

# Ansys Software Tutorial

## Introduction to Ansys Mechanical #4: Loading of Structures and Irregular Sections

Ashleigh Kirkland

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 the loading of 3-D structures within Ansys Mechanical, as well as the creation and application of cross sections.

This tutorial is #4 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.

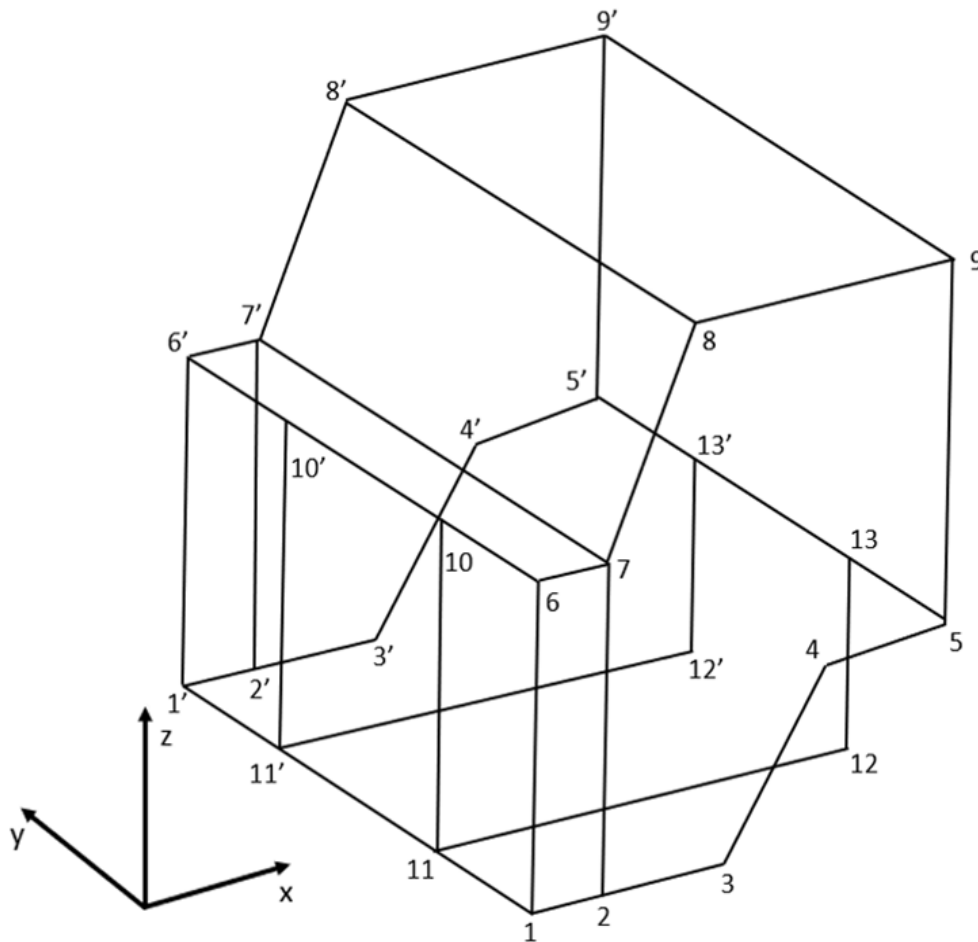
\*This tutorial was created using the 2022R2 Student Version of Ansys Workbench. Some screens may look different, depending on your version. Check the [Ansys Learning Forum](#) if you have any questions.

## Table of Contents

Exercise 1: Loaded Structures.....	3
Scenario 1:.....	3
Scenario 2:.....	4
Exercise 2: Creating and Loading Irregular Sections.....	5
Exercise 3: Challenge.....	8

## Exercise 1: Loaded Structures

The geometry uploaded with this tutorial represents a simplified model of a truck cabin. The model consists of 26 nodes, x-z is a symmetry plane – with symmetrical nodes denoted by a '.



The structure is being used in a safety test – where it will collide with a reinforced concrete wall in 2 different scenarios at low speeds.

The structure is made entirely from structural steel, and the testing procedure has a peak impact force of approximately 2 kN. In each scenario the nodes 11, 11', 12 and 12' can be assumed to be fixed.

You can change the view in Ansys mechanical from a wireframe to a cross section by navigating to the display tab and selecting 'Cross Section' instead of the default 'Thick Shells and Beams'.

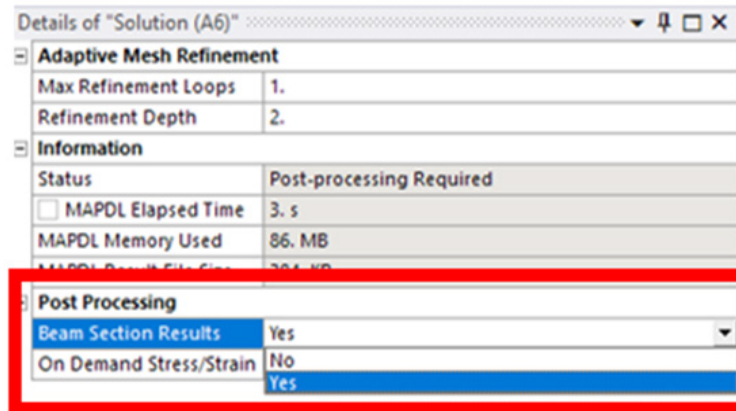
### Scenario 1:

The first impact test occurs solely on the front corner of the cab (*i.e* between nodes 1 and 6). Apply a 2 kN force at 45 degrees heading into the body of the structure. In this case the load should be applied to the vertices.

Determine the maximum displacement and stress in the structure.

**What do these results tell you given the yield point of the steel used in the structure is 250 MPa?**

To allow a stress analysis to be run using beam elements, in the solutions menu be sure to enable beam section results.



### Scenario 2:

The second impact test occurs across the front face of the cab. It impacts directly into nodes 1,6,10 and 11 as well as their mirrored counterparts.

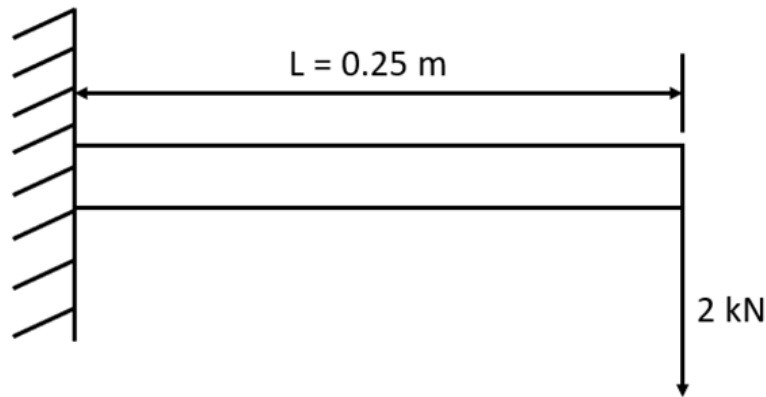
Apply a 2 kN force to each of these nodes.

Repeat the analysis as per scenario 1 to determine the maximum displacement and stress in the structure.

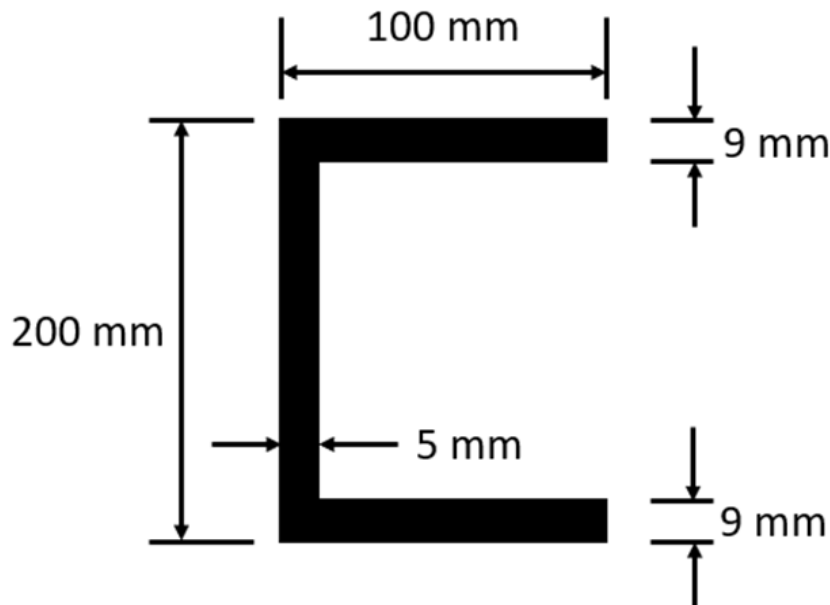
**What do these results tell you given the yield point of the steel used in the structure is 250 MPa?**

## Exercise 2: Creating and Loading Irregular Sections

The figure below shows a cantilever beam loaded with a point load on the end of approximately 200 kg.



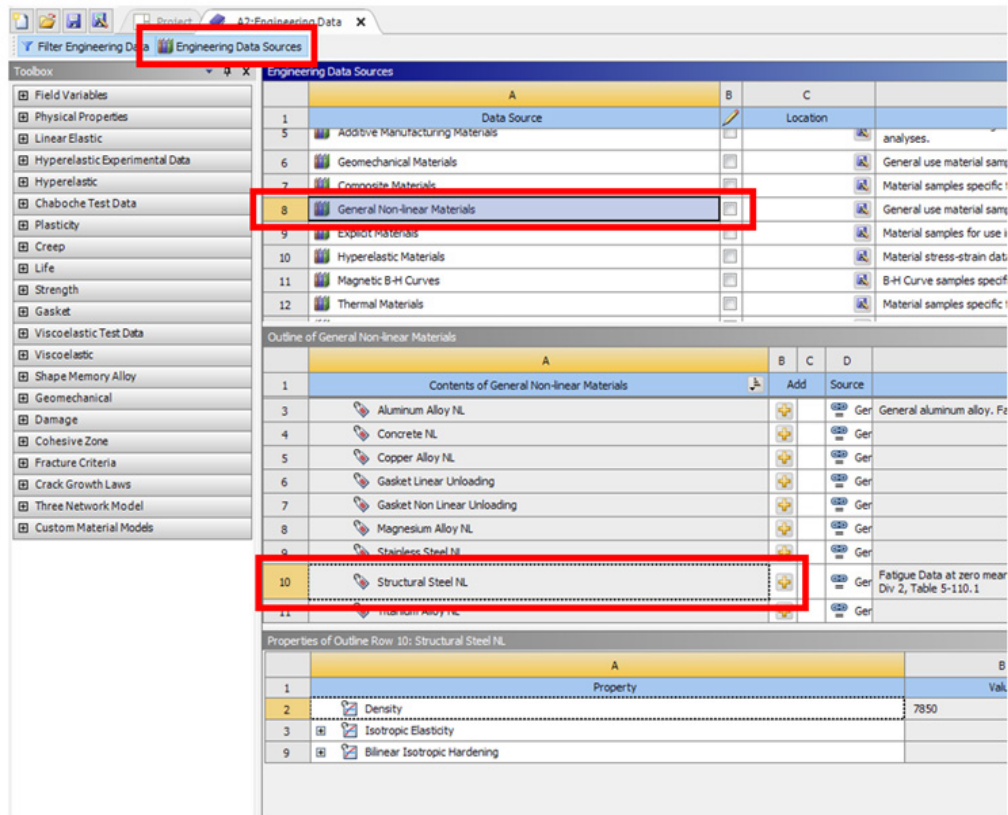
The beam has the same cross section as question 1 in week 4's tutorial, as shown below.



**Create the cross section of the beam as shown and determine if the beam will reach its yield point. Determine the maximum load the beam can carry before it yields.**

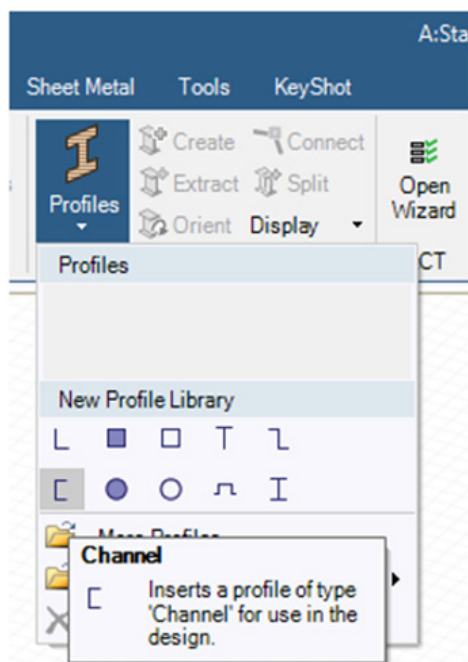
To begin this setup, the first step is to set the material property. As the goal of this exercise is to analyze the plastic deformation in the beam, it is required that the material properties are representative of how the material would behave in an actual test, i.e., elasticity would change as the load is applied, as well as changes due to hardening.

To incorporate this, you should set the material property to be non-linear structural steel. This can be found in the Engineering Data Sources tab, under General Non-linear Materials.

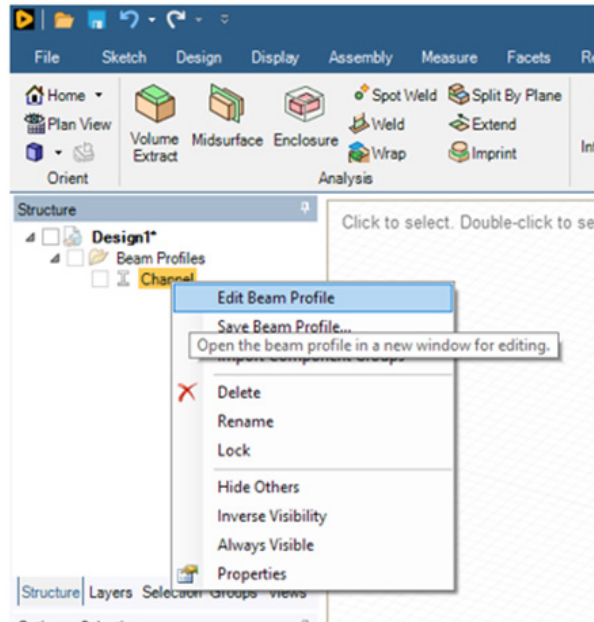


The next step is to create a custom beam profile within SpaceClaim. This can be done by modifying an existing profile within SpaceClaim.

Navigate to Prepare – Profiles and select the channel from the drop-down menu.



From this point the dimensions of the channel can be edited by selecting 'Edit Beam Profile'.



Now that the custom cross section has been created, the geometry for this exercise is able to be completed. Create a new SpaceClaim geometry and sketch a line of length 250 mm.

Assign this cross section to the geometry by creating a beam as you did in lab 3.

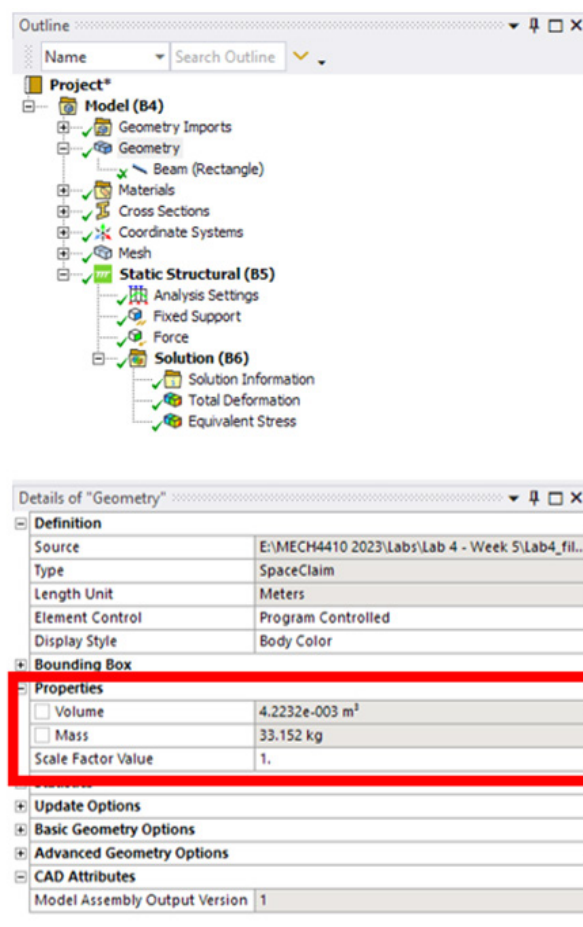
### Exercise 3: Challenge

This part of the exercise can be done in a variety of different ways. It is up to you to decide if your solution is sensible/ an efficient way to complete this objective.

Using the provided geometry of the truck cabin – alter the cross section of the beam to create a structure that would pass the safety test.

To pass the safety test, none of the beams within the structure should exceed 75% of their yield strength. The overall mass of the skeleton structure should also not exceed 1200 kg.

The weight of a structure can be checked by navigating to properties within the geometry tab of the model setup.





© 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.