Computational Fluid Dynamics (CFD) Study of Centrifugal Compressor using blockMesh Meshing in openFOAM

Caesar Wiratama[1, a)]

Author Affiliations
[1]*Aeroengineering Solutions, Sleman, Yogyakarta, Indonesia*

Author Emails
[a)] Corresponding author: caesar@aeroengineering.co.id

Abstract. The ERCOFTAC centrifugal pump is a benchmark experimental case to study the behavior of centrifugal pump. Numerous numerical Computational Fluid Dynamics (CFD) studies using Reynold Averaged Navier-Stokes (RANS) turbulent model has been caried out to study this case by utilizing openFOAM software. In this study, the same simulation was carried out using blockMesh, then compared to the previous similar work done using ICEM-CFD Meshing. The resulted velocity and pressure distribution of blockMesh study has a good agreement with the ICEM-CFD result but, slight velocity distribution error, static pressure poor local suction at the vane prediction, and some geometry shape errors were still observed for the low-resolution mesh.

1. INTRODUCTION

The relative motion between the rotor and stator at the small gap responsible to make an unsteady force created by high pressure fluctuation in that vicinity. The experimental work contributes to the understanding of the complex flow caused by rotator-stator interaction were caried out by European Research Community on Flow Turbulence and Combustion (ERCOFTAC) [1] together with Ubaldi [2], adopted the centrifugal test rig which then widely used to verify and validate theoretical and experimental study of centrifugal compressor.

Computational Fluid Dynamics (CFD) have been shown in the recent decade to be a useful complement to experiment. CFD result provides more extensive solutions in the whole domain, giving better overall understanding. The improvement of the software, hardware, and computational algorithms in the recent years have shown promising evidence that CFD is a reliable tool to study unsteady flow [3].

To capture a detailed complex flow, the large meshes number and short time steps are often used, which making the simulations computationally heavy and becomes costly when proprietary software is used, where there is a license cost. To overcome that problem, the community-driven openFOAM [4] turbomachinery working group extends and validates openFOAM for turbomachinery applications [5]. openFOAM is finite volume method based open-source library written in C++ which is proven to be as accurate as proprietary codes for many applications [4-8].

2D CFD simulation of ERCOFTAC centrifugal pump were previously made using openFOAM by Petit et al. [9]. The general behavior of the flow was well captured, but the detailed unsteadiness of the flow was not well captured. Then, unsteady flow using k-epsilon, k-epsilon realizable, k-epsilon RNS, and the k-omega SST was caried out by Petit et al. [10] and some detailed finite volume scheme selection study was carried out by Xie [13] which utilize ICEM-HEXA meshing technology.

In this study, the same case and computational setup will be implemented, but blockMesh meshing will be used and compare the result with ICEM-HEXA mesh to examine the feasibility of blockMesh in incompressible turbomachinery simulation.

2. METHOD

In this study, the flow through 2D centrifugal pump is investigated using incompressible, transient, Reynolds-Averaged Navier-Stokes (RANS) equations. To solve the equations, finite-volume method openFOAM software is used with pimpleFoam solver, which is transient, incompressible, PIMPLE (combined PISO and SIMPLE)-based algorithm solver with k-omega turbulent model. The finite volume schemes used are based on the Xie's [13] study.

Transient case with 5e-6 s time step is used to do 5e-2 s total time to ensure the flow is fully developed. The blockMesh, an openFOAM meshing utility with four different sizes are used to simulate this case. The sizes used in this study is summarized in the **TABLE 1**.

TABLE 1. Meshing parameter used in blockMesh

Mesh category	Average cell size		Cells
	mm	% Diffuser radius	
A	4,5	1,35 %	21.664
B	3	0,9 %	48.736
C	2	0,6 %	96.597
D	1	0,3 %	386.724

Four arcs are used to generate the diameter of the cylinder and 1 element is used to extrude it as 2D mesh to be the basic blockMesh shape, then snappyHexMesh utility is used with zero refinement to snap the rotor and stator shape in the mesh.

The time used to perform the simulation for each mesh are also recorded to investigate the feasibility of the size used with respect to the computational effort. The number of cells versus time is plotted in **FIGURE 1**. This study is carried out using daily used PC core i7 with 32 GB RAM.

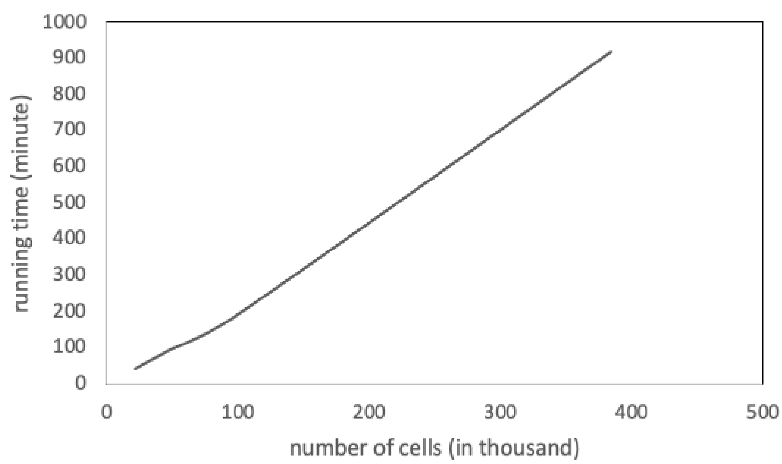


FIGURE 1. Running time versus number of cells

The mesh used in the previous study was generated using ICEM-HEXA is shown in **FIGURE 2**, and the mesh created with blockMesh are shown in **FIGURE 3.**

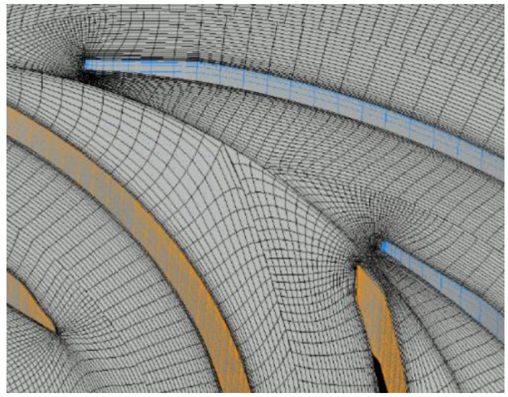
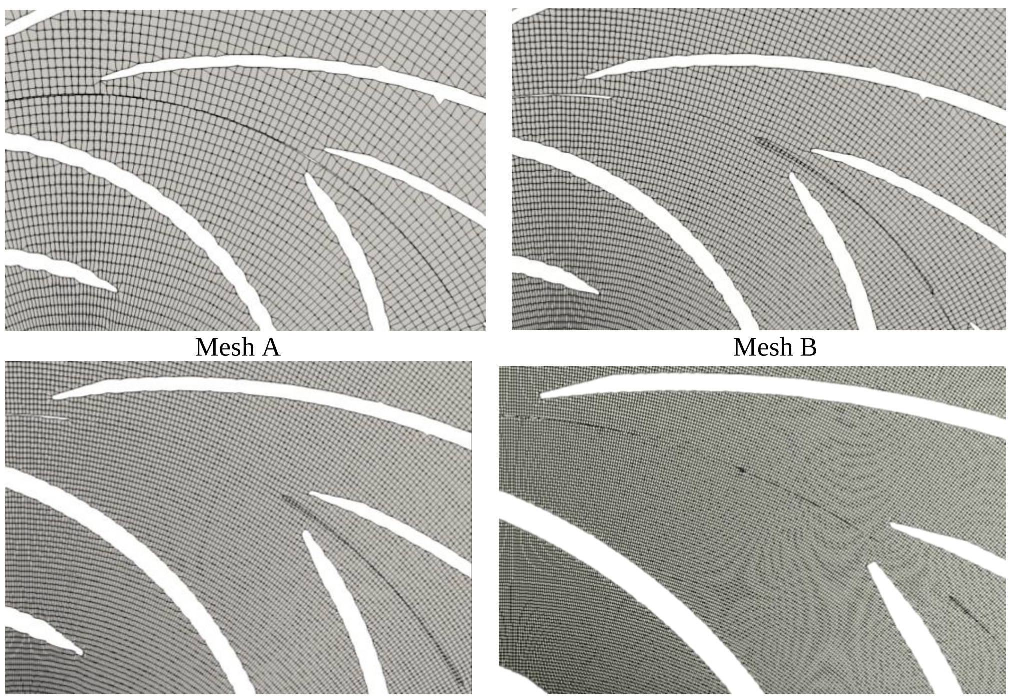


FIGURE 2. Previous study mesh created using ICEM-HEXA [12]



Mesh A Mesh B

Mesh C Mesh D

FIGURE 3. Various blockMesh used from coarsest (mesh A), to finest (mesh D)

3. THE ERCOFTAC CENTRIFUGAL PUMP CASE STUDY

The ERCOFTAC (European Research Community on Flow Turbulence and Combustion) centrifugal pump is a simplified model of a centrifugal turbomachinery to be used as a benchmark model which was presented by Combes [1] at a Turbomachinery Flow Prediction ERCOFTAC workshop in 1999. This model has 7 impeller blades, 12 diffuser vanes and 6% vaneless radial gap as shown in **FIGURE 4.** The geometry and operating conditions data are shown in **TABLE 2.**

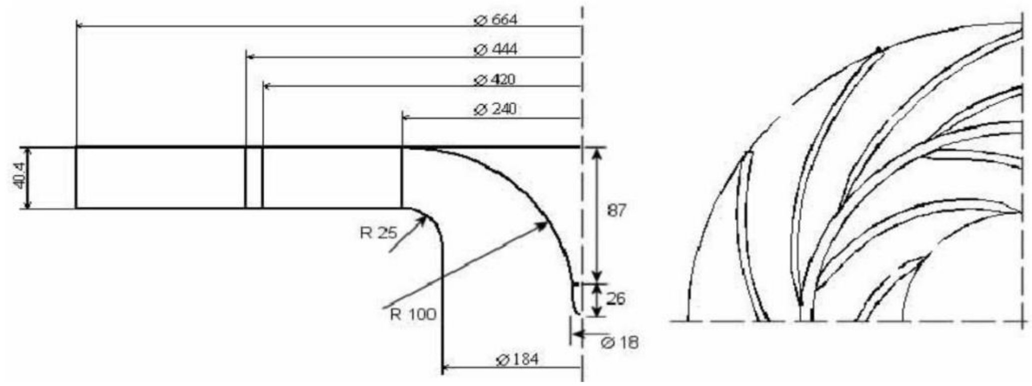


FIGURE 4. Impeller and vaned diffuser geometry of ERCOFTAC pump [11]

TABLE 2. Geometric data and operating conditions of ERCOFTAC pump [11]

Impeller	
Leading edge diameter	240mm
Trailing edge diameter	420mm
Number of blades	7
Blade span	40,4 mm
Diffuser	
Leading edge diameter	444mm
Trailing edge diameter	664
Number of vanes	12
Outlet diameter	750mm
Operating Conditions	
Rotational speed	2000rpm
Impeller tip speed	43,98m/s
Flow rate coefficient	0,048
Total pressure rise coefficient	0,65
Reynolds number	$6{,}5 \times 10^5$

The ERCOFTAC centrifugal pump was also used as a case study for the Fourth openFOAM workshop, in Montreal, Quebec, 2009. In this present study, a 2D representation of the geometry is used, and the boundary conditions are summarized in **TABLE 3.**

TABLE 3. Boundary conditions used in the simulation

Variable	Value
Inlet diameter	200mm
Thickness	1mm
Flow rate	$0{,}292 m^3/s$
Inlet radial speed	11,4m/s
Viscosity ratio	10
Inlet k	$0{,}48735 m^2/s^2$
Outlet static pressure	0Pa

4. RESULTS

The simulations have done without any Courant number problems. The fully developed flow pattern take place at time about 2.e-2 to 3.e-2s, but the simulations were not stopped until time 5.e-2s to ensure the consistency of the transient flow pattern from the initialization effect. The velocity and pressure distribution at t/Ti = 0,126 which indicates the relative angular position of the rotor.

The velocity distribution of ICEM-HEXA mesh and various blockMesh sizes are shown in **FIGURE 5.**

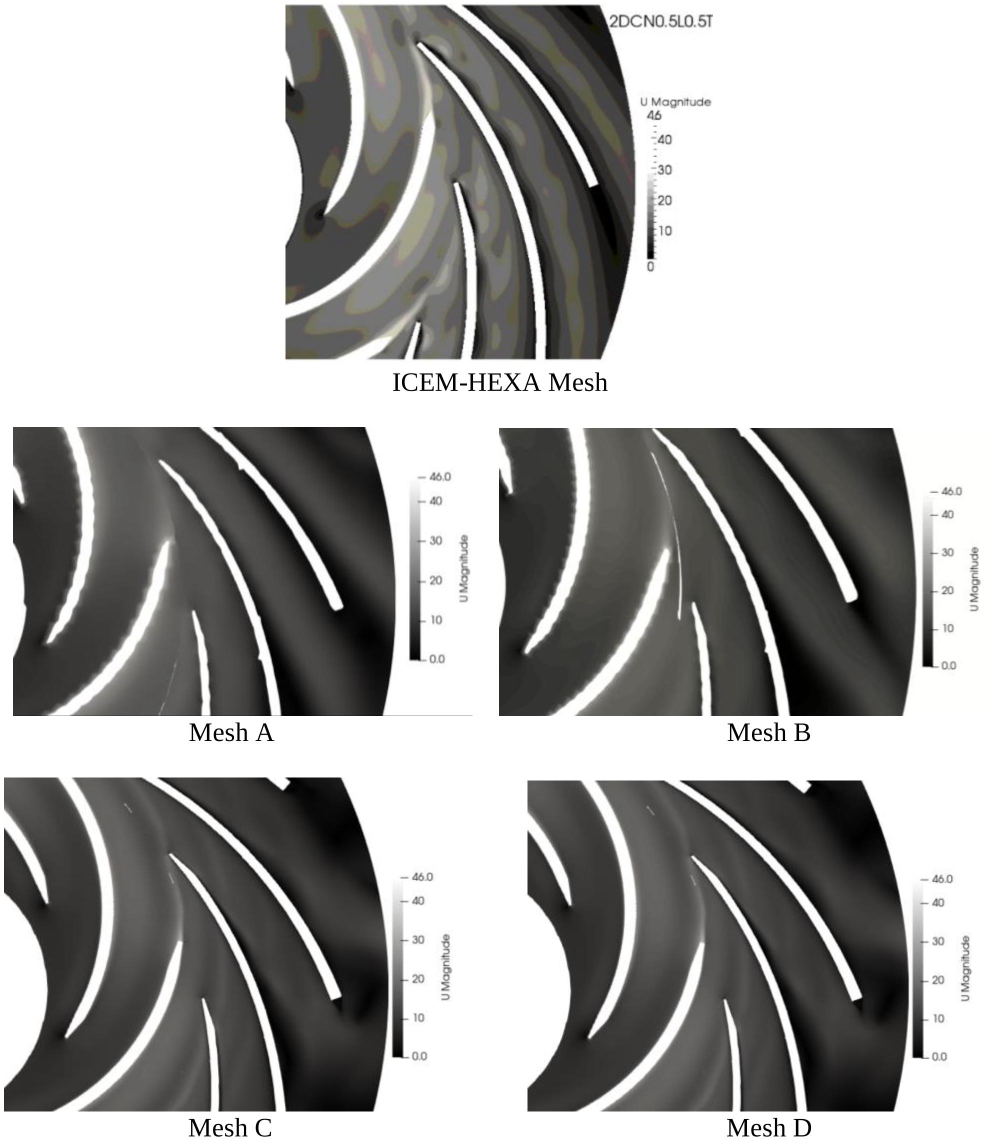


FIGURE 5. The resulted velocity contour from the previous study using ICEM-HEXA [13], and various blockMesh settings

From the velocity distribution, we can see a good agreement between ICEM-HEXA mesh from previous study and blockMesh at the various sizes globally, but slight differences at the detailed flow near the walls are observed for low resolution blockMesh (mesh A and mesh B). The shape of the snapped rotor and vane geometry for mesh A and mesh B also not well-fitted with the expected geometry curve which is affect the flow near those regions. This is happened because the refinement used at snappyHexMesh setting is zero.

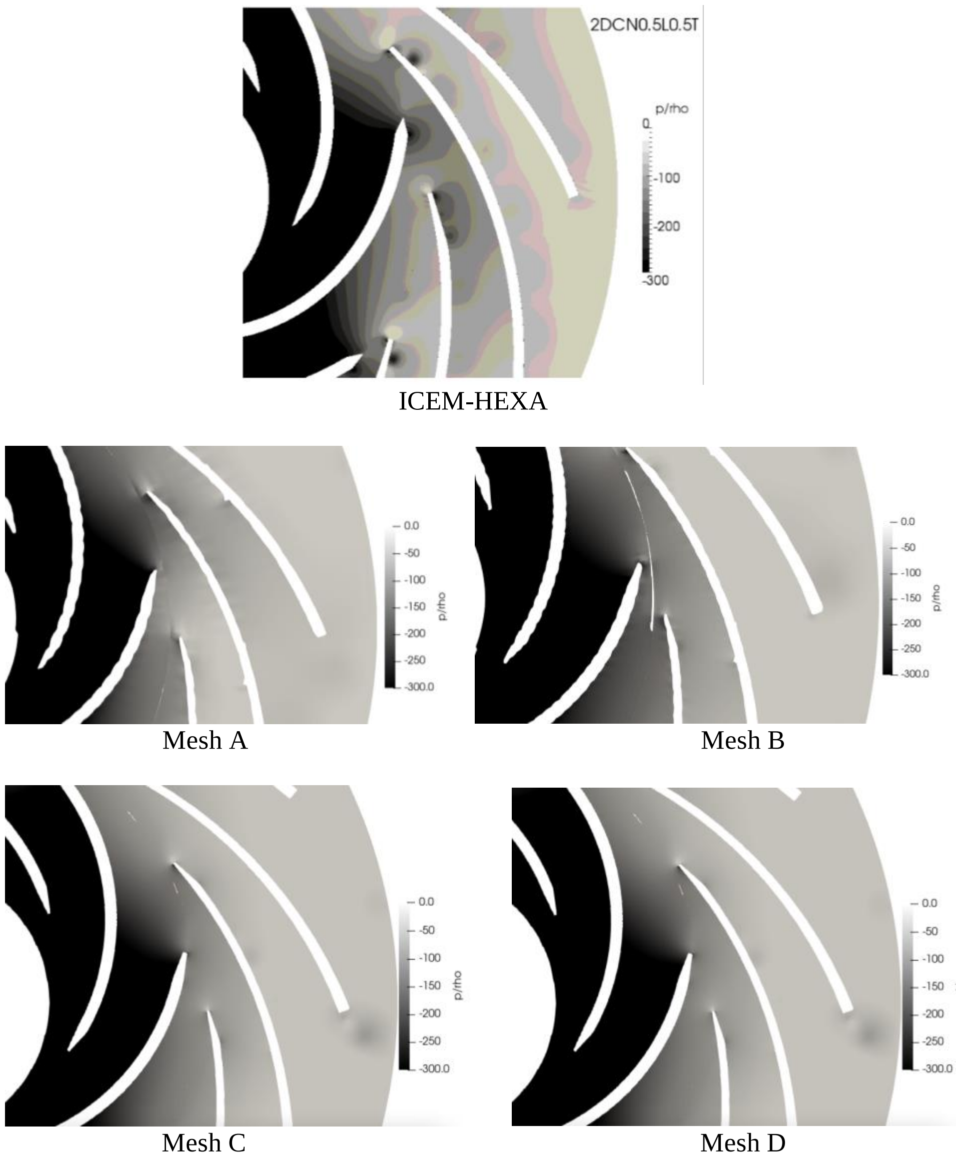


FIGURE 6. The static pressure contour from the previous study using ICEM-HEXA [13], and various blockMesh settings

In openFOAM, the pressure is generally divided by the density, which is indicated the kinematic pressure. From the kinematic pressure distribution, we can see a good agreement between the ICEM-HEXA mesh and the various blockMesh sizes globally. The similar trend with the velocity distribution also observed in this static pressure distribution. The local suction pressure in the vicinity of vane Is not observed in the mesh A, this is because the mesh size is too large and cannot capture the local swirl flow occurs at that location which generate the local suction pressure.

5. CONCLUSION

The incompressible, transient, turbomachinery simulation have been investigated using OpenFOAM with various mesh type and size. From the velocity and static pressure gradient, the blockMesh meshing for coarse and fine size generated a good agreement with previous study done with ICEM-HEXA meshing at the global level. Slight difference

Is observed at velocity distribution for mesh A and mesh B, and local swirling flow at the vane leading edge were not observed for mesh A.
The mesh A and mesh B were not well-handled the curvature of the blade and vane geometry because of the zero refinement at the snappyHexMesh setup. The suggestion for the next study is to use the one or higher snappyHexMesh refinement at the rotor and vane walls.

6. ACKNOWLEDGEMENT

Author would like to acknowledge that the computational resources and data used have been supplied by the Aeroengineering solution from the opensource simulation software project, which is a project of introducing, explore, and supply the sufficient resources for the opensource CFD and FEA software as robust and feasible solver for industrial applications.

7. DISCLAIMER

8. REFERENCES

[1] J.F. Combes, "Test case U3: centrifugal pump with a vaned diffuser", in *Proceedings of the ERCOFTAC Seminar and Workshop on Turbomachinery Flow Prediction VII,* 1996.

[2] M. Ubaldi, P. Zunin, G. Barigozzi, and A. Cattanei, "An experimental investigation of stator induced unsteadiness on centrifugal impeller outflow", *Journal of Turbomachinery,* vol 118, no. 1, pp. 41-51, 1996.

[3] H. Keck and M. Sick, "Thirty years of numerical flow simulation in hydraulic turbimachines", Acta Mechanica, vol. 201, no. 1-4, pp. 211-229, 2008.

[4] H.G. Weller, G. Tabor, H. Jasak, and C. Fureby, "A tensorial approach to computational continuum mechanics using object-oriented techniques", *Computers in Physics,* vol. 12, no. 6, 1998.

[5] H. Nilsson, M. Page, M. Beaudoin, and H. Jasak, "The openFOAM turbomachinery working group, and conclusions from the turbomachinery session of the tird openFOAM worshop", in *Proceedings of the 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and System,* Brno, Czech Republic, 2009.

[6] O. Petit, H. Nilsson, T. Vu, O. Manole, and S. Leonsson, "The flow in the U9 Kaplan turbine-preliminary and planned simulations using CFX and openFOAM", In *Proceedings of the 24th IAHR Symposium on hydraulic Machinery and System,* 2008.

[7] S. Muntean, H. Nilsson, and R.F. Susan-Resiga, "3D Numerical analysis of the unsteady turbulent swirling flow in a conical diffuser using fluent and openFOAM", in *Proceedings of the 3rd IAHR International Meeting of the workgroup on cavitation and Dynamic Problems in Hydraulic Machinery and Systems,* Brno, Czech Republic, 2009.

[8] H. Nilsson, "Evaluation of openFOAM for CFD of turbulent flow in water turbines", in *Proceedings of the 23rd IAHR Symposium on Hydraulic Machinery and Systems,* 2006.

[9] O. Petit, M. Page, M. Beaudoin, and H. Nilsson, "The ERCOFTAC Centrifugal pump openFOAM case-study", in *Proceedings of the 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and System,* Brno, Czech Republic, 2009.

[10] O. Petit, and H. Nilsson, "Numerical Investigations of Unsteady Flow in a Centrifugal Pump with a Vaned Diffuser", in *International Journal of Rotating Machinery,* vol 2013, London, 2013.

[11] Ubaldi M., Zunino P., Barigozzi G. and Cattanei A., "*An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal Impeller Outflow",* Journal of Turbomachinery, vol.118, 41-54, 1996.

[12] M. Beaudoin and H. Jasak, "Developent of a Generalized Grid Interface for Turbomachinery simulations with OpenFOAM,' in *Open source CFD International Conference,* 2008.

[13] Shasha Xie, "Studies of the ERCOFTAC Centrifugal Pump with OpenFOAM", *Department of Applied Mehanics, Division of Fluid Dyanmics, Chalmers University of Technology,* Sweden, 2010.