International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 04
February 2016

Finite Element Analysis of a Pressure Cylinder Cover

Mohit Kumar[1]; Shubham Singhmar[2]; Ranveer singh jamwal[3]& Rohit Chauhan[4]

[*1-4] student, Department of mechanical engineering, Chandigarh University, India

Abstract:

A 3D solid model for the cover of a pressure cylinder is designed which can hold up a pressure of 0.5 MPa. The model is made up of aluminum alloy. It is required to find alternative material to manufacture pressure cylinder cover. This case deals with material – Engineering Plastic being substituted for aluminum alloy. The geometry is created in Catia and analysis performed in Ansys.

Keywords: ANSYS; CATIA; FEA

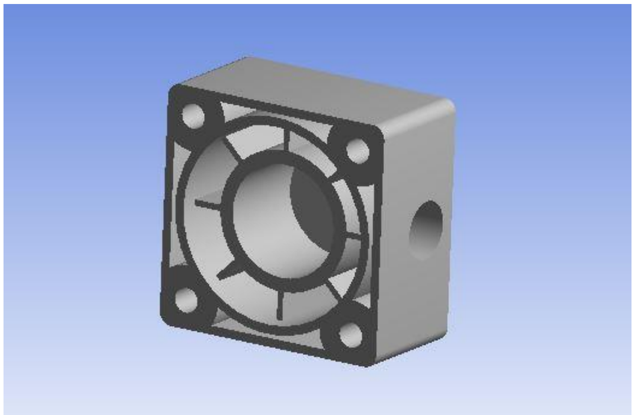
Introduction

The purpose of the simulation is to assess the possibility of replacing the aluminum alloy by a new type of engineering plastic. In this case study various parameters and technical segments come into picture which will form our areas of interest.

1. Circularity
2. Cylindrical Coordinate System
3. Mesh Refinement
   a) Tetrahedral Meshing
      i. Patch Conforming
      ii. Patch Independent
   b) Multi zone Meshing
      i. Hex/Hex-Dominant Method
   c) Sweep Method
   d) Skewness mesh metric
4. Silvers.

Circularity: Circularity of a model is defined as the difference between the radii of two concentric circular areas. It is used to control the roundness of circular parts or features. The circularity should be less than 10 micrometer; excess of circularity beyond this value may impair the tightness and cause the gas leakage.

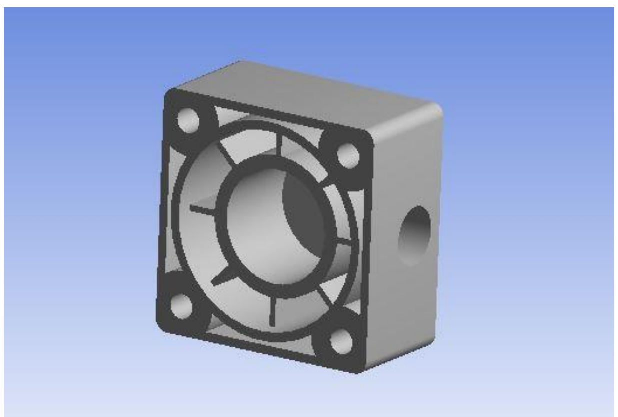
Model:



This model is made up of aluminum alloy and having following material properties:

- E = 71 GPa
- υ = 0.33

Fig. 1 Cover of Pressure cylinder (Aluminum)



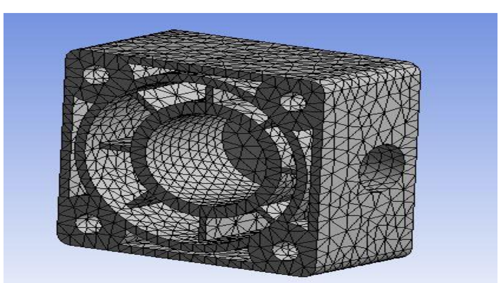
This model is made up of engineering plastic and having following material properties:

- E = 22 GPa
- υ = 0.33

Fig. 2 Cover of Pressure cylinder (Engineering Plastic)

At the onset static structural analysis is carried out with default mesh settings, a 0.5 MPa pressure is applied on the inner faces of the circularity and the frictionless support is applied to the space provided for the bolt at all four corners of the geometry. The problem was solved for Total deformation, Equivalent Stress, Structural error and Directional deformation.

Meshed Model:



The mesh generated by the default settings and the value of relevance center is given as 100.

Fig. 3 Auto meshed model with relevance

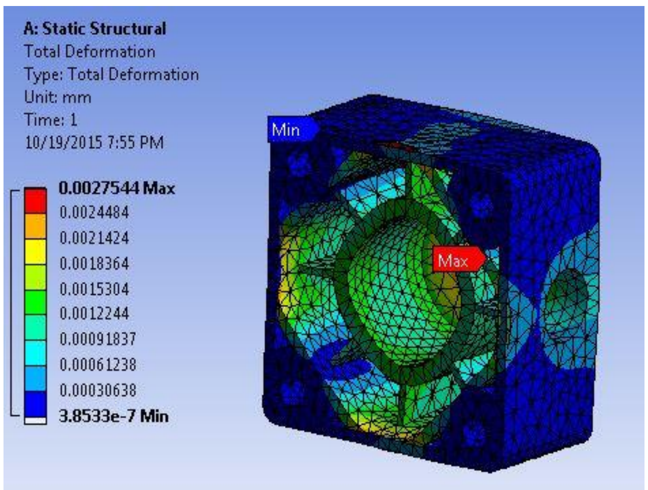
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 04
February 2016

The boundary conditions and the mesh parameters were same for both the cases and the problem was solved one by one for both materials.

Results:

For Aluminum Model:

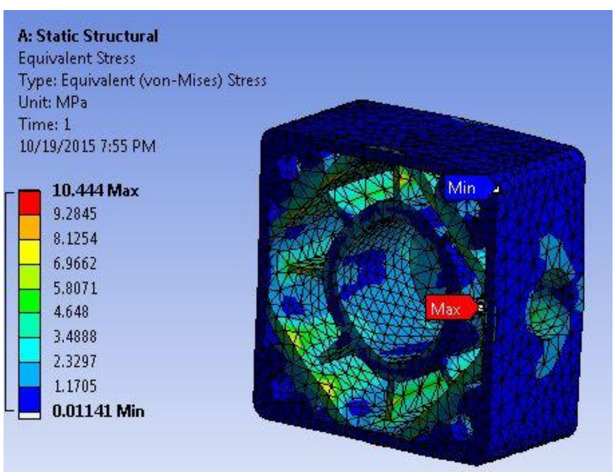
Total deformation:



The maximum deformation value obtained is 0.0027544 mm, the location of the max. and min. deformation can be seen in fig. 4.

Fig. 4 Total Deformation Plot

Equivalent Stress:

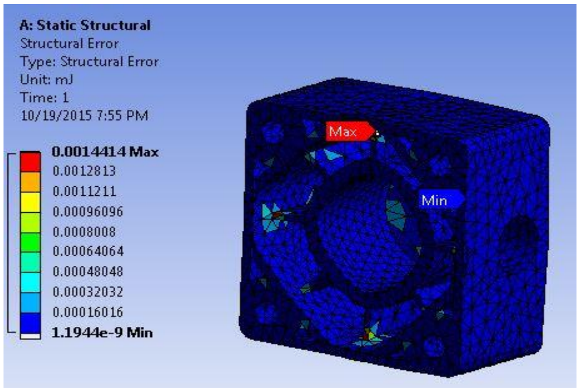


The maximum stress obtained is 10.444 MPa, and the location of max. and min. stress concentration is shown in the fig. 5.

Fig. 5 Equivalent Stress Plot

International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 04
February 2016

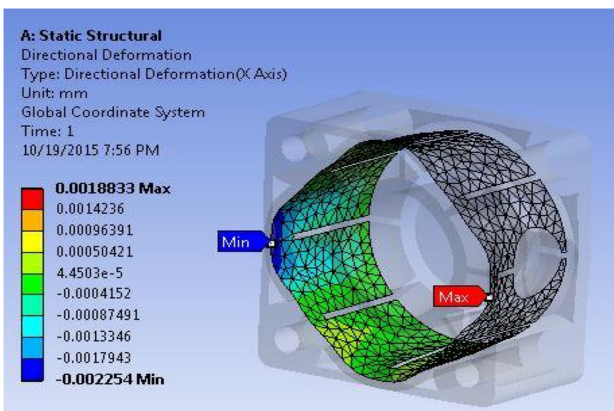
Structural Error:



The maximum structural error obtained is 0.0014414 mJ.

Fig. 6 Structural Error Plot

Directional Deformation:

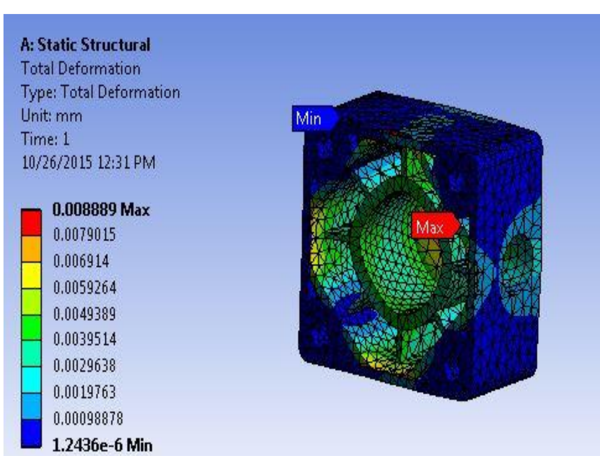


Directional deformation along x-axis of inner surface of the circularity is obtained as 0.0018833 mm.

Fig. 7 Directional Deformation Plot

For Engineering Plastic Model:

Total Deformation:



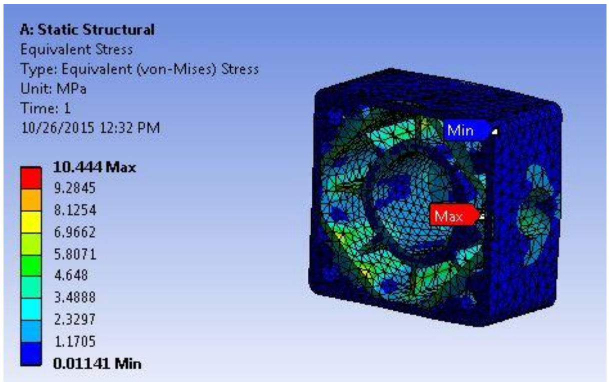
The maximum deformation value obtained is 0.008889 mm, the location of the max. and min. deformation can be seen in fig. 8.

Fig. 8 Total Deformation Plot

International Journal of Research
Available at https://edupediapublications.org/journals

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 04
February 2016

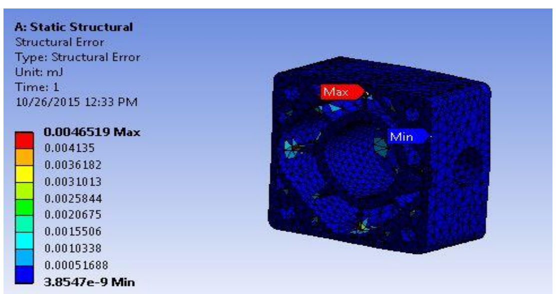
Equivalent Stress:



The maximum stress obtained is 10.444 MPa, and the location of max. and min. stress concentration is shown in the fig. 9.

Fig. 9 Equivalent Stress Plot

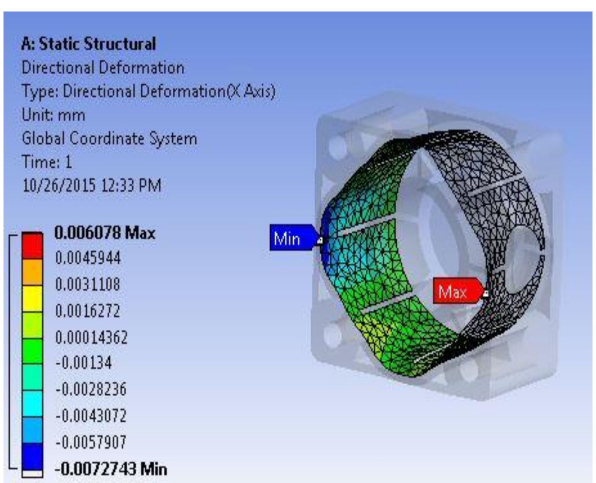
Structural Error:



The maximum structural error obtained is 0.0046519 mJ.

Fig. 10 Structural Error Plot

Directional Deformation:



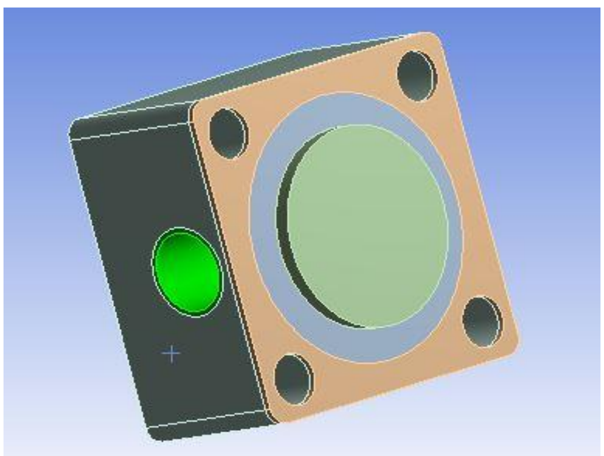
Directional deformation along x-axis of inner surface of the circularity is obtained as 0.006078 mm.

Fig. 11 Directional Deformation Plot

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 04
February 2016

When the analysis was carried out both using Aluminum Alloy and Engineering Plastic without mesh refinement, we could observe similar results with the circularity criterion satisfied. However the mesh doesn't seen to be appropriate so as to provide a final conclusion, therefore the analysis with Engineering Plastic is further carried out after refining the mesh appropriately as discussed in the section below:

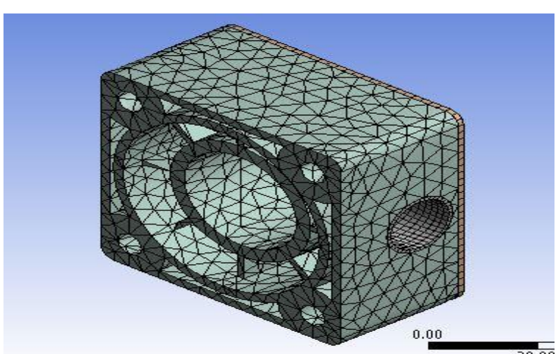
For mesh refinement, first off all the model of the pressure cylinder cover is divided into 5 sub-parts so that each part will be meshed separately and the surface distortion will be avoided. The geometry of the 5 parts is shown in fig. 12.



The five different parts of the cover of pressure cylinder can be seen in fig. 12.

Fig. 12Cover of pressure cylinder

In the next step the contacts between all five parts defined accordingly, then "Sweep" meshing method is provided to four parts, {i.e. 4 plates and one inlet except wall} and the wall is meshed with the "Patch Independent with tetrahedron" method, after generating the mesh with the above mentioned methods, we found the mesh discontinuity on inlet in the form of "silver", so we repaired this mesh defect in "Design Modeler" and then re-meshed the cover and we found that all the silver defects were removed and the mesh was appropriate. The meshed model of the cover with mesh details is shown below in fig. 13.



The mesh refinement is done by using "sweep" and "Patch Independent" method. The mesh details are:

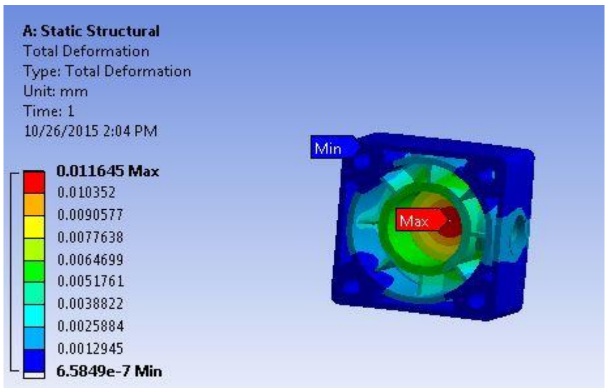
- Nodes = 22726

Fig. 13Meshed Model of the Cover of

After mesh refinement the number of nodes and elements were reduced and the maximum value of skewness mesh metric was 0.89 which was less than 0.95, it means the mesh is appropriate.

The problem is solved again for the same results and the results are shown below:

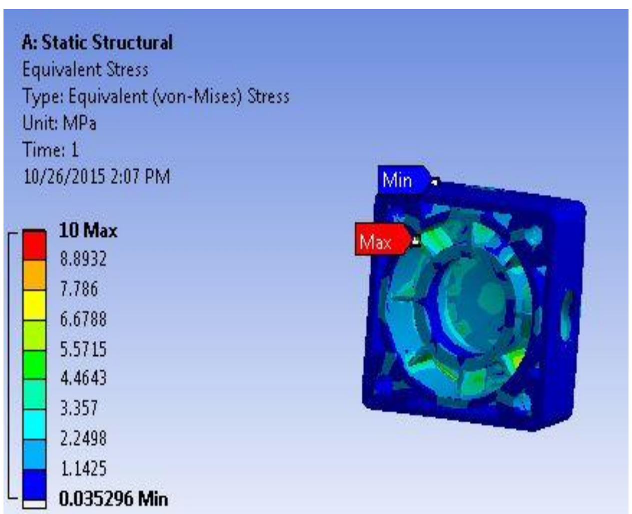
Total deformation:



The maximum deformation value obtained is 0.011645 mm, the location of the max. and min. deformation can be seen in fig. 14.

Fig. 14 Total Deformation Plot

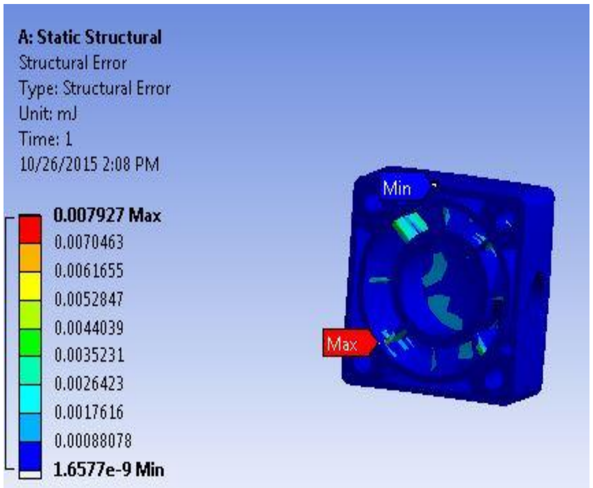
Equivalent Stress:



The maximum stress obtained is 10MPa, and the location of max. and min. stress concentration is shown in the fig. 15.

Fig. 15 Equivalent Stress Plot

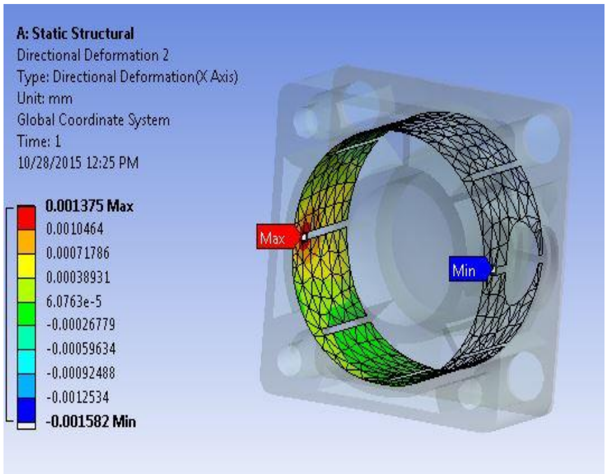
Structural Error:



The maximum structural error obtained is 0.007927mJ.

Fig. 16 Structural Error Plot

Directional Deformation:



Directional deformation along x-axis of inner surface of the circularity is obtained as 0.001375 mm.

Fig. 17 Directional Deformation Plot

Conclusion:

After analyzing all the above results for Aluminum Alloy and Engineering Plastic, we observed that the directional deformation of the circularity region reduces in case of engineering plastic which also reduces the risk of gas leakage and the value of 0.001375mm is much lesser than the value of 10 micron.

Hence the Engineering Plastic material is safe for the cover of pressure cylinder and it can easily replace the aluminum alloy.

References

[1] Stresses in Large Horizontal Cylindrical Pressure Vessels on Two Saddle Supports by L. P. Zick.Original paper published in September 1951 "THE WELDING JOURNAL RESEARCH SUPPLEMENT."

[2] Location of 'reboiler return inlet nozzle' and its effect on Distillation Column article@processenggservices.com, 005-2009 Process Engineering Services.

[3] Optimal wall-thickness of the spherical pressure vessel with respect to criterion about minimal mass and equivalent stressdražankozak, josipsertić.ANNALS OF THE FACULTY OF ENGINEERING HUNEDOARA – 2006 TOME IV.

[4] Fascicole 2 Understanding How Changes in Pressure Vessel Codes May Impact Pressure Vessel Performance .Patrick J. Sullivan Highlander Engineering Services 2008, PLLC Argyle, NY

[5] Results of fea analyses at nozzle/shell junctions subjected to external loads steven R. Massey charlie S. HsiehBlack & veatch pritchardInc-2009

[6] Singhmar, S., Sahil, S. G., & Sharma, A. GLOBAL JOURNAL OF ENGINEERING SCIENCE AND RESEARCHES.

[7] Biswal, V., Kumar, R., Singhmar, S., & Kumar, M. INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY FINITE ELEMENT ANALYSIS OF STACKED VESSELS.

[8] Kumar, Mohit, Shubham Singhmar, and Shubham Gupta. "Computational Fluid Dynamics Analysis of Two Dimensional Condenser." *International Journal of Research* 2, no. 11 (2015): 746-752.