HO CHI MINH CITY UNIVERSITY OF TECHNOLOGY AND EDUCATION

FACULTY OF MECHANICAL ENGINEERING

# APPLICATION OF CAE IN DESIGN



Lecturers: Tran Mai Van

Group: 9

Members: Dang Ngoc Dung

Do Nguyen Nhan

Pham Nhat Thanh


# MID-TERM REPORT

## SPOON SIMULATION WITH ANSYS 2020 R1

*Ho Chi Minh City, 9th July 2020*

# Contents

# Chapter 1:

# Introduction

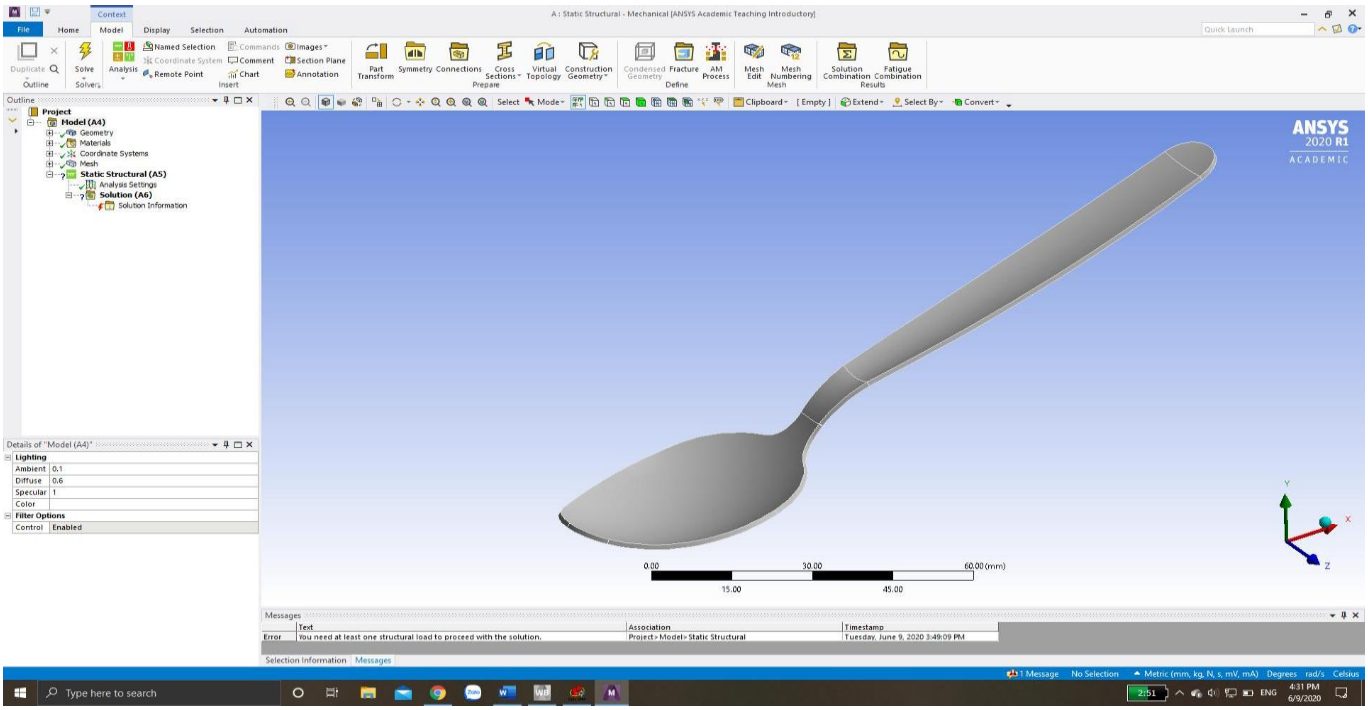
### 1.1 Whereis the Spoonused:

Spoons are used primarily for eating liquid or semi-liquid foods, such as soup, stew or ice cream, and very small or powdery solid items which cannot be easily lifted with a fork, such as rice, sugar, cereals and green peas. In Southeast Asia, spoons are the primary utensil used for eating; forks are used to push foods such as rice onto the spoon as well as their western usage for piercing the food.

Spoons are also widely used in cooking and serving. In baking, batter is usually thin enough to pour or drop from a spoon; a mixture of such consistency is sometimes called "drop batter". Rolled dough dropped from a spoon to a cookie sheet can be made into rock cakes and other cookies, while johnnycake may be prepared by dropping spoonfuls of cornmeal onto a hot greased griddle.

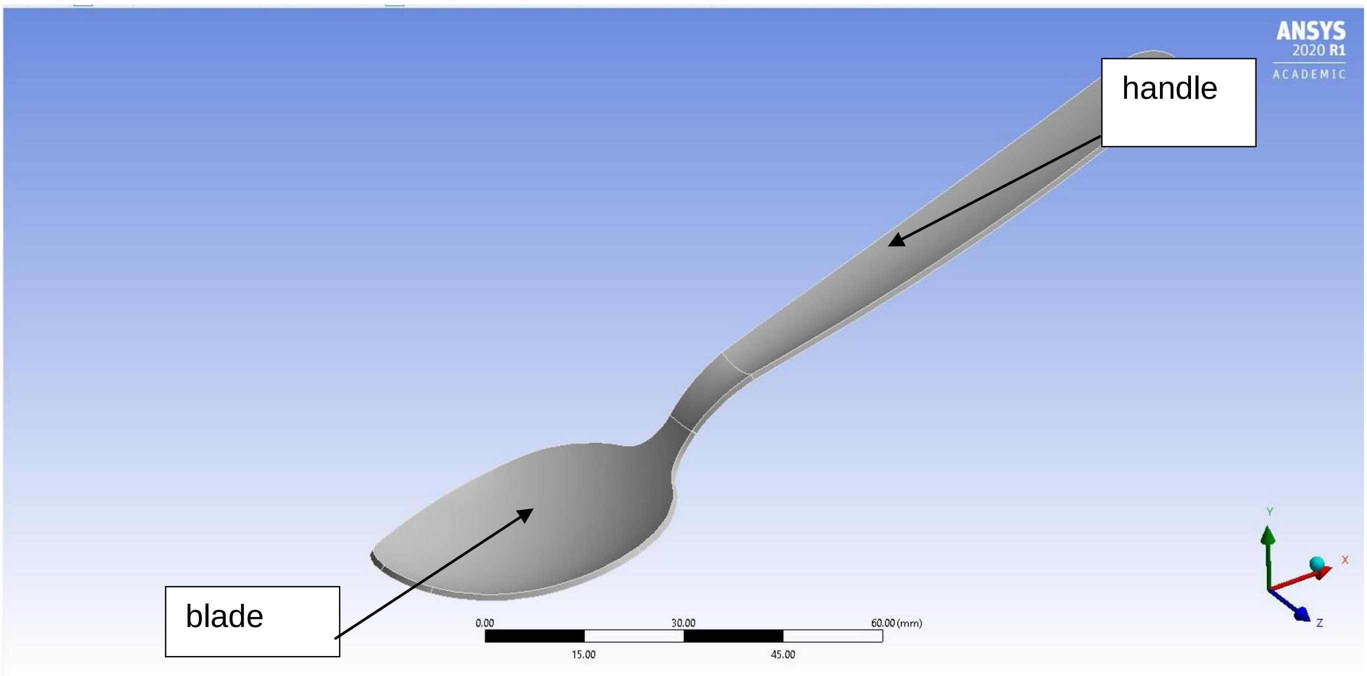
A spoon is similarly useful in processing jelly, sugar and syrup. A test sample of jelly taken from a boiling mass may be allowed to slip from a spoon in a sheet, in a step called "sheeting". At the "crack" stage, syrup from boiling sugar may be dripped from a spoon, causing it to break with a snap when chilled. When boiled to 240 °F. and poured from a spoon, sugar forms a filament, or "thread". Hot syrup is said to "pearl" when it forms such a long thread without breaking when dropped from a spoon.

Used for stirring, a spoon is passed through a substance with a continued circular movement for the purpose of mixing, blending, dissolving, cooling, or preventing sticking of the ingredients. Mixed drinks may be "muddled" by working a spoon to crush and mix ingredients such as mint and sugar on the bottom of a glass or mixer. Spoons are employed for mixing certain kinds of powder into water to make a sweet or nutritious drink. A spoon may also be employed to toss ingredients by mixing them lightly until they are well coated with a dressing.



### 1.2 Construction of the Spoon:

Spoon is a device consisting of two parts: the handle and the blade, which may be round or oval in shape, firmly attached to a handle.




### 1.3About ANSYS 2020:

Facing increasing demands to streamline product development life cycles and boost product performance, engineering teams in all industries require the most advanced simulation technology. For optimal performance, simulation solvers must operate seamlessly across all physics and among all members of an engineering team.

Ansys 2020 R1 continues to integrate simulation throughout product life cycles, from ideation to virtual testing to operation, with Ansys Minerva. This cutting-edge platform spurs collaboration within global engineering teams and increases data sharing to innovate product designs and slash development costs.

Minerva enables unprecedented access to the broadest selection of the most accurate, physics-based simulation software in the world. Ansys 2020 R1 introduces many trailblazing enhancements to these technologies, driving unrivaled engineering productivity.

Ansys 2020 R1 includes enhancements to Ansys Mechanical to help engineers design complex, highly nonlinear and extremely large models. This release also features a greatly simplified workflow in Ansys Fluent, so even novice engineers can execute complex multiphase simulations quickly and easily. Other portfolio upgrades include dynamic new tools for Ansys HFSS SBR+ and Ansys Maxwell that substantially improve processes for electronic/electromagnetic design.

### 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.

### 2.1 Input

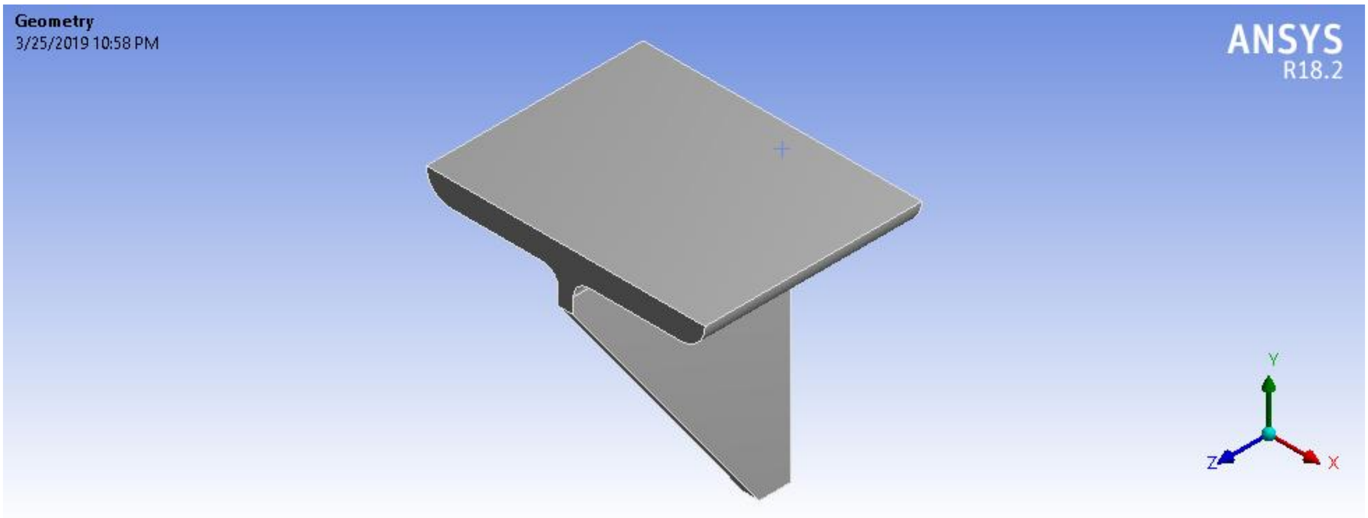
*Input in analysis this component includes: analysis type, geometry, boundary condition and numerical parameters.*

<u>2.1.1 Analysis type:</u>

- Analysis system: Static Structural

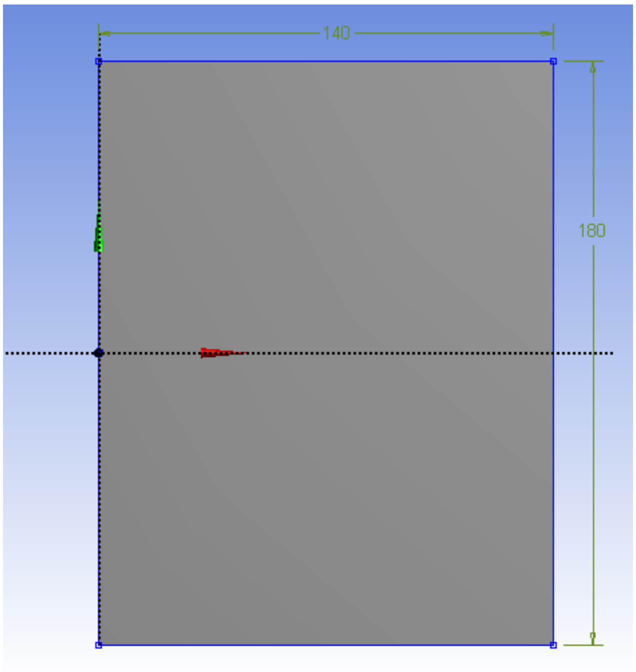
- Analysis type: 3D

<u>2.1.2 Geometry:</u>

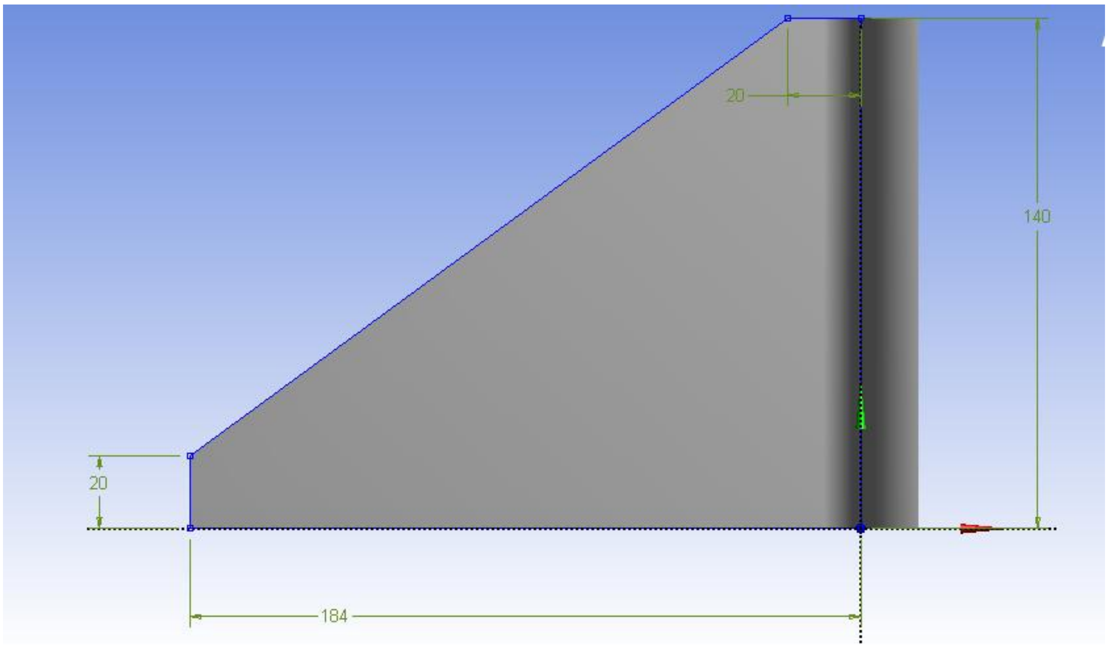



- Unit: mm

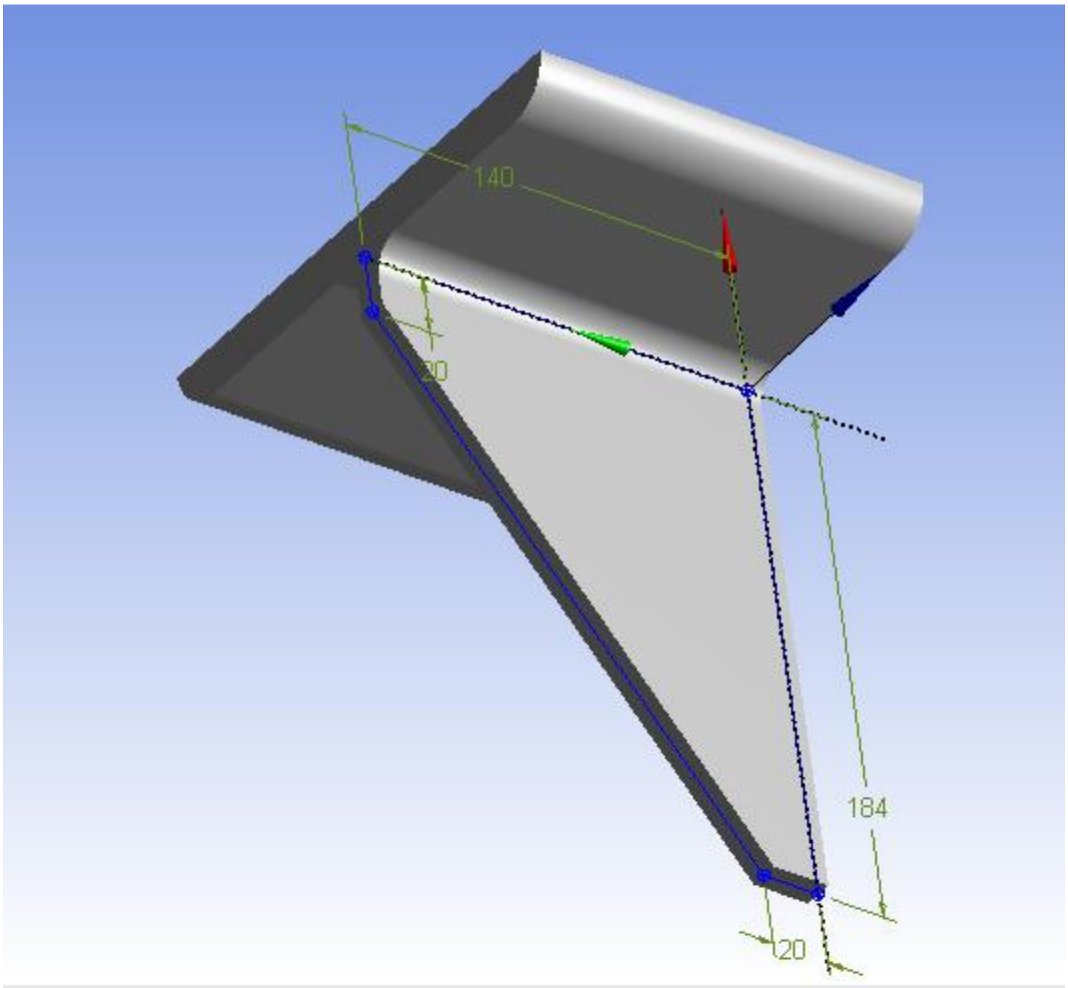
- Seat plate dimension:




- Web plate dimension:




- Create Fillets R10:




## 2.1.3 Material:

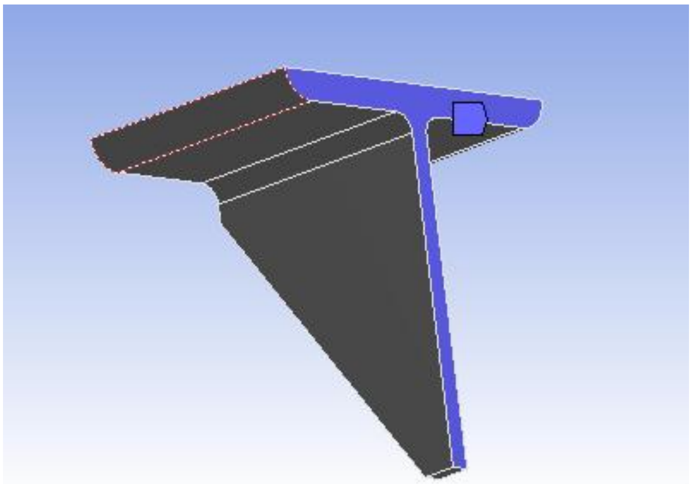
**Structural Steel**

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:

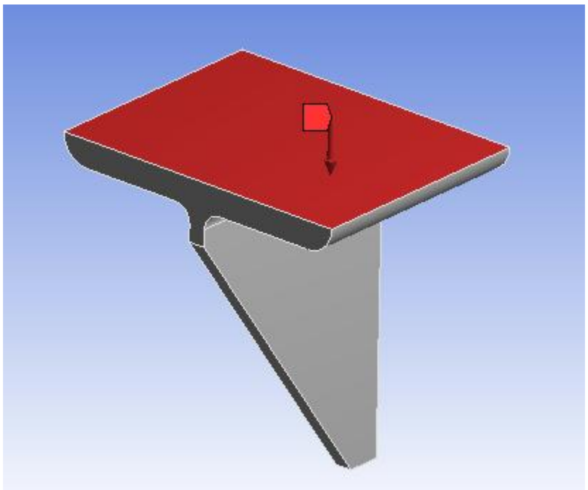
- Fixed support:

  **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

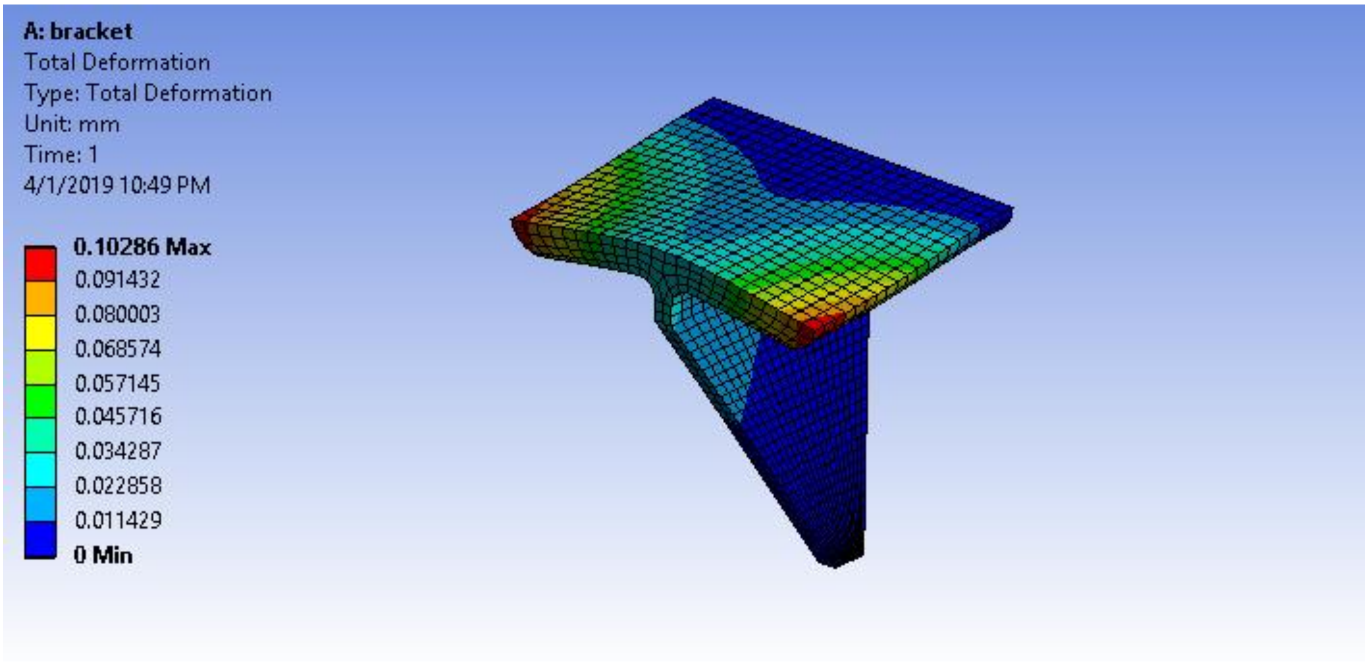


## 2.2 Output

*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.

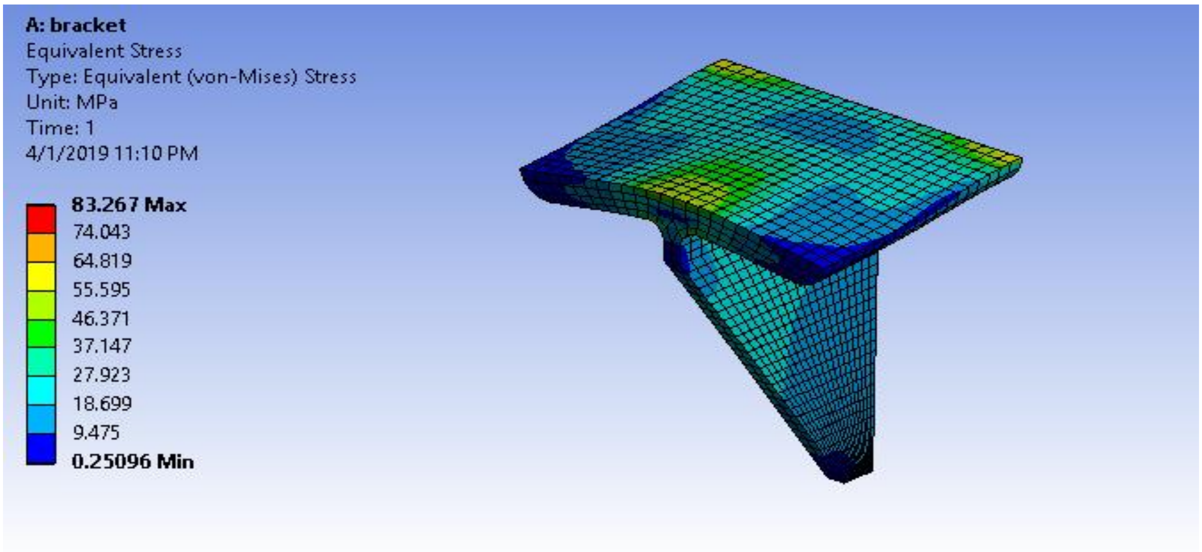



**Scope:**
Scoping Method: Geometry Selection
Geometry: All bodies
**Definition:**
Type: Total deformation
Calculate history time: Yes
Suppressed: No

**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.

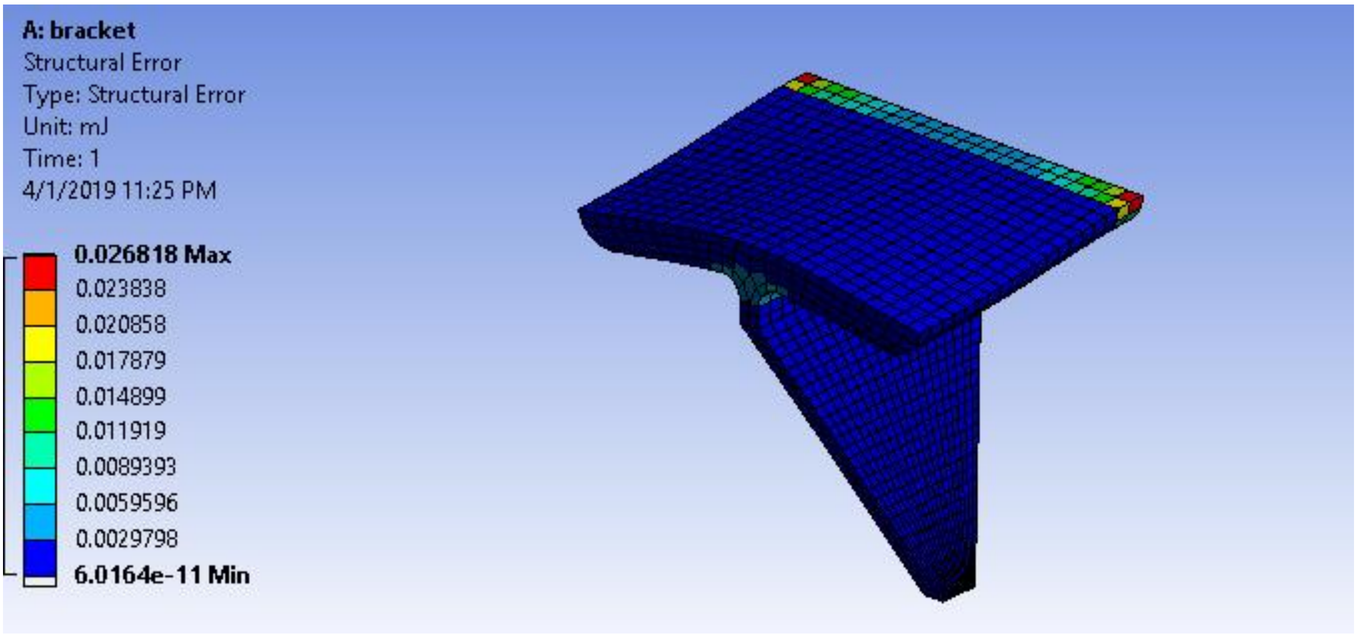


**Scope:**
Scoping Method: Geometry Selection
Geometry: All bodies
**Definition:**
Type: Equivalent stress
Calculate history time: Yes
Suppressed: No
**Intergration point result:**
Display option: Averaged
Averaged across body: No

**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

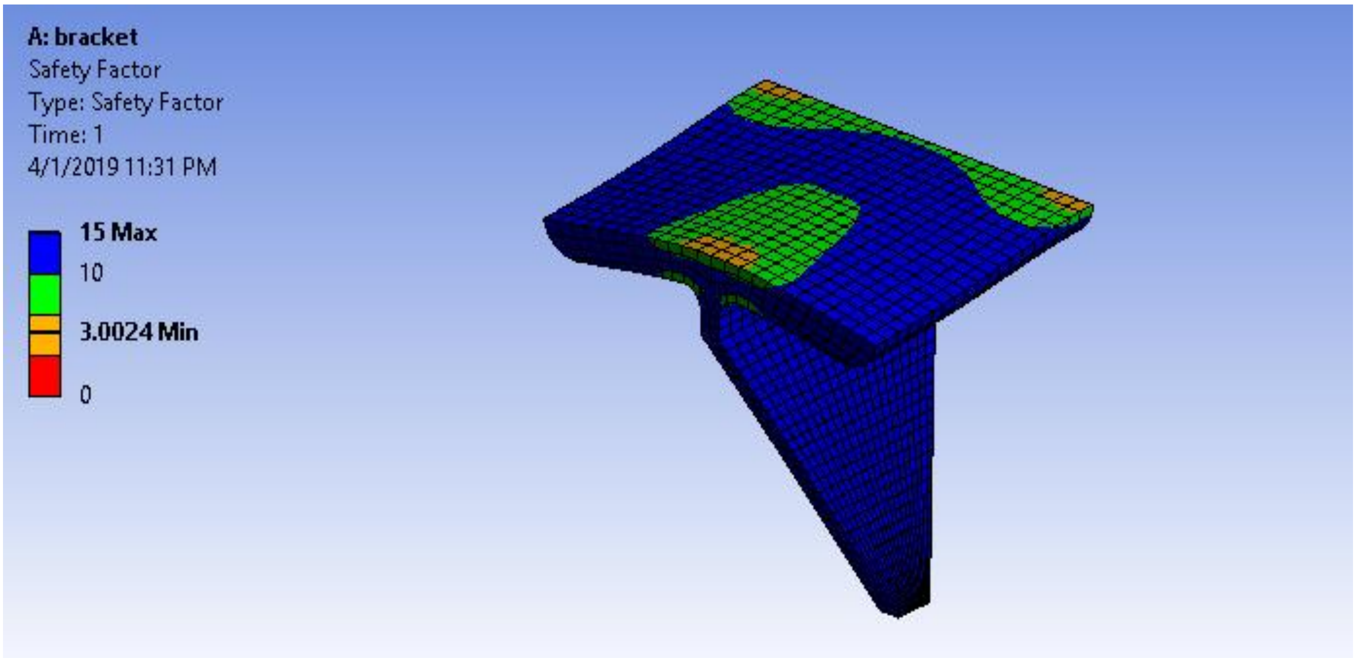
You can insert an Error result based on stresses to help you identify regions of high error and thus show where the model would benefit from a more refined mesh in order to get a more accurate answer. You can also use the Error result to help determine where Mechanical will be refining elements if Convergence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

**Result:**
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.

### 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.

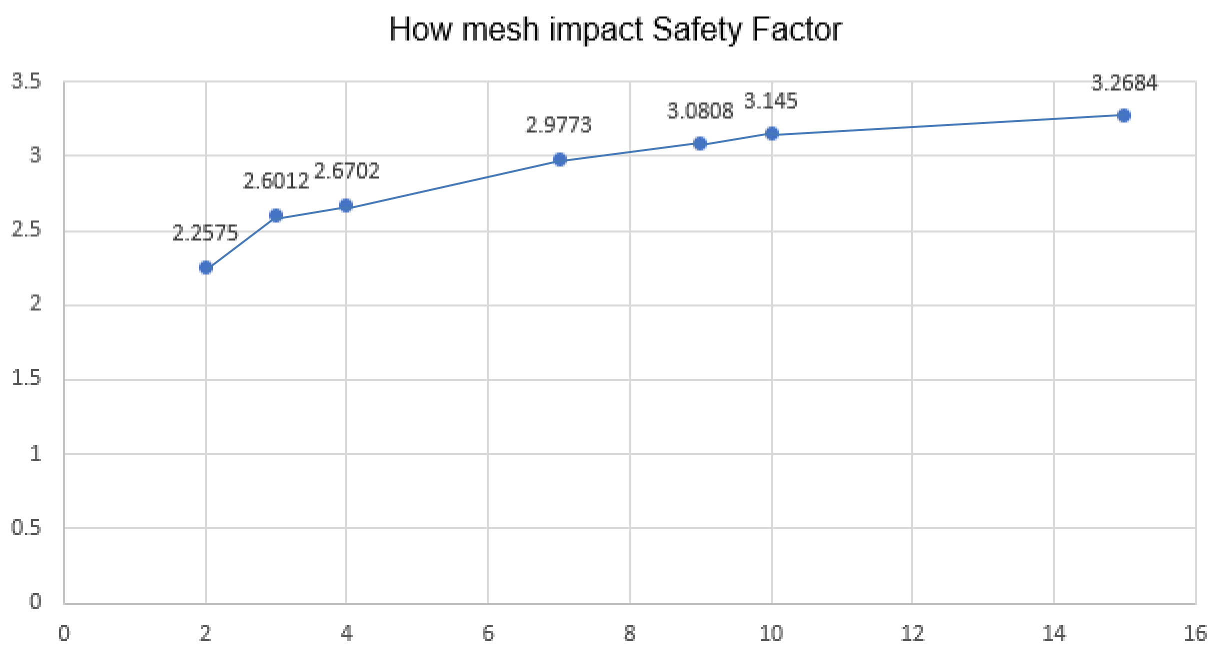
The maximum element size is chosen is 15, because a coarse mesh will require less computational resources to solve and, while it may give a very inaccurate solution, it can still be used as a rough verification and as a check on the applied loads and constraints.

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

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

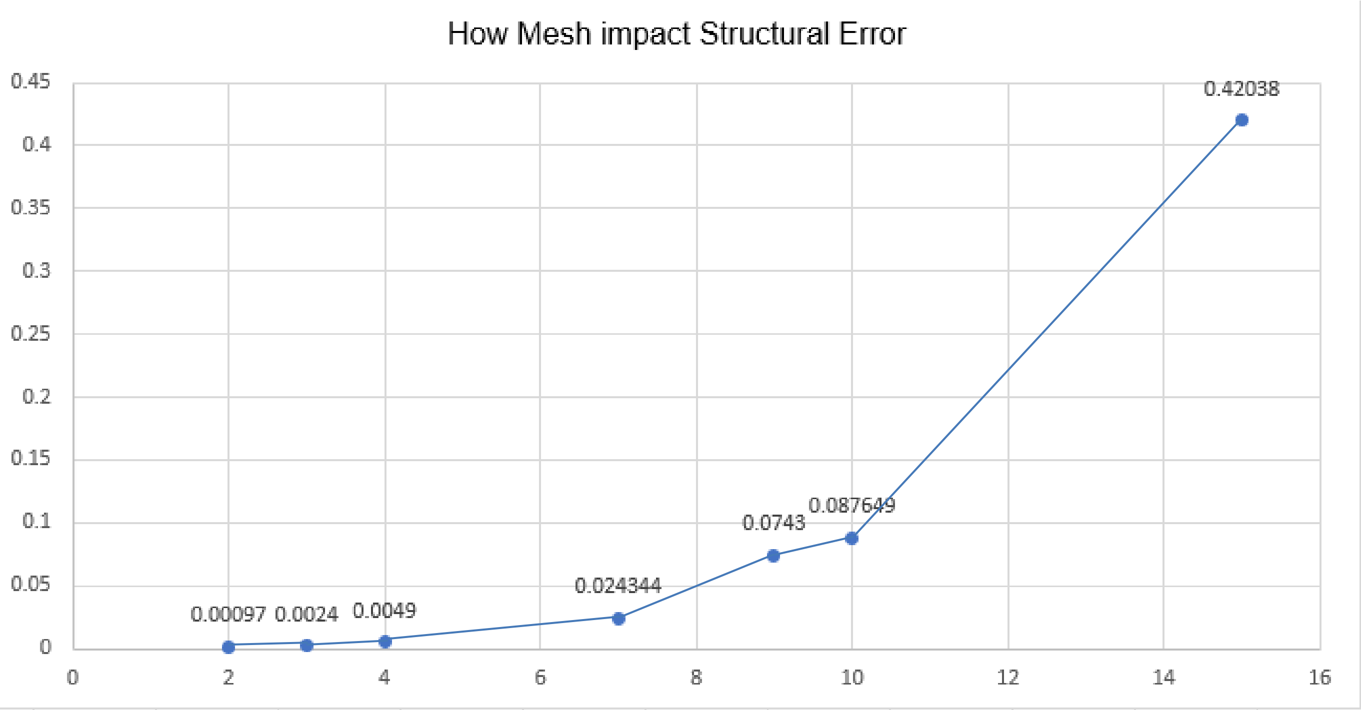
**Safety Factor**




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.

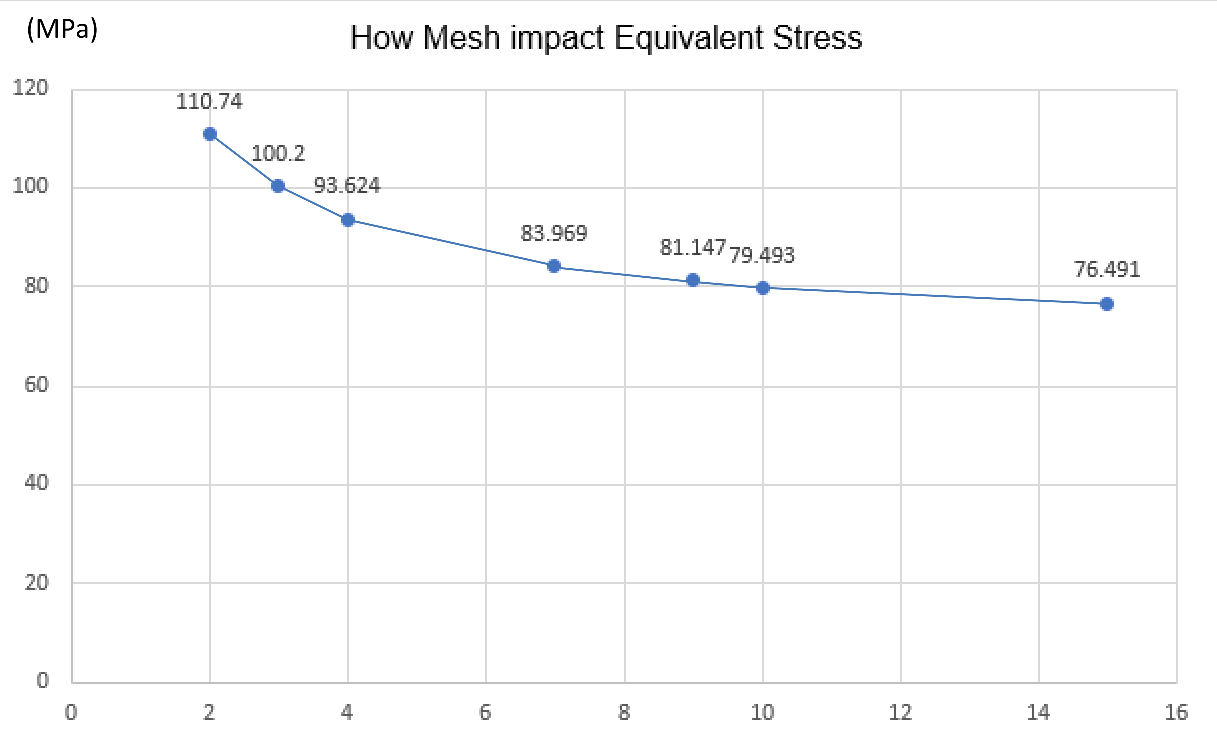
**Structural Error**




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.

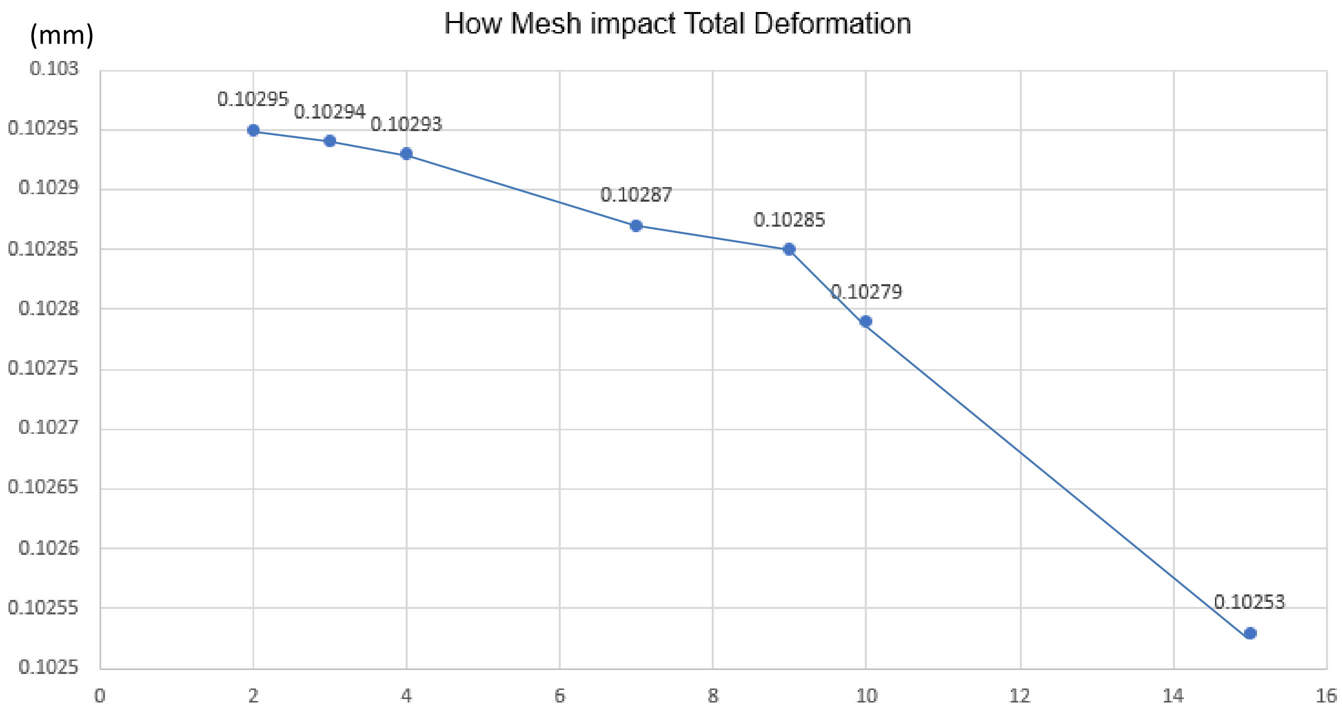
In small mesh value, the accuracy is ensured. Therefore, it will lead to less structural error. But when the mesh value is out of control, it will lead to the large structural error

**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

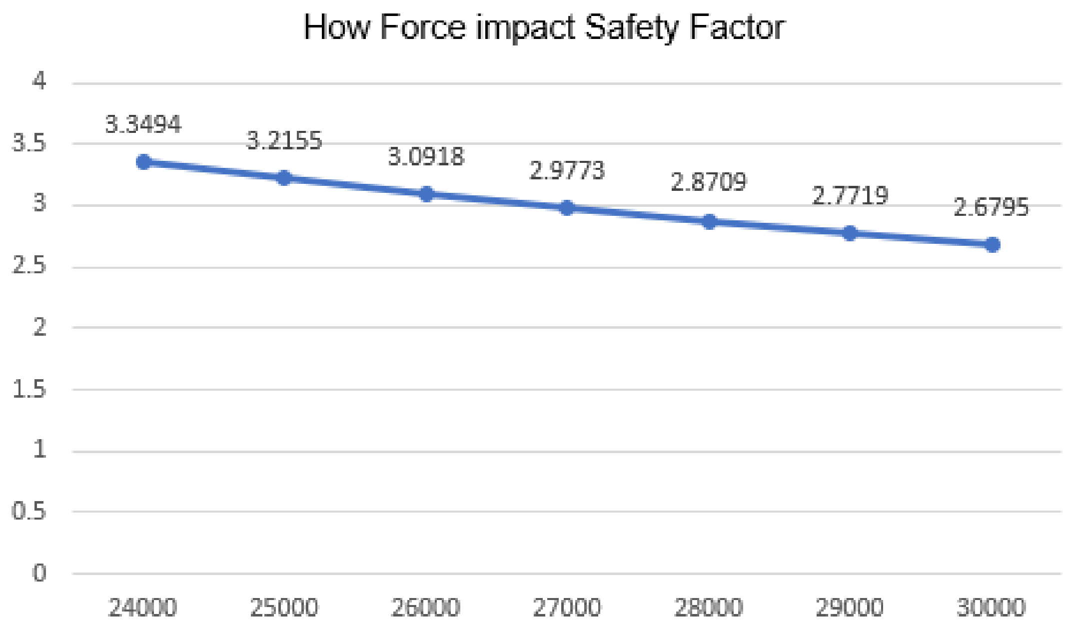
The beam brackets are used mostly in civil engineering industry. They have to be stand will a large force in the seat plate and the force will be spreaded to the net plate (the rib).

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

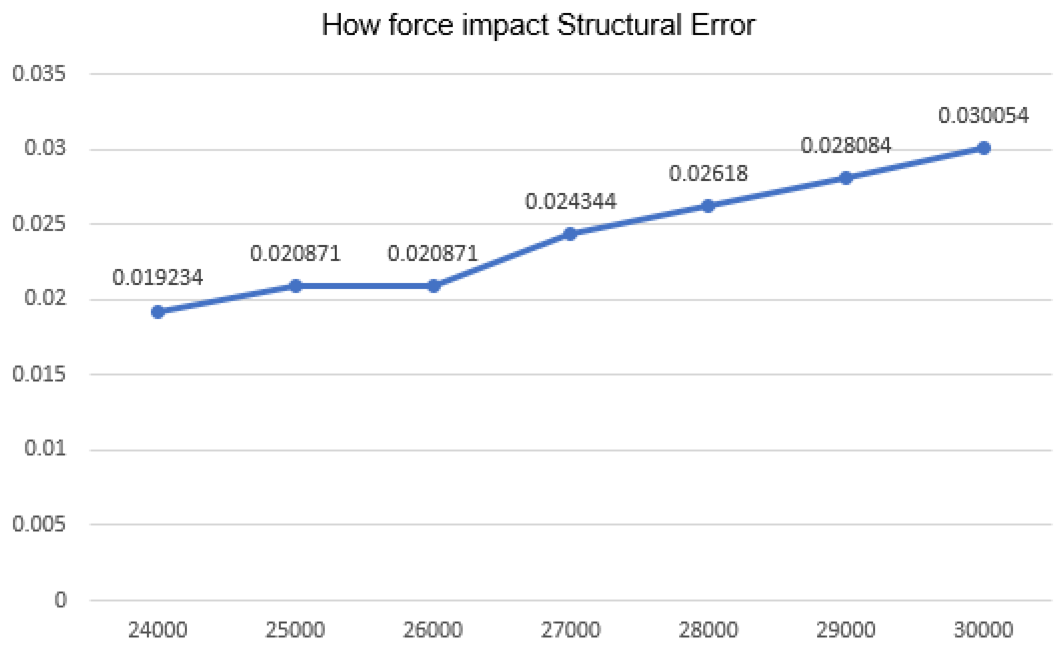
**Safety Factor**



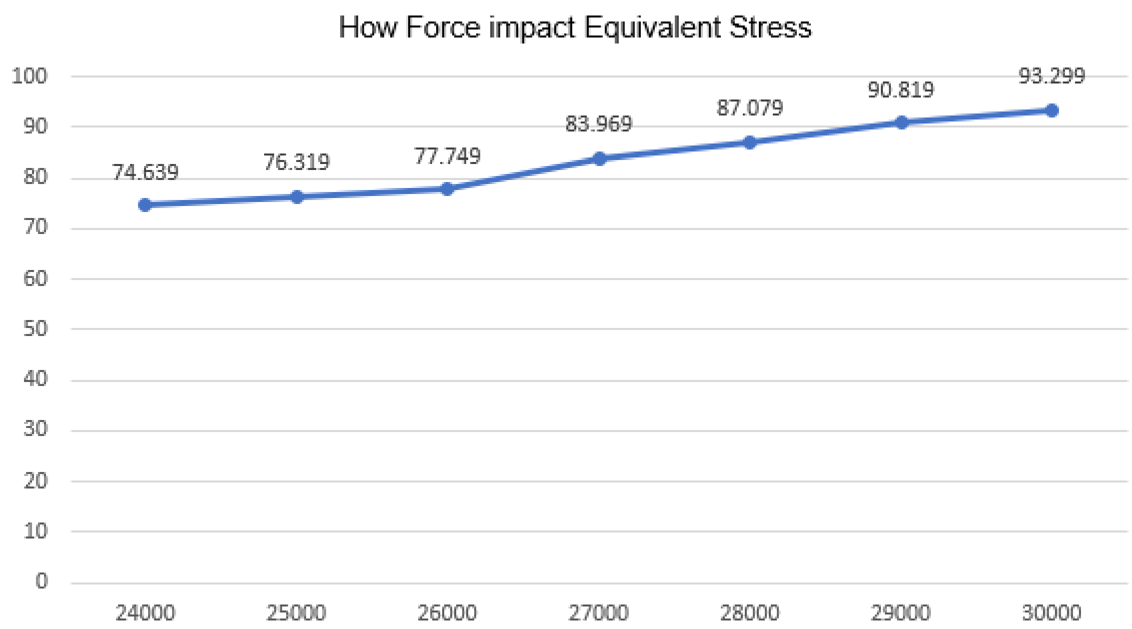

The line graph shows how force impact Safety Factor. The number decrease steadily in the force range from 24000N to 30000N, safety facetor goes down frm 3.3494 to 2.6795, decrease nearly 1 unit.

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**

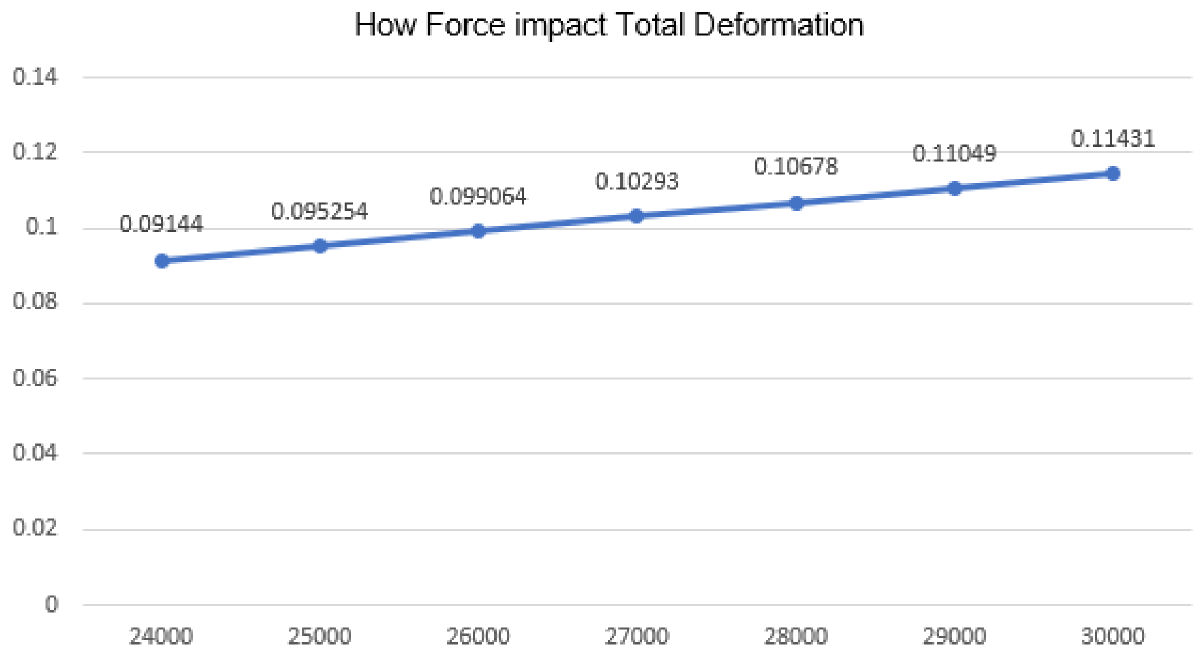



There are a rise number in 2 graphs about how force impacts structural error and equivalent stress.

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.

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