TRƯỜNG ĐẠI HỌC
SƯ PHẠM KỸ THUẬT TP. HỒ CHÍ MINH
HCMC University of Technology and Education
HCMUTE

KHOAĐÀO TẠO CHẤT LƯỢNG CAO

BỘMÔNỨNG DỤNG CAE TRONG CƠ KHÍ

TIỂU LUẬN

PHÂN TÍCH MÔ PHỎNG CHI TIẾT SPOON

Thựchiện: Nhóm 9

- **Đặng Ngọc Dũng**
- **ĐỗNguyênNhân**
- **PhạmNhậtThành**

Giảngviênhướngdẫn: Th.STrần Mai Văn

-TP. HồChí Minh, tháng 6/2020

Contents

Chapter 1:

Introduction

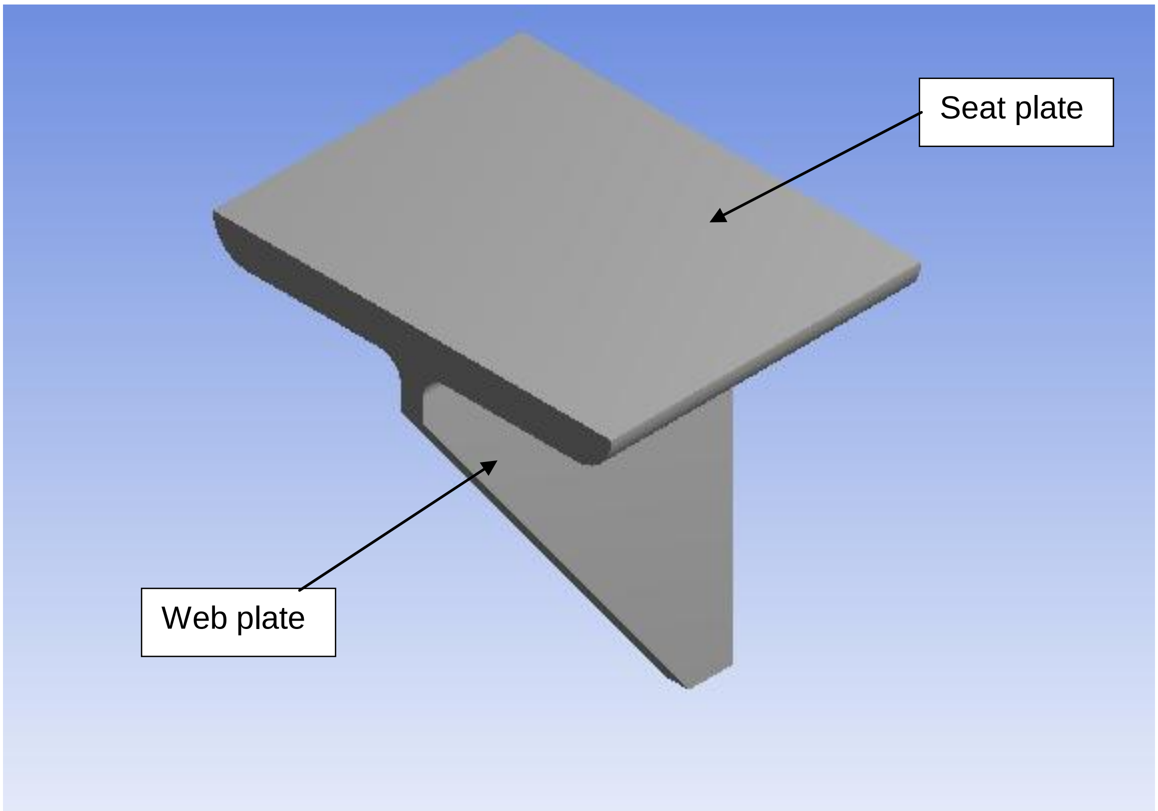
Nhom 9

1.1 Whereis the Beam Bracketused:

In construction steel beam-column structures, such as high-rise building or manufacturing plants, column are erected before beams can be elevated, positioned and welded. The functions of beam brackets are to precisely position the beam during the construction, and safety transfer the loads from the beam to the column. The loads are determined by a thorough analysis of the entire struture subject to design loads, such as dead load, live load, earthquake, wind load,…

1.2 Construction of the Beam Bracket:

The beam bracket consists of a seat plate (the flange) and a web plate.



1.3About ANSYS 18.2:

Many industries face a fundamental technological change. Digitization, which has already changed many business models in the consumer field, now also penetrates industrial products and processes. Networking through the Internet of Things brings new comrades on the scene and intensifies the competitive pressure. At the same time, networked products create new opportunities to get to know customers, their application scenarios and the actual requirements more effectively, thus tailoring products and services to the real needs. The simulation plays a central role - be it in design, design and verification, where it already opens up potential for optimization during the development or whether it is using an individual product.

These many new scenarios mean that the use of simulation technology is increasingly supported by users who are not simulation experts, but rather use the advantages of physical simulation as design engineers, development engineers or service engineers. On the other hand, there is a growing need to describe the ever-increasing complexity of products in an adequate manner so that experts can implement the methods that ensure the required reality of the simulation. Thus, simulation takes place everywhere and through: Pervasive Engineering Simulation.

This development is favored by a continuous simulation solution, which combines all the components - physical and logical - but also all work steps in an efficient work process. ANSYS, Inc. is responsible for the development of ANSYS R18.

1.4Solved problem in this research:

The problem in this research is about the beam bracket.

Beam Brackets are used in a lot of large industry such as civilization, machinary, airspace and even in military, …

They have been solved by mechanic theory such as machine elements and mechanics of materials until now. But the precise results have not been examined. To solve this case, numerical methods by using ANSYS is applied to analyze the stress and strain of this component.

I choose this component to analyse some capacities: Total Deformation, Equivalent Stress, Structure Error, Safety Factor, … From that I will give some recommendations for the usage of this component.

The design considerations include:

- Hypothesis 1: Would the maximum stress excess the allowable stress?
- Hypothesis 2: Would the web buckle under the load?

Chapter 2:

Input- Output

This chapter is about theory of deformation of the bracket and numerical parameters of simulation using ANSYS 18.2. It includes Input and Output of analysis progress and some evaluations, results of this analysis.

Nhom 9

2.1 Input

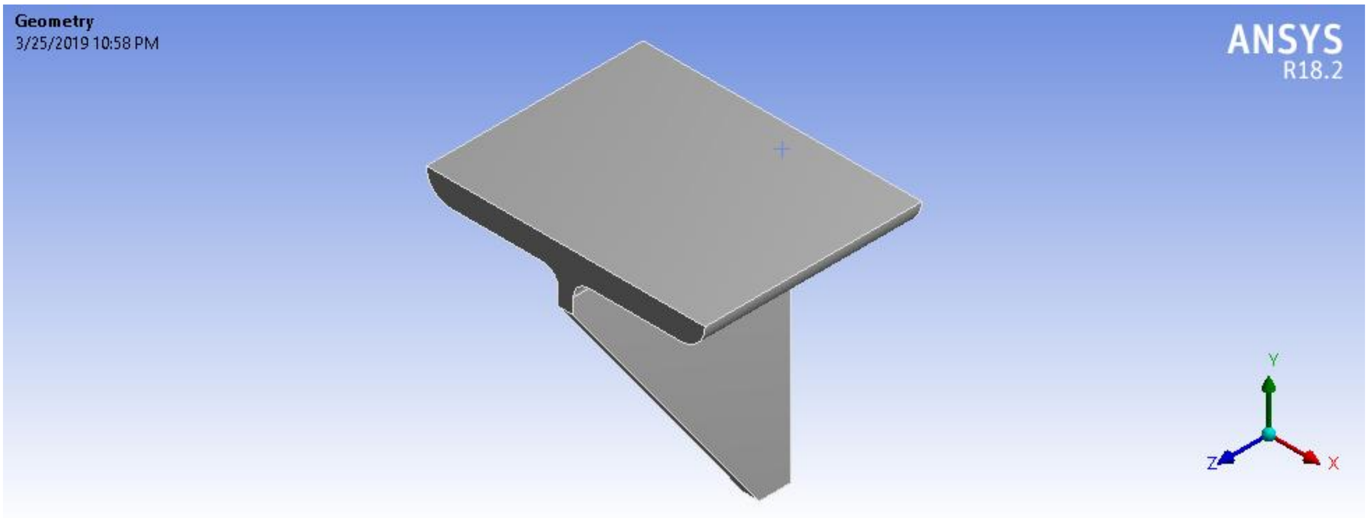
Input in analysis this component includes: analysis type, geometry, boundary condition and numerical parameters.

2.1.1 Analysis type:

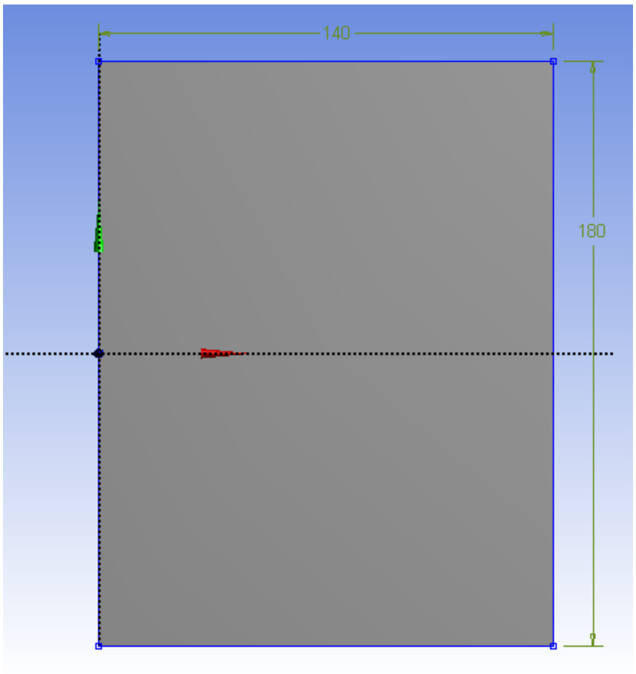
- Analysis system: Static Structural

- Analysis type: 3D

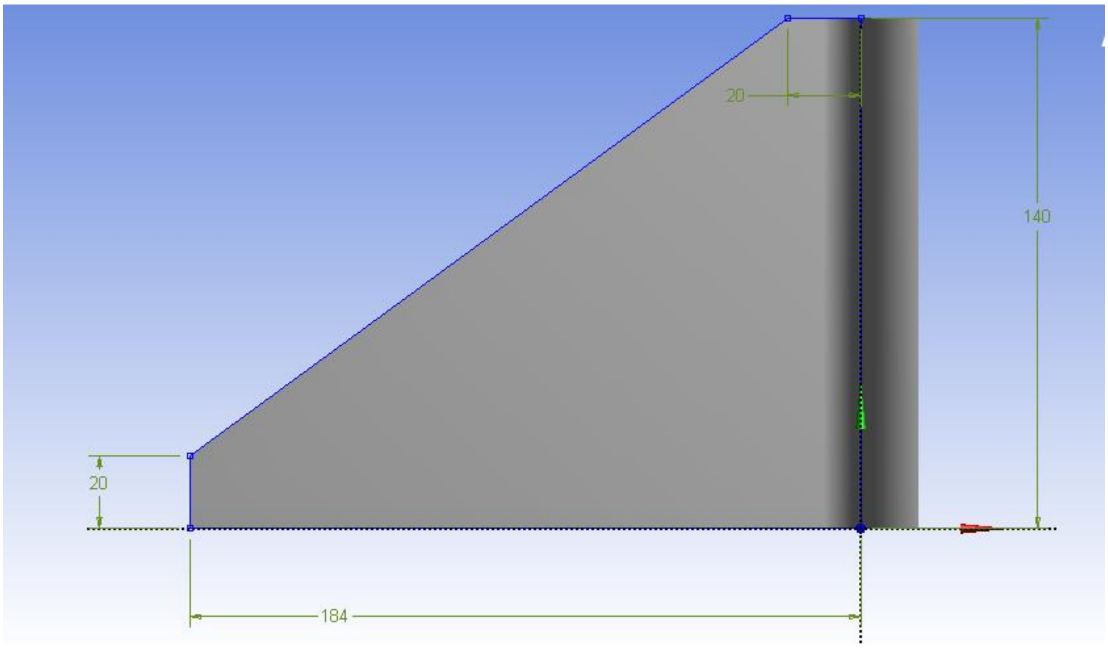
2.1.2 Geometry:



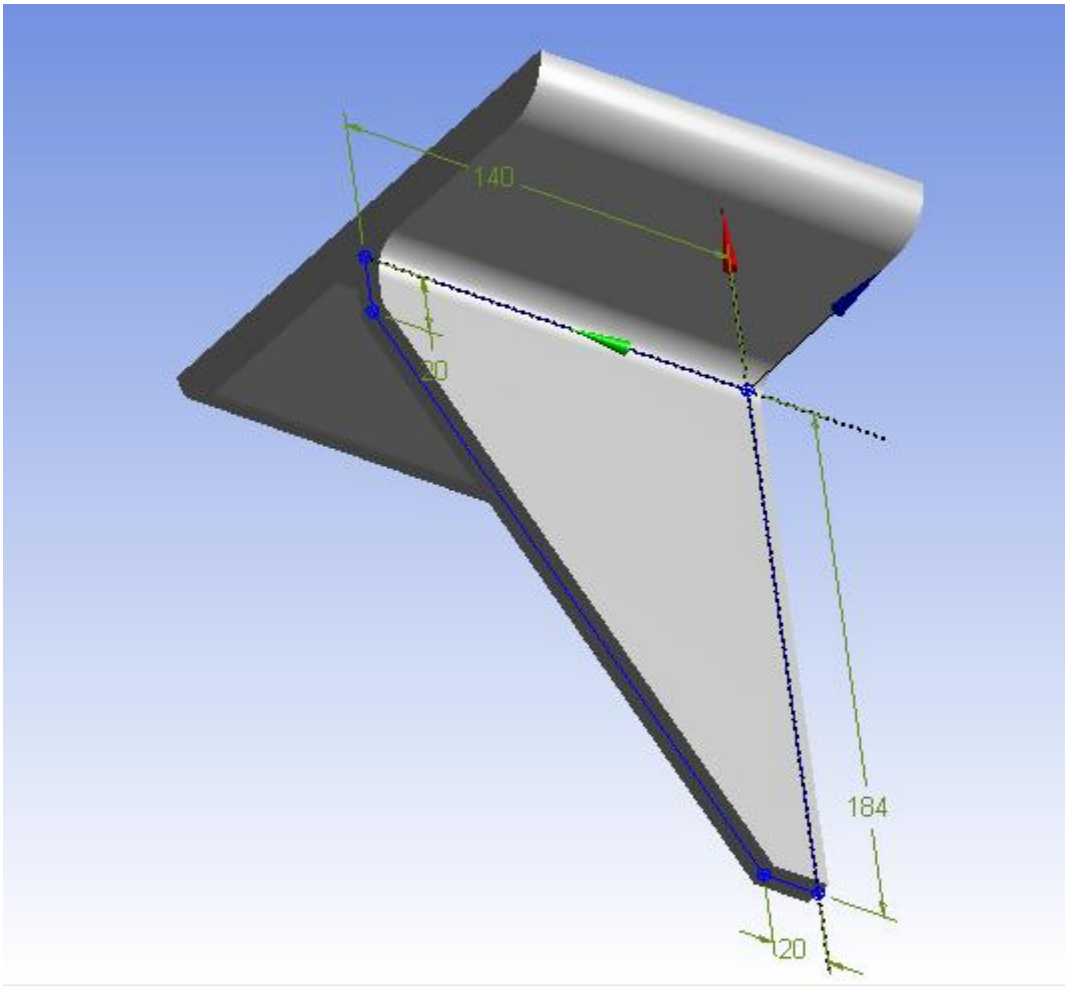
- Unit: mm
- Seat plate dimension:



- Web plate dimension:



- Create Fillets R10:



2.1.3 Material:

Nhom 9

Structural Steel

Density: 7850 kg/m^3

Coefficent of Thermal Expansion:1.2E-05 C^{-1}

Young's Modulus: 2E+11 Pa

Poisson's Ratio: 0.3

Bulk Modulus: 1.6667E+11 Pa

Shear Modulus: 7.6923E+10 Pa

Tensile Yield Strength: 2.5E+08 Pa

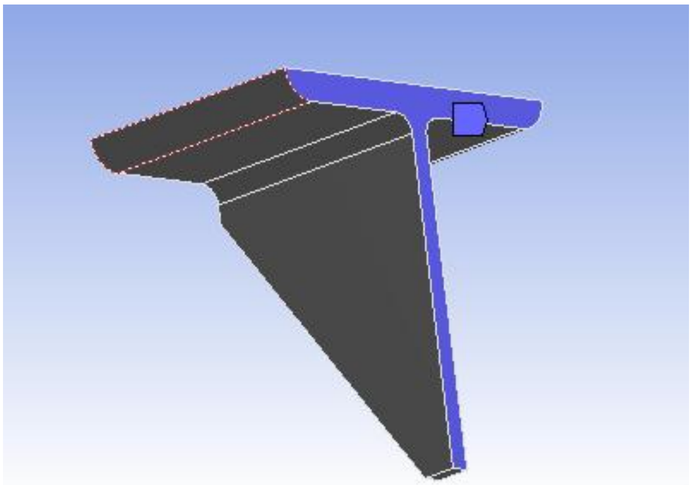
Compressive Yield Strength: 2.5E+08 Pa

Tensile Ultimate Strength: 4.6E+08 Pa

2.1.4 Boundary Condition:

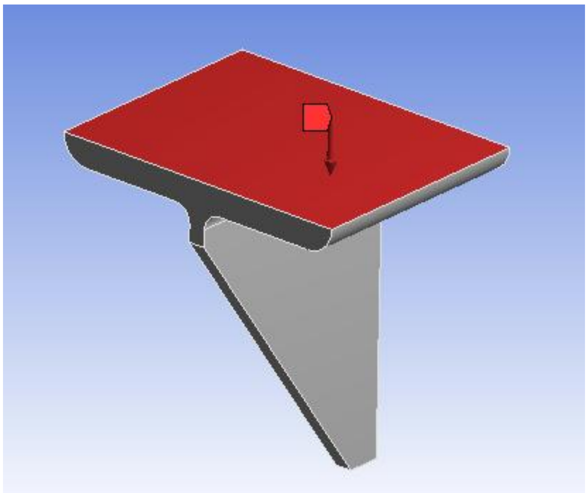
- Fixed support:

Scope:
Scoping Method: Geometry Selection
Geometry: 1 Face
Definition:
Type: Fixed Support
Suppressed: No



- Force:

Scope:
Scoping Method: Geometry Selection
Geometry: 1 Face
Definition:
Type: Force
Defined by: component
Global coordinate system
Y component: -27000N
Suppressed: No

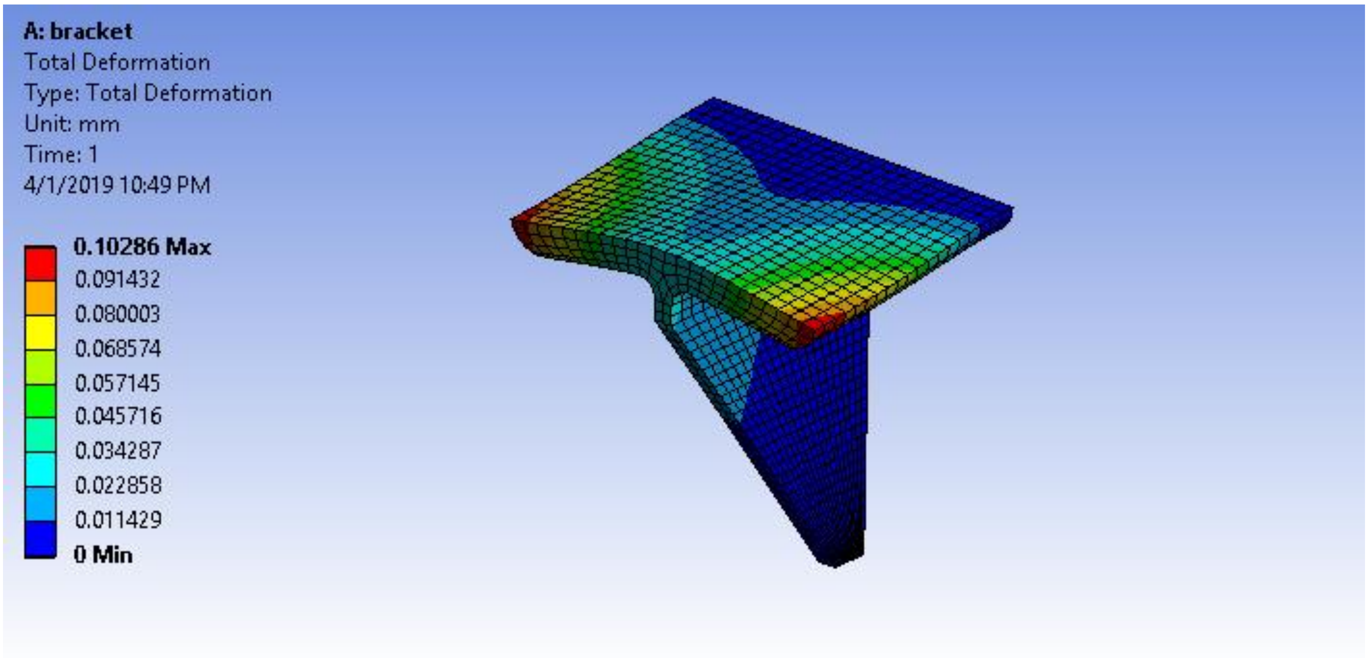


2.2 Output

Output in this chapter consists of stress distribution, force convergence and deformation of the beam bracket.

2.2.1 Total Deformation

They are used to obtain displacements from stresses. It gives a square root of the summation of the square of x-direction, y-direction and z-direction.

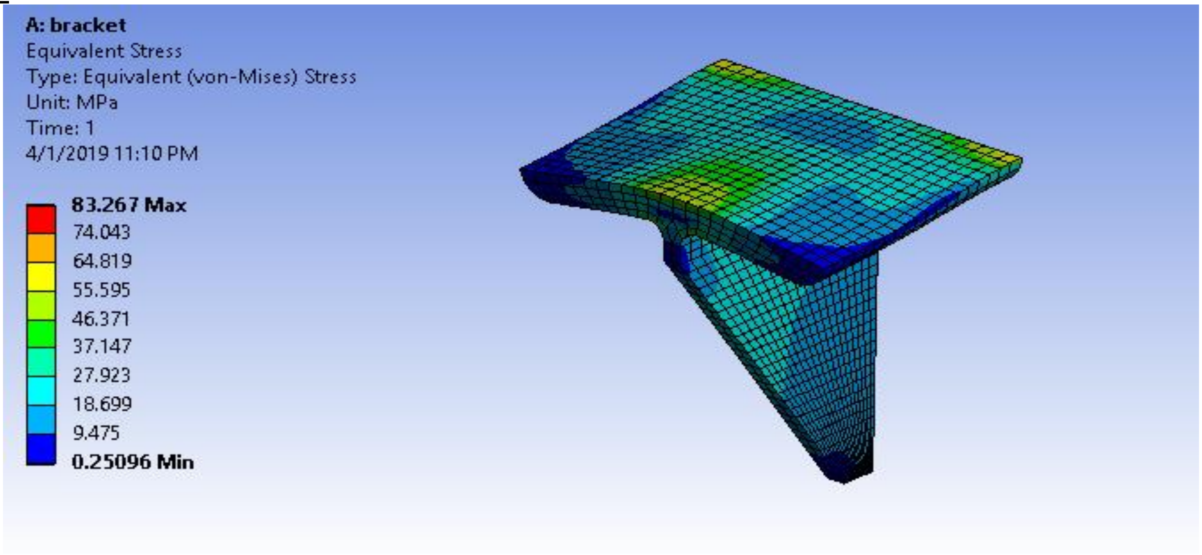


Scope:
Scoping Method: Geometry Selection
Geometry: All bodies
Definition:
Type: Total deformation
Calculate history time: Yes
Suppressed: No

Result:
Minimum: 0mm
Maximum: 0.10286mm
Minimum occurs on: Solid
Maximum occurs on: Solid
Information:
Time, load step, substep, iterison number: 1

2.2.2 Equivalent Stress

Equivalent stress (also called von Mises stress) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximumequivalent stress failure theory used to predict yielding in a ductile material.

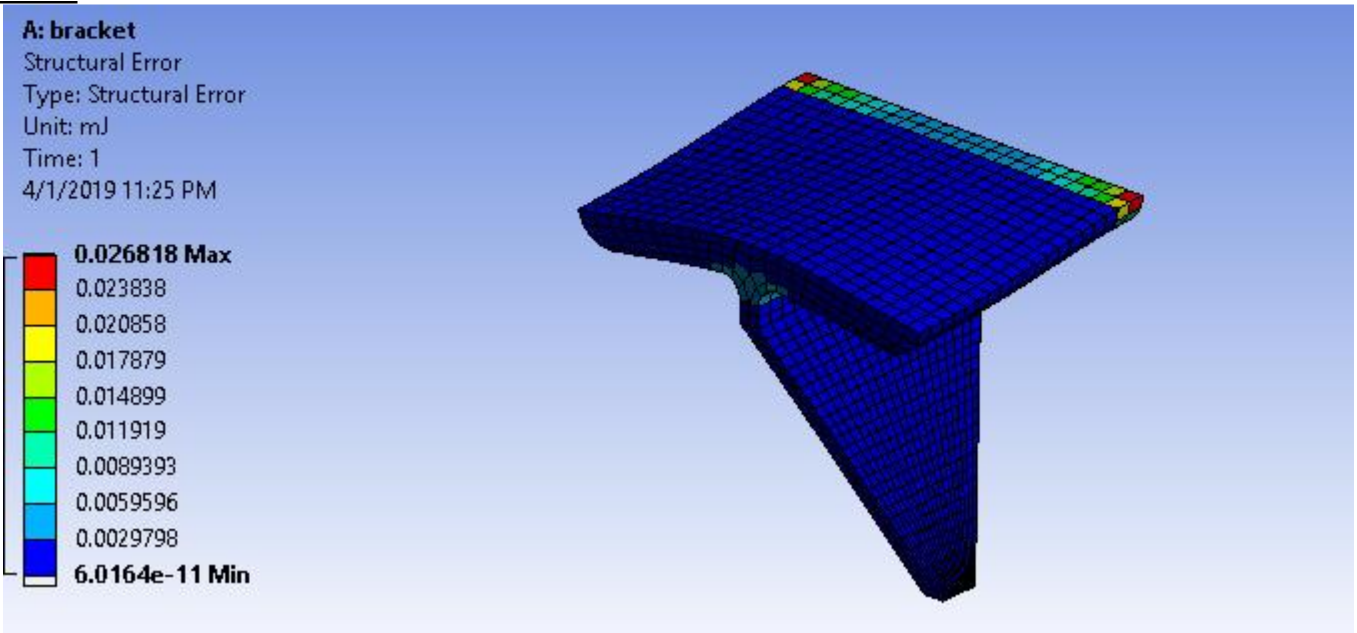


Scope:
Scoping Method: Geometry Selection
Geometry: All bodies
Definition:
Type: Equivalent stress
Calculate history time: Yes
Suppressed: No
Intergration point result:
Display option: Averaged
Averaged across body: No

Result:
Minimum: 0.25096MPa
Maximum: 83.267MPa
Minimum occurs on: Solid
Maximum occurs on: Solid
Information:
Time, load step, substep, iterison number: 1

2.2.3 Structural errors

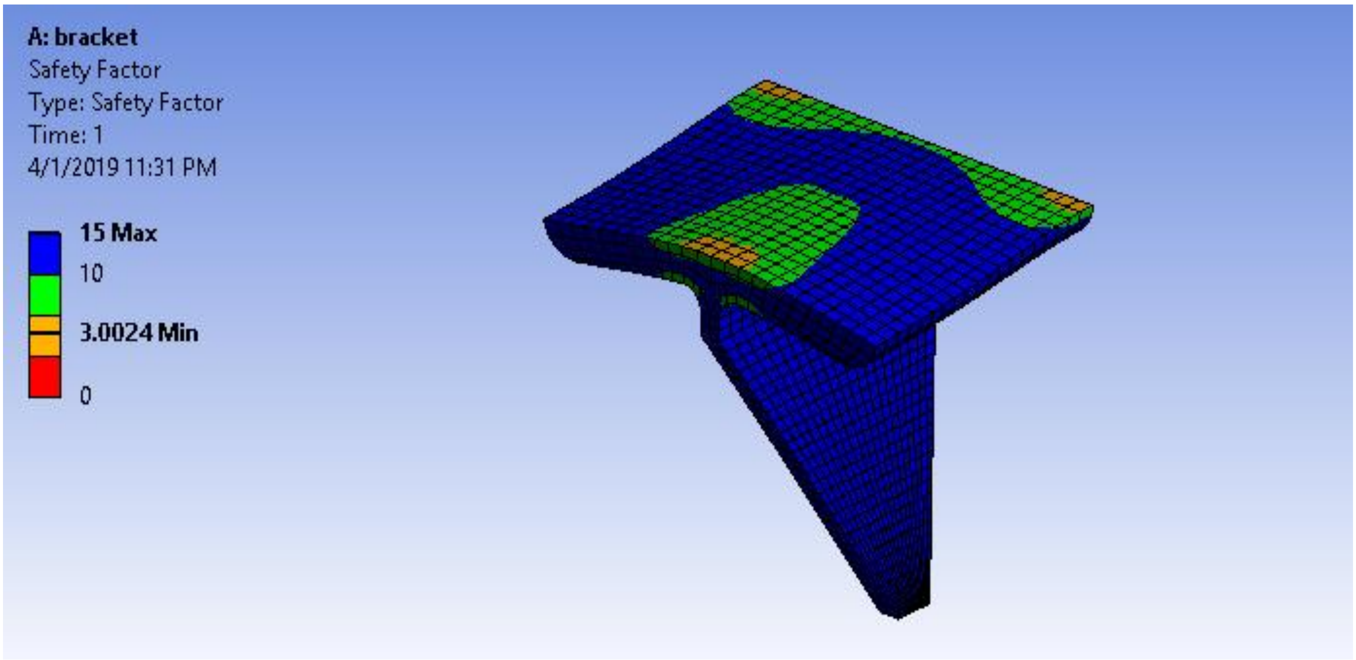
You can insert an Error result based on stresses to help you identify regions of high error and thus show where the model would benefit from a more refined mesh in order to get a more accurate answer. You can also use the Error result to help determine where Mechanical will be refining elements if Convergence is active.



Scope:
Scoping Method: Geometry Selection
Geometry: All bodies
Definition:
Type: Structural error
Calculate history time: Yes
Suppressed: No

Result:
Minimum: 6.0164e-011mJ
Maximum: 2.6818e-002mJ
Minimum occurs on: Solid
Maximum occurs on: Solid
Information:
Time, load step, substep, iterison number: 1

2.2.4 Safety factors



Scope:

Result:

Nhom 9

Scoping Method: Geometry Selection
Geometry: All bodies
Definition:
Type: Safety Factor
Calculate history time: Yes
Suppressed: No
Intergration point result:
Display option: Averaged
Averaged across body: No

Minimum: 3.0024
Minimum occurs on: Solid
Information:
Time, load step, substep, iterison number: 1

Chapter 3:

Analysis

This chapter analyzes and evaluates reacts of the beam bracket in various aspects. By calculating the beam bracket in various meshing, different force andfrictioness, we can suggest the function and construction and strength of material in the beam bracket.

Nhom 9

3.1 Table of comparision

Case	Mesh	Force	Safety Factor	Structural Error	Equivalent Stress	Total Deformation
1	2	27000	2.2575	0.00097897	110,74	0.10295
2	3		2.6012	0.0024937	100.2	0.10294
3	4		2.6702	0.0049925	93.624	0.10293
4	7		2.9773	0.024344	83.969	0.10287
5	9		3.0808	0.074339	81.147	0.10279
6	10		3.145	0.087649	79.493	0.10285
7	15		3.2684	0.42038	76.491	0.10253
8	7	24000	3.3494	0.019234	74.639	0.09144
6		25000	3.2155	0.020871	96.319	0.095254
7		26000	3.0918	0.020871	77.749	0.099064
8		27000	2.9773	0.024344	83.969	0.10293
9		28000	2.8709	0.02618	87.079	0.10678
10		29000	2.7719	0.028084	90.819	0.11049
11		30000	2.6795	0.030054	93.299	0.11431

3.2 Mesh value change

3.2.1 Mesh value choosen:

Mesh size is one of the most common problems in Ansys. There are: bigger elements give bad results, but smaller elements make computing so long you don't get the results at all. You hardly really know where exactly is your mesh size on this scale.

To solve mesh problem in this analysis, many element size calculations are carried out by concerning some tips below:

- Perform chosen analysis for several different mesh sizes.
- Notice where high deformations or high stresses occur, perhaps it is worth to refine mesh in those regions.
- Collect data from analysis of each mesh: outcome, number of nodes in the model, computing time.

Some values have been chosen around the "default" value, because the beam bracket is quite small, so the changing value to be compared will be changed within a small number.

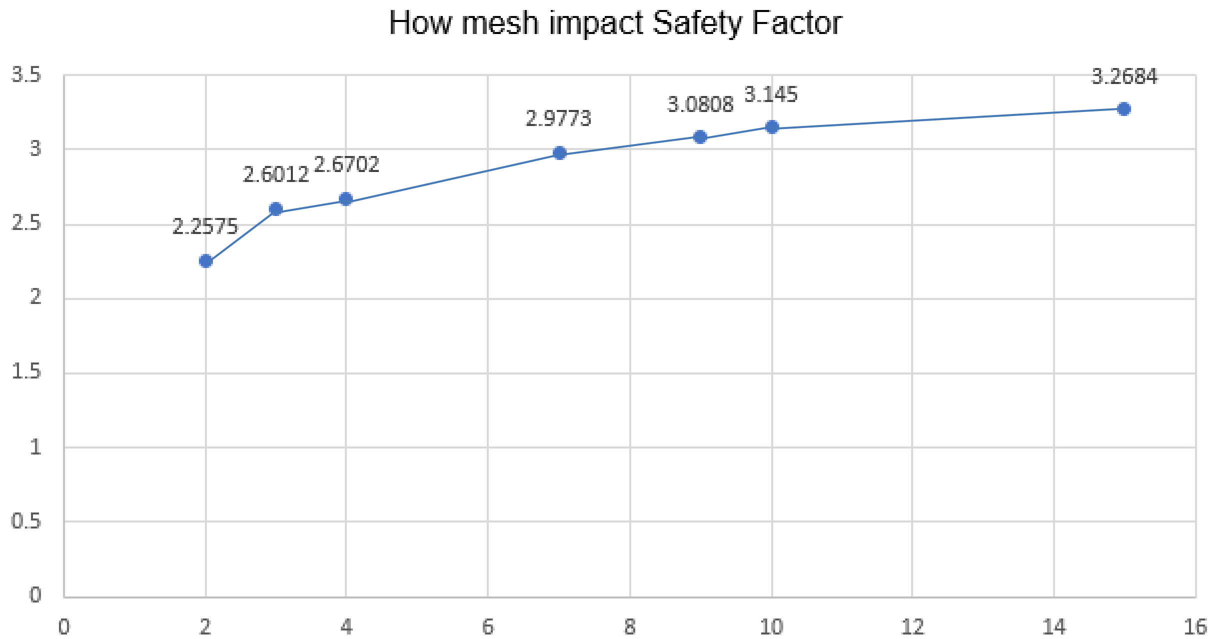
The maximum element size is chosen is 15, because a coarse mesh will require less computational resources to solve and, while it may give a very inaccurate solution, it can still be used as a rough verification and as a check on the applied loads and constraints.

The minimum element size is chosen is 2, because higher the number of elements (small sized), higher will be the time taken to solve that problem and, amount of system space required for solving. Which will of course will lead to greater extent of accuracy. But during meshing process, there is no benefit in trying to mesh a model to greater accuracy than the input data admits.

3.2.2Analysis through chart

Nhom 9
After Meshing,the entire structure is divided into number of elements and each element having its own stiffness while loading.

Safety Factor

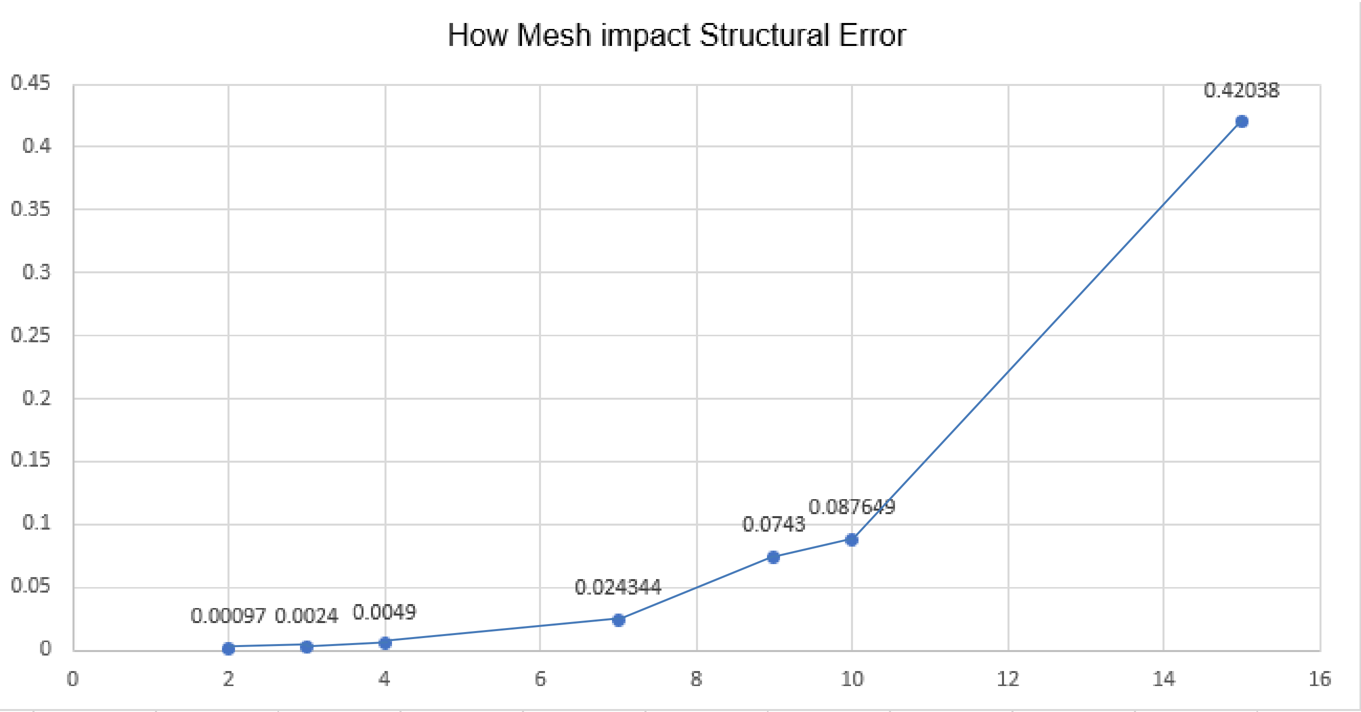


The line graph shows how mesh impact the safety factor,it can be seen that the value increases significantly in 15 meshing value from 2.2575 to 3.2684.

It means that the larger the element size meshing is, the more safety factor needs. Because the accuracy value decreases, more safety factor needed.

Nhom 9

Structural Error

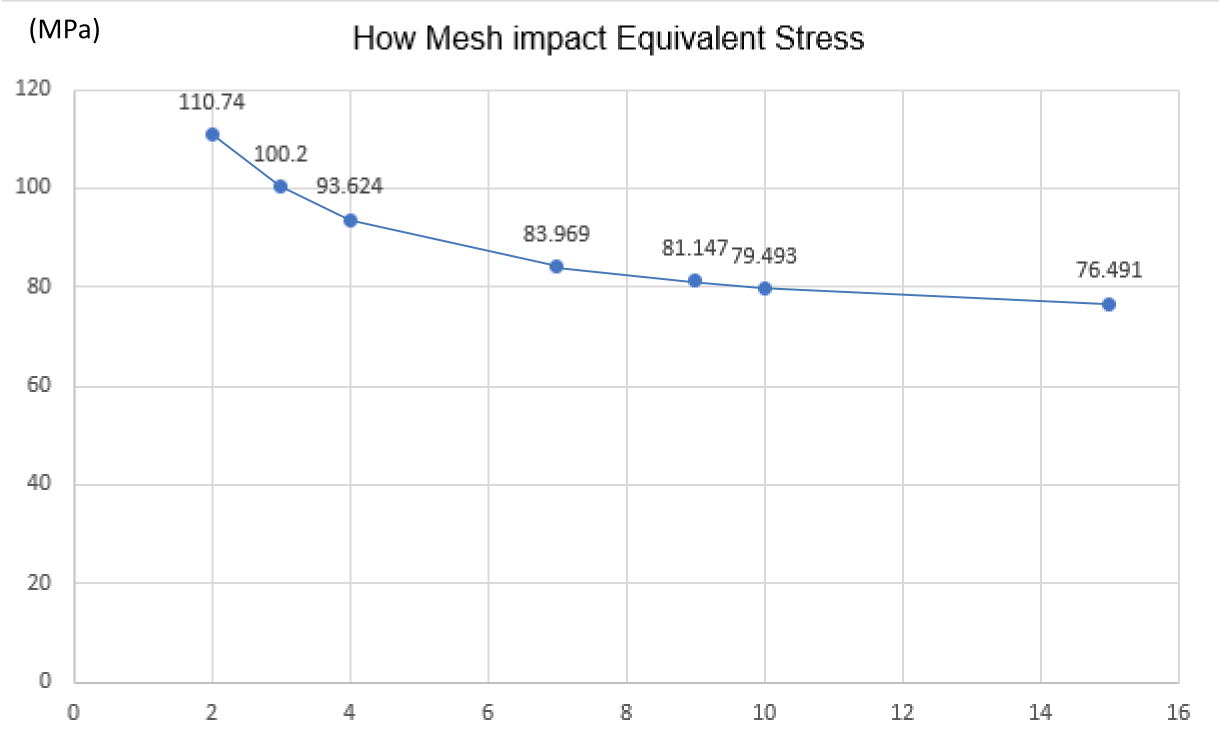


In the line chart about how mesh impact structural error, there is a significant increase in mesh value from 2 to 10. But from value 10 on, there is a dramatically increase.

In small mesh value, the accuracy is ensured. Therefore, it will lead to less structural error. But when the mesh value is out of control, it will lead to the large structural error

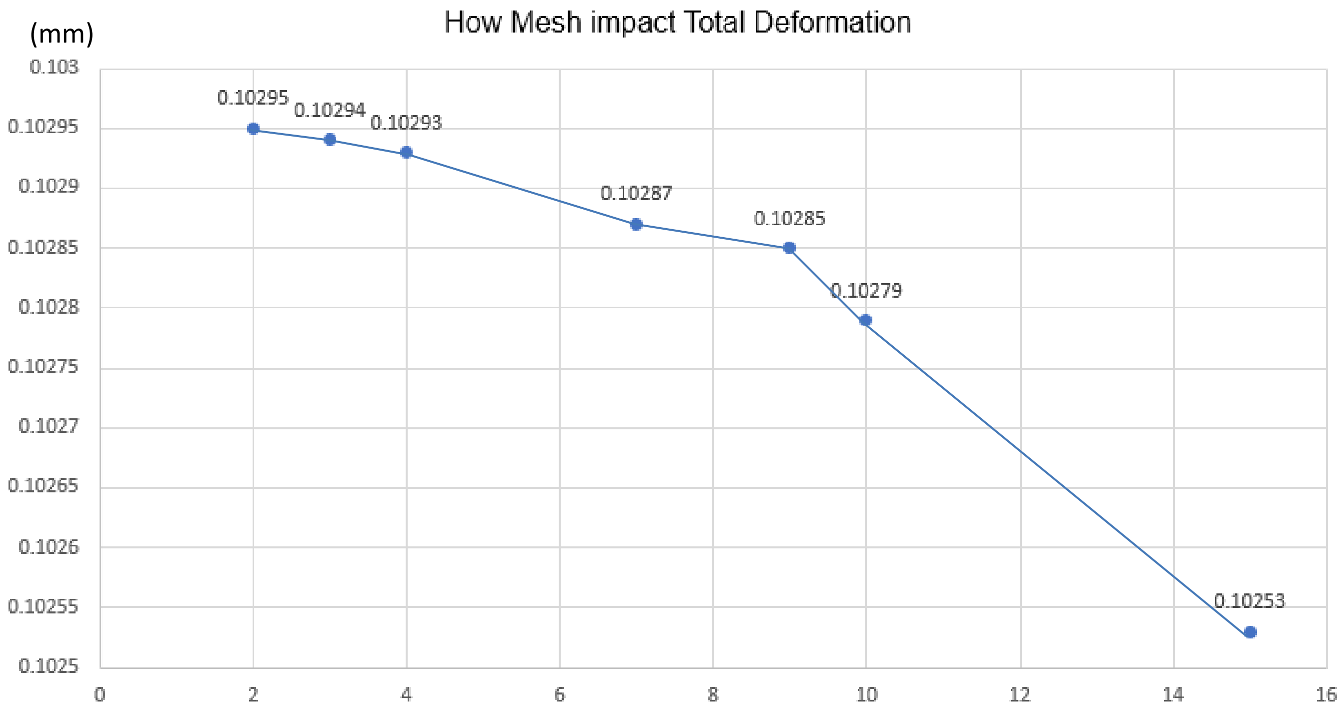
Nhom 9

Equivalent Stress



Different from 2 line graphs above, in “how mesh impact equivalent stress”, the values of equivalent stress go down from 110.74 to 76.491 MPa.

Total Deformation



In this graph, there is a significantly fall in the Total deformation when the mesh value decreases, the number of total deformations falls from 0.10295mm to 0.10253mm.

The reason is that when the mesh size is large, the accuracy will be small, so there is a small number 0.10253 in the mesh value 15, the largest value.

3.3 Force change

3.3.1 Force value chosen

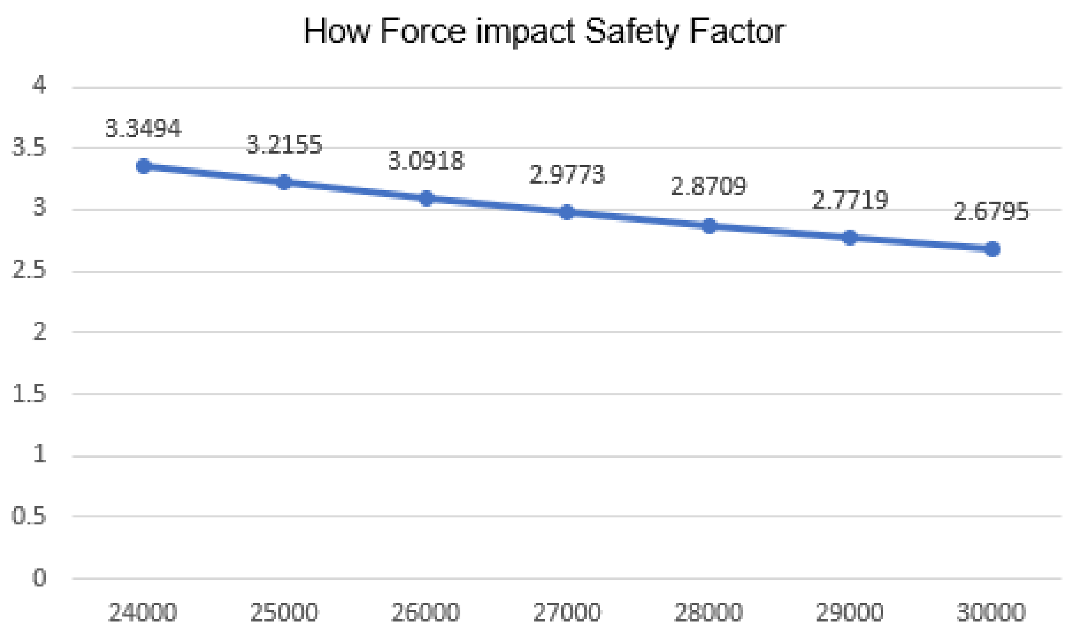
The beam brackets are used mostly in civil engineering industry. They have to be stand will a large force in the seat plate and the force will be spreaded to the net plate (the rib).

They always hold up pillars or bars in house or building construction, not only pillars or bars but even a net of pillars and bars.

The mass value of pillar and bar nets are very various, they often range from hundreds Newton to thousands Newton. To ensure the reliability of the beam bracket, the force of thousand Newton has been chosen to put in the beam bracket. More specifically, the value will be ranged from 24000N to 30000N because the mass of the pillar and bar net will vary from 2000kg to 2500kg, so the force impacts in the beam bracket will vary from 24000N to 30000N.

3.3.2 Analysis throung chart

Safety Factor

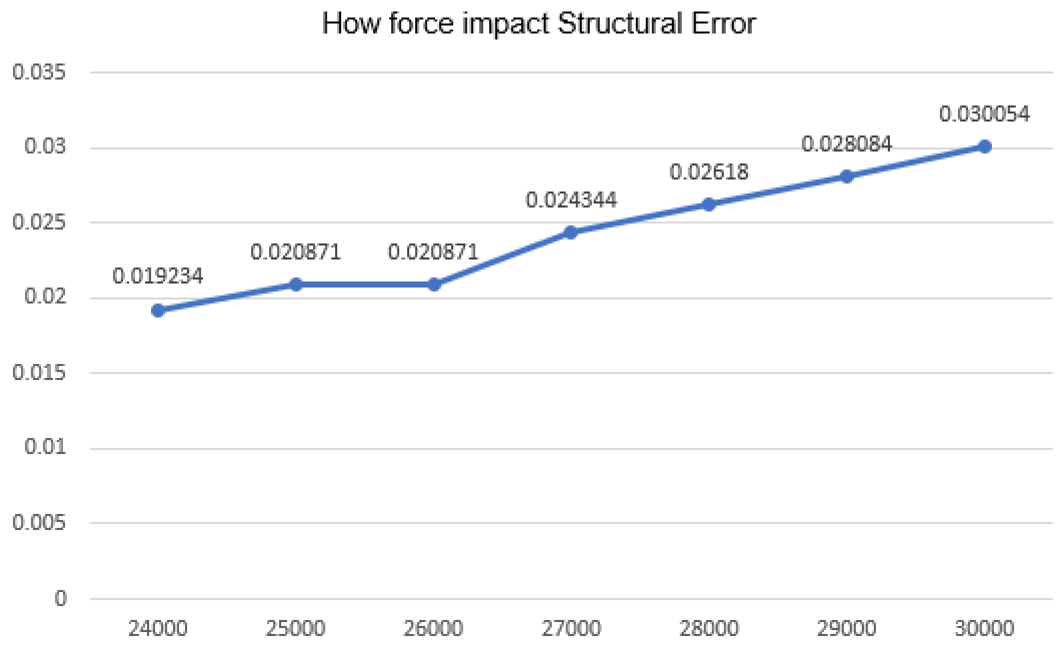


The line graph shows how force impact Safety Factor. The number decrease steadily in the force range from 24000N to 30000N, safety facetor goes down frm 3.3494 to 2.6795, decrease nearly 1 unit.

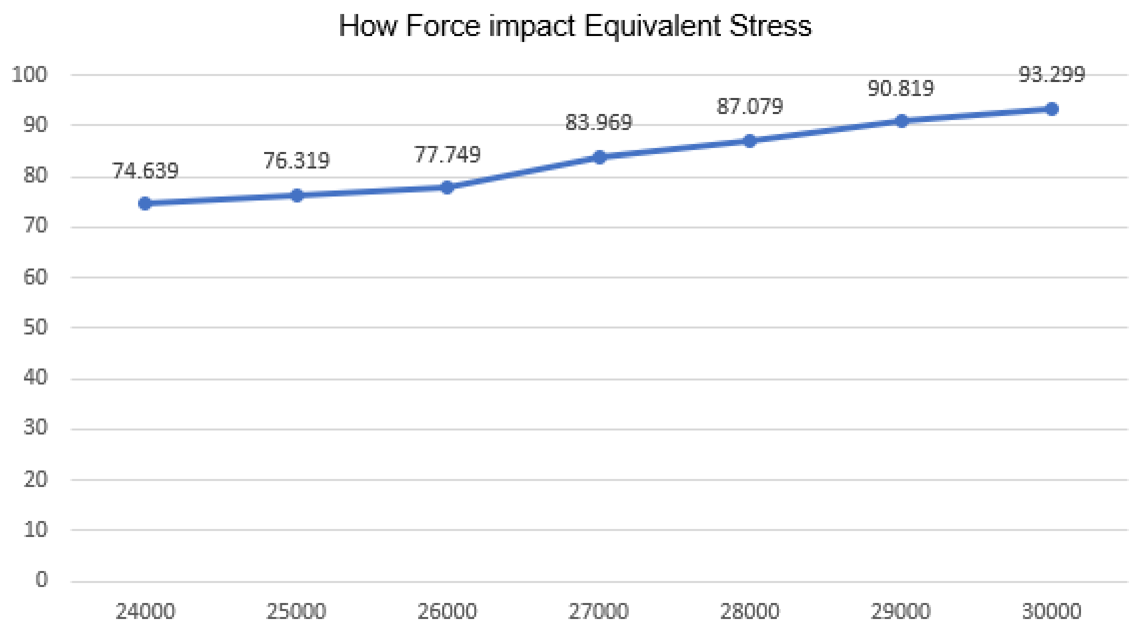
Nhom 9

When the larger force is put on the beam bracket, the less safety factor value becomes because the dangerous limit is nearer, and it’s very easy to reach the maximum stress value.

Structural Error



Equivalent Stress



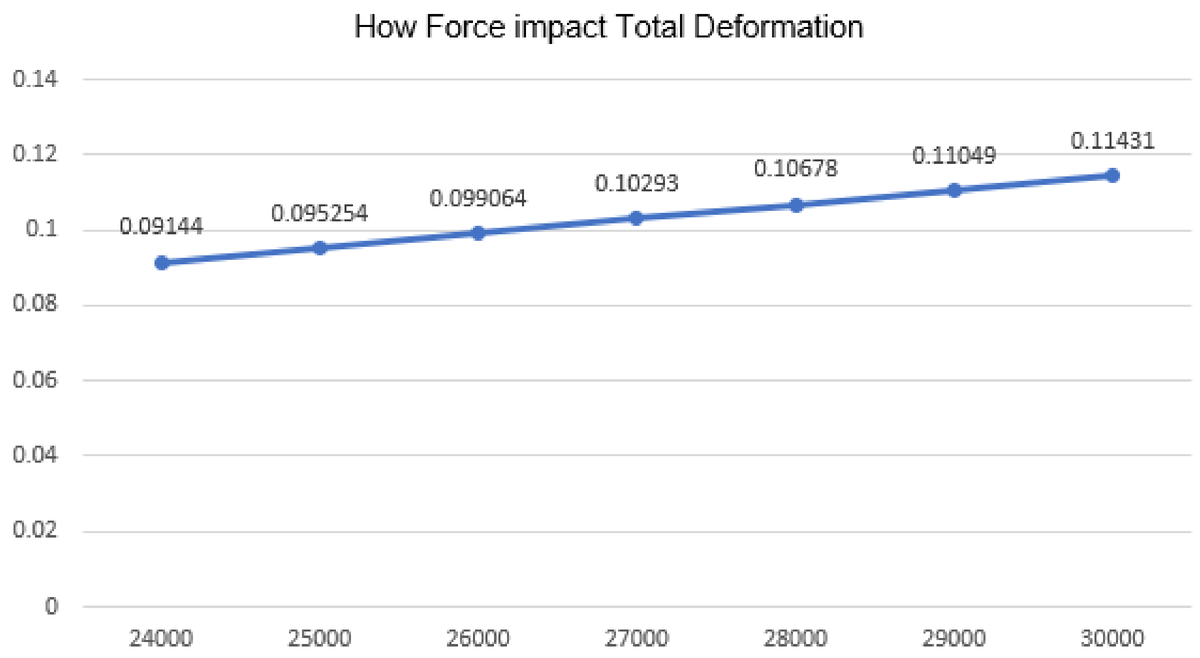
There are a rise number in 2 graphs about how force impacts structural error and equivalent stress.

The structural error increases steadily from 0.019234 to 0.030054 when the force goes up from 24000N to 30000N. And the equivalent stress also climbs from 74.639Mpa to 93.299MPa.

Nhom 9

The reason is that when the force increases, the sustainabilities of the material become weaker and weaker, it leads to the deformation and other factors. Hence, the structural error and equivalent stress increases.

Total Deformation



In the graph about how force impact total deformation, it can be seen that there is a steadily increase in the number. It varies from 0.09144mm to 0.11431mm

To explain for this graph, we should consider about Hook's Law.

"The extension of a material or a spring is its increase in length when pulled. Hooke's Law says that the extension of an elastic object is directly proportional to the force applied to it. In other words:

- If the force applied is doubled, the extension doubles.
- If no force is applied, there is no extension."

Therefore, the graph specifically increases by the extension of material when a force is applied on the beam bracket.

Chapter 4:

Conclusion

Conclusion sums up what have been analysis above so as to give recommendations about how to use the beam bracket components by providing some informations about the strain, flexibility, safety factor and its deformation.

To solve problem in the beam brackets, many researchs and calculations have been carried out, but only by mathematics and other means in paper and calculator.

The born of ANSYS helps these analysises become more easier by caculating in computer and we can carry out many analysises with many aspects to evaluate the components.

In this report, many analysises have been gone through and we have come out with some conclusions below:

- It's very important to consider about the force applied on the beam bracket. Because when the force is applied, it will lead to change many things such as: structural error, equivalent stress, total deformation and safety factor. If everything isn't simulated and calculated carefully, it will lead to the damage of many constructions, buildings or even manufacturing systems.
- Some problems about the beam bracket have been solved in this report. Through analysis:
  - We know how to choose the element size in mesh and decide which one is the best for every purpose
  - How mesh and force impact structural error, safety factor, equivalent stress and total deformation
  - Which value of force can be applied to the beam bracket, and with every force, the beam bracket will change in which way.

But there's still some problems aren't solved in this analysis, I hope in the future, there will be another breakthrough analysis about the beam bracket to contribute more to the industry.

References

1. https://www.ansys.com
2. https://opentextbc.ca/physicstestbook2/chapter/elasticity-stress-and-strain/
3. http://www.bautsystem.com/en/baut-fixing-system/brackets/
4. https://www.quora.com
5. https://enterfea.com