Static Analysis of Compass Prototype (Part A) Build with FDM Process

JEETENDRA KUMAR
Department of Mechanical Engineering RPSIT, Patna (Bihar)

Abstract- ***Main objective of the research work is to develop a newly designed compass consisting of two parts i.e. Part A and Part B and find the optimum build orientation of the R.P. model of compass build with FDM process. General compass don't have scale in it. For drawing the circle, scale is needed to measure the length of the radius or diameter. R.P. models of compass are produced with scale provided in it. To check the functionality of the model, prototype is developed. It is very necessary that prototype should be tough and strong enough to bear load acting on it while using and it should be durable. FEM analysis is conducted to find the strength of the model material. Stress analysis was done using ANSYS software for ABS P400 and Polycarbonate – ISO (Plastic material) to choose the best model material for the R.P. model. In this article static analysis is shown only for Part A.***

Indexed Terms- ***FEM- Finite Element Method. RP- Rapid Prototype***

I. FINITE ELEMENT ANALYSIS (STATIC ANALYSIS OF MODEL)

Static analysis has been carried out on the CAD model to check the strength of the R.P model. Analysis is conducted using ANSYS software. ABS material and polycarbonate-ISO is considered as the material for the analysis. Properties of both the materials are provided as input for getting the stress result.

- Mechanical properties of Polycarbonate- ISO which are required for the analysis are

a) Young's modulus= 2 Gpa
b) Poisson's ratio= 0.37

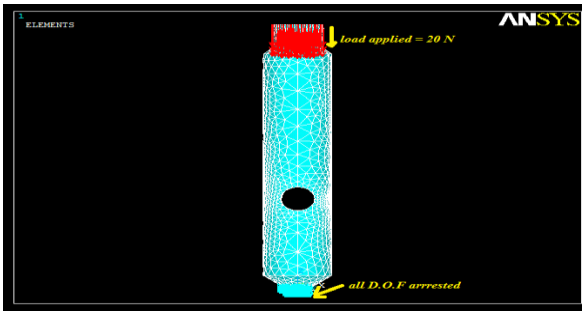
- Mechanical properties of ABS which are required for the analysis are

a) Young's modulus = 2.3Gpa
b) Poison's ratio = 0.35

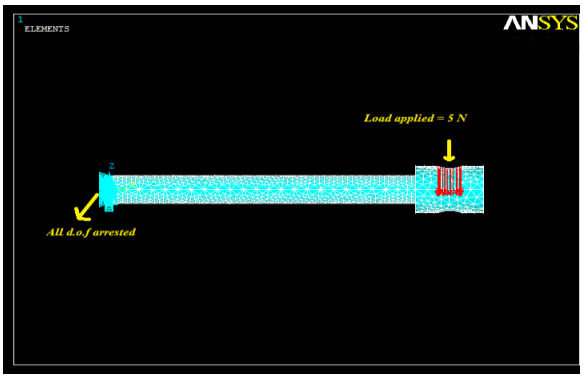
Load and Boundary Condition

Analysis is performed on part A only in this article.

a) Part A (Upper Portion)
✓ Load applied = 20 Newton,
✓ All degree of freedom has been arrested at the lower portion of Part A as shown is fig.



Part A



Part B

Fig.: Load and boundary condition

II. STATIC ANALYSIS PROCEDURE FOR 3-D SOLID MODEL

- Preferences - Structural
- Pre-Processor
- Element Type - Solids - Options
- Real Constants - No real constants
- Material Properties - Material Models.

✓ (Young's Modulus & Poisson's Ratio).

- Modeling – Create – Volume-

Or create KP's and then create volume.

- Meshing– Mesh tool - Mesh.
- Loads - Define Loads – Apply load
- ✓ Displacement,
- ✓ Force and momentum
- Solution - Solve - Current LS
- General post-processor

- *Deformed Shape.*

Element Table-Define-Plot-List.
PR-Contour-Line Element Result.
List Results
Query Results-
Graph- Path Operations-Define-By
Nodes-Map onto Path-Plot on Path-On
Graph.

- Apply Displacements Select – Entities – Nodes – By location – X co. Or – Min. Max – Apply – Plot Nodes – and then apply displacements.
Then Select – Everything – Plot Elements.

III. RESULT AND DISCUSSION

Stress analysis is done on part A of compass to check the strength. Result is shown through plots.

Static Analysis Carried Out By Using ANSYS for PART A

- Material used – ABS P400
- Load applied – 20 Newton
- Output requested: Nodal solution
- Von Misses stress
- Principal stress
- stress intensity
- z -component of stress
- Displacement vector sum
- z- component of displacement

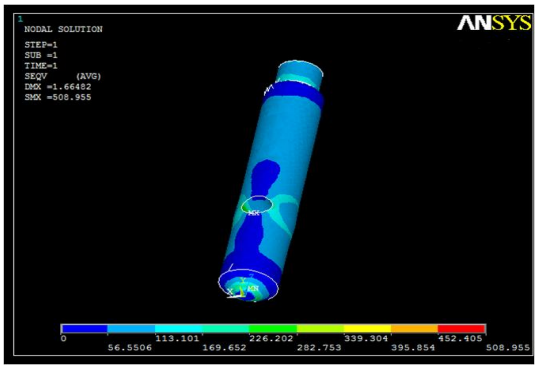


Fig. : Simulation diagram of Von mises stress

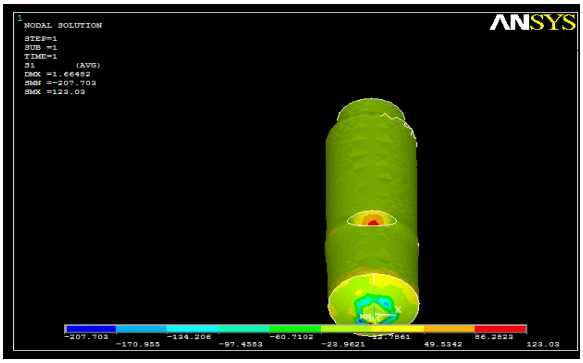


Fig. : Simulation diagram of 1st principal stress

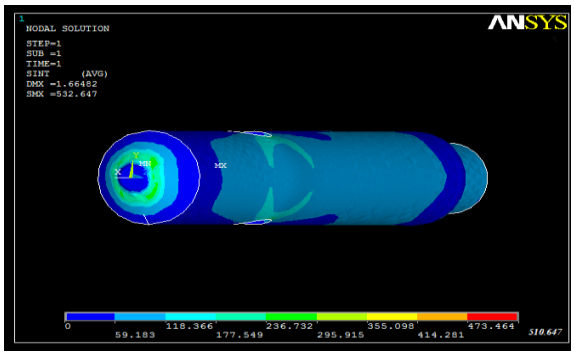


Fig. : Simulation diagram of Stress intensity

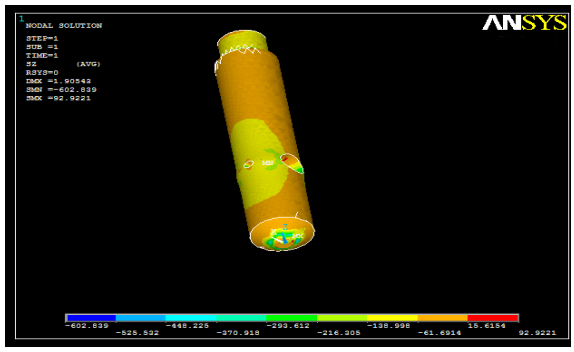


Fig. : Simulation diagram of Z- component of stress

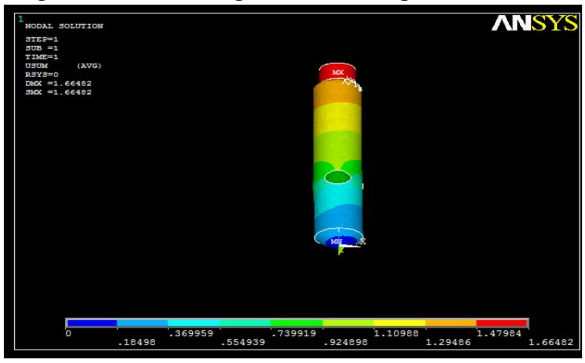


Fig. : Simulation diagram of Displacement vector sum

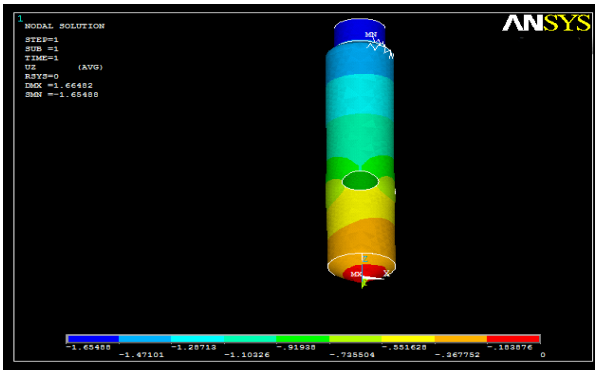


Fig. : Simulation diagram of Z - Component of displacement

IV. STATIC ANALYSIS CARRIED OUT BY USING ANSYS FOR PART A

- Material used – Polycarbonate-ISO
- Load applied – 20 Newton
- Output requested: Nodal solution
- Von Misses stress
- Principal stress
- stress intensity
- z -component of stress
- Displacement vector sum
- z- component of displacement

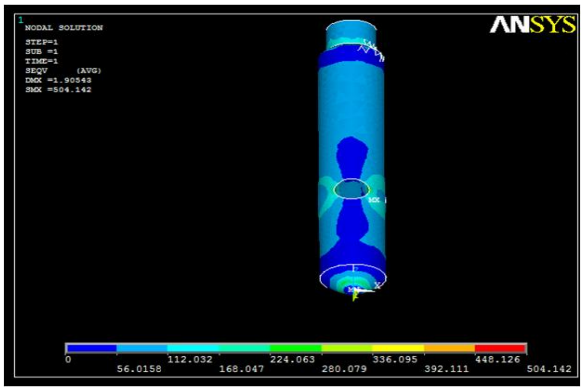


Fig. : Simulation diagram of Von mises stress

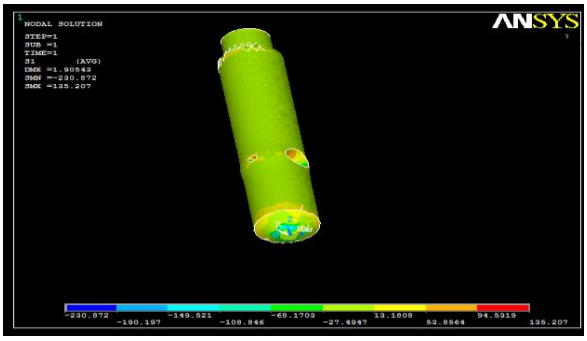


Fig. : Simulation diagram of 1ST Principal stress

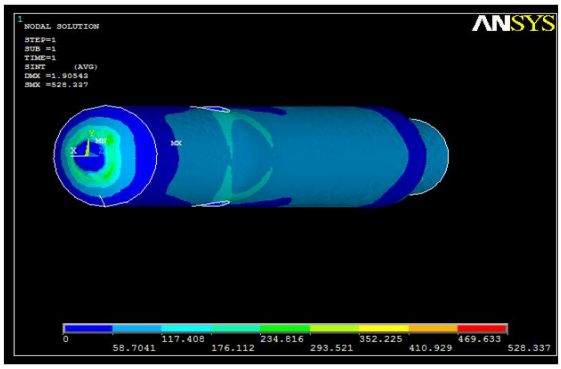


Fig. : Simulation diagram of Stress intensity

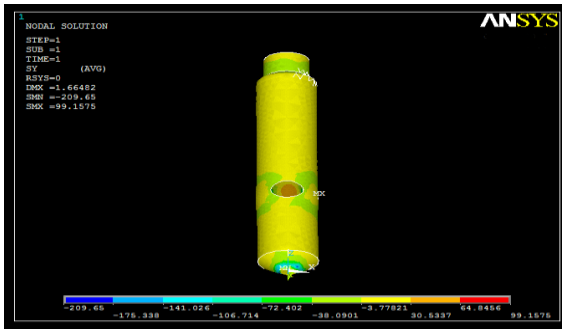


Fig. : Simulation diagram of Z- Component of stress

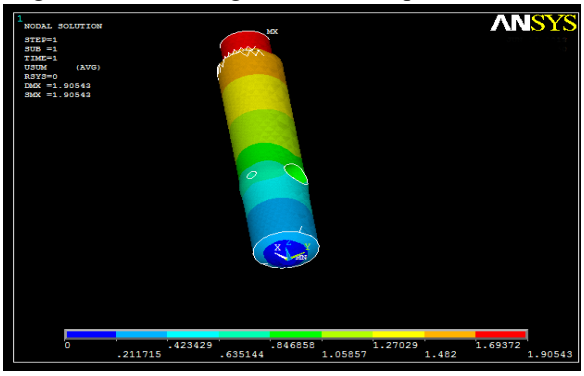


Fig. : Simulation diagram of Displacement vector sum

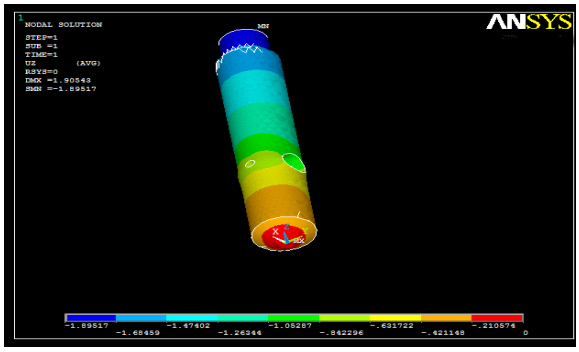


Fig. : Simulation diagram of Z- Component of displacement

Table: Static Analysis Result for Part A

PROPERTIES	ABS P400	POLYCARBONATE -ISO
Von mises stress	508.955	504.142
Vector sum of displacement	1.66482	1.90543
Stress intensity	510.647	528.337
Z component of stress	92.9221	99.1575
Z component of displacement	0	0

V. FUTURE WORK

Static analysis of Part B of compass will be available in the next article.

REFERRENCES

[1] Wohler's T.2008 Wholer's Report 2008 – State of industry Annual worldwide progress Report Wholer's Associates inc.

[2] CzajkiewiczZ. 2008 " Direct Digital Manufacturing – New Product Development and Production technology", ORGAMSZ,2(2),pp.29-37.

[3] http://finance.yahoo.com/news/RedEye-On-Demand-is-First-bw-14255281.html

[4] Fused deposition modeling (FDM) Material Properties Characterization Me 222 Final Project, Hohn Mechael, Brock Michael Montero Dan Odell, shad roundry Spring Semester.

[5] P.T. Lang, S.Y. Chou, L.L. Chent, D, Gemmill, Determing fabrication orientation for rapid prototyping with stereolithography apparatus, Computer – Aided design 2(1) (1997) 53-61.

[6] P.Alexander, S. Allen, D.Duttta, Part orientation and build cost determination in layered manufacturing, Computer –Aided Desgin 30 (5) (1998) 343-356

[7] Hur J, Lee K. The development of a CAD environment to determine the preferred build up direction for layered manufacturing, Int J Adv Manufa Technol 1998: 14 (4):2 47-54