

# **Ansys Software Tutorial**

## **Introduction to Ansys Fluent #6: Heat Transfer with Fluent**

Mitchell Boots

University of Newcastle

Edited by the Ansys Academic Development Team

[education@ansys.com](mailto: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 be on the thermal analysis capabilities in Ansys Fluent, such as coupling heat transfer and fluid flow. Our focus will be on mastering the setup of boundary conditions for thermal analysis, including essential aspects like the energy equation and precise boundary condition configuration.

This tutorial is #6 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.

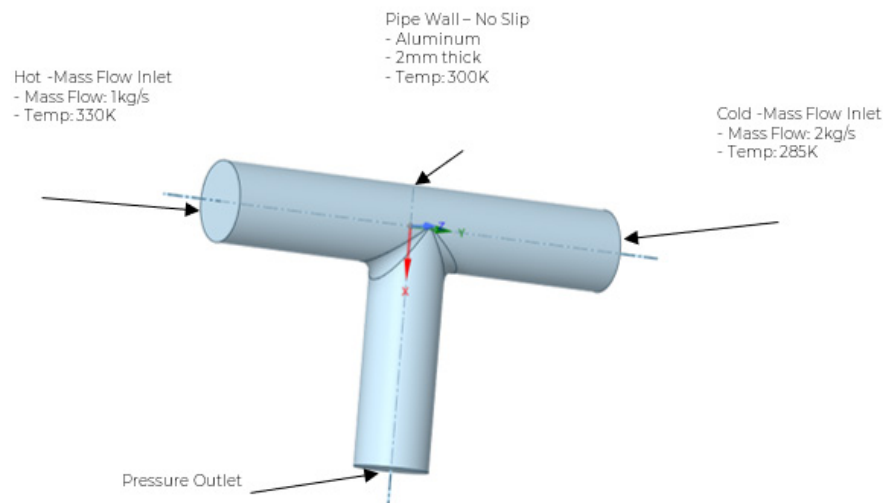
## Table of Contents

Exercise 1: 3D Shower Mixing T-pipe:.....	3
Exercise 2: 3D Pipe Cross Flow Heat transfer: .....	8

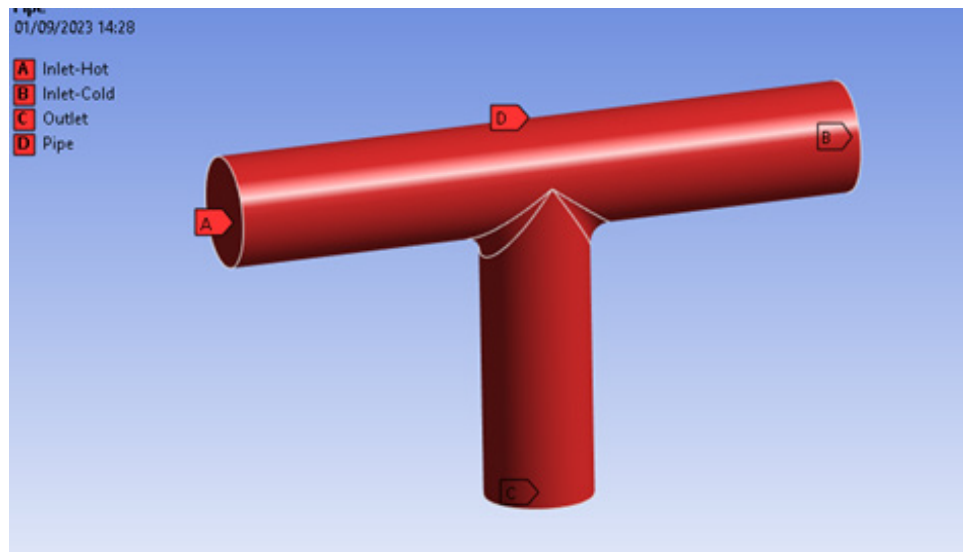
In this tutorial, we delve into the thermal analysis with Fluent. In this session, we will explore Ansys's ability to couple heat transfer and fluid flow using Fluent. Our focus will be on mastering the setup of boundary conditions for thermal analysis, including essential aspects like the energy equation and precise boundary condition configuration.

### Exercise 1: 3D Shower Mixing T-pipe:

In the first exercise, we will utilize the energy equation to perform thermal analysis on a shower mixing valve. Please import the "3D-T-Pipe.step" file into a new instance of Fluent and load SpaceClaim<sup>1</sup>. In this lab, we will investigate the mixing of hot and cold fluids within a shower pipe.

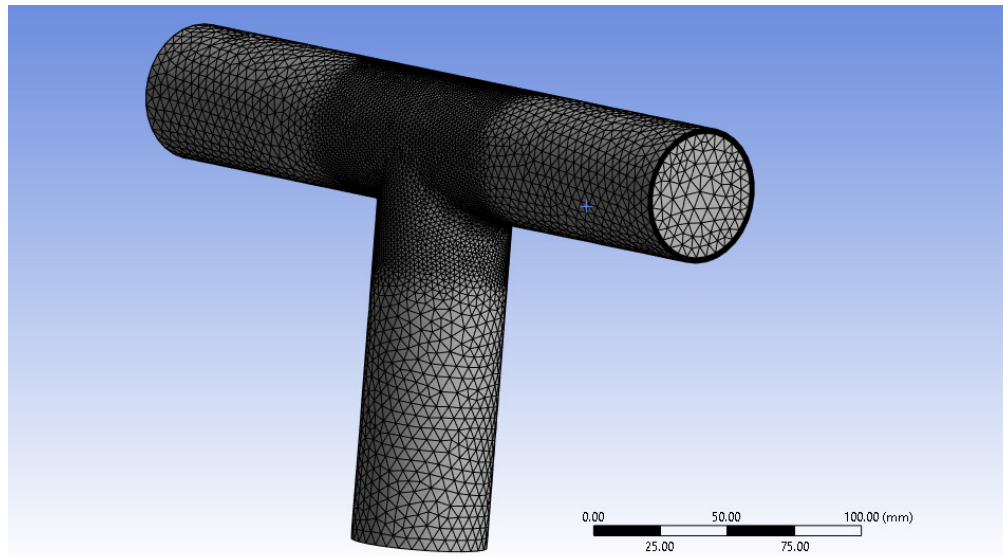


Utilizing your knowledge, establish the named selections for "Inlet-hot," "Inlet-cold," "Pipe," and the "Outlet" of the pipe.

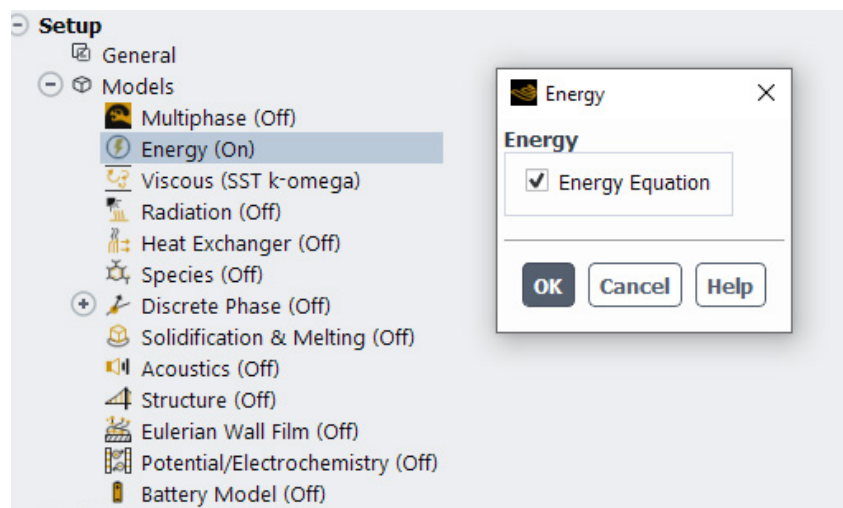


<sup>1</sup> Ansys Discovery is our go forward CAD tool, providing an improved user experience, it can be used to handle CAD instead of SpaceClaim

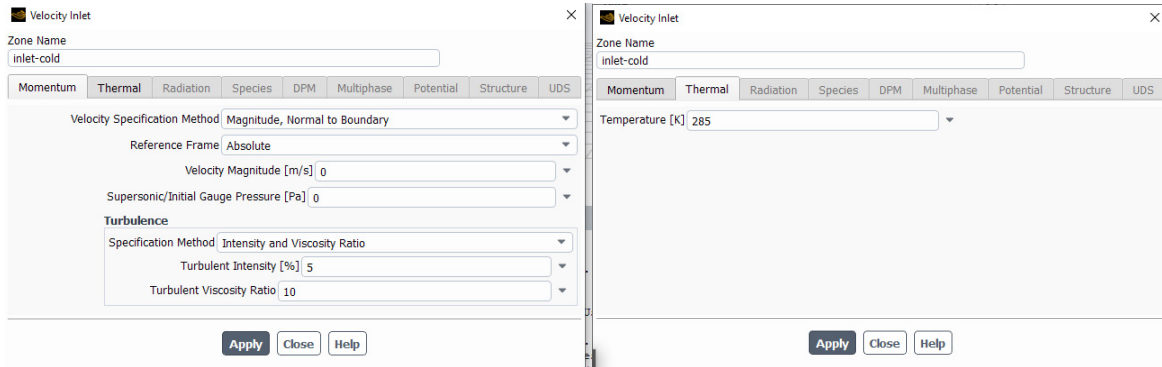
Move on to the Mesh section and generate a suitable mesh. The following image provides an example of what the mesh should look like. (Do your best to optimize solve time vs quality of mesh)



Load the setup/solution and let's begin by configuring the model to use k-omega SST and activating the energy equation. Enabling the energy equation is essential for conducting a thermal analysis.



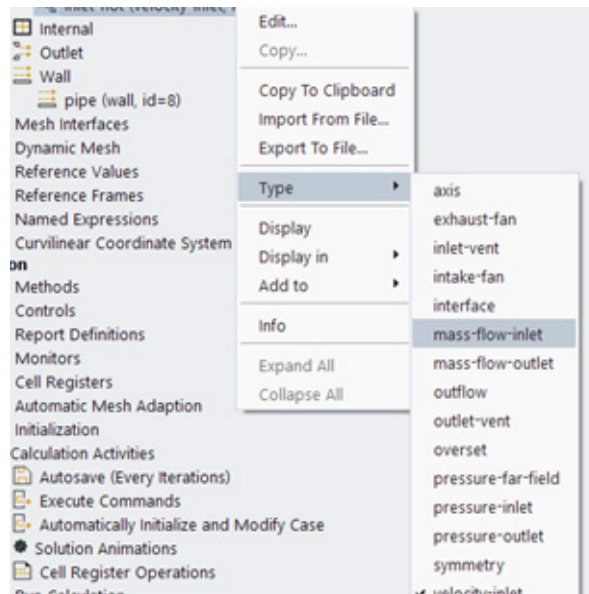
We will now proceed to configure our boundary conditions. With the energy equation enabled, you should observe a new “Thermal” tab under our boundary conditions.



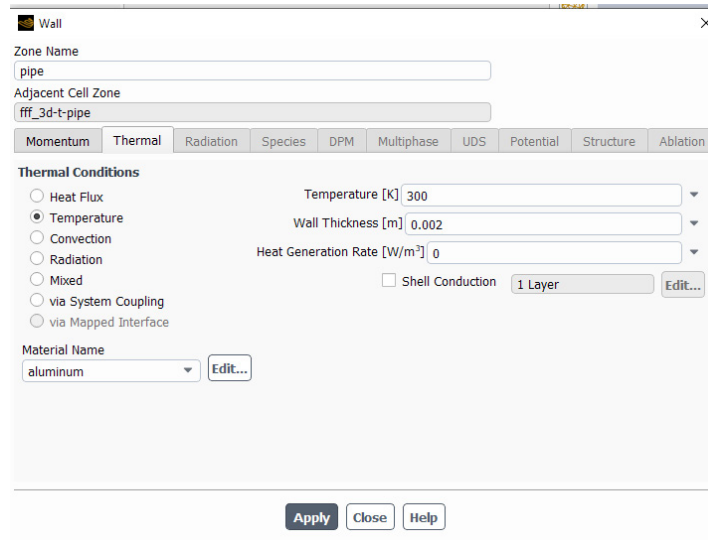
Configure the following conditions in Fluent:

1. Set the Cell-Zone Fluid to 'Water-liquid.'
2. Assign a flow rate of 1 kg/s to both 'Inlet-Hot' and 'Inlet-Cold.'
3. Specify a temperature of 330 K for 'Inlet-Hot.'
4. Specify a temperature of 285 K for 'Inlet-Cold.'

For most of this semester, we have configured our inlet conditions using velocity. To set the inlet condition based on a flowrate, right-click and select "Type -> Mass Flow Inlet."

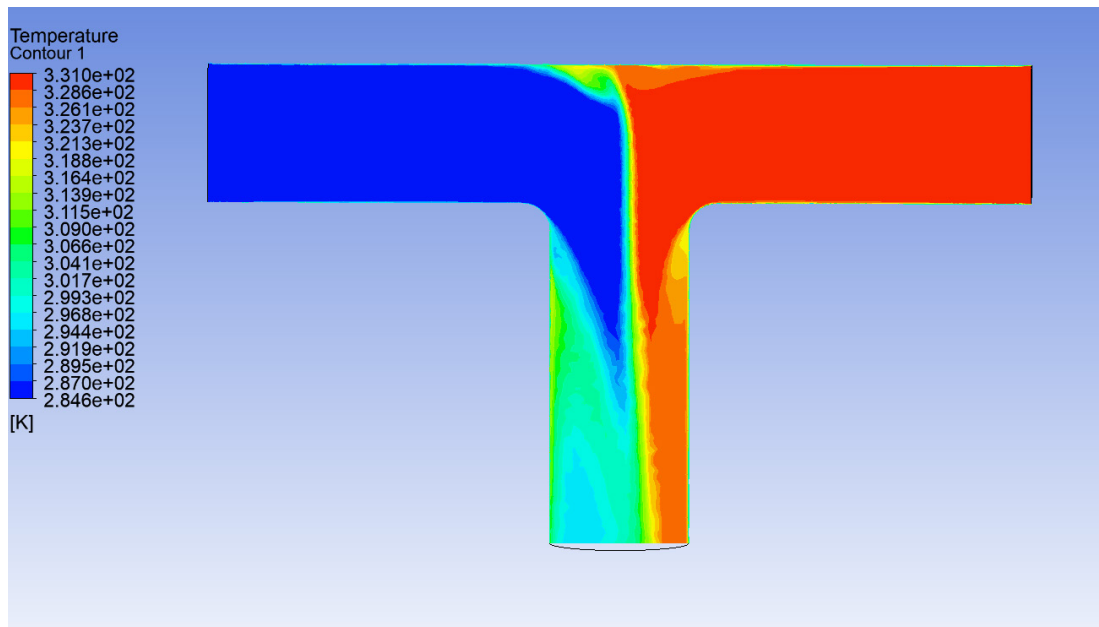


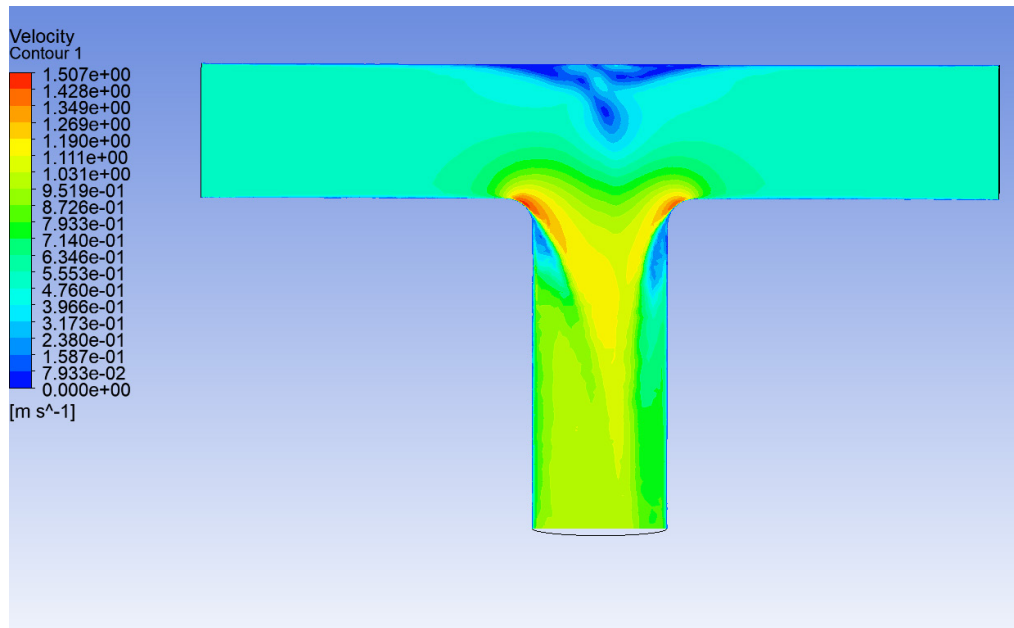
Before initializing the simulation, we must define the thermal conditions for the 'Pipe-Wall.' Click on the thermal properties, set the type to temperature, and configure the temperature to 300 K, with a wall thickness of 2 mm. You can keep the material as aluminum.



Now, let's proceed with initializing and running the simulation for 200 iterations.

After that, we will display the velocity and temperature contours along the centerline of the pipe.



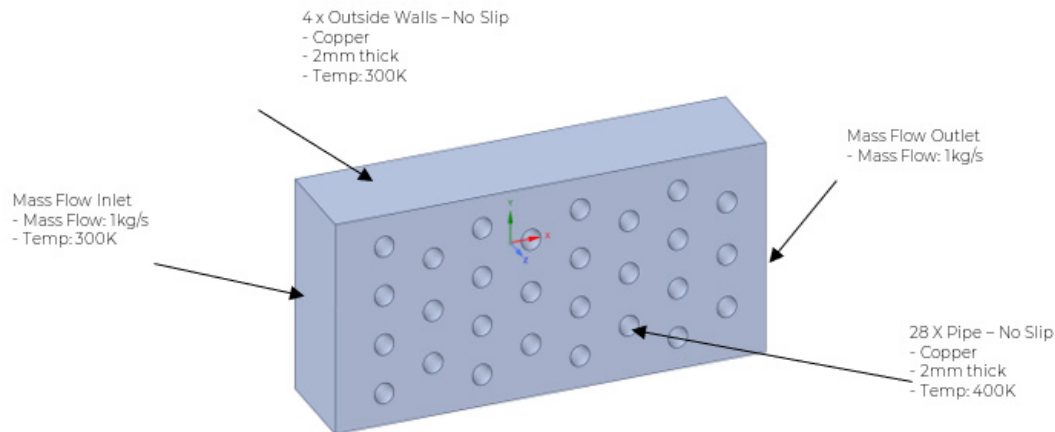


Ensure you have checked your Yplus to capture what is happening at the boundary.

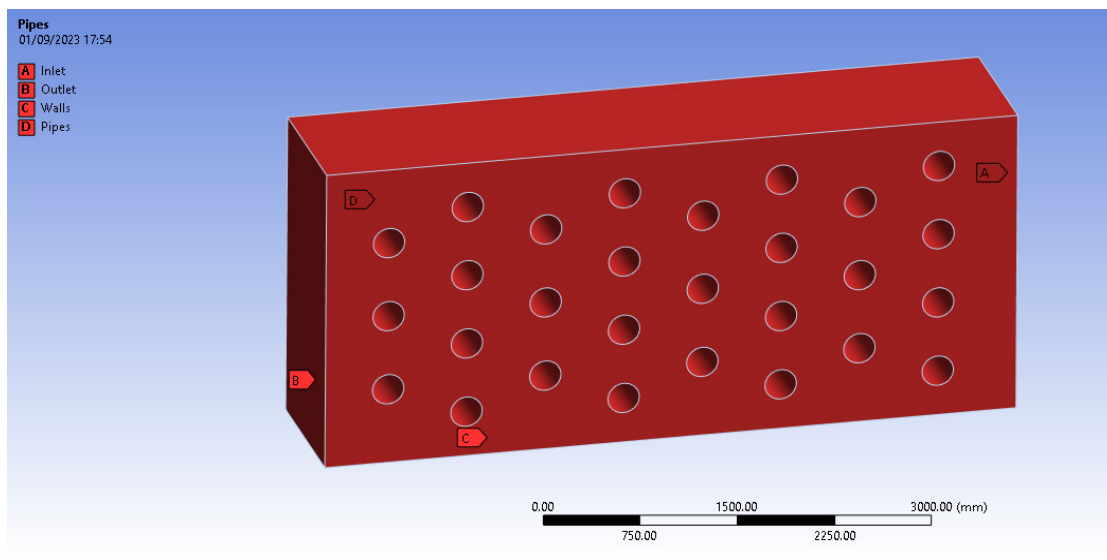
Do these results make sense? How could we optimize the geometry to ensure better mixing of the hot and cold fluids?

## Exercise 2: 3D Pipe Cross Flow Heat transfer:

In the second exercise, we will analyze heat transfer over an array of pipes. Please import the '3D-Crossflow.step' file and open it in SpaceClaim. The setup we will be using for this simulation is illustrated in the following image.

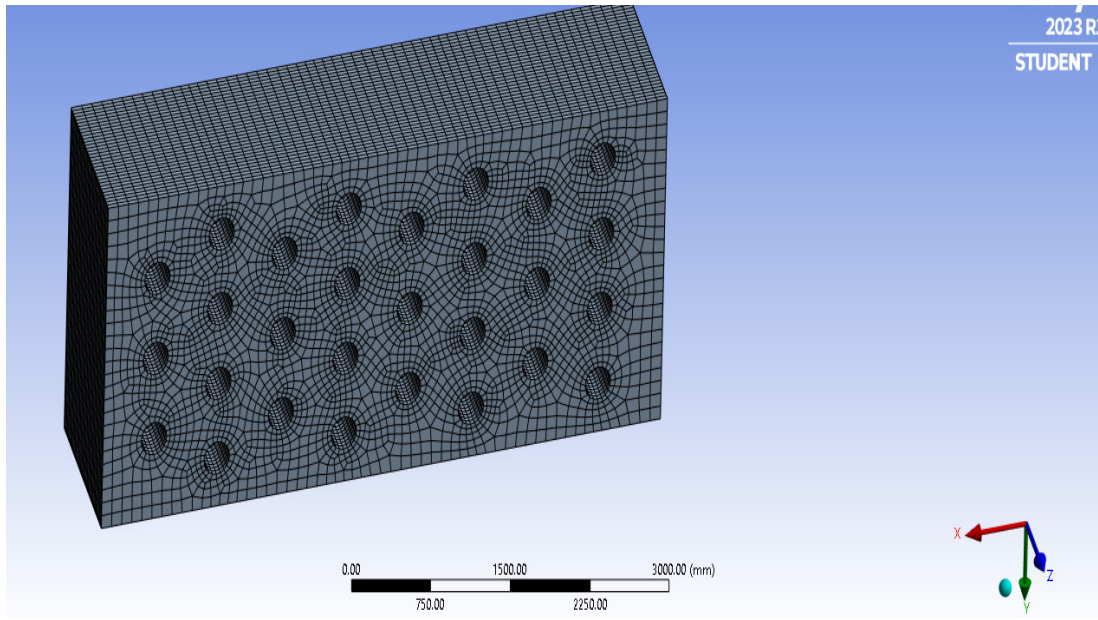


Configure the named selections as follows: Inlet, Outlet, Wall, and Pipes.

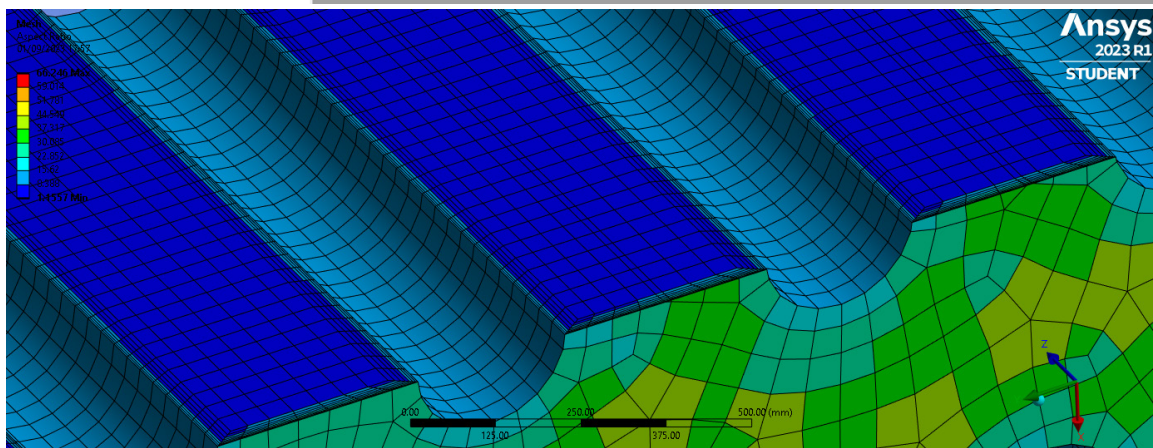
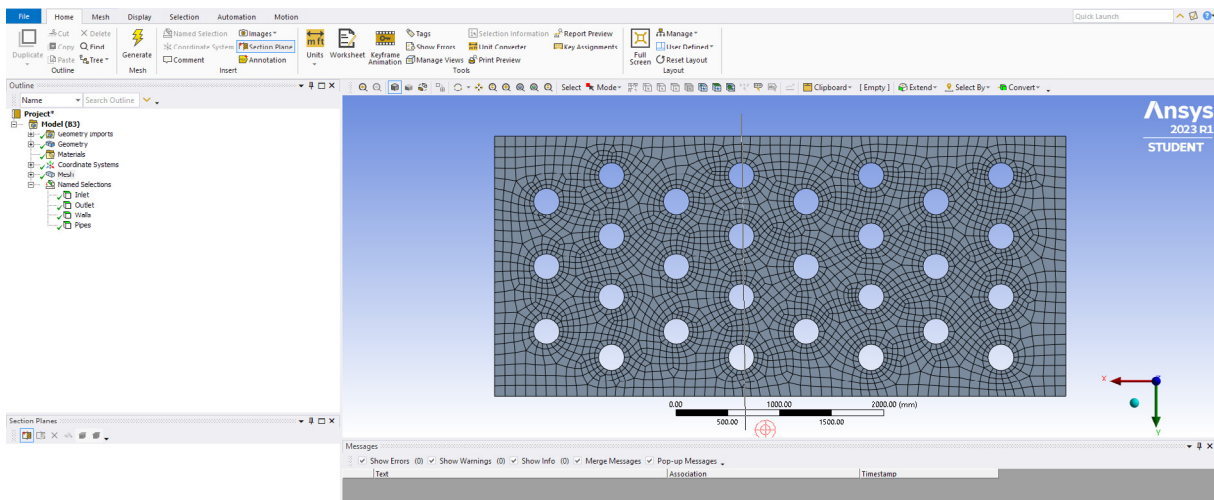


Generate an appropriate mesh, and please refer to the example provided. Aim to find a balance between computational efficiency and mesh quality, with a focus on the areas of greatest interest.

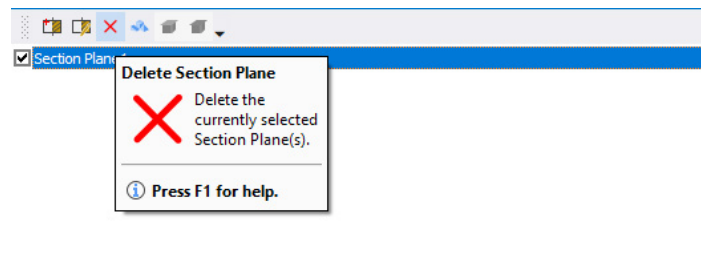




Before finalizing the mesh, we can inspect the interior using the section plane tool. Navigate to 'Home' and click on 'Section Plane.' You can then drag a line across the body to create a section, like the example shown.



You can delete the section plane by clicking on the red cross located in the bottom left corner.

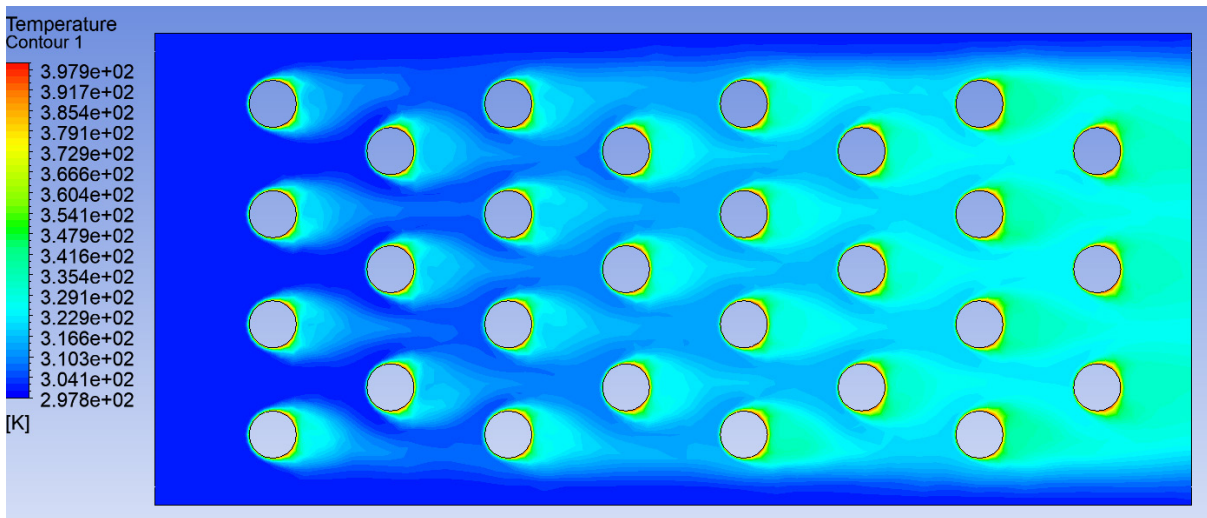


Move over to Fluent and configure the solution with the following conditions:

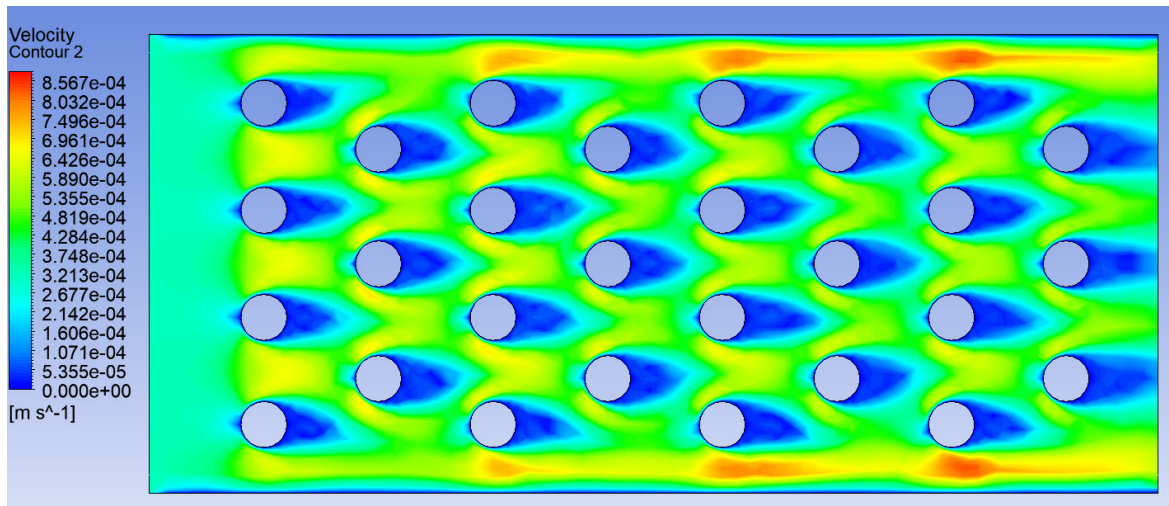
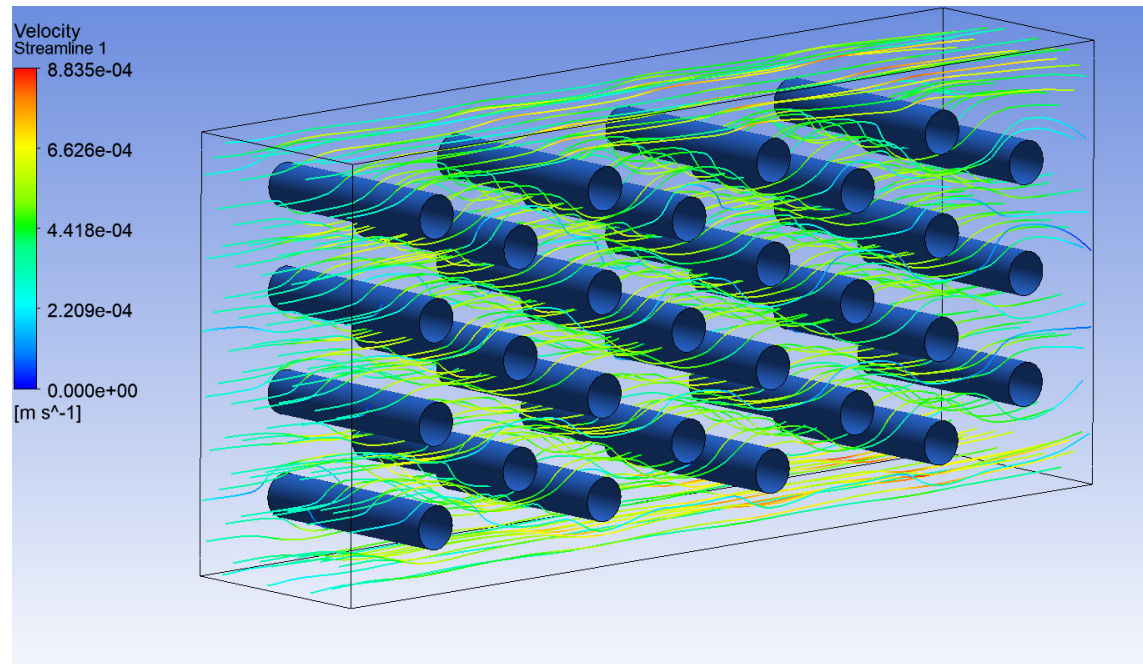
1. Set the Model to 'Laminar.'
2. Activate the 'Energy Equation.'
3. Set the cell zone fluid to 'Water-liquid.'
4. Configure the Inlet with a mass flow of 1 kg/s and a temperature of 300 K.
5. For the 'Pipes,' set a temperature condition of 400 K, a wall thickness of 2 mm, and use copper as the wall material.
6. For the 'Walls,' set a temperature condition of 300 K, a wall thickness of 2 mm, and use copper as the wall material.

Now, initialize and solve the simulation over 200 iterations.

Examine the streamlines, velocity distribution, and temperature contours.







How can we improve the heat transfer of this system? Would the material of the pipes make a difference in this simulation?

© 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](http://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](mailto:education@ansys.com).

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto: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](http://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.