



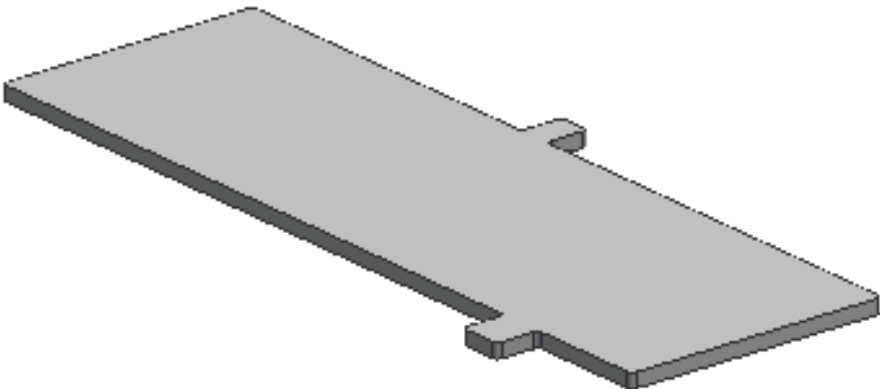
# Tab overview

Overview

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**Tab** constructs a flat feature on a sheet metal part. You can use this command to construct a base feature or to add material to existing faces of a sheet metal part.



## Where do I find it?

|             |                                                                                                        |
|-------------|--------------------------------------------------------------------------------------------------------|
| Application | NX Sheet Metal                                                                                         |
| Toolbar     | NX Sheet Metal→Tab  |
| Menu        | Insert→Tab                                                                                             |

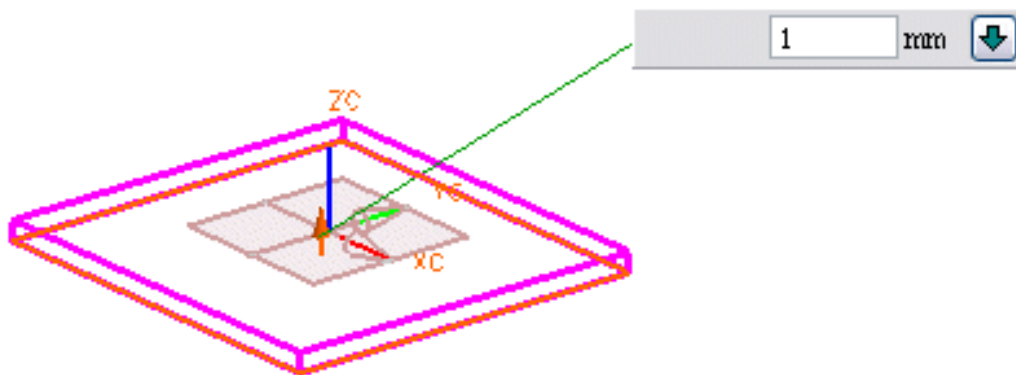


## Create a tab

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To create a **Base Tab**:

1. Click **Tab** . If there are no other bodies in the part file, **Base Tab** is selected by default.
2. Define the profile plane.
3. Draw a closed profile , or select a section  from an existing sketch.
4. Pick the side of the profile you want the material to be on: accept the default (above the profile) or double-click on the drag handle to reverse the direction.
5. Accept the default thickness for the tab, or specify another. (In the latter case, deselect **Use Global Value**) Here, we specify thickness of 1 mm.



6. Click **OK** to finish the feature.

To create a secondary tab:

1. Click **Tab** . If you have already created a base feature, **Secondary Tab** is selected by default.
2. Define the profile plane by selecting a part face.
3. Draw an open profile in any 2D shape. If the ends of the open profile sketch are not

coincident with the edges of the part, they will be extended to those edges. An arc with open ends is extended to form a circle.



4. Click **OK** to finish the feature.

The tab is assigned the material thickness of the part.



## Tab options

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The following options are available for **Tab**:

- **Base Tab** and **Secondary Tab** let you specify which kind of tab to create. If there is no body in the part file, the system defaults to construction of a base feature. If there is already a body in the part file, the system defaults to creating a secondary feature.



- **Curve** and **Sketch Section**  let you specify whether to construct the feature from an existing sketch or to create a new sketch from within the feature. For more information on these sketch options, see [Using Sketches in NX Sheet Metal](#).