MAE 656 - Advanced Computer Aided Design

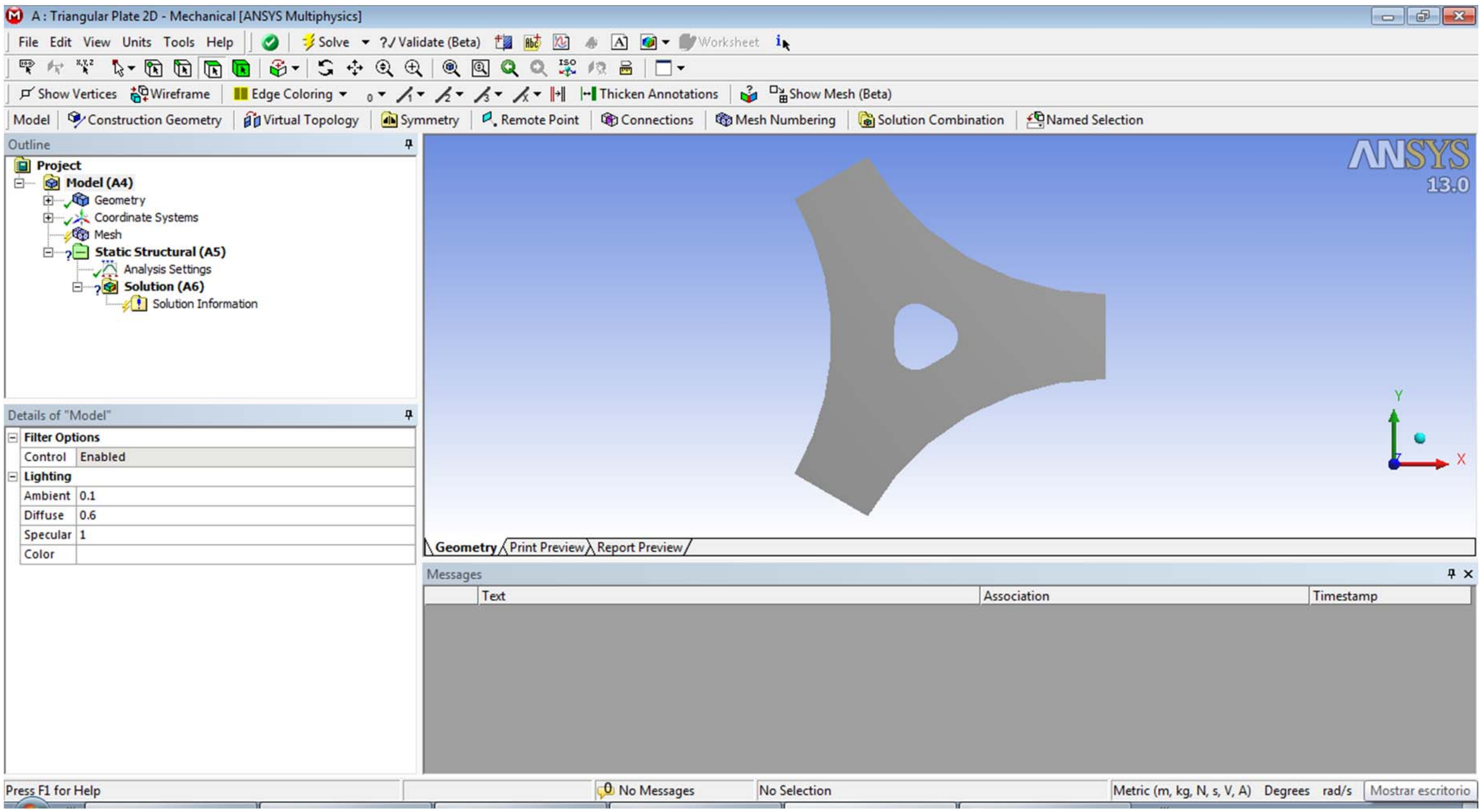
02. Ansys Workbench – Doc 02

Introduction to
Ansys Workbench - 2

Main Screen

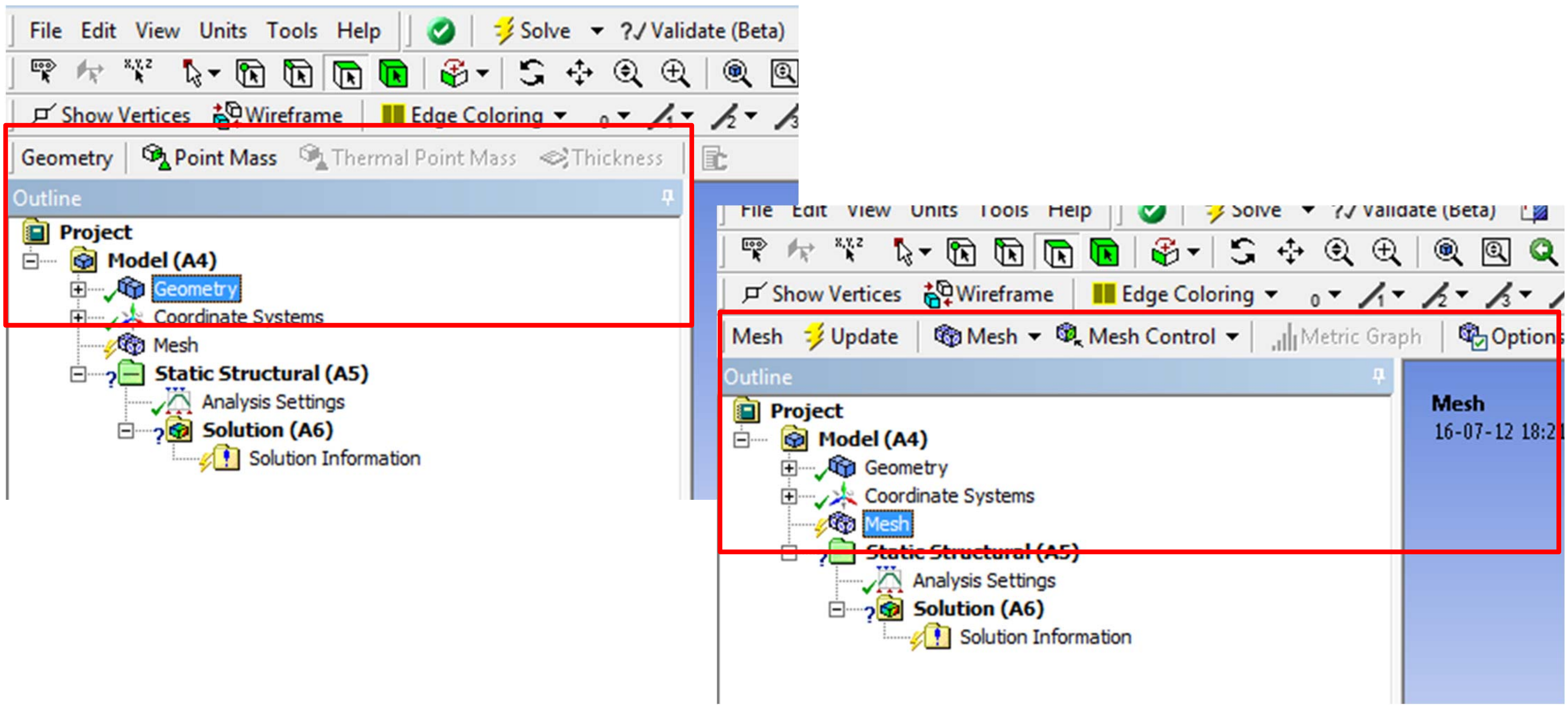
Components:

Top menu – Toolbars – Outline – Details – Visualization - Messages



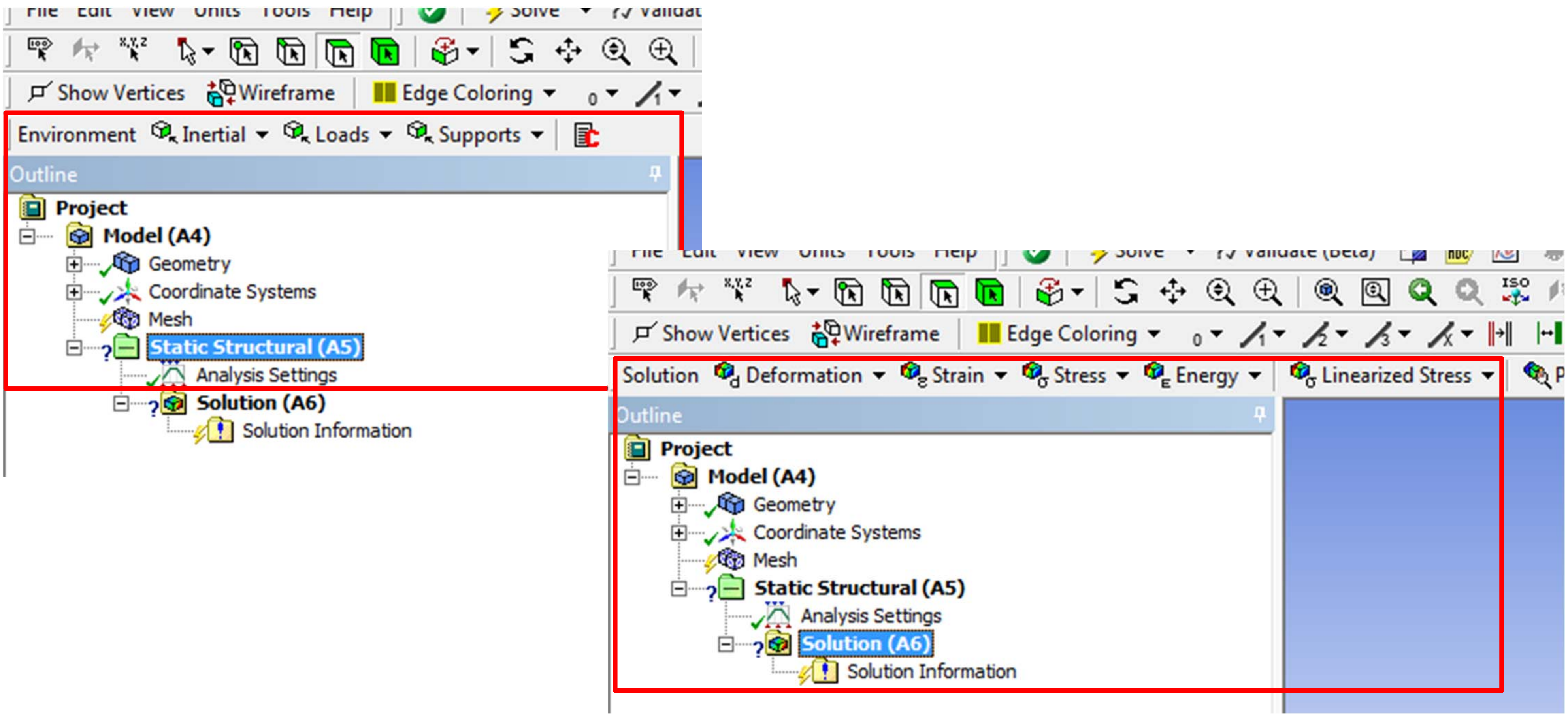
Toolbars

Toolbars change depending on where we are:



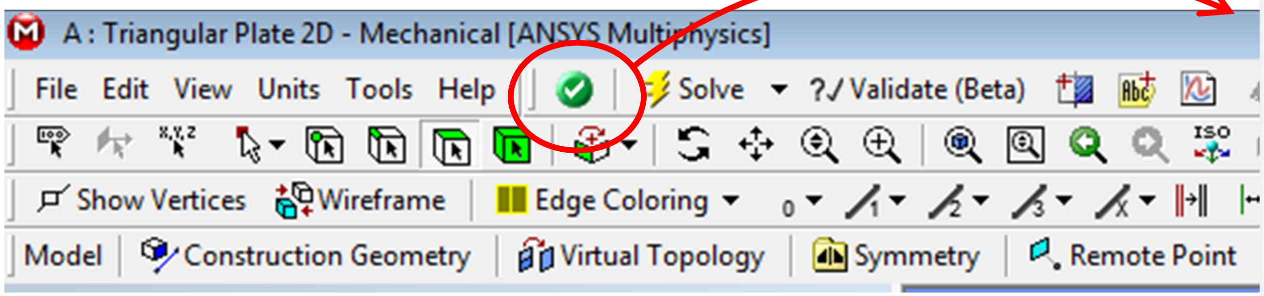
Toolbars

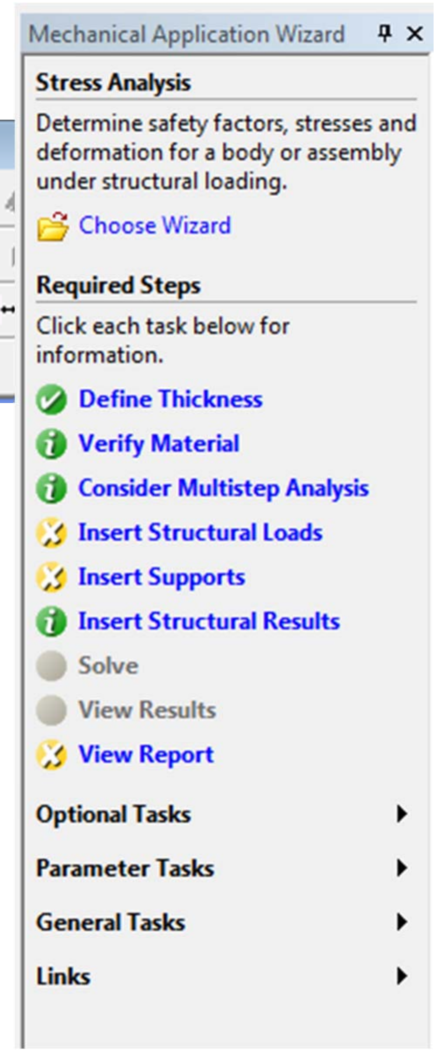
Toolbars change depending on where we are:



Mechanical Wizard

Mechanical Wizard



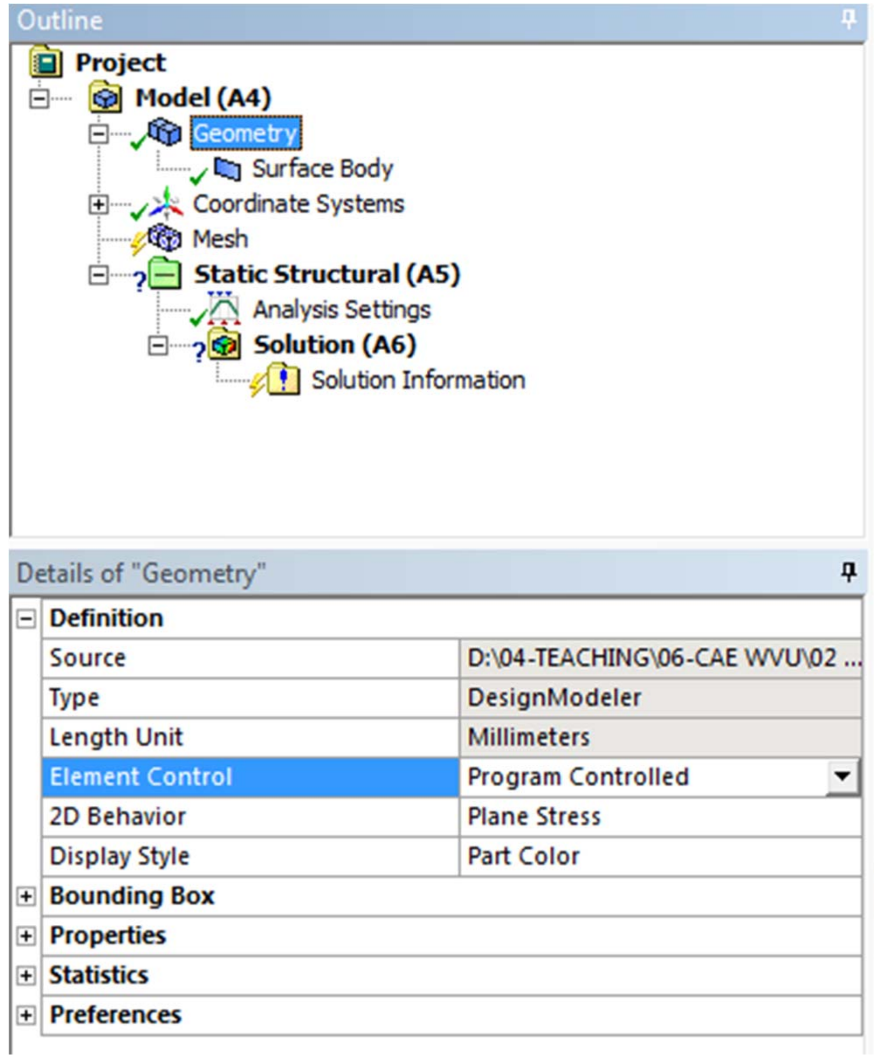


- The Mechanical Wizard provides a list of required steps and the status of them
- Green checkmark indicates the item is complete
- Green "i" shows an informational item
- A greyed symbol shows that the step cannot be performed yet
- A red question mark means that there is an incomplete item
- An "x" means that the item is not performed yet
- A lightning bolt means that the item is ready to be solved or updated

Geometry

If we place ourselves on the *Geometry* section, we can define common parameters of the model geometry.

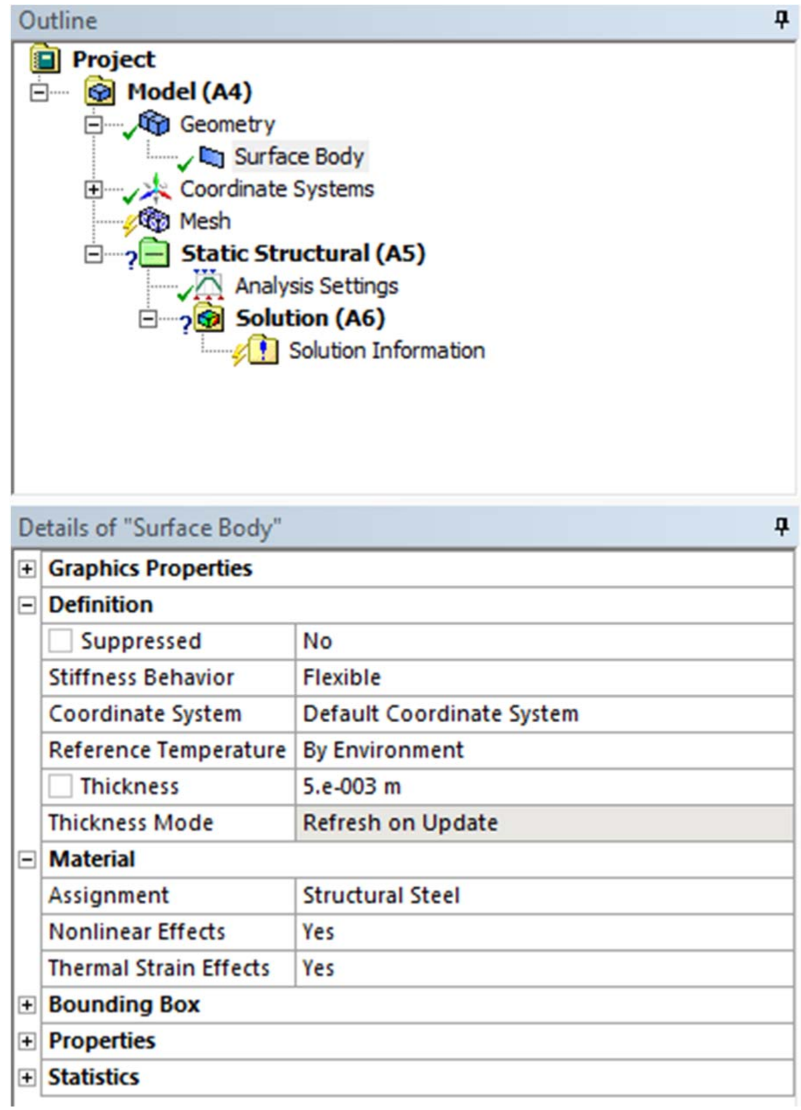
We also can look at different properties and statistics of the structure that will be solved.



Geometry

We can also look into the different components of the defined geometry.

We can (and have to) define different parameters (i.e. material that will be assigned) for each one of the components.

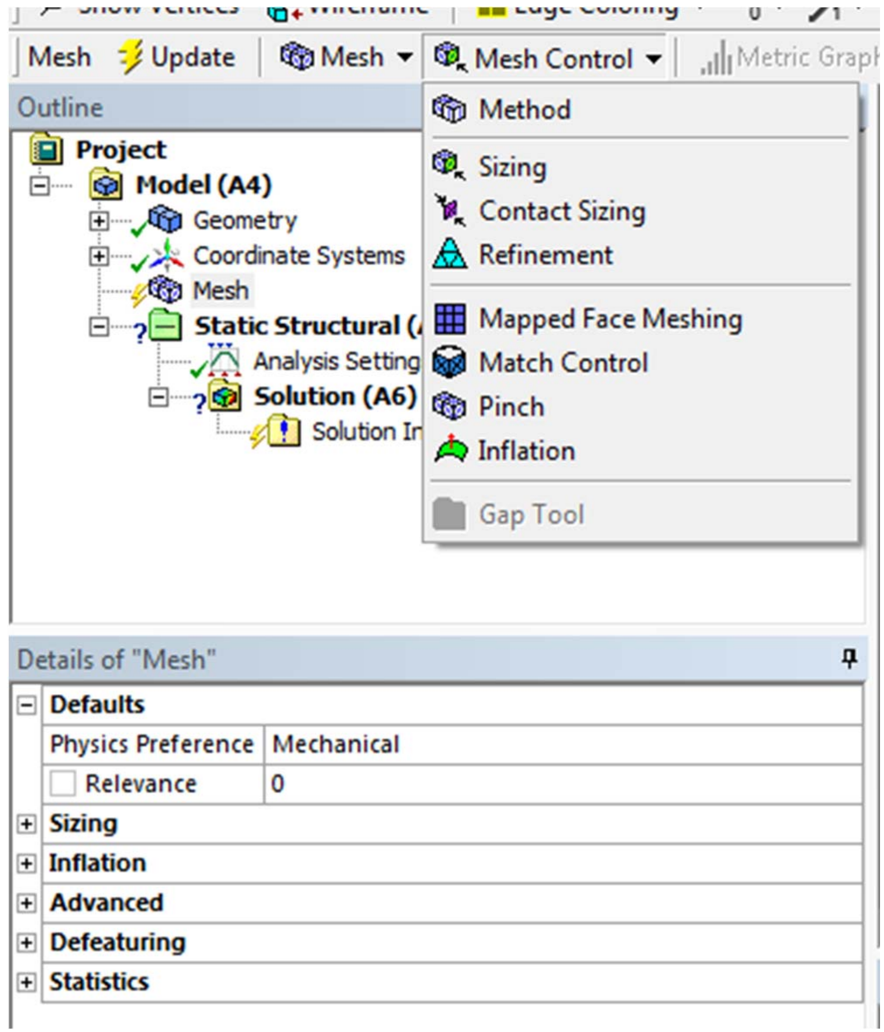


Mesh

We can define several mesh properties using the mesh control option.

When meshing we have to take into account that the results will be more accurate if the element is smaller. However, smaller elements require larger computational cost.

It is also recommended to have elements with all their dimensions as similar as possible.



Mesh

In *Method* we can define the type of elements that will be used.

In *Sizing* we can give instructions regarding the size of the elements (either giving a general size, or defining the number of elements in certain lines/faces)

More information about meshing options and configurations can be obtained from the Ansys tutorial:

M-13.0 - 02 - Meshing.pdf

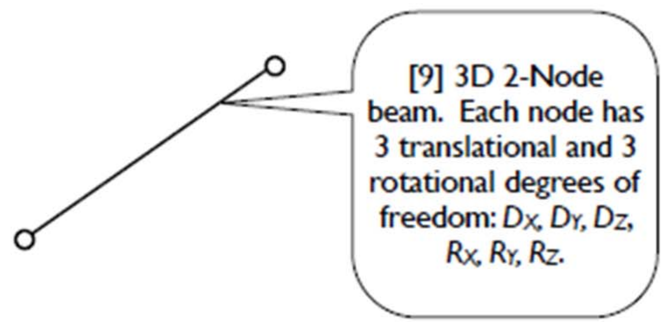
Elements Available

Ansys Workbench does not have all Mechanical APDL elements available.

The ones that are available are the following. More information on these elements can be obtained from the Ansys manual and Ansys Help.

3D LINE BODIES

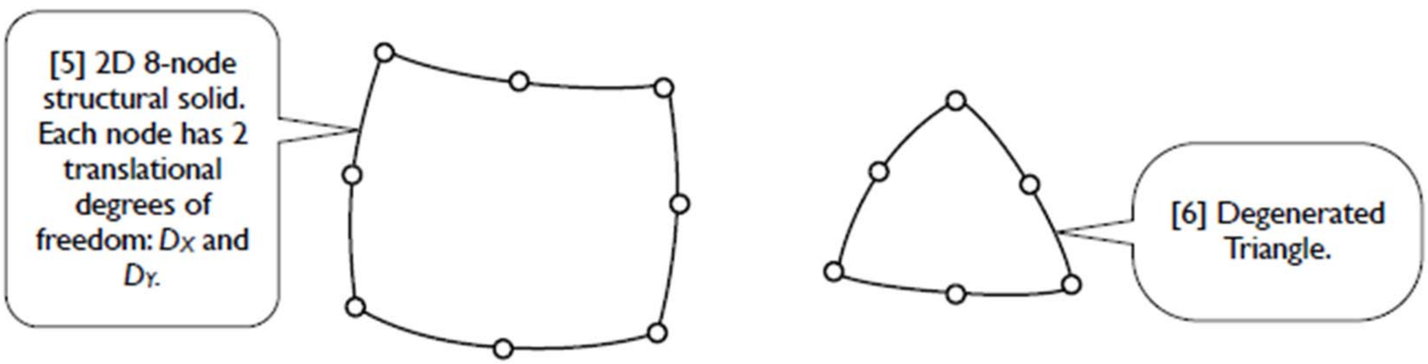
Element BEAM 188



Elements Available

2D Solid Bodies

Element PLANE 183

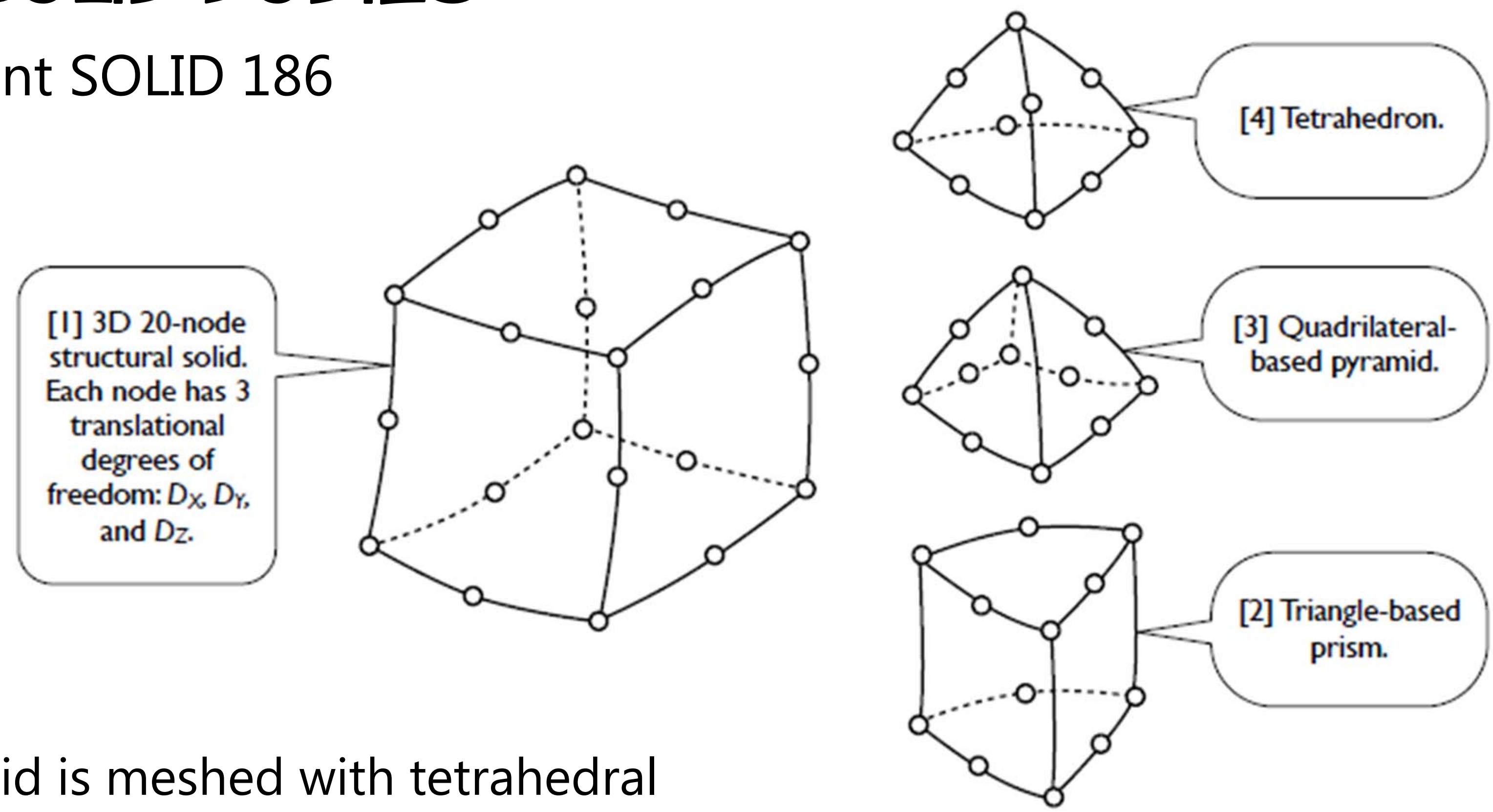


It is possible to drop the midside nodes. In this case, the element used is PLANE 182

Elements Available

3D Solid Bodies

Element SOLID 186

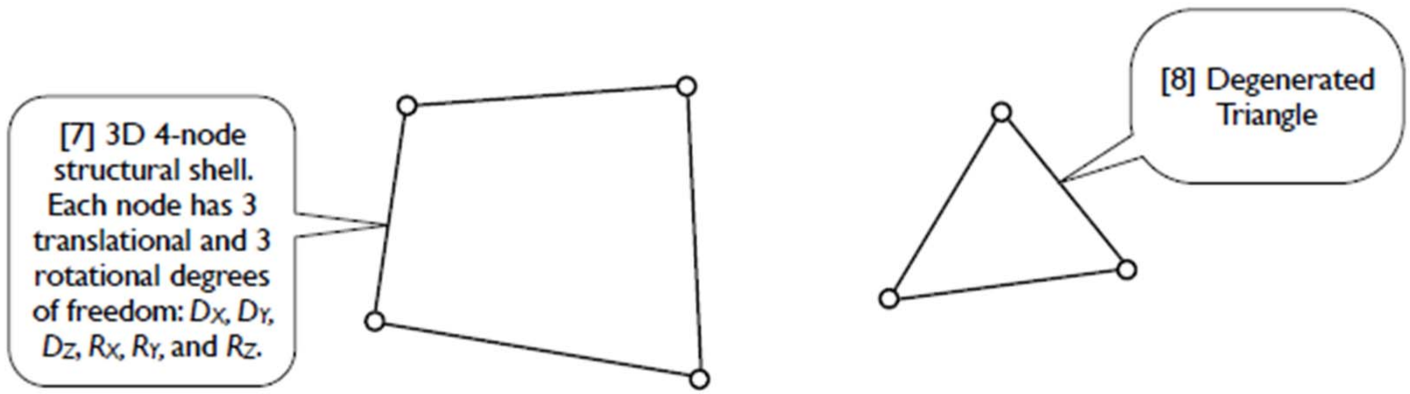


If the solid is meshed with tetrahedral elements exclusively, Ansys uses SOLID 187

Elements Available

3D Surface Bodies

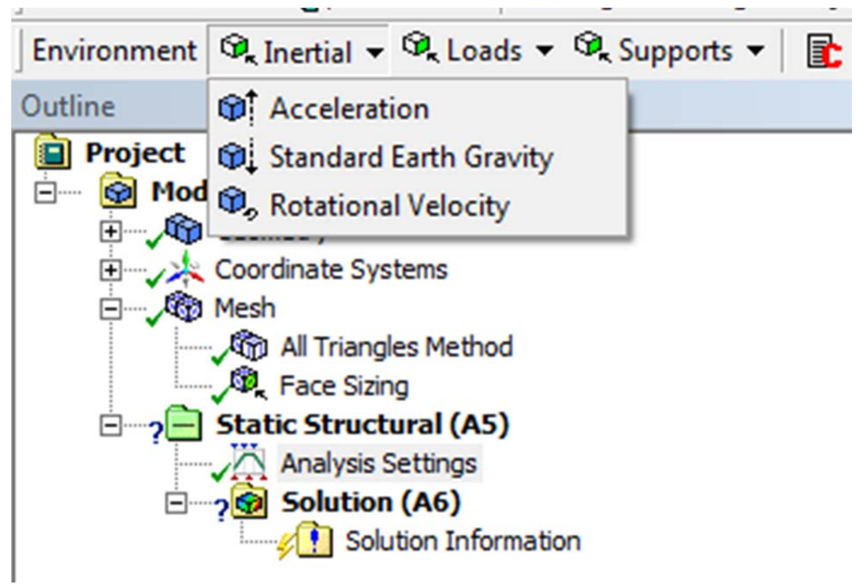
Element SHELL 181



Setup - Inertial

We can define different inertial forces that may affect the structure.

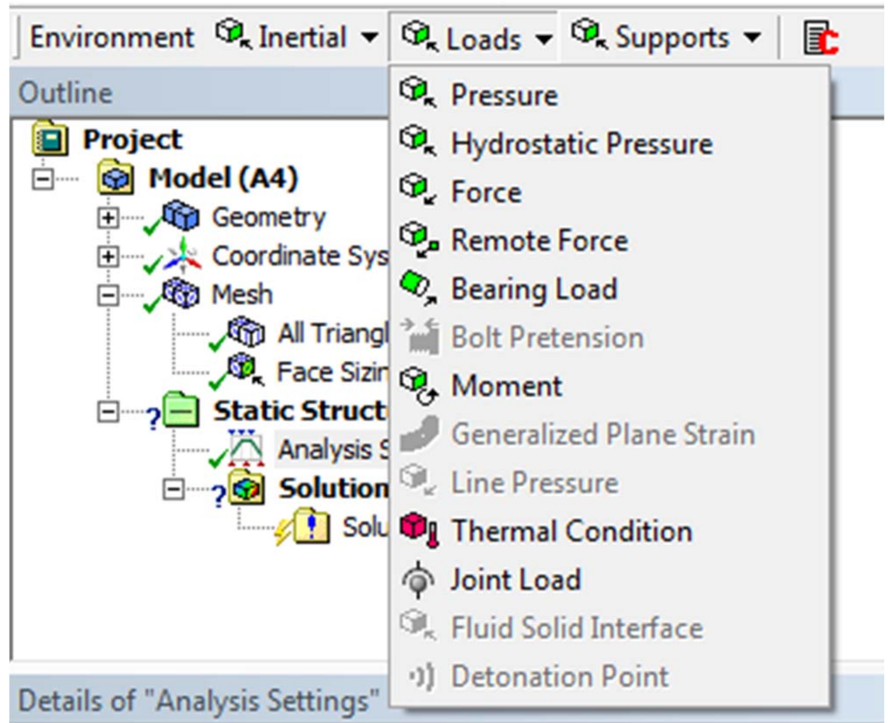
These are: Acceleration, Gravity and Rotational Velocity



Setup - Loads

Here we can select the loads that will be applied to the different elements of the structure.

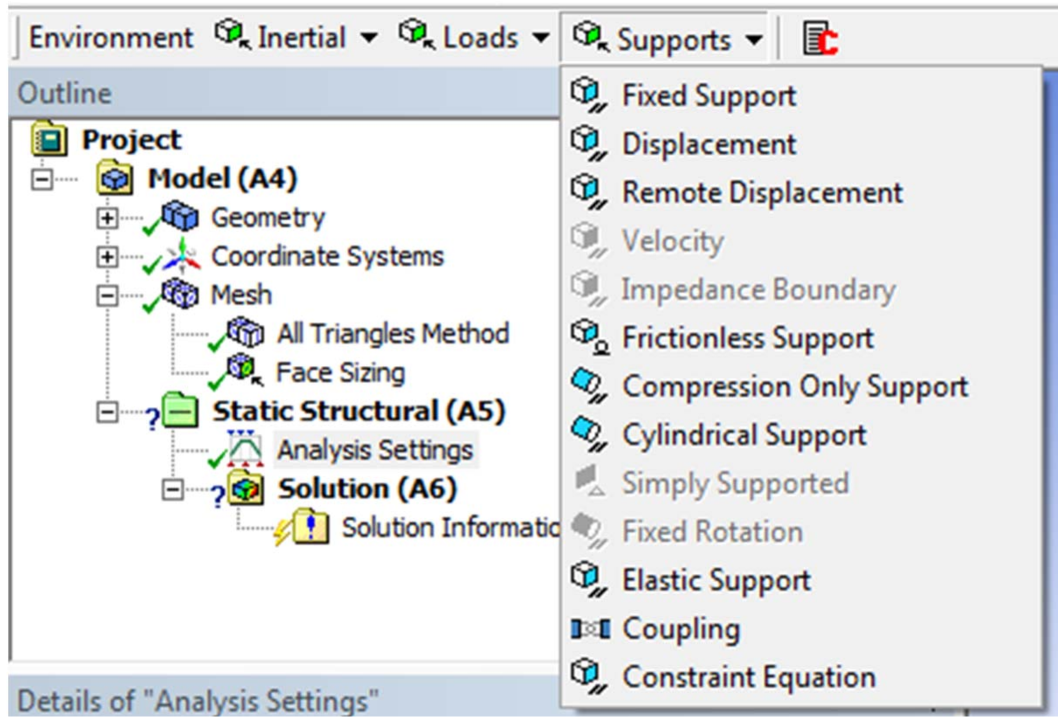
The most common are pressures and forces.



Setup - Supports

Fixed displacements can also be used as supports if the displacement length is defined as zero.

If close to the support appear stress concentrations, a good solution is to add an elastic support with a very high stiffness.



Setup – Weak Springs

Weak Springs:

If a structure has not enough supports, it becomes unstable (it moves!)

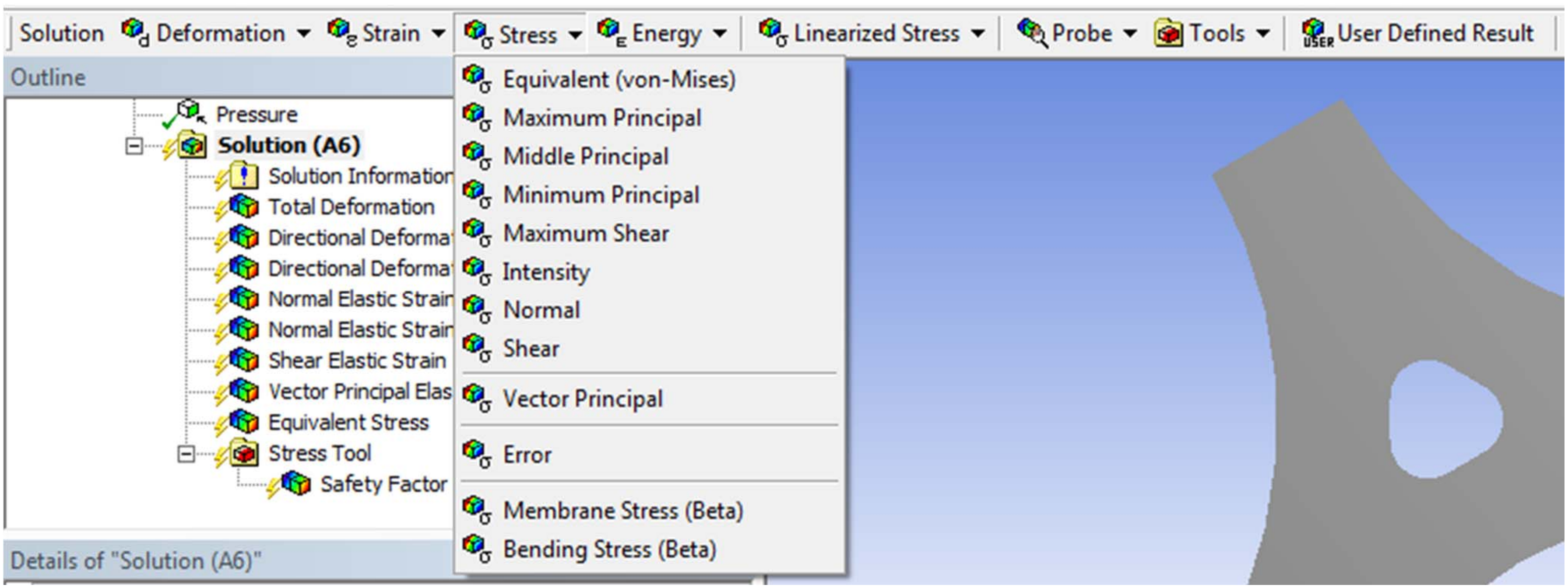
If Ansys Workbench detects an unstable structure, it adds "weak springs" on it to make it capable of withstanding small external forces.

Although with this procedure it is possible to obtain a solution, it is **strongly recommended to give enough supports to the structure** to avoid errors in the solution.

Results

Before performing the calculation, we have to tell the code what results we want to see afterwards.

Among the different possibilities are the ones shown below:



Results

Averaged vs. Unaveraged results:

Contour results in the Mechanical application are displayed, by default, as averaged results.

Averaged contours will average elemental nodal results across element and geometric discontinuities but will never average results across bodies.

Unaveraged contours display as element nodal contours that vary discontinuously even across element boundaries. These contours are determined by linear interpolation within each element and are unaffected by surrounding elements. Those results provide the *"exact"* solution obtained by the solver.