

# Ansys Software Tutorial

## Introduction to Ansys Mechanical #1: Basics of FEA Simulation

Ashleigh Kirkland & Stacey Dyer

University of Newcastle

Edited by the Ansys Academic Development Team

[education@ansys.com](mailto:education@ansys.com)

## Summary

Ansys Mechanical is a finite element analysis (FEA) software used to perform structural analysis using advanced solver options, including linear dynamics, nonlinearities, thermal analysis, materials, composites, hydrodynamic, explicit, and more.

In this set of tutorials, we will introduce basic functionalities of Ansys Mechanical through the Ansys Workbench interface. Ansys Workbench is the integration and workflow platform that connects Ansys products.

This tutorial will cover how to create or import geometry, creating a basic mesh, applying boundary conditions and loads, solving your first FEA simulations, and changing the material of your model.

This tutorial is #1 of a four-part tutorial series that serves as an introduction to Ansys Mechanical. Details of the topics covered and the order can be found in the table below. These tutorials build on one another, so it is recommended that they are followed in order. Other tutorials can be found on the [Ansys Education Resources site](#).

<b>Tutorial Order</b>	<b>Tutorial Topic</b>
1	Basics of FEA
2	Bar Elements and Meshing
3	Truss Elements and Mesh Quality
4	Loading of Structures and Irregular Sections

\*This tutorial was created using the 2022R2 Student Version of Ansys Workbench. Some screens may look different, depending on your version. Check the [Ansys Learning Forum](#) if you have any questions.

## Table of Contents

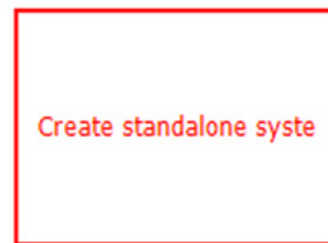
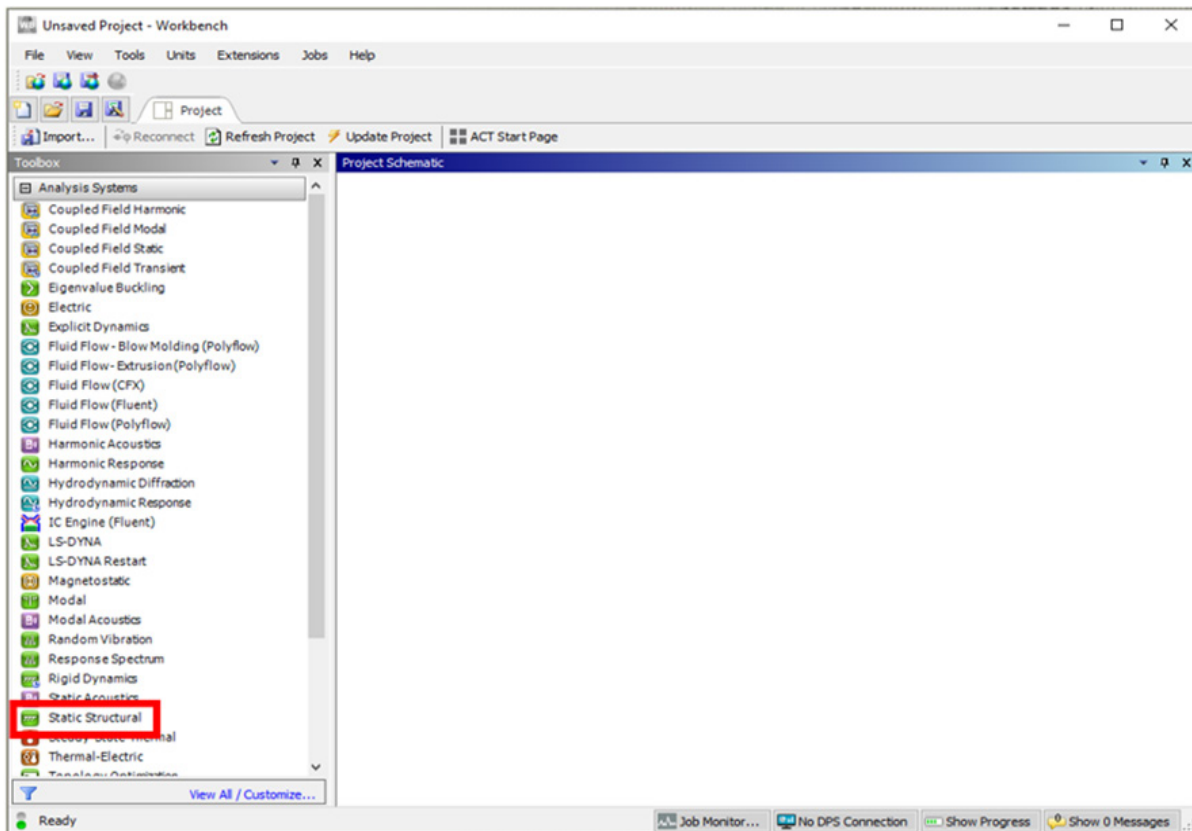
Starting a Project in Workbench.....	3
Exercise 1: Bending of a simply supported beam.....	5
Exercise 2: Bending of a simply supported beam (Aluminium Alloy).....	15

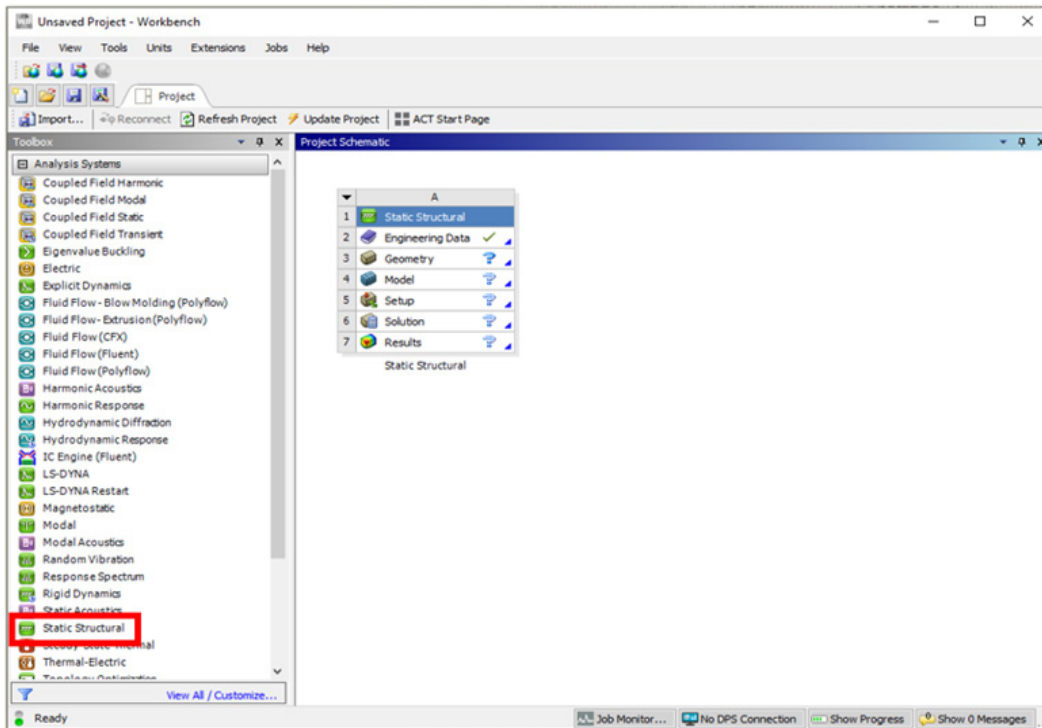
## Starting a Project in Workbench

Begin by opening Ansys Workbench.

On the left-hand side of the window in the “Analysis Systems” drop down menu you can see Ansys has a wide range of tools that can be used to analyze input geometry. For this exercise a static structural, finite element analysis will be used. To select “Static Structural” drag the list option into the workspace denoted by the green dotted lines – dragging the option into the space will cause the box to change into a message that reads “Create standalone system”.

On the left-hand side of the window in the “Analysis Systems” drop down menu you can see Ansys has a wide range of tools that can be used to analyze input geometry. For this exercise a static structural, finite element analysis will be used. To select “Static Structural” drag the list option into the workspace denoted by the green dotted lines – dragging the option into the space will cause the box to change into a message that reads “Create standalone system”.





To complete a static structural solve, you will need to provide the following:








1. Engineering data (a material allocation)
2. Geometry
3. Modeling information (a mesh dividing the geometry into a finite number of elements)
4. Setup information (such as boundary conditions and loads)

After solving, information such as stresses, displacements, elongations, *etc.* are provided by the program, both graphically and numerically

It is important to remember that the accuracy of the simulation is dependent on your understanding of the finite element method, including its limitations and applicability for each individual case that you have setup.

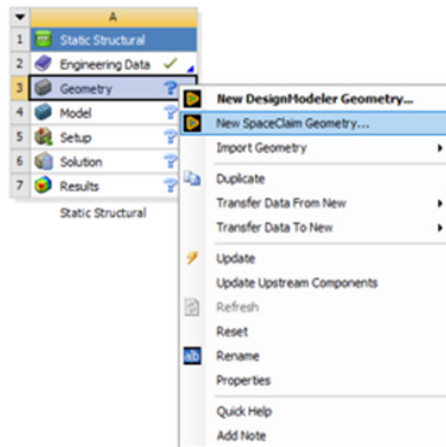
## Exercise 1: Bending of a simply supported beam

After dragging static structural into the workspace, each cell within it will contain one of the following symbols:

-  Filled question mark: data is required before continuing
-  Unfilled question mark: this section is unfilled, however upstream data is required before this can be resolved
-  Tick: the required minimum data has been input
-  Refresh: section requires review or modification
-  Lightning bolt: section requires an update due to changes in upstream data
-  Lightning bolt with pause: solution was interrupted and needs to be rerun
-  Tick with arrow: changes pending, project schematic may require further inputs

After dragging “Static Structural” into the workspace note that “Engineering Data” will show a tick, opposed to the other cells displaying a question mark. Ansys automatically selects structural steel for use in analysis. This can be changed but for this example we will leave the material as structural steel.

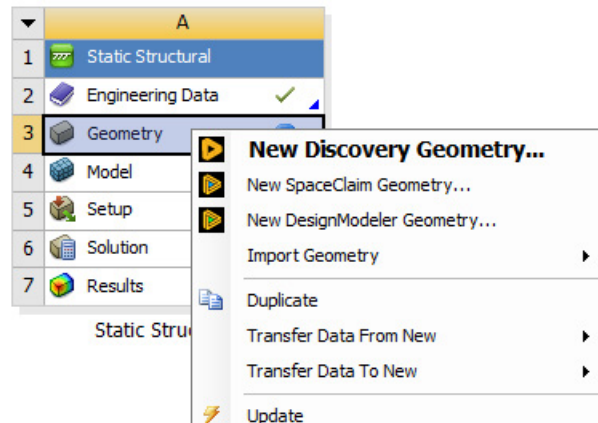
Geometry can either be made using one of the inbuilt programs or imported from an external CAD program. Right click on the “Geometry Cell” and select “New SpaceClaim Geometry”.



Ansys SpaceClaim is a non-parametric modeling program, which can be used to construct new or alter externally constructed models. Being a non-parametric program makes it quick and easy to adjust complex geometry.

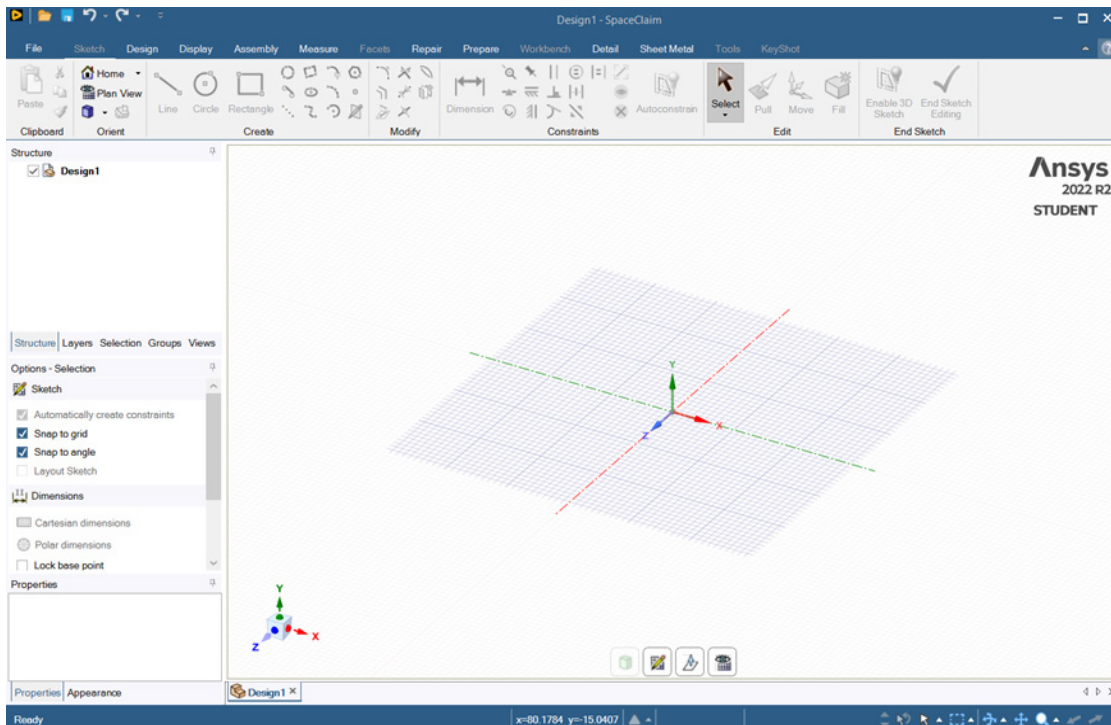
**NOTE:**

This tutorial was made using the 22R2 Student Version of Ansys Workbench. After the 2023R1 release, Ansys SpaceClaim became a legacy product and Ansys Discovery (a simulation-driven design tool that combines instant physics simulation, high fidelity simulation and interactive geometry modeling in a single easy-to-use experience) became the primary built-in Geometry tool. An example of what the screen will look like if using 23R1 or later is shown below.



If you want to learn more about specifically modeling in Ansys Discovery, check out this Ansys Innovation Course [“Learn Solid Modeling with Ansys Discovery”](#).

After navigating to the SpaceClaim window, select the sketch plane you wish to work in by clicking on the corresponding arrow in the bottom left corner. Alternatively, you can select the arrow located on the coordinate system in the center of the window.

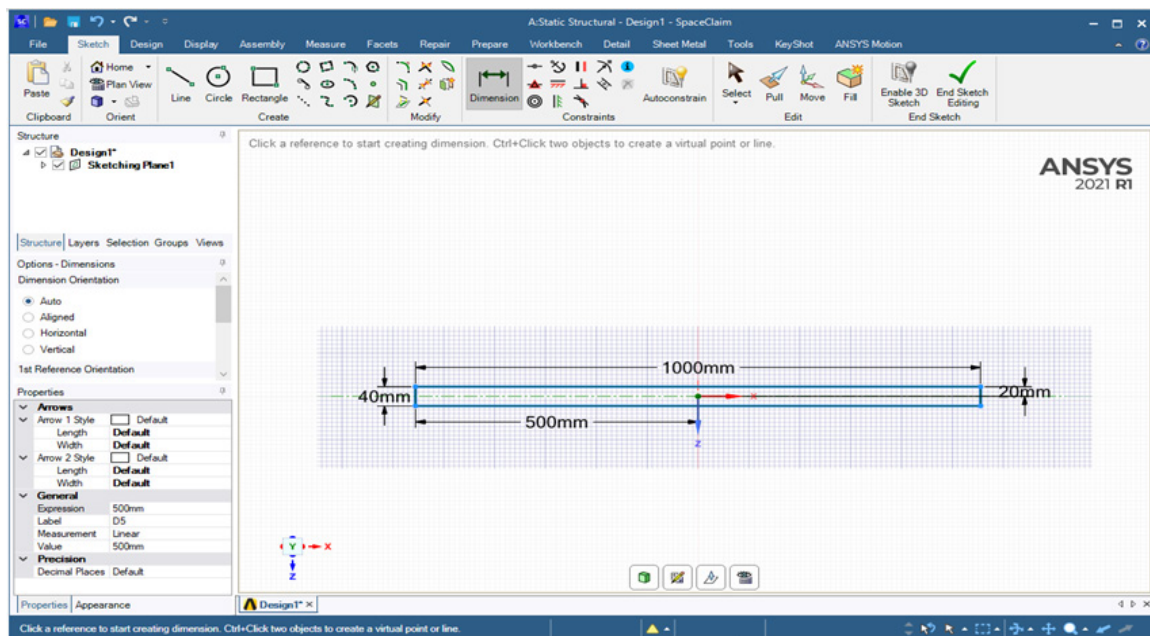


For this example, create a sketch on the Y plane by clicking on the green arrow beside “Y” on the coordinate system.

Select the “Rectangle” tool from the sketch tab.

Draw a rectangle with the dimensions shown below. The dimension tool can be used by clicking on the line of interest, typing the required dimensions in, and pressing enter.

To create dimensions between the coordinate system or between two lines, hold Ctrl and select the line/s of interest. Selecting non-parallel lines will allow you to dictate the angle that is made between them.



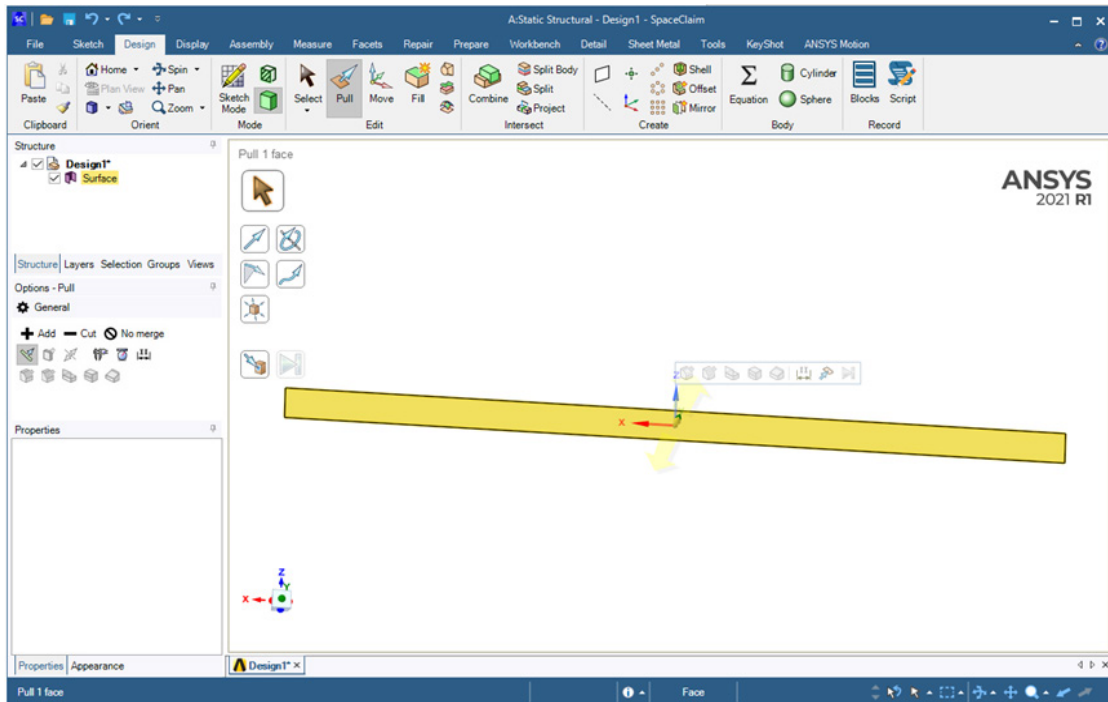
Select the rectangle by drawing a box around it using the “Select” tool (located on the right hand side of the sketch tab).

After the rectangle is selected press the “Fill” button (located to the right of select). This will convert the selection of lines into a surface once the sketch is complete.

Once you have completed the sketch you will need to click the “End Sketch Editing” button. This will take you to the design page/menu.

You can rotate the sketch/part in the window by holding the mouse scroll button down and moving the mouse. Use this to orient the rectangle in such a way where the Y coordinate at the center of the rectangle can be seen.

Select the “Pull” tool from the design menu and hover over the Y-coordinate (green arrow) at the center of the rectangle until a yellow arrow appears over it as shown below.



You can now drag the mouse upwards to extrude the surface into a solid. Type in 40 and press enter to create a beam that is 1000 mm x 40 mm x 40mm.

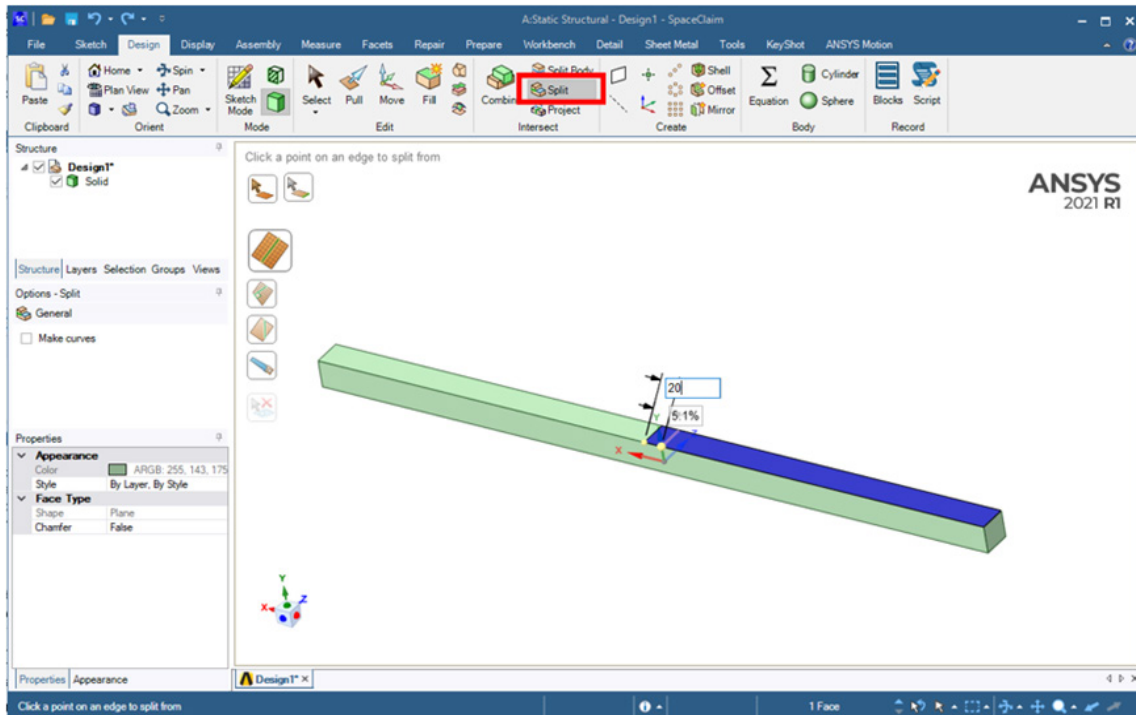
You can see in the structure tree on the left-hand side SpaceClaim that the surface has become a solid.

The dimensions of the beam can be checked by selecting the measure tool from the measure tab at the top of the page and selecting the dimension of interest.

For this example, we are required to load the beam in the center. Ansys does not allow for a “point loading” scenario, where loads can be applied through a single point (as this is not realistic). To account for this we will split one face to enable loading across an area of the geometry.

Use the “Split” tool from the intersect menu under the design tab. Select the top face of the beam, the selected face will turn blue.





Run the mouse along the edge of the face and split the face at 490 mm.

The percentage is automatically selected, allowing you to split the face as a ratio of its length. Pressing tab will change the selection to the dimension, allowing you to type the required dimension before pressing enter.

Perform a second split at the other end of the beam, so that a 20 mm section of the beam has been split along one face.

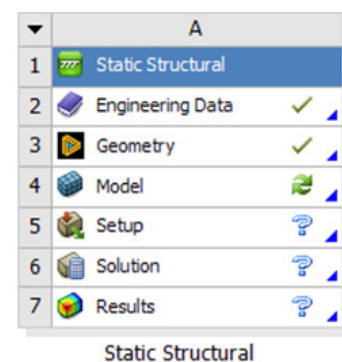
Use the “Measure” tool to ensure that your beam is split on one face into two sections of 490 mm x 40 mm, with a piece of 20 mm x 40 mm at the center.

Upon completion of splitting your beam, close SpaceClaim. Saving is not required unless you wish to reuse this geometry in future projects. Ansys will import the geometry into Workbench automatically.

A green tick should now be seen in the geometry cell, shown to the right.

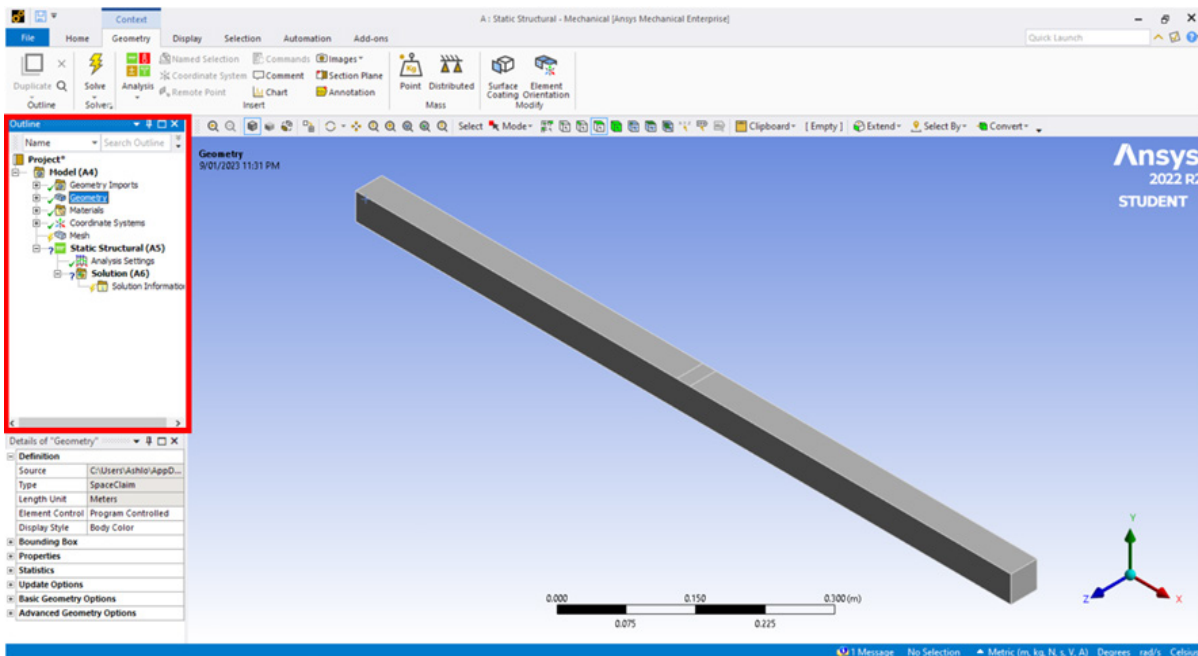
Double click on “Model”, this will open Ansys Mechanical and allow you to progress to the first stage of the simulation setup.

The basic controls for Ansys Mechanical are the same as SpaceClaim; you can zoom in and out using the mouse scroll wheel, rotate by pressing the scroll wheel and moving the mouse, and you can pan by holding Ctrl, while pressing the scroll wheel and moving the mouse.

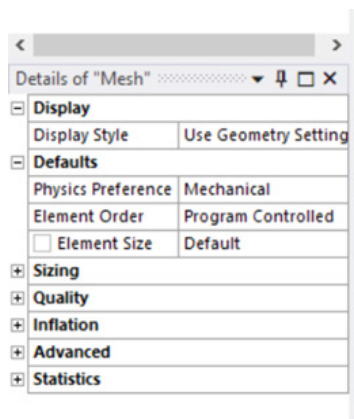


The next step is to create a mesh for the beam.

If opening Ansys mechanical for the first time, the window may open with an instructional worksheet. To navigate out of this and to view the geometry imported select “Geometry” in the project tree on the left-hand side of the window.

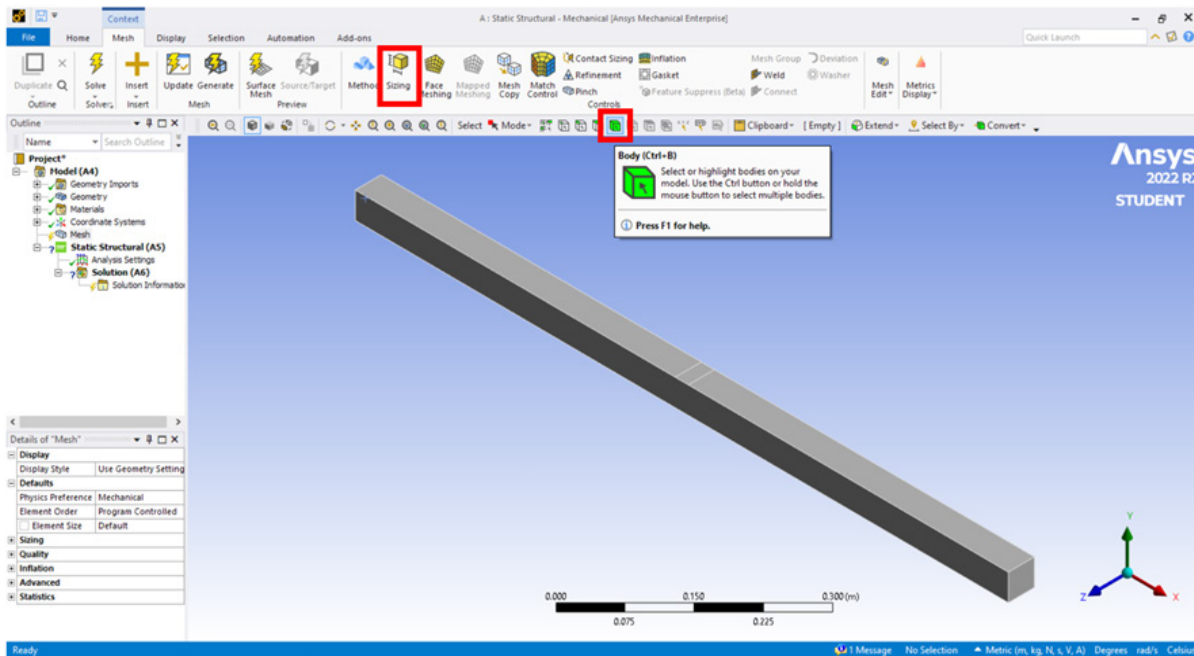


In the “Outline” window on the left-hand side, click on “Mesh”. This will open the meshing options in the “Details” menu beneath the “Outline” window.



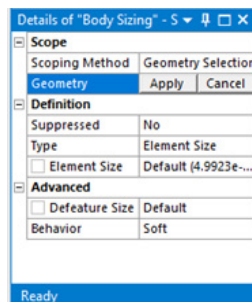
Create a mesh sizing feature by selecting “Sizing” from the “Mesh” tab at the top of the program menu.

After selecting the sizing feature the “Geometry” and “Type” cells will become highlighted in the details window.



Select the body selection tool and click on the beam.

Within the details window, select the highlighted cell next to Geometry. If the body has been selected correctly, the “No Selection” message should change to an “Apply” or “Cancel” option. Select apply.



This will dictate to Ansys that the selected body is the one that the sizing details will be applied to.

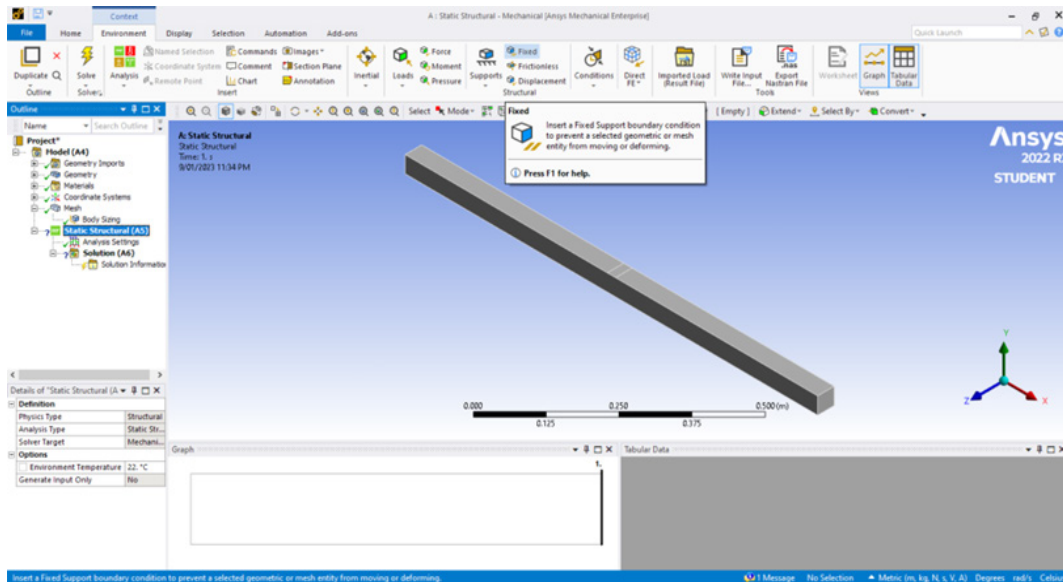
In the cell next to “Element Size” create a 10 mm sizing by enter 0.01 (as the units are in meters) and hitting enter.

Press the “Generate” button in the mesh menu at the top of the screen to implement the sizing.

After generating a mesh, supports and a load are required before we can solve the simulation.

Click on “Static Structural” in the project tree to change the context menu.

Select “Fixed” from the supports menu at the top of the page.



Rotate the geometry and click one of the square ends of the beam. Within the details menu click apply to allocate the constraint to the end of the beam. Repeat this process for the other end of the beam. We now have a simply supported beam. The load is applied in the same fashion as the supports.

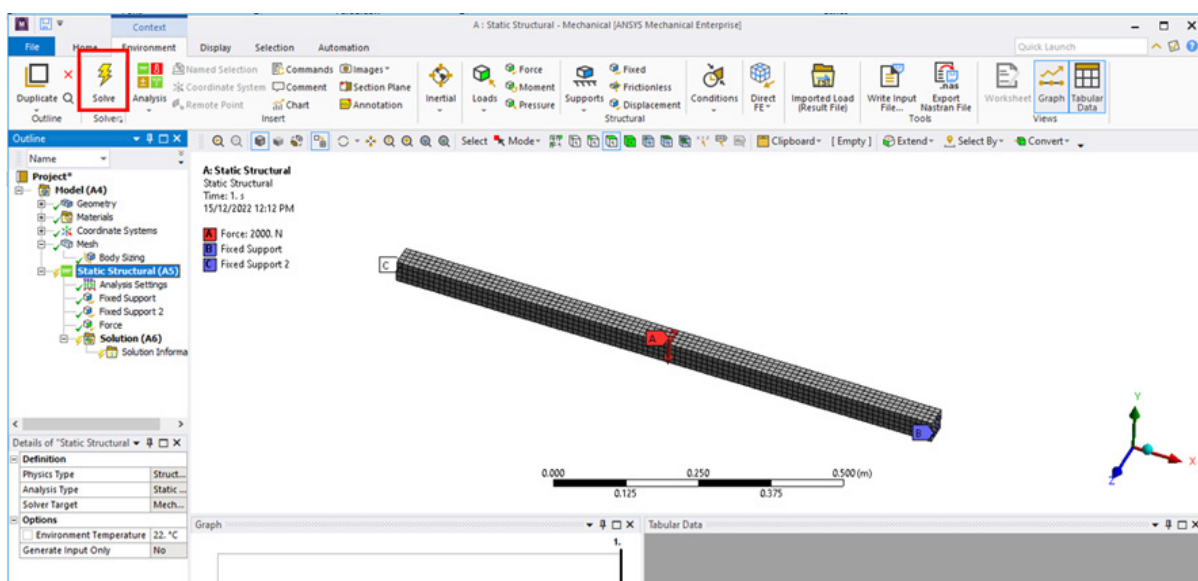
Rotate the beam so the split face is facing up.

From the “Loads” menu at the top of the page, select “Force” and click on the split made in SpaceClaim.

Forces can be applied as a vector or in constituent components. In the details window of the force click on the “Define By” drop down and select “Components”.

Beside the Y-component, apply a 2 kN force acting downwards by entering -2000 and hitting enter. You now have everything required to solve your first FEA simulation.

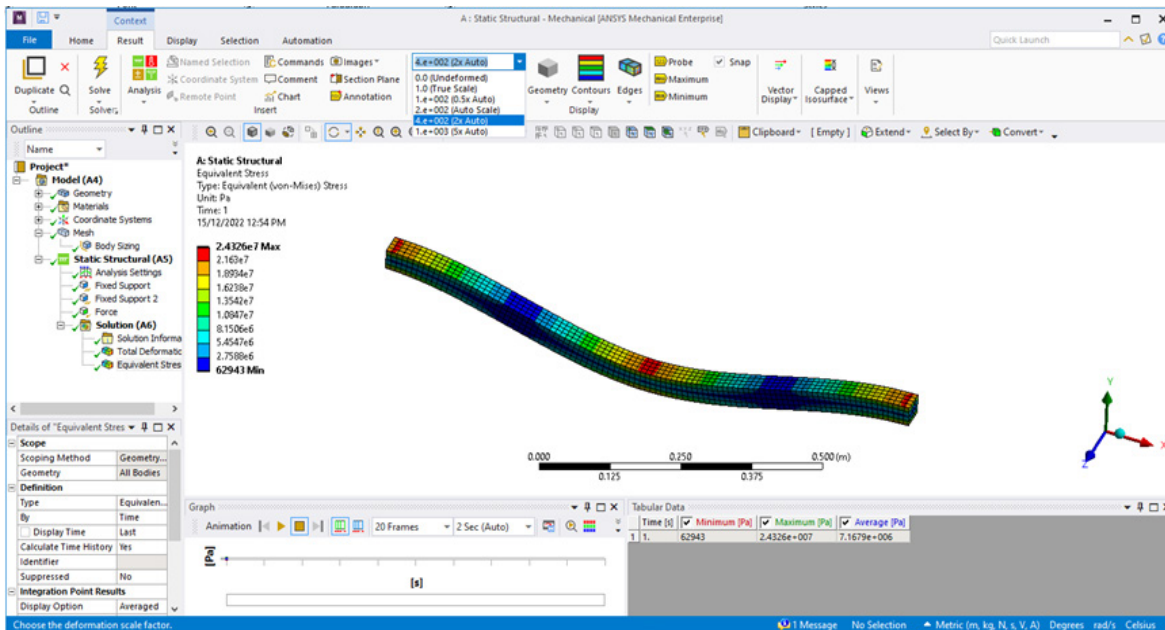
Click the “Solve” button under the Environment tab.



Click on the “Solution” drop down under the Outline tree.

This will change the tab (Context menu) at the top of the program. Here you will find the analysis options provided by the simulation.

Select “Total” from the deformation drop down and “Equivalent (Von-Mises)” from the stress drop down. Press the “Solve” button again to populate these analyses.



Note that the deflection of the beam is exaggerated. You can change this by selecting the drop down depicted and changing it to show a greater/lesser exaggeration or selecting “True Scale” to show the precise amount of deflection displayed on the beam.

We see the maximum stresses are seen at the center of the beam, and at the ends near the fixed supports.

The contour map shows that the maximum stress seen by the beam is 24.3 MPa. As we know this is well beneath the yield strength of structural steel, no permanent deformation will occur during bending of the beam. By comparing maximum stresses to material moduli, we can check the safety of designed structures using FEA.

**Always check to ensure you simulation results intuitively make sense:**

- Is the maximum stress where you would expect it to be?
- Is the magnitude in the right range?

In this case we would expect the stresses experienced to be low, as the bar has a 40 mm x 40 mm solid cross section with a load applied of approximately 200 kg. We see the deflection is also appropriately low, with a maximum deflection of 0.25 mm seen at the center of the beam.

The unit system is displayed at the bottom, right of the program. These can easily be changed by clicking on them.

A variety of unit systems are displayed. For the purposes of most FEA simulations, it is best to work with mm. This will change the units of stress to MPa.

Closing Ansys mechanical will result in the simulation analysis being saved to your Workbench project.

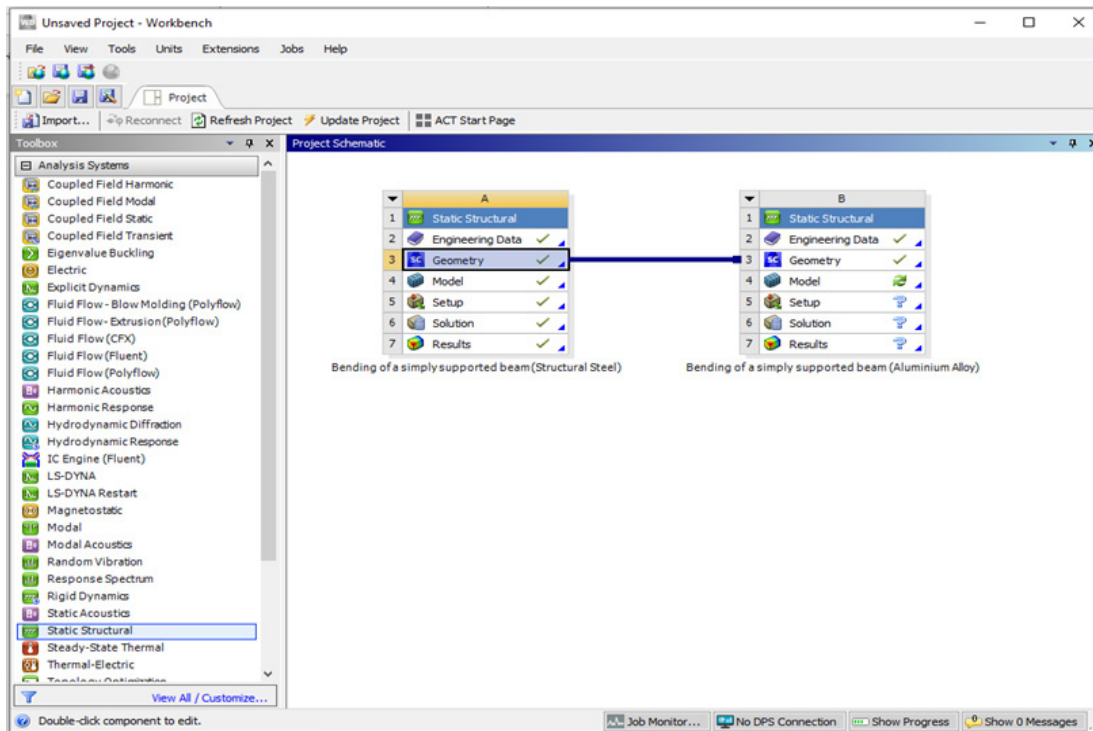
## Exercise 2: Bending of a simply supported beam (Aluminum Alloy)

The second exercise in this lab is to repeat exercise 1 using a different material model and shared data.

Drag a second Static Structural analysis into your Workbench space, into the “Create standalone system” red box.

Within Ansys it is possible to couple systems or share information between standalone systems by dragging components onto each other.

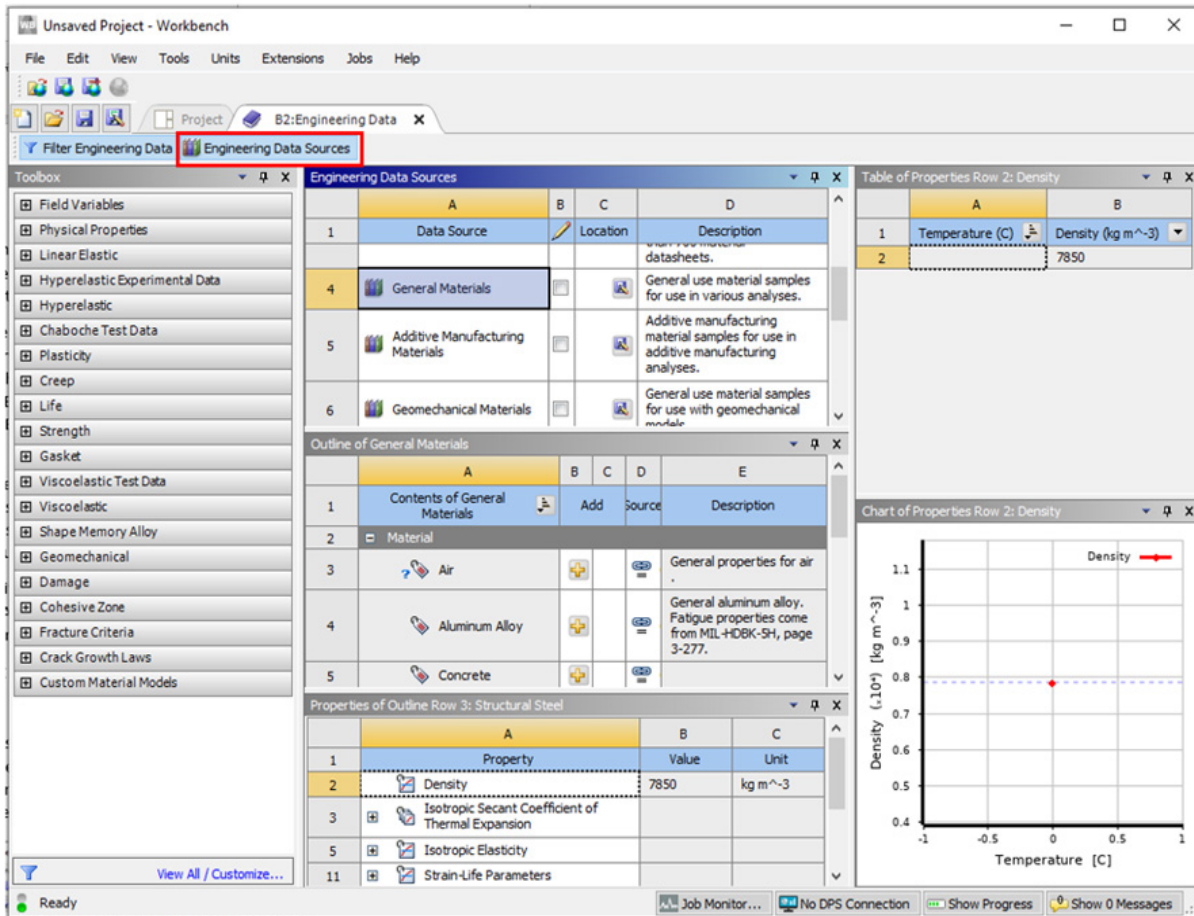
Click and drag the geometry cell from the first system onto the second; a line will join the two geometries, indicating that they are now shared.




Before solving, the material for the beam needs to be changed.



Double click on the “Engineering Data” cell in the second system, this will open the engineering data tab.

Select the “Engineering Data Sources” tab, this will open the library of engineering materials stored within Ansys.



Under the “Engineering Data Sources” window, click on “General Materials”.

The window beneath will display the library of general materials. Structural steel is automatically selected, scroll up until you see “Aluminum Alloy” and press the  button beside it. This will add the material to the list of materials that can be applied in the simulation.

All materials that are available for use in simulations will have a  symbol beside the  symbol. Materials can also be added to the library if material properties such as elastic modulus, Poisson’s ratio, etc. are known.

Close the “B2: Engineering Data” tab or select the “Project” tab to return to Workbench.

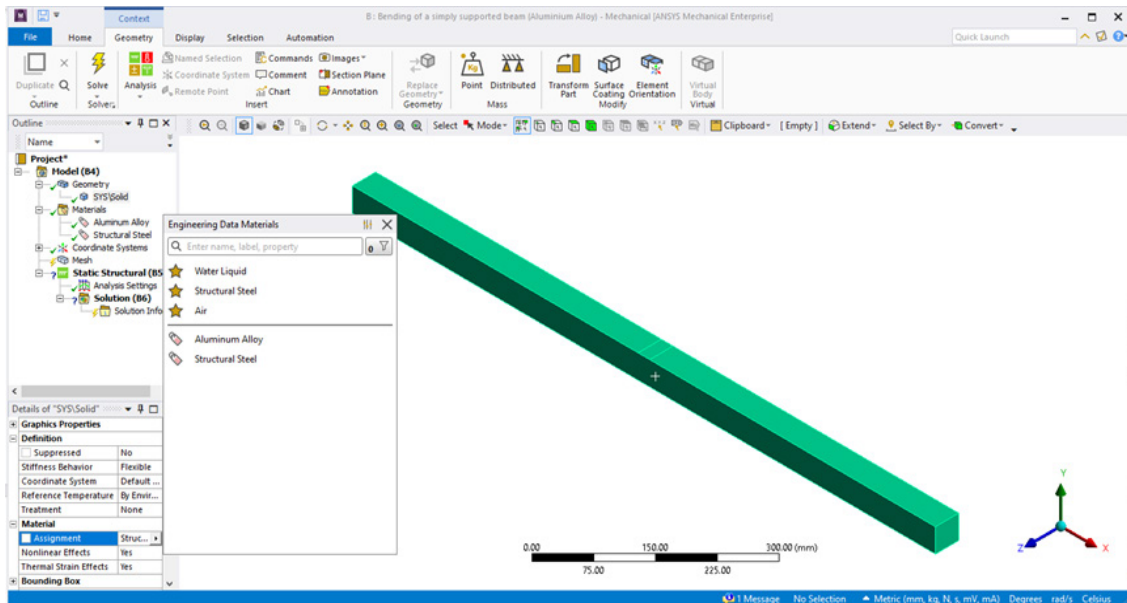
Open Ansys Mechanical by double clicking on the “Model” cell in the second system.

Click on the + beside “Geometry” to open the drop down menu and select “SYS/Solid”.

Under the Details tree, click on the + beside “Material”, making the material properties alterable.

In the box beside “Assignment” click on the drop down, you will see that Structural Steel is currently selected. Change this to the aluminum alloy that was added in the previous step.





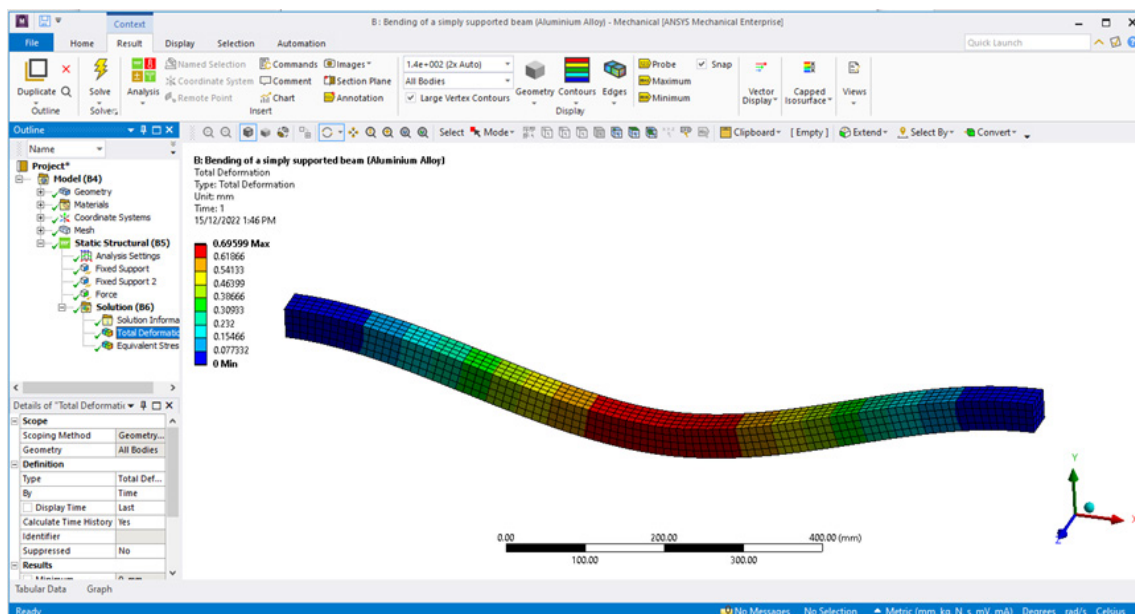
As per the first exercise, implement a 10 mm mesh sizing, before supporting and loading the beam as before.

NOTE: If you changed the unit system to be in mm you will now enter 10 as the body sizing rather than 0.01.

Add a “Total” deformation and a “Equivalent (Von-Mises)” stress as before, so that we can compare the two models of different materials.

As is to be expected, the stresses seen in both beams are the same (as the same loading and support conditions are applied beams of the same size).

The change in material properties implies that the deformation will be different. For the aluminum beam we see a total deflection of 0.7 mm.



© 2024 ANSYS, Inc. All rights reserved.

## Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

## Document Information

This case study is part of a set of teaching resources to help introduce students to topics focused on structures and structural simulations.

## Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, MicroProjects, real life examples and more, visit [www.ansys.com/education-resources](http://www.ansys.com/education-resources).

## Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at [education@ansys.com](mailto:education@ansys.com).

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit [www.ansys.com](http://www.ansys.com) for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.