

SHEET METAL ENHANCEMENTS

BUMP BEND: Bump bend is small incremental bends in a specific area to achieve a radius. The more hits done the smoother the contour.

Where Feature would be shown and created:

Feature would be created in flat pattern view of a part. Option would be grayed out in 3D view but when selecting flat pattern it would be active.

Purpose of feature:

When creating a sheet metal part that contains a bend radius (a radius that is large in size compared to standard tooling radius) that cannot be achieved by standard tooling a technique called bump bend at the press brake is used to achieve the radius.

Using the bump bend technique you can achieve a course or fine radius depending on the amount of bends are made in this area

To help the press brake operator I laser etch (scribe) this area with center lines that go from tangent to tangent lines of the radius in small increments (evenly space). I have to manually add these lines in the .dxf file. The angle to hit each line is an educated guess done at the press brake.

The bump bend feature would automatically place these lines in the flat pattern and give the angle to which to hit each bend line to achieve the radius.

Real World application:

Scenario:

If a sheet metal manufacturer does not own a press brake tooling to achieve a large bend radius.

Values to enter to create feature:

When selecting, to execute this feature ,you would get the following options in the dialog box to enter values to create the bump bend lines.

Choose one of the following:

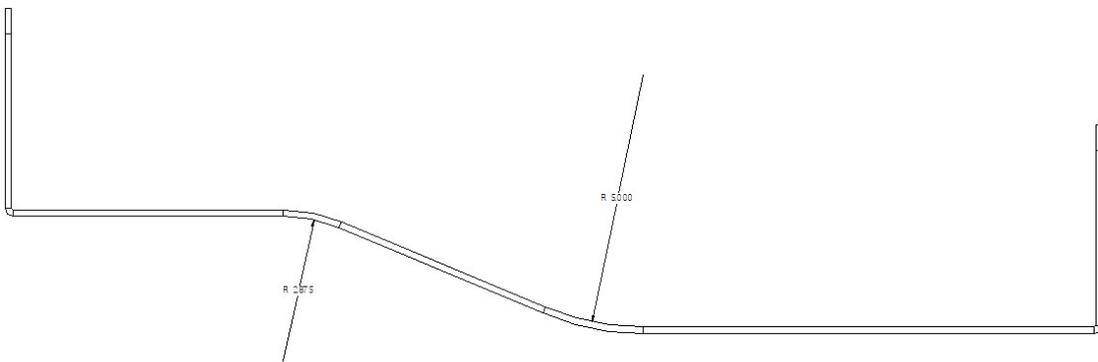
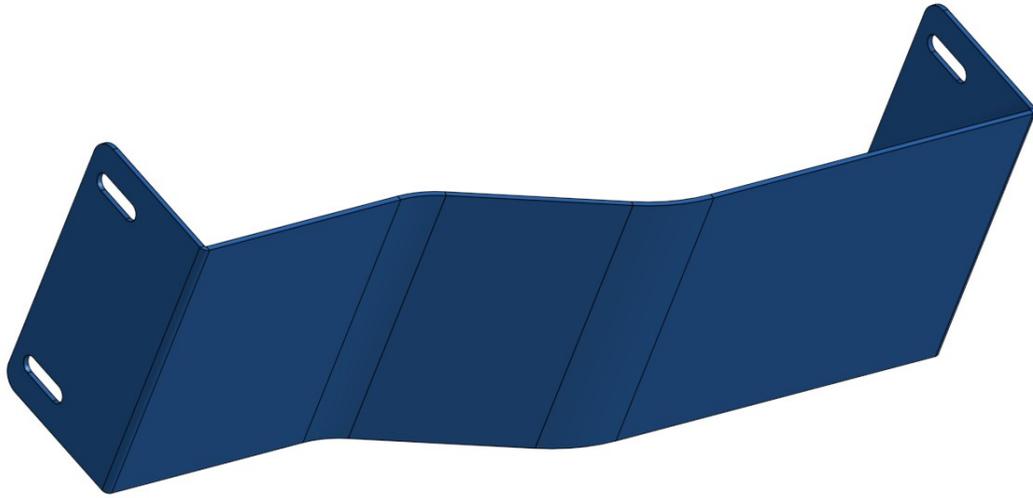
- Number of hits. The software would take the area and divide by hits and place that many bend lines evenly spaced.
- OR**
- Select course or fine. The software would use programmed algorithms to achieve a course radius or fine radius. Fine would give you more bend lines in this area than course.

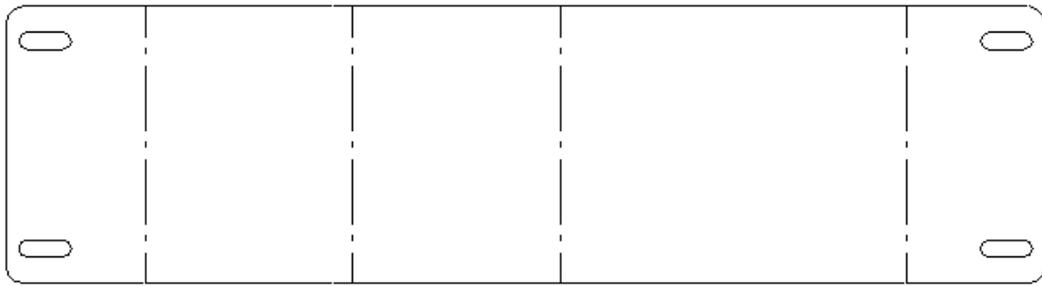
Profiles to select to create feature:

You would select the area to apply the bump bends. Bend lines would be placed as per the amount you enter or by a process of selecting course or fine.

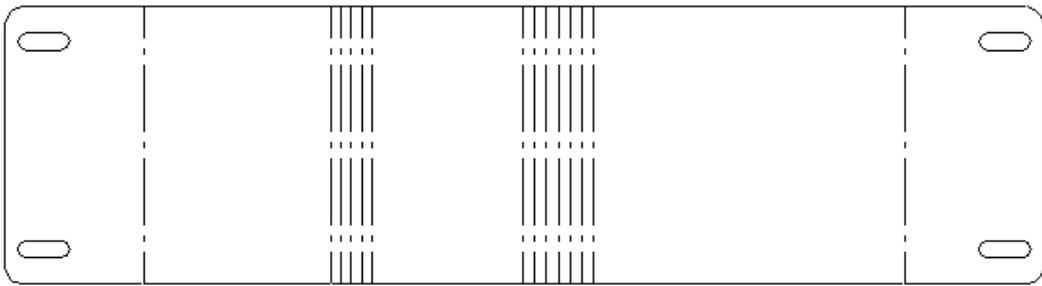
In the drawing of the flat pattern you would get the bend lines and the bend angle for each line needed to achieve the overall radius.

Example:





WHAT YOU CURRENTLY GET



WHAT THE BUMP BEND COMMAND WOULD GIVE YOU