University of Science and Technology - Zewail City

Computer-Aided Design and Engineering - Fall 2024

Final Project - Group 7

Prepared by:

Phelopater Ramsis	202001171
Manar Tarek	202100501
Marwa Ali	202001052

Under the supervision of:

Dr. Mohamed Abdelnaby

Eng. Omar Khattab

Contents

1 Problem Description

The following project is a detailed CFD analysis of a Horizontal Axis Wind Turbine blade that was designed in a previous assignment. This uses ANSYS for the aeromechanical performance simulation of the blade and to predict structural stresses and modal responses under specified operational conditions. The main goal of the CFD simulation at this moment is to analyze the performance of the blade from an aeromechanical, structural, and vibrational point of view under prescribed wind velocity of 9 m/s and rotational speed of 400 rpm. Steady-state analysis shall be performed in order to predict the performance of the blade regarding power generation capability, stress distribution along the blade structure, and resonant frequency that could indicate susceptibility to operational vibrations. Given that the material of the blade is wood, whose mechanical properties are predefined, the focus of the project will be specifically on the following:

1. **3D Steady CFD Analysis:** The way the blade will interact with the wind at operational speeds and its interaction with the turbine's efficiency.

2. **Structural Analysis through Fluid-Structure Interaction:** This should be conducted to find out the operational stresses and strains in operation.

3. **Modal Analysis:** Through this, one will have the natural frequencies and the mode shapes; these have to be as far from the operating speed as possible to avoid resonance.

The focus of this project will therefore shift from the blade performance under normal conditions to the feasibility and reliability of the design in long-term deployment in wind turbines.

2 Proposed Solution

Following represents the proposed solution for an ANSYS-based CFD simulation for a HAWT blade:

1. **CFD Analysis:** A complete 3-D, steady-state CFD-thermal simulation has to be conducted in a way that realistically models airflow around a HAWT blade while calculating resultant aerodynamic forces, torque, and power output. Advanced models of turbulence must be involved in this kind of simulation to provide realistic dynamics of airflow.

2. **FSI Simulation:** Pressure distributions obtained from CFD analysis should be used to conduct fluid-structure interaction simulation, which will take into account the structural responses of the wooden blade during operational conditions, thereby helping to evaluate the stress distribution and deformation.

3. **Modal Analysis:** Modal analysis is important to get the natural frequencies and mode shapes of the blade.

4. **Iterative Optimization:** Do iteration in the design of the blade based on the CFD, FSI, and modal analyses results to achieve an optimized performance by minimizing the stress and avoiding resonance.

5. **Integration and Assessment:** Integrate all the results of simulation in order to understand, in an overall trend, the performance of the blade; assess the power coefficient and make sure that such a design will be viable and efficient concerning the existent benchmarks in literature.

3 Methods

3.1 Geometry of Wind Blade

1- Finalizing the Design Phase of Assignment 3 in inventor Software

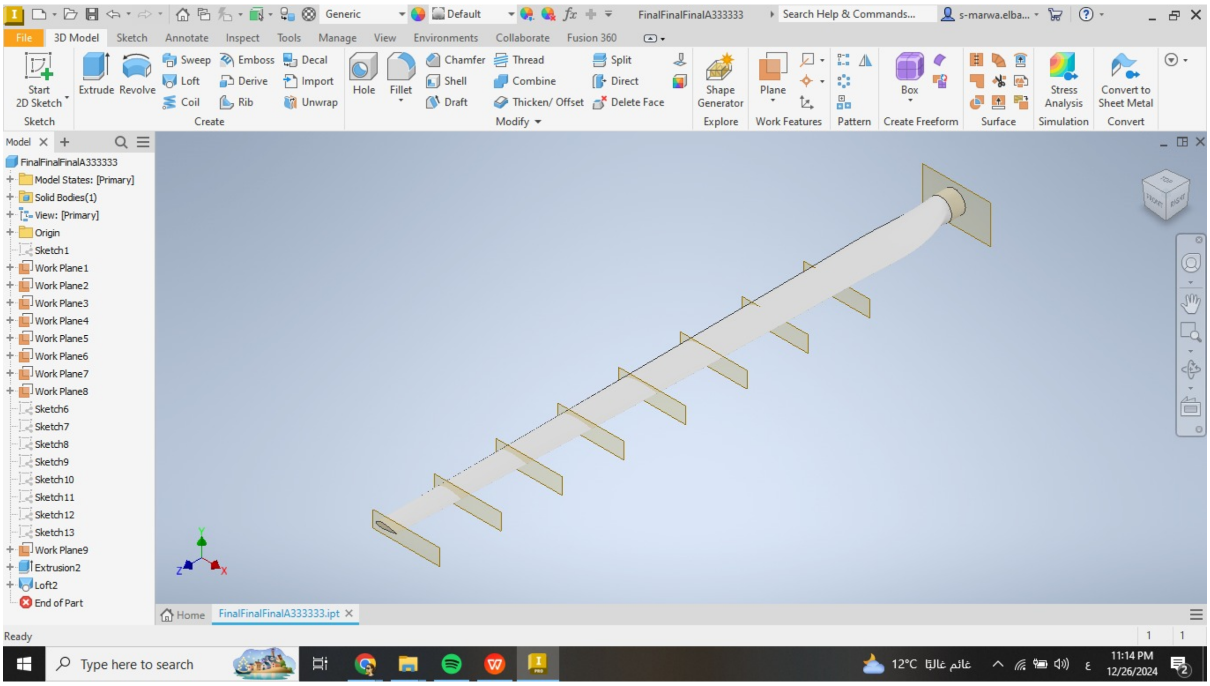

Figure 1: the Design Phase of Assignment 3

2- Fillet to avoid sharp edges and Add curvature Analysis

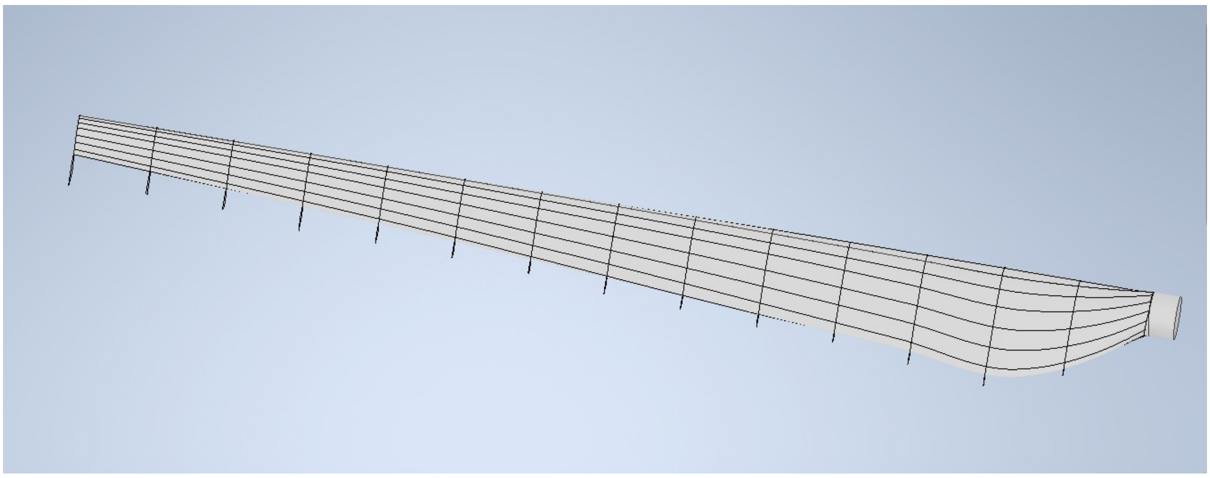

Figure 2: After fillet

3- Edit the geometry coordinates and directions to fit in spaceClaim Ansys

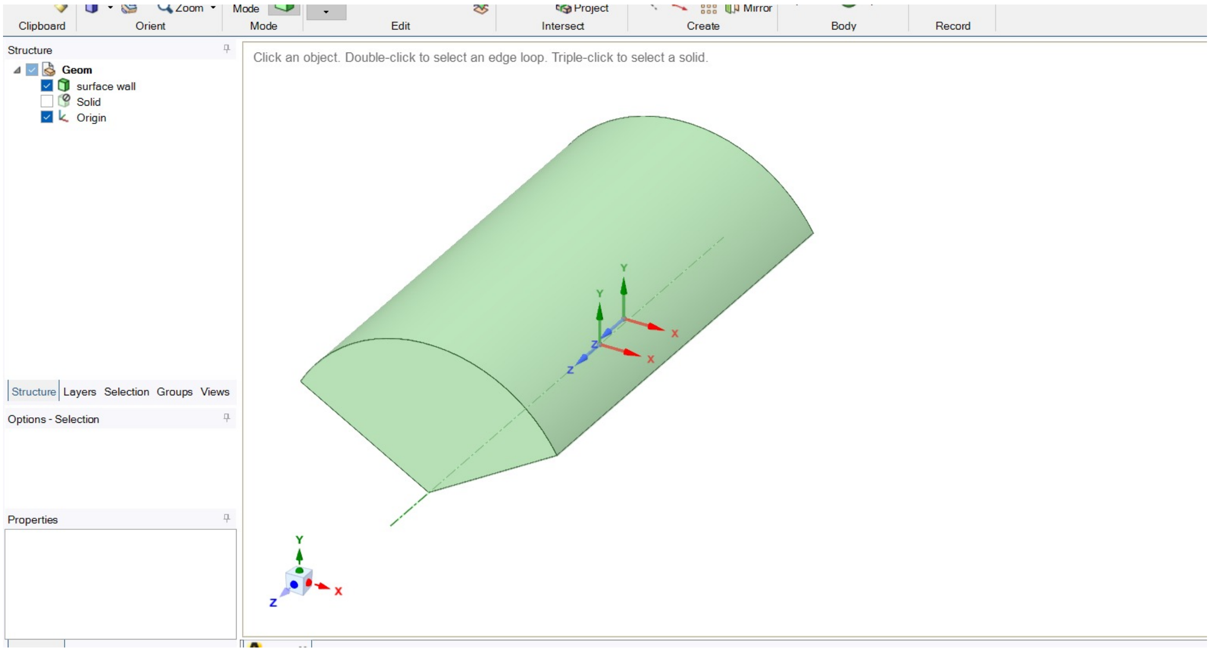

Figure 3: Editing the geometry coordinates and directions

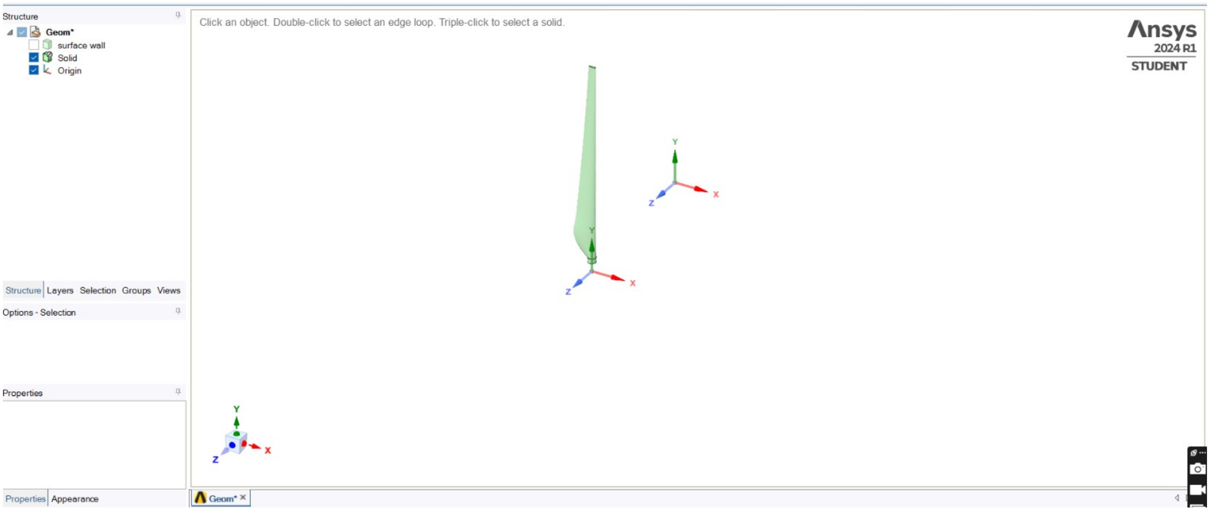

Figure 4: Editing the geometry coordinates and directions

3.2 Mesh (Grid Dependency Study Results.)

1- Import geometry in meshing

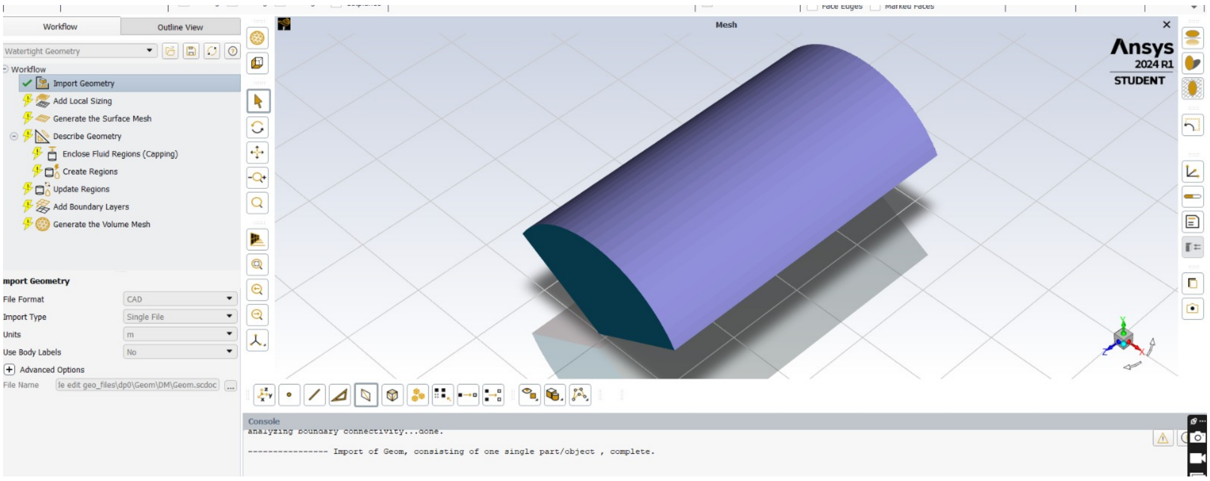

Figure 5: Geometry in meshing

2- Add local sizing

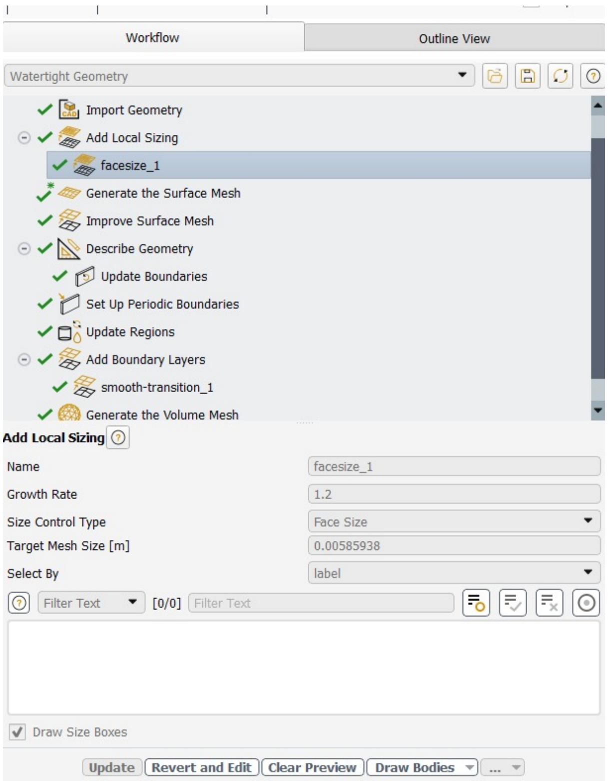

Figure 6: local sizing

3-After generating surface mesh

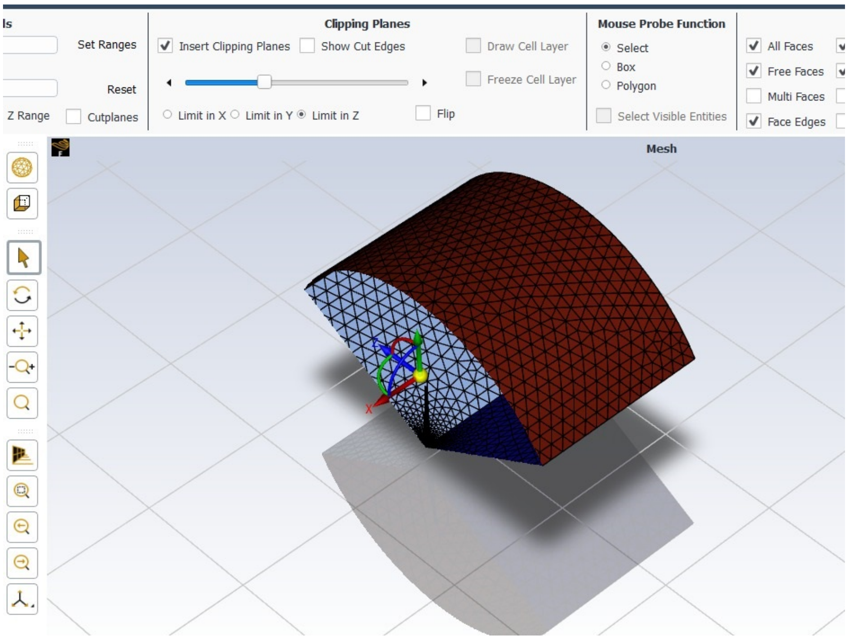

Figure 7: Surface Mesh

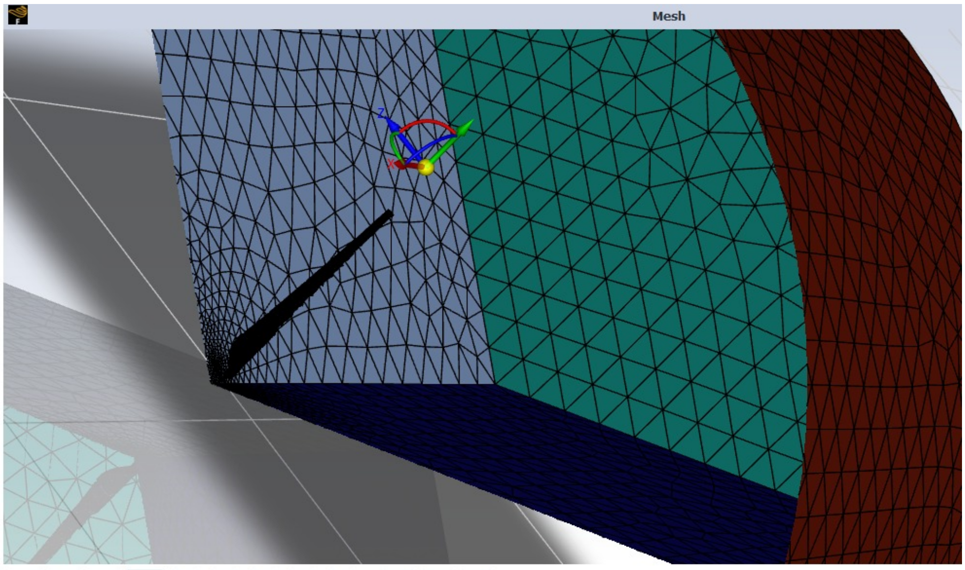

Figure 8: Surface Mesh

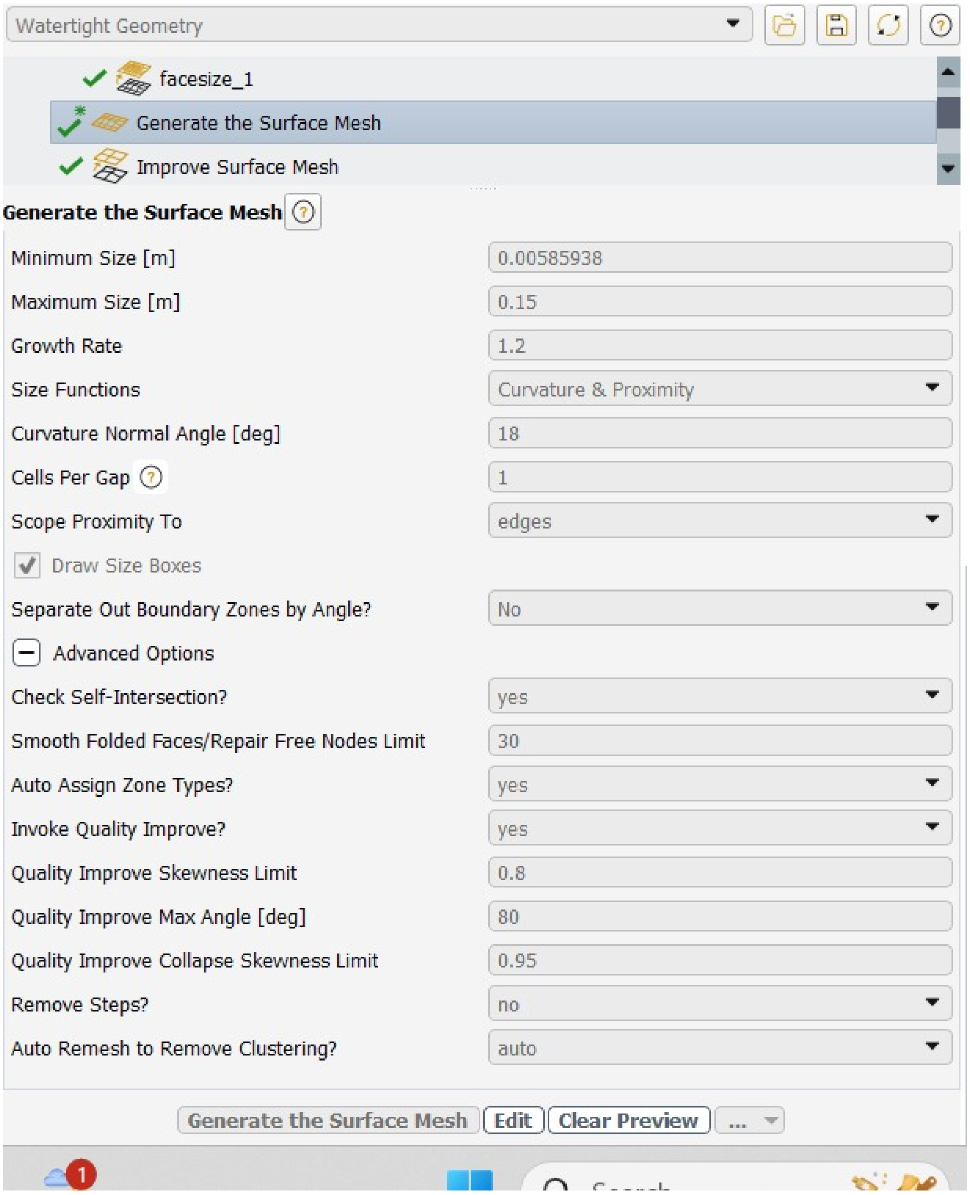

Figure 9: generate the surface mesh.

4-Setting up periodic boundaries

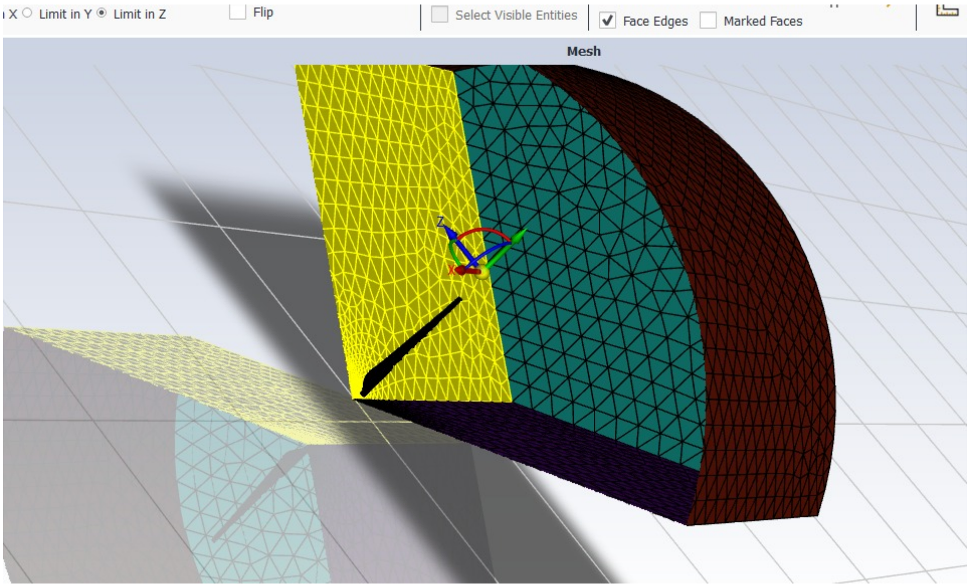

Figure 10: Setting up periodic boundaries

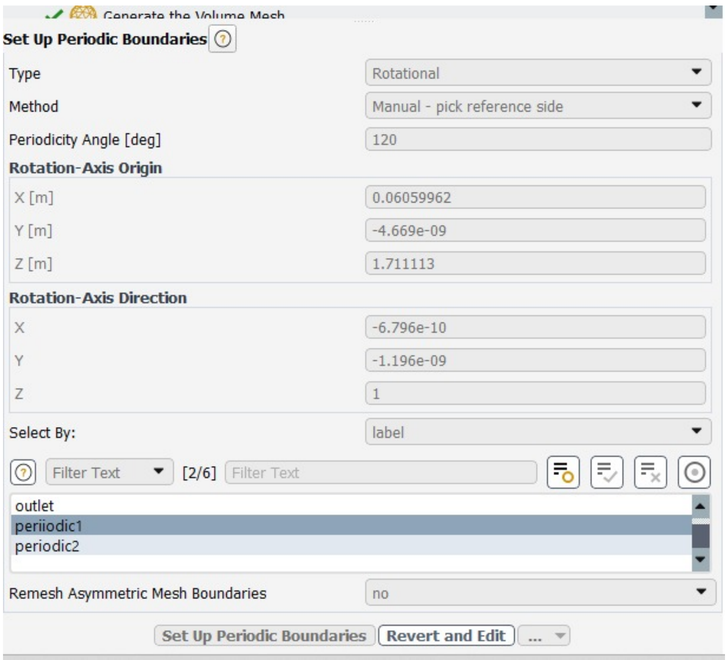

Figure 11: Setting up periodic boundaries

5- Generate the Volume Mesh

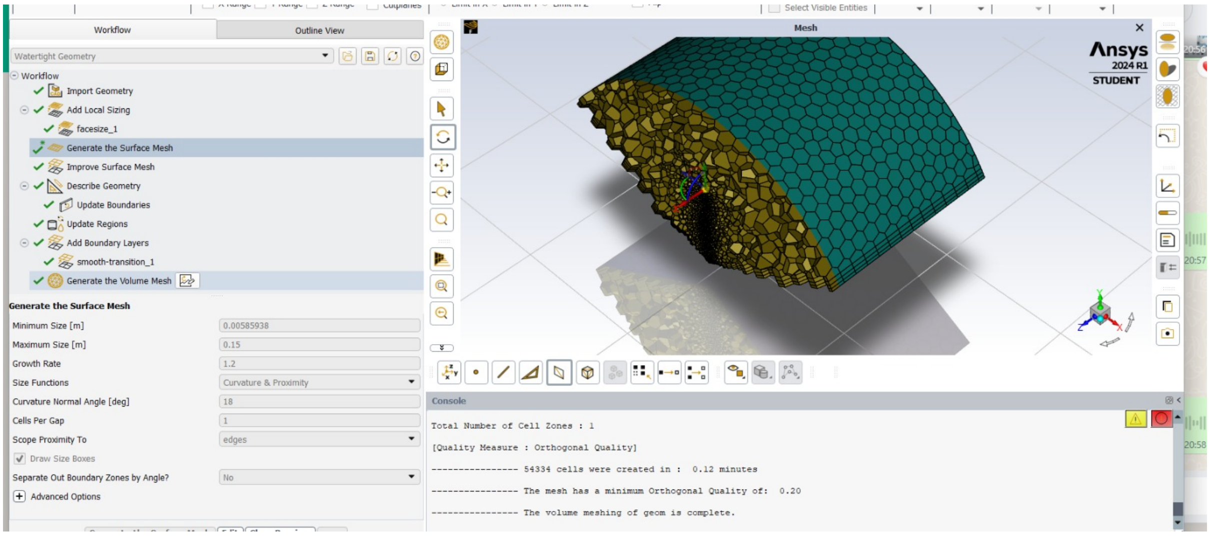

Figure 12: General Orthogonal Quality Mesh

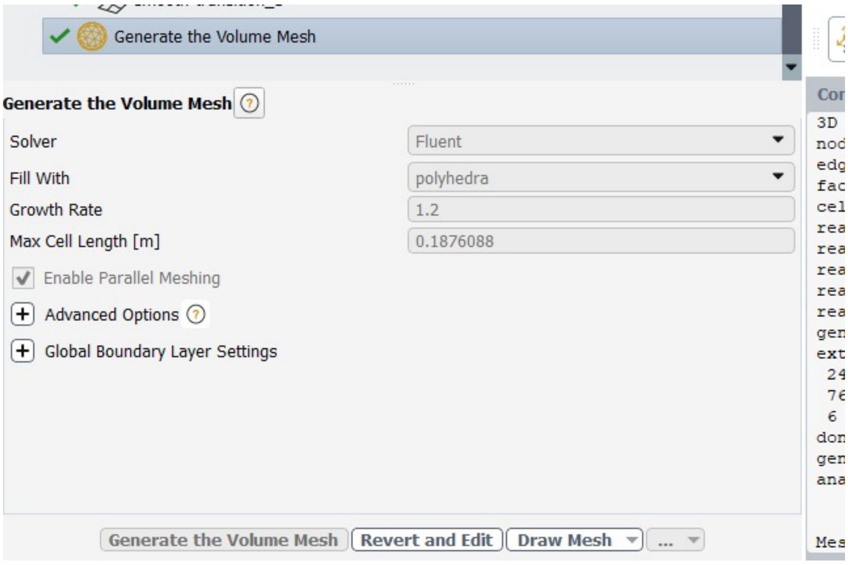

Figure 13: Generate volume mesh

6- Report Mesh size data

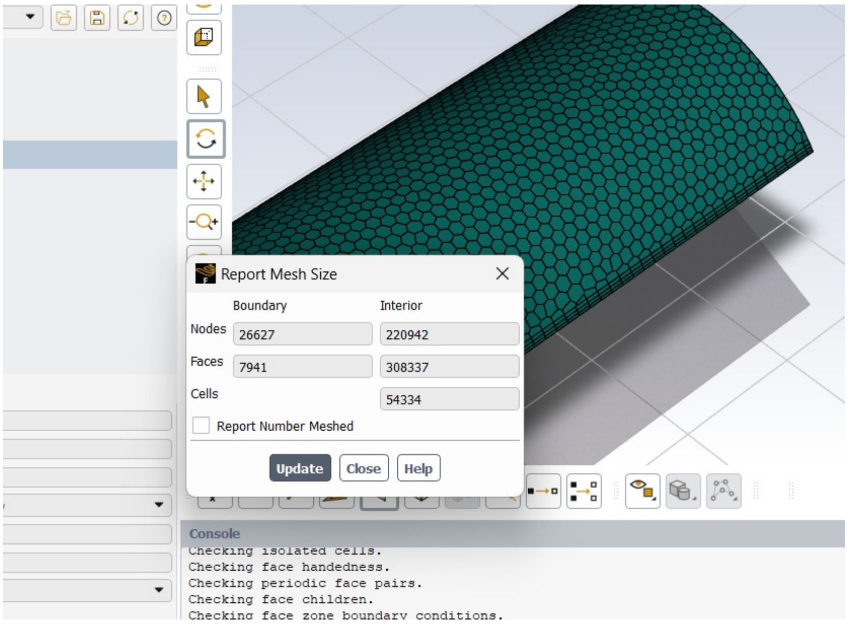

Figure 14: Report Mesh size data

7- Data about the Mesh

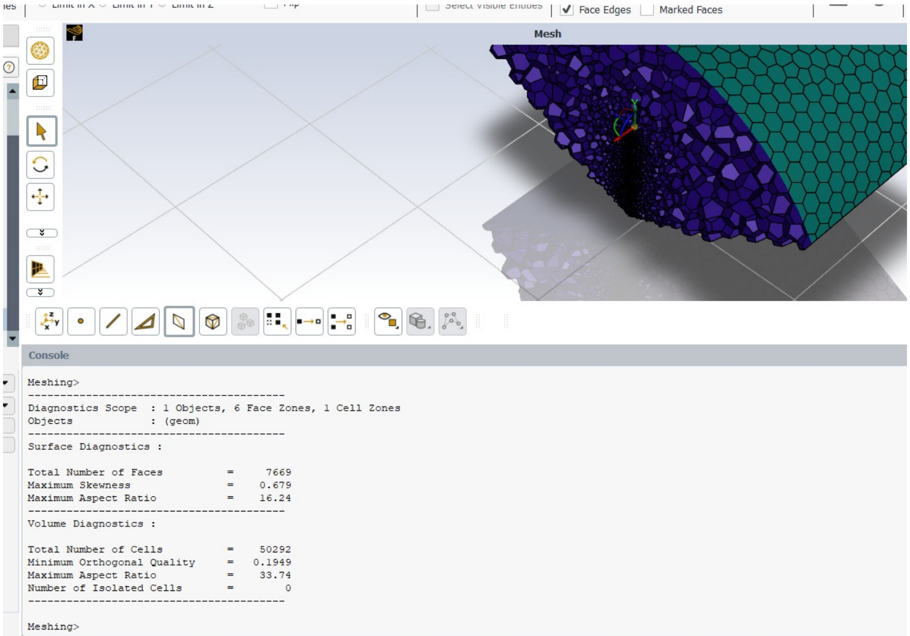

Figure 15: Data about the Mesh

Note: Enhanced meshing has been implemented to improve the quality of the general mesh, as we encountered several errors, such as sharp edges, fillets, etc. After approximately five trials, we needed to improve the data for both surface and sizing meshes. Additionally, we displayed the grid to identify weak points, which led us to regenerate the geometry for repairs. Finally, we added surface mesh and increased the face quality limit to 0.9.

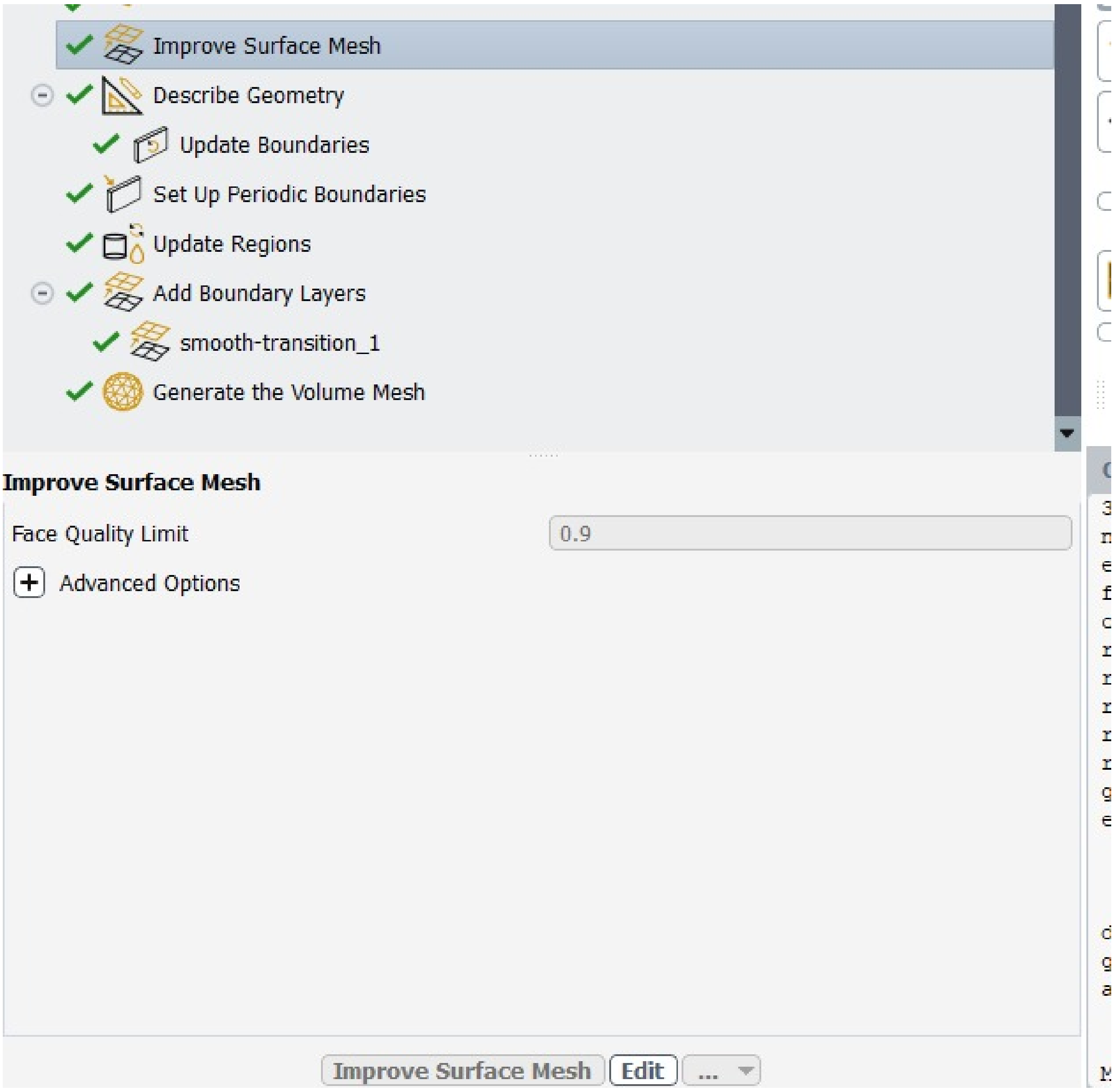

Figure 16: Enhanced Mesh

4 Solution.

4.1 Viscous Model

Choosing K-omega (2 equ.) with SST Model

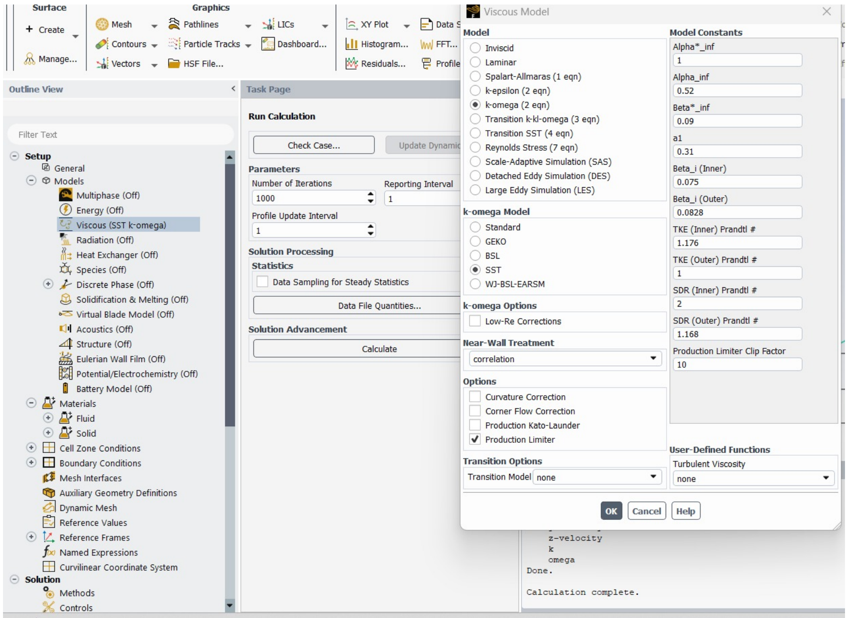

Figure 17: Viscous Model

4.2 Fluid flow motion

Cell zone condition "Velocity rpm to rad/sec"

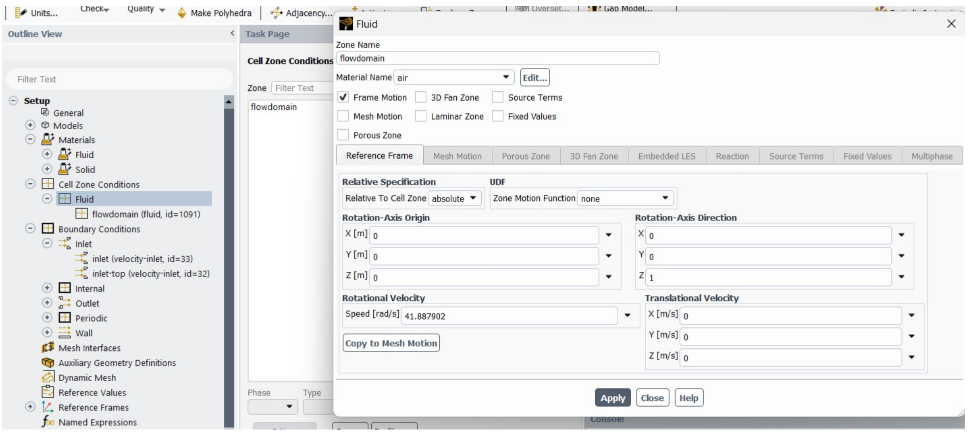

Figure 18: cell zone condition

4.3 Used Boundary Conditions.

Boundary condition with Z velocity of 9 m/s (inlet), and outlet pressure

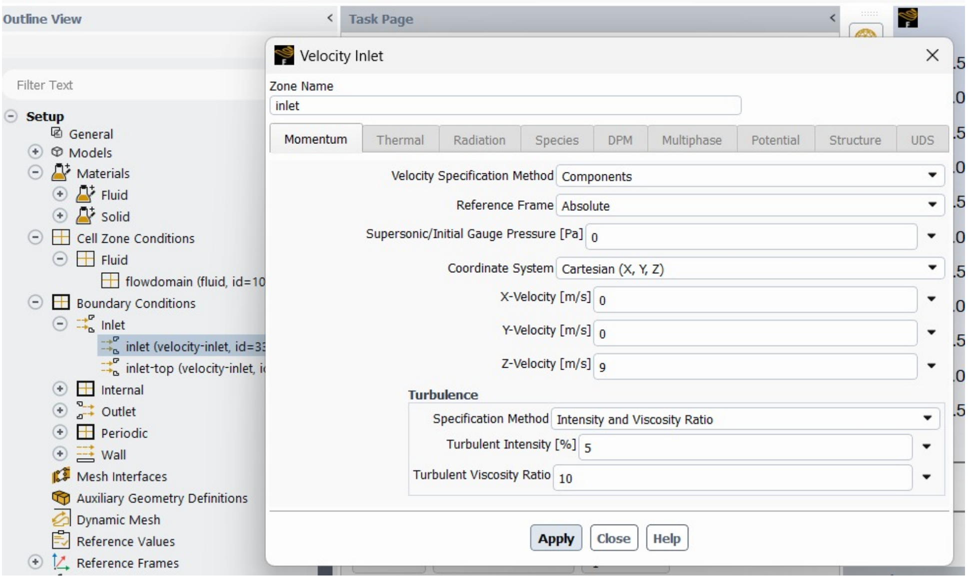

Figure 19: inlet velocity

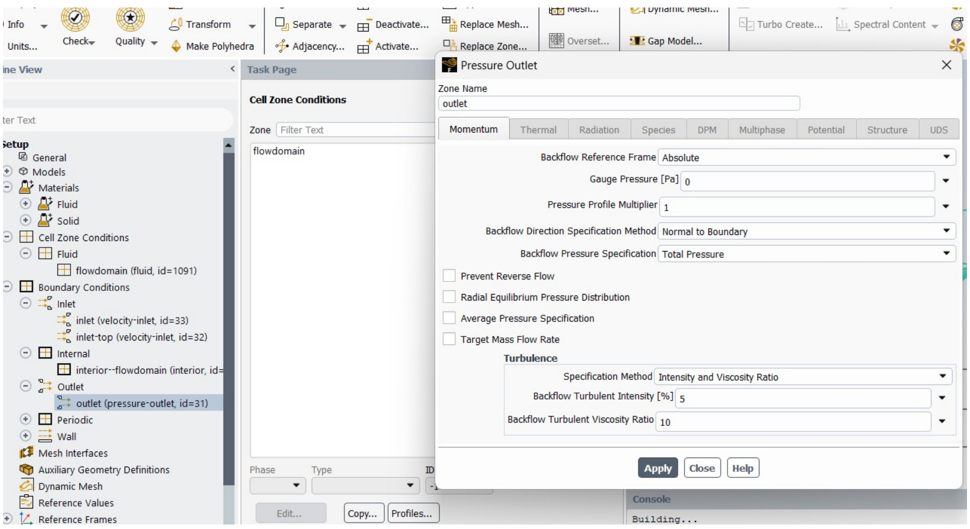

Figure 20: Pressure outlet

4.4 Residual Monitors

Residual Monitors Number of iterations, Equations, and the graph

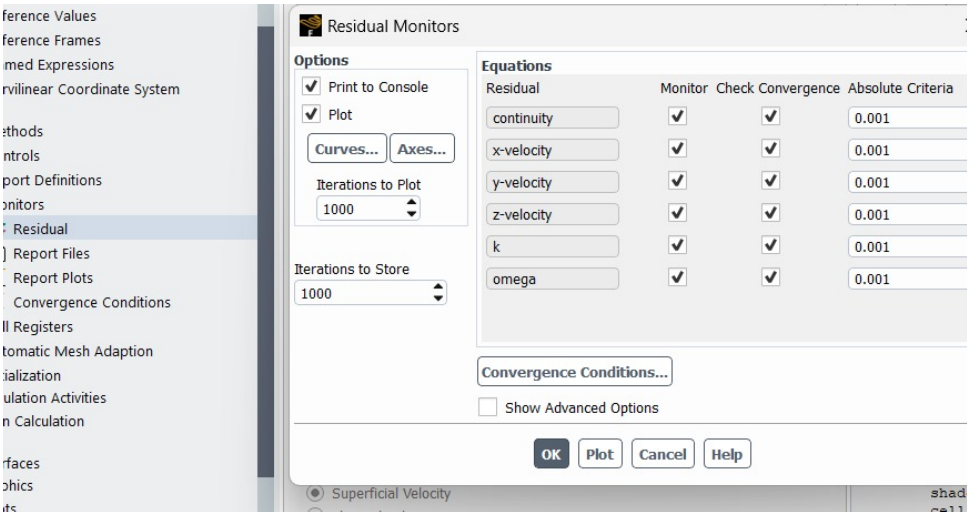

Figure 21: Residual Monitors

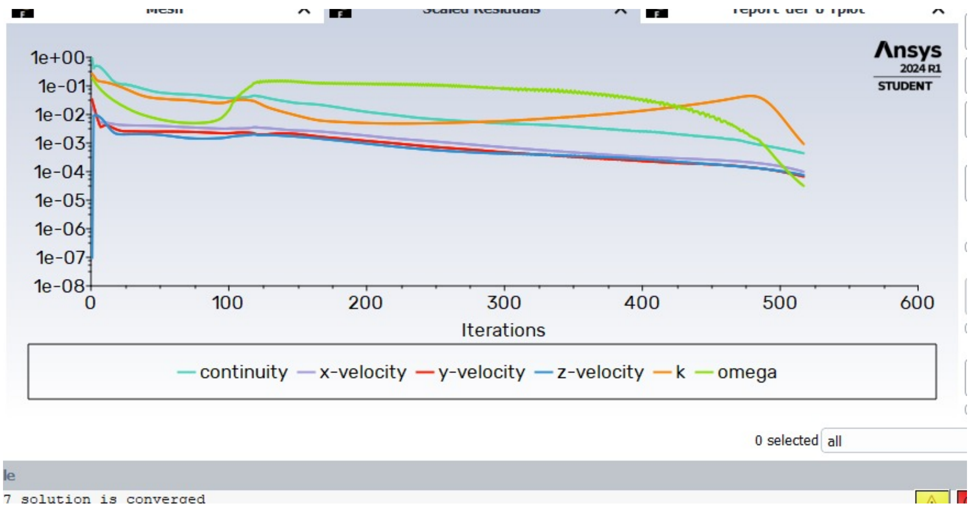

Figure 22: Residual graph

4.5 Surface Report Definition

Static Pressure on the blade

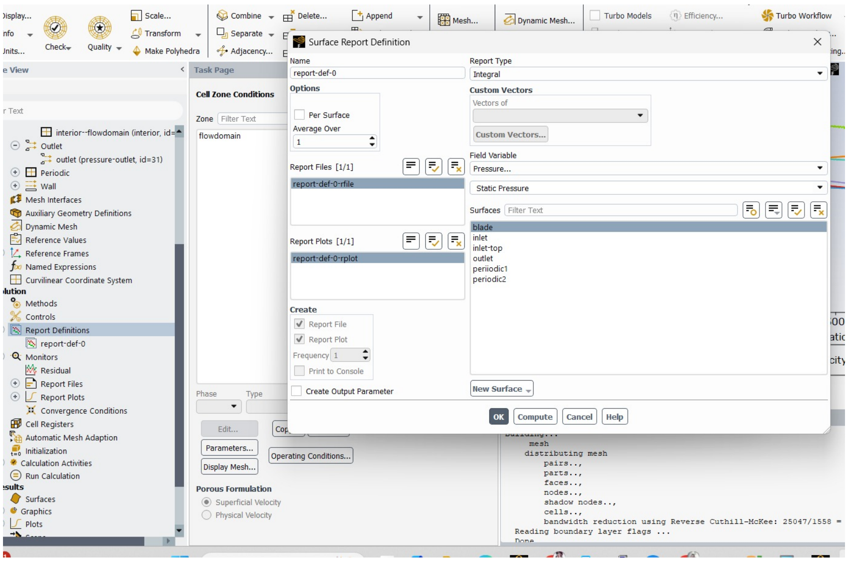

Figure 23: Static Pressure

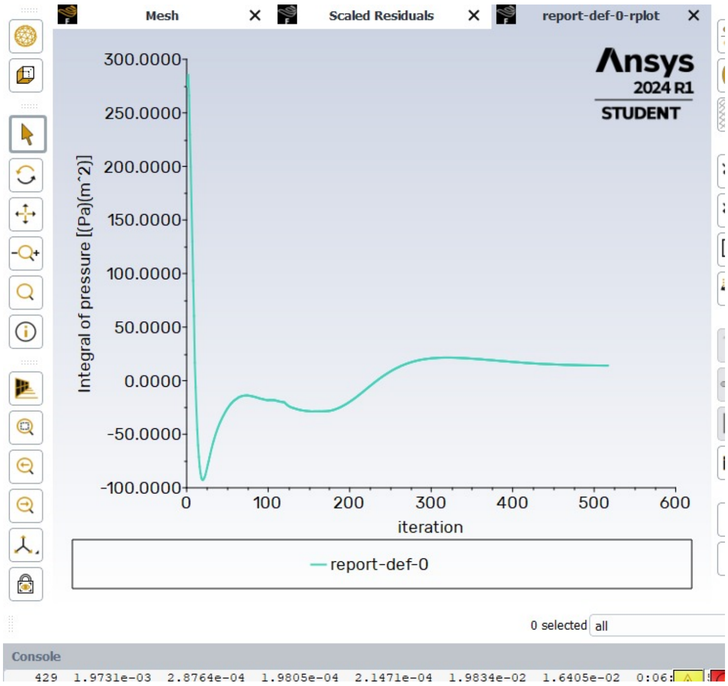

Figure 24: Graph of Static Pressure

4.6 Solution Initialization

The solution initialization was computed from the inlet with the following settings: Gauge Pressure (0), X Velocity (0 m/s), Y Velocity (0 m/s), Z Velocity (9 m/s), Turbulent Kinetic Energy (0.30375), and Specific Dissipation Rate (2079.433).

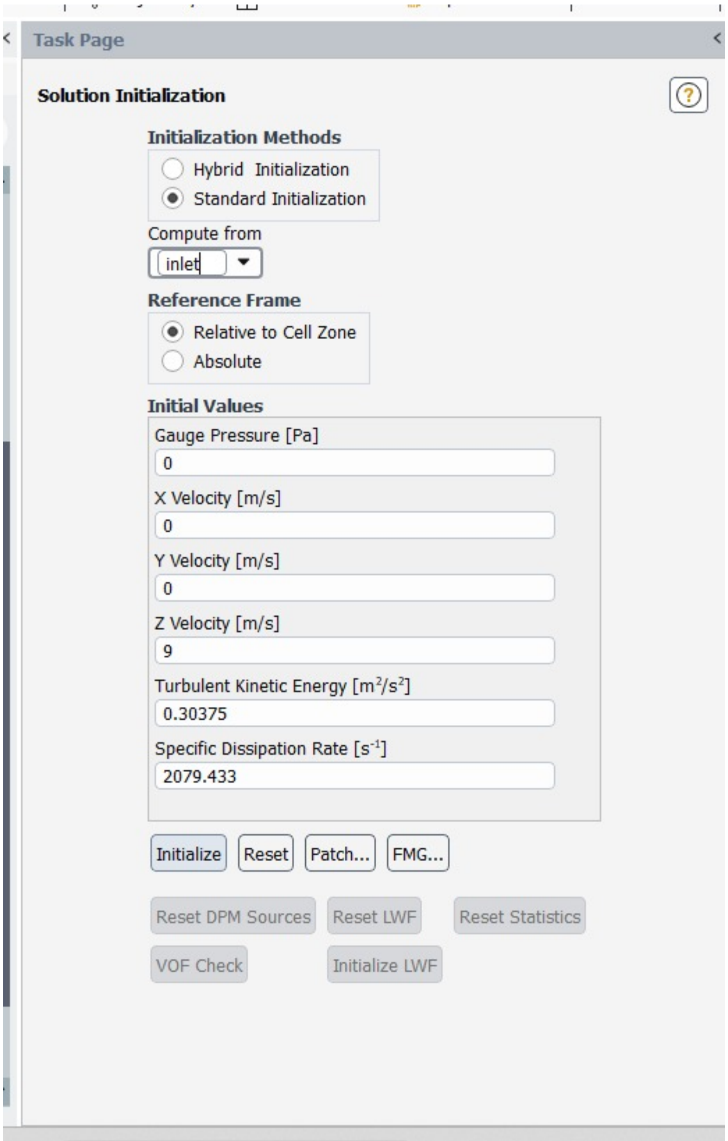

Figure 25: Solution initialization Data

5 Results.

5.1 Contours Results

5.1.1 Pressure Contours

1- Pressure on the blade contours (outlet)

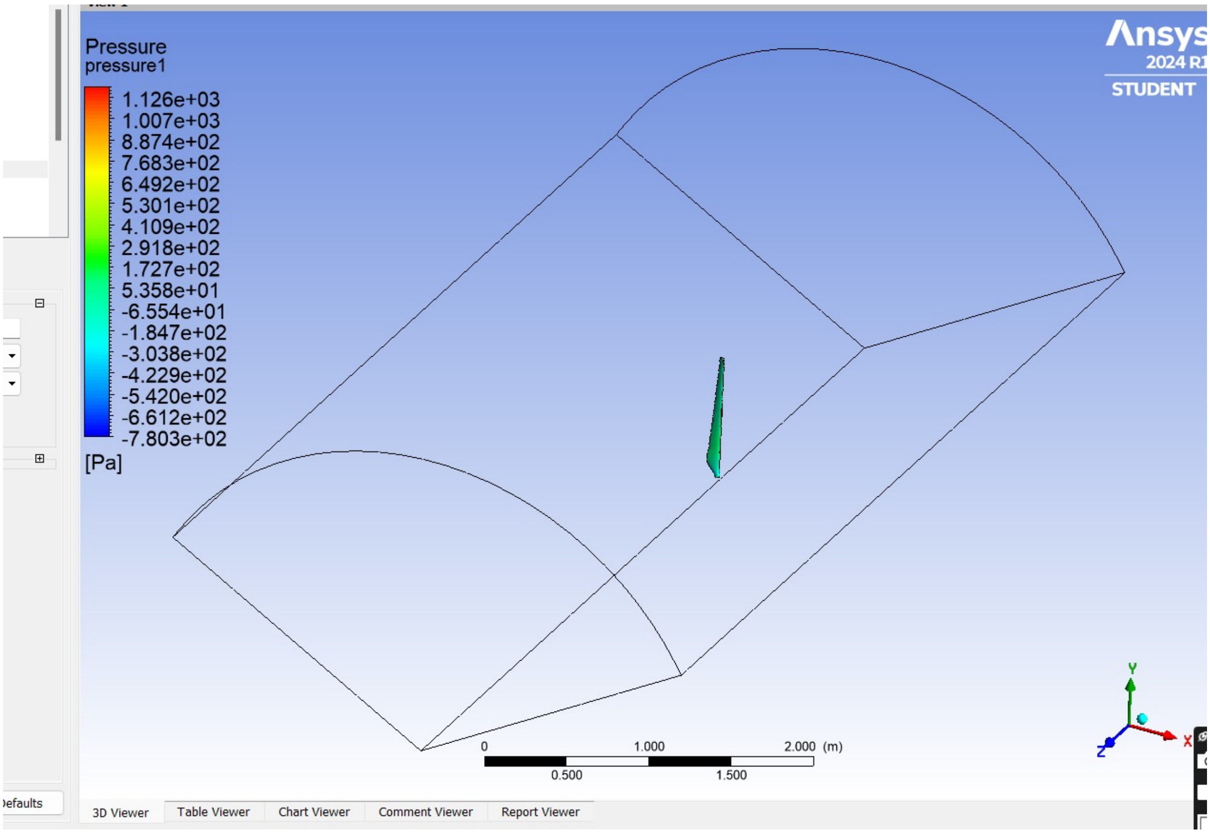

Figure 26: Outlet Pressure

2- Pressure on the blade contours (inlet)

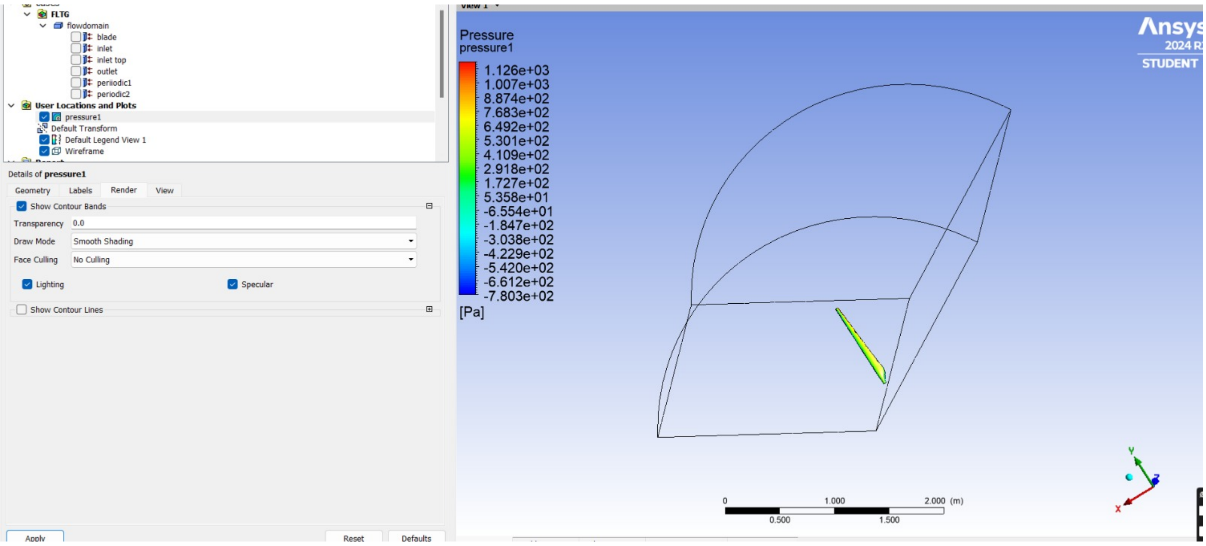

Figure 27: Inlet Pressure

3- Pressure on the blade contours (XZ-Plane)

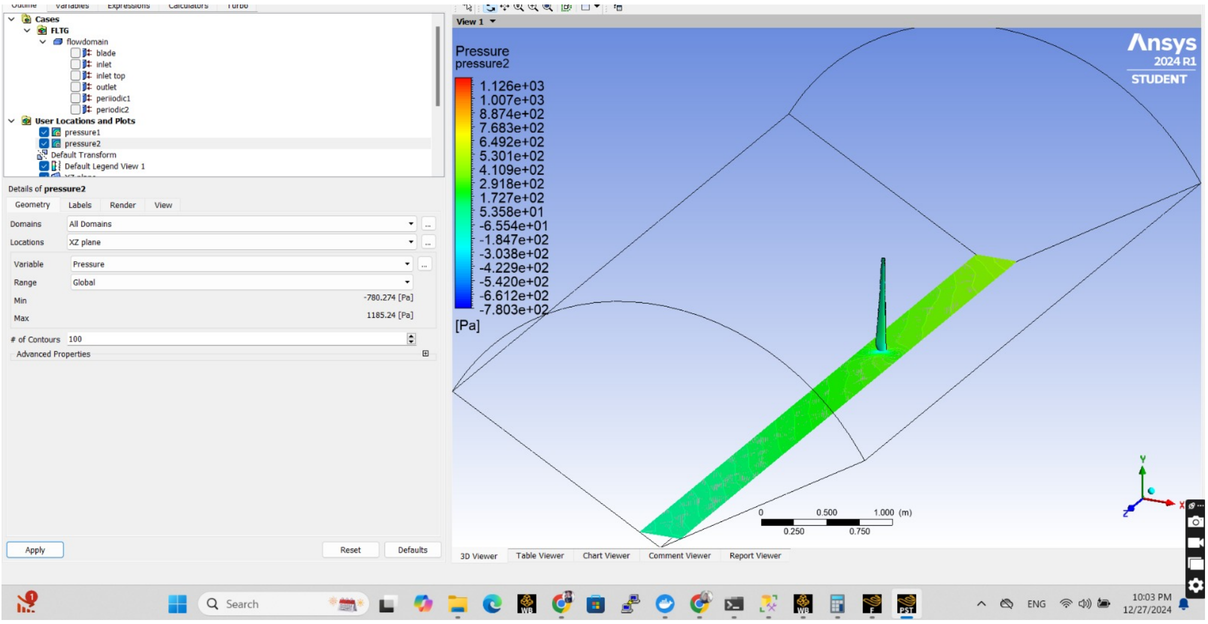

Figure 28: Pressure on the blade contours (XZ-Plane)

4- Pressure blade on top

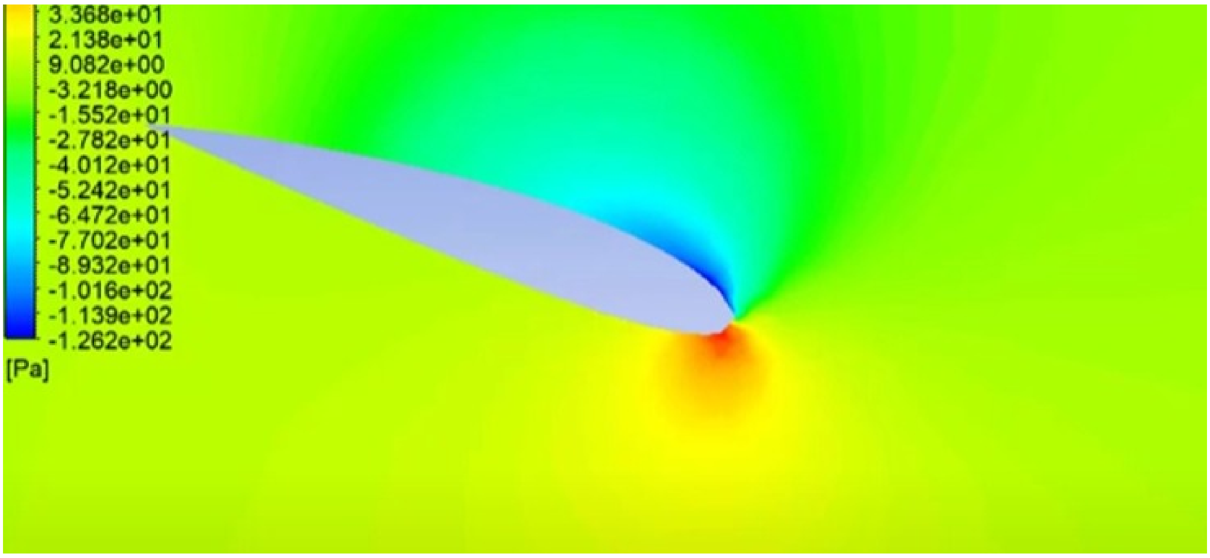

Figure 29: Pressure on the blade contours (Top)

5.1.2 Velocity Contours

1- Velocity Contour on the blade

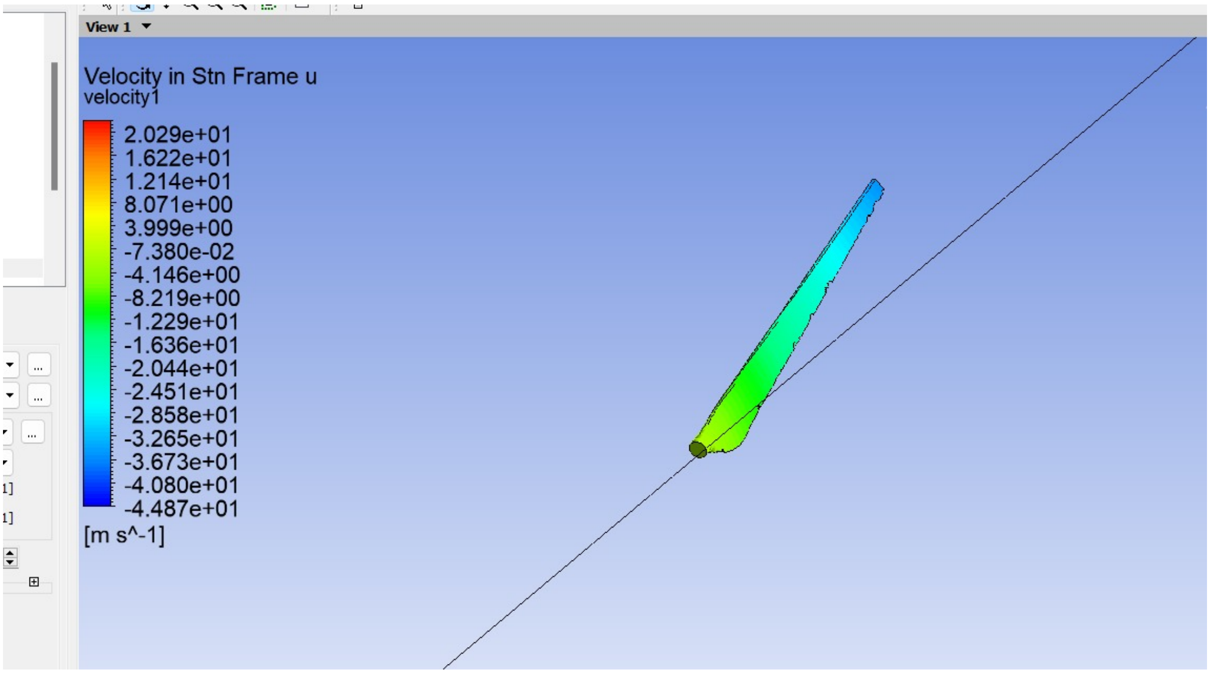

Figure 30: Velocity Contour

2- Velocity Streamlines

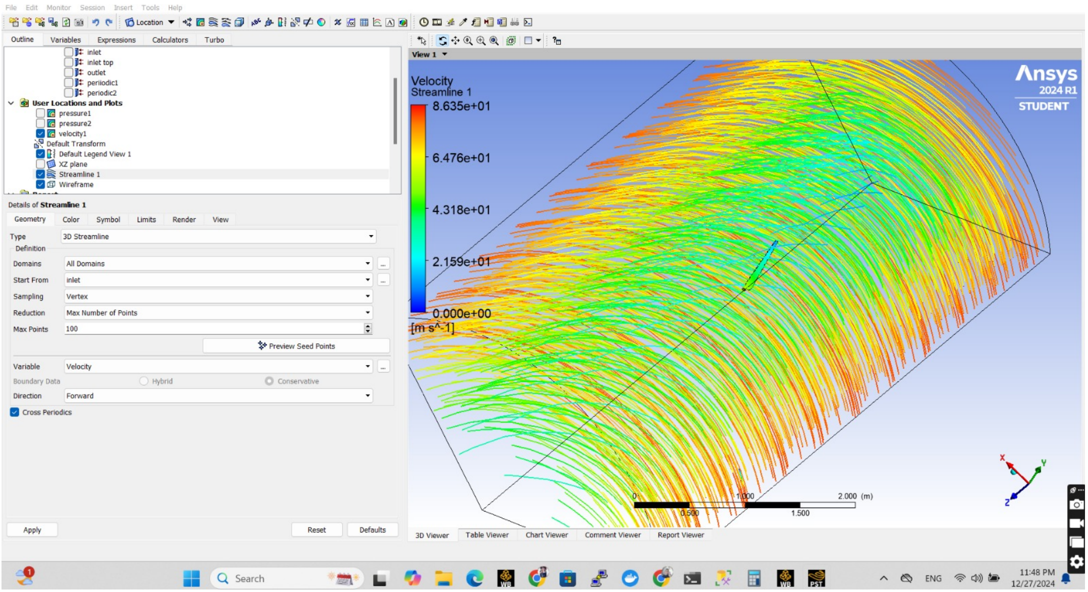

Figure 31: Velocity Streamlines

3- Velocity vector

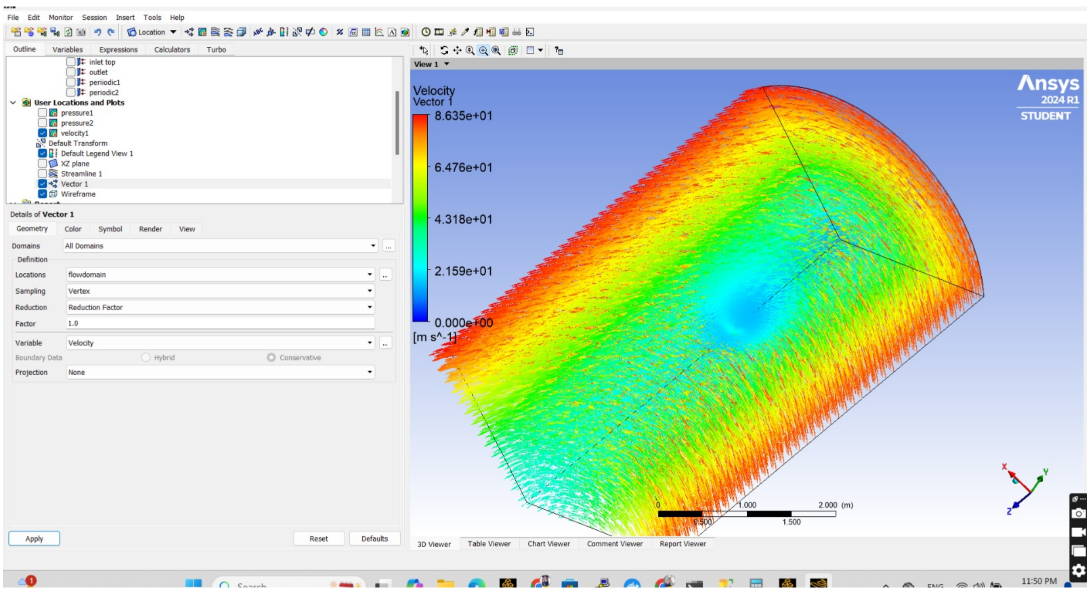

Figure 32: Velocity vector

5.2 Torque generated by the blade.

Torque on the blade = -2.29432 [N m]

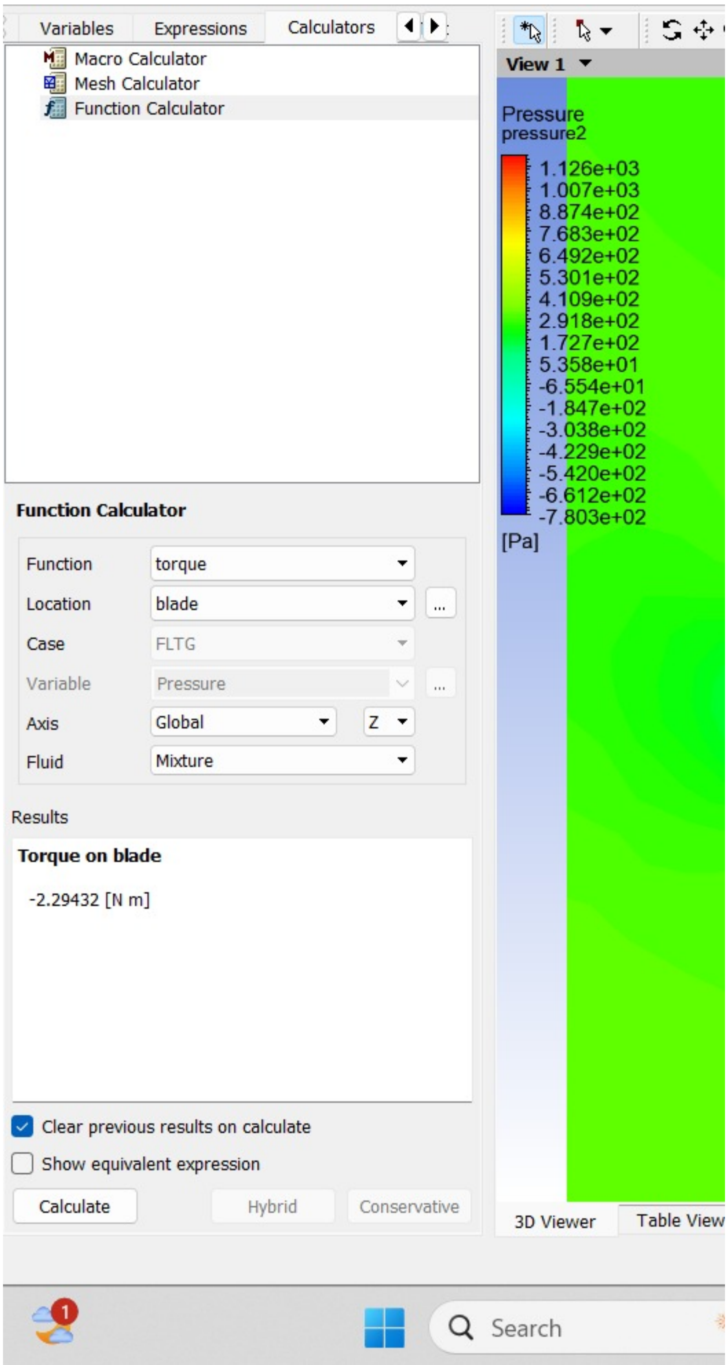

Figure 33: Velocity vector

5.3 The Total power output of the turbine.

Use the equation:

$$P = T \cdot \omega$$

where:

- P is the power (Watts),
- T is the torque (Nm),
- ω is the angular velocity (rad/s), calculated as:

$$\omega = \frac{2\pi \cdot \text{RPM}}{60}$$

The formula to calculate total power is given by:

$$\text{Total Power} = \text{Torque} \times \text{Velocity (rad/sec)} \times 3 \text{ Blades} = 287.7W$$

5.4 Power Coefficient of the blade.

The power coefficient is a dimensionless number that measures the efficiency of the turbine in extracting energy from the fluid.

$$C_p = \frac{P}{\frac{1}{2}\rho A V^3}$$

where:

- P is the power extracted by the turbine (Watts),
- ρ is the air density (kg/m^3),
- A is the swept area of the turbine blades (m^2),
- V is the velocity of the fluid (m/s).

1- Cp of total blade:

$$C_p = \frac{TotalPower}{0.5 \cdot \pi \cdot \rho \cdot r^2} = \frac{287.7}{0.5 \cdot \pi \cdot 1.225 \cdot 25^2} = 0.2392235$$

2- Cp of Each blade:

$$C_p = \frac{TotalPower}{3 * 0.5 \cdot \pi \cdot \rho \cdot r^2} = \frac{287.7}{3 * 0.5 \cdot \pi \cdot 1.225 \cdot 25^2} = 0.0797411$$

5.5 Stresses and deformation on the blade

In the presentation

5.6 The blade's natural frequencies and mode shapes.

In the presentation

6 Identify any potential resonance risks.

Different strategic analyses that are done by ANSYS in the identification of possible resonance risks in a HAWT blade, not limited to but ensuring that the blade operates safely and efficiently, include the following:

(a) **Modal Analysis:** It is one of those important analyses that help a blade understand its natural frequency and mode shape. It helps verify what the inherent vibrational characteristics will be when the blade is not under the application of any external forces.

(b) **Operational Conditions Analysis:** Natural frequencies obtained through the modal analysis are compared with the operational frequencies of the turbine for any overlap that could result in resonance.

(c) **Environmental Conditions:** Wind fluctuations in speed will cause variation in the operational speeds of the turbines to reach within the blade's natural frequencies, which can lead to resonance.

(d) **Damping Assessment:** It is very important to determine the damping properties of the material of the blade, since the less the damping, the greater the resonance problem.

(e) **FEA Integration:** FEA models are used to predict the behavior of the blade under realistic load conditions, coupling the aerodynamic loads coming from CFD analysis with other forces like gravity and inertia.

(f) **Identification of Risk Zones:** It identifies those zones where the operational frequencies come closer to the natural frequency of that zone. These must be carefully monitored and require early planning of a prevention strategy.

(g) **Strategies for Mitigation:** Some of the mitigation strategies include blade design modifications, adjustment of operational parameters, and installation of dampers, etc.

The outcome will be to ensure that the design of the turbine blade is optimized in performance and the resonance risk is mitigated to improve further the reliability and safety of turbine operation.

7 Reflect on your findings in relation to the attached or other relevant literature.

Aspect	**Your CAD Project Findings**	**Relevant Literature**
Fluid Structure Interaction (FSI) Analysis	Used FSI to evaluate aerodynamic and structural responses under operational conditions.	Used FSI to assess blade stresses with different environmental conditions. [1]
Mesh Dependency and Quality	the quality and dependency of the mesh to get accurate CFD results.	Focus on enhancing mesh resolution to improve airflow simulation accuracy around blades.[1]
Modal Analysis and Resonance Risks	Identified natural frequencies and mode shapes to avoid resonance risks.	Stressed the importance of avoiding operational frequencies that overlap with natural frequencies to prevent resonance.[2]
Power Output and Efficiency	Calculated power output and efficiency to know blade performance.	Compared efficiency under varying operational conditions to evaluate blade design effectiveness.[1]
Stress Distribution Analysis	Analyzed stress distribution along the blade during operation to ensure design durability.	Focused on ensuring blade materials and design withstand operational stresses, enhancing turbine reliability.[2]
Viscous Modeling	Used the $k-\omega$ SST model.	Used the $k-\omega$ SST model [1]

Table 1: Comparison of CAD Project Findings with Relevant Literature

8 References

i. Zezatti Flores, K., Castro Gómez, L., & Urquiza, G. (2021). *Fluid Structure Interaction Analysis of Wind Turbine Rotor Blades Considering Different Temperatures and Rotation Velocities*. IntechOpen. doi: 10.5772/intechopen.96495

ii. Wang, L., Quant, R., & Kolios, A. (2016). *Fluid structure interaction modelling of horizontal-axis wind turbine blades based on CFD and FEA*. Journal of Wind Engineering and Industrial Aerodynamics, 158, 11–25. doi:10.1016/j.jweia.2016.09.006