International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

IJIEMR Transactions, online available on 16th Sept 2019. Link :http://www.ijiemr.org/downloads.php?vol=Volume-08&issue=ISSUE-09

Title **DESIGN AND ANALYSIS OF AERODYNAMICS NOZZLE FOR MAXIMUM THRUST USING MULTIDISCIPLINARY DESIGN OPTIMIZATION**

Volume 08, Issue 09, Pages: 684–692.

Paper Authors

T.V.D SAI RAMKRISHNA, ANIL KUMAR.G

G V R & S COLLEGE OF ENGINEERING & TECHNOLOGY - [GVR&S], *GUNTUR*, ANDHRA PRADESH

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

DESIGN AND ANALYSIS OF AERODYNAMICS NOZZLE FOR MAXIMUM THRUST USING MULTIDISCIPLINARY DESIGN OPTIMIZATION

[1]T.V.D SAI RAMKRISHNA, [2]ANIL KUMAR.G
[1]M-TECH SCHOLAR, DEPARTMENT OF MECHANICAL ENGINEERING IN *G V R & S COLLEGE OF ENGINEERING & TECHNOLOGY* - [*GVR&S*], *GUNTUR*, ANDHRA PRADESH
[2]ASST PROFESSOR DEPARTMENT OF MECHANICAL ENGINEERING IN *G V R & S COLLEGE OF ENGINEERING & TECHNOLOGY* - [*GVR&S*], *GUNTUR*, ANDHRA PRADESH

ABSTRACT

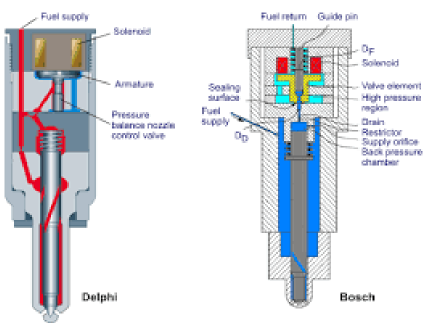
A multidisciplinary analytic model of a aero spike rocket nozzle has been developed, this model includes predictions of nozzle thrust, nozzle weight, and effective vehicle gross-liftoff weight (GLOW). The linear aero spike rocket engine is the propulsion system proposed for the X33. The model has been developed to demonstrate multidisciplinary design optimization (MDO) capabilities for relevant engine concepts, assess performance of various MDO approaches, and provide a guide for future application development.This paper details the multidisciplinary design optimization (MDO) of a strut-braced wing aircraft and its benefits relative to the cantilever wing configuration. The multidisciplinary design team is subdivided into aerodynamics, structures, aero elasticity and synthesis of the various disciplines. The aerodynamic analysis consists of simple models for induced drag, wave drag, parasite drag and interference drag. The interference drag model is based on detailed computational fluid dynamics (CFD) analyses of various wing-strut intersection flows. The wing structural weight is partially calculated using a newly developed wing bending material weight routine that accounts for the special nature of strut-braced wings. The remain-ing components of the aircraft weight are calculated using a combination of NASA's Flight Optimization Sys-tem (FLOPS) and Lockheed Martin Aeronautical System formulas. The strut-braced wing and cantilever wing con-figurations are optimized using Design Optimization Tools (DOT). CFD analysis to determine the pressure drop, velocity, heat transfer coefficient, mass flow rate and heat transfer rate for different aerodynamic nozzle (rectangular, circular and hexagonal), the aerodynamic nozzle models modeling using CREO parametric software and analysis in ANSYS software. In ANSYS using analysis modules for aerodynamic nozzle CFD & thermal analysis
.

INTRODUCTION

The primary challenges towards developing new diesel engines for traveler cars be the strict future emission legislation together with the customer's demands for steady rising performance. for instance, the emission limitations of Tier a pair of Bin

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

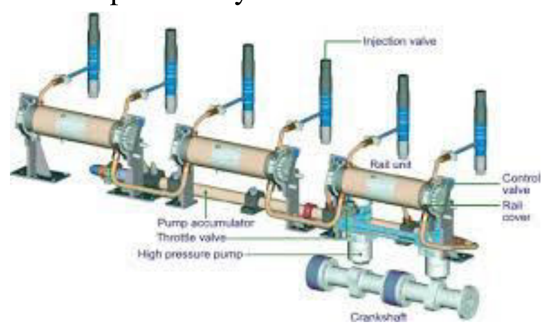
five needs a complicated once treatment system and a sturdy combustion method that minimizes emissions within the method of them being shaped. Advancements within the technology of Diesel Injection (DI) systems have contend in necessary role within the enhancements that are created up to the current purpose . Combining the reduction in nozzle passage diameters through increased flow characteristics with inflated injection pressures provides a chance to develop engines that includes high power density and reduced emissions. the first downside to those fashionable spray hole geometries is that they typically suffer a discount of power output throughout long run operation. alternative studies have known these important formations of deposits because the main reason for this behavior.



Basic mechanisms are often wont to make a case for the formation and removal of deposits in burning engines These mechanisms act severally of the situation of shaped deposits (e.g. injection nozzles, heat changer) and of the combustion method (e.g. IDI, DI; diesel or gasoline).

Objectives

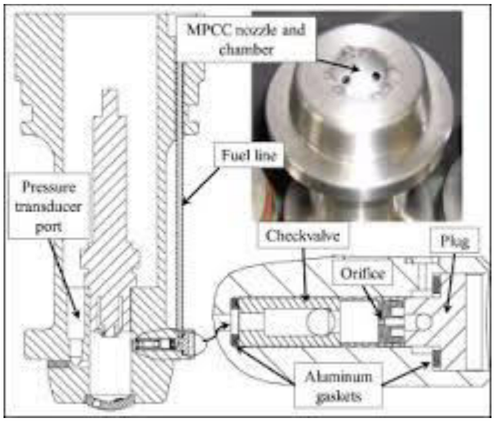
The primary goals of this study were to research the key causes of the formation of deposits within the spray hole and to ascertain the leading parameters that promote the formation and therefore the decay of deposits. the subsequent step was to seek out measures that will cut down or maybe inhibit the formation of those deposits. it absolutely was determined to divide the study into 2 sub-projects. the primary sub-project handled the experimental investigations, as well as a close deposit analysis.



The pay attention of the opposite sub-project was on simulating the results of assorted nozzle sorts (cylindrical, ks-nozzle) on heat transfer and fluid flow, utilizing coupled CFD- and thermal modeling. additionally, analysis was conducted on the thermal state of affairs of the nozzle, including a sensitivity analysis concerning the thermal conditions at the gismo tip. Cavitation and thermal effects were additionally enclosed as a vicinity of the investigation. This document highlights the experimental results, as well as the deposit analysis. The results of the opposite sub-project area unit part printed

EFFECTS OF DISEL NOZZLE

The four-stroke direct-injection diesel engine typical was measured and sculptured by Bakar et al victimization GT-POWER machine model and has explored of diesel performance result based mostly on engine speeds. GT-POWER is the leading engine simulation tool employed by engine and vehicle manufacturers and suppliers and is appropriate for analysis of a large vary of engine problems.



LITERATURE REVIEW

M. Volmajer et al [4] had numerical and experimental results of the nozzle fuel flow analysis for a four-hole injection nozzle Bosch DLLA 148 S 311376 are presented. The fuel flow coefficients obtained from the experimental results at steady flow conditions in the nozzle are compared with the results of the CFD analysis. The fuel flow coefficients obtained from the experimental results at steady flow conditions in the nozzle are compared with the results of the CFD analysis. From the presented results the following conclusions could be made. Flow coefficient testing device constructed at the ERL yields sufficiently precision, with reasonable uncertainties of the measurement. To refine the precision of the measurement, by defining the exact value of the pressure difference, the pressure downstream of the nozzle should be measured, or the nozzle position should be changed so, that the fluid would be injected directly into the measuring Plexiglas. For the same purpose, Plexiglas cylinder with high ovalness should be replaced with the glasslPlexiglas cylinder with proper circle cross-section

INTRODUCTION TO CAD:

Pc-aided design (CAD) is making use of computer buildings (or workstations) to useful resource within the appearance, amendment, analysis, or optimization of a structure. CAD application software is used to increase the productiveness of the trend dressmaker, give a boost to the nice of structure, fortify communications by way of documentation, and to create a database for manufacturing. CAD output is customarily inside the form of electronic records for print, machining, or distinct manufacturing operations.

INTRODUCTION TO CREO:

Present CREO, earlier known as professional/ENGINEER, is 3-D modeling program software applied in mechanical engineering, design, manufacturing, and in CAD drafting provider companies. It grew to become one of the vital first 3-d CAD modeling packages that used a rule-headquartered parametric gadget. Using parameters, dimensions and capabilities to grab the habits of the product, it might optimize the advance product furthermore to the design itself.

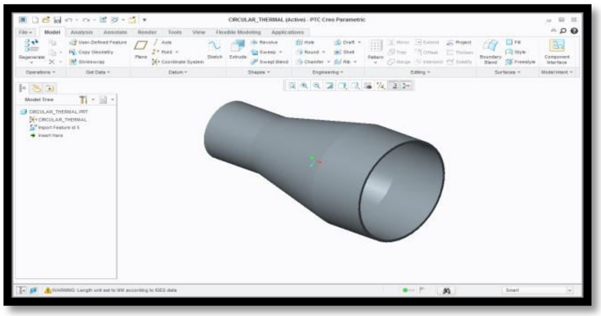
MODELING OF AN AERODYNAMIC NOZZLE

The modeling of an AERODYNAMIC NOZZLE is done in CREO Parametric 3.0 modeling software. The three cross-sections such as hexagonal, square and circular

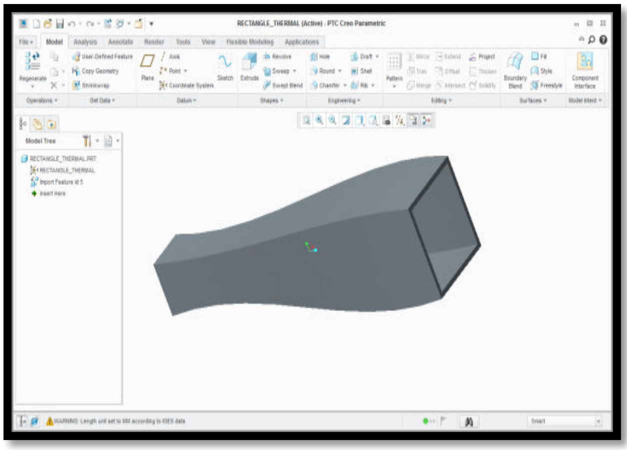
International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

models are taken. The model of a circular AERODYNAMIC NOZZLE

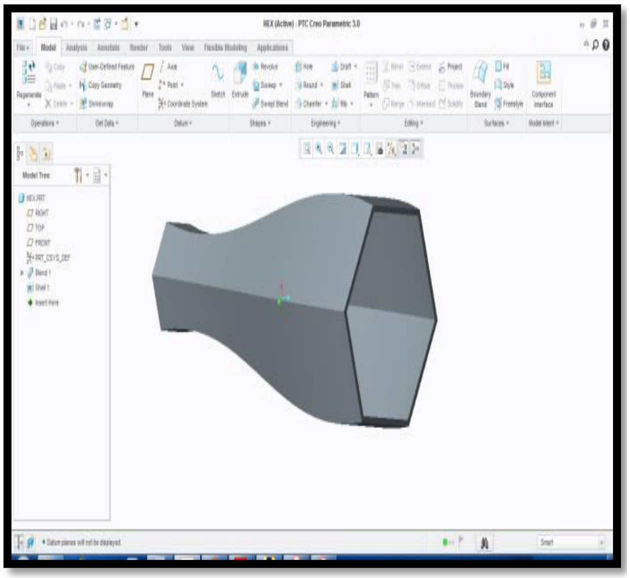
CIRCULAR TYPE



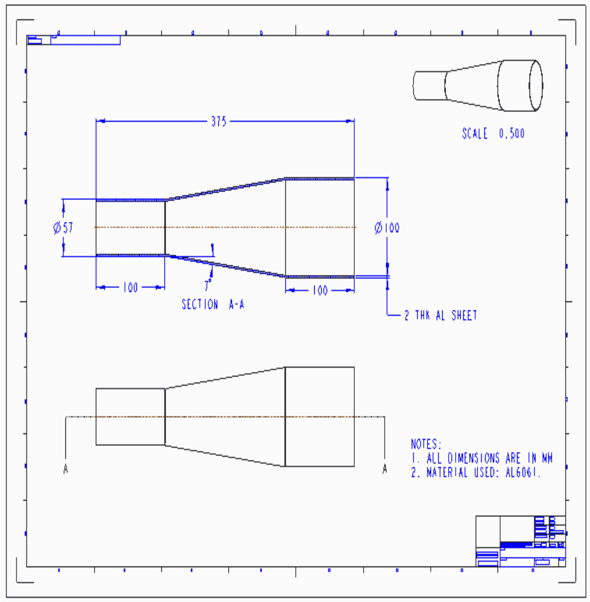
RECTANGULAR TYPE



HEXAGONAL TYPE



Model of a circular AERODYNAMIC NOZZLE the drawing specifications of a circular exhaust diffuse



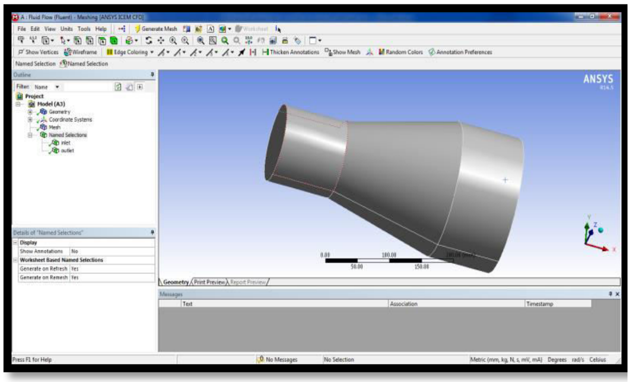
INTRODUCTION TO FEA:

Finite element evaluation is a method of fixing, ordinarily approximately, nice disorders in engineering and science. It's used specifically for troubles for which no actual resolution, expressible in just a few mathematical forms, is available. As such, it is miles a numerical as an alternative of an analytical procedure. Approaches of this variety are wanted because analytical techniques cannot care for the real, complex disorders which might be met with in engineering. For illustration, engineering force of drugs or the mathematical suggestion of elasticity can be utilized to calculate analytically the stresses and traces in a dishonest beam, nonetheless neither can also be very a success in finding out what's taking field in part of a car suspension device throughout the path of cornering.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

CASE; 1 CIRCULAR TYPE AERODYNAMIC NOZZLE

Imported model



Meshed model

MESHING

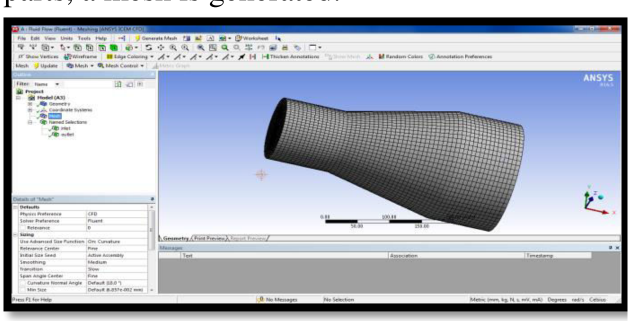
ü The model is created using ICEM CFD software.

ü The whole model is divided into different parts namely inlet, divergent inlet section, divergent exit section, outlet and wall.

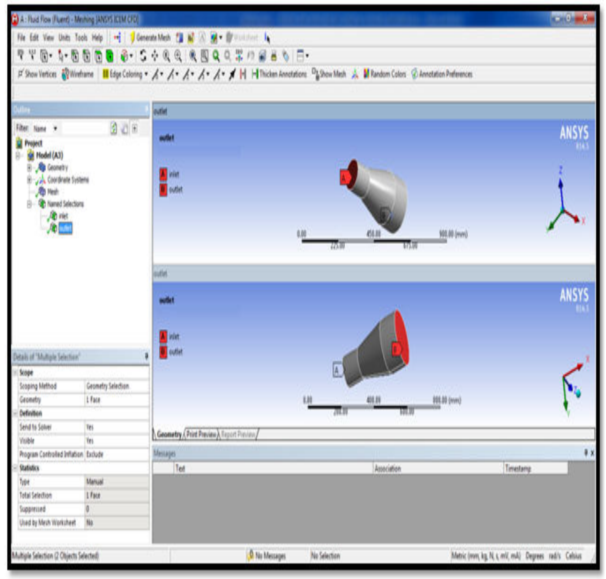
ü Global Mesh parameters are defined which gives information regarding type of mesh. The global element seed size, part parameters are setup and mesh is computed which gives the mesh information regarding total number of elements.

ü Anstructured hexahedral mesh is generated in order to perform computations with the Octree approach.

After setting up part parameters for various parts, a mesh is generated.



Boundary conditions



BOUNDARY CONDITIONS

The relevant boundary conditions for the computation of the divergent AERODYNAMIC NOZZLE are as follows:

ü Inlet: The inlet parameter is defined as inlet-velocity and the value at the inlet of a conical exhaust
diffuser set as 45m/s.

ü Outlet: At the outlet section the parameter is defined as pressure-outlet and the value is set as 101325
Pascal.

ü Wall: The stationary wall with no slip condition is defined. Also the smooth surface is assumed i.e the roughness height and the roughness constants are set to be zero For the Inlet zone, the type would be velocity inlet. The Velocity inlet boundary conditions include the velocity of 45 m/s and a temperature of 1773 K. For the boundary, stationary wall conditions are taken. For the Outlet zone, the type would be pressure outlet. The pressure outlet boundary conditions are taken for standard

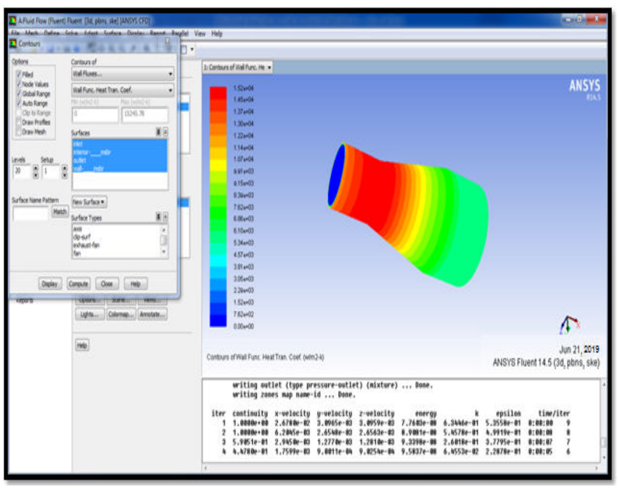
International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

temperature conditions and operating pressure conditions of 101325 Pa.

The boundary condition for the AERODYNAMIC NOZZLE

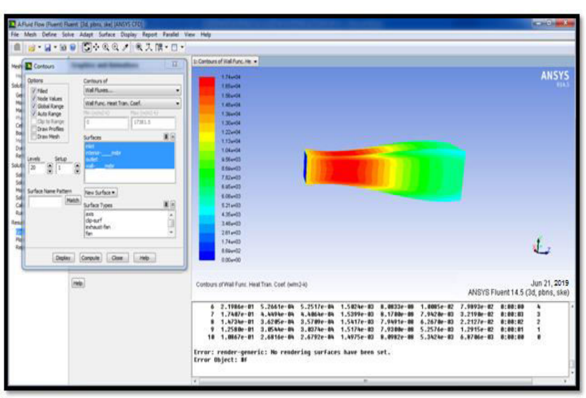
S. No	ZONE	TYPE
1	Inlet	Velocity Inlet
2	Outlet	Pressure outlet
3	In_inner_wall	Wall
4	In_outer_wall	Wall
5	Boundary	Wall

Heat transfer coefficient



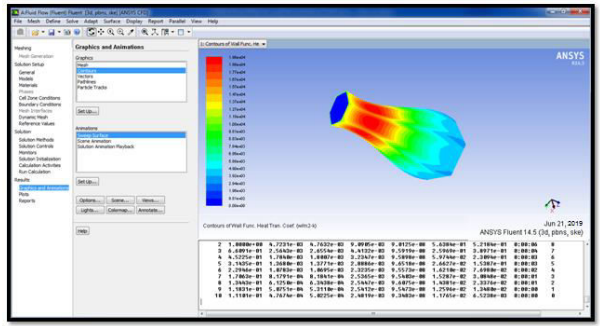
CASE; 2 RECTANGULAR TYPE AERODYNAMIC NOZZLE

Heat transfer coefficient



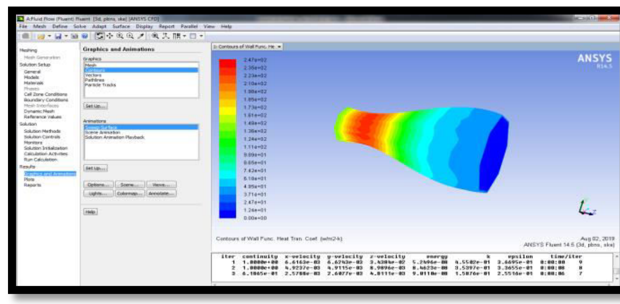
CASE; 3 HEXAGONAL TYPE AERODYNAMIC NOZZLE

Heat transfer coefficient



Case: 3 octagon type nozzle

Heat transfer coefficient

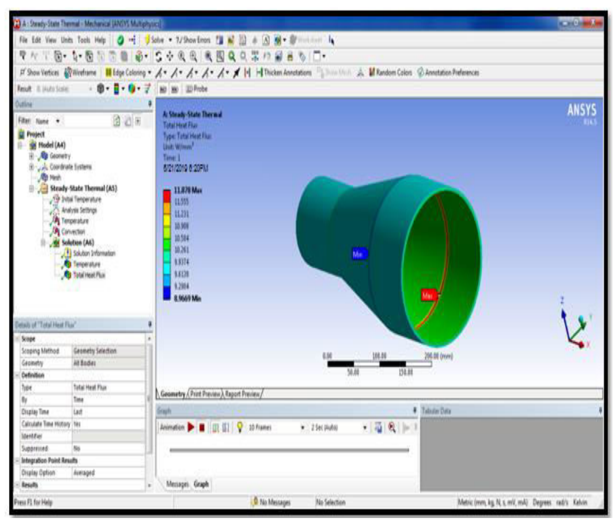


THERMAL ANALYSIS OF AERODYNAMIC NOZZLE

MATERIAL- STEEL

CASE; 1 CIRCULAR TYPE AERODYNAMIC NOZZLE

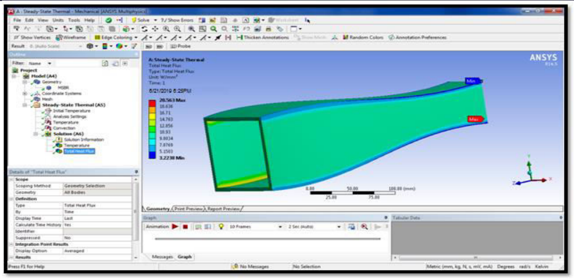
Heat flux



CASE; 2 RECTANGULAR TYPE AERODYNAMIC NOZZLE

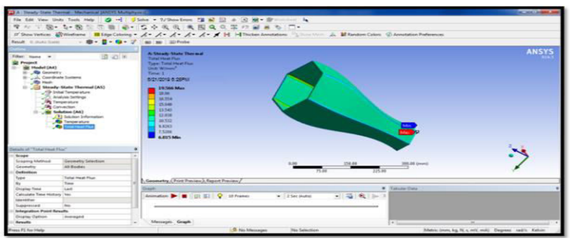
Heat flux

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org



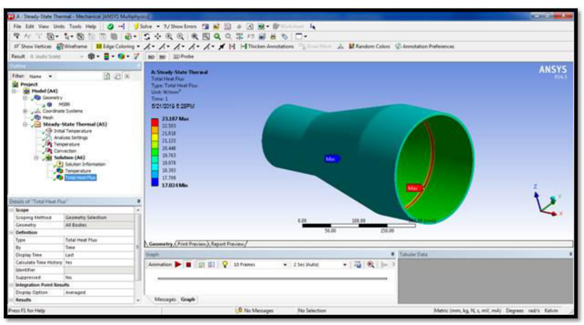
CASE; 3 HEXAGONAL TYPE AERODYNAMIC NOZZLE

Heat flux



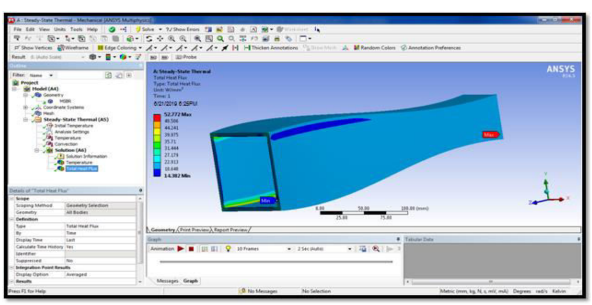
MATERIAL- COPPER

CASE; 1 CIRCULAR TYPE AERODYNAMIC NOZZLE



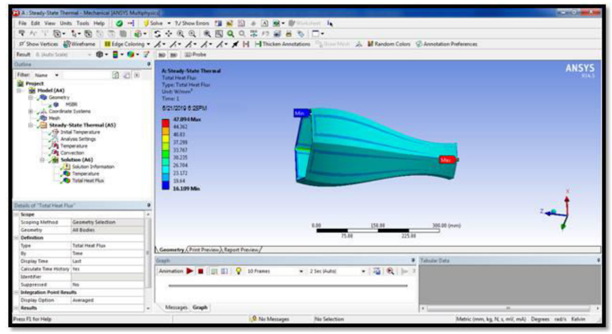
CASE; 2 RECTANGULAR TYPE AERODYNAMIC NOZZLE

Heat flux



CASE; 3 HEXAGONAL TYPE AERODYNAMIC NOZZLE

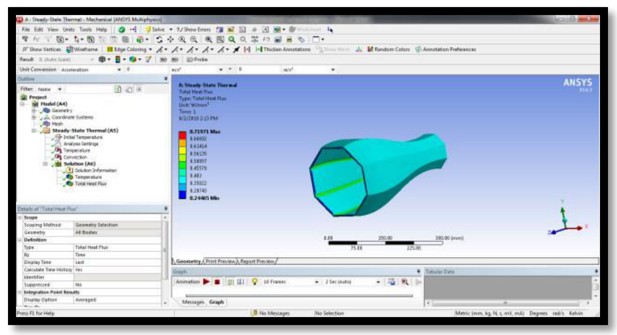
Heat flux



CASE; 4 OCTAGONAL TYPE AERODYNAMIC NOZZLE

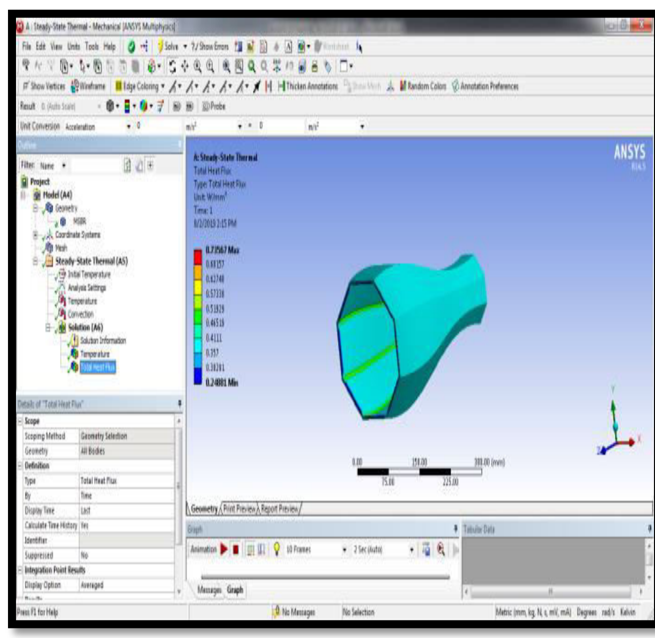
Material- steel

Heat flux



Material- copper

Heat flux



International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Result tables

CFD analysis results table

Aerodynamic nozzle models	Pressure (Pa)	Velocity(m/s)	Heat transfer coefficient (w/m2-k)	Mass flow rate (kg/s)	Heat transfer rate (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)		Heat flux(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 can be outlined by considering the analysis on different aerodynamic nozzles. The modeling is done in CREO Parametric 3.0 modeling software. The thermal analysis is performed in ANSYS workbench. The solutions obtained were then converted to plots and contours using the post processing interface of FLUENT. Computational analysis was performed on various shapes of diffusers and their co-efficient of pressure recovery were calculated using the data obtained. The velocity plots and contours depict an exactly opposite trend, owing to the conversion of kinetic energy into pressure energy. Also, it can be seen that the centerline velocity is higher than the velocity at the boundary due to friction effects at the boundary layer. It was found that the pressure, velocity, heat transfer coefficient, mass flow rate and heat transfer rate and using this type of diffuser we can improve turbine efficiency and turbine performance.By observing the cfd analysis results the heat transfer coefficient value and heat transfer rate values are more for hexagonal type aerodynamic nozzle. By observing the thermal analysis the heat flux value maximum at copper material. So it can be concluded the copper material is better material for aerodynamic nozzle.

REFERENCES

[1] R. Prakash, D. Christopher, K. Kumarrathinam, "CFD Analysis of flow through a AERODYNAMIC NOZZLE", International Journal of Research in

Engineering and Technology, Volume 3, Issue 11, NCAMESHE-2014.
[2] Parameshwar Banakar, Dr. Basawaraj, "Computational Analysis of flow in after burner diffuser mixer having different shapes of struts", International Journal of Engineering Research, Vol. 3, Issue 6, 2015.
[3] Venugopal M M, Somashekar V, "Design and Analysis of Annular AERODYNAMIC NOZZLE for Jet Engines", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 4, Issue 7, July 2015.
[4] Nikhil D. Deshpande, Suyash S. Vidwans, Pratik R. Mahale, Rutuja S. Joshi, K. R. Jagtap, "Theoretical & CFD Analysis Of De Laval Nozzle", International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 2, Issue- 4, April-2014.
[5] Ali Asgar S. Khokhar, Suhas S. Shirolkar, "Design And Analysis Of Undertray Diffuser For A Formula Style Race car", International Journal of Research in Engineering and Technology, Volume: 04 Issue: 11 | Nov-2015
[6] Manoj Kumar Gopaliya, Piyush Jain, Sumit Kumar, Vibha Yadav, Sumit Singh, "Performance Improvement of S-shaped Diffuser Using Momentum Imparting Technique", IOSR Journal of Mechanical and Civil Engineering, Volume 11, Issue 3 Ver. I (May- Jun. 2014), PP 23-31.