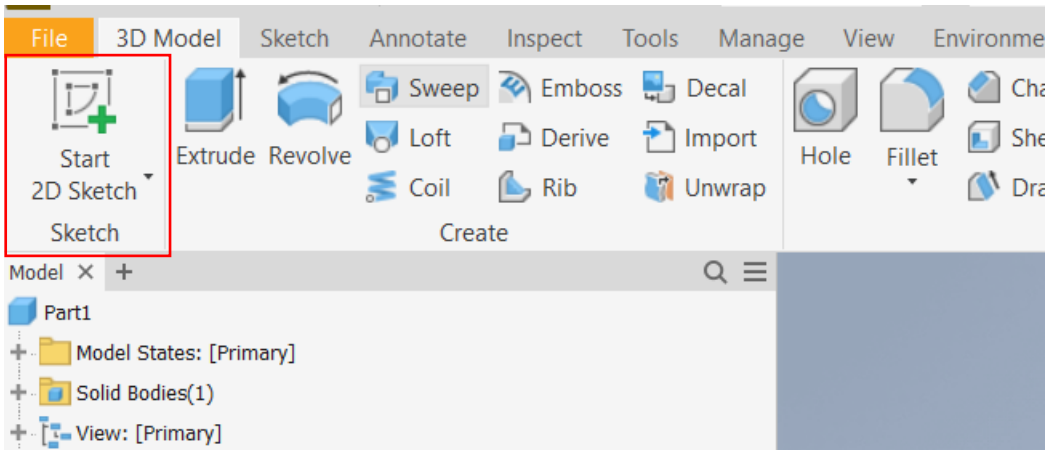
Sketch – Features

Introduction

After you have started a project and created a **Part** file. Then make a **Sketch**, as **Sketches** in Inventor are the foundation for the 3D model. You start a **Sketch** by locating "**Start 2D Sketch**" at the top of the screen to the far left. Then the **Sketch** menu will be activated and here you can draw in 2D.



Index:

Shortcut Keys

Selections:

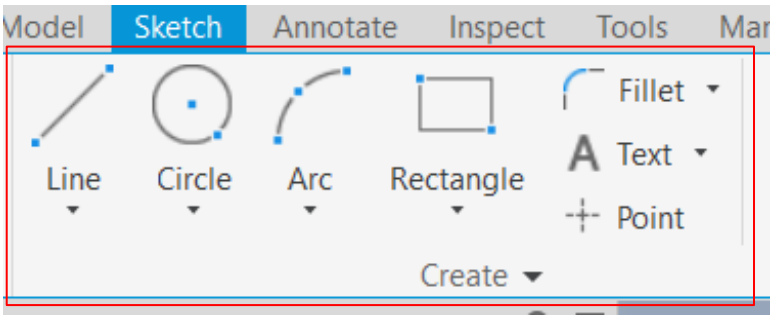
Control (CTRL) = Adds more than one selection. ***Press and hold.*** *(It is done correctly when there is a plus icon next to the mouse).*

SHIFT (↑) = Removes selections. ***Press and hold.*** *(It is done correctly when there is a minus icon next to the mouse).*

Create:

Introduction:

In order to use the **Create** features, locate the tap at the top of the screen:



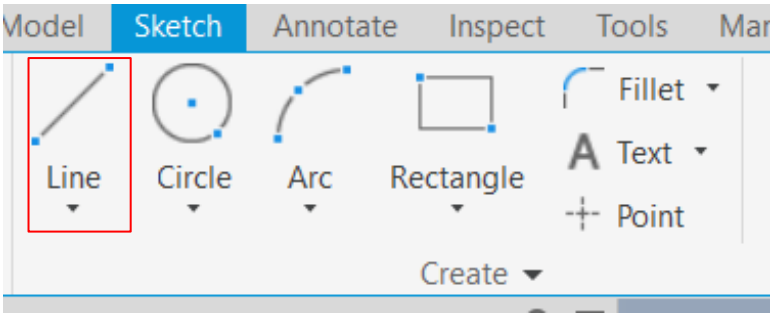
The **Create** features, make shapes and lines.

Lines

Introduction:

Lines is one of the most used **Create** features. You can use the **Lines** features to create straight and curved lines.

In order to use the **Lines**, locate **Create** tab at the top of the screen:



To locate the other **Lines** features; click on the arrow below "**Line**", then a drop-down menu will roll down, where you can see all the **Line** features.

Line:

- Makes straight lines.
- **Short key** = L

Spline:

Makes curves in 2 different ways:

Control Vertex:

Start by selecting the first point. Then choose where the control points, that control the curve should be. Two exit one's **Control Vertex**, make a closed **Loop**, or click on:

Interpolation:

Start by selecting the first point. Then choose how many points you want to use. Now finish the line, after you can change how the curves behave. The gray lines can be moved and the gray points change the curve. To get the gray lines, remember to select your curve first.

Equation Curve:

When you click, this box appears:

Here you can create a line based on an equation.

Bridge Curve:

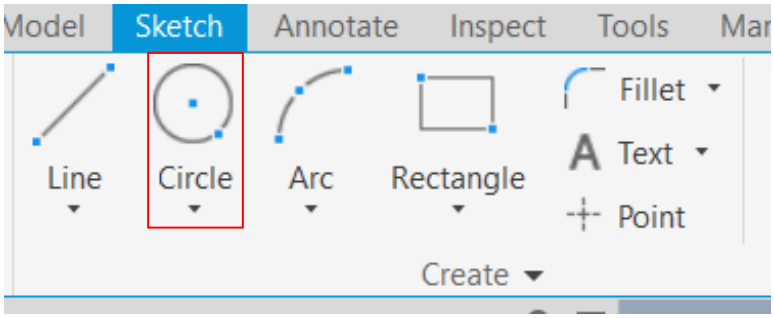
Makes a bridge between 2 lines, with a **Smooth** curve **(G2)**.

Circles

Introduction:

The **Circles** features are used to create circles and ovals etc.

In order to use the **Circles**, locate **Create** tab at the top of the screen:



To locate the other **Circles** features; click on the arrow below "**Circle**", then a drop-down menu will roll down, where you can see all the **Circle** features.

Circle – Center Point:

Creates a circle from a center point, start by selecting a center point and then specify the diameter. Two specify in radius, right click with the mouse and select **Radius**, this also applies from radius to diameter.

- **Short key** = CTRL + SHIFT (↑) + C

Circle – Tangent:

Makes a tangent circle between 3 lines.

Ellipse:

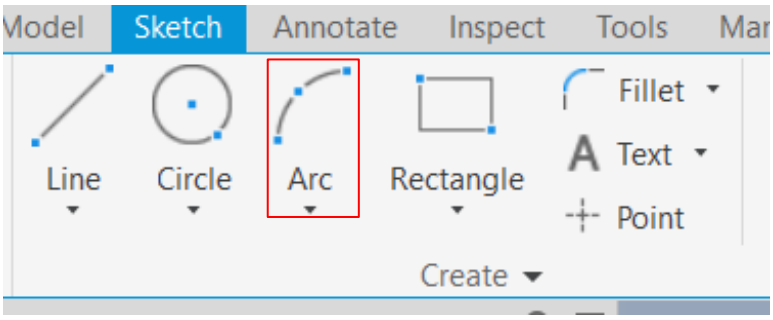
Makes ovals and ellipses. Start by choosing a center point, then choose the length (it's symmetrical) and finally choose the height.

Arcs

Introduction:

The **Arcs** features are used to create bridges and arches.

In order to use the **Arcs** features, locate **Create** tab at the top of the screen:



To locate the other **Arcs** features; click on the arrow below "**Arc**", then a drop-down menu will roll down, where you can see all the **Arc** features.

Tree Point:

Creates an arc from 3 points. Two of the points are start (first) / end (second) and the point in the middle (third) is the radius.

Tangent:

Creates an arc that is tangent to, for example, a line. Start at an existing point.

Center Point:

Creates an arc defined by a center point and a start/end point. Start by choosing the center point, then you can choose the length and angle of the first point, then you choose the end point. You can choose to specify an angle.

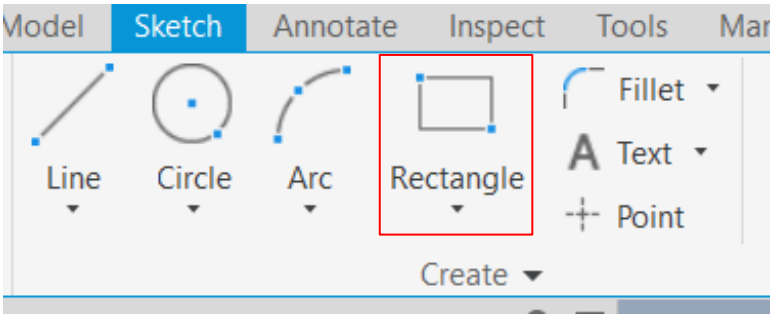
- **Short key =** A

Rectangles

Introduction:

The **Rectangles** features are used to create squares, slots, and polygons.

In order to use the **Rectangles** features, locate **Create** tab at the top of the screen:



To locate the other **Rectangles** features; click on the arrow below "**Rectangle**", then a drop-down menu will roll down, where you can see all the **Rectangle** features.

Rectangle:

Makes a rectangle.

Two Point

Creates a rectangle with 2 diagonal points. Select the first point, then you can specify the length. Select the diagonal point and here you can also specify length.

Three Point

Creates a rectangle with 3 points. Select the first point and choose the placement, here you can specify length. Then select the direction of the rectangle with the mouse. Then select the third point of the rectangle, here you can also specify length.

Two Point Center

Creates a rectangle with a center point. Select a center point and then define length and width.

Three Point Center

Creates a rectangle with 3 points. Choose a center point, and then enter the length/height. Then select the third point of the rectangle, here you can specify the height/length.

Slot:

A **Slot** is 2 parallel lines, they are rounded (2 arches) on each side:

Center to Center

Creates a **Slot** that defines the placement and length of the center points, of the **Slot's** arcs. Start by selecting the center points of the 2 arcs, and then the width.

Overall

Creates a **Slot** that defines the placement, length and width. Start by selecting the start and end point, then select the width or enter the diameter/radius of the arcs.

Center Point

Creates a **Slot** that defines the placement of the center point, the placement of one arc and its width. Start by selecting the center point, then select the width or specify the diameter/radius of the arcs.

Tree Point Arc

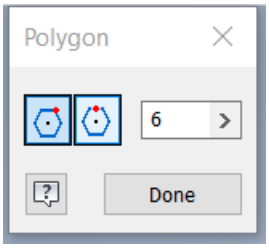
Makes 3 point **Slot** arc. Start with the start point, then the end point. Then the third point, which is the radius. Then choose the thickness of the **Slot**.

Center Point Arc

Creates a **Slot** arc defined by a center point and start/end point. Start by selecting the center point. Then you can choose the length and angle of the first point. Then select the end point, here you can select and specify an angle. Then choose the thickness of the **Slot**.

Polygon:

A polygon makes shapes with a certain number of sides.



In the box you indicate how many sides you need. Max 120 sides.

The 2 symbols are **Circumscribed** or **Inscribed**; the difference is where your mouse is stuck. The 2 symbols show visually where the mouse gets stuck.

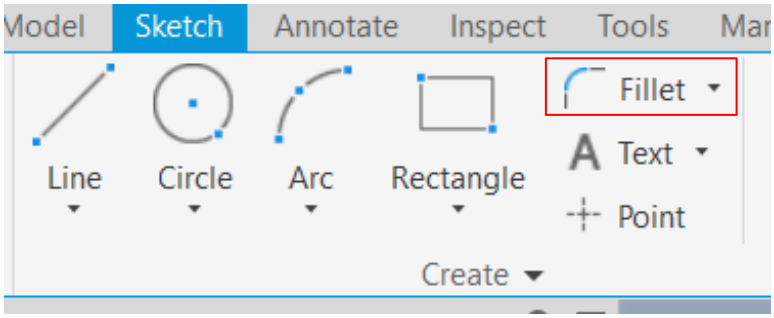
After "**Done**", you cannot specify the length of the sides, it must be done after the feature is not in use.

Fillet/Chamfer

Introduction:

The **Fillet and Chamfer** features are used to modify corners.

In order to use the **Fillet/Chamfer** features, locate **Create** tab at the top of the screen:



To locate the **Chamfer** feature; click on the arrow below "**Fillet**", then a drop-down menu will roll down, where you can see the **Chamfer** feature.

It's **not recommended** to use **Fillet** or **Chamfer** in **Sketch** but instead in 3D **Modify**. Since Inventor has better conditions in 3D.

Fillet

- Rounds corners with a specified radius.

Chamfer

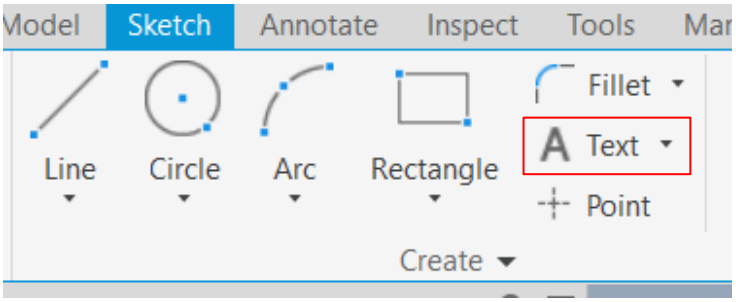
- Cuts corners with a specific angle.

Text

Introduction:

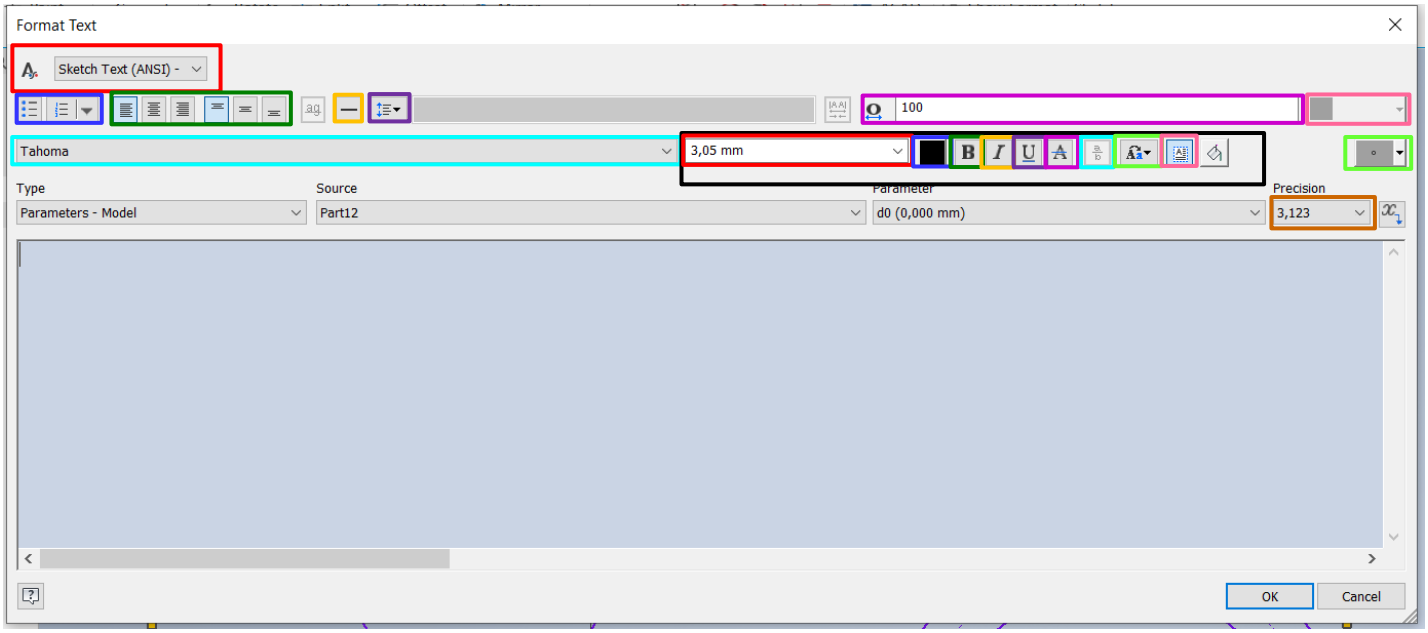
The **Text** features are used to create text.

In order to use the **Text** features, locate **Create** tab at the top of the screen:



Text

This is what the **Properties** looks like:



Red = Profiles. A text format can be saved here.
Blue = Lists. Arranges text in a list format.
Green = Placement of the text.
Yellow = Single Line Text.
Purple = Text Spacing.
Magenta = Letter spacing. The width between the letters.
Cyan = Font.

Black:

- Red = Size.
- Blue = Colour.
- Green = Bold.
- Yellow = Italic.
- Purple = Underline.
- Magenta = Strikethrough.
- Cyan = Stack.
- Lime = lowercase / UPPERCASE / Title Case.
- Pink = Turned on, makes the direction horizontal up (so it's readable). If turned off, then the second Pink box unlocks.

Lime = Insert symbols.
Pink = To open → turn off the first Pink box. Then it's the direction of the text.
Brown = Number of decimal there are.

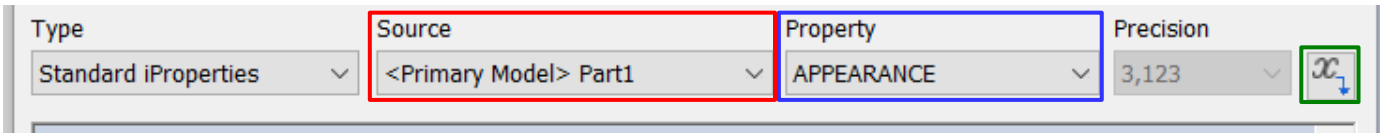
If a change doesn't work, remember to highlight the text.

Standard iProperties

Standard iProperties is Inventor's way of using **Part** values. So, for example you can have a **Material** value engraved in the **Part**. If you change **Material** in the **Part**, the text is also changed. (They are related).

Property example = **Designer**/**Part Number**/**Material**.

In order to use, select "**Standard iProperties**" under **Type**:

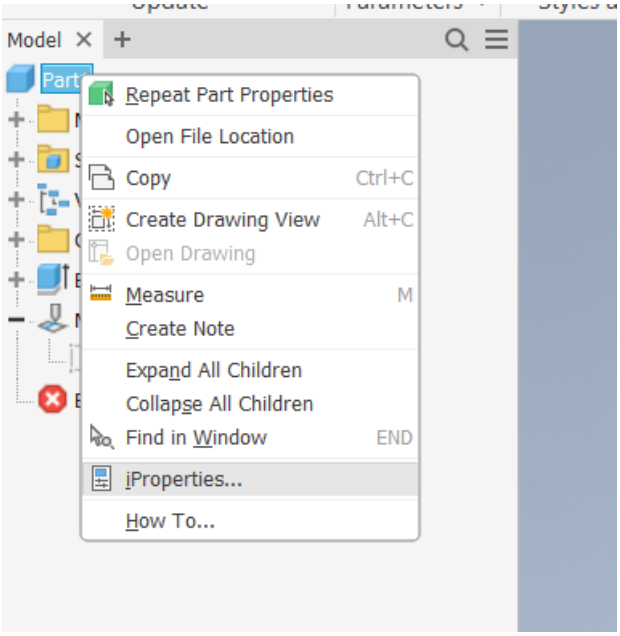


Source = Which **Part** Inventor takes **Property** from.
Property = Which **Property** to insert.
Green = To add, select the ↑ 2 and then press here.

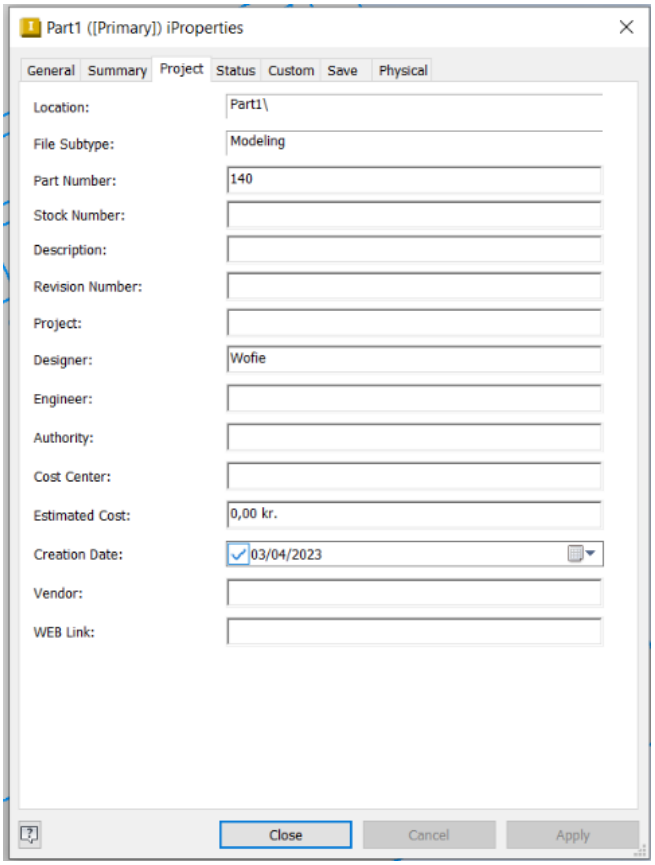
If a particular Property doesn't work:

You can always change/edit for example **Part Number** under **iProperties**. To change open **iProperties**.

Start by right-clicking on one's **Part** in the history →



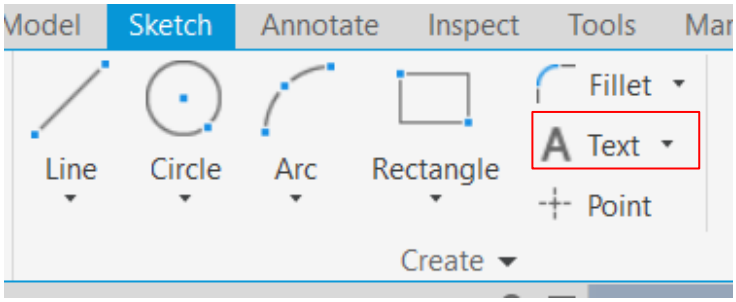
Then click on **Project** at the top of the new window ↓.



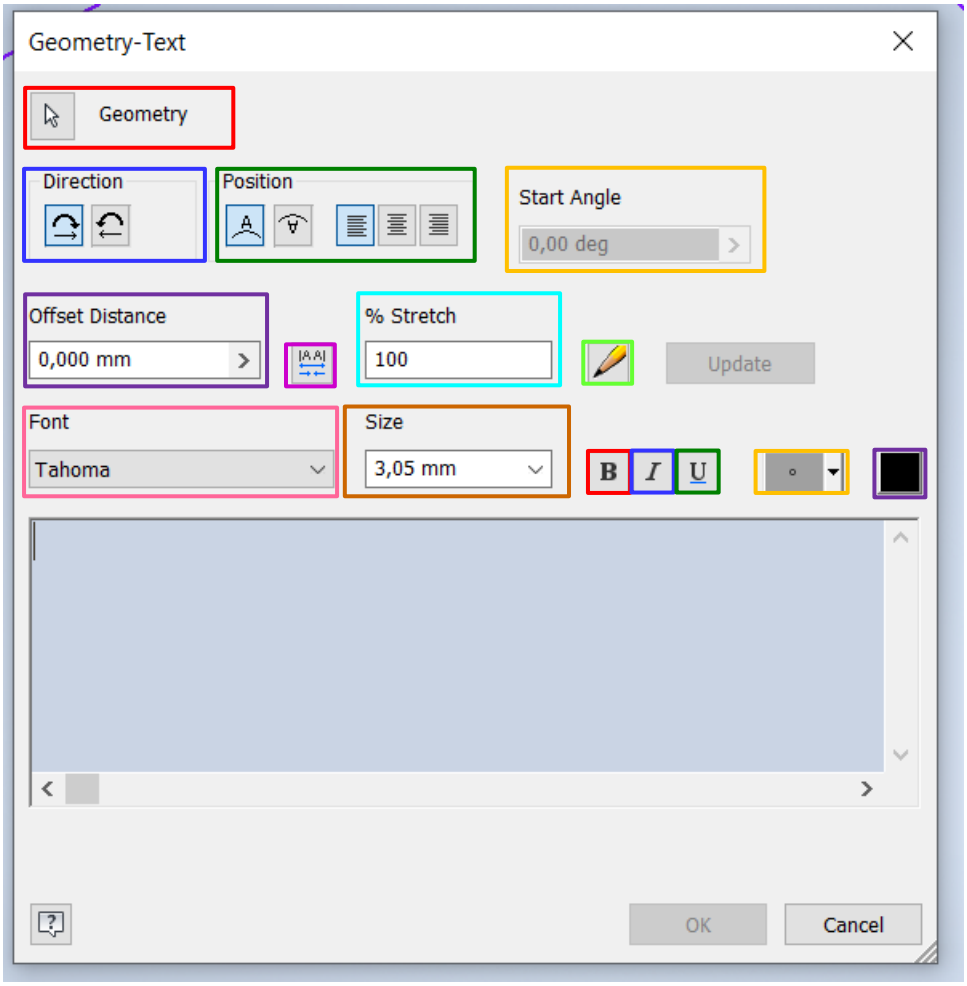
See more under **Drawings** → **Template** → **iProperties**.

Geometry Text

Geometry Text er text that follows a shape.



To locate the **Geometry Text** feature; click on the arrow below "**Text**", then a drop-down menu will roll down, where you can see the **Geometry Text** feature.



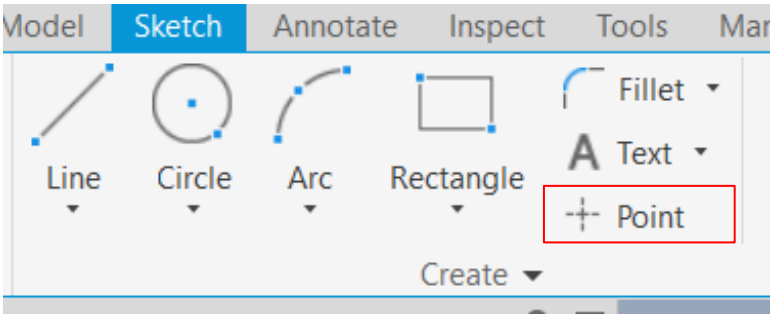
Red = Select the geometry the text should follow.
Blue = The direction of the text, towards you or away from you. (Mirror).
Green = Placement of the text.
Yellow = What angle the text starts.
Purple = How much the text is offset by.
Magenta = Fills in the text on the geometry.
Cyan = Stretches the text onto the geometry.
Lime = Opens text settings.
Pink = Font.
Brown = Size.
Red 2 = Bold.
Blue 2 = Italic.
Green 2 = Underline.
Yellow 2 = Insert symbols.
Purple 2 = Colour.

Point

Introduction:

The **Point** feature is used to define a single **Point** in 3D space. This point is a kind of reference point, and can for example, be used for positioning a **Hole**. **Sketch Point** is used to reference in 2D space. You can change the **Point** type under the **Format** tab.

In order to use the **Point** feature, locate **Create** tab at the top of the screen:



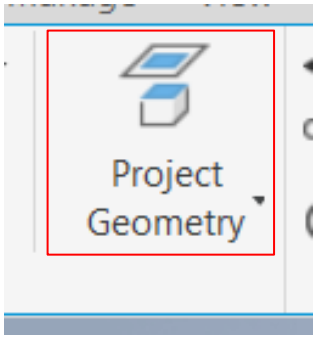
Points are good when you need dimensions. It is recommended that you focus on using **Points** and **Constraints** to create your **Sketches**.

Project Geometry

Introduction:

The **Project Geometry** feature is used to project a geometric shape from one **Plane** to another **Plane** or surface. Here you choose yourself what is projected.

In order to use the **Project Geometry** feature, locate tab at the top of the screen:



To locate the other **Project Geometry** features; Click on the arrow below "**Project Geometry**", then a drop-down menu will roll down, where you can see all the **Project Geometry** features.

Project Cut Edges

This feature is used to create a flat projection of the outer edges of a three-dimensional object.

Project to 3D Sketch

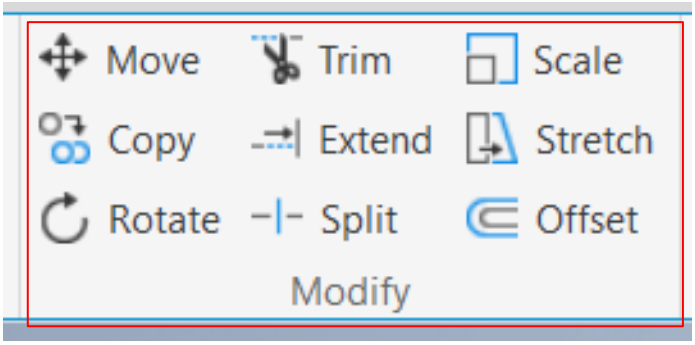
Project the **Sketch** to a **3D Sketch** on a selected surface.

Project DWG Geometry

In order to use this feature, there must be an imported DWG file. Since it is projected from a DWG file.

Modify

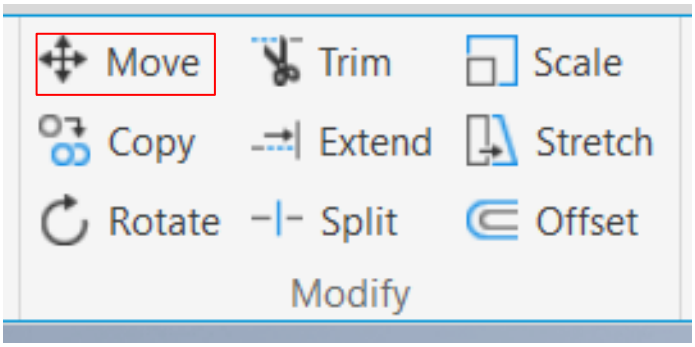
In order to use the **Modify** features, locate tab at the top of the screen:



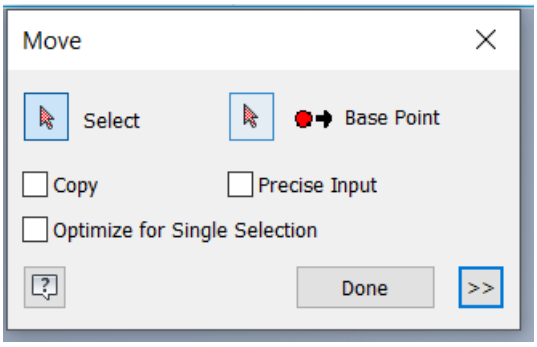
Modify features, changes shapes and lines.

Move

In order to use the **Move**, locate **Modify** tab at the top of the screen:



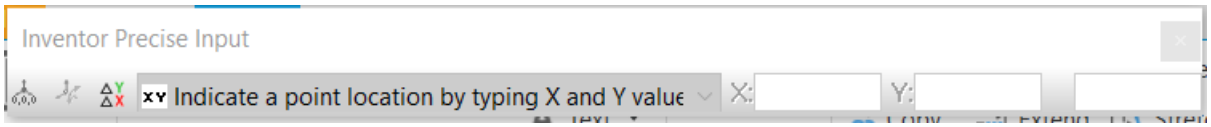
With this feature you can move shapes and lines:



First you select the lines you want to move, and then the point you want to move from.

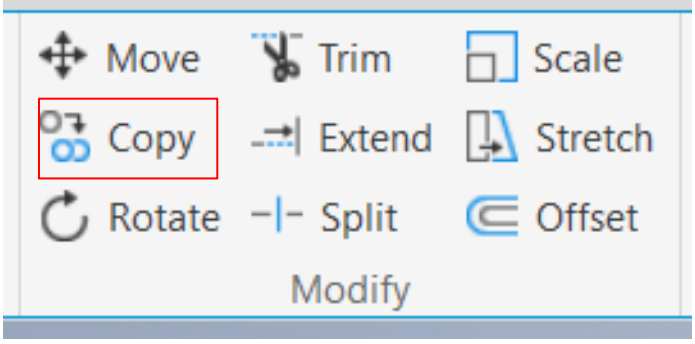
Copy = Here, you copy while moving.

Precise Input = Here, a box appears where you can enter precise coordinates:



Copy

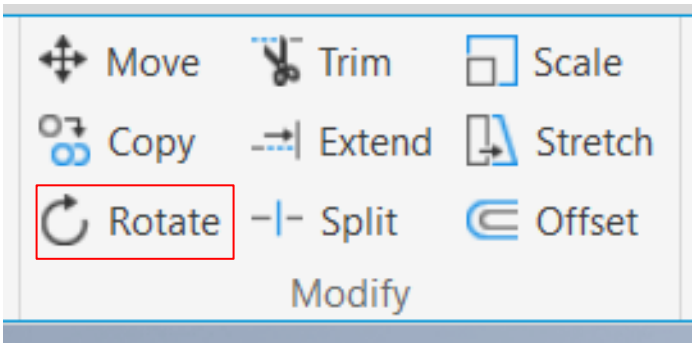
In order to use the **Copy**, locate **Modify** tab at the top of the screen:



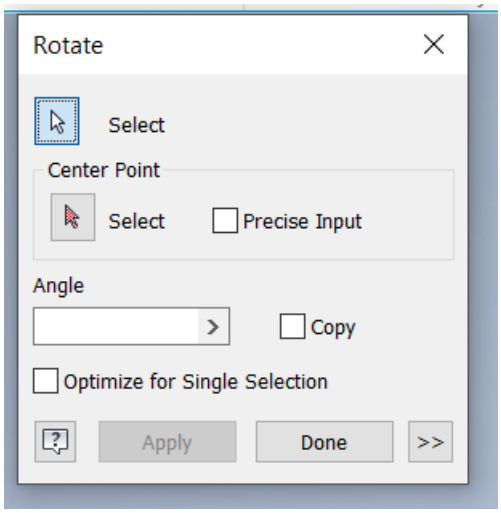
Works the same way as **Move**, just for each placement a copy is made.

Rotate

In order to use the **Rotate**, locate **Modify** tab at the top of the screen:



Rotate a shape or line from a specific point.

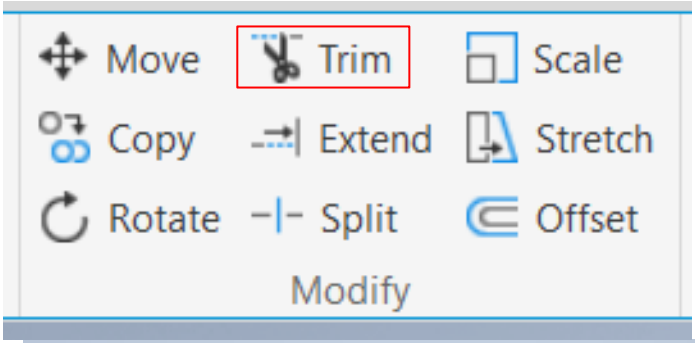


The box works almost in the same way as **Move** and **Copy**. Here "**Base Point**" = "**Center Point**".

Angle = Determines the number of degrees rotated.

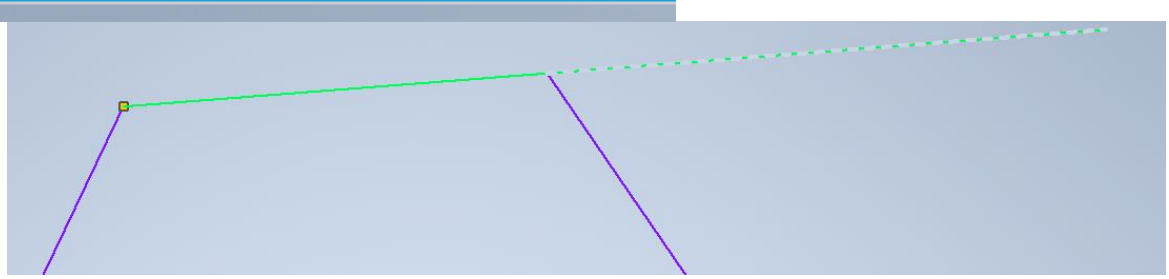
Trim

In order to use the **Trim**, locate **Modify** tab at the top of the screen:



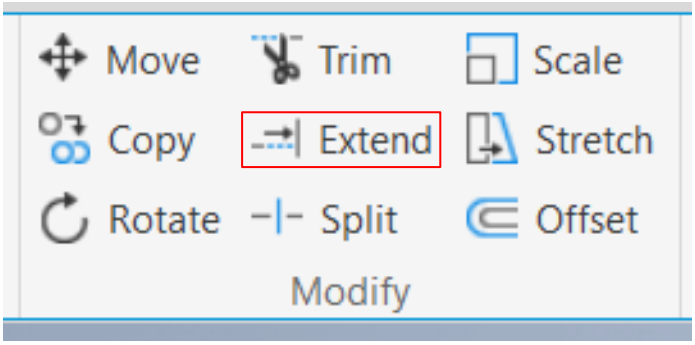
Trims lines when you click on them:

Example:



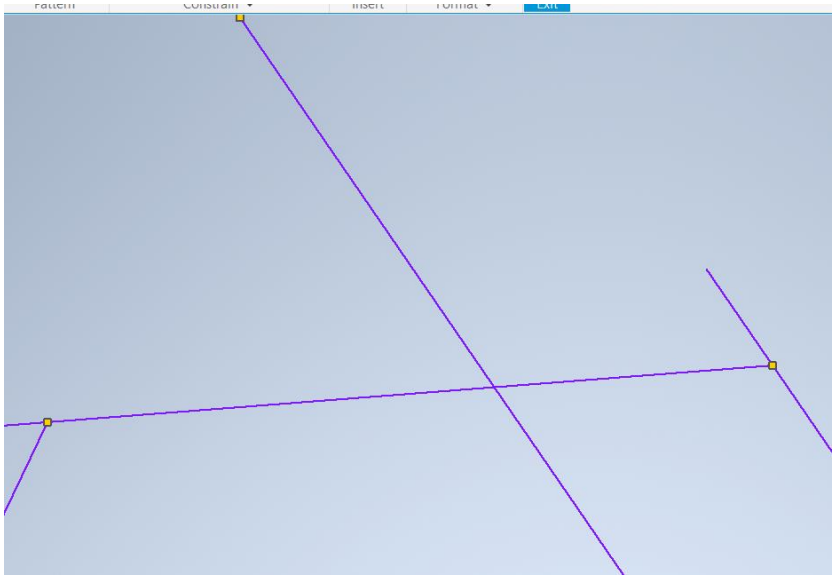
Extend

In order to use the **Extend**, locate **Modify** tab at the top of the screen:



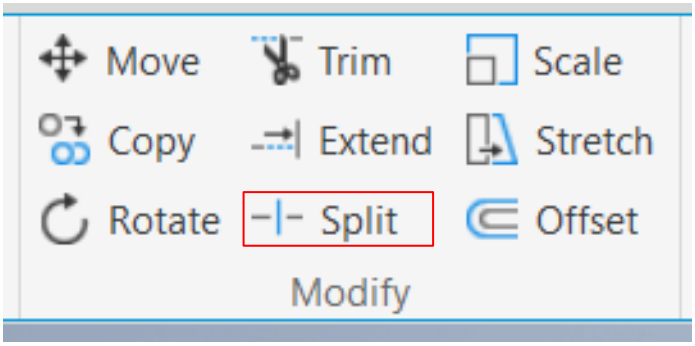
Extends lines to the next line. If there is no line that stops the feature, it will be **Extended** to the edge of the window.

Example:



Split

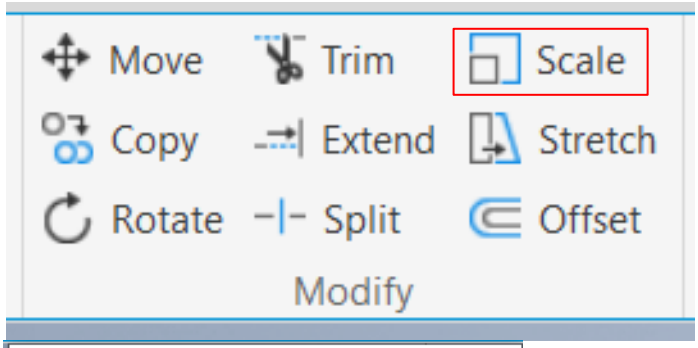
In order to use the **Split**, locate **Modify** tab at the top of the screen:



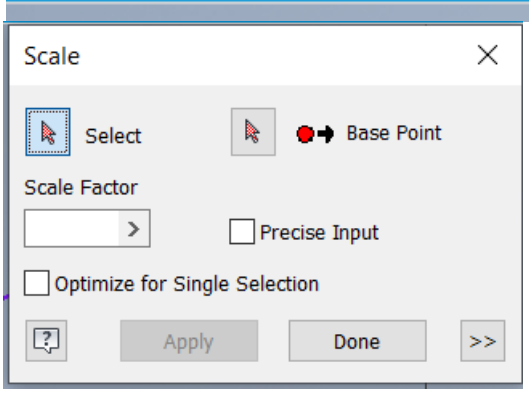
Splits a line across, via a cross/edge.

Scale

In order to use the **Scale**, locate **Modify** tab at the top of the screen:



Scales shapes.

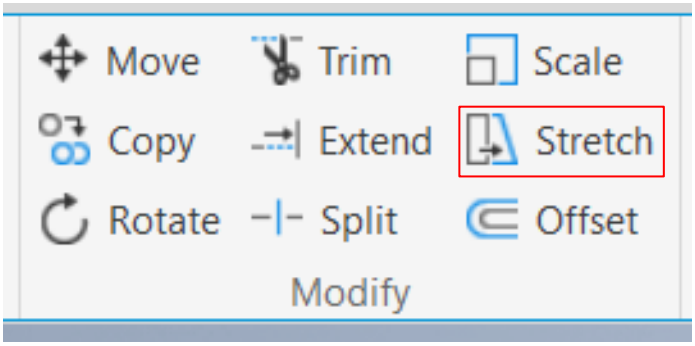


Works the same way as **Move**.

Scale Factor = How much you scale with.

Stretch

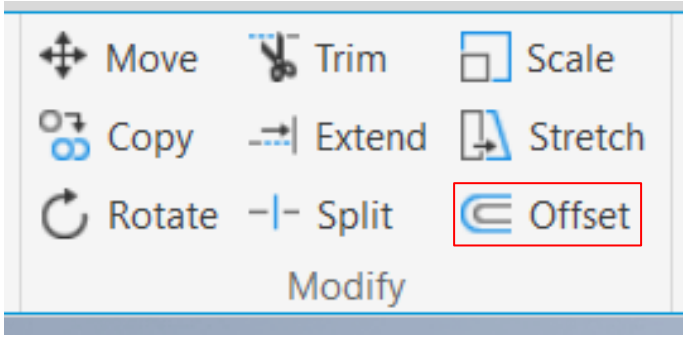
In order to use the **Stretch**, locate **Modify** tab at the top of the screen:



Works the same way as **Move**. here you don't have to mark the whole shape, but the part you want to pull out.

Offset

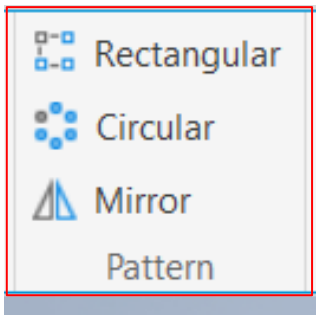
In order to use the **Offset**, locate **Modify** tab at the top of the screen:



Offsets a shape, makes a copy of the shape that is smaller or larger than the original. Most often with a defined dimension.

Pattern

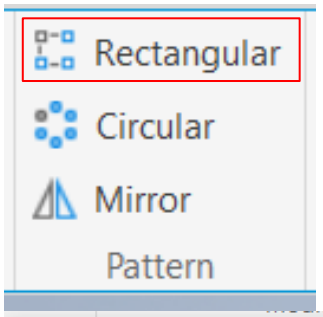
In order to use the **Pattern** features, locate tab at the top of the screen:



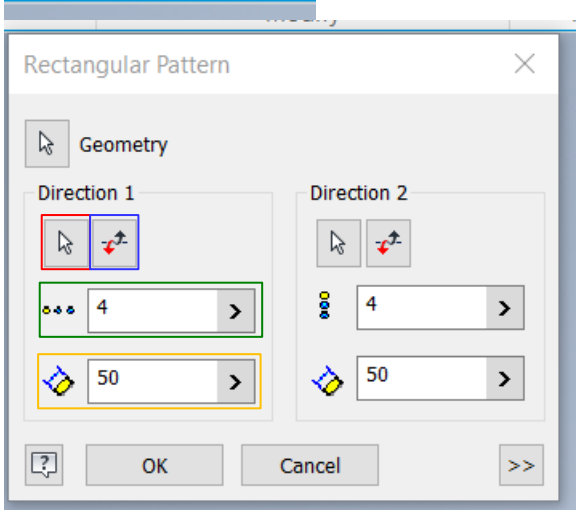
Under **Pattern** there are 2 patterns and a mirror feature.

Rectangular

In order to use the **Rectangular**, locate **Pattern** tab at the top of the screen:



Select **Rectangular**, when you press **Rectangular** a box will appear:



Geometry = Activates, so you can mark your shape.

Direction 1 = Is left or right.

Direction 2 = Is up and down.

Red = Activates placement of a **Direction**.

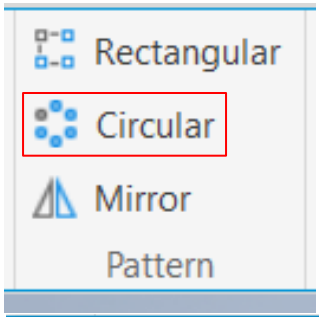
Blue = Flips **Direction**.

Green = Amount.

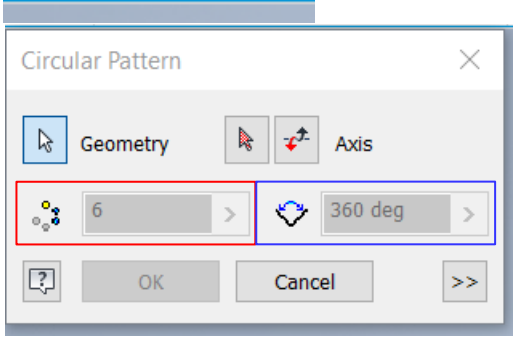
Yellow = Space.

Circular

In order to use the **Circular**, locate **Pattern** tab at the top of the screen:



Select **Circular**, when you press **Circular** a box will appear:



The icons mean the same as **Rectangular**, here **Direction** = **Axis**.

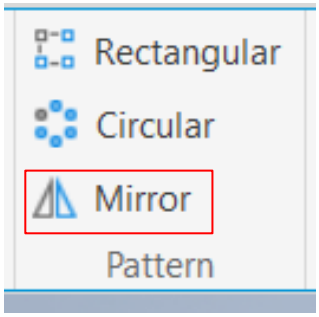
The axis is a point that you place.

Red = Amount.

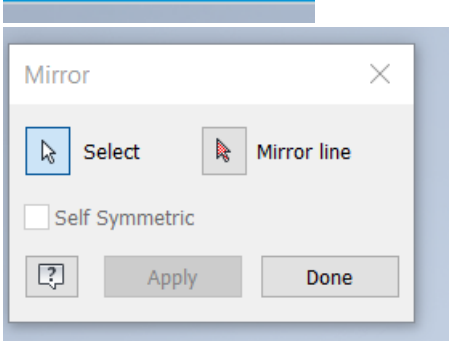
Blue = Angle.

Mirror

In order to use the **Mirror**, locate **Pattern** tab at the top of the screen:



Select **Mirror**, when you press **Mirror** a box will appear:



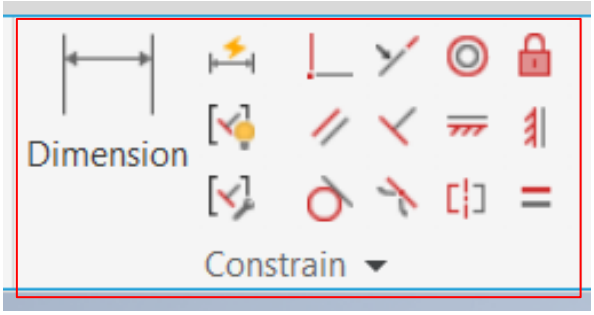
The **Mirror** feature, mirrors a figure over a line.

Select = Select a shape.

Mirror Line = Select mirror line.

Constrain

In order to use the **Constrain** features, locate tab at the top of the screen:



The **Constrain** feature locks lines so that they do not move. As a **recommendation**, your lines in the **Sketch** should be either black or yellow.

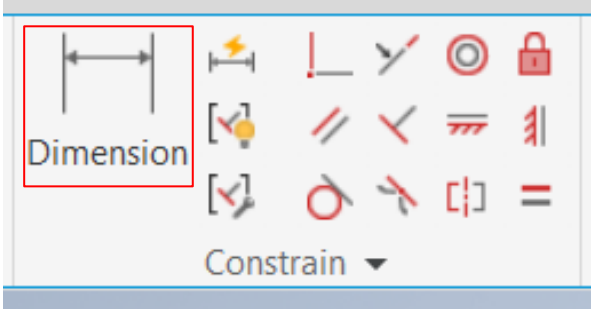
Purple lines = Not locked and able to move.

Black lines = Is locked and has been given a Constrain.

Yellow lines = "Project Geometry" lines.

Dimension

In order to use the **Dimension**, locate **Constrain** tab at the top of the screen:



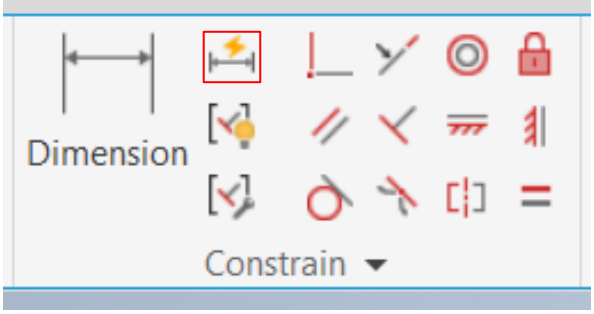
The **Dimension** feature is used to measure one's lines, circles, and shapes, etc.

You can enter numbers, variables, equations, and parameters in the **Dimension** box.

- **Shortcut key** = D

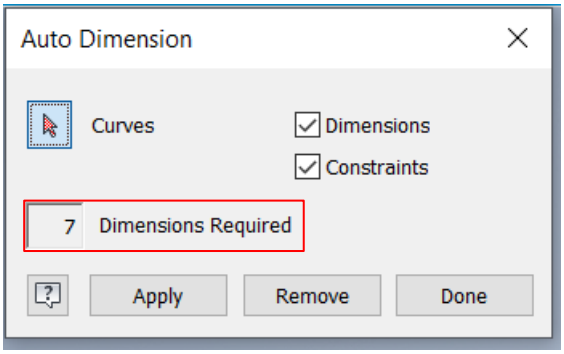
Automatic Dimensions and Constraints

In order to use the **Automatic D & C**, locate **Constrain** tab at the top of the screen:



It is not **recommended** to use this feature, as Inventor often does not use them correctly. However, use them if you are in doubt about which **Constrain** to use.

Select **Automatic D & C**, if you press on **Automatic D & C** a box will appear:



Curves = Turns marking on/off, which lines are used.

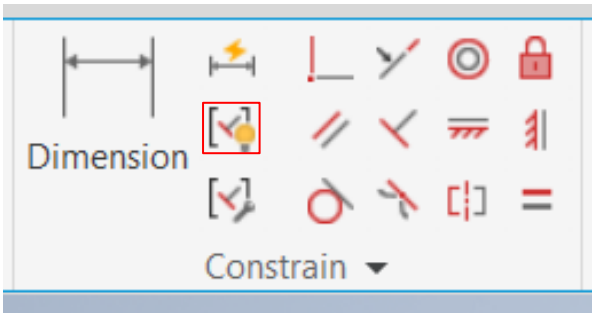
Dimensions = Turns on/off.

Constraints = Turns on/off.

Red = How many Inventor thinks are missing.

Show Constraints

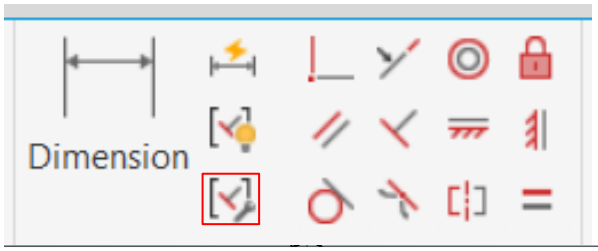
In order to use the **Show Constraints**, locate **Constrain** tab at the top of the screen:



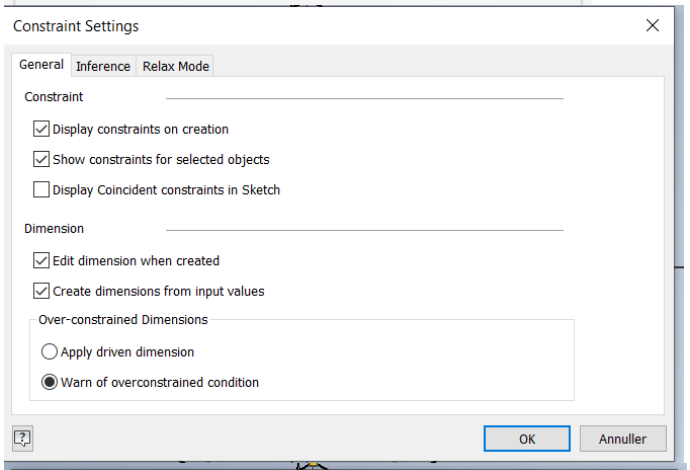
Will display the used **Constraints** on a line. You can also display all **Constraints** with the *"**F8**"* key on your keyboard. Two remove all **Constraints** it is *"**F9**"*.

Constraint Settings

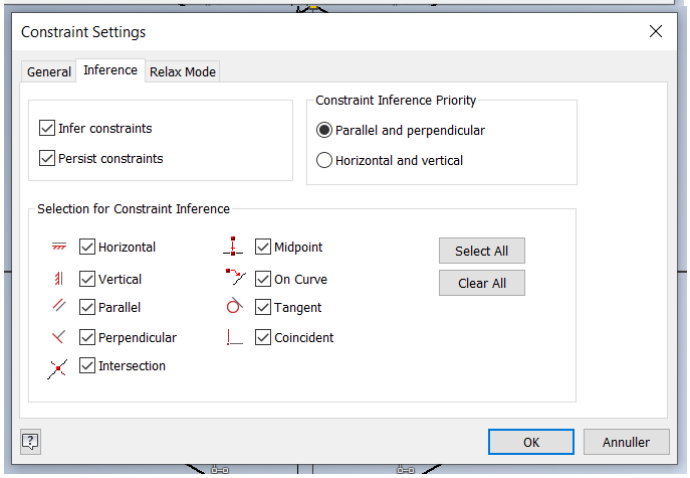
In order to use the **Constraint Settings**, locate **Constrain** tab at the top of the screen:



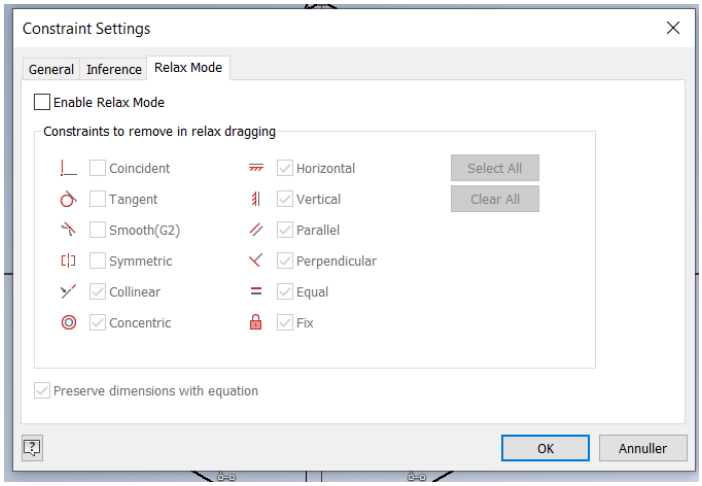
Select **Constraint Settings**, if you press **Constraint Settings** a box with 3 tabs appears:



← Here is the "**General**" tab.



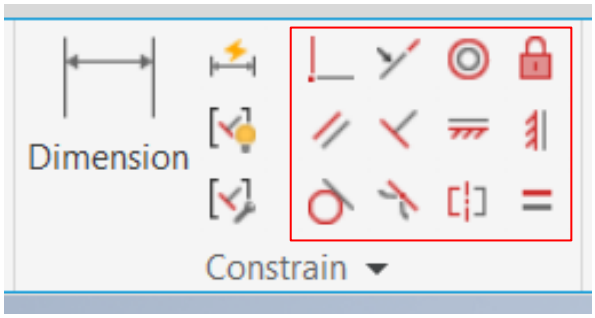
← Here is the "**Inference**" tab.



← Here is the "**Relax Mode**" tab.

Constraints:

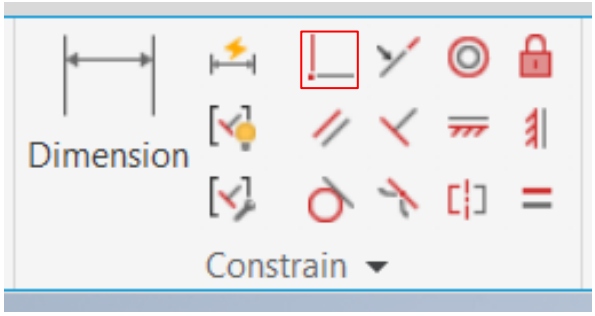
In order to use the **Constraints**, locate **Constrain** tab at the top of the screen:



All features in the red box are **Constraints**, they are shown with icons, that you can hover over to get an English explanation of what they do.

Coincident

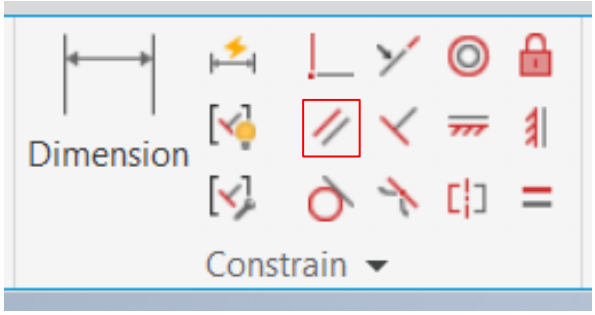
In order to use the **Coincident**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks a point to a line but can still move on the given line. Also used to lock 2 points to each other.

Parallel

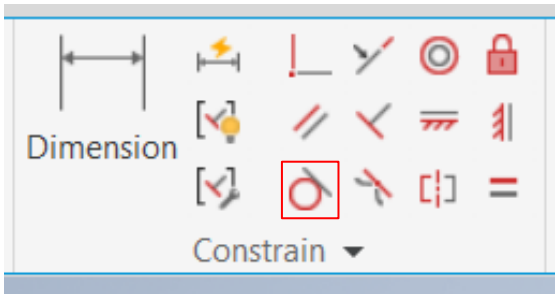
In order to use the **Parallel**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks 2 things (e.g. lines) in parallel with each other.

Tangent

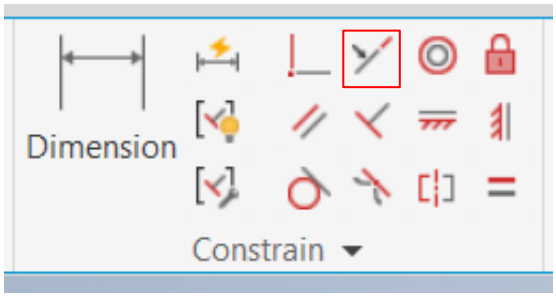
In order to use the **Tangent**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks 2 arcs tangent to each other, this also applies if you have to extend an arc.

Collinear

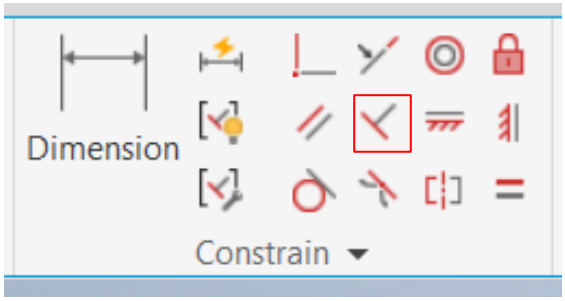
In order to use the **Collinear**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks 2 or more lines to lie on top of each other.

Perpendicular

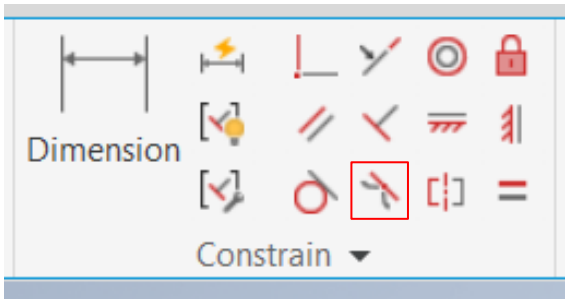
In order to use the **Perpendicular**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks linear geometry to lie 90° to each other.

Smooth (G2)

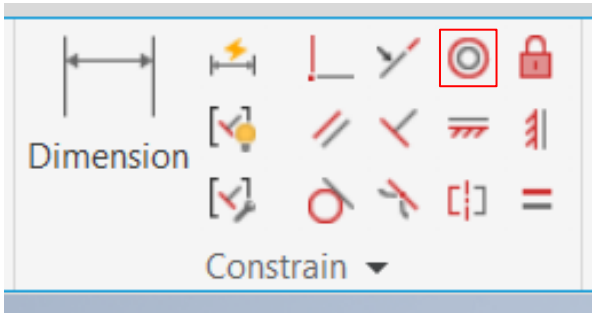
In order to use the **Smooth**, locate **Constrain** tab at the top of the screen:



The **Constraint**, softens arcs and curves against other geometry. In order to use the 2 lines, they must be marked: First the arc/curve and then an adjacent line.

Concentric

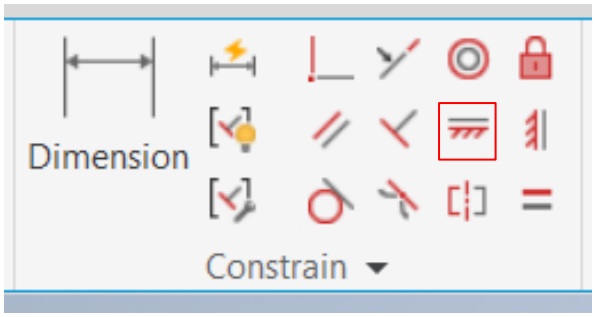
In order to use the **Concentric**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks 2 round shapes center point together.

Horizontal

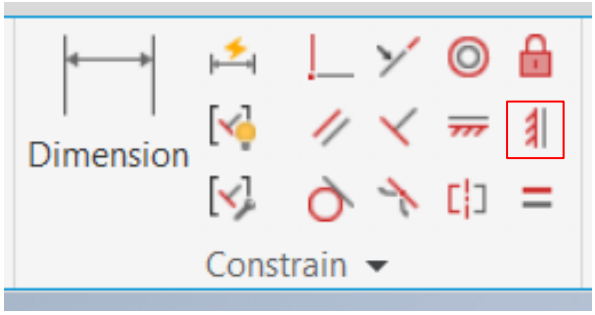
In order to use the **Horizontal**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks a line parallel to the **X-Axis**.

Vertical

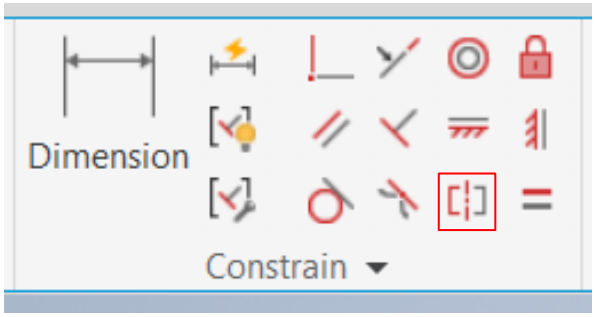
In order to use the **Vertical**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks a line parallel to the **Y-Axis**.

Symmetric

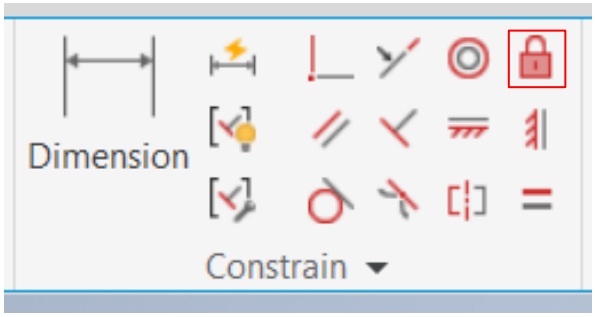
In order to use the **Symmetric**, locate **Constrain** tab at the top of the screen:



The **Constraint**, locks symmetrically over a line.

Fix

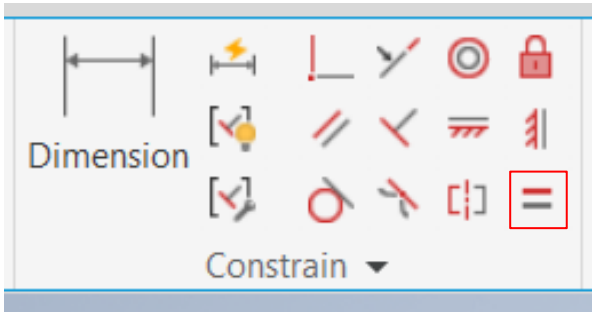
In order to use the **Fix**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks geometry, in the given position.

Equal

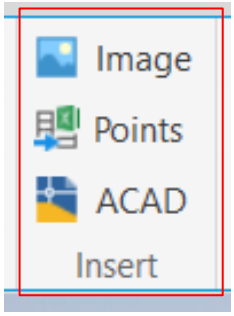
In order to use the **Equal**, locate **Constrain** tab at the top of the screen:



The **Constraint** locks equal **Dimensions** and automatically adjusts the **Dimensions** so that they are equal.

Insert

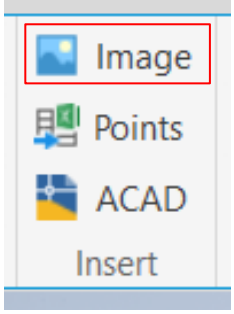
In order to use the **Insert** features, locate tab at the top of the screen:



Imports from other filetypes/programs.

Image

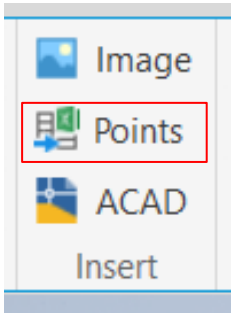
In order to use the **Image**, locate **Insert** tab at the top of the screen:



Here you can import an image file. This is used for e.g. The **Decal** feature.

Points

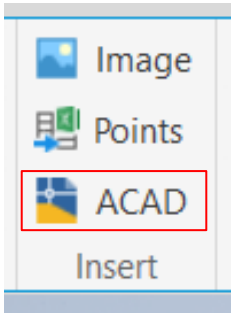
In order to use the **Points**, locate **Insert** tab at the top of the screen:



Here you can import coordinate points from a spreadsheet such as **Excel**.

ACAD

In order to use the **ACAD**, locate **Insert** tab at the top of the screen:

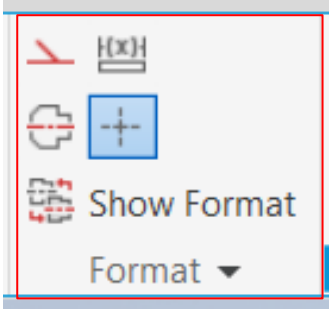


ACAD = AutoCAD

Here you can import 2D **AutoCAD** data into the **Sketch**.

Format

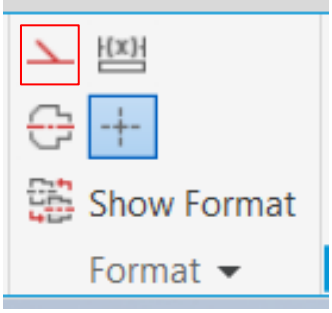
In order to use the **Format** features, locate tab at the top of the screen:



Under **Format** there are some frequently used features such as; **Centerline**, **Construction Line** and **Center Point**.

Construction

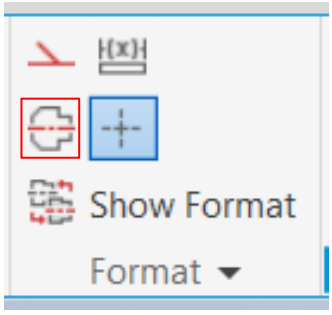
In order to use the **Construction**, locate **Format** tab at the top of the screen:



Construction Line is a temporary line that helps build dimensions in a model.

Centerline

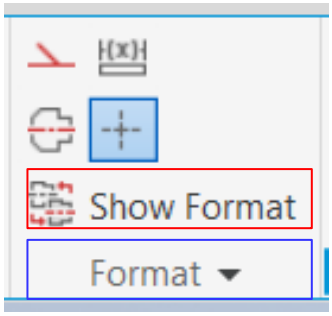
In order to use the **Centerline**, locate **Format** tab at the top of the screen:



Centerline is a line that represents the center of a geometry

Show Format

In order to use the **Show Format**, locate **Format** tab at the top of the screen:



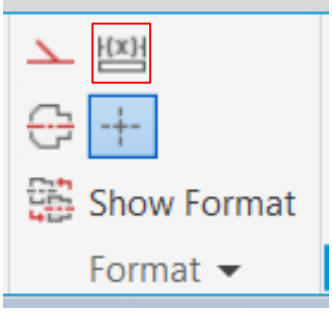
Toggles between:

- Your own changes to the lines (eg line type, color and thickness).
- Inventor's **Standard** for all subjects.

Blue = Here you can change the line-settings. ***Remember*** *to mark the line before changing it.*

Driven Dimension

In order to use the **Driven Dimension**, locate **Format** tab at the top of the screen:



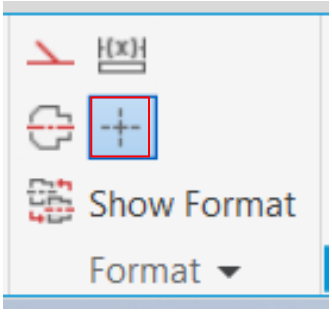
Locks and unlocks **Dimension** changes.

”**123**” = Can be changed.

”**(123)**” = Can't be changed.

Center Point

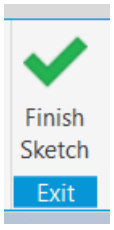
In order to use the **Center Point**, locate **Format** tab at the top of the screen:



Changes one's point (from the **Create** tab) between **Center Point** and a **Sketch Point.**

Center Point = Is used automatically for the **Hole** feature (under the 3D **Modify** tab).

Sketch Point = Is not used automatically for the **Hole** feature (under the 3D **Modify** tab).



Create – Features

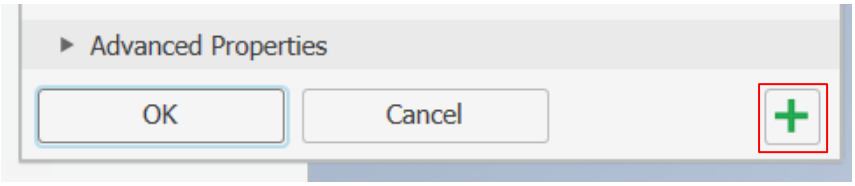
Introduction

After starting a project and creating a **Part** file, you can create a **Sketch**, which is the fundamental building block for creating 3D models in Inventor. Inventor constructs **Solids**, which is what Inventor refers to a shape.

General

Green +

The Green + on all the **Create** features. You use them, to make several different "**Properties**" settings, but on the same **Sketch**.



Index:

Shortcut Keys

Selections:

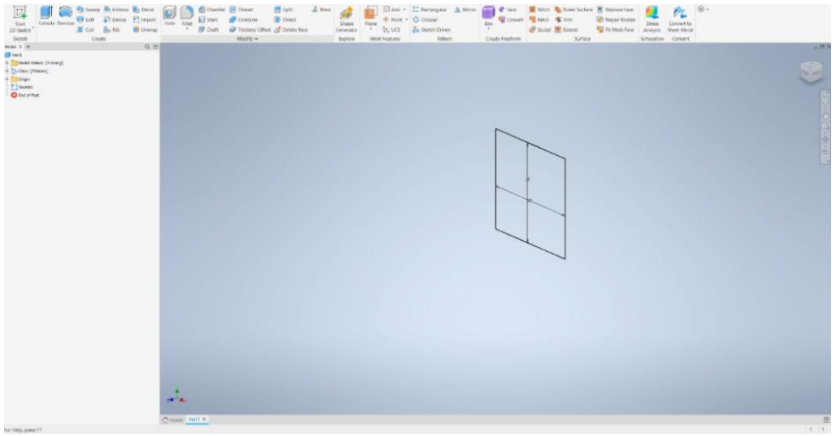
Control (CTRL) = Adds more than one selection. ***Press and hold.*** *(It is done correctly when there is a plus icon next to the mouse).*

SHIFT (↑) = Removes selections. ***Press and hold.*** *(It is done correctly when there is a minus icon next to the mouse).*

Extrude

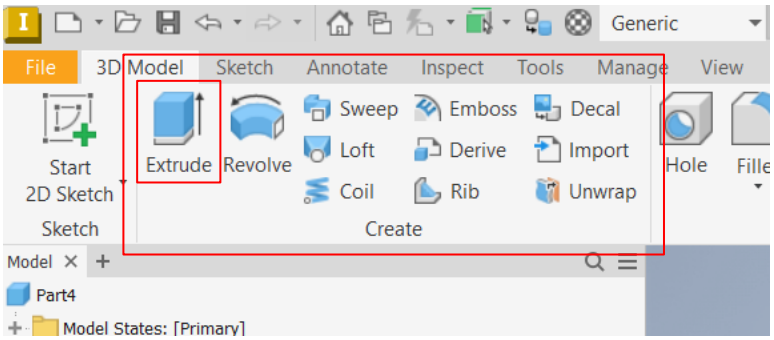
Introduction:

The **Extrude** feature is one of the most used. **Extrude** is a way to create three-dimensional objects by pulling in a two-dimensional **Sketch**. When you finish your **Sketch**, it can look like this:



Here is an example of, what a **Sketch** might look like when it is finished.

In order to use **Extrude**, locate **Create** tab at the top of the screen:

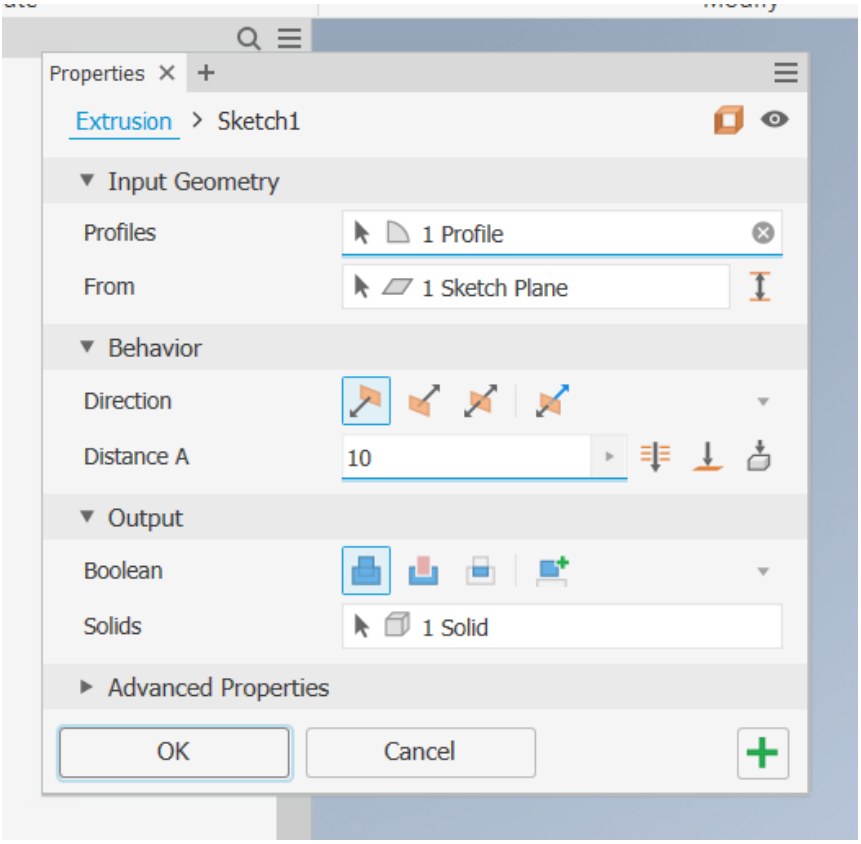


Select **Extrude**. When you press **Extrude** a box will appear. (Inventor can decide to auto select your **Sketch**, e.g. if your **Sketch** is as simple as a square).

- **Short key** = E

How to use the Box (First)

This is what **Properties** looks like, the first time you press **Extrude**:



Input Geometry

Profiles:

Profile is the surface used. In the box, Inventor informs you of how many **Profiles** are selected. The blue line under the word "**Profile**" shows that "**Profile**" is active. if it is active, you can select or off select surfaces. If it's not active, you cannot.

If it is not active, you can make it active by clicking on "**Profile**". But Inventor auto enables "**Profile**".

From:

Informs about which **Plane** is selected, this is automatically on a "**Sketch Plane**" (*this means that Inventor selects to proceed from the **Plane** you selected for your **Sketch***).

Behavior

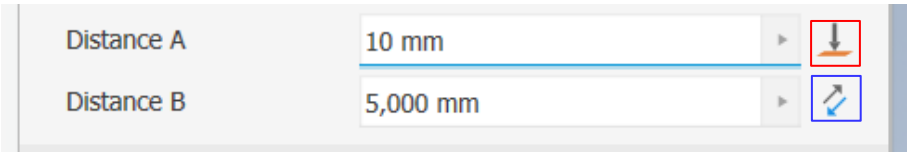
Direction:

Here are 4 icons.

- *First (from the left):*
  - This is Inventor's auto selection. Here it **Extrudes** from your **Plane**.
- *Second:*
  - **Extrudes** it backwards from your **Plane**.

- *Third:*
  - **Extrudes** it both ways symmetrically from your **Plane**.
- *Last:*
  - **Extrudes** it both ways asymmetrically from your **Plane**.

Distance:



With the first, three there are 1 box called Distance A, on the last there are 2 boxes Distance A and B.

The boxes indicate how much Inventor **Extrudes** with.

Red - ICON (To):
Extrudes to a **Plane**.

Blue - ICON (Flip Direction):
Flips **Distance A** and **B**.

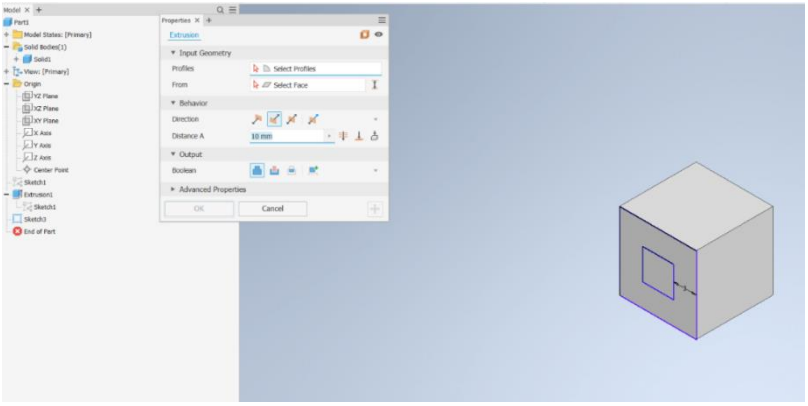
Output

When a **CREATE** feature is activated, Inventor creates a new **Solid**, which is what Inventor calls a shape. You can have more than one **Solid** in a **Part** file. E.g. 2 cubes can be 2 **Solid's**.

Body Name:
Rename "**Solid**" here.

How to use Properties (Other times)

When using **Extrude** more than once. it looks like this:

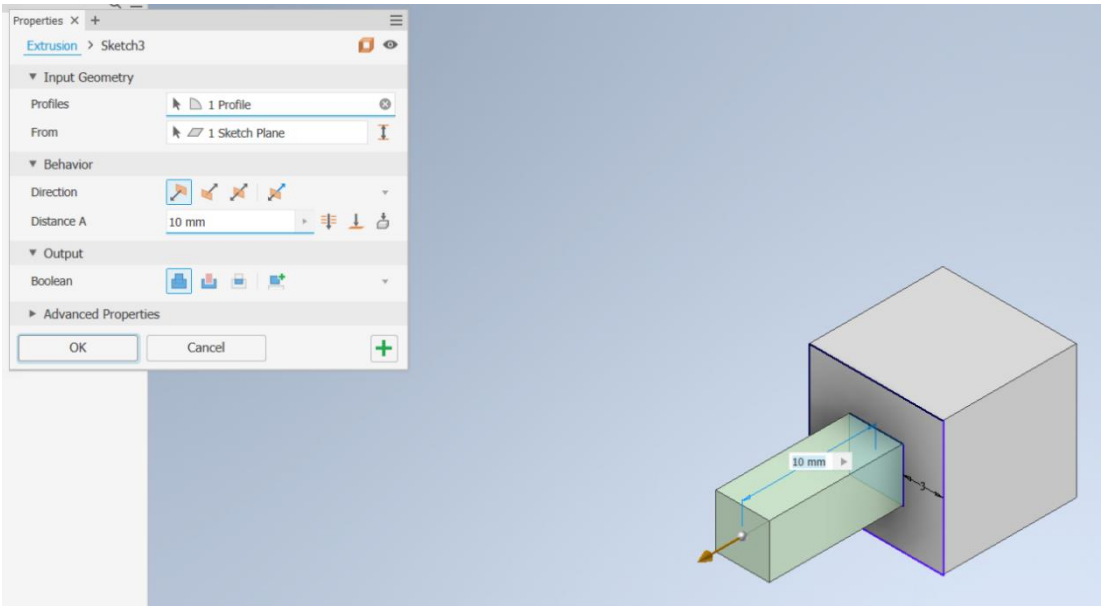


Here, as an example, I have made a square on the front of the first. I offset it by 3 mm towards the center.

Input Geometry

Profiles:

Works the same way as **Extrude → How to use... (First) → Input Geometry → Profiles**. Here, Inventor has not auto selected a surface. To select, ensure that "**Profile**" is active and click on a surface you want to use.



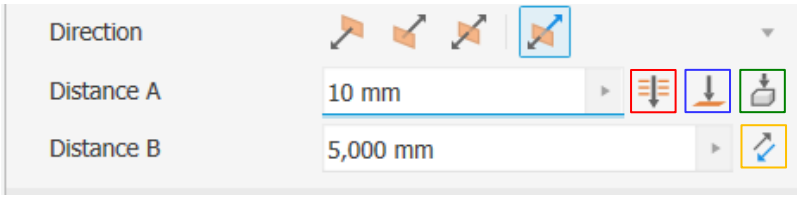
From:

Works in the same way as **Extrude → How to use... (First) → Input Geometry → From**.

Behavior

Direction:

Works in the same way as **Extrude → How to use... (First) → Behavior → Direction**.



Distance:

On the three first there is 1 box called Distance A, on the Last there are 2 boxes Distance A and B.

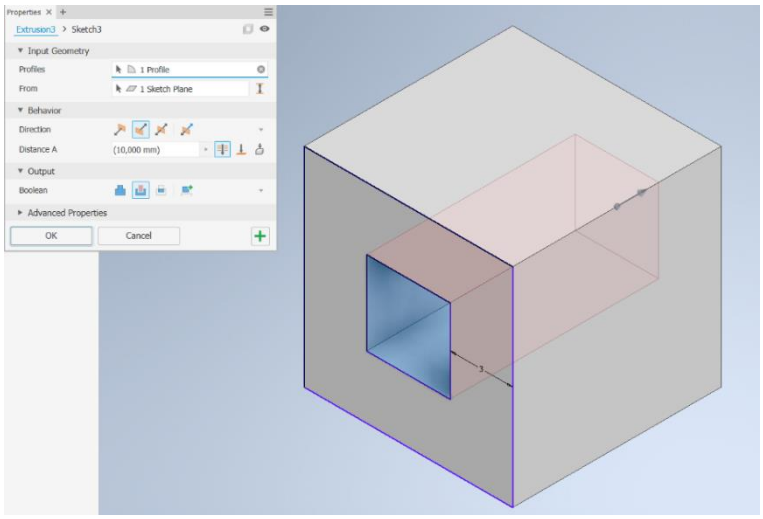
In the boxes you indicate how much Inventor **Extrudes** with.

Red - ICON (Through All):
Extrudes through a **Solid**.

Blue - ICON (To):
Extrudes to a **Plane**.

Green - ICON (To Next):
Extrudes too next **Solid**.

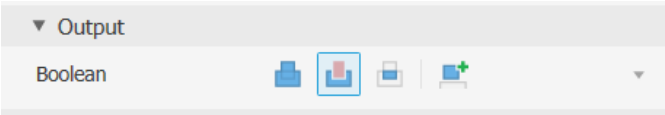
Yellow – ICON Asymmetry:
Flips **Distance A** and **B**.



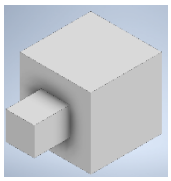
Output

Works in the same way as Extrude → How to use... (First) → Output.

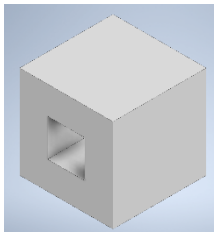
Boolean:



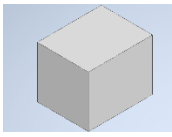
- *First (from the left):*
  - **Join**. Combines the 2 **Extrusions** together into one **Solid**:



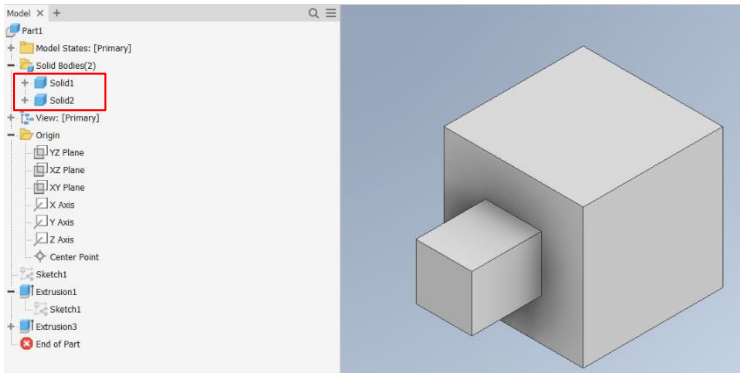
- *Second:*
  - **Cut**. The 2nd. **Extrusion** gets removed from the 1st. **Extrusion**, as a hollow **Solid**:
  - (you usually want to flip it into the first **Extrusion**)



- *Third:*
  - **Intersect**. the 1st. **Extrusion** gets removed and the 2nd. **Extrusion**, becomes its own **Solid**:



- *Last:*
  - **New Solid**. Makes a new **Solid**, then there are 2, one for each **Extrusion**:



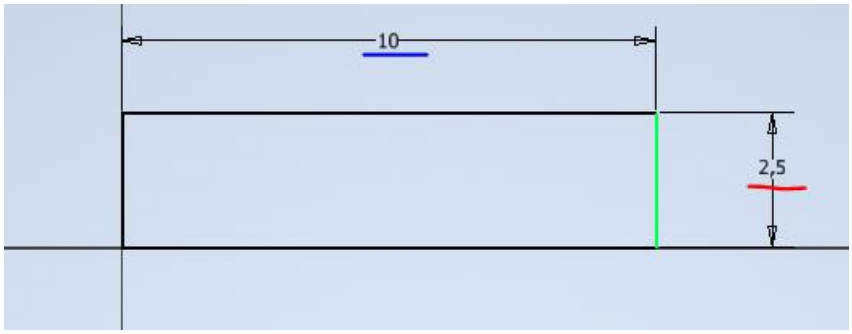
  - Then the **Body Name** section appears: Rename your "**Solid**".

Revolve

Introduction:

The **Revolve** feature is used to make for example cylinder. This is done by rotating a **profile** around an axis.

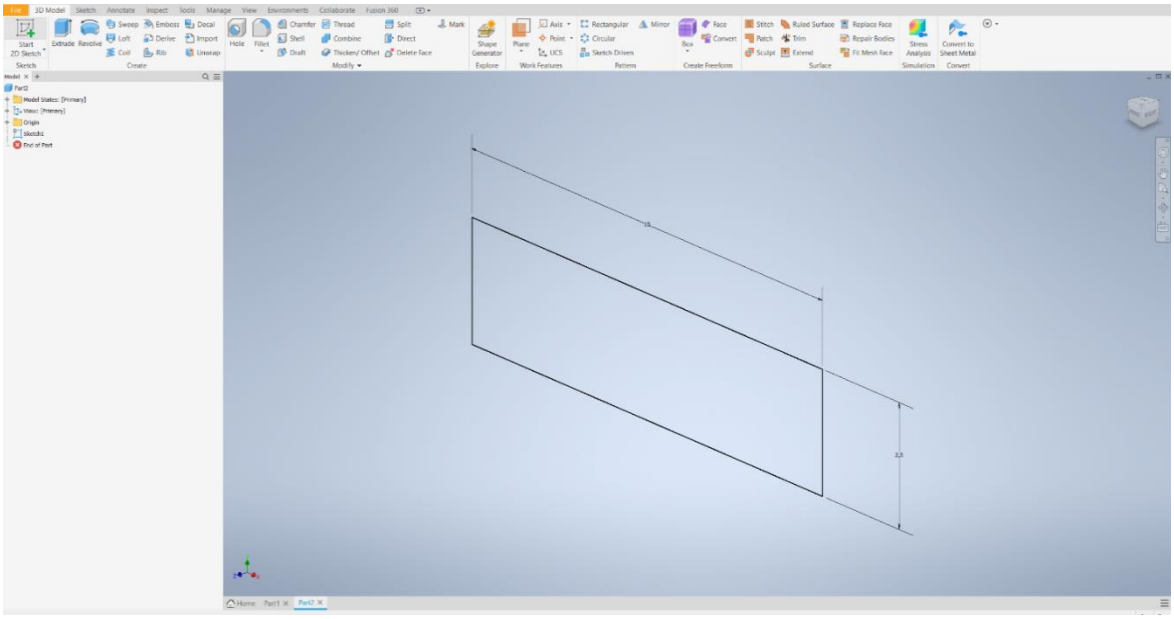
To use **Revolve** you always have to start with a **Sketch**.



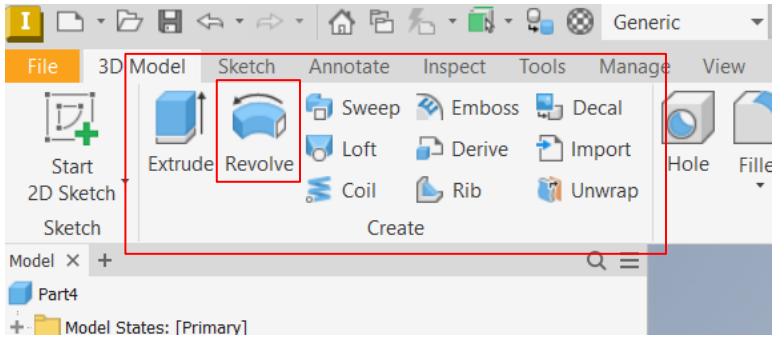
To make a cylinder remember to use radius and not diameter:

Here the number blue is the length of the cylinder, and the number red is the radius.

When the **Sketch** is complete, your screen will look like this:



In order to use **Revolve**, locate **Create** tab at the top of the screen:

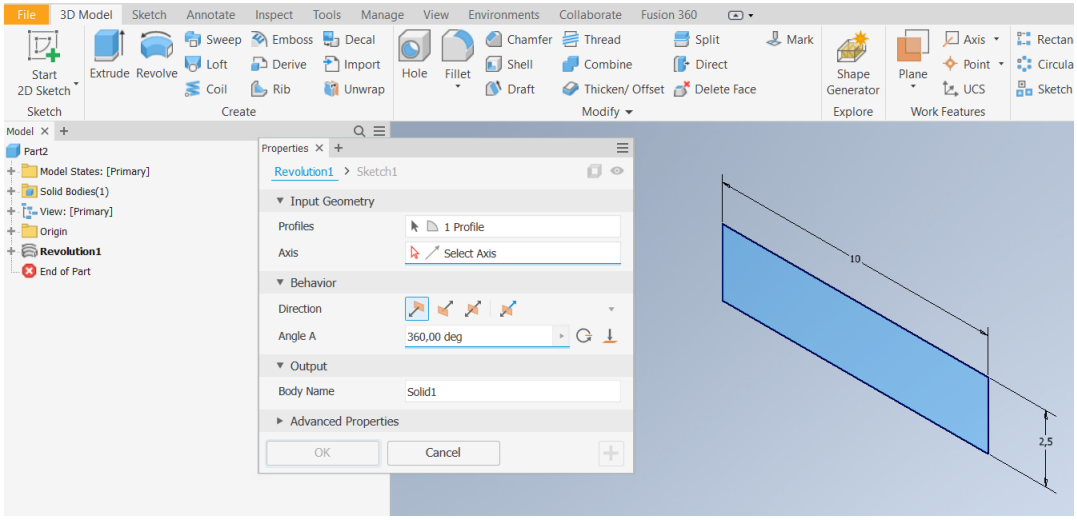


Select **Revolve**. When you press **Revolve** a box will appear. (Inventor can decide to auto select your **Sketch**, e.g. if your **Sketch** is as simple as a square).

- **Shortcut key** = R

How to use Properties (First)

This is what **Properties** looks like, the first time you press **Revolve**:



Input Geometry

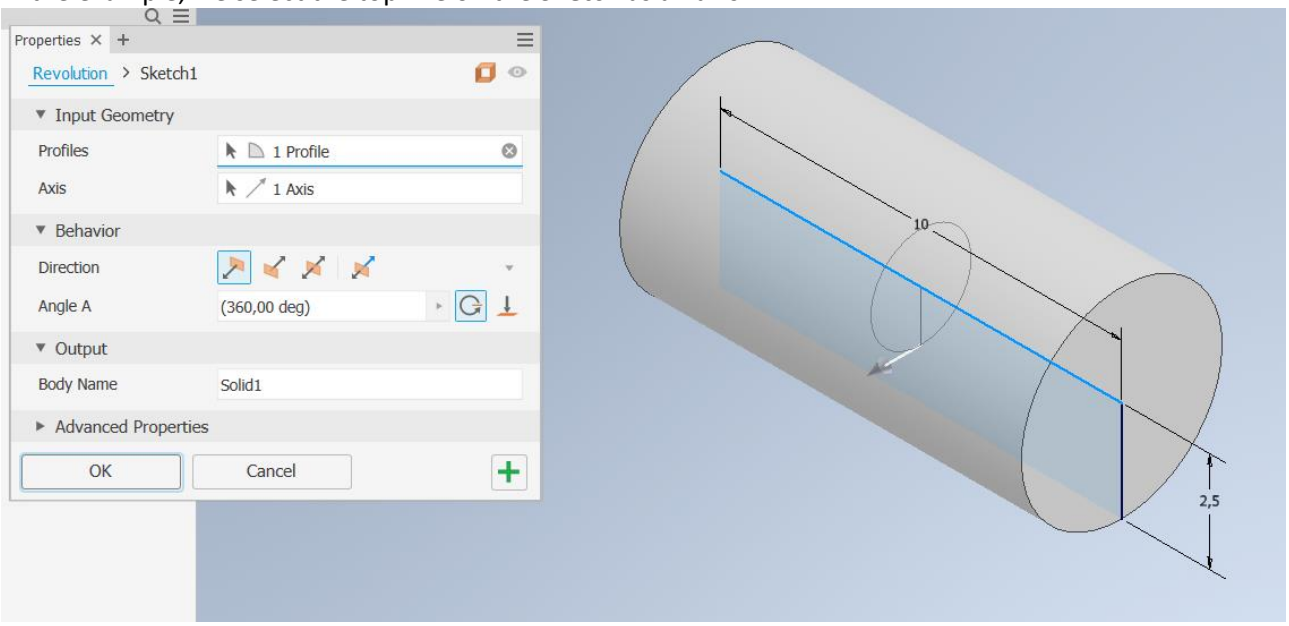
Profiles:

Profile is the geometry that is rotated around the axis. In the box, Inventor informs you how many **Profiles** (this is a surface/**Sketch**) are selected.

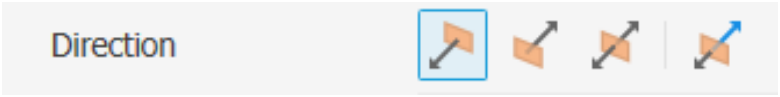
Axis:

Axis is what the geometry rotates around. Is the mouse icon red, that means no axis has been selected. The blue line below the word "**Select Axis**", shows that "**Select Axis**" is active. If the blue line is not there, you cannot click on the desired axis. You can make it active by clicking on "**Select Axis**". But Inventor auto activates "**Select Axis**".

In the example, we select the top line on the **Sketch** as an axis.



Behavior

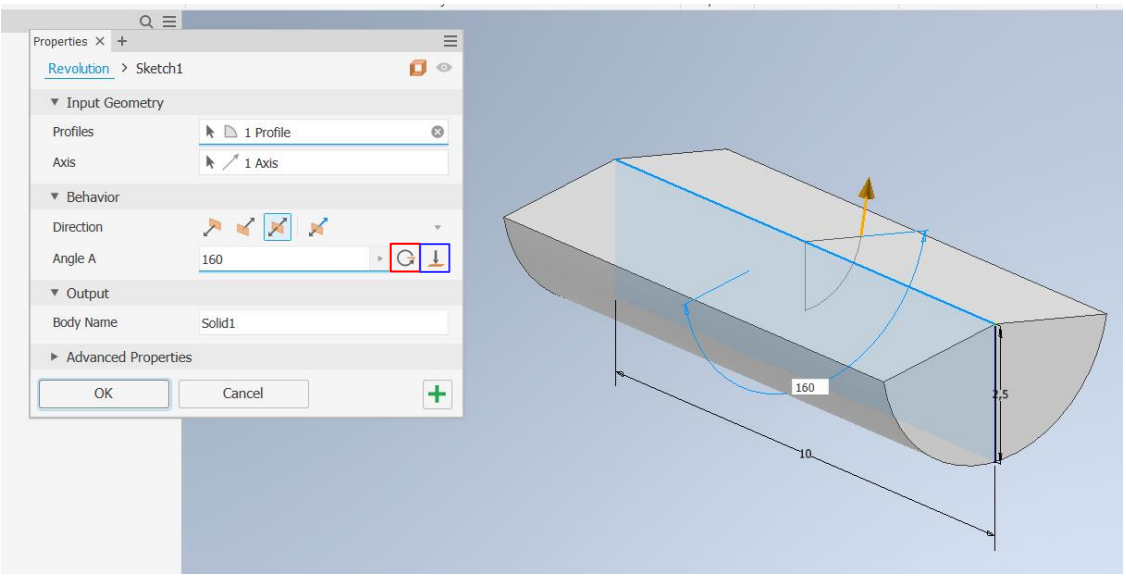


Direction:

Here are 4 Icons.

- *First (From the left):*
  - This is Inventor's auto selection. Here **Extrude** it from your **Plane**.
- *Second:*
  - **Extrudes** it backwards from your **Plane**.
- *Third:*
  - **Extrudes** it both ways symmetrically from your **Plane**.
- *Last:*
  - **Extrudes** it both ways asymmetrically from your **Plane**.

Angle:



Red - ICON (To):
Specifies your **Solid** to 360, that means, it rotates all the way around the axis.

Blue - ICON (Full):
Revolve to a **Plane**.

Output

Body Name:
Rename "**Solid**" here.

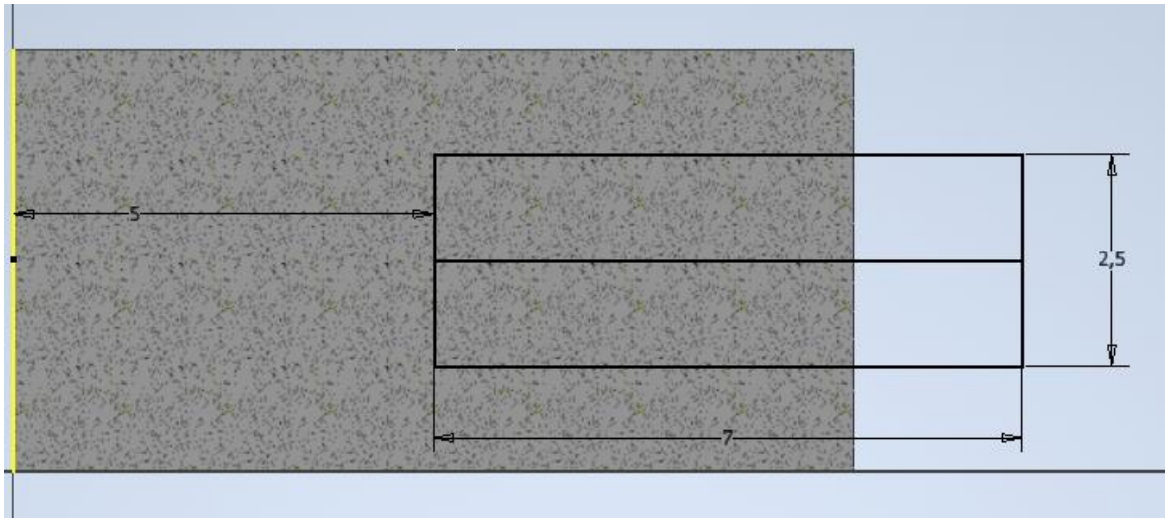
How to use Properties (Other times)

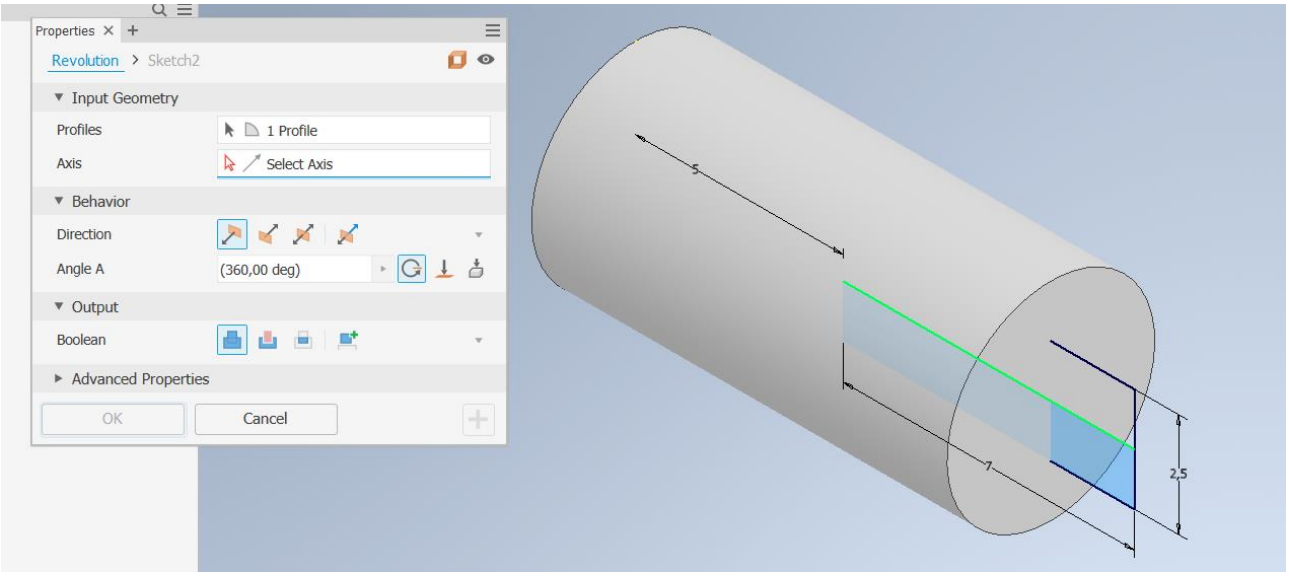
Inventor has more options when using **Revolve** more than once.

Here we will use an example to explain the different options. Start by making a smaller version of the first example, the **Sketch** will look like this:

*Note: here I have used the F7 key (on the keyboard) to cut through the first **Solid**.*

In this example are there 2 squares, and we will only use one. (***Remember**, when making a **Revolve**, that you need to use radius, and not diameter*).





Select the bottom/top surface. And then the line in the middle of the cylinder. (***Remember**, to choose the middle line*).

Input Geometry

Profiles:

Works the same way as **Revolve → How to use... (First) → Input Geometry → Profiles**. Here you can see that Inventor has not selected a surface, so you can do that by ensuring that "**Profile**" is active and clicking on the surface you would like to use. *In the example we select the small cylinder.*

Axis:

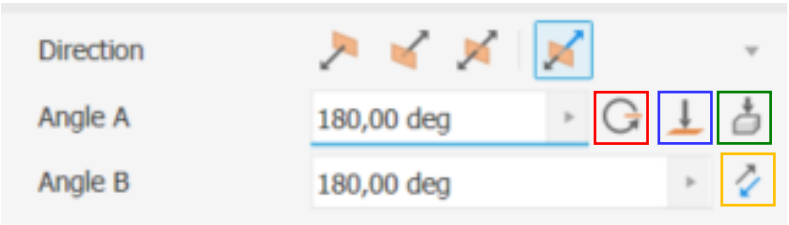
Works the same way as **Revolve → How to use... (First) → Input Geometry → Axis** Here you can see Inventor didn't selected an axis, so you can do that by ensuring that "**Axis**" is active and then click on the surface you want to use. *In the example we select the line in the middle.*

Behavior

Direction:

Works the same way as **Revolve → How to use... (First) → Behavior → Direction**.

Angle:



Red - ICON (Full):
Specifies your **Solid** to 360, that means, it rotates all the way around the axis.

Blue - ICON (To):
Select **Revolve** to a **Plane**.

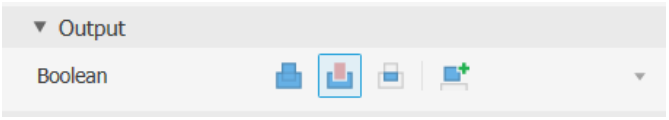
Green - ICON (To Next):
Select **Revolve** the next **Solid**.

Yellow – ICON Asymmetry:
Flips **Distance A** and **B**.

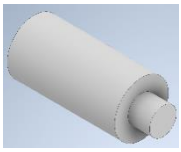
Output

When a **CREATE** feature is activated, Inventor creates a new **Solid**, which is what Inventor calls a shape. You can have more than one **Solid** in a **Part** file. E.g. 2 Squares can be 2 **Solid's**.

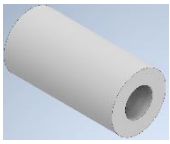
***Boolean**:*



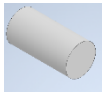
- *First (from the left):*
  - **Join**. Combines the 2 **Extrusions** together into one **Solid**:



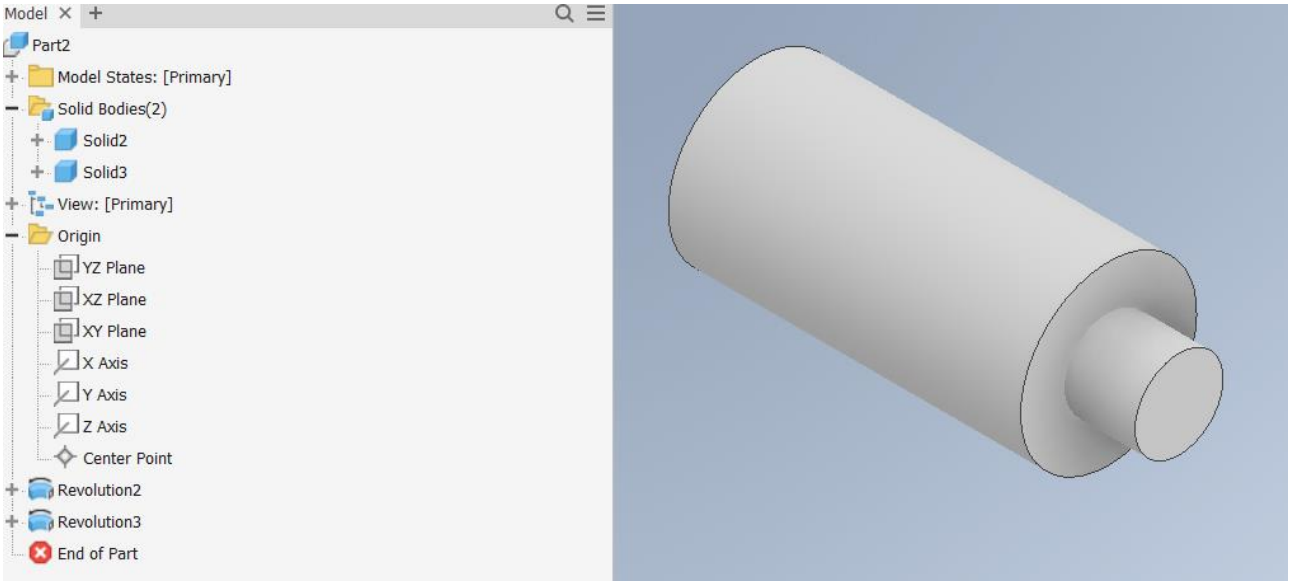
- *Second:*
  - **Cut**. The 2nd. **Extrusion** gets removed from the 1st. **Extrusion**, as a hollow **Solid**:
  - (you usually want to flip it into the first **Extrusion**)



- *Third:*
  - **Intersect**. the 1st. **Extrusion** gets removed and the 2nd. **Extrusion**, becomes its own **Solid**:



- *Last:*
  - **New Solid**. Makes a new **Solid**, then there are 2, one for each **Extrusion**:



  - Then the **Body Name** section appears: Rename your "**Solid**".

Loft

Introduction:

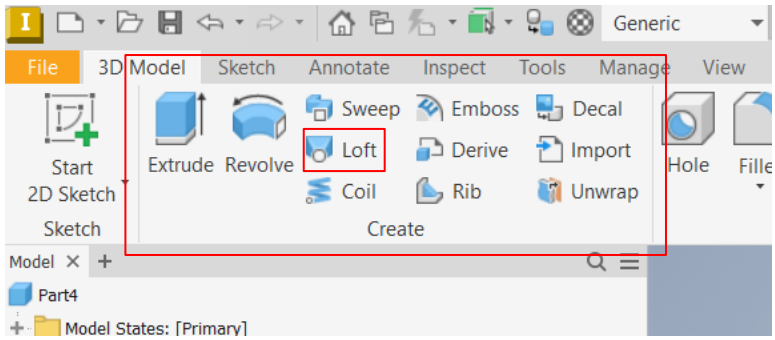
The **Loft** feature creates a transition shape between two **Sketches**. E.g. a pyramid.

There are 3 different methods to make a **Loft**:

- **Rails Loft**.
- **Center line**.
- **Area Loft**.

You have to start your **Sketches** differently per method. **Work Planes** are used here, so find the **Work Planes** document.

In order to use **Loft**, locate **Create** tab at the top of the screen:

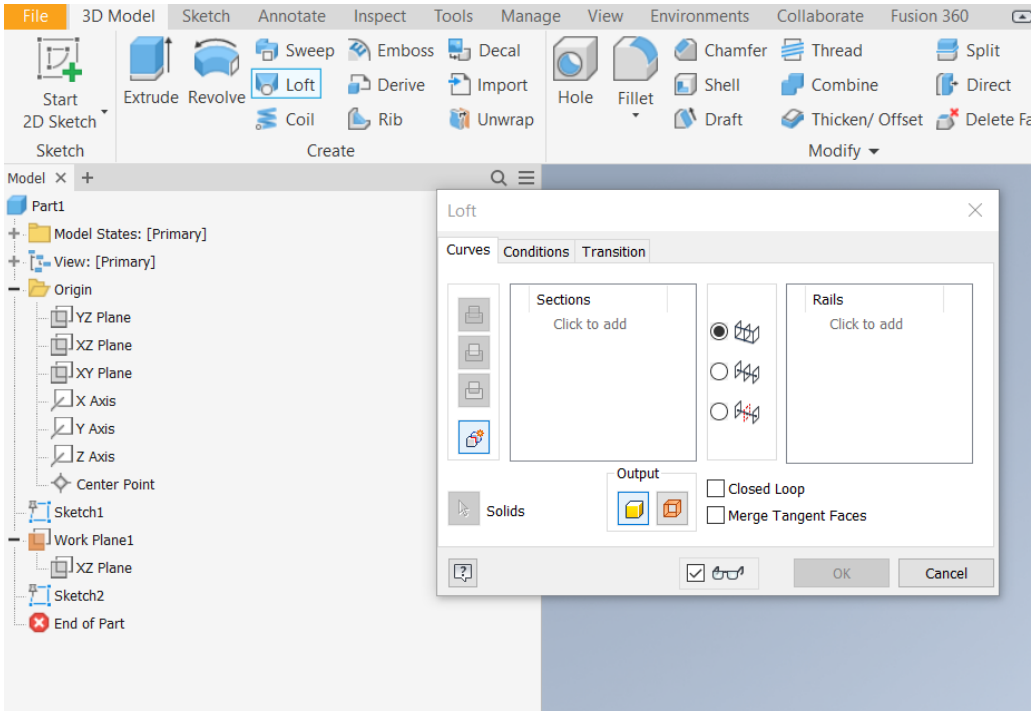


Select **Loft**. When you press **Loft** a box will appear.

- **Shortcut key** = Ctrl + Shift + L

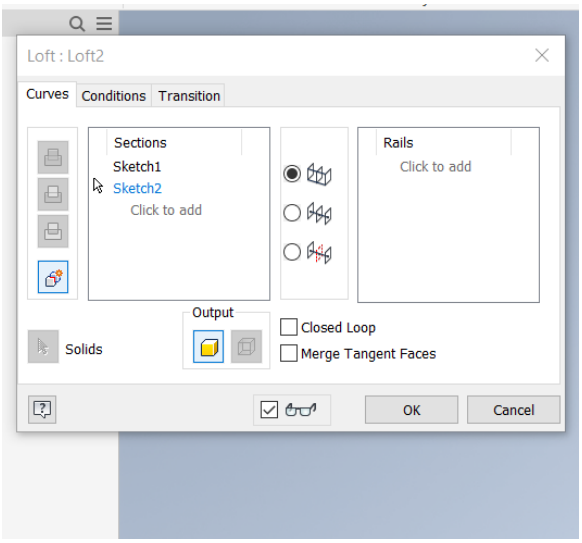
How to use Properties

This is what **Properties** looks like, the first time you press **Loft**:



Curves:

Sections:



Select **Sketches**.

*<u>Note:</u> If you make a mistake, you can in the **Sections** box, **Select** the mistake and delete it by pressing the "**Delete**" key on the keyboard.*

If you press on a **Sketch**, then **remember** that you have not selected the surface. To press the desired surface, first press the **Sketch** and then press the desired surface.

You can also change the order in the **Sections**.

Boolean:

All the way to the left, are all the **Boolean** symbols. They work in the same way as in the other "**Create**" features.

The first time, are all unavailable until you make 2 or more.

Output:

Solid or surface.

Conditions

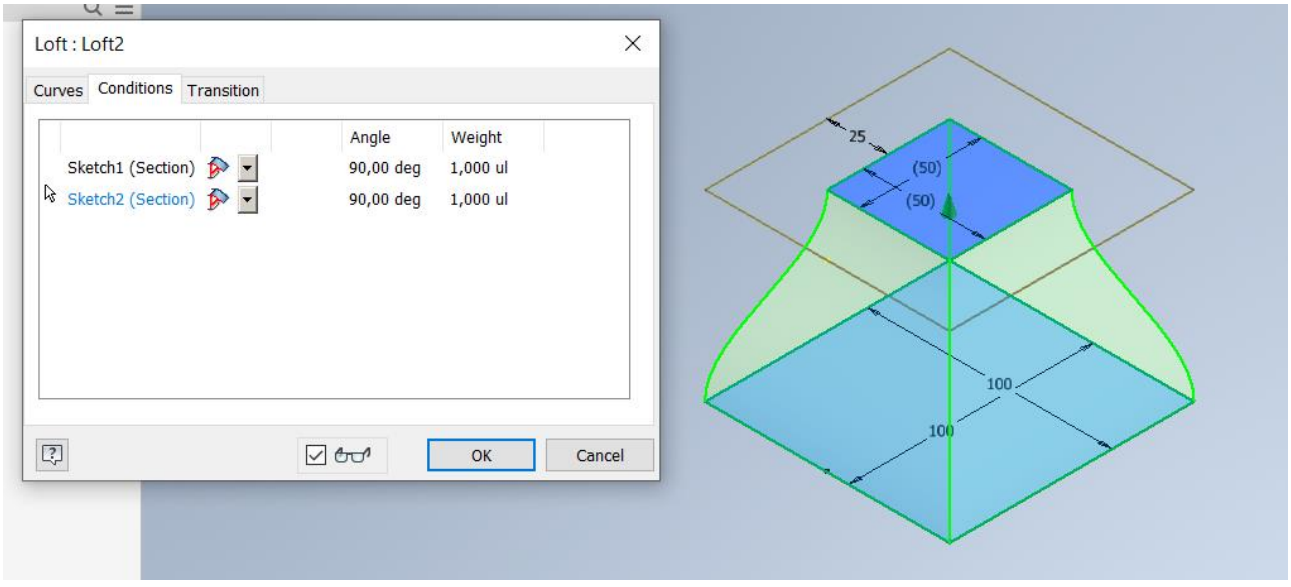
Here is a list of the **Sketches** in use. Next to the name are there a symbol with a down arrow. <u>The symbol is automatically on "**Free Condition**"</u>, The second option is "**Direction Condition**". And here are 2 columns; first is "**Angle**" and second is "**Weight**".

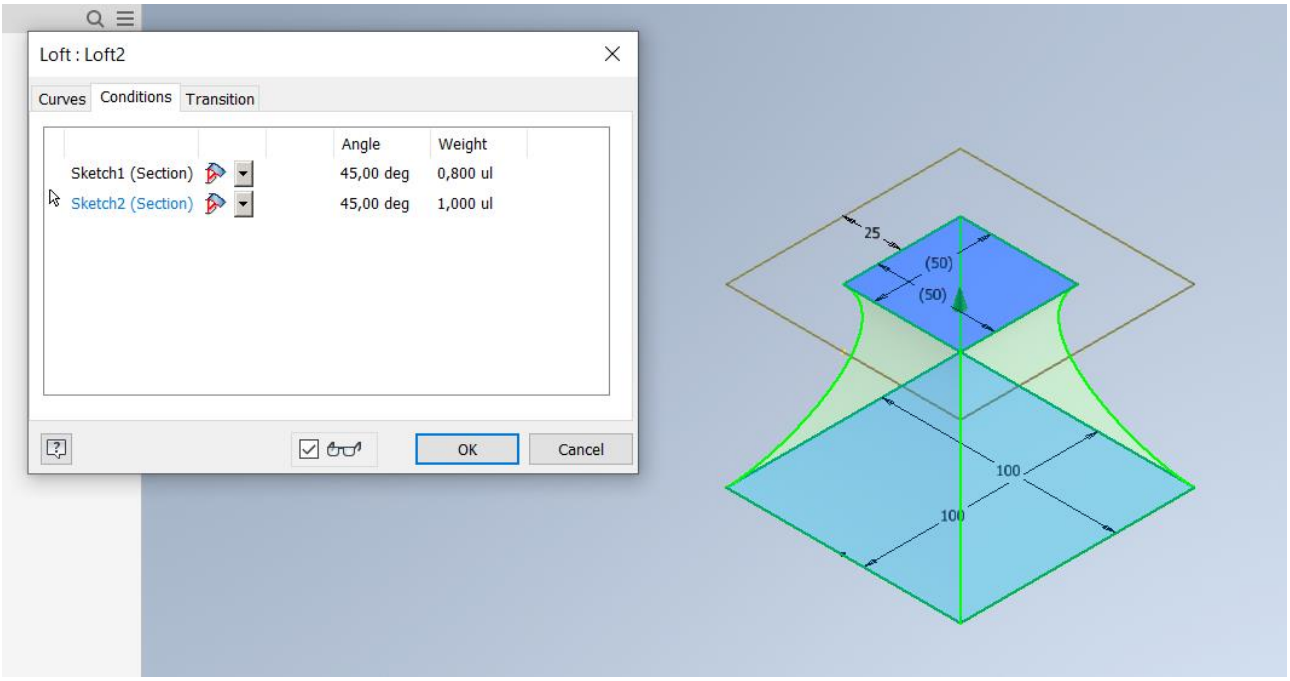
Free Condition

Here "**Angle**" locks to 90°, and "**Weight**" to 0.

Direction Condition

Here "**Angle**" and "**Weight**" will open so you can specify them. "**Angle**" is how much the sides curve. "**Weight**" is how much the curve weighs. This sounds both simple and complicated. If you try it yourself, it's easier to understand.





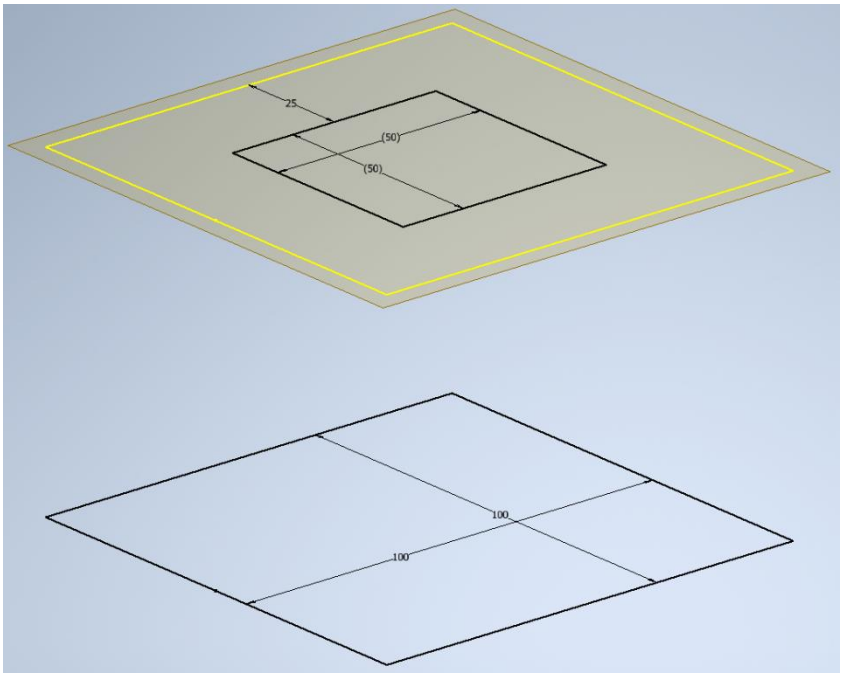
Rails Loft

To make a **Rails Loft**, you must have at least 2 parallel **Planes**, each with its own **Sketch**/shape.

Example:

Here make a 100x100 mm square on the **XZ-Plane**. And then make a **Work Plane** offset from **XZ**. Offset with 75 mm.

On that **Plane**, make a square of 50x50 mm, offset towards the centre. If the "**Offset**" feature doesn't work, use "**Project Geometry**". (Use 25 mm to offset with, it gives a square of 50x50).

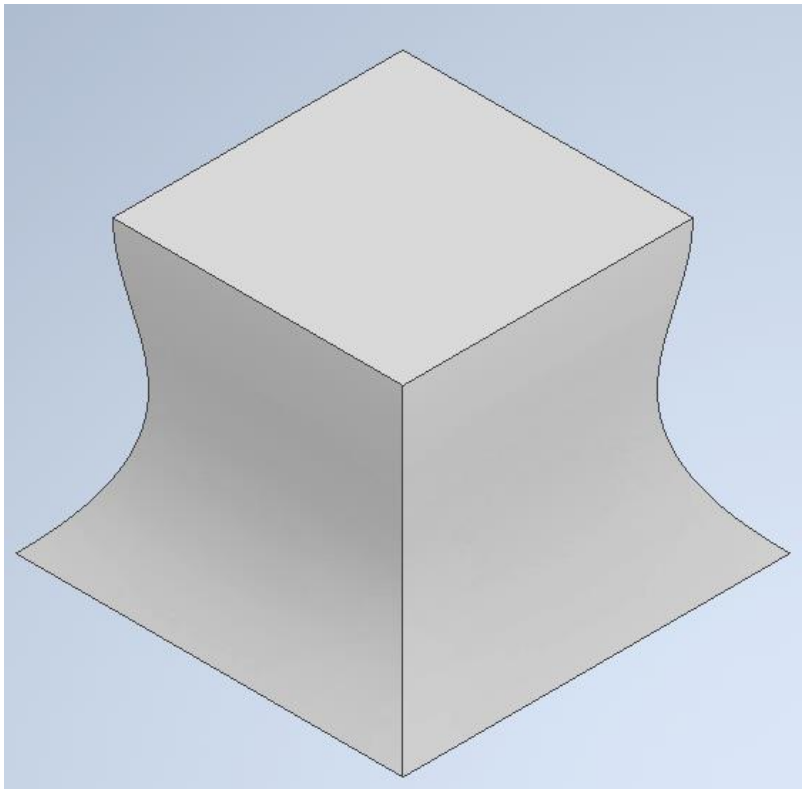


Sections:

In this example, start by selecting the bottom **Sketch**. Then select the top one. When you select the top **Sketch**, Inventor thinks there are 2 squares, since "**Project Geometry**" has been used, here you must first press the **Sketch** and then the square you want to use (the small one).

Conditions:

Now those who followed the example can try out some different **Conditions**, under **Conditions** on page 15, there are some proposals for what you could try. (*Note: When you change the degrees and the weight, Inventor cannot give you a preview of the result. So, remember to press enter for each attempt*).



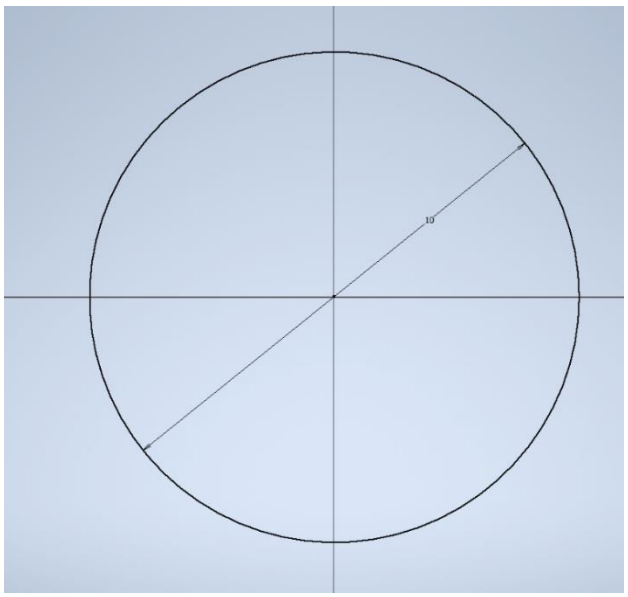
This is the finished result.

Center line 

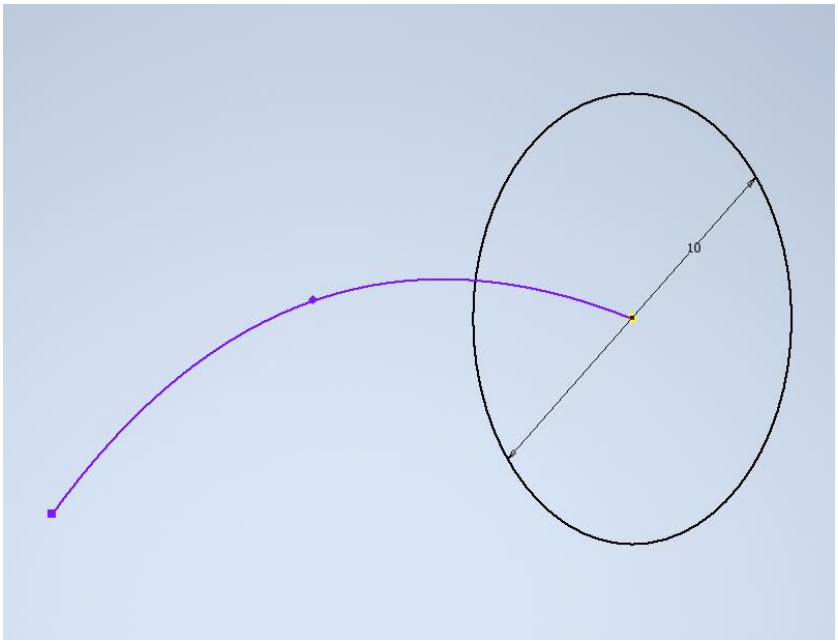
Here you have to make 2 or more **Planes**. Afterwards, make a centreline. It doesn't have to be straight, it can be curved.

Example:

In the example start by making a **Sketch** in the **XY-Plane**. Make a circle or another simple shape. In this example, a circle is used:



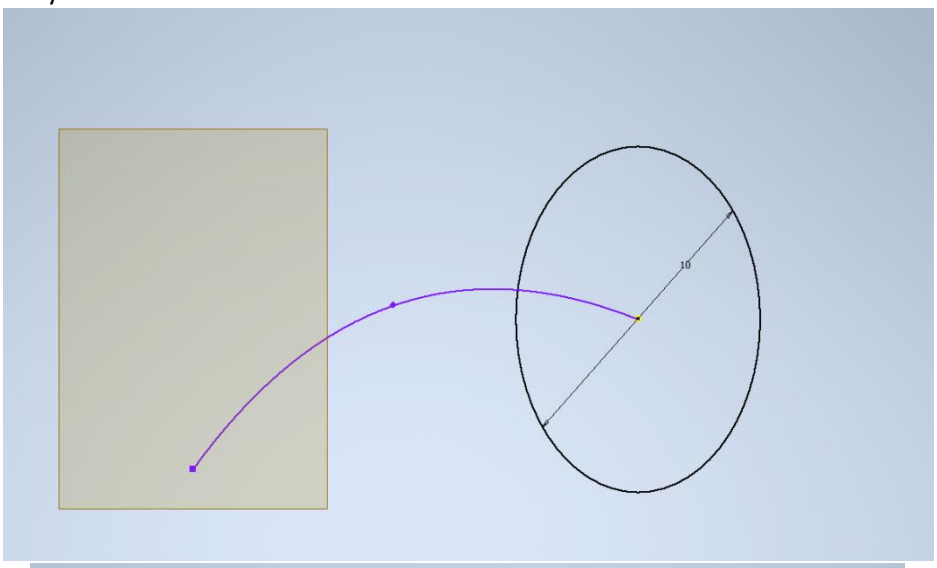
Next step, make a centreline, a curve is best. Start on the **Plane** that cuts through the first **Sketch**. If you started on the **XY-Plane**, it is **YZ**. Make a **Sketch** on that **Plane**.



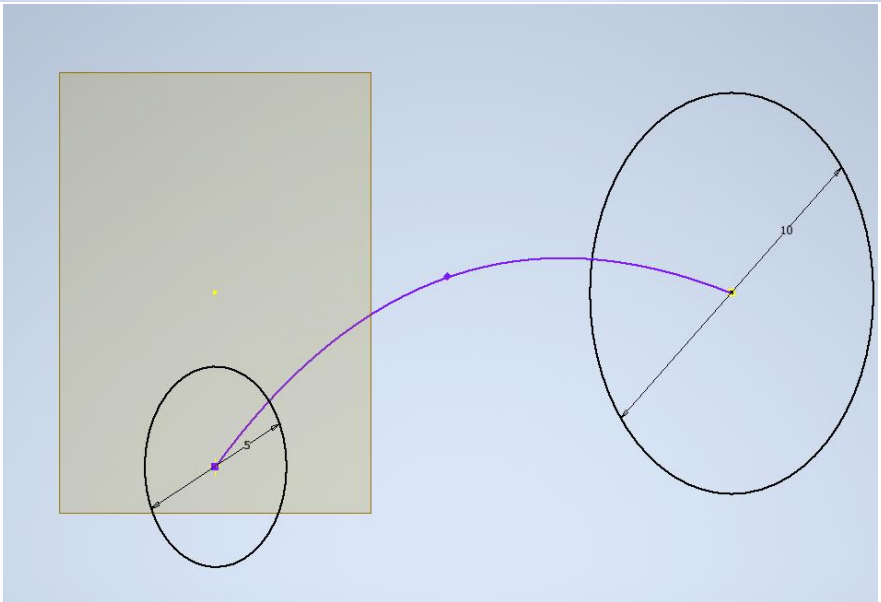
The curve must start from the first **Sketch**, whether you lead the curve to the right or left, is up to you. But in the example the curve leads to the right. To make a curve use "**Interpolation Spline**", left click three times and then click on the green check:

The next step, make a shape that's smaller than the first circle. We need to use a **Plane** parallel to the first (**XY**), it has to end at the curve.

You do this by clicking on "**Work Plane**", then click on the **XY-Plane** in your history and then the last point of your curve.



On the new **Plane**, make a **Sketch** and draw a shape that resembles the first. In the example, a smaller circle is used.



Then use "**Loft**", and click on the 2 shapes, and press the centreline symbol: And then press the centreline you created, and then "**OK**".

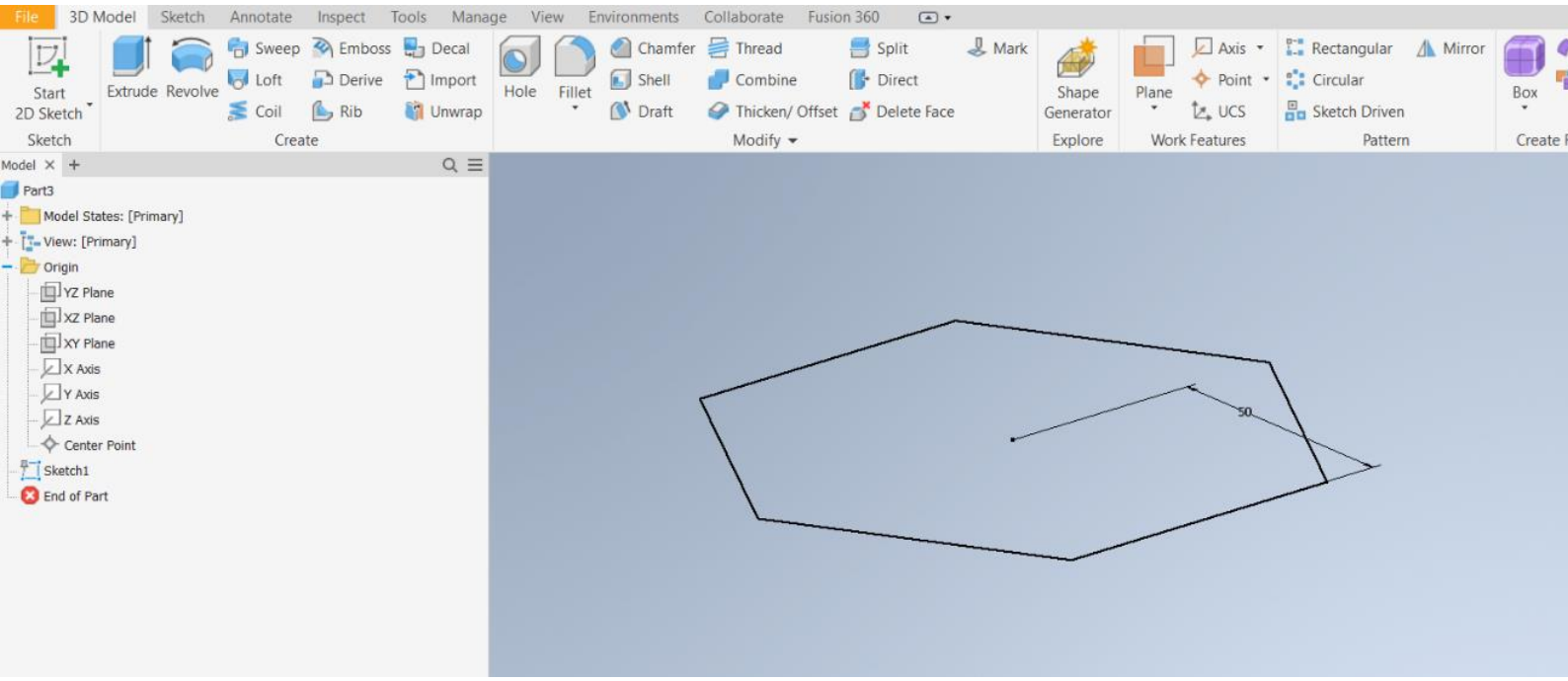
This is the finished result.

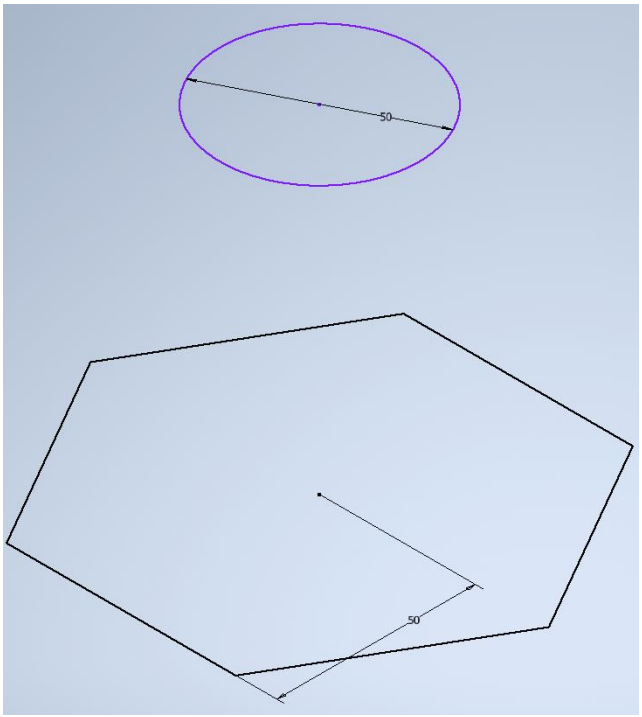
Area Loft

To make an **Area Loft** you need to have at least 2 parallel offset **Planes**, each with its own **Sketch**/shape.

Example:

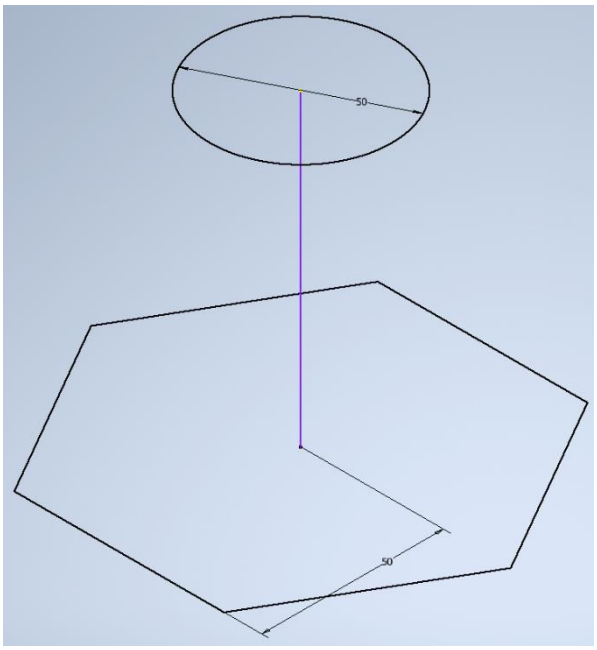
The example starts in the **XZ-Plane**, with a **Sketch**. Make a simple shape. In this example, a hexagon is used:





The next step is to make a centreline, in relation to the example. Create a new **Sketch** on the **XY-Plane** and connect the 2 shapes in the middle.

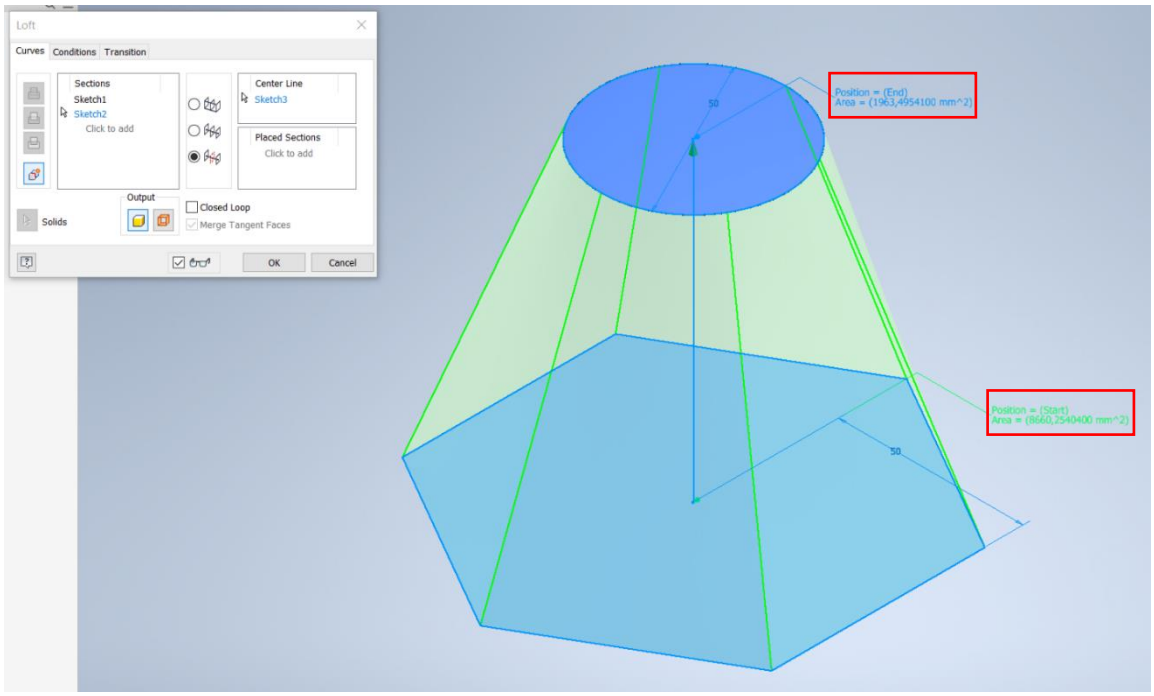
The next step is to make a new **Plane**. Offset from the **XZ-Plane**, then make a **Sketch**. Make a shape that's smaller than the first. In this example, a circle is used.



The next step is to use **Area Loft**. Start by selecting the 2 **Sketches** (***Remember****, the Inventor* ***Loft*** *feature does not like that there are extra lines in the shape, you actually want to use*).

Then select the **Area Loft** symbol: and click the box.
("Click to add") Bellow the text "**Center line**". Then click on the centreline:

The red is an overview of the areal, at the bottom and top of the shape ↓.

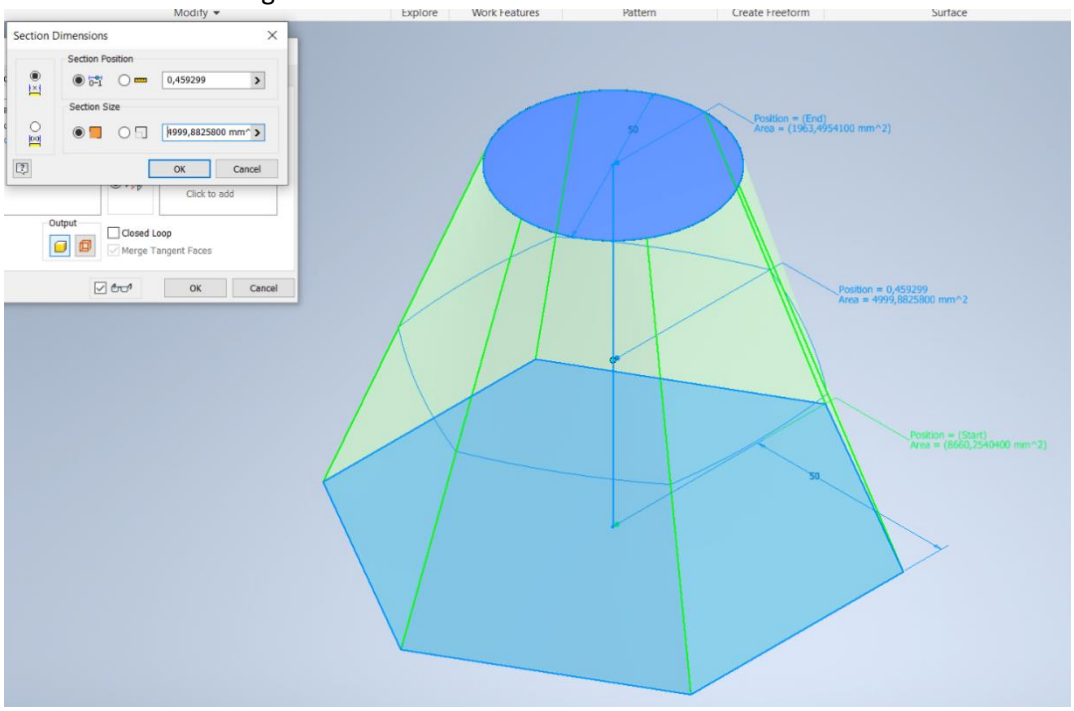


Placed Sections

It is a feature that adds an extra layer that you can manipulate as you wish.

How:

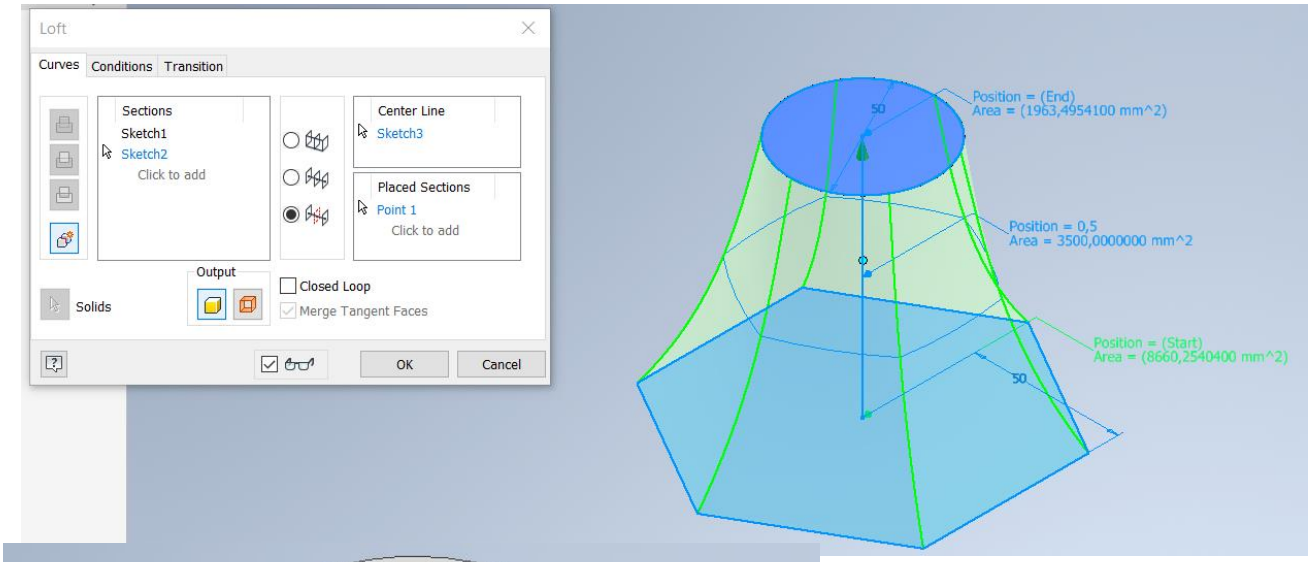
Then click on the ("Click to add") box below the text "**Placed Sections**". Then click on the centreline, it doesn't mean it can't be changed afterwards.

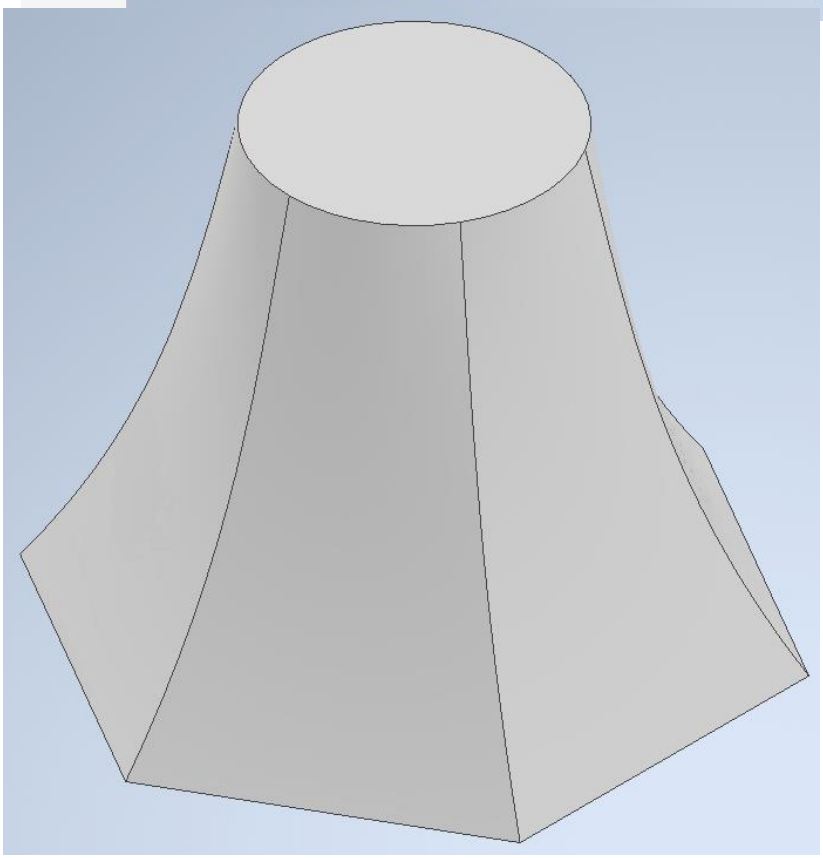


When finished, a box will appear. You can always, when it's set, double-click on the numbers/words for the box to appear again.

- **Section Position**
  - Determines the height of your centreline. It goes from 0 (Start) to 1 (End).
- **Section Size**
  - Here you determine the size of the yield height.

In the example, 0.5 height and 3500 area were selected:





This is the finished result.

Sweep

Introduction:

The **Sweep** feature is used to create a shape that follows the selected **Path**.

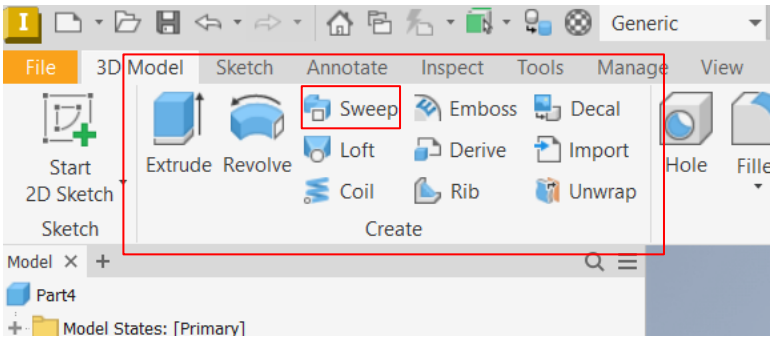
There are different methods to make a **Sweep**:

- **Solid Sweep** off:
    - The default is **Follow Path.**
    - **Fixed.**
    - **Guide.**

- **Solid Sweep** on:
    - **Follow Path.**
    - **Fixed.**
    - **Aligned.**

You start your **Sketches** differently per method.

In order to use **Sweep**, locate **Create** tab at the top of the screen:



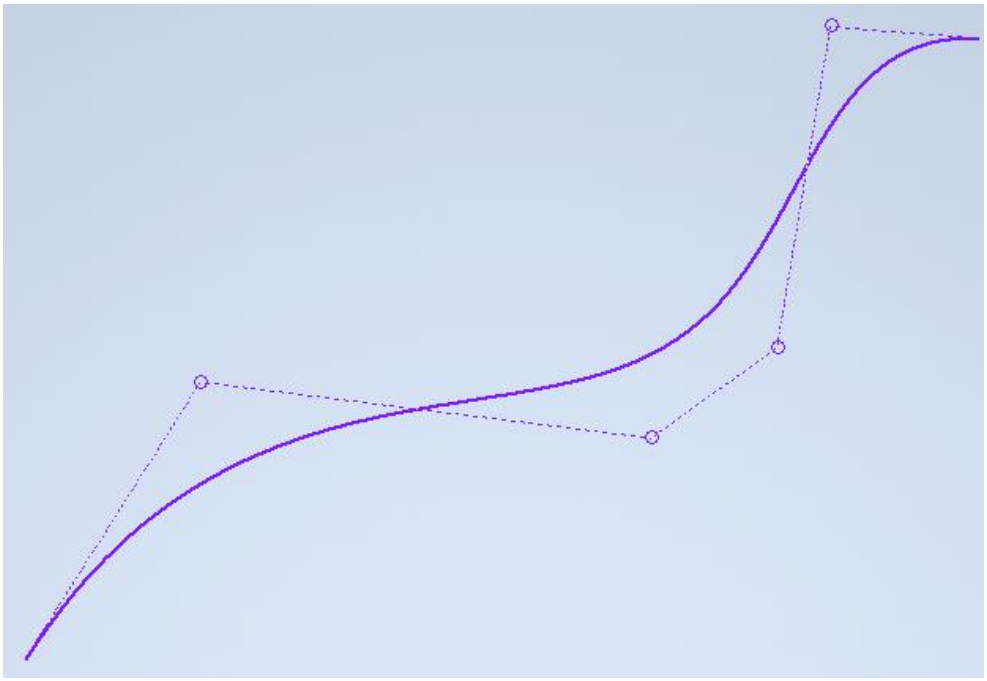
Select **Sweep**. When you press **Sweep** a box will appear.

- **Shortcut key =** Ctrl + Shift + S

Follow Path (Off):

To make a standard **Sweep** with "**Follow Path**", you have to start a **Sketch** in any **Plane**. Then make a **Profile**, (a square is used in the example).

In the first **Sketch** make a **Path**, it doesn't matter what it looks like.



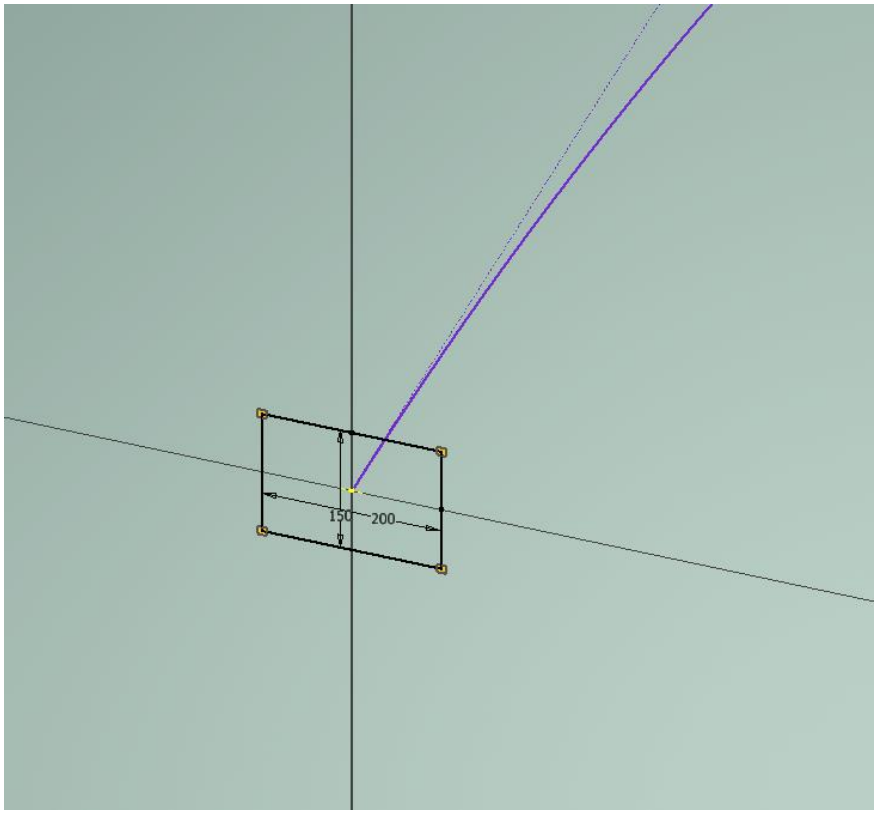
The example.

Then make a new **Sketch**, either on the perpendicular **Plane** or at the end of your **Path**.

For setting a Plane on a line:

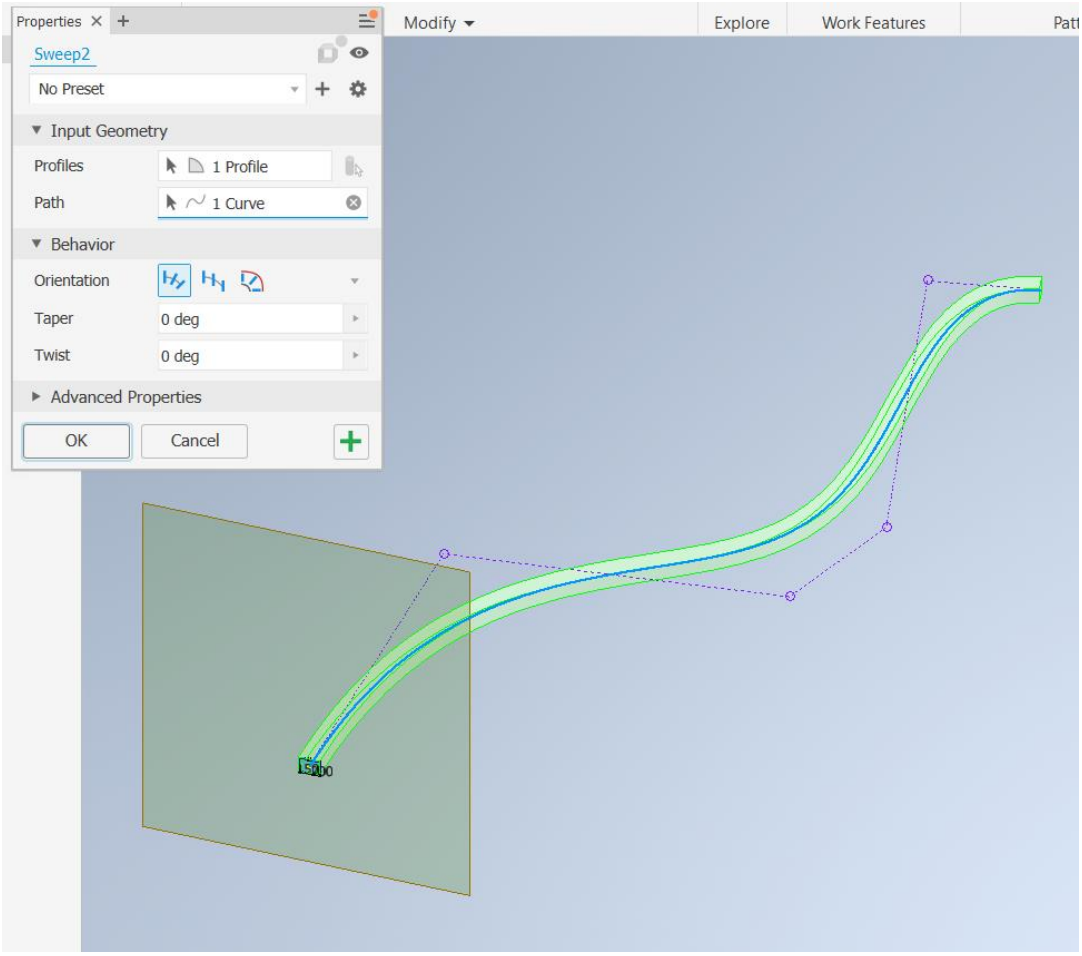
This can be done by first selecting "**Plane**". Then press one end of the **Path** and finally the **Path**, (then your **Sweep** side will stick to your **Path**). Which means that when you change your **Path**, the rest will follow. (Remember that one's **Plane** can be difficult to see, try zooming in).

After you have chosen/created a **Plane**. Then make a **Sketch** on the new **Plane**, create a shape with the center point; **Path**. (Use "**Project Geometry**", to select the end of the **Path**).



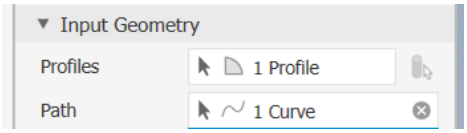
Then click on **Sweep**.

This is what **Properties** looks like, the first time you press **Sweep**:



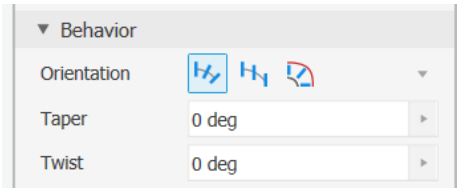
Input Geometry

Works the same way as **Extrude → How to use... (First) → Input Geometry**, here **Path** is where you select a **Path**, in sted of **Profiles**.



Behavior

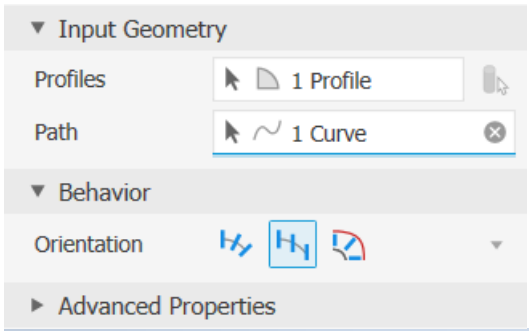
- **Orientation =** The different ways to use **Sweep**. (It's on the **Follow Path**).
- **Taper =** The degree the **Sweep** grows from the **Profile**.
- **Twist =** The degree to which it revolves on itself.

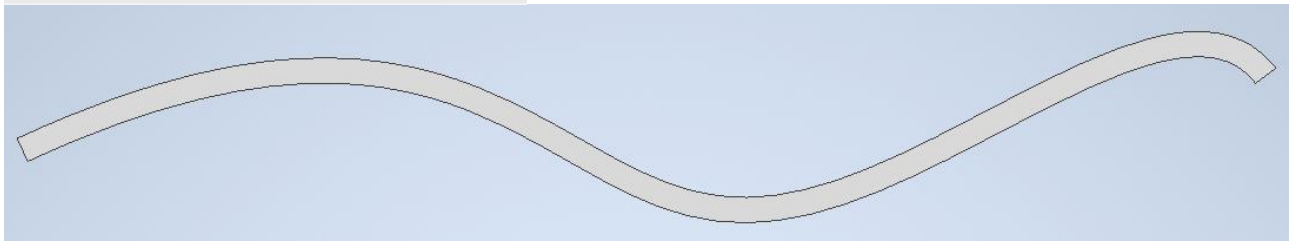


(***Remember*** *Inventor doesn't like your* ***Sweep*** *crossing each other, that means* ***Taper*** *and* ***Twist*** *won't work if the shape crosses itself*).

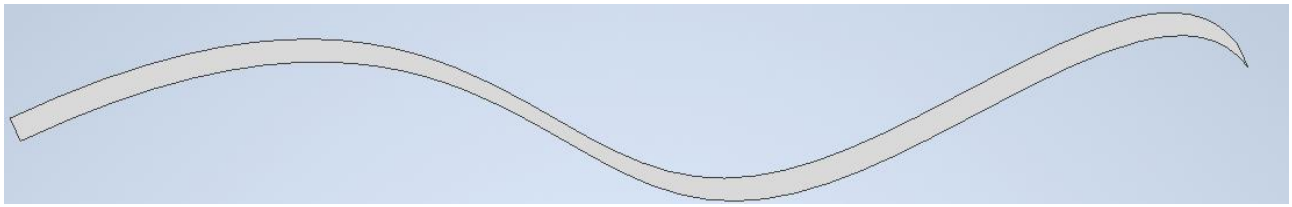
Fixed (Off):

Here it is the second under "**Orientation**". What "**Fixed**" does differently than "**Follow Path**" is that, "**Follow Path**" follows the profile around the **Path**, and "**Fixed**" follows the profile around with a fixed angle. (It makes it look italic).





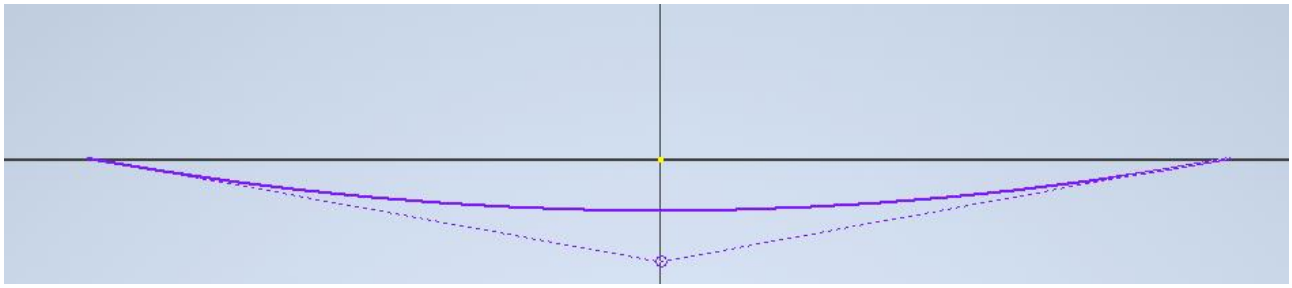
Follow Path↑ **Fixed**↓



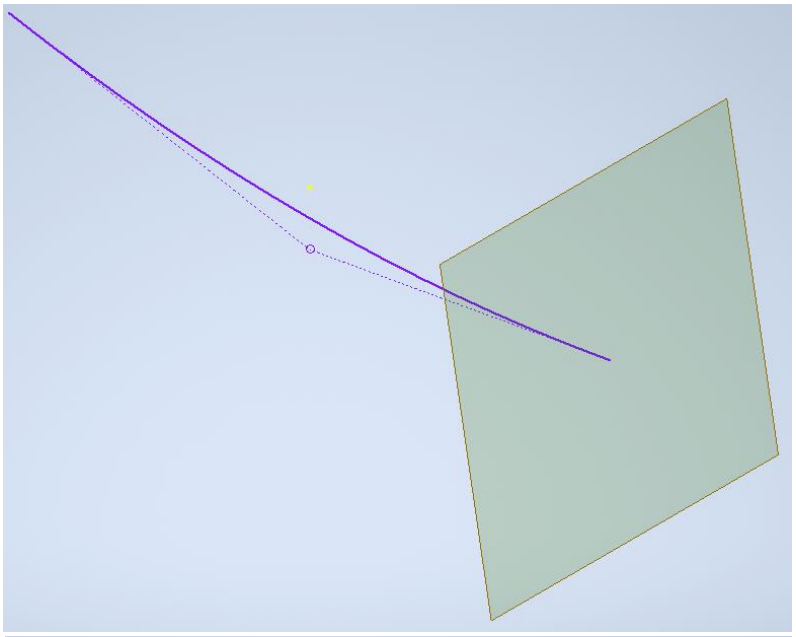
Guide (Off):

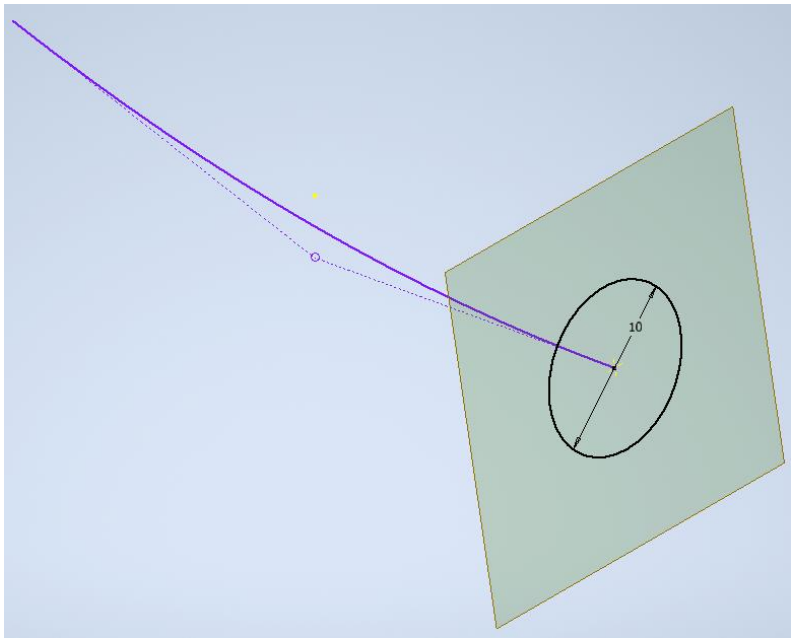
In the **Guide (Off)** you need 3 **Sketches**, 2 of them are **Paths** and a **Profile**. The first **Path**, guides the **Sweeps** shape.

In the example, the **XY-Plane** is used for the 2 **Paths** and a "**Work Plane**" as in "Follow Path". First, make a downward going curve on the **XY-Plane**:



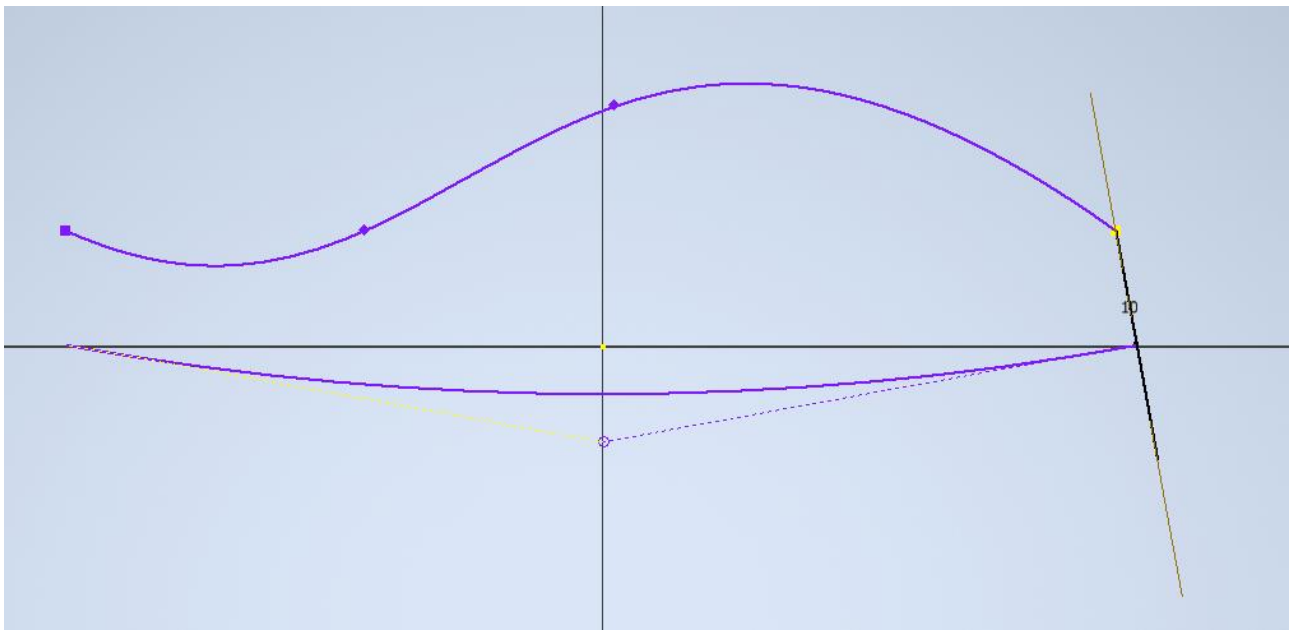
Then create a "**Work Plane**" by clicking on "**Plane**" at the top of the screen, then the end of the curve and finally the curve:



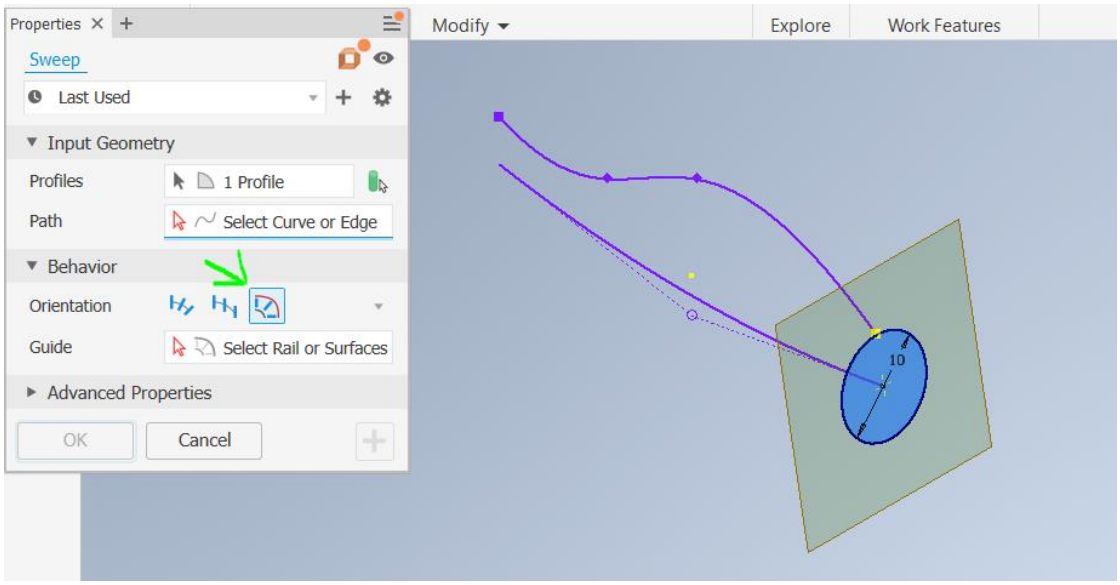


On this **Plane**, create a circle with a diameter of 10 mm (***Remember*** *that the center of the circle must be the curve, this is done with "****Project Geometry****").*

Then make the second path (on the **XY-Plane**). **Remember** to start at the edge of the circle. And that it must not be longer than the first **Path**:



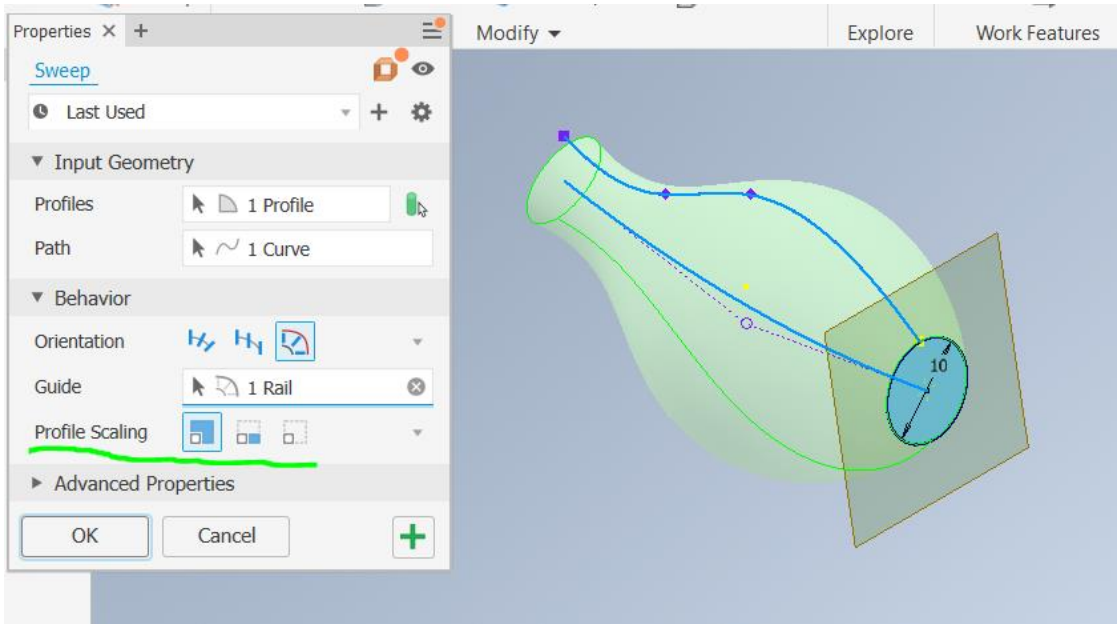
Only then, can you use the **Sweep Guide** feature. Click on **Sweep** and then on the guide icon (by the green arrow).



Then click on "**Select Curve or Edge**" (it's active when the blue line is visible), and then on the bottom curve.

Then click on "**Select Rail or Surfaces**" (when it is active), and then the upper curve.

Afterwards, some settings will appear:



Profile Scaling

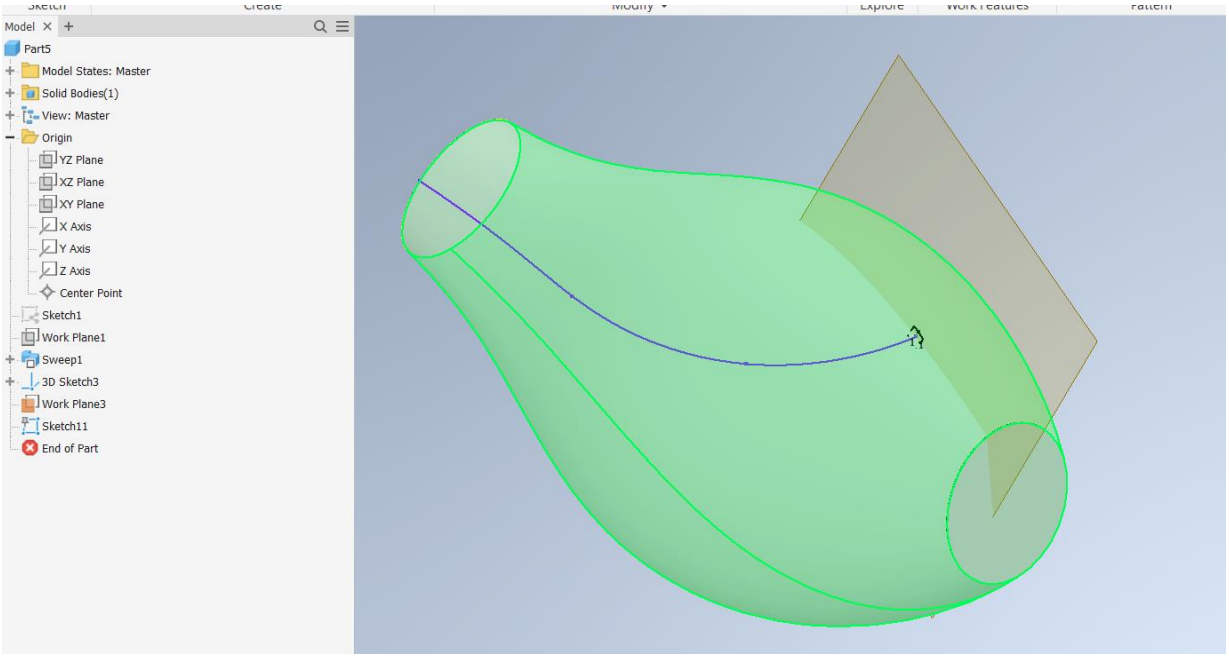
Below here you can select 3 different methods the guide will use, which are "**X & Y Scaling**", "**X Scaling**" and "**No Scaling**".

- **X & Y Scaling** = Both up/down and left/right, fills out.
- **X Scaling** = Only top/bottom filling.
- **No Scaling** = Nothing happens (Doesn't matter since "**Follow Path**" does the same thing).

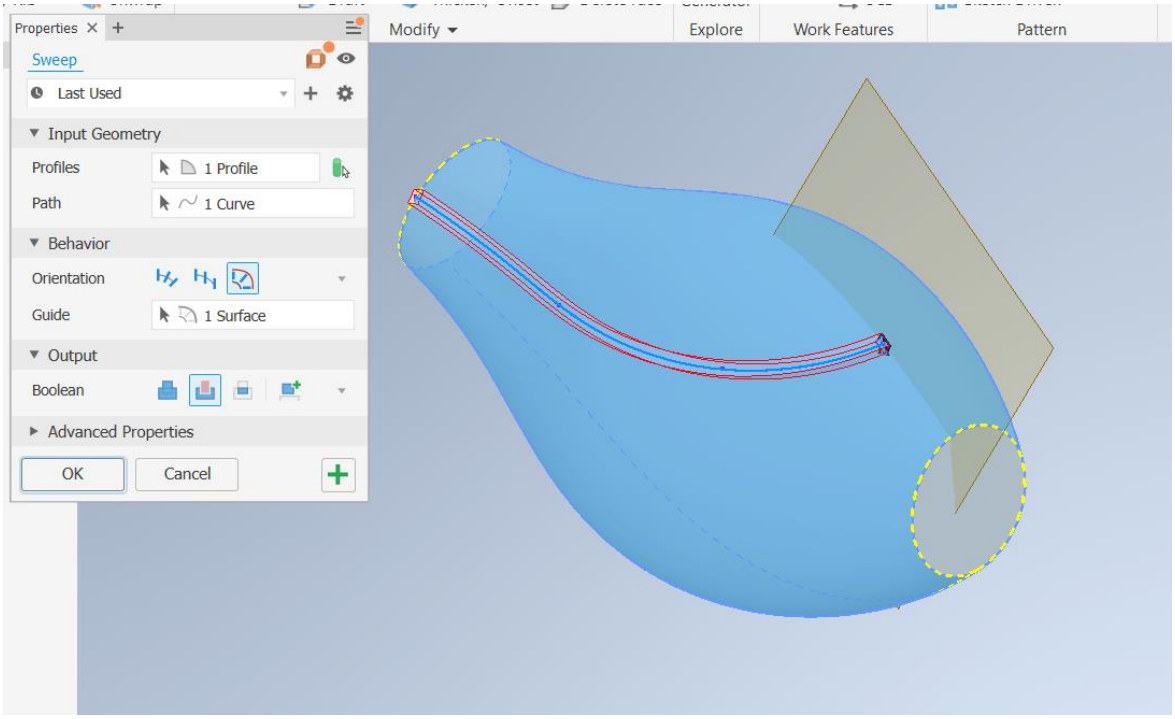
Profile Cut

This is not an Inventor feature with a symbol, but **Part** of what the "**Guide**" feature can do. To use, make a **Path** the **Cut Profile** can follow. The **Path** must be created on the surface of an existing shape. This can be done by using the **3D Sketch** feature.

Locate the feature = "**Curve On Face**", next create a **Path** for the **Cut** to follow. Then make a **Plane** at the end of the new curve, on that **Plane** make a **Cut Profile**:



Use the **Sweep** feature to make a **Cut**. Select the curve. Then select "**Surface**"; this is the shape. (***Remember to activate Guide***). Afterwards, the **Boolean** must be on **Cut** to remove.



Boolean:

Like the other "**Create**" features, "**Boolean**" is the same 4 symbols:

- **Join**.
- **Cut**.
- **Intersect**.
- **New Solid**.

Which is only activated the second time the feature is used.

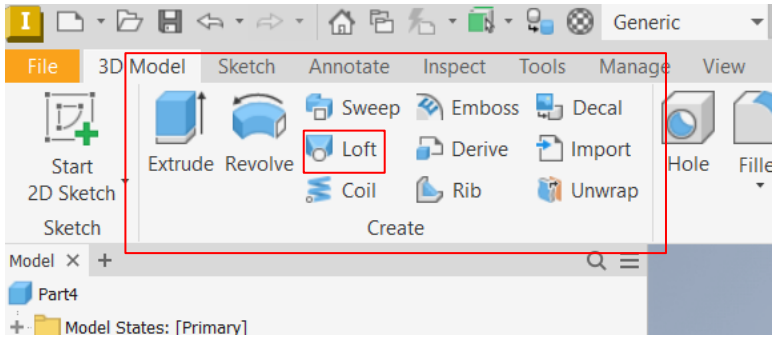
Coil

Introduction:

The **Coil** feature is used to make a shape around another, for example a drill head.

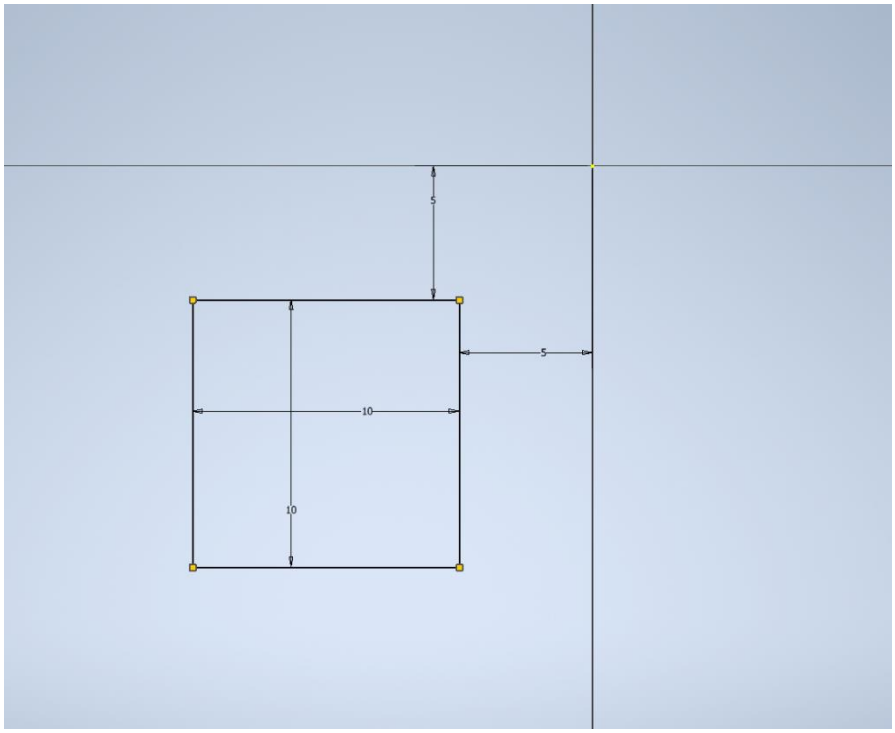
Before you can use **Coil**, a **Sketch** must exist. It should ideally be offset from the center as we use the standard axes in Inventor.

In order to use **Coil**, locate **Create** tab at the top of the screen:



Select **Coil**. When you press **Coil** a box will appear.

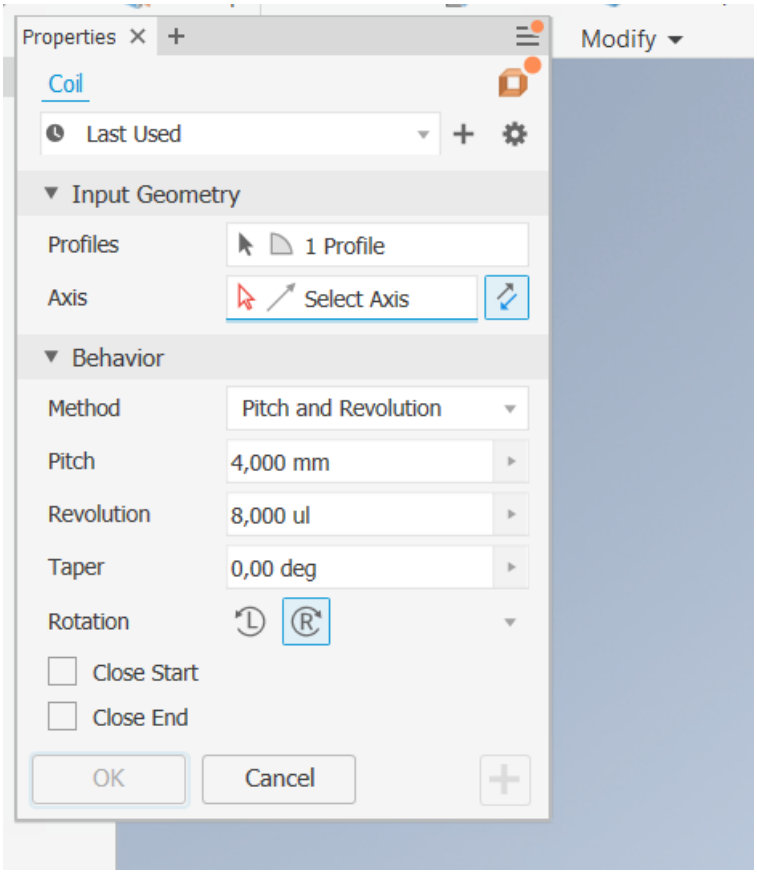
In the example, a small square is used as a **Sketch**, offset from the center:



When the **Sketch** is done, then use "**Coil**".

How to use Properties

This is what **Properties** looks like, the first time you press **Coil**:



Input Geometry

Profiles:

Profile is rotated around the axis. In the box, Inventor informs about the amount of **Profiles** (this is a surface/**Sketch**) that are selected.

Axis:

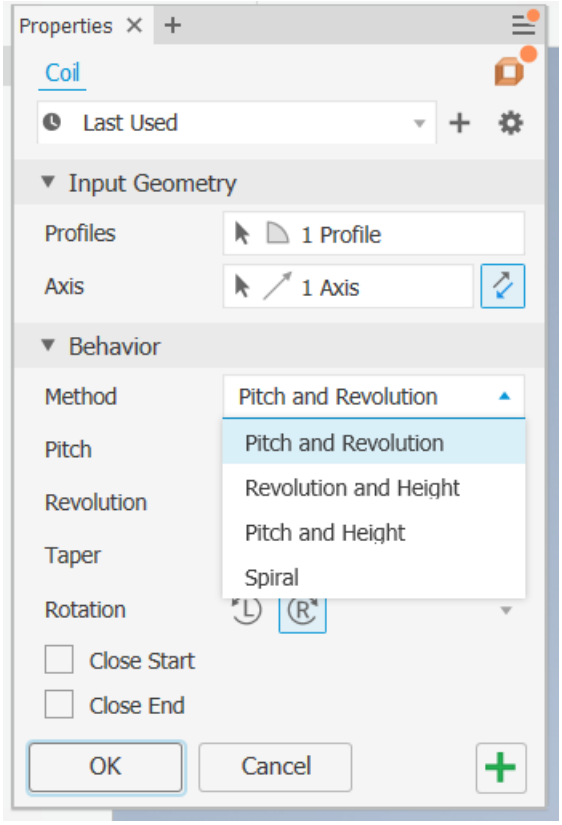
Axis is where the **Coil** feature rotates around.

Axis is what the geometry rotates around. Is the mouse icon red, that means no axis has been selected. The blue line below the word "**Select Axis**", shows that "**Select Axis**" is active. If the blue line is not there, you cannot click on the desired axis. You can make it active by clicking on "**Select Axis**". But Inventor auto activates "**Select Axis**".

In the example we select the **Y-axis** (You locate it in your history under **Origin**), we select the **Axis** that is perpendicular to the **XZ-Plane** (It is the letter that was not in your **Plane**).

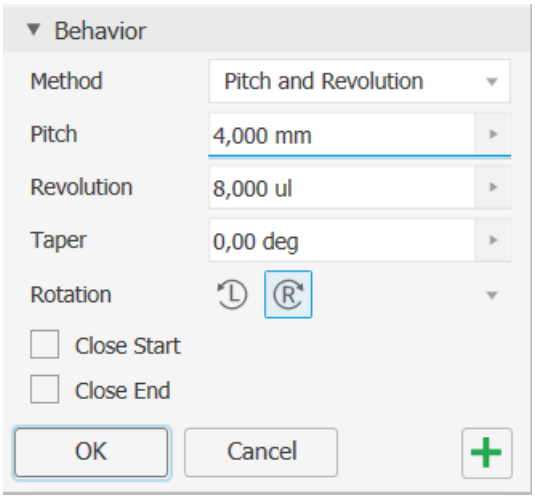
Behavior

In **Behavior** there are 4 different methods to make a "**Coil**", You can change the method by clicking on "**Method**" (Box on the right).



← Flips the way it **Coils** (Up or down).

Pitch and Revolution



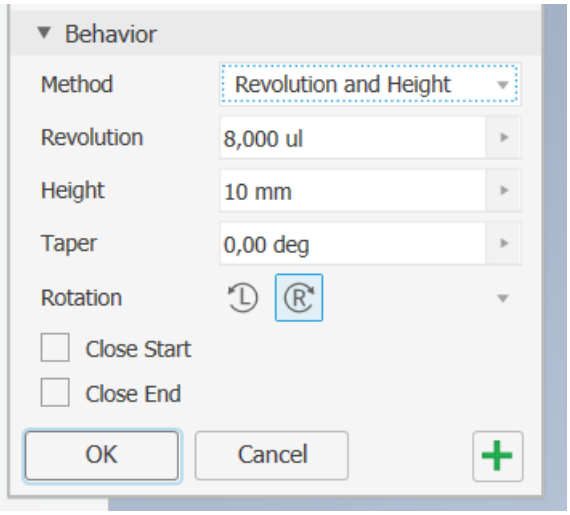
Pitch = The height between the **Coils**.

Revolution = How many times it **Coils**.

Taper = The degree the **Coil** grows, from the **Profile**.

Rotation = Which way it rotates (default clockwise).

Revolution and Height



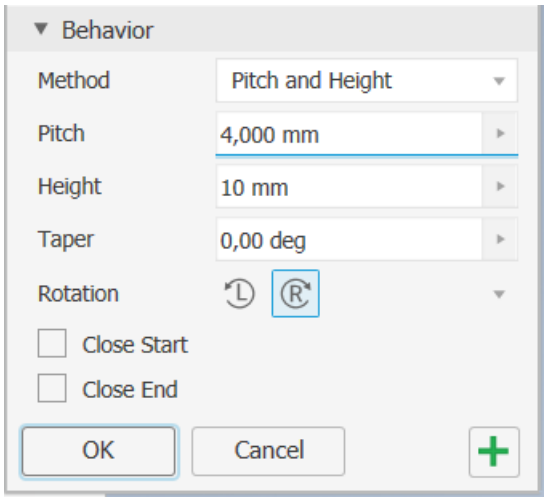
Revolution = How many times it **Coils**.

Height = How far from **Sketch Plane** to the end **Coil**.

Taper = The degree the **Coil** grows, from the **Profile**.

Rotation = Which way it rotates (default clockwise).

Pitch and Height



Height = How far from **Sketch Plane** to the end **Coil**.

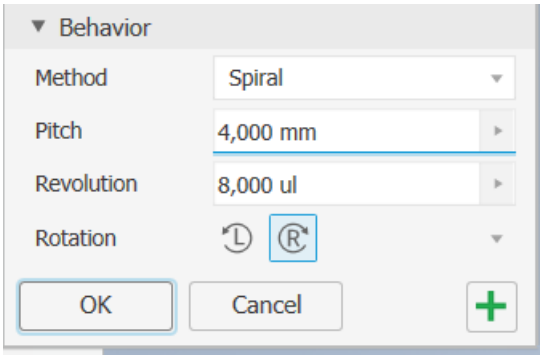
Pitch = The height between **Coils**.

Taper = The degree the **Coil** grows, from the **Profile**.

Rotation = Which way it rotates (default clockwise).

Spiral

Is different as it is flat and does not have a height.



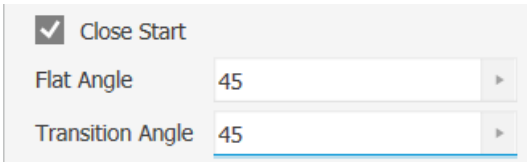
Pitch = The height between **Coils**.

Revolution = How many times it **Coils**.

Rotation = Which way it rotates (default clockwise).

Close Start/End

Closes the ends, gives 2 settings:



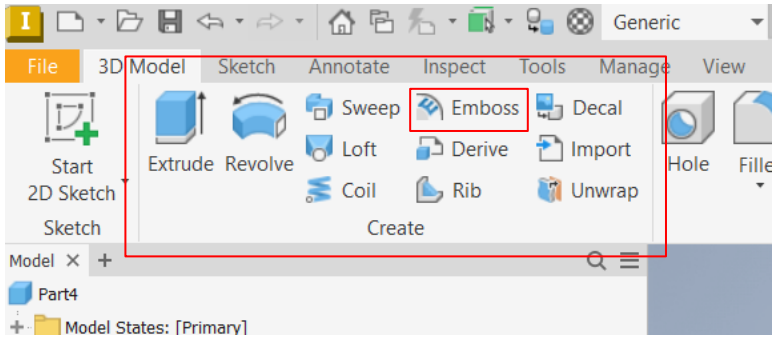
Decides which degrees the ends end in.

Emboss

Introduction:

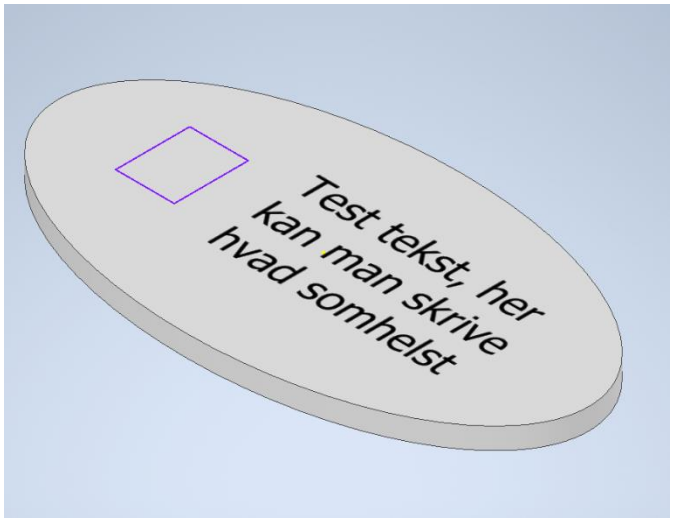
The **Emboss** feature is used to create engravings, in other words, raises or lowers a profile.
Before you can use **Emboss**, a **Sketch** and a **Solid** must exist.

In order to use **Emboss**, locate **Create** tab at the top of the screen:



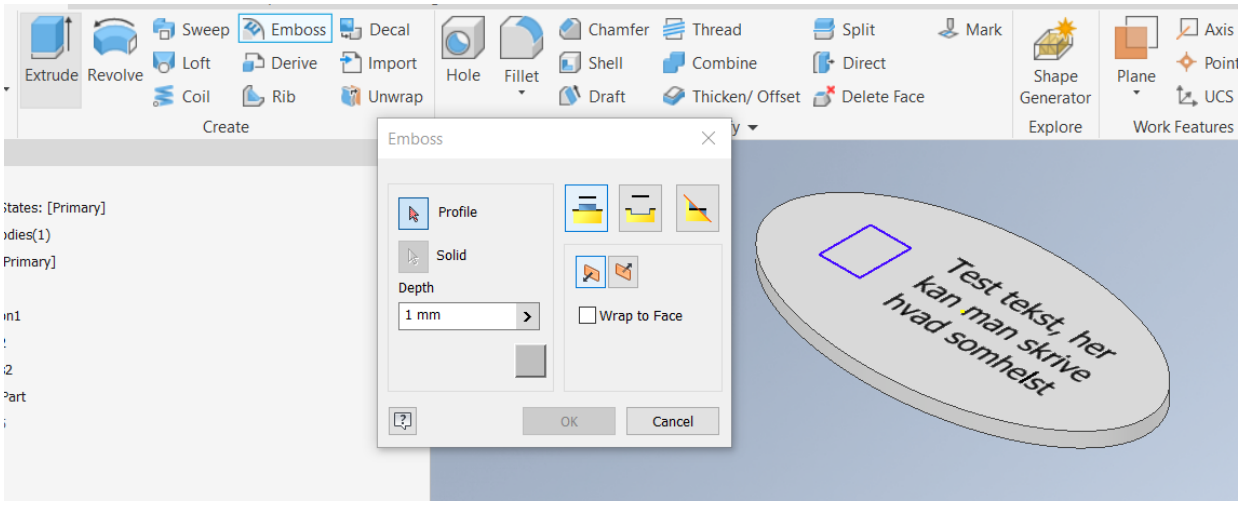
Select **Emboss**. When you press **Emboss** a box will appear.

An example of how you can use **Emboss**:



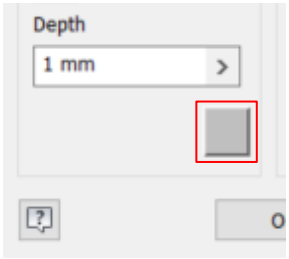
How to use Properties

This is what **Properties** looks like, the first time you press **Emboss**:

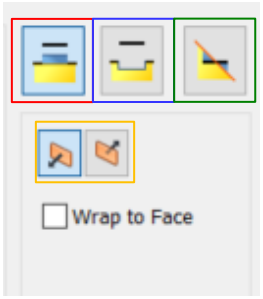


Profile = Select the **Profiles** (Note: Text counts as a profile).

Depth = How deep the **Emboss** is.



You can click on the square to change the appearance of the **Emboss**.

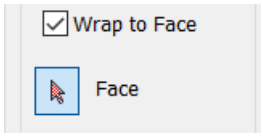


Red = Works like "**Join**" (pulls the **Profile** out of **Solid**).

Blue = Works like "**Cut**" (cuts the **Profile** from **Solid**).

Green = Doesn't work.

Yellow = Flips.



Checks "**Wrap to Face**" on, remember to select a **Face**.

Face = Here select **Faces** (aka **Surfaces**).

Rib

Introduction:

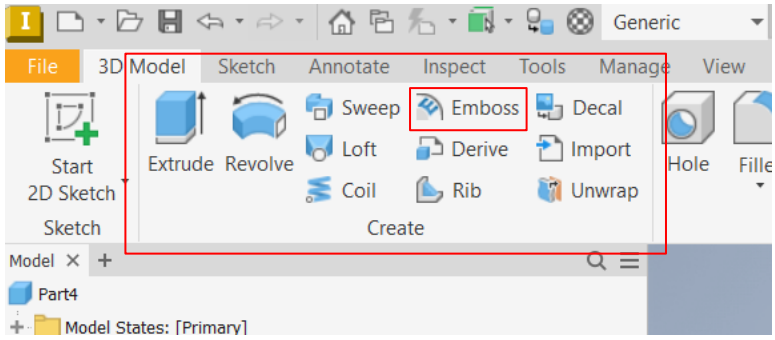
The **Rib** feature is used to make thin-walled supports.

Before you can use **Rib**, there must exist 2 **Solids** that meet each other at an angle, and there must be a **Sketch** that shapes your **Rib**.

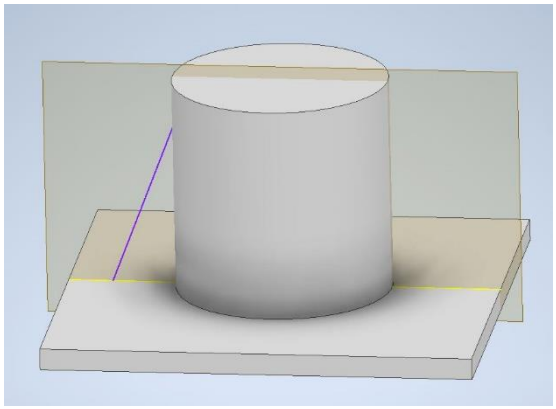
There are 2 different ways to make a **Rib**:

- **Normal to Sketch Plane**  .
- **Parallel to Sketch Plane**  .

In order to use **Rib**, locate **Create** tab at the top of the screen:



Select **Rib**. When you press **Rib** a box will appear.

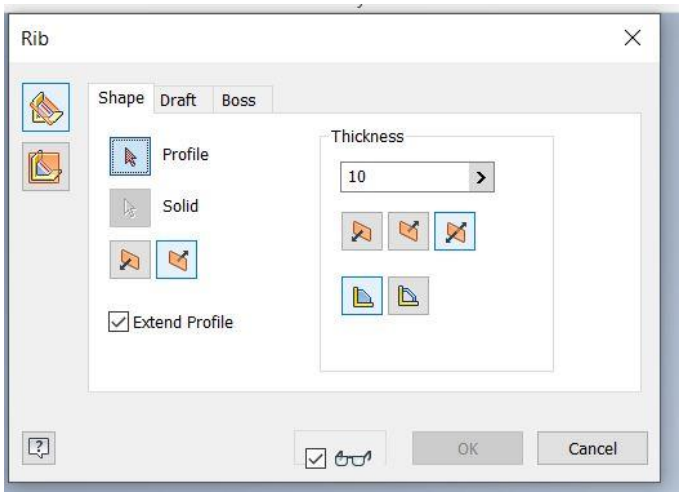


An example of how to use **Rib**:

This example uses two shapes: a cylinder and a square. Connect the 2 shapes with a line at an angle.

How to use Properties

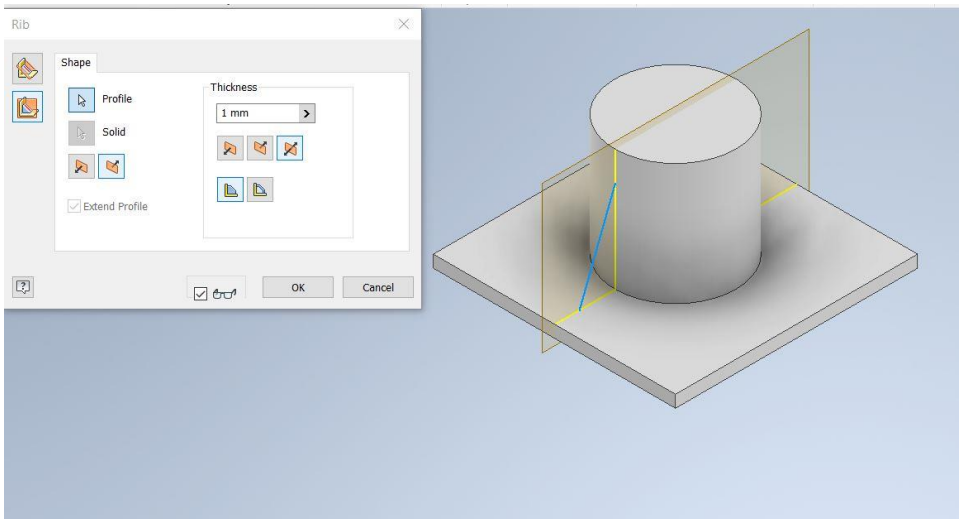
This is what **Properties** looks like, the first time you press **Rib**:



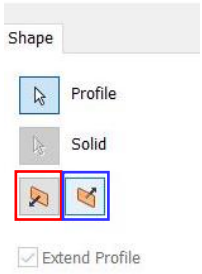
Parallel to Sketch Plane

This is the second icon; we are starting with this feature as it is the easiest method to use.

Start by clicking on: , this will open "**Parallel to Sketch Plane**":



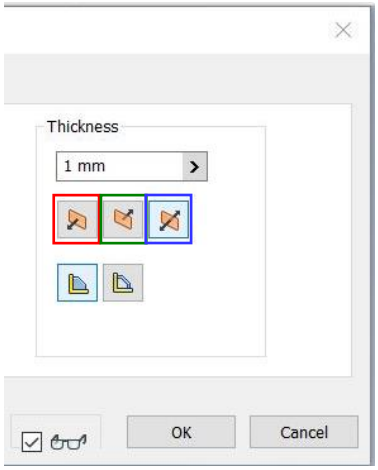
This is how it looks from the start ↑.



Profile = Select the **Profiles** (Note: Can also be a line instead of surfaces).

Red = Extrudes forward. **NOTE**: This feature is automatically switched off and the second has difficulty working. ***Remember*** *to turn on.*

Blue = Extrudes back. **Ignore**, works badly.



Thickness:

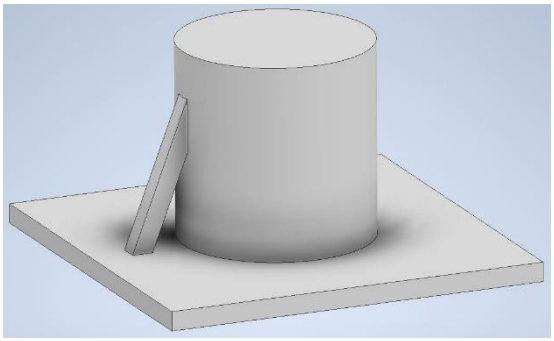
Box = Specify thickness, width.

Red = Extrudes to the right.

Blue = Extrudes to the left.

Blue = Extrudes both ways.

= Fills out the corner.

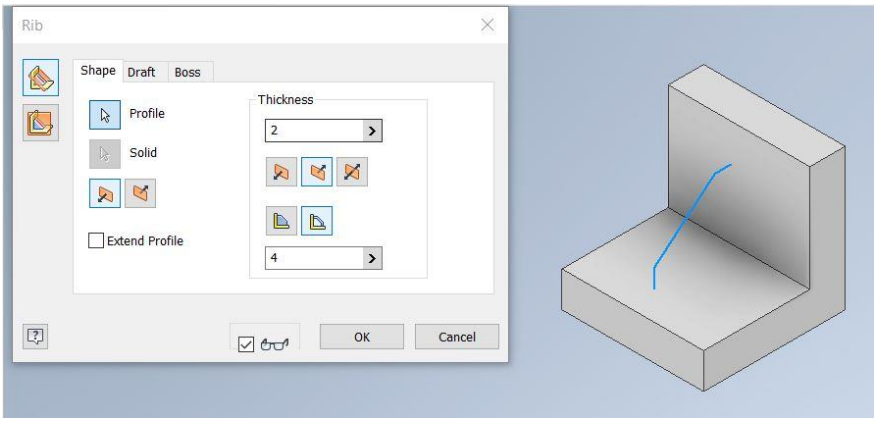


← = Does not fill out the corner. (Box = thickness).

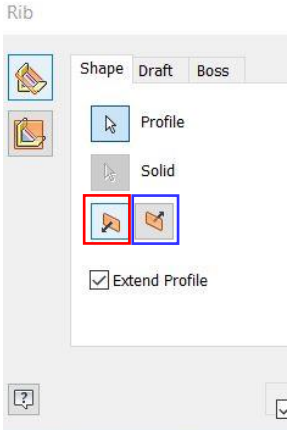
Normal to Sketch Plane

Here, the function is based on the **Sketch** and not parallel. i.e. the feature is based on the line.

Start by clicking on: , to open "**Normal to Sketch Plane**":



This is how it looks from the start ↑.

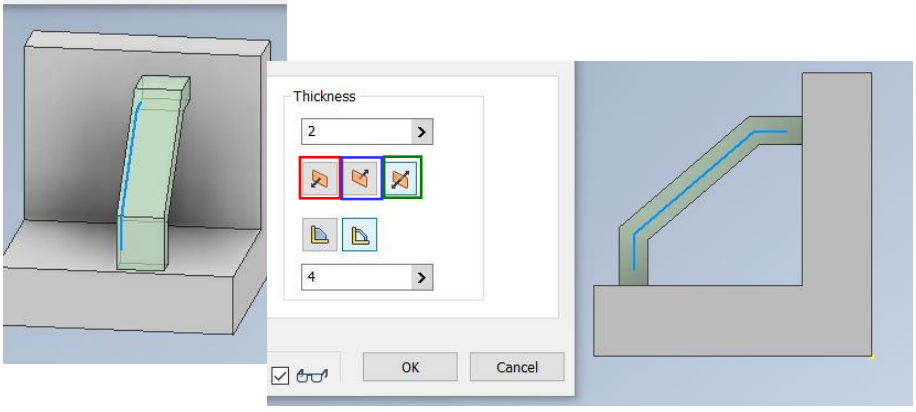


Profile = Select the **Profiles** (***Note****: Can also be a line instead of surfaces*).

Red = Extrudes to the right.

Blue = Extrudes to the left.

Note: When using "**Normal to Sketch Plane**" you cannot center it from the line/**Sketch**, therefore it is not normally used.



Thickness:

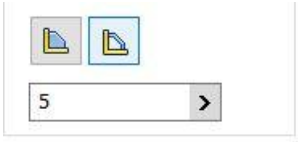
Box = Thickness from **Sketch**.

Red = Extrudes into the corner.

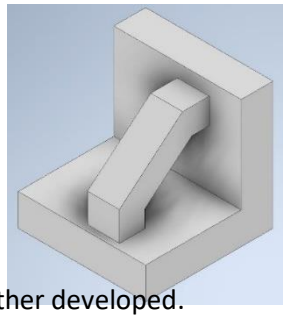
Blue = Extrudes away from the corner.

Green = Extrudes both ways.

= *Doesn't work here.*



← = Does not fill out the corner. (**Box** = thickness, width).



Derive

Introduction:

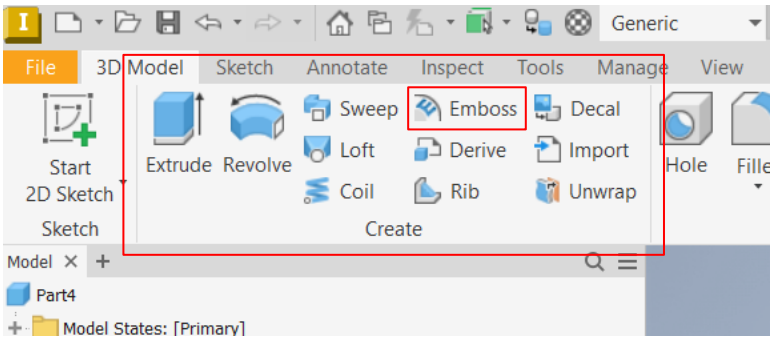
The **Derive** function is used to retrieve a **Part** that needs to be changed / further developed.

1. E.g. to build a box on the outside of the existing **Part**.

2. E.g. to have an original product and remove some material, to save money.

Before you can use **Derive**, you must have an original **Part** and start from it.

In order to use **Derive**, locate **Create** tab at the top of the screen:



Select **Derive**. When you press **Derive** a box will appear.

In the guide, we start with **Properties**, and then 2 assignments. That's the easiest way to explain **Derive**.

How to use Properties

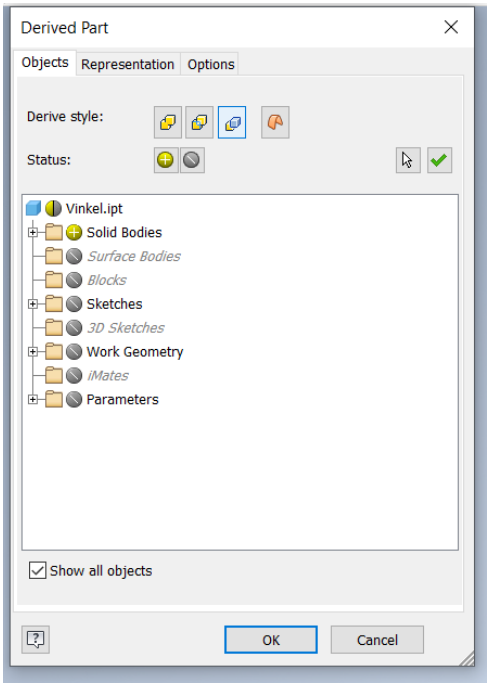
How to access **Derive Properties**:

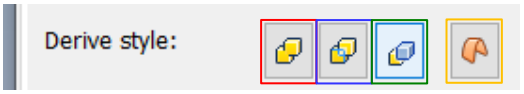
- Start by clicking on **Derive**.
- Then a **Pathfinder** emerges; here you select the desired **Part**.

When you have done that ↑ **Properties** will appear and look like this:

Objects

Derive style:



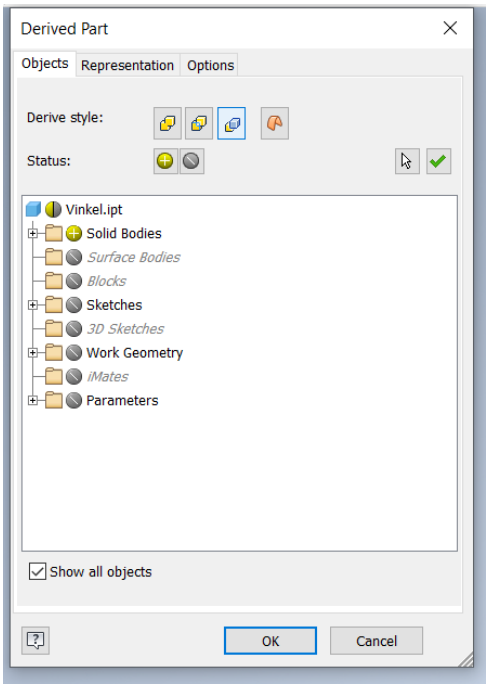


Red = It combines all the **Part's Solids** into 1 **Solid** (without stitching).

Blue = Retains the stitching between the **Parts Solids**.

Green = Retains all **Solids** separate (more than one **Solid**).

Yellow = Turns the **Parts Solids** into "**Work Surfaces**".



In the box with the work tree menu, you can select which components should be included in the new **Part** file.

There are 2 ways to remove and add from the menu.

1. Click on ' ' next to the thing you want to change.
2. Click on the name of the thing you want to change, then go under **Status** and click on the desired symbol.

 = Turned on, select which objects are used on the 3D model.

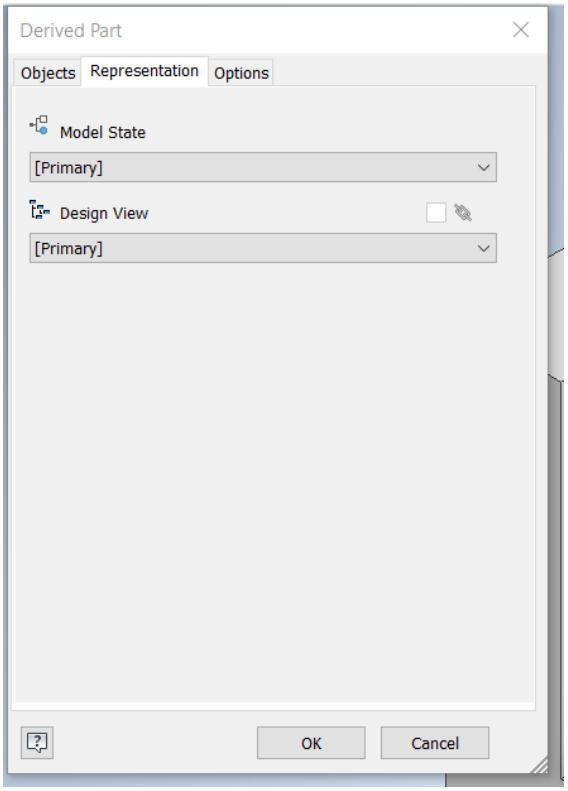
 = Activates ↑.

Show all objects = Turns on/off, that the selected objects can be seen.

Note

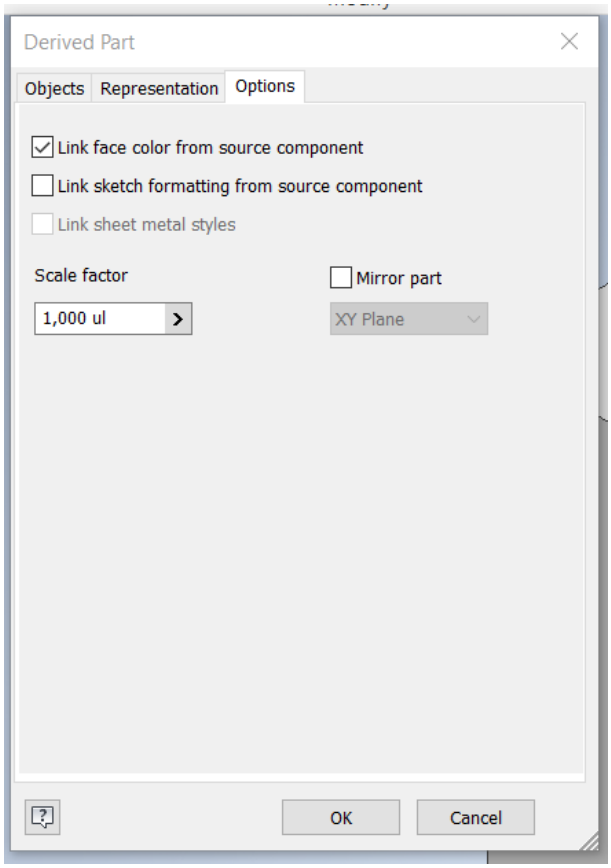
The next settings are located differently if you are using Inventor version 2021 or younger. This guide is based on the 2023 version.

Representation



Here you can change which "**View**" is used. It is recommended not to change them from the default **[Primary]**.

Options



Link face color from source component =
Links the original **Part** and the **Derived Part** color.

Link sketch formatting from source component =
Links the original **Part** and the **Derived Parts Sketch** formatting.

Link sheet metal styles =
Links the original **Part**, and the **Derived Parts Sheet Metal** style, only works on a **Sheet Metal Part**.

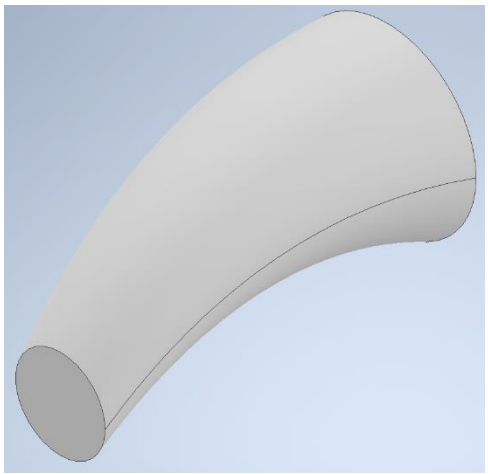
Scale factor =
Box where you can specify how much your original **Part** will be scaled by (1.000 ul; means that it does not scale, that measurements remain the same as the original **Part**).

Mirror Part =
The **Derived Part** is rotated/mirrored in the **Plane** you selected.

1. Assignment – Packaging

In this assignment we go over how to use **Derive**, to create a box/ packaging for an item. Start by creating any item:

Example:



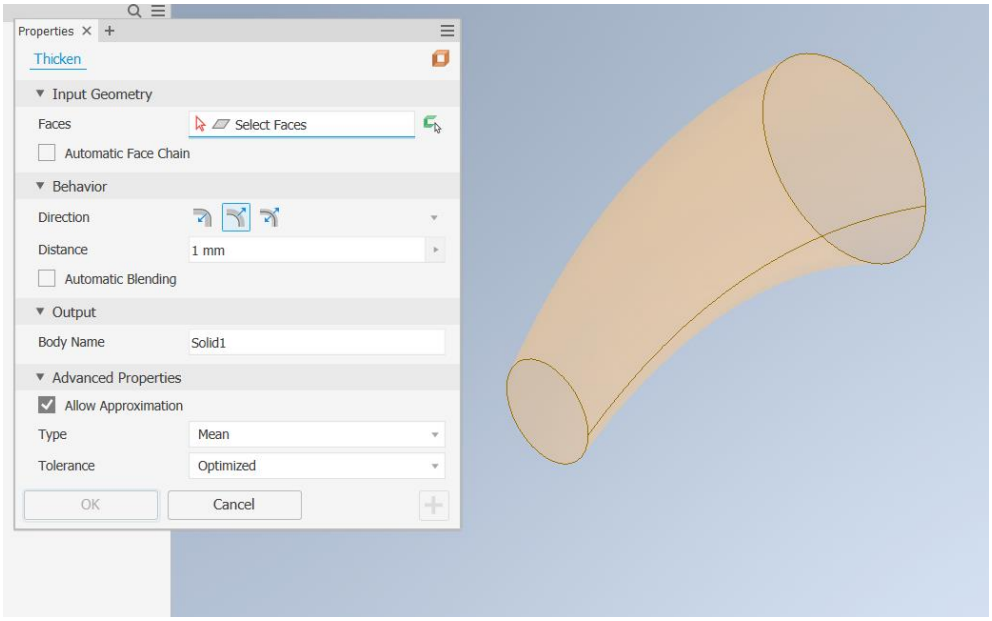
*Uses the "Loft" **Part** as an example.*

1. Create new **Part** file.
2. Insert an item with **Derive**.

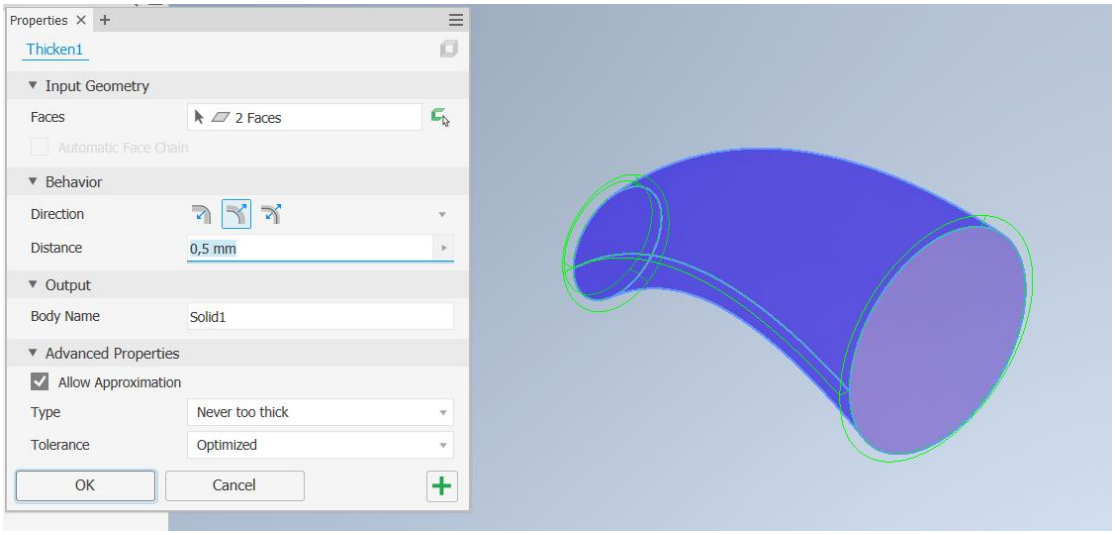
3. Select **Derive style:** **.**

4. Click **OK.**

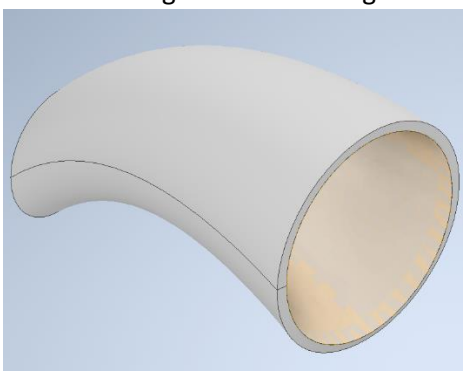
5. Select **Thicken / Offset** which is under the "**Modify**" tab (see more under the **Modify** document):



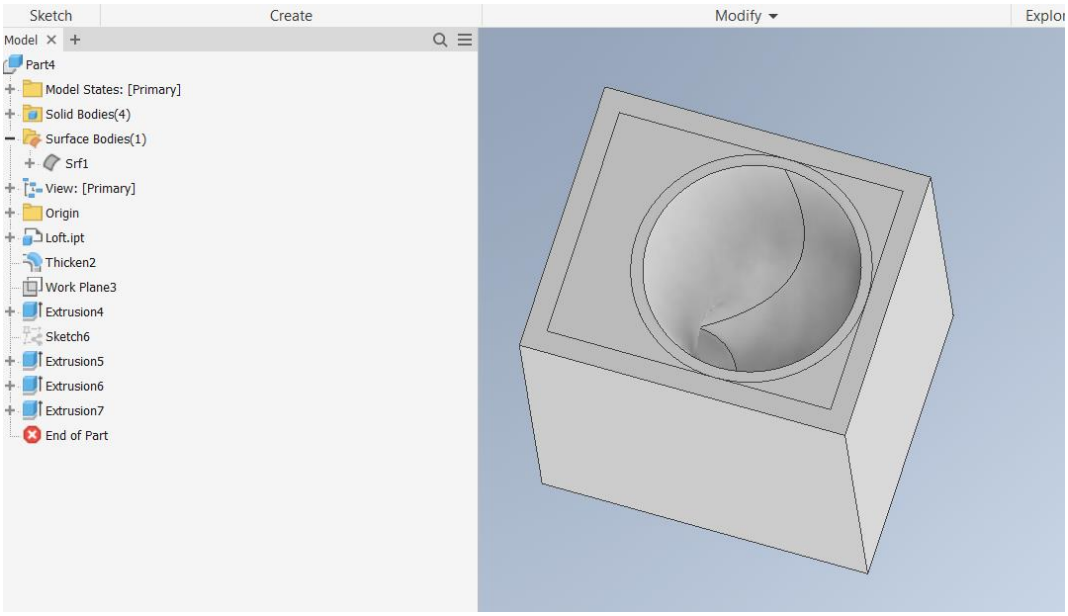
6. Select all but 1 of your **Faces**, select the largest end of the shape, it then becomes a hollow version of the previous one:



7. Set the settings so that the original **Part**" can be in the new **Part**:

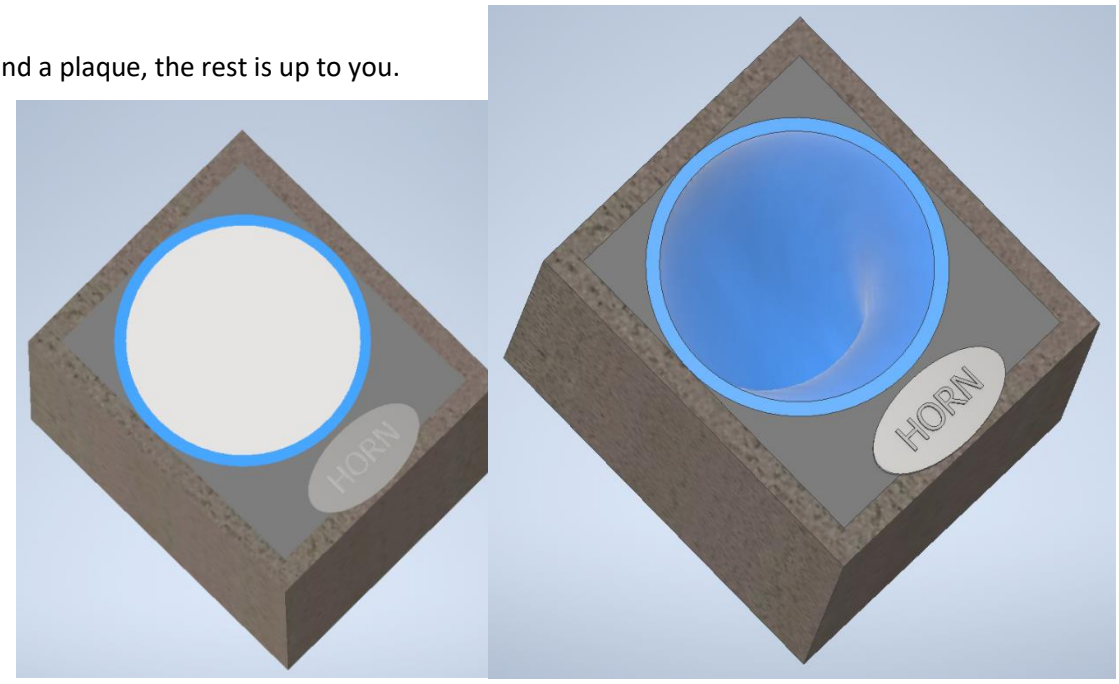


8. Create a box around the new shape (feel free to split **Solids**):



9. Give it details, and a plaque, the rest is up to you.

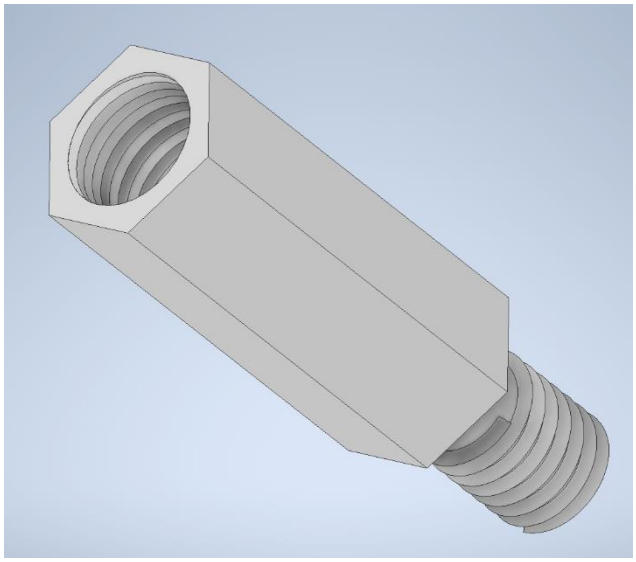
The finished product:



2. Assignment – Modification of an existing product

In this assignment we go over how to use **Derive** to make changes to an existing product. It can, for example, be a standoff bolt or something similar. With **Derive** you can cut some of your standoff bolt. You do this when you buy the raw material, and then cut it to the desired shape. This can be a good idea to save money.

As written above, we use a standoff bolt as an example:



How to make a standoff bolt:

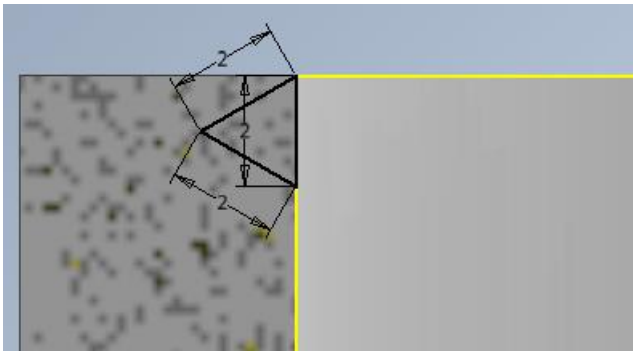
1. **Extrude** the bottom cylinder of the **Part**:
    - It's 15 mm in diameter.
    - It's **Extruded** with 30 mm

2. Create a **Coil** on the bottom of the cylinder:
    - The **Sketch** is an equilateral triangle of 2 mm going away from the cylinder.
    - Flip the axis.
    - Use **Pitch Height**.
    - 3 mm **Pitch**.
    - 20 mm **Height**.

3. **Extrude** (70 mm) a hexagon, with a 15 mm side.
4. **Extrude** Cut (65 mm) from the top, a circle of 20 mm.

5. Create a **Coil** from the top of the hexagon inside the cylinder hole:

- The **Sketch** is an equilateral triangle of 2 mm going away from the cylinder.
- Flip the axis.
- Use **Pitch Height**.
- 4 mm **Pitch**.
- 60 mm **Height**.
- Close Start/End med FA at 60° and TA at 60°.
- Boolean = **Cut**.



1 & 2 is one **Solid** and 3, 4 & 5 is the 2nd **Solid**.

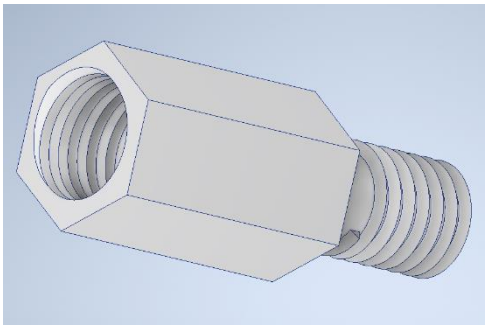
After the Part:

When you have your **Part**, whether you made it or found it online.

1. Start by creating a new **Part fil**.
2. Insert your **Part** with **Derive**.
3. **Derived Style**:
    a. 
    b. You can also select:  if you like working with one **Solid**.
4. Make sure "**Solid Bodies**" is turned on.

5. Press "**OK**", is there a message, then click "**OK**".
6. Make a **Sketch** on top of the hexagon.
7. Use "**Project Geometry**" and select the sides of the hexagon.
8. **Extrude Cut** with 25 mm.

The finished product:



Decal

Introduction:

The **Decal** function, is used to conform an image file or **Word/Excel** document to a surface. Before you can use the **Decal**, there must be a **Sketch** with the desired image and its location.

1. Start a **Sketch**.
    a. Insert an image.
    b. Place image.
    c. Finish **Sketch**.
2. Use **Decal** which are under the **Create** tab.
3. Now select the image (Often Inventor will auto-select).
4. Select **Face**, the primary (Often Inventor will auto-select).

"**Automatic Face Chain**" = *Turns off/on, that* surfaces can connect to the primary.

"**Wrap to Face**" = *Turns off/on, a* change on how your **Decal** behaves. It is recommended that you try both to see which one you like best.

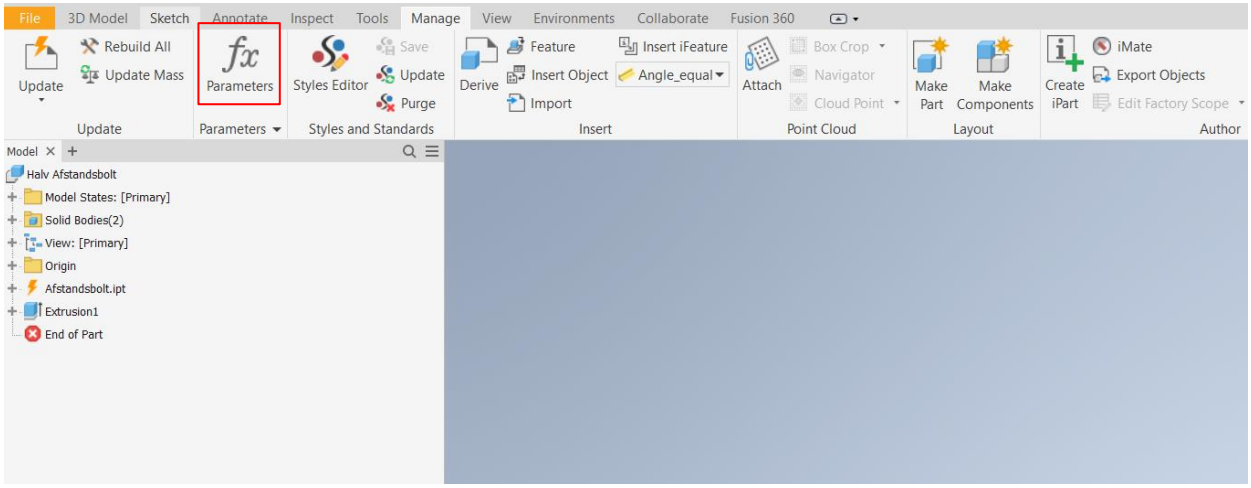
5. Click "**OK**".

Parametric

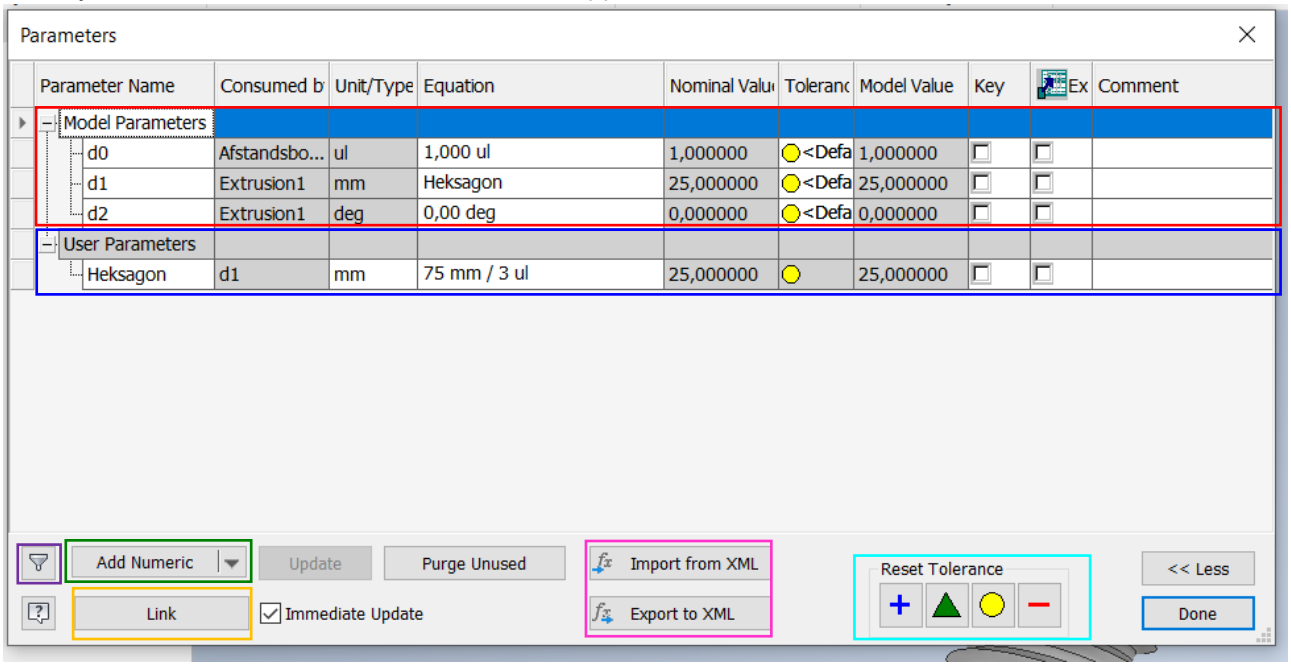
Introduction:

Parametric is a method to make measurements easier, especially if you have to write the same thing many times. The method is called **Parameters** in Inventor.

Here is **Parameters** located:



When you have clicked on **Parameters**, this box appears:



Menu

Red = This is a list of **Parameters** that Inventor automatically adds, as Inventor creates its own parameter in the background when drawing.

Blue = Here you can see an overview of your own **Parameters**.

A **white box** means that you can change it.

A **grey box** means that you cannot change it.

To make your own:

When entering a measurement into **Sketch** or **Feature**. Can you make a **User** "**Parameter**" by writing "NAME = XX" instead of specifying the measurement.

- NAME = the name of the **Parameter**. You cannot use spaces in the name, so use "_".
- XX = Equation.

Buttons

Green = Here you create new **Parameters**, there are three different types.

- **Add Numeric =** Here you can add a number or formula as a **Parameter**.
- **Add Text =** Here you can add text as a **Parameter**.
- **Add True/False =** Here you can add true/false as a **Parameter**.

Yellow = Here you link the **Parameters** from a 2nd **Part**. However, this doesn't work if you use it as an **iPart**.

Purple = Here you can filter the menu, so it's easier to find the desired **Parameter**.

Pink = Here you import/export **XML**-files.

Cyan = Here you change the Tolerance.

IPart

Introduction:

An **iPart** is used to make the work process easier and faster, if a product has many different sizes, widths, etc. A good example of an **iPart**, could be the bolts and nuts in the "**Content Center**". In addition, **iPart** can be used amongst many different **Projects**, if you add them to the "**Content Center**".

In this guide we start with, how to create an **iPart**, and place it in the "**Content Center**".

This guide also contains some assignments, it is best to start a new project, where you create the **iPart** and insert it into the "**Content Center**".

When creating an **iPart**, start with a normal **Part** and build it up parametrically, since you edit/create new **iParts** by changing the various **Parameters**. See more about how to use **Parametric** here.

There will be two different assignments, that deals with **iParts**, one easy and one difficult. In the easy one, we create a simple bolt, here you'll choose how it should appear. The **iPart** has to consist of 4 different sizes. In the difficult assignment, we create an **ikLego Brick**, here the **iPart** must consist of a 1X1, 2X1, 2X2, 3X2. You can choose which of the two assignments, you want to complete.

REMEMBER

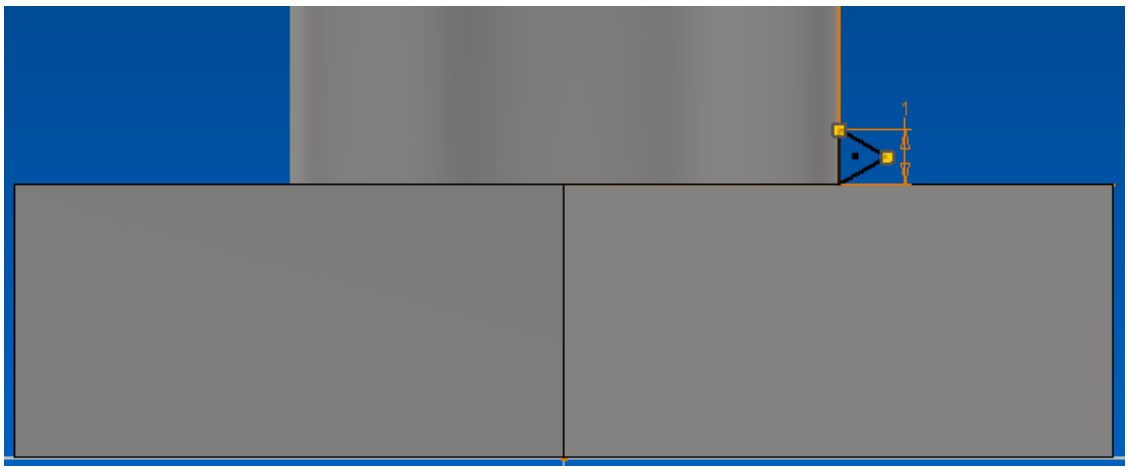
Before you create an **iPart**, you must open **Excel**. You don't have to start any assignment, just have it open in the background (since Inventor sometimes can't locate **Excel**, then you'll have to start over, when selecting your **Parameters**).

Easy Assignment - Bolt

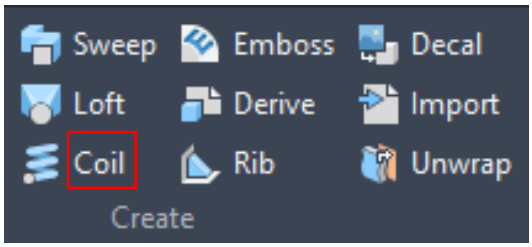
You decide how the bolt should look, and what dimensions it should have. When measuring, name it something you can remember, and is easy to find in the **Parameter** list. Then create a bolt with the **Parametric** method. E.g. in "**Extrude Properties**", you can write "**Lenght=10**", instead of just writing 10. See more about how to use **Parametric** here.

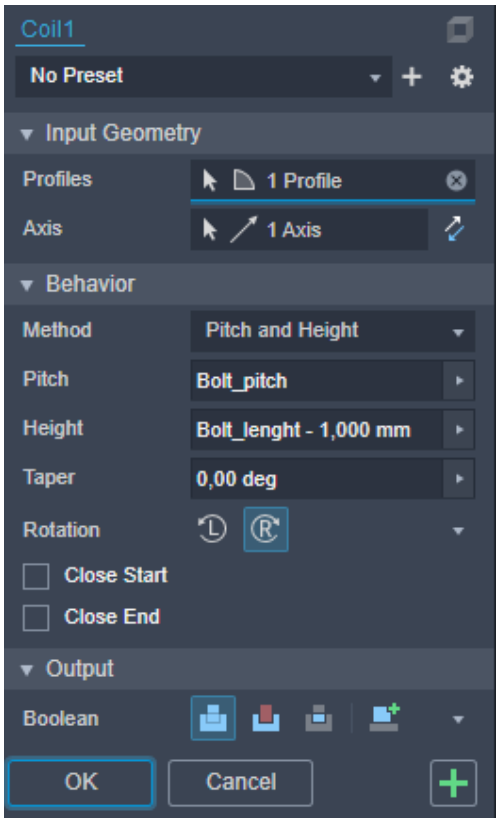
More examples of what you could name your **Parameters** is further down.

To make the thread itself, start by making a **Sketch** on the side of the bolt. Create the thread's desired shape here.



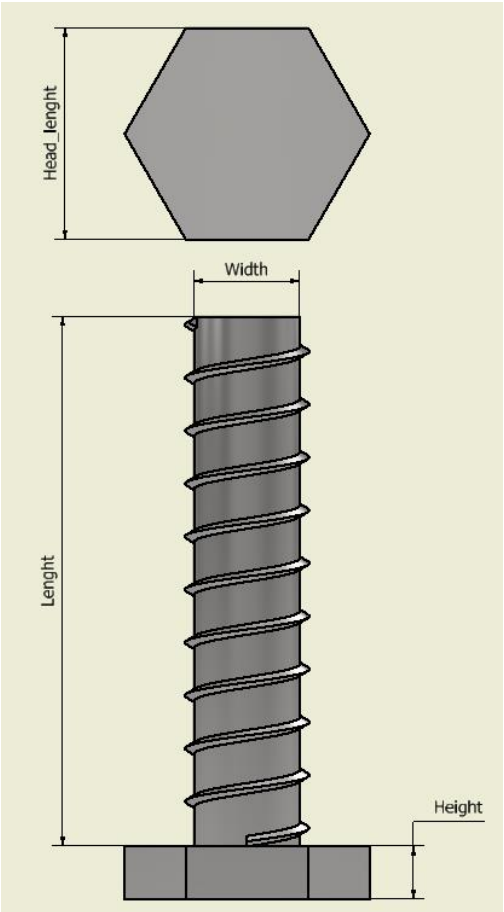
When the **Sketch** is made. Press **Coil** under the **Create** tab. Afterwards, the **Properties** comes up. See more about [Coil] here.



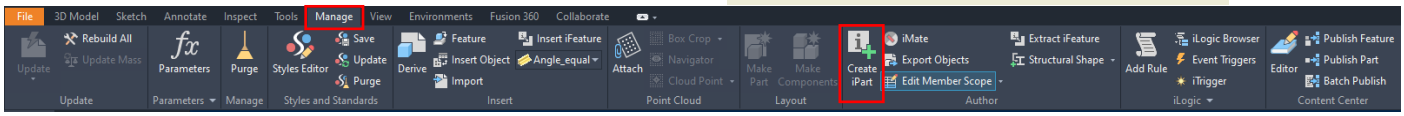


Then use **Parameters** in the **Behavior** box. In the assignment, we use the **Pitch** and **Height** method.

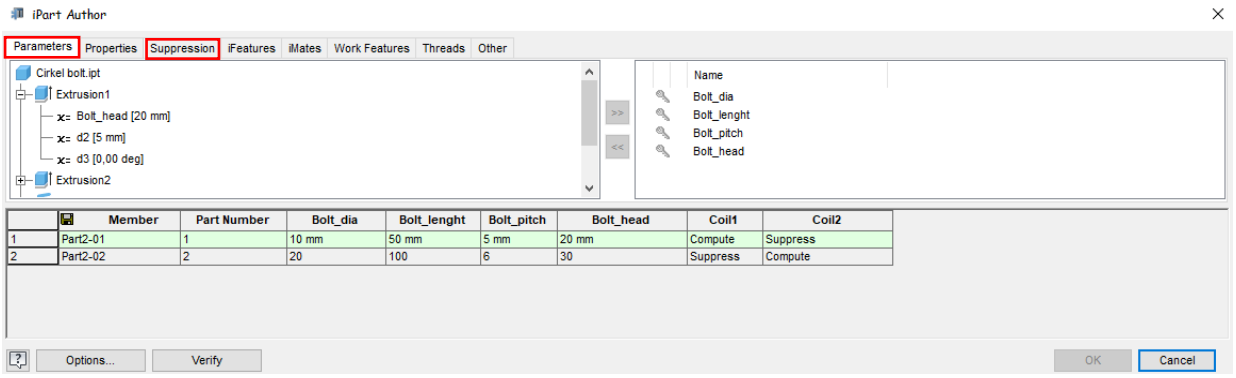
Below are examples of how your **Coil** could look, as well as what you could name your measurements.



- After the creation of the main model (the bolt), then select "**Manage**". Under "**Manage**" press "**Create iPart**".



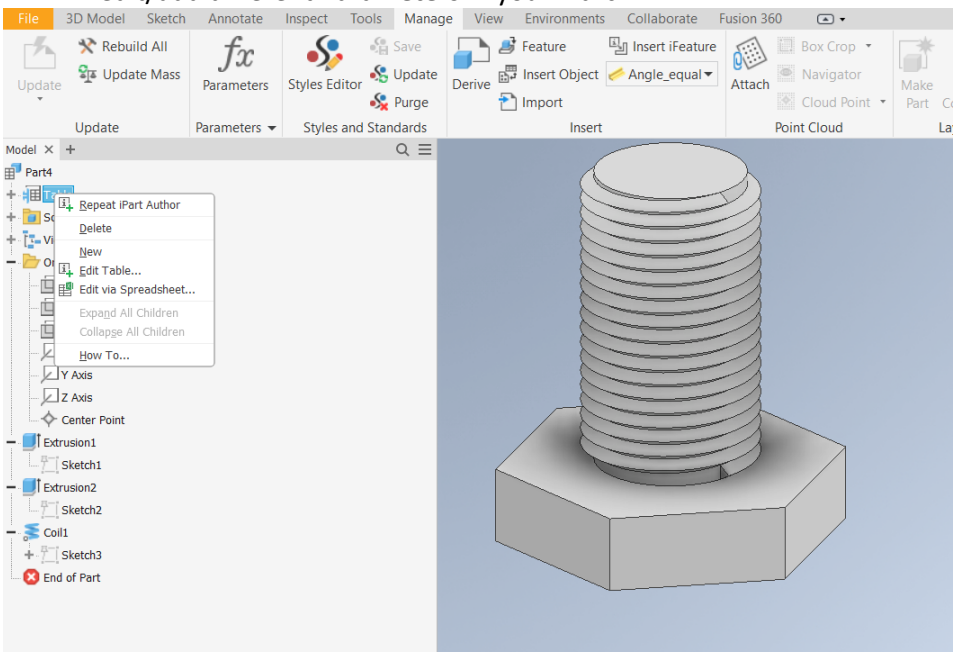
Go to "**iPart Author**", here you choose which **Parameters** are utilized in the **iPart**. In these assignments we will only need to use two different windows, **Parameters**, and suppression.



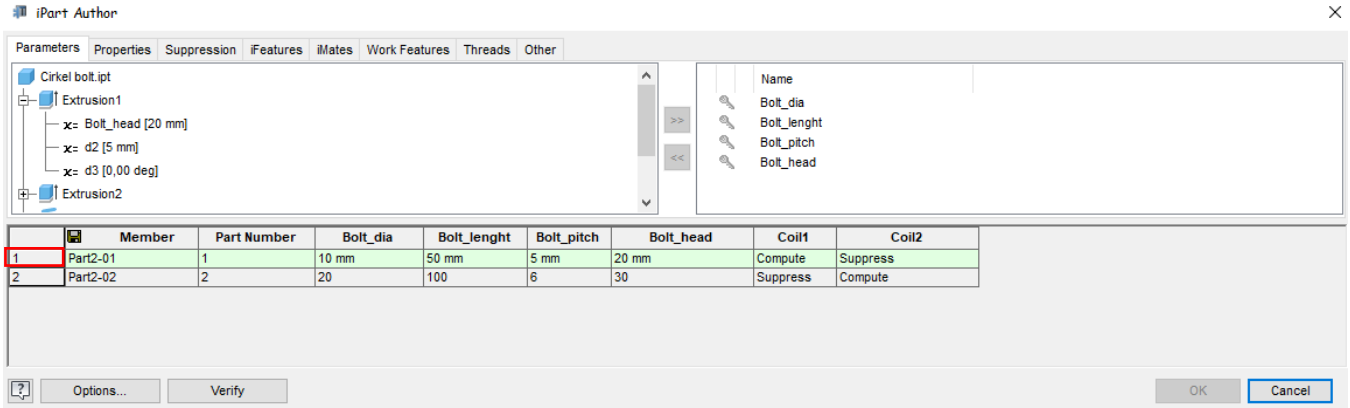
Parameters = Select which measurements/features should be able to be made larger/smaller.

Suppression = Select which measurements/features should be able to be switched on/off.

- After you have selected the measurements/features to be altered, click "**OK**". Over in the History, right click on **Table**, and select **Edit Table**. Here you can open up the "**iPart Author**" window and edit/add different **Parameters** in your **iPart**.

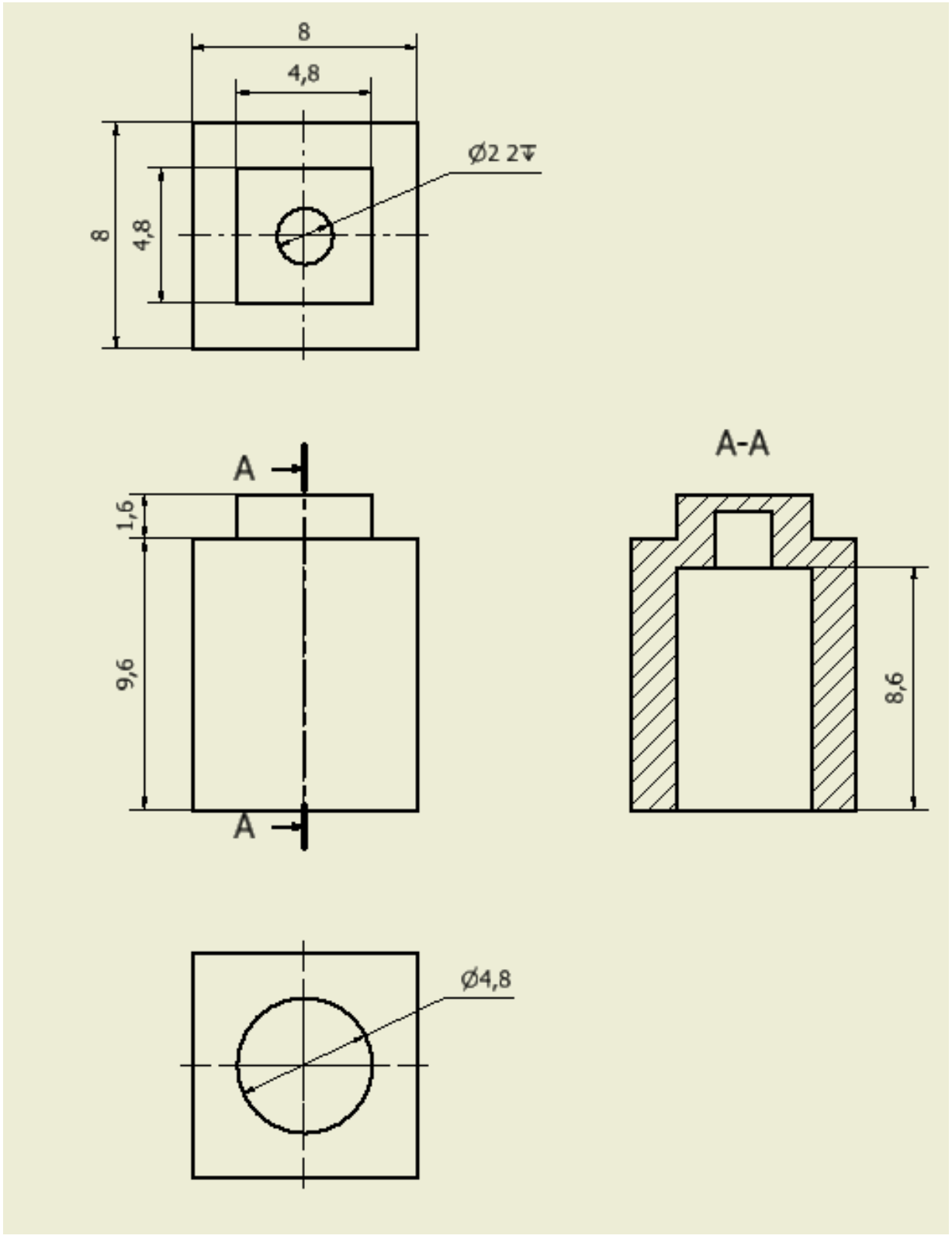


You can also press "**Edit via Spreadsheet**", where it opens the table in **Excel**, you can edit from there. If you wanted to create a new parameter set, you must right-click on the first parameter row and select "**Insert Row**".

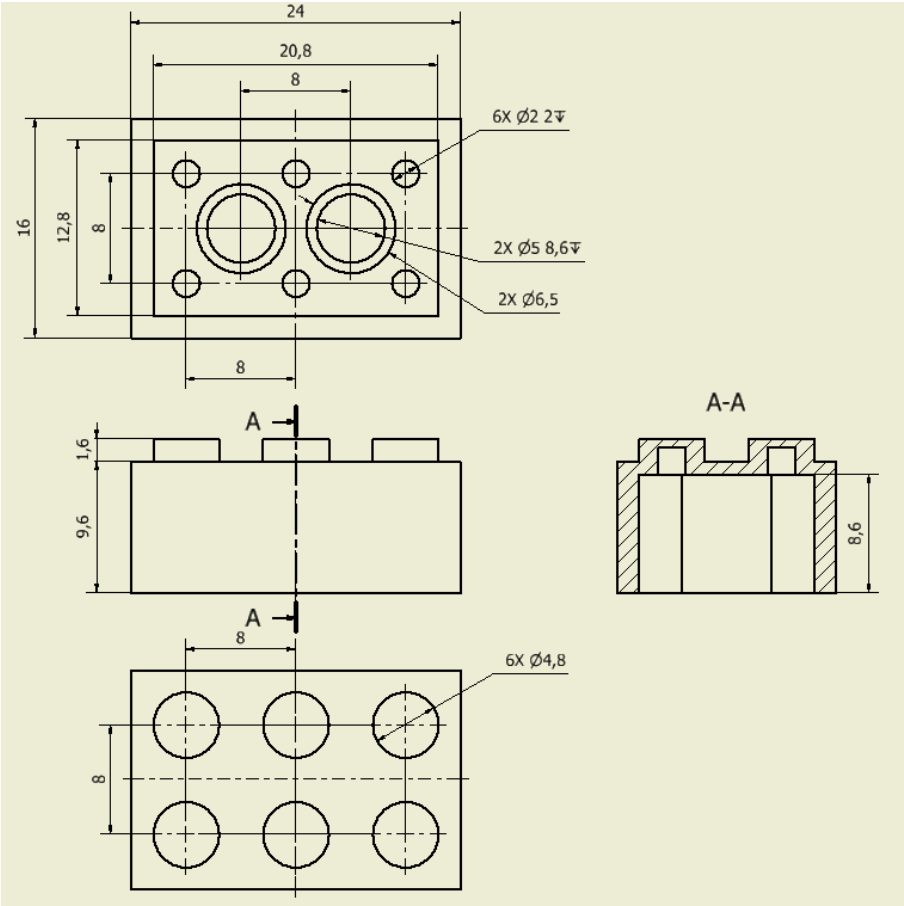


Difficult Assignment - ikLego Brick

In this assignment, we make an **ikLego Brick**, which has 4 different sizes. When you make an **iPart**, start with a base and work from there. Therefore, start by making **ikLego Brick** 1X1. If you haven't completed the easy assignment, we advise that you at least read through it.



After it's created, we have to work our way to subsequently making; 2X1 and 2X2, and finish with 3X2. These can be seen below.



Here you need to find out of which measurements/features, are parametrically and used in the **iPart**. All the 4 different pieces must be made in the same **Part**, using the **iPart** function.

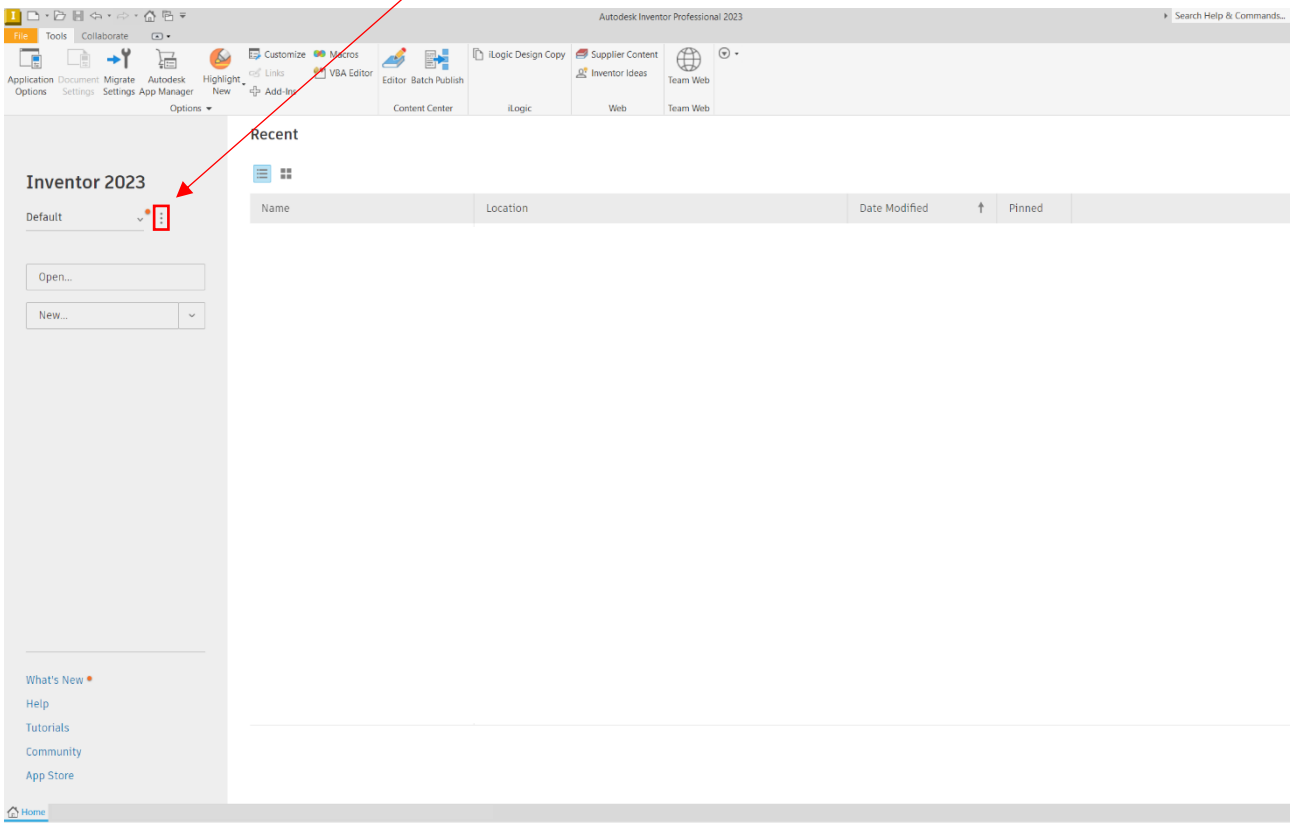
Content Center

When adding a self-made **Part**/**iPart** to the "**Content Center**", you must start by creating your own "**Content Center**" library (it does not matter in which project you create your library, as it would be shared in the same way as the other libraries).

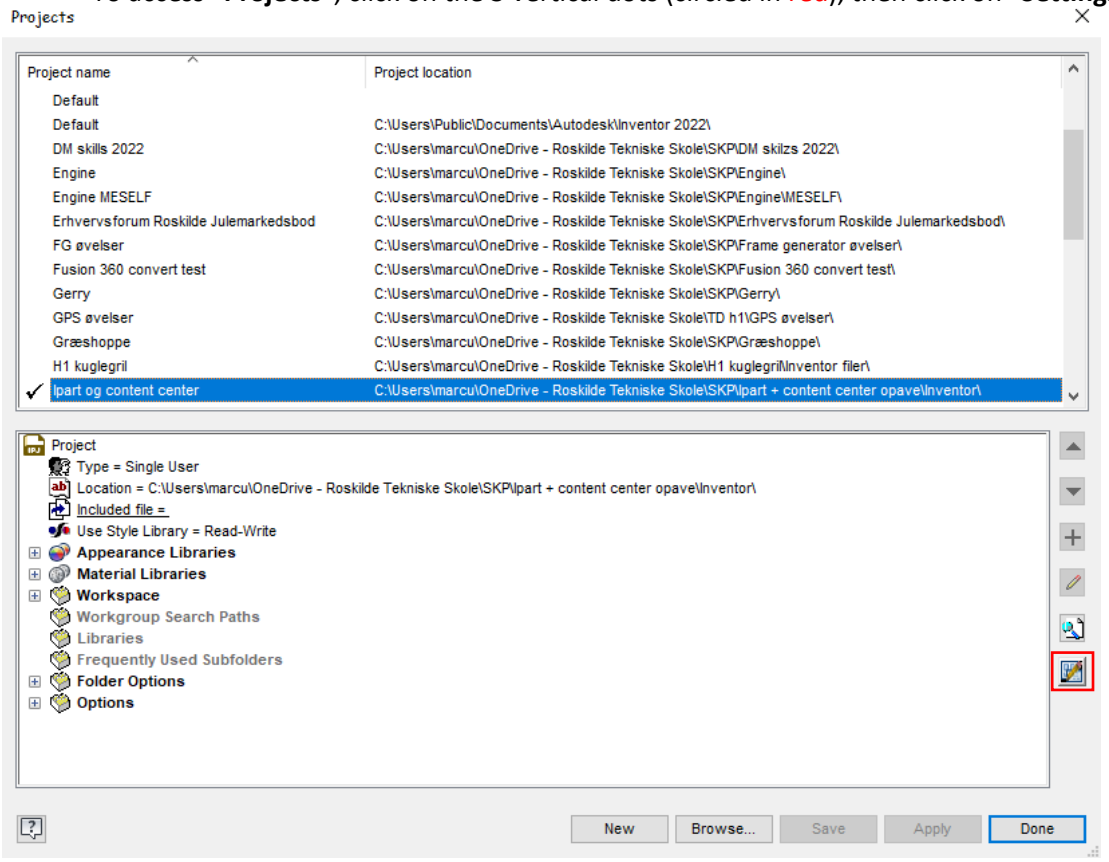
Create your library

1. To create your library, open "**Projects**" and click on " **Configure Content Center Libraries**".

"**Projects**" is on the front page:



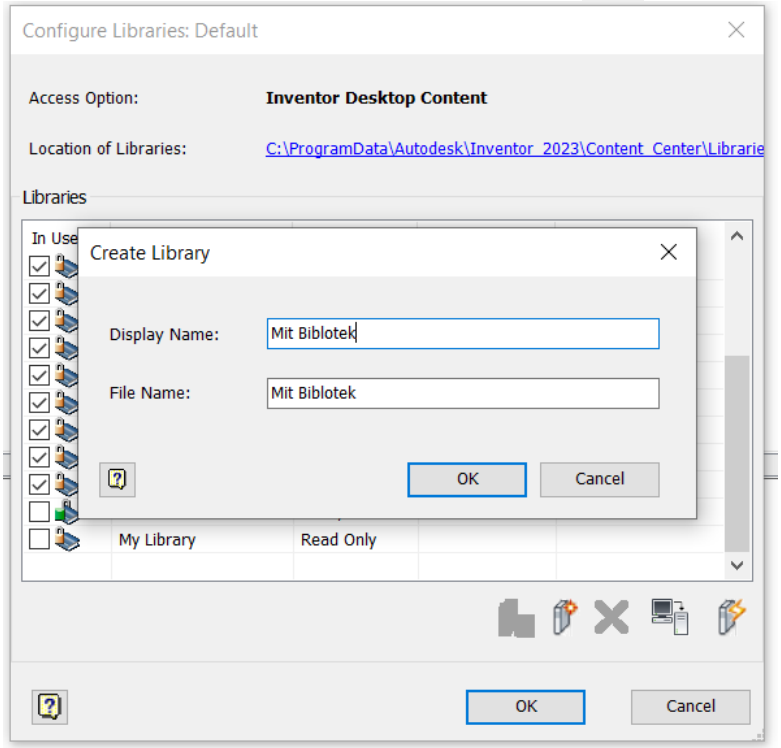
To access "**Projects**", click on the 3 vertical dots (circled in red), then click on "**Settings**":



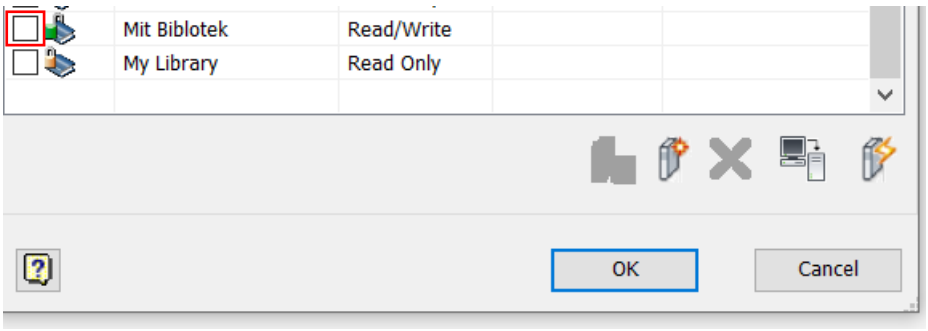
Click here to enter the "**Content Center**".

Here you can see all the different "**Content Center**" libraries that are in Inventor.

2. In here, press "**Create Library**" and name your library.



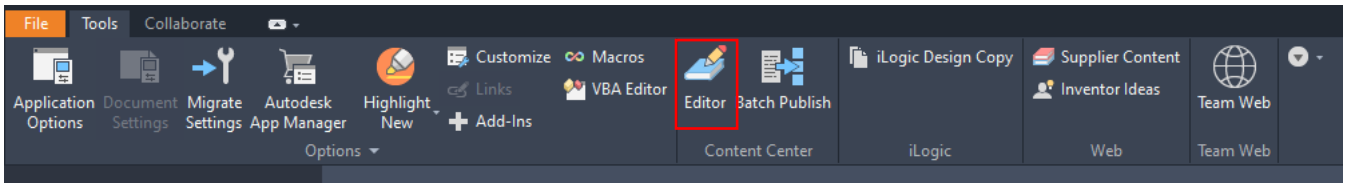
3. Remember to check your "**Library**" on:



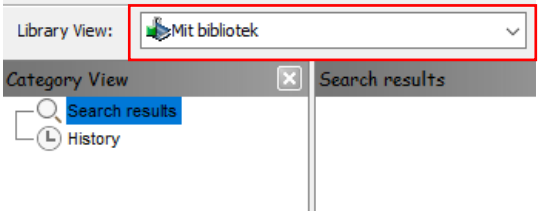
4. When it's done, you click on "**OK**" and "**Done**" in **Project** (***remember*** *to press save first*). Now you have created your very own library, here you can drop your **iPart**/**Part** in.

Insert your own Part

1. Before you can insert your **iPart/Part** into the library, you must modify your library. To do that, go to Inventor's home screen and press **Editor** under the "**Tools**" menu.

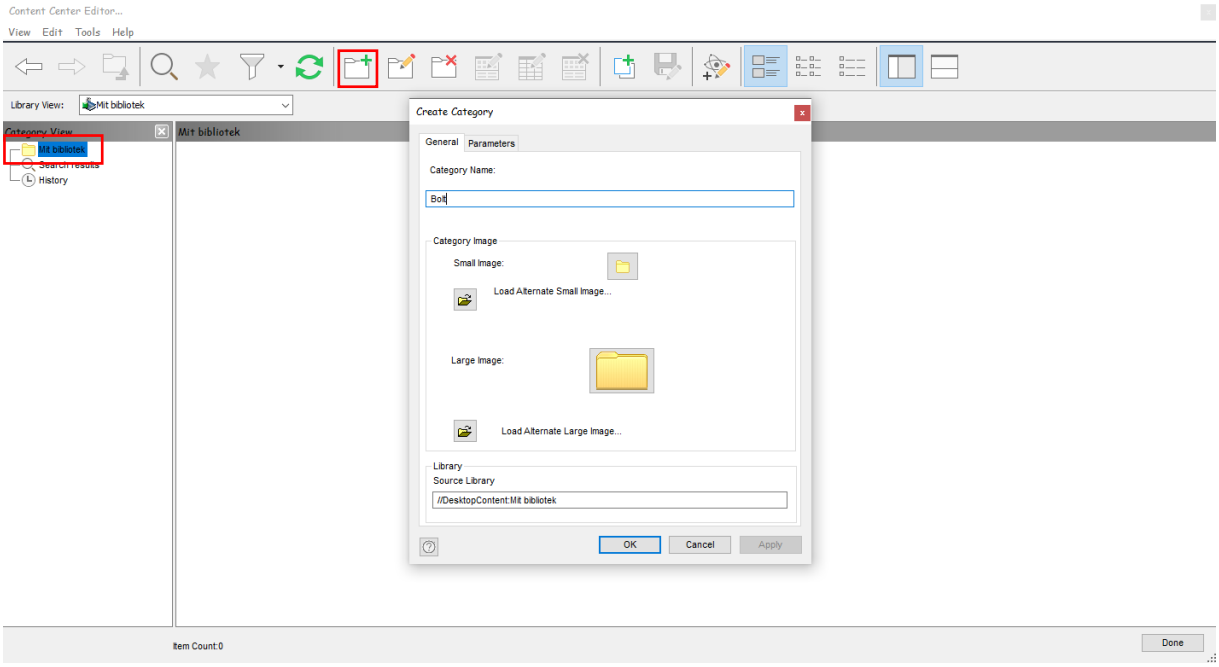


2. In here, click on **Library View**, and select the library you have created (it makes the next work process easier if you can only see your own library).

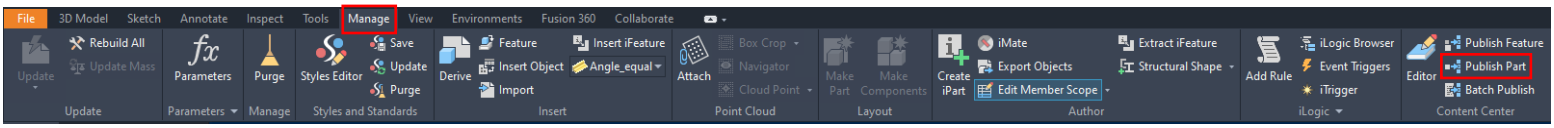


3. In the library, create a category and subcategory, in which add your various **iParts**/**Parts**. To do this, right-click in the "**Category View**" box and select "**Create Category**". Here you get a window where you can name the category and select an image for the folder.

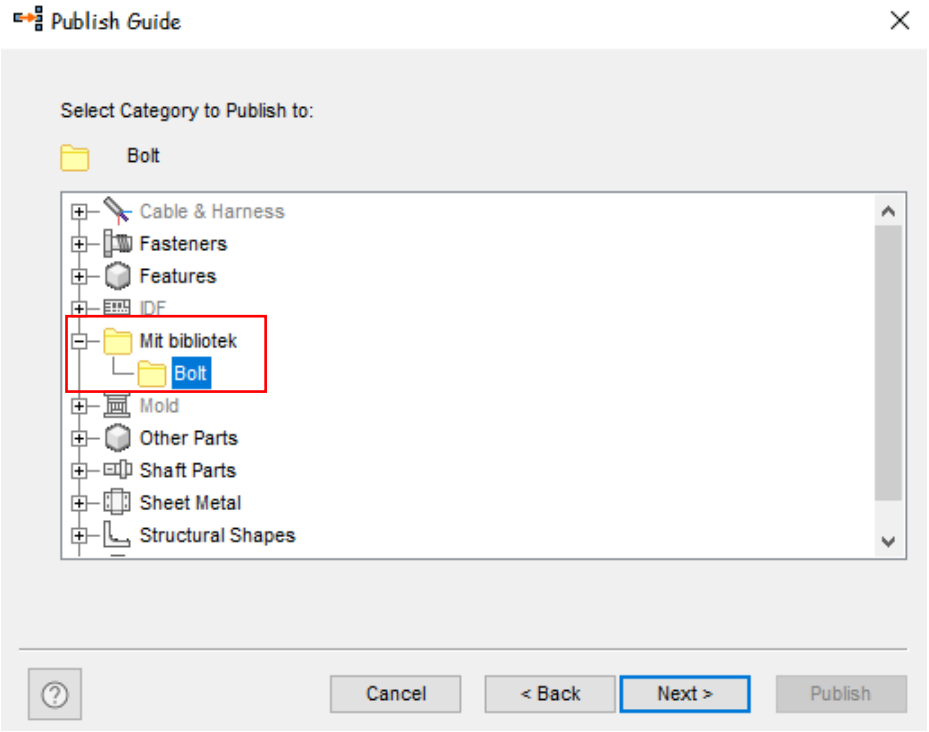
4. Then you click on "**OK**" after which, it would be on the left side, under "**Category View**". When you need to create your subcategory, click on your library, and then click on the "**Create Category**" icon. Where the same options are, as in the first category.



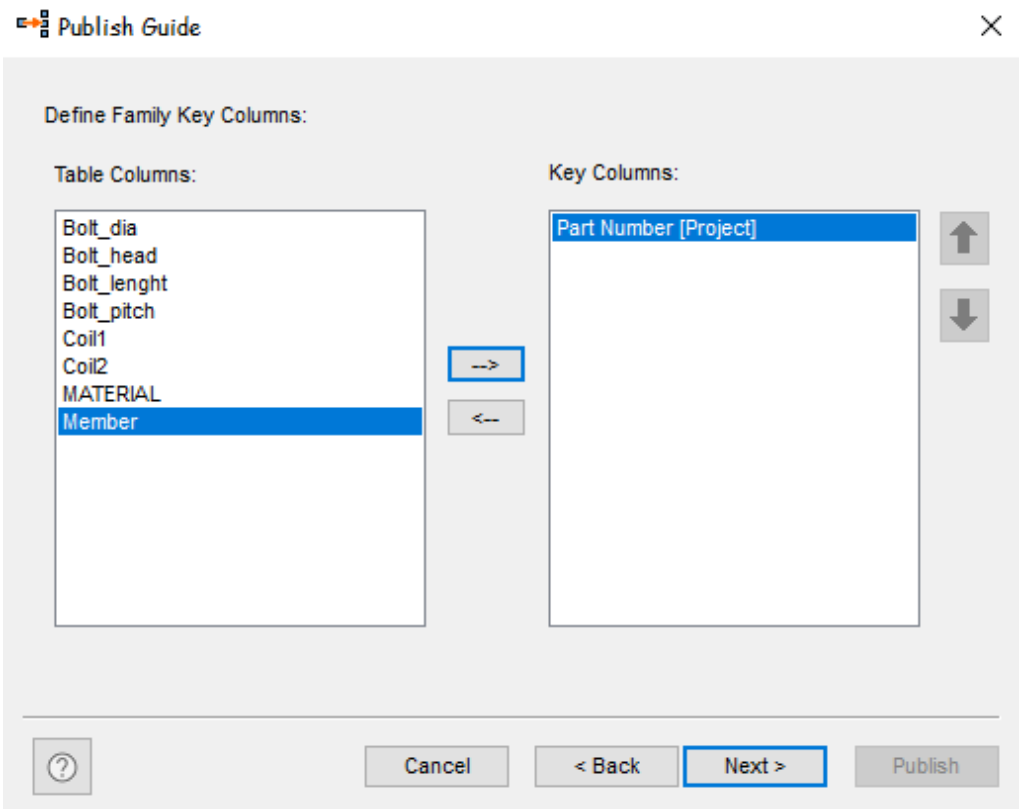
5. Now that you have created and modified your library, insert the various **Parts/iParts**. When inserting your **Part/iPart**, start by opening it. Then go to **Manage** (at the top of the screen) and click on **Publish Part**.



6. Here select which library the **Part**, should be inserted into. Since there is only one library that you are allowed to edit in, there would only be one option to select.
Then click "**Next >**", then select which category, and subcategory, your **Part** should be under.



7. When selecting the folder and clicking on "**Next >**", you will get to a menu, where you can select some **Parameters** (this menu is not important now, so you just move on).
Afterwards, define how the **Content Center** should name the **Part**. Here you select "**Part Number [Project]**", as the name is on the **Part**, and what you have saved it as.



8. (Click "**Next >**") Here you can give the **Part** different properties, such as whether it is ISO or ANSI (standards). In the next menu you can, select which image is displayed in the "**Content Center**". Remember you have to locate it. When finished, click on "**Publish**".

If you wanted to double-check that it's done correctly, start a new project. Then open an **Assembly** file and select your **iPart** from the "**Content Center**" menu. If done correctly, one's category with subcategory would appear. Where you placed the **iPart**.

Solid Bodies

Introduction:

Method where you construction geometry in a **Part** file called the skeleton **Part**.

Put in another way; Instead of making the **Part** files separately and then assemble them in an **Assembly**, you make them in one **Part** file.
That is, **Part** file = 1 "**Assembly**", and the **Parts** = "**Solids**".

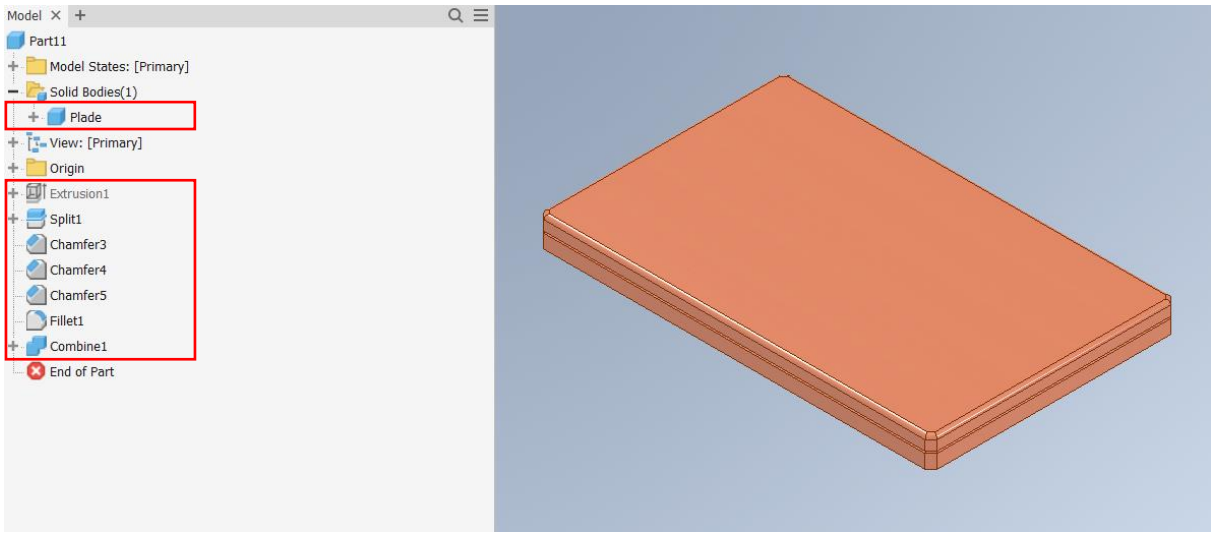
In the example, we make a table.

***Note**: You can change the name of your "**Solid Body**" by pressing the F2 key (keyboard).*

If you do not use the "**Solid Bodies**" method, you would create the **Board** → **Legs** → **Edges**, all of these would have a separate **Part** file. And then assembled in an "**Assembly**".

Table

Wooden board:



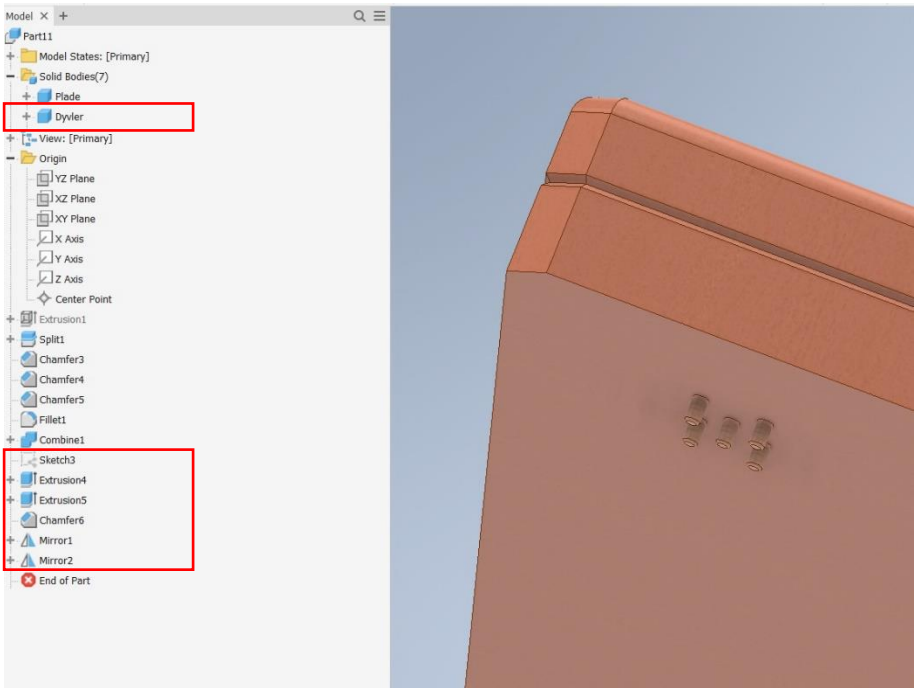
Remember *when you add a new section of your model (the table), remember to make it a* ***Solid***.

Leg:

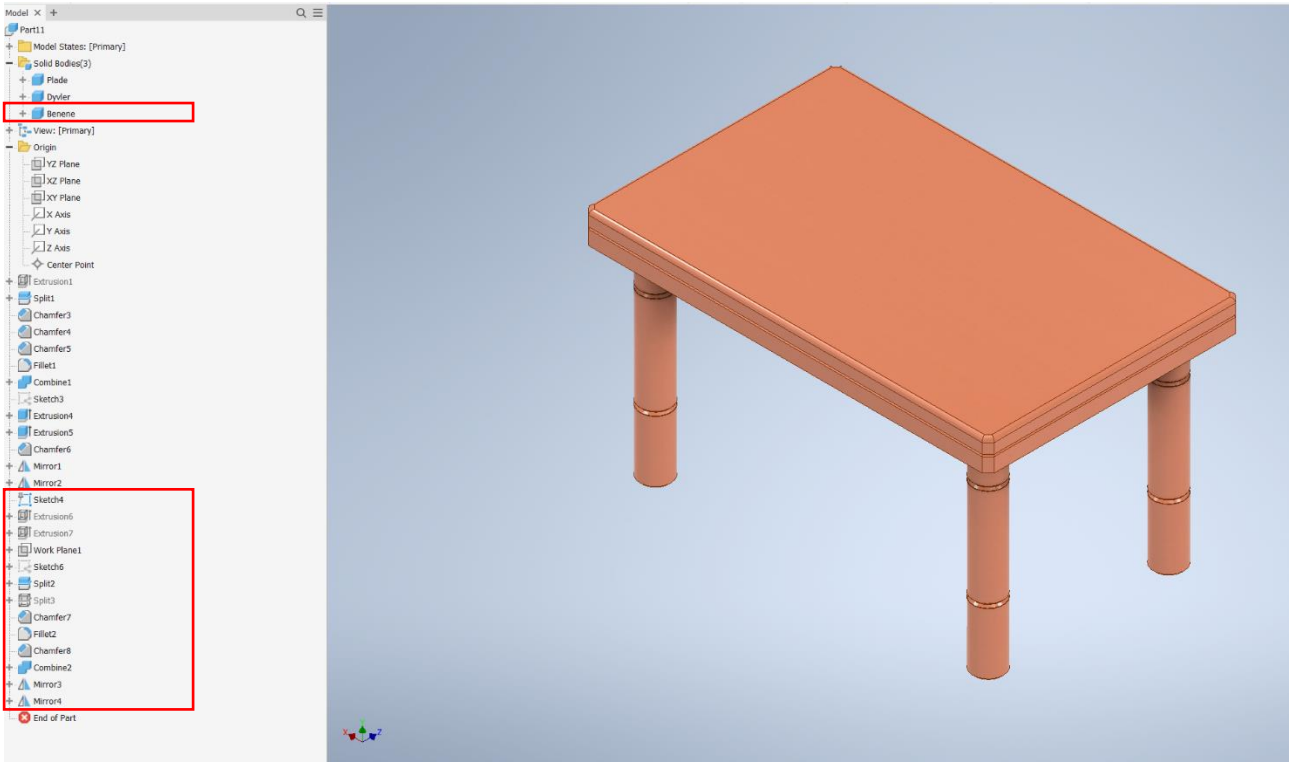
Make the joint between board and legs. Combination between 2 things can e.g. be dowels and glue, brackets, screws, and pockets. In the example we use dowels and glue.

Here in the example we use glue and dowels, as it is an invisible assemblage.

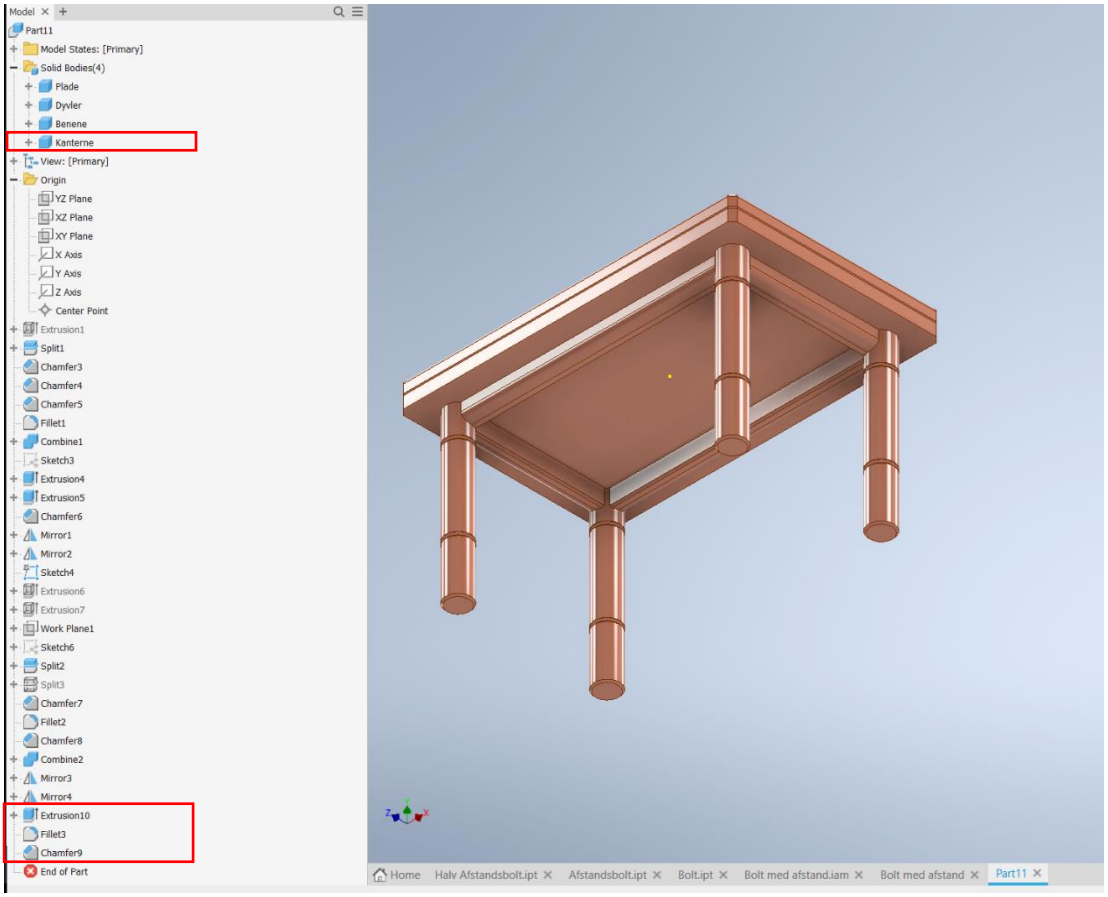
Assembly (Dowels):



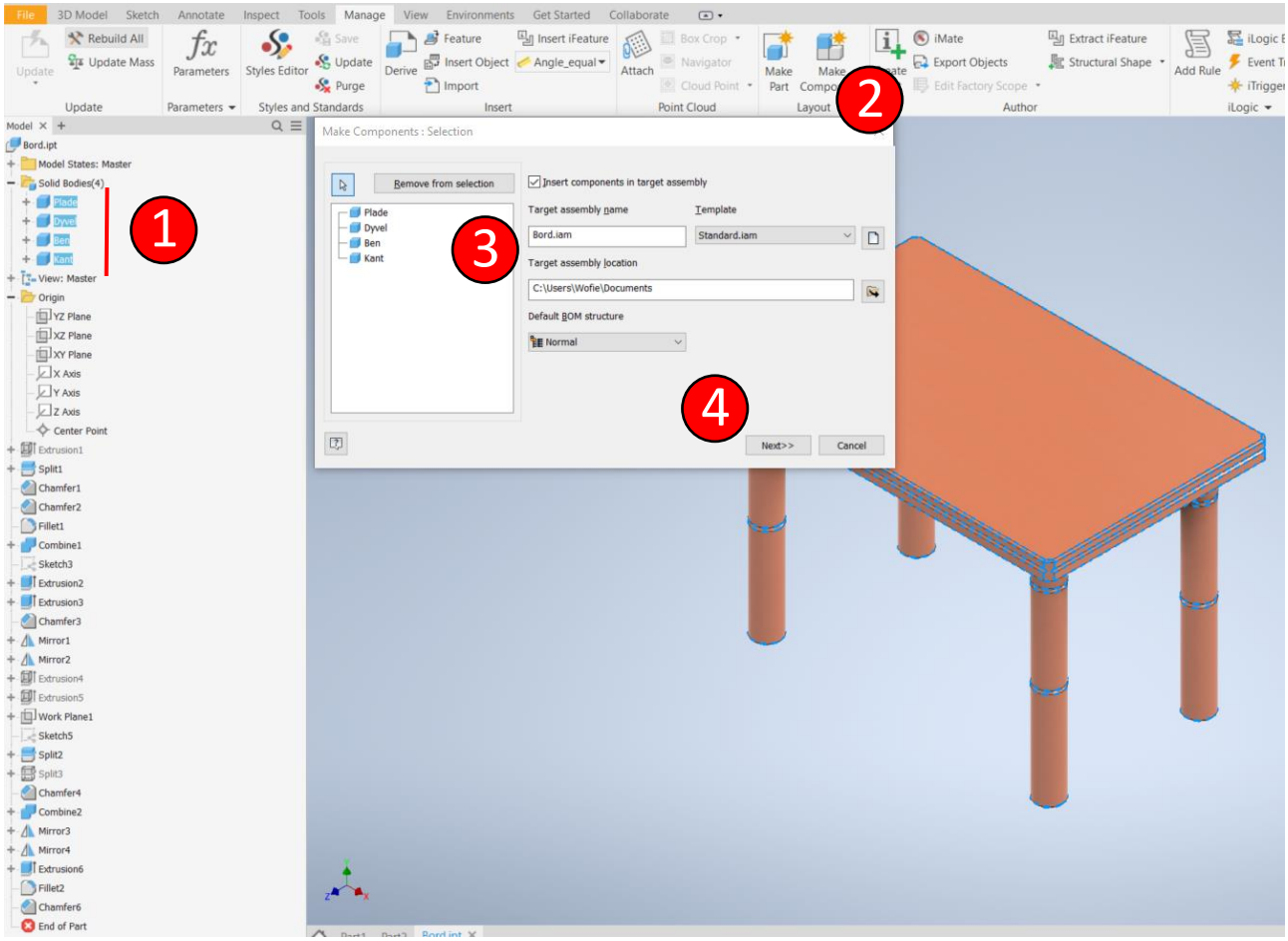
Legs:

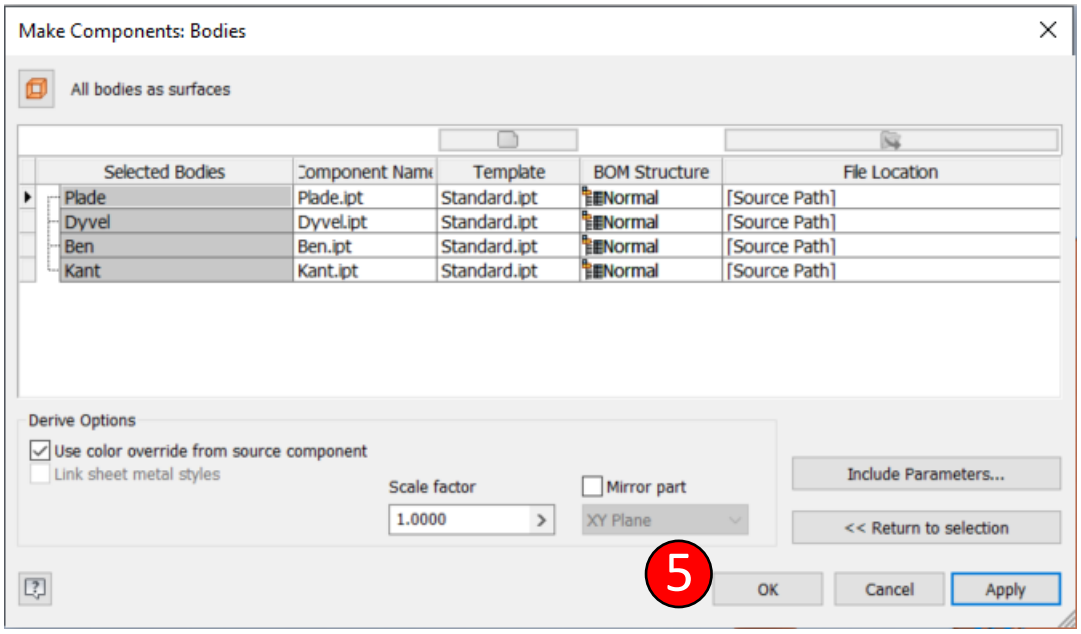

Model
Part11
Model States: [Primary]
Solid Bodies(3)
Plade
Dyvler
Benene
View: [Primary]
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Extrusion1
Split1
Chamfer3
Chamfer4
Chamfer5
Fillet1
Combine1
Sketch3
Extrusion4
Extrusion5
Chamfer6
Mirror1
Mirror2
Sketch4
Extrusion6
Extrusion7
Work Plane1
Sketch6
Split2
Split3
Chamfer7
Fillet2
Chamfer8
Combine2
Mirror3
Mirror4
End of Part

Edges:

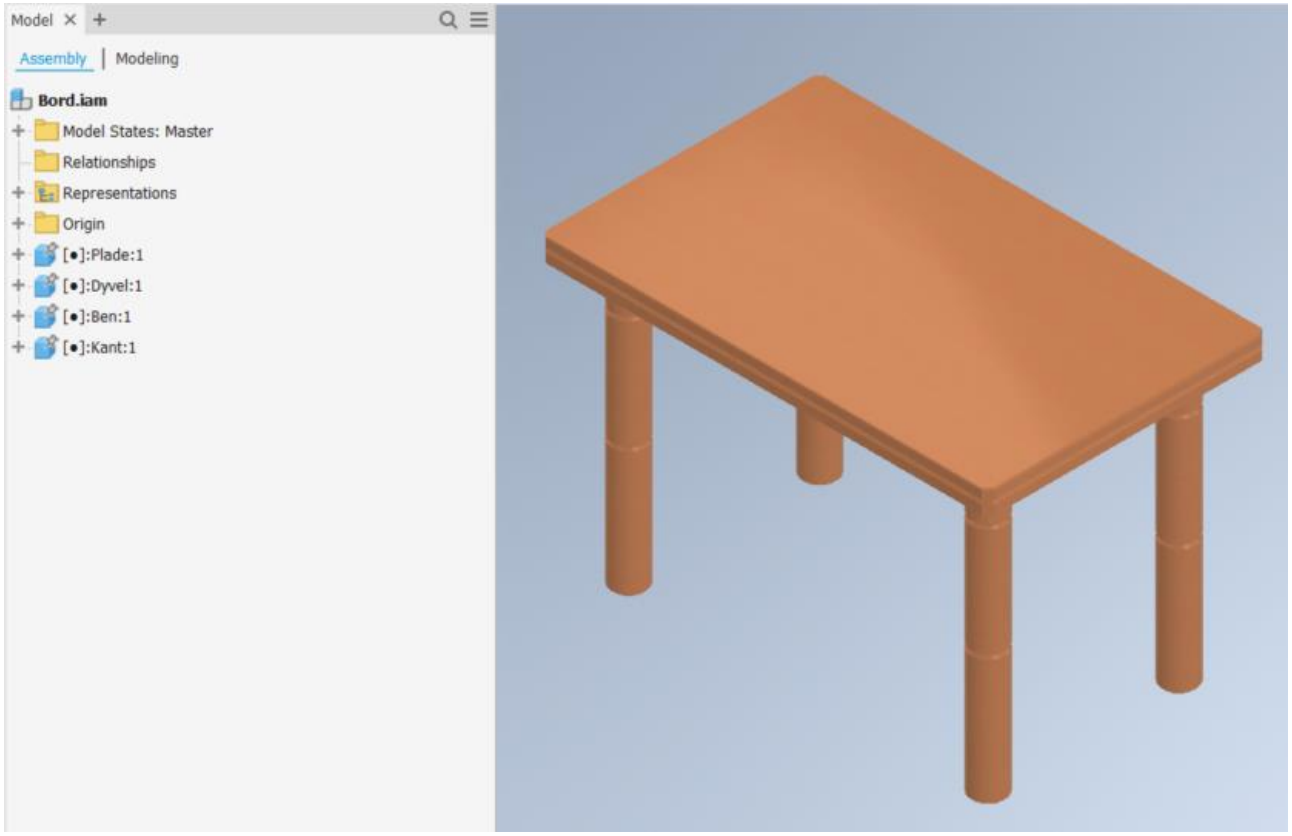

Model
Part11
Model States: [Primary]
Solid Bodies(4)
Plade
Dyvler
Benene
Kanterne
View: [Primary]
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Extrusion1
Split1
Chamfer3
Chamfer4
Chamfer5
Fillet1
Combine1
Sketch3
Extrusion4
Extrusion5
Chamfer6
Mirror1
Mirror2
Sketch4
Extrusion6
Extrusion7
Work Plane1
Sketch6
Split2
Split3
Chamfer7
Fillet2
Chamfer8
Combine2
Mirror3
Mirror4
Extrusion10
Fillet3
Chamfer9
End of Part
Home
Halv Afstandsbolt.ipt
Afstandsbolt.ipt
Bolt.ipt
Bolt med afstand.iam
Bolt med afstand
Part11

Convert Skeleton Part file to Assembly:





Finished result:



Modify – Feature

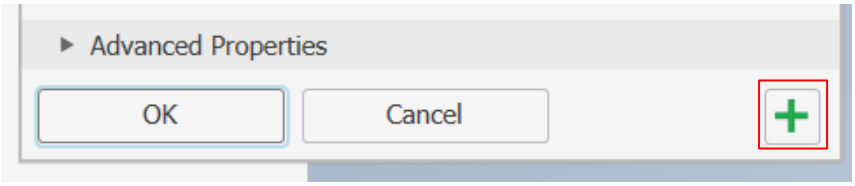
Introduction

After starting a project and creating a **Part** file, **Sketch** and **Solid**: you can use the **Solid** for different **Modify** features, here you can see an overview of all the features, under the **Modify** tab.

General

Green +

The Green + on all the **Modify** features. You use them, to make several different "**Properties**" settings, but on the same **Sketch**.



Index:

Shortcut Keys

Selections:

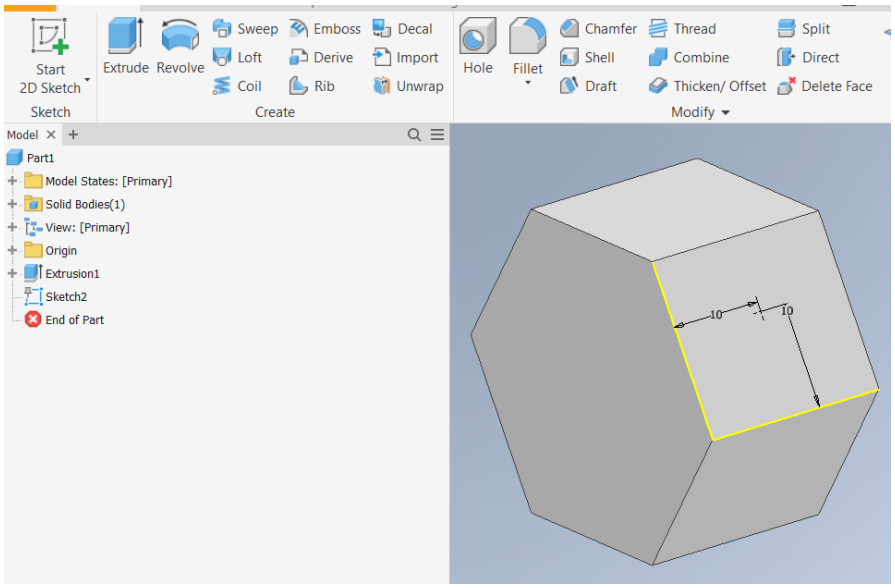
Control (CTRL) = Adds more than one selection. ***Press and hold.*** *(It is done correctly when there is a plus icon next to the mouse).*

SHIFT (↑) = Removes selections. ***Press and hold.*** *(It is done correctly when there is a minus icon next to the mouse).*

Hole

Introduction:

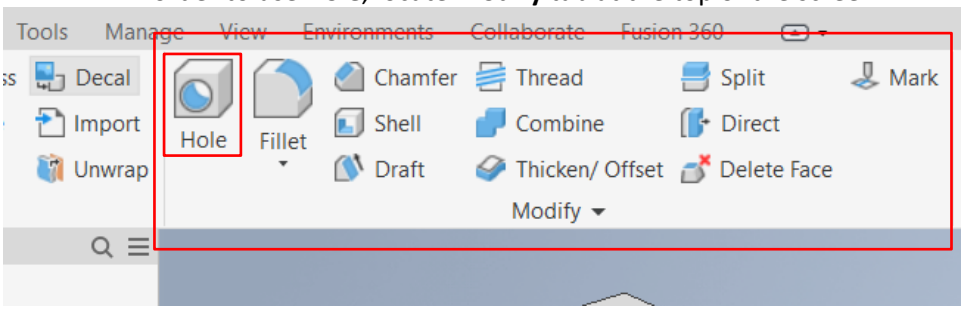
The **Hole** feature is one of the most used **Modifiers**, It makes holes from a **Center Point**, on a **Solid**, in a **Sketch**.



Here is an example of what a **Sketch** might look like when it is finished.

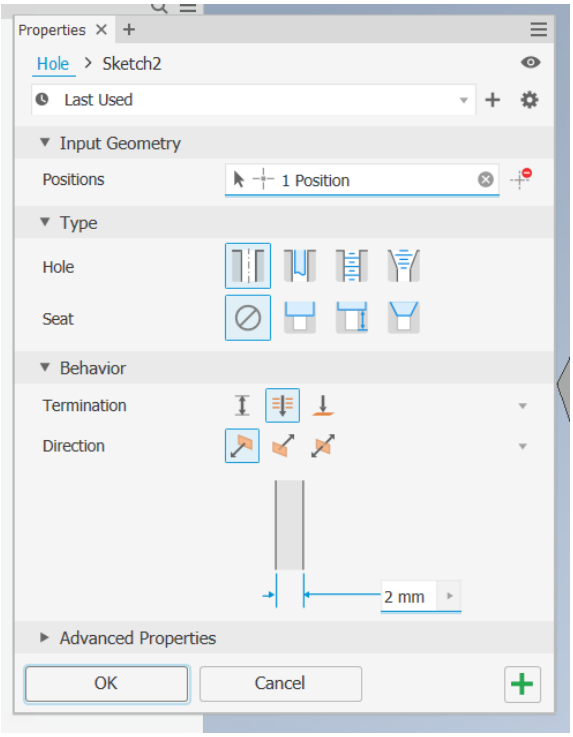
- **Short key** = H

In order to use **Hole**, locate **Modify** tab at the top of the screen:



Select **Hole**. When you press **Hole** a box will appear:

The box is called "**Properties**". This is what **Properties** looks like the first time you make a **Hole**:



Input Geometry

Positions:

Positions is the position of the **Hole**. In the box, Inventor informs you of how many **Positions** (This = Center Points) are selected. The blue line below the word "**Position**" shows you that **Position** is active, if it's active, you can choose your **Center Points**, if the blue line isn't there, you can't. To make it active, click on **Position**. But Inventor auto Activates **Position**.

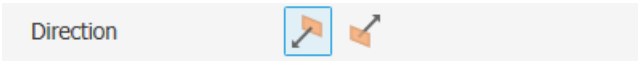
Center Point Creation:

This feature must be Activatesd, when it is Activatesd you can place "**Center Points**" on the **Sketch**.

Behavior

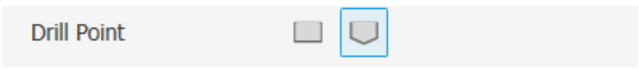
Termination: 

1. Distance 
   - Stops at a specified length.
   - If this is Activatesd, **Drill Point** appears.
2. Through All 
   - Goes through all.
3. To 
   - The **Hole** stops at a **Plane/Surface**.
   - Then this box comes up. Here you select **Plane/Surface**:

Direction: 

Here are 2 icons.

- *First (from the left)* 
  - This is Inventor's auto selection. The Hole protrudes forwards from the **Sketch Plane**.
- *Second* 
  - The Hole protrudes backwards from the **Sketch Plane**.

Drill Point: 

Here are 2 icons.

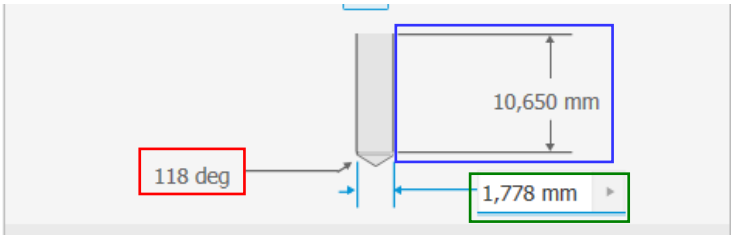
- *First (from the left)* 
  - **Flat**, means the end of the drill is flat.

- *Second* 
    - **Angle**, means the end of the drill is pointed at a specified angle.

Types

Simple Hole:

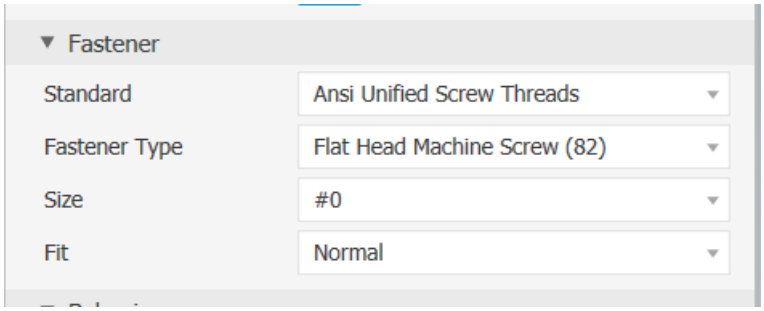
Behavior:



Red = Angle of the tip.
Blue = The depth of the **Hole**.
Green = The diameter of the **Hole**.

Clearance Hole:

Behavior:

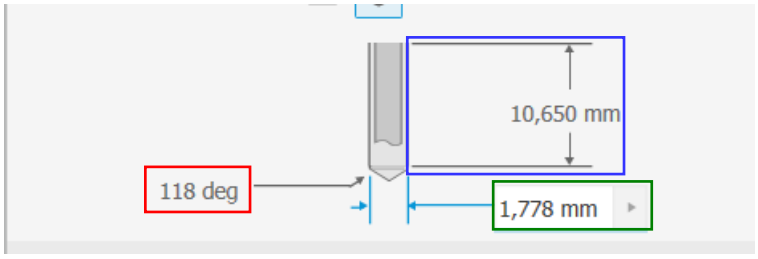


Standard = Select the standard for clearance. (ISO/DIN).

Fastener Type = The types of clearance, from the different standards.

Size = The size of the hole.

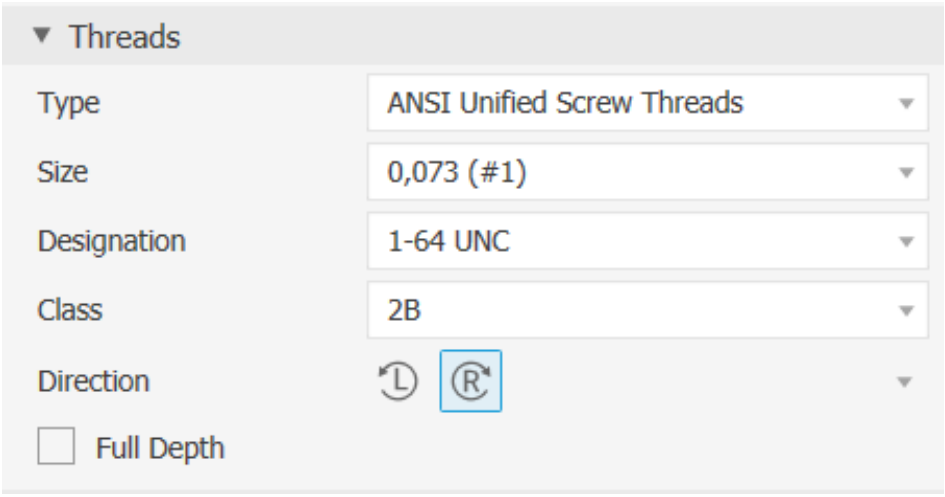
Fit = How bolt/rod and hole fit against each other.



Red = Angle of the tip.
Blue = The depth of the **Hole**.
Green = The diameter of the **Hole**.

Tapped Hole:

Behavior:



Type = Select the standard & type. (ISO/DIN)

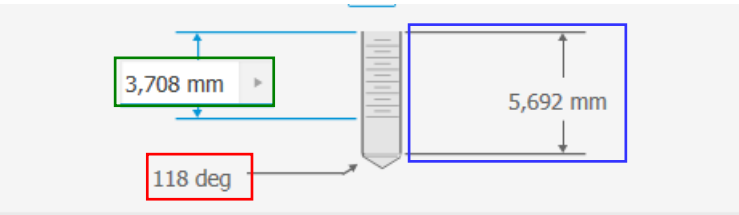
Size = The size of the **Hole**.

Designation = The thread **Pitch**.

Class = Hole class, e.g. 6H (ISO).

Direction = Which way the thread turns.

Full Depth = Thread to tip.

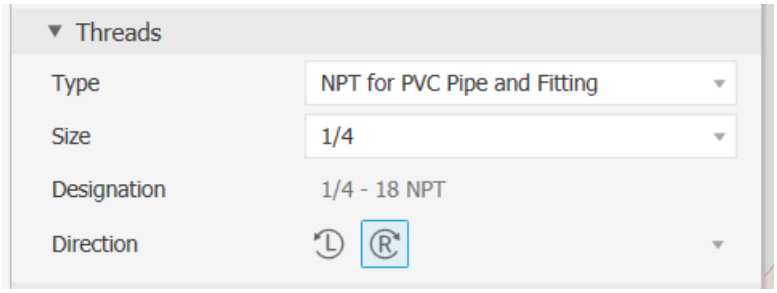


Red = Angle of the tip.
Blue = The depth of the **Hole**.
Green = Thread depth.

Taper Tapped Hole:

Taper = The degree hole grows.

Behavior:

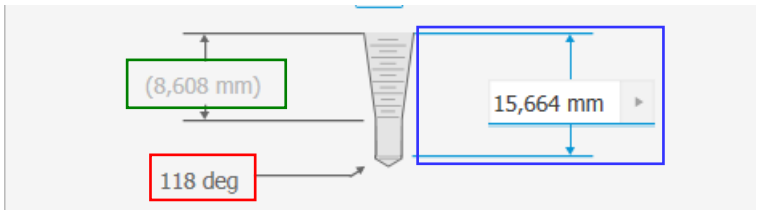


Type = Select the standard & type. (ISO/DIN)

Size = The size of the **Hole**.

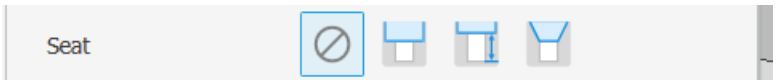
Designation = The thread **Pitch**.

Direction = Which way the thread turns.



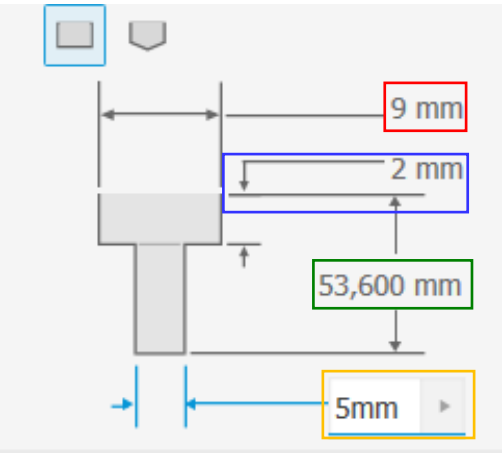
Red = Angle of the tip.
Blue = The depth of the **Hole**.
Green = Thread depth.

Seat



Seat is an indentation on **Holes**, that makes room for the screw heads, so that it lies flush with the surface.

Counterbore



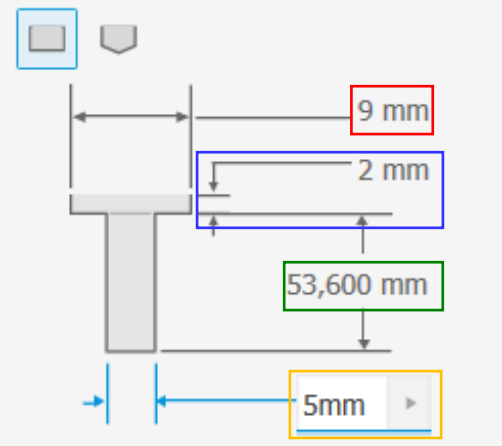
Red = The diameter of the indentation.

Blue = The depth of the indentation.

Green = Total depth.

Yellow = Hole diameter.

Spotface



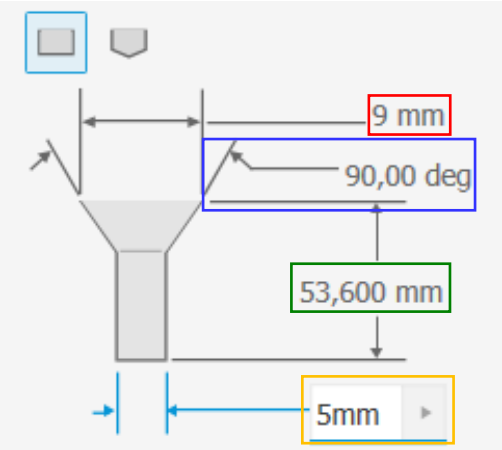
Red = The diameter of the indentation.

Blue = The depth of the indentation.

Green = Dybde fra fordybningen.

Yellow = Hole diameter.

Countersink



Red = The diameter of the indentation.

Blue = Angle of the tip.

Green = Total depth.

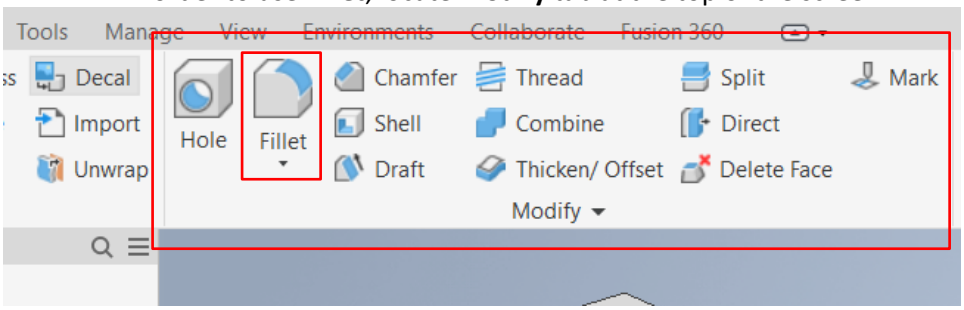
Yellow = Hole diameter.

Fillet

Introduction:

The **Fillet** feature is used to round the edges of an existing shape, with a specified radius.

In order to use **Fillet**, locate **Modify** tab at the top of the screen:



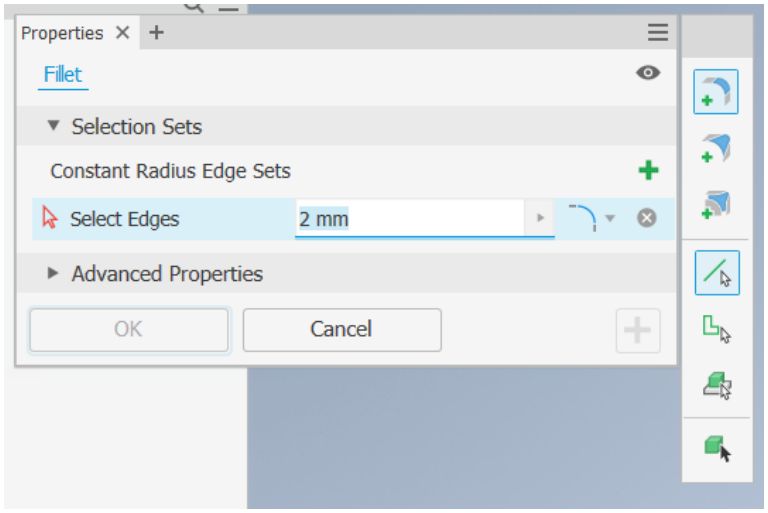
Select **Fillet**. When you press **Fillet** a box will appear.

- **Short key** = F

There are 3 different **Fillets**:

Fillet

This is what **Fillet Properties** looks like:

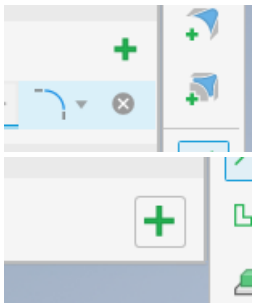


Selection Sets

Tells which **Set** is in use.

Symboler

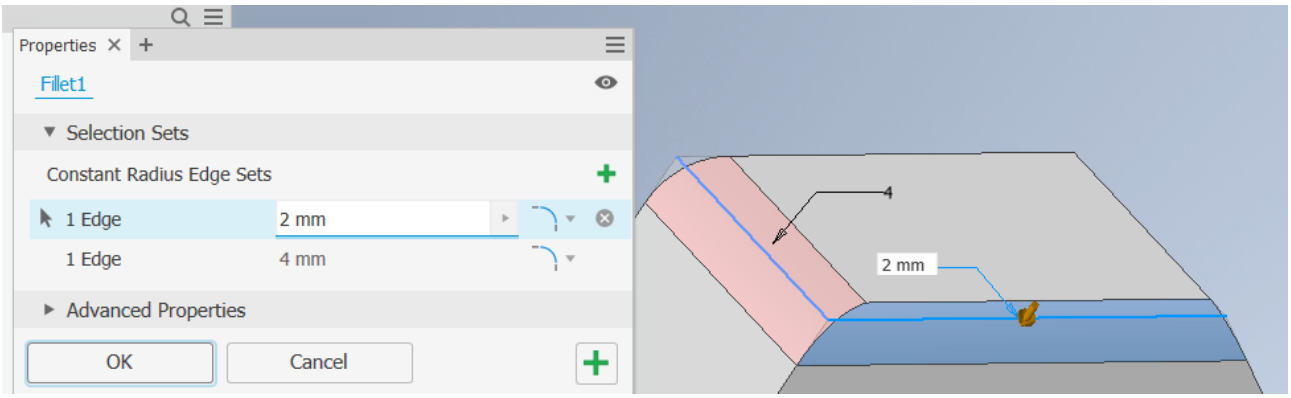
The 2 green +:



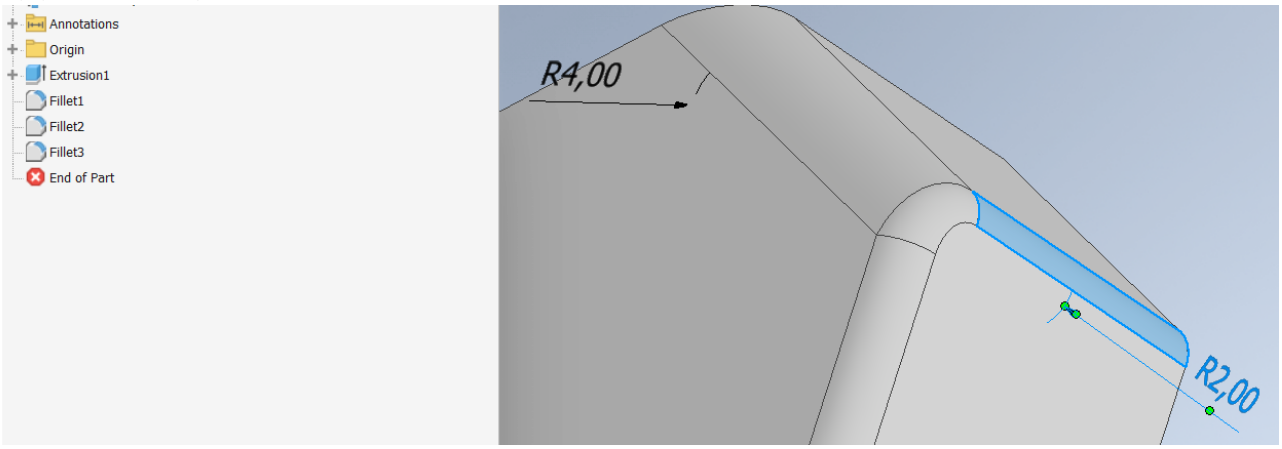
The 2 green + symbols, behave differently than before. The **small** one creates a new **Fillet** "**Select Edges**" box, where you can specify with a different number. The **big** one makes a new **Fillet** feature, like the [others](#).

The difference between the 2 symbols is the relationship they have, e.g. the **small** one will make a soft transition on a corner of 4 mm and 2 mm. The **big** one will first make a rounding and the second time; it will select the entire line with an arc as you can see here:

Small green + example:



Big green + example:



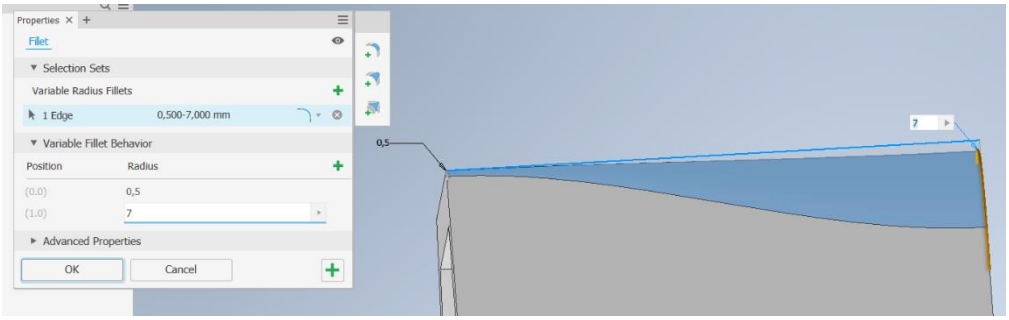
Radii

Constant Radius Edge:

The standard for **Fillets**, it is a completely normal edge radius.

Variable Radius:

A **Fillet** that varies from one end to the other (with this you can only choose edges):

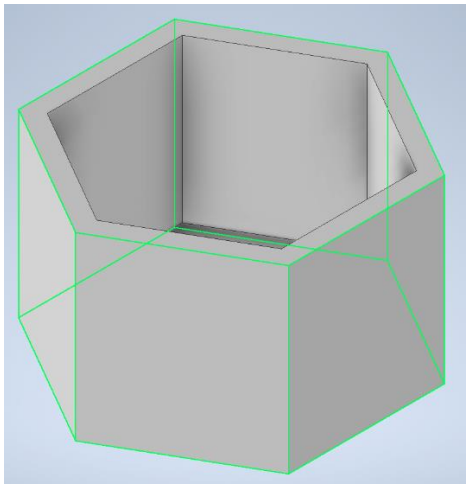


Selection Priority

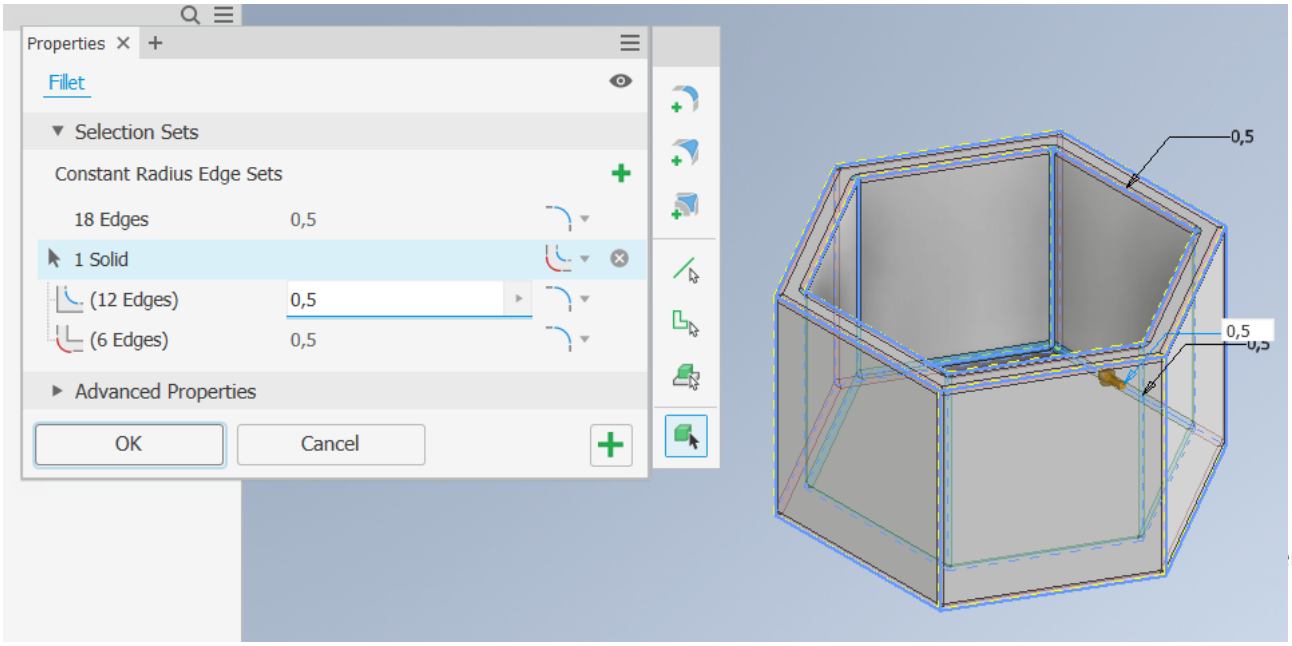
Edge:
Here you can only select edges.

Edge Loops:
Here you can only select edges that are connected on a **Plane**/**Face** (surface).

Features:
Here you can only select one **Feature**. A **Feature** is all edges that connect without being broken:



Associative Fillets:
Here Inventor selects all edges and separates them into outer and inner. It is possible to choose only one of the 2 and to specify the 2 (outer/inner) differently, if desired.



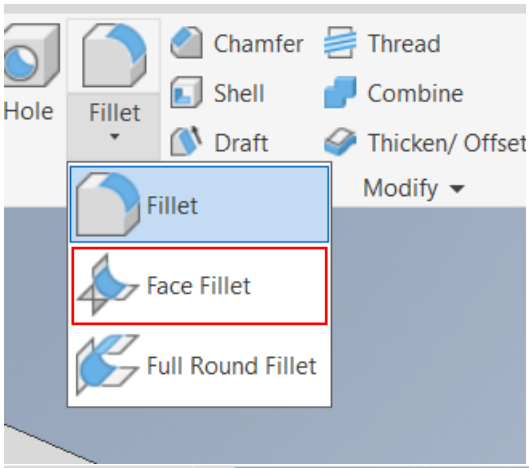
r en

Axis:

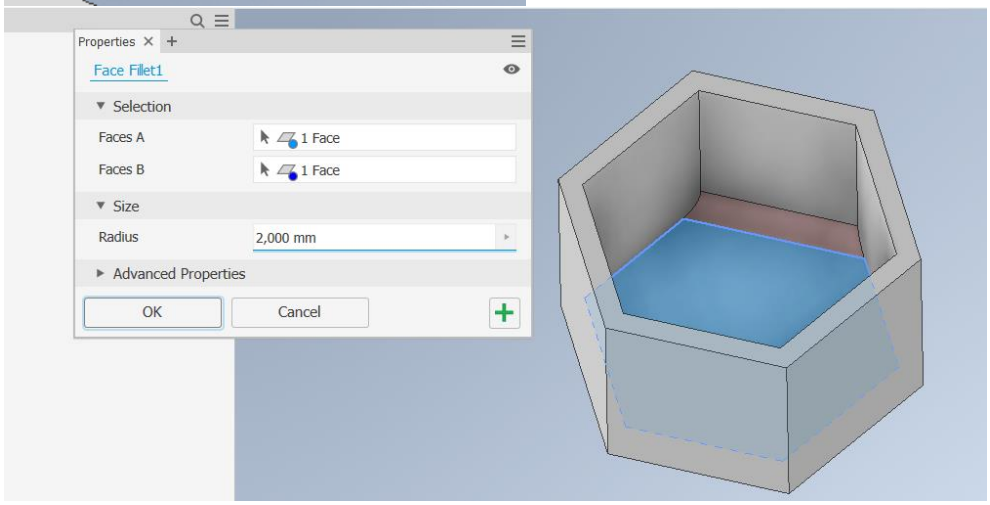
Axis is the one the **Solid** rotates around.

Face Fillet

In order to use **Face Fillet**, locate **Fillet** tab at the top of the screen, and then hover the mouse over it:



Select **Face Fillet**. When you press **Face Fillet** a box will appear:



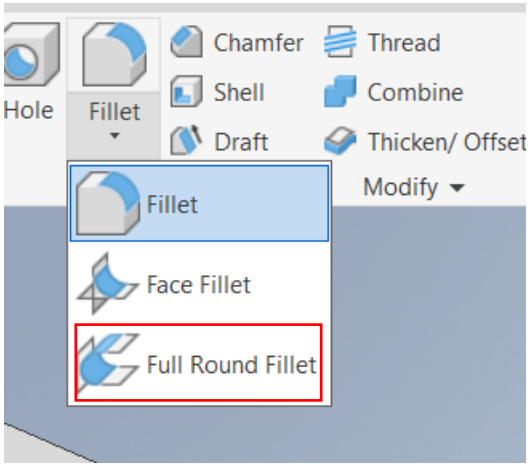
Input Geomerty

Selection:

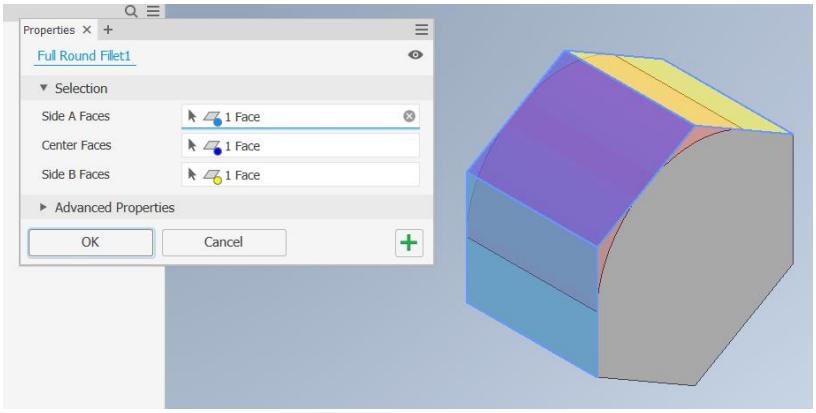
Here you choose 2 surfaces (**Faces**). If you choose more than 2, it's not certain 1 will work as you want.

Full Round Fillet

In order to use **Full Round Fillet**, locate **Fillet** tab at the top of the screen, and then hover the mouse over it:



Select **Full Round Fillet**. When you press **Full Round Fillet** a box will appear:



Selection:

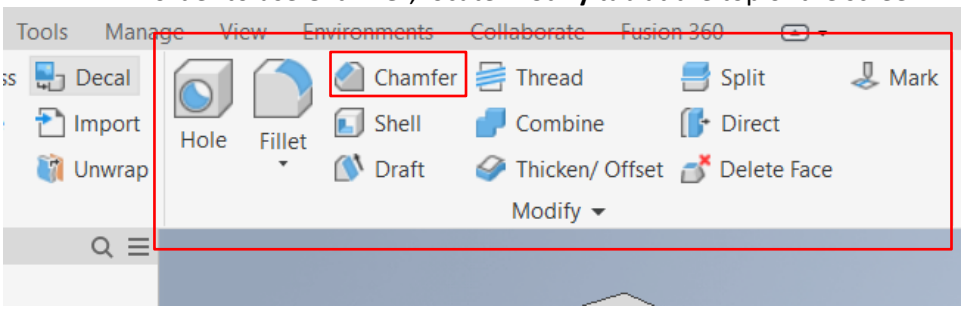
Here you choose 3 surfaces (**Faces**). First one side, then the center side (the rounded surface), and then the last side.

Chamfer

Introduction:

The **Chamfer** feature is used to cut corners of an existing shape at a specified angle.

In order to use **Chamfer**, locate **Modify** tab at the top of the screen:

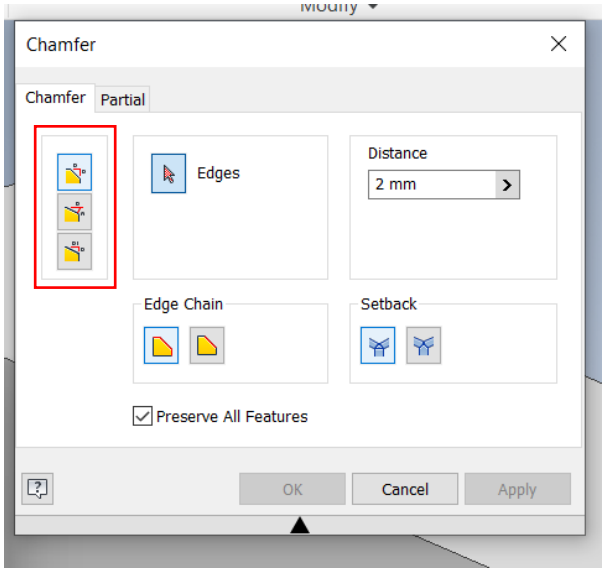


Select **Chamfer**. When you press **Chamfer** a box will appear:

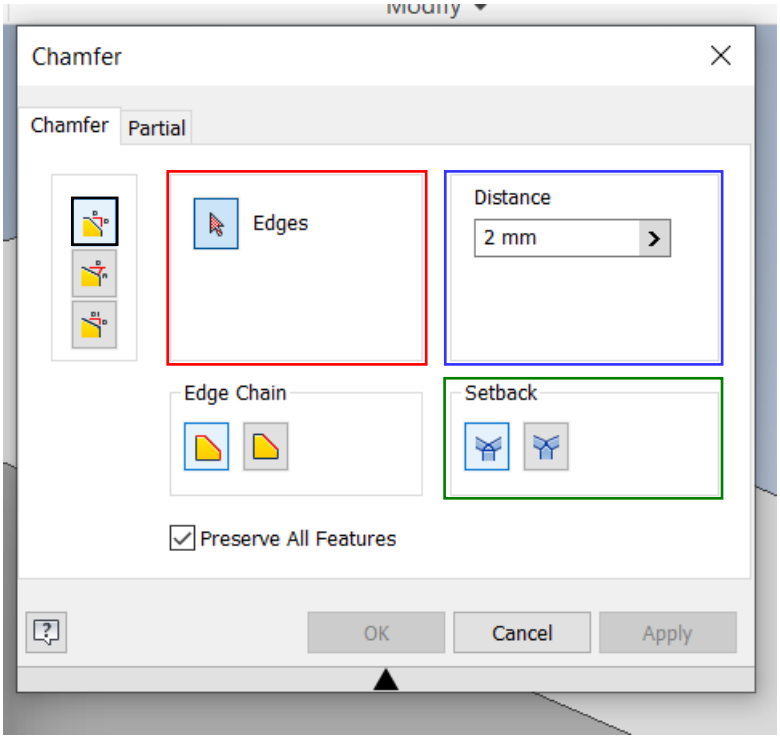
- **Short key** = Ctrl + Shift + K

How to use Properties

This is what **Chamfer Properties** looks like. There are 3 different **Chamfers**:



Distance:

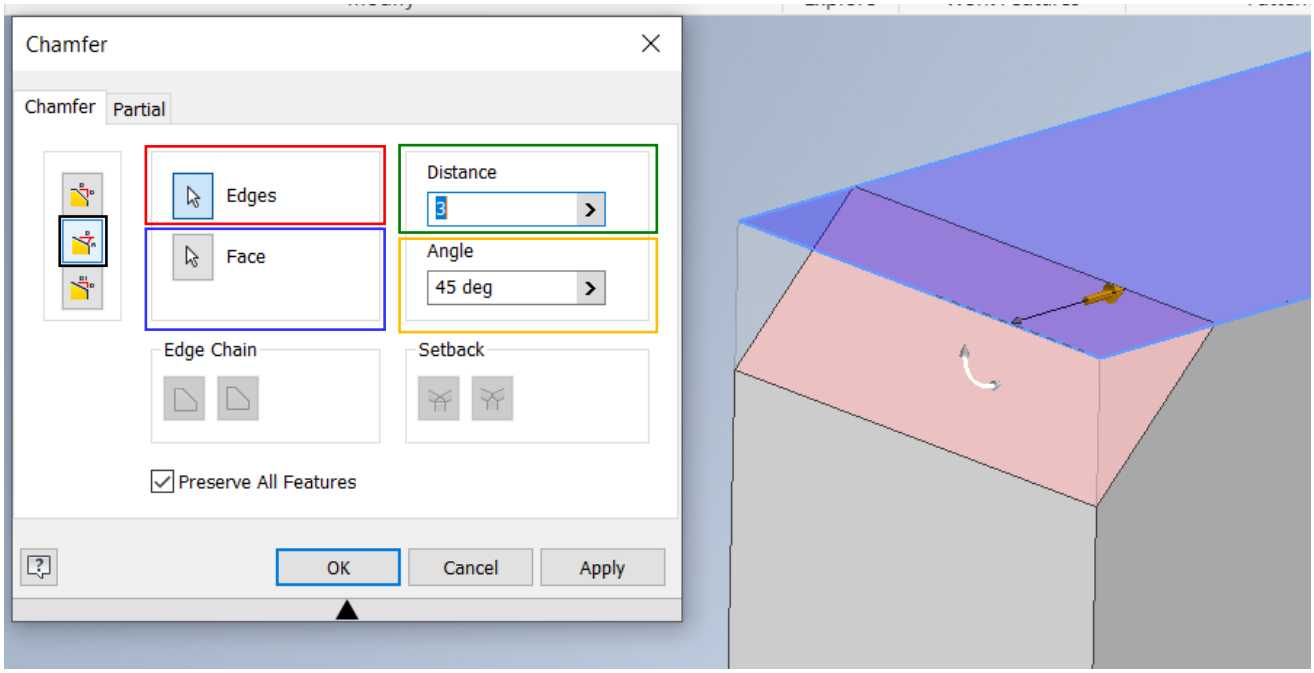


Red = Activates placement of **Edges**.

Blue = Length.

Green = What a **Chamfer** corner looks like.

Distance and Angle:



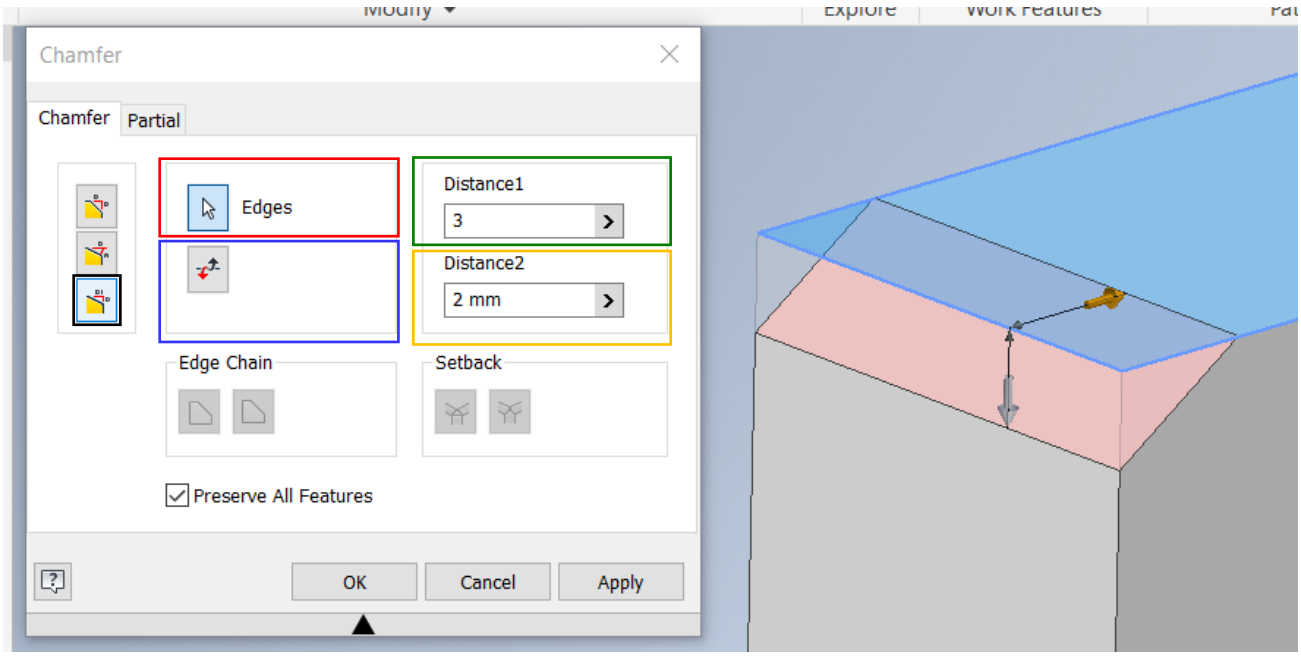
Red = Activates placement of **Edges**.

Blue = Activates placement of a **Face** (surface).

Green = Length.

Yellow = Angle.

Two Distances:



Red = Activates placement of **Edges** (kanter).

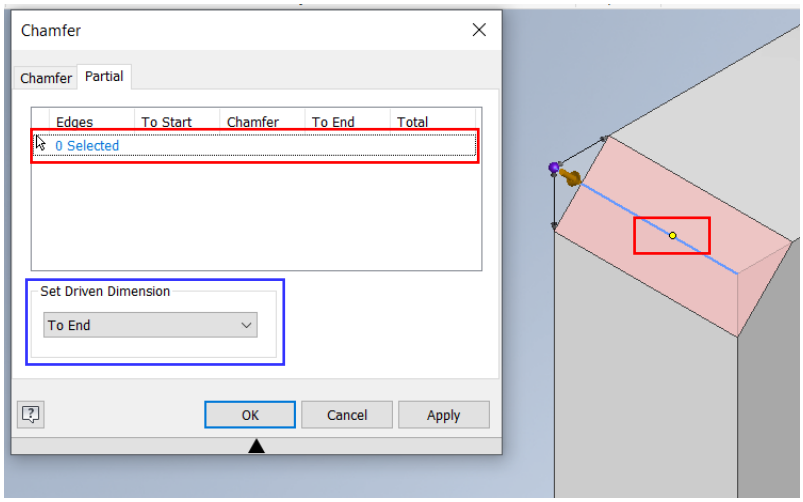
Blue = Flips **Direction 1** & **Direction 2**.

Green = Length for **Direction 1**.

Yellow = Length for **Direction 2**.

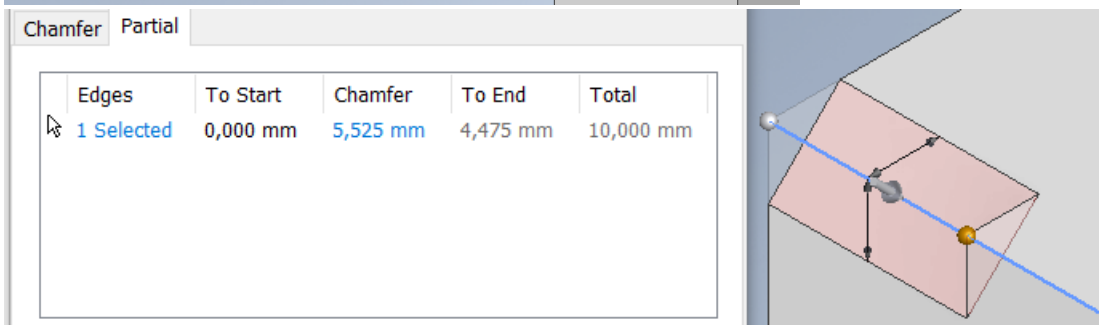
Partial:

Here you can make a half or partial **Chamfer**.



To Activates click on ”**0 Selected**”, This is auto selected by Inventor.

Then click on the edge (blue line), here a yellow dot will appear:



Blue = Here you choose how "**Partial**" should behave:

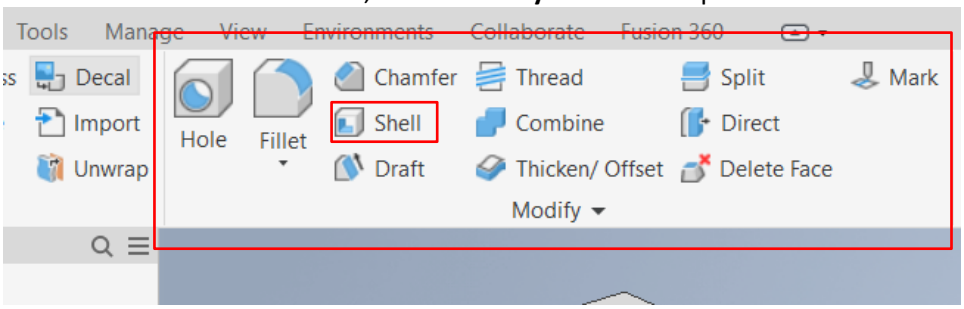
1. **To End =** Here you can change **Start** og **Chamfer** the length.
2. **Chamfer =** Here you can change **Start** og **End** the length.
3. **To Start =** Here you can change **End** og **Chamfer** the length.

Shell

Introduction:

The **Shell** feature is used to make an existing shape hollow or for a shell shape.

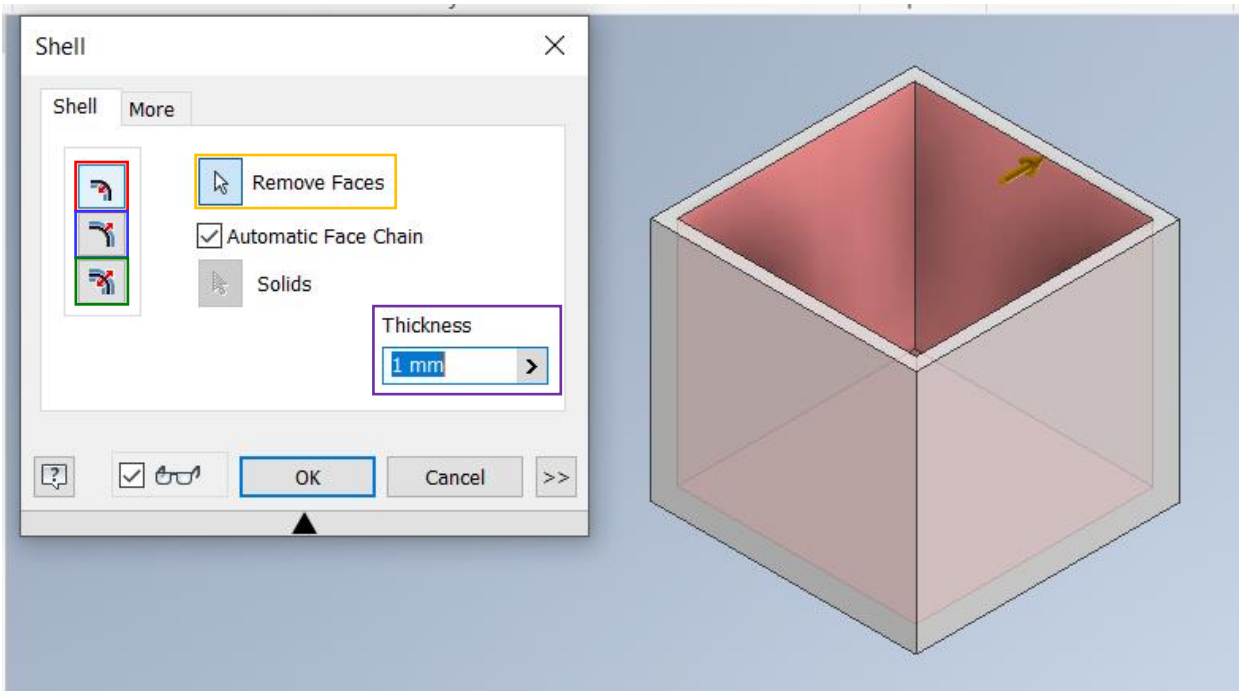
In order to use **Shell**, locate **Modify** tab at the top of the screen:



Select **Shell**. When you press **Shell** a box will appear:

How to use Properties

This is what **Shell Properties** looks like:



Red = Flips the shell shape **towards** the center point.

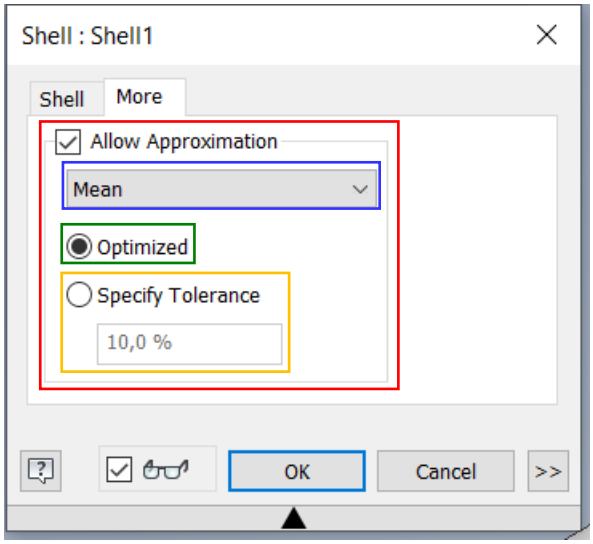
Blue = Flips the shell shape **away from** the center point.

Green = Flips the shell shape **both ways**.

Yellow = Removes the surfaces that open into the cavity.

Purple = The thickness of the shell shape's edges.

More:



Red = Activates placement of **Edges**.

Blue = Flips **Direction 1** & **Direction 2**.

Green = Length for **Direction 1**.

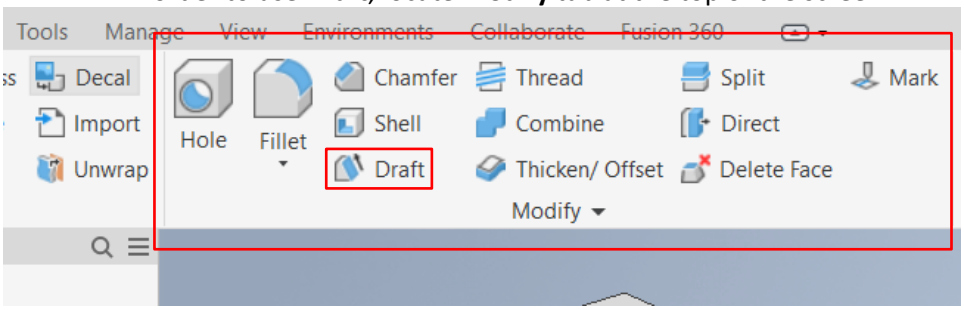
Yellow = Length for **Direction 2**.

Draft

Introduction:

The **Draft** feature is used to make an existing shape with a slope.

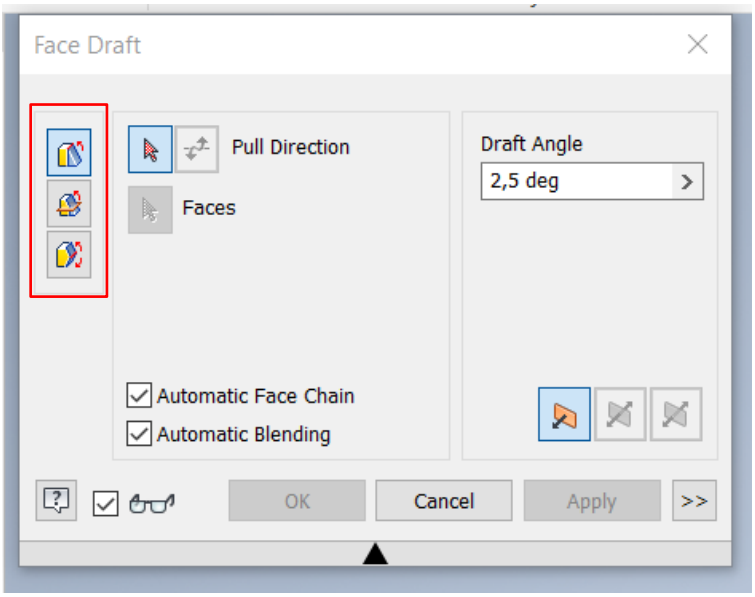
In order to use **Draft**, locate **Modify** tab at the top of the screen:



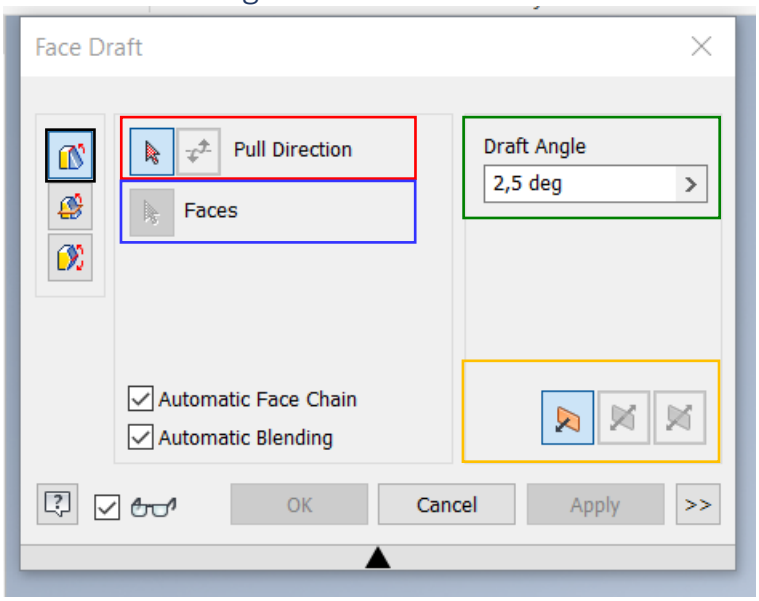
Select **Draft**. When you press **Draft** a box will appear:

How to use Properties

This is what **Draft Properties** looks like. There are 3 different **Drafts**:



Fixed Edge



Red = Activates placement of an **Edge**. You can then flip, the symbol on the right.

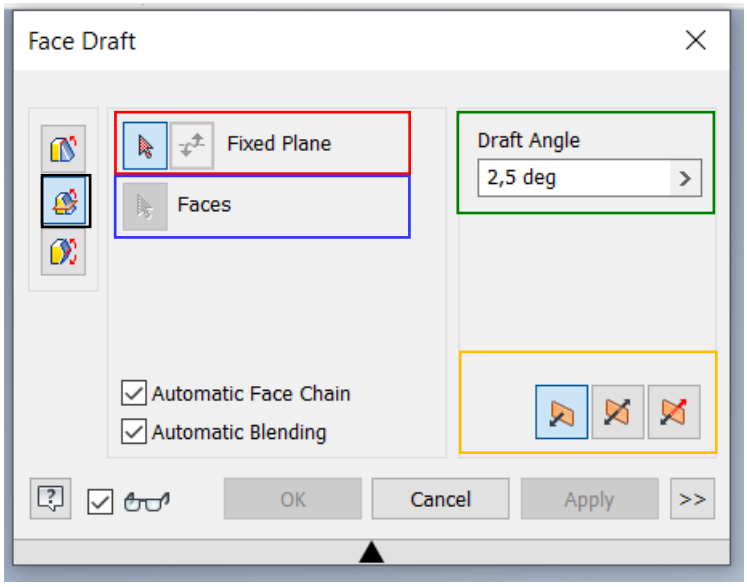
Blue = Activates placement of **Faces** (surfaces).

Green = Angle on **Draft**.

Yellow = Symbol on the left **= One Way**.

Changes the direction of the **Draft**.

Fixed Plane



Red = Activates placement of **Fix Plane**. You can then flip, the symbol on the right.

Blue = Activates placement of **Faces** (surfaces).

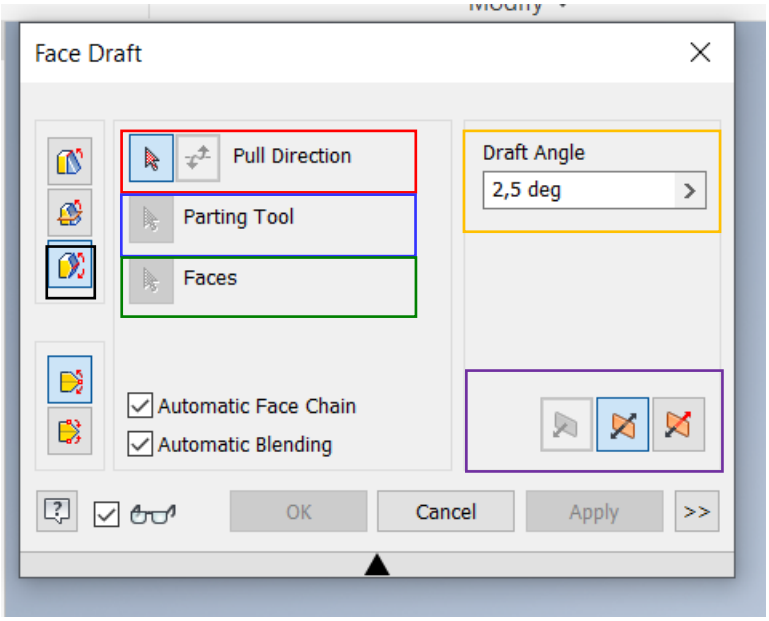
Green = Angle on **Draft**.

Yellow = Symbol in the middle **= Symmetric**.

The **Draft** is both above and below a **Plane**. Use it together **Draft** angle.

Fixed Plane is best to uses with a **Work Plane**.

Parting Line



Red = Activates placement of **Faces** (surfaces). You can then flip, the symbol on the right.

Blue = Activates placement of a **Parting Edge**.

Green = Activates placement of **Faces** (surfaces).

Yellow = Angle on **Draft**.

Purple = Symbol on the right **= Asymmetric**.

This feature works like **Symmetric**.

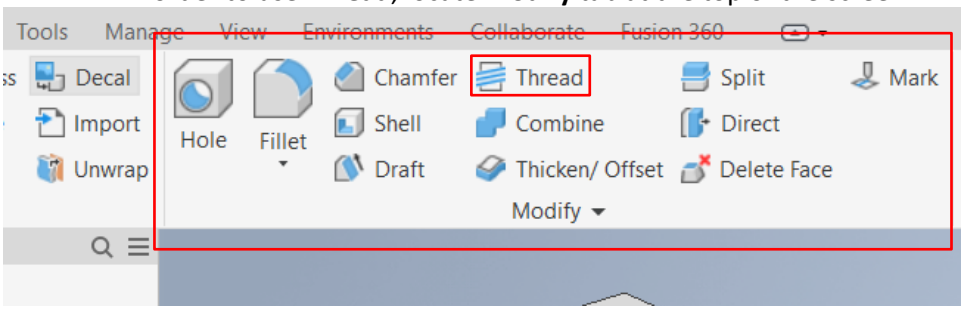
The difference is that you get a 2 **Draft Angles** (**Draft Angle 2**). Here, two different angles can be changed in one's **Draft**.

Thread

Introduction:

The **Thread** feature is used to create a **Thread** on a cylinder shape/forms.

In order to use **Thread**, locate **Modify** tab at the top of the screen:

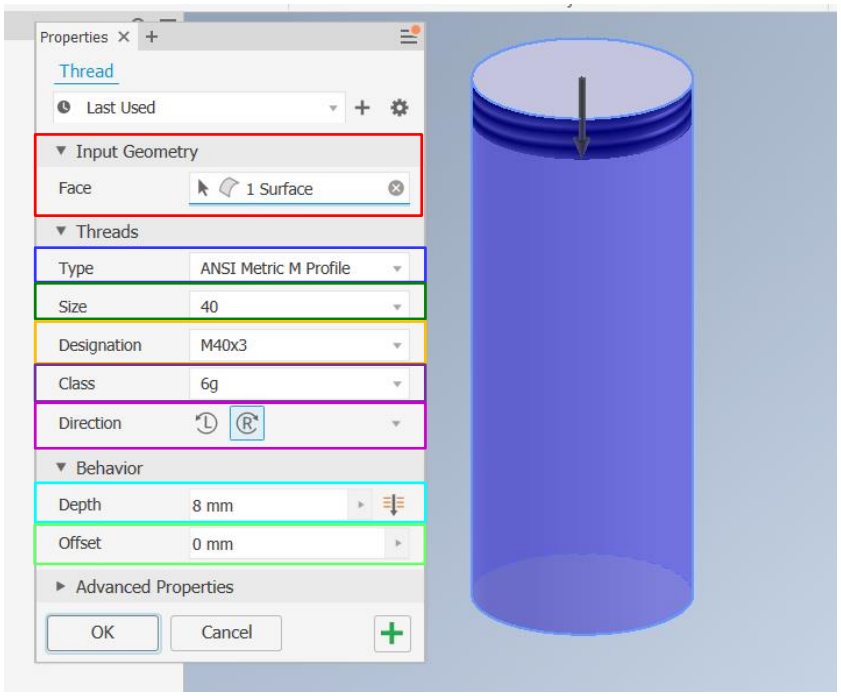


Select **Thread**. When you press **Thread** a box will appear:

Remember *to start by making a cylinder shape*.

How to use Properties

This is what **Thread Properties** looks like:



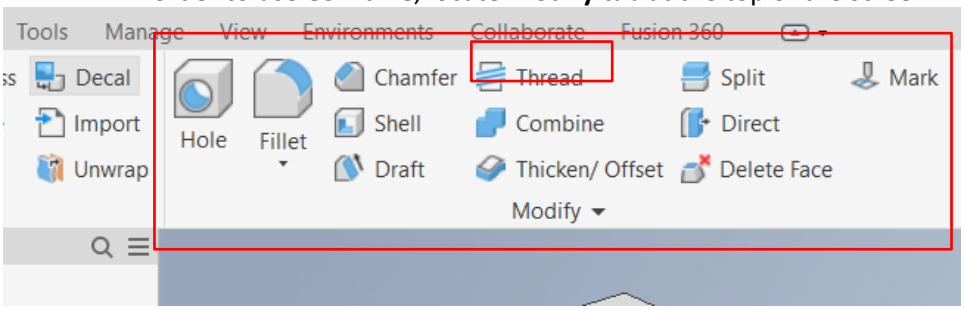
Red = Activates placement of en **Face** (surface).
Type = Select the standard & type. (ISO/DIN)
Size = The size of the **Hole**.
Designation = The thread **Pitch**.
Class = Hole class, e.g. 6H (ISO).
Direction = The way the thread turns.
Cyan = Depth, for **Thread**. Symbol = the hole cylinder.
Lime = **Offset**, for **Thread**.

Combine

Introduction:

The **Combine** feature is used to join 2 **Solids** into 1 **Solid**.

In order to use **Combine**, locate **Modify** tab at the top of the screen:

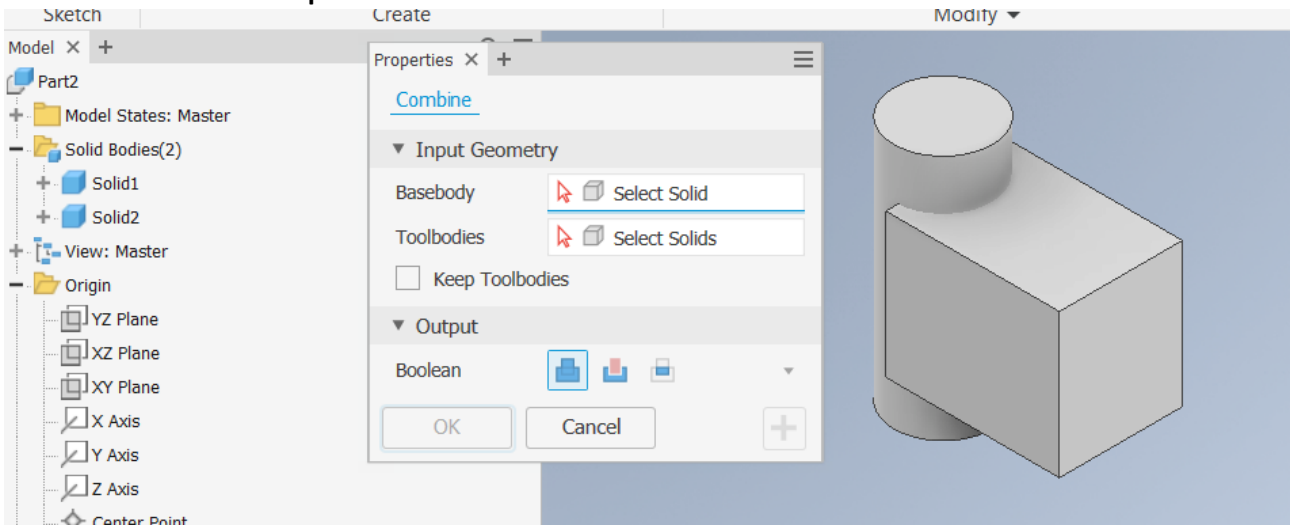


Select **Combine**. When you press **Combine** a box will appear:

***Remember** this only works if you have more than one **Solid**.*

How to use Properties

This is what **Combine Properties** looks like:



Basebody = Activates selection of a base **Solid**.

Toolbodies = Activates selection of a second **Solid**.

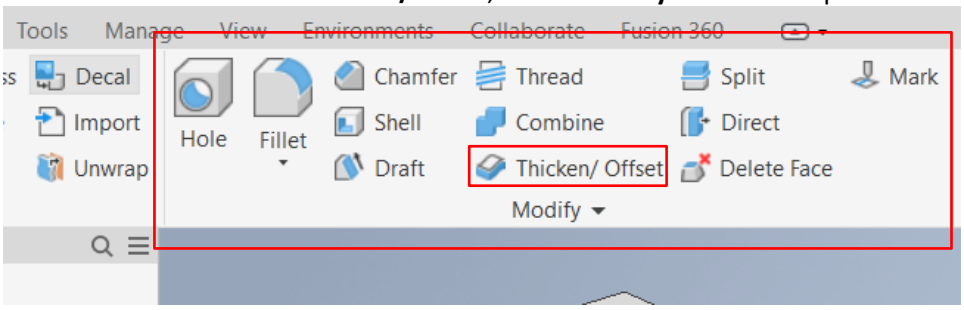
***Reminder**: To choose your **Solids** in the History.*

Thicken / Offset

Introduction:

The **Thicken/Offset** feature is used to make a desired **Solid** thicker or add an **Offset**.

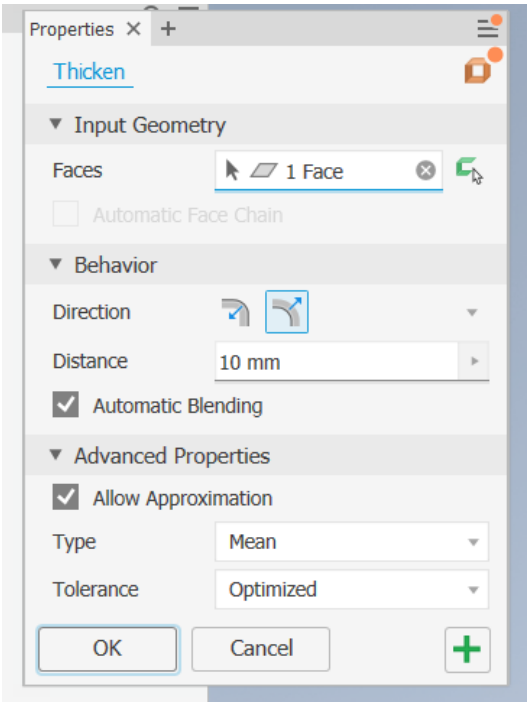
In order to use **Thicken/Offset**, locate **Modify** tab at the top of the screen:



Select **Thicken/Offset**. When you press **Thicken/Offset** a box will appear:

How to use Properties

This is what **Thicken/Offset Properties** looks like:



Input Geometry

Profiles:

Works the same way as **Hole → Input Geometry → Positions**. Here, Inventor has not selected a surface, so you can do that by ensuring that "**Profile**" is active and clicking on the surface, you would like to use. In the example, we choose the small square.

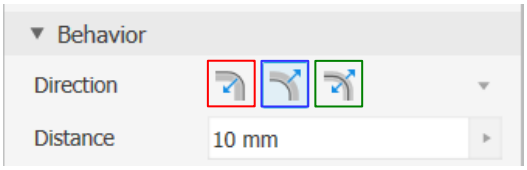
Behavior

Direction:

Red = Flips the shell shape **towards** the center point.

Blue = Flips the shell shape **away from** the center point.

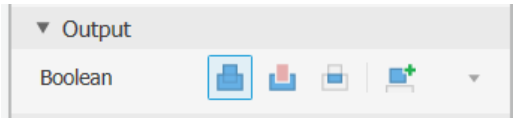
Green = Flips the shell shape **both ways**.



Distance:

Thickness for **Thicken**.

Output



Boolean:

1. *First (from the left):*
   - **Join**. Combines the 2 into one **Solid**.
2. *Second:*
   - **Cut**. The new feature is removed from the existing one, to a hollow **Solid**.
3. *Third:*
   - **Intersect**. The existing feature is deleted. And the new feature becomes a **Solid**.
4. *Last:*
   - **New Solid**. It will make the new feature into a new **Solid**, so there are e.g. 2.

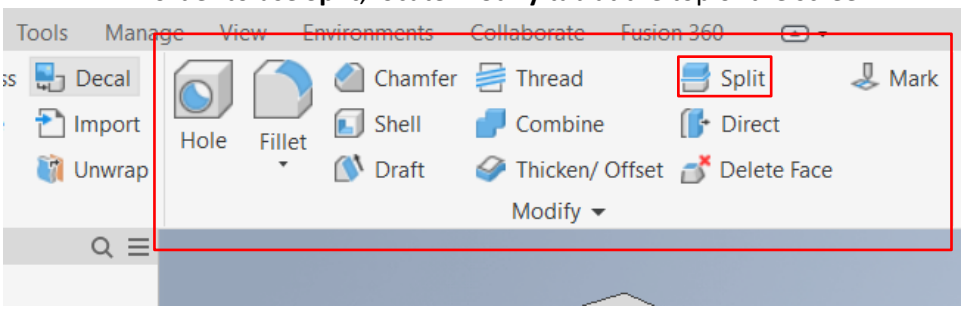
- **Body Name** = Renames **Solid**.

Split

Introduction:

The **Split** feature is used to split 1 **Solid** into 2 **Solids**.

In order to use **Split**, locate **Modify** tab at the top of the screen:

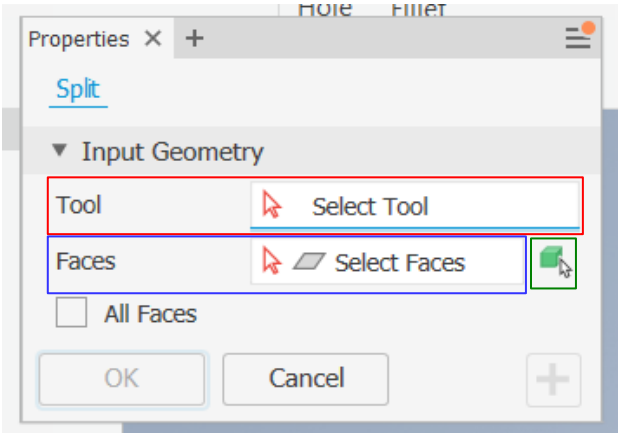


Select **Split**. When you press **Split** a box will appear:

***Remember**, here you must use a **Sketch/Plane** before you can use the feature*.

How to use Properties

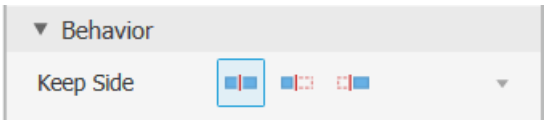
This is what **Split Properties** looks like:



Red = Activates selection of a **Plane** or **Sketch**.

Blue = Activates selection of **Faces** (surfaces), or **Solids**.

Green = Changes the blue sides to **Solids**, hereafter:



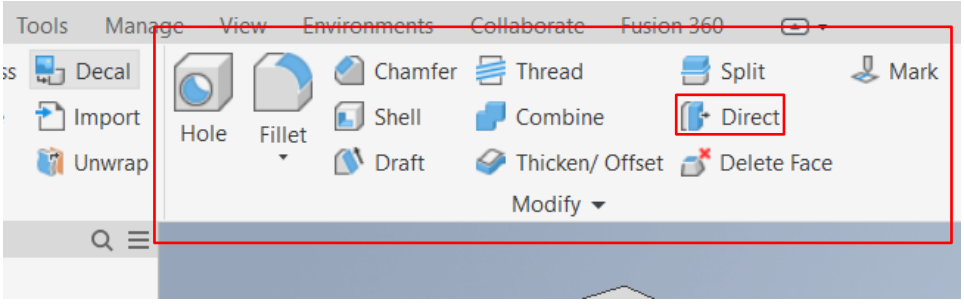
You can select the desired page(s) to save.

Direct

Introduction:

The **Direct** feature is used to move, resize, scale, rotate or delete in 3D. In other words, it's used to manipulate in a 3D space.

In order to use **Direct**, locate **Modify** tab at the top of the screen:

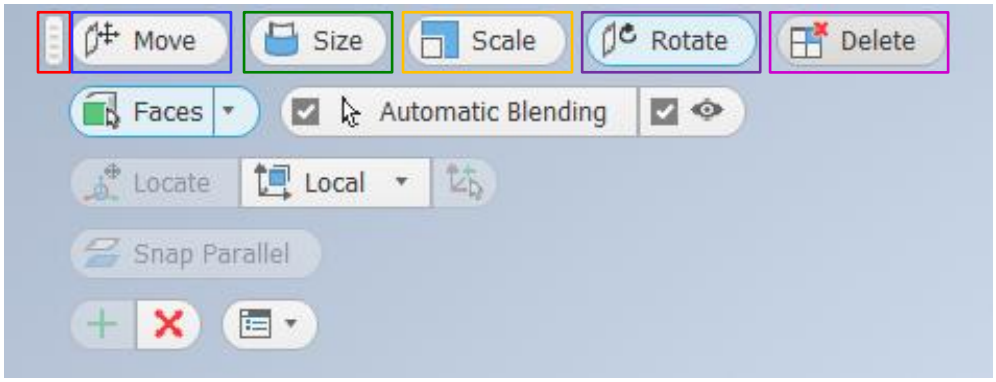


Select **Direct**. When you press **Direct** a box will appear:

***Strongly recommend** that if you have to use **MOVE** and **ROTATE**, that you <u>use</u> **SOLID** and <u>not</u> **Faces**.*

How to use Properties

This is what **Direct Properties** looks like:



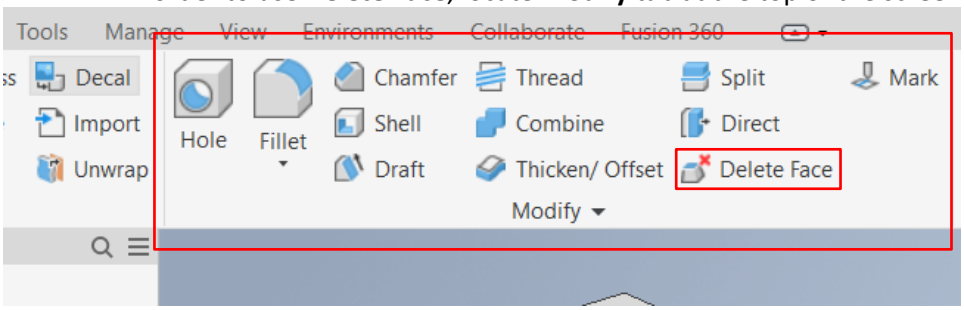
Red = Moves the **Direct Properties**.
Move = Moves **Solids** (and **Faces**, but doesn't work). So, use **Solids**.
Size = Adjusts the size of **Faces** (surfaces).
Scale = Scales the **Solids** to a specified size.
Rotate = Rotates both **Faces** and **Solids**, recommend using **Solids**. Since it has a better relationship.
Delete = Deletes surfaces. Don't use **Delete**, use **Delete Face** instead (The separate feature).

Delete Face

Introduction:

The **Delete Face** feature is used to delete surfaces (**Faces**).

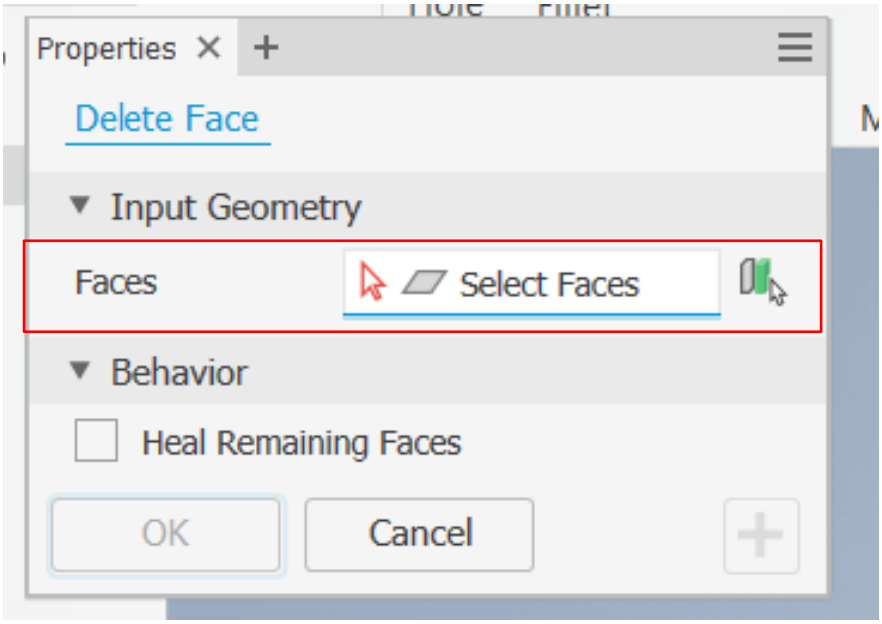
In order to use **Delete Face**, locate **Modify** tab at the top of the screen:



Select **Delete Face**. When you press **Delete Face** a box will appear:

How to use Properties

This is what **Delete Face Properties** looks like:



Red = Activates placement of **Faces** (surfaces).

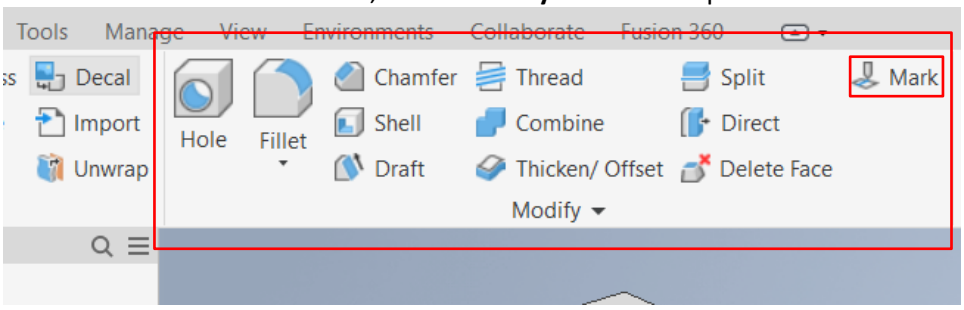
Mark

Introduction:

The feature is only available in version 2023 of Inventor.

The **Mark** feature is used to make markings for e.g. laser engraving with a laser cutter.

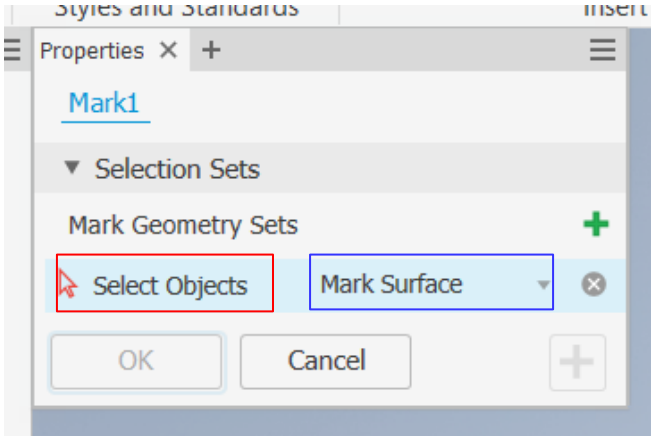
In order to use **Mark**, locate **Modify** tab at the top of the screen:



Select **Mark**. When you press **Mark** a box will appear.

How to use Properties

Sådan kommer man ind på **Mark Properties**:



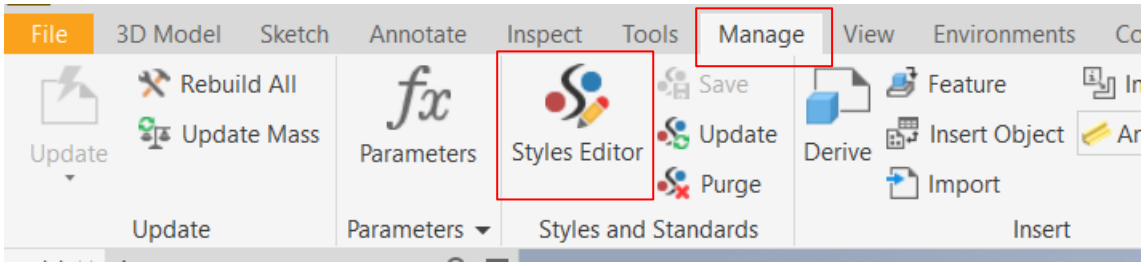
Red = Activates selection of a **Sketch**.

Blue = List/selection of **Mark** style.

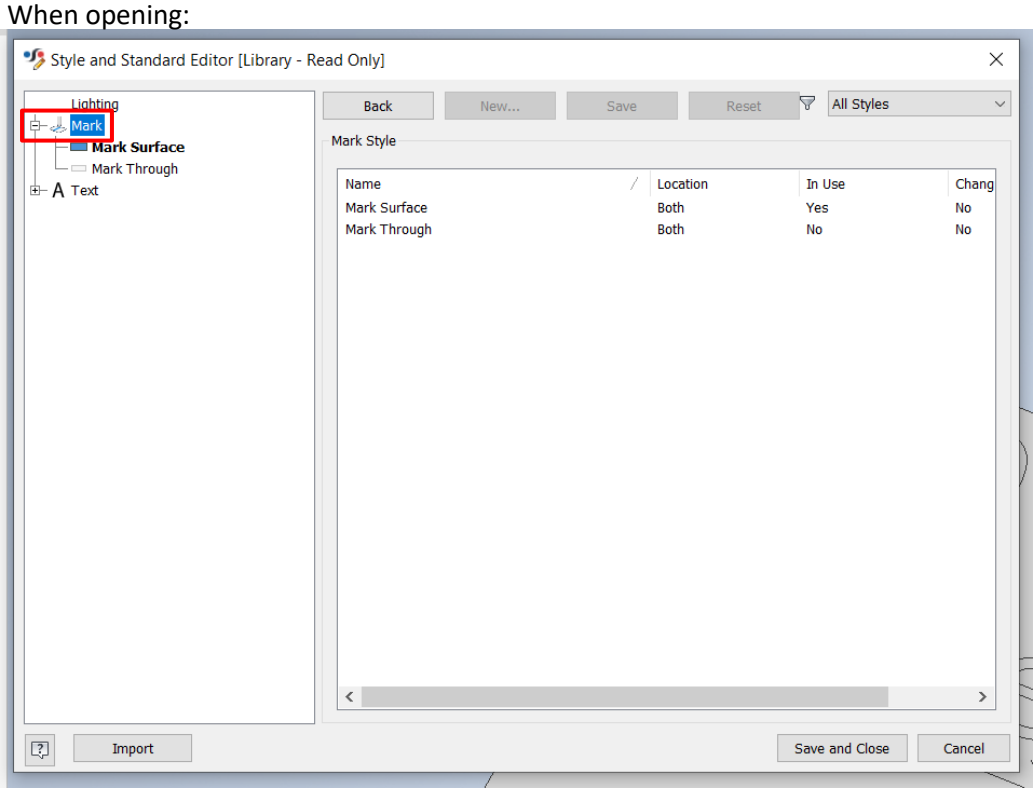
Styles Editor

Here you can make more/change the **Mark** styles.

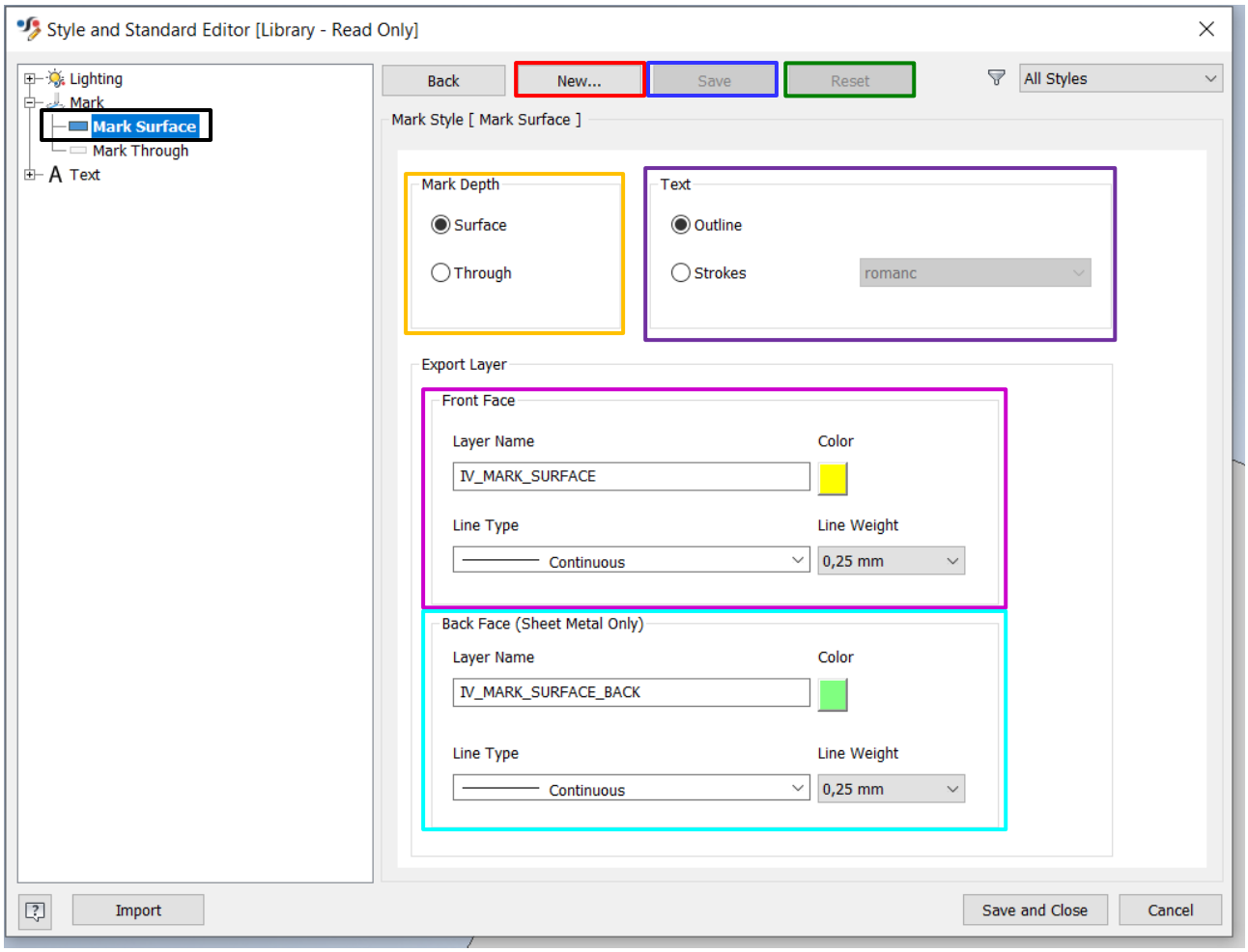
This is how to access the Styles Editor.



When opening:



How to use?



Red = Makes a ny **Mark** style.
Blue = Saves the **Mark** style.
Green = Resets the **Mark** style.
Yellow = The choice between, surface depth or full depth (Goes through the entire subject).
Purple = The choice between, the outermost lines of **Text** from the **Sketch**. Or choosing what kind of font it should be.
Magenta = Which line type, color and weight the **Export** layer has.
Cyan = Same as ↑, but on the reverse side and indents only with **Sheet Metal**.

Template – Drawings

Introduction

After you have finished with either a **Part** file, **Sheet Metal**, **Assembly** or **Weldment**, you can create a drawing file, so e.g. an Industrial Technician can make the item on their machines. You can make your own drawing template or use your company's. If you make your own **Template**, it is recommended to use the **Default** project folder, as Inventor saves better here.

There are 2 types of drawing files:

- DWG = Used for **Part** file and **Assembly**.
- IDW = Used for **Sheet Metal** and **Weldment**.

Index:

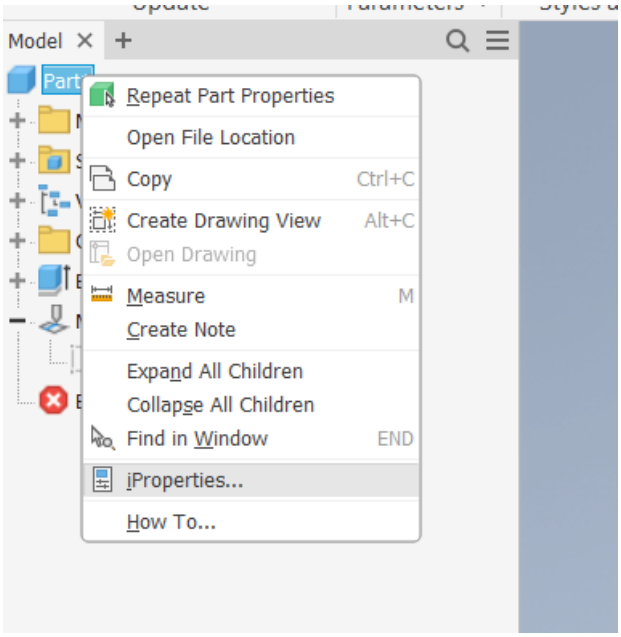
iProperties:

Standard iProperties is Inventor's way of using **Part** values. So, for example you can have a **Material** value engraved in the **Part**. If you change **Material** in the **Part**, the text is also changed. (They are related).

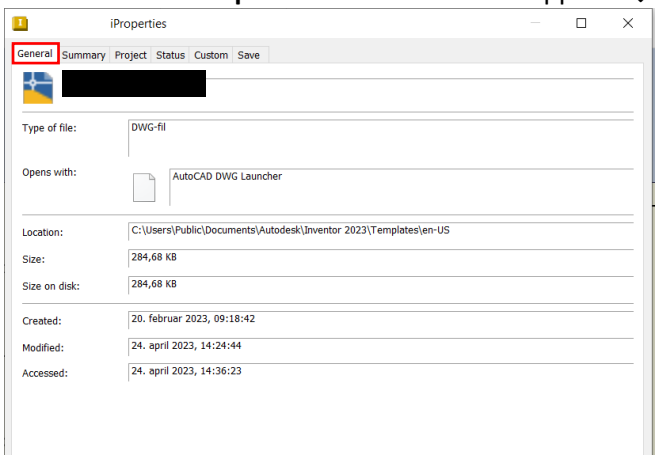
How to change one's iProperties:

You can always add, a **Part** number or similar to your **Part**, by opening **iProperties** and typing it in there.

Open the desired **Part** and right-click on "**Part**" (The name may be different) in the history: →

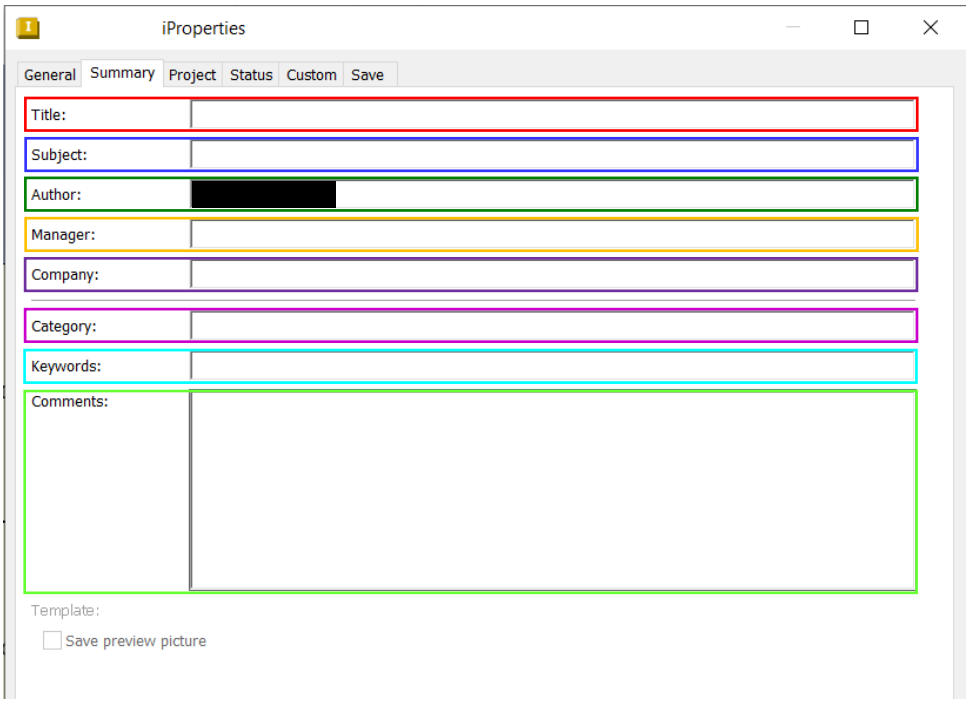


Then click on "**iProperties…**" and a box will appear: ↓



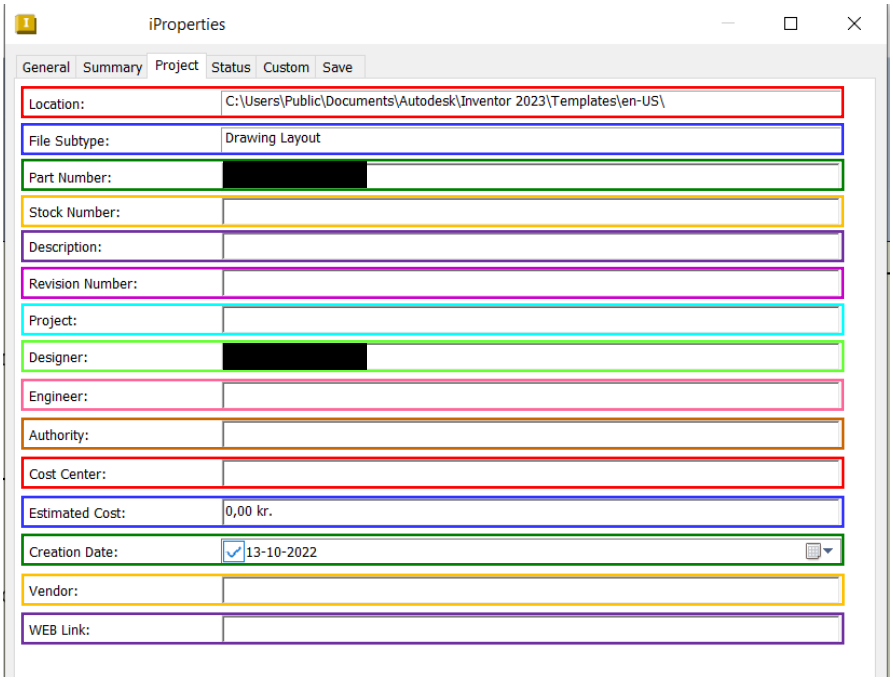
Under **General**, you cannot change anything, as this is an overview.

Summary



Red = Title / heading.
Blue = Subject.
Green = Author.
Yellow = Manager.
Purple = Company.
Magenta = Category.
Cyan = Keywords.
Lime = Comments.

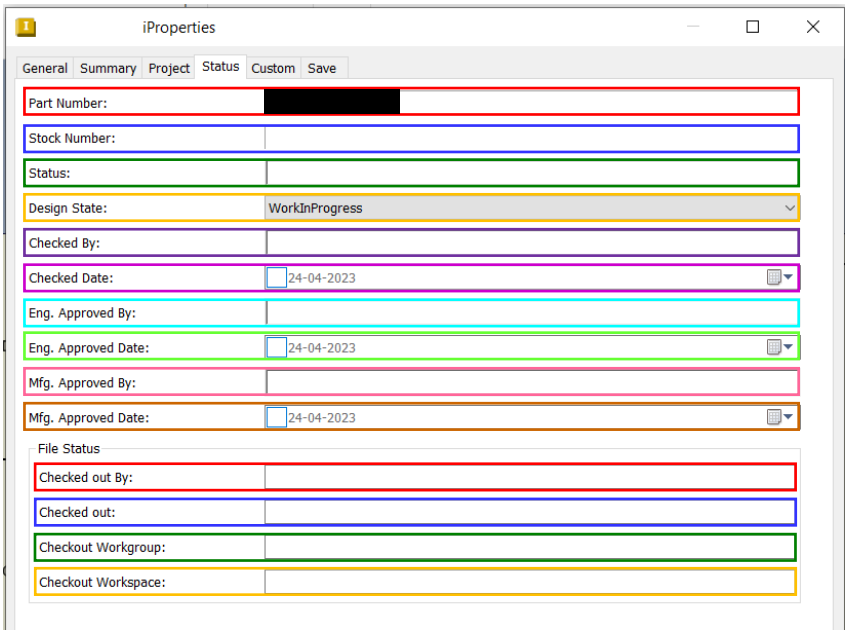
Project



Red = File location.
Blue = File subtype.
Green = Part number.
Yellow = Stock number.
Purple = Description.
Magenta = Revision umber.
Cyan = Project.
Lime = Designer.
Pink = Engineer.
Brown = Authority.

Red 2 = Cost center.
Blue 2 = Estimated cost.
Green 2 = Creation date.
Yellow 2 = Vendor.
Purple 2 = URL (Web link).

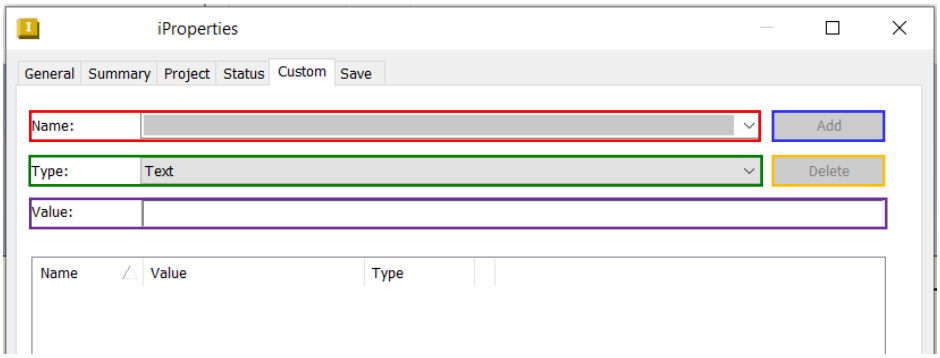
Status



Red = Part number.
Blue = Stock number.
Green = Status.
Yellow = Design state.
Purple = Checked by.
Magenta = Checked date.
Cyan = Eng. approved by.
Lime = Eng. approved date.
Pink = Mfg. approved by.
Brown = Mfg. approved date.

Red 2 = Checked out by.
Blue 2 = Checked out.
Green 2 = Checkout workgroup.
Yellow 2 = Checkout workspace.

Custom



Red = Name.
Blue = Add.
Green = Type.
Yellow = Delete.
Purple = Value.

Save

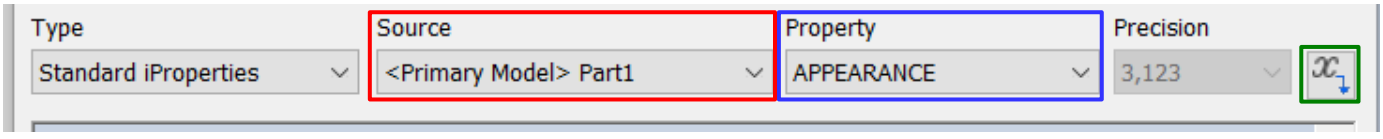
Red = Save preview picture.
Blue = Active component Iso view on save.
Green = Active window on save.
Yellow = Active window.
Purple = Import from file.
Magenta = Takes a pictures / saves.
Cyan = Imports.

Text:

Under **Text** you can insert **Standard iProperties**.

Property example = **Designer**/**Part Number**/**Material**.

In order to use, select "**Standard iProperties**" under **Type**:



Source = Which **Part** Inventor takes **Property** from.
Property = Which **Property** to insert.
Green = To add, select the ↑ 2 and then press here.

What is Trailing Zeros and Leading Zeros:

Trailing Zeros adds the last 0 from decimal numbers, so if you have 3.123 (Precision) selected:

- **With** = 1,200.
- **Without** = 1,2 (even if 3,123 (Precision) is selected).

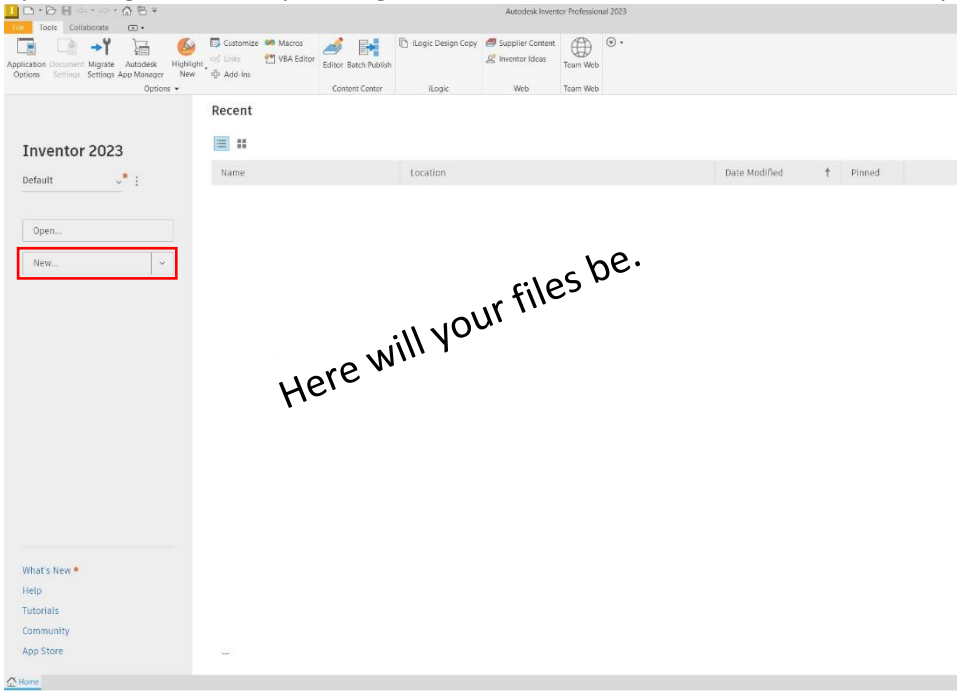
It's recommended that you don't have **Trailing Zeros** on - but with **ISO** it's on by default.

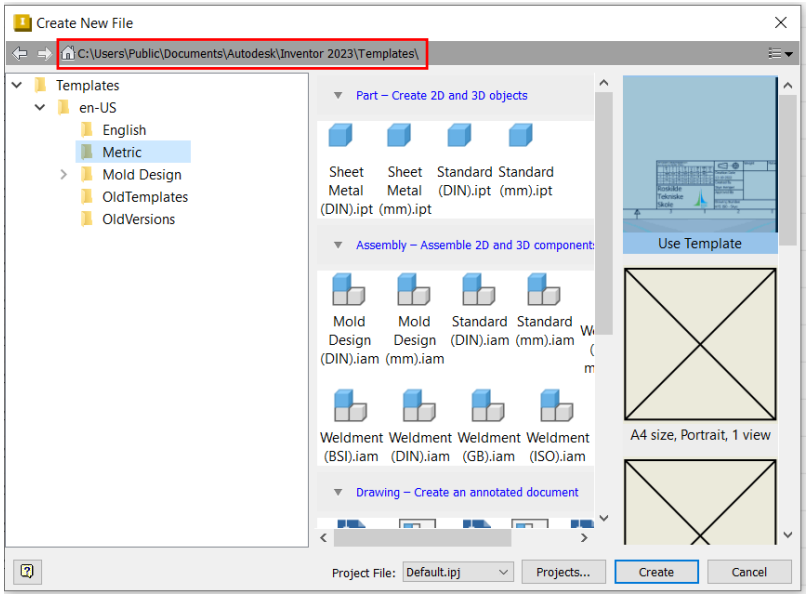
Leading Zeros adds the first 0 from the decimal number, so if you have 3.123 (Precision) selected:

- **With** = 0,123
- **Without** = ,123 (even if 3,123 (Precision) is selected).

Insert Template

To insert a prefabricated **Template**, you have to look for a certain folder and put your **Template** there. Start by creating a new file by clicking on "**New...**", which can be found on the front page:





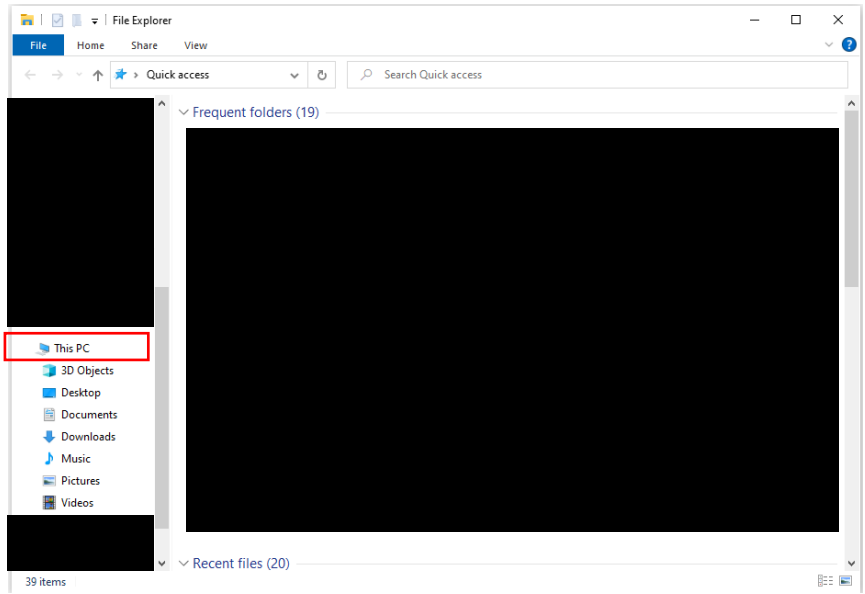
Afterwards, this box appears. These are the folders where **Templates** are located. Here you can e.g. select "**en-US**" → "**Metric**" here there are standard **Templates** for the **Metric** system. Here you can only see an overview of the folders and **Templates** that exists. You cannot change, add, or remove. Up at the top you can see where the folders are located in **Windows**. To find the folder, follow:

Brief:

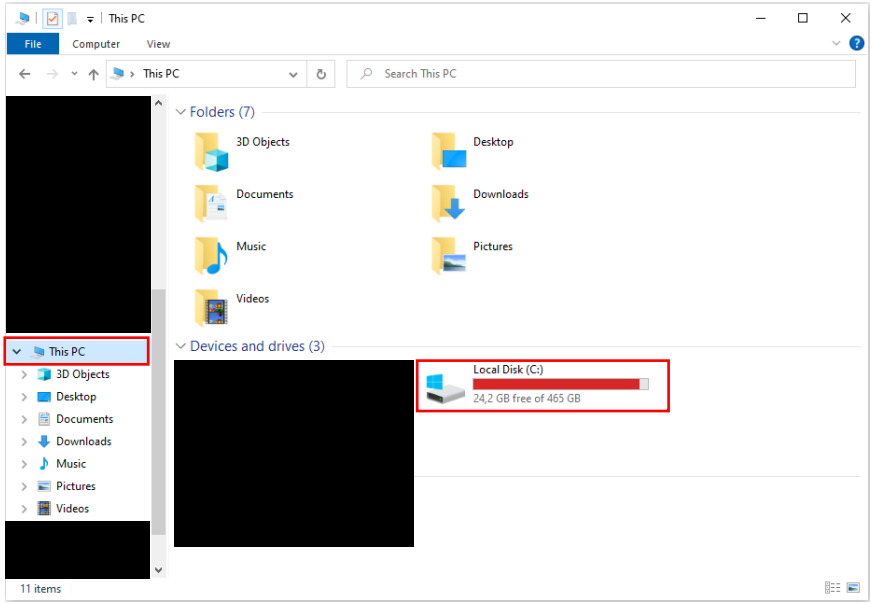
1. Open a pathfinder.
2. Click on **"This PC"**. *(Denne pc)*
3. Click on **"Local Disk"**. *(Lokal disk)*
4. Click on **"Users"**. *(Brugere)*
5. Click on **"Public"**. *(Delte Filer)*
6. Click on **"Public Documents"**. *(Delte Dokumenter)*
7. Click on **"Autodesk"**.
8. Click on **"Inventor XXXX"**. (XXXX = version)
9. Click on "**Template**".
10. Here are all the folders and **Templates** that exists, when you press "**New...**".

Throughout:

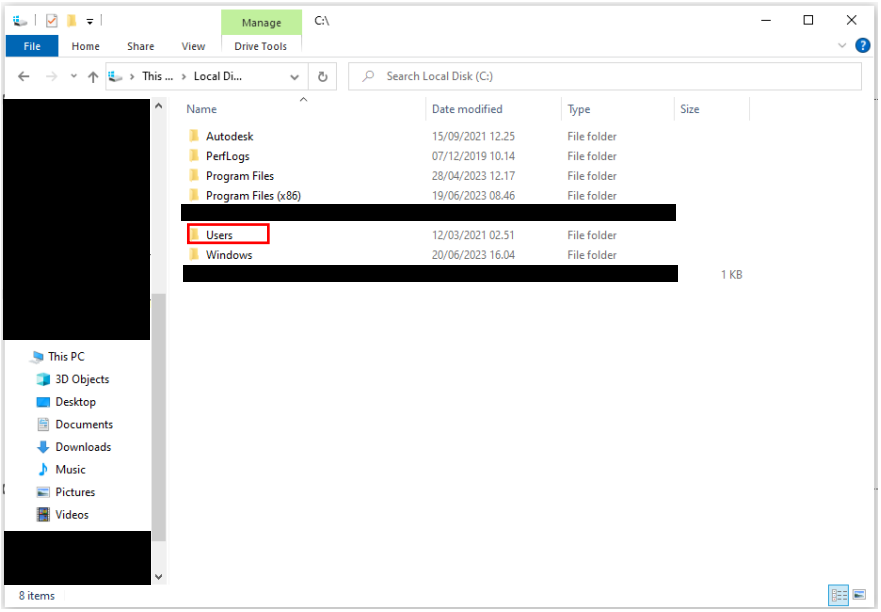
1. Open a pathfinder and click on "**This PC**".



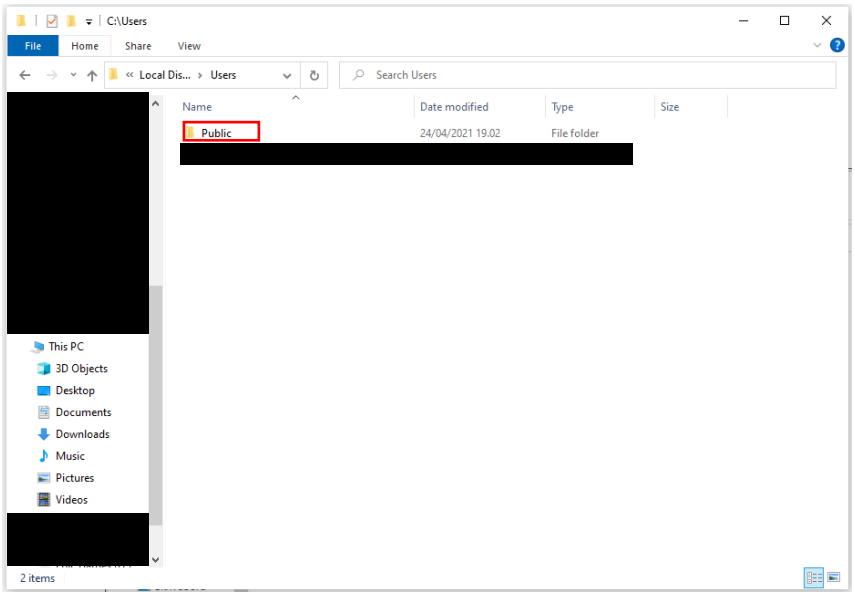
1. Click on “**Local Disk (C:)**”.



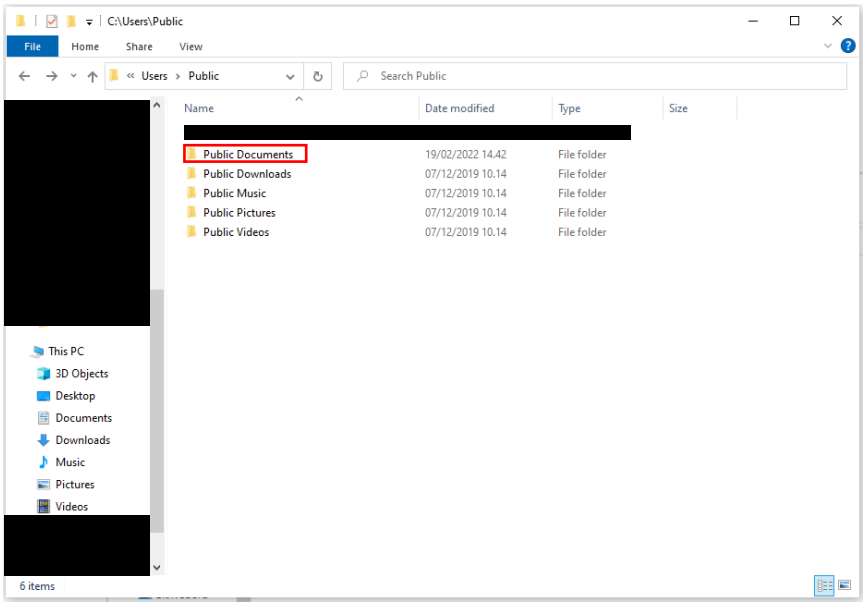
2. Click on “**Users**”.



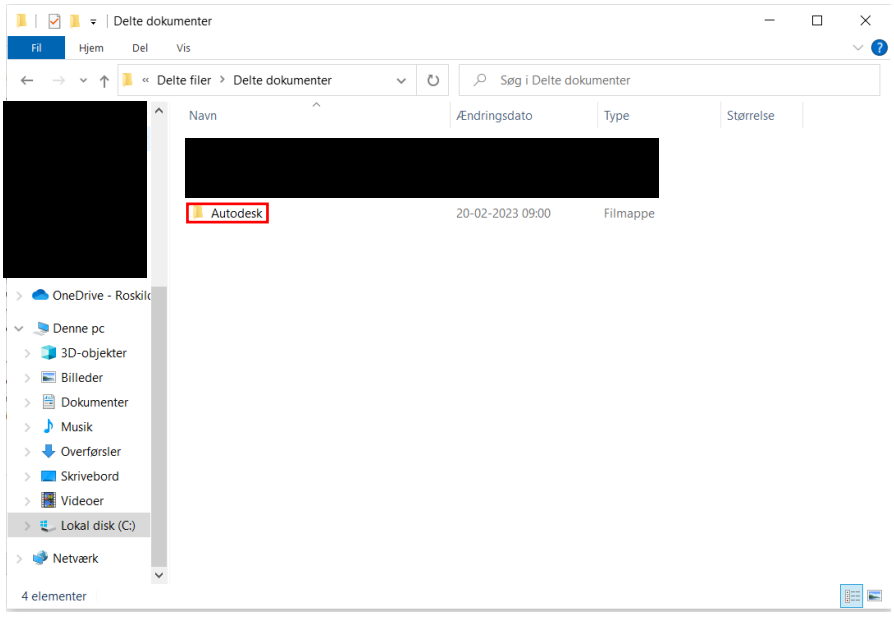
3. Click on “**Public**”.



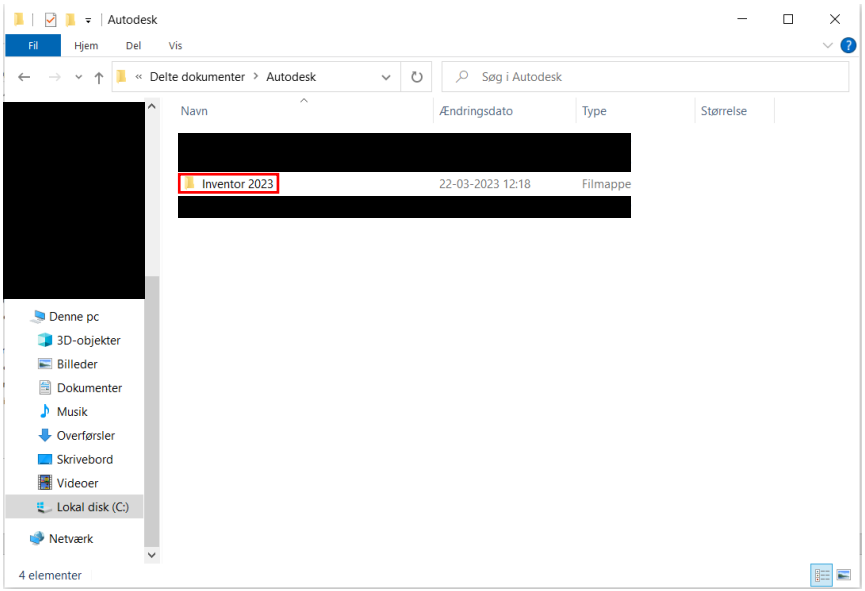
4. Click on “**Public Documents**”.



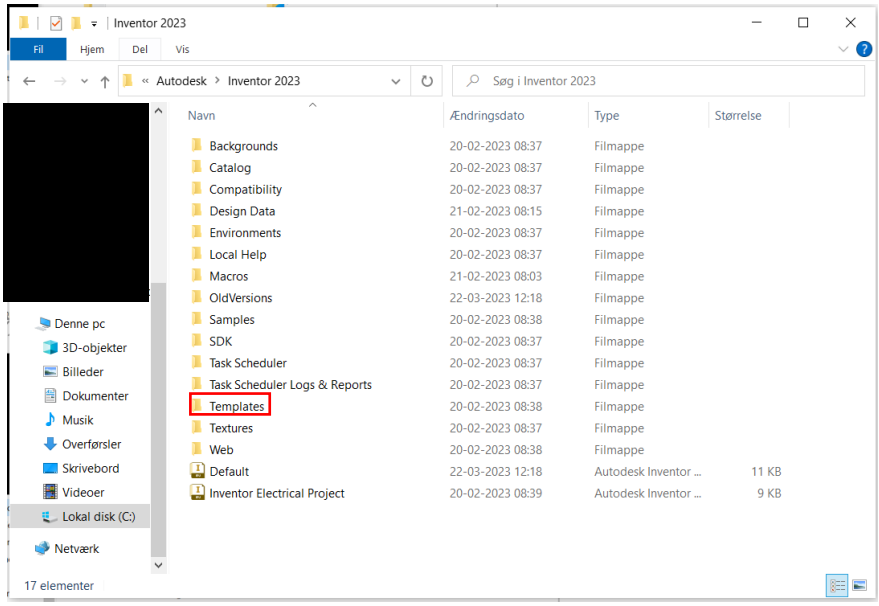
5. Click on “**Autodesk**”.



6. Click on “**Inventor XXXX**”. (XXXX = version).

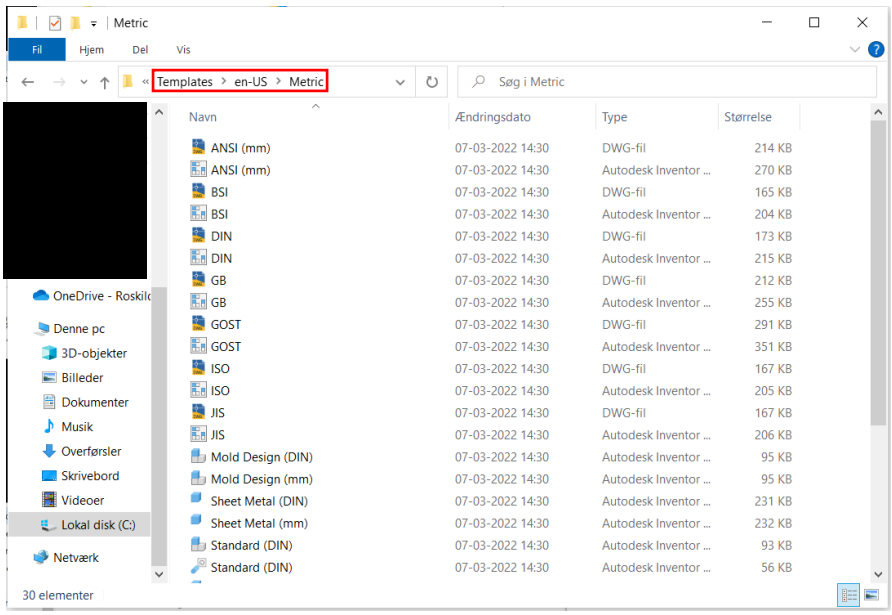


7. Click on “**Template**”.



Now you are in the first folder when you click “**New…**”.

8. Here are all the folders and **Templates** that exists, when you press “**New…**”.



Here we can create a new folder for our own **Templates** or insert prefabricated **Templates**.

Here in the example, we create our own folder under "**en-US**".

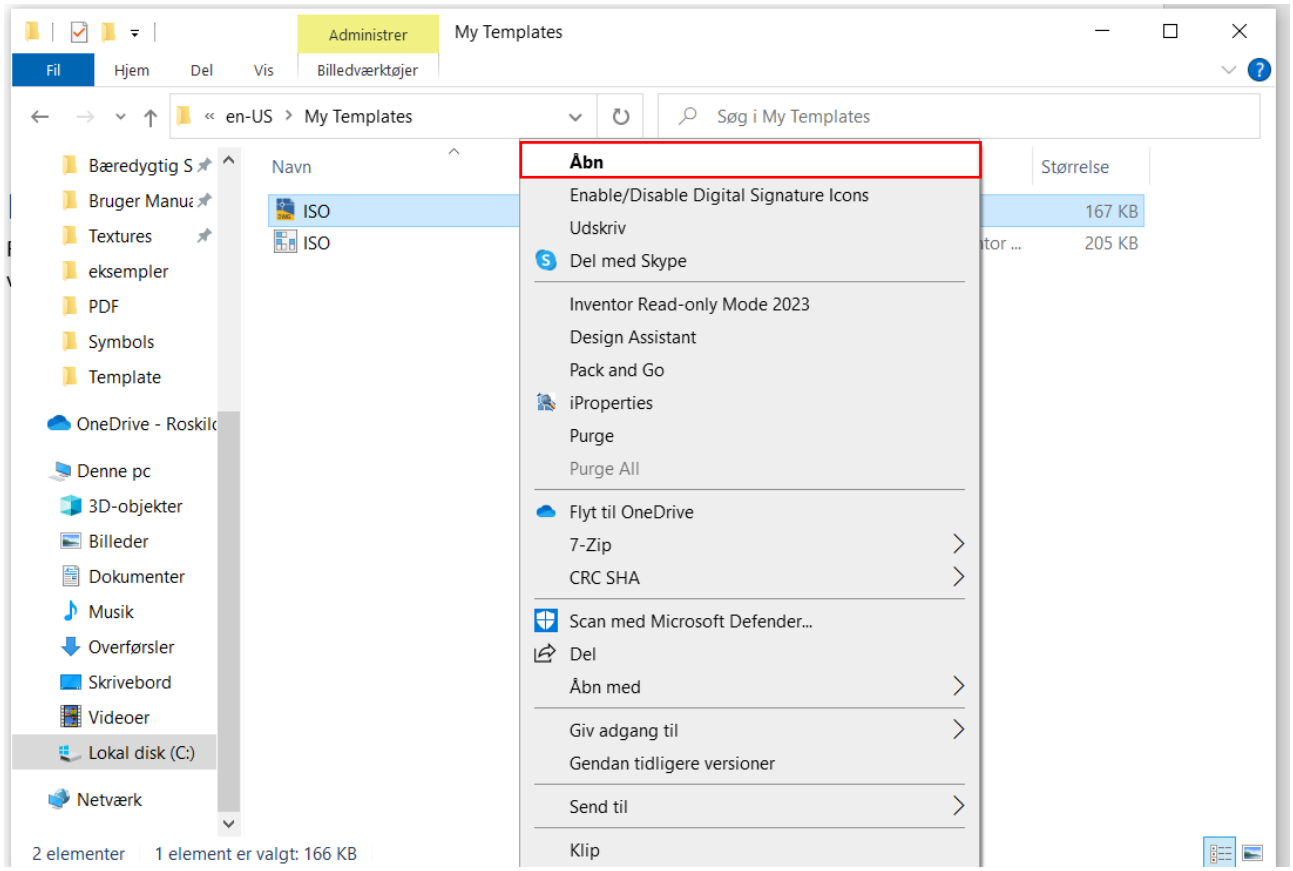
New Templates

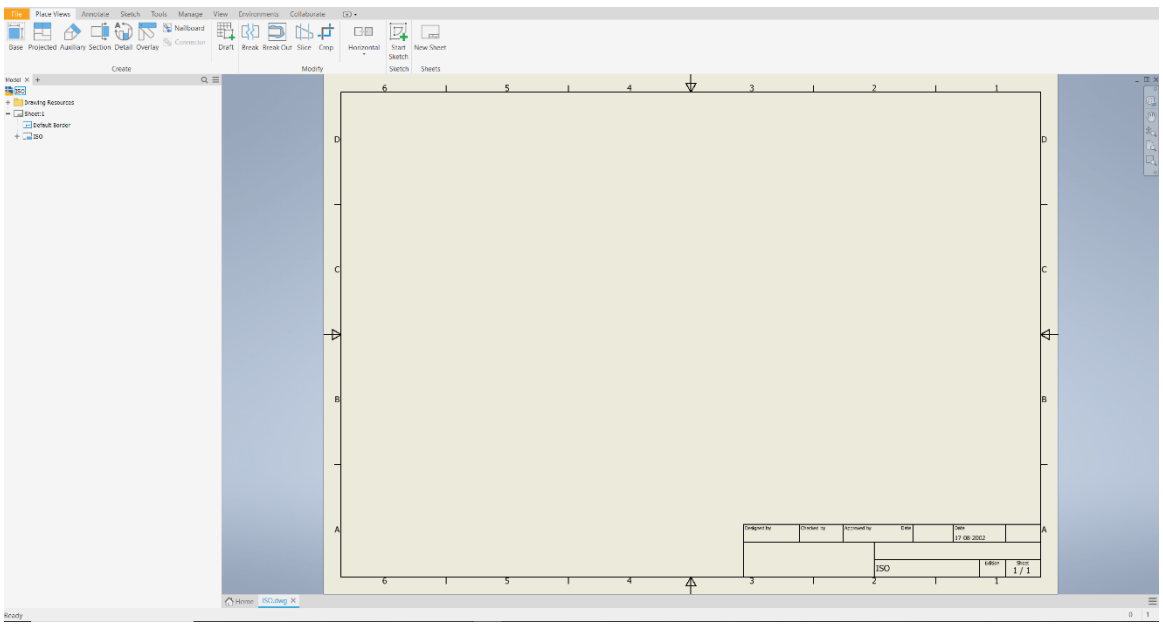
To create a **Template** from scratch, it is a good idea to do it in the **Default** project folder. Here is a guide that goes over how to change **Trailing Zeros** and **Leading Zeros** off and on, by default. And how to change the **Title Block** with **iProperties**.

Title Block

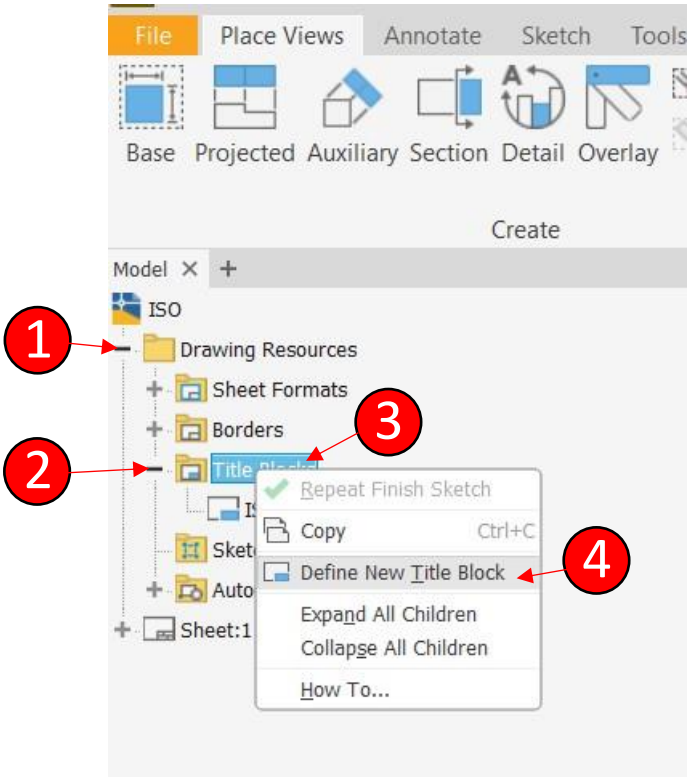
Template first (.dwg):

1. Start by opening the **Template** file you want to change.
   This is done by finding and opening with a right-click (In this guide, "**ISO**" is selected as the basis):



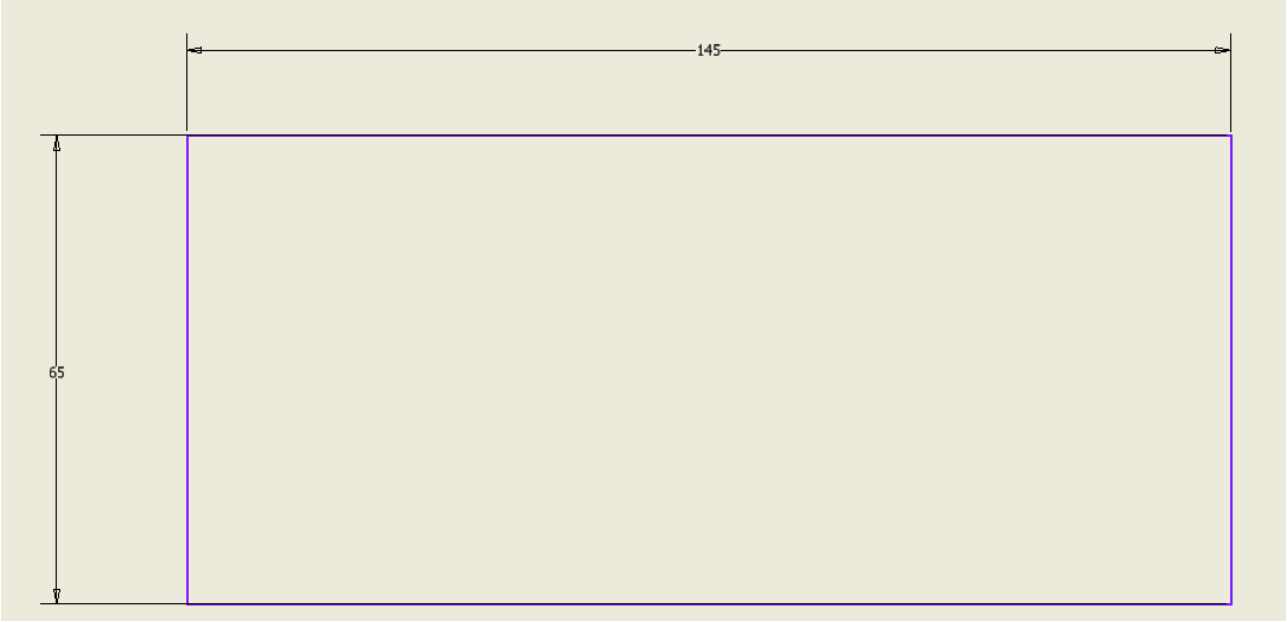


2. Make a new **Title Block**:

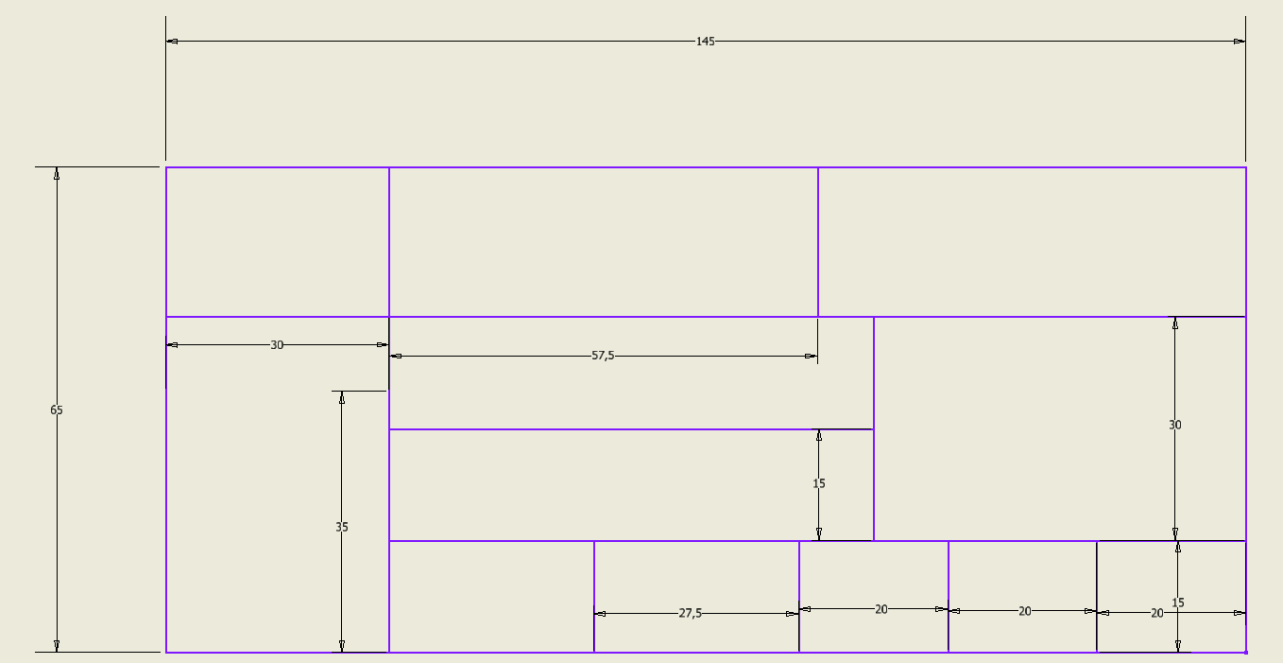


1) Click on + at "**Drawing Resources**".
2) Click on + at "**Title Blocks**".
3) Right-click on "**Title Blocks**".
4) Select "**Define New Title Block**".

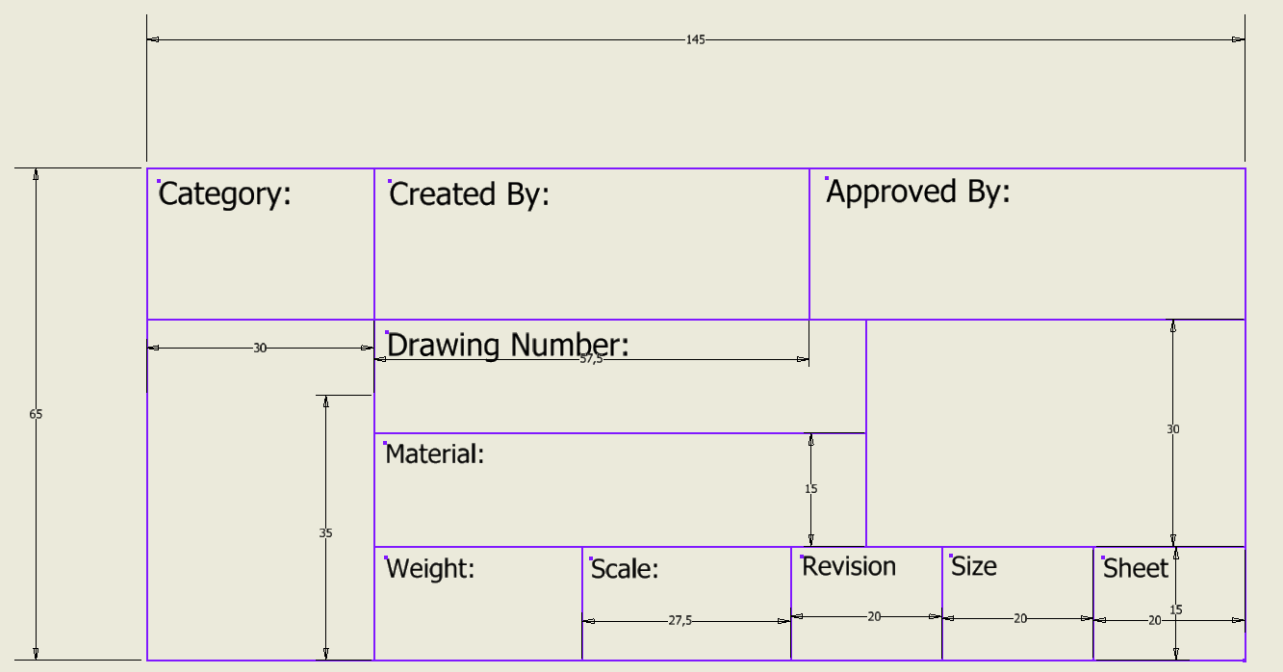
3. Draw an outer border of your **Title Block**:



4. Draw some boxes:



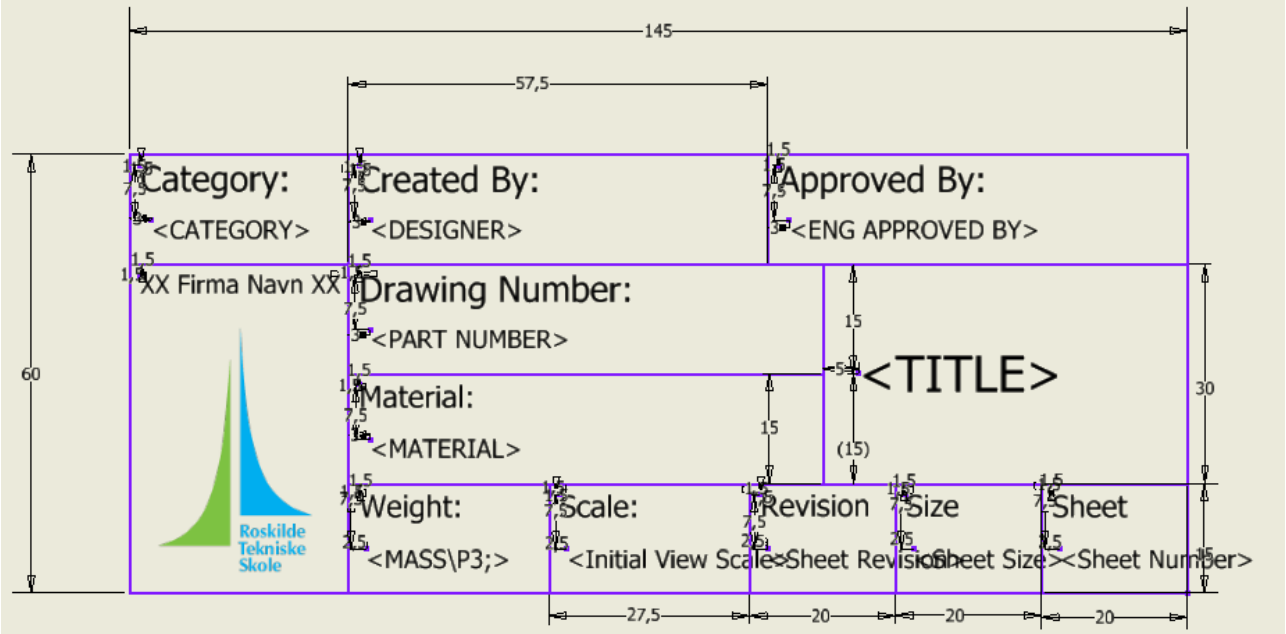
5. Headings for the boxes:



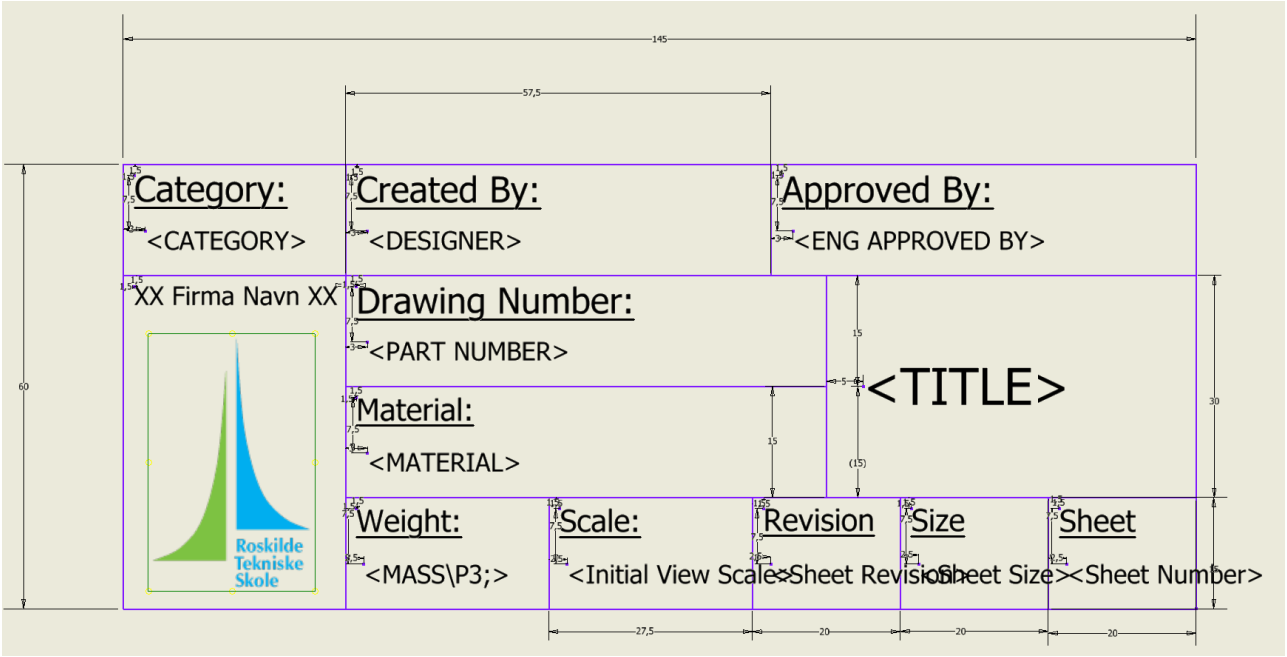
Here, the 2 empty boxes do not have headings, the one on the left is space for the company logo, and the other title, is a large font size.

6. Insert **iProperties** values:
   - *Weight, areal, density and volume* is under the type: **Physical Properties – Model.**
   - *Scale, sheet number, size and revision* is under the type: **Sheet Properties.**

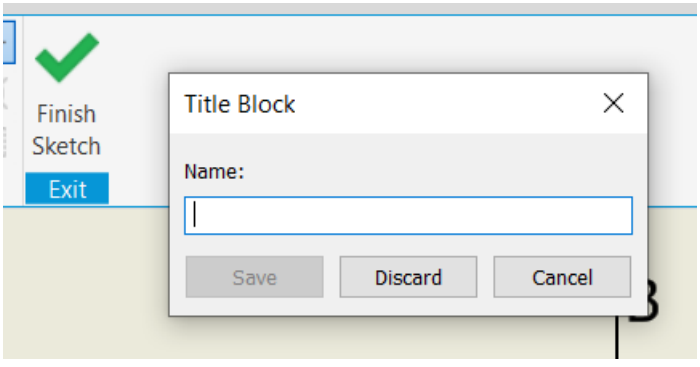
Tip: It's okay that the value name crosses a line, as the value itself, probably doesn't.



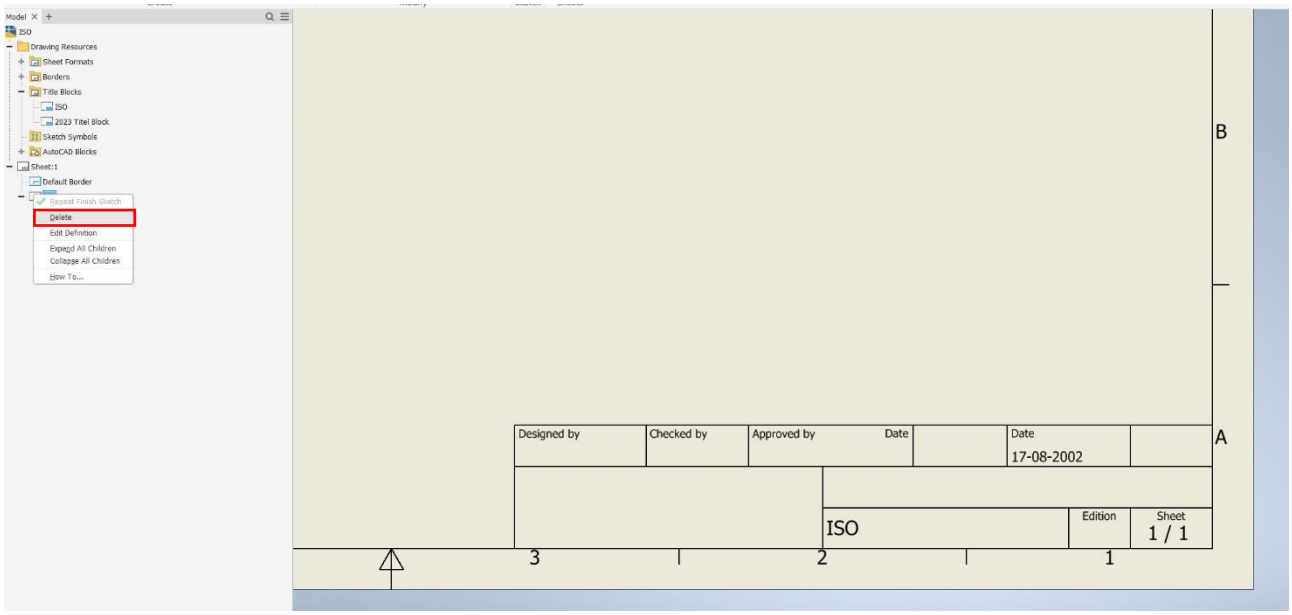
7. Fixes:
   - It is recommended to dimension the texts, as it gives a more professional appearance.



8. Finish and name.

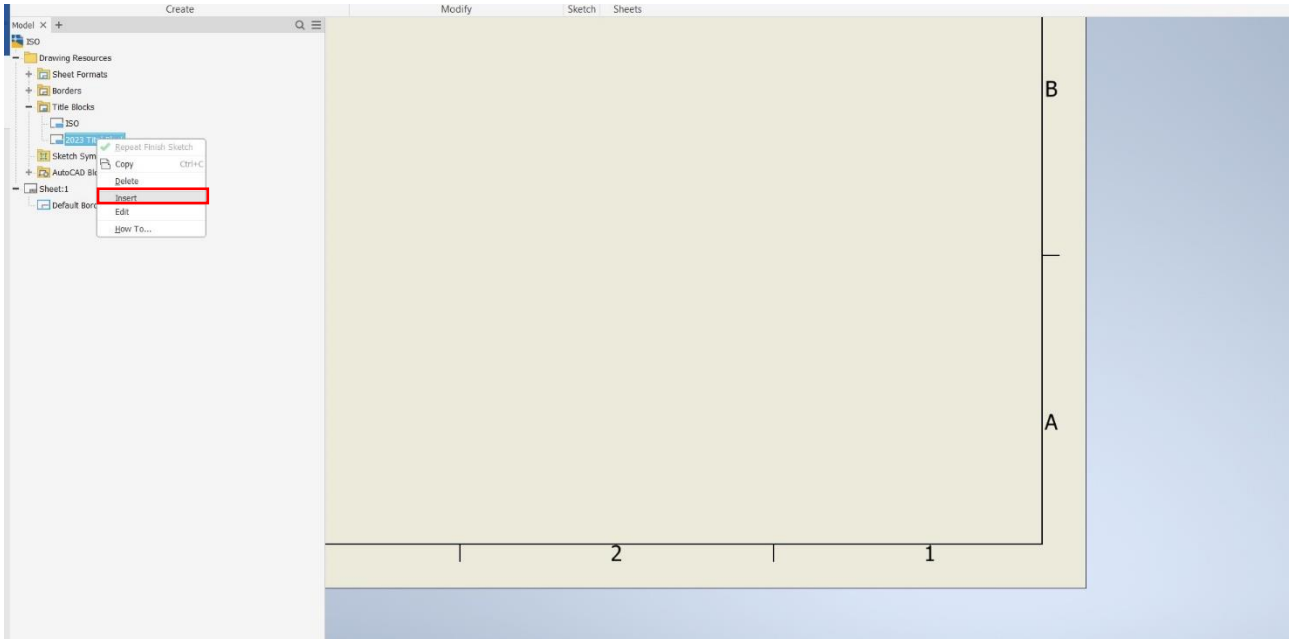


9. Right click on the previous **Title Block** and select "**Delete**":

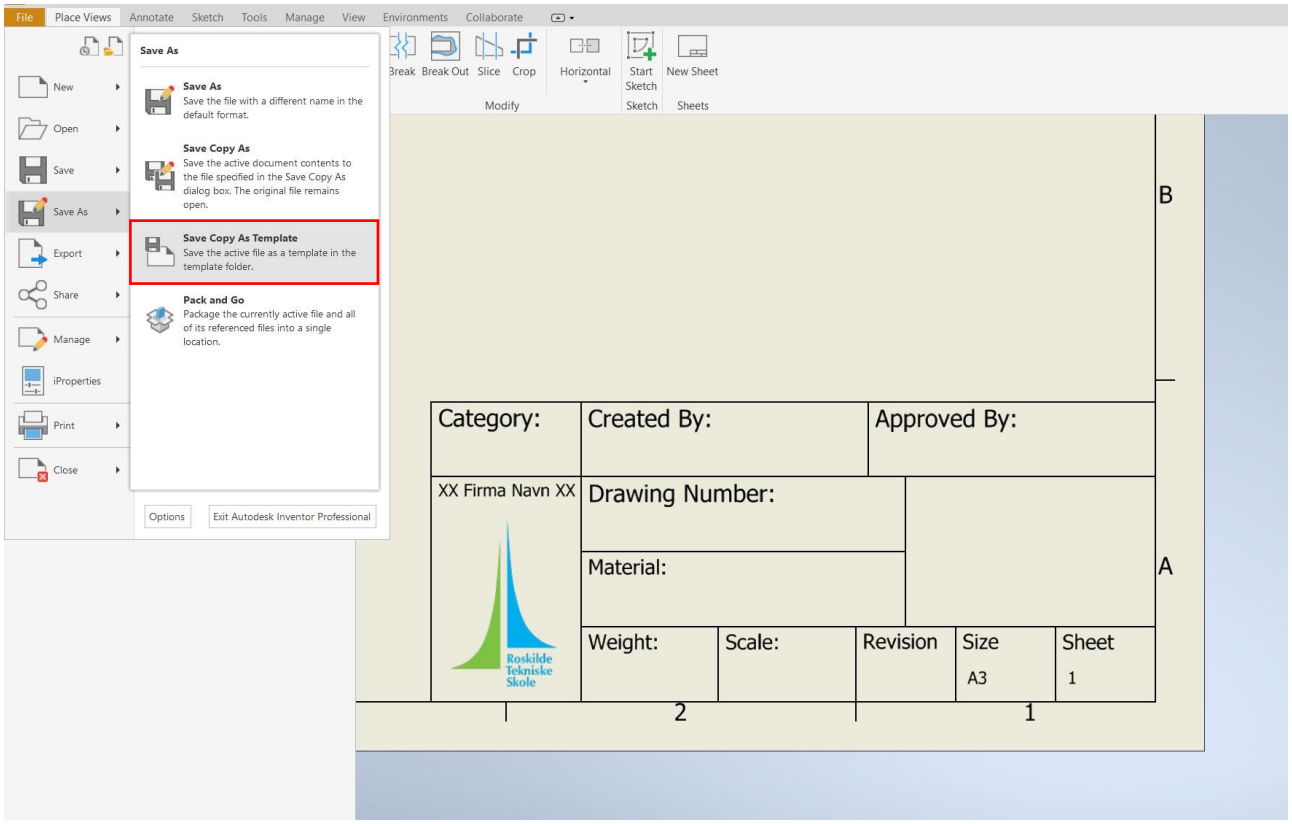


***Remember** to delete the **Standard Title Block (ISO)**.*

10. Insert the new **Title Block**. Right-click on the newly created **Title Block** and select "**Insert**":

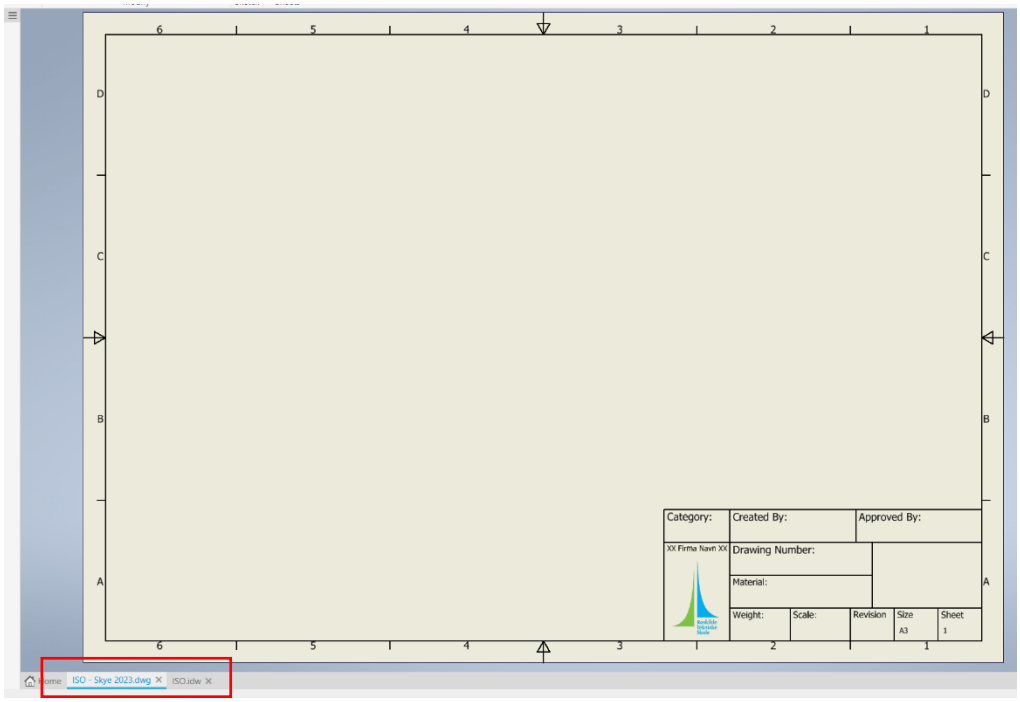


11. Save **Template** (***Remember*** *to select the desired folder*).

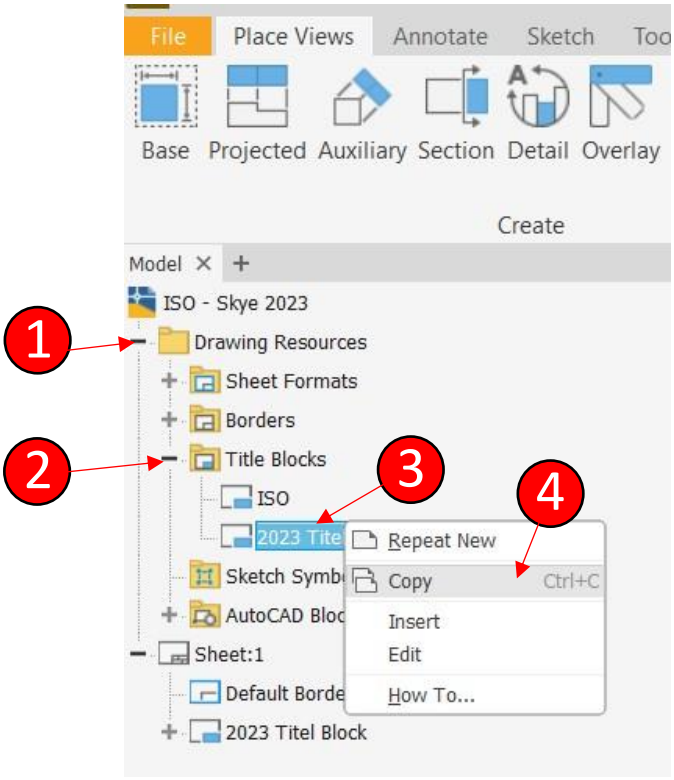


Template secondary (.idw):

1. <u>Open</u> the newly made **Template** and open a "**.idw**" fil (**ISO.idw**):

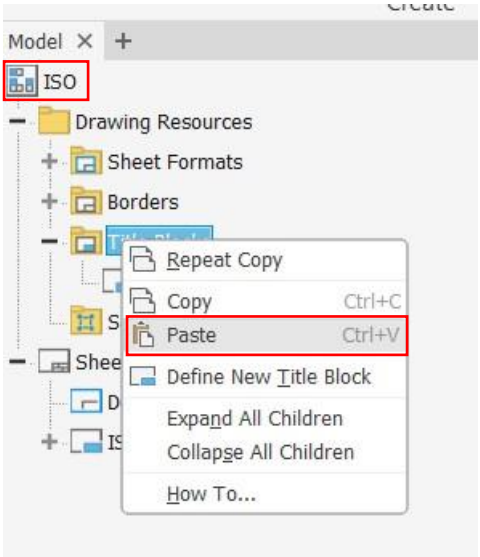


2. Copy the drawing head from the newly created **Template**:

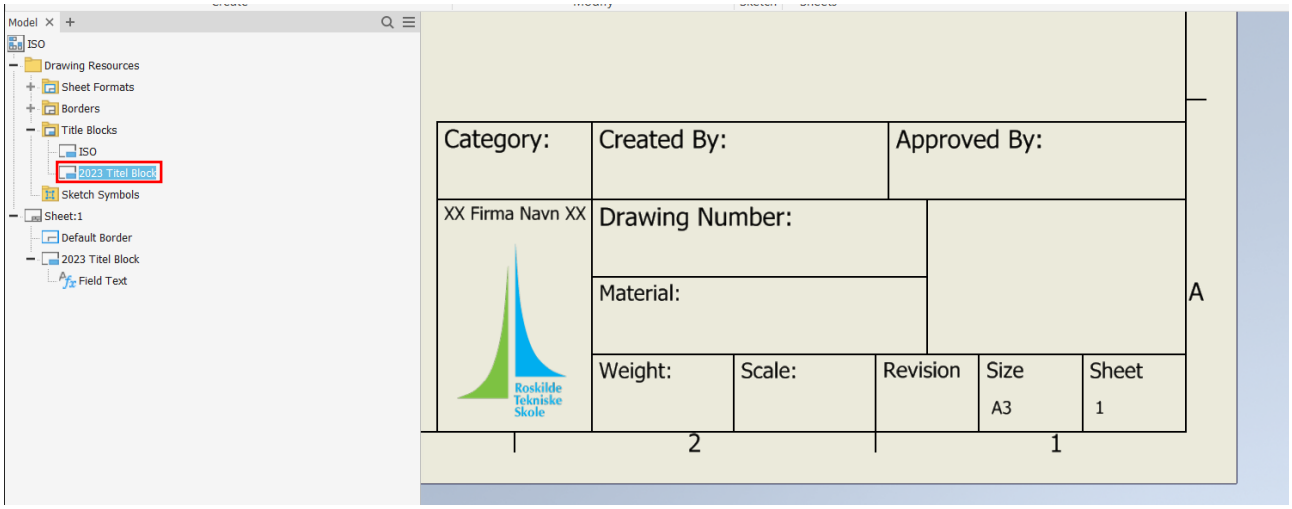


1) Click on + at "**Drawing Resources**".
2) Click on + at "**Title Blocks**".
3) Right-click on "**Title Blocks**".
4) Select "**Copy**".

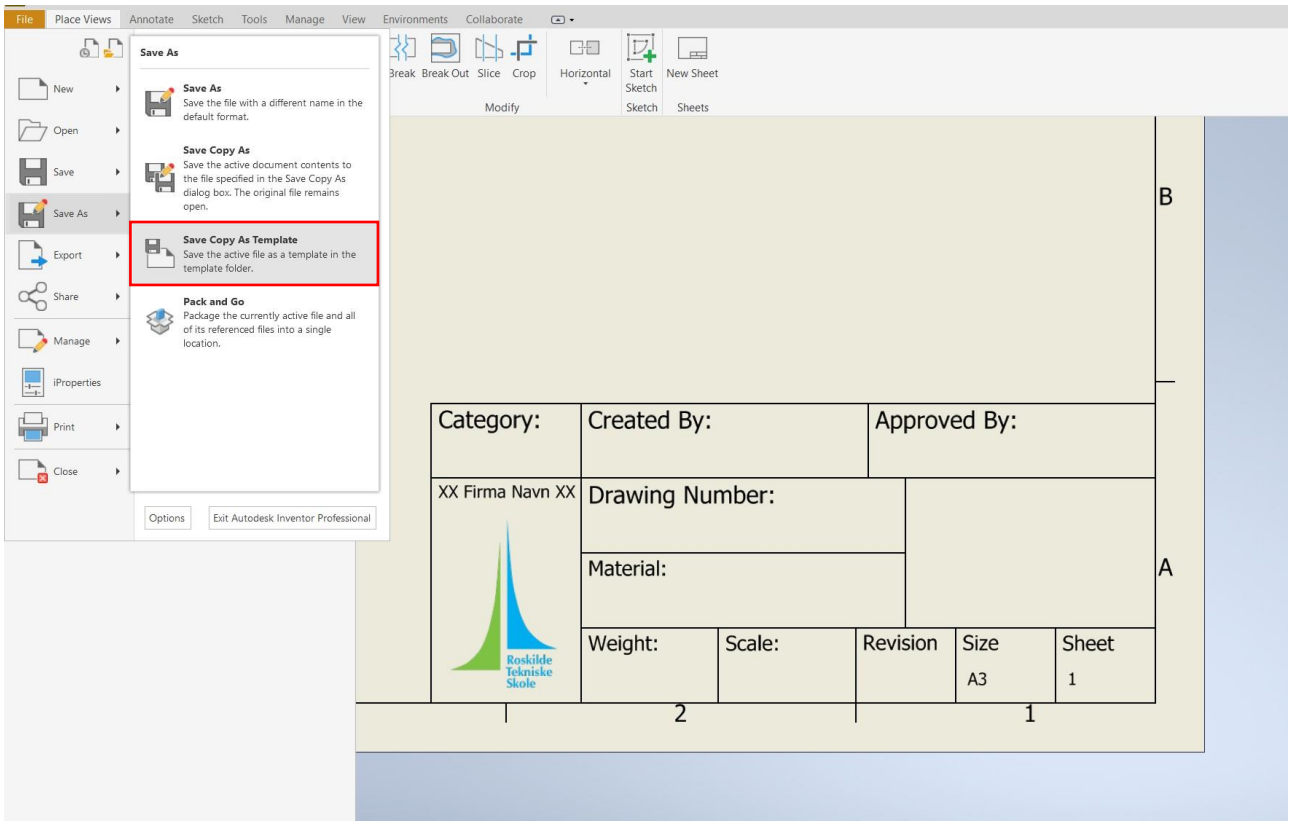
3. Open **ISO.idw** and locate the **Title Block** folder, like before. Right-click and select "**Paste**":



4. Delete now the **Standard Title Block (ISO)**. Insert the **Title Block** you made yourself:

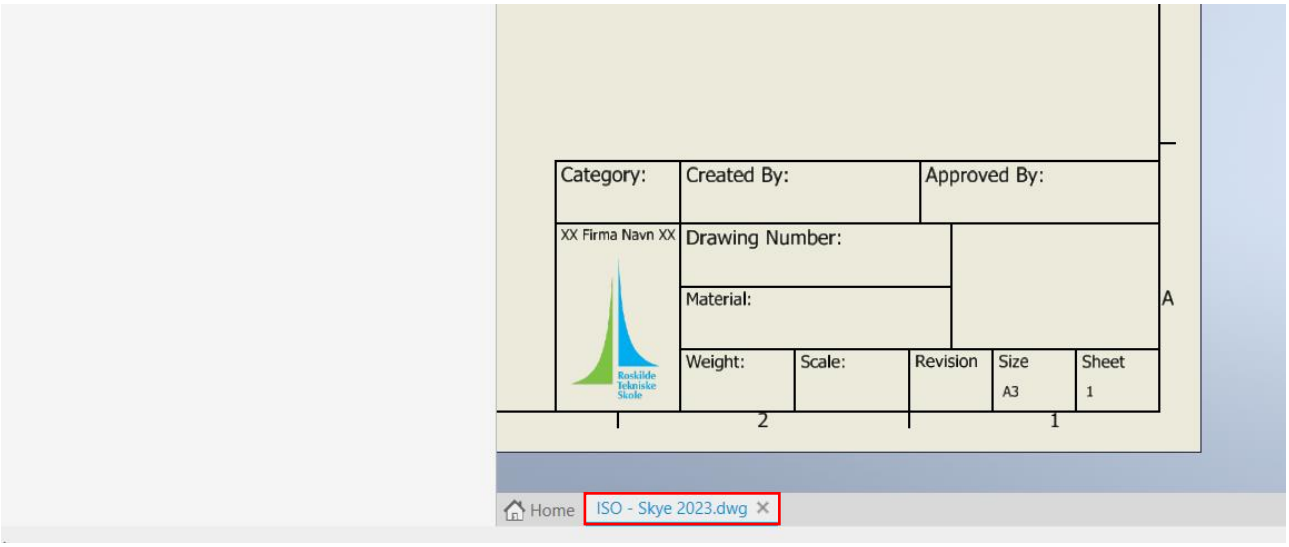


5. Save **Template** (***Remember*** *to select the desired folder and save as* ***.idw***).

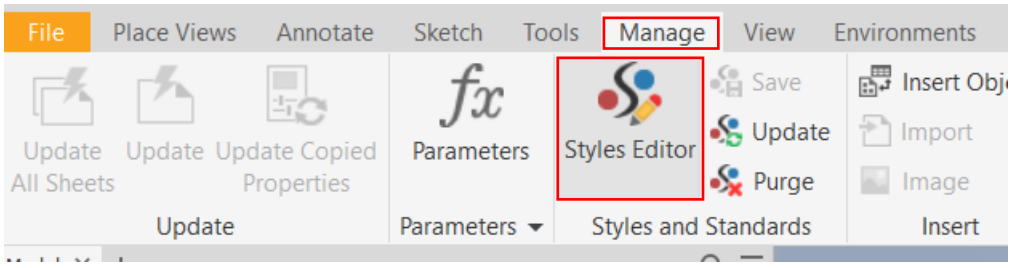


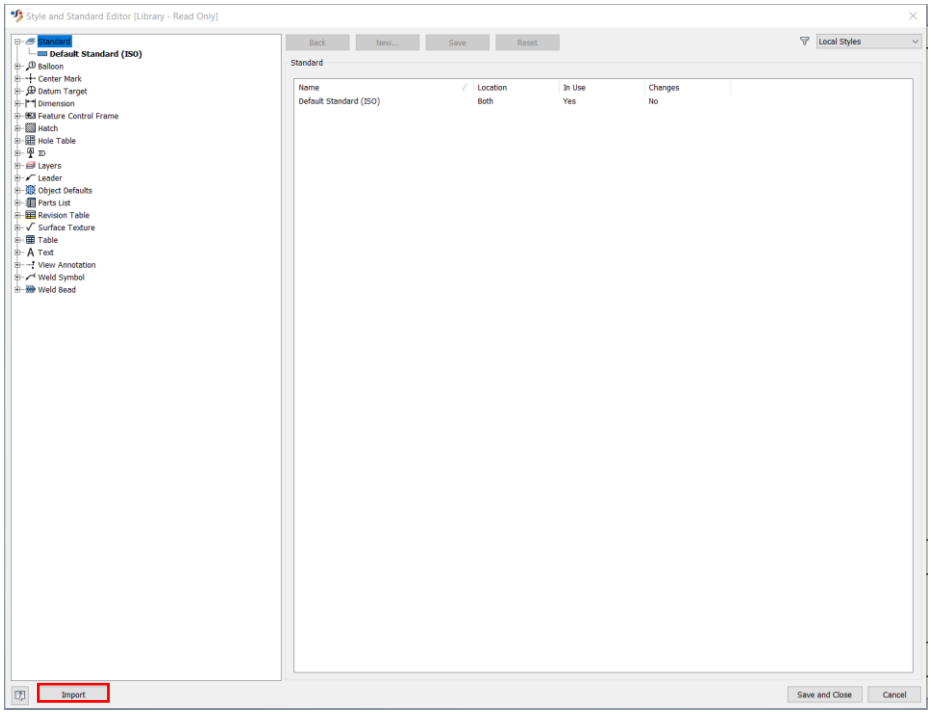
Changes to Trailing Zeros og Leading Zeros

1. Open the desired **Template**:



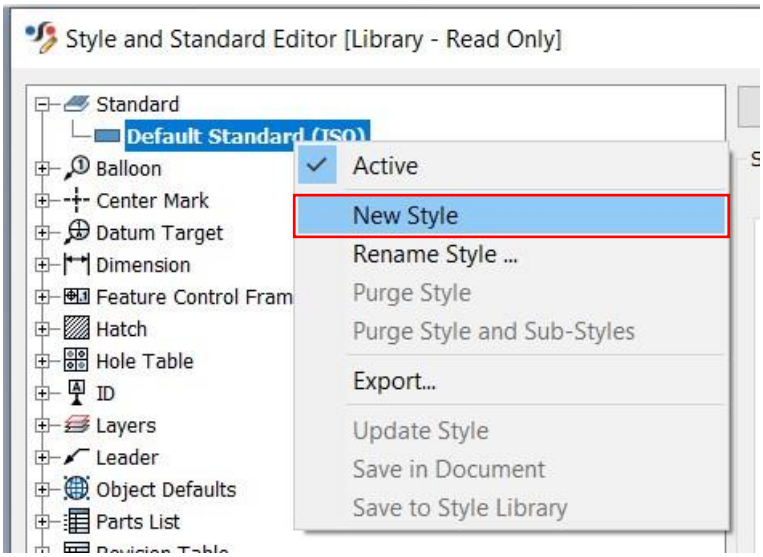
2. Open **Styles Editor**:



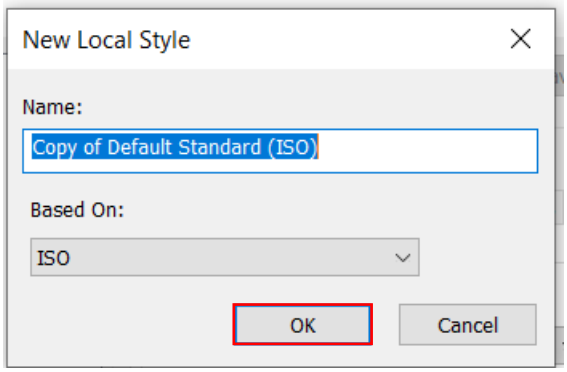


If you have a prefabricated **Styles Editor** / **Standard**, press on Import.

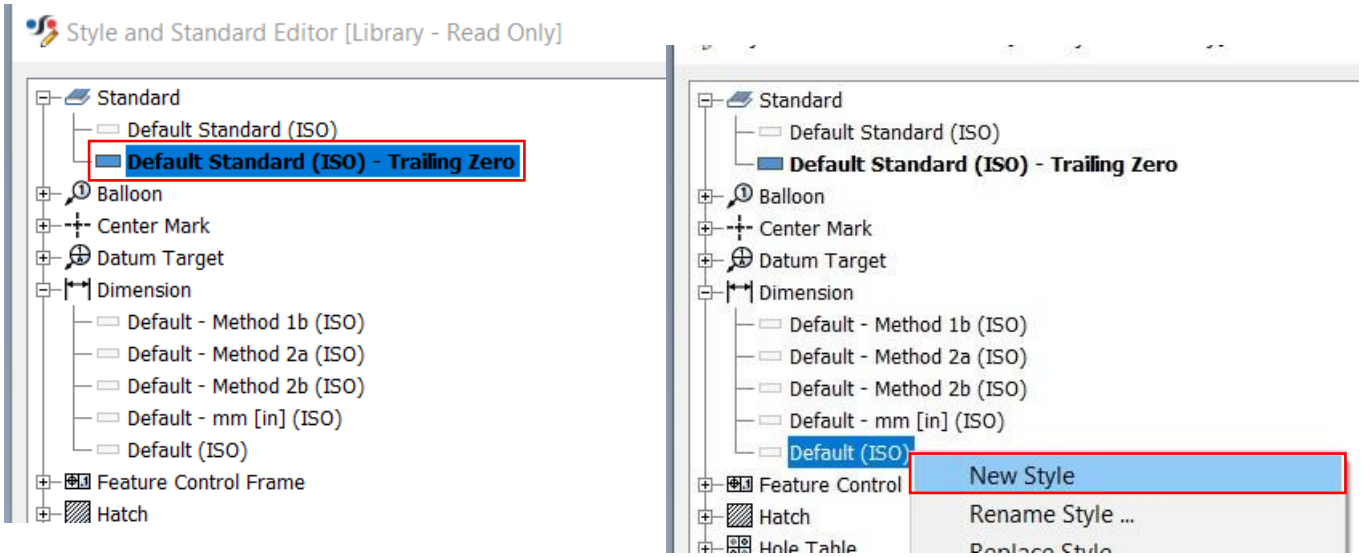
3. Right-click on **Default Standard (ISO)** and select "**New Style**":



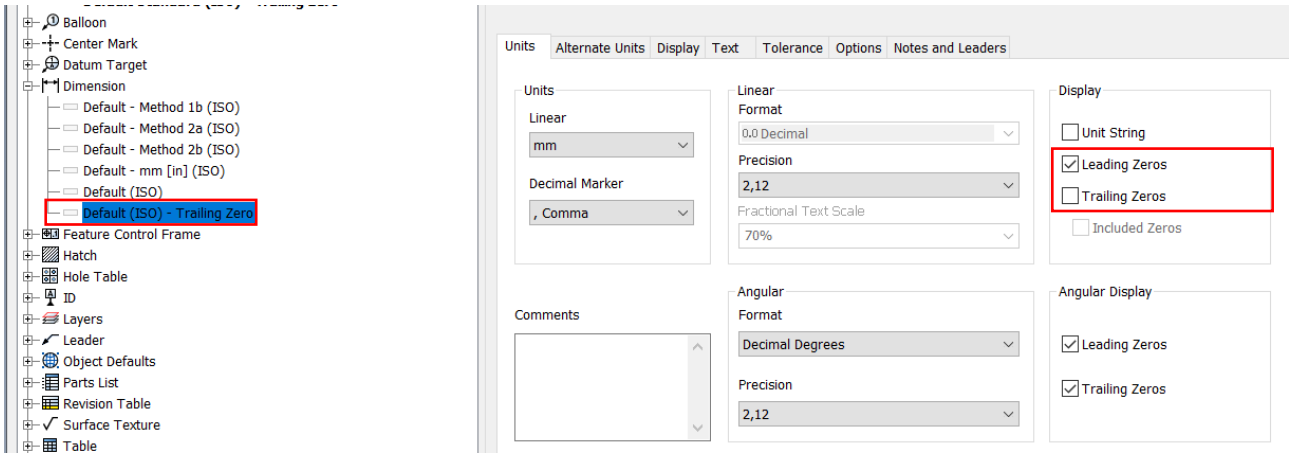
4. Name the **Standard** and click **OK**.



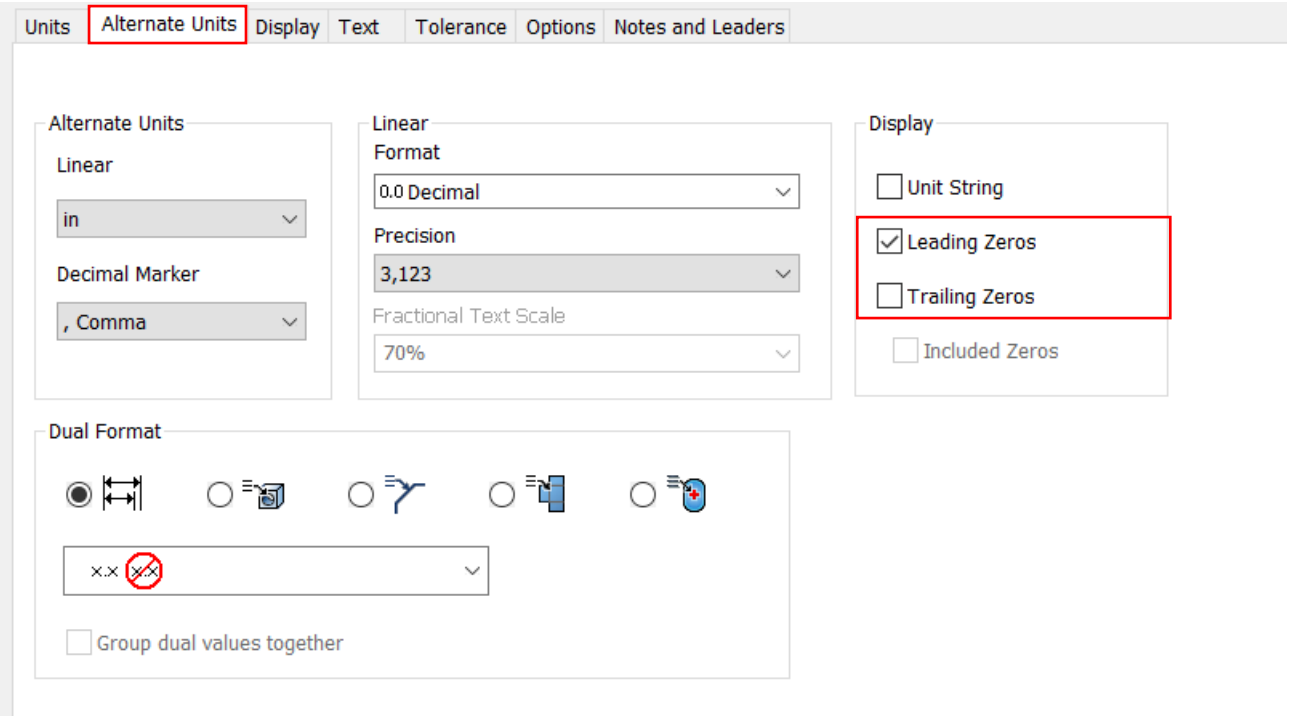
5. Make sure you have selected the new **Standard** and expand **Dimension**. Then make a new **Default** (**ISO**) like before:



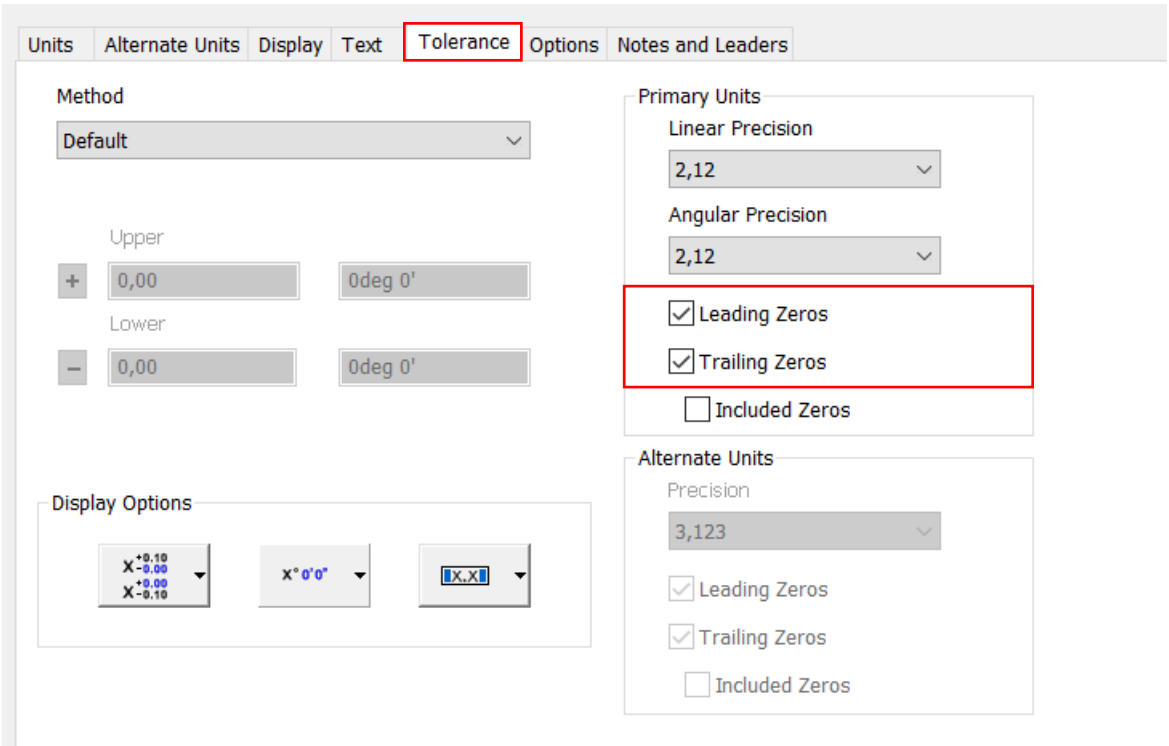
6. Turn [Trailing Zeros / Leading Zeros] off (in the example we only turn off *Trailing Zeros*):



7. **Alternate Units**, turn off:

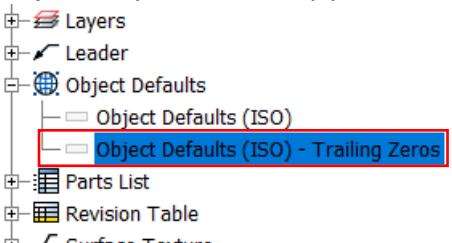


8. **Tolerance**, here you decide for yourself whether your tolerances should have **Trailing Zeros** on or off.

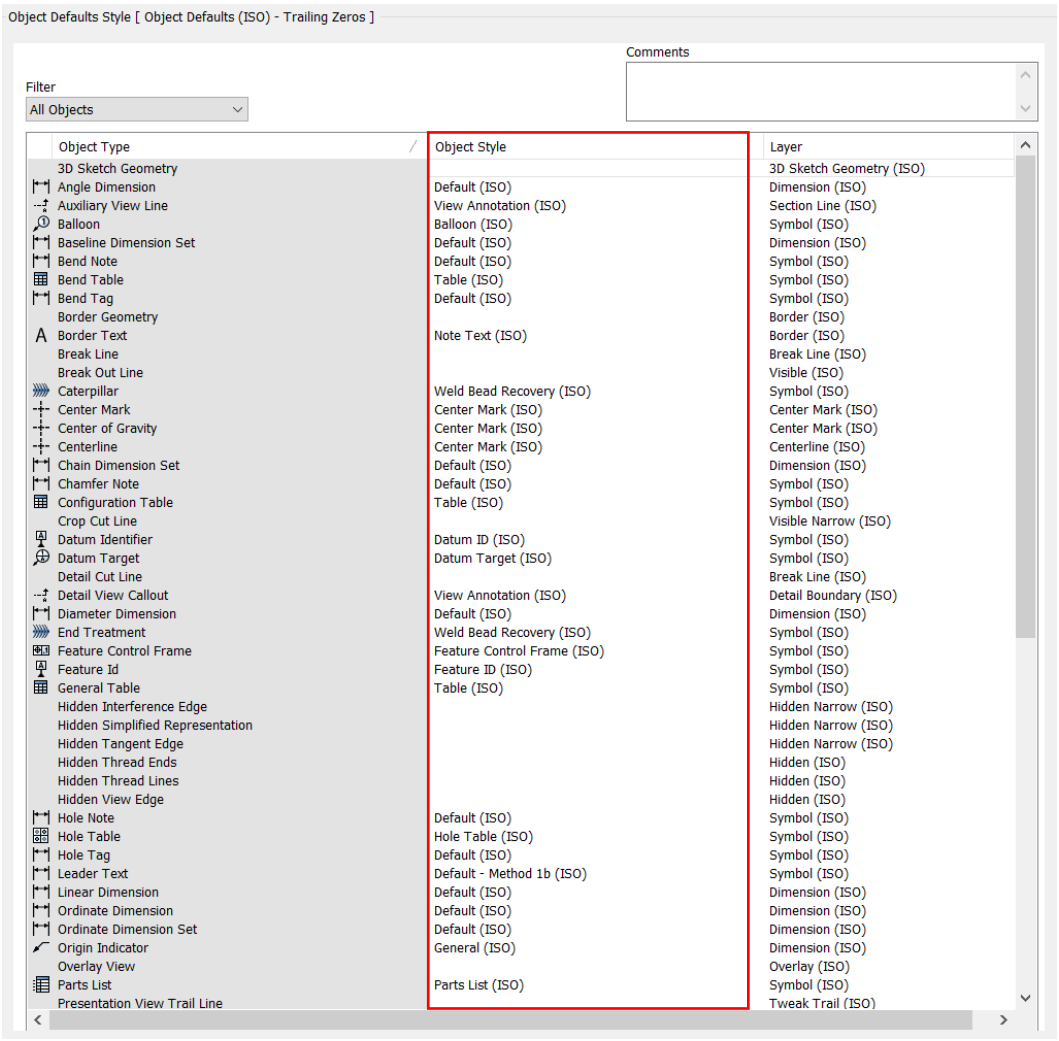


9. Now the different categories/sections must be assigned an **Object Style**, which is the **Default (ISO) without Trailing Zeros**. Expand "**Object Defaults**" and make a new:

- *If it asks you to save, say yes.*

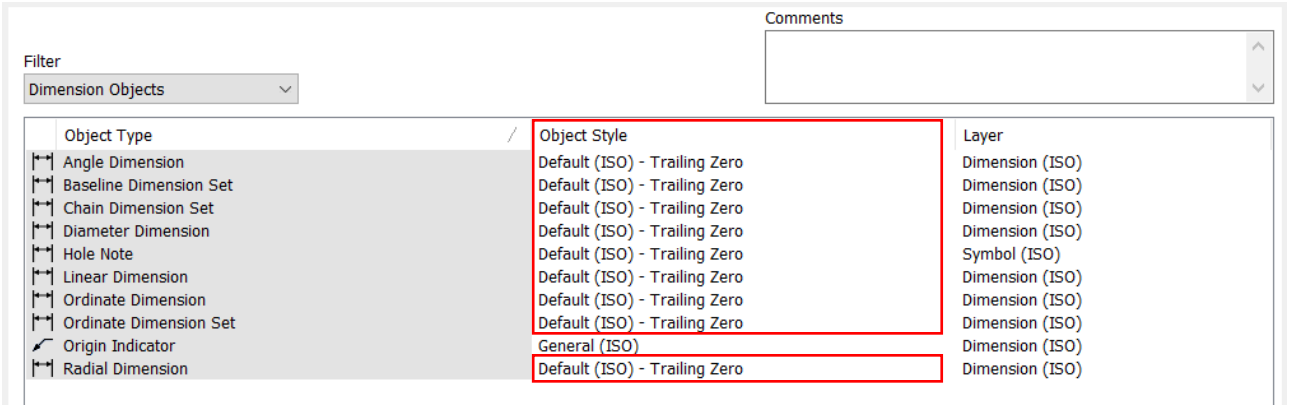


10. Below, assign the **Object Style**, to **Default (ISO) without Trailing Zeros**:
    - *As a starting point, you must change all the places that say "**Default (ISO)**" under **Objects Style**.*

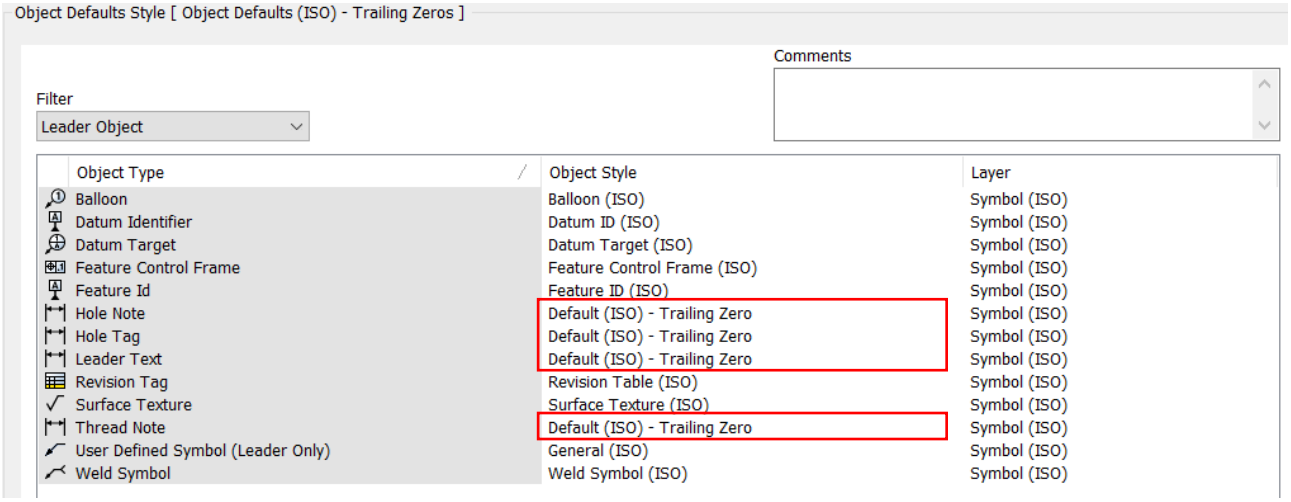


Review of allocation:

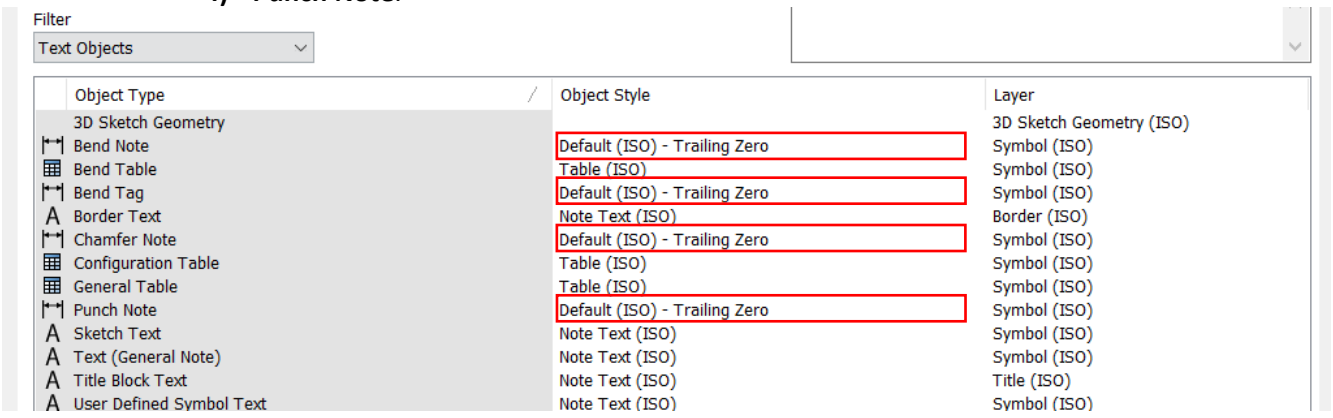
11. Change the filter to "**Dimension Objects**", here change all, except "**Origin Indicator**", to the **Dimension Default** you created:



12. Change the filter to "**Leader Objects**", here change the following to the **Dimension Default** you created:
    - *If it asks you to save, say yes.*
    1) **Hole Note**.
    2) **Hole Tag**.
    3) **Leader Text**.
    4) **Thread Note**.

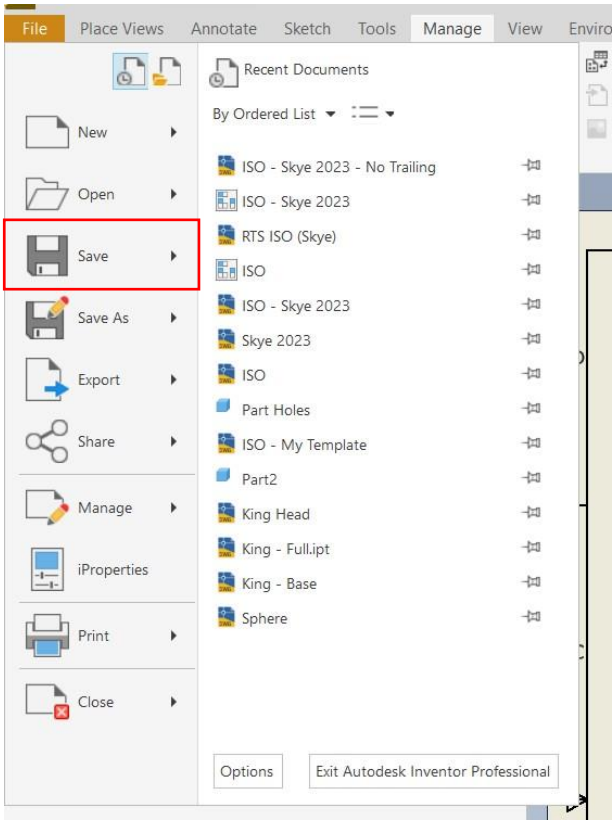


13. Change the filter to "**Text Objects**", here change the following to the **Dimension Default** you created:
    - *If it asks you to save, say yes***.**
    1) **Bend Note**.
    2) **Bend Tag**.
    3) **Chamfer Text**.
    4) **Punch Note**.



14. Press **Save** and **Close.**

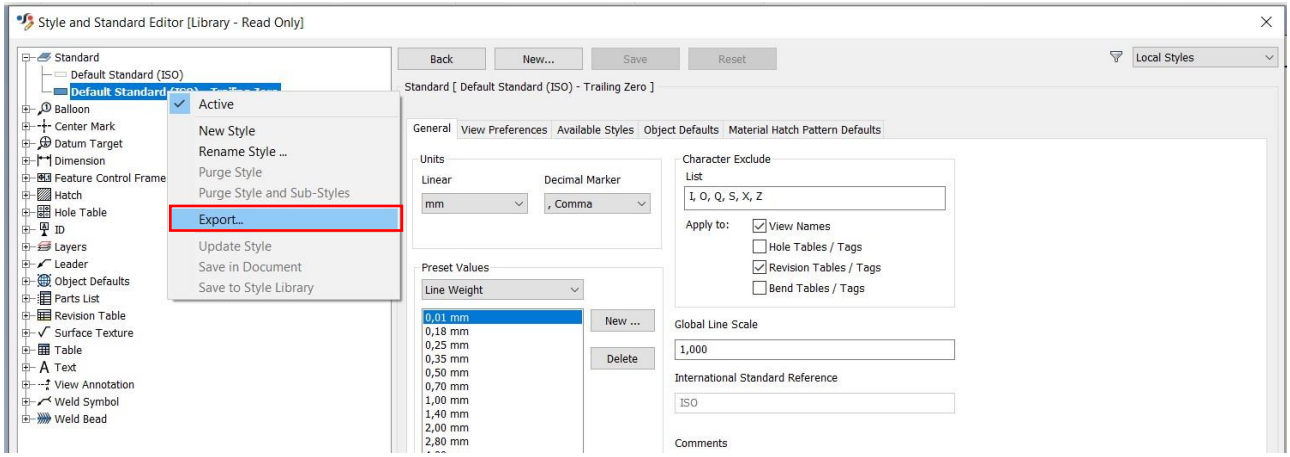
15. Save **Template** (***Remember*** *to select the desired folder*).



Exporting and importing of the Styles Editor

Exporting:

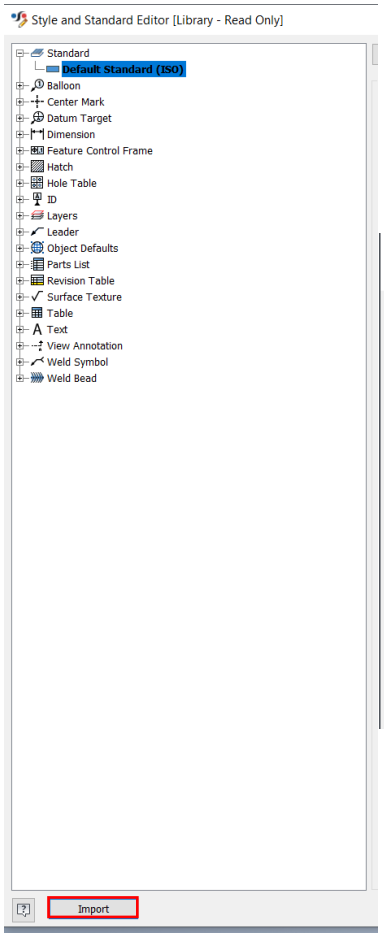
1. Open a changed **Styles Editor**, and right-click on the desired, here choose "**Export…**":



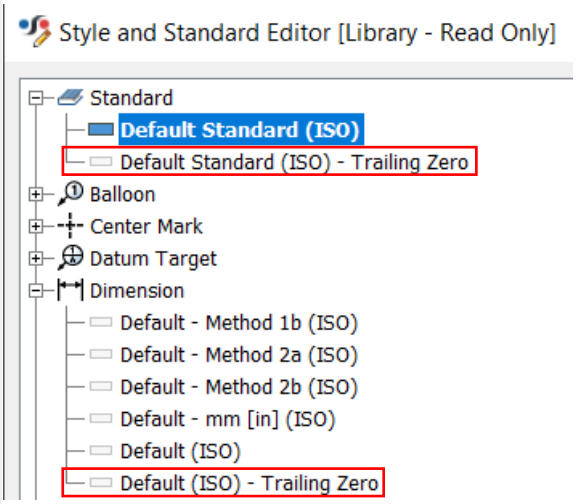
2. Save in a desired place.

Importing:

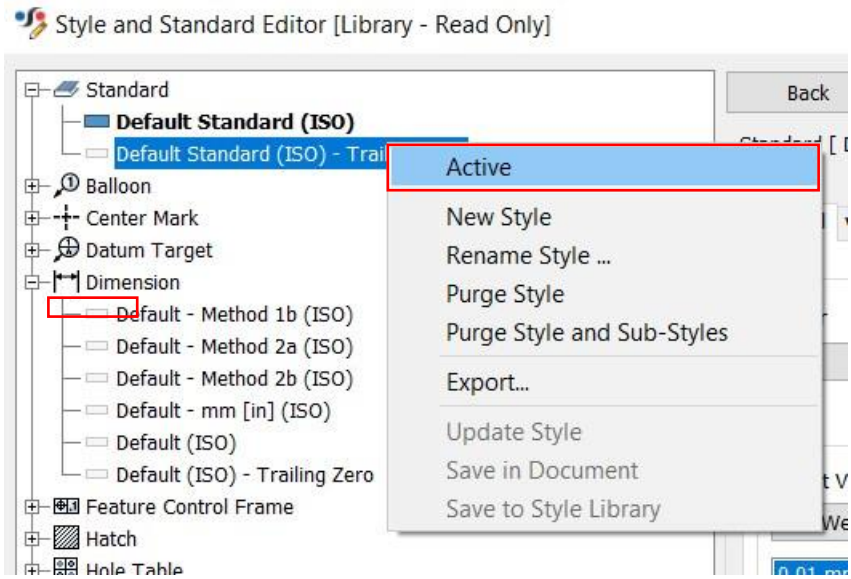
1. Open a desired **Styles Editor**, and click on "**Import**":



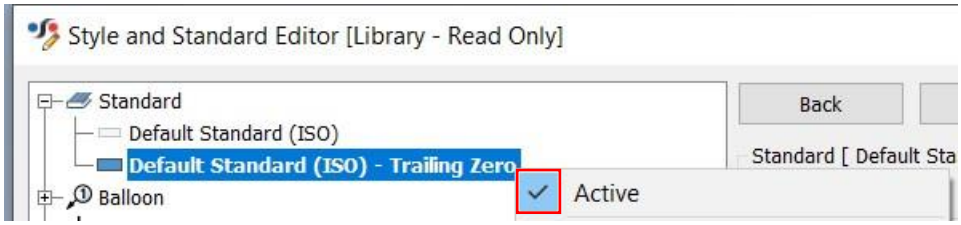
2. Locate the desired file, then you can see it has been imported, and all its **Sub-Styles**:



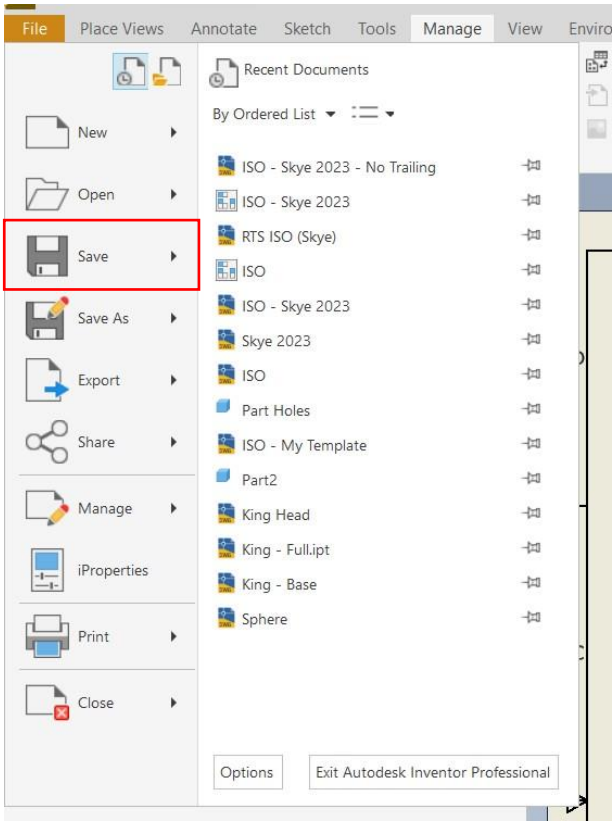
3. Right-click on the imported and select "**Active**".



4. Check that it is active (right-click). Now complete with "**Save and Close**".



15. Save **Template** (***Remember*** *to select the desired folder*).

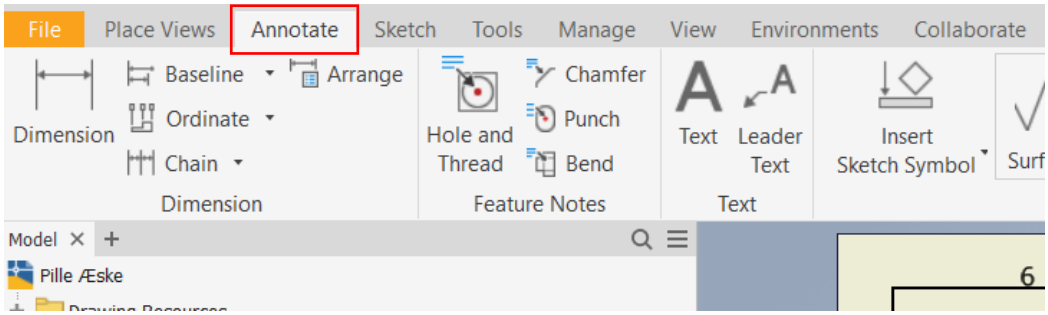


Annotate – Drawings

Introduction

When done with either a **Part** or an **Assembly**, then make a drawing file/technical drawing. With that can (for example) an Industrial Engineer make an item on their machines. When starting a **DWG-file** / **IDW-file** you need to insert your object as a **View**. Afterwards, can you either **Annotate** or **Modify**.

In order to use **Annotate** look under the **Annotate** Tab:



Index:

Shortcut Keys

Selections:

Control (CTRL) = Adds more than one selection. *Press and hold.* *(It is done correctly when there is a plus icon next to the mouse).*

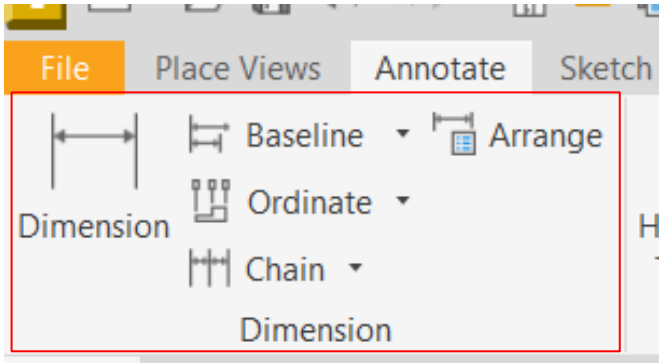
SHIFT (↑) = Removes selections. *Press and hold.* *(It is done correctly when there is a minus icon next to the mouse).*

Dimensions:

Introduction:

Dimensions the features are used to provide measurements so that an Industrial Engineer can make the part on their machines.

In order to use the **Dimensions** features, locate the **Annotate → Dimensions** tab at the top of the screen:

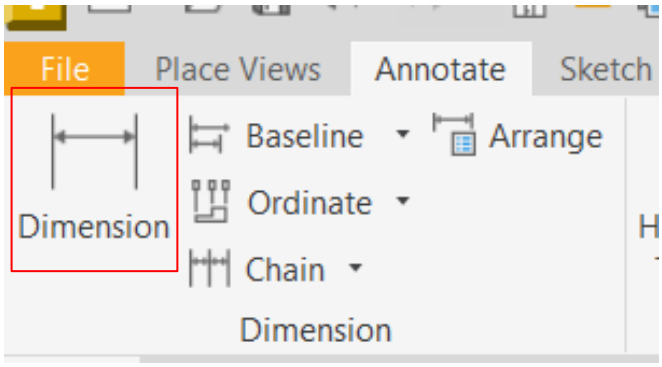


Dimension

Introduction:

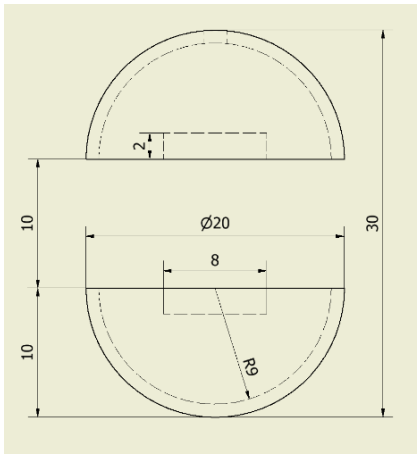
Dimension is one of the most frequently used **Dimensions** features, you use the **Dimension** feature to provide basic dimensions so that an Industrial Engineer can make the part on their machines.

In order to use **Dimension**, locate the **Dimension** tab at the top of the screen:



After you have **Projected** your object, you can use **Dimension** to create basic measurements. Such as length, width, depth, height, radius & diameter.

- **Shortcut Key** = D



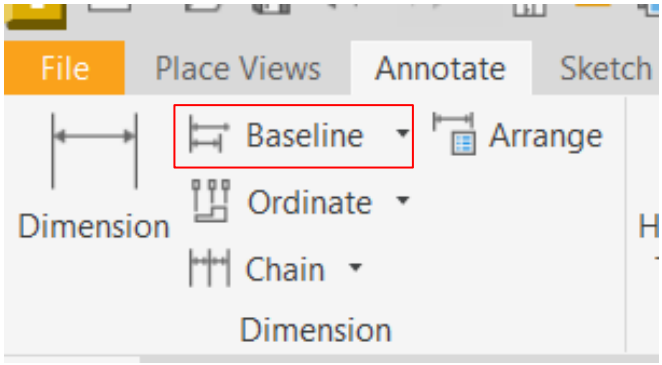
Here is an example of how to use **Dimension**.

Baseline

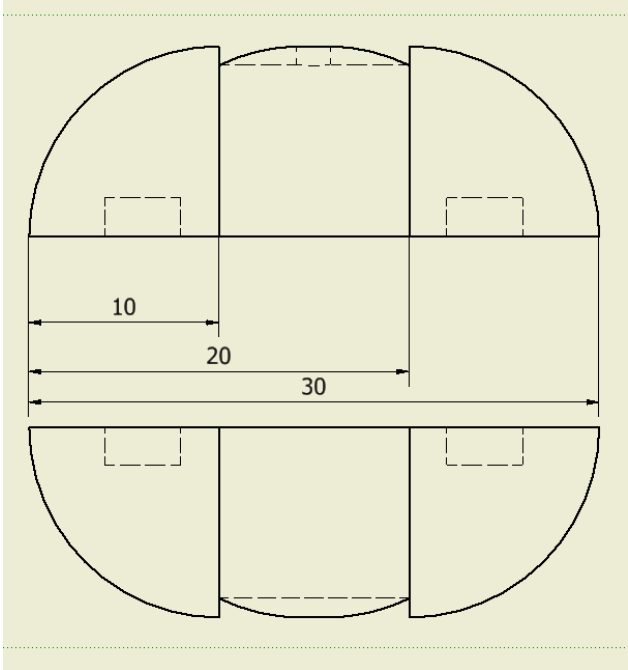
Introduction:

The **Baseline** feature is used to establish a dimension **Set** based on an origin line. Start by selecting a line from which all dimensions should originate. It goes from that line to the next point, and from the origin line to the next point, and further onwards.

In order to use **Baseline**, locate the **Dimension** tab at the top of the screen:



To locate the other **Baseline** features; Click on the arrow under "**Baseline**", and a drop-down menu will appears where you can see all the **Baseline** features.



Here is an example of what a **Baseline** might look like.

Select **Baseline**, then select your lines then right click and select "**Continue**". Then select the **placement** with a left click.

Select your **Origin** by right-clicking close to the desired line and select "**Make Origin**", as **Origin** is the line the dimensions are based on. Then right-click when you're done and select "**Create**".

If you want to **Rearrange** your **Baseline**, select all the dimensions → right-click and select "**Arrange Dimensions**".

Baseline Set:

The **Baseline Set** feature is used to make the **Baseline** in a different way with grouping. Start by selecting the start and end line. Then "**Continue**". Then select **placement**. Then select all the desired geometries and finish with "**Create**".

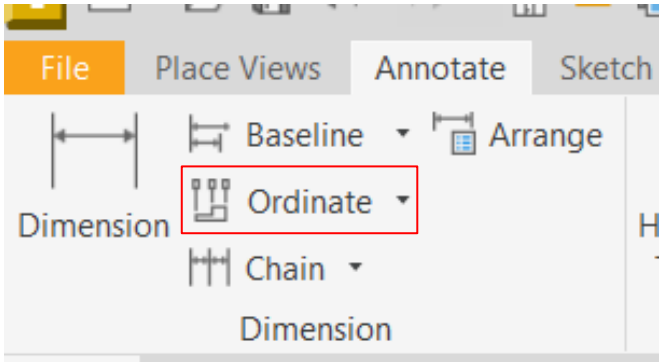
Now when you select a **Baseline** / **Dimension** you select everyone, as a group. But if you choose top or bottom, you only move one **Dimension**. To delete only one **Dimension**, right-click on the desired one and select "**Delete Member**". You can also "**Deatach Member**", "**Add Member**", "**Arrange**", "**Make Origin**".

Ordinate

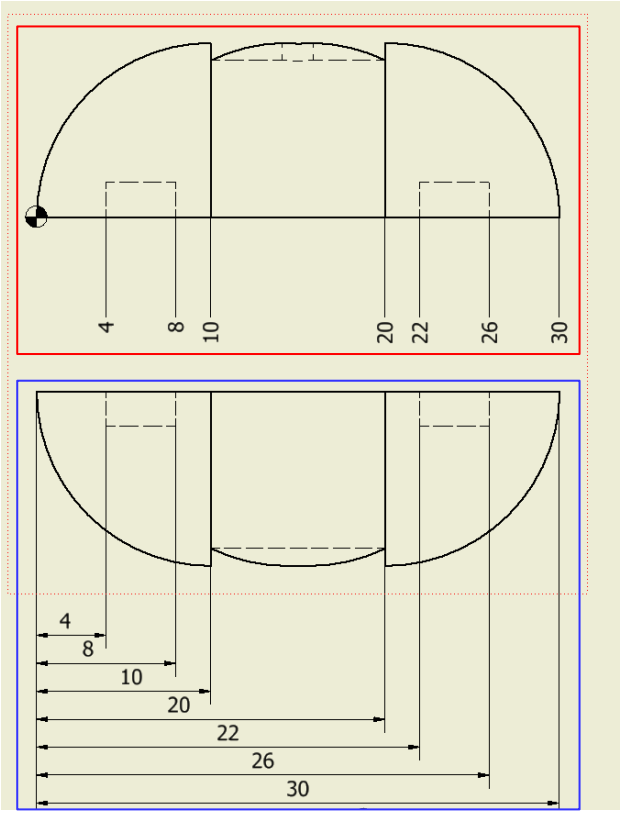
Introduction:

The **Ordinate** feature is used to create dimensions that resemble the **Baseline**. Here, instead of an **Origin**, a **Reference Point** is used. Your **Dimensions** become smaller and more manageable. And rotated 90°.

In order to use **Ordinate**, locate the **Dimension** tab at the top of the screen:



To locate the other **Ordinate** features; Click on the arrow under "**Ordinate**", and a drop-down menu will appears where you can see all the **Ordinate** features.



The top one is an example of an **Ordinate** and the bottom one is the **Baseline**.

Select **Ordinate**, then select the desired **View**. Then you choose your **Reference Point**.

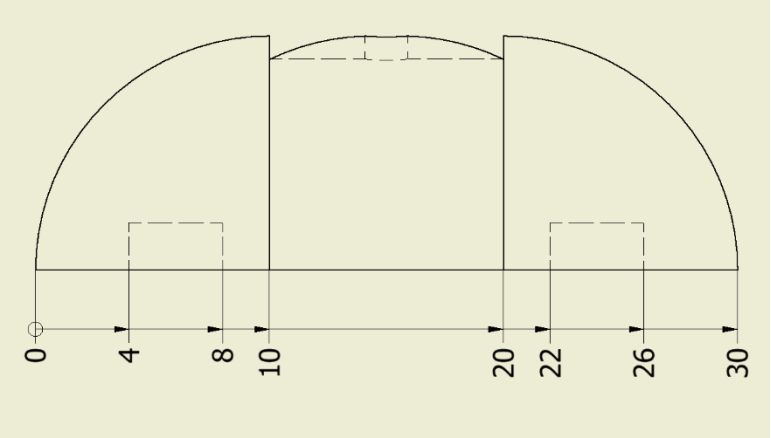
Select the desired geometries, then select **placement** with left click. Then right-click and select "**OK**" to complete.

Ordinate Set:

The **Ordinate Set** features is used to create a group of **Ordinates**. That means when you select one, all are selected.

In order to use, select the desired geometries, → right-click and select "**Continue**". Then select **placement** with left click. Then select your **Origin** by right-clicking close to the desired line and select "**Make Origin**". Then right-click when you're done and select "**Create**".

Example of an **Ordinate Set**:

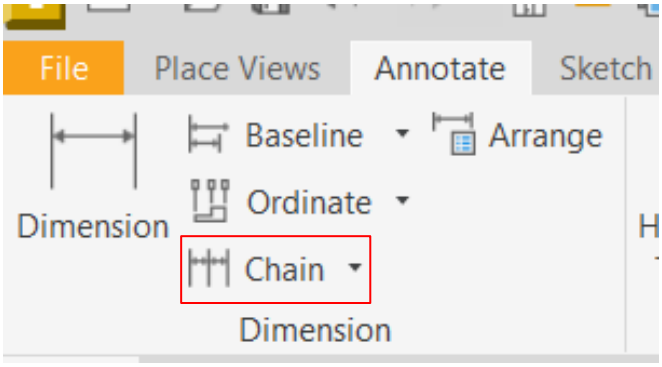


Chain

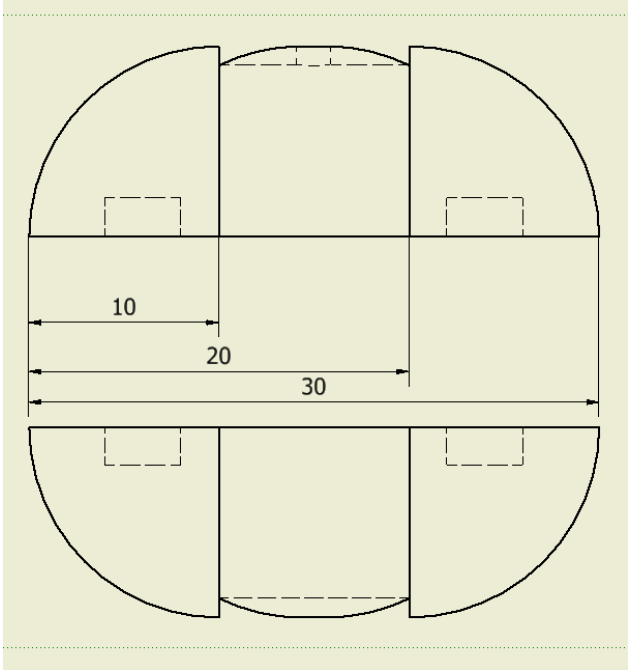
Introduction:

The **Chain** feature is used to make a chain of **Dimensions**.

In order to use **Chain**, locate the **Dimension** tab at the top of the screen:



To locate the other **Chain** features; Click on the arrow under "**Chain**", and a drop-down menu will appears where you can see all the **Chain** features.



Here is an example of what a **Chain** might look like.

Chain, select the desired lines. Then right-click and select "**Continue**". Then select the **placement** with a left click.

Select your **Origin** by right-clicking close to the desired line and select "**Make Origin**", as **Origin** is the line the dimensions are based on. Then right-click when you're done and select "**Create**".

If you want to **Rearrange** your **Chain**, select all the dimensions and right-click and select "**Arrange Dimensions**".

Chain Set:

Chain Set feature is used to make a group of **Chain**. That means when you select one, all are selected.

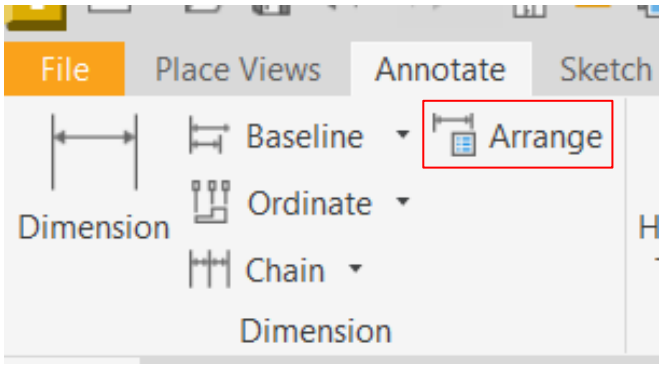
Now when choosing a **Chain Set**. To delete only one **Dimension**, right-click on the desired one and select "**Delete Member**". You can also "**Add Member**", "**Arrange**".

Arrange

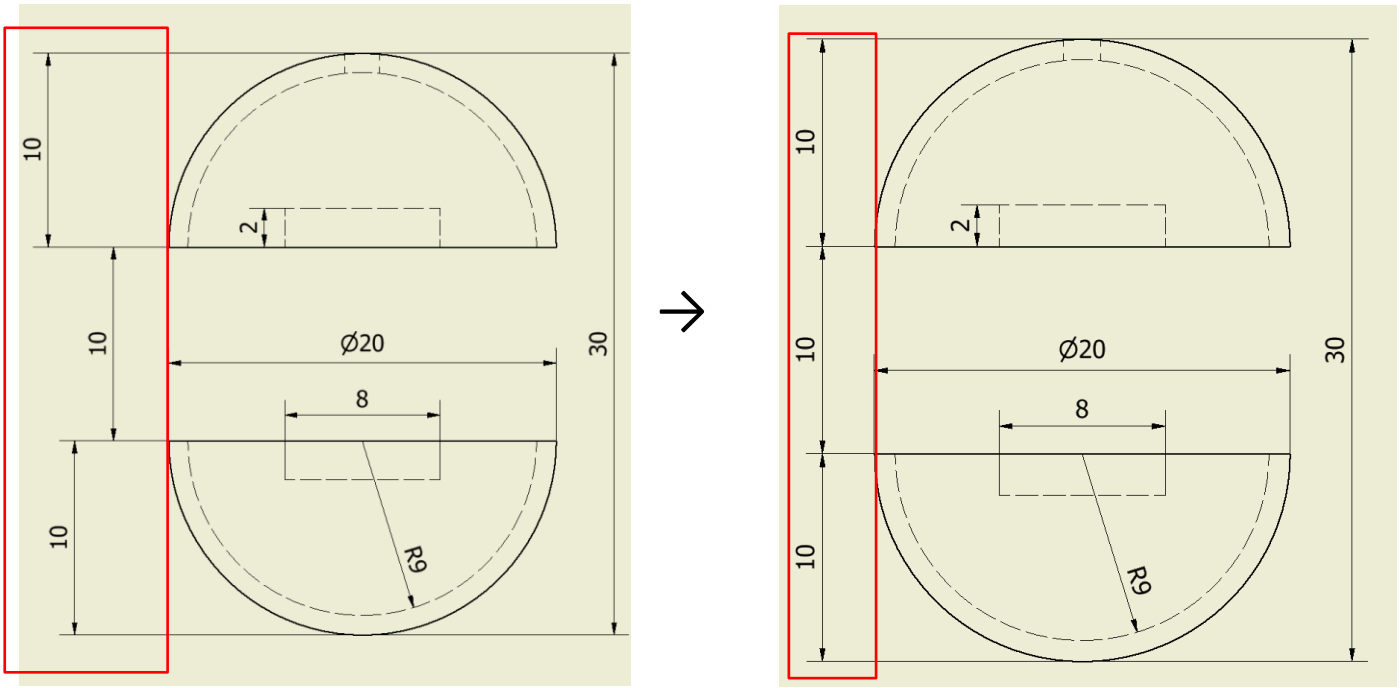
Introduction:

The **Arrange** feature is used to set up the **Dimensions** nicely, but this does not work on a **Dimension Set**.

In order to use **Arrange**, locate the **Dimension** tab at the top of the screen:



Select some desired **Dimensions** on one of the sides of your **View**. Then right-click and select "**OK**".

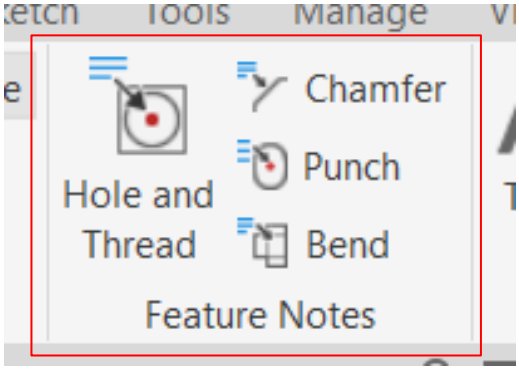


Feature Notes

Introduction:

The **Feature Notes** functions are used to set the **Dimension** of a **Modify** feature.

In order to use the **Feature Notes** functions, locate the **Annotate** → **Feature Notes** tab at the top of the screen:

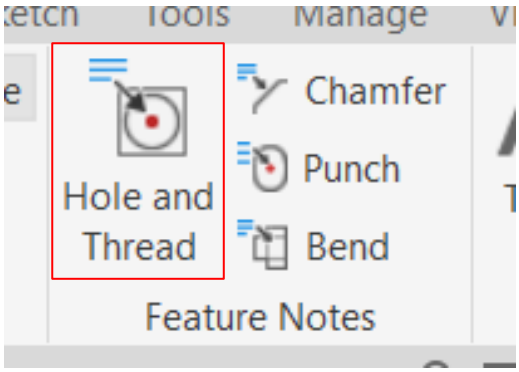


Hole and Thread

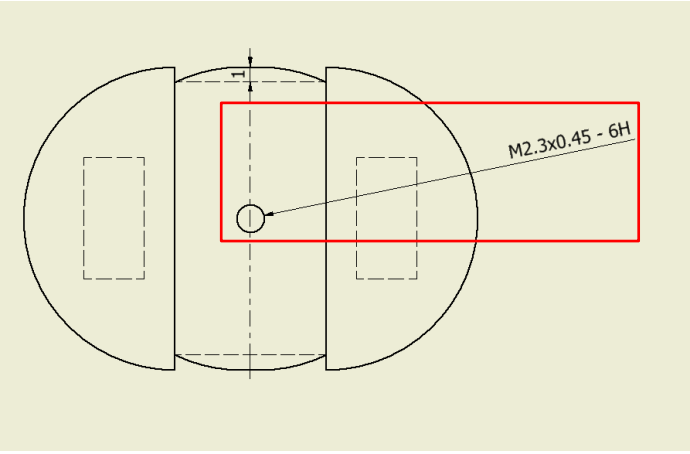
Introduction:

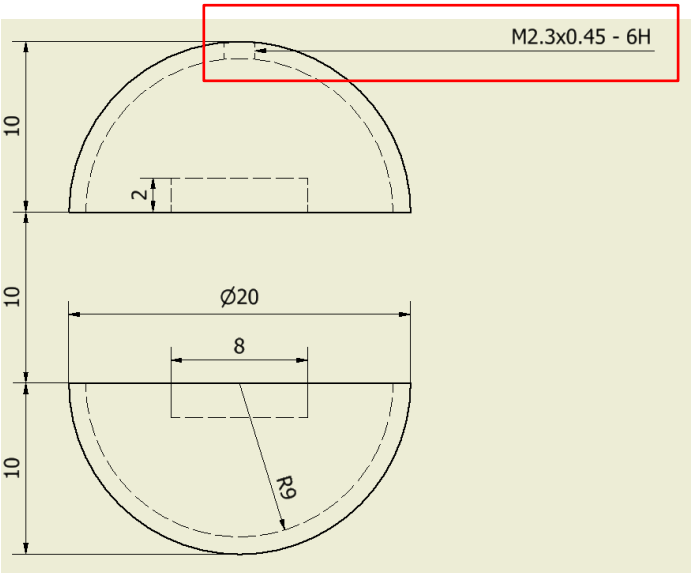
The **Hole and Thread** feature is used to set **Dimension** on a **Hole** or **Thread**.

In order to use **Hole and Thread**, locate the **Feature Notes** tab at the top of the screen:



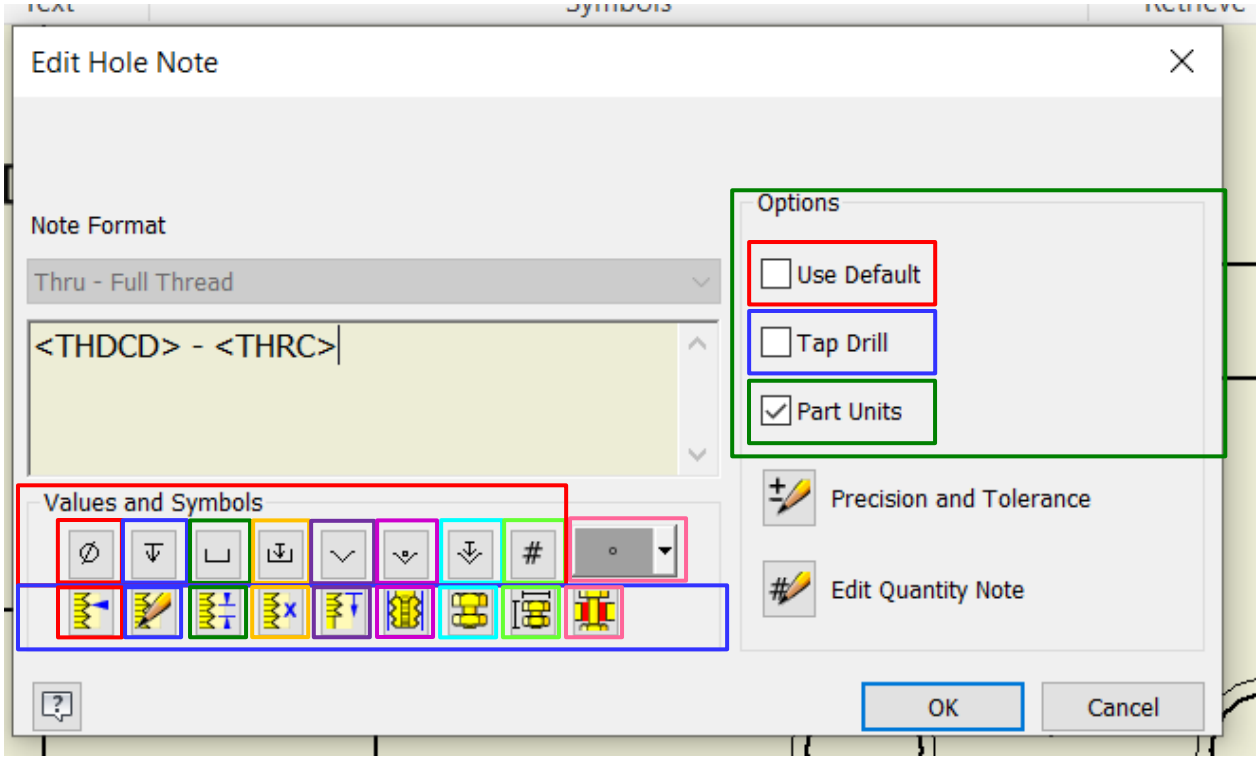
Select **Hole and Thread**. Then choose a **Hole** or **Thread**, below there are 2 examples of how. In the example you can see that you can select the hole in 2 different **Views**.





Note:

To modify the **Hole and Thread Note**; right click on a **Dimension** and select "**Edit Hole/Thread Note**". Afterwards, a box appearss, it looks like this:



Red = Values 1

- Red = **Hole Diameter** value.
- Blue = **Hole Depth** value.
- Green = **Counterbore / Spotbore Diameter** value.
- Yellow = **Counterbore / Spotbore Depth** value.
- Purple = **Countersink Diameter** value.
- Magenta = **Countersink Angle** value.
- Cyan = **Countersink Depth** value.
- Lime = **Quantity** value**.**

Pink = Symbols.

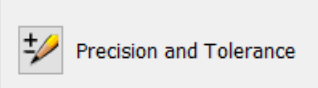
Blue = Values 2

- Red = **Thread Designation** value.
- Blue = **Custom Designation** value.
- Green = **Thread Pitch** value.
- Yellow = **Thread Class** value.
- Purple = **Thread Depth** value.
- Magenta = **Tap Drill Diameter** value.
- Cyan = **Fastener Type** value.
- Lime = **Fastener Size** value.
- Pink = **Fastener Fit** value.

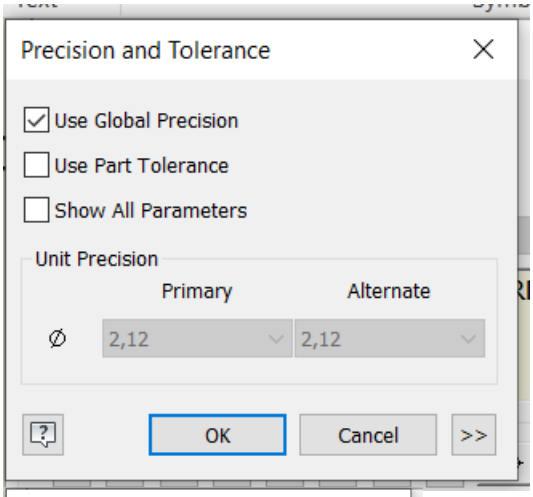
Green = Placement of the tekt.

- Red = Size.
- Blue = Colour.
- Green = **Bold** font.

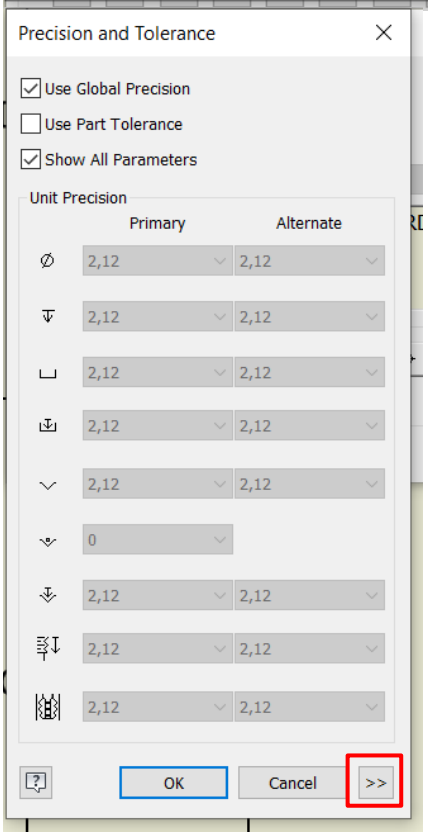
Precision and Tolerance:



Here, a box appearss after you press ↑:

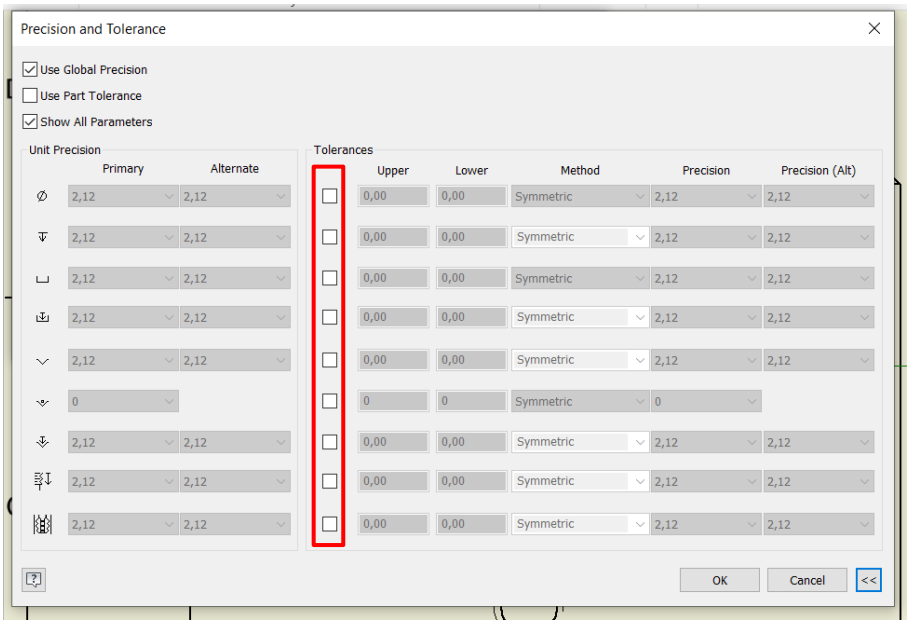


Here you can choose your **Unit Precision** and **Tolerance**. To display all parameters, select "**Show All Parameters**":



Here you can choose your **Parameters Precision**, but only if you select **Global Precision** off, you do that by deselecting "**Use Global Precision**".

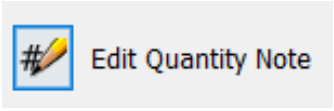
To select a precise **Tolerance**, click on the symbol in the red box:



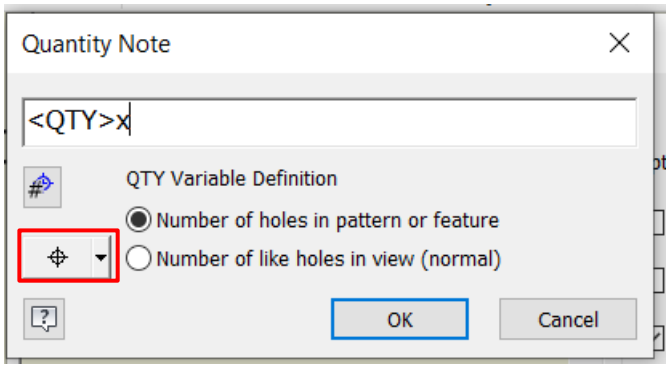
To change a specific **Tolerance**, select the red boxes on.

To finish select OK.

Quantity Note:



Can only be used if a **Quantity Note** value has been inserted. Select ↑, then a box appearss with some options:



Here you can change "x" to a "*" symbol instead, or you can insert a symbol = Red.

Otherwise, you have 2 possibilities:

1 = The number of holes that are in a **Pattern** or **Feature**.

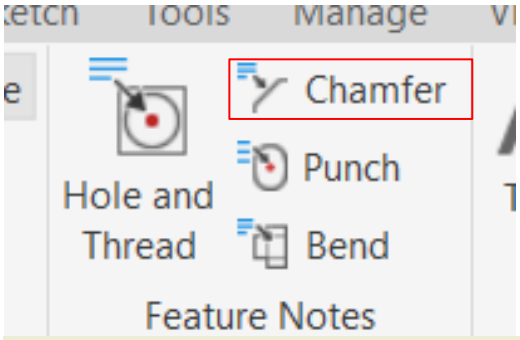
2 = The number of holes there are in a **View**.

Chamfer

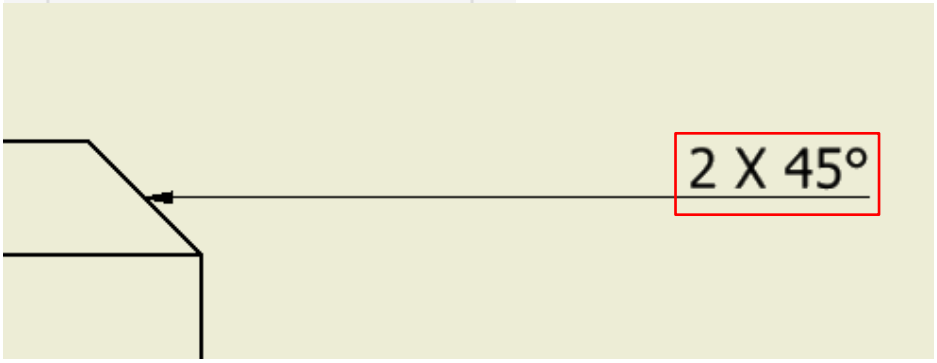
Introduction:

The **Chamfer** feature is used to cut corners of an existing shape at a certain angle.

In order to use **Chamfer**, locate the **Feature Notes** tab at the top of the screen:



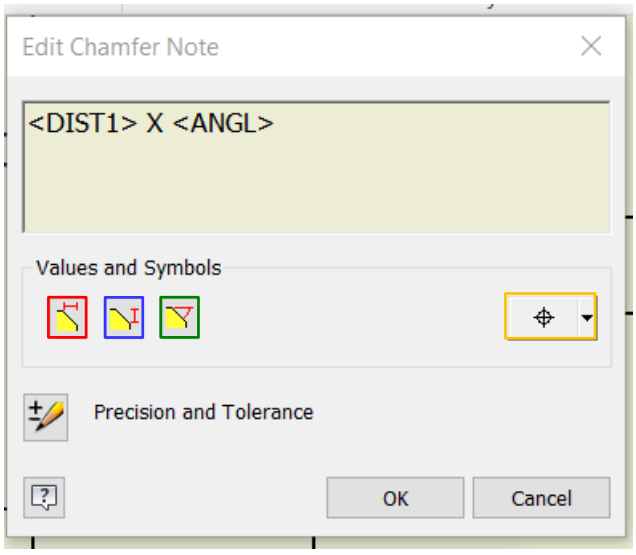
Select the **Chamfer**. Then choose a line that meets the **Chamfer**. Then right-click and select "**OK**" to finish. Below is an example of a **Chamfer Dimension**.



Here it is 2 millimetres long. And has a 45° degree slope.

Note:

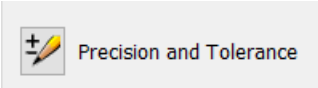
To modify the **Chamfer Note**; right click on **Dimension** and select "**Edit Chamfer Note**". Afterwards, a box appearss, it looks like this:

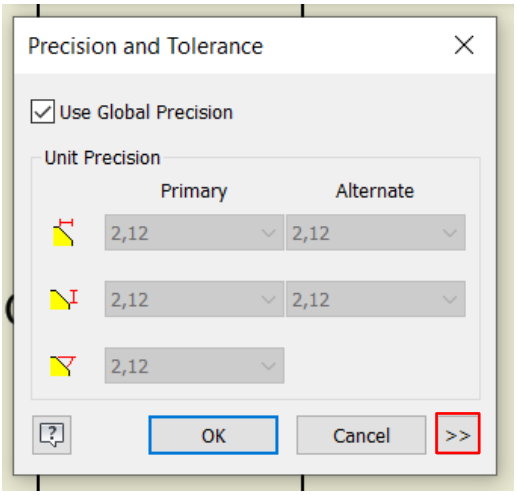


There are 3 values here:

Red = Length 1.
Blue = Length 2.
Green = Angle.
Yellow = Symbols.

Precision and Tolerance:





Here, a box appearss after you press ↑:

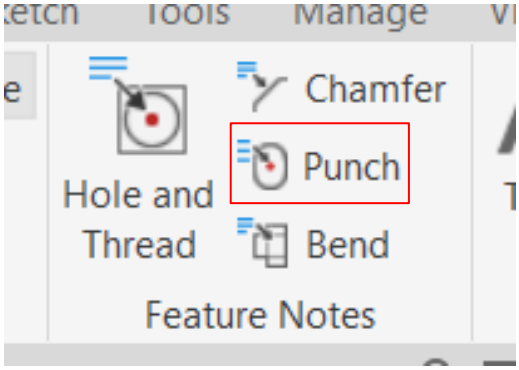
To adjust, deselect "**Use Global Precision**". To change the **Tolerance**, press the red symbol.

Punch

Introduction:

The **Punch** feature is used to inform about **Punches**. This only works with a **Sheet Metal Drawing** and **Part** (**idw**.).

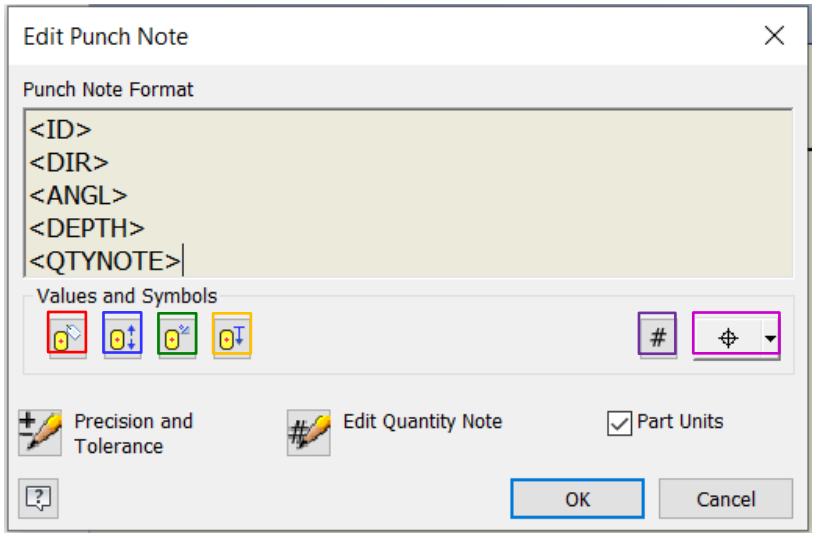
In order to use **Punch**, locate the **Feature Notes** tab at the top of the screen:



Select **Punch** and click on the desired **Punch** geometry in a **Flat Pattern View**.

Note:

To modify the **Punch Note**; right click on **Dimension** and select "**Edit Punch Note**". Afterwards, a box appearss, it looks like this:

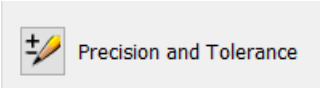


There are 4 values here:

Red = **ID**. (Name)
Blue = Direction.
Green = Angle.
Yellow = Depth.

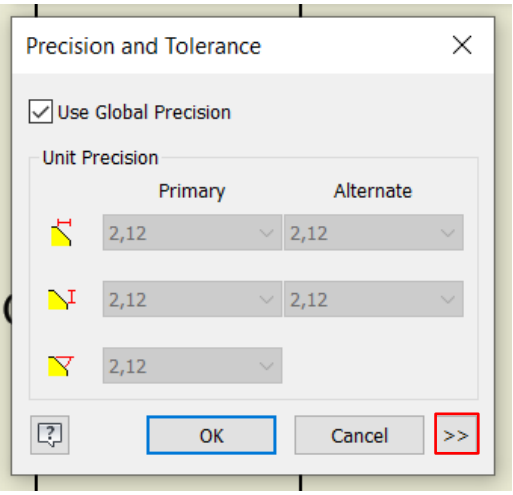
Purple = Quantity.
Magenta = Symbols.

Precision and Tolerance:

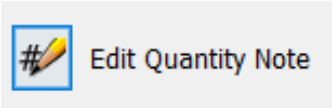


Here, a box appearss after you press ↑:

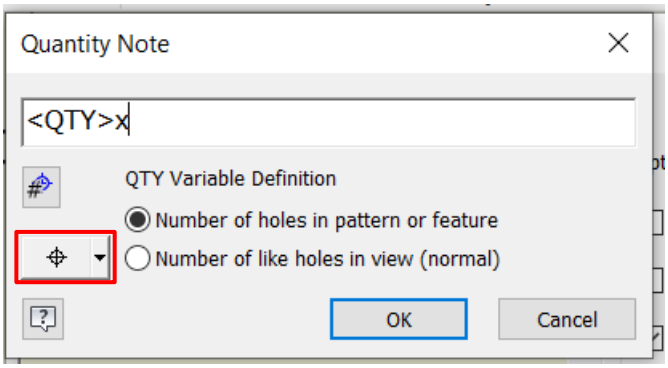
To adjust, deselect "**Use Global Precision**". To change the **Tolerance**, press the red symbol.



Quantity Note:



Can only be used if a **Quantity Note** value has been inserted. Select ↑, a box appearss with some options:



Here you can change "x" to a "*" symbol instead, or you can insert a symbol = Red.

Otherwise, you have 2 possibilities:

1 = The number of holes that are in a **Pattern** or **Feature**.

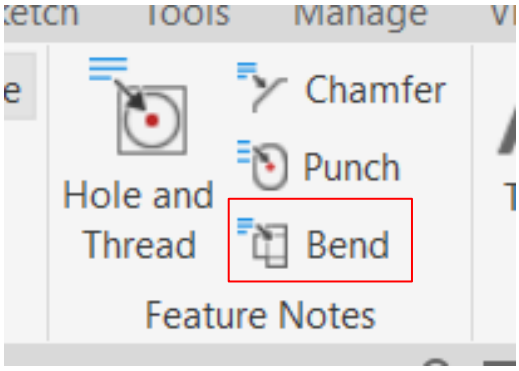
2 = The number of holes there are in a **View**.

Bend

Introduction:

The **Bend** feature is used to inform about **Bend**. This only works with **Sheet Metal Drawing** and **Part** (**idw.**).

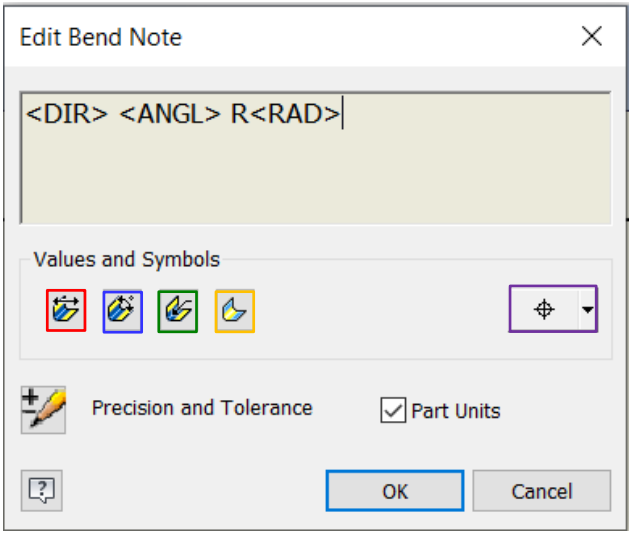
In order to use **Bend**, locate the **Feature Notes** tab at the top of the screen:



Select **Bend** and click on the desired **Bend** geometry (dashed lines) in a **Flat Pattern View**.

Note:

To modify the **Bend Note**; right click on **Dimension** and select "**Edit Bend Note**". Afterwards, a box appearss, it looks like this:

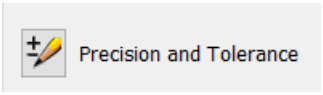


There are 4 values here:

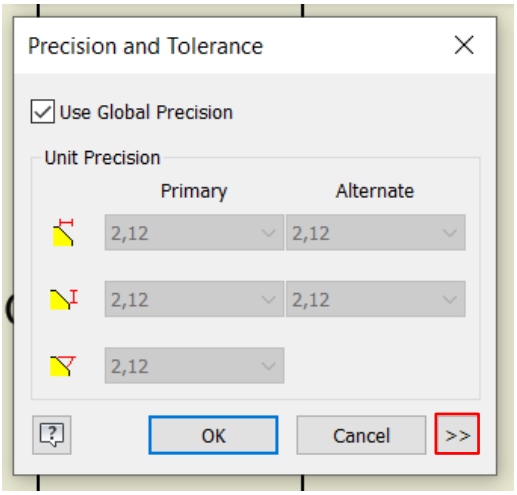
Red = Direction.
Blue = Angle.
Green = Radius.
Yellow = kfactor.

Purple = Symbols.

Precision and Tolerance:



Here, a box appearss after you press ↑:



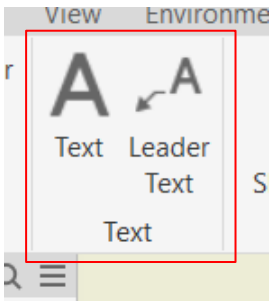
To adjust, deselect "**Use Global Precision**". To change the **Tolerance**, press the red symbol.

Text

Introduction:

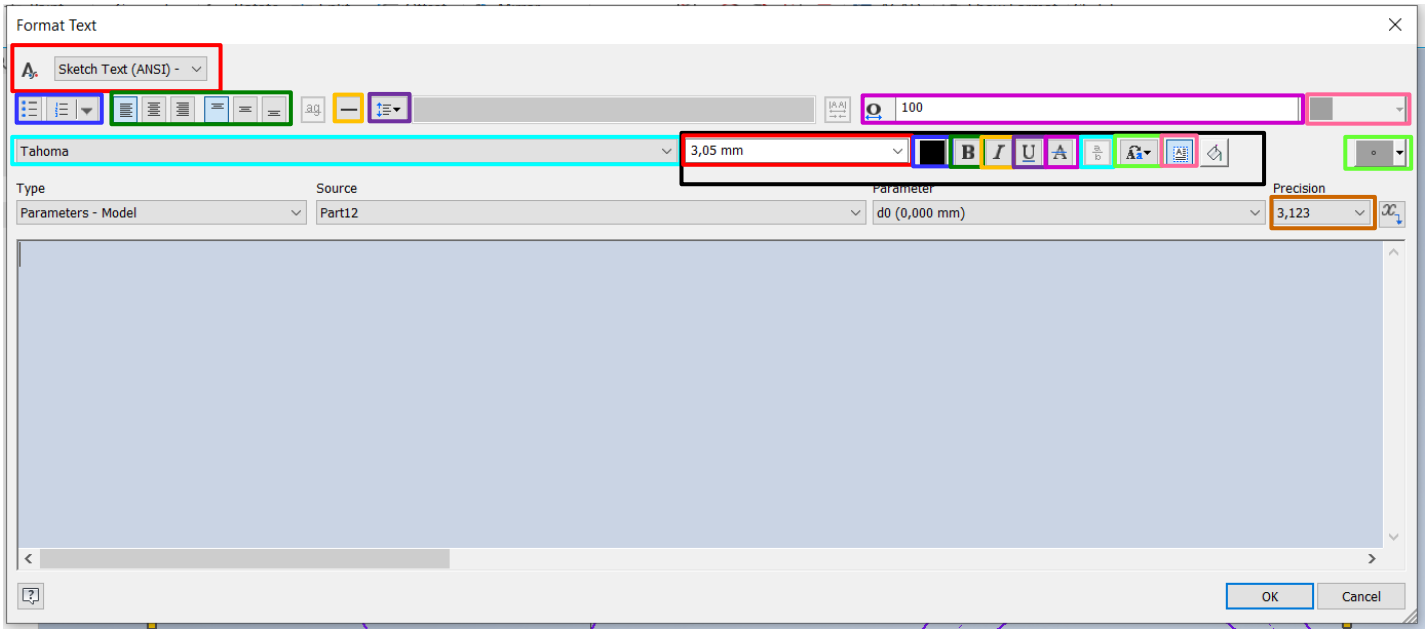
The **Text** features are used to create text.

In order to use the **Text** features, locate the **Annotate → Text** tab at the top of the screen:



- **Shortcut key** = T

This is what **Properties** looks like:



Red = Profiles. A text format can be saved here.
Blue = Lists. Arranges text in a list format.
Green = Placement of the text.
Yellow = Single Line Text.
Purple = Text Spaceing.
Magenta = Letter spacing. The width between the letters.
Cyan = Font.

Black:

- Red = Size.
- Blue = Colour.
- Green = Bold.
- Yellow = Italic.
- Purple = Underline.
- Magenta = Strikethrough.
- Cyan = Stack.
- Lime = lowercase / UPPERCASE / Titel Case.
- Pink = Turned on, makes the Direction horizontal up (so it's readable). If turned off, then the second Pink box unlocks.

Lime = Insert Symbols.
Pink = To open → turn off the first Pink box. Then its the Direction of the text.
Brown = Number of decimal there are.

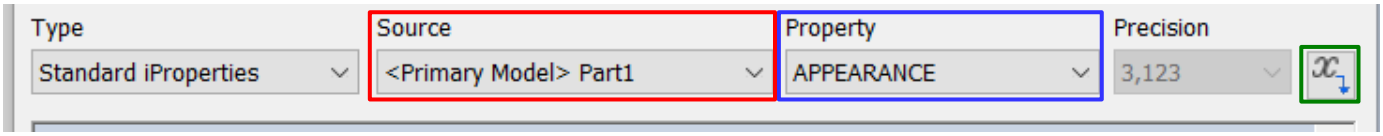
If a change doesn't work, remember to highlight the text.

Standard iProperties

Standard iProperties is Inventor's way of using **Part** values. So, for example can you have a **Material** value engraved in the **Part**. If you change **Material** in the **Part**, the text is also changed. (They are related).

Property example = **Designer/Part Number/Material**.

In order to use, select "**Standard iProperties**" under **Type**:

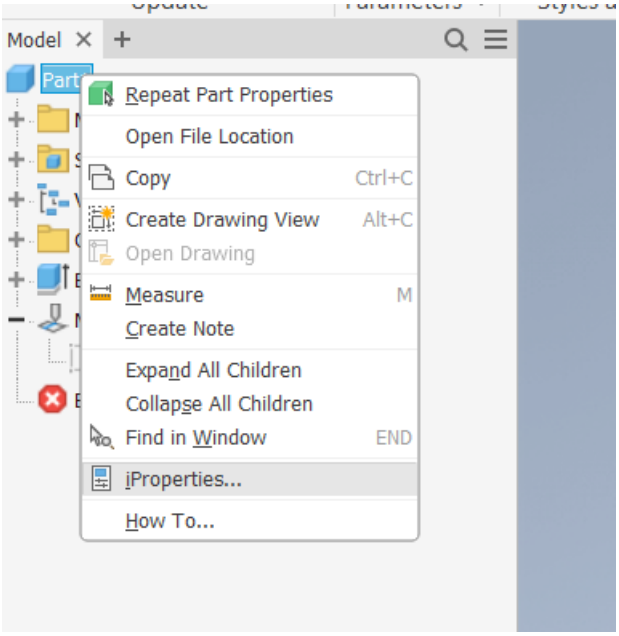


Source = Which **Part** Inventor takes **Property** from.
Property = Which **Property** to insert.
Green = To add, select the ↑ 2 and then press here.

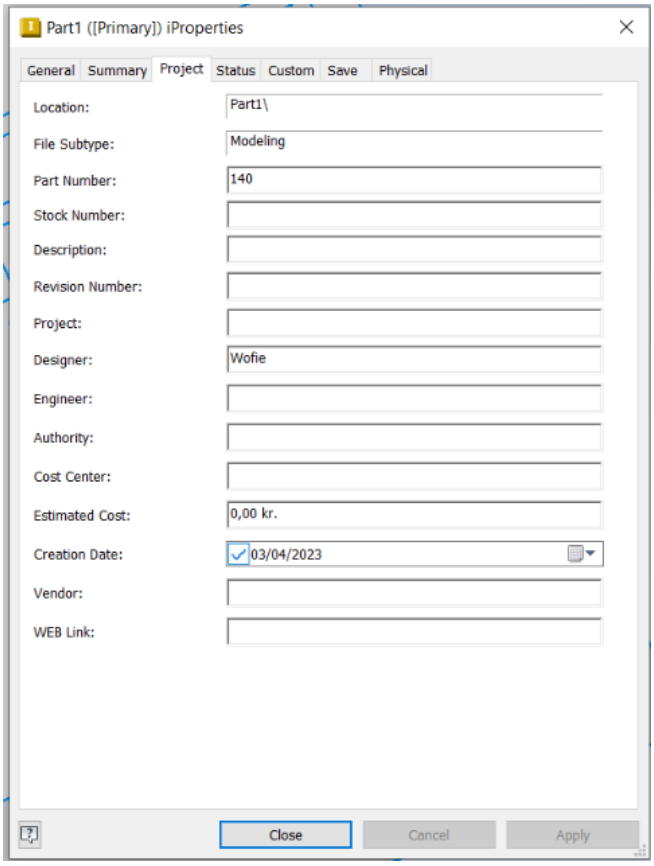
If a particular Property doesn't work:

You can always change/edit for example **Part Number** under **iProperties**. To change open **iProperties**.

Start by right-clicking on one's **Part** in the history →



Then click on **Project** at the top of the new window ↓.



See more under **Drawings** → **Template** → **iProperties**.

Leader Text

Leader Text is text that follows a shape.

To apply, select a geometry/or any placement. Here a **Leader** is created (A line with an arrow). Then left-click with the desired elbow points (normally you choose 1, we recommend highest 2). Then right-click and select "**Continue**".

The **Text** features works in the same way as in **Text**.

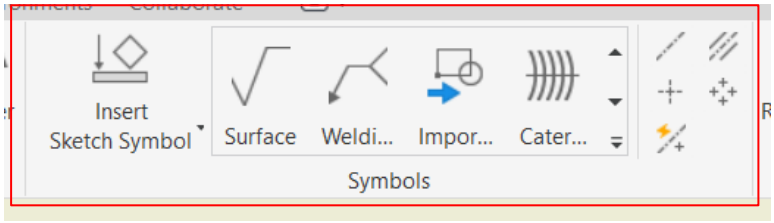
- **Shortcut key** = Ctrl + Shift + T

Symbols:

Introduction:

The **Symbols** features are used to insert complicated symbols.

In order to use the **Symbols** features, locate the **Annotate → Symbols** tab at the top of the screen:

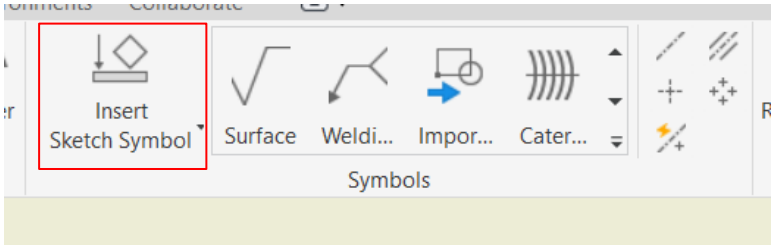


Insert Sketch Symbol

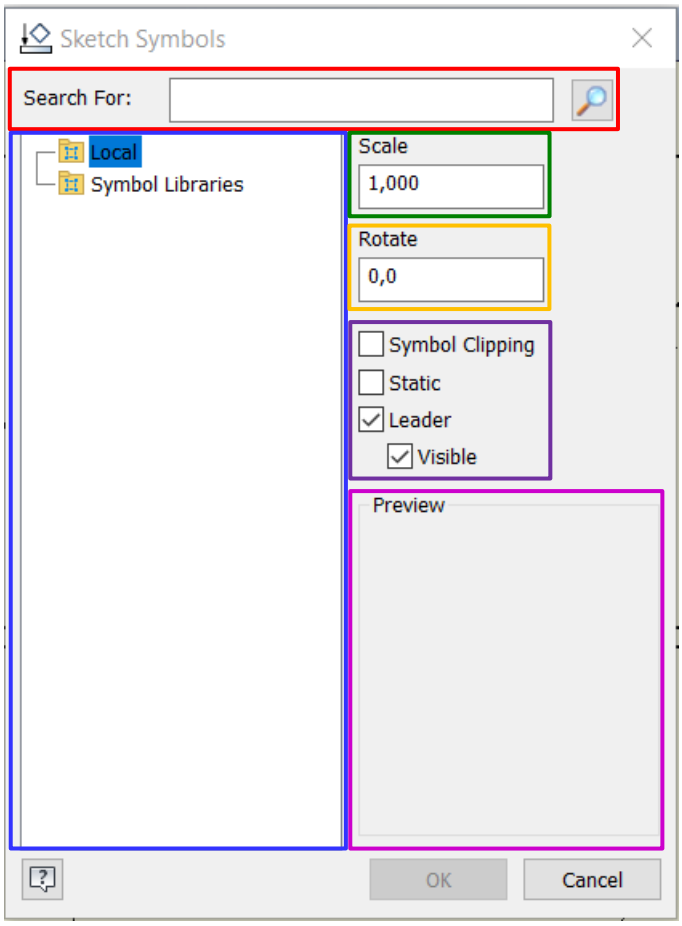
Introduction:

The **Insert Sketch Symbol** feature is used to insert **Sketch Symbols** that you create yourself.

In order to use the **Insert Sketch Symbol**, locate the **Symbols** tab at the top of the screen:



Select **Insert Sketch Symbol**. When you press **Insert Sketch Symbol**, a box appearss:



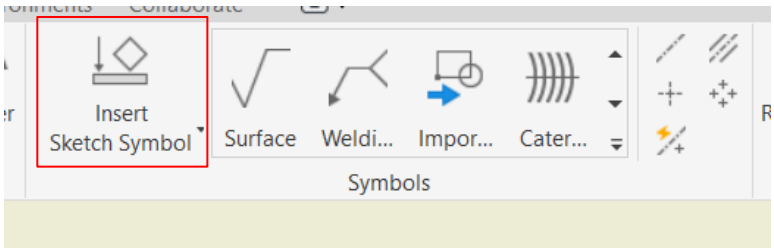
Red = Search box.
Blue = The menu / overview of Symbols.
Green = Scale.
Yellow = Rotation.
Purple = Options.
Magenta = Preview.

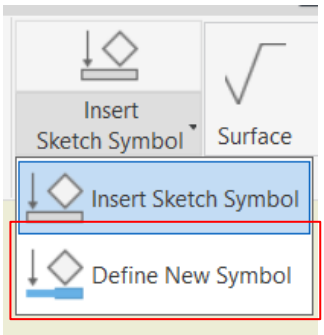
Define Sketch Symbol

Introduction:

The **Define Sketch Symbol** feature is used to create Sketch Symbols.

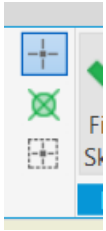
In order to use the **Define Sketch Symbol**, locate the **Symbols** tab at the top of the screen:





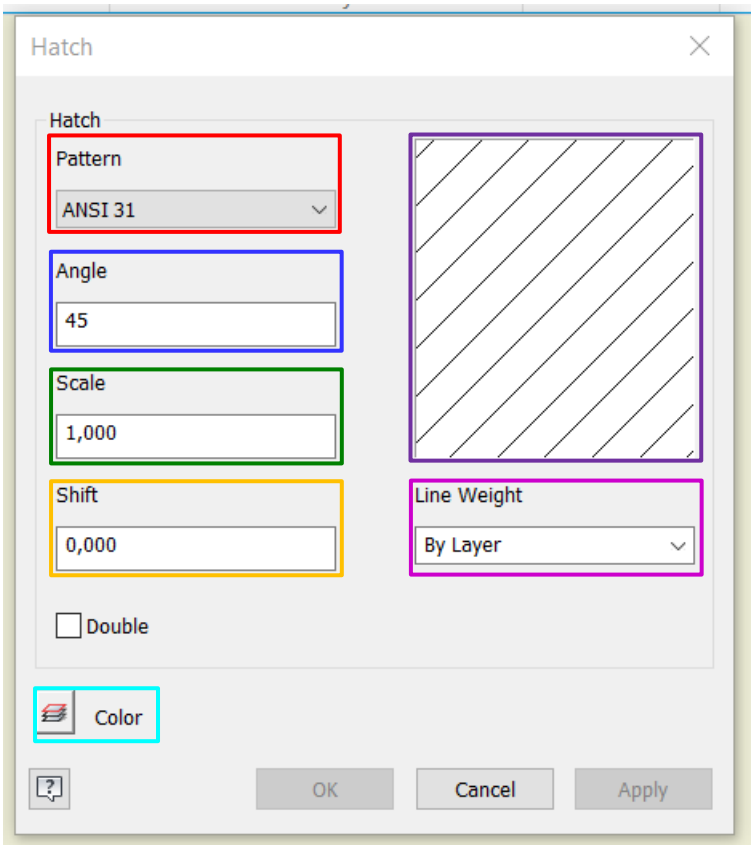
Select **Define New Symbol**. Afterwards, the **Sketch** tab becomes activated, and then you can draw your own symbol.

There are now 2 new types of **Points**:



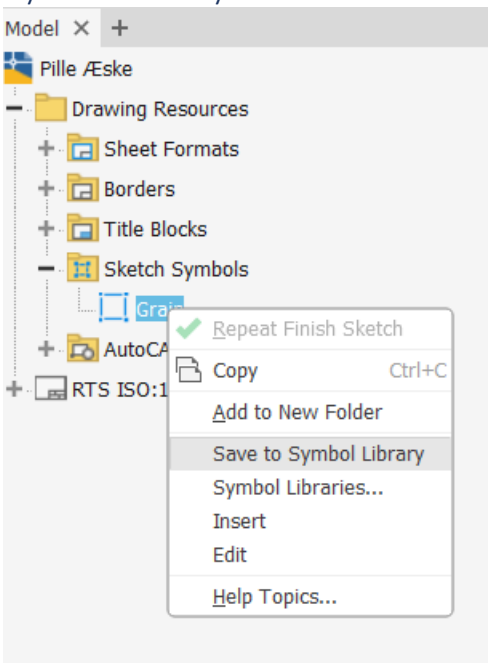
1. **Center Point**.
2. **Connection Point Grip** = defines a point that can snap to other geometry.
3. **Insertion Point Grip** = defines a point that snaps to the mouse when inserting a symbol.

There is also a new feature **Hatch Region**, this feature fills in regions with pattern or colour:



Red = Search box.
Blue = Angle.
Green = Scale.
Yellow = Shift.
Purple = Preview.
Magenta = Line Weight.
Cyan = Colour.

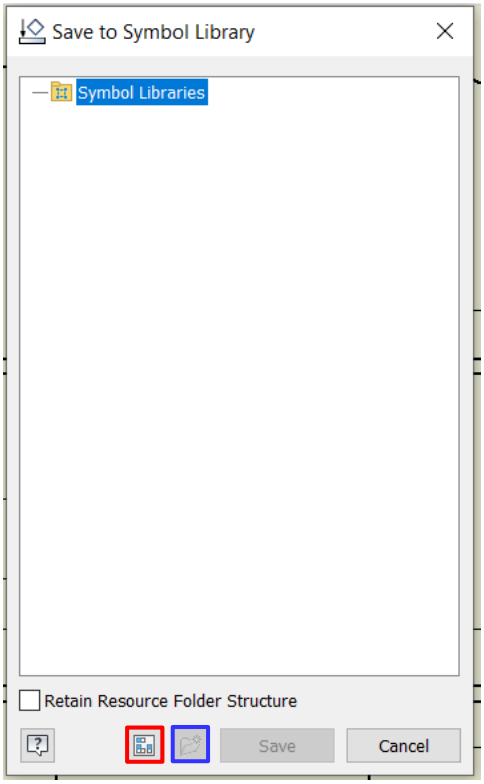
Symbol Library:



To use, go to history and expand **Drawing Resources**, then expand **Sketch Symbols** → select the desired **Sketch Symbol** → right click and select "**Save to Symbol Library**".

1. Create new library: press red.
2. Name library.
3. Click save.

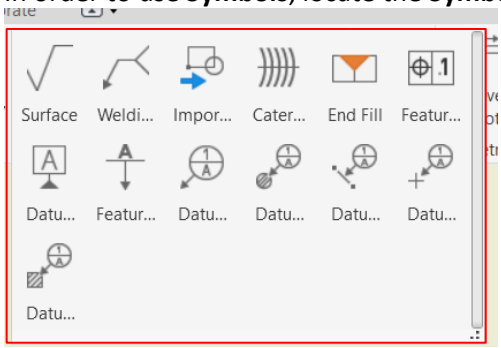
Blue = New folder.



Symbols:

Introduction

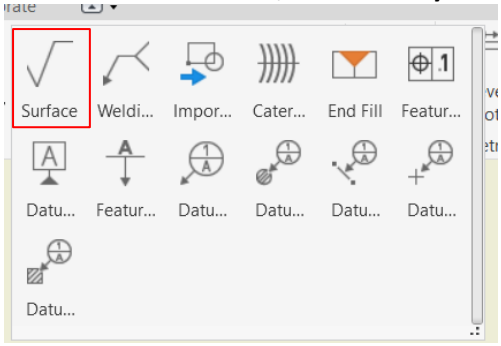
In order to use **Symbols**, locate the **Symbols** tab at the top of the screen:



All features in the red box are **Symbols**. If you hover the mouse over a symbol, an English explanation of the symbol will appears.

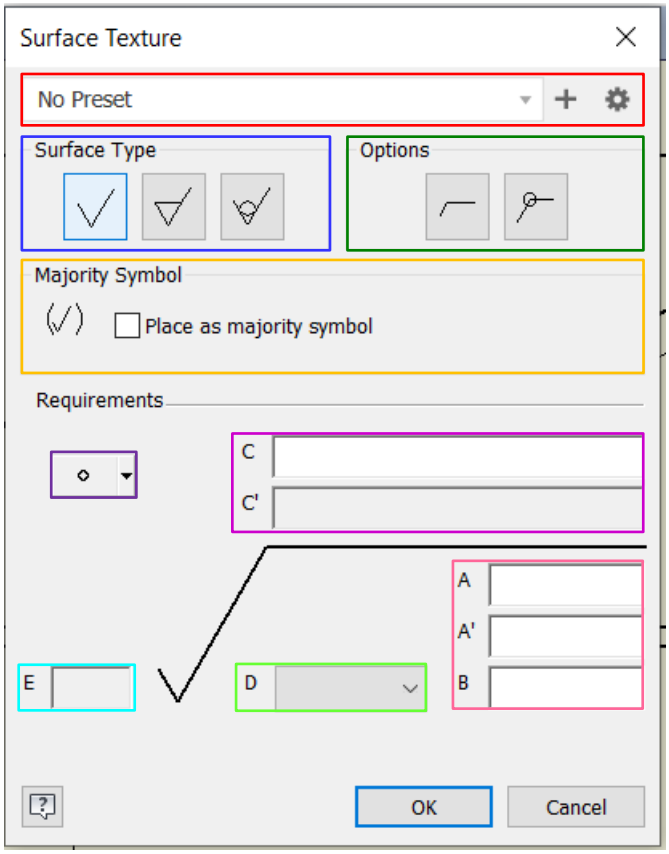
Surface

In order to use **Surface**, locate the **Symbols** tab at the top of the screen:



Select **Surface** and a geometry. Afterwards, you have a choice whether you want a **Leader** or not. When you want to finish, right-click and select "**Create**". Afterwards, a box will appears:

If a **Leader** is desired; left click to place and then right click and select "**Create**" to finish.



Red = Here you can create new, change and save **Presets**.

Blue = Choose between surface types:

- Basic Texture.
- Removal of material Required.
- Removal of material Prohibited.

Green = Choose between:

- Forced Tail.
- All-around.

Yellow = Adds and removes, symbols to the right of the primary.

Purple = Symbols.

Magenta = Process requirement.

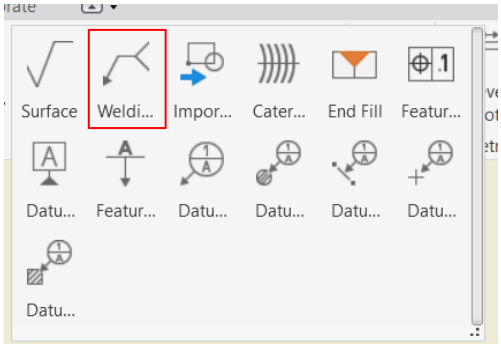
Cyan = Machine allowance.

Lime = Requirement.

Pink = Surface Symbols.

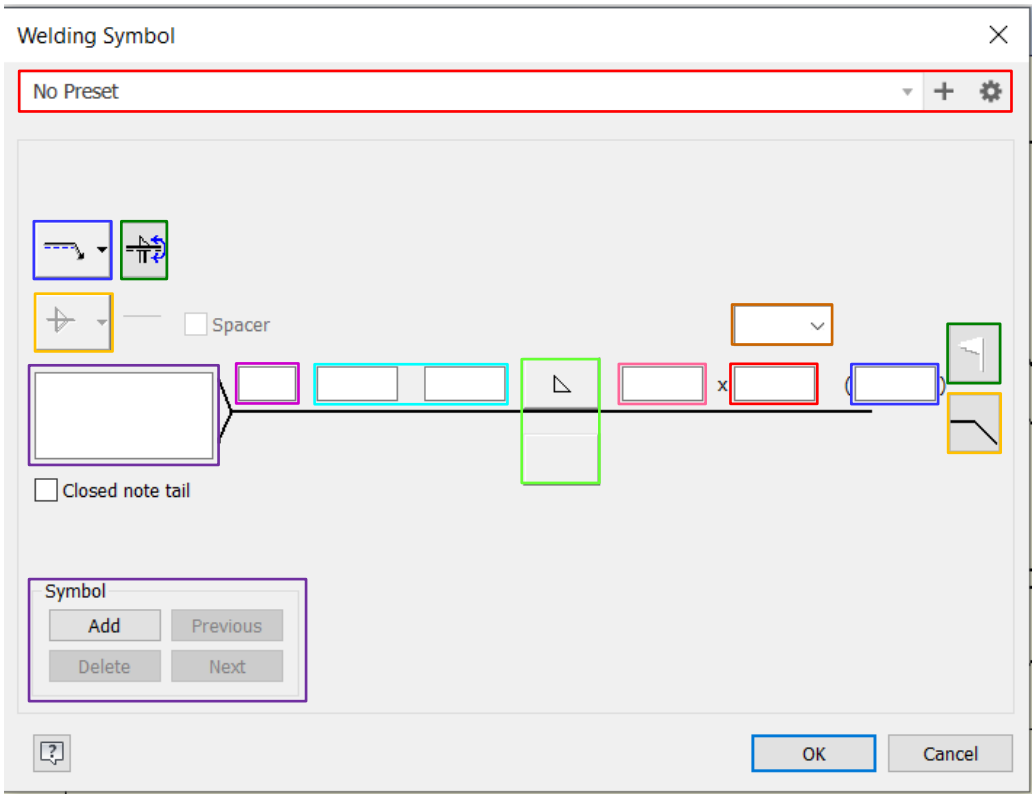
Welding Symbol

In order to use **Welding Symbol**, locate the **Symbols** tab at the top of the screen:



Select **Welding** and a geometry. Afterwards, you have a choice whether you want a **Leader** or not. When you want to finish, right-click and select "**Create**". Afterwards, a box will appears:

If a **Leader** is desired; left click to place.

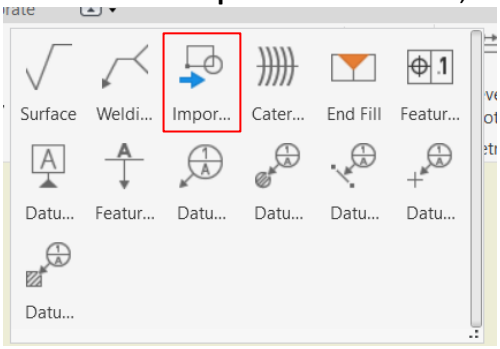


Red = Here you can create new, change and save **Presets**.
Blue = Change Identification Line.
Green = Flip.
Yellow = Tail Note.
Purple = Prefix.
Magenta = No Stagger.
Cyan = Leg 1 & 2.
Lime = Welding Symbol.
Pink = Number.
Brown = Contour.

Red 2 = Length.
Blue 2 = Spaceing.
Green 2 = Box weldig symbol.
Yellow 2 = Peripheral weldig symbol.
Purple 2 = Symbol: Add, Delete, Previous, Next.

Import AutoCAD Block

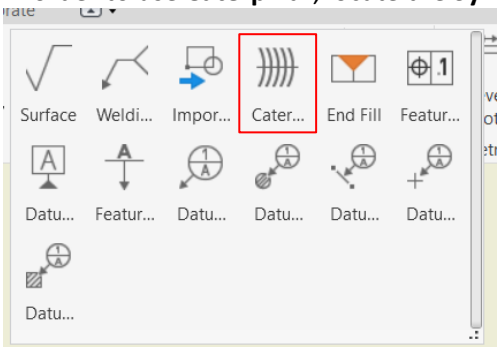
In order to use **Import AutoCAD Block**, locate the **Symbols** tab at the top of the screen:



Import **AutoCAD Symbols** → locate the desired file → click ”Open”.

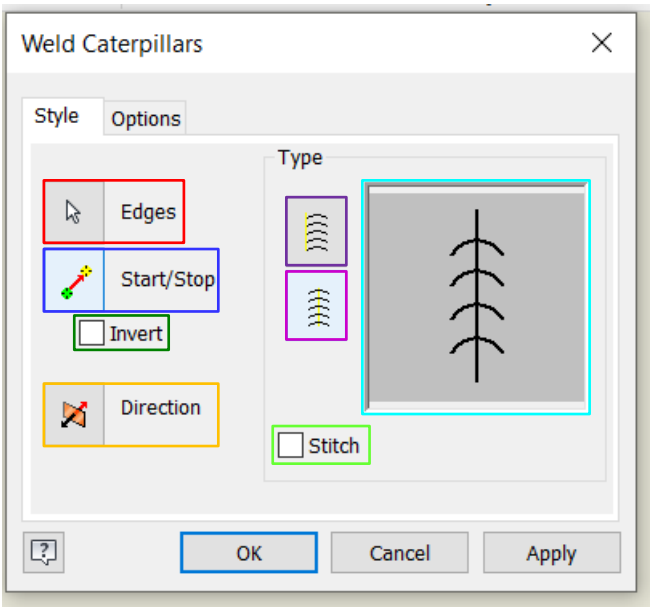
Caterpillar

In order to use **Caterpillar**, locate the **Symbols** tab at the top of the screen:



Select **Caterpillar**, here a box will appears with 2 tabs:

1. Style.
2. Options.



Style:

Red = Enable edge selection.

Blue = Set a start and stop point on the selected edge.

Green = Flips start and stop.

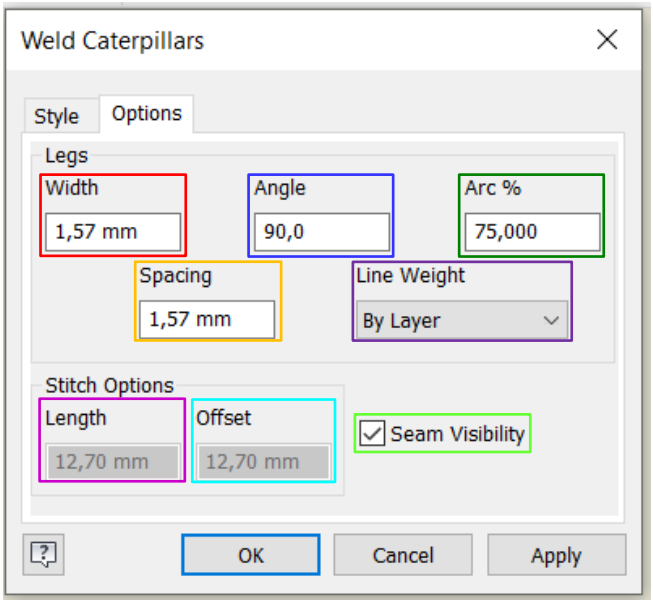
Yellow = Flips the way the **Caterpillar** turns.

Purple = Select **Caterpillar** type **Partial**.

Magenta = Select **Caterpillar** type **Full**.

Cyan = **Caterpillar** type preview.

Lime = On and off, choose whether to include **Stitch**.

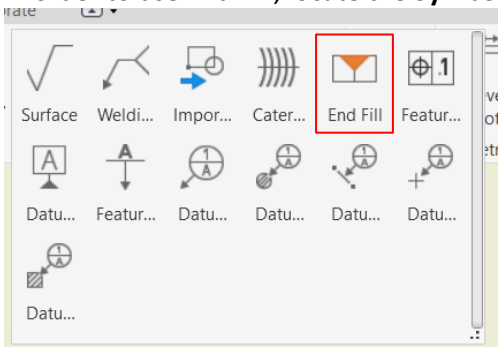


Options:

Red = Width of **Caterpillar** legs.
Blue = The angle of the **Caterpillar** legs.
Green = Arch percentage on the **Caterpillar** legs.
Yellow = Spaceing between **Caterpillar** legs.
Purple = Line weight on the **Caterpillar** legs.
Magenta = Length of **Stitch**.
Cyan = Offset **Stitch**.
Lime = **Seam** visibility.

End Fill

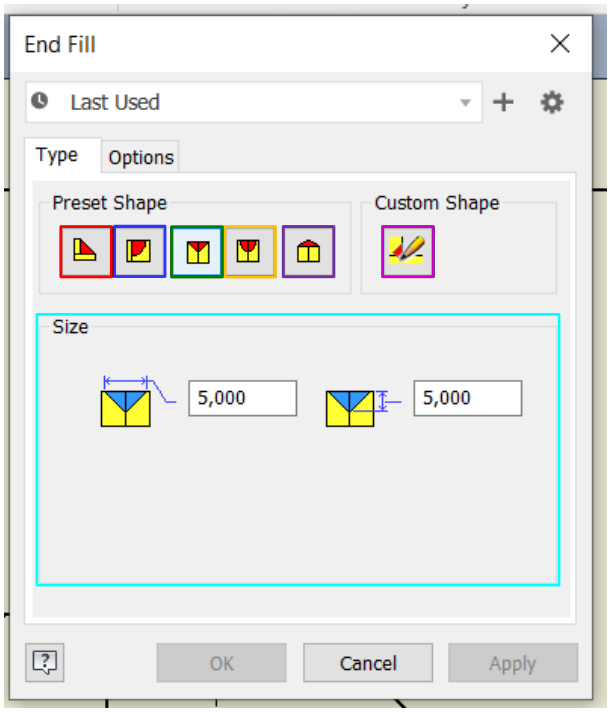
In order to use **End Fill**, locate the **Symbols** tab at the top of the screen:



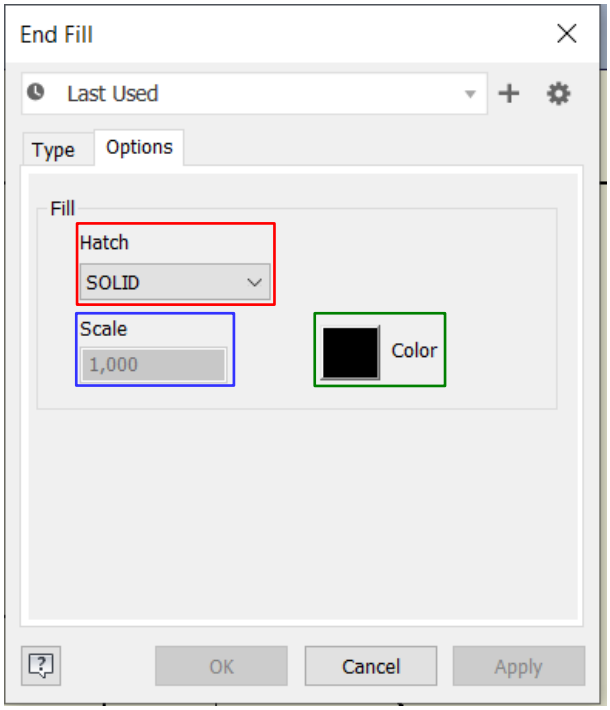
Fills ends, with a solid color / pattern. Select **End Fill**, here a box with 2 tabs appearss:

If a predefined shape is used, select placement (e.g. 2 lines).

If **Custom** shape is used, draw the desired shape.



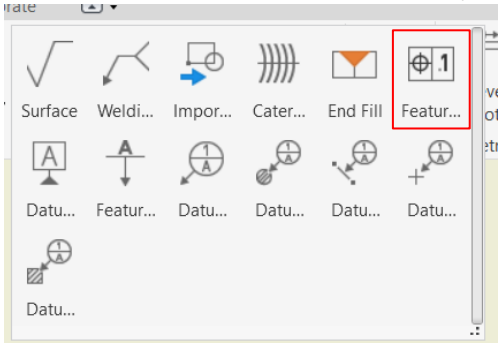
Red = **Fillet** predefined shape.
Blue = **J-Type** predefined shape.
Green = **V-Type** predefined shape.
Yellow = **U-Type** predefined shape.
Purple = **Seam/Dome** predefined shape.
Magenta = **Region/Brugeredineret** shape.
Cyan = Size.



Red = Hatch Type.
Blue = Scale.
Green = Colour.

Feature Control Frame

In order to use **Feature Control Frame**, locate the **Symbols** tab at the top of the screen:

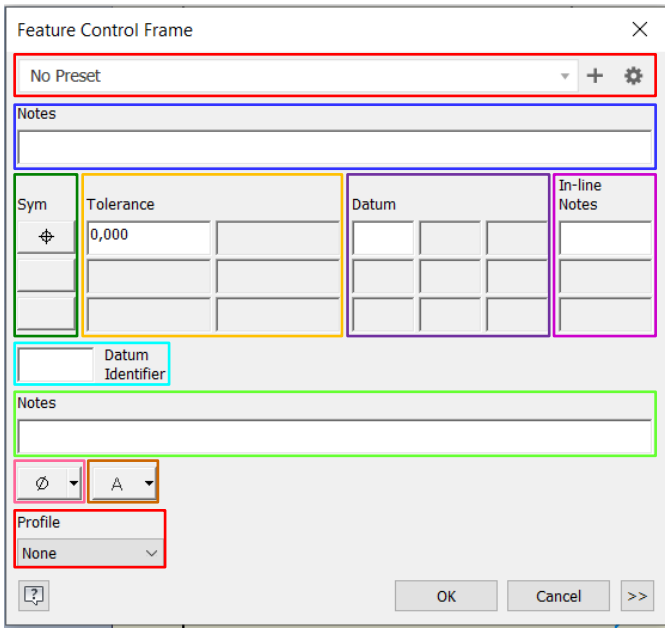


Select **Feature Control Frame** and a geometry. Afterwards, you have a choice whether you want a **Leader** or not. When you want to finish, right-click and select "**Create**". Afterwards, a box will appears:

If a **Leader** is desired; left click to place.

*To modify double click on **Feature Control Frame**.*

- **Shortcut key** = F

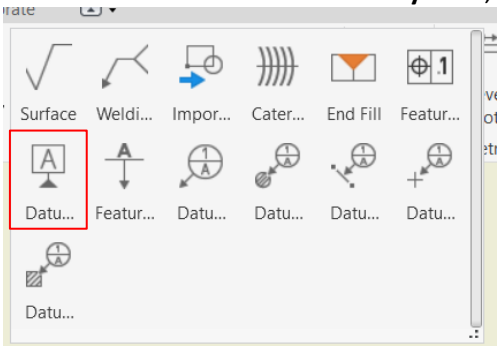


Red = Here you can create new, change and save **Presets**.
Blue = Note.
Green = Geometric characteristic symbol.
Yellow = Tolerance.
Purple = Datum.
Magenta = On line note.
Cyan = Datum Identifier.
Lime = Notes 2.
Pink = Symbol.
Brown = Letter.

Red 2 = Profile.

Datum Identifier Symbol

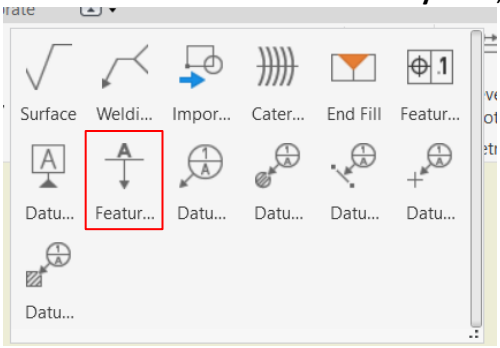
In order to use **Datum Identifier Symbol**, locate the **Symbols** tab at the top of the screen:



Select **Datum Identifier Symbol**, and a geometry or a **Feature Control Frame**. Afterwards, a text box appearss. For more look under **Text**.

Feature Identifier Symbol

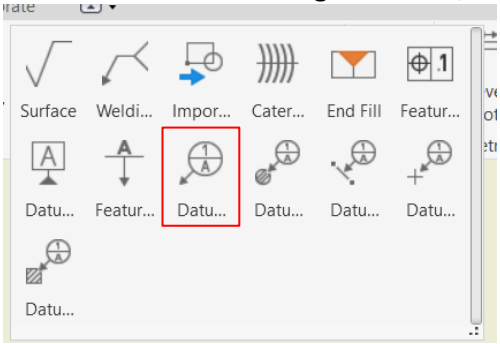
In order to use **Feature Identifier Symbol**, locate the **Symbols** tab at the top of the screen:



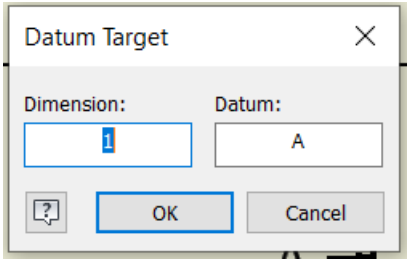
Is used to identify the **Feature Control Frame**, if you have a list.

Datum Target - Leader

In order to use **Datum Target - Leader**, locate the **Symbols** tab at the top of the screen:

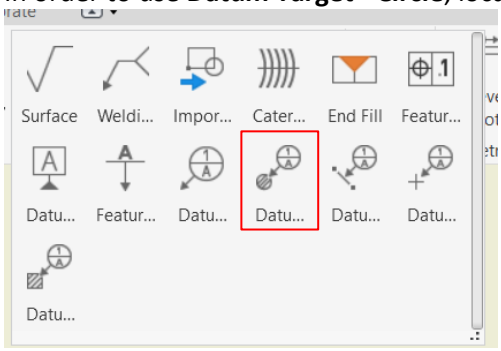


Here a **Datum Target** symbol with Leader is inserted. Select geometry and place Leader with left click. Then right-click and select "**Continue**", a box will appears where you can change the **Dimension** and **Datum** letter:



Datum Target - Circle

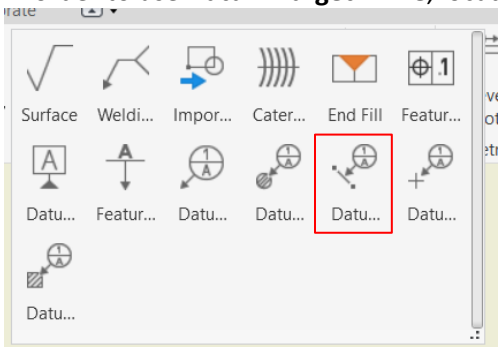
In order to use **Datum Target - Circle**, locate the **Symbols** tab at the top of the screen:



Here, a **Datum Target** with circle marking is inserted. Select the center of a circle → then the edge of the same circle. Now place a **Leader** with left click. Then right-click and select "**Continue**", a box will appears where you can change the **Dimension** and **Datum** letter.

Datum Target - Line

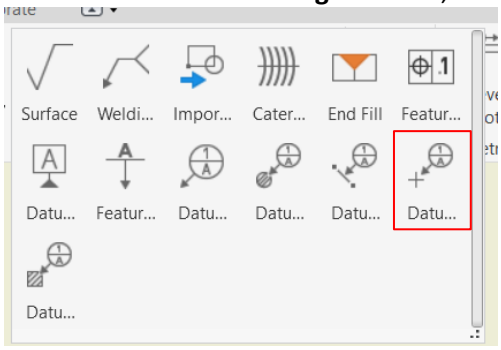
In order to use **Datum Target - Line**, locate the **Symbols** tab at the top of the screen:



Here, a **Datum Target** with a line is inserted. Select start and end point on the desired line and place the Leader with left click. Then right-click and select "**Continue**", a box will appears where you can change the **Dimension** and **Datum** letter.

Datum Target - Point

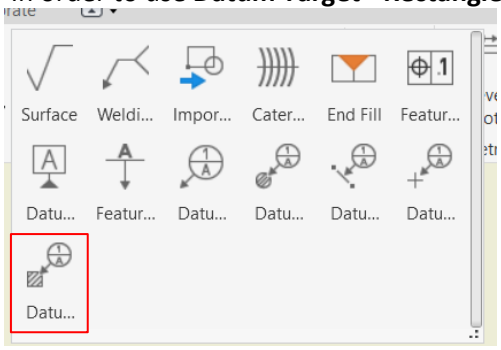
In order to use **Datum Target - Point**, locate the **Symbols** tab at the top of the screen:



Here, a **Datum Target** with a point is inserted here. Select the **Point** and place the Leader with left click. Then right-click and select "**Continue**", a box will appears where you can change the **Dimension** and **Datum** letter.

Datum Target - Rectangle

In order to use **Datum Target - Rectangle**, locate the **Symbols** tab at the top of the screen:



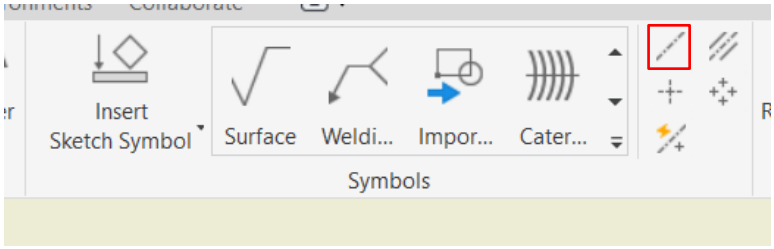
Here, a **Datum Target** with a rectangle is inserted. Select the center of a rectangle → then the edge. Now place a **Leader** with left click. Then right-click and select "**Continue**", a box will appears where you can change the **Dimension** and **Datum** letter.

Centerline

Introduction:

The **Centerline** feature is used to create center lines / lines of symmetry.

In order to use **Centerline**, locate the **Symbols** tab at the top of the screen:



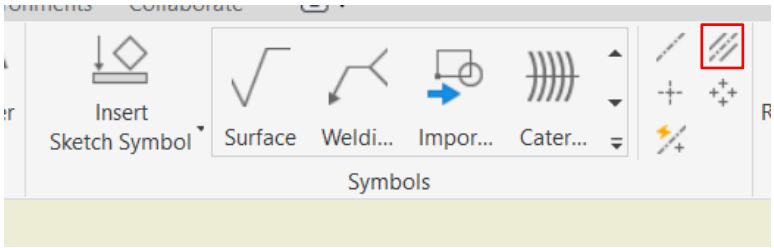
Select, for example 2 points and right click and select "**Create**".

Centerline Bisector

Introduction:

The **Centerline Bisector** feature is used to create center lines / lines of symmetry. Here it makes an automatic line, between 2 lines.

In order to use **Centerline Bisector**, locate the **Symbols** tab at the top of the screen:



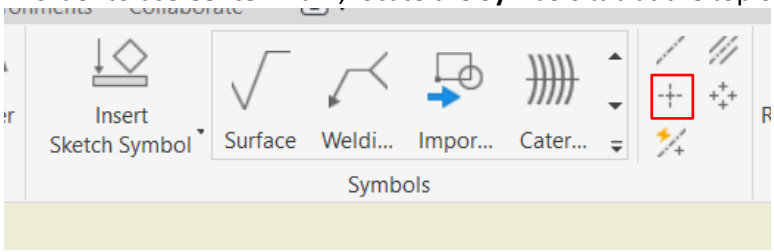
Select 2 lines.

Center Mark

Introduction:

The **Center Mark** feature is used to mark the center of circles or circular shapes.

In order to use **Center Mark**, locate the **Symbols** tab at the top of the screen:



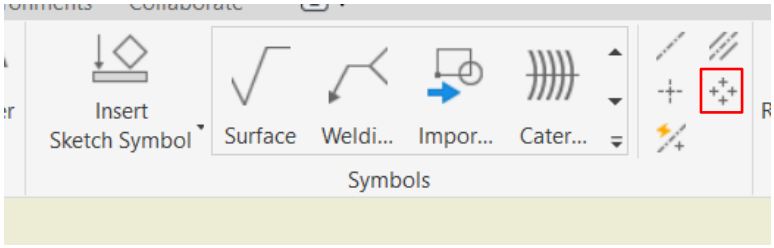
Select a circle / circular shape.

Centered Pattern

Introduction:

The **Centered Pattern** feature is used to make a center mark in a **Pattern**.

In order to use **Centered Pattern**, locate the **Symbols** tab at the top of the screen:



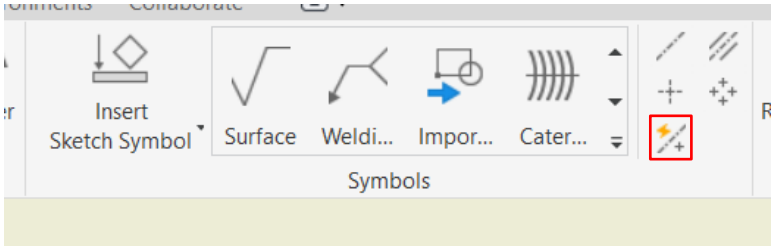
Select the center point / axis. It is used for **Pattern**. And then select the other circles in the **Pattern**.

Automated Centerlines

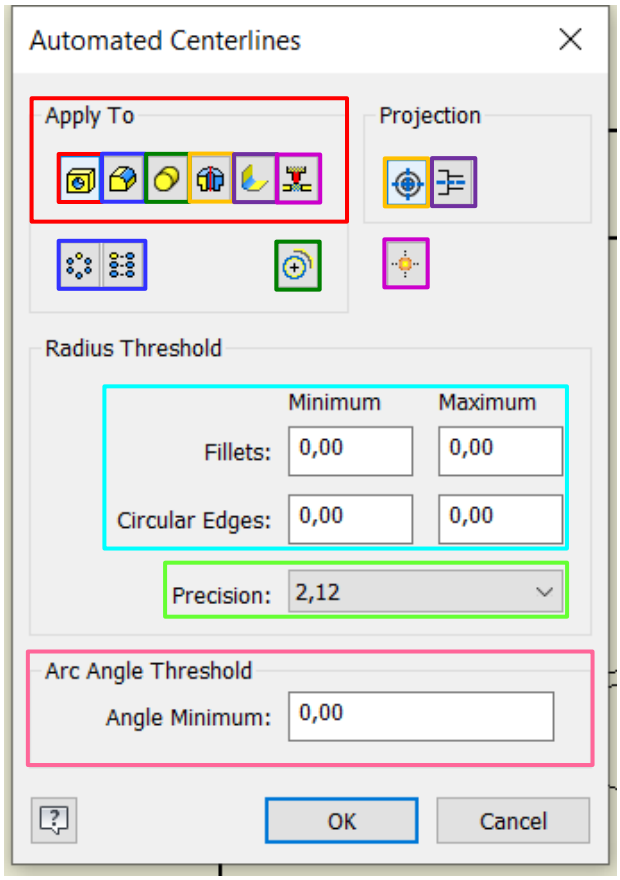
Introduction:

The **Automated Centerline** feature is used to create an automatic **Centerline** over a **View**.

In order to use **Automated Centerline**, locate the **Symbols** tab at the top of the screen:



When you select **Automated Centerlines**, now select the desired **View**. Then a box appearss that looks like this:



Here you can select the **Features** to add center lines and center point markings. Whether **Patterns** should be turned on or off.

Red = Apply To:
- Red = Hole Features.
- Blue = Fillit Features.
- Green = Cylindrical Features.
- Yellow = Revolved Features.
- Purple = Bends (Sheet metal).
- Magenta = Punches (Sheet metal).

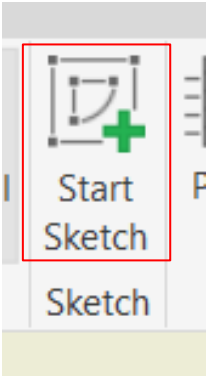
Blue = Patterned Features: Circular, RectanYellowar.
Green = Sketch Geometry.
Yellow = Projection: Object in View, Axis Normal.
Purple = Projection: Object in View, Axis Parallel.
Magenta = Work Features.
Cyan = Specify Radius: Fillets, Cicular Edges.
Lime = Precision.
Pink = Specify Arc Angle Minimum.

Sketch

Introduction:

Start Sketch, the feature is used to start a **Sketch** in a **View**.

In order to use the **Start Sketch** feature, locate the **Annotate → Sketch** tab at the top of the screen:



Select **Start Sketch**, then select the desired **View**.

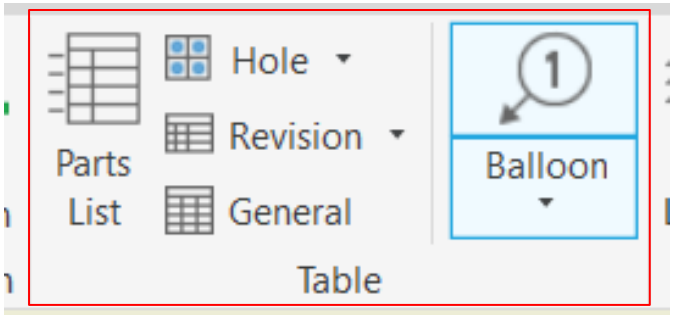
- **Shortcut key** = S

Table:

Introduction:

The **Table** functions are used to create tables.

In order to use the **Table** features, locate **Annotate → Table** tab at the top of the screen:

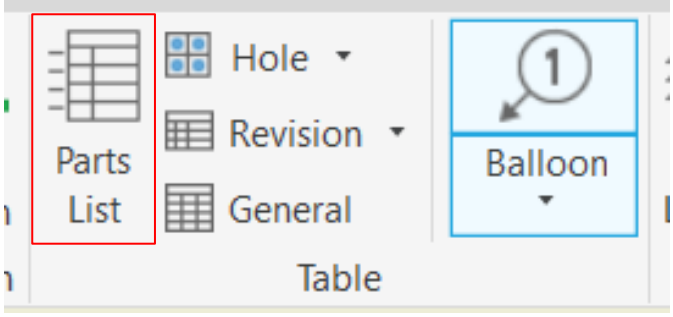


Parts List

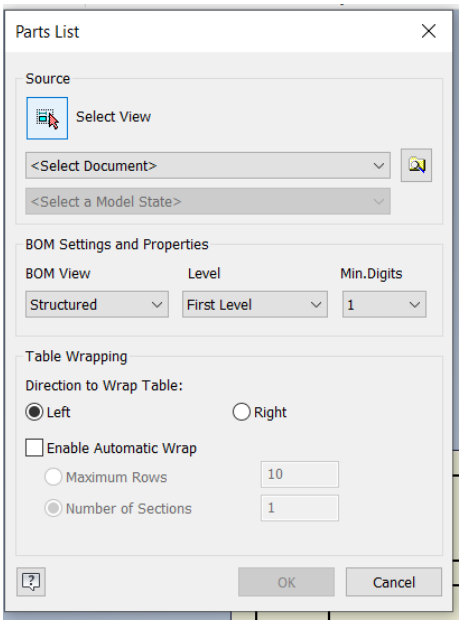
Introduction:

The **Part List** feature is used to create a list of **Parts**. This only works with an **Assembly**.

In order to use the **Part List**, locate **Table** tab at the top of the screen:



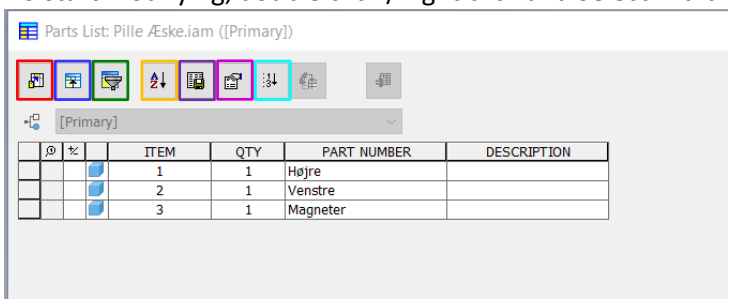
Select **Part List**. When you press **Part List**, a box appearss:



Select **Part List**. Select then the primary **View** you started with placing. Then press **OK**, after a box will appears again (a warning). Press **OK** at a **BOOM View** warning. Now place your **Part List**.

Edit:

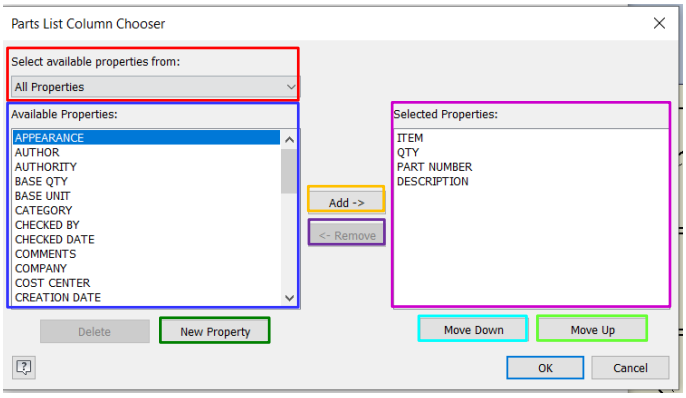
To start modifying, double click / right click and Select "**Edit Partlist**".



Colourr:

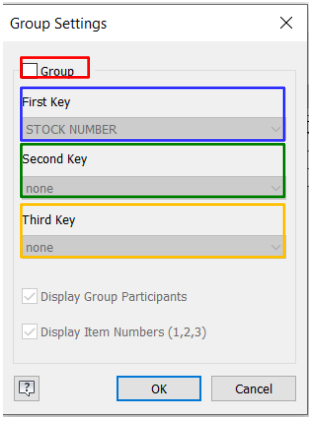
- Red
- Blue
- Green
- Yellow
- Purple
- Magenta
- Cyan

Red:



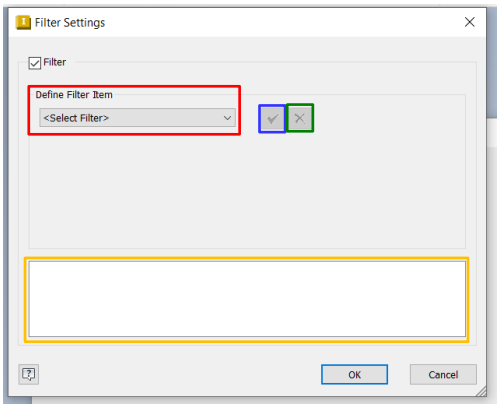
Red = Search/look for type. You don't need to.
Blue = The various **Properties**. Column type.
Green = Create new **Property**.
Yellow = Add.
Purple = Remove.
Magenta = List of the selected **Properties**.
Cyan = Move down.
Lime = Move up.

Blue:



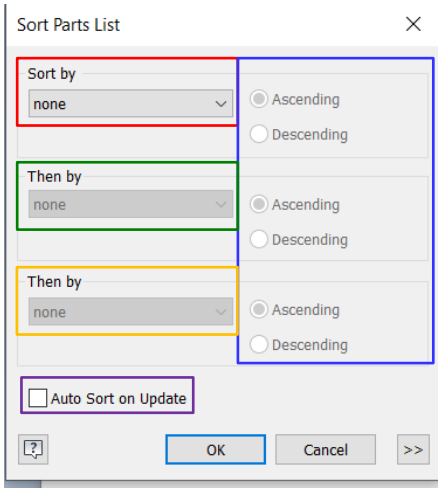
Red = Check grouping on and off.
Blue = First grouping/priority.
Green = Second grouping/priority.
Yellow = Third grouping/priority.

Green:



Red = Type of filter.
Blue = Yes to filter.
Green = No to filter.
Yellow = History, where you can remove or change previous filters.

Yellow:

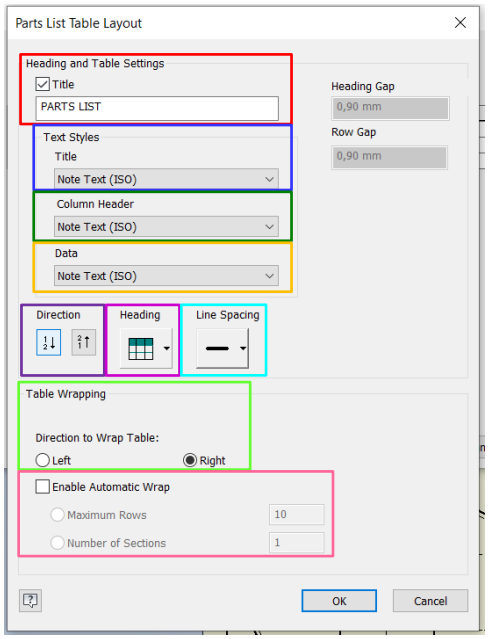


Red = Sort by type (first priority).
Blue = Choice between ascending or descending.
Green = Sort by type by first (second priority).
Yellow = Sort by type by second (third priority).
Purple = Auto update after updating **Part**.

Purple:

Exports / saves **Part List** to table.

Magenta:



Red = Title.
Blue = Title text style.
Green = Column text style.
Yellow = Data text style.
Purple = Direction of new **Parts**.
Magenta = Title, position & removal**.**
Cyan = Line Spacing: Single, Double, Triple.
Lime = Direction of wrapping.
Pink = Auto Wrap.

Cyan:

Renumber Items.

Tips

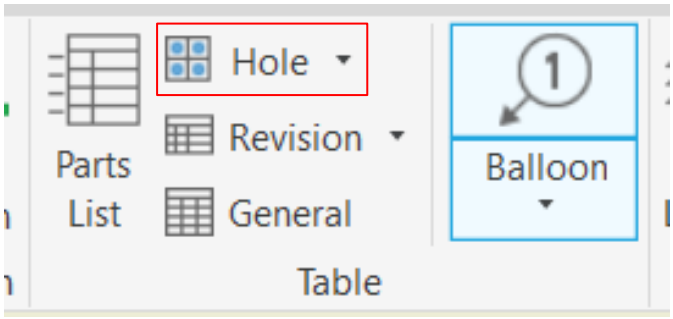
- Visibility:
  - You can change visibility: Select the desired row by right-clicking and select "**Visible**".
- **Custom Part**:
  - You can add and remove custom **Parts**: Right-click on any row and select "**Insert / Remove Custom Part**".
- To change the Name / number:
  - If you want to change a specific name / number, you can do it by: Left-clicking twice on the desired box.

Hole

Introduction:

The **Hole** feature is used to make a table of holes.

In order to use **Hole**, locate **Table** tab at the top of the screen:

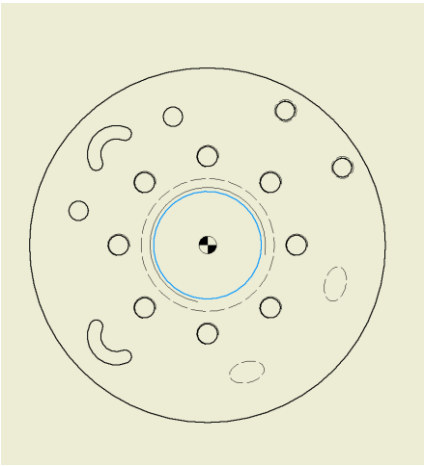


There are 3 methods on how to make a **Hole Table**.

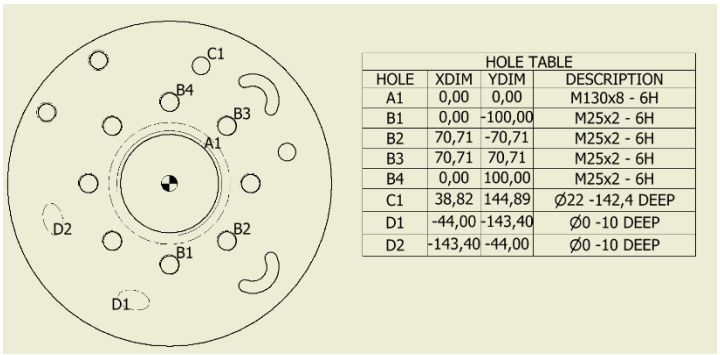
Hole Selection:

Here you select the desired holes you want to include.

To use select **Hole Selection**, then start by setting a **Reference Point** (recommended center point of subject):



Then select the desired holes, then right-click and select "**Create**".



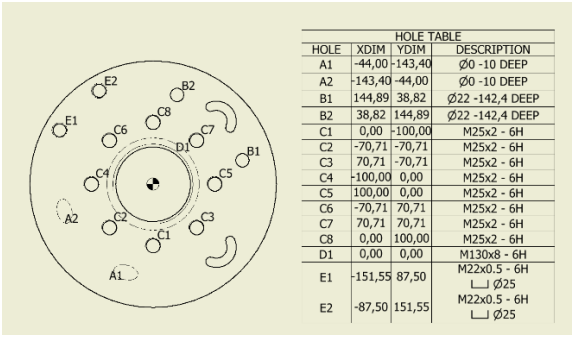
HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	0,00	0,00	M130x8 - 6H
B1	0,00	-100,00	M25x2 - 6H
B2	70,71	-70,71	M25x2 - 6H
B3	70,71	70,71	M25x2 - 6H
B4	0,00	100,00	M25x2 - 6H
C1	38,82	144,89	Ø22 -142,4 DEEP
D1	-44,00	-143,40	Ø0 -10 DEEP
D2	-143,40	-44,00	Ø0 -10 DEEP

Hole View:

Hole View can be found by clicking "**Hole**" and pressing the arrow pointing down.

To use select **Hole View**, start by setting a **Reference Point** (recommended center point of subject).

Then place the table, then right-click and select "**Create**".



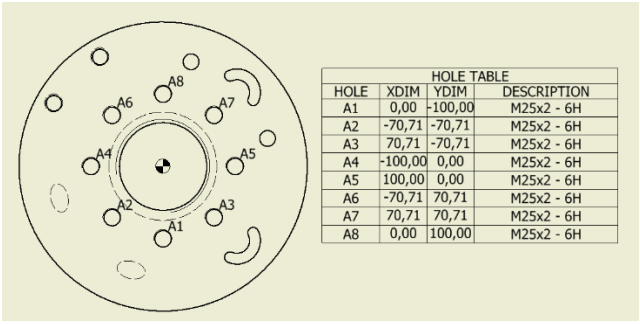
HOLE TABLE

HOLE	XDIM	YDIM	DESCRIPTION
A1	-44,00	-143,40	Ø0 -10 DEEP
A2	-143,40	-44,00	Ø0 -10 DEEP
B1	144,89	38,82	Ø22 -142,4 DEEP
B2	38,82	144,89	Ø22 -142,4 DEEP
C1	0,00	-100,00	M25x2 - 6H
C2	-70,71	-70,71	M25x2 - 6H
C3	70,71	-70,71	M25x2 - 6H
C4	-100,00	0,00	M25x2 - 6H
C5	100,00	0,00	M25x2 - 6H
C6	-70,71	70,71	M25x2 - 6H
C7	70,71	70,71	M25x2 - 6H
C8	0,00	100,00	M25x2 - 6H
D1	0,00	0,00	M130x8 - 6H
E1	-151,55	87,50	M22x0.5 - 6H ⌴ Ø25
E2	-87,50	151,55	M22x0.5 - 6H ⌴ Ø25

Hole Features:

Hole Features can be found by clicking "**Hole**" and pressing the arrow pointing down.

To use select **Hole Features**, then start by setting a **Reference Point** (recommended center point of subject). Then select one of the holes that are in a **Feature**, it could be 2 or more **Mirrored** holes, or many holes that are in a **Pattern**. Then right-click and select "**Create**".



HOLE TABLE

HOLE	XDIM	YDIM	DESCRIPTION
A1	0,00	-100,00	M25x2 - 6H
A2	-70,71	-70,71	M25x2 - 6H
A3	70,71	-70,71	M25x2 - 6H
A4	-100,00	0,00	M25x2 - 6H
A5	100,00	0,00	M25x2 - 6H
A6	-70,71	70,71	M25x2 - 6H
A7	70,71	70,71	M25x2 - 6H
A8	0,00	100,00	M25x2 - 6H

Tips:

To change your **Hole Table**, right click and here are some options, they only work if you right-click on a specific box:

- Precision = Changes the number of decimal places. (Right click on a number).
- Table = Use "**Export Table**" to export.
- Row = Adds and removes holes. To add, right click and select "**Create**". This does not work with **Hole Features**.
- Visibility:
  - **Origin** = Hides the **Reference Point**.
  - **Tag** = Hides a specific **Tag**.
  - **Hide all Tags** = Hides all **Tags**.
  - **Show all Tags** = Shows all **Tags**.

- Edit:
  - Tag = This will bring up a **Text** box. Where you can change the **Tag Text**. It's the name Inventor gives holes.

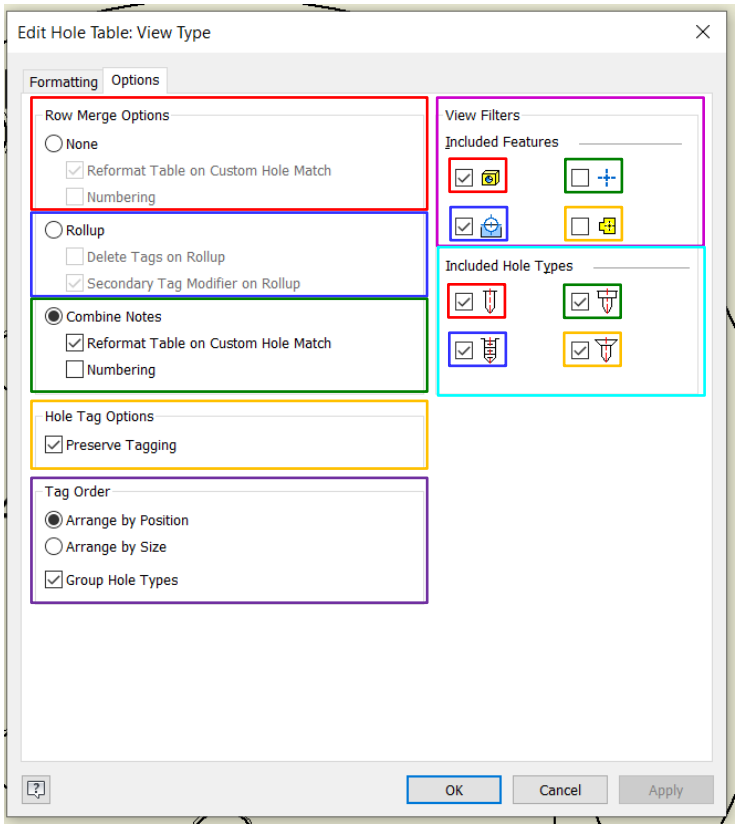
- Hole Note = This will bring up the **Hole Note**: works the same way as Hole and Thread. It's the text under the **DESCRIPTION** column that describes the hole.
- Description Text = This will bring up a **Text** box: You can write here before and after the **Hole Note info** (<<>>), This is not recommended, it is better to write it under the **Hole Note**.

To give one's **Tag** a **Leader,** click and drag the green dot next to the desired **Tag**.

Edit

In order to use the **Edit**, right-click and select "**Edit Hole Table...**" Afterwards, a box with 2 tabs appearss:

Options:



Red = **None**:

- Reshape table on custom hole match.
- Numbering. Meaning that instead of letters and numbers at a **Tag**, it's only numbers over all of them.

Blue = **Rollup**. Means that it compiles similar types of holes:

- *It is recommended to include a **QUANTITY** column.*
- Delete **Tags** on Rollup, similar holes in the same type only have one of them **Tagged**.
- Second **Tag** modifier on Rollup, similar holes of the same type will have letters and numbers like in **None**.

Green = **Combine Notes**. Means it compiles the **Hole Note DESCRPTION** if they are the same:

- Reformat table on custom hole match.
- Numbering. Meaning that instead of letters and numbers at **Tag**, it's only numbers over all of them.

Yellow = Keep **Tagging**.
Purple = **Tag** order:

- Arrange by position.
- Arrangere by size.
- Group type.

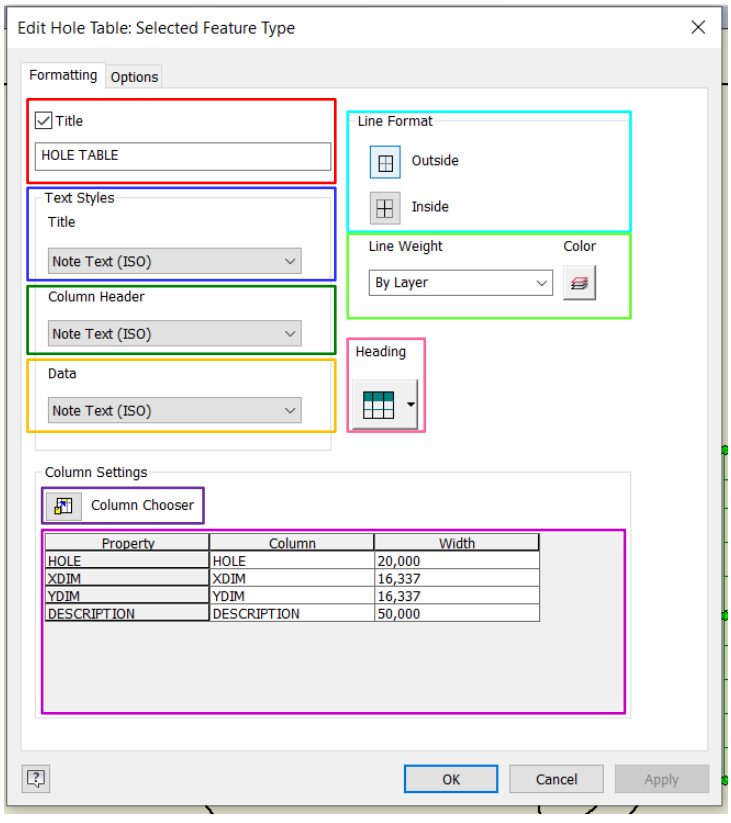
Magenta:

- Red = Off & on, **Hole Features**.
- Blue = Off & on, **Circular Cuts**.
- Green = Off & on, **Center Marks**.
- Yellow = Off & on, **Recoverd Punch Centers**.

Cyan:

- Red = Off & on, **Drill Holes**.
- Blue = Off & on, **Threaded Holes**.
- Green = Off & on, **Counterbore / Spotface Holes**.
- Yellow = Off & on, **Countersink Holes**.

Formatting:



Red = Title.
Blue = Text style: Title.
Green = Text style: Column header.
Yellow = Text style: Data.
Purple = **Column Chooser**. Functions as: **Part List → Edit → Red.**
Magenta = **Part List → Edit → Tips.**

Cyan = Line format.

- Outer edge.
- Inner edge.

Lime = Line weight. Here you change the arrow (↑).

- Line thickness.
- Line colour.

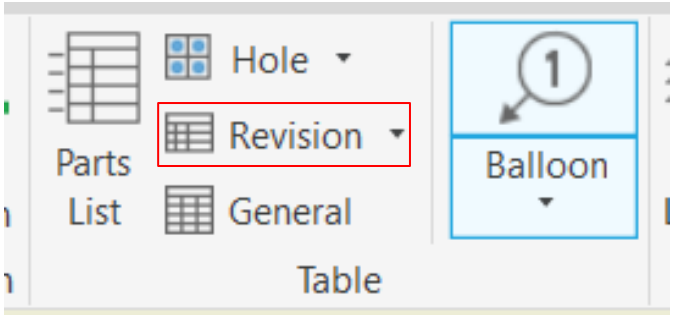
Pink = Title, position & removal.

Revision

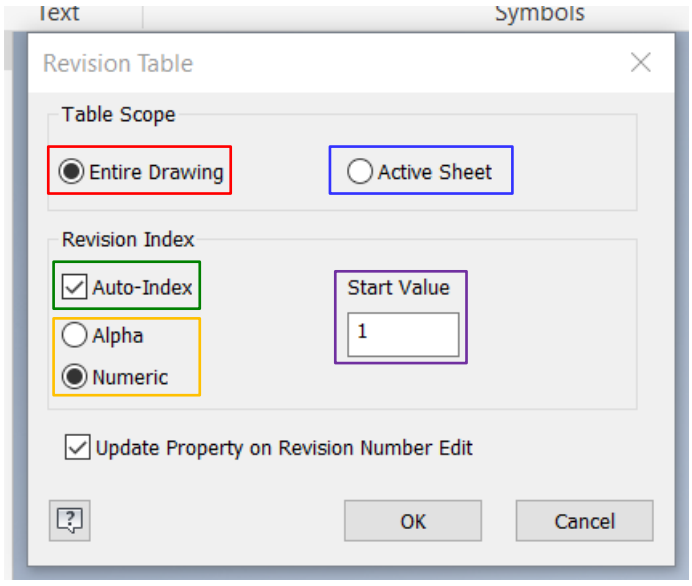
Introduction:

The **Revision** feature is used to create a history of changes you subsequently make to a subject.

In order to use the **Revision**, locate **Table** tab at the top of the screen:



Select **Revision**. When you press Revision, a box appearss:

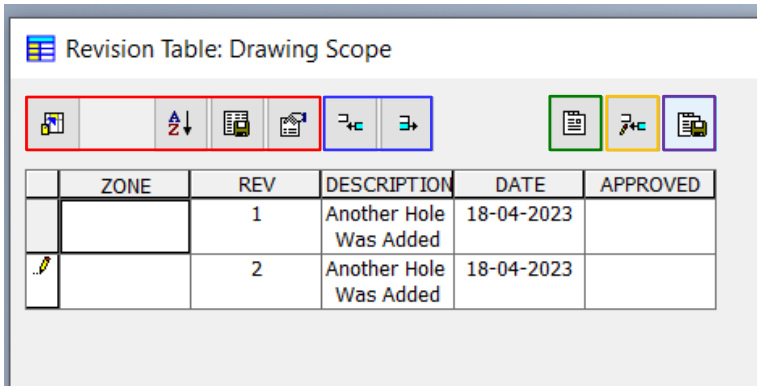


Red = Select the whole drawing.
Blue = Select current sheets.
Green = Select off & on: **Auto Indeks**.
Yellow = Select between **Alpha** and **Numeric**.
Purple = Start value.

To add a new **Revision**. Right-click and select "**Add Revision Row**".

Edit / Add:

After pressing **Edit** or adding a **Revision Row**, this box appearss:



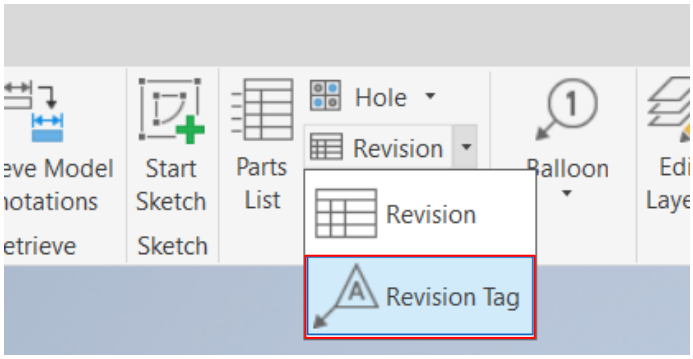
Red = Functions as: **Part List → Edit**.
Blue = Inserts or removes row.
Green = **iProperties**.
Yellow = Adds a **Revision Row**.
Purple = Updates **Property** to **Revision** number.

Revision Tag:

Introduction:

The **Revision Tag** feature is used to tell where your **Revision** is located.

In order to use the **Revision Tag**, locate **Table** tab at the top of the screen:



Select **Revision Tag**, and then click on the desired geometry.

To change the **Revision** number: Right-click and select "**Tag**" and then the desired number.

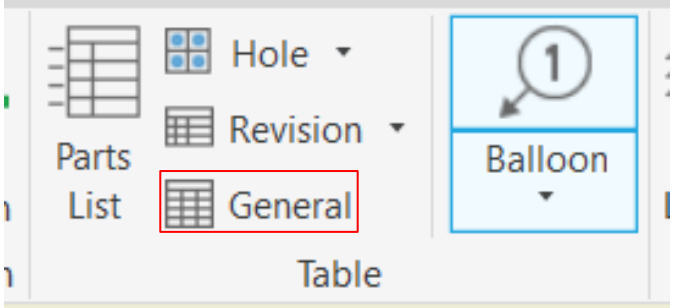
With right click you can also change the arrow head or add an extra Vertex / Leader. And delete the Tag / Leader.

General

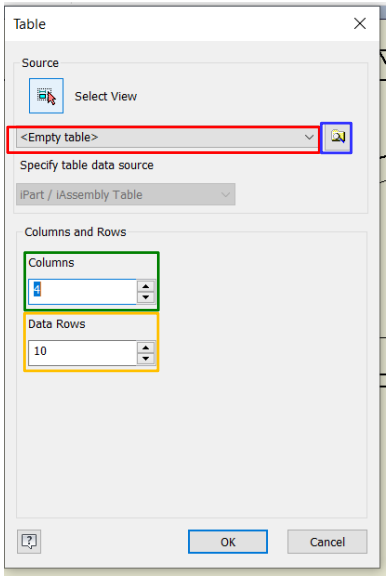
Introduction:

The **General** feature is used to create a general table, or to import **Excel**, **iPart** or **Sheet Metal**; tables.

In order to use the **General**, locate **Table** tab at the top of the screen:



Select **General**. When you press General, a box appearss:



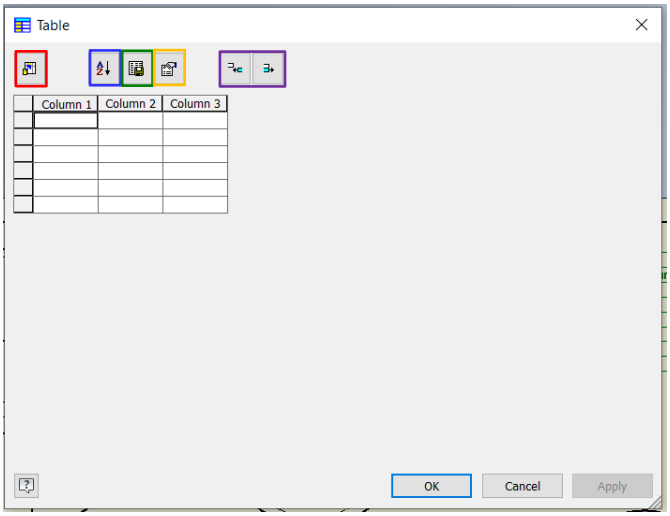
Red = Here choose your, **iPart** or **Sheet Metal**. If not ignore.
Blue = Here select an **Excel** file or similar via **Explorer**.

Again, you don't need to use a Source, you can make a bare table with **General**.

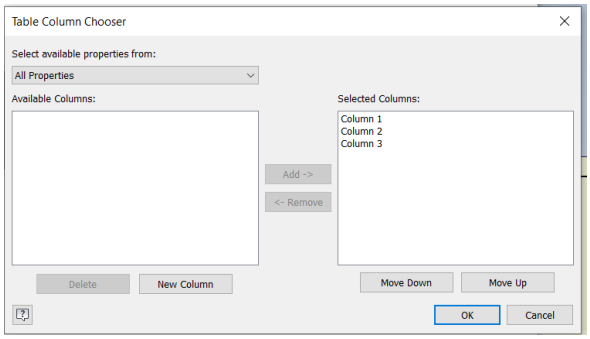
Green = The amount: Columns.
Yellow = The amount: Data Rows.

Edit:

To start modifying, double click / right click and select **Edit Partlist**.



Red: it's like **Part List Edit Column Chooser**, just without **Properties**:



Can only be used to create new columns. Or to temporarily remove a column. And move up and down columns.

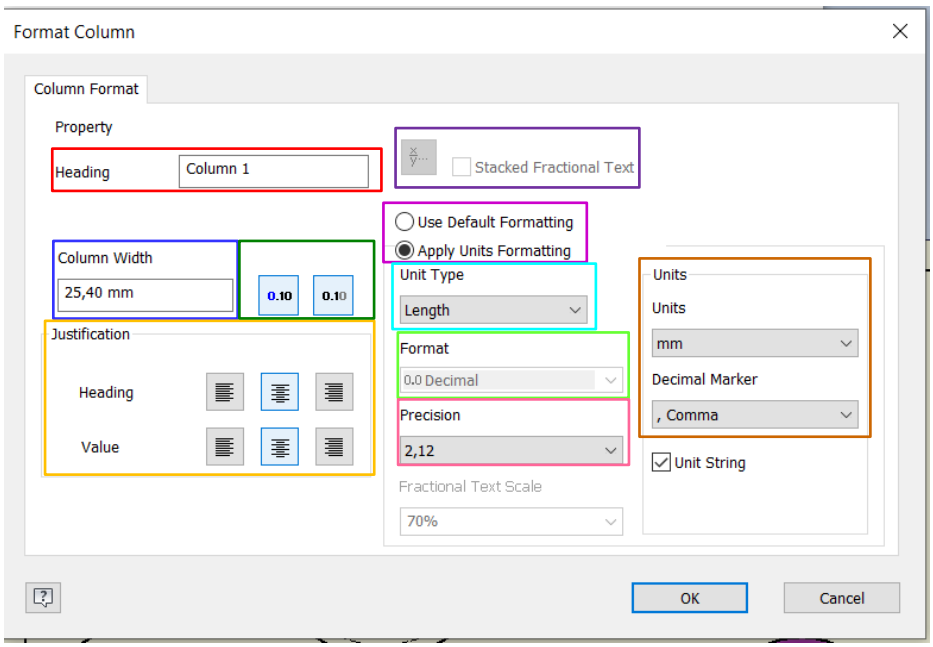
Blue: Sorting tool, works in the same way as: **Part List → Edit →** Yellow.
Green: Export /saves, the table.
Yellow: Layout tool, work in the same way as: **Part List → Edit →** Magenta.
Purple: Inserts or removes row.

Tips

- Visibility:
  - You can change visibility: Select the desired row by right-clicking and select "**Visible**".
- To change the Name / number:
  - If you want to change a specific name / number, you can do it by: Left-clicking twice on the desired box.
- Change columns:
  - Right-click on the desired column and select "**Format Column**". Afterwards, a box appearss → see more below **Format Column**.
- Change the width of columns:
  - Right-click and select "**Column Width**".

Format Column:

Changes one's column you have to format, this can also be done in the **Part List** but is not recommended.



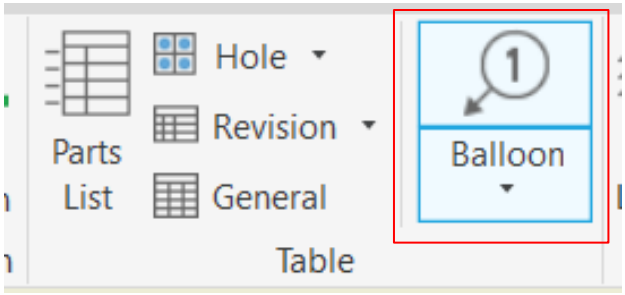
Red = Title.
Blue = Column width.
Green = **Leading / Trailing Zeros**: removes zero before the decimal point or after the decimal point.
Yellow = Text position.
Purple = Stacked fractional text. To stack to a fraction, first write X/Y in one of the data boxes. After that you can select the style.
Magenta = Select default formatting or unit specific formatting.
Cyan = **Unit** type.
Lime = Format. For example that you can only write in fractions.
Pink = How many decimal places are used.
Brown = Which **Unit** is used. And whether a comma or period is used as a decimal marker.

Balloon

Introduction:

The **Balloon** feature is used to assign a specific number to a **Part**. So, you can understand the **Part List**.

In order to use the **Balloon**, locate **Table** tab at the top of the screen:



Select **Balloon**, then the **Part** you want to give its number. Now place the **Balloon**.

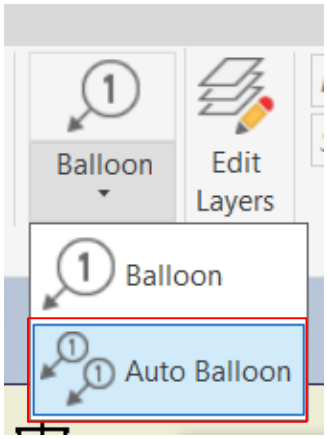
- **Shortcut key** = B

Auto Balloon:

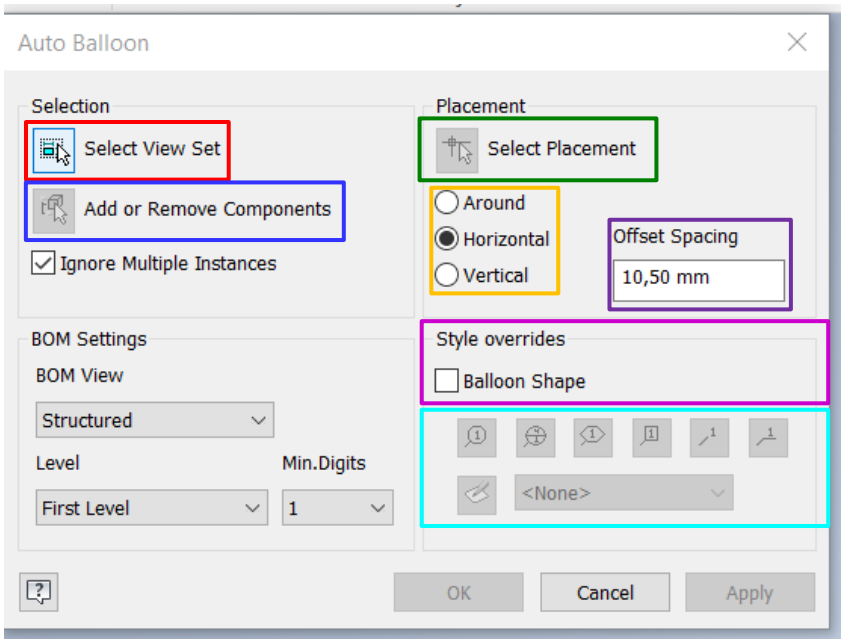
Introduction:

The **Auto Balloon** feature is used to automatically assign a specific number to **Parts**. So, you can read the **Part List**.

In order to use the **Auto Balloon**, locate **Table** tab at the top of the screen:



Select **Auto Balloon**. When you press **Auto Balloon**, a box appearss:



Red = Select a **View Set**.
Blue = Add or remove components, here select the disired **Parts**.
Green = Select placement. You can move them afterwards.
Yellow = **Around. Horizontal**. **Vertical**.
Purple = How big the distance is between **Balloons**.
Magenta = Select whether you want to change the appearsance of the **Balloons Shape**.
Cyan = Here change the appearsance of the **Balloons Shape**.

FINISH with **OK.**

Work Features – Features

Introduktion

Efter man har startet et projekt, og man har opret en **Part fil**, **Sketch** og **Solid**: først her kan **Work Features** bruges som reference elementer, så den ønskede form opnås. Nogle **Modify** og **Create** funktioner bruger **Work Features** for at **virke**.

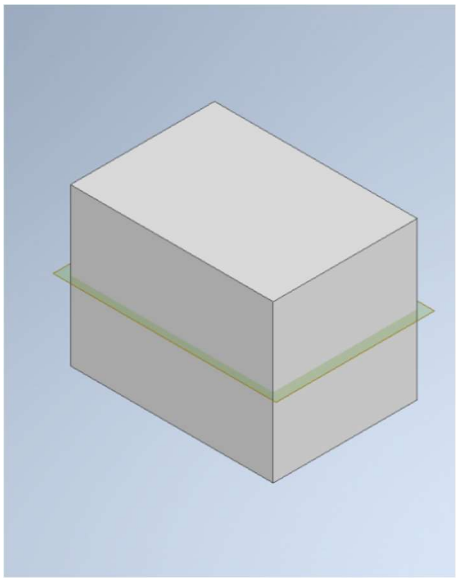
Indholdsfortegnelse:

Work Planes

Introduktion:

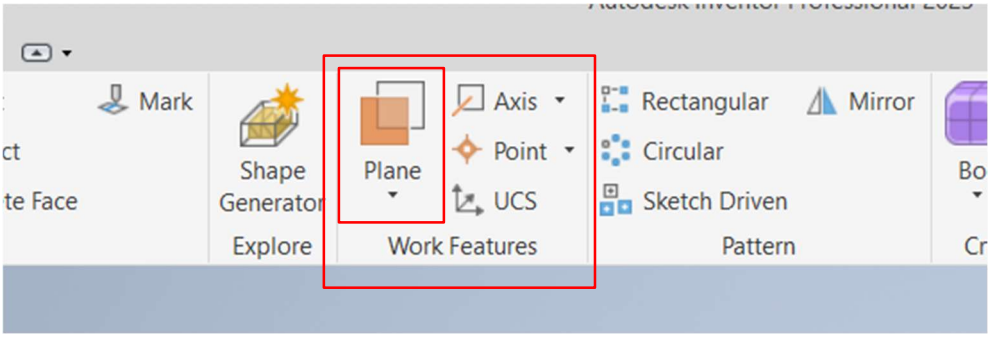
Work Planes bruges som en reference når man 3D-modellerer. Det kan være en stor hjælp når man skal lave en **Sketch** på en vinkel.

Work Planes laver et nyt **Plane** der ikke er en del af YZ, XZ og XY-planene.



Eksempel på et **Work Plane**, gennem en 3D-model.

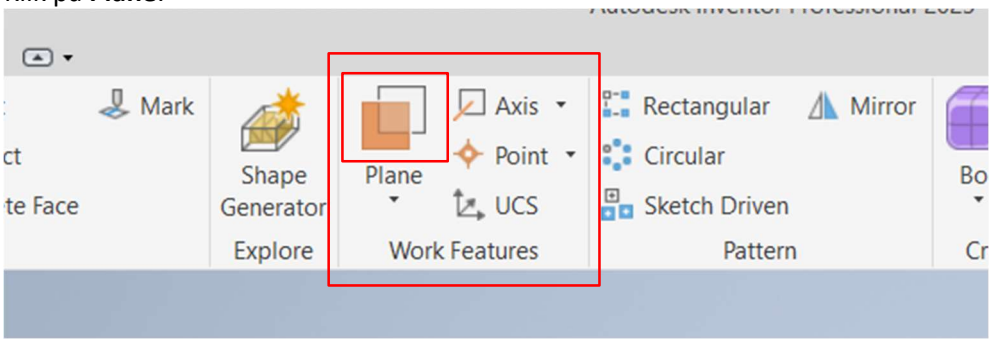
For at bruge **Work Planes**, find **Work Features**, fanen oppe i toppen af skærmen:



Her er der en lang liste af forskellige **Work Planes**:

Plane

Plane er den mest brugte af alle **Work Planes** mulighederne, da den er fleksibel og kan bruges til meget.
Klik på **Plane**:



For at offsette med Plane og ikke "Offset From Plane":

Vælg **Plane**, klik så på den **Face** (overflade) man vil offsette fra, herefter vælg den (blå) kant af **Work** planenet og hiv hen imod den ønskede vej. Og derefter angiv længden.

Work Planes man også kan lave med Plane:

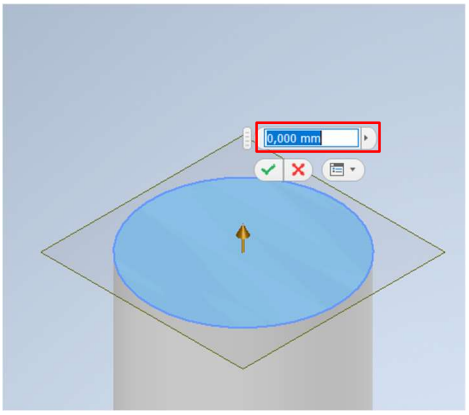
Bemærk: man kan nøjes med at vælge **Plane**, For at bruge, disse plan funktioner ↓:

- **Midplane between Two Plane.** (Vælg 2 **Faces/Planes** der ligger imodsætning til hinanden).
- **Midplane of Torus.** (Vælg torussen).
- **Parallel to Plane Through Point**. (Sæt **Point** og vælg det. Herefter vælg 1 **Face/Plane**).
- **Angle to Plane around Edge**. (Vælg 1 kant og 1 **Face** der ligger vinkelret med kant).
- **Three Points**. (Vælg 3 punkter).
- **Two Coplanar Edges**. (Vælg 2 diagonale kanter).
- **Tangent to Surface through Edge**. (Vælg cylinder/afrunding og herfter kant).
- **Tangent to Surface and Parallel to Plane**. (Vælg cylinder/afrunding og herefter et **Face/Plane**).

De andre Work Planes

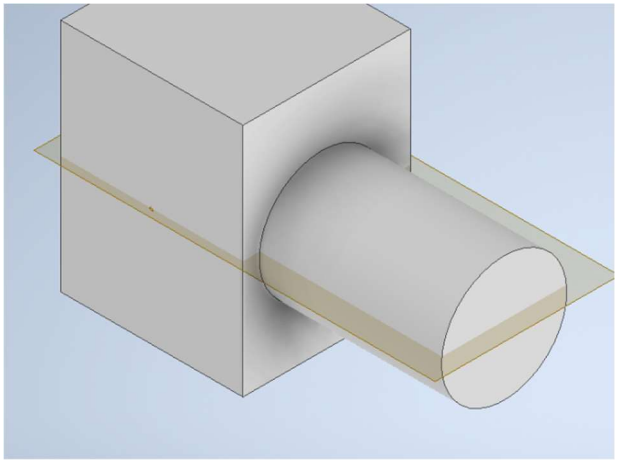
Offset from Plane

Laver et **Plane** offsat fra en overflade. Klik på det **Plane** man vil offsette fra. Derefter angiv i det røde felt; længden. →



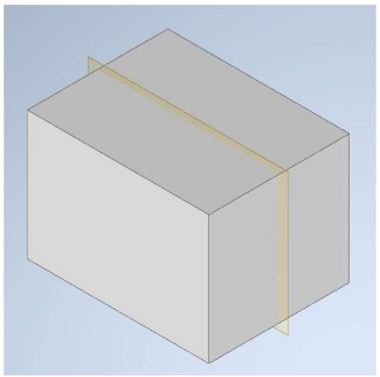
Parallel to Plane Through Point

Laver et parallelt **Plane** igennem et punkt. For at anvende, sæt et punkt med enten **Sketch Point** eller **Work Feature Point**. Tryk så på en overflade. ↓



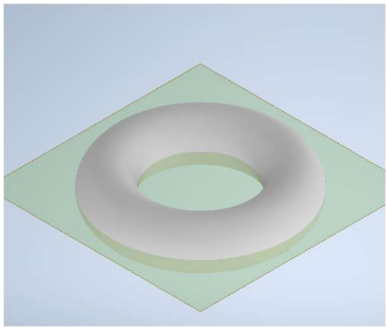
Midplane between Two Plane

Laver et **Plane** i midten af 2 overflader. For at anvende, vælg 2 overflader. Det kan f.eks. være 2 parallelle overflader eller 2 symmetriske. ↓



Midplane of Torus 

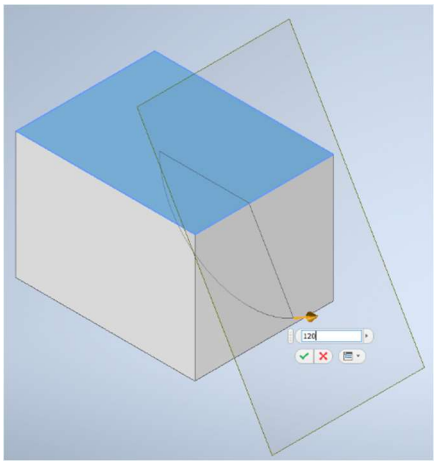
Laver et **Plane** i midten af en **Torus** figur. En **Torus** figur minder om et bildæk eller en donut. For at anvende, vælg torussen. ↓



Angle too Plane around Edge 

Laver et **Plane** i en vinkel ud fra en kant og overflader. For at anvende, vælg en kanten planet støder imod og en overflade hvor fra vinklen bliver stødt imod. Man kan også vælge 2 overflader.

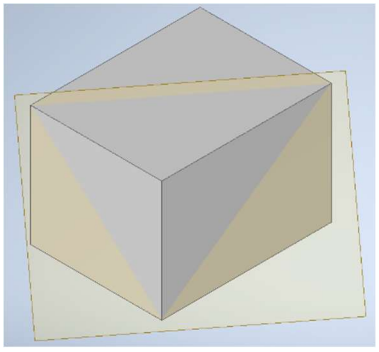
↓



Three Points 

Laver et **Plane** ud fra 3 punkter. For at anvende, vælg 3 punkter.

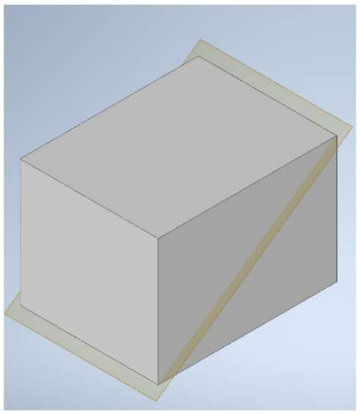
↓



Two Coplanar Edges 

Laver et **Plane** diagonalt i mellem 2 kanter. For at anvende, vælg 2 diagonale kanter.

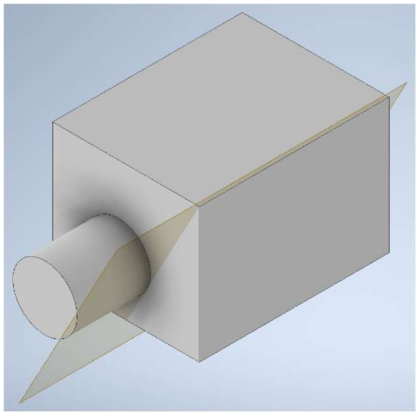
↓



Tangent to Surface through Edge 

Laver en **Plane** tangent med en kant. For at anvende, vælg en afrunding/cylinder, vælg herefter en kant.

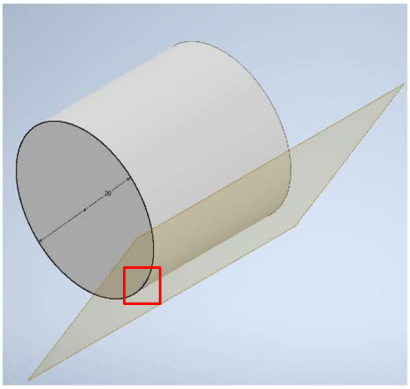
↓



Tangent to Surface through Point 

Laver en **Plane** tangent med et punkt. For at anvende, først en afrunding/cylinder, herefter vælg et punkt.

↓

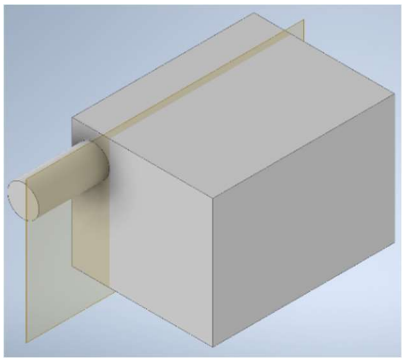


Rød = Punkt.

Tangent to Surface and Parallel to Plane 

Laver et **Plane** parallelt og tangent med en afrunding/cylinder. For at anvende, vælg først en afrunding/cylinder, herefter vælg et punkt.

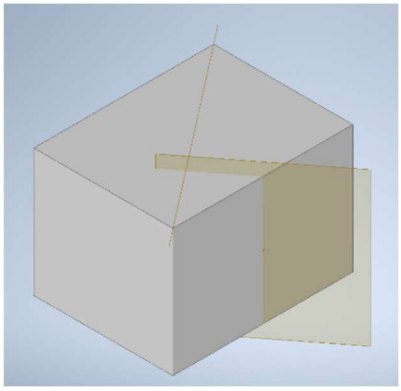
↓



Normal to Axis through Point 

Laver et **Plane** i forhold til **1 akse** og **1 punkt**. For at bruge, sæt en akse eller brug X/Y/Z akserne. sæt et punkt, herefter brug **Plane** funktionen. Vælg så punktet og aksen.

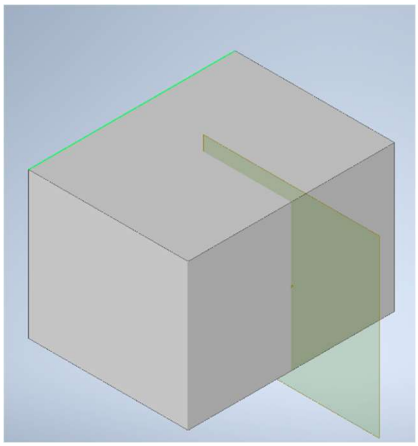
↓



Normal to Curve at Point 

Laver et **Plane** der drejer omkring et punkt. For at anvende, vælg et punkt, herefter en kant / overflade.

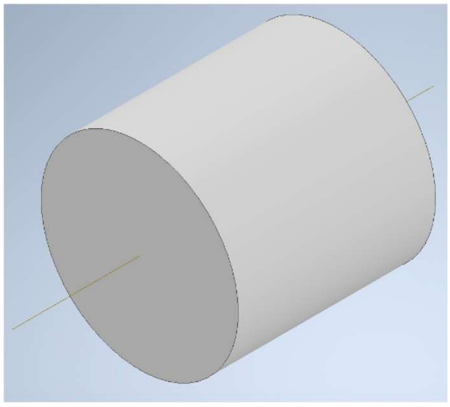
↓



Axes

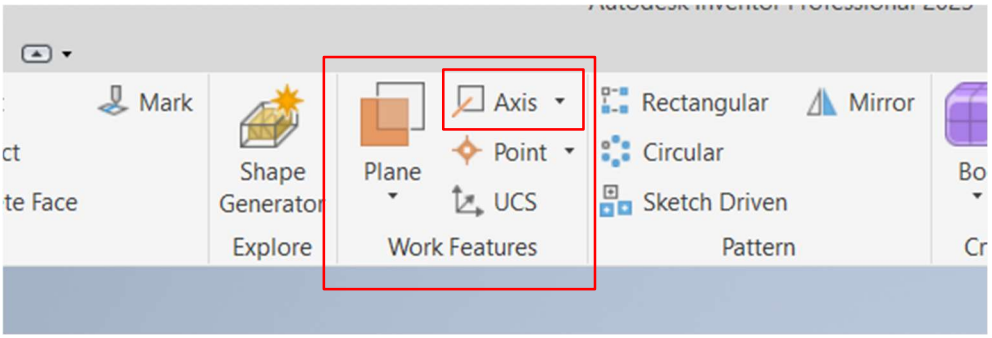
Introduktion:

Axes er de **Work Features**, som laver nye akser der ikke er en del af X, Y og Z akserne. Man ville bruge **Axes**, hvis man skal f.eks. bruge en bestemt akse til en **Revolve**:



Her er et eksempel på hvordan en **Axis** kan se ud, når den er færdig.

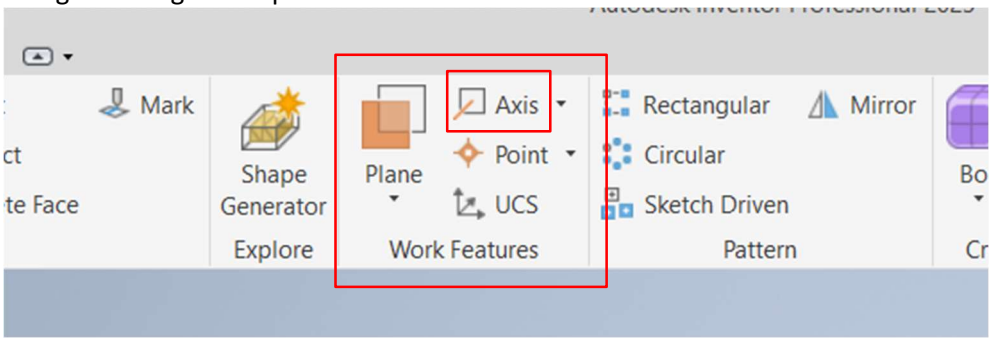
For at bruge de forskellige **Axes**, find **Work Features**, fanen oppe i toppen af skærmen:



Her er der en lang liste af forskellige **Axes**:

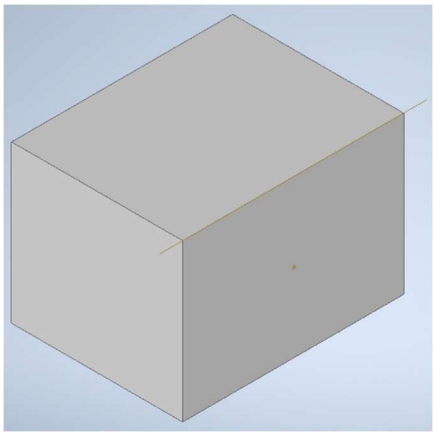
Axis

Med **Axis** kan man lave kanter om til akser og sætte akser igennem punkter. Da den er fleksibel og kan bruges til meget. Klik på **Axis**:



For at sætte akser igennem et punkt; vælg punktet og de overflader/**Planes** retningen af aksen skal skære igennem.

Bemærk: Man behøver ikke at trykke på pilen og vælge en specifik **Axis** man kan nøjes med at vælge **Axis**. Hvis man ikke kan få **Axis** til at gøre som man vil, så brug de specifikke.



De Andre Axes

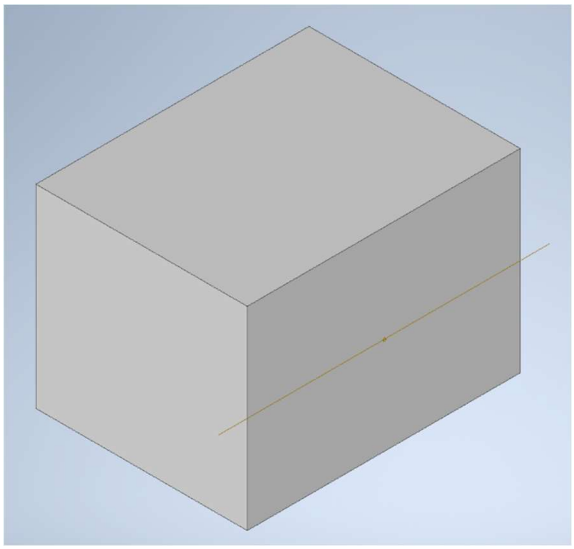
On Line or Edge 

Laver et akse på en kant. Klik på den kant man lave til en akse. →

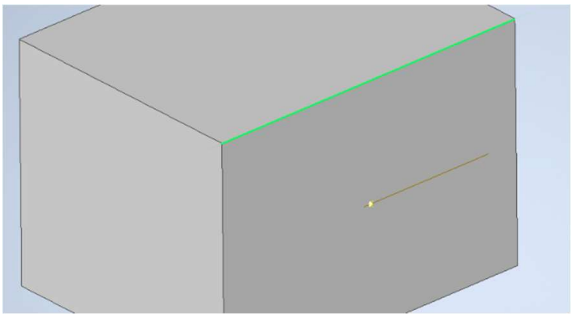
Parallel to Line through Point 

Laver en akse igennem et punkt parallelt med en kant. For at anvende, sæt et punkt med enten **Sketch Point** eller **Work Feature Point**. Brug derefter funktionen og vælg punktet, herefter vælg en kant.

Bemærk forhåndsvisningen viser kun en halv streg (Akse). når man færdiggøre bliver den hel.
↓

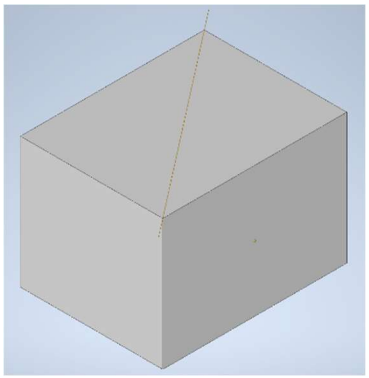


Halv streg eksempel.



Through Two Points 

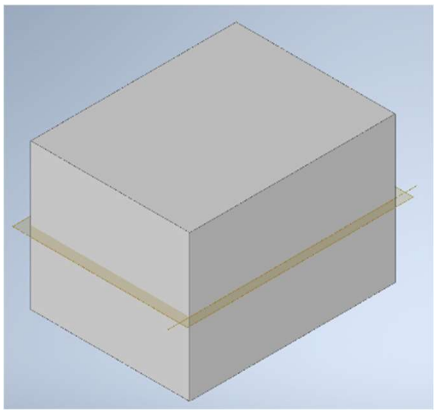
Laver en akse over 2 punkter. For at anvende, vælg 2 punkter.
↓



Intersection of Two Planes

Laver en akse i krydspunktet af 2 planer eller overflader. For at anvende, vælg 2 **Planes**/overflader.

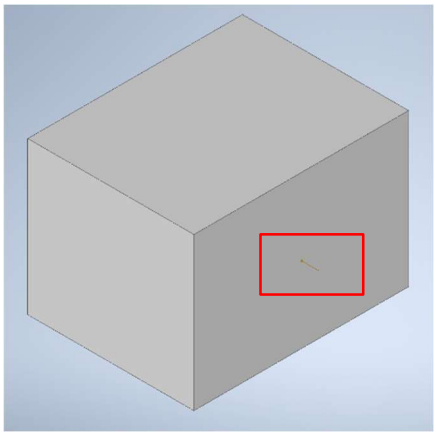
↓



Normal to Plane through Point

Laver en akse til et **Planes**/overflader igennem et punkt. For at anvende, vælg et punkt, herefter vælg et **Planes**/overflader.

↓

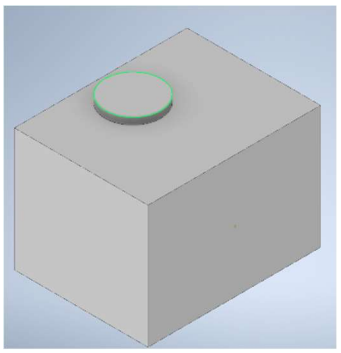


Går igennem emnet.

Through Center of Circular or Elliptical Edge

Laver en akse i centeret af en cirkulær / elliptisk kant. For at anvende, vælg en cirkulær / elliptisk kant.

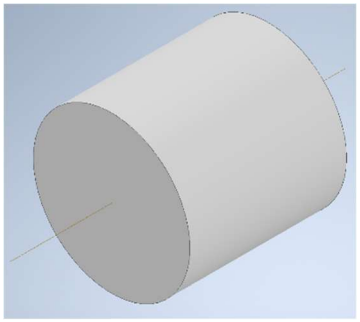
↓



Through Revolved Face or Feature (Cylinder)

Laver en akse i centeret af en cylinder/ **Revolved** overflade. For at anvende, vælg en cylinder/ **Revolved** overflade.

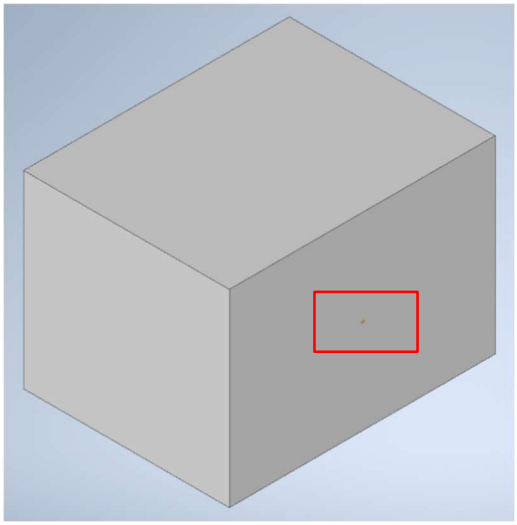
↓



Points

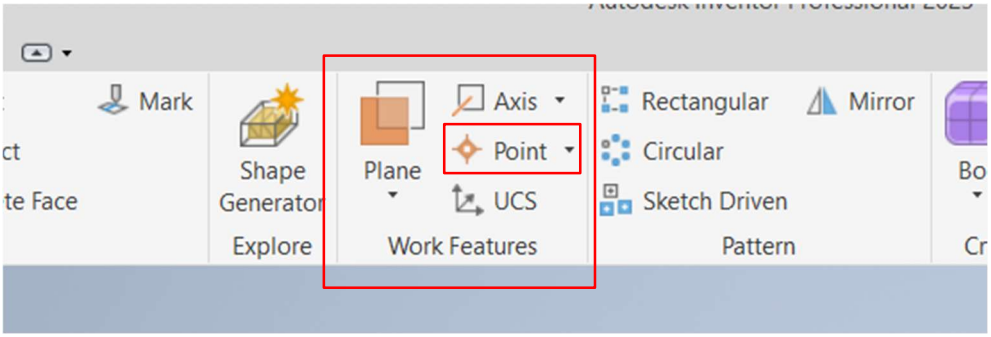
Introduktion:

Points er de **Work Features**, som laver nye punkter. **Points** er en anderledes funktion end **Sketch Point**. Men disse punkter virker også til **Hole** funktionen.



Her er et eksempel på hvordan et **Point** kan se ud, når den er færdig. (Kan være svær at se men kan altid findes ude i historikken).

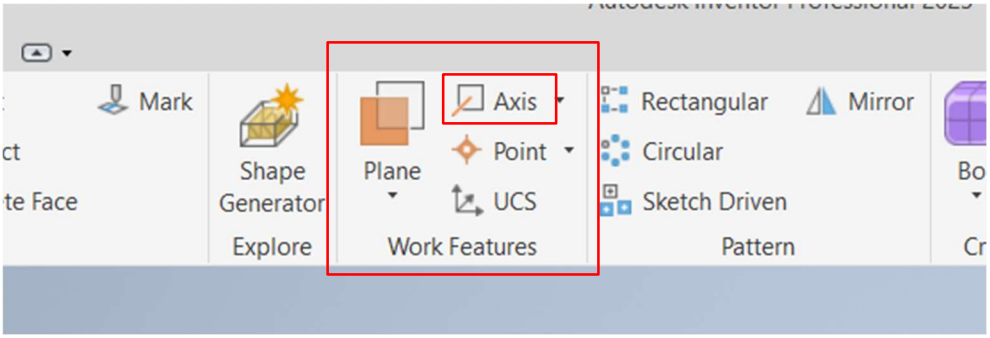
For at bruge **Points**, find **Work Features**, fanen oppe i toppen af skærmen:



Her er der en lang liste af forskellige **Points**:

Point

Point er den mest brugte af alle **Points** funktioner, da den er fleksibel og kan bruges til meget:

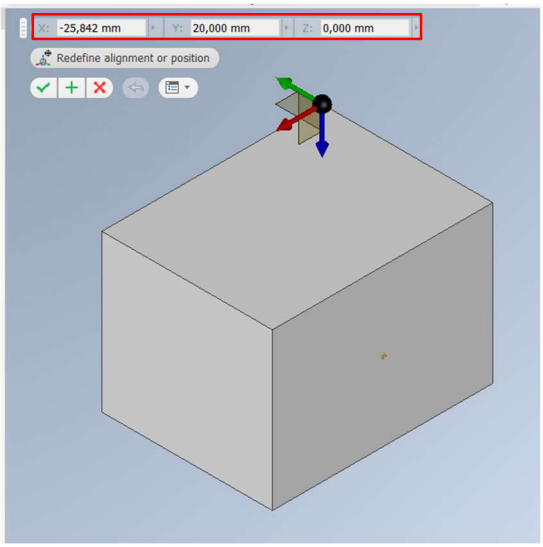


Med **Point** kan man vælge **Faces** (overflader), **Edges** (kanter) og hjørner / **Points** (punkter).

Points man også, kan lave med Point:

Bemærk: man kan nøjes med at vælge **Point**, For at anvende, disse plan funktioner ↓:

- **On Vertex, Sketch point, or Midpoint.** (Vælg et hjørne / midtpunktet på en linje / **Sketch** Punkt).
- **Intersection of Three Planes.** (Vælg 3 **Planes**/overflader der ligger op ad hinanden).
- **Intersection of Two Lines.** (Vælg 2 linjer).
- **Intersection of Plane/Surface and Line.** (Vælg et **Plane**/overflade og herefter en linje).
- **Center Point of Torus.** (Vælg en **Torus**).
- **Center Point of Sphere.** (Vælg en **Sphere**).



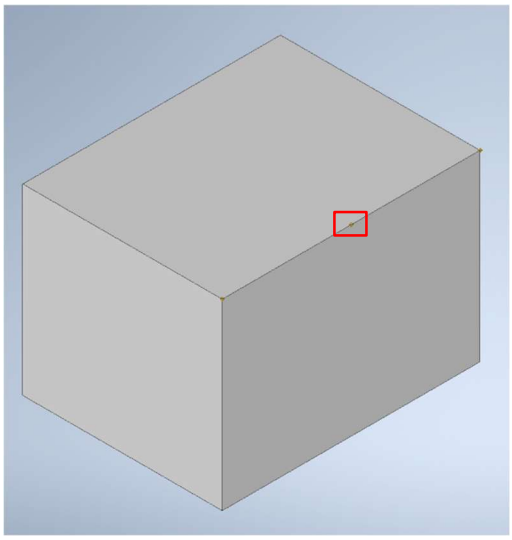
De Andre Points

Grounded Point

Laver et Punkt der kan flyttes på med koordinater, og der efter bliver låst. Man kan angiv koordinater i det røde felt. Man kan også hive i de farvede pile →

On Vertex, Sketch point, or Midpoint 

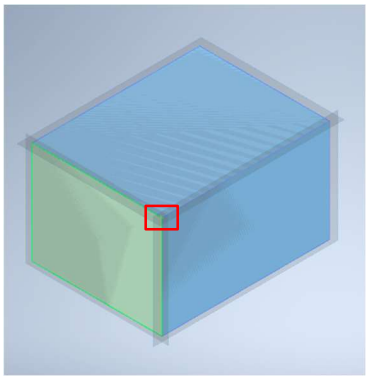
Laver et punkt ved hjørner, **Sketch** punkter og midt på linjer/kanter. For at anvende, vælg punktet; enten hjørner, **Sketch** punkter, midt på kanter/linjer. ↓



Intersection of Three Planes 

Laver et punkt i krydset af 3 planer. For at anvende, vælg 3 overflader.

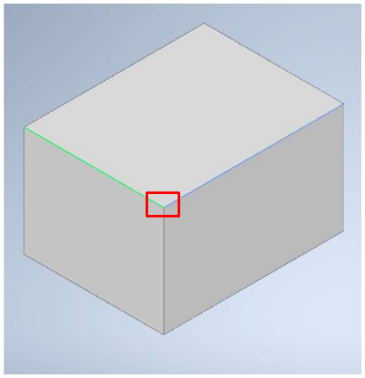
↓



Intersection of Two Lines 

Laver et punkt i krydset af 2 kanter/linjer. For at anvende, vælg 2 kanter/linjer.

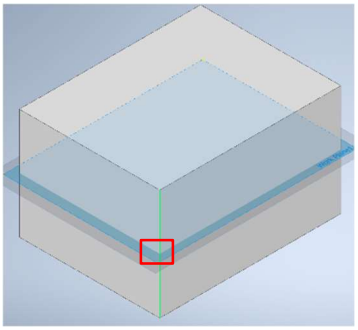
↓



Intersection of Plane/Surface and Line 

Laver et punkt i krydset imellem et **Plane**/overflade og en kant/linje. For at anvende, vælg 1 **Plane**/overflade og 1 kant/linje.

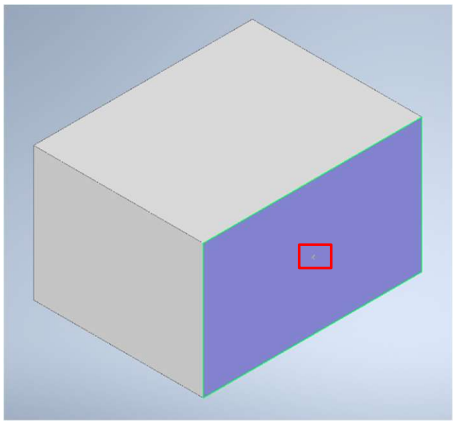
↓



Center Point of Loop of Edge 

Laver et punkt i midten af et **Loop** eller løkke af kanter. For at anvende, vælg et **Loop**. For at få fat i et **Loop** prøv at gå tæt på kanten og vælg så den ønskede oplyste overflade.

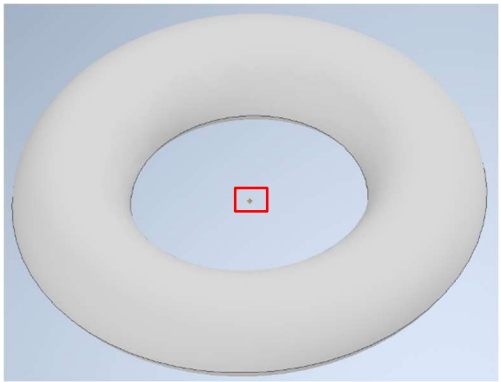
↓



Center Point of Torus 

Laver et punkt i midten af en **Torus** figur. En **Torus** figur minder om et bildæk eller en donut. For at anvende, vælg torussen.

↓



Center point of Sphere

Laver et punkt i midten af en **Sphere** / kugle form. En kugle form kan være en boldt. For at anvende, vælg kuglen. Her kan man ikke se punktet da, det er i midten af en hel figur. For at finde kig i historikken.

↓