RESEARCH ARTICLE OPEN ACCESS

Numerical Simulation of Industrial Centrifugal Blower

Prathamesh V. Gurav[1], Sanjay V. Kadam[2]
[1]PG- student Mechanical- Heat Power Engineering, RIT
Email: prathamesh2718@gmail.com
[2]Professor Mechanical- Heat Power Engineering, RIT
Email:sanjay.kadam@ritindia.edu

--************************************----------------------------------

Abstract:

Accuratesimulation of Turbomachinery will help you to evaluate alternative design iterations more quickly. As a result,increase efficiency, reliability and durability. Providing solutions for Centrifugal blower design and performance optimization through computer-aided simulations has been easier and more cost-effective. Through a reduction in measurements, CFD helps to reduce the turn-around time for development and improve the product much faster.

Keywords: CFD,Centrifugal blower, turbomachinery

--************************************----------------------------------

1.Introduction :

Computational fluid elements (CFD) is a part of fluid mechanics that utilizes numerical examination and information structures to dissect and tackle issues that include fluid streams. CFD has been effectively applied in different zones of fluid mechanics. This incorporates optimal design of vehicles and aeroplane, hydrodynamics of boats, siphons and turbines and thermodynamics of fan, blower, and blower. Turbomachines are a mix of fixed and pivoting sections. Fluid going through these entries either gets or delivers vitality. Turbo hardware streams are normally temperamental mostly because of the overall movement of rotors and stators and the regular stream dangers present in tip holes and optional streams. There are numerous strategies to mimic fixed and pivoting sections together by comprehending their mass, force and vitality move conditions. Computational fluid elements are one of the most amazing assets to reenact such complex stream framework. CFD gives a doable arrangement of the difficulties which is under dangerous conditions in genuine. It additionally expands its answer past the useful limits of the issue. It permits checking arrangement ideally by offering a wide scope of variable boundaries for an issue under investigation. Any sensible stream reenactment must be done on a three-dimensional premise.

These days virtual three-dimensional stream investigations are practical and permitting fashioner to have a better gauge of the impact of spatial boundaries on the execution of the turbomachine. Computational fluid elements (CFD) gives a precise choice to scale model testing, with varieties accessible in reproduction boundaries because of ongoing advances in processing power, along with incredible illustrations and intelligent three-dimensional controls of models. Propelled solvers contain calculations which empower powerful arrangements of the stream field in a sensible time. Because of these variables, Computational Fluid Dynamics is presently a set up mechanical plan apparatus, assisting with diminishing structure time scales and improve forms all through the designing scene

2.Modelling:

Blower Geometry is created using SolidWorks CAD Software. Blower casing, Inlet duct, Impeller is created using available drawing. The cad model is imported to the workbench as a step file format. The fluid domain is extracted from this geometry using the design modeller tool. The blower impeller having a backward inclined blade that rotates with rotating speed of 1755 rpm.

- Volume flow rate = 4000 CFM
- Pressure difference = 6.25 inch WC
- Speed of fan= 1755 rpm
- Power = 4.22 kw
- Impeller Diameter=584 mm
- Impeller Width= 118 mm
- Types of blades= Backward Inclined

Quality Parameter	Actual Value	Required value
Element Quality	0.17048	> 0.05
Aspect Ratio	14.729	< 80
Skewness	0.85	< 0.95

- No of Blades=11

To compute the answer for a given simulation case, the simulation domain must be discretized. It essentially means to separate up one large problem into multiple smaller mathematical problems. The whole domain is meshed using unstructured type mesh.

Table 1 :Mesh quality parameters

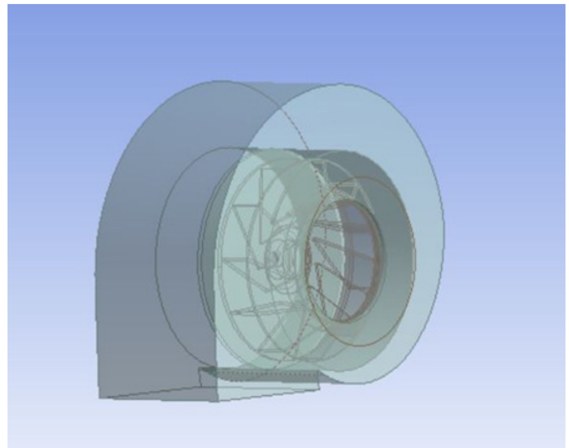

Fig 1: Fluid Domaim

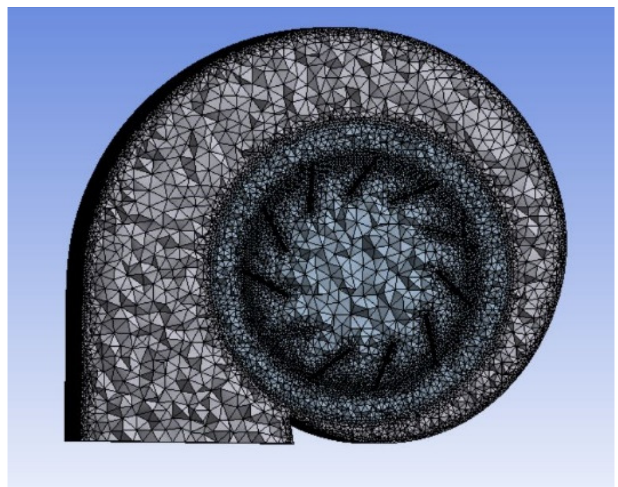

Fig 2. Meshed Model

Component	Nodes	Element
Inlet zone	55460	51262
Rotating zone	186399	902820
Casing zone	188941	902097
Total	430800	1856179

Table 2: Number of Mesh Element

3.Case setup:

After meshing meshed blower is transferred to the Ansys fluent. For our simulation transient analysis is used to compute result more accurately.

3.1 Properties of Material :

Air with standard properties such as

- Density =1.2 kg/m^3
- Viscosity = 1.81e-05 kg/ms

3.2 Boundry conditions:

The inlet boundary is considered as pressure inlet. Which is atmospheric in this case. The outlet boundary condition is mass flow outlet and it is calculated from the volume outlet of blower.

Boundary Condition	Type
inlet	Pressure inlet
outlet	Mass flow outlet

Table 3:Boundary conditions

3.3 Turbulence model:

Various turbulence models are available for our analysis out of which Ke, Sa and Kw are suitable for our analysis. Ke and w are two-equation models and Sa is one equation model.

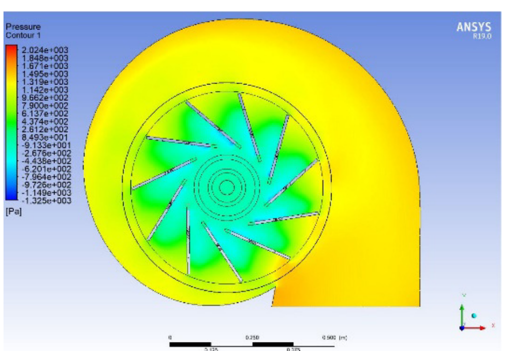

Fig. 3: Pressure Contour for the k-e model

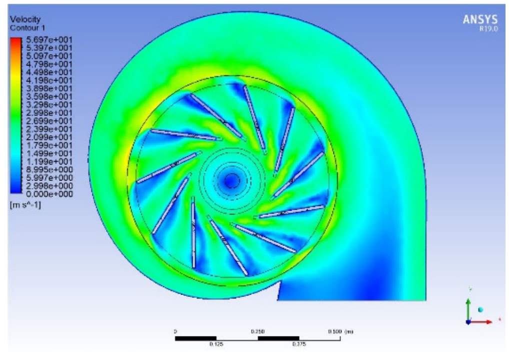

Fig. 4: Velocity Contour for the k-e model

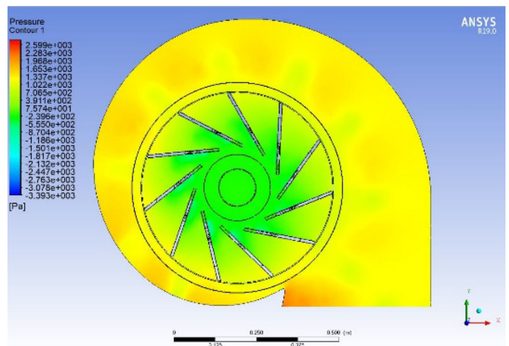

Fig. 5: Pressure Contour for the S-A model

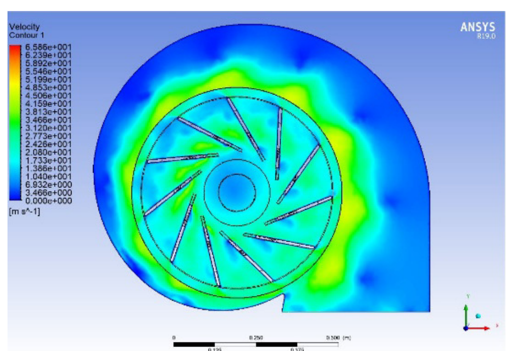

Fig. 6: Velocity Contour for the S-A model

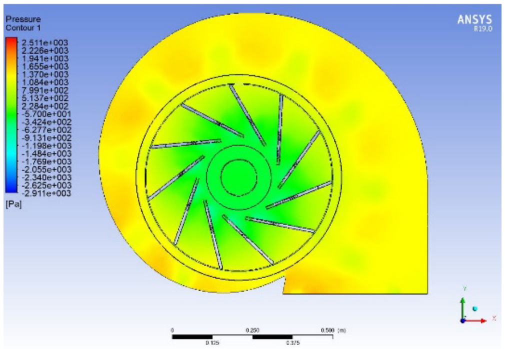

Fig. 7: Pressure Contour for k-w model

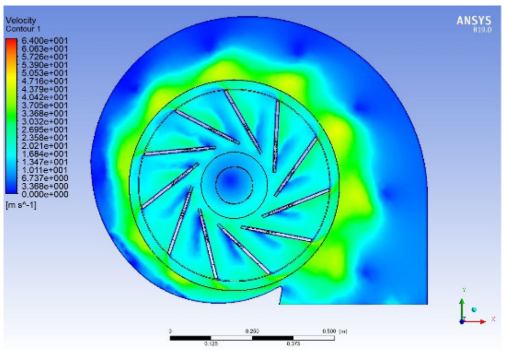

Fig. 8: Velocity Contour for k-w model

4.Computational results:

The simulation is carried out in Ansys fluent 19. The simulation is carried out using different turbulence models.The pressure difference in case of spalart-allamaras and k-omega model is less compared to the k-epsilon model. Spalart-allamaras and k-omega are unable to model flow behaviour near the casing wall while the k-epsilon model gives good correlation to theoretical values.

Model	Pressure Difference (inch wc)	Volume flow Rate (m3/s)
Spalart-Allamaras	5.46	1.9
k-epsilon	6.36	1.89
k-omega	5.44	1.9

Table 3:Computational Results

5. Conclusion:

The k-epsilon model provides better results than spalart-allamaras and k-omega model. The pressure difference in case of spalart-allamaras and k-omega model is less compared to the k-epsilon model. Our analysis shows that result obtained with the k-epsilon model provides better results than the k-omega and spalart-allamaras model. CFD provides a good correlation to theoretical values. It helps us to analyze the flow behaviour inside the blower.

References:

1.Beena D. Baloni, Yogesh Pathak, S.A. Channiwala. "Centrifugal blower volute optimization based on Taguchi method." Computers & Fluids 112 (2015) 72–78.

2.Sunil R.Patil, Sandip T.Chavan, Nitin S. Jadhav, Shivaraj S. Vadgeri. "Effect of Volute Tongue Clearance Variation on Performance of Centrifugal Blower by Numerical and Experimental Analysis." Materials Today: Proceedings 5 (2018) 3883–3894.

3.V.H.Chaudhari, Varun Parmar, M.J.Zinzuvadia. "Parametric Study and Geometrical Optimization of Centrifugal Blower." International Conference on Research and Innovations in Science volume 1, 2017.

4.Young-Tae Lee, Hee-Chang Lim. "Performance assessment of various fan ribs inside a centrifugal blower." Energy 94 (2016) 609-622.

5.Utkarsh Diliprao Charapale , Arun Tom Mathew. "Prediction of Flow in the Industrial Blower using Computational Fluid Dynamics." Materials Today: Proceedings 5 (2018) 12311–12319.

6.Li Chunxi, Wang Song Ling , Jia Yakui. "The performance of a centrifugal fan with enlarged impeller." Energy Conversion and Management 52 (2011) 2902–2910.

7.Sheam-Chyun Lin, Ming-Lun Tsai. "An integrated performance analysis for a backward-inclined centrifugal fan." Computers & Fluids 56 (2012) 24–38.

8.Jason Stafford, Ed Walsh, Vanessa Egan. "A study on the flow field and local heat transfer performance due to geometric scaling of centrifugal fans." International Journal of Heat and Fluid Flow 32 (2011) 1160–1172.

