International Journal of Scientific Research & Engineering Trends
Volume 8, Issue 5, Sep-Oct-2022, ISSN (Online): 2395-566X

Development and Analysis of Flow through Annular Curved Diffuser with Fins

K. Manoj Kumar, S. Deepthi, Dr. P. Srikar
Department Of Mechanical Engineering
Mallareddy College Of Engineering And Technology
Medchal, Telangana, India
konatham.manojkumar1@gmail.com, hdeepthi2004@mrcet.ac.in, srikarpotnuru@mrcet.ac.in

Abstract – Diffusers are largely used in centrifugal compressors, axial flow compressors, combustion chambers, ram jets, gas turbine engines, inlet portions of jet engines, and so on. A minimal change in pressure recovery will increases the efficiency of the machinery. Hence diffusers are absolutely essential for good turbo machinery performance. The internal part of gas turbine engines are curved annular diffusers at high speed air craft. By decreasing the entire pressure loss the diffuser facilitate effective working of the combustor. Performance of those diffusers depends on the geometrical dimensions of diffuser and inlet conditions. In the present investigation, distribution of static pressures inside the diffuser and velocity of the fluid at outlet are studied with help of CFD on a curved annular diffuser of 7^0 angle of turn, circular hub of diameter to 20mm and also by varying the fins on circular hub of height to 5mm and 10mm with a thickness of 3mm and allowing the air as fluid to pass through the diffuser. Annular curved diffuser is modeled by using CREO Parametric. Modifications are done by adding fins to the model and CFD analysis is done in ANSYS Fluent to determine the flow characteristics.

Keywords – Annular Curved Diffuser, Cero, Ansys Fluent.

I. INTRODUCTION

Diffusers are mainly used in the centrifugal compressors, combustion chambers, ram jets, gas turbine engines, inlet chamber of jet engines, and so on. Consequently, very massive ranges of residual kinetic power often fulfil the work enter and work extrusion methods, sometime as a great deal as fifty percent of the entire energy being transferred. Small alternate in pressure recovery will increase the performance extensively. Consequently diffusers are in reality important for turbo machinery overall performance.The geometric conditions in plane programs where diffusers want specially designed on the way to reap maximum stress restoration inside the shortest viable duration brought about the development of curved annular diffusers.

Diffusers may be located in most of the turbo machines. In particular widespread, a part of the period of the gasoline turbine engines is taken up with the aid of diffuser attributable to the excessive integral velocities which it develops in some sections. In the advanced aeronautical engine diffusers will be positioned within the intake, between compressor spools, at combustor inlet, between turbines and in the exhaust duct. Most people of these diffusers can be of annular configuration and designed with the intent of making them feasible. The quintessential components of the gasoline turbine engine are curved annular diffusers at high steam air craft. By decreasing the overall pressure loss the diffuser facilitate powerful operations of the combustor. The overall performance of those diffusers depends on their geometry and inlet conditions. Diffuser are works on exactly opposite of venturi impact because of this while excessive speed air is exceeded from smaller location to large vicinity then its speed decreases and static pressure is increase this procedures on which diffusers are works. The velocity is increasing as the cross closer to outlet (larger area) from inlet (smaller area) this shows that pressure is lowering from inlet to outlet in nozzle that is precisely contrary to that of diffuser manner diffuser are opposite of nozzle.

II. LITERATURE REVIEW

Annular curved diffuser is a common mechanical device, however, their performance and flow characteristics depends on their geometrical dimensions and inlet conditions. Annular curved diffuser have been investigated by many researchers due to their wide spread usage some of which are listed in brief.

Sangappa Hadapad et.al [1], has investigated the distribution of stationary pressure and general stress are studied with the assist of Computational Fluid Dynamics on a annular diffuser of 30^0,35^0 and 40^0 perspective of turn and also via varying the circular hub of diameterto5mm, 7.5mm, 10mm, 12.5mm,15mm and 20mm and permitting the air as fluid to pass through the diffuser. ANSYS FLUENT 14.5is used for this assessment and Turbulence model as general k- ε extensively utilized. On this investigation it turned into observed that for a small perspective of turn the static pressure recuperation is more

© 2022 IJSRET

and as the perspective of turn increases static stress recuperation reduces and additionally the overall strain loss is extra. And it became also found that for an angle of three hundred with 20mm hollow diameter the static stress is better and general pressure is less.

Mohammad Maqsood et.al [2], has studied absolutely high-quality shapes are used and outcomes have been as compared for strain restoration. 4 actually awesome discretionary volumes of diffusers are attempted & outcomes are conferred. Experimental set-up being surprisingly high priced& time-phase is overwhelming, these strategies are idea-approximately right here that is for experimental. It is a commercial tool using for this task. From CFD effects it is determining that the 0.33 version with the diffuser has giving the only stress healing as differenced to the alternative elements. finally the 1/3 model is notion-about because of maximum beneficial version.

Raghul. M et.al [3], Has pursued a few latest advances in computational gasoline dynamics of exhaust diffusers are provided. Numerical research were done for an annular kind fuel turbine exhaust diffuser with inlet manual vanes and with and without. Diffuser is the final expansion level of the gasoline turbine used to increase the pressure by using eating kinetic strength. A symmetric version of the whole section is considered and simulations are achieved to look at the struts placement on turbine performance. The simulations are executed to calculate the stress healing coefficient, for an most fulfilling divergence attitude of 13o by using maintaining the diffusion duration as constant. With a view to investigate this impact of go with the flow improvement in the passage, diverse drift conditions are implemented and effects are proven. The variant in inlet velocity is in steps of 40m/s from 80m/s to 200m/s. The final results for with and without struts implies how the pressure healing coefficient effect the overall efficiency of turbine. Some character thoughts also are offered for attention.

P.K. Sinha et.al [4], has studied The velocity distribution of mean , strain & general stress with the use of experiments on a diffuser of 37.05° perspective of turn a ratio at 1.273 at Reynolds no 1.84 x 10^5on the idea of inner diameter & common inner tempo. This experiment effect had been demonstrated with the help of fluid-fluent so a chain of steady researches are completed with the duration of centred line &inlet diameter but with completely particular place ratios changes from 1.25 to 2.01 with change in mind-set of flip from 30º to 75º. The distribution of speed suggests fluids of excessive pace shifted & gathered on the exit phase conical wall due to the mixed effect of centrifugal motion &tempo diffusivity. The consistent research it's decided will recovers of static stress which booms as much as location percentage of two or more and stress restoration decreases steadyup to attitude of flip seventy fiveº. The entire strain loss co-efficient stays steady with the changes in areas and ratios & perspective of comparable inlet situations..

O.H. Jacobsen et.al [5], has researched on a systematic series of exams are done on a own of diffusers anywhere the outer diameter is maintained at ordinary casing. These kinds of diffusers are traditional of rotary motor exhausts. Records, in the type pressure healing regular is standard towards diffusers period for masses without a doubt certainly a type varieties of centre body closure. It's miles been confirmed that such diffusers could have quick duration centre our bodies that are fashion publications has been mounted.

R. Keerthana et.al [6], Studied on Diffusers are substantially utilized in centrifugal compressors. The energy switch in those faster machineries entails the trade of tremendous levels of kinetic electricity so as to accomplish the supposed reason. Very huge ranges of residual kinetic electricity frequently accompany the paintings input and paintings extraction processes, sometime 50% of the whole strength transferred. A small change in stress recovery can increases the efficiency appreciably.

Consequently diffusers are virtually critical for accurate turbomachinery performance the geometric barriers in plane programs in which the diffusers want to be particularly designed which will obtain maximum pressure restoration inside the shortest feasible length brought about development of annular diffusers. Within the present observe the entire performance of the sequence of diffusers of angles (9º, 15º, 21º and 27º) is calculated. A 3D diffuser version turned into carried out in seasoned-E and the analysis also executed in ansys. These final-results are shows the form of pace diagram, pressure diagram, static stress distributaries alongside casing wall and hub. Outcomes display some hanging functions, the divergent angles is increased the stress restoration.

S.N Singh et.al [7], This Author studied the S formed rectangular diffuser with the aid of momentum injection via CFD. During analysis they found glide distribution and stress healing progressed significantly because they modified the diffuser having component ratio of 6 at inlet. And 3cm diameter positioned inflexion plane throughout width of diffuser and specific speeds to inject momentum to the decelerating glide because of boundary layer separation.

Ram Kumar Raman et.al [8], this project refers about CFD is well established as a tool of for solving issues that indicates one or many of the following phenomena: flow of gases, heat transfer, mass transfer and chemical reaction. The present work reviews the salient features as well as practical application of CFD techniques carried out in the last more than 50 years in various sections related to industry with the objective of how and where

© 2022 IJSRET

it can be applied. It is concluded that CFD presents a number of opportunities in industrial work analysis, and the scope of this opportunity will further develop as both computational hardware and software resources becomes more advanced.

Ambarita et.al [9], studied on CFD is a numerical technique that solves fluid flow and related governing equations the usage of a computational device. The research on CFD, its technique and its software as a studies device, are growing. In this look at, utility of CFD by using Indonesian researcher is in brief reviewed. The main goal is to explore the characteristics of CFD packages in Indonesian researchers. Considering the dimensions and reputation, this study uses Scopus courses listed statistics base.

All the documents in Scopus related to CFD which is affiliated via as a minimum one of Indonesian researcher are accumulated to be reviewed. Research topics, CFD method, and simulation effects are reviewed in quick. The consequences display that there are 260 documents found in literature indexed by means of Scopus. Those files divided into research articles one hundred twenty five titles, conference paper 135 titles, e-book 1 title and overview 1 name. Within the research articles, best constrained researchers centered on the improvement of CFD technique. Almost all the articles awareness on using CFD in a selected software, as a research device, including plane utility, wind strength and warmth exchanger. The topics of the one hundred twenty five studies articles divided into 12 precise packages and 1 miscellaneous application. The maximum famous application is Heating Ventilating and air con and observed via Reactor, Transportation and warmth Exchanger programs. The maximum famous business CFD code used is ANSYS Fluent and only several researchers' use CFX.

III. METHODOLOGY

1. Problem Statement- To increase the overall performance of the annular diffuser, the velocity of the inlet fluid must be reduced and static pressure of fluid should be increased. The performance of this diffuser depends on their geometrical dimensions of diffuser and inlet conditions. Thus, with the help of CFD on a annular curved diffuser of 7^{o} angle of turn, circular hub of diameter to 20mm and also by varying the fins on circular hub of height to 5mm and 10mm with a thickness of 3mm and allowing the air as fluid to pass through the diffuser based on the properties of the diffuser material and the analytical analyses on the design.

2 Solution-

The problem is solved in two steps
Modelling in CAD Software
CFD Analysis in FEA Software

In this thesis, annular curved diffuser is modelled by using Creo Parametric 6.0. The references for the models are taken from journals specified as [6] [7], in References chapter. Annular curved diffusers are modelled by changing angle of turn 7^{0}. Modifications are done by adding fins to the models.

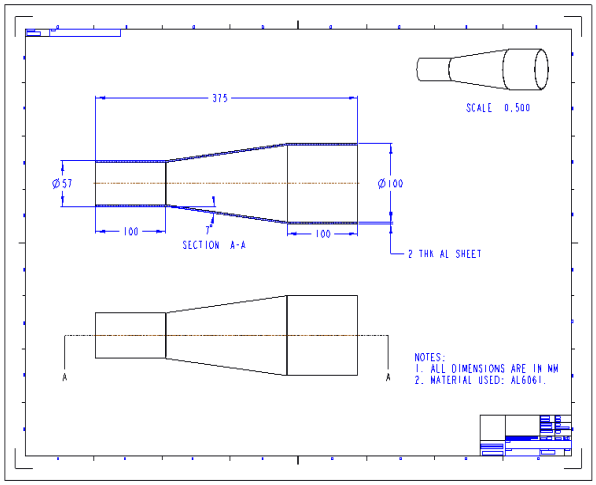


Figure 1 Basic geometry of annular curved diffuser

3. MODELLING IN CAD SOFTWARE:

Model of Annular curved diffuser without fins designed in CREO 6.0 is shown in below figure 2,

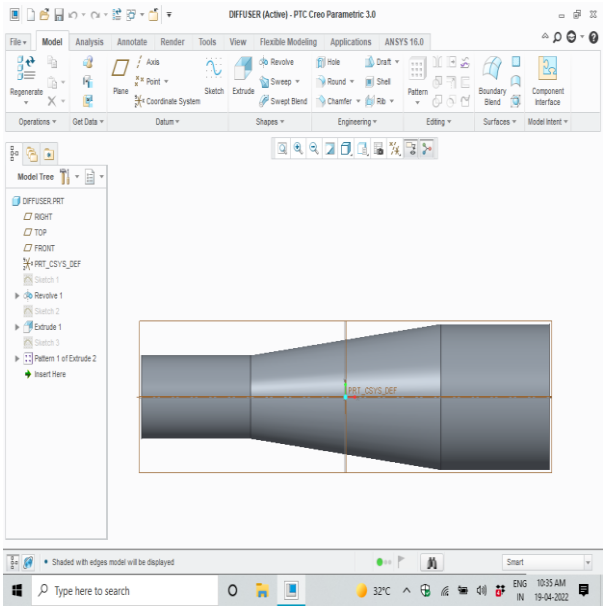


Figure 2 – Annular curved diffuser without fins

Model of Annular curved diffuser with fins (5mm) designed in CREO 6.0 is shown in the figure 3,

International Journal of Scientific Research & Engineering Trends
Volume 8, Issue 5, Sep-Oct-2022, ISSN (Online): 2395-566X

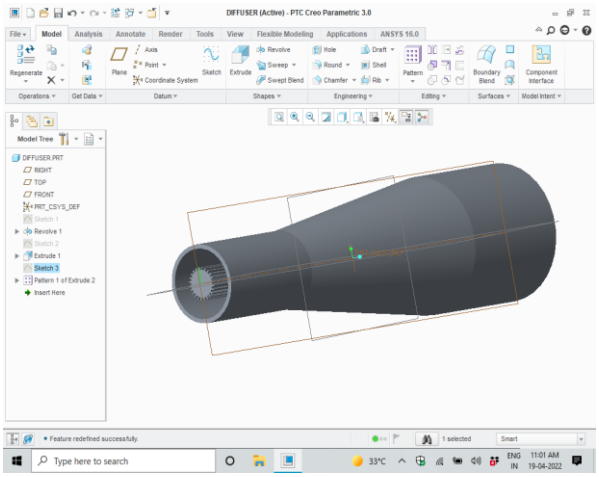


Figure 3 Annular curved diffuser with fins (5mm)

Model of Annular curved diffuser with fins (10mm) designed in CREO 6.0 is shown in the figure,

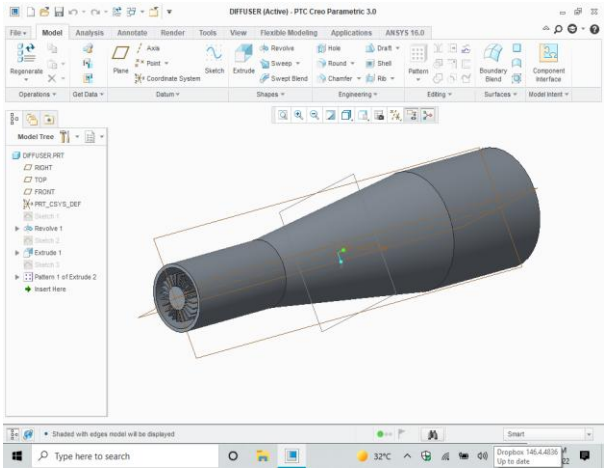


Figure 4 – Annular curved diffuser with fins (10mm)

4. Boundary Conditions

Material: Aluminium 1050A
Mesh Model Solver type: Pressure-based
Problem model: Standard k- ε
Fluid: Air
Velocity: 47m/sec
Outlet gauge pressure: 0 Pa
Wall condition: stationary and no slip condition
Reynolds number: 2.45 x 10^5
Standard k-ε model is used for all model analysis. Air is the working fluid. At the inlet, constant velocity condition is given for all model analysis and ambient pressure is for the exit boundary condition. Stationary and No-slip conditions are assumed on the wall of all models.

53 Meshing

The model is meshed by automated method for meshing shown in the figure 3.8 to get better and appropriate results.

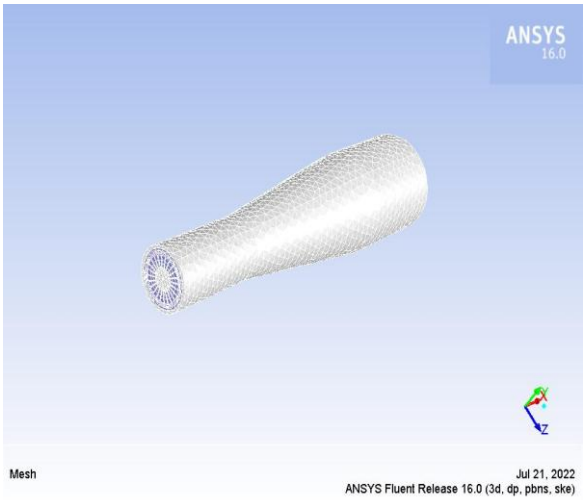


Figure 5 Meshing of Diffuser

.

6. Boundary Conditions

The Inlet and Outlet walls are fixed (constant) and we are assuming air as passing fluid in the diffuser, inlet and outlet named in ansys as shown in figure 6 and 7.

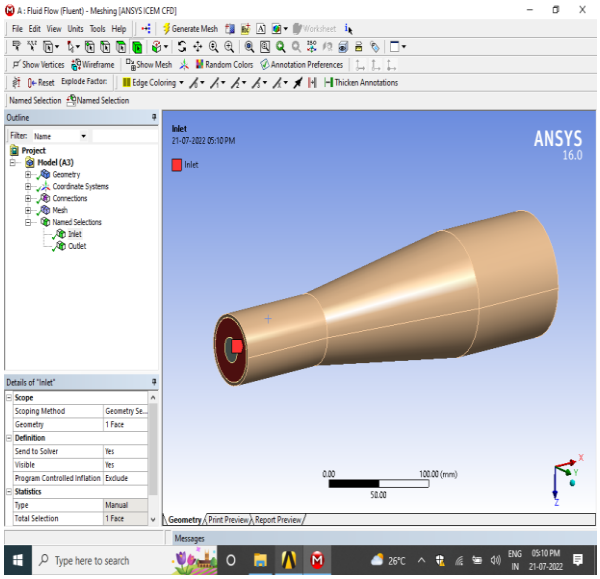


Figure 6 Inlet of annular curved diffuser

© 2022 IJSRET

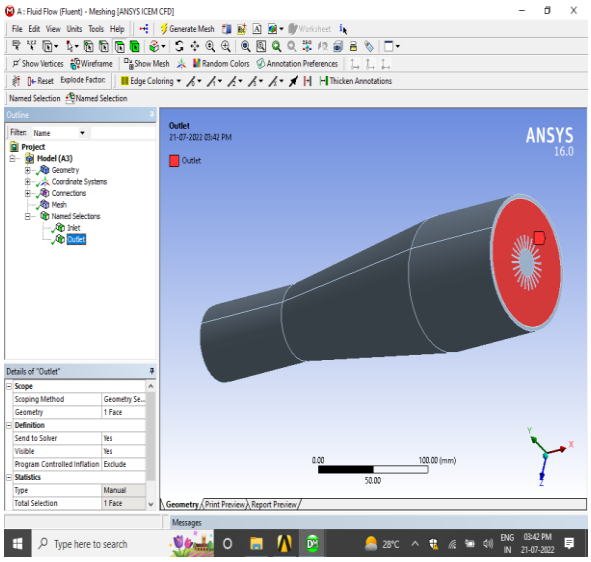


Figure 7 Outlet of annular curved diffuser g.

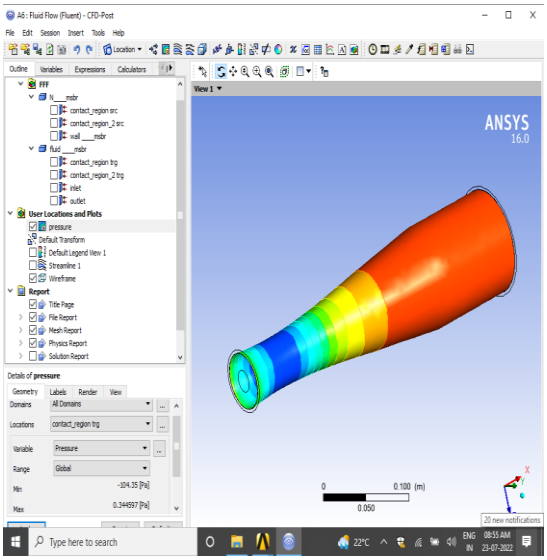


Figure 8 -Pressure contours of annular curved diffuser with hub

IV. RESULTS

The three dimensional annular curved diffuser designed in Creo software and imported to ansys CFD. The modifications at the primary geometry had been investigated to optimize the overall performance of the diffuser. First a model for the base diffuser changed into advanced by means of taking its geometric records from literature and the overall performance statistics function a reference for evaluating the performance of the changed. So that to find the optimal overall performance results of the annular diffuser geometric parameters has been numerous and those consequences are projected below. It's far assumed that the flow is exhausted to environment, so pressure at exit of diffuser is believed to be atmospheric.

1. Pressure Contours

Figures 8 to 10 show the pressure contours of all geometric models for different circular hub without and with fins 5mm and 10mm. In the pressure contour the colour code indicates the pressure variation from inlet of diffuser to outlet. From all the pressure contours initially the pressure is less due to the high velocity at inlet. As the fluid is flowing along the diffuser the pressure gradually increases due to decrease in velocity. Inserting the circular hub with different fin sizes on the circular hub of the diffuser it gives more pressure recovery than the basic annular curved diffuser. From all the pressure contours it is observed that when the hollow section starts the pressure drops and then it recovers as the geometry increases

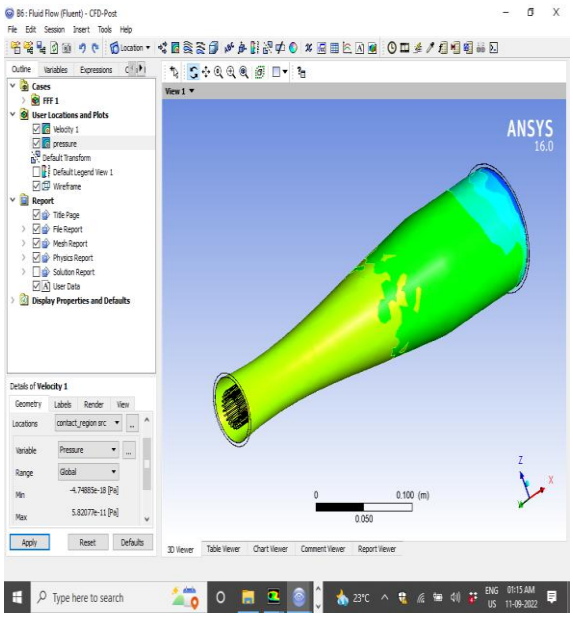


Figure 9 -Pressure contours of annular curved diffuser with hub and fins of 5mm

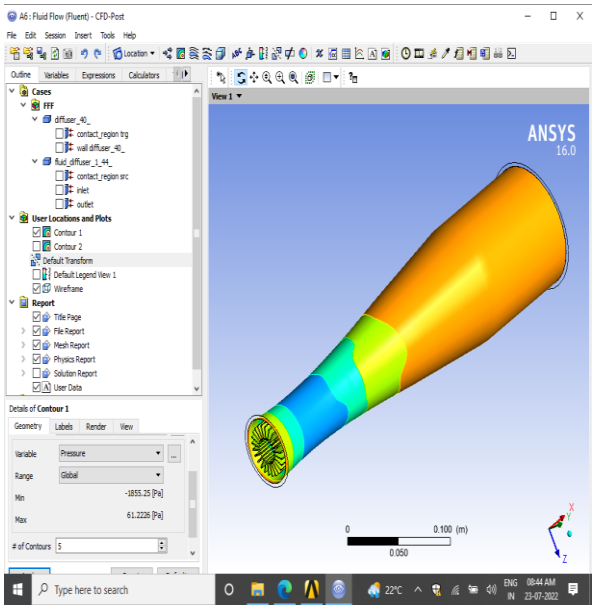


Figure 10 -Pressure contours of annular curved diffuser with hub and fins of 10mm

2. Velocity Contours

Figures 11 to 13 Shows the velocity contours of all annular curved diffuser models. In the velocity contours the colour code indicates the variation of velocity in the diffuser. From the below figures it is observed that velocity uniformly varies at inlet of the diffuser, when the hollow hub starts velocity increases gradually but at the outlet it again decreases due to increase in area towards the outlet of the diffuser.

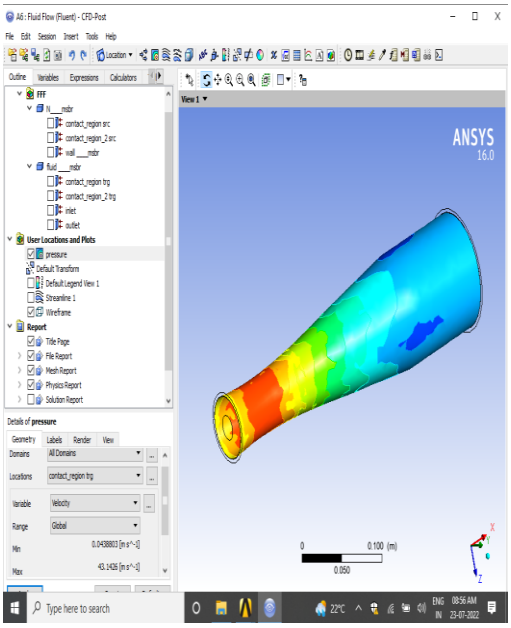


Figure 11 Velocity contour of annular curved diffuser with hub

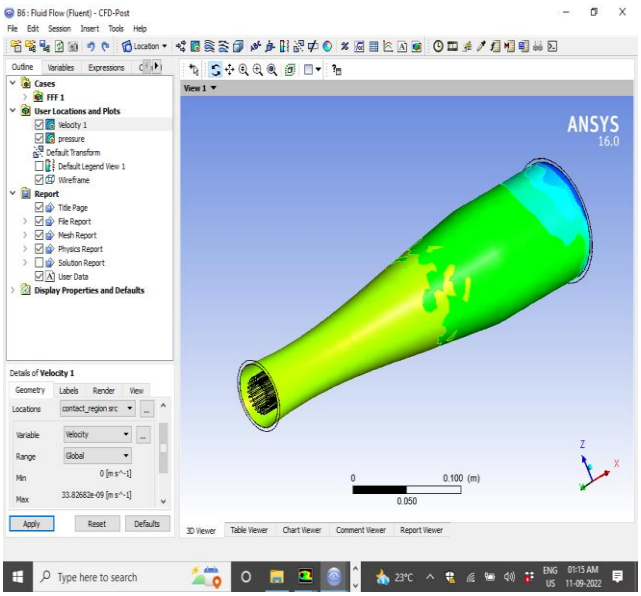


Figure 12 –Velocity contour of annular curved diffuser with hub and fins of 5mm

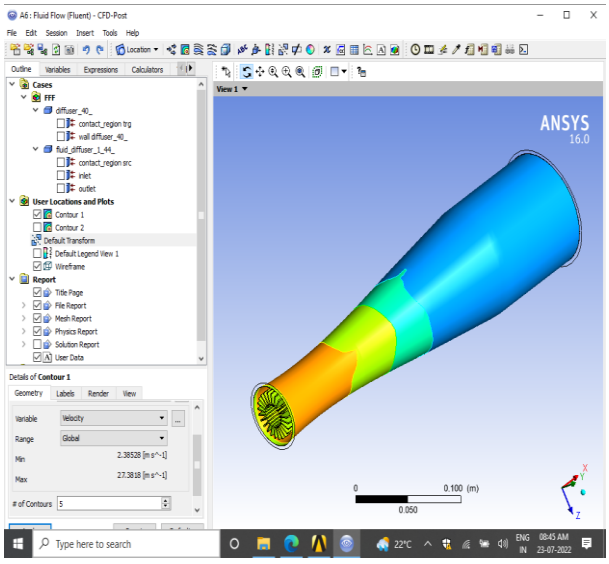


Figure 13 Velocity contour of annular curved diffuser with hub and fins 10mm respectively

Analysis Report Of Diffusers

Table 1 Result sheet of Annular curved diffuser.

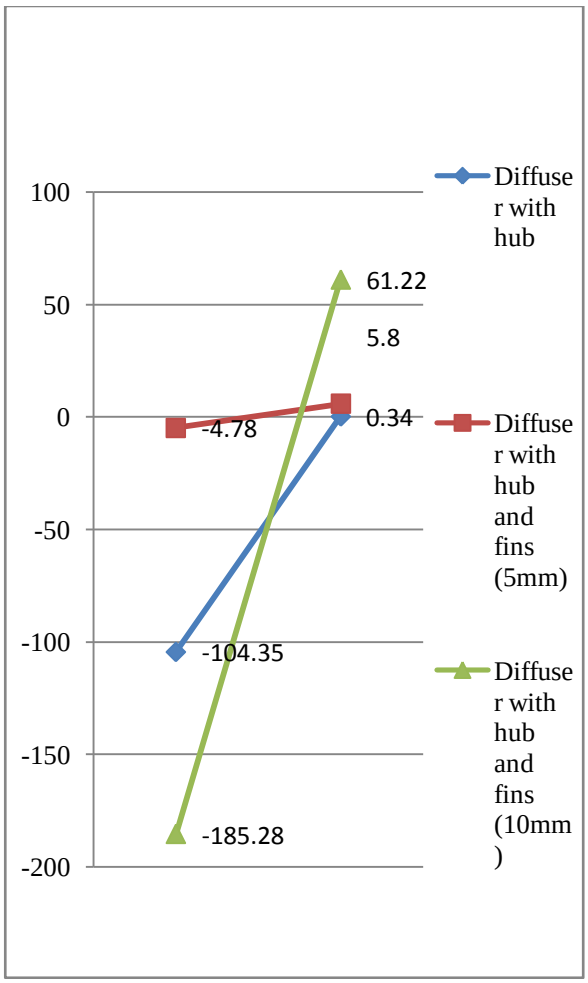
Model	Pressure (Pa)		Velocity (M/Sec)	
	Min	Max	Min	Max
Diffuser with hub	-104.35	0.34	0.043	43.14
Diffuser with hub and fins(5mm)	-4.78	5.8	0	33.28
Diffuser with hub and fins(10mm)	-185.28	61.22	2.38	27.38

© 2022 IJSRET

3. Graph Comparisions

Comparison of Pressure values for Annual diffuser with hub and variation in fins heights of 5mm, 10mm:

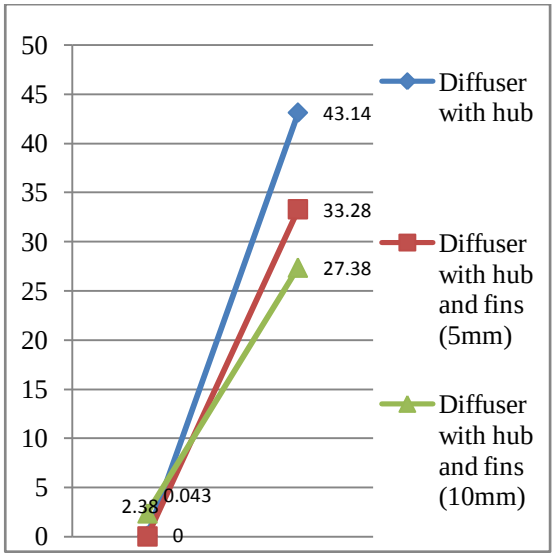
Graph 1 shows the comparison between pressure recovery annular curved diffuser with hub and fins 5mm and 10mm height. The graph predicts that annular curved diffuser with fin 10mm fin height gives the more pressure recovery than the other models.



Graph 1-Pressure recoveries for Annular curve diffuser with hub and different fin sizes (5mm, 10mm)

Comparison of Velocity for Annual diffuser with hub and variation in fins heights of 5mm, 10mm:

Graph 2 shows the comparison between velocity of air in the annular curved diffuser with hub and fins 5mm and 10mm height. The graph predicts that annular curved diffuser with fin 10mm fin height gives the more velocity than the other models.



Graph 2-Velocity for Annular curved diffuser with hub and different fin sizes (5mm, 10mm) font.

VI. CONCLUSION

Based upon the present investigation the following conclusions have been drawn. In the present investigation the CFD values almost matches with experimental results for basic curved annular diffuser model.

1. The modified model gives more static pressure than the basic annular curved diffuser.
2. By inserting fins on circular hub in the hollow section of the annular curved diffuser also gives more static pressure recovery.
3. The fin height of 10mm, diameter 20mm of circular hub of 7^0 angle gives the more static pressure (61.22Pa) recovery than the other annular curved diffusers.
4. The fin height of 10mm, diameter 20mm of circular hub of 7^0 angle reducing the velocity of air from 47m/sec to velocity 27.38m/sec which is less than the other annular curved diffusers.
5. In this analysis observed that as the height of fin decreases the static pressure recovery goes on decreases and total pressure loss is also decreases.
6. From the present investigation it is concluded that inserting the fins on circular hub inside hollow section of the annular curved diffuser gives more satisfactory values of velocity and static pressure recovery than the basic annular curved diffuser model

REFERENCES

[1]. Sangappa Hadapad, Basavaraj Dotihal, Analysis of flow on annular curved diffuser by using CFD, International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308, Volume: 06 Issue: 02 | Feb-2017

© 2022 IJSRET

[2]. Mohammed Maqsood, Kailash B A, Gowreesh Subramanya, Design And Optimization Of Flow Performance For Annular Curved Diffusing Duct Using CFD, International Research Journal of Engineering and Technology (IRJET), Volume: 02, Issue: 09, Dec-2015.
[3]. Raghul., Danushkotti., Nanda Gopal, Flow analysis of effect of struts in an annular diffuser, International Journal of Advanced Research in Biology Science and Technology, Vol. 2, Special Issue 13, April 2016
[4]. P. K. Sinha, A.K.Das , B. Majumda, Numerical Investigation of flow through Annular Diffusing Duct, International Journal of Engineering & Technology, Volume- 11, Issue- 03, June 2011.
[5]. O.H. Jacobsen, Chevalier. P, A Preliminary study of annular diffusers with constant diameter outer walls, THE American society of Mechanical Engineers, 1983.
[6]. R.KEERTHANA, G.JAMUNA RANI / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 3, May-Jun 2012, pp.2348-2351 2348 | P a g e "flow analysis of annular diffusers"
[7]. S.N Singh, V. Seshadri, Sunil Chandel, Mahendra Gaikwad."Analysis of the Improvement in Performance Characteristics of S-Shaped Rectangular Diffuser by Momentum Injection using CFD" Engineering Applications of Computational Fluid Mechanics Vol 3,No.1, pp. 109-122 (2009).
[8]. Ram Kumar Raman, Yogesh Dewang and Jitendra Raghuwanshi. "A review on applications of computational fluid dynamics", International Journal of LNCT, Vol 2(6)ISSN (Online): 2456-9895 137 .
[9]. Ambarita H, Siregar M R , Kishinami K , Daimaruya M , Kawai H. "Application of CFD in Indonesian Research: A review", 5th International Seminar of Aerospace Science and Technology,IOP Conf. Series: Journal of Physics: Conf. Series 1005 (2018) 012014.
[10]. www.wikipedia.com
[11]. www.google .com

© 2022 IJSRET