ISSN (Online): 2455-3662
EPRA International Journal of Multidisciplinary Research (IJMR) - Peer Reviewed Journal
Volume: 9| Issue: 9| September 2023|| Journal DOI: 10.36713/epra2013 || SJIF Impact Factor 2023: 8.224 || ISI Value: 1.188

STRESS ANALYSIS USING SOLIDWORKS SIMULATION

Yazeed Mohammed Ahmed Almashani[1], Abdullah Ahmed Alamri[2], Muhab Mohammed Faraj Al-Ghassani[3], Mithun V Kulkarni[4*]

[1,2,3] *Student, University of Technology and Applied Sciences – Salalah, Oman*

[4] *Senior Lecturer, University of Technology and Applied Sciences -Salalah, Oman*

Article DOI: https://doi.org/10.36713/epra14295
DOI No: 10.36713/epra14295

ABSTRACT

Stress analysis using SolidWorks Simulation is a powerful tool for predicting the behaviour of components and structures under various loading conditions. The methodology involves creating a 3D model of the component, assigning material properties, defining boundary conditions, meshing, setting up the analysis, and analysing the results. SolidWorks Simulation includes a wide range of analysis types, such as static, dynamic, and thermal, and provides a comprehensive suite of tools to visualize and analyse the results. The paper focusses on performing Solidworks simulation studies and hence comparing the results with the hand calculated results. The paper is prepared based on the second-year diploma project, in which the students were taught the basics of stress analysis and use of Solidworks 2021 software for performing stress analysis on bars and beams subjected to axial, bending, torsion and combined loads. However, in this paper only bending related problem has been discussed. The concept of stress analysis was also taught to the students and hence an example related to stress concentration has also been discussed. The results of the comparison reveal that the percentage variation between the hand calculated, and simulation results is less than 1%.

KEYWORDS: ***Solidworks, Stress analysis, 3D modelling, Engineering design, Stress concentration, FEA***

1. INTRODUCTION

Stress analysis is an essential aspect of engineering design, as it helps ensure that the designed components can withstand the loads and stresses, they are subjected to in their operating environment. SolidWorks Simulation is a widely used computer-aided engineering (CAE) tool that enables engineers to simulate and analyse the behaviour of designs under various load and boundary conditions. By using SolidWorks Simulation, engineers can predict and optimize the performance of their designs, improve product quality, reduce development time, and minimize costs associated with physical testing.

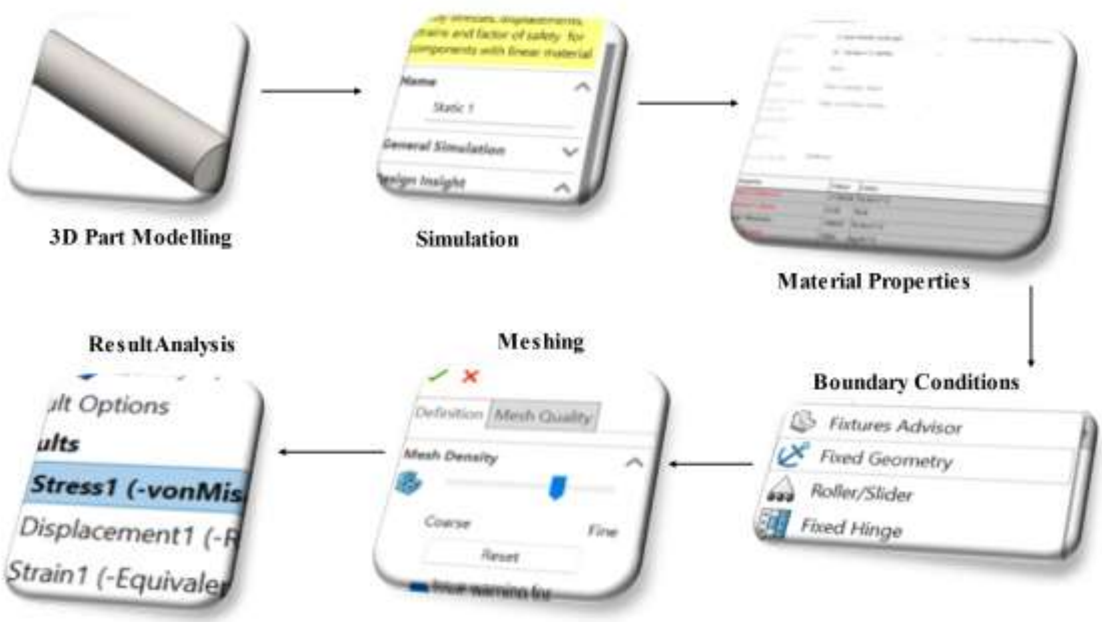
SolidWorks Simulation is a finite element analysis (FEA) software that uses the finite element method (FEM) to discretize complex models into smaller, more manageable elements. The software then solves the equations of motion and calculates the stresses, strains, and displacements of each element based on the applied loads and boundary conditions. The results of the analysis can then be used to evaluate the performance of the design and make any necessary modifications before physical prototyping.

SolidWorks Simulation can perform a wide range of stress analyses, including static analysis, dynamic analysis, thermal analysis, and fatigue analysis. Static analysis is used to determine the stresses and deformations of a design under steady-state loads, while dynamic analysis is used to simulate the response of a design to time-varying loads. Thermal analysis is used to analyse the temperature distribution and heat transfer within a design, while fatigue analysis is used to predict the lifespan of a design under cyclic loading.

The process of stress analysis using SolidWorks Simulation can be understood with the help of Fig.1 and is elaborated in the paragraphs to follow.

ISSN (Online): 2455-3662
EPRA International Journal of Multidisciplinary Research (IJMR) - Peer Reviewed Journal
Volume: 9| Issue: 9| September 2023|| Journal DOI: 10.36713/epra2013 || SJIF Impact Factor 2023: 8.224 || ISI Value: 1.188

Fig. 1 The steps followed during stress analysis using SolidWorks Simulation



Modelling: The first step is to create a 3D model of the design in SolidWorks. This can be done by either importing an existing CAD model or creating a new one from scratch. The model should be fully defined and include all necessary components, materials, and boundary conditions.

Meshing: Once the model is complete, it needs to be meshed. Meshing involves dividing the model into smaller elements, each of which can be analysed individually. The mesh should be dense enough to capture the important features of the design but not so dense that the analysis becomes computationally expensive.

Applying loads and boundary conditions: The next step is to apply the loads and boundary conditions to the model. Loads can include forces, pressures, and moments, while boundary conditions can include fixed or sliding supports, symmetry, or periodicity.

Running the analysis: With the loads and boundary conditions defined, the analysis can be run. SolidWorks Simulation will solve the equations of motion and calculate the stresses, strains, and displacements of each element in the model.

Interpreting the results: Once the analysis is complete, the results can be reviewed and interpreted. The engineer can use the results to evaluate the performance of the design and identify any areas of concern. If necessary, modifications can be made to the design, and the analysis can be rerun.

To summarise, stress analysis using SolidWorks Simulation is a powerful tool that enables engineers to predict and optimize the performance of their designs. By simulating the behaviour of designs under various load and boundary conditions, engineers can identify and resolve potential issues before physical prototyping, reducing development time and costs. SolidWorks Simulation is a comprehensive FEA software that can perform a wide range of stress analyses, from static and dynamic to thermal and fatigue. With its intuitive interface and powerful features, SolidWorks Simulation is an indispensable tool for any engineer involved in product design and development.

The present work explains how to use Solidworks software for performing stress analysis on elements subjected to bending and stress concentration. Solidworks 2021 has been used for performing simulation studies and also the results obtained from the simulation have been verified by hand calculations.

2. Literature Review

The use of computer-aided engineering (CAE) software for stress analysis has become increasingly common in recent years, with SolidWorks Simulation being one of the most widely used programs. SolidWorks Simulation is a finite element analysis (FEA) software that uses the finite element method (FEM) to simulate and analyse the behaviour of designs under various load and boundary conditions. In this literature review, we will explore the existing research on stress analysis using SolidWorks Simulation and its applications in various fields.

One of the earliest studies on the use of SolidWorks Simulation for stress analysis was conducted by (Koirala et al., 2021) in their research on the deformation and stress analysis of a bridge structure. The authors used SolidWorks Simulation to model the bridge and performed static and dynamic analyses under various load and boundary conditions. They found that the software provided accurate results and was able to predict the behaviour of the structure under different loading scenarios.

In the field of aerospace engineering, SolidWorks Simulation has been used to analyse the stress and deformation of aircraft components. For example, (Grodzki & Le, n.d.) used SolidWorks Simulation to analyse the stress distribution and deformation of an unmanned aerial vehicle under different flight conditions. The authors found that the software was able to accurately predict the behaviour of the carbon fabric epoxy resin laminate and thus used in developing the wings of the aircraft.

SolidWorks Simulation has also been used in the field of biomechanics to analyse the stress and strain on various parts of the human body. For example, (Dorji et al., 2019) used SolidWorks simulation and abacus to analyse a new type of brace using three types of polymers. Stress, strain and the rate of displacement of the brace was analysed. In another work by (Landines Jiménez et al., 2019), simulation of forces using Solidworks program on human femur was performed. For performing simulation studies, the real values of mechanical properties of cortical bone were used.

In the field of civil engineering, SolidWorks Simulation has been used to analyse the stress and deformation of various structures, such as buildings and bridges, infrastructure development, office stationeries and furniture's. For example, (Ardita, 2018) used SolidWorks program for modelling and design optimisation of folding table. These types of table are extensively preferred in residential buildings that have space constraints.

SolidWorks Simulation has also been used in the field of mechanical engineering to analyse the stress and deformation of various components, such as gears and bearings. (Kadhim Zarzoor et al., 2018) used SolidWorks Simulation to analyse the stress distribution and deformation of a spur gear under different load and boundary conditions. The authors found that the software was able to accurately predict the behaviour of the gear and identified areas where the design could be improved.

In addition to its applications in various fields, SolidWorks Simulation has also been used in research on design by analysis(DBA) and design by formula(DBF). (Altinbalik & Kantur, 2020) used SolidWorks for modelling and analysis of a water storage tank. The results of the study indicated that the use of DBA is more effective than using DBF for reducing the cost of construction and materials by 50%.

Overall, the existing research on stress analysis using SolidWorks Simulation demonstrates the software's accuracy and versatility in simulating and analysing the behaviour of designs under various load and boundary conditions. The software has been used in various fields, including aerospace engineering, biomechanics, civil engineering, and mechanical engineering, and has been applied to analyse the stress and deformation of various components, structures, and even the human body. In addition, SolidWorks Simulation has also been used in research on optimization techniques, allowing engineers to identify the optimal design for their application.

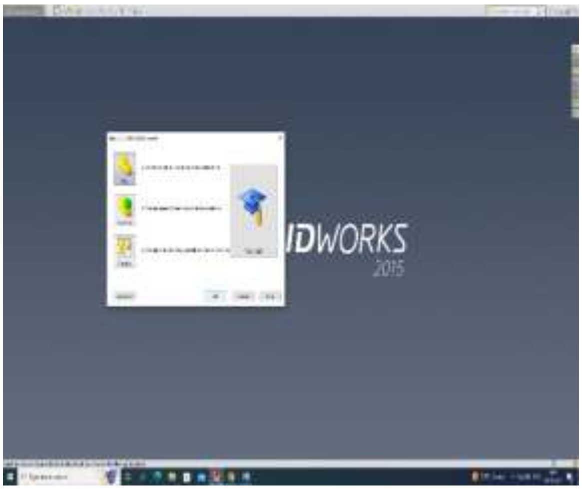
3. METHODOLOGY

The steps followed in the stress analysis using Solidworks simulation has been explained with the help of an example in this section. The results of the simulation has been compared with the hand calculation results, which has been discussed in the next section Also, an example related to analysing stress concentration in a plate with hole has been addressed in the results and discussions section.

Example 1: Beam subjected to pure bending

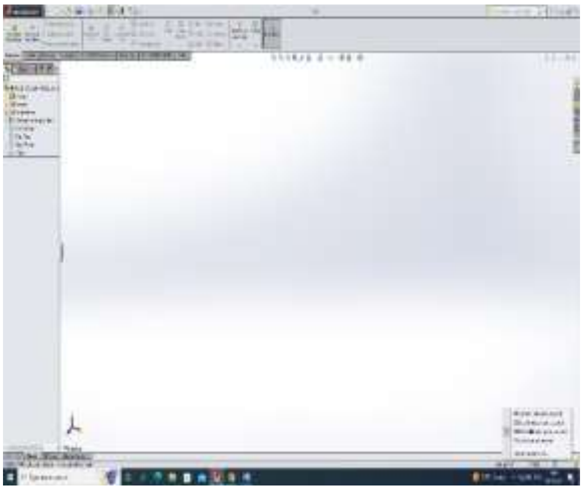
A short Cantilever beam of width 30 mm and depth 40 mm supports a load of 20 kN at a distance of 40 mm from the fixed end. Determine the maximum bending stress in the beam.

Step 1: Open a new file.

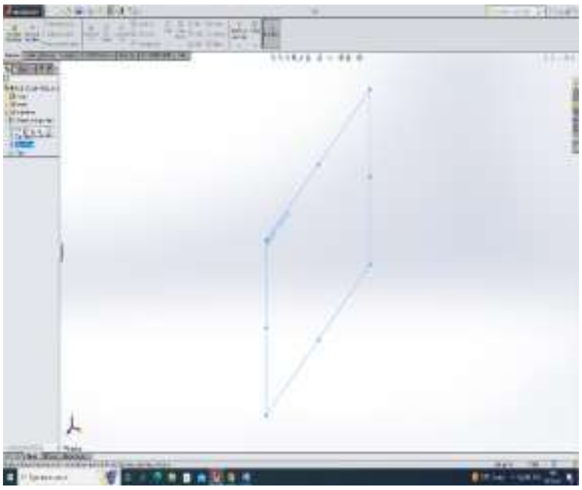


Step 2: Click on the part module → Ok.

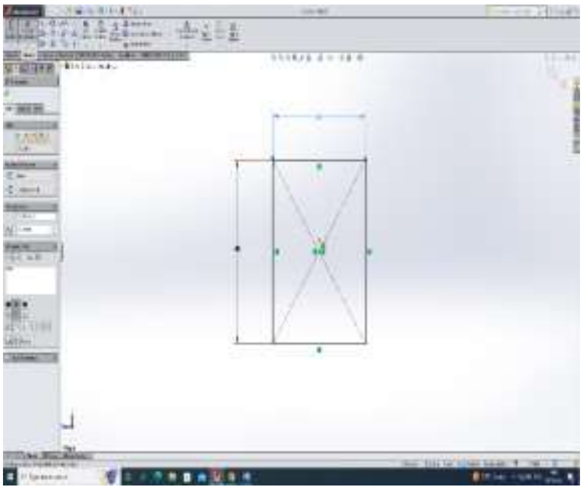
ISSN (Online): 2455-3662
EPRA International Journal of Multidisciplinary Research (IJMR) - Peer Reviewed Journal
Volume: 9| Issue: 9| September 2023|| Journal DOI: 10.36713/epra2013 || SJIF Impact Factor 2023: 8.224 || ISI Value: 1.188



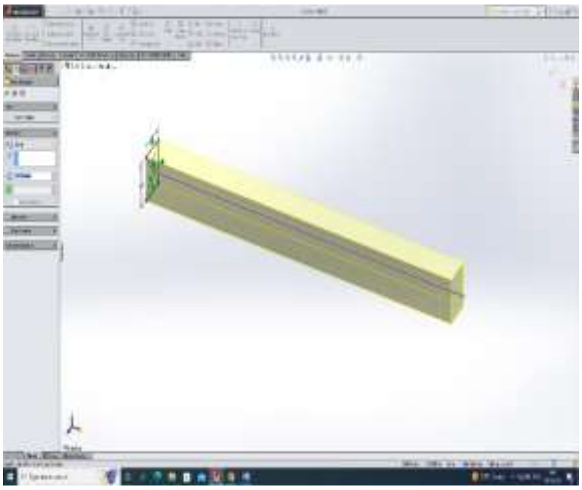
Step 3: Check the units for MMGS.



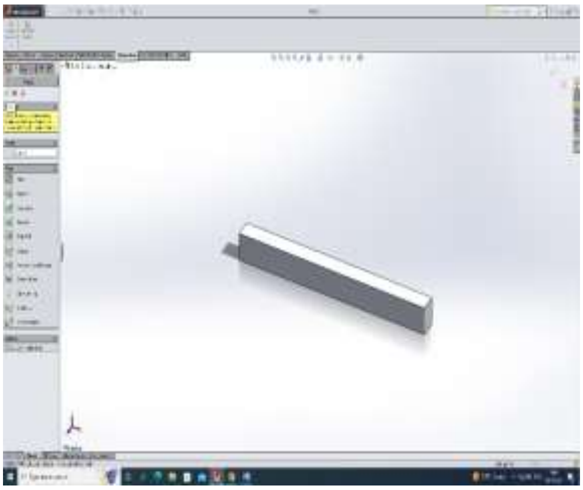
Step 4: Choosing the right plan → Normal to.



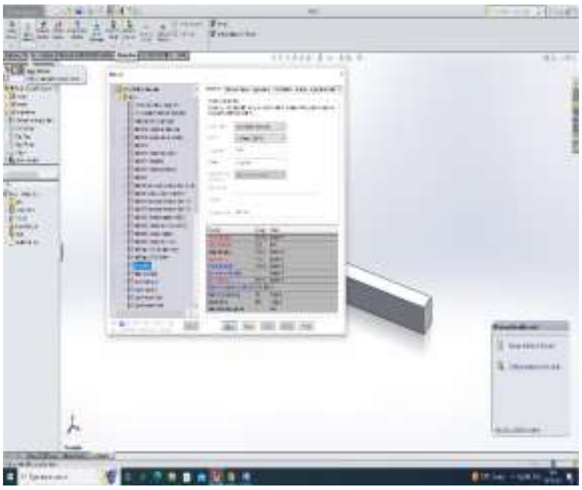
Step 5: Choose Sketch → Centre Rectangle → Smart Dimension → set the width to 30 mm and height to 40 mm.



Step 7: Features → Extrude boss/base → set D1 = 400 mm.

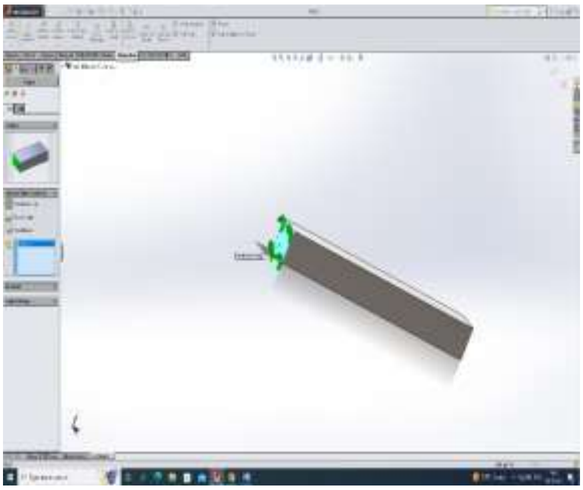


Step 8: Simulation → New Study→ Static Analysis → Ok (√).

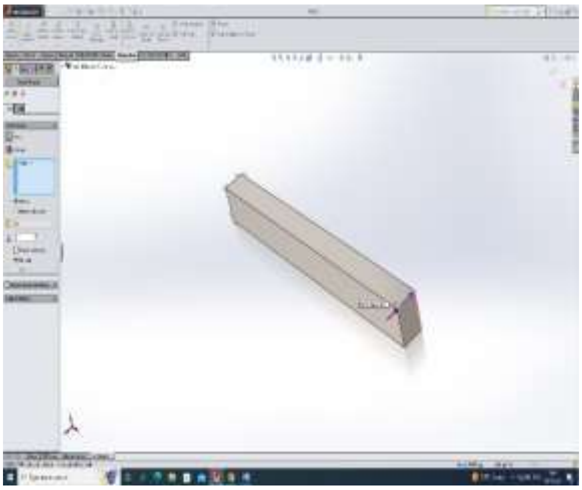


Step 9: Apply material → Chose Alloy Steel → Apply → Close.

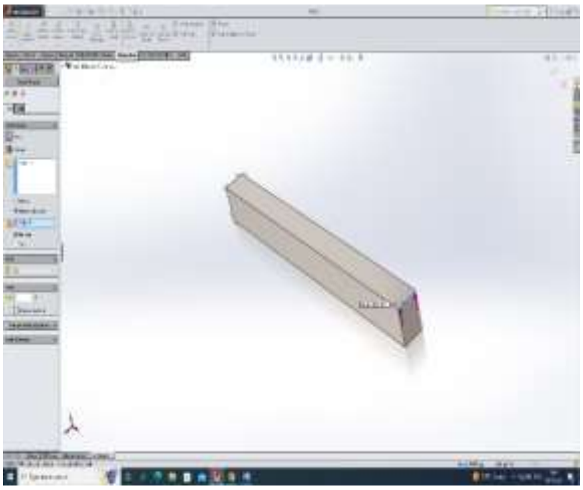
ISSN (Online): 2455-3662
EPRA International Journal of Multidisciplinary Research (IJMR) - Peer Reviewed Journal
Volume: 9| Issue: 9| September 2023|| Journal DOI: 10.36713/epra2013 || SJIF Impact Factor 2023: 8.224 || ISI Value: 1.188



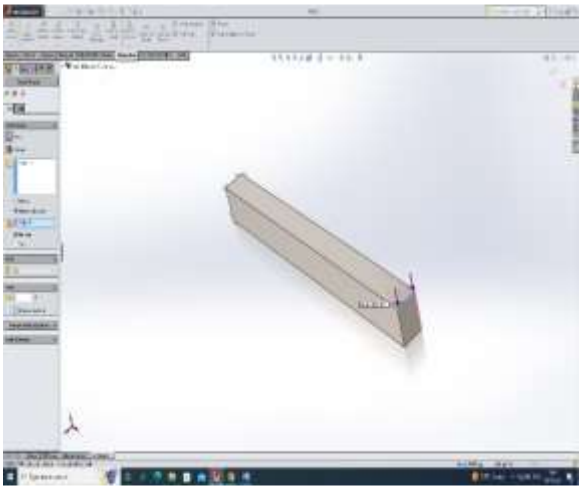
Step 10: Fixture Advisor → Fixed Geometry→ Select the end face to apply fixed geometry.



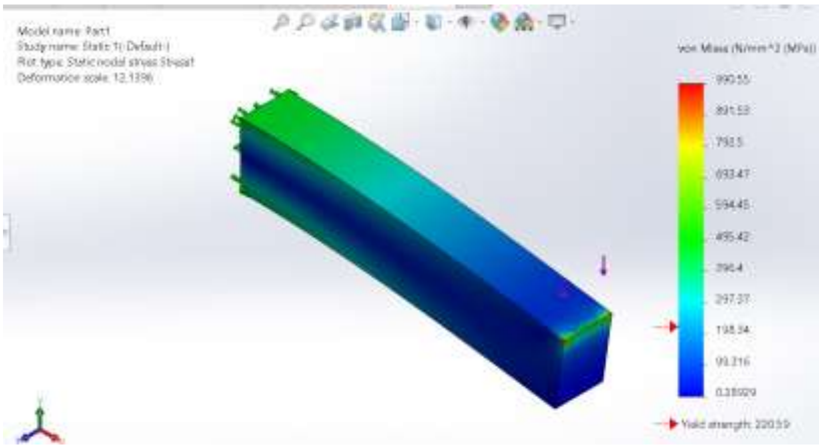
Step 11: External loads Advisor → Force→ Select the top edge → edge <1>.



Step 12: Force → selected direction → Edge <2>.



Step 13: Force → 20e3 → check the reverse direction.



Step 14: Run this study→Note the stress, strain and deformation values.

ISSN (Online): 2455-3662
EPRA International Journal of Multidisciplinary Research (IJMR) - Peer Reviewed Journal
Volume: 9| Issue: 9| September 2023|| Journal DOI: 10.36713/epra2013 || SJIF Impact Factor 2023: 8.224 || ISI Value: 1.188

4. RESULTS AND DISCUSSIONS

For the example problem solved in the methodology section, the results are verified with hand calculations as below:

Since the example 1 problem is subjected to pure bending, the bending stress equation is given by Eq. 1

$$\sigma_b = \frac{M_b}{Z} \quad (1)$$

Where σ_b= Bending stress, MPa
M_b=Bending moment, N.mm
Z=Sectional modulus, mm^3 (refer Eq. 2)
M_b=(Force × distance) = $20\times10^3 \times 400 = 8\times10^6$ N.mm

For rectangle section, $$Z = \frac{bh^2}{6} \quad (2)$$

Where b = width of the beam = 30 mm
h = height of the beam = 40 mm

$$\therefore Z = \frac{30 \times 40^2}{6} = 8000\, mm^3$$

$$\therefore \sigma_b = \frac{8 \times 10^6}{8000} = 1000 MPa$$

% variation between the simulation results and hand calculations = 0.945% (generally, the acceptable range is taken as 5% and the results obtained are in the acceptable range.

However, it is clearly evident from the simulation that beam fails. If we go back to the properties of a Plain carbon steel, its tensile strength is 339.826 MPa and yield strength is 220.594 MPa as seen from the figure 2 and as we can see in step 14, extremely high stresses of 990.55 MPa is obtained, thus indicating the **beam will break.**

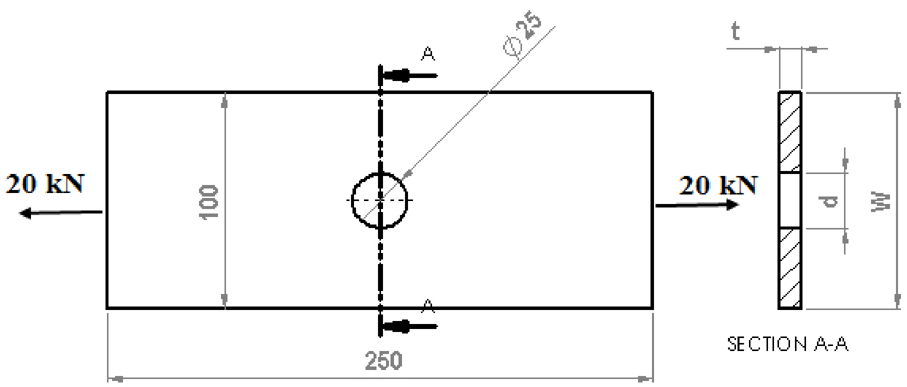
Fig. 2 Material Properties table for plain carbon steel from Solidworks

Property	Value	Units
Elastic Modulus	210000	N/mm^2
Poisson's Ratio	0.28	N/A
Shear Modulus	79000	N/mm^2
Mass Density	7800	kg/m^3
Tensile Strength	399.826	N/mm^2
Compressive Strength		N/mm^2
Yield Strength	220.594	N/mm^2

Example 2: Problem on Stress Concentration

To find the maximum stress in the steel plate shown in the Fig. 3, which is subjected to an axial load of 20 kN. The thickness and width are 10 mm and 100 mm, respectively. The plate has a central hole of 25 mm diameter.

Fig. 3 A plate with a hole



In order to solve the above problem, it is first necessary to under the meaning of "stress concentration". Stress concentration is a phenomenon that occurs when a machine element has a localized area where stress is significantly higher than the average stress in the rest of the component. This can be caused by a geometric irregularity, such as a notch, a hole, or a sharp corner, or due to the sudden change in cross-section of the component.

When a load is applied to a machine element, the stress is distributed throughout the material, and the maximum stress occurs at the point of maximum load. However, when there is a

ISSN (Online): 2455-3662
EPRA International Journal of Multidisciplinary Research (IJMR) - Peer Reviewed Journal
Volume: 9| Issue: 9| September 2023|| Journal DOI: 10.36713/epra2013 || SJIF Impact Factor 2023: 8.224 || ISI Value: 1.188

stress concentration point, the stress at that point can be much higher than the average stress in the component, and this can lead to cracks and failure over time.

Stress concentration is an important consideration in the design and analysis of machine components, as it can significantly affect their strength and durability. Designers and engineers must take into account the possibility of stress concentration in their designs and apply appropriate measures to mitigate its effects. This may involve minimizing sharp corners, using fillets or rounding edges, and avoiding abrupt changes in cross-section. Additionally, materials with high strength and ductility can help reduce the effects of stress concentration.

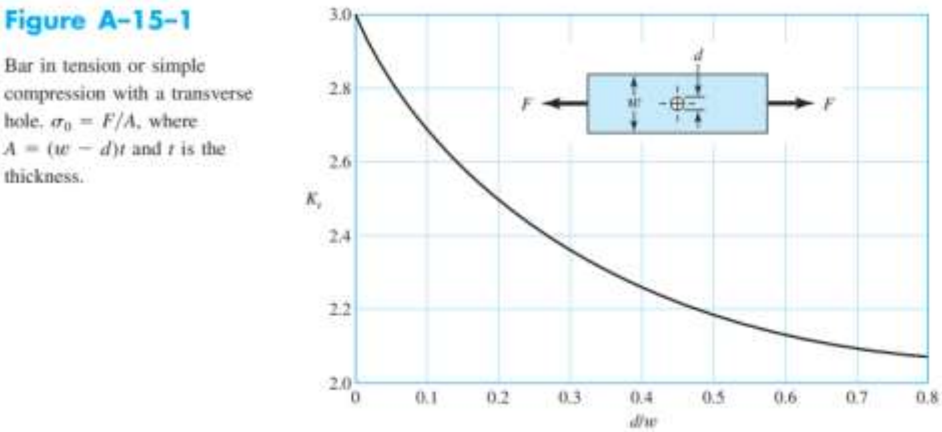
The effect of stress concentration in a machine element is described by a dimensionless factor known as stress concentration factor (k_t). It is defined as the ratio of the maximum stress at the stress concentration point to the nominal stress in the component, which is the stress that would occur in the absence of the stress concentration.

The stress concentration factor is used to calculate the actual stress at the stress concentration point, which is important in determining whether a machine component will fail due to stress. The higher the stress concentration factor, the greater the effect of stress concentration, and the more likely the component is to fail.

$$\therefore k_t = \frac{\text{Maximum stress}(\sigma_{max})}{\text{Nominal stress}(\sigma_o)} \qquad (3)$$

The value of k_t can be obtained from the stress concentration charts. For the above example, the applicable stress concentration chart is shown in the fig. 4.

Fig. 4 Stress concentration chart for a bar / plate in tension or compression with a transverse hole (Photo source: Shigley's Mechanical Engineering Design 10th Edition)



As per equation 3, to find the maximum stress the value of k_t and nominal stress should be calculated, and these can be obtained from the chart shown in Fig. 4.

Nominal stress, $\sigma_o = \frac{F}{A} = \frac{F}{(w-d)t} = \frac{20\times10^3}{(100-25)10} = 26.67\ MPa$

$$\frac{d}{w} = \frac{25}{100} = 0.25$$

$$\therefore\ K_t = 2.4.$$

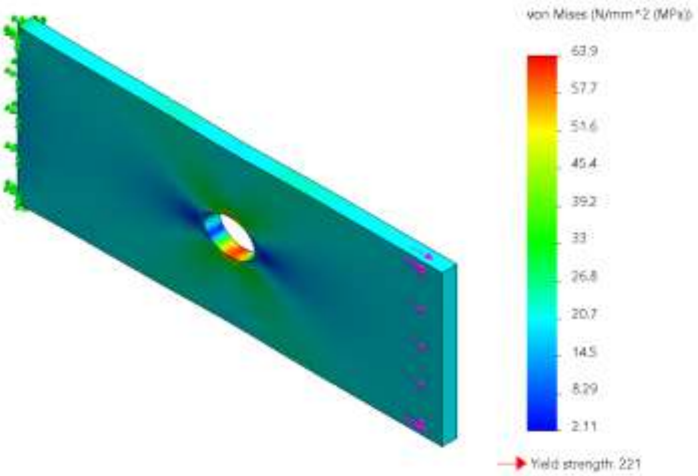
Thus, as per Eq. 3, the maximum stress is $\sigma_{max} = \sigma_{max} \times K_t = 26.67 \times 2.4 = 64\ Mpa$

Simulation Results

The maximum stress obtained from the stress analysis using solidworks is very close to the hand calculation results and infact the percentage variation observed is only 0.15% and thus the results obtained are with the acceptable limit. Also, from Fig. 5 it is learnt that the plate will not fail under the loading condition as the maximum stress (63.9 MPa) is well below the yield strength (221 MPa) of the plain carbon steel (material assumed).

ISSN (Online): 2455-3662
EPRA International Journal of Multidisciplinary Research (IJMR) - Peer Reviewed Journal
Volume: 9| Issue: 9| September 2023|| Journal DOI: 10.36713/epra2013 || SJIF Impact Factor 2023: 8.224 || ISI Value: 1.188

Fig. 5 Simulation result of a plate with a transverse hole subjected to axial loading.



5. CONLUSIONS

- SolidWorks Simulation's stress analysis is a valuable tool for predicting how components and structures will behave under different loading circumstances.
- The paper discusses two examples, one on pure bending and another on stress concentration.
  - The results of the pure bending example have been verified with hand calculations, it is found that the percentage variation between the simulation results and hand calculations is only 0.945%. However, the simulation results show that the beam will fail due to extremely high stresses.
  - In the stress concentration example, the maximum stress obtained from the simulation is very close to the hand calculation results, and the percentage variation observed in this case is 0.15% only.

Acknowledgment
This paper represents the culmination of the academic efforts undertaken during the 2022-23 academic year, specifically in Semester 1, as part of the second-year diploma project at the University of Technology and Applied Sciences – Salalah. The authors would like to extend their sincere gratitude to the university for its invaluable support, which played a pivotal role in facilitating the successful execution of this project by the students.

REFERENCES

1. *Altinbalik, M. T., & Kantur, S. (2020). Available online www.jsaer.com Research Article Comparison of DBA vs. DBF by Designing a Storage Tank for Different Head and Control Types. 7(9), 109–122.*
2. *Ardita, E. (2018). MODELING AND DESIGN OPTIMIZATION OF FOLDING TABLE FRAME By. 15.*
3. *Dorji, L., Jullamate, P., Subgranon, R., & Rosenberg, E. (2019). Predicting Factors of Successful Aging among Community Dwelling Older Adults in Thimphu, Bhutan. The Bangkok Medical Journal, 15(1), 38–43. https://doi.org/10.31524/bkkmedj.2019.02.007*
4. *Grodzki, W., & Le, K. (n.d.). Modelling of Uav ' S Composite Structures and Prediction. 67–75.*
5. *Kadhim Zarzoor, A., Almuramady, N., & Hussein, E. S. (2018). Stress analysis for spur gears using solid works simulation. International Journal of Mechanical Engineering and Technology, 9(11), 927–936.*
6. *Koirala, R., Aacharya, A., & Chitrakar, S. (2021). Structural optimization of simple span bridge by adding truss structure. 15(2), 5–10.*
7. *Landines Jiménez, K., Nieves Pimiento, N., & Toledo Bueno, C. A. (2019). Simulation of forces applied to the human femur: Analysis of finite elements. Revista Vínculos, 16(1), 73–81. https://doi.org/10.14483/2322939x.1*5575