

Solid Edge Sketch Orientation

This Tech Tip will look at how to orientate a sketch view to a part edge for easier placement of lines.

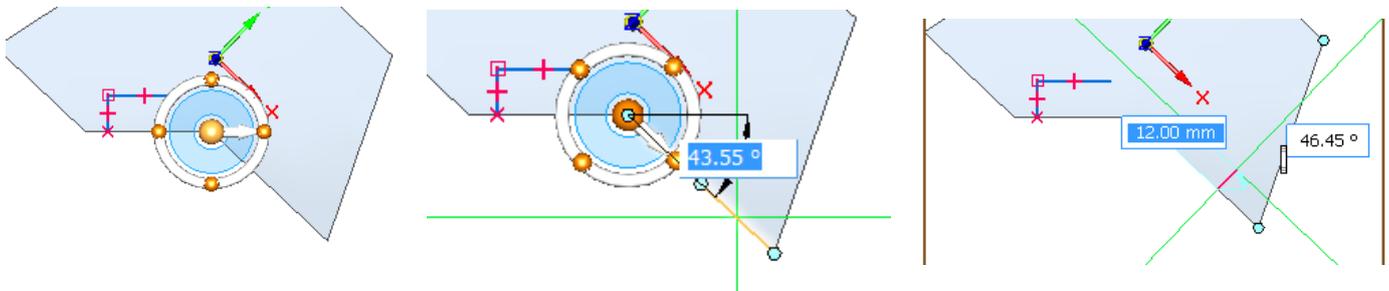
There are times when a part will have edges that are non-orthogonal (see highlighted edge in this part), but you wish to create a cutout perpendicular from that edge.

The first step is to orient the sketch plane to the face that you require. The appearance is different with Synchronous and Ordered, but the commands are the same.

In Synchronous mode, select the 2D command that you wish to use (which will typically be line) and position the cursor over the face you wish to draw on (but don't click or press the F3 key). One of the edges on the part face will be highlighted in green and this is the edge the x-axis will orient to. To change the orientation, click the "N" to move the orientation edge to the next straight edge in an anti-clockwise direction or "B" for a clockwise direction. In this case we need to press the N key once to get to the required face. Once you have the edge highlighted, press the F3 key to lock the plane. Use Ctrl H to switch to the Sketch View, if required.

Now with the cursors orientated correctly, you can create your lines for the cutout as perpendicular from the selected edge using normal horizontal/vertical alignments from intellisketch. Notice that the horizontal/vertical relationships are still shown, even though they don't correspond to the global orientation.

To complete this profile we need to change the orientation to the edge to the right of the initial one selected. Click the F8 key to reposition the origin and the steering wheel will be displayed. Use the central knob to position the steering wheel at the end of the line (shown left). Click the Steering Wheel Torus and click on the desired line. The alignment now changes to the new orientation.



In ordered, you can use the N and B key to orient a sketch plane, but once set, it is not possible to re-orient the plane like you can in synchronous mode.