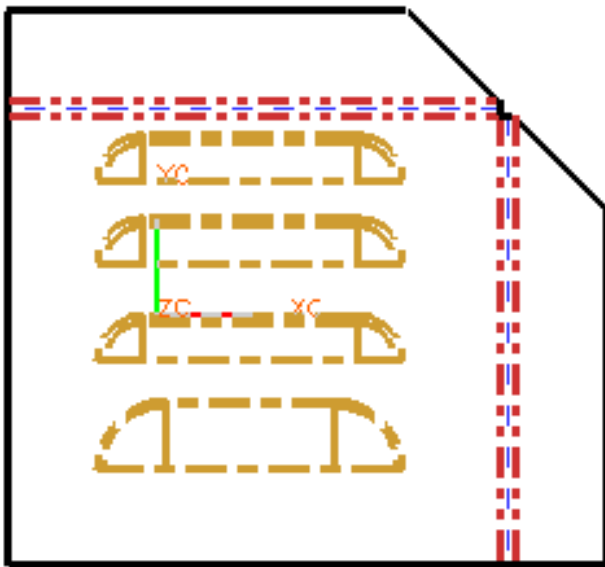




Flat Pattern overview

[Overview](#)[How To](#)[Options](#)

Use the **Flat Pattern** command to create a solid or wireframe flat representation of a 3D sheet metal part. You can:

- Include annotations for bending, punching, or cutting manufacturing instructions.
- Include annotations for automatic bend regions.
- Specify line color, layer, and fonts for the various line types (outline, bend centerline, bend tangents, and so on).
- Export the flat pattern through a translator to a cutting tool for the manufacture of the part.

Note: The recommended translator for exporting flat patterns to tools is **2D Exchange**.

Where do I find it?

Application

NX Sheet Metal

Toolbar	NX Sheet Metal→Flat Pattern list  →Flat Pattern
Menu	Insert→Flat Pattern→Flat Pattern
Menu	File→Export→2D Exchange



Create a flat pattern

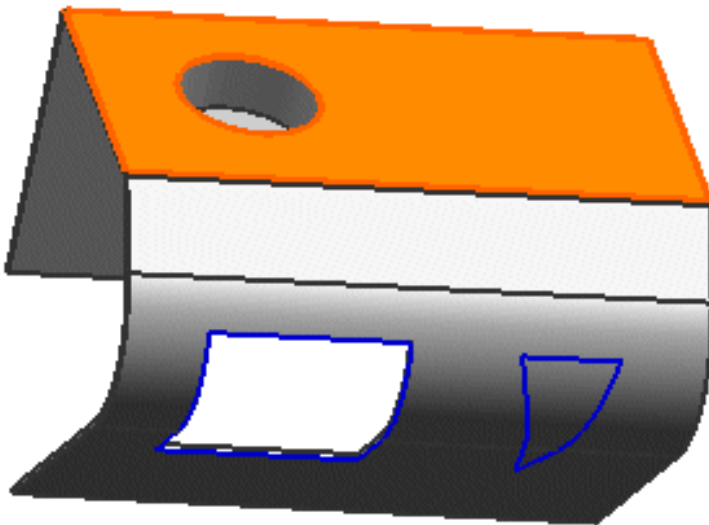
[Overview](#)**How To**[Options](#)

1. Open your part.

2. On the **NX Sheet Metal** toolbar, select **Flat Pattern** , or choose **Insert**→**Flat Pattern**→**Flat Pattern**.

In the **Flat Pattern** dialog box, **Select Face**  is active in the **Upward Face** group.

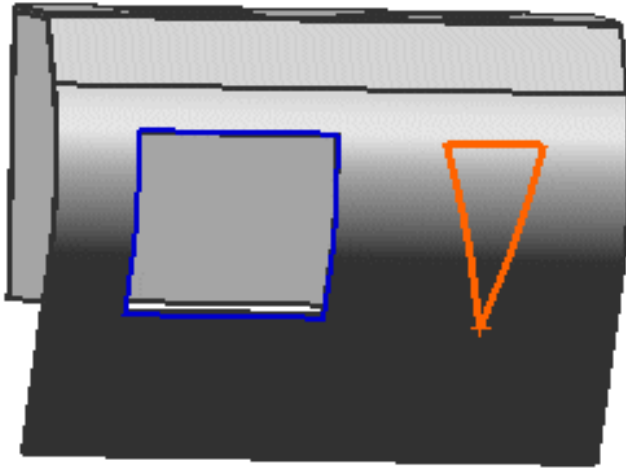
3. Select an upward face to orient the flat pattern. This face can either be on a formed sheet metal body, or on a flat solid body. The normal direction associated with this face is used to specify the upward direction for bend regions. In this example, the upper tab face was selected.



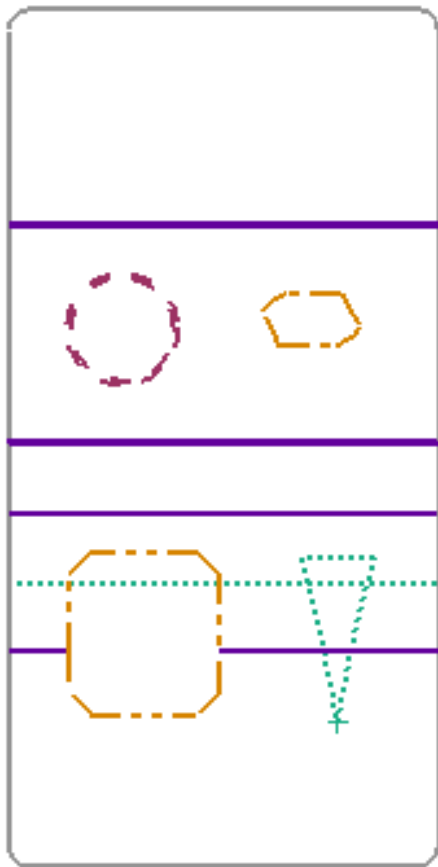
Note: A **Flat Pattern** feature needs a flat solid body as the basis for the flat pattern curves. So if you select a face on a formed sheet metal body as the base face for the Flat Pattern feature, a Flat Solid feature will be automatically created as the basis for the flat pattern curves. When you create a Flat Solid feature, the face you select on the formed sheet metal body will be used as the placement face for the Flat Solid feature and the base face for the Flat Pattern feature. As a result, this placement face eventually defines the upward direction

for the bend region annotations on the Flat Pattern feature.

4. Select **Additional Curves**, if any, that you want to display on your Flat Pattern. In this example, the 3D curves that make the triangle were selected.



5. You can round or chamfer interior or exterior corners to accommodate machining tools. In the **Outer Corner Properties** and **Inner Corner Properties** groups, clear the **Use Global** check boxes and supply values for **Corner Treatment**. In this example, the outer corners were given a radius of 5 mm, and the inner corners were chamfered 5 mm.
6. Click **OK** to create the feature.
7. To see the FLAT-PATTERN view, in the **Part Navigator**, expand the **Model Views** node, then double-click the FLAT-PATTERN. In this example, **Show Interior Feature Curves** was selected, so you can see the dimple and normal cutout with inner corners rounded. You can also see the **Additional Curves** in cyan.



Edit a flat pattern

To edit the orientation or placement of the Flat Pattern feature, you must edit the orientation of the Flat Solid feature that it references. This is true both for flat patterns created from flat solids and those created from a formed sheet metal body. The Flat Pattern feature is simply a grouped set of curves and associated annotations. You can add, remove, or edit curves in this group. However, any time you update the model, the curves associated with the Flat Pattern feature will be regenerated and will overwrite any changes. To make permanent changes to a Flat Pattern feature, you must first make a non-associative copy of the flat pattern curves and then edit that set of curves.

Export a Flat Pattern feature

Exporting a **Flat Pattern** feature through a translator requires a proper translator license. This license is not included as part of NX Sheet Metal.

You can export a flat pattern as follows. With the part that contains the Flat Pattern feature open:

1. Choose **File**→**Export**→**2D Exchange**. Select the **Files** tab. (This is the default.)
2. On the **Files** page, in the **Export to** group, select **DXF** or **DWG** from the **Output As** list. Specify where you want to save the new file.
3. Click the **Data to Export** tab. In the first **Export** list, choose **Selected Objects**.

4. In the **Model Data** group, click **Select Object** , then in the graphics window select the part.

5. In the second **Export** list, choose **Selected View**, select FLAT-PATTERN from the view list, and then click **OK** to export.



Flat Pattern options

Overview	How To	Options
--------------------------	------------------------	---------

Upward Face

Select Face	Lets you select a face on the part. The normal direction associated with this face is used to specify the upward direction for bend regions. It also defines top and bottom faces for Additional Curves .
-------------	--

Additional Curves

Select Curve or Point	Lets you select additional curves or points that you want to have displayed on the Flat Pattern feature. This option is useful for displaying features such as part numbers or company logos. It is also useful for representing other manufacturing instructions such as weld points. Edges of solids cannot be included. You can choose different display options for curves on top and bottom faces of the part (Added Top and Added Bottom on NX Sheet Metal Preferences → Flat Pattern Display).
-----------------------	---

Outer Corner Properties control the shape of outer corners of the part.

Corner Treatment	Use Global uses the values you set for the Outside corner treatments and Outside value options in the NX Sheet Metal Preferences dialog box. To find these options, choose Preferences → NX Sheet Metal . In the dialog box, click the Flat Pattern Treatments tab. The options are available under Corner treatment options .
	Treatment: • None leaves the corners as specified in the part. • Chamfer trims the outer corner edges a specified distance in from the corner. • Radius rounds the outer corners with a radius of the value specified.

	Value specifies the chamfer or radius of outer corners.
Inner Corner Properties control the shape of interior corners of the part.	
Corner Treatment	Use Global uses the values you set for the Inside corner treatments and Inside value options in the NX Sheet Metal Preferences dialog box. See the entry for Outer Corner Properties above for the path to these options
	Treatment: • None leaves the corners as specified in the part. • Chamfer trims the sharp angles of inside corners a specified distance in from the corner. • Radius rounds the inside corner with a radius of the value specified.
	Value specifies the chamfer or radius of inside corners.
Settings	
Show Interior Feature Curves	Displays the curves that define interior features such as dimples, punches, or cutouts.

Because most companies have standards for their flat patterns, you may want to establish default settings for flat pattern curve display and annotation. See the following topics for instructions:

- [Set Flat Pattern Display options](#)
- [Use Flat Pattern Custom Callouts](#)



Set Flat Pattern Display options

[Overview](#)[How To](#)[Options](#)

NX Sheet Metal provides display options for a wide range of curve types, including bend center lines (up and down), bend tangent lines, inner and outer mold lines, and so on. You can establish defaults in the **Customer Defaults** dialog box, and you can change settings for each part individually in the **NX Sheet Metal Preferences** dialog box.

To change the customer defaults for the curves displayed by the **Flat Pattern** command:

1. Choose **File**→ **Utilities**→ **Customer Defaults**.
2. Select **Sheet Metal**→ **Flat Pattern**.
3. Click the **Curves** tab to manage which curves will be displayed and how. The same group of options is displayed for each curve type.
 - To change the color of a curve, click the color swatch in its group. The color palette is displayed. Click a color, then click **OK**.
 - Set line type and line width for the curve you want to display by choosing from the **Font** and **Width** lists.
 - Set the layer of the curve by typing a new value in the **Layer** box, or accept the default (1).
 - Repeat these steps for each type of curve you want to display on your **Flat Pattern** feature.
4. Select the **Enabled** check box for each type of curve you want displayed. Then click **OK**.

To set non-default display options for a Flat Pattern feature in a part file:

1. Choose **Preferences**→ **NX Sheet Metal**.
2. Click the **Flat Pattern Display** tab. The same options available for each Flat Pattern feature curve in the **Customer Defaults** dialog box are also available here. The options from left to right are: Color, font, width, and layer.
3. Change color, font, and width in the same way you did it in **Customer Defaults**. Set a layer.
4. Clear the check boxes for the curve types you do not want displayed.
5. Click **OK** to set the options.

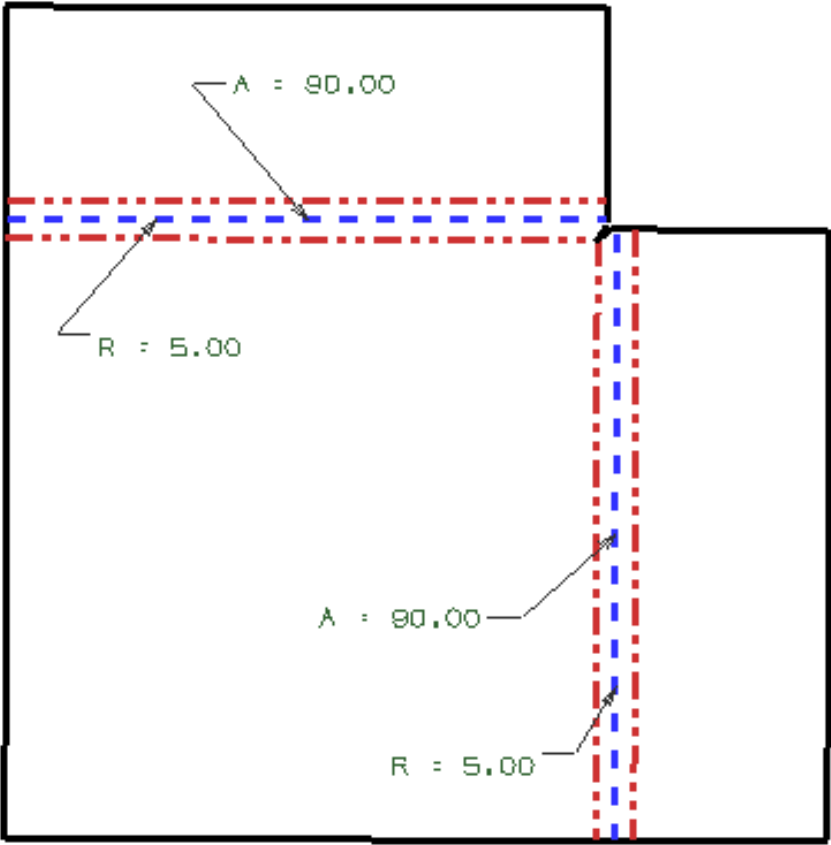


Use Flat Pattern Custom Callouts

[Overview](#)[How To](#)[Options](#)

NX Sheet Metal has six templates that you can use to create callouts for display on your Flat Pattern feature. You can customize them and control their availability in the **Customer Defaults** dialog box. Each callout has the following:

- **Name** specifies the name of the callout as it appears in the **NX Sheet Metal Preferences** dialog box. You can change this name.
- **Object Types** indicates the types of objects to which particular callouts can be associated, in order of preference. For example, **Bend Radius** would be associated with **Bend Center Line** first. If bend center lines are not shown on the Flat Pattern feature, **Bend Radius** would be associated with **Outer Mold Line**, which is the second item in the list, and so on. You cannot change the object types, but you can control their display on the Flat Pattern feature by:
 - Making them unavailable in the **Customer Defaults** dialog box.
 - Turning them off in the **NX Sheet Metal Preferences** dialog box.
 - Changing the order of the types or deleting ones that you do not want to display in the **Object Types** box.
- **Content** specifies the text that appears in the callout in your Flat Pattern feature. You can change portions of this, for example, the displayed name, and the number of decimal places. In the following Flat Pattern feature, we changed Bend Radius to R and Bend Angle to A.



- **Available** controls whether the callout is visible in the **NX Sheet Metal Preferences** dialog box.
- **Enabled** controls whether the callout is created by default on your Flat Pattern feature.

NX Sheet Metal Template Defaults for Custom Callouts		
Name	Object Types	Content
Bend Radius	Bend Center Line, Outer Mold Line, Inner Mold Line	Bend Radius = <!KEY=0,3.2@UGS.radius>
Bend Angle	Bend Center Line, Outer Mold Line, Inner Mold Line	Bend Angle= <!KEY=0,3.2@UGS.angle>
Bend Direction	Bend Center Line, Outer Mold Line, Inner Mold Line	Bend Direction= <!KEY=0,3.2@UGS.direction "up" "down">
Aerospace Sheet Metal only		
Hole Diameter	Hole Diameter	Hole Diameter = <!KEY=0,3.2@UGS.diameter>
Joggle Runout	Joggle Line	Joggle Runout = <!KEY=0,3.2@UGS.joggleRunout>

Joggle Depth

Joggle Line

Joggle Depth = <!KEY=0,3.2@UGS.joggleDepth>

For each part file, you can create six new callouts in the **Customer Defaults** dialog box. NX Sheet Metal remembers up to a maximum of five sets of six new custom callouts.

To customize the default callouts, choose **File**→**Utilities**→**Customer Defaults**. Select **Sheet Metal**→**Flat Pattern**. Click the **Annotations** tab. You see six **Custom Callout** templates. For each callout template:

1. (Optional) Change the name that is displayed in the **NX Sheet Metal Preferences** dialog box by typing a new one in the **Name** box.
2. Customize the callout that is displayed on the Flat Pattern feature by changing the text string in the **Content** box. For example, to change **Bend Angle** to BA, you would type the following: BA = <!KEY=0,3.2@UGS.angle>.
3. Repeat steps 1–2 for as many of the **Custom Callout** templates as you want to enable.
4. Select the **Available** check box for each callout that you want to display in the **NX Sheet Metal Preferences** dialog box. Select **Enabled** for all the callouts that you want to have appear by default on your Flat Pattern feature.
5. Click **OK** to save your changes. You have to start a new NX session for these defaults to take affect.

Customizing callout display in a sheet metal part file

After you create custom callouts, they are listed in the **New callouts from defaults** group on the **Flat Pattern Display** tab in the **NX Sheet Metal Preferences** dialog box. Select the check boxes next to the ones you want to see and clear the check boxes next to the callouts you want to hide. If you do not, you may have two sets of callouts for each item on your Flat Pattern feature, your customized label and the default.

Formatting custom callout text

To create your own custom callouts in NX Sheet Metal, you can use the special keyword language shown in the default callouts. A key looks like this:

<!KEY=0,3.2@UGS.angle>

These keys get resolved into values from the individual bend regions. The above key is an angle key and will be replaced by the angle value of the associated bend region. All keys begin with a <!KEY=0 tag that specifies the beginning of a key to be replaced. The next section consists of an integer, a period, and another integer, which defines the format of the number that replaces the key. If this section is 2.3, then the number that replaces the key will be in the format XX.YYY. As an example, for angle values, the possible range is 0 to 180, so you would logically specify 3 for the first digit, and for the second, the number of decimal digits you want displayed.

Following the number format is an @ sign and then the actual name of the value to replace the key. For NX Sheet Metal, possible values are UGS.angle, UGS.radius, and UGS.direction. These would be replaced, respectively, with the angle, radius, and direction of the associated bend region.

The default direction key is slightly different than the normal angle and radius keys. The direction key is a text-only key, so the number format is ignored. Also, the direction key is a conditional key, in that it shows one of two values. The first value is the “true” value, and the second value is the “false” value. The default direction key looks like this:

Bend Direction = <!KEY=0,3.2@UGS.direction "up" "down">

This is how it works. The software takes the face normal from the face you selected as the **Upward Face** and assigns that to the “up” direction. Then, it compares this direction to the direction of the bend region. If they are both “up,” then the condition is “true” and “up” is displayed. If the bend direction is not equal to “up,” then the condition is “false” and “down” is displayed.

Carriage returns are also allowed in the definition of an annotation (the text string that appears in the **Content** box of the **Annotations** tab in the **Customer Defaults** dialog box). Here are the annotation definitions with carriage returns for a bend region with an angle of 30, radius of 5, and a down direction:

Annotation definitions:

A = <!KEY=0,3.2@UGS.angle>

R = <!KEY=0,3.2@UGS.radius>

<!KEY=0,3.2@UGS.direction "up" "down">

Resulting annotation:

A=30.00

R=5.00

down

You can also combine annotations. For example, you could create a callout that combines bend angle and bend direction with the label Bend. The annotation definition would be:

Bend = <!KEY=0,3.2@UGS.angle><!KEY=0,3.2@UGS.direction "up" "down">

The resulting annotation (with a bend angle of 90 and an upward bend) would be:

Bend = 90.00 up