2581-4575

International Journal For Recent Developments in Science & Technology

Crossref

A Peer Reviewed Research Journal

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

[1] CHENNABOINA RAJESH KUMAR, [2]SK RESHMA BEGAM, [3]L. VIJAY

[1]Department of mechanical engineering M-Tech student (CAD/CAM) in Eswar College of Engineering Kesanupalli Village, Narasaraopet, Andhra Pradesh 522601

[2]Department of mechanical engineering Assistant Professor (CAD/CAM) in Eswar College of Engineering Kesanupalli Village, Narasaraopet, Andhra Pradesh 522601

[3]Department of mechanical engineering Assistant Professor (CAD/CAM) in Eswar College of Engineering Kesanupalli Village, Narasaraopet, Andhra Pradesh 522601

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).

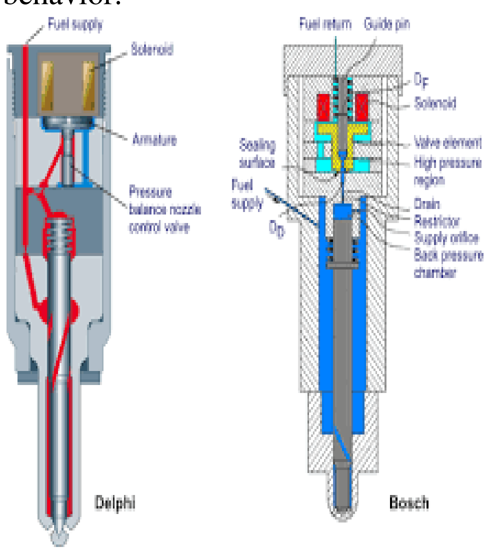
CFD analysis to determine the pressure drop, velocity, heat transfer coefficient, mass flow rate and heat transfer rate for different aerodynamic nozzle (rectangular, circular and hexagonal), the aerodynamic nozzle models modeling using CREO parametric software and analysis in ANSYS software. In ANSYS using analysis modules for aerodynamic nozzle CFD & thermal analysis.

INTRODUCTION

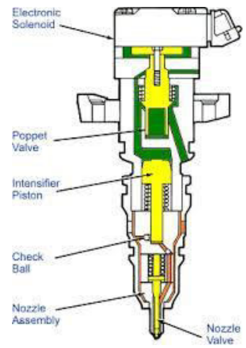
The primary challenges towards developing new diesel engines for traveler cars be the strict future emission legislation together with the customer's demands for steady

2581-4575
International Journal For Recent Developments in Science & Technology
Crossref
A Peer Reviewed Research Journal

rising performance. for instance, the emission limitations of Tier a pair of Bin 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.



This method results is associate degree increasing concentration of deposit-building particles close to the wall.



High a pair of turbulence close to the wall could scale back the force of the aerosol once more to a norm, compensating for associate degree inflated temperature distinction. The deposits area unit composed of connected particles (solid and liquid) and gas (Figure 1).

A **propelling nozzle** is a nozzle that converts the internal energy of a working gas into propulsive force; it is the nozzle, which forms a jet, that separates a gas turbine, being gas generator, from a jet engine.

Propelling nozzles accelerate the available gas to subsonic, transonic, or supersonic velocities depending on the power setting of the engine, their internal shape and the pressures at entry to, and exit from, the nozzle. The internal shape may be convergent or convergent-divergent (C-D). C-D nozzles can accelerate the jet to supersonic velocities within the divergent section, whereas a convergent nozzle cannot accelerate the jet beyond sonic speed.

Principles of operation

- A nozzle operates according to the Venturi effect to bring the exhaust gasses to ambient pressure, and thus form them into a coherent jet; if the pressure is high enough, the flow may choke, and the jet may be supersonic. The role of the nozzle in back-pressuring the engine is explained below.

- The energy to accelerate the stream comes from the temperature and pressure of the gas. The gas expands adiabatically with low losses and hence high efficiency. The gas accelerates to a final exit velocity which depends on the pressure and temperature at entry to the nozzle, the ambient pressure it exhausts to, and the efficiency of the expansion.[5] The efficiency is a measure of the losses due to friction, non-axial divergence as well as leakage in C-D nozzles.[6]

APPLICATION OF A thermocouple junction AT THE NOZZLE TIP

- Initial take a look at cell investigations measured nozzle tip temperatures throughout operation. Production elements were custom-made throughout the project with thermocouples. The conception understood the flexibility to run the engine at rated power and to avoid any influence on the combustion method. Thus, the wants LED to a style, wherever the thermocouples area unit integrated within the body of the nozzle holder. The thermocouple junction was then set between nozzle and warmth defend. The tip of the thermocouple junction was welded to the tip of the nozzle (detailed image of welded thermocouple junction in Figure three fuels and stuff

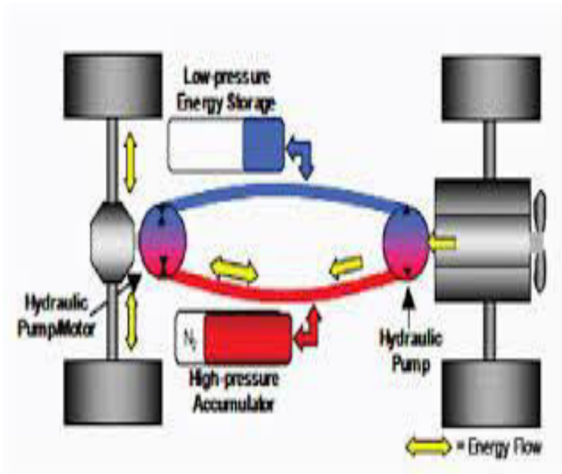
- **EFFECTS OF DISEL NOZZLE**

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.

- **Modeling of gismo Nozzle**

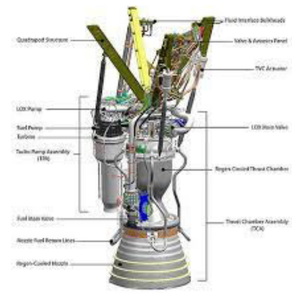
Holes The four-stroke direct-injection (DI) diesel engine was bestowed in this chapter. The specification of the chosen diesel engine was bestowed in Table one. To develop the four-stroke direct-injection diesel modeling is step by step, the primary step is open all of the chosen diesel engine elements to live the engine elements half size. Then, the engine elements size knowledge can be input to the computer code library of the all engine elements knowledge. to make the model, choose window then tile with example library from the menu. this can place the example library on the hand aspect of the screen.

2581-4575
International Journal For Recent Developments in Science & Technology
Crossref
A Peer Reviewed Research Journal



- The example library contains all of the out there templates that are often used in machine modeling. a number of these templates those who are going to be required within the project got to be traced into the project before they will be wont to produce objects and elements. For the aim of this model, click on the icons listed and drag them from the example library into the project library. Some of those area unit examples and a few area unit objects that have already been outlined and enclosed in the template library (Gamma Technologies, 2004). This chapter targeted on fuel nozzle hole of fuel gismo.
- **Injection pattern**

The rate of injection of fuel typically varies throughout the injection length amount. Figure a pair of shows 3 common rate shapes: boot, ramp and sq.. gap rate and shutting rate refers to the gradients within the rate of injection throughout needle nozzle gap and shutting events, severally.



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. the presented testing device also enables the

International Journal For Recent Developments in Science & Technology
2581-4575
Crossref
A Peer Reviewed Research Journal

measurement of the flow coefficient separately for each nozzle hole, which brings better comparison with the results of CFD analysis when the simplified models, introducing only one hole, are applied. Zhijun Li et al [5]had investigates the effects of manufacturing variations in fuel injectors on the engine performance with emphasis on emissions. **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. The time interval CADD (for computer Aided Design and Drafting) is also used. Its use in designing virtual techniques is called digital design automation, or EDA.

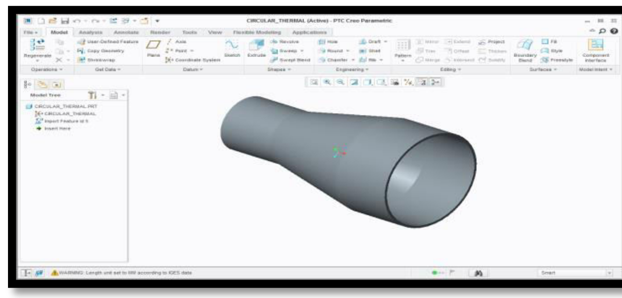
INTRODUCTION TO CREO:

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.

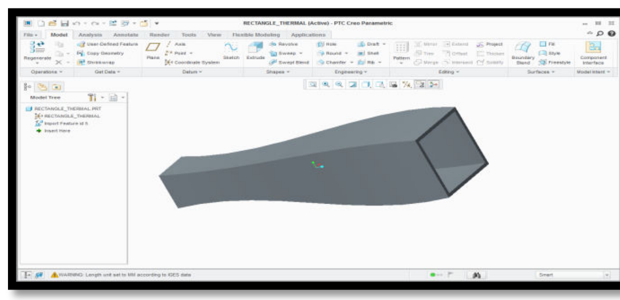
MODELING OF AN AERODYNAMIC NOZZLE

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 models are taken. The model of a circular AERODYNAMIC NOZZLE

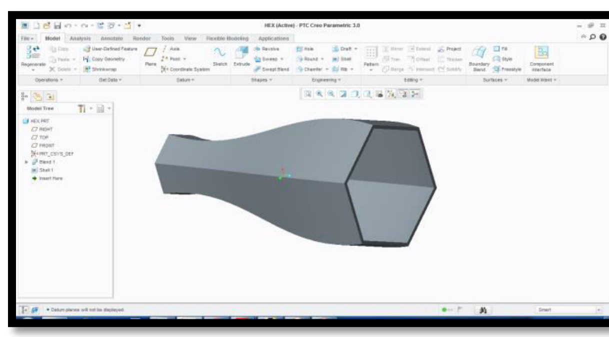
CIRCULAR TYPE



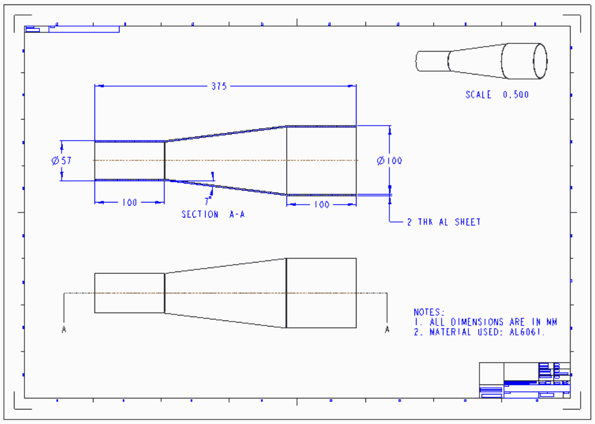
RECTANGULAR TYPE



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.

INTRODUCTION TO ANSYS

Structural evaluation

ANSYS Autodyne is computer simulation gadget for simulating the response of substances to transient period immoderate loadings from influence, high strain or explosions. ANSYS Mechanical ANSYS Mechanical is a finite element analysis gadget for structural assessment, alongside linear, nonlinear and dynamic study. This pc simulation product offers finite factors to version conduct, and helps cloth models and equation solvers for a wide style of mechanical design problems. ANSYS Mechanical also consists of thermal comparison and matched-physics abilities involving acoustics, piezoelectric, thermal–structural and thermo-electrical evaluation.

INTRODUCTION TO CFD

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

CFD ANALYSIS OF AERODYNAMIC NOZZLE

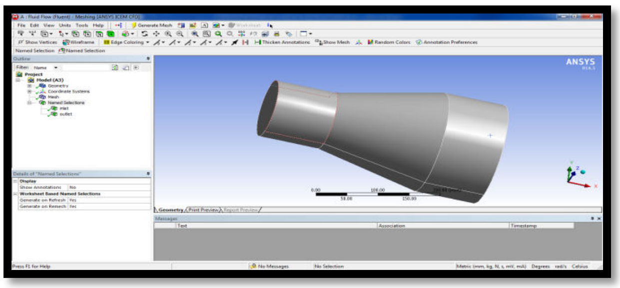
Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reaction (e.g., combustion), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer. The technique is

2581-4575
International Journal For Recent Developments in Science & Technology
Crossref
A Peer Reviewed Research Journal

very powerful and spans a wide range of industrial and non-industrial application areas.

CASE; 1 CIRCULAR TYPE AERODYNAMIC NOZZLE

Imported model



Meshed model

MESHING

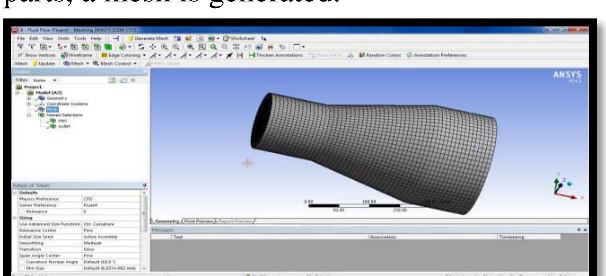
ü The model is created using ICEM CFD software.

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

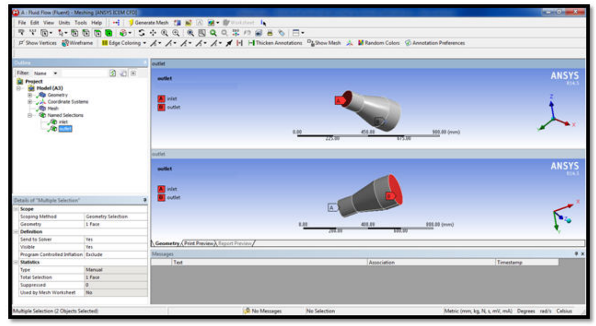
ü Global Mesh parameters are defined which gives information regarding type of mesh. The global element seed size, part parameters are setup and mesh is computed which gives the mesh information regarding total number of elements.

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

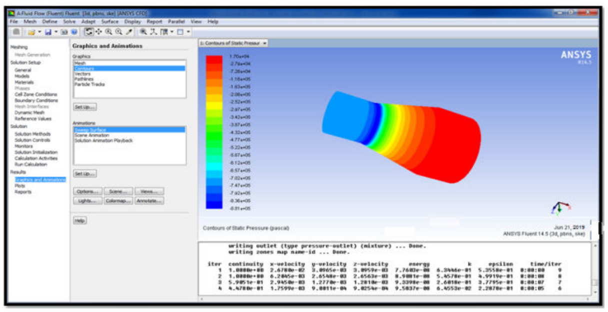
For the Inlet zone, the type would be velocity inlet. The Velocity inlet boundary conditions include the velocity of 45 m/s and a temperature of 1773 K. For the boundary, stationary wall conditions are taken. For the Outlet zone, the type would be pressure outlet. The pressure outlet boundary conditions are taken for standard temperature conditions and operating pressure conditions of 101325 Pa.

The boundary condition for the AERODYNAMIC NOZZLE

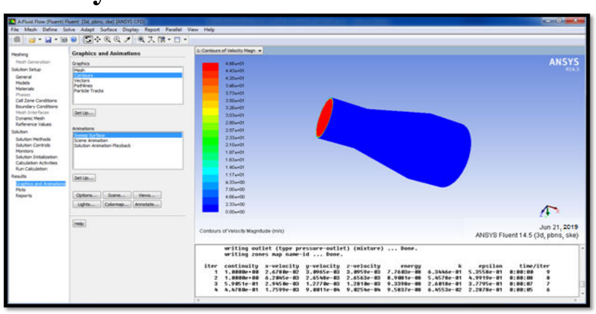
2581-4575
International Journal For Recent Developments in Science & Technology
Crossref
A Peer Reviewed Research Journal

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

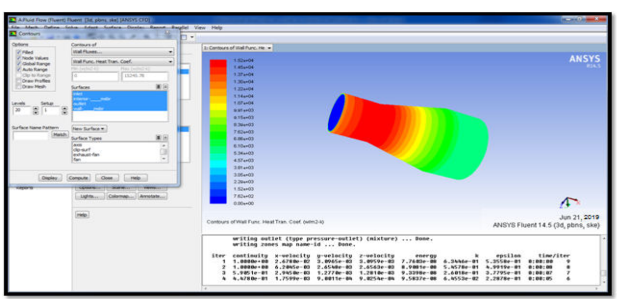
Pressure



Velocity



Heat transfer coefficient



Mass flow rate

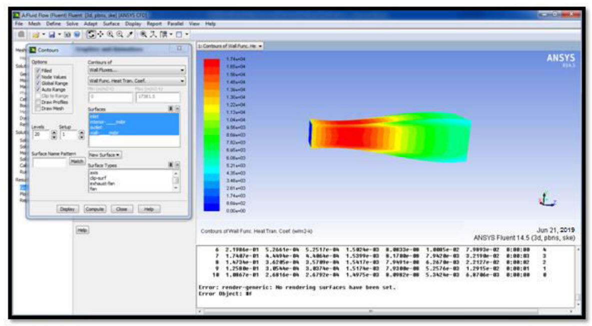
```
        Mass Flow Rate                 (kg/s)
-------------------------- --------------------
                 inlet               380.69006
      interior-____msbr             -20916.734
                outlet                -380.853
          wall-____msbr                      0
     ----------------- --------------------
                   Net             -0.16293335
```

Heat transfer rate

```
Total Heat Transfer Rate                    (w)
------------------------- --------------------
                  inlet          1.1509969e+09
                 outlet         -1.1514876e+09
           wall-____msbr                     0
       ---------------- --------------------
                    Net                -490752
```

CASE; 2 RECTANGULAR TYPE AERODYNAMIC NOZZLE

Heat transfer coefficient

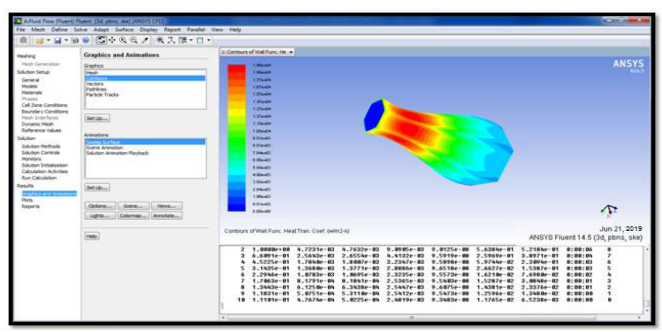


Heat transfer rate

```
Total Heat Transfer Rate                    (w)
------------------------- --------------------
                  inlet          3.6689498e+08
                 outlet         -3.6719926e+08
           wall-____msbr                     0
       ---------------- --------------------
                    Net                -304288
```

CASE; 3 HEXAGONAL TYPE AERODYNAMIC NOZZLE

Heat transfer coefficient

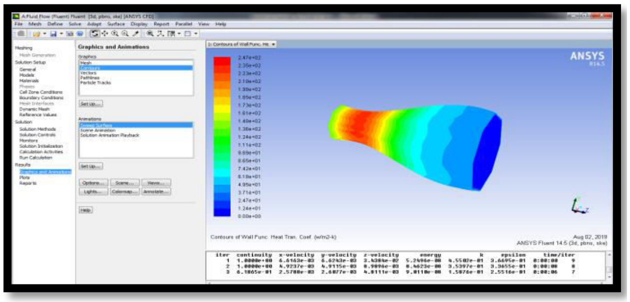


Heat transfer rate

Total Heat Transfer Rate	(w)
inlet	7.3347258e+08
outlet	-7.3450573e+08
wall-____msbr	0
Net	-1033152

Case: 3 octagon type nozzle

Heat transfer coefficient

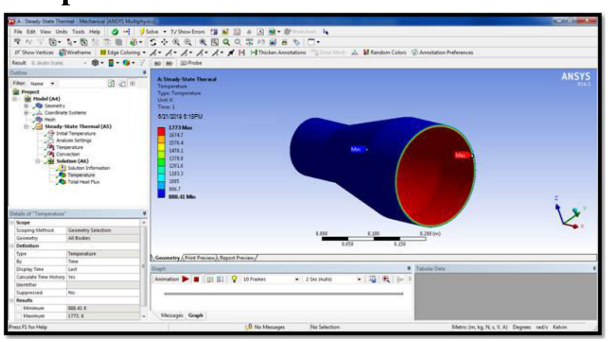


THERMAL ANALYSIS OF AERODYNAMIC NOZZLE

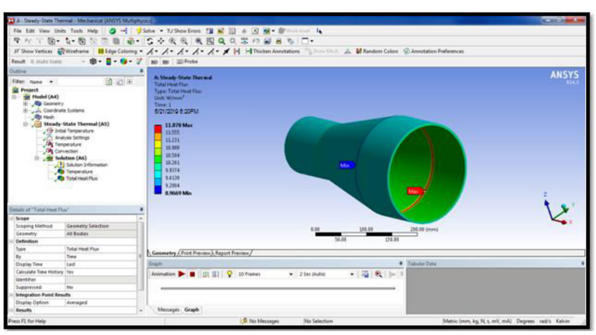
MATERIAL- STEEL

CASE; 1 CIRCULAR TYPE AERODYNAMIC NOZZLE

Temperature distribution

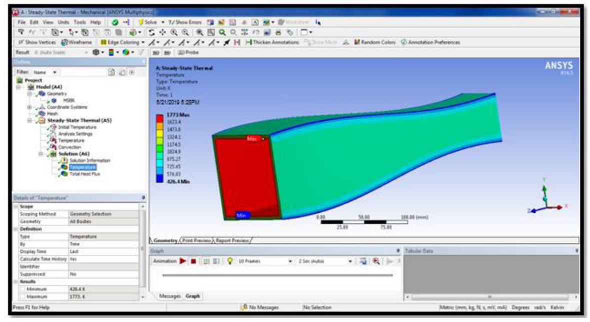


Heat flux

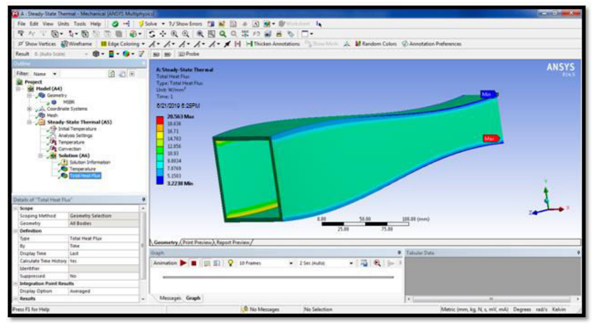


CASE; 2 RECTANGULAR TYPE AERODYNAMIC NOZZLE

Temperature distribution

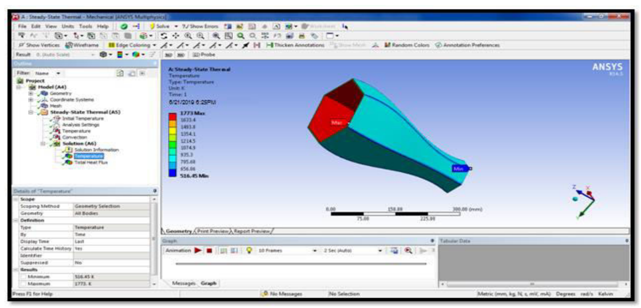


Heat flux

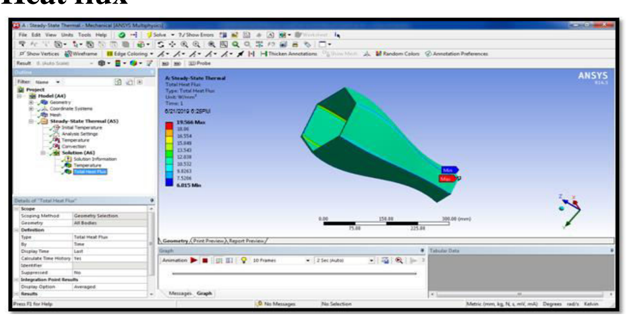


CASE; 3 HEXAGONAL TYPE AERODYNAMIC NOZZLE

Temperature distribution



Heat flux

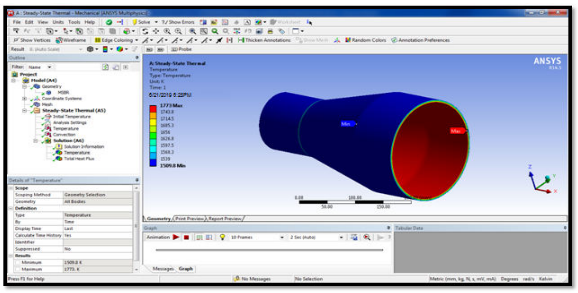


2581-4575
International Journal For Recent Developments in Science & Technology
Crossref
A Peer Reviewed Research Journal

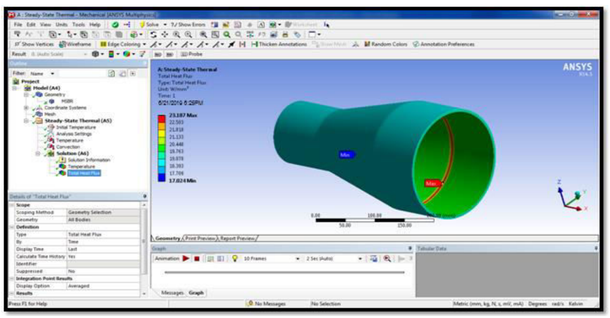
MATERIAL- COPPER

CASE; 1 CIRCULAR TYPE AERODYNAMIC NOZZLE

Temperature distribution

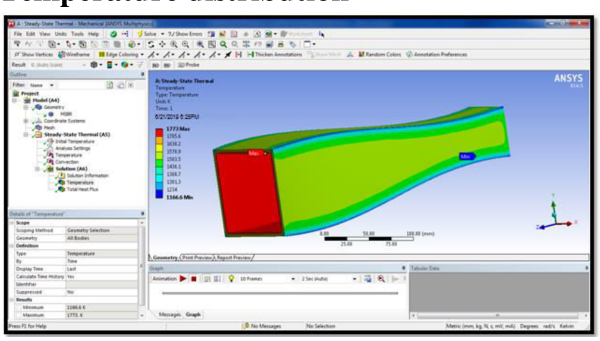


Heat flux

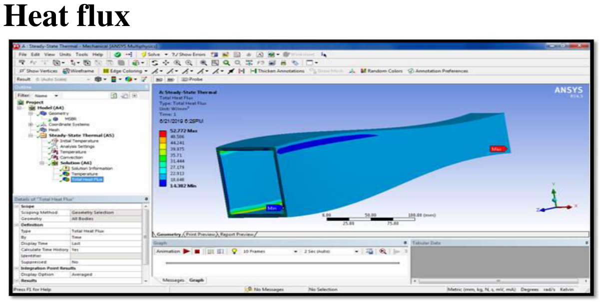


CASE; 2 RECTANGULAR TYPE AERODYNAMIC NOZZLE

Temperature distribution

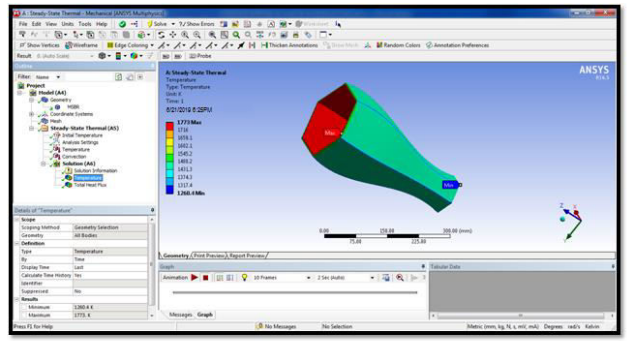


Heat flux

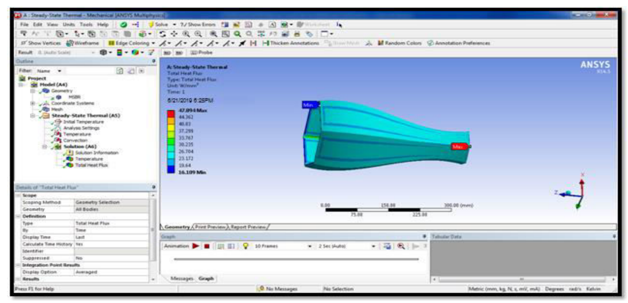


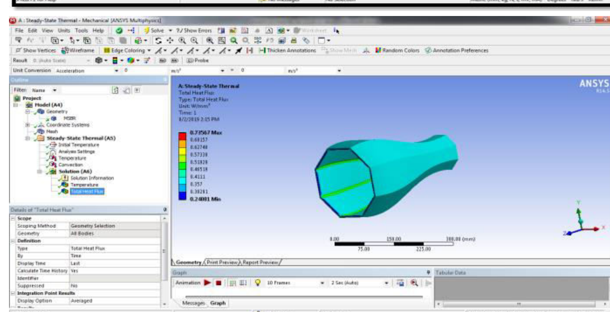
CASE; 3 HEXAGONAL TYPE AERODYNAMIC NOZZLE

Temperature distribution



Heat flux

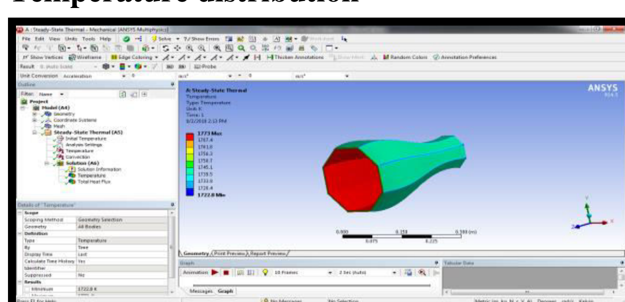




CASE; 4 OCTAGONAL TYPE AERODYNAMIC NOZZLE

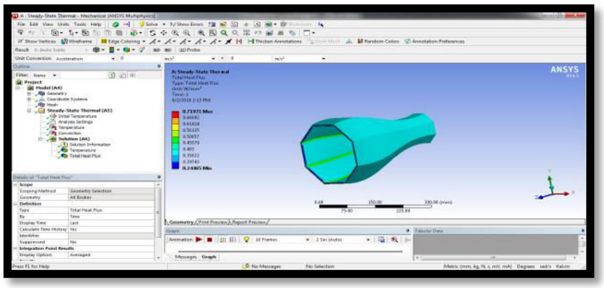
Material- steel

Temperature distribution



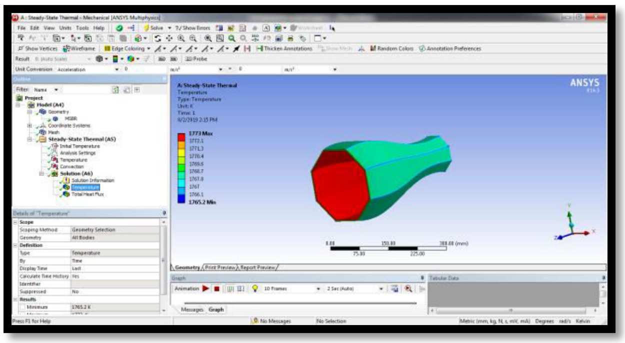
2581-4575
International Journal For Recent Developments in Science & Technology
Crossref
A Peer Reviewed Research Journal

Heat flux

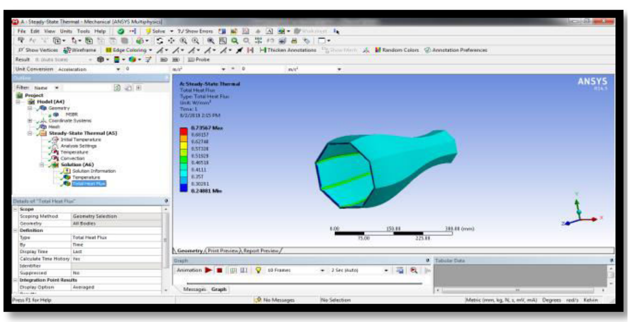


Material- copper

Temperature distribution



Heat flux



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.

[7] Masafumi Nakagawa, Atsushi Harada, "Analysis of Expansion Waves Appearing in the Outlets of Two-Phase Flow Nozzles", International Refrigeration and Air Conditioning Conference at Purdue, July 14-17, 2008.

[8] K.M. Pandey, Member IACSIT and A.P. Singh, "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software", IJCEA, Vol.1, No.2, August 2010.

[9] Vinod kumar S Hiremath, S. Ganesan, R. Suresh, "Design And Analysis Of AERODYNAMIC NOZZLE Of Gas Turbine Afterburner Using CFD",

Proceedings of 27th IRF International Conference, 26th June, 2016.
[10] Santhosh Kumar Gugulothu and Shalini Manchikatla, "Experimental and Performance Analysis of Single Nozzle Jet Pump with Various Mixing Tubes", International Journal of Recent advances in Mechanical Engineering (IJMECH) Vol.3, No.4, November 2014.
[11] Karna S. Patel, "Flow Analysis and Optimization of Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 6 Ver. IV (Nov- Dec. 2014), PP 01-10.
[12] Ms. B.Krishna Prafulla, Dr. V. Chitti Babu and Sri P. Govinda Rao, "Cfd Analysis of Convergent- Divergent Supersonic Nozzle", International Journal of Computational Engineering Research||Vol, 03||Issue, 5.
[13] Bogdan-Alexandru Belega, Trung Duc Nguyen, "Analysis of Flow in Convergent-Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics", International Conference of Scientific Paper a fases 2015 Brasov, 28-30 May 2015.