Industrial Engineering Journal
ISSN: 0970-2555
Volume : 50, Issue 7, No. 1, July 2021

CONICAL EXHAUST ON DIFFERENT USERS CFD AND THERMAL STUDY USING ANSYS ANALYSIS MODULES

[1]S. S. RAVISHANKAR, [2]V.VARUN KUMAR, [3]SK. ZAKIRHUSSIAN, [4]A.BHARGAV
[1,2,3,4]Dept of Mechanical Engineering, Sree Venkateswara College Of Engineering, Nellore (Dt), Andhra Pradesh, India.

Abstract
In a fluid machine, such as a gas turbine, the exhaust diffuser recovers static pressure by slowing the flow and transforming kinetic energy into pressure energy. This makes it an essential part of a turbo machine's environment and a key factor in determining the performance of a turbo machine. As a result, if the diffuser design is improved for efficient pressure recovery, the fluid machine's efficiency can be increased. Based on the findings of a computational fluid dynamics (CFD) research on diffusers with different half-cone angles, the form that gave the greatest pressure recovery was selected. The diffuser's optimal form was then manufactured and tested. CFD analysis to calculate mass flow rate, heat transfer coefficient, pressure drop, and velocity, and heat transfer rate for various conical exhaust diffusers (rectangular, circular, and hexagonal), conical exhaust diffuser models modeling using CREO parametric software, and analysis in ANSYS software for different conical exhaust diffusers (rectangular, circular, and hexagonal). CFD and thermal study of onical exhaust diffusers using ANSYS analysis modules
Keywords: CFD, shapes, thermal analysis, pressure drop and ANSYS.

INTRODUCTION
A blown diffuser is a device that modifies diffuser airflow by interacting with exhaust gases. The definition of a diffuser is "a device for enhancing the static pressure while lowering the velocity of a fluid passing through a system." Diffusers are used to increase static pressure while reducing fluid flow. The phrase "pressing factor recovery" refers to how the static pressing factor of a liquid rises as it passes through a pipe. However, a spout is designed to increase release speed, decrease pressure, and specifically coordinate the stream.
Despite the fact that investigations might have considerable frictional consequences, these effects are typically disregarded. Typically, Bernoulli's rule can be used to inspect pipes carrying low-speed liquids. When investigating conduits that are streaming at higher rates with mach values greater than 0.3, compressible stream relations are frequently used.

Supersonic Diffusers
A supersonic diffuser is a duct that shrinks in size as it moves in the flow direction. Fluid temperature, pressure, and density rise as the duct gets smaller, while velocity falls. Compressible flow is the reason for these variations in temperature, density, pressure, and velocity. Shockwaves may also serve a vital purpose in a personic diffuser. There are many different forms available for diffusers, such as circular, rectangular, and linear slot diffusers (LSDs, for instance). The components of linear slot diffusers are one or more long, narrow slots, which are typically partially concealed in a fixed or suspended ceiling.
[1] Diff users are occasionally employed in the other direction, as air inlets or returns. This is especially true for' perf' and linears lot diffusers. Grilles are most typically utilized as return or exhaust air inlets. A gas turbine

engine's divergent exhaust diffuser is a critical component. It essentially lowers the fluid velocity that exits the low-pressure turbine stage and raises the static pressure.

LITERATUREREVIEW

Singh et al. [2] directed a CFD examination of an annular exhaust diffuser with different shapes while keeping up with the gulf half cone point steady. At long last, they guaranteed that the exhibition of an annular exhaust diffuser with an equal wandering center and packaging outflanked twirl.

The pressing factor recuperation of an exhaust diffuser can be upgraded by raising the channel disturbance force,as per Hoffman[3].

The disturbance force of an exhaust diffuser can be changed to further develop the diffuser pressing factor recuperation. This paper proposed orchestrate on the construction of disturbance in a cone shaped exhaust diffuser by P A C Okwuobi [4]. They found that close to the edge of the divider layer, which stretches out to the mark of maximal change,the tempestuous energy rate arrives at a most extreme worth.The greatness of energy convective dissemination because of pressing factor and active impacts is equivalent to that of energy creation, as indicated by the fierce motor energy balance.

The fumes diffuser of a liquid machine, like a gas turbine, recuperates static pressing factor by decelerating the stream and transforming motor energy into pressure energy, as indicated by R.Prakash [5]. Accordingly, it is a fundamental segment in a turbo machine climate and has a significant impact in choosing a turbo machine's exhibition.Therefore, if the diffuser plan is upgraded for ideal pressing factor recuperation, the liquid machine's proficiency can be expanded.

The subsonic stream study is done in diffuser blenders with and without swaggers, as indicated by Parameshwar Banakar [6]. For the two situations, all out pressure misfortune, pressure acquire,and significant stream boundaries like Mach number,speed, statics pressing factor, and twirl are analyzed.The investigation was done utilizing a 45-degree are a model of the diffuser blender without swaggers and with swaggers,considering the math's periodicity.

In his examination, Venugopal M [7] tended to what the presentation of the fumes turbine diffuser means for the force and effectiveness of gas turbines. We should address the course through unsound contact with the turbine's high and low pressing factor rotational stages, which cause twirl stream, to foster high-proficiency diffuser gas turbine. A survey of the writing uncovers that there is opportunity for development as far as turbine execution. Since the diffuser is the focal point of the framework, it should manage variable levels of twirl. Twirl stream in the diffuser segment will cause issues like as pressing factor misfortune while streaming across wagers and a lessening in momentary stream. The higher the Reynolds number, the more fierce the stream becomes, which is unfortunate.Reynolds'digit. At the point when mathematical discoveries are contrasted with exploratory outcomes, unmistakably the aftereffects of single-stage investigation are very near the trial results.

MODELINGAND ANALYSIS

CREOis3-DdemonstratingPCprogrammingutilized in mechanical designing, plan, assembling,and CAD drawing administration associations. It was recently known as expert/ ENGINEER. It was one of the soonest three-dimensional CAD displaying applications that utilization a standard based parametric framework.It might streamline the development item just as the actual plan by utilizing boundaries, measurements, and abilities to catch the item's practices. In 2010, the choice to change from genius/ ENGINEER Wildfire to CREO was made. It became made accessible via the organization that made it, Parametric Science Corporation (PTC),sooner or later during the dispatch of its set-up of plan items, which incorporates applications like get together displaying, second orthographic perspectives for specialized drawings, limited detail examination, and others.As a result of nitty-gritty abilities, which contains the mix of parametric and direct displaying in solitary stage, P.C.CREO

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 50, Issue 7, No. 1, July 2021

claims it will give a more noteworthy powerful format information than exceptional demonstrating programme. T The whole scope of capacities covers the whole range of item advancement, giving originators alternatives at each phase of the interaction.The application programming likewise offers a superior client decent interface, which gives creators a superior encounter.It likewise includes cooperation capacities that simplify it to share and adjust plans.

Fig1:circular type

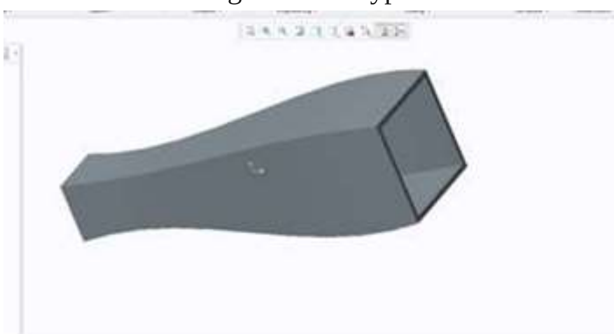

Fig:2 rectangular type

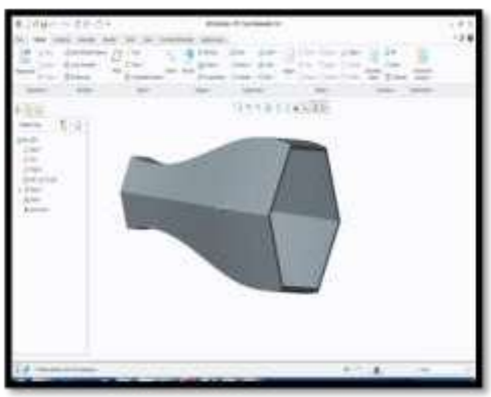

Fig: hexagon type

CFD

Computational Fluid dynamics(CFD) is a part of liquid mechanics that settles and examinations issues including liquid streams utilizing mathematical strategies and calculations.The calculations important to display the communication of fluids and gases with surfaces characterized by limit conditions are performed on PCs. Better arrangements are conceivable with high velocity super computers.Momentum research is yielding programming that builds the exactness and speed of trouble some recreation situations like transoceanic or fierce streams.The underlying exploratory approval of such programming is completed in an air stream, with full-scale testing,for example,flight tests,giving a definitive approval.

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 50, Issue 7, No. 1, July 2021

METHODOLOGY

In ANSYS ICEM CFD, the math is worked for each model dependent on the given information, and a space is framed to incorporate the stream inside the are a to the body's dividers.To examine space freedom, three tube shaped areas are assessed utilizing an experimentation strategy that includes estimating distances between the model's nose and last parts and computing the span from the model's hub.

BOUNDARYCONDITIONS

Coming up next are the applicable limit conditions for registering the dissimilar exhaust diffuser:

Delta: The gulf boundary is bay speed, with a worth of 45m/s at the channel of a cone like exhaust diffuser.

The boundary pressure-outlet is characterized in the power source area, and the worth is set to 101325.Pascal.

The fixed divider with no slip condition is characterized as the divider.The unpleasantness tallness and harshness constants are additionally set to 0, suggesting a smooth surface.

The sort for the Inlet zone would be speed delta. A speed of 45 m/s and a temperature of 1773 K are the Velocity channel limit conditions. Fixed divider conditions are utilized for the line. The sort for the Outlet zone would be pressure outlet.Standard temperature and working pressing factor states of 101325 Pa are utilized to decide the pressing factor outlet limit conditions.

RESULTS AND DISCUSSIONS

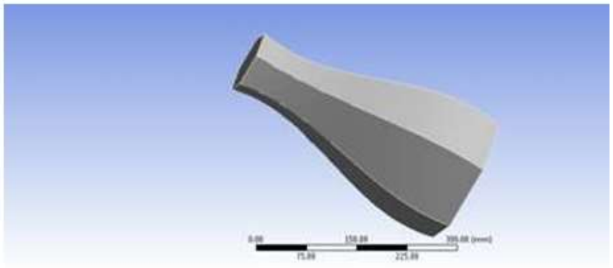

Fig:5 imported model

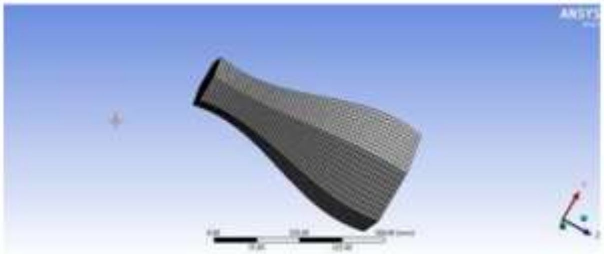

Fig:6 meshed model

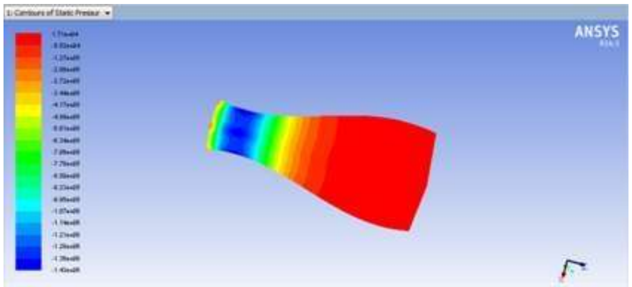

Fig:7 pressure drop

Industrial Engineering Journal
ISSN: 0970-2555
Volume : 50, Issue 7, No. 1, July 2021

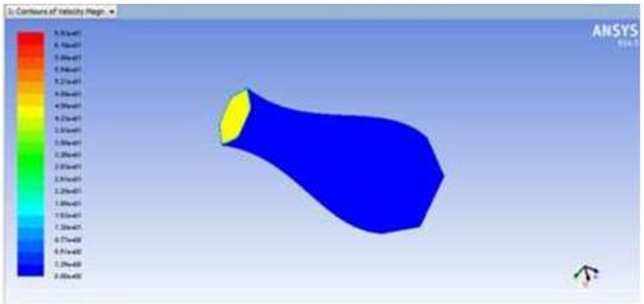

Fig:8 velocity counter

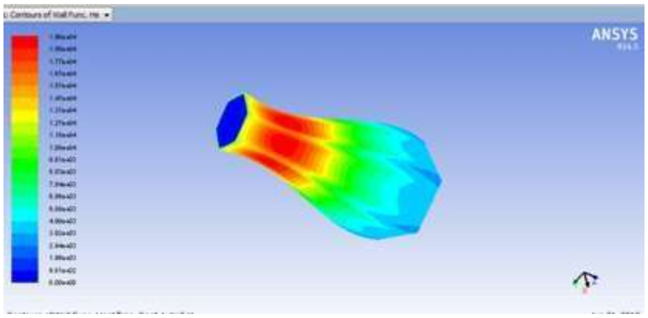

Fig:9 Heat transfer coefficient

THERMAL ANALYSIS OF CONICAL EXHAUST DIFFUSER

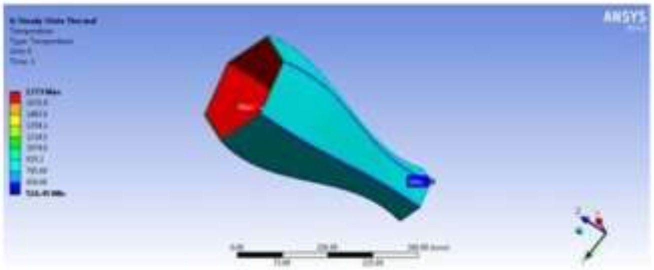

Fig:10 temperature distribution

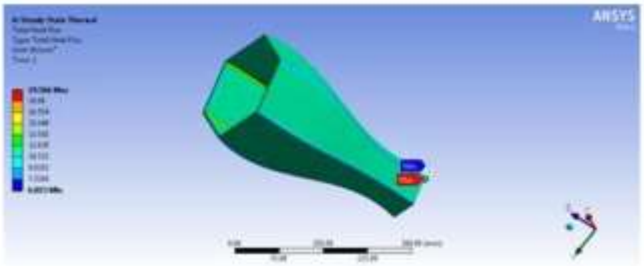

Fig11:heat flux

Result tables

CFD analysis result stable

Conicalexhaustdiffusermodels	Pressure (Pa)	Velocity(m/s)	Heattransfercoefficient (w/m2-k)	Massflowrate (kg/s)	Heattransferrate(W)
Circular	1.70e+04	4.66e+01	1.52e+04	0.1629	490752
Rectangular	2.24e+04	5.74e+01	1.74e+04	0.10063	304288
hexagonal	1.71e+04	6.51e+01	1.96e+04	0.3409	1033152

Thermal analysis result table

Models	**Materials**	**Temperature(K)**		**Heatflux(w/mm2)**
		Max.	Min.	
Circular	**Steel**	1773	888.41	11.878
	copper	1773	1509.8	23.187
Rectangular	**Steel**	1773	426.4	20.563
	copper	1773	1166.6	52.772
Hexagonal	**Steel**	1773	516.45	19.566
	copper	773	1260.4	47.894

CONCLUSION

The following conclusions may be taken from the examination of the various exhaust diffusers. The modeling was done using CREO Parametric 3.0 software. The thermal analysis is done in ANSYS Workbench. The results were then converted to plots and contours using FLUENT's post processing interface. Different diffuser forms were computationally analyzed, and the findings were used to calculate the co-efficient of pressure recovery. The velocity plots and contours exhibit a completely different pattern as a result of the conversion of kinetic energy into pressure energy. Additionally, the center line velocity is greater than the border velocity due to friction effects at the boundary layer. It was discovered that improving the turbine's pressure, velocity, heat transfer coefficient, mass flow rate, and heat transfer rate were shown to improve turbine efficiency and performance when utilizing this type of diffuser.The heat transfer coefficient and heat transfer rate values for hexagonal type conical exhaust diffusers are higher based on the CFD analysis results. The heat flux value is highest at copper material, according to the thermal analysis. As a result, the copper material is a preferable choice for a conical exhaust diffuser.

REFERENCES

[1] R. Prakash, D. Christopher, K. Kumarrathinam,"CFDAnalysisofmovethroughataperedfumesdiffuser",InternationalJournalofResearchinEngineering and Technology, Volume 3, Issue 11,NCAMESHE-2014.

[2] ParameshwarBanakar,Dr.Basawaraj,"ComputationalAnalysisofstream inmaxenginepropulsion diffuser blender having various states ofswaggers",InternationalJournalofEngineeringResearch,Vol. 3, Issue 6, 2015.

[3] Venugopal M, Somashekar V, "Plan and AnalysisofAnnularExhaustDiffuserforJetEngines",InternationalJournalofInnovativeResearchinScience, Engineering and Technology,Vol. 4, Issue7, July2015.

[4] NikhilD.Deshpande,SuyashS.Vidwans,Pratik R.Mahale,RutujaS.Joshi,K.R.Jagtap,"HypotheticalandCFDAnalysisOfDeLavalNozzle",InternationalJournalofMechanicalAndProduction Engineering, ISSN: 2320-2092, Volume-2,Issue-4,April-2014.

[5] Ali Asgar S. Khokhar, Suhas S. Shirolkar, "PlanAnd Analysis Of Undertray Diffuser For A FormulaStyleRacevehicle",InternationalJournalofResearch in Engineering andTechnology, Volume:04 Issue: 11|Nov-2015

[6] ManojKumarGopaliya,PiyushJain,SumitKumar,VibhaYadav,SumitSingh,"ExecutionImprovementofS-moldedDiffuserUsingMomentum Imparting Technique", IOSR Journal ofMechanical and Civil Engineering, Volume 11, Issue3 Ver. I(May-Jun. 2014),PP23-31.

[7] MasafumiNakagawa,AtsushiHarada,"Investigation of Expansion Waves Appearing in theOutlets of Two-Phase Flow Nozzles", InternationalRefrigerationandAirConditioningConferenceatPurdue, July14-17, 2008.

[8] K.M. Pandey, Member IACSIT and A.P. Singh,"CFDAnalysisofConicalNozzleforMach3at Various Angles of Divergence with Fluent Software",IJCEA,Vol.1, No.2, August2010.

[9] Vinod kumar S Hiremath, S. Ganesan, R. Suresh,"PlanAndAnalysisOfExhaustDiffuserOfGasTurbineAfterburnerUsingCFD",Proceedingsof27thIRFInternationalConference,26thJune, 2016.

[10] SanthoshKumarGugulothuandShaliniManchikatla, "Exploratory and Performance AnalysisofSingleNozzleJetPumpwithVariousMixingTubes", International Journal of Recent advances inMechanicalEngineering(IJMECH)Vol.3,No.4,November 2014.