

Ansys Software Tutorial

Introduction to Ansys Fluent #7: Compressible Flows

Mitchell Boots

University of Newcastle

Edited by the Ansys Academic Development Team

education@ansys.com

Summary

Ansys Fluent is a comprehensive computational fluid dynamics (CFD) software that allows you to model fluid domains.

In this set of tutorials, we will introduce basic functionalities of Ansys Fluent through the Ansys Workbench interface. Ansys Workbench is the integration and workflow platform that connects Ansys products.

This tutorial will delve into the realm of compressible flow using Ansys Fluent. Specifically, we will explore two distinct case types and illustrate the process of configuring boundary conditions and solution settings required to perform these simulations.

This tutorial is #7 of a seven-part tutorial series that serves as an introduction to Ansys Fluent. Details of the topics covered and the order can be found in the table below. These tutorials build on one another, so it is recommended that they are followed in order. Other tutorials can be found on the [Ansys Education Resources site](#).

Tutorial Order	Tutorial Topic
1	Introduction to Ansys Fluent
2	Mesh Sensitivity
3	Steady State vs. Transient
4	Aerodynamic Analysis Part 1
5	Aerodynamic Analysis Part 2
6	Heat Transfer with Ansys Fluent
7	Compressible Flows

*This tutorial was created using the 2023R1 Student Version of Ansys Workbench. Some screens may look different, depending on your version. Check the [Ansys Learning Forum](#) if you have any questions.

Table of Contents

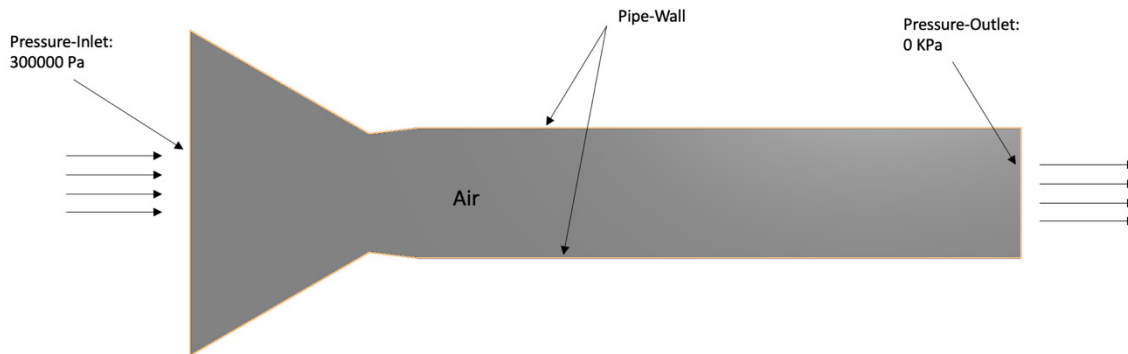
Exercise 1 – 2D Convergent-Divergent Nozzle:	3
Exercise 2 – 2D Simplified High Speed Rocket:	7

In this tutorial, we will delve into the realm of compressible flow using Ansys Fluent. Specifically, we will explore two distinct case types and illustrate the process of configuring boundary conditions and solution settings required to perform these simulations.

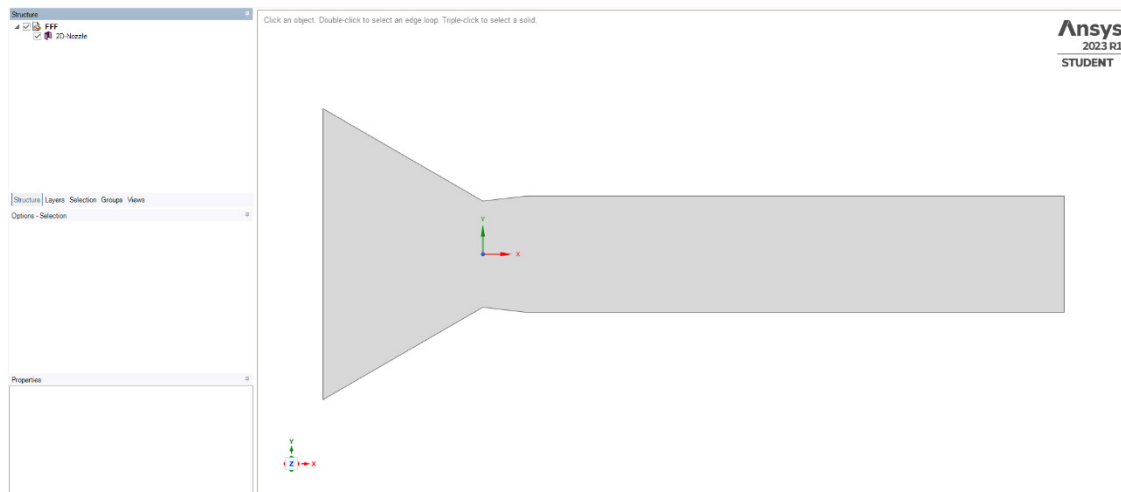
Exercise 1 – 2D Convergent-Divergent Nozzle:

In this exercise, we will analyze compressible flow in a convergent-divergent nozzle. This exercise will follow a similar analysis to the problems addressed previously.

The following image illustrates the solution conditions.

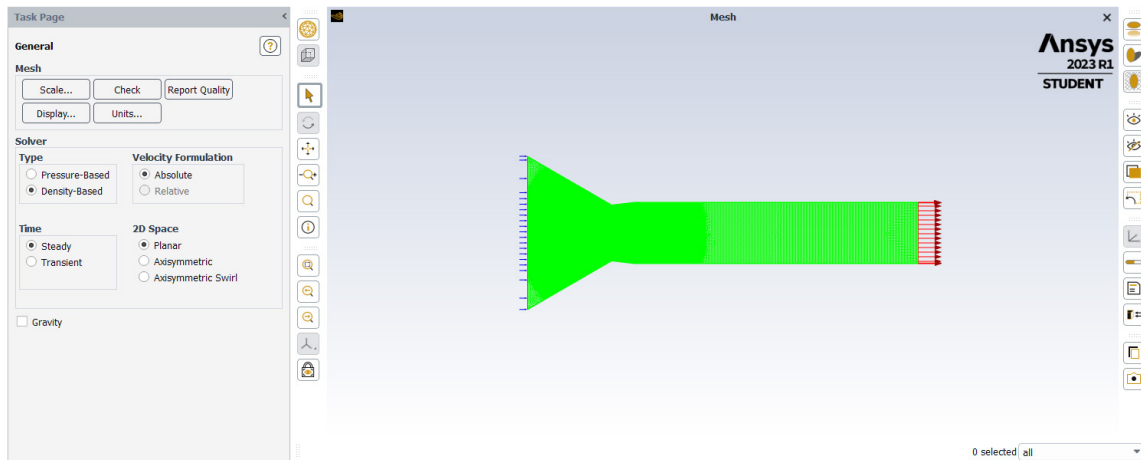


Load the file '2D-Nozzle.step' and open SpaceClaim¹ to verify its correct loading. Once you have confirmed the successful loading, the first step is to apply an appropriate mesh. After you are satisfied with your mesh, proceed to assign the relevant named selections.

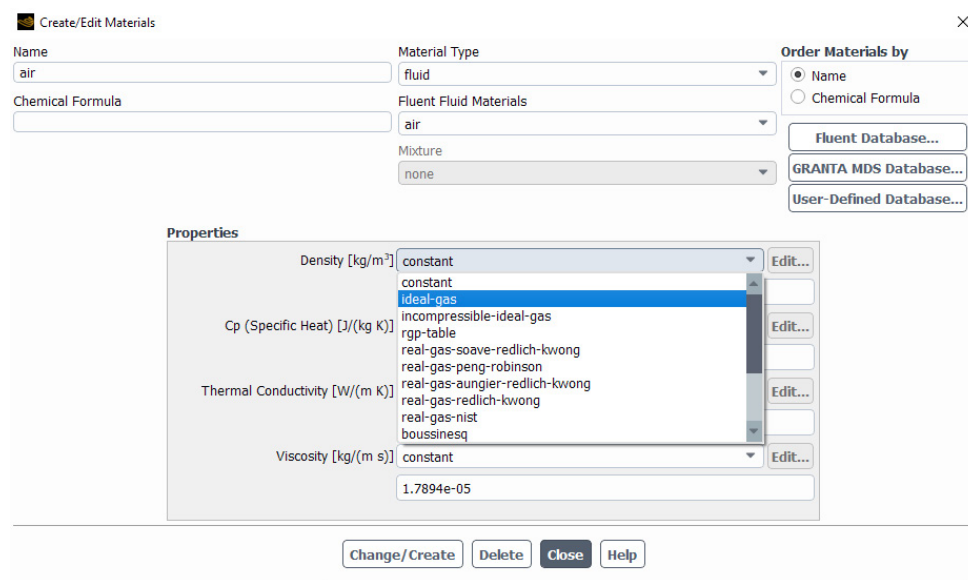


After completing the previous steps, open Fluent. In this simulation, we will introduce some different conditions compared to our usual setups. The initial change required to simulate compressible flow is to enable a density-based simulation in the general settings. Density-based simulation is designed for high-speed compressible flow applications, whereas pressure-based simulation is typically used for incompressible simulations.

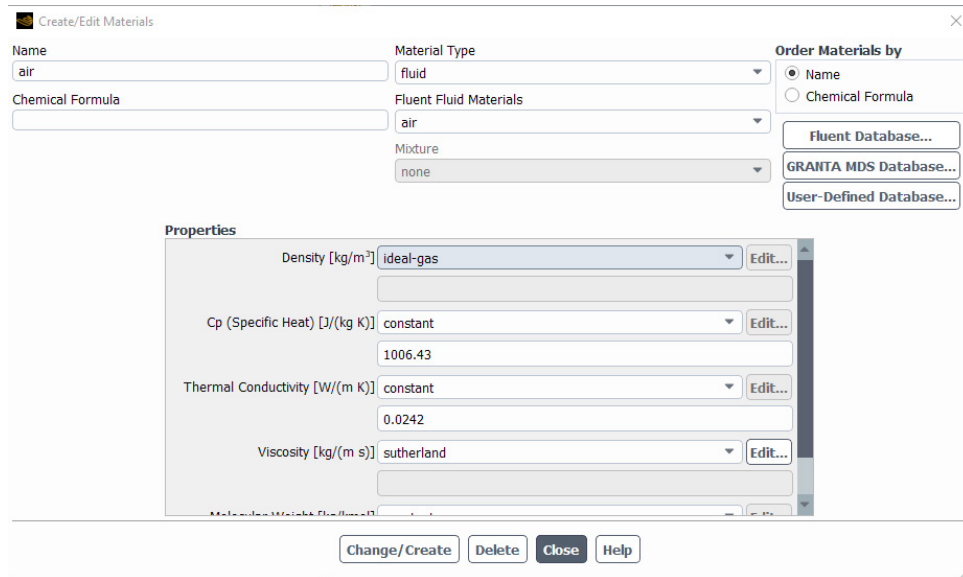
¹ Ansys Discovery is our go forward CAD tool, providing an improved user experience, it can be used to handle CAD instead of SpaceClaim



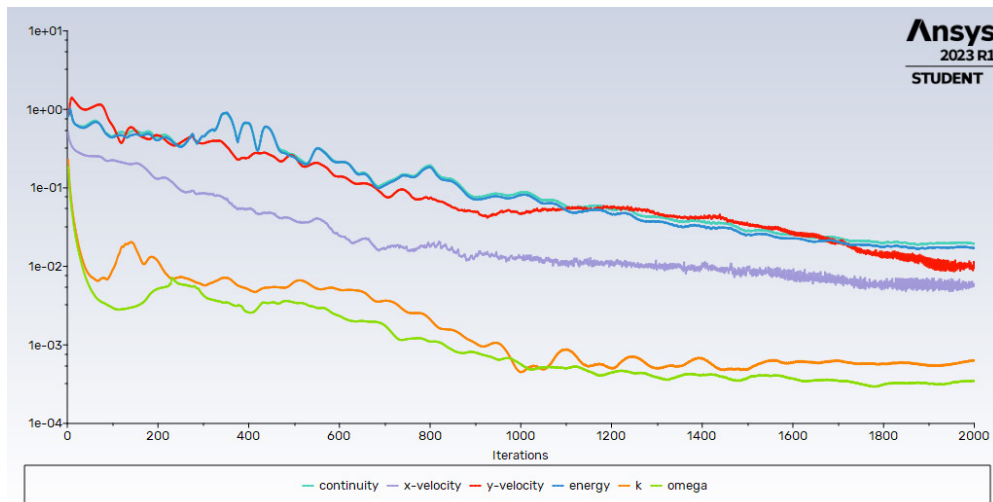
In addition, we will activate the energy equation while keeping the turbulence model set to k-omega SST. Another crucial modification involves adjusting the cell-zone material settings to utilize the ideal gas law for calculating density. It's essential to specify this setting; otherwise, the density will remain constant throughout the simulation.



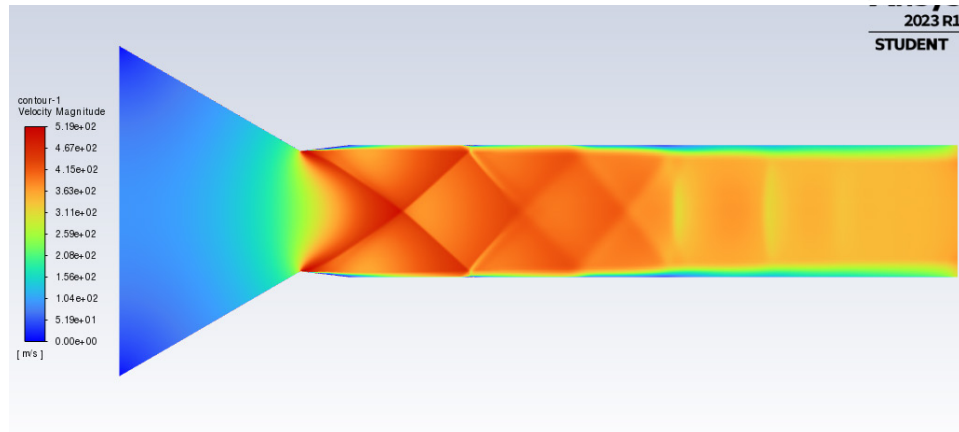
Furthermore, we will customize our viscosity so that it varies with changes in temperature. We have various options for defining this temperature-dependent variation as a function. For today's lab, we will employ the Sutherland model. However, if you have your own expression, whether it's polynomial, linear, or any other mathematical expression, you can input it into Ansys for viscosity modeling.



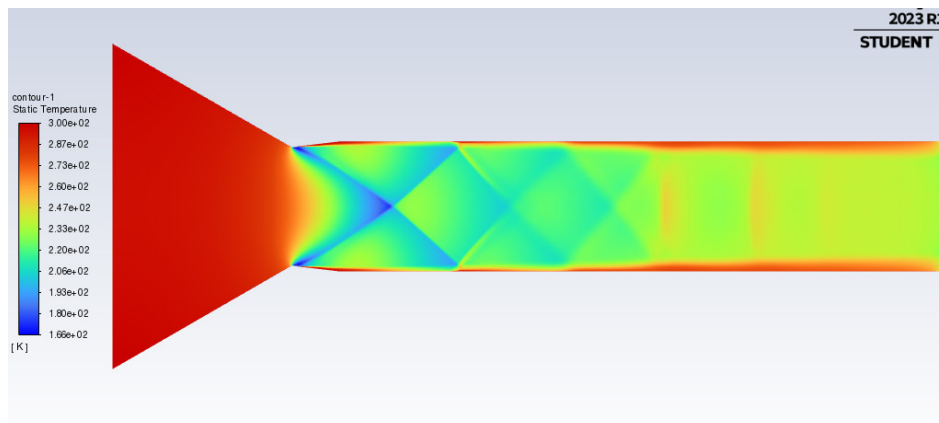
Configure the remaining conditions as indicated in the solution condition image shared earlier. After setting these conditions, initialize the simulation and commence the run. Specify the number of iterations as 2000 to allow the simulation ample time to converge. To confirm convergence, closely monitor the residuals and ensure that they are progressively decreasing toward values below zero.



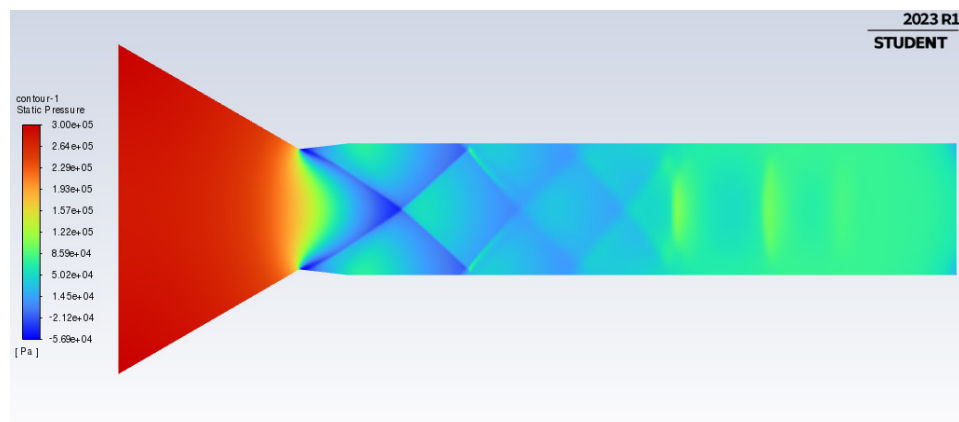
After the simulation has completed, let's examine the velocity contour to visualize the flow within the pipe.



Given that we are simulating compressible flow with the energy equation enabled, we can also analyze the temperature distribution and observe how the temperature varies within a shock wave. This will provide valuable insights into the thermal characteristics of the flow.



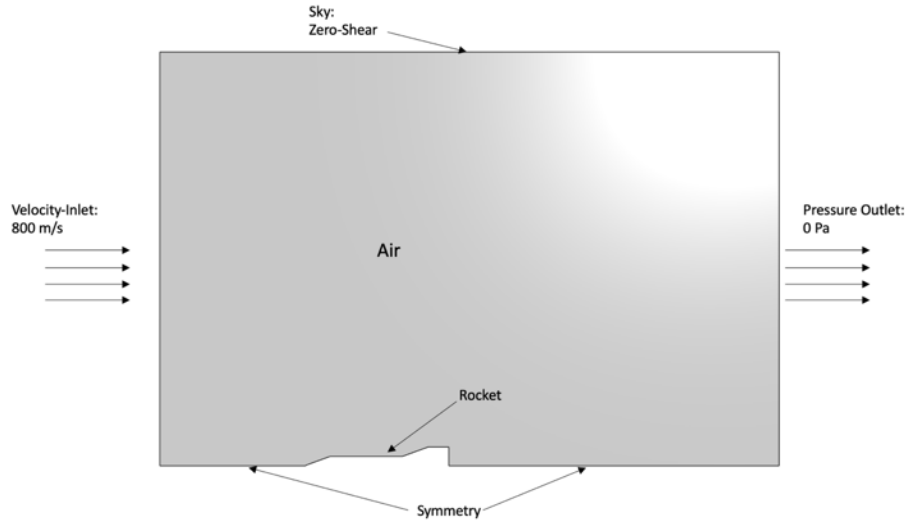
Additionally, we can visualize the pressure contour throughout the simulation. This will help us gain a comprehensive understanding of the pressure distribution within the system, which is crucial for analyzing compressible flow behavior.



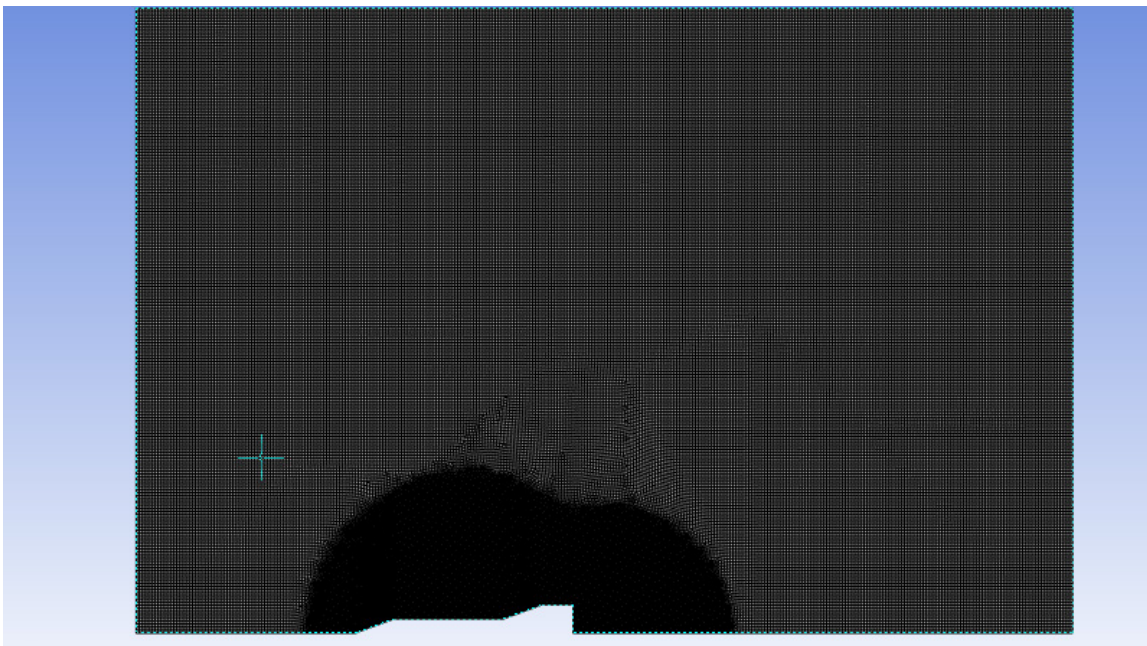
What can you conclude about the flow through this pipe? What would be the effect of increasing the inlet pressure? Are these considered valid?

Exercise 2 – 2D Simplified High Speed Rocket:

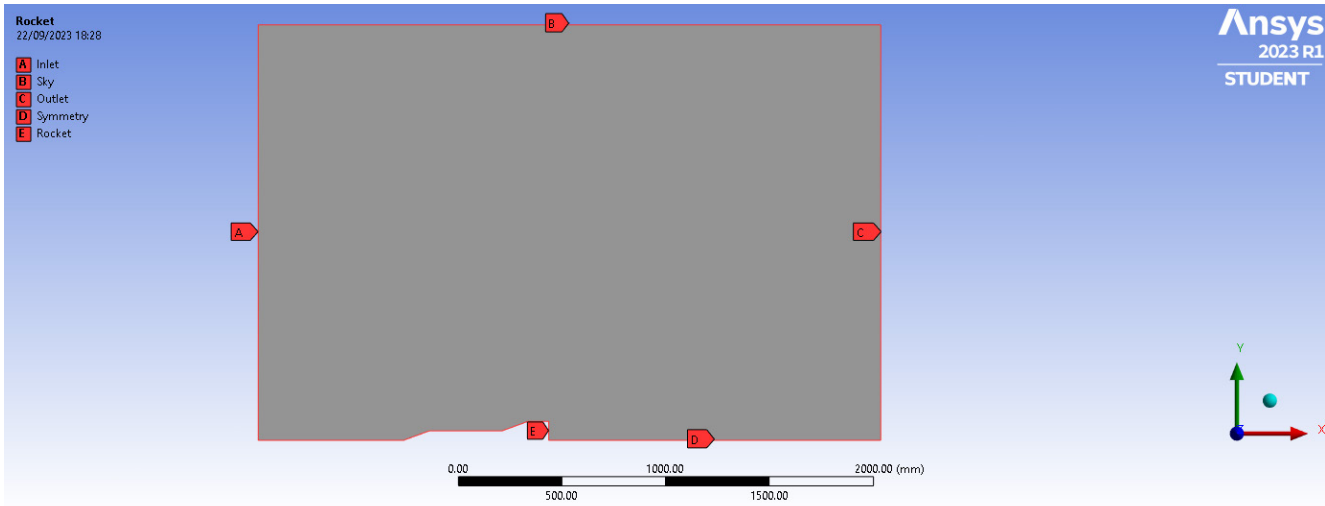
Now, let's proceed with a simplified 2D simulation of a rocket, which will enable us to investigate symmetry and external compressible flows. The following is the case study that we will analyze in this context.



Load the provided 2D surface and verify that it loads into SpaceClaim correctly. Apply an adequate mesh. The following is an example mesh.



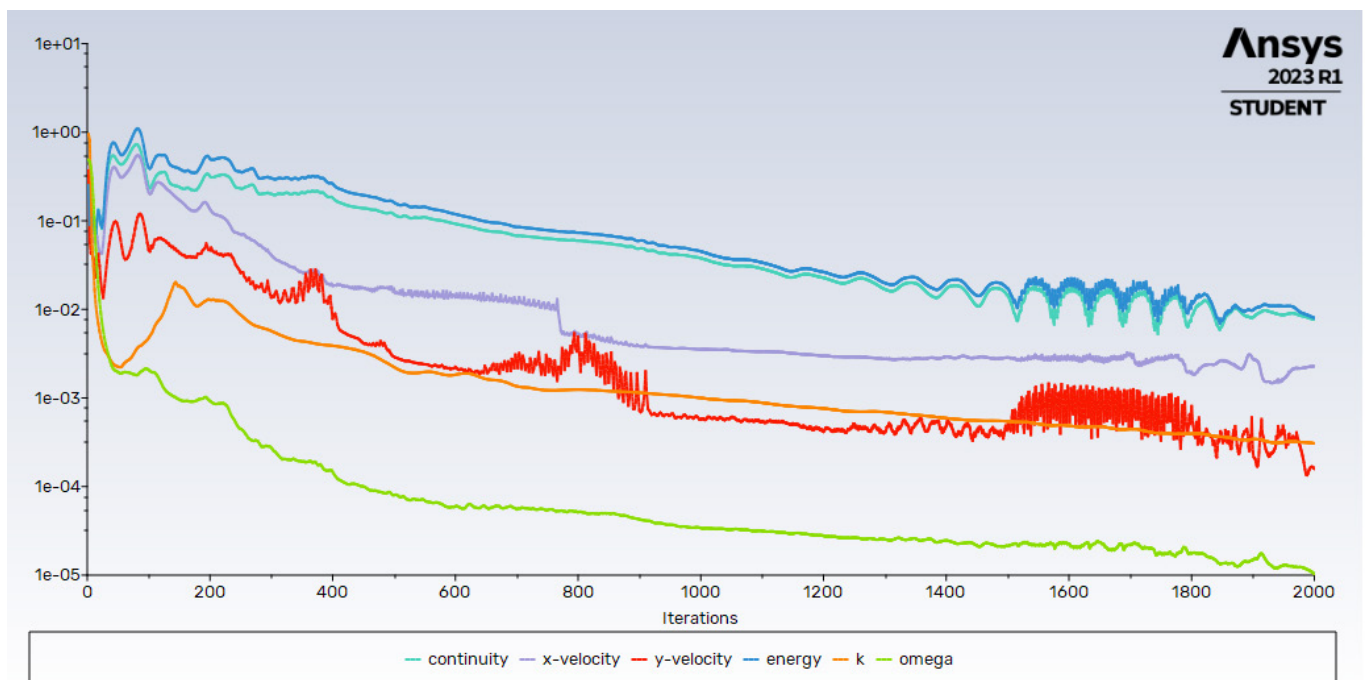
Before closing the mesh tab, ensure that you have applied your named selections. The following image depicts how they should be set up.



Once you're happy with your mesh and named selections move over to fluent, for this simulation we will set the solution up as follows:

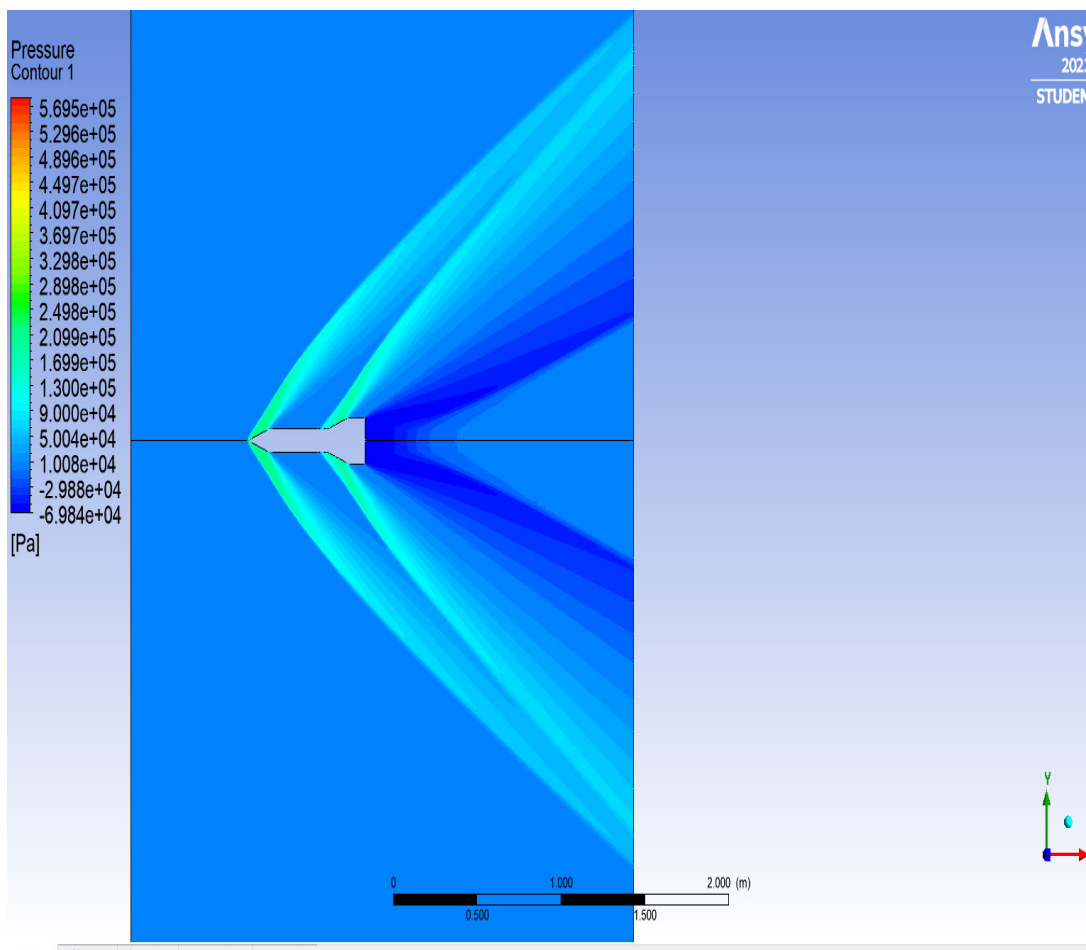
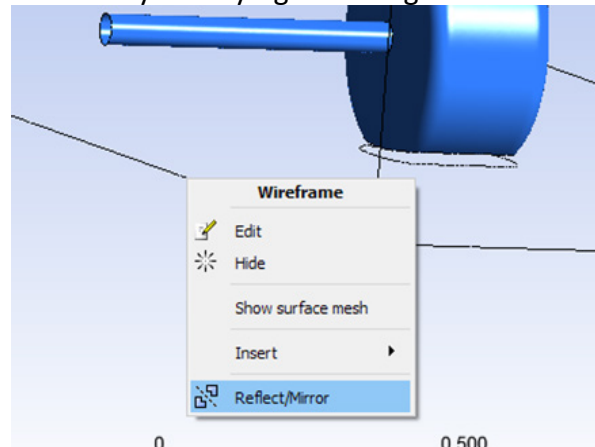
1. Density-based simulation
2. Cell zone - Air
3. Air - Density - Ideal gas
4. Air - Viscosity - Sutherland
5. Velocity Inlet - 800 m/s
6. Sky - Zero Shear

Once you have set your simulation up, initialize the calculation and solve over 2000 iterations. Ensure the residuals are acceptable.

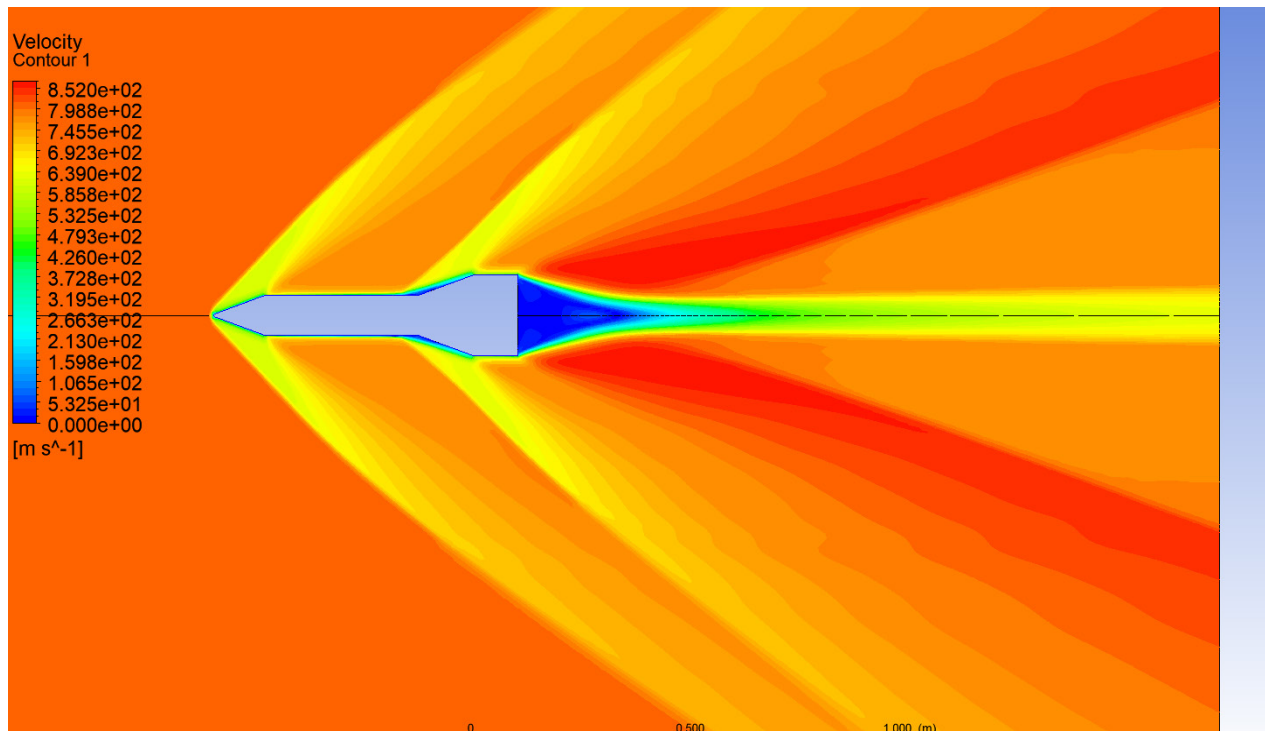
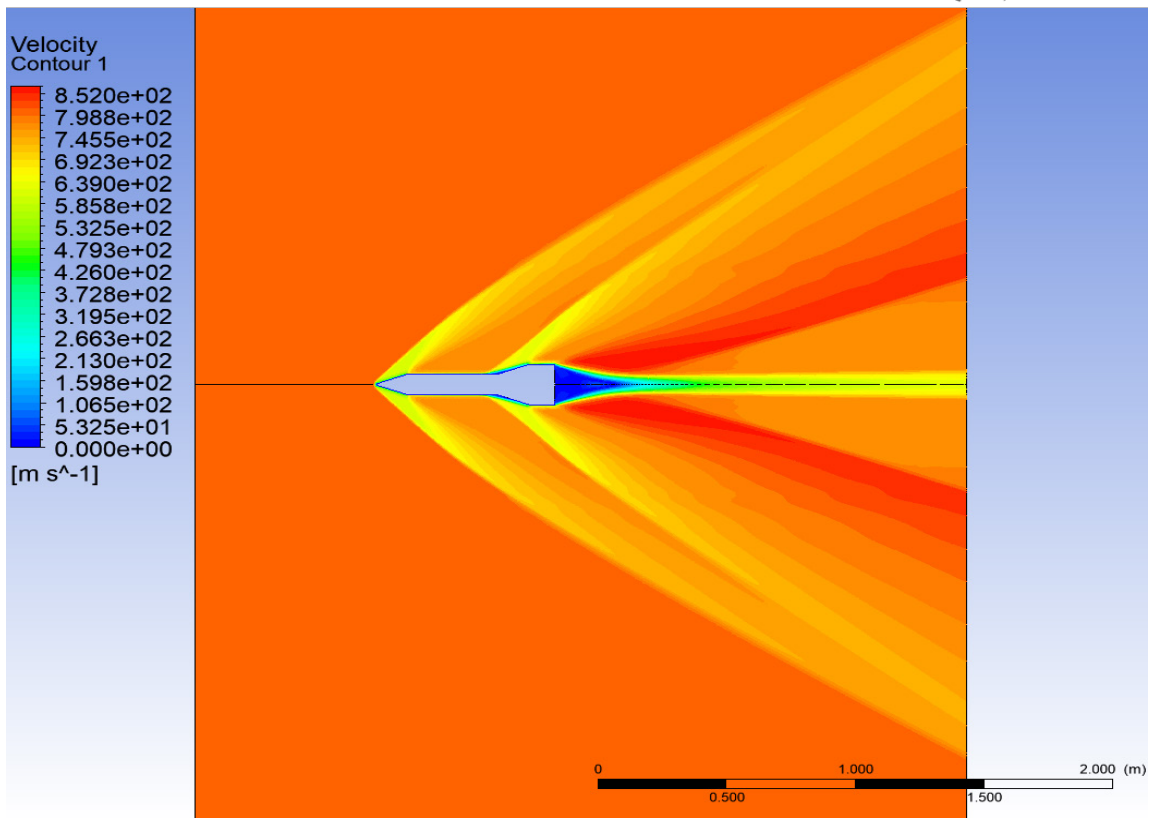


Let's examine the pressure contour to visualize the shock waves experienced by the rocket.

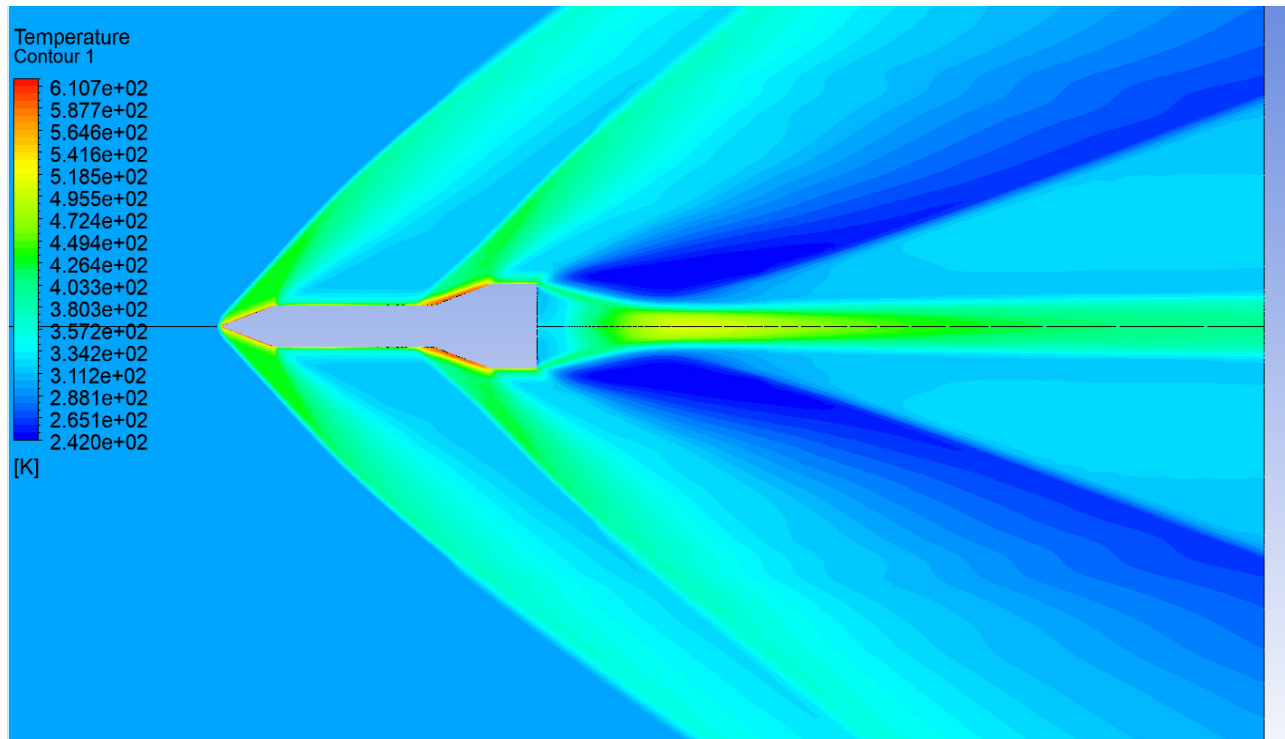
Remember we can mirror about the y-axis by right clicking the wireframe



We can also examine the velocity contour to visualize the wake and fluctuations in velocity caused by the shock.



We can also create a temperature contour plot to observe how the temperature fluctuates around the rocket.



Is this an optimal rocket design? How does the shock wave affect temperature and pressure?

© 2024 ANSYS, Inc. All rights reserved.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics focused on structures and structural simulations.

Ansyes Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, MicroProjects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit www.ansys.com for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.