

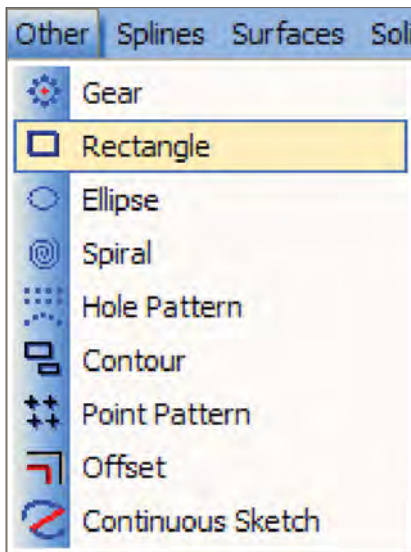
## FAQ #2: What are contours, and how do I use them?

A contour entity in BobCAD-CAM is an overlay type of entity that gathers selected geometry into “groups” that can be easily controlled. They are the simplest way to ensure chain direction for profile cuts and make sure that tool offsets occur on the correct side of a profile.

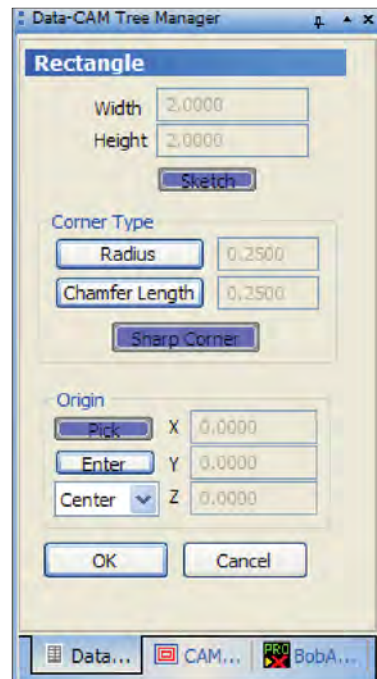
To create a contour entity from a wireframe shape, follow these simple steps.

**Step 1.** There must be pre-existing geometry on the screen to create a contour entity. If there already is, skip ahead to **Step 2**. Otherwise, draw a sample rectangle in the **Workspace**.

Click on **Other** in the main menu and then on **Rectangle**.

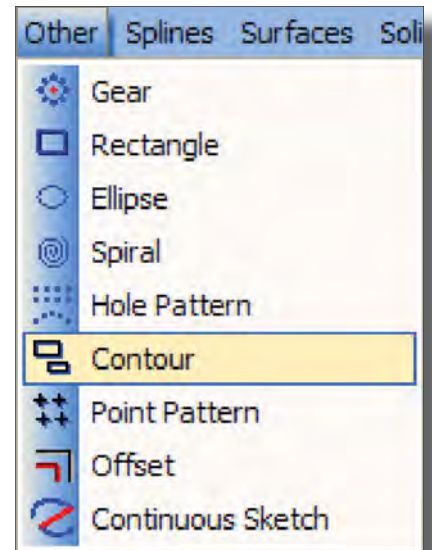
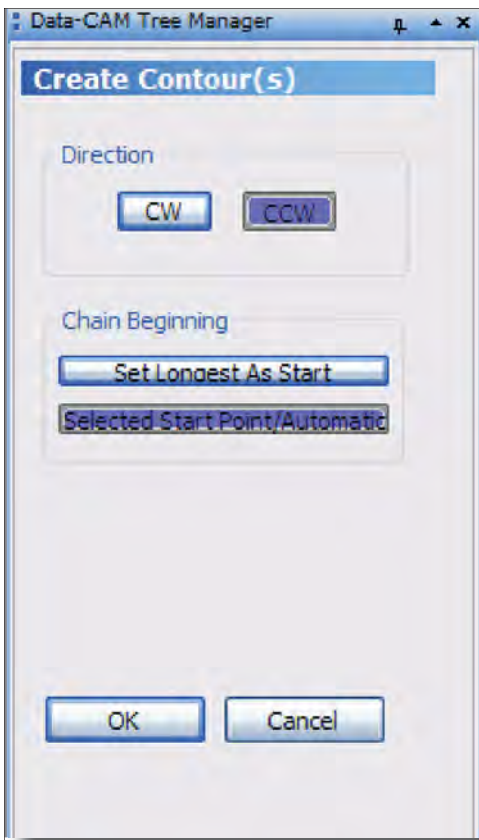


Make sure that both **Sketch** and **Pick** are selected since the actual placement of the rectangle doesn't matter.



Click somewhere in the **Workspace** for the first corner of the rectangle, then again to set the opposite corner as illustrated here. Click **Cancel** in the **Data-CAM Tree Manager** to the left to quit the function.

**Step 2.** Click on **Other** in the main menu and then on **Contour**.



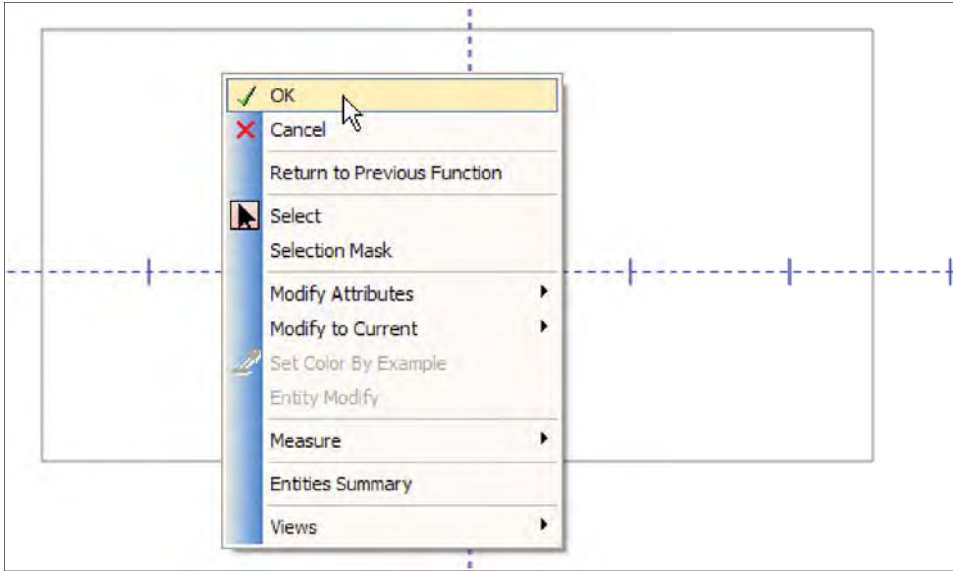
In the dialog that will appear in the **Data-CAM Tree Manager**, click on **CCW** under direction and choose **Selected Start Point/Automatic** under **Chain Beginning** to start the contour entity where the mouse is clicked on the shape.

The default selection under **Chain Beginning** is **Set Longest as Start**. If that were left chosen, the contour would have started at the beginning of the longest line on the rectangle after the start of the selection.

Hold down the Shift key on the keyboard and click the rectangle as below. An important item to remember is that the selection will end where the cursor is on the entity, so the contour entity will effectively end where the cursor is, not start there.



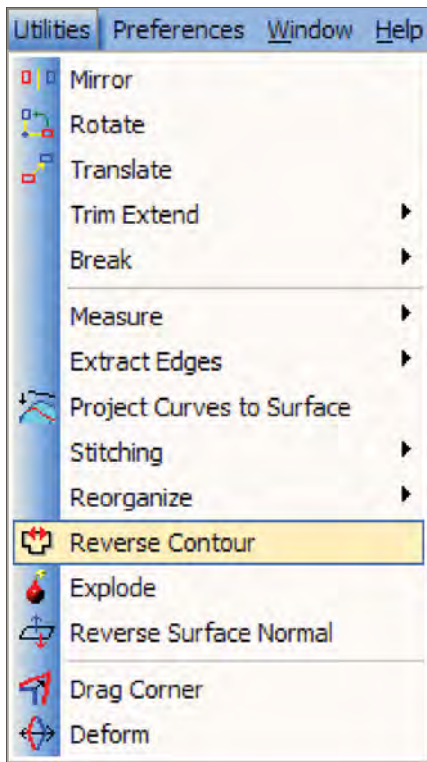
Once the rectangle has been highlighted, right-click in the **Workspace** and click **OK**. The contour will have been created as shown here.



### Reversing the Contour Direction

In the example above, the contour was left with a counter-clockwise direction. If this were an inside profile to be cut, this would be correct. However, if a climb-mill on an outside shape is required, the direction of this contour entity will need to be reversed. This is simple to do and takes only one step as shown here.

**Step 1.** In the main menu, select **Utilities** and then **Reverse Contour** as shown here.



Click on the contour to be reversed to highlight it, then right-click in the **Workspace** and select **OK**.



BobCAD-CAM will reverse the contour as shown, and will maintain the same start position on the contour if possible.



This contour would now be ready to select as the geometry for a **Profile** feature in the CAM tree or any other feature that requires a contour entity to operate properly.