FMM ANSYS REPORT "CFD OF AIRPLANE"

OJASVI GOYAL AND HARDIK SHARMA

SUMMARY

This Report is made with help of Youtube videos and without in depth knowledge of ANSYS and FLUENT software.
Project here depicts the pathlines, contours and a ANSYS WORKBENCH report of the CFD done on FLUENT software.
The Plane Model was inspired from the Youtube video and made in Solid Works software with basis geometry.
Please neglect the tolerance and closure defects if found any in this report and CFD cause it may have been a geometrical error

INTRODUCTION

There's an experimental Model of airplane taken from inspiration from an youtube video.
This project is still in the conceptual design stage and therefore a lot of details are not fully known. Nevertheless, it is necessary to get an indication of the basic aerodynamic properties. It is not desirable to obtain these properties by a wind tunnel experiment in this stage. In the design stage the geometry of the plane will be changed constantly and so it would be very time consuming if after each small adaption a new model must be made. Therefore, there is chosen to perform the analysis by making use of a CFD analysis. The basic aerodynamic properties to be investigated are the lift and the drag.
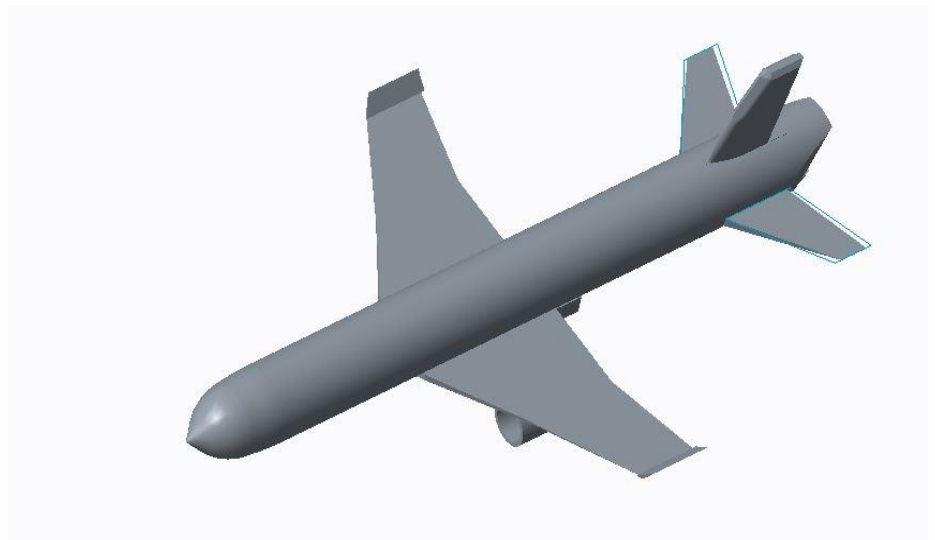

Figure 1

MANUAL

A I M

To get Full Analysis of velocity and Pressure variations over the airplane contour through the help of CFD simulation on ANSYS (FLUENT).

A P P A R A T U S   R E Q U I R E D

A capable pc, knowledge for ANSYS, ANSYS workbench, ANSYS FLUENT, ANSYS Design Modeler, ANSYS Mesher
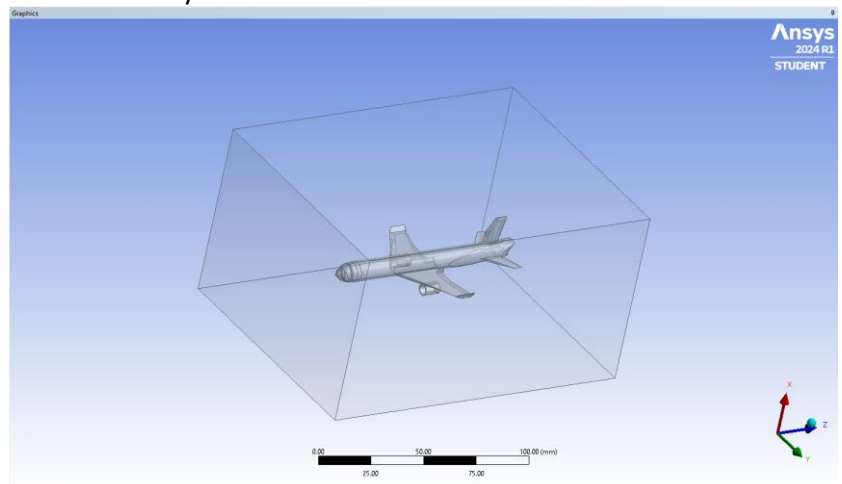
T H E O R Y

ANSYS FLUENT has the advantage of coupled solvers, which can contribute to a better convergence for pressure-based solvers. Therefore, will the analysis be performed in ANSYS FLUENT. The meshing can be done in ANSYS MESHING, but also in ICEM CFD. ICEM CFD can generate a structured mesh, which is an orientated mesh and therefore of good quality for complex geometries. While ANSYS FLUENT has the benet that only one type of software will be used which will reduce the amount of time to get familiar with the software. Because of the very complex geometry a structured mesh would lead to a mesh with a lot of different orientated groups of elements which must be orientated manually and is very time consuming to create. So therefore, an unstructured is used, which makes it unnecessary to make use of ICEM CFD, so ANSYS MESHING is used for the meshing There is already a model of the airplane present, which is made in CATIA V5. This model contains a highly detailed representation of the engine, which is not very interesting for this type of analysis. Therefore, will this detailed representation be simplified to an 'actuator disk'. An actuator disk is a mathematical model mainly used for propellers. An actuator disk is an infinitely thin disk where the fluid can flow through while experiencing a pressure jump (discontinuity). In this way we can represent the effects of the engine on the surrounding flow in an easier way, which is more suited for this analysis

PROCEDURE

1. Create a Geometry using Solid Works.
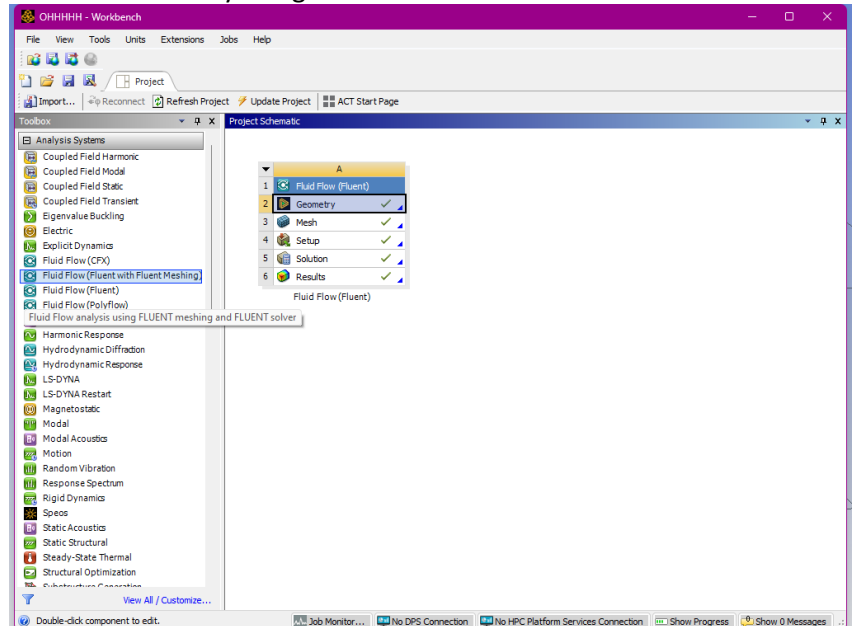
2. Open Work Bench R1 and choose Cello FLUID FLOW (FLUCENT).
3. Then Import the geometry.
4. Open the Design Modeller for Geometry.
5. Create/Generate the Geometry and create a suitable Enclosure to it for better analysis.

6. Now update the workbench project and open mesher.
7. Create the Namespaces "INLET" and "OUTLET".

8. Create the mesh which will look like this (in wireframe view).

9. Finally save and upgrade the project and open the setup for FLUENT.

10. Finally set the parameters and use the Outlined Report to generate a report.
11. The REPORT is Attached to next page→

CFD of a airplane

Analyst	FR34K
Date	4/21/2024 02:40 AM

Table of Contents

System Information

Application	Fluent
Settings	3d, double precision, pressure-based, realizable k-epsilon
Version	24.1.0-10184
Source Revision	5b3f9fb3c8
Build Time	Nov 22 2023 10:32:41 EST
CPU	11th Gen Intel(R) Core(TM) i5-11300H @ 3.10GHz
OS	Windows

Mesh Size

Cells	Faces	Nodes
325818	787636	61591

Mesh Quality

Name	Type	Min Orthogonal Quality	Max Aspect Ratio
solid	Tet Cell	0.0076112454	221.87089
_____msbr	Tet Cell	0.00024701575	189.28503

Orthogonal Quality

Simulation Setup
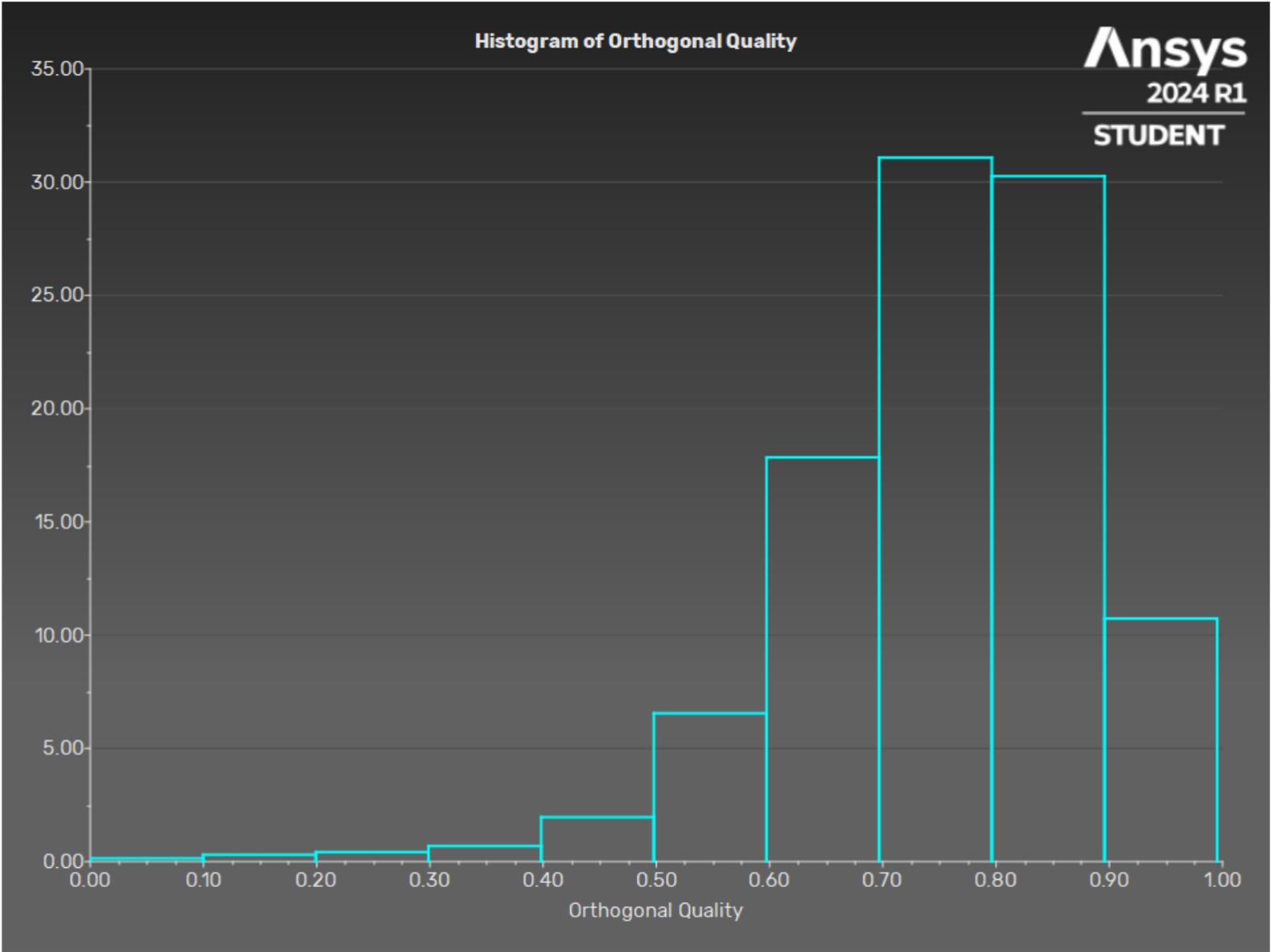
Physics

Models

Model	Settings
Space	3D
Time	Steady
Viscous	Realizable k-epsilon turbulence model
Wall Treatment	Standard Wall Functions

Material Properties

Fluid	
air	
Density	1.225 kg/m^3
Cp (Specific Heat)	1006.43 J/(kg K)
Thermal Conductivity	0.0242 W/(m K)
Viscosity	1.7894e-05 kg/(m s)
Molecular Weight	28.966 kg/kmol
Solid	
aluminum	
Density	2719 kg/m^3
Cp (Specific Heat)	871 J/(kg K)
Thermal Conductivity	202.4 W/(m K)

Cell Zone Conditions

Fluid	
solid	
Material Name	air
Specify source terms?	no
Specify fixed values?	no
Frame Motion?	no
Laminar zone?	no
Porous zone?	no
3D Fan Zone?	no
Solid	
____msbr	
Material Name	aluminum
Frame Motion?	no
Solid Motion?	no

Boundary Conditions

Inlet

inlet

Velocity Specification Method	Magnitude, Normal to Boundary
Reference Frame	Absolute
Velocity Magnitude [m/s]	60
Supersonic/Initial Gauge Pressure [Pa]	0
Turbulent Specification Method	Intensity and Viscosity Ratio
Turbulent Intensity [%]	5
Turbulent Viscosity Ratio	10

Outlet

outlet

Backflow Reference Frame	Absolute
Gauge Pressure [Pa]	0
Pressure Profile Multiplier	1
Backflow Direction Specification Method	Normal to Boundary
Turbulent Specification Method	Intensity and Viscosity Ratio
Backflow Turbulent Intensity [%]	5
Backflow Turbulent Viscosity Ratio	10
Backflow Pressure Specification	Total Pressure
Build artificial walls to prevent reverse flow?	no
Radial Equilibrium Pressure Distribution	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no

Wall

wall-7-shadow

Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Wall Roughness Height [m]	0
Wall Roughness Constant	0.5

wall-7	wall

wall-13

Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Wall Roughness Height [m]	0
Wall Roughness Constant	0.5

wall-12	wall

wall-solid

Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Wall Roughness Height [m]	0
Wall Roughness Constant	0.5

Reference Values

Area	1 m^2
Density	1.225 kg/m^3
Enthalpy	0 J/kg
Length	1 m
Pressure	0 Pa
Temperature	288.16 K
Velocity	60 m/s
Viscosity	1.7894e-05 kg/(m s)
Ratio of Specific Heats	1.4
Yplus for Heat Tran. Coef.	300
Reference Zone	solid

Solver Settings

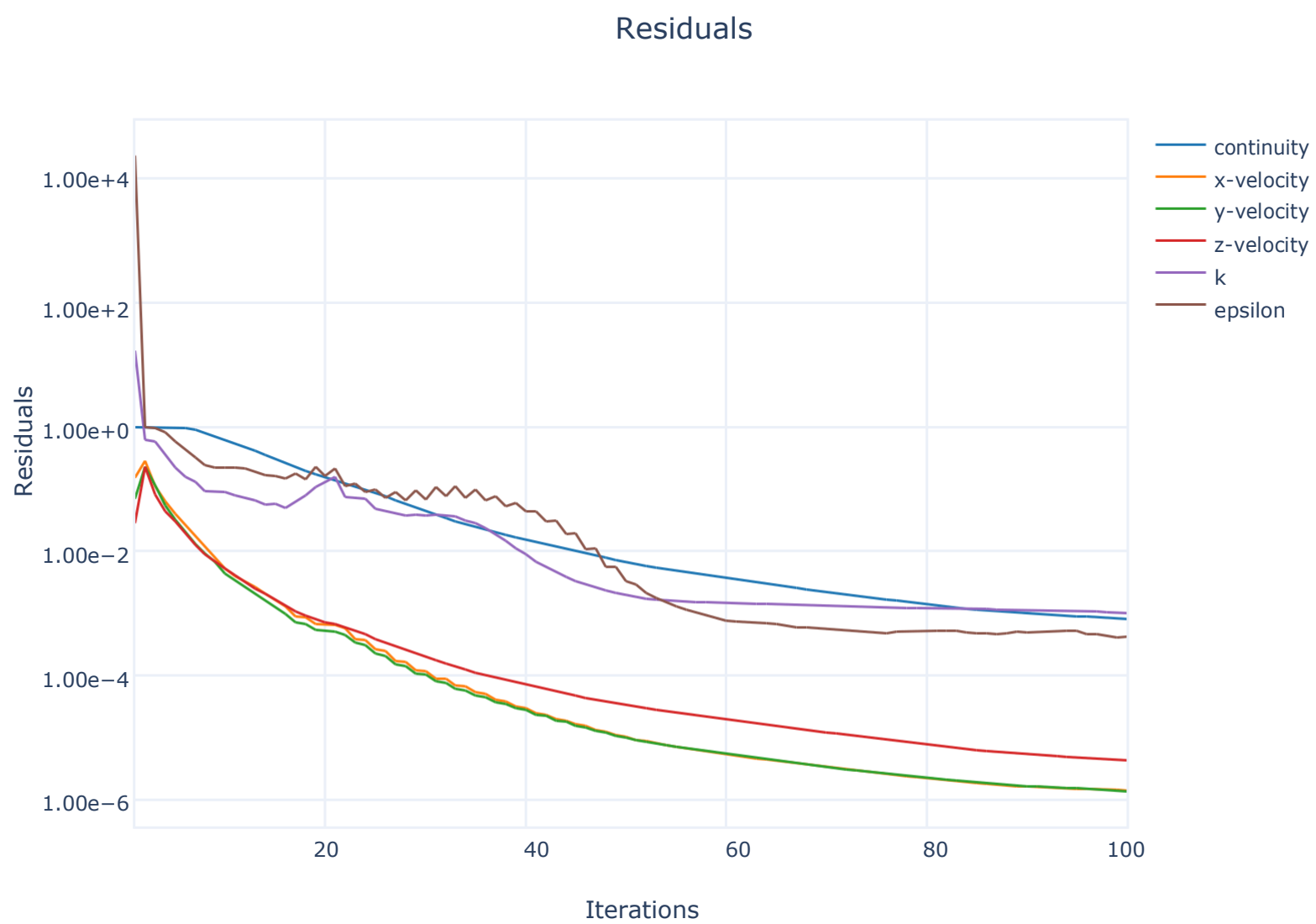| Equations	
Flow	True
Turbulence	True
Numerics	
Absolute Velocity Formulation	True
Pseudo Time Explicit Relaxation Factors	
Density	1
Body Forces	1
Turbulent Kinetic Energy	0.75
Turbulent Dissipation Rate	0.75
Turbulent Viscosity	1
Explicit Momentum	0.5
Explicit Pressure	0.5
Pressure-Velocity Coupling	
Type	Coupled
Pseudo Time Method (Global Time Step)	True
Discretization Scheme	
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind
Solution Limits	
Minimum Absolute Pressure [Pa]	1
Maximum Absolute Pressure [Pa]	5e+10
Minimum Static Temperature [K]	1
Maximum Static Temperature [K]	5000
Minimum Turb. Kinetic Energy [m^2/s^2]	1e-14
Minimum Turb. Dissipation Rate [m^2/s^3]	1e-20
Maximum Turb. Viscosity Ratio	100000

Run Information

Number of Machines	1
Number of Cores	1
Case Read	8.017 seconds
Iteration	251.953 seconds
AMG	165.482 seconds
Virtual Current Memory	1.53702 GB
Virtual Peak Memory	1.86441 GB
Memory Per M Cell	2.99554

Solution Status

Iterations: 100

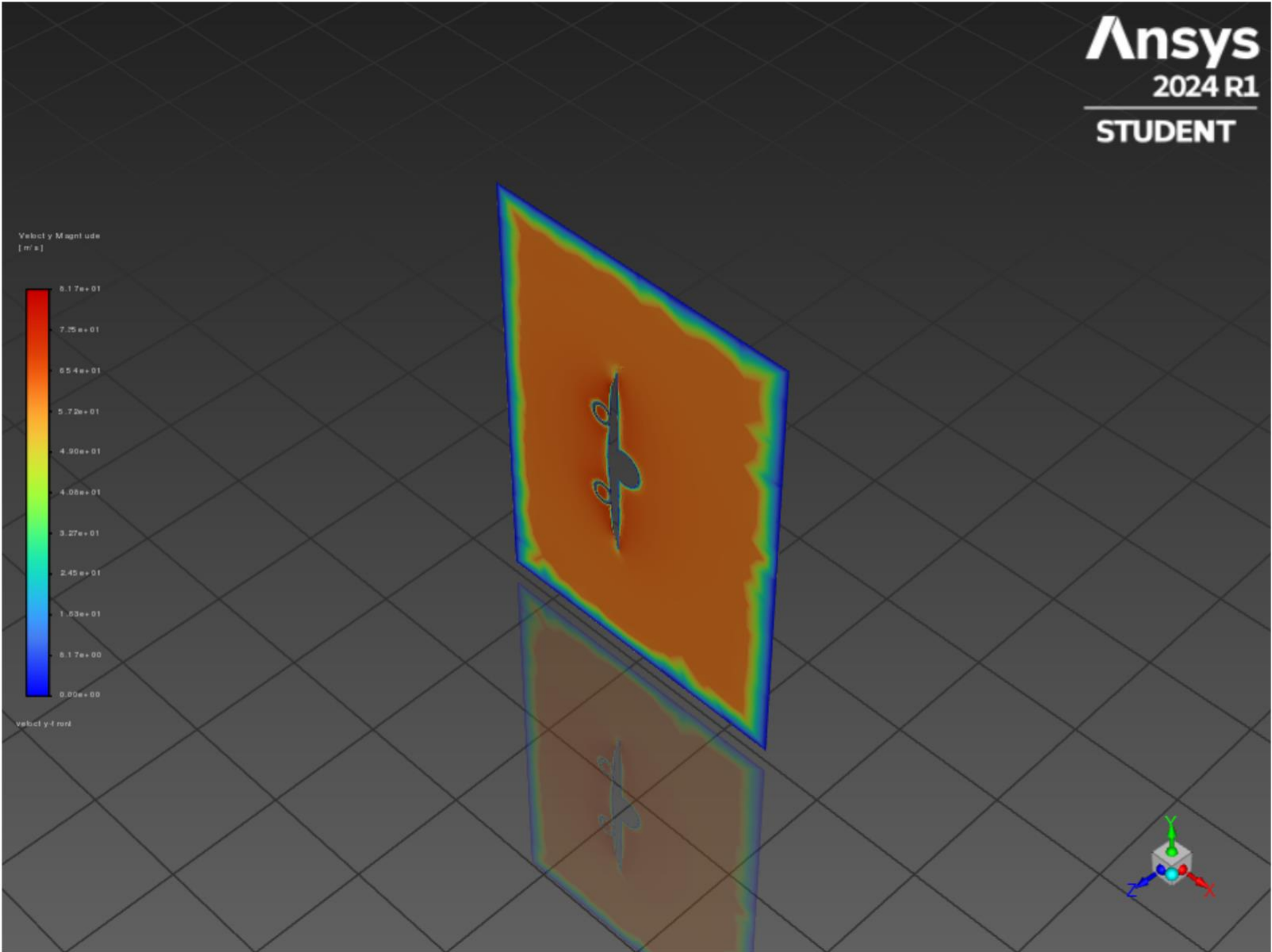|	Value	Absolute Criteria	Convergence Status
continuity	0.0008184233	0.001	Converged
x-velocity	1.371341e-06	0.001	Converged
y-velocity	1.348966e-06	0.001	Converged
z-velocity	4.219996e-06	0.001	Converged
k	0.001006511	0.001	Not Converged
epsilon	0.0004191457	0.001	Converged

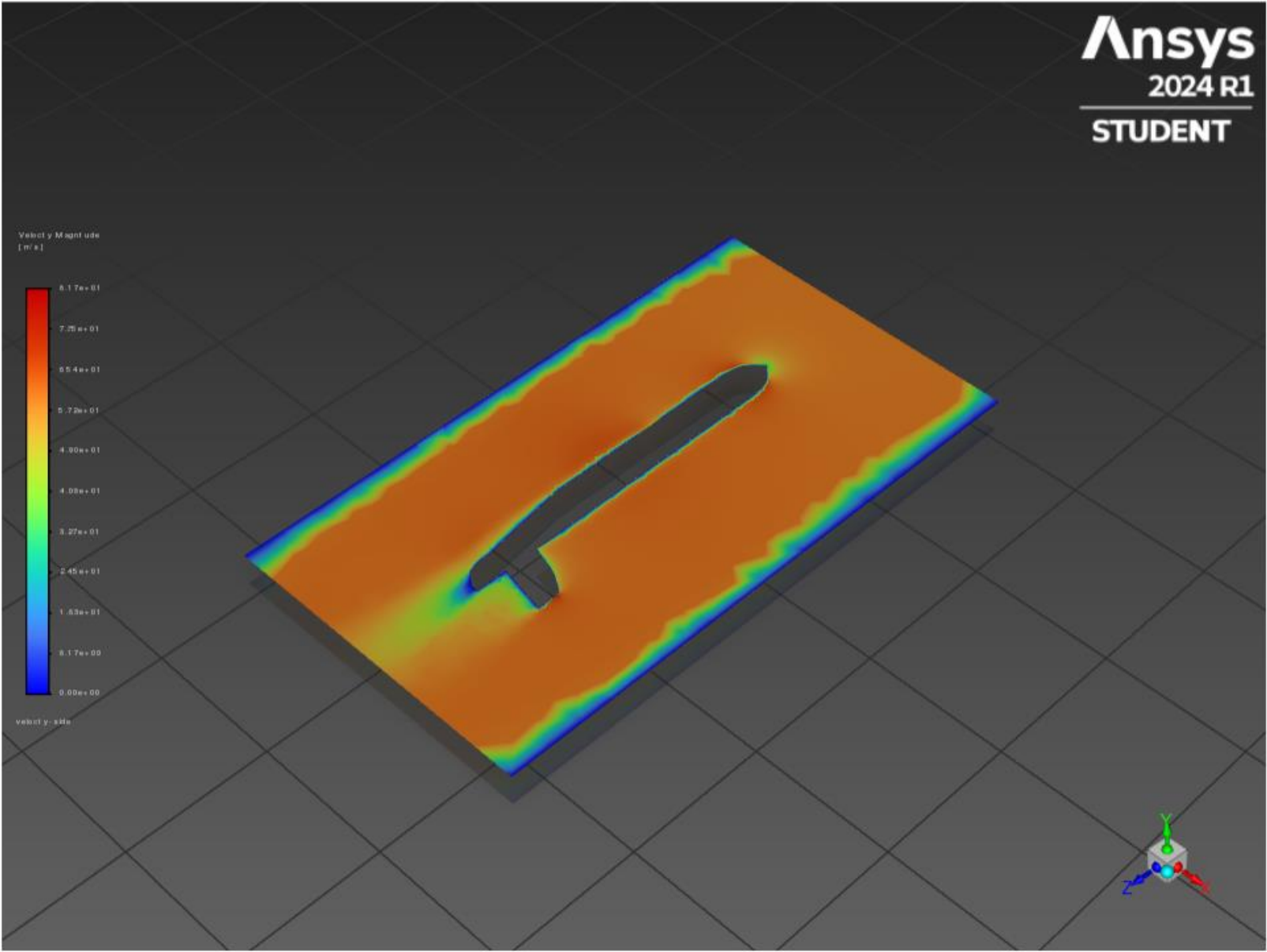# Plots

Residuals

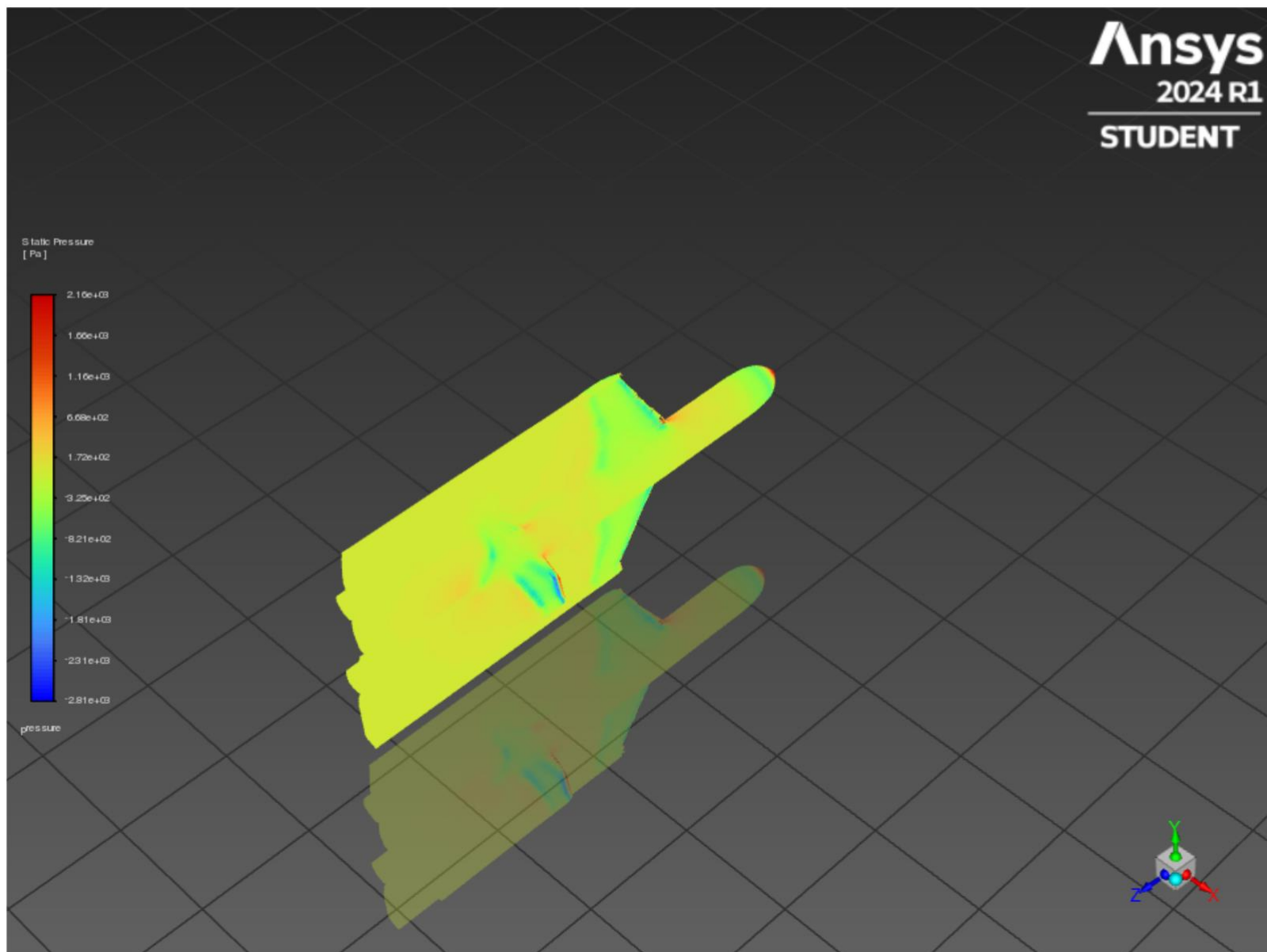

Residuals

Mesh

mesh-1

Contours
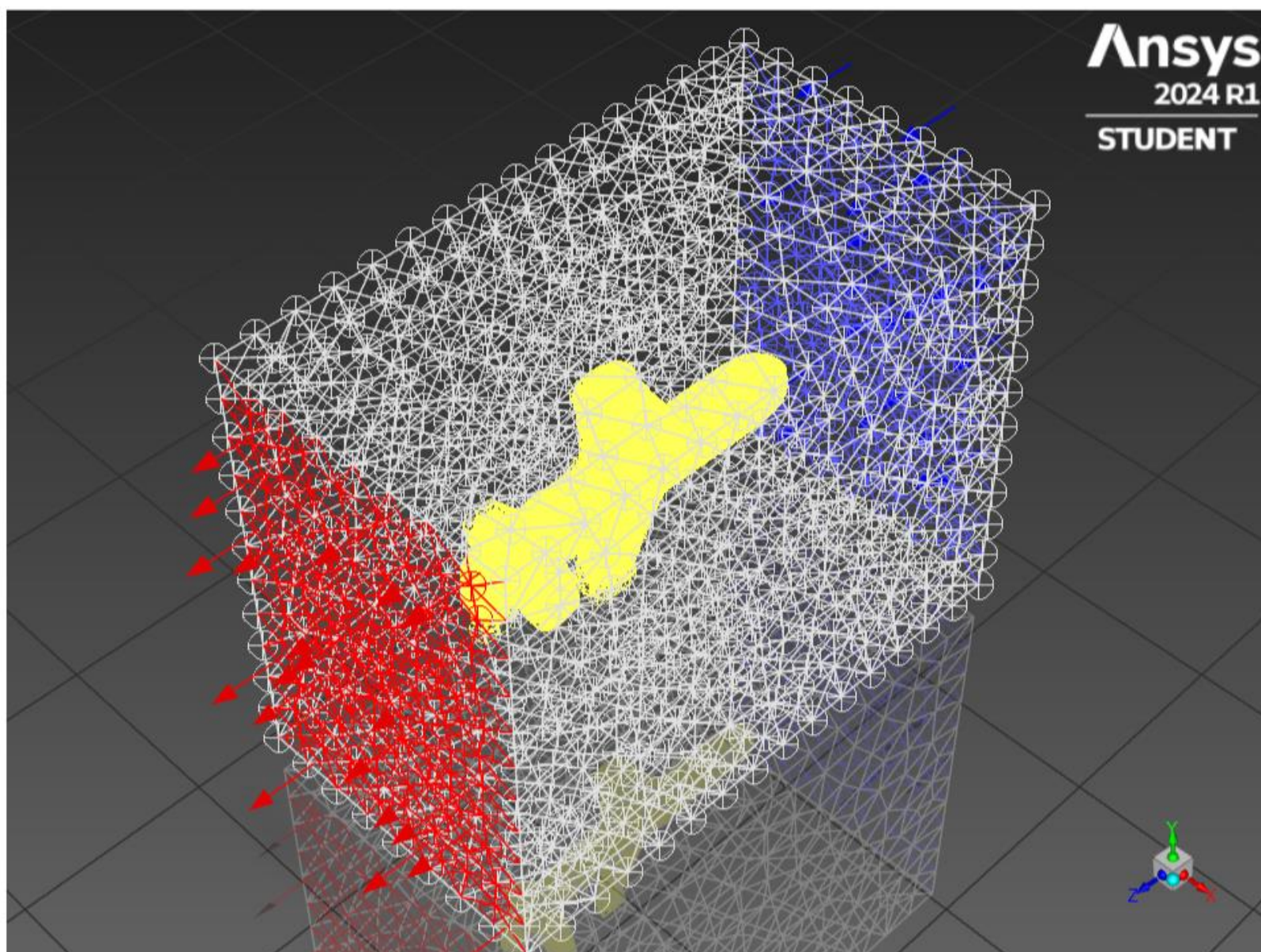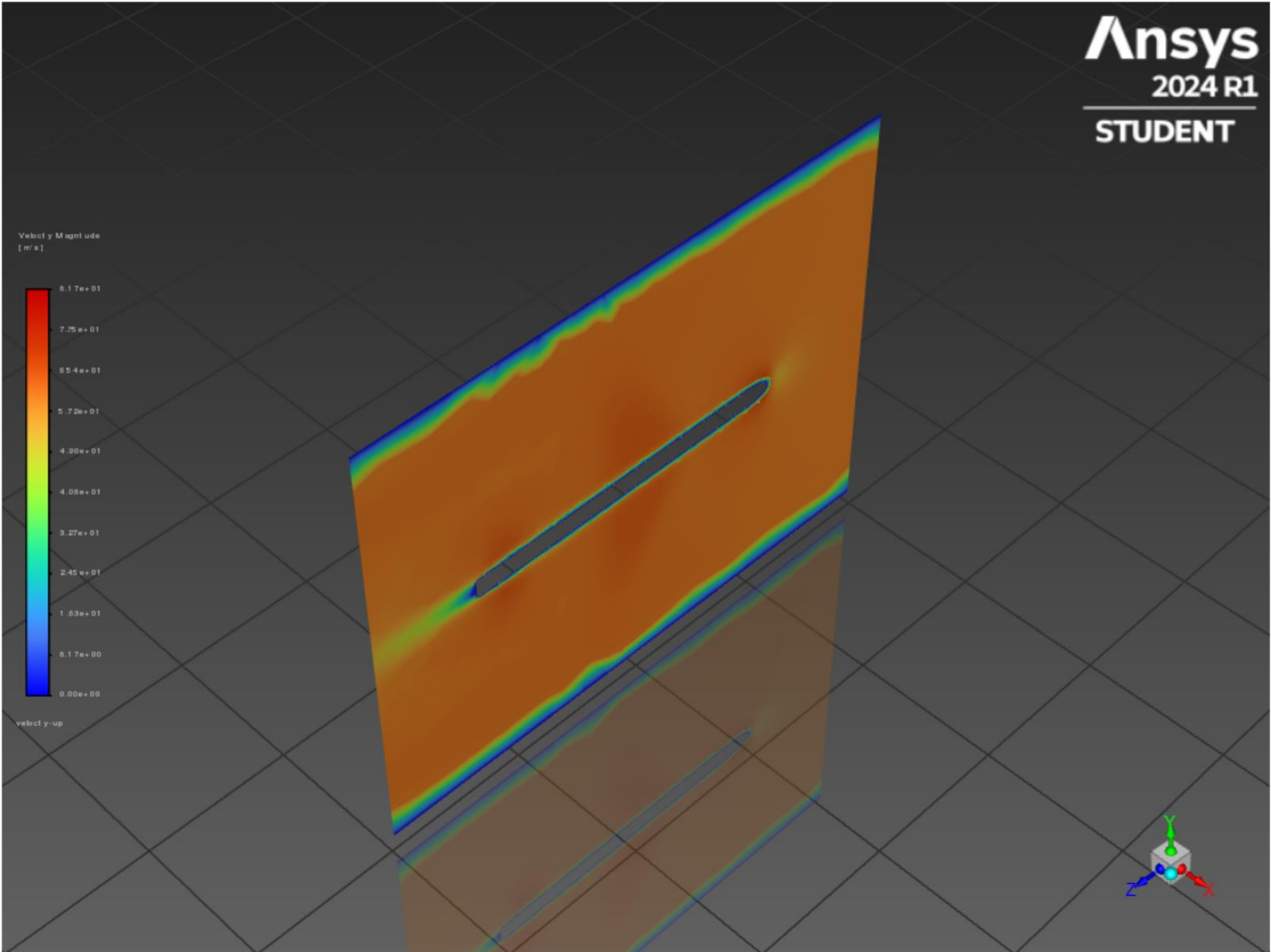
velocity-up

velocity-front

velocity-side

pressure

velocity

velocity-plot

Velocity Magnitude

(legend: plane-13)

X-axis: Position (m)
Y-axis: Velocity Magnitude (m/s)
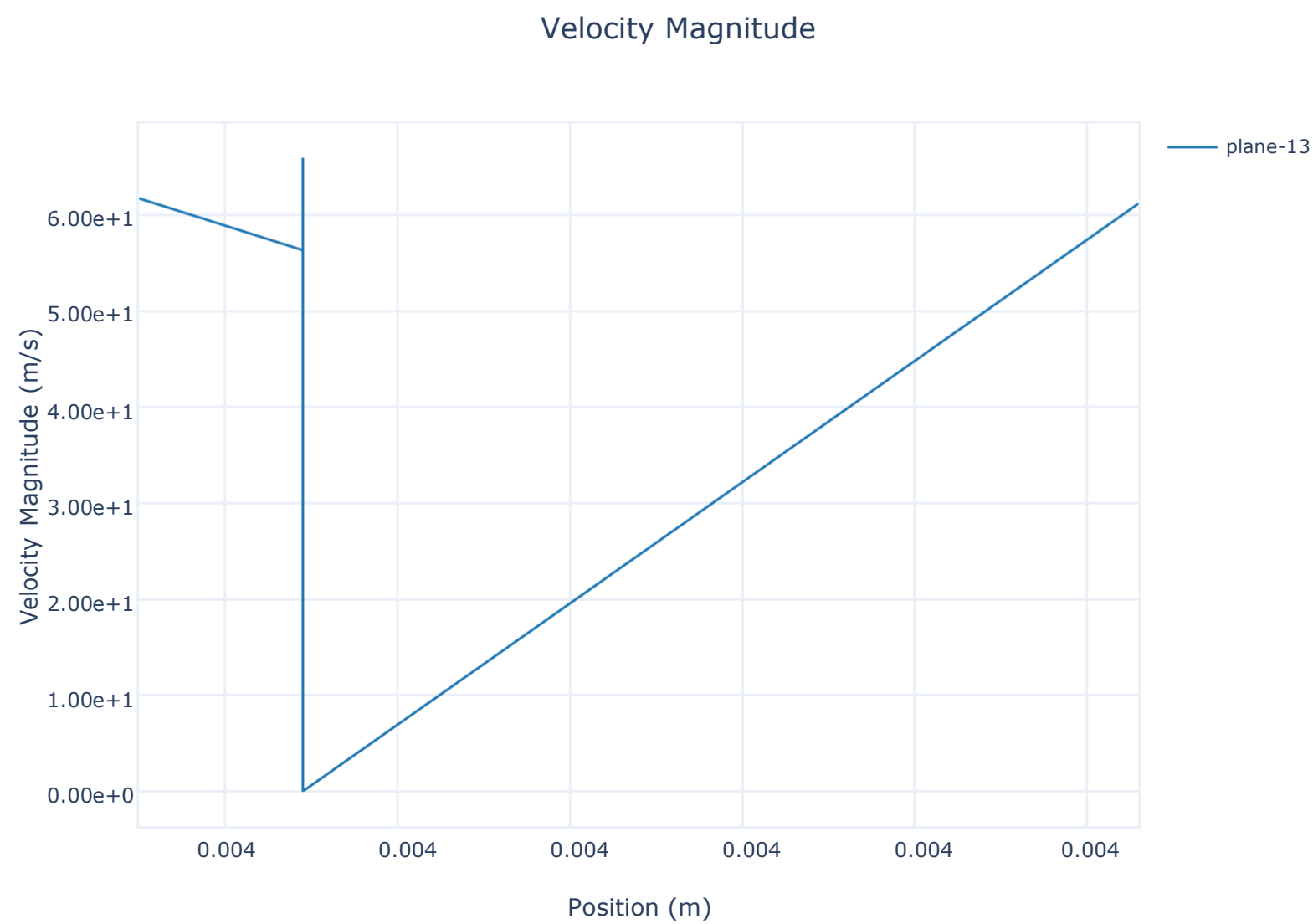
pressure-plot

Static Pressure

(legend: plane-1)

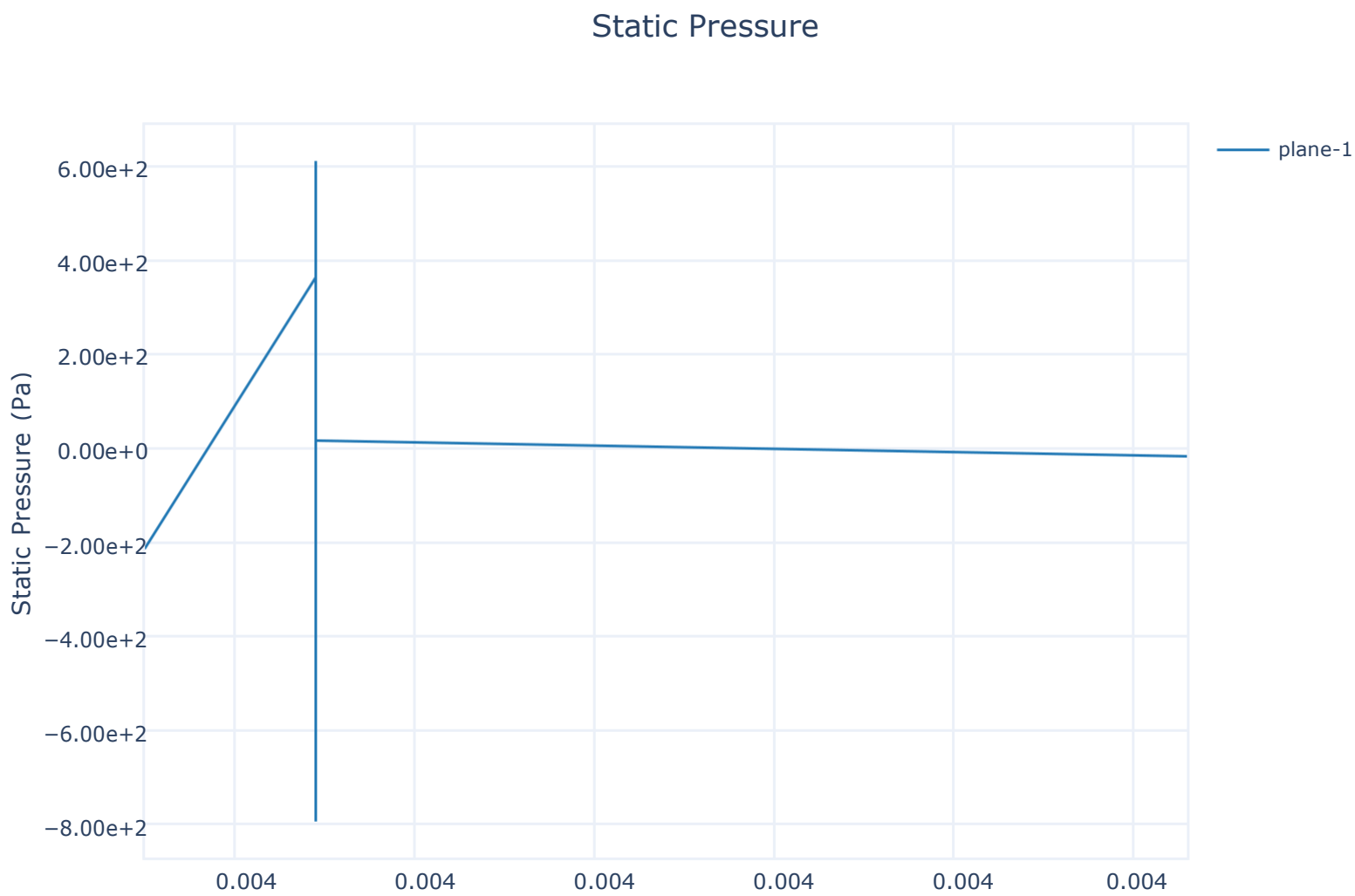Y-axis: Static Pressure (Pa)
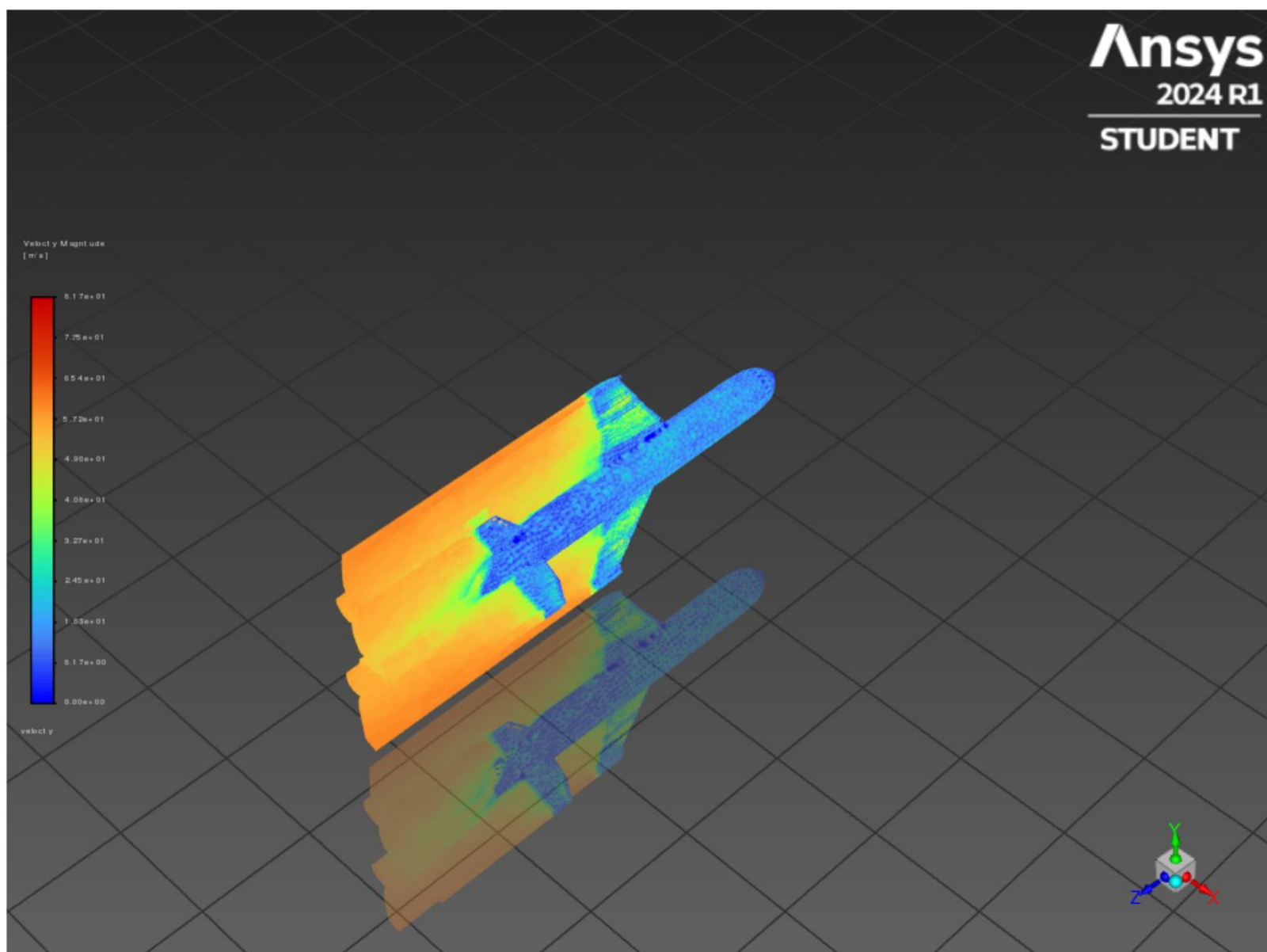
RESULTS

1. We found out the variations of velocity and pressure around a airplane wings and body.
2. We got tolerance issue as can be seen in calculation graph due to presence of geometry issue.
3. Some sudden disturbance at places of jet propulsion engine can be seen due to its contour.
4. The graphs and readings a satisfactorily explaining the fluid flow around a airplane.