PREDICTION OF FLOW PHYSICS IN TURBINE UNIT OF TURBOCHARGER BY DYNAMIC MESH MOTION

MANIMARAN RENGANATHAN[1]
[1]*Associate Professor, Thermal and Automotive Research Group, School of Mechanical and Building Sciences, Vellore Institute of Technology, Chennai, India 600127*

Keywords: *Turbocharger, Dynamic mesh, snappyHexMesh, rhoPimpleDymFOAM, Grid Independence, Performance.*

A turbocharger is a device composing turbine, compressor linked and connected by a shaft with bearings. The three dimensional drawing of the entire device is shown in Fig. 1. Energy is extracted using a turbine mounted in the device (shown as left) and the compressor is coupled to a turbine by a coupling shaft with bearings. The compressor supplies pressurised air to the engine for increasing the charge density during the suction stroke. An intercooler is usually present after the turbocharger for maintaining proper temperature.

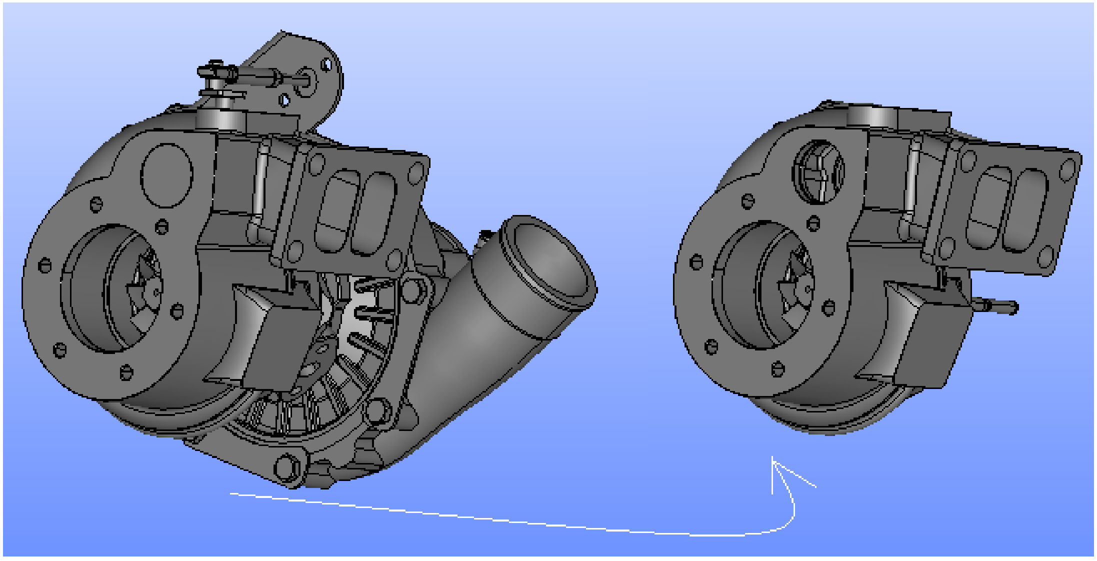


Fig. 1. Turbocharger (left) and turbine part of the turbocharger (right)

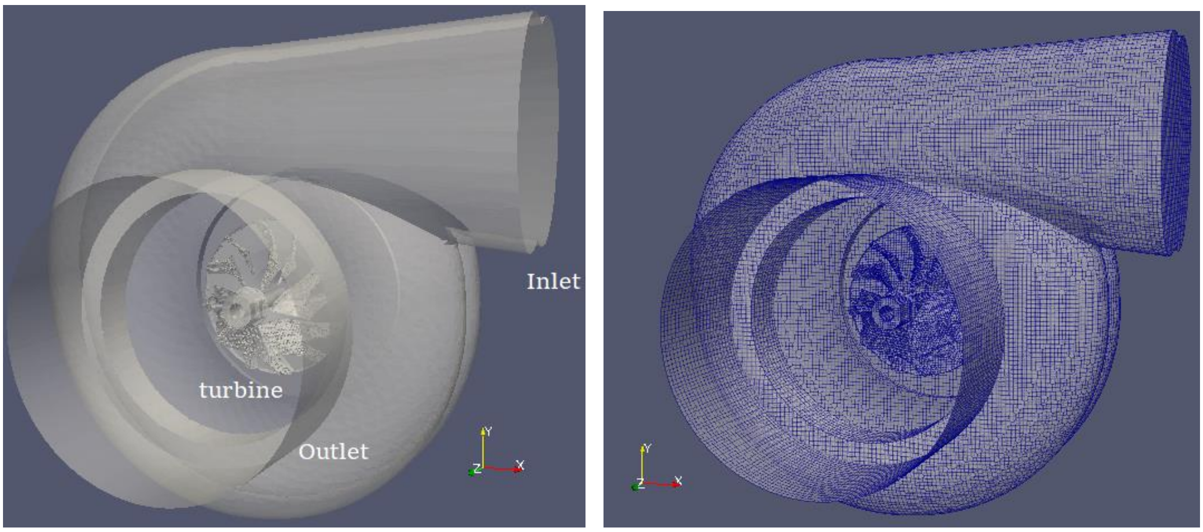


Fig.2. Fluid domain of the turbine (left), meshed model of turbine flow (right)

The objective of the present study is to simulate the flow field inside the turbine fluid domain with the rotation of turbine wheel imposed as dynamic mesh motion considered in the study. The geometry is created (Fig. 2) with the use of Salome software and the meshing is completed using snappyhexmesh grid generator. One of the solvers in OpenFOAM [1] namely rhoPimpleDymfoam solver is used in this work. This solver takes care of the flow field with temperature variations in a dynamic mesh environment. The mesh elements are carefully selected such that grid convergence criteria is satisfied with the experimental study. Inlet conditions are maintained at 463 K and 123 m/s with a blade velocity of 20,000 rpm. A test section plane is created normal to +Z axis and the streamlines emanating at the outlet are colored with angular velocity as shown in Fig. 3.

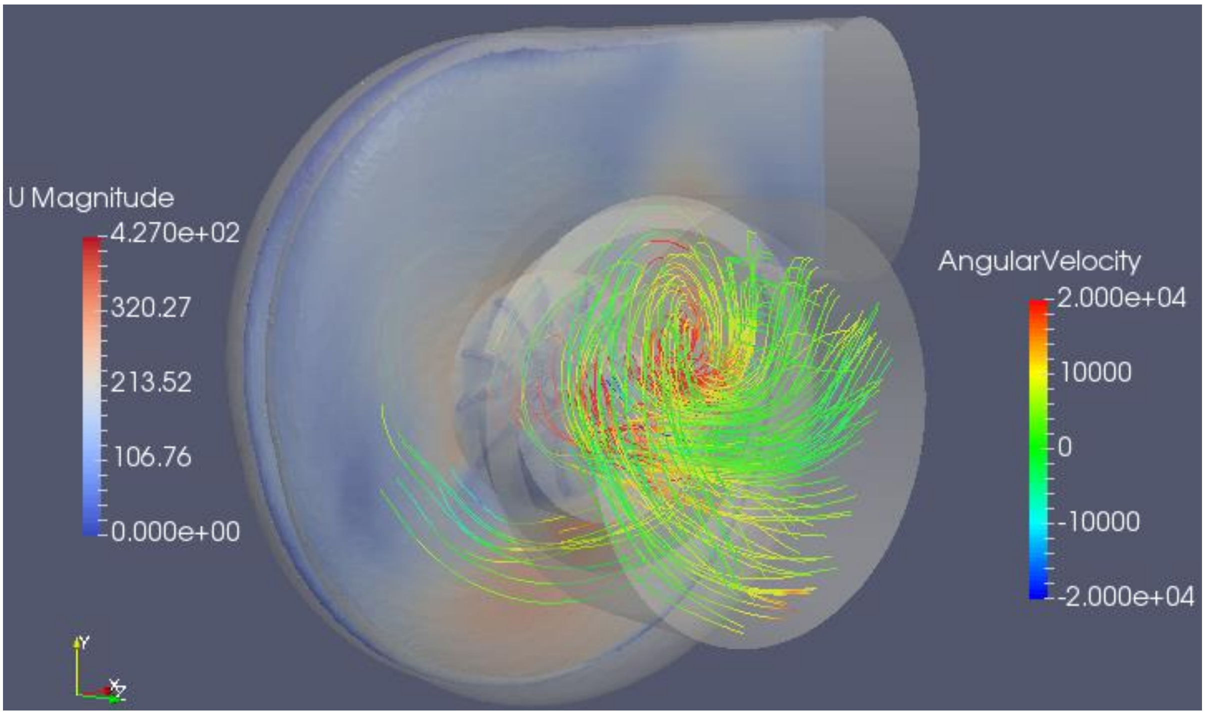
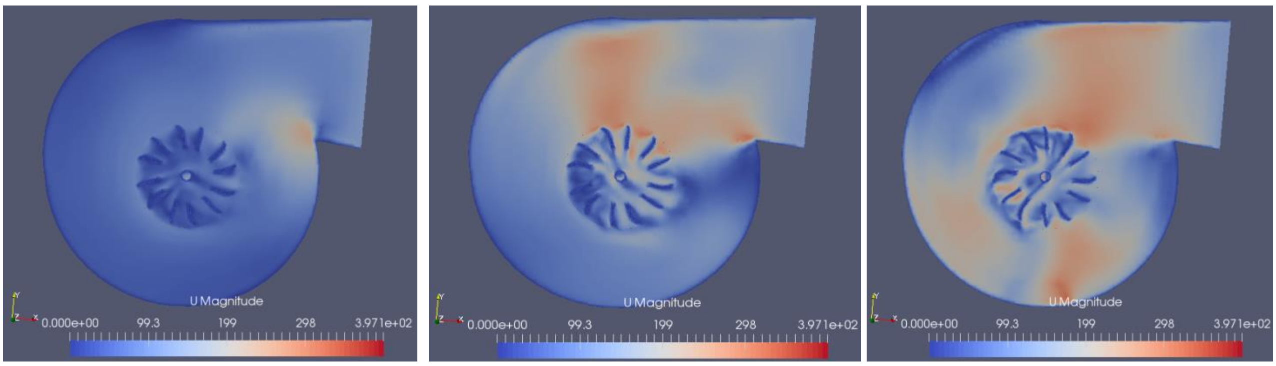


Fig. 3. Test section plane (shown in blue) with streamlines colored by angular velocity at the exit

Grid independence study is carried out at total mass flow parameter of 0.0105 kg/s.$K^{0.5}$kPa and the resulting non-dimensional mass flow rate parameter is compared as shown in Table 1. The convergence is reached as the number of elements increases beyond 2 million (in number) where the aspect ratio is almost close to 1.

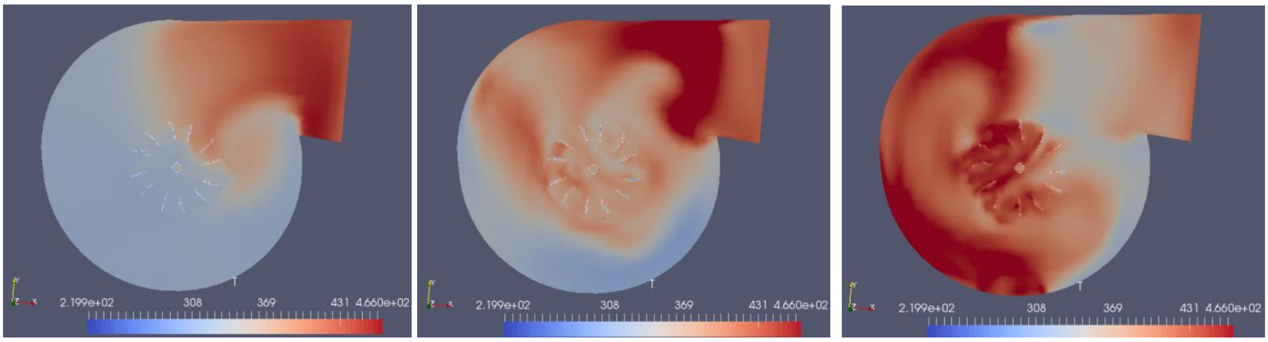
Table 1: Grid Independence study

Grid Size (no. of elements)	Total pressure at inlet-to-exit static pressure ratio
0.2 million	1.21
0.5 million	1.29
1.0 million	1.42
2.0 million	1.47



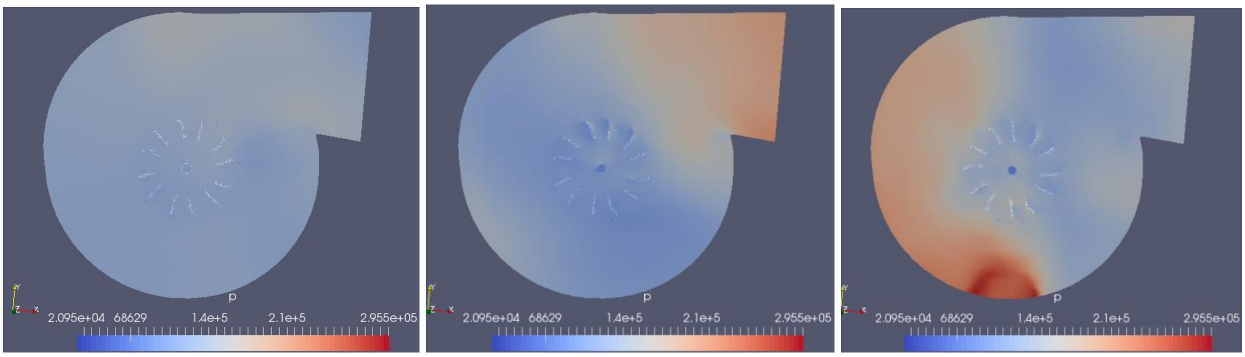
(a) (b) (c)

Fig. 4. Velocity magnitude in the test section at 15 mS (a), 30 mS (b) and 45 mS (c)



(a) (b) (c)

Fig. 5. Static Temperature contour in the test section at 15 mS (a), 30 mS (b) and 45 mS (c)



(a) (b) (c)

Fig. 6. Static Pressure magnitude in the test section at 15 mS (a), 30 mS (b) and 45 mS (c)

The contours of the velocity magnitude, static temperature and static pressure are presented in figures 4,5 and 6 respectively. It is clear that the flow motion from the inlet is influenced by the rotating turbine blade. The temperature distribution and flow expansion is also evident. The output parameters are compared against the experimental readings [2]. The output parameter from the study is compared and found to be within lower degree of variation with respect to experimental measurement as reported in literature[2]. The mass flow rate of the turbine is varied as per the following formula in equation (1) and the output parameter (pressure ratio in equation 2) is observed using Paraview.

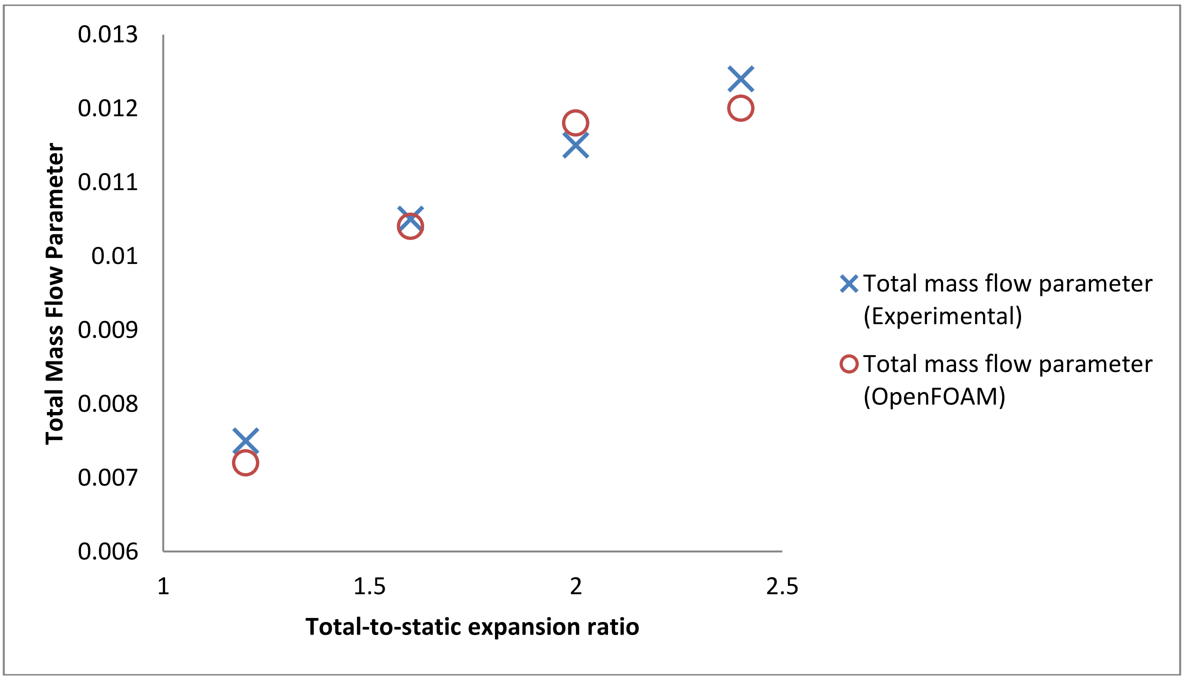


Fig. 7. Comparison of different cases by varying total-to-static expansion ratio

Total-to-static expansion ratio : P_o/P_{exit} (1)
Total mass flow parameter : $\dot{m}(T_o)^{0.5}/P_o$ (2)

T_o – Turbine inlet total temperature
P_o – Turbine inlet total pressure
P_{exit} – static pressure at the exit of turbocharger
$\dot{m}$ – mass flow rate in the turbine

This study is possible with the help of blueCFD software [3], an OpenFOAM source in windows distribution. The future of this work is aimed at coupling the turbine with the compressor fluid domain. Further, a realistic analysis is focussed on the fluid structure interaction by the expanding flow on the turbine blades.

Acknowledgements
The authors thank all those involved in the organisation of OFW13 and blueCFD-core team.

References

[1] OpenCFD, OpenFOAM: The Open Source CFD Toolbox. User Guide Version 1.4, OpenCFD Limited. Reading UK, Apr. 2007.
[2] Kevin John Fogarty, Turbocharger turbines: An experimental study on the effects of wastegate size and flow passage design, M.S. Thesis, Mechanical Engineering, The Ohio State University, 2013.
[3] BlueCFD-core project, http://bluecfd.github.io/Core.