Benha Faculty of Engineering
Mechanical Engineering Department

M1382 : **Computer Aided Design CAD**
First Semester 2018, Y3

Lecture No. 04

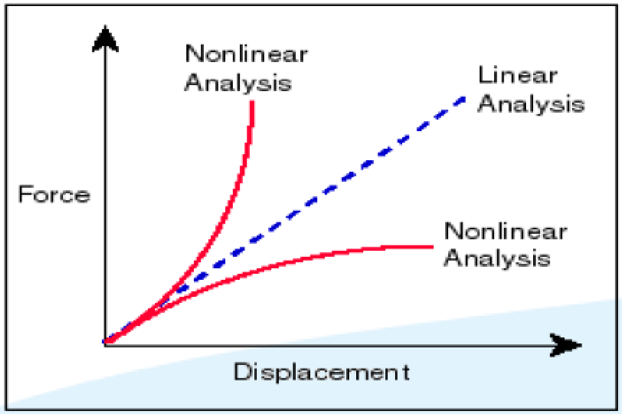
Presented by:
Mahmoud Magdy

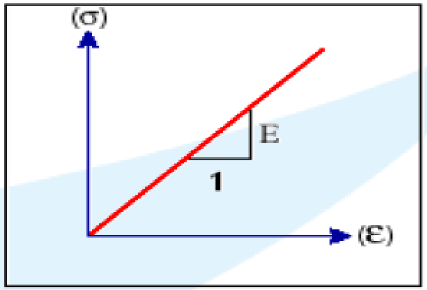
Week	Topics
1	Introduction
2	Introduction to CAD (Solid Modeling)
3	Part modeling
4	**Finite element analysis (FEA)**
5	Parts assembly using SolidWorks
6	Basic concepts of engineering drafting
7	**Linear Static Analysis**
8	Adaptive Analysis and Mesh Control
9	Modal Analysis
10	Design Optimization
11	Case study 1
12	Case study 2
13	Co-simulation SolidWorks and ADMS software
14	Project Discussion

Linear Static Stress Analysis

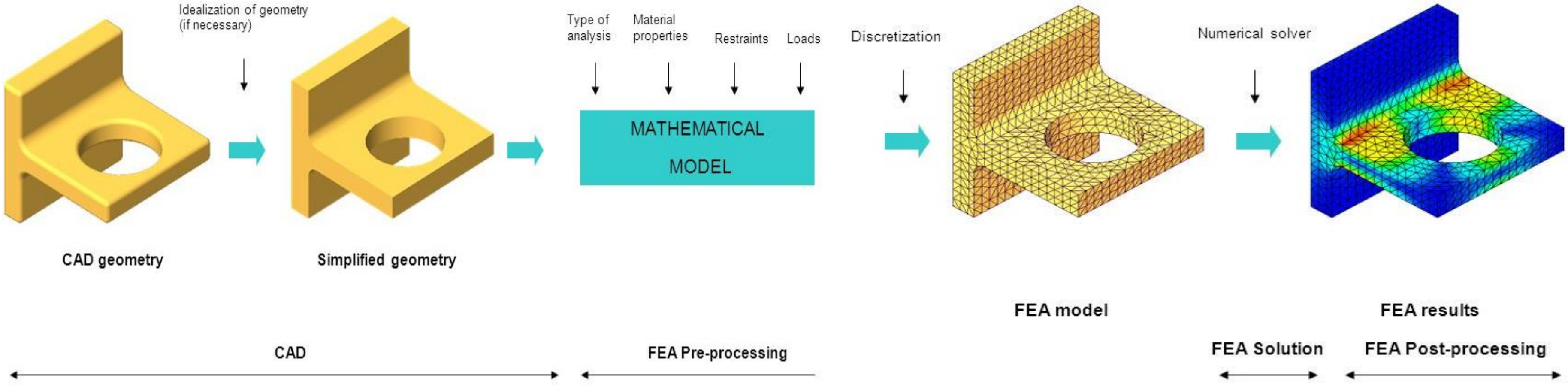
Key Assumptions:

- Loads applied slowly, no inertia, no time-dependence etc.
    - Dynamic analysis overcomes this assumption
- Linear material behavior
- Small deformations – constant stiffness matrix
    - Nonlinear Analysis overcomes this assumption



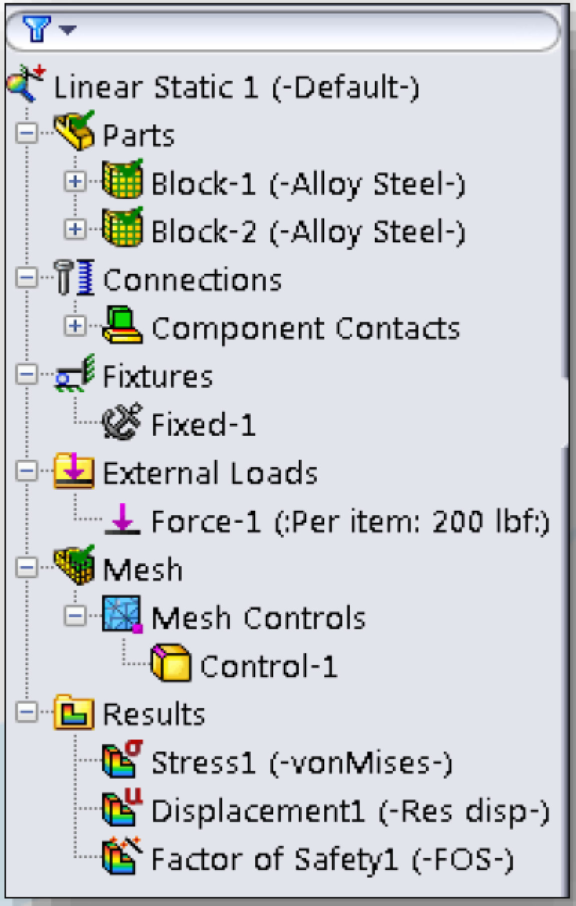


Building the FEA Model



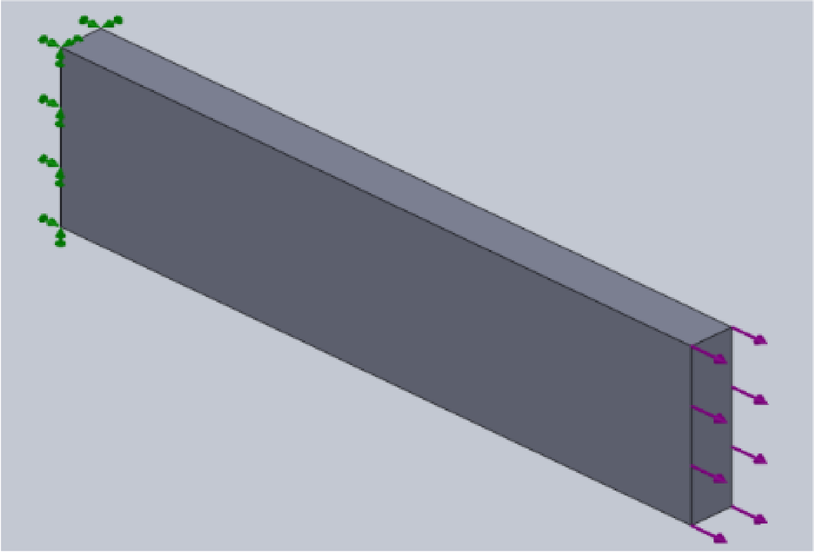
Analysis Process and Considerations

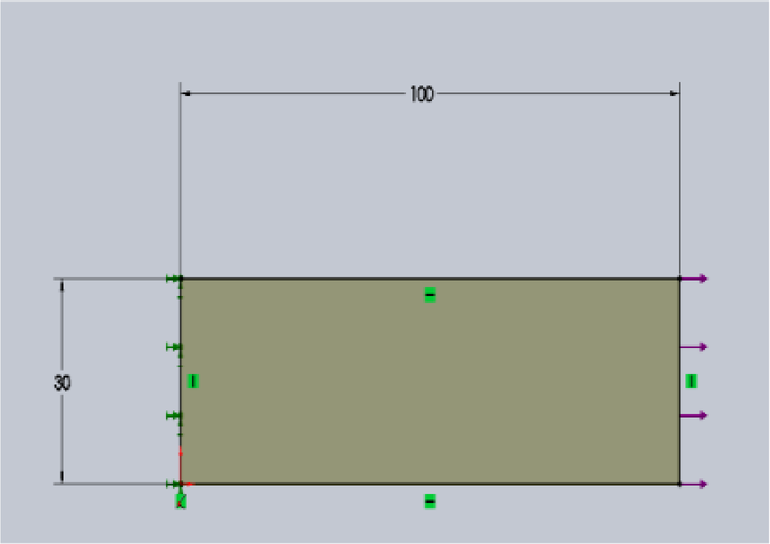
- Material Definition
- Contact
- Connectors
- Fixtures
- Loads
- Meshing
- Solving
- Results/Post Processing



Case study

A rectangular alloy steel plate has one of its ends fixed to the wall. The plate has a width w = 5 mm and a height h = 30 mm. The length of the plate is l = 100 mm. A load of 65 KN acts horizontally on the other end of the plate as shown in the figure. Find the optimal dimensions of the plate using Shape optimization.

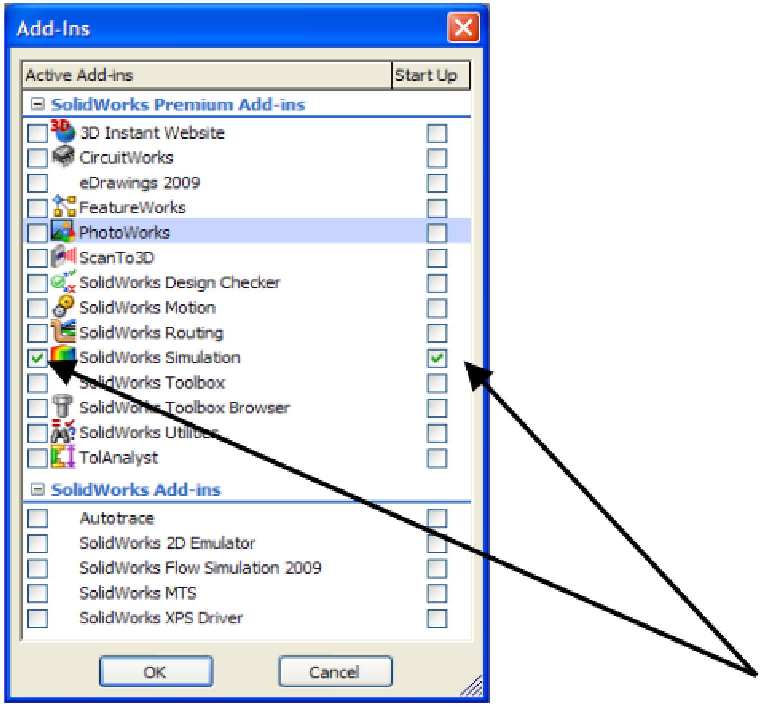




Case study

Step 1: Enabling SolidWorks Simulation

In order to perform FEM analysis, it is necessary to enable the FEM component, called SolidWorks Simulation, in the software.



Check “SolidWorks Simulation” boxes

1. Pre-Processing

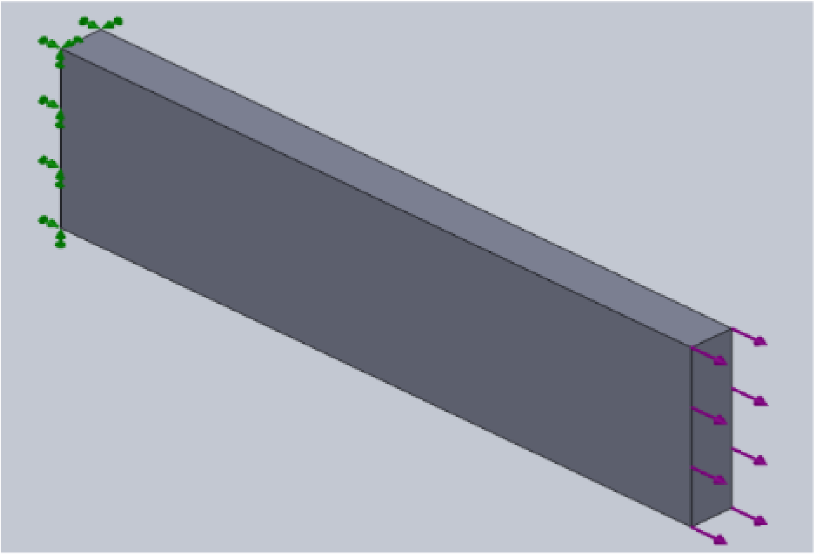
The purpose of pre-processing is to create an FEM model for use in the next step of the simulation, Solution. It consists of the following sub-steps:

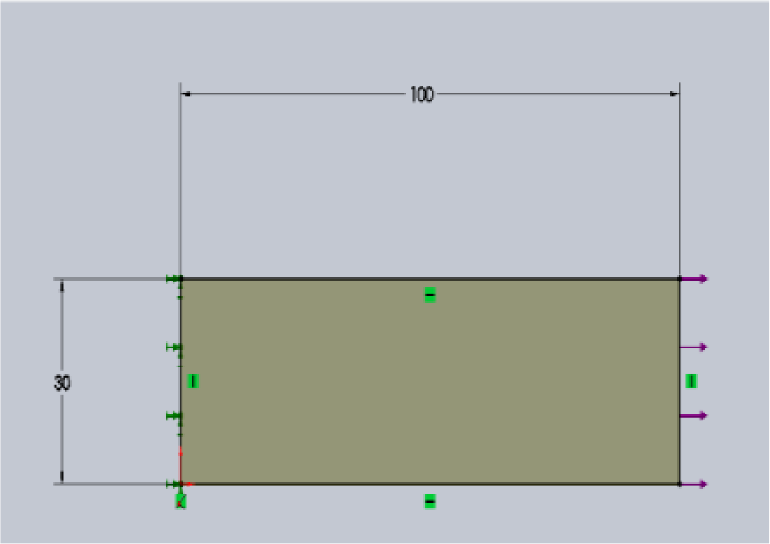
- **Geometry creation**
- **Material property assignment**
- **Boundary condition specification**
- **Mesh generation.**

1.1 Geometry Creation

The purpose of Geometry Creation is to create a geometrical representation of the solid object or structure to be analyzed in FEM. In SolidWorks such a geometric model is called a part

.



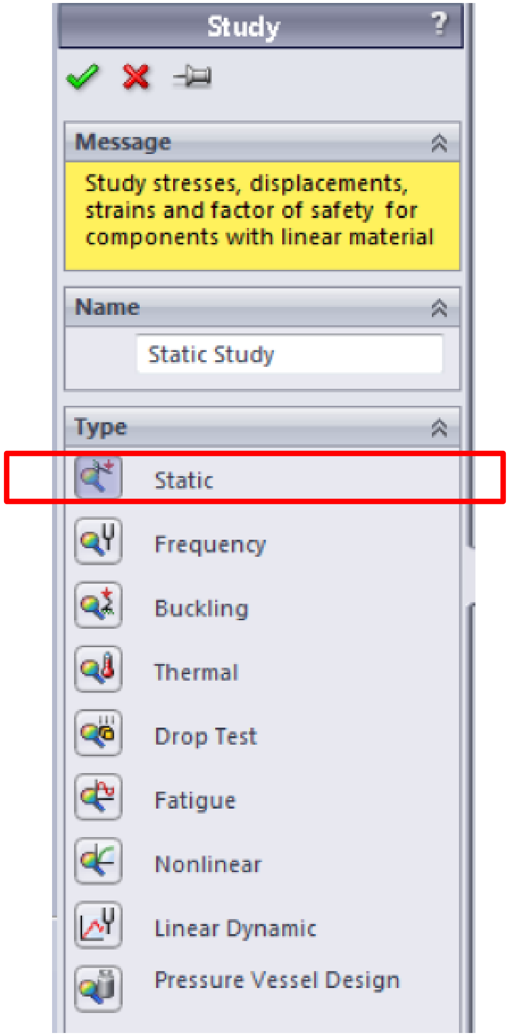


Case study

Step 1: *Opening the part for simulation.*

Step 2: *Creating a Study*

- Click the “Simulation” tab above the model tree
- Click on the drop down arrow under “Study” and select “New Study” as in the Figure
- In the “Name” panel, give the study the name “Static Study”
- Select “Static” in the “Type” panel to study the static equilibrium of the part under the load
- Click “OK”  to accept and close the menu



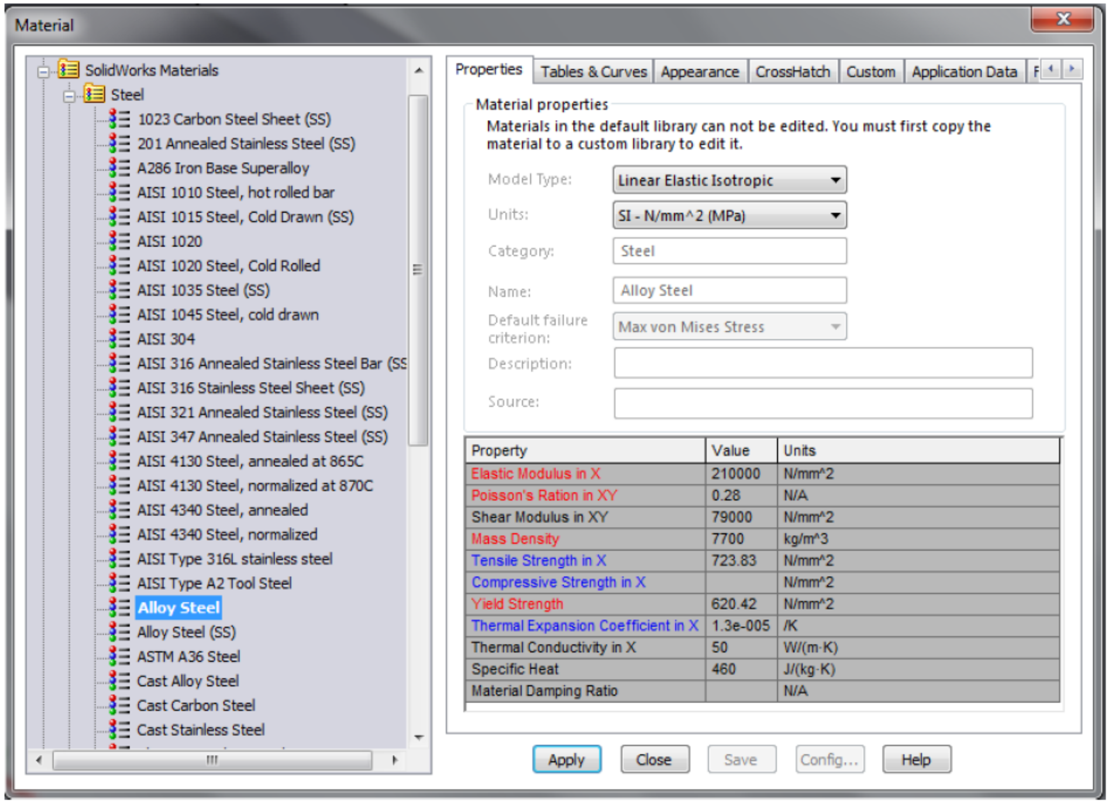
Case study

1.2 Material Property Assignment

Step 3: *Opening the material property manager*

- In the upper left hand corner, click “Apply Material”.
- The “Material” window appears as shown in the Figure.

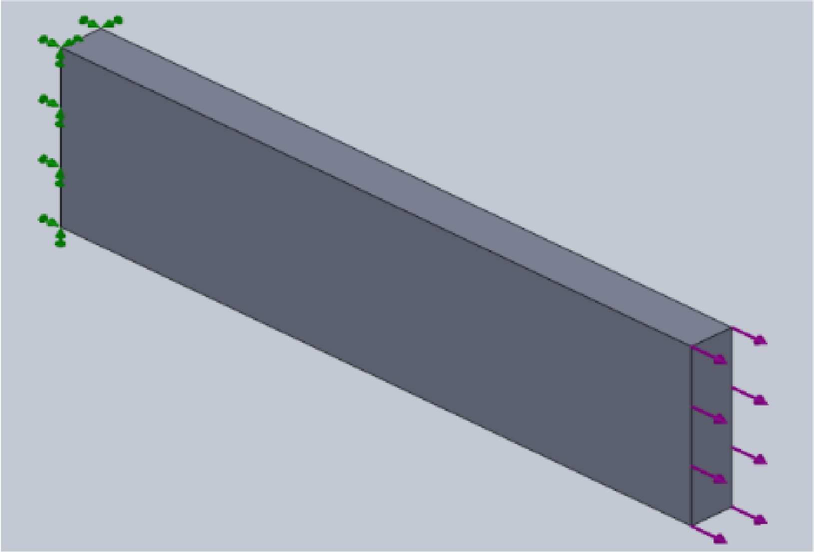
This will apply one material to all components. If the part is made of several components with different materials, open the model tree and apply this process to individual components.

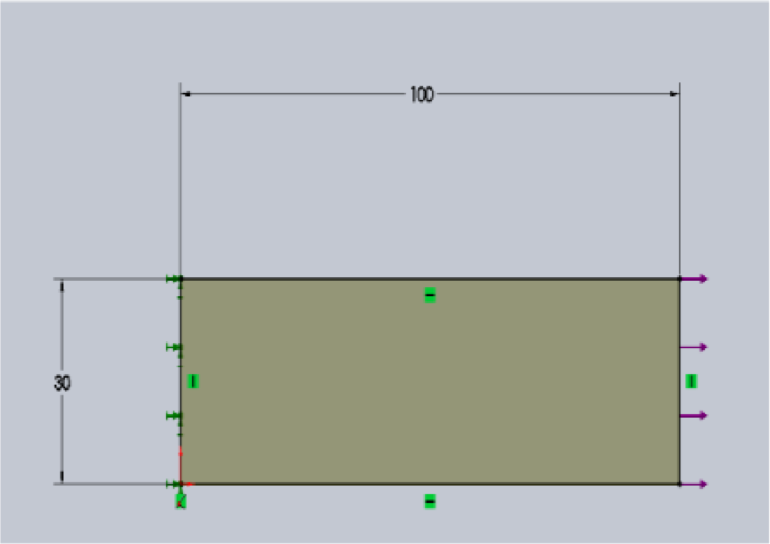


Case study

1.3 Boundary Condition Specification

In the Boundary Condition Specification sub-step, the restraints and loads on the part are defined. Here, the face of the beam attached to the wall needs to be restrained, and the force in the proper direction needs to be applied on the other end of the beam.





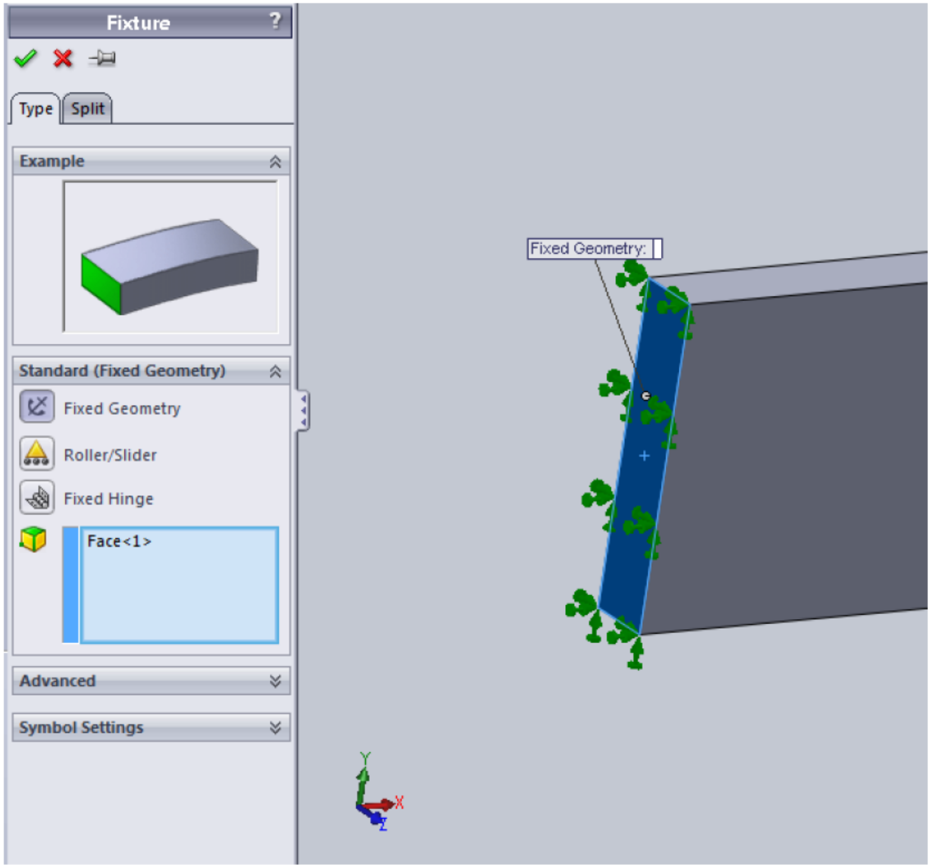
Case study

Step 4: *Opening the fixtures property manager*

- Right click on “Fixtures” in the model tree and select “Fixed Geometry”
- Move the cursor into the graphic window.

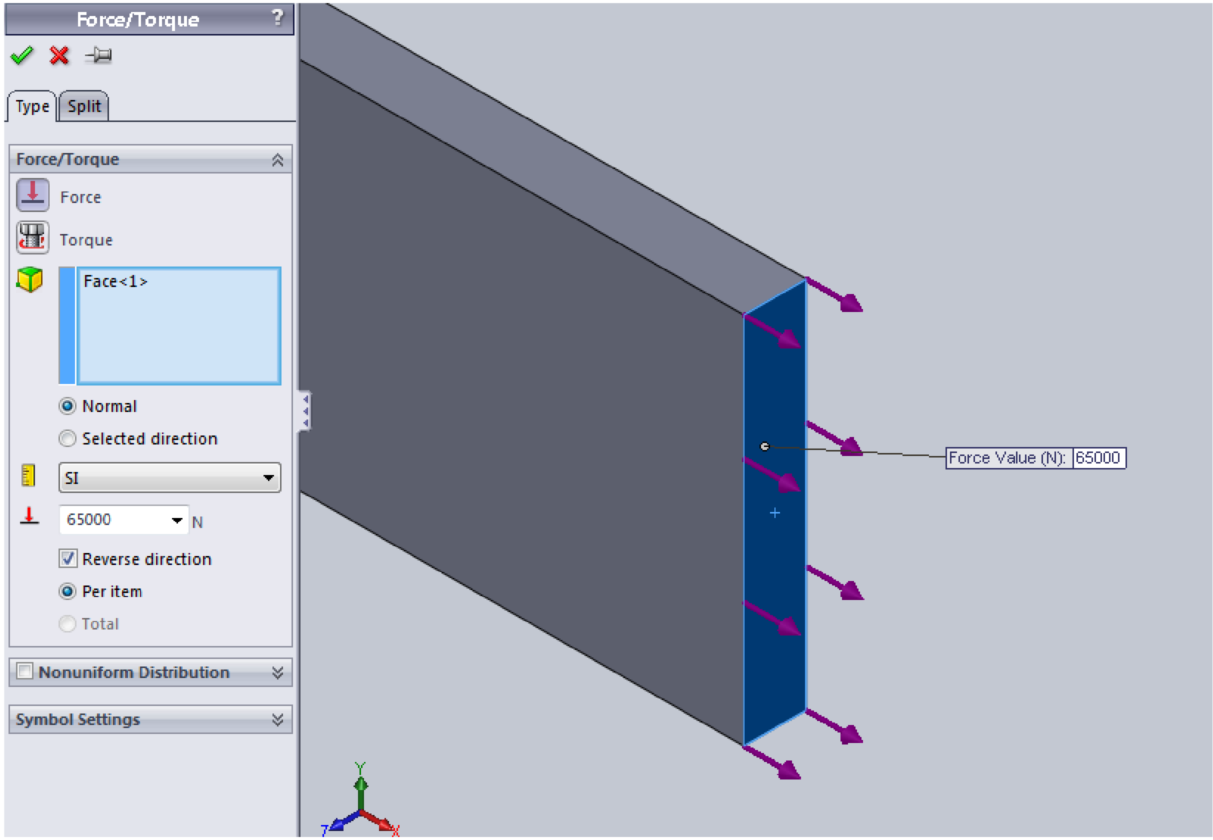
Step 5: *Restraining the member*

- Select the face as in Figure 5
- Once the face has been selected, click the green check mark to close the “Fixture” menu



Case study

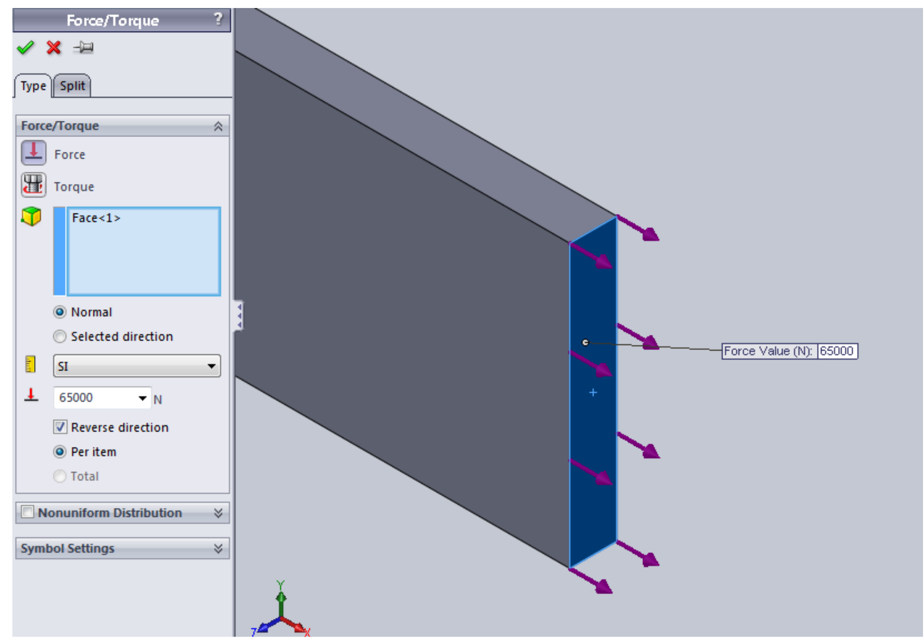
Step 6: The next step is to load the beam with the applied force. The total force applied is 65000 N in the direction as shown in the figure 6.



Case study

***Step 6 :** Applying the Tensile Force:*

- Right click on “External Loads” in the model tree and select “Force”.
- Under the “Force/Torque” tab, click the “Faces, Edges, Vertices, Reference Points for Force” input field box to activate it, if not already active.
- Click on the face on which the force is applied in the graphics window. Make sure the face is highlighted (turns blue) and appears in the input field box.
- Use SI units and type in a force of 65000 N.
- Check the “Reverse direction” box if the force is pointing in the wrong direction, as shown in Figure.
- Click “OK” to close the menu.



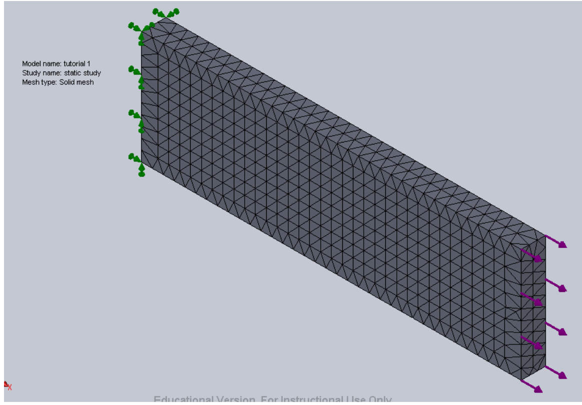
Case study

1.4 Mesh Generation

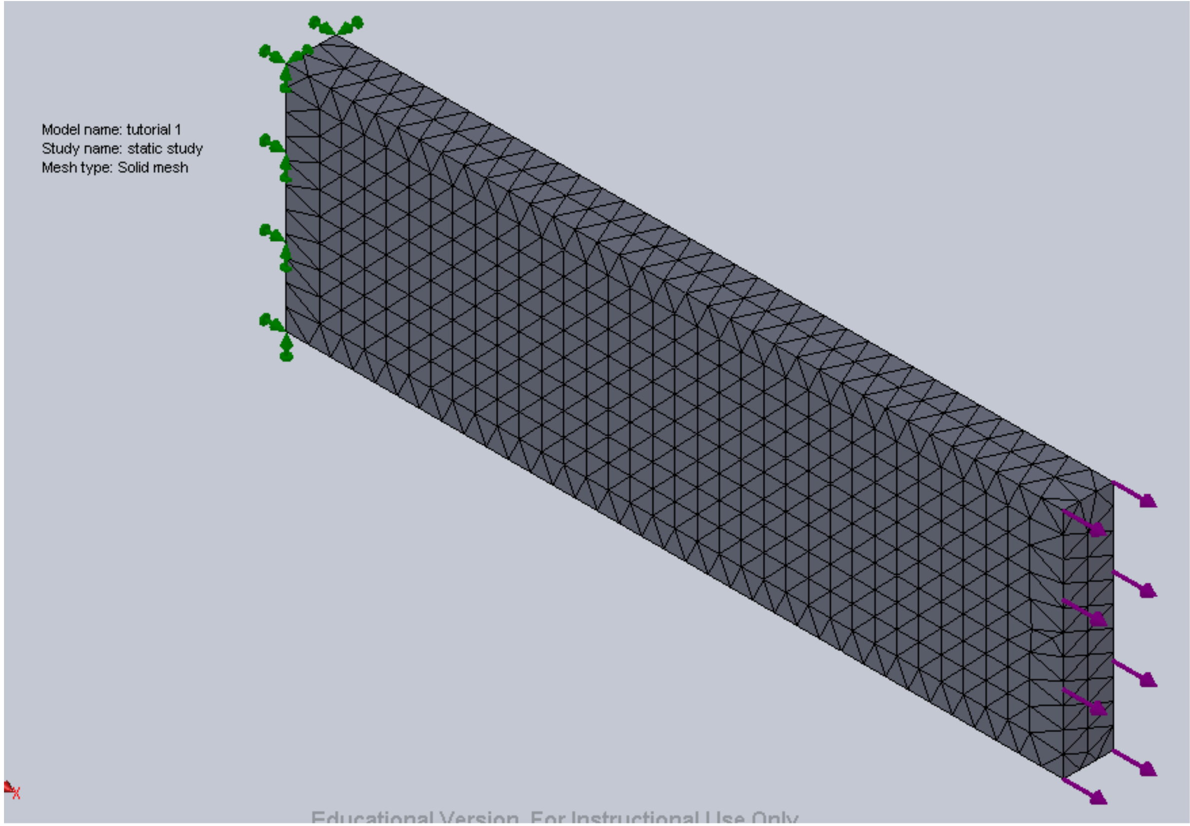
Purpose: The purpose of the Mesh Generation sub-step is to discretize the part into elements. The mesh consists of a network of these elements.

Step 9: Creating the mesh

- Right click “Mesh” in the model tree and select “Create mesh”
- Leave the mesh bar on its default value
- Drop down the “Advanced” menu and make sure the mesh is high quality, not draft quality, by making sure the “Draft Quality Mesh” checkbox is not clicked
- Figure 7 shows the completed mesh
- Click “OK” to close the menu and generate the mesh.



Case study



“Mesh Control” in SolidWorks may be used to refine the mesh locally. The guiding principle is **to refine mesh at locations of high stress gradient**, such as regions around stress concentrators and locations of geometric changes. For the current problem, local mesh refinement is not pursued.

2. Solution

Purpose: The Solution is the step where the computer solves the simulation problem and generates results for use in the Post-Processing step.

Step 1: Running the simulation

- At the top of the screen, click "Run"
- When the analysis is finished, the "Results" icon will appear on the model tree

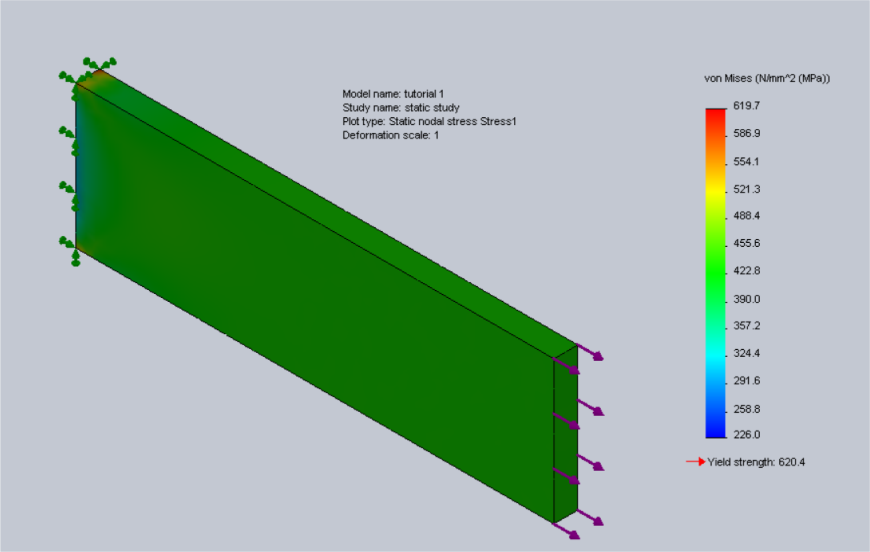
Case study

3. Post-Processing

Purpose: The purpose of the Post-Processing step is to process the results of interest. For this problem, the von Mises stress and the displacement is of interest.

Step 1: Creating a stress plot

- Right click “Results” on the model tree and select “Define Stress Plot”
- Select “von Mises” as the stress type and “Mpa” as the unit
- Unclick the “Deformed Shape” box and click “OK” to close the menu

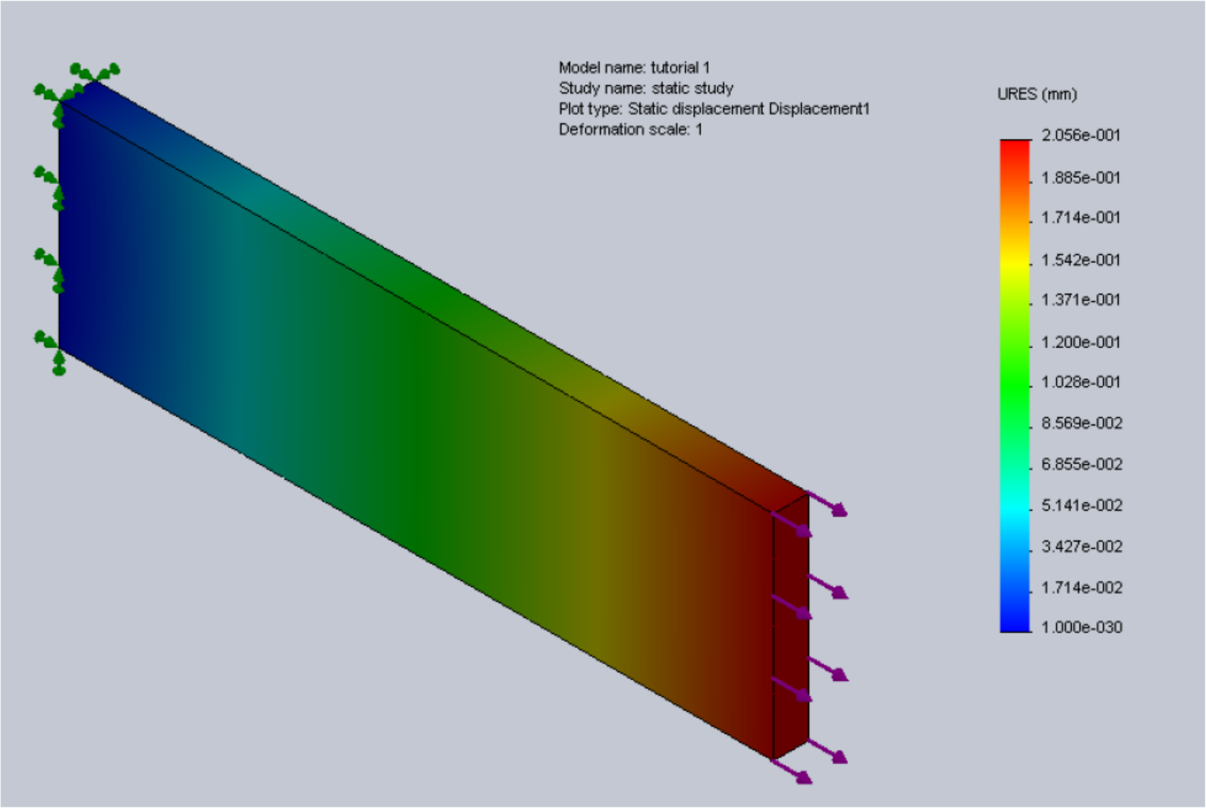


The von Mises stress plot.

Case study

Step 2: Plotting Displacement plot:

- Select the plot for Resultant displacement.

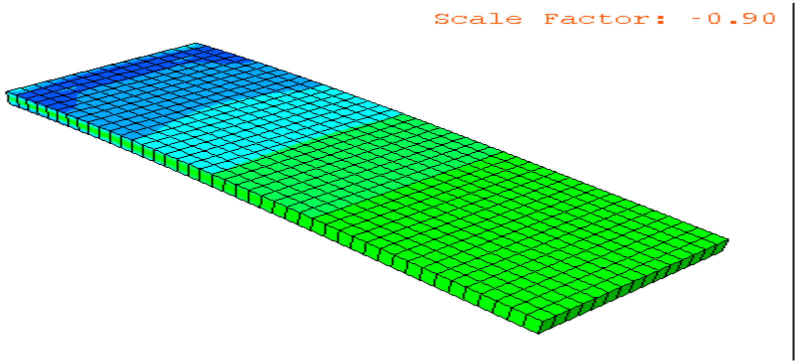


The displacement plot.

Result Quantities Available

Stress

- Stress in X, Y, Z
- Shear stress about X, Y, Z
- Principal Stresses - 1, 2, 3
- Von Mises
- Stress Intensity (P1-P3)
- Energy Norm Error
- Contact Pressure



Displacement

- Displacement in X, Y, Z, and resultant
- Reaction Forces

Strain

- Strain in X, Y, Z, resultant
- Shear Strain about X, Y, Z
- Principal Strains
- Strain Energy Density

How to Create Stress Strain Curves

Matching Real-Life Results: Conditions

- ✓ Are you comparing to physical tests, or real-world usage?
- ✓ Do you have a document that explains the experiment setup?
- ✓ What materials are being used? Do the mechanical properties of your material match that which is found in the SOLIDWORKS material database?
- ✓ How realistic are your restraints/fixtures?

Thank You for Attention !!

Any Questions