

ANSYS 11 - Crank Step 2

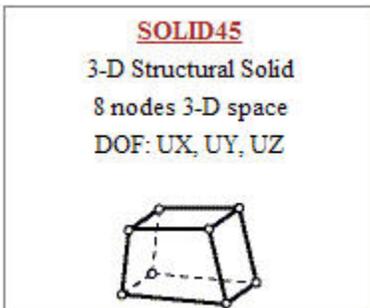
Problem Specification

1. Start-up and preliminary set-up
- 2. Specify element type and constants**
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Step 2: Specify element type and constants

Specify Element Type

We next select the appropriate element type(s) for our problem from a large list of about 200 candidates. Consider this as equivalent to rifling through a sizable toolbox, picking out one or more tools and placing them on a table for later use (in step 5, in our case). To see which element types are appropriate for this problem, bring up the pictorial summary of element types: **Utility menu > Help > Help Topics**. Search for "pictorial summary" and double-click on the search result titled **3.2 Pictorial Summary**. Click on the link to **SOLID Elements**. These are the element types you can use to mesh a solid volume. Check out the **SOLID45** element type. It is a brick-shaped element of the type referred to as "hexahedral" or "hex". It has a node at each corner and each node has three degrees of freedom: displacement in the x, y and z directions.

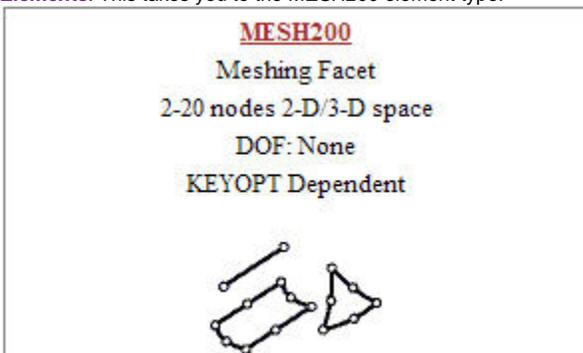


Click on the **SOLID45** link in the pictorial summary. This takes you to the help page for this element. Read through the juicy information at the beginning of this help page. Note that there are no real constants to be defined in our case. Click on **SOLID45** in the statement "See SOLID45 in the Theory Reference for ANSYS and ANSYS Workbench for more details about this element" which appears near the top of the help page. Click on **Equation 12-188**. This shows you the shape function for the element i.e. the equation used to determine the displacement at a general point within the element from the displacement values at the 8 nodes.

We will create our volume mesh in two steps:

1. Mesh the front surfaces of the crank as well as the pedal shaft.
2. Extrude these surface meshes to get the corresponding volume meshes.

This is analogous to creating a sketch and then extruding while making a solid in a CAD package. Since we cannot mesh *surfaces* with **SOLID45**, we need an additional element type called **MESH200**. Go back to the pictorial summary of element types. Scroll to the top of the page and click on the link to **MESH Elements**. This takes you to the **MESH200** element type.



Click on the **MESH200** link in the pictorial summary to see the help page for this element. You see the following information:

"MESH200 is a "mesh-only" element, contributing nothing to the solution. This element can be used for the following types of operations:

- Multistep meshing operations, such as extrusion, that require a lower dimensionality mesh be used for the creation of a higher dimensionality mesh"

In our case, meshing the two front surfaces with *MESH200* elements can be thought as going to these surfaces and marking out points and lines with a pen to show ANSYS where to put the corresponding *SOLID45* nodes and element faces. The *SOLID45* nodes and elements are actually placed at these pre-marked locations in the extrusion step. In effect, *MESH200* provides greater control over the mesh without actually contributing to the solution.

Referring to Figure 200.1 in the *MESH200* help page, we see that this element type comes in 12 different flavors. For our purposes, we will be using the **3-D quadrilateral with 4 nodes** option since this corresponds to an element face of *SOLID45*. The help page indicates that this option is selected by setting **KEYOPT(1) = 6**. Note that there are no real constants to be defined for *MESH200*.

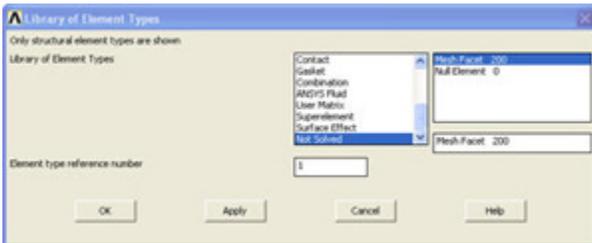
Minimize the ANSYS *Help* window. Select

Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add...

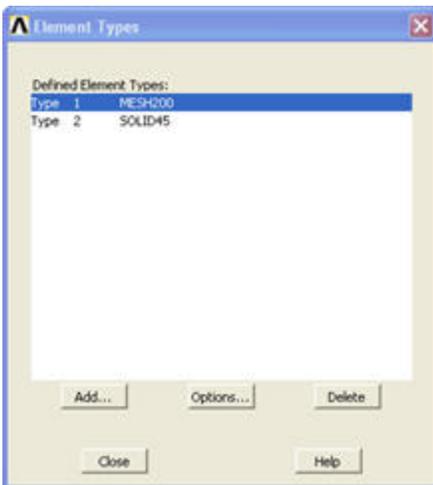
Pick **Structural Mass Solid** in the left field and **Brick 8node 45** in the right field. Click **Apply** to select this element.



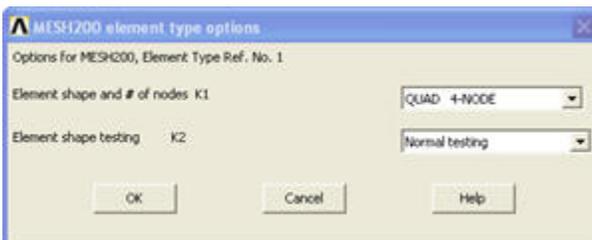
Scroll down the left field, pick **Not Solved** in this field and then **Mesh Facet 200** in the right field. Click **OK** to select this element.



The *Element Types* window should list two types of elements: **MESH200** and **SOLID45**.



In order to set the **3-D quadrilateral with 4 nodes** option for *MESH200*, click on **MESH200** and then **Options ...** in the above menu. Then, select **QUAD 4-NODE** next to **Element shape and # of nodes K1**. Click **OK**.



Close the *Element Types* menu.

Specify Element Constants

There are no real constants to be set for either element type as noted above.

Save your work

Toolbar > SAVE_DB

[Go to Step 3: Specify material properties](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)