

**People's Democratic Republic of Algeria**  
**The Ministry of Higher Education and Scientific Research**

**University Center Of Ain Temouchent**  
**Institute Of Science And Technology**  
**Civil Engineering Department**



**Graduation project for Obtaining the Master's Degree**

**Branch: Civil Engineering**

**Specialty: Structural Engineering**

Theme:

**MODELING BEHAVIOR OF A SIMPLY SUPPORTED BEAM BY THE  
PROGRAMME ANSYS WORKBENCH**

Presented in September 2016 by:

**LAFEE Abdallah Atieh Mohammad**

Graduation jury members:

<b>Dr</b>	<b>Mme .DERBAL Ahlem</b>	<b>supervisor</b>
<b>Dr</b>	<b>Mme .ATIA</b>	<b>President</b>
<b>Dr</b>	<b>Mme .ABDESSELAM</b>	<b>Examiner</b>
<b>Dr</b>	<b>Mme .MOUSSI</b>	<b>Examiner</b>

## ***Dedication***

*With great pleasure I dedicate this humble work to those who mean more than life to me...my dear parents and all my sisters.*

*To my long path best friends Yazan, Dahrawi , Mohammed Swaitat, Moath, Wael, Amin, Mehdelli, Montasser, Hassan, Humam...all my friends and family.*

*Dedicated to whoever lay his hands on this work.*

LAFEE Abdallah

## *Acknowledgement*

*First of all, praise and thanks be to Allah for all his graces that righteous is done by, and for guiding me through my years of study.*

*This work would not have been possible without the direction and encouragement of my supervisor **Dr.DERBELE Ahlem** and I would like to extend my deepest gratitude and sincere appreciation to her.*

*I would also like to sincerely thank the president of the graduation jury and the members of the jury for accepting and evaluating my project.*

*My special thanks to my second country **Algeria** for holding me as a student and providing me all the resources to finish this study.*

*I would also like to thank all my teachers and professors through all my period of study.*

## Contents

---

<b>Chapter I:</b>	<b>ANSYS program/</b>	<b>1</b>
1.	Introduction	2
2.	Difinition	2
2.1.	programs work with ANSYS	3
2.2.	ANSYS workbench	5
2.3.	Workbench example	7
2.3.1.	New project	7
2.3.2.	Geometric creation modeling	7
2.3.3.	Creating the Geometry	8
2.3.4.	Material properties	19
2.3.5.	Model simulation	20
2.3.6.	Meshing	22
2.3.7.	Boundary conditions	25
2.3.8.	Model resolution	27
<b>Chapter II:</b>	<b>Study</b>	
case/		28
1.	Introduction	28
2.	Distributed load	28
2.1.	Beam characteristics	28
	Hand calculation solutions	29
2.2.	Modeling	30
2.2.1.	Engineering data	30
2.2.2.	Geometry	31
2.2.3.	Model	32
2.2.4.	Setup and solution	33
2.2.5.	Results	35
3.	Center load	37
3.1.	Beam characteristics	37
	Hand calculation solutions	38
3.2.	Modeling	39
3.2.1.	Geometry	39
3.2.2.	Model	41
3.2.3.	Setup and solution	42
3.2.4.	Results	43

<b>Figure 1.1:</b> ANSYS logo.....	2
<b>Figure 1.2:</b> ANSYS products.....	3
<b>Figure 1.3:</b> workbench interface.....	5
<b>Figure 1.4:</b> workbench toolbar .....	5
<b>Figure 1.5:</b> workbench situation bar.....	5
<b>Figure 1.6:</b> static structural cells.....	6
<b>Figure 1.7:</b> creating system. ....	7
<b>Figure 1.8:</b> entering geometry .....	7
<b>Figure 1.9:</b> creating new sketch 1.....	8
<b>Figure 1.10:</b> creating new sketch 2.....	8
<b>Figure 1.11:</b> sketching toolbox .....	9
<b>Figure 1.12:</b> sketching scale .....	9
<b>Figure 1.13:</b> draw line selection .....	10
<b>Figure 1.14:</b> drawing line .....	10
<b>Figure 1.15:</b> creating dimensions.....	11
<b>Figure 1.16:</b> adding dimensions.....	11
<b>Figure 1.17:</b> selecting dimensions.....	12
<b>Figure 1.18:</b> editing dimensions.....	12
<b>Figure 1.19:</b> selecting sketch1 .....	13
<b>Figure 1.20:</b> lines from sketches selection.....	13
<b>Figure 1.21:</b> apply sketch on the line.....	13
<b>Figure 1.22:</b> generate sketch .....	14
<b>Figure 1.23:</b> creating rectangular cross section.....	14
<b>Figure 1.24:</b> creating rectangular.....	15
<b>Figure 1.25:</b> editing rectangular dimensions.....	15
<b>Figure 1.26:</b> selecting the line body.....	16
<b>Figure 1.27:</b> adding rectangular section to line body.....	16
<b>Figure 1.28:</b> showing section step (1) .....	16
<b>Figure 1.29:</b> showing section step (2).....	17
<b>Figure 1.30:</b> rectangular section adding result.....	17
<b>Figure 1.31:</b> saving and quitting geometry .....	18
<b>Figure 1.32:</b> system after closing geometry.....	18

<a href="#"><u>Figure 1.33:</u></a> engineering data's options .....	19
<b>Figure 1.34:</b> structural steel properties.....	19
<b>Figure 1.35:</b> closing engineering data. ....	19
<b>Figure 1.36:</b> geometry properties .....	20
<b>Figure 1.37:</b> checking line bodies.....	20
<b>Figure 1.38:</b> importing new geometry .....	21
<b>Figure 1.39:</b> entering model cell.....	21
<b>Figure 1.40:</b> mesh sizing. ....	22
<b>Figure 1.41:</b> sizing details. ....	22
<b>Figure 1.42:</b> edge selection.....	22
<b>Figure 1.43:</b> applying mesh sizing .....	23
<b>Figure 1.44:</b> number of mesh divisions.....	23
<b>Figure 1.45:</b> mesh in geometry window .....	24
<b>Figure 1.46:</b> generating mesh .....	24
<a href="#"><u>Figure 1.47:</u></a> meshing of the beam .....	24
<a href="#"><u>Figure 1.48:</u></a> adding fixed support.....	25
<a href="#"><u>Figure 1.49:</u></a> vertex selection.....	25
<a href="#"><u>Figure 1.50:</u></a> selected point.....	25
<a href="#"><u>Figure 1.51:</u></a> fixed support.....	26
<a href="#"><u>Figure 1.52:</u></a> adding inertial (standard earth gravity).....	26
<a href="#"><u>Figure 1.53:</u></a> beam with standard earth gravity.....	27
<a href="#"><u>Figure 1.54:</u></a> solving the solution.....	27
<b>Figure 2.1:</b> simply supported beam with distributed load.....	29
<b>Figure 2.2:</b> beam cross-section.....	29
<b>Figure 2.3:</b> displacement1 .....	30
<b>Figure 2.4:</b> moment 1 .....	30
<b>Figure 2.5:</b> shear 1 .....	30
<b>Figure 2.6:</b> engineering data properties .....	31
<b>Figure 2.7:</b> creating surface.....	32
<b>Figure 2.8:</b> full 3D beam. ....	32
<b>Figure 2.9:</b> mesh sizing. ....	33
<b>Figure 2.10:</b> boundary conditions.....	33

<b>Figure 2.11:</b> directional deformation.....	34
<b>Figure 2.12:</b> directional deformation with mesh .....	34
<b>Figure 2.13:</b> equivalent stress .....	35
<b>Figure 2.14:</b> equivalent stress with mesh .....	35
<b>Figure 2.15:</b> cross section showing stress.....	35
<b>Figure 2.16:</b> directional deformation.....	36
<b>Figure 2.17:</b> equivalent stress in center.....	36
<b>Figure 2.18:</b> normal stress.....	37
<b>Figure 2.19:</b> centerload.....	38
<b>Figure 2.20:</b> displacement 2.....	39
<b>Figure 2.21:</b> moment 2.....	39
<b>Figure 2.22:</b> shear 2.....	39
<b>Figure 2.23:</b> sketch projection.....	40
<b>Figure 2.24:</b> face split step.....	41
<b>Figure 2.25:</b> top faces after splitting.....	41
<b>Figure 2.26:</b> center load force adding.....	42
<b>Figure 2.27:</b> boundary conditions.....	42
<b>Figure 2.28:</b> convergence percentage .....	43
<b>Figure 2.29:</b> equivalent stress.....	44
<b>Figure 2.30:</b> directional deformation.....	44
<b>Figure 2.31:</b> force reaction at the fixed support.....	45
<b>Figure 2.32:</b> force reaction at the displacement.....	45
<b>Figure 2.33:</b> equivalent stresses with mesh refining.....	46
<b>Figure 2.34:</b> directional deformation with mesh refining.....	46

---

تصميم العناصر الإنشائية بواسطة برنامج (ANSYS workbench 14.5) ستقوم بهذا المشروع بتصميم كمرّة بناء بواسطة البرنامج وذلك عن طريق التوضيح بمثال بسيط ثم عرض حالتين للدراسة و حلها مع توضيح الخطوات.  
بعد مقارنة النتائج للحسابات اليدوية مع النتائج التي يعرضها البرنامج وتوافقهما نستنتج فوائد برامج التصميم في توفير التكلفة والجهد في محاكاة هذه التجارب مع ضمان سرعة المعالجة و دقة النتائج.

**Abstract:**

Modeling a simply supported beam by (ANSYS workbench) software

Modeling construction elements (a simply supported beam, in our case) by computer software like the ANSYS workbench decreases the cost and ensures the accuracy of the results.

This project present two study cases; comparing the software results with the results of hand calculations showing some software's benefits.

**Résumé :**

Modélisation d'une poutre simplement appuyée par (ANSYS Workbench) logiciel.

Modélisation des éléments de construction (une poutre simplement appuyée, dans notre cas) par des logiciels comme (ANSYSworkbench) diminue le coût et assure la précision des résultats.

Je présente deux cas d'étude dans le mémoire comparant les résultats de logiciels avec les résultats des calculs à la main montrant les avantages des certains logiciels.



### 1. Introduction

ANSYS (finite element analysis) engineering simulation software and training. It allows you to solve your engineering problems with a low-budget by simplifying the application of finite element analysis to iteratively test your design by 40 years of experiences, pioneering and applying FEA (finite element analysis) methods to solve the most complex engineering design challenges.

### 2. ANSYS software

It is a generally purposed software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers.



Figure 1.1 ANSYS logo

ANSYS, which enables to simulation tests or working conditions, enables to test in virtual environment before manufacturing prototypes of products. Furthermore, determining and improving weak points, computing life and foreseeing probable problems are possible by 3D simulation in virtual environment.

ANSYS can import CAD data and also enables to build geometry with its “preprocessing” abilities. Similarly in the same preprocessor, finite element model (a.k.a mesh) which is required for computation is generated. After defining loadings and carrying out analyses. Results can be viewed as numerical and graphical.

It also can carry out advanced engineering analyses quickly, safely and practically by its variety of contact algorithms, time based loading features and nonlinear material models.

## 2.1. Programs work with ANSYS software

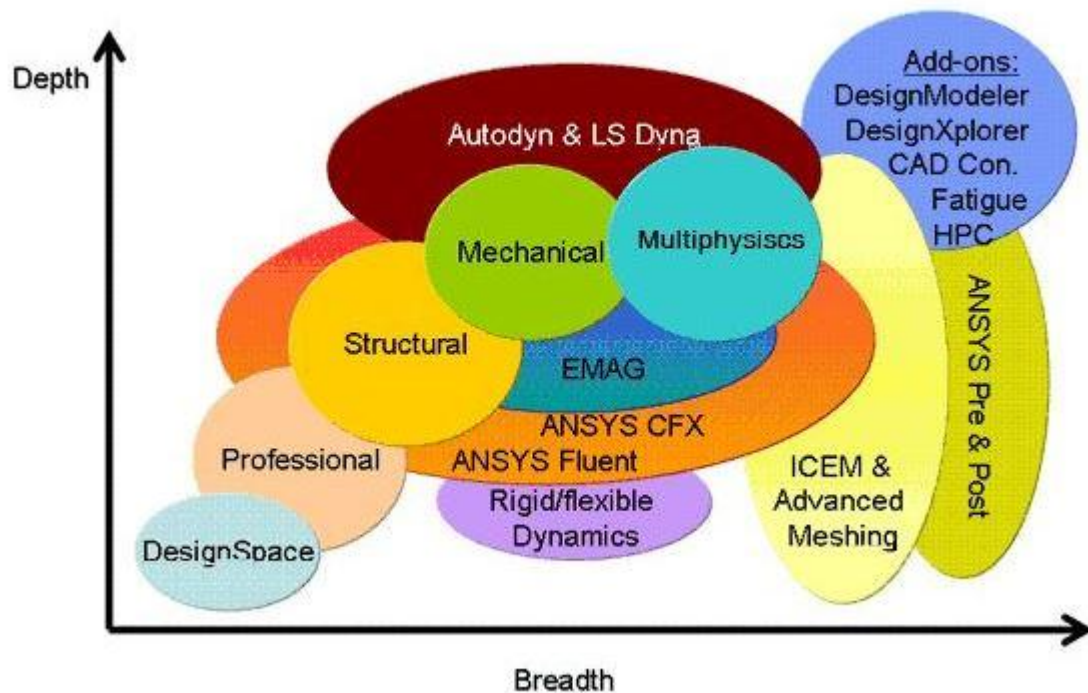


Figure 1.2 ANSYS products

### ANSYS Structural

The **ANSYS structural** analysis software suite enables the user to solve complex **structural** engineering problems and make better, faster design decisions. With finite element analysis (FEA) tools, that can customize and automate the simulations, and parameterize them to analyze multiple design scenarios. This product is capable to perform mechanical simulations in the structural analysis such as;

- Static/modal analysis
- The temporal analysis
- Harmonic analysis or forced response
- The management of different nonlinear situations (contacts, materials plasticity, large displacements and large deformations).

### ANSYS Mechanical

This product has the same capacities of the ANSYS structural including adding thermal solver, with modeling of radiation. It provides a complete set of elements behavior, material models and equation solvers for a wide range of mechanical design problems.

### ANSYS Fluent

ANSYS Fluent is the most-powerful computational fluid dynamics (CFD) software tool available; Fluent includes well-validated physical modeling capabilities to deliver fast, accurate results across the widest range of CFD and multiphysics applications.

## **ANSYS CFX**

ANSYS CFX is a high-performance computational fluid dynamics (CFD) software tool that delivers reliable and accurate solutions quickly and robustly across a wide range of CFD and multi-physics applications. CFX is recognized for its outstanding accuracy, robustness and speed with rotating machinery such as pumps, fans, compressors, and gas and hydraulic turbines.

## **ANSYS multiphysics**

ANSYS multiphysics solutions can help engineers examine; (Fluid forces, thermal effects, structural integrity and electromagnetic radiation) all in combination and isolation with an accurate prediction of behavior.

## **ANSYS professional.**

This software provides a first step into linear dynamics and the nonlinear capabilities of ANSYS structural mechanics. It comes in two versions:

- **ANSYS professional NLT**

Includes linear structural analysis, dynamic analysis and nonlinear thermal capability.

- **ANSYS professional NLS**

Includes the thermal calculations, calculation of linear structures and the basic calculation in nonlinear capability.

## **ANSYS Design Space**

This software is an easy-to-use simulation package that provides you with tools to conceptualize design and validate your ideas right on the desktop using the same solver technology as ANSYS Mechanical. With DesignSpace, you don't need to have advanced analysis knowledge to perform real-world static structural and thermal, dynamic, weight optimization, vibration mode, and safety factor simulations.

Design Space was developed for the exclusive purpose of being used to complement a 3D CAD tool to provide designers and engineers to design a fast return and a better quality performance of their products during the design cycle.

## 2.2. ANSYS workbench:

The ANSYS Workbench platform is the backbone for delivering a comprehensive and integrated simulation system to designers. Using Workbench for the product development simulations will result in higher productivity from integrated applications leveraging common and compatible data models. Workbench gives designers access to multiphysics and systems level insights that could not be attained before.

### Main menu of ANSYS workbench

Opening the software displays the main menu shown below:

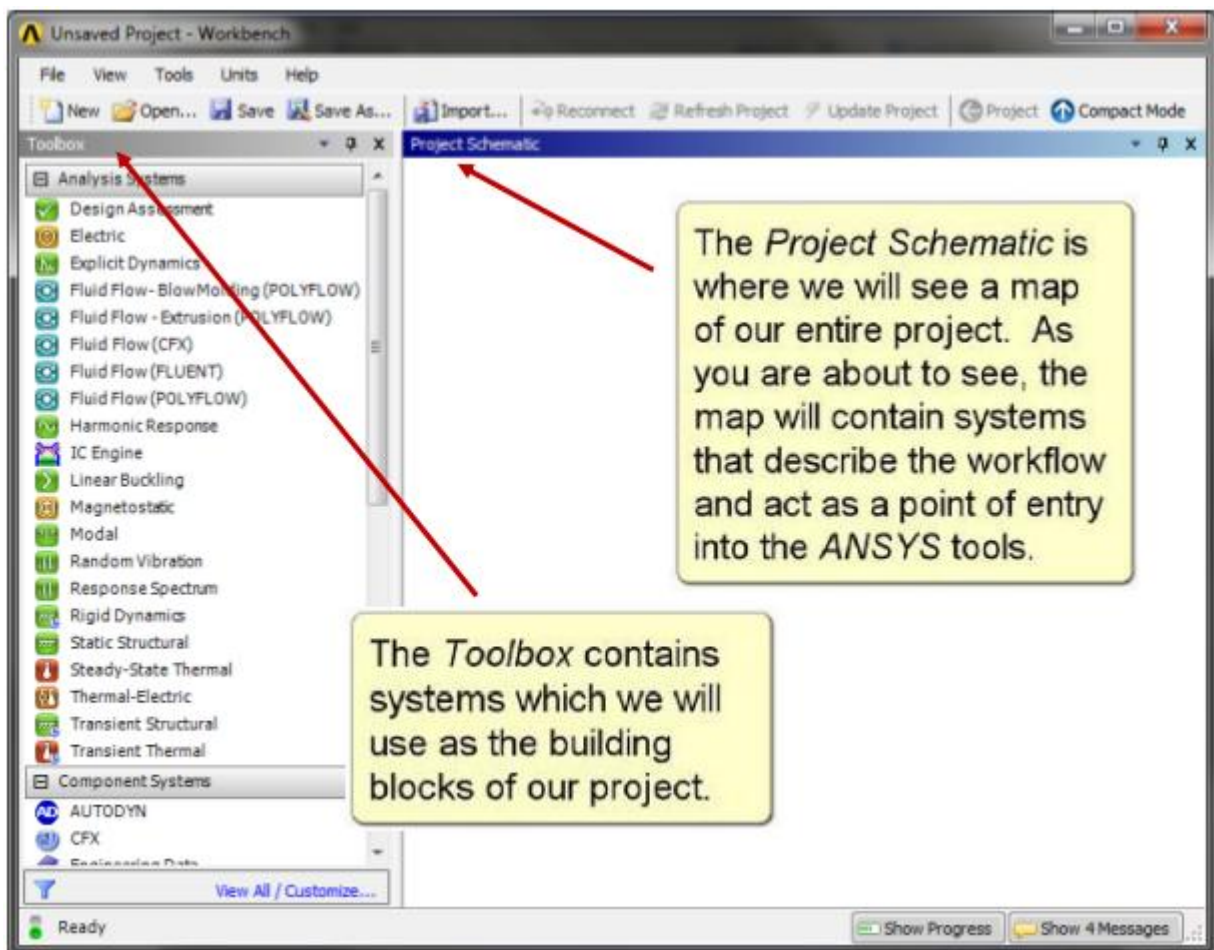


Figure 1.3 ANSYS workbench interface

In the main menu there is the tool bar which allows an easy access to many commands and also to the software options. Clicking on the save as button will gather all the project files in on folder

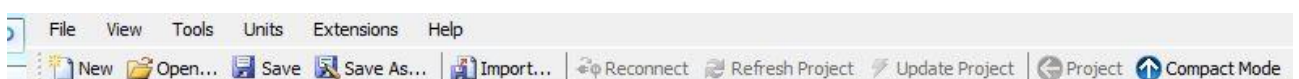
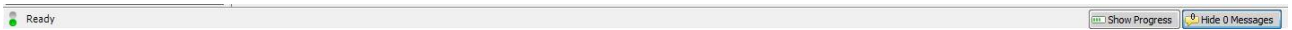


Figure 1.4 workbench toolbar

It also contains the situation bar which shows the messages and the readiness of the software



**Figure 1.5** situation bar

The system includes various cells either: engineering data, geometry, model, setup, solution and results. To facilitate their comprehensions, they can be grouped according to the already studied Mechanical APDL software (ANSYS)

- Pre-processing: Engineering Data, Geometry and Model
- Resolution: Setup and Solution
- Post-processing: Result

It is possible to get a description of each cell in the ANSYS Workbench's help file:

### **Engineering Data**

The Engineering Data cell is designed to allow the designer to create, save, and retrieve material models, as well as to create libraries of data that can be saved and used in subsequent projects and by other users. Double-click on engineering data cell or RMC then edit to enter the environment.

### **Geometry**

Use the Geometry cell to import, create, modify or update a model that can be used for analysis, allowing seamless transfer of existing data including parameters. The parameters can then be adjusted and the design updated, and any feature removal or simplification is maintained. This results in rapid turnaround of any design changes and updates. Double-click or RMC then edit to enter the environment.

### **Model/mesh**

Model cell is associated with the definition of the geometry, the systems of coordinates, connections and mesh in the mechanical simulation module. It also allows you to access a meshing application or share a mesh with another system. Model corresponds to the contents of the Model branch within the ANSYS Mechanical application and allows you to perform physics-based meshing capabilities, such as spot welds, contact, etc. Mesh contains just node coordinates and mesh connectivity.

### **Setup**

Setup cell lets you define loads, boundary conditions and other configuration analysis

### **Solution**

Solution cell allows access to data resolution

### **Results**

Results cell includes the result of the analysis. This is often referred to a post-processing cell.

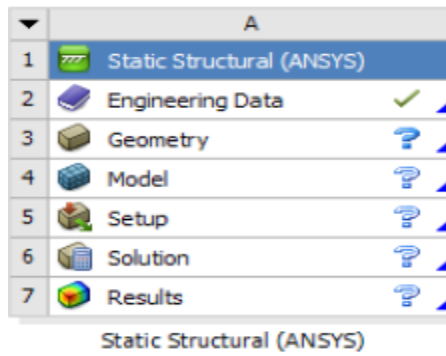


Figure 1.6 static structural cells

### 2.3. WORKBENCH example:

#### 2.3.1. New project

For starting a new project LMC on new then using dragging and dropping on the static structural in the toolbox to the project schematic area

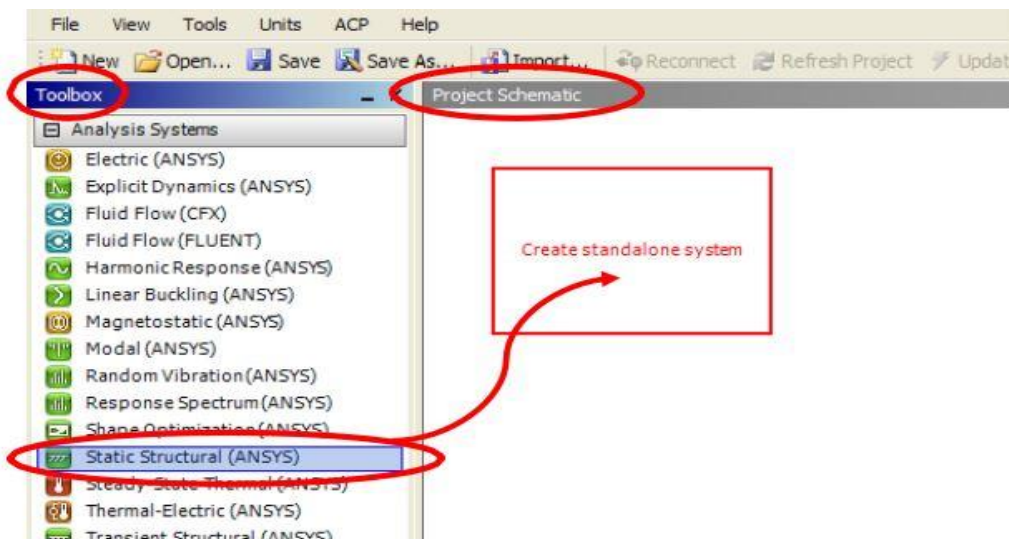
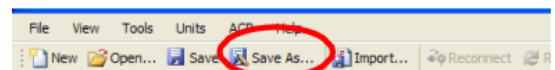


Figure 1.7 creating system

Save the project by LMC on save as: →



#### 2.3.2. Geometric creation modeling

Double click on the geometry to open the creation modeling. After opening 'Geometry creation modeling, Select Millimeter as the unit of dimension.



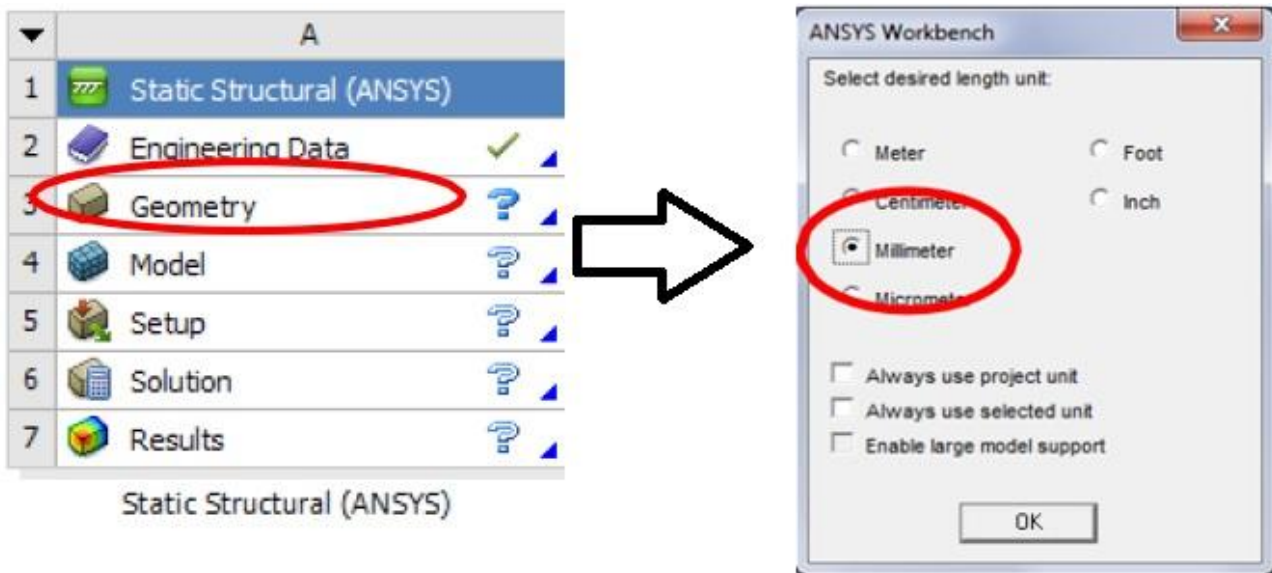
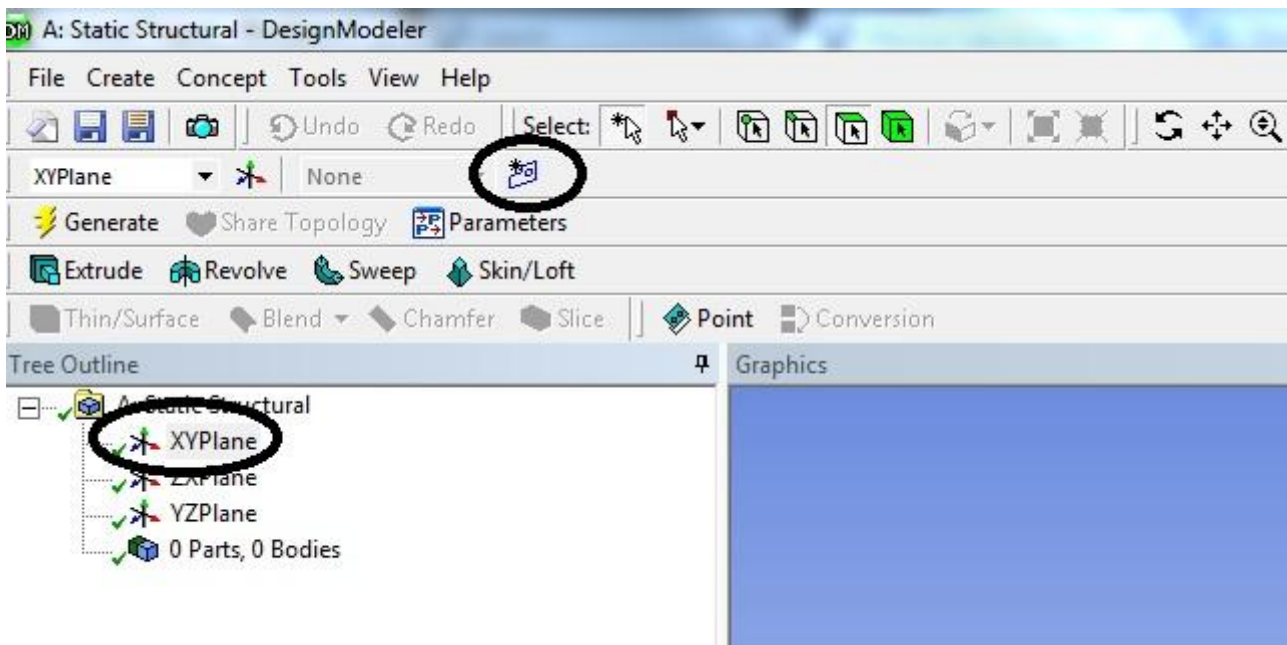


Figure 1.8 entering geometry

### 2.3.3. Creating the Geometry

Afterwards, LMC on XY plane under (Tree Outline) to select the plane and LMC on (New Sketch).



**Figure 1.9** creating a new sketch 1

A new drawing will appear under the XY plane in the tree outline, LMC on Sketch1 and LMC on the tape Sketching



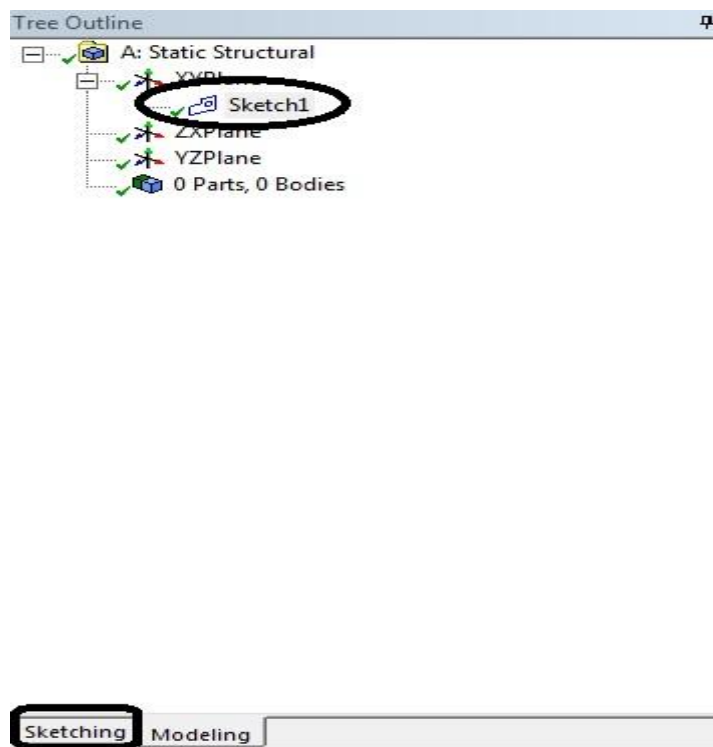


Figure 1.10 creating a new sketch 2

Sketching Toolboxes will appear on the left. This toolbox to create the geometry, is used for adding the dimensions and forcing it

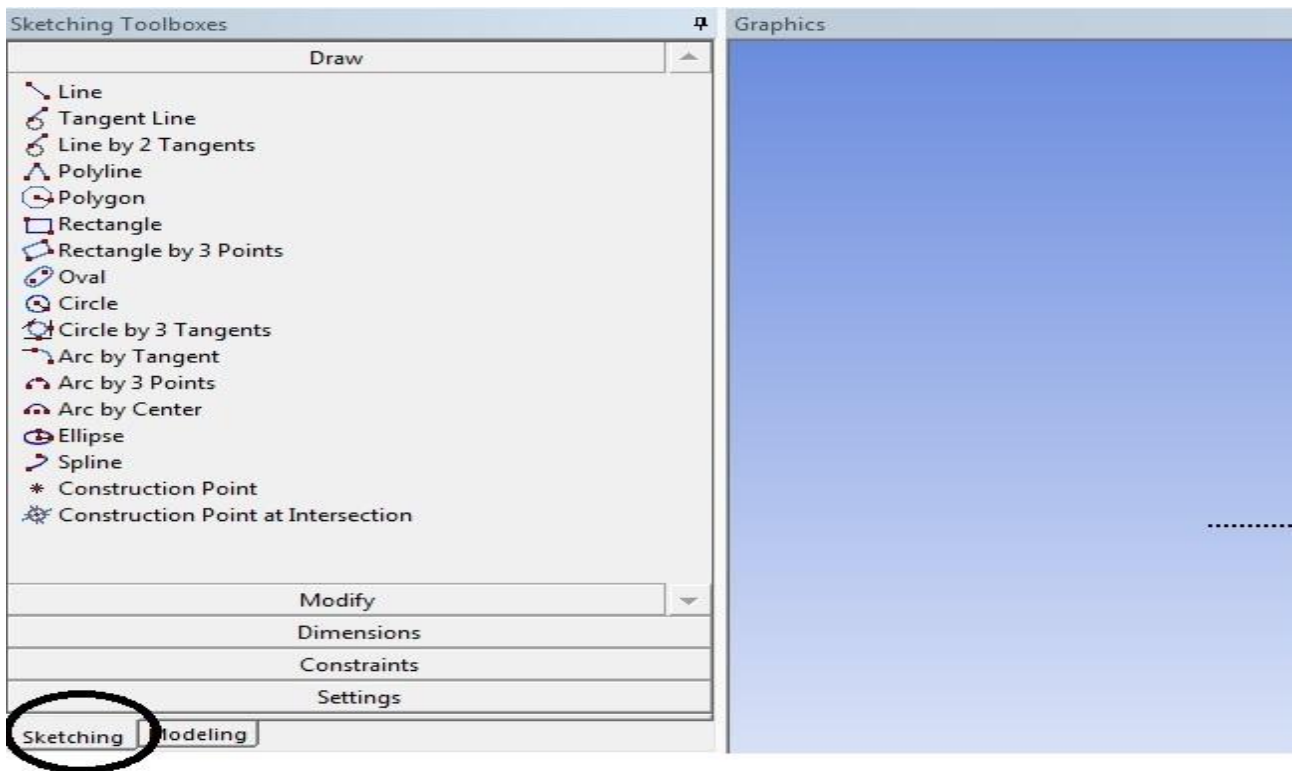


Figure 1.11 sketching toolbox

Before drawing, make sure that the scale is representing the problem in precision. At the following case, the length of the beam is 1000mm. there are several ways to perform this operation. The most accessible is the combination of the key Ctrl+MMS.

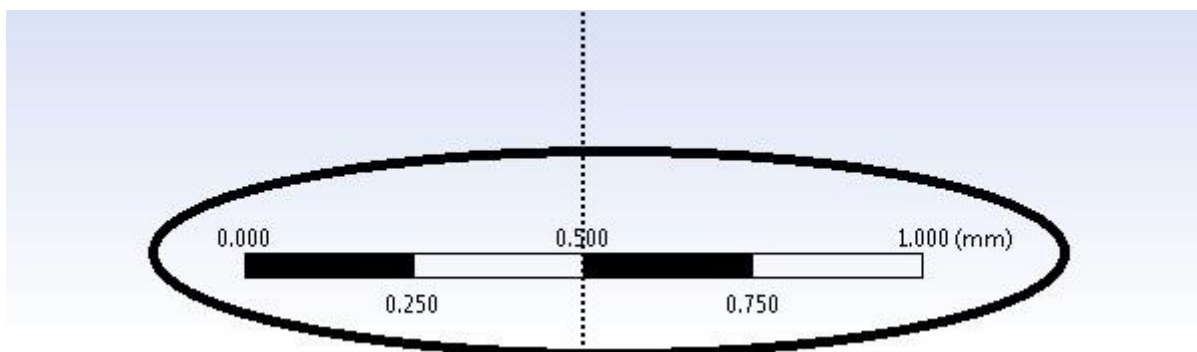


Figure 1.12 sketching scale

Select line in the Sketching Toolboxes under Draw.

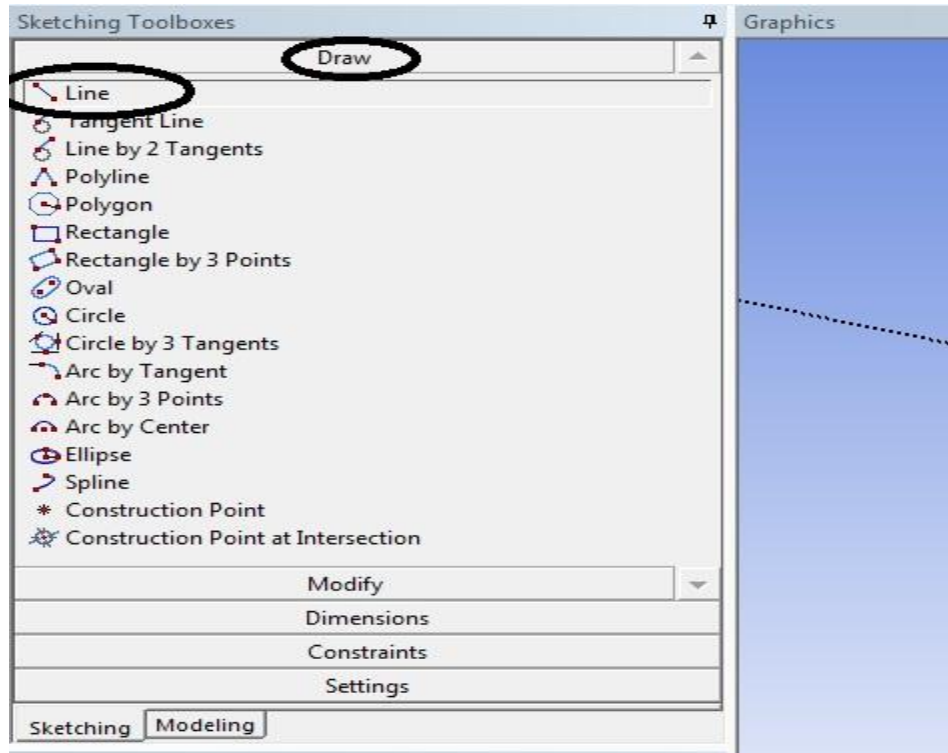


Figure 1.13 draw line selection

Place the cursor of the mouse at the point of origin of the plan. The letter P should appear to signify that the point is fixed to the center. LMC is to fix the first point of the line and move to the bottom right in order to get the notation H for Horizontal. Once again, obtaining LMC, Then press Esc button on the keyboard.

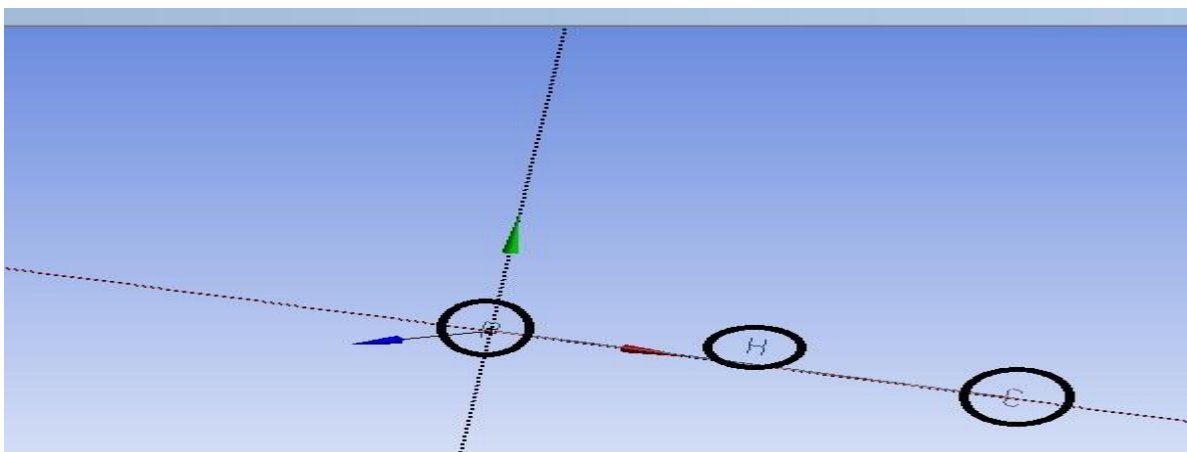
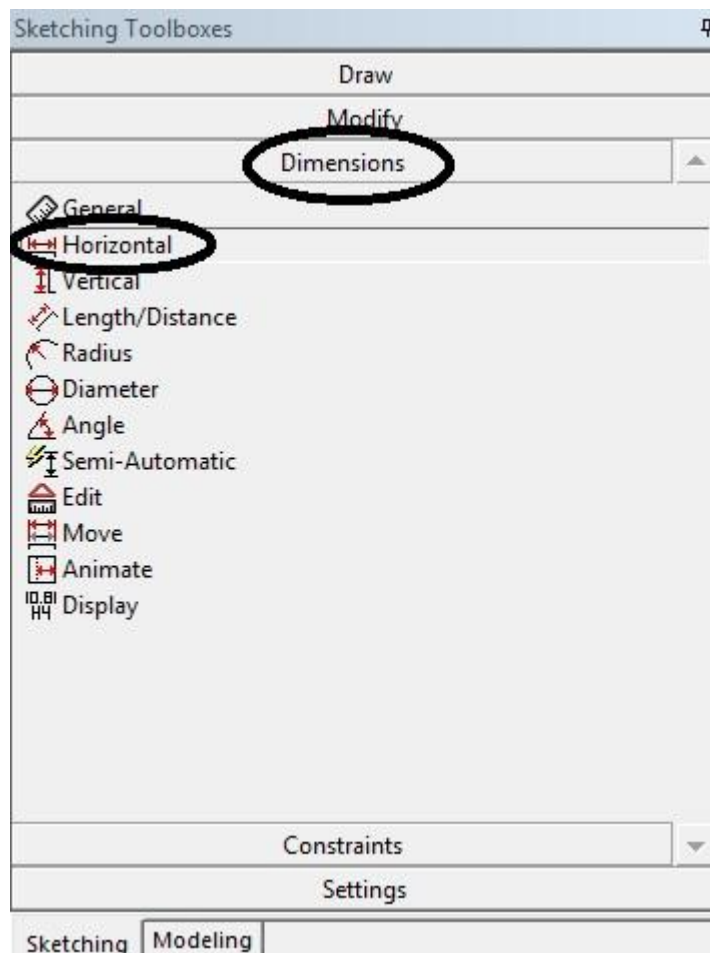


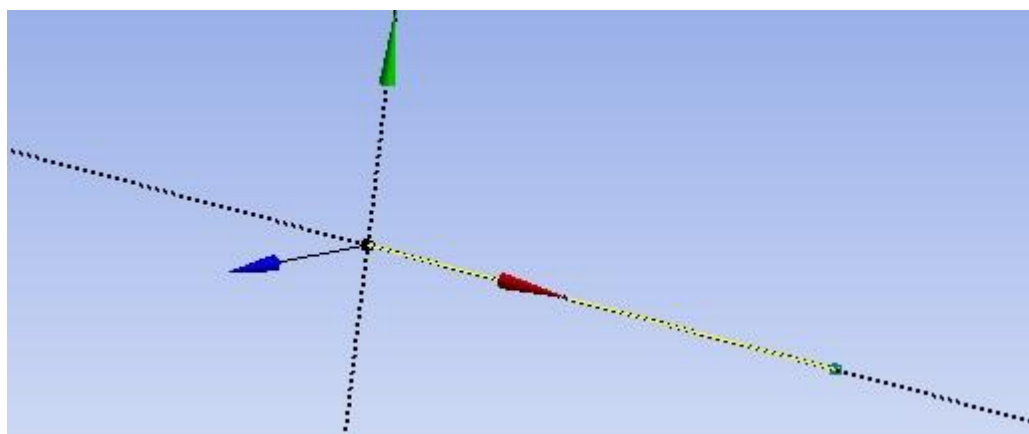
Figure 1.14 drawing line

Select Horizontal in Sketching Toolboxes under the tape Dimensions



**Figure 1.15** creating dimensions

Click the top of the line to add the horizontal dimension. Which should be yellow once selected.



**Figure 1.16** adding dimensions

LMC to fix it to the desired location. then, LMC on the dimension to select, it should be yellow once selected.

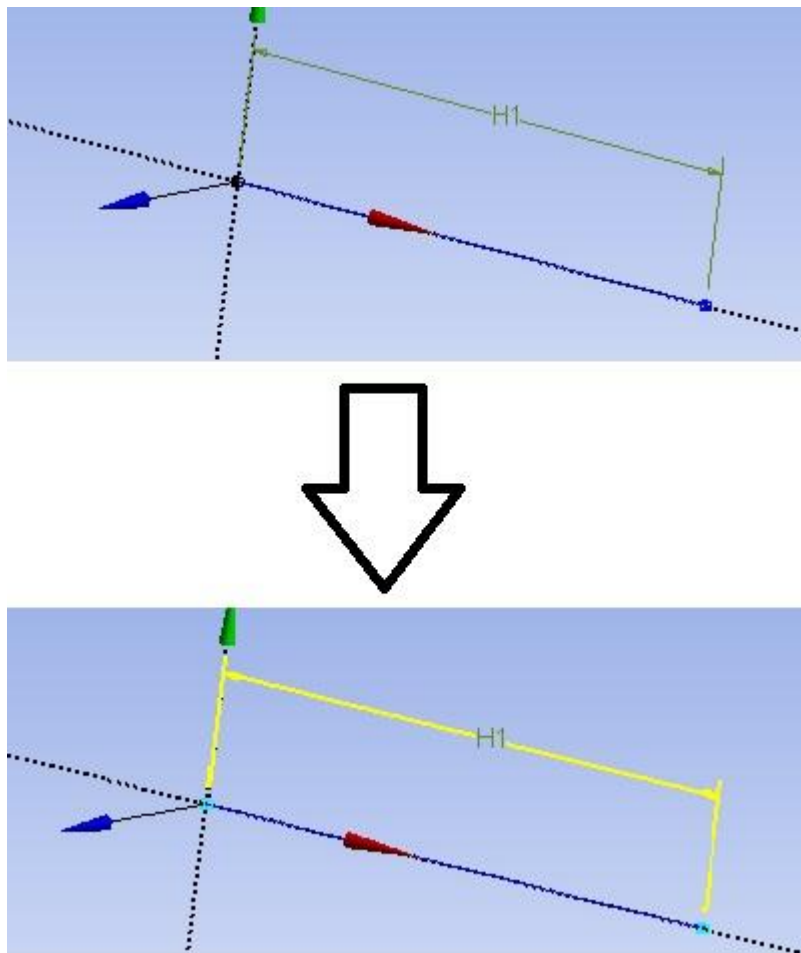
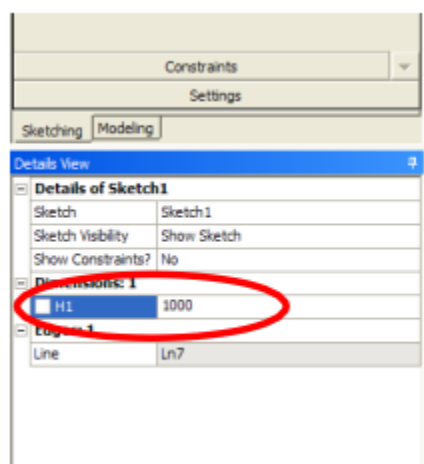


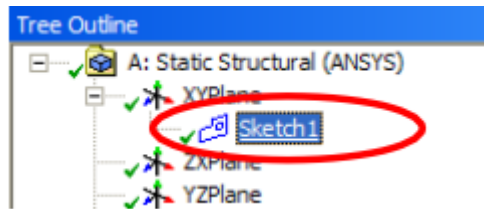
Figure 1.17 selecting dimension

The selection allows you to edit the dimension in Details View under Sketching Toolboxes. Enter 1000 for the dimension 1000mm (length of the beam).



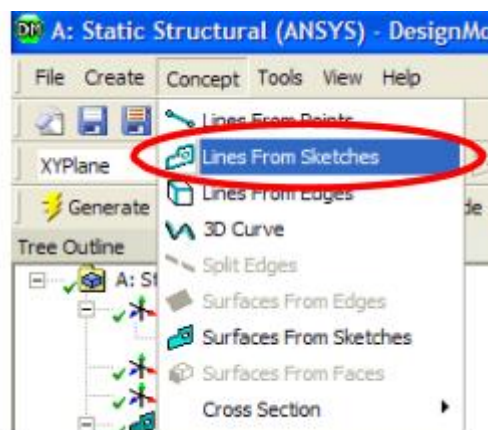
**Figure 1.18** editing dimensions

LMC on modeling tab to return to Tree Outline and LMC on Sketch1



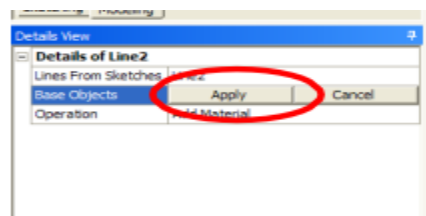
**Figure 1.19** selecting sketch1

In the menu of the module of geometric creation LMC on Concept/Lines from Sketches.



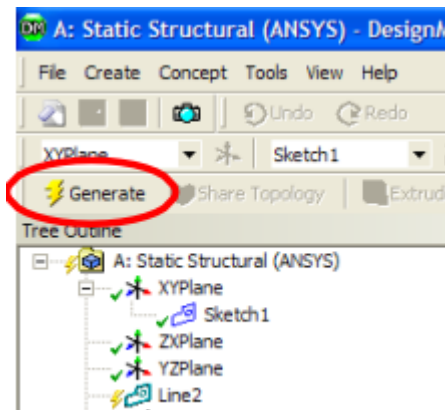
**Figure 1.20** lines from sketches selecting

Under Details View RMC on Apply; to apply your Sketch1 on the line.



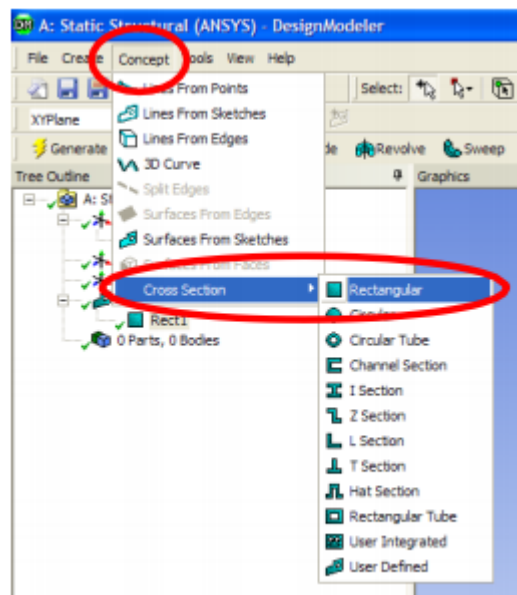
**Figure 1.21** apply sketch on the line

Afterward, click on Generate, this operation has the effect of updating your project.



**Figure 1.22** generate the sketch

In the menu of the module of geometric creation LMC at Concept/Cross Section/Rectangular.



**Figure 1.23** creating the rectangular cross section

The rectangular beam section is now created, it should appear under Tree Outline.

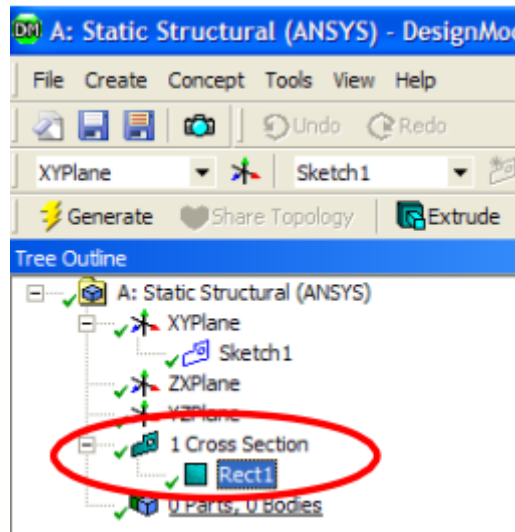


Figure 1.24 created rectangular

LMC to edit the section, always in Details View, modify the section to obtain a rectangular section of the dimensions 50mm\*10mm. note that according to the initial place only B and H can be reversed. It is possible to verify this step later.

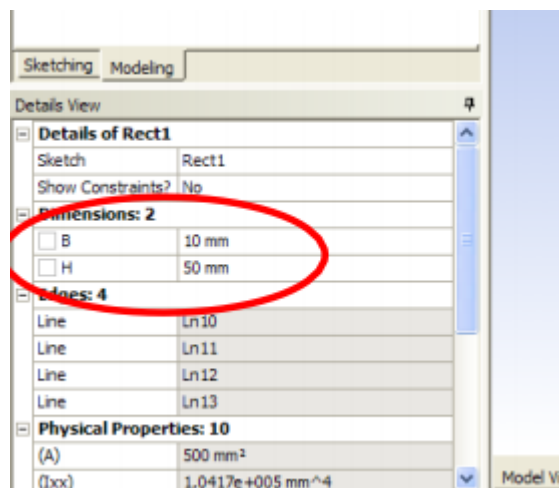
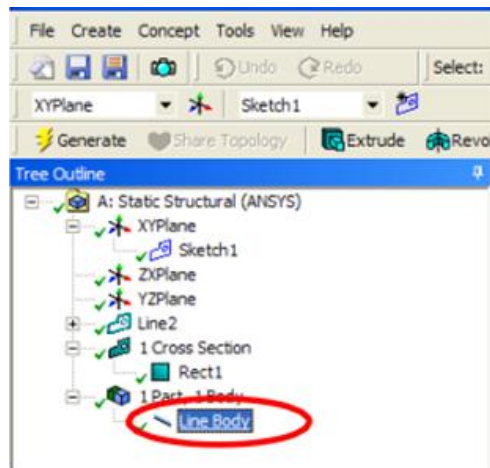


Figure 1.25 editing rectangular section dimensions

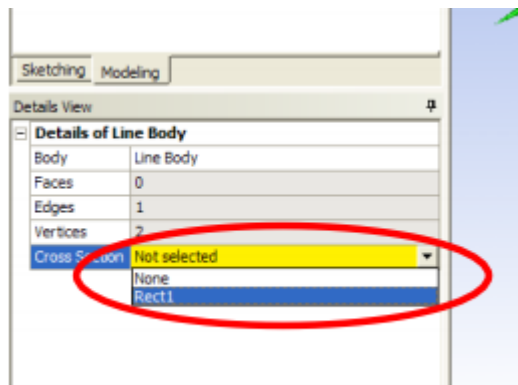


LMC on Line Body under tree outline.



**Figure 1.26** selecting the line body

Under Details View, select your rectangular Rect1.



**Figure 1.27** adding rectangular section to the line body

And you should obtain the following, to view your section.

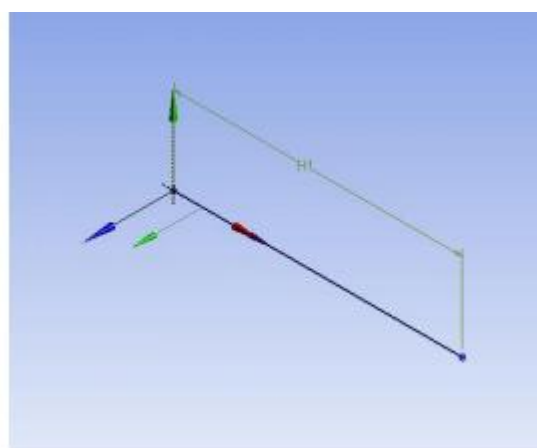


Figure 1.28 showing section step (1)

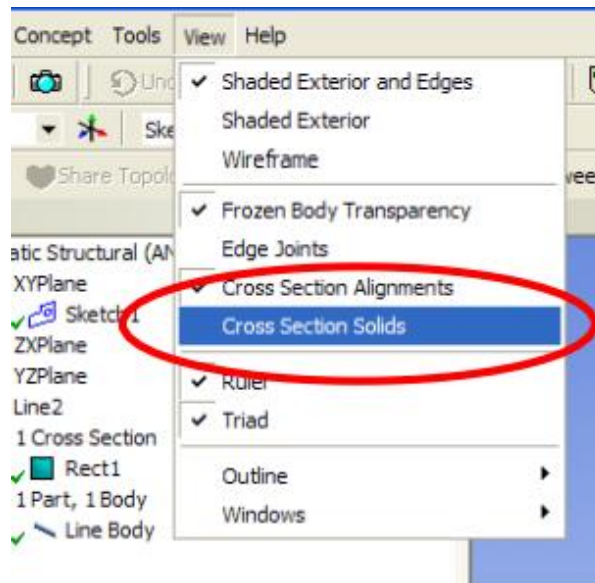


Figure 1.29 showing section step (2)

And you should obtain the following

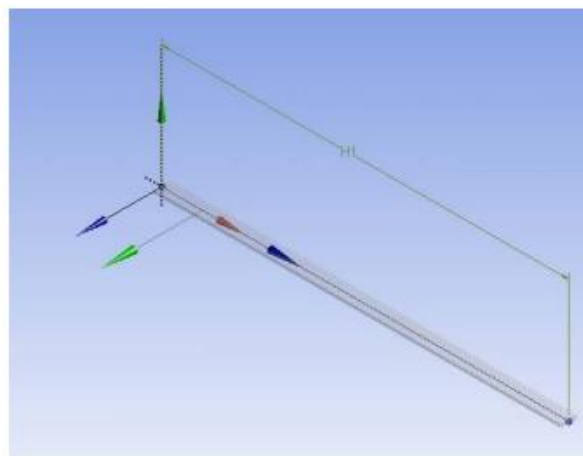
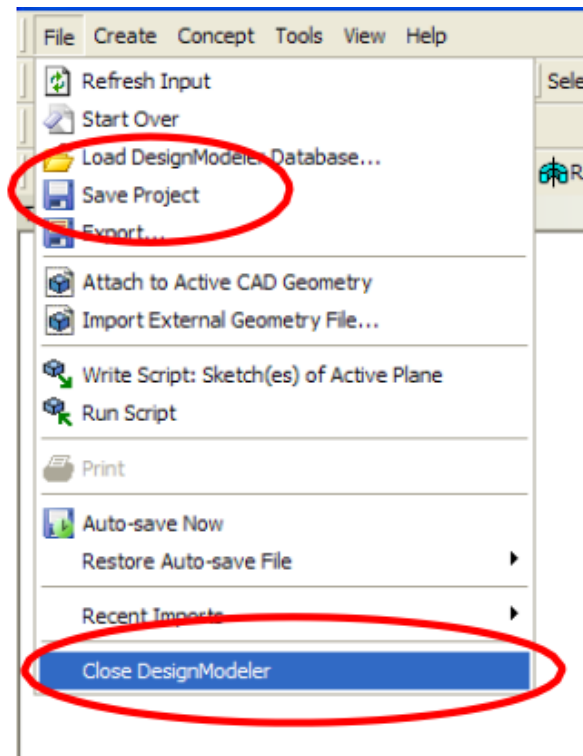


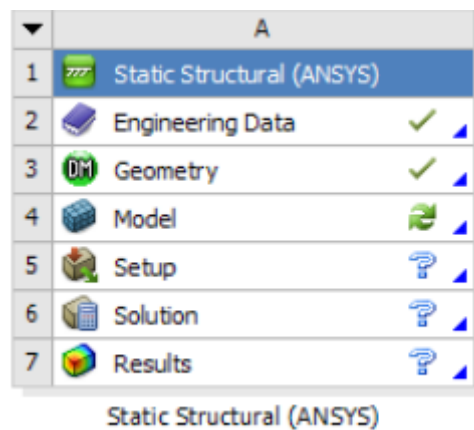
Figure 1.30 rectangular section adding result

Save your project and quite the module of geometric creation.



**Figure 1.31** saving and quitting the geometry

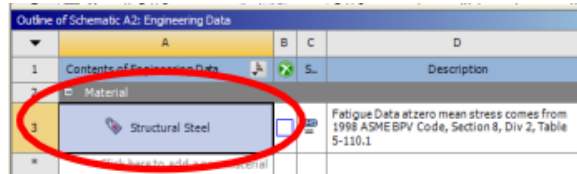
Display the main menu of Workbench. Your system now should have green check marks for the completed elements.



**Figure 1.32** system after closing geometry

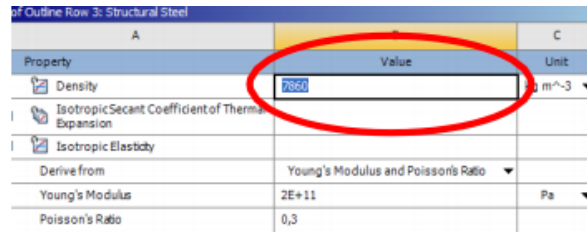
**2.3.1. Material properties**

Workbench used for steel structure by default where the green chick marks are used for Engineering Data. However, we have to change the properties. Double LMC on Engineering Data. LMC on Structural Steel.



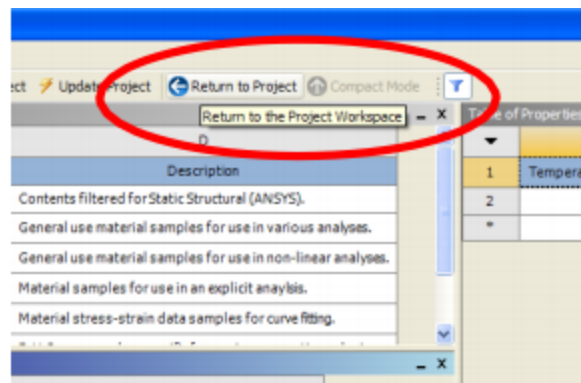
**Figure 1.33** engineering data's options

LMC on the Density value and enter 7860.  
Young module is 200 GPa and Poisson' Ratio is 0.3



**Figure 1.34** structural steel properties

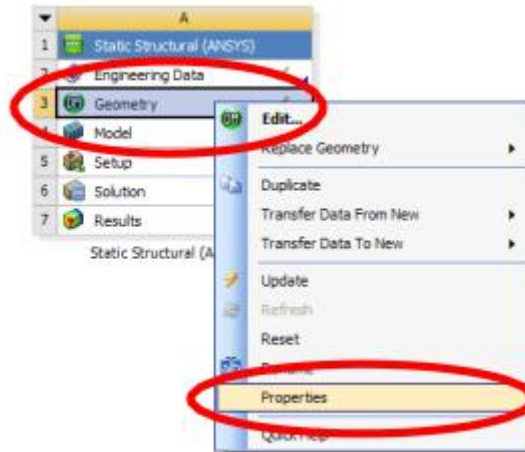
LMC on Return to Project.



**Figure 1.35** closing engineering data

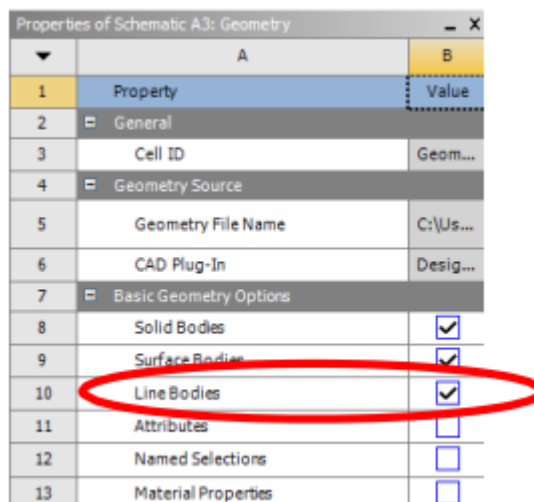
**2.3.2. Model simulation**

RMC on Geometry and LMC on Properties.



**Figure 1.36** geometry properties

The properties of the geometry should appear on the right of the main menu. Check Line Bodies.



**Figure 1.37** checking line bodies

RMC on Module and LMC on Refresh to import the geometry of the beam in the simulation module.

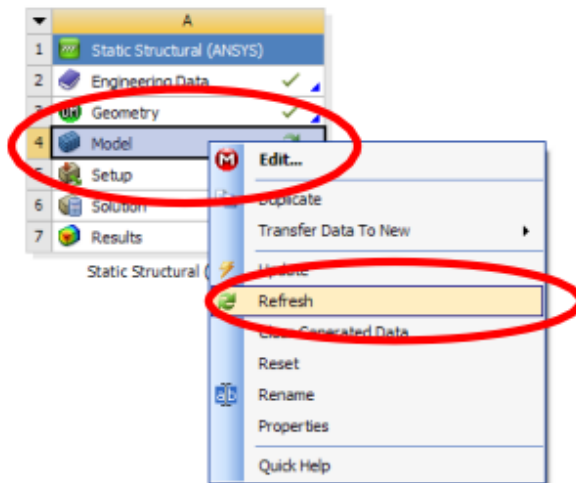


Figure 1.38 importing new geometry

You should have a lightning sign next to model. Double on LMC to open the simulation module.

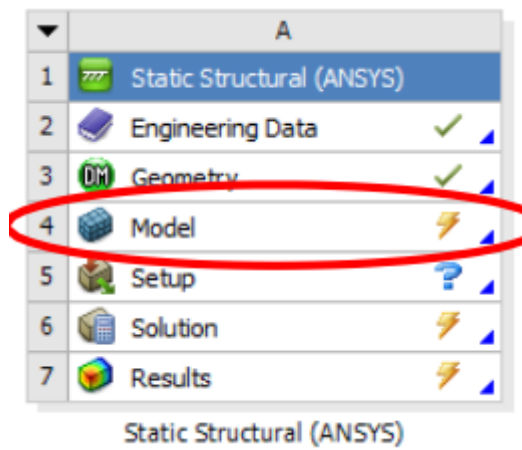


Figure 1.39 entering model cell

### 2.3.3. Meshing of the model

LMC on Mesh, LMC on Mesh Control and On Sizing.

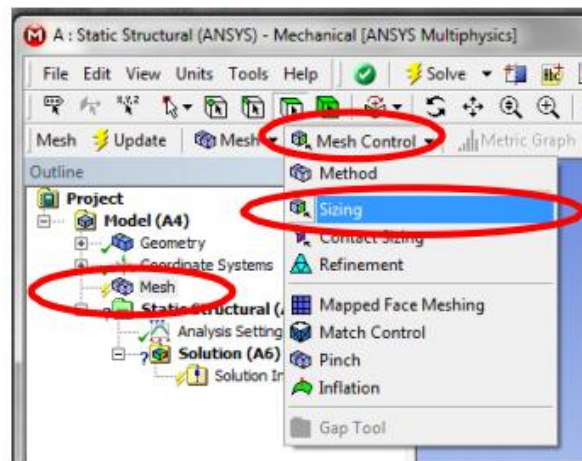


Figure 1.40 mesh sizing

Details of Sizing should appear on the left bottom.

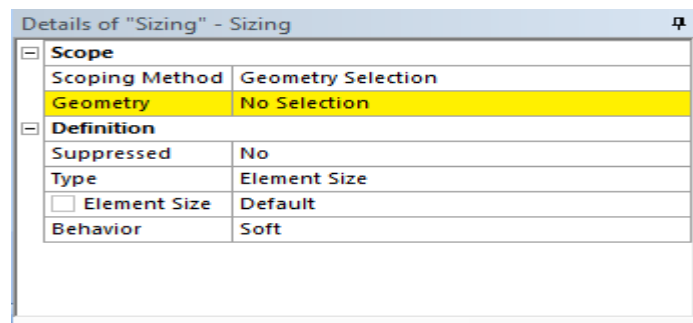


Figure 1.41 sizing details

Now select the line using the selection tool (Edge) available in the menu. Then select the line body

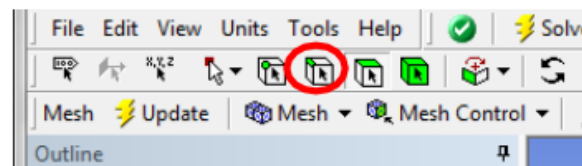
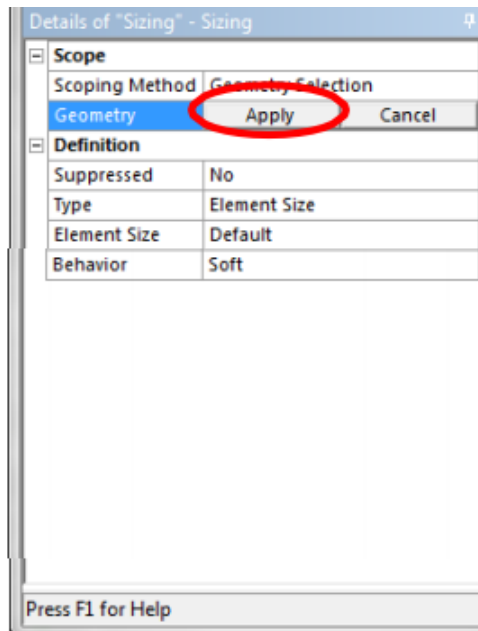


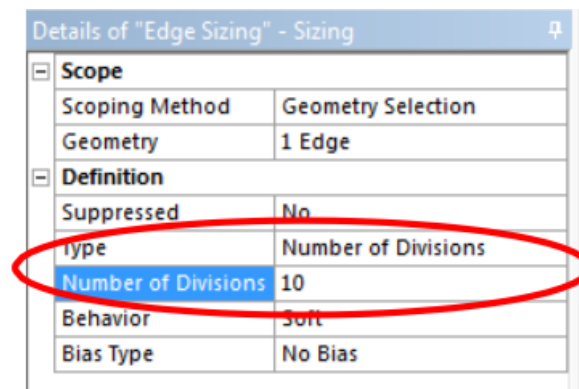
Figure 1.42 edge selection

Afterwards click on Apply.



**Figure 1.43** applying mesh sizing

Select Number of Divisions in type of sizing and enter 10 as Number of Divisions. And that is to devise the selected line into 10 elements.



**Figure 1.44** number of mesh divisions

Now you should obtain as the following figure in Geometry window.



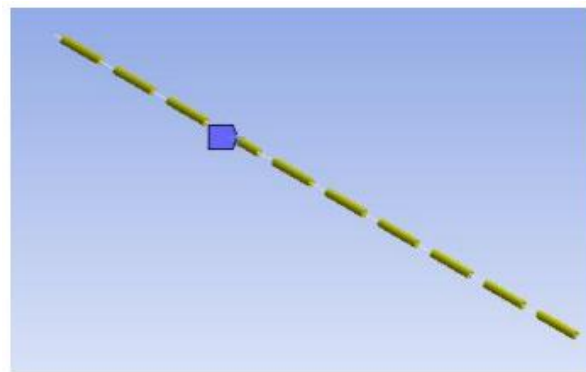


Figure 1.45 mesh in geometry window

RMC on Mesh and LMC on Generate Mesh.

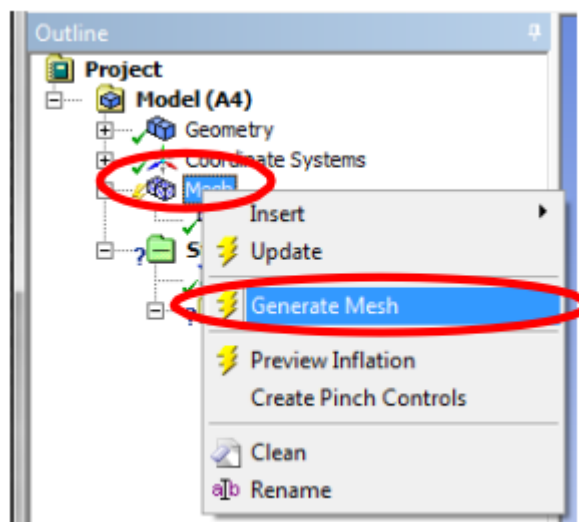


Figure 1.46 generating mesh

The Meshing of the beam should be displayed in Geometry window.

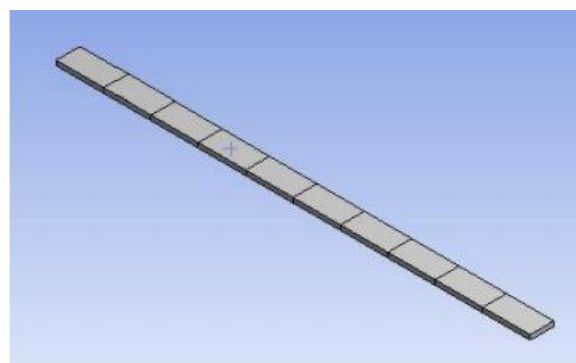
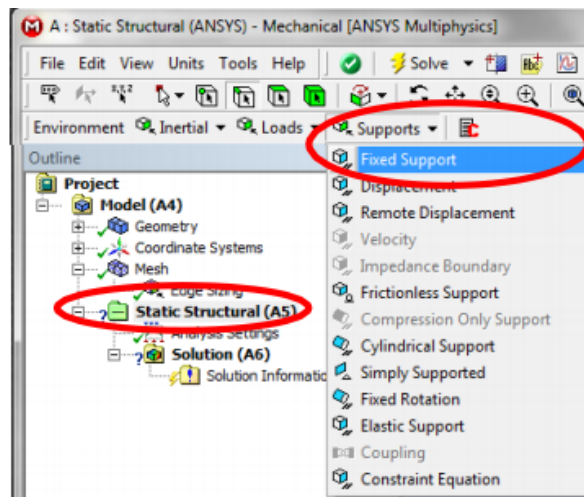


Figure 1.47 meshing of the beam

**2.3.4. Boundary conditions**

LMC on Static Structural, LMC on Supports and LMC on Fixed Support.



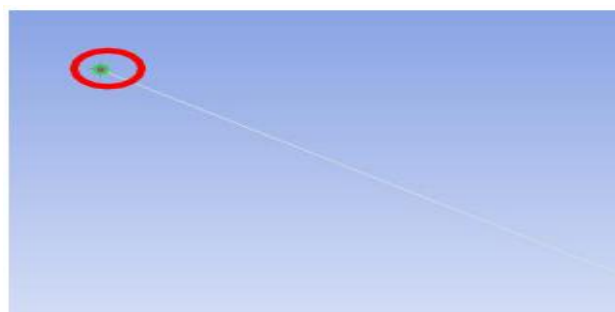
**Figure 1.48** adding a fixed support

You should now select the top built-in the beam using the selection tool Vertex available in the menu



**Figure 1.49** vertex selection

LMC on the top at the right, until Selection this turns green.



**Figure 1.50** selected point

LMC on apply in Details of Fixed Support and you should have the following.

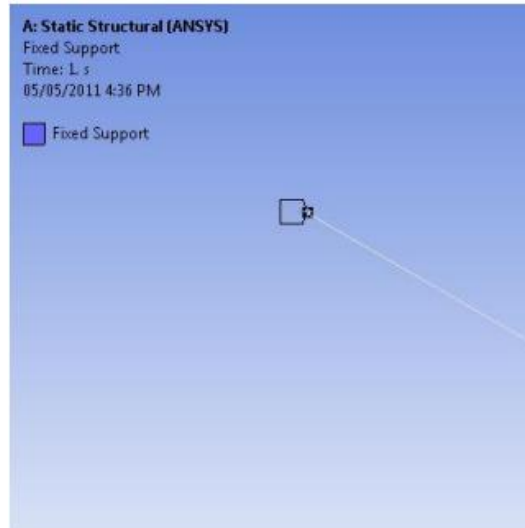


Figure 1.51 fixed support

Select the line, LMC on Inertial and LMC on Standard Gravity.

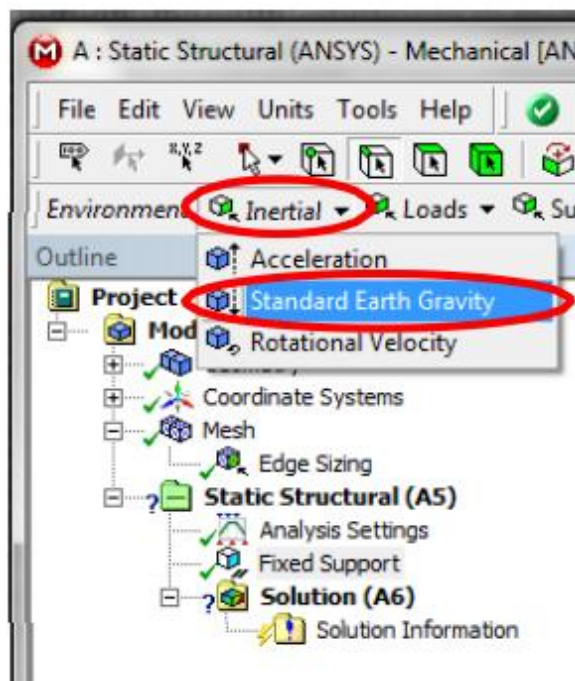


Figure 1.52 adding inertial (standard earth gravity)

In Details of Standard Earth Gravity, change the direction of – Y Direction (3). A yellow check mark should be displayed in the gravity of the geometry.

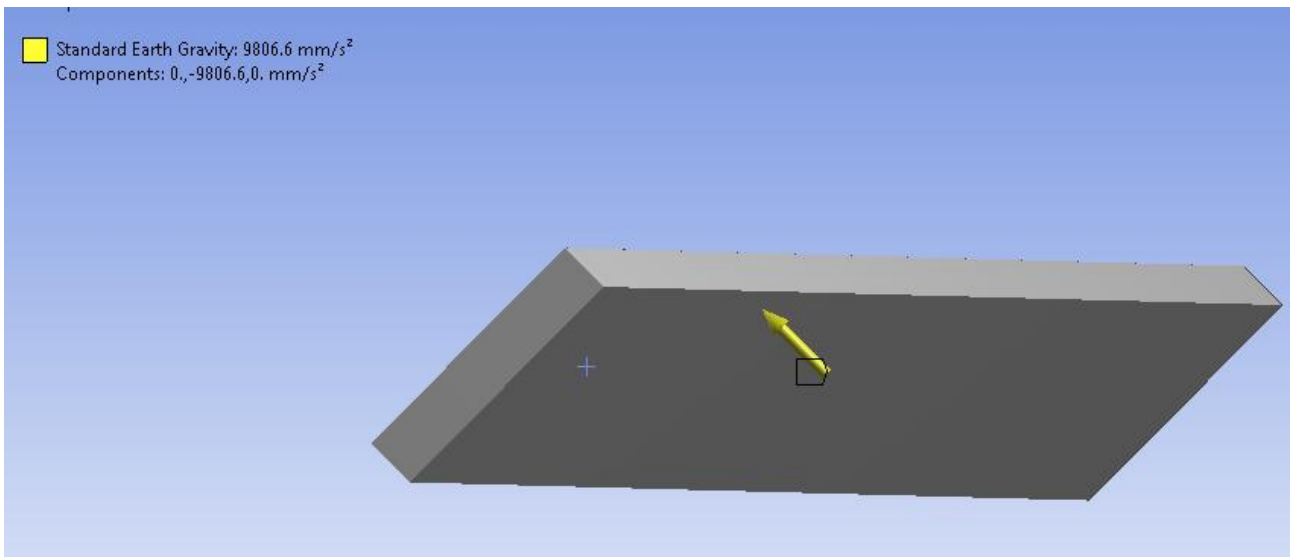


Figure 1.53 beam with standard earth gravity

### 2.3.5. Model Resolution

To solve the model, RMC on Solution and LMC on Solve.

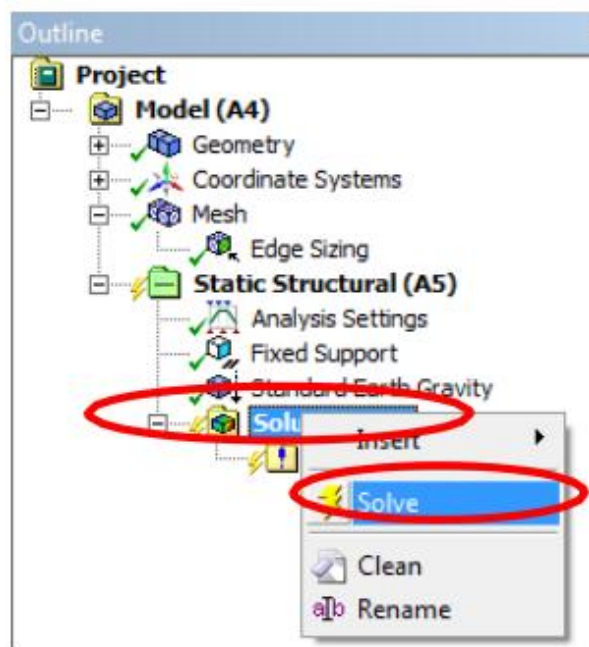


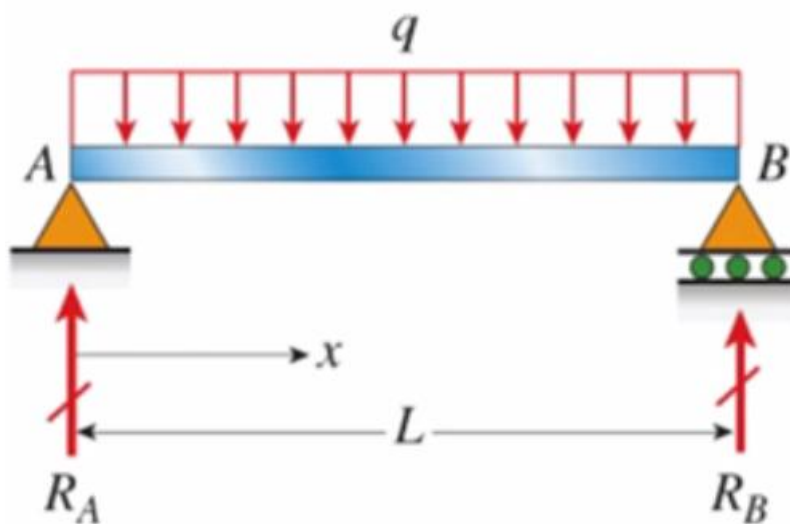
Figure 1.54 solving the solution

## 1. Introduction

In our study case, we are modeling a simply supported beam with a distributed load in the first part, and the same beam with a center load in the second part showing the differences between these two cases in several options; stress normal, stress equivalent and directional deformation .

## 2. Distributed load (case 1)

### 2.1. Beam characteristics



**Figure 2.1:** simply supported beam with distributed load

We have a fixed support at point (A), and a Hench with roller at point (B), with a uniformly distributed load across the beam. For this case we're going to use beam length of one meter, and for the load we used 5 N/mm. we use steel in this case so, Youngs modulus 210 Gpa. For the section we using; it's 40 mm by 40 mm.

Moment of inertia is going to be calculated in the formula for cross-section:  $I = 1/12 \cdot a^4$

Section modulus Z is going to be calculated by the formula:  $Z = a^3/6$ .

$L = 1000 \text{ mm}$

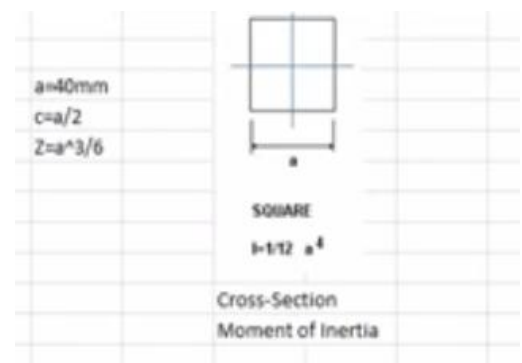
Line pressure load on beam ( $q$ ) = 5 N/mm

Young's modulus E: 210000 N/mm<sup>2</sup>

Distance from neutral axis to extreme fibers  $c = 20 \text{ mm}$

Moment of inertia;  $I = 213333.\text{mm}^4$

Section modulus  $Z = 10667 \text{ mm}^3$



**Figure 2.2:** beam cross section

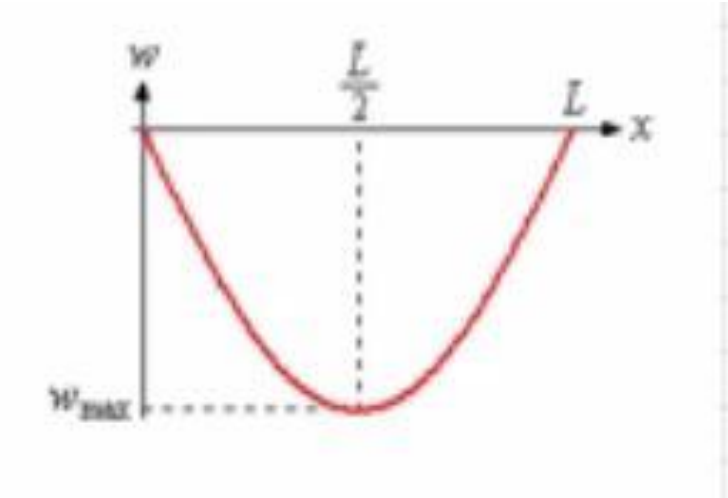
**Hand calculation solutions:**

**Displacement:**

$$W(x) = -q (L^3 - 2x^2L + x^3) / 24 EI$$

$$W_{\max} = W(L/2) = -1.453\text{mm}$$

**Figure 2.3** displacement 1



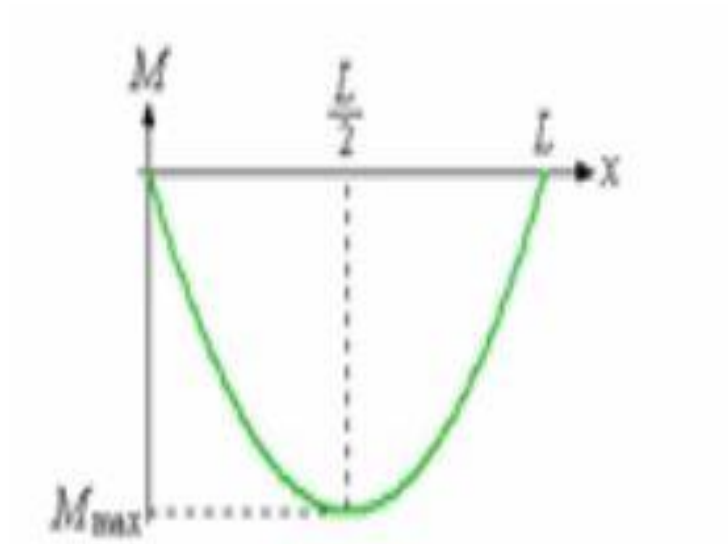
**Moment :**

$$M(x) = -\frac{1}{2} q (L - x) x$$

$$M_{\max} = M(L/2) = -qL^2/8$$

$$\sigma_{\max} = 58.59 \text{ Mpa}$$

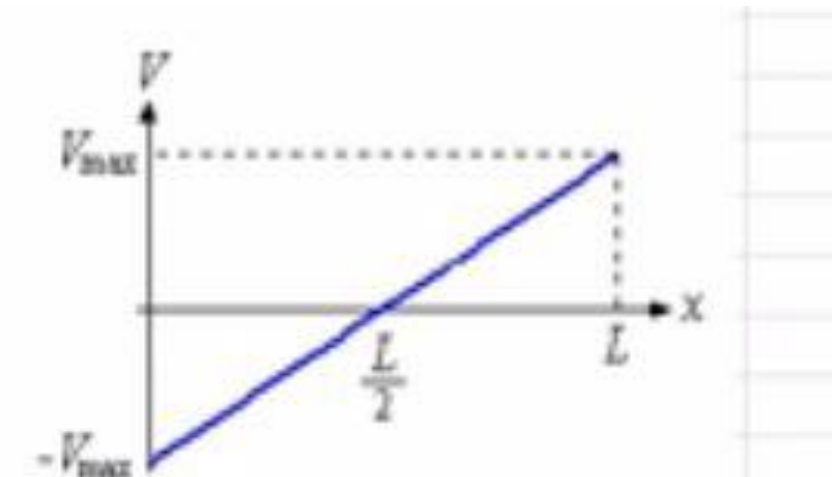
**Figure 2.4** moment 1



**Shear:**

$$V(x) = -\frac{1}{2} q (L - 2x)$$

$$V_{\max} = V(0 \ \& \ L) = -2500 \text{ N}$$



---

**Figure**

**2.5**

shear

1

## 2.2. Modeling

### 2.2.1. Engineering data

After opening the ANSYS workbench interface page, drop a static structural from toolbox to the project schematic area, and renaming the project. Enter the engineering data cell and specify the young’s modulus as the case that we have.

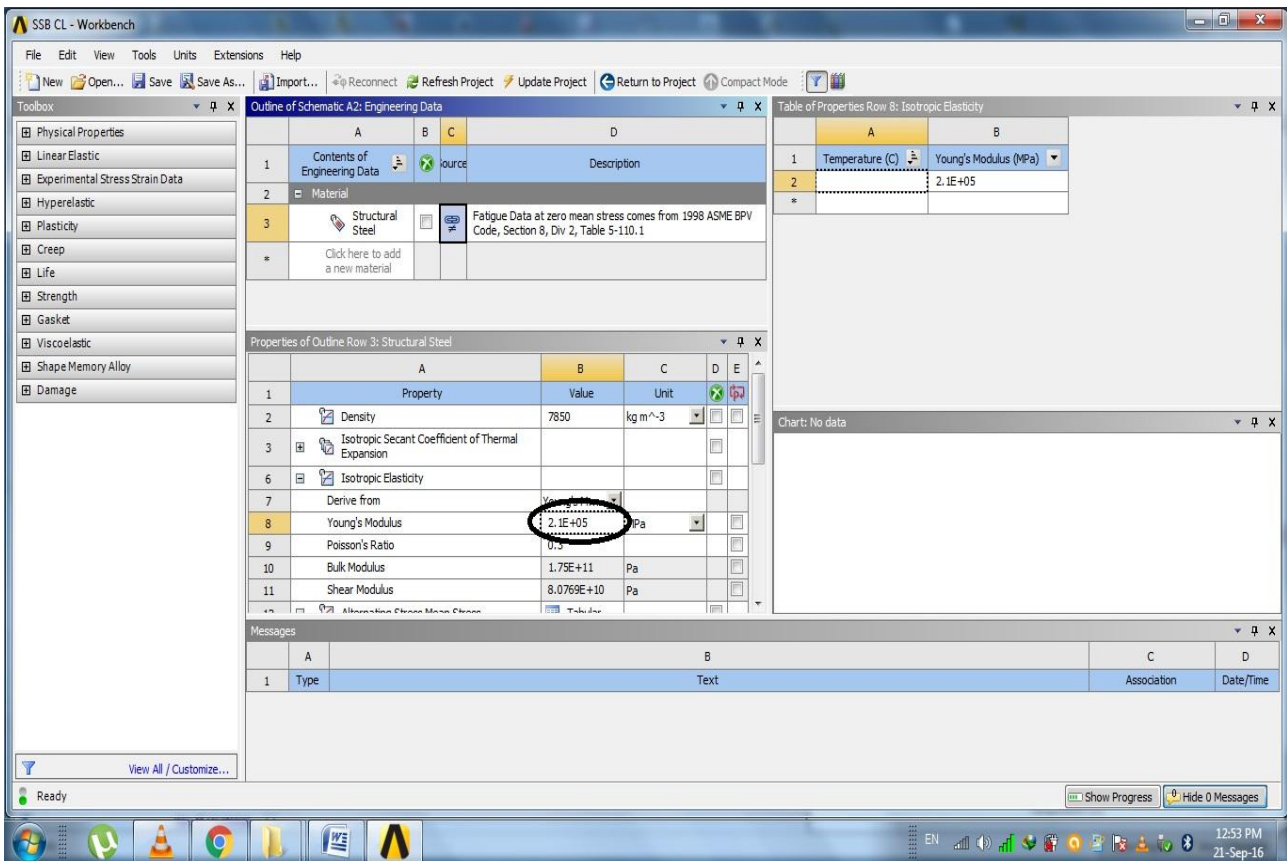


Figure 2.6 engineering data properties



### 2.2.2. Geometry

In this case we are doing the full 3D model, by modeling a surface then extruding it

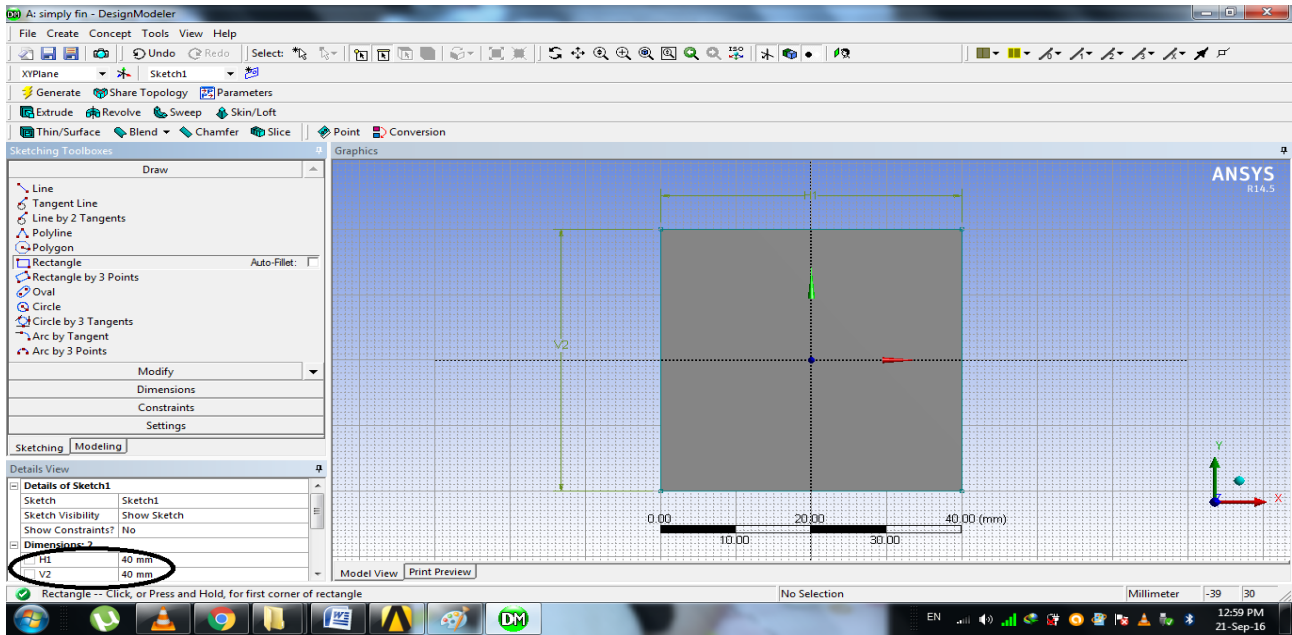


Figure 2.7 creating a surface

After extruding the surface to the exact dimensions we have the beam done.

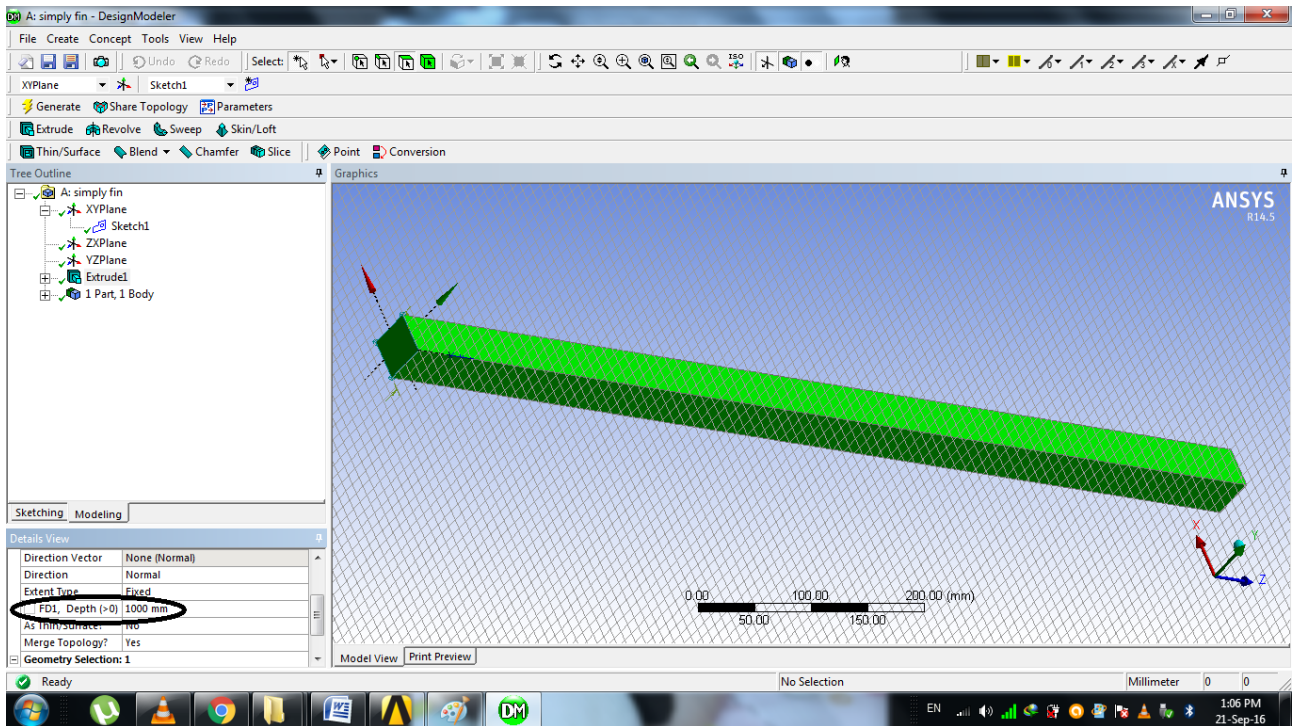


Figure 2.8 full 3D beam

### 2.2.3. Model/mesh

Defining and generating mesh by modify the sizing to 20 mm

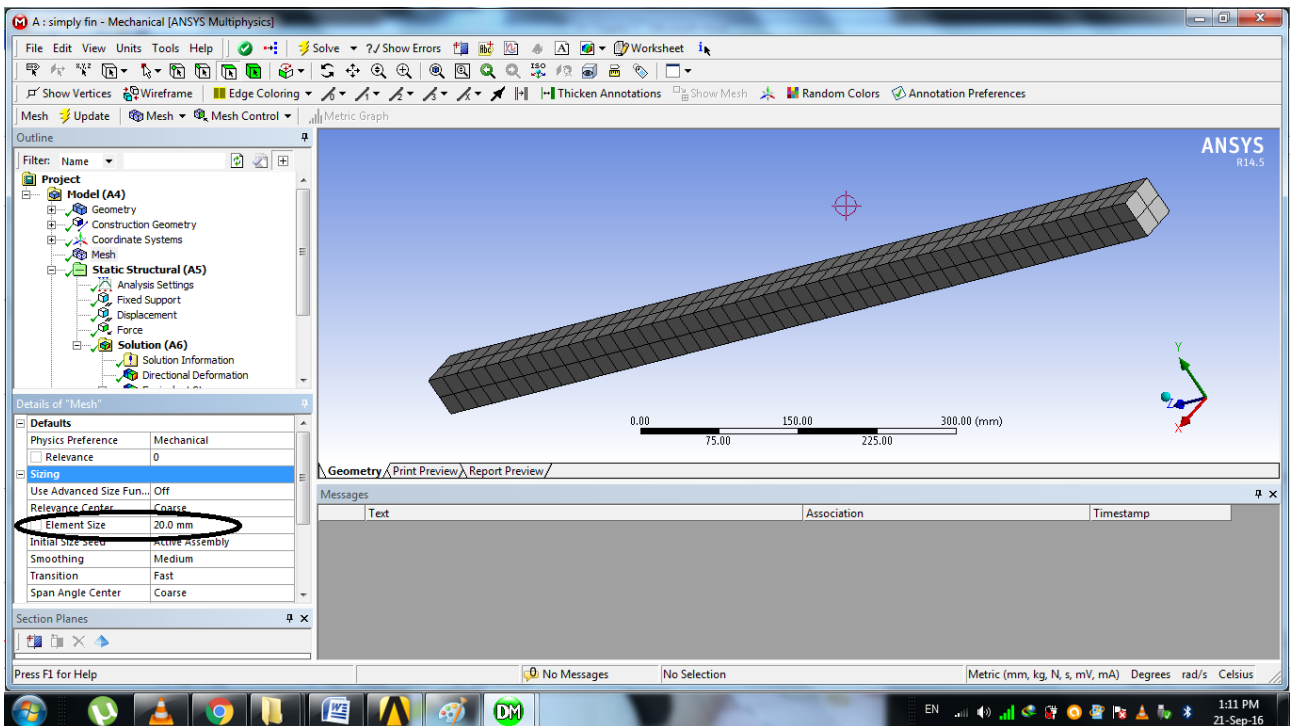


Figure 2.9 mesh sizing

After applying our Boundary conditions for this case

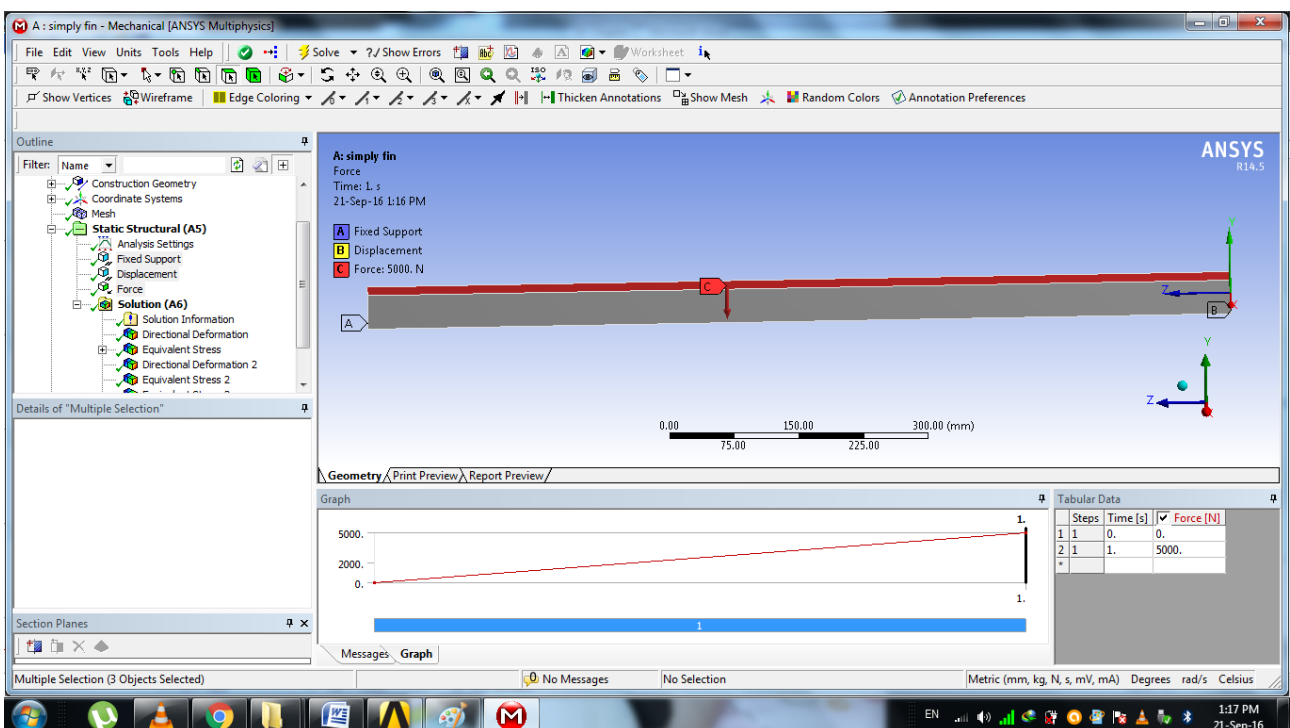


Figure 2.10 boundary conditions

### 2.2.4. Setup and solution

Adding our analysis: in this case the equivalent stress normal stress and directional deformation. Under the equivalent stress, we're going to add a convergence that basically gives us the accuracy to any percent I want which I have specified it with 3 percent.

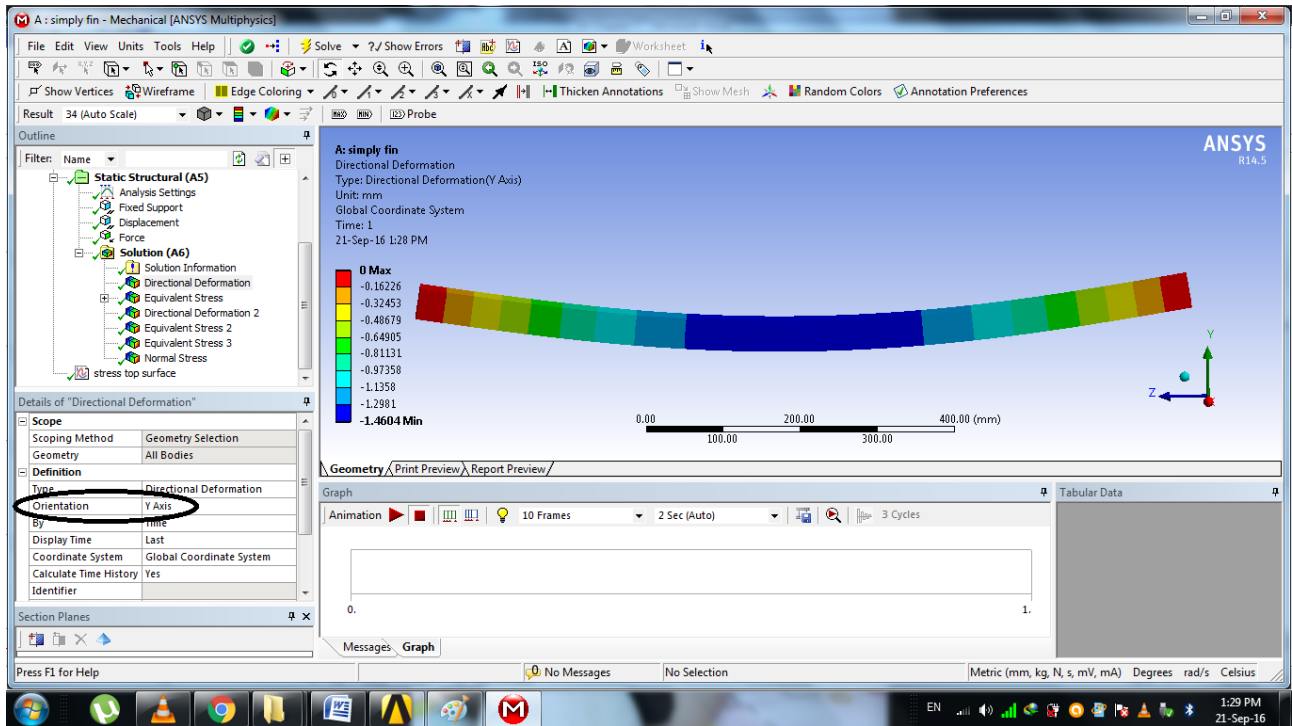


Figure 2.11 directional deformation

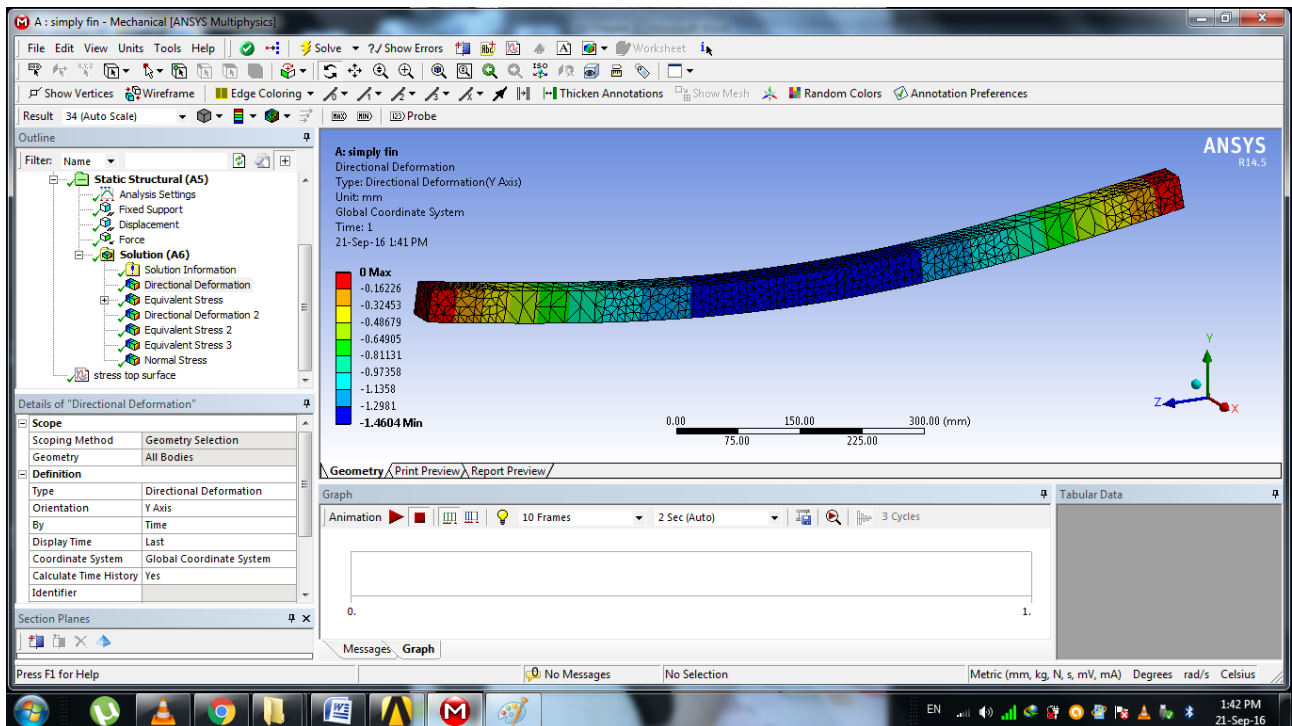


Figure 2.12 directional deformation with showing elements

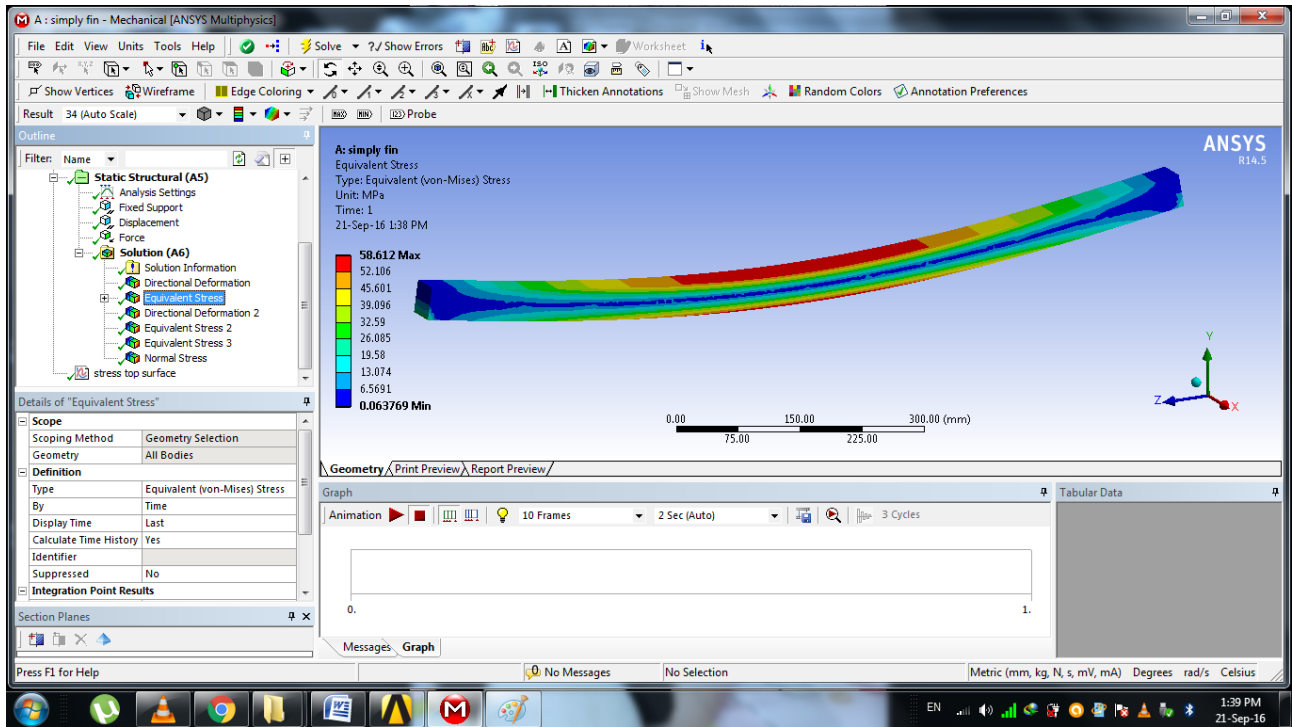


Figure 2.13 equivalent stress

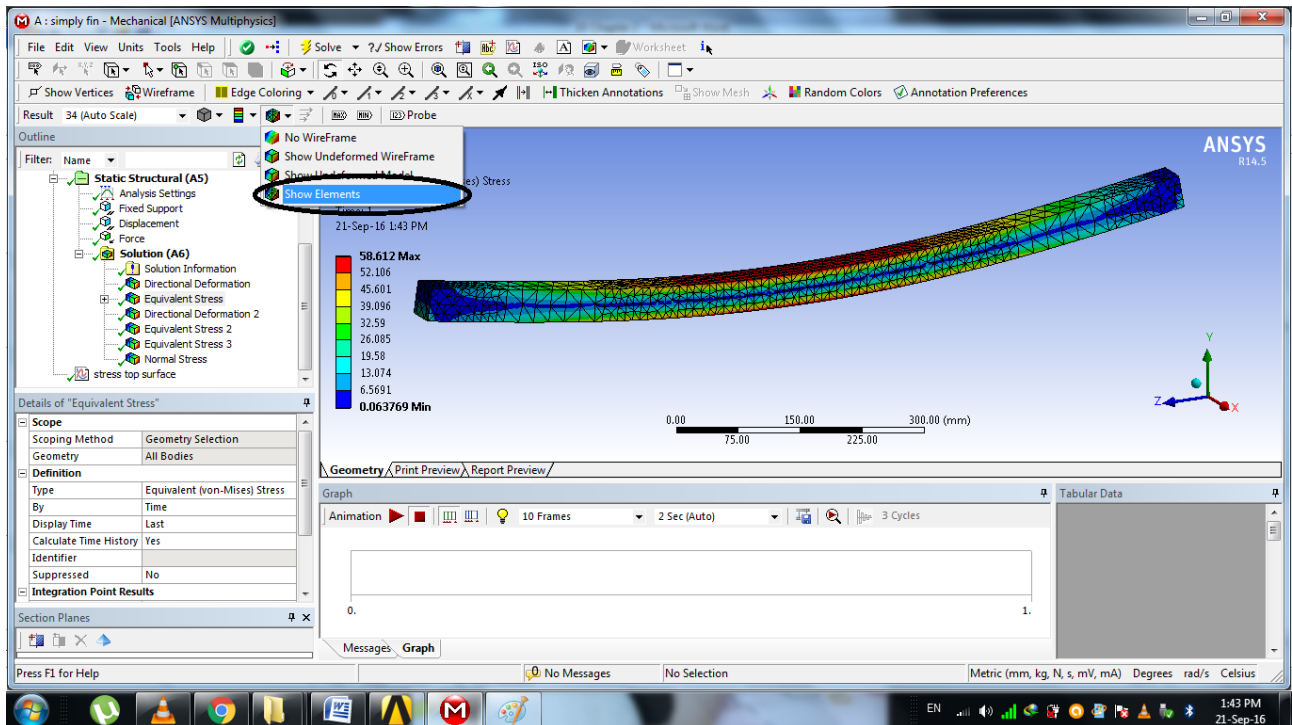
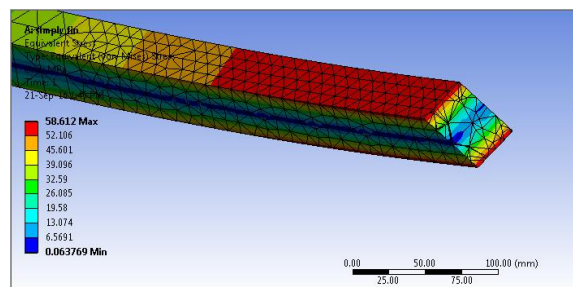


Figure 2.14 equivalent stresses with showing mesh

Figure 2.15 cross section showing stress





### 2.2.5. Results

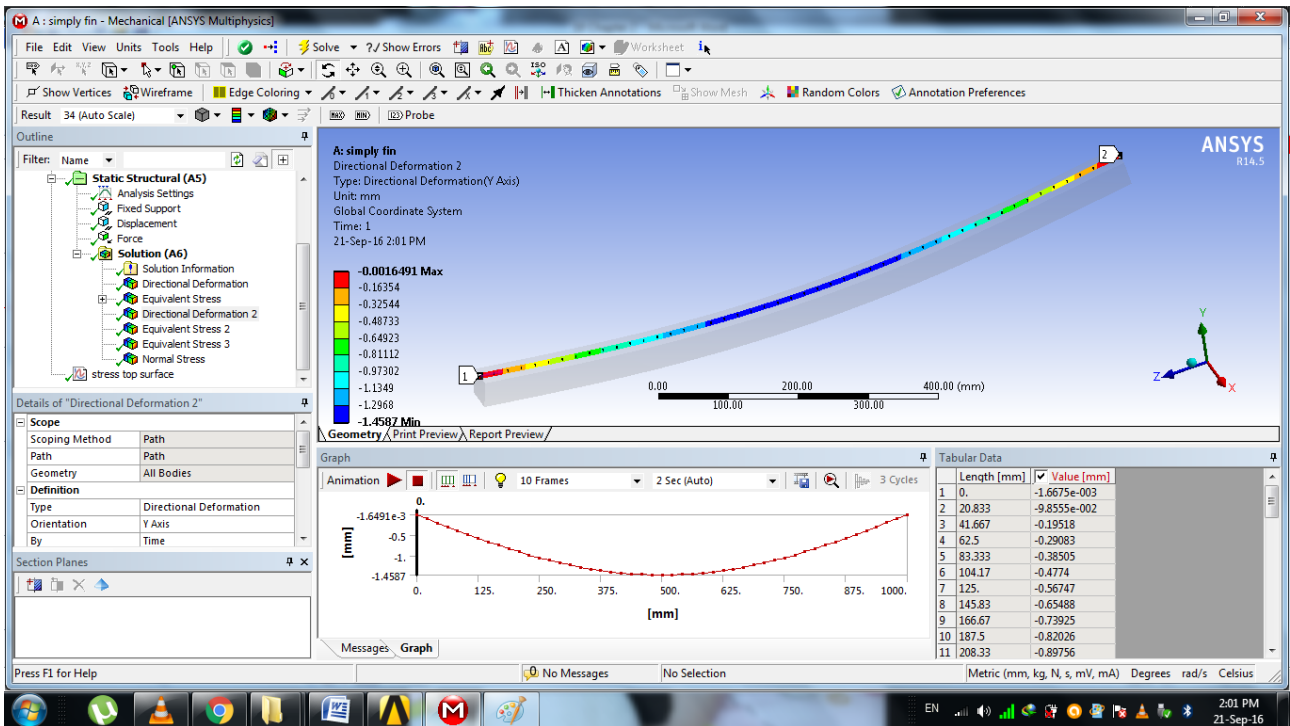


Figure 2.16 directional deformation of the top surface

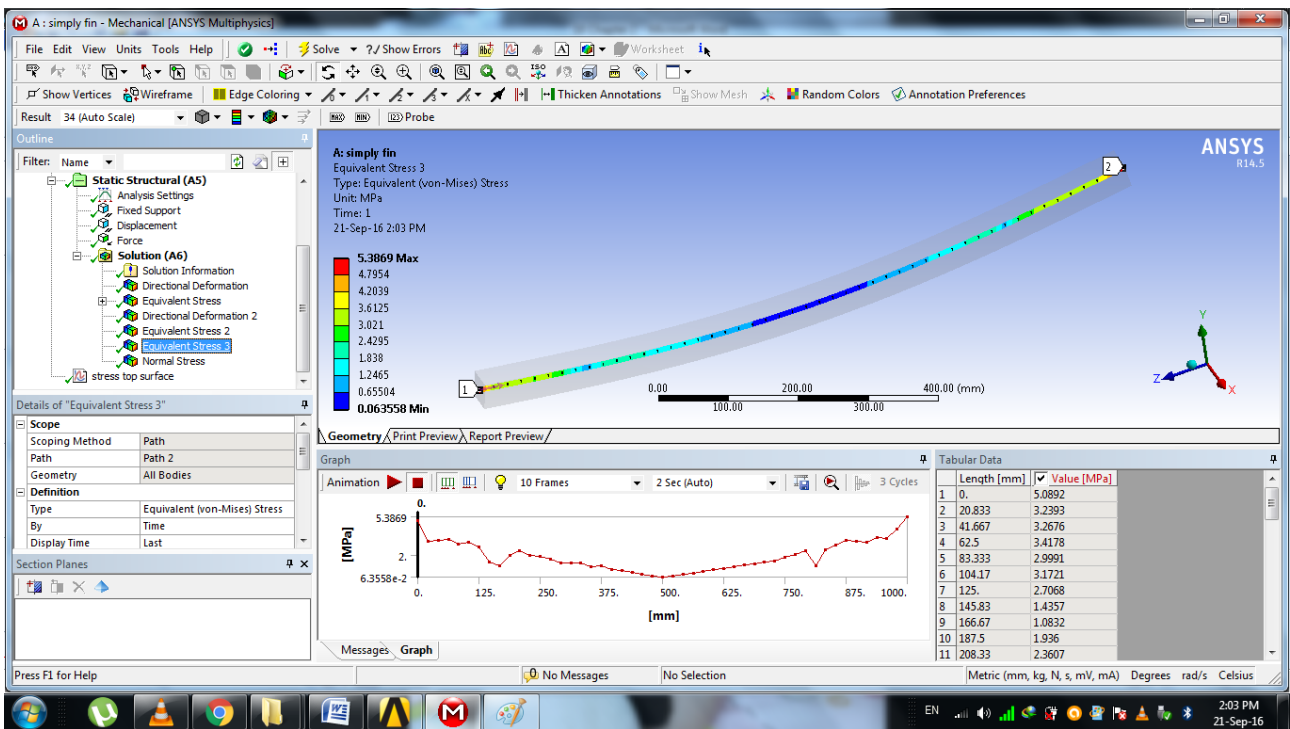


Figure 2.17 equivalent stress of the center of the section

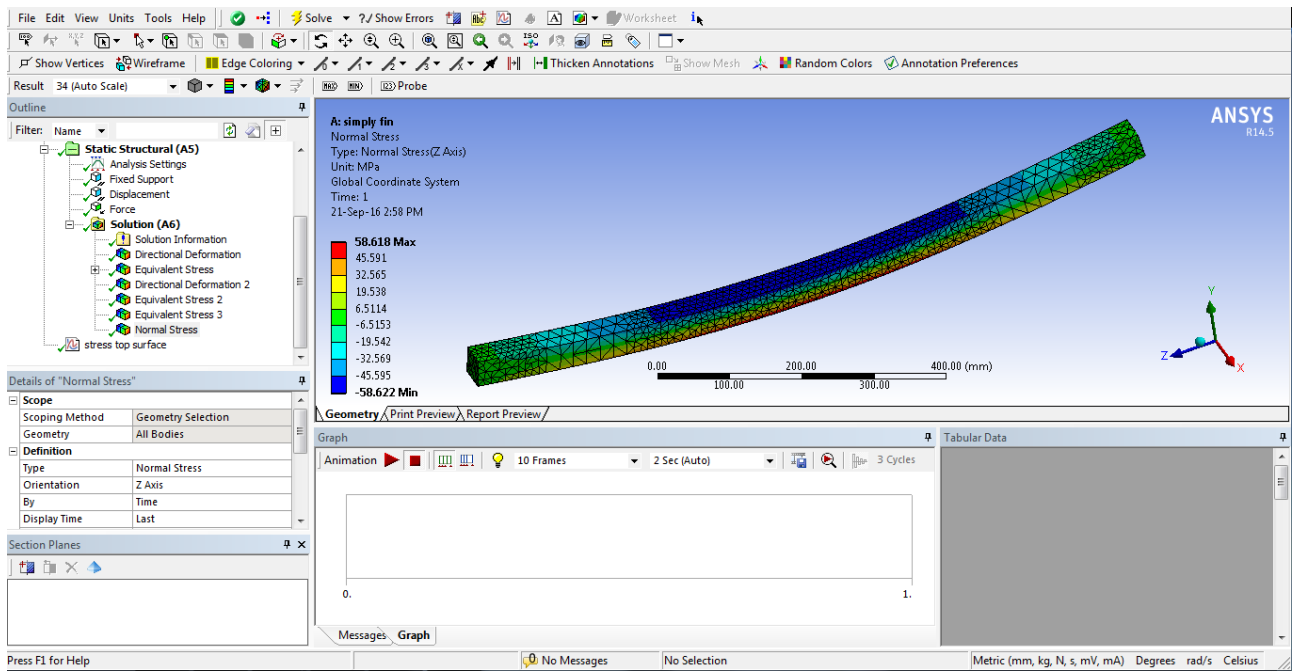
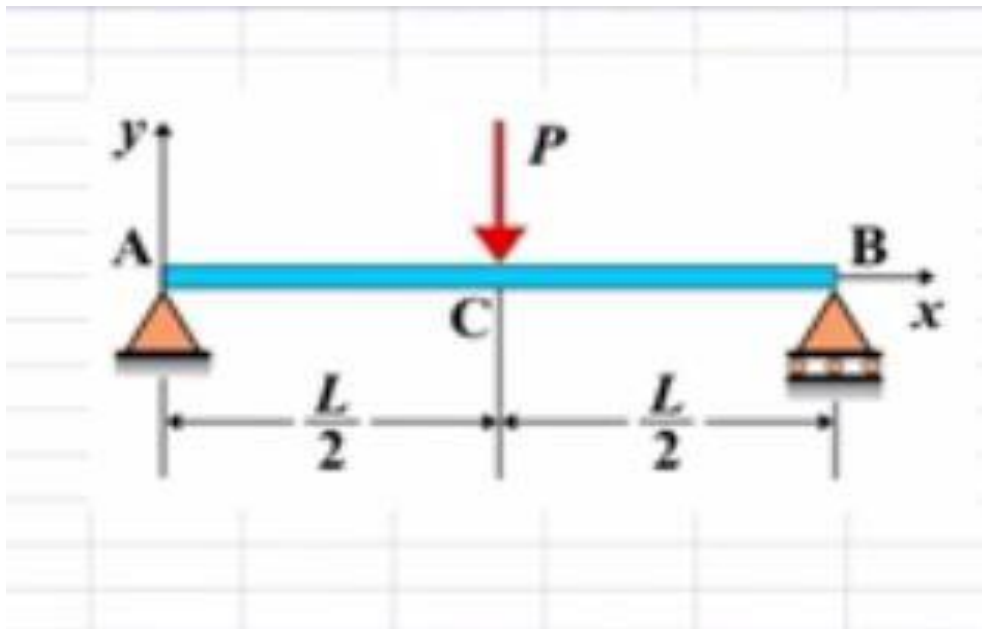


Figure 2.18 normal stress showing the mesh

### 3. Center load

#### 3.1. Beam characteristic:



**Figure 2.19** simply supported beam with center load

We have the same beam characteristics just like the beam with distributed load in case (1), but this time it's loaded with a center load in the middle of the beam.

$L = 1000 \text{ mm}$

Load on beam ( $q$ ) = 5000 N

Young's modulus  $E$ : 210000 N/mm<sup>2</sup>

Distance from neutral axis to extreme fibers  $c = 20 \text{ mm}$

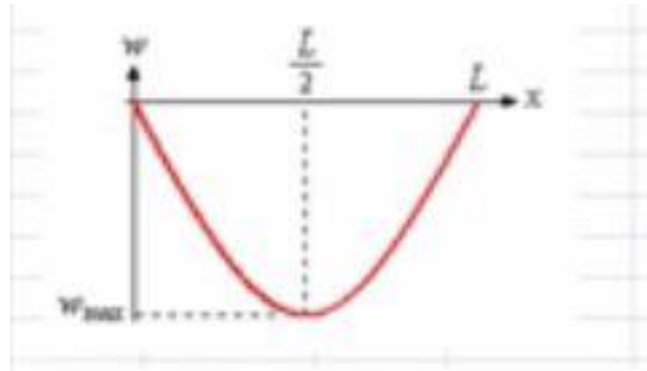
Moment of inertia;  $I = 213333 \text{ mm}^4$

Section modulus  $Z = 10667 \text{ mm}^3$

**Hand calculation solutions:**

**Displacement**

$$W_{\max} = W(L/2) = PL^3 / 48 EI = -2.33 \text{ mm}$$

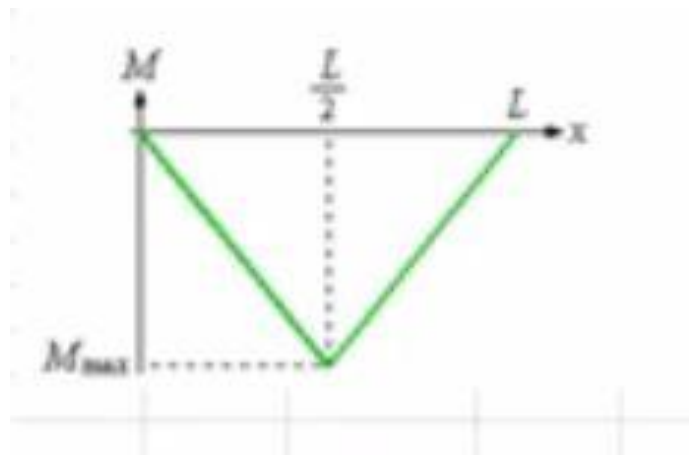


**Figure 2.20** displacement 2

**Moment**

$$M_{\max} = M(L/2) = - PL/4$$

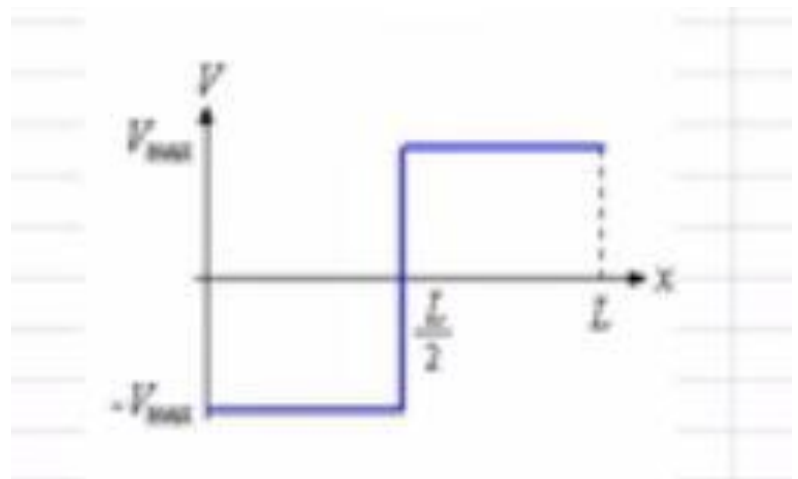
$$\sigma_{\max} = 117 \text{ Mpa}$$



**Figure 2.21** moment 2

**Shear**

$$V_{\max} = V(x) = \pm P/2 = -2500 \text{ N}$$



**Figure 2.22** shear 2



### 3.2. Modeling

In order to add a center load to the beam, we have to split the face because ANSYS needs a line or an edge to apply the load.

#### 3.2.1. Geometry

Selecting the face we want to split then add a new plane on the top face and insert a sketch projection, which will extract all the edges in this face. Then, with modifying the new sketch with the offset tool we can draw a line on the top face splitting it in the middle.

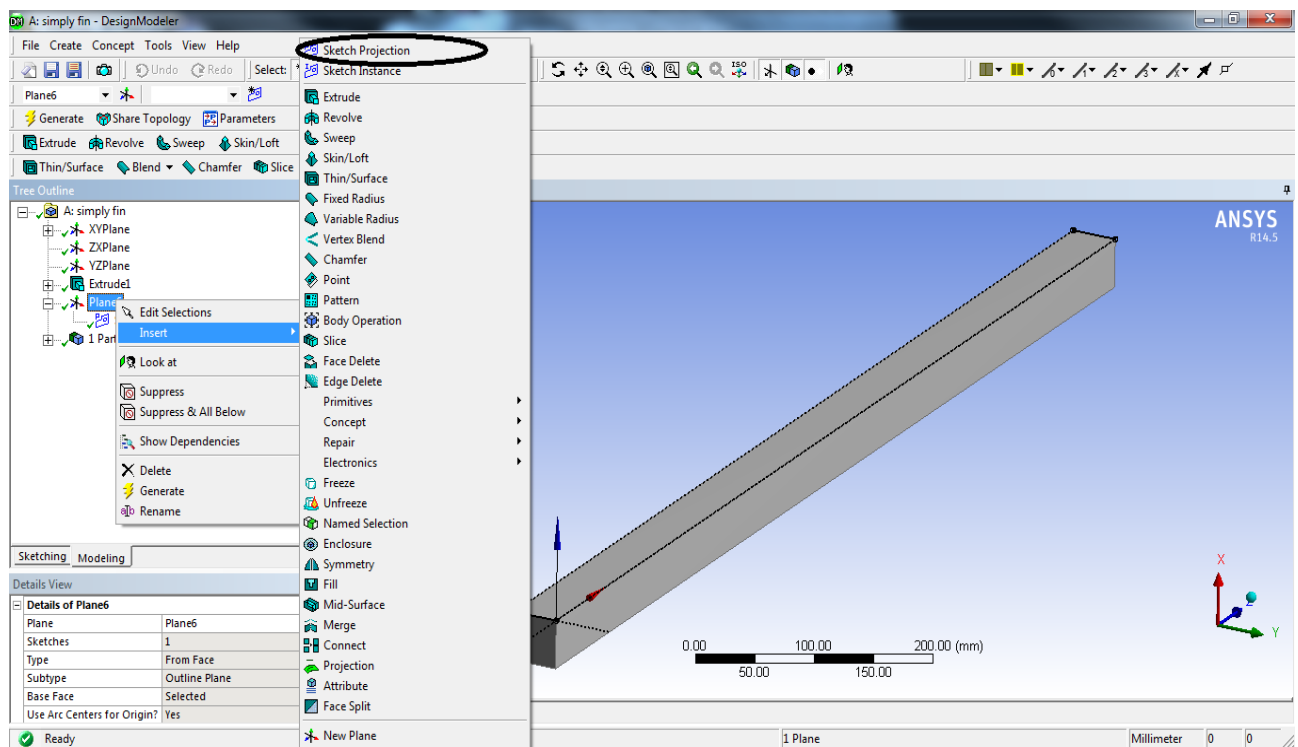


Figure 2.23 sketch projection

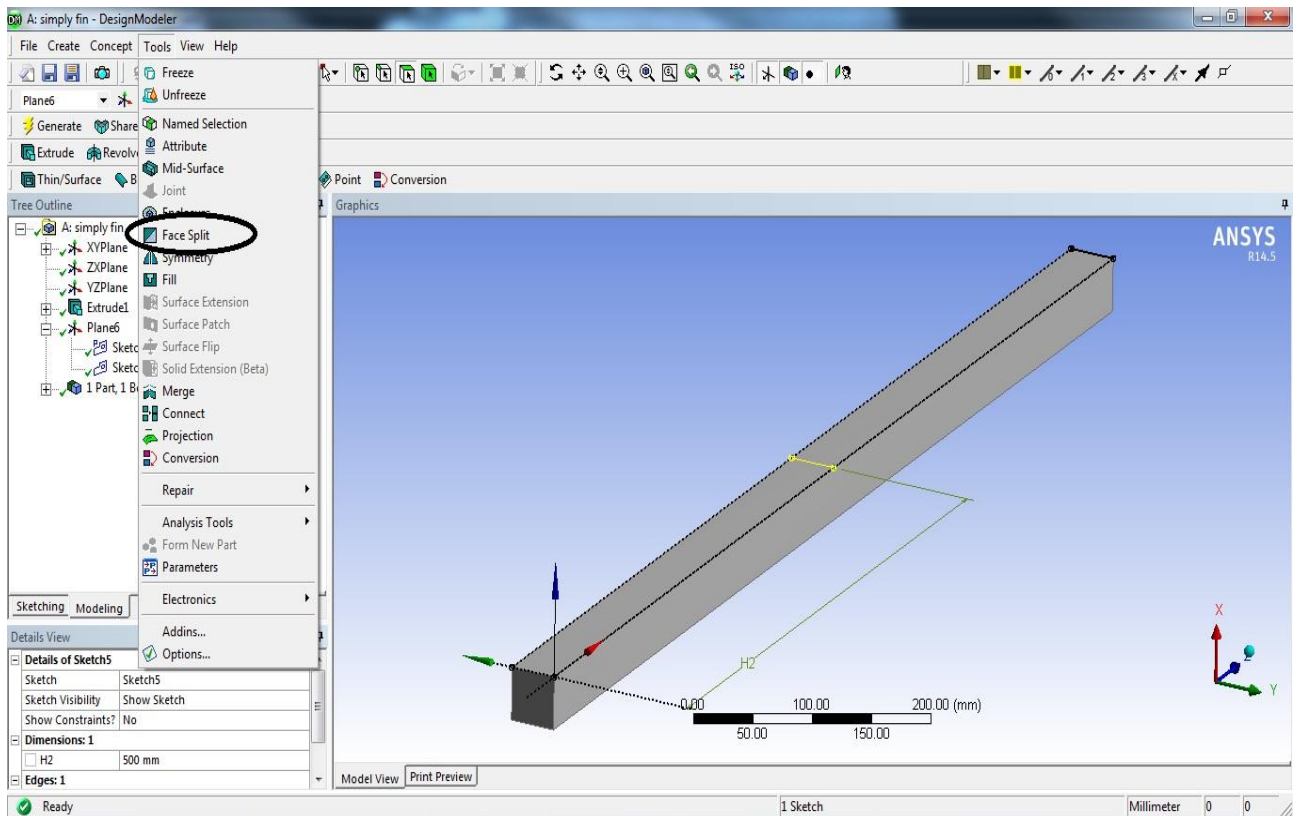


Figure 2.24 face split step

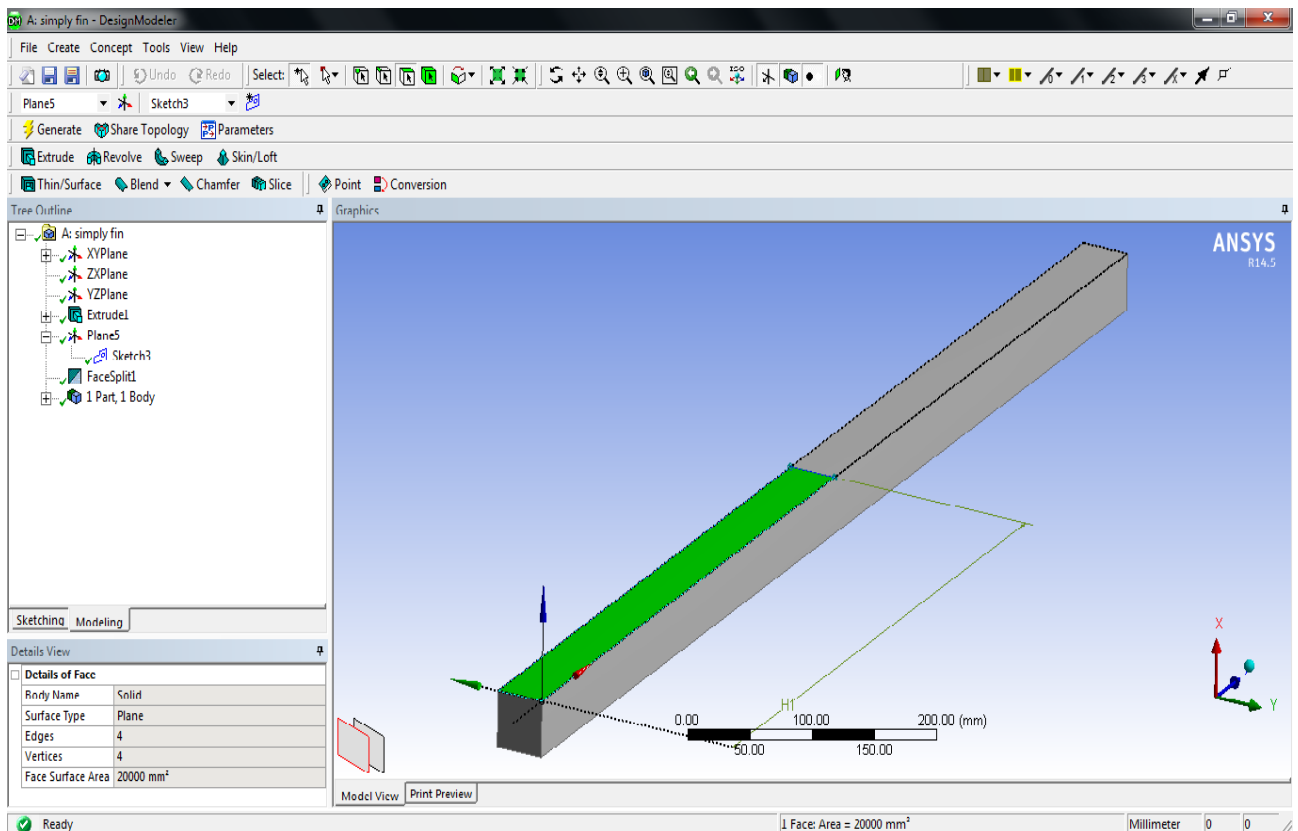


Figure 2.25 top faces after splitting

3.2.2. Model

After splitting the top surface we have to redefine our force in the model cell under the force at the tree outline and put it in the right place which is in the middle of the beam.

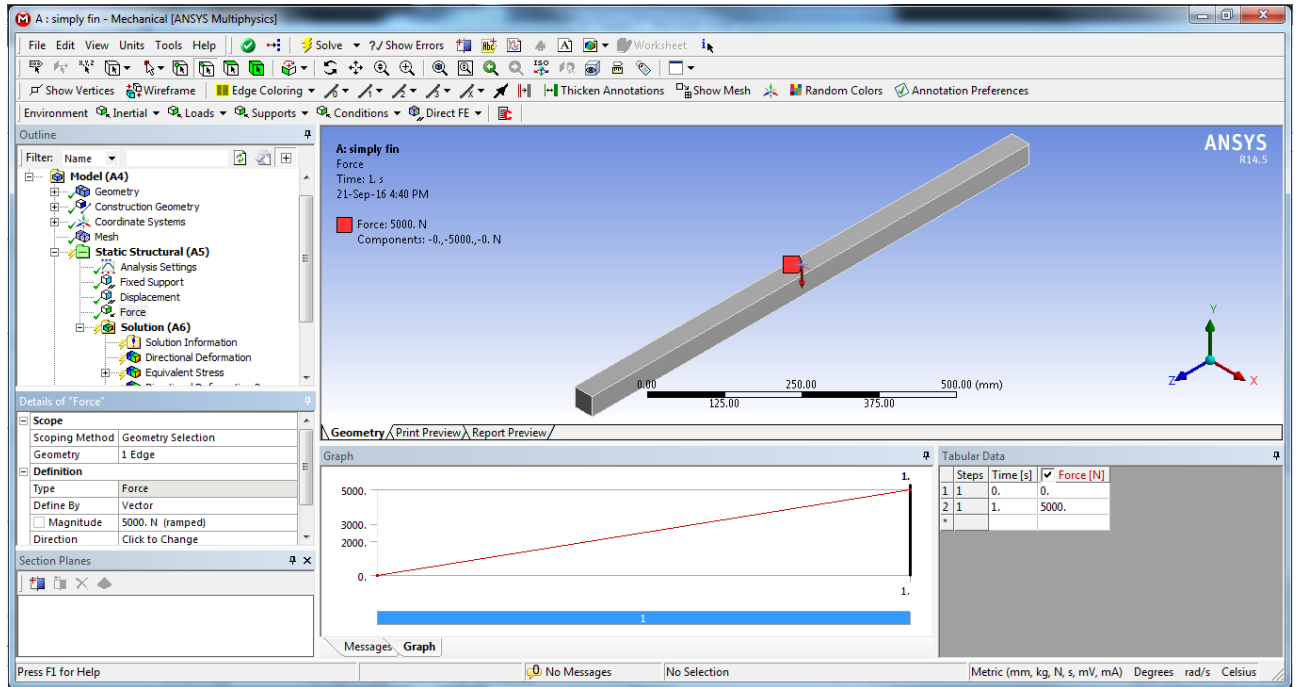
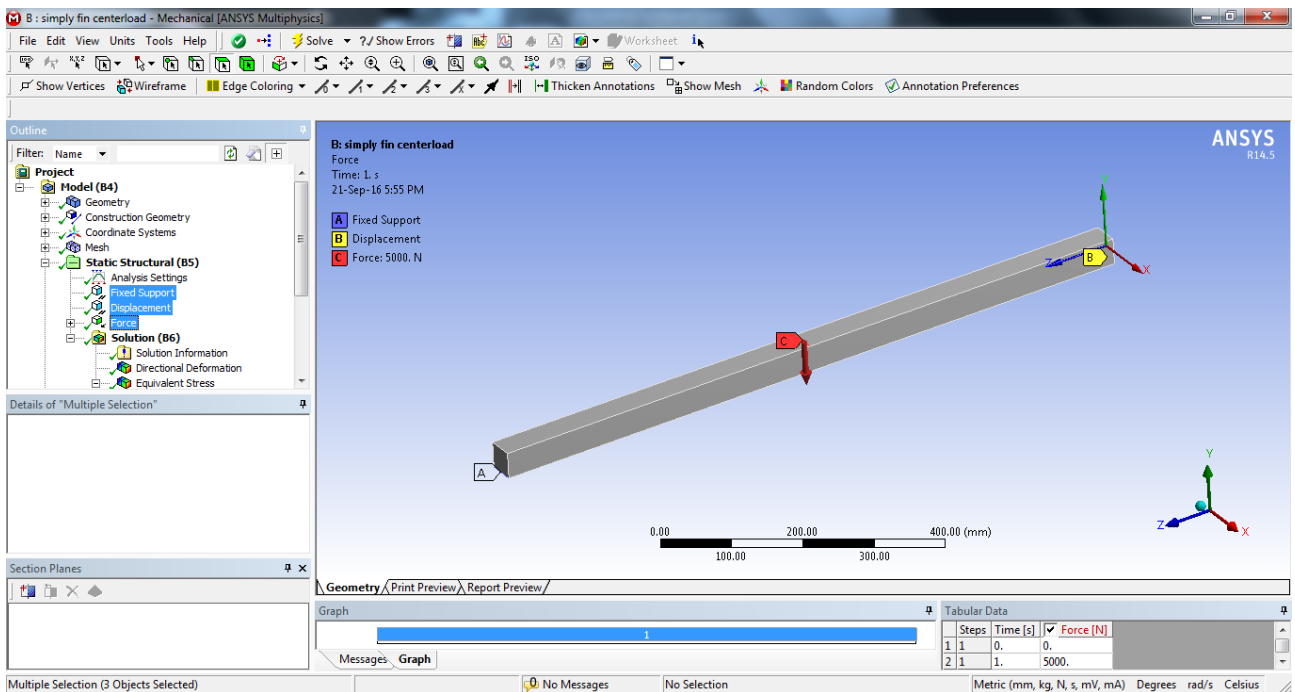


Figure 2.26 center load force adding



---

**Figure 2.27** boundary conditions

### 3.2.3. Setup and solution

With the center load and solving the project to refine the mesh and showing the results of the analysis we used an allowable convergence of 3% and the results came in this order:

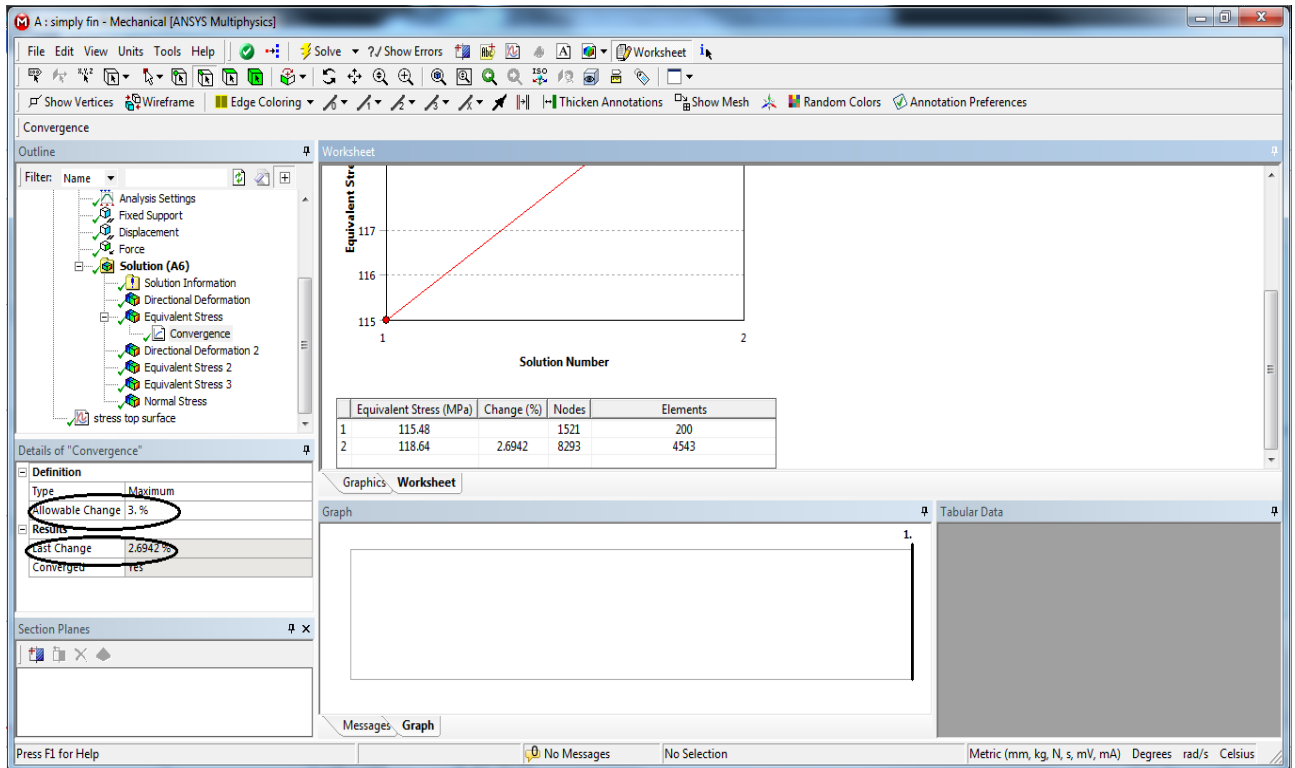


Figure 2.28 convergence percentage and results of change

3.2.4. Results

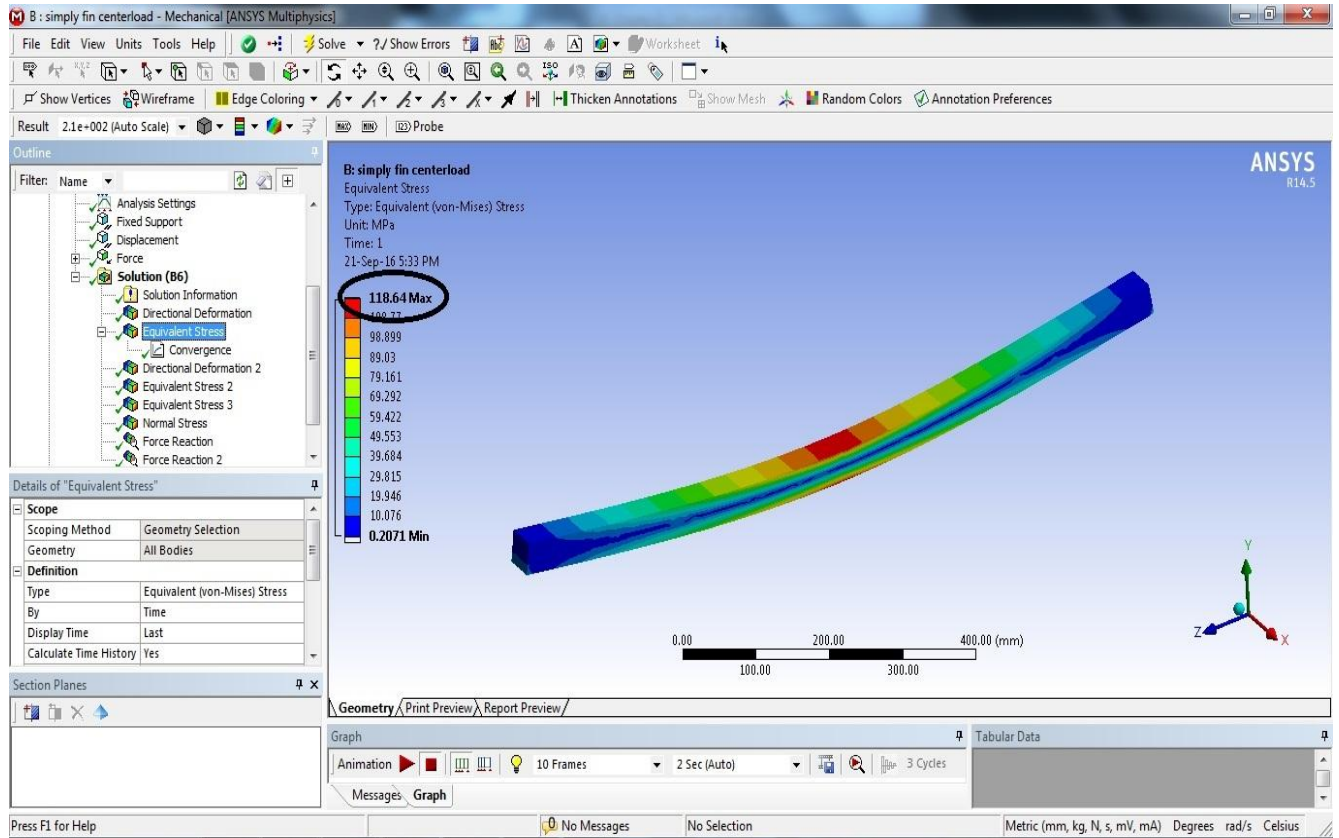


Figure 2.29 equivalent stress

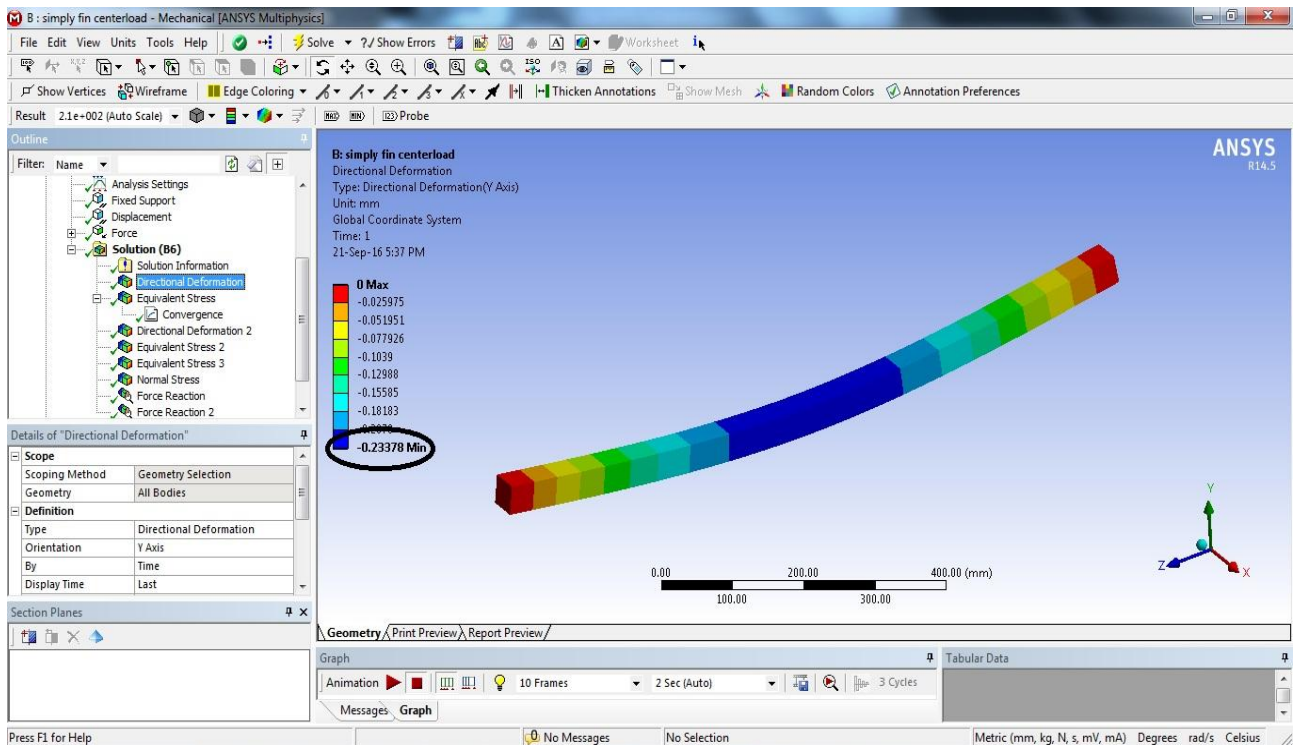


Figure 2.30 directional deformation

Force reactions at the supports:

Fixed support:

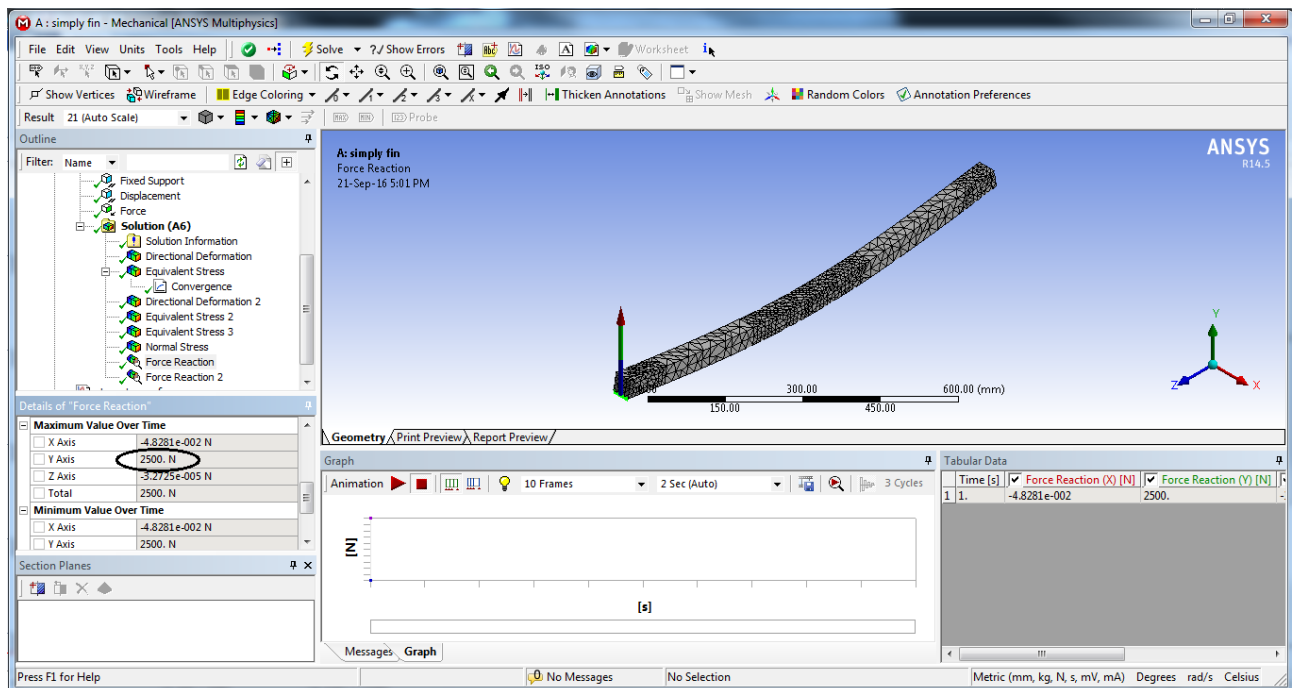


Figure 2.31 force reaction at the fixed support

At the rollers support:

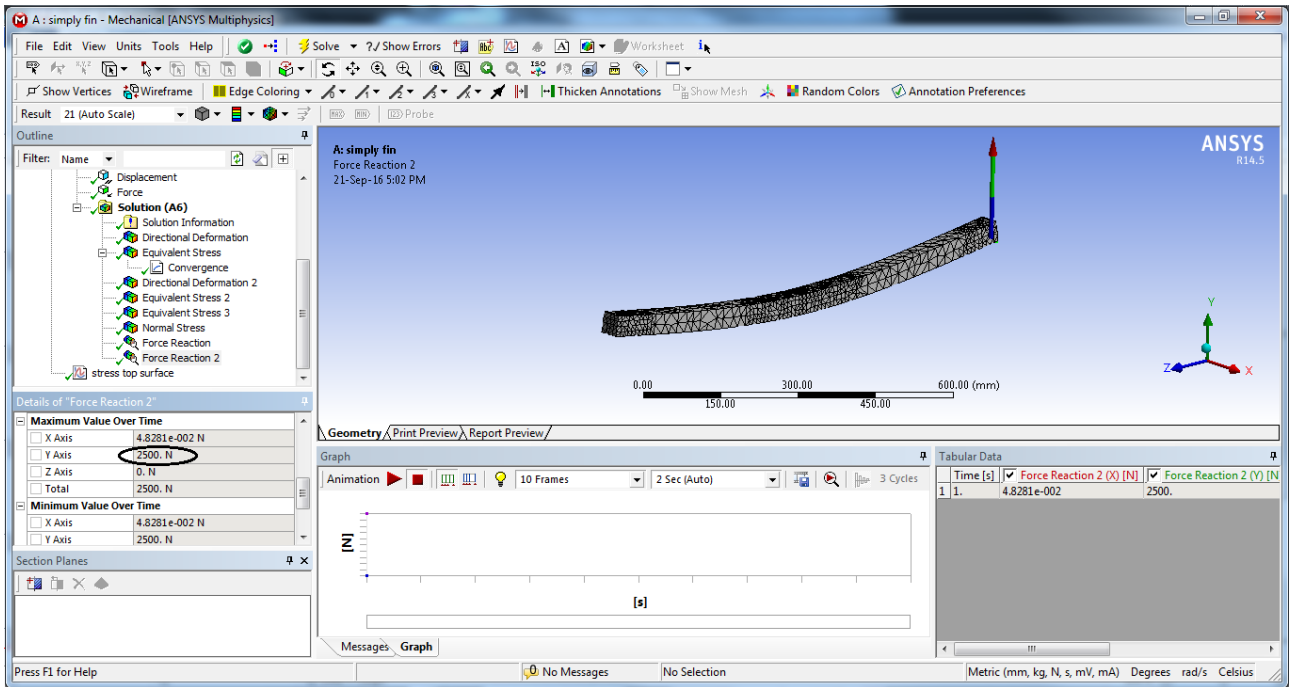


Figure 2.32 force reaction at the displacement

After refining the mesh; 2.6 % the percentage of mesh changing.

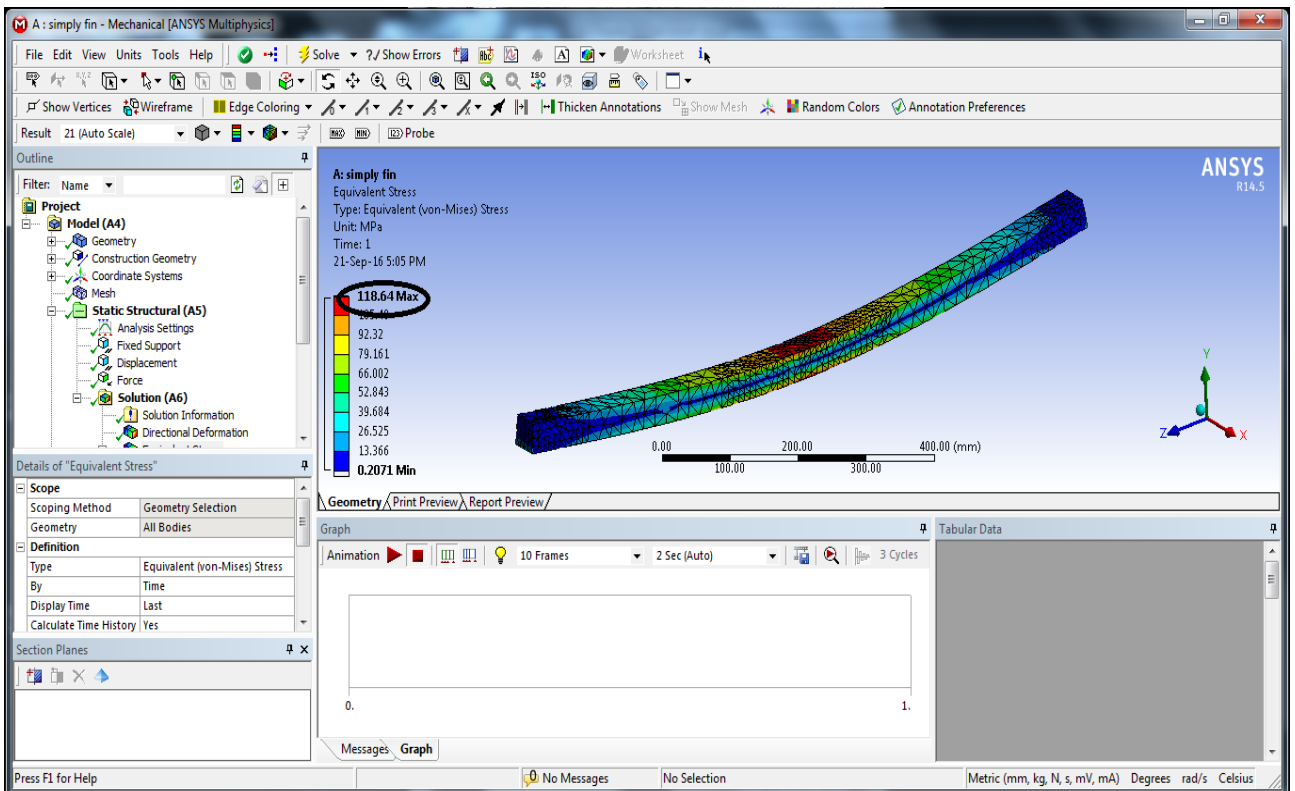


Figure 2.33 equivalent stresses with mesh refining



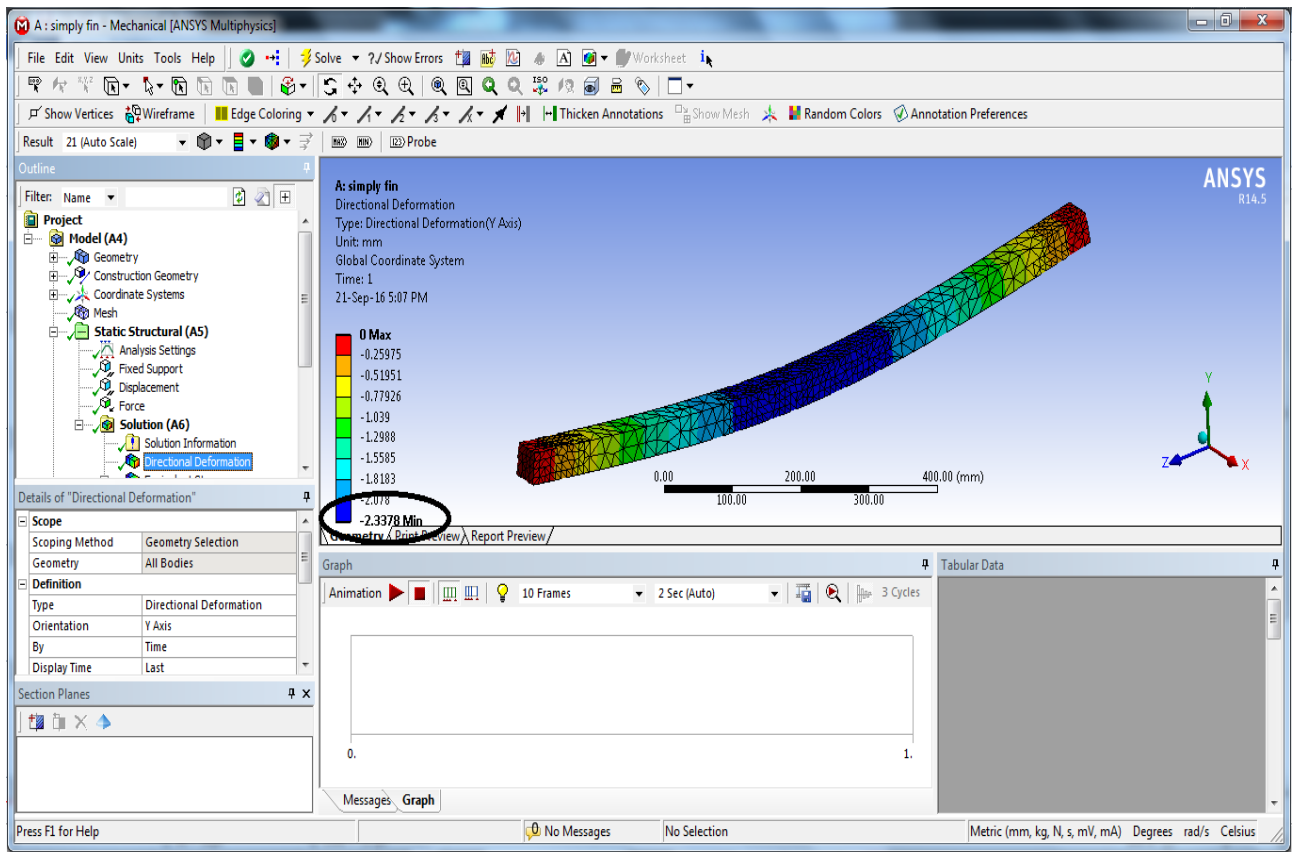


Figure 2.34 directional deformations with mesh refining

# Conclusion

---

## Comparing results

### First case

	Analytical results	Hand calculations
Displacement	<b>-1.46 mm</b>	<b>-1.453 mm</b>
Stress	<b>58.61 Mpa</b>	<b>58.59 Mpa</b>
Shear	<b>-2500 N</b>	<b>-2500 N</b>

### Second case

	Analytical results	Hand calculations
Displacement	<b>-2.33 mm</b>	<b>-2.33 mm</b>
Stress	<b>118.64 Mpa</b>	<b>117 Mpa</b>
Shear	<b>-2500 N</b>	<b>-2500 N</b>

This humble graduation project has allowed me to improve my knowledge about designing. It also improved my skills of using software applications to do the analyses and to design structures' elements to confront the problems that face the engineers at the level of testing and modeling with different construction materials.

The major mission in this document is to analyze and compare hand calculation results with the software results.

Firstly we talked in general about the software I used which is ANSYS workbench and other products of ANSYS Inc.

Finally, modeling a simply supported beam in two cases ; distributed load in the first case and center loaded in the second comparing my hand calculation results with the software result which are nearly the same finding that the software analysis are accurate and saving time and effort..

At the end I hope that I introduced suitable and effective solutions for the problems. It's important to note that the details of the solution are able to change by the situation and the used materials and many other variables.

(1)<http://www.mece.ualberta.ca/tutorials/ansys/IT/GST/GST.html>

(2)ANSYS Help V 14.5

(3) Engineering analysis with ANSYS software. Y. NAKASONE and s. YOSHIMOTO, T.A. STOLARSK. Department of mechanical engineering Brunel University, Middlesex,

