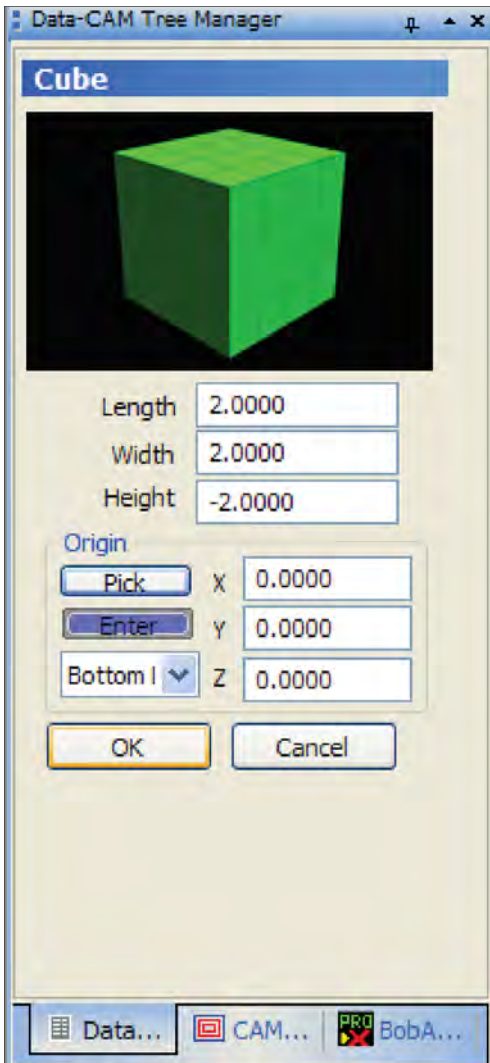


## FAQ #15: How do I set a boundary for 3D toolpath?

When machining it is not always desirable to allow the toolpath to extend all the way and cover an entire surface or solid. Often only a section should be machined at one time, be it because of fixture constraints or some other reason. These are the times when it's useful to use a boundary to keep the generated toolpath within a certain area. BobCAD-CAM allows the user to select any 2d boundary shape for use as a boundary.

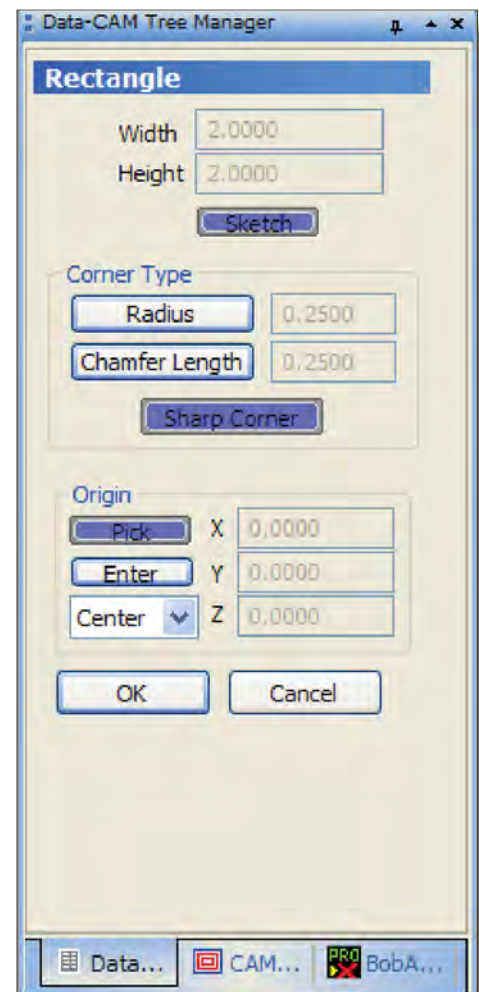
**Step 1.** A 3D surface or solid is required, so if there is already a 3D shape on the screen, skip to **Step 2**. Otherwise, draw a simple cube as shown here.



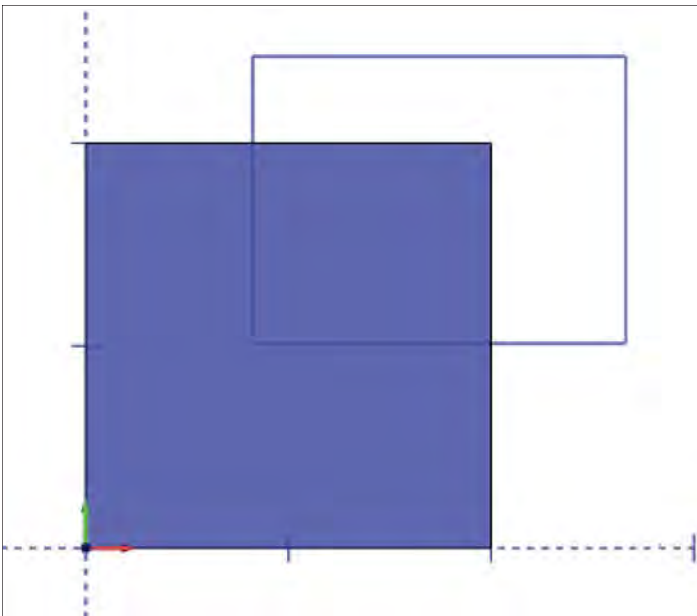
Click on **Solids** in the main menu and choose **Cube**. In the dialog to the left, enter in 2 for both **Length** and **Width**, and -2 for **Height**. The -2 ensures that the bottom of the cube is below Z0. Leave the **X**, **Y**, and **Z** under **Origin** all at 0, and choose **Bottom Left** for the corner location. Click **OK** and a cube will be drawn in the **Workspace**. The cube will serve as the part model for the toolpath.

**Step 2.** The next step is to draw a boundary. Click on **Other** in the main menu and choose **Rectangle**.

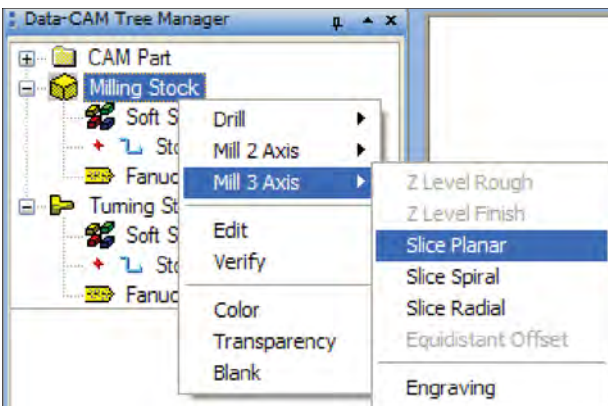
In the dialog to the left (shown to the right here), click on **Sketch** near the top of the box, and then on **Pick** in the **Origin** section. For this example it's fine to leave the **Corner Type** on **Sharp Corner**.



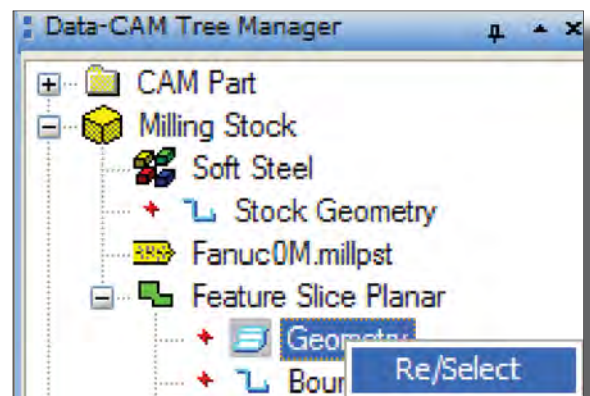
Click somewhere off of the cube to start the rectangle, then again when the box covers part of the cube drawn in the previous step. The rectangle will serve as the boundary for the toolpath generated in **Step 3**. The part in the **Workspace** should resemble the illustration below, but if using an existing part and boundary the important idea to remember is that any closed 2D contour can be used as the boundary for any 3D surface of a solid part. The steps remain the same even if the geometry itself is completely different.

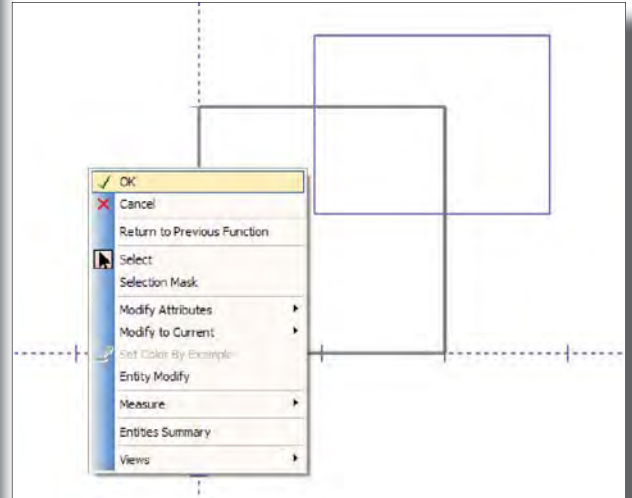
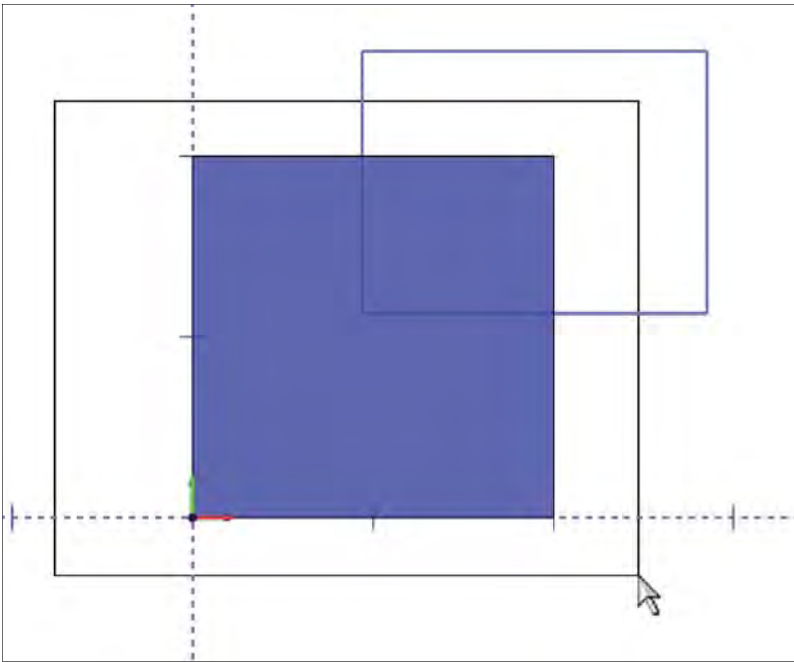


**Step 3.** Click on the **CAM Tree** tab of the **Data-CAM Tree Manager**. Right-click on **Milling Stock**, hover the mouse cursor over **Mill 3 Axis** and choose **Slice Planar** from the fly-out menu. A “Feature Slice Planar” feature will be added to the CAM tree.

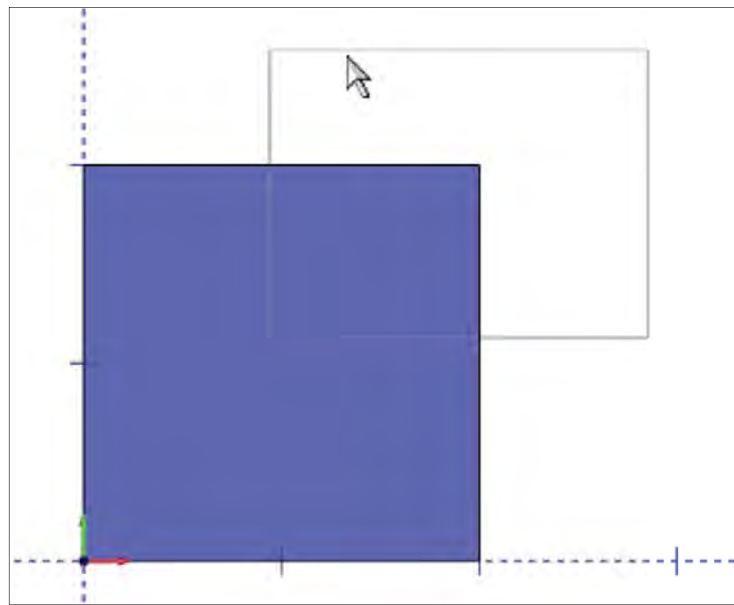
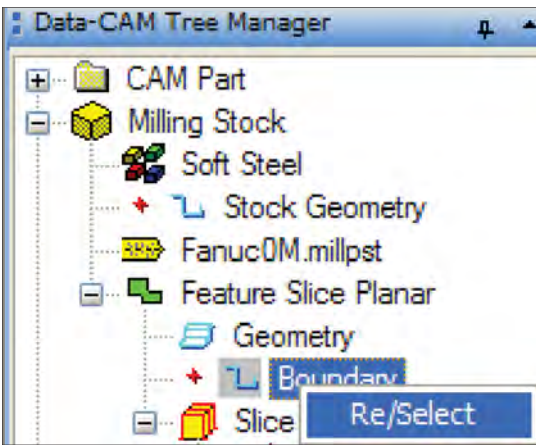


Right-click on **Geometry** and choose **Re/Select**. Then click somewhere off of the cube (or choose the appropriate surfaces if using a real part), and while holding the mouse button down, cover the entire cube with the selection rectangle and release it. The cube should highlight. Right-click anywhere in the **Workspace** and choose **OK** from the menu that will appear.

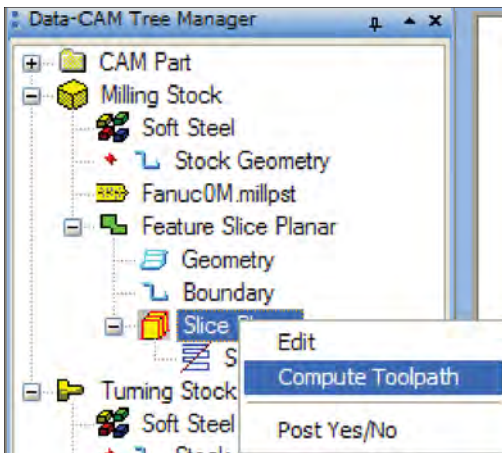




Next, right-click on **Boundary** and choose **Re/Select**. Hold the Shift key on the keyboard and click anywhere on the rectangle (or any other boundary geometry that may be needed). The entire boundary should highlight. If it only partly highlights, make sure the boundary is a closed 2D shape before continuing on. Right-click in the **Workspace** and choose **OK** again.



The actual parameters for the feature are irrelevant for the exercise, but at this point the user would normally right-click on the feature and edit the parameters. For this example, simply right-click on **Slice Planar** in the tree and choose **Compute Toolpath**.



When the toolpath finishes computing, it will be drawn over the cube *but only inside the boundary*. For more illustrative viewing, this example has been rotated in 3D space.

