Changing the angle of a ST sheet metal bend

This Tech Tip looks at two methods of changing the angle of a sheet metal bend while using synchronous technology.
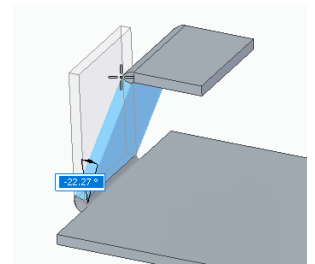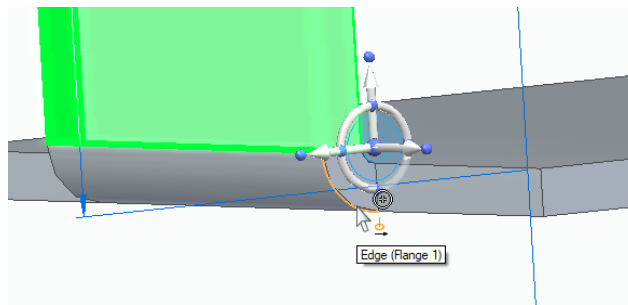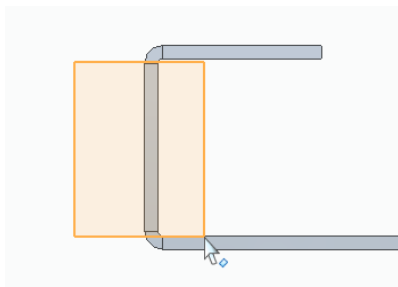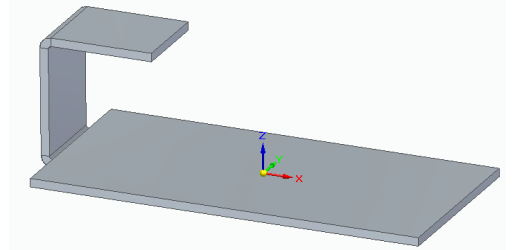
Option 1: Using the Steering Wheel

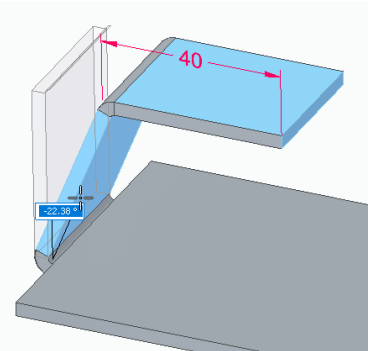In this example, there are two flanges, one horizontal and one vertical.

If you only wish to adjust the vertical flange, you need to fence select
the flange only, making sure that the bends are not included on the selection.

Then use the central knob of the steering wheel to position the centre of the rotation. By clicking on the edge of the bend (as shown), it should snap to the centre of the bend.

Finally, click on the torus of the steering wheel and rotate your flange to the new location. Note that the horizontal flange will not move, but just become shorter or longer, depending whether the angle is decreased or increased.
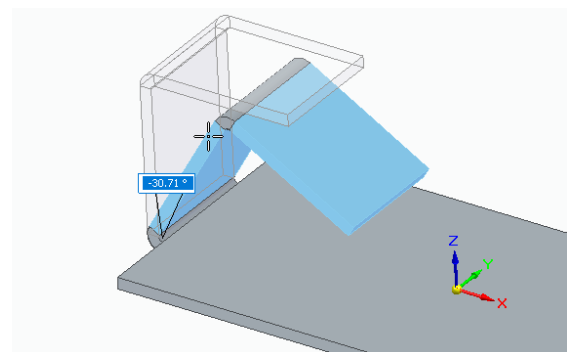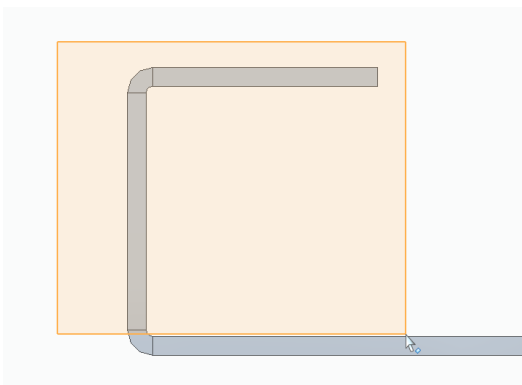
Note that if you want the length of the top flange to stay the same, simply add a locked dimension to the top flange.

If you wish to keep the horizontal flange perpendicular to the vertical flange, select both the side and top flange, including the bend in between, but not the bend linking it to the tab (as shown).
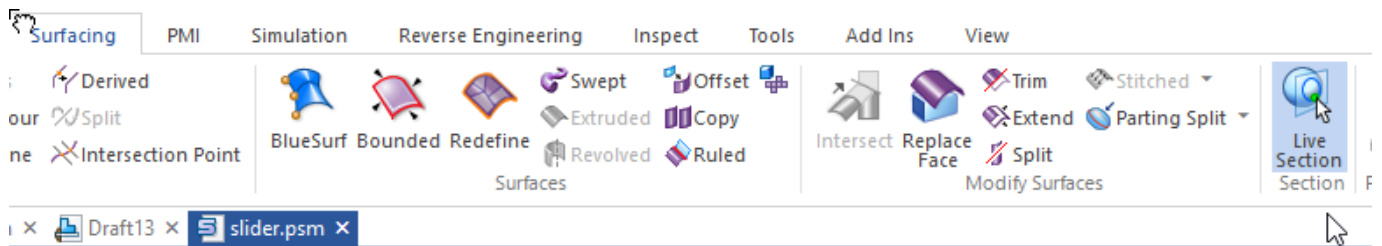
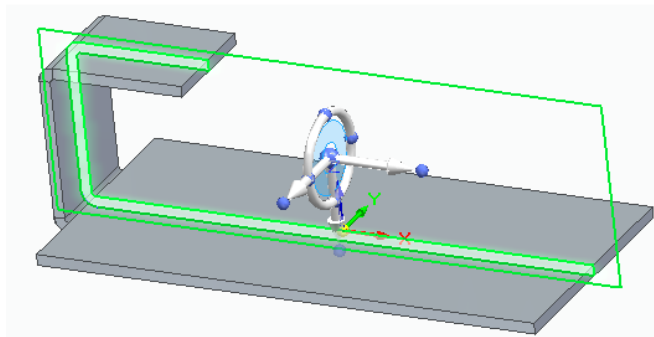Position the steering wheel as above and rotate the flange.

Option 2: Using Live Sections

An alternative to using the steering wheel is to create a live section, which enables the creation of dimensions easier, especially if the flanges are not orthogonal.
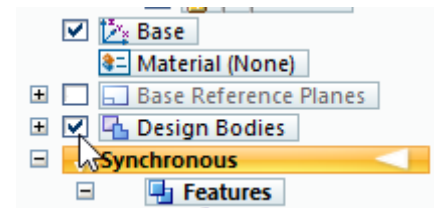
The Live Section command is found on the surfacing Tab.
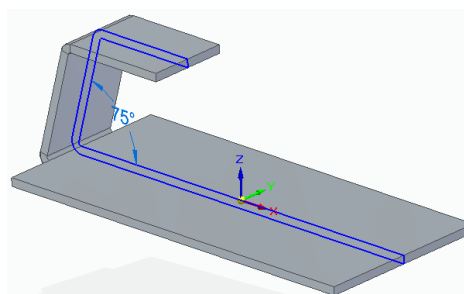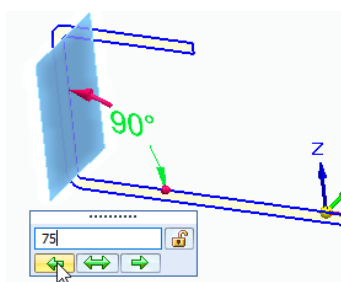

Start by selecting the plane that cuts perpendicularly through the flanges. Remember that you have the steering wheel if you need to reposition it.

If you wish, you can turn off the model display (untick Design Bodies in the pathfinder) and just use the Live Section (similar to a sketch in ordered mode) if you want to make things clearer.

Place the angular dimensions on the Live Section and alter to suit.

Once complete, turn on the display of the Design Bodies.