Static Stress Analysis of a Beam Fixed at One-End and Subjected to a Uniform Pressure on a Segment of Its Surface at The Free-End Using ANSYS Workbench

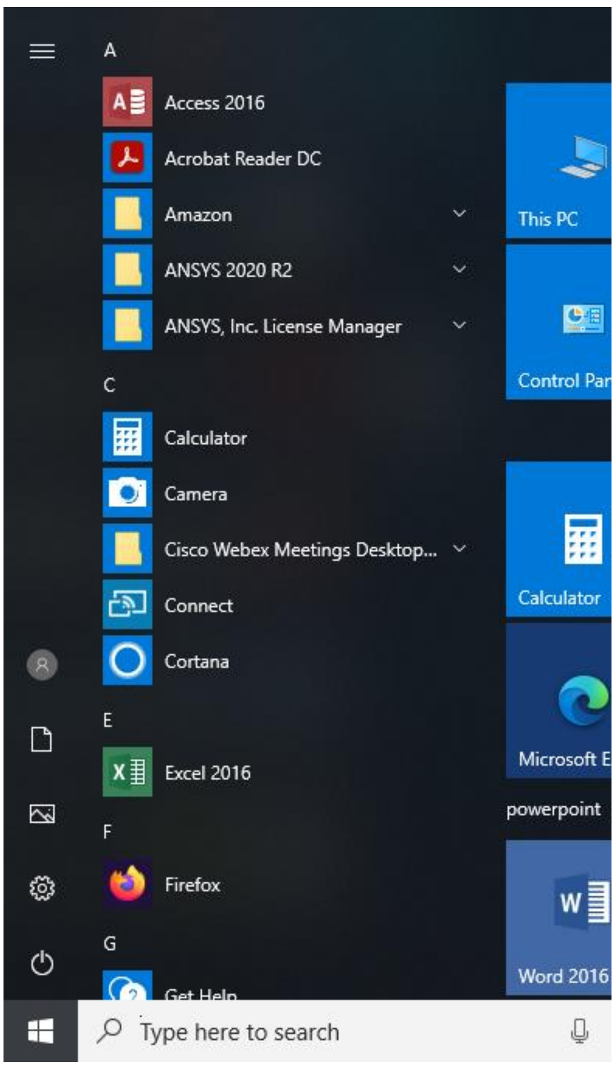
Cyrus Hagigat, Ph.D., PE
Professor of Engineering Technology at University of Toledo

Introduction:

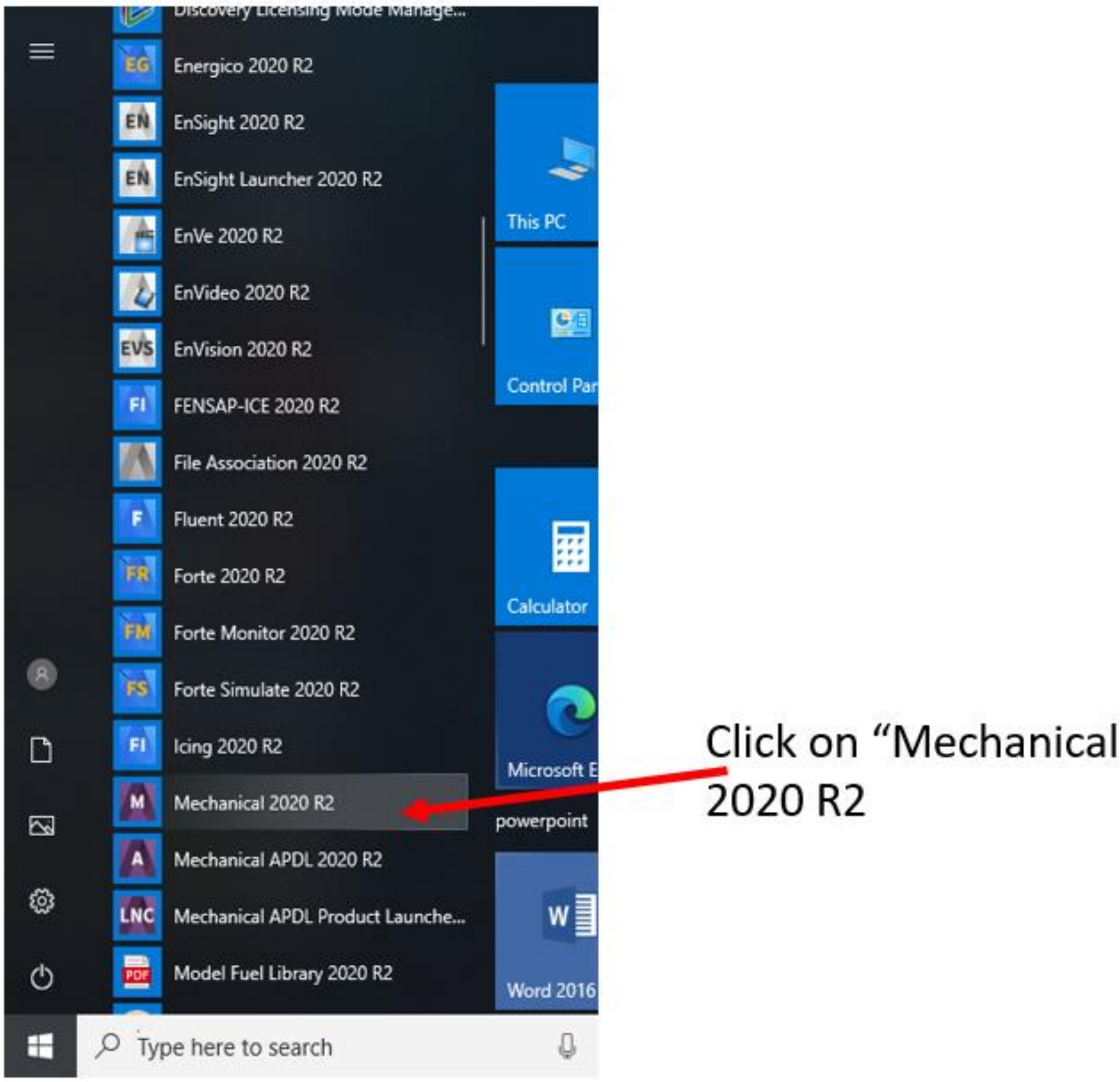
This article contains step-by-step instructions for analyzing a beam using the 2022 Rev 2 of ANSYS workbench. It must be emphasized that as future revisions of ANSYS workbench are released, the instructions presented in this article will not exactly match the software. In fact, the YOUTUBE tutorial that is reference 1 of this article solves the problem using an earlier version of ANSYS, and the instructions presented in this article do not exactly match the tutorial.

Step-by-step instructions:

Click on Windows at the bottom left corner, and the following pops on.



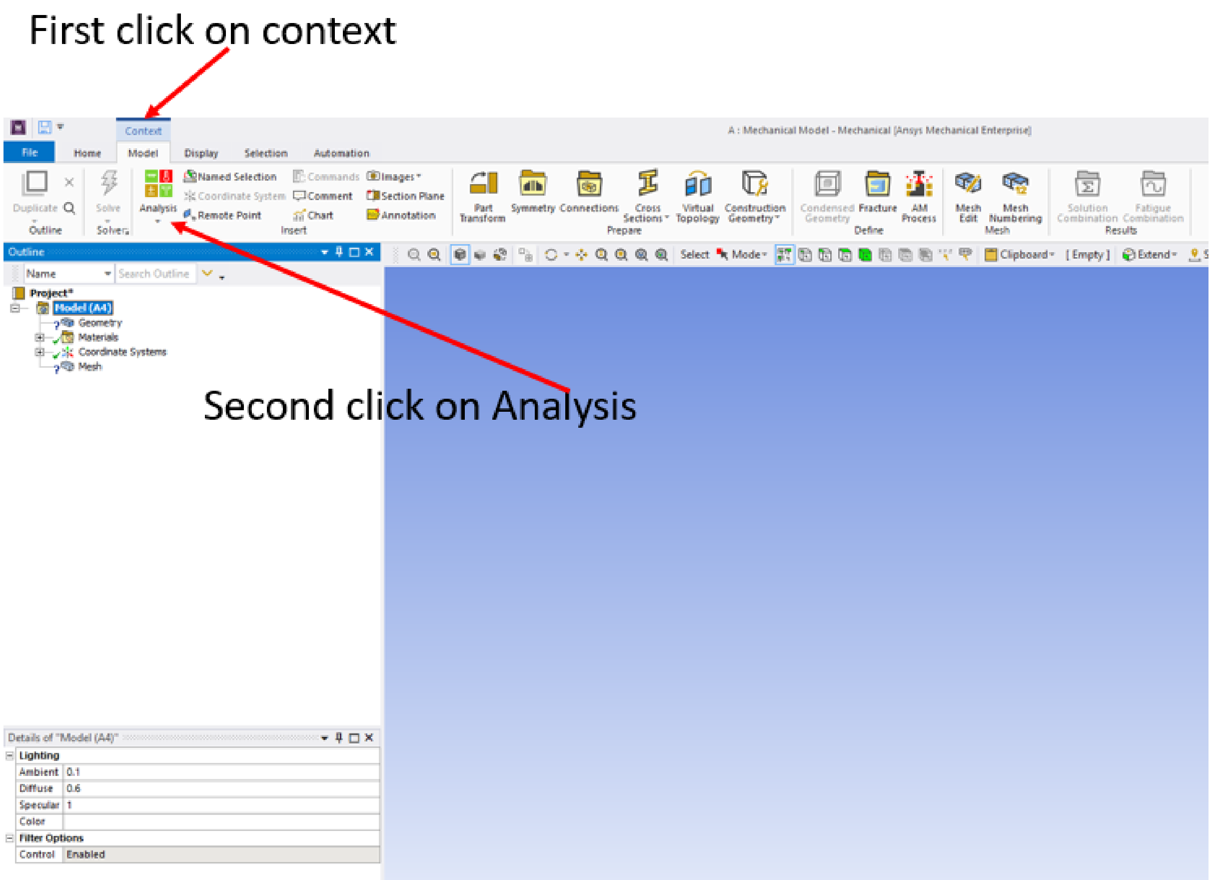
Click on “ANSYS 2020 R2,” and the following pops on.

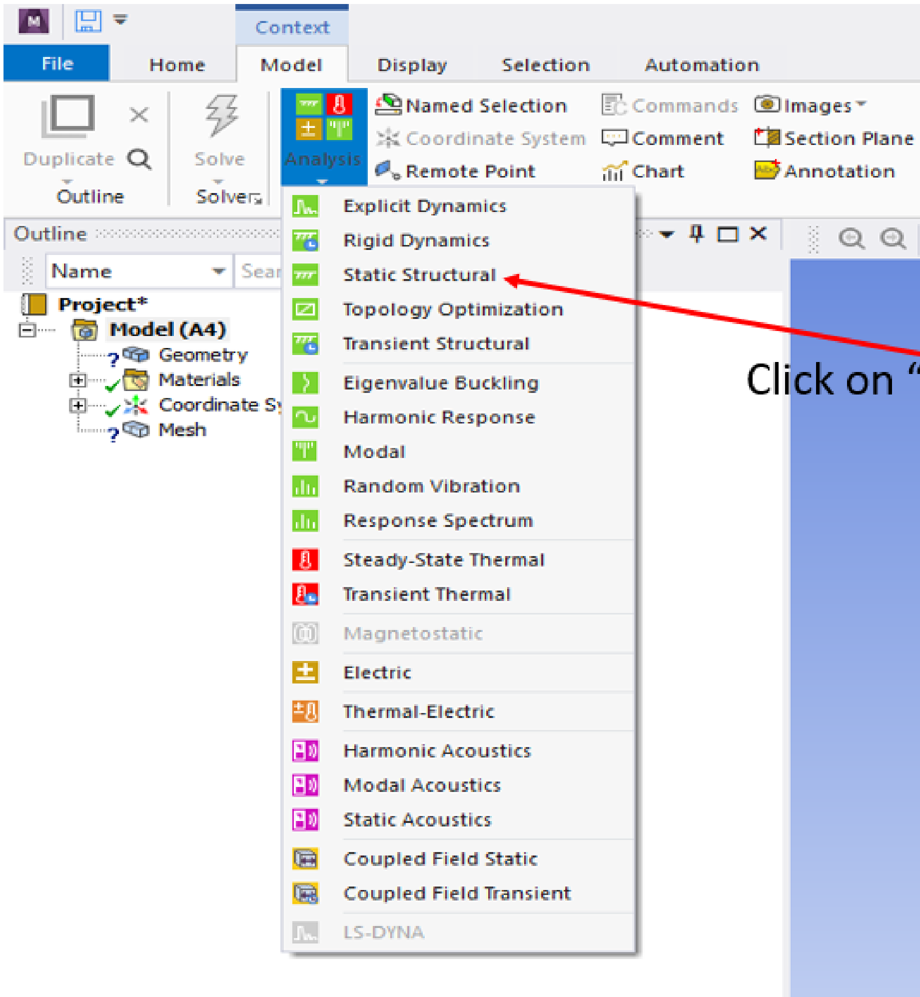


After doing the above, two separate and distinct screens become available. They can be accessed from the bottom of the Windows screen. They are like having two different programs open at the bottom. One is the project window, and the other is the “mechanical interface” window. The projects window is called WB (at the bottom of the screen). WB stands for Work Bench. The other window is called M. M stands for Mechanical.

Please don’t assume the program is not working when it takes a long time for both windows, especially the second window, to appear.

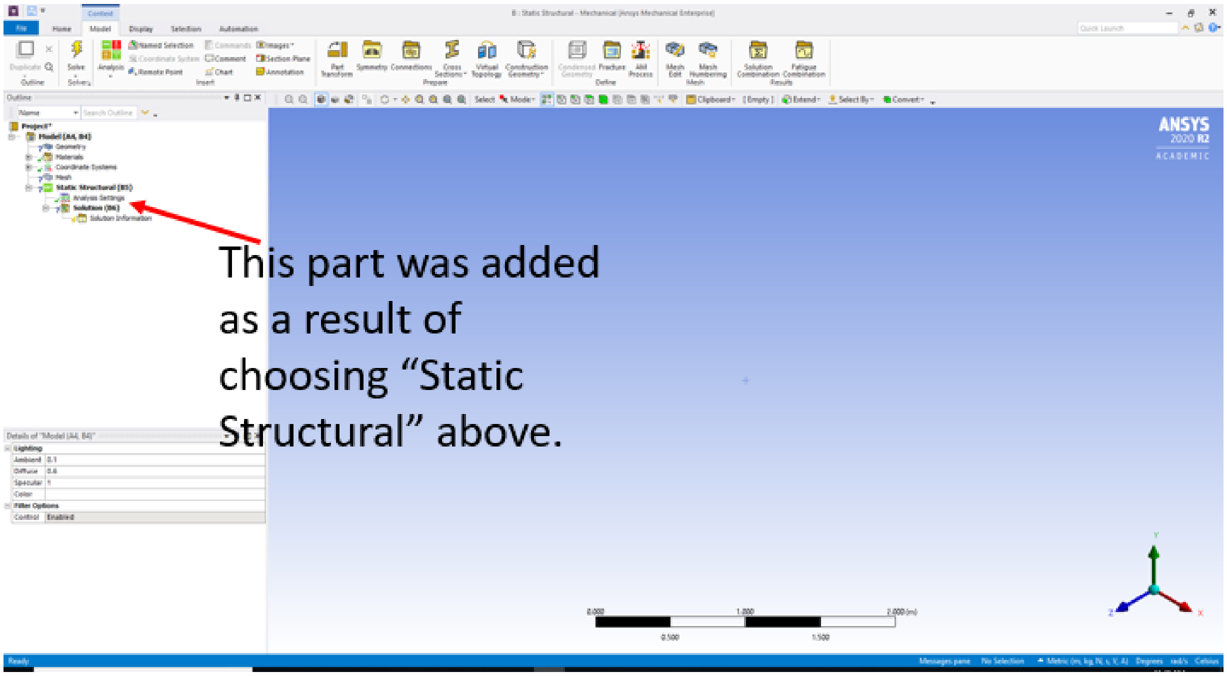
The following is the M (Mechanical) window.



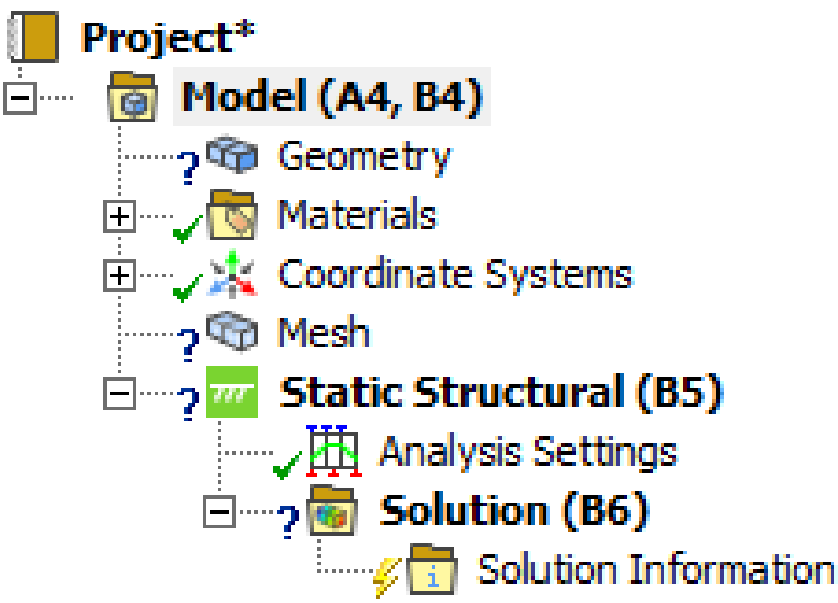


Click on “static Structural”

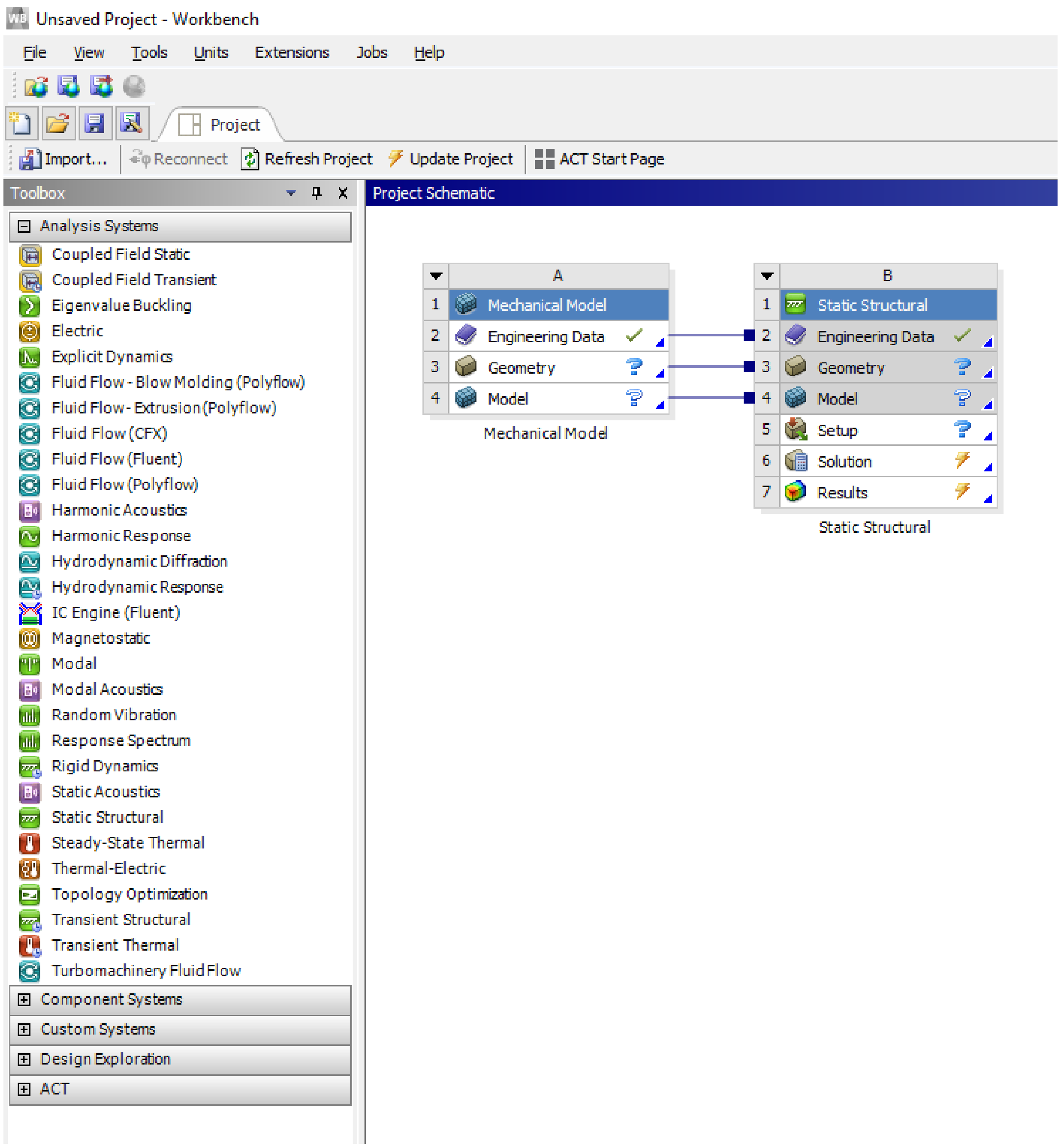
After clicking on “Static Structural,” the following appears.



The above is small. The following is the enlarged version.



By minimizing the above interface by clicking on its "_" on the upper right-hand side, the added part becomes visible, as shown below on the "projects screen." The project screen is the same as the "WB" screen at the bottom of the screen. Recall that "WB" stands for Work Bench.

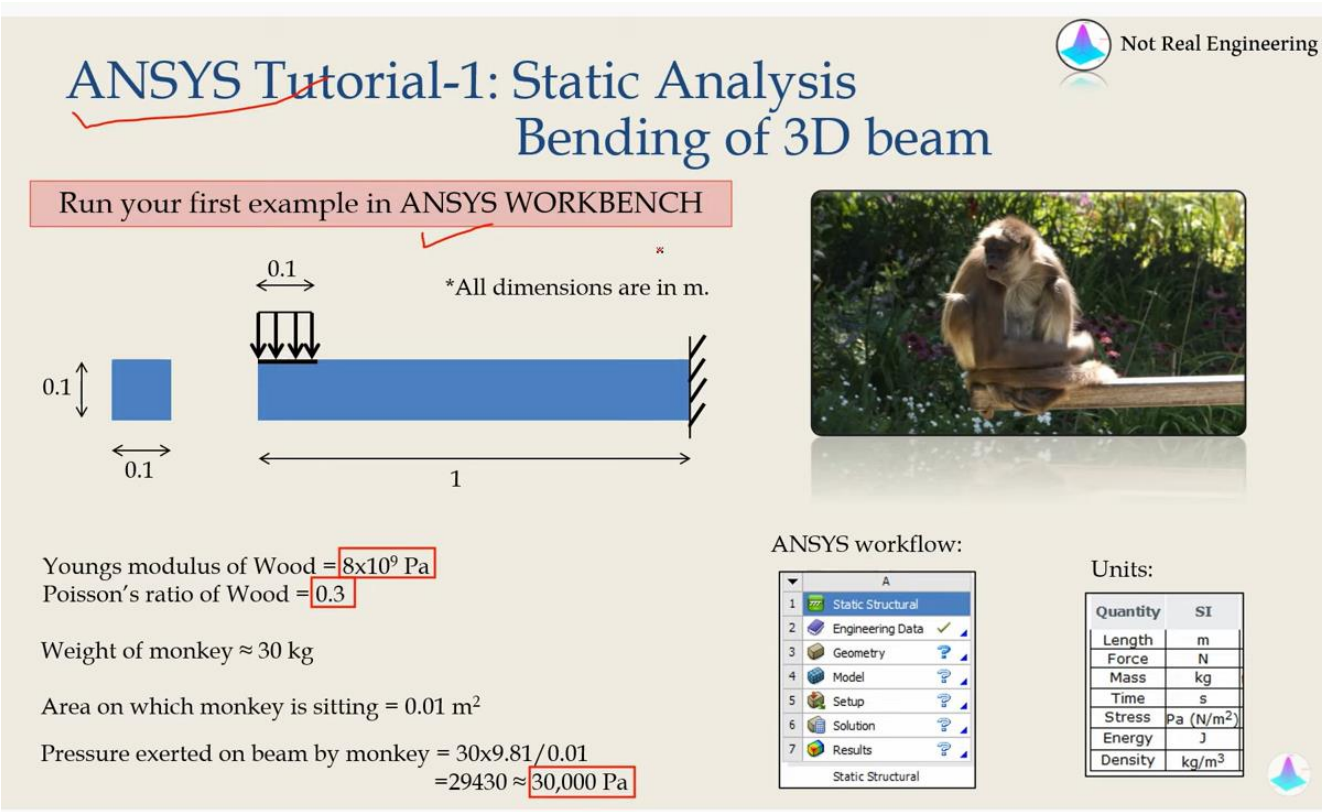


As shown above, "Static Structural" consists of 6 parts. They are

- Engineering Data
- Geometry
- Model
- Setup
- Solution
- Results

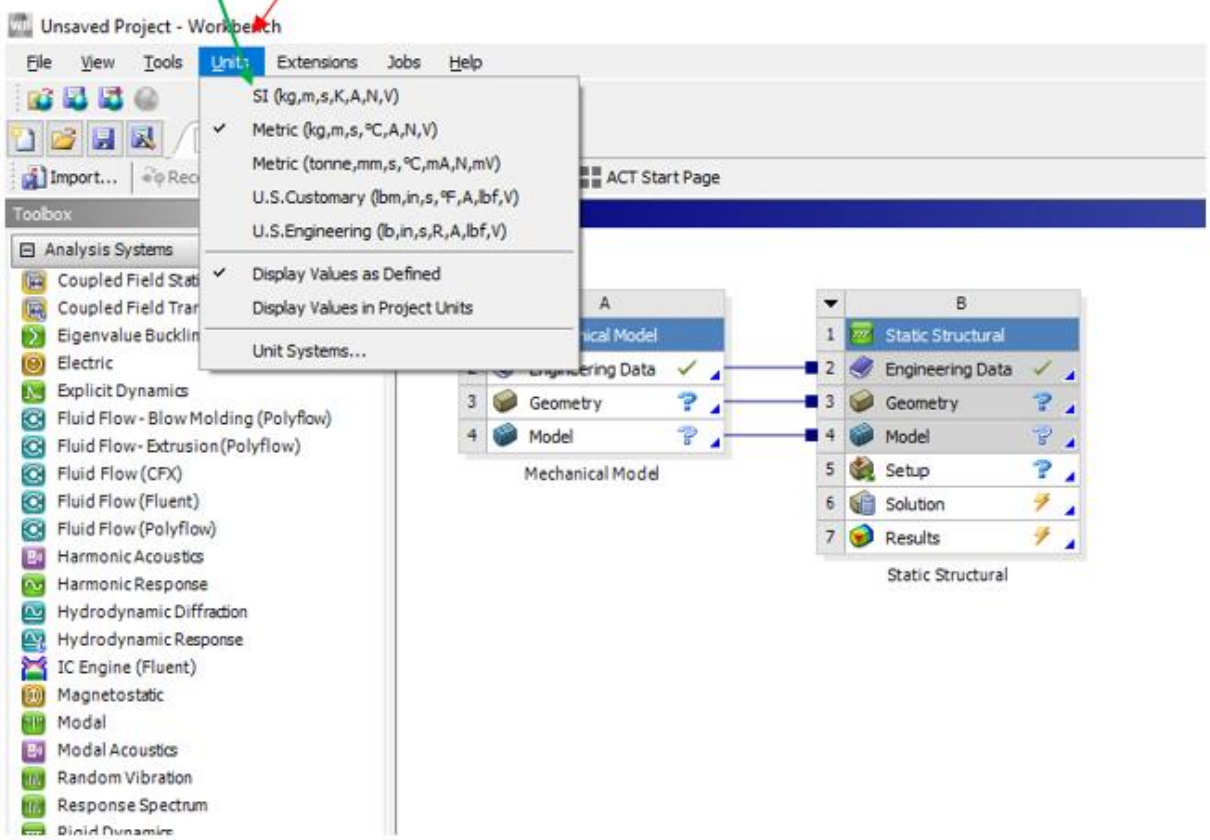
Note that "Engineering Data," "Model," and "Setup" can only be accessed through the "Projects" window and cannot be accessed in the "Mechanical" window, as shown by the fact that they are grayed out in the "Mechanical" window.

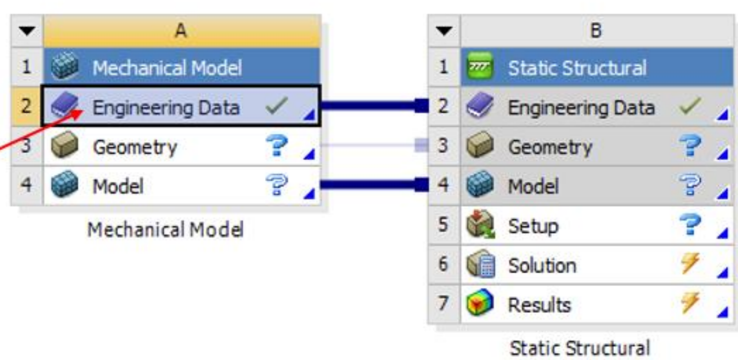
The following defines the problem that is going to be solved[1].



The following steps are implemented on the two distinctly separate screens (Project and Mechanical). The content clarifies which screen should be used.

Click on "Units" on top of screen and then click on "SI"

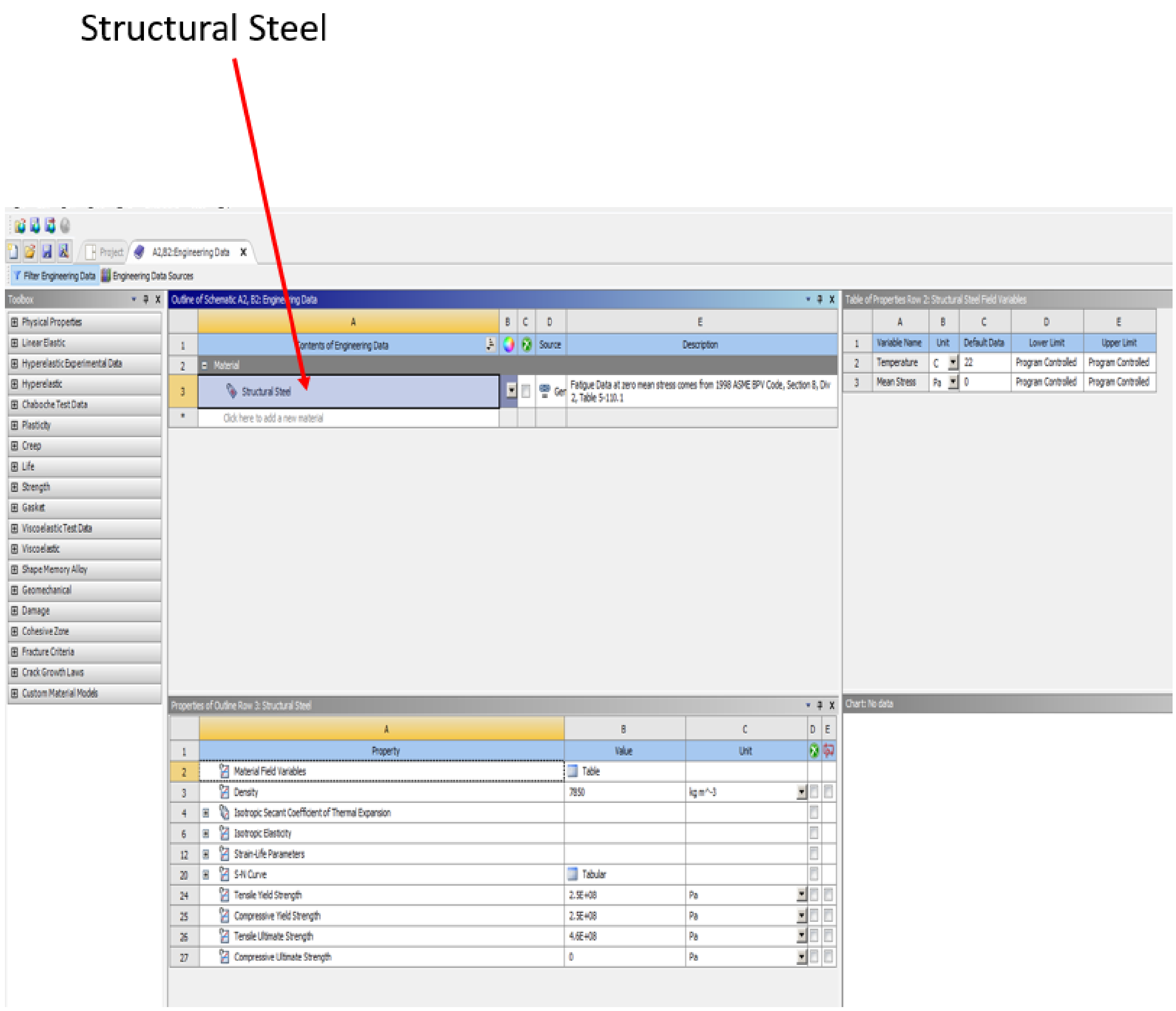




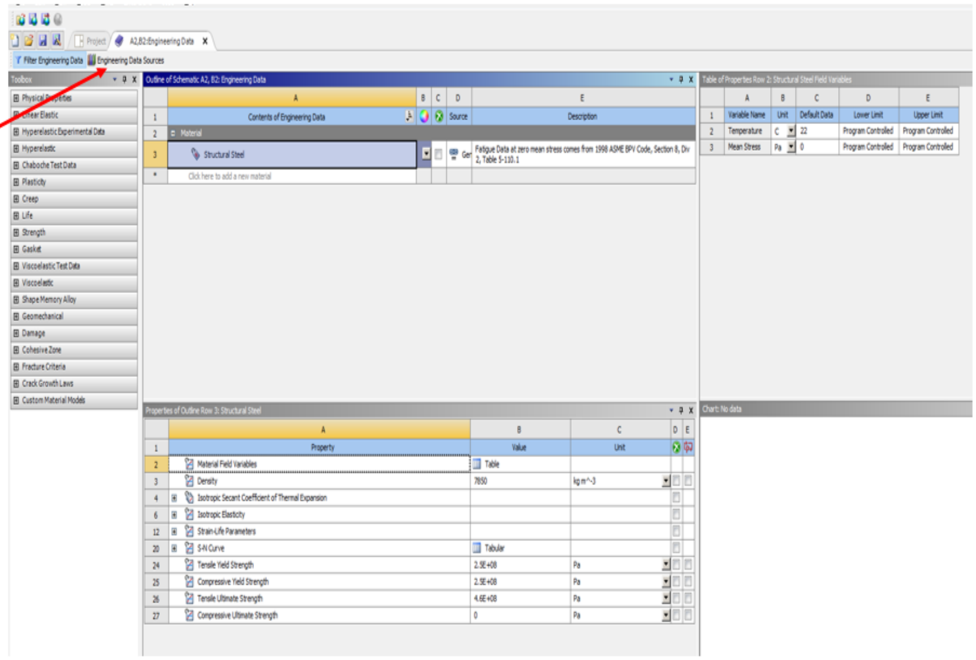
Double click on "Engineering Data".

I think once a "Project" is defined, a "Static Structural" or another type of structural analysis, such as vibration analysis, can be performed on the "Project." The "Project" consists of 1: Engineering Data; 2: Geometry; 3: Model.

The following appears. Structural Steel properties in SI units are listed as shown below.

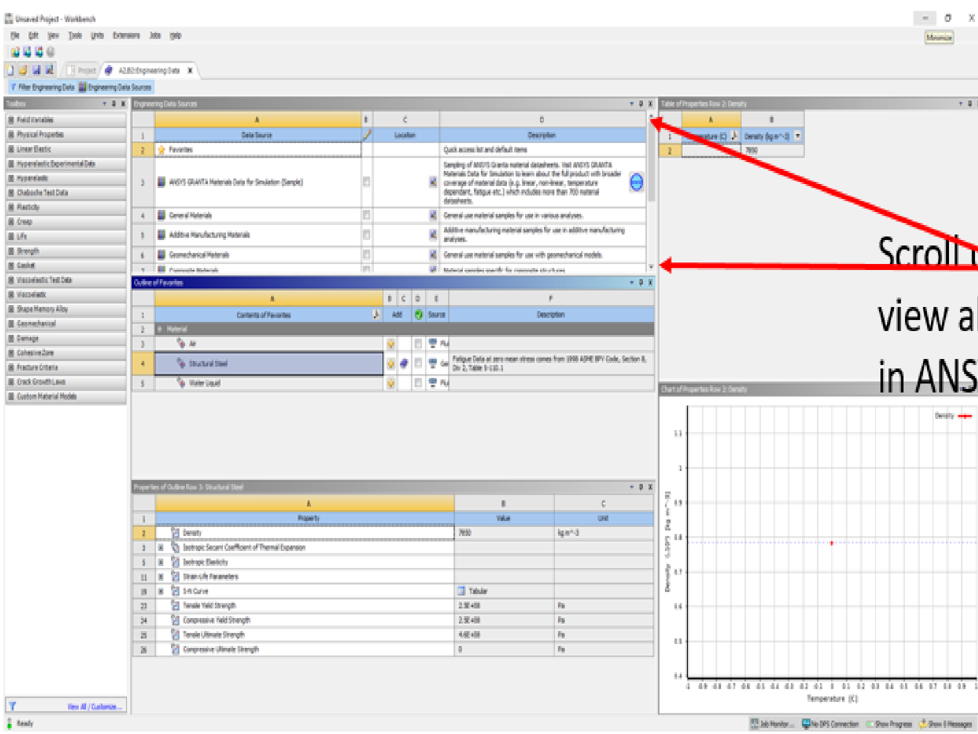


Other materials can be added, as shown below.



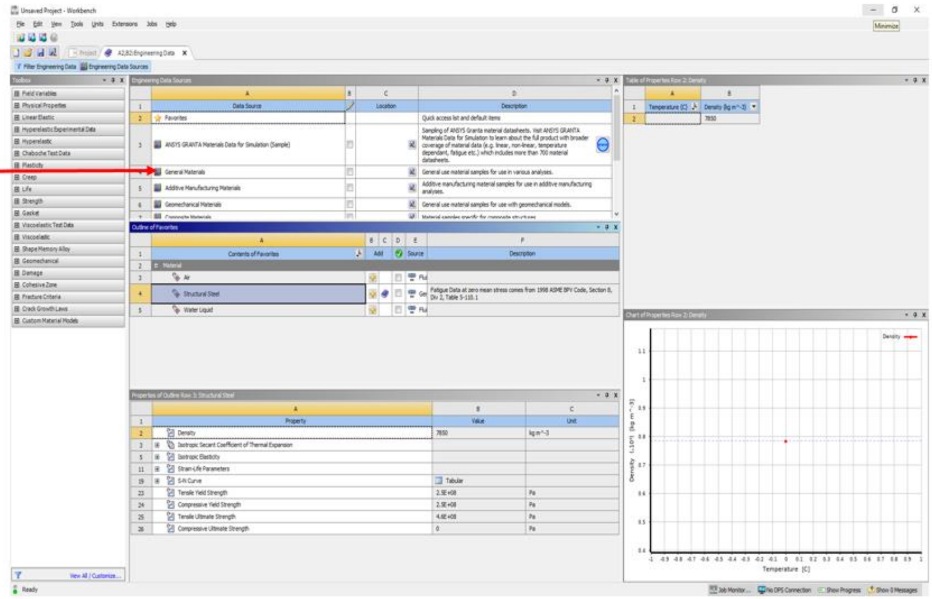
Click on "Engineering Data Sources" to bring up other material properties. Make sure you click only once. More than once, creates problems.

After the above, the following screen pops on. It is small, and it cannot be read.



Scroll up and down to view all the available data in ANSYS data base.

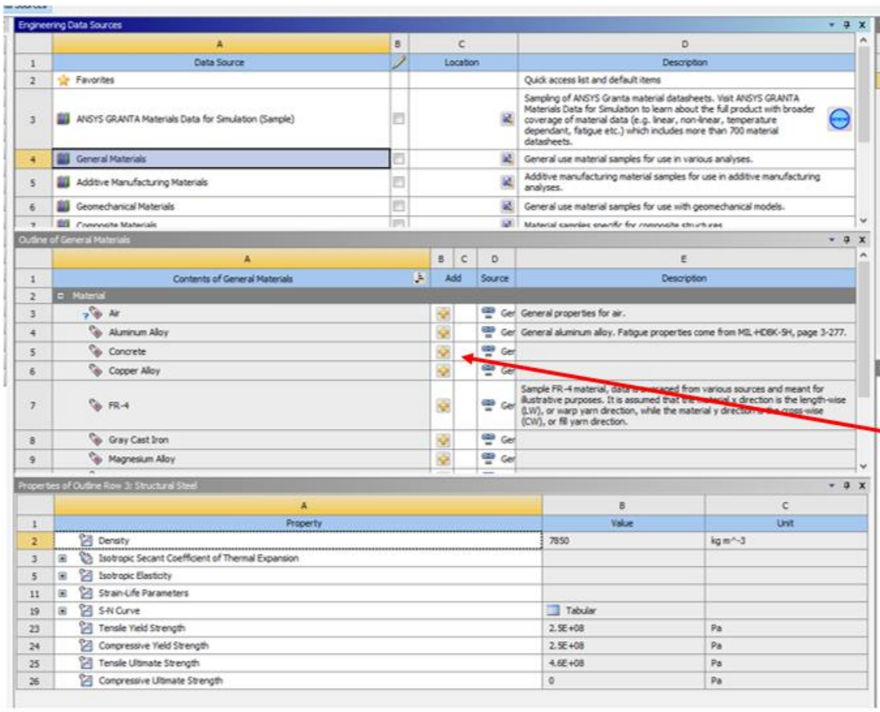
Click on "General Materials"



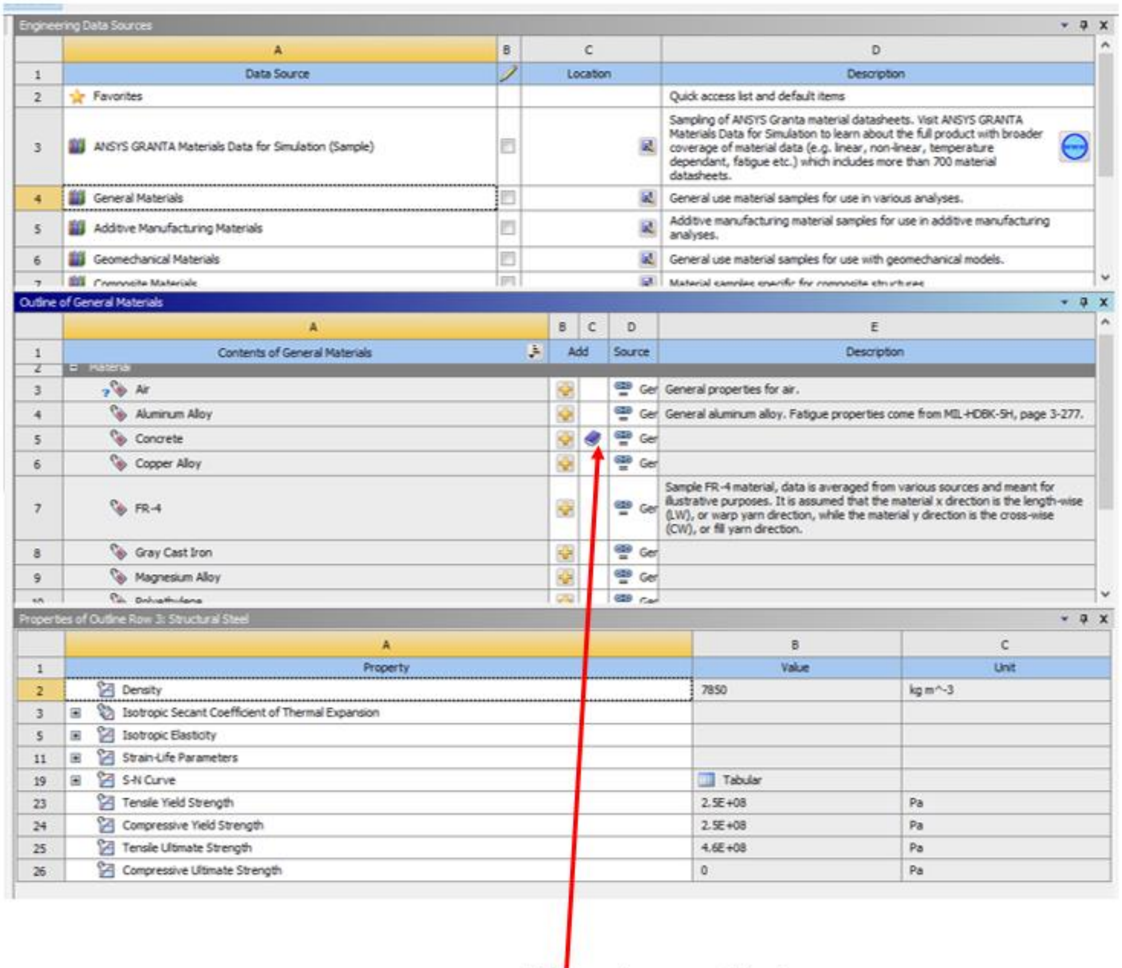
To see all available material available on the ANSYS database, scroll up and down.

After clicking on "General Materials," as shown above, common materials such as Gray Cast iron and Magnesium Alloys appear. Click on the "+" sign to add the particular material to the first material table.

Below is an example of clicking on the "+" sign next to concrete.

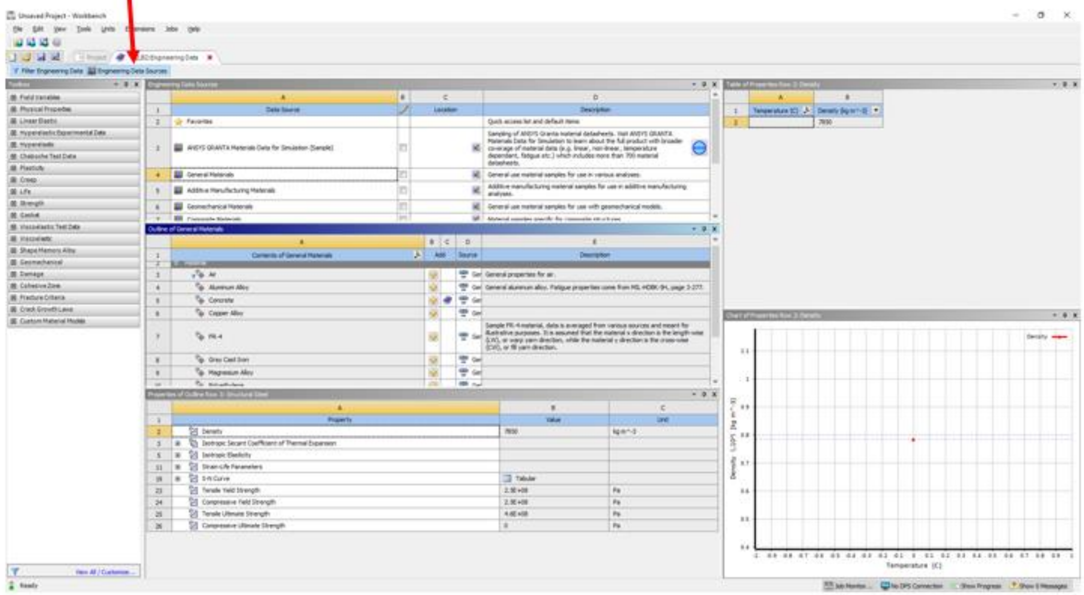


Click on"+" next to concrete and concrete will be added to the original material table.

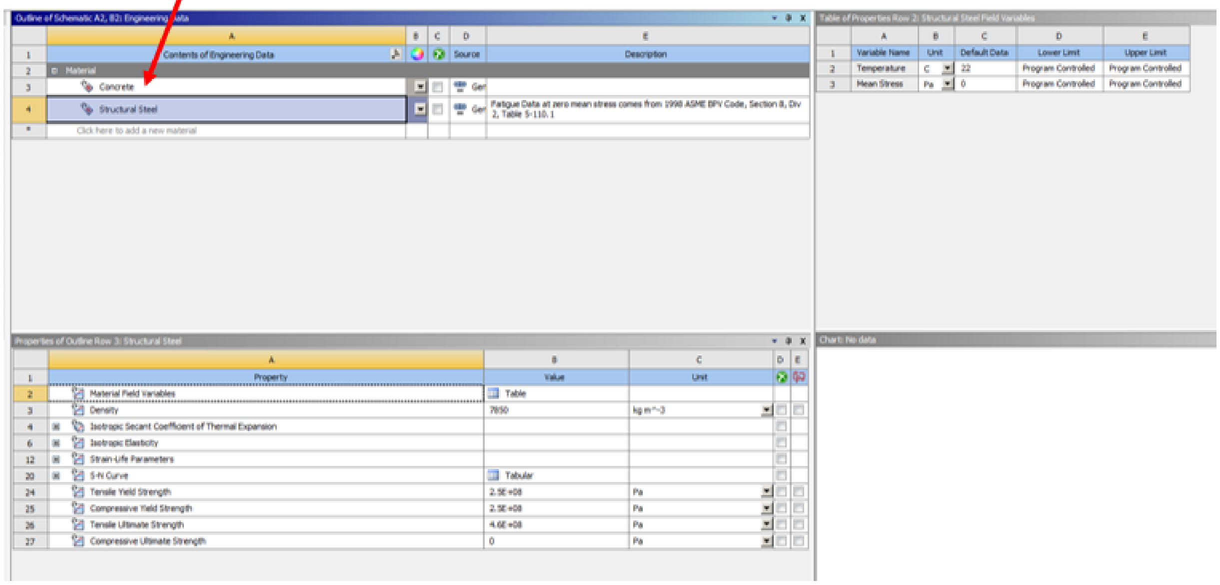


This shows that "Concrete" has been added to "Engineering Data Source"

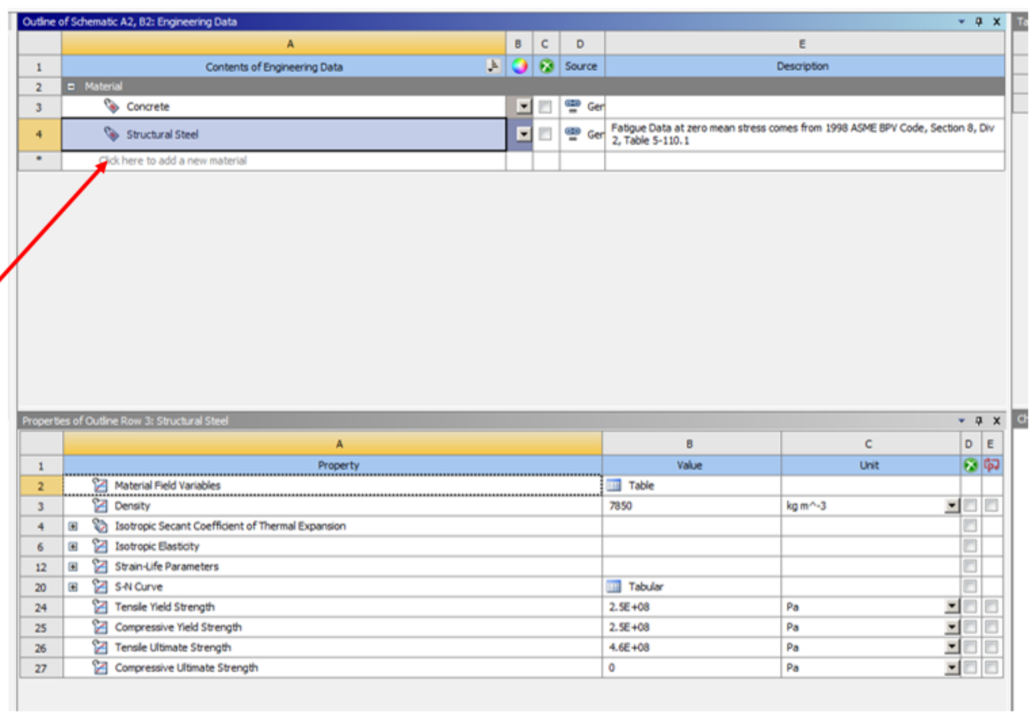
Click on "Engineering Data Source" to see that concrete has been added.



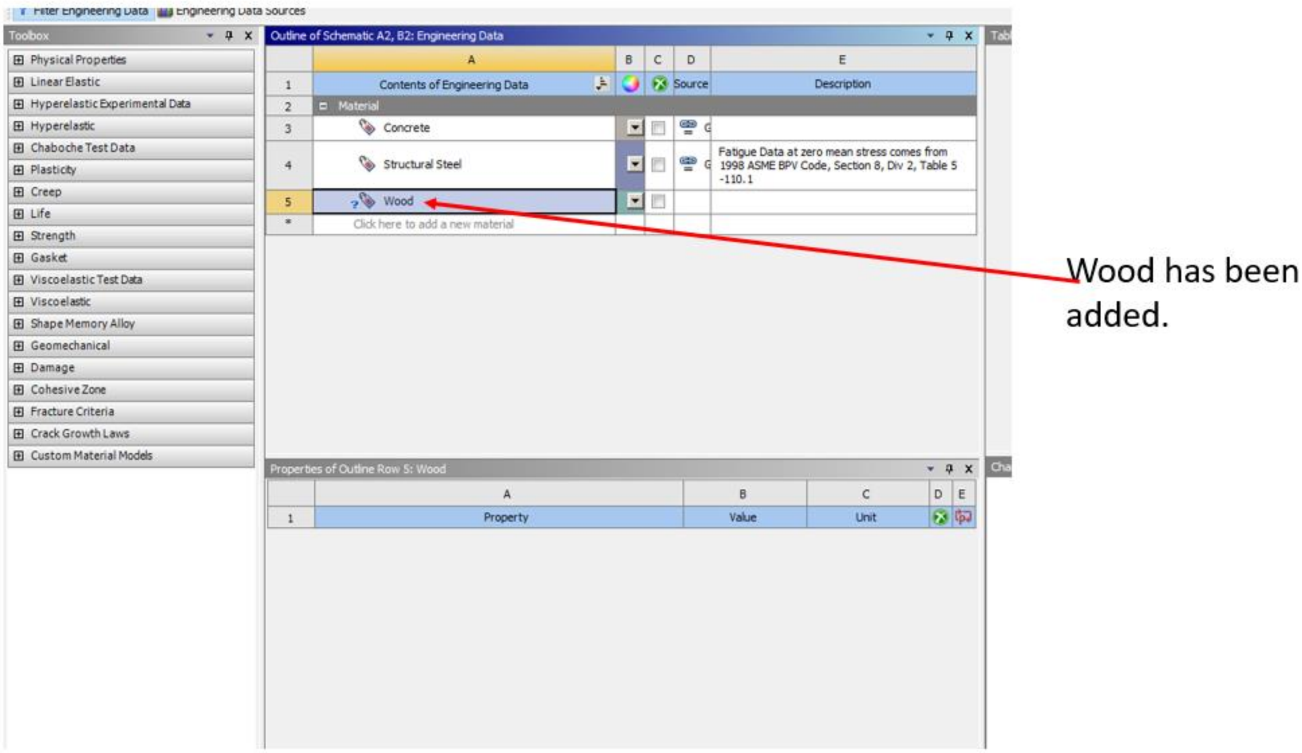
Concrete has been added.

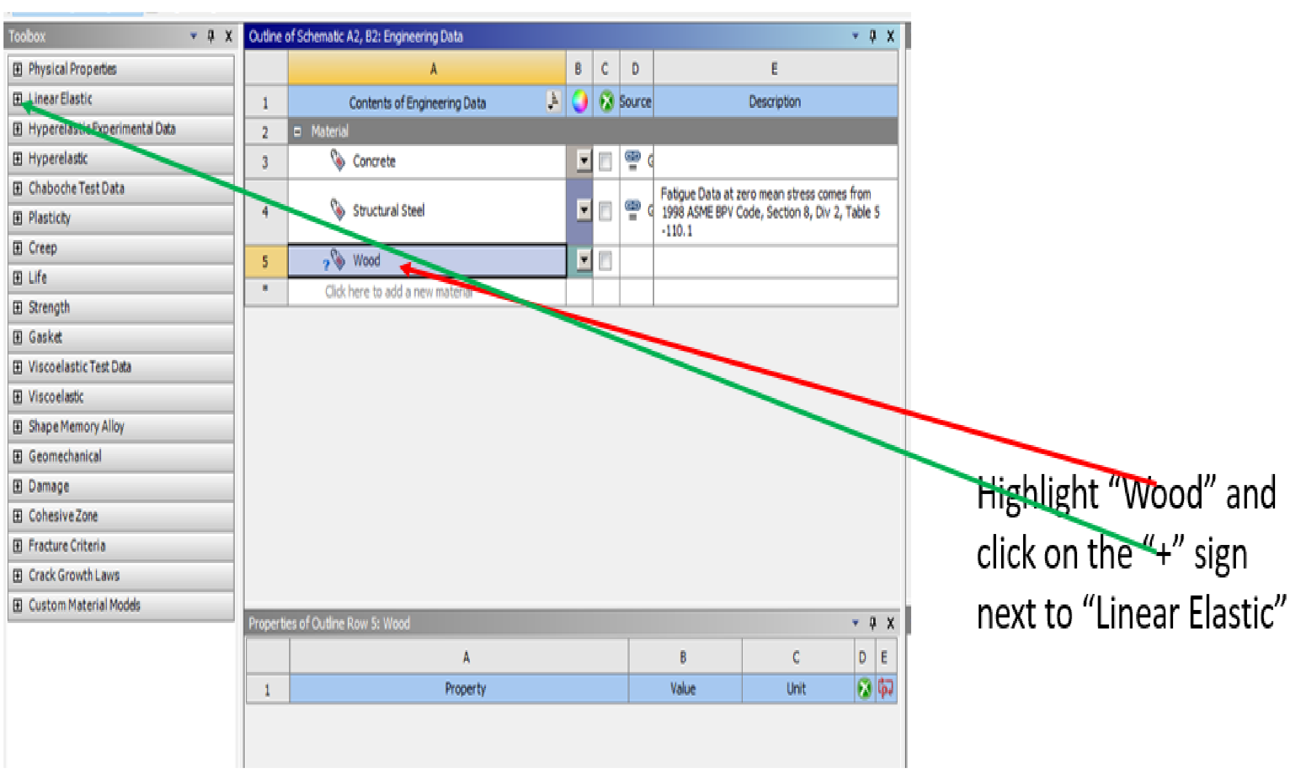


Material that is not a part of ANSYS data base can also be added by clicking here

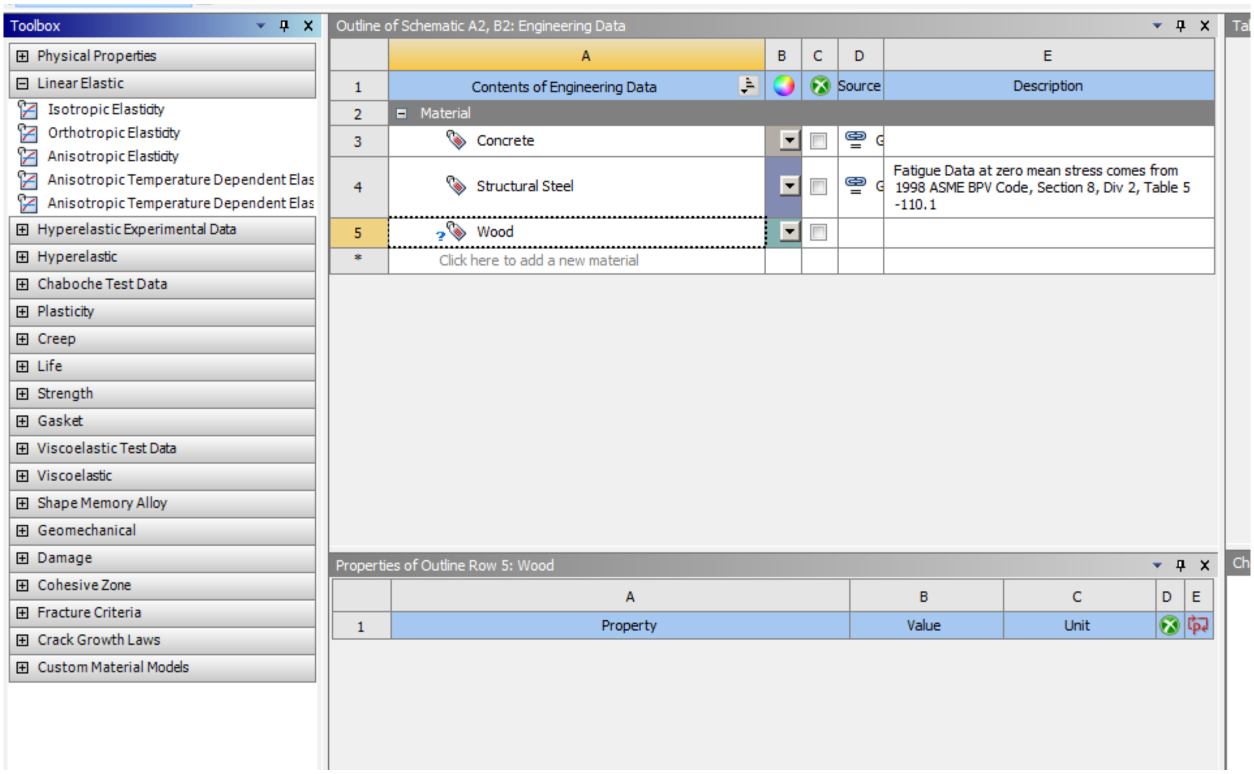


Click on the box shown above, and type wood. The following appears.

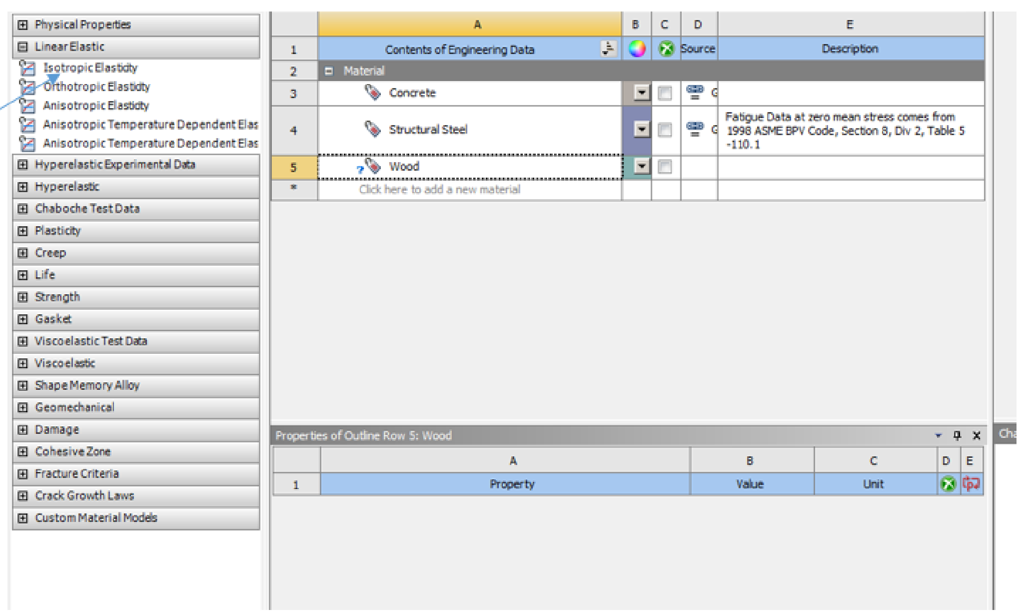




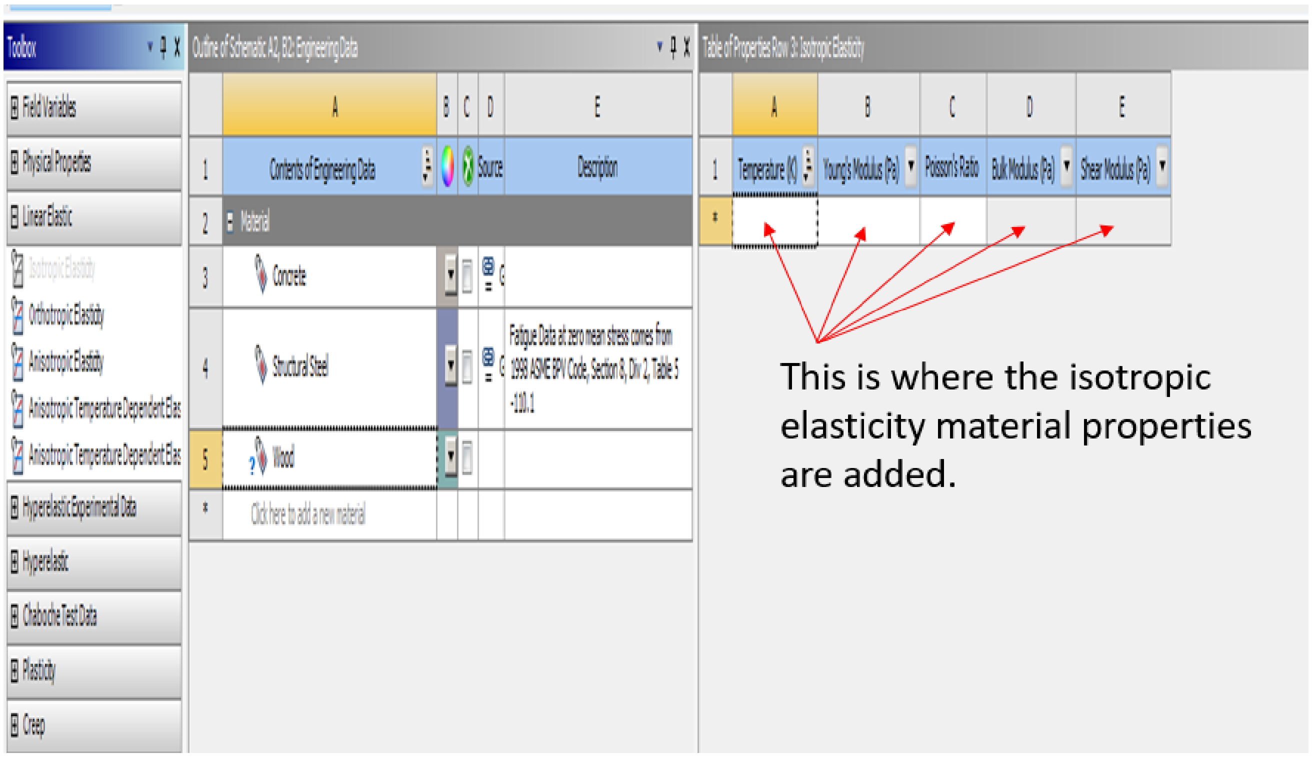
After clicking on the "+" sign next to "linear Elastic," the following appears.

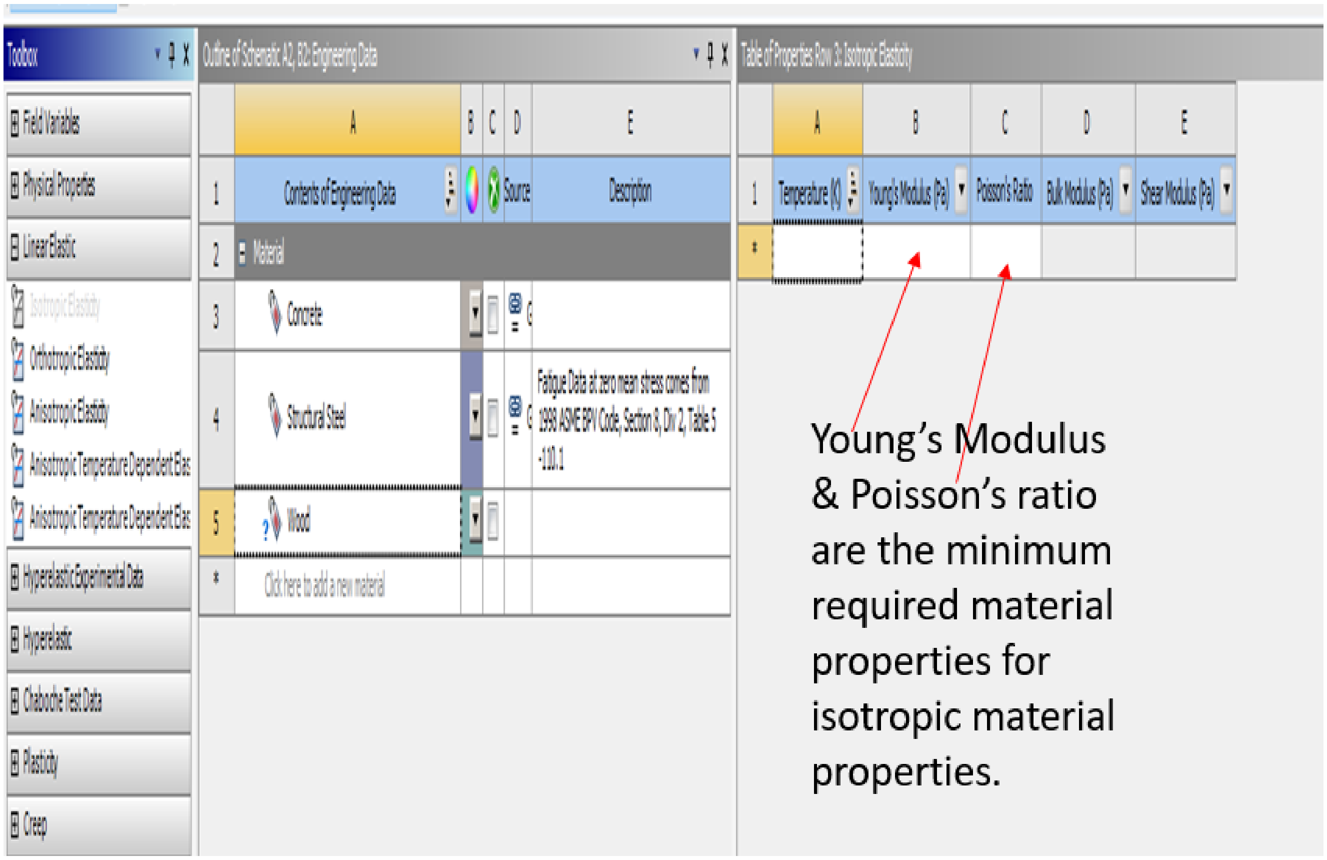


Double Click on "Isotropic Elasticity"



The following appears after double-clicking.





Recall from the problem description that “Young’s Modulus” and “Poisson’s ratio” are defined as shown below.

Youngs modulus of Wood = $\boxed{8 \times 10^9 \text{ Pa}}$

Poisson’s ratio of Wood = $\boxed{0.3}$

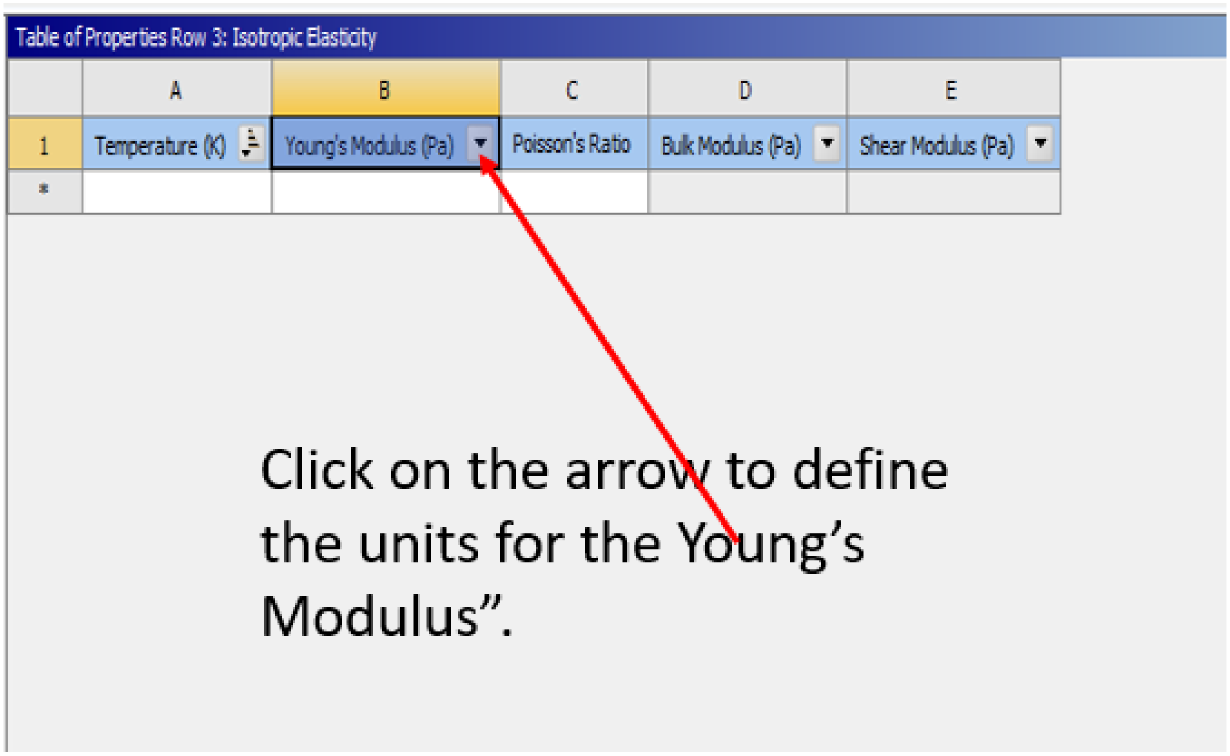
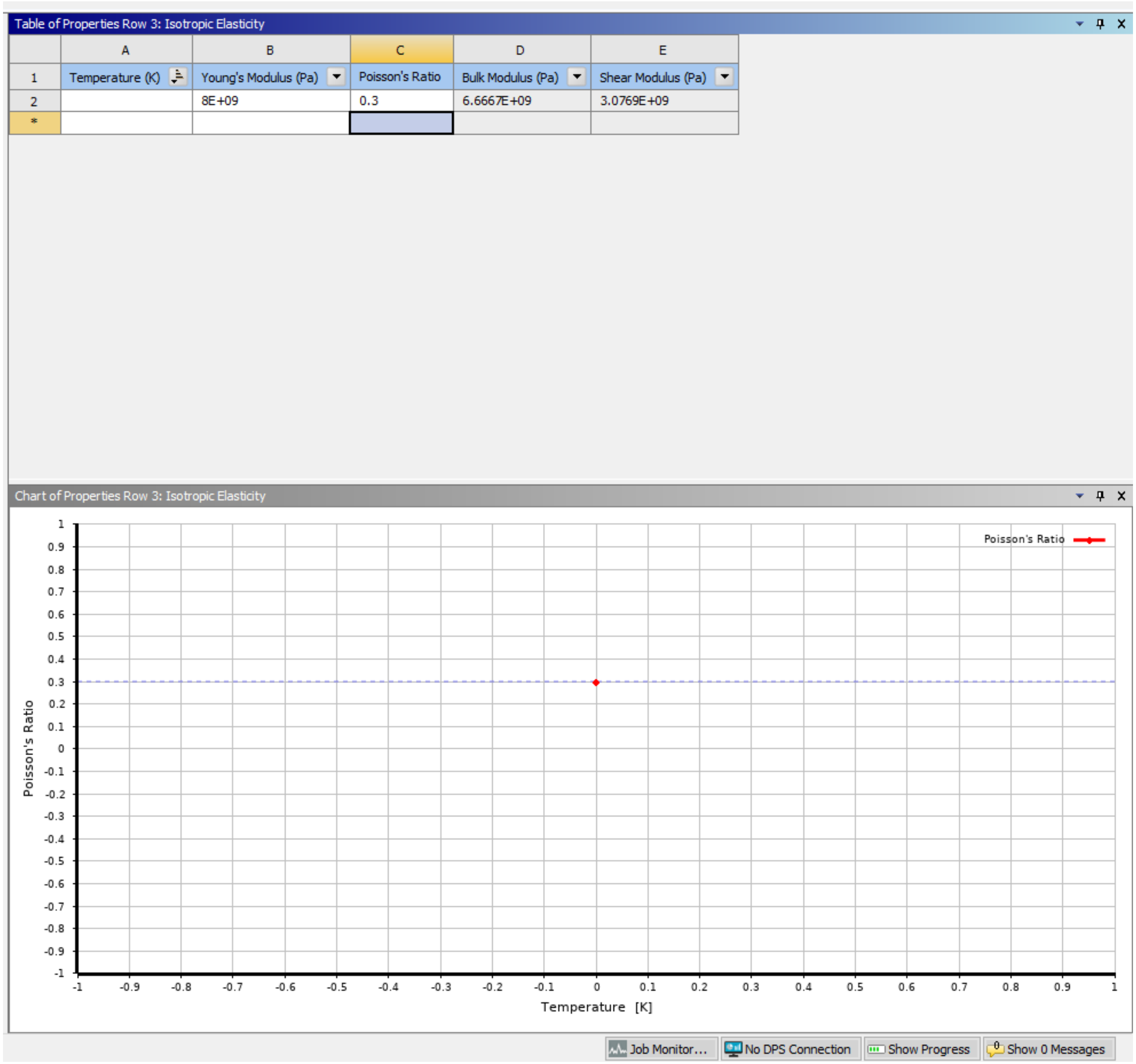


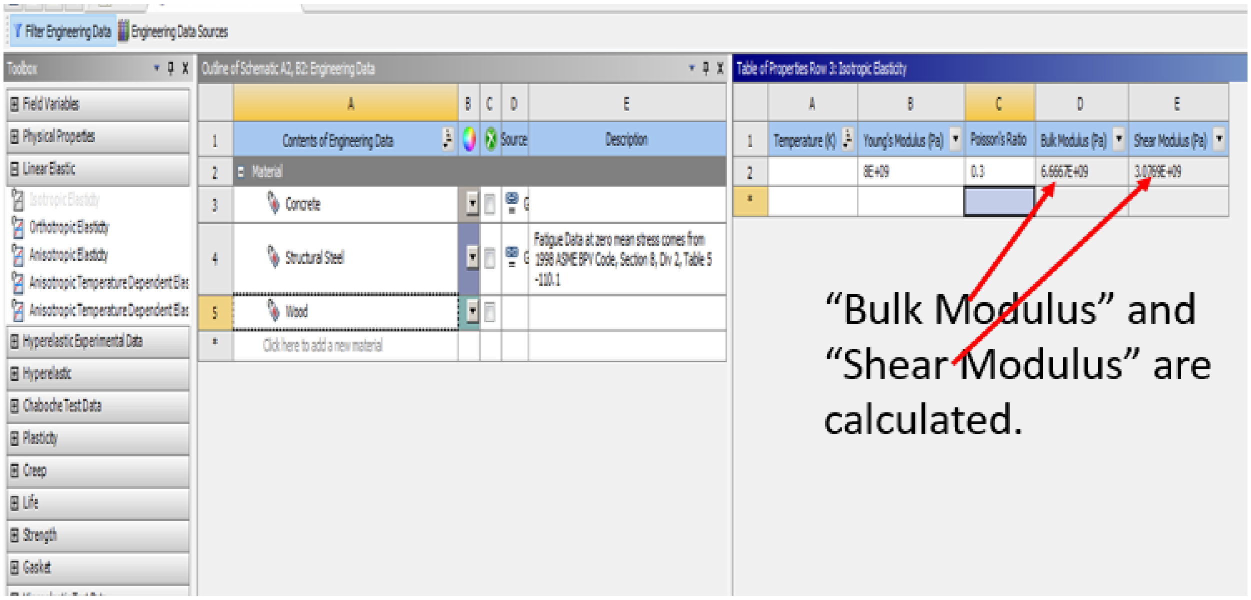
Table of Properties Row 3: Isotropic Elasticity
A
B
C
D
E
1
Temperature (K)
Young's Modulus (Pa)
Poisson's Ratio
Bulk Modulus (Pa)
Shear Modulus (Pa)
*
Click on the arrow to define the units for the Young's Modulus".

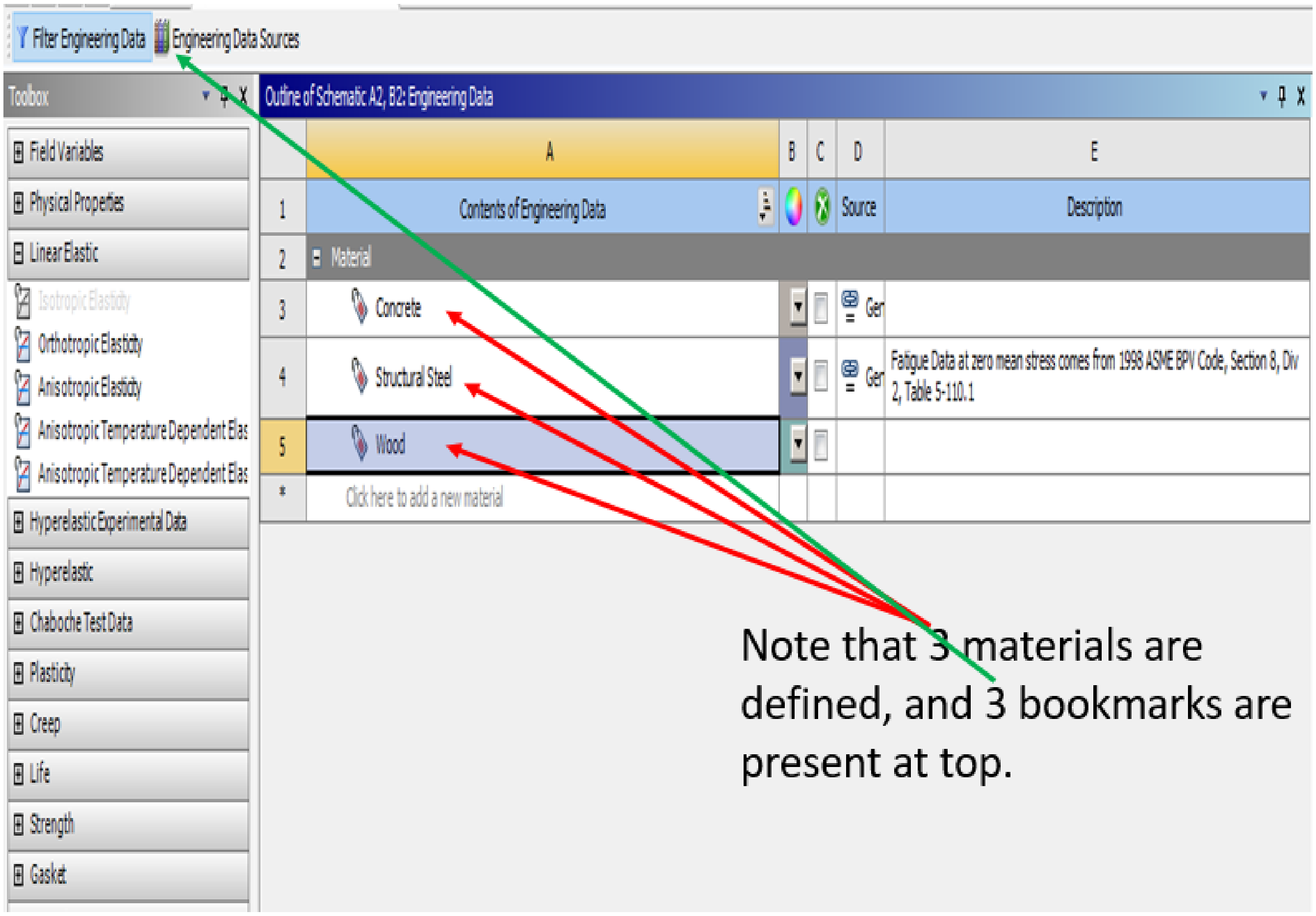
Fill in Young's Modulus and Poisson's ratio as shown below.



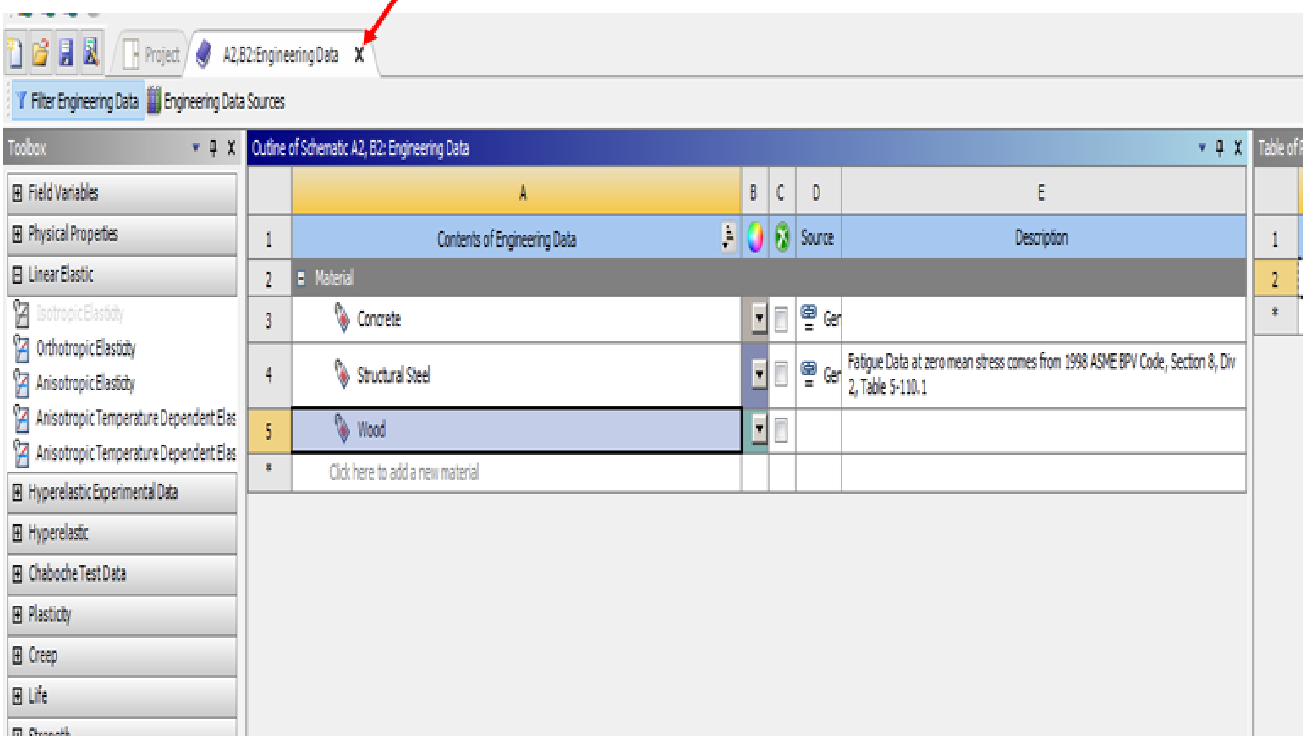
As seen above, Poisson's ratio can be filled as a function of temperature. However, for this example, Poisson's ratio is the same for all temperatures, and temperature is not even defined.

After defining “Young’s Modulus” and “Poisson’s ratio,” “Bulk Modulus” and “Shear Modulus” are automatically calculated in units of Pa, which was the input unit for “Young’s Modulus.”

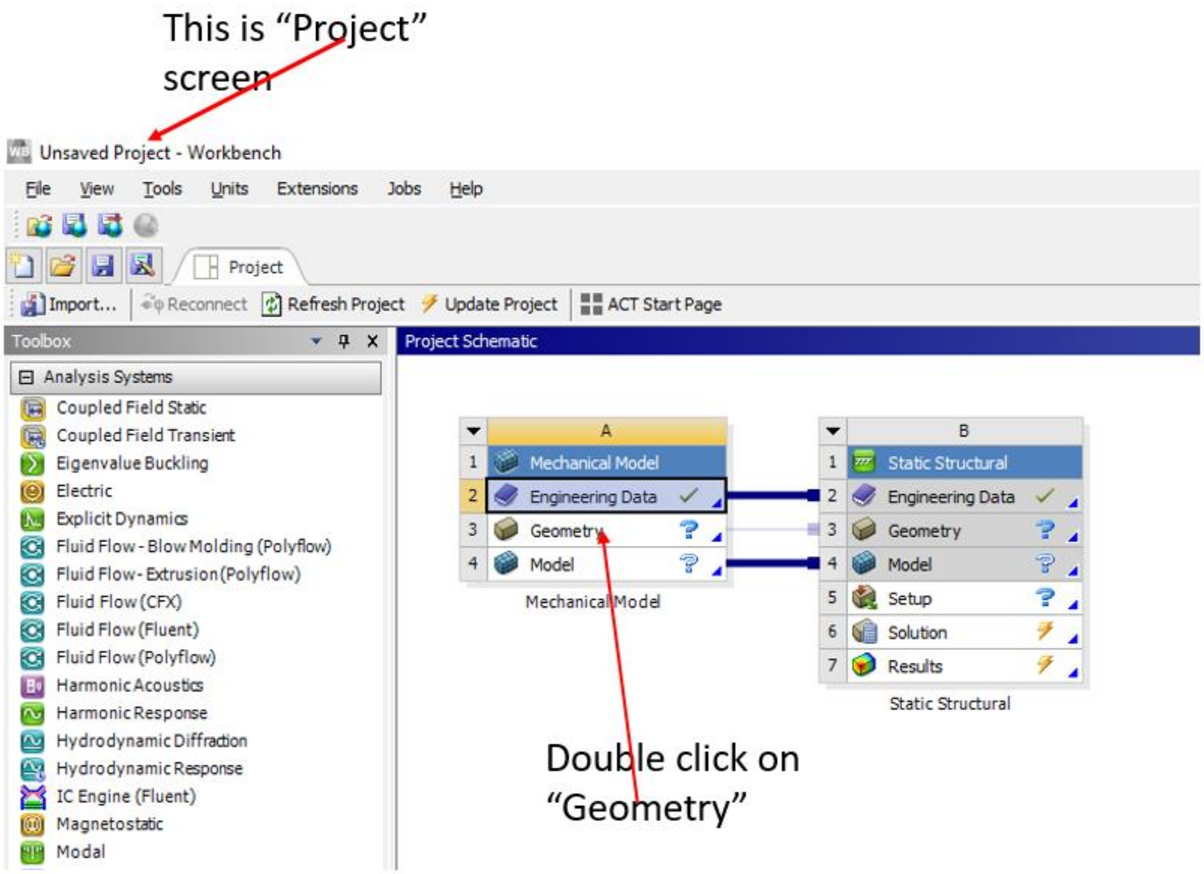



Filter Engineering Data
Engineering Data Sources
Toolbox
Outline of Schematic A2, B2: Engineering Data
Field Variables
Physical Properties
Linear Elastic
Isotropic Elasticity
Orthotropic Elasticity
Anisotropic Elasticity
Anisotropic Temperature Dependent Elas
Anisotropic Temperature Dependent Elas
Hyperelastic Experimental Data
Hyperelastic
Chaboche Test Data
Plasticity
Creep
Life
Strength
Gasket
A
B
C
D
E
Contents of Engineering Data
Source
Description
Material
Concrete
Structural Steel
Wood
Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
Click here to add a new material
Note that 3 materials are defined, and 3 bookmarks are present at top.

Click on “X” to close the “Engineering Data” tab.

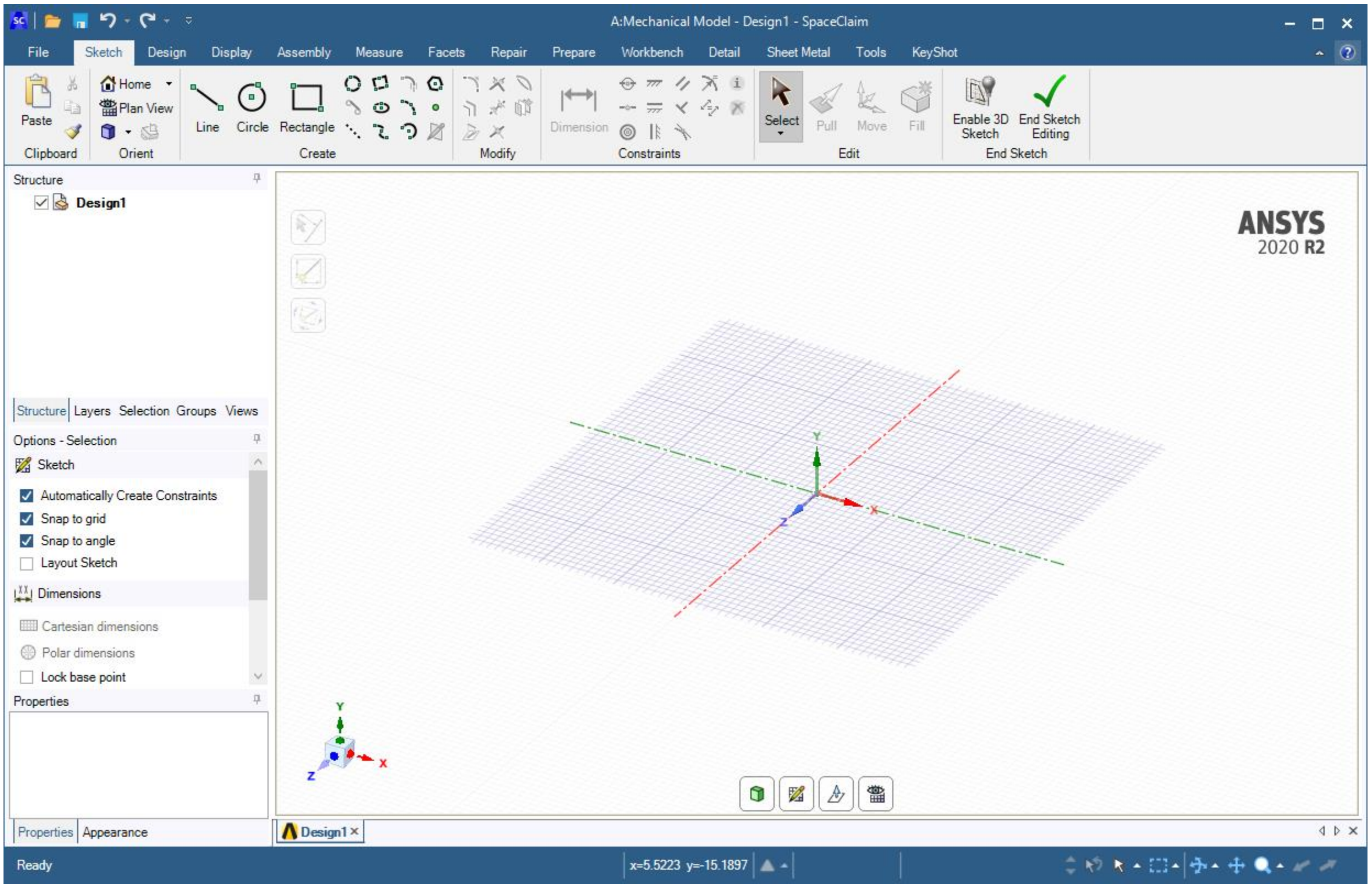


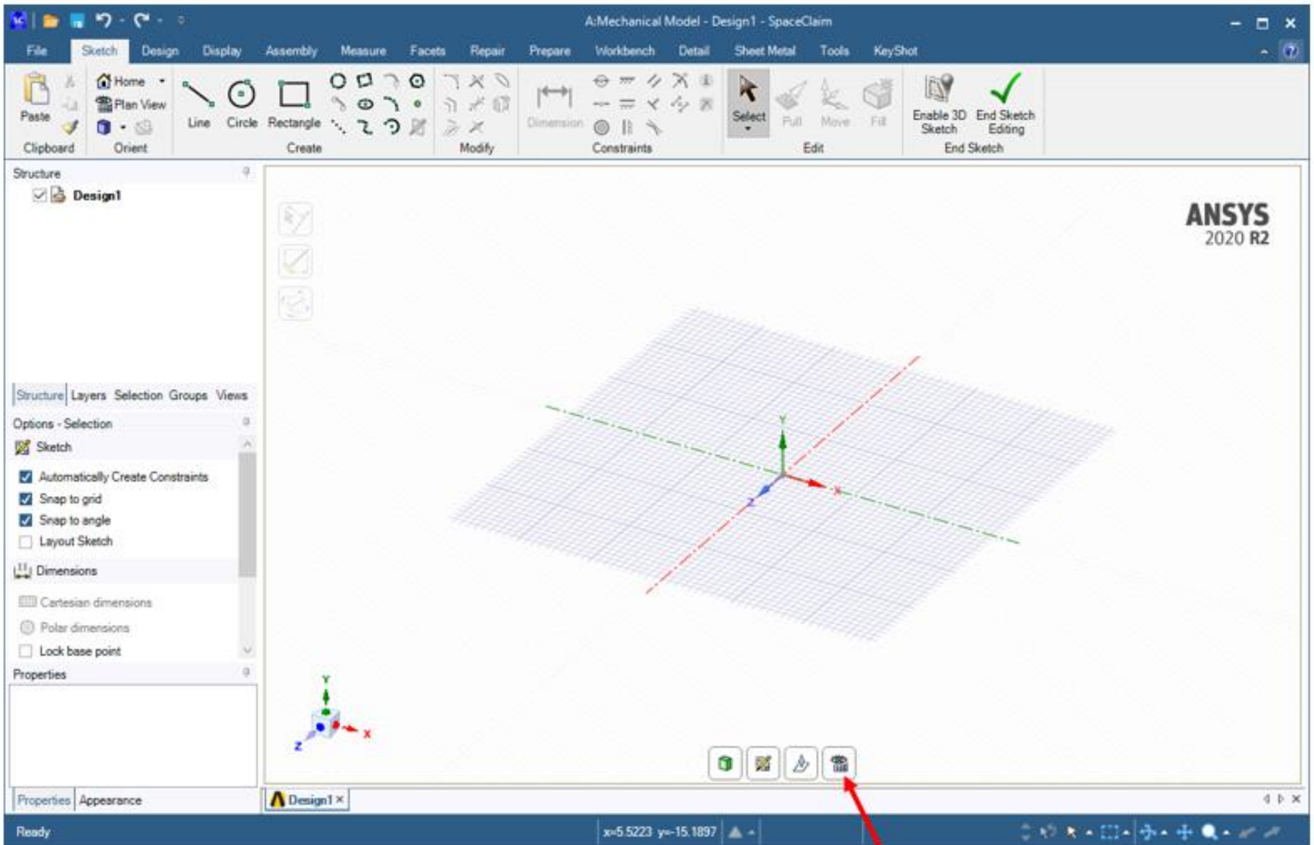
On the projects screen, double-click on “Geometry.”



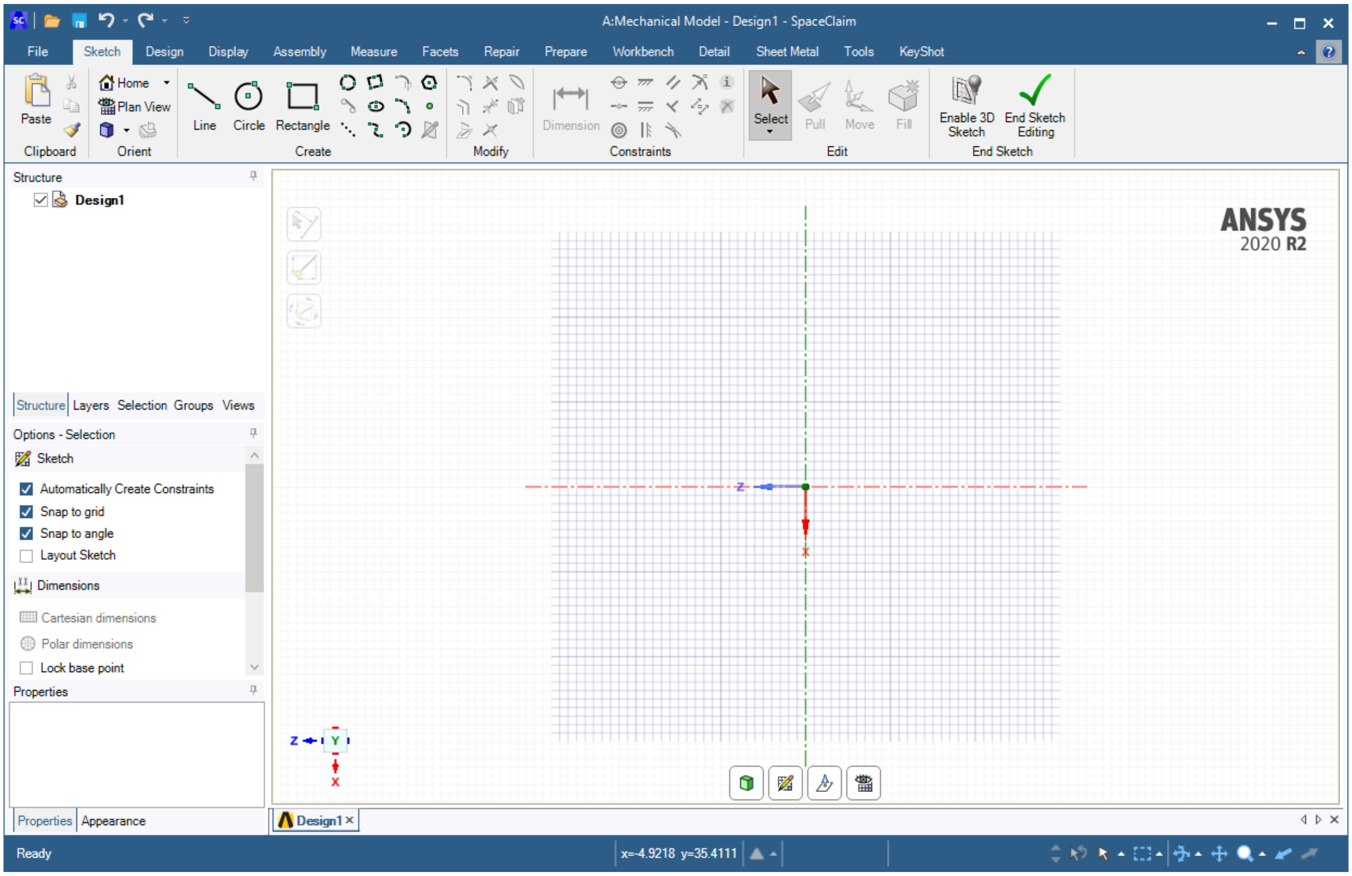
After "Double Clicking" on Geometry, a new window, as shown below, will pop up. The appearance of the new window will take a while.

This new window will be the third independent screen that will appear at the bottom of the screen. The name of this new window at the bottom of the screen is "SC." "SC" stands for "Space Claim."

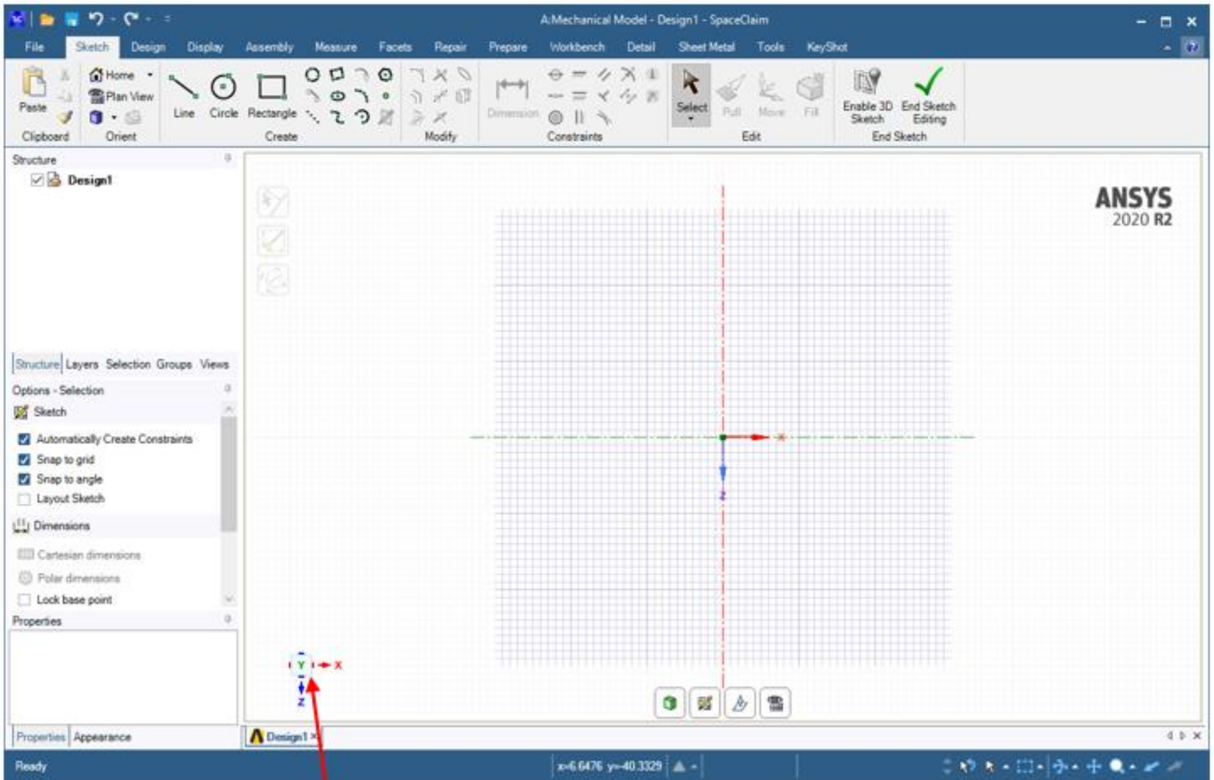




Click on this to change
the coordinate to XZ as
shown below.

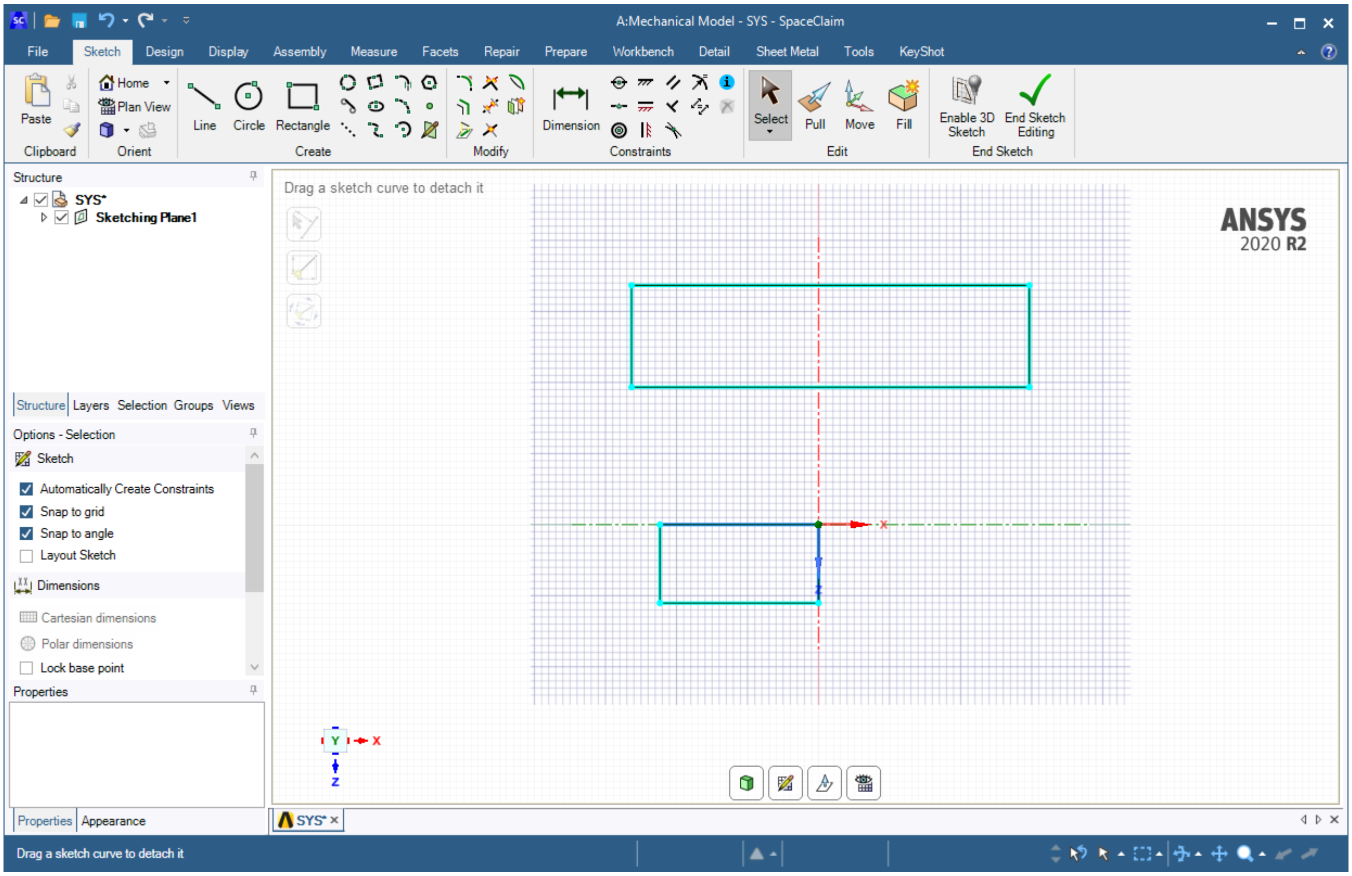


The wheel on the mouse can be used to zoom in and out.

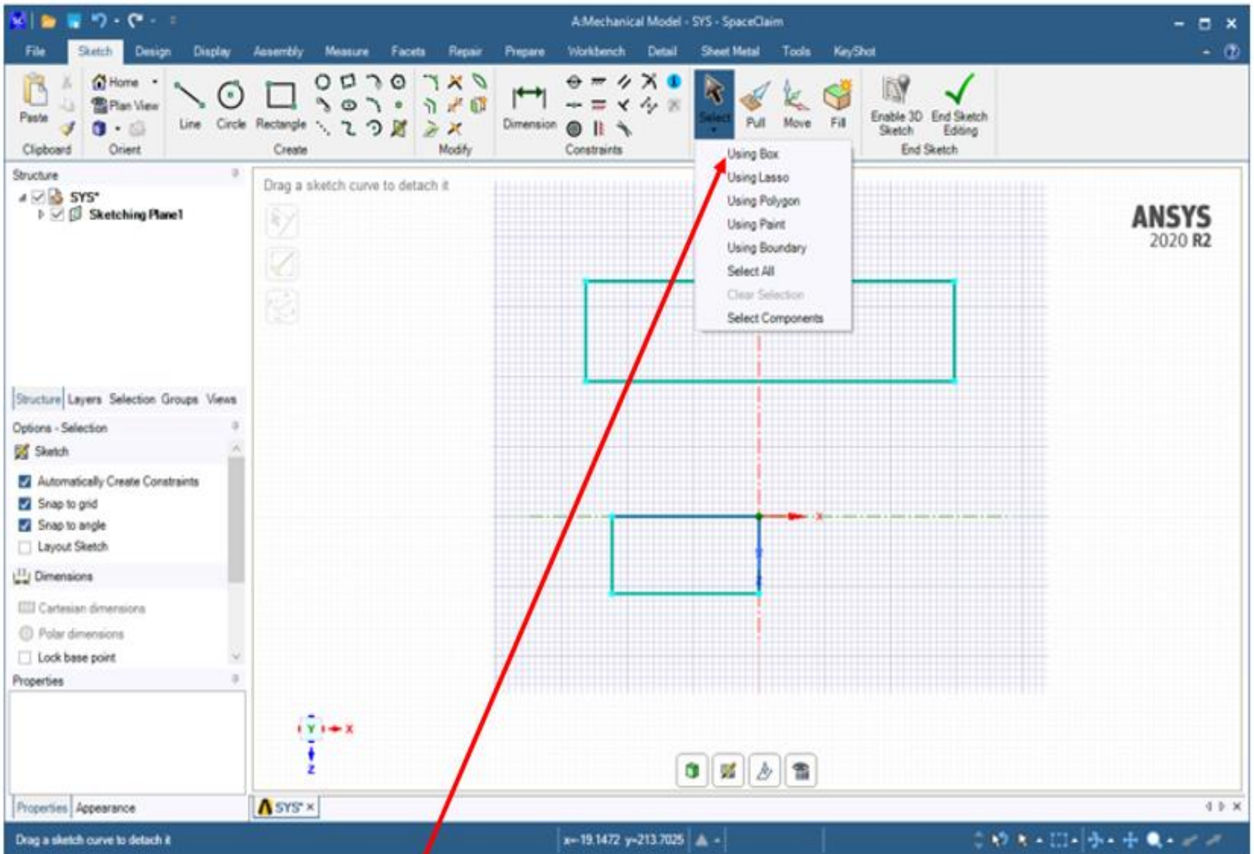


The sketching plane can be changed by clicking on this.

The following shows how to erase entities once created. Consider the following screen, which is independent of the FE example problem. A mistake was made during the process of repeating what the YOUTUBE video said, and this was created as a result of attempting to correct the mistake.



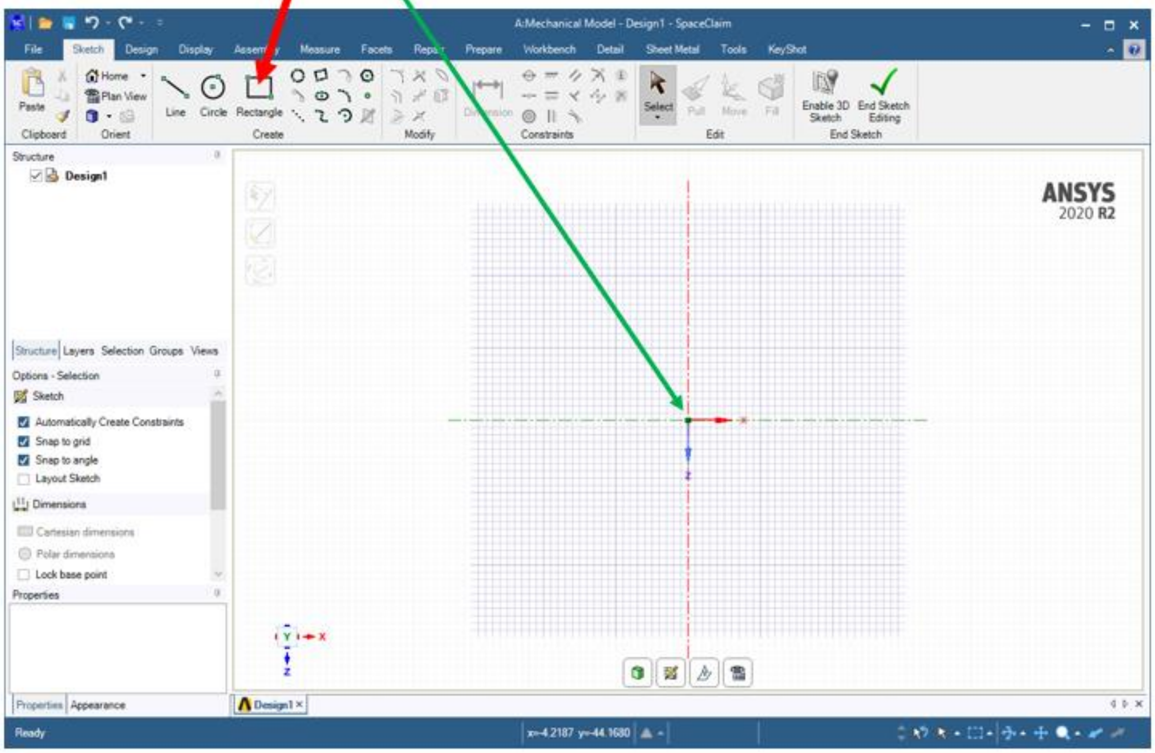
Click on “Select” in the “Edit” box, as shown below.



Then select using for example “Using Box” option. Then click the “Delete” button on the keyboard.

The sketch screen will appear with nothing on it.

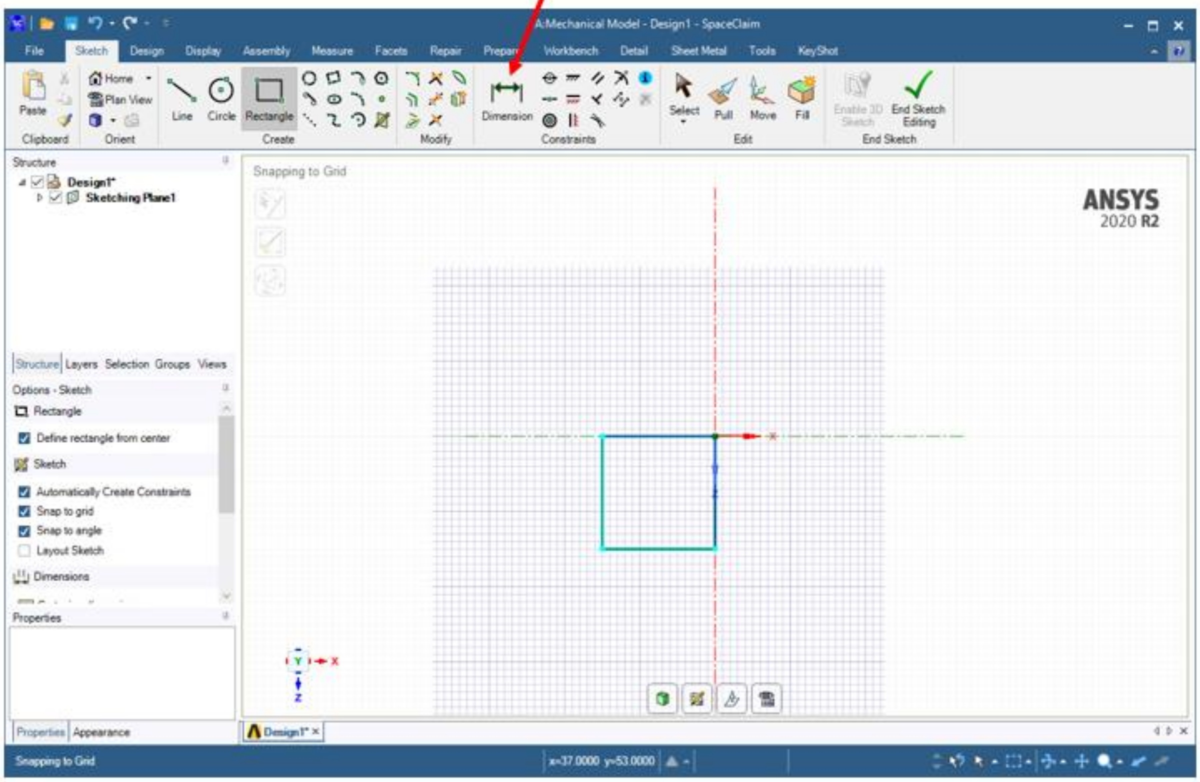
Create a rectangle by clicking on this. Then clicking here. Then dragging.



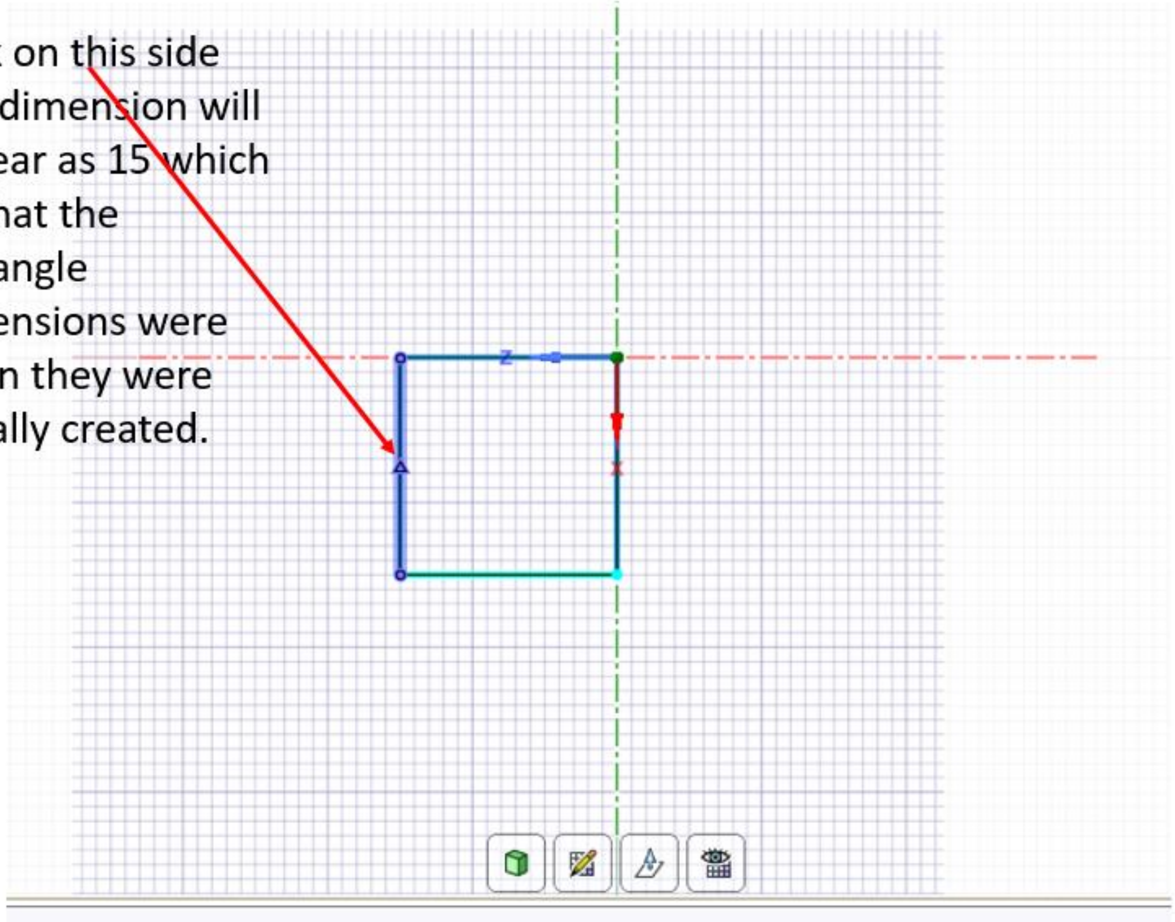
Once the rectangle is created, hit the “Esc” button.

The following shows the screen after the rectangle has been created.

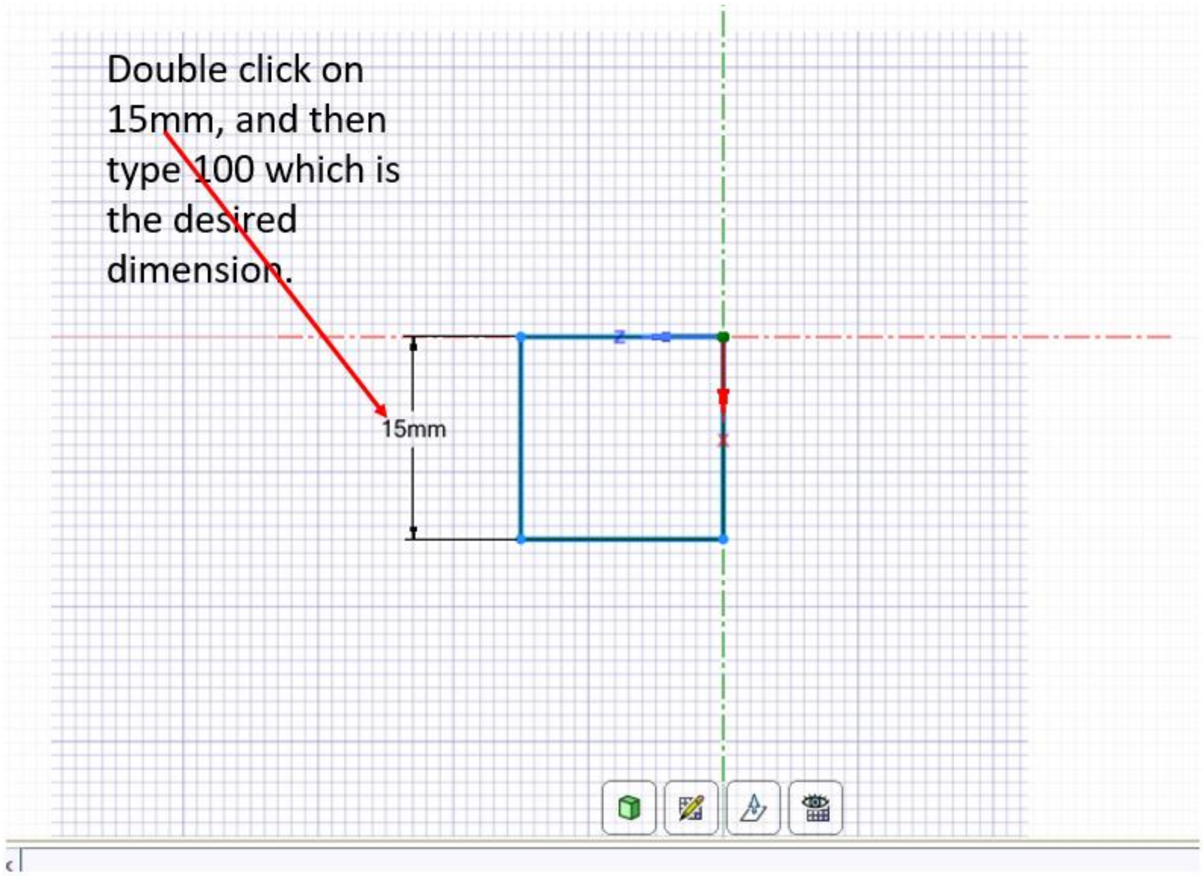
Click on “Dimension” to dimension the newly created rectangle.



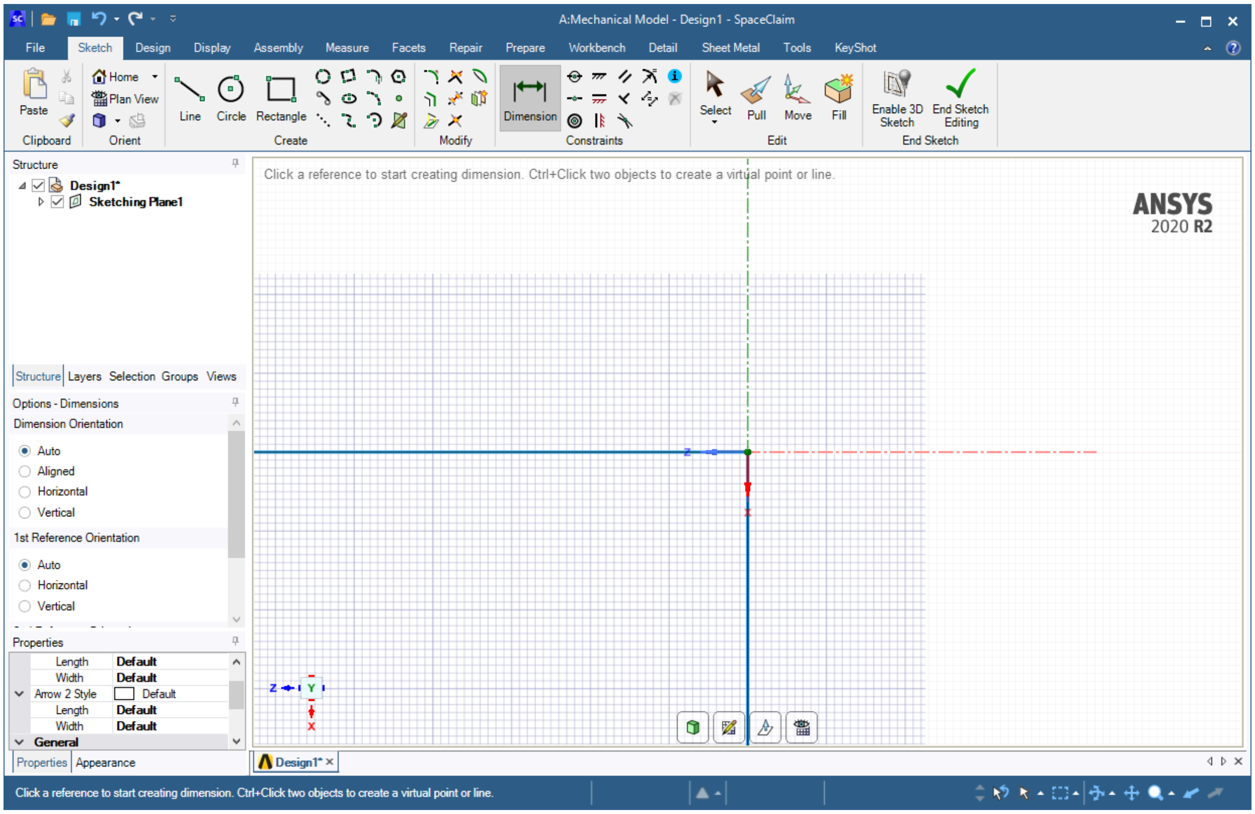
Click on this side and dimension will appear as 15 which is what the rectangle dimensions were when they were initially created.



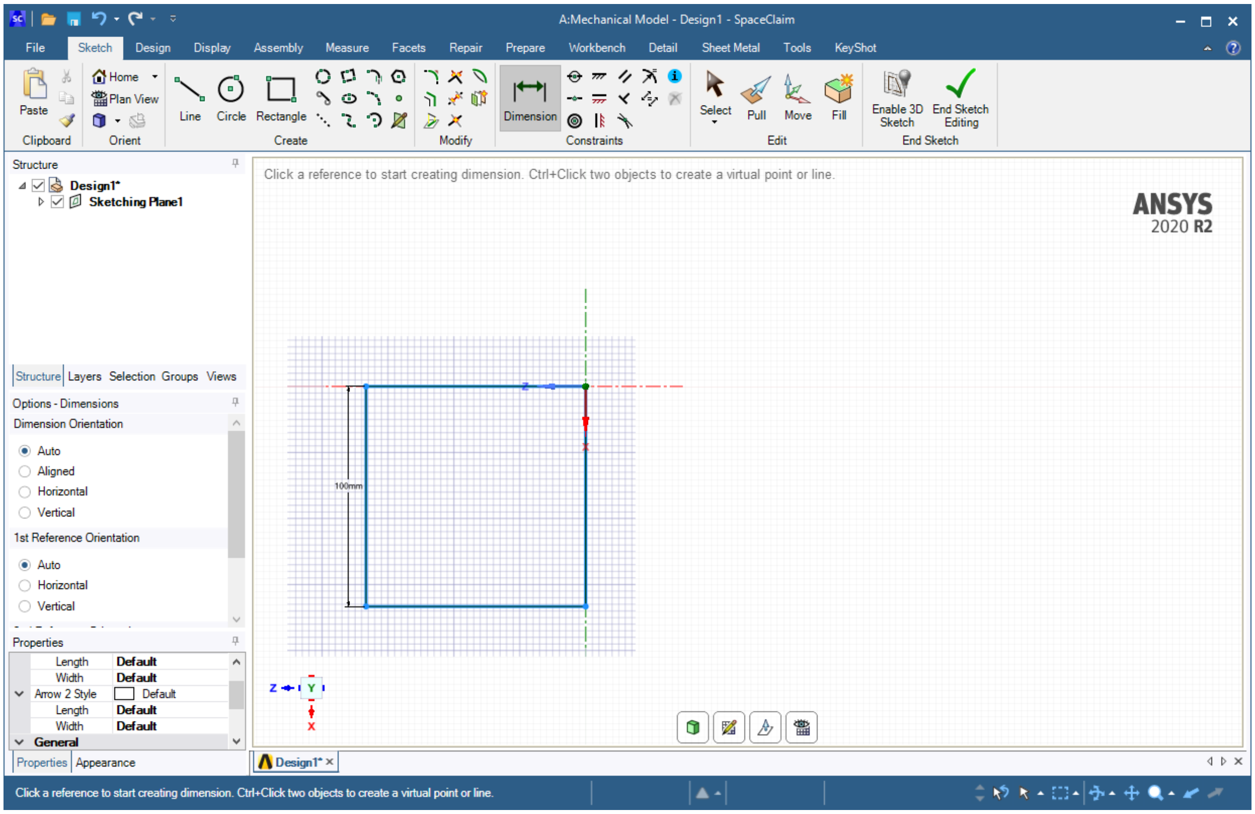
The screen will appear as shown below.



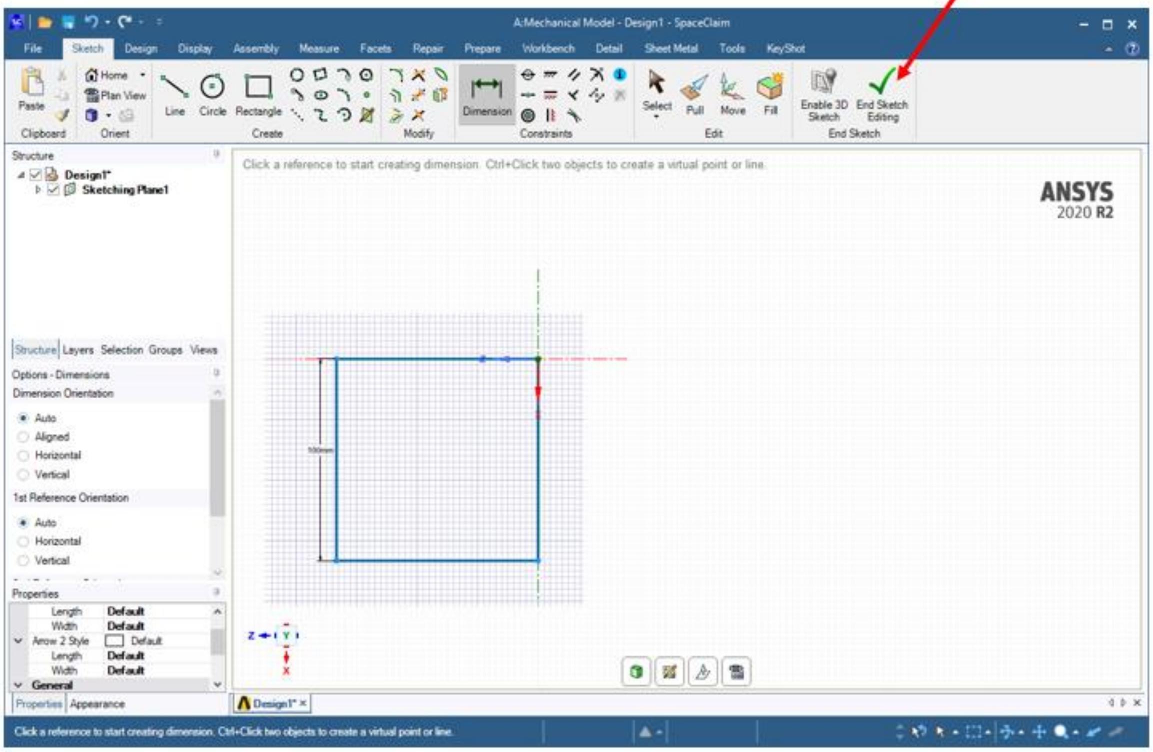
After changing the 15 to 100, the rectangle won't fit, and the screen appears as shown below.



Use the mouse wheel and fit the rectangle as shown below.

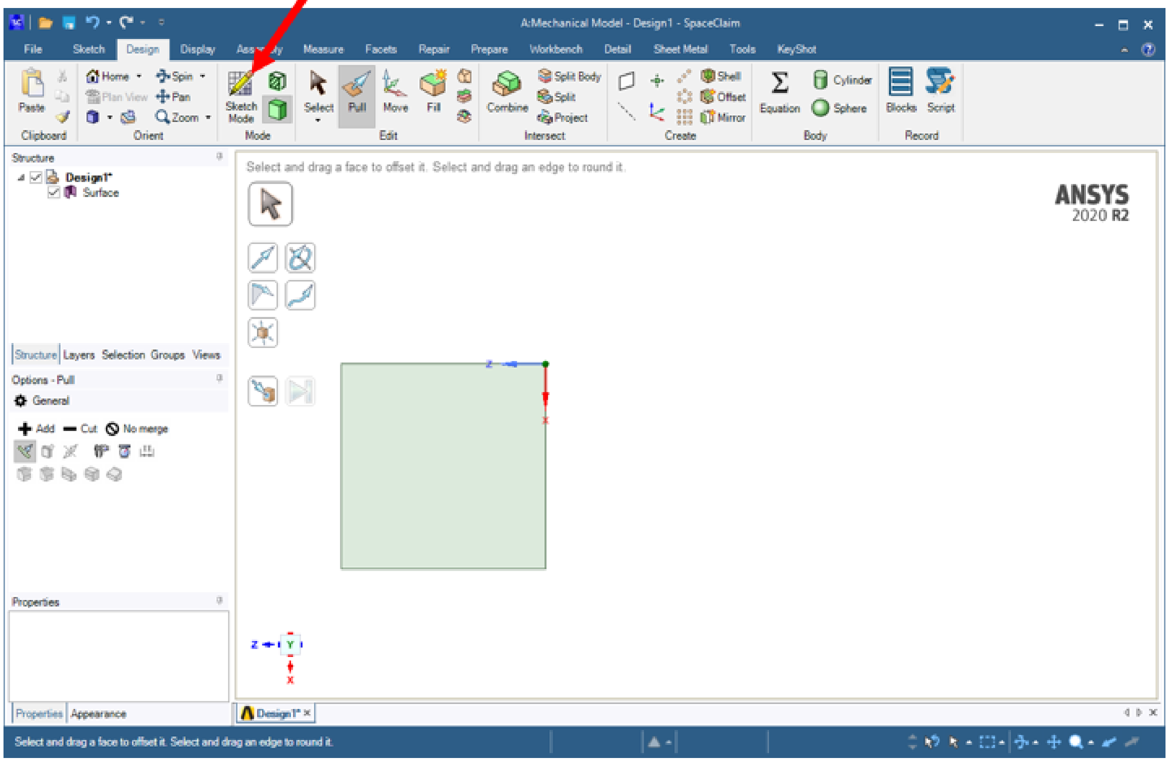


Click on “End Sketch Editing”

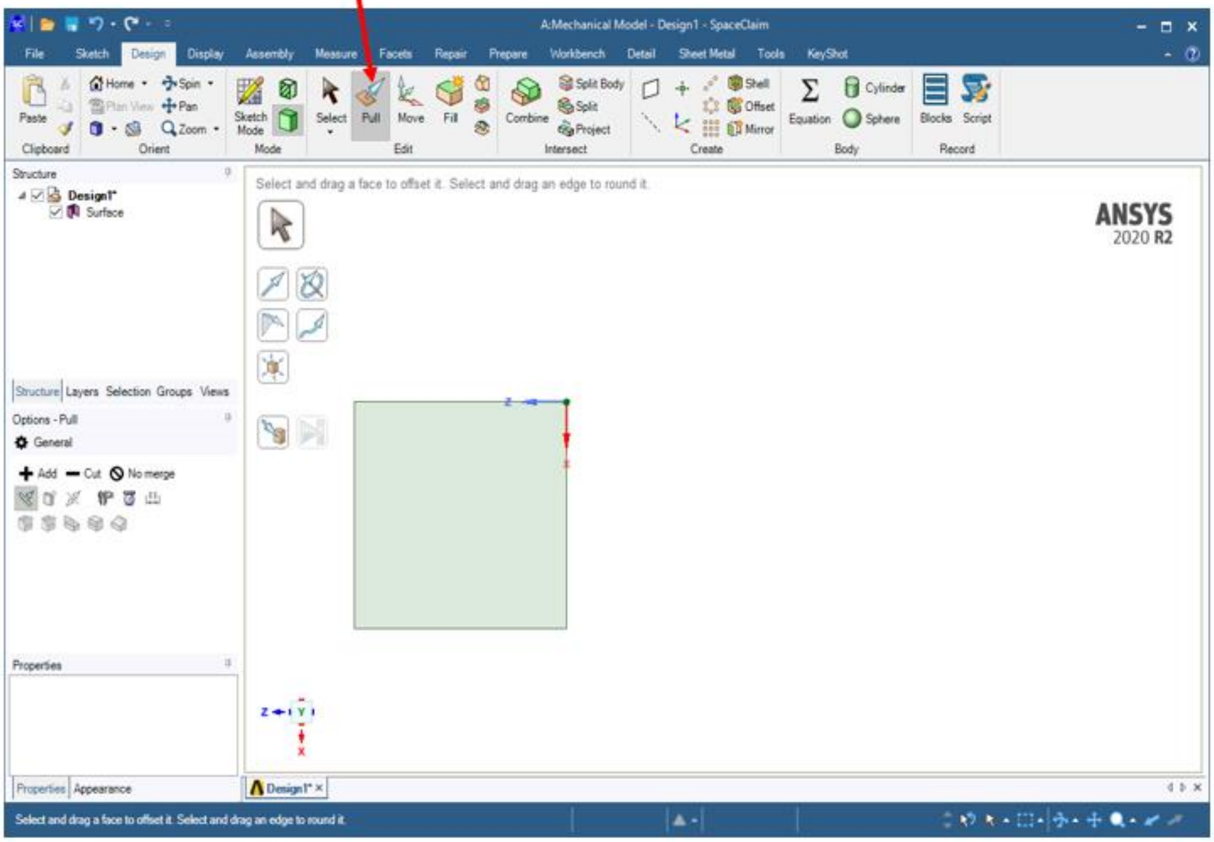


Clicking on “End Sketch Editing” will bring up the following screen, as shown below.

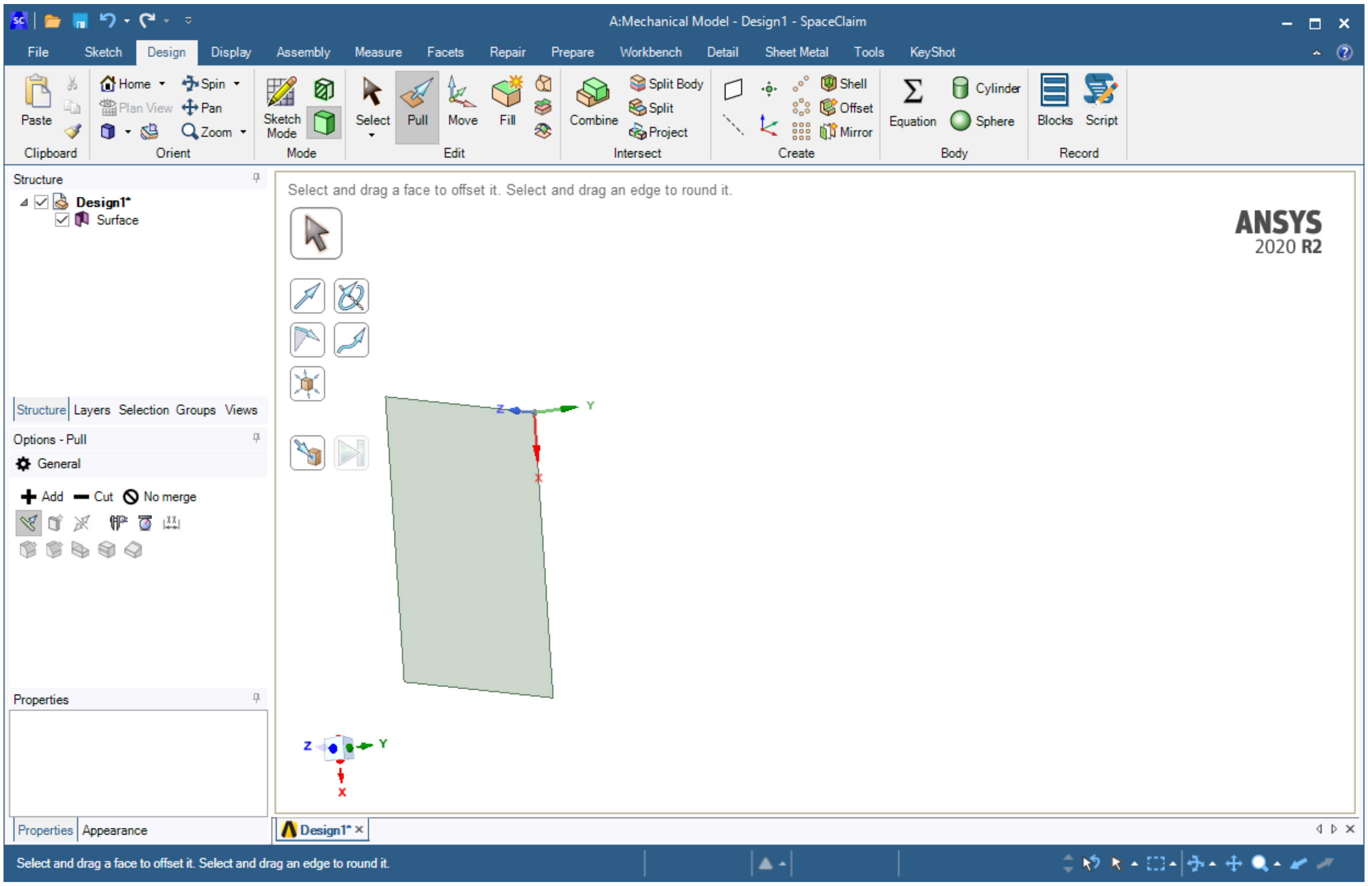
Clicking on “Sketch Mode” will bring up the previous screen. Do this only if necessary.



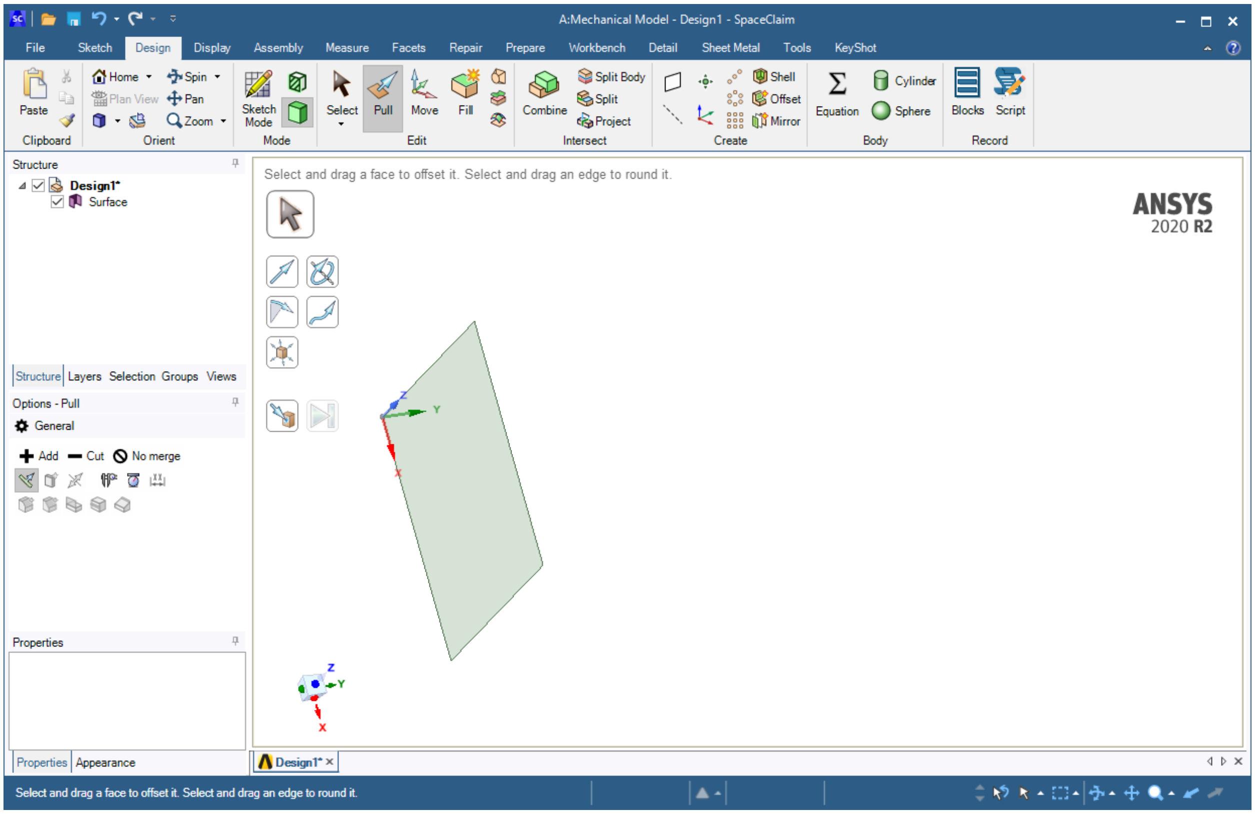
At this point, the “SC” screen is in “Pull” tab.

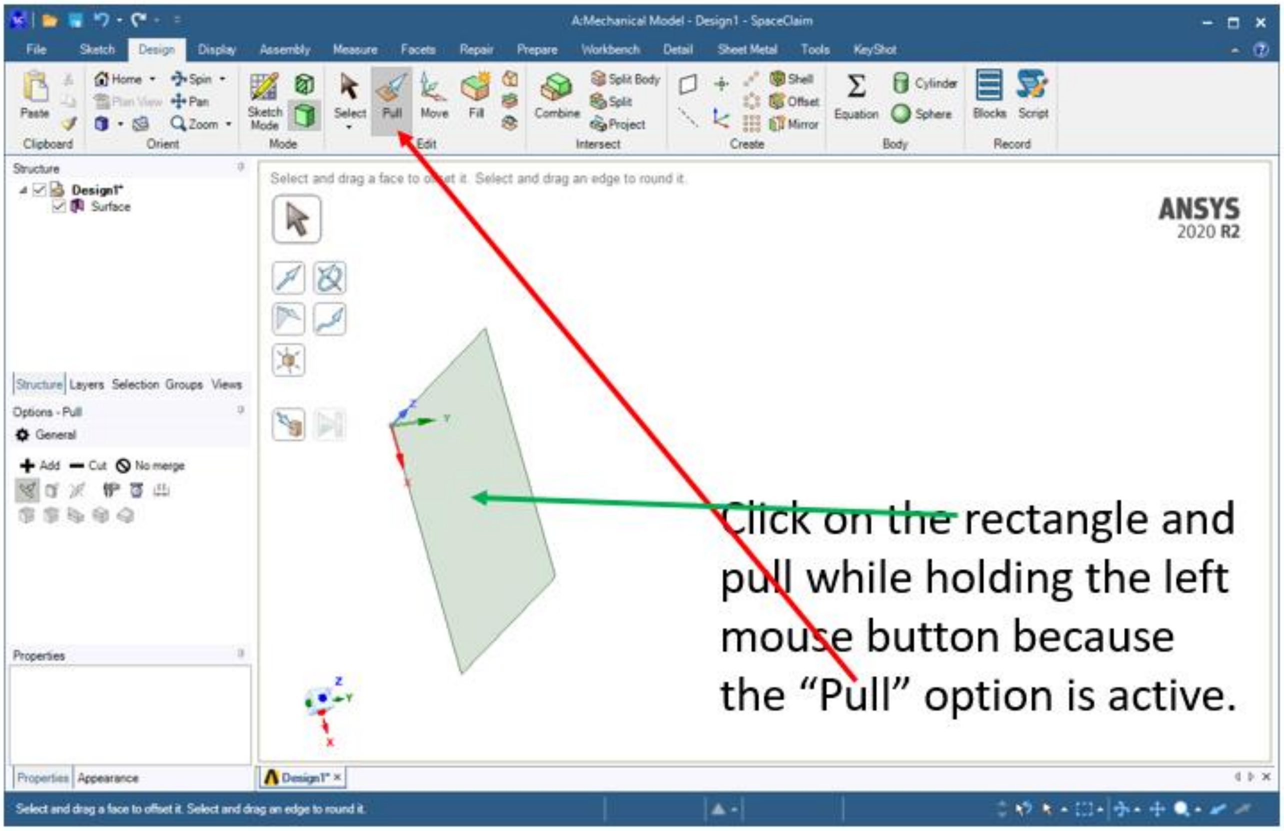


Clicking on the mouse wheel and moving the mouse pointer on the screen will rotate the rectangle, as shown below. An example is shown below.



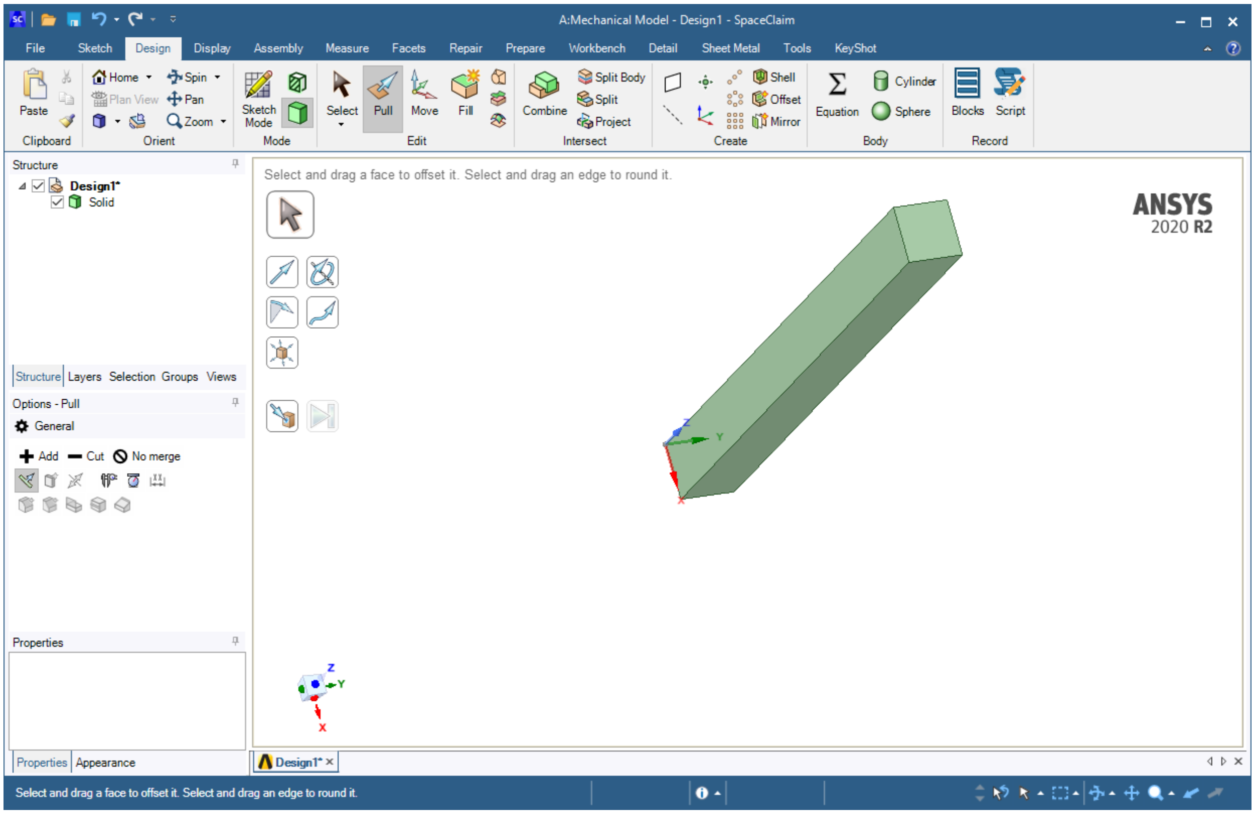
Rearrange the screen as shown below.





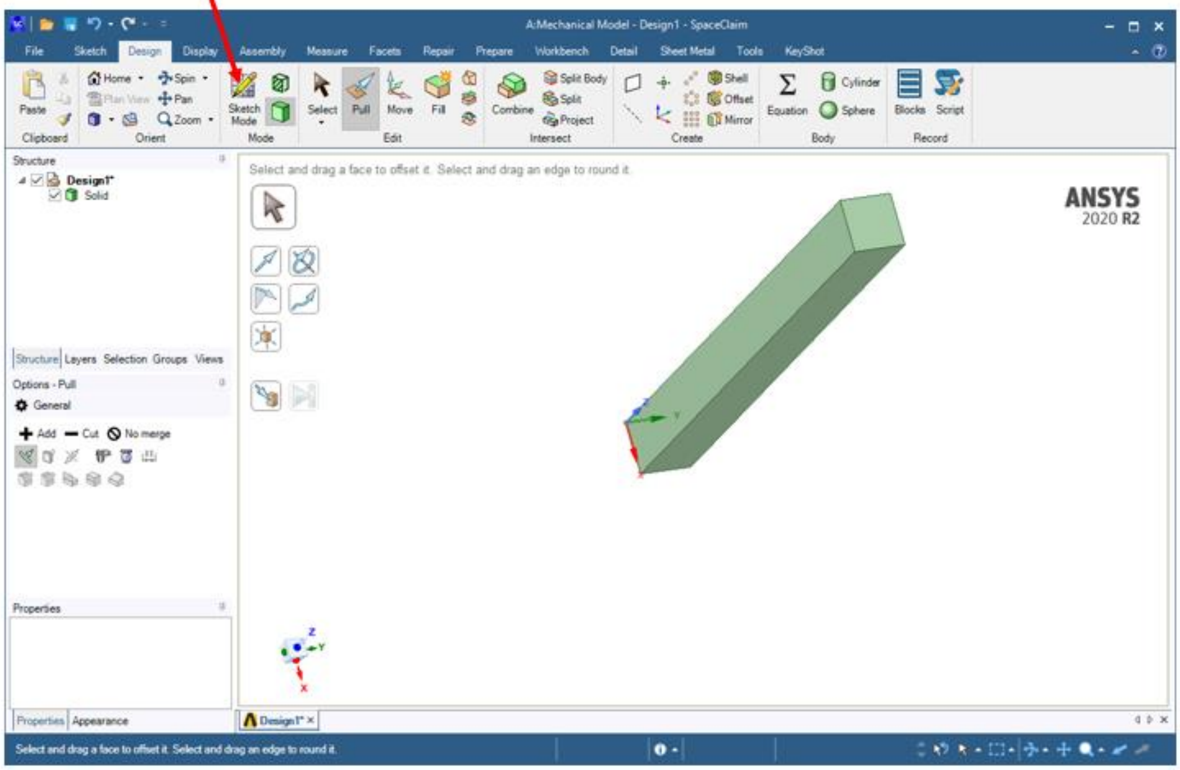
As it is pulled while holding the "left mouse button," type 1000, which is the length. The following appears.

The following is a beam with a length of 1000 mm and a rectangular cross-section with a side of 100 mm.

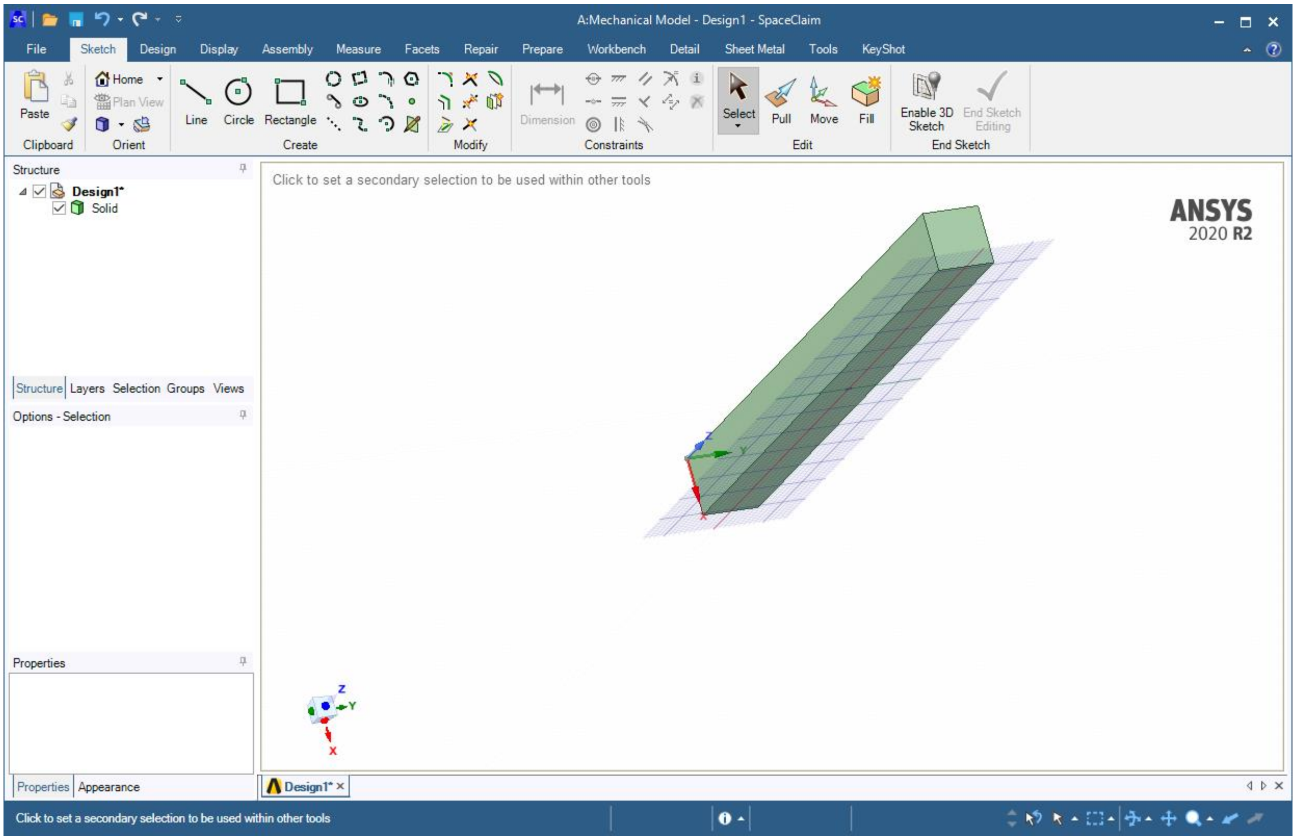


A section must be defined for the monkey to sit on.

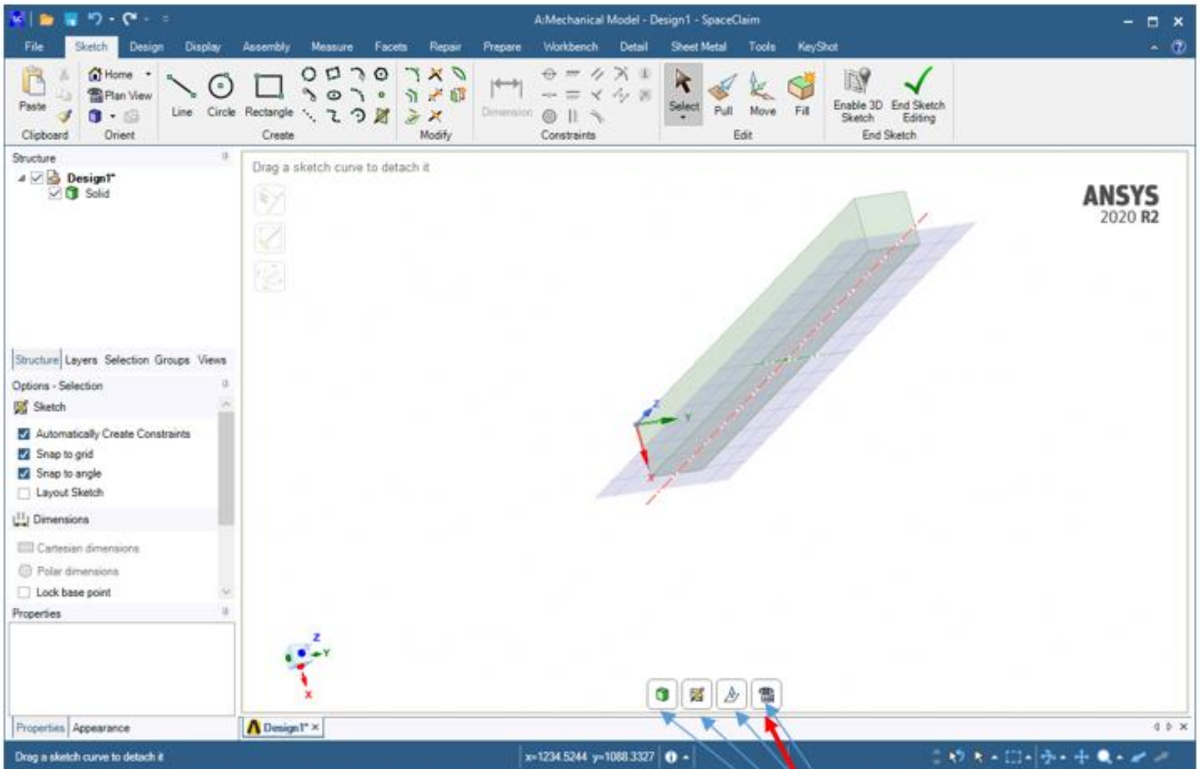
Click on “Sketch Mode” to go to sketching screen.



After entering “Sketch Mode,” the screen appears as shown below.

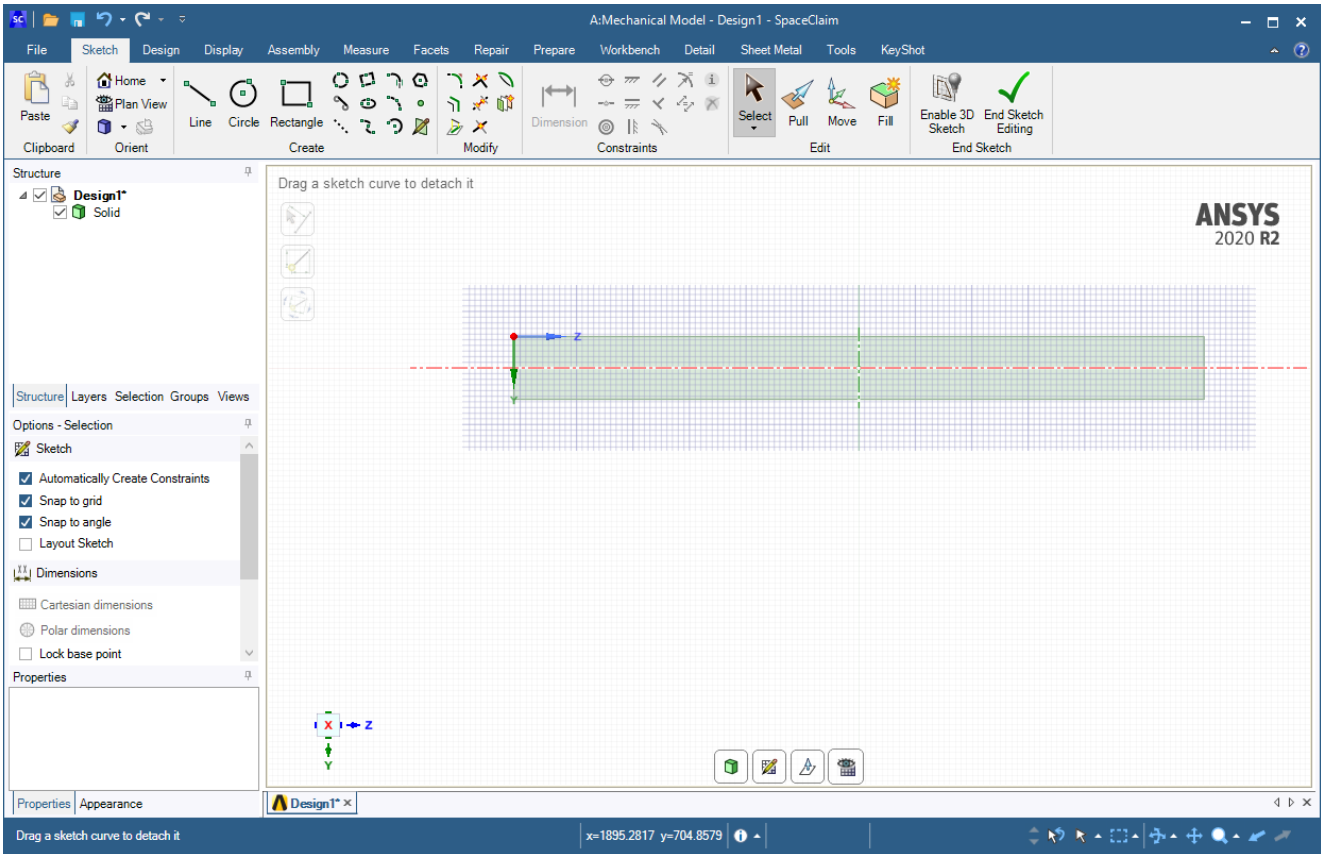


Select a face by putting the mouse pointer on it and clicking on it.

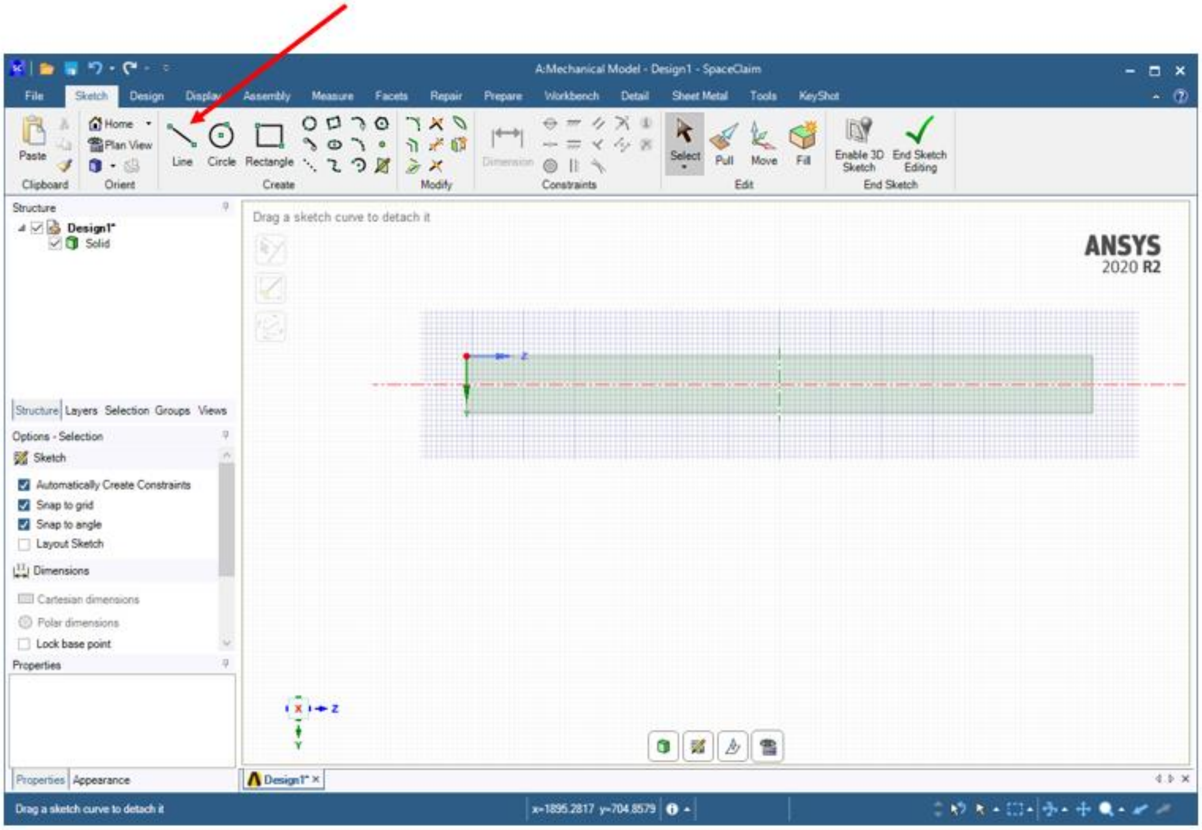


Once the face is selected, these buttons appear. Click on this button to get a top view of the selected face.

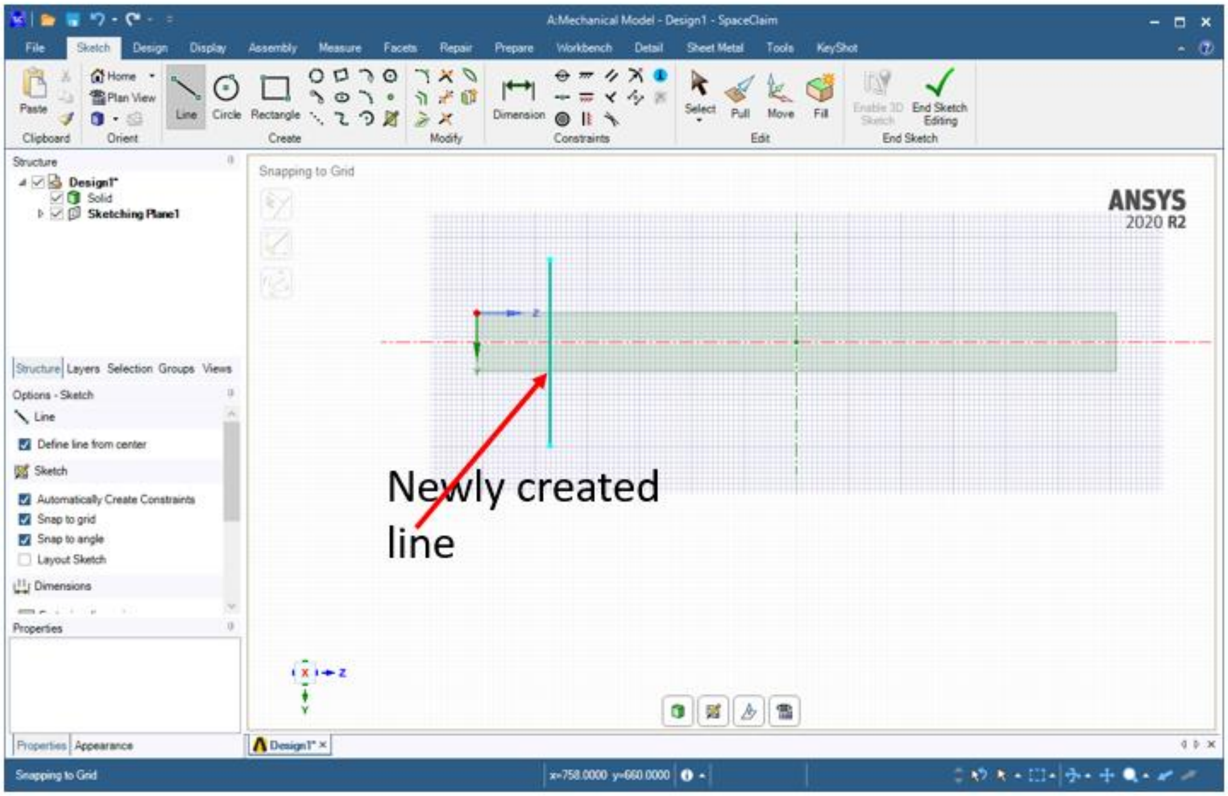
The “SC” screen appears as shown below.



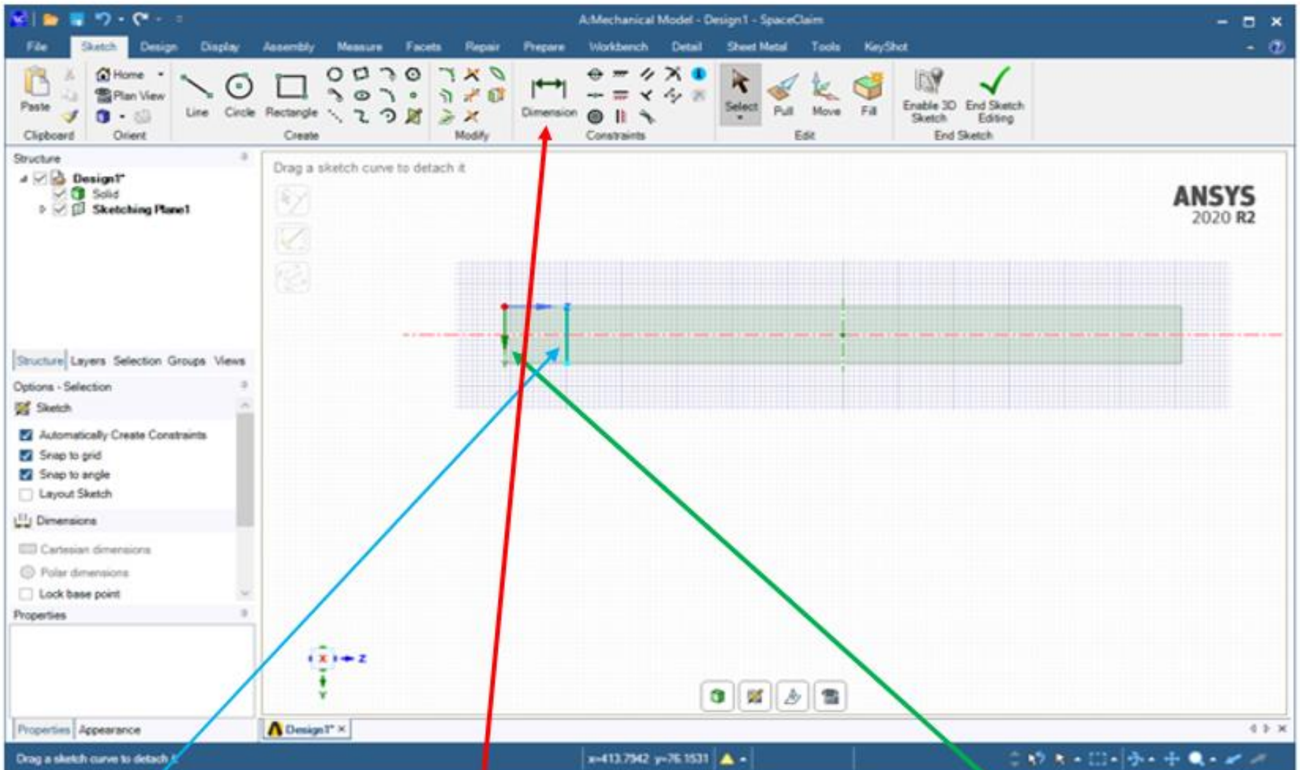
Click on "Line"



Create a line as shown below. After the creation of the line, hit "Esc" on keyboard to get out of line creation mode.

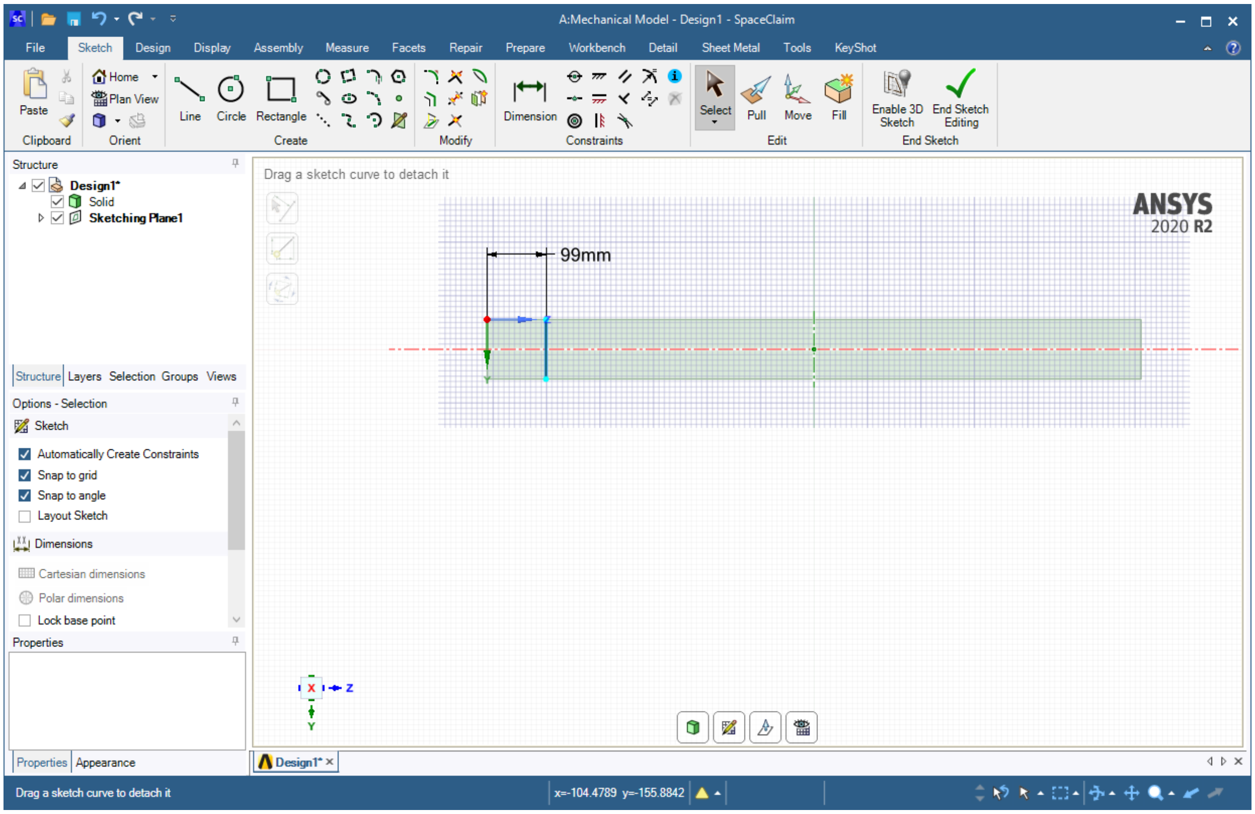


The above line is too long. Erase it and redraw it as shown below.

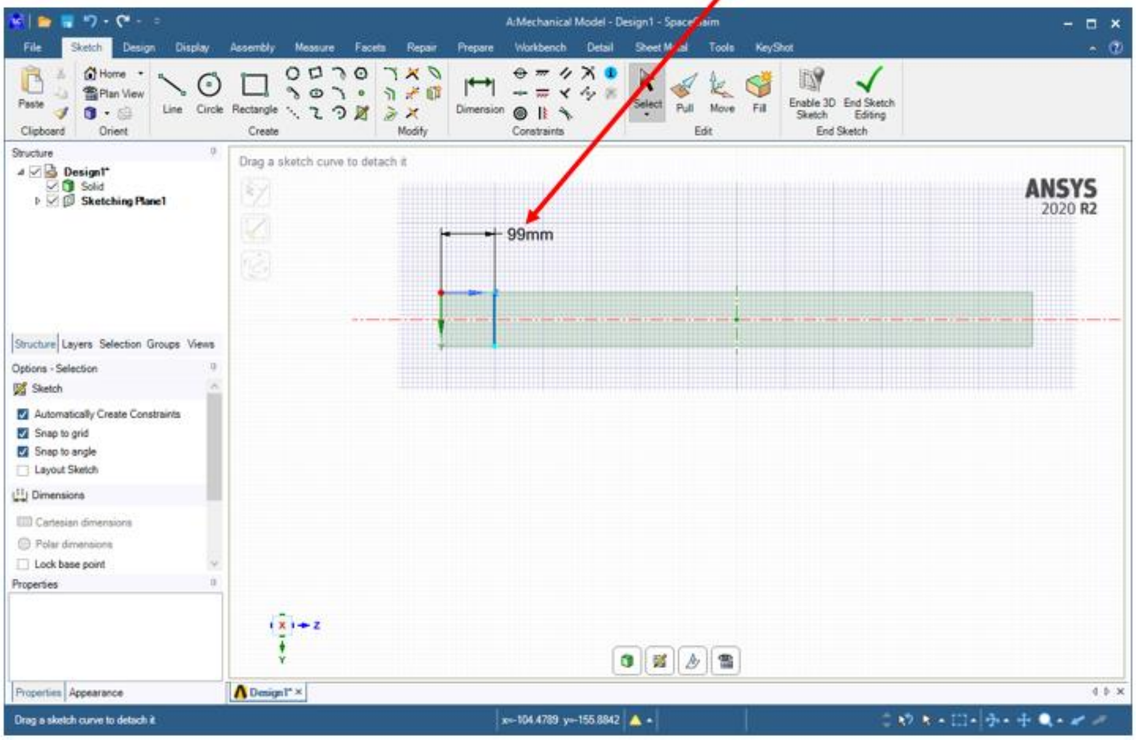


Create a "Dimension from the surface to the left to the newly drawn line. This is done by using the "Dimension" option at the top of the screen. It is tricky to dimension. Start from the left edge and drag to line after having invoked the "Dimension" command.

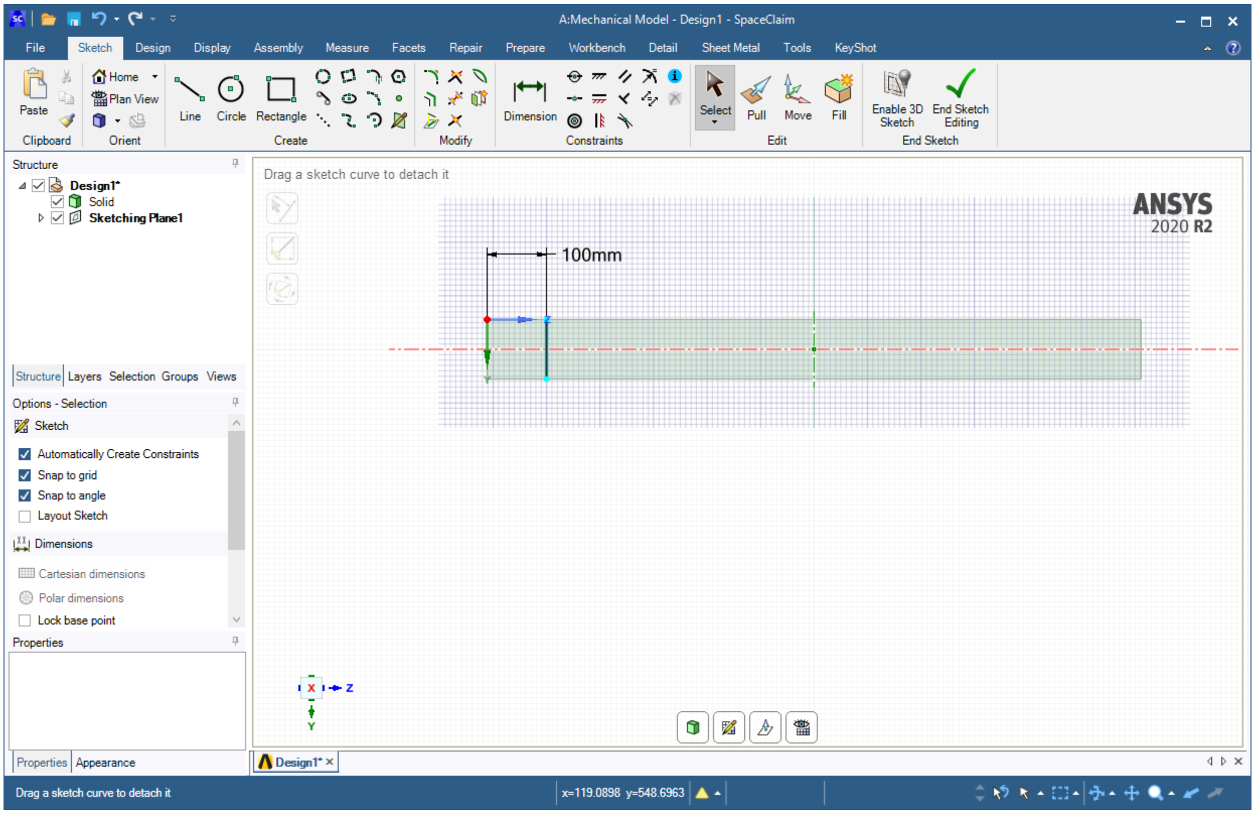
After dimensioning, the sketch appears as shown below.



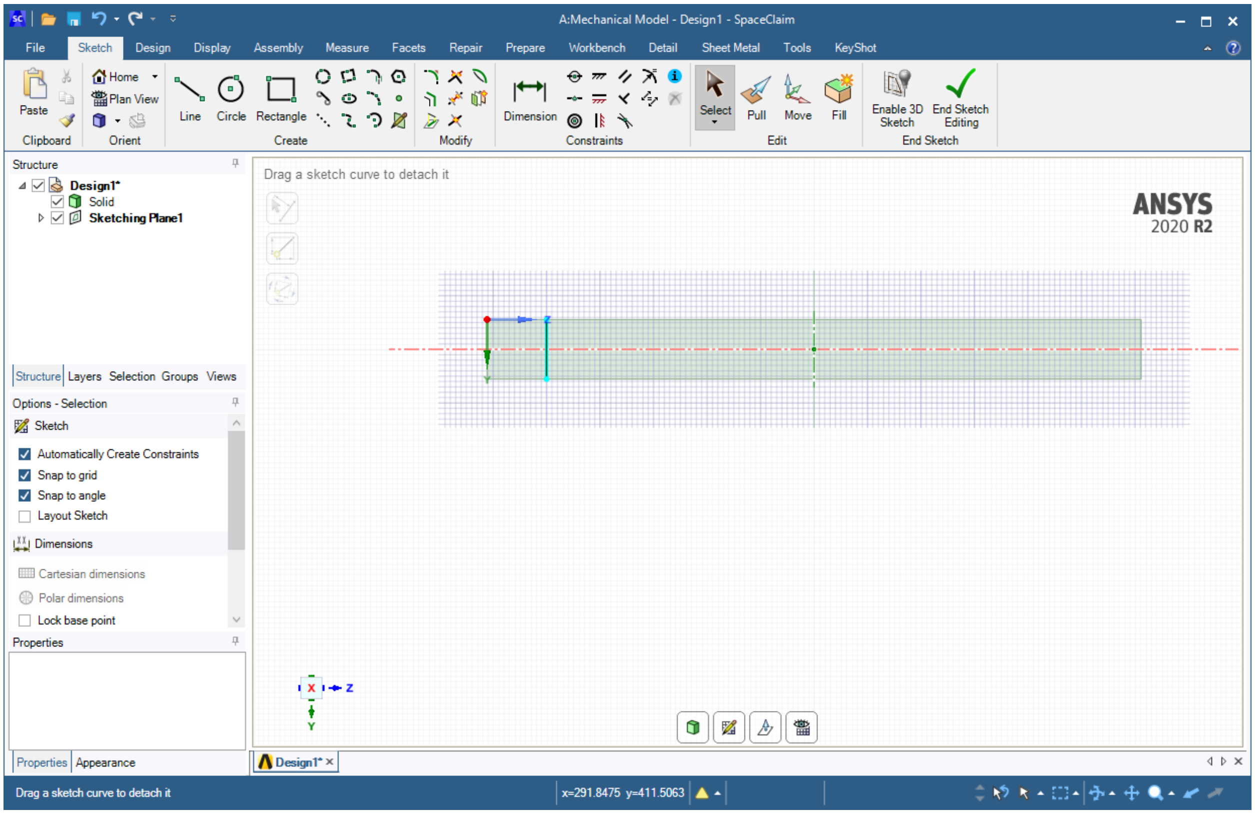
Double click and change to "100"



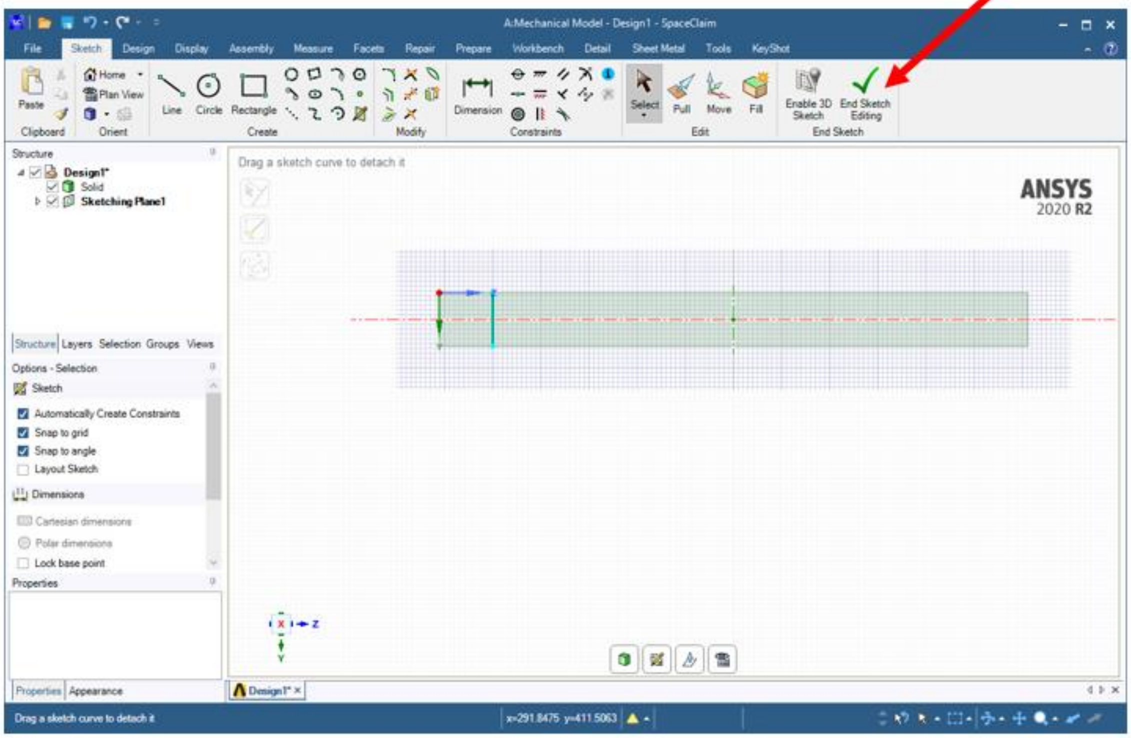
After changing the “Dimension,” the plot appears as shown below.



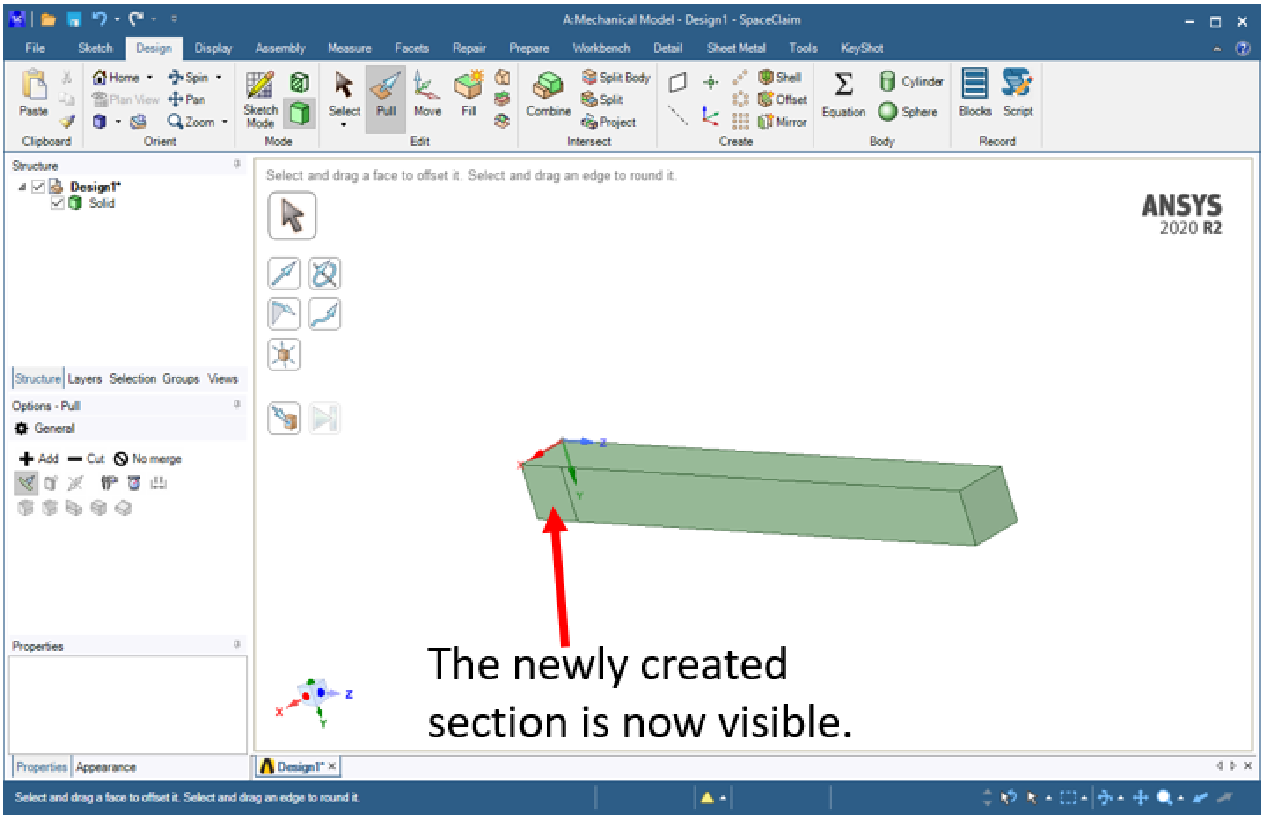
After erasing the extra dimension lines and dimension text, the plot appears as shown below.



Click on "End Sketch Editing"

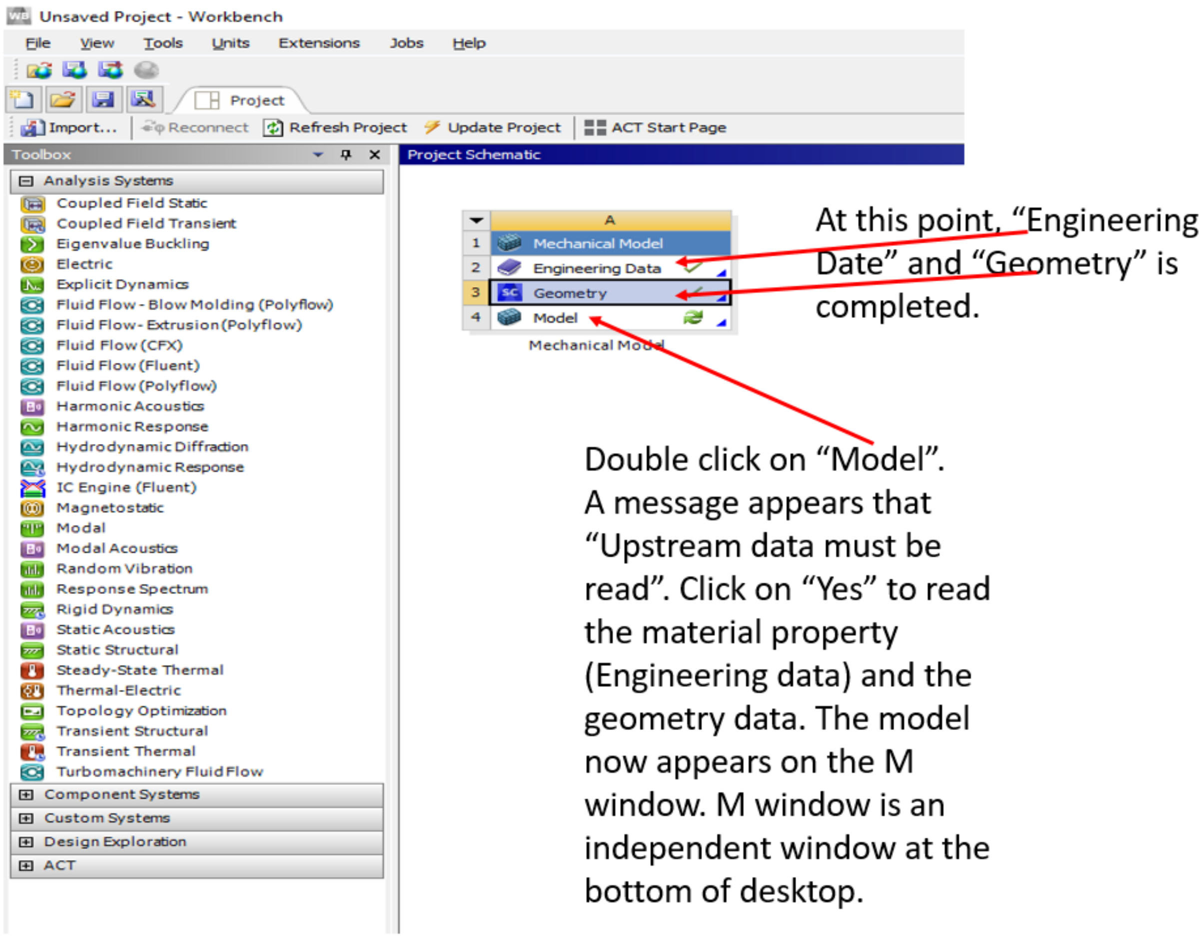


After exiting the “Sketch mode” and using the “Spin,” “pan,” and “zoom” commands and using the mouse wheel for rotation, the beam appears as shown below.

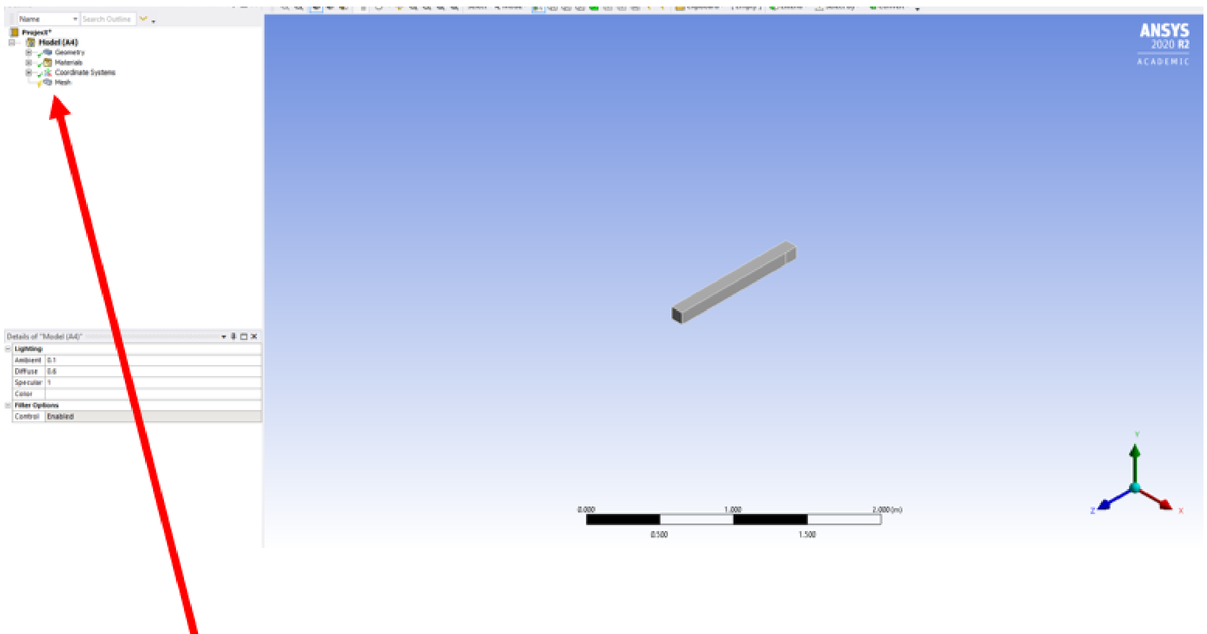


At this point, the geometry creation is completed. Minimize (but not close) the “S.C.” window.

Bring up the project window. Follow the instructions shown below.

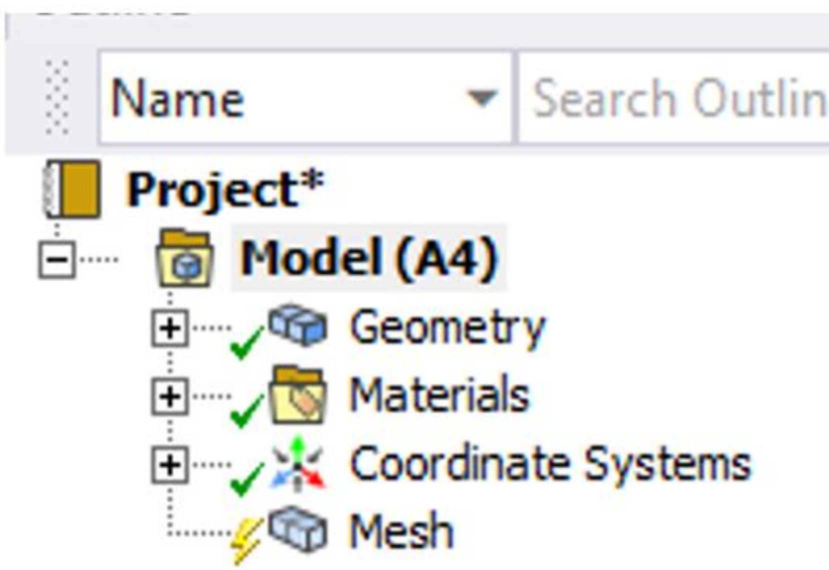


M window appears as shown below.

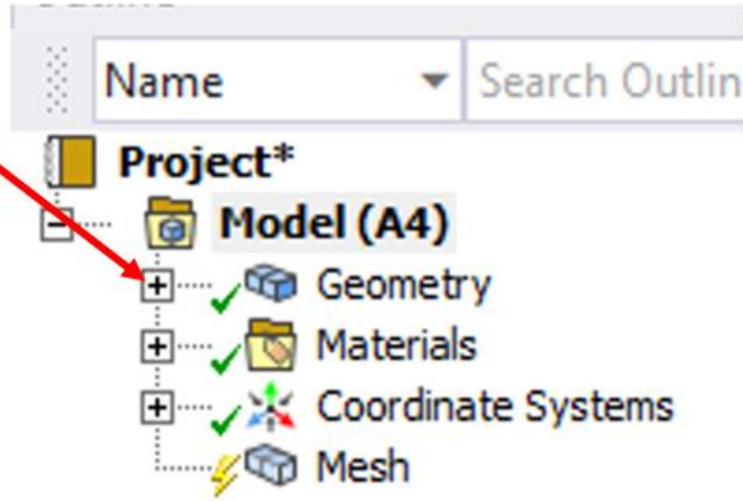


This portion is critical, but it is too small to see.

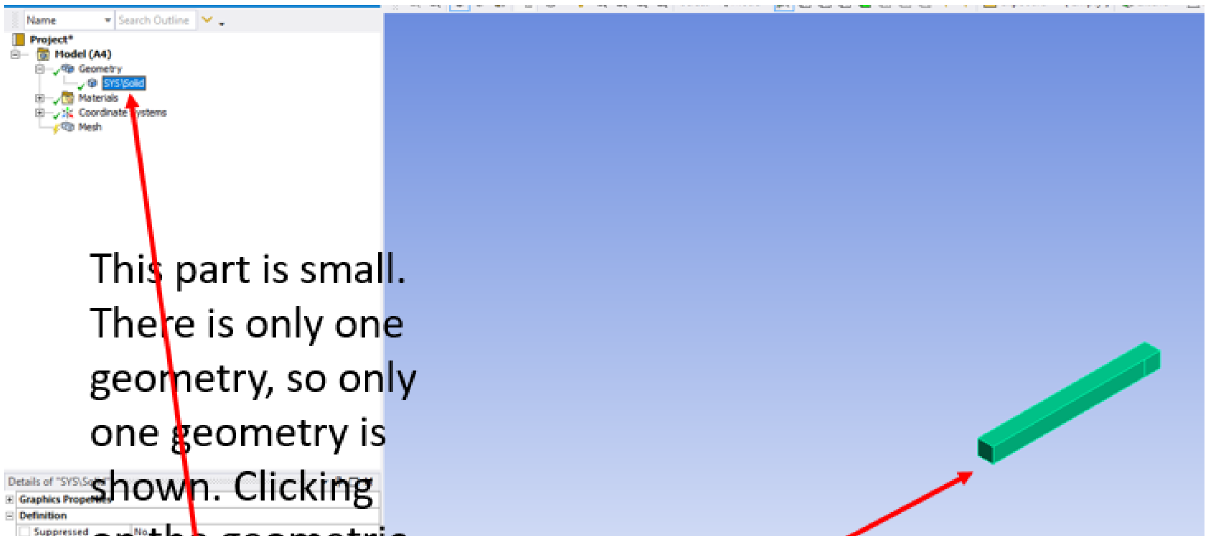
The following shows the small part in detail.



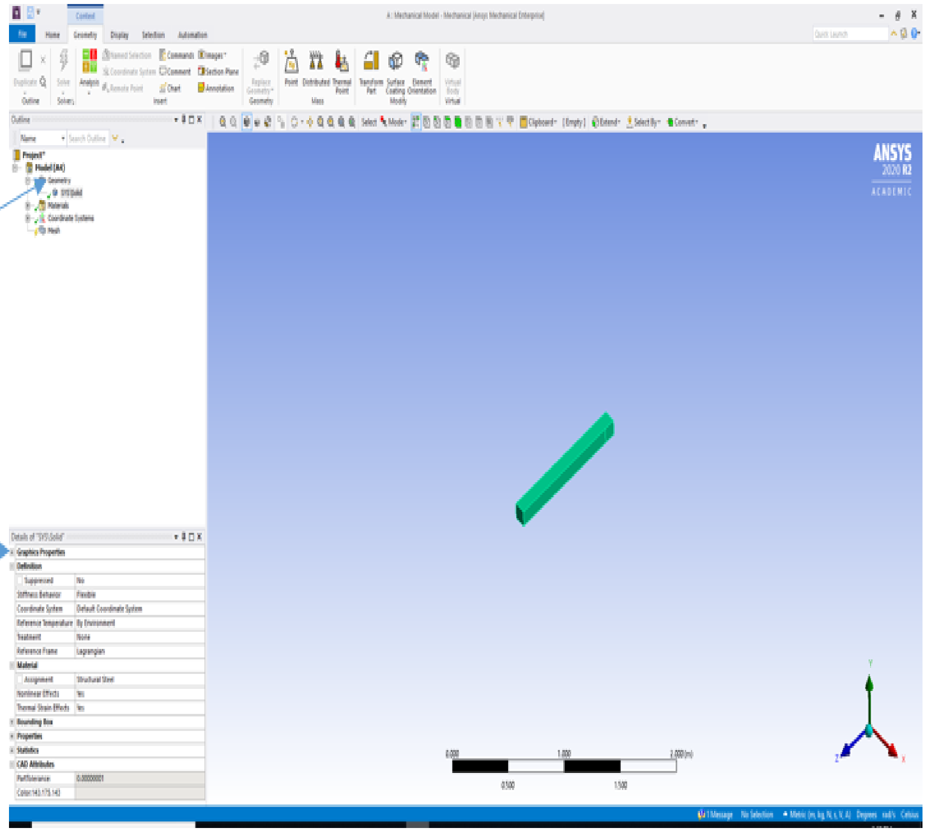
Click on The "+" sign next to geometry.



The screen will change as shown below after the "+" sign next to geometry is clicked on.

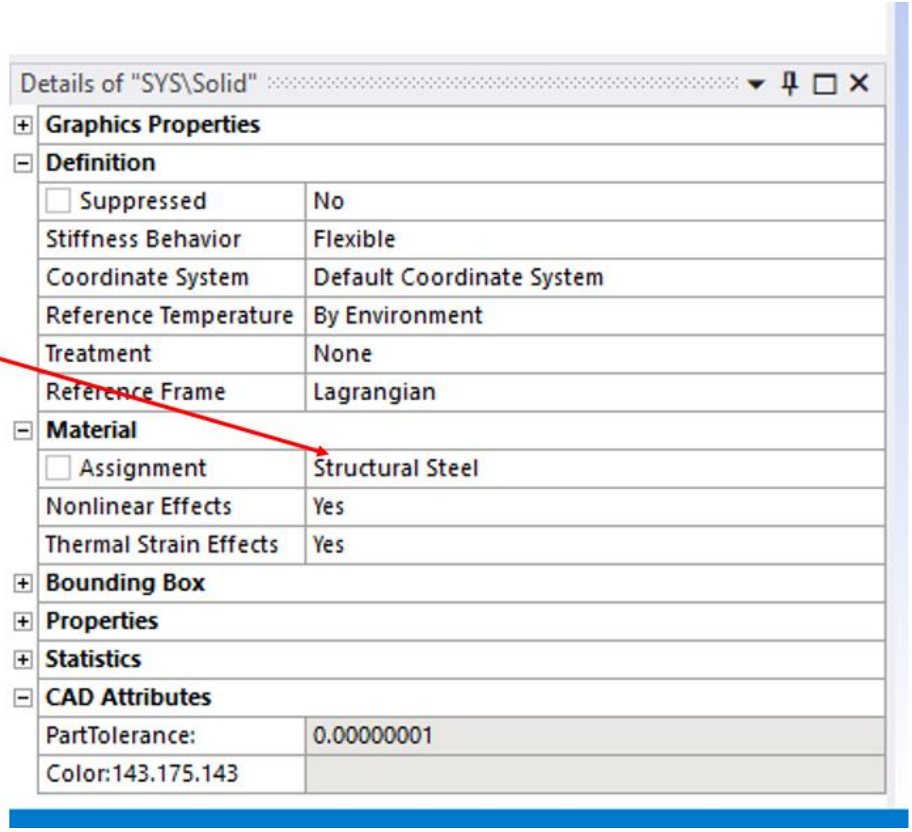


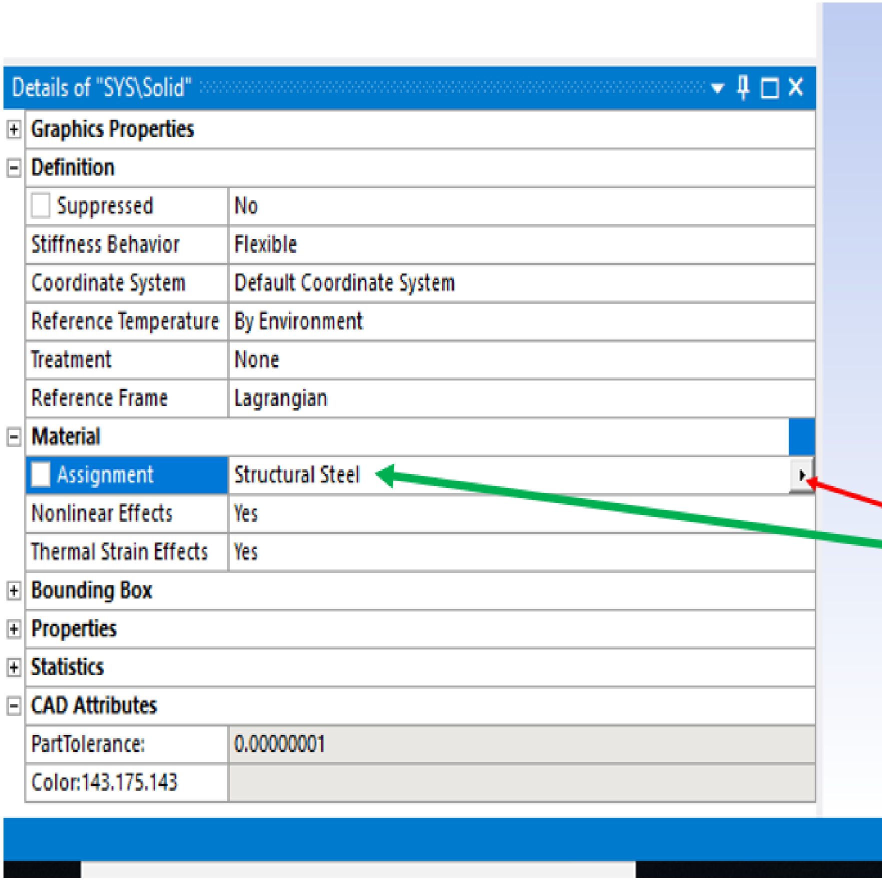
This part is small. There is only one geometry, so only one geometry is shown. Clicking on the geometric element highlights it on the tree on the left and highlights it on the M screen.



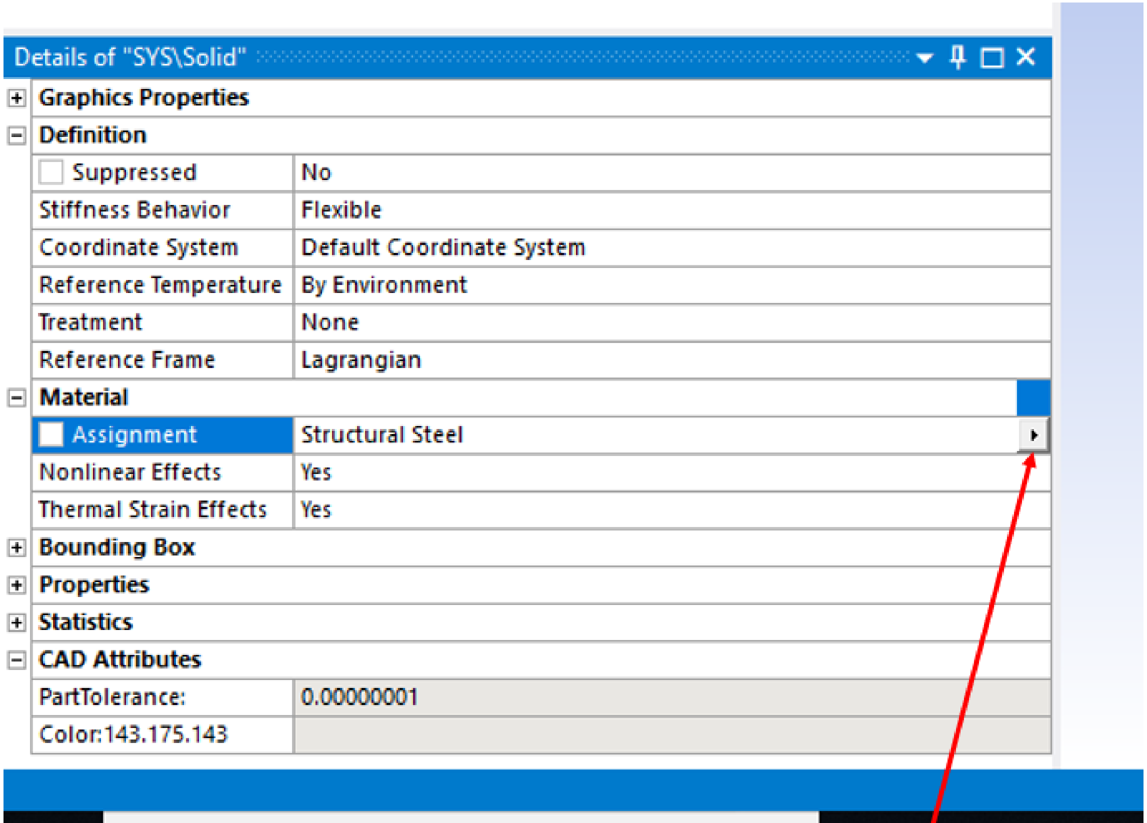
When "Geometry" was clicked, The "Material" window appeared on the bottom left of the screen.

The default material assigned is structural steel.



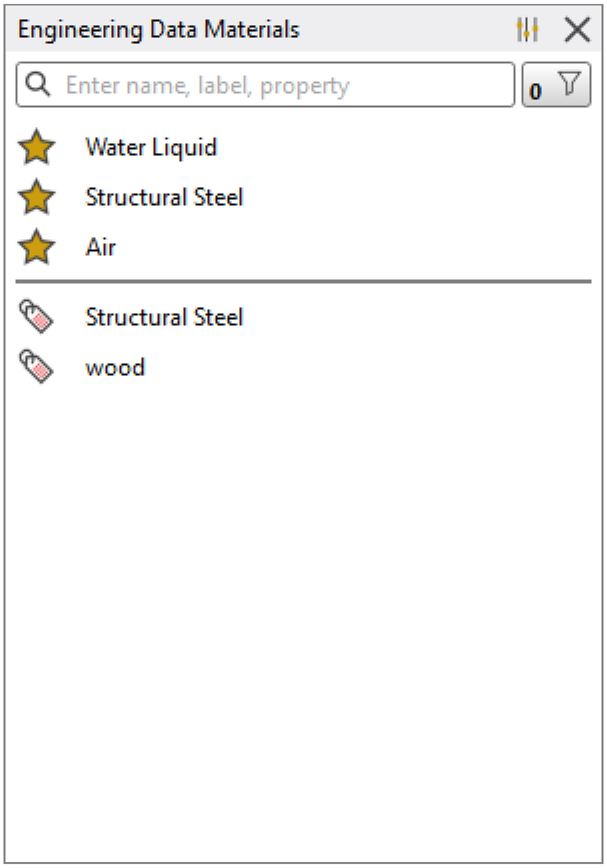


Click on "Structural Steel", and this arrow appears.



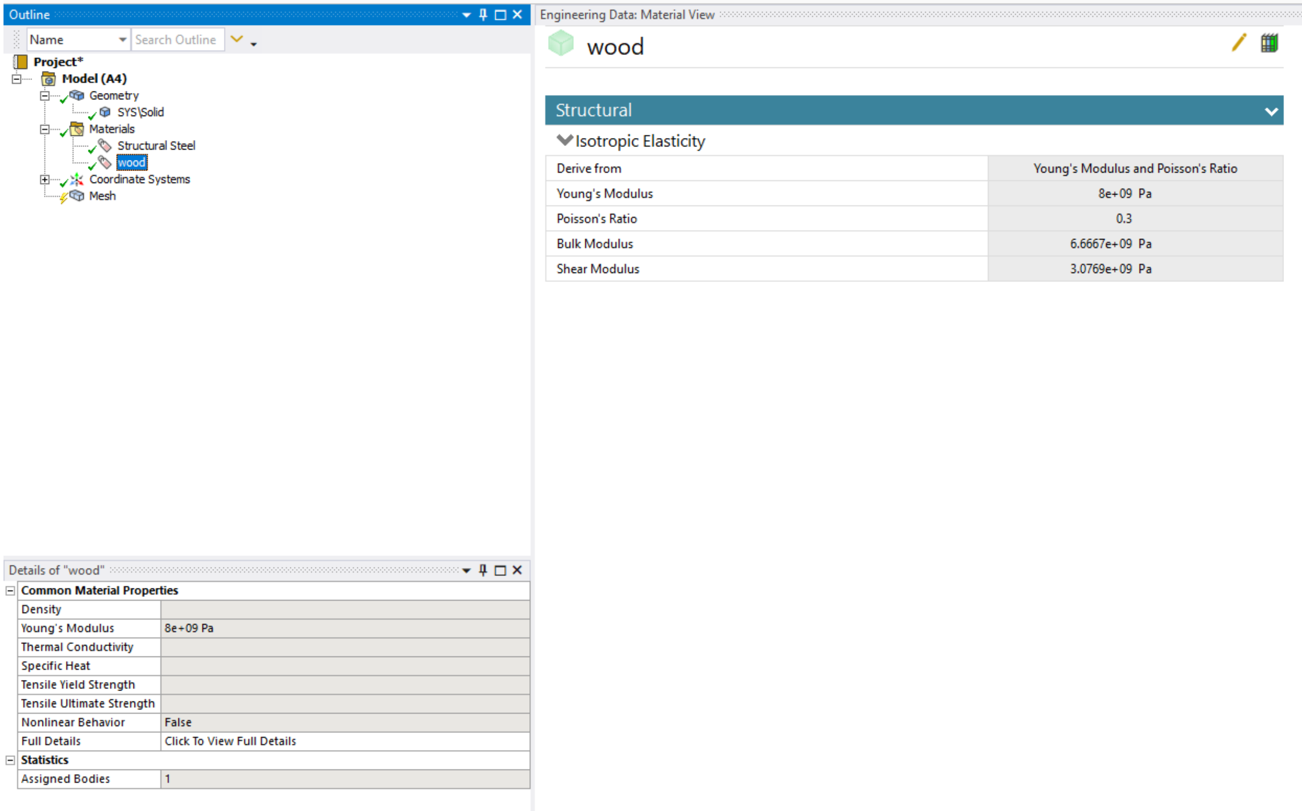
Click on this arrow and the material property box appears.

The following is the screen appearance after clicking on the arrow.



Click on "Wood" above.

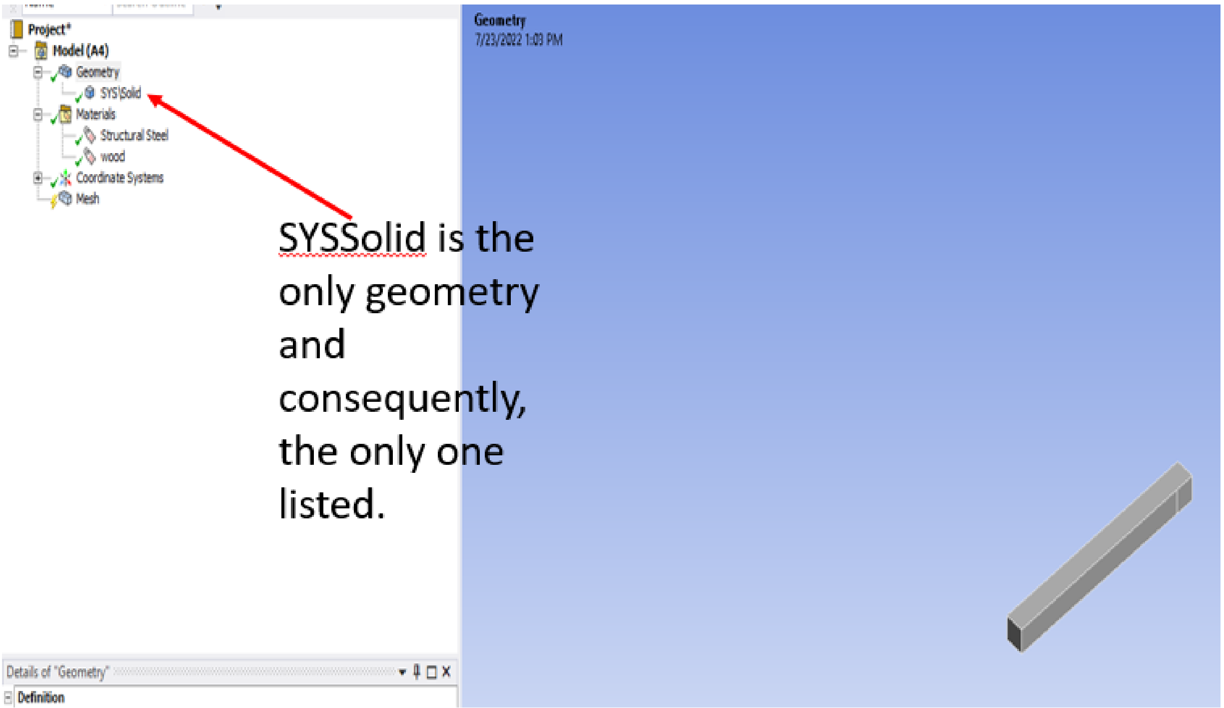
Wood is available under "Material" now.



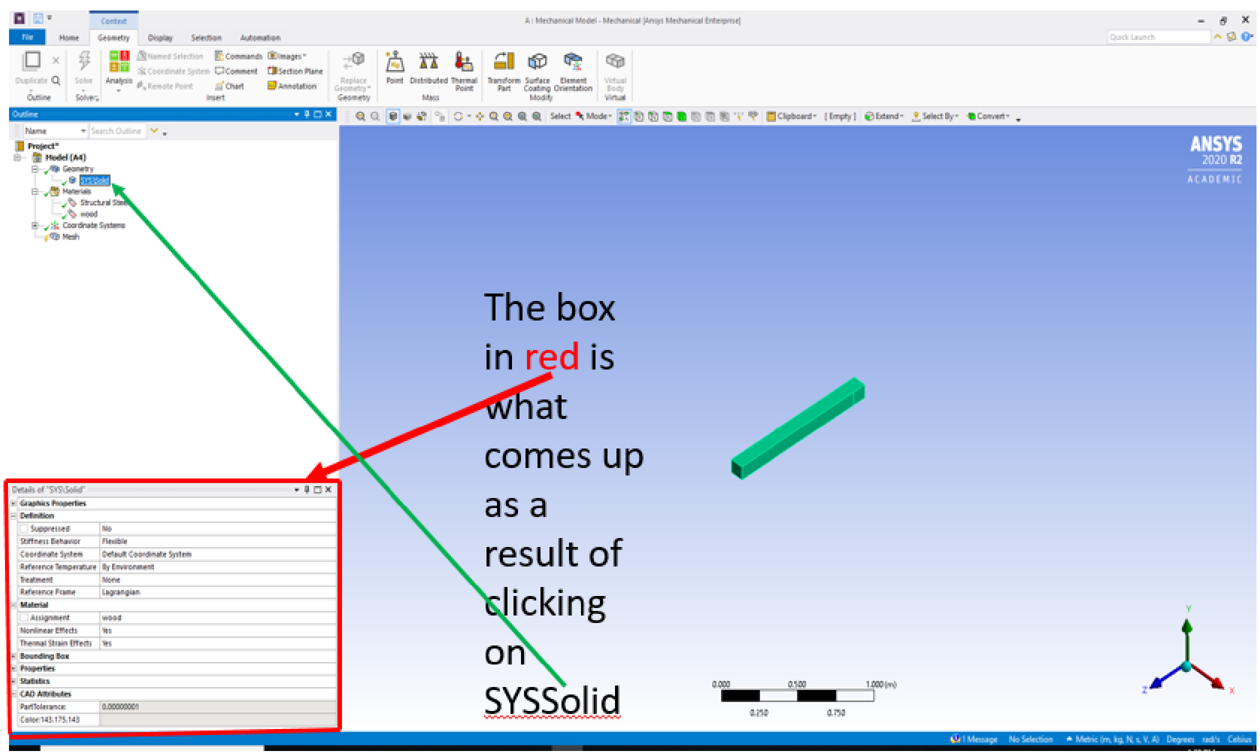
I had much difficulty assigning the material. Then I noticed that it is listed under Geometry.

In the "M" window, click on "Geometry." The following appears.

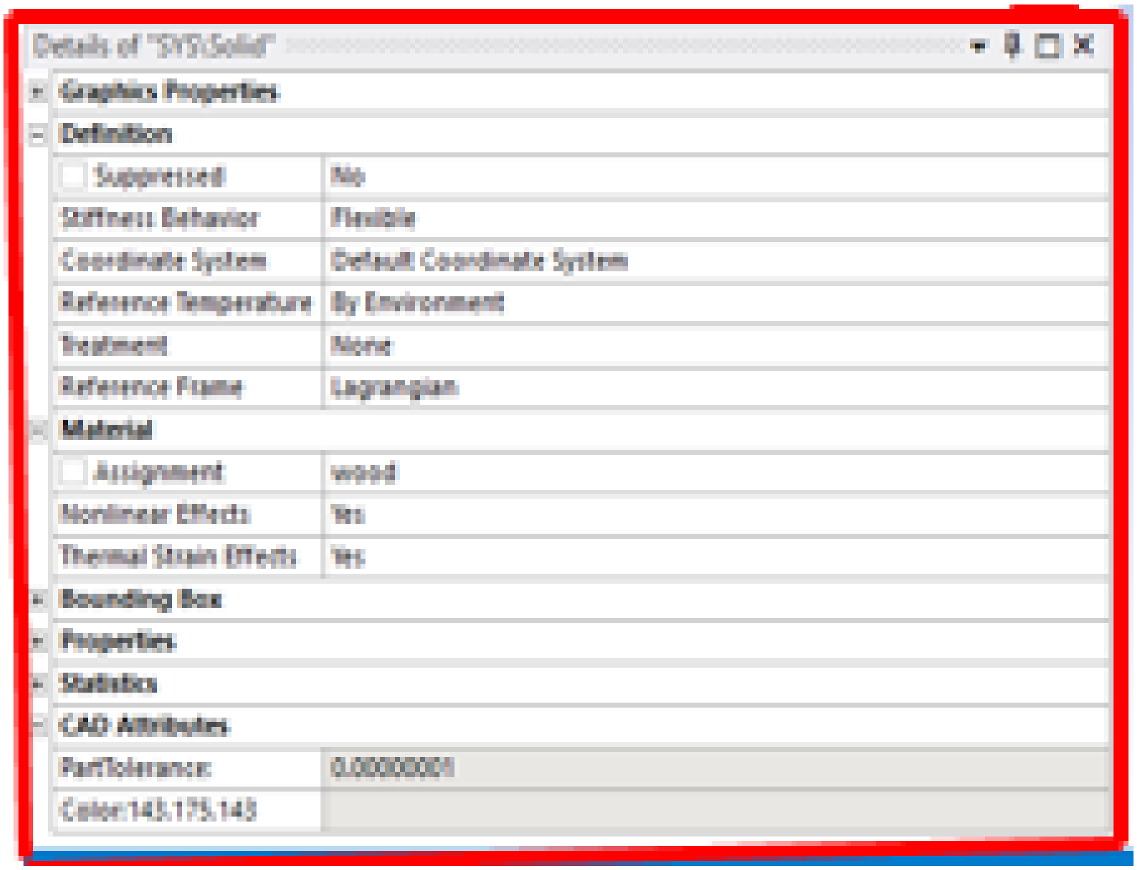
In the case of this example, there is only one Geometry, called “SYSSolid.”

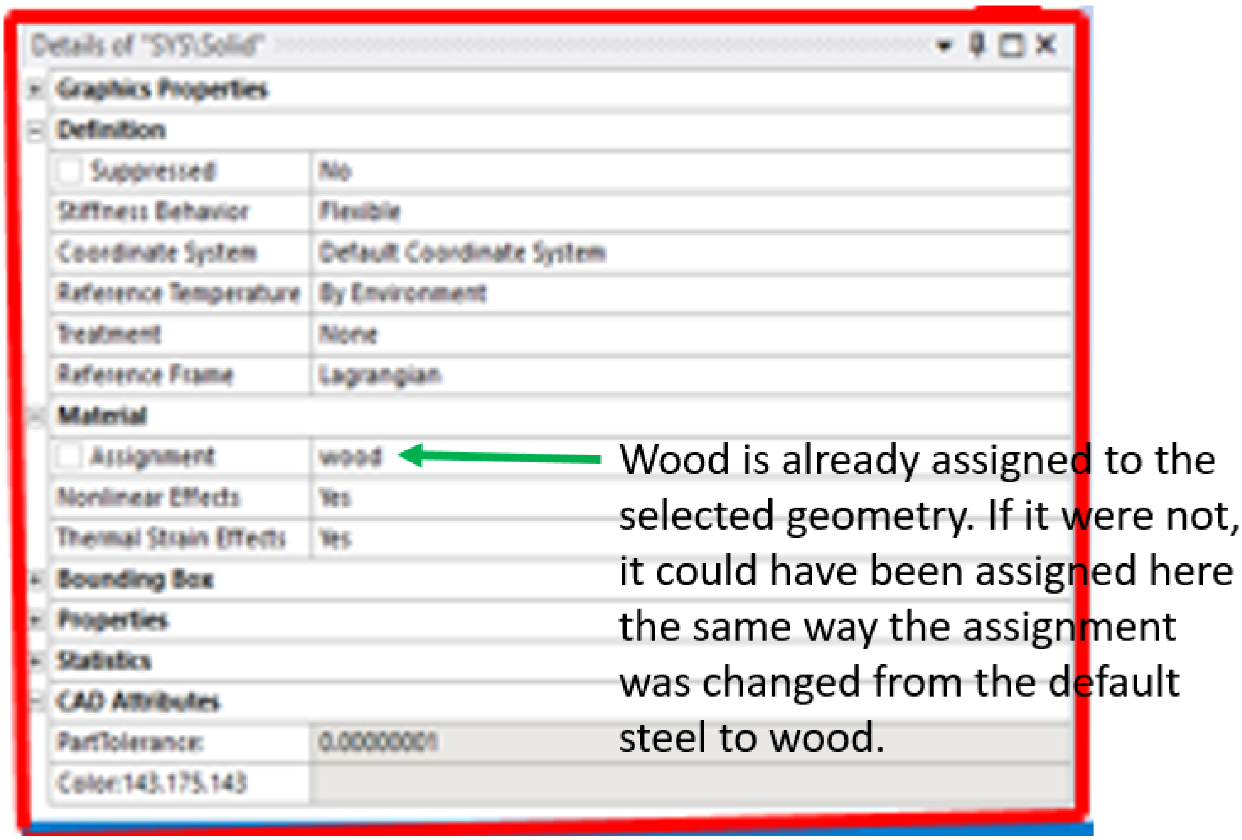


After clicking on “SYSSolid” above, a box appears in the lower left part of the screen.



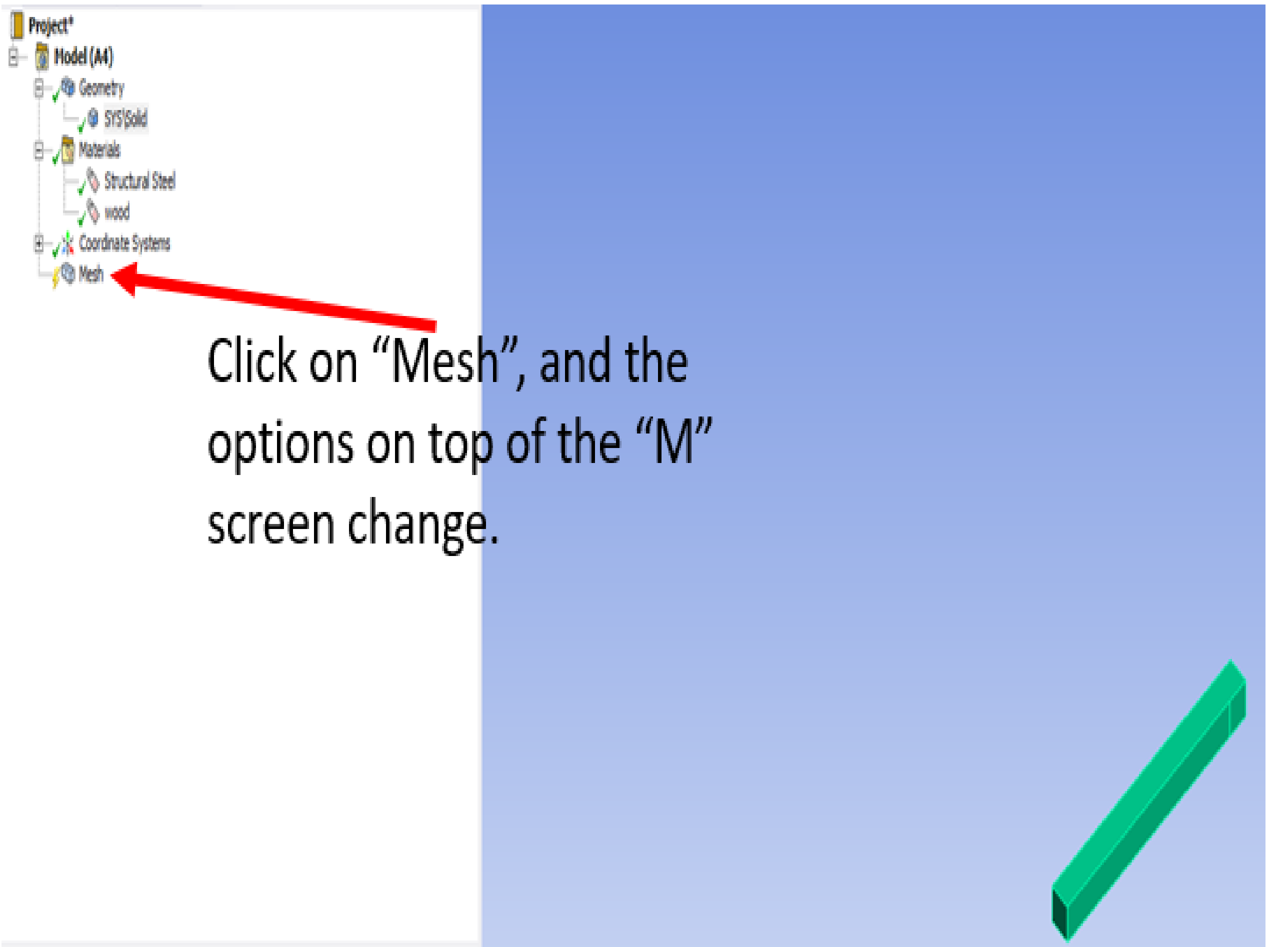
The following is the enlarged red box from the previous screen.



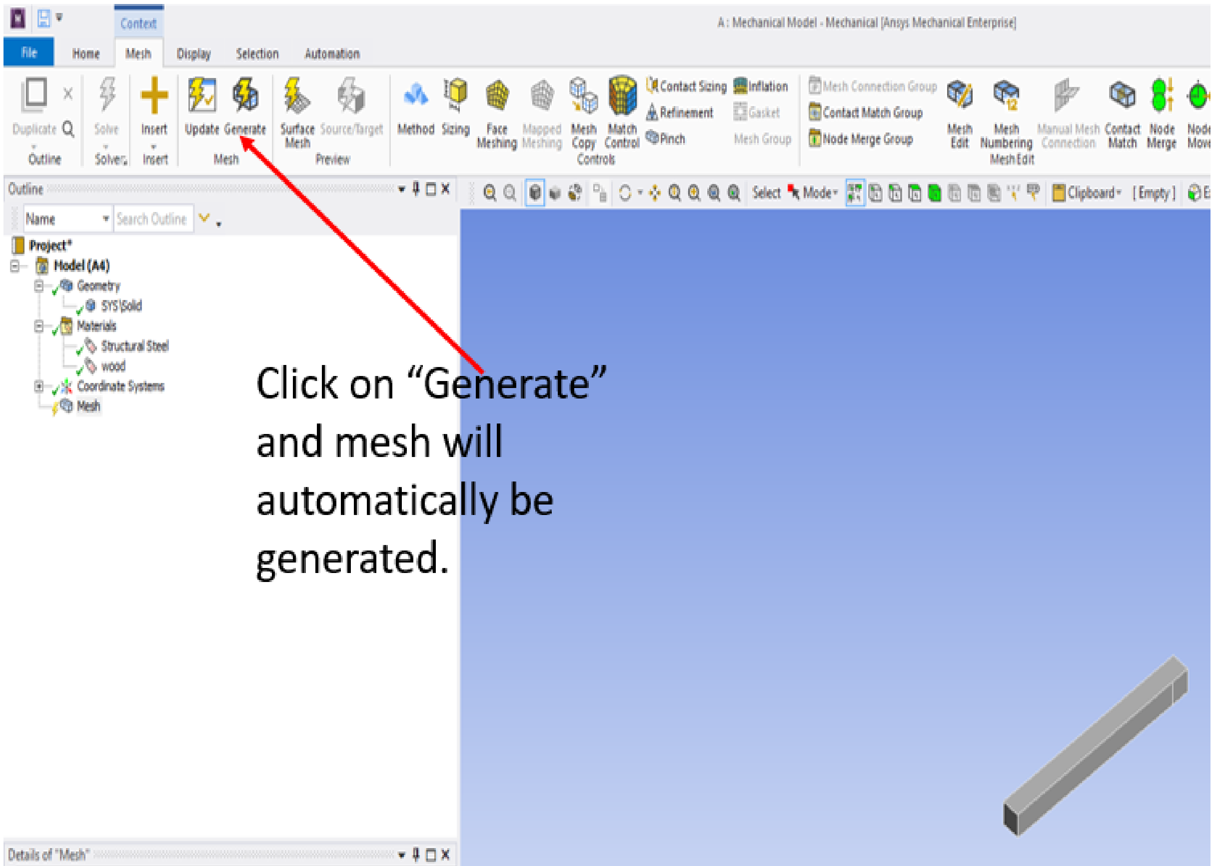


Note that different materials can be assigned to different geometries by the technique described.

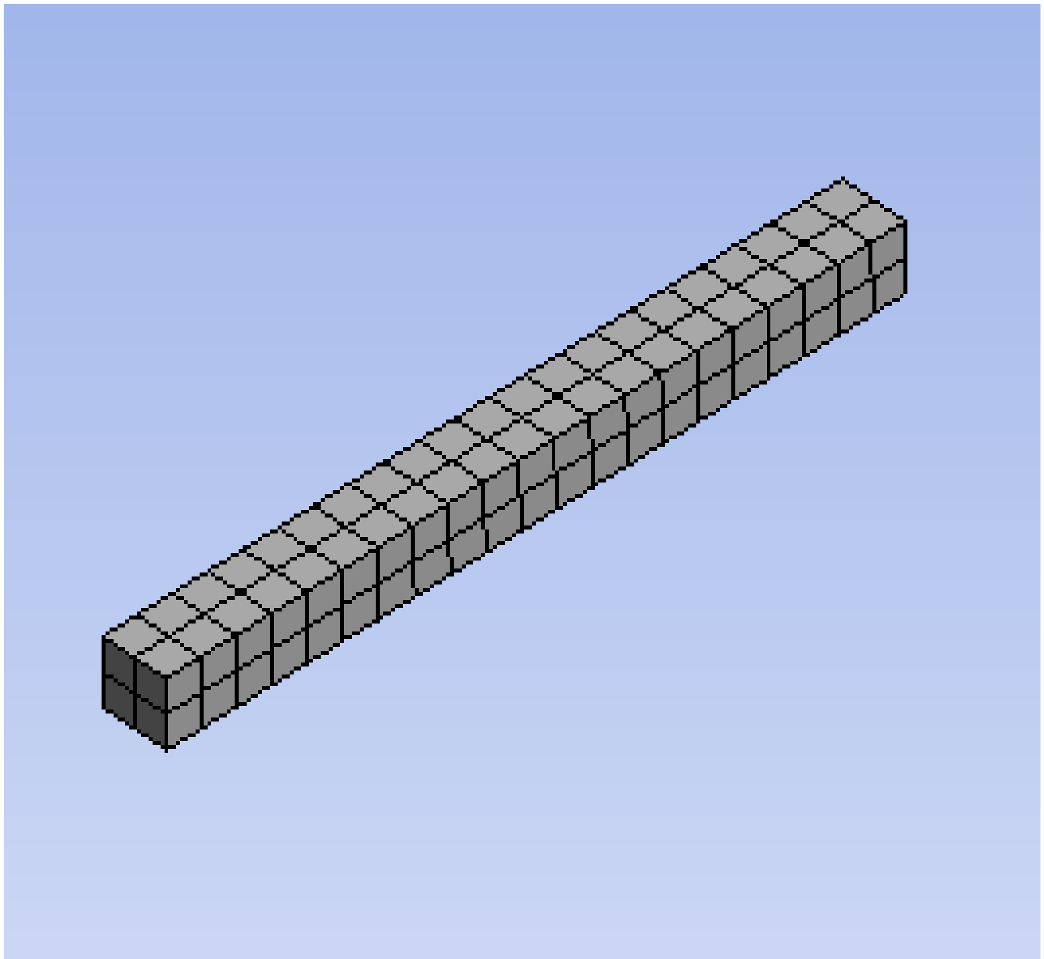
At this point, the geometry has been completed, and the material has been defined and assigned to the Geometry. The next step is to create the Mesh.

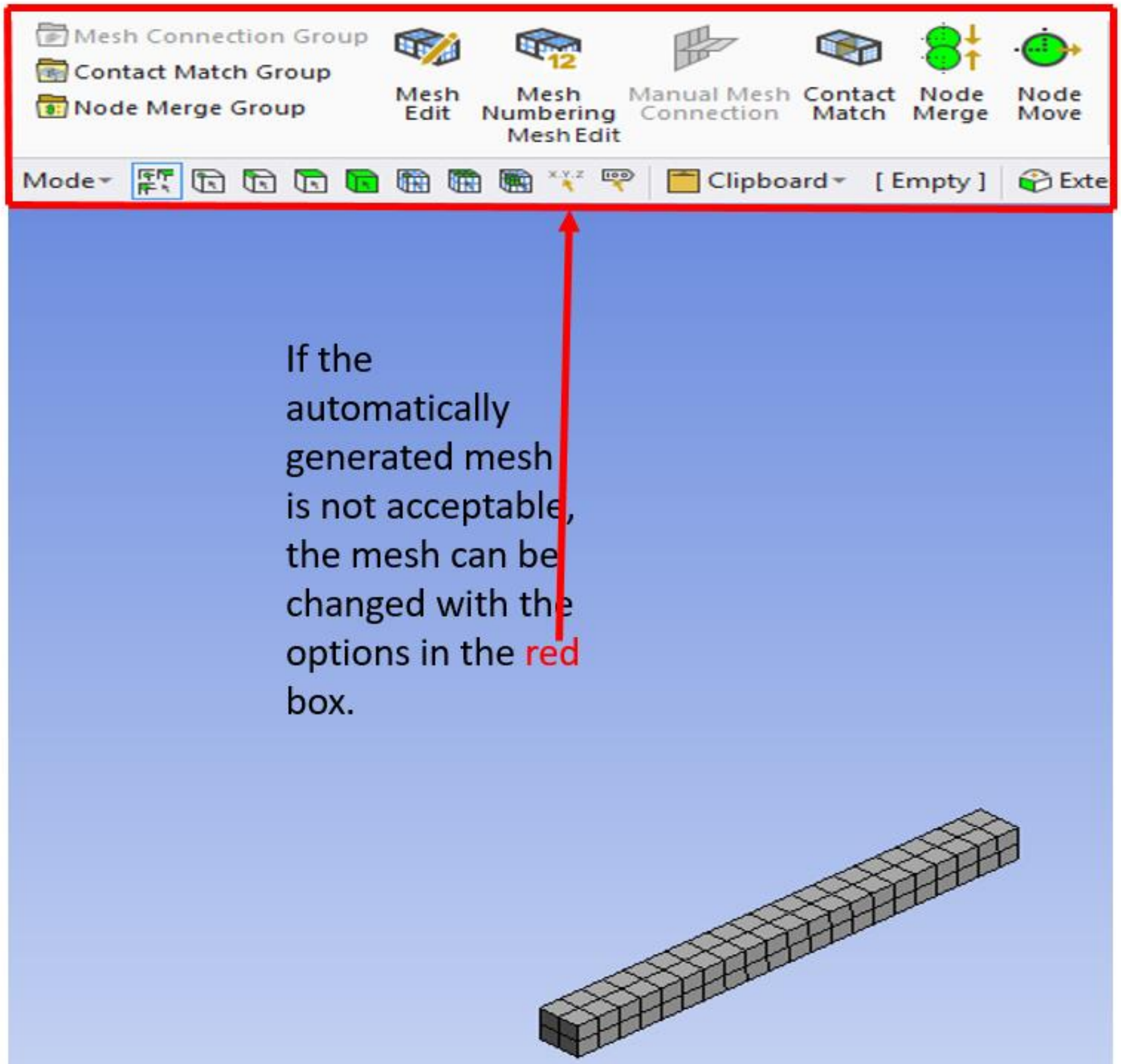


If the Geometry is simple as it is here, Clicking on "Generate" on top of the screen will automatically generate a mesh.

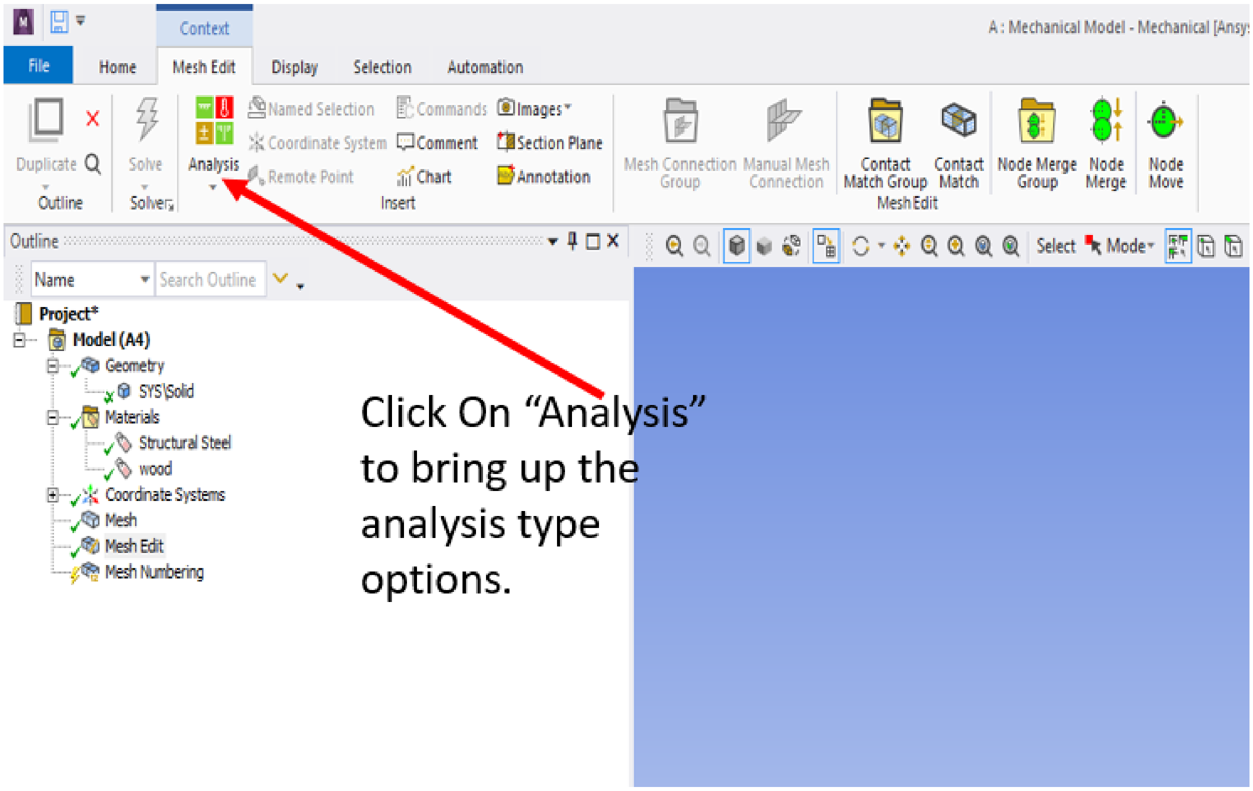


After clicking on "Generate," the Geometry is meshed as shown below.

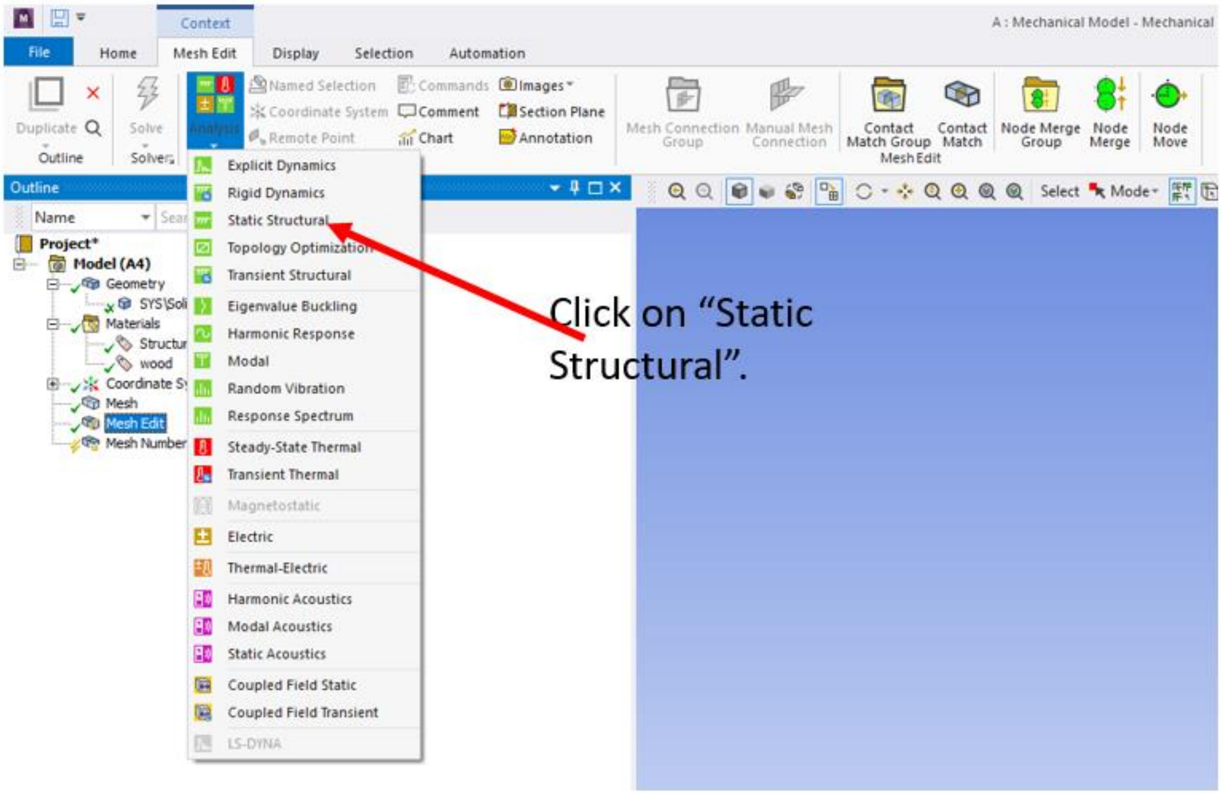


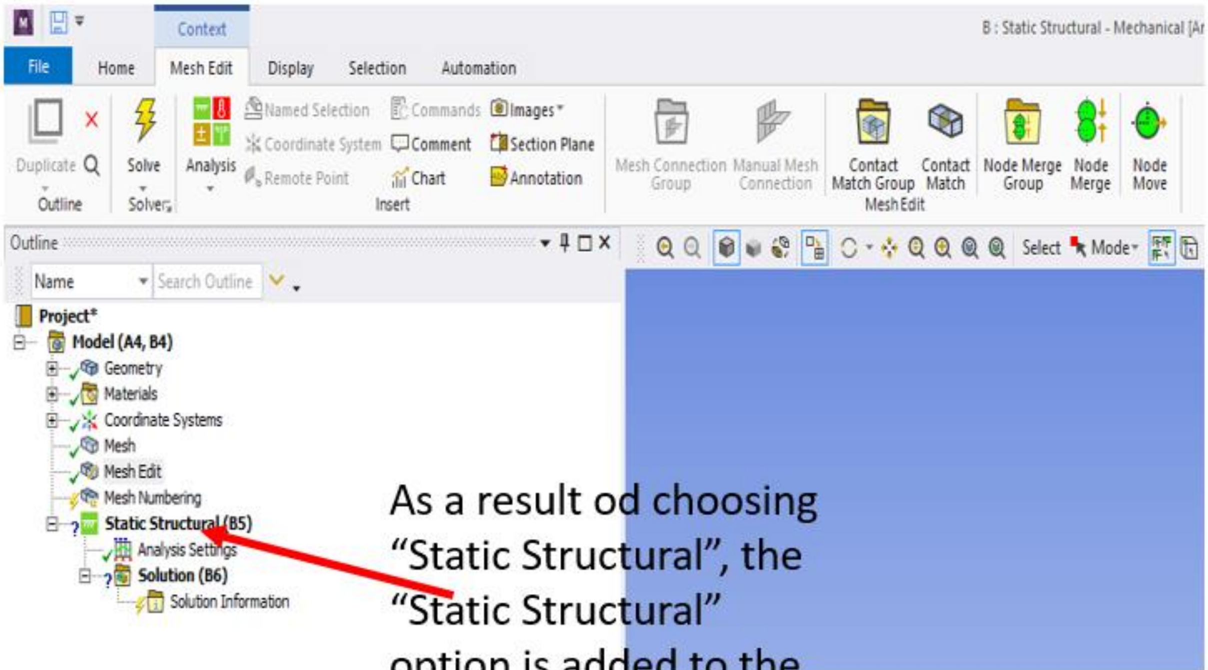

Mesh Connection Group
Contact Match Group
Node Merge Group
Mesh Edit
Mesh Numbering
Manual Mesh Connection
Contact Match
Node Merge
Node Move
Mesh Edit
Mode
Clipboard
[Empty]
Exte
If the automatically generated mesh is not acceptable, the mesh can be changed with the options in the red box.

For some reason, the Static Structural option has disappeared on both WB (Work Bench) and M (Mechanical) screens.

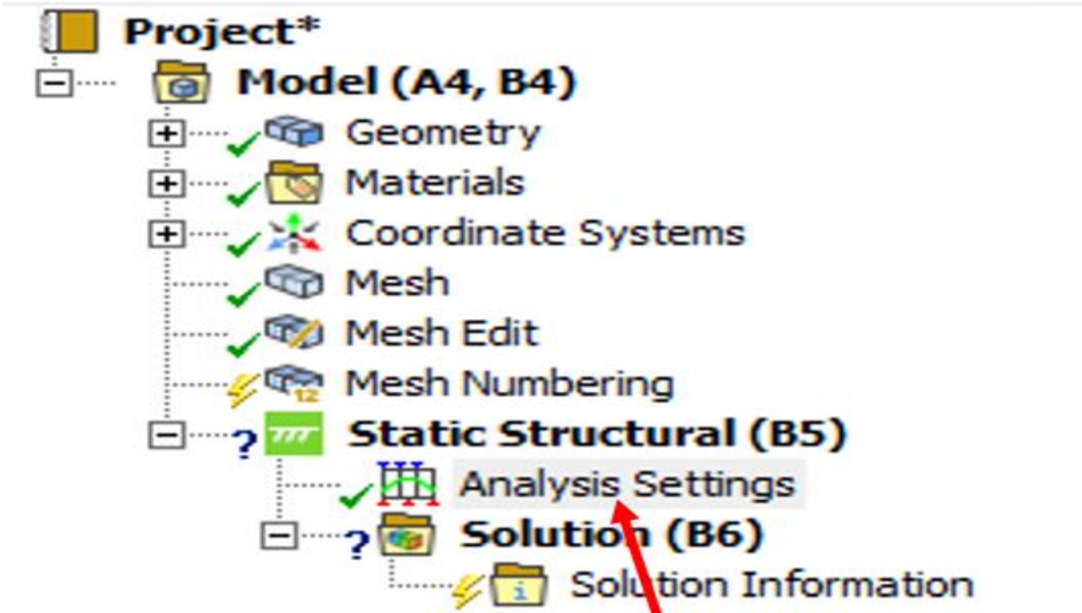


After clicking on “Analysis,” the screen appears as shown below.

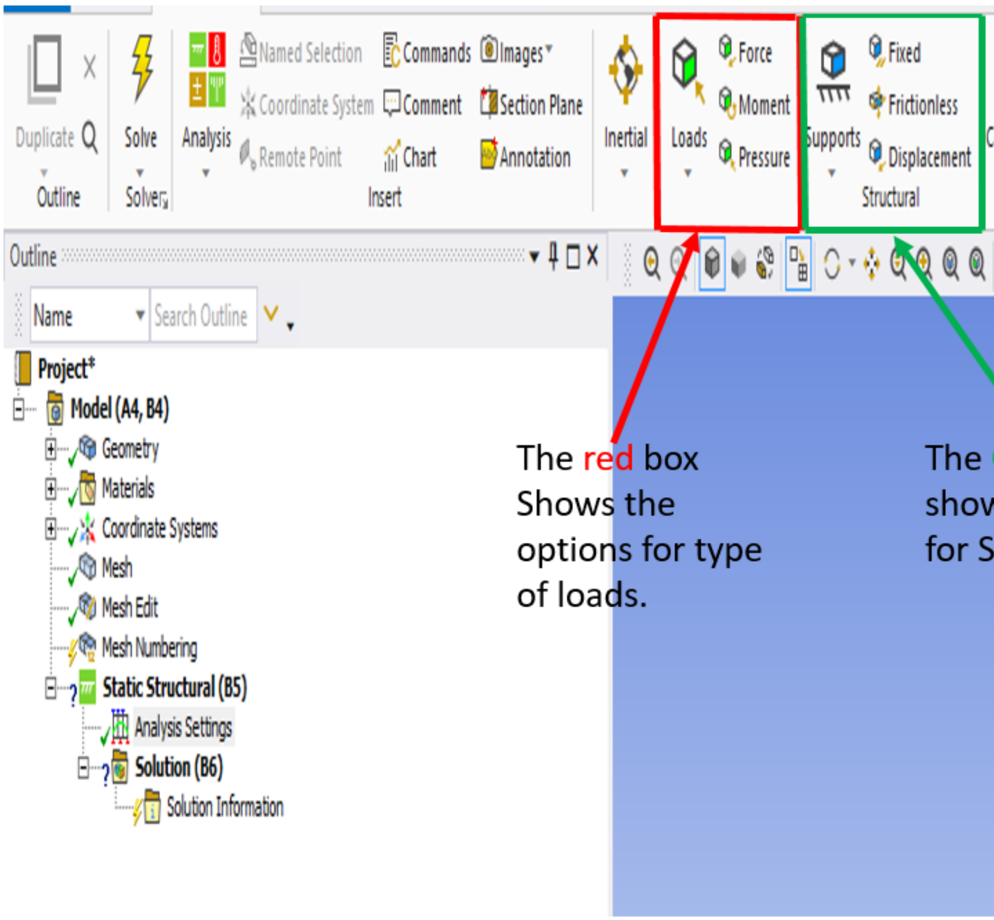




As a result od choosing "Static Structural", the "Static Structural" option is added to the tree on the upper left of "M" screen.



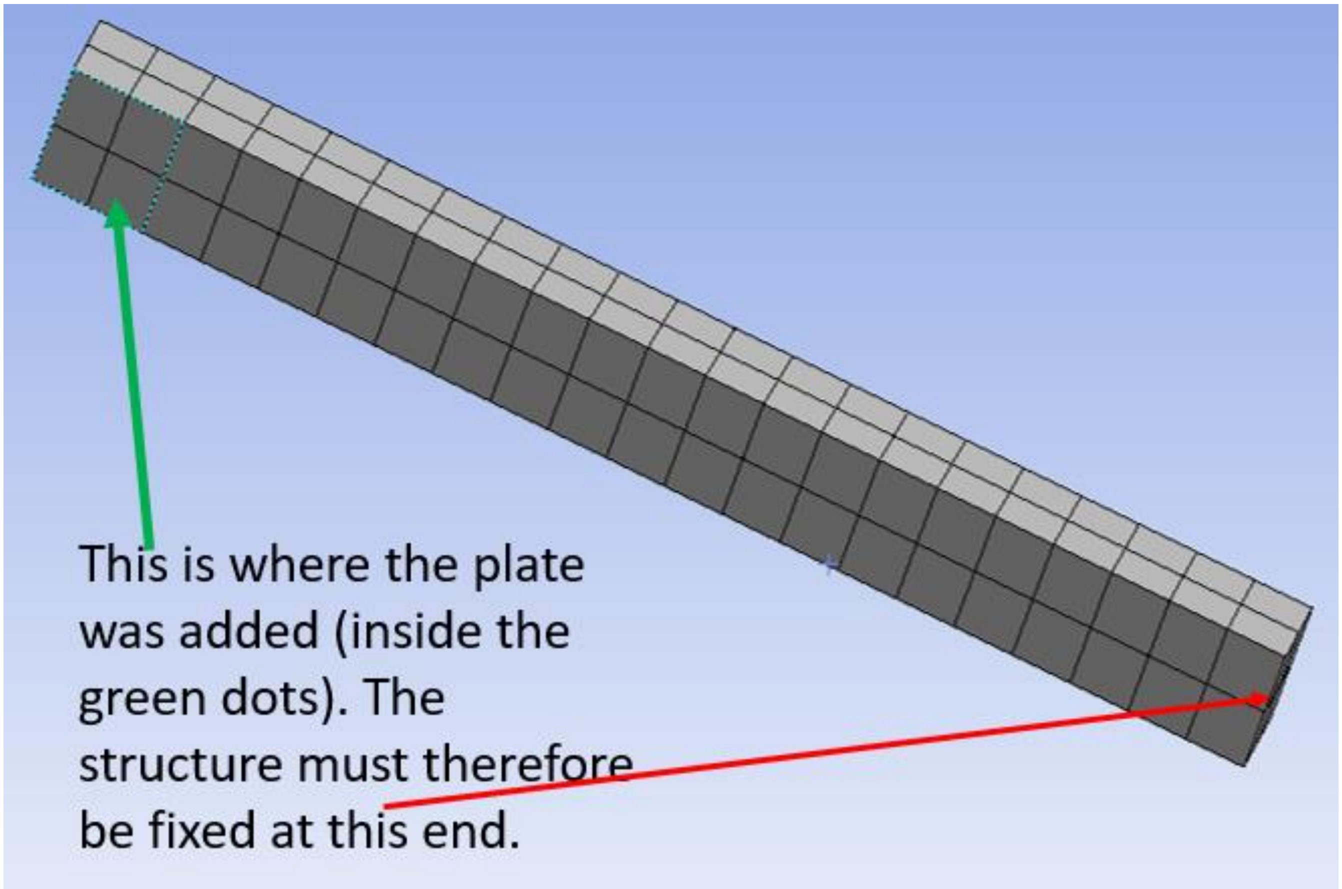
Click on 'Analysis Settings" and the menu on top of "M" screen changes. Among the options that appear on top of "M" screen are "Loads" and "Supports".



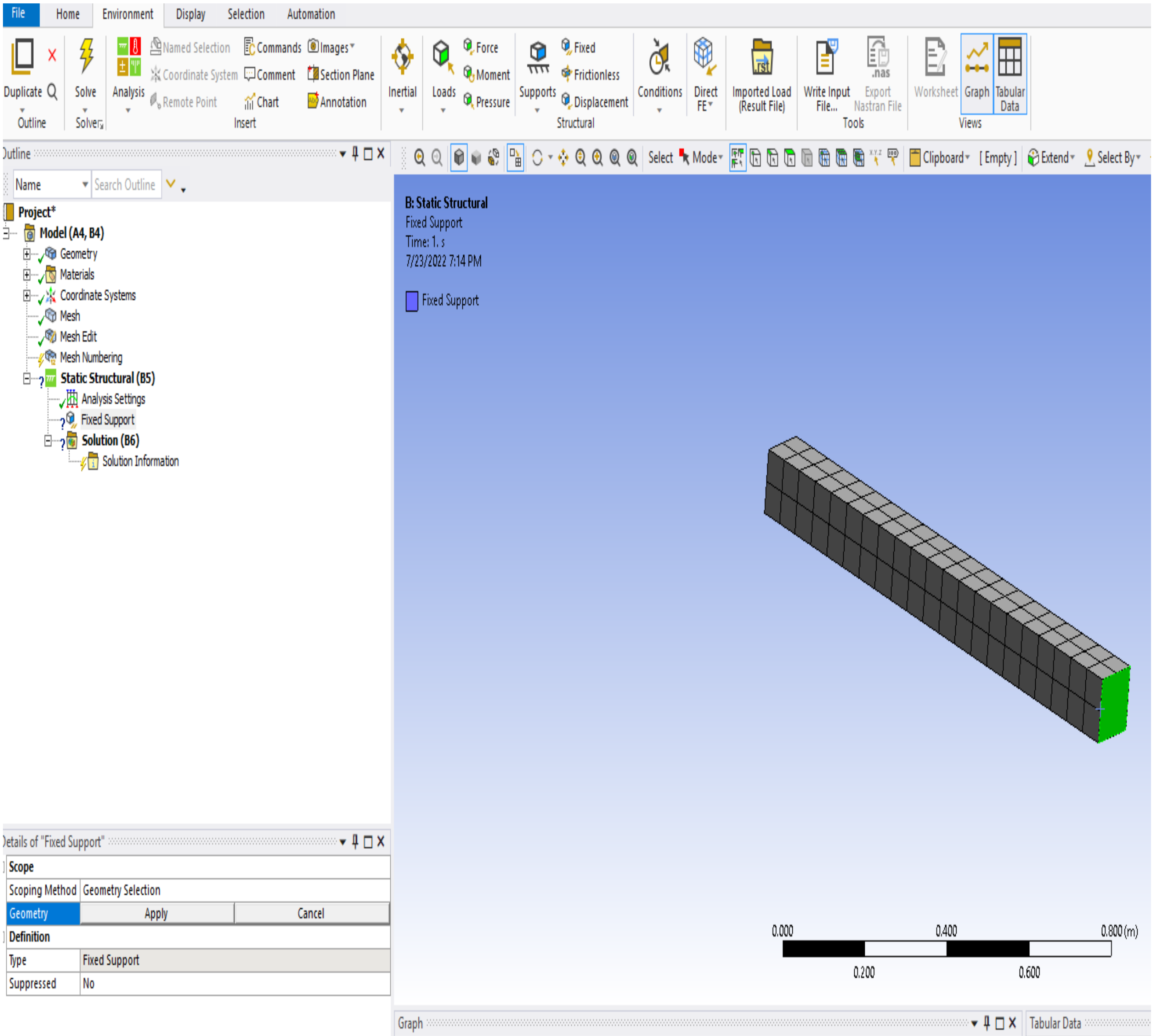
The red box Shows the options for type of loads.

The Green box shows the options for Support types

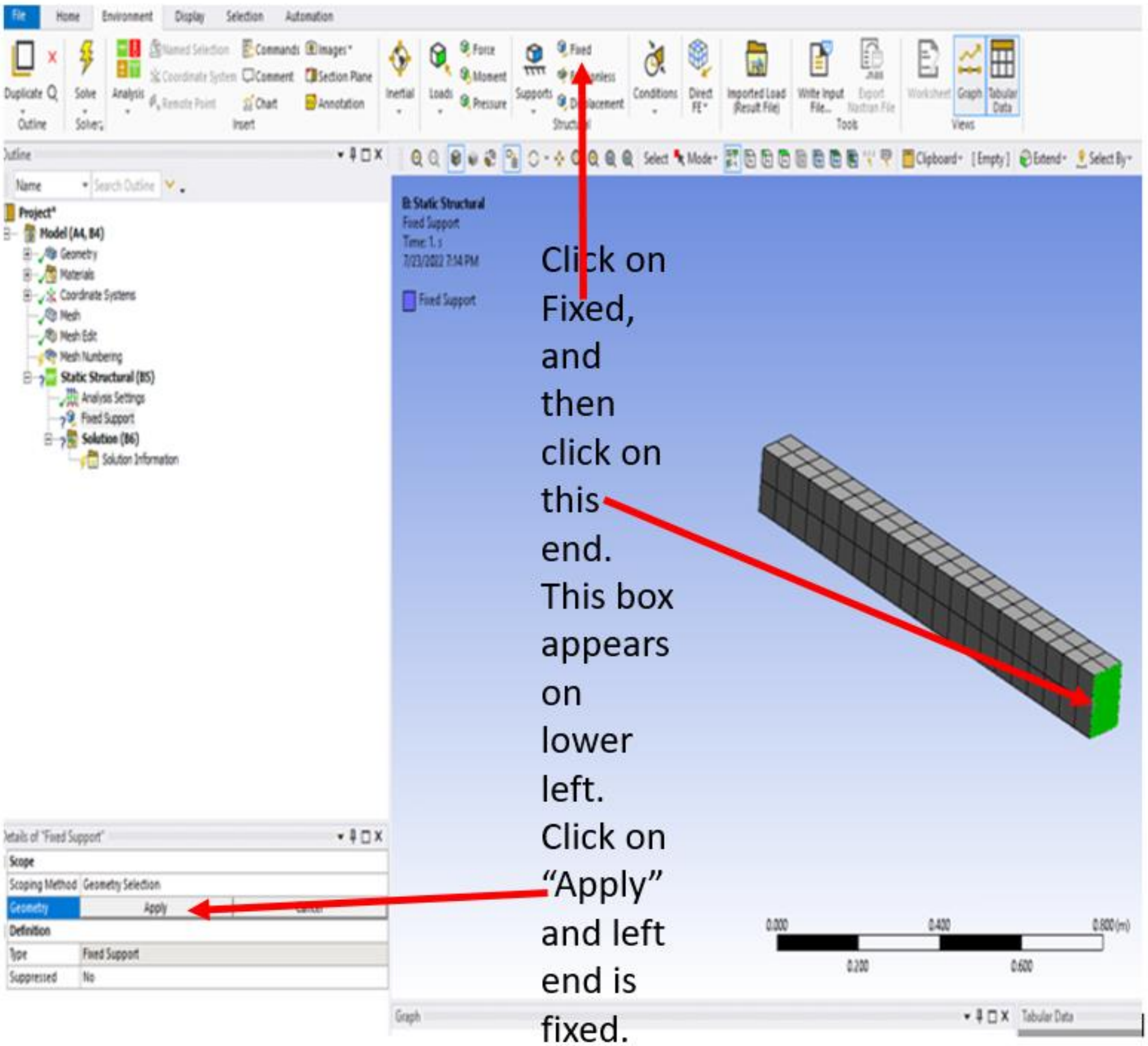
Play with Mouse Wheel and enlarge the geometry. Hold down the Mouse Wheel and rotate the geometry as necessary. Move the Mouse Pointer on the Geometry until the location of the plate that was added for pressure application is identified.



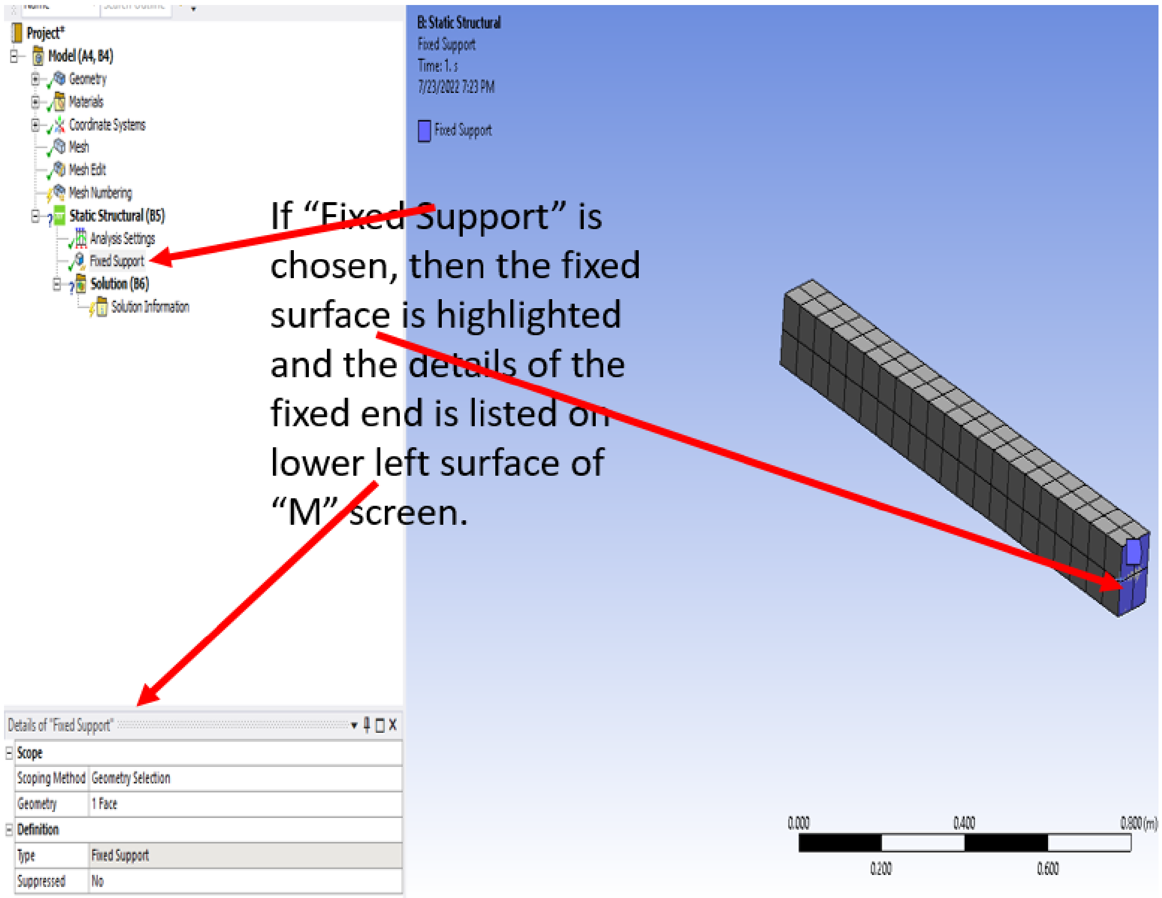
Square Geometry is being shown as rectangular because of stretching in Powerpoint.



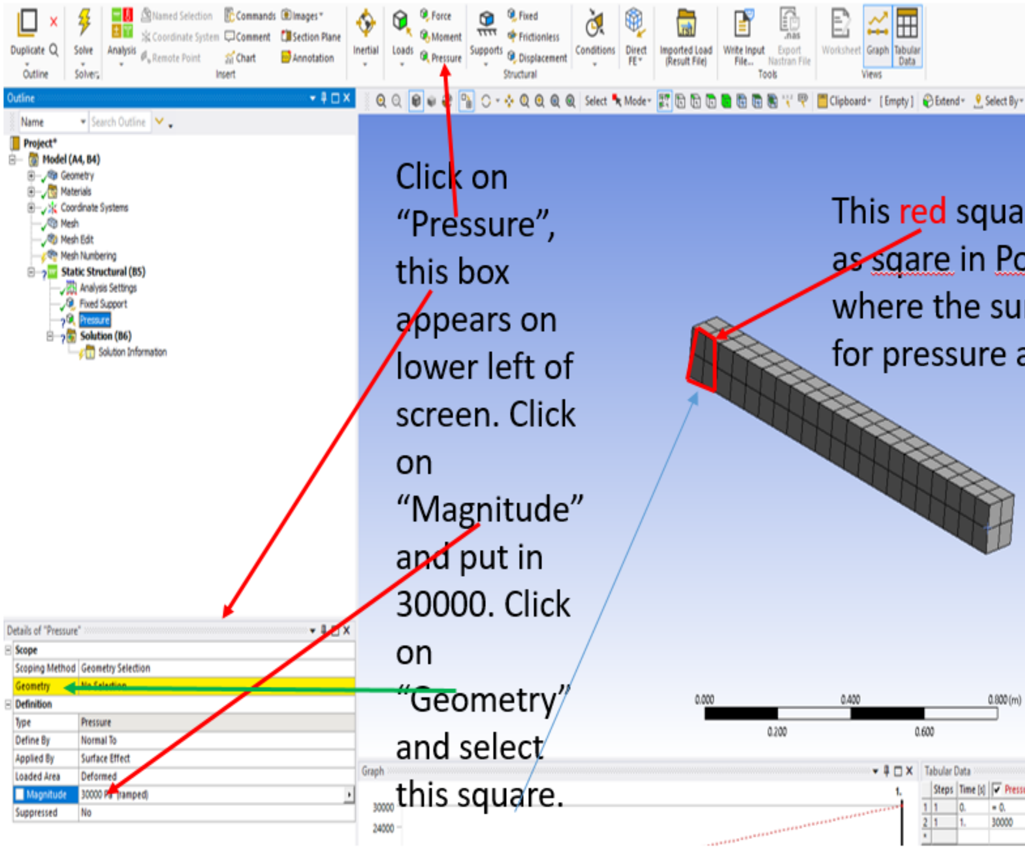
Square Geometry is being shown as rectangular because of stretching in Powerpoint.



Square Geometry is being shown as rectangular because of stretching in Powerpoint.

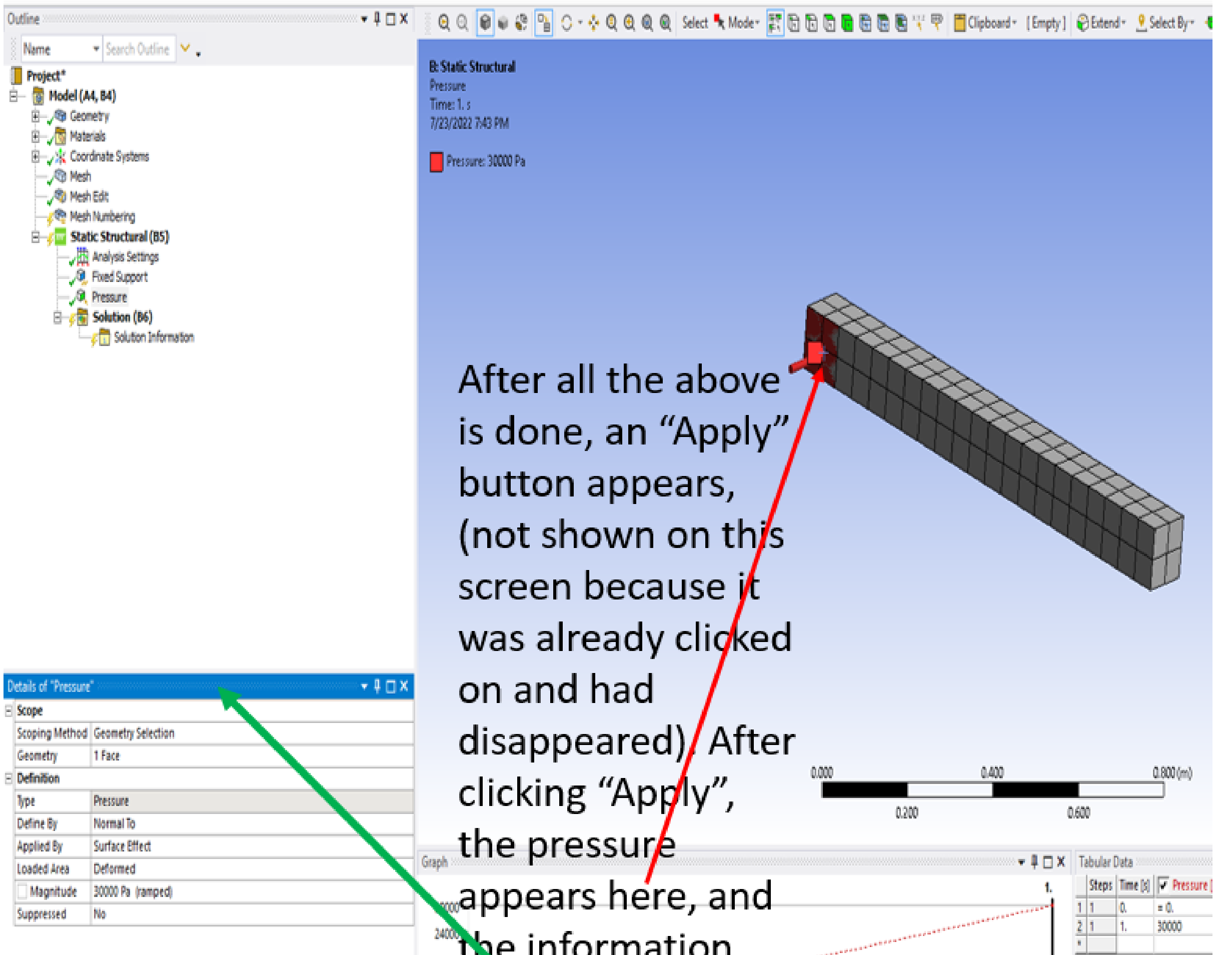


Square Geometry is being shown as rectangular because of stretching in Powerpoint.

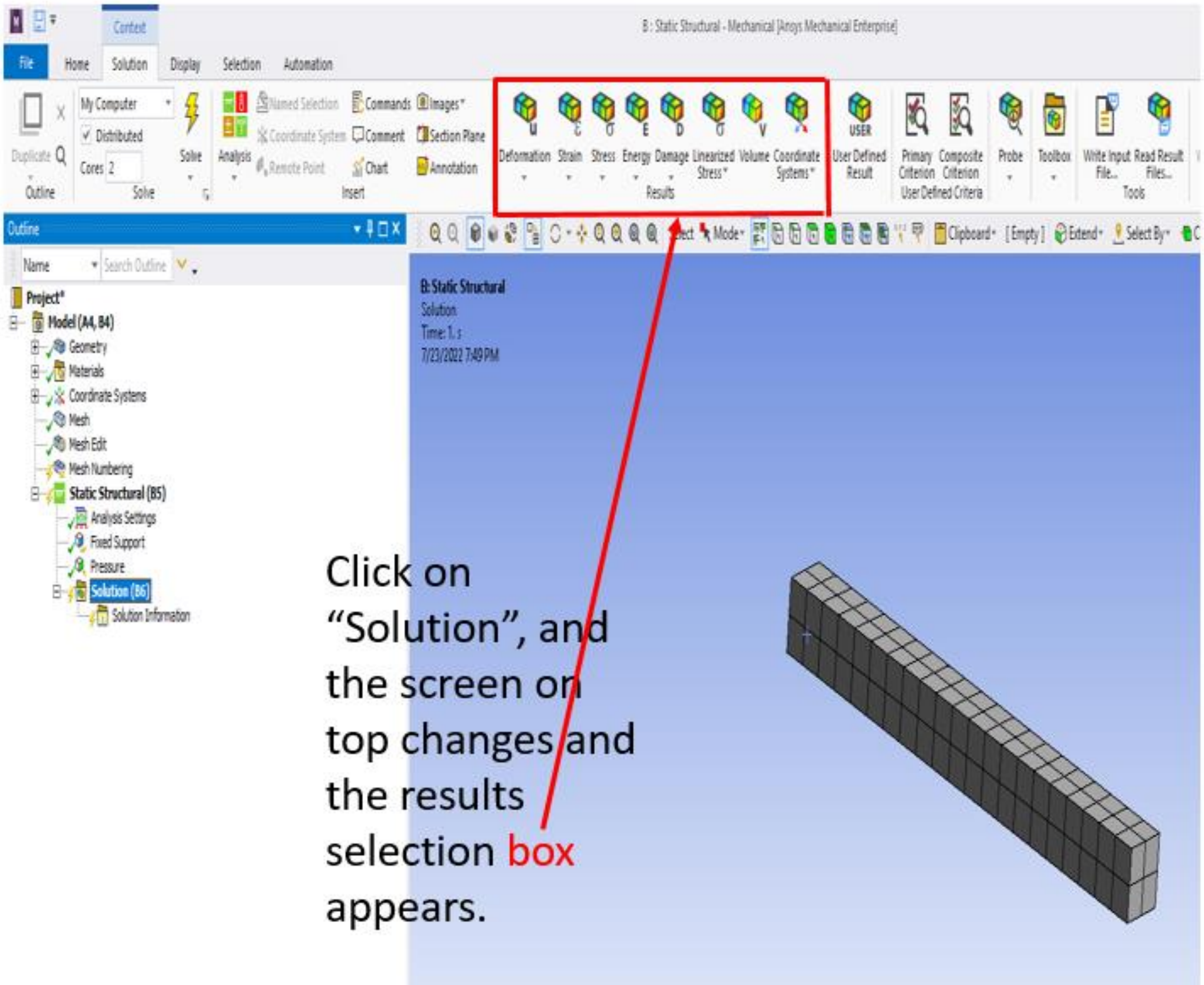


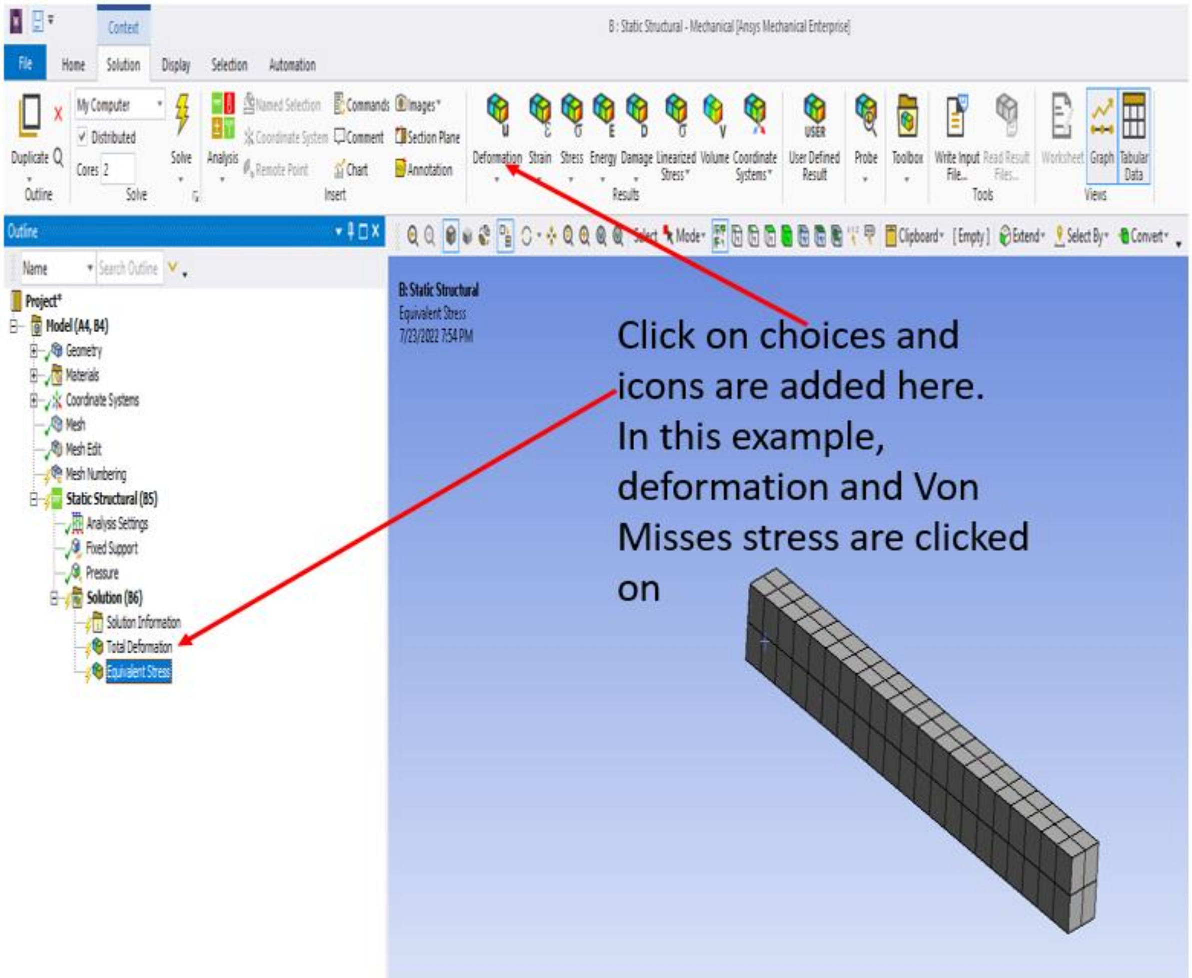
Click on "Pressure", this box appears on lower left of screen. Click on "Magnitude" and put in 30000. Click on "Geometry" and select this square.

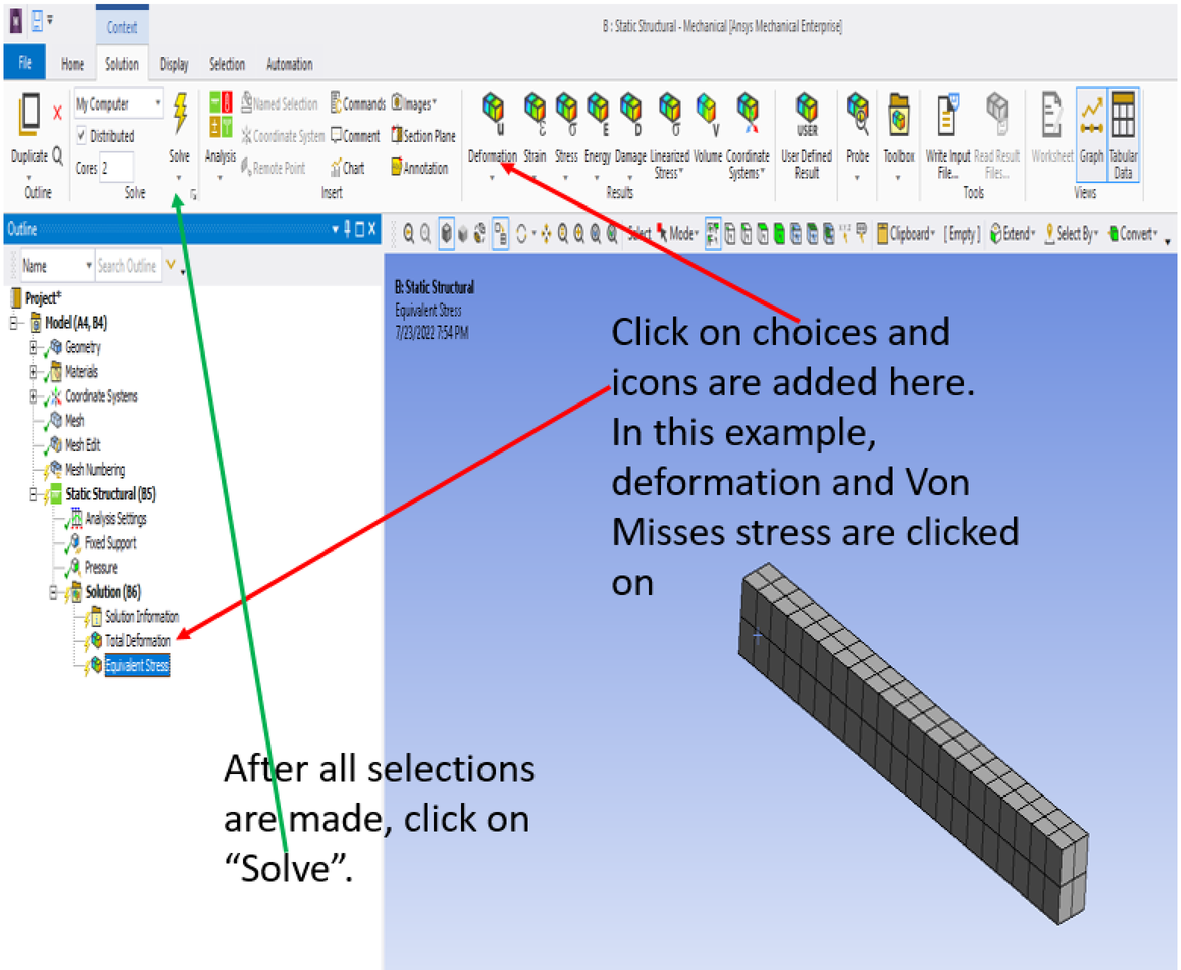
This red square cannot be drawn as sqare in Powerpoint. This is where the surface was created for pressure application.



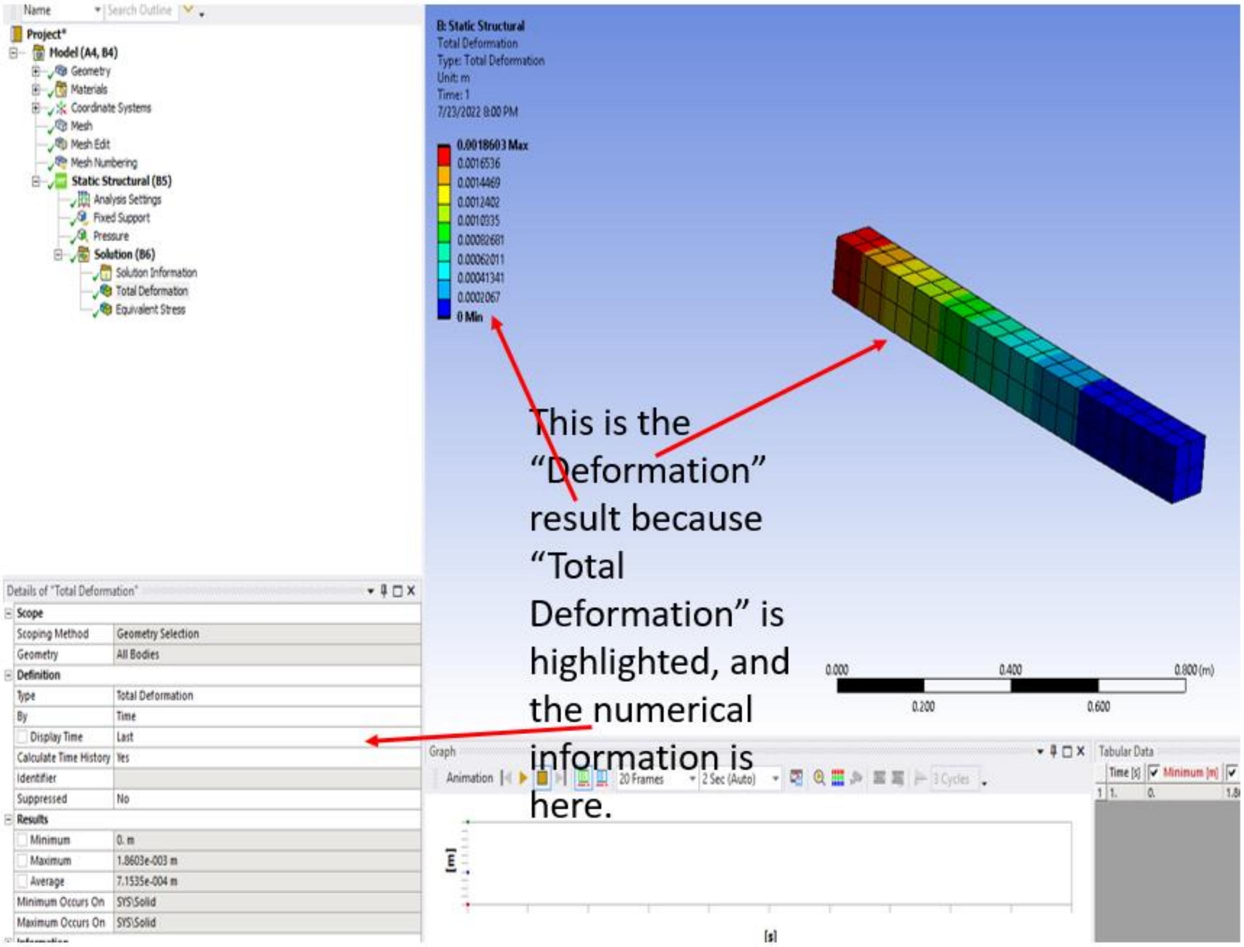
After all the above is done, an "Apply" button appears, (not shown on this screen because it was already clicked on and had disappeared). After clicking "Apply", the pressure appears here, and the information box appears in lower left part of screen.

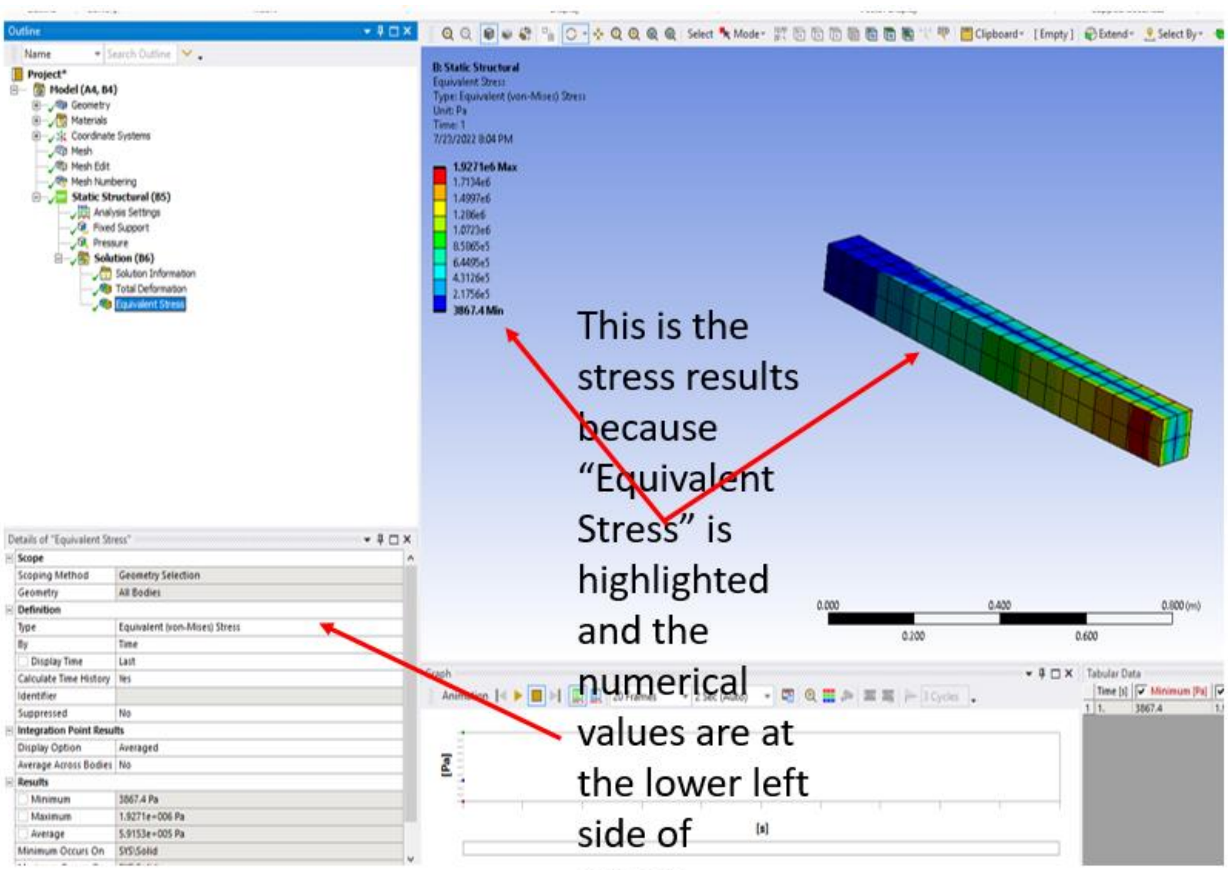

B : Static Structural - Mechanical [Ansys Mechanical Enterprise]
Click on "Solution", and the screen on top changes and the results selection box appears.


B : Static Structural - Mechanical [Ansys Mechanical Enterprise]
File
Home
Solution
Display
Selection
Automation
Outline
Name
Project*
Model (A4, B4)
Geometry
Materials
Coordinate Systems
Mesh
Mesh Edit
Mesh Numbering
Static Structural (B5)
Analysis Settings
Fixed Support
Pressure
Solution (B6)
Solution Information
Total Deformation
Equivalent Stress
B: Static Structural
Equivalent Stress
7/23/2022 7:54 PM
Click on choices and
icons are added here.
In this example,
deformation and Von
Misses stress are clicked
on



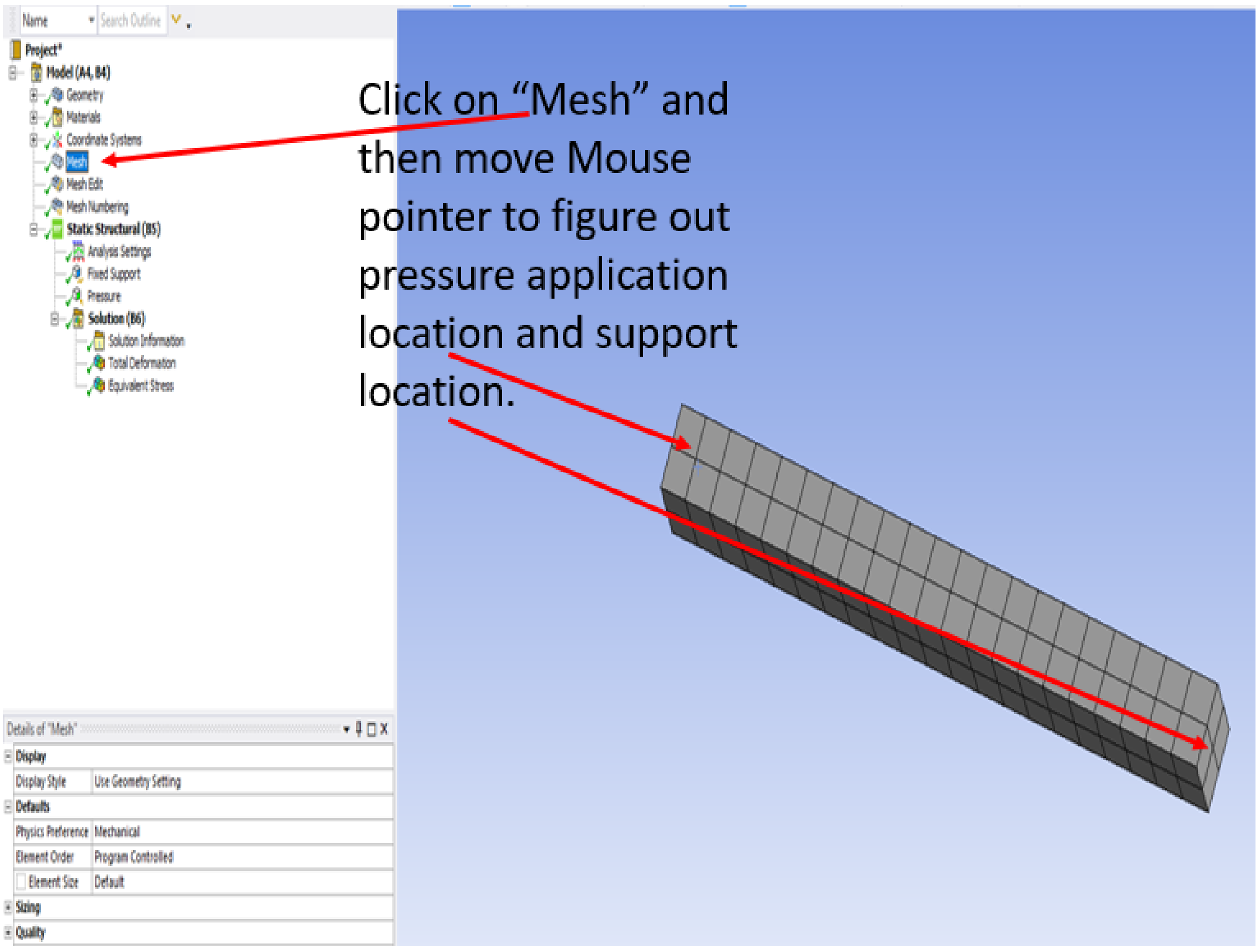
The solution will be obtained after some security warnings by the computer. Allow the access the software requires.

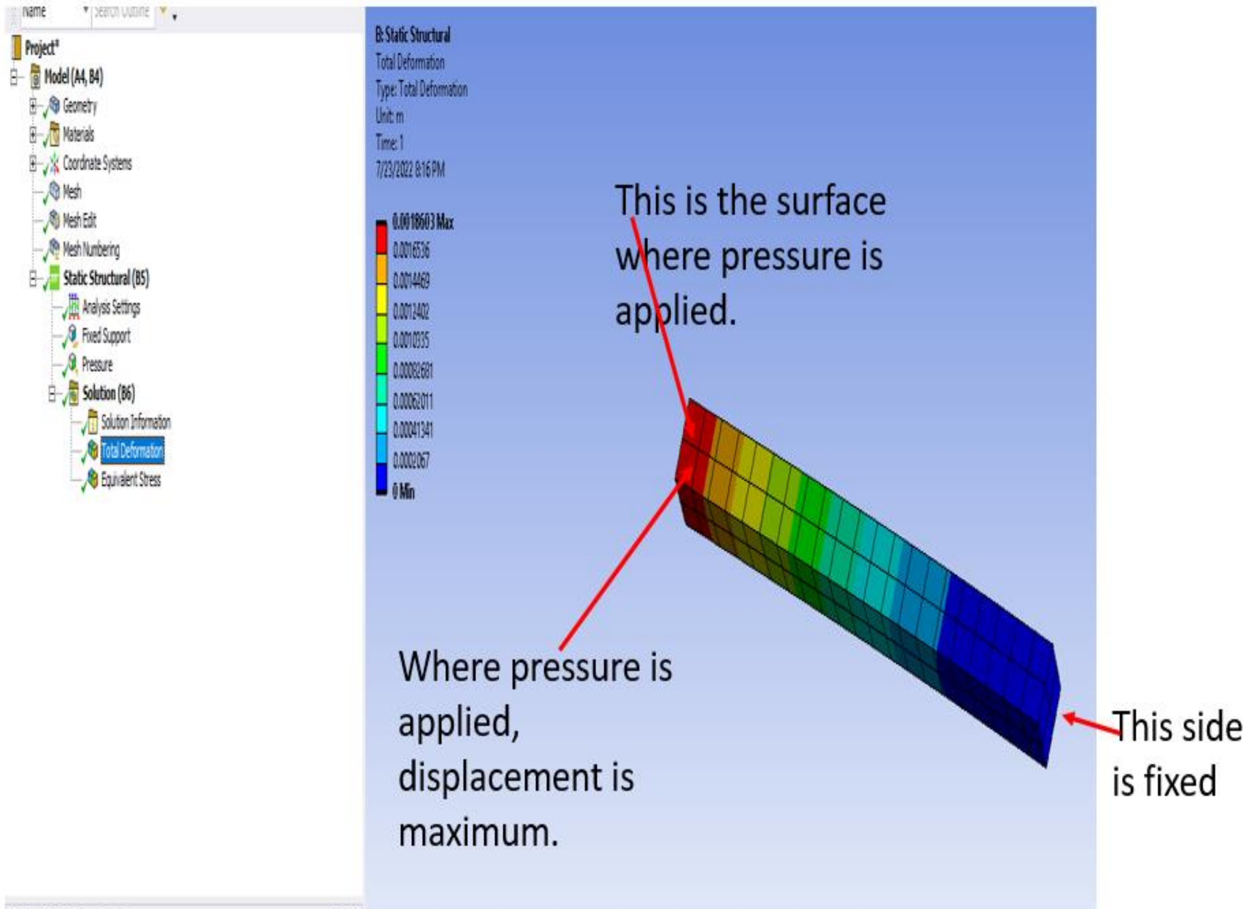

Project*
Model (A4, B4)
Geometry
Materials
Coordinate Systems
Mesh
Mesh Edit
Mesh Numbering
Static Structural (B5)
Analysis Settings
Fixed Support
Pressure
Solution (B6)
Solution Information
Total Deformation
Equivalent Stress
B: Static Structural
Total Deformation
Type: Total Deformation
Unit: m
Time: 1
7/23/2022 8:00 PM
0.0018603 Max
0.0016536
0.0014469
0.0012402
0.0010335
0.00082681
0.00062011
0.00041341
0.0002067
0 Min
This is the "Deformation" result because "Total Deformation" is highlighted, and the numerical information is here.
Details of "Total Deformation"
Scope
Scoping Method Geometry Selection
Geometry All Bodies
Definition
Type Total Deformation
By Time
Display Time Last
Calculate Time History Yes
Identifier
Suppressed No
Results
Minimum 0. m
Maximum 1.8603e-003 m
Average 7.1535e-004 m
Minimum Occurs On SYS\Solid
Maximum Occurs On SYS\Solid
0.000
0.200
0.400
0.600
0.800 (m)
Graph
Animation
20 Frames
2 Sec (Auto)
Tabular Data
[m]
[s]

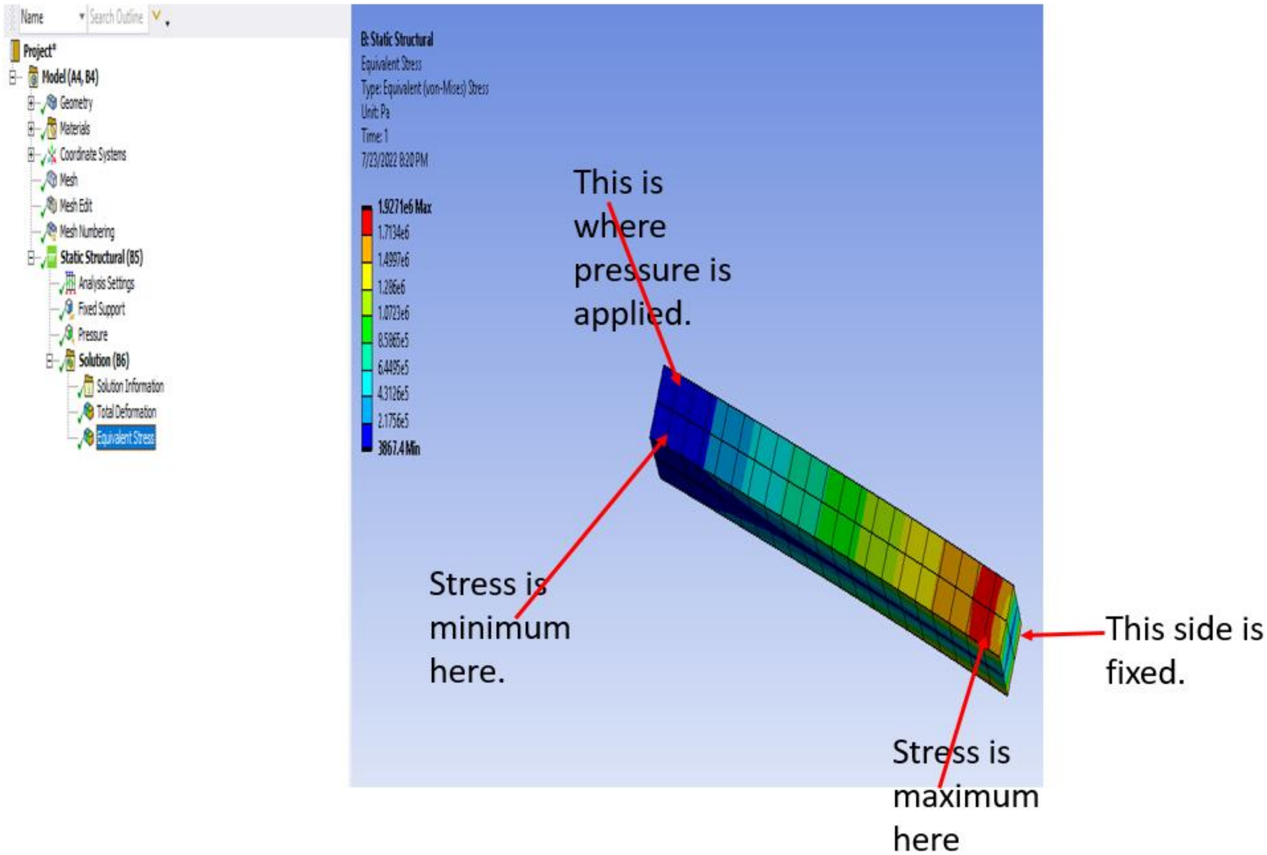


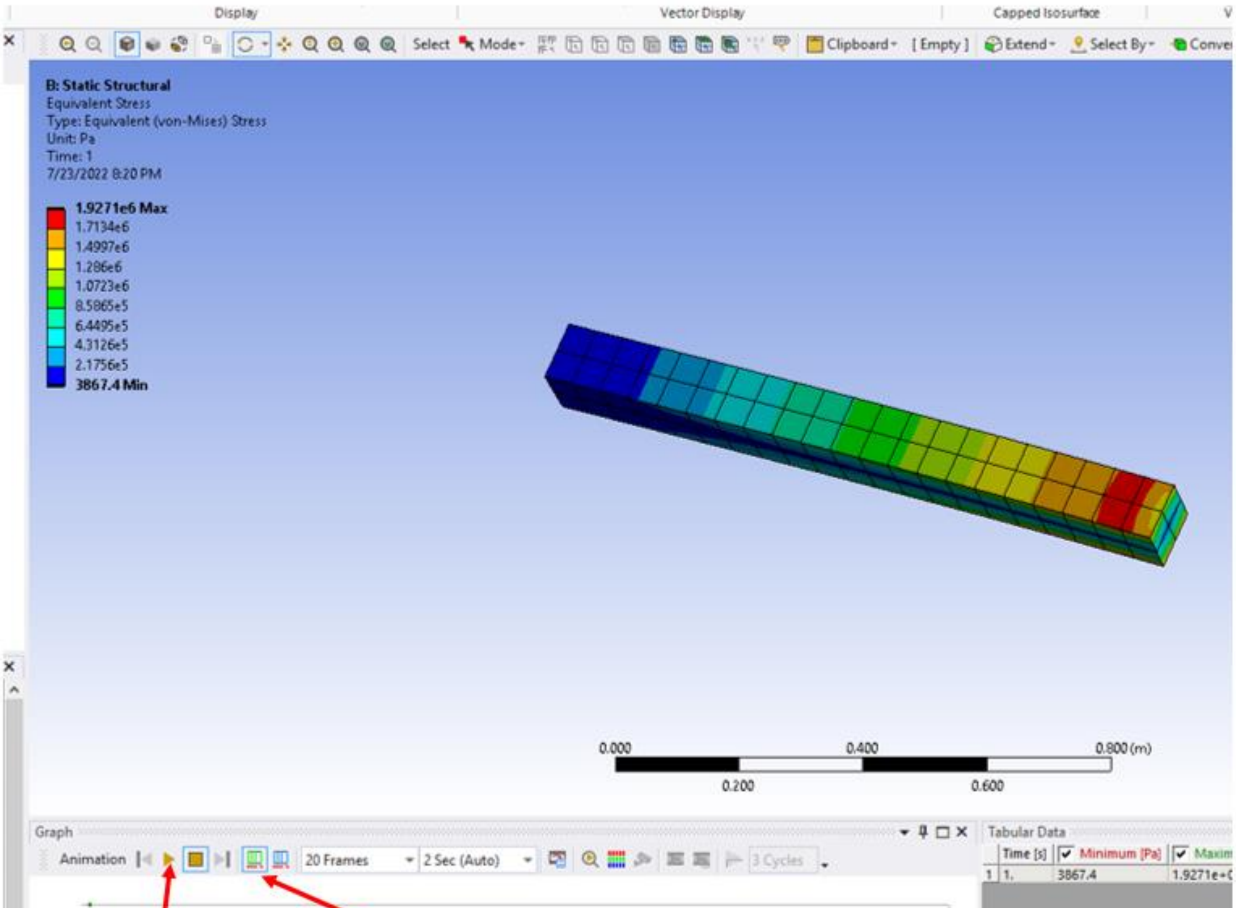
This is the stress results because "Equivalent Stress" is highlighted and the numerical values are at the lower left side of screen.

The model can be rotated by a mouse wheel to make the results more apparent. The following is a description of the process.



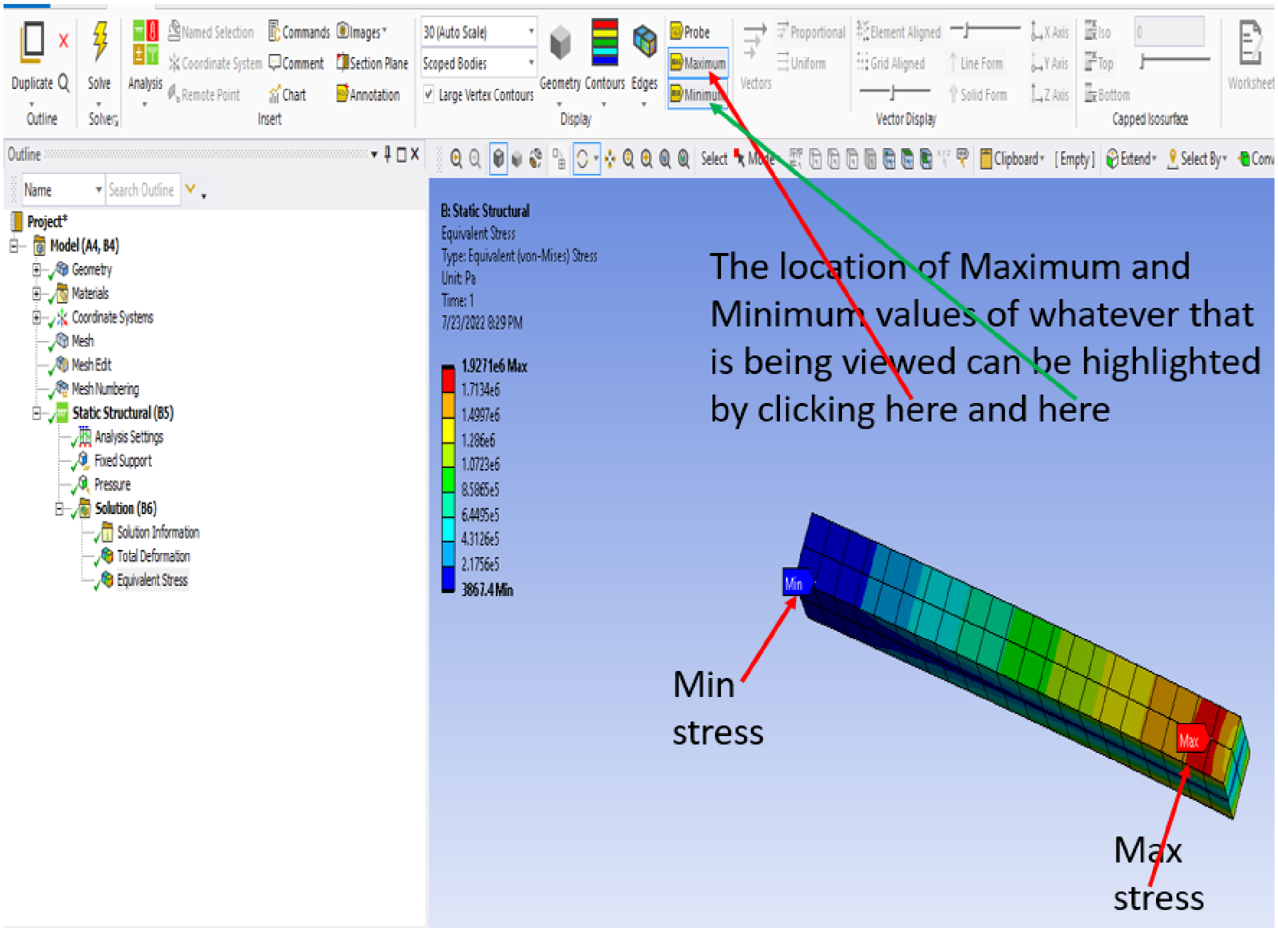

Name
Project*
Model (A4, B4)
Geometry
Materials
Coordinate Systems
Mesh
Mesh Edit
Mesh Numbering
Static Structural (B5)
Analysis Settings
Fixed Support
Pressure
Solution (B6)
Solution Information
Total Deformation
Equivalent Stress
B: Static Structural
Total Deformation
Type: Total Deformation
Unit: m
Time: 1
7/23/2022 8:16 PM
0.0018603 Max
0.0016536
0.0014469
0.0012402
0.0010335
0.00082681
0.00062011
0.00041341
0.0002067
0 Min
This is the surface where pressure is applied.
Where pressure is applied, displacement is maximum.
This side is fixed


Name
Search Outline
Project*
Model (A4, B4)
Geometry
Materials
Coordinate Systems
Mesh
Mesh Edit
Mesh Numbering
Static Structural (B5)
Analysis Settings
Fixed Support
Pressure
Solution (B6)
Solution Information
Total Deformation
Equivalent Stress
B: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: Pa
Time: 1
7/23/2022 8:20 PM
1.9271e6 Max
1.7134e6
1.4997e6
1.286e6
1.0723e6
8.5865e5
6.4495e5
4.3126e5
2.1756e5
3867.4 Min
This is where pressure is applied.
Stress is minimum here.
This side is fixed.
Stress is maximum here

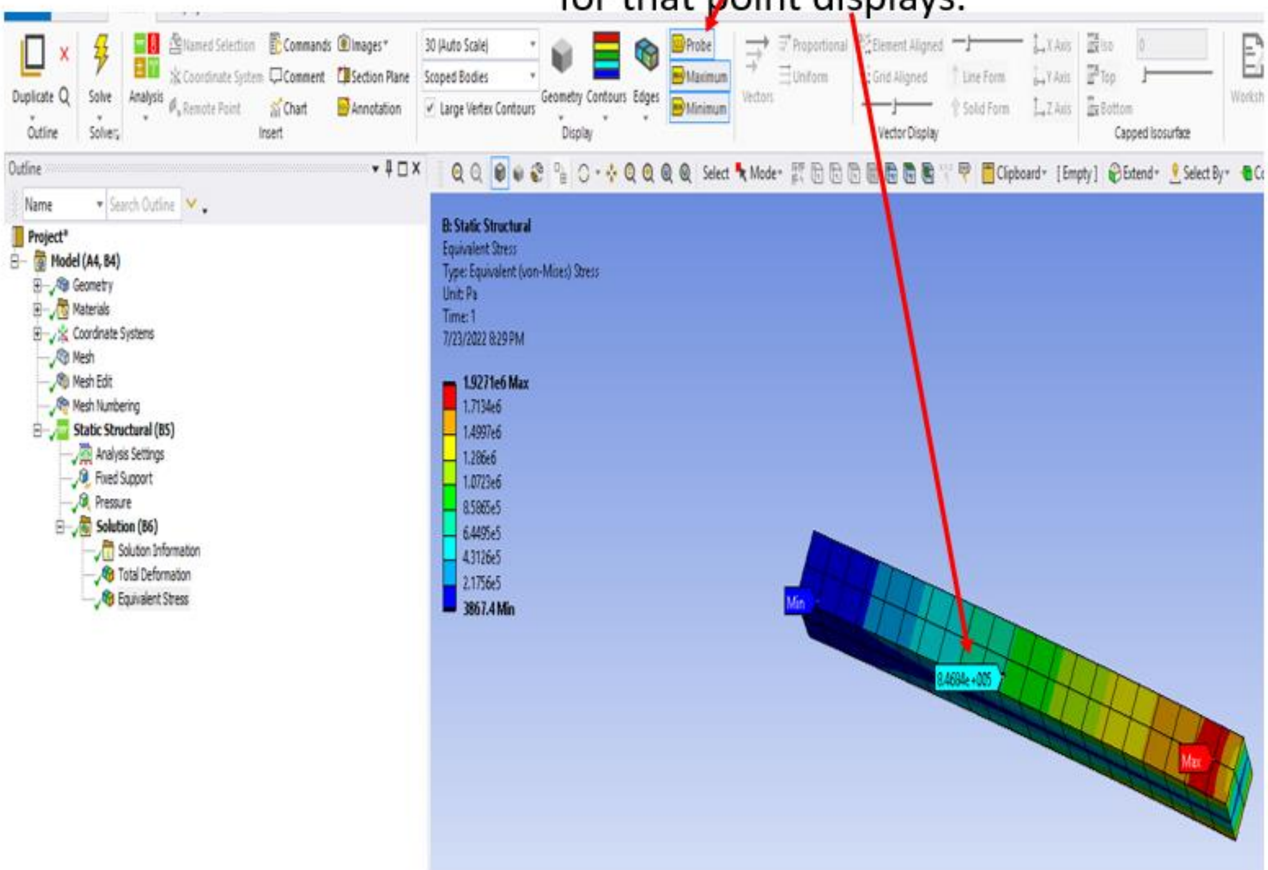


Animation can be performed by clicking here.

Animation can be stopped by clicking here

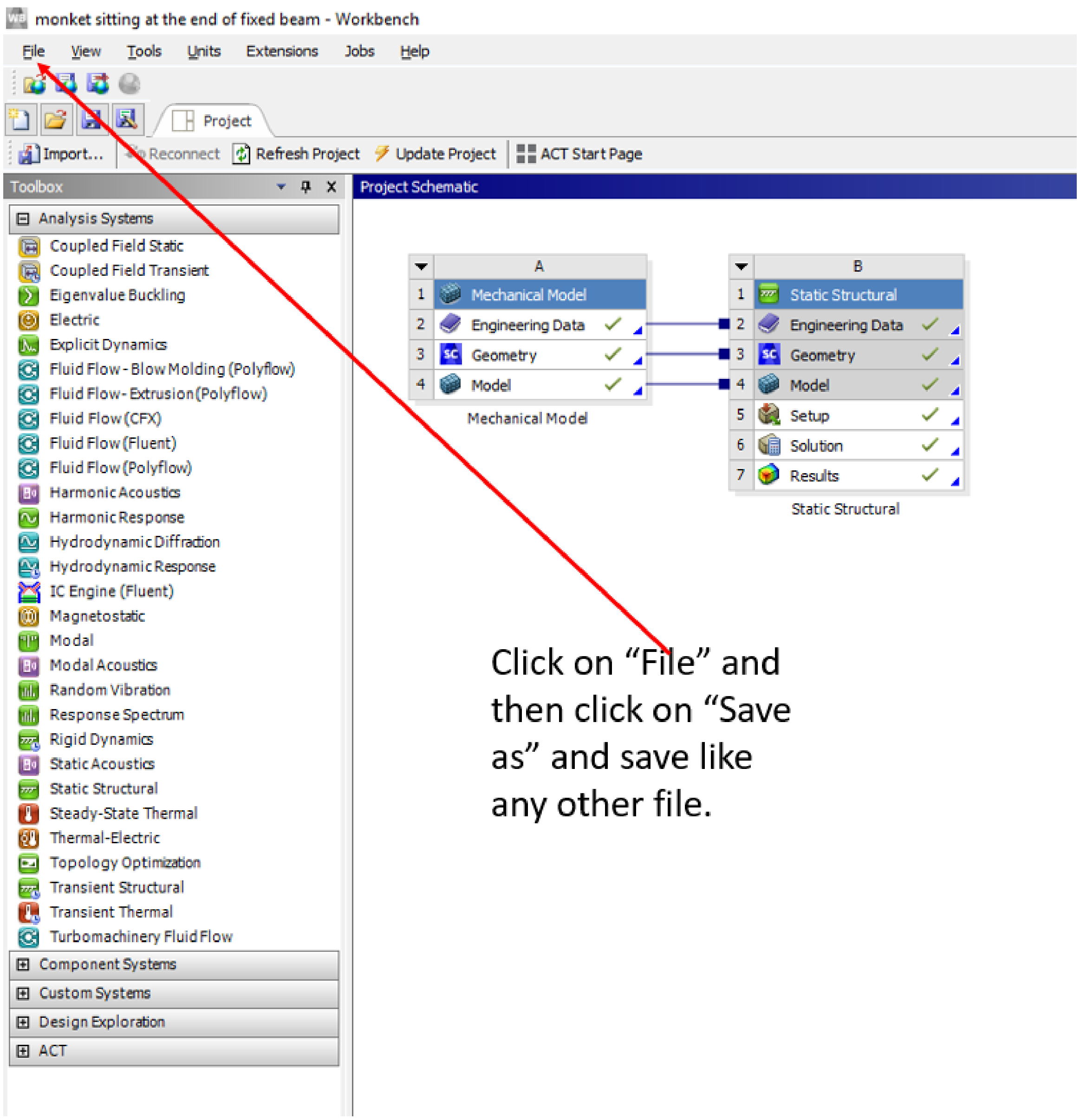

B: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: Pa
Time: 1
7/23/2022 8:29 PM
1.9271e6 Max
1.7134e6
1.4997e6
1.286e6
1.0723e6
8.5865e5
6.4495e5
4.3126e5
2.1756e5
3867.4 Min
The location of Maximum and Minimum values of whatever that is being viewed can be highlighted by clicking here and here
Min stress
Max stress

Click on "Probe" and then click at any point on the model. The result for that point displays.



To save the project, create the folder where you want to save it.

Go to WB (Work Bench) screen.



After saving, close the project by clicking “X” in the upper right corner. I am unsure if the “M” & “SC” windows will close automatically after this or if they must be closed manually.

To recover the job, go to the subdirectory (folder) where the project is saved and double-click on it. To reach various parts of the project, click on the appropriate sections on the project tree.

Class use:

These instructions were provided to students in a statics class in the author's institution as an aid in an extra credit assignment involving ANSYS static stress analysis.

Summary and conclusion:

This article provided instructions for students that had never used ANSYS or another finite element software. The detailed instructions helped the beginning students use the elementary stress analysis software.

References:

[1] YOUTUBE video titled: ANSYS Tutorial-1: Static Analysis Bending of 3D beam
[2] ANSYS software help function.
[3] A first course in the Finite Element Method; Logan