DOI: http://dx.doi.org/10.4314/star.v4i4.00
ISSN: 2226-7522 (Print) and 2305-3372 (Online)
Science, Technology and Arts Research Journal
Sci. Technol. Arts Res. J., April-June 2018, 5(2): 029-043
Journal Homepage: http://www.starjournal.org/

Original Research

Finite Element Analysis of a Base Stand Using Different Materials

Aisha Muhammad[1, 2*], Ibrahim Haruna Shanono[3, 4]

[1]Department of Mechatronics, Faculty of the Engineering, Bayero University Kano, Nigeria
[2]Department of Electrical, Faculty of Engineering, Universiti Malaya, 50603 Kuala Lumpur, Malaysia
[3]Department of Electrical, Faculty of the Engineering, Bayero University Kano, Nigeria
[4]Faculty of Electrical and Electronics Engineering, Universiti Malaysia Pahang, 26600 Pekan, Malaysia

Abstract

In this paper, deformation, Von-Mises stress and failure analysis were carried out on a base stand using ANSYS workbench. The investigation is performed using four different materials, i.e. structural steel, aluminium alloy, carbon fibre, and copper alloy with an applied loading force of 1000N at the tip of the structure. The results obtained were compared and analysed, to identify the best material capable of withstanding the subjected force. This technique highlights a complete reaction of the structure to the loading force, therefore providing the room for structural optimisation to reduce the risk of unexpected failure and unnecessary material wastage. The structural design was carried out using SolidWorks software and then imported into the ANSYS workbench for analysis.

Article Information

Article History:
Received : 10-01-2016
Revised : 16-03-2016
Accepted : 23-03-2016

Keywords:
Finite Element Analysis,
Base Stand,
ANSYS,
Deformation,
Von-Mises Stress

***CorrespondingAuthor:**
E-mail: Aisha Muhammad[1]ayshermuhd@gmail.com

INTRODUCTION

A base stand is a structurally designed pedestal for holding or supporting an object in a specific desired position. Depending on the weight of the object it supports, it can be classified as lightweight or heavy-duty base stand, which could be use for domestics or industrial applications. Its area of application and operating conditions are the vital factors considered by the production companies to maximize profits without compromising product quality and reliability. These factors ranges from environmental, stress and loading properties etc., which are used to determine the base stand shape, weight, size and the type of material used for production.

Finite Element Analysis (FEA) involves the simulation of a physical engineering structure using numerical technique called Finite Element Method (FEM). It involves sub-dividing the structure into smaller elements called Mesh. Several numbers of design analysis under different constraints are performed with FEA. In designing a complex structure, computer-aided design (CAD) is used,and then the analysis is performed using Computer-aided engineering (CAE) software such as ANSYS, which is the most widely used software amongst others. Further investigation can also be doneto improve the design for optimal performance and lifespan with regards to design failure (Xiaolin & Yijun, 2014). ANSYS Workbench has a user-friendly platform. It provides two-way access to most used computer-aided design systems.

The solid modelling software defines the geometry of the solid structure. The meshing of the elements of the structure is done using the ANSYS workbench. For a more accurate result, finer meshing sizes are used. The design constraints are applied in this study and then finally the simulation results are retrieved after being solved by the ANSYS software (Qiongying & Yushi, 2014).

This study aims at analysing the deformation, stress (Von-Mises) and failure of a base stand. The analysis is carried out in ANSYS static structural, mechanical solver with a tensile force of 1000N acting along the three legs of the fixed positioned base stand. The said analysis was conducted on structural steel, aluminium alloy, carbon fibre, and copper alloy to get the most suitable material for the said application.

MATERIALS AND METHODS

Finite Element Method

A numerical method for solving a problem to an approximately exact solution described by partial differential equations is known as Finite Element Method (FEM) (Xiaolin & Yijun, 2014). FEM is used to solve the problem by subdividing the engineering structure into smaller elements. An excellent, desirable feature of FEM is its ability in dealing with complex engineering structures and their constraints.

The procedure and steps used in solving the problem using FEM approach are depicted in Figure 1 below.

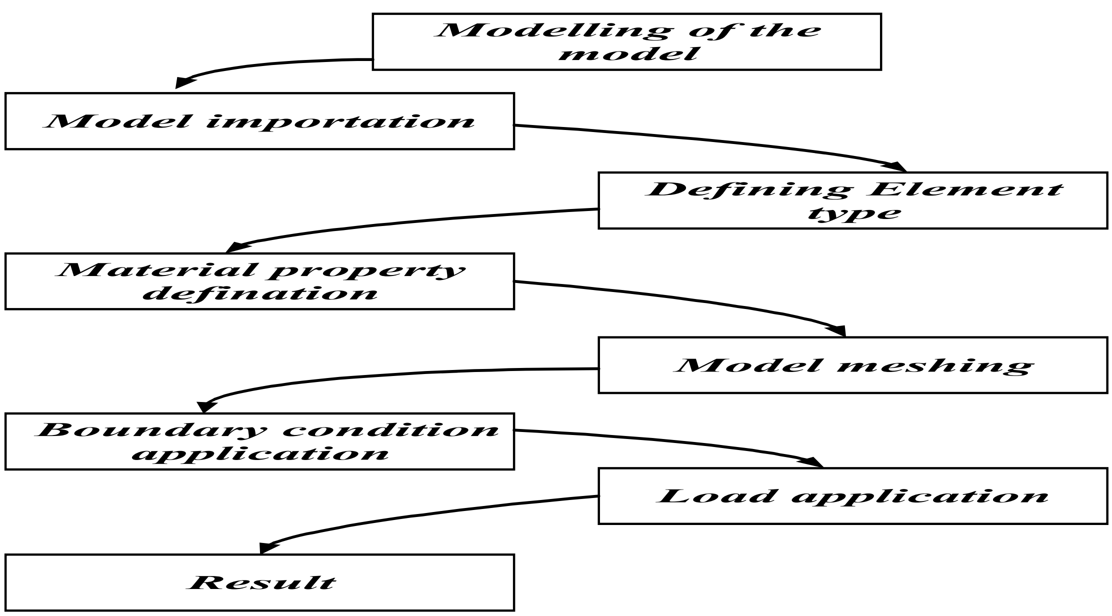


Figure 1: FEA flowchart.

Base Stand Modelling in CAD

The CAD design of the structure is done using Solid-Works software, which is then exported into ANSYS workbench for further analysis. Solid-Works have a user-friendly interface, which allows the design/ modelling of very complex structures. Figure 2 below shows the sketched base stand structure with the head, pin and base assembled to one piece.

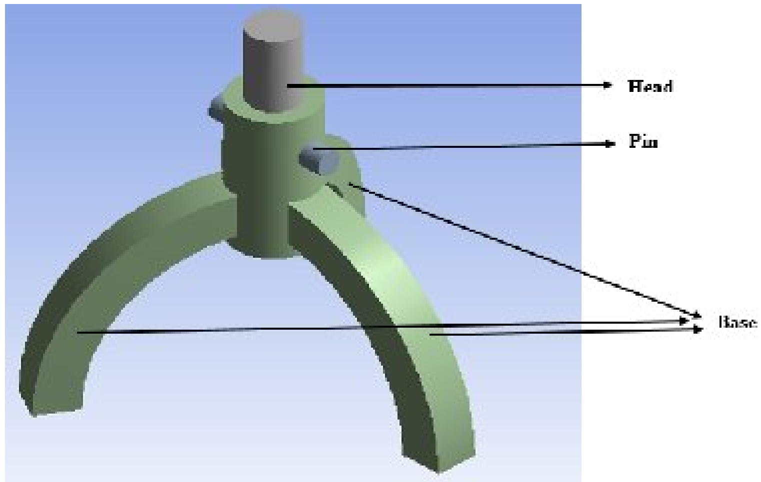


Figure 2: Base stand model in Solid works

Fınıte Element Analysıs usıng ANSYS

Analysis of any structure begins with the geometry definition which is defined based on the type of simulation analysis that is to be carried out. Since our analysis focuses on stress and deformation, the exact FEA model used is a substructure. There are two main ways of introducing a three-dimensional structure into ANSYS software. First is by saving it as an Initial Graphics Exchange Specification (IGES) format then import it into the ANSYS workbench, the second method is by building the entire structure in the ANSYS workbench (Wang*et al*., 2012) (Kim *et al*., 2007) (Janq & Lee, 2002) (Han *et al*., 2002).In this paper, the analysis is performed via geometry cell by importing the geometry from a CAD in the IGES format into the software as shown in figure 3. The hub is 35mm from the ground level, and 18mm tall. The holder is 36mm long with radius of 6 and 3mm for the holder and pin respectively. The geometrical construction showing its dimension is shown in figure below.

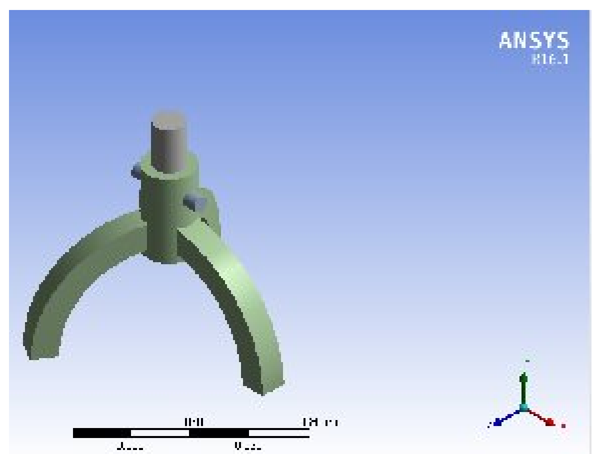


Figure 3: Imported Geometry in ANSYS window

Material properties are defined based on the type of simulation analysis to be performed.

For an efficient and qualitative analysis of material, the material properties need to be correctly and carefully entered. These materials which can be either linear or nonlinear, isotropic or orthotropic, constant or temperature dependent needs to be define correctly. Depending on the aim of the analysis, some mechanical properties such as density, strength and coefficient of thermal expansion definition is optional (Barbero, 2014). Knowing and declaring the correct value of material property is very useful for design analysis purpose. The different types of material indicated by different density will show different analysis result outcome as well. The young's modulus of a material alternatively called modulus of elasticity is a numerical constant that describes the elasticity and measures the capability of a solid to withstand changes when subjected to tension or compression in a certain direction. The higher the young's modulus, the stiffer (i.e. how it deflects under load) is the material which will require a much higher amount of load to deform it. Poisson's ration which is the ratio of compression to expansion of material together with the young's modulus (ratio of stress to strain) defines the strength and nature of how a material structure deforms based on a certain constraint. Material deformation to due uniform volume and opposing forces are described by the bulk and shear modulus respectively Two other important properties that determines when the material losses its elastic behavior and the maximum stress a material can undergo are the yield and tensile strength respectively. After importing the geometry, the definition of an element and material properties is carryout. As outlined earlier, four materials are used in the analysis. Eachmaterial has seven essential properties useful in the analysis and are obtainedfrom the engineering data. Table 1 below presents those properties.

Table 1: Properties of materials

Material Type/ Property	**Structural steel**	**Aluminum alloy**	**Carbon fibre**	**Copper alloy**
Young's Modulus	2E+11 Pa	7.1E+10 Pa	1E+11Pa	1.1E+11Pa
Density	7850 kgm^-3	2770 kgm^-3	1600 kgm^-3	8300 kgm^-3
Poisson's ratio	0.30	0.33	0.10	0.34
Bulk modulus	1.6667E+11Pa	6.9608E+10Pa	4.1667E+10Pa	1.1458E+11Pa
Shear Modulus	7.6923E+10 Pa	2.6692E+10Pa	4.5455E+10Pa	4.1045E+10Pa
Tensile strength	2.5E+08 Pa	2.8E+08Pa	75.85N/mm^2	2.8E+08Pa
Ultimate shear strength	4.6E+08 Pa	3.1E+08Pa	600MPa	4.3E+08Pa

The breaking down of model or structure into smaller elements to analyse each of the components is known as meshing (Talikoti *et al*., 2016). It is a discrete realisation of the structure, which helps in solving the exact model solutions. For a smaller meshing size, the computational time is higher so also the accuracy of the analysis result (Qiongying & Yushi, 2014). ANSYS is a great control tool for meshing(Yu & Huang, 2005). The classification of meshing tools in ANSYS is (Wang *et al.,* 2012).

a. unit size control
b. Level control of intelligent division
c. Thinning grid control
d. Shape settings of meshing
e. Grid partition.

Taking into consideration the setup time and computational expense, with the speed and ease of use, a free mesh type is used. The default meshing control is used having a relevance value of +100 with a medium smoothing number of iteration. Using Table 1 above, the model properties are set in the software followed by dividing the structure or model into finer number of nodes and elements.The meshing size is set to default as determined by the ANSYS software.

The figure below shows the meshed structure of the base stand.

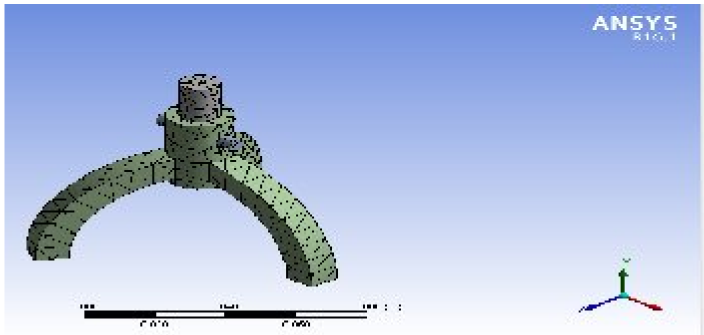


Figure 4: Meshed structure

Constraints such as fixed support and forces are applied to the model after the meshing is done, this is very important and serve as a primary step required in the analysis (Talikoti *et al.*, 2016). Constraints are set in a manner conforming to the real-life situations (Wang*et al.,* 2012). The boundary condition used in this analysis includes support and loads. The support types used for the 3D structure are a fixed support that prevents movement or deformation of the structure geometry. The type of load used is a static force. To determine how the contact bodies are going to move relative to each other, a no separation contact is used for the connection contact for frictionless sliding of the faces without separation of the faces in contact.The based stand is set fixed on it three base legs while a tensile force of 1000N is applied to the top holder face as shown in the figure 5a and 5b below.

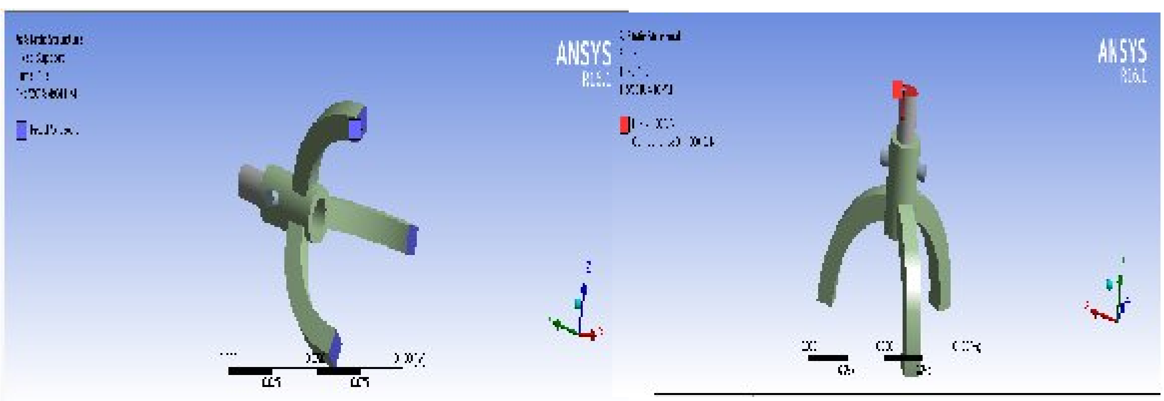


Figure 5: (a) Fixed support application (b) Load constraints application.

RESULTS AND DISCUSSION

Due to the unexpected failure of engineering structures, analysis becomes very significant to ascertain the safety of these structures. Such failures or deformations when exposed to loads over a specific duration of time result in the sudden collapse of the structure. Hence, this work assesses the deformation, Von-Mises stress and safety factor of a base stand model against different materials. Under the currentconstraints, the analysis results are shown in the figures below.

To assess the reason for failure, how and where it occurs, and the way to prevent it, simulation in Finite Element analysis is done.

Therefore, an essentialgoalof performing the analysis is determining a safer design. Failure of the whole or part of the structure leads to high risk of life and financial loss. Just like in the human context, when the human body does a lot of work, it becomes stressed, sick and finally nervous breakdown may occur. Also, in an engineering structure, failure may occur when a structure is subjected to a high amount of stress. The amount of pressure in an engineering model that happens when it is exposed to external force or load is termed as stress, which indicates that the applied load is a function of the amount of stress.

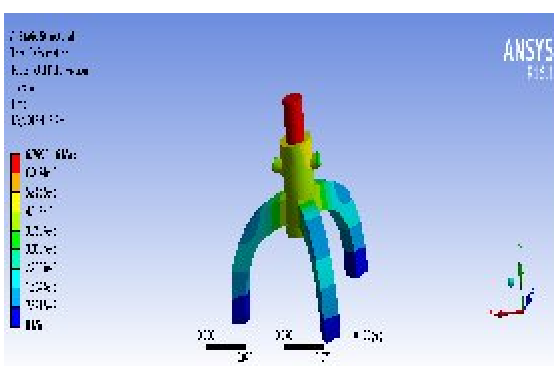
A designer uses Von-Mises stress analysis to ascertain the failure of his design structure. Failure is inevitable when the strength of the material used is less than the maximum value of the stress. Von-Mises stress is ideally used for ductile materials.

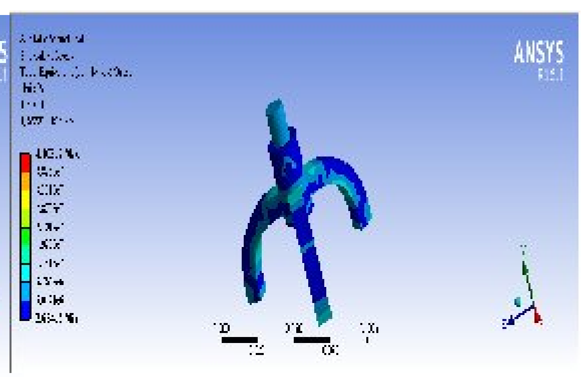
The safety factor is the ratio of material strength to the maximum stress in a structure. It describes the load carrying ability of a structure to the actual loads. Designers use it in assessing if a design is safe. It indicates that the stress of a point in a structure or model is higher than the material strength. The calculation of safety factor involves the yield strength; therefore, it becomes necessary to declare this parameter prior the simulation in the material properties.

Simulation results for structural steel

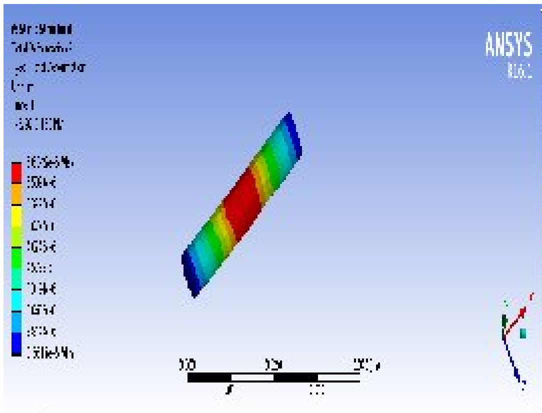
Below are the results obtained for structural steel material. Figure 6a shows the deformation plot when the base stand is subjected to the required load. The maximum deformation stands at 6.7695e-006m which is indicated by the red colour at the tip of the holder and the minimum deformation of 0Pa located at the base legs. The next image (6b) shows the stress distribution level across the base stand surface. There appeared to have no overstressed visible region. That results in the need for analysing the deformation and stress level on the non-visible/hidden portions like the pin and holder. Figure 6c to 6f depicts the pin and hole deformations and stress respectively. The pin maximum deformation and stress occurs at it centreregion, which is the point of contact with holder hole. Similarly, the holder, experience maximum deformation at its tip and maximum stress at the vicinity of contact with the pin.

Figure 6g is the safety factor (SF) level, which shows that no region of the material is overstressed, most of it lies within the maximum SF level of 15.

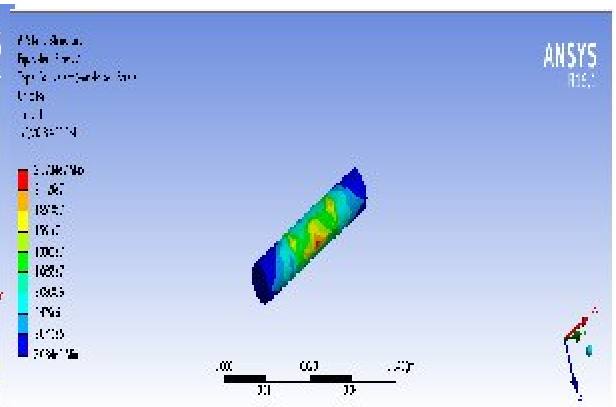




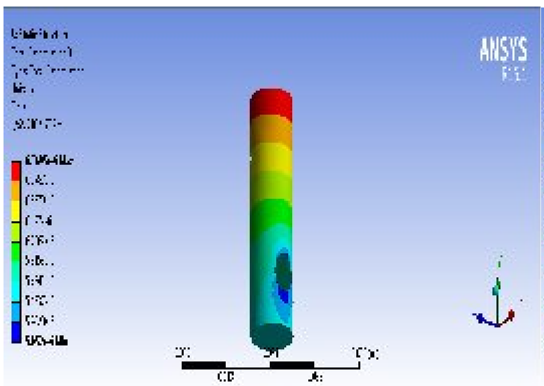
(a) (b)



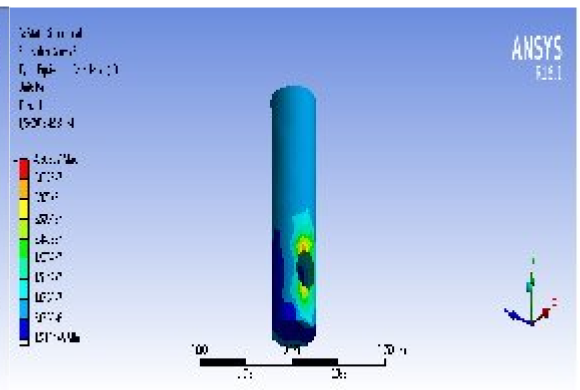
(c)



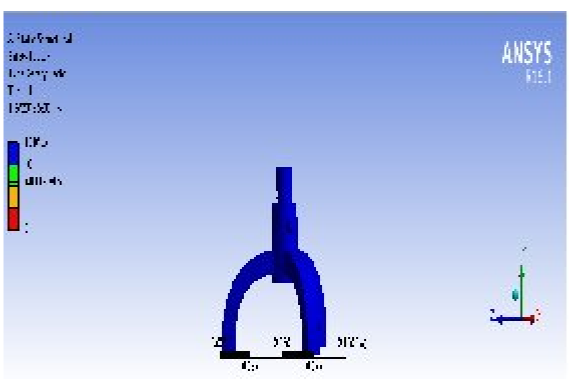
(d)



(e)



(f)



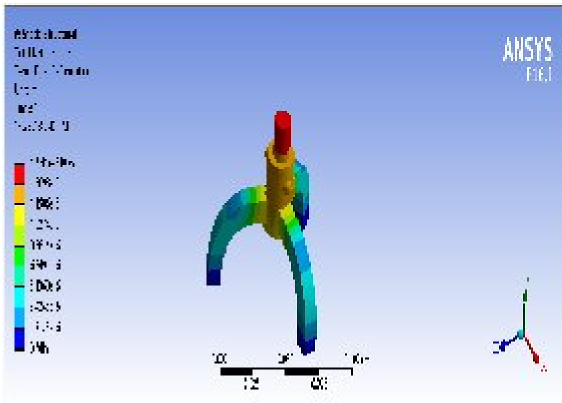
(g)

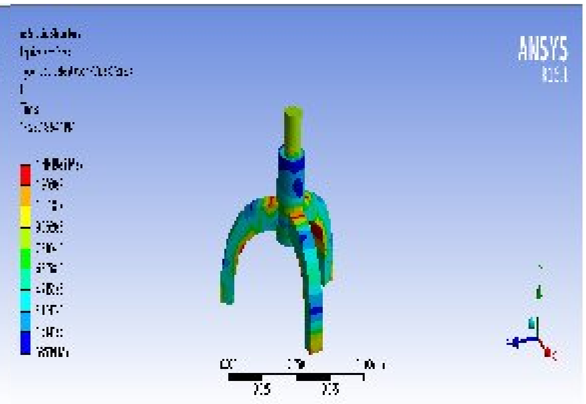
Figure 6 (a) Base Total deformation (b) Base Equivalent stress(c) Pin Total deformation (d) Pin Equivalent stress (e) Holder total deformation(f) Holder equivalent stress (g) Safety factor

Simulation results for Aluminium alloy

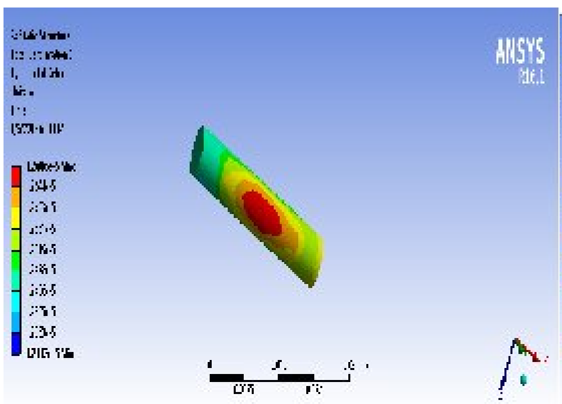
This section presents the results when Aluminum alloy is used as the structural material. There appeared to see significant changes concerning the prior (structural steel), even though it is subjected to the same amount of loading force. Figure 7a shows the deformation property of the material having 1.541e-005m as the maximum deformation, which occurs at the same points with the previous material. Figure 7b shows the stress distribution in the material which indicatesmaximum stress of 1.4019e+007Pa as shown by the red spots regions.

The pin and the holder were also analysed, indicating the area of maximum and minimum deformation and stress level as depicted in figure 7c down to 7f. Figure 7g is the safety factor plot, which suggests that the applied external stress is higher than the material strength. Therefore it is not safe to used such material under such load. The structure has safety factor less than unity.

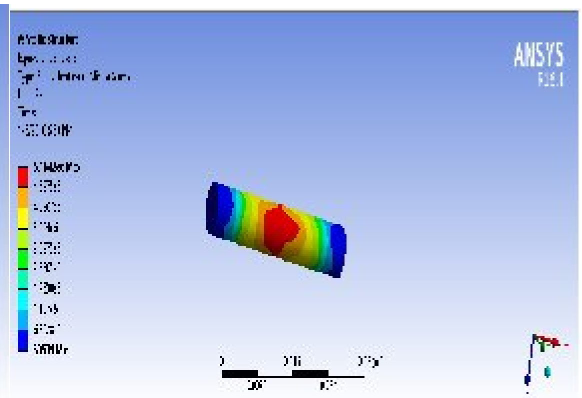




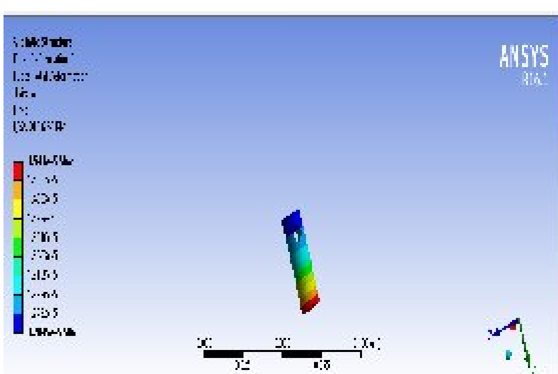
(a) (b)



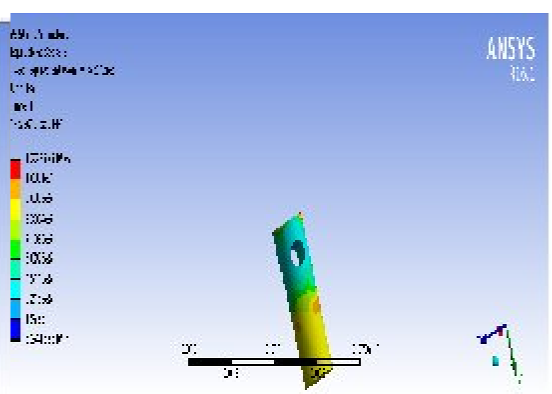
(c)



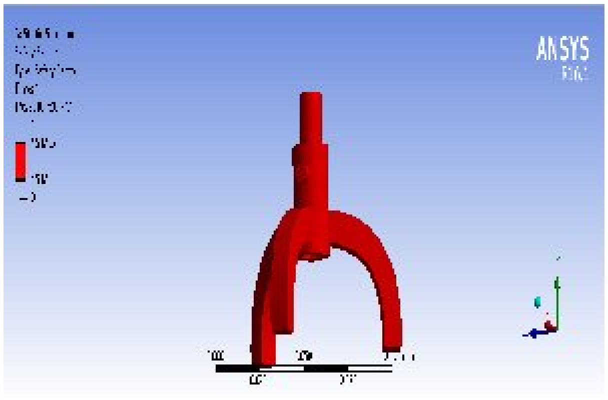
(d)



(e)



(f)

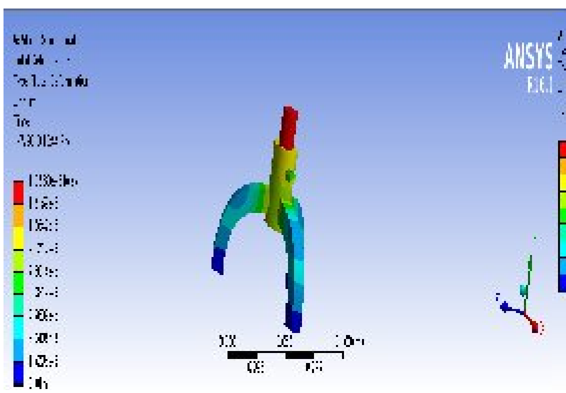


(g)

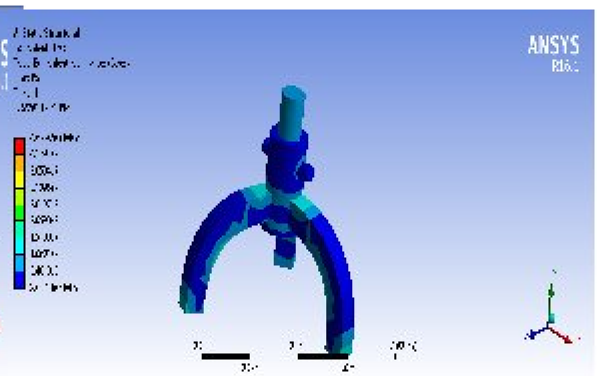
Figure 7. (a) Base Total deformation
(b) Base Equivalent stress
(c) Pin Total deformation
(d) Pin Equivalent stress
(e) Holder total deformation
(f) Holder equivalent stress
(g) Safety factor

Simulation results for Carbon fibre

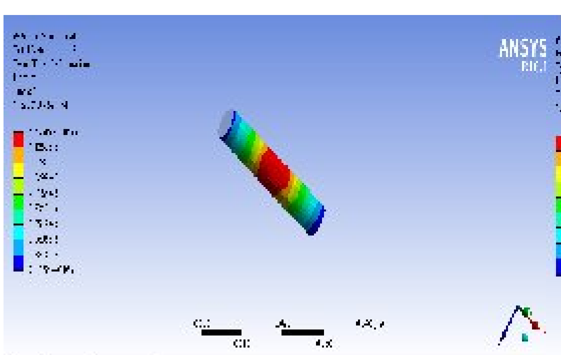
This section presents the results with Carbon fibre used as the structural material. Figure 8a shows the deformation property of the material having 1.3683e-005m as the maximum deformation, which occurs at the pin tip and base legs respectively. Figure 8b shows the stress distribution in the material with the maximum stress of 4.6992e+007Pa. The pin and the holder were also analysed, indicating the region of maximum and minimum deformation and stress level as depicted in figure 8c down to 8f. Figure 8g is the safety factor plot, showing the minimum SF to lies within the range of 5-10 at the holder tip and base legs



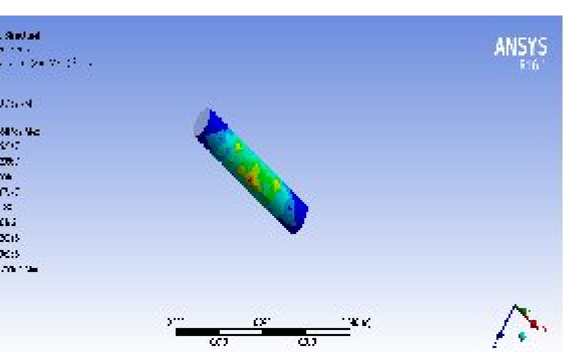
(a)



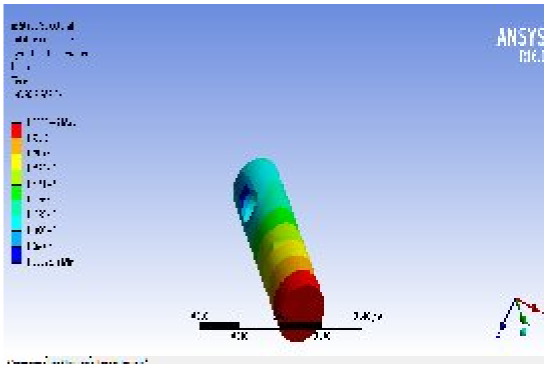
(b)



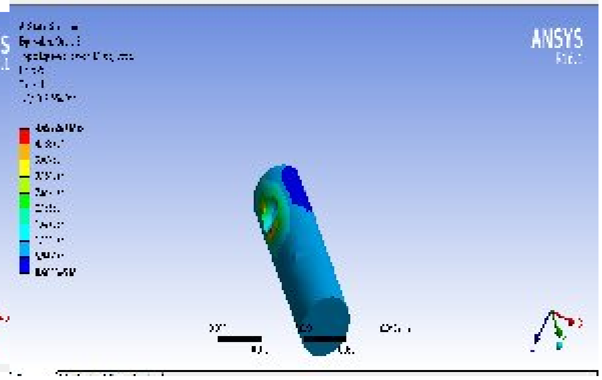
(c)



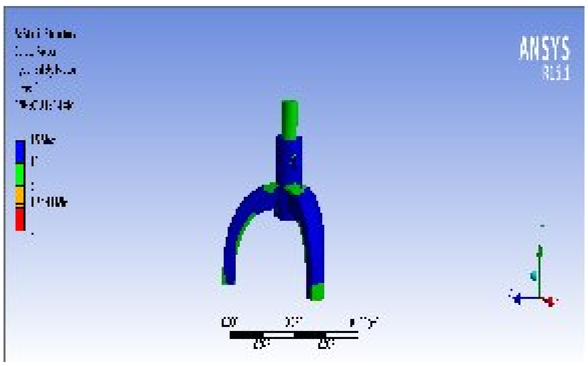
(d)



(e)



(f)

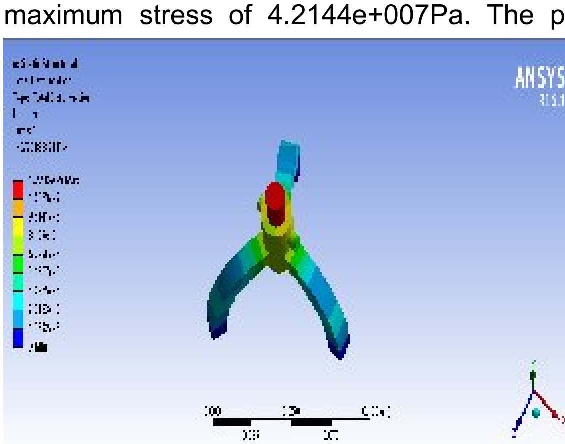


(g)

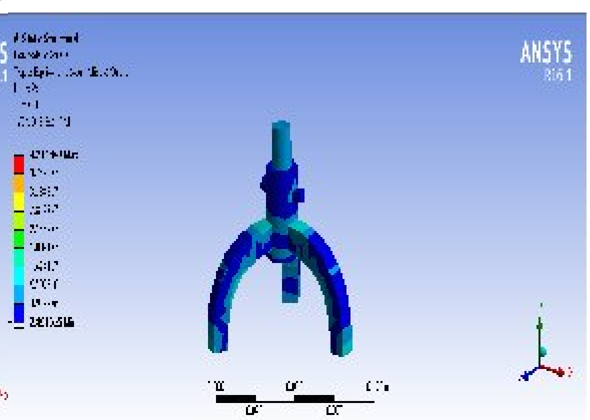
Figure 8:
(a) Base Total deformation
(b) Base Equivalent stress
(c) Pin Total deformation
(d) Pin Equivalent stress
(e) Holder total deformation
(f) Holder equivalent stress
(g) Safety factor

Simulation results for Copper Alloy

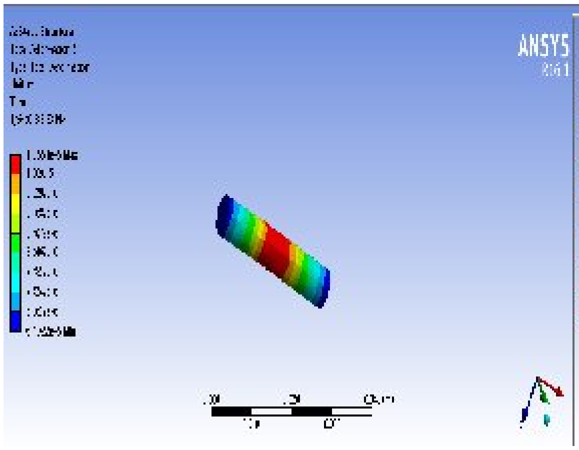
Lastly, Copper Alloy is used as the material. Figure 9a shows the deformation property of the material having 1.2232e-005m as the maximum deformation occurring at the pin tip and base legs respectively. Figure 9b shows the stress distribution in the material with the maximum stress of 4.2144e+007Pa. The pin and the holder were also analysed with the region of maximum and minimum deformation and stress occurring at the centre and tip respectively, as seen in Figure 9c to 9f. Figure 9g shows the safety factor plot, which indicates that the material strength can withstand the maximum applied stress, making it safe to be used in the said application.



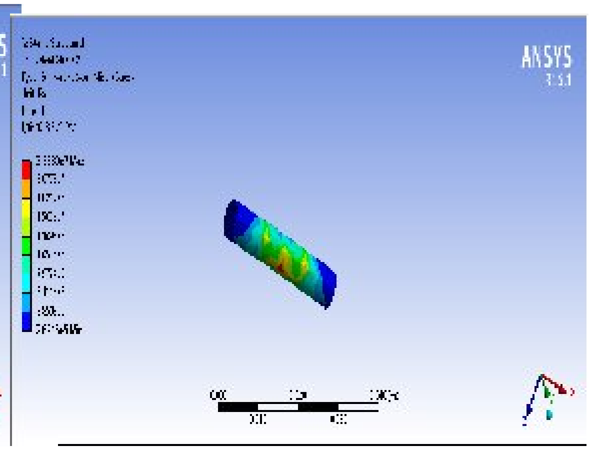
(a)



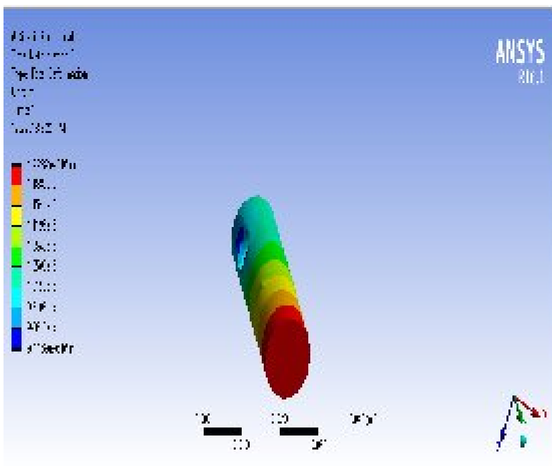
(b)



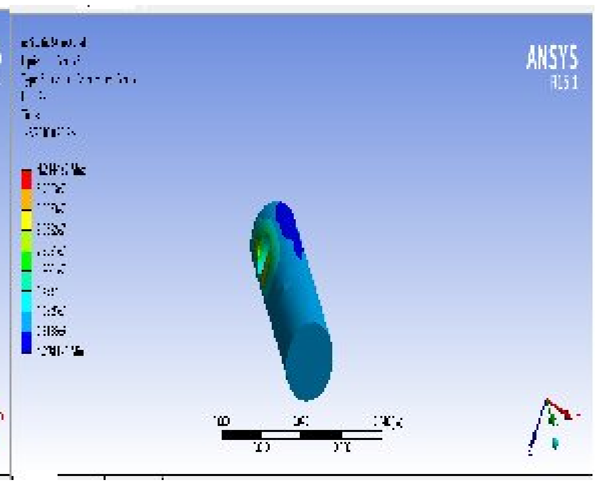
(c)



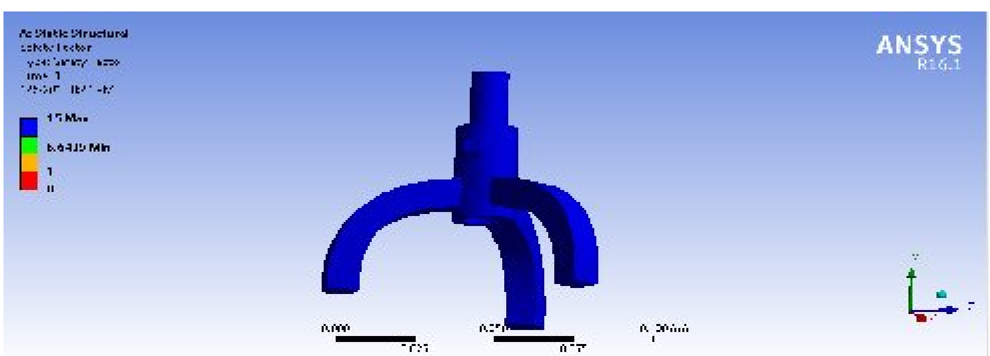
(d)



(e)

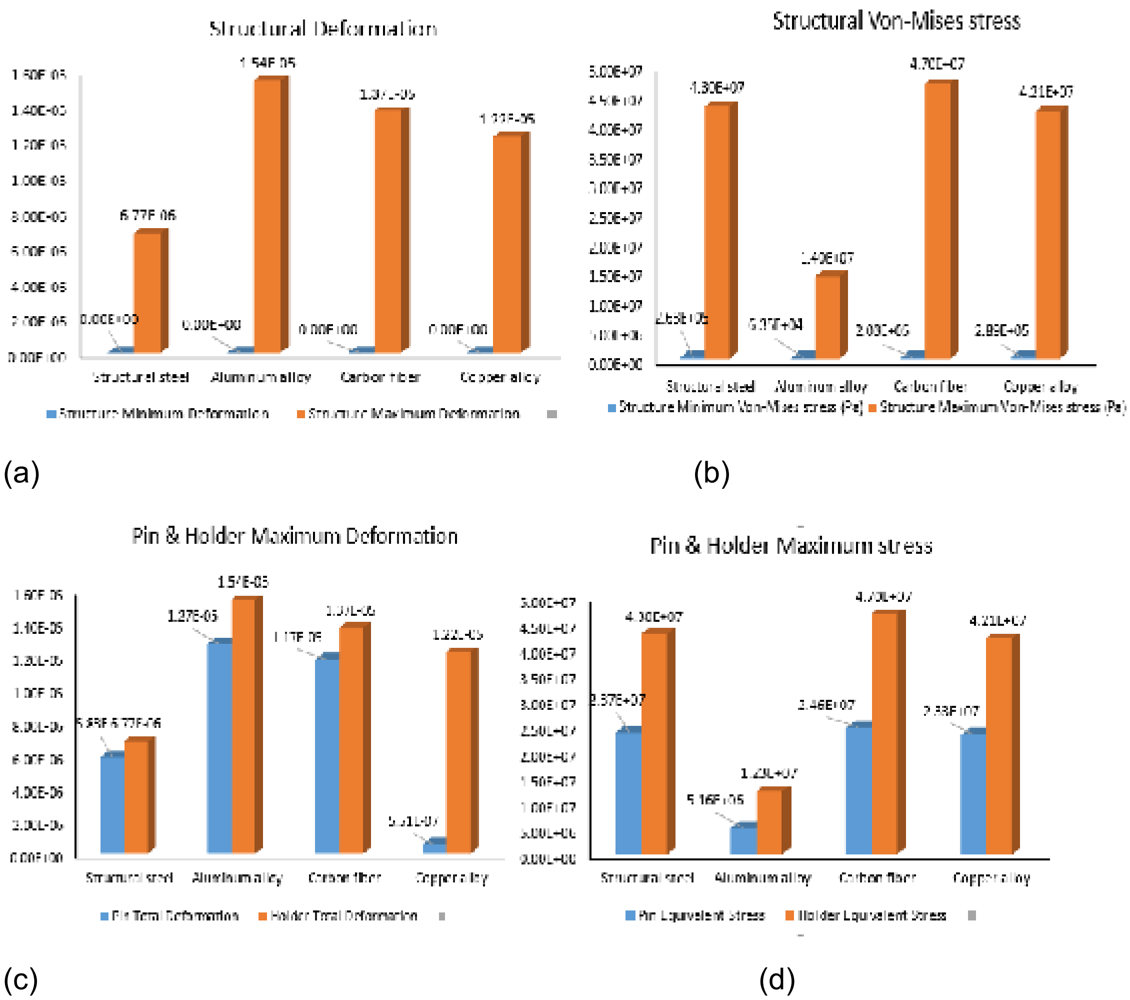


(f)



(g)

Figure 9: (a) Base Total deformation
(b) Base Equivalent stress
(c) Pin Total deformation
(d) Pin Equivalent stress
(e) Holder total deformation
(f) Holder equivalent stress
(g) Safety factor



(a) (b)

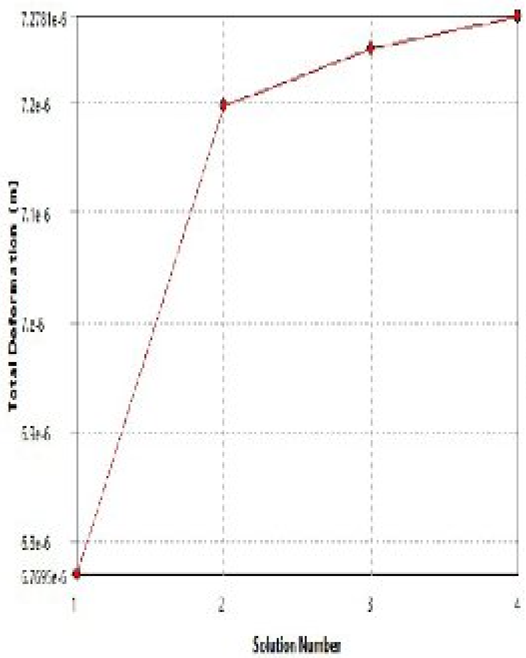
(c) (d)

Figure 10: (a) Base deformation (b) Base equivalent stress

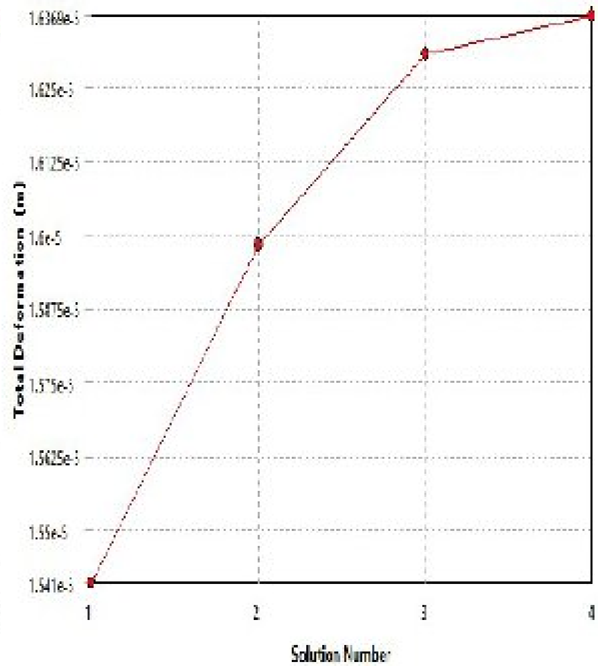
(c) Pin and Holder deformation (d) Pin and holder stress

CONVERGENCE HISTORY

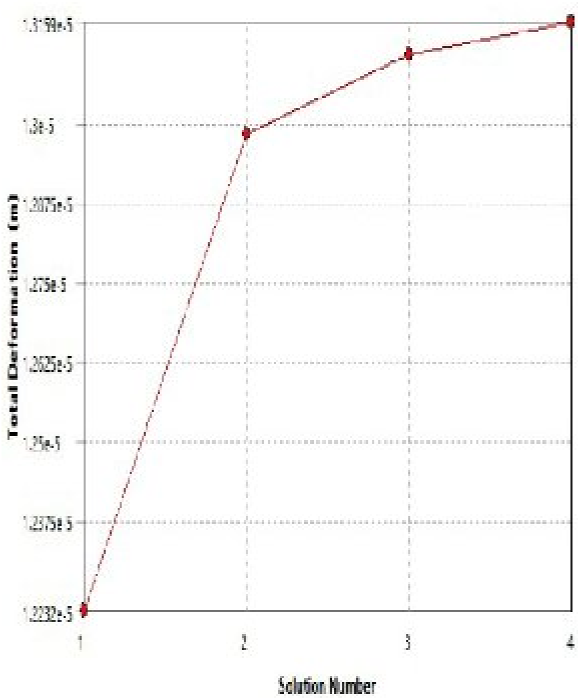
Further repeated refinement of the an FEA structure into much smaller elements results in the convergence of the solution to an exact solution of the mathematical problem. Some of these refinement types formulated in FEA which converges the FEA solution to an exact solution of the mathematical model include: h-refinement (with h referring to the element size), p-refinement (p referring to the polynomial highest order), r-refinement (which rearrange the nodes in the mesh) and hp-refinement (which combines h and p-refinement) (Xiaolin & Yijun, 2014). In this analysis, the convergence criteria are based on the total deformation with 1% tolerance limit. Convergence was obtained for the static analysis solution after a maximum number of 10 refinement loops as shown in figure 11. Maximum total deformations that results at different mesh iteration are displayed and shown in table 2 below.



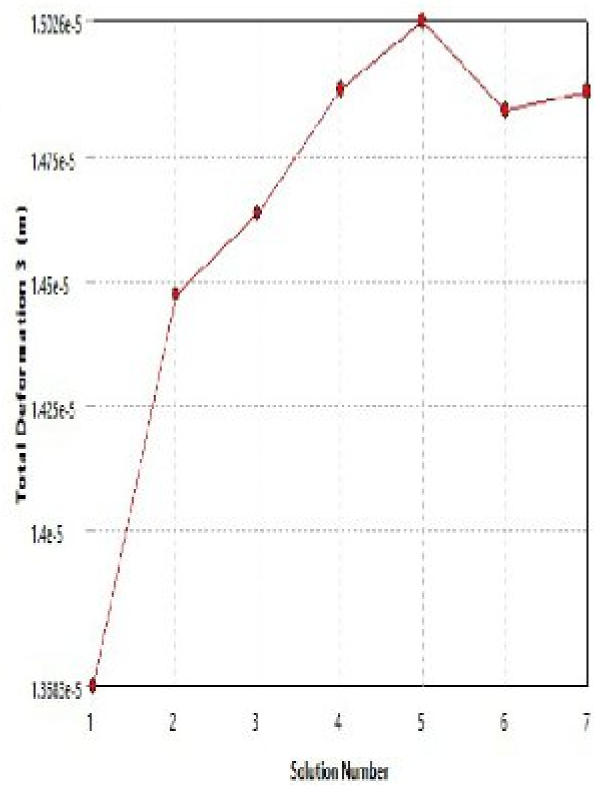
(a)



(b)



(c)



(d)

Figure 11: (a) Structural steel (b) Aluminium(c) Copper(d) Carbon

Table 1: Mesh iteration table

Material type	Solution number	Total Deformation (m)	Change (%)	Nodes	Elements
Structural steel	1	6.77E-06		2273	915
	2	7.20E-06	6.1169	11640	6.64E+03
	3	7.25E-06	0.71455	47255	30350
	4	7.28E-06	0.40977	69672	45754
Aluminium	1	1.54E-05		2273	915
	2	1.60E-05	3.6221	10965	6085
	3	1.63E-05	2.0181	45171	28522
	4	1.64E-05	0.39121	118304	78931
Copper	1	1.22E-05		2273	915
	2	1.30E-05	5.9419	11640	6637
	3	1.31E-05	0.97735	47505	30521
	4	1.32E-05	0.38327	70128	46058
Carbon	1	1.37E-05		2273	915
	2	1.45E-05	5.6218	11405	6498
	3	1.46E-05	1.1259	38424	24535
	4	1.49E-05	1.7006	85533	56310
	5	1.50E-05	0.91879	205681	139809
	6	1.48E-05	-1.2307	249490	170689
	7	1.49E-05	0.26973	346247	238254

CONCLUSIONS

Conclusively, based on the results obtained from the four different cases, it can be seen from Figure 10a that Aluminum alloy has the highest deformation level, followed by Copper alloy, then Carbon fibre and lastly Structural steel. This indicates that Structural steel is having the least deformation when subjected to the same loading condition. Based on the Von-mises stress analysis obtained as seen in 10b, Carbon fibre has the highest value of maximum stress, followed by structural steel, then Copper alloy and Aluminum alloy. While aluminium alloy has the least minimum Von-Mises.The entiremaximum Von-Mises stress in the material structures are less than the material tensile and ultimate shear stress, indicating that the applied stress is less than the material yield point value. The safety factors obtained for all the materials appeared to have enough safety band exceptaluminiumalloys that has a value close to unity.

Lastly, structural steel is the best choice based on the results obtained. It safety factor is far superior to unity in all the model, which indicates that the structure has been over-engineered. The materials are too excessive, and this shows wastage of resources and cost. Therefore, it is recommended that the structure should be optimized.

.

REFERENCE

Kim, B.S., Lee, S.H., Lee, M.G., Ni,J., Song, J.Y., & Lee,C.W.(2007). A comparative study on damage detection in speed-up and coast-down process of grinding spindle-typed rotor-bearing system. *Journal of Materials Processing Technology, 187*(188), 30-36.

Talikoti,B., Kurbet,S.N., Kuppast, V.V, & Arvind, M.(2016). Harmonic analysis of a two cylinder crankshaft using ANSYS.Inventive Computation Technologies. *IEEE* International Conference on Inventive Computation Technologies *(ICICT), 1*(3), 1-6.

Barbero, E.J. (2014). *Finite Element Analysis of Composite Materials Using ANSYS* (Second Edition). London: CRC Press.

Han, X., Zhong, L., & Li, B.. (2002). Application of finite element analysis in structure analysis and computer simulation. *Journalof Chongqıng,4*, 124-126.

Janq, G.H., & Lee,S.H. (2002). *Free vibration analysis of a spinning flexible disk-spindle system sup- ported by ball bearing and flexible shaft using the finite element method and substructure synthesis.* UK: Academic Press.

Qiongying, L.V., & Yushi, M. (2014). Modal analysis of a magnetic climbing wall car frame based on the ANSYS. *IEEE Workshop on Electronics Computer and Applications 1*(1), 938-940.

Wang, G., Zhou, G., & Yang, J. (2012). Modal analysis of high-speed spindle based onANSYS. *IEEE International Conference onComputer Science and Education (ICCSE), 7*, 475-478.

Xiaolin., C., & Yijun, L. (2014). *Finite Element Modeling and Simulation with ANSYS*. Workbench.CRC Press, London NewYork.

Yu, B., & Huang, Z. (2005). Finite Element Modality Analysis of Rotor in Centrifugal Pump. *Journal of Gansu Sciences*. *16*(2), 32-35.