

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Design and Analysis of Knuckle Joint Utilized in Structures by using Polylactic Acid Material with Different Infill Percentage

Siddharaj Barge ¹, Dr. Zheng Li ²

Research Assistant, Department of Mechanical Engineering, University of Bridgeport, Bridgeport, Connecticut, USA¹

Associate Professor, Department of Mechanical Engineering, University of Bridgeport, Bridgeport, Connecticut, USA²

ABSTRACT: This project is based on analyzing the behavior and reaction of the polylactic acid material generally termed as PLA which is mostly used in Fused Deposition Modeling (FDM) type printing method. The analysis is performed by 3D printing this material with different infill percentage property like 15, 20 and 100 percent. The model used for performing the analysis is a knuckle joint assembly whose fork end would be fixed and the other end would have tensile force of 1000 N. The study of this type of project would in future result in different substitutes of material but at same time gives us an idea of the extreme conditions in which the PLA can be used in.

KEYWORDS: Reaction, PLA, Fused deposition Modeling (FDM), infill, knuckle joint, substitute, extreme.

I. INTRODUCTION

There are a few unique techniques for 3D printing, yet the most generally utilized is a procedure known as Fused Deposition Modeling (FDM). FDM printers utilize a thermoplastic fiber, which is warmed to its liquefying point and afterward expelled, layer by layer, to make a three dimensional article^[1]. In order to study this several infill percent and it's capability which can be done by the means of analysis can be done if we are having exact physical properties^[2]. In this project, the 3D printer used to print the model is Makerbot 5th generation. While printing this project the print quality is set to high in order to ensure fine printing and using this setting the model is printed with three different infill percentages which are 10, 15 and 100 percent^{[2][3]}. Further using these three different models they are subjected to FEA analysis in order to evaluate performance of the material under different testing conditions^[4]. At the end of this project study it can be successfully evaluated that which physical characteristics is suitable for the model under different conditions depending upon the application.^[5]

II. RELATED WORK

The steps to be performed in this project study is based on the previous research projects provided in the references where a 35mm fork eye diameter knuckle joint is used for structure purposes are:

1. Based on the previous research, design a CAD model suitable for the application.
2. To print three different models of 10, 20 and 100 percent infill in order to study and determine the behaviour of the material and at the same time evaluate their performances under given conditions.
3. For the analysis this model is imported into ANSYS Workbench and as the model has three different infill so the material properties are different from each other provided in[2].
4. Study the analysis results, compare them and determine their capabilities in order to choose the most appropriate one. Based on that we can determine which infill would be best suited for what conditions.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

III. CALCULATIONS

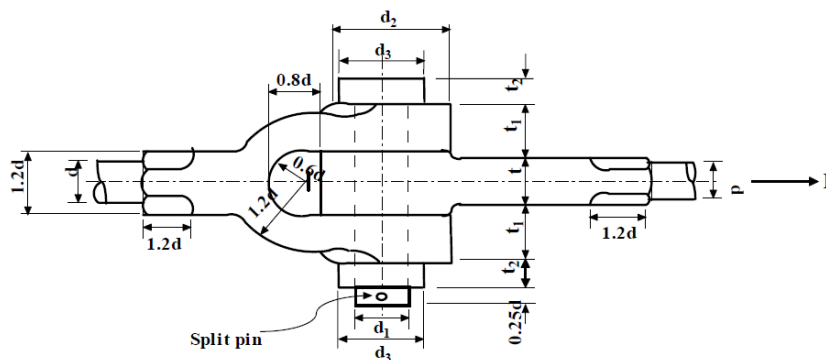


Fig.1 Schematic diagram of knuckle joint assembly showing dimension calculations^[2].

where, d = diameter of rod

$$\begin{aligned} d_1 &= d & t &= 1.25d \\ d_2 &= 2d & t_1 &= 0.75d \\ d_3 &= 1.5d & t_2 &= 0.5d \\ a &= 0.75d & b &= 1.25d \end{aligned}$$

The dimensions for the model used in this project is as follows :

$$\begin{aligned} d = d_1 &= 35\text{mm} & t &= 43.75\text{mm} \\ d_2 &= 70\text{mm} & t_1 &= 26.25\text{mm} \\ d_3 &= 52.5\text{mm} & t_2 &= 17.5\text{mm} \\ a &= 26.25\text{mm} & b &= 43.75\text{mm} \end{aligned}$$

where, a = thickness of fork b = thickness of eye

Mean diameter of split pin = 0.25d

Equation for strength ratio^[6] :

$$\text{Strength Ratio (SR)} \stackrel{\text{def}}{=} \frac{F_{max}}{mass} / \frac{F_{max, solid}}{mass_{solid}}$$

Equation for increase in strength ratio :

$$\text{Strength Ratio (SR)} \stackrel{\text{def}}{=} \frac{F_{max}}{mass} / \frac{F_{max, solid}}{mass_{solid}}$$

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Sr.No	Infill geometry	Percent infill	Max force (N)	Strength ratio	Increase in strength ratio
1.	Rectilinear	15	1026.764	0.640	0.265
2.	Rectilinear	20	1574.099	0.926	1.065
3.	Solid	100	1114.584	0.863	N/A

Table 1. Strength increase caused due to infill changes^[4]

IV. EXPERIMENTAL RESULTS

Design of Knuckle joint assembly:

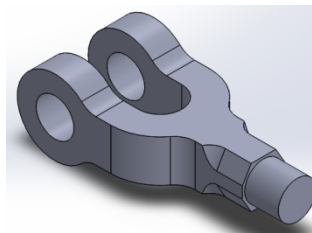


Fig.2 CAD model for fork end

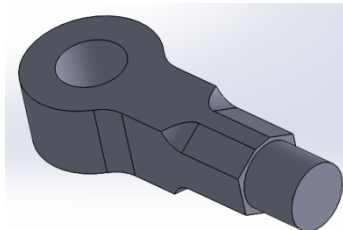


Fig.3 CAD model for eye end

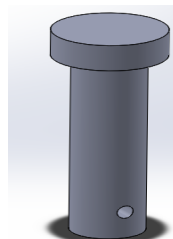


Fig.4 Eye pin

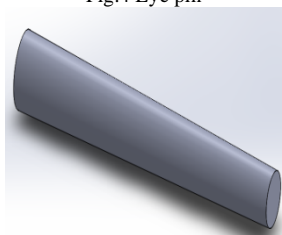


Fig.5 Taper pin

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

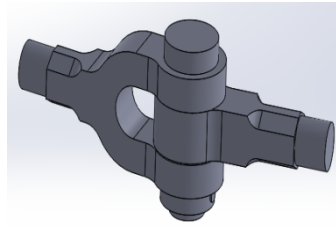


Fig.6 Assembly of Knuckle joint

FEA Analysis of Knuckle joint :

FEA analysis for this project is done using ANSYS Workbench. The analysis is carried out on three different conditions of the solid model, which are 10, 15 and 100 percent infill. FEA analysis is carried out in order to evaluate how PLA reacts under different inner structural conditions^[6].

The model used to study in this project is a knuckle joint assembly which is printed using a Makerbot Replicator 5th gen and the filament used is having 1.75mm diameter. The boundary conditions applied during three different conditions is same where, one end of the assembly which is fork end is fixed while the free end which is the eye end is under force. Considering this type of model being used for structural purposes it is been subjected to a load of 1000N^{[3][7]}.

Material properties for 10 percent infill:

- Modulus of Elasticity - 615 MPa
- Poison's ratio - 0.35
- Ultimate tensile strength - 14.45

Mesh settings:

- Relevance center - Fine
- Smoothing - High
- Transition - Slow
- Minimum edge length - 4.09 mm

Boundary conditions:

Force applied : 1000 N ^[2]

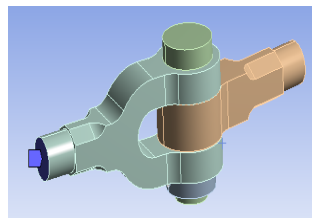


Fig.7 Fixed fork end

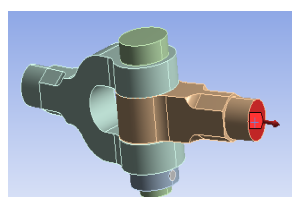


Fig.8 Force applied eye end

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

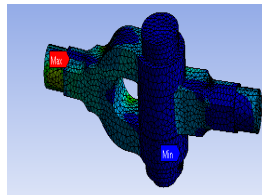


Fig.9 Equivalent Von Mises stress

As it can be evaluated from the above analysis result the stress is maximum on the left side i.e. the fixed end of the model. This can be considered as a safe model as the most critical section of the geometry which is an intersection and the entire assembly is connected through that point is undergoing minimum stress for the maximum part and though the stress varies along the length of the assembly it is on the lower side of stress scale. The minimum stress induced is **0.018 MPa** and the maximum is **72.86 MPa**.

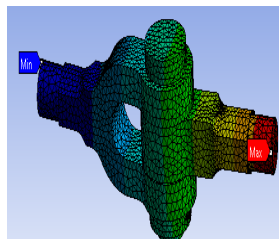


Fig.10 Total deformation

From the analysis above it can be evaluated that the deformation along the assembly is changing continuously but not rapidly or invariably. The deformation is minimum which is **0 mm** at the fixed end which is understandable as that section has minimum disturbance due to force while the deformation is maximum at the end where force is applied. The maximum deformation occurred is **129.72 mm**.

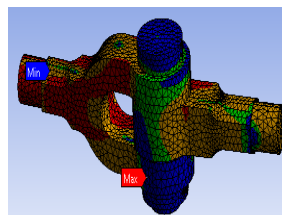


Fig.11 Safety factor for 10% infill

After analyzing the results from the safety factor it can be said that using this material with the given specifications can be considered safe design as well as safe infill percentage. As the safety factor is higher where the stress concentration would be maximum. The minimum safety factor here is **0.198** while the max is **15**.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Material properties for 20 percent infill:

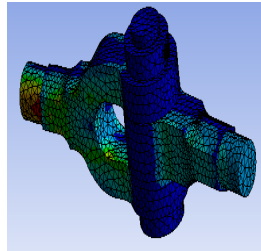


Fig.12 Equivalent von mises stress

From the above analysis it can be evaluated that the design and the material properties used is safe. It can also be stated that, stress is maximum on the fixed side of fork end and further along the length of fork reduces by small value. The most important thing to be noted in this analysis is the stress value is almost around minimum at the joint of the assembly, which means the most vulnerable part that is most likely to fail is well under the limit. The minimum stress generated here is **0.023 Mpa** and maximum is **73.66 MPa**.

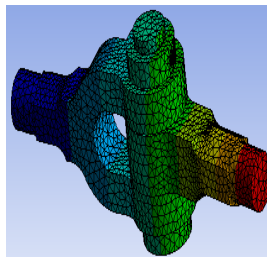


Fig.13 Deformation

Studying from the above analysis it can be said that deformation is minimum at the fixed end while it goes increasing along the length of the assembly. The deformation at the joint of the assembly is at the mid range of the scale which is still considered to be safe. The minimum deformation is **0 mm** and maximum is **129.6 mm**.

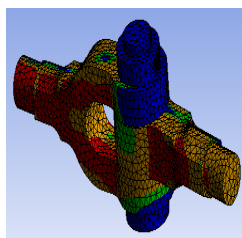


Fig.14 Safety factor

Safety factor analysis is the deciding analysis whether the component is safe or not. From the above analysis it can be stated that the component is safe under given conditions. Where the minimum safety factor is **0.272** and maximum is **15**.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

Material properties for 100 percent infill:

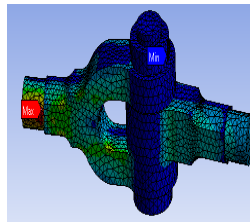


Fig.15 Equivalent Von Mises stress

The above analysis is carried out on same model but with infill percent to be 100. So this component can be considered as completely solid and more heavier than previous two conditions. As it can be seen the difference between stress generated in this case and previous case is minor. The minimum stress induced here is **0.023 MPa** and maximum is **73.65 MPa**.

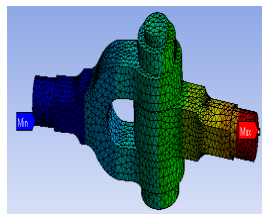


Fig.16 Deformation

Though there is minute difference between stress analysis of this and previous cases but after studying the analysis in this case there is major difference between the deformation in this case and previous cases. In this case the minimum deformation is **0 mm** and maximum is **81.93 mm**. So it can be said that, this type of structure should be used where stability is the most vital factor.

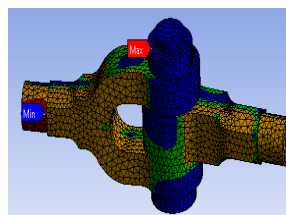


Fig.17 Safety factor

It can be evaluated from the above analysis that there is minor increase in the safety factor compared to other cases. The safety factor is minimum near to the fixed end while it is maximum almost throughout the joint of the assembly but is maximum near the neck of the knuckle pin. The minimum value is **0.63** and maximum is **15**.

V. CONCLUSION

Referring to the above study and the analysis various points can be stated that are, if compared with the stress analysis the difference between the values of 10, 20, 100 percent infill is almost negligible. This can be due to the fact that the design of the geometry remains the same for all the cases the characteristic that changes is inner structure. On the other hand if the deformation results are compared there is major difference between 10, 20 and 100 percent infill. The analysis proves that the deformation caused in 100 percent infill is very less compared to previous two cases. The reason for this is as having 100 percent infill the model indirectly can be considered as solid while the other two cases

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 3, March 2019

are 10 and 20 percent in terms of solid material thus can be deflected easily and also due to its hollowness it can get easily disturbed during operation^[7]. Finally we can conclude that if knuckle joint has to be used using polylactic acid material that would be a fine option but depending upon the application we have to decide what exact infill do we need^[8]. If we need such model for light and stable operation then choosing less infill would be best option thereby saving weight, cost, time in terms of model, material and printing but if we need such model for heavy and varying application then 100 percent infill is the best option but that would be leading to higher cost.

REFERENCES

- [1]. Shaik John Bhasha, Hari Sankar Vanka, "Modelling and analysis of knuckle joint", International Journal & Magazine of Engineering, Technology, Management and Research, pp. 292-298, November 2015.
- [2]. Chris D. McCoy, "Materials testing of 3D printed ABS and PLA samples to guide mechanical design", Conference paper Research gate, 10.1115/MSEC2016-8668, pp. 1-12, June 2016.
- [3]. Abdul Farhan, Kaustubh Khoptikar, Dhananjay R Dolas, " Design and optimization of knuckle joint using trusses", iosr journal of mechanical and civil engineering, May 2016.
- [4]. Sajid Khan, Nausad Khan, Abdul Quadir, "Design and Analysis of Knuckle joint of different material using the CAE tools", International Journal of Engineering Technology Science and Research, Vol.5, Issue 5, pp. 278-285, May 2018.
- [5]. Suraj Yadav et.al, "Design and analysis of knuckle joint using FEA", Technical Research Organization India, Vol.4 Issue 6, pp. 70-77, 2017.
- [6]. Gabriel A. Johnson, Jesse J. French, "Evaluation of Infill Effect on Mechanical Properties of Consumer 3D Printing Materials", Advances in technology innovation, Vol.3 No.4, pp 179-184, December 2017.
- [7]. A. Lanzotti, M. Grasso, G. Staiano, and M. Martorelli, "The impact of process parameters on mechanical properties of parts fabricated in PLA with an open-source 3-D printer." Rapid Prototyping Journal, vol. 21, no. 5, pp. 604-617, 2015.
- [8]. Jonathan torres et.al. "Mechanical property optimization of FDM PLA in shear with multiple objectives", JOM, Vol.67, No.5, pp 1183-1193, 2015.