

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

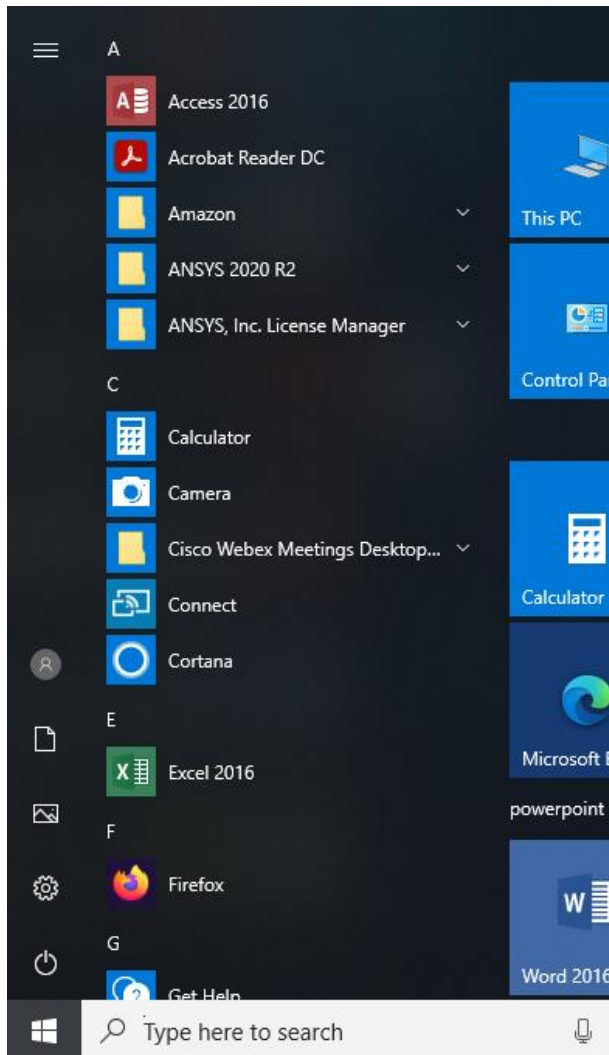
Professor of Engineering Technology at University of Toledo

Introduction:

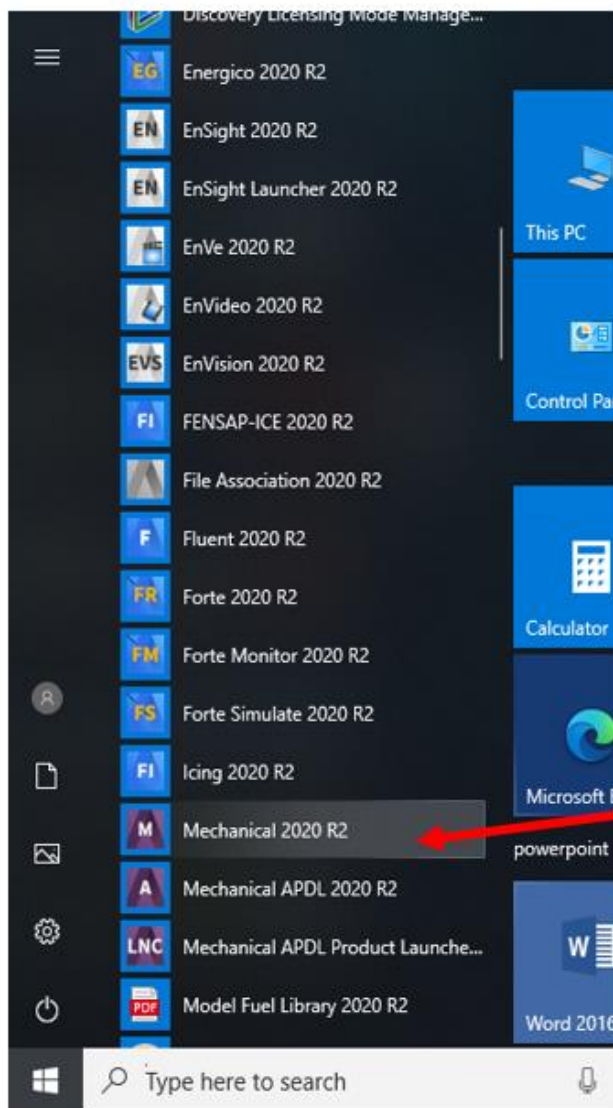
This article contains step-by-step instructions for analyzing a beam using the 2022 Rev 2 of ANSYS workbench. It must be emphasized that as future revisions of ANSYS workbench are released, the instructions presented in this article will not exactly match the software. In fact, the YOUTUBE tutorial that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



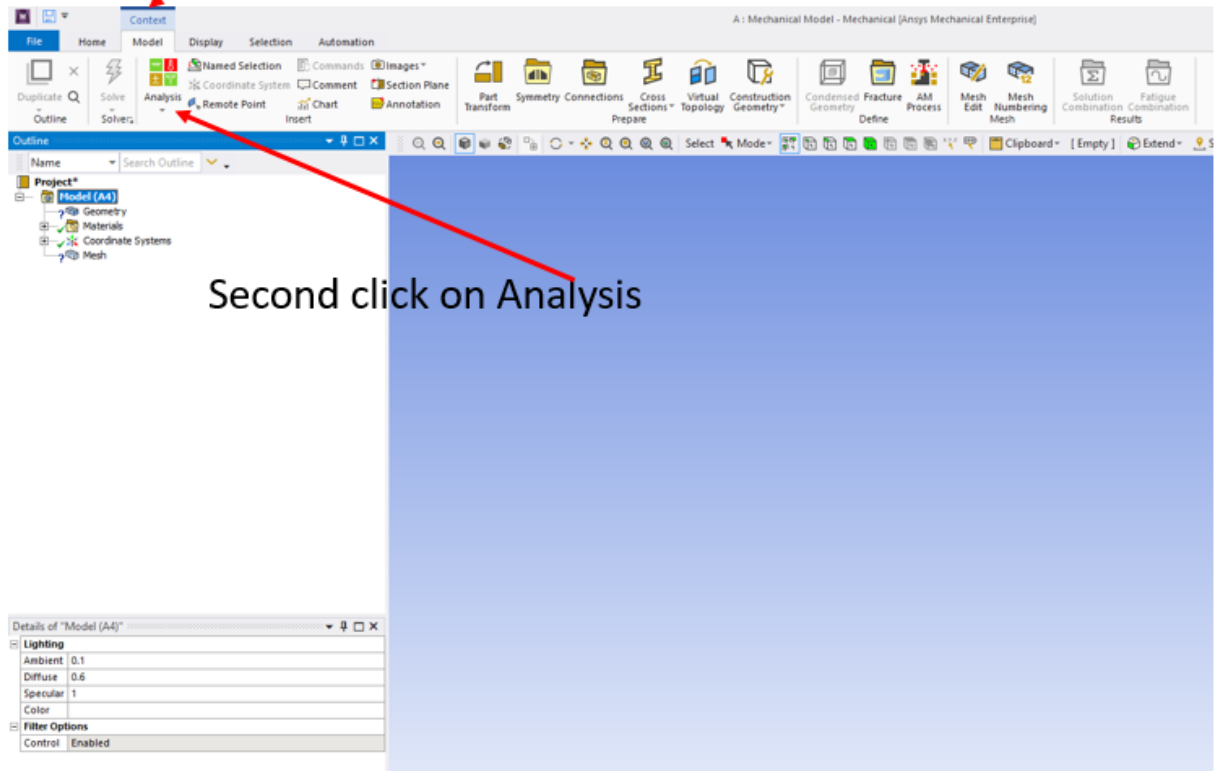
Click on “Mechanical
2020 R2

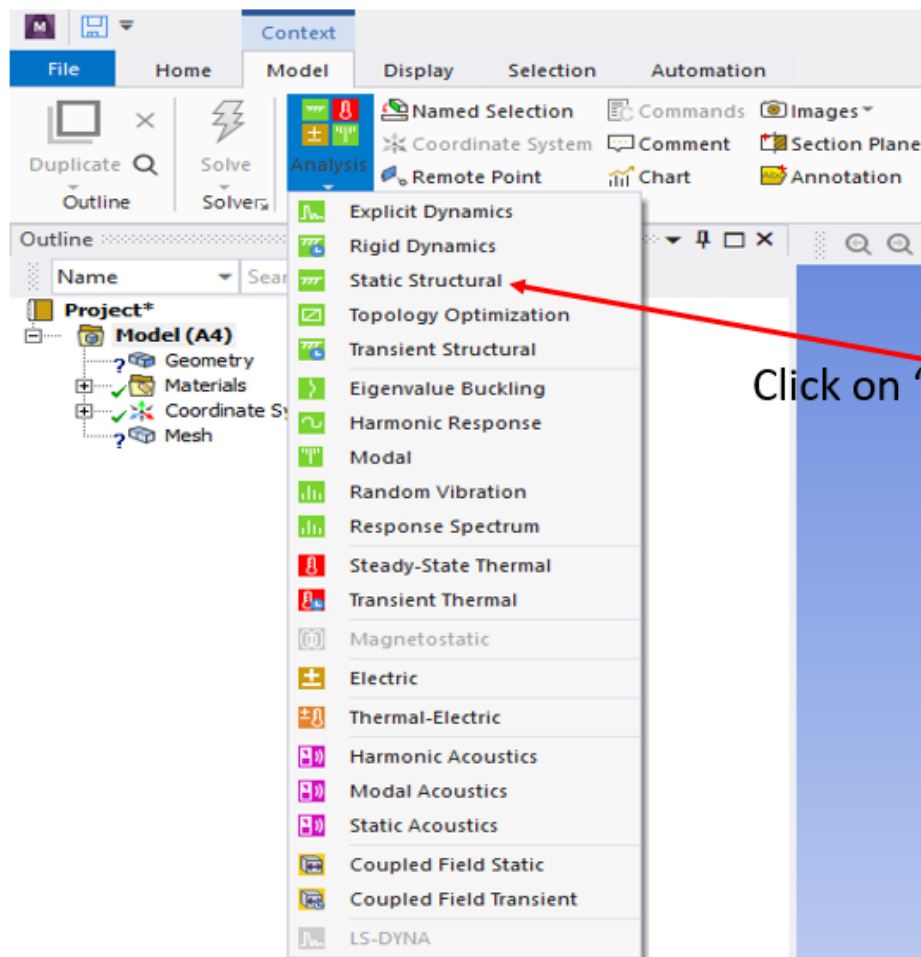
After doing the above, two separate and distinct screens become available. They can be accessed from the bottom of the Windows screen. They are like having two different programs open at the bottom. One is the project window, and the other is the “mechanical interface” window. The projects window is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

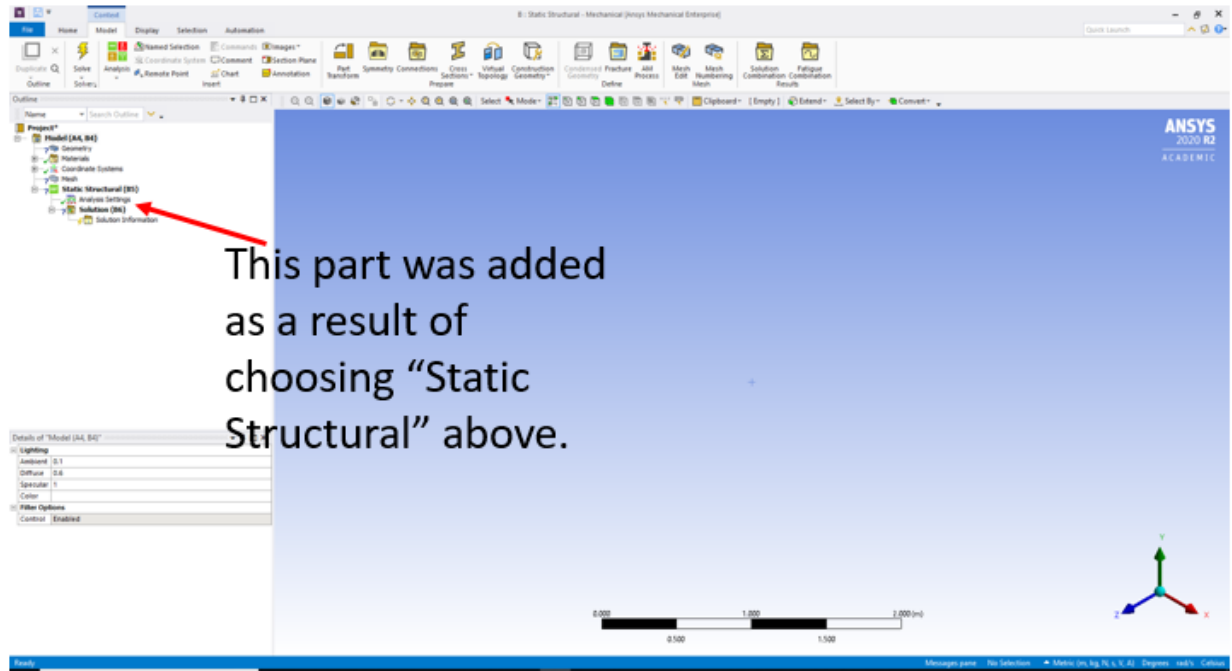
First click on context





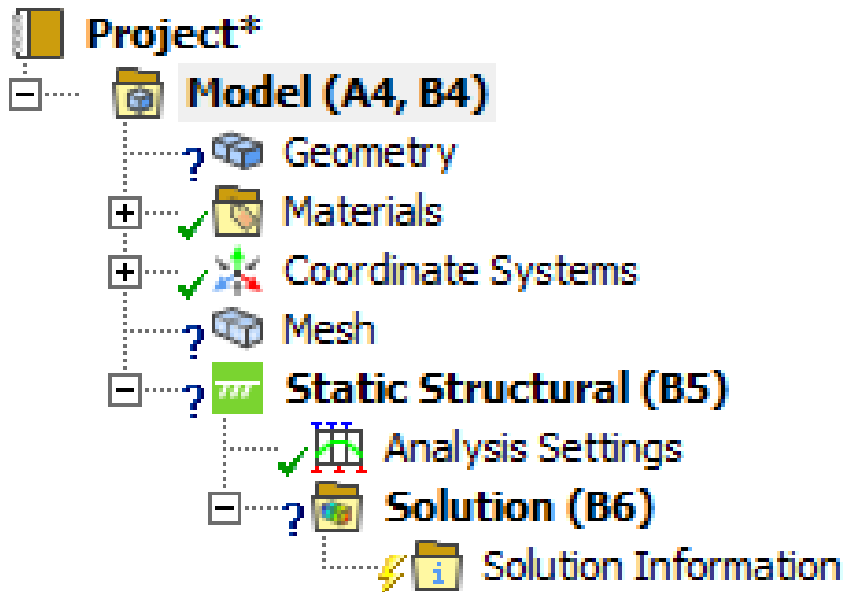
Click on “static Structural”

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

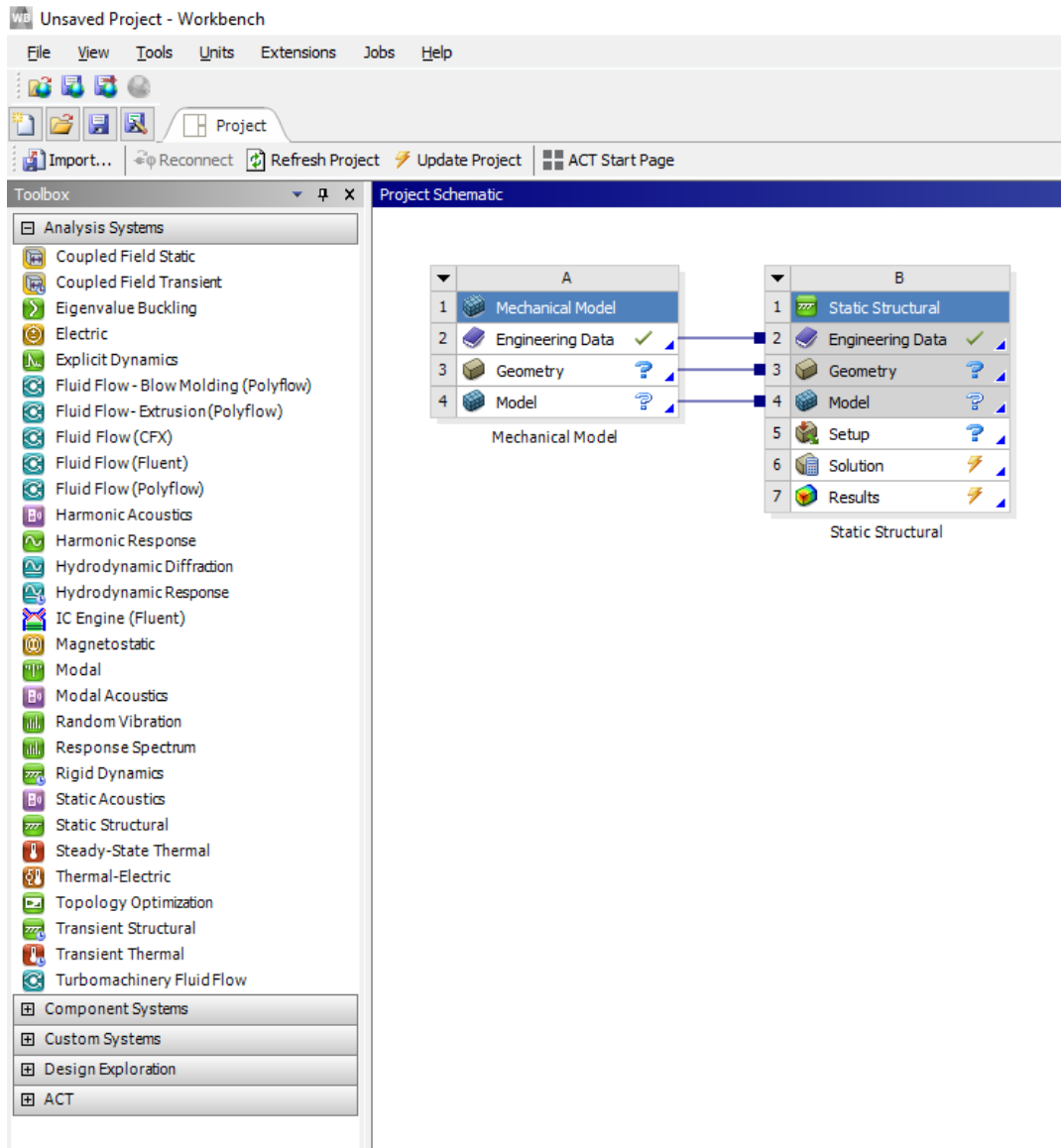


This part was added as a result of choosing “Static Structural” above.

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.

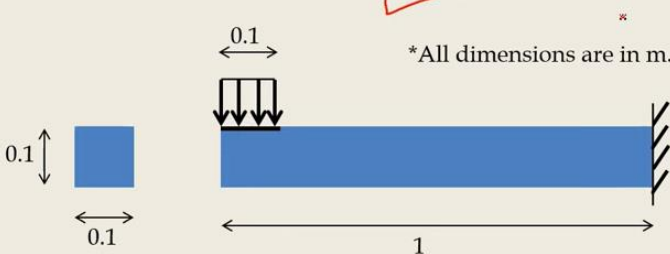
The following defines the problem that is going to be solved¹.


Not Real Engineering


ANSYS Tutorial-1: Static Analysis

Bending of 3D beam

Run your first example in ANSYS WORKBENCH



*All dimensions are in m.



Young's modulus of Wood = 8×10^9 Pa


Poisson's ratio of Wood = 0.3

Weight of monkey ≈ 30 kg

Area on which monkey is sitting = 0.01 m^2

Pressure exerted on beam by monkey = $30 \times 9.81 / 0.01$
 $= 29430 \approx 30,000 \text{ Pa}$

ANSYS workflow:

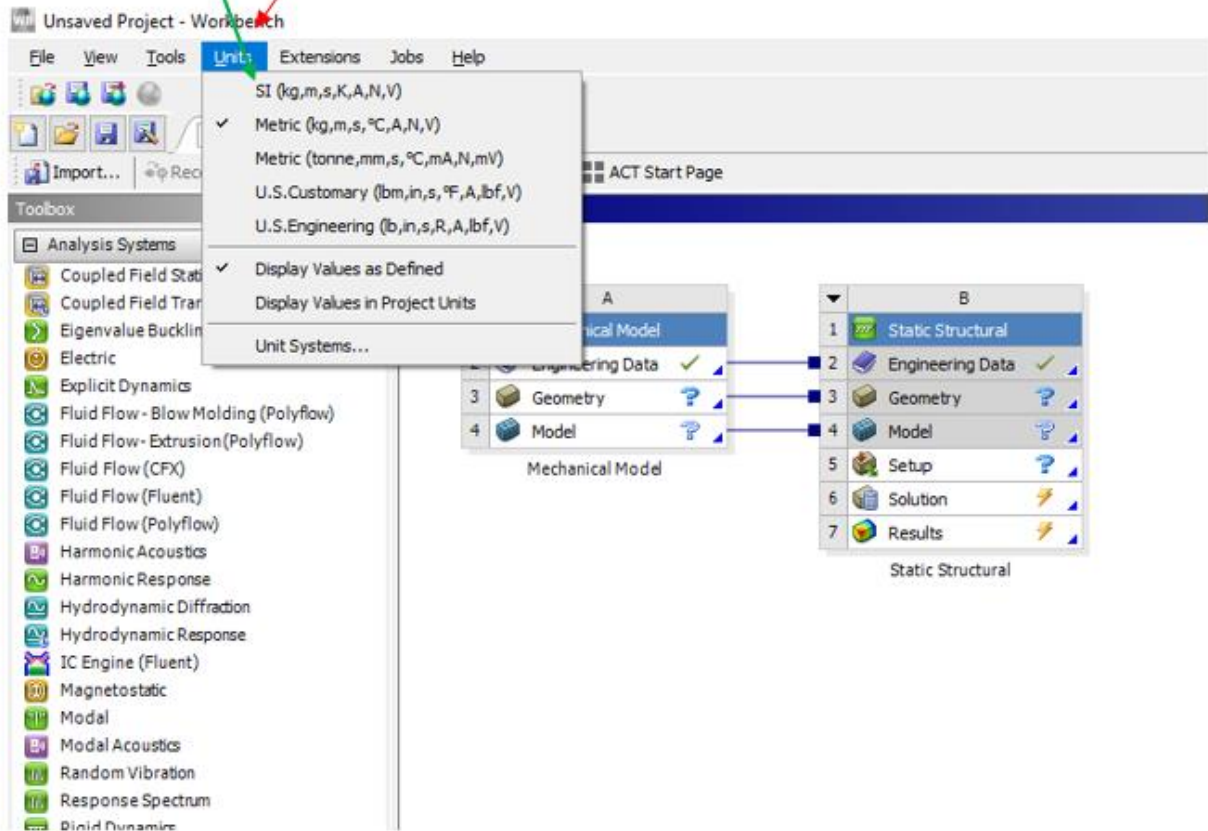


Units:

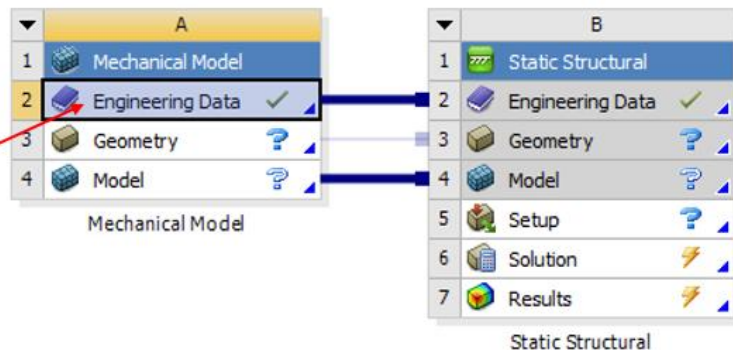
Quantity	SI
Length	m
Force	N
Mass	kg
Time	s
Stress	Pa (N/m ²)
Energy	J
Density	kg/m ³

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.

Structural Steel

The screenshot shows the 'A2/B2-Engineering Data' software interface. A red arrow points to the 'Structural Steel' material selected in the 'Outline of Engineering Data' table. The 'Properties of Outline Row 3: Structural Steel' table is expanded, showing various material properties in SI units.

Outline of Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data				Source
2	Material				
3	Structural Steel				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				

Table of Properties Row 2: Structural Steel Field Variables

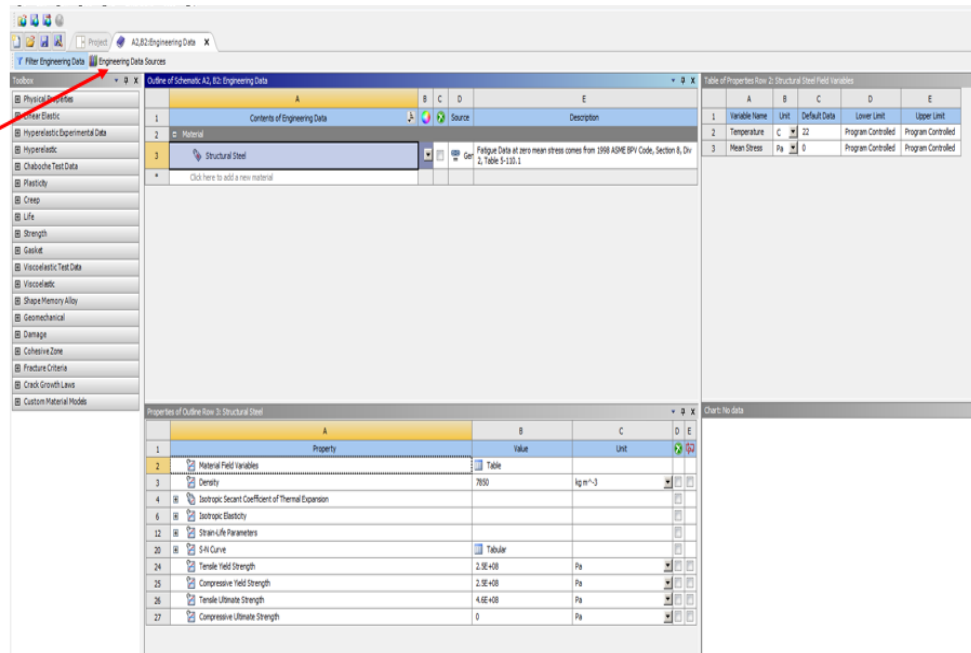
	A	B	C	D	E
1	Variable Name	Unit	Default Data	Lower Limit	Upper Limit
2	Temperature	C	22	Program Controlled	Program Controlled
3	Mean Stress	Pa	0	Program Controlled	Program Controlled

Properties of Outline Row 3: Structural Steel

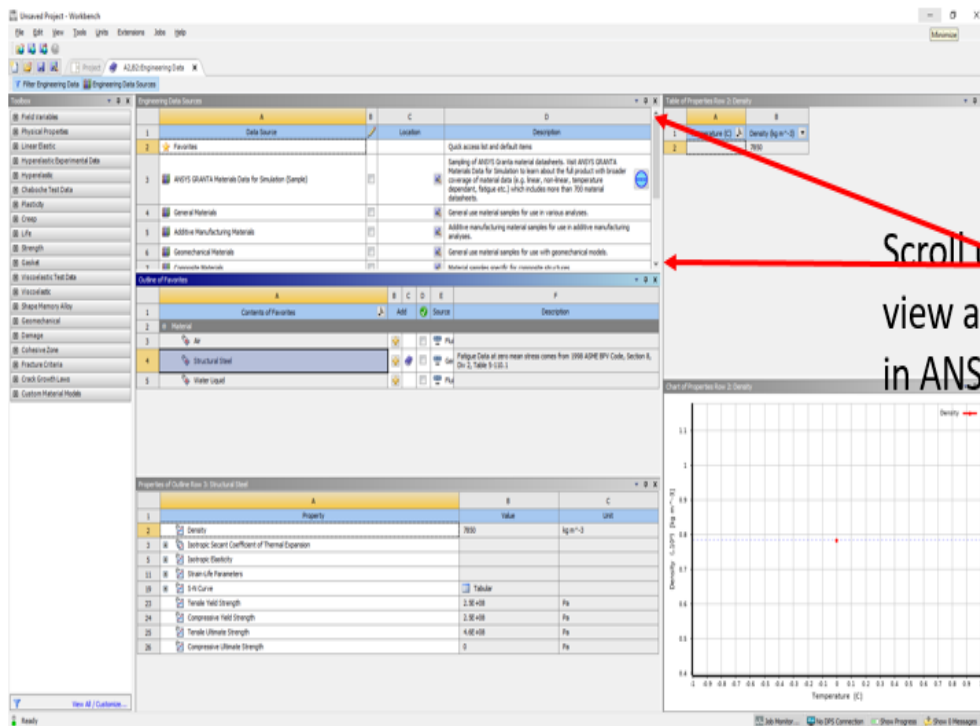
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m ⁻³		
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
12	Strain-Life Parameters				
20	S-N Curve	Tabular			
24	Tensile Yield Strength	2.9E+08	Pa		
25	Compressive Yield Strength	2.9E+08	Pa		
26	Tensile Ultimate Strength	4.6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

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.

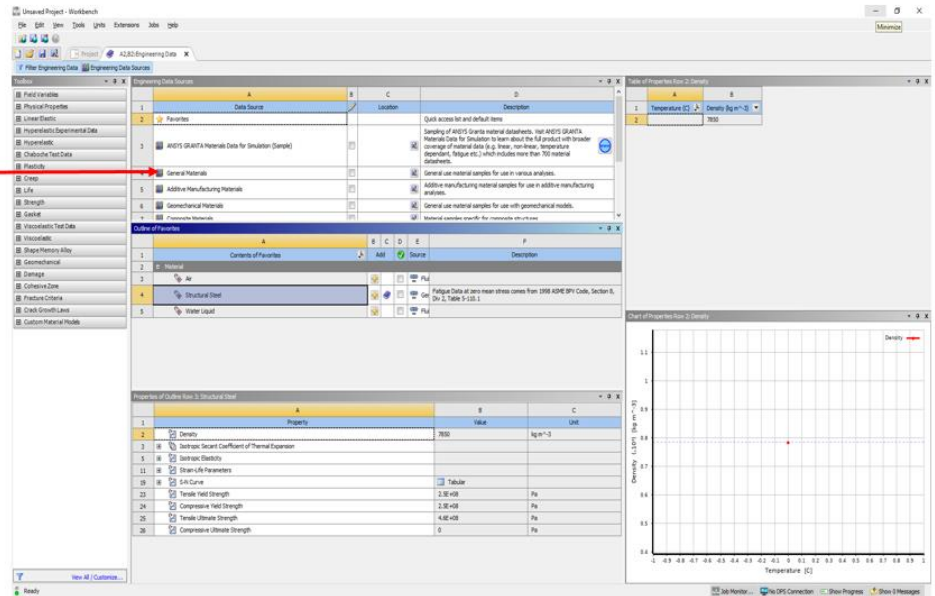


After the above, the following screen pops on. It is small, and it cannot be read.



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.

Engineering Data Sources				
	A	B	C	D
1	Data Source		Location	Description
2	★ Favorites			Quick access list and default items
3	ANSYS GRANTA Materials Data for Simulation (Sample)			Sampling of ANSYS Granta material datasheets. Visit ANSYS GRANTA Materials Data for Simulation to learn about the full product with broader coverage of material data (e.g. linear, non-linear, temperature dependant, fatigue etc.) which includes more than 700 material datasheets.
4	General Materials			General use material samples for use in various analyses.
5	Additive Manufacturing Materials			Additive manufacturing material samples for use in additive manufacturing analyses.
6	Geomechanical Materials			General use material samples for use with geomechanical models.
7	Creoniche Materials			Material samples specific for creoniche structures
Outline of General Materials				
	A	B	C	D
1	Contents of General Materials	Add	Source	Description
2	Material			
3	Air		Ger	General properties for air.
4	Aluminum Alloy		Ger	General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
5	Concrete		Ger	
6	Copper Alloy		Ger	
7	FR-4		Ger	Sample FR-4 material, data is averaged from various sources and meant for illustrative purposes. It is assumed that the material x direction is the length-wise (LW), or warp yarn direction, while the material y direction is the cross-wise (CW), or fill yarn direction.
8	Gray Cast Iron		Ger	
9	Magnesium Alloy		Ger	
Properties of Outline Row 3: Structural Steel				
	A	B	C	
1	Property	Value	Unit	
2	Density	7850	kg m ⁻³	
3	Isotropic Secant Coefficient of Thermal Expansion			
5	Isotropic Elasticity			
11	Strain-Life Parameters			
19	S-N Curve	Tabular		
23	Tensile Yield Strength	2.5E+08	Pa	
24	Compressive Yield Strength	2.5E+08	Pa	
25	Tensile Ultimate Strength	4.6E+08	Pa	
26	Compressive Ultimate Strength	0	Pa	

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

Engineering Data Sources				
	A	B	C	D
1	Data Source		Location	Description
2	★ Favorites			Quick access list and default items
3	ANSYS GRANTA Materials Data for Simulation (Sample)			Sampling of ANSYS Granta material datasheets. Visit ANSYS GRANTA Materials Data for Simulation to learn about the full product with broader coverage of material data (e.g. linear, non-linear, temperature dependant, fatigue etc.) which includes more than 700 material datasheets.
4	General Materials			General use material samples for use in various analyses.
5	Additive Manufacturing Materials			Additive manufacturing material samples for use in additive manufacturing analyses.
6	Geomechanical Materials			General use material samples for use with geomechanical models.
7	Compositional Materials			Material samples specific for composite structures
Outline of General Materials				
	A	B	C	D
1	Contents of General Materials	Add	Source	Description
2	Material			
3	Air		Ger	General properties for air.
4	Aluminum Alloy		Ger	General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
5	Concrete		Ger	
6	Copper Alloy		Ger	
7	FR-4		Ger	Sample FR-4 material, data is averaged from various sources and meant for illustrative purposes. It is assumed that the material x direction is the length-wise (LW), or warp yarn direction, while the material y direction is the cross-wise (CW), or fill yarn direction.
8	Gray Cast Iron		Ger	
9	Magnesium Alloy		Ger	
10	Orthotropic Steel		Ger	
Properties of Outline Row 3: Structural Steel				
	A	B	C	
1	Property	Value	Unit	
2	Density	7850	kg m ⁻³	
3	Isotropic Secant Coefficient of Thermal Expansion			
5	Isotropic Elasticity			
11	Strain-Life Parameters			
19	S-N Curve	Tabular		
23	Tensile Yield Strength	2.5E+08	Pa	
24	Compressive Yield Strength	2.5E+08	Pa	
25	Tensile Ultimate Strength	4.6E+08	Pa	
26	Compressive Ultimate Strength	0	Pa	

This shows that
 “Concrete” has been
 added to “Engineering
 Data Source”

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

The screenshot shows the ANSYS Workbench Engineering Data Source interface. A red arrow points to the 'Engineering Data Source' tab in the top toolbar. The main window displays a tree view on the left with categories like Field Variables, Physical Properties, and Material Models. The central pane shows a table of material properties for 'Concrete' (Material ID: 1). The table includes columns for Property, Value, and Units. The right pane shows a graph of Density vs. Temperature.

Property	Value	Units
Density	1500	kg/m ³
Thermal Expansion Coefficient of Thermal Expansion	12.1e-6	1/K
Thermal Conductivity	1.7	W/m-K
Thermal Life Parameters		
S-N Curve	Tabular	
Tensile Yield Strength	2.5e+08	Pa
Compressive Yield Strength	2.5e+08	Pa
Tensile Ultimate Strength	4.4e+08	Pa
Compressive Ultimate Strength	0	Pa

Concrete has been added.

Outline of Schematic A2, B2: Engineering Data				
	A	B	C	D
1	Contents of Engineering Data		Source	Description
2	Material			
3	Concrete		Ger	
4	Structural Steel		Ger	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table S-110.1
*	Click here to add a new material			

Table of Properties Row 2: Structural Steel Field Variables				
	A	B	C	D
1	Variable Name	Unit	Default Data	Lower Limit
2	Temperature	C	22	Program Controlled
3	Mean Stress	Pa	0	Program Controlled

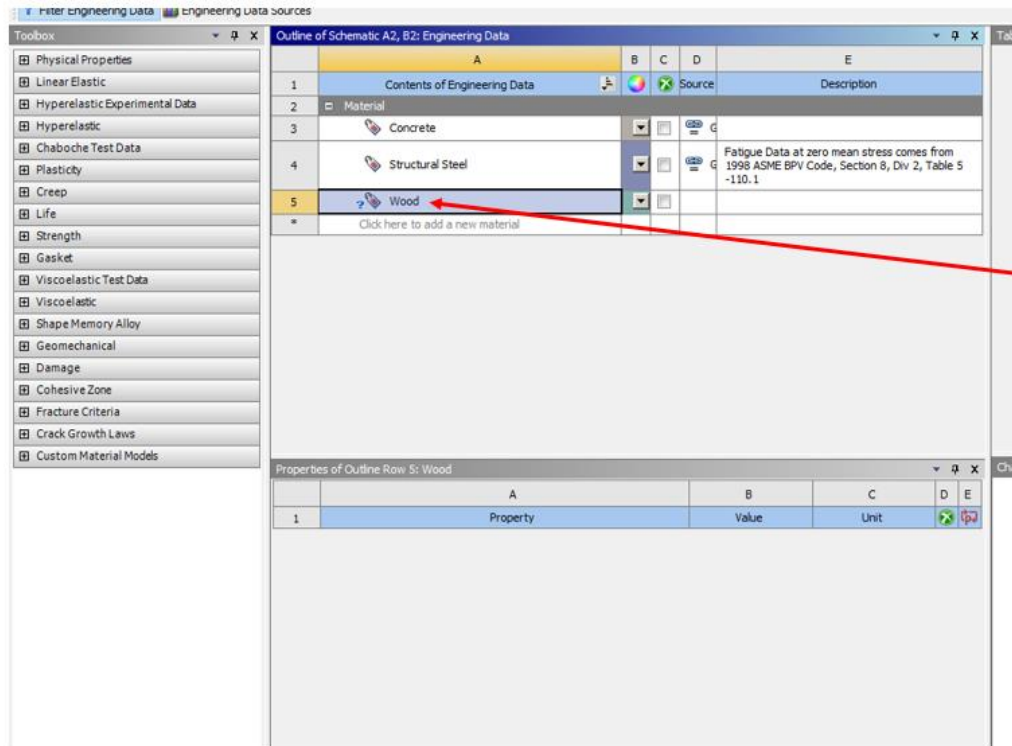
Properties of Outline Row 3: Structural Steel			
	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7850	kg m ⁻³
4	Isotropic Secant Coefficient of Thermal Expansion		
6	Isotropic Elasticity		
12	Strain-Life Parameters		
20	S-N Curve	Tabular	
24	Tensile Yield Strength	2.5E+08	Pa
25	Compressive Yield Strength	2.5E+08	Pa
26	Tensile Ultimate Strength	4.6E+08	Pa
27	Compressive Ultimate Strength	0	Pa

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

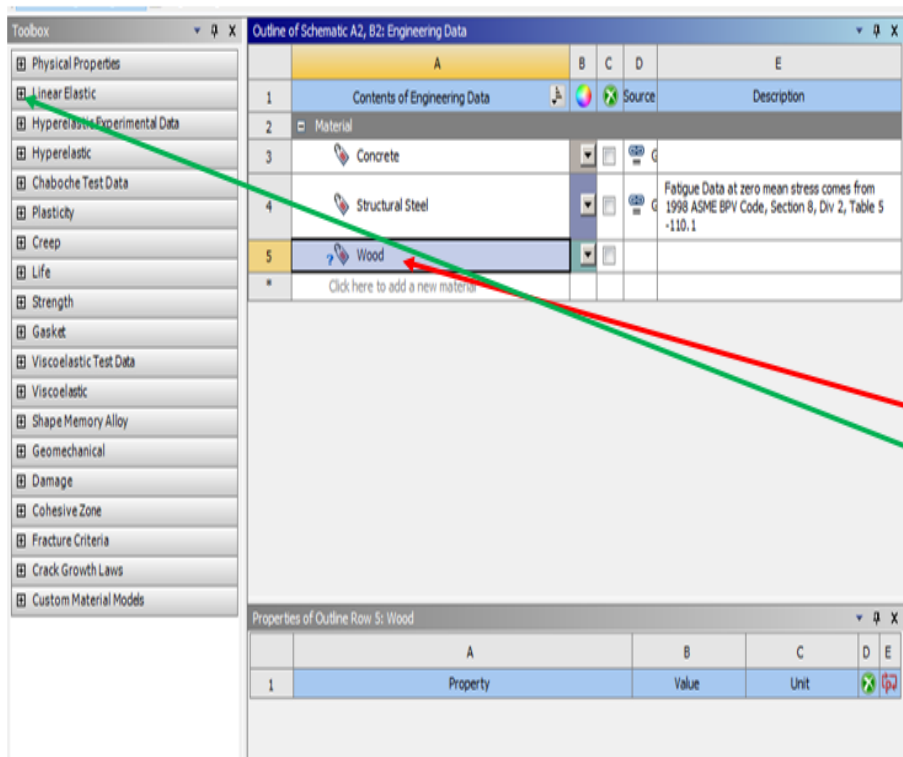
Outline of Schematic A2, B2: Engineering Data				
	A	B	C	D
1	Contents of Engineering Data		Source	Description
2	Material			
3	Concrete		Ger	
4	Structural Steel		Ger	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table S-110.1
*	Click here to add a new material			

Properties of Outline Row 3: Structural Steel			
	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7850	kg m ⁻³
4	Isotropic Secant Coefficient of Thermal Expansion		
6	Isotropic Elasticity		
12	Strain-Life Parameters		
20	S-N Curve	Tabular	
24	Tensile Yield Strength	2.5E+08	Pa
25	Compressive Yield Strength	2.5E+08	Pa
26	Tensile Ultimate Strength	4.6E+08	Pa
27	Compressive Ultimate Strength	0	Pa

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



Wood has been added.



Highlight "Wood" and click on the "+" sign next to "Linear Elastic"

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

Toolbox

- Physical Properties
 - Linear Elastic
 - Isotropic Elasticity
 - Orthotropic Elasticity
 - Anisotropic Elasticity
 - Anisotropic Temperature Dependent Elasticity
 - Anisotropic Temperature Dependent Elasticity
 - Hyperelastic Experimental Data
 - Hyperelastic
 - Chaboche Test Data
 - Plasticity
 - Creep
 - Life
 - Strength
 - Gasket
 - Viscoelastic Test Data
 - Viscoelastic
 - Shape Memory Alloy
 - Geomechanical
 - Damage
 - Cohesive Zone
 - Fracture Criteria
 - Crack Growth Laws
 - Custom Material Models

Outline of Schematic A2, B2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Concrete				
4	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1
5	Wood				
*	Click here to add a new material				

Properties of Outline Row 5: Wood

	A	B	C	D	E
1	Property	Value	Unit		

Double Click on
“Isotropic Elasticity”

Toolbox

- Physical Properties
 - Linear Elastic
 - Isotropic Elasticity
 - Orthotropic Elasticity
 - Anisotropic Elasticity
 - Anisotropic Temperature Dependent Elasticity
 - Anisotropic Temperature Dependent Elasticity
 - Hyperelastic Experimental Data
 - Hyperelastic
 - Chaboche Test Data
 - Plasticity
 - Creep
 - Life
 - Strength
 - Gasket
 - Viscoelastic Test Data
 - Viscoelastic
 - Shape Memory Alloy
 - Geomechanical
 - Damage
 - Cohesive Zone
 - Fracture Criteria
 - Crack Growth Laws
 - Custom Material Models

Outline of Schematic A2, B2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Concrete				
4	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1
5	Wood				
*	Click here to add a new material				

Properties of Outline Row 5: Wood

	A	B	C	D	E
1	Property	Value	Unit		

The following appears after double-clicking.

The screenshot displays the 'Engineering Data' environment. On the left, the 'Toolbox' contains categories like 'Field Variables', 'Physical Properties', 'Linear Elastic', 'Isotropic Elasticity', 'Orthotropic Elasticity', 'Anisotropic Elasticity', 'Anisotropic Temperature Dependent Elasticity', 'Hyperelastic Experimental Data', 'Hyperelastic', 'Chaboche Test Data', 'Plasticity', and 'Creep'. The 'Outline of Schematic A2: Engineering Data' pane shows a tree structure with 'Contents of Engineering Data' (Source: Description), 'Material', 'Concrete', 'Structural Steel' (with a note: 'Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -110.1'), and 'Wood'. The 'Table of Properties Row 3: Isotropic Elasticity' pane shows a table with the following structure:

	A	B	C	D	E
1	Temperature (K)	Young's Modulus (Pa)	Poisson's Ratio	Bulk Modulus (Pa)	Shear Modulus (Pa)
*					

Red arrows point to the empty cells in the row marked with an asterisk (*), indicating where isotropic elasticity material properties are added.

This is where the isotropic elasticity material properties are added.

Toolbox Outline of Schematic A2, 3D Engineering Data Table of Properties Row 3: Isotropic Elasticity





	A	B	C	D	E
1	Contents of Engineering Data		Source		Description
2	Material				
3	Concrete				
4	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5 -100.1
5	Wood				
*	Click here to add a new material				

	A	B	C	D	E
1	Temperature (K)	Young's Modulus (Pa)	Poisson's Ratio	Bulk Modulus (Pa)	Shear Modulus (Pa)
*					

Young's Modulus & Poisson's ratio are the minimum required material properties for isotropic material properties.

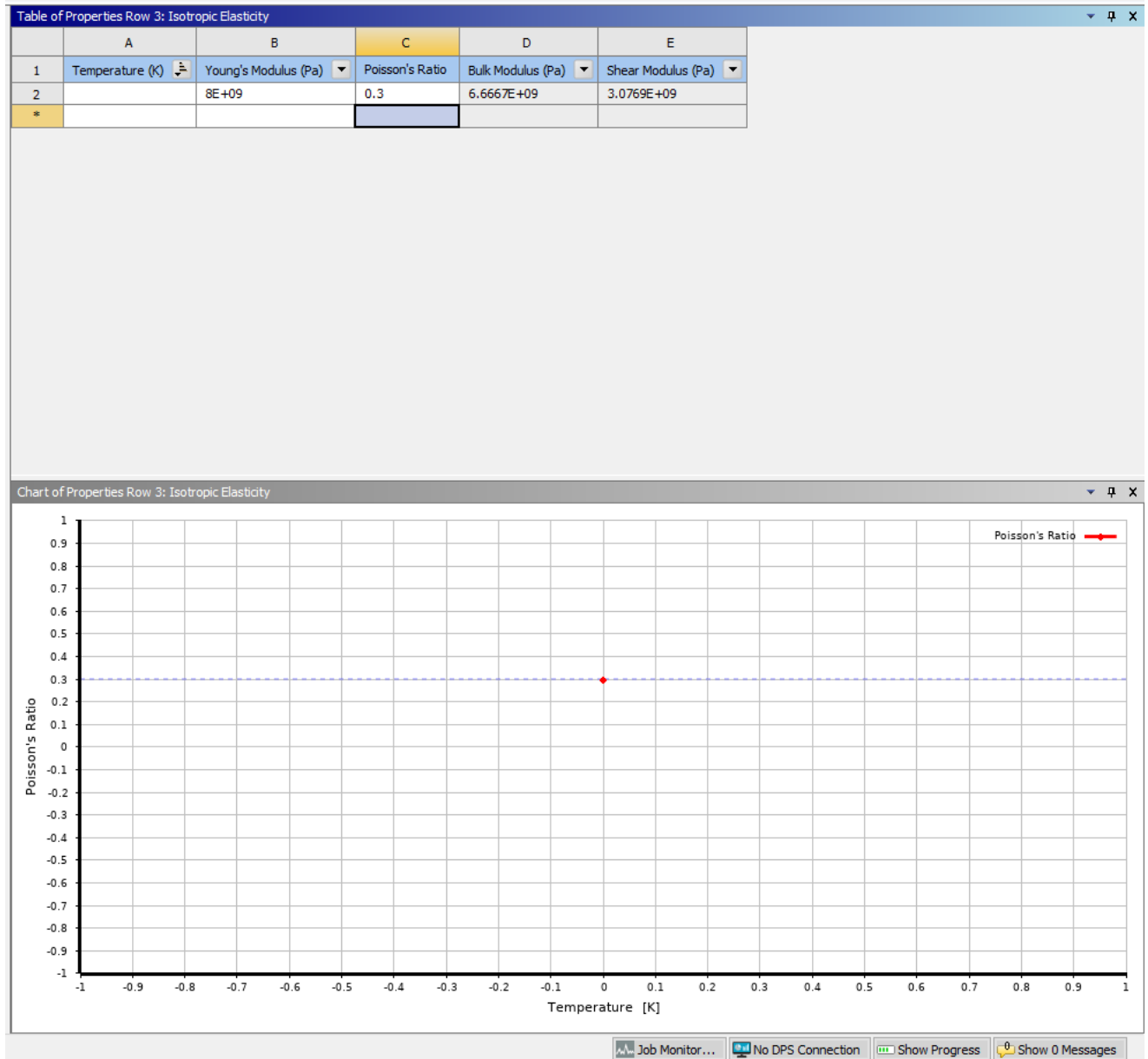
Recall from the problem description that “Young’s Modulus” and “Poisson’s ratio” are defined as shown below.

Young's modulus of Wood = 8×10^9 Pa
 Poisson's ratio of Wood = 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.”

The screenshot displays the Engineering Data Sources panel on the left, listing various material models. The main area shows a table of material properties. The 'Table of Properties Row 3: Isotropic Elasticity' is expanded, showing the following data:

	A	B	C	D	E
1	Temperature (K)	Young's Modulus (Pa)	Poisson's Ratio	Bulk Modulus (Pa)	Shear Modulus (Pa)
2		8E+09	0.3	6.6667E+09	3.0769E+09
3					

Red arrows point from the text “Bulk Modulus” and “Shear Modulus” are calculated. to the cells containing 6.6667E+09 and 3.0769E+09 respectively.

Filter Engineering Data Engineering Data Sources

Toolbox Outline of Schematic A2, B2: Engineering Data

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Concrete			Ger	
4	Structural Steel			Ger	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
5	Wood				
*	Click here to add a new material				

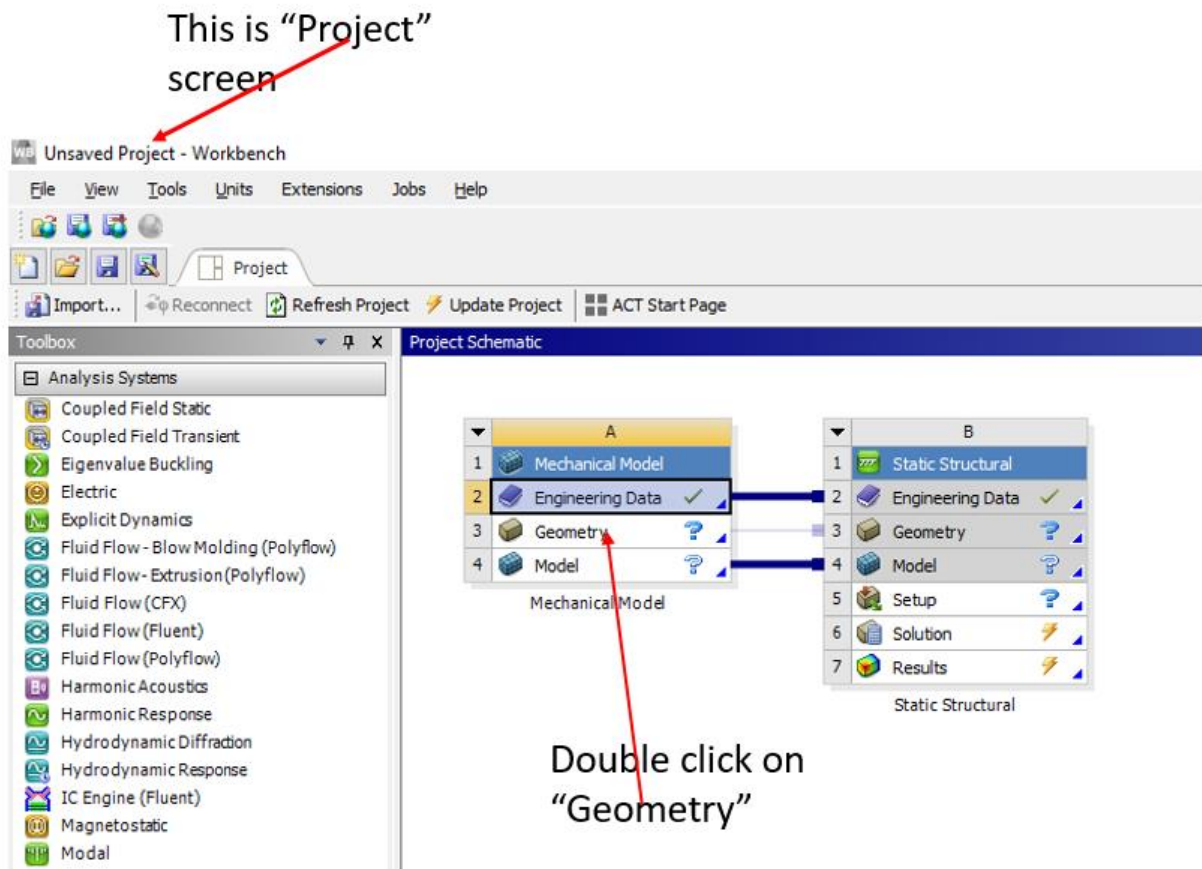
Note that 3 materials are defined, and 3 bookmarks are present at top.

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

The screenshot shows the ANSYS Workbench Engineering Data interface. The top toolbar contains a tab labeled "A2, B2: Engineering Data" with a red arrow pointing to its close button ("X"). Below the toolbar, the "Filter Engineering Data" and "Engineering Data Sources" sections are visible. The "Toolbox" on the left lists various material models, including Linear Elastic, Hyperelastic, and Plasticity. The main area displays a table titled "Outline of Schematic A2, B2: Engineering Data" with columns A, B, C, D, and E. The table lists various material models and their associated data sources.

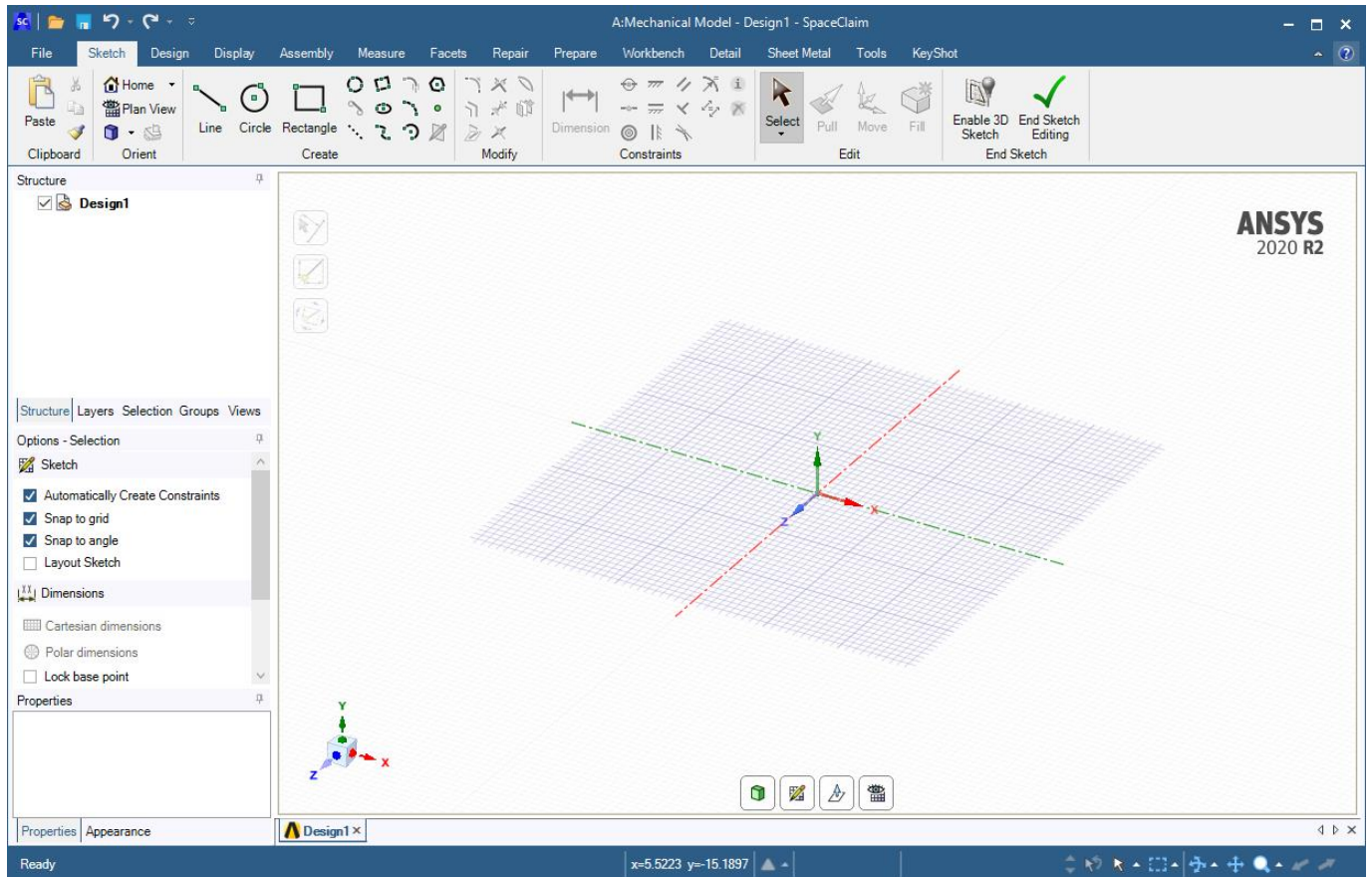
	A	B	C	D	E	
1	Contents of Engineering Data			Source	Description	1
2	Material					2
3	Concrete			Gen		*
4	Structural Steel			Gen	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table S-110.1	
5	Wood					
*	Click here to add a new material					

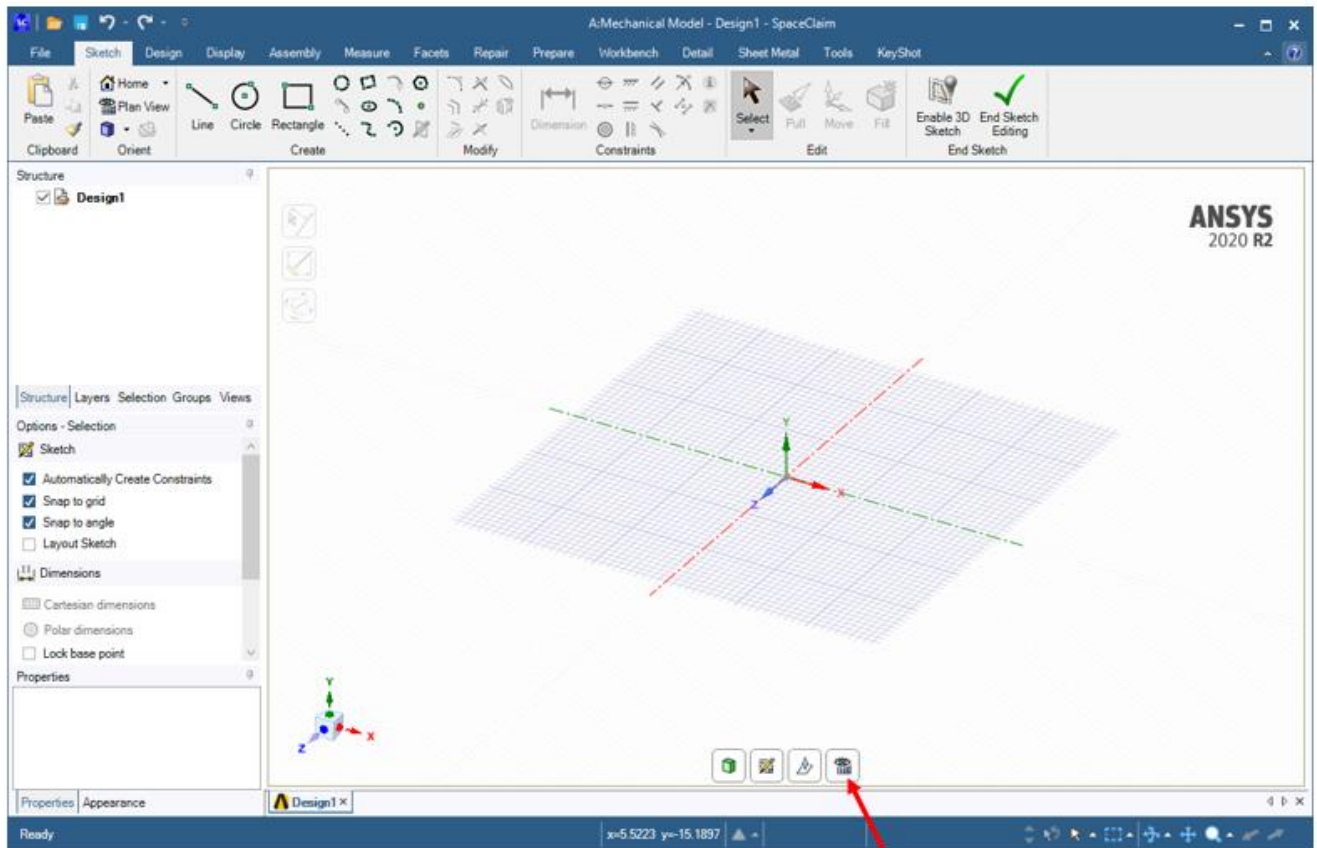
On the projects screen, double-click on “Geometry.”



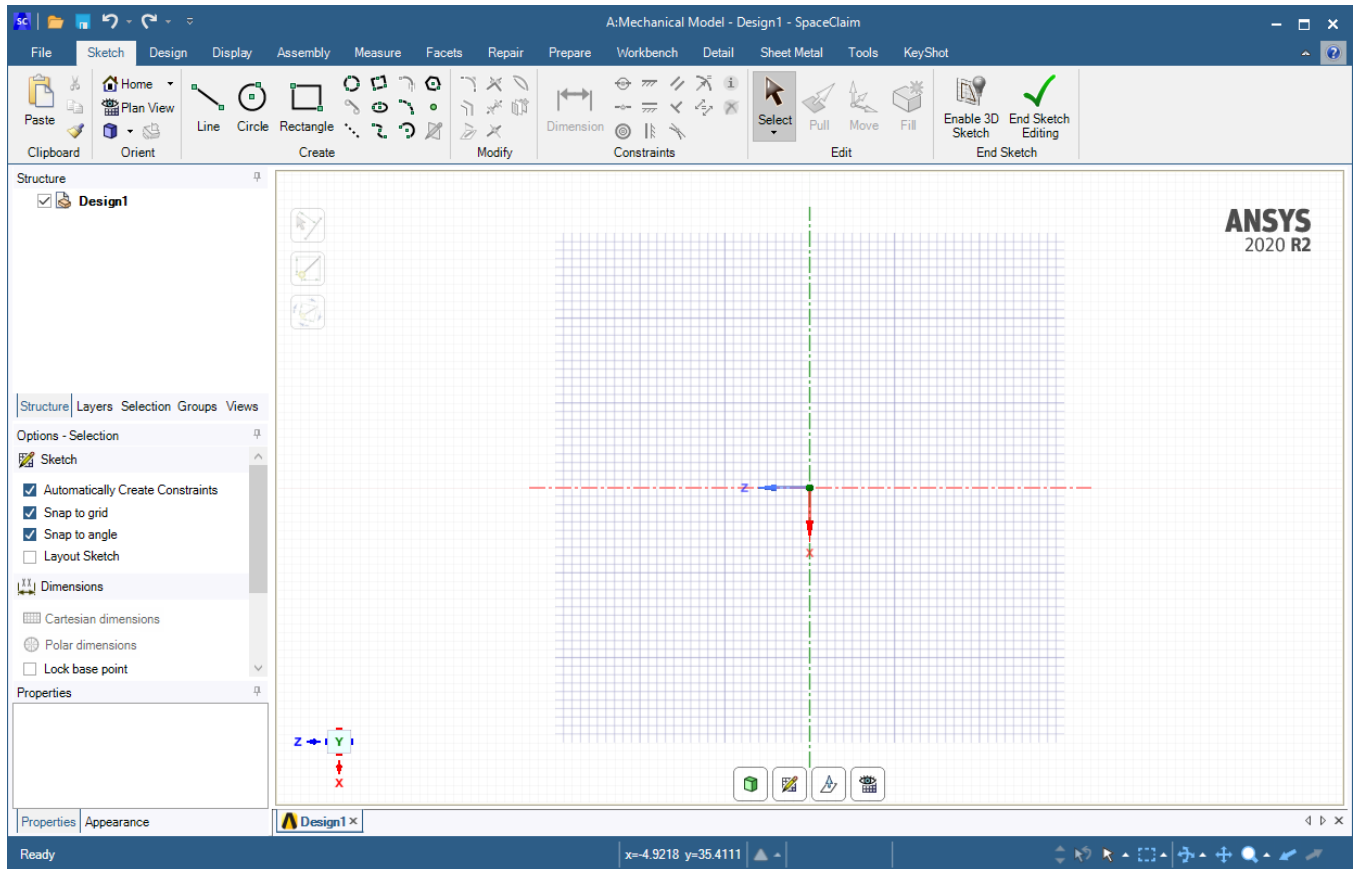
After “Double Clicking” on Geometry, a new window, as shown below, will pop up. The appearance of the new window will take a while.

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.”

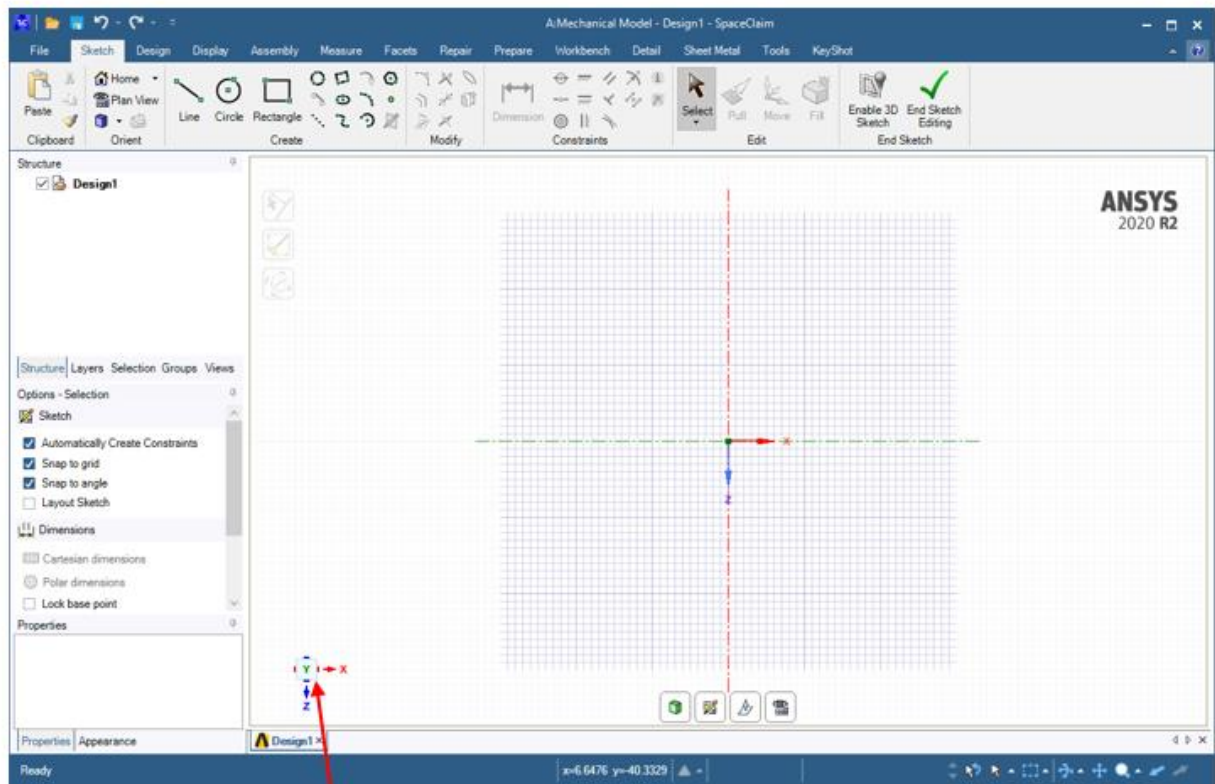




Click on this to change the coordinate to XZ as shown below.

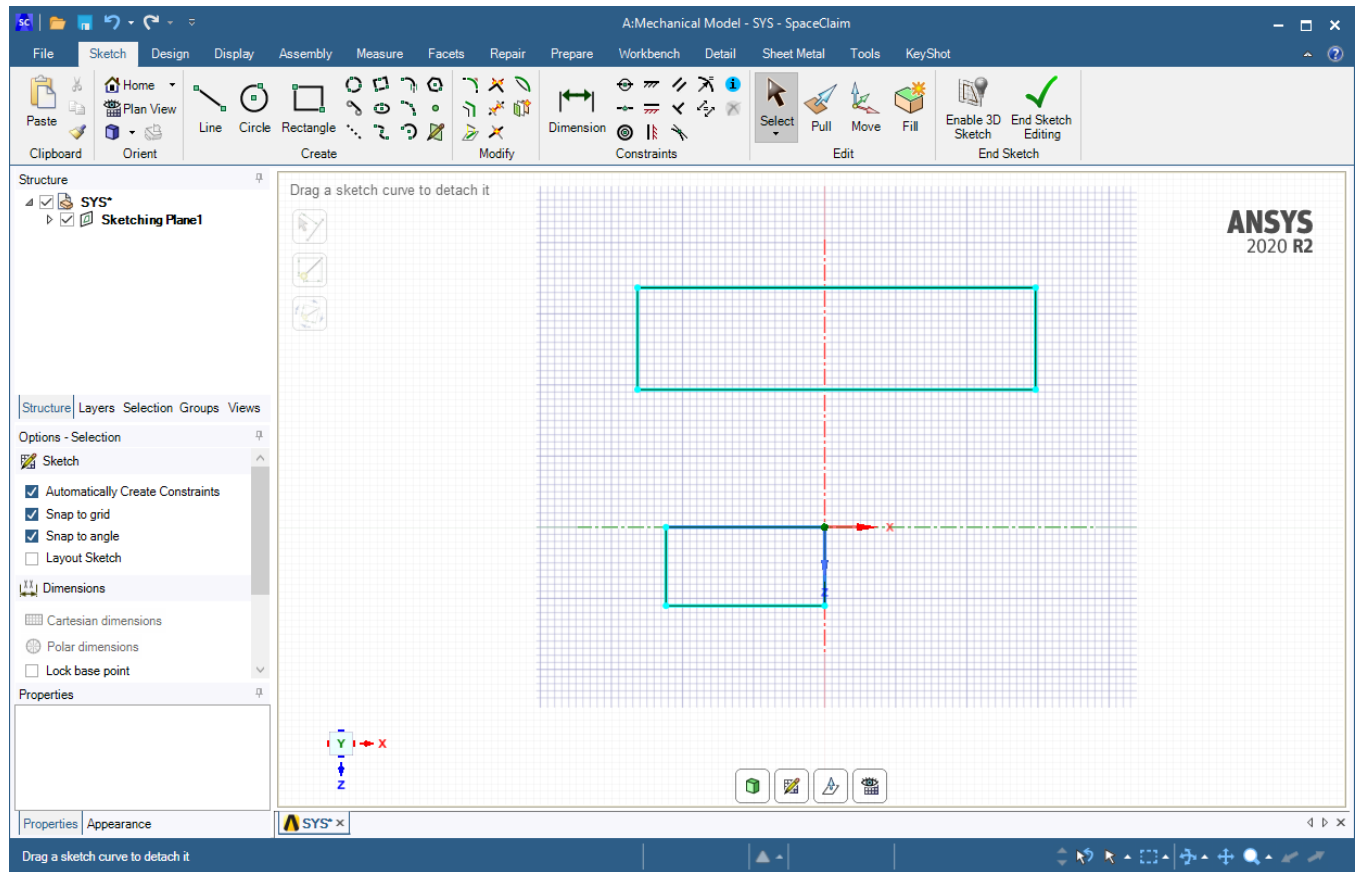


The wheel on the mouse can be used to zoom in and out.

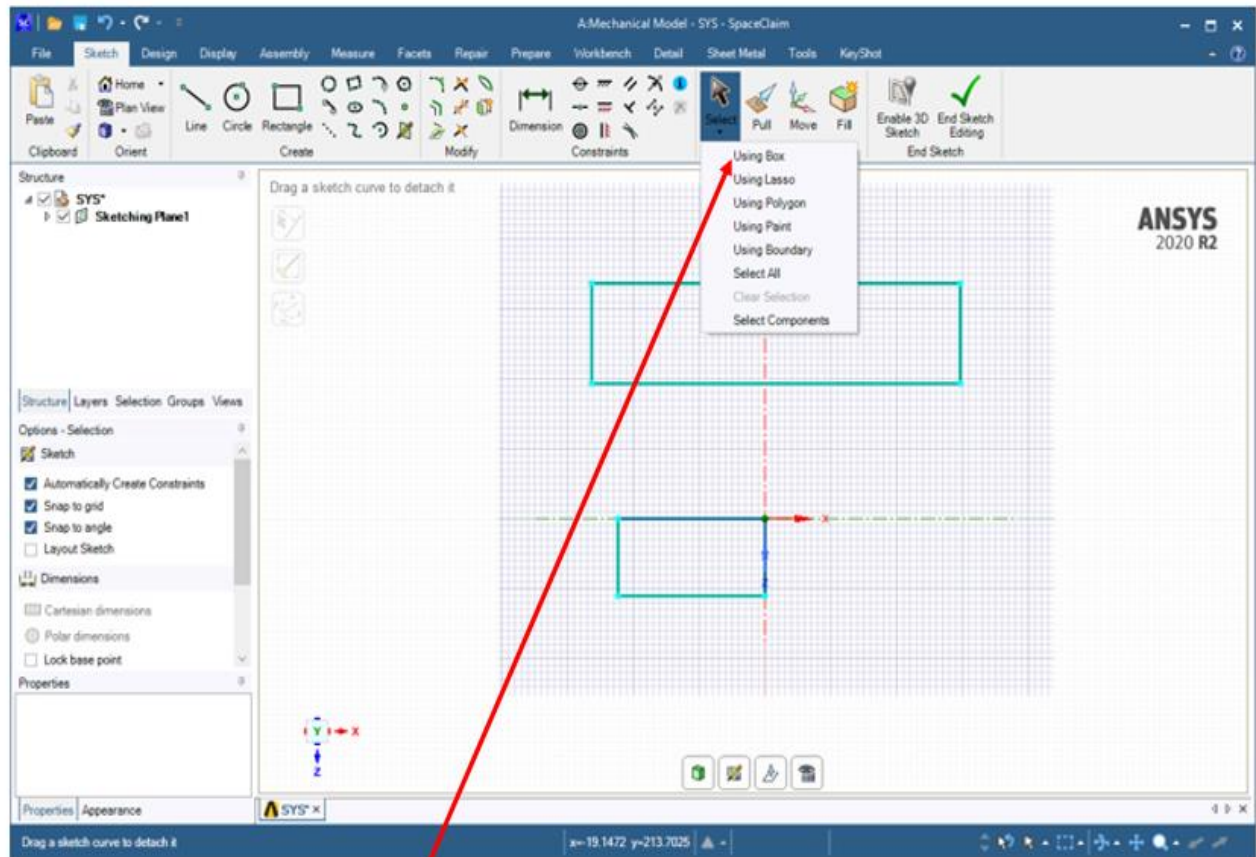


The sketching plane can be changed by clicking on this.

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.



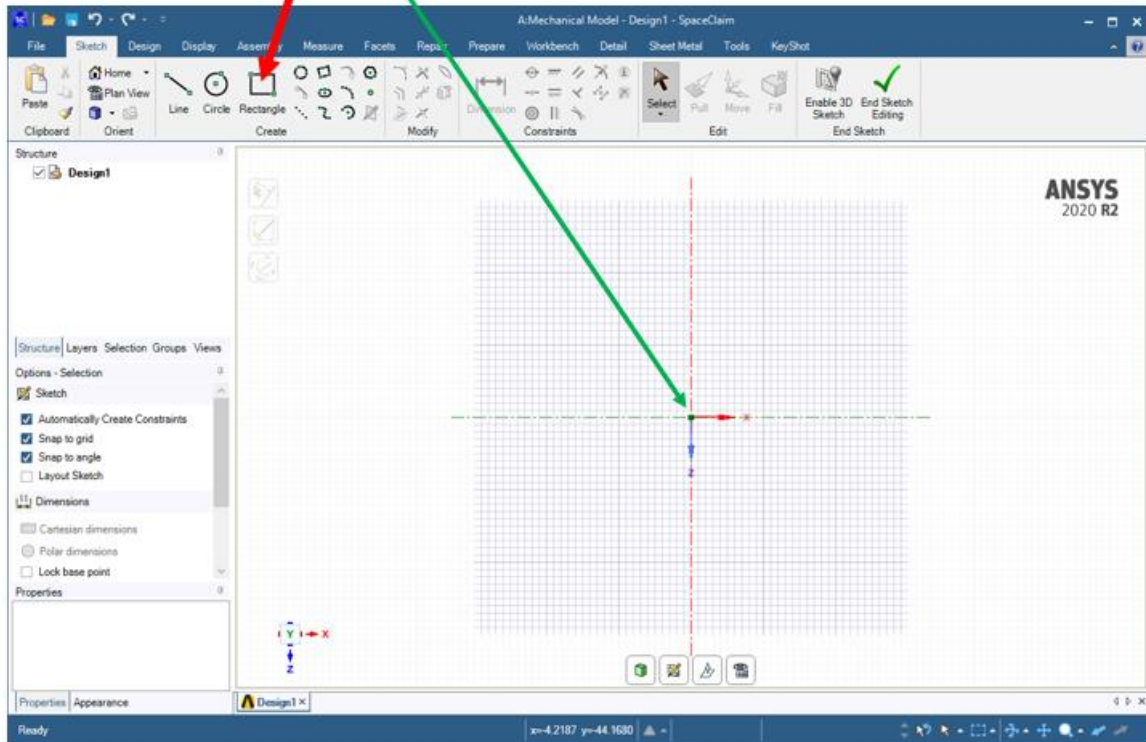
Click on “Select” in the “Edit” box, as shown below.



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.

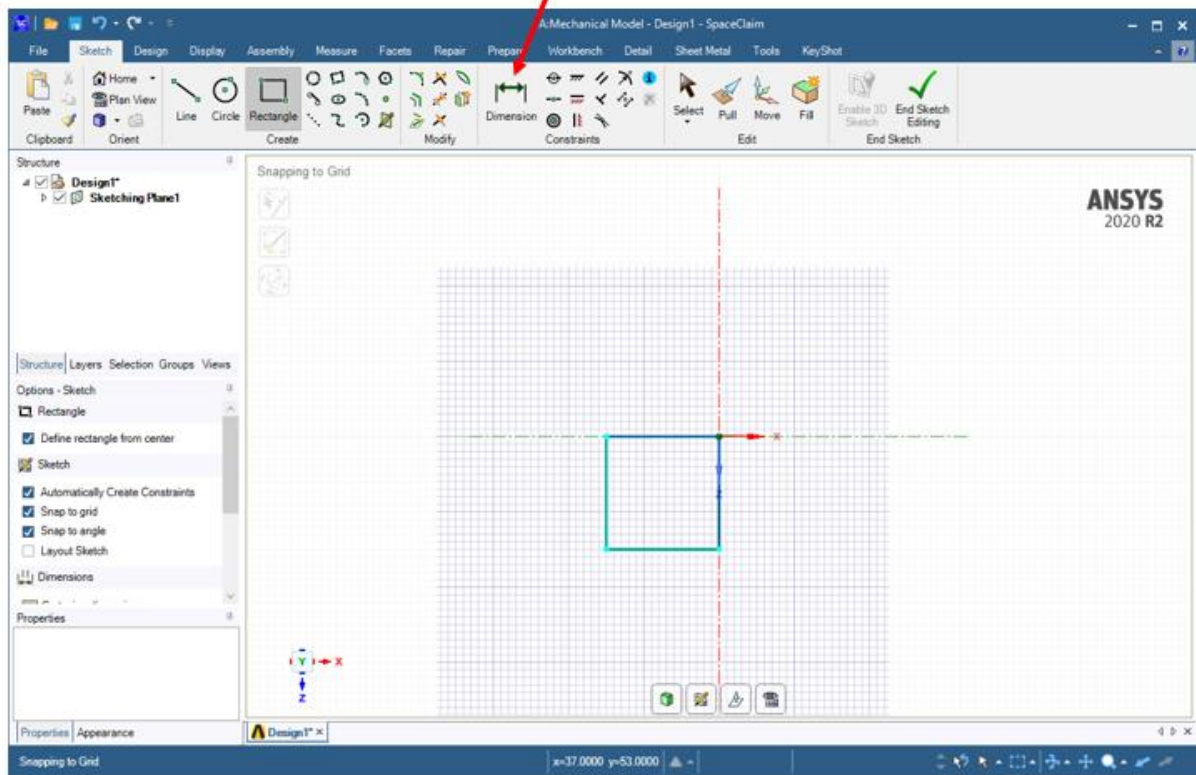
Create a rectangle by clicking on this. Then clicking here. Then dragging.



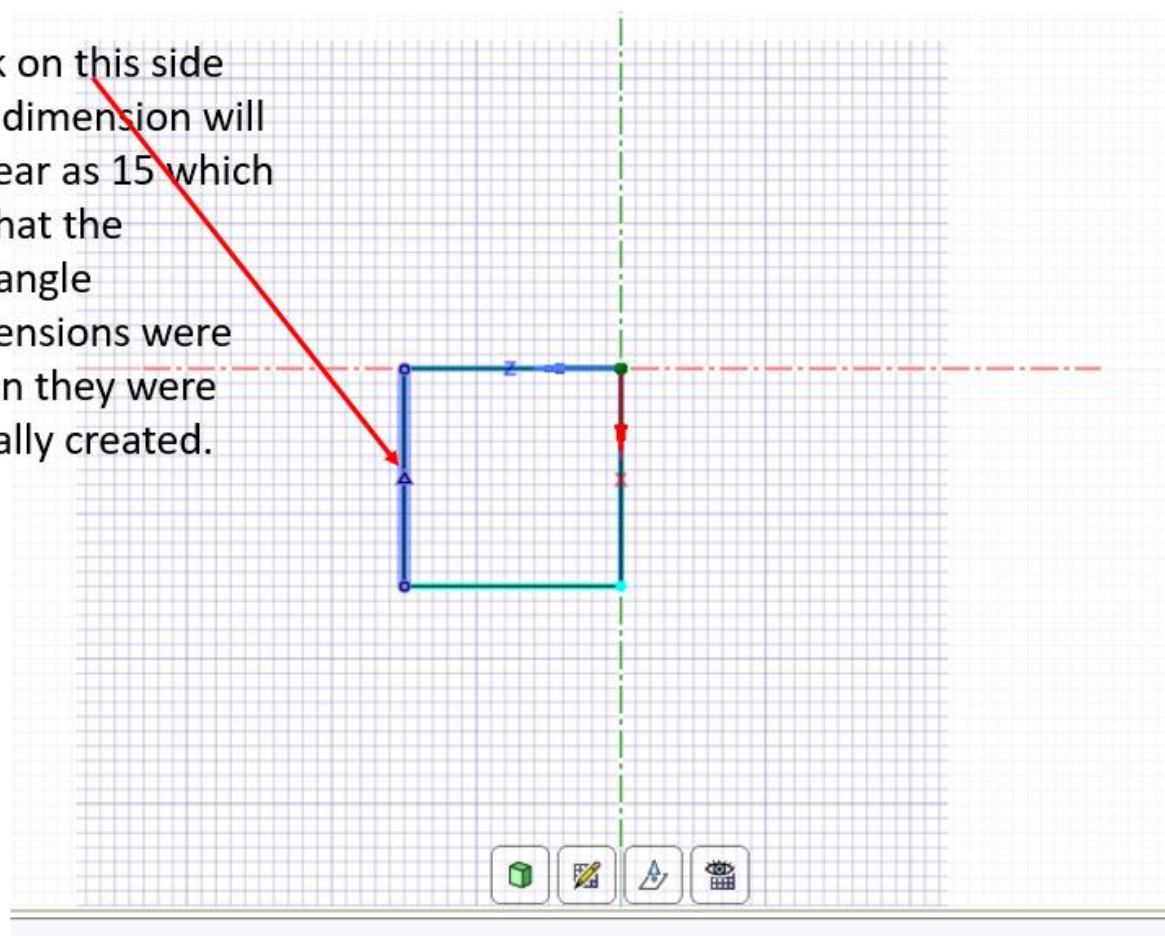
Once the rectangle is created, hit the “Esc” button.

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

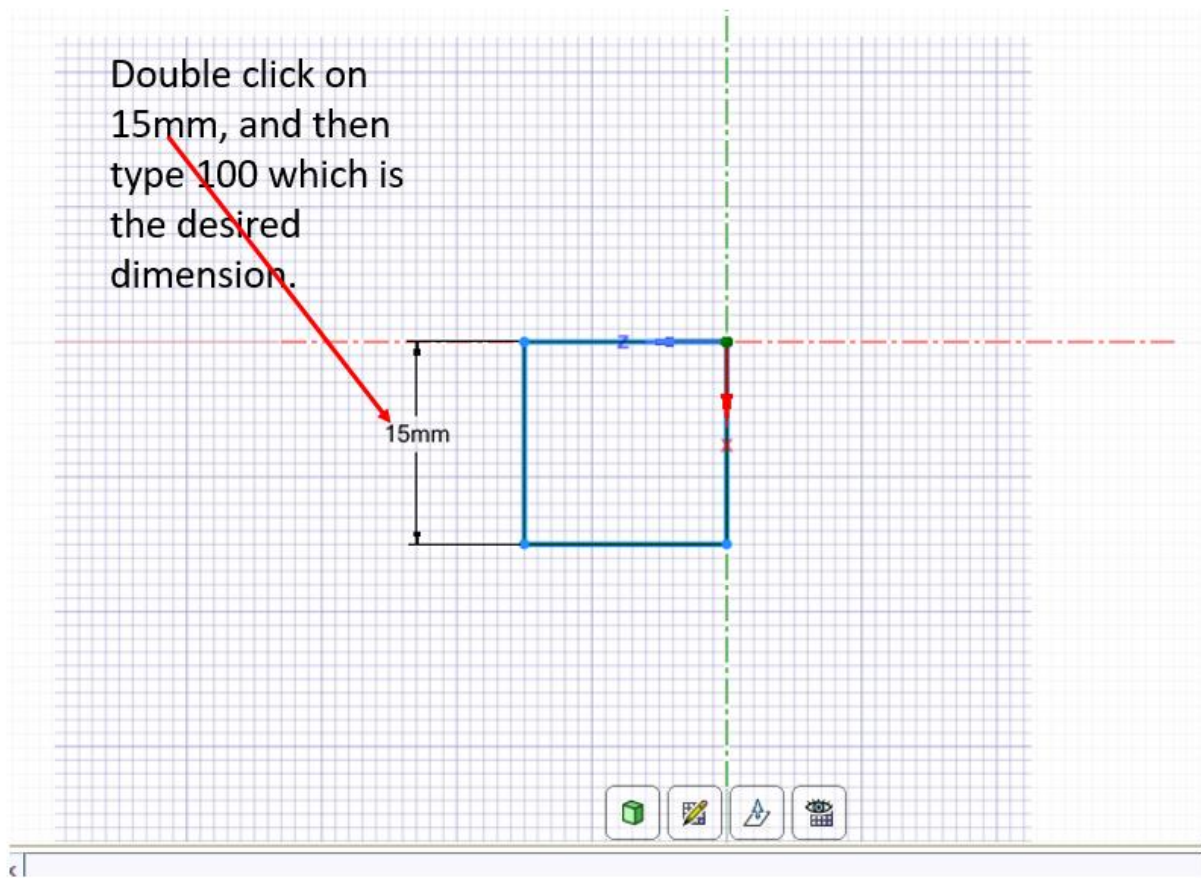
Click on “Dimension” to dimension the newly created rectangle.



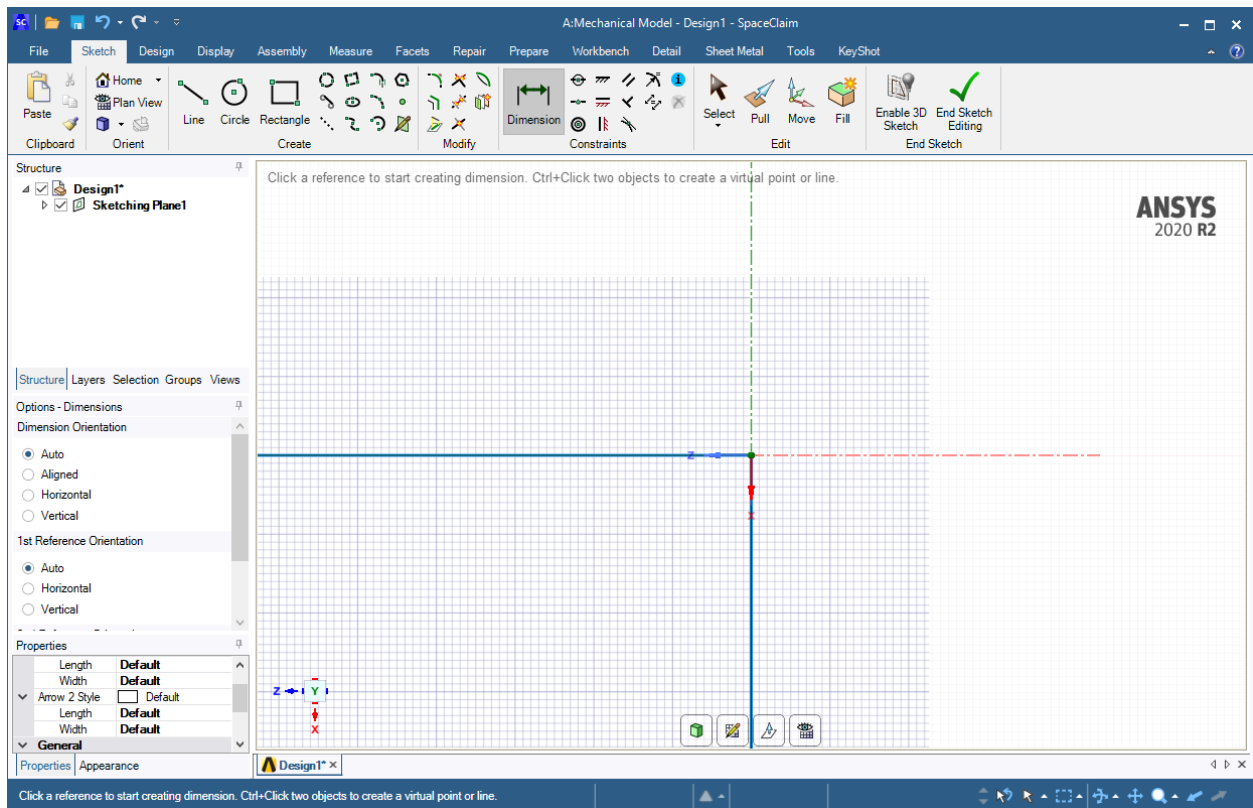
Click on this side
and dimension will
appear as 15 which
is what the
rectangle
dimensions were
when they were
initially created.



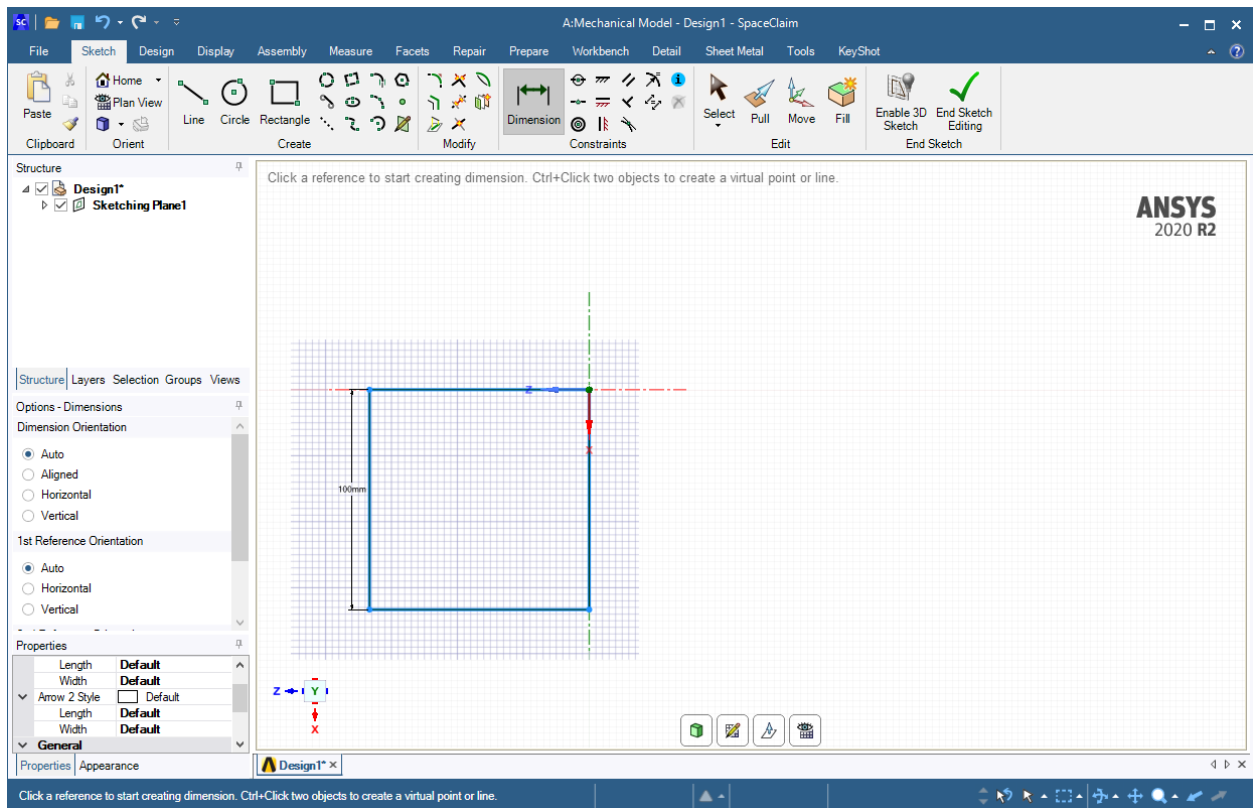
The screen will appear as shown below.



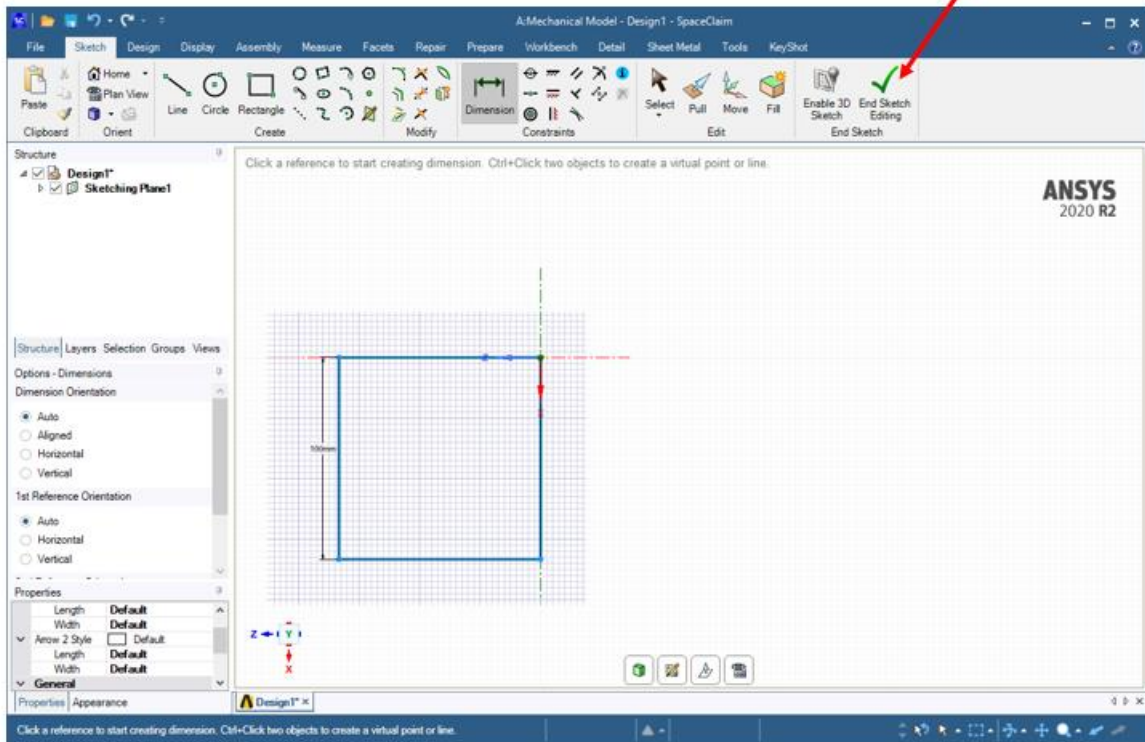
After changing the 15 to 100, the rectangle won't fit, and the screen appears as shown below.



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

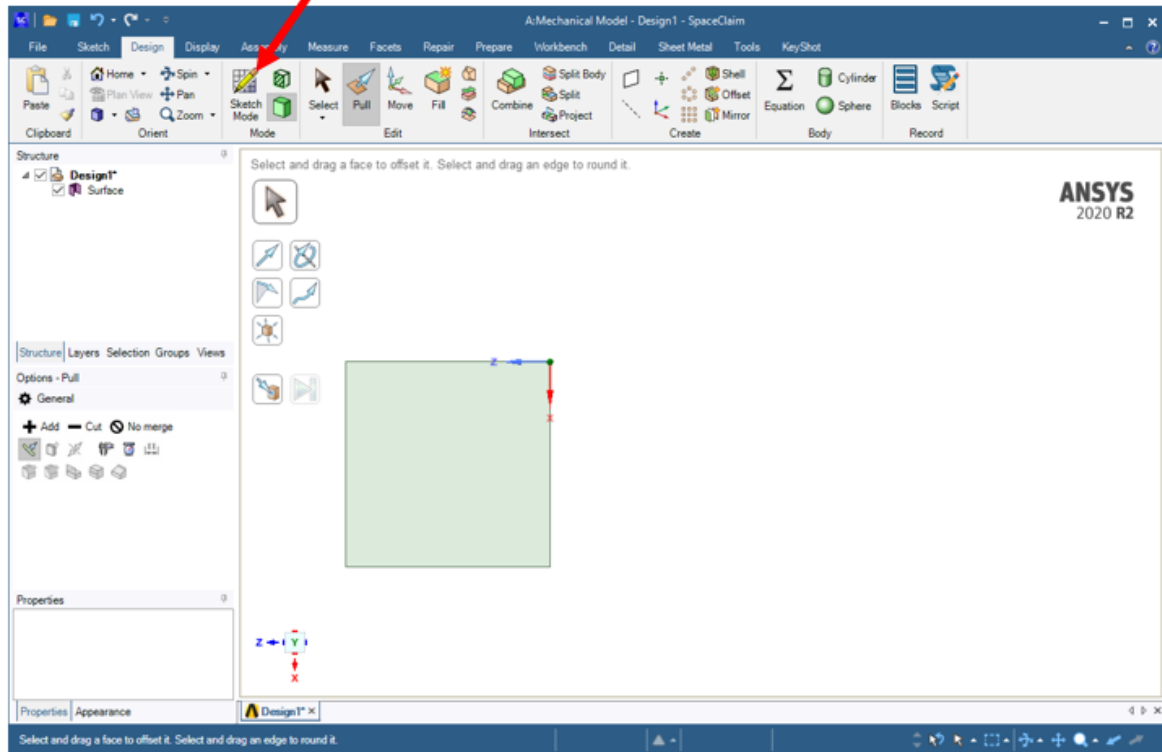


Click on “End Sketch Editing”

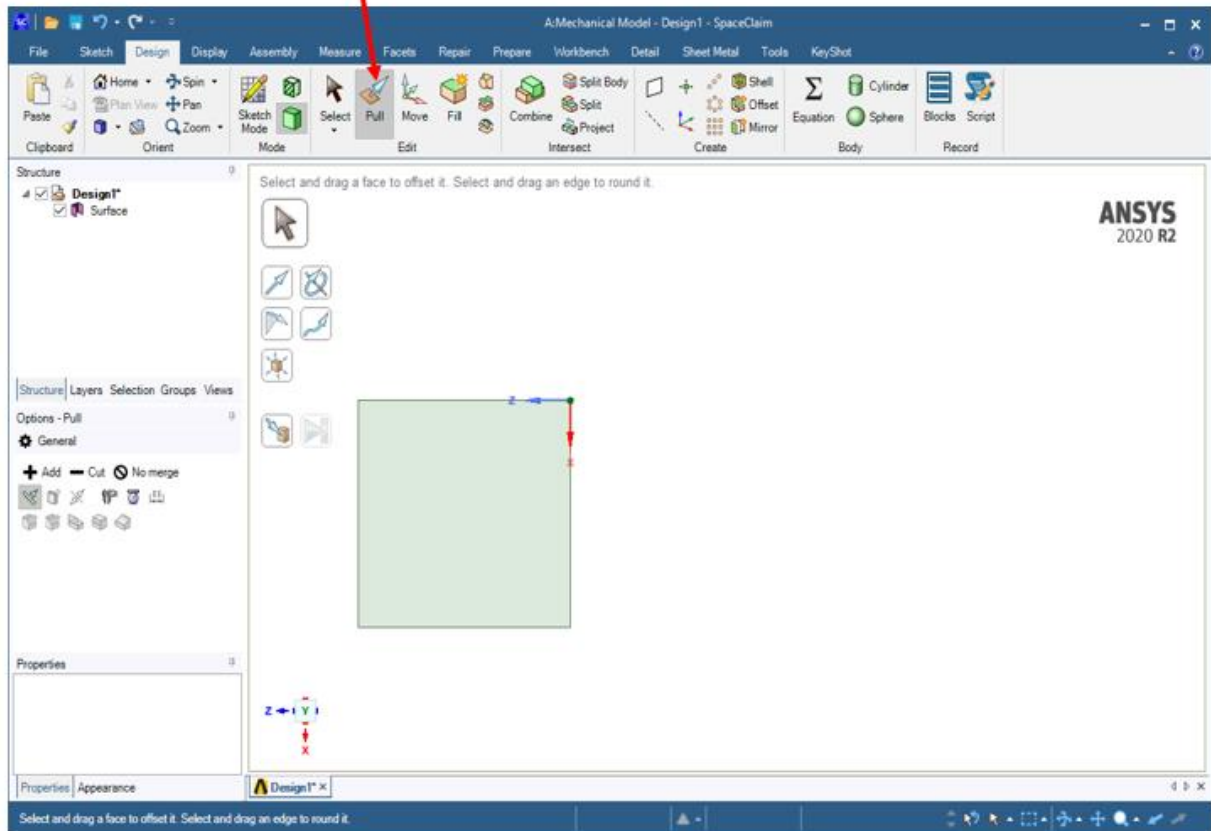


Clicking on “End Sketch Editing” will bring up the following screen, as shown below.

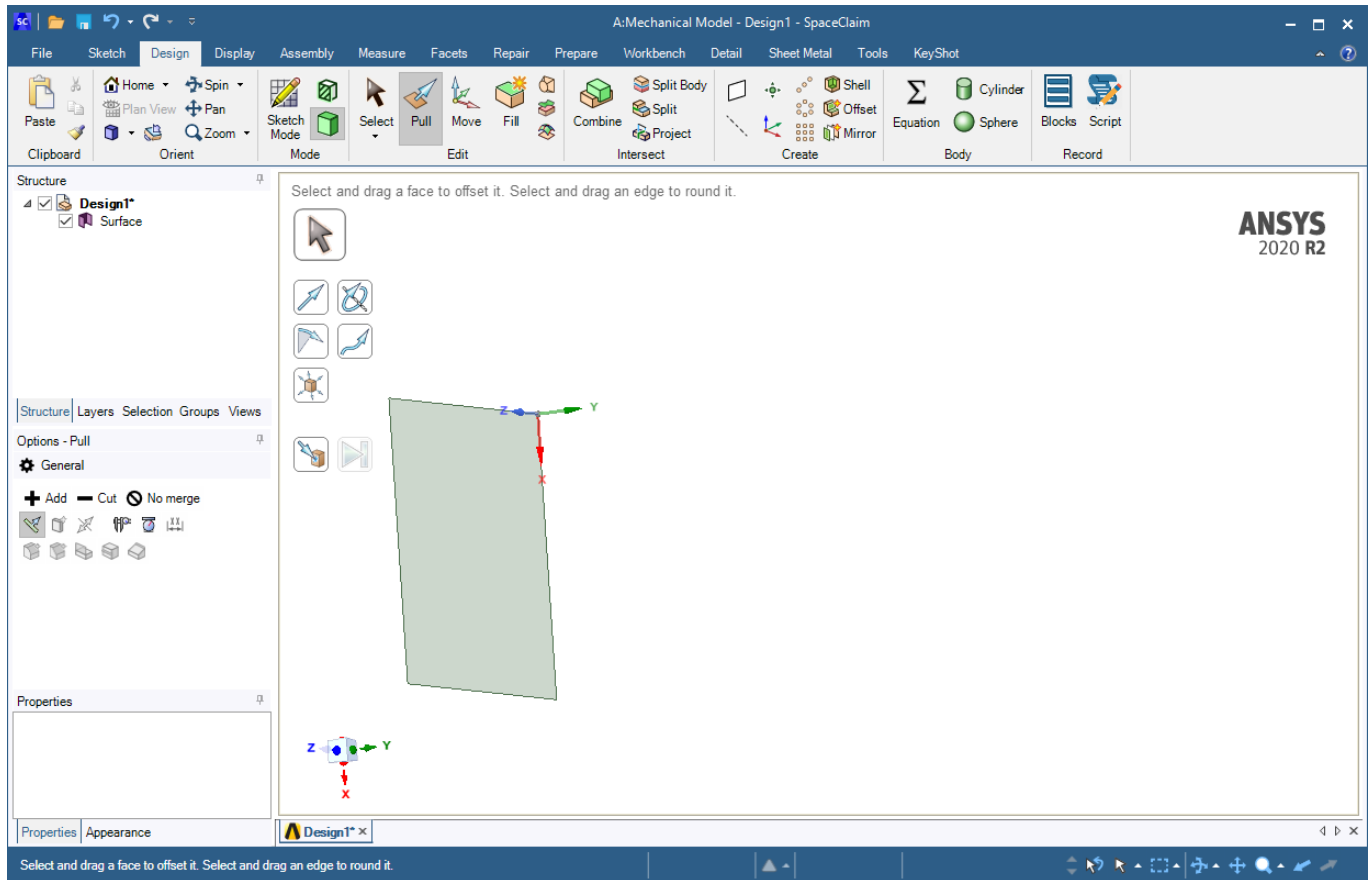
Clicking on “Sketch Mode” will bring up the previous screen. Do this only if necessary.



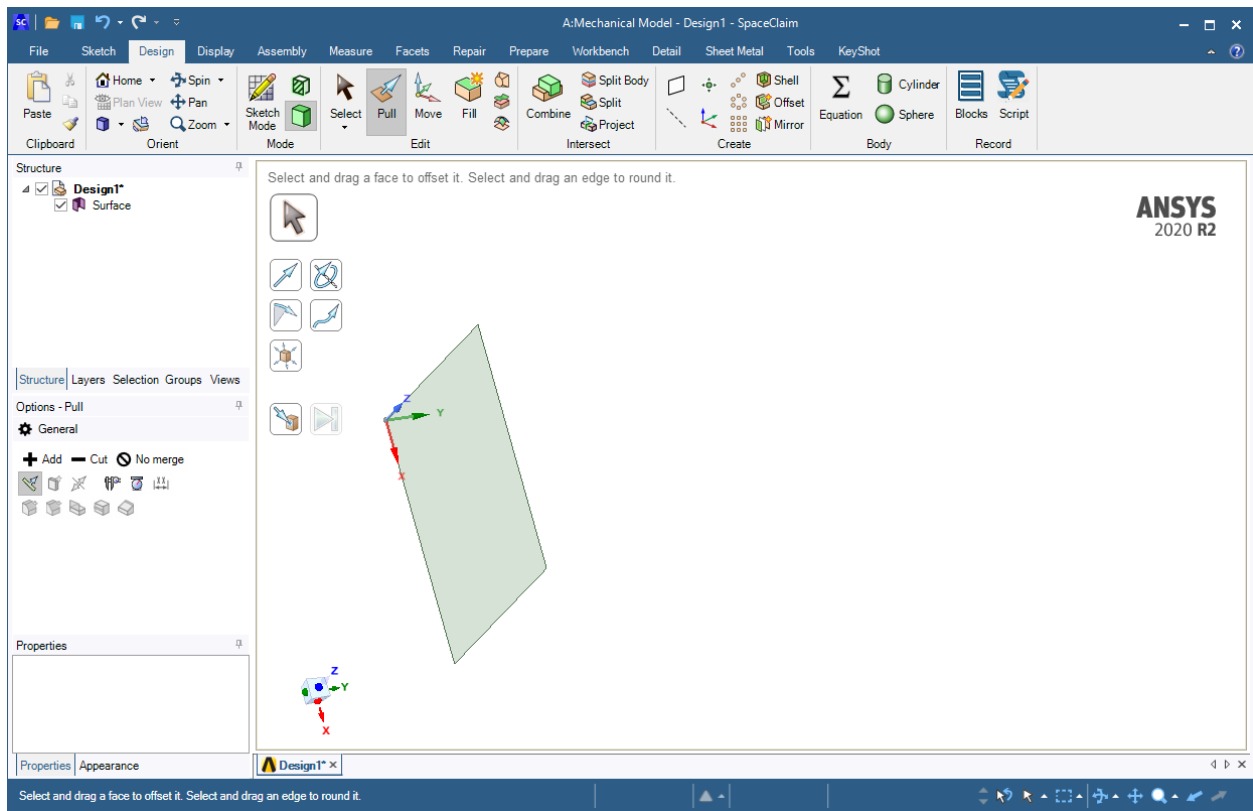
At this point, the
“SC” screen is in
“Pull” tab.

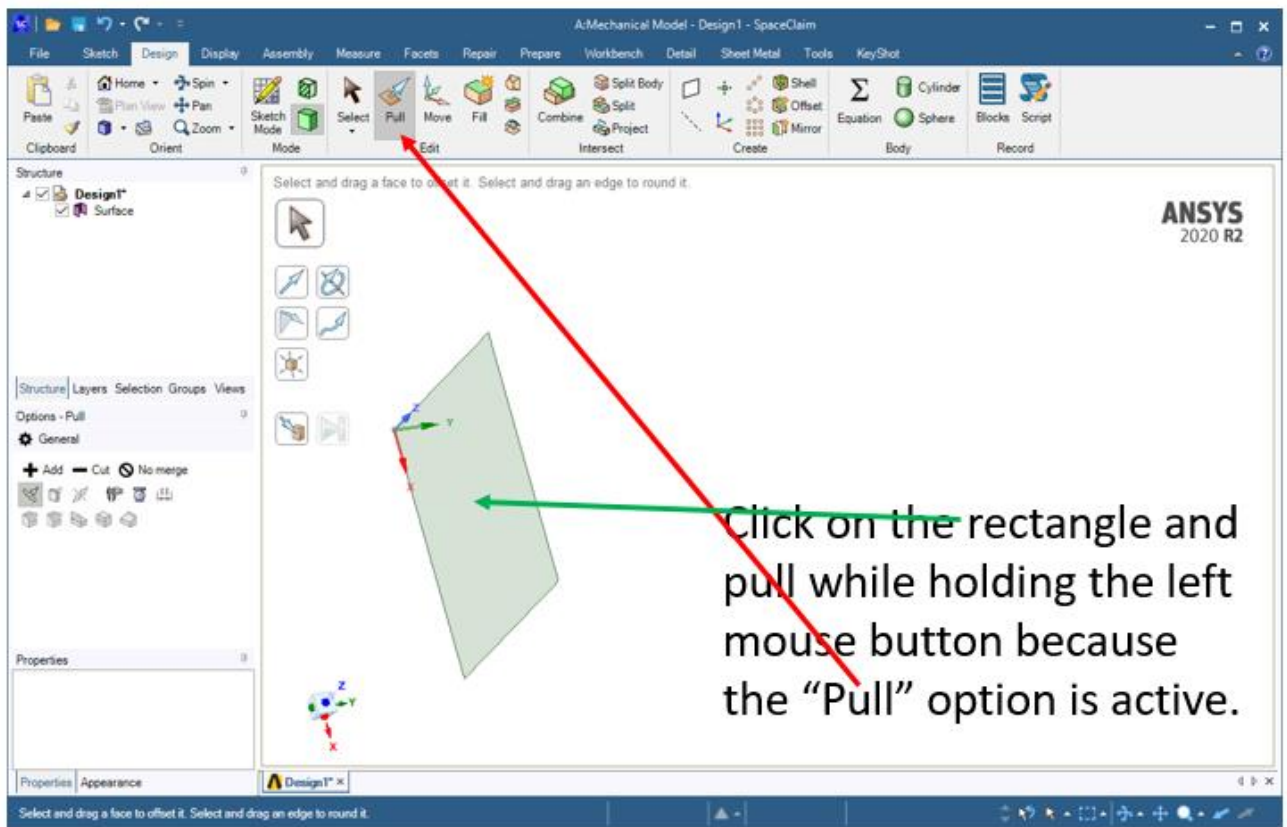


Clicking on the mouse wheel and moving the mouse pointer on the screen will rotate the rectangle, as shown below. An example is shown below.



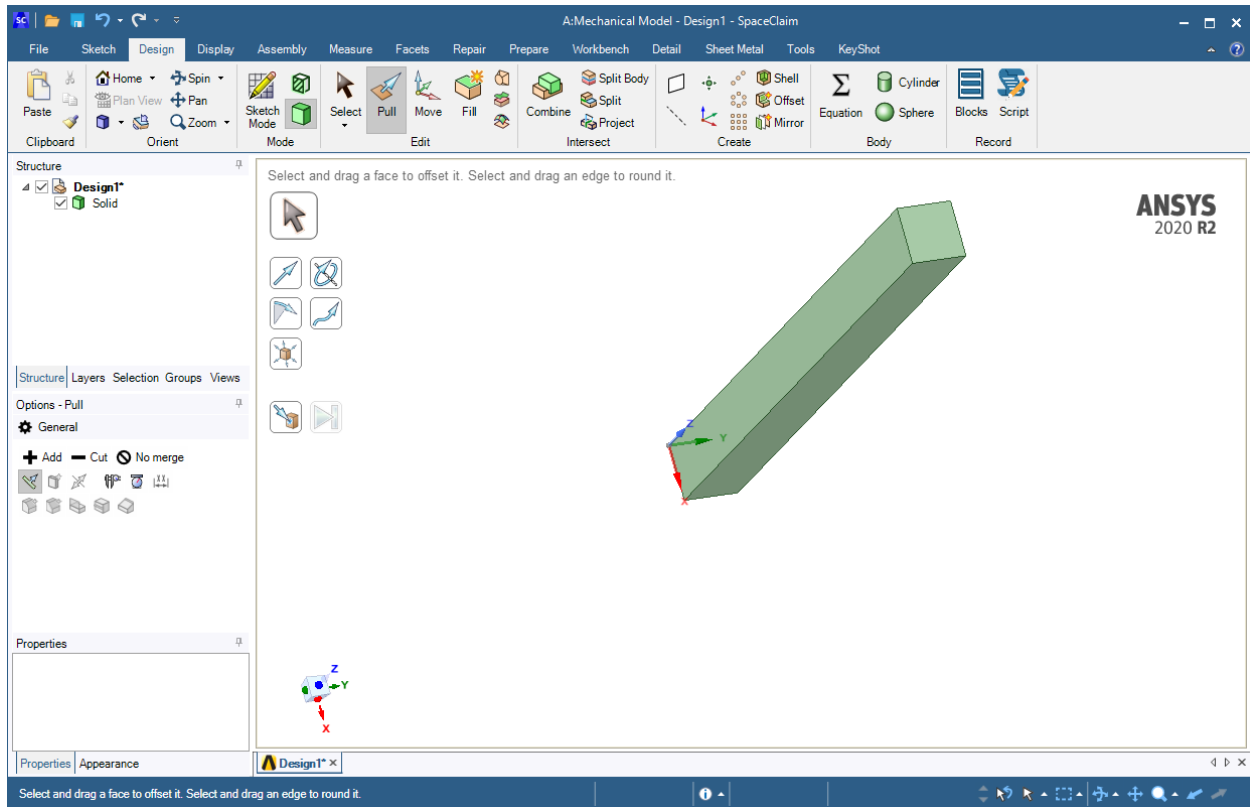
Rearrange the screen as shown below.





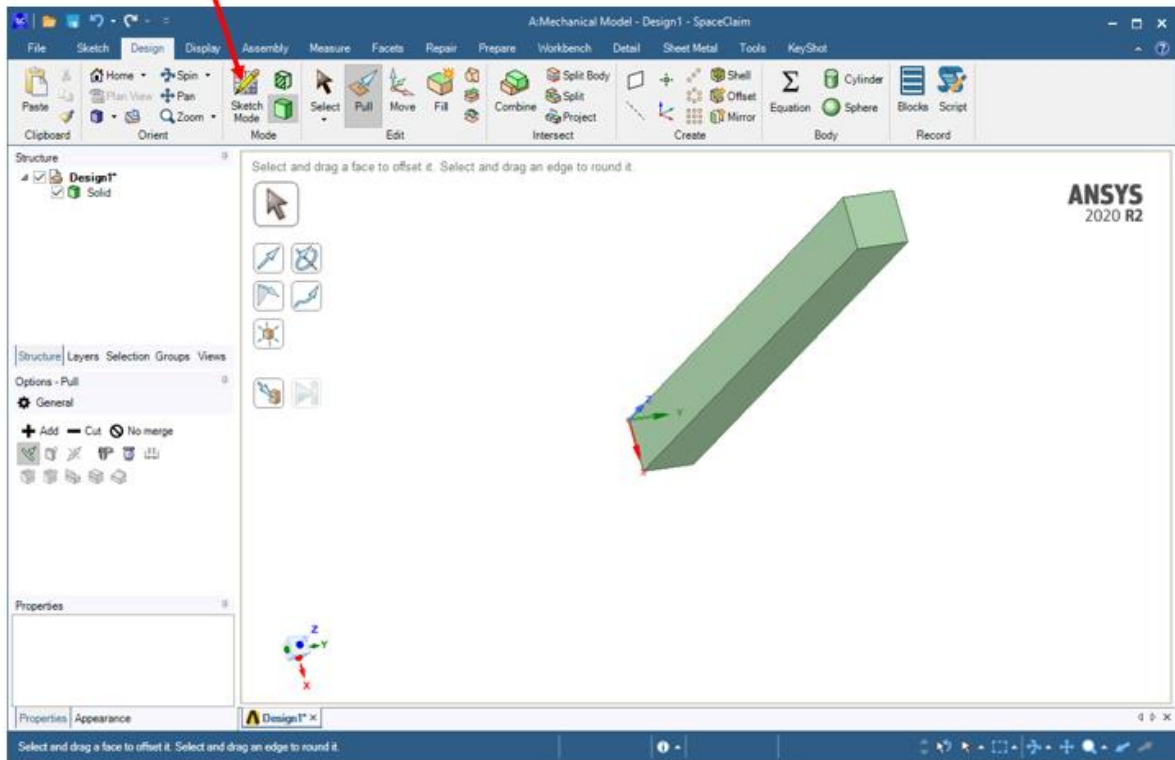
As it is pulled while holding the "left mouse button," type 1000, which is the length. The following appears.

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

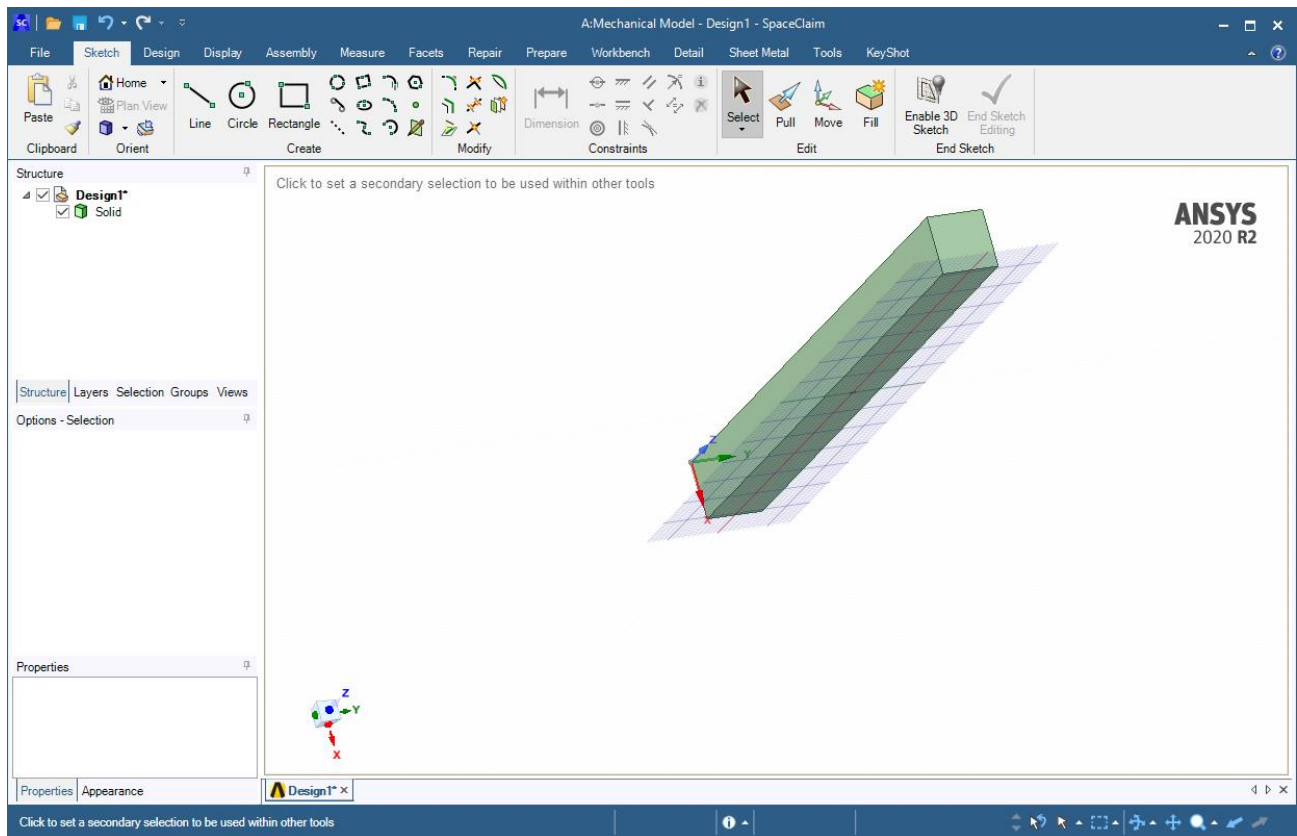


A section must be defined for the monkey to sit on.

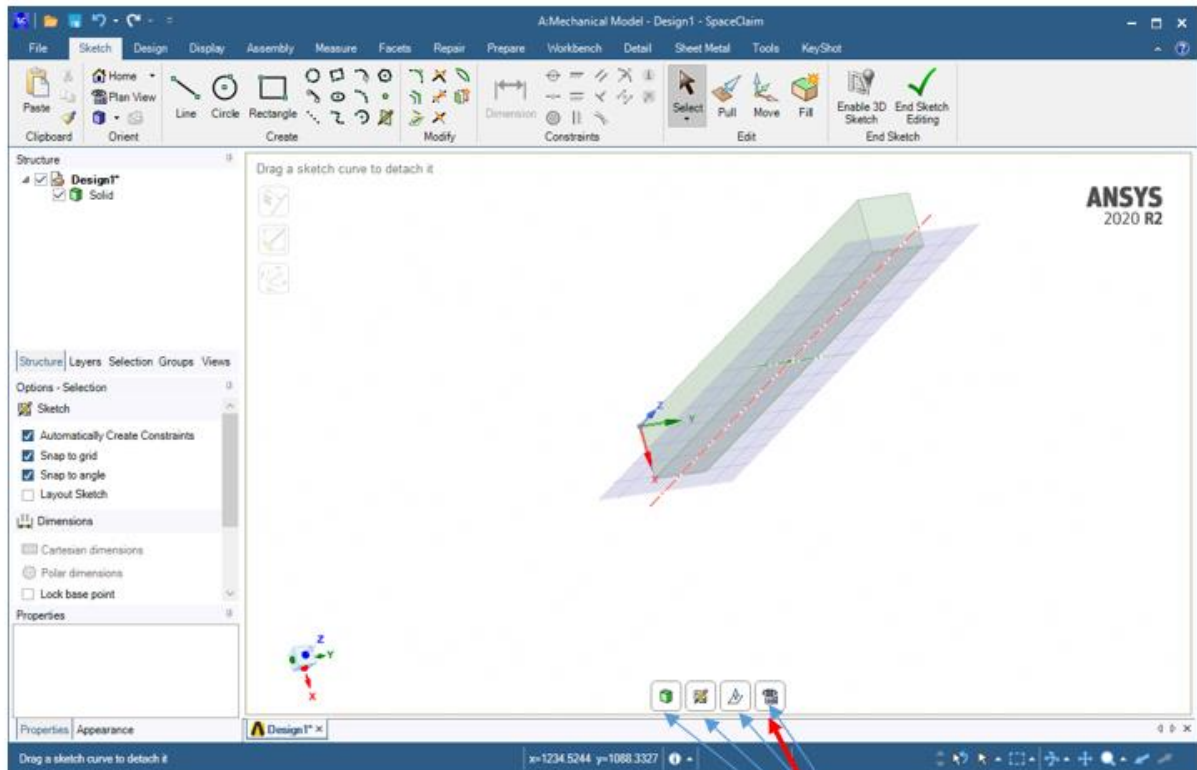
Click on “Sketch
Mode” to go to
sketching screen.



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

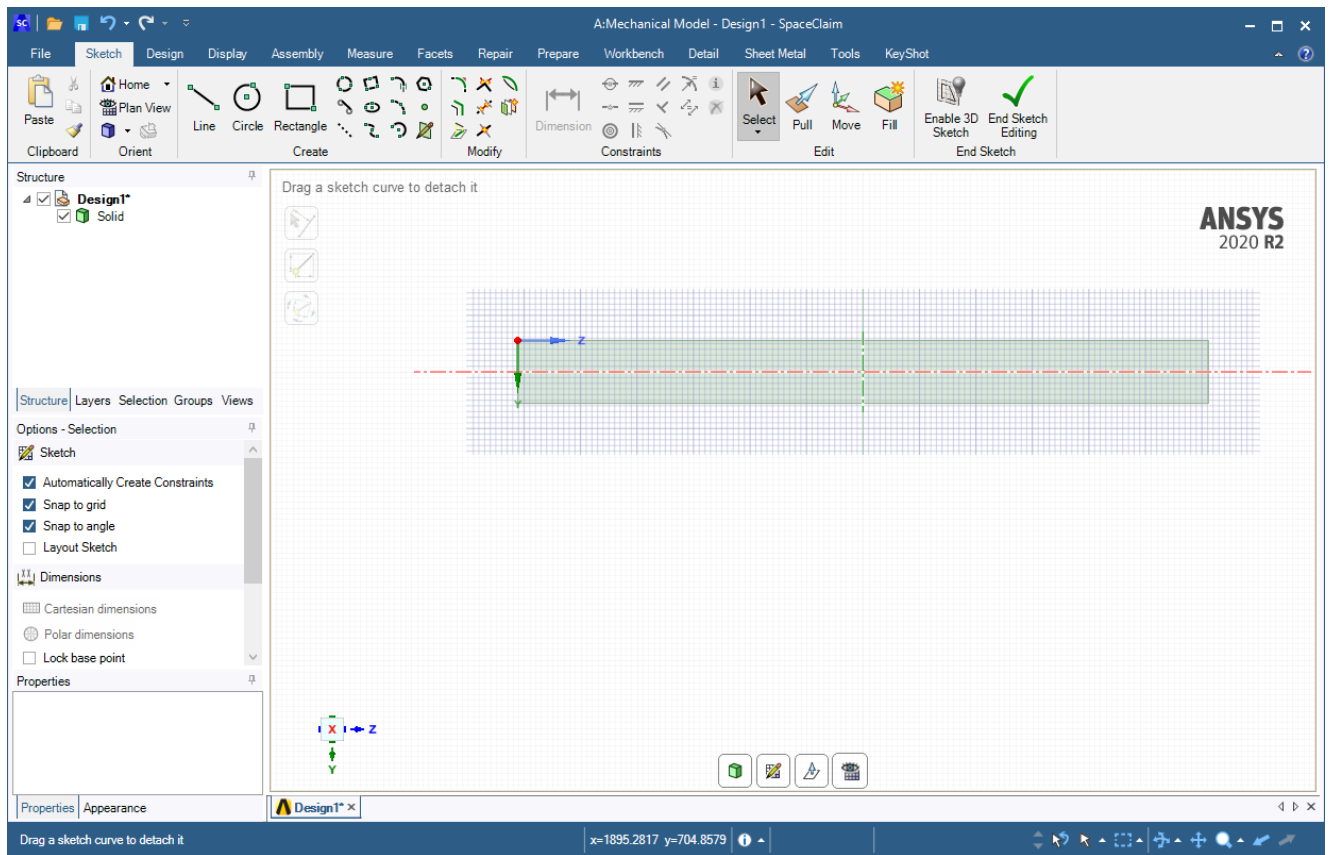


Select a face by putting the mouse pointer on it and clicking on it.

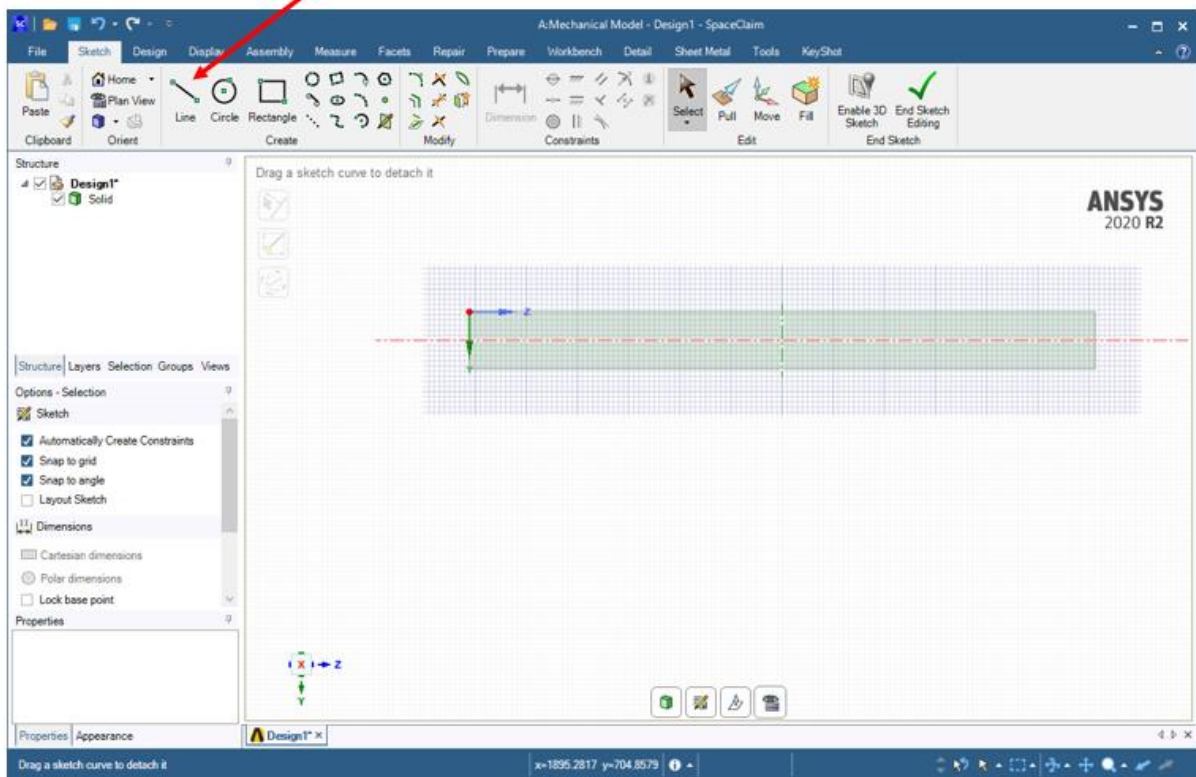


Once the face is selected, these buttons appear. Click on this button to get a top view of the selected face.

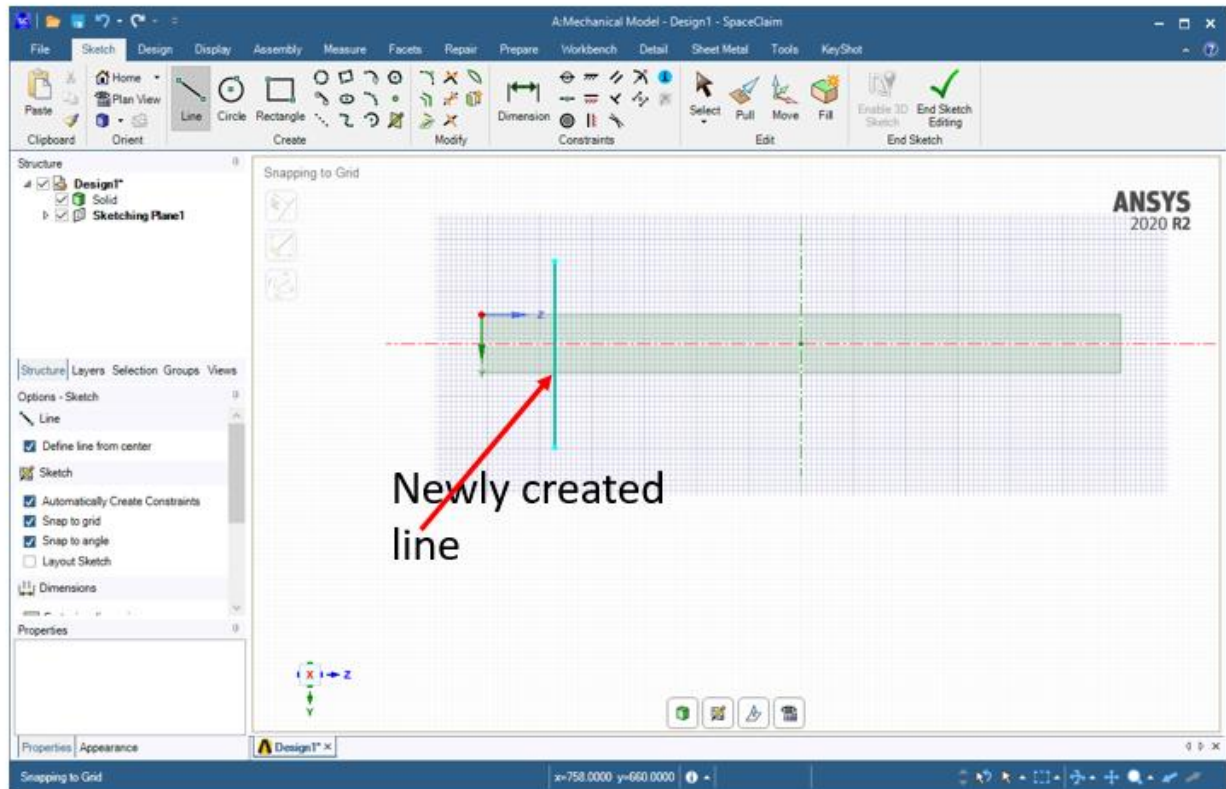
The “SC” screen appears as shown below.



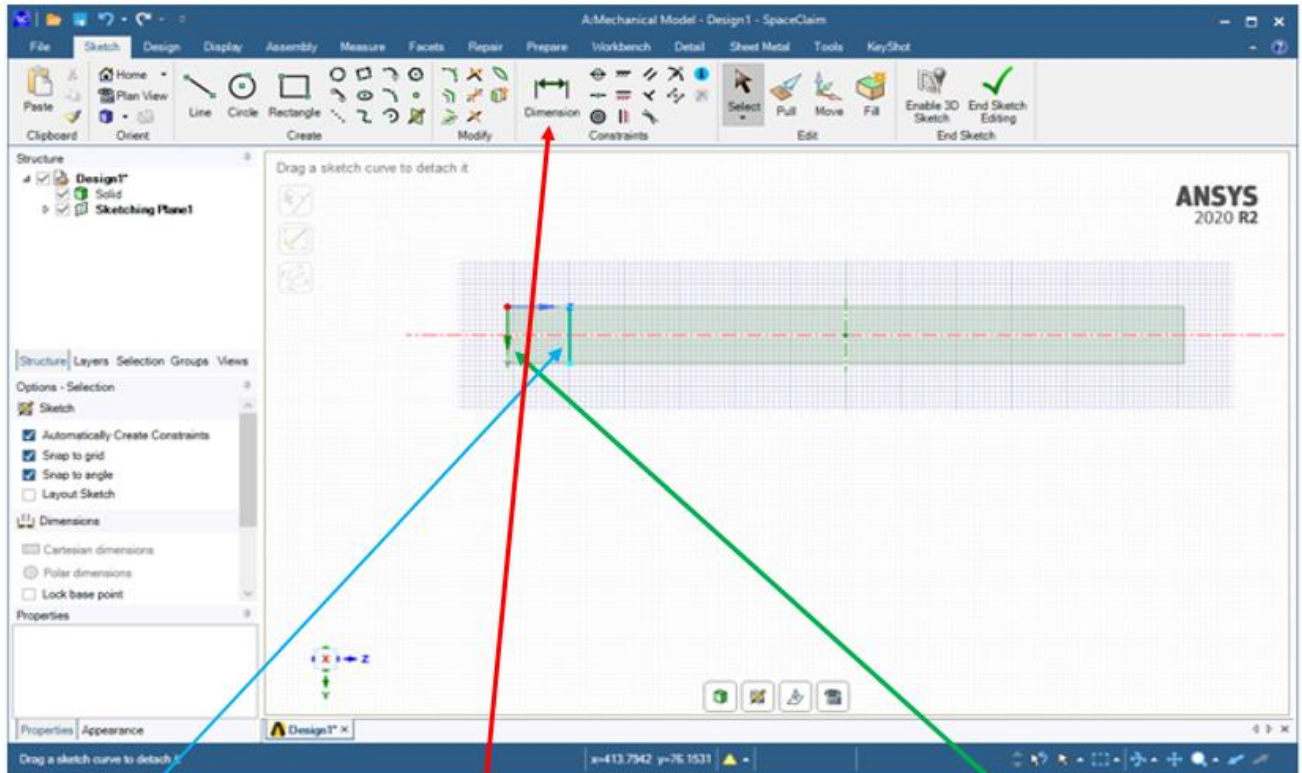
Click on "Line"



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

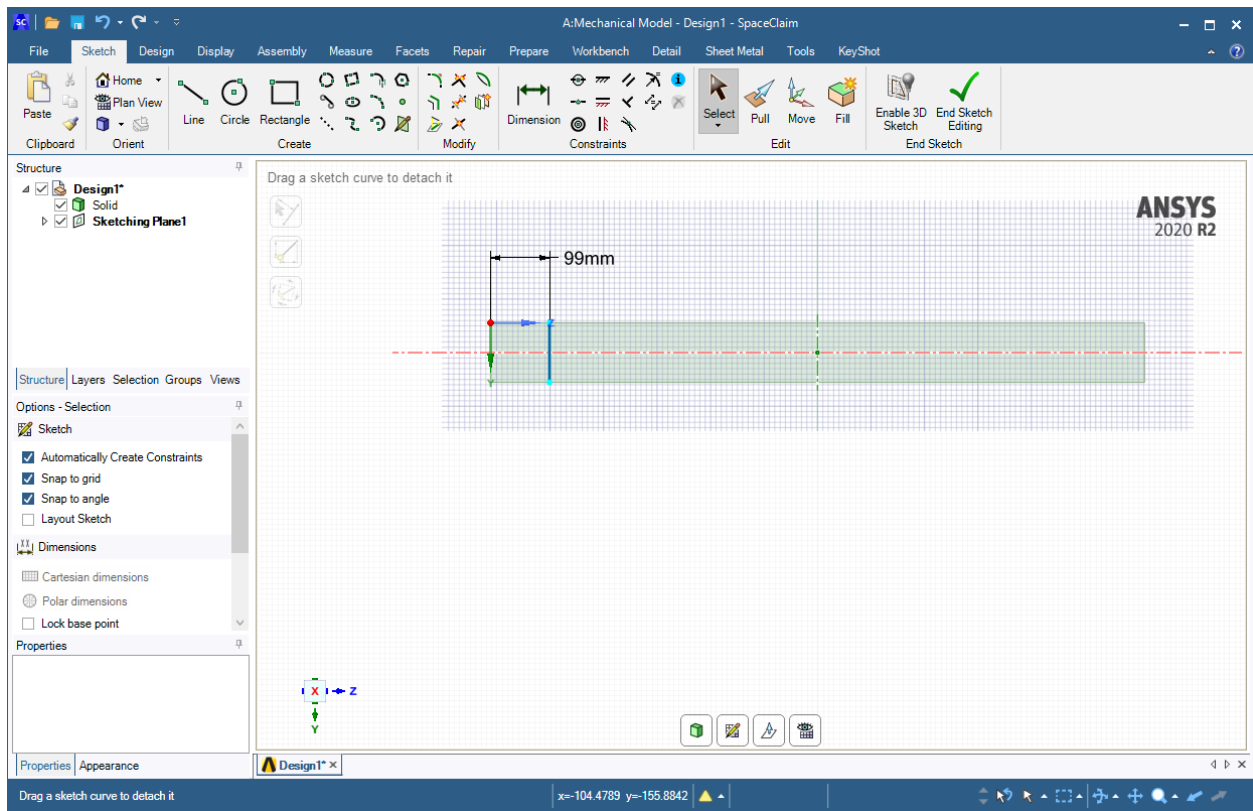


The above line is too long. Erase it and redraw it as shown below.

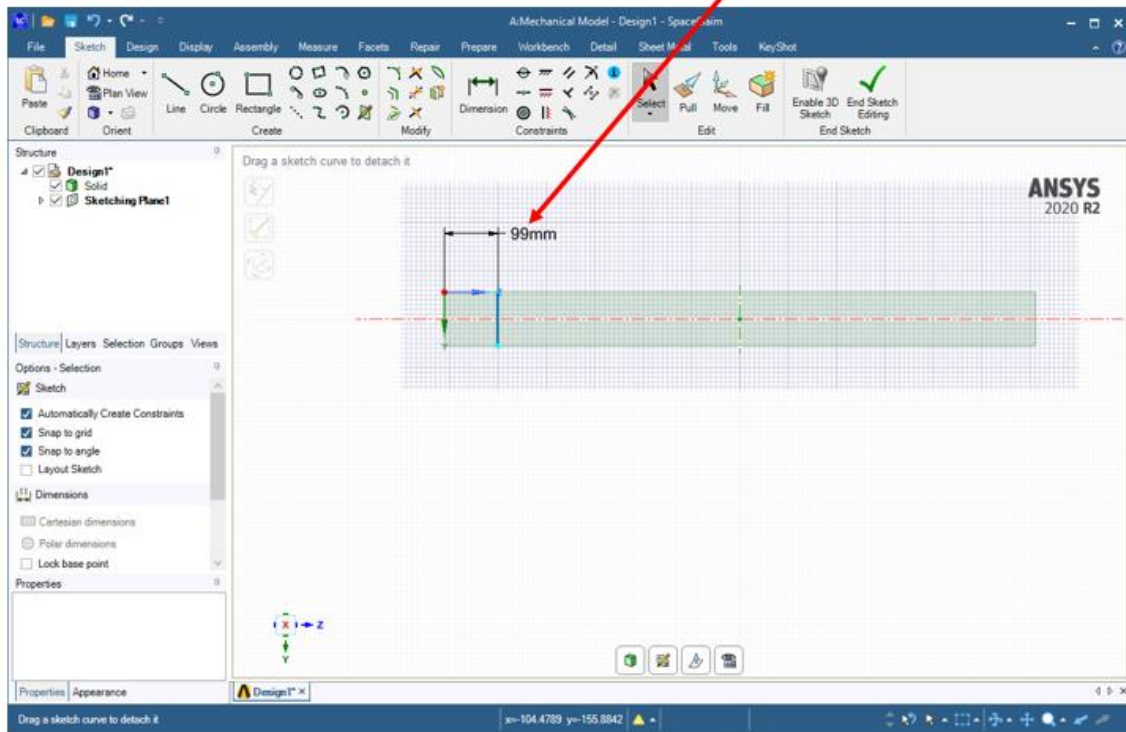


Create a "Dimension from the surface to the left to the newly drawn line. This is done by using the "Dimension" option at the top of the screen. It is tricky to dimension. Start from the left edge and drag to line after having invoked the "Dimension" command.

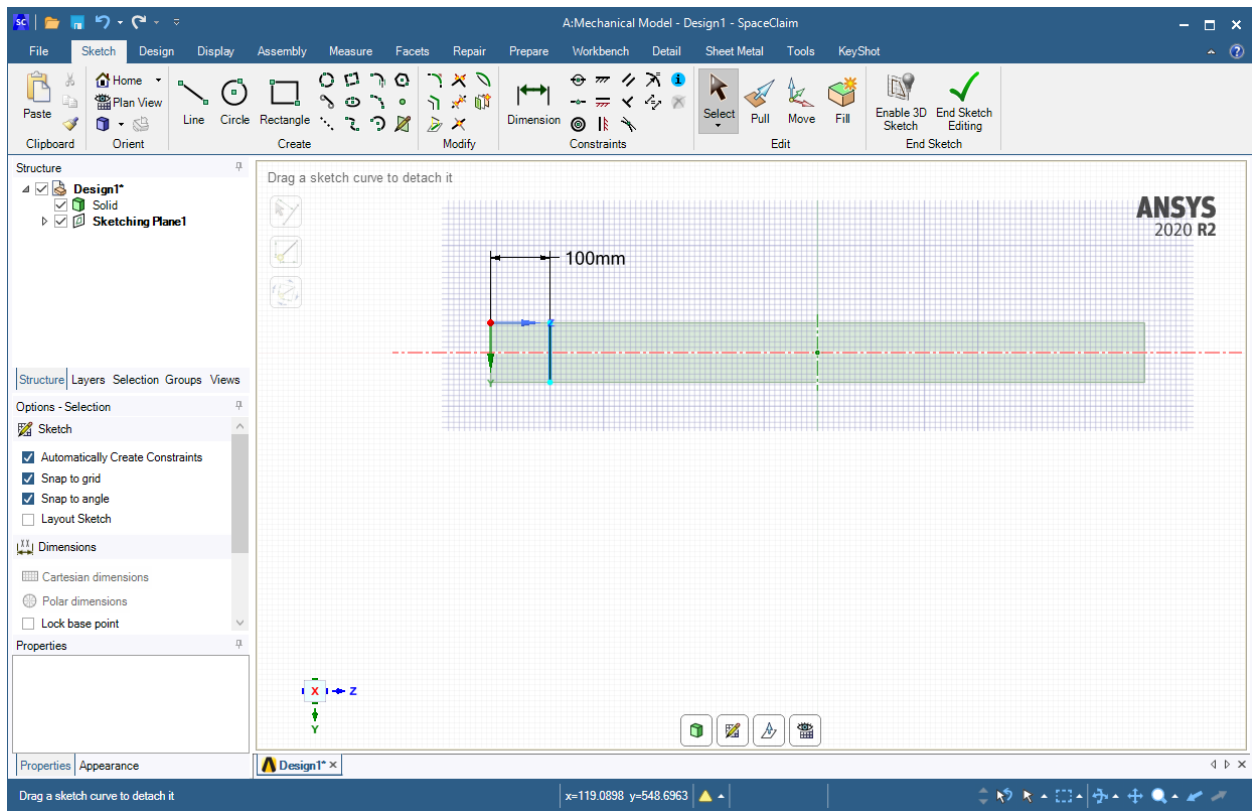
After dimensioning, the sketch appears as shown below.



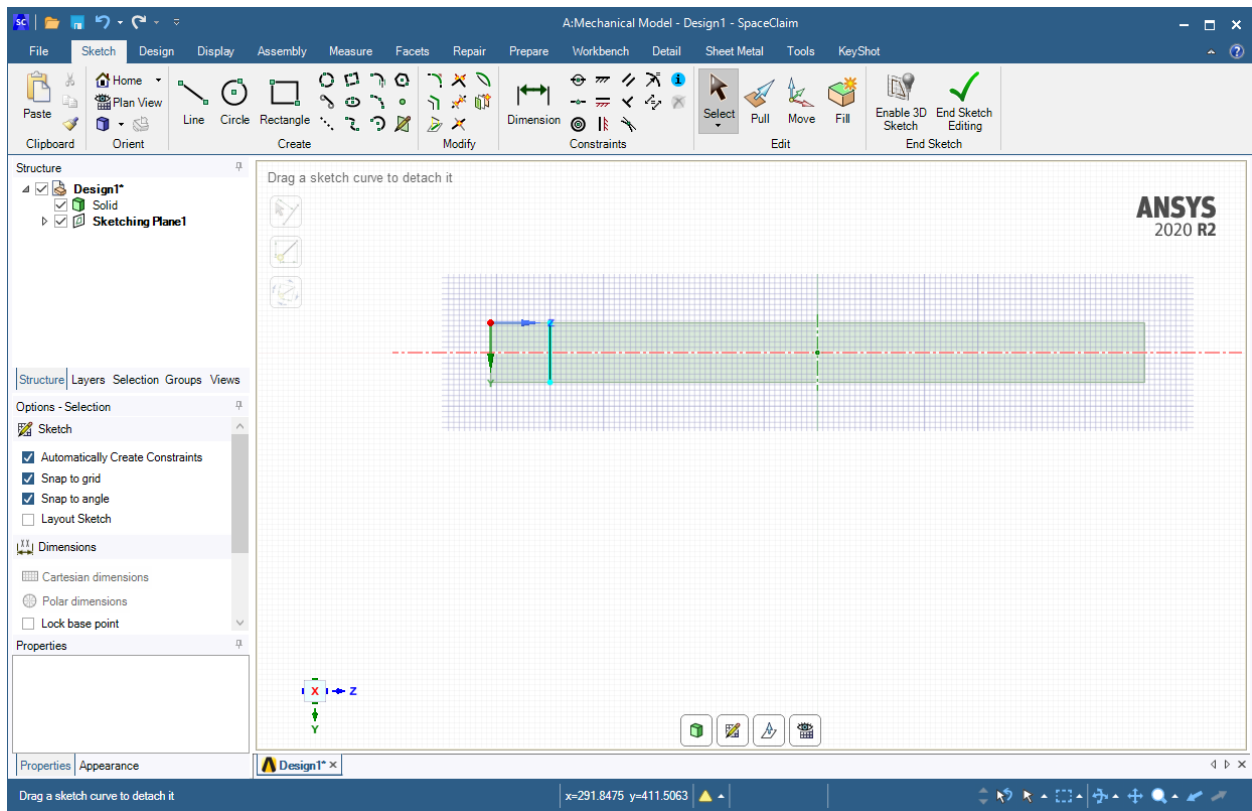
Double click and
change to "100"



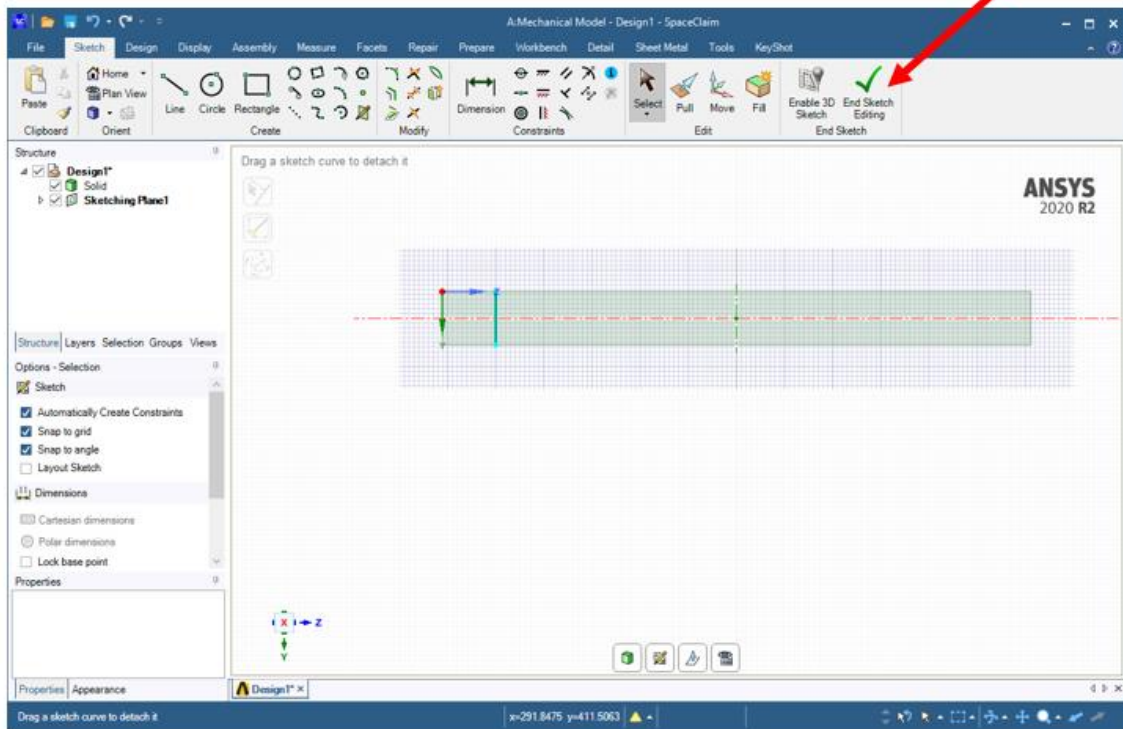
After changing the “Dimension,” the plot appears as shown below.



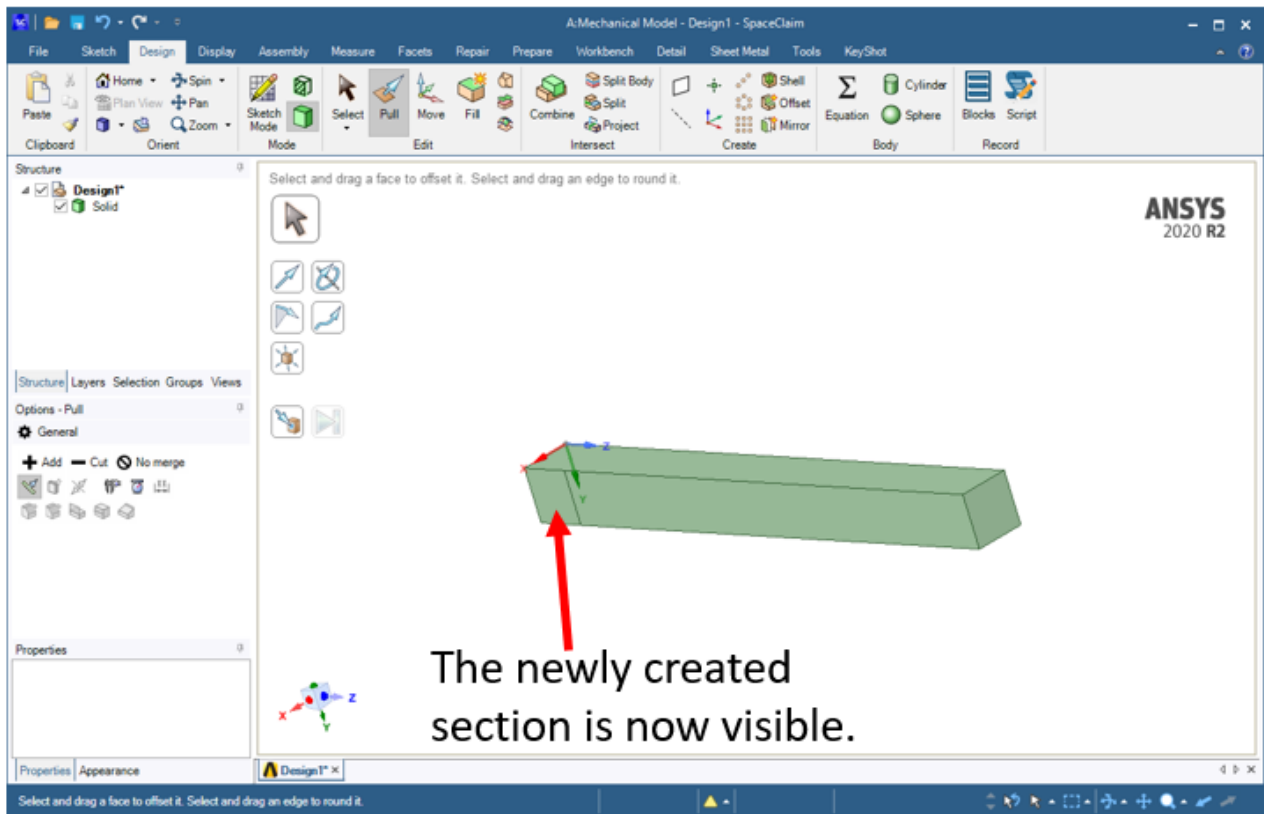
After erasing the extra dimension lines and dimension text, the plot appears as shown below.



Click on “End Sketch Editing”

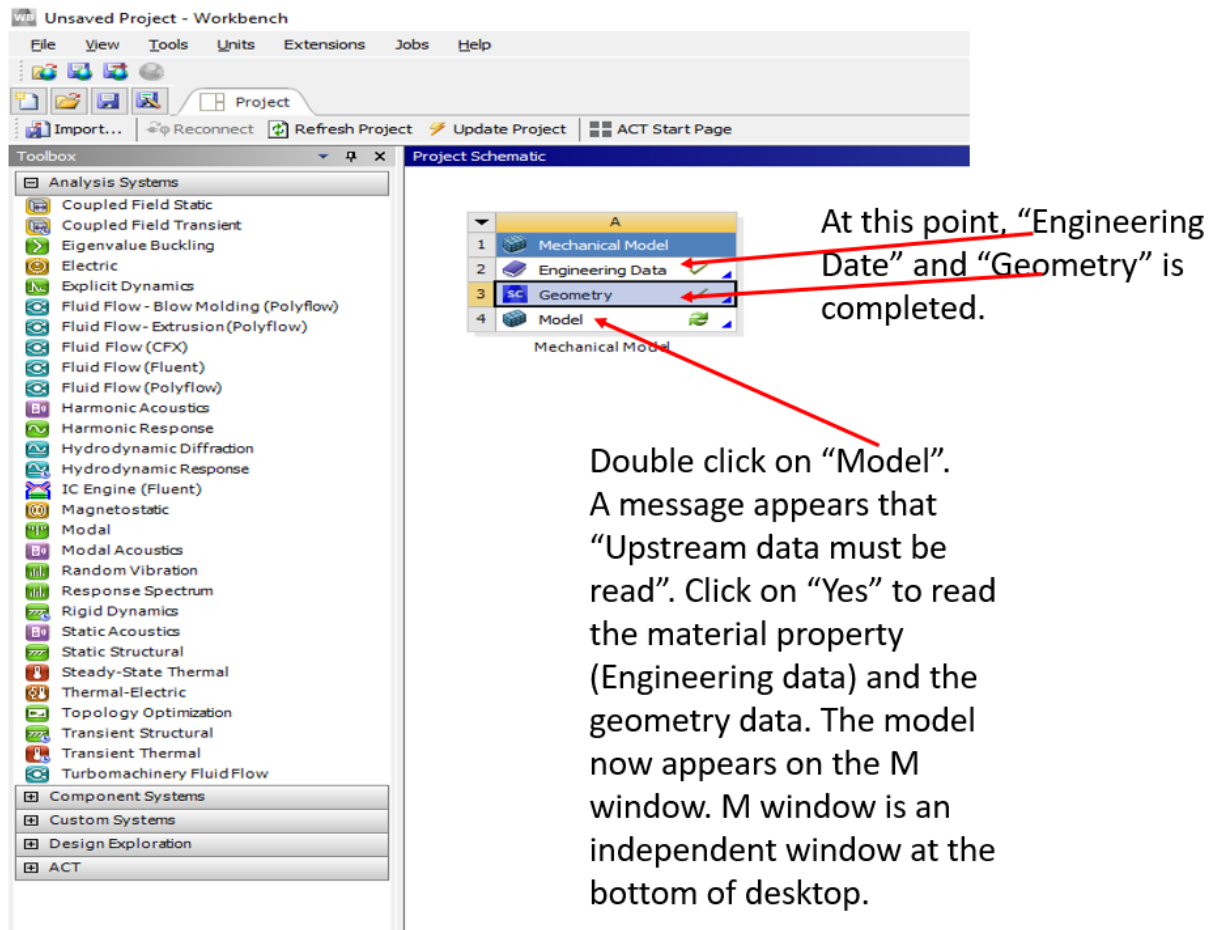


After exiting the “Sketch mode” and using the “Spin,” “pan,” and “zoom” commands and using the mouse wheel for rotation, the beam appears as shown below.

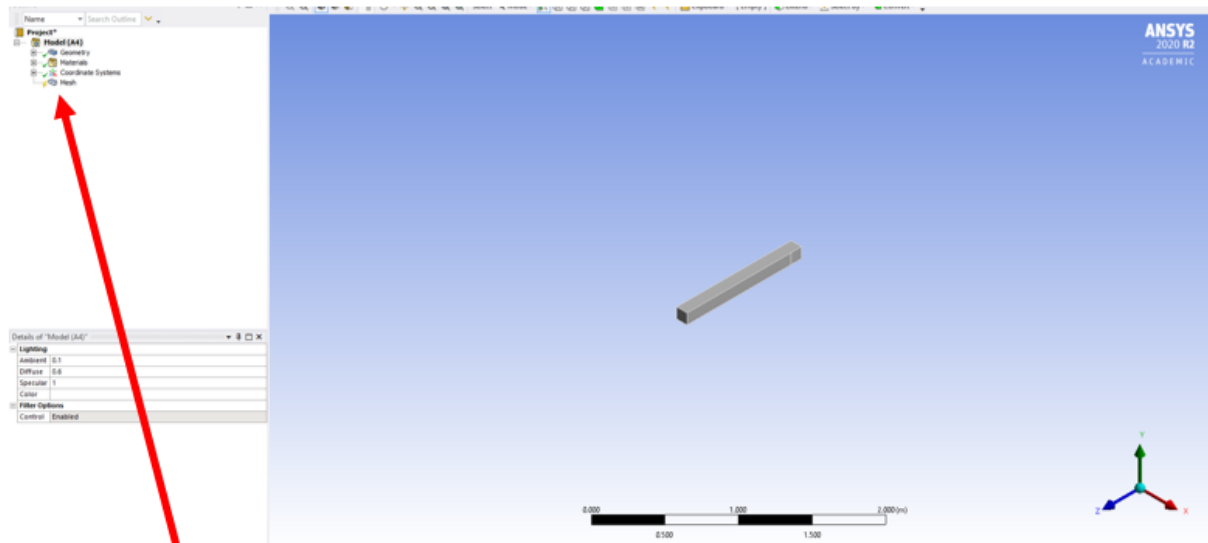


At this point, the geometry creation is completed. Minimize (but not close) the “S.C.” window.

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

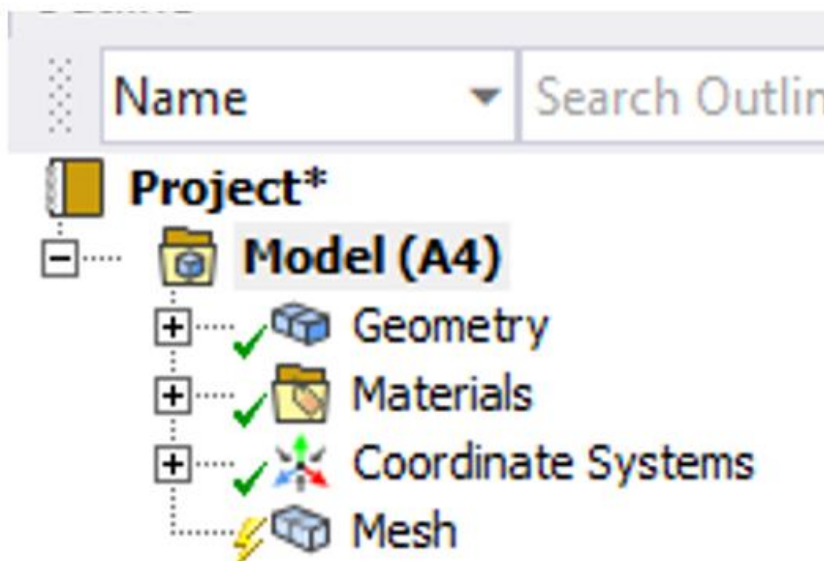


M window appears as shown below.

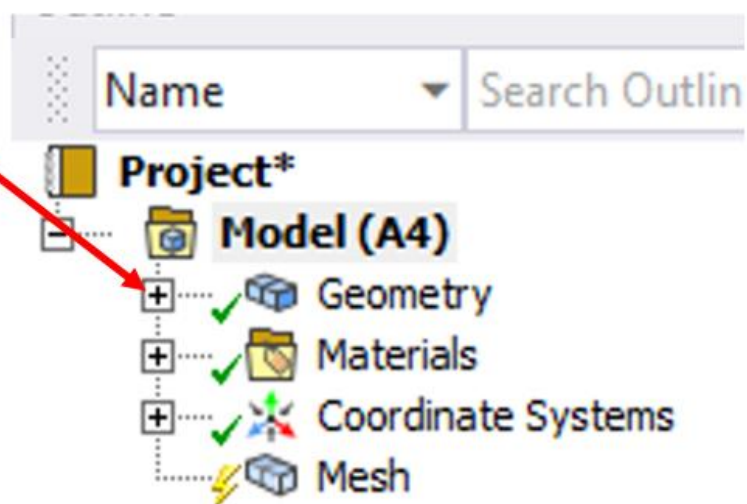


This portion is critical,
but it is too small to
see.

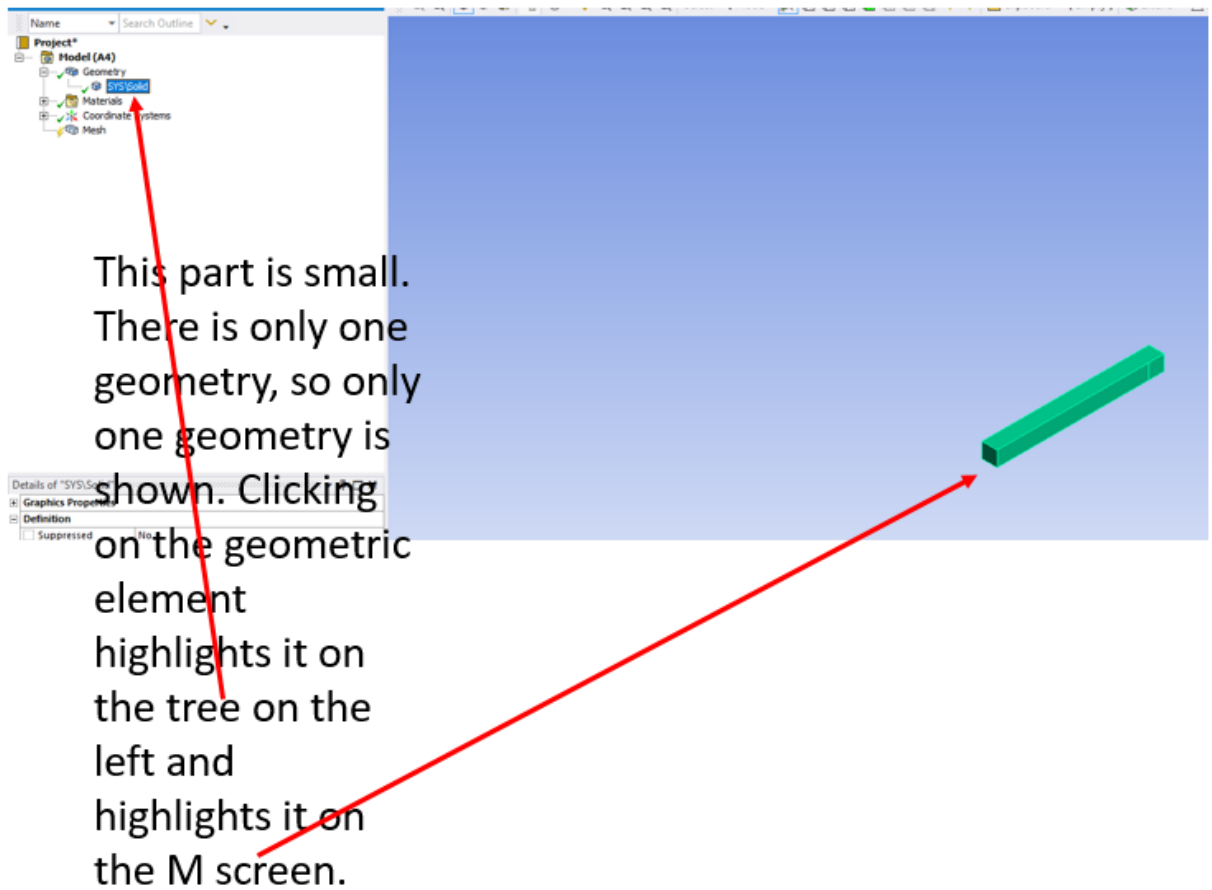
The following shows the small part in detail.



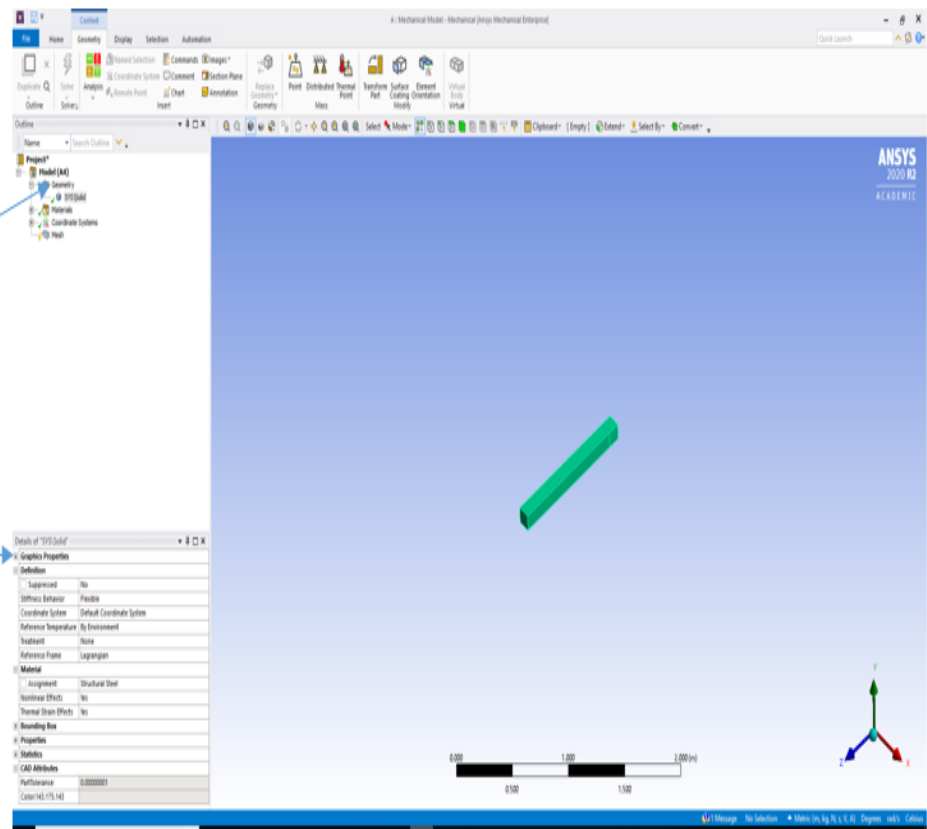
Click on The “+”
sign next to
geometry.



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



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



The default material assigned is structural steel.

Details of "SYS\Solid" ▾ 🔍 □ ×

+ Graphics Properties	
- Definition	
<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Treatment	None
Reference Frame	Lagrangian
- Material	
<input type="checkbox"/> Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
+ Bounding Box	
+ Properties	
+ Statistics	
- CAD Attributes	
PartTolerance:	0.00000001
Color:143,175,143	

Details of "SYS\Solid" ▾ □ ×

+ Graphics Properties

- Definition

<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Treatment	None
Reference Frame	Lagrangian

- Material

<input checked="" type="checkbox"/> Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes

+ Bounding Box

+ Properties

+ Statistics

- CAD Attributes

PartTolerance:	0.00000001
Color:143.175.143	

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

Details of "SYS\Solid" ▾ 🔍 □ ×

+ **Graphics Properties**

- **Definition**

<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Treatment	None
Reference Frame	Lagrangian

- **Material**

<input checked="" type="checkbox"/> Assignment	Structural Steel	▶
Nonlinear Effects	Yes	
Thermal Strain Effects	Yes	

+ **Bounding Box**

+ **Properties**

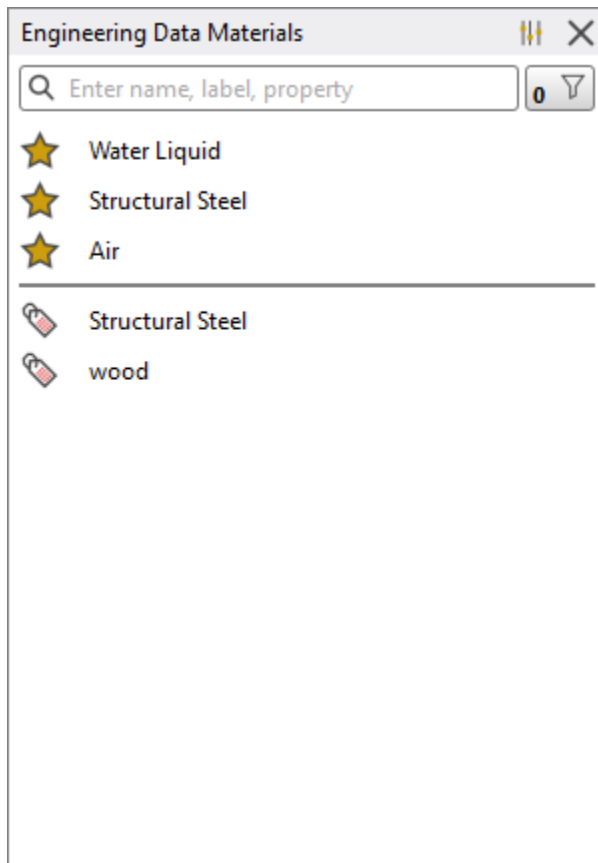
+ **Statistics**

- **CAD Attributes**

PartTolerance:	0.00000001
Color:143.175.143	

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.

Outline

Name Search Outline

Project*

Model (A4)

- Geometry
- SYS/Solid
- Materials
 - Structural Steel
 - wood
- Coordinate Systems
- Mesh

Engineering Data: Material View

wood

Structural

Isotropic Elasticity

Derive from	Young's Modulus and Poisson's Ratio
Young's Modulus	8e+09 Pa
Poisson's Ratio	0.3
Bulk Modulus	6.6667e+09 Pa
Shear Modulus	3.0769e+09 Pa

Details of "wood"

Common Material Properties

Density	
Young's Modulus	8e+09 Pa
Thermal Conductivity	
Specific Heat	
Tensile Yield Strength	
Tensile Ultimate Strength	
Nonlinear Behavior	False
Full Details	Click To View Full Details

Statistics

Assigned Bodies	1
-----------------	---

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.

Project*

Model (A4)

- Geometry
- SYS/Solid
- Materials
 - Structural Steel
 - wood
- Coordinate Systems
- Mesh

Geometry

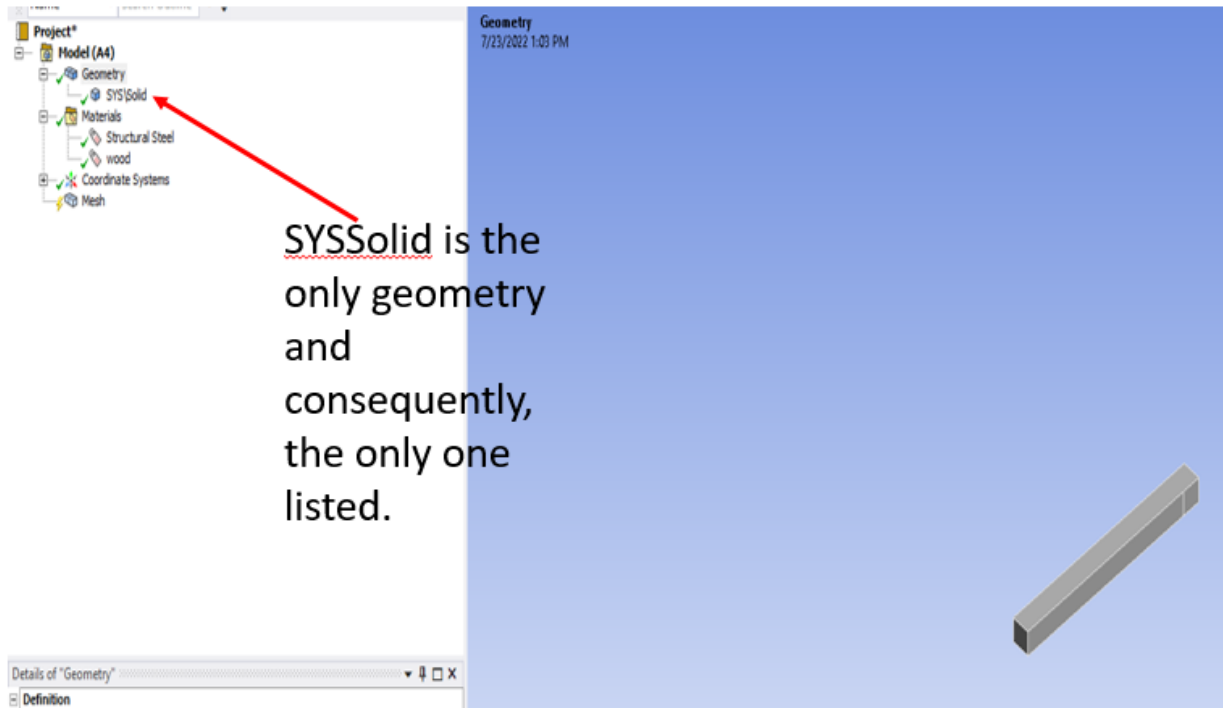
7/23/2022 1:03 PM

Details of "Geometry"

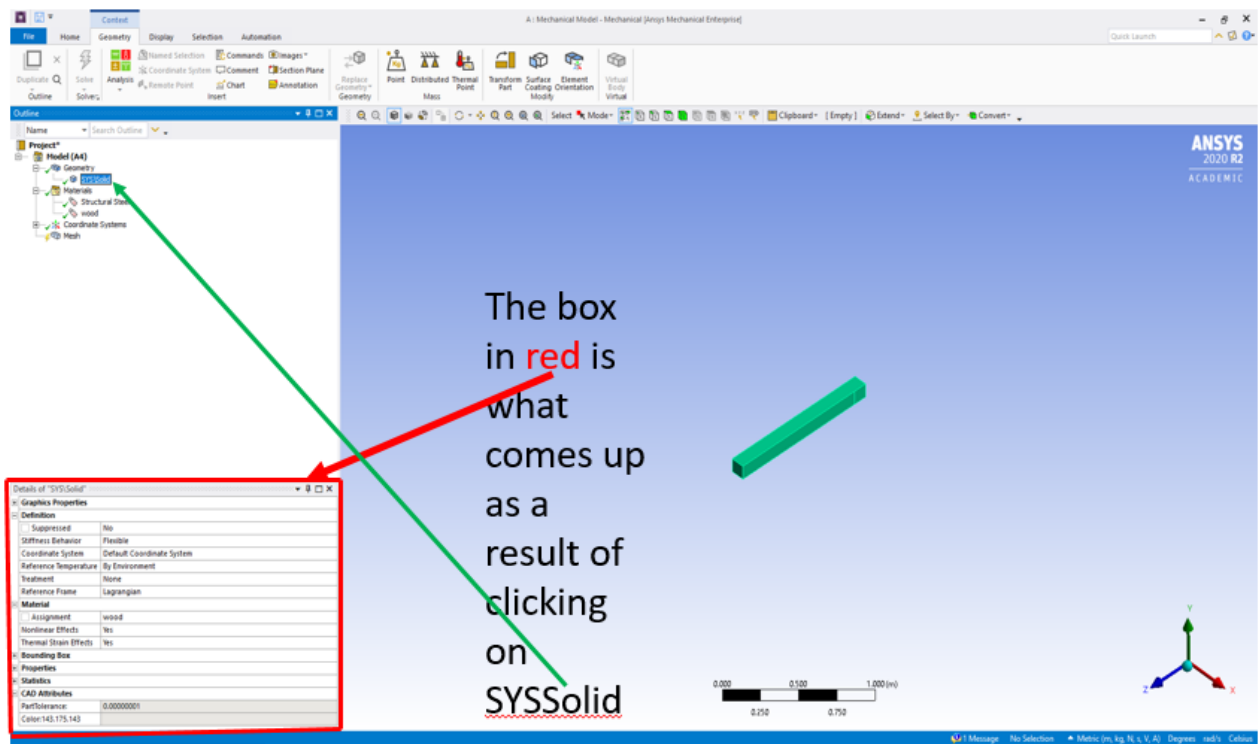
Definition



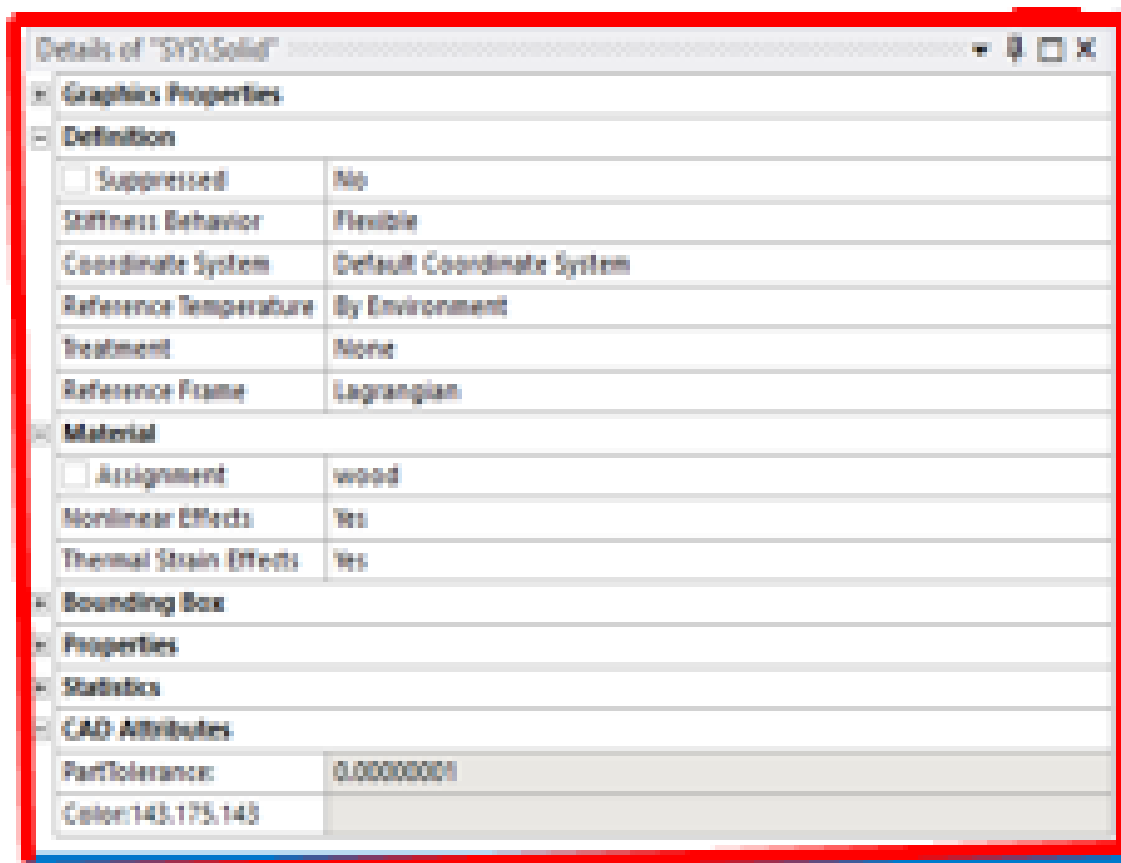
In the case of this example, there is only one Geometry, called “SYSSolid.”



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

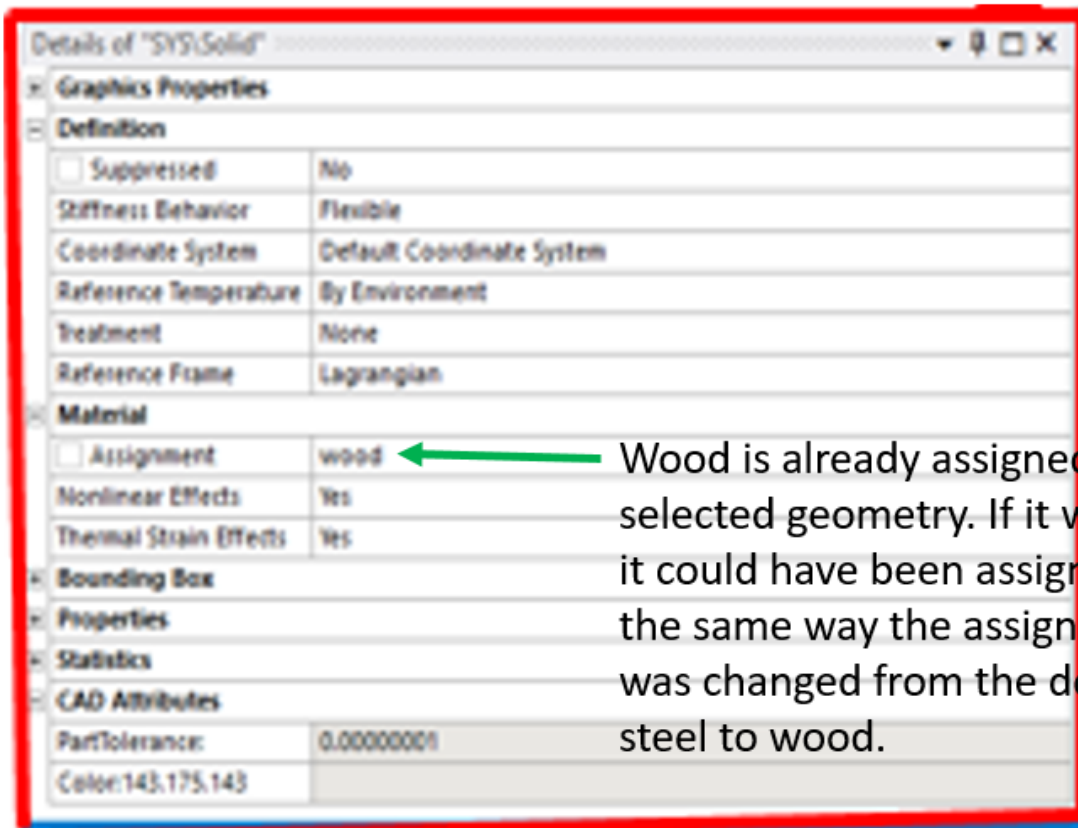


The following is the enlarged red box from the previous screen.



The image shows a software dialog box titled "Details of 'SYS:Solid'". It contains several expandable sections. The "Definition" section is expanded, showing a table of properties. The "Material" section is also expanded, showing another table of properties. The "CAD Attributes" section is expanded, showing a table of attributes. The dialog box has a red border and standard window controls in the top right corner.

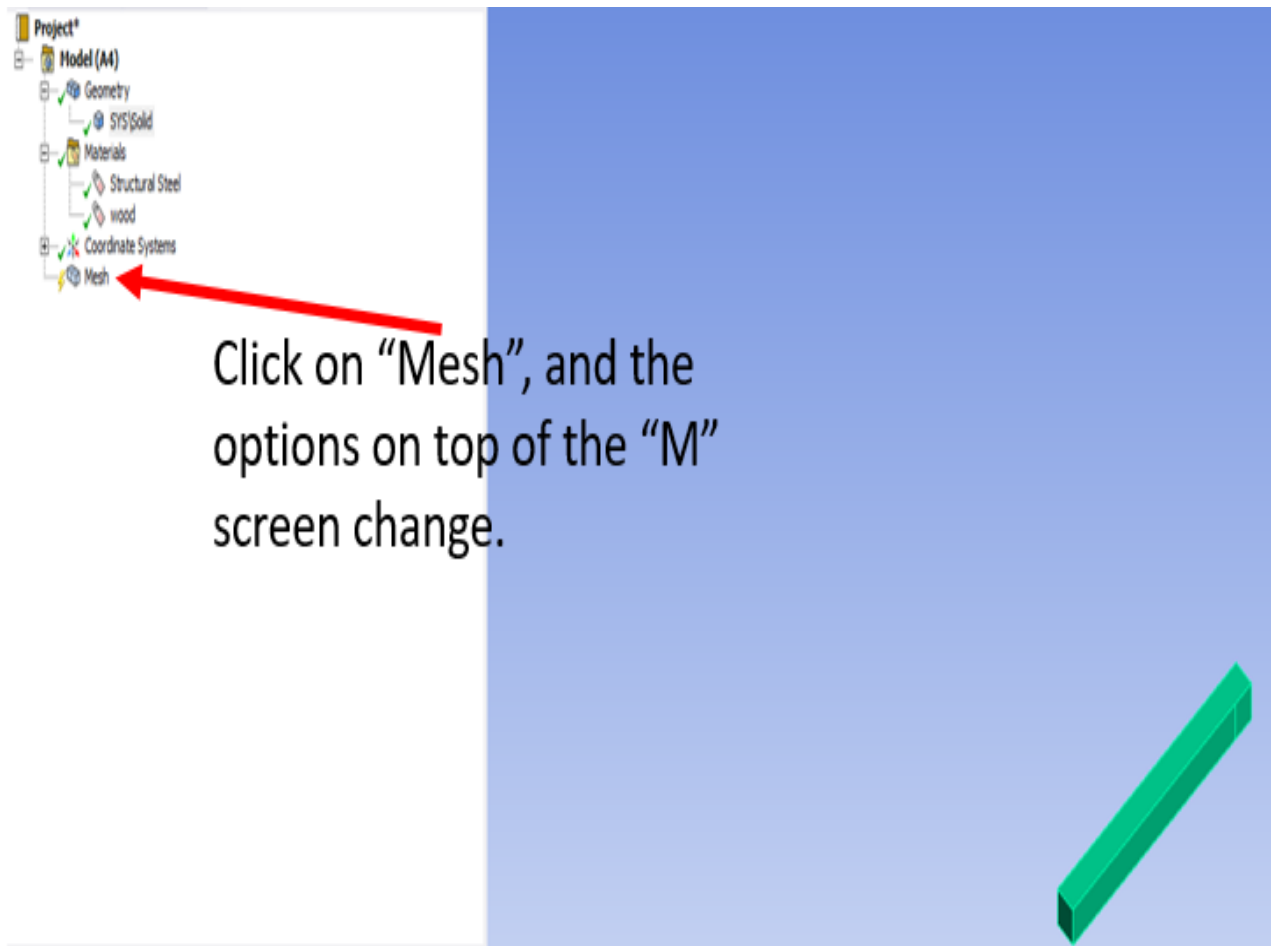
Details of "SYS:Solid"	
Graphics Properties	
Definition	
<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Treatment	None
Reference Frame	Lagrangian
Material	
<input type="checkbox"/> Assignment	wood
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	
CAD Attributes	
PartTolerance:	0.00000001
Color:143,175,143	



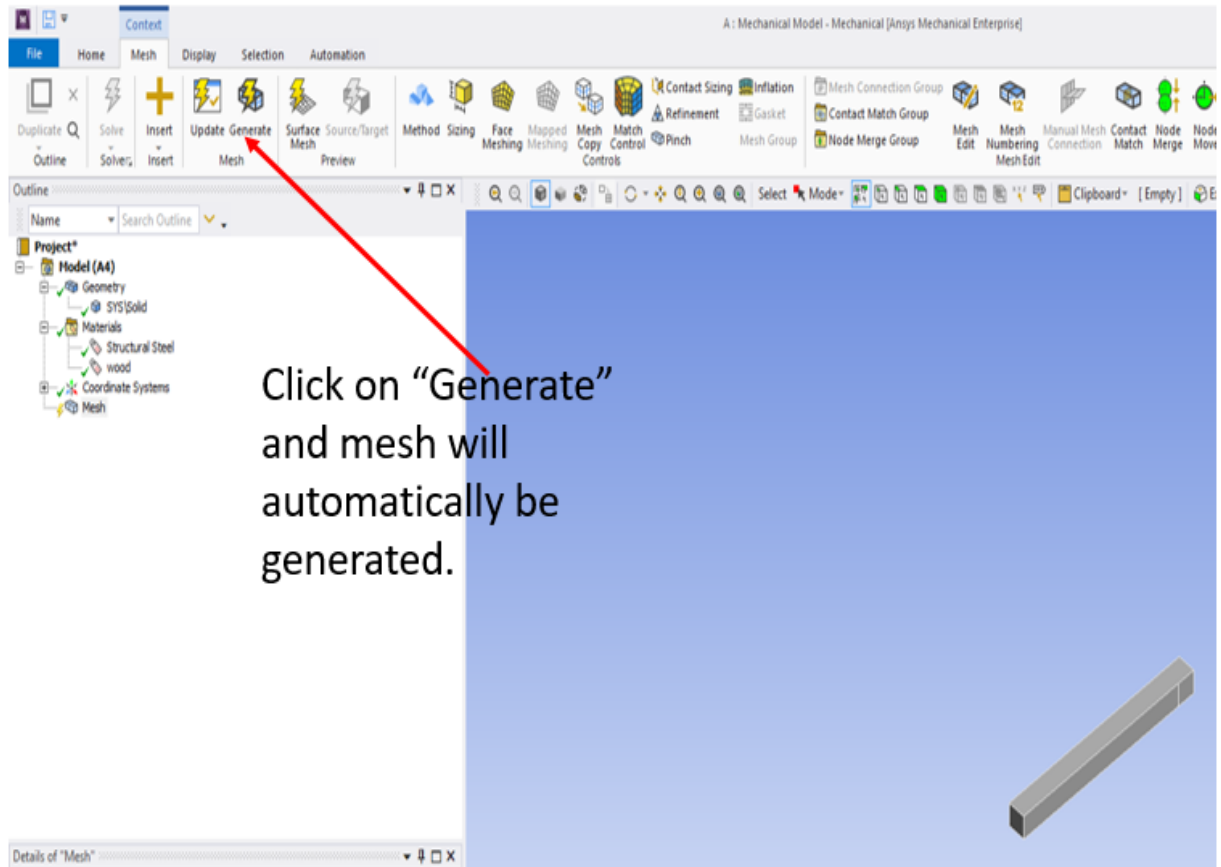
Wood is already assigned to the selected geometry. If it were not, it could have been assigned here the same way the assignment was changed from the default steel to wood.

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

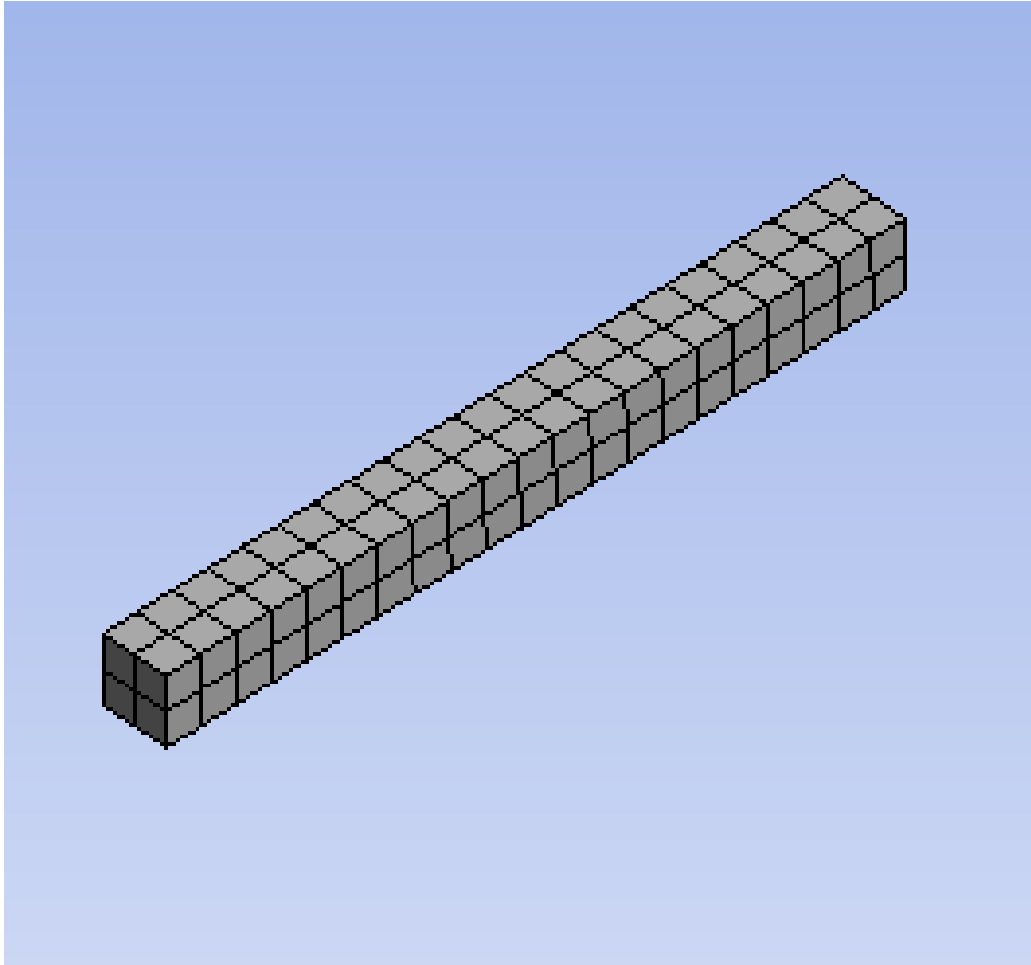
At this point, the geometry has been completed, and the material has been defined and assigned to the Geometry. The next step is to create the Mesh.

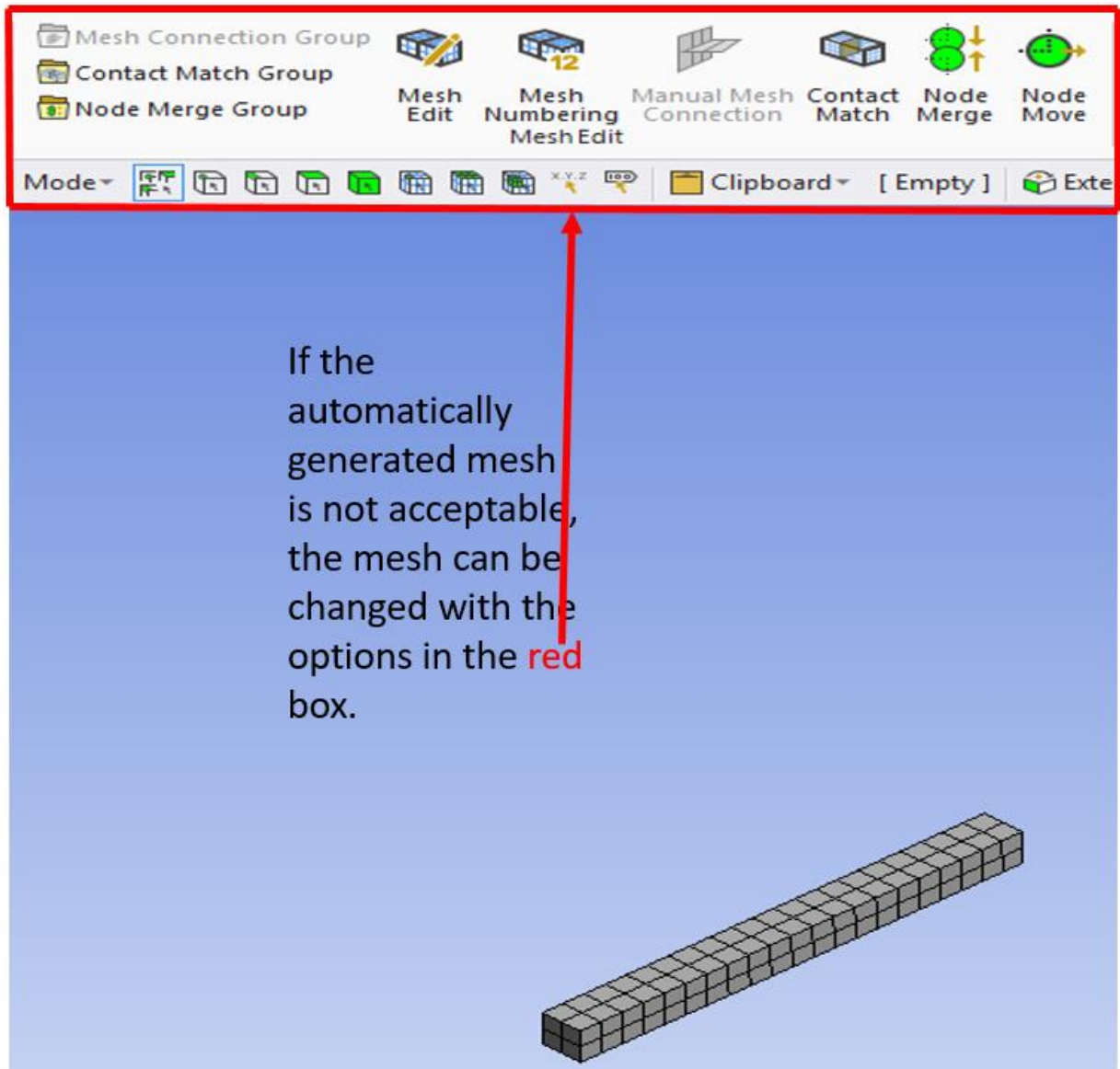


If the Geometry is simple as it is here, Clicking on “Generate” on top of the screen will automatically generate a mesh.

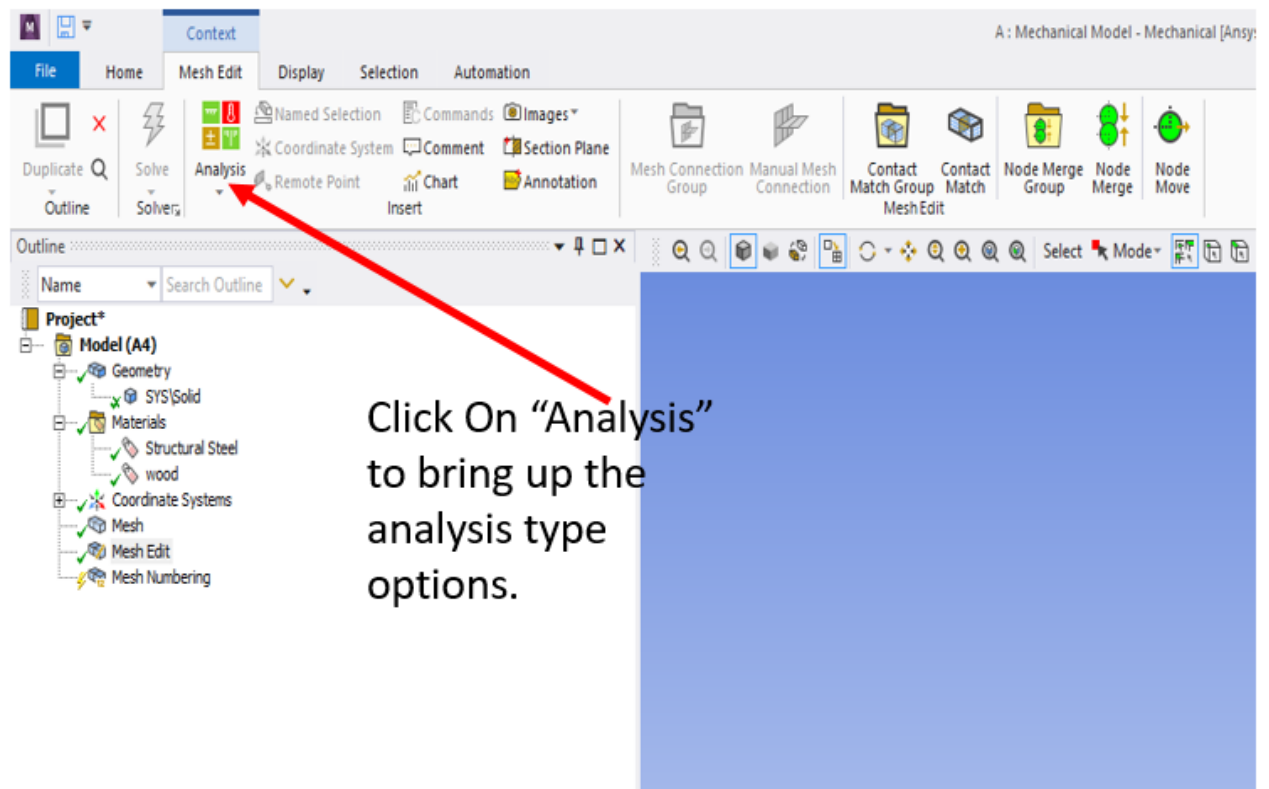


After clicking on “Generate,” the Geometry is meshed as shown below.

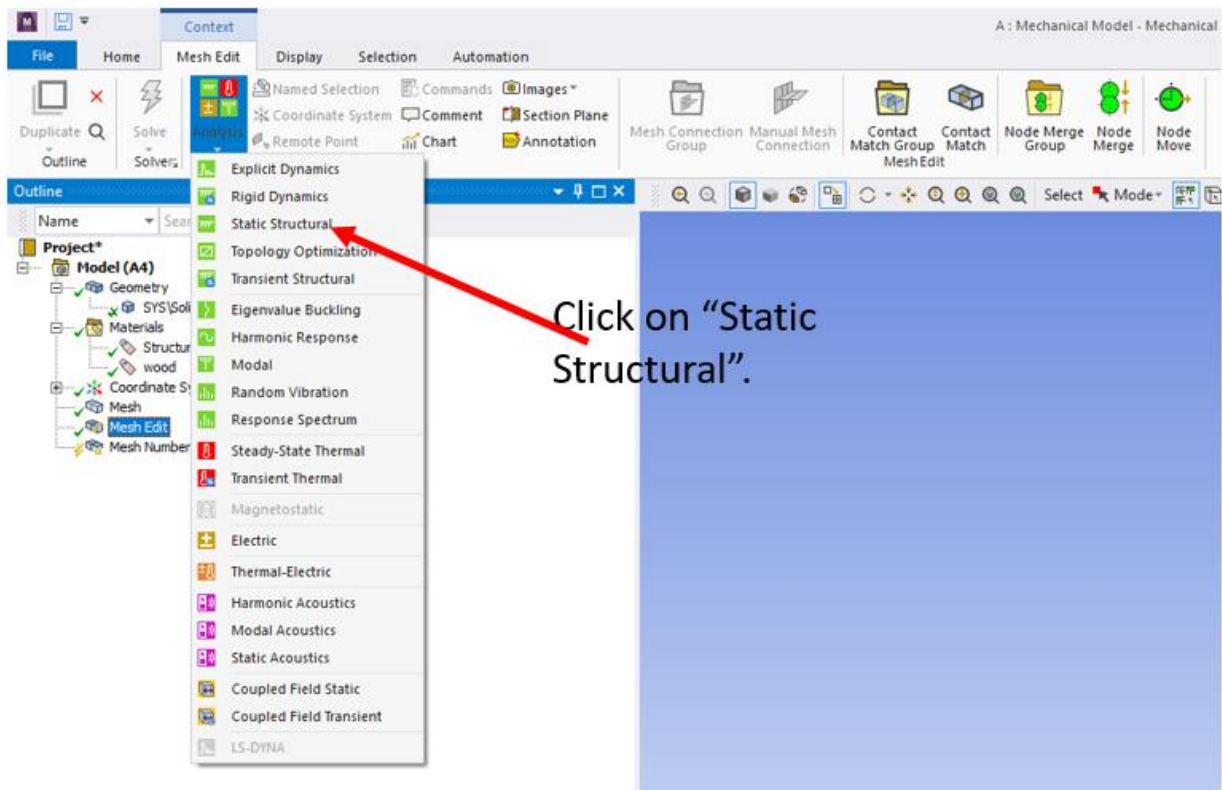


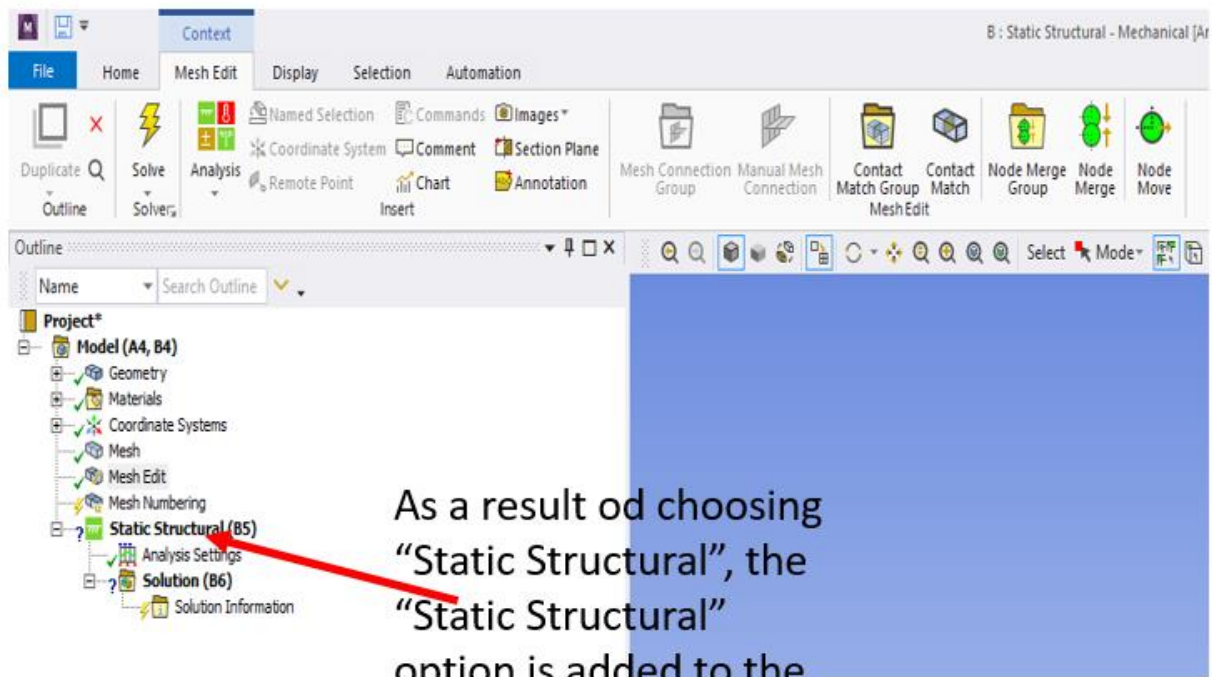


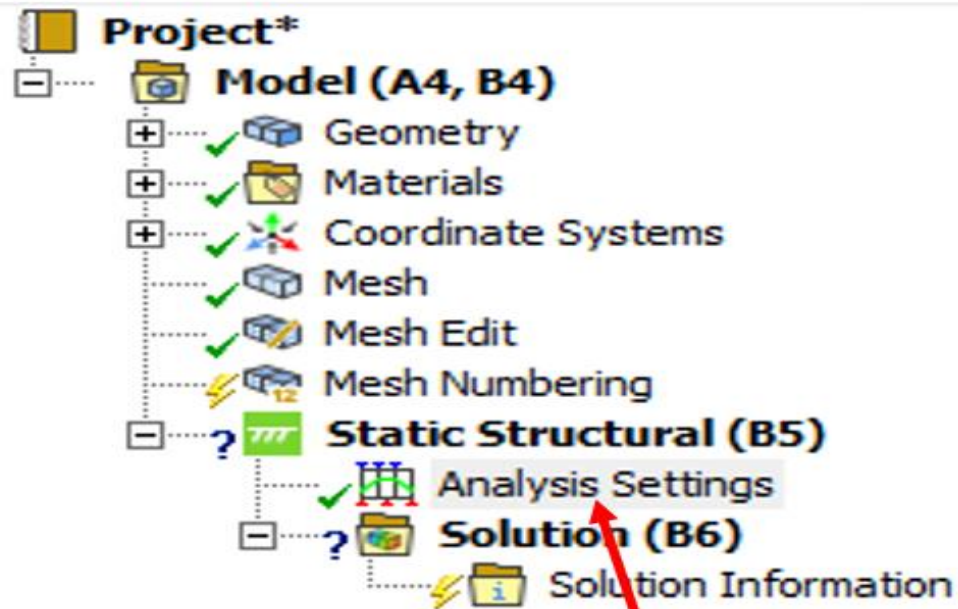
For some reason, the Static Structural option has disappeared on both WB (Work Bench) and M (Mechanical) screens.



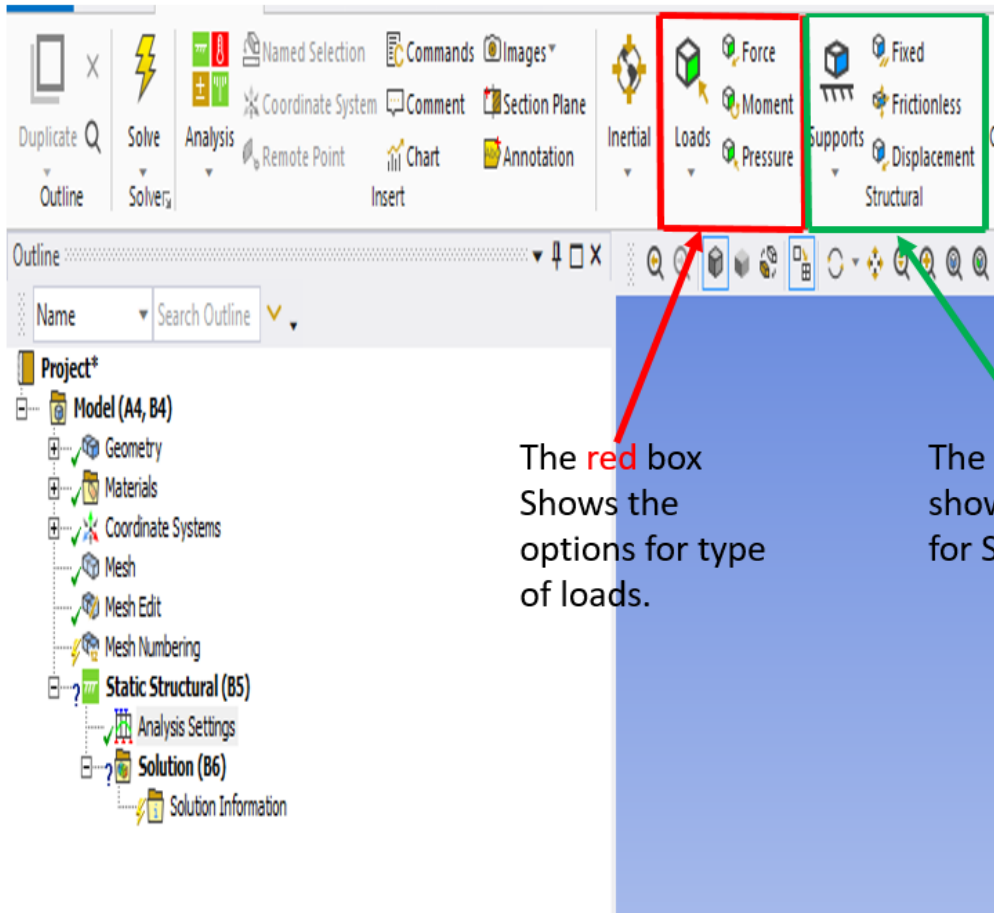
After clicking on “Analysis,” the screen appears as shown below.



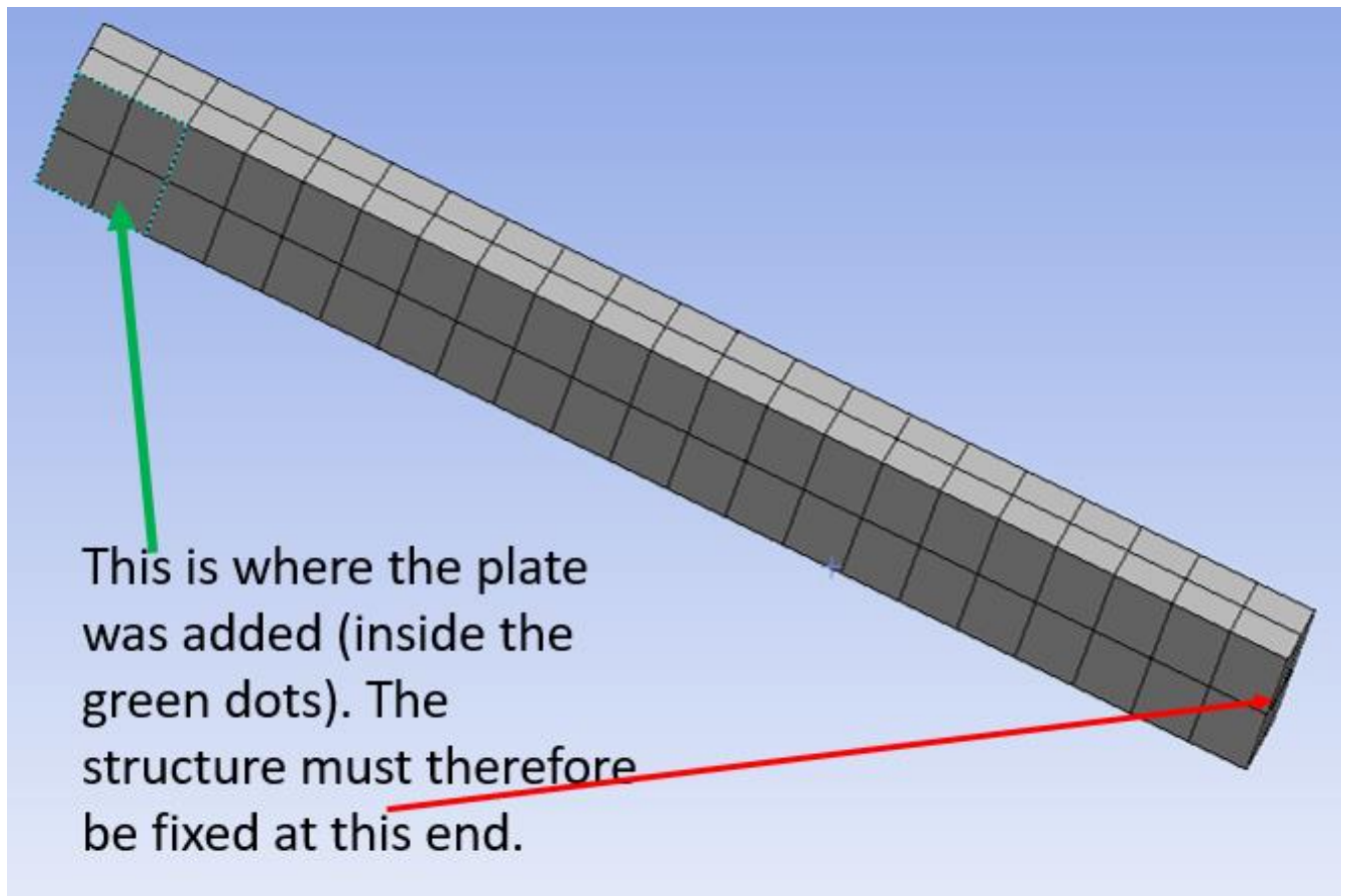




Click on 'Analysis Settings' and the menu on top of "M" screen changes. Among the options that appear on top of "M" screen are "Loads" and "Supports".



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.

File Home Environment Display Selection Automation

Duplicate Outline Solve Analysis Remote Point Chart Annotation Inertial Loads Force Moment Supports Frictionless Displacement Structural Conditions Direct FE Imported Load (Result File) Write Input File... Export Nastran File Tools Worksheet Graph Tabular Data Views

Outline Name Search Outline

Project*

- Model (A4, B4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
 - Mesh Edit
 - Mesh Numbering
 - Static Structural (B5)
 - Analysis Settings
 - Fixed Support
 - Solution (B6)
 - Solution Information

B: Static Structural

Fixed Support

Time: 1. s

7/23/2022 7:14 PM

Fixed Support

Details of "Fixed Support"

Scope

Scoping Method: Geometry Selection

Geometry Apply Cancel

Definition

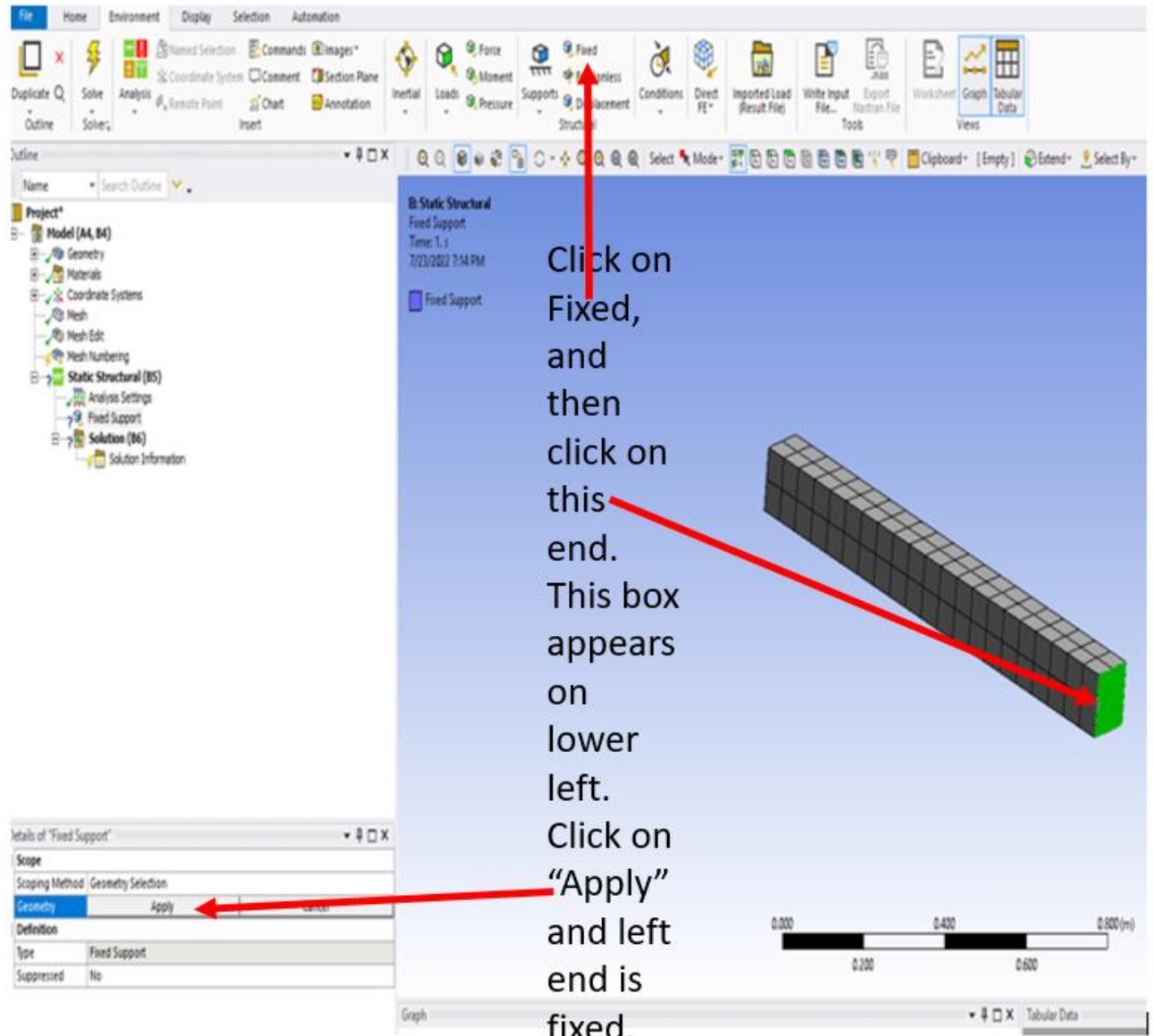
Type: Fixed Support

Suppressed: No

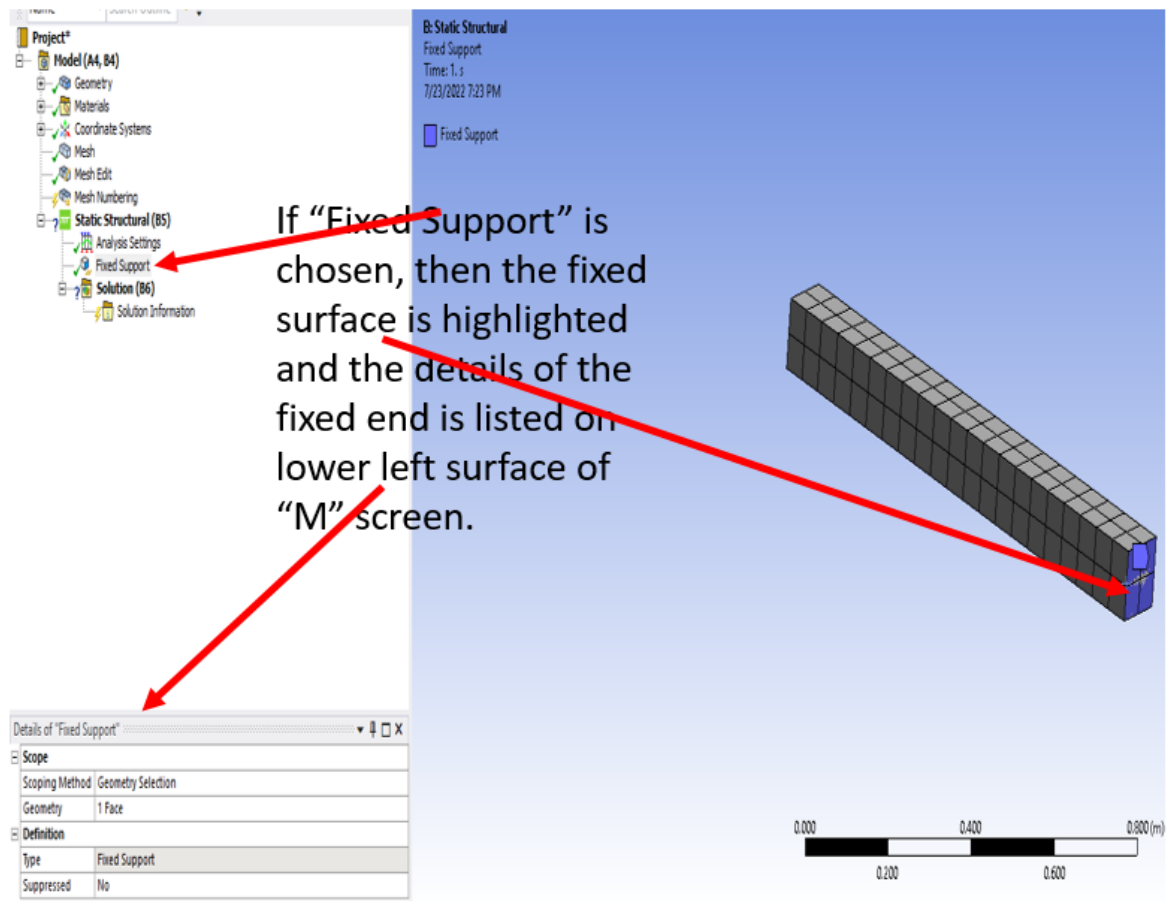
Graph Tabular Data

0.000 0.200 0.400 0.600 0.800 (m)

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.

The screenshot displays the ANSYS Workbench environment. The top toolbar includes icons for various analysis types, with 'Pressure' highlighted. The left sidebar shows the project hierarchy, including 'Model (AA, B4)', 'Geometry', 'Coordinate Systems', 'Mesh', 'Static Structural (B5)', 'Analysis Settings', 'Fixed Support', 'Pressure', and 'Solution (B6)'. The main workspace shows a 3D model of a rectangular block with a red square on its top surface, indicating the area where pressure is applied. A red arrow points from the 'Pressure' icon in the toolbar to the red square on the model. Another red arrow points from the 'Magnitude' field in the 'Details of Pressure' panel to the value '30000'. A third red arrow points from the 'Geometry' field in the same panel to the red square. A blue arrow points from the 'Geometry' field to the red square. A scale bar at the bottom right indicates dimensions from 0.000 to 0.600 (m). A 'Tabular Data' window is open at the bottom right, showing a table with columns for 'Steps', 'Time (s)', and 'Pressure'. The table has two rows: the first row shows '1' for Steps, '0' for Time, and '0' for Pressure; the second row shows '2' for Steps, '1' for Time, and '30000' for Pressure.

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 square in Powerpoint. This is where the surface was created for pressure application.

Steps	Time (s)	Pressure
1	0	0
2	1	30000

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.

Details of “Pressure”

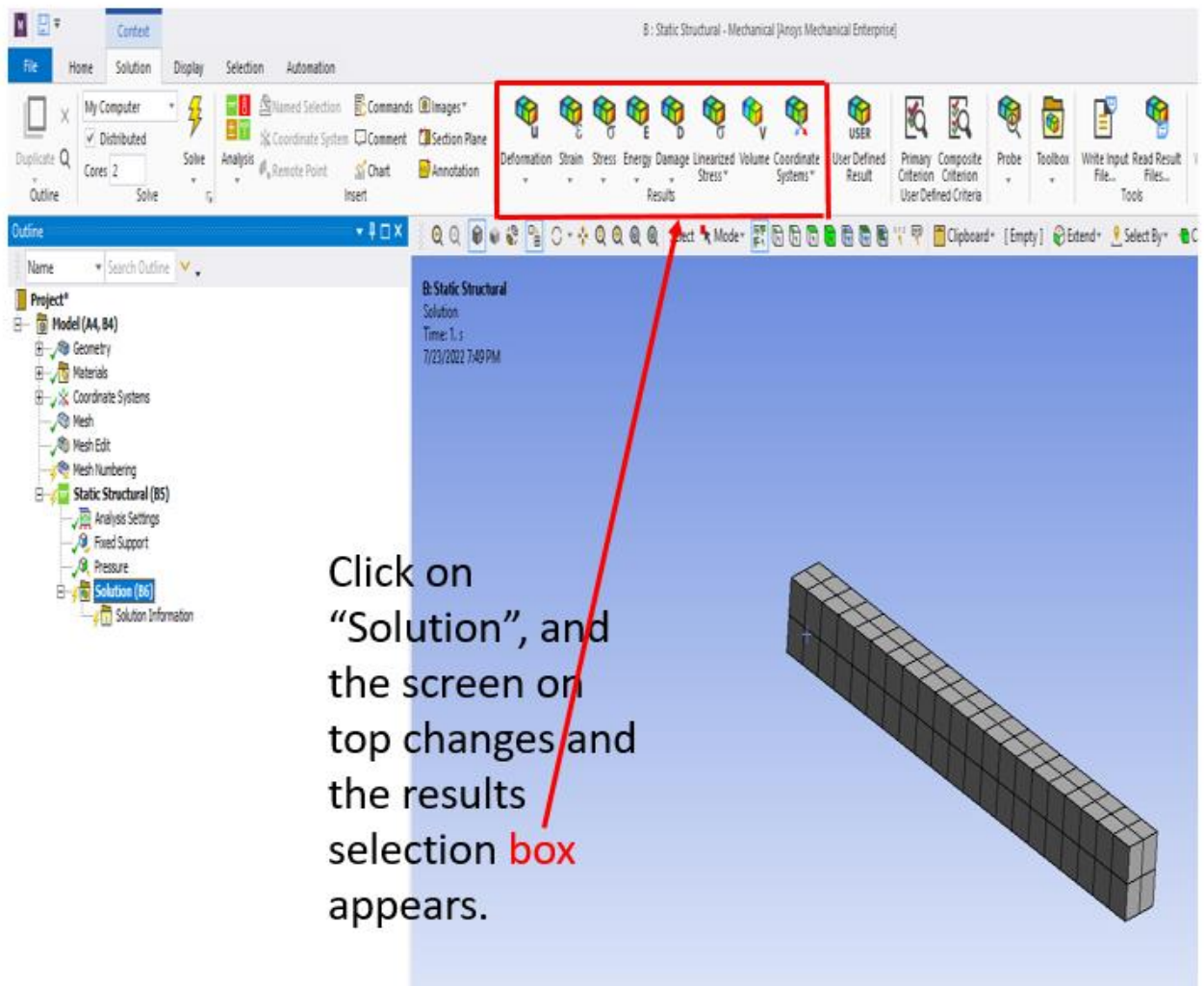
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Pressure
Define By	Normal To
Applied By	Surface Effect
Loaded Area	Deformed
<input type="checkbox"/> Magnitude	30000 Pa (ramped)
Suppressed	No

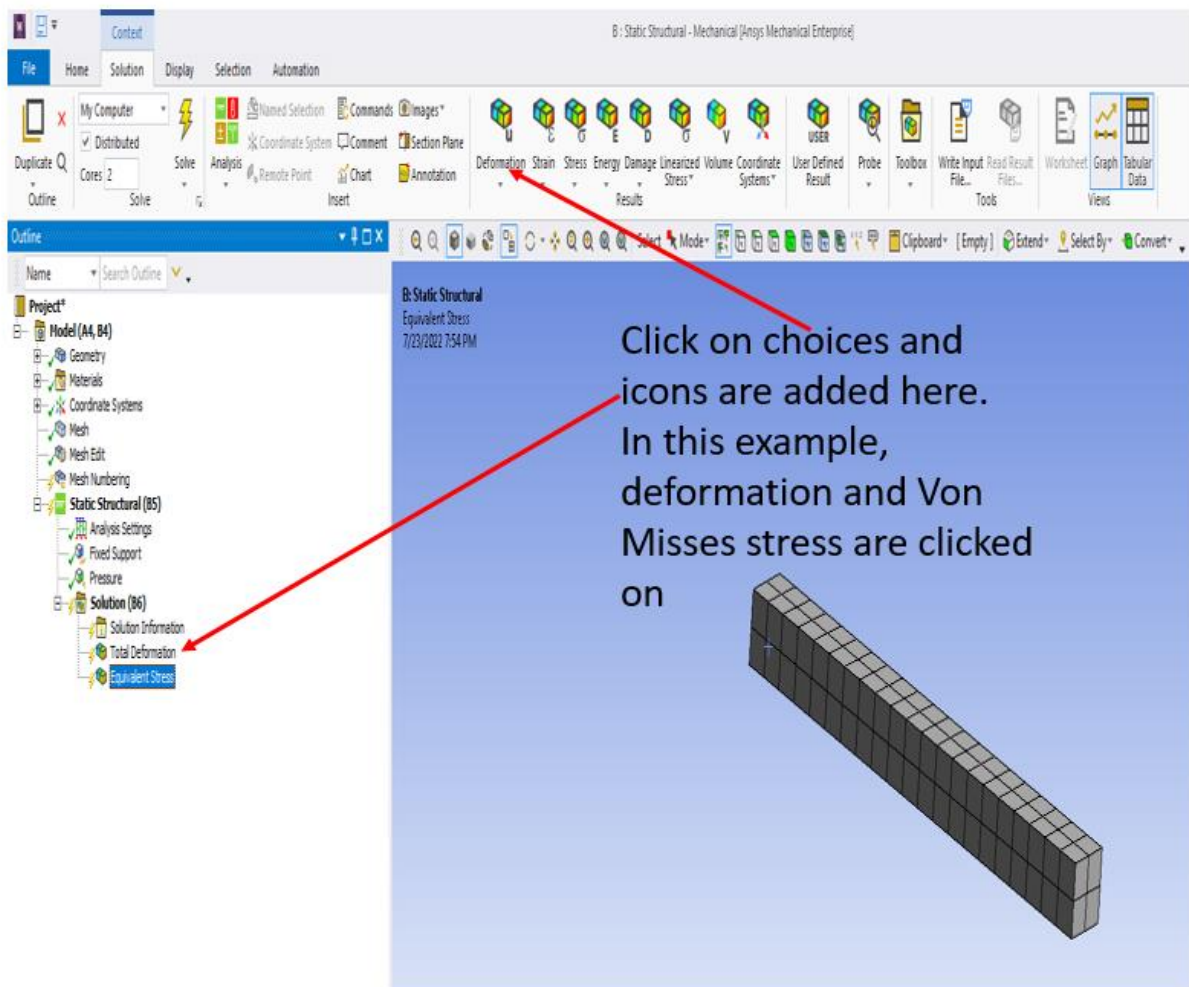
Graph

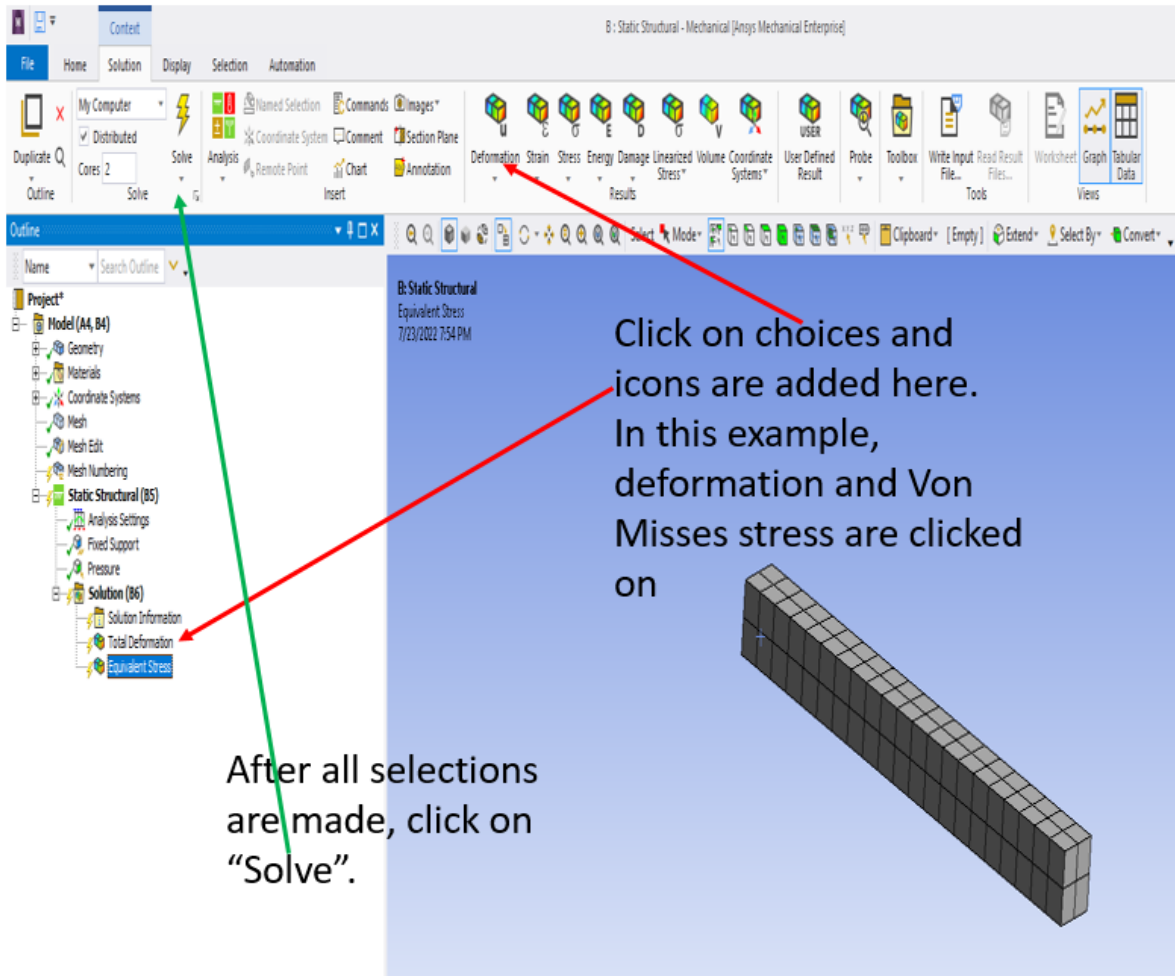
240000

Tabular Data

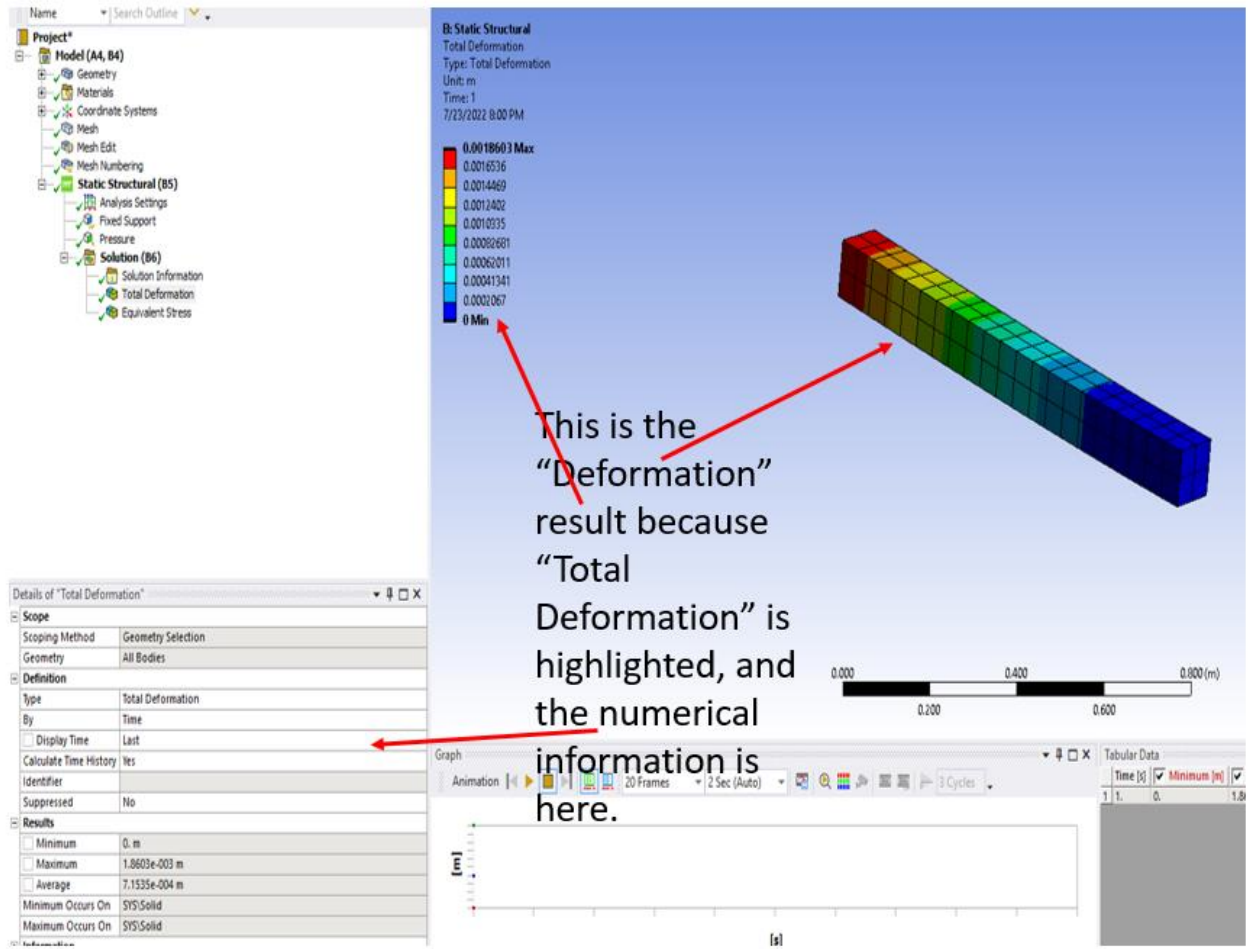
Steps	Time [s]	Pressure
1	0.	0.
2	1.	30000

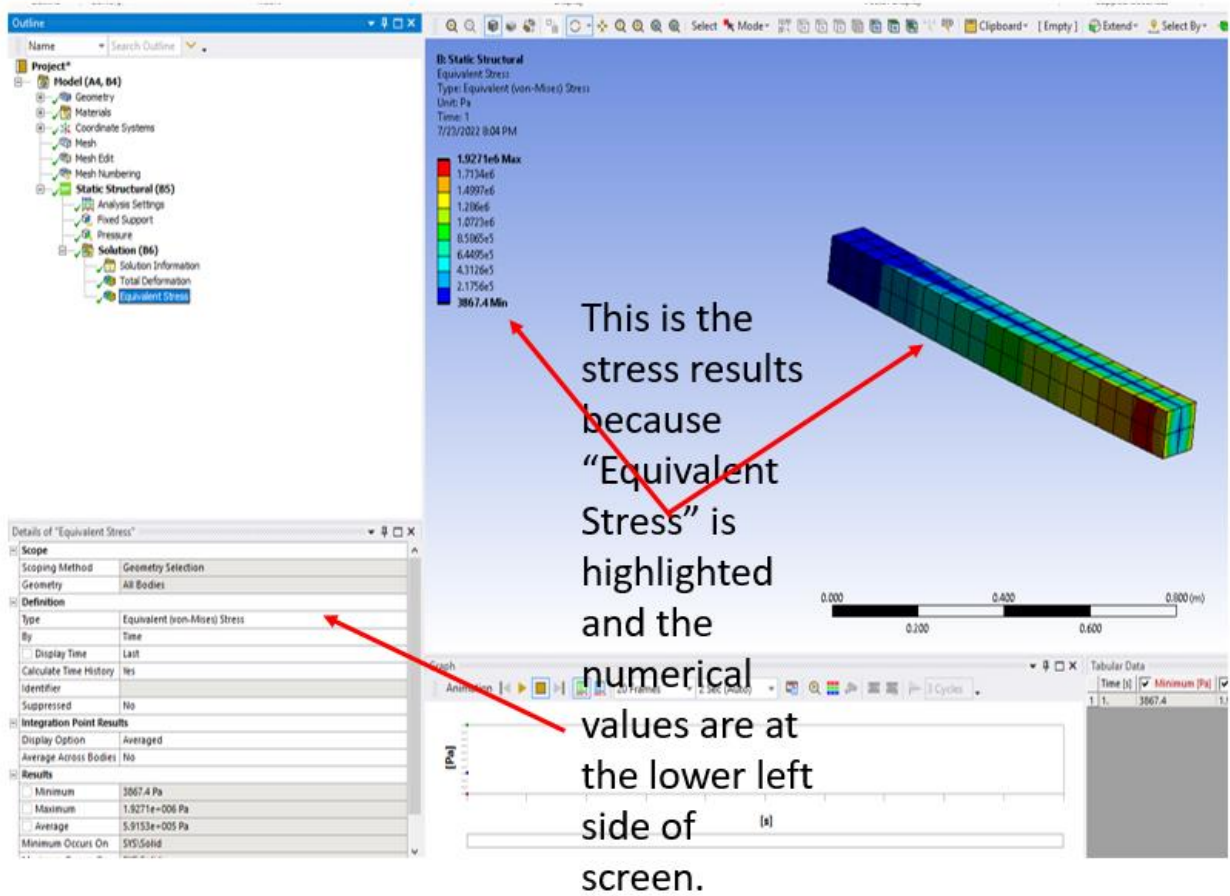




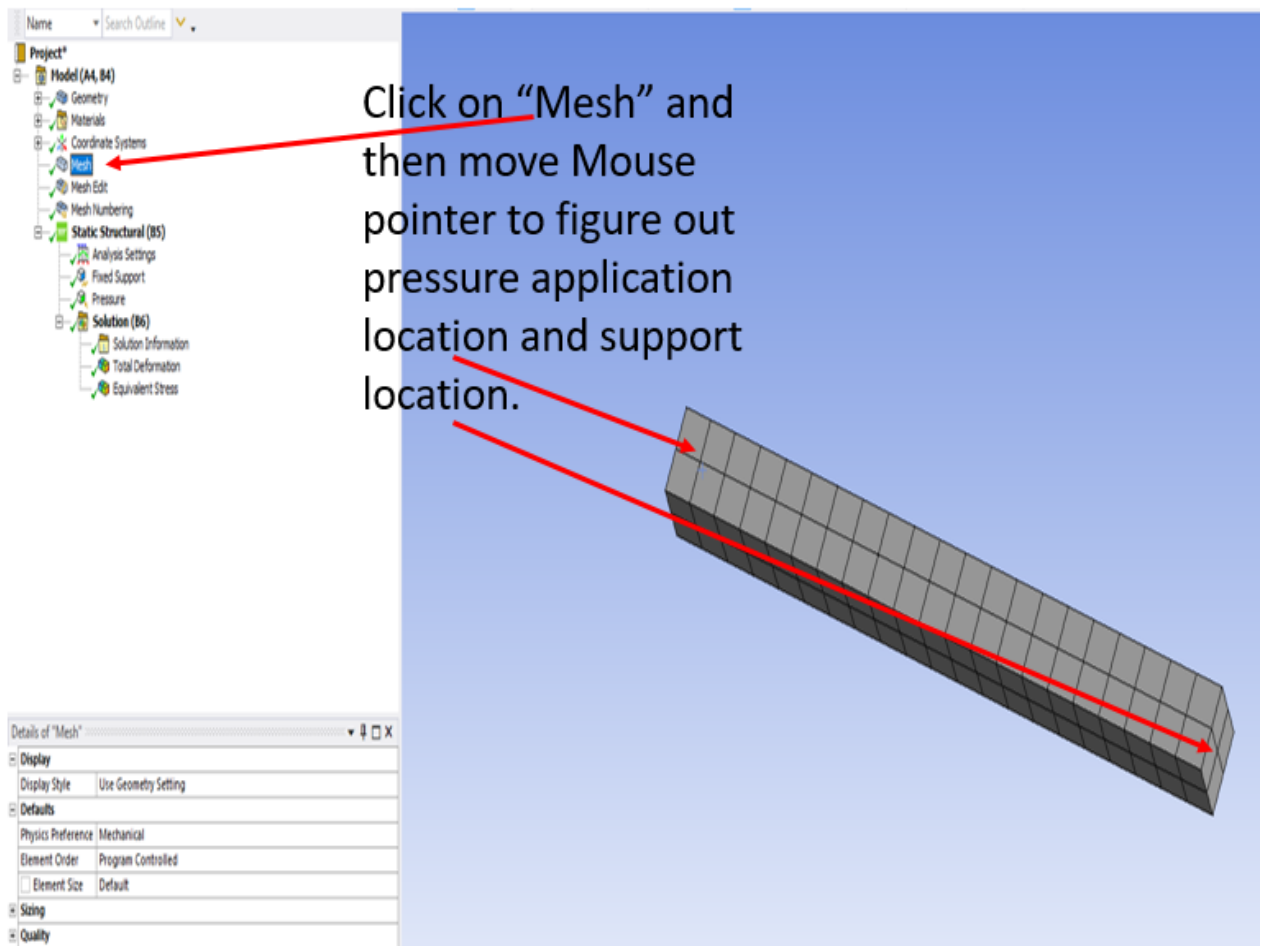


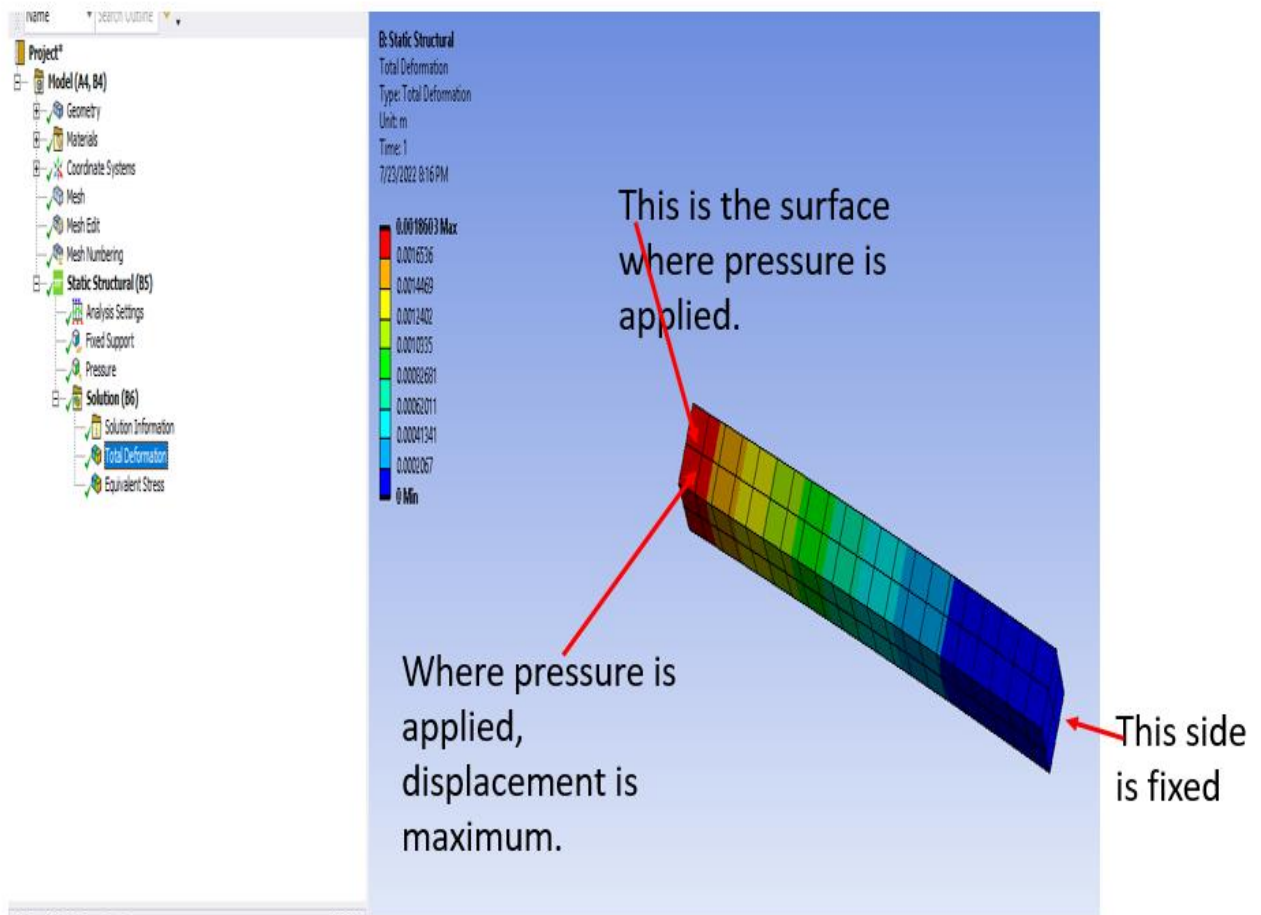
The solution will be obtained after some security warnings by the computer. Allow the access the software requires.

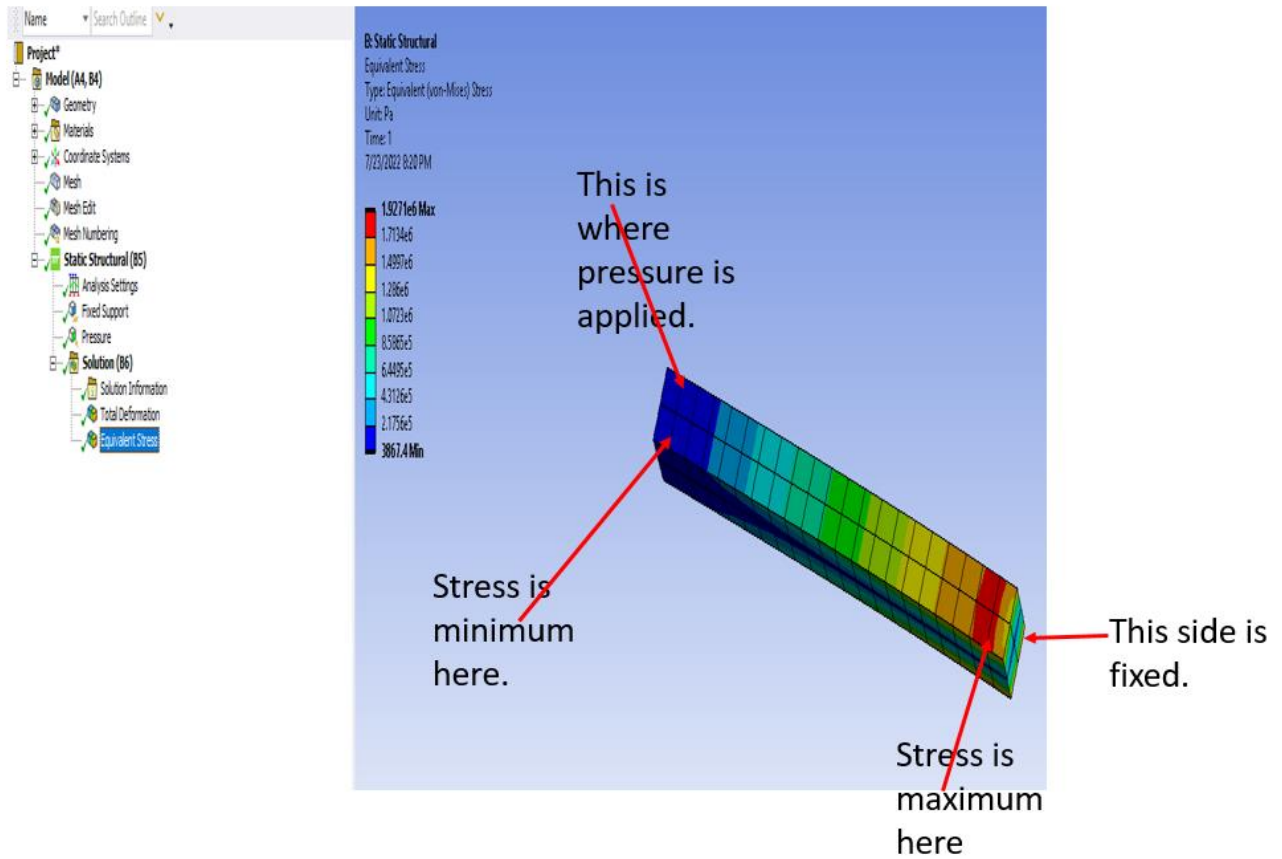


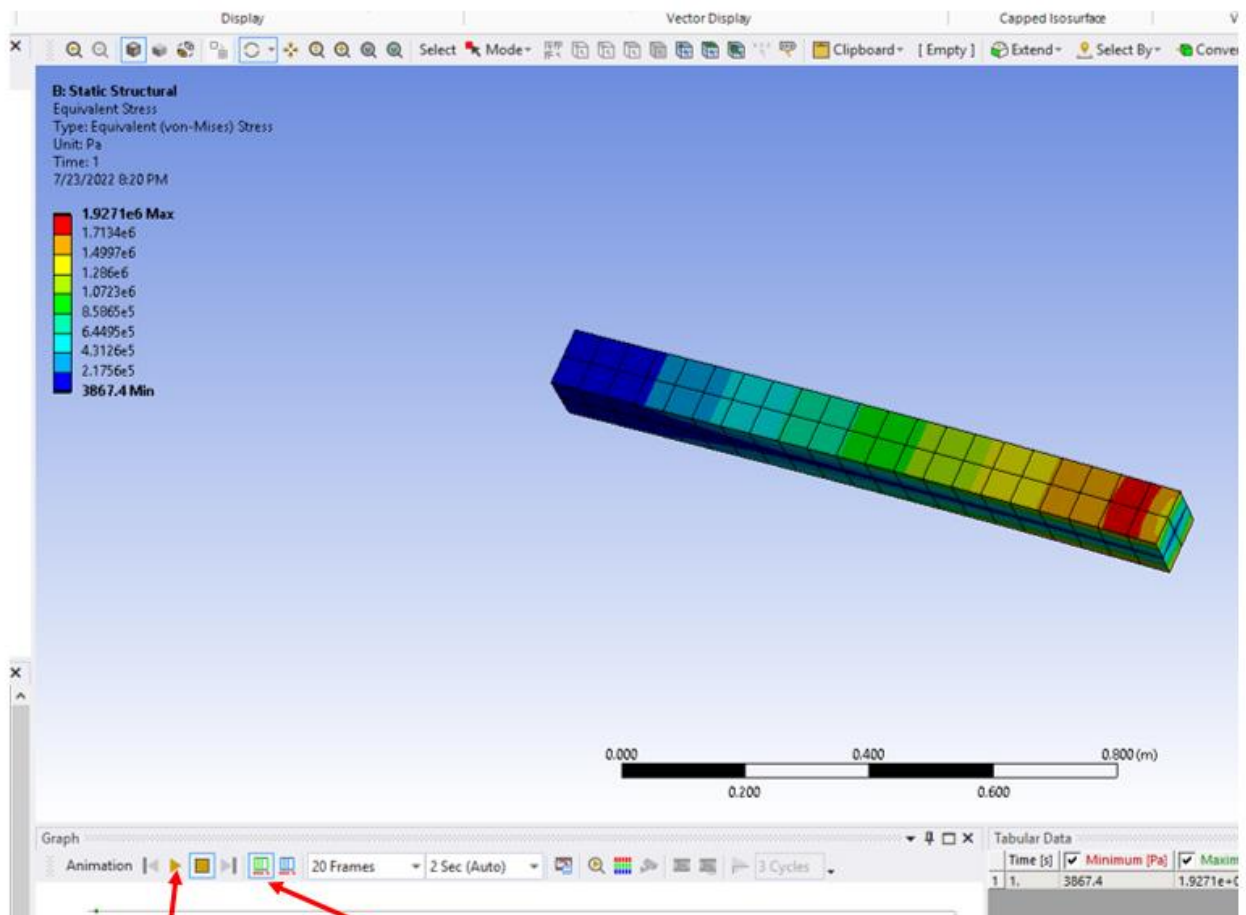


The model can be rotated by a mouse wheel to make the results more apparent. The following is a description of the process.



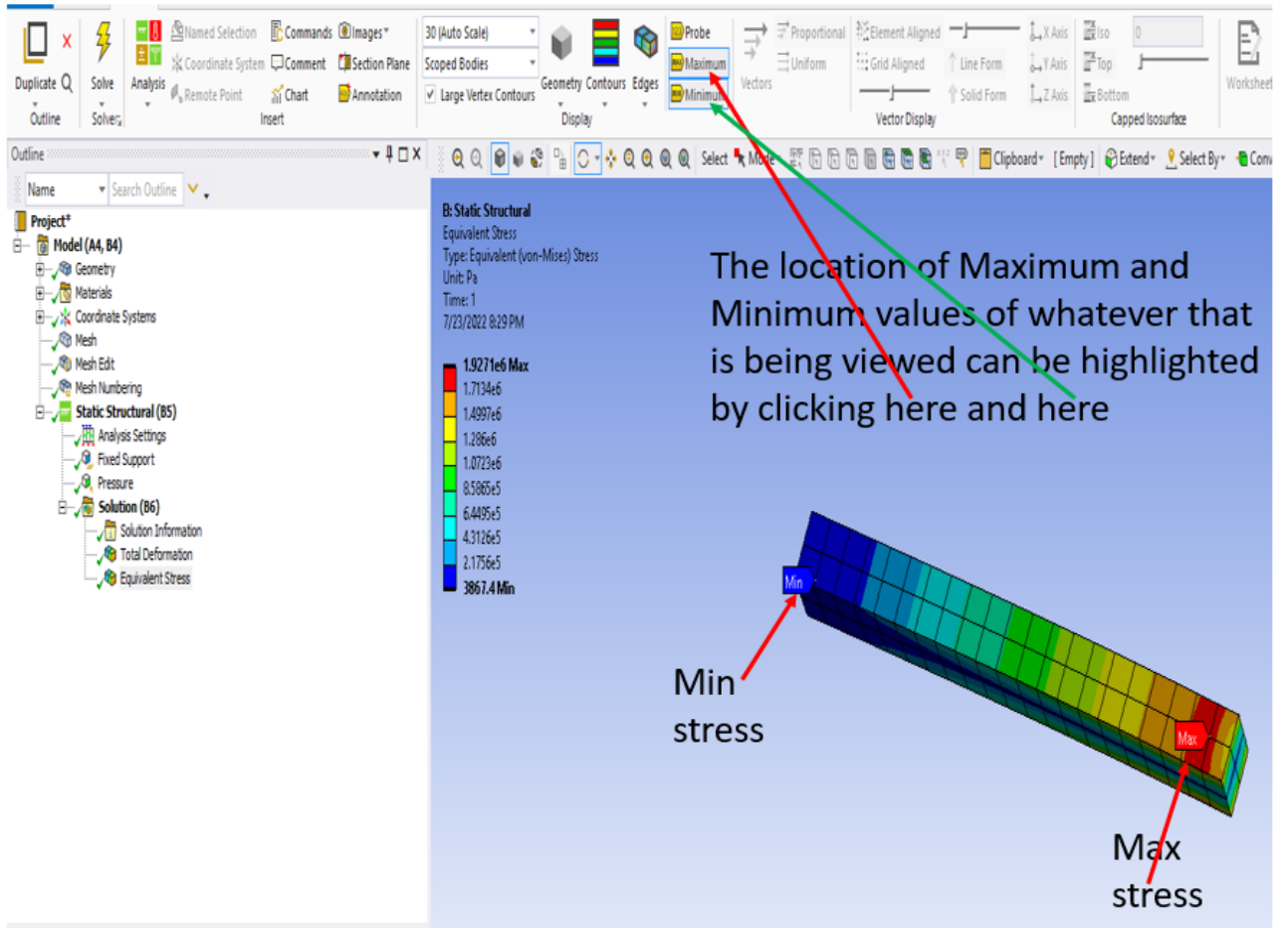




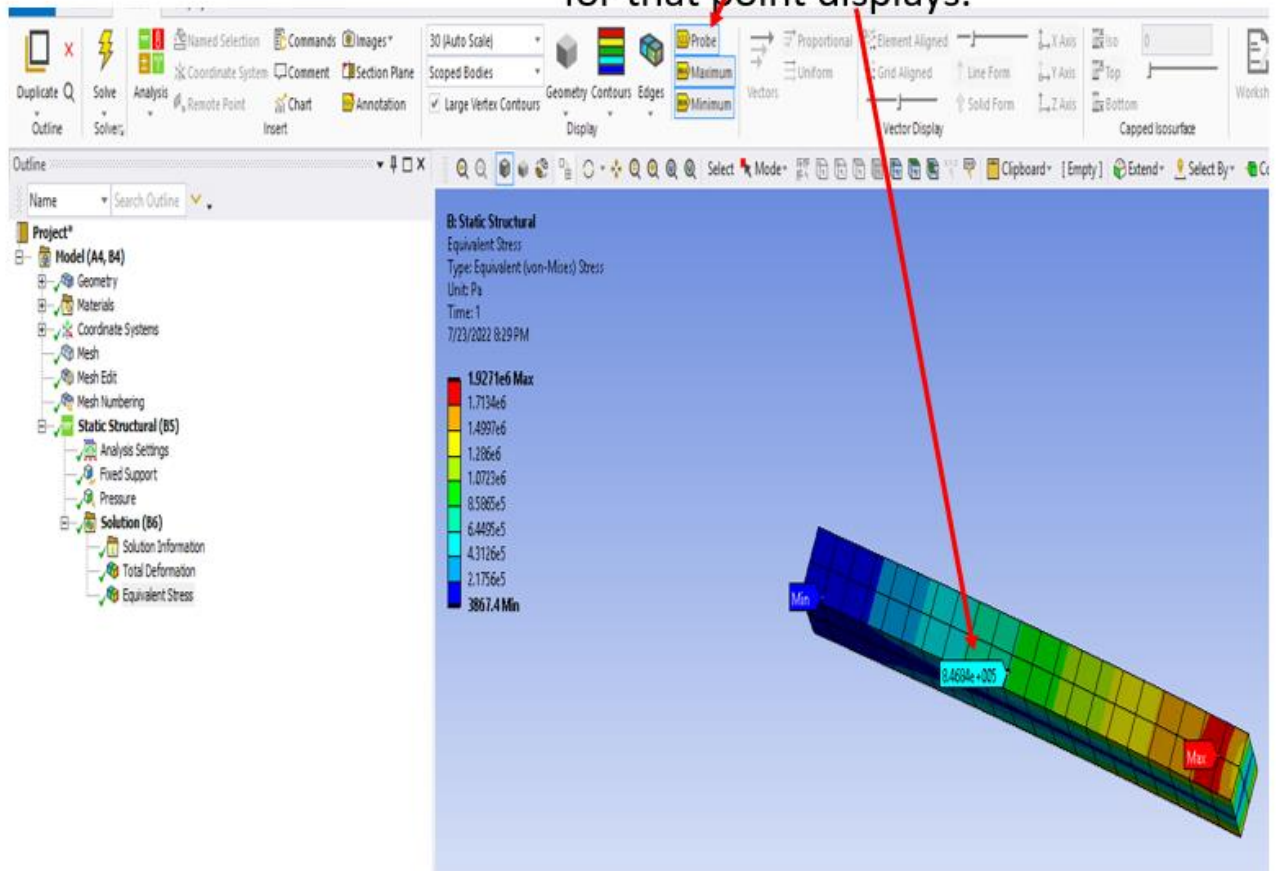


Animation can be performed by clicking here.

Animation can be stopped by clicking here

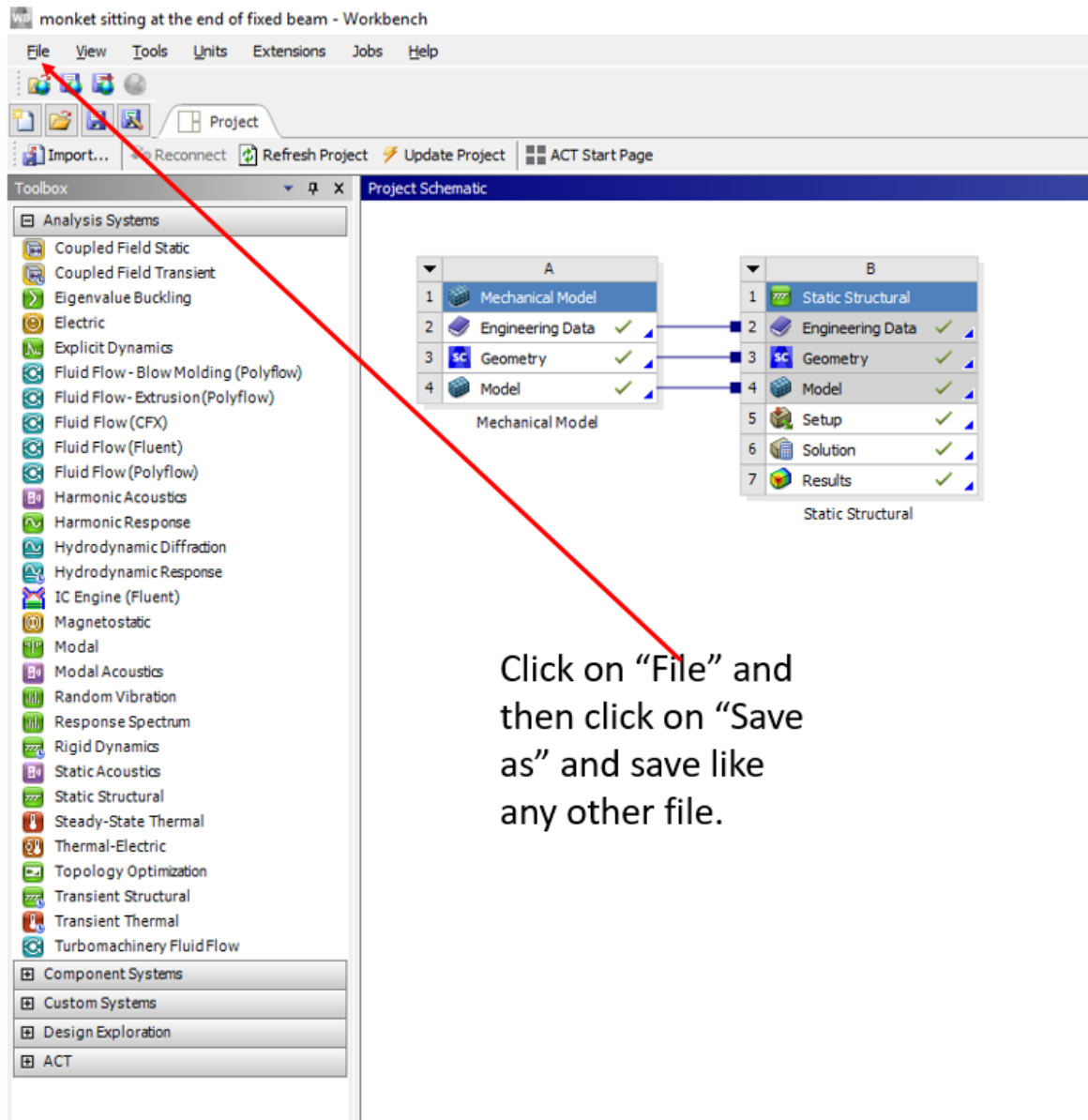


Click on “Probe” and then click at any point on the model. The result for that point displays.



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