

Ansys Software Tutorial

Introduction to Ansys Fluent #5: Aerodynamics Analysis Part 2

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 expand upon the concepts from Tutorial #4, delving into the details of symmetry and moving domains using a simplified wheel and axle model. Furthermore, we will cover and make use of another fluid flow system from Ansys, Fluid Flow (Fluent with Meshing).

This tutorial is #5 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.

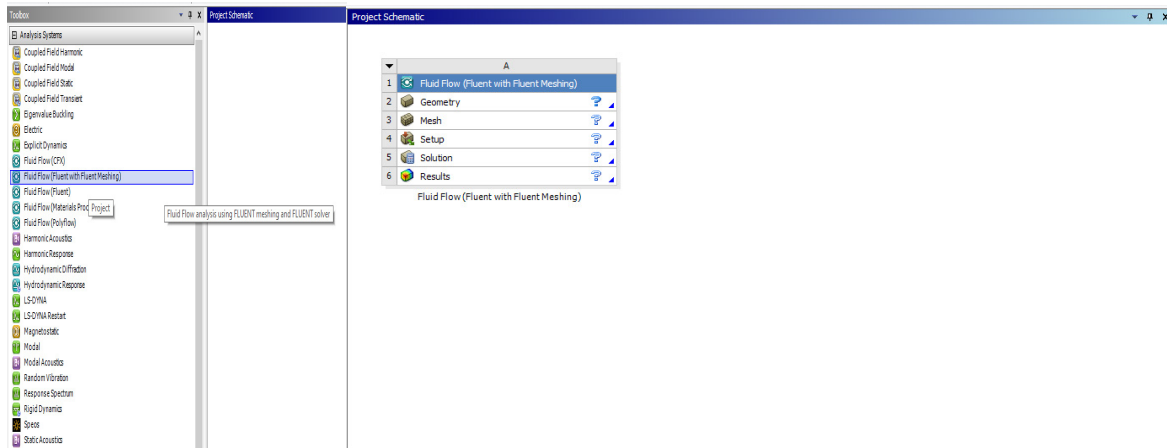
Table of Contents

Exercise 1: 3D Wheel and Axle – Symmetry:	3
Exercise 2: 3D Simplified Car Model – Moving Domains:	13

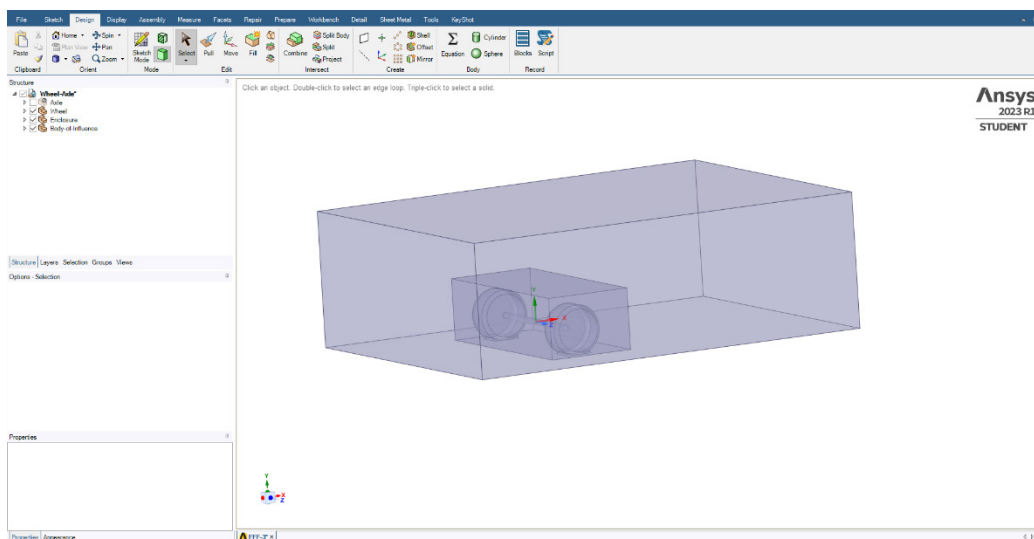
In this tutorial, we continue on from Intro to Ansys Fluent Tutorial #4, delving into the details of symmetry and moving domains using a simplified wheel and axle model. Furthermore, we will cover and make use of another fluid flow system from Ansys, Fluid Flow (Fluent with Meshing).

Exercise 1: 3D Wheel and Axle – Symmetry:

In this exercise, we will go through the process of creating and utilizing a **symmetry** boundary condition, utilizing Fluent with Meshing. Add a new instance of this tool to the project schematic. The only distinction when using Fluent with Meshing is the meshing interface. Nevertheless, all other elements such as geometry, results, solutions, and setup will remain unchanged. After placing the system, import the provided geometry file and open SpaceClaim¹.

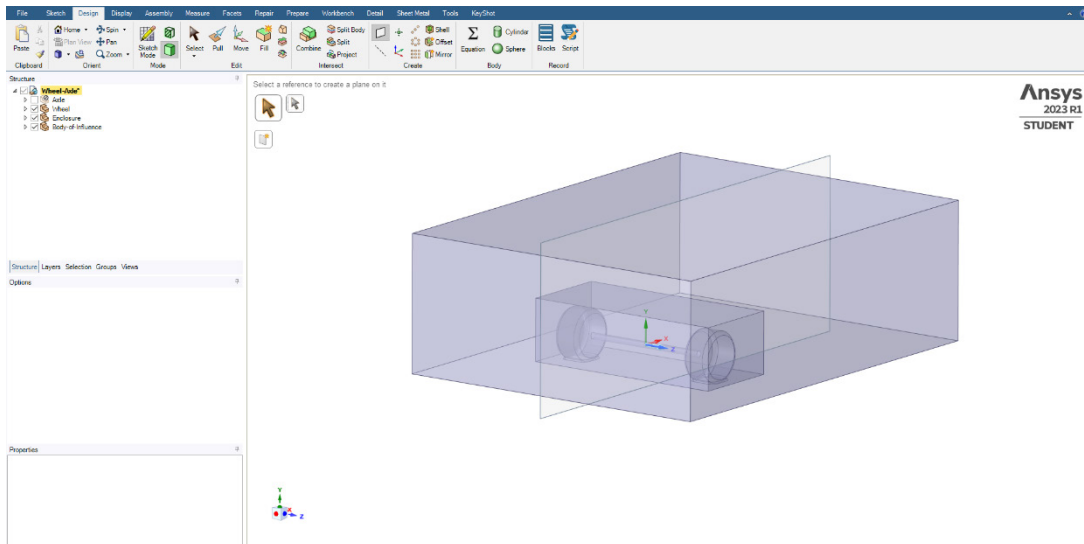


The model should resemble the following image, and the named selections should have been loaded with the CAD document. To verify, refer to the 'Groups' section. You should observe that 'Sky,' 'Ground,' 'LeftWheel,' 'RightWheel,' 'Inlet,' 'Outlet,' and 'Axle' are all listed.

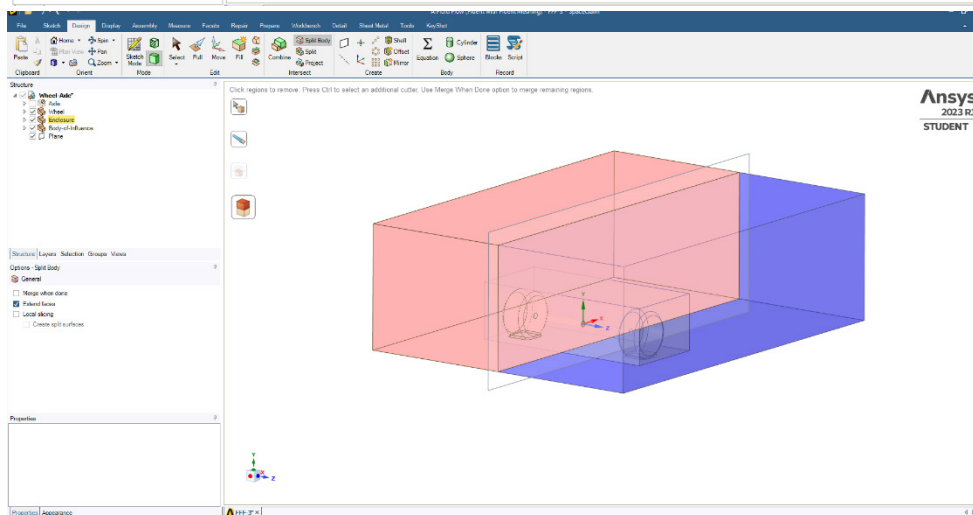
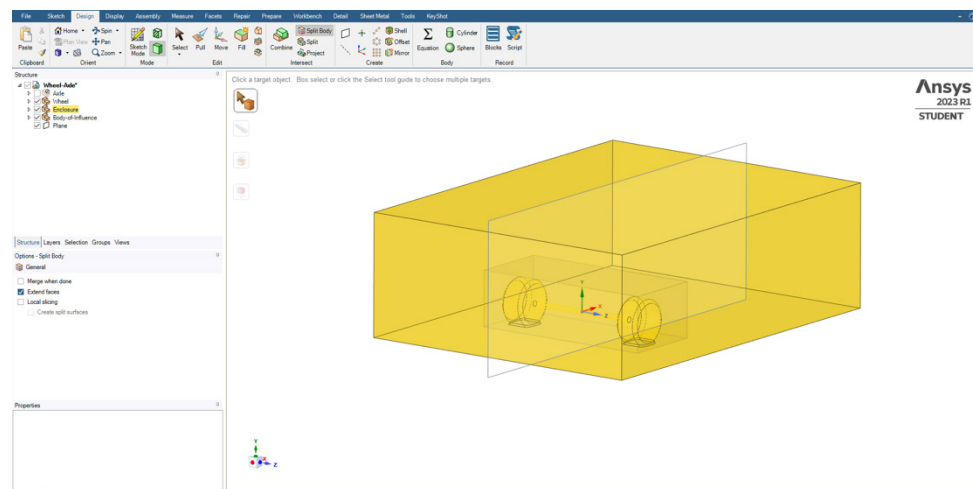


Now, let's proceed to create our 'Symmetry' named selection. To begin, we need to generate a plane on the XY plane. Navigate to the plane tool located under Design -> Plane. Click on the Z arrow, and a plane should appear as shown below.

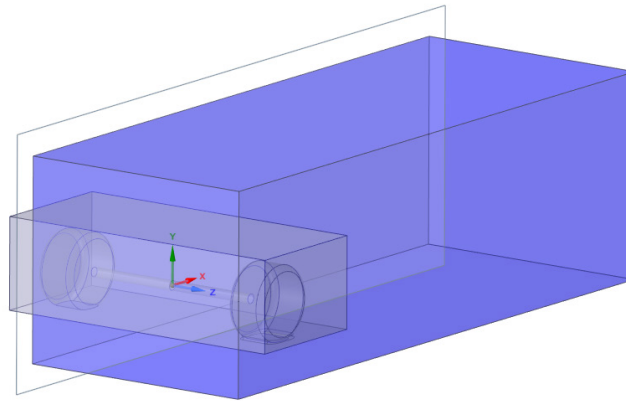
¹ Ansys Discovery is our go forward CAD tool, providing an improved user experience, it can be used to handle CAD instead of SpaceClaim



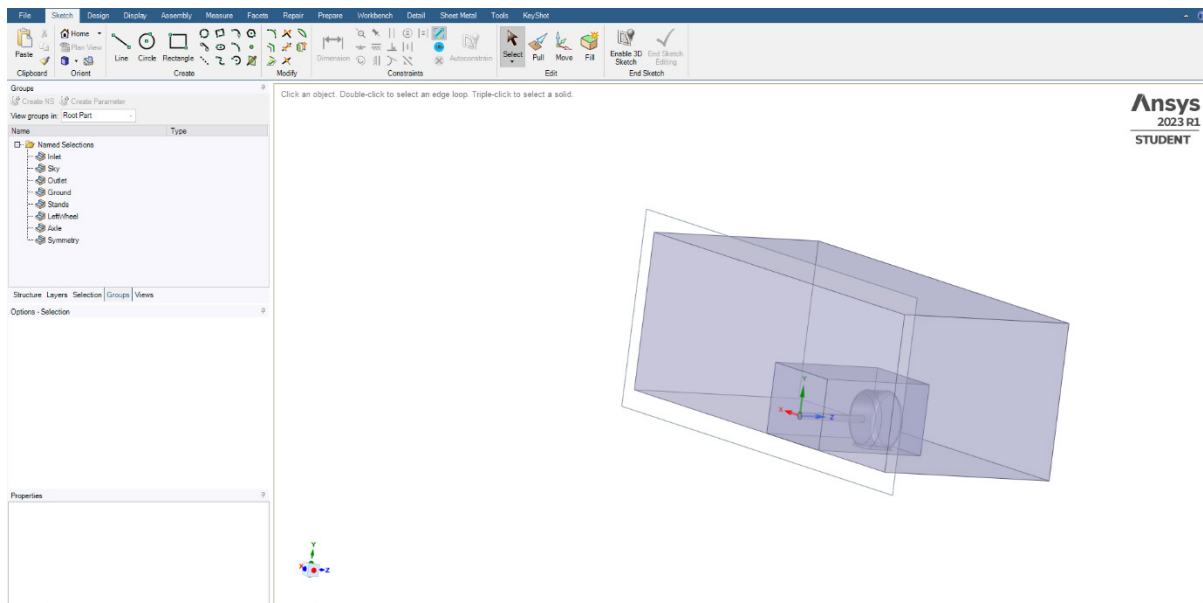
We will now utilize this plane to remove one half of the enclosure. Proceed by employing the 'Split Body' tool found under Design -> Split Body, which allows us to eliminate a single side of the car. Select the body, then choose the cutting plane, and finally, indicate the side you wish to delete.



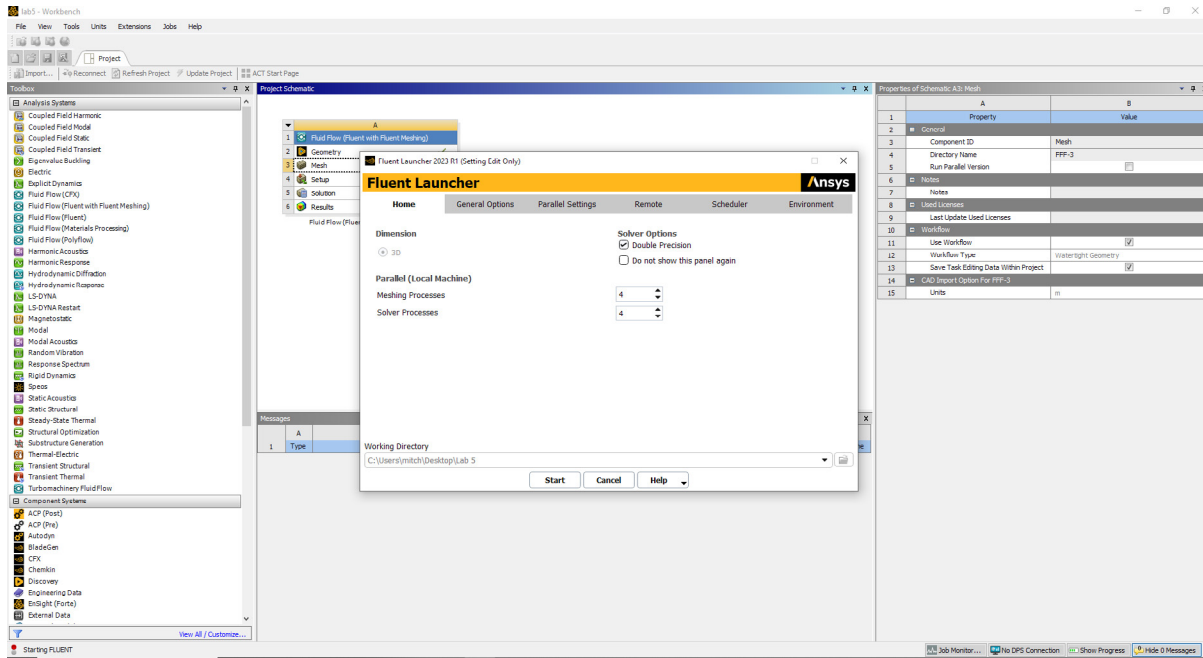
After deleting half of the main enclosure, it should resemble the following image. Repeat this process with the body of influence. So that anything on the right side of the plane is gone.



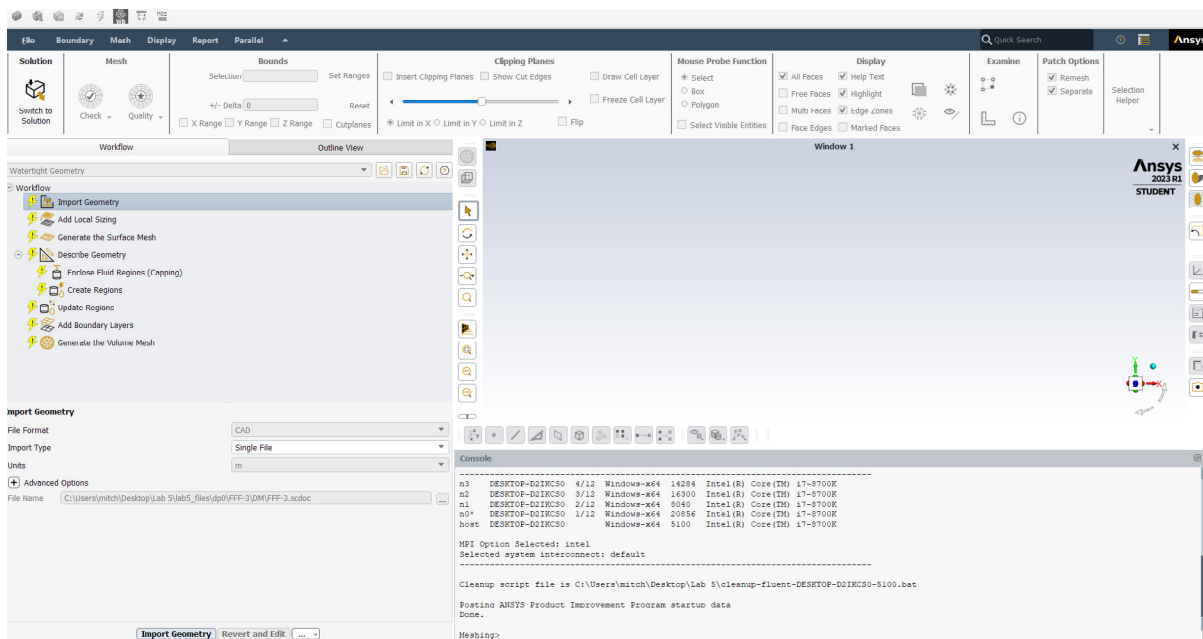
Having split the car in half, it's now time to assign a name to the symmetry face. Just as we did last week, name this face 'Symmetry'. Pay attention to the coordinate system and take note of the wheel's position in relation to the origin.



After splitting the geometry and naming the symmetry face, close SpaceClaim and open the Mesh module. You will now be prompted to allocate computational resources here. Enter '4' for the number of meshing processors and enable double precision. Then, click on the 'Start' button.



This interface differs quite a bit from the one we have been using for the past few weeks. Like Fluent, we will progress through each step of this meshing process gradually to develop our fluid domain. The interface should resemble the following image.

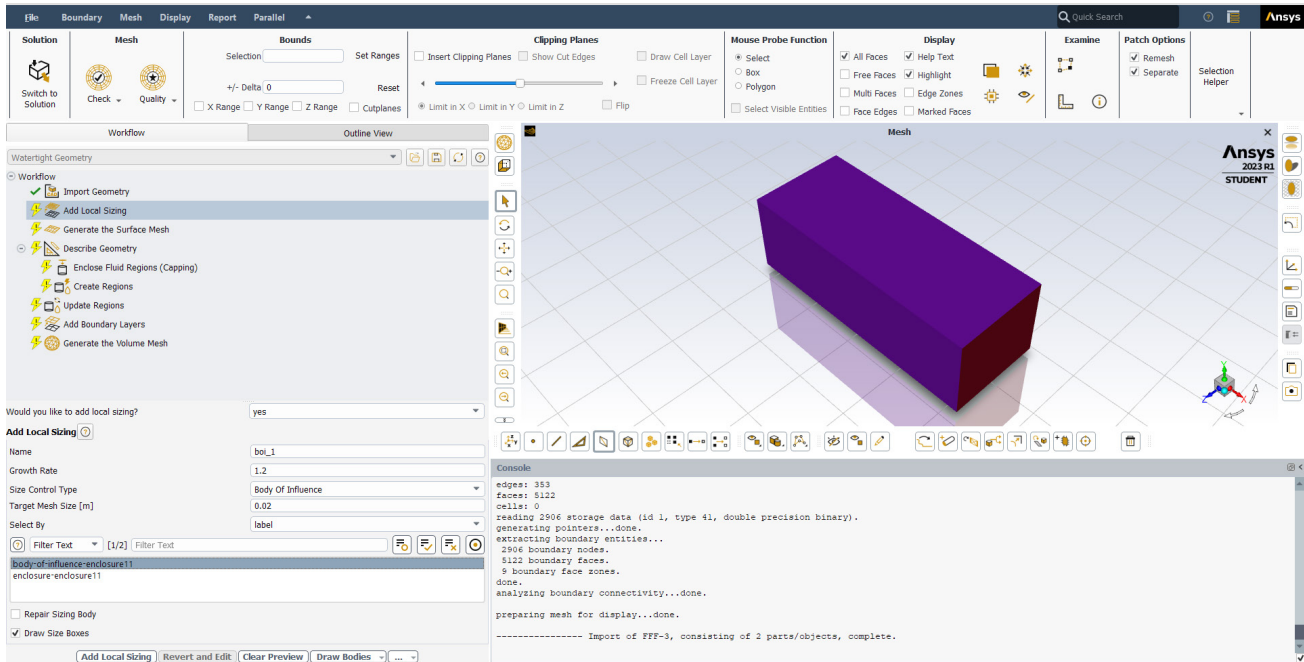


First, import the geometry. Select the 'Import Geometry' section as shown above and press 'Import Geometry' in the bottom left corner.

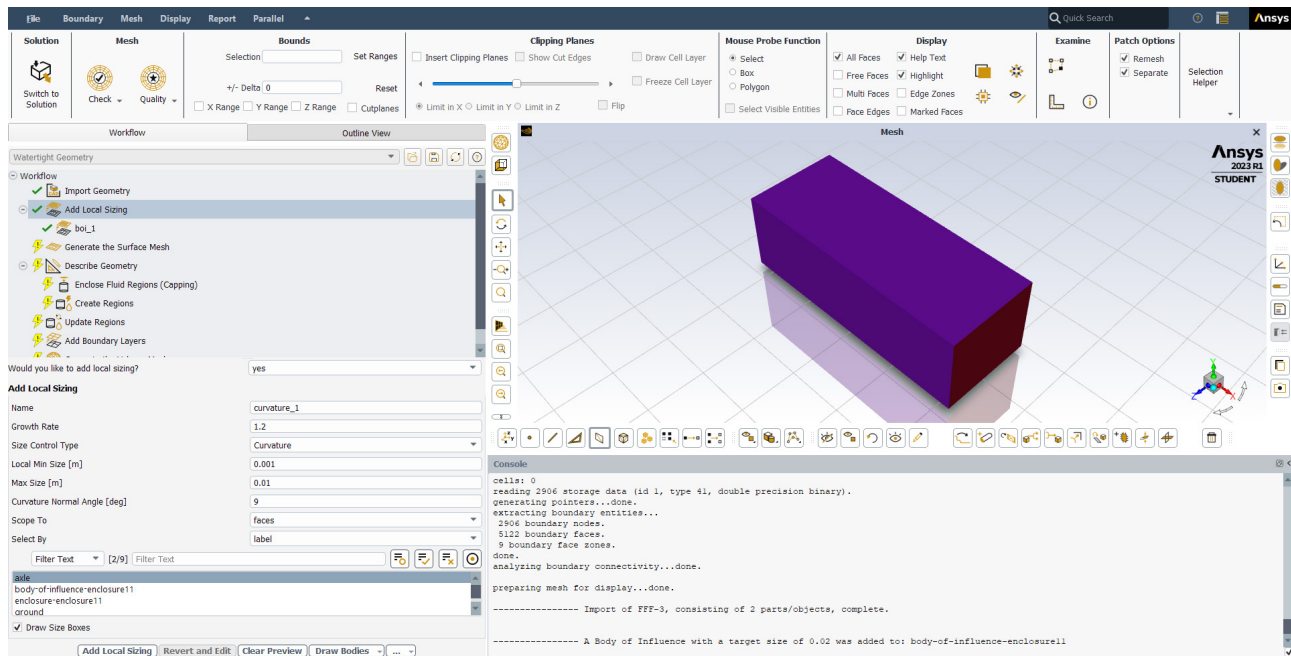
Move over to 'Local Sizing.' We will start by clicking the dropdown box and selecting 'Yes.'

Next, we will create a 'Body of Influence.' Click on 'Size Control Type' and choose 'Body of Influence.'

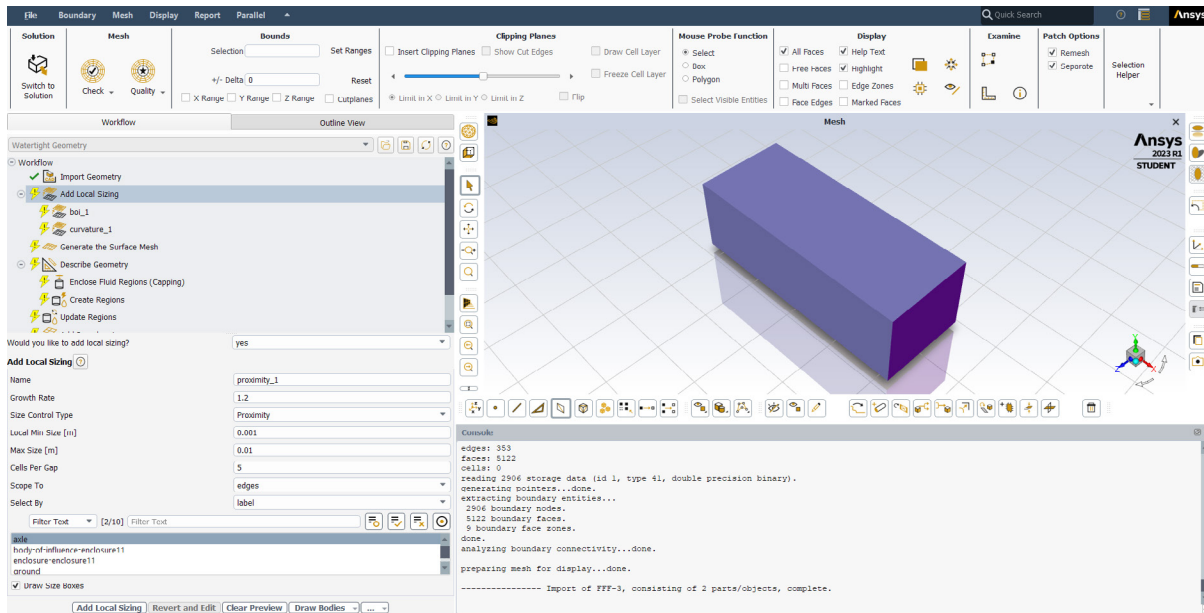
Set the target mesh size to 20mm and select the body of influence from the bottom list, as shown in the image below. Once you have completed these steps, click 'Add Local Sizing'.



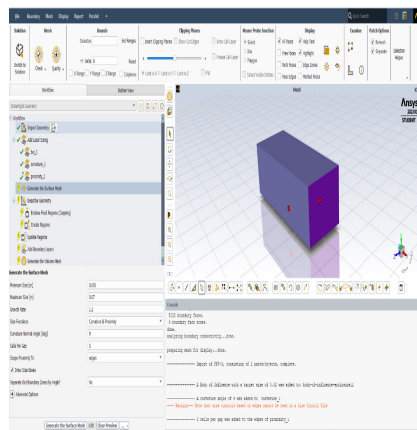
Now, let's apply curvature sizing to the faces of the Stands, Axle and Wheel. Select the control type as 'Curvature' and set the minimum mesh size to 1mm and the largest size to 10mm. Additionally, set the angle to 9 degrees. After adjusting these settings, click 'Add Local Sizing'.



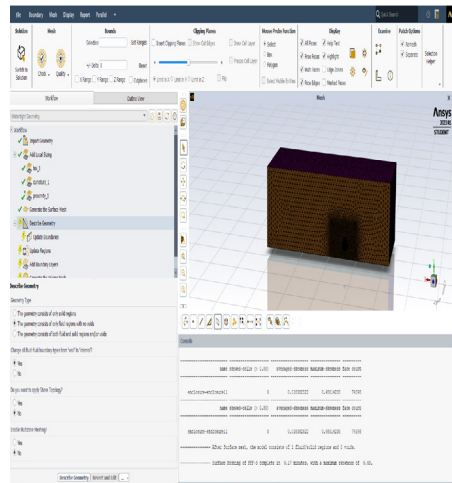
Next, we will create proximity sizing for the edges of the wheel, axle and stands. Ensure to scope to the edges. Set the minimum size again to 1mm and the maximum size to 10mm. Additionally, set the cells to 5 per gap, and then add the local sizing.



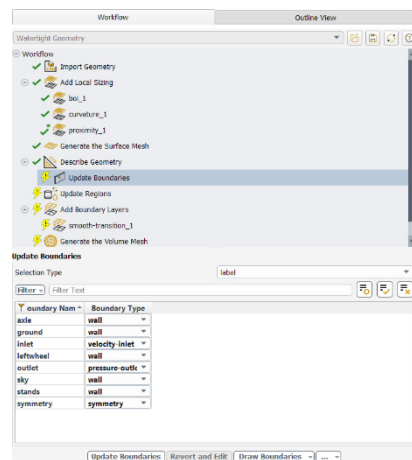
Now, let's generate our surface mesh. Configure the settings as following depicts and then click 'Generate the surface mesh'.



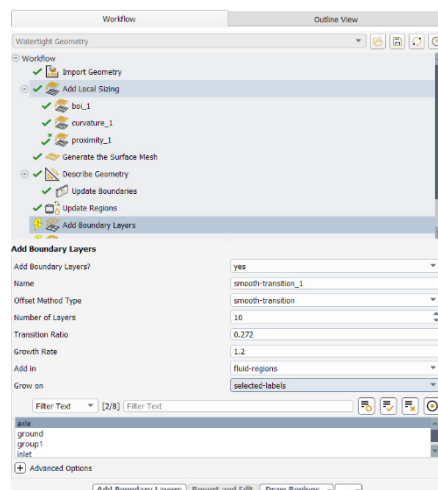
After that, we will proceed to 'Describe Geometry' of our geometry. Tick the options 'The geometry consists of only fluid domains with no voids' and 'Change all fluid-fluid boundary types from 'wall' to 'internal'' to 'Yes.' Then, click 'Describe Geometry.'



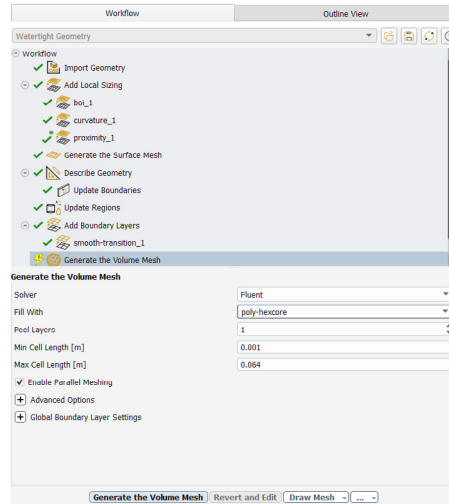
Next, let's proceed to update our boundaries. You should observe the following loaded settings. Ensure that our inlet, outlet, and symmetry boundaries are configured correctly.



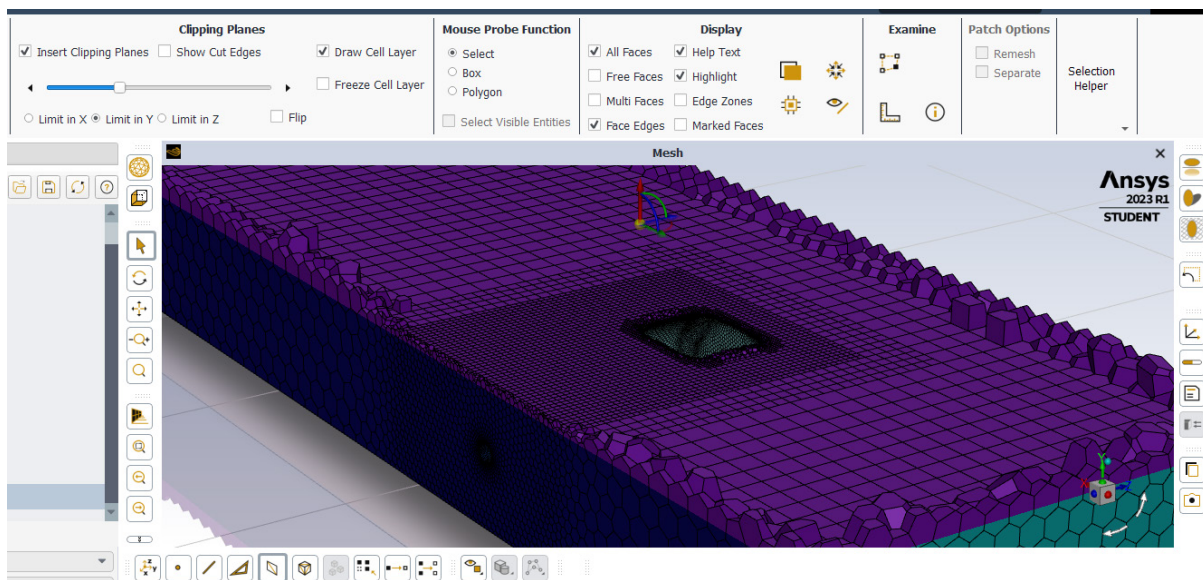
We will then add a smooth transition boundary layer with ten layers, grown on selected labels Wheel and axle. Select the options as shown in the provided image.

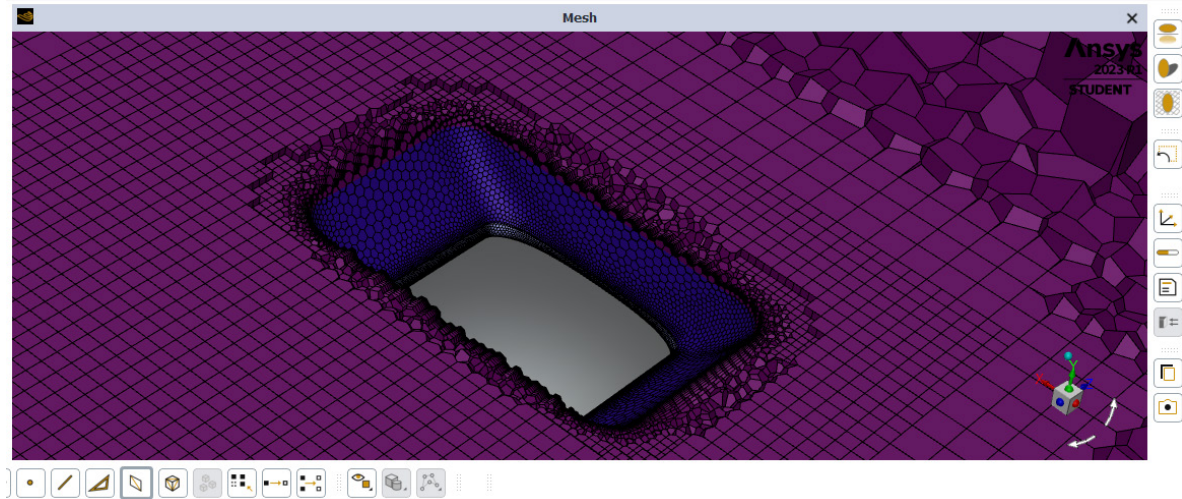


After adding the boundary layers, we can proceed with generating our volumetric mesh. Configure the following options and then click 'Generate the Volume Mesh'.



Once you've generated and updated the mesh, you can inspect it by using the clipping plane to visualize the internal mesh. Drag the slider as indicated in the following image.



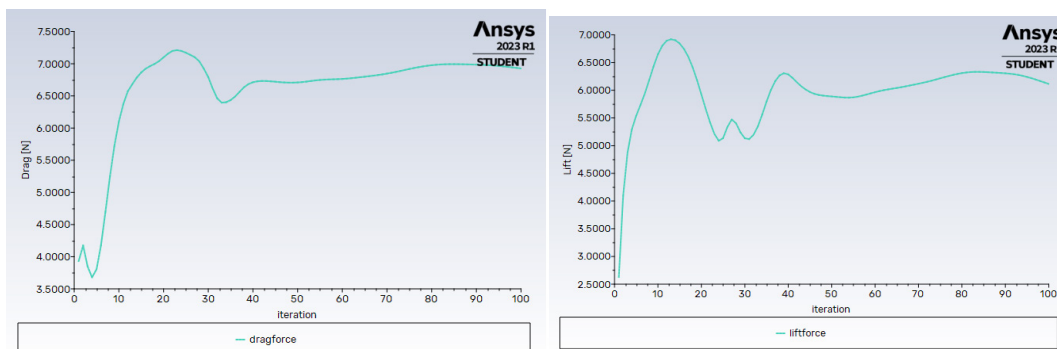


Once you are done move over to the setup/solution. If you look under boundary, you should see a symmetry condition, this means our named selection has successfully loaded into our simulation. Now set the solution up with the following conditions.

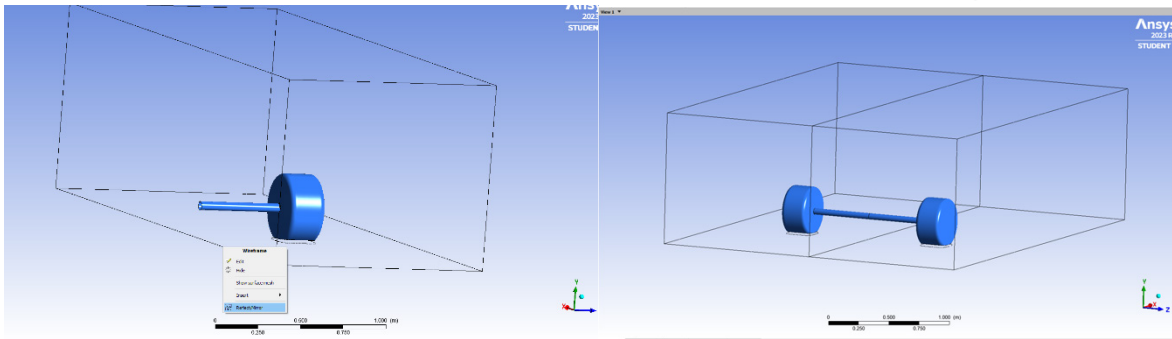
1. Set the model to k omega SST.
2. Set the cell zone to 'air'.
3. Set the Inlet velocity to 14 m/s.
4. Set the sky to zero shear.
5. Set the Stands to zero shear.
6. Drag and Lift force reports on the wheel and axle.

Initialize and perform calculations for the solution over 100 iterations. Please note that the solution might not have fully converged, but due to computational limitations, we will restrict the number of iterations to 100. Once the solution calculation is complete, proceed to CFD post.

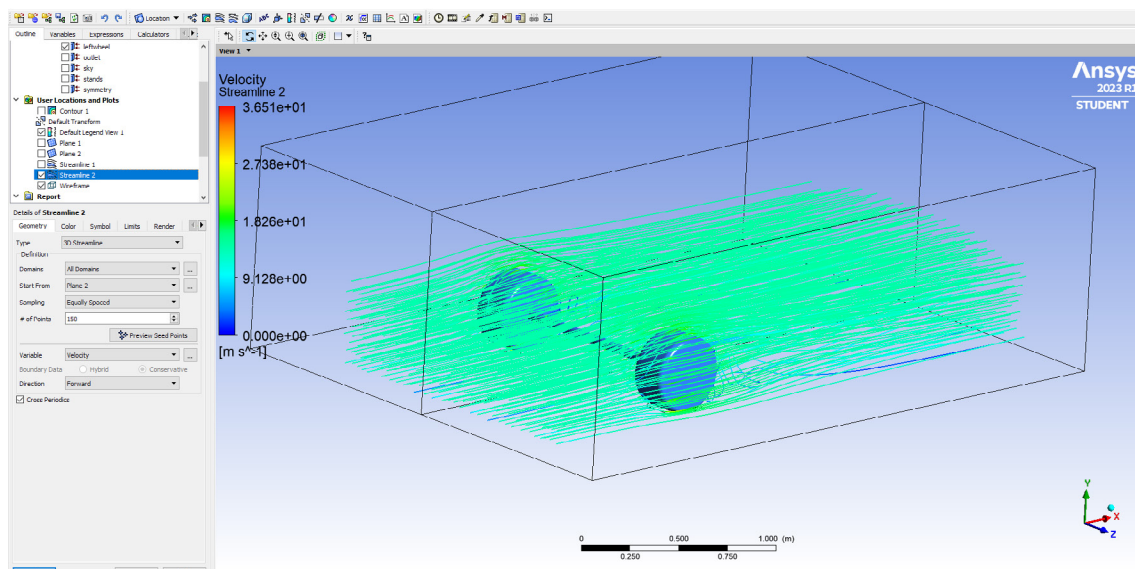
Pay attention to the lift and drag forces observed on the wheels, even when they are not rotating.



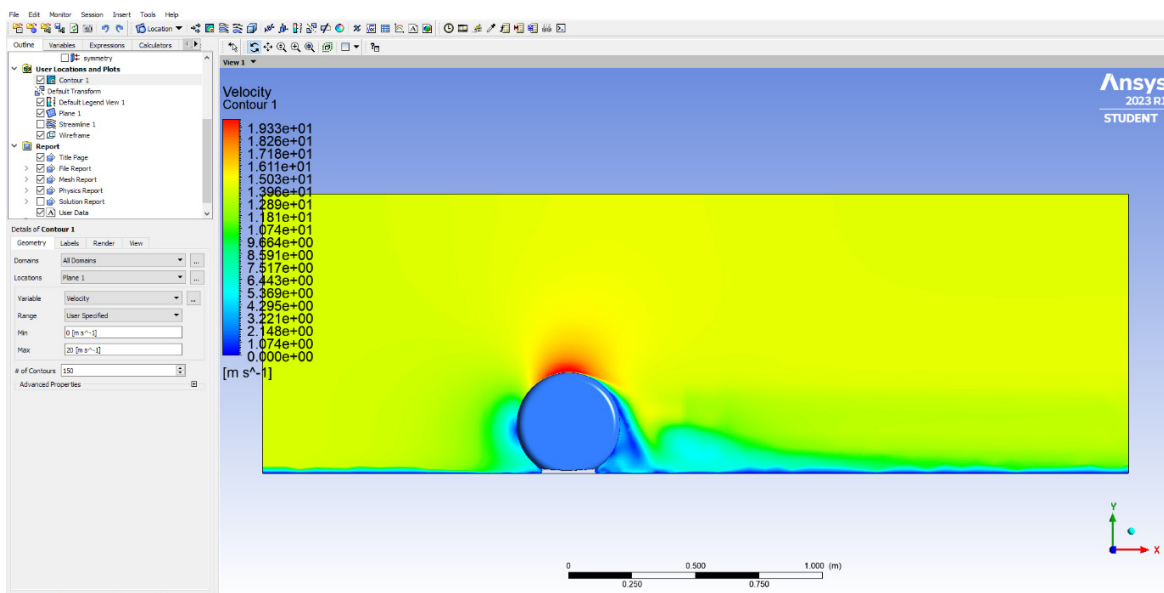
Open the Results section and proceed to create a mirrored view of our simulation. Right-click on the wireframe, then select Reflect/Mirror -> Z axis.



Generate a streamline starting from the inlet. Zoom in to closely observe the flow around the tire and take note of its behavior.



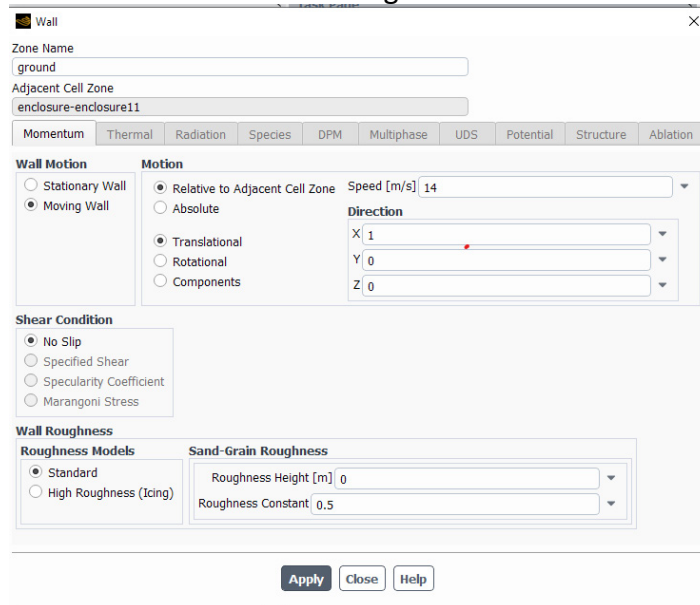
Produce a velocity contour at the center of the wheel, with the Z-coordinate set to 0.608 meters. Analyze the flow patterns around the wheel based on the contour.



Exercise 2: 3D Simplified Car Model – Moving Domains:

We will now advance with the same simulation, introducing rotating domains to observe their influence on result variations. In a simulation, it's common to consider the air in motion while keeping everything else stationary. This situation mirrors a moving car scenario, where we need to simulate the spinning motion of the ground and wheels.

To achieve this, we will modify the boundary conditions for the ground and wheels. Start by accessing the boundary condition for the ground. Select 'Moving Wall' -> 'Translational' and set the speed along the x-axis to 14 m/s. This modification simulates the ground's movement.



Now, let's proceed to configure the rotational origin and speed for the wheel. Set up the wheel as a 'Moving Wall' with the 'Rotational' option. To define a moving rotational wall, we need to specify the angular velocity, the location from the origin, and the axis around which it rotates. We will be rotating around the Z-axis, and depending on the specific wheel you've chosen, you will need to apply the right-hand torque rule to determine the direction—either -1 or 1. The distance from the origin is $X=Y=0$ and $Z=0.608$. Depending on the side of the car you have selected, it might be -0.608 ; you may need to revisit SpaceClaim to confirm your coordinate system. The angular velocity is 69 rad/s, calculated based on the fluid velocity and wheel diameter.

Wall

Zone Name

leftwheel

Adjacent Cell Zone

enclosure-enclosure11

Momentum

Thermal

Radiation

Species

DPM

Multiphase

UDS

Potential

Structure

Ablation

Wall Motion

Stationary Wall

Moving Wall

Relative to Adjacent Cell Zone

Speed [rad/s] 69

Absolute

Translational

Rotational

Components

Rotation-Axis Origin

X [m] 0

Y [m] 0

Z [m] 0.608

Rotation-Axis Direction

X 0

Y 0

Z 1

Shear Condition

No Slip

Specified Shear

Specularity Coefficient

Marangoni Stress

Wall Roughness

Roughness Models

Standard

High Roughness (Icing)

Sand-Grain Roughness

Roughness Height [m] 0

Roughness Constant 0.5

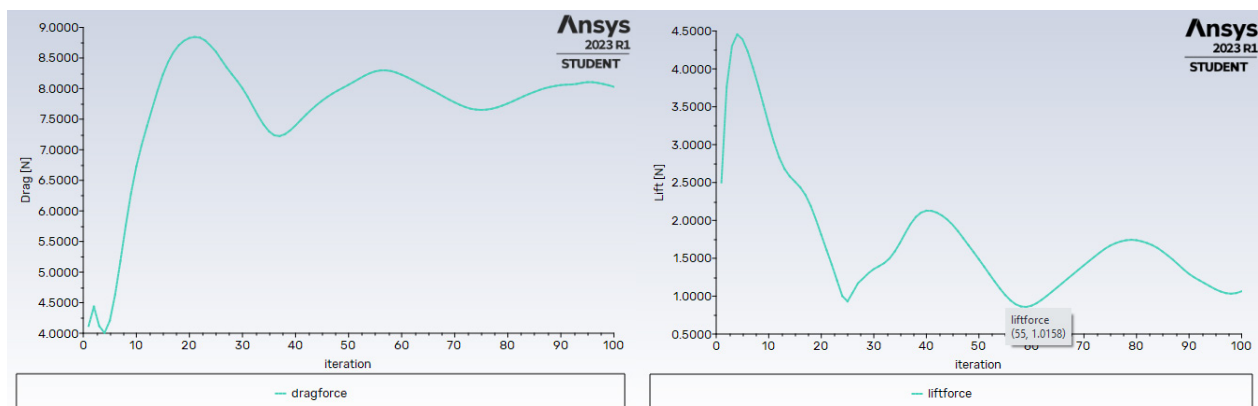
Apply

Close

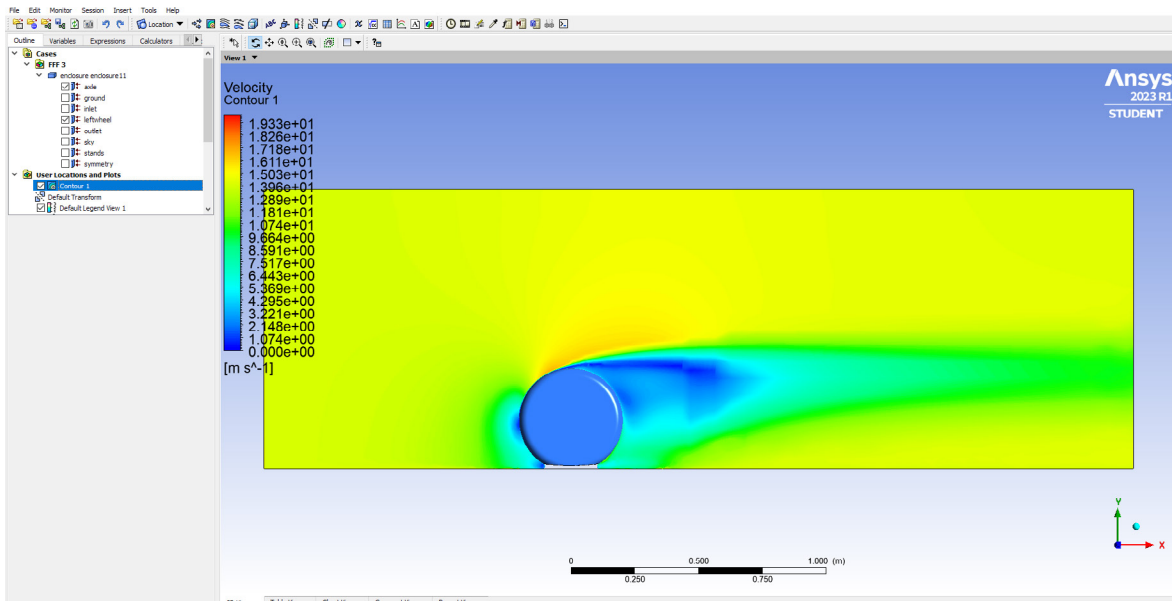
Help

Initialize and run the simulation once more. Compare the drag and lift forces to those obtained in our previous simulation.

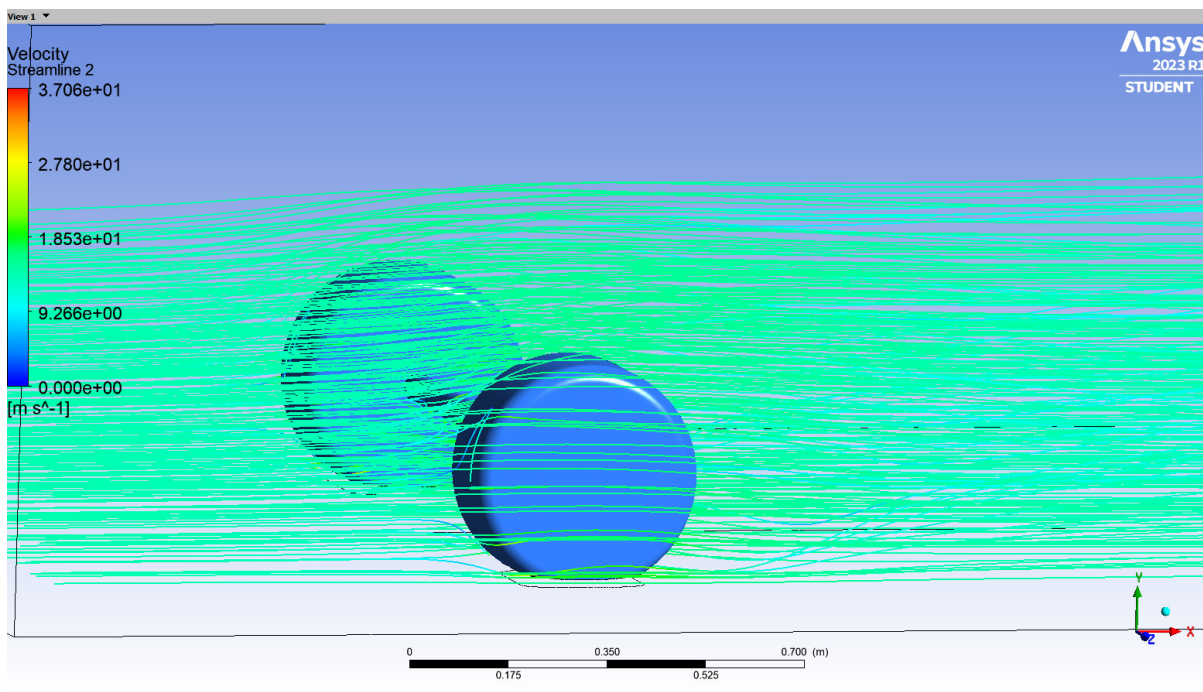
Does the motion of the wheels and ground affect the flow in terms of drag and lift forces?



Let's revisit the velocity contour once more.



Once more, let's examine the streamlines.



Do the contours exhibit any differences? What observations can you make about the flow patterns?

© 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.