# THERMAL AND STRUCTURAL VARIATIONS OF A ROCKET NOZZLE USING ANSYS SIMULATION

**B. LAXMI NARASIMHA**
M. Tech Student, Dept of Mechanical-THERMAL, Brilliant grammar school educational institutions group of institutions integrated campus, T.S, India

**Mr. K. SRINIVASA RAO**
Associate Professor, Dept of Mechanical-THERMAL, Brilliant grammar school educational institutions group of institutions integrated campus, T.S, India

## ABSTRACT

*A pressure vessel typically consists of large cylindrical and / or spherical containers with nozzles through which the reactants flow in and out. While plain cylindrical or spherical containers can be analyzed for internal pressure using thin/ thick cylinder formulae, the ones with nozzles are difficult to analyze. This is in view of complicated stress concentrations that arise at the interface of the nozzle and pressure vessel junction. The calculations have become complicated because of forces that arise at the free end of the nozzle. The forces include those of piping, wind forces, earth quake forces in addition to the internal pressure. In spite of these, strict adherence to safety codes is to be followed. ASME, Section VIII specifies the stress limits to be adhered to. One of the criterions is the stress intensity, which is not possible to compute by simple analytical procedures.*

*FEM can be used for computing the deformation and stress at the nozzle-vessel junction in the structure and also at all other points on the pressure vessel. Quite often the geometric models are imported from CAD files for mesh generation with tetrahedral elements. Engineers generally use shell elements or tetrahedral elements while modeling the reactor vessels. But, precise estimation of stress intensity is not possible with these elements for a structure with nozzles. A method is developed for a precise structured modeling and for estimating the stress intensities at the junction of nozzles and pressure vessels. The structure of a reactor vessel of diameter 1900mm and length of 3600mm with a nozzle will be modeled and analyzed. The pressure load is 7 MPa. Stresses have been estimated. The induced stresses are compared with allowable stress. Based on the induced stress in the Pressure vessel two additional design variants have been studied to bring the stresses within allowable limits.*

***Keywords:*** *Pressure vessel, finite element analysis, stress intensity, nozzle.*

## INTRODUCTION

### Introduction to Nozzle

Nozzle is the vital component of a Missile, Rocket or air-breathing Engine that Produces thrust required for the rocket or missile movement. The main purpose of the nozzle is to convert the thermal energy of the flue gases into Kinetic energy. The generated high velocity gases move along the Nozzle axis, as illustrated below, accomplish this.

Typically propellant is composition of the liquid $h_2$ and oxidizer. Liquid $H_2$ and oxidizers are sent into the combustion chamber with some mass flow rate where proper amount of liquid $H_2$ and oxidizer mixed together and burning takes place. The generated flue gases are sent through the nozzle. In the nozzle the area of will continuously varies and the area cross section variation will be different for different nozzles. In C-D nozzle the least area portion is said to be throat.

Since the throat is of less cross-sectional territory than whatever is left of the portion, the gasses are compacted to a higher weight. The spout itself continuously Increases in cross sectional territory permitting the gasses to grow. As the gasses do as such, they push against the dividers of the spout making push. Mathematically, a definitive motivation behind the spout is to extend the gasses as productively as conceivable in order to augment the leave speed ($v_{exit}$). This procedure will amplify the push (F) create by the framework since the two are straightforwardly related by the condition.

Thrust force = Force due to momentum + Force due to the pressure difference

## MATERIAL AND ITS PROPERTIES

## HIGH TEMPERATURE MATERIALS FOR NOZZLES AND THEIR REQUIREMENT

Recent developments in nuclear power, jet aircrafts, ballistic missiles and rocketry have increased the demand for materials that have good corrosion resistance, strength characteristics and particularly, creep resistance at high temperatures.

High temperature use of materials can give rise to several problems such as:

1. Accelerated oxidation and/or corrosion.
2. Creep
3. Grain boundary other weakening
4. Allotropic and other phase changes
5. Modification of conventional properties.

Therefore, the material for high temperature use most be such that it can withstand these difficulties and perform its functions satisfactory during service.

The nozzle components are made of 18 Maraging steel 250 (ASTM A579 Code 72). This material used in annealed condition to acquire high strength, good machinability and the high temperature. The composition used for this material is given in table

## FINITE ELEMENT ANALYSIS OF NOZZLE

## FINITE ELEMENT MODELING

Finite Element Method in short known as FEM has a vital technique and present it is one of the most widely used techniques, for analyzing mechanical loading characteristics in modern engineering components. Traditional analysis techniques can only be satisfactory applied to a range of conventional component shapes and specific loading conditions. Unfortunately, the majority of engineering loading situations are not simple and straight forward therefore traditional techniques often need to be modified and compromised to suit situations for which they were not intended. The instability hence made, normally prompts the creator applying exorbitantly high factor of security to the mechanical loads thus to over plan segments by determining either pointlessly cumbersome cross areas or astounding materials, unavoidably the cost of item is unfavorably influenced.

Finite Element Method in short known as FEM is one of the numerical methods that process certain Characteristics that take advantage of special facilities, offered by the high speed computers. Specifically the limited component technique can be deliberately modified to suit such perplexing and troublesome issues as non-homogeneous materials, non straight anxiety strain conduct and convoluted limit conditions.

## FEA PROCEDURES:

The basic steps involved in the FEA are

**Step 1:** Modelling, Descretization of the given domain using finite elements of different types, shapes and orders.

**Step 2:** Approximation of field variables over each element domain

**Step 3:** Element matrix generation

**Step 4:** Assembly of element matrices

**Step 5:** Imposition of boundary and constraint conditions
**Step 6:** Solution of global matrix equations
**Step 7:** Post processing of the results

## LITERATURE REVIEW

Convergent Divergent Nozzles are mostly used for Rockets, Jets and High speed Missiles which operate at relatively high Mach speeds ranging from Mach 2 to Mach 5. The motivation behind the CD spout is to change over warm or warmth vitality to motor vitality to acquire a fast fumes which thusly gives an equivalent measure of pushed according to Newton's Third Law of movement. Spout shapes assume a huge part in its execution and requires an in profound examination to outline for a particular utilize or application.

The Convergent piece of the spout is for the most part outlined such that it gets greatest measure of liquid mass stream thus it is intended to work for greater volumes. The size and measurements of the dissimilar part assumes an essential part in choosing the spout execution. A few parameters that best depict the execution attributes of a CD spout are Nozzle Pressure Ratio (NPR), Nozzle Area Ratio (NAR) or Expansion Ratio and the Divergence point. The adjustments in any of these parameters will incredibly influence the spout execution relying upon its outline.

**[1] G.Satyanarayana, Ch.Varun and S.S.Naidu** (2012) have done the investigation of stream inside merged disparate supersonic spout for various cross areas like rectangular, square and round by utilizing ANSYS FLUENT 12.0. They played out the examination relying upon the state of the supersonic spout keeping a similar info conditions. They utilized the utilization of a spout as torpedo and done the work for most appropriate spout among those which are considered. Results have been appeared for weight, speed and temperature varieties for various supersonic spouts and they found that rectangular spout gives high leave speed contrasted with roundabout and square spouts and they likewise specified that liquid properties are to a great extent reliant on the cross area of the spout which incredibly influences the liquid stream.

**[2] AfshinAbrishamkar, and AliakbarJoneidijafari** (2013) have completed a reproduction examination on a focalized different spout which is utilized as a part of supersonic breeze burrows. They made utilization of programming devices like Ansys Fluent and Gambit to complete the recreation examination. They completed the recreation on a 2D space stage and made a numerical examination between CFD demonstrating comes about and accessible measured information. They embraced two unique models to be specific K-$\mathcal{E}$ model and K-$\omega$ demonstrate and saw that k-$\mathcal{E}$ display gets higher normal estimations of Mach no as opposed to k-$\omega$ show. They watched that the computational esteems are not precisely coordinating with measured information and finished up by saying that the deviation is because of material property variety, slip factor and divider erosion. At long last they finished it by demonstrating that the genuine outcome gotten in one plan may vary with that of other and it would be a spitting picture of the past plan.

**[3] Steffen et al.** [2015] have directed tests on different CD spouts by changing the difference points from 7o 10' to 50o and spout region proportions from 1.39 to 3.81over an extensive variety of spout weight proportions to decide the impacts of changing the disparity edge on the spouts inside execution. They watched that the push coefficient diminished from 0.973 at a uniqueness point of 7o to 0.93 for a dissimilarity edge of 50o . The additionally watched that the weight proportions where division happened an expansion in difference edge affected the spout execution all things considered. They at long last closed by saying that for a steady weight proportion, the adjustment in spout region proportion has a negligle impact on the execution of a spout

**[4] Badrinarayanan** (1961) researched tentatively the base streams at supersonic rates. Itemized estimations in the wake stream behind limit 7 based 2-d and 3-d bodies were made at M = 2. The outcomes toss some light on the behavio of isolated stream and shows the significance of stream inversion. The impact of air infusion at the base demonstrates that the base weight increments essentially with air infusion. contemplated the impact of construct seep with respect to an intermittent wake. He presumed that Base drain decreases the drag of an aerofoil, by deferring the onset of insecurity in the isolated shear layers. The extent of vorticity which really enters the vortex sheet in the wake of being shed from the model tumbles from an underlying estimation of around 0.5 as the shear layers increment long. In his trial, the ideal drain was given by a drain coefficient of 0.125. This gives a drag diminishment equivalent to that delivered by a long splitter plate and it was imagined that little further change is conceivable by any strategy for wake obstruction. No endeavor was made by Wood to clarify either how base drain balance out the shear layer or why little drain amounts seem to have the switch impact. The strategy utilized by the Wood to decide the properties of the vortex road was a backhanded one, in light of the accepted legitimacy of the VON KARMAN vortex road.

**[5] Hall and Orme** (1956) contemplated compressible course through sudden extension in a pipe, both hypothetically and tentatively, and demonstrated a decent assention amongst hypothetical and trial comes about. They built up a hypothesis to foresee the Mach number in a downstream area of sudden amplification for known esteems and Mach number at the exit of the gulf tube, with incompressible stream presumption. They likewise accepted that the weight over the substance of the extension was equivalent to the static weight in the little tube just before the development. However, this supposition is far from reality, the weight over the face in the distribution area, in particular the base weight is especially unique in relation to the weight in the littler tube just before the extension.

## MODELING

### 5.1GEOMETRIC MODELLING:

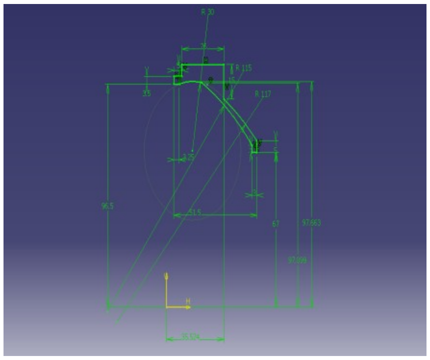
Modelling has going to workout in CATIA V5 R 15 Software, which is very easy modelling software for modelling of critical and complex component and also the interface of the CATIA package is user friendly.

Mainly Nozzle consisting of 3 components as follows:

1. End dish of the Nozzle
2. Neck of the nozzle
3. Nozzle Cone

### NOZZLE END DISH:

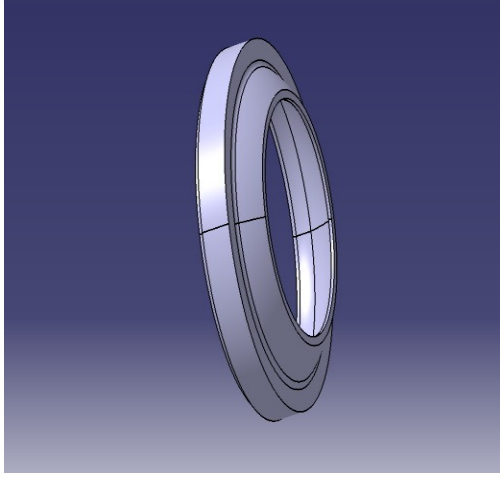
End dish of the nozzle is the vital part to component model. End dish of the Nozzle inner wall has different curvature radius and the outer surface has different curvature, where it becomes complicated to model. Nozzle end dish is cylindrical object. The 2-d cross sectional view of drawing of the nozzle end dish is given the Fig 5.1detailed drawing of Nozzle End Dish (mm)



**NOZZLE END DISH DIMENSIONS (MM)**

Nozzle end dish is modelled by the following procedure:

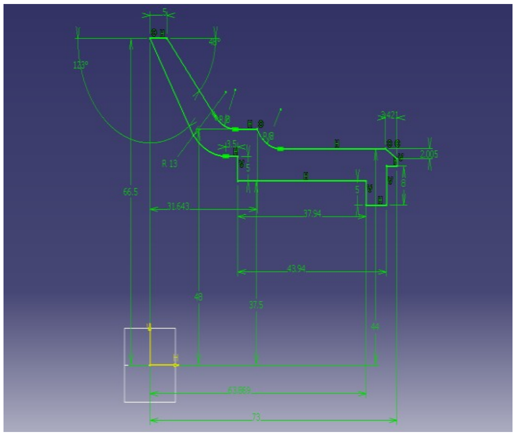
1. The sectional drawing of the nozzle end dish is revolved to 360 about the Central axis as Shown in the sectional drawing is revolved to obtain the 3 D model of nozzle end dish shown in the fig below



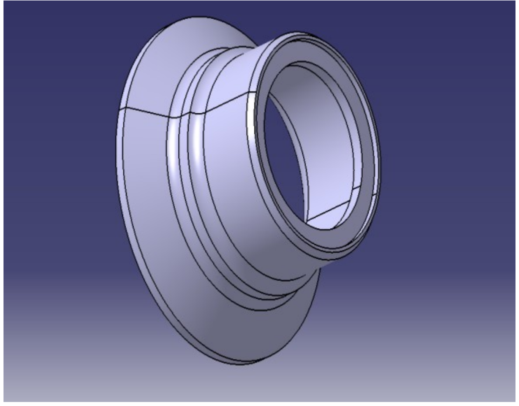
**3-D MODEL OF NOZZLE END DISH**

## NOZZLE NECK

Nozzle neck is the critical part to model, Nozzle neck inner wall has a curvatures with inclined angle of 48 and outside curvature with a angle of33, where it becomes complicated to model. Nozzle neck is cylindrical object. The detailed drawing of the nozzle neck is given the fig-5.3



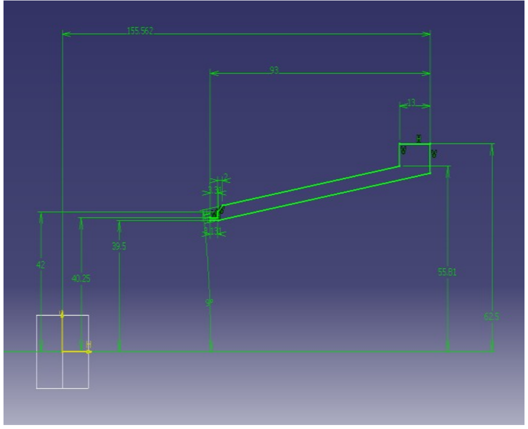
**NOZZLE NECK DIMENSIONS (MM)**



**3D MODEL OF NOZZLE NECK**

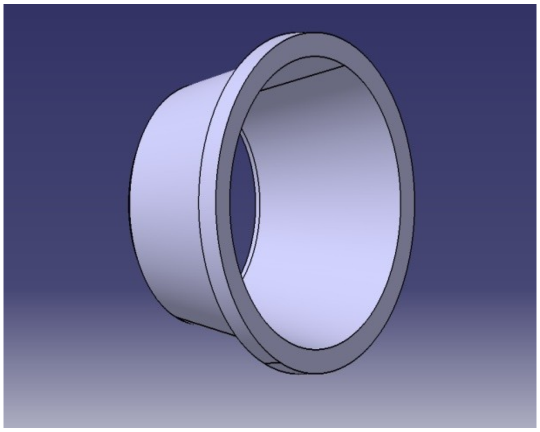
## NOZZLE CONE

Nozzle cone is easy to mode l, Nozzle cone inner wall is inclined angle of 9 Nozzle cone is cylindrical object. The detailed drawing of the nozzle cone is given the fig-5.5.



**DETAILED DRAWING OF NOZZLE CONE (MM)**

Nozzle cone is modelled by the following procedure:

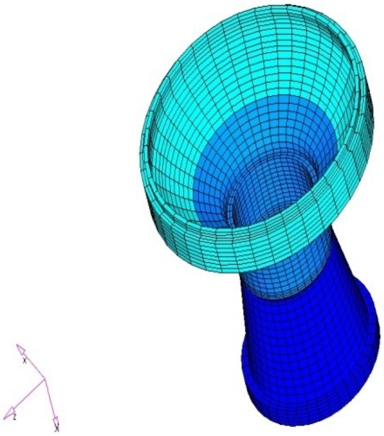


**3 D MODEL OF NOZZLE CONE**

## MESHING AND ANALYSIS

### MESH GENERATION:

According to the specific quality and required specifications Fine sized hexahedral shaped elements and shell elements were generated in the mesh generation. All the quality criteria's were maintained within the default values in the hyper mesh. Quality criteria are mentioned as follows.



**MESHED MODEL**

### BOUNDARY CONDITIONS

Meshed model is imported to ansys11.0 .In ansys 3D 20 noded solid element (solid 95) is defined. Solid 95 is higher order element.

## RESULTS AND DISCUSSIONS

### STRUCTURAL ANALYSIS

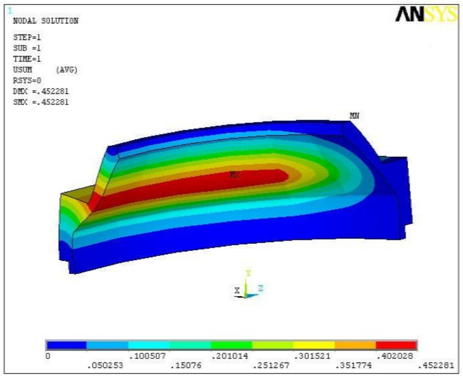
Analysis is carried out for each component with pressure loads given in the table-6.2.Results are tabulated in table.

Resultant displacement and von-Mises stress contours subjected to load case 1 are shown in Figures.

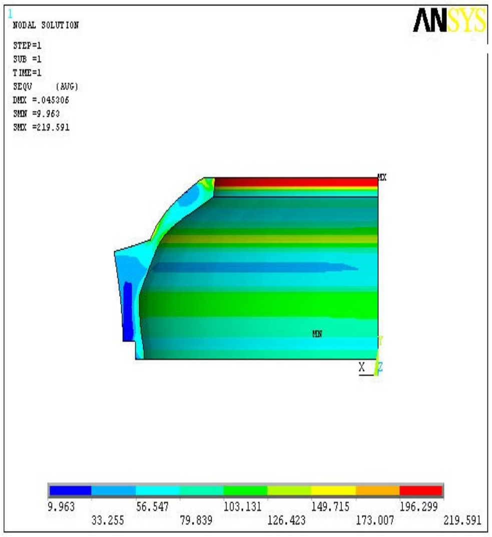
Resultant displacement and von-Mises stress contours subjected to load case 2 are shown in Figures.

Resultant displacement and von-Mises stress contours subjected to Load case 3 are shown in Figures.

### RESULTANT DISPLACEMENT AND VON-MISES STRESS CONTOURS FOR LOAD CASE1

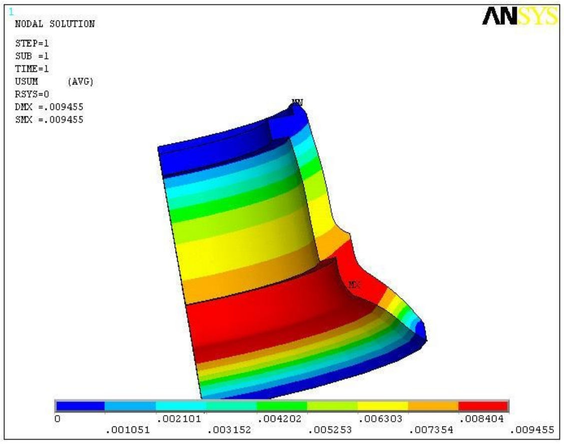


**RESULTANT DISPLACEMENT OF END DISH FOR LOAD CASE1 (MM)**



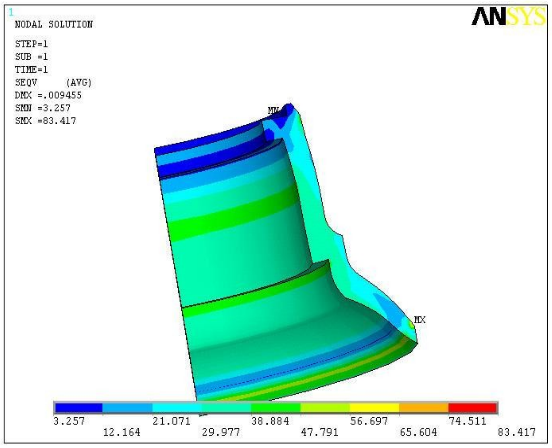
**VON-MISES STRESS OF END DISH FOR LOAD CASE 1 ($N/MM^2$)**

From Maximum stress is 219.591 $N/mm^2$.Maximum Stress is at the Nozzle end dish curvature. Minimum Stress is 9.963$N/mm^2$



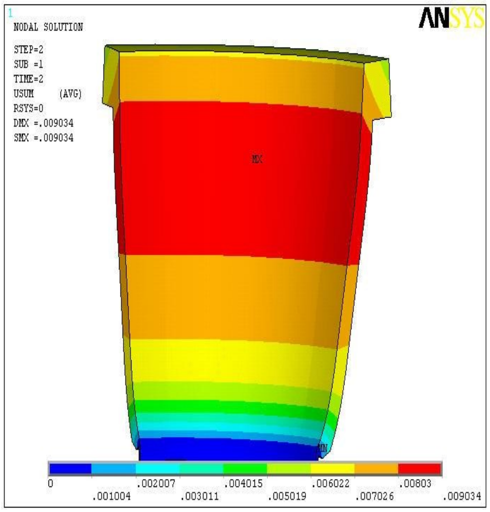
**RESULTANT DISPLACEMENT OF NECK FOR LOAD CASE 1(MM)**

From the Resultant displacement contour figure maximum displacement is 0.0094 mm. This end is assembled to the nozzle end dish. Minimum displacement is 0.0105 mm in the nozzle neck



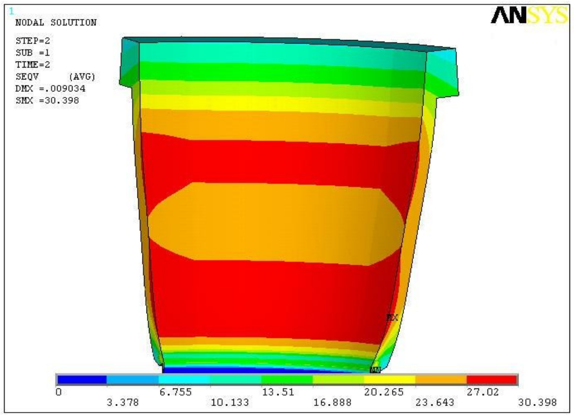
**VON-MISES STRESS OF NECK FOR LOAD CASE 1(N/MM$^2$)**

From the von-Mises Stress contour maximum stress is167.059 N/mm$^2$. And that end of the nozzle neck is assembled to the nozzle end dish. Minimum Stress is 3.257 N/mm$^2$.



**RESULTANT DISPLACEMENT OF CONE FOR LOAD CASE 1(MM)**

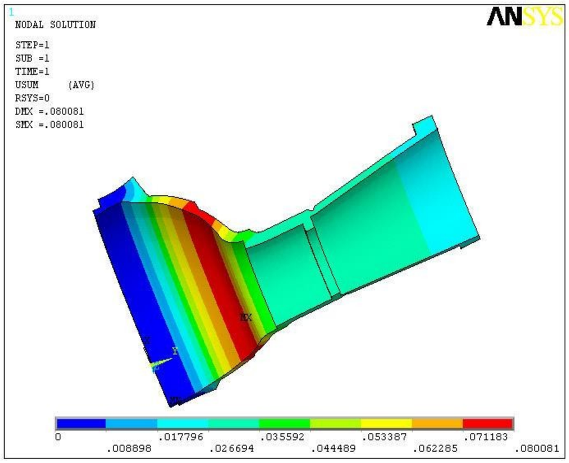
From the Resultant displacement contour maximum displacement is 0.0903 mm. Maximum displacements in the Nozzle cone is due to, component is assembled at the free end of Nozzle. Minimum displacement is 0.00100 mm in the nozzle cone.



**VON-MISES STRESS OF CONE FOR LOAD CASE1 (N/MM$^2$)**

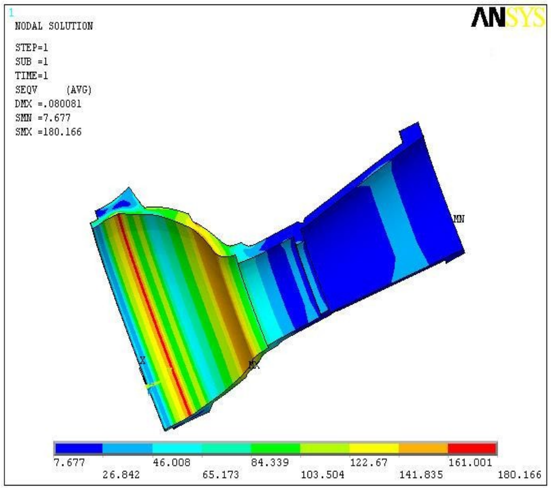
From the von-Mises Stress contour maximum stress is 30.398 N/mm$^2$ .Minimum Stress is 3.378 N/mm$^2$ is at the end of the nozzle cone.

**RESULTANT DISPLACEMENT AND VON MISES STRESS FOR NOZZLE ASSEMBLY LOAD CASE 1**



**RESULTANT DISPLACEMENT CONTOUR OF NOZZLE ASSEMBLY FOR LOAD CASE 1(MM)**

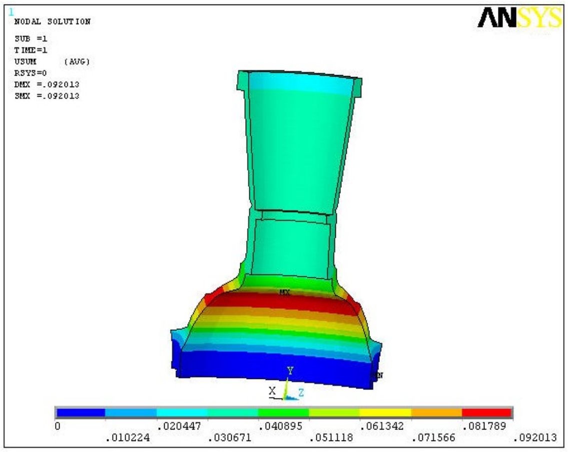
maximum displacement is 0.0800 mm. Here in the nozzle assembly the maximum displacement is in the nozzle end dish component. And the minimum displacement is 0.008898.



**VON-MISES STRESS COUNTER OF NOZZLE ASSEMBLY FOR LOAD CASE 1(N/MM$^2$)**

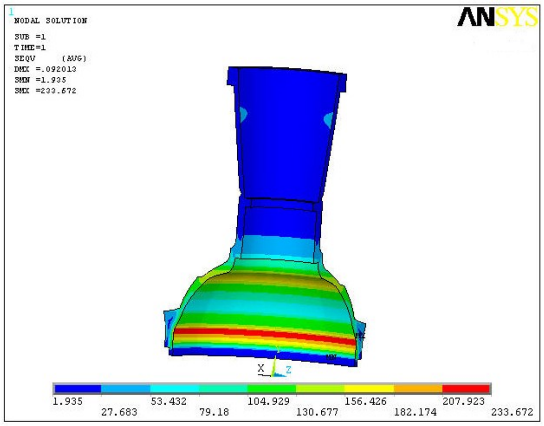
From the Von mises stress contour maximum stresses are occurred in End dish component of Nozzle and the stress is 180.166 N/mm$^2$. Maximum stress is due to nozzle end dish curvature thickness is low and that end is assembled to the nozzle neck.

### THRUST OF ASSEMBLY FOR LOAD CASE 1



**RESULTANT DISPLACEMENT OF NOZZLE ASSEMBLY APPLYING THRUST FOR LOAD CASE1 (MM)**

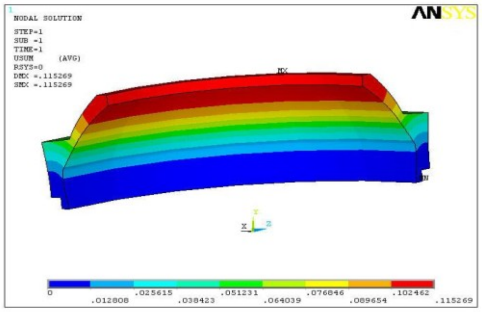
If we apply thrust on nozzle assembly (load case1) maximum displacement is 0.092013 and minimum displacement is0.010224



**VON – MISES STRESS OF NOZZLE ASSEMBLY APPLY THRUST FOR LOAD CASE1 (N/MM$^2$)**

Stress maximum is 233.672and minimum stress is 1.935.That means nozzle can with stand for this load case 1 by applying thrust also.

### RESULTANT DISPLACEMENT AND VON-MISES STRESS CONTOURS FOR LOAD CASE2



**RESULTANT DISPLACEMENT OF END DISH FOR LOAD CASE 2(MM)**

## CONCLUSIONS

Static Analysis and Thermal analysis is done on the nozzle. For the different load conditions nozzle is analyzed. A global Finite Element Analysis (FEA) is applied to arrive at the stresses and deformations in Nozzle.

a. Maximum displacement .317081 occurs in load case 4
b. Maximum stress 793.798 $N/mm^2$ occurs in load case 4
c. Stresses in Load Cases 1(179.9 $N/mm^2$) , Load Case 2 (332.906 $N/mm^2$) and load case3 $(378.974n/mm^2)$ are comparatively less value than the yield strength of the material 750 $N/mm^2$ .So Nozzle is safe up to the load the load case 3.
d. Stresses are maximum in Load Case 4 and the stress value is 793.798, which is greater that yield strength of the material 750 $N/mm^2$.This causes failure of the Nozzle.
e. Maximum deformation is occurred in the nozzle end dish curvature.
f. It is absorbed that the maximum heat flux is developed at the end dish curvature
g. Stress at the given condition is 378.974 is less that yield strength 750 by this observation of results, it is concluded that **"The Structure is safe under the given loading conditions".**
h. Nozzle assembly can with stand by applying thrust also. Up to load case 3
i. Thermal analysis should carried to observe the Heat flow and convention results.

## References

*[1]. P. Parthiban, M. Robert Sagayadoss, T. Ambikapathi, Design And Analysis Of Rocket Engine Nozzle by using CFD and Optimization of Nozzle parameters, International Journal of Engineering Research, Vol.3., Issue.5., 2015 (Sept.-Oct.).*

*[2]. Bogdan-AlexandruBelega, TrungDuc Nguyen, Analysis of Flow in Convergent-divergent rocket engine nozzle using Computational Fluid Dynamics, International Conference Of Scientific Paper Afases 2015 Brasov, 28-30 May 2015.*

*[3]. BalajiKrushna.P, P. SrinivasaRao, B. Balakrishna , Analysis Of Dual Bell Rocket Nozzle Using Computational Fluid Dynamics, IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308.*

*[4]. Dumonov, G., Ponomaryov, N.B. and Voinov, A.L.,(1997)``Dual-Bell Nozzles for Rocket Engines of Launch Vehicle Upper Stages and Orbital Transfer Vehicles,'' AIAA Paper 97-3089, 33rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, USA, Luglio.*

*[5]. AbdulnaserSayma ,Computational Fluid Dynamics.*

*[6]. Hagemann, G., Immich, H. and Preuss, A,(3-6 December 2002,)``Advanced Nozzle Concepts for Future Rocket EngineApplications'', 4th International Conference on Launcher Technology, Liege, Belgium.*

*[7]. K.M.Pandey and S.K.Yadav, ,CFD Analysis of a Rocket Nozzle with Two Inlets at Mach .1,vJournal ofEnvironmental Research and Development, Vol 5, No 2,2010, (pp 308-321).*

*[8]. P. Padmanathan, Dr. S. Vaidyanathan, Computational Analysis of Shockwave in Convergent Divergent Nozzle, International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 2, Issue 2,Mar-Apr 2012, pp.1597-1605.*

*[9]. Natta, Pardhasaradhi.; Kumar, V.Ranjith.; Rao, Dr. Y.V. Hanumantha; Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd), International Journal of Engineering Research and*

*Applications (IJERA), ISSN: 2248-9622,Vol. 2, Issue 5, September- October 2012, pp.1226-1235.*

*[10]. Pandey,K.M.; Singh, A.P.; CFD Analysis of Conical Nozzle for Mach 3 at V arious Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010, ISSN: 2010-0221*