iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 13 **Issue:** IV **Month of publication:** April 2025

DOI: https://doi.org/10.22214/ijraset.2025.69924

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue IV Apr 2025- Available at www.ijraset.com

Advanced Simulation and Flow Analysis of a Centrifugal Pump Using MSC scFLOW

P V Mallari[1], E Kumara Gouda[2], Kiran D V[3], Amruth Kumar S[4], Anil Kumar[5], K Prathibha[6]
UG Students, Department of Mechanical Engineering, RYM Engineering College, Ballari, Karnataka, India

Abstract: In MSC scFLOW, the centrifugal pump is modelled to analyze its performance. Centrifugal pumps are widely used to transport fluids by converting rotational kinetic energy into the hydrodynamic energy of the fluid flow. This rotational energy typically comes from a motor or electric engine. The fluid enters the impeller near the centre and is accelerated by the rotating impeller, then radially flows outward into a diffuser or volute chamber (casing), where it exits. Common applications of centrifugal pumps include pumping water, sewage, oil, and petrochemical fluids, with a centrifugal fan often used to create a vacuum in systems like vacuum cleaners.
Keywords: CFD simulations of fluid and air mixer interactions are essential for optimizing flow characteristics, turbulence, and mixing efficiency in various industrial applications

I. INTRODUCTION

MSC Software is recognized as one of the pioneering companies in the software industry, standing out as a global leader in enabling manufacturers to enhance their engineering processes through simulation software and services. As a trusted collaborator, MSC Software assists organizations in improving product quality, reducing design and testing time, and cutting costs associated with product development. The company's technology is widely used by academic institutions, researchers, and students to broaden their understanding and further the capabilities of simulation.

MSC Software's simulation tools are utilized by top manufacturers for a wide range of applications, including linear and nonlinear finite element analysis (FEA), advanced material modelling, acoustics, fluid-structure interaction (FSI), multi-physics, optimization, fatigue and durability analysis, multi-body dynamics, controls, and manufacturing process simulations. The company's products provide accurate and dependable predictions of real-world product behaviour, empowering engineers to design more innovative solutions.

MSC SCFLOW Overview:

Computational Fluid Dynamics (CFD) using MSC scFLOW is a powerful simulation instrument for examining heat transfer and fluid flow in complex systems. This allows engineers to predict and verify fluid dynamic behaviour for both steady-state and transient conditions. Users can model and analyze fluid flow, interactions and energy transfer within the system with MSC scFLOW to achieve accurate results. scFLOW allows users to work with single-phase flows and more complex multiphase interactions, simultaneously providing powerful features such as thermal management, cavitation simulation, and free surface modelling. This makes it an essential tool in a variety of industries, including aerospace and automotive industry. The software solver engine is automatically run to solve fluid dynamics equations, provide reliable analysis results, and at the same time ensure the best performance results.

Centrifugal Pump Operation:

Most centrifugal pumps are not self-priming. In other words, the pump casing must be pre-filled with fluid before the pump is started, or it won't function. If the casing becomes filled with vapors or gases, the pump impeller becomes gas-bound and is unable to pump [1]. SolidWorks mechanical design automation software is a feature-based, parametric solid modeling design tool that takes advantage of an easy-to-learn Windows graphical user interface [2]. It allows for the creation of fully associative 3D solid models, with or without the use of automatic or user-defined relations to capture design intent. Parameters refer to constraints whose values determine the shape or geometry of the model or assembly [3]. These parameters can be numeric, such as line lengths or circle diameters, or geometric, such as tangency, parallelism, concentricity, or alignment. Numeric parameters can be linked to each other using relations, which enable them to capture design intent. Design intent is how the designer expects the part to respond to changes and updates [4]. Several factors contribute to how we capture design intent, such as automatic relations, equations, added relations, and dimensioning.

II. LITERATURE REVIEW

1) Centrifugal pumps are widely used in industries ranging from water treatment to oil and gas for transporting fluids by converting mechanical energy into fluid motion [5].
2) Their efficiency and performance heavily depend on design parameters like impeller geometry, flow rate, and rotational speed. Computational Fluid Dynamics (CFD) simulations have become essential for predicting pump behavior under different operating conditions without the need for costly prototypes [6].
3) Several studies have highlighted the importance of accurate meshing, turbulence modeling, and boundary condition settings in achieving reliable CFD results [7].
4) Moving mesh methods, such as those used in MSC scFLOW, enable precise modeling of rotating impellers and fluid-structure interactions, critical for understanding real-world pump behavior [8].
5) Researchers have shown that the turbulence model selection (e.g., k-ε, k-ω SST) significantly affects the prediction of flow separation and cavitation within centrifugal pumps [9].
6) MSC scFLOW provides capabilities such as free surface modeling, multiphase flow analysis, and automatic handling of rotating regions, making it a powerful tool for pump simulation [10].
7) Detailed studies using scFLOW have demonstrated the advantages of transient analysis in capturing unsteady phenomena like vortex shedding and pressure pulsations [11].
8) Tutorials and case studies available online, including CFD simulations using ANSYS Fluent and MSC scFLOW, further demonstrate practical workflows for modeling centrifugal pumps [12][13].

III.PROBLEM DEFINITION & CALCULATION

A centrifugal pump has a water inlet velocity of 2 m/s, and the impeller is rotating at an angular velocity of 1000 rpm. Conduct a CFD simulation to visualize the velocity distribution on the impeller blades and the flow of water from the inlet. The model is first created in two dimensions using various AutoCAD commands and the centrifugal pump's isometric view, as seen in figure 1

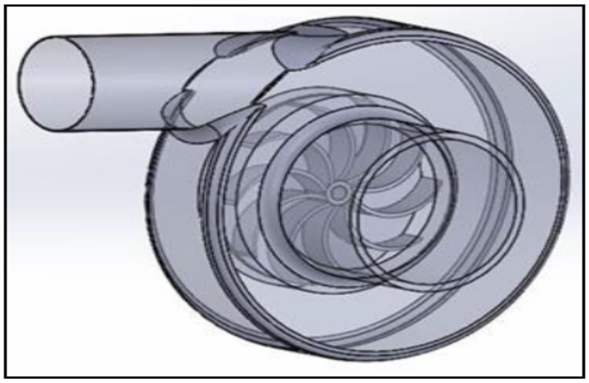


Figure 1 Relative Velocity Turbulence Model

Modeling With Calculation

Theoretical calculations were performed before conducting the CFD simulation to validate the initial design of the centrifugal pump.

Flow Rate (Q)

Given In Below

Inlet velocity, V=2 m/s

Inlet pipe diameter, D=0.1 m (assumed typical small pump)

Cross-sectional area of the centrifugal pump

$$A = \frac{\pi D^2}{4} = \frac{3.1416 \times (0.1)^2}{4} = 0.00785\ m^2$$

Flow rate of centrifugal pump

$$Q = A \times V = 0.00785 \times 2 = 0.0157\ m3/s$$

or

$$Q = 15.7\ L/s$$

Pump Head (H)
Assuming standard head generated by centrifugal pumps is around 15 meters for small impeller designs rotating at 1000 RPM.

$$H \approx 15 \text{ m}$$

Theoretical Pump Power (P)
The hydraulic power needed to lift a water.

$$P = \rho g Q H P$$

Where:
ρ=1000 kg/m3 (density of water),
g=9.81 m/s2 (gravitational acceleration),
Q=0.0157 m3/s
H=15 m.

$$P = 1000 \times 9.81 \times 0.0157 \times 15$$
$$P = 2311.6 \text{ W} = 2.31 \text{kw}$$

Impeller Tip Speed (U)
Tip speed formula give in data hand book (DHB) [1]

$$U = \frac{\pi D N}{60}$$

where:
D=0.15 m (assumed impeller diameter in (DHB)),
N=1000 rpm

$$U = \frac{3.1416 \times 0.15 \times 1000}{6} = 7.85\ m/s$$

Specific Speed (Ns)
Specific speed helps in pump design classification:

$$Ns = \frac{N\sqrt{Q}}{H^{3/4}}$$

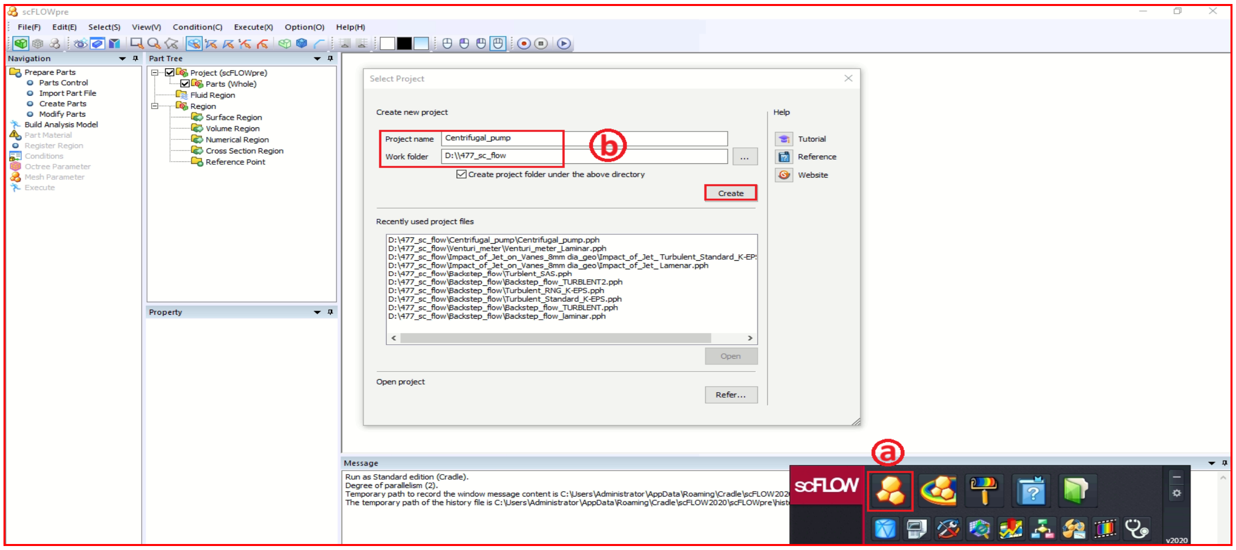
$$Ns = \frac{1000 \times \sqrt{0.0157153}}{15^{3/4}}$$

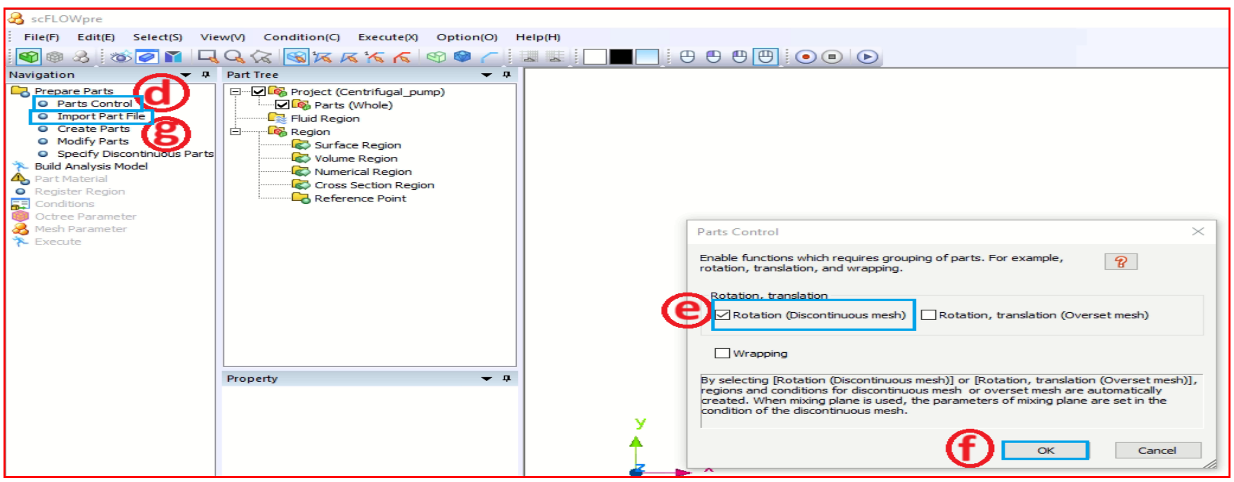
$$Ns = \frac{1000 \times 0.125}{7.62} \approx 16.4$$

The low specific speed (~16.4) indicates a radial flow type centrifugal pump, which matches typical design expectations.

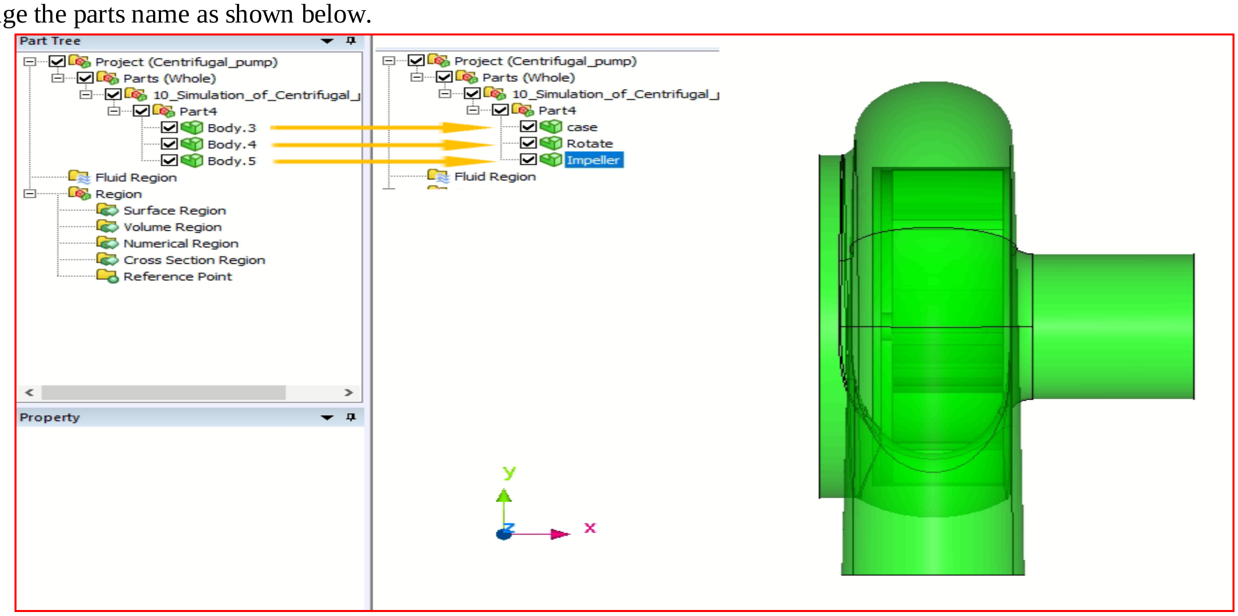
IV. SETUP METHODOLOGY AND SIMULATION

1) *Step 1: Open scFLOW Software. In the Kicker, launch Pre-processor*
- Click launch Pre-processor
- Project Name: Centrifugal Pump
- Click Create
- Click on Parts Control
- Select Rotation
- Click Ok.

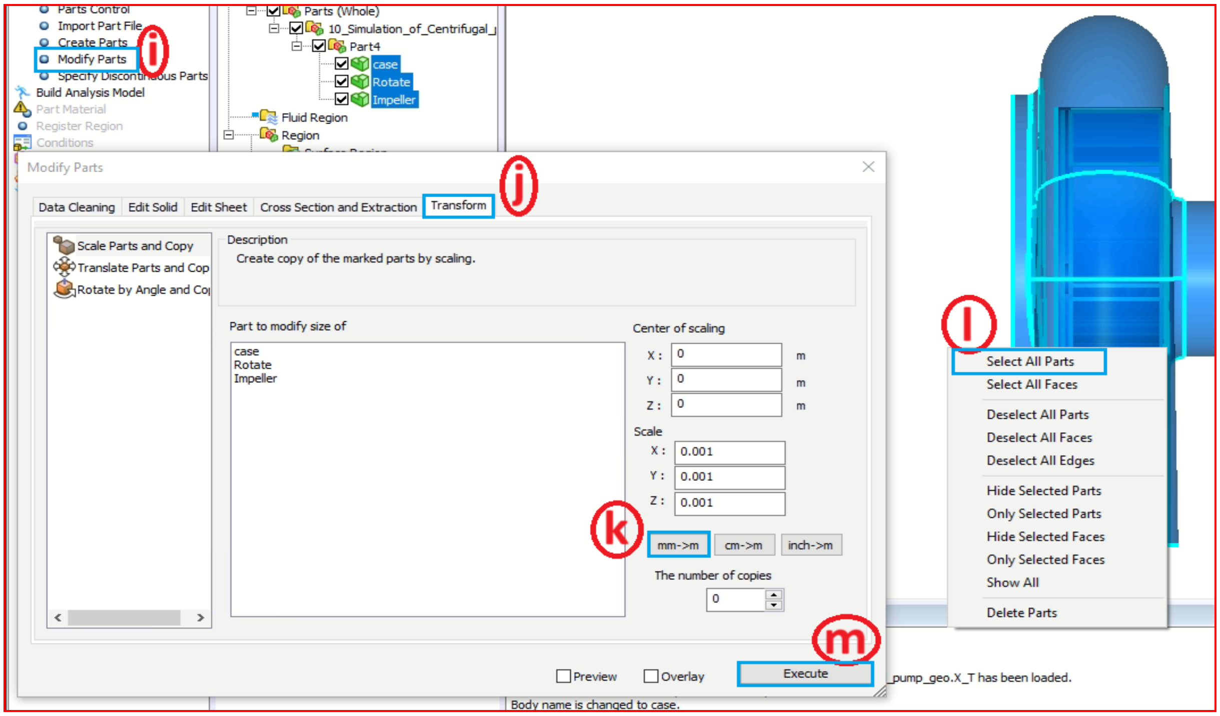




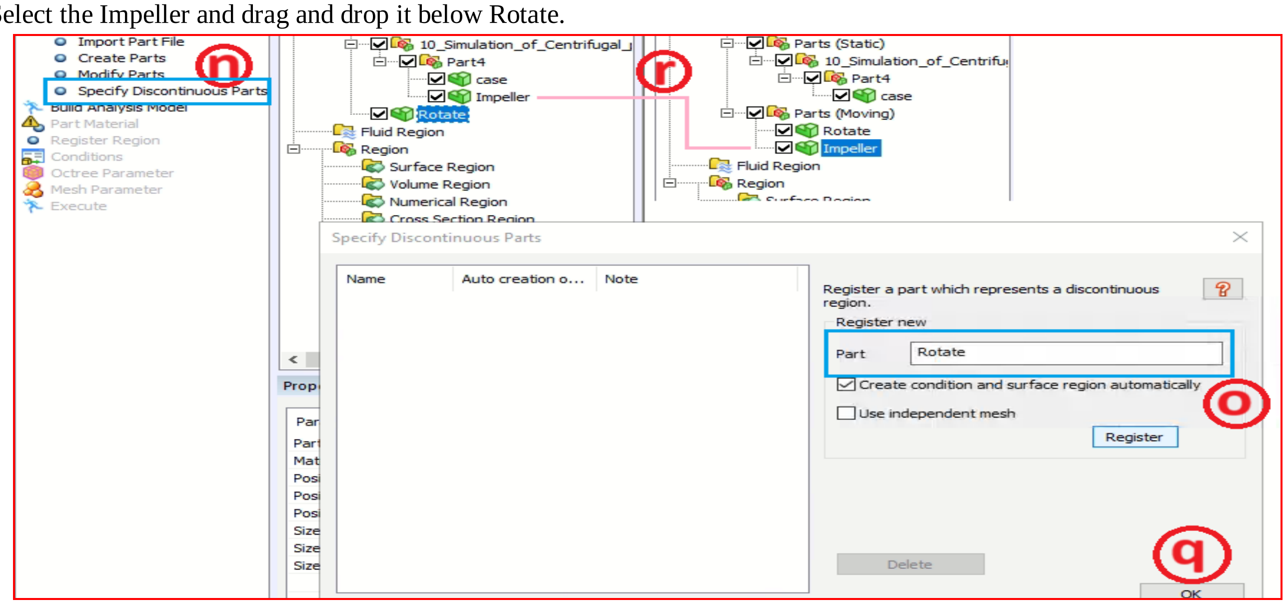
- Click import part file (browse the directory to import the CAD).
- Change the parts name as shown below.



- Click Modify Parts in Navigation Panel
- Click on Transforms
- Click on mm>m
- Right click in Draw window and select all the Parts. Make sure both the parts are displayed in Selection tab.
- Click on Execute



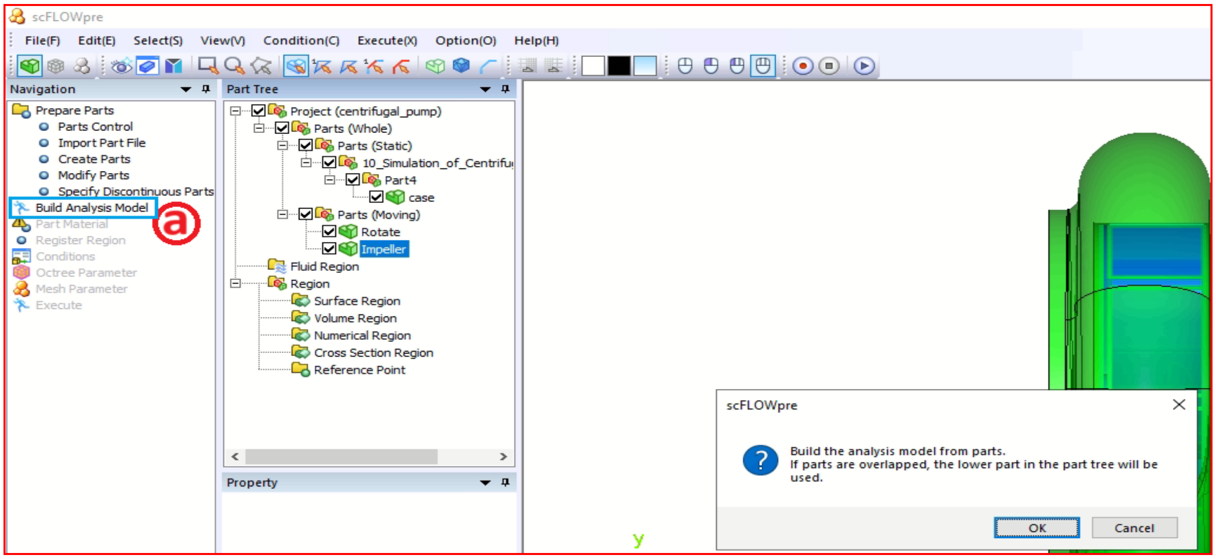
- Click on Specify Discontinues Parts.
- Select Rotate from part tree. Ensure Rotate is displayed in the form and click Register.
- Click Ok in new message window.
- Click Ok.
- Select the Impeller and drag and drop it below Rotate.



2) *Step 2: Click on Build Analysis Model*

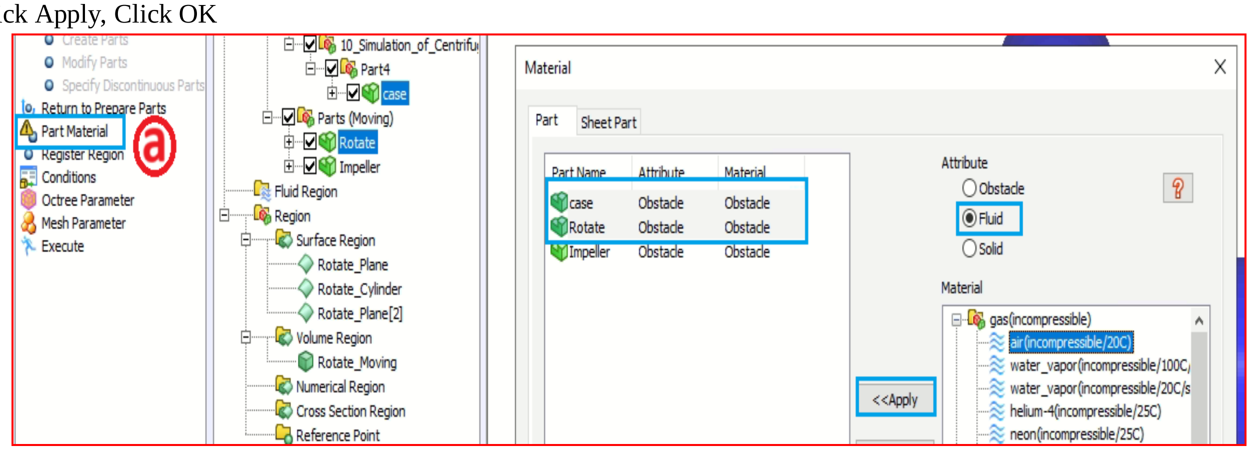
- Click on Build Analysis Model

(Click OK in the pop-up dialogue)



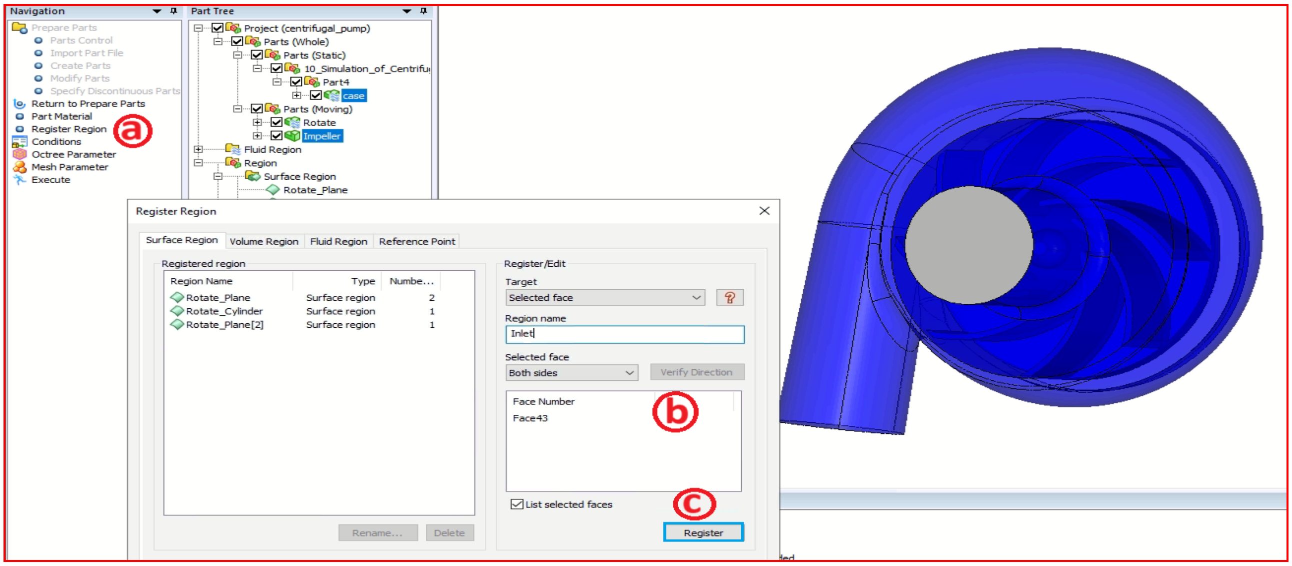
3) *Step 3: Click on Part Material*

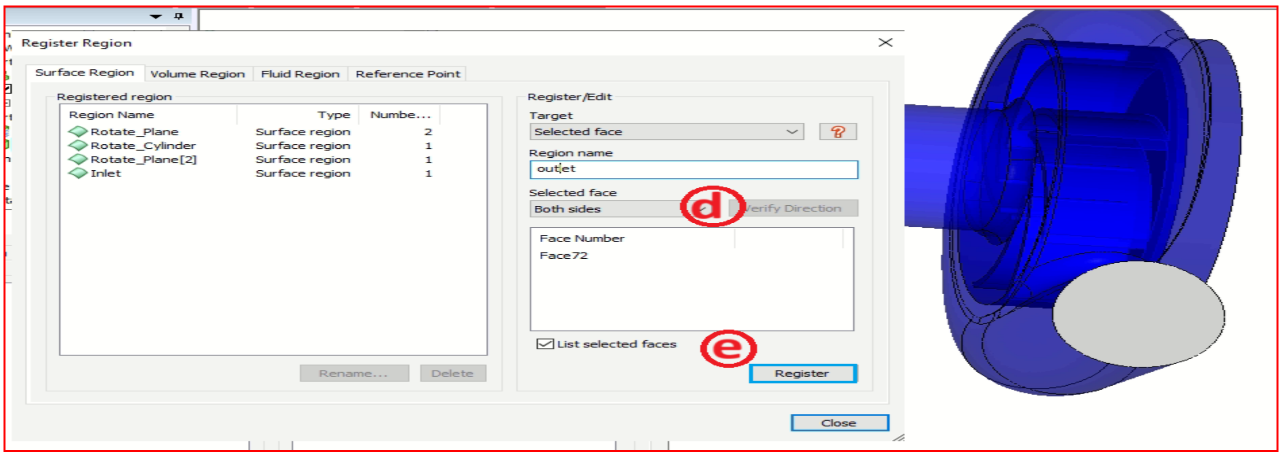
- Click on Part Material
- Select Case and Rotate in Part list
- Select Fluid in attribute and select air.
- Click Apply
- Select Impeller in Part list
- Select Obstacle in Attribute
- Click Apply, Click OK



4) *Step 4: Click Register Region*

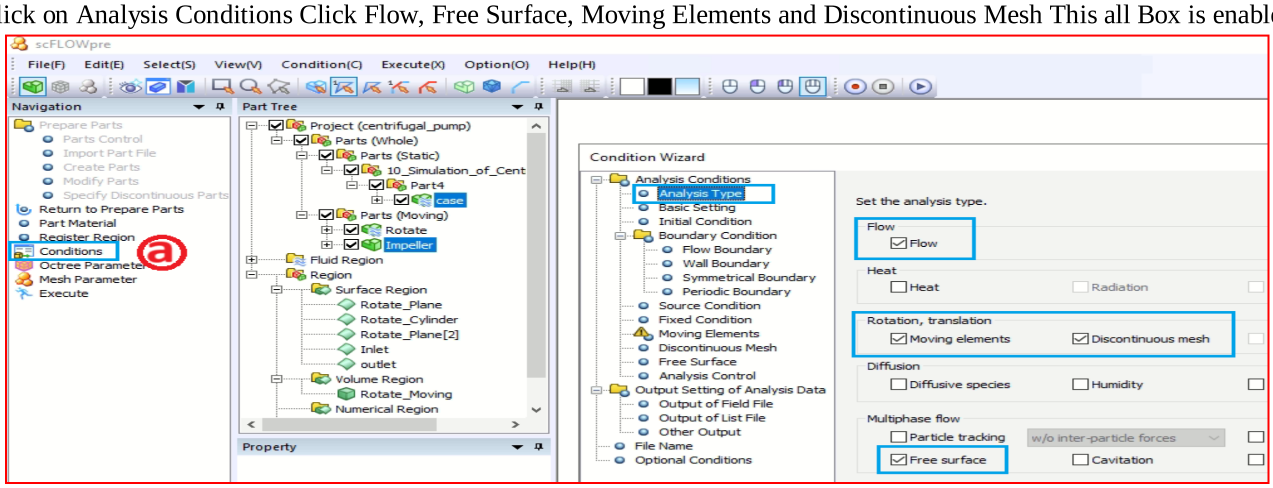
- Click Register Region
- Insert Inlet in Region name Select X Max face
- Click Register
- Insert Out in Region name Select Y Min surface
- Click Register and Click Close



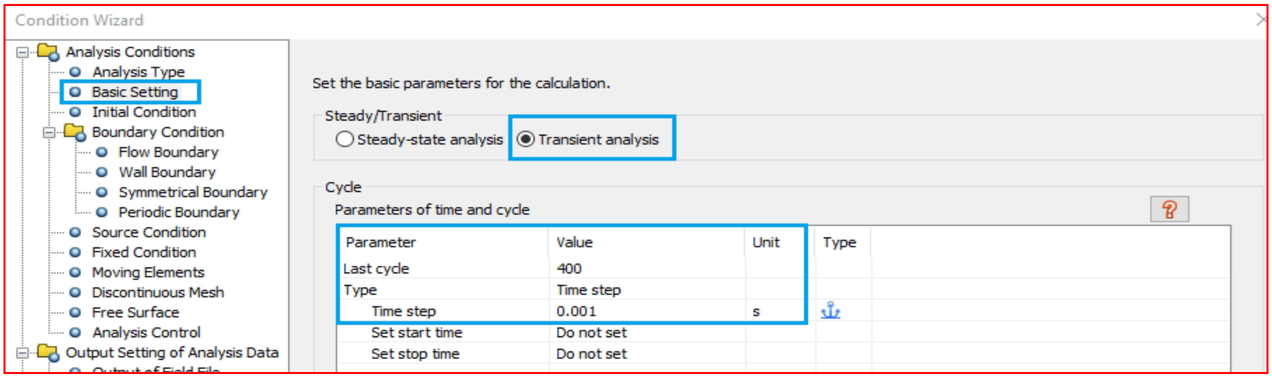


5) *Step 5: Click on Conditions*

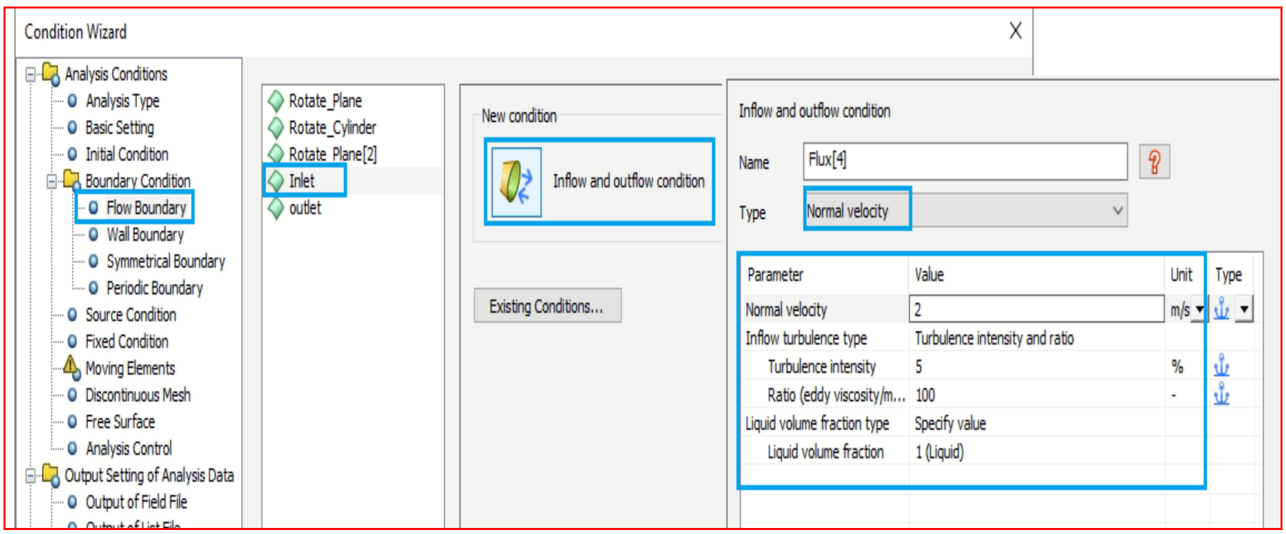
- Click Conditions
- Click on Analysis Conditions Click Flow, Free Surface, Moving Elements and Discontinuous Mesh This all Box is enabled



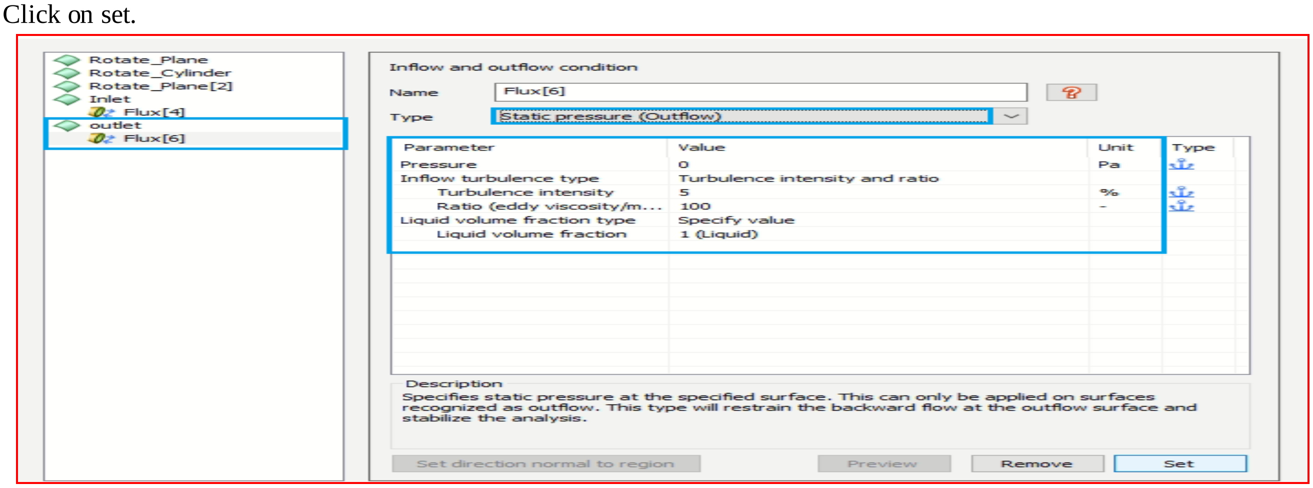
- Click Basic Setting and select Transient Analysis
- Input last cycle as 400.
- Type to Time step and Time step = 0.001 s



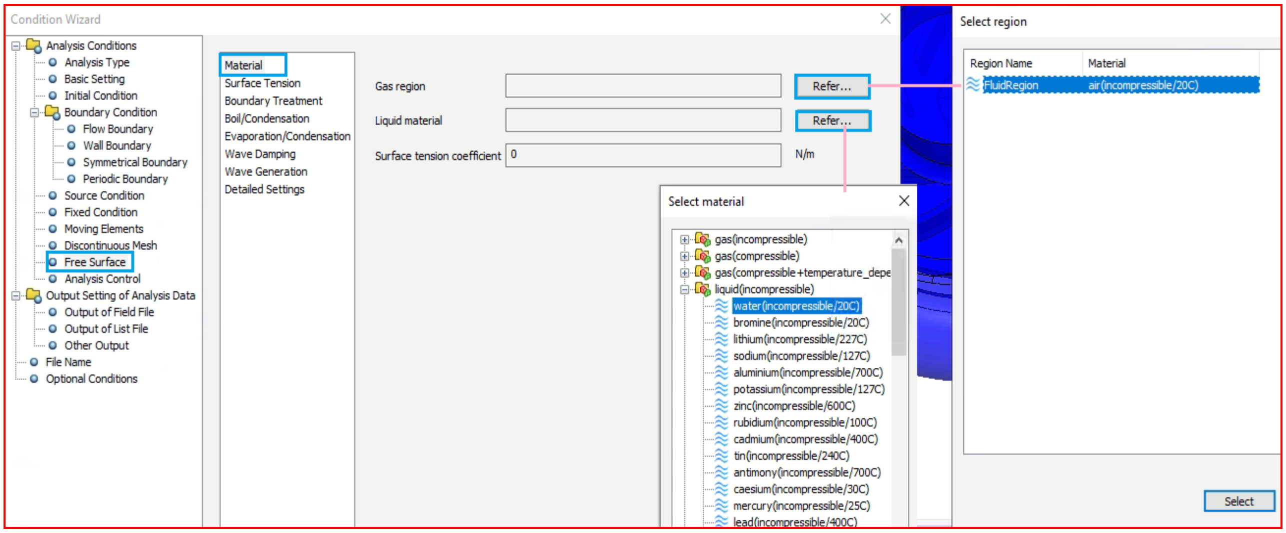
- Click on Flow Boundary condition
- Click on Inlet and click Inflow and Outflow Condition
- Select type as Normal Velocity and value as 2 m/s.
- Liquid Volume fraction to 1(Liquid)
- Click on Set.



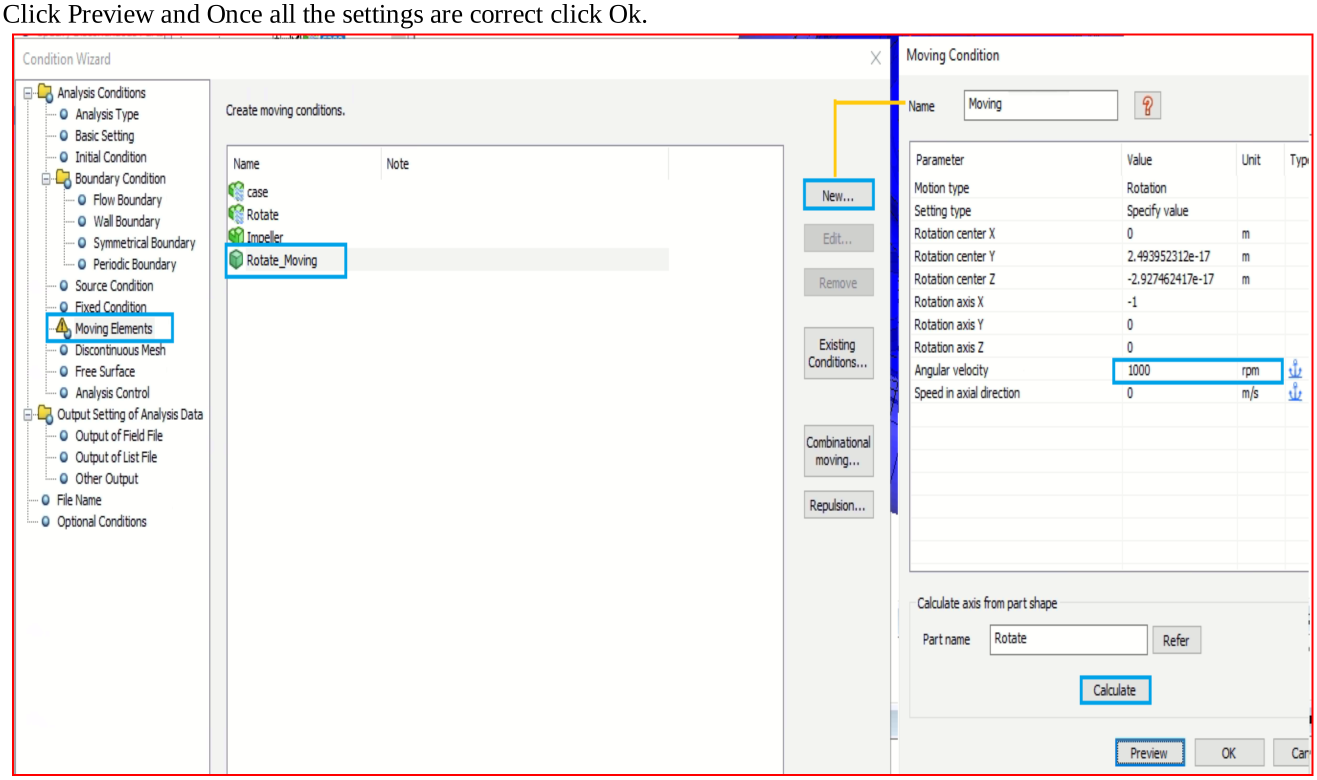
- Click on Outlet and click Inflow and Outflow Condition
- Select type as Static Pressure
- Pressure value as 0.
- Liquid Volume fraction to Natural
- Click on set.



- Click on Free-surface. In material tab for gas region refer and select air, in liquid material refer and select water.

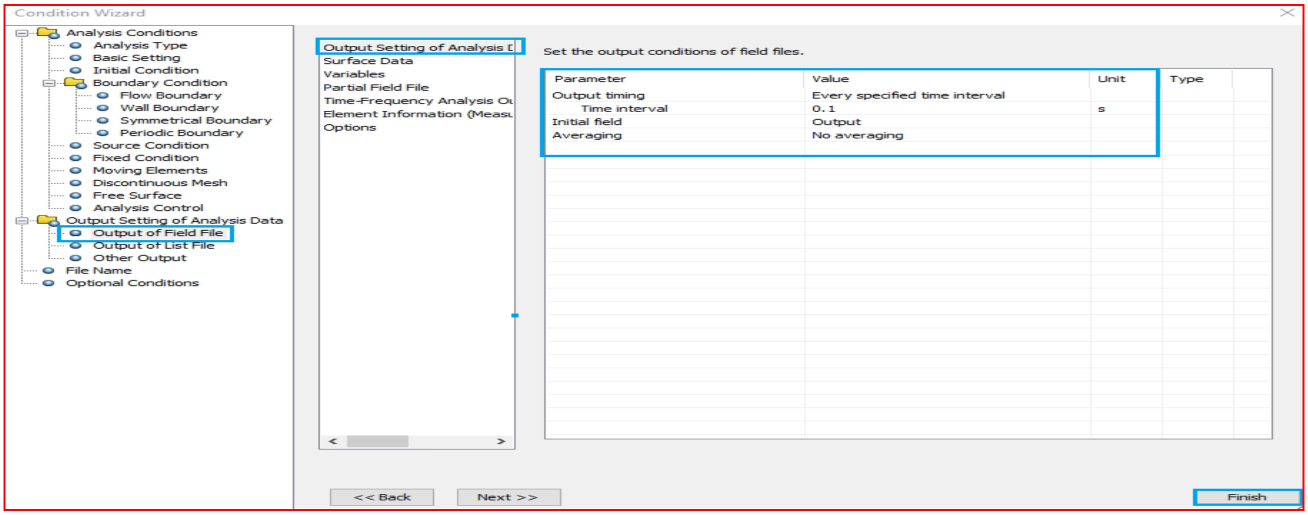


- Select on moving elements
- Select Rotate-Moving.
- Click on New.
- In the motion type select rotation and for Setting type select Specify Value. In part name select Rotate and Click refer and calculate. The rotation axis will automatically get calculated.
- Set Angular velocity to 1000
- Click Preview and Once all the settings are correct click Ok.



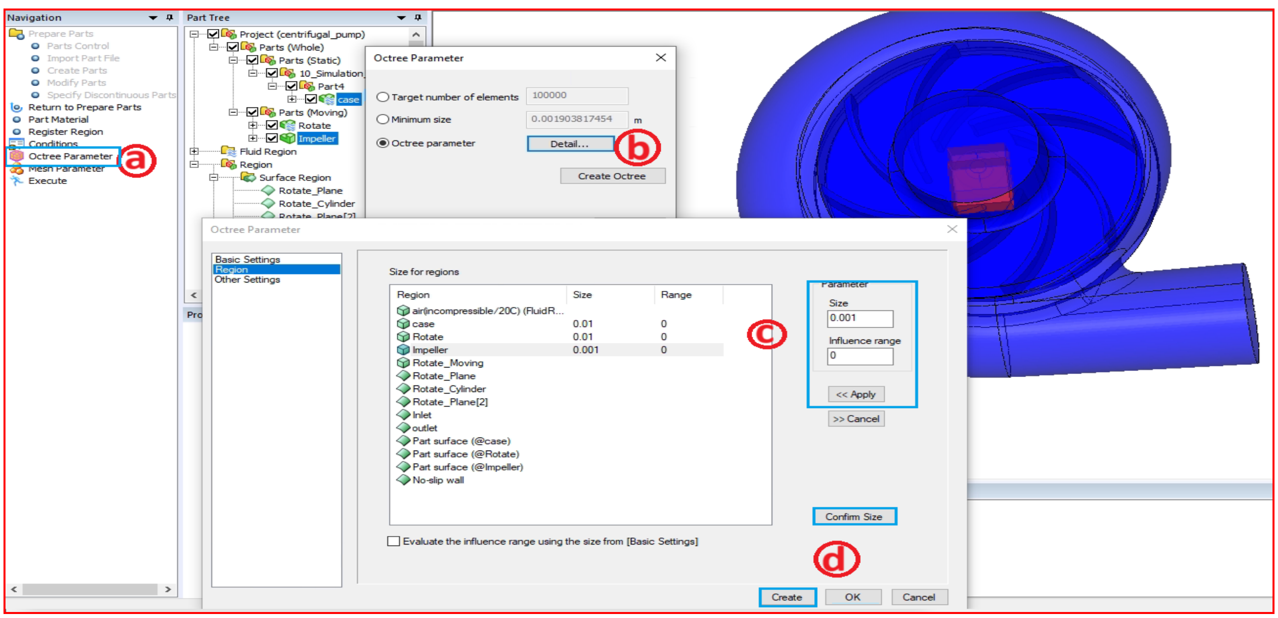
- Click on Output of field file Select Output setting of Analysis data
- In output timing select every specified time interval, time interval = 0.1s and Initial field = Output
- Click Finish

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue IV Apr 2025- Available at www.ijraset.com



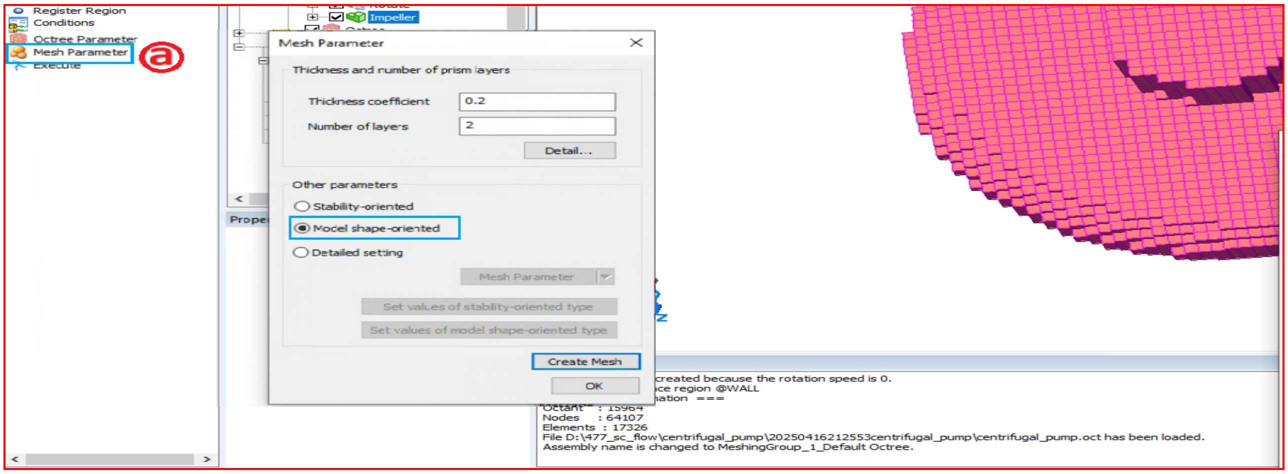
6) *Step 6: Click on Octree Parameter*

- Click on Octree parameter and click Detail
- Select Region Fill Parameter
- Select Enter Case Size 0.01 Rotate Size 0.01 Impeller Size 0.001 Click Apply.
- Click Confirm size and Click Create and Click Ok
- Click Ok



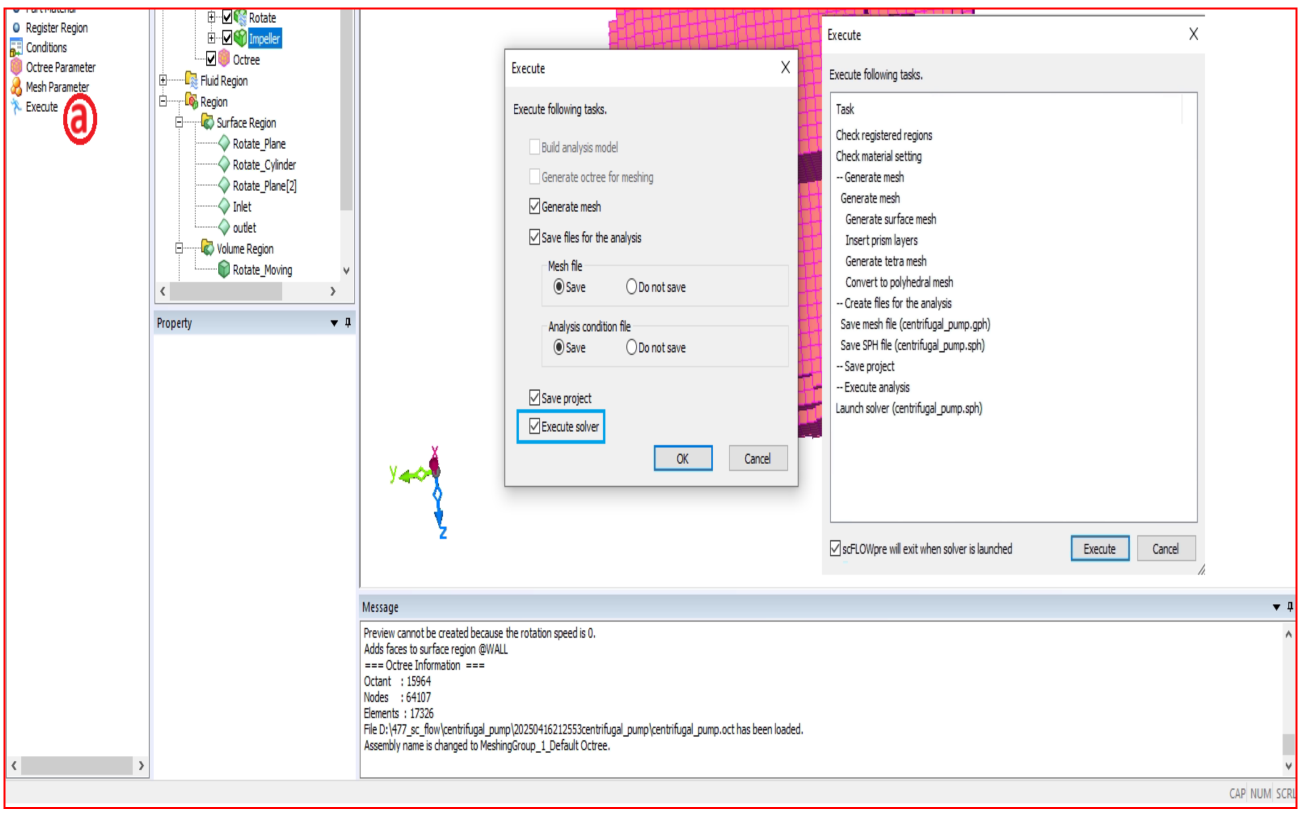
7) *Step 7: Click on Mesh Parameter*

- Click on Mesh Parameter
- Select Model shape-oriented
- Click on Create Mesh and Click Ok



8) Step 8: Click on Execute

- Click on Execute.
- Make sure you enable Execute solver.
- Click Ok.



9) Step 9: Click on SC Monitor

- Click on SC Monitor Execute
- Click import SPH file path.
- Click Execute and waiting for Run
- Click  Post Processor icon.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 13 Issue IV Apr 2025- Available at www.ijraset.com

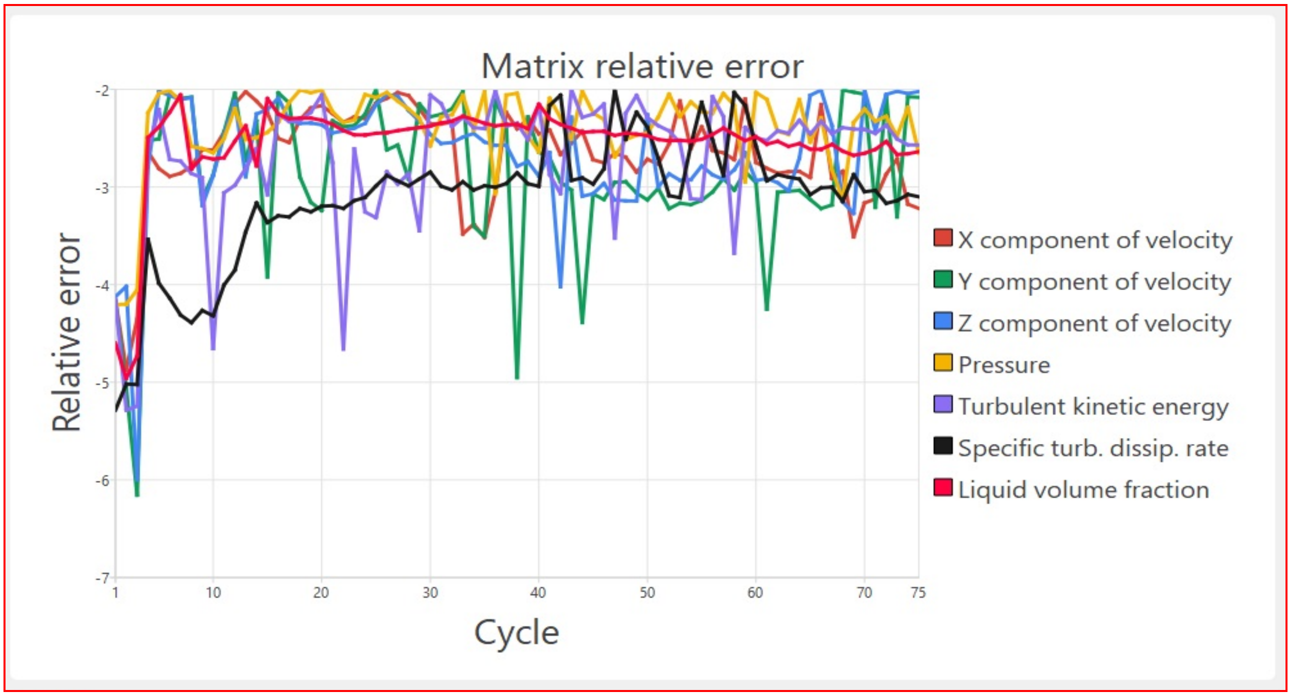


Figure 2 Check the Matrix Relative Error Status of Simulation.

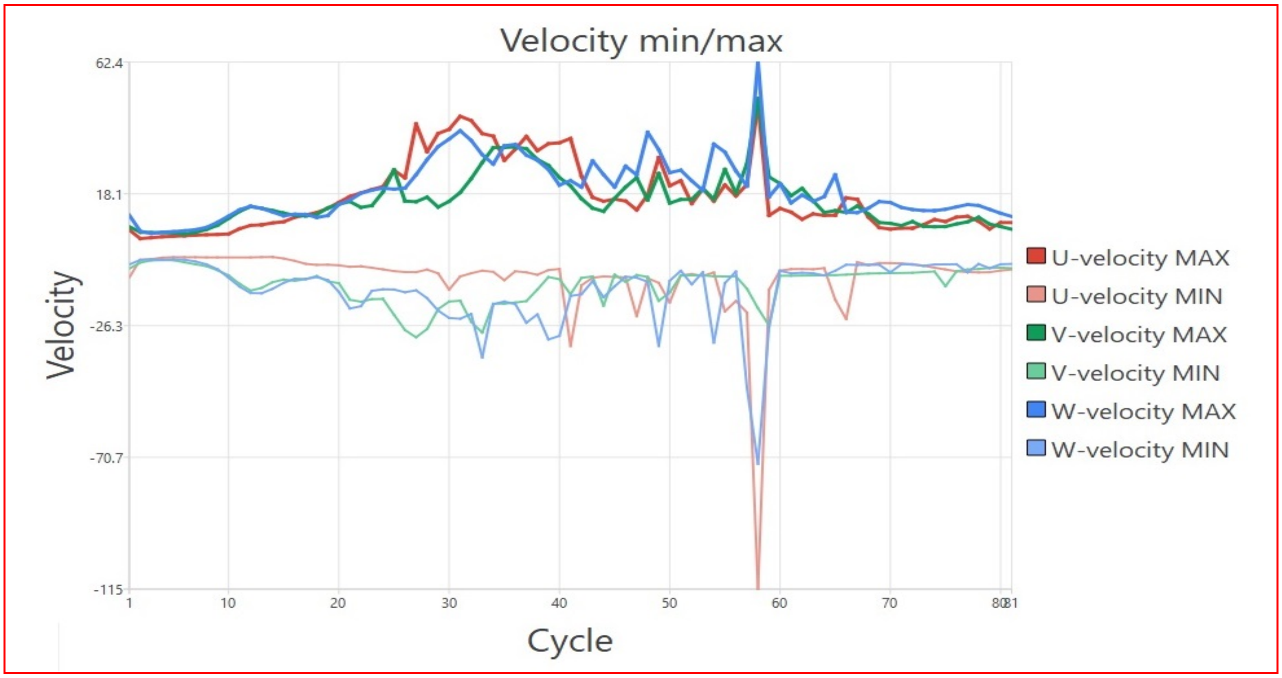


Figure 3 Check the Velocity Status of Simulation.

10) Step 10: Post Processing

- confirm the status of job in [Job schedular] as 'Reached steady state normally'.
- Select to launch post processor.
- Select [Plane (1)] from control window. In [Contour] select Magnitude of velocity for Variable and click Redraw icon
- Contour of Liquid volume

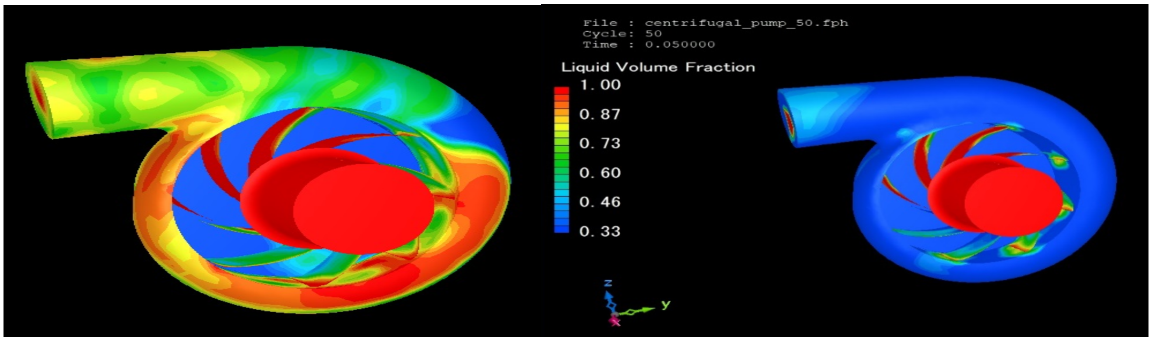
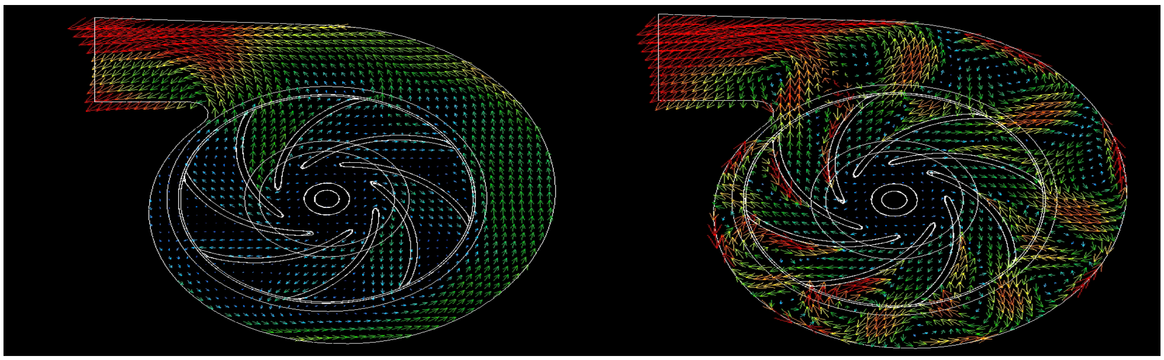


Figure 4 Contour of Liquide Fraction

- Contour of Velocity Vector



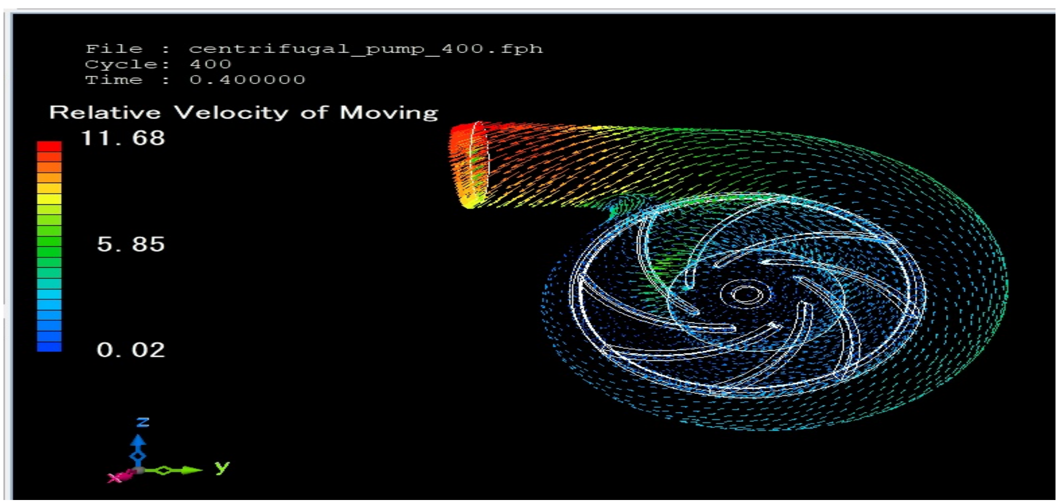


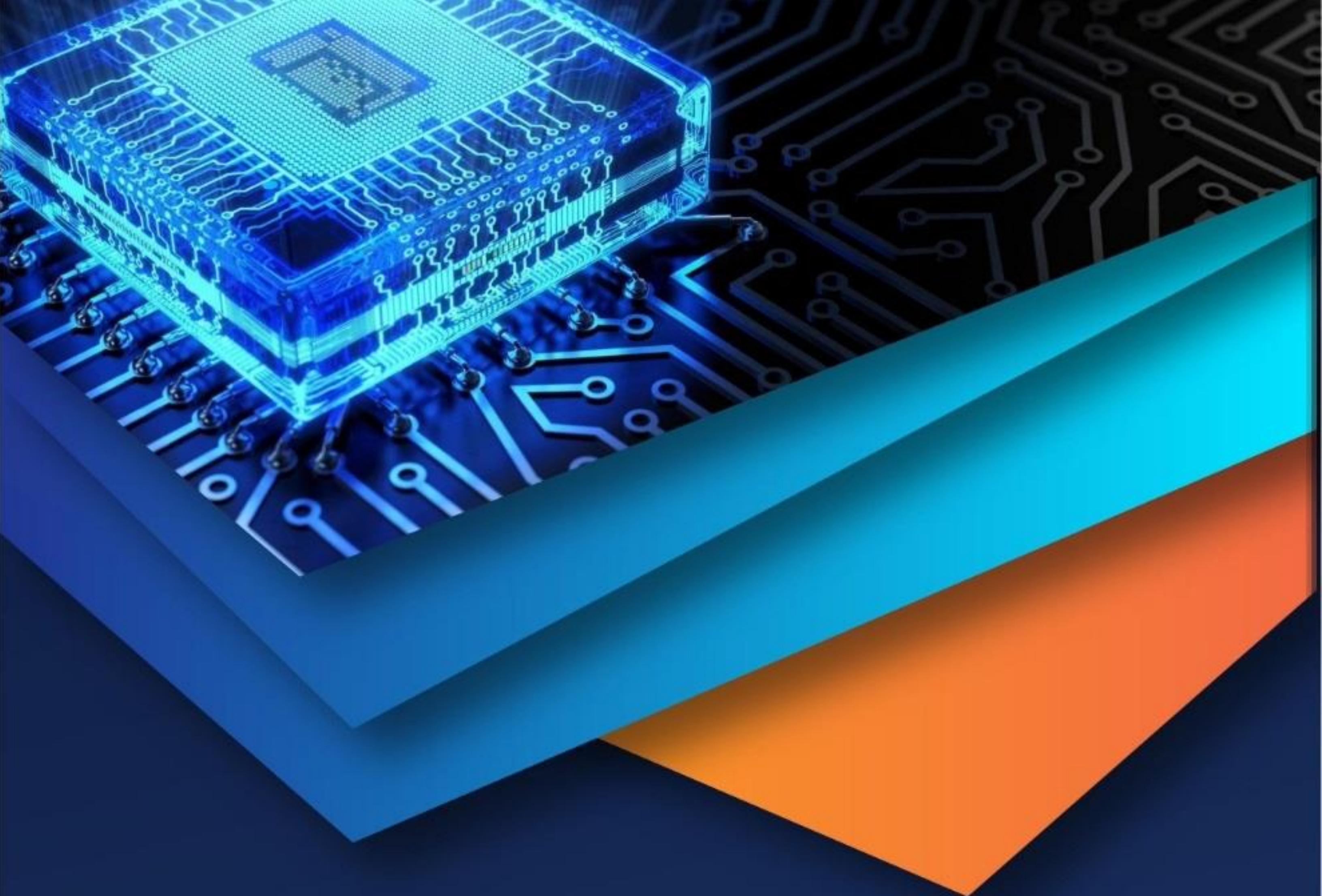
Figure 5 Relative Velocity Turbulence Model

V. CONCLUSION

This report modeled and analyzed the centrifugal pump with MSC scFLOW for its internal fluid dynamics. The computational fluid dynamics (CFD) workflow illustrated the velocity distributions, flow patterns, and turbulence in the pump at transient conditions. The workflow consisted of pre-processing, the assignment of materials, setup of boundary conditions, meshing and executing the solver. With regards the simulation results, the overall pump performance characteristics were provided with critical insights into the high velocity and turbulence around the impeller blades. Collectively this use of scFLOW provided a full picture of the operational use of the pump and demonstrated how powerful CFD can be in understanding centrifugal pump agriculture design to enhance reliability and improve efficiency.

REFERENCES

[1] K. Mahadevan and Balaveera Reddy, "Design Data Hand Book".
[2] Finite Element Analysis", Chandra Pautla.
[3] Fatigue Life Analysis of Aluminum Wheels by Simulation of Rotary Fatigue Test Liangmo Wang* - Yufa Chen - Chenzhi Wang - Qingzheng Wang School of Mechanical Engineering, Nanjing University of Science & Technology, China.
[4] Trupti G. Raut and M.Y. Shinde, "A review on optimization of machining parameters in EDM", Int. Journal of Innovative Research in Science, Engg. And Tech., Vol. 4(3), (2015), pp. 893-896.
[5] Karassik, I. J., Messina, J. P., Cooper, P., & Heald, C. C. (2001). Pump Handbook (3rd ed.). McGraw-Hill.
[6] Anagnostopoulos, J. S. (2009). "Numerical study of the flow in a centrifugal pump impeller using CFD tools." Journal of Mechanical Engineering Science, 223(5), 1045–1056.
[7] Kumar, A., & Saini, R. P. (2010). "CFD applications in the design of hydraulic turbines: A review." Renewable and Sustainable Energy Reviews, 14(9), 374–383.
[8] Aoki, T., Inaba, K., & Shibata, T. (2018). "Application of Moving Mesh Techniques in CFD Simulation of Pumps Using MSC scFLOW." Proceedings of ASME Turbo Expo 2018, GT2018-75123.
[9] Baskharone, E. A. (2006). Principles of Turbomachinery in Air-Breathing Engines. Cambridge University Press.
[10] MSC Software Corporation. (2022). MSC SCFLOW User's Guide. Hexagon Manufacturing Intelligence.
[11] Stepanski, B., & Ogrizek, D. (2020). "Advanced CFD analysis of fluid machinery: Current trends and future challenges." International Journal of Fluid Machinery and Systems, 13(1), 1–11.
[12] CAE Associates. (2021). CFD Analysis of a Centrifugal Pump – ANSYS Fluent Tutorial. [YouTube Video]. Available at: https://www.youtube.com/watch?v=W88M6Y2N5jg
[13] MSC Software Official. (2023). MSC SCFLOW Simulation Tutorial: Centrifugal Pump Flow Analysis. [YouTube Video]. Available at: https://www.youtube.com/watch?v=d-sJDcdXFA8



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY