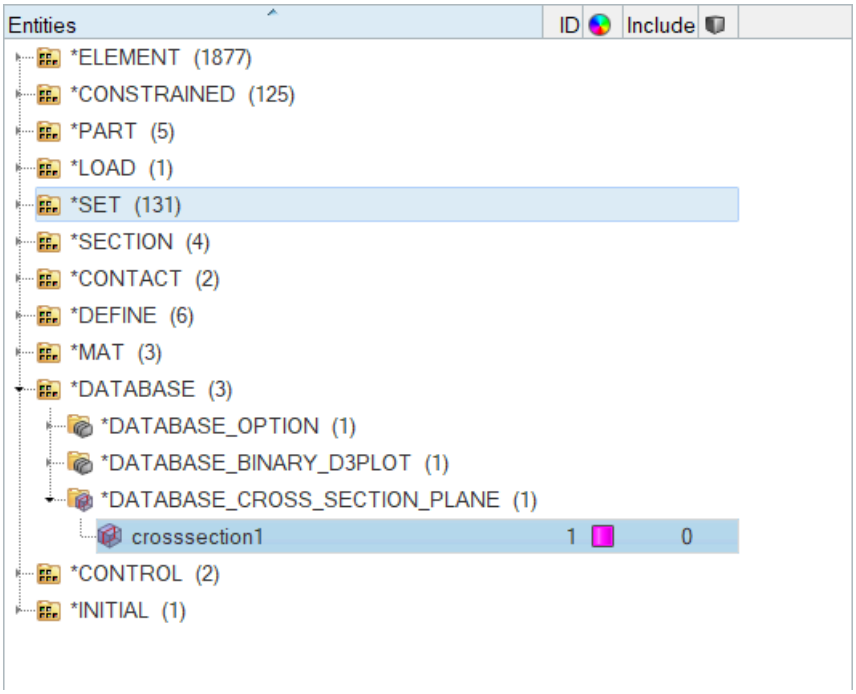


Figure 1017:

A new group opens in the Entity Editor.

2. For Name, enter `CrossSection_Plane`.

5.4.16 Define the Location and Size of the Section Plane

In this step, you will define the location and size of the section plane.

In this step the planes origin (the tail of the normal vector) is defined by a base node. The Entity Editor should be open for the CrossSection_Plane cross section.

1. Create a base node.
   a) Open the Create Nodes panel by clicking **Geometry** > **Create** > **Nodes** > **XYZ** from the menu bar.

      Tip: You can also open the Create Nodes panel by pressing **F8**.

   b) In the x field, enter `-320`.
   c) In the y field, enter `-500`.
   d) In the z field, enter `100`.

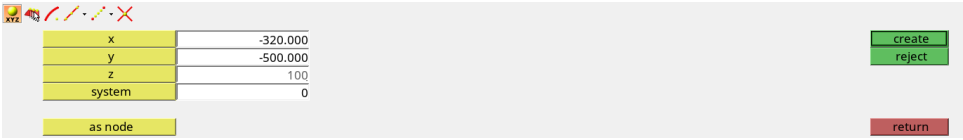


Figure 1018:

e) Click **create**.
A new node displays as seen in Figure 1019.

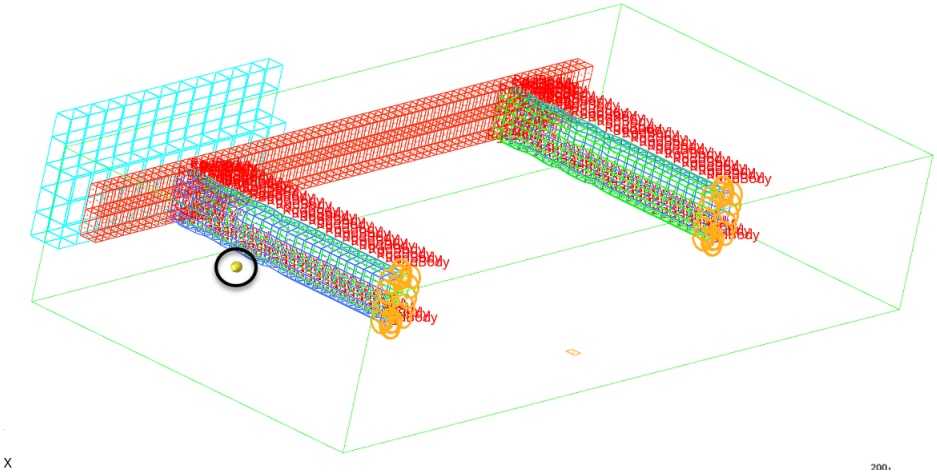


Figure 1019:

f) Click **return**.

2. In the Entity Editor, define the XTAIL, YTAIL, ZTAIL (base node) for the section.

a) Click **XTAIL, YTAIL, ZTAIL** (base node), and then click .

b) In the graphics area, select the base node you created in step 1 as seen in Figure 1020.

Note: If the base node is not visible, click on the Visualization toolbar to display elements as a wireframe (skin only).

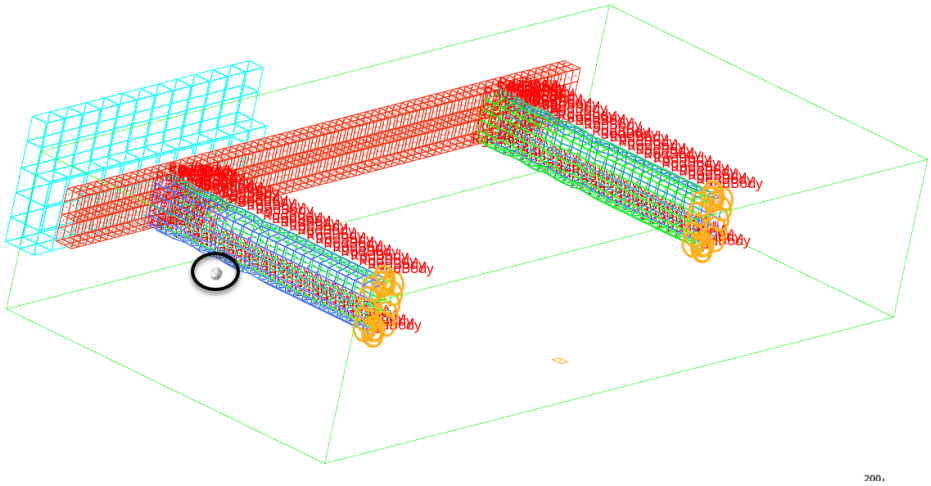


Figure 1020:

c) Click **proceed**.
The Entity Editor displays the coordinates of the base node in the XTAIL, YTAIL, ZTAIL field as seen in Figure 1021.

Name	Value
Solver Keyword	*DATABASE_CROSS_SECTION_PLANE_ID
Name	crosssection1
ID	1
Color	
Include	[Master Model]
Defined	☑
CrossSection Config	plane
Geometry type	Infinite plane
PSID	<Unspecified>
XTAIL,YTAIL,ZTAIL	-320, -500, 100
Normal	0, 0, 0
XHEAD	-320.0
YHEAD	-500.0
ZHEAD	100.0
ID	<Unspecified>
ITYPE	0

Figure 1021:

3. Set Geometry type to **Finite plane**.

4. Define the normal vector.

a) Click **Normal**, and then click .

b) In the panel area, set the orientation selector to **x-axis**.

c) Click **proceed**.

5. Define the edge vector.

a) Click Edge, and then click .

b) In the panel area, set the orientation selector to **y-axis**.

c) Click **proceed**.
The Entity Editor displays the coordinates of the edge vector L in the Normal field.

6. For LENL (length of edge a, in the L direction), enter `100`.

7. For LENM (length of edge b, in the M direction), enter `200`.

> **Tip:** If you know the coordinates of the base node, edge, and normal, you can manually enter them in the Entity Editor.

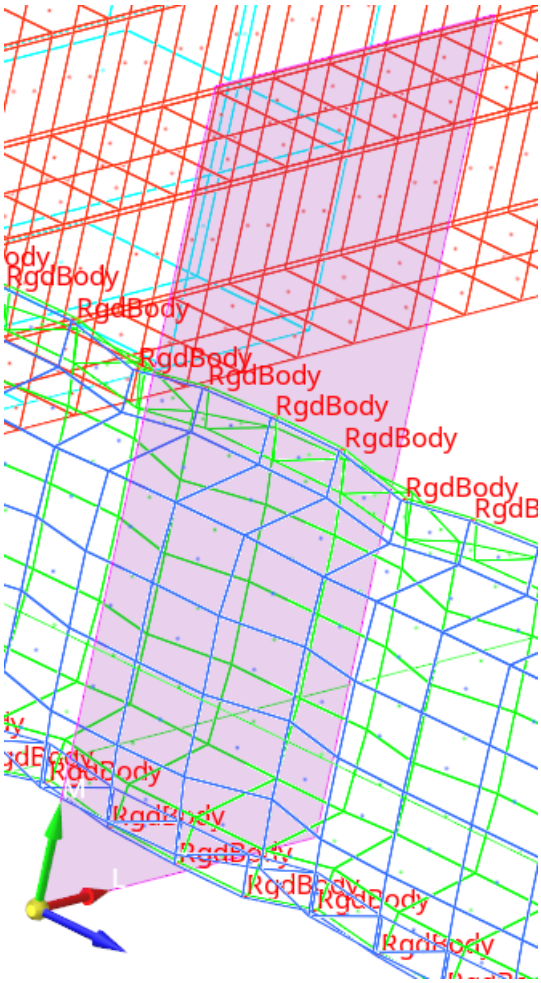


Figure 1022:

5.4.17 Specify the Parts Secondary to the Cross Section

In this step, you will specify the parts secondary to the cross section.

In this step the Entity Editor should be open for the CrossSection_Plane cross section.

1. For PSID, click **Unspecified** > **Set**.
2. In the **Select Set** dialog, select **CrossSectionPlane-Parts** and then click **OK**.

5.4.18 View the Entities Secondary to the Rigid Wall

In this step, you will view the entities secondary to the rigid wall.

1. In the Solver Browser, right-click on **CrossSection_Plane** and select **Review** from the context menu.
   The secondary entities and rigid wall highlight and all of the other entities temporarily display grey as seen in Figure 1023.

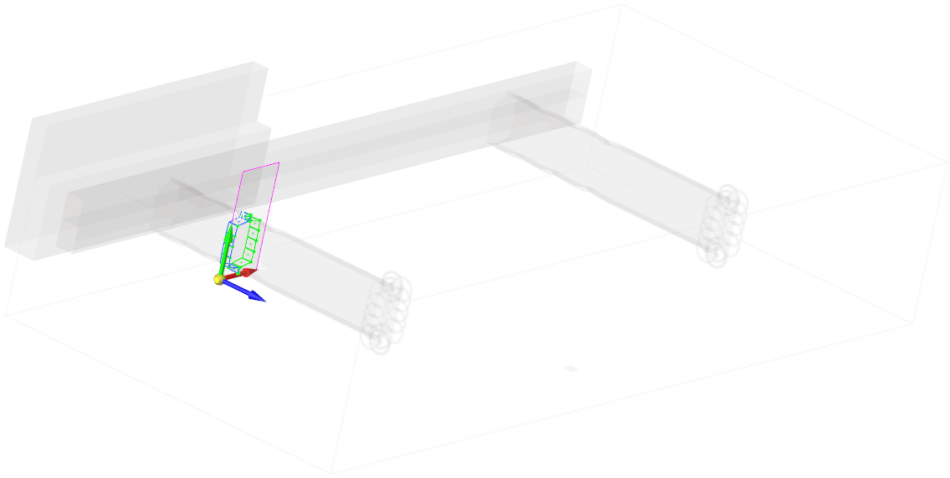


Figure 1023:

2. Return the entities to their original display color by right-clicking on **CrossSection_Plane** and selecting **Review** from the context menu.

5.4.19 Create a *DEFINE_BOX that Contains Barrier and Bumper Nodes

In this step, you will create a *DEFINE_BOX containing the nodes making up the barrier and the left side of the bumper.

These nodes will be secondary to the rigid wall.

1. In the Solver Browser, right-click and select **Create** > ***DEFINE** > ***DEFINE_BOX** from the context menu.

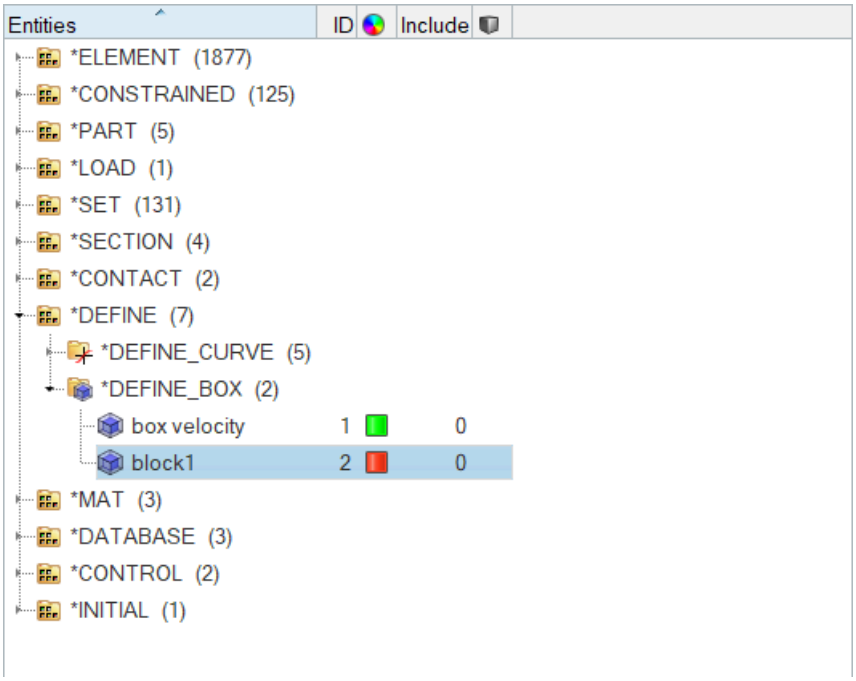


Figure 1024:

A new block opens in the Entity Editor.

2. In the Entity Editor, define the block.

a) For Name, enter `half model`.

b) Optional: Click the **Color** icon and select a color to display the block.

c) For Xmin Ymin Zmin, enter `-600, -800, 0`.

d) For Xmax Ymax Zmax, enter `-460, 0, 400`.

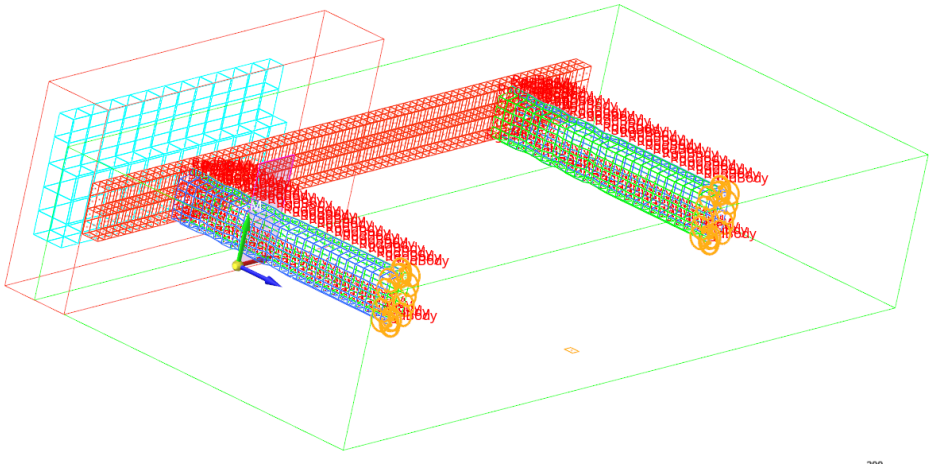


Figure 1025:

5.4.20 Create a *RIGIDWALL_PLANAR_FINITE

In this step, you will define a HyperMesh group by creating *RIGIDWALL_PLANAR_FINITE.

*RIGIDWALL are created from the Solver Browser or Model Browser, Create Cards menu (access from the Tools pull-down menu), or the Quick Access tool (Ctrl + F) when a keyword is entered.

1. In the Solver Browser, right-click and select **Create** > ***RIGIDWALL** > ***RIGIDWALL_PLANAR_FINITE** from the context menu.

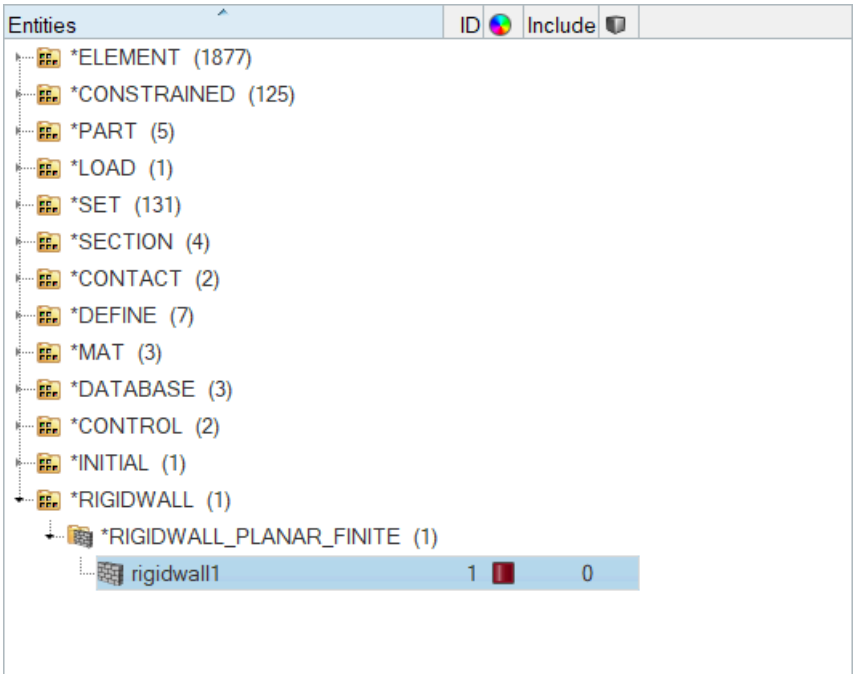


Figure 1026:

A new rigid wall opens in the Entity Editor.

2. For Name, enter `wall`.

5.4.21 Define the Location and Size of the Rigid Wall

In this step, you will create a node from the create nodes panel and then select it for the base node.

In the Create Nodes panel, XYZ sub-panel, the rigid wall's origin (the tail of the normal vector) is defined by a base node. In this step the Entity Editor should be open for the rigid wall

1. Create a base node.
   a) Open the Create Nodes panel by clicking **Geometry** > **Create** > **Nodes** > **XYZ** from the menu bar.

   > **Tip:** You can also open the Create Nodes panel by pressing **F8**.

   b) Click  to open the XYZ subpanel.

c) In the x field, enter `-600`.

d) In the y field, enter `-750`.

e) In the z field, enter `90`.

f) Click **create**.

> **Tip:** If the base node is not visible, click on the Visualization toolbar to display elements as a wireframe (skin only).

g) Click **return**.

2. For XT, YT, and ZT enter `-600`, `-750`, and `90` as seen in Figure 1027.

> **Note:** You can also select the node created in step 1 for the rigid wall base.

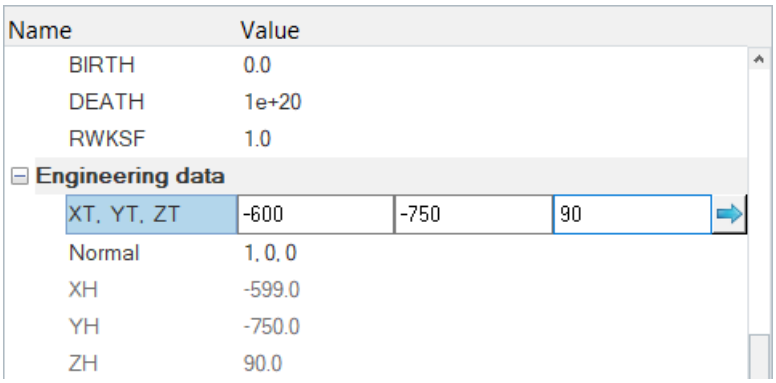


Figure 1027:

3. Set Geometry type to **Finite plane**.

4. Define the normal vector.

a) Click **Normal**, and then click .

b) In the panel area, set the orientation selector to **x-axis**.

c) Click **proceed**.

5. Define the edge vector.

a) Click **Edge**, and then click .

b) In the panel area, set the orientation selector to **y-axis**.

c) Click **proceed**.

6. For Length LENL, enter `165`.

7. For Length LENM, enter `250`.

> **Note:** The input values for LENL and LENM are the length of the edges a and b in the L and M directions, respectively. These values define the extent of the rigid wall.

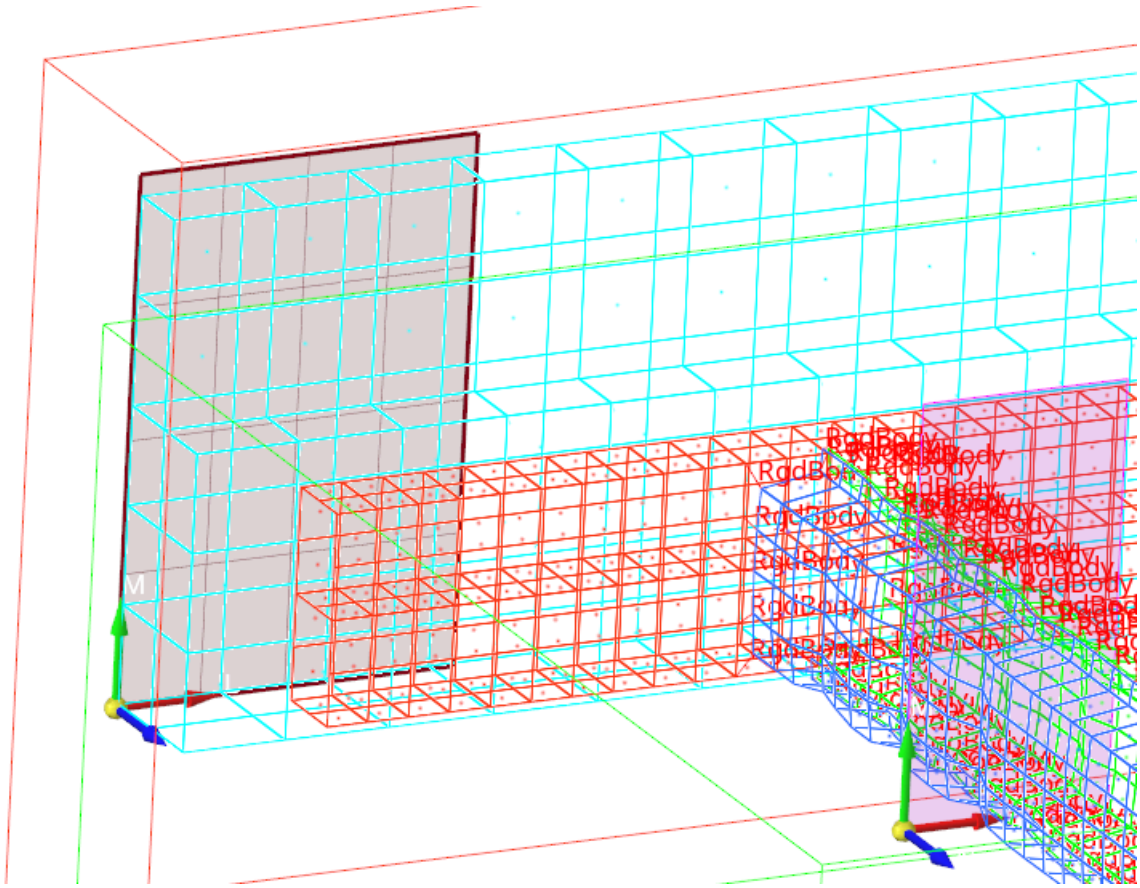


Figure 1028:

5.4.22 Specify Nodes as Secondary to the Rigid Wall

In this step, you will use the Entity Editor for the rigid wall to specify the nodes in the *DEFINE_BOX half model as secondary to the rigid wall.

In this step the Entity Editor should be open for the rigid wall

1. Click **BOXID** > **Block**.
2. In the **Select Block** dialog, select **half model** and then click **OK**.
3. For FRIC (Interface friction), enter `1.0`.

5.4.23 Specify Nodes to be Output

In this step, you will specify some nodes to be output to the ASCII NODOUT file with *DATABASE_HISTORY_NODE.

1. In the Solver Browser, right-click and select **Create** > ***DATABASE** > ***DATABASE_HISTORY_NODE** from the context menu.

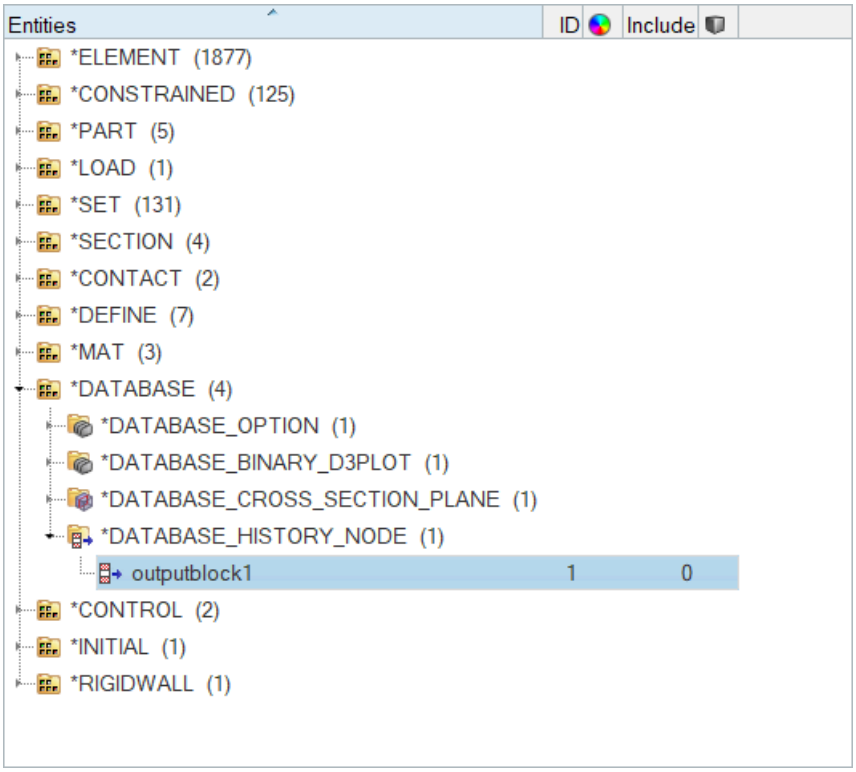


Figure 1029:

A new output block opens in the Entity Editor.

2. In the Entity Editor, define the output block.

a) For Name, enter `nodeth`.

b) For Entity IDs, click **0 Nodes** > **Nodes**.

c) In the graphics area, select a few nodes of interest as seen in Figure 1030.

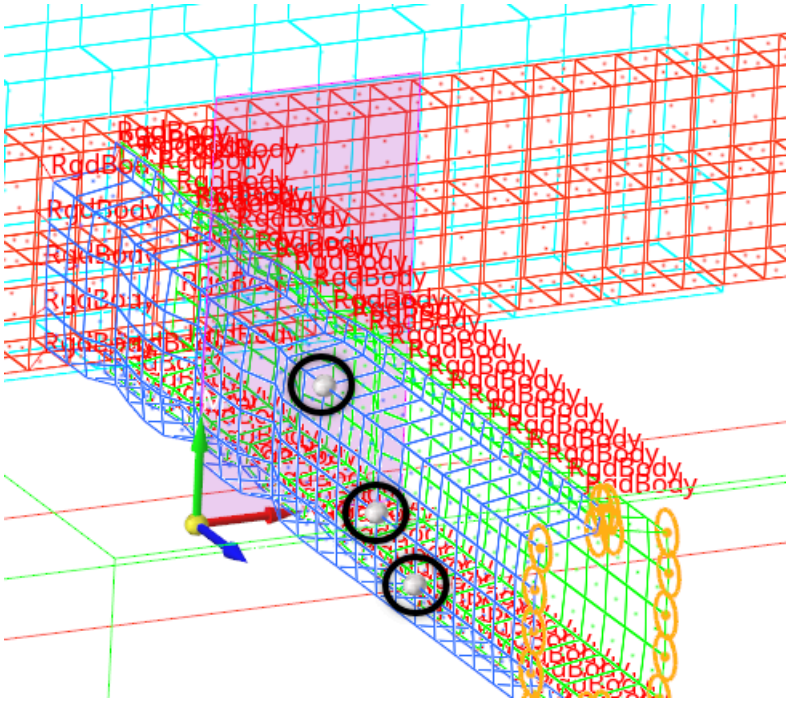


Figure 1030:

d) Cick **proceed**.

5.4.24 Export the Model

In this step, you will export the model to an LS-DYNA 971_R# formatted input file.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
   The Export - Solver Deck tab opens.
2. Set File type to **LsDyna**.
3. In the File field, navigate to your working directory and save the file as `Bumper_complete.key`.
4. Click **Export**.

5.4.25 Submit the Input File

In this step, you will submit the LS-DYNA input file to the LS-DYNA 970 solver.

1. From the Start Menu, open the LS-DYNA Manager program.
2. From the solvers menu, select **Start LS-DYNA analysis**.
3. Load the `bumper_complete.key` file.
4. Start the analysis by clicking **OK**.

5.4.26 View the results in HyperView

In this step, you will view the results in HyperView.

This exercise is complete.

Save your work to a HyperMesh file.

5.5 HM-4625: Assemblies Using DYNA

In this tutorial, you will learn how to weld between geometry surfaces and shell elements, welding using a master connectors file and duplicating and reflecting connectors, create connectors from existing welds to create new welds of a different type, swap welded part, and understand why connectors may fail to realize and how to correct the problems.

Before you begin, copy the following files from `<hm.zip>/interfaces/lsdyna/` to your working directory:

- `frame_assembly_1.hm`
- `frame_assembly_2.hm`
- `frame_assembly_3.hm`
- `frame_assembly_4.hm`
- `body_side_assembly.hm`

The part assembly used in the first four exercises of this tutorial is depicted in the following image.

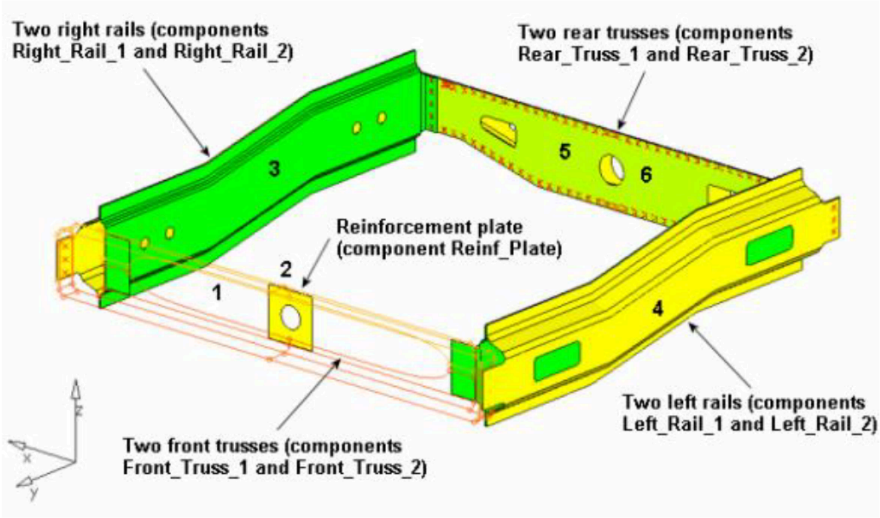


Figure 1031:

5.5.1 Exercise 1: Weld Between Geometry Surfaces and Shell Elements

In this exercise, you will become familiar with creating welds at pre-defined weld points between geometry surfaces and shell elements.

In this exercise, you will first weld the two front trusses depicted in Figure 1032 by creating connectors between the geometry surfaces at pre-defined weld points and realizing the connectors into two node weld elements. Second, you will weld the two front trusses to the reinforcement plate depicted in Figure 1033 by creating connectors between their shell elements at pre-defined weld points and realizing the connectors into two node weld elements.

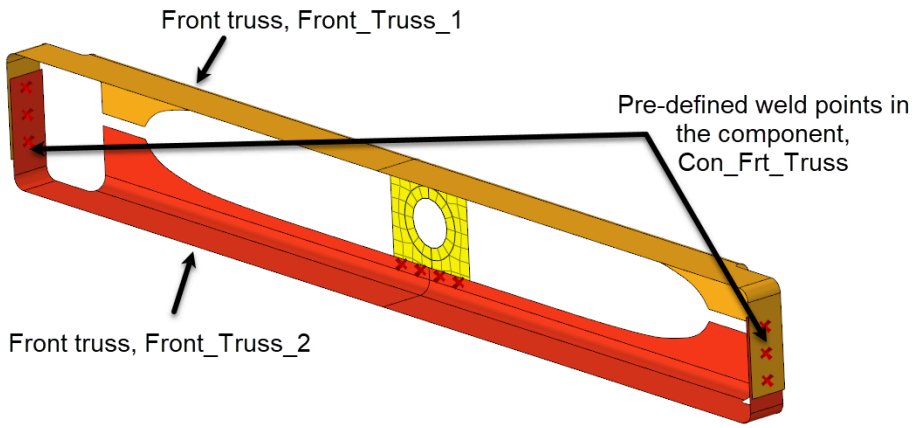


Figure 1032:

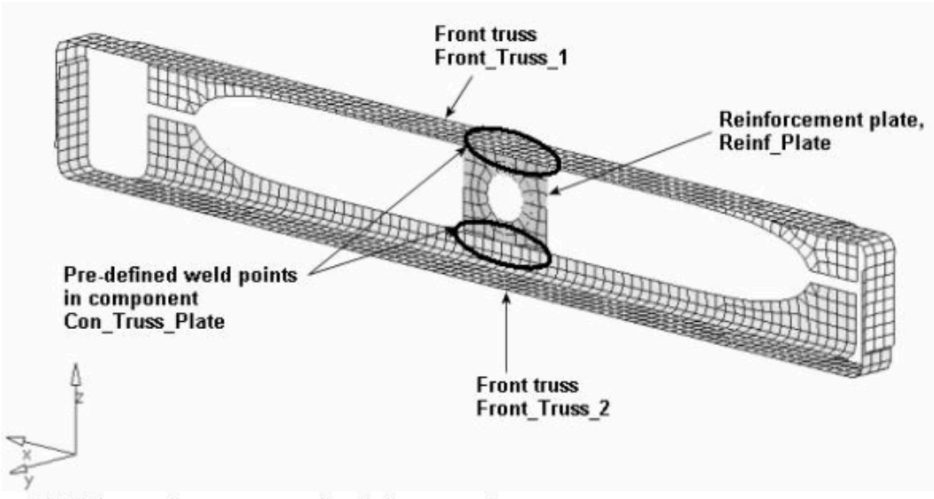


Figure 1033:

Retrieve and View the Model File

In this step, you will retrieve and view the model file in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.
3. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
4. In the **Open Model** dialog, open the `frame_assembly_1.hm` file.
5. On the Visualization toolbar, click to shade your model's geometry and surface edges.

6. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).

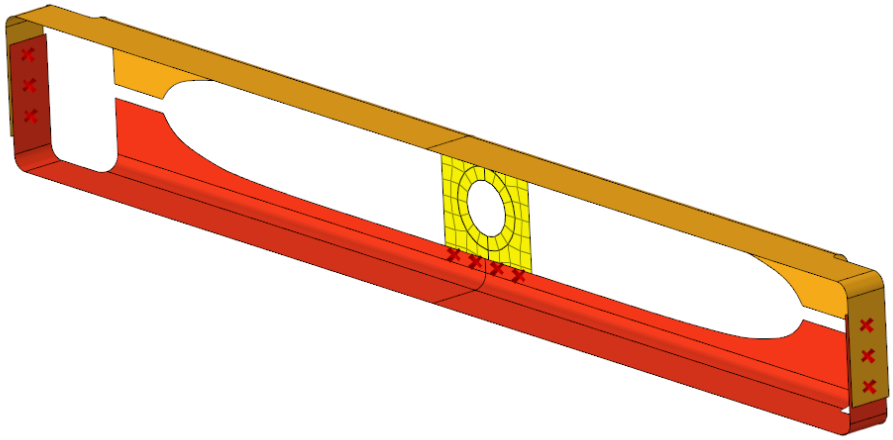


Figure 1034:

Create Welds Between the Geometry

In this step, you will create welds between the geometry for the two front trusses at the pre-defined weld points.

1. In the Model Browser, verify that the current component is **Con_Frt_Truss**.

> **Tip:** The current component is always boldfaced in the Model Browser, Component folder.

2. Open the connector Spot panel by clicking **Connectors** > **Create** > **Connectors** > **Spots** from the menu bar.
3. Set the location selector to **points**.
4. Select the six pre-defined weld points by clicking **points** > **by collector**.
5. Select the **Con_Frt_Truss** checkbox.

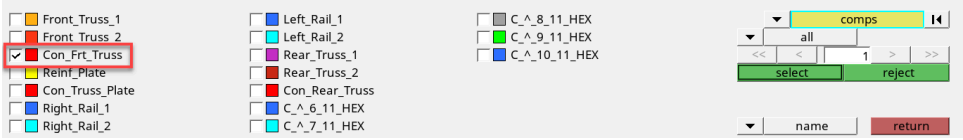


Figure 1035:

6. Click **select**.
7. Double-click **connect what: comps**.
8. Select the **Front_Truss_1** and **Front_Truss_2** checkboxes.
9. Click **select**.

10. In the tolerance= field, enter `5`.

11. Click **type=** and select **weld**.

12. Under the connect what selector, toggle from **elems** to **geom**.

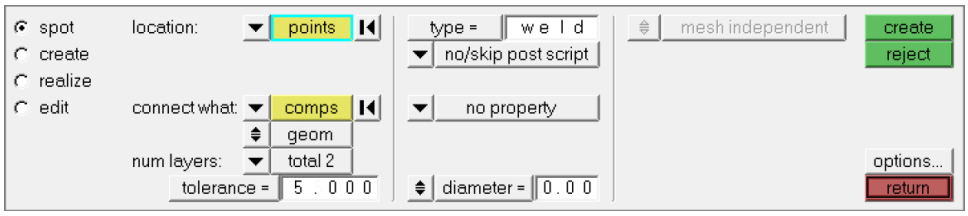


Figure 1036:

13. Click **create**.

A green connector indicates that the creation of the weld entity was successful. There are three states of connectors: realized (green), unrealized (yellow), and failed (red). The color of the connectors can change from yellow to green (if created manually), indicating they are realized into weld elements. If you create connectors automatically, they will be green immediately as there is no interim unrealized (yellow) state.

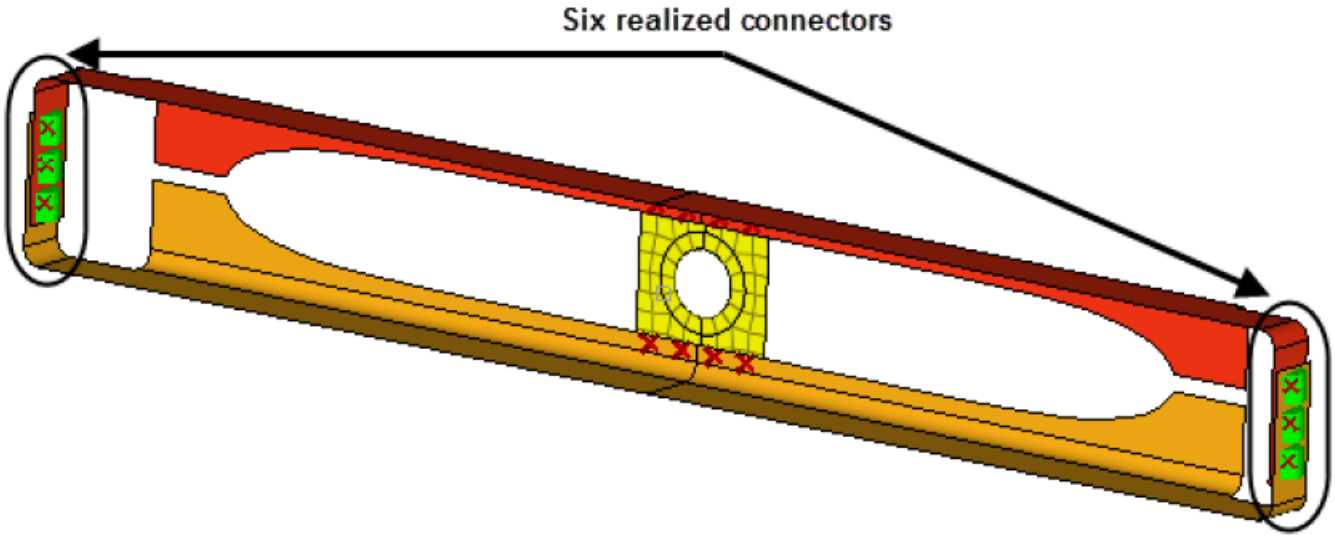


Figure 1037:

HyperMesh also adds fixed points to the surfaces at the ends of the weld elements to guarantee connectivity between the weld elements and the shell mesh that will be created on the surfaces.

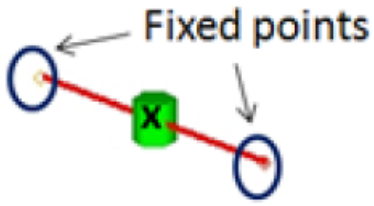


Figure 1038:

HyperMesh creates and realizes six spot connectors, and organizes them as geometry (not elements) in the current component collector, Con_Frt_Truss.

14. Click **return**.

Create A Shell Mesh

In this step, you will create a shell mesh on the front truss components of the model.

1. Open the Automesh panel by completing one of the following:
   - Click **Mesh** > **Create** > **2D Automesh** from the menu bar.
   - Press **F12**.
2. Open the **size and bias** subpanel.
3. Click **surfs** > **by collector**.
4. Select the **Front_Truss_1** and **Front_Truss_2** checkboxes.
5. Click **select**.
6. In the elem size= field, enter `10`.
7. Set the mesh type to **mixed**.
8. Set the elems to surf comp/elems to current comp toggle to **elems to surf comp**.
9. Switch the mesh mode from interactive to **automatic**.
10. Click **mesh**.
    HyperMesh meshes the surfaces.

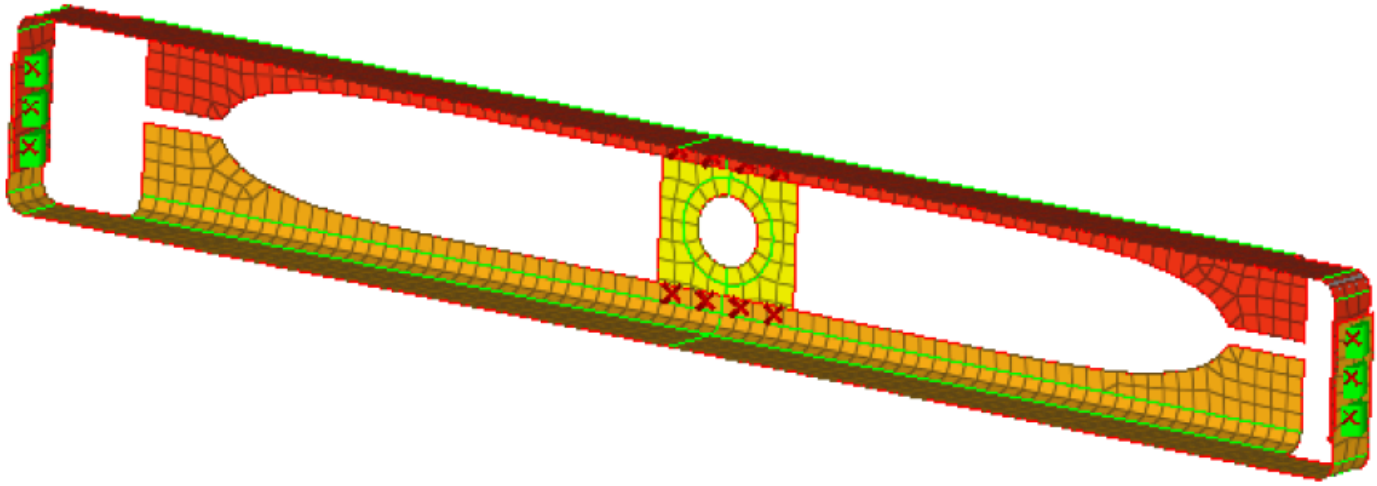


Figure 1039:

11. On the 3D View Controls toolbar, left-click on to zoom in on the area with a connector.

See how the fixed point created from the weld has ensured the mesh seeding passes through the weld.

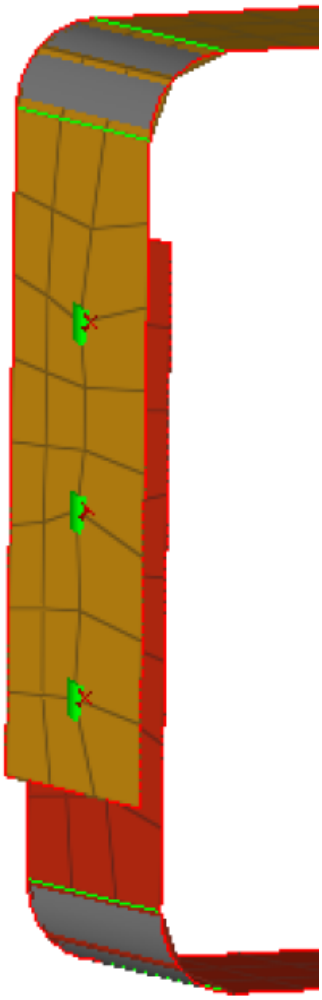


Figure 1040:

12. Click **return**.

Create Connectors

In this step, you will create connectors between the shell mesh for the front trusses and the reinforcement plate at pre-defined points.

In this step you will create and realize the connectors manually.

1. In the Model Browser, Component folders, right-click on **Con_Truss_Plate** and select **Make Current** from the context menu.
2. Open the connector Spot panel.
3. Open the **create** subpanel.
4. Set the location selector to **points**.
5. Click **points** > **by collector**.
6. Select the **Con_Truss_Plate** component.

7. Click **select**.
8. Set connect when to **now**.
9. Double-click **connect what: comps**.
10. Select the **Front_Truss_1**, **Front_Truss_2**, and **Reinf_Plate** components.
11. Click **select**.
12. Under the connect what selector, toggle from geom to **elems**.

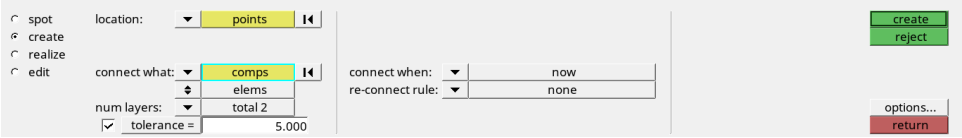


Figure 1041:

13. Click **create**.
    HyperMesh creates eight spot connectors at the selected weld points, and organizes them into the current component collector, Con_Truss_Plate.

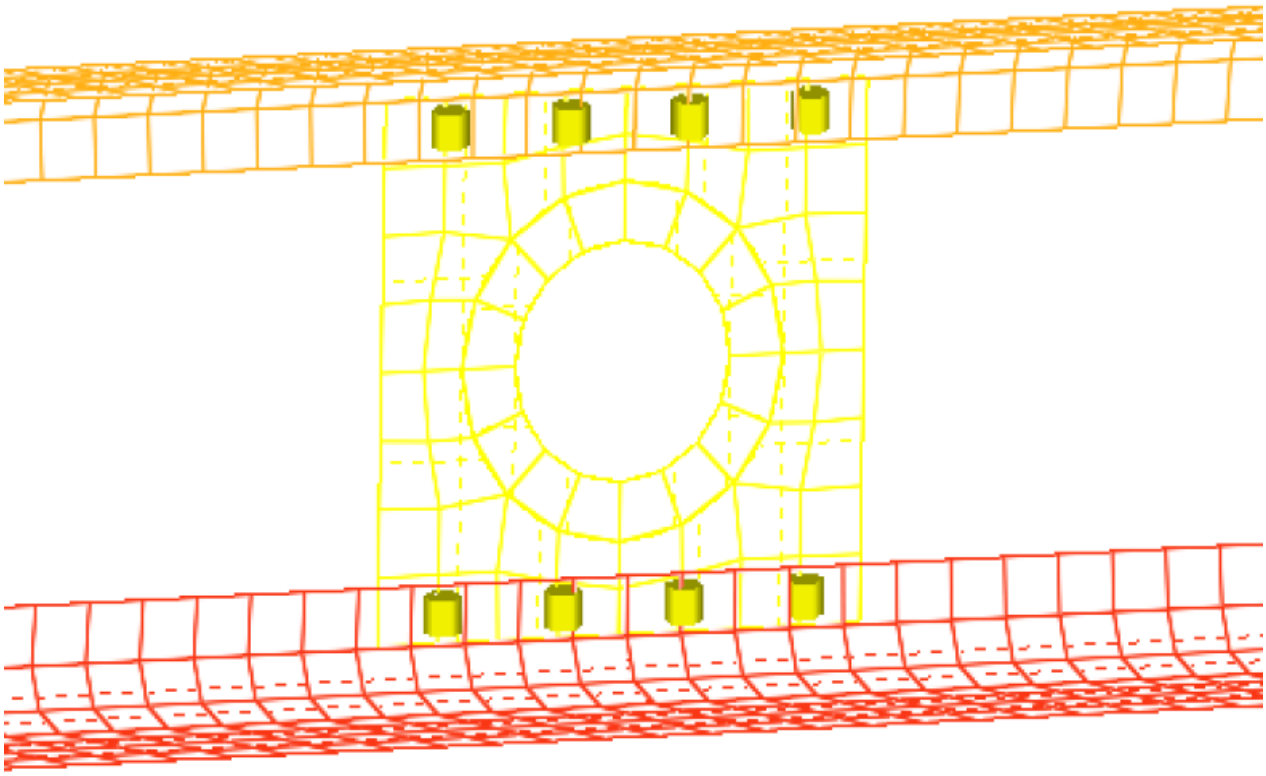


Figure 1042:

Realize the Connectors into Weld Elements

In this step, you will realize the connectors into weld elements.

1. Open the **realize** subpanel.
2. Click **location: connectors** > **by collector**.
3. Select the **Con_Truss_Plate** component.
4. Click **select**.
5. Click **type=** and select **weld**.
6. In the tolerance= field, enter `5`.
7. Set the mesh independent/mesh dependent toggle to **mesh dependent**.

> **Note:** When mesh dependent is active, HyperMesh equivalences the nodes if the realized finite element of the connector is coincident to a node of the shell mesh it is being connected to. If there are no suitable nodes present, this option partitions the mesh accordingly to ensure the mesh seeding passes through the weld point.

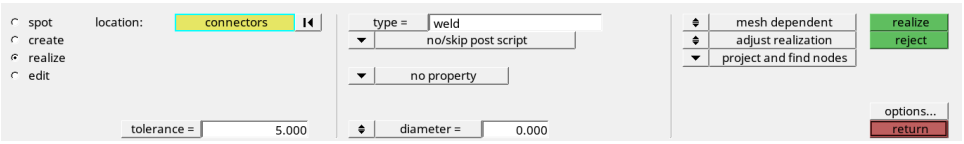


Figure 1043:

8. Click **realize**.
   HyperMesh realizes the selected connectors into weld elements.

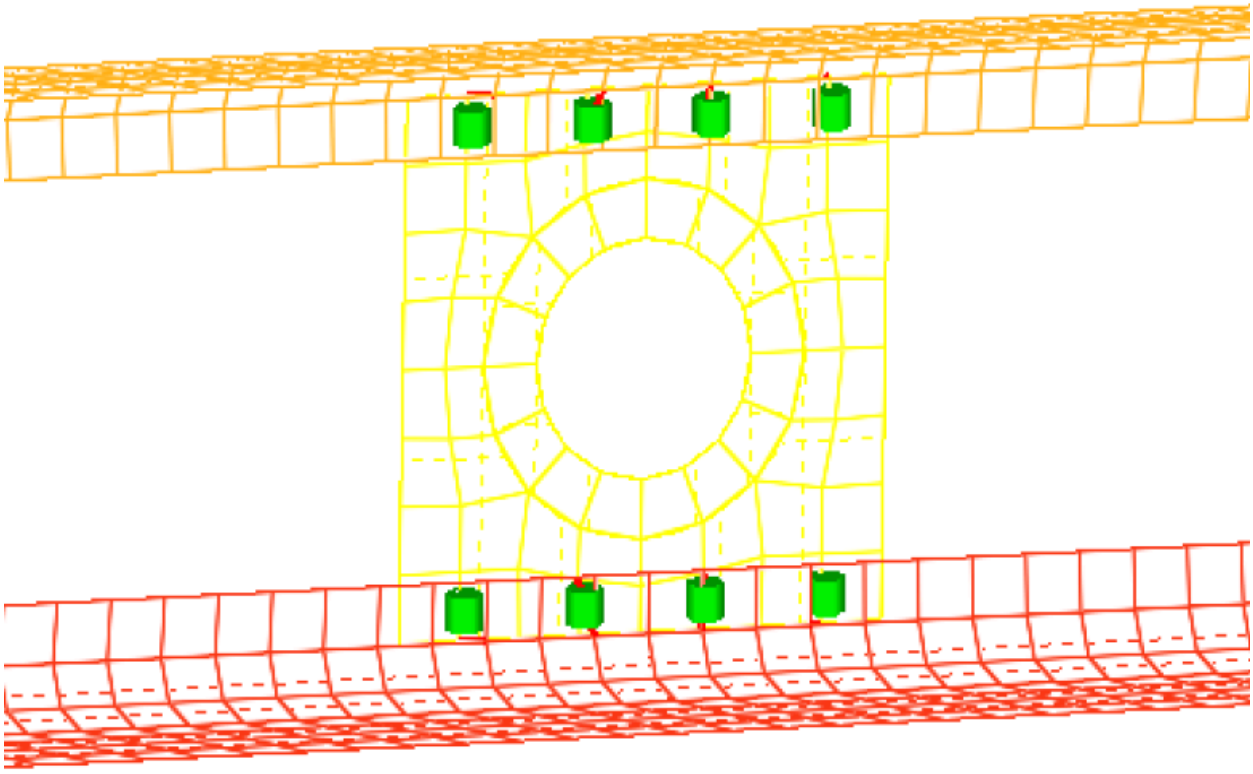


Figure 1044:

9. Click **return**.

Save Your Work

In this step, you will save your work.

This exercise is complete.

Save your work as a HyperMesh file.

5.5.2 Exercise 2: Weld Using a Master Connectors File and Duplicate and Reflect Connectors

In this exercise, you will becoming familiar with defining weld locations in HyperMesh by importing a master connectors file and Duplicating, reflecting, and updating connectors to create welds.

In this exercise, you will first weld the two right rails to each other and to the two front trusses depicted in Figure 1045. To do this you will import weld point data from a master connectors file, create connectors, and realize the connectors into LS-DYNA 100 Mat100 (beam) welds.

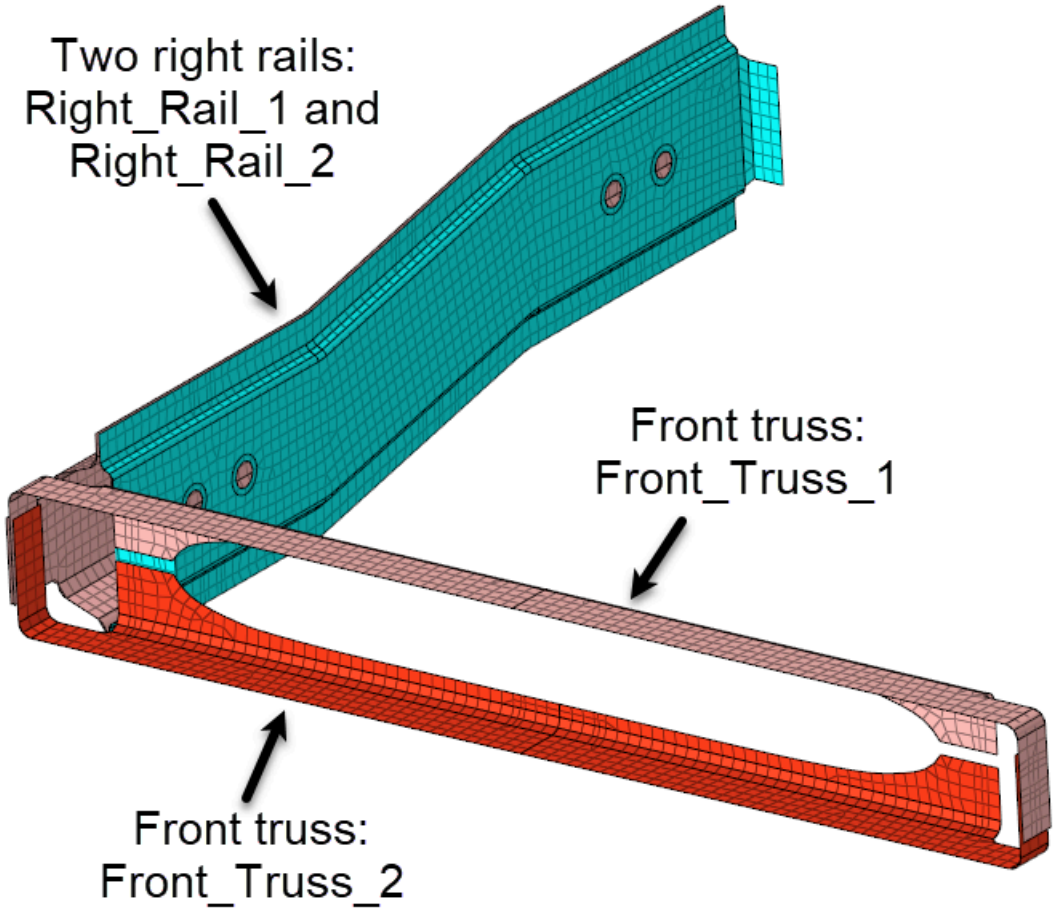


Figure 1045:

Second, you will weld the two left rails to each other and to the two front trusses depicted in Figure 1046. To do this you will duplicate and reflect the connectors that were created by importing the master connectors file, update the link information for the reflected connectors, and realize the connectors into LS-DYNA 100 Mat100 (beam) welds.

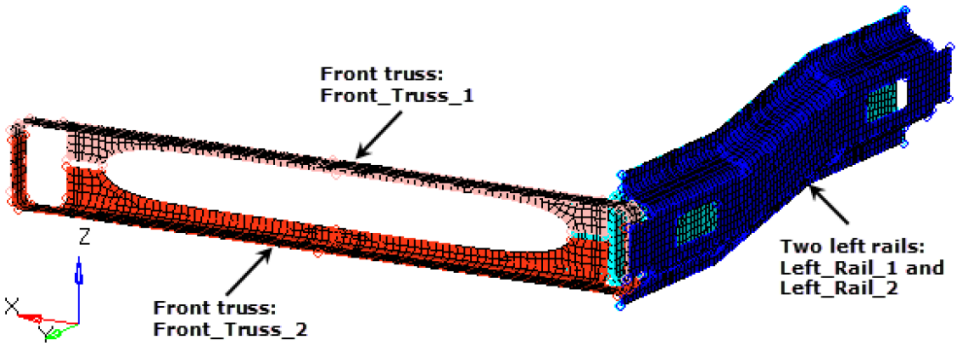


Figure 1046:

Third, you will combine adjacent 2T connectors into 3T connectors in order to update adjacent 2T welds to 3T welds.

Retrieve and View the Model File

In this step, you will retrieve and view the model file in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.
3. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
4. In the **Open Model** dialog, open the `frame_assembly_2.hm` file.
5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).

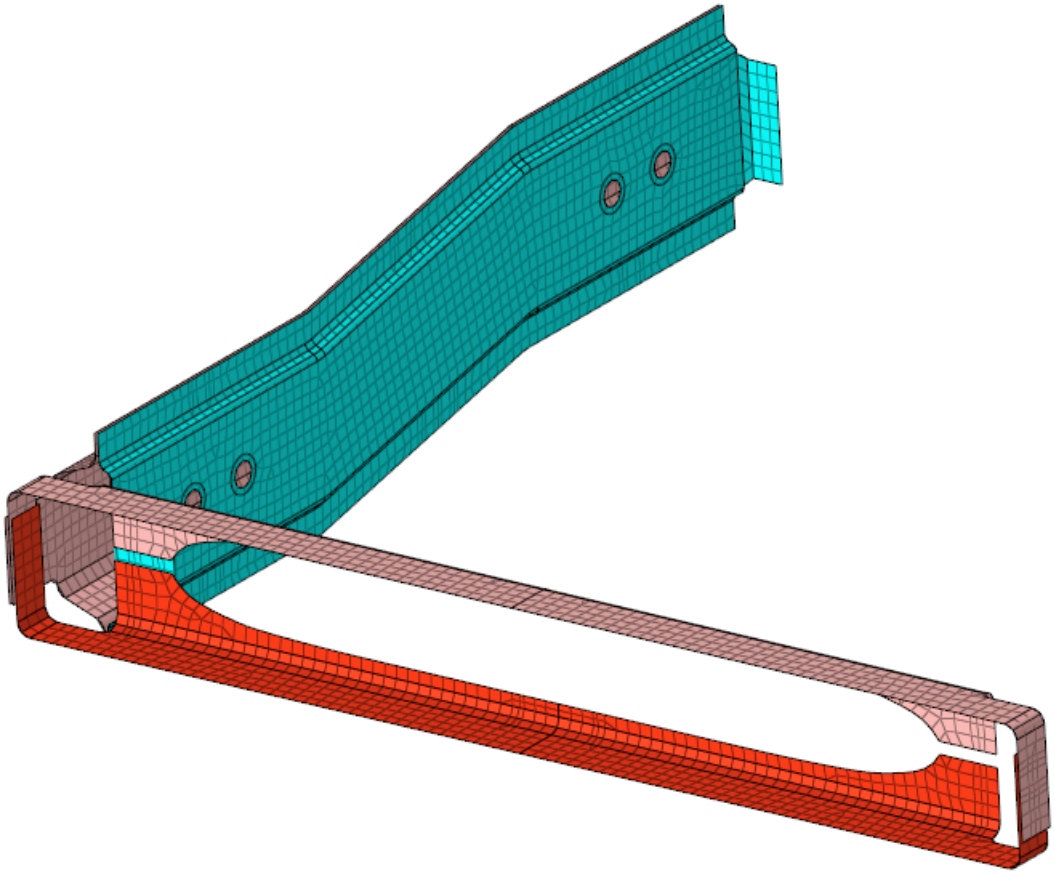


Figure 1047:

Create Connectors

In this step, you will create connectors to connect the right rails to each other and to the front trusses by importing a master connectors file.

1. From the menu bar, click **File** > **Import** > **Connectors**.
2. Set File type to **Connectors**.
3. In the File field, open the `rails_frt_truss.mwf` file.
4. Click **Import**.

HyperMesh imports the connectors from the file, and organizes them into a new component, CE_Locations.

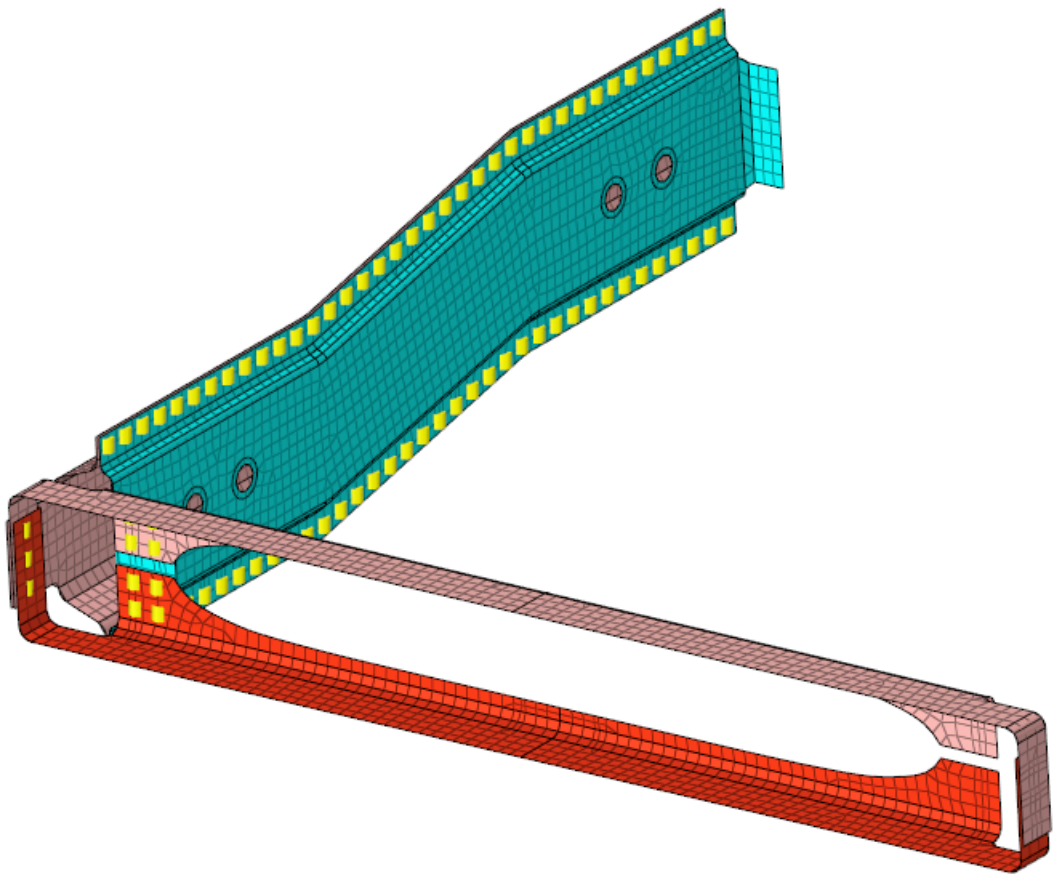


Figure 1048:

Realize the Connectors into LS-DYNA 100 Mat100 Welds

In this step, you will realize the connectors in the component, CE_Locations, into LS-DYNA 100 Mat100 (beam) welds.

1. In the Model Browser, Component folder, right-click on **CE_Locations** and select **Make Current** from the context menu.

2. Open the realize subpanel within the connector Spot panel by clicking **Connectors** > **Realize** > **Spots** from the menu bar.
3. Click **location: connectors** > **by collector**.
4. Select the **CE_Locations** component.

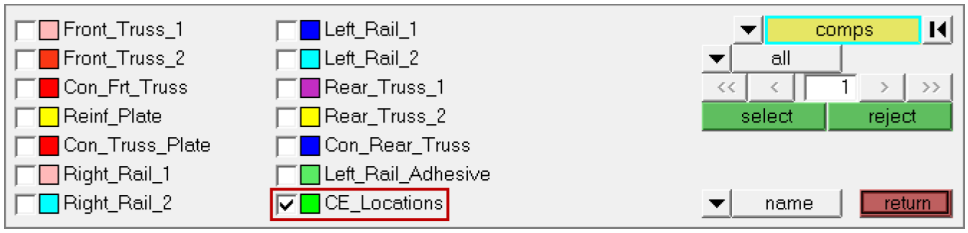


Figure 1049:

5. Click **select**.
6. In the tolerance = field, enter `5`.
7. Set the mesh independent/mesh dependent toggle to **mesh independent**.
8. Click **type =** and select **mat100**.

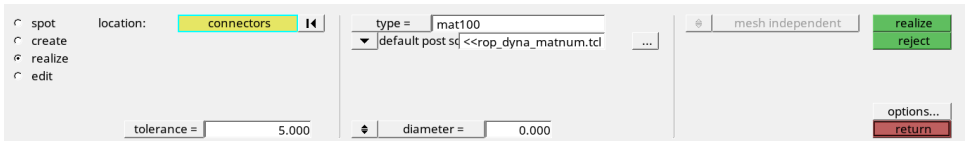


Figure 1050:

9. Click **realize**.

> **Note:** The naming convention for these components is C_^_[id of comp 1]_[id of comp 2].

HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds and organizes them into the following new component collectors:

- C_^_1_7_BEAM_100
- C_^_6_7_BEAM_100
- C_^_2_7_BEAM_100
- C_^_1_6_BEAM_100

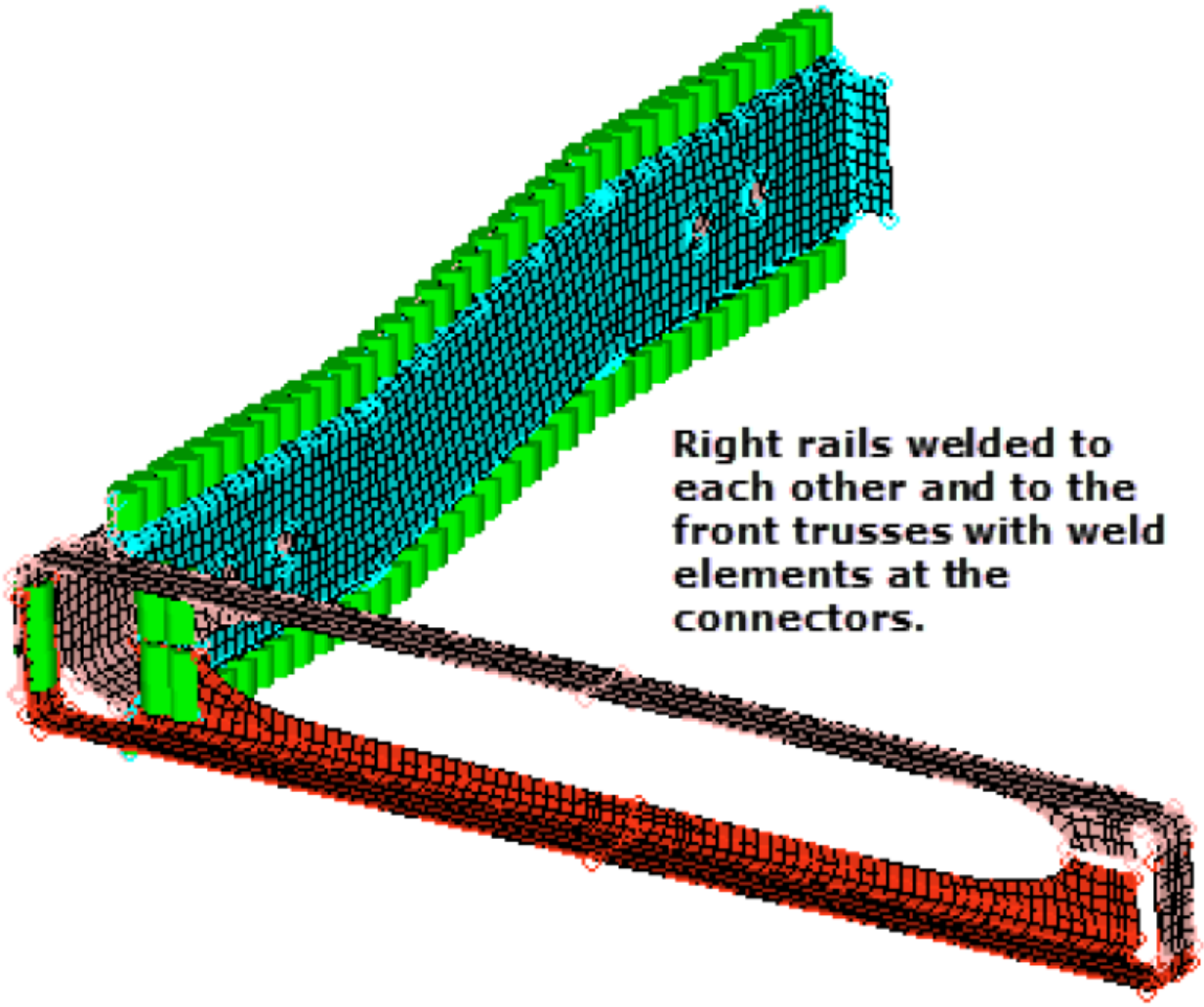


Figure 1051:

10. Click **return**.

Verify Materials were Automatically Created

In this step, you will verify materials were automatically created when the LS-DYNA 100 Mat100 (beam) welds were created.

1. In the Model Browser, expand the **Material** folder.
2. Review the materials with IDs 2 through 5, that were created when the LS-DYNA 100 Mat100 (beam) welds were created.

> **Note:** The naming convention for the materials is M_^_[id of comp 1]_[id of comp 2]. One material is created for every two components that are connected.

3. Select one of the materials with IDs 2 through 5.

   Note: The solver keyword is set to *MAT_SPOTWELD, and the values are automatically specified for the density, Young's Modulus, and Poisson's Ratio parameters.

   The Entity Editor opens and displays the material's card details.

Verify Properties were Automatically Created

In this step, you will verify that properties were automatically created when the LS-DYNA 100 Mat100 (beam) welds were created.

1. In the Model Browser, expand the **Property** folder.
2. Review the properties with IDs 1 through 6, that were created when the LS-DYNA 100 Mat100 (beam) welds were created.

   Note: The naming convention for the properties is P_^_[id of comp 1]_[id of comp 2]. One property is created for every two components that are connected

3. Select one of the properties with IDs 1 through 6. The Entity Editor opens and displays the property's card details.

   Note: The solver keyword is set to *SECTION_BEAM, and the default values are specified for the property's parameters.

   The Entity Editor opens and displays the property's card details.

Verify a Contact was Automatically Created

In this step, you will verify that a contact was automatically created when the LS-DYNA 100 Mat100 (beam) welds were created.

1. In the Model Browser, Group folder, click **C_Contact_Spotweld_1**.

   Note: The solver keyword is set to *CONTACT_SPOTWELD_ID.

   The Entity Editor opens and displays the group's card details.
2. Open the Interfaces panel by clicking **BCs** > **Assign** > **Interface Entities** from the menu bar.
3. In the panel area, click **name=** and select **C_Contact_Spotweld_1**.
4. Click **review** to review the contacts.

   The contact's master elements temporarily display blue and its slave elements temporarily display red.

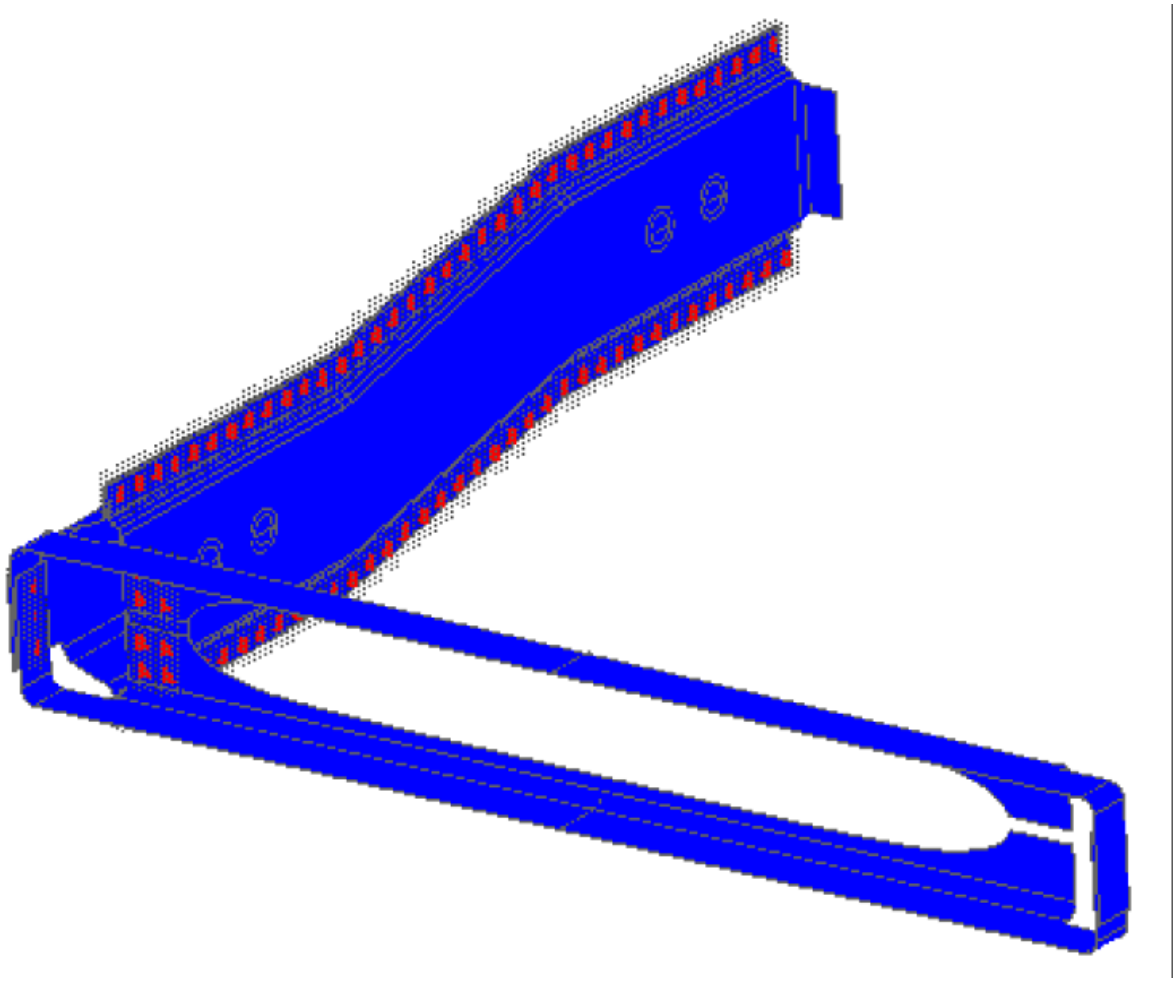


Figure 1052:

5. Click **return**.

Display the Rail Components

In this step, you will display the components Left_Rail_1 and Left_Rail_2 for elements.

In the Model Browser, Component folder, right-click on **Left_Rail_1** and **Left_Rail_2** and select **Show** from the context menu.

> **Tip:** Select multiple components by pressing **Ctrl** while left-clicking on the components.

The components display in the graphics area.

Duplicate and Reflect the Connectors

In this step, you will duplicate and reflect the connectors created from the master connectors file.

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu.
   A new component opens in the Entity Editor.
2. For Name, enter `CE_Locations_Dup`.

   Note: You do not need to assign any materials or properties.
3. Open the Reflect panel by clicking **Connectors** > **Reflect** > **Connectors** from the menu bar.
4. Click **connectors** > **by collector**.
5. Select the **CE_Locations** component.
6. Click **select**.
7. Click **connectors** > **duplicate** > **current comp**.
   HyperMesh duplicates the displayed connectors, are organizes them into the current component, CE_Locations_Dup.
8. Set the orientation selector to **x-axis**.

   Note: This is the axis normal to the plane of interest.
9. Reflect about the base node by clicking B.
10. Click **x =**.
    HyperMesh activates the x =, y =, and z = fields. By default their values are 0.000, which is the base point you want to reflect about.

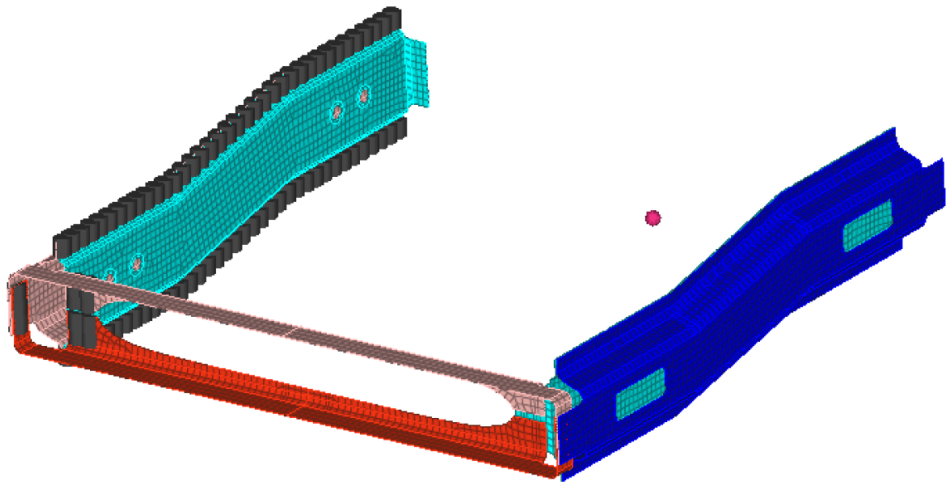


Figure 1053:

11. Click **return**.
12. Click **reflect**.
    HyperMesh reflects the connectors.
13. Click **return**.

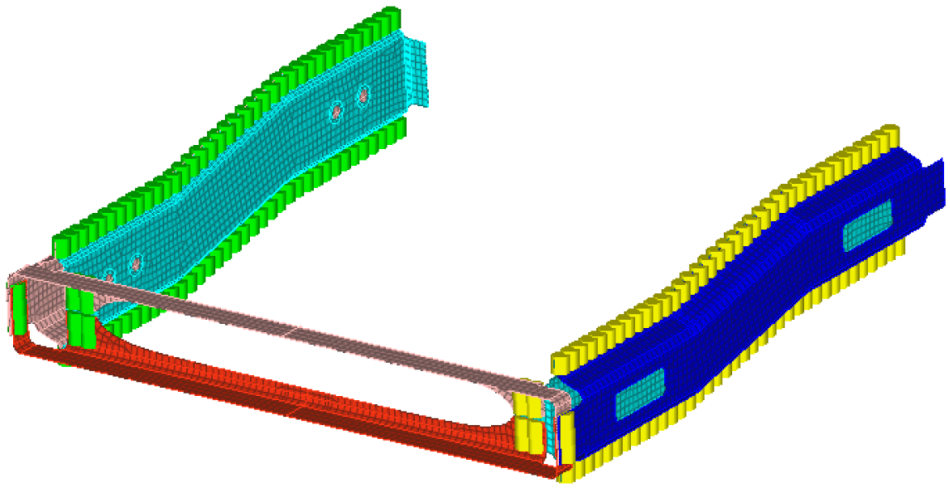


Figure 1054:

Update the Connectors

In this step, you will update the connectors for the left rails to link them to the left rail components.

1. Open the Connector Browser by clicking **View** > **Browsers** > **HyperMesh** > **Connector** from the menu bar.
2. In the Connector Entity Browser, expand the **mat 100** folder, which contains the connectors in the model with material mat100.

   Note: The reflected connectors are displayed in yellow, which indicates that they are unrealized.

3. Select all of the unrealized connectors in the list.

Entities	Layer	Tolerance	Link1	Link2	Link3	RealizeTo	State
188	2	5.0	Right_Rail_1	Right_Rail_2		property script	realized
187	2	5.0	Right_Rail_1	Front_Truss_1		property script	realized
186	2	5.0	Right_Rail_1	Front_Truss_1		property script	realized
185	2	5.0	Front_Truss_1	Right_Rail_1		property script	realized
270	2	5.0	Front_Truss_1	Right_Rail_2		property script	unrealized
271	2	5.0	Front_Truss_1	Right_Rail_2		property script	unrealized
272	2	5.0	Front_Truss_1	Right_Rail_2		property script	unrealized
273	2	5.0	Front_Truss_1	Right_Rail_2		property script	unrealized
274	2	5.0	Front_Truss_2	Right_Rail_2		property script	unrealized
275	2	5.0	Front_Truss_2	Right_Rail_2		property script	unrealized
276	2	5.0	Front_Truss_2	Right_Rail_2		property script	unrealized
277	2	5.0	Right_Rail_2	Front_Truss_2		property script	unrealized
278	2	5.0	Right_Rail_2	Right_Rail_1		property script	unrealized
279	2	5.0	Right_Rail_1	Right_Rail_2		property script	unrealized
280	2	5.0	Right_Rail_1	Right_Rail_2		property script	unrealized

Figure 1055:

4. In the Entities column, right-click on the selected connectors and select **Update Link** from the context menu.

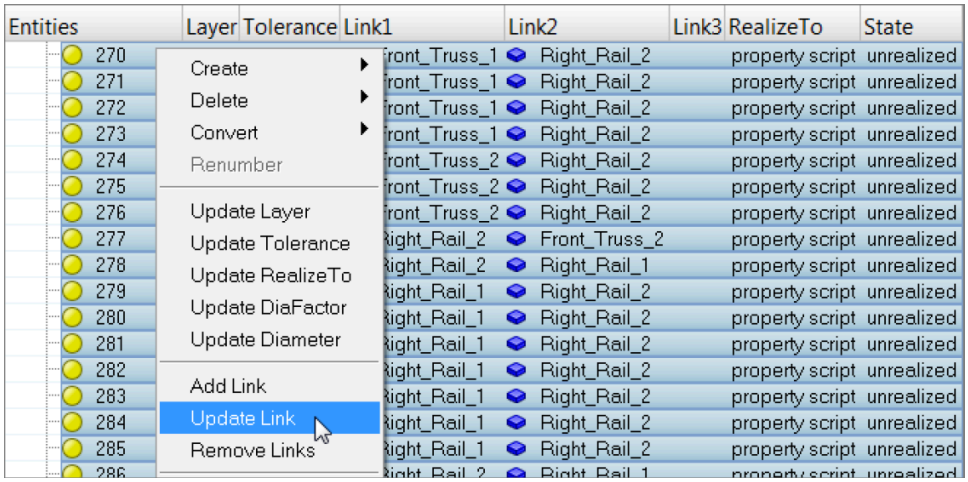


Figure 1056:

The Update window opens.

5. In the Search column, set the Link Type to **comps**.

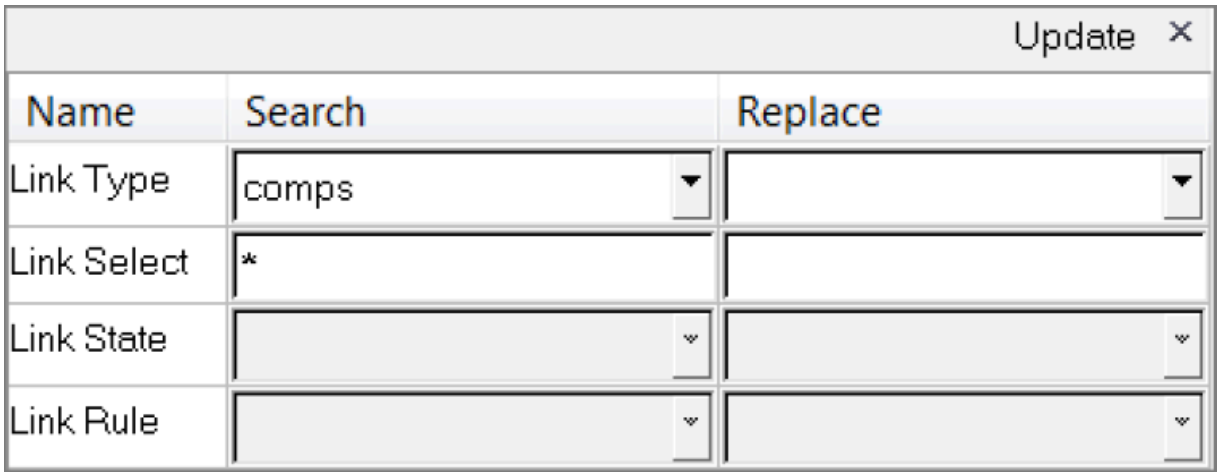


Figure 1057:

6. In the Search column, click the **Link Select** field.
7. In the panel area, click **component**.
8. Select the **Right_Rail_1** component.
9. Click **proceed**.
   The Link Select field displays Right_Rail_1.
10. In the Replace column, set the Link Type to **comps**.
11. In the Replace column, click the **Link Select** field.
12. In the panel area, click **component**.
13. Select the **Left_Rail_1** component.
14. Click **proceed**.
    The Link Select field displays Left_Rail_1.

15. Click **Update**.
    HyperMesh updates the connector's links.
16. Repeat 9 through 15, except search for the Right_Rail_2 component and replace it with the Left_Rail_2 component.
17. Ensure none of the connectors in the list of unrealized connectors are linked to the right rail components.
18. Close the Update window by clicking **X** next to Update.
19. Right-click on the selected connectors and select **Rerealize** from the context menu.
    HyperMesh realizes the connectors.

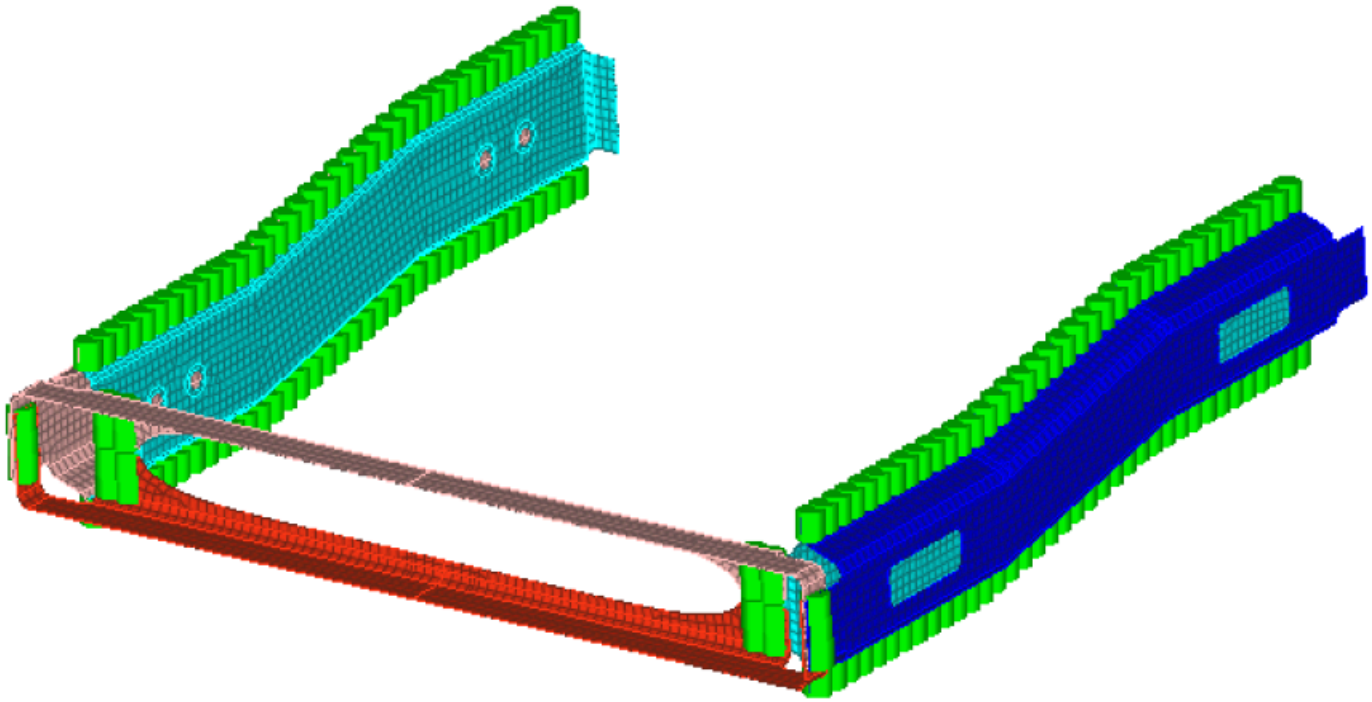


Figure 1058:

Verify Connectors are Realized and Identify Ajacent Connector Pairs

In this step, you will verify that all connectors are realized and identify the pairs of adjacent connectors.

All of the connectors listed in the Connector Entity Browser are displayed in green in the State column, which indicates that they are realized.

1. Zoom in on one of the two areas where the front trusses are connected to the rail components.

   Note: In these two areas, pairs of adjacent connectors exist.

2. On the Visualization toolbar, click .

3. In the Visualization tab, click .
4. Under Color by, select **Layer**.

> **Note:** This option indicates that each of these connectors link two components. Because each pair of connectors creates a series of two weld elements, you can combine each pair into a single connector, which links the three components together.

HyperMesh changes the connectors color to purple because under Layers, 2t (two thickness) is defined by the color purple.

Isolate the Pairs of Adjacent 2t Connectors

In this step, you will isolate the pairs of adjacent, 2t connectors.

1. In the Model Browser, turn off the geometry display for all of the components.
2. In the Connector Browser, Link Entity Browser, while pressing **Ctrl**, select **Front_Truss_1, Front_Truss_2**, **Right_Rail_1**, and **Left_Rail_1**.
3. Right-click and select **Find Between** from the context menu.

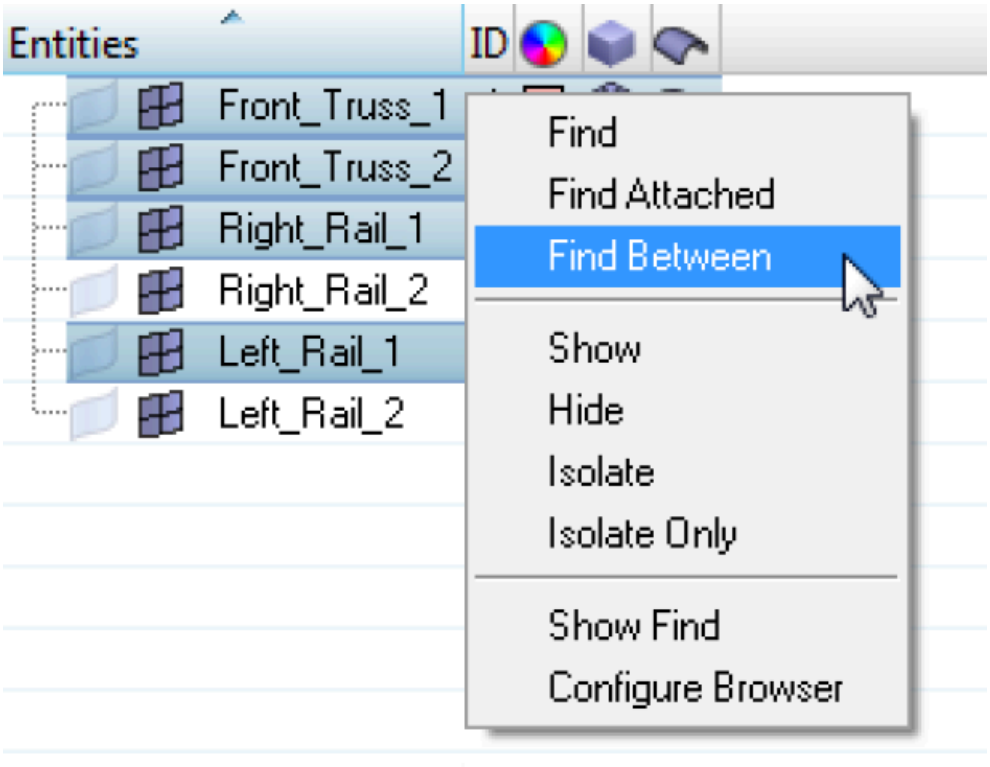


Figure 1059:

HyperMesh finds and displays the connectors between the four components you selected.

Unrealize the Selected Connectors

In this step, you will unrealize the selected connectors.

The connectors you found in Isolate the Pairs of Adjacent 2t Connectors should still be selected in the Connector Entity browser.

In the Connector Entity Browser, right-click on the selected connectors and select **Unrealize** from the context menu.

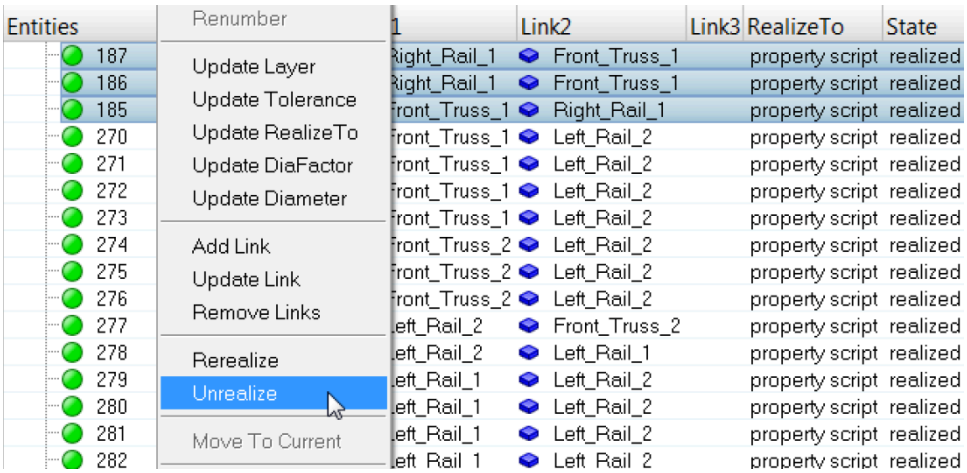


Figure 1060:

Save Your Work

In this step, you will save your work in HyperMesh.

This exercise is complete.

Save your work as a HyperMesh file.

5.5.3 Exercise 3: Create Connectors from Existing Welds to Create New Welds of a Different Type

In this exercise, you will become familiar with absorbing existing finite element welds into connectors in order to create new finite element welds of a different type.

In this exercise, LS-DYNA 101 Mat100 (hexa) welds already connect the rear trusses to each other.

You will first update the weld type to LS-DYNA 100 Mat100 (beam) welds. To do this you will create connectors from the existing LS-DYNA 101 Mat100 (hexa) welds, as seen in Figure 1061, and then realize the connectors into LS-DYNA 100 Mat100 (beam) welds.

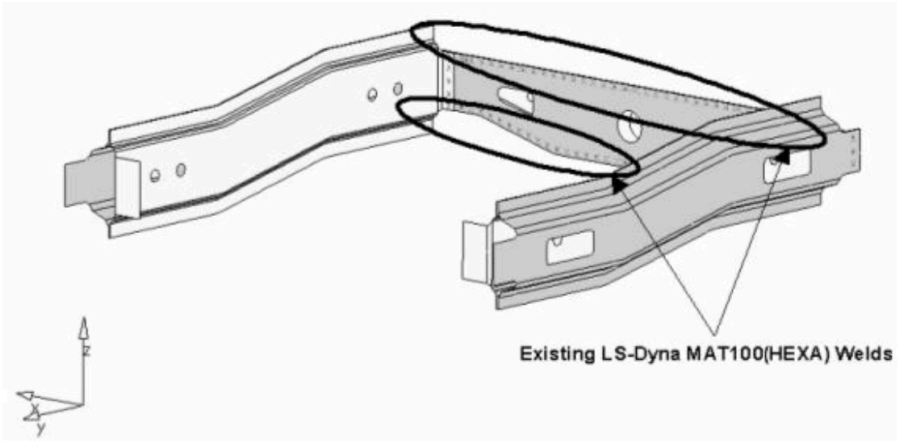


Figure 1061:

Second, you will update the existing LS-DYNA 101 MAT100 (hexa) welds to LS-DYNA 100 Mat100 (beam) welds.

Retrieve and View the Model File

In this step, you will retrieve and view the model file in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.
3. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
4. In the **Open Model** dialog, open the `frame_assembly_3.hm` file.
5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).

Create Connectors from Existing Welds

In this step, you will create connectors from existing LS-DYNA 101 Mat100 (hexa) welds.

1. From the menu bar, click **Connectors** > **Fe Absorb**.
2. In the **Automated Connector Creation and FE Absorption** dialog, set FE configs to **custom**.
3. Set FE type to **dyna 101 mat 100 (hexa)**.
4. Set the Elem filter toggle to **select**.
5. Double-click the **Elem filter: Elements** selector.
6. In the panel area, click **elems** > **by collector**.
7. Select the following components:
   - **C_^_6_11_HEX**

- **C_^_7_11_HEX**
- **C_^_8_11_HEX**
- **C_^_9_11_HEX**
- **C_^_10_11_HEX**

8. Click **select**.
9. Click **proceed** to return to the dialog.
10. Set FE connectivity to **Mesh independent** (projs).
11. Select the **Move connectors to FE component** checkbox.
12. Click **Absorb**.
    HyperMesh absorbs the elements into connectors at the locations of the LS-DYNA 101Mat100 (hexa) welds, and organizes them into the respective components to which the LS-DYNA 101 Mat100 (hexa) welds belong.
13. Click **Close**.

Isolate the 2t Connectors

In this step, you will isolate the 2t connectors between the Rear_Truss_2 component and the Right_Rail_2 and Left_Rail_2 components.

1. Open the Connector Browser by clicking **View** > **Browsers** > **HyperMesh** > **Connector** from the menu bar.
2. In the Link Entity browser, while pressing **Ctrl**, select **Rear_Truss_2**, **Right_Rail_2**, and **Left_Rail_2**.
3. Right-click and select **Find Between** from the context menu.
   HyperMesh finds and displays ten connectors between the three components you selected.

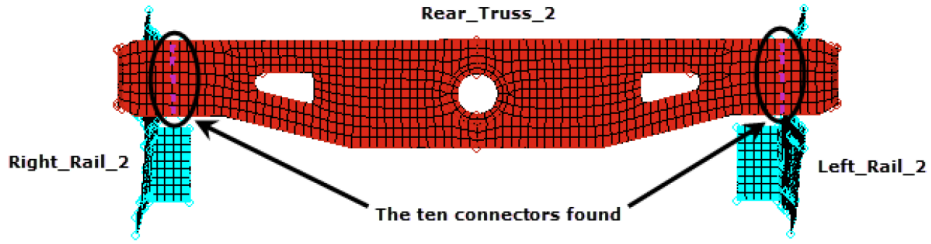


Figure 1062:

Add Rear_Truss_1 as a Link

In this step, you will add Rear_Truss_1 as a third link to four of the ten displayed 2t connectors.

1. In the Model Browser, Component folder, while pressing **Ctrl**, select **Rear_Truss_2**, **Left_Rail_2**, and **Right_Rail_2**.
2. Right-click on the selected components and select **Show** from the context menu.

3. Turn on the geometry display for the **C^9_11_HEX** component .
4. Open the Add Links panel by clicking **Connectors** > **Assign** > **Links** from the menu bar.
5. Activate the **location: connectors** selector.
6. Select the four connectors as indicated in Figure 1063.

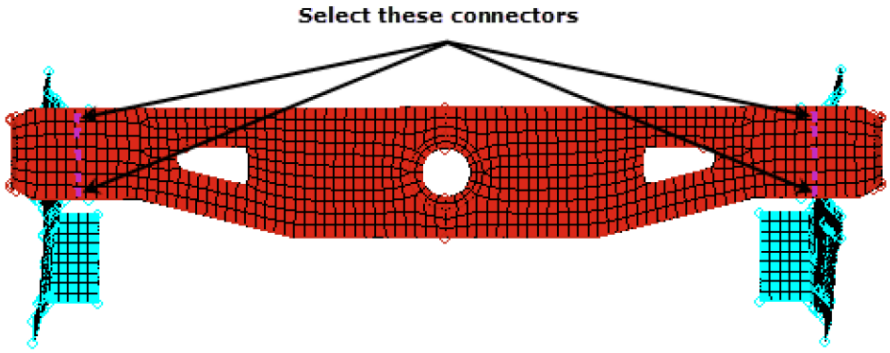


Figure 1063:

7. Set the connect what selector to **comps**.
8. Click **comps**.
9. Select the **Rear_Truss_1** component.
10. Click **select**.
11. Set the connect what toggle to **elems**.
12. Select the checkbox next to search tol =.
13. In the search tol= filed, enter `5`.
14. Set # of layers to **total 3**.

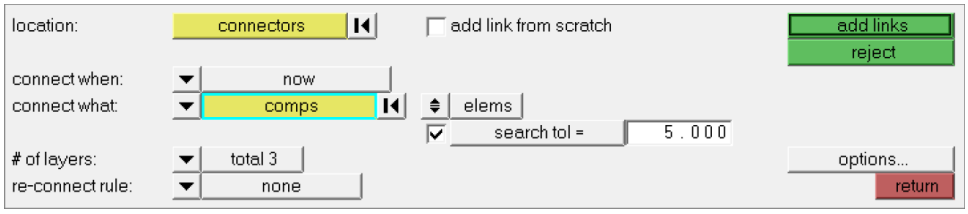


Figure 1064:

15. Click **add links**.
    HyperMesh updates the connector's definitions, and changes their display color to blue, as they are now 3t connections.

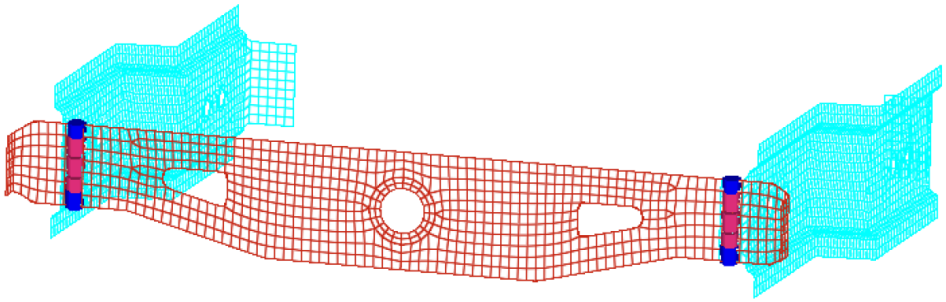


Figure 1065:

16. Click **return**.

Unrealize the Connectors for the Welds

In this step, you will unrealize the connectors for the LS-DYNA 101 Mat100 (hexa) welds.

1. In the Model Browser, turn on the geometry display for the following components only:
   - **C_^_6_11_HEX**
   - **C_^_7_11_HEX**
   - **C_^_8_11_HEX**
   - **C_^_9_11_HEX**
   - **C_^_10_11_HEX**
2. Open the Unrealize panel by clicking **Connectors** > **Unrealize** from the menu bar.
3. Click **connectors** > **displayed**.
4. Click **unrealize**.
   HyperMesh unrealizes the connectors, and deletes the weld elements associated to these connectors.
5. Click **return**.
6. On the Visualization toolbar, click .
7. In the Visualization tab, click .
8. Under Color by, select **State**.
   HyperMesh displays the connectors in yellow, which indicates that they are not realized into FE elements.



Figure 1066:

Realize the Unrealized Connectors

In this step, you will realize the unrealized connectors into LS-DYNA 100 Mat100 (beam) welds.

1. Open the **realize** subpanel of the connector Spot panel by clicking **Connectors** > **Realize** > **Spots** from the menu bar.
2. Click **location: connectors** > **displayed**.
3. Click **type=** and select **mat 100**.
4. In the tolerance = field, enter `5`.
5. Click **realize**.
   HyperMesh realizes the connectors.

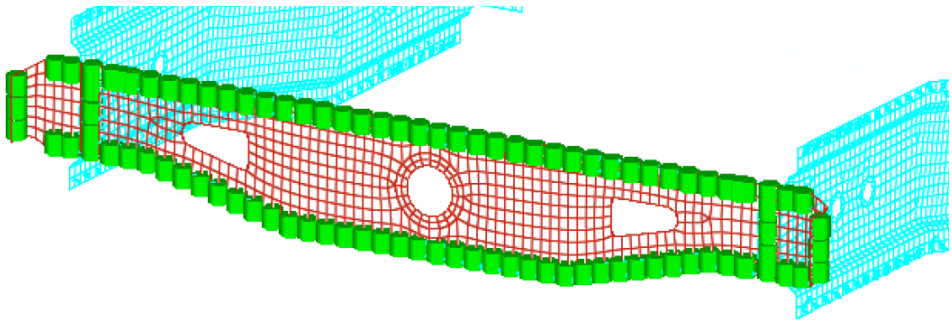


Figure 1067:

6. Click **return**.

Save Your Work

In this step, you will save your work in HyperMesh.

This exercise is complete.

Save your work as a HyperMesh file.

5.5.4 Exercise 4: Swap Welded Part

In this exercise, you will become familiar with swapping welded parts and updating their affected connections (welds).

In this exercise, you will replace the component Rear_Truss_1 with a new, similar part and update its affected connections (welds). To do this you will update the connectors to use the "use name" rule, delete the old part, import the new part, and then realize the corresponding connectors into LS-DYNA 100 Mat100 (beam) welds.

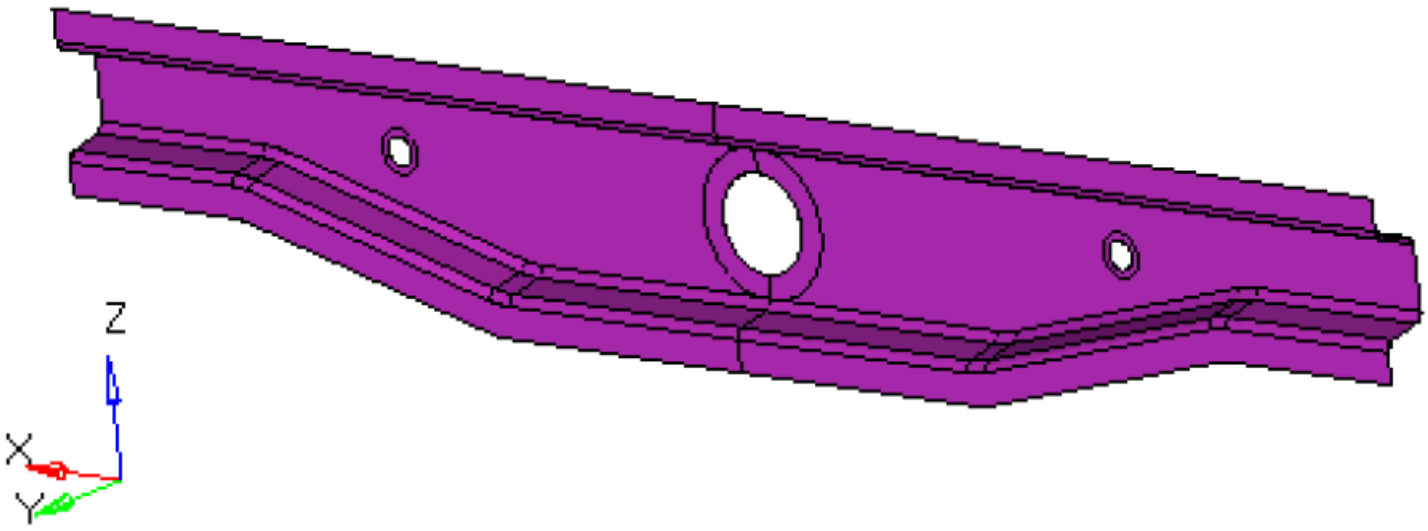


Figure 1068:

Retrieve and View the Model File

In this step, you will retrieve and view the model file in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.
3. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
4. In the **Open Model** dialog, open the `frame_assembly_4.hm` file.
5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).

Import the Model and Update the Connectors

In this step, you will import the new model and update the connectors to use the "use name" rule when reconnecting parts.

1. Open the Connector Browser by clicking **View** > **Browsers** > **HyperMesh** > **Connector** from the menu bar.
2. In the Connector Entity Browser, right-click and select **Configure Browser** from the context menu.
3. In the **Browser Configuration** dialog, click the **Options** tab.
4. Under Local connector options, select the **Extended information** checkbox.

Note: This option allows you to edit connector attributes.

5. Click **OK**.
6. From the menu bar, click **File** > **Import** > **Model**.
7. In the Import tab, click .
8. In the **Open** dialog, navigate to your working directory and open the `rear_truss_1_new.hm` file.
9. Click **Import**.
10. In the Link Entity Browser, right-click on **Rear_Truss_1** and select **Find** from the context menu.
    HyperMesh isolates the component Rear_Truss_1 and the connectors attached to it in the graphics area. Also, HyperMesh highlights the corresponding connectors in the Connector Entity browser.
11. In the Connector Entity Browser, right-click on the selected connectors and select **Update Link** from the context menu.

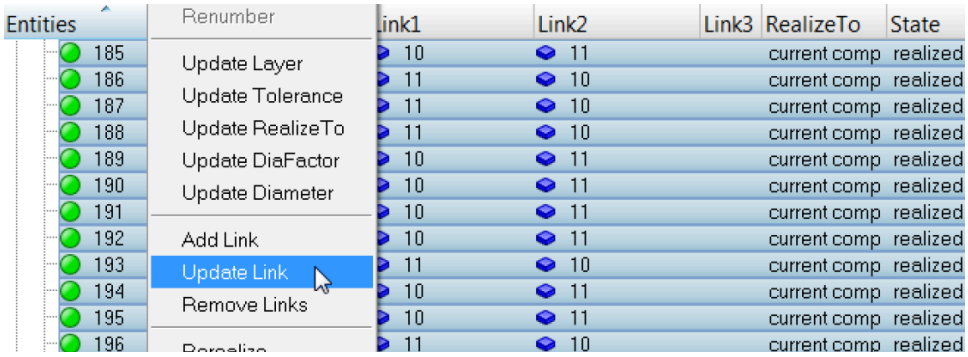


Figure 1069:

12. In the Update window, Search column, click the **Link Select** field.
13. In the panel area, click **component**.
14. Select the **Rear_Truss_1** component.
15. Click **proceed**.
    The Link Select field displays Rear_Truss_1.
16. In the Search column, set Link Rule to **none**.



Figure 1070:

17. In the Replace column, click the **Link Select** field.
18. In the panel area, click **component**.
19. Select the **Rear_Truss_1.1** component.
20. Click **proceed**.
    The Link Select field displays Rear_Truss_1.1.
21. In the Replace column, set Link Rule to **use-name**.



Figure 1071:

22. Click **Update**.
    HyperMesh updates the connector links.
23. Close the Update window by clicking **X** next to Update.

Delete the Rear_Truss_1 Component

In this step, you will delete the Rear_Truss_1 component.

1. Open the Delete panel by pressing **F2**.
2. Set the entity selector to **comps**.
3. Click **comps**.
4. Select the **Rear_Truss_1** component.
5. Click **select**.
6. Click **delete entity**.
   The display color of the connectors associated to the deleted component changes from green (realized state) to yellow (unrealized state), because the deleted component was one of the connectors' links. Also, HyperMesh deletes the existing finite element welds.

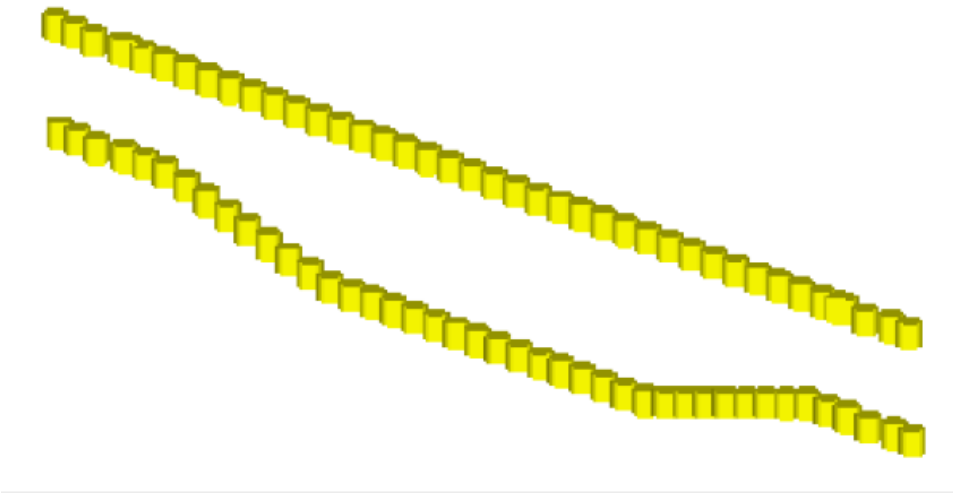


Figure 1072:

7. Click **return**.

Realize the Unrealized Connectors

In this step, you will realize the unrealized connectors.

1. Open the connector Spot panel by clicking **Connectors** > **Realize** > **Spots** from the menu bar.
2. Click **location: connectors** > **displayed**.
3. Click **type=** and select **mat100**.
4. In the tolerance =, enter `5`.
5. Click **realize**.
   HyperMesh realizes the connectors.
6. Click **return**.

Save Your Work

In this step, you will save your work in HyperMesh.

This exercise is complete.

Save your work as a HyperMesh file.

5.5.5 Exercise 5: Troubleshoot Failure of Connectors to Realize

In this exercise, you will become familiar with troubleshooting the failure of connectors to realize. This exercise will help you identify issues related to small projection tolerance and missing link definitions.

In this exercise, you will realize connectors to weld parts of a vehicle door frame. The model is depicted in Figure 1073.

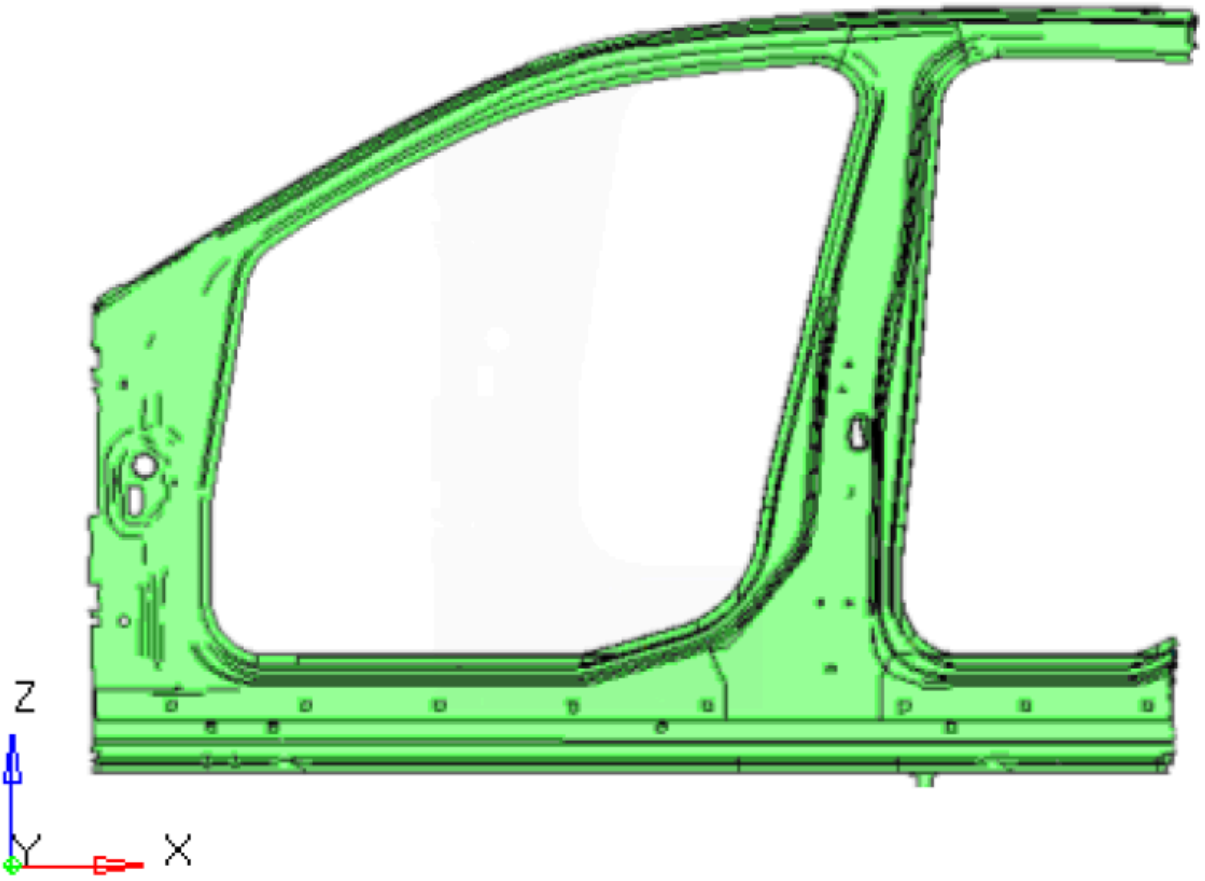


Figure 1073:

Retrieve and View the Model File

In this step, you will retrieve and view the model file in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.
3. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
4. In the **Open Model** dialog, open the `body_side_assembly.hm` file.
5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).

Realize the Connectors

In this step, you will realize the connectors using a projection tolerance of 1.0.

1. Open the connector Spot panel by clicking **Connectors** > **Realize** > **Spots** from the menu bar.
2. Click **location: connectors** > **all**.
3. Click **type =** and select **mat100**.
4. In the tolerance =, enter `1.0`.
5. Click **realize**.

   Note: The Status bar reads, "257 connectors realized (9 failed)".

   HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds.
6. Click **return**.

Review the Information Table and Determine Reasons for Failure

In this step, you will review the information table listing the connectors that failed to realize and determine the reasons for failure.

1. On the Visualization toolbar, click  .
2. In the Visualization tab, click  .
3. Under State, clear the **Realized** checkbox.
   The display of the realized (green) connectors turns off.

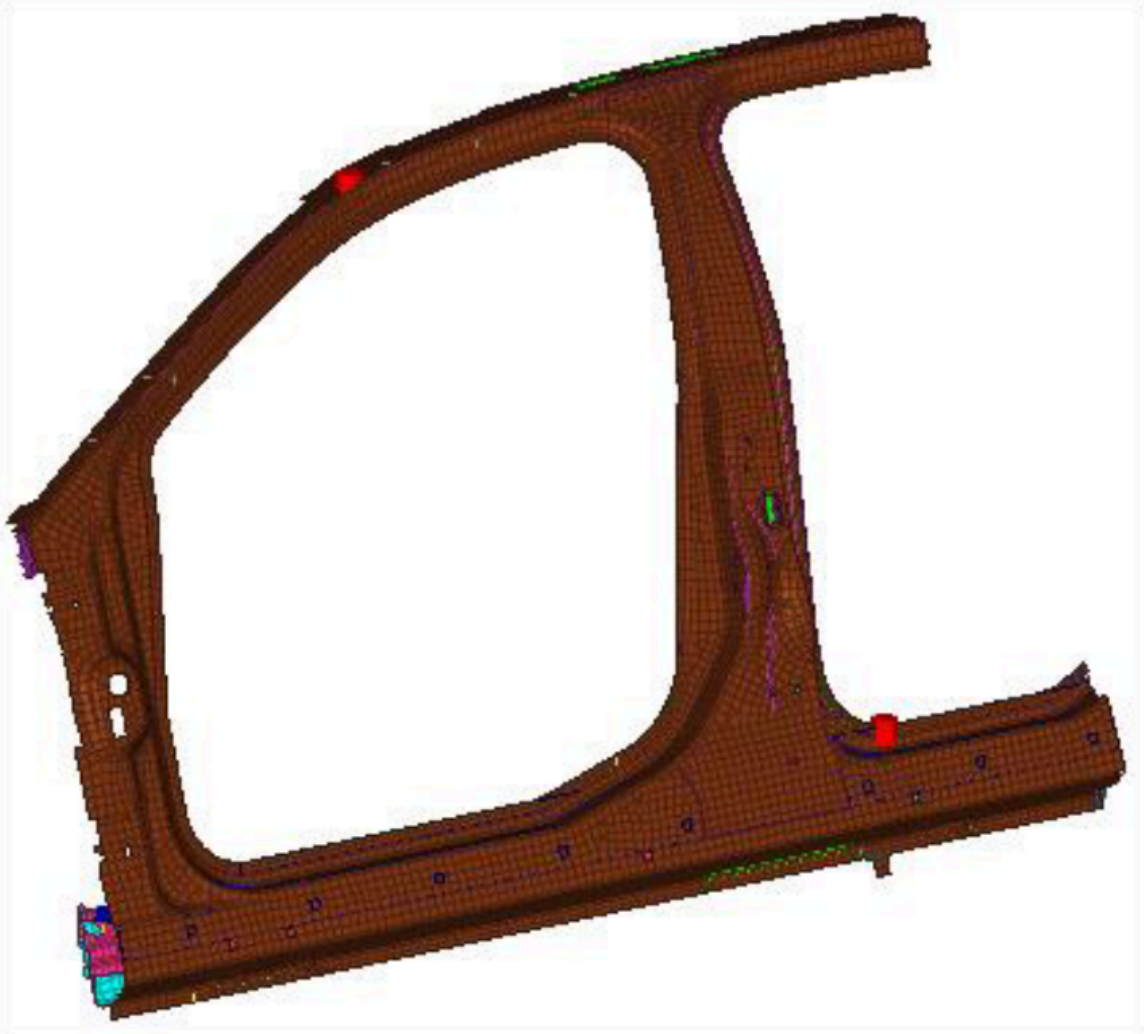


Figure 1074:

4. Open the Connector Browser by clicking **View** > **Browsers** > **HyperMesh** > **Connector** from the menu bar.
5. In the Connector Entity Browser, review the five red connectors with the IDs: 96, 155, 223, 261, and 262.

   Note: In the Layer column, 2 layers are specified, and in the Link1 and Link2 columns, a link is defined. Because the number of layers and links that were defined match, a possible cause for the connectors not realizing is a small projection tolerance.

6. Review the four connectors with the IDs: 152, 153, 154, and 156.

   Note: In the Layer column, 3 layers are specified. In the Link1 column, one link is defined; in the Link2 and Link3 columns, no links are defined. Because the numbers of layers and links do not match, the likely cause for the connectors not realizing is undefined link definitions.

Realize the Failed Connectors

In this step, you will realize the failed connectors using a larger projection tolerance.

1. In the Connector Entity Browser, right-click and select **Create** > **Spot** from the context menu.
2. Click **location: connectors** > **displayed**.
3. Click **type =** and select **mat100**.
4. In the tolerance = field, enter `2.0`.
5. Click **realize**.

> **Note:** The Status bar reads, "5 connectors realized (4 failed)".

HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds.

Define the Missing Second Link for the Failed Connectors

In this step, you will define the missing second link for the failed connectors.

1. In the Connector Entity Browser, select the four red, unrealized connectors.

> **Tip:** Select multiple connectors by pressing **Ctrl** while left-clicking on **connectors**.

2. Right-click on the selected connectors and select **Add Link** from the context menu.

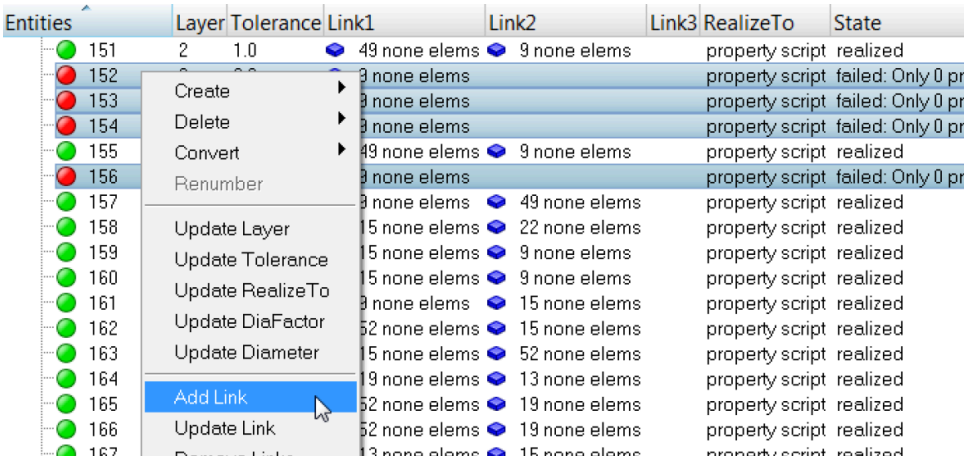


Figure 1075:

3. In the Add Link window, set Link Type to **Components**.

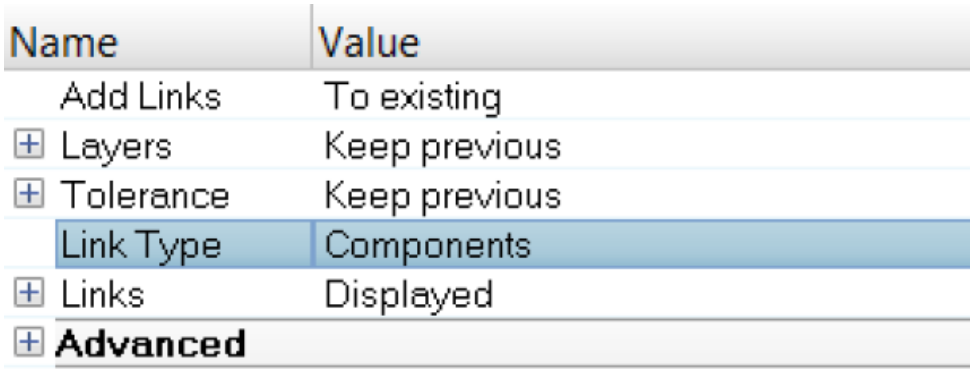


Figure 1076:

4. Set Links to **Select**.
5. Under Links, for Entities, click **0 Component** > **comps**.

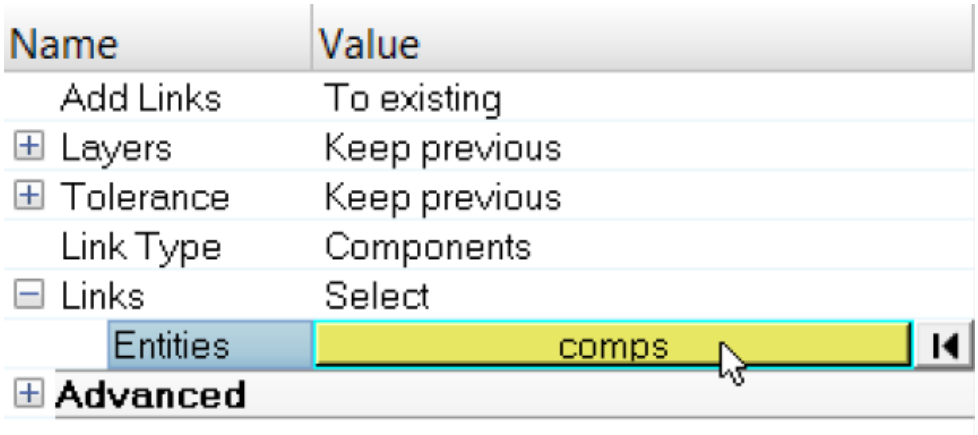


Figure 1077:

6. In the panel area, click **comps**.
7. Select the component, **Comp2**.
8. Click **select**.
9. Click **proceed**.
10. At the top of the Add Links window, click **Add**.
11. In the **confirm** dialog, click **Add Link**.
    HyperMesh adds the specified link entity to the connector.

Define the Missing Third Link for the Failed Connectors

In this step, you will define the missing third link for the failed connectors.

Repeat Define the Missing Second Link for the Failed Connectors, except select **Comp10** in step 7.

Realize the Failed Connectors

In this step, you will realize the failed connectors.

For this step, you should remain in the realize subpanel of the Spot panel.

1. Click **location: connectors** > **displayed**.
2. Click **type =** and select **mat100**.
3. In the tolerance =, enter `4.0`.
4. Click **realize**.
   HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds.
5. On the Visualization toolbar, click  .
6. In the Visualization tab, click  .
7. Under Color by, select **State**.
8. Under State, select the **Realized** checkbox.

   Note: All of the connectors are realized into FE elements.

   HyperMesh displays all of the connectors, which are green.

Save Your Work

In this step, you will save your work in HyperMesh.

This exercise is complete.

Save your work as a HyperMesh file.

5.6 HM-4630: Interfacing for Arbitrary - Lagrangian - Eulerian Capability Using DYNA

In this step, you will become familiar with defining LS-DYNA cards related to the ALE capability. You will set up the model data for an LS-DYNA analysis of an ink cartridge falling through air and onto the ground.

Before you begin, copy `cartridge.hm` from `<hm.zip>/interfaces/lsdyna/` to your working directory.

The ink and ground model is shown in Figure 1078.

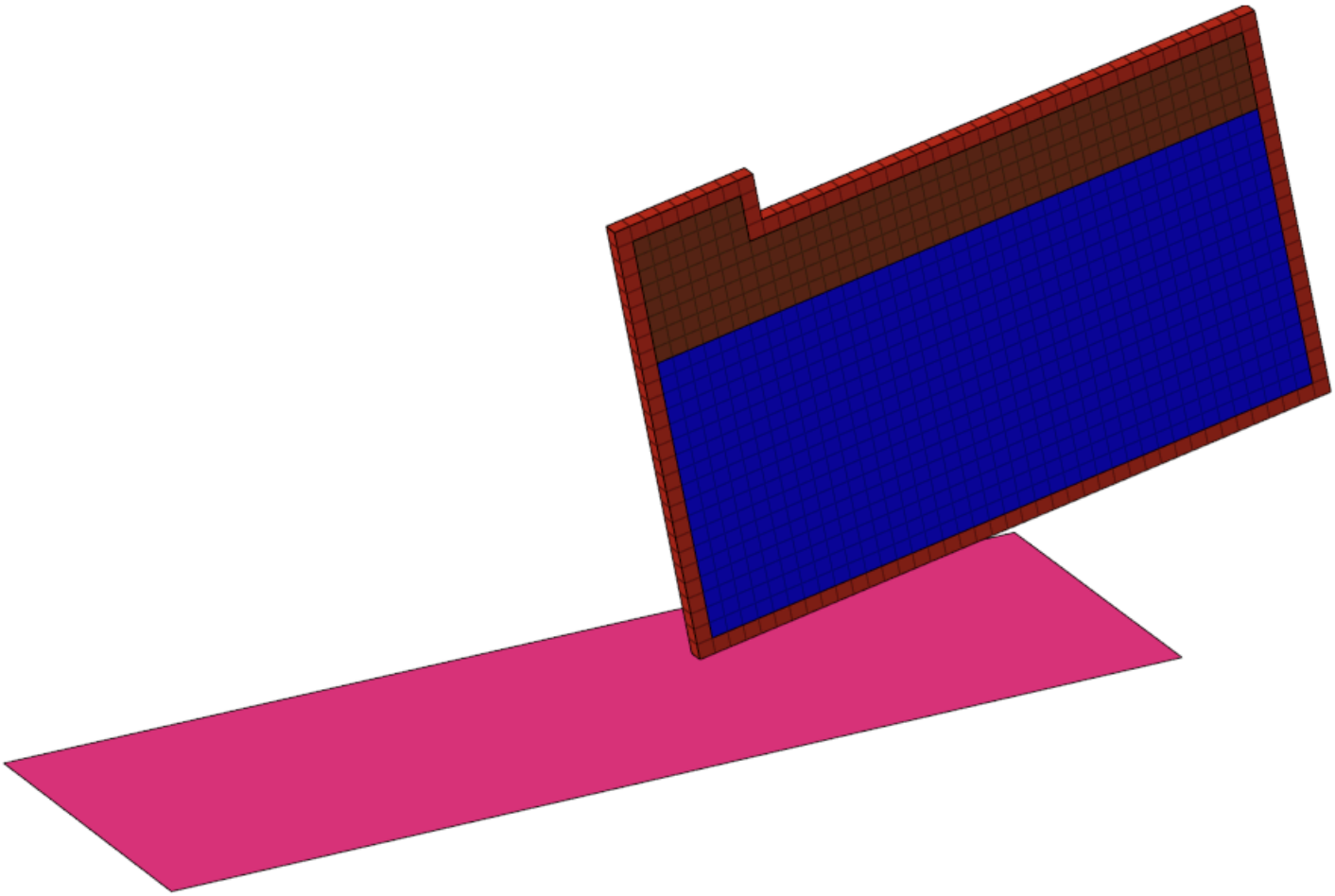


Figure 1078:

5.6.1 Tools Overview

In this section, you will learn about the ALE setup panel, Card Edit feature, and the Control Cards panel.

ALE Setup Panel

The ALE setup panel can be accessed from the Analysis page. Use the ALE setup panel to create and modify input data pertaining to the Arbitrary-Lagrangian Eulerian LS-DYNA capability.

Card Edit Feature

The Card Edit feature can be accessed by clicking on the Collectors toolbar. Use the Card Edit panel to select the entities that are viewed in the card image subpanel. The card images are defined in the template file.

Access the Control Cards feature by clicking **Setup** > **Create** > **Control Cards** from the menu bar or by clicking Control Cards from the Analysis page.

Control Cards Panel

Use the Control Cards panel to set up job-level, solver specific data. The available Control Cards are defined in the template file.

5.6.2 Load the LS-DYNA Profile

In this step, you will load the LS-DYNA profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

5.6.3 Retrieve and View the Model File

In this step, you will retrieve and view the model file in HyperMesh.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
2. In the **Open Model** dialog, open the `cartridge.hm` file.
3. In the Model Browser, expand the Component folder.
   Notice the model is divided into four component collectors: AIR, CARTRIDGE, INK, and PLATFORM.

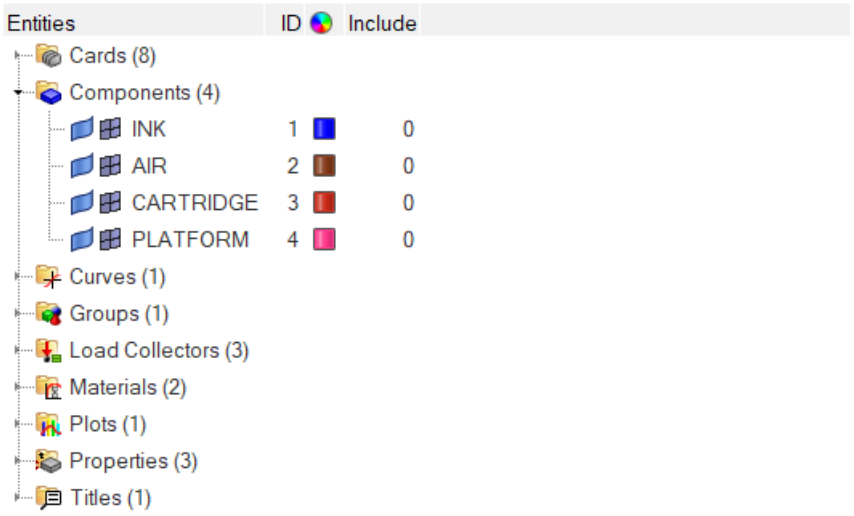


Figure 1079:

4. Use the Model Browser or the Solver Browser to review the defined materials and properties.

5.6.4 Update the *SECTION_SOLID Card

In this step, you will update the *SECTION_SOLID card used by the ALE components INK and AIR to *SECTION_SOLID_ALE using three methods: Solver Browser, Model Browser, and Card Editor.

1. Method 1: Use the Solver Browser
   a) Optional: From the menu bar, click **View** > **Browsers** > **HyperMesh** > **Solver** to open the Solver Browser if it is closed.
   b) Expand the *SECTION folder, expand the *SECTION_SOLID folder, and then click **SOLID PROP FOR ALE**.
   The Entity Editor opens, and displays the property's corresponding data.

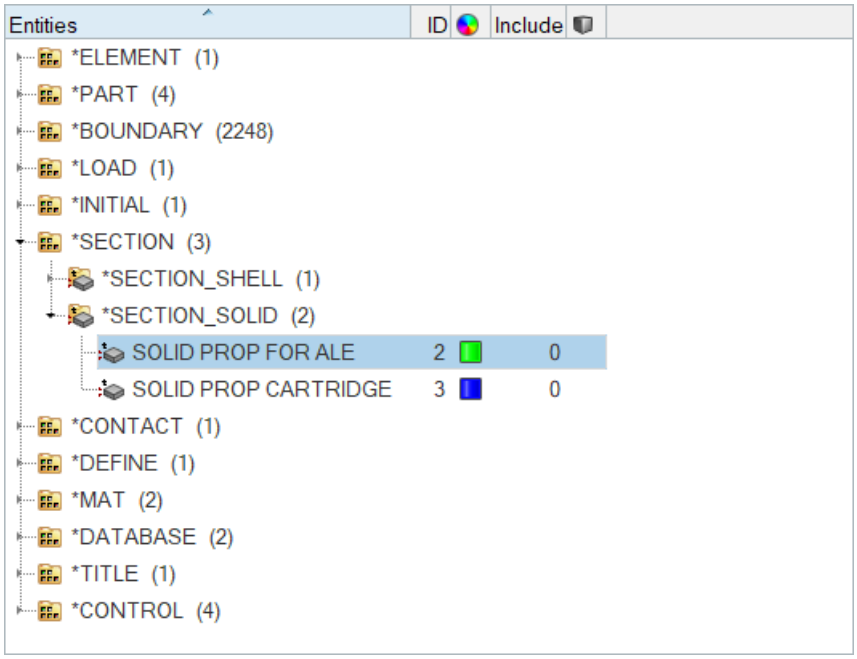


Figure 1080:

c) For ELFORM (Element formulation options), verify that it is set to **12**.

d) Set Options to **ALE**.

2. Method 2: Use the Model Browser

a) In the Model Browser, Property folder, click **SOLID PROP FOR ALE**.
The Entity Editor opens and displays the property's corresponding data.

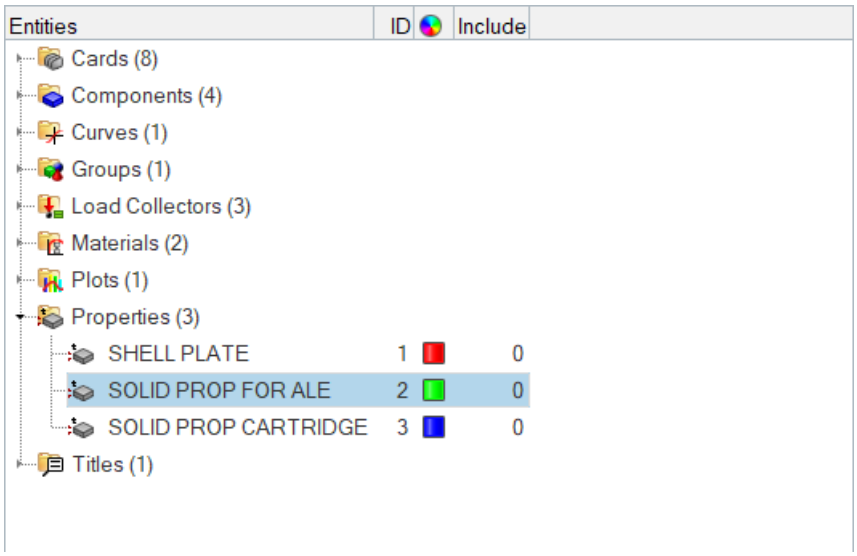


Figure 1081:

b) For ELFORM (Element formulation options), verify that it is set to **12**.

c) For Options, select **ALE**.

3. Method 3: Use the Card Editor
   a) In the Model Browser or Solver Browser, right-click on the property **SOLID PROP FOR ALE** and select **Card Edit** from the context menu.
   The **Card Image** dialog opens.

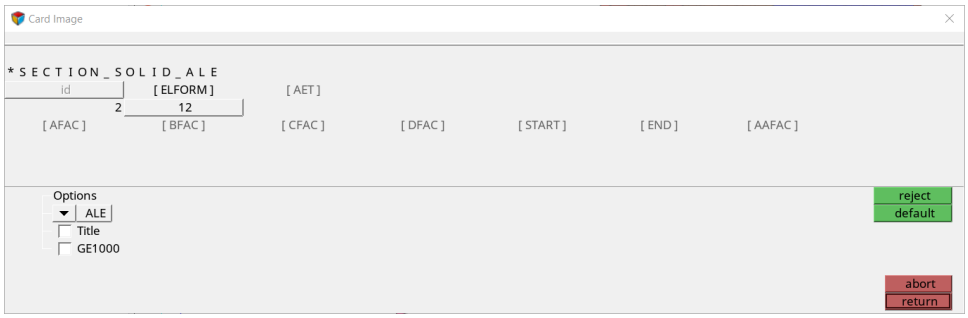


Figure 1082:

   b) For ELFORM (Element formulation options), verify that it is set to **12**.
   c) For Options, select **ALE**.
   d) Click **return**.

5.6.5 Assign *EOS_LINEAR_POLYNOMIAL

In this step, you will assign *EOS_LINEAR_POLYNOMIAL for the ALE components.

1. In the Solver Browser, click ***PART** > ***PART folder** > **INK**.
   The Entity Editor opens, and displays the component's corresponding data.
2. For EOSID, click **Unspecified** > **Property**.

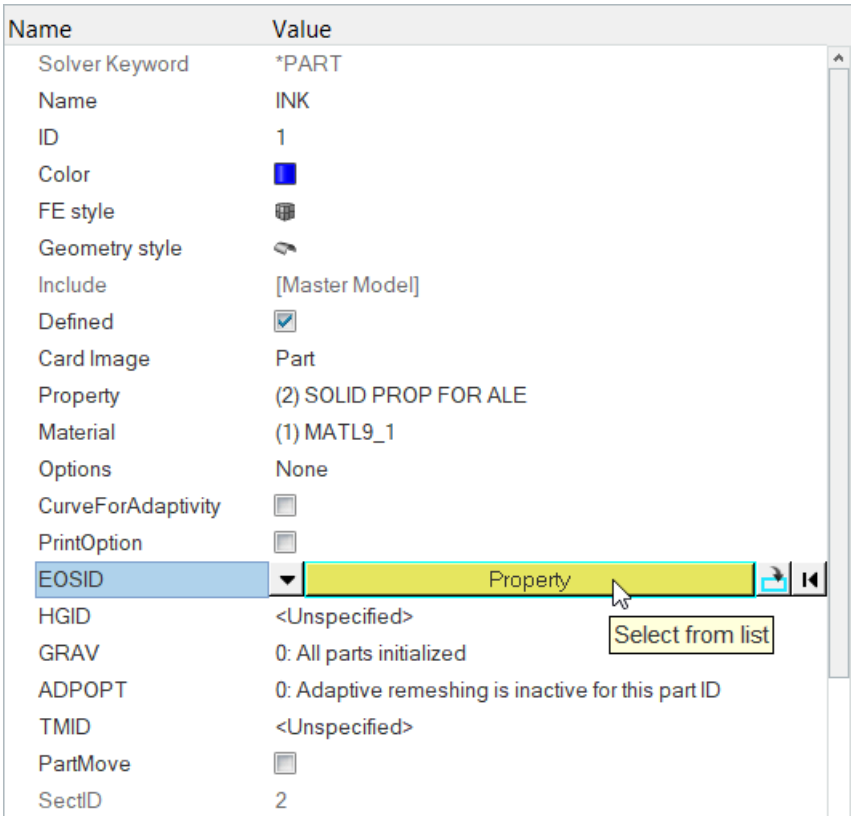


Figure 1083:

3. In the **Select Property** dialog, select **EOS_LINEAR** and then click **OK**.

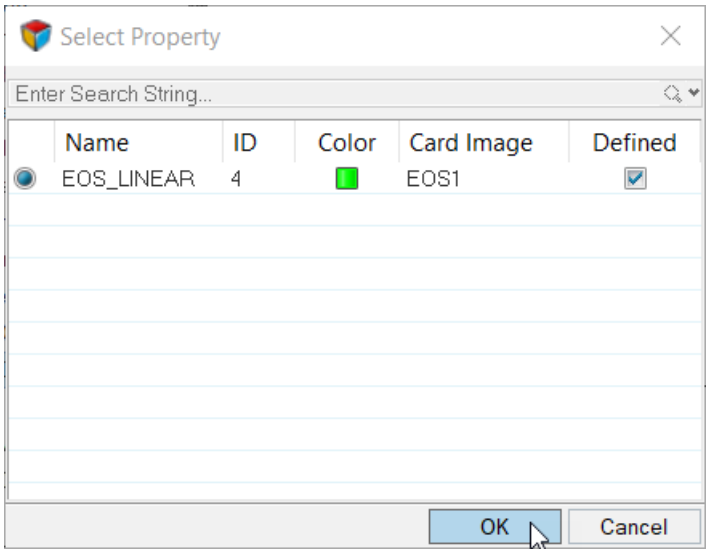


Figure 1084:

4. In the Solver Browser, click ***PART** > ***PART folder** > **AIR**.
5. Repeat steps 2 and 3.

5.6.6 Define *EOS_LINEAR_POLYNOMIAL

For this step, you will define *EOS_LINEAR_POLYNOMIAL for the ALE components.

1. In the Solver Browser, right-click and select **Create** > ***EOS** > ***EOS_LINEAR_POLYNOMIAL** from the context menu.
   A new property opens in the Entity Editor.

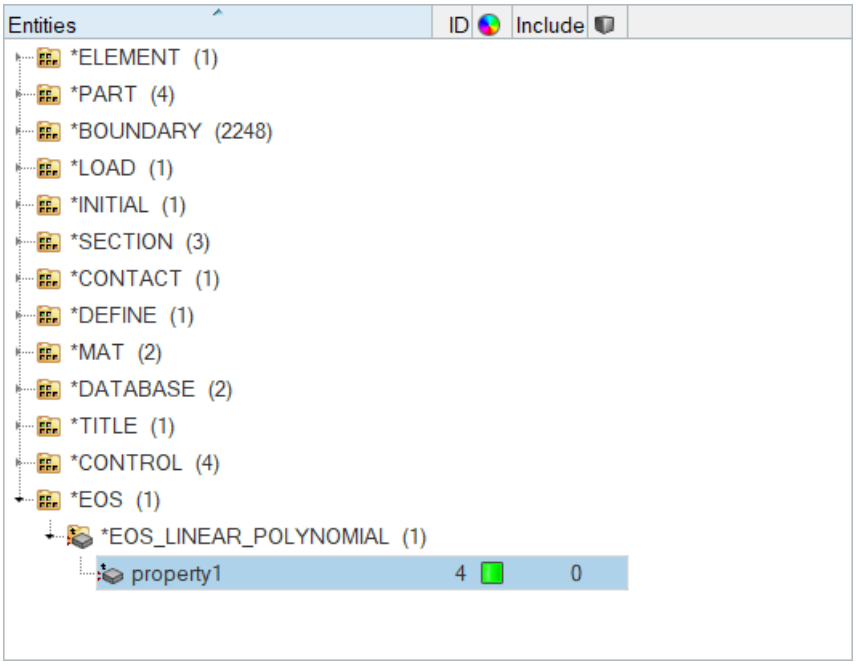


Figure 1085:

2. For Name, enter `EOS_LINEAR`.
3. For c1 (1st polynomial equation coefficient), enter `1.5E+9`.
4. For V0 (Initial relative volume), enter `1`.

5.6.7 Define *INITIAL_VOID_PART

In this step, you will define *INITIAL_VOID_PART to define the AIR component as the void in the fluid interaction.

1. In the Solver Browser, right-click and select **Create** > ***INITIAL** > ***INITIAL_VOID** from the context menu.
   A new group opens in the Entity Editor.
2. For Name, enter `VOID`.
3. For PSID/PID, click **0 Components** > **Components**.
4. In the **Select Components** dialog, select **AIR** and then click **OK**.

5.6.8 Define *ALE_REFERENCE_SYSTEM_NODE

In this step, you will define *ALE_REFERENCE_SYSTEM_NODE to control the motion of the ALE mesh.

1. In the Solver Browser, select **Create** > **ALE** > ***ALE_REFERENCE_SYSTEM_NODE** from the context menu.
   A new *ALE_REFERENCE_SYSTEM_NODE opens in the Entity Editor.
2. For Name, enter `ALE_NODE`.
3. For NID1, click **Unspecified** > **Node**.
4. In the panel area, click **node**.
5. In the id= field, enter `5` and then press **Enter**.
6. Click **proceed**.
7. For NID2, click **Unspecified** > **Node**.
8. Repeat step 4.
9. In the id= field, enter `6` and then press **Enter**.
10. Repeat step 6.
11. For NID3, click **Unspecified** > **Node**.
12. Repeat step 4.
13. In the id= field, enter `7` and then press **Enter**.
14. Repeat step 6.

5.6.9 Define *ALE_REFERENCE_SYSTEM_GROUP

In this step, you will define *ALE_REFERENCE_SYSTEM_GROUP to specify the reference system type for the fluid interaction of the ALE components.

1. In the Solver Browser, select **Create** > ***ALE** > ***ALE_REFERENCE_SYSTEM_GROUP** from the context menu.
   A new *ALE_REFERENCE_SYSTEM_GROUP opens in the Entity Editor.
2. For Name, enter `SYSTEM_GROUP`.
3. Click **SID**.
4. Set the entity selector to **Sets**.
5. Assign a set.
   a) Right-click on **SID** and select **Create** from the context menu.
   b) For part Name, enter `Set_ALE_Components`.
   c) For card image, select **Part**.
   d) For Entity IDs, click **0 Components** > **Components**.
   e) In the **Select Components** dialog, select **INK** and **AIR**.
   f) Click **OK**.
   g) Click **Close**.
6. Set PRTYPE to **5**.
7. Click **PRID** > **Alereferencesystemnode**.

8. In the **Select Alereferencesystemnode** dialog, select **ALE_NODE** and then click **OK**.

5.6.10 Review the *CONTROL_ALE Card

In this step, you will review the pre-defined *CONTROL_ALE card.

1. In the Solver Browser, ***CONTROL** > ***CONTROL_ALE folder**, click ***ALE1**.
   The Entity Editor opens, and displays the control card's corresponding data.
2. Review the *CONTROL_ALE parameters.

5.6.11 Export the Model

In this step, you will export the model to an LS-DYNA 971 formatted input file.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
2. In the Import - Solver Deck tab, set Template to **Keyword971**.
3. In the File field, navigate to your working directory and save the file as `cartridge_complete.key`.
4. Click **Export**.

5.6.12 Submit the LS-DYNA Input File

In this step, you will submit the LS-DYNA input file to LS-DYNA 971.

1. From the desktop's Start Menu, open the LS-DYNA Manager program.
2. From the solvers menu, select **Start LS-DYNA analysis**.
3. Load the `cartridge_complete.key` file.
4. Start the analysis by clicking **OK**.

5.6.13 View the Results in HyperView

The results for ALE data are stored as Extra Solid History Variable in HyperView.

5.6.14 Save Your Work

In this step, you will save your work as a HyperMesh file.

The exercise is complete.

Save your work as a HyperMesh file.

5.7 HM-4635: Create Joints and Check Minimum Time Step

In this tutorial, you will create joints and check the minimum time step in HyperMesh.

Before you begin, copy `joints.hm` and `pene_dyna.hm` from `<hm.zip>/interfaces/lsdyna/` to your working directory.

Also, it is recommended that you complete HM-1000: Getting Started with HyperMesh.

To fix and check penetrations, refer to the Penetration - HM-3320 tutorial.

5.7.1 Exercise 1: Create Joints

In this exercise, you will create a spherical joint and a revolute joint.

You can create joint definitions in the FE Joints panel, which can be accessed from the 1D page. HyperMesh supports the following standard joint types: spherical, revolute, cylindrical, planar, universal, translational, and locking. HyperMesh also supports LS-DYNA's *CONSTRAINED_JOINT_STIFFNESS_OPTION property to define friction, damping, stop angles, and so on. The LS-DYNA solver interface supports the creation of joints in the FE Joints panel. The PAM-CRASH solver interface currently supports the creation of Joints as rod elements (see HM-4700: Using the PAM-CRASH Interface in HyperMesh).

A spherical joint consists of two coincident nodes. During analysis, the two coincident nodes are forced to remain coincident, but the bodies attached to each coincident node are allowed to rotate freely about the joint location.

Load the LS-DYNA Profile

In this step, you will load the LS-DYNA profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.
3. Select the **Keyword971_R10.1** template from the drop down list.
4. Select **OK** to close the **User Profile** dialog.

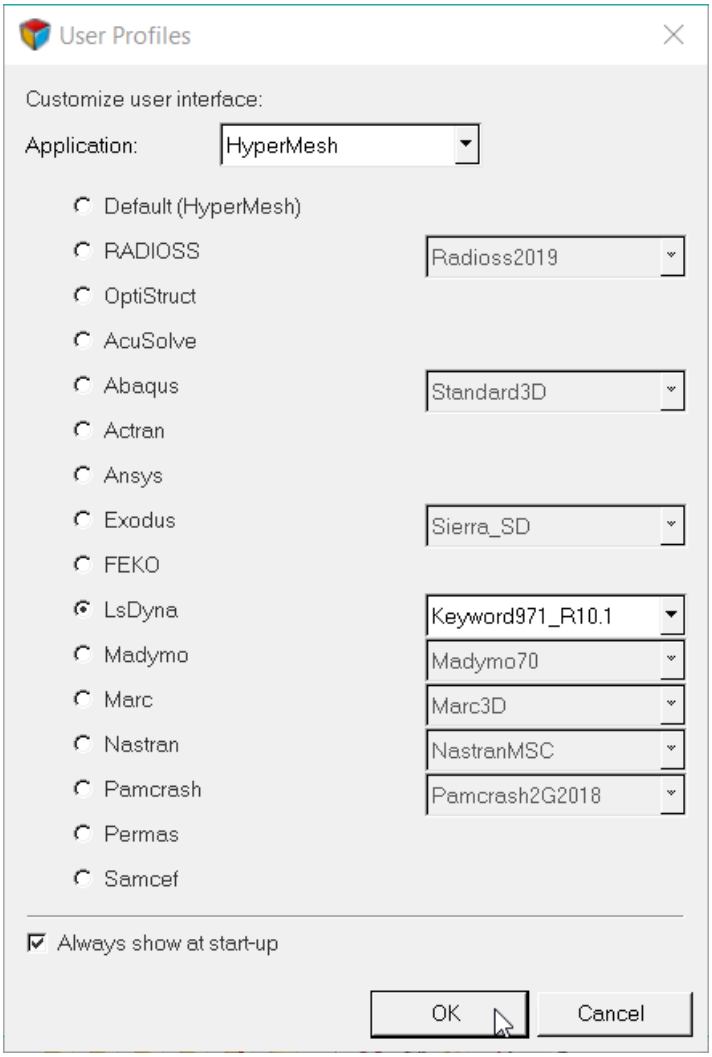


Figure 1086:

Retrieve and View the Model File

In this step, you will open the model file and view it in HyperMesh.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
2. In the **Open Model** dialog, open the `joints.hm` file.

The model appears in the graphics area.

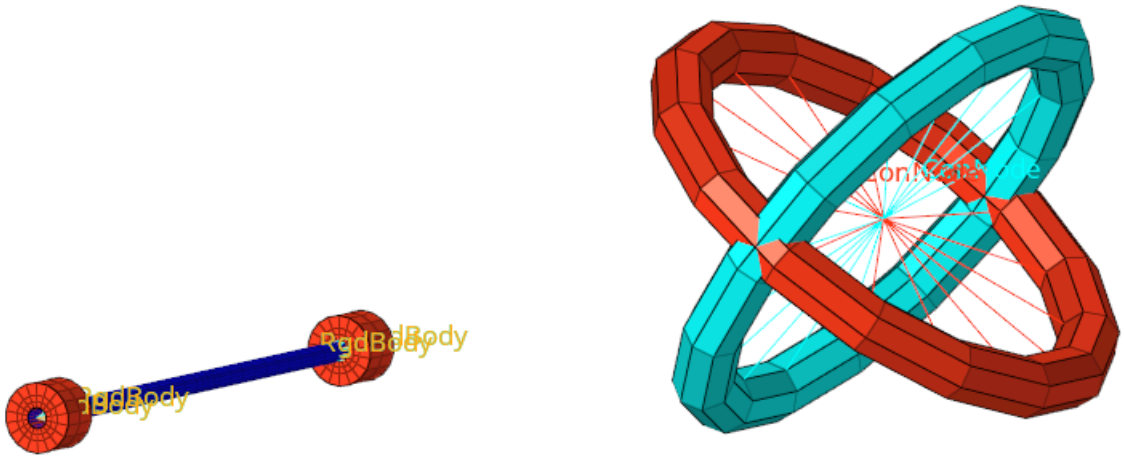


Figure 1087:

Activate Coincident Node Picking

In this step, you will activate coincident picking within the Graphics panel.

According to the LS-DYNA specification, a joint needs to reference a pair of coincident nodes. The creation of this element requires the selection of coincident nodes. To select coincident nodes, activate the coincident node picking option in HyperMesh, and then select a coincident node. HyperMesh will present you with a selection circle identifying the coincident node ID.

1. Open the Graphics panel by clicking **Preferences** > **Graphics** from the menu bar.
2. Select the **coincident picking** checkbox.
3. Click **return**.

Change the Display

In this step, you will change the display of the model in the graphics window by disabling components in the Model Browser.

1. In the Model Browser, expand the Components folder to display the contents.
2. Click next to each component except blue torus, orange torus, and New Joint to disable the display of those elements.

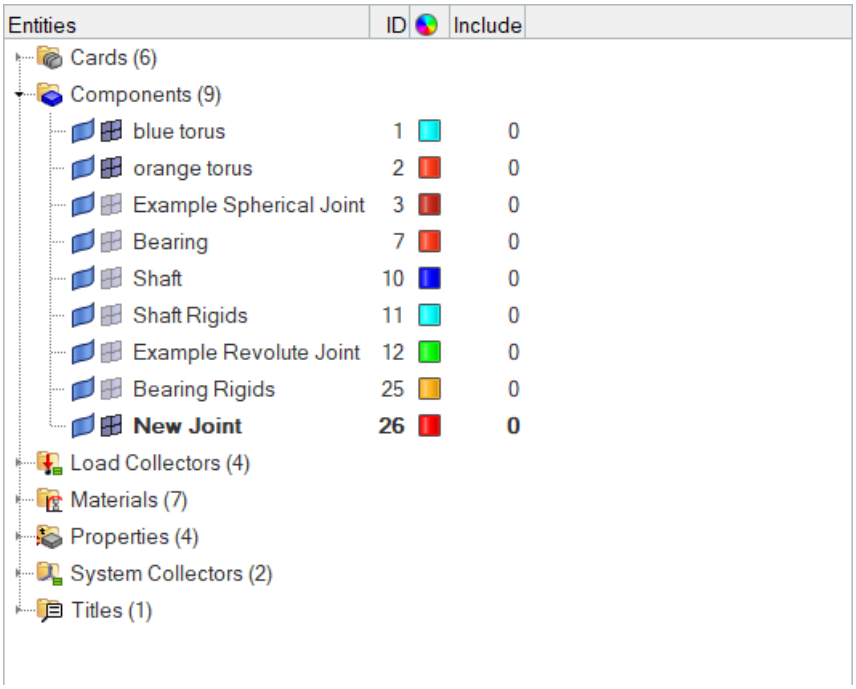


Figure 1088:

The blue torus, orange torus, and New Joint are the only components displayed in the graphics window as seen in Figure 1089.

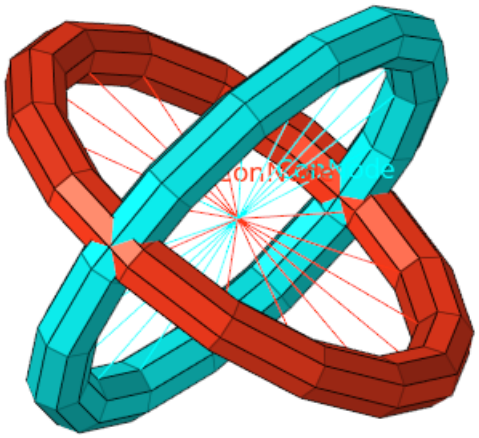


Figure 1089:

Create a Spherical Joint

In this step, you will create a spherical joint using the Joints panel.

1. Open the Joints panel by completing one of the following options:

- Click **Mesh** > **Create** > **1D Elements** > **Joints** from the menu bar.
- In the Solver Browser, right-click and select **Create** > ***CONSTRAINED** > ***CONSTRAINED_JOINT_SPHERICAL** > ***CONSTRAINED_JOINT_SPHERICAL** from the context menu.

2. Set the joint type to **spherical**.
3. Use the node 1 selector to select a node in the center of both tori as indicated in Figure 1090.
   The coincident node picking mechanism appears and displays two nodes, node 598 and node 1.

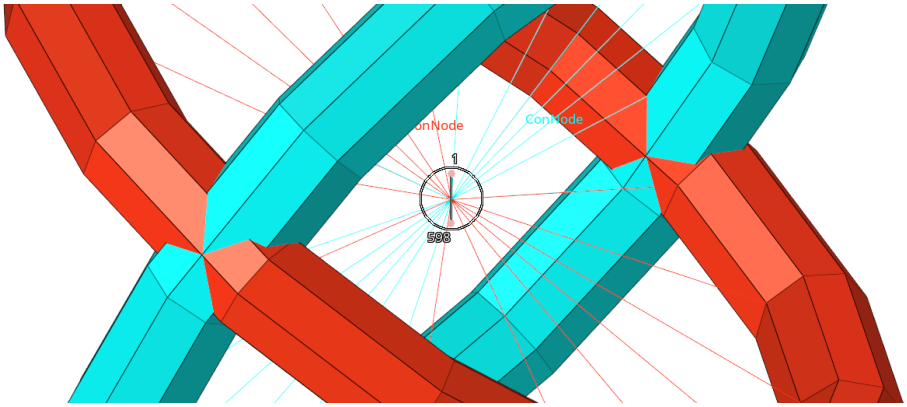


Figure 1090:

4. Select the node labeled **598**.

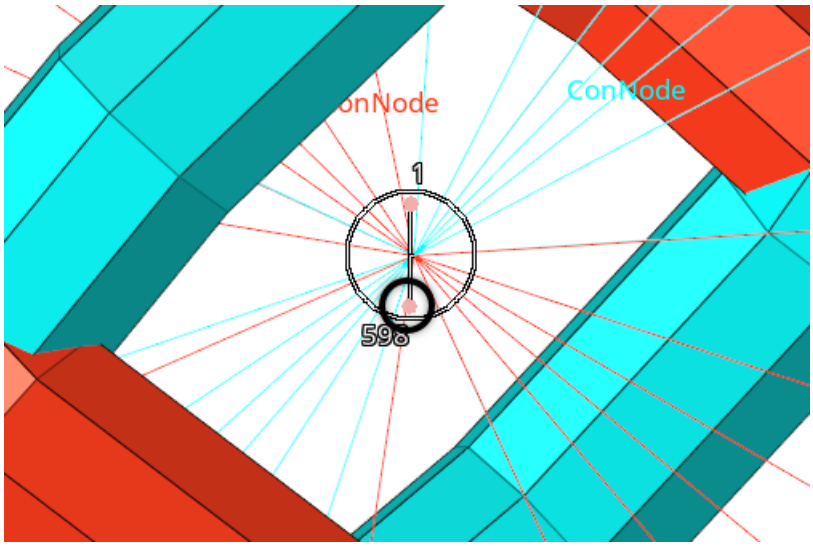


Figure 1091:

5. Using the node 2 slector, select the node created in step 4.
   The coincident node picking mechanism appears and displays two nodes, node 598 and node 1.
6. Select the node labeled **1**.

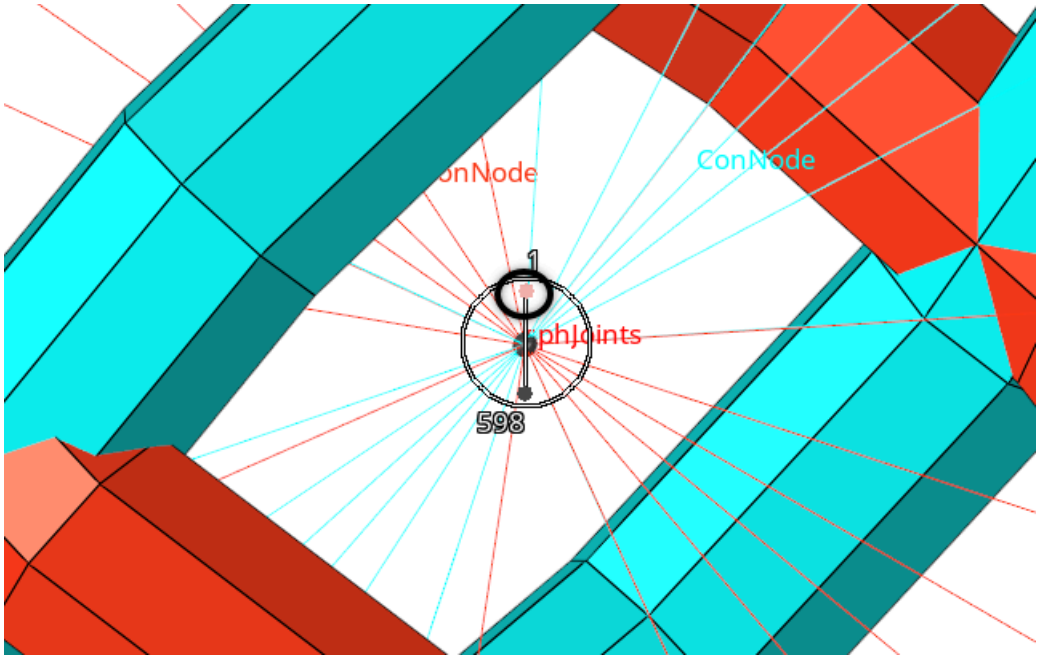


Figure 1092:

7. From the the Joints panel, click **create**.
HyperMesh generates the spherical joint element seen in Figure 1093.

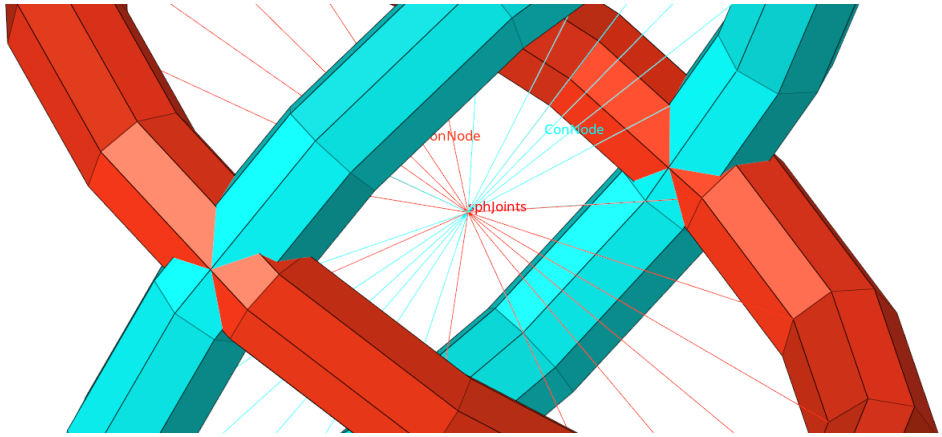


Figure 1093:

8. Click **return** to exit the panel.

Change the Display

In this step, you will change the display of the model using the component folder of the Model Browser.

In the Model Browser, Component folder, turn off the display of all of the components, except Bearing, Shaft, Bearing Rigids, Shaft Rigids, and New Joint.

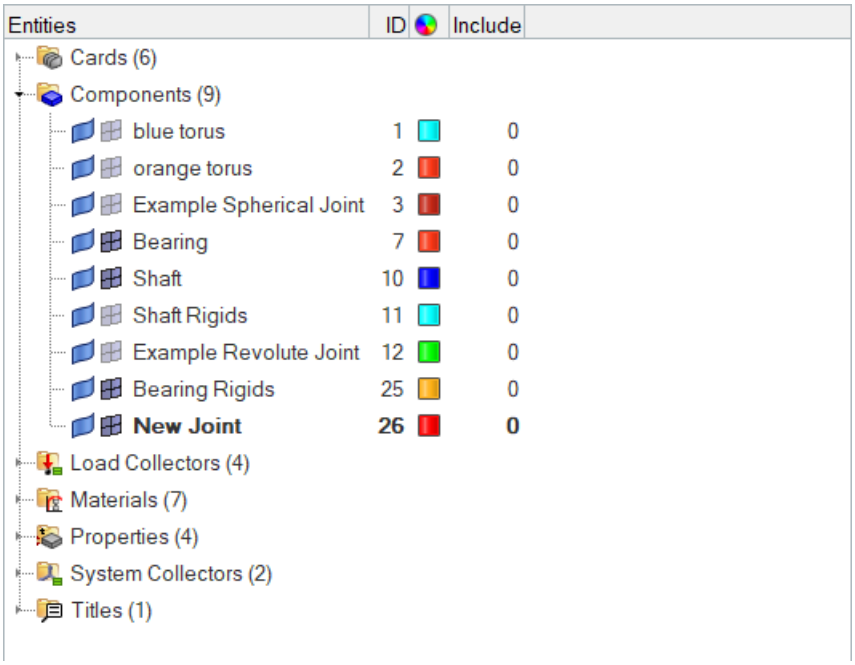


Figure 1094:

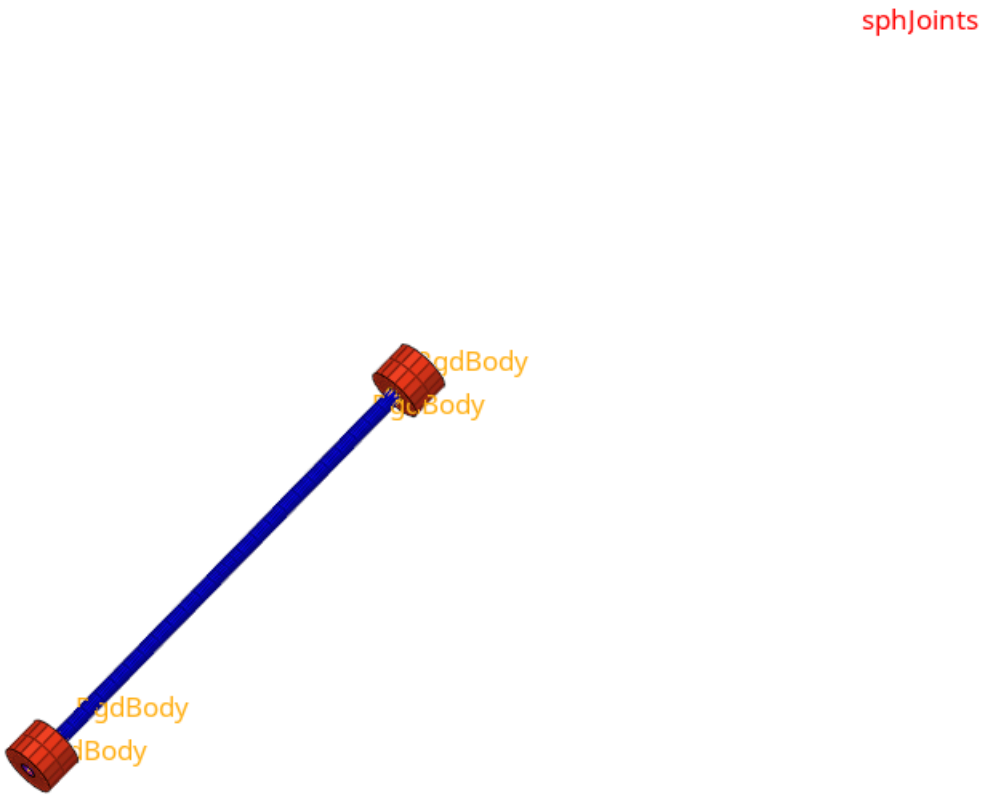


Figure 1095:

The Bearing, Shaft, Bearing Rigids, Shaft Rigids, and New Joint are the only components displayed in the graphics window as seen in Figure 1095.

Create a Revolute Joint

In this step, you will create a revolute joint using the Joints panel.

A revolute joint consists of four nodes, two sets of two coincident nodes. During analysis, all four of the revolute joint's nodes remain at the same location with respect to each other. The bodies attached to the nodes are free to rotate about the axis that lies along the length of the revolute joint.

1. Open the Joints panel by completing one of the following options:
   - Click **Mesh** > **Create** > **1D Elements** > **Joints** from the menu bar.
   - In the Solver Browser, right-click and select **Create** > ***CONSTRAINED** > ***CONSTRAINED_JOINT_REVOLUTE** > ***CONSTRAINED_JOINT_REVOLUTE** from the context menu.
2. Zoom in on one end of the shaft assembly as indicated in Figure 1096.

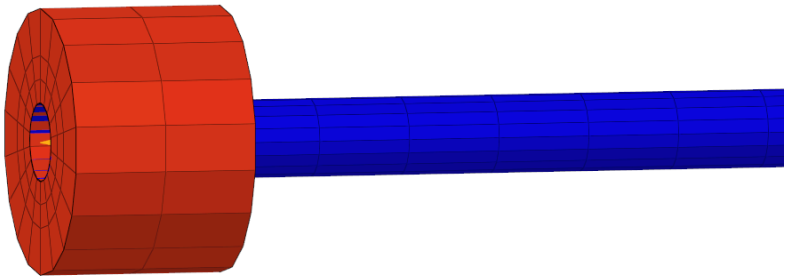


Figure 1096:

3. Set the joint type to **revolute**.
4. Using the node 1 selector, select a node at the center of one of the rigid link elements.
   The coincident node picking mechanism appears.

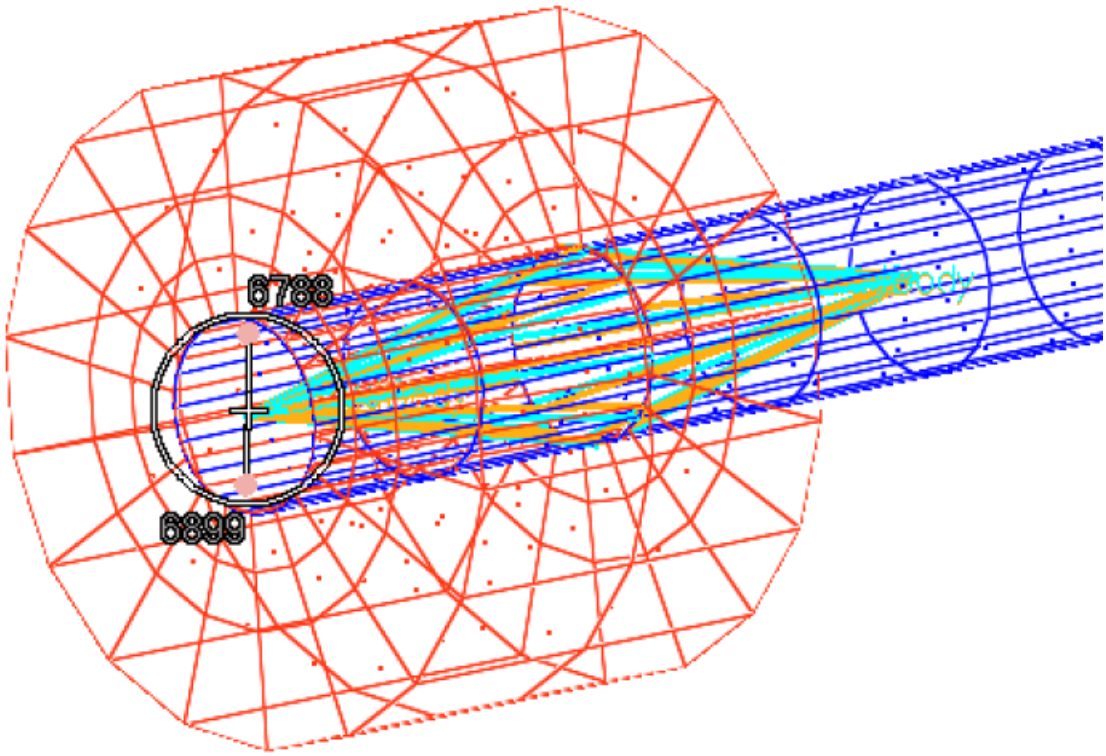


Figure 1097:

5. Select node **6788**, which is attached to the blue rigid link element.

> **Tip:** Drag your cursor over a node in the coincident node picking mechanism to highlight the attached element.

6. Using the node 2 selector, select the node that was created in step 5.
7. Select node **6899**, which is attached to the orange rigid link element.
8. Using the node 3 selector, select a node at the center of the opposing pair of blue and orange rigid link elements.
   The coincident node picking mechanism appears.

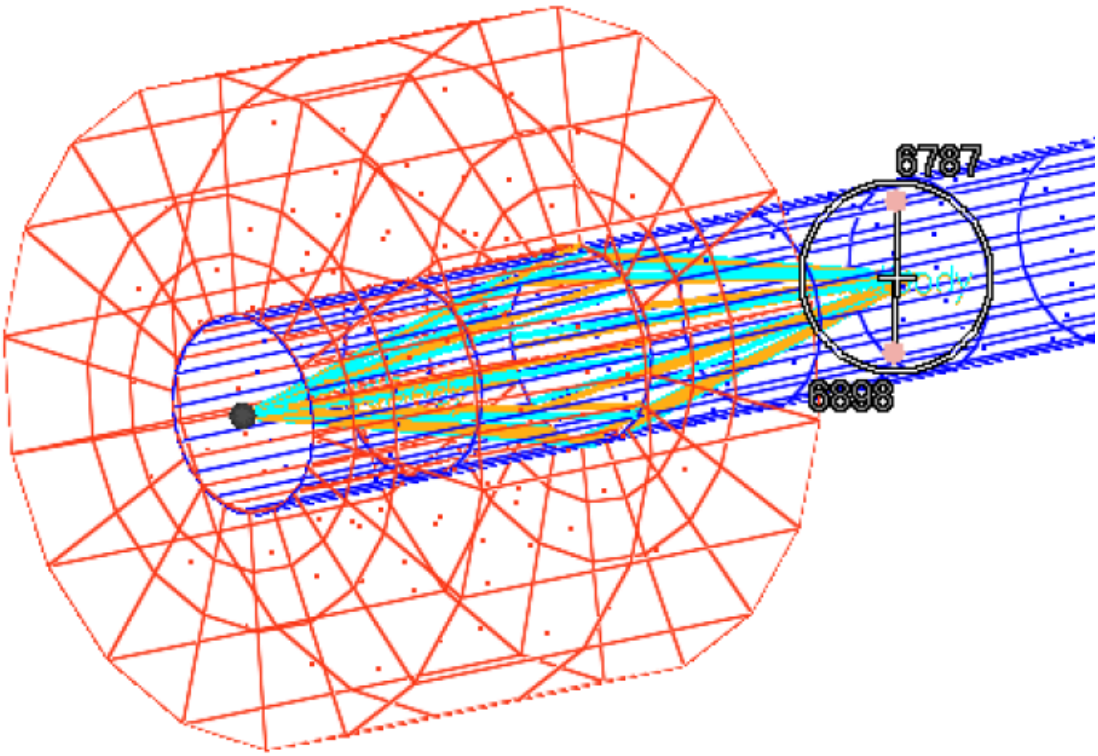


Figure 1098:

9. Select node **6787**, which is attached to the blue rigid link element.

10. Using the node 4 selector, select the node that was created in step 9.

11. Select node **6898**, which is attached to the orange rigid link element.

12. Click **create**.

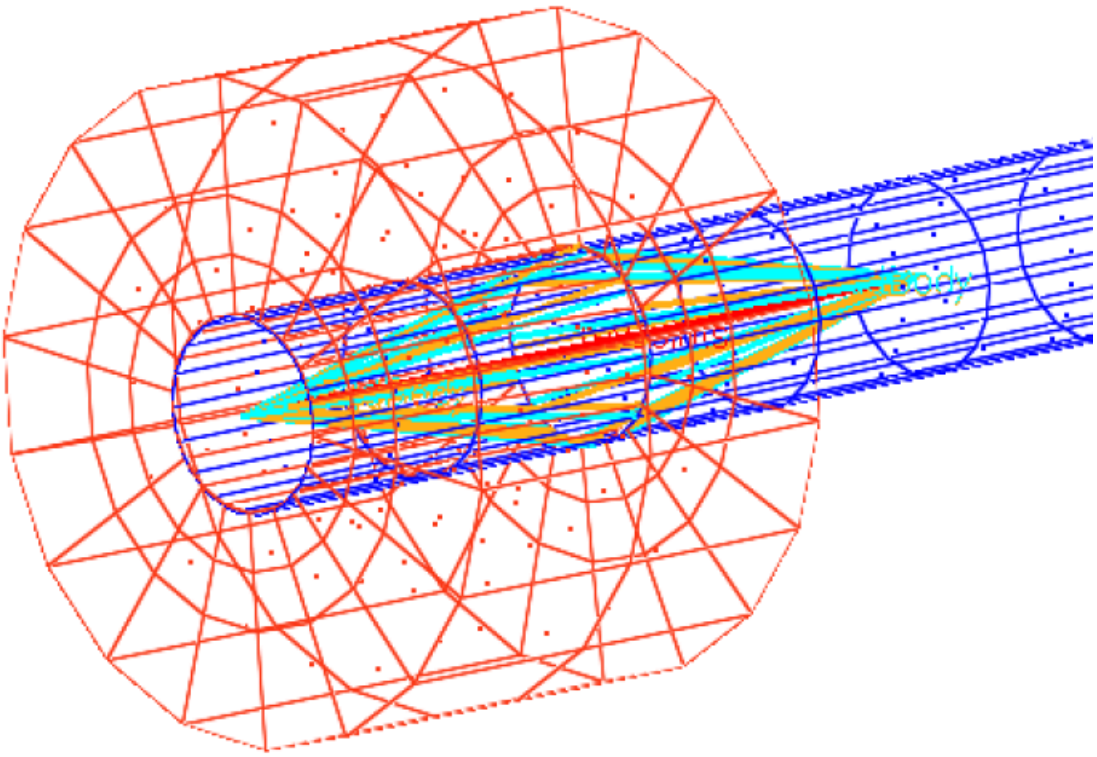


Figure 1099:

13. Click **return**.

5.7.2 Exercise 2: Check the Minimum Time Step

In this exercise, you will check the minimum time step.

The Time subpanel in the check elems panel calculates element time steps, based on the FEA solver, and allows you to check for time steps that fall below a specified value. In explicit codes such as LS-DYNA, it is sufficient that a single element would have a small time step to drastically reduce the total CPU time of the entire job. For this reason the check is used to identify those elements.

Load the LS-DYNA Profile

In this step, you will load the LS-DYNA profile in HyperMesh.

Only complete this step if you did not complete Load the LS-DYNA Profile in Exercise 1: Create Joints.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.
3. Select the **Keyword971_R10.1** template from the drop down list.
4. Select **OK** to close the **User Profile** dialog.

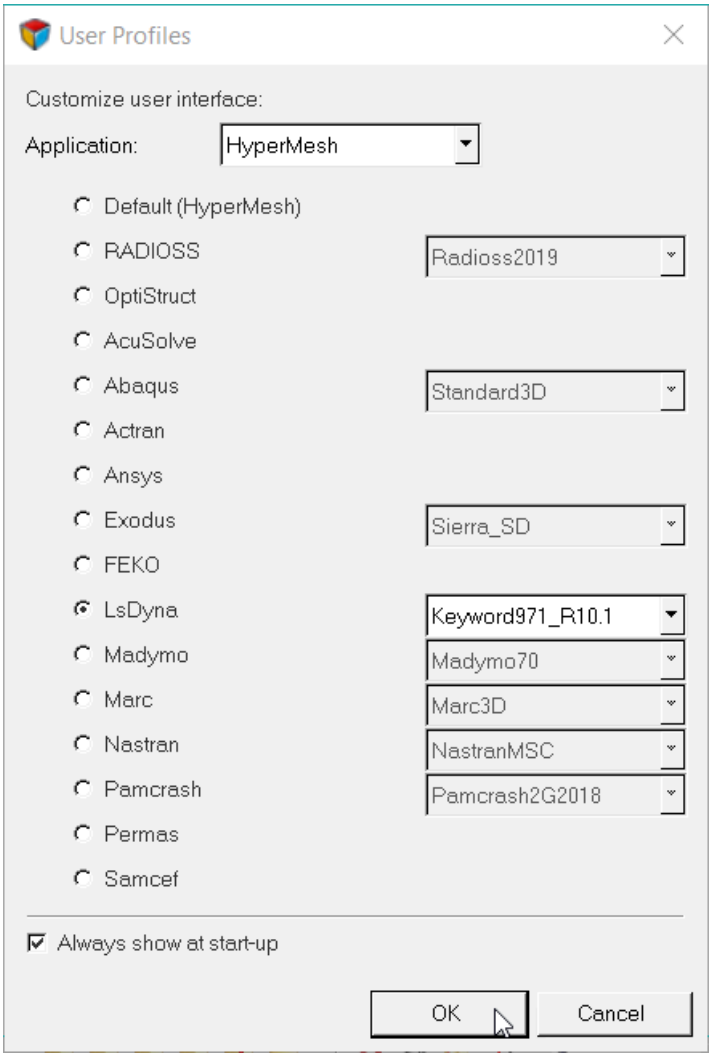


Figure 1100:

Retrieve and View the Model File

In this step, you will open the model file and view it in HyperMesh.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click on the Standard toolbar.
2. In the **Open Model** dialog, open the `pene_dyna.hm` file.

The model appears in the graphics area.

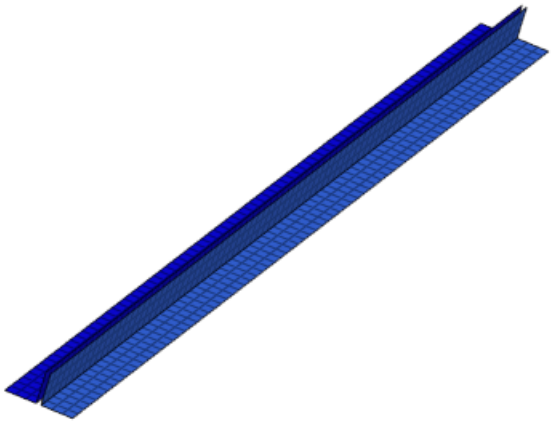


Figure 1101:

Specify the Template

In this step, you will specify the template in HyperMesh.

The time step associated to an element is related to its geometric characteristic and to its material properties such as density and young modulus. For this reason, a material and a valid template need to be associated to the elements. The time step is dependent upon the user profile, so you may want to load a different profile to see how results may change.

1. Load a solver template by clicking **File** > **Load** > **Solver Template** from the menu bar.
2. In the **Open** dialog, navigate to the LS-DYNA folder and open the `dyna.key` file.

Check the Time Steps

In this step, you will check the time steps.

1. Open the Check Elements panel by clicking **Mesh** > **Check** > **Elements** > **Check Elements** from the menu bar.
2. Open the **time** subpanel.
3. Click **check elems**.
   The total number of elements failing the check displays in the status bar as seen in Figure 1102.

   0 elements failed check. 0.561408 min time step. (Incomplete material for 90 elements)

   Figure 1102:

4. Click **return**.

MADYMO

6

MADYMO tutorials.

This chapter covers the following:

- 6.1 HM-4900: Create a MADYMO Model (p. 857)
- 6.2 HM-4910: Set up a MADYMO Occupant Safety Analysis in HyperMesh (p. 875)

6.1 HM-4900: Create a MADYMO Model

In this tutorial, you will create the MADYMO leg model and define one of the body local coordinate systems.

Before you begin, copy the `leg_geom.hm` from `<hm.zip>/interfaces/madymo/` to your working directory.

6.1.1 Load the MADYMO User Profile

In this step, you will load the MADYMO profile in HyperMesh.

The MADYMO user profile configures the user interface to be MADYMO specific. The user profile also loads the XML import reader and MADYMO template file. The template file defines MADYMO specific entity types and attributes versus generic entities.

1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to **Madymo**.
3. Set the template drop-down to **Madymo 70**.
4. Click **OK**.

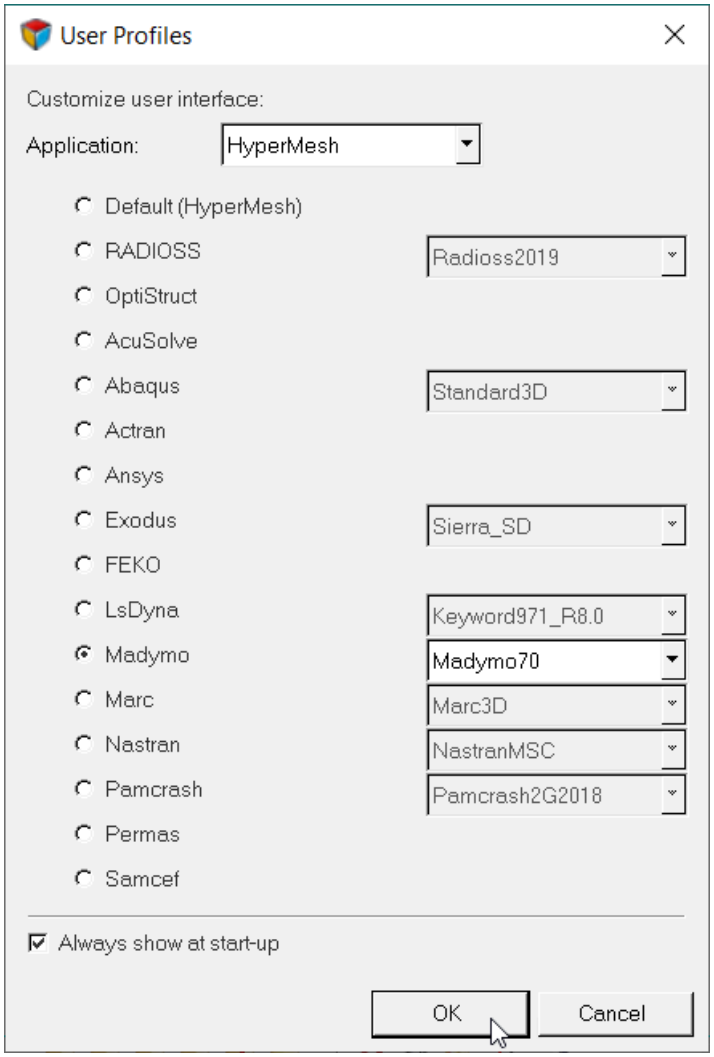


Figure 1103:

5. Open the Utility menu by clicking **View** > **Browsers** > **HyperMesh** > **Utility** from the menu bar.

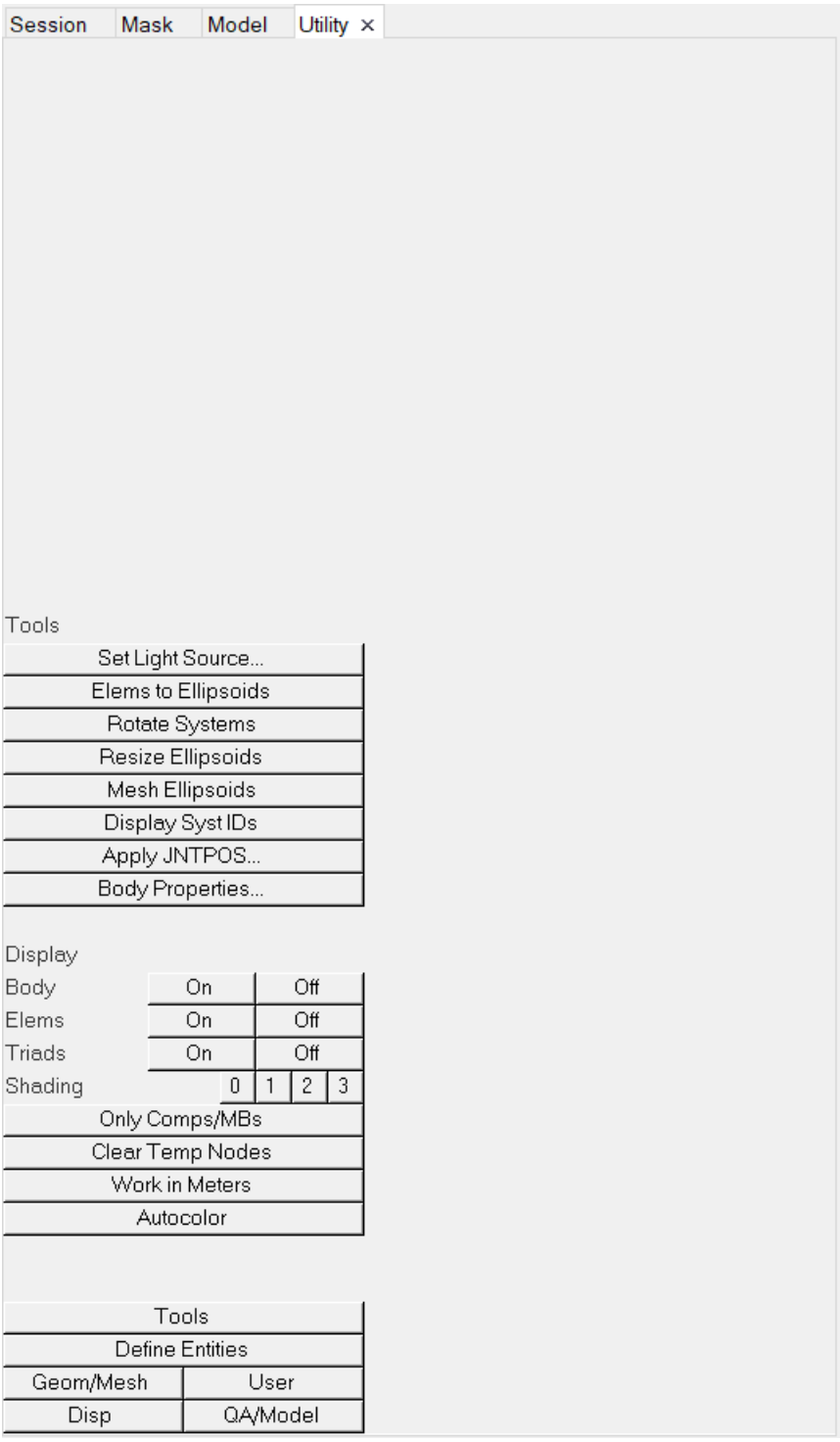

Session
Mask
Model
Utility
Tools
Set Light Source...
Elems to Ellipsoids
Rotate Systems
Resize Ellipsoids
Mesh Ellipsoids
Display Syst IDs
Apply JNTPOS...
Body Properties...
Display
Body
On
Off
Elems
On
Off
Triads
On
Off
Shading
0
1
2
3
Only Comps/MBs
Clear Temp Nodes
Work in Meters
Autocolor
Tools
Define Entities
Geom/Mesh
User
Disp
QA/Model

Figure 1104:

6.1.2 Retrieve and View the Model File

In this step, you will open the model file and view it in HyperMesh.

The `leg_geom.hm` file contains IGES lines representing the layout of the leg and coordinate systems for bodies and joints.

1. Open the model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
2. In the **Open Model** dialog, open the `leg_geom.hm` file.

The model appears in the graphics area.

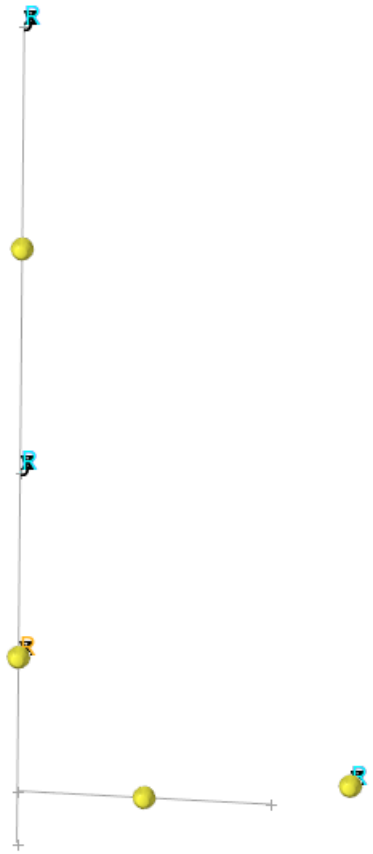


Figure 1105:

6.1.3 Create Coordinate Systems

In this step, you will create coordinate systems.

The `leg_geom.hm` file contains all the coordinate systems for defining the leg model, except the coordinate system for the femur body.

1. From the menu bar, click **Geometry** > **Create** > **Systems** > **Axis Direction**.
2. In the panel area, use the origin selector to select the center node of the femur area.
3. Using the x-axis selector, select a node along the length of the leg.
4. Using the xy-plane selector, select the node along the foot.
5. Click **create**.
6. Click **return** to exit the Systems panel.

6.1.4 Create the Femur Body

In this step, you will create the femur body.

1. In the Model Browser, right-click and select **Create** > **Multibody** from the context menu.
2. In the **Create multibody** dialog, Name field, enter `femur`.
3. Set card image to **RIGID**.
4. Using the Center of gravity: Node selector, select the highest yellow temp node (at the center of the femur).
5. Set Creation method to **Body system**.
6. Click **Create**.
7. In the **Card Image** dialog, enter `6.0` in the MASS field and `0.1` in the moments of inertia field.
8. Click **return** to save changes to the body's card image and close the Card Editor.
9. Remain in the Multibody: Create panel.

6.1.5 Create the Tibia Body

In this step, you will create the tibia body.

You will complete these steps from within the Multibody: Create panel from Create the Femur Body.

1. Enter `tibia` in the name= field.
2. Click **card image =** and select **RIGID** from the pop-up list.
3. Double-click **N1** to enter the Node Vector Edit sub-panel.
4. Enter the following:
   a) For x, enter `0.0`.
   b) For y, enter `0.0`.
   c) For z, enter `-0.85`.
5. Click **return**.
6. Click the yellow **system selection** box under use this system: to make it active.
7. Select the orange coordinate system at the center of the tibia.
8. Click **create/edit** to create the body and bring it up in the Card Editor.
9. Set MASS to **5.0** and the moments of inertia to **0.1**.
10. Click **return** to save the changes to the body's card image and close the Card Editor.

11. Remain in the Multibody: Create panel.

6.1.6 Create the Ball Body

In this step, you will create the ball body.

You will complete these steps from within the Multibody: Create panel from Create the Tibia Body.

1. Enter `ball` in the multibody name = field.
2. Click **card image =** and select **RIGID** from the pop-up list.
3. Click once on **N1** to make it the active selector.
4. Select the yellow temp node at the center of the ball.
5. Click the yellow **system selection** box under use this system: to make it active.
6. Click the **system** box again.
   a) In the Id= field that pops up, enter `7` to select the orange coordinate system at the ball center.
7. Click **create/edit** to create the body and bring it up in the Card Editor.
8. Set MASS to **0.1** and the moments of inertia to **0.01**.
9. Click **return** to save changes to the body's card image and close the Card Editor.
10. Remain in the Multibody: Create panel.

6.1.7 Create the Ground Body

In this step, you will create the ground body.

You will complete these steps from within the Multibody: Create panel from Create the Ball Body.

1. Enter `ground` in the multibody name= field.
2. Click **card image =** and select **RIGID** from the pop-up list.
3. Double-click **N1** to enter the Node Vector Edit subpanel.
4. Enter the following:
   a) For x, enter `0.0`.
   b) For y, enter `0.0`.
   c) For z, enter `-1.1`.
5. Click **return**.
6. Click the switch to change the value to use body’s local system.
7. Click **create** to create the body.
8. Click **return** to leave the Multibody: Create panel.

6.1.8 Create Geometry

In this step, you will create geometry.

1. Create the ground plane.

   SURFACE.PLANE is defined in the Planes panel.

   a) Click the **Analysis** page and within the safety subpanel, click **planes**.
   b) Enter `ground` in the name = field.
   c) Click **color** and select a color for the plane.
   d) Double-click **N1** to enter the Node Vector Edit subpanel.
   e) Under N1, enter the following:
      - For x, enter `-0.25`.
      - For y, enter `0.25`.
      - For z, enter `-1.1`.

      Tip: Use the **Tab** key to quickly move through the fields.

   f) Under N2, enter the following:
      - For x, enter `-0.25`.
      - For y, enter `-0.25`.
      - For z, enter `-1.1`.
   g) Under N3, enter the following:
      - For x, enter `0.50`.
      - For y, enter `-0.25`.
      - For z, enter `-1.1`.
   h) Click **return**.
   i) Click **multibody=** and select **ground** from the pop-up list.
   j) Click **create** to create the plane.
   k) Click **return** to leave the panel.

2. Create the femur ellipsoid.

   SURFACE.ELLIPSOID and SURFACE.CYLINDER are created in the Ellipsoids panel.

   a) While still in the safety module, click on **ellipsoids**.
   b) Enter `femur` in the base name = field.
   c) Click **color** and select a color for the femur.
   d) Click the triangle next to center: and select entities from the pop-up list.
   e) With the node selector active, select the node at the center of the femur.
   f) Enter the following:
      - For A, enter `0.3`.
      - For B, enter `0.1`.
      - For C, enter `0.1`.

g) Set orientation: to **body system**.

> **Note:** If an ellipsoid is created while referencing the wrong body, it can be easily updated in the Organize panel.

h) Click **multibody=** and select **femur** from the body list.

i) Activate the **ellipsoid radio** button next to create:.

j) Click **create** to create the ellipsoid.

k) Remain in the Ellipsoids panel.

3. Create the Tivia Ellipsoid

While still in the Ellipsoids panel from the previous exercise, complete the following steps:

a) Enter `tibia` in the base name = field.

b) Click **color** and select a color for the tibia.

c) With the node selector active, select the node at the center of the tibia.

d) Enter the following:

- For A, enter `0.25`.
- For B, enter `0.06`.
- For C, enter `0.06`.

e) Keep orientation: set to **body system**.

f) Click **multibody=** and select **tibia** from the body list.

g) Activate the **ellipsoid radio** button next to create:.

h) Click **create** to create the ellipsoid.

i) Remain in the Ellipsoids panel.

4. Create the Foot Ellipsoid

a) Enter `foot` in the base name = field.

b) Click **color** and select a color for the foot.

c) With the node selector active, select the node at the center of the foot.

d) Enter the following:

- For A, enter `0.16`.
- For B, enter `0.1`.
- For C, enter `0.06`.

e) Change orientation: to **system** and click the **system** button to make it active.

f) Select the global triad at the lower left corner of the model window.

g) Keep multibody= set to **tibia**.

h) Activate the **ellipsoid radio** button next to create:.

i) Click **create** to create the ellipsoid.

j) Remain in the Ellipsoids panel.

5. Create the Ball Ellipsoid

a) Enter `ball` in the base name = field.

b) Click **color** and select a color for the ball.
c) Click the triangle switch next to center: and select entities from the pop-up list.
d) Select the node at the center of the ball.
e) Enter the following:
- For A, enter `0.1`.
- For B, enter `0.1`.
- For C, enter `0.1`.

f) Change orientation: to **body system**.
g) Click **multibody=** and select **ball** from the body list.
h) Activate the **ellipsoid radio** button next to create:.
i) Click **create** to create the ellipsoid.
j) Click **return** to leave the Ellipsoids panel.

6.1.9 Creating Joints

In this step, you will create joints using the create joints panel.

Joints connecting bodies in a multi-bodied system are created in the create joints panel.

1. Create the Knee Joint.
a) While still in the safety subpanel, click on **create joints**.
b) Enter `knee` in the name = field.
c) Click **type =** and select **REVO** from the pop-up list.
d) Click the **syst entity selection** box under parent: and select the joint parent coordinate system by clicking on the **coordinate system origin at the knee** and selecting the **purple parent coordinate system** from the coincident selection circle.
You can also single-click the **syst** button and enter `4` in the id= field that appears.
e) Click the **syst entity selection** box under child: to make it active.
f) Select the joint child coordinate system by clicking on the **coordinate system** origin at the knee and selecting the **cyan child coordinate system** from the coincident selection circle.
You can also single-click the **syst** button and enter `6` in the id= field that appears.
g) Click the **multibody entity selection** box under parent: to make it active.
h) Click on the femur ellipsoid in the model window to select its body as the parent body.
i) Click the **multibody entity selection** box under child: to make it active.
j) Click on the tibia ellipsoid in the model window to select its body as the child body.
k) Click **create** to create the joint.
l) Remain in the Multibody Joints panel.

2. Create the hip joint.
a) Enter `hip` in the name = field.
b) Keep type = set to **REVO**.
c) Click the **syst entity selection** box under parent: to make it active.

d) Select the joint parent coordinate system by clicking on the **coordinate system** origin at the hip and selecting the **purple parent coordinate system** from the coincident selection circle.

You can also single-click the **syst** button and enter `3` in the id= field that appears.

e) Click the **syst entity selection** box under child: to make it active.

f) Select the joint child coordinate system by clicking on the **coordinate system origin** at the hip and selecting the **cyan child coordinate system** from the coincident selection circle.

You can also single-click the **syst** button and enter `5` in the id= field that appears.

g) Click the toggle next to the multibody selection box under parent: to set the joint parent body to reference space.

h) Click the **syst** entity selection box under child: to make it active.

i) Click on the femur ellipsoid in the model window to select its body as the child body.

j) Click **create** to create the joint.

k) Remain in the Multibody Joints panel.

3. Create the Ball Joint

a) Enter `ball` in the name = field.

b) Click **type =** and select **FREE** from the pop-up list.

c) Click the **syst entity selection** box under parent: and enter `8` in the id= field.

d) Click the **syst entity selection** box under child: and enter `9` in the id= field.

e) Keep the parent body set to **reference space**.

f) Click on the **ball ellipsoid** in the model window to select its body as the child body.

g) Click **create** to create the joint.

h) Click **return** to leave the panel.

6.1.10 Define Systems of Bodies

In this step, you will define systems of bodies.

1. Define the leg SYSTEM.MODEL.

SYSTEM.MODEL, SYSTEM.REF_SPACE, and MADYMO are defined in the Assemblies panel. HyperMesh assemblies are collections of collectors. The SYSTEM, BODY, FE_MODEL structure can also be defined in the Model Browser. Launching the Model Browser provides a visual reference for the model hierarchy.

a) While still in the safety subpanel, click on **assemblies**.

b) Enter `leg_system` in the assem name = field.

c) Click **type =** and select **SYSTEM** from the pop-up list.

d) Click the **multibodies entity selection** box to make it active.

e) Include the femur and tibia ellipsoids in the system by clicking the femur and tibia ellipsoids in the graphics window.

> **Note:** You can also include the femur and tibia ellipsoids in the system by clicking the **multibodies** button again, selecting the **femur** and **tibia** bodies from the list, and clicking **select**.

f) Click **create** to create the assembly.

g) Remain in the Assemblies panel.

2. Define the ball SYSTEM.MODEL.

a) Enter `ball_system` in the assem name = field.

b) Click **type =** and select **SYSTEM** from the pop-up list.

c) With the multibodies selection box active, select the ball ellipsoid to include its body in the system.

> **Note:** You can also include the ball ellipsoid in the system by clicking the **multibodies** button again, selecting **ball** from the list, and clicking **select**.

d) Click **create** to create the assembly.

e) Remain in the Assemblies panel.

3. Define the SYSTEM.REF_SPACE.

a) Enter `ref_space` in the assem name = field.

b) Click **type =** and select **SYSTEM** from the pop-up list.

c) With the multibodies selection box active, select the ground plane to include its body in the system.

> **Note:** You can also include the ground plane in the system by clicking the **multibodies** button again, selecting **ground** from the list, and clicking **select**.

d) Click **create/edit** to create the assembly and bring it up in the Card Editor.

e) 5. Click **MODEL** and select **REF_SPACE** from the pop-up list.

f) Click **return** to save changes made to the card image and close the Card Editor.

g) Remain in the Assemblies panel.

4. Define the MADYMO MODEL assembly.

a) Enter MADYMO_Model in the assem name = field.

b) Click **type=** and select MADYMO from the pop-up list.

c) Click the **assems entity selection** box to make it active.

d) Click **assems** again to bring up the assemblies list.

e) Activate the **leg_system,**, **ball_system**, and **ref_space** checkboxes and click **select**.

f) Click **create/edit** to create the assembly and bring it up in the Card Editor.

g) Click on **TIME_STEP** and enter `1e-4`.

h) Click **return** to save changes made to the card image and close the Card Editor.

i) Click **return** to leave the Assemblies panel.

6.1.11 Define the Initial Conditions and Position the Leg

In this step, you will define the initial conditions and position the leg.

1. While still in the safety subpanel, click on **dummy**.
2. Activate the **incremental radio** button.
3. Click on the triangle switch next to the yellow selection box and select **multibodies** from the pop-up list.

   Multibodies is used for positioning ellipsoid rigid body models. Comp is used to position finite element and facet dummy models.
4. Enter `10` in the increment field.
5. Select the **femur** ellipsoid.

   HyperMesh automatically searches up to the next joint in the hierarchy and down through all the joints, bodies, and ellipsoids below in the hierarchy.
6. Rotate the hip by completing one of the following:
   - Use the < and > buttons next to x rot to rotate the hip child joint coordinate system about the x axis of the hip parent joint coordinate system.
   - Enter rotation values in the current fields.
7. Select the **tibia** or **foot** ellipsoids.

   HyperMesh automatically searches up to the next joint in the hierarchy.
8. Rotate the knee by completing one of the following:
   - Use the < and > buttons next to x rot to rotate the knee child joint coordinate system about the x axis of the knee parent joint coordinate system.
   - Enter rotation values in the current fields.
9. Click **return** to leave the Dummy panel.

6.1.12 Apply Gavity and Create Vector Data

In this step, you will apply gravity and create vector data in HyperMesh.

A gravity vector is applied by defining the vector in HyperMesh then applying the vector to a system of bodies in the Curve Editor.

1. Click **XY Plots** > **Curve Editor**.
2. Click on **New...**.
3. Enter `function_data` in the Name= field.
4. Click **proceed**.
5. Select **function_data** in the top left corner.
6. In the first row of the table, enter:
   - In the X column, enter `0`.
   - In the Y column, enter `-9.81`.
7. In the second row of the table, enter:
   - In the X column, enter `2`.

- In the Y column, enter `-9.81`.

8. Select a color, and activate the **Display** checkbox.
9. Set the Symbol field to **Triangle**.
10. Set the Line Style field to **Solid**, and activate the **Thick line** checkbox.
11. Click **Update**.
12. Click **Close** to close the Curve Editor.

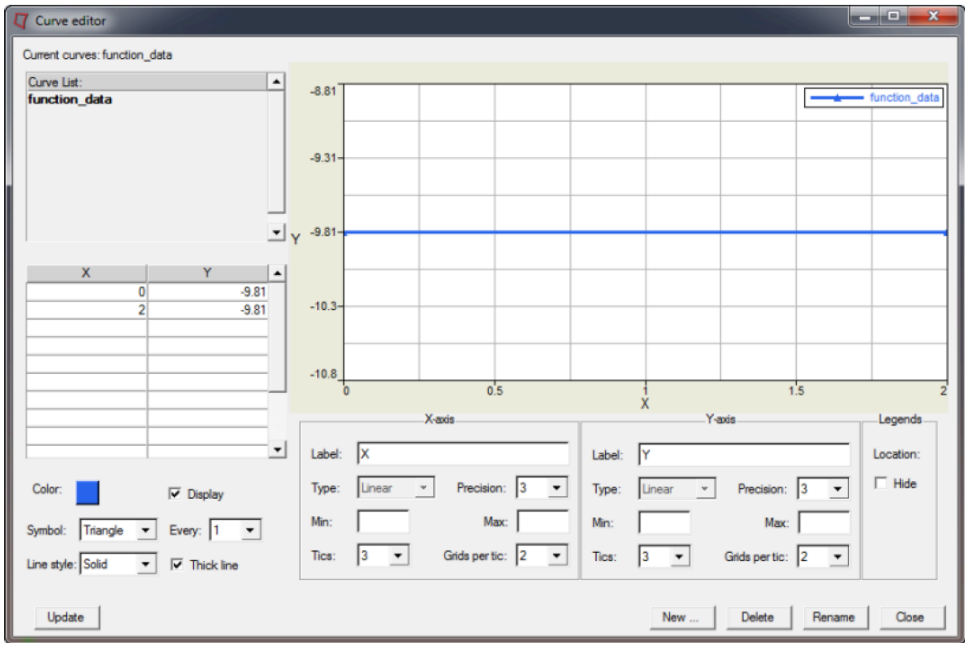


Figure 1106:

6.1.13 Apply the Gravity Vector to the Dummy

In this step, you will apply the gravity vectory to the dummy.

The gravity vector is applied by card editing the HyperMesh assembly which represents the dummy system of bodies.

1. Right-click on the **leg_system assembly** in the Model Browser and pick **Card Edit**.
2. Activate the **LOAD.SYTEM_ACC** checkbox.
3. Click once on the yellow **AZ_FUNC curve selection** box to make it active.
4. Click on **AZ_FUNC** and select **function_data**.
5. Click **return** to save the changes and leave the card image.
6. Click **return** to exit the Card Editor panel.
7. Repeat steps 1 through 5 for ball_system.

6.1.14 Create Ball Ellipsoid Characteristics

In this step, you will create ball ellipsoid characteristics.

Ellipsoid characteristics are properties of ellipsoids and are defined as HyperMesh properties cards.

1. Create the function curve using the Curve Editor.
   a) Click **XY Plots** > **Curve Editor**.
   b) Click on **New...**.
   c) Enter `curve2` in the Name= field.
   d) Click **proceed**.
   e) Select **curve2** in the top left corner.
   f) In the for row of the table, enter:
      - In the X column, enter `0`.
      - In the Y column, enter `0`.
   g) In the second row of the table, enter:
      - In the X column, enter `0.01`.
      - In the Y column, enter `1000`.
   h) In the third row of the table, enter:
      - In the X column, enter `0.02`.
      - In the Y column, enter `10000`.
   i) In the fourth row of the table, enter:
      - In the X column, enter `0.03`.
      - In the Y column, enter `100000`.
   j) Set the Symbol field to **Triangle**.
   k) Set the Line style field to **Solid**.

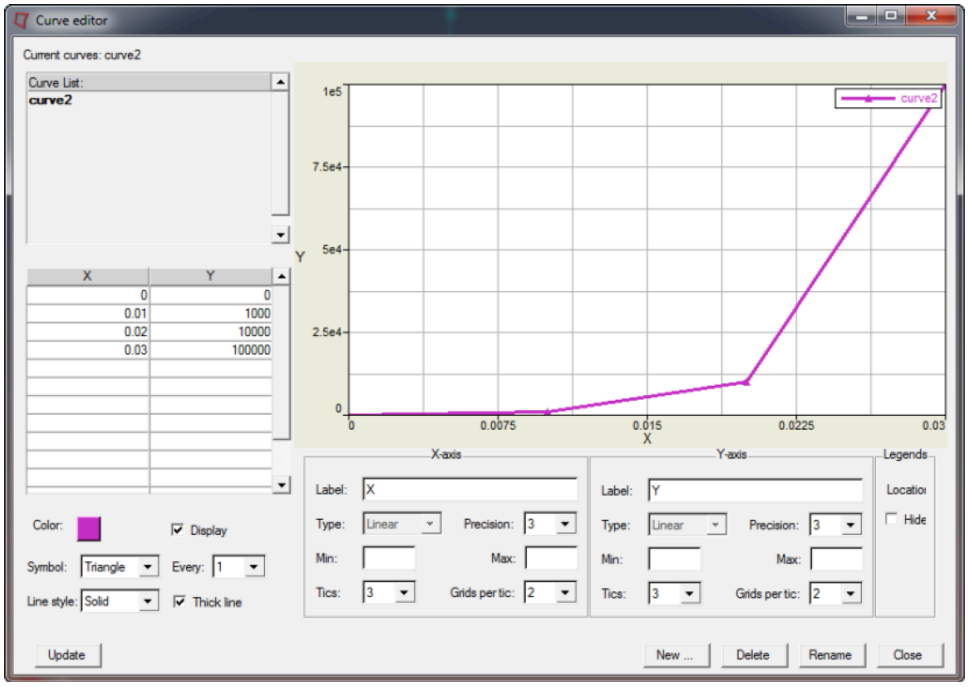


Figure 1107:

a) Click **Update**.
b) Click **Close**.

2. Create the property for the ellipsoid.

a) Right-click in the Model Browser and pick **Create** > **Property**.
b) Enter `ball_char` in the Name= field.
c) Click **card image=** and select **CHAR** from the pop-up list.
d) Click the **Card edit property** upon creation to activate the option
e) Click **Create** to create the property and open the Card Editor.
f) Click **LOAD_FUNC** and select the created characteristic **curve2**.

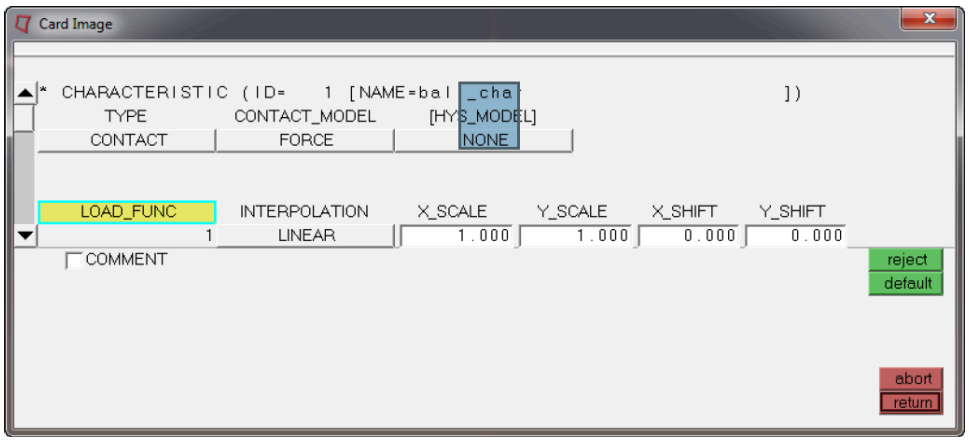


Figure 1108:

g) Click **return** to save the changes and exit the panel.

3. Assign the characteristic.
   a) Click the **Card Edit** icon and click **ellipsoids**.
   b) Select the **ball ellipsoid**.
   c) Click **edit**.
   d) Click twice on **CHAR** to bring up the property list.
   e) Select **ball_char**.

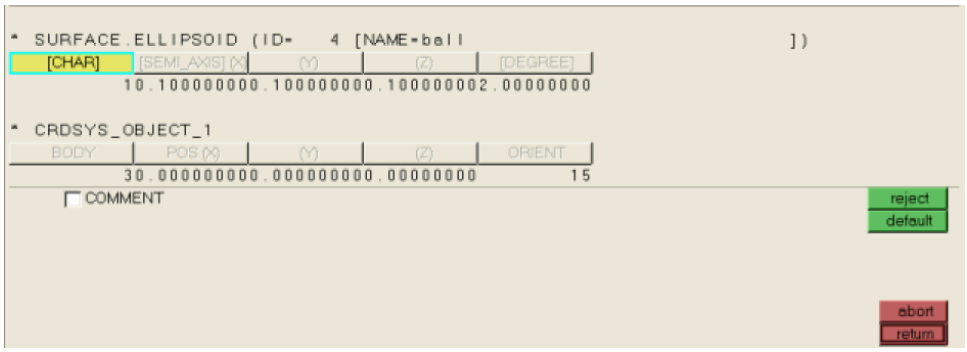


Figure 1109:

   f) Click **return** to save the changes and exit the card image.
   g) Click **return** to exit the Card Edit panel.

6.1.15 Define Contacts

In this step, you will define Contacts in the Contacts panel.

Contacts are created in the Contacts panel. A MADYMO GROUP consists of a single entity LIST or multiple LIST cards. LIST and GROUP cards are both defined in the HyperMesh Entity Sets panel.

1. Create a GROUP for the foot contact surface.
   a) Click **Tools** > **Create** > **Sets**.
   b) Enter `foot_ellipsoid_list` in the name= field.
   c) Click the toggle next to entity to set the selector to ellipsoids.
   d) Graphically select the **foot** and **tibia** ellipsoids.
      You can also click **ellipsoids** and click on **foot** and **tibia** in the list.
   e) Click **create** to create the entity set.
   f) Click the toggle next to entity to set the selector to sets.
   g) Enter `lower_leg_group` in the name= field.
   h) Click on the **sets selection** box.
   i) Activate the **foot_ellipsoid_list** checkbox and click **select**.
   j) Click **create** to create the entity set.
   k) Remain in the Entity Sets panel.

2. Create a GROUP for the ball contact surface.
   a) Enter `ball_ellipsoid_list` in the name= field.
   b) Click the toggle next to entity to set the selector to ellipsoids.
   c) Select the **ball** ellipsoid.
   d) Click **create** to create the entity set.
   e) Click the toggle next to entity to set the selector to sets.
   f) Enter `ball_group` in the name= field.
   g) Click on the **sets selection** box.
   h) Activate the **ball_ellipsoid_list** checkbox and click **select**.
   i) Click **create** to create the entity set.
   j) Remain in the Entity Sets panel.
3. Create a GROUP for the ground contact surface.
   a) Enter `ground_plane_list` in the name= field.
   b) Click the toggle next to entity to set the selector to mbplanes.
   c) Select the **ground plane** by clicking on it in the model window.
      You can also click **mbplanes** > **ground** > **select**.
   d) Click **create** to create the entity set.
   e) Click the toggle next to entity to set the selector to sets.
   f) Enter `ground_group` in the name= field.
   g) Click on the **sets selection** box.
   h) Activate the **ground_plane_list** checkbox and click **select**.
   i) Click **create** to create the entity set.
   j) Click **return** to leave the panel.
4. Define the Ball to Foot contact.
   a) Click **BCs** > **Create** > **Interfaces**.
   b) Activate the **create radio** button.
   c) Enter `ball_to_foot` in the name= field.
   d) Click **type=** and select **MB_MB** from the pop-up list.
   e) Click **create** to create the contact definition.
   f) Activate the **add radio** button.
   g) Click twice on the **master sets** selection box.
   h) Activate the **ball_group** checkbox and click **select**.
   i) Click the **master update button**.
   j) Click twice on the **slave sets selection** box.
   k) Activate the **lower_leg_group** checkbox and click **select**.
   l) Click the **slave update** button.
   m) Remain in the Contacts panel.
5. Define the Ground to Ball contact.
   a) Click the **create radio** button to open the subpanel.

b) Enter `ground_to_ball` in the name= field.
c) Click **type=** and select **MB_MB** from the pop-up list.
d) Click **create** to create the contact definition.
e) Activate the **add radio** button.
f) Click twice on the **master sets selection** box.
g) Activate the **ground_group** checkbox and click **select**.
h) Click the **master update** button.
i) Click twice on the **slave sets selection** box.
j) Activate the **ball_group** checkbox and click **select**.
k) Click the **slave update button**.
l) Activate the **card image radio** button in the Contacts panel.
m) Click the green **edit** button.
n) Enter `0.1` for FRIC_COEF.
o) Click **return** to leave the card image panel.
p) Click **return** to exit the panel.

6.1.16 Export and Run the Model

In this step, you will export and run the model.

1. Right-click on the **MADYMO_Model** assembly in the Model Browser and click **Card Edit**.
2. In the card image, enter `0.05` in the TIME_END field.
3. Click **return** to save the changes and exit the card image.
4. Click  to export the MADYMO file.
5. Use the File Browser to specify an XML filename and click **Save**.
6. Solve the saved file using the MADYMO solver.
7. View the results by loading the KIN3 file.

6.2 HM-4910: Set up a MADYMO Occupant Safety Analysis in HyperMesh

In this tutorial, you will load the MADYMO user profile and import XML files.

Before you begin, copy the following files from `<hm.zip>/interfaces/madymo/` to your working directory:

- `truck_model.xml`
- `steering_column.xml`
- `seatbelt_system.xml`
- `dummy_pulse.dat`

The MADYMO HyperMesh interface supports the import and export of MADYMO files. A MADYMO XML file can be read using the Import tab. The model will be displayed. The model can be altered and the finished model can be written using the Export tab.

6.2.1 Load the MADYMO User Profile

In this step, you will load the MADYMO profile in HyperMesh.

The MADYMO user profile configures the user interface to be MADYMO specific. The user profile also loads the XML import reader and MADYMO template file. The template file defines MADYMO specific entity types and attributes versus generic entities.

1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to **Madymo**.
3. Set the template drop-down to **Madymo 70**.
4. Click **OK**.
5. Open the Utility menu by clicking **View** > **Browsers** > **HyperMesh** > **Utility** from the menu bar.

6.2.2 Retrieve and View the Model File

In this step, you will open the model file and view it in HyperMesh.

The `truck_model.xml` file was created by converting an NCAC LS-DYNA truck model to MADYMO within the HyperMesh interface.

1. Click **File** > **Import** > **Solver Deck**.
2. In the File: field, select the `truck_model.xml` file and click **Open**.
3. Click **Import Options** to expand the panel and display the advanced importing options.
4. Click on the **Display Import Errors** checkbox to deactivate this function.
5. Leave all other default settings as they are and click **Import** to import the model.
6. Remain in the Import tab.

6.2.3 Import the Steering Column File

In this step, you will open the steering column file and view it in HyperMesh.

The `steering_column.xml` file was created from scratch within the HyperMesh interface.

1. In the File: field, select the `steering_column.xml` file and click **Open**.
2. Leave all other default settings as they are and click **Import** to import the model.

6.2.4 Position the Steering Column

In this step, you will position the steering column.

The Set Joints panel contains functionality for defining joint rotations, joint orientations for all joints, and global positioning for the highest joint in any system of bodies.

1. From the Analysis page, click **safety** and then select the **set joints** subpanel.
2. Select the **H-point** subpanel.

   The H-point subpanel is used for defining global positioning for the highest joint in the system of bodies.
3. Ensure the yellow entity selection is set to **multibody**.

   To change, click on the triangle switch button next to the yellow entity selection box and select **multibodies** from the pop-up list. Comp is used to position finite element and facet dummy models. Multibody is used for positioning ellipsoidal rigid body models.
4. Click on any ellipsoid in the steering column.
5. In the Position fields, enter the following:
   - For x, enter `3.30`.
   - For y, enter `0.41`.
   - For z, enter `0.98`.
6. Click the green **position** button to set the position of the steering column.
7. Enter `180.00` in the increment field.
8. Click the triangle switch next to N1, N2, N3 and select **z-axis** from the pop-up window.
9. Click the green **rotate+** button to rotate the steering column.

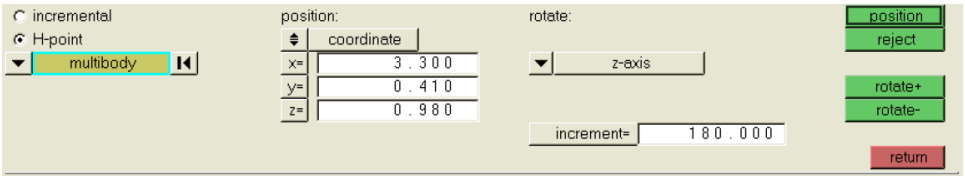


Figure 1110:

10. Click **return** to exit the panel.

6.2.5 Import File

In this step, you will import the file.

The `d_hyb350el_usr.xml` file is taken directly from your MADYMO installation. These files can be found in the path `...\share\dbs\dummies\3d` folder in typical MADYMO installation.

1. Click **File** > **Import** > **Solver Deck**.
2. In the File: field, select the `d_hyb350el_usr.xml` file and click **Open**.
3. Click **Import** to import the model.

6.2.6 Position the Dummy H-Point

In this step, you will position the dummy H-point.

1. From the Analysis page, click **safety** and then select the **dummy** subpanel.
2. Select the **H-point** subpanel.
3. Click on any **ellipsoid** in the dummy.
4. In the Position fields, enter the following:
   - x = 2.16
   - y = 0.41
   - z = 0.88
5. Click the green **position** button to set the position of the dummy.
6. Enter `18` in the increment field.
7. Click the triangle switch under rotate: and select **y-axis** from the pop-up window.
8. Click the green **rotate-** button to rotate the dummy back against the seat.
9. Remain in the Dummy Positioning panel.

6.2.7 Position the Dummy Limbs

In this step, you will position the dummy limbs.

You can quickly find the dummy limbs used in the steps below by using the Model Browser. Right-click on any entity and select **Isolate** to display only the part being manipulated.

You will use the Dummy Positioning panel for this step.

1. Rotate the femur ellipsoids.
   a) Select the **incremental** subpanel.
   b) Enter `10` in the increment field.
   c) Click on a femur ellipsoid **(FemurR_bod)**.
      HyperMesh automatically searches up to the next joint in the hierarchy.
   d) Rotate the hip child joint coordinate system about the x, y, and z axes of the hip parent joint coordinate system by using y rot >.

e) Click the **reset** button next to the multibody selection option.

f) Repeat steps 1.a through 1.e for the other femur ellipsoid (FemurL_bod).

2. Rotate the upper tibia.

a) Select the right upper tibia ellipsoid **(tibiaUpR_bod)**.

b) In the y rot row, set the current field to **-60** and click **Enter**.

c) Repeat steps 2.a and 2.b for the left upper tibia ellipsoid (tibiaUpL_bod).

3. Rotate the upper arm.

a) Select left upper arm ellipsoid **(ArmUpL_bod)**.

b) In the x rot row, set the current field to **-30** and click **Enter**.

c) Repeat steps 3.a and 3.b for the right upper arm ellipsoid (ArmUpR_bod).

4. Rotate the lower arm.

a) Select the left lower arm ellipsoid **(ArmLowL_bod)**.

b) In the x rot row, set the current field to **-5** and click **Enter**.

c) In the y rot row, set the current field to **-70** and click **Enter**.

d) Select the right lower arm ellipsoid **(ArmLowR_bod)**.

e) In the x rot row, set the current field to **5** and click **Enter**.

f) In the y rot row, set the current field to **-70** and click **Enter**.

g) Remain in the Dummy Positioning panel.

6.2.8 Set the Steering Wheel Tilt

In this step, you will set the tilt of the steering wheel.

You will complete this step in the Dummy panel.

1. Activate the **incremental radio** button.

2. Enter `10` in the increment field.

3. Select an ellipsoid in the steering wheel.

HyperMesh automatically searches up to the next joint in the hierarchy.

4. Click the **rotate:** toggle to change the joint dof to joint orientation.

Joint dof rotates the child joint coordinate system locally about its axes. Joint orientation rotates the parent joint coordinate system. Rotating the child joint coordinate system instead of the parent joint coordinate system to set the tilt will initiate an undesired initial torque. Rotating the parent joint coordinate system will temporarily "unlock" the joint.

5. Use the **<** button next to x rot to set the tilt.

6. Set the rotate: toggle back to **joint dof**.

7. Click **return** to exit the panel.

6.2.9 Define the MADYMO/SYSTEM/FE MODEL Hierarchy

In this step, you will define the MADYMO/SYSTEM/FE MODEL Hierarchy.

Since three running MADYMO models were imported, extra INERTIAL_SPACE and MADYMO_MODEL blocks were defined.

1. Click on the Model Browser.
2. Drag and drop **truck_system** and **steering_system** from ASSEMBLY.MADYMO and ASSEMBLY.MADYMO.1 respectively into the ASSEMBLY.MADYMO.2 folder.
3. Right-click and select **delete** on ASSEMBLY.MADYMO and ASSEMBLY.MADYMO.1.

6.2.10 Apply a Pulse Function

In this step, you will apply a pulse function.

A pulse function is applied by reading vector data from an external file source and applying the vector to a system of bodies in the Card Editor.

1. Read the pulse data from an external file.

   HyperMesh can read vector data from the same file formats supported by HyperGraph. HyperMesh recognizes file types based on the file's contents and automatically selects the correct reader to import the data.

   a) Click **XYPlots** > **Create** > **Plots**.
   b) In the plot= field, enter `acceleration_data`.
   c) Click **create plot** to create an empty plot window.
   d) Click **return** to leave the Plots panel.
   e) Click **XYPlots** > **Edit** > **Plots**.
   f) Click on **create** to open the create subpanel.
   g) In the file field, click **load**.
   h) Browse to the `/hm/interfaces/madymo` directory and select the `dummy_pulse.dat` file.
   i) Activate the **y= radio** button.
   j) Click the **+** button next to comp= to set the y component to Column 2.
   k) Click **create**.
   l) Remain in the Edit Curves panel.
2. Apply the pulse vector to the dummy.

   The pulse vector is applied by card editing the HyperMesh assembly which represents the dummy system of bodies.

   a) Right-click **Hybrid_III_50th** in the Model Browser and click **Card Edit**.
   b) Activate the **LOAD.SYSTEM_ACC** checkbox.
   c) Click once on the yellow **AX_FUNC curve selection** box.
   d) Click on the pulse curve **(curve1)** in the plot window.
   e) Click **return** to save the changes and leave the card image.

6.2.11 Apply Gravity

In this step, you will apply gavity using a gravity vector.

A gravity vector is applied by defining the vector in HyperMesh then applying the vector to a system of bodies in the card image.

1. Create vector data in HyperMesh.
   a) From the XY Plots menu, click on **Curve Editor....**
   b) Click on **New...** at the bottom of the window.
   c) In the Name= field, enter `gravity`.
   d) Click **proceed**.
   e) In the Curve Editor window, select **gravity** on the top left corner.
   f) In the first row of the table, enter:
      - In the X column, enter `0`.
      - In the Y column, enter `-9.81`.
   g) In the second row of the table, enter:
      - In the X column, enter `2`.
      - In the Y column, enter `-9.81`.
   h) Select a color.
   i) Activate the **Display** checkbox.
   j) Set the Symbol field to **Triangle**.
   k) Set the Line style field to **Solid**, and activate the **Thick line** checkbox.
   l) Click **Update**.
   m) Click **Close** to close the Curve Editor.
2. Apply the gravity vector to the dummy.

   The gravity vector is applied by card editing the HyperMesh assembly which represents the dummy system of bodies.
   a) Right-click the **Hybrid_III_50th** assembly in the Model Browser and click **Card Edit**.
   b) Click once on the yellow **AZ_FUNC curve selection** box.
   c) Click on the gravity curve **(gravity)** in the plot window.
   d) Click **return** to save the changes and leave the card image.

6.2.12 Define Contacts

In this step, you will define the Contacts using the contacts panel.

Contacts are created in the contacts panel. A MADYMO GROUP consists of a single entity LIST or multiple LIST cards. LIST and GROUP cards are defined in the HyperMesh Entity Sets panel.

1. Create a contact entity.
   a) Click **BCs** > **Create** > **Interfaces**.
   b) In the name= field, enter `seat_to_dummy`.

c) Click **type=** and select **MB_MB** from the pop-up list.
d) Click **create/edit** to create the entity and bring up the Card Editor.

2. Define contact attributes.
a) In the FRIC_COEF field, enter `0.1`.
b) Click on the box under CONTACT_TYPE and select **MASTER** from the pop-up list.
c) Click **return** to save the changes and close the Card Editor.

3. Add master and slave surfaces.
a) Click **add** to open the add subpanel.
b) Next to the master: field, click the **yellow sets entity selection** box twice to bring up a list of sets.
c) Activate the check box next to seat_gmb to make seat_gmb the master surface.
d) Click **select**.
e) Click **update** to the right of master: to update the master surface definition.
f) Next to the slave: field, click the **yellow sets entity selection** box twice to bring up a list of sets.
g) Activate the checkbox next to thorax_gmb and pelvis_gmb to use these as the slave surface.
h) Click **select**.
i) Click **update** to the right of slave: to update the slave surface definition.
j) Repeat steps 1 through 3.i for the other contacts.

6.2.13 Add Seatbelts

In this step, you will create a seatbelt system using the seatbelt routing panel.

A seatbelt system can be created by scratch, or by importing and modifying an existing system. This exercise focuses on creating a seatbelt system in the seatbelt routing panel. When creating seatbelts from scratch, create a component of type BELT for each BELT_TYING or RETRACTOR point then connect the belt points in the seatbelt routing panel.

1. Import a file.
The `seatbelt_system.xml` contains a running model of a single belt system.
a) Click **File** > **Import** > **Solver Deck**.
b) In the File: field, select the `seatbelt_system.xml` file and click **Open**.
c) Click **Import** to import the model.

2. Delete existing belt elements.
a) Press the **F2** key to enter the Delete panel.
b) Set the yellow entity selector to **elements**.
c) Click **elements** and select **by collector** from the pop-up list.
d) Activate the check boxes next to fe_shoulder_belts and fe_lap_belts and click **select**.
e) Right-click on the last lap belt LINE2 element attached to the constraint in the floor (left end of pelvis) to deselect it.

f) Click **delete entity** to delete the selected entities.

g) Click **return** to exit the panel.

3. Route the should belt.

a) Enter the Seatbelt panel from the Safety panel.

b) Click **G** to open the Global panel.

c) Click **component=** and click **fe_shoulder_belts** in the component list.

d) Click **return** to leave the Global panel.

e) Select the node at the end of the BELT_SEGMENT element near the left shoulder.

f) Select the node at the end of the higher BELT_SEGMENT element near the buckle.

g) Click the **ellipsoids entity selection** button to make it active.

h) Click the **ellipsoids** in the thorax.

Alternatively click the **ellipsoids** button again to display the ellipsoid list, select **ellipsoids by collector**, and click on all the bodies in the thorax.

i) Click **orient** to display the belt routing line segments.

j) Click on one of the red end segments to activate it.

k) While holding the left mouse button, drag the mouse up and down to adjust the location of the belt segment.

l) Repeat steps 3.j and 3.k for the other end segment and the middle segment.

m) Click **mesh** to create the mesh.

n) Set the using size= to **0.03**.

o) Click **return**.

p) Remain in the Seatbelt panel.

4. Route the lap belt.

a) Click **G** to open the Global panel.

b) Click **component=** and click **fe_lap_belts** in the component list.

c) Click **return** to close the Global panel.

d) In the Seatbelt panel, select the node at the end of the LINE2 element near the floor constraint.

e) Select the node at the end of the lower BELT_SEGMENT element near the buckle.

f) Select the ellipsoids in the pelvis and abdomen.

g) Click **orient** to display the belt routing line segments.

h) Click on one of the red end segments to activate it.

i) While holding the left mouse button, drag the mouse up and down to adjust the location of the belt segment.

j) Repeat steps 4.h and 4.i for the other end segment and the middle segment.

k) Click **mesh** to create the mesh.

l) Set the using size= to **0.03**.

m) Click **return** to close the Seatbelt panel.

6.2.14 Equivalence Duplicate Nodes

In this step, you will equivalence duplicate nodes.

Many times it is preferred to use a duplicate node as the to or from node to define BELT_TYING. However, sometimes this feature is not preferred. Because inserting a coincident node is more difficult than equivalencing nodes, the seatbelt panel always uses a duplicate node. This training exercise does not use the duplicate node option in BELT_TYING so the duplicated nodes must be equivalenced.

1. Replace nodes.
   a) Click **O** to open the Options panel.
   b) Click on the **graphics** subpanel.
   c) Activate the **coincident picking** checkbox.
   d) Click **return** to leave the Options panel.
   e) Press **F3** to enter the Replace panel.
   f) Click a node selected as a to or from node in the Seatbelt panel.
   g) In the selection circle, select the node with the lower ID.
   h) Click on the same node location and select the other node from the coincident selection circle.
   i) Repeat steps 1.a through 1.h for the other three to and from nodes.
   j) When finished, click **return** to close the panel.
2. Create a group multi-body for Contacts.
   a) Click **Tolls** > **Create** > **Sets**.
   b) Set the yellow entity selector to **nodes**.
   c) In the name= field, enter `seatbelt_node_list`.
   d) Click on the **nodes selection** box and select **by collector** from the pop-up list.
   e) Activate the **fe_shoulder_belts** and **fe_lap_belts** checkboxes and click **select**.
   f) Click **create** to create the entity set.
   g) Set the yellow entity selector to **sets**.
   h) In the name= field, enter `seatbelt_group`.
   i) Click on the **sets selection** box.
   j) Activate the **seatbelt_node_list** checkbox and click **select**.
   k) Click **create** to create the entity set.
   l) Click **return** to leave the Entity Sets panel.
3. Define the contact.

   Applying MB_FE Contacts is very similar to applying MB_MB Contacts.
   a) Click **BCs** > **Create** > **Interfaces**.
   b) In the name= field, enter `dummy_to_belts`.
   c) Click **type=** and select **MB_FE** from the pop-up list.
   d) Click **create/edit** to create the contact definition and bring it up in the Card Editor.
   e) Set CONTACT_FORCE to **KINEMATIC**.
   f) Under FRIC_COEF., enter `0.1`.

g) Click **return** to save the changes and close the Card Editor.
h) Click the **add** subpanel.
i) Click the **master sets selection** box.
j) Activate the **Shoulders_gmb**, **Thorax_gmb**, **Abdomen_gmb**, and **Pelvis_gmb** checkboxes and click **select**.
k) Click the **master update** button.
l) Click the **slave sets selection** box.
m) Activate the **seatbelt_group** checkbox and click **select**.
n) Click the **slave update** button.
o) Click **return** to close the Contacts panel.

4. Update the MADYMO/SYSTEM/FE Model hierarchy.

   Since three running MADYMO models were imported, extra INERTIAL_SPACE and MADYMO_MODEL blocks were defined.

   a) Click on the **Model Browser**.
   b) Click and drag the **Vehicle folder** from its ASSEMBLY.MADYMO folder to the top ASSEMBLY.MADYMO.2 folder.
   c) Click and drag **slipring_seg** and **buckle_seg** from their ASSEMBLY.MADYMO folder to the top ASSEMBLY.MADYMO.2 folder.
   d) Right-click and select **delete** on ASSEMBLY_MADYMO.

6.2.15 Export and Run the Model

In this step, you will export and run the model.

1. Right-click on the **ASSEMBLY.MADYMO.2** assembly in the Model Browser and click **Card Edit**.
2. In the card image, enter `0.05` in the TIME_END field.
3. Click **return** to save the changes and exit the card image.
4. Click **File** > **Export** > **Solver Deck**.
5. In the file: field, enter an XML file name.
6. Click **Export**.

Nastran

7

Nastran tutorials.

This chapter covers the following:

- 7.1 HM-4200: Set Up Nastran Static Analysis in HyperMesh (p. 886)

7.1 HM-4200: Set Up Nastran Static Analysis in HyperMesh

In this tutorial, you will learn how to define a model in HyperMesh, apply boundary conditions in HyperMesh, write the Nastran input deck, and view the results.

Before you begin, copy the `plate_hole.hm` from `<hm.zip>/interfaces/nastran/` to your working directory.

You will use the HyperMesh Nastran interface to create finite elements on the geometry of a plate with a hole, apply boundary conditions, and perform finite element analysis.

7.1.1 Load the Nastran User Profile

In this step, you will load the Nastran profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Nastran**.

7.1.2 Retrieve and View the Model File

In this step, you will open the model file and view it in HyperMesh.

1. Open a model file by completing one of the following:
   - Clicking **File** > **Open** > **Model** from the menu bar.
   - Clicking  on the Standard toolbar.
2. In the **Open Model** dialog, open the `plate_hole.hm` file.

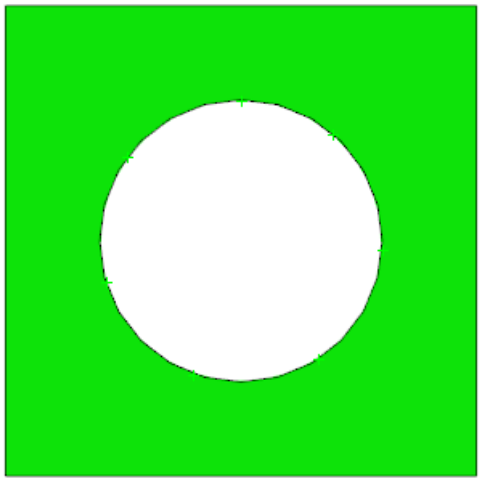


Figure 1111:

The model appears in the graphics area.

7.1.3 Create Material Collectors and Components

In this step, you will create material collectors and components in HyperMesh.

1. In the Model Browser, right-click and select **Create** > **Material** from the context menu.

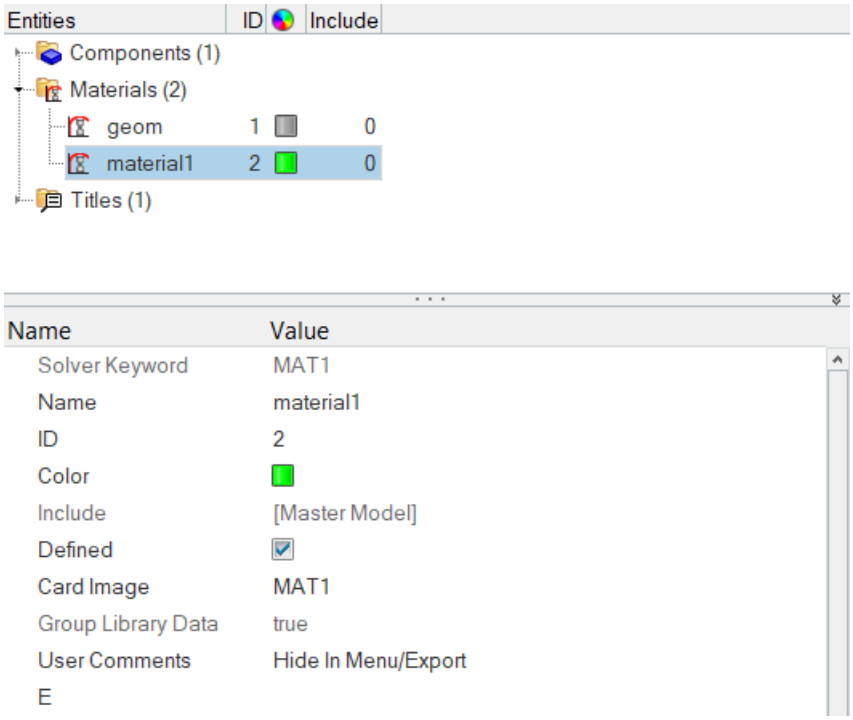


Figure 1112:

   A new material opens in the Entity Editor.
2. For Name, enter `steel`.
3. Set card image to **MAT1**.
4. For E, enter `2e5`.
5. For NU, enter `0.30`.
6. For RHO, enter any number (if needed).
7. In the Model Browser, right-click and select **Create** > **Component** from the context menu.

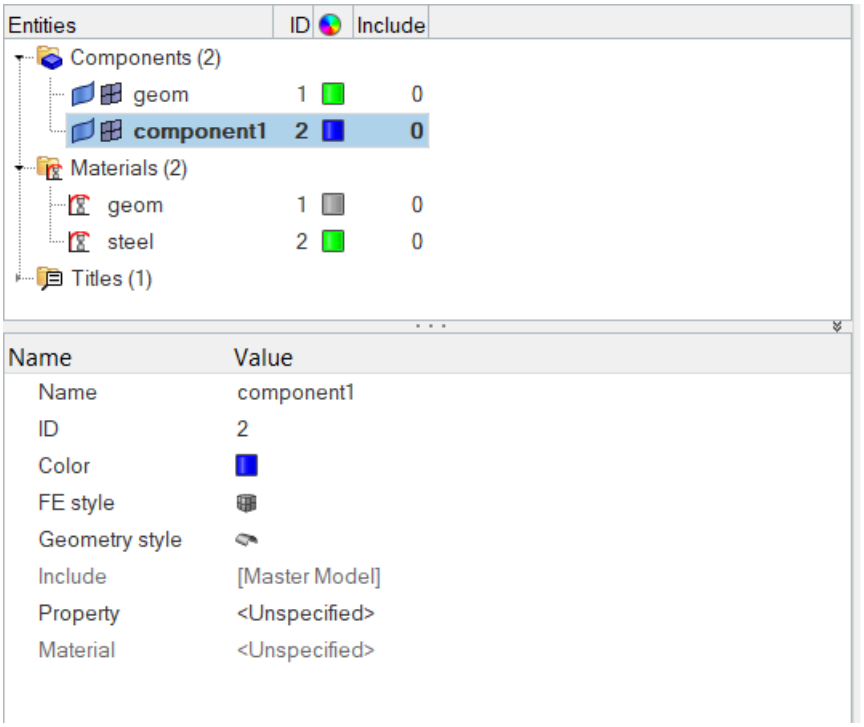


Figure 1113:

A new component opens in the Entity Editor.

8. For Name, enter `shells`.
9. Right-click on **Property** and select **Create** from the context menu.

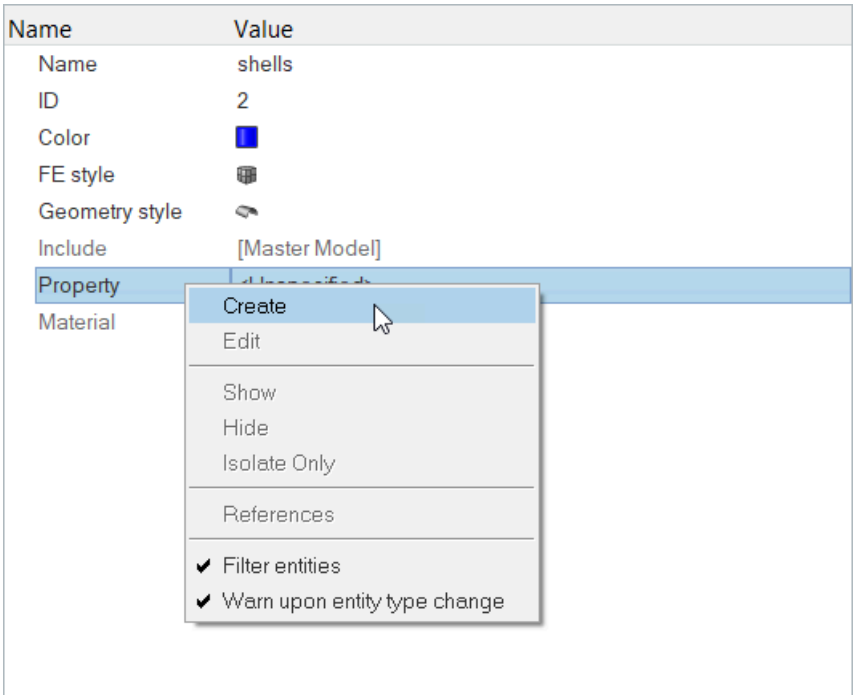


Figure 1114:

10. In the **Create Properties** dialog, Name field, enter `pshells`.
11. Set card image to **PSHELL**.
12. For T (thickness), enter `1.0`.
13. Click **Close**.

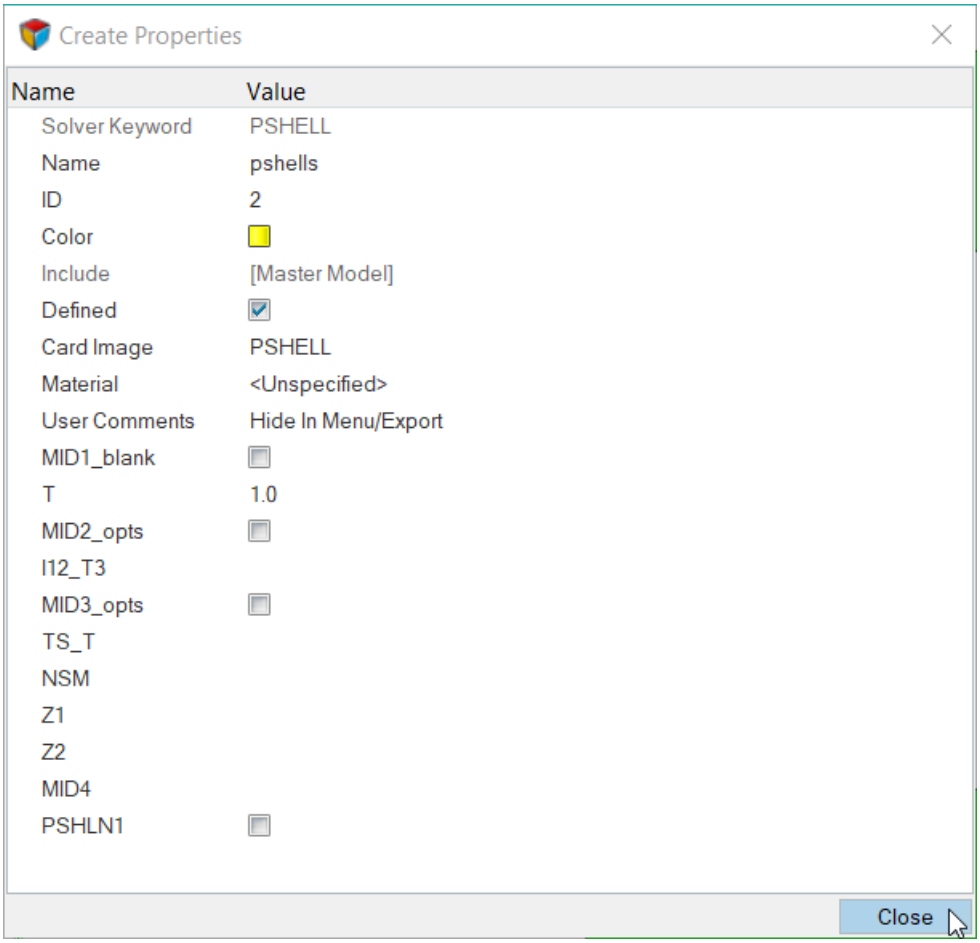


Figure 1115:

HyperMesh assigns the property pshells to the component shells.

14. For Material, click **Unspecified** > **Material**.

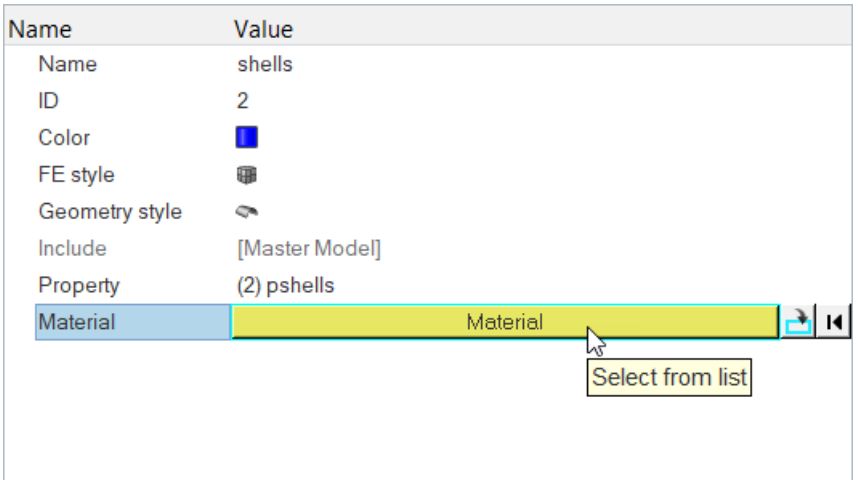


Figure 1116:

15. In the **Select Material** dialog, select **steel** and then click **OK**.

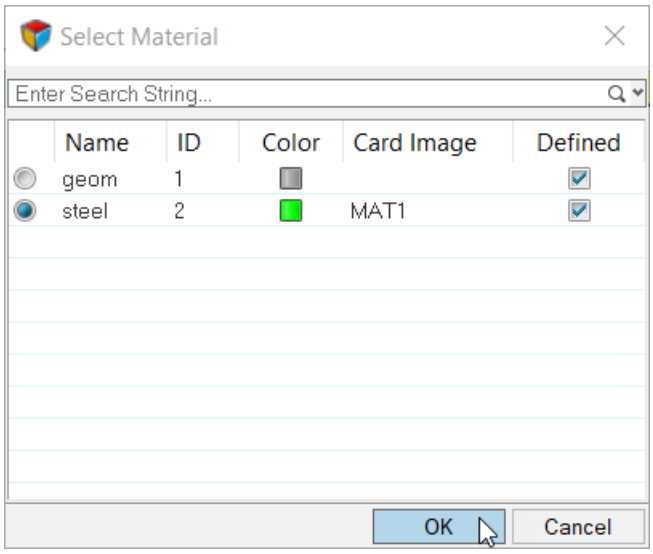


Figure 1117:

HyperMesh assigns the material steel to the component shells.

7.1.4 Mesh the Geometry

In this step, you will mesh the geometry using the Automeshing panel.

Use the Automeshing panel to mesh interactively on surfaces. This panel contains tools for manipulating surface edges and meshing fixed points (locations where the mesher is required to place a node.) The elements generated are organized into the current component and shells.

1. Open the Automesh panel by pressing **F12**.
2. Go to the size and bias subpanel.
3. Click **surfs** > **displayed**.
4. In the element size field, enter `40`.
5. Set the elems to surf comp/elems to current comp toggle to **elems to current comp**.
6. Click **mesh**.

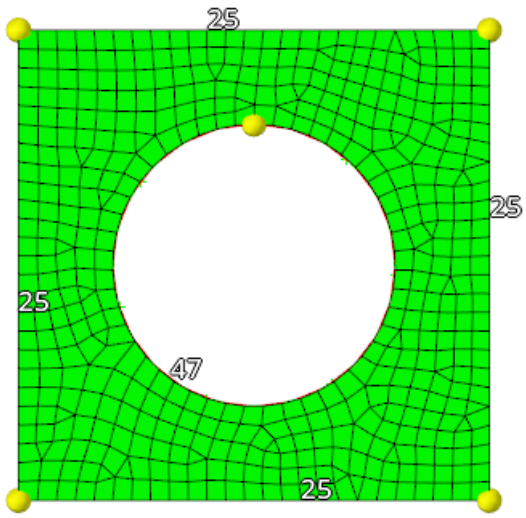


Figure 1118:

HyperMesh meshes the selected surfaces, and the meshing module opens.

7. Accept the mesh in the shells component by clicking **return**.
8. Close the Automesh panel by clicking **return**.

7.1.5 Apply Boundary Conditions to the Model

In this step, you will create collectors, constraints, and forces in order to apply boundary conditions to the model.

In this section, the model is constrained so that two of the four edges cannot move. A total lateral load of 1000N is applied at the edge of the hole so that all forces point in the positive z-direction.

1. Create collectors.

   Load collectors are created before boundary condition and loads. These load collectors are used for boundary conditions and loads.

   a) In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.

      A new load collector opens in the Entity Editor.

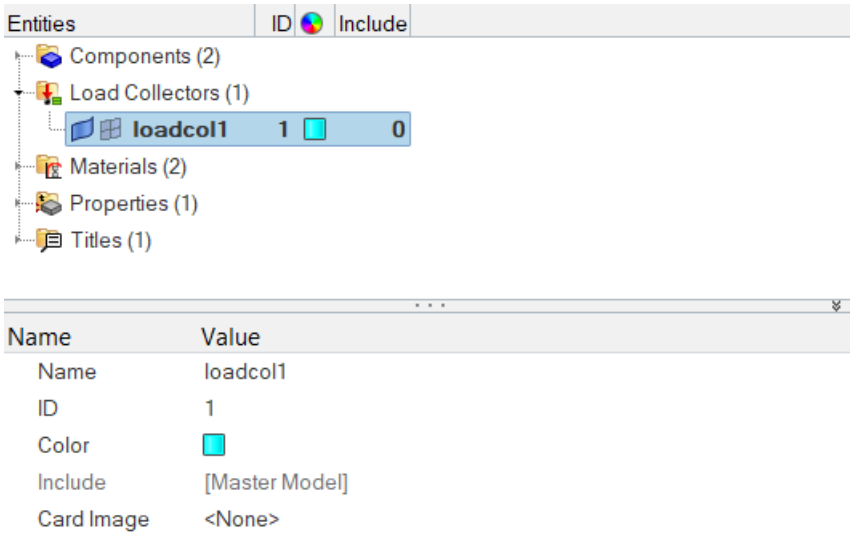


Figure 1119:

b) For Name, enter `spcs`.
c) Click the **Color** icon and select a color to display the load collector.
d) Repeat step 1.a.
e) For Name, enter `forces`.

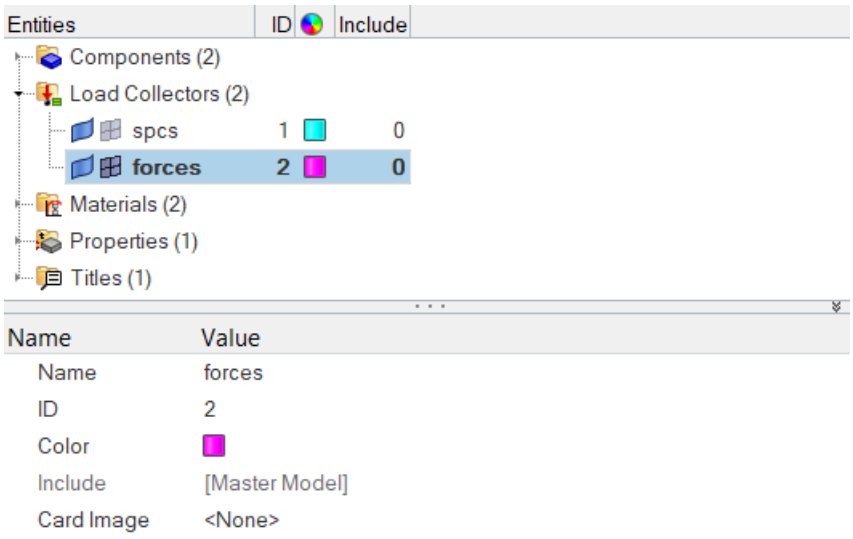


Figure 1120:

2. Create constraints.

a) In the Model Browser, Load Collector folder, right-click on **spcs** and select **Make Current** from the context menu.

b) Open the Constraints panel by clicking **BCs** > **Create** > **Constraints** from the menu bar.

c) Go to the create subpanel.

d) Set the entity selector to **nodes**.

e) Click **nodes** > **by window**.

f) Select the **exterior** checkbox.

g) Draw a box around all of the displayed nodes while excluding the four corners of the model.

> **Tip:** Since each side of the box must be drawn individually, draw the box by selecting each corner as indicated in Figure 1121. Select the top, right corner first and continue clockwise.

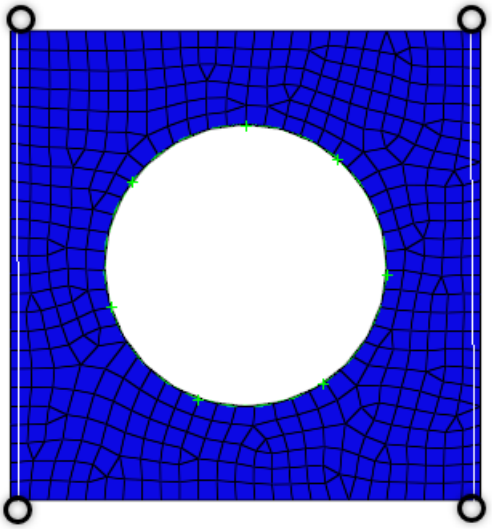


Figure 1121:

h) Click **select entities**.

HyperMesh selects all of the nodes outside of the window you drew.

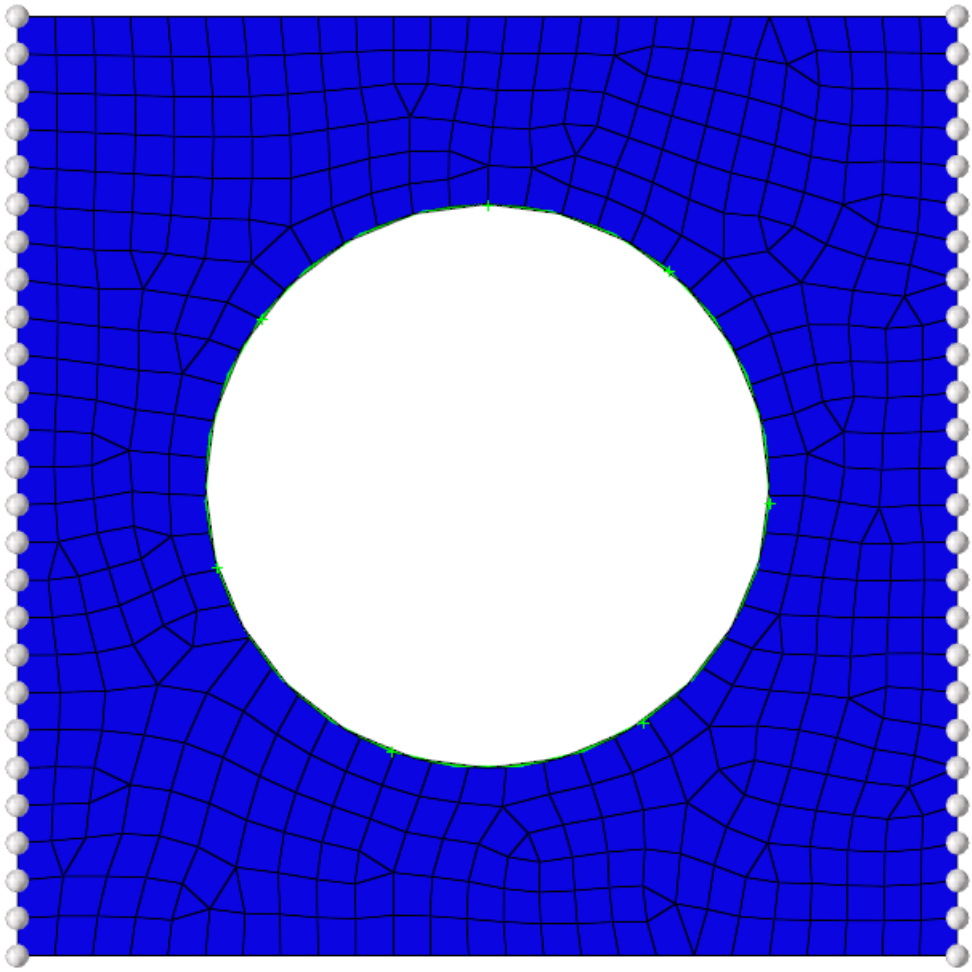


Figure 1122:

i) Select all of the **dof** (degree of freedom) checkboxes.

> **Note:** Dofs that are checked are constrained. Dofs 1, 2, and 3 are x, y, and z translation degrees of freedom, and dofs 4, 5, and 6 are x, y, and z rotational degrees of freedom.



Figure 1123:

j) Apply these constraints to the selected nodes by clicking **create**.

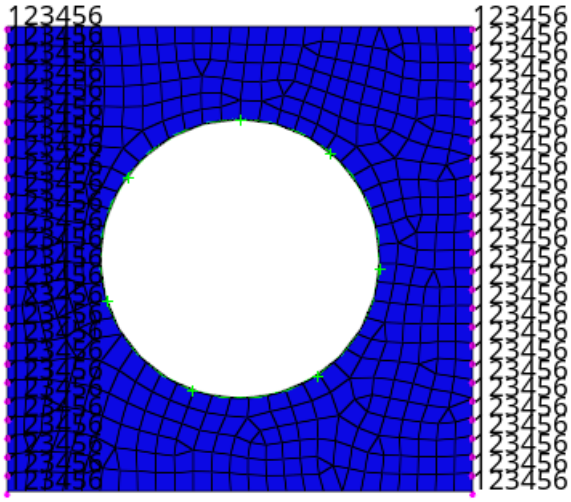


Figure 1124:

k) Click **return** to exit the Constraints panel.

3. Create forces on the nodes around the hole.

a) In the Model Browser, Load Collector folder, right-click on **forces** and select **Make Current** from the context menu.

b) Open the Forces panel by clicking **BCs** > **Create** > **Forces** from the menu bar.

c) Click **nodes** > **by path**.

d) Select all of the nodes around the hole of the model.

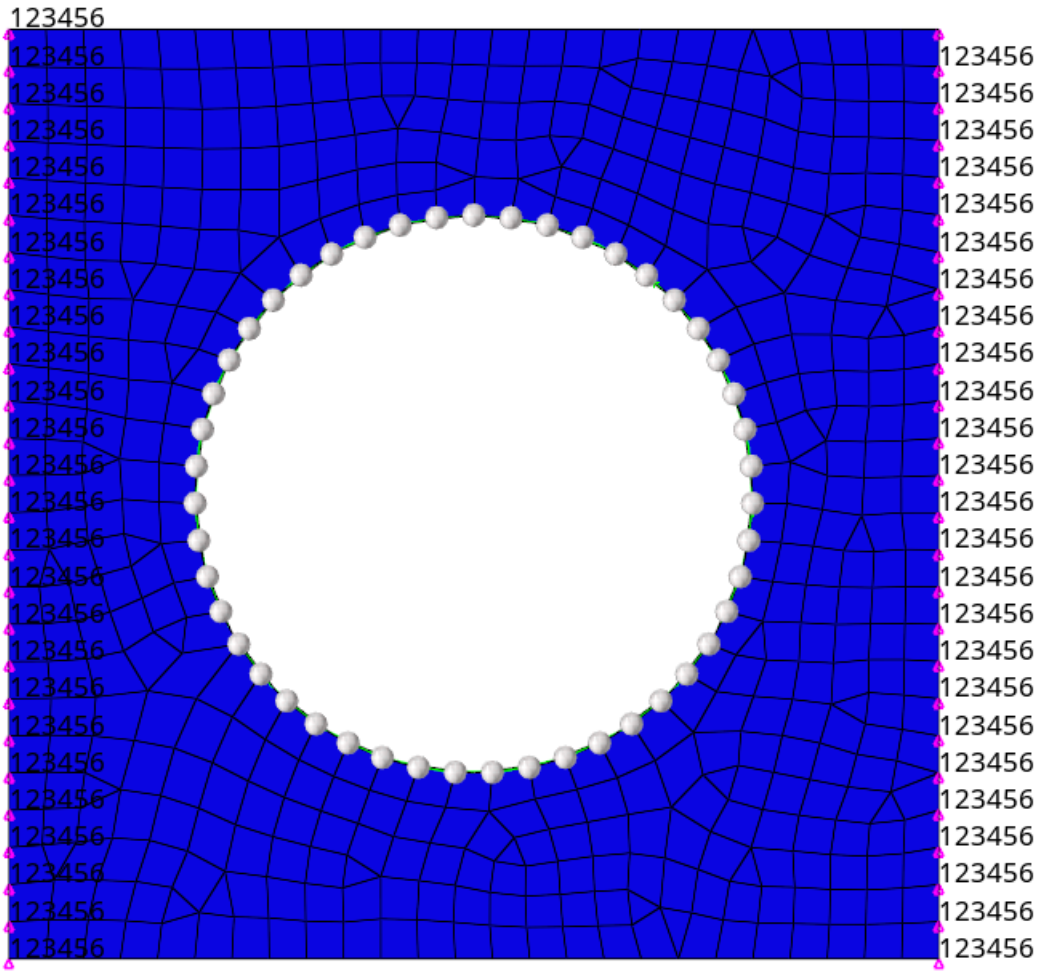


Figure 1125:

e) Click **nodes** > **save**.
f) Click **return**.
g) Open the Count panel by going to the Tool page and clicking **count**.
h) Go to the FE entities subpanel.

> **Note:** Nodes are automatically counted so that a calculation can be made to create a total force of 1000N.

i) Set the entity selector to **nodes**.
j) Retrieve the nodes you saved in the Forces panel by clicking **nodes** > **retrieve**.
k) Click **selected**.
HyperMesh counts the number of nodes around the hole.
l) Click **return**.
m) Open the Forces panel by clicking **BCs** > **Create** > **Forces** from the menu bar..
n) Click **nodes** > **retrieve**.

o) In the magnitude = field, enter `21.277` (this is 1000/47).

> **Note:** The total load on the nodes around the hole is 1000N.

p) Set the orientation selector to **z-axis**.

q) Click **create**.

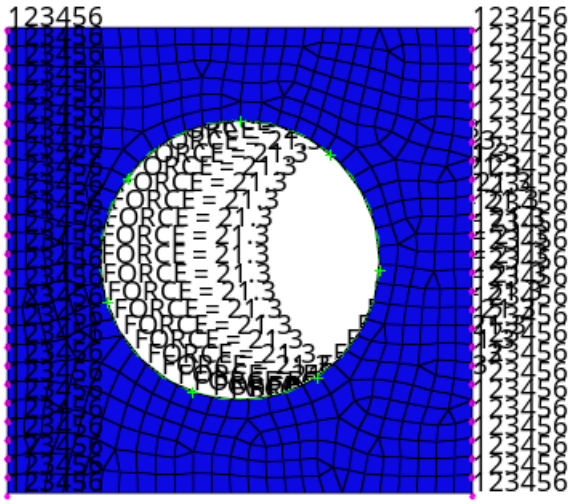


Figure 1126:

r) Click **return**.

7.1.6 Create a Nastran Subcase

In this step, you will creat a Nastran subcase (a load step in HyperMesh).

1. Create the load step.
   a) In the Model Browser, right-click and select **Create** > **Load Step** from the context menu.
   A new loadstep opens in the Entity Editor.

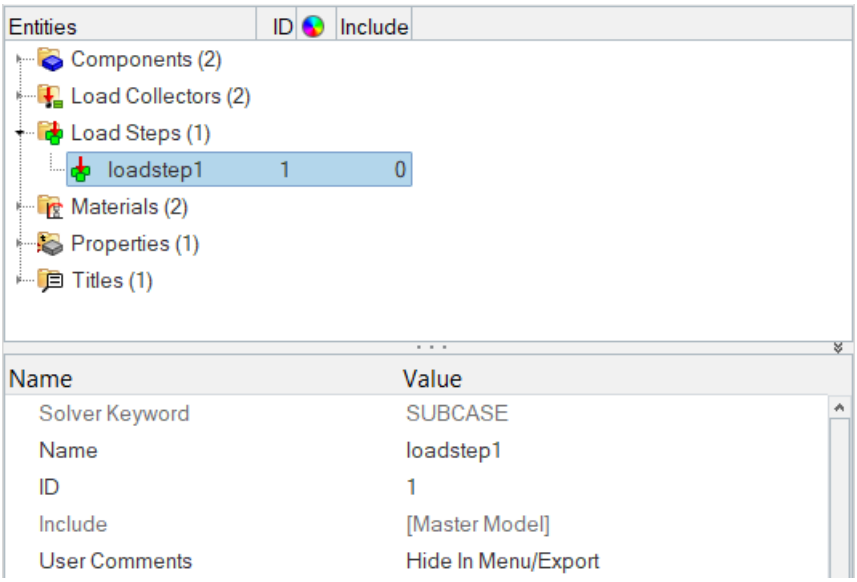


Figure 1127:

b) For Name, enter `lateral force`.

c) For SPC, click **Unspecified** > **Loadcol**.

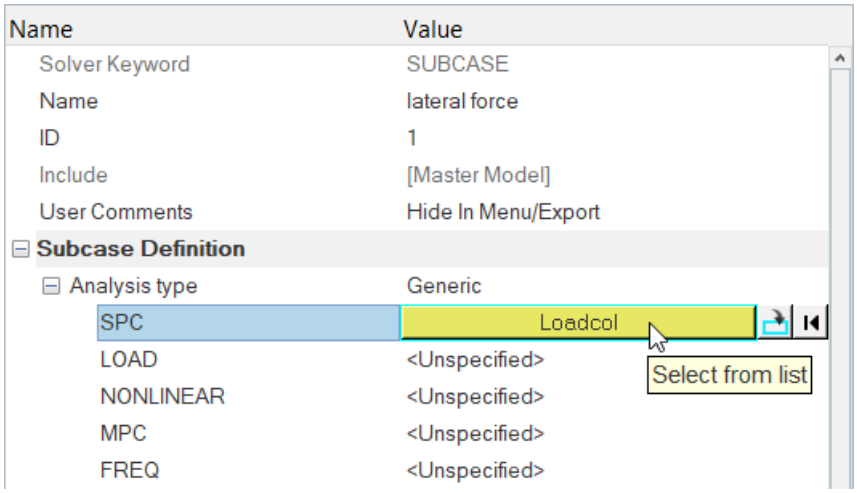


Figure 1128:

d) In the **Select Loadcol** dialog, select **spcs** and then click **OK**.

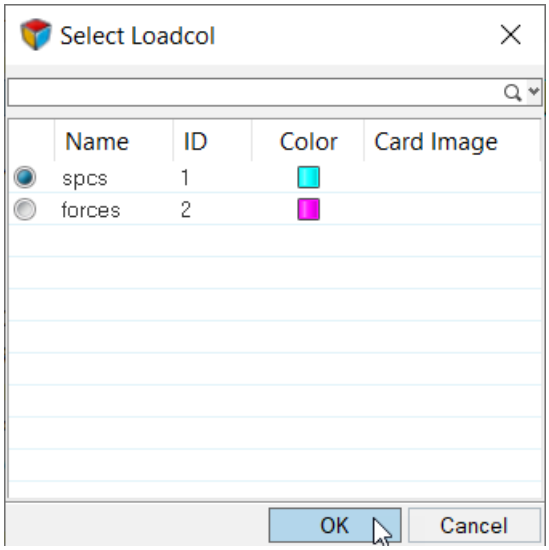


Figure 1129:

e) For LOAD, click **Unspecified** > **Loadcol**.
f) In the **Select Loadcol** dialog, select **forces** and then click **OK**.
g) Under SUBCASE OPTIONS, select the **OUTPUT** checkbox.
h) Under OUTPUT, select the **DISPLACEMENT** checkbox.
i) Under OUTPUT, select the **STRESS** checkbox.

2. Create control cards.
a) From the menu bar, click **Setup** > **Create** > **Control Cards**.
b) In the card image panel, click **SOL**.
c) Set Analysis to **Statics**.



Figure 1130:

d) Click **return**.
e) Click **PARAM**.
f) Select the **AUTOSPC** and **POST** checkboxes.
g) In the card edit field, click **POST_V1** and enter `-2` in the editable field.

Note: This option specifies that an op2 file should be created.



Figure 1131:

h) Click **return** twice.

7.1.7 Write the Nastran Input Deck

In this step, you will write the Nastran input deck.

1. Write your file.
   a) From the menu bar, click **File** > **Export** > **Solver Deck**.
   The Export - Solver Deck tab opens.
   b) Set File type to **Nastran**.
   c) In the File field, navigate to your working directory an save the file as `plate_hole.dat`.
   d) Click **Export**.
   HyperMesh writes your HyperMesh database as a Nastran ASCII input deck.
   e) Click **Close**.
2. Save your file and exit HyperMesh.
   a) From the menu bar, click **File** > **Save As** > **Model**.
   b) In the **Save Model As** dialog, navigate to your working directory and save the file as `plate_hole_new.hm`.

7.1.8 View the Results

In this step, you will view the results and the deformed shape.

After running Nastran, the result file `plate_hole.op2` is created. This file contains displacement and stress results for your linear static analysis. This section describes how to view those results in HyperMesh.

1. Add a HyperView page to the session and load the fem and op2 files.
   a) On the Page Controls toolbar, click  .

b) Set the Client Selector toolbar to **HyperView**.

c) From the menu bar, click **File** > **Open** > **Model**.

The Load Model panel opens.

d) In the Load model field, navigate to your working directory and open the `plate_hold.dat` file.

e) In the Load results field, navigate to your working directory and open the `plate_hold.op2` file.

f) Click **Apply**.

HyperView loads the model and results.

2. View a deformed shape.

a) Open the Deformed panel by clicking **Results** > **Plot** > **Deformed** from the menu bar.

b) Set Result type to **Displacement**.

c) Set Scale to **Model units**.

d) Set Type to **Uniform**.

e) For Value enter `25`.

f) Under Undeformed shape, set Show to **Wireframe**.

g) View a deformed plot of your model overlaid on the original, undeformed mesh by clicking **Apply**.

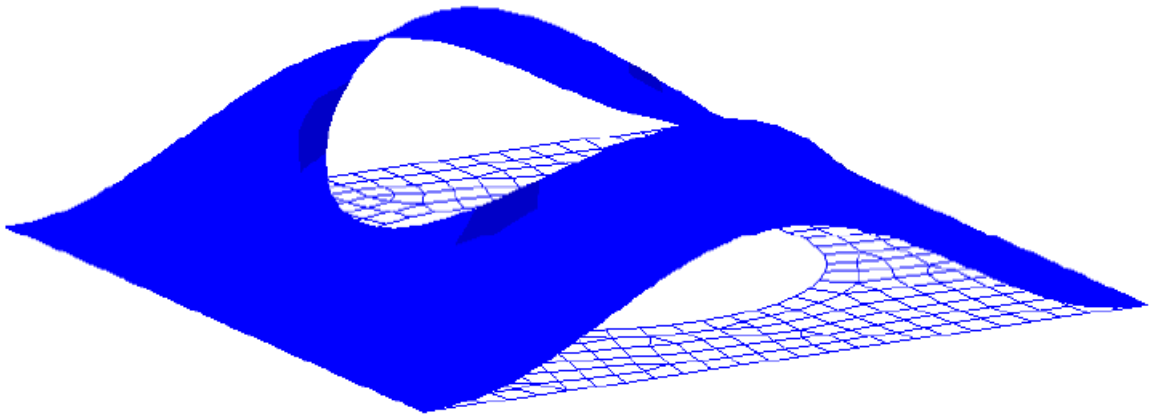


Figure 1132:

3. View a contour plot of stresses and displacements.

a) Open the Contour panel by clicking **Results** > **Plot** > **Contour** from the menu bar.

b) Set Result type to **Displacement (v)**.

c) On the Standard Views toolbar, click  .

d) Click **Apply**.

e) Set Result type to **Stress (t)**.

f) Set Averaging method to **Simple**.

g) Click **Apply**.

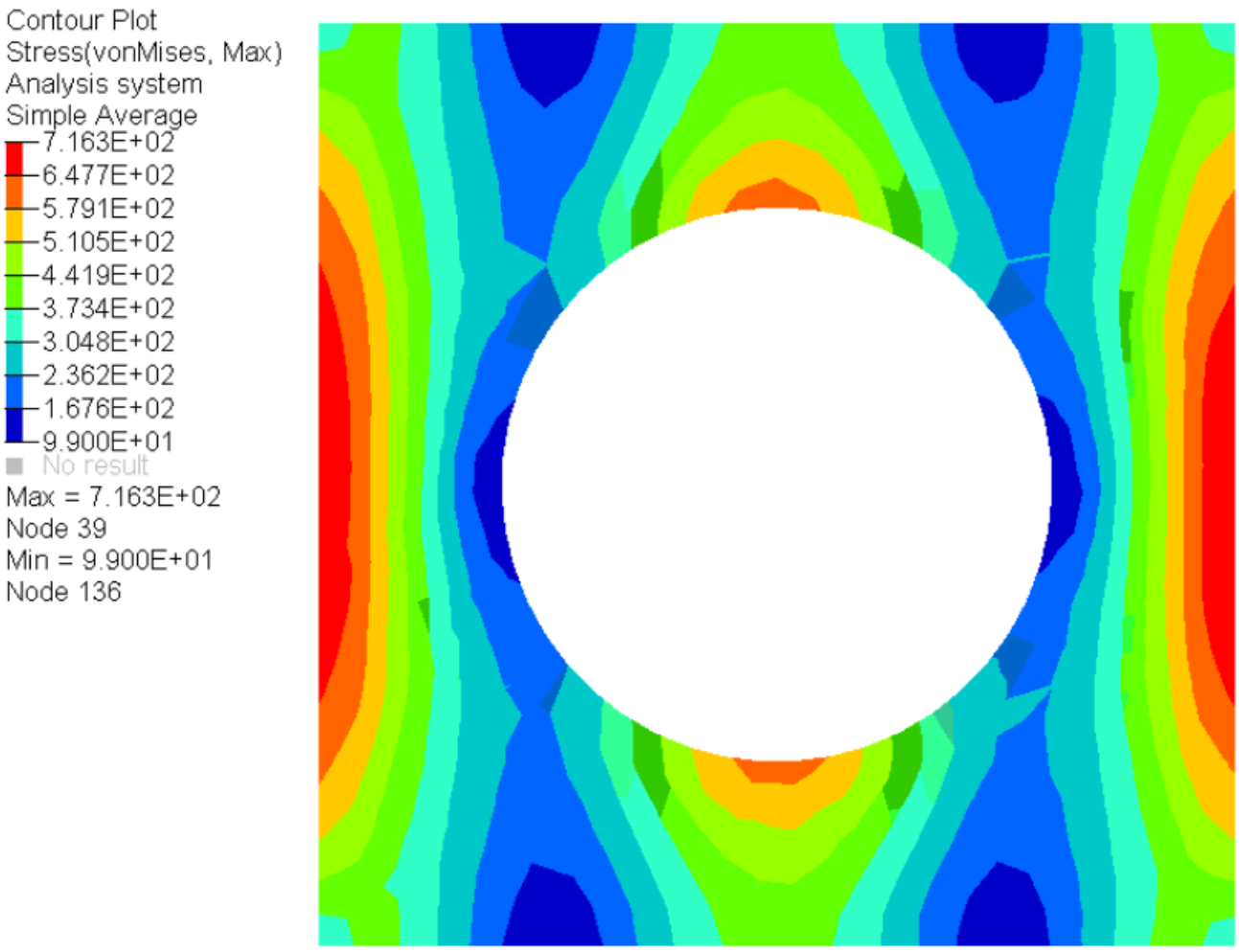


Figure 1133:

PAM-CRASH 2G

8

PAM-CRASH 2G tutorials.

This chapter covers the following:

- 8.1 HM-4700: Using the PAM-CRASH 2G Interface in HyperMesh (p. 905)

8.1 HM-4700: Using the PAM-CRASH 2G Interface in HyperMesh

In this tutorial, you will: create Control Cards, boundary conditions, time histories, functions, and sensor cards; define materials, HyperMesh groups, and rigid walls; assign element types; and export a PAM-CRASH 2G data deck.

Before you begin, copy the `rail-dyna.hm` file from `<hm.zip>/interfaces/pamcrash/` to your working directory.

Also, it is recommended that you complete the introductory tutorial, HM-1000: Getting Started with HyperMesh.

8.1.1 Load the PAM-CRASH User Profile

In this step, you will load the PAM-CRASH profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **Pamcrash**.
3. Set the subprofile to **Pamcrash2G2012**.

   Selecting the PAM-CRASH user profile and the Pamcrash2G2012 subprofile sets the FE input reader to PAM-CRASH 2G and loads the PAM-CRASH 2G 2012 FE output template. It also loads the PAM-CRASH 2G Utility menu, which contains numerous tools specific to this interface. The graphical user interface is tailored to PAM-CRASH 2G users.



Figure 1134:

8.1.2 Retrieve and View the Model File

In this step, you will open the model file and view it in HyperMesh.

The `leg_geom.hm` file contains IGES lines representing the layout of the leg and coordinate systems for bodies and joints.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
2. In the **Open Model** dialog, open the `rail-dyna.hm` file.

The model appears in the graphics area.

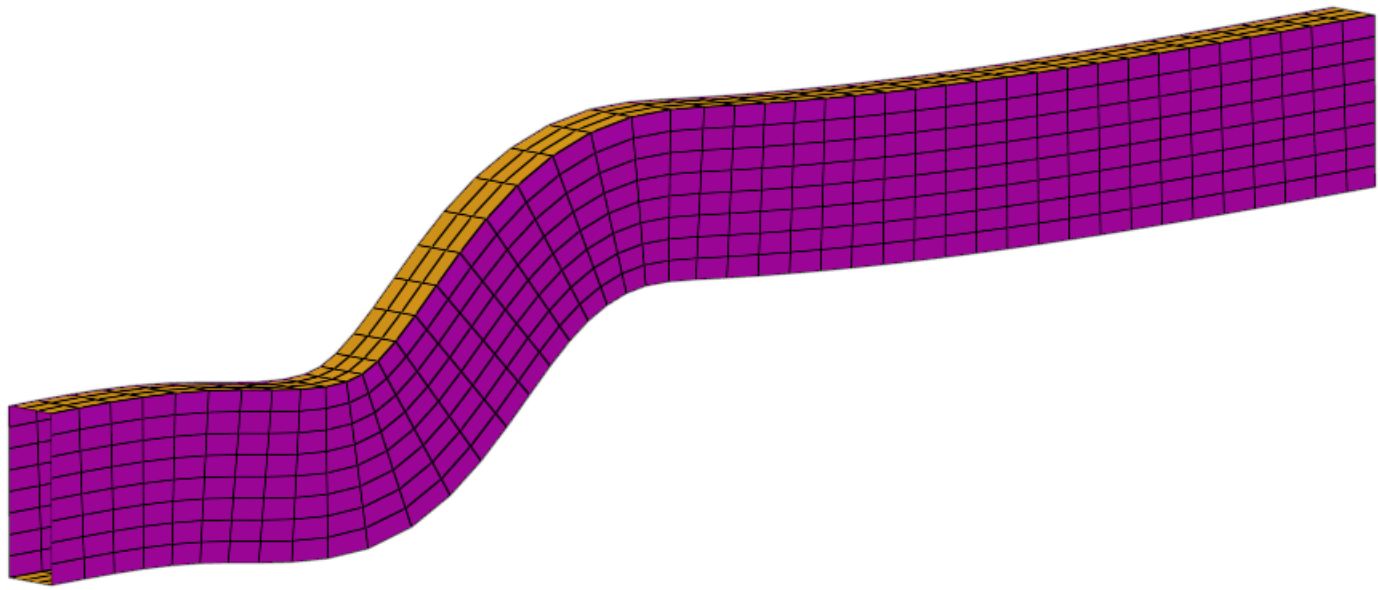


Figure 1135:

8.1.3 Create Control Cards

In this step, you will create Control Cards for PAM-CRASH 2G.

The settings of the Control Cards influence the default values for defining materials. PAM-CRASH 2G decks cannot be executed without error if Control Cards are undefined.

1. Define the title card.
   a) In the Solver Browser, right-click and select **Create** > **CONTROL CARDS** > **TITLE** from the context menu.
   A new Control Card opens in the Entity Editor.
   b) In the TITLE field, enter `This is my first PAM-CRASH example`.

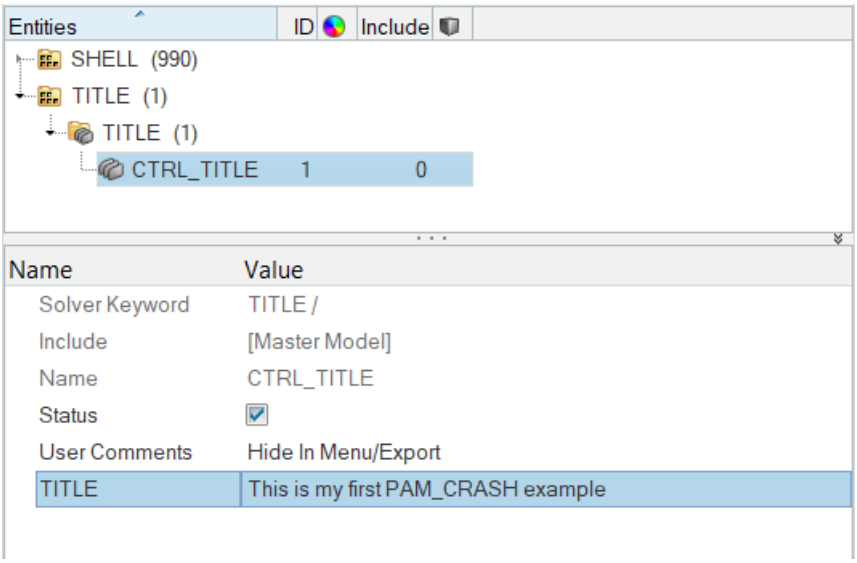


Figure 1136:

2. Create and define the output Control Cards.

a) In the Solver Browser, right-click and select **Create** > **CONTROL CARDS** > **OCTRL** from the context menu.

The OCTRL Control Card defines output control parameters.

A new Control Card opens in the Entity Editor.

b) For VALUE, under THPOUTPUT_OPT, enter `0.005`.

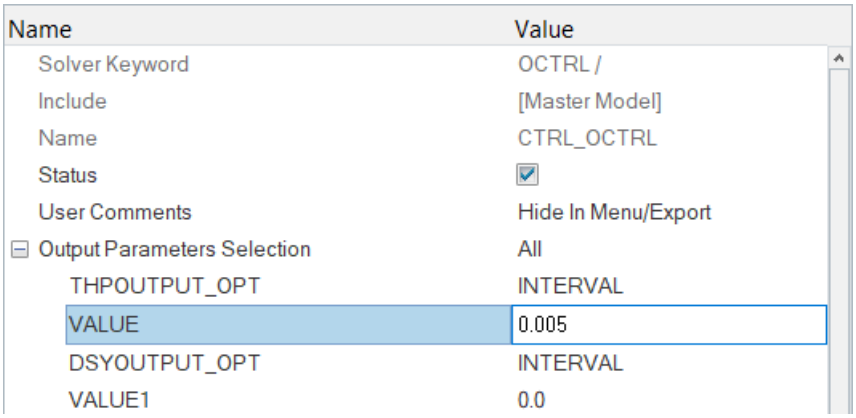


Figure 1137:

c) For VALUE, under DSYOUTPUT_OPT, enter `0.005`.

d) In the Solver Browser, right-click and select **Create** > **CONTROL CARDS** > **RUNEND** from the context menu.

The RUNEND Control Card defines end of run parameters.

A new Control Card opens in the Entity Editor.

e) For TIO2, enter `0.06`.

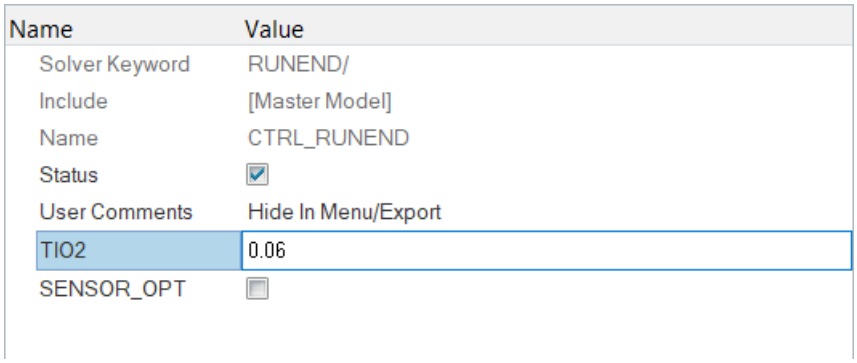


Figure 1138:

3. Define the file optional keyword.

a) In the Solver Browser, right-click and select **Create** > **CONTROL CARDS** > **FILE** from the context menu.
A new Control Card opens in the Entity Editor.

b) For NAME, enter `rail-dyna.hm`.

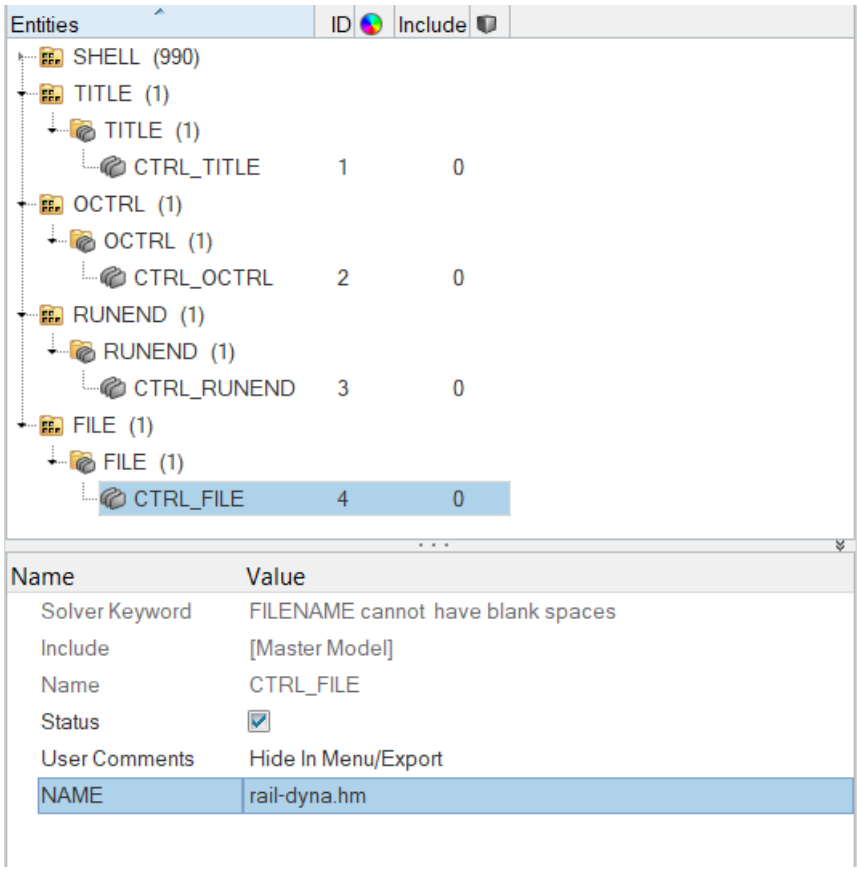


Figure 1139:

4. Define the time step optional keyword.

a) In the Solver Browser, right-click and select **Create** > **CONTROL CARDS** > **TCTRL** from the context menu.
A new Control Card opens in the Entity Editor.
b) Select the **SHELL_TIMESTEP_OPT** checkbox.
c) Set TIMESTEP_OPT_LS to **LARGE**.
d) Set TIMESTEP_OPT_BN to **BEND**.

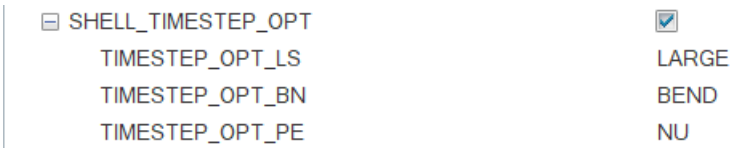


Figure 1140:

8.1.4 Assign Element Types

In this step, you will assign element types for PAM-CRASH 2G.

Depending on the analysis requirement, the HyperMesh basic element type can be changed.

For example, a quad4 can be a SHELL or a MEMBR element. The tria3 element can be a TRIA_C, SHELL, or MEMBR element. The tetra4, the penta6, and the hexa8 elements define the SOLID elements of PAM-CRASH. Properties can be added for the selected element type using Control Cards.

1. Open the Element Type panel by clicking **Mesh** > **Assign** > **Element Type** from the menu bar.
2. Open the **2D & 3D** subpanel.
3. Click **quad4 =** and select **SHELL**.
4. Click **elems** > **all**.
5. Click **update**.
6. Click **return** to exit the panel.

8.1.5 Define a Material and /PART Cards

In this step, you will define a material and /PART cards for PAM-CRASH 2G.

1. Rename the tmp component.
   a) In the Model Browser, Component folder, select the **tmp** component.

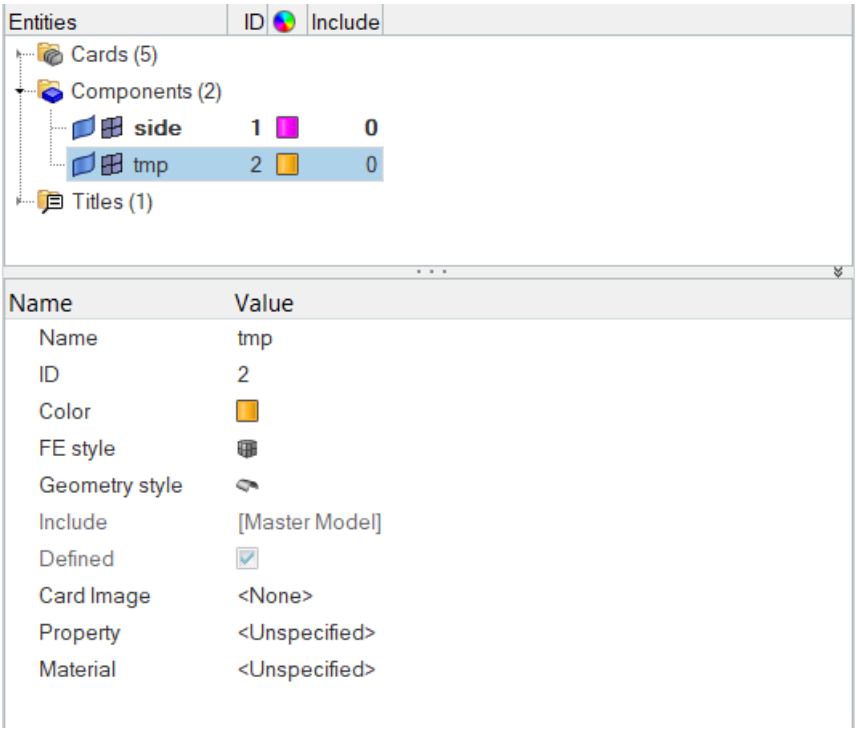


Figure 1141:

The Entity Editor opens, and displays the component's card data.

b) For Name, enter `topbottom`.

2. Create and define a type 102 material .

a) In the Model Browser, right-click and select **Create** > **Material** from the context menu.
A new material opens in the Entity Editor.

b) For Name, enter `new mat`.

c) Set card image to **MAT_2D**.

> **Note:** The template provides MAT_1D, MAT_2D, and MAT_3D dictionaries. Material types from 200 to 230 are defined with MAT_1D. Materials types from 100 to 151 are defined with MAT_2D. Material types from 1 to 41 are defined with MAT_3D. Use the card previewer to switch the material type.

d) Set Material Type to **Type 102**.

e) For RHO, enter `7.85e-9`.

f) For E, enter `20000`.

g) For SIGMay, enter `250`.

h) For NU, enter `0.3`.

3. Assign a material and thickness to the side and topbottom collectors.

a) In the Model Browser, Component folder, select **side** and **topbottom**.
The Entity Editor opens and displays the selected component's common card data.

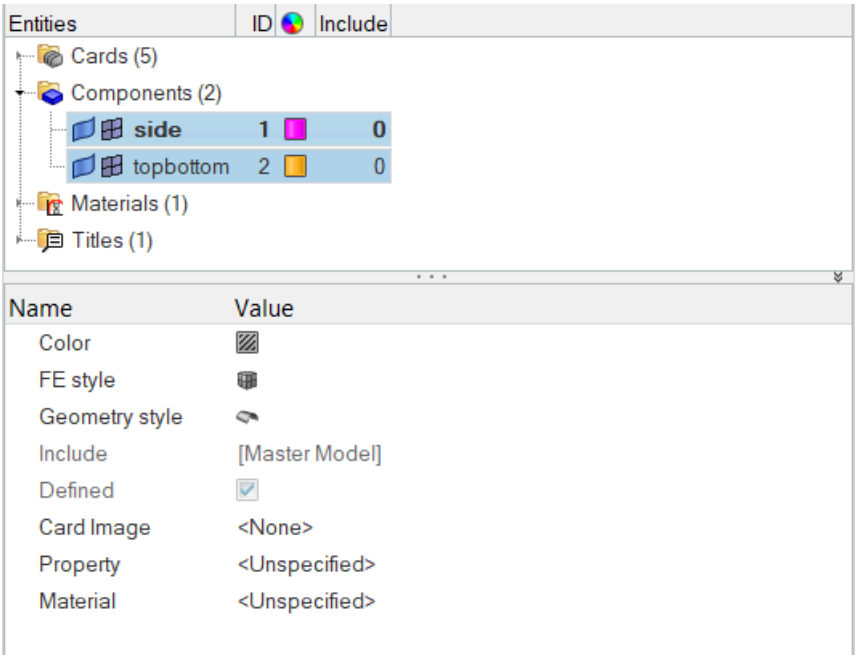


Figure 1142:

b) Set card image to **Part_2D**.

c) For Material, click **Unspecified** > **Material**.

d) In the **Select Material** dialog, select **new mat** and then click **OK**.
HyperMesh assigns the new mat material to the selected components.

e) For h (thickness), enter `2.5`.

8.1.6 Define HyperMesh Groups

In this step, you will define HyperMesh groups: sliding interface for PAM-CRASH 2G.

This step describes how to define a type 36 self contacting sliding interface. A second interface is defined only for tutorial purposes.

1. Define the group.

a) In the Solver Browser, right-click and select **Create** > **CONTACT** > **CNTAC36** from the context menu.
A new group opens in the Entity Editor.

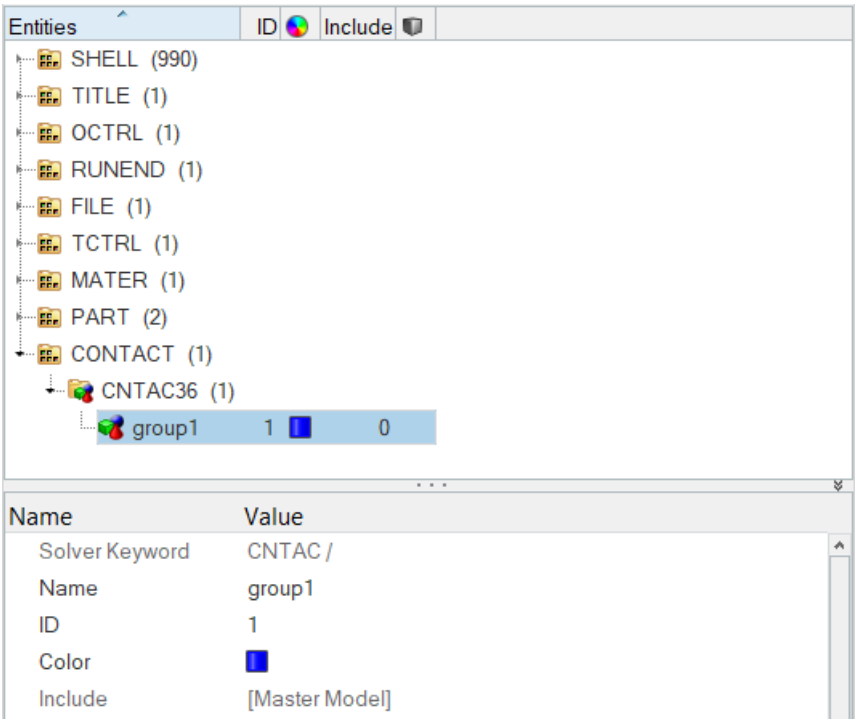


Figure 1143:

b) For Name, enter `self_impact`.

c) Select a new color for the group.

d) For SLFACM, enter `1.0`.

2. Add the slave components.

In this step, the Entity Editor should be open for the self_impact group.

a) Click **Slave entity IDs**.

b) Set the entity selector to **Components**.

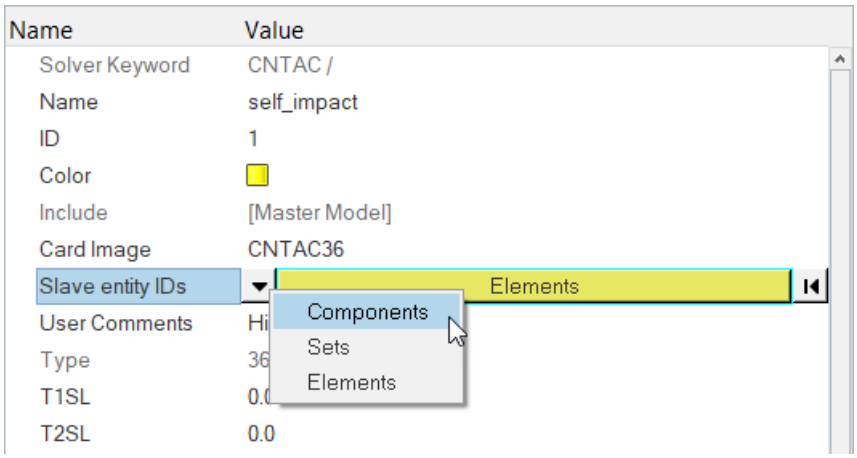


Figure 1144:

c) Click **Components**.

d) In the **Select Components** dialog, select **side** and **topbottom** and then click **OK**.

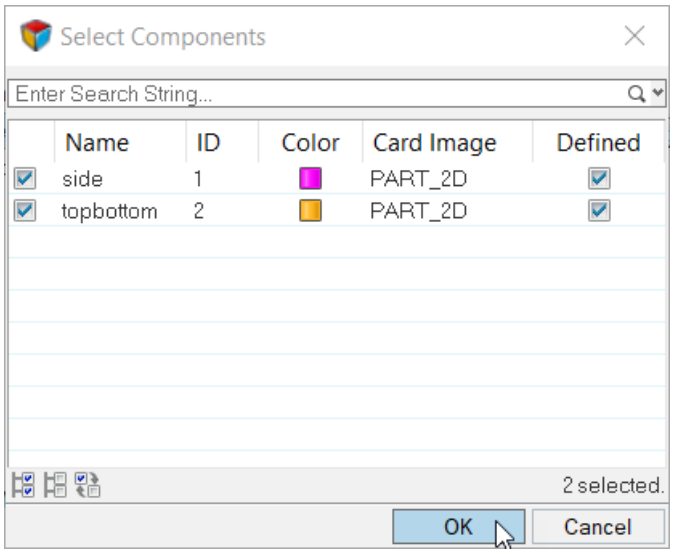


Figure 1145:

3. Define an additional contact.

This procedure explains how to define a type 34 master slave (element - node) contact.

a) In the Solver Browser, right-click and select **Create** > **CONTACT** > **CNTAC34** from the context menu.

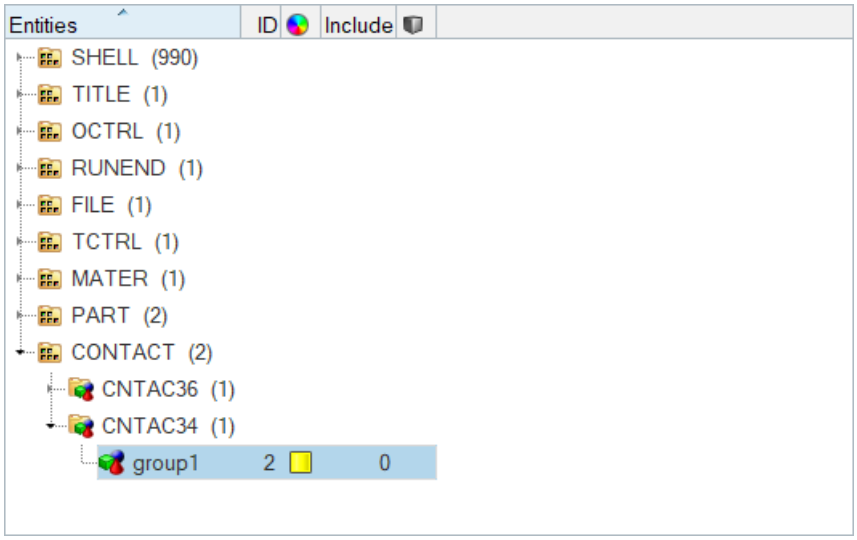


Figure 1146:

A new group opens in the Entity Editor.

b) For Name, enter `masterslave`.

c) Select a new color for the group.

d) For SLFACM, enter `1.0`.

4. Add the master elements and slave nodes.

In this step, the Entity Editor should be open for the masterslave group.

a) For Master entity IDs, click **0 Elements** > **Elements**.

b) In the panel area, set the switch to **elems**.

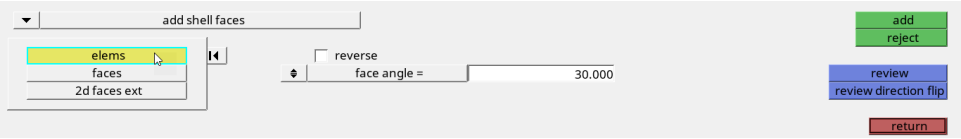


Figure 1147:

c) Using the elems selector, select two elements as seen in Figure 1148.

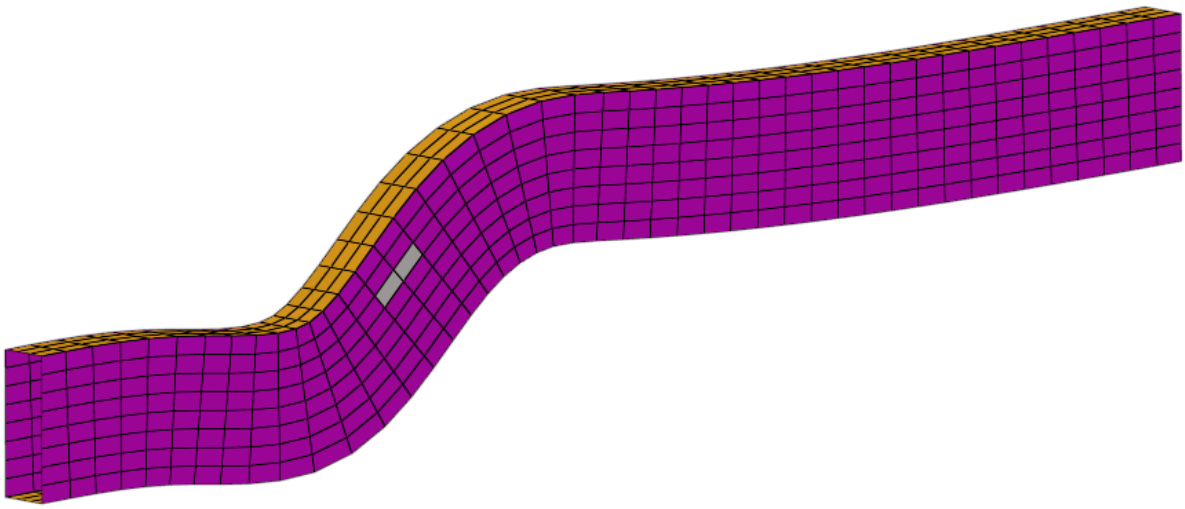


Figure 1148:

d) Click **add**.

e) Click **return**.

f) For Slave entity IDs, click **0 Nodes** > **Nodes**.

g) In the panel area, set the switch to **nodes**.

h) Using the nodes selector, select two nodes as seen in Figure 1149.

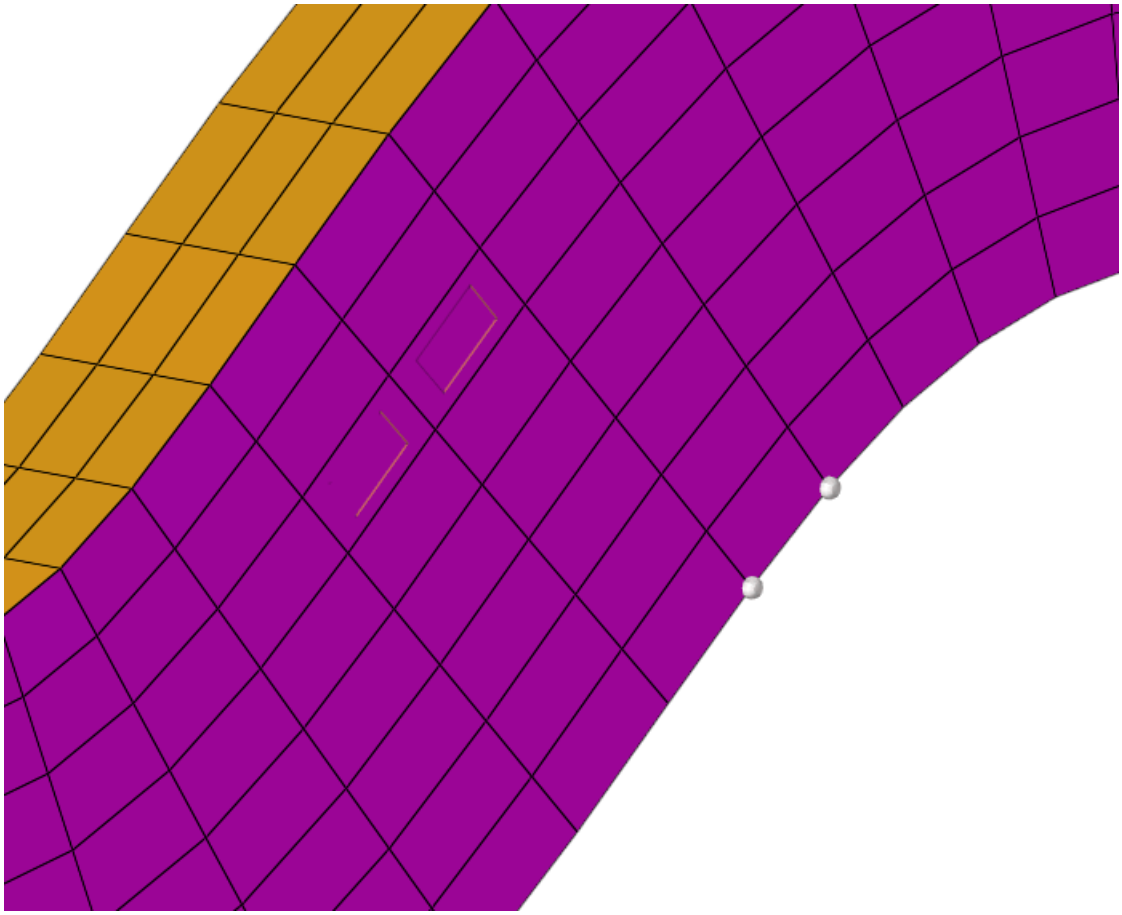


Figure 1149:

i) Click **add**.

j) Click **return**.

The master elements are marked with square indents, and the slave nodes displays as seen in Figure 1150.

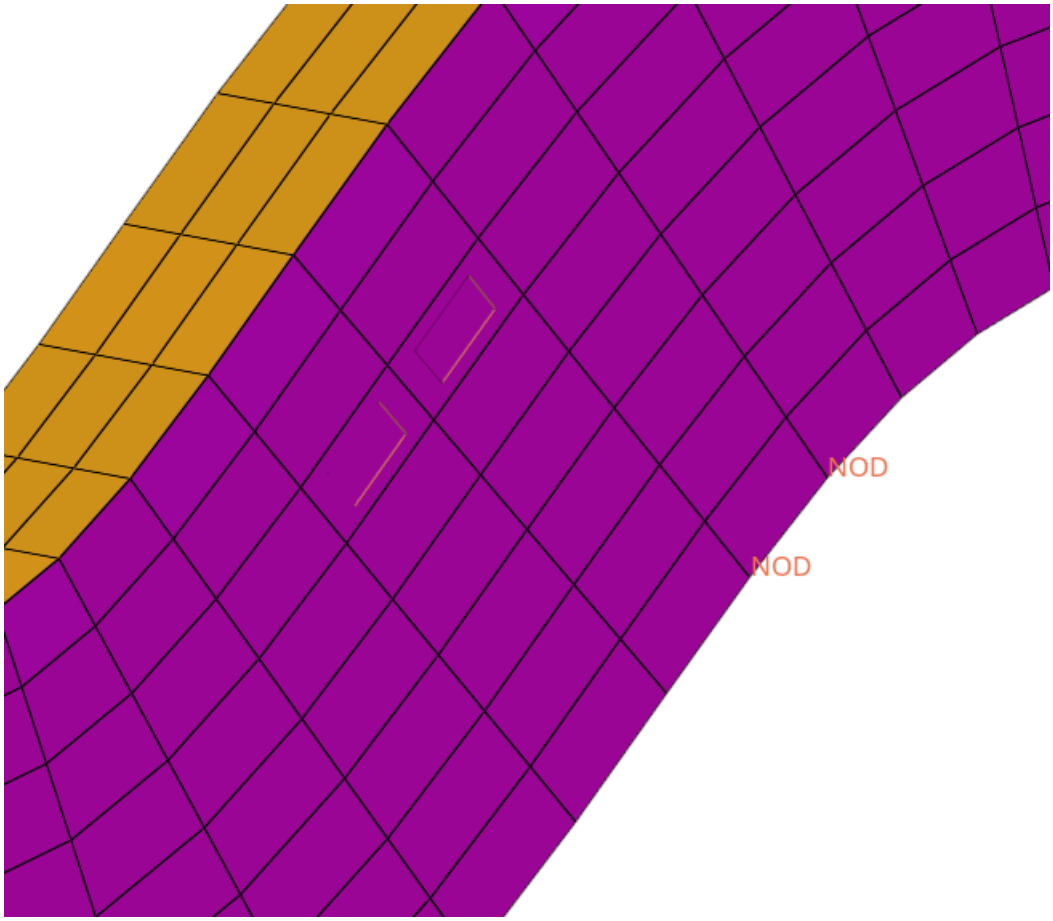


Figure 1150:

8.1.7 Define a Rigid Wall

In this step, you will define a rigid wall for PAM-CRASH 2G.

This step explains how to define a type 4 infinite rigid wall with a base node at -1.00, 0.0, 0.0.

1. Create a base node for the rigid wall.
   a) From the menu bar, click **Geometry** > **Create** > **Nodes** > **XYZ**.
   b) For x, enter `-1.0`.
   c) For y, enter `0.0`.
   d) For z, enter `0.0`.
   e) Click **create**.
   HyperMesh generates the base node as seen in Figure 1151

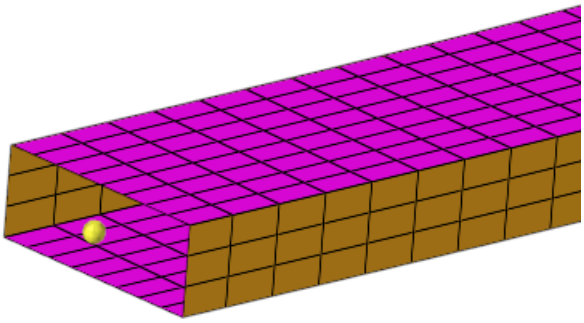


Figure 1151:

f) Click **return** to exit the panel.

2. Create and define the rigid wall card.

a) From the menu bar, click **BCs** > **Create** > **Rigid Walls**.

b) In the Rigid Walls panel, Name field, enter `rwall1`.

c) Click **type =** and select **RWALL**.

d) Select a color.

e) In the size =, enter `100`.
The display size of the rigid wall is specified.



Figure 1152:

f) Click **create**.

3. Define rigid wall geometry.

a) Open the **geom** subpanel.

b) Double-click **name=**, and select **rwall1**.

c) Set the switch after shape to **plane**.

d) Set the toggle after shape to **infinite**.

e) Under normal vector, set the switch to **vectors**.

f) Click the second switch and select **x-axis**.



Figure 1153:

g) Using the base node selector, select the node you created in step 1.

h) Click **update**.
The rigid wall displays as seen in Figure 1154.

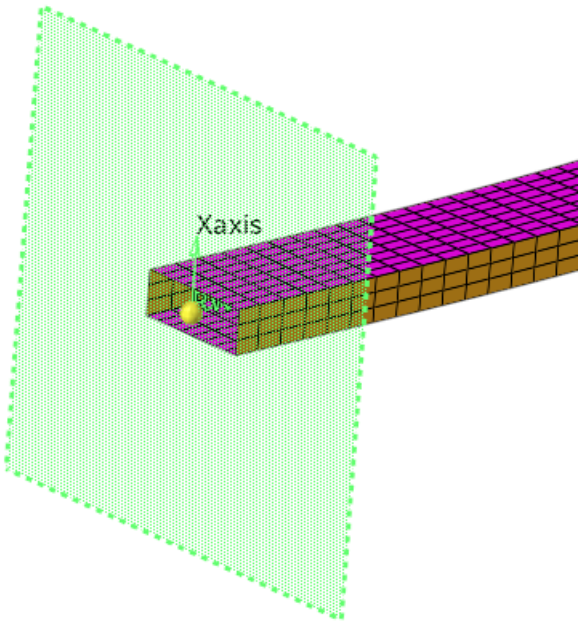


Figure 1154:

4. Add secondary nodes for the rigid wall.

a) Go to the add subpanel.

b) Set slaves to **nodes**.

c) Click **nodes** > **by id**.

d) In the id field, enter `1-21` and then press **ENTER**.
21 nodes at the interface of the rail and the rigid wall highlight. One of the nodes is not selected.

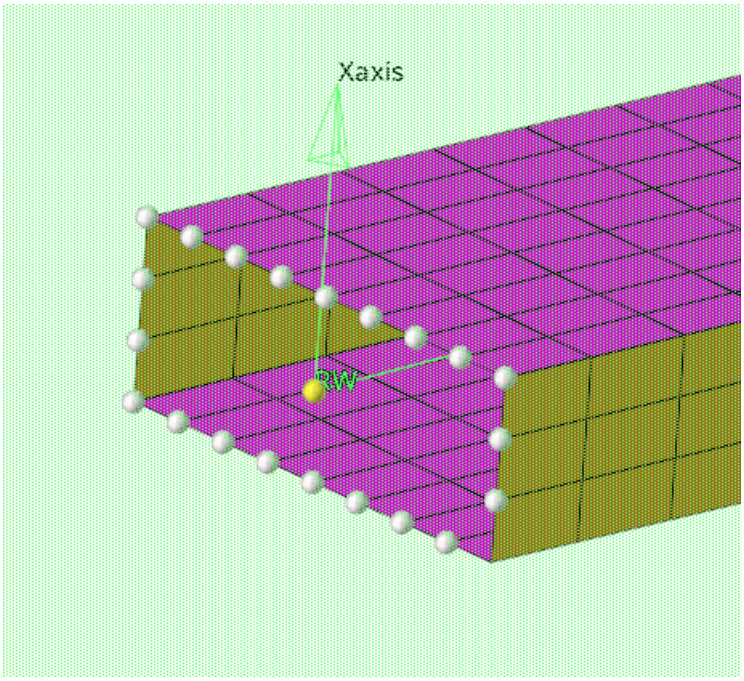


Figure 1155:

e) Click the unhighlighted node.

> **Note:** You can also click **nodes** > **id** and enter `1012` in the id field.

f) Click **add**.
The selected nodes are now set as secondarys.

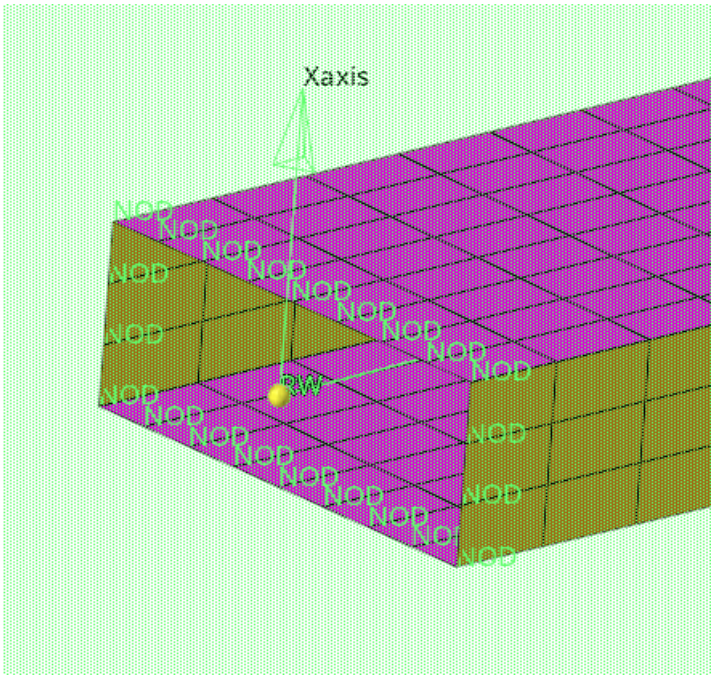


Figure 1156:

5. Add motion to the rigid wall.
   a) Open the **motion** subpanel.
   b) Set the switch below name to **components**.
   c) In the x comp field, enter `1.0`.
   d) Set type of motion to **velocity**.

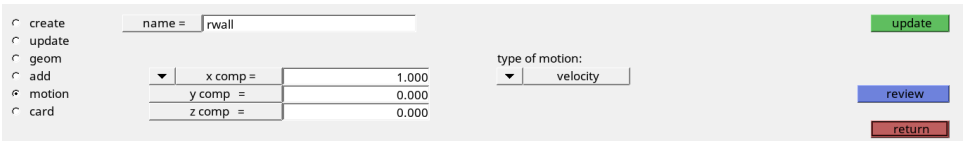


Figure 1157:

   e) Click **update**.
   f) Click **return** to exit the panel.

8.1.8 Define Attributes in the Card Previewer

In this step, you will define attributes in the card previewer.

1. In the Model Browser, Group folder, click **rwall1**.
   The Entity Editor opens and displays the group's card data.
2. Set Friction type flag to **No Sliding**.
3. Set Rigid Wall Descriptor – Plane Type to **Type 4**.

> **Tip:** The parameters in the Entity Editor are according to the definitions made. It is now possible to define the mass and the initial velocity for moving rigid wall with finite mass.

4. For mRW, enter `1`.
5. For VINIT, enter `2000`.

8.1.9 Create Boundary Conditions

In this step, you will create model boundary conditions for PAM-CRASH 2G.

1. Create a load collector.
   a) In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.

      The new load collector becomes the current collector. Any new loads created will be placed in this collector.

      A new load collector opens in the Entity Editor.
   b) For Name, enter `boundary conditions`.
   c) Select a **card image**.
   d) Select a color.
2. Specify the constraints.
   a) Open the Constraints panel by clicking **BCs** > **Create** > **Constraints** from the menu bar.
   b) Click **load types**, and select **BOUNC**.

      All constraints that are now created will be displacement boundary conditions.
   c) Click **nodes** > **by id**.
   d) In the id field, enter `990-1011`.
   e) Press **ENTER**.
   22 nodes highlight.

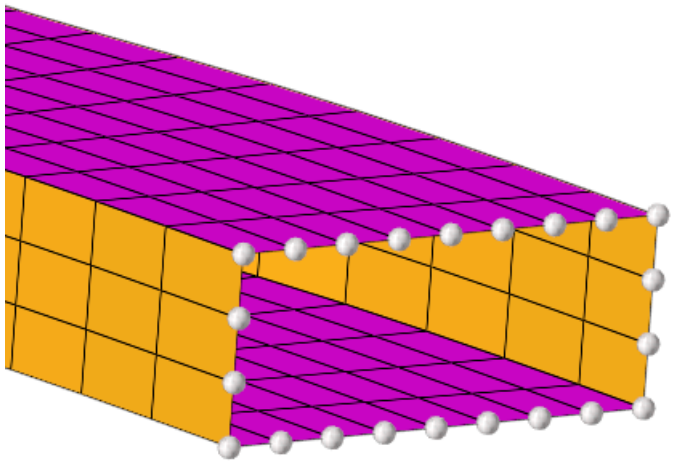


Figure 1158:

f) In the relative size= field, enter `10`.

g) Click **create**.
Constraints are now added to the selected nodes.

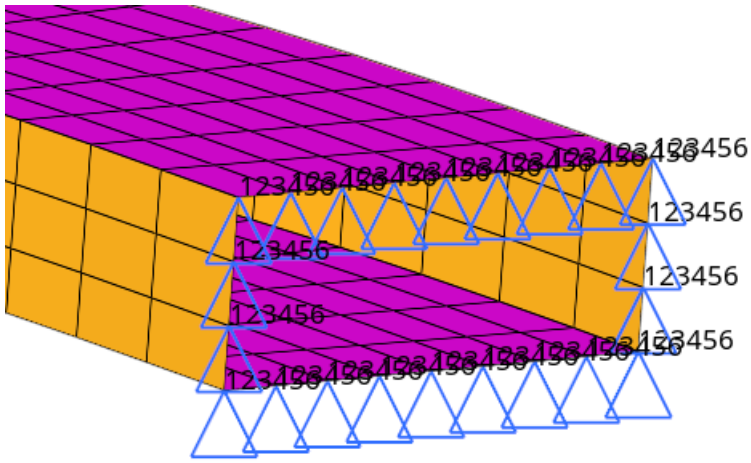


Figure 1159:

h) Click **return** to exit the panel.

8.1.10 Create Time Histories

In this step, you will create time histories for PAM-CRASH 2G.

For PAM-CRASH 2G, time histories may be defined for nodes, elements, and local coordinate systems. For this step, you will only create time histories for nodes and elements. The operation is the same for any type of time history that is created.

1. Create a node time history card.
   a) In the Model Browser, Load Collector folder, right-click on **Boundary Conditions** and select **Hide** from the context menu.
   The display of loads turns off.
   b) From the menu bar, click **Setup** > **Create** > **Output Blocks**.
   c) In the name field, enter `node_thp`.
   d) Using the nodes select, select a few nodes.
   e) Click **create**.
   The time history for nodes is created.
2. Create an element time history card.
   a) In the name field, enter `elem_thp`.
   b) Set the entity selector to **elems**.
   c) Select a few elements.
   d) Click **create**.
3. Review time histories entities.
   a) Click **review**.
   b) Select **elem_thp**.
   The entities associated with this time history highlight.
   c) Click **return** to exit the panel.
4. View the time history card image.
   a) In the Model Browser, Output Blocks folder, right-click on **elem_thp** and click **Card Edit**.
   The time history card is displayed as it will look in the output.
   b) Click **return** twice to exit the panel.

8.1.11 Create a Function

In this step, you will generate a curve and a function.

This step describes how to generate curves, which corresponds to the function cards /FUNCT in PAM-CRASH 2G. This curve should serve as a function for a logical sensor switching on and off. At time=0, the sensor is on. At time=0.01, the sensor is switched off.

1. From the menu bar, click **XY Plots** > **Create** > **Curves** > **Single Curve**.
2. Select **math**.
3. In the x = field, enter `{0, 0.01, 0.1}`.

> **Tip:** Include the brackets in the x = field.

4. In the y = field, enter `{1, 0, 0}`.

> **Tip:** Include the brackets in the x = field.



Figure 1160:

5. Click **create**.
   HyperMesh generates a curve.

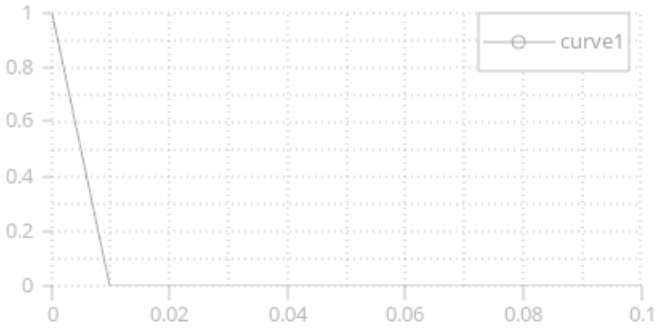


Figure 1161:

6. Click **return**.

8.1.12 Create a Sensor Card

In this step, you will create a sensor card.

Sensors are implemented as properties in HyperMesh. In this example we refer to the curve defined in Create a Function.

1. From the Analysis page, click **safety** > **sensors**.
2. In the name = field, enter `sensor`.
3. Click **card image =**, and select **SENSOR**.
4. Click **create/edit**.
   The **Card Image** dialog opens.
5. Set Sensor type to **Type 5** (logical function switch Sensor).
6. Click **LCS** > **curve1**.
7. Click **return** twice to exit the panels.

8.1.13 Export a Data Deck

In this step, you will export a PAM-CRASH 2G data deck from HyperMesh.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
   The Export - Solver Deck tab opens.
2. In the File field, enter `rail.pc`.
   `rail.pc` is the PAM-CRASH 2G file you will create.
3. Set Template to **Pamcrash2G2012**.
4. Click **Export**.
   HyperMesh writes the deck. A message in the Status bar indicates the process is completed.

Permas

9

Permas tutorials.

This chapter covers the following:

- 9.1 HM-4800: Setup a Permas Analysis (p. 928)

9.1 HM-4800: Setup a Permas Analysis

In this tutorial, you will complete the full setup of a Permas analysis.

Before you begin, copy `permastube_clamping.hm` from `<hm.zip>/interfaces/permas/` to your working directory.

9.1.1 Load the Permas User Profile

In this step, you will load the Permas profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to **Permas**.

9.1.2 Retrieve and View the Model File

In this step, you will open the model file and view it in HyperMesh.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
2. In the **Open Model** dialog, open the `permastube_clamping.hm` file.

> **Note:** This file contains pre-defined model data.

The model appears in the graphics area.

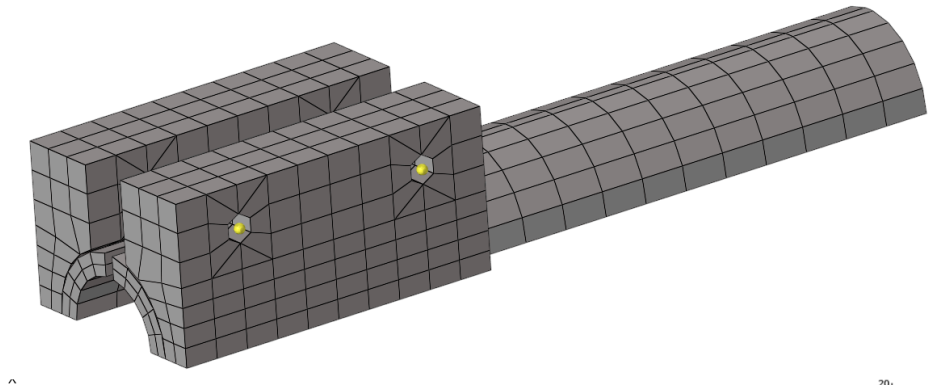


Figure 1162:

9.1.3 Create a Permas $COMPONENT and $SYSTEM

In this step, you will create a component and system card in HyperMesh.

As a first recommended step for each model set up in the Permas user profile, you should create a component and a system card. If this is not done, a warning message is displayed in the message bar on export. Currently only one component and system per model is supported in HyperMesh.

1. From the menu bar, click **Setup** > **Create** > **Control Cards**.
   The **Card Image** dialog opens.
2. In the **Card Image** dialog, click **COMPONENT**.



Figure 1163:

3. In the Name field, enter `comp1`.
4. Click **return** to create the card and close the card image.
5. Click **SYSTEM**.
6. In the Name field, enter `sys1`.
7. Click **return** to create the card and exit the card image.
8. Click **return** again to close the **Card Image** dialog.

9.1.4 Create a Component Collector

In this step, you will create a new component collector.

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu.



Figure 1164:

A new component opens in the Entity Editor.

2. Define the component properties in the Entity Editor.
   a) For Name, enter `clamps1`.
   b) Select a color for the component.
   c) Set card image to **<None>**.

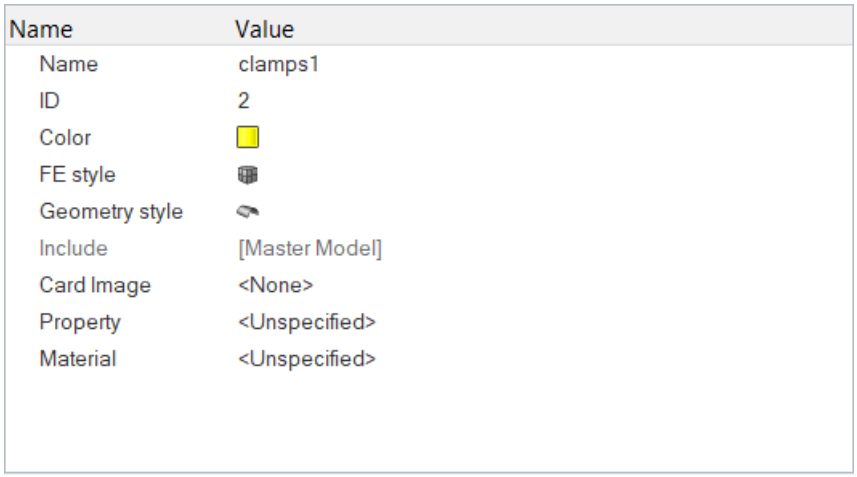


Figure 1165:

9.1.5 Organize the Elements

In this step, you will organize the elements of the model.

1. From the menu bar, click **Mesh** > **Organize** > **Elements** > **To Component**.
   The Organize panel opens.
2. Select one element on one side of the clamp as seen in Figure 1166.

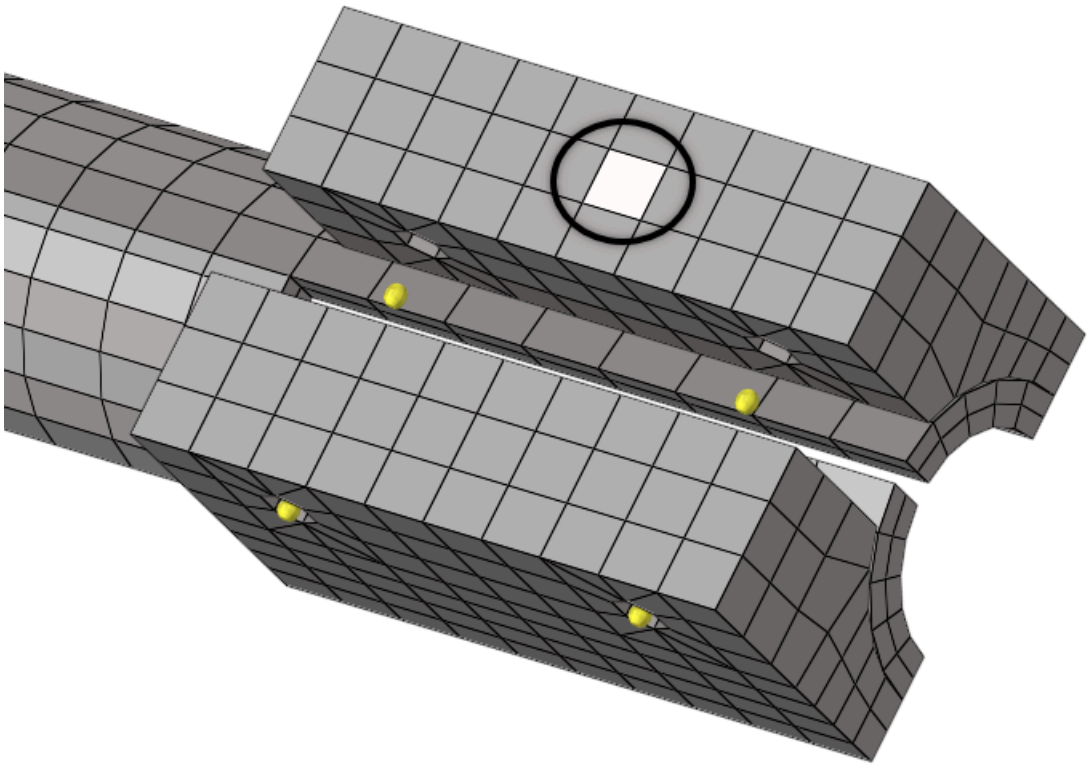


Figure 1166:

3. In the Organize panel, click **elems** > **by attached**.

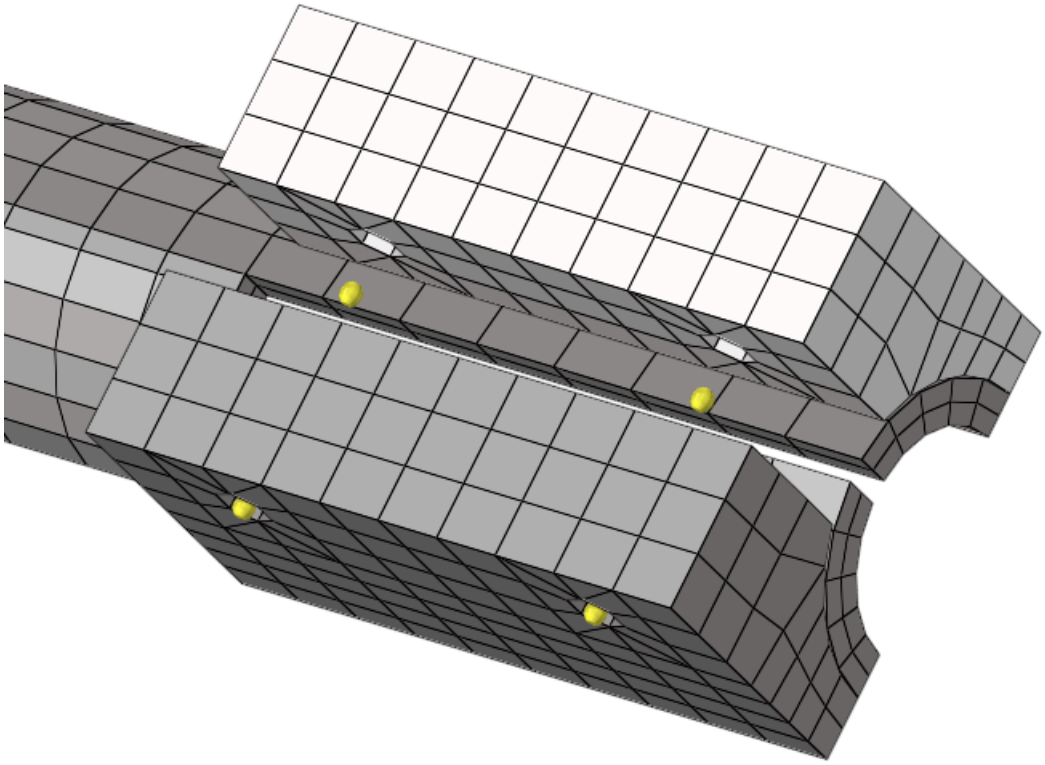


Figure 1167:

The entire side of the clamp becomes selected as seen in Figure 1167.

4. Repeat steps 2 and 3 to select the elements on the other side of the clamp.
The other side of the clamp becomes selected as seen in Figure 1168.

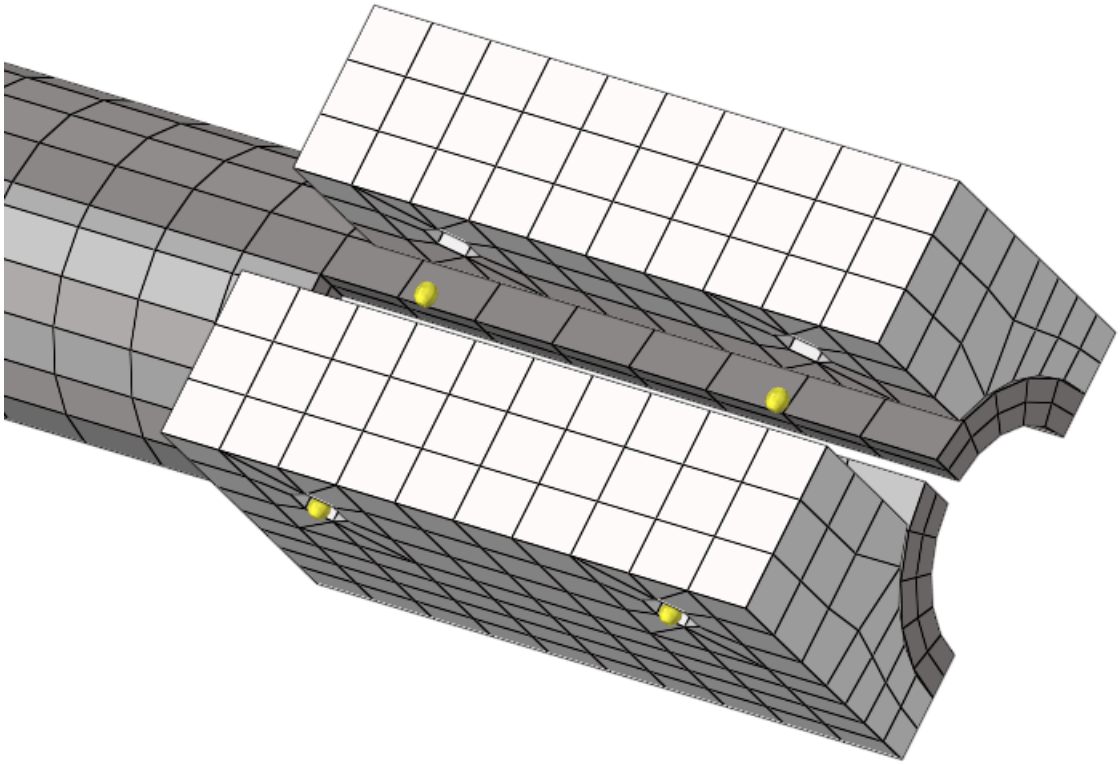


Figure 1168:

5. In the dest component = field, ensure that `clamps1` is entered.
6. Click **move**.
   HyperMesh organizes the selected elements into the clamps1 component as seen in Figure 1169.

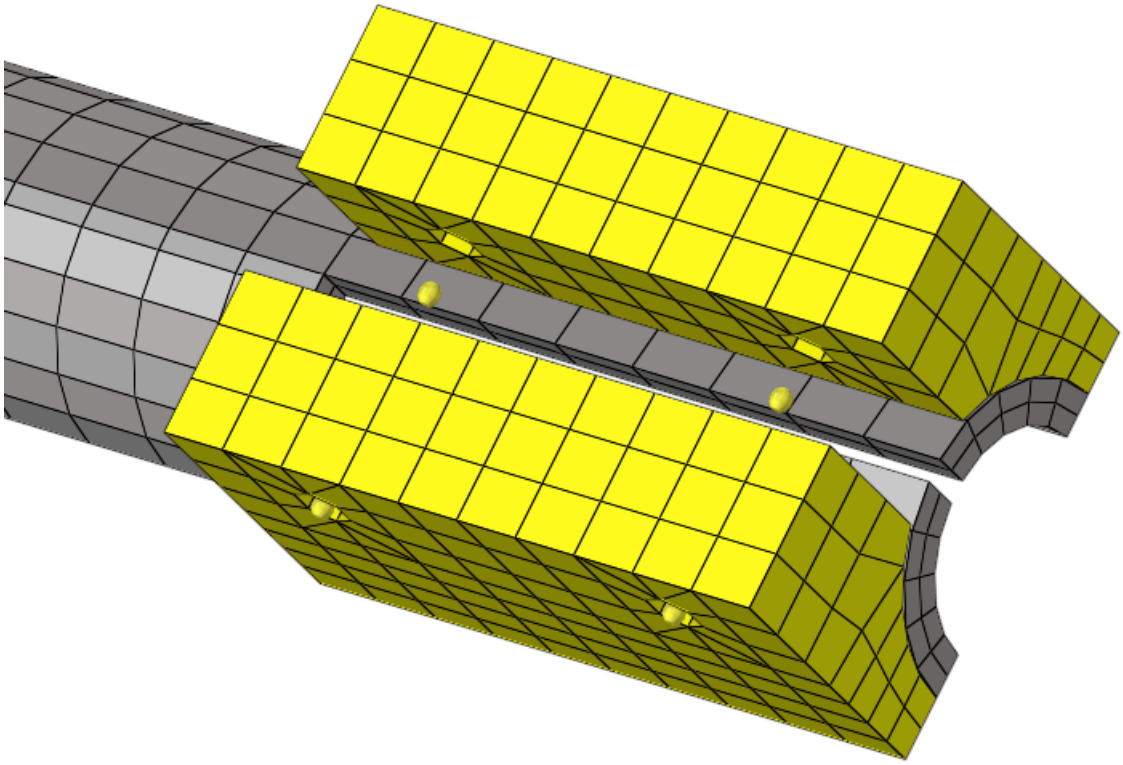


Figure 1169:

7. Click **return** to exit the panel.
8. In the Model Browser, Component folder, click **misc1**.
   The Entity Editor opens and displays the component's card data.
9. Define the misc1 properties in the Entity Editor.
   a) For Name, enter `tube`.
   b) Click the **color** box, and select a new display color for the component.

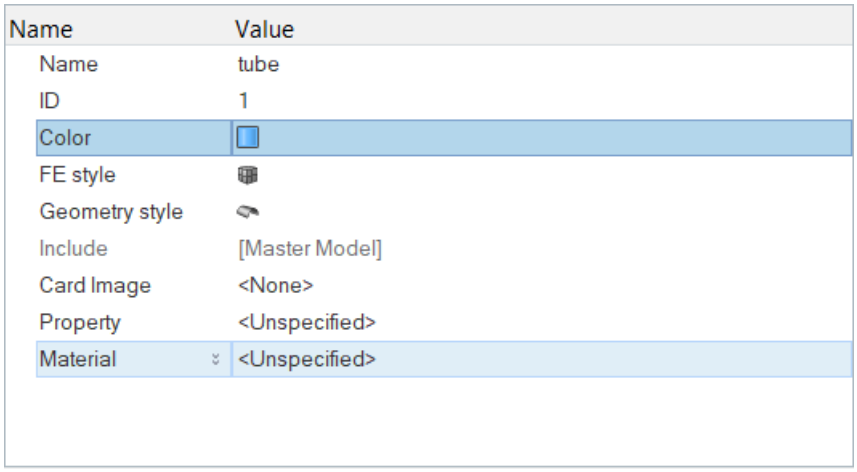


Figure 1170:

9.1.6 Create Material and Elemental Properties

In this step, you will create a solid property and two materials for the existing elements. You will also assign the property and materials to the previously created components.

Assigning a property/material to a component is equivalent to a property to set assignment in Permas. For 2D and 3D elements, this is currently the only method of property assignment.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu.
   A new property opens in the Entity Editor.

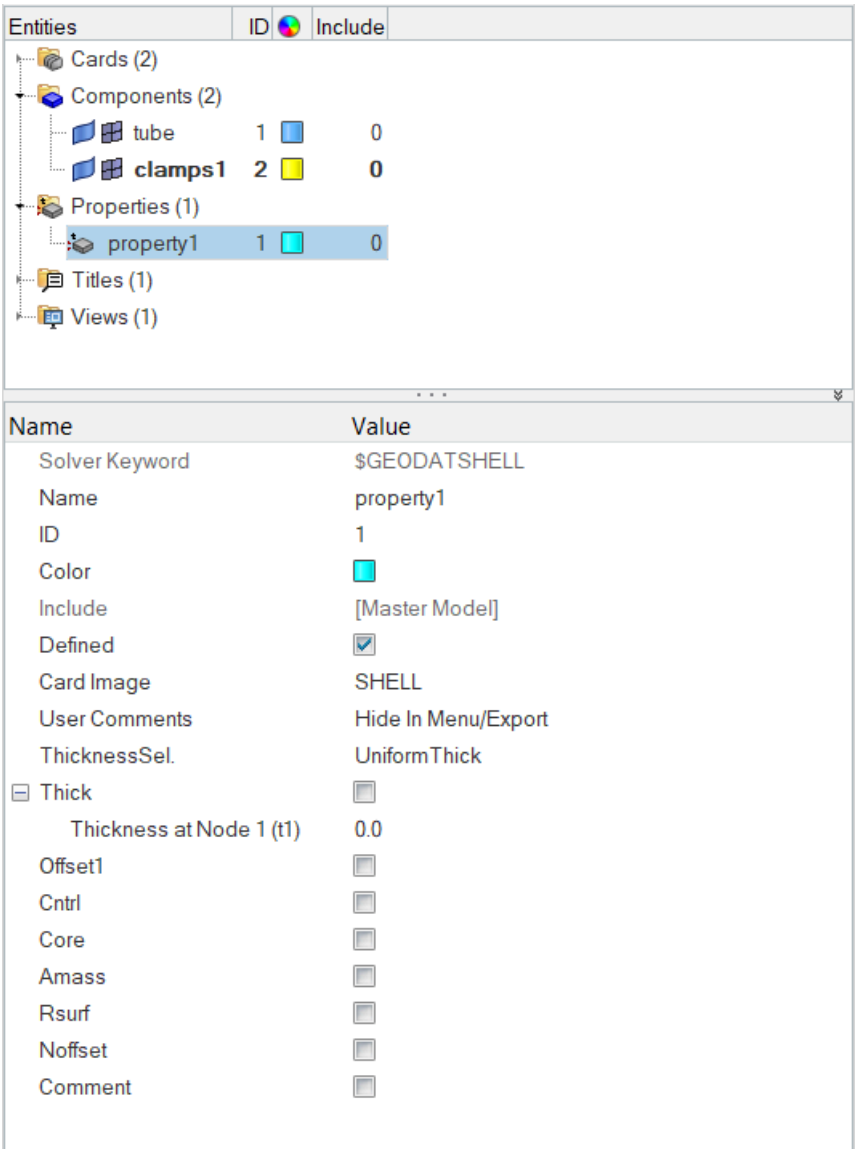


Figure 1171:

2. Define the property properties in the Entity Editor.

a) For Name, enter `Solid`.

b) Set card image to **SOLID**.

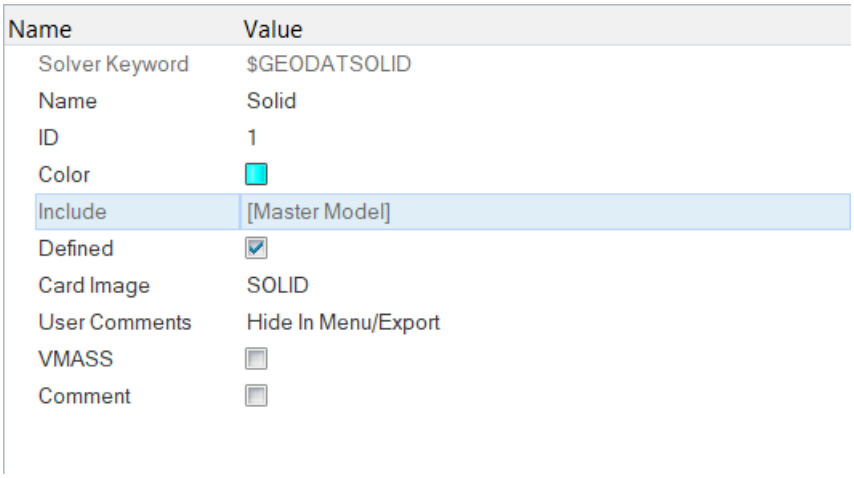


Figure 1172:

3. In the Model Browser, right-click and select **Create** > **Material** from the context menu.
A new material opens in the Entity Editor.

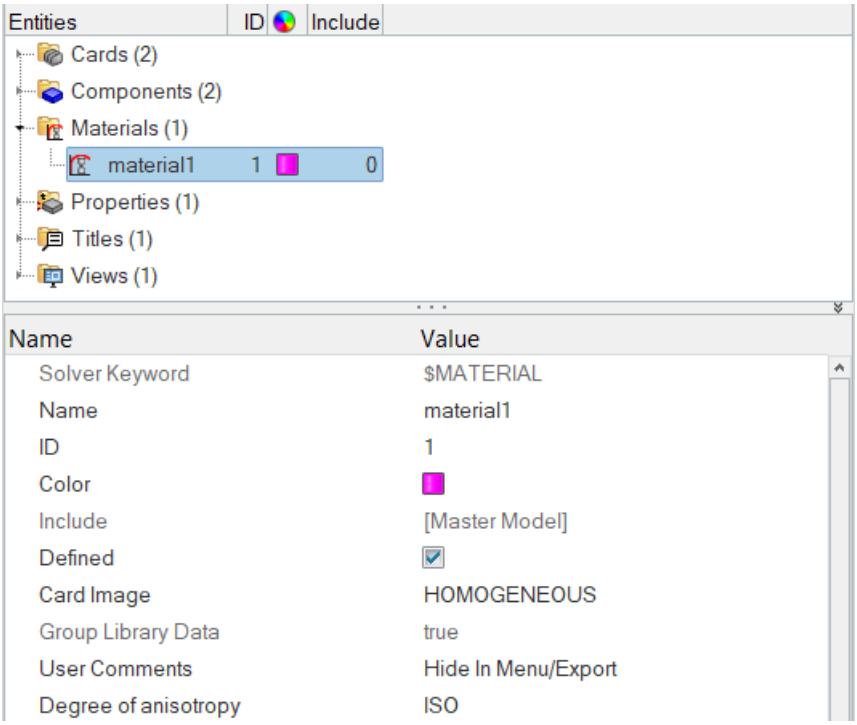


Figure 1173:

4. Define the material properties in the Entity Editor.
   a) For Name, enter `Mat_1`.
   b) Set card image to **HOMOGENEOUS**.

5. In the Model Browser, Materials folder, right-click on **Mat_1** and select **Card Edit** from thecontext menu.

6. Define settings in the card image.
   a) Select the **Elastic** checkbox.
   b) In the E (elasticity modulus field) field, enter `210000`.
   c) In the nu field, enter `0.3`.

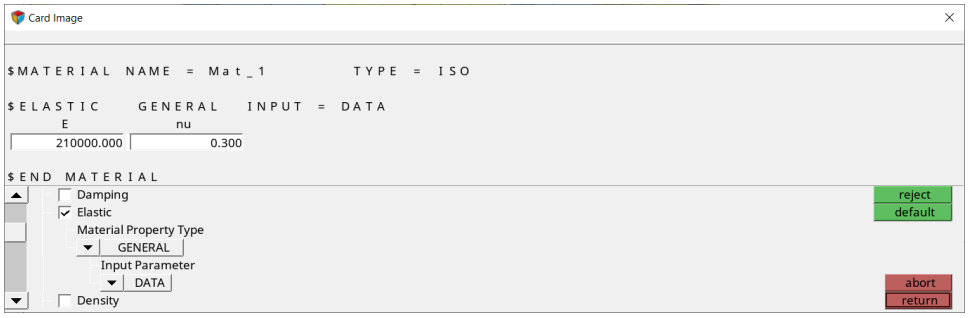


Figure 1174:

7. Click **return** to close the card image.
8. In the Model Browser, right-click and select **Create** > **Material** from the context menu.
   A new material opens in the Entity Editor.

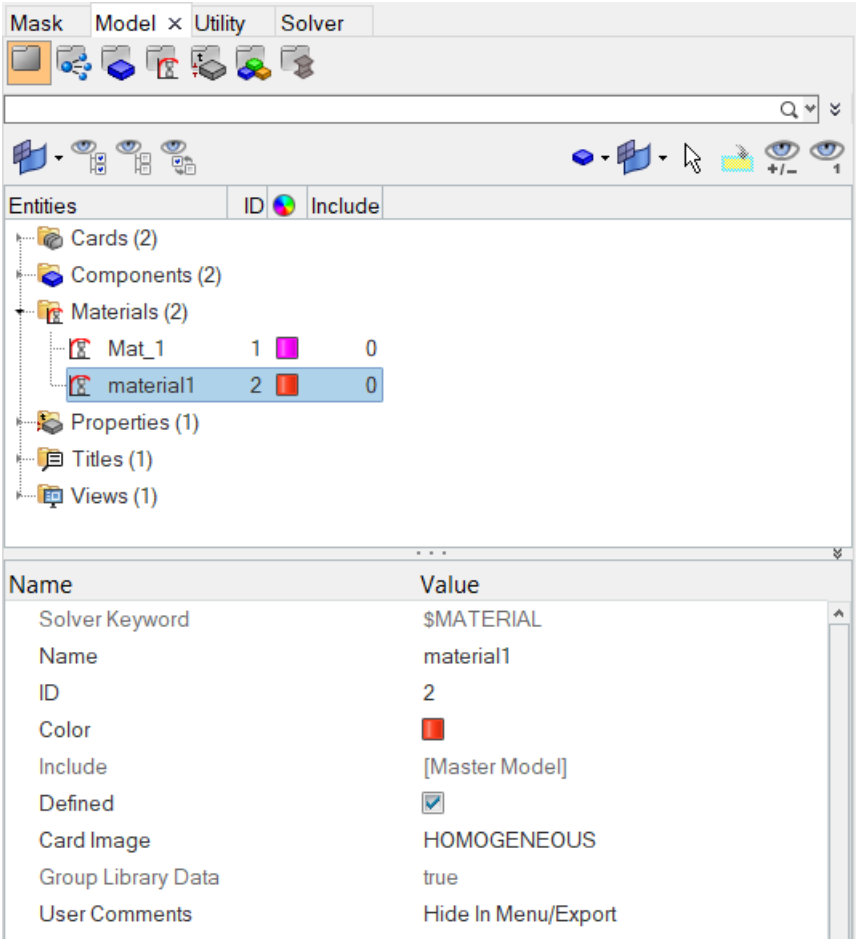


Figure 1175:

9. Define the material properties in the Entity Editor.
   a) For Name, enter `Mat_2`.
   b) Set card image to **HOMOGENEOUS**.

10. Open the card image by right-clicking on **Mat_2** in the Model Browser, Materials folder and selecting **Card Edit** from the context menu.

11. Define settings in the **Card Image** dialog.
   a) Select the **Elastic** checkbox.
   b) In the E (elasticity modulus field) field, enter `7000`.
   c) In the nu field, enter `0.330`.



Figure 1176:

12. Click **return** to close the card image.
13. On the Collectors toolbar, click  .
    The Components panel opens.
14. Open the **update** subpanel.
15. Click **comps**.
16. Select the **clamps1** checkbox.
17. Click **select**.

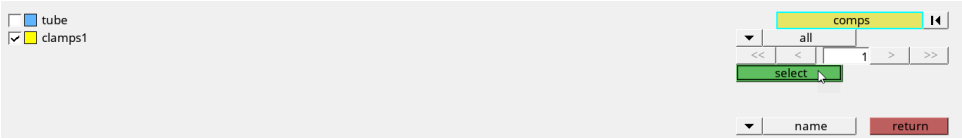


Figure 1177:

18. Click **card image**, and select **PART_ELPROP**.
19. Set the property/no property toggle to **property**.
20. Click **property=** and select **solid**.
21. Set the material/no material toggle to **material**.
22. Click **material=** and select **Mat_1**.



Figure 1178:

23. Click **update**.
24. In the Model Browser, Component folder, right-click on **clamps1** and select **Card Edit** from thecontext menu.
    The card image opens, and the name of the component, property, and material you assigned is shown.

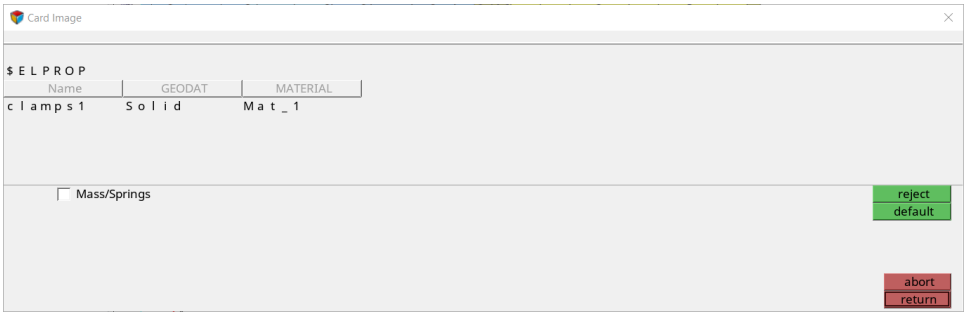


Figure 1179:

25. Click **return** to close the card image.
26. Edit the tube component.
    a) Right-click on **tube** in the Model Browser, Component folder and select **Make Current** from the context menu.
    b) On the Collectors toolbar, click  .
    The Components panel opens.
    c) Open the **update** subpanel.
    d) Click **comps**.
    e) Select the **tube** checkbox.
    f) Click **select**.
    g) Click **card image**, and select **PART_ELPROP**.
    h) Set the property/no property toggle to **property**.
    i) Click **property=** and select **solid**.
    j) Set the material/no material toggle to **material**.
    k) Click **material=** and select **Mat_2**.
27. Click **Update** to complete the changes.
28. Click **return** to exit the Components panel.

9.1.7 Create Beam Elements

In this step, you will create beam elements and assign a property to individual elements during creation.

In comparison to 2D and 3D elements, you can assign mutually exclusive properties to elements or sets (components).

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu.
   A new component opens in the Entity Editor.
2. Define the component properties in the Entity Editor.
   a) For Name, enter `Beam`.
   b) Set card image to **<None>**.

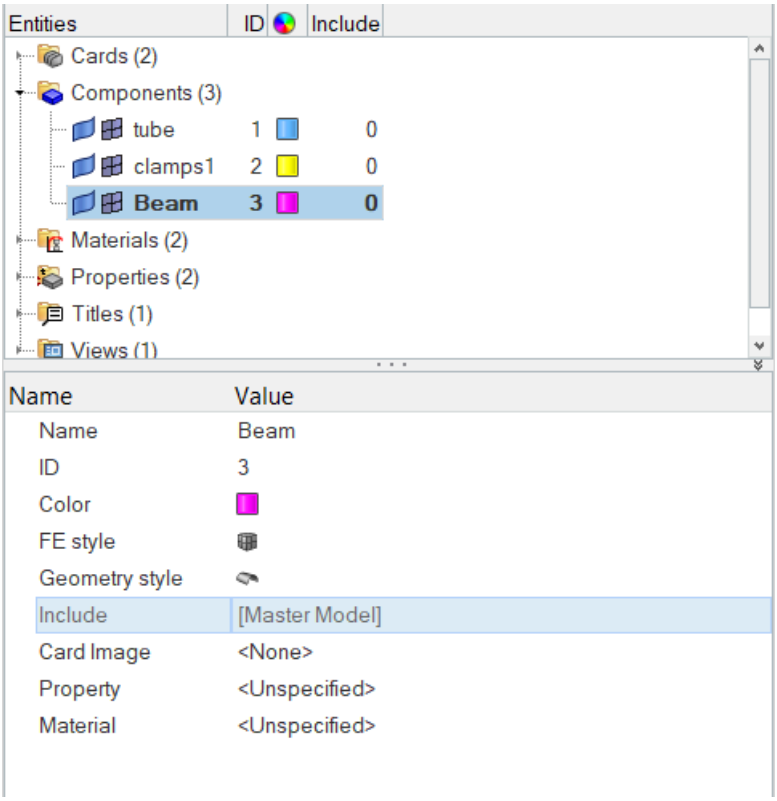


Figure 1180:

3. In the Model Browser, right-click and select **Create** > **Property** from the context menu. A new property opens in the Entity Editor.
4. Define the property properties in the Entity Editor.
   a) For Name, enter `XBA`.
   b) Set card image to **BEAM**.

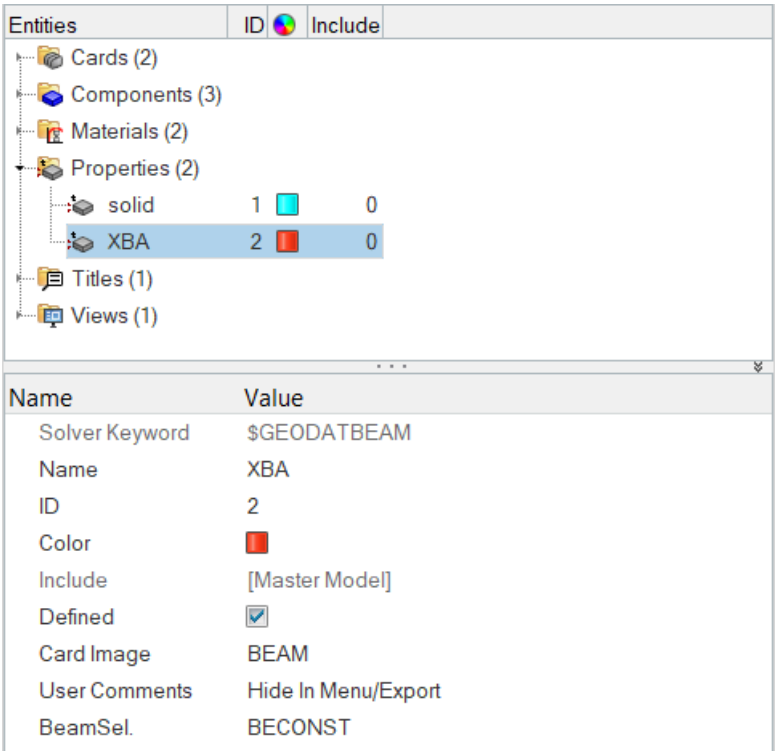


Figure 1181:

5. In the Model Browser, Property folder, right-click on **XBA** and select **Card Edit** from thecontext menu.
   The card image opens.
6. In the card image, enter the following parameters:
   - For A-1, enter `12.6`.
   - For Ixx-1, enter `12.6`.
   - For Iyy-1, enter `12.6`.
   - For J-1, enter `25.1`.

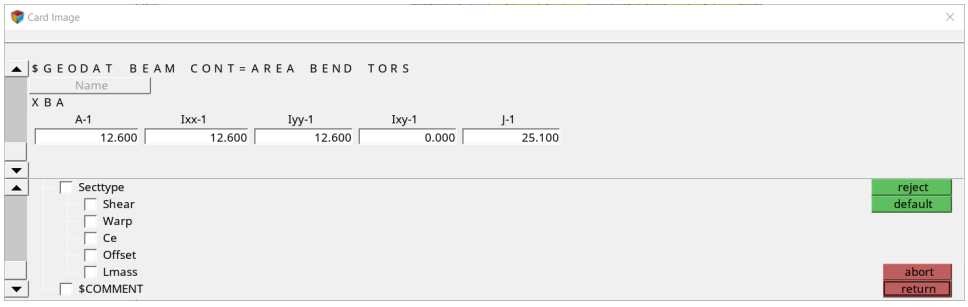


Figure 1182:

7. Click **return** to exit the card image.
8. Open the Bars panel by clicking **Mesh** > **Create** > **1D Elements** > **Bars** from the menu bar.
9. Set the orientation selector to **z-axis**.

Note: Because HyperMesh is a generic preprocessor, you have to provide an orientation vector to create the element, although this vector does not go anywhere in the Permas deck. For beams, use the node option in the same panel where a direction vector needs to be defined.

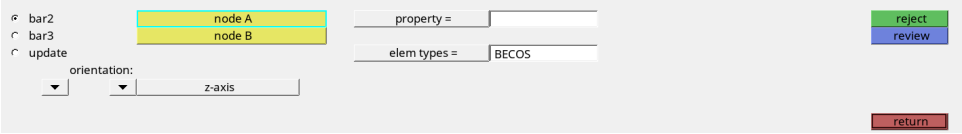


Figure 1183:

10. Click **property =** and select **XBA**.

11. Using the node A selector, select the node on the top of one side of the clamp as seen in Figure 1184.

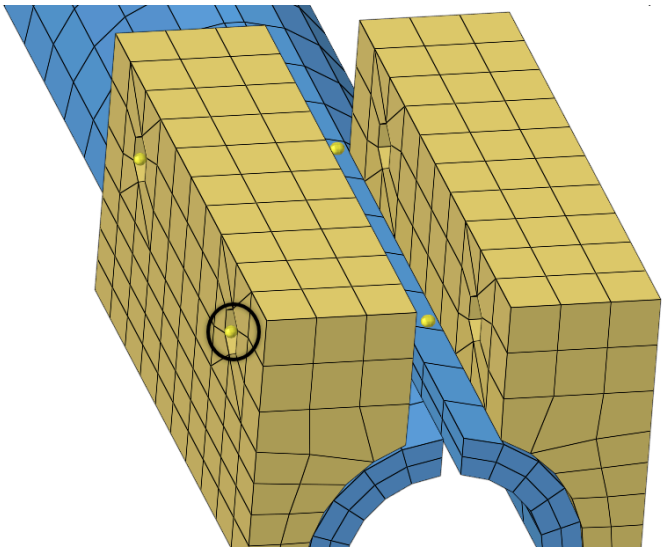


Figure 1184:

12. Using the node B selector, select the node in the middle of the clamp that is closest to the node you selected as node A as seen in Figure 1185.

Tip: You may need to zoom in on the model to properly select the the node in the middle of the clamp which is closest to the node you selected as node A.

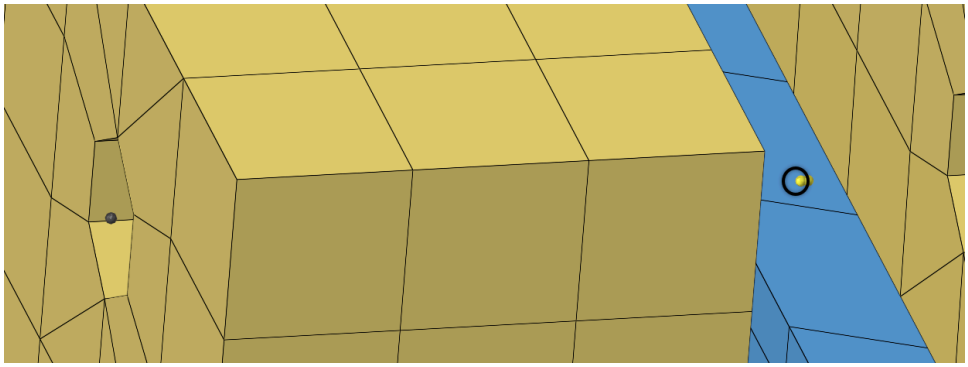


Figure 1185:

The beam is created automatically.

13. Repeat steps 11 and 12 to connect the other node on the top of the clamp with the other node in the middle of the clamp.
    The beam is created automatically.
14. Repeat steps 11 and 12 for both sets of nodes on the other side of the clamp.
15. In the Model Browser, Component folder, right-click on **clamps1** and select **Hide** from the context menu to see the four sets of bars connecting the six nodes.

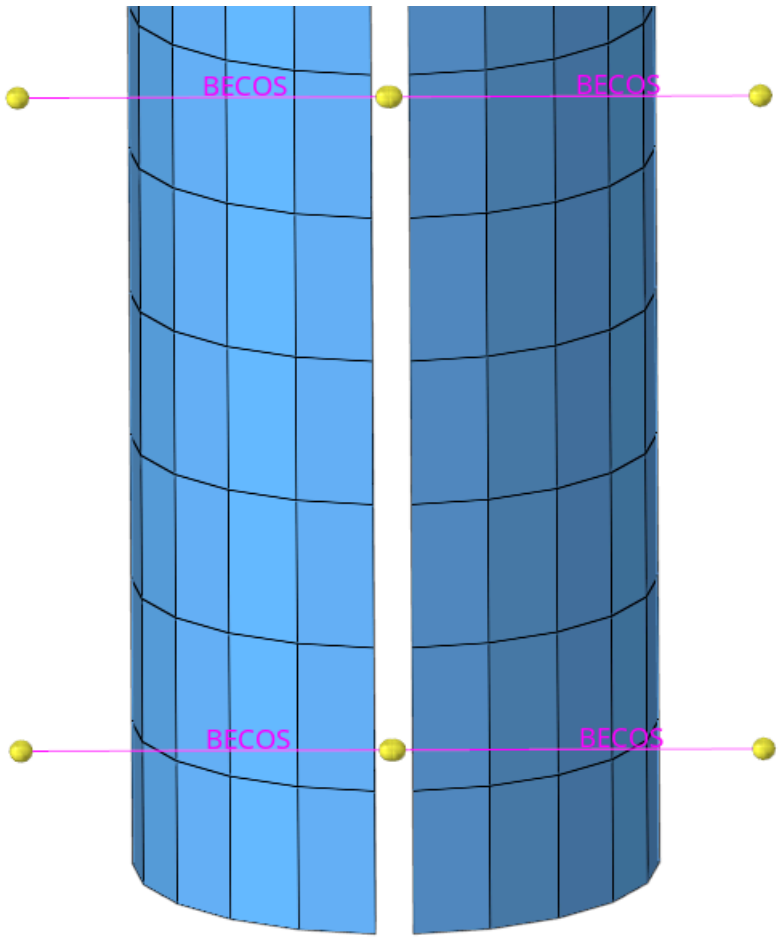


Figure 1186:

16. In the Model Browser, click  to display the elements in the color of the property that has been assigned to them.

> **Tip:** This is a good way to ensure that you have assigned the correct property to the elements.

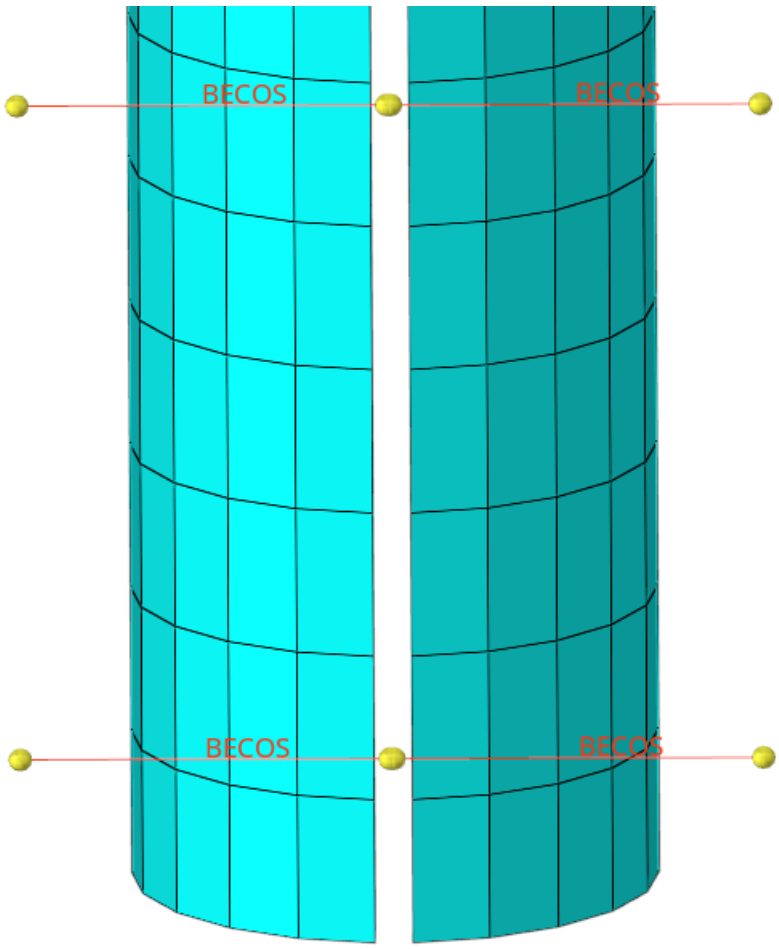


Figure 1187:

17. In the Model Browser, click to return to the default display settings.

a) Optional: If necessary, right-click on the **clamps1** component and select **Show** to display the clamps again.

18. Click **return** to exit the panel.

9.1.8 Create a Spring Element

In this step, you will create a spring element and change the element type used when elements are created. You will also use the coincident picking mechanism to help you precisely select nodes.

The coincident picking mechanism is useful in cases where entities, such as nodes and elements, are close to each other.

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu.

> **Note:** Springs should now be your current component collector, which ensures that newly created elements are put into this collector.

A new component opens in the Entity Editor.

2. Define the component properties in the Entity Editor.
   a) For Name, enter `Springs`.
   b) Set card image to **PART_ELPROP**.

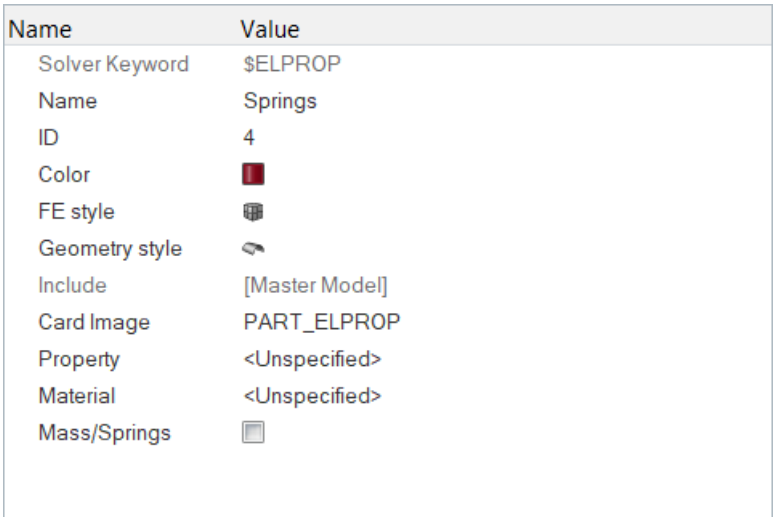


Figure 1188:

3. Open the Element Type panel by clicking **Mesh** > **Assign** > **Element Type** from the menu bar.
4. Open the **1D** subpanel.
5. Click **spring =**, and select **X2STIFF3**.
6. Click **return** to exit the panel.
7. From the menu bar, click **Preferences** > **Graphics**.
   The Graphics panel opens.
8. In the Graphics panel, select the **coincident picking** checkbox.

   Note: Coincident picking gives you more control during node selection. See the online help regarding the graphics subpanel for more information.

9. Click **return** to exit the panel.
10. In the Model Browser, right-click and select **Create** > **Property** from the context menu.
    A new proptery opens in the Entity Editor.
11. Define the property properties in the Entity Editor.
    a) For Name, enter `XST`.
    b) Set card image to **SCALAR**.
12. In the Model Browser, Property folder, right-click on **XST** and select **Card Edit** from thecontext menu.
13. Define settings in the card image.
    a) Select the **DOF** checkbox to enable the degrees of freedom at the first and second node.
    b) In the coeff field, enter `100`.

c) In the DOF1 and DOF2 fields, enter 2.

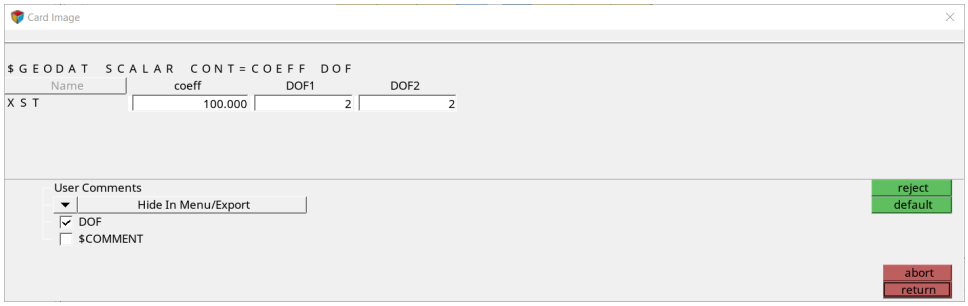


Figure 1189:

d) Click **return** to close the card image.

14. Open the Springs panel by doing one of the following:

- Open the **ID** page and then selecting **springs**.

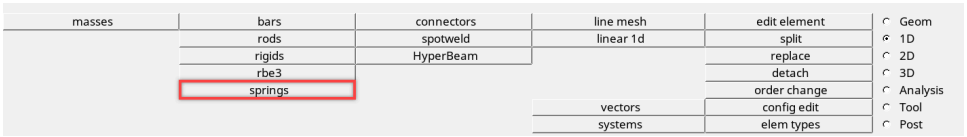


Figure 1190:

- Click **Mesh** > **Create** > **1D Elements** > **Springs** from the menu bar.

15. Click **property =** and select **XST**.

16. Using the node selector, select the node shown in Figure 1191.
The coincident picking tool appears, and displays two nodes.

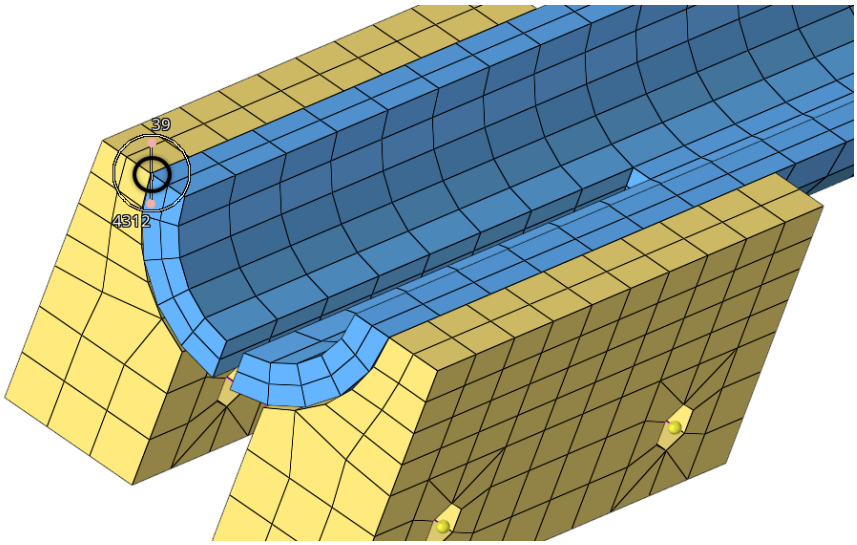


Figure 1191:

17. Click the top node displayed by the coincident picking tool as seen in Figure 1192.

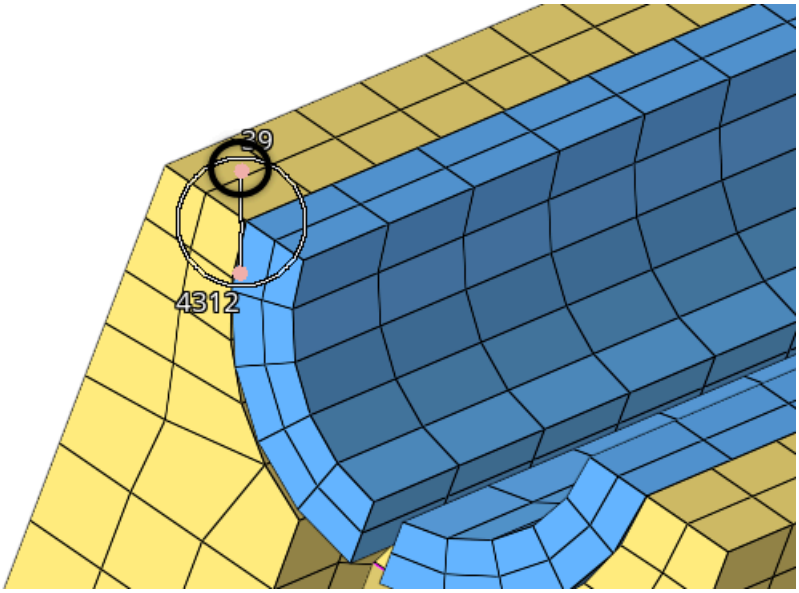


Figure 1192:

18. Using the node selector, click the node selected in step 17.

19. Using the coincident picking tool, click the bottom node as seen in Figure 1193.

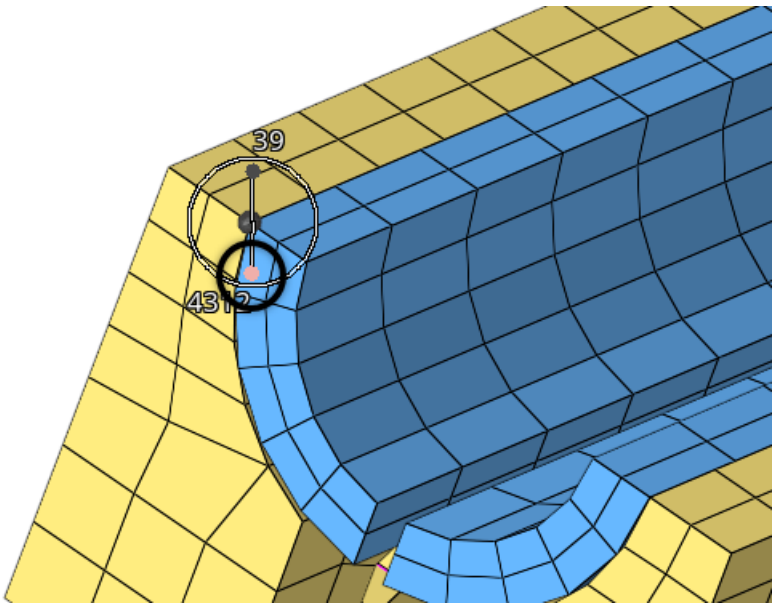


Figure 1193:

HyperMesh creates a spring as seen in Figure 1194.

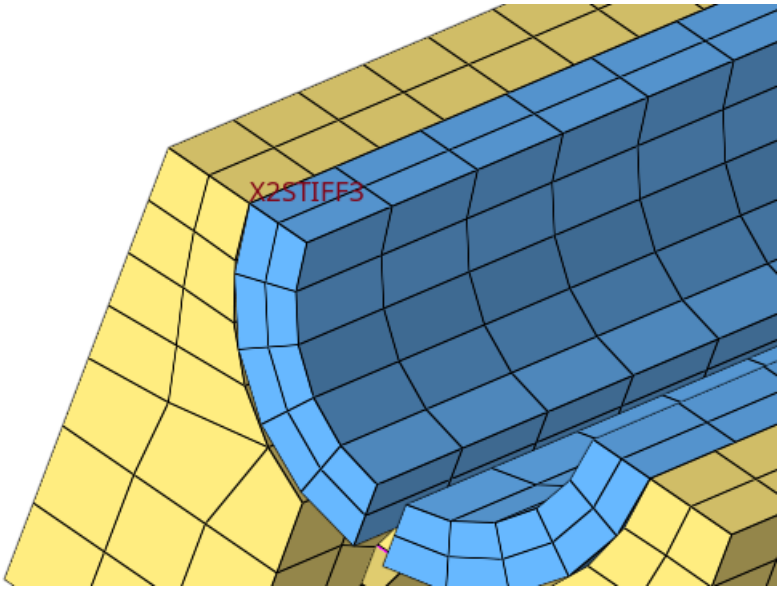


Figure 1194:

20. Repeat steps 16 through 19 to create a spring on the other side of the model.

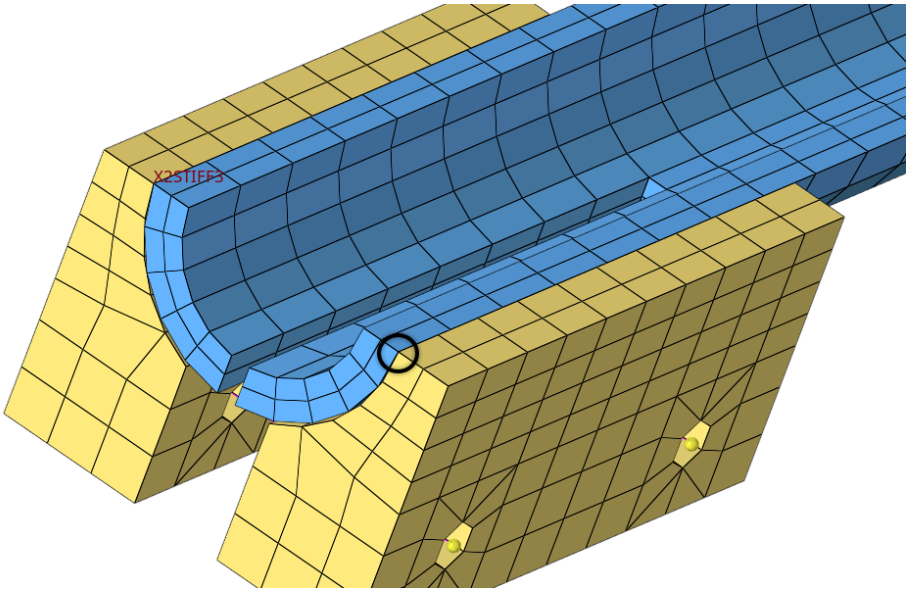


Figure 1195:

HyperMesh creates a spring on the other side of the model as seen in Figure 1196.

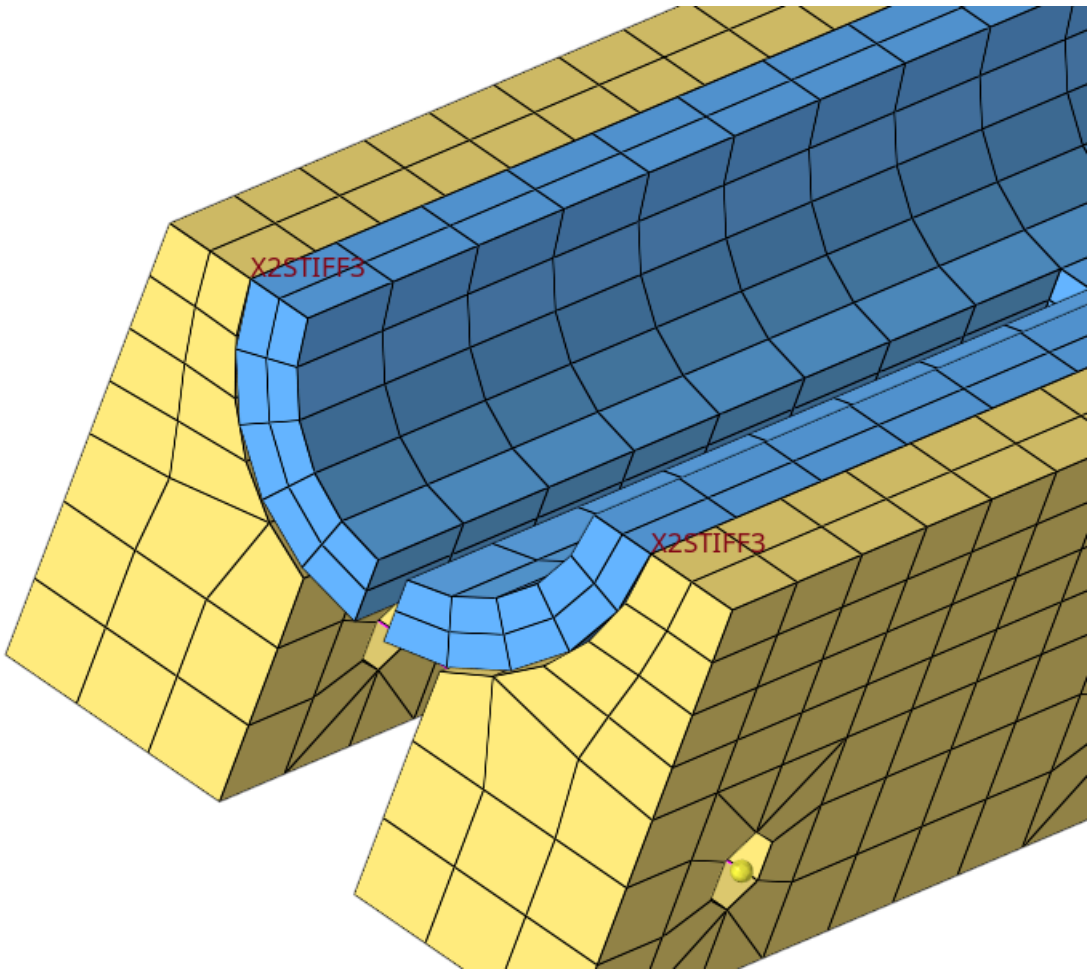


Figure 1196:

21. Click **return** to exit the panel.

9.1.9 Create Multi-Point Constraints

In this step, you will create two different types of multi-point constraints (MPCs). You will also create a component collector for each type of MPC so you can switch their display on and off separately.

Because MPCs do not need a property assignment in HyperMesh, a card image ($ELPROP) is needed for these components.

1. In the Model Browser, right-click and select **Create** > **Component** from the context menu.
   A new component opens in the Entity Editor.
2. Define the component properties in the Entity Editor.
   a) For Name, enter `MPC_Rigids`.
   b) Set card image to **<None>**.
3. Open the Rigids panel by clicking **Mesh** > **Create** > **1D Elements** > **Rigids** from the menu bar.
4. Set the dependent switch to **multiple nodes**.

5. Clear the **dof4**, **dof5** and **dof6** checkboxes.

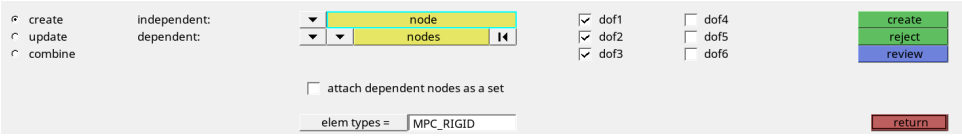


Figure 1197:

6. Using the independent node selector, select one of the nodes at the end of the beam elements that were created in Create Beam Elements.

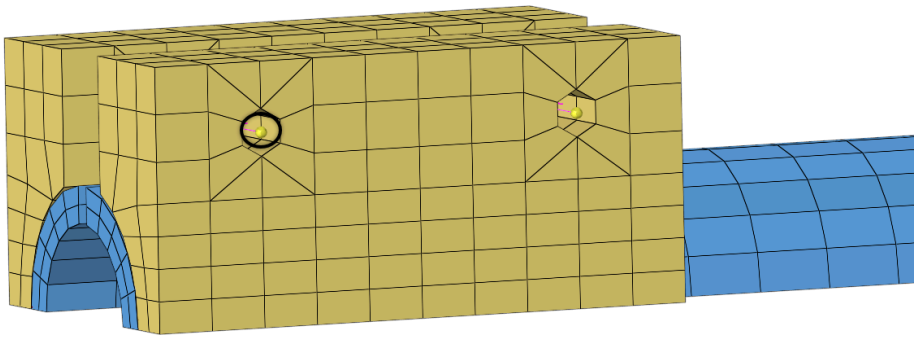


Figure 1198:

7. Using the dependent node selector, select the nodes surrounding the first node as seen in Figure 1199.

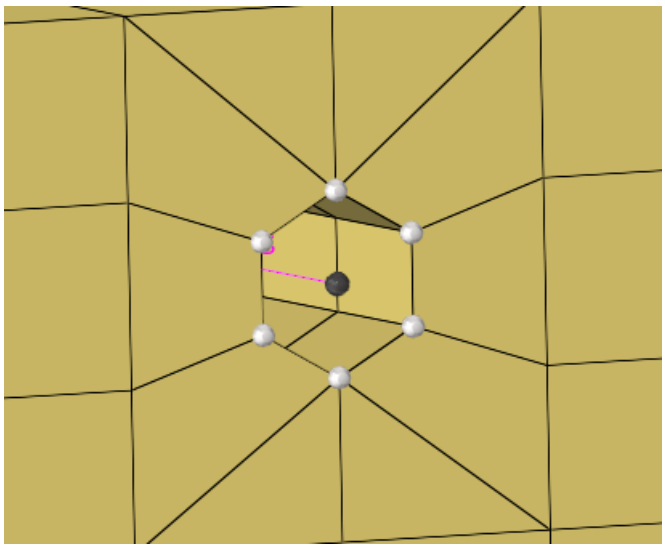


Figure 1199:

8. Click **create**.
HyperMesh generates the MPC.

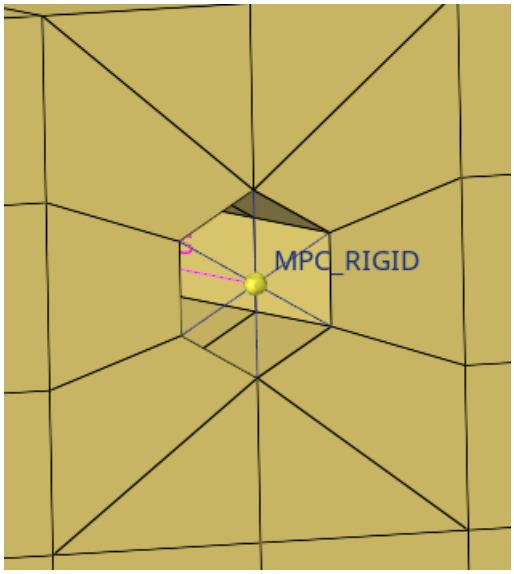


Figure 1200:

9. Repeat steps 6 through 8 to create elements for the other three ends of the beam elements.
10. Click **return** to exit the panel.
11. Open the Utility menu by clicking **View** > **Browsers** > **HyperMesh** > **Utility** from the menu bar.
12. In the Utility menu, click **Disp**.
13. Click **Clear Temp Nodes**.
14. In the Model Browser, right-click and select **Create** > **Component** from the context menu.
    A new component opens in the Entity Editor.
15. Define the component properties in the Entity Editor.
    a) For Name, enter `MPC_Same`.
    b) Set card image to **<None>**.
16. Open the Element Type panel by clicking **Mesh** > **Assign** > **Element Type** from the menu bar.
17. Click **rigid =**, and select **MPC_SAME**.
18. Click return to close the panel.

> **Note:** MPC_Same should now be your current component collector. If not, right-click on **MPC_Same**, in the Model Browser, and select **Make Current** from the context menu.

19. Open the Rigids panel by repeating step 3.
    A MPC Same rigid will be used to connect the two beams created in Create Beam Elements, where there is a small gap between each vertical section. You will use the Rigids panel to connect these two sections with a MPC Same rigid.
20. Select all of the **DOF** checkboxes, except dof2.
    Dof2 is set free for the pretension load which is applied later in the tutorial.
21. Set the dependent switch to **single node**.

22. Using the independent node selector, select the inner end of one of the beams created in Create Beam Elements.

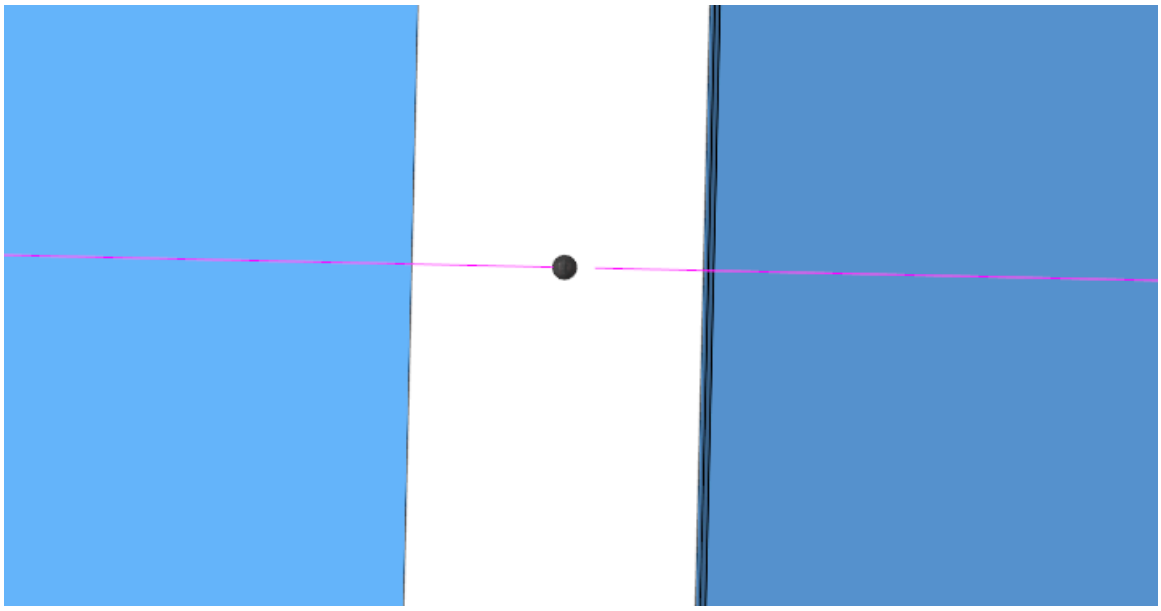


Figure 1201:

23. Using the dependent node selector, select the inner end of the other beam.
The MPC Same rigid is created.

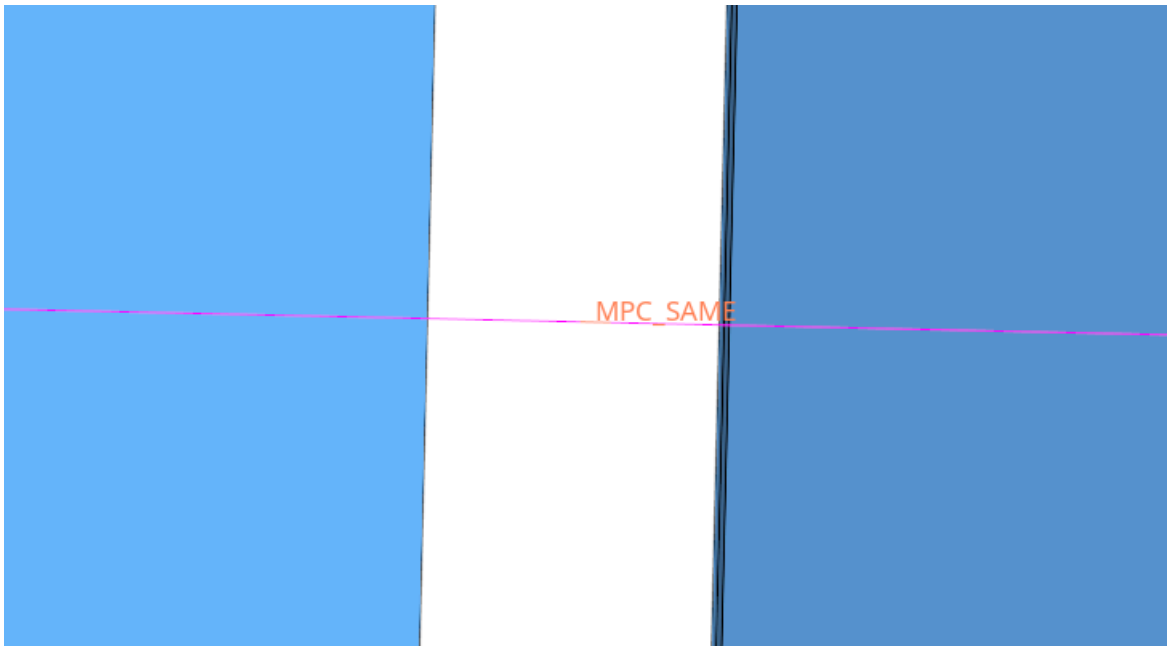


Figure 1202:

24. Click **return** to close the panel.

9.1.10 Set Definition

In this step, you will create sets that will later be used for contact definition.

1. In the Model Browser, Views folder, right-click on **set_def** and select **Show** from the context menu.



Figure 1203:

2. From the menu bar, click **Tools** > **Create** > **Sets**.
   The **Create set** dialog opens.
3. Define the set within the **Create set** dialog.
   a) In the name field, enter `SET_1`.
   b) For Entity IDs click **0 Nodes** > **Nodes**.
   The Nodes panel opens.
   c) While pressing **Shift**, draw a box around the lower edge of the split portion of the gap as seen in Figure 1204.

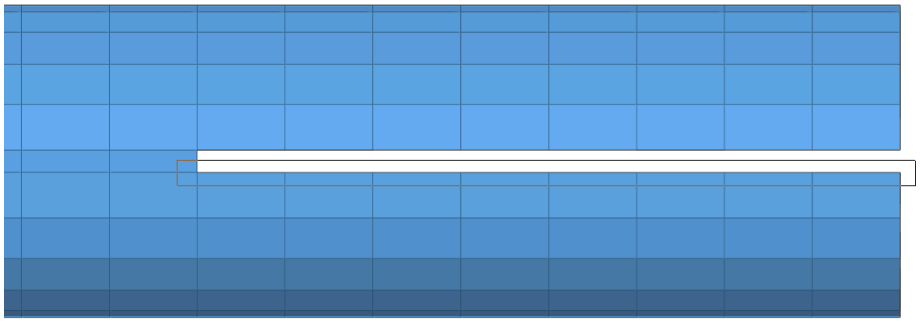


Figure 1204:

All nodes within this section highlight as seen in Figure 1205.

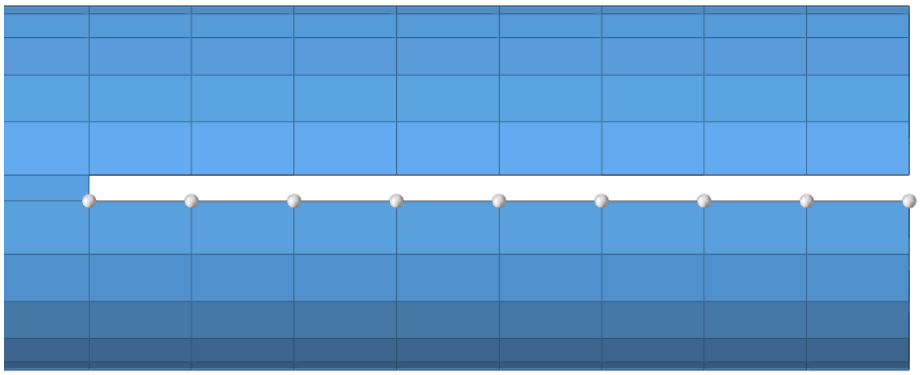


Figure 1205:

d) Click **proceed**.
HyperMesh organizes all of the highlighted nodes into SET_1.

e) Click **Close** to close the **Create set** dialog.

4. In the Model Browser, Component folder, turn on the display of **Beam** and **MPC Same**.

5. Repeat steps 3.a through 3.d to create SET_2, SET_11, SET_12, SET_21, and SET_22 using the following definitions:

- SET_2: upper edge of the split portion of the tube.
- SET_11: lower node of 1st bolt connection.
- SET_12: upper node of 1st bolt connection.
- SET_21: lower node of 2nd bolt connection.
- SET_22: upper node of 2nd bolt connection.
- Clamps: nodes of the contact surfaces at the clamps to the tube.

Tip: Use the following parameters to create the sets:

- SET_2 – Select the top set of nodes on the opening that was selected.
- SET_11 – Zoom in on the MPC SAME rigid, and select one node on the bottom of the first MPC SAME rigid created.
- SET_12 – Zoom in on the MPC SAME rigid, and select one node on the top of the first MPC SAME rigid created in.
- SET_21 – Zoom in on the MPC SAME rigid, and select one node on the bottom of the second MPC SAME rigid created.
- SET_22 – Zoom in on the MPC SAME rigid, and select one node on the top of the second MPC SAME rigid created.

6. Click **return**.

7. In the Model Browser, Component folder, right-click on **clamps1** and select **Isolate Only** from the context menu.

8. On the Standard Views toolbar, click  .
9. From the menu bar, click **Tools** > **Create** > **Sets**.
10. In the name field, enter `Clamps`.
11. Click **nodes** > **by window**.
12. Draw a box to select the nodes.
13. Click **create**.
    HyperMesh creates the set.
14. Click **return** to close the panel.

> **Tip:** When you have a large amount of sets, use the Set Browser to create sets.

9.1.11 Create Surfaces from Solid Elements

In this step, you will create surfaces from solid elements.

The surface definition and the sets from above will be used later for contact definition.

1. In the Model Browser, Component folder, right-click on **Tube** and select **Isolate Only** from the context menu.
2. Open the Contact Surfaces panel by clicking **BCs** > **Create** > **Contact Surfaces** from the context menu.
3. Open the **solid faces** subpanel.
4. In the name field, enter `Surf_tube`.
5. Click **card image =**, and select **$SURFACE**.
6. Set the switch to **elems**.

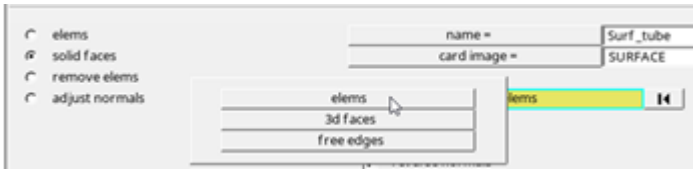


Figure 1206:

7. Click **elems** > **by collector**.
8. Select the **tube** component.
9. Click **select**.
10. Using the nodes selector, select two nodes on the tube component.

> **Tip:** These nodes should be on one solid element, diagonal from each other.

These nodes will be used to specify the faces of the solids to be taken into account for this surface creation.

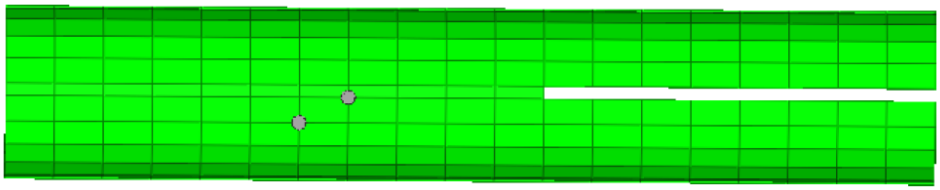


Figure 1207:

11. Click **create**.
    HyperMesh creates the contact surface.
12. Click **return** to exit the panel.

9.1.12 Define the Contact Partners

In this step, you will define the contact partners using $CONTACT.

In this analysis, four contacts will be established.

Table 4:

Contact Name	Type	Master	Slave
Bolt_1	Node to Node	SET_11	SET_12
Bolt_2	Node to Node	SET_21	SET_22
Gap	Node to Node	SET_1	SET_2
Tube_clamps	Surface to Node	Surf_Tube	Clamps

1. Open the Interfaces panel by clicking **BCs** > **Create** > **Interfaces** from the menu bar.
2. Open the create subpanel.
3. In the name field, enter `tube_clamps`.
4. Click **type =**, and select **CONT_SURFACE_NODE** to create a surface to node contact.
5. Choose a color for the interface.
6. Click **create**.
7. Open the **add** subpanel.
8. Set the master switch to **csurfs**.
9. Using the contactsufs selector, select **surf_tube**.
10. Click **update**.
11. Set the slave switch to **sets**.
12. Using the sets selector, select **Clamps**.
13. Click **update**.

14. Click **review**.

    You should now be able to see if you have set up master and slave surfaces correctly for your contact definition. Master surfaces display in blue, and slaves display in red. If one (or both) do not display, then you might have missed clicking update. If this happens, select the missing entity (master or slave) again and click **update**.

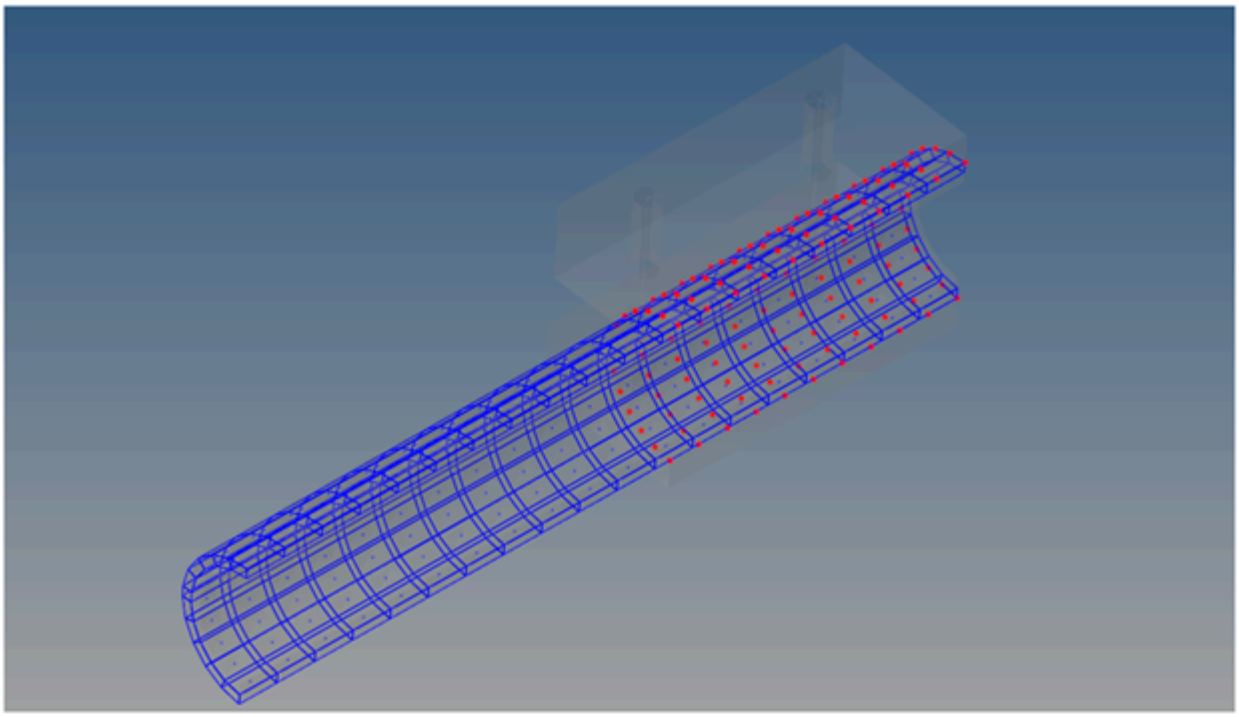


Figure 1208:

15. Click **return** to close the panel.
16. In the Model Browser, Component folder, turn on the display of **Beam**, **MPC Rigid**, and **MPC Same**.
17. From the menu bar, click **BCs** > **Create** > **Interfaces**.
18. Open the **create** subpanel.
19. In the name field, enter `Bolt_1`.
20. Click **type =**, and select **CONT_NODE_NODE** for a node to node contact.
21. Choose a color for the interface.
22. Click **create**.
23. Open the **add** subpanel.
24. Set the master switch to **sets**.
25. Using the sets selector, select **SET_11**.
26. Click **update**.
27. Set the slave switch to **sets**.
28. Using the sets selector, select **SET_12**.
29. Click **update**.

30. Click **review**.

31. Rotate the model to view the interface.

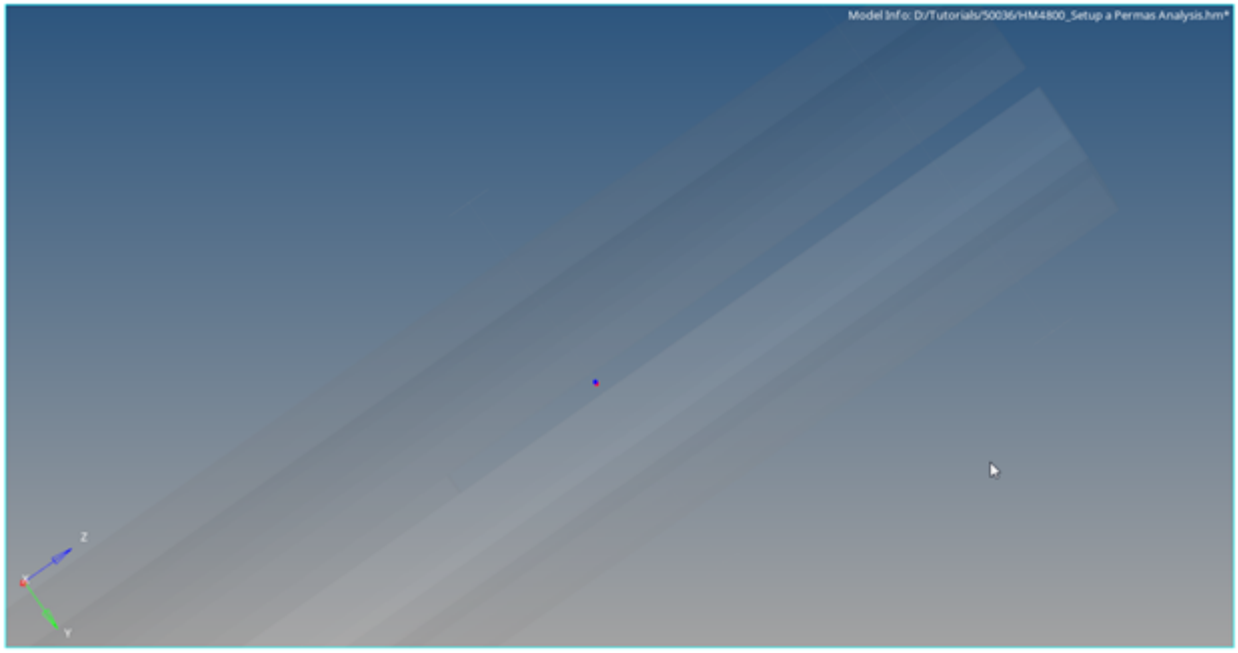


Figure 1209:

32. Click **return** to close the panel.

33. Create the Bolt_2 interface using the following criteria:

a) Set Interface type to **CONT_NODE_NODE**.

b) Set Master set to **SET_21**.

c) Set Slave set to **SET_22**.

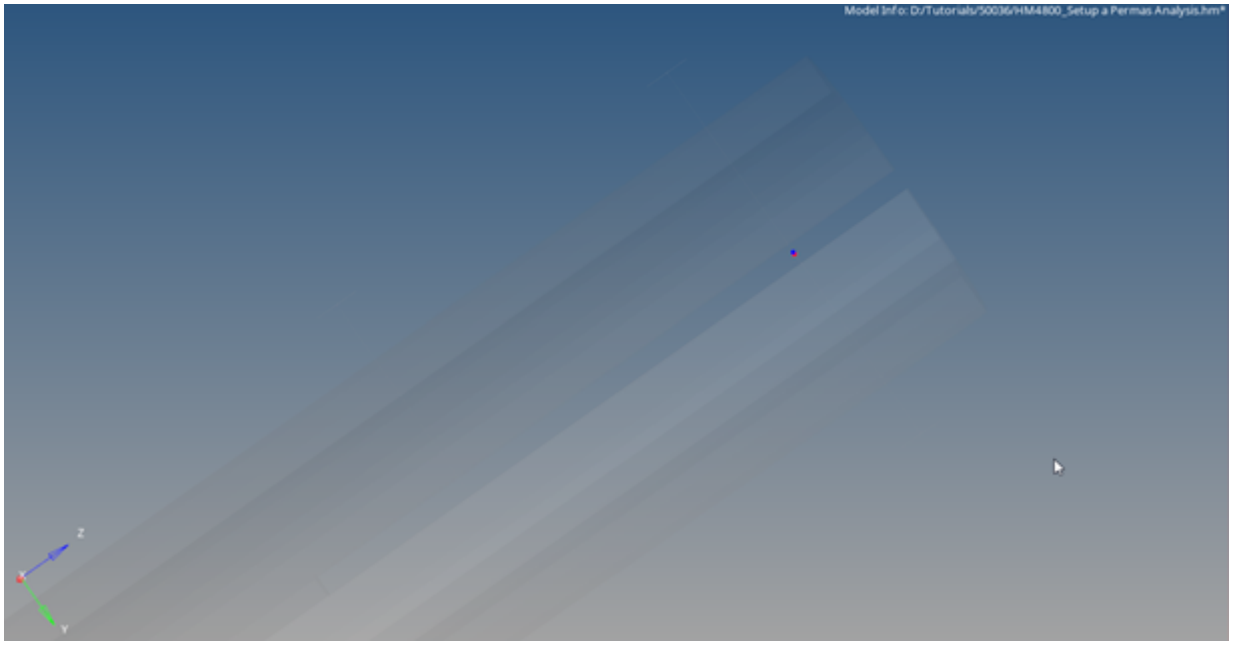


Figure 1210:

34. Isolate the display of the tube component and create the Gap interface using the following cirteria:

a) Set Interface type to **CONT_NODE_NODE**.

b) Set Master set to **SET_1**.

c) Set Slave set to **SET_2**.

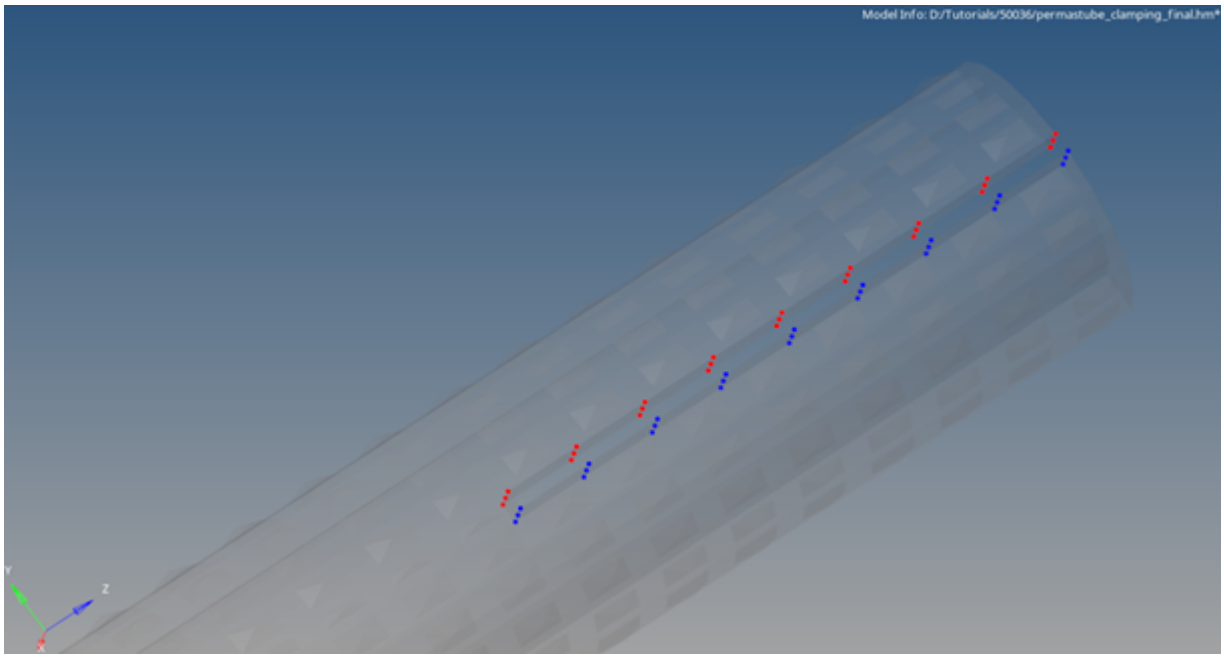


Figure 1211:

35. In the Model Browser, Group folder, right-click on **Gap** and select **Card Edit** from the context menu.
36. In the card image of the $CONTACT card, click **CONTSYS** and select **DIRECT**.

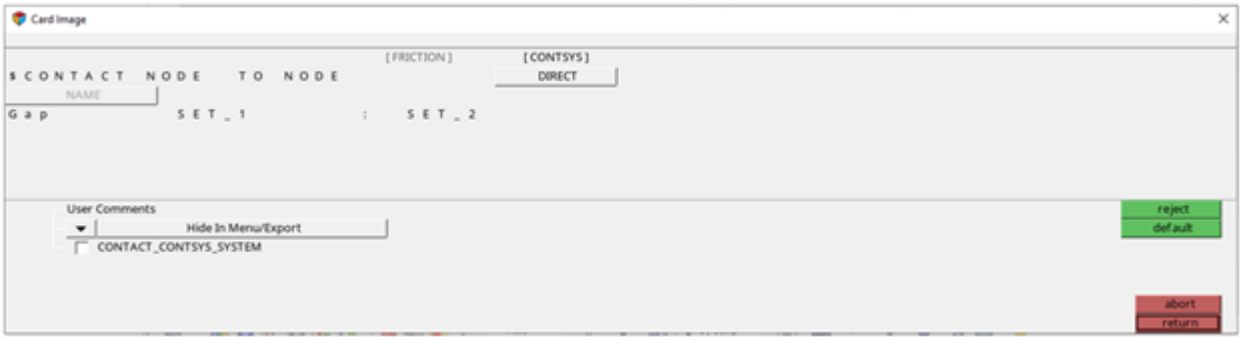


Figure 1212:

37. Click **return**.
38. Repeat steps 35 and 36 for the Bolt_1 and Bolt_2 interfaces.

9.1.13 Create the Boundary Conditions

In this step, you will create constraints and group them into one load collector. You will also create a $CONSTRAINTS variant (load step in HyperMesh) and attach the load collector with the previously created contact constraints.

1. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.
2. Define settings in the **Create Load Collector** dialog.
   a) For Name, enter `Boundary`.
   b) Select a color for the load collector.
   c) Set the card image to **SUPPRESS**.
   d) Click **Create**.
3. On the Standard Views toolbar, click  .
4. Open the Constraints panel by clicking **BCs** > **Create** > **Constraints** from the menu bar.
5. Clear the **dof4**, **dof5**, and **dof6** checkboxes.
6. Verify that the Clamps1 component is displayed.
7. While pressing **Shift**, draw a box around the bottom row of elements to select the nodes.

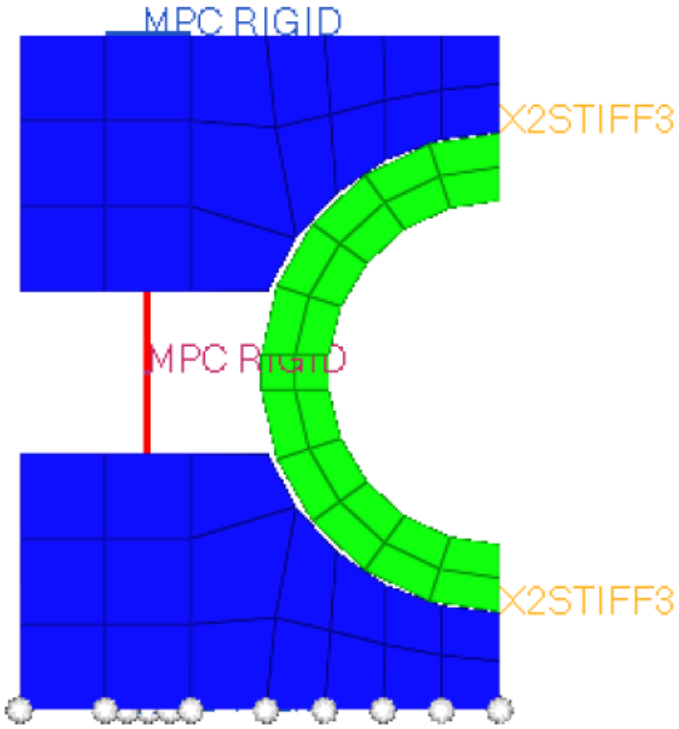


Figure 1213:

8. Rotate the model so that the underside of the bottom clamp is shown.

9. Right-click on the nodes around the MPC Rigids to deselect them.

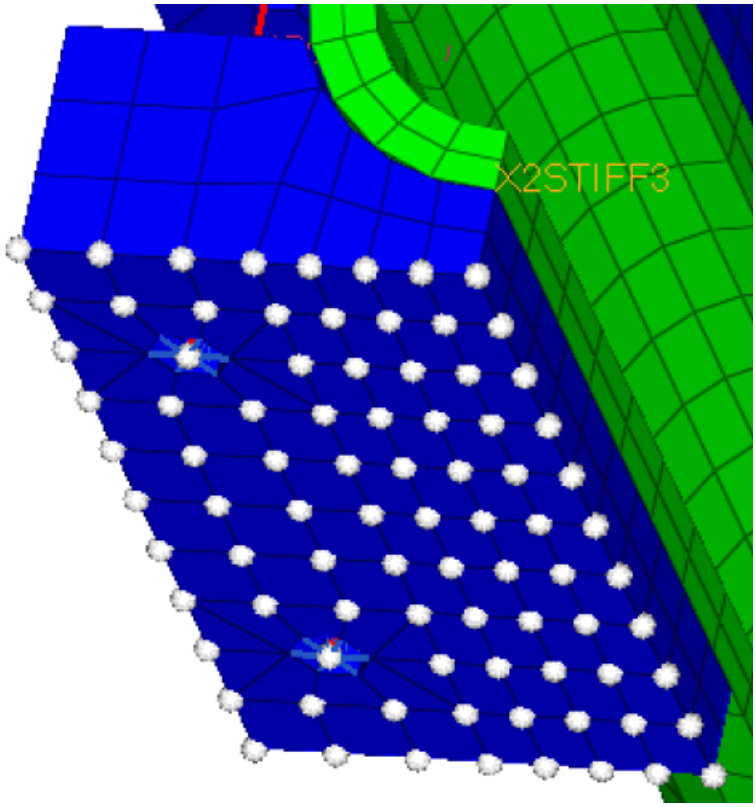


Figure 1214:

10. Deselect the nodes surrounding the other MPC Rigid on this side of the clamp.

11. Click **create**.

12. In the Model Browser, Component folder, display the **tube** component.

13. In the size field, enter `1.0` to reduce the size of the constraints in the graphics area.

14. While pressing **Shift**, draw a box around the far end of the tube to select those nodes in order to include them in the constraint.

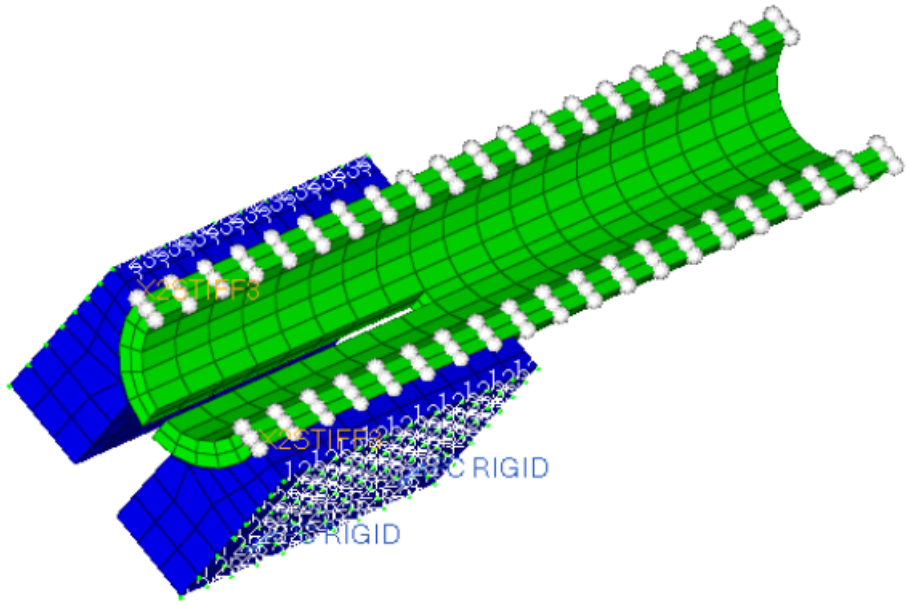


Figure 1215:

15. Click **nodes** > **on plane**.
16. Select three nodes on the narrow edge of the tube, and then click **select entities**.
    All of the nodes on that plane highlight.

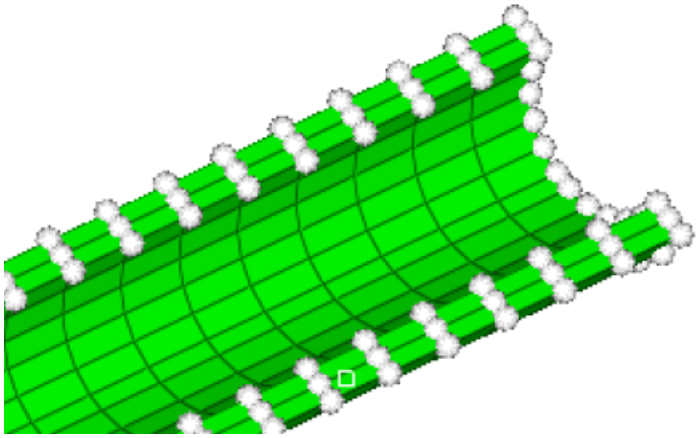


Figure 1216:

17. Click **create** to create the constraints.
18. Click **return** to exit the panel.
19. Turn on the display of all of the components.

All of the constraints have been created.

9.1.14 Assign the Boundary Conditions to a Load Step

In this step, you will assign the boundary conditions to a load step.

1. From the menu bar, click **Setup** > **Create** > **Load Steps**.
2. In the name field, enter `Con_1`.
3. Using the loadcols selector, select **Boundary**.
4. Using the groups selector, select all four groups.
5. Click **create**.
6. Click **edit**.
7. In the card image, set AnalysisProcedure to **CONSTRAINTS**.

   Note: If the Analysis Procedure field is not set to constraints, the load collectors are included, but will not be exported.

8. Click **return** twice to close the panels.

   Note: You can only attach a load collector card image SUPPRESS to a load step with analysis procedure CONSTRAINTS and a load collector with card image LOADS in one with analysis procedure LOADING. If this is not followed correctly your loads and boundary conditions will not be exported. Also, you will receive a warning in the card image of the load step as well as a comment in the exported deck.

9.1.15 Define the Load Collector and Contact Property

In this step, you will create one load pattern (LPAT) for a force applied to the tip of the tube as well as the contact properties for the contacts.

In HyperMesh a load pattern is represented by a load collector with card image LOADS. Finally these load patterns will be included into a $LOADING variant. The LOADING variant is mapped to a load step in HyperMesh.

1. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.
2. Define settings in the **Create Load Collector** dialog.
   a) For Name, enter `Single_Load`.
   b) Select a color for the load collector.
   c) Set card image to **LOADS**.
   d) Click **Create**.
3. Open the Forces panel by clicking **BCs** > **Create** > **Forces** from the menu bar.
4. Using the nodes selector, select the node.

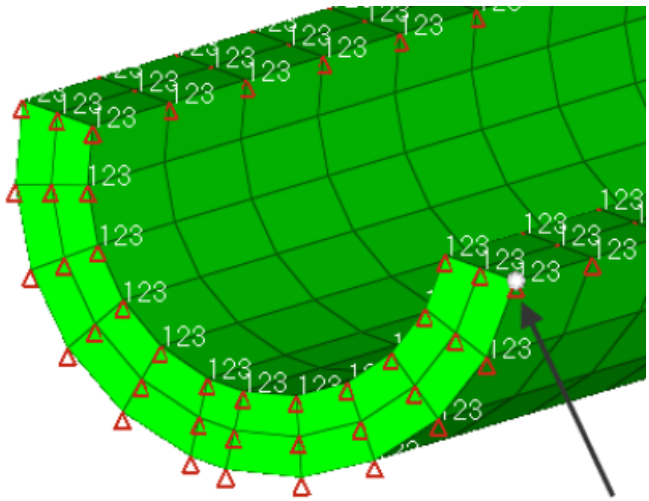


Figure 1217:

5. Set the orientation selector to **y-axis**.
6. In the magnitude= field, enter `-12000`.
7. Click **create**.
8. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.
9. Define settings in the **Create Load Collector** dialog.
   a) For Name, enter `Pretension_Bolt1`.
   b) Select a color for the load collector.
   c) Set card image to **LOADS**.
   d) Select the **Card edit loadcollector upon creation** checkbox.
   e) Click **Create**.
10. In the card image, select the **CONTVAL** checkbox.
11. Under No_of_Contacts, enter `1`.
12. Double-click **contid(1)** and select **Bolt_1**.
13. Click **GAPWIDTH** and select **ABS**.
14. Click **NORMAL** and select **FORCE_LOCK**.
15. In the ContactForce(1) field, enter `10000`.



Figure 1218:

16. Click **return** to close the card image.
17. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.
18. Define settings in the **Create Load Collector** dialog.
    a) For Name, enter `Pretension_Bolt2`.
    b) Select a color for the load collector.
    c) Set Same as to **Pretension_Bolt1**.
    d) Click **Create**.
19. In the card image, double-click **contid(1)** and select **Bolt_2**.
20. Click **return** to close the card image.
21. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.
22. Define settings in the **Create Load Collector** dialog.
    a) For Name, enter `Gap_Tube_Clamps`.
    b) Select a color for the load collector.
    c) Set card image to **LOADS**.
    d) Select the **Card edit loadcollector upon creation** checkbox.
    e) Click **Create**.
23. In the card image, select the **CONTVAL** checkbox.
24. Under No_of_Contacts, enter `2`.
25. Double-click **contid(1)** and select **Gap**.
26. Double-click **contid(2)** and select **tube_clamps**.
27. Click **GAPWIDTH** and select **ABS**.
28. Click **NORMAL** and select **CALCULATE**.
29. Click **return** to close the card image.

9.1.16 Define the $NLLOAD Cards

In this step you will create a $NLLOAD card to specify how individual load patterns are acting over an artificial time.

1. Open the Load Steps panel by clicking **Setup** > **Create** > **Load Steps** from the menu bar.
2. In the name field, enter `NLLOAD`.
3. Using the loadcols selector, select the following load collectors: **Single_Load**, **Pretension_Bolt1**, **Pretension_Bolt2**, and **Gap_Tube_Clamps**.
4. Click **create**.
5. Click **edit** to open the card image.
6. Define settings in the card image.
   a) Select the **NLLOAD** checkbox.
   b) In the TimeSteps field, enter `4`.
   c) In the NoOfLPATS field, enter `4`.
   d) Using the LPAT selectors, select the following load collectors:
      - For LPAT(1), select **Gap_Tube_Clamps**.
      - For LPAT(2), select **Pretension_Bolt1**.
      - For LPAT(3), select **Pretension_Bolt2**.
      - For LPAT(4), select **Single_Load**.
   e) Enter the information for each of the load patterns.

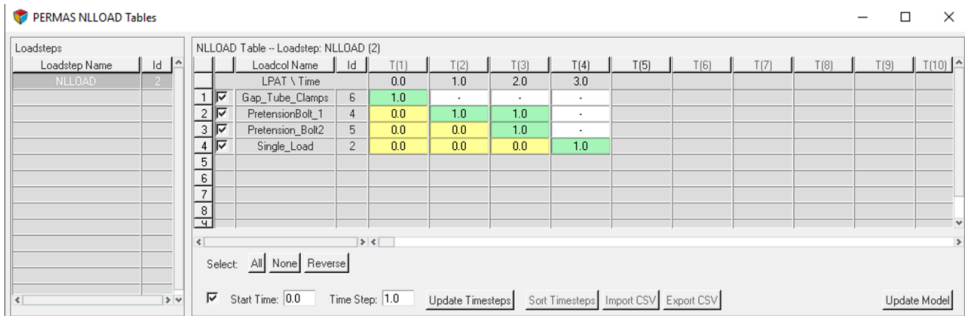


Figure 1219:

> **Note:** Time values should be between 0 and 1.

7. Click **return** twice to close the panels.

> **Tip:** Use the Plot NNLOAD tool, located on the Utility menu, to review the contents of the card. You can see the load history of each load pattern plotted as a graph. Also, you can edit values within the table of the tool. See the topic Creating an NLLOAD Card in the HyperMesh online help for more information.The entry fields for the NLLOAD entries require real numbers, and cannot understand hyphens, which are used commonly by PERMAS users. However, to define a hyphen for export you can write -999 which will display as '-' in the exported deck. This also applies to imported files. A hyphen will convert to '-999' on import.

The NLLOAD card is defined.

9.1.17 Define the $SITUATION Card

In this step, you will build a $SITUATION card which specifies which loads, boundary conditions, and system definitions are combined for this analysis.

1. Open the Load Steps panel.
2. In the name field, enter `SIT_NLLOAD`.
3. Click **create**.
4. Click **edit** to open the card image.
5. Define the card image settings.
   a) Set AnalysisProcedure to **SITUATION**.
   b) Using the CONSTRAINTS selector, select **Con_1**.
   c) Using the LOADING selector, select **NLLOAD**.
6. Click **return** twice to close the card image and the panel.

9.1.18 Export the Deck

In this step, you will export the deck as a `.dat` file.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
   The Export - Solver Deck tab opens.
2. In the File field, enter `tube_clamp_final.dat`.
3. Next to Export Options, click .
4. Set Export to **All**.
5. Click **Export**.

9.1.19 Acknowledgements

Altair thanks INTES for their assistance and support during the creation of this tutorial.

Samcef

10

Samcef tutorials.

This chapter covers the following:

- 10.1 HM-4850: Introduction to Samcef (p. 973)

10.1 HM-4850: Introduction to Samcef

In this step you will: define the hypothesis on the elements; create isotropic materials, shell properties, and loads and load cases; assign properties and materials to elements; and export the model.

Before you begin, copy the `hook.hm` from `<hm.zip>/interfaces/samcef/` to your working directory.

10.1.1 Load the Samcef User Profile

In this step, you will load the Samcef profile in HyperMesh.

1. Start HyperMesh Desktop.
2. In the **User Profiles** dialog, set the user profile to **Samcef**.

10.1.2 Retrieve and View the Model File

In this step, you will retrieve and view the model file in HyperMesh.

1. Open a model file by completing one of the following options:
   - Click **File** > **Open** > **Model** from the menu bar.
   - Click  on the Standard toolbar.
2. In the **Open Model** dialog, open the `hook.hm` file.
3. On the Visualization toolbar, click  to shade the models elements and mesh lines, if they are not already shaded.
   The model appears in the graphics area.

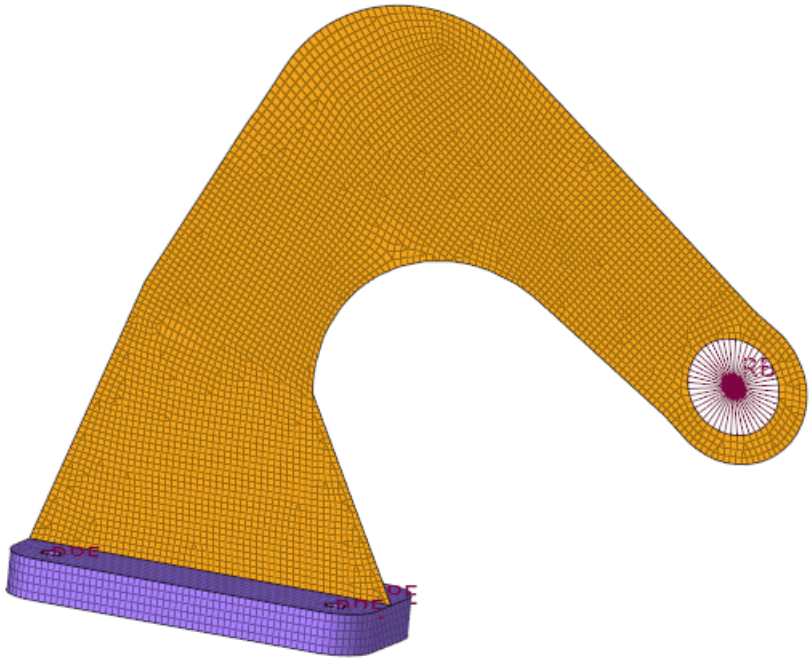


Figure 1220:

10.1.3 Check the Hypothesis

In this step, you will check the Samcef hypothesis using the 2D & 3D subpanel.

1. Open the Element Types panel by clicking **Mesh** > **Assign** > **Element Type** from the menu bar.
2. Go to the **2D & 3D** subpanel.

   In this panel, you have the link between the keyword of HyperMesh (tria3, quad4, ...) and the hypothesis of Samcef. You can change the hypothesis for each element configuration and also update each element with a different hypothesis.

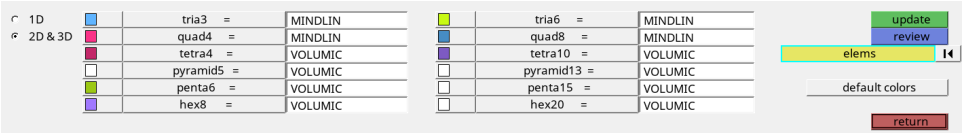


Figure 1221:

3. Click **return** to close the panel.

10.1.4 Create a Material

In this step, you will create a new material named steel.

1. In the Model Browser, right-click and select **Create** > **Material** from the context menu.

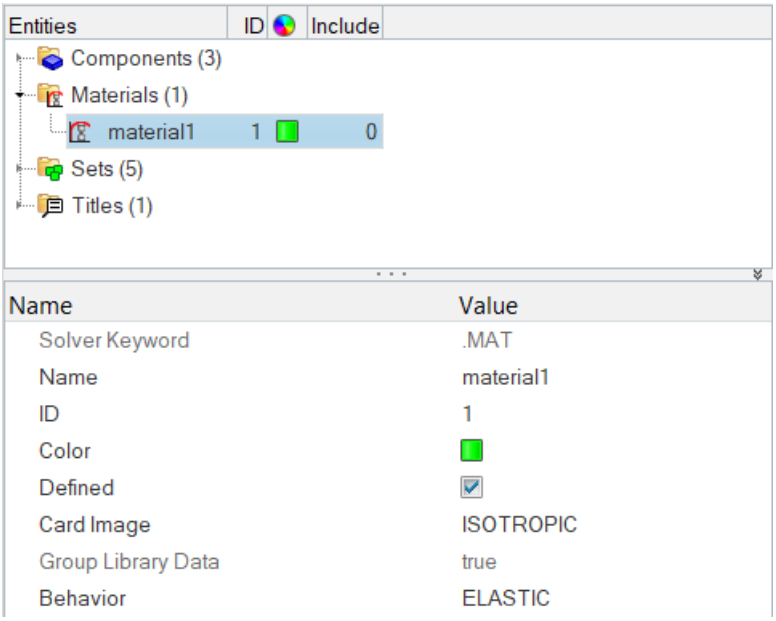


Figure 1222:

A new material opens in the Entity Editor.

2. Define the material properties in the Entity Editor.
   a) For Name, enter `steel`.
   b) Set card image to **ISOTROPIC**.

   > **Note:** You can create different types of elastic material (anisotropic, isotropic or orthotropic).

   c) Select a color for the material.
3. In the Model Browser, Property folder, right-click on **steel** and select **Card Edit** from thecontext menu.
   The card image opens.
4. Define the properties in the card image.
   a) For YT, enter `2.1e5`.
   b) For NT, enter `0.3`.
   c) For M, enter `7.9e-9`.

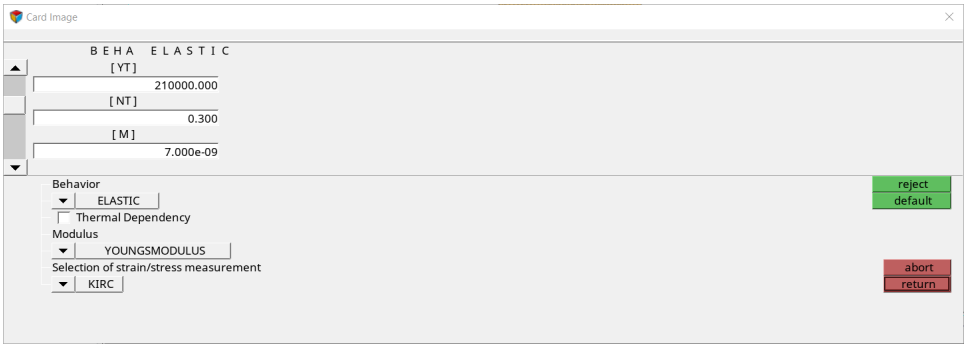


Figure 1223:

5. Click **return** to exit the card image.

10.1.5 Create and Assign Properties

In this step, you will create the properties and assign them to the elements.

1. In the Model Browser, right-click and select **Create** > **Property** from the context menu.

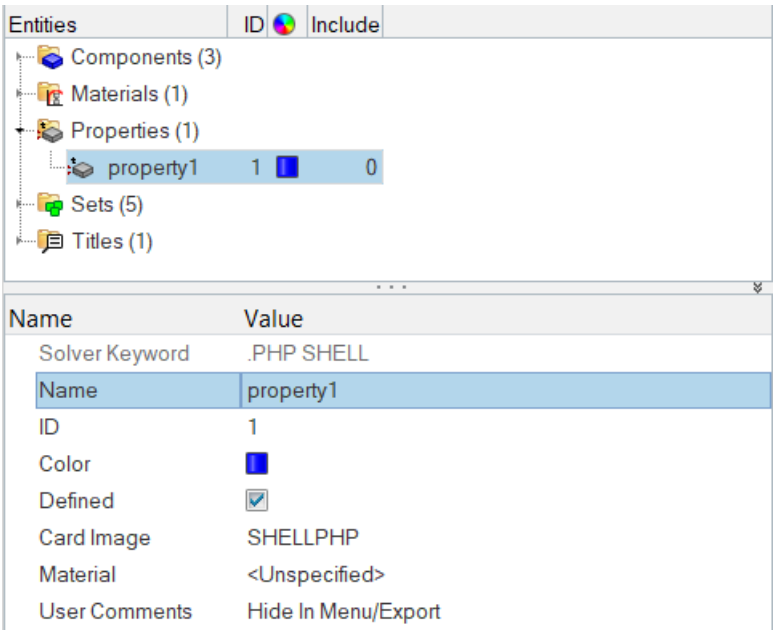


Figure 1224:

A new property opens in the Entity Editor.

2. Define the property properties in the Entity Editor.
   a) For Name, enter `shell`.
   b) Set card image to **SHELLPHP**.
   c) Select a color for the property.
   d) For Material, click **Unspecified** > **Material**.

e) In the **Select Material** dialog, select **steel** and then click **OK**.

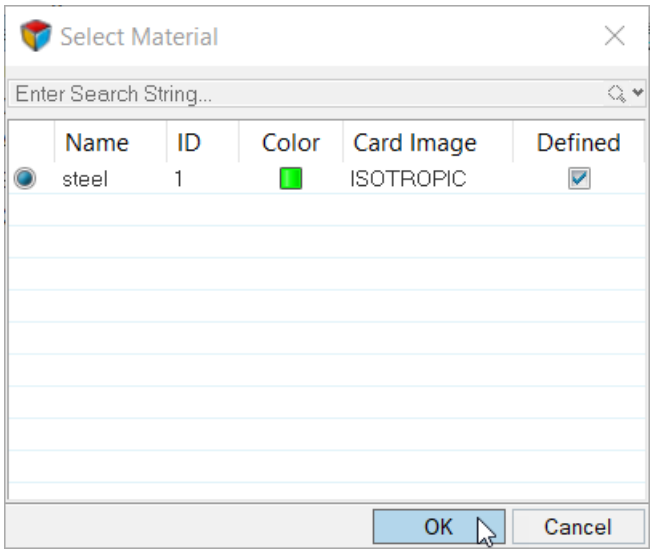


Figure 1225:

3. In the Model Browser, Property folder, right-click on **shell** and select **Card Edit** from thecontext menu.
4. Define settings in the card image.
   a) In the card image, select the **THICK** checkbox.
   b) In the THICK field, enter `1.5`.

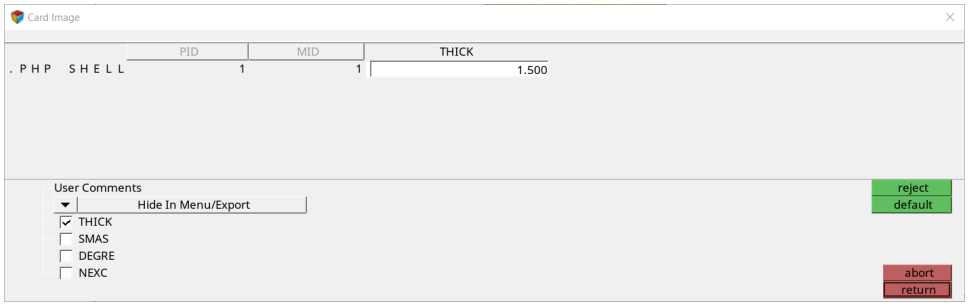


Figure 1226:

   c) Click **return** to close the card image.
5. In the Model Browser, Component folder, right-click on **shell** and select **Assign** from the context menu.
6. Define settings in the **Assign to Components** dialog.
   a) Set Property to **shell**.
   b) Set Material to **steel**.
   c) Click **OK**.

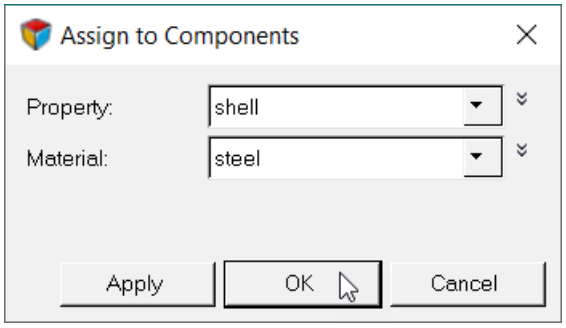


Figure 1227:

The property is now assigned to the elements.

7. Repeat step 1.
8. Define the property settings in the Entity Editor.
   a) For name, enter `solid`.
   b) Set card image to **SOLIDMAT**.
9. In the Model Browser, Component folder, right-click on **solid** and select **Assign** from the context menu.
10. Define settings in the **Assign to Components** dialog.
    a) Set Property to **solid**.
    b) Set Material to **steel**.
    c) Click **OK**.
    The property is now assigned to the elements.
11. Ensure the properties are correctly assigned to the elements by clicking By Prop .
    The elements should have the same color as the properties.

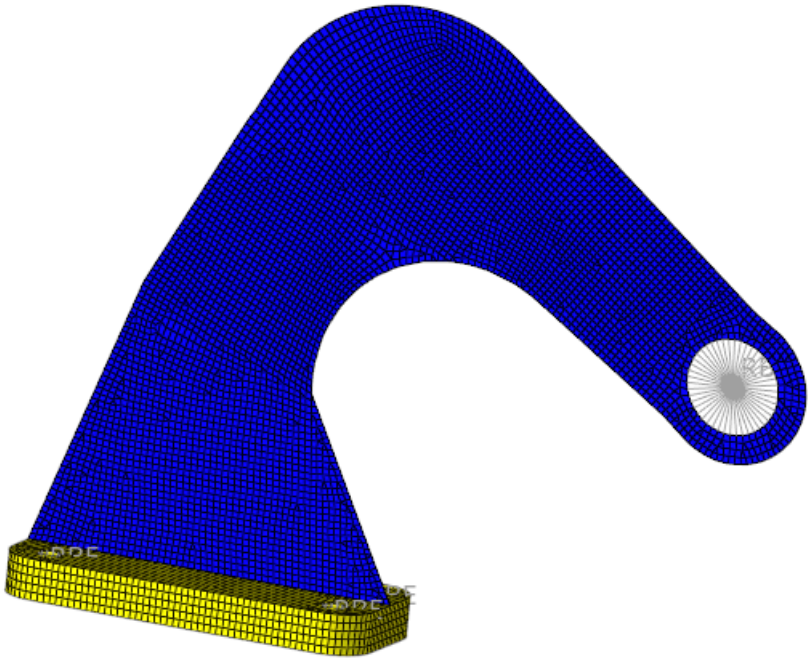


Figure 1228:

10.1.6 Create Load Collectors

In this step, you will create two load collectors named spc and force.

1. In the Model Browser, right-click and select **Create** > **Load Collector** from the context menu.

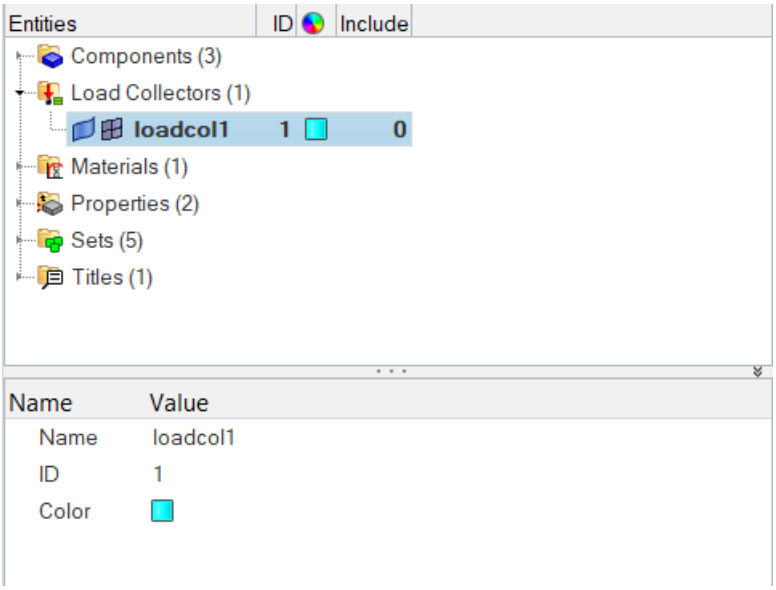


Figure 1229:

A new load collector opens in the Entity Editor.

2. Define the load collector properties in the Entity Editor.
   a) For Name, enter `spc`.
   b) Select a color for the load collector.
3. Repeat step 1.
4. Define the load collector properties in the Entity Editor.
   a) For Name, enter `force`.
   b) Select a color for the load collector.

10.1.7 Create Constraints

In this step, you will create constraints.

1. In the Model Browser, Load Collectors folder, right-click on **spc** and select **Make Current** from the context menu.
   This sets the spc load collector as the current load collector.
2. Open the Constraints panel by clicking **BCs** > **Create** > **Constraints** from the menu bar.
3. Go to the **create** subpanel.
4. Using the nodes selector, select the main nodes of the rigid elements.

> **Tip:** The main nodes of the rigid elements are defined in the holes of the component solid.

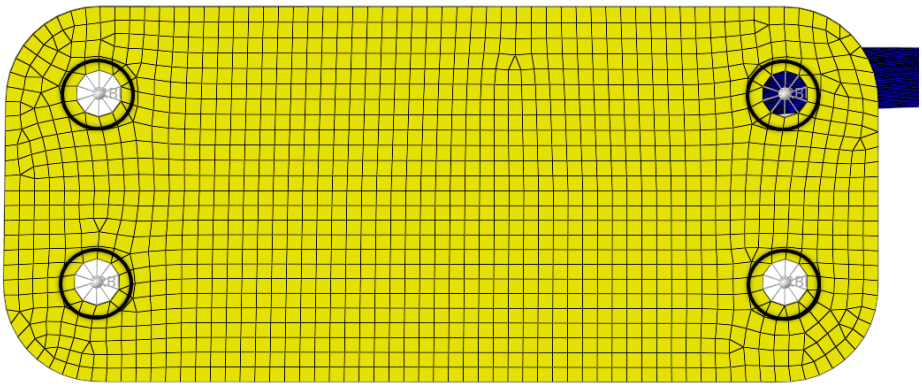


Figure 1230:

5. Constrain **dof1**, **dof2**, **dof3**, **dof4**, **dof5**, and **dof6** by checking the corresponding checkboxes.
6. Set dof1, dof2, dof3, dof4, dof5 and dof6 to a value of `0.0`.

   DOFs with a check will be constrained, while DOFs without a check will be free. DOFs 1, 2 and 3 are x, y, and z translational degrees of freedom. DOFs 4, 5 and 6 are x, y an z rotational degrees of freedom.



Figure 1231:

7. Click **create** to apply the constraints to the selected nodes.
8. Click **return** to close the panel.

10.1.8 Create Forces

In this step, you will create forces.

1. In the Model Browser, Load Collector folder, right-click on **force** and select **Make Current** from the context menu.
   This sets the force load collector as the current load collector.
2. Open the Forces panel by clicking **BCs** > **Create** > **Forces** from the menu bar.
3. Go to the **create** subpanel.
4. Using the nodes selector, select the main nodes of the rigid element as seen in Figure 1232.

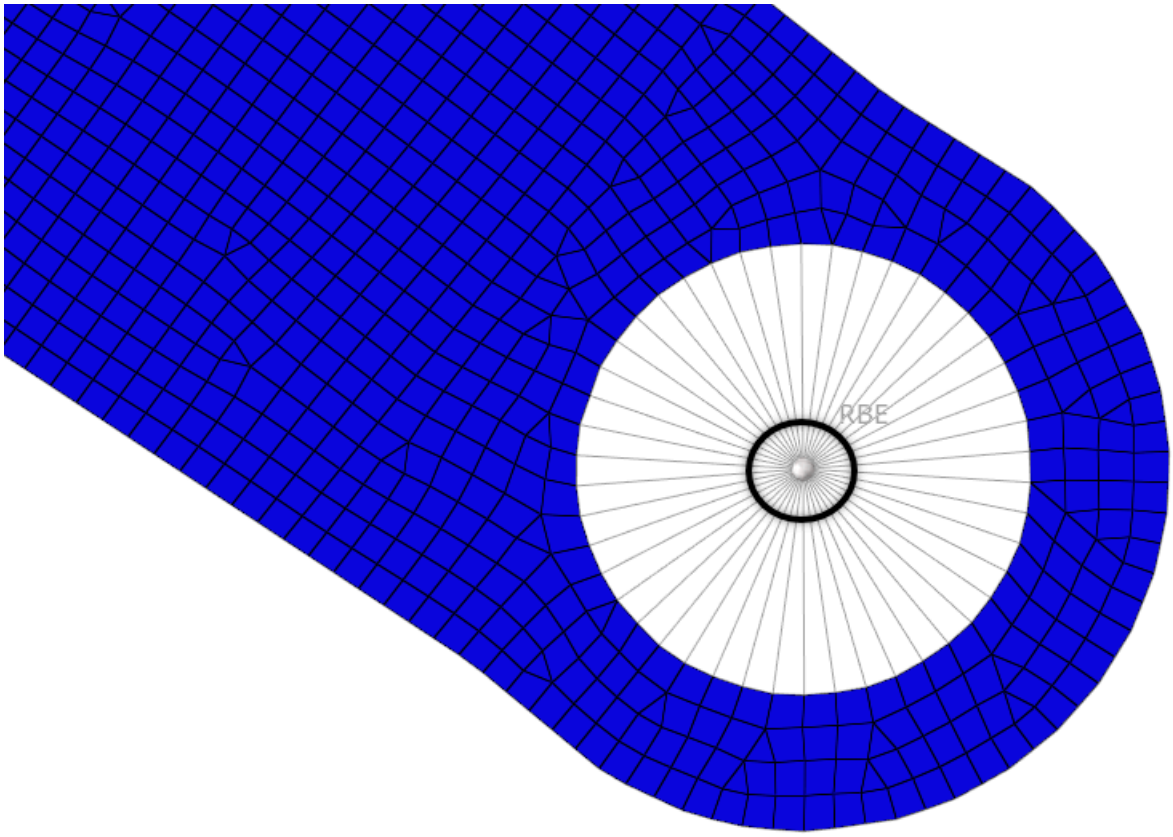


Figure 1232:

5. Set the local system / global system toggle to **global system**.
6. Set the vector definition switch to **constant vector**.
7. In the magnitude = field, enter `250`.
8. Set the orientation selector to **z-axis**.



Figure 1233:

9. Click **create**.
10. Click **return**.

10.1.9 Create a Samcef Database

In this step, you will create a Samcef database using the Load Steps panel.

1. Open the Load Steps panel by clicking **Setup** > **Create** > **Load Steps** from the menu bar.
2. In the name field, enter `z-force`.
3. Using the loadcols selector, select the **force** and **spc** load collectors.
4. Click **create**.
5. Click **return** to close the panel.

10.1.10 Export the File

In this step, you will export the file.

1. From the menu bar, click **File** > **Export** > **Solver Deck**.
   The Export - Solver Deck tab opens.
2. In the File field, enter `hook.dat`.
3. Set template to **Samcef**.
4. Click **Export**.

Index

A

ACU-T: 1000 HyperWorks UI Introduction (HyperMesh) *104*
ACU-T: 2000 Turbulent Flow in a Mixing Elbow (HyperMesh) *143*
ACU-T: 2300 Atmospheric Boundary Layer Problem – Flow Over Building (HyperMesh) *152*
ACU-T: 3000 Enclosed Hot Cylinder: Natural Convection (HyperMesh) *167*
ACU-T: 3100 Conjugate Heat Transfer in a Mixing Elbow (HyperMesh) *186*
ACU-T: 3101 Transient Conjugate Heat Transfer in a Mixing Elbow (HyperMesh) *197*
ACU-T: 3110 Exhaust Manifold Conjugate Heat Transfer - CFD Data Mapping using acuOptiStruct (HyperMesh) *209*
ACU-T: 3200 Radiation Heat Transfer in a Simple Headlamp using the Enclosure Radiation Model (HyperMesh) *234*
ACU-T: 3201 Solar Radiation and Thermal Shell Tutorial (HyperMesh) *255*
ACU-T: 3202 Heat Transfer Between Concentric Spheres – P1 Radiation Model (HyperMesh) *276*
ACU-T: 3203 Heat Transfer Between Concentric Spheres – Discrete Ordinate Radiation Model (HyperMesh) *289*
ACU-T: 3204 Radiation Heat Transfer in a Simple Headlamp using the Discrete Ordinate Model (HyperMesh) *301*
ACU-T: 3300 Modeling of a Heat Exchanger Component (HyperMesh) *324*
ACU-T: 3310 Single Phase Nucleate Boiling (HyperMesh) *334*
ACU-T: 3311 Multiphase Nucleate Boiling Using the Algebraic Eulerian Model (HyperMesh) *350*
ACU-T: 4000 Transient Dam Break Simulation (HyperMesh) *364*
ACU-T: 4001 Water Filling in a Tank (HyperMesh) *375*
ACU-T: 4002 Sloshing of Water in a Tank (HyperMesh) *386*
ACU-T: 4100 Disperse Case – LPipe (HyperMesh) *398*
ACU-T: 4200 Humidity – Pipe Junction (HyperMesh) *413*
ACU-T: 4201 Condensation & Evaporation - Air Box (HyperMesh) *426*
ACU-T: 5000 Centrifugal Air Blower with Moving Reference Frame (Steady) (HyperMesh) *449*
ACU-T: 5001 Blower - Transient (Sliding Mesh) (HyperMesh) *459*
ACU-T: 5100 Modeling of a Fan Component: Axial Fan (HyperMesh) *473*
ACU-T: 5201 Coupled Simulation of a Check Valve using AcuSolve and MotionSolve (HyperMesh) *484*
ACU-T: 6010 Flow Through Porous Medium (HyperMesh) *507*
ACU-T: 7000 Parametric Optimization with AcuSolve (HyperMesh) *519*
ACU-T: 7001 Shape Optimization using HyperMorph (HyperMesh) *540*
ACU-T: 7010 Shape Optimization using HyperStudy (HyperMesh) *561*

D

define *STEP using Abaqus step manager *61*
define Abaqus contacts for 2D models *29*
define Abaqus contacts for 3D models *43*

P

pre-processing for bracket and cradle analysis *82*

S

set up an Abaqus analysis in HyperMesh *17*