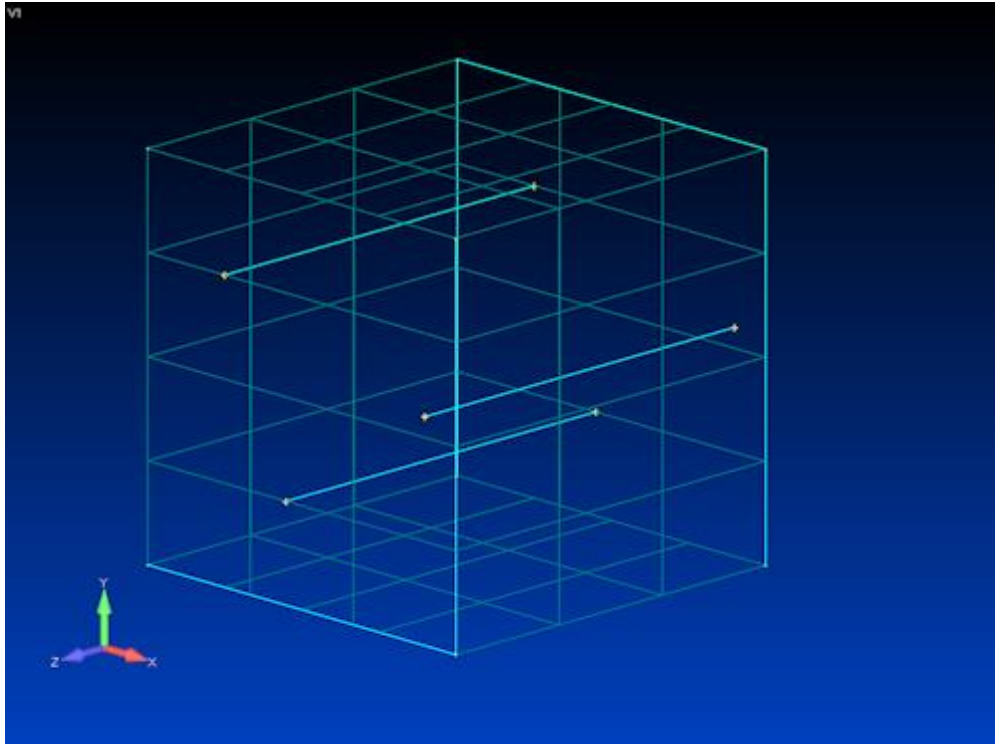


## Femap Tips and Tricks: Solid Mesh Internal Beams

Structures like reinforced concrete where steel rebar is encased in solid concrete can be represented by 1-D rod or beam elements for the rebar inside solid 3-D tet-elements for the concrete. When setting up such a model you have to take care to ensure that the mesh is contiguous and that the 1-D elements connect with the 3-D elements correctly.


For clarity, we'll demonstrate how this may be set up in Femap using a simple model of a solid block that will represent the concrete that contains three internal curves that will represent the rebar.




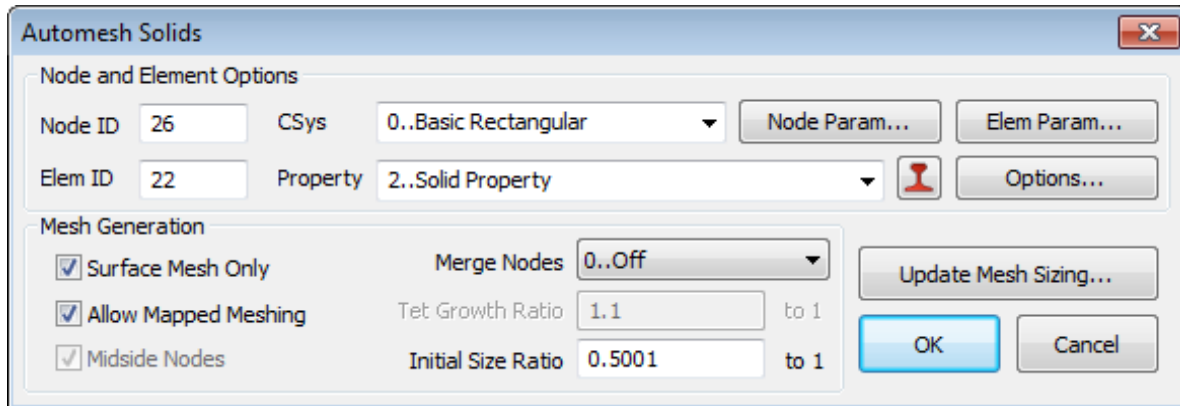
First, we need to create Mesh Points on the surfaces where the internal curves connect to ensure that the surface mesh will create nodes at the internal curve positions.

Select **Mesh / Mesh Control / Mesh Points on Surface...** and pick the first surface to which the internal curves connect. Select **Use Existing Points** and click **OK**, then pick all of the curve end points on the selected surface and click **OK**. Now repeat for any other surfaces that connect with curve end points.


Now we can mesh the internal curves to create rod elements in this case, as they will adequately represent the tension / compression properties of the rebar.

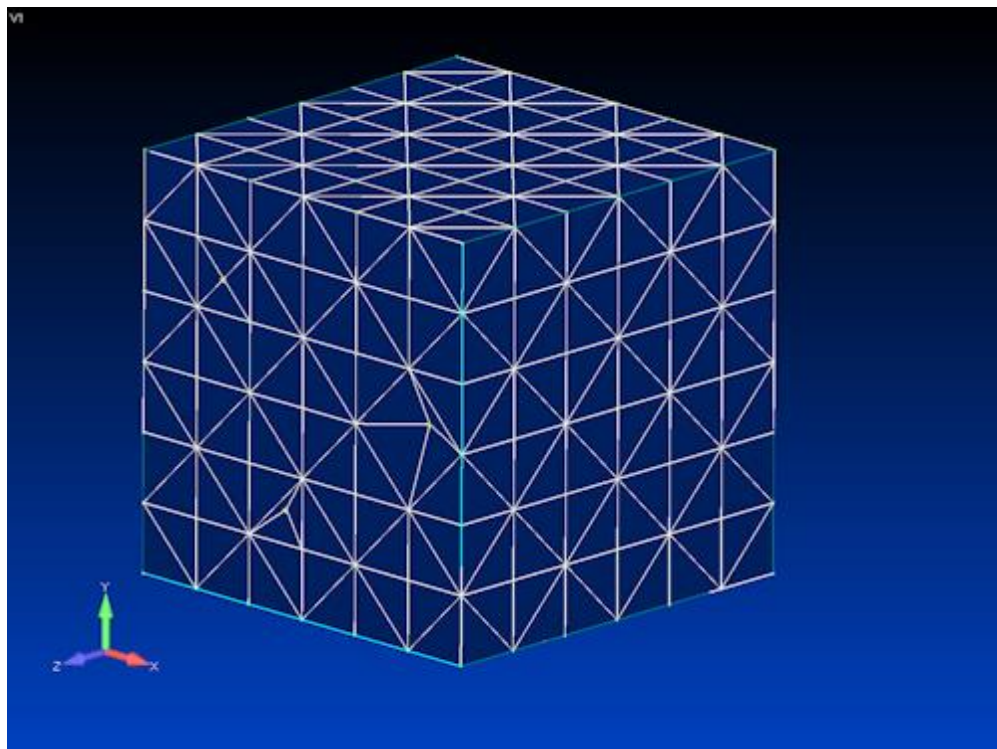
Select the **Mesh Curve** icon , and select all internal curves. Enter an appropriate property type and click **OK**.

The next step is to mesh the bounding surfaces. Select the **Tet Mesh Solid** icon , and pick all the bounding surfaces and click **OK** to close the *Entity Selection* dialog. In the *Automesh Solids* dialog activate the **Surface Mesh Only** checkbox.



Ensure that an appropriate property is entered and click **OK**.

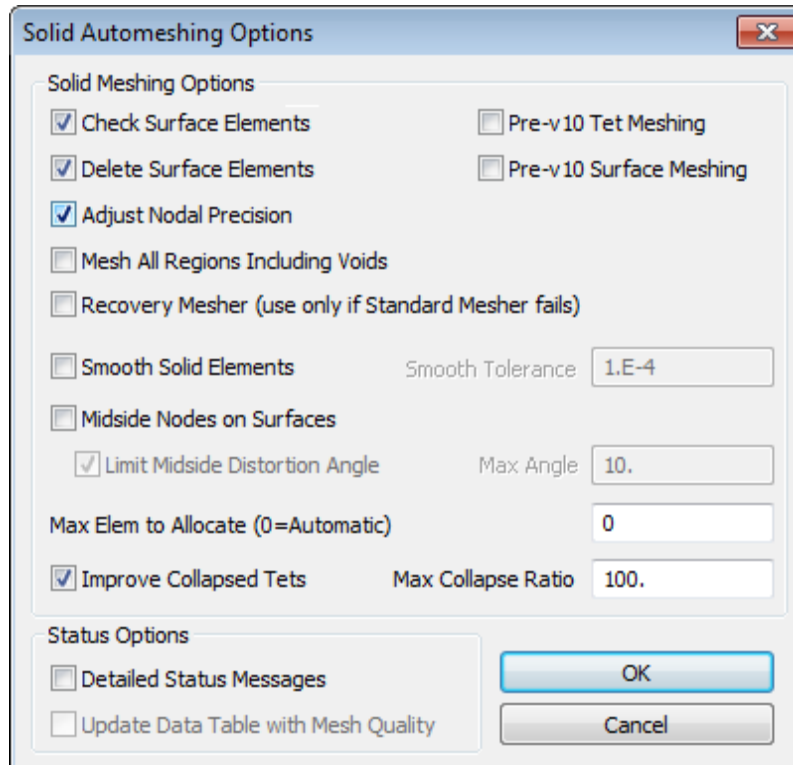
To verify that the meshing points have been picked up correctly select the **Wireframe** icon  in the dropdown menu of the *View Style* icon in the *View* toolbar.



Click the Undo icon  to revert to the previous view style.


Now with the curves and surfaces meshed in separate commands there will be coincident nodes at the mesh points that will have to be removed. Select **Tools / Check / Coincident Nodes...** and Click the **Select All** and **OK** buttons. Clicking the **Preview** button at this point will show all of the coincident nodes found. Click **OK** to complete the command.

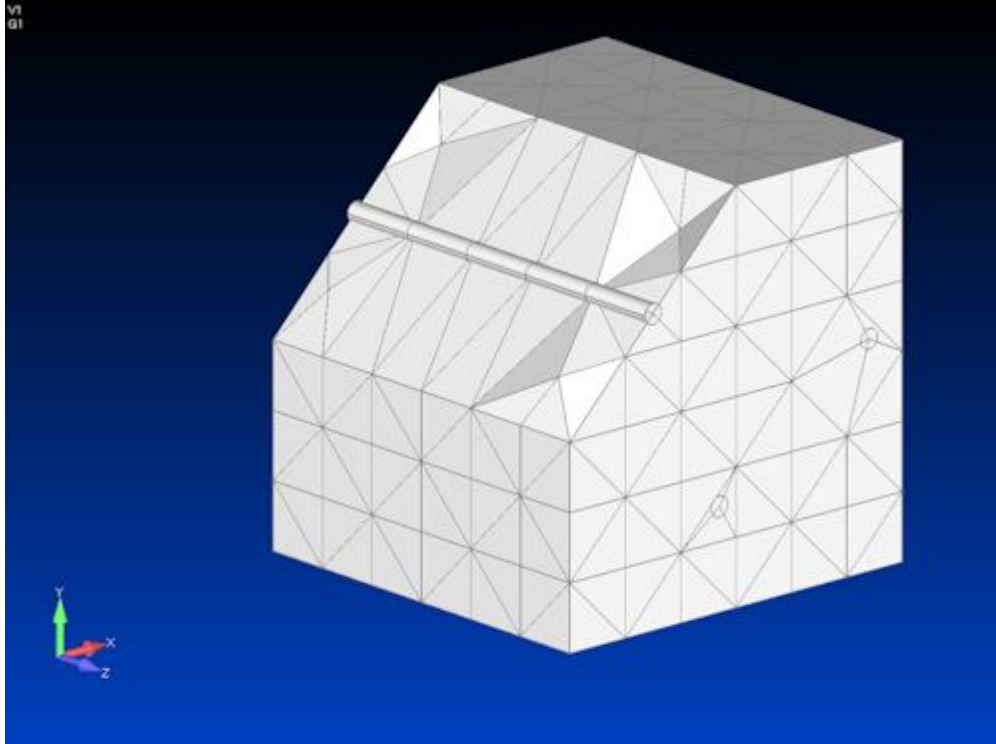
Now we can create the solid internal mesh. Select **Mesh / Geometry / Solid from Elements...** and click **Select All** and **OK**. In the *Automesh Solids* dialog click **Options** then in the *Solid Meshing Options* dialog verify that **Delete Surface Elements** is checked.



Click **OK** and **OK** again to complete the mesh. If a warning message appears requesting to proceed with the mesh, click **Yes**.

Verifying the interior of a solid model visually can be challenging, but in Femap we can use the grouping function to create a subset of the model that can be blanked off. In this case, we'll put the corner elements of the block into a group such that when they are blanked off one of the interior lines of rod elements will be revealed.

Select the **Solid** icon  in the dropdown menu of the *View Style* icon in the *View* toolbar. Now we orient the model such that a selection of elements can be made that can reveal a line of the internal rod elements. Select **Group Create/Manage...** to activate the *Group Manager* dialog, click **New Group...** and enter a **Title** then click **OK** and **Done**. Selecting **Group / Element / ID...** will allow a group of elements to be selected and added into the new group. In the *Model Info* tree expand the **Groups** entity, right click on the new group that's just been created and select **Show Active Group**.



It can be seen that a contiguous mesh has been created correctly linking the internal rod elements.

You can watch the video of this Femap tip on [YouTube](#).