

Ansys Software Tutorial

Introduction to Ansys Mechanical #3: Truss Elements and Mesh Quality

Ashleigh Kirkland

University of Newcastle

Edited by the Ansys Academic Development Team

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 creating and analyzing truss structures using Ansys Mechanical, as well as reviewing the effects of mesh quality and sizing on the results of a simulation.

This tutorial is #3 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](#).

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

NOTE: This lab follows a more question-based approach, under the assumption that students have completed the first two Intro to Ansys Mechanical tutorials. Documentation will include detail around how to apply new techniques within Ansys Mechanical, however previously used techniques and methods are assumed knowledge. Revisit the first two tutorials from the table above as needed.

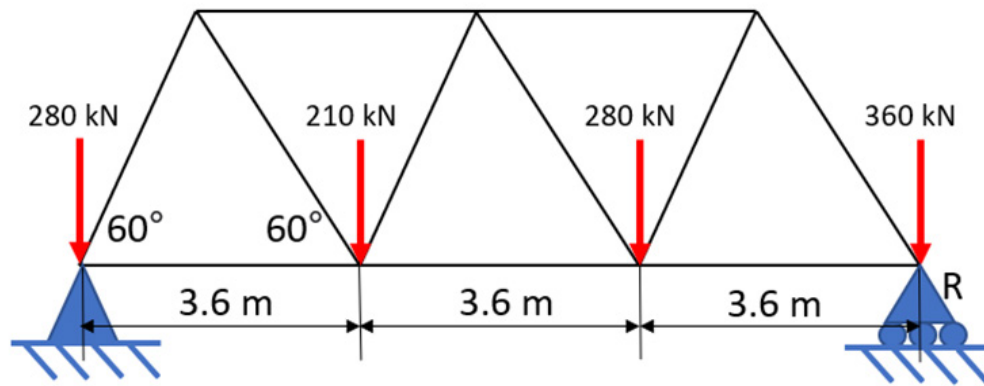
*This tutorial was created using the 2022R2 Student Version of Ansys Workbench. Some screens may look different, depending on your version. Check the [Ansys Learning Forum](#) if you have any questions.

Table of Contents

Exercise 1: Loaded Railway Bridge.....	3
Exercise 2: Analysis of Mesh Quality	12
Applying an Inflation Layer	13
Applying a Sphere of Influence.....	14

Exercise 1: Loaded Railway Bridge

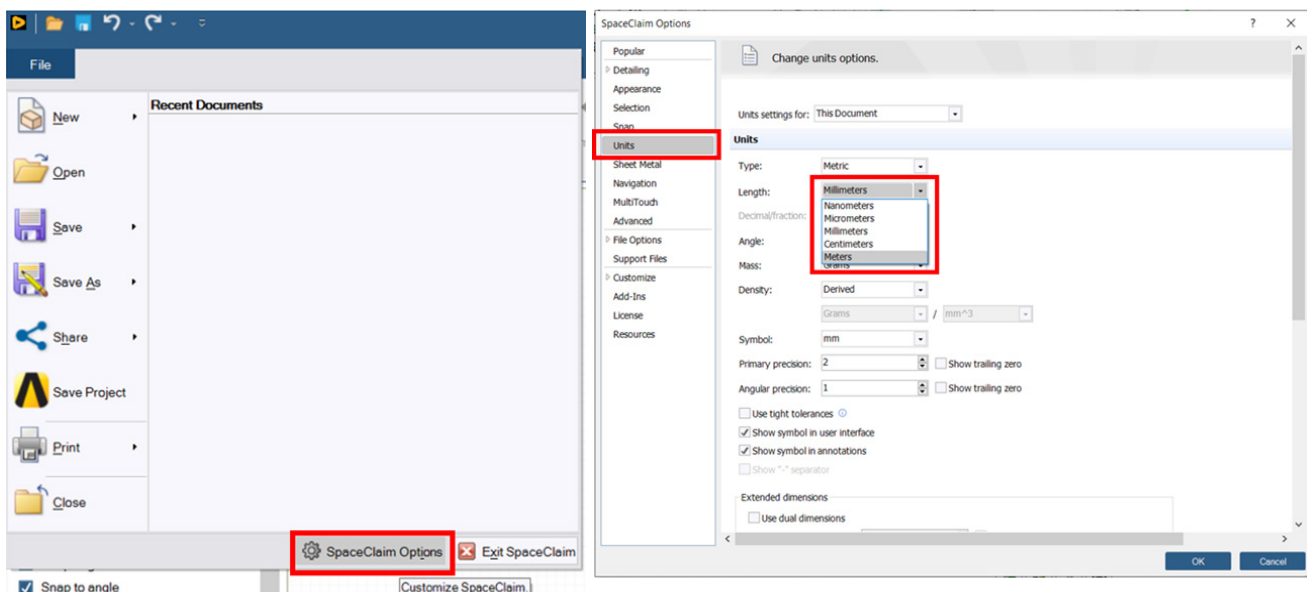
The figure below shows a small railway bridge constructed from round steel members, all of which have a radius of 40 mm. A train stops on the bridge and applies the loads shown in the figure to one side of the truss. A train stops on the bridge and applies the loads shown in the figure to one side of the truss.



Determine how far point R moves as well as the displacements and stress in each element.

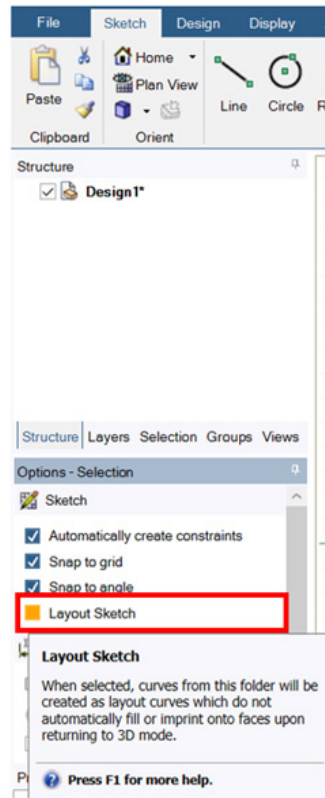
Start by creating a sketch in SpaceClaim¹.

Hint: it may be easier to change the units in SpaceClaim to meters when dealing with large geometry. This can be done by navigating to File – SpaceClaim Options – Meters



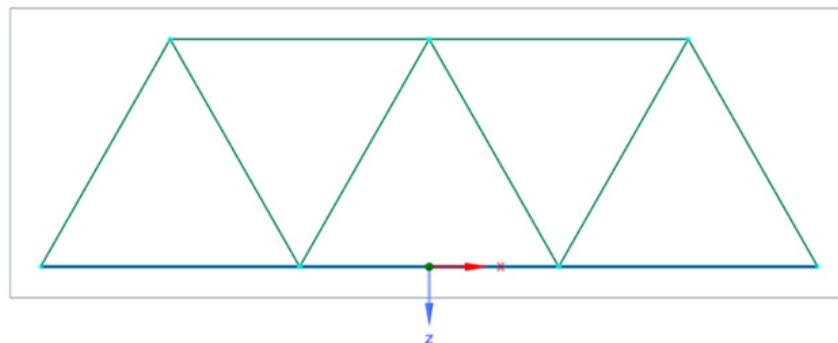
¹ This tutorial was made using the 22R2 Student Version of Ansys Workbench. After the 2023R1 release, Ansys SpaceClaim became a legacy product and Ansys Discovery (a simulation-driven design tool that combines instant physics simulation, high fidelity simulation and interactive geometry modeling in a single easy-to-use experience) became the primary built-in Geometry tool. If you want to learn more about specifically modeling in Ansys Discovery, check out this Ansys Innovation Course [“Learn Solid Modeling with Ansys Discovery”](#).

To start drawing a truss make sure that Layout Sketch is selected. This lets SpaceClaim know to not assign the completed shape as a surface for extrusion. This option will need to be selected for all exercises where a frame is used.

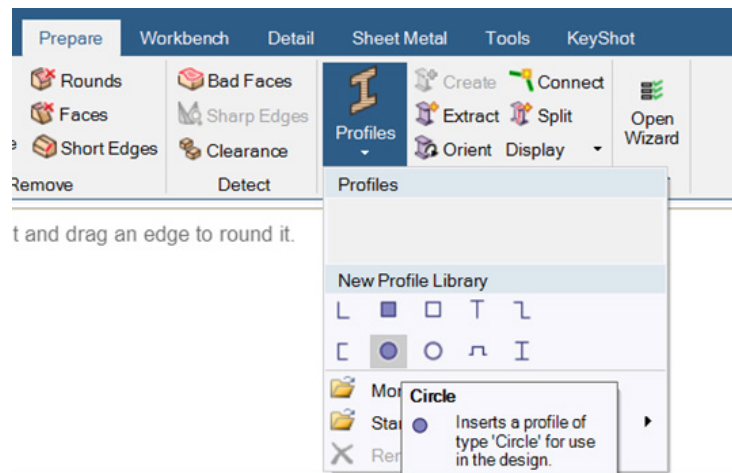


If you complete any lines of the truss before Layout Sketch is selected, the line will be allocated as to be part of a surface. You will need to go back and replace these lines as needed.

After completing the truss layout select complete sketch, the completed layout should appear similar to the image below. If the triangles appear shaded, check that all lines were drawn with layout sketch selected.

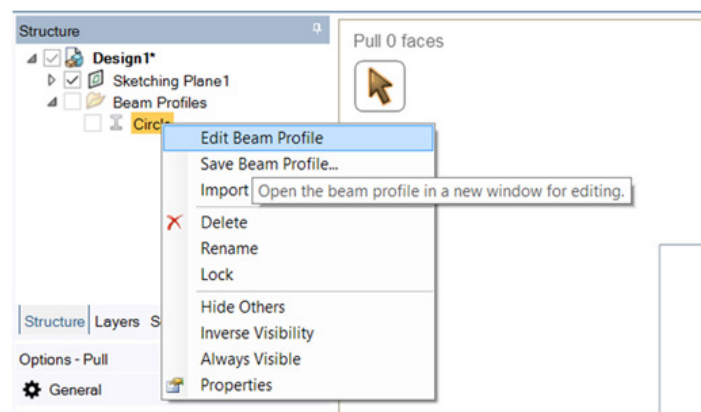


Select the relevant cross section for the truss by navigating to Prepare – Profiles – Circle.

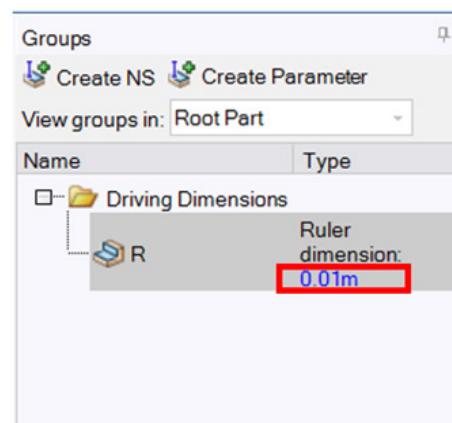


The beam profile should then appear in the Structure window on the left side of the screen.

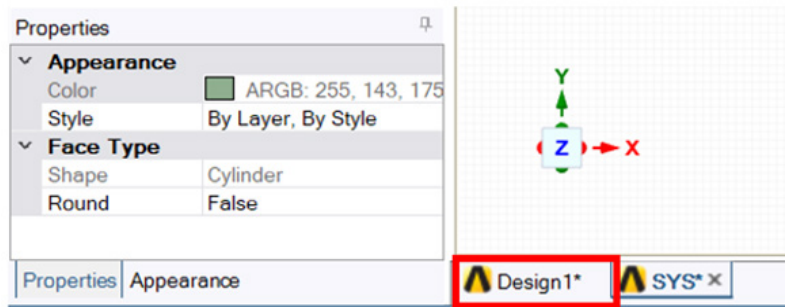
Edit the beam profile to set a radius of 40mm.



To change the dimensions of a predetermined beam profile, the dimension for each feature can be seen in the groups window, under driving dimensions. The dimensions can be changed by double clicking on the highlighted existing dimension as shown below.

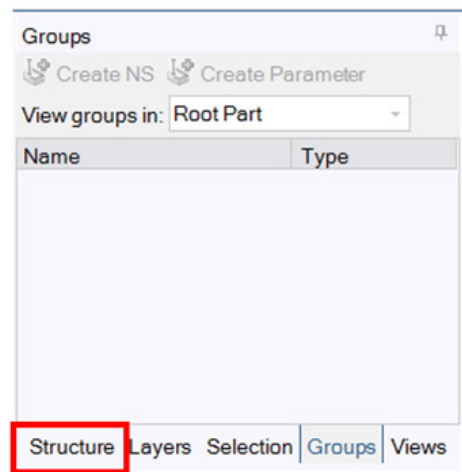


After completing the cross section, navigate back to the main window via the tabs at the bottom of the window.

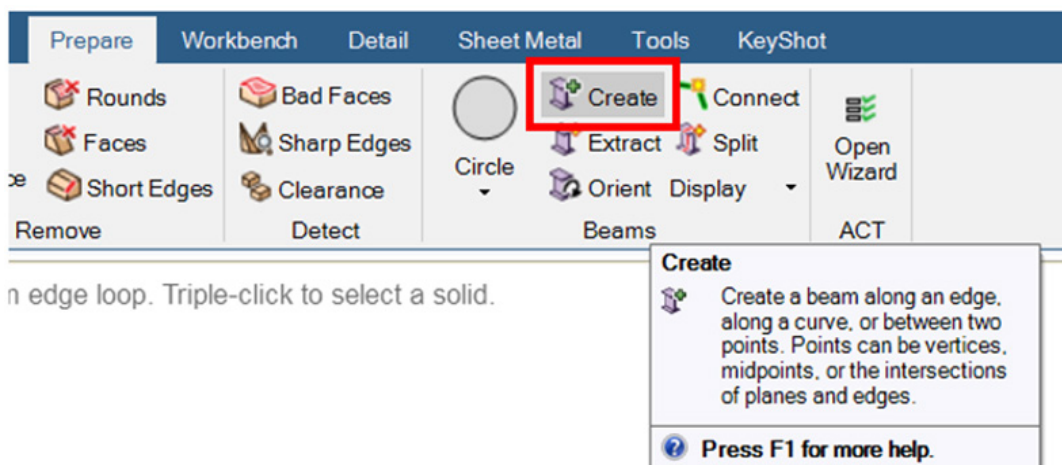


The edited cross section will appear under the tab SYS, the geometry will appear under the file name given to the project or Design1 if this has not been previously set.

When navigating back to the main design window, the groups tab will still be visible to the left of the window. To navigate back to the structure window, select structure from the list of tabs underneath the panel.



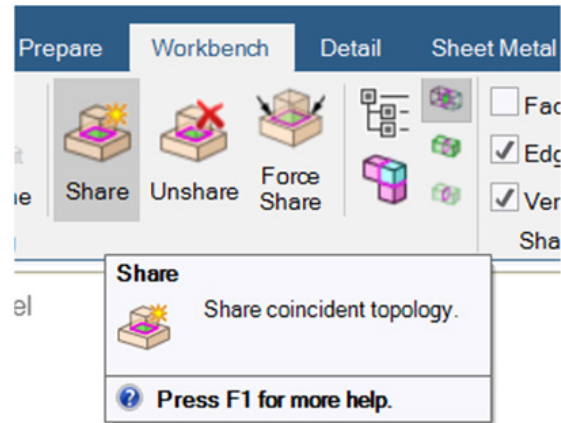
To apply the section to the truss layout, select the sketch by clicking and dragging a selection box over each of the proposed beams, followed by clicking create.



If this operation was successful, the truss should appear to be highlighted green.

The final step in SpaceClaim is to indicate to Ansys which connections between the beams are shared. This will ensure that the truss behaves as a cohesive structure as opposed to separate elements.

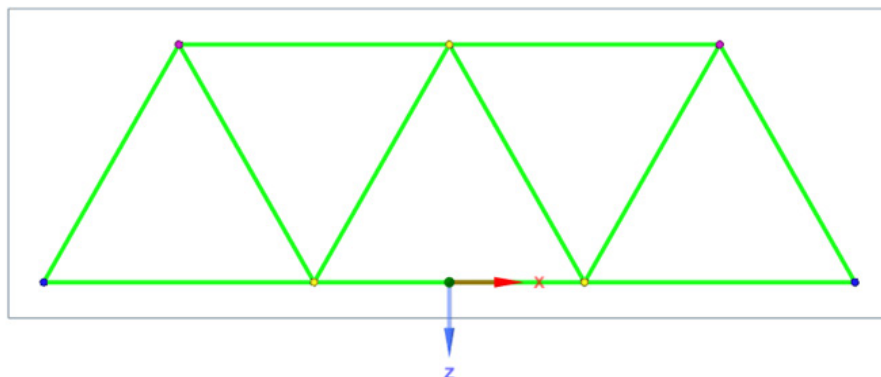
To do this navigate to the workbench tab and select the “Share” operation.



The common nodes between the beam elements will then appear highlighted.

In the top left of the working window a green tick will appear, clicking this will confirm the selection and share the highlighted nodes.

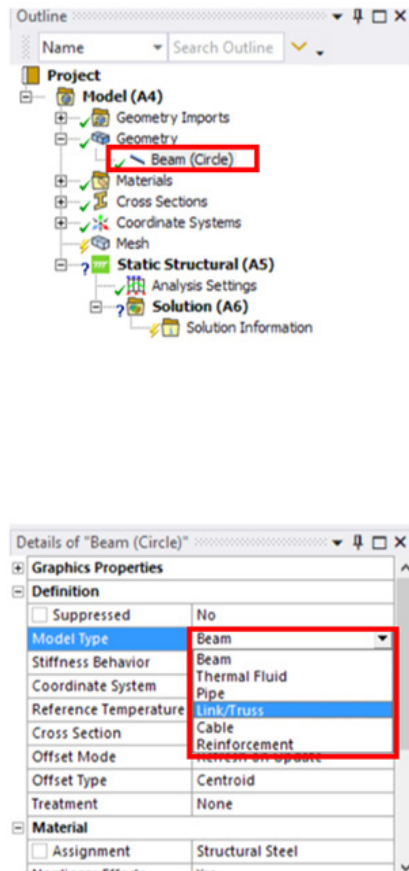
The completed geometry should then appear as below.



The geometry for this exercise is now complete, you may close SpaceClaim.

Open Ansys Mechanical.

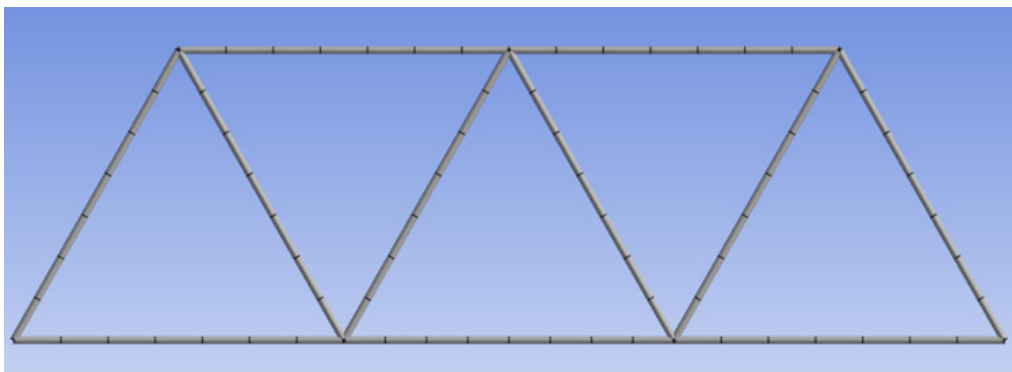
To allow the program to recognize this geometry as a truss, the model type must be changed in the outline window. This is done by selecting the beam element under the geometry drop-down box and changing the model type to truss.



NOTE: A truss system can only carry axial forces, where as a beam system can carry axial, shear and bending forces. For this exercise we only wish to look at axial forces.

The next step in this method is to generate the mesh.

If you generate a default mesh, you will notice that the beams will be broken up into smaller sections as per the image below.

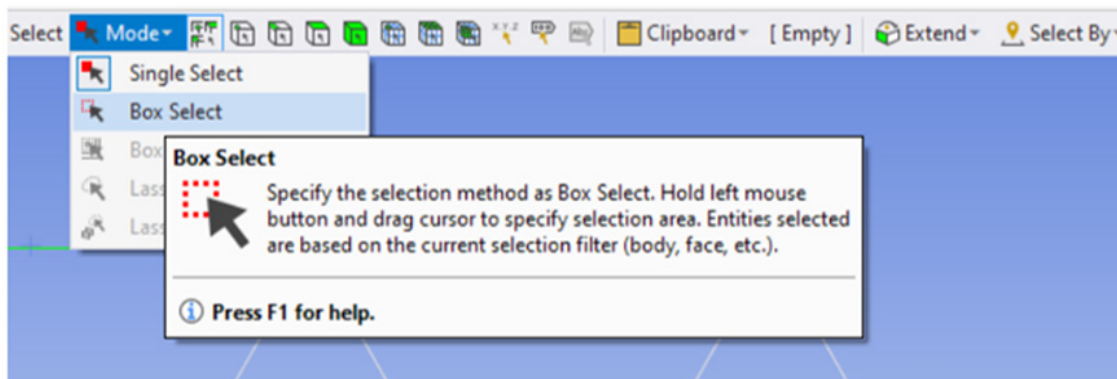


This will not solve for a truss analysis as it will be impossible to solve for equilibrium in the connections between the elements that are not located at the connection points between the beams.

To fix this each strut within the truss will need to be made of a single element.

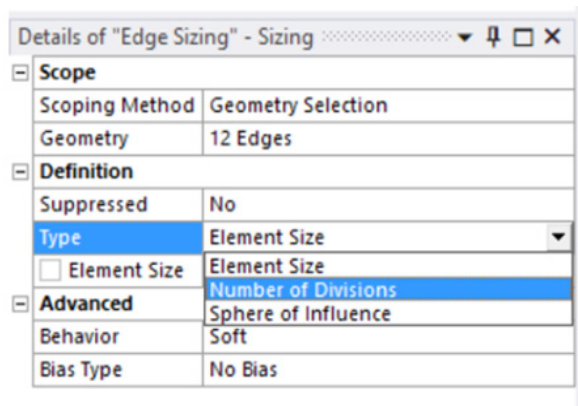
Apply a sizing constraint to the mesh.

Select the entire truss by using the box selection tool or another method of your own choosing and apply the sizing constraint.



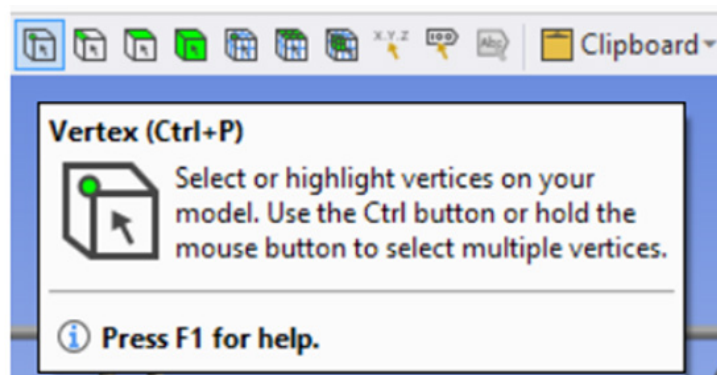
Ansys classifies struts within a truss as edges. After selecting box select ensure that the edge tool is selected.

Change the type of the sizing constraint to “Number of Divisions”. Set the number of divisions per edge to 1.

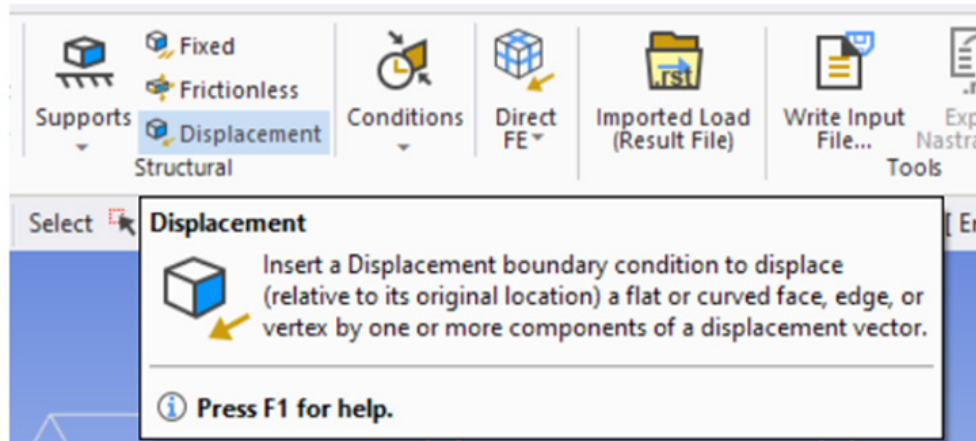


Regenerating the mesh will create vertexes at the joins between the struts, and single elements along the length of the beams.

To apply loads and conditions to these vertexes (or nodes), switch from edge selection to vertex selection and add the required boundary conditions and loads as per the image at the top of the document.



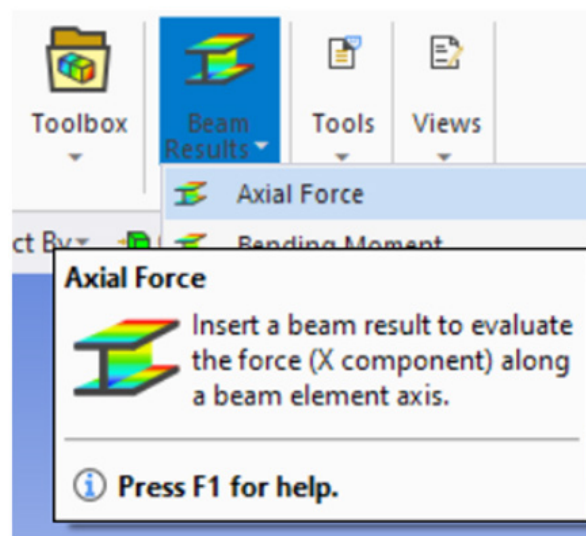
NOTE: To apply a roller joint, select the displacement option from structural panel under fixed support. You will need to adjust the condition depending on the coordinate system you are using for the geometry. The roller should be unable to leave the ground plane but should be free in all other directions. To stop movement in a specific direction, set this component to 0.



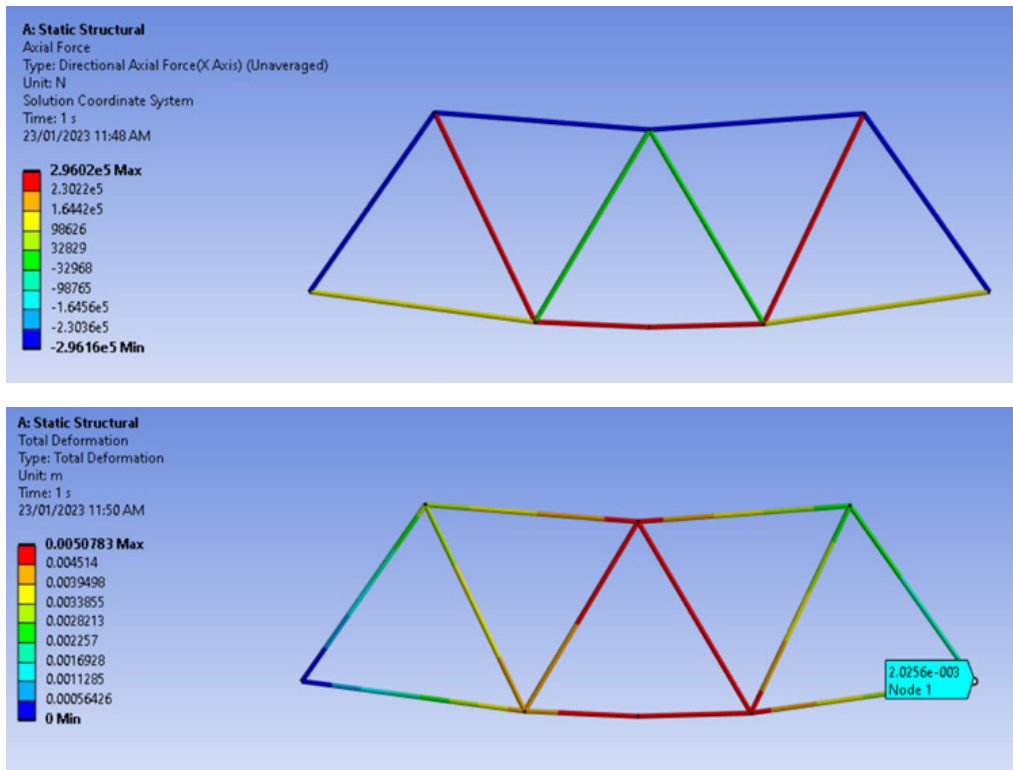
After applying loads and conditions to the geometry, each node will need to be constrained using a displacement condition to stop movement out of plane. This keeps the analysis in two dimensions. This can be done using the same method as the roller joint; however, the direction will again depend on the coordinate system used.

After adding all relevant constraints, the system can be solved.

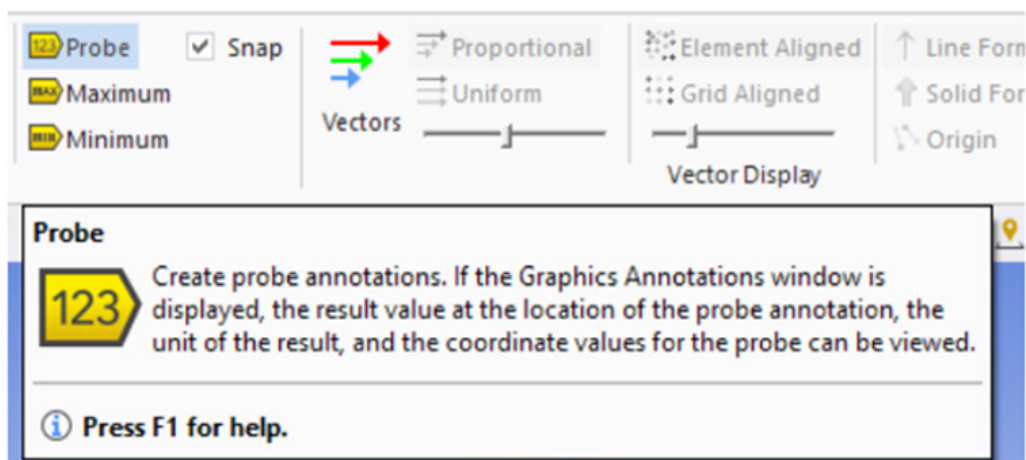
Add a total deformation analysis and a beam axial stress analysis to the solution. The beam axial stress can be added by selecting beam results in the solutions tab.



The solution to the solve can be seen in the images below, keep in mind that answers within 5-10% are still considered acceptable, as the algorithm used for solving will vary slightly with the coordinate system used, mesh quality etc.



NOTE: To look at specific points within the structure use the probe tool within the solution panel.



Exercise 2: Analysis of Mesh Quality

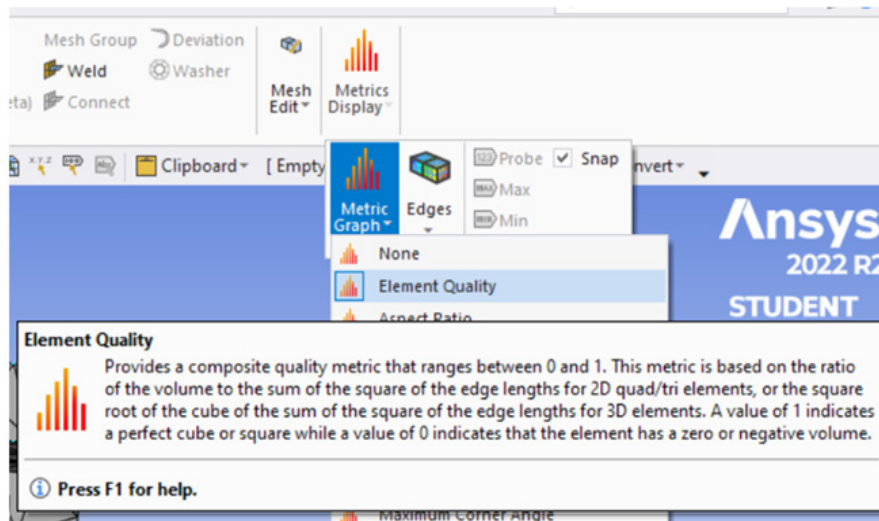
This exercise focuses on the effects of mesh quality and mesh density on solver results.

Import the geometry from Canvas under a new static structural system, leaving the model as structural steel.

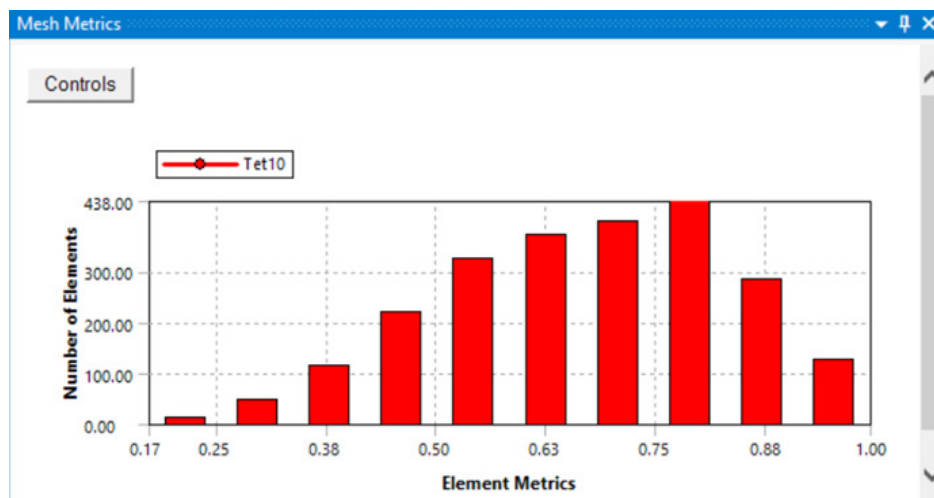
Open Ansys mechanical.

Generate the default mesh for the geometry.

Open a mesh metric graph by selecting mesh quality from the drop-down list



The results from the graph should appear similar to below.

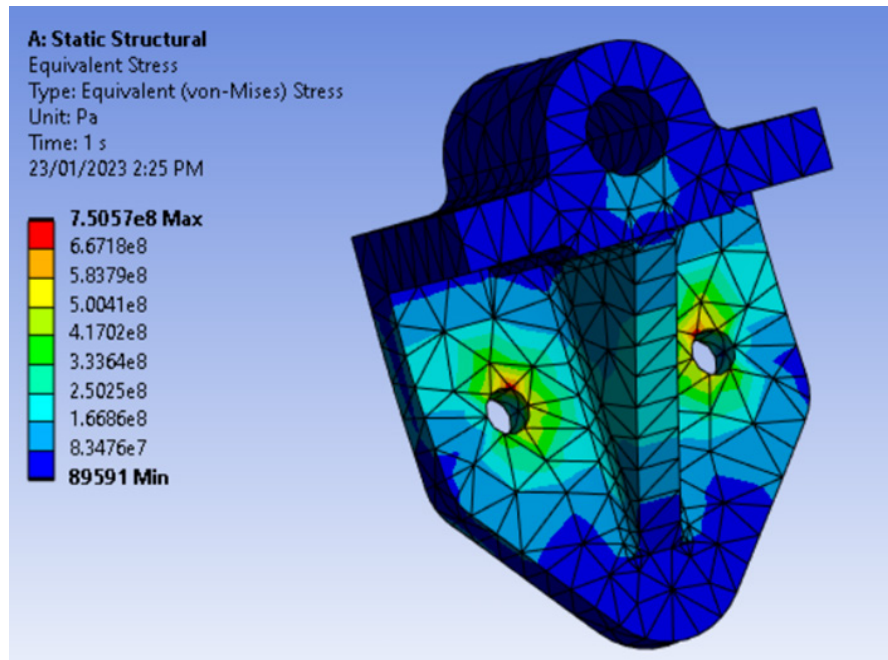


Higher quality elements will be closer to 1 on the graph, whilst lower quality elements will be closer to 0.

Apply a fixed boundary condition to the bolt holes of the geometry (the two holes on the thinner plate) and apply a downwards force of 50 kN to the top mounting hole.

Hint: You can apply a load or condition across multiple faces by choosing the face select option and holding the control key.

The result should resemble the following for a Von-Mises stress solution.



From the results, the detail captured around the bolt holes and mounting holes may not be sufficient to validate the solution.

At this point, feel free to experiment with different meshing techniques, sizes and element types to see how this influences the results of the simulation.

If using an Ansys Student license, be mindful of the node limitations. Check out the [Ansys Student Product Page](#) for more information.

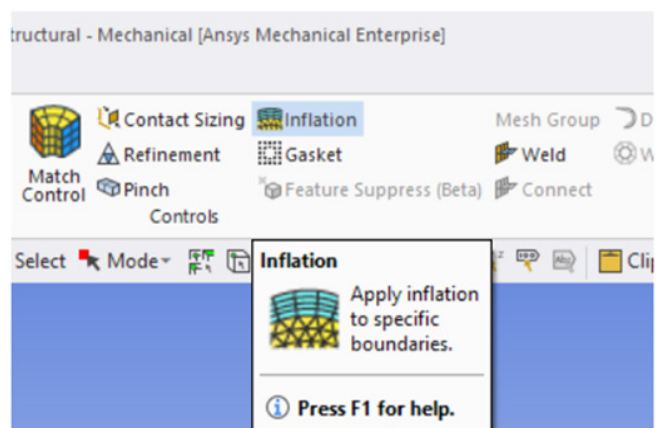
Two options to increase the accuracy of the solution whilst also keeping an appropriate number of elements is to apply an inflation layer to the features of interest, or to apply a sphere of influence to an appropriate region.

Applying an Inflation Layer

Applying an inflation layer can be done from the meshing menu.

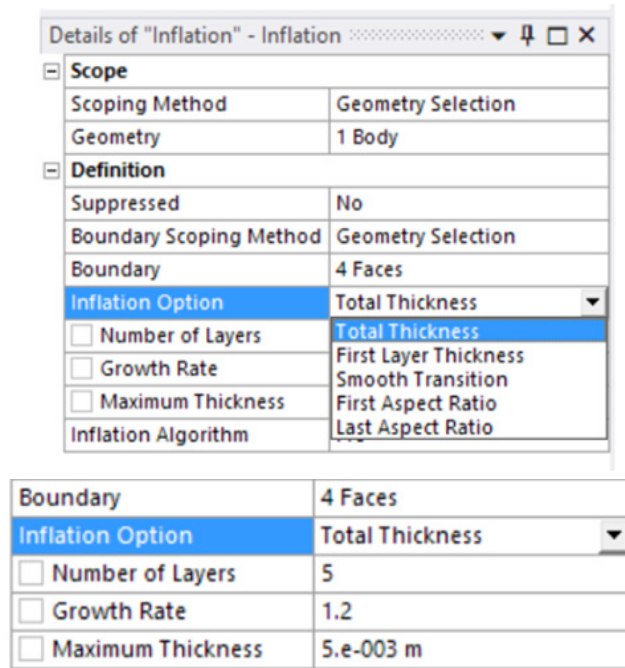
Select the inflation layer option and select the body it will be applied to.

Within the “Details” window, select the relevant edges or faces from where you want the inflation layer to start.



There are many different settings that can be applied within the inflation layer.

The number of layers and growth rate for each layer can also be set.

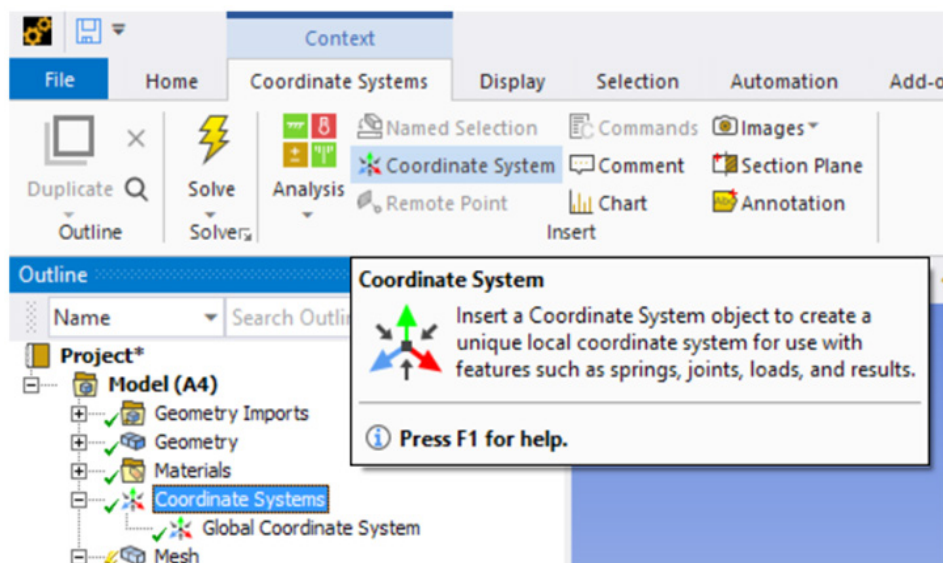


After applying a relevant inflation layer, regenerate the mesh to see it applied to the body.

Applying a Sphere of Influence

Lab 1 includes details on how to apply a sphere of influence through a sizing constraint on the mesh, however a relevant coordinate system has to be applied to the body to attach the sphere to.

To add coordinate systems to a body, select the coordinate system option from the project tree, and add a new one from the context menu.



Add a coordinate system to the center of each of the through holes before applying the relevant spheres of influence.

© 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.

Ansyes 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.

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.

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
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 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.