MAE 656 - Advanced Computer Aided Design

05. Shells and Membranes – Doc 03

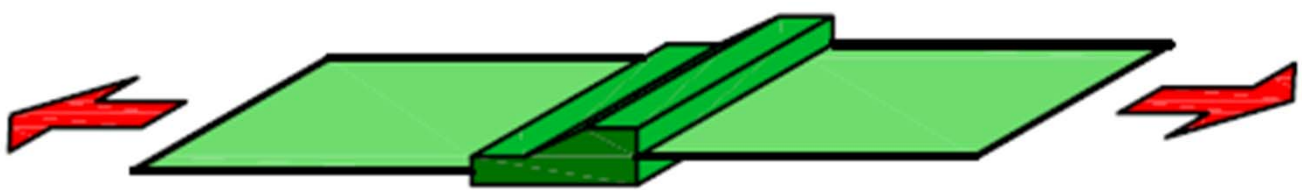
Coupling Shell Elements with Solid Elements

Introduction

In this session we will learn how to create shell structures from solid elements.

We will also see how we can couple solid structures with shell structures.

This last option can be used when some structural component, attached to a shell structure, wants to be studied in detail. i.e. ply drop-off



Introduction

Specifically, we will learn:

1. How to create shells structures from the mid-plane of solid elements

2. How to create shell structures from the outer surfaces of solid elements

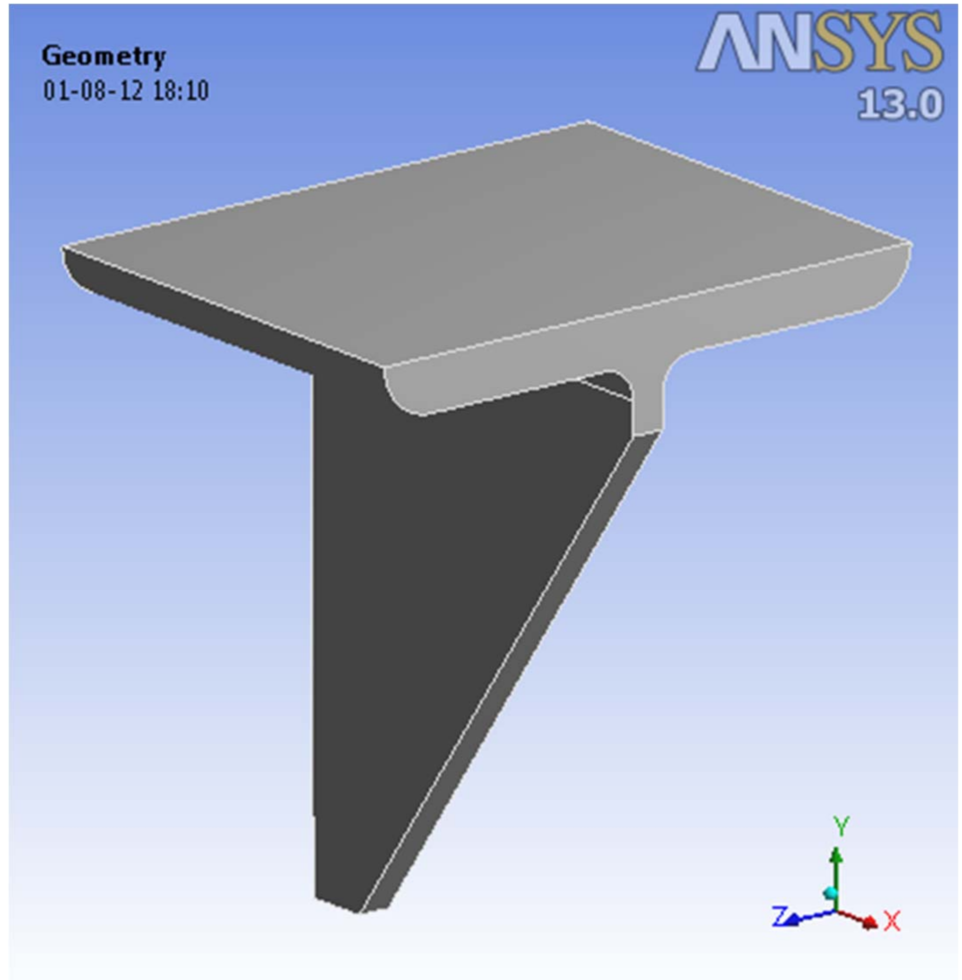
3. How solid elements can be coupled with shell elements

Base Model

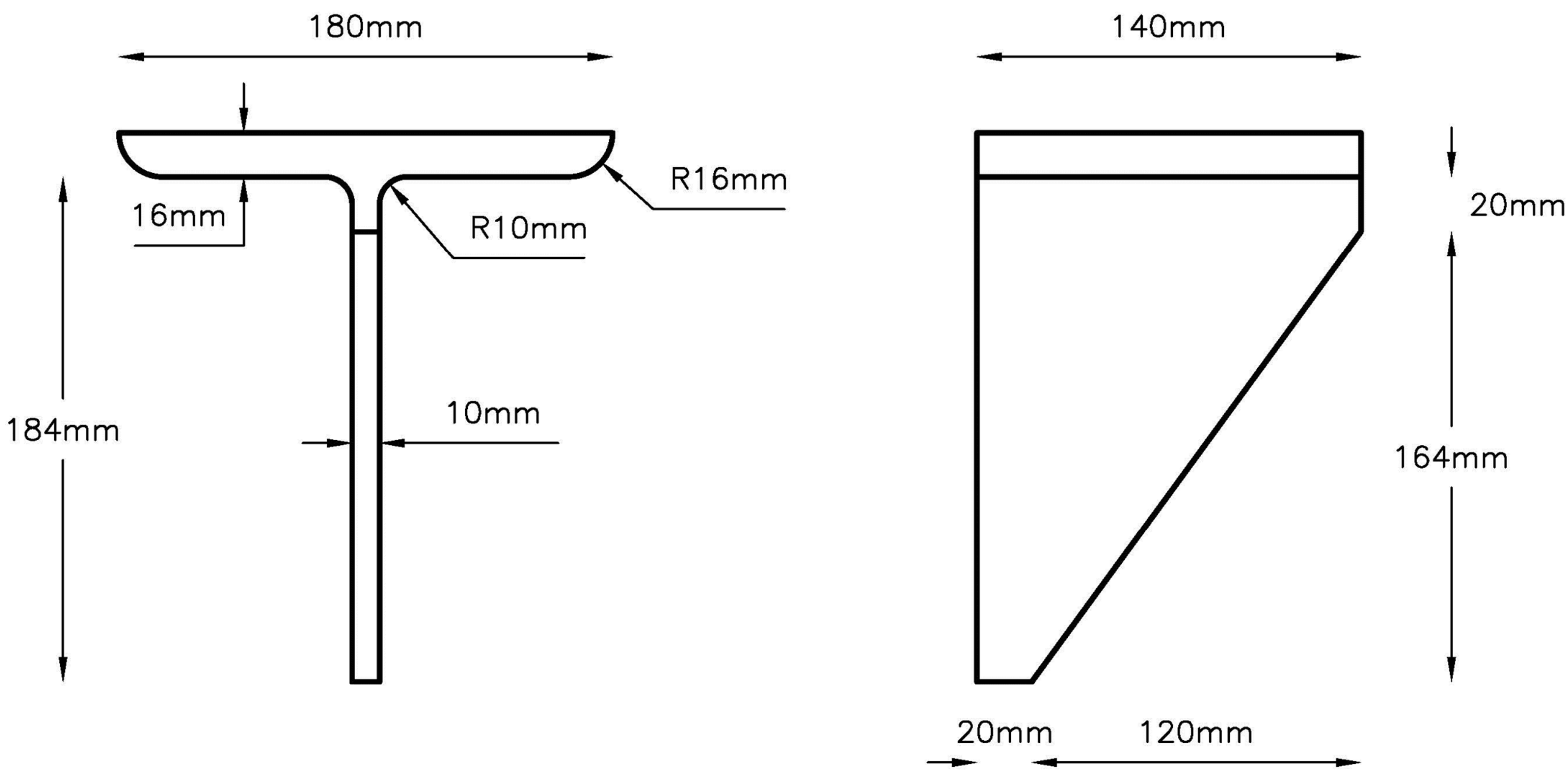
As base model we will use the following beam support. We will first solve it as a solid structure.

The support is made of *Structural Steel*

The support dimensions and loads have been obtained from Huei-Huang Lee. *Finite Element Simulations with ANSYS Workbench 13. Theory – Applications – Case Studies*. SDC Publications (2011)



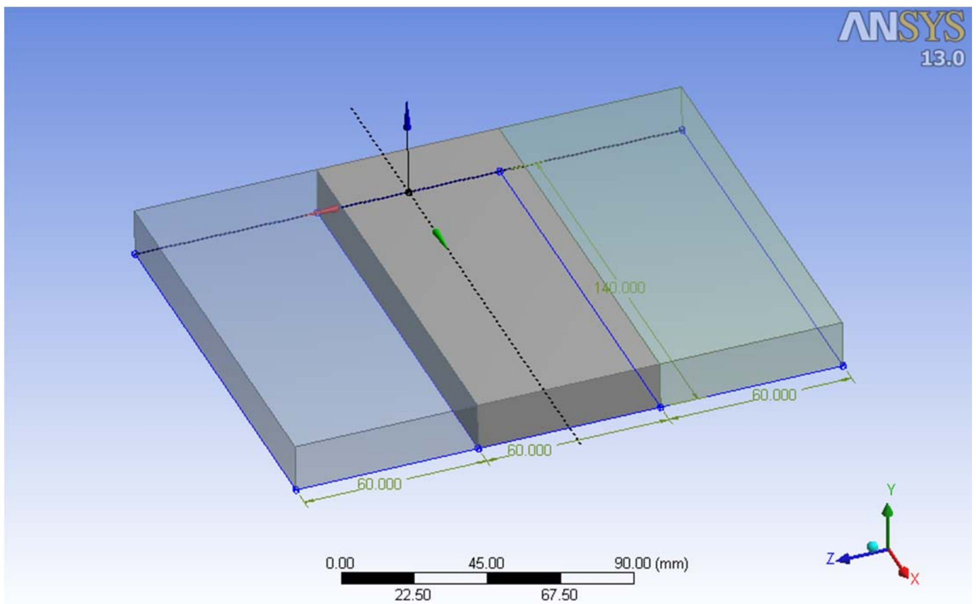
Base Model - Geometry



Base Model - Geometry

To create the solid geometry, we will first define the support seat plate, in plane xz.

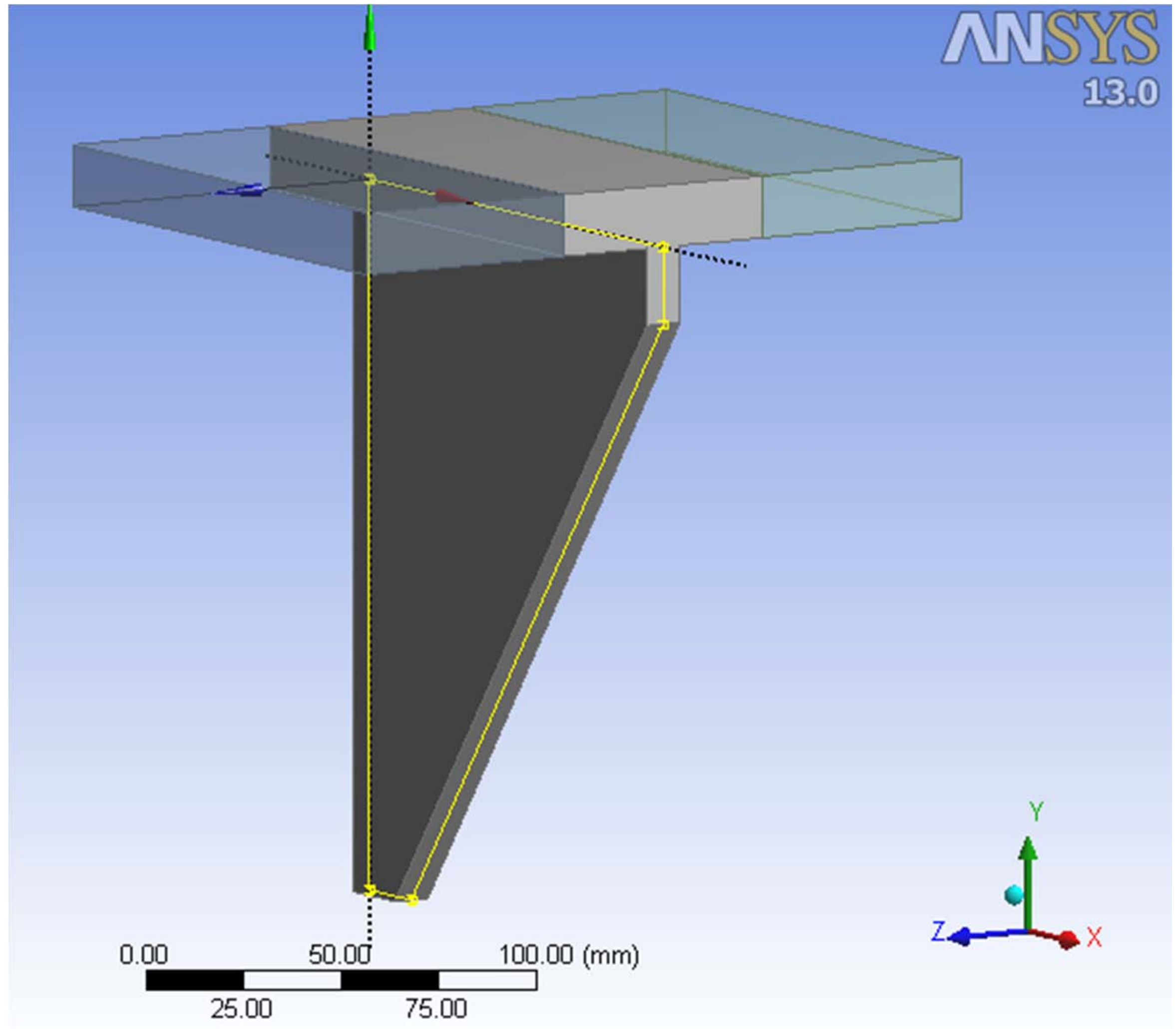
We will create it as three different sketches that will be extruded 16mm as *frozen*.



Base Model - Geometry

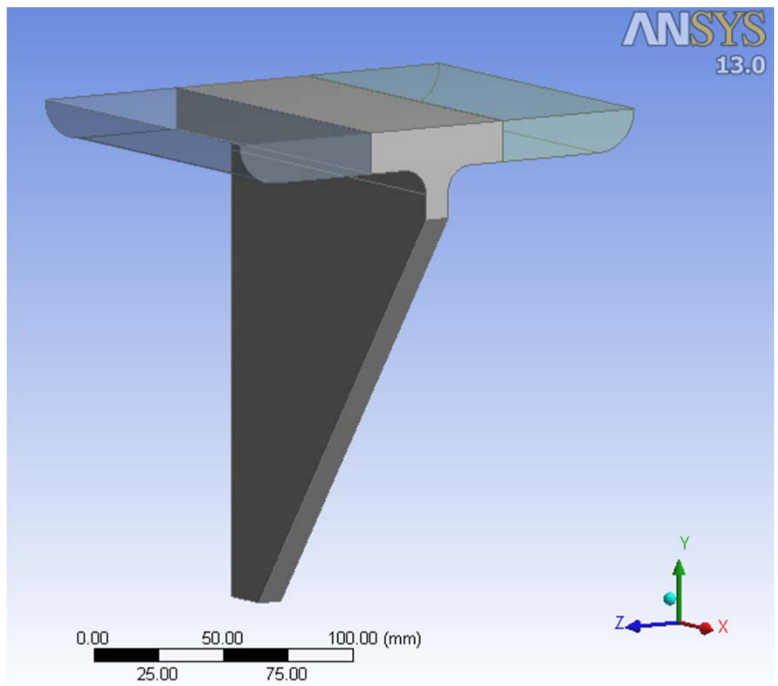
The web plate is defined extruding 5mm in direction *Both - Symmetric* a sketch drawn in xy plane.

The web plate is defined as *Add Material*



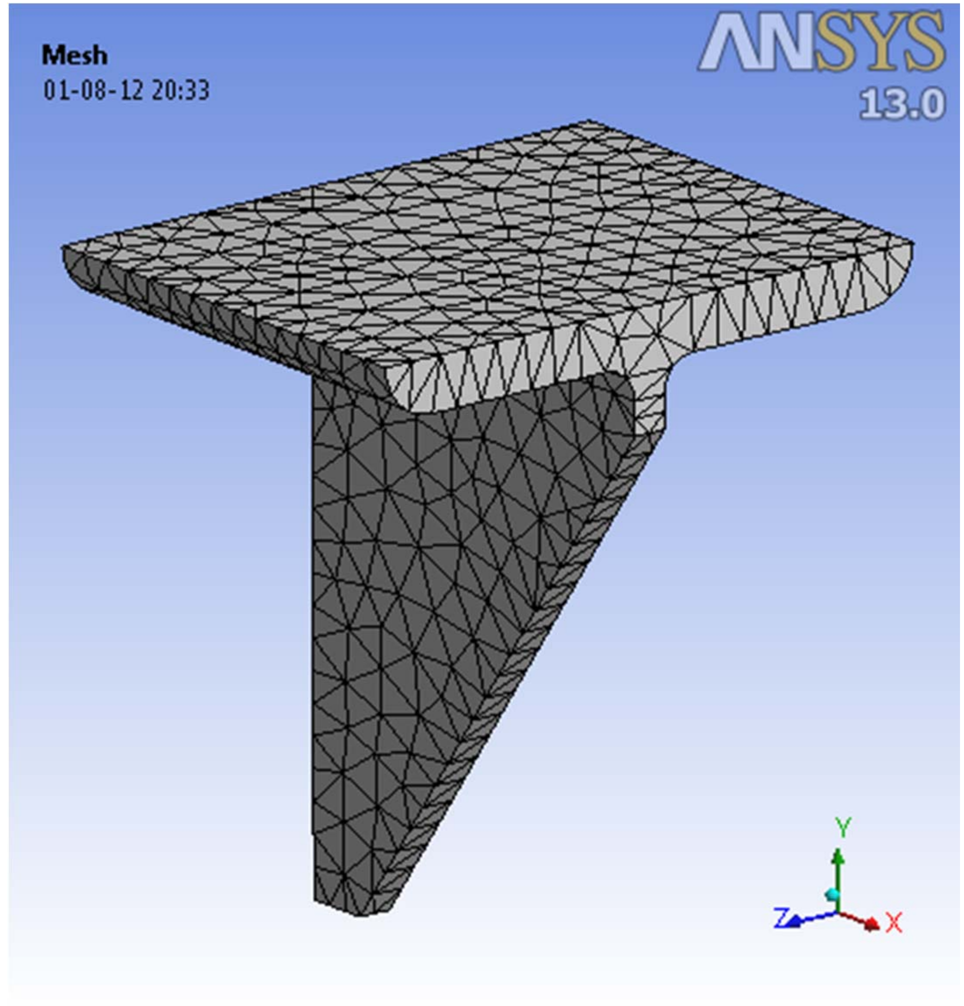
Base Model - Geometry

The last step consists in blending the external corners with a radius of 16mm and the connection of the seat plate with the web with a radius of 10mm



Base Model - Mesh

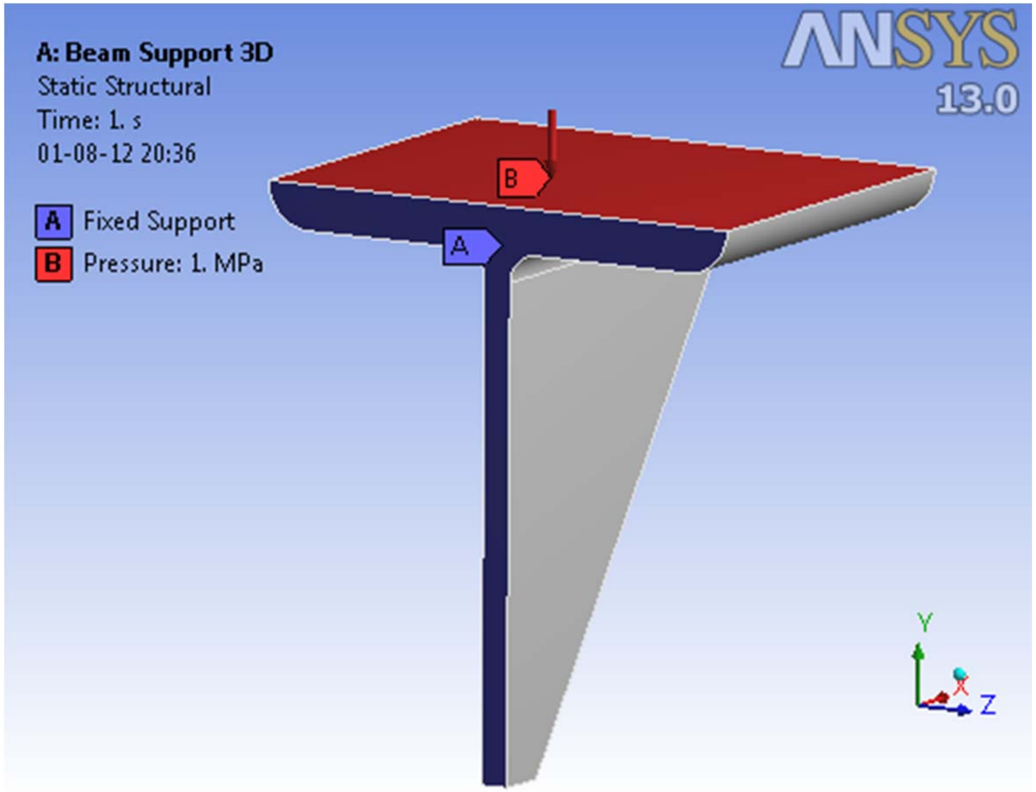
The solid will be meshed with the automatic mesh suggested by Ansys, using a *relevance* value of 100



Base Model – Boundary Conditions

The boundary conditions applied are a pressure of 1.0MPa in the seat plate.

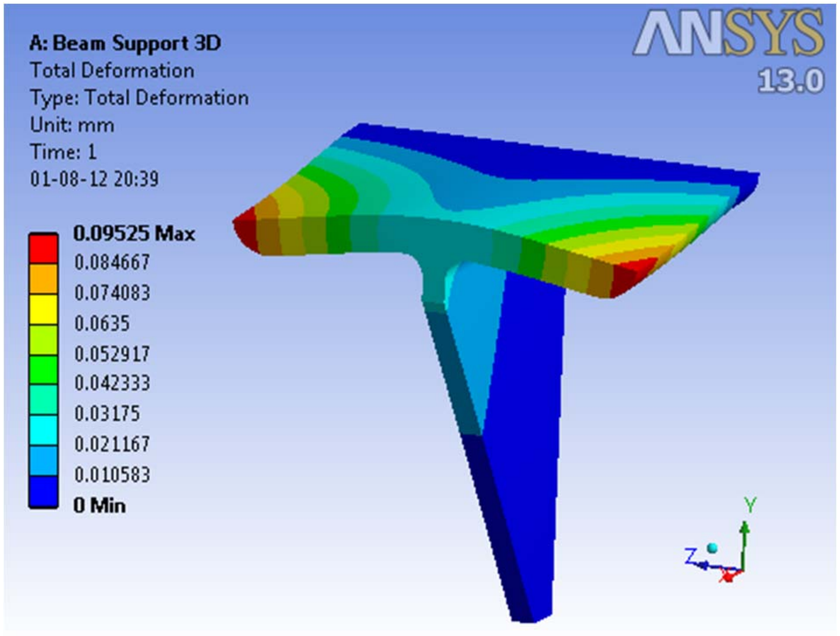
The edge of the web plate is fixed supported.



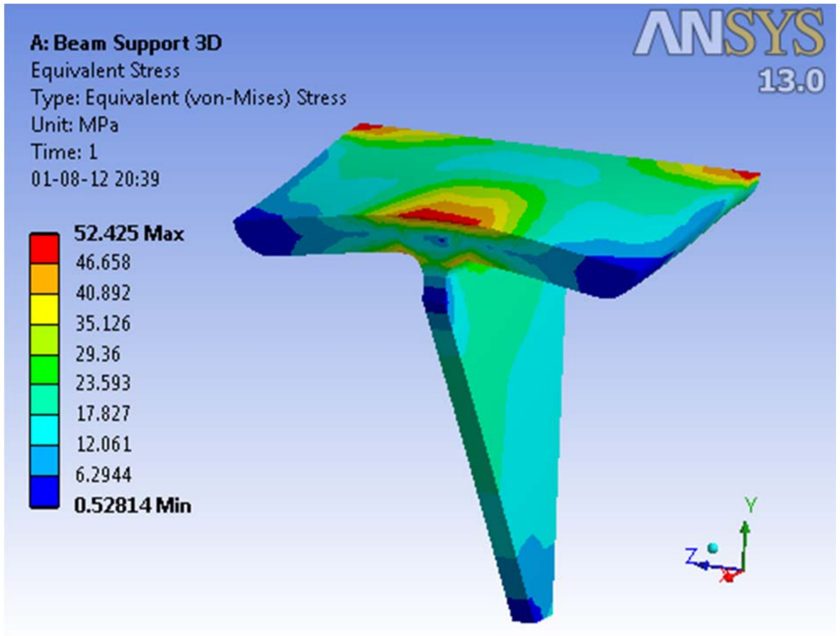
Base Model – Results

The main two results that will be studied are the total displacement and the Von-Mises stresses

Total Deformation



Von-Mises Stress

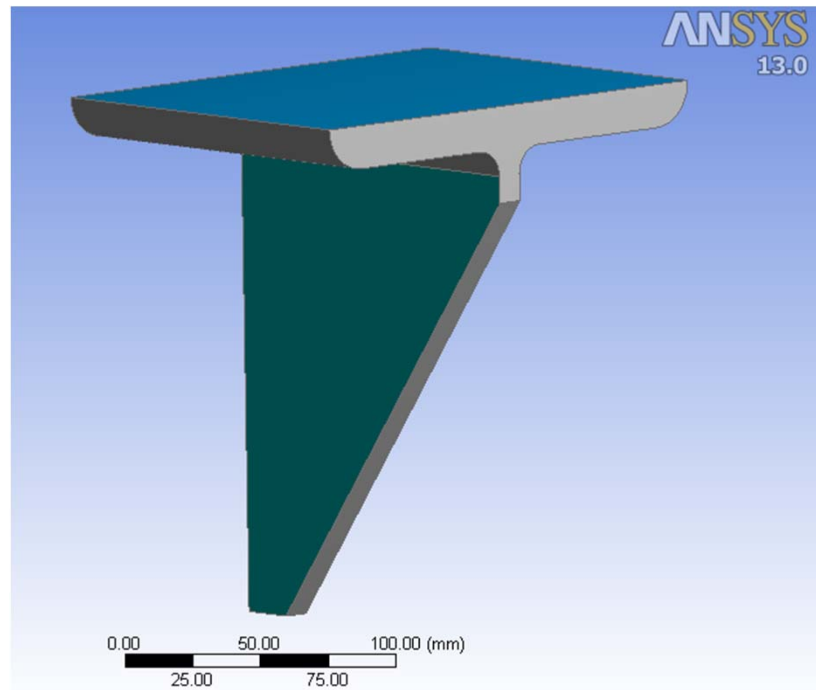


Shell – Create Mid-plane Surfaces

In *Design Modeler > Tools*, there is an option that creates surface elements at the mid-surface of solids. This process can be done manually or automatic

Details View

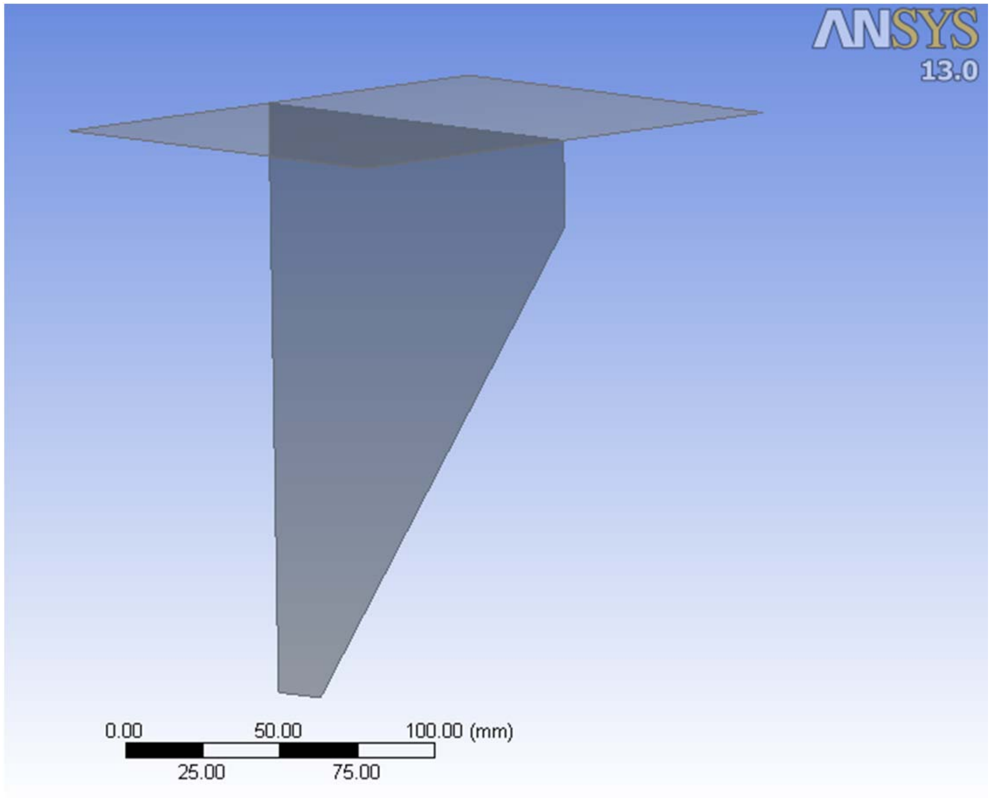
Details of MidSurf1	
Mid-Surface	MidSurf1
Face Pairs	3
Selection Method	Automatic
Bodies To Search	Visible Bodies
Minimum Threshold	10 mm
Maximum Threshold	16 mm
Find Face Pairs Now?	No
FD1, Thickness Tolerance (>=0)	0.01 mm
FD2, Sewing Tolerance (>=0)	0.02 mm
Extra Trimming	Intersect Untrimmed with Body
Ambiguous Face Delete	All
Preserve Bodies?	No



Shell – Create Mid-plane Surfaces

The result is the shell model sought.

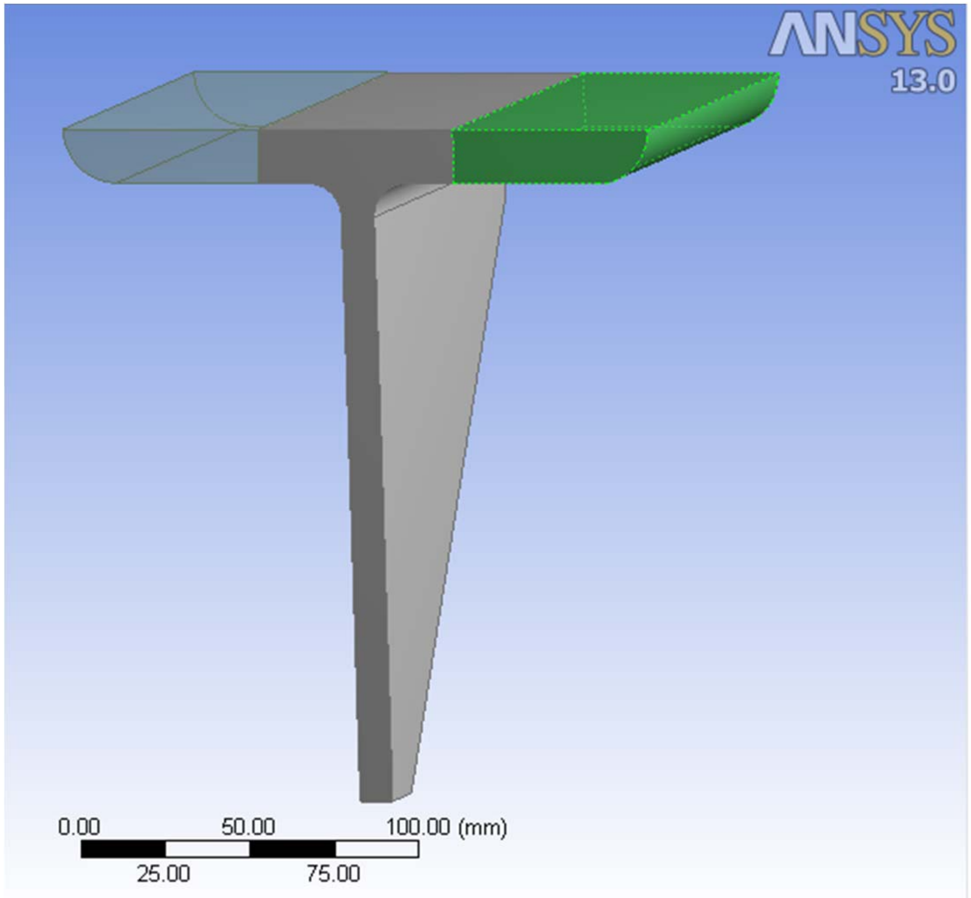
We will apply the same boundary conditions to this model and we will compare the results with those obtained from the solid simulation



Shell – Create Surfaces from Solids

The solid contour can be used to create 3D surfaces. This is done with the command *Thin/Surface* that can be found in *Design Modeler > Create*.

This command asks us to define the surfaces of the solid geometry that we want to use as surfaces.

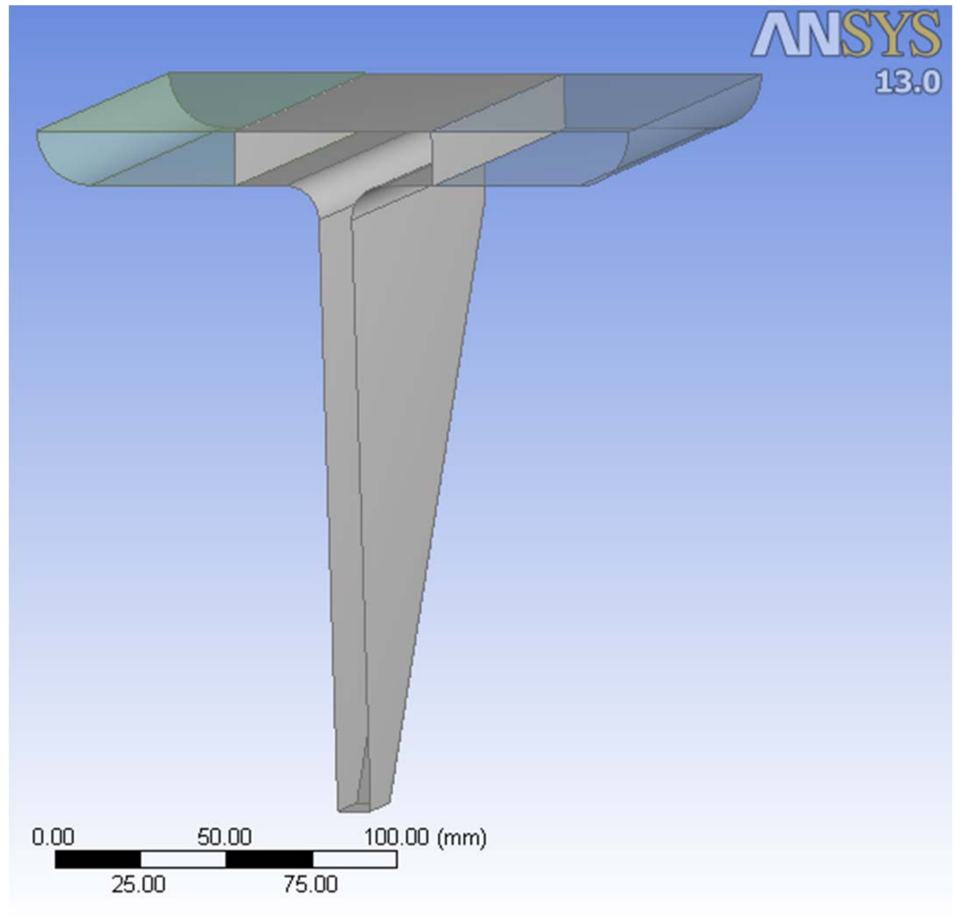


Shell – Create Surfaces from Solids

If a thickness is specified, the result is a thin walled solid. If thickness is defined as zero, the result is a surface element.

Details View

Details of Thin1	
Thin/Surface	Thin1
Selection Type	Faces to Keep
Geometry	5 Faces
Direction	Inward
FD1, Thickness (>=0)	0 mm
FD2, Face Offset (>=0)	0 mm



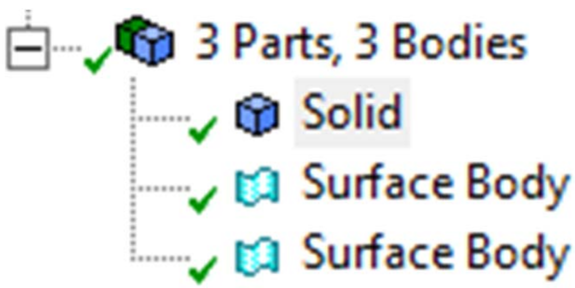
Shell – Create Surfaces from Solids

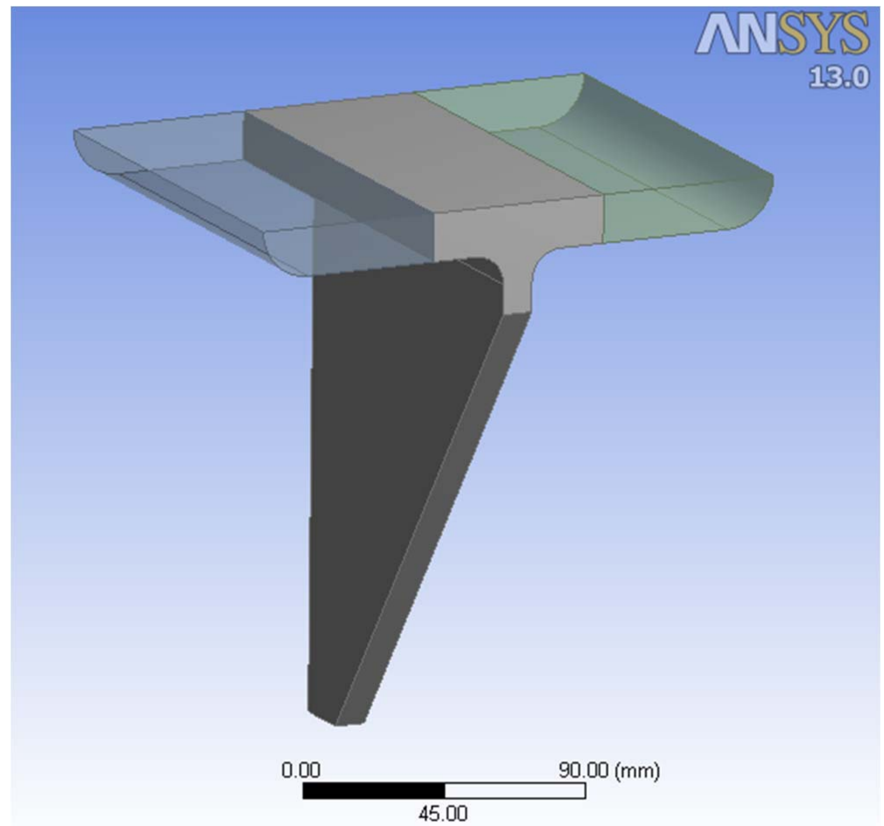
As we have done with the other shell model, we will apply the same boundary conditions defined for the 3D case and we will analyze the results obtained.

In this case, the results are substantially different, as the structural element studied is also different.

Shell-Solid model

If the center solid is not transformed into shell elements, we will end up with three different parts: a solid body and two surface bodies

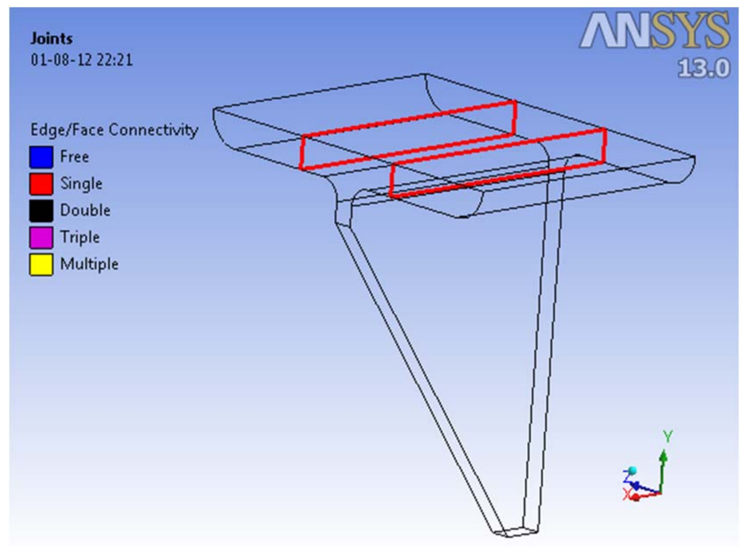




Shell-Solid model

We will not group these three bodies in a single part; because Ansys Workbench does not recognize a single part with surfaces and solids (only the solid will be displayed).

The union of these three parts is done in the Design Modeler, using the *body to body connection*.



Shell-Solid model

Once all bodies are connected, the mesh is defined, and the boundary conditions are applied, it is possible to calculate the solution.

The solution obtained with this model will be compared with the solution obtained with all previous models.