Comparative Deflection Analysis of a Cantilever Beam Fixed at One End by Using Finite Element and Analytical Method

Capstone Project – II Report submitted in partial fulfillment for the award of the degree of

Degree of B. Tech (Mechanical Engineering)

Submitted by

ADITYA PRATAP SINGH

(18021011960)

AMRESH KUMAR

(19021012110)

IN

MECHANICAL ENGINEERING

Under the Supervision of

PROF. P. SURESH

PROF. SHRIKANT VIDYA

SESSION – JUNE - 2022

DEPARTMENT OF MECHANICAL ENGINEERING

GALGOTIAS UNIVERSITY

(Established under Galgotias University Uttar Pradesh Act No. 14 of 2011)

GALGOTIAS
UNIVERSITY

DEPARTMENT OF MECHANICAL ENGINEERING

BONAFIDE CERTIFICATE

Certified that this project report "**Comparative Deflection Analysis of a Cantilever Beam Fixed at One End by Using Finite Element and Analytical Method"** is the Bonafide work of "**ADITYA PRATAP SINGH (18021011960), AMRESH KUMAR (19021012110)"** who carried out the project work under my supervision.

SIGNATURE OF SUPERVISOR

APPROVAL SHEET

This thesis/dissertation/project report entitled titled **Comparative Deflection Analysis of a Cantilever Beam Fixed at One End by Using Finite Element and Analytical Method** is approved for the degree of Bachelor of Technology in Mechanical Engineering.

Examiners

____

____

____

Supervisor

____

Dean

____

Date: _____________

Place: ____________

Statement of Project Report Preparation

1. Project report titled: **"Comparative Deflection Analysis of a Cantilever Beam Fixed at One End by Using Finite Element and Analytical Method".**
2. Degree for which the report is submitted**: BACHELOR DEGREE OF TECHNOLOGY.**
3. Project Supervisor was referred to for preparing the report.
4. Specifications regarding thesis format have been closely followed.
5. The contents of the thesis have been organized based on the guidelines.
6. The report has been prepared without resorting to plagiarism.
7. All sources used have been cited appropriately.
8. The report has not been submitted elsewhere for a degree.

(Signature of the student)

Name: (ADITYA PRATAP SINGH) **Roll No: (18021011960)**

Statement of Project Report Preparation

1. Project report titled: **"Comparative Deflection Analysis of a Cantilever Beam Fixed at One End by Using Finite Element and Analytical Method".**
2. Degree for which the report is submitted: **BACHELOR DEGREE OF TECHNOLOGY.**
3. Project Supervisor was referred to for preparing the report.
4. Specifications regarding thesis format have been closely followed.
5. The contents of the thesis have been organized based on the guidelines.
6. The report has been prepared without resorting to plagiarism.
7. All sources used have been cited appropriately.
8. The report has not been submitted elsewhere for a degree.

(Signature of the student)

Name: (AMRESH KUMAR) **Roll No: (19021012110)**

ACKNOWLEDGEMENT

I have taken efforts in this project. However, it would not have been possible without the kind support and help of many individuals and organizations. I would like to extend my sincere thanks to all of them.

I am highly indebted to **Dr. P.K.S. NAIN (DEAN S.O.M.E., G.U.), PROF. SHRIKANT VIDYA (S.O.M.E., G.U.)** & **PROF. P. SURESH (S.O.M.E., G.U.)** for their guidance and constant supervision as well as for providing necessary information regarding the project & also for their support in completing the project.

I would like to express my gratitude towards member of **GALGOTIAS UNIVERSITY** for their kind co-operation and encouragement which help me in completion of this project.

My thanks and appreciations also go to my colleague in developing the project and people who have willingly helped me out with their abilities.

Aditya Pratap Singh (18021011960)

Amresh Kumar (19021012110)

(of Mechanical engineering)

ABSTRACT

In mechanical, and civil engineering, beams are considered as a common feature of many buildings, structures buildings, and the bending beams studies are also a significant part of a comprehensive field of structural mechanics and mechanics of materials. Under the activity of a uniformly dispersed load along its own weight and an outside vertical accumulated load at the free end, the old style issue of deflection of linear elastic material of a cantilever beam, is being analytically and numerically analysed. Material is being presumed to be isotropic with the material AISI1020 Stainless Steel is being taken for the study. For the analytical evaluation of the system and for calculating beam material deflection the SOLIDWORKS program is being used. Finally, finite elements analysis is used to compare the numerical results with the analytical ones.

\

TABLE OF CONTENT

List of abbreviations

1. FEA --- Finite Element Analysis
2. AISI --- American Iron and Steel Institute.
3. URES --- Resultant Displacement
4. CAD --- Computer- Aided Design

List of figures

List of tables

CHAPTER 1

INTRODUCTION

Civil engineers and industrialists are responsible for the safety of buildings. Anticipating and measuring the deflection of the support beams is one of their most important safety functions. Learning the different types of deviations, and how to calculate them can help you decide if this is a good job. In this report, we discuss the definition of deflection in engineering, the different types of deflections, the main causes, and how to calculate the deflection rate for different geometries.

Deflection is the movement of the beam from its original position. Some refer to engineering deflection as migration. This refers to the movement from an engineering force, from an object itself or an external source, such as the weight of a wall or roof. Many buildings are at risk of collapse, including beams and frames. Deflection is average length. Calculating beam deflection gives an angle or distance, which refers to the range of motion of the beam.

In terms of structural engineering, beams are considered to be a component made of a number of materials (including steel, wood aluminum) to withstand loads - commonly used in each metal case. Beams can also be referred to as members, elements, planks, shafts, or purlins. To carry a direct load, shear load, and sometimes horizontal beam load as a horizontal structure is used. It is a very important part of the architecture. It is often used in the development of extensions, brackets, and various designs that convey a specific load. Compared to the height of any part of a building where the cross-section is very small, it carries a side load known as a beam. A beam horizontal bar that carries a rear load or a couple that tends to bend or a horizontal bar facing a curved pressure is known as a beam. In building a building and failing to use deflection is an important consideration that can be catastrophic. Beam deviations can be calculated in the structure in a variety of ways: mathematical methods and finite object analysis etc. [1]

According to engineering terms, the shape of the building changes at the rate of the deflection when a load is applied. The Changes that occurs includes the angle or distance which can be invisible or visible, that depends on following factors: severity of load, part shape, also included the material from which they are obtained. The deflection mainly caused by different types of load, in which uniform distributed loads, shear loads, point loads, wind loads, and ground

pressures along with earthquakes, are included. The part may fail when too much deviation is produced by load. Beams, floors, columns, bridge floors, walls, dams, tunnel walls and more are included in detachable sections. San Francisco's Golden Gate Bridge can move about 15 feet [4 m] alongside strong winds.[2]
Deviation is observed in the non-built parts, for example, building lock panels may deviate inside if they are under high air load. In order to ensure the building users safety and the integrity of the structure as a whole while considering the potential for failure related to structures, construction code generally decide maximum deviation to be allowed. In the beam, this is often expressed as part of a span, e.g. beam deviation should not be greater than 1/360 span; therefore, if span is 5m, then deviation should not exceed 13.9 mm. In the center of the beam it will typically be measured. [3]

It is important to know the different types of deviations, as this can affect the direction and movement of the beams. There are two main types of deflections, including:

1.1 Angular Deflection

The angular deflection, which we can also refer to as the rotational deflection, measures the rotation of the structure from its original position. Heavy loads can cause buildings to turn from their original position. Calculation of angular deflection measures the deflection of the angular movement from the starting point to the point after rotation.

1.2 Linear Deflection

A line deflection, which you can also refer to as a translation deflection, measures the movement of a particular point of a building from its actual position. Linear movement can be a few millimeters away from where you start. A line deviation can also be a straight or horizontal deviation, which refers to the direction of your movement. The deviation of the line is usually the limit of the distance between the curve and its tessellation, which is the idea of repeating the same shape over and over again.

1.3 Causes of Deflection

There are four primary causes of deflection, which include:

- The weight of the load that sits on the structure affects how much it moves or bends. A heavier load on top of a beam may cause it to deflect more.
- The moment of Inertia refers to the size of the opposite phase. It is a standard that engineers use primarily to measure deviation.
- The size of the structure without support also contributes to the deviation. This refers to the extent to which the structure lacks support.
- What materials are made of it also affects its deviation. Some materials are stronger than others, and aluminum deviates more than steel.

A lot of research had been published related to the analysis of deflection on different types of engineering structures.

1.4 Project Background

In this project, there is a comparative deflection analysis of a cantilever beam fixed at one end by using finite elements and mathematical methods. The point load of 4000 N supported by beam at one of the end of the cantilever beam that is manufactured of stainless steel AISI1020. And the results are compared with the different types of load that are acting on the fixed end of the cantilever beam. The comparison has been developed by the help of finite element analysis and mathematical formula. SolidWorks is one of the advanced and used software for the geometric analysis. Finite Element Analysis (FEA) is a numerical method for solving engineering and mathematical physics problems. It is useful for complex problems geometry, loading, and visual structures where analytical solutions may not be widely accepted. There is a different load (4000N, 5000N, 6000N) acted on the structure and has been compared for the more accurate result.

CHAPTER 2

LITERATURE REVIEW

Deviation and the distribution of stress on the long, thin cantilever of a rectangular cross-shaped cross-sectional and isotropic was being examined by Ashis Kumar Samal. Deviation of the cantilever beam in actual is a 3-D problem. The stress in perpendicular direction accompanies the stretching on one side. Under the action of three different loading conditions the beam is being modelled: vertical focused. He concluded by using Ansys the deviation is very accurate when using the 10 node Tetrahedral feature but due to the stress, the 8node brick feature gives better results. Therefore, the 10-node Tetrahedral element being used, to detect deviations, while the stresses 8-node element of the brick is best and most suitable [4].

GT Beam is a graphical analysis system that is being developed by Georgia Tech which is used in undergraduate studies. In order to, allow students to do 'what if ’design conditions in deciding shear/minute and complex deviation beam diagrams, program was being designed that reduces the instructor time which he spent for showing the analysis of beam. This kind of program allows the reader to visualize the impact of the moving foundations or loads in the real-time that has its own design next to the Macintosh graphical interface [5].

Kyungwoo Lee has presented that a large deviation of the cantilever beams made of Ludwick-type material under the combined load that includes load which is distributed uniformly and a single direct load at free end was being examined. Dominant figures were obtained by using the shape of the shear instead of the flexible temporal structure because, in the case of a large deformed limb, the shear strength formation has certain calculation advantages over the temporal curvature. Since the problem involves both non-linear geometry and equipment, the rule number is the complexity of dividing a random number, for determining the maximum derivation of the given load numerical solution is required. The fifth Butcher method Runge – Kutta is used to obtain numerical solution that is presented in a table format [6].

Saeed Moaveni's research has been dedicated to the analysis of problems, stress, and conversion of other members of the structure by such examination. The numerical tests reported here will introduce students to the importance of boundary conditions, power equilibrium conditions, stress/weight relationship of a given object, axial loading, pressure focus, pure bending, opposite loading, and integrated loading. ANSYS and parametric programs are used to set these tests [7].

S. S. Oueini and A. H. Nayfeh verified the result from theoretical analysis by testing. The cantilever steel beam is fitted with piezoceramic actuators and subject to parametric excitation has a frequency equal to twice the natural frequency of its original mode. A computer and a series of analog filters are used to generate a response signal of cubic velocity. Frequency test and power turning curves are in excellent agreement with quality and theoretical results.[8]

Kishan H. Joshi and Chetankumar M. Patel has presented that Euler beam method is used successfully to determine tapered deviation cantilever beam, and verified using FEA method and CREO SIMULATION way. An analysis method designed for tapered cantilever beam by Euler. The beam method can be used to determine the deviation and stiffness of the tapper a cantilever beam with a tendency to load. This theory is simple, and can be and can be used to detect tapered beam deviations with loaded loading non-rectangular section too.[9]

Al-Gahtani and Khan used Euler Bernoulli's standard equation to determine the non-prismatic beam deviation of the flexible parabolic beam and the opposite loading position, and this method was used to determine the deviation of the bridge with the opposite parabolic phase. Direct analysis of non-prismatic planks with standard boundary conditions is presented. The analysis is based on a borderline approach. Basic solutions for non-prismatic beams of direct and parabolic profiles are found.[10]

Bhavikatti [11], and Daryl [12] have described the finite element method to determine the deflection of any mechanical structure or element.

CHAPTER 3

METHODOLOGY AND FORMULATION

3.1 ANALYTICAL METHOD

There are several ways to determine the deviation of a beam or frame. The choice of a particular method depends on the loading condition and the type of problem being solved. Some of the methods used in this chapter include the dual integration method, the unity method, the temporary location method, the unit load method, the optical operating method, and the power methods. Cantilever beams are special types of beams that are constrained by only one support, as shown in the below figure. These members would naturally deflect more as they are only supported at one end.

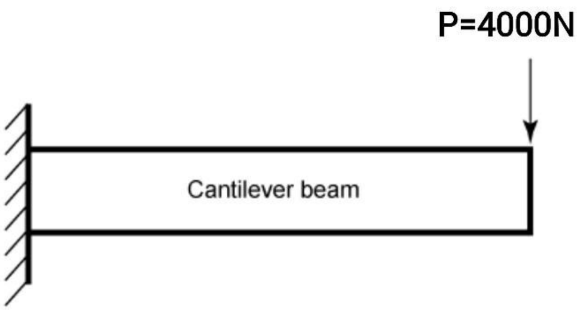


Fig.1. Cantilever beam diagram loaded at the free end

In a cantilever beam, the maximum deflection is experienced only in the free end and is calculated using the below formula.

$$\delta_{max} = \frac{PL^3}{3EI}$$

Where

P is a point load applied (in N)

L represents the beam length (in mm)

E represents the material Young's Modulus (MPa or N/mm^2)

I represents the moment of inertia of the cross-section (in mm^4)

δ_{max} is maximum deflection at free end (in mm)

3.2 FINITE ELEMENT ANALYSIS

FEA is used by engineers to help simulate real-world scenarios and thus reduce the need for portable prototypes while allowing component development as part of the project design process. In complex geometry, FEA is being most commonly used in which conventional analysis methods can solve issues as FEA partitions complex calculations into more modest parts (components). Examination of the limited feature of the cantilever beam under the free endpoint is done using SOLIDWORKS 19.0. Imitation can provide accurate, reliable results for a wide range of types of research from basic vertical line analysis to complex and flexible indirect analysis. Speed up the repetition and prototyping phase of your Imitation design process. The model is built on Solid Tasks and there is a point load operating at the beam of 4000 N. Accurate results in the FEA model is being obtained from Meshing. The information related to Meshing is shown in Figure 3. The best meshing is done in the fillet area.

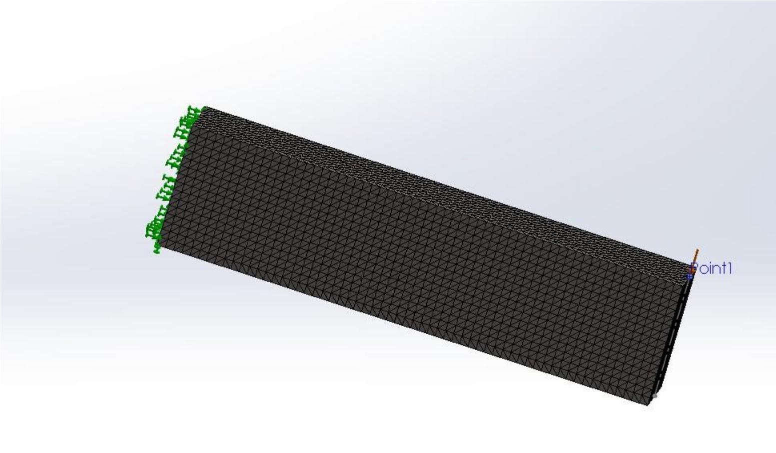


Fig.2. Meshed model of cantilever beam

The analytical and finite element methods are being used to calculate the maximum deflection value. The material is presumed as being isotropic that has been taken for the review is AISI1020 Stainless Steel. The low carbon steel is AISI 1020 carbon steel that has as a minimum 0.17%C and 0.3%Mn. It’s strength and ductility combination is good which can be carburized. It is difficult to install a solid or fire-resistant due to its low carbon content and also it is not suitable for nitriding because of the absence of alloying elements. With the help of traditional methods the steel can easily be machined and welded.

The results of FEA are found and compared for cantilever beam. Overall three different point loads are being prepared, analysed and simulated. Figure 4 to 6 shows FEA results of a cantilever beam with free end point loading for the different point loading. Here, the applied load is considered for examination purpose, the point load as 4000 N, 5000N and 6000N for all iterations with different point loads, and all the variables are same for all iterations except loading. The result is being calculated as the change in maximum deflection from all 3 various point loading. The resultant deflection denoted as URES (mm). There is a minimum value and maximum value on the side of colour scale and is labelled URES (mm).

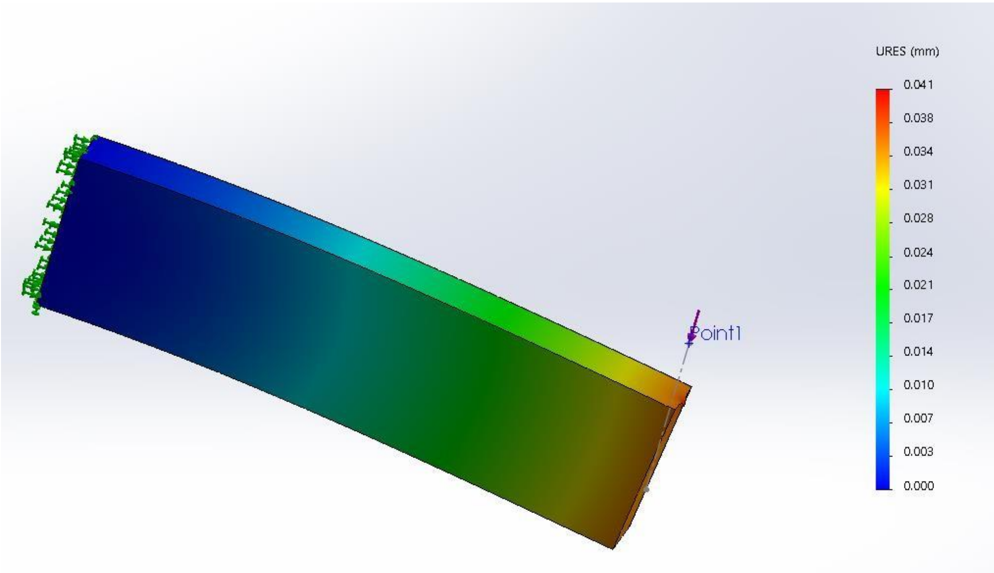
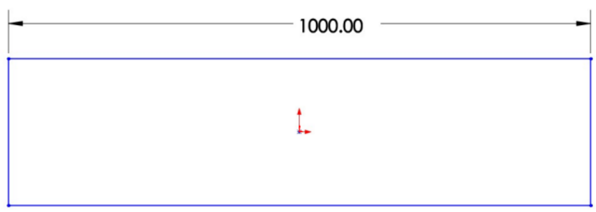
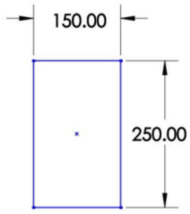


Fig.3. Graphical plot of the numerical simulation by FEA, corresponding to displacements on the beam

3.3 CALCULATION

First, let's define the problem to be solved. This consists on a single cantilever beam 1000 mm length (L) with a rectangular section (a = 150 mm, b = 250 mm). This beam will support a point load of 4000N at one of the end of cantilever beam which is manufactured of stainless steel AISI1020. In these conditions, the maximum deflection (δ_{max}) must be calculated. A scheme of the problem is as follows:





For this particular problem, P = 4000 N, L = 1000 mm, For the value of E, this value can be obtained by searching the technical datasheet (TDS) of a stainless steel (E = 200 GPa). For the calculation of Moment of Inertia, First we have to calculate the centroid coordinates of this structure.

Segment	**Area A (mm^2)**	$\underline{x}$ **(mm)**	$\underline{y}$ **(mm)**	$\underline{x}$**A (mm^3)**	$\underline{y}$**A (mm^3)**
1	$150 \times 250 = 37500$	75	125	2812550	4687500
Total	37500			281250	4687500

Centroid Calculation

$$\underline{x} = \frac{\sum xA}{\sum A} = \frac{2812500}{37500} = 75\ mm$$

$$\underline{y} = \frac{\sum yA}{\sum A} = \frac{4687500}{37500} = 125mm$$

The moment of inertia of a rectangle with respect to an axis passing through its centroid, is given by the following expression:

$$I_x = \frac{1}{12} bh^3 = \frac{1}{12} \times 0.15 \times (0.25^3) = 1.953 \times 10^{-4}\ m^4$$

The following step is to substitute all the above-mentioned numerical values into the general expression(equation1) to calculate the maximum vertical deflection.

$$\delta_{max} = \frac{PL^3}{3EI} = \frac{4000 \times 1^3}{3 \times 2 \times 10^{11} \times 1.953 \times 10^{-4}} = 3.41 \times 10^{-5} \text{ m} = 0.0341 \text{ mm}$$

CHAPTER 4

PROBLEM DESCRIPTION

While starting with the project we came across many problems related to software, bolt specification some of them are listed below:

- The dimension shape and size of the design should be accurate for better simulation results.
- One should have detailed knowledge of engineering software for proper simulation of Cantilever Beam as analyses on SolidWorks is difficult.
- While calculating the theoretical value for the cantilever beam, there should be proper calculation and analysis of the geometry.
- The latest version of SolidWorks is required.

CHAPTER 5

RESULT AND DISCUSSION

In this work, a point load has been developed at free end of a cantilever beam. The maximum deflection's value being calculated with the help of finite element analysis. The chart has been shown below between the parametric distance and deflection. The red curve represents the maximum deflection at the end and it is represented by the horizontal line.

Table 2 : Comparison of maximum deflection acquired from FEA, and theoretical maximum deflection acquired from the analytical method under point loading at free end.

Load P (N)	Theoretical maximum deflection (mm)	Equivalent (ures) deflection (mm)	Percentage deviation (%)
4000	0.0341	0.0358	4.98
5000	0.0426	0.0448	5.16
6000	0.0512	0.0539	5.27

From the table 2, the both result of the maximum deflection from the FEM and theoretical method have approached toward same as different loading.

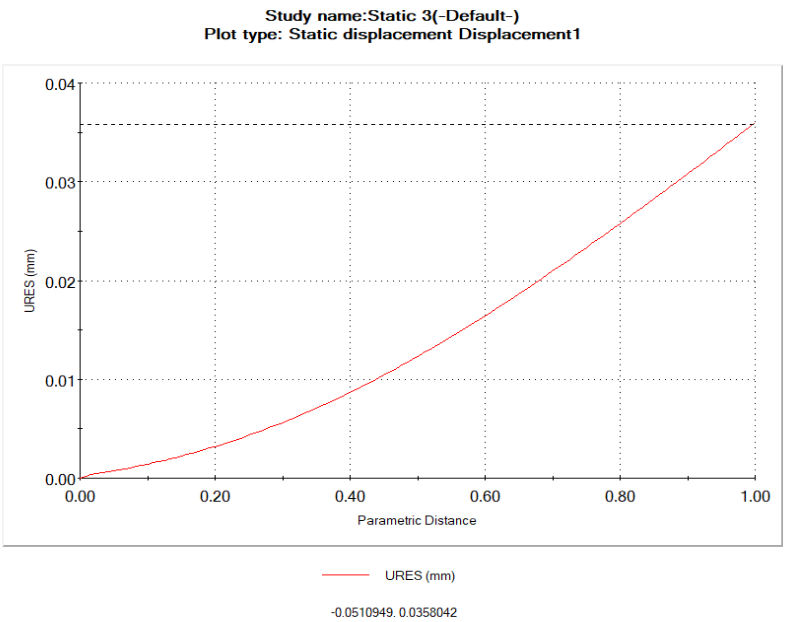


Fig.4. Maximum deflection graph at the free end for P = 4000N

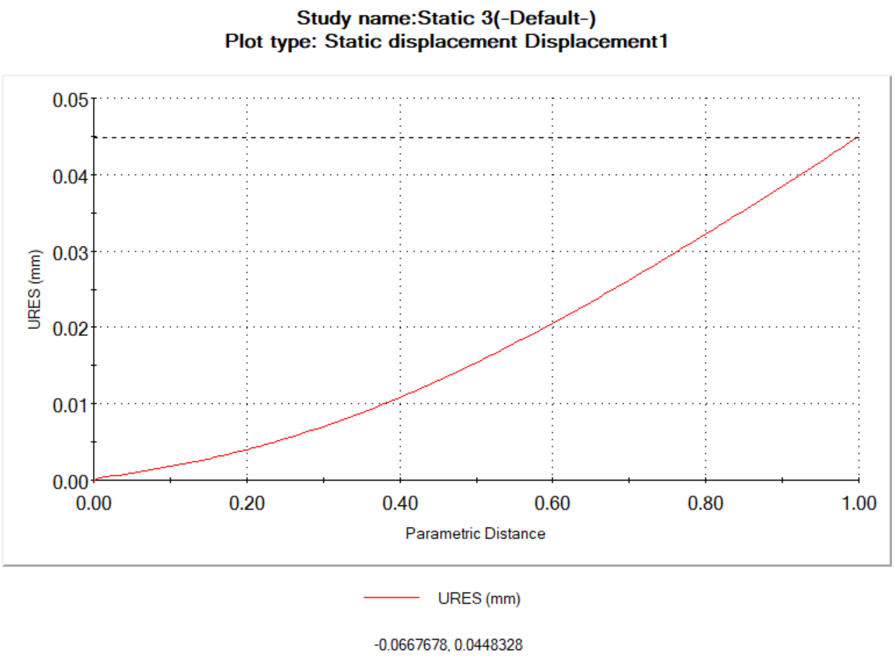


Fig.5. Maximum deflection graph at the free end for P = 5000N

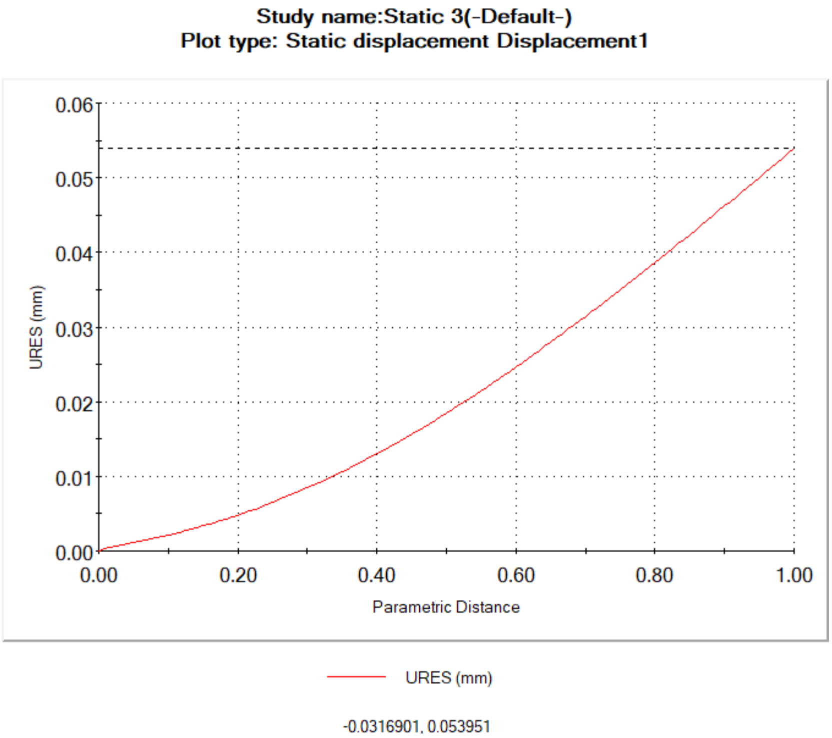


Fig.6. Maximum deflection graph at the free end for P = 6000N

CHAPTER 6

CONCULSION

The conclusions given below which are mainly based on contrast between the maximum deflection result that is obtained by the theoretical method FEA for a cantilever beam point loading at free end.

- The maximum deflection value decreases when the value of point load is increased.
- As you have seen in this article, the solution of a simple engineering problem (single cantilever beam) can be solved by either analytical expression and Finite Elements Analysis (FEA). The solution is almost the same but some deviation can occur by using numerical methods by FEA.
- These complex engineering systems are not easy to be solved by analytical expressions and it is there where FEA takes some advantages. FEA allows simulating complex parts in a reasonable time so FEA represents an important tool for engineers. The use of FEA tools can positively contribute to reducing the design and development time of an engineering part or assembly.

CHAPTER 7

LIST OF PUBLICATION

Paper 1: - Aditya Pratap Singh, Amresh Kumar, Shrikant Vidya, P. Suresh, K. S. Shrikant

"Comparative stress analysis of round shaft with shoulder fillet using finite element and analytical method"

Materials Today: Proceedings,

Volume 56, Part 1,

2022,

Pages 497-501,

ISSN 2214-7853,

https://doi.org/10.1016/j.matpr.2022.02.168.
https://www.sciencedirect.com/science/article/pii/S2214785322007453

Paper 2: - Aditya Pratap Singh, Amresh Kumar, Shrikant Vidya, P. Suresh,

"Comparative Deflection Analysis of a Cantilever Beam Fixed at One End by Using Finite Element and Analytical Method"

Status: - Submitted in ICEAME-2022 (Scopus Based Conference)

CHAPTER 8

REFERENCES

1. F. W. Riley and L. D. Sturges, Engineering Mechanics: Statis, Jonh Wiley & Sons New York (1993)
2. A. Bedford and W. Fowler, Engineering Mechanics: Statics, Addison Wesley, Massachusetts (1996)
3. D. J. McGill and W. W. King, Engineering Mechanics: Statics, PWS Publishing Company, Boston (1995)
4. Samal, A. K., & Rao, T. E. (2016). Analysis of stress and deflection of cantilever beam and its validation using ANSYS. Int. J. Eng. Res. Appl., 6, 119-126.
5. Wu, T. M. (2006). Computer-aided Deflection and Slope Analyses of Beams. Journal of Applied Sciences, 6(2), 333-339.
6. Lee, K. (2002). Large deflections of cantilever beams of non-linear elastic material under a combined loading. International Journal of Non-Linear Mechanics, 37(3), 439-443.
7. Moaveni, S. (1998). Numerical experiments for a Mechanics of Materials course. International Journal of Engineering Education, 14(2), 122-129.
8. Oueini, S. S., & Nayfeh, H. A. (1999). Single-mode control of a cantilever beam under principal parametric excitation. Journal of Sound and Vibration, 224(1), 33-47.
9. Joshi, K. H., & Patel, C. M. (2016). Development of Analytical method to determine the deflection of tapered cantilever beam with inclined loading condition using software simulation. In *Proceedings of International Conference on ICT for Sustainable Development* (pp. 281-288). Springer, Singapore.
10. Al-Gahtani, H. J., & Khan, M. S. (1998). Exact analysis of nonprismatic beams. *Journal of engineering mechanics, 124*(11), 1290-1293.
11. Bhavikatti, S. S. (2005). *Finite element analysis*. New Age International.
12. Logan, D. L. (2016). *A first course in the finite element method*. Cengage Learning.

App

ORIGINALITY REPORT

8%
SIMILARITY INDEX

PRIMARY SOURCES

1	Oueini, S.S.. "Single-Mode Control of a Cantilever Beam Under Principal Parametric Excitation", Journal of Sound and Vibration, 19990701 Crossref	41 words — 2%
2	hdl.handle.net Internet	36 words — 1%
3	Aditya Pratap Singh, Shrikant Vidya, P. Suresh, Amresh Kumar, K.S. Srikanth. "Comparative stress analysis of round shaft with shoulder fillet using finite element and analytical method", Materials Today: Proceedings, 2022 Crossref	26 words — 1%
4	calcresource.com Internet	22 words — 1%
5	pdfs.semanticscholar.org Internet	17 words — 1%
6	technicaljournalsonline.com Internet	16 words — 1%
7	lib.buet.ac.bd:8080 Internet	11 words — < 1%
8	Ibrahim Eren. "Determining large deflections in rectangular combined loaded cantilever beams	10 words — < 1%

made of non-linear Ludwick type material by means of different arc length assumptions", Sadhana, 02/2008
Crossref

---

9 baixardoc.com — 10 words – < 1%
Internet

---

10 2y39.com — 8 words – < 1%
Internet

---

11 Osezua Obehi Ibhadode, Ishaya Musa Dagwa, Jude Odianosen Asibor, Eghosa Omo-Oghogho. "Development of a Computer Aided Beam Deflection Analysis (CABDA) Program for Simply Supported Loaded Beams", International Journal of Engineering Research in Africa, 2017 — 8 words – < 1%
Crossref

---

12 Bhaveshkumar Prahladbhai Patel, Dhaval Punamchand Patel. "Deflection analysis of uniform cross sectional hollow rectangular cantilever beam with transverse stiffeners under UDL and concentrated load", Engineering Research Express, 2021 — 6 words – < 1%
Crossref

EXCLUDE QUOTES	ON	EXCLUDE SOURCES	< 6 WORDS
EXCLUDE BIBLIOGRAPHY	ON	EXCLUDE MATCHES	OFF

ARTICLE IN PRESS

Materials Today: Proceedings xxx (xxxx) xxx

Contents lists available at ScienceDirect

Materials Today: Proceedings

journal homepage: www.elsevier.com/locate/matpr

Comparative stress analysis of round shaft with shoulder fillet using finite element and analytical method

Aditya Pratap Singh, Shrikant Vidya *, P. Suresh, Amresh Kumar, K.S. Srikanth

Department of Mechanical Engineering, Galgotias University, Uttar Pradesh, India

ARTICLE INFO

Article history:
Available online xxxx

Keywords:
Finite element analysis
Round shaft
Stress concentration
Solid works

ABSTRACT

In the design of machine elements asymmetry and sudden changes in dimensions of surface are unpreventable due to some characteristics of the element such as oil cavity and grooves, splines and keyways, shoulders and screw threads. But such discontinuities often lead to stress concentration near the irregularity due to which the stress near the irregularity is higher than the average stress in the whole member. In this study, rounded shaft with shoulder fillet is analyzed by theoretical method and finite element analysis. The round shaft is subjected to tensile loading and the stress concentration factor for different values of fillet radius is calculated. Further, the maximum stress is calculated theoretically and is compared with maximum stress computed by using Solid works. The static study shows that the level of error among analytical and fillet component arrangements has been viewed as <5%. And the error percentage decreases as the value of fillet radius increases.

Selection and peer-review under responsibility of the scientific committee of the International Conference on Materials, Machines and Information Technology-2022.

1. Introduction

In developing many building materials as well equipment, it is unpreventable to avoid changes in the section, notches, groove, holes, etc. to reduce the force of material to give access to one more part of machines or equipment. In those cases, the distribution of average force at the top access reach will be too large for greater than normal stress above part. The concentration of stress can be determined on the shaft in various ways: photo elasticity, Finite Element Analysis, Analytic mathematics methods, etc. By suggesting the behavior of this focus, an architect can change its pattern to grow the service span of the feature and the safety of the individual working it. Its effectiveness be dependent on the amount of force Concentration Factor handed down, because when it is large, the feature often fails.

Stress concentration (commonly referred to as stress producers) is the location of the object where the main stress is located. The structure is very stable when energy is evenly distributed in its area. Restriction of geometry, reduction of the geographical area is increase stress when loaded out. Stress distribution may be localized or may vary. Nearly all engineering use has a flexible distribution, and that is why it is prime to forecast or have a clearity of how the attribute will behave under certain load conditions.

The presence of fillet radius on a shoulder shaft locates the stress concentration and we cannot avoid all possibilities that affect the stress concentration. Several researches had been published on the analysis of stress concentration from the analytical method. Fillets are frequent applications in mechanical frequent to give a plane transition in fields where available unexpected switch on rod as in that case of the shoulders. Shoulders dispense in bars for a variation of motive, cam profile, to give support, etc. However, this is sometimes the reason for the growth in local stress levels.

K.S. Babulal has presented the maximum force generated at three dissimilar types of notches that are found in steel plates that is; semi-circular notch, U-shaped notch and V-shaped notches. The fixed analysis reflects that the error percentage between analytical method and finite element solutions is <2% [1].

The stress concentration factor (SCF, Kt) of a particular non-persistent structure such as; U-shaped notched in shaft, plates and fillet shaft are considered under stress and bending. High stress on fillets and notches is calculated using of ABAQUS software. The results show, when compared with the theoretical results of the theories obtained by the mathematical solutions found in the literature, also shows that, there is an inverse

* Corrresponding author.
E-mail address: skvrsm@gmail.com (S. Vidya).

https://doi.org/10.1016/j.matpr.2022.02.168
2214-7853/
Selection and peer-review under responsibility of the scientific committee of the International Conference on Materials, Machines and Information Technology-2022.

Please cite this article as: Aditya Pratap Singh, S. Vidya, P. Suresh et al., Comparative stress analysis of round shaft with shoulder fillet using finite element and analytical method, Materials Today: Proceedings, https://doi.org/10.1016/j.matpr.2022.02.168

relationship between values (Kt) and (r/d) estimation values, where as a relationship between (Kt) and (D/d) values is positive [2–4].

Hiren Prajapat [5] analysed the stress concentration factor (SCF) is calculated for the shoulder filleted shaft and also set side by side with the theoretical outcome acquire from derived modified Pilkey's equations, S. M. Tipton equations and Roark's equations. The outcome shows that for lower value of D/d ratio result identical in nature but for higher value D/d ratios, the outcome found different. With increasing D/d ratios the SCFs are decreased.

Sonmez [6] studied about the global shape increase in efficiency of shoulder fillets in flat bars and round bars was achieved. Although most appropriate fillet sketch of flat bars was a big issue that was regarded by other researchers and scholars before, improved outcome were result in this shaping and studying of bars by making use of a global shape optimization procedure improvement is achieved by making using a numerical search algorithm called direct search simulated simulation. The exact design was found in flat bars and circular bars below to the bend, torsional, axial, or integrated loads. The obtained results show that the concentration factors were near to one can be accessed when in bars with notable changes of different categories. Otherwise, the region density of the optimum fillets is very small compared to round or elliptical.

Troyani et al. [7] analyzed the result of the shaft length as the focus attribute of the shafts with shoulder fillet under the same tension. The results showed that the length of the member has an important effect on the stress caused by the shoulder fillet.

Duris [8] has presented the numerically calculated value of stress concentration factor with selected numbers cases of circular shaft with shoulder fillet higher compared the values presented in the books. The biggest difference between numerical results and analysis was 5.4% for a bar with D = 150 mm and a diameter D/d = 1.5. As a result, numerical stress concentration factor for fatigue statistics will not decrease level of security.

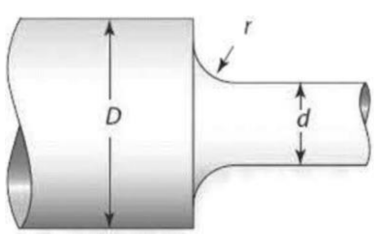


Fig. 1. Round shaft with shoulder fillet.

Augusto Ajovalasit [9] introduced the historical evolution of technique that was used for the study of torsion forces on shafts. Experimental technique based on structural and mathematical models are also evaluated, and a number of analyses based on distinct approaches are collected and compared to the two standard approaches. Case study: a static section shaft with a key and a symmetric axis shaft with a shoulder fillet.

The shape-improvement problem of reduce stress concentration factors can be regarded as the mathematical programming problem, the structural analysis is obtained by the use of finite-element method. The pliability of finite elements in action with contrasting kinds of shapes and weights, permit an extremely usual and automatic use of the method to different problems [10].

Gujar [11] studied a life of prediction is made based on Finite element method and analytical method using a constant amplitude load, fatigue life of the dynamometer shaft be predicted. This research will help to understand more conduct dynamometer shaft and give manufacturer knowledge to develop fatigue life of the dynamometer shaft using FEA tools. It is clear from the above results, the difference between both result is around 10%, which is an acceptable distance.

There are several research had been published on the analysis of stress concentration from the analytical method. In this present study, there will be comparative study between finite element method and analytical method. While increasing the fillet radius, there will be change in maximum stress on the body and the maximum stress is defined as the multiplication of SCF (stress concentration factor) and nominal stress. Normal stress is defined same as the stress that is the force on the object divided by the original area. The calculation for maximum stress on the body can be calculated from analytical and finite element method.

2. Materials and method

A bar of roundcross-segment with a shoulder and oppressed is displayed in Fig. 1. The shoulder makes an adjustment of cross-segment of the shaft, which brings about stress concentration. Stress concentration factor Kt, which is the proportion of the greatest stress (ґ_max) to the nominal stress (ґ_0) as introduced in below Eq. (1)

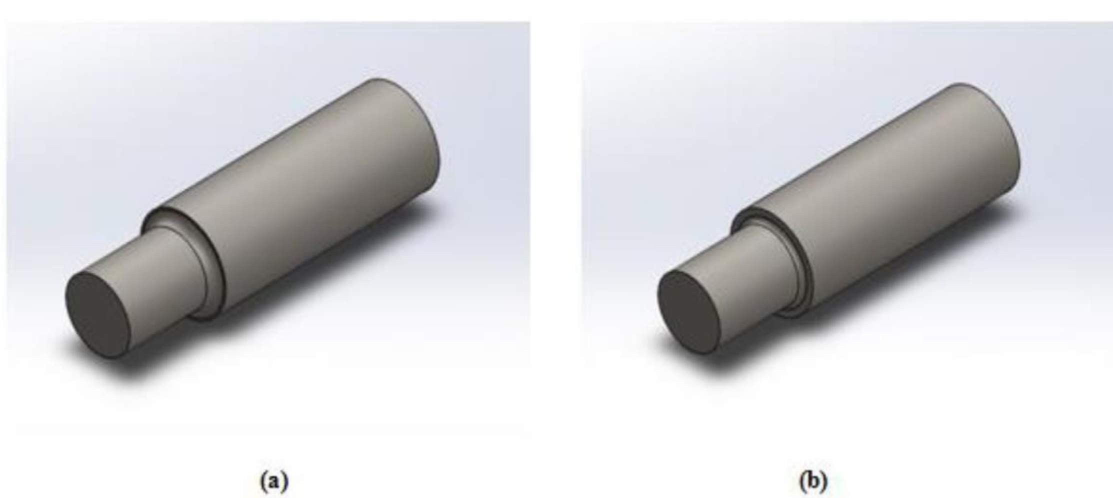


Fig. 2. Round Shaft with (a) r = 0.01 m, D = 0.2 m, d = 0.15 m (b) r = 0.02 m, D = 0.2 m, d = 0.15 m.

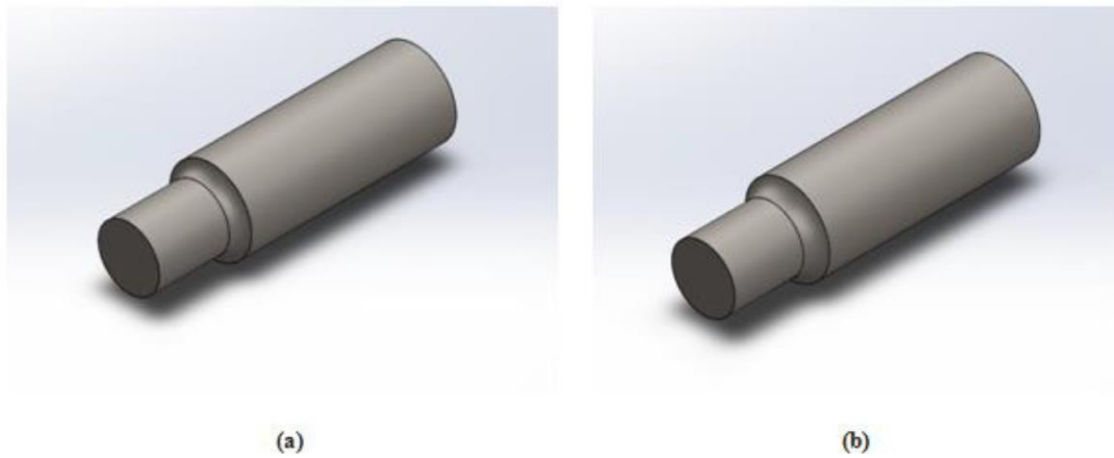


Fig. 3. Round Shaft with (a) r = 0.03 m, D = 0.2 m, d = 0.15 m (b) r = 0.04 m, D = 0.2 m, d = 0.15 m.

$$K¼ \frac{r_{max}}{r_0} \quad ð1Þ$$

where K_t ¼ Stress concentration factor

The stress concentration factor is calculated with the help of graph of the stress concentration factor of round shaft in tension [12]. As we know the smaller the fillet radius (r) or larger the D/d ratio results in high SCF. It can be easily found from Fig. 4. Stress concentration factors for a round shaft of shoulder fillet radius can be found in many standard textbooks of mechanics of materials. The diagrams for stress concentration factors for various mathematical shapes and states of loading were initially evolved by RE Peterson [13]. So far, we use Peterson charts for stress concentration factors in case of geometry of shaft, bar, and plate to calculate value of stress concentration factor. The curves given by Peterson for round shaft with shoulder fillet radius were based on four different parameters i.e. larger diameter (D), smaller diameter (d), shoulder fillet radius (r), and stress concentration factor (Kt). In the conventional approach, to get the stress concentration factor using Peterson's curves, the ratios radius/diameter, and D/d are to be considered as known. As mentioned by Peterson, the curves were developed from the results obtained from test work and scientific work did by a few specialists before. Fig. 1 shows the shoulder with a fillet of radius r. This outcome is in steady progress from little breadth to an enormous width. The fillet radius ought to be just about as extensive as conceivable to lower stress concentration. By and by, the fillet range is restricted by the plan of mating parts.

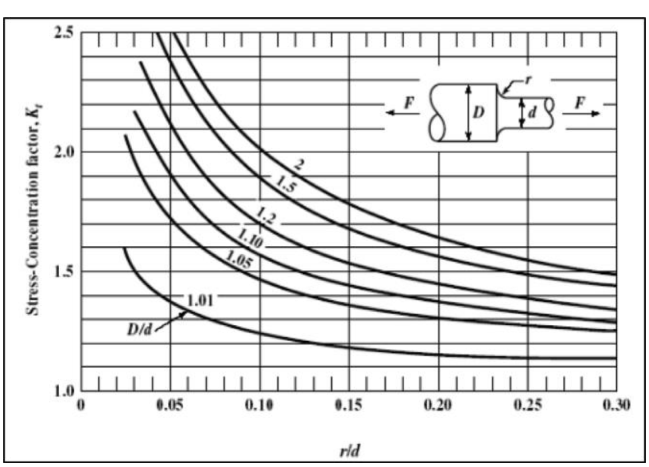


Fig.4. Change in stress concentration factor with radius/diameter for a filleted shaft with tensile loading.

In this study, by changing the value of fillet radius, there is huge reduction in maximum stress from the both method (analytical and finite element solution). The shape has been taken as round shaft with fillet radius (r = 0.01 m, 0.02 m, 0.03 m and 0.04 m), larger diameter (D = 0.2 m), smaller diameter (d = 0.15 m), tensile force (F = 15000 N) and has been modeled in solid works (Figs. 2 and 3). The value of maximum stress has been calculated from the analytical method and finite element method. The material is assumed to be isotropic and material has been taken for the study is 1023 Carbon Steel sheet (SS). Steels (linear elastic isotropic) containing carbon as the principle alloying component are known as carbon steels having up to 0.4% Si and 1.2% Mn. Components, for example, Cr, Ni, Al, Cu and Mo are additionally present in little amounts. The mechanical properties of the material are mentioned in the Table 1.

2.1. Finite element analysis

FEA is utilized by architects to assist with reproducing actual peculiarities and in this way lessen the requirement for actual models, while taking into consideration the advancement of parts as a feature of the plan cycle of an undertaking. The Finite component investigation (FEA) is broadly utilized for the perplexing calculations where customary examination techniques can't tackle the issues as FEA partitions the difficult calculations into more modest parts (components). Finite component investigation of shoulder filleted shaft exposed to loading (axial) is done utilizing SOLIDWORKS 19.0. Simulation can give exact, solid outcomes for a wide scope of study types from fundamental direct static examination to more perplexing nonlinear and dynamic investigation. Accelerate the emphasis and prototyping period of your configura-

Table 1
Material properties of 1023 Carbon steel sheet (SS).

Property	Value	Unit
Density	7858.000032	kg/m^3
Poisson's coefficient	0.29	–
Modulus of rigidity	79999.99987	MPa
Elastic modulus	204999.9984	MPa
Tensile strength	425.0000032	MPa
Yield strength	282.68049	MPa

tion interaction with Simulation Model has been developed in the solid Works and there is external load acting on the body is 15,000 N. Meshing is one of the critical parts to getting exact outcomes from a FEA model. The meshing information is shown in Fig. 5. The much fine meshing is created on the fillet surface.

The after effects of the finite element study are introduced and analyzed for shoulder filleted shaft. Complete four diverse fillet range models are ready, dissect and mimicked. Figs. 6, 7, 8 and 9 shows different FEA result of a shaft with shoulder filet under axial loading. Here, for analytical reason, the applied burden is viewed as steady as 15,000 N for all cycles with various fillet range, and the D/d ratio is same for all iteration. The result has been calculated from the change in maximum stress from all four different fillet radiuses. The maximum stress represented as von Mises stress. Von Mises stress is a quantity used to decide whether a given material will yield or crack. It is for the most part utilized for flexible materials, like metals. The Von Mises yield model expresses that if the Von Mises stress of a material under load is equivalent

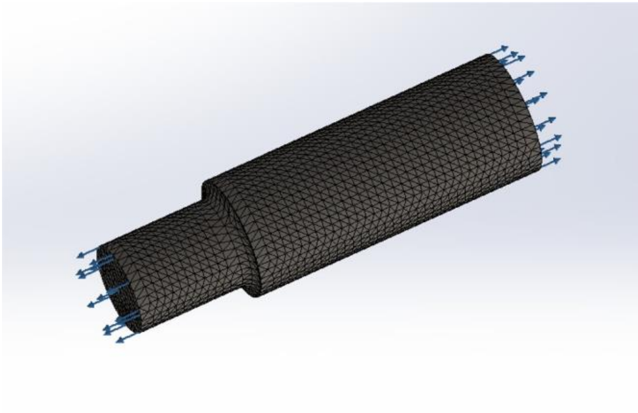


Fig. 5. Meshed model of shoulder shaft.

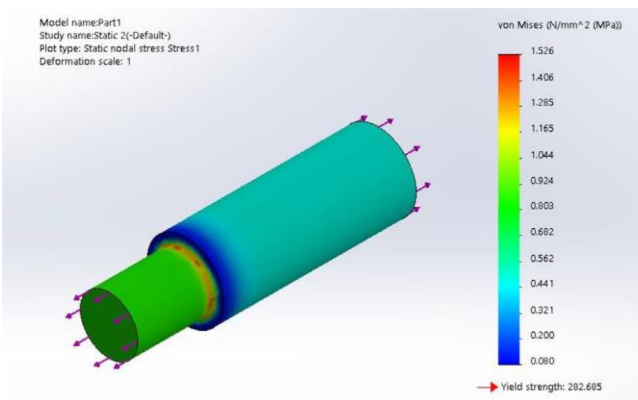


Fig. 6. Equivalent stress (Von Mises) for r = 0.01 m (D = 0.2 m and d = 0.15 m).

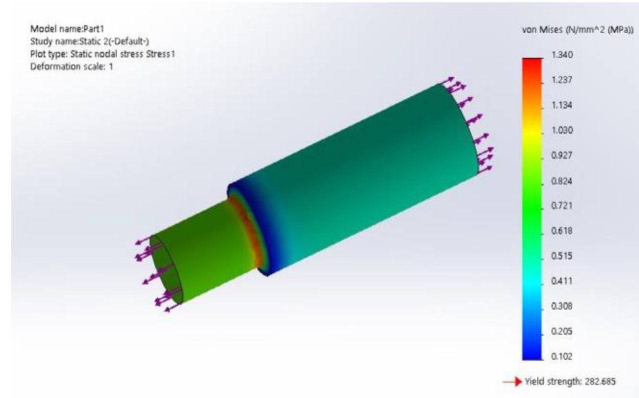


Fig. 7. Equivalent (Von Mises) stress for r = 0.02 m (D = 0.2 m and d = 0.15 m).

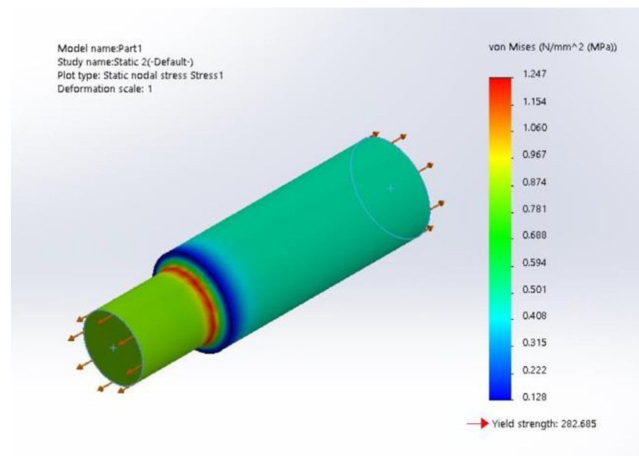


Fig. 8. Equivalent (Von Mises) stress for r = 0.03 m (D = 0.2 m and d = 0.15 m).

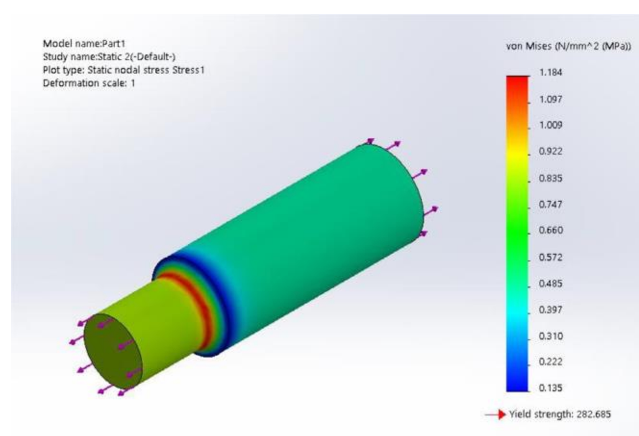


Fig. 9. Equivalent (Von Mises) stress for r = 0.04 m (D = 0.2 m and d = 0.15 m).

Table 2
Maximum stress comparison obtained by FEA &theoretical maximum stress obtained with the help of Peterson SCF graph [5] under axial loading.

D (m)	D (m)	Radius of fillet, r (m)	Load value P (N)	Theoretical SCF (Kt)	Theoretical stress (maximum) (MPa)	Equivalent stress (von misses)(MPa)	Percentage deviation (%)
0.20	0.15	0.01	15,000	1.93	1.601	1.526	4.68
0.20	0.15	0.02	15,000	1.66	1.394	1.340	3.87
0.20	0.15	0.03	15,000	1.52	1.276	1.247	2.27
0.20	0.15	0.04	15,000	1.43	1.201	1.184	1.41

or more noteworthy than the yield furthest reaches of a similar material under straightforward strain then the material will yield.

From the FEA results, identical Von Mises stresses of shoulder filleted shaft are recognized and used them for comparative study from the maximum stress obtained from analytical method. All the results of FEA for equivalent von misses and the result calculated from the analytical method are showed in the Table 2, also its contrast are made with maximum stress obtained from FEA and analytical method.

3. Results and discussion

There is no such thing as a personal choice. The appropriate fillet radius of the shoulder shaft will provide minimum element of stress of a given geometry. Therefore, an effort was made to improve the condition preference of radius of fillet of the filleted shaft (shoulder) to perform give the idea about the maximum stress concentration. The calculation for the maximum stress from the both method mentioned in Table 2, By increasing the value of fillet radius there is less amount of stress being applied from the both method (theoretical and FEA).

From Table 2, the results obtained by the finite element analysis and theoretical analysis are approaching toward the same value when the value of radius increased.

4. Conclusion

Following observations are made dependent on the examination between the after effects of greatest stress acquired from hypothetical strategy and FEA in case of shoulder filleted shaft (circular bar) under loading (axial) condition.

- By increasing value of fillet radius the value of maximum stress decreasing.
- The static study shows that the level of error among analytical and fillet component arrangements has been viewed as <5%. And the error of percentage is decreasing by increasing the value of fillet radius.
- The value of theoretical stress concentration factor have been calculated by the RE Peterson chart, this could be the reason for the variation in the both result that have been calculated by the theoretical and the analytical method.

CRediT authorship contribution statement

Aditya Pratap Singh: Conceptualization, Methodology. Shrikant Vidya: Supervision. P. Suresh: Supervision. Amresh Kumar: Data curation, Writing – original draft. K.S. Srikanth: Data curation, Visualization.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] K.S. Babulal, I.V. Kannan, A. Pradeep, Int. J. Curr. Res. Aca. Rev. 3 (2015) 144–154.
[2] L.S. Jabur, Am. Int. J. Res. Sci., Technol., Eng., Math. (2016) 2328–3629.
[3] A. Santos, Journal of Physics, Conference Series, vol. 1, 466, IOP Science Publishing, 2013.
[4] A. Kotousov, Int. J. Solids Struct. 2010 (47) (1916) 1923.
[5] B. Patel, H. Prajapati, Trends Mech. Eng. Technol. 8 (2018) 2.
[6] F.O. Sonmez, Proc. Inst. Mech. Eng., Part C J. Mech. Eng. Sci. 223 (8) (2009) 1741–1754.
[7] N. Troyani, G. Sterlacci, C. Gomes, Int. J. Fat. 25 (4) (2003) 353–355.
[8] R. Duris, E. Labasova, Ann. DAAAM Proc. (2011) 181–183.
[9] A. Ajovalasit, V. Nigrelli, G. Pitarresi, G. Virzi Mariotti, J. Str. Ana. Engg. Des. 49 (6) (2014) 452–466.
[10] A. Francavilla, A.C. Ramakrishnan, O.C. Zienkiewicz, J. Str. Ana. 10 (2) (1975) 63–70.
[11] R.A. Gujar, S.V. Bhaskar, Int. J. Engg. Res. App. 3 (4) (2013) 1061–1066.
[12] W.D. Pilkey, D.F. Pilkey, Z. Bi, Peterson's Stress Concentration Factors, John Wiley & Sons, 2020.
[13] R.E. Peterson, Stress Concentration Design Factors, John Wiley, New York, 1974.