International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 07 Issue: 03 | Mar 2020 www.irjet.net p-ISSN: 2395-0072

STRESS ANALYSIS OF SOLID ROCKET MOTOR CASE

Nevin Nelson[1], A. Yezhil Arasu[2], B. Kiran Kumar[3], Thomas Kurian[4]

[1]Asst. Professor, Dept. Of Mechanical Engineering, Bishop Jerome Institute, Kerala, India
[2]Indian Space Research Organisation, Vikram Sarabhai Space Centre, Kerala, India

---

Abstract - *With the help of finite element analysis the actual stress distributions in the different part of a rocket motor case were studied. It is found that element size affects the results of finite element analysis but after particular size of the element it does not effect on the analysis results. Stress obtained for cylinder shell and hemispherical dome were analyzed. Stress analysis helps us to find out stresses at different points, which were difficult to calculate from empirical relations.*

***Key Words*: Solid Rocket Motor Case, FEA, Circular Shell, Hemispherical Dome, Hoop stress, Meridional stress.**

1. INTRODUCTION

Solid rocket motors are used as boosters for satellite launchers. In a solid rocket motor, the fuel and oxidizer are mixed together into a solid propellant, which is packed into motor case. When the mixture is ignited by the igniter, combustion takes place on the surface of the propellant. A flame front is generated which burns into the mixture. The combustion produces great amounts of exhaust gas at high temperature and pressure. There exists a thermal insulation between propellant and motor case to limit the temperature of motor case. The hot exhaust gas is passed through a nozzle which accelerates the flow. Thrust is then produced according to Newton's third law of motion [1]. A solid propellant rocket is formed by four main components as shown in Fig -1.

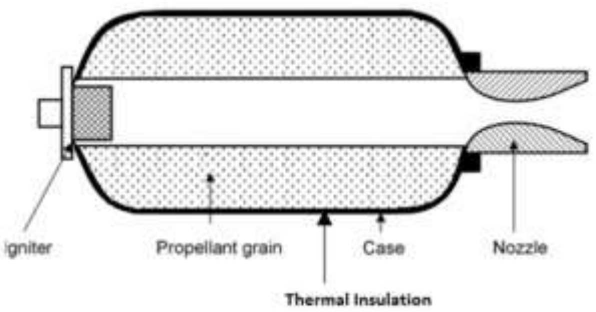

Fig -1: Basic Solid Rocket Motor [1]

A case containing the solid propellant and withstanding internal pressure when the rocket is operating. The solid propellant charge (or grain), which is usually bonded to the inner wall of the case, and occupies before ignition the greater part of its volume. When burning, the solid propellant is transformed into hot combustion products. The nozzle channels the discharge of the combustion products and because of its shape accelerates them to supersonic velocity. The igniter, which can be a pyrotechnic device or a small rocket, starts the rocket operating when an electrical signal is received.

2. DESIGN OF SOLID ROCKET MOTOR CASE

To illustrate the design approach used for stress analysis and to bring out the intricacies a case study is shown in the section. The procedure followed for design calculations and the following finite element analysis is typical for any rocket motor case stress analysis [2, 3].

2.1 Cylindrical Shell Design

To illustrate the design approach used for stress analysis and to bring out the intricacies a case study is shown in the section. The procedure followed for design calculations and the following finite element analysis is typical for any rocket motor case stress analysis [2, 3].

Operating pressure, P = 6 MPa
Inner diameter = 88.70 mm
Inner radius, r = 44.35 mm
Factor of Saftey (F.S) on yield = 1.1
Material – High Strength Steel
Young's modulus, E = 206 MPa
Poisson's ratio, υ= 0.3
Yield strength, σ_{ys} = 835 MPa
Welding efficiency, η_w = 0.9
Mismatch Factor, M.F = 1.15 for long seam weld joint
Mismatch Factor, M.F = 1.3 for cir-seam weld joint

Hoop stress, $\sigma_h = (P*r)/t$ (2.1)
Meridional stress, $\sigma_m = (P*r)/2t$ (2.2)

Considering the fact that hoop stress is the max. Principal stress (σ_1) in the cylindrical shell we will find out the thickness required as

Max. Principal stress (σ_1) = σ_{ys}/F.S (2.3)
$\sigma_1 = (P*r)/t = \sigma_{ys}/F.S$
$t = [(P*r)/\sigma_{ys}] *F.S$

Inherently there will be some mismatches in the weld in cylindrical shell. When mismatch comes an additional bending stress will introduced in our structure, this will get added to our hoop stress. This can be accounted by introducing some factors called Mismatch factor and Weld efficiency factor.

Mismatch Factor, M.F = 1+(3e/t) (2.4)

Where 'e' is center line mismatch, which will be different for different type of welds and depends on thickness of the material. Usually we come across long seam weld, cir-seam weld and T-joint.

e = 5% of t for long seam weld.

e = 10% of t for cir-seam weld.

While doing welding process the strength of the parent material will get affected, this can be taken care by introducing weld efficiency factor. Generally strength of weld material is 90% of strength of parent material.

Mismatch Factor for long seam weld, M.F = 1+(3e/t)
M.F = 1+[(3*0.05t)/t] = 1.15

t = [(P*r)/σ_{ys}]*F.S*(M.F/ η_w)
t = [(6*44.35)/835]*1.1*(1.15/ 0.9)
t = 0.448 mm

Considering some manufacturing allowances our adopted cylindrical shell thickness = 0.6 mm
Radius of cylindrical shell = 44.35 mm

Table -1: Stresses at various locations of cylindrical shell.

Locations	Hoop Stress (MPa)	Meridional Stress (MPa)	Effective Stress (MPa)
Away from welds	443.50	221.75	384.08
Near Long Seam joint	510.03	221.75	442.93
Near Cir-Seam joint	443.50	288.28	389.80
Near T- joint	510.03	288.28	442.93

2.2 Hemispherical Dome Design

Whenever process design or storage conditions permit or high design pressure requires, a spherically shaped vessel is used. Although it is more difficult to fabricate than a cylindrical shell, it requires only half the wall thickness of the cylindrical shell under the same pressure, with minimum exposed area. On large-diameter cylindrical vessels, hemispherical heads will introduce negligible discontinuity stress at junctures.

A spherical shell is developed by rotation of a circle around an axis. Both stresses are uniform across the shell thickness in tension [4, 5].

Hoop stress, σ_h = (P*r)/2t (2.5)
Meridional stress, σ_m = (P*r)/2t (2.6)

Considering hoop stress is the max. Principal stress (σ_1) in the cylindrical shell we will find out the thickness required for design purpose by equating eqn. (2.5)

Max. Principal stress (σ_1) = σ_{ys}/F.S
σ_1 = (P*r)/2t = σ_{ys}/F.S
t = [(P*r)/2σ_{ys}] *F.S

Due to some mismatch in weld joints,

Mismatch Factor, M.F = 1+(3e/t)
e = 5% of t for long seam weld.

Mismatch Factor for long seam weld, M.F = 1+(3e/t)
M.F = 1+[(3*0.05t)/t] = 1.15

t = [(P*r)/2σ_{ys}]*F.S*(M.F/ η_w)
t = [(6*44.35)/(2*835)]*1.1*(1.15/ 0.9)
t = 0.224 mm

From the above formula the recommended thickness for hemispherical dome is 0.3 mm, but the discontinuity stress at the cylinder-hemispherical head junction can be minimized by a design taper between the head and the cylindrical shell.
So the adopted hemispherical dome thickness = 0.6 mm
Radius of hemispherical dome = 44.35 mm

Table -2: Stresses at various locations of hemispherical dome end.

Locations	Hoop Stress (MPa)	Meridional Stress (MPa)	Effective Stress (MPa)
Away from welds	221.75	221.75	221.75
Near Long Seam joint	255.01	221.75	240.11
Near Cir-Seam joint	221.75	288.28	261.44
Near T- joint	255.01	288.28	273.17

3. FINITE ELEMENT ANALYSIS OF SOLID ROCKET MOTOR CASE

Because of the complicated shape of the rocket motor case, stress analysis done by any other method is difficult, so finite element method is obviously the best choice. Hence finite element technique has been selected for the analysis purpose. There are different types of commercial FEM software's available in the market. ANSYS FEM software is one of the most popular commercial software and is used for the Finite Element Analysis of the rocket motor case [6].

In ANSYS there are three processors that are used in finite element analysis;

i) Preprocessor
ii) Solution or Processor.
iii) General Postprocessor.

3.1 Preprocessor

In preprocessor following steps involved.
-Create model or geometry
-Define element type
-Define element real constant
-Define material properties
-Define meshing controls
-Mesh the object created
-Applying boundary conditions
-Applying loads.

3.2 Solution or Processor

The solution processor has the commands that allow you to apply boundary conditions and loads and it solves for nodal solutions.

3.3 General Postprocessor

It contains the commands that allow you to list and display the result of analysis.

Geometric modeling is the process of generating three dimensional objects of the real world for the purpose of analysis, design, drafting and manufacture etc. Geometry of the part can be created in several different ways. These include starting from primitives. A key philosophy in geometric modeling is to sketch sections on work plane or on a face of an object and modify the dimensions later. Here we seek the help of AutoCAD for making sketch of a solid rocket motor case. After that we import the sketch in ANSYS as an IGES file. Then we have to create an area by picking lines. An axisymmetric model we are trying to create, so we don't create it as a full model.

ANSYS provides more than 150 various elements to be used to analyze different problems. Selecting the correct element type is a very important part of the analysis process. In ANSYS each element type is identified by category name followed by a number. PLANE183 is a higher order 2-D, 8-node or 6-node element. PLANE183 has quadratic displacement behavior and is well suited to modeling irregular meshes. This element is defined by 8 nodes or 6 nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress, plane strain and generalized plane strain) or as an axisymmetric element.

Element real constants are quantities that are specific to a particular element. It is important to realize that real constants vary from one element type to another; not all the element requires real constants.

At this point we have to define the material physical properties of the material used for the given problem. The material use for the construction of pressure vessel is Steel and its properties are as shown in Table 3.

Table -3: Material Properties of High Strength Steel.

SL No.	Property	Value
1	Modulus of Elasticity (MPa)	206
2	Poisson's Ratio	0.3

While defining mesh controls the created geometrical model is discretized into nodes and elements. The process is called meshing. The ANSYS program can automatically generate the nodes and elements, provided that you specify the element size. The element size controls the fineness of the mesh. The smaller the element size the finer the mesh.

Next we have to apply loads and boundary conditions in processor. Internal pressure and displacement constrains applied as per the deformation takes place in the motor case. In this problem we have applied the boundary conditions as shown in fig-3 and those are at lower section line DY=0 and other sections are automatically constrained because of axisymmetric modeling. The pressure applied on the internal line is 6 MPa as shown in fig-3.

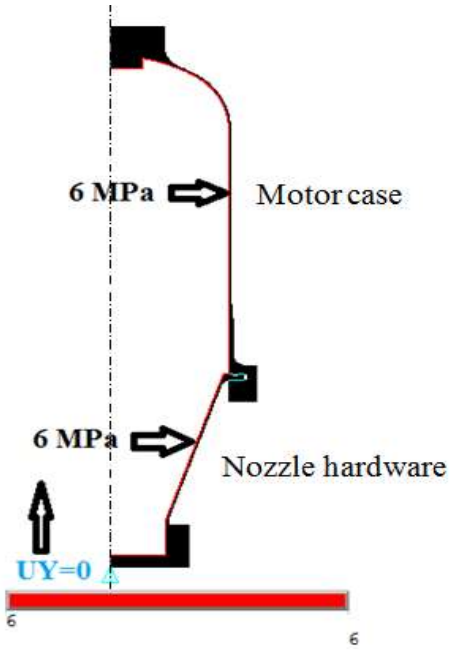

Fig -2: Internal Pressure applied and displacement constrain.

In the solution step the problems get solved. In General Postprocessor step we get the results and its display in the deformed shape and contour displays.

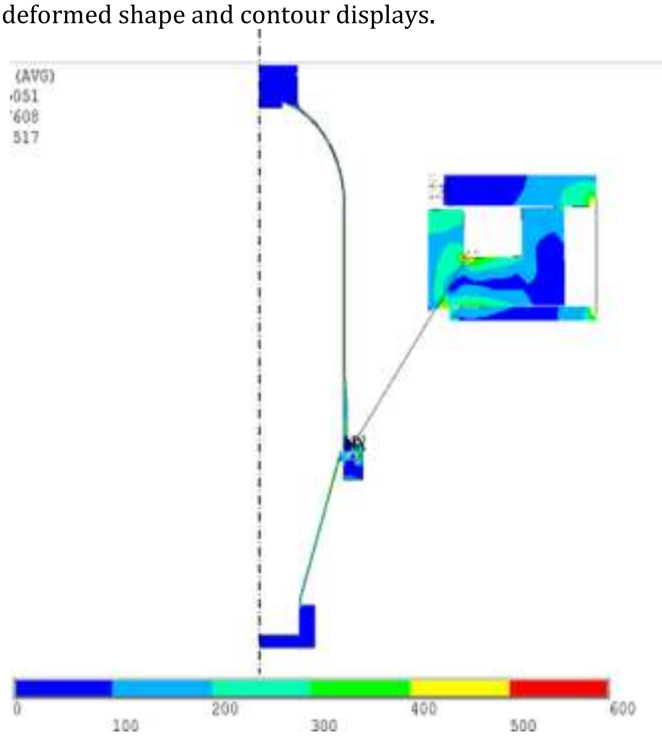

Fig -3: VonMises Stress in MPa in Solid Rocket Motor Case.

4. NUMERICAL RESULTS

Table -4: Stresses at various locations of cylindrical shell & hemispherical dome end.

Locations	Hoop Stress (MPa)	Meridional Stress (MPa)	Effective Stress (MPa)
Away from welds	444.24	220.90	392.92
Away from welds	222.07	221.88	221.75

Table -5: Result Comparison.

Stress on a Rocket Motor Case (MPa)	Circular Shell		Hemispherical Dome	
	FEA	Theoretical	FEA	Theoretical
Hoop Stress	444.24	443.5	222.07	221.75
Meridional Stress	220.90	221.75	221.88	221.75
Effective Stress	392.92	384.08	221.75	221.75

Table 5 shows the comparison of stresses in cylindrical shell and hemispherical dome for locations far away from weld joints. Close match between theoretical and FEA shows the correctness of FE Model, loads and boundary conditions.

5. CONCLUSION

With the help of finite element analysis the actual stress distributions in the different part of a rocket motor case were studied. It is found that element size affects the results of finite element analysis but after particular size of the element it does not effect on the analysis results. There is less than 3 % variation between the results by finite elements and analytical method. The effective stress is less than the yield strength of the material, so the design is safe. Stress obtained for cylinder shell and hemispherical dome end shows good agreement with numerical results. Stress analysis helps us to find out stresses at different points, which were difficult to calculate from empirical relations.

ACKNOWLEDGEMENT

I would like to thank my guides A. Yezhil Arasu, B. Kiran Kumar and Thomas Kurian for their valuable advice and technical assistance.

REFERENCES

[1] P.G. Sutton and O. Biblarz, Rocket Propulsion Elements, John Wiley & Sons, New York, 2001.

[2] D. Kozakl, I. Samardzic, A. Stoic and Z. Ivandic, “Stress Analysis of Cylindrical Vessel with Changeable Head Geometry”, Machine Manufacturing Technology, Vol.23, pp.93-104, 2006.

[3] N.D. Mundhe, “Analysis of Cracked Cylindrical Pressure Vessel by Using Experimental Approach, International Journal of Latest Trends in Engineering and Technology”, Vol.2, pp.155-160, 2013.

[4] H.H. Bednar, Pressure Vessel Design Handbook, Krieger Publishing Company, Florida, 1991.

[5] J.F. Harvey, Theory and Design of Pressure Vessel, CBS Publishers & Distributors, New Delhi, 1987.

[6] B. Praneeth and T.B.S. Rao, “Finite Element Analysis of Pressure Vessel and Piping Design”, International Journal of Engineering Trends and Technology”, Vol.3, pp.567-570, 2012.

BIOGRAPHY

“Nevin Nelson received the B.Tech degree from University of Kerala, India, in 2011. M.Tech degree from Cochin University of Science and Technology, India, in 2014. His Primary research interest is in Structural Analysis and Design”