

EDU[**CAD**]TION **STUDY GUIDE**

How to Configure and Prepare Bends in CAD

Chapter 5, Lesson 8

SendCutSend

Starting the Design

- Begin with a simple sketch, for example, a 5×5 inch square profile.
- Navigate to the Sheet Metal workspace and select '**Create Flange**' (see Figure 1.1).
- Choose the **profile** and adjust the dialog box to define the flange **orientation** (Side 1, Side 2, or Center) and select **Sheet Metal Rules** (see Figure 1.2). **Note:** For consistency, most designers stick with Side 1 orientation.
- When creating flanges, always make a **new component** instead of adding to an existing body.
- Each component retains its own sheet metal rules, allowing you to edit thickness, K-factors, and radii independently. If you only create a body, these rules will apply globally and limit flexibility.

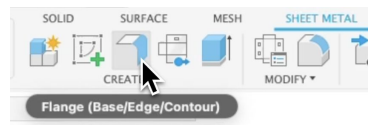


Figure 1.1

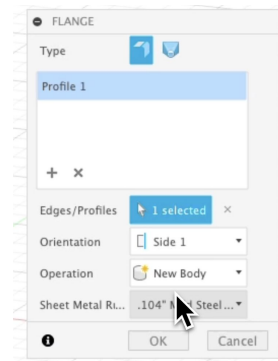


Figure 1.2

Switching Sheet Metal Rules

- One easy way to switch Sheet Metal Rules is in your **Browser Tree** → **Rule** → **Switch Rule** (see Figure 1.3).
- A new Dialog box will pop up where you can select a new rule (see Figure 1.4)
- Always select the correct rule for your material: **e.g.**, '0.135" Mild Steel'. You can switch rules later if you change materials or thickness.

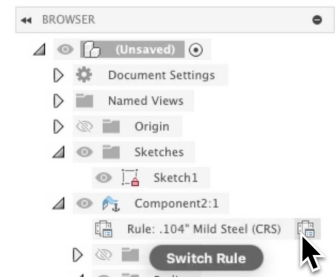


Figure 1.3



Figure 1.4

Create or Modify Sheet Metal Rules

To create or modify Sheet Metal Rules, go to **Modify** → **Sheet Metal Rules** (see Figure 1.5).

- A dialog box will appear showing the rules used in your current design and your rules library (see Figure 1.6).
- To edit an existing rule, for example, in this lesson we're using 0.104" Mild Steel, scroll through your library and click the **pencil icon** next to the rule name (see Figure 1.6).
- Once you've clicked the pencil icon, a new dialog box will appear where you can edit parameters such as (see Figure 1.7):
 - **Rule Name:** Custom name for identification.
 - **Thickness:** Must match your material specification.
 - **K-Factor:** Defines where the neutral axis lies during bending.
 - **Bend Conditions:** Includes bend radius, relief shape (round, tear, or straight), and relief dimensions.
 - **Corner Conditions:** Controls corner reliefs for intersecting bends (square or round).



Figure 1.5



Figure 1.6

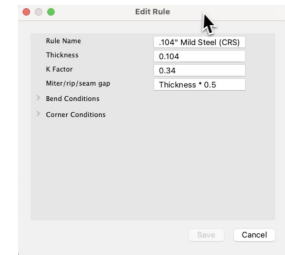


Figure 1.7

Creating Additional Flanges

- Use **Create** → **Flange** again to add more bends from the edges of your base flange (see Figure 1.8).
- Once you've added the bend, new options such as Height and Angle will appear in the dialog box (see Figure 1.9).
- You can specify the flange height and bend position relative to the part edge:
 - **Inner Face:** Bend starts inside the sketch line.
 - **Outer Face:** Bend offset outward.
 - **Tangent to Bend:** Bend originates from the bend's tangent point.

Measure bends from the outer face for consistent flange dimensions.

Note: Jake recommends going off the the outer face since this is the standard way to measure these bends.

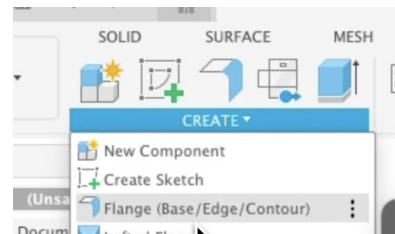


Figure 1.8

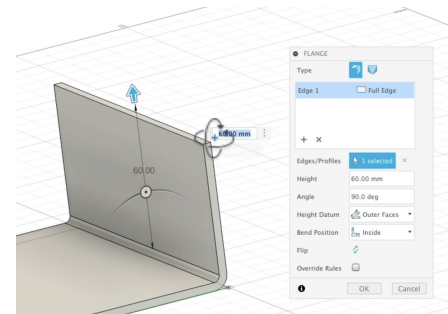


Figure 1.9

Working with Bend Reliefs in CAD

- To add bend reliefs, go to the **Timeline** → **right-click Flange** → **Edit Feature** (see Figure 1.10).

- A new dialog box will appear, where you can open the **Full Edge** dropdown to view additional options (see Figure 1.11).

Note: When you adjust the flange to 'Two Sides' instead of 'Full Edge', CAD automatically generates a bend relief slot.

- The reliefs are configured by the selected sheet metal rule.
- To edit them, return to **Modify** → **Sheet Metal Rules** and change the relief width or depth parameters.

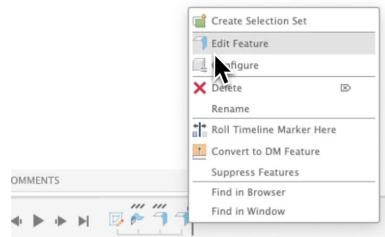


Figure 1.10

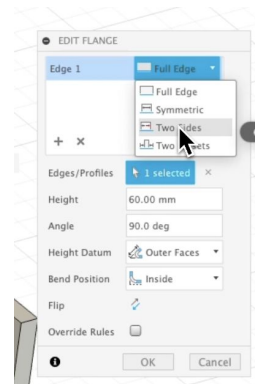


Figure 1.11

Corner Reliefs and Multi-Flange Corners

- When creating flanges that meet at a corner, CAD applies both bend and corner reliefs automatically, in this lesson it you can see we got both square reliefs (see Figures 1.12 and 1.13).
- There are two corner relief styles:
 - **Square Corner Relief:** Easier to weld, provides a tight corner.
 - **Round Corner Relief:** Smooth finish, often used for aesthetic parts.

Creating multiple flanges simultaneously can generate different relief results than creating them individually.

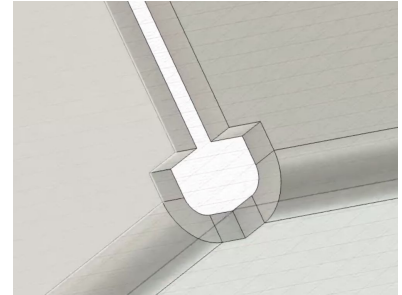


Figure 1.12

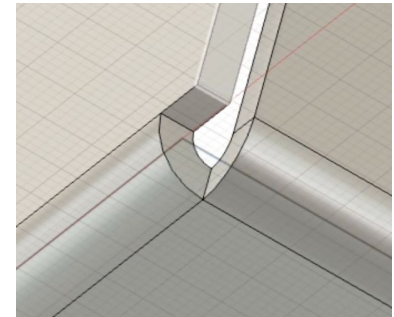


Figure 1.13

Adjusting Corner Conditions

- To modify corner relief type or size:

1. Open **Modify** → **Sheet Metal Rules**.
2. Select your material rule (e.g., 0.135 Mild Steel).
3. Under '**Corner Conditions**', switch between Square or Round (see Figure 1.14).
4. Click **Save** and your model will update automatically. In this lesson we switched from square to round (see Figure 1.15).

- Different materials use different relief values.

- Thicker materials require larger relief dimensions to accommodate bending deformation. For instance, 0.250" steel will show wider reliefs compared to 0.104" steel.



Figure 1.14

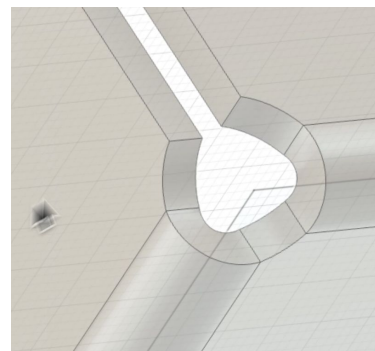


Figure 1.15

Exporting for Manufacturing

- Once your bends are properly configured, you can prepare your model for production in two ways:
 - **SendCutSend Step File:** Save the part as a .STEP file and upload directly to SendCutSend.
 - **Flat Pattern DXF:** Create a flat pattern (**Create** → **Flat Pattern**) and export it as a DXF file. This file includes bend lines and reliefs for laser cutting.
- To generate a **flat pattern** in Autodesk Fusion:
 1. Go to the **Sheet Metal** tab → **Create** → **Flat Pattern**.
 2. Select a stationary face (usually your base sketch face).
 3. Click OK, the part unfolds, showing bend lines, radii, and reliefs (see Figure 1.16).
 4. Export as DXF for SendCutSend or another manufacturer.

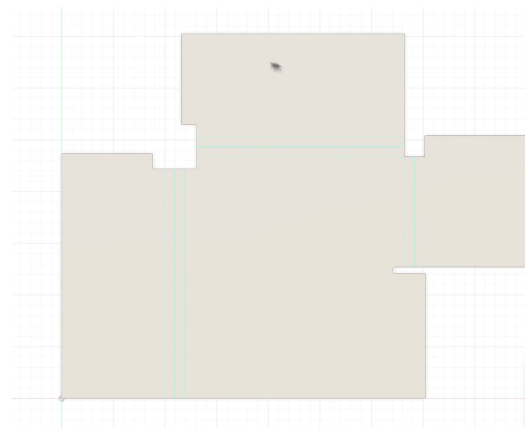


Figure 1.16

Bending Limitations

- Even with CAD automation, bending has physical limits:
 - **Bend Length:** Determined by machine tooling width.
 - **Material Thickness:** Thicker materials require greater tonnage.
 - **Tonnage:** Too much force can damage punches or dies.
 - **Angle Limits:** Some materials can't bend beyond specific angles without cracking or deforming.

U-Channel Ratio & Parallel Edges

- When creating U-channel bends (two opposite bends), maintain safe spacing (see Figure 1.17):
 - **Metal:** 2:1 base-to-flange ratio.
 - **Plastic:** 3:1 base-to-flange ratio.
- These ratios prevent tooling interference and ensure proper bending.

Parallel edges to bends:

- Machines require a parallel reference edge to align bends correctly.

Note: If your part lacks a parallel edge, add temporary tabs or geometry for alignment. If SendCutSend's DFM check requests a parallel edge, this is the reason why.

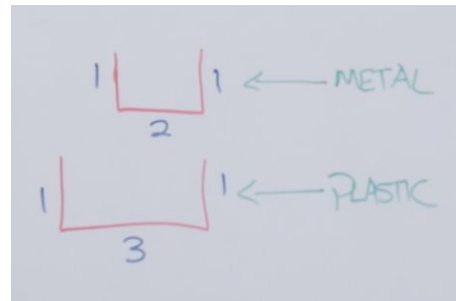


Figure 1.17

Summary

Configuring bends in CAD ties together everything learned about bend mechanics, reliefs, and tolerances. By using correct sheet metal rules, reliefs, and material settings, you can ensure parts that are accurate, manufacturable, and ready for SendCutSend fabrication.

Learn more at <https://sendcutsend.com/education/>