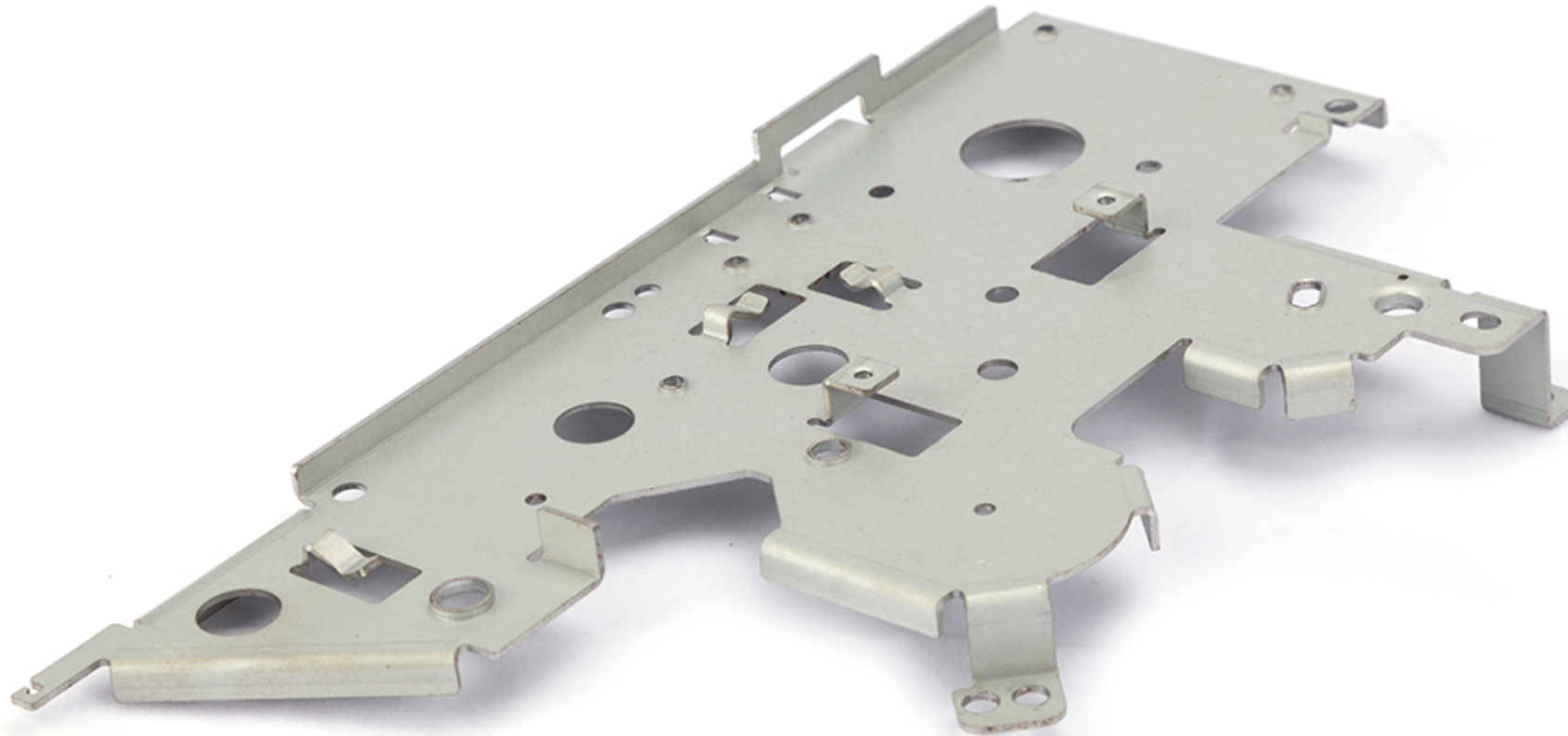


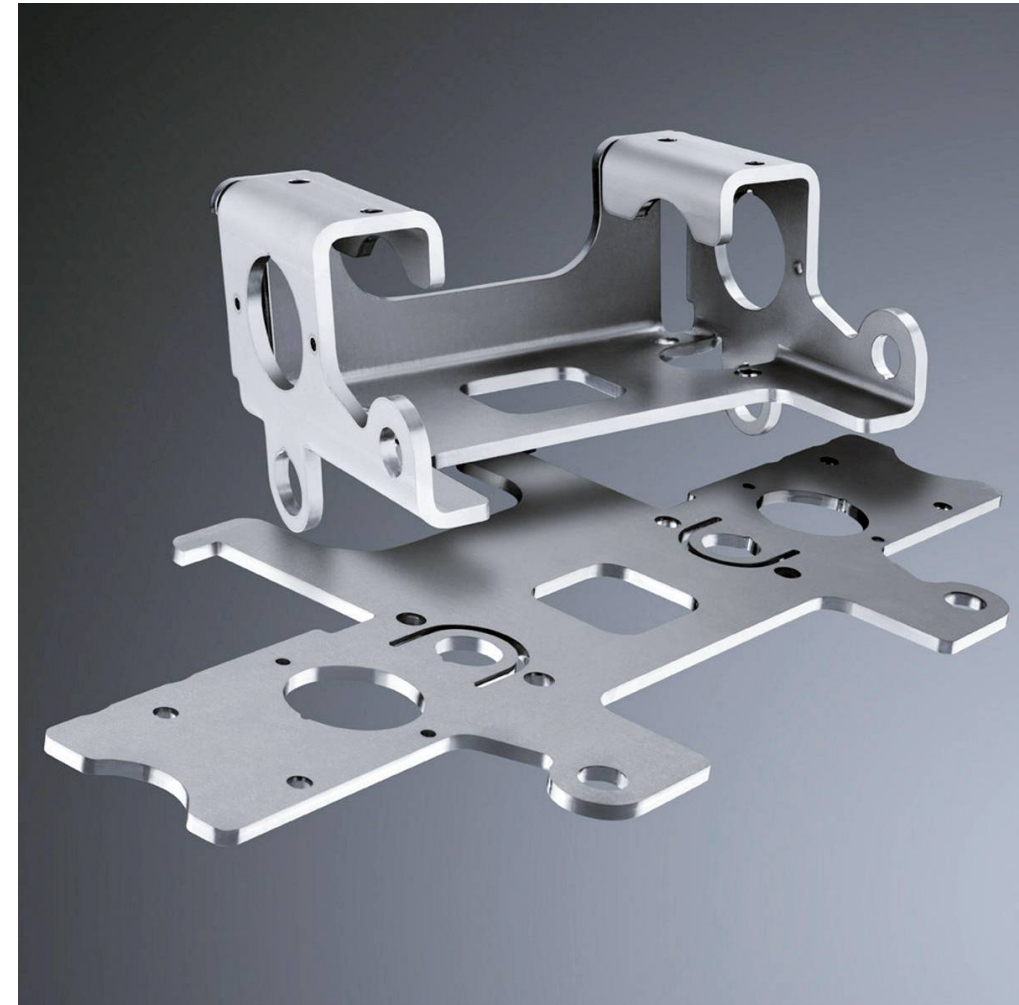
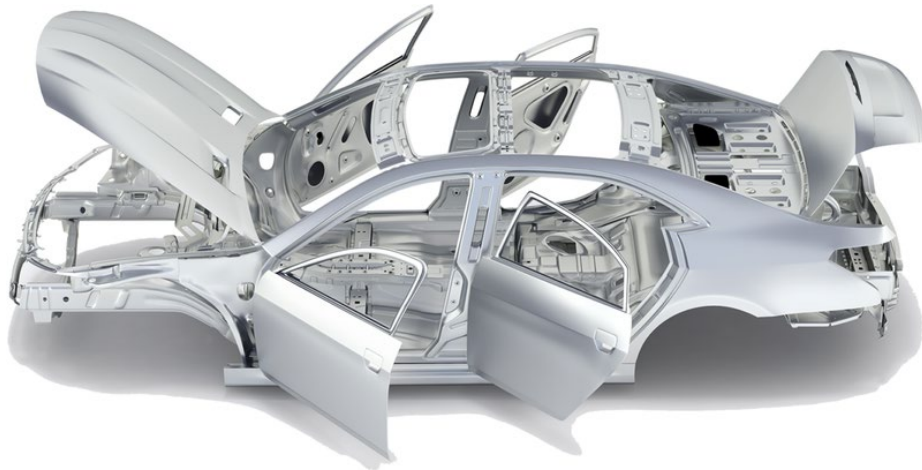
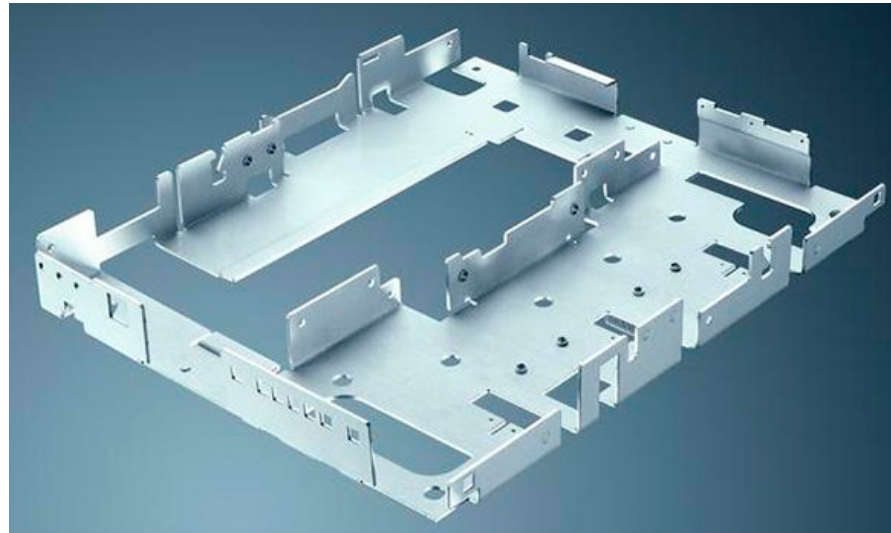


# Sheet Metal Design

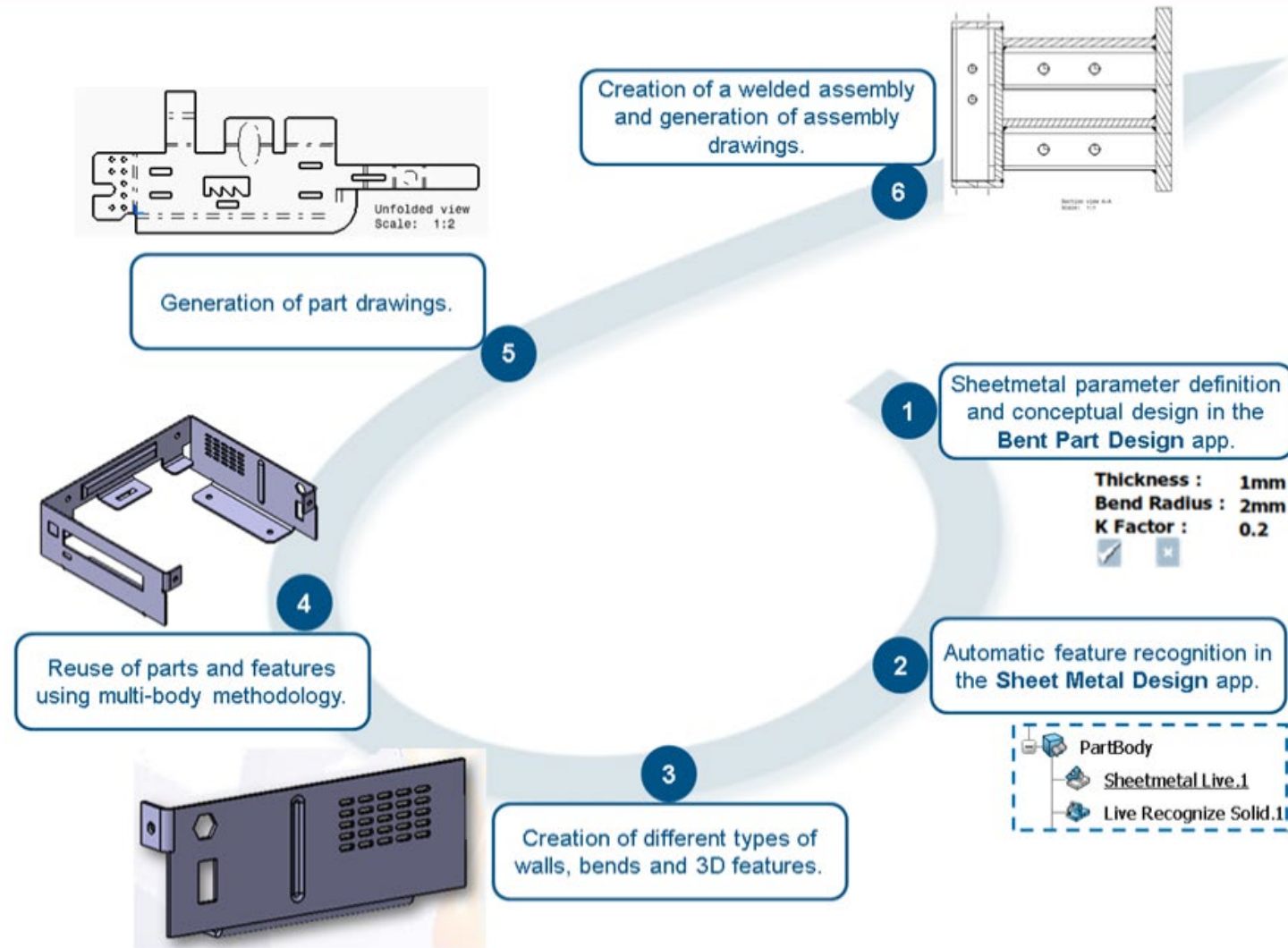




# Sheet Metal Design



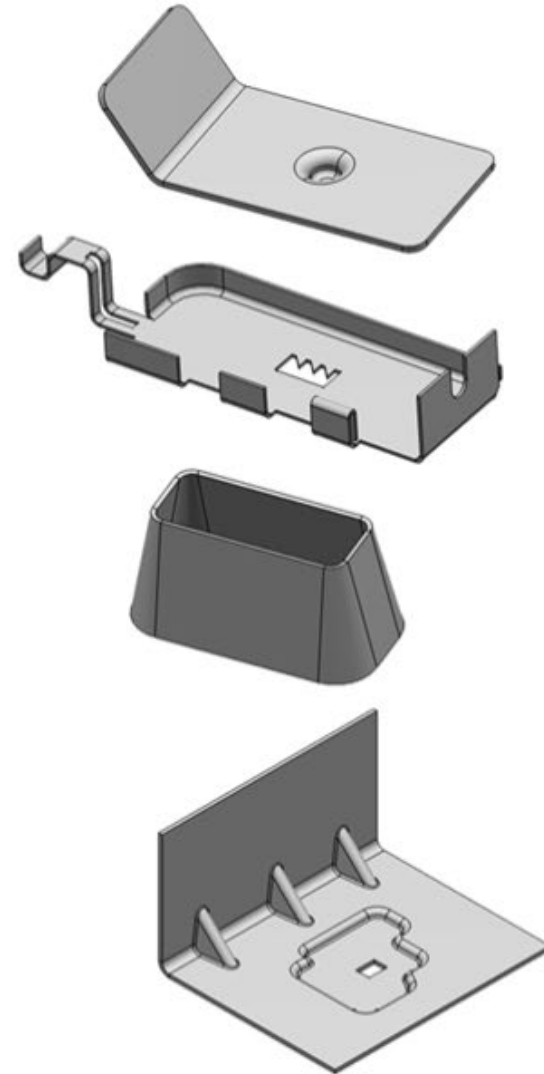
# General Process Sheet Metal Design



# Sheet Metal Design Process

The **Sheet Metal Design** app supports the following processes:

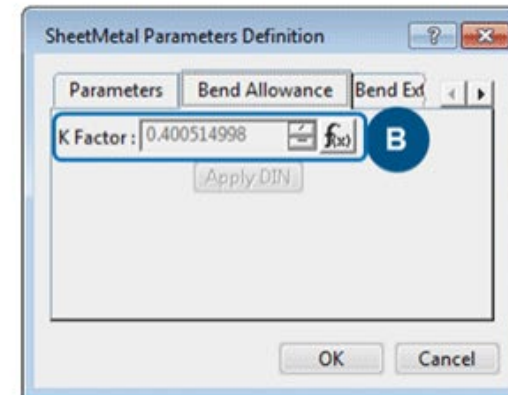
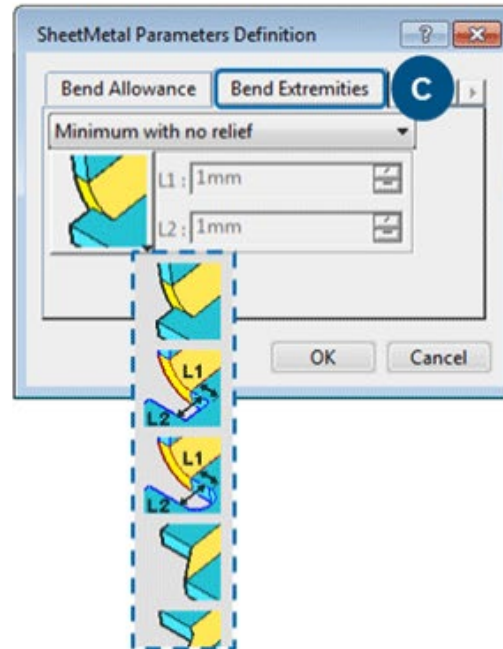
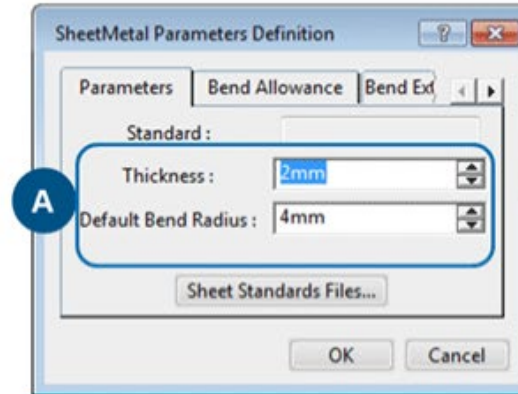
- ▶ Blanking
- ▶ Drawing
- ▶ Deep Drawing
- ▶ Bending / Folding
- ▶ Punching
- ▶ Forming / Stamping
- ▶ Shearing
- ▶ Slitting



# Sheet Metal - Parameters

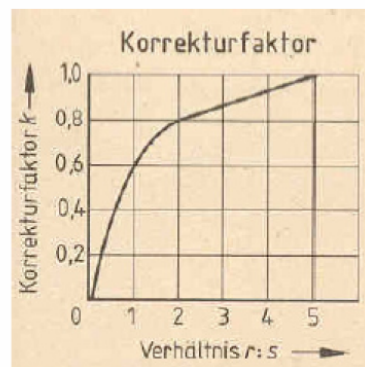
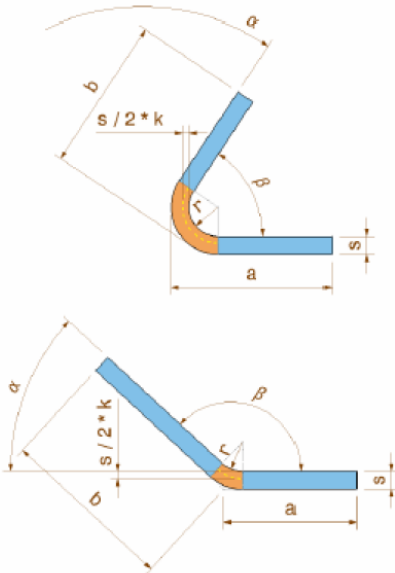
You can specify parameters using the tabs in the **SheetMetal Parameters Definition** dialog box.

- A. **Thickness** and **Default Bend Radius** in the **Parameters** tab are mandatory.
- B. The option in the **Bend Allowance** tab is optional.
- C. The options in the **Bend Extremities** are also optional.



# Sheet Metal - Parameters

## EINFÜHRUNG – K-Faktor und Ausgleichswerte nach DIN 6935



L = gestreckte Länge

a, b = Länge der Schenkel

v = Ausgleichswert

s = Blechdicke

r = Biegeradius

$\beta$  = Öffnungswinkel

k = K-Faktor

**Gestreckte Länge:**  $L = a + b - v$

**Ausgleichswert für  $\beta = 0^\circ$  bis  $90^\circ$**

$$v = 2(r+s) - \pi * ((180^\circ - \beta) / 180^\circ) * (r + s / 2 * k)$$

**Ausgleichswert für  $\beta > 90^\circ$  bis  $165^\circ$**

$$v = 2(r+s) * \tan((180^\circ - \beta) / 2) - \pi * ((180^\circ - \beta) / 180^\circ) * (r + s / 2 * k)$$

Für  $\beta > 165^\circ$  bis  $180^\circ$ ;  $v=0$  (vernachlässigbar klein)

**Beispiel K-Faktor:**

$$r = 4\text{mm}, s = 2\text{mm} \quad k = 0,8$$

Der K-Faktor ist nur für ein Verhältnis zwischen Biegeradius und Blechstärke von  $>0$  bis  $<5$  definiert.

# Sheet Metal - Parameters

## EINFÜHRUNG – Biegetabellen

Da der Korrekturwert neben dem Material auch von der Biegemaschine abhängig ist, werden in der Praxis häufig empirisch ermittelte Korrekturwerte verwendet.

In Abhängigkeit von Material, Materialstärke, und Biegeradius wird für jeden Biegewinkel ein Korrekturfaktor ermittelt. Diese werden in Biegetabellen eingetragen.

In Biegetabellen können zusätzlich Informationen über Stempel- und Stanzwerkzeuge abgelegt werden. Auf diese Weise lassen sich firmenspezifische Werkzeuge verwalten und Fehlerquellen minimieren.

SheetMetalStandard	Thickness (mm)	DefaultBendRadius (mm)	BendTable
Material: DC01 2mm	2	1	bend_allowance\BA_T2.txt
Material: DC01 1mm	1	0.7	bend_allowance\BA_T1.txt

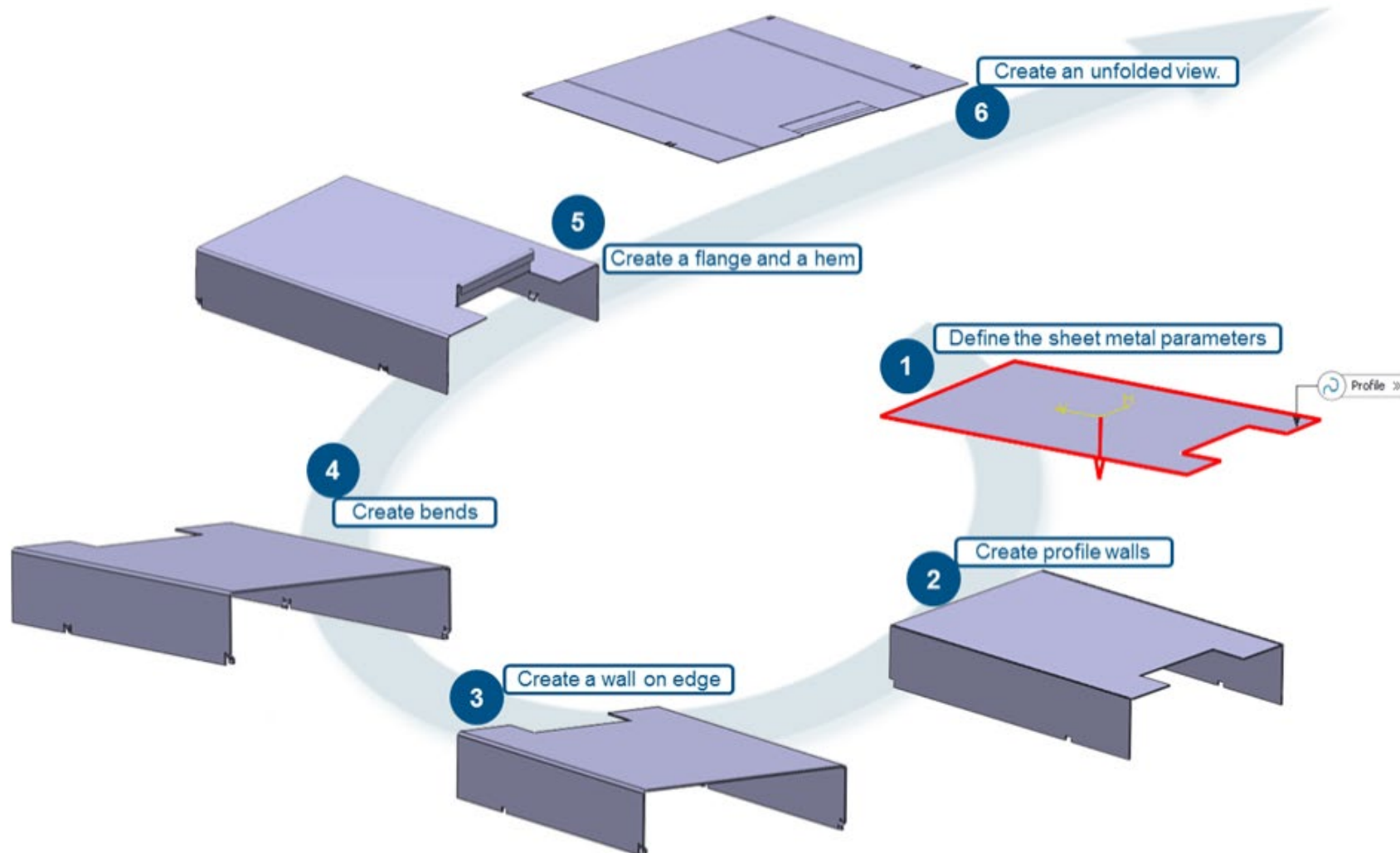
OpenAngle (deg)	Deduction (mm)
0	0
5	0
5.001	-0.1
15	-0.1
15.001	-0.21
25	-0.21
25.001	-0.34

Deduction ist der Korrekturwert v.

Die Zwischenwerte werden linear interpoliert.

Die Tabellen werden vom Administrator verwaltet und den Anwendern zur Verfügung gestellt.

# Stages in Process



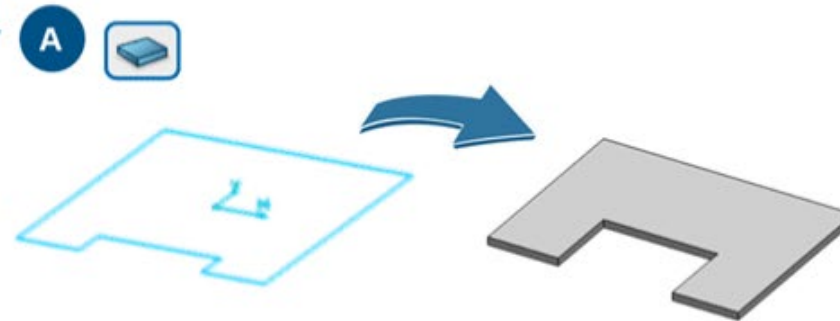


# About Sheet Metal Walls

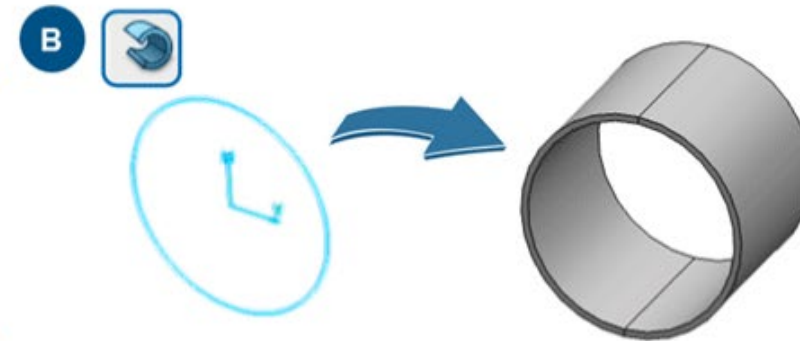
A sheet metal wall is a side or face of a sheet metal part. It is generally but not always a non-deformable feature.

Following are the different types of walls:

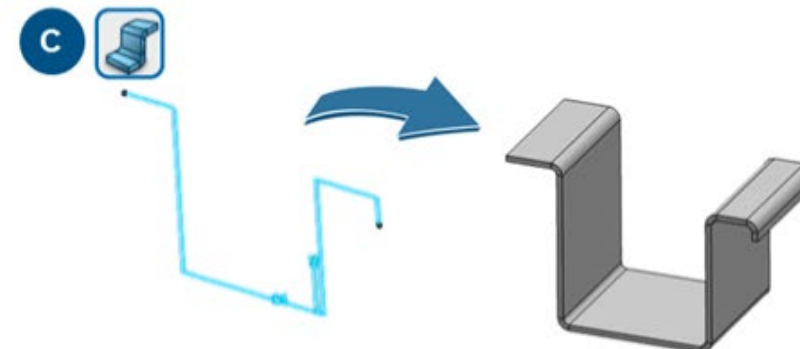
A. A single wall is created by extruding a closed profile.



B. A simple circular profile is extruded to create a rolled wall.



C. An open profile is extruded to create several walls at once.



# Main Tools

## Model Section (1/2)

1. **Extrusion:** Creates walls and bends using an open sketch profile perpendicular to the sketch plane.
2. **Hopper:** Creates a sheet metal feature between two sketches which are continuous in tangency and on parallel planes.
3. **Rolled Wall:** Creates a wall using simple open or closed circular sketch profile perpendicular to the sketch plane.
4. **Bend:** Creates a constant radius between adjacent wall features.
5. **Conical Bend:** Creates a variable radius between adjacent wall features.
6. **Bend from Flat:** Creates a constant radius bend by folding an existing wall using a sketched fold line.
7. **Unfolding:** Applies folding/unfolding to cylindrical faces.



# Main Tools

## Model Section (2/2)

---

8. **Fold/Unfold:** Allows you to see the sheet metal part in its flattened state before folding.
9. **Views Management:** Displays the *Flat View* and the *3D View* simultaneously.
10. **Flange:** Creates a uniform length wall on an existing edge.
11. **Hem:** Creates a specific form of a simple flange where the swept wall is parallel to the wall to which it is attached.
12. **Tear Drop:** Creates a specific form of a simple flange where the swept wall is folded so as to touch the wall to which it is attached.
13. **User Flange:** Creates a flange in which you can define your own profile.



# Main Tools

---

## Refine Section

---

1. **Corner:** Rounds off sharp edges.



2. **Chamfer:** Trims the corners of sharp edges.



# Main Tools

## Refine Section (1/2)

---

1. **Cut Out:** Creates openings in a sheet metal part.
2. **Hole:** Creates a hole in a sheet metal part.
3. **Circular Cutout:** Creates a circular openings in a sheet metal part.
4. **Corner Relief:** Creates cut in a sheet metal part at the intersection of walls.
5. **Surface Stamp:** Creates a stamp.
6. **Bead:** Creates a bead.
7. **Curve Stamp:** Creates a curve stamp.
8. **Flanged Cut Out:** Creates a flanged cutout.



# Main Tools

## Refine Section (2/2)

---

9. **Louver:** Creates a louver.



10. **Bridge:** Creates a bridge.



11. **Flanged Hole:** Creates a flanged hole.



12. **Circular Stamp:** Creates a circular stamp.



13. **Stiffening Rib:** Creates a stiffening rib.

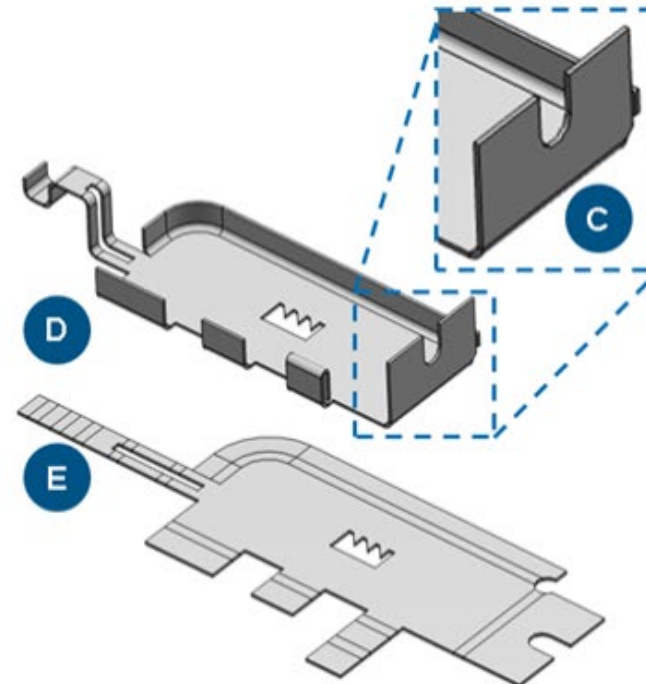
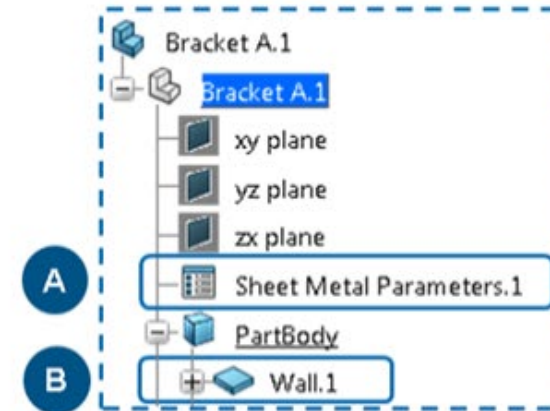


14. **Dowel:** Creates a dowel.



# Terminology

- A. Parameters: These are the default values for the thickness, the bend radii and the type of bend relief used in the part.
- B. Wall: This is generally a non-deformable feature and is created from a sketch. It is the first sheet metal geometric feature – the reference feature – in the sheet metal part.
- C. Bend: This is a deformable feature and is usually created between the adjacent wall features.
- D. Folded View: This is the view of the part as it would appear in its final manufactured state.
- E. Unfolded view: This is the view of the part in its flattened state before folding.

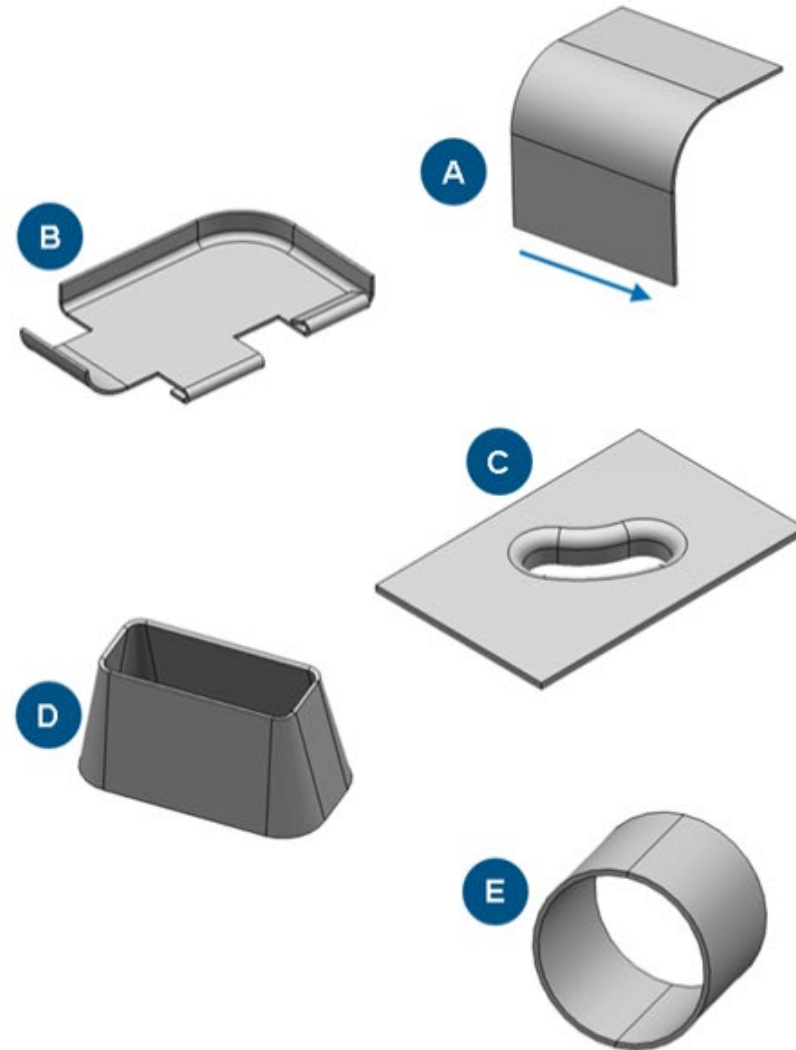


Sheetmetal Parameters that are defined in the **Sheet Metal Design** app will remain the same in the **Bent Part Design** app.

# Terminology

Other terms that are used are as follows:

- A. Extrusion: This is a single feature which can be created from a sketch and which can include several walls and bends.
- B. Flange: This can be a pre-defined or user-defined profile which is swept along an edge and which can include several walls and bends.
- C. Stamp: This is a feature that cuts or deforms existing deformable or non-deformable features.
- D. Hopper: This is a single feature which can include several walls and bends.
- E. Rolled wall: This is a specific case where a wall can be created from a circular profile.



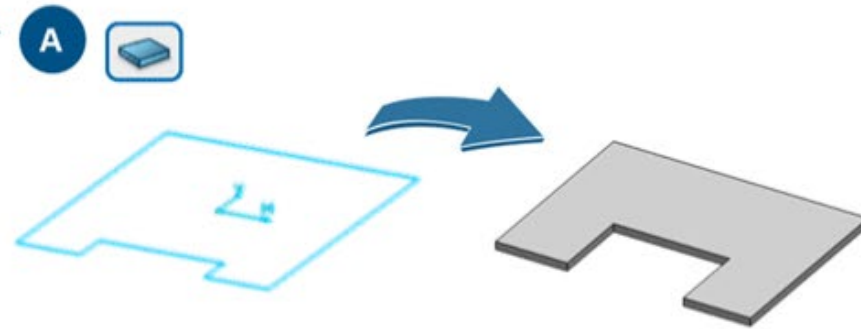


# About Sheet Metal Walls

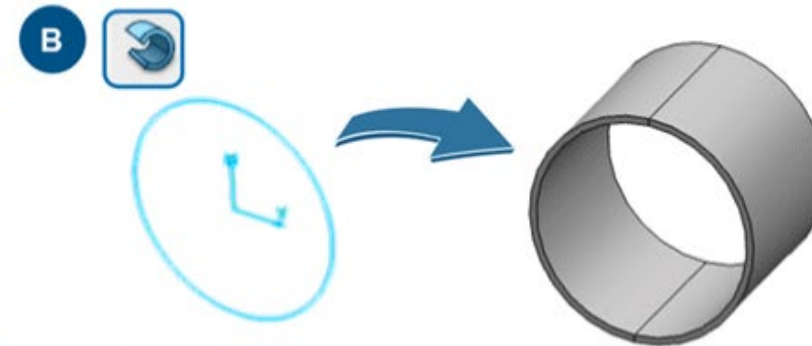
A sheet metal wall is a side or face of a sheet metal part. It is generally but not always a non-deformable feature.

Following are the different types of walls:

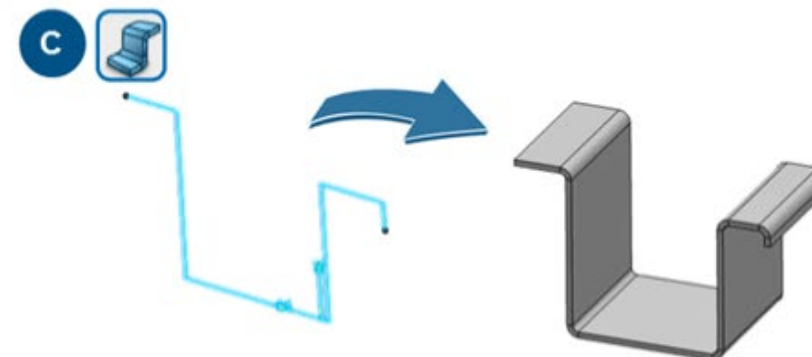
A. A single wall is created by extruding a closed profile.



B. A simple circular profile is extruded to create a rolled wall.

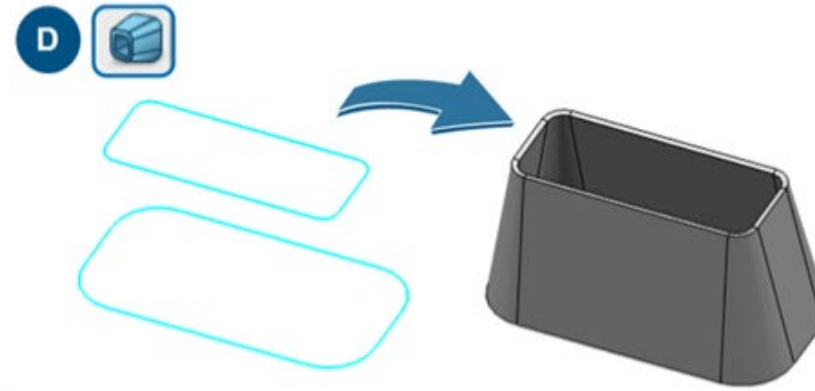


C. An open profile is extruded to create several walls at once.

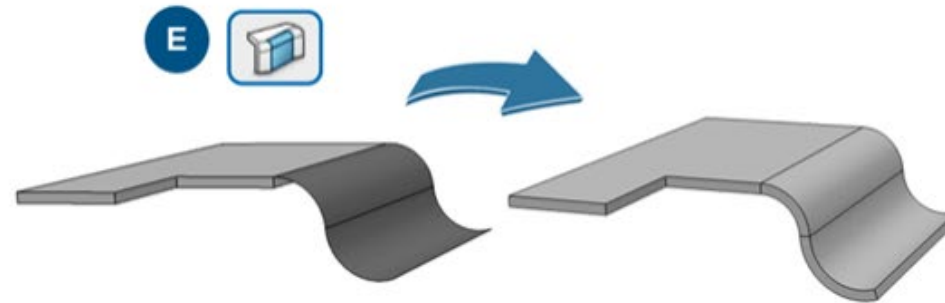


# About Sheet Metal Walls

D. Two closed profiles are used to create a hopper.



E. The surface is connected to an already existing sheet metal features with free form surface.



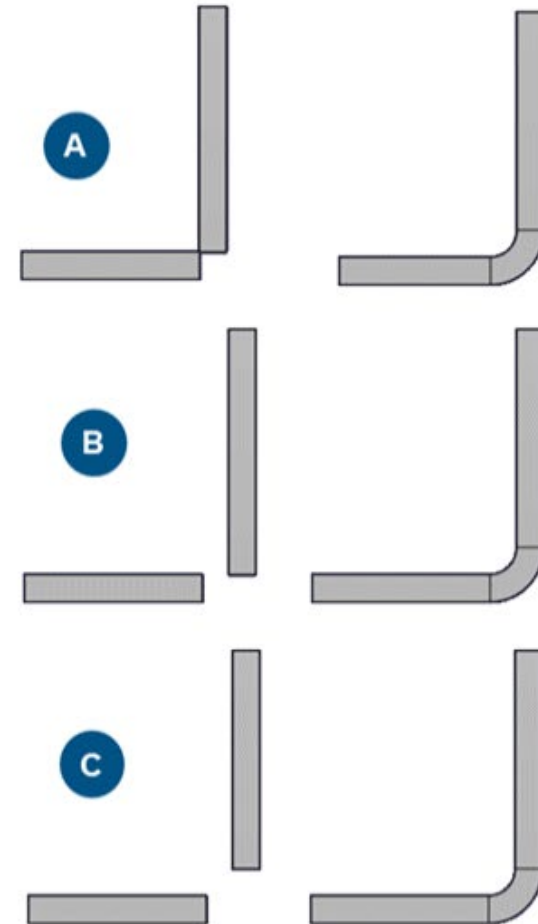
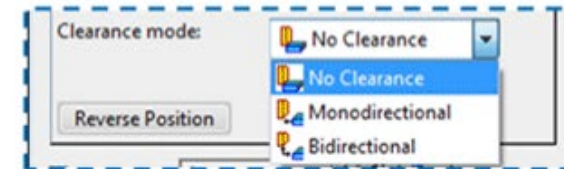
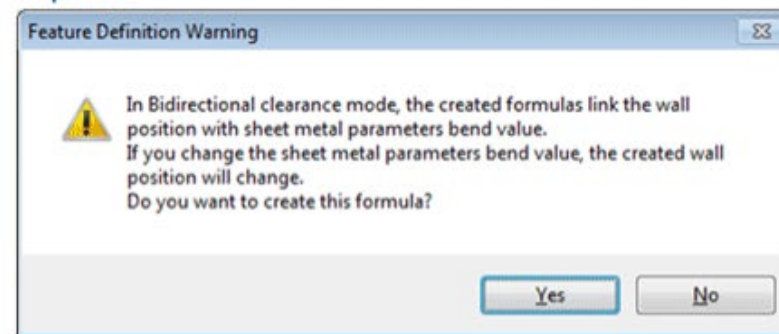
# Wall on Edge Terminology

The *Clearance* defines how the wall is positioned with respect to the reference wall and the selected edge.

Select the **Height & Inclination** tab to access the **Clearance mode** options:

- A. **No Clearance:** Creates a wall on edge without clearance.
- B. **Monodirectional:** Creates a wall on edge by moving the wall in one direction.
- C. **Bidirectional:** Creates a wall on edge by moving the wall in two directions.

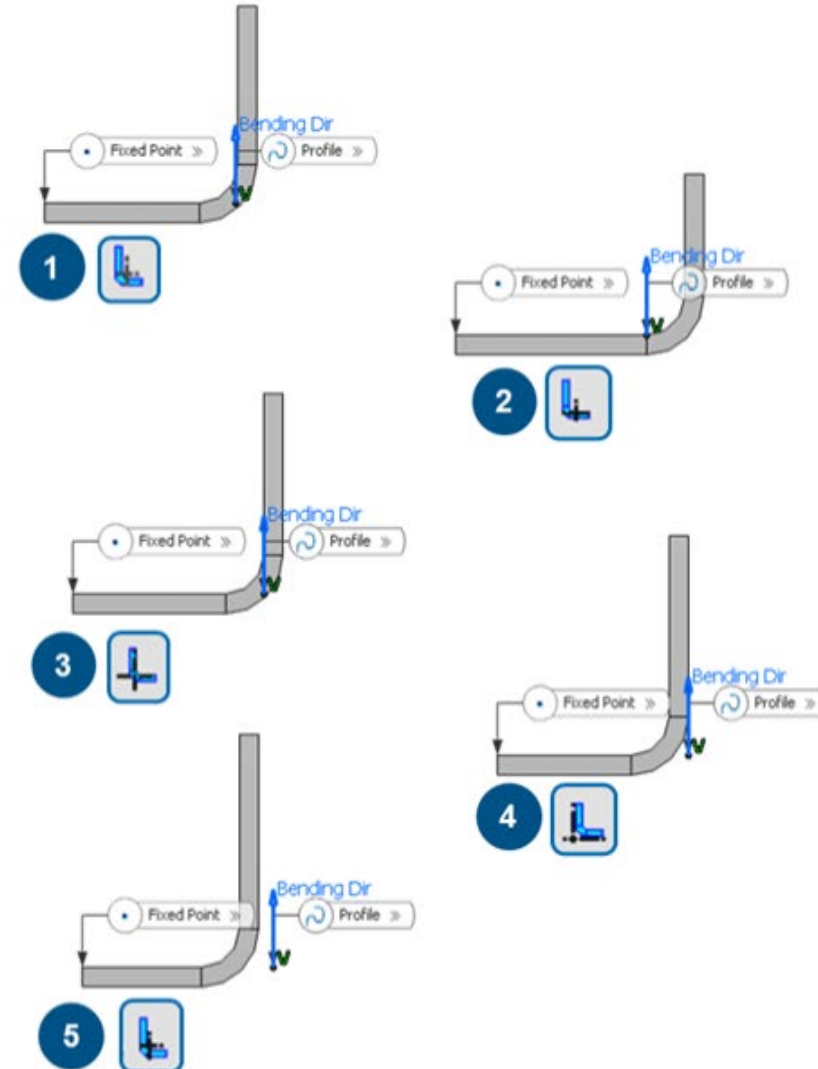
A warning message will be displayed when you select **Bidirectional** clearance to warn about the possible impacts.



# Wall on Edge Terminology

The line sketch can be used as the:

1. The **Axis** of the bend.
2. The **BTL Base Feature** (Bent Tangent Line Feature): The line is present on the wall and corresponds to the limits of the bend's fillet.
3. The **IML** (Inner Mold Line): The line becomes the intersection the internal surfaces of the bend (before filleting) and the wall.
4. The **OML** (Outer Mold Line): The line becomes the intersection of the bend support and a plane perpendicular to the wall and normal to the OML.
5. The **BTL Support** (Bent Tangent Line Support): The line is present on the bend support and corresponds to the limits of the bend's fillet.



# Übungen

- 7\_UL\_01 – 7\_UL\_14