Introduction to Ansys SpaceClaim

Module 02: Creating Geometry

Release 2022 R1

Please note:
- These training materials were developed and tested in Ansys Release 2022 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.

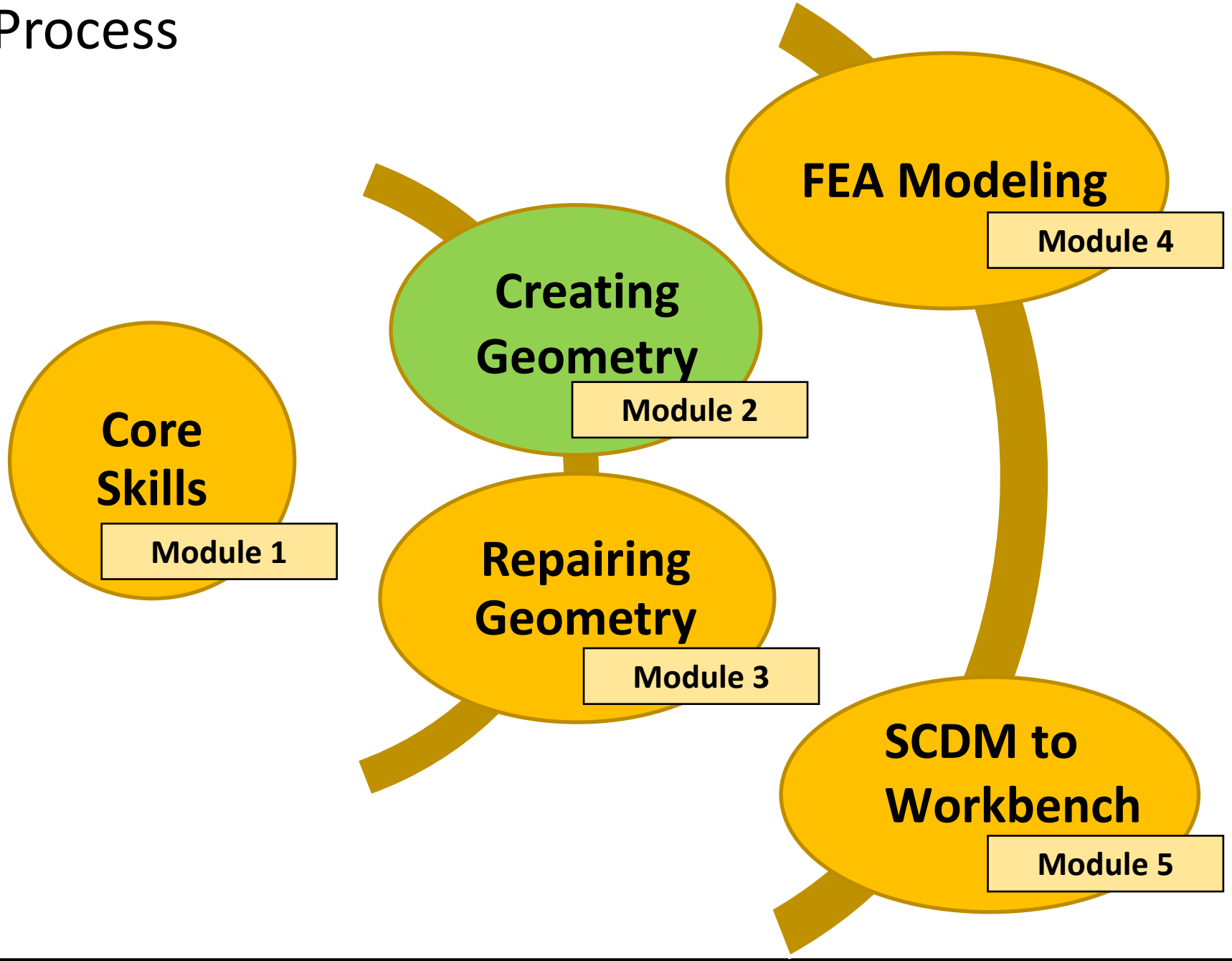
Learning Outcomes

- Discover the Sketch tab with its different functions
- Learn how to draw a sketch in SpaceClaim
- Discover the 3 modes in Ansys SpaceClaim
- Learn how to use the 3D modeling tools to extend the sketch to a 3D geometry
- Learn how to draw a drawing sheet in Ansys SpaceClaim

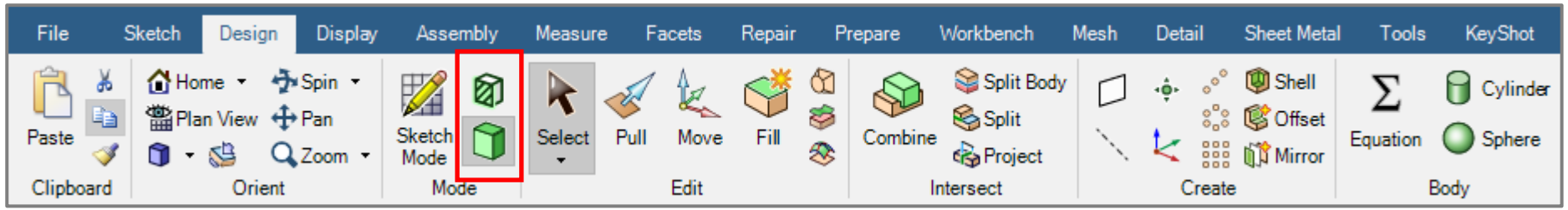
Overview

- In this module we will learn about:
  - Modes in SCDM
  - Sketching Tools
  - 3D Modeling Tools
  - Detailing

Overall Process



Modes in SpaceClaim Direct Modeler



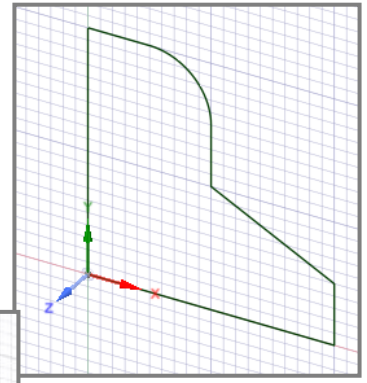
- 3 modes available for designing:
  - Sketch mode 

    Enables sketch grid for creating 2D sketch
  - Section mode 

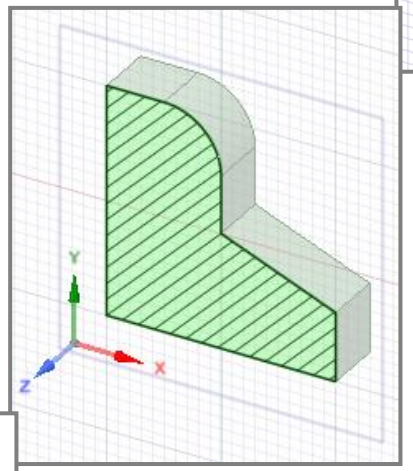
    Enables editing of solid and surface bodies by working with their edges and vertices in “cross-section” view
  - 3D mode 

    Enables 3D modeling mode for creating/editing geometries

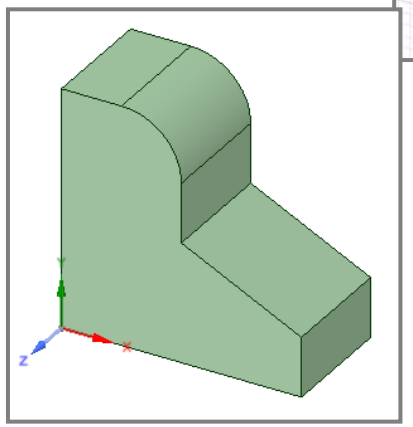
Sketch Mode



Section Mode



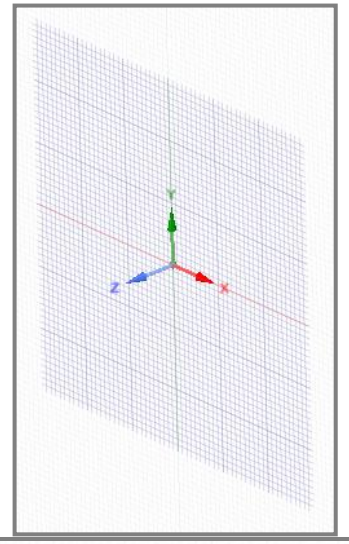
3D Mode



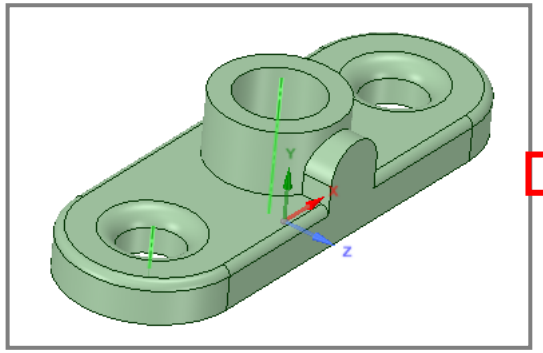
Creating Sketches (1)

Sketch Grid

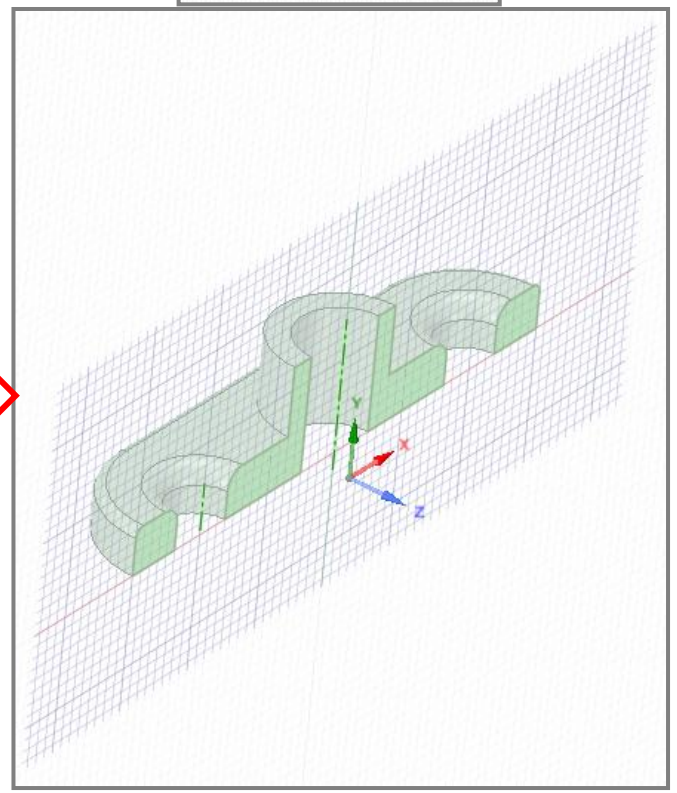
- Sketches are created on a “Sketch Grid”
- Sketch Grid can be created on
  - Any planar surface
  - Any existing Plane
    - (details about plane creation discussed later)
  - Combination of any geometrical entities which define a planar surface
    - 2 coplanar lines
    - Line and 1 point
    - Coordinate axes etc



Sketch Grid



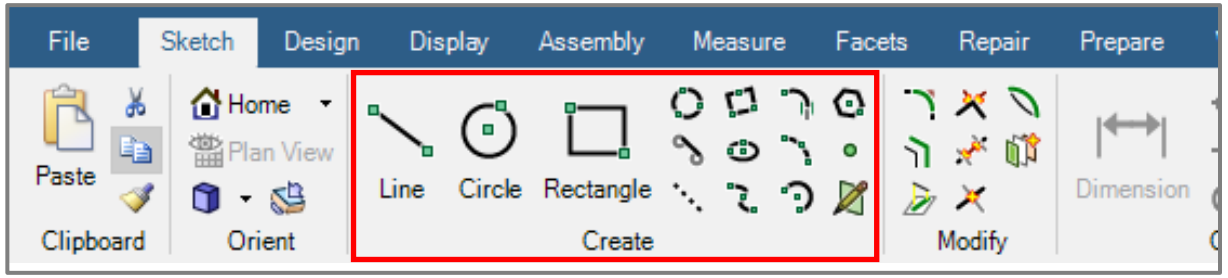
2 coplanar axes selected

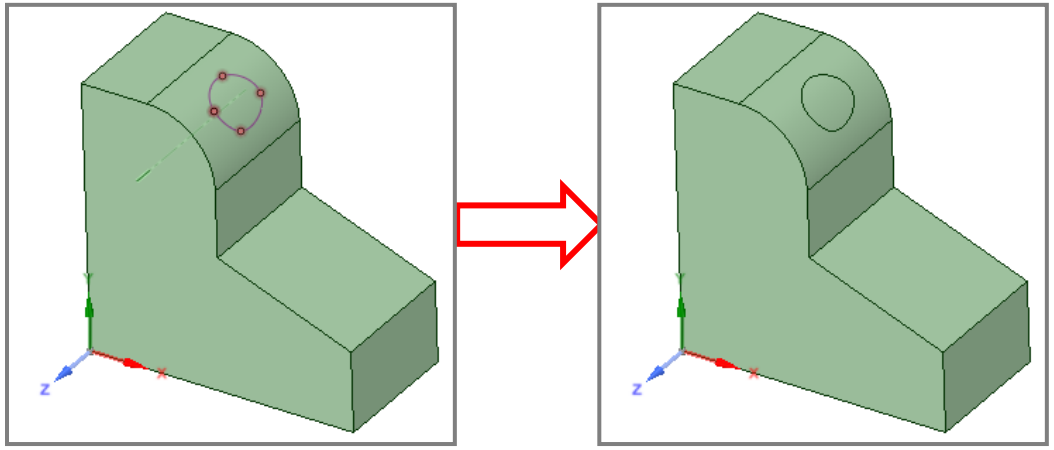


Creating Sketches (2)

Sketch Creation tools

- Hosts standard tools for creating sketches
  - Line
    - Single, polyline, tangent line, construction line
  - Rectangle
    - 2 point, 3 point
  - Circle
    - Center, 3 point
  - Ellipse
  - Polygon
  - Arc
    - Tangent, 3 point, sweep arc
  - Spline
  - Point
  - Face Curve
    - Sketch curve directly on face of body
    - Face need not be planar

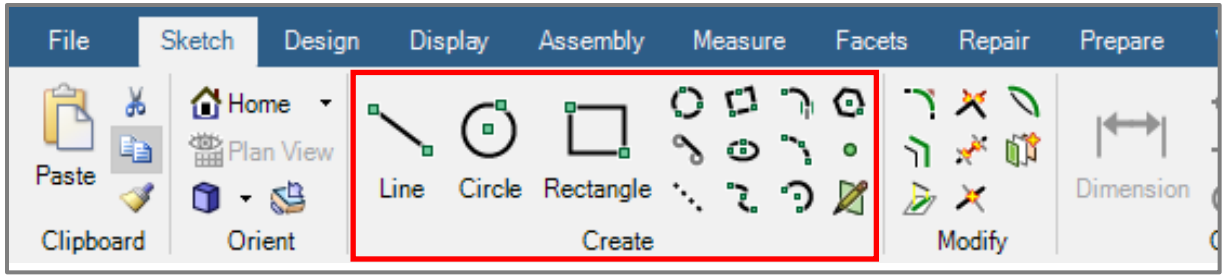


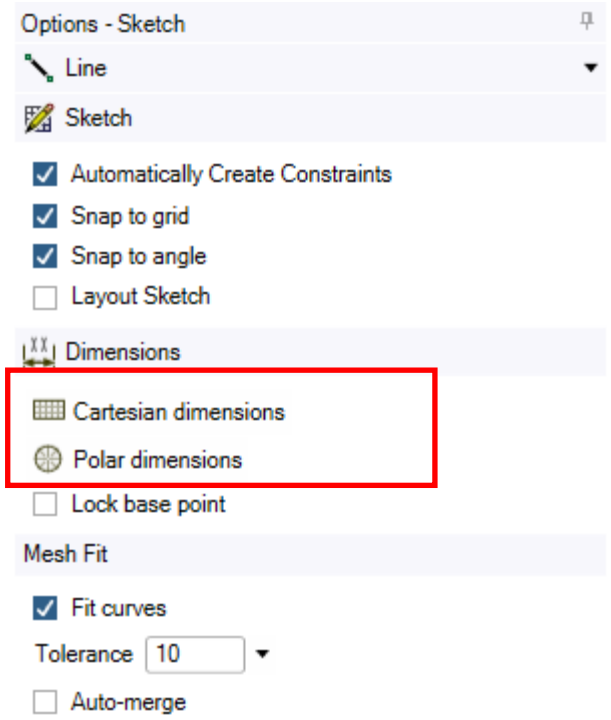


Face Curve

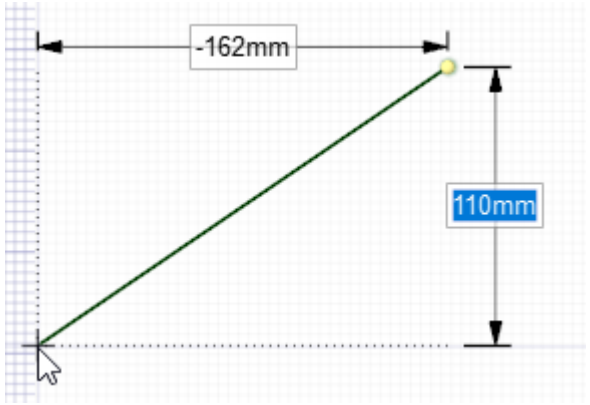
Creating Sketches (3)

- Line Tool

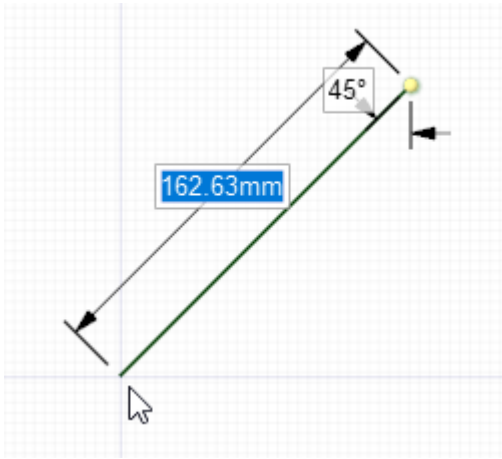




Cartesian Dimensions

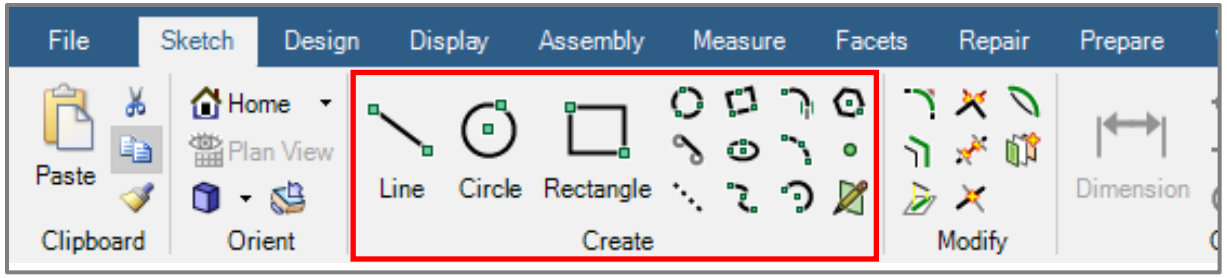


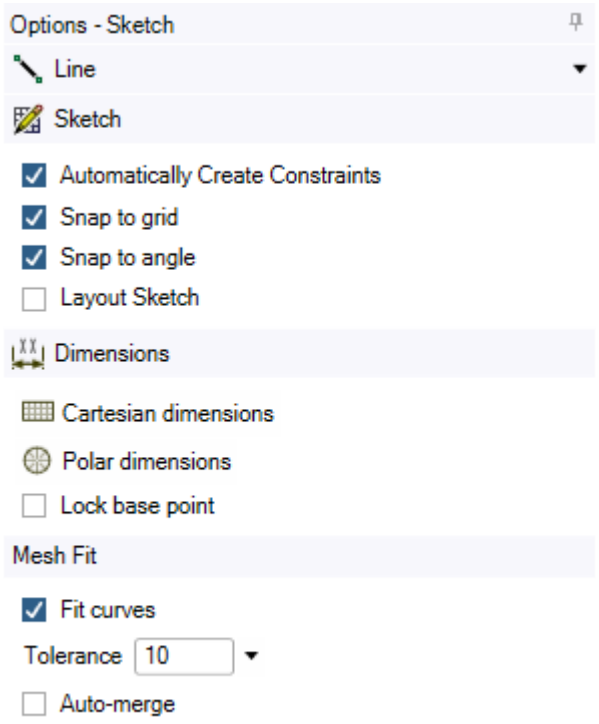
Polar Dimensions



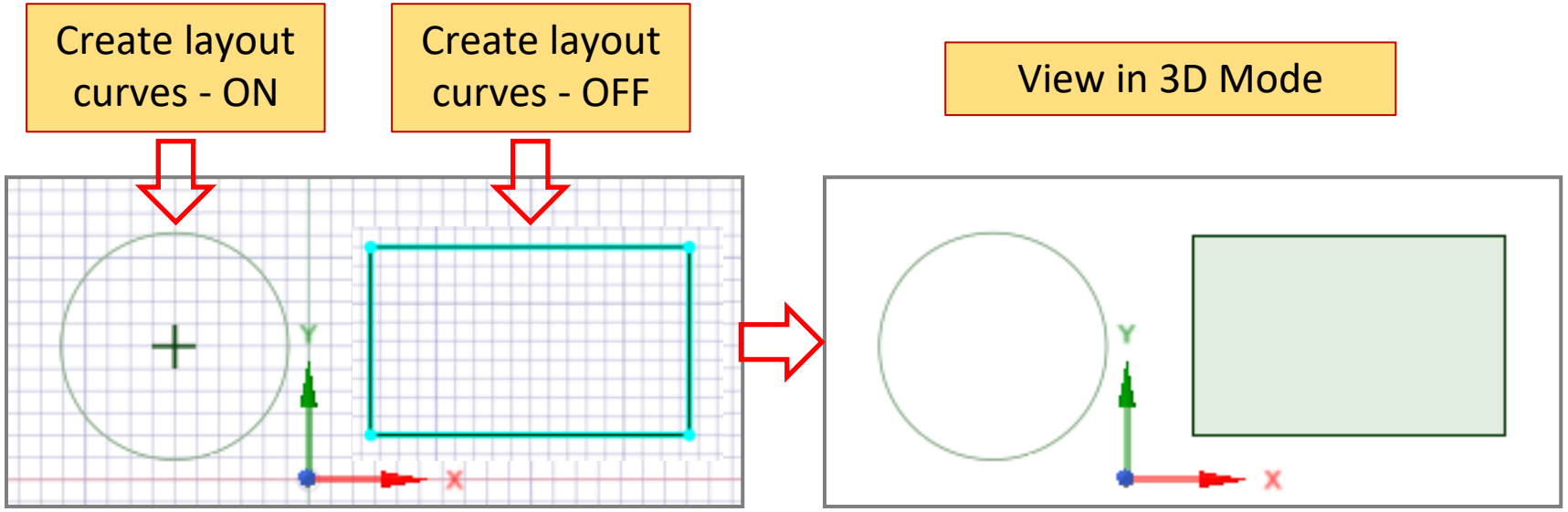
Creating Sketches (4)

- Line Tool

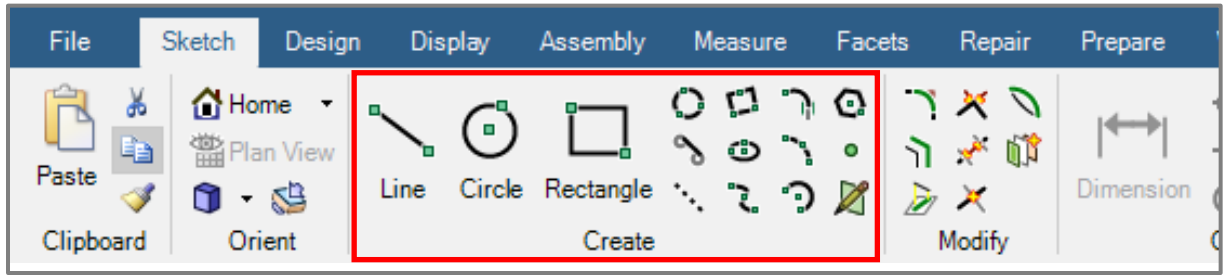




- Create layout curves
  - Create curves which are not immediately needed to generate 3D objects
  - Curves created are similar to a pencil drawing made on design

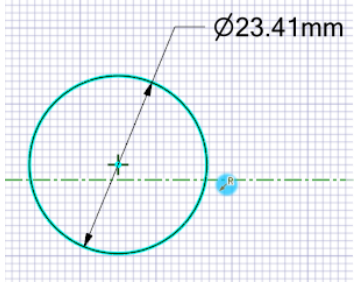


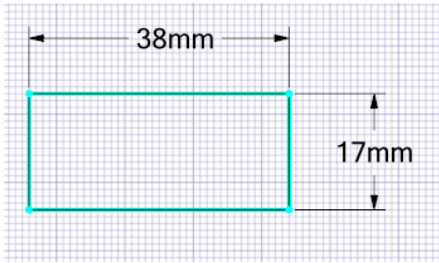
Dimensioning Sketches



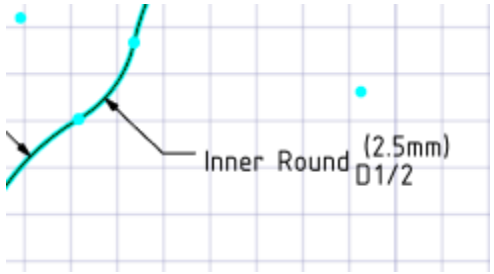
Dimensioning

- Specify sketch dimensions on-the-fly while creating sketches
  - Radius, Length, Width
- Use “Tab” key to switch between dimensions

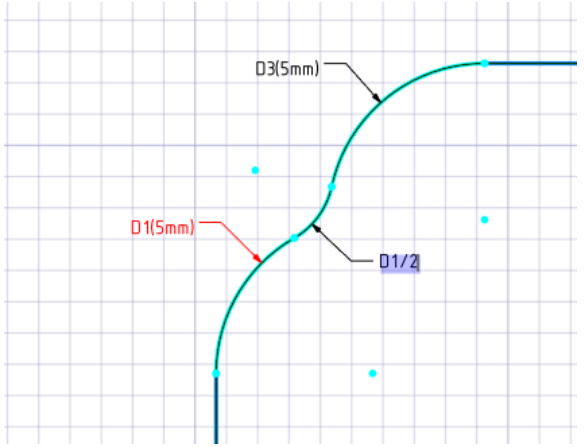




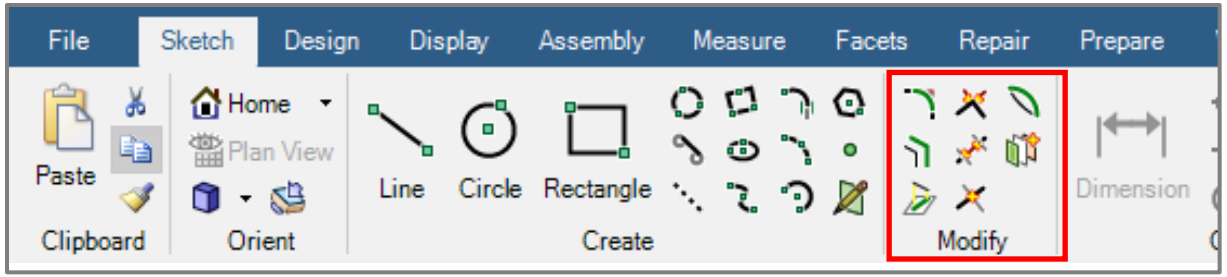
- Dimensional relationships can be created using expressions
  - The expression, label, or value can be changed in the property panel
  - When a dimension is selected, expressions, labels, and values are seen in the design window



General	
Expression	D1/2
Label	Inner Round
Measurement	Radial
Value	2.5mm

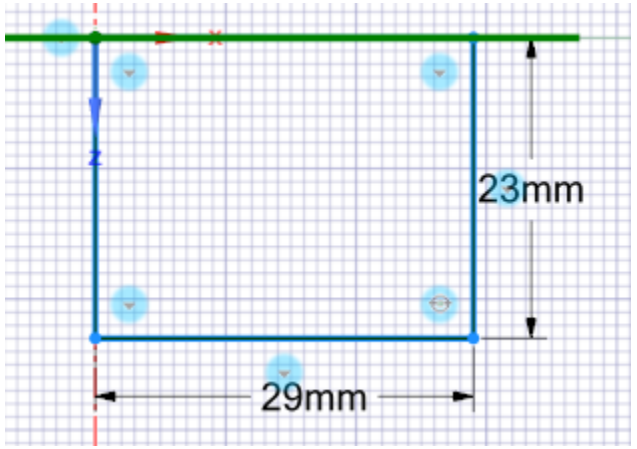


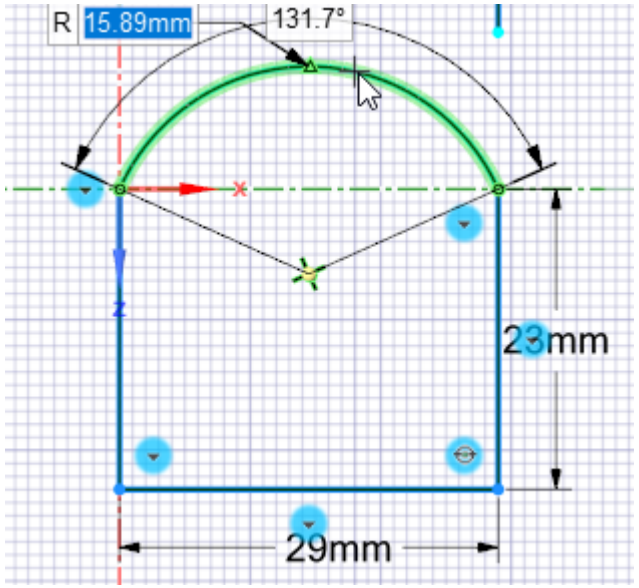
Editing Sketches



Sketch Editing tools

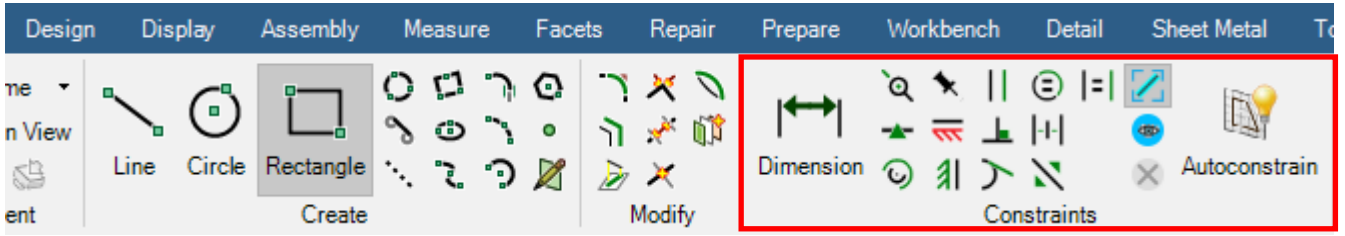
- Hosts standard tools for editing sketches
  - Fillet / Chamfer
  - Offset
  - Project
  - Create Corner
  - Trim
  - Split
  - Bend
  - Scale



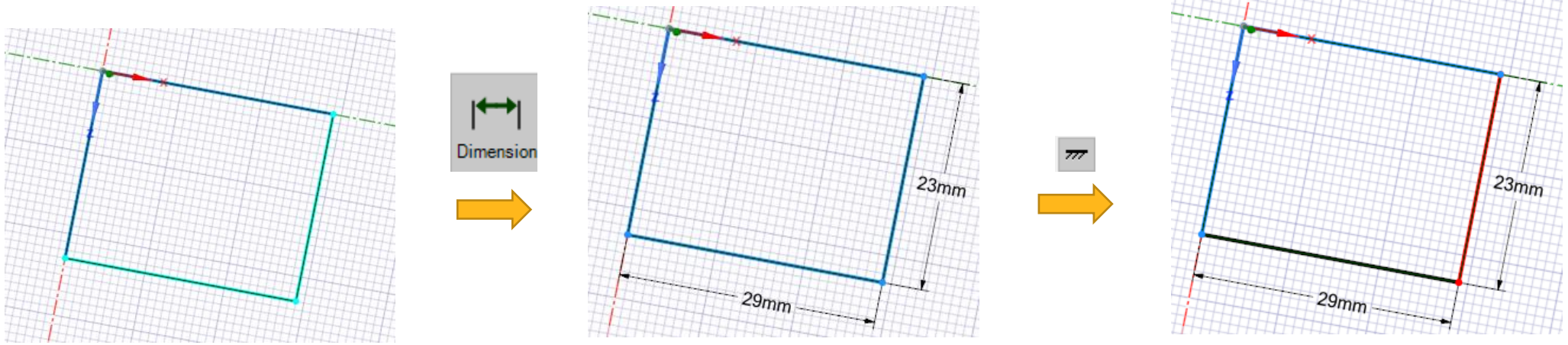


Bend tool

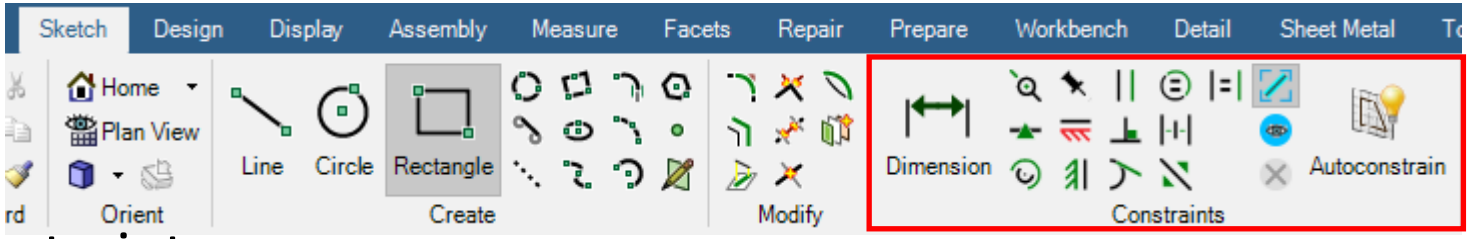
Constraints (1)



- Force curves to be vertical, horizontal, fixed, coincident, tangent, etc.
- Dimensional constraints make creation of exact sketches easier
- Coded colors: dark blue for constained, light blue for non constrained, black for fixed, red for overconstrained



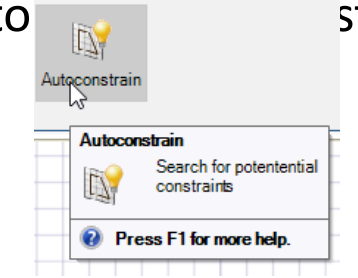
Constraints (2)



- You can find below the different constraints:
  - Dimension
  - Coincident
  - Midpoint
  - Concentric
  - Fixed
  - Horizontal
  - Vertical
  - Parallel
  - Perpendicural
  - Tangent
  - Equal Radius
  - Equal Distance
  - Symmetric
  - Equal Length

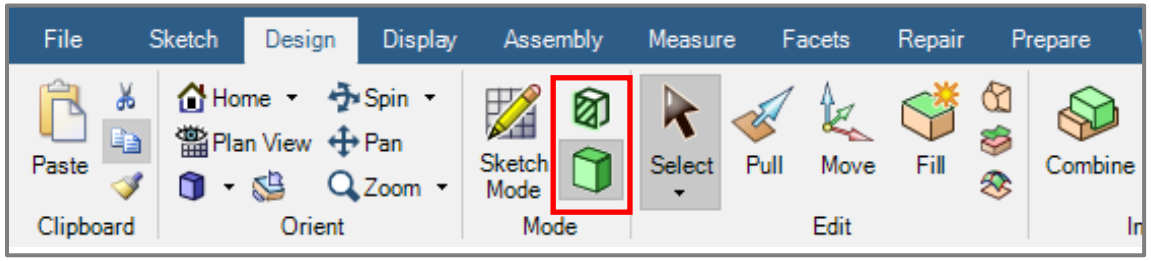
- Other actions:
  - Show Constraint Colors
  - Show Constraints Tips
  - Delete Constraint

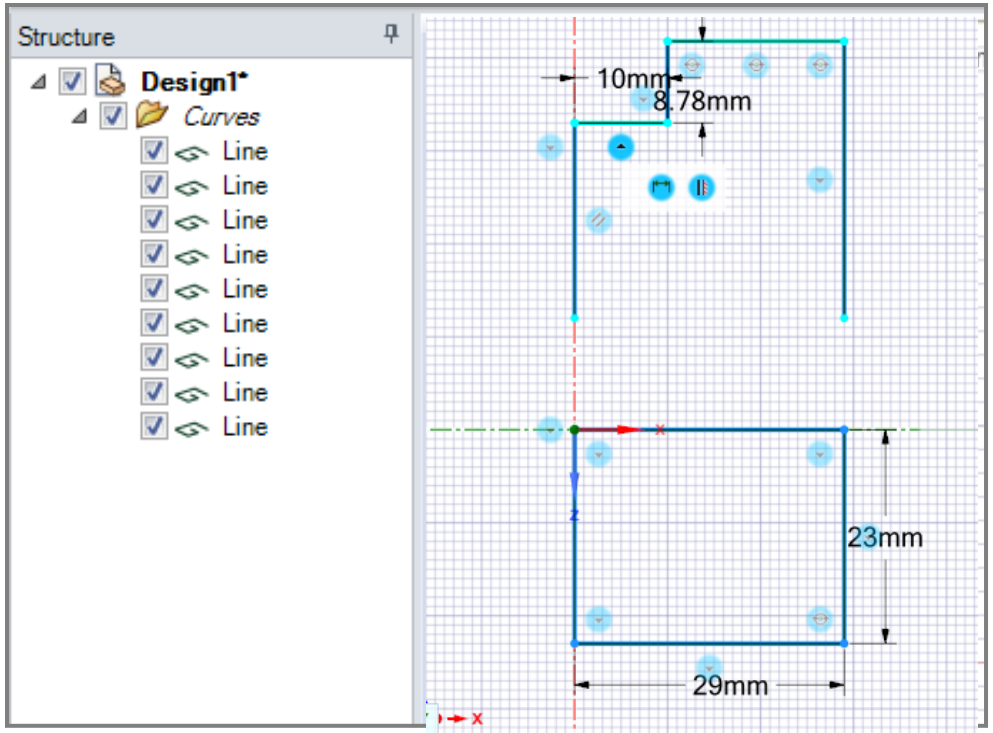
- “Autoconstrain” tool
  - Finds constraints that can be added to a sketch following a find-> fix paradigm
  - Useful when reverse engineering or adding constraints to strained sketch



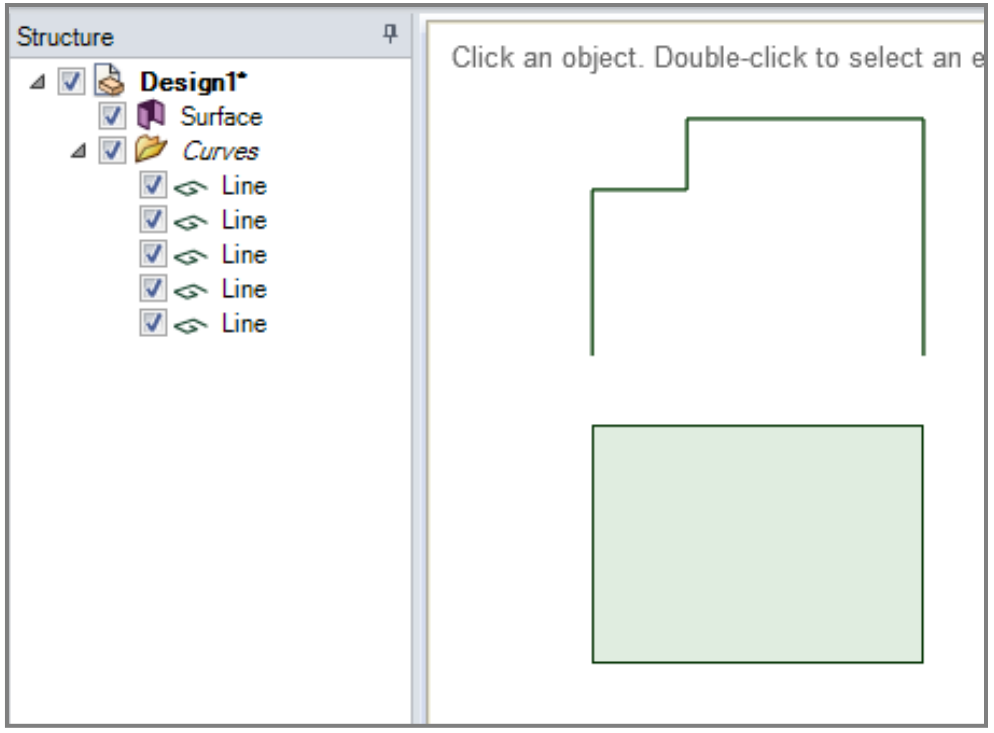
Switching to 3D Mode

- Switching to 3D mode converts
  - All closed sketch objects to surface bodies
  - All open sketch objects to curves



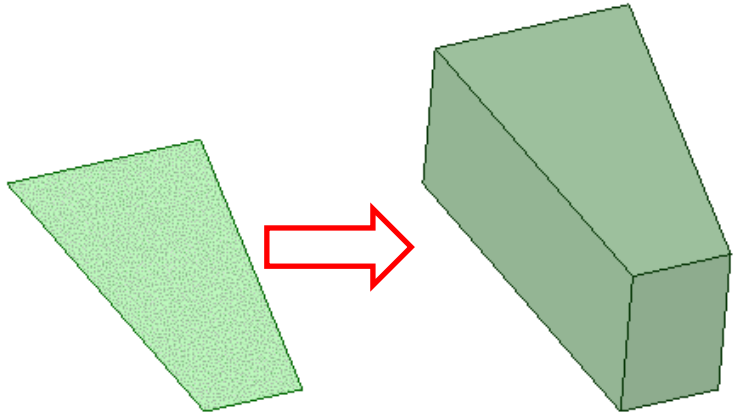


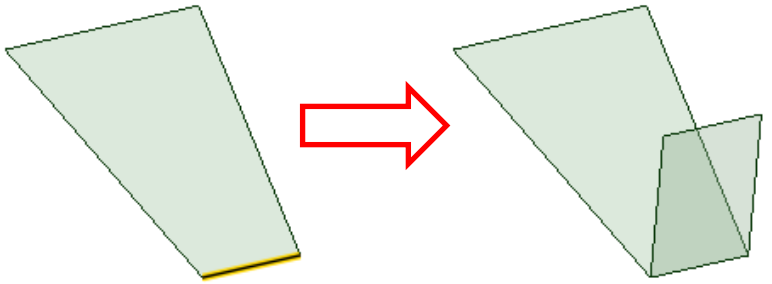
Switching to 3D Mode

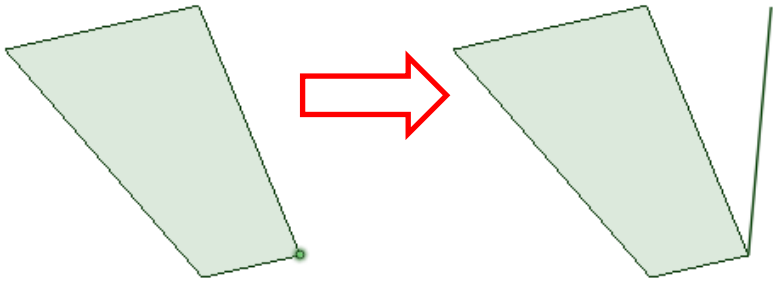


Modeling Tools – Pull (1)

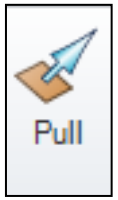
- Used to convert sketches to 3D
  - Pulling a line creates a surface
  - Pulling a surface creates a solid
- Distort or deform existing geometry
- Drag the selected object in a chosen direction when Pull tool is active
- Multi-functional tool
  - Extrude, Revolve, Sweep, Offset and Draft faces
  - Create Rounds (Fillets), Chamfers or Extrude edges
  - Pull a point to create Line
- Several tool guides available to alter its behaviour

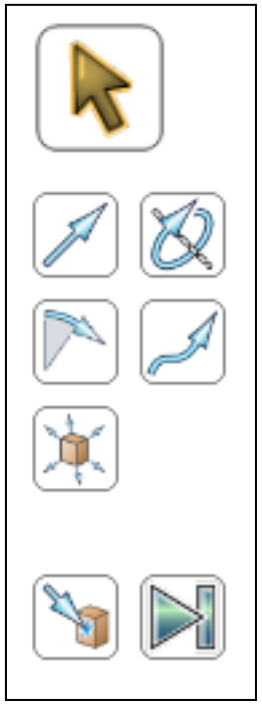






Modeling Tools – Pull (2)

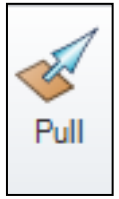


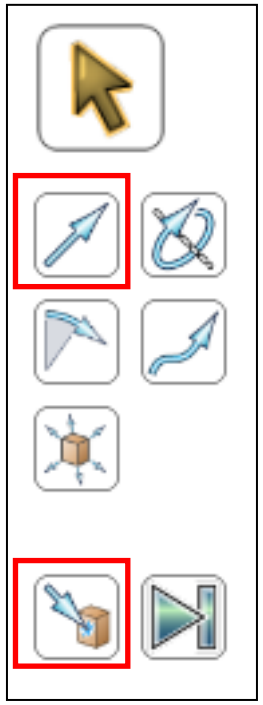


Tool Guide

	**Select -** Select objects to pull
	**Pull Direction -** Set direction of pull
	**Revolve –** Set axis of revolution
	**Draft -** Select entities (plane, planar face or edge) as pivot to create draft
	**Sweep -** Select entities (edges, lines) to sweep along
	**Scale –** Scale selected entities
	**Up To -** Select destination object
	**Full Pull –** Revolve 360 deg or pull upto trajectory end

Modeling Tools – Pull (3)





Tool Guide

Face selected for Pull

Blue Edge selected for Pull direction

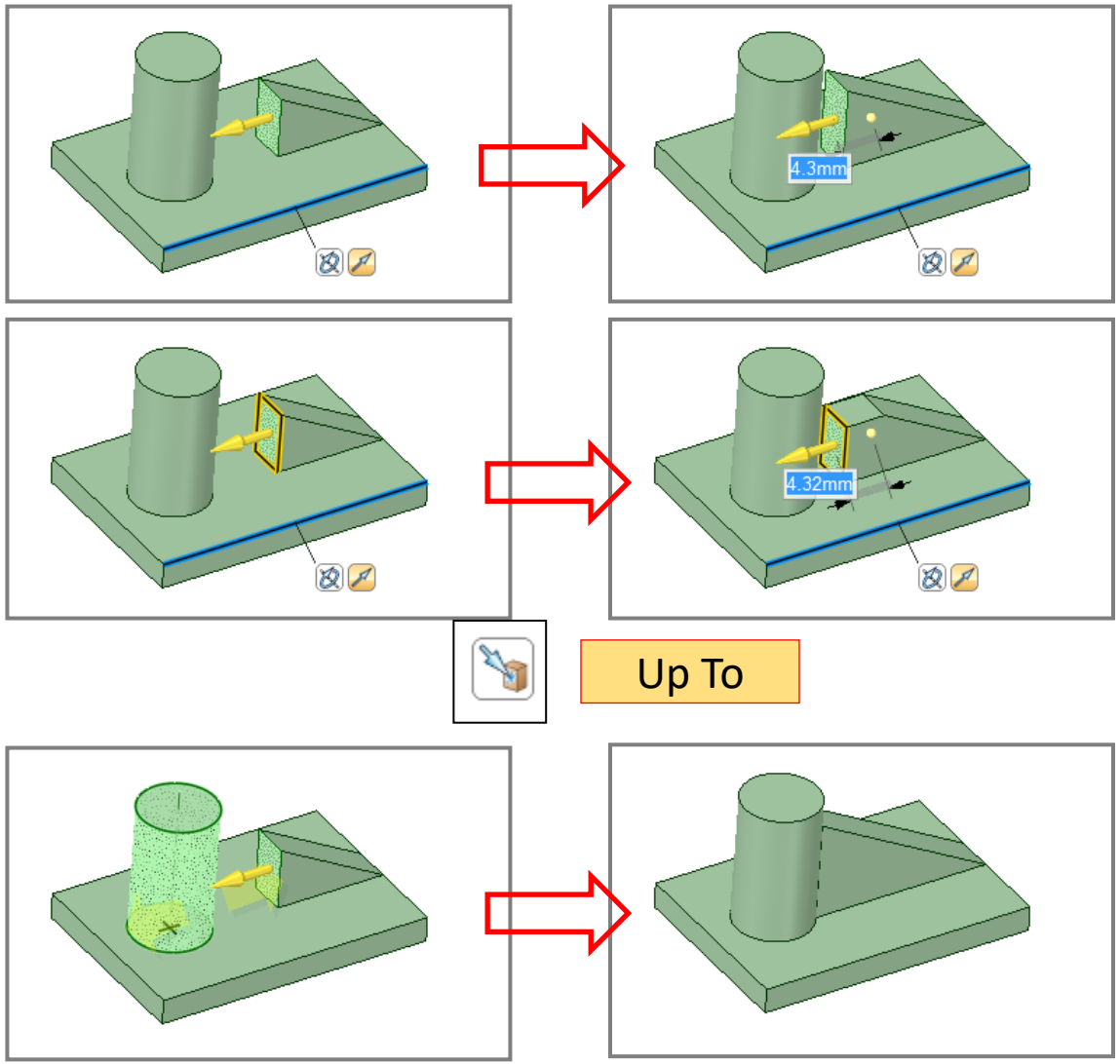
Face and its edges selected for Pull

Blue Edge selected for Pull direction

Face selected for Pull

Cylindrical face selected as destination.
The selection used for Up To can come from the Structure Tree

Pull Direction



Up To

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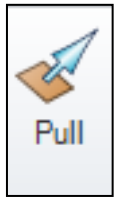
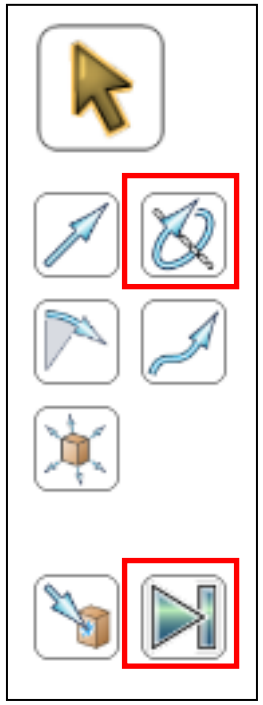
Modeling Tools – Pull (4)



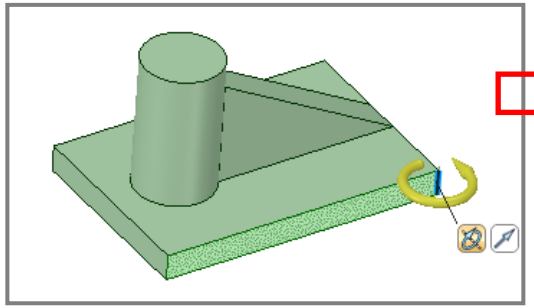


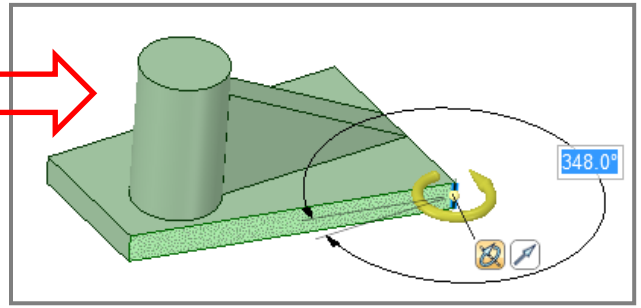
Tool Guide

Revolve

Face selected for Pull

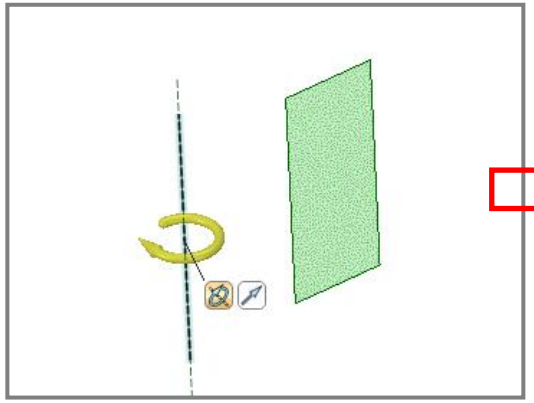
Blue Edge selected as axis of revolution

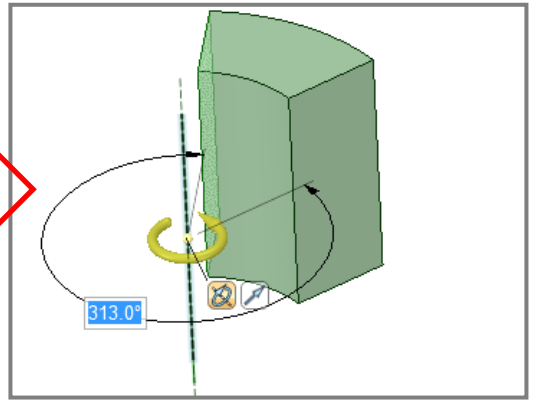




Face selected for Pull

Blue Axis selected as axis of revolution

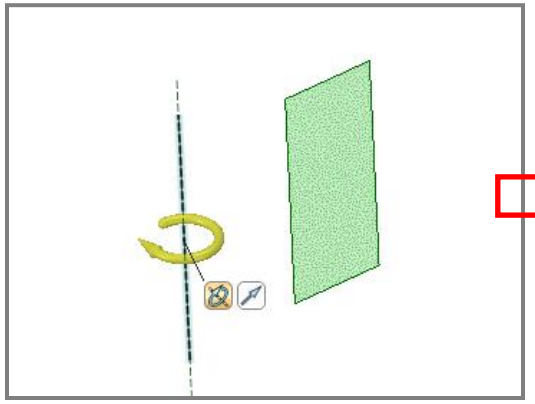


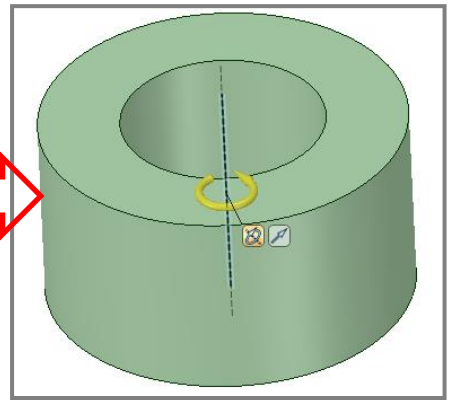


Revolve with Full Pull

Face selected for Pull

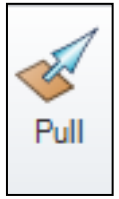
Blue Axis selected as axis of revolution



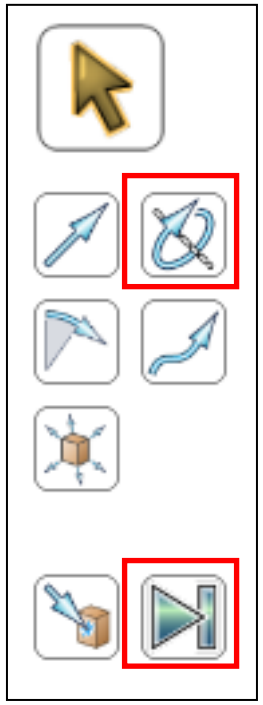


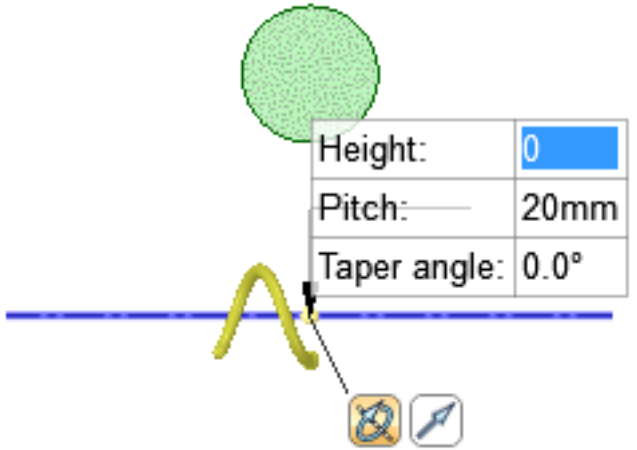
Ansys

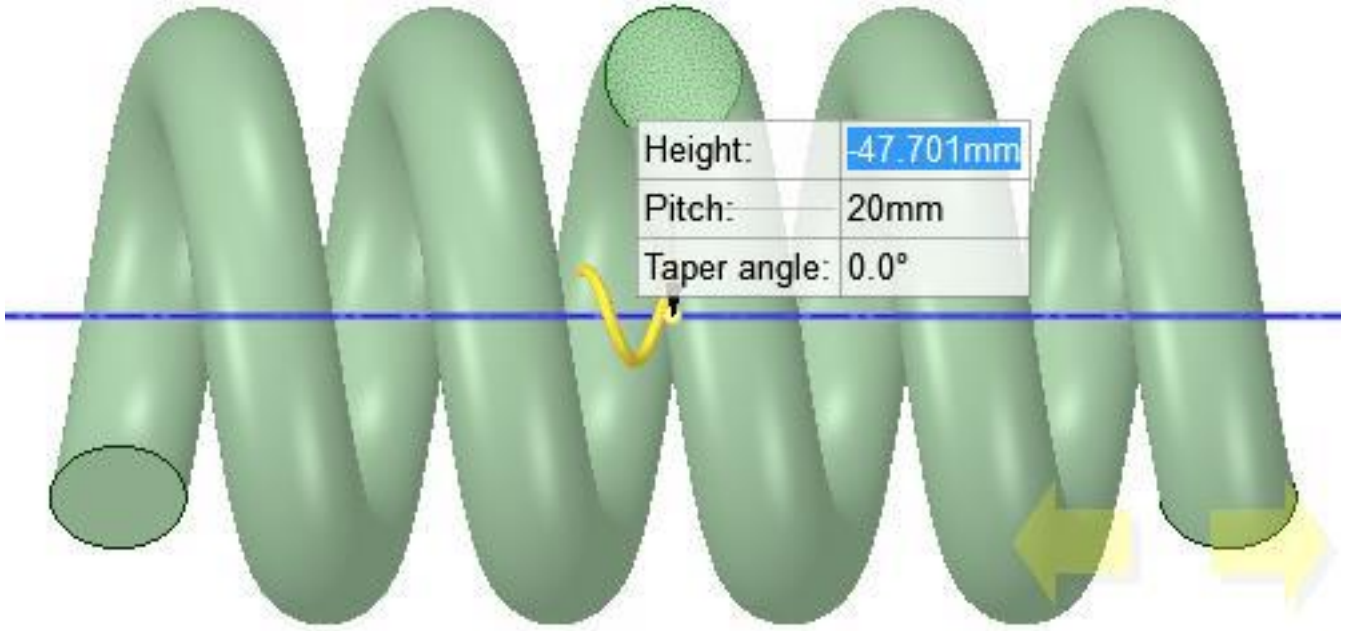
Modeling Tools – Pull (5)



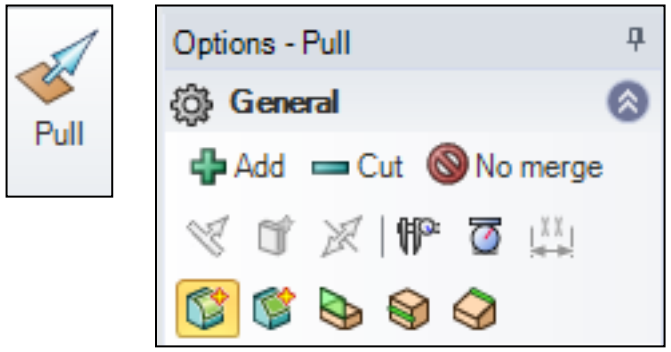
In the **Pull** tool, the **Pull Both Sides** Option works for Helices.



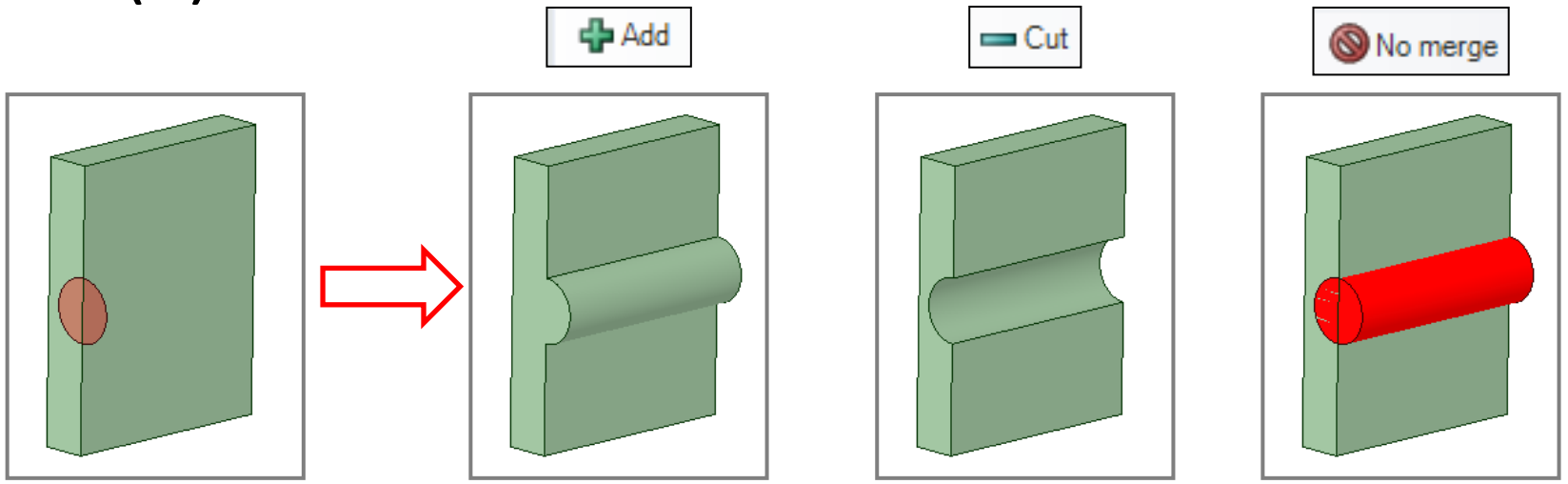


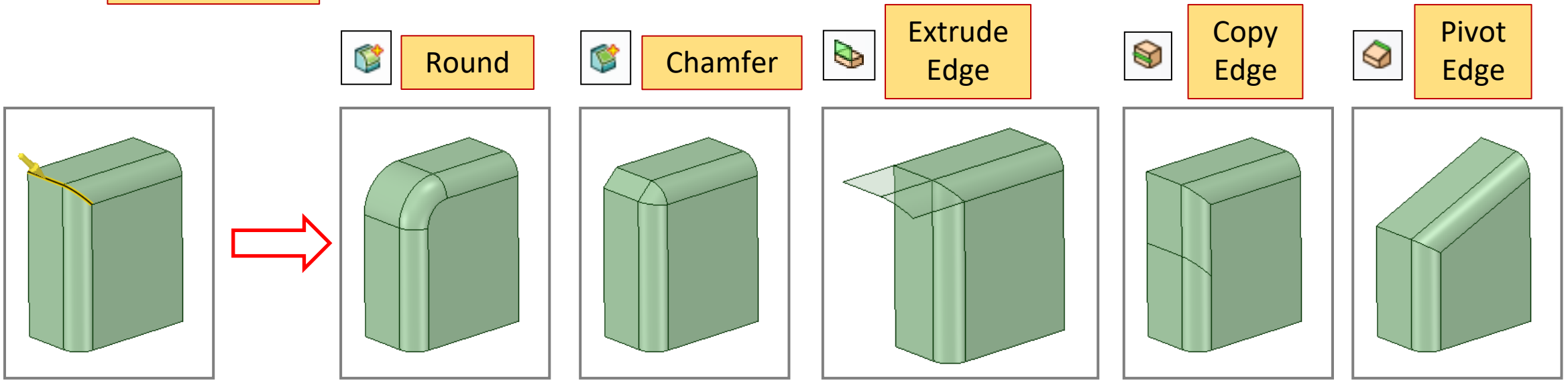


Modeling Tools – Pull (6)



Tool Options

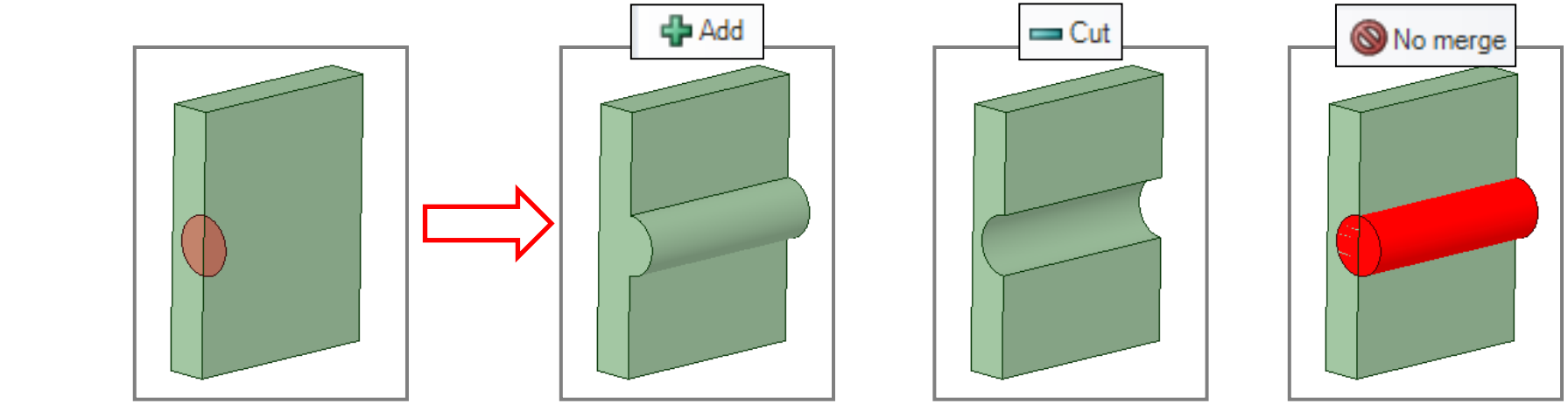




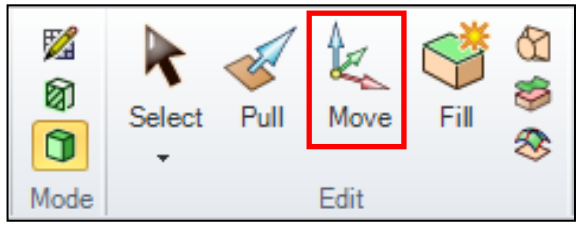
Face gets split

Quick Workshop 02.1

- Import the ‘QuickWorkshop02.1’ geometry and try the different ‘Pull’ options yourself

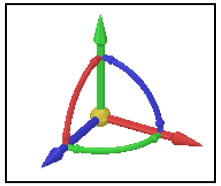


Modeling Tools – Move (1)

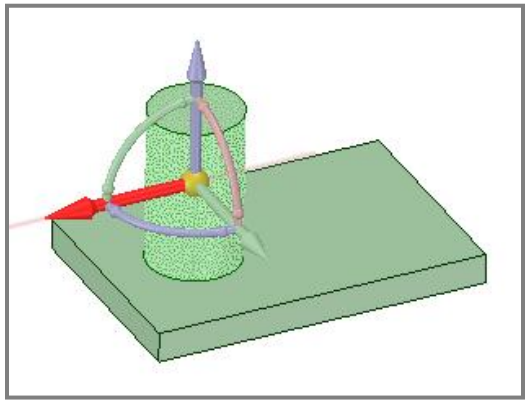


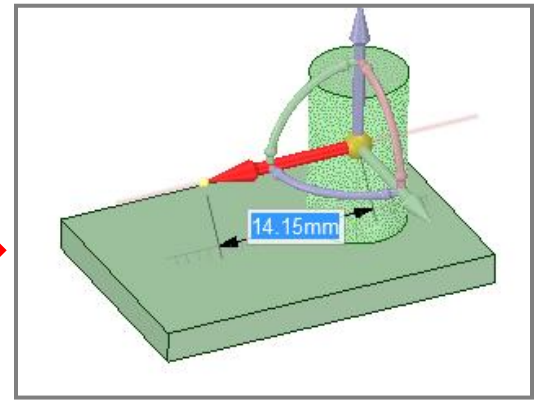
- Move tool
- Multi functional tool
  - Translate
  - Rotate
  - Pattern
- Move handle guides the direction of movement
  - Translational movement along 3 “linear” axes
  - Rotational movement along 3 “curved” axes
- Drag the selected object along the Move handle axis (linear, curved) to facilitate Move
- Distort or deform existing geometry
- Several tool guides available to alter tool behaviour

Move Handle

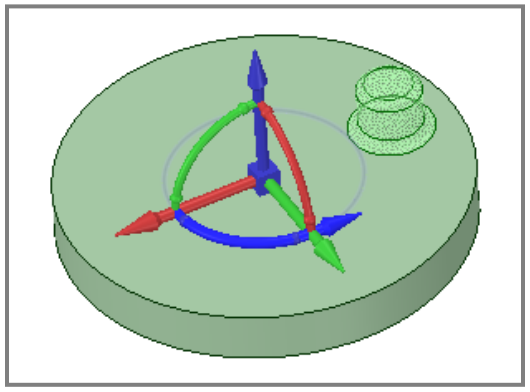


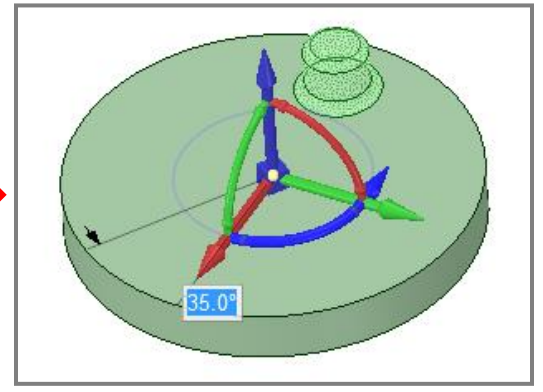
Translation movement



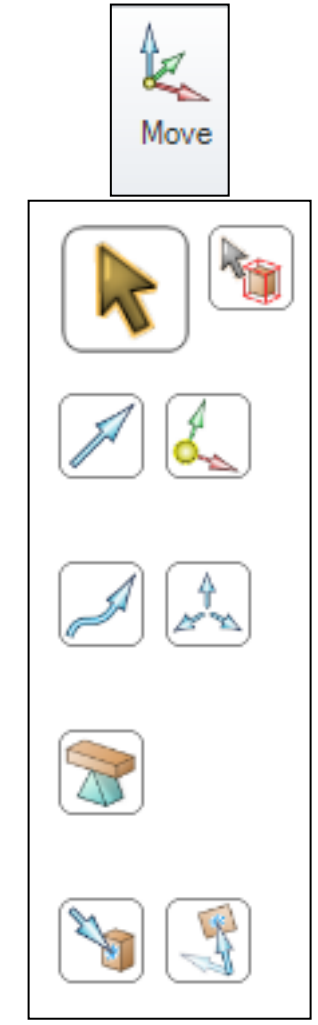


Rotational movement





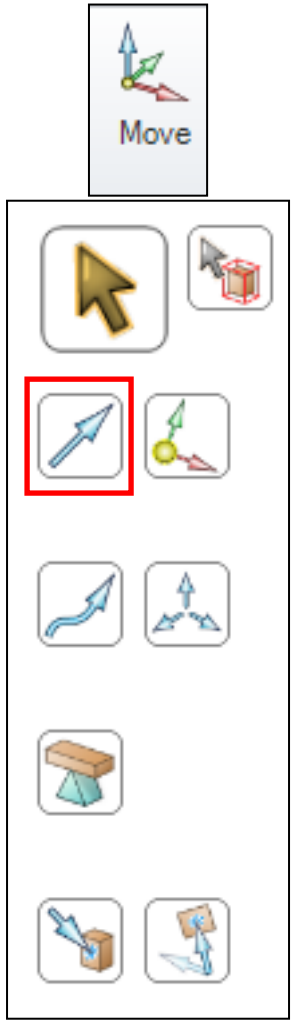
Modeling Tools – Move (2)



Tool Guide

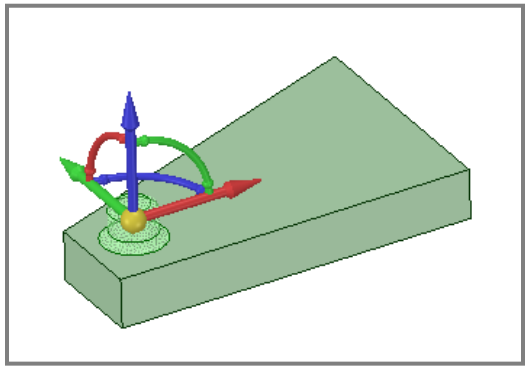
	Select - Select entities (faces, surfaces, solids, or components)
	Select Component - Select solid
	Move Direction - Set direction of move
	Anchor - Change location of move handle
	Move Along Trajectory - Select trajectory (edges, lines) to move along
	Move Radially about axis - Move object radially about axis, line or linear edge
	Fulcrum - Select plane or edge as pivot
	Up To - Select destination object
	Orient To Object - Orient object in selected direction

Modeling Tools – Move (3)

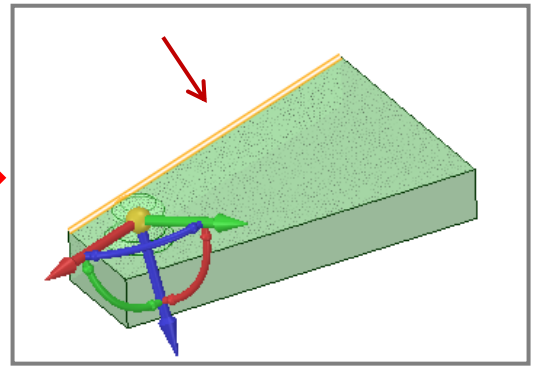


Tool Guide

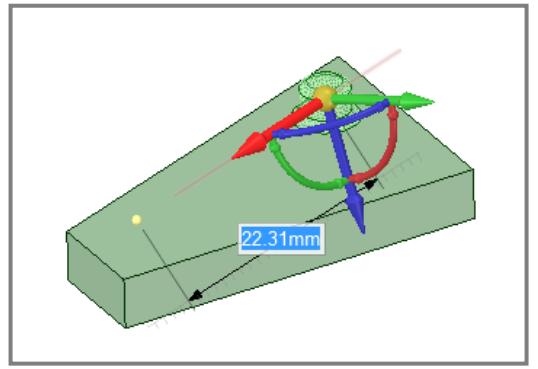
Move Direction



Select object for moving

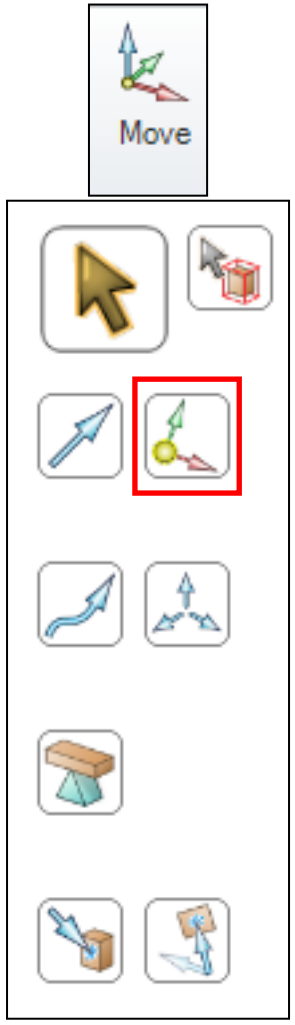


Select Edge as direction. Move handle orients to align Red axis with selected edge



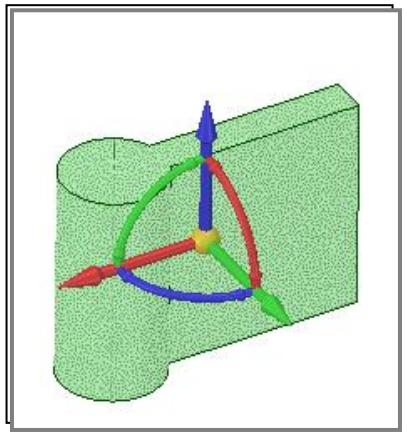
Drag selected object along Red axis to move

Modeling Tools – Move (4)

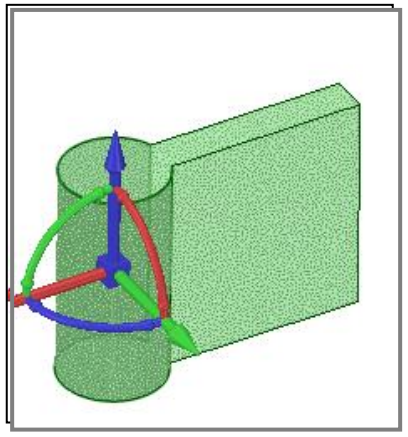


Tool Guide

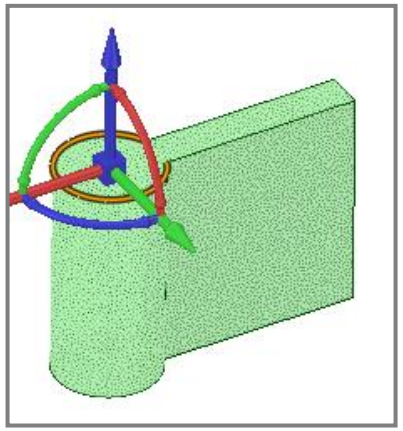
Anchor



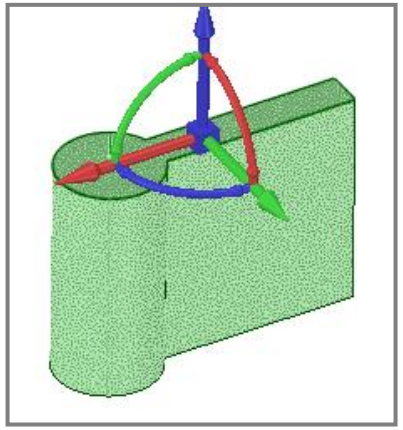
By default, Move handle located at centroid of selected entity



Move handle shifted to centroid of selected “face”



Move handle shifted to centroid of selected “edge”

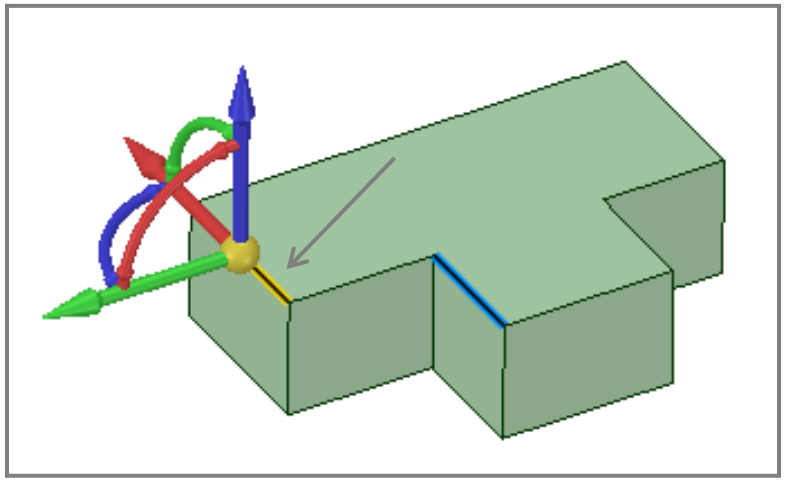


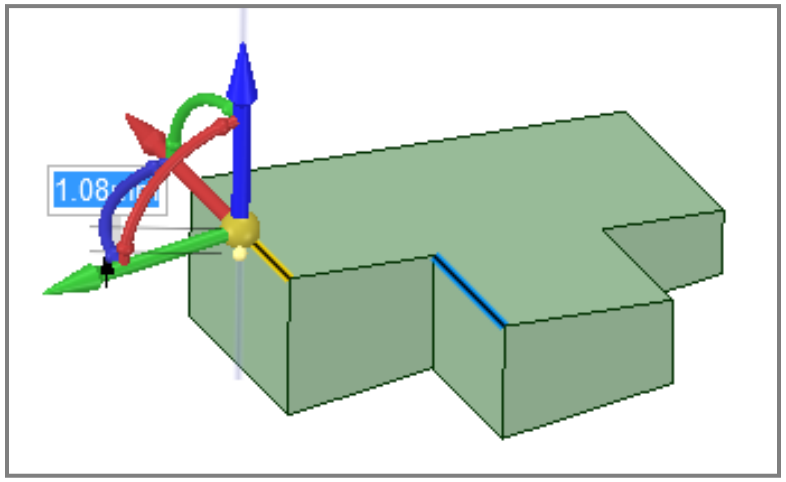
Move handle shifted to centroid of selected “face”

Modeling Tools – Move (5)

Tool Guide

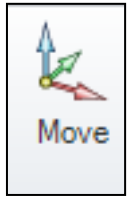
Fulcrum

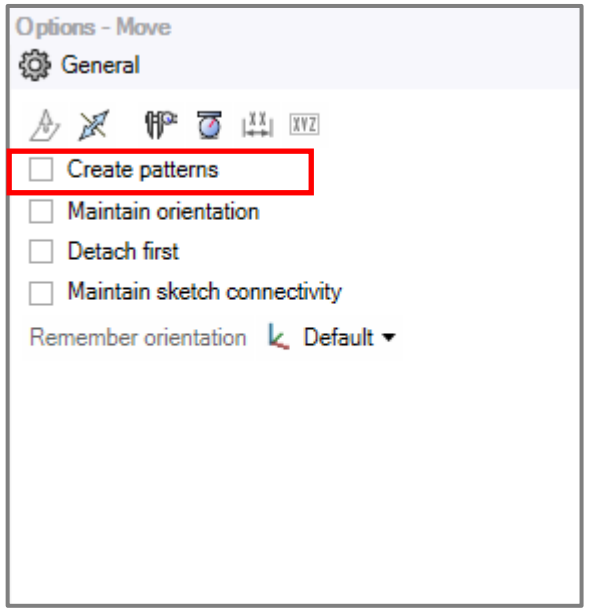




Highlighted edge moved along “Blue” axis with “Blue edge” as fulcrum point

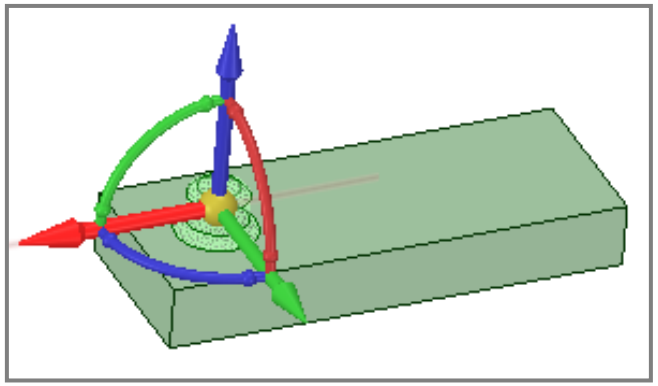
Modeling Tools – Move (6)

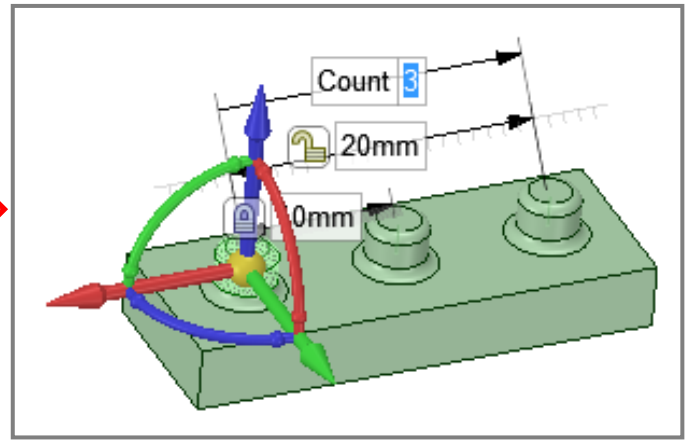




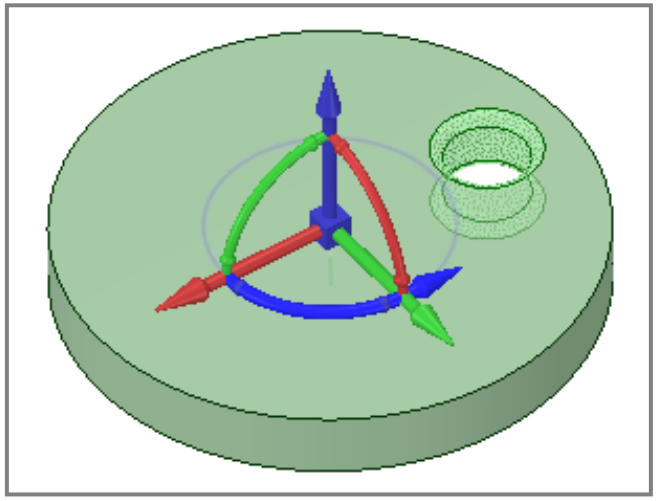
Tool Options

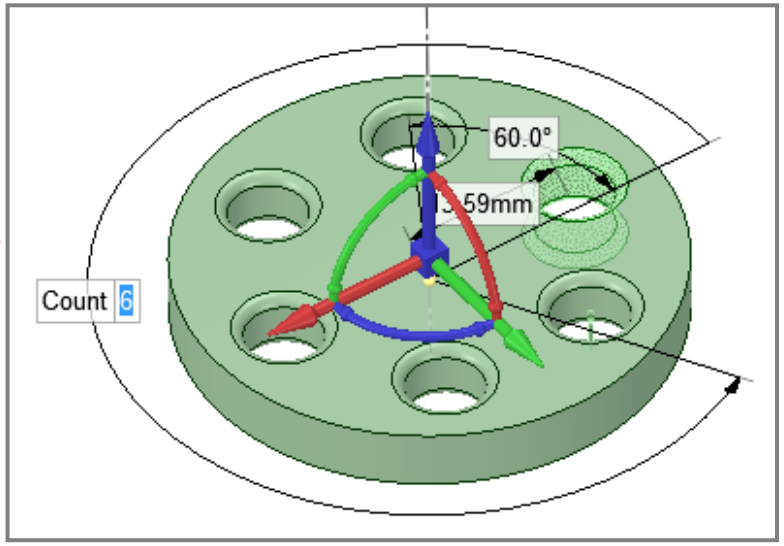
Rectangular Pattern





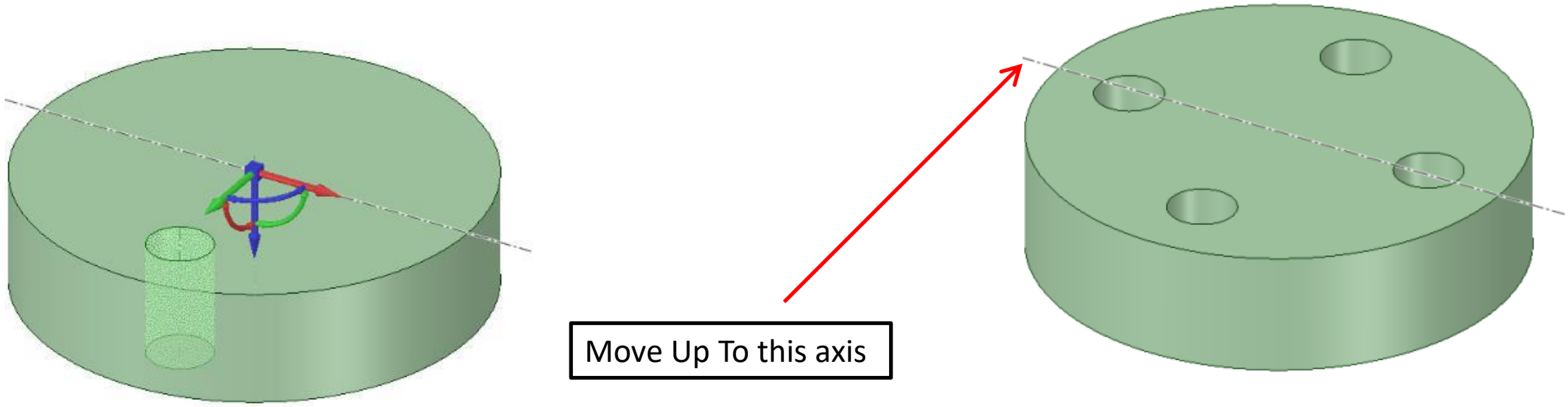
Circular Pattern





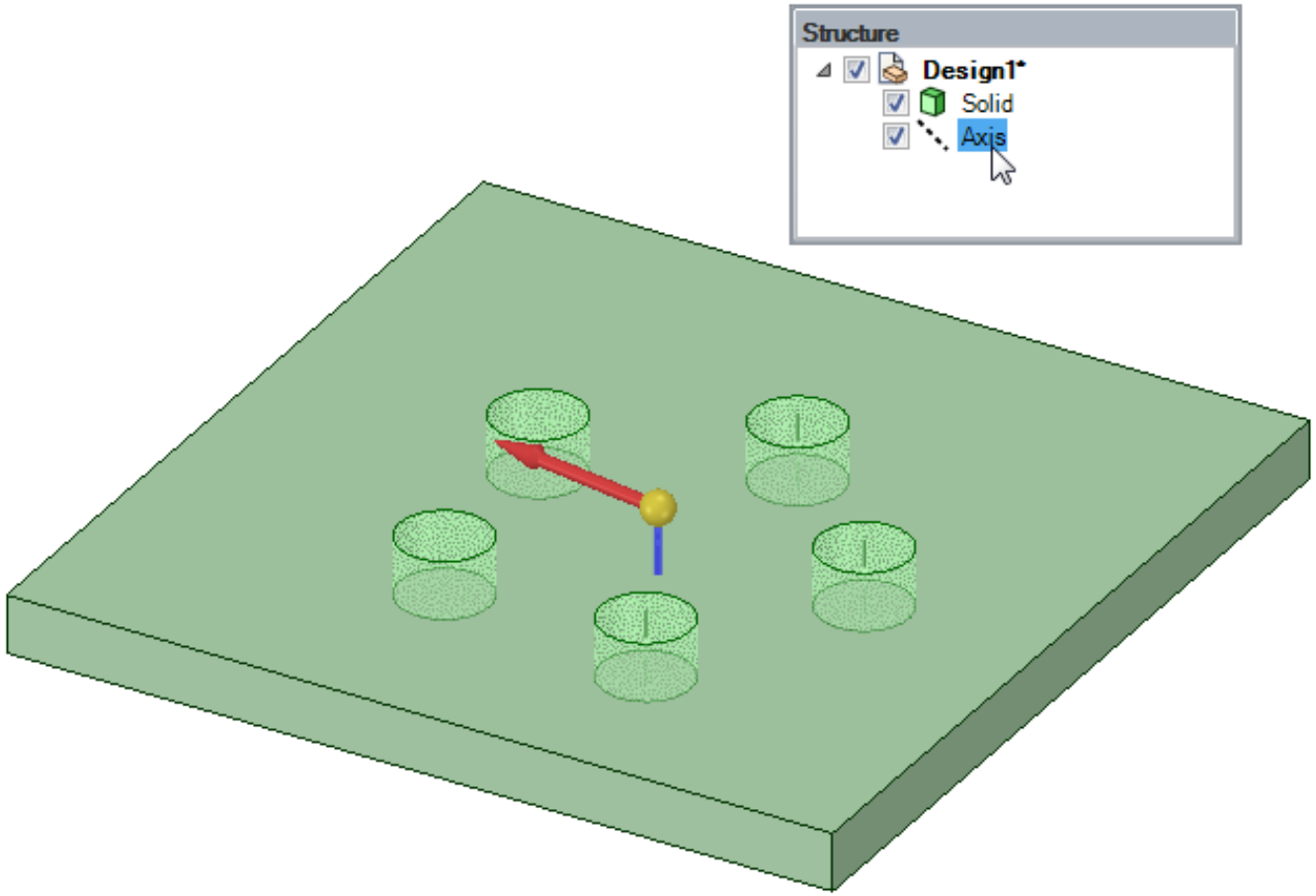
Modeling Tools – Move (7)

- You can create a circular pattern, using the **Up To** tool guide in the **Move** tool, by rotating up to a linear entity passing through the Move Handle origin. Relocate the Move Handle to the axis of the cylinder. Then choose the rotation handle about the cylinder's axis.



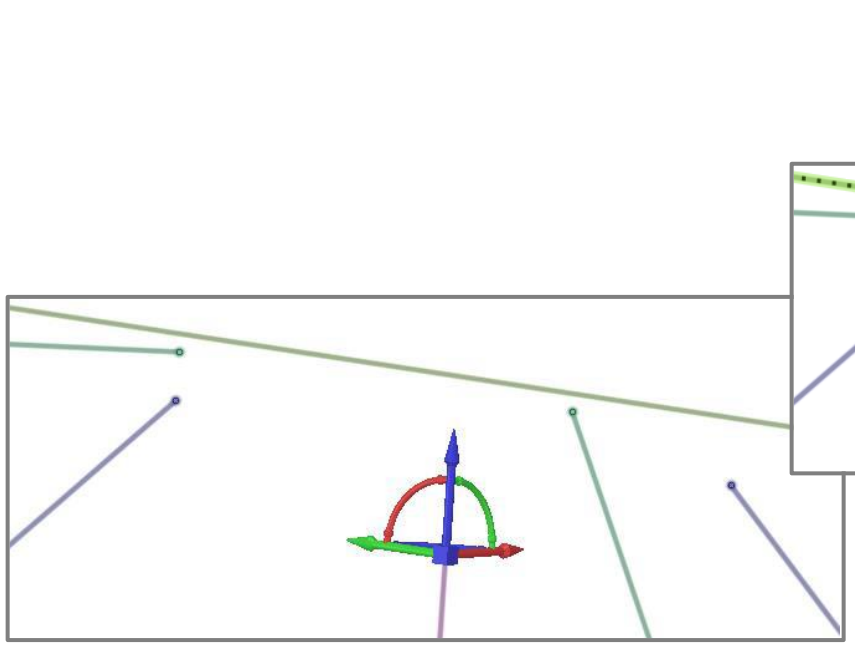
Modeling Tools – Move (8)

- Radial Move allows you to select the Axis in the Structure Tree

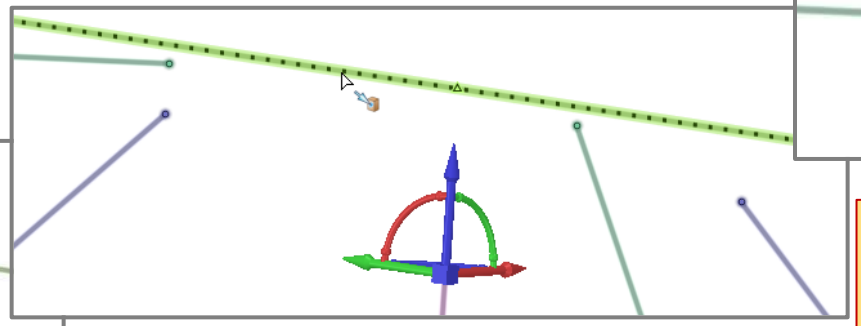


Modeling Tools – Move (9)

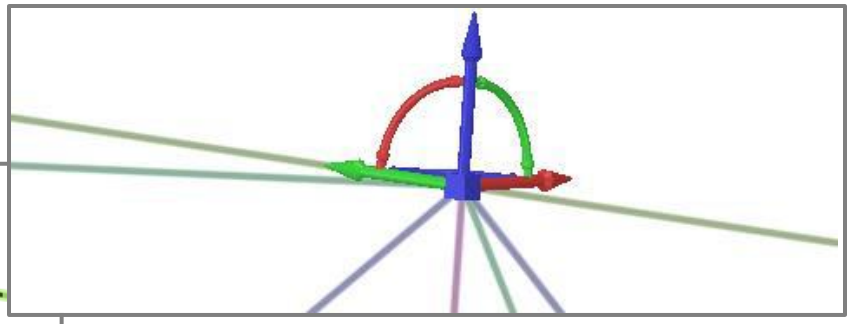
- With the move tool, Curve/Beam endpoints can be moved to a single point



Select all the endpoints and anchor the Move Handle



Click the Up to tool guide and select the Up to reference

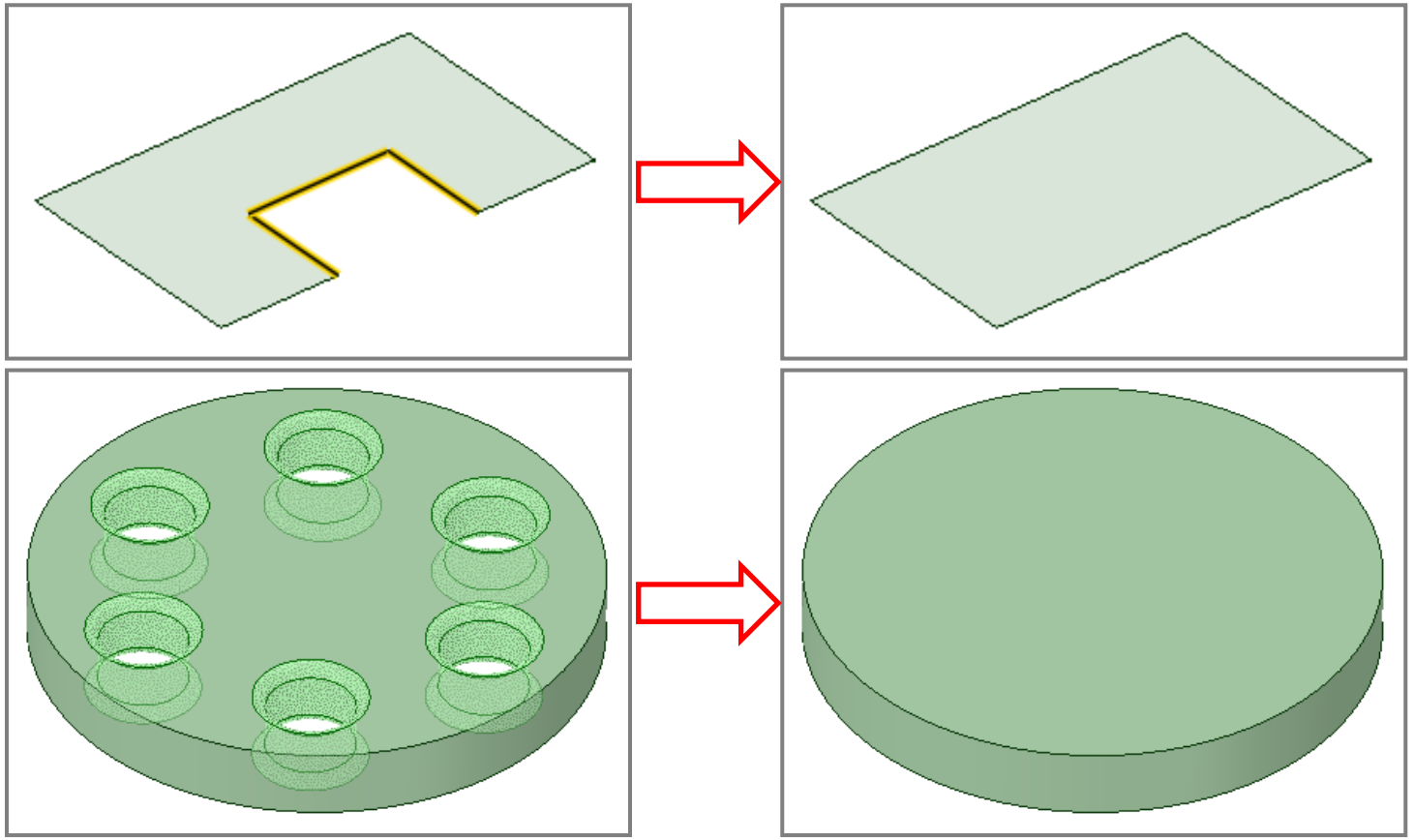


All the beams have been moved into the same point

Modeling Tools – Fill (1)

Fill tool

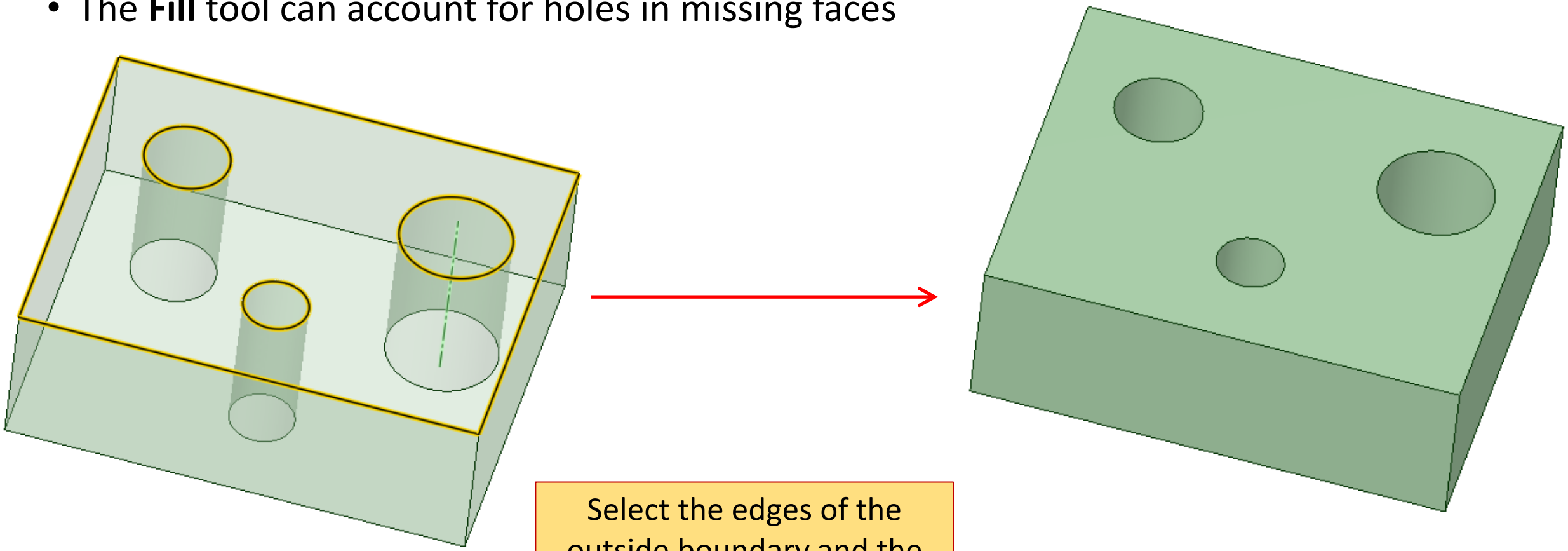
- Fill selected region with surrounding surface or solid
- Acts as a “healing” tool to remove:
  - Fillets
  - Chamfers
  - Holes
  - Protrusions
  - Depressions



Notice: Sometimes a simple delete after selecting the entities can replace the Fill Tool

Modeling Tools – Fill (2)

- The **Fill** tool can account for holes in missing faces

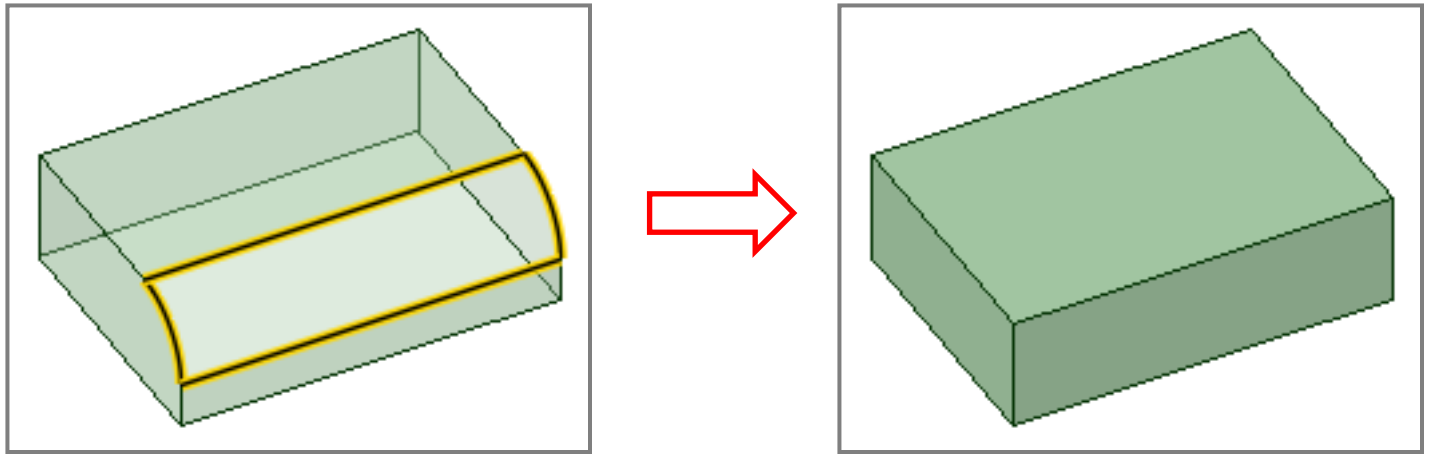


Select the edges of the outside boundary and the interior holes and execute

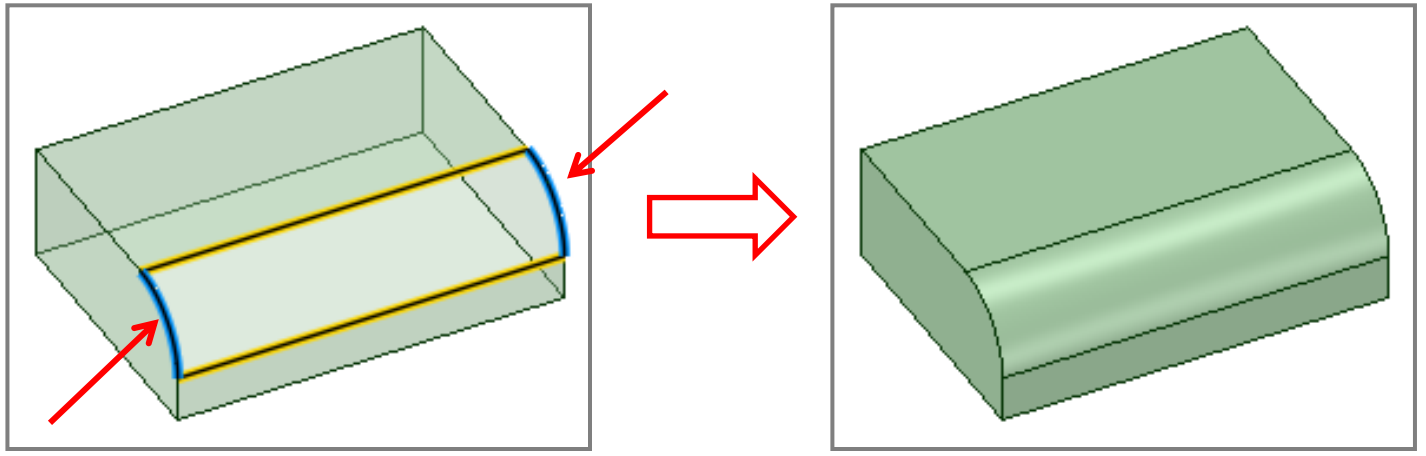
Modeling Tools – Fill (3)

Tool Guide

Fill (Default behavior)



Fill – With Guide Curves

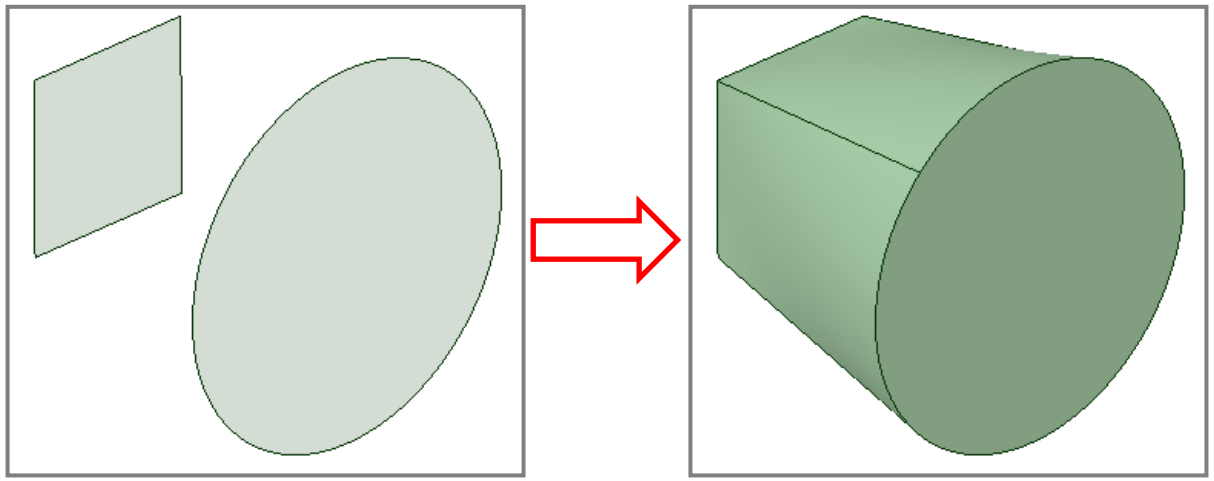


2 curve edges selected as guide curves

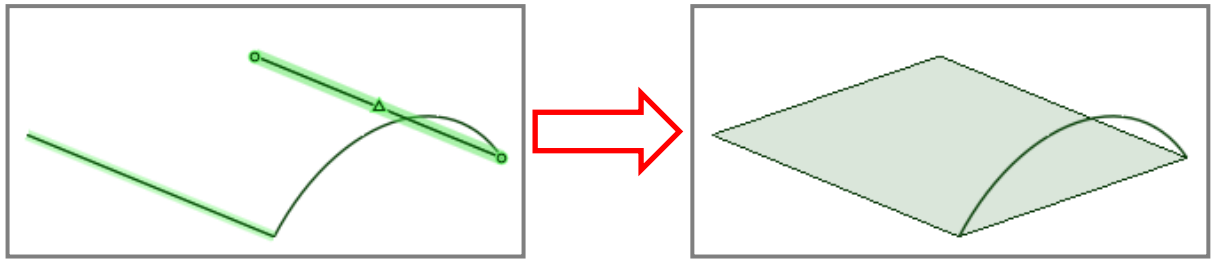
Modeling Tools – Blend (1)

Blend tool

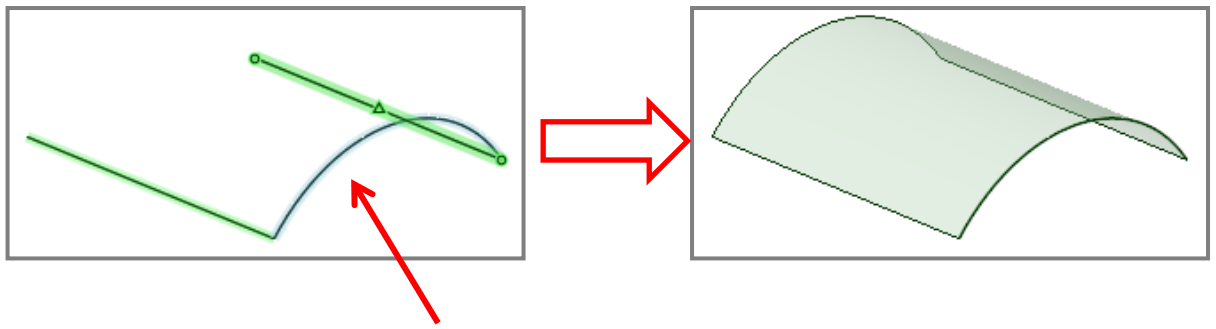
- Create blend between faces, surfaces, edges and curves



Without Guiding curve



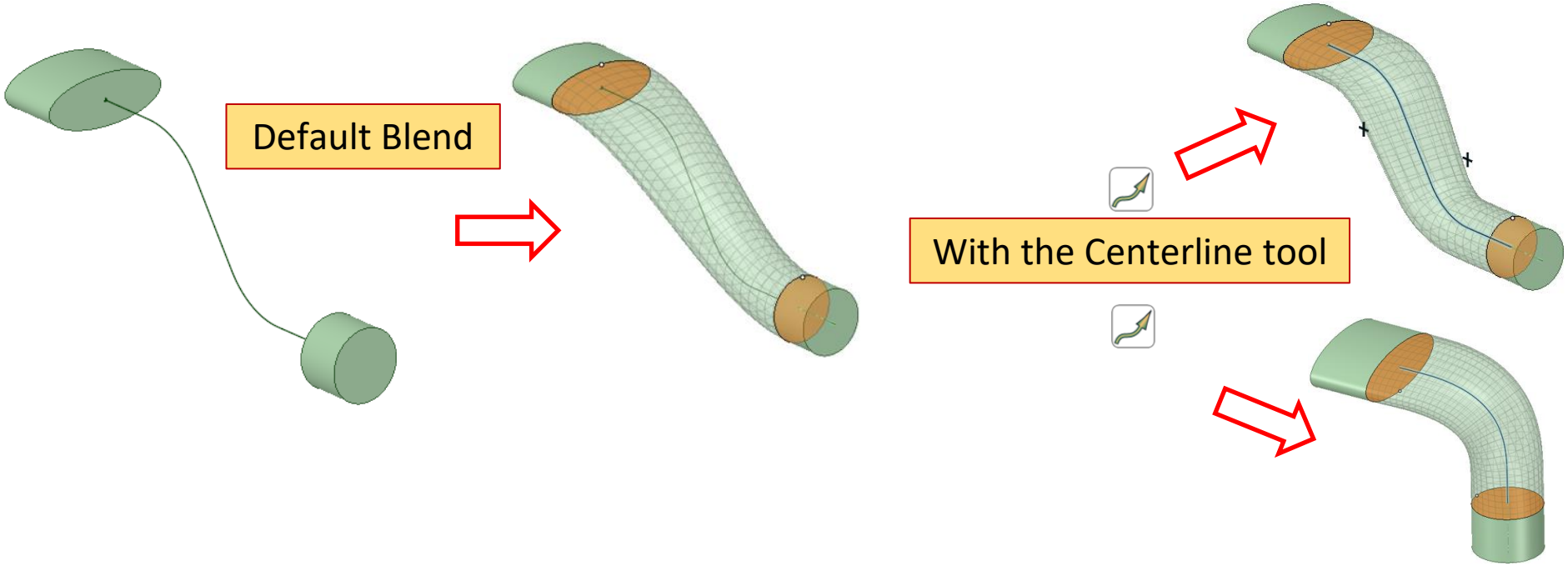
With Guiding curve



Modeling Tools – Blend (2)

Centerline Blends

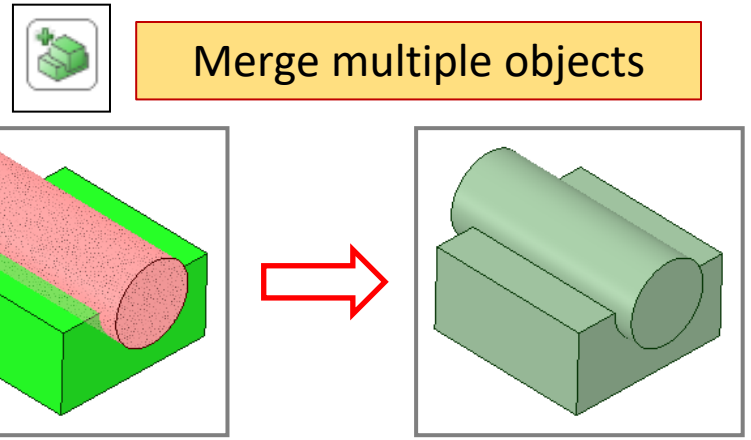
- Allows you to select a centerline path to follow for the Blend. The centerline does not have to touch either of the profiles, but the extension of the curves must pass through the profile. Multi-segmented curves or edges can be used as centerline provided the meet the following criteria: meet end-to-end without gaps and are piece-wise continuous or tangent

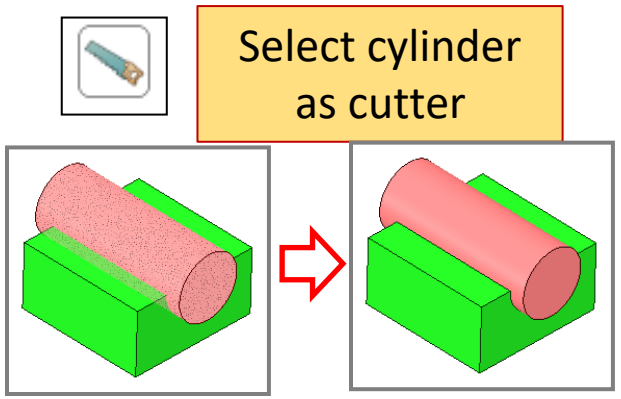


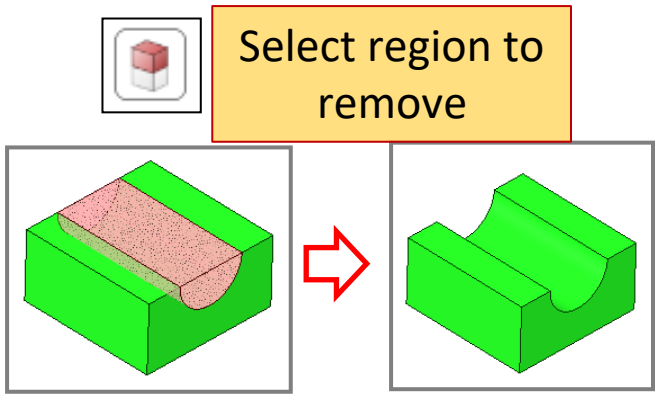
Modeling Tools – Combine

Combine tool

- Perform Boolean operations
  - Add
  - Subtract
- By default, second selected object acts as a cutter to perform subtract operation
  - Cutter tool guide gets activated once a body is selected
- Select multiple objects using “Ctrl” key to add them
  - “Merge” tool guide gets activated automatically
- Option to delete or retain left over region after subtract operation
  - “Regions to Remove” tool guide automatically gets activated after subtract operation





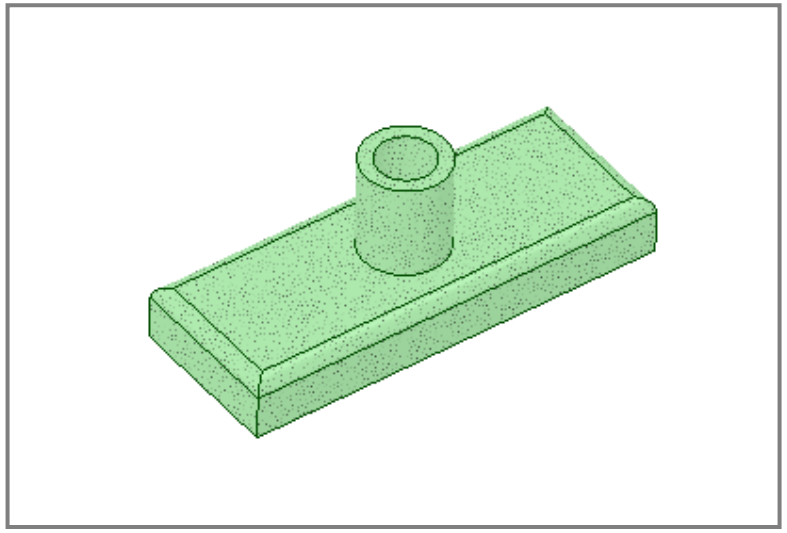


Modeling Tools – Split Body (1)

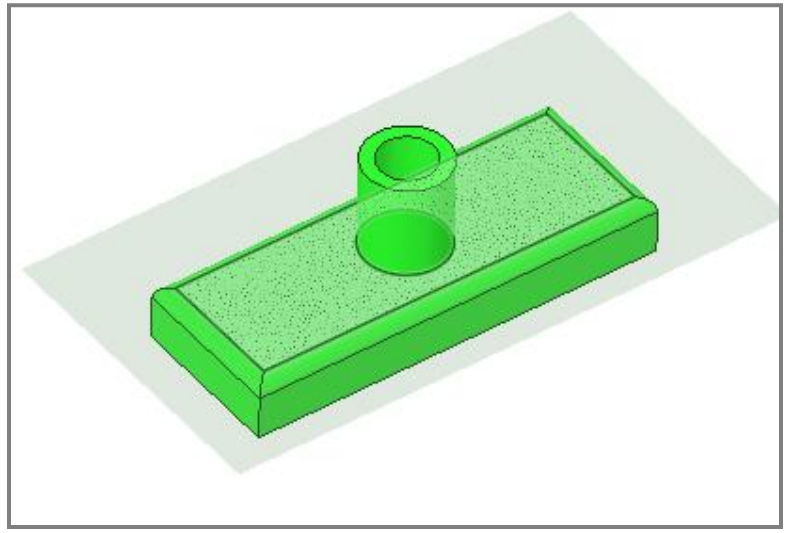
Split Body Tool

- Split solid body using:
  - Face
  - Plane
  - Planar or Cylindrical surface
- Split surface body using edge

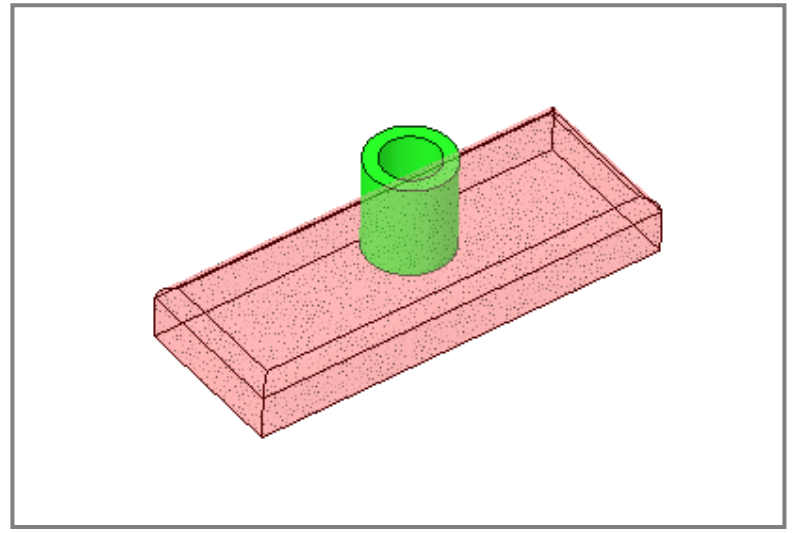
Select body to split



Select face to cut body

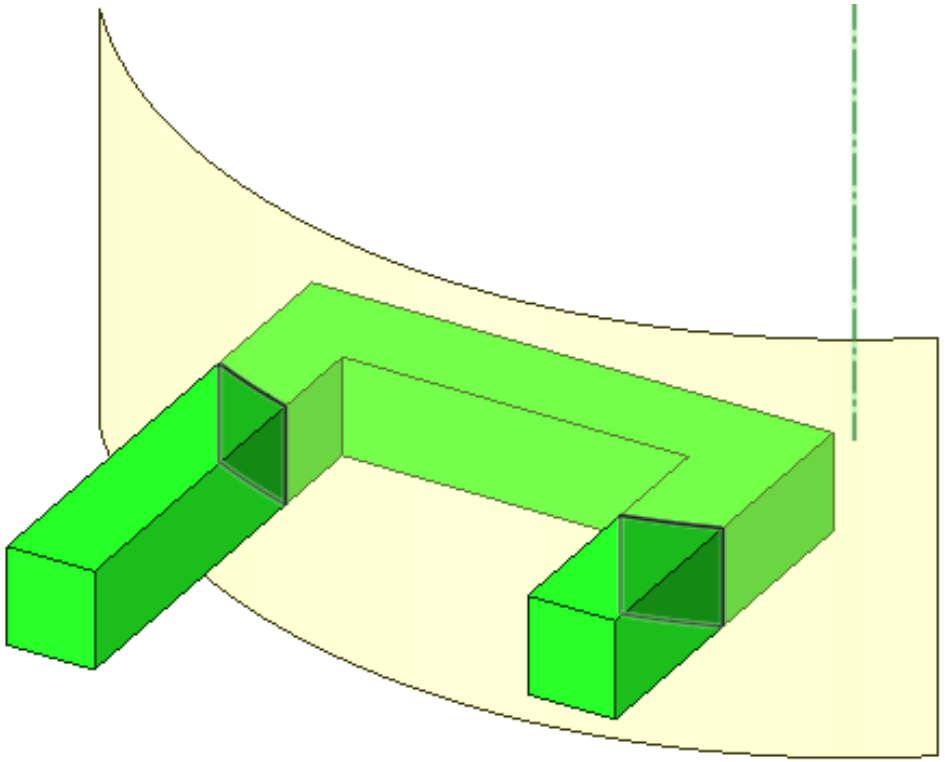


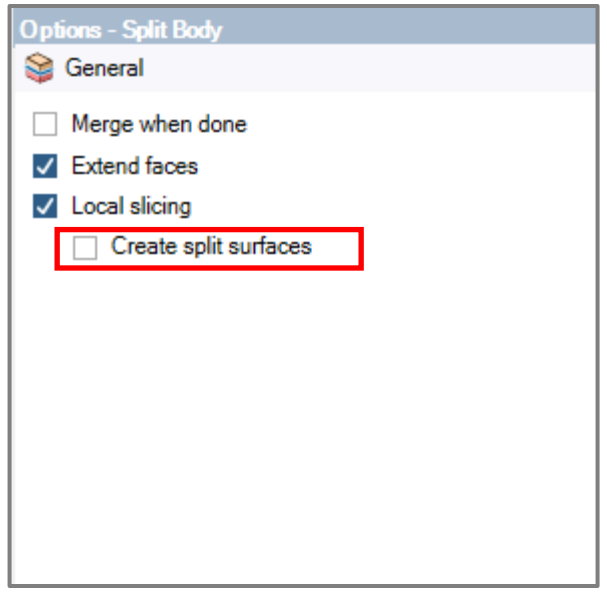
Select region to remove (optional)



Modeling Tools – Split Body (2)

- The **Create split surfaces** option has been added. It creates surfaces at the intersection of the target and cutter





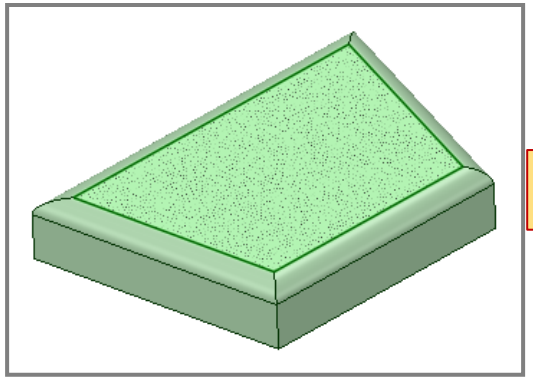
Modeling Tools – Split (1)

The Split tool works on Edges, Curves and Beams will be added later

Split Face Tool

- Split face using:
  - UV Cutter point
  - Perpendicular cutter point
  - 2 cutter points
  - Face

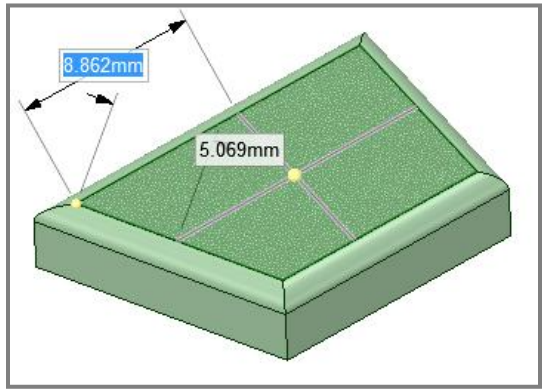
UV cutter point

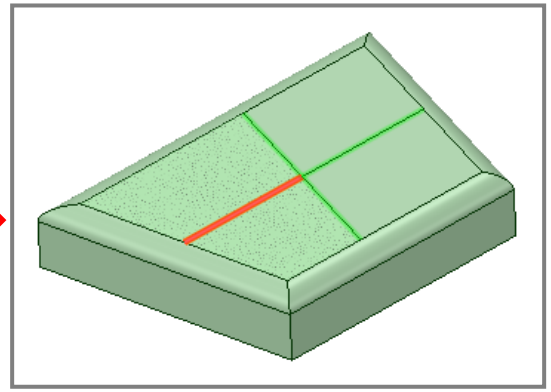


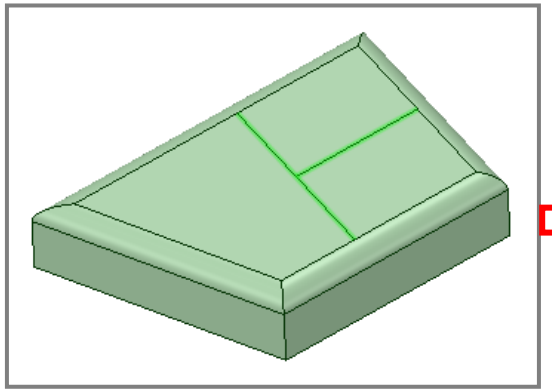
Face to be split

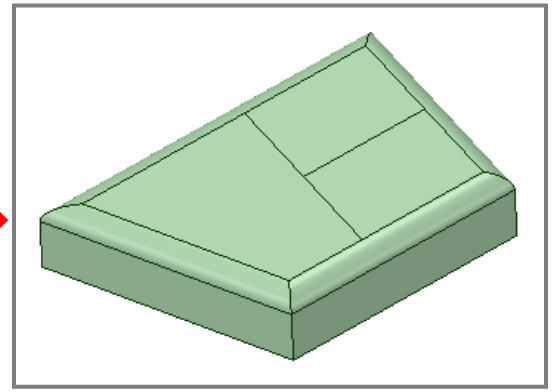
Move mouse over face to preview split.
Click to split face at selected location

Select unwanted edges (red) to remove





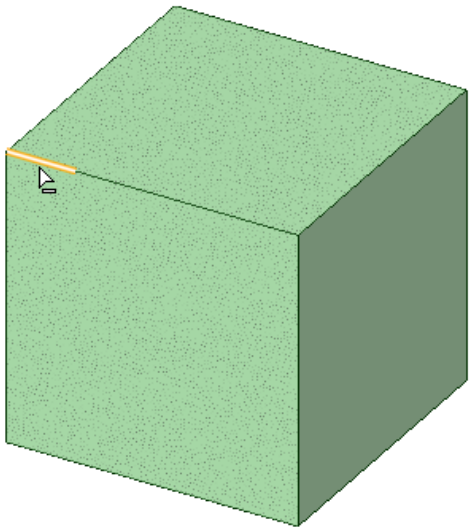


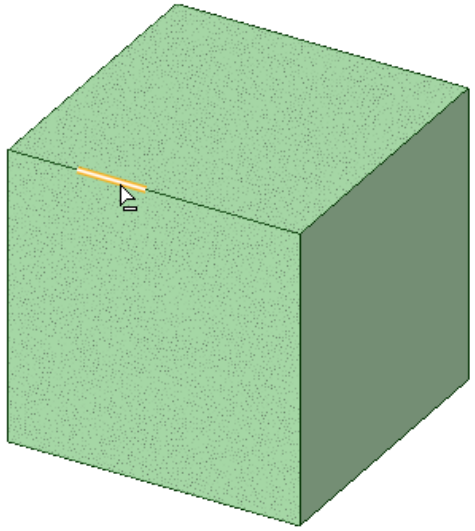


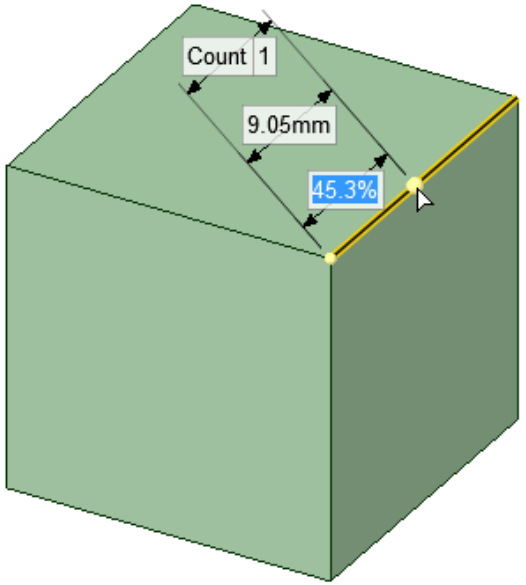
Modeling Tools – Split (2)

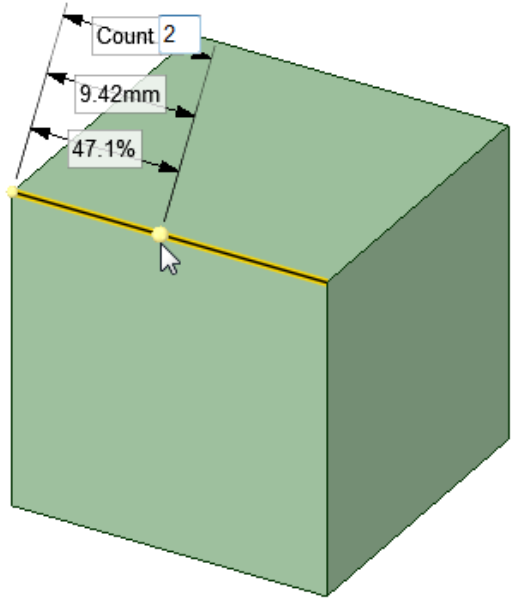
Split Edge Tool

- With the Split Edge tool guide active, the cursor changes to indicate that only Edges are selectable.
- The Split location can be adjusted by:
  - Dragging the yellow ball
  - Entering a Percentage
  - Pressing the Tab key twice and entering a length







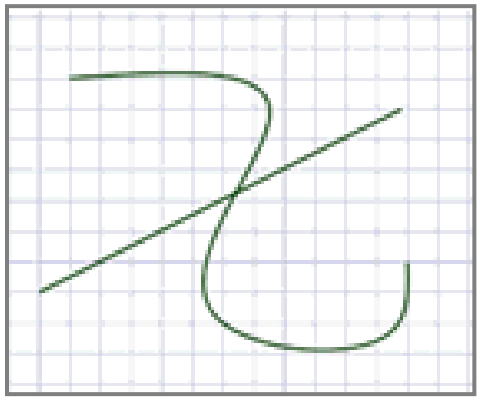


Modeling Tools – Split (3)

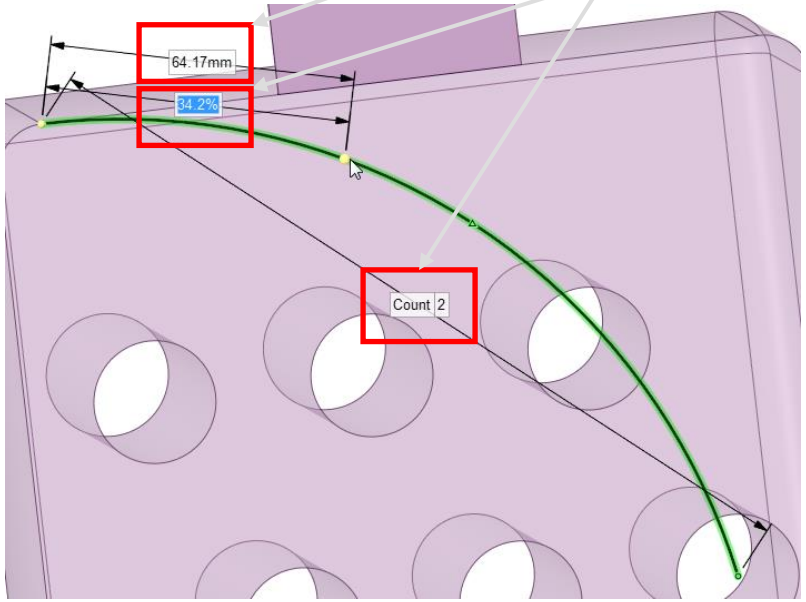
Split Curve

2 ways: split curve in the sketch group or split edge

- Click Split Curve in the Sketch group
- Click the curve you want to split
- Click a curve or point that intersects the curve you want to split. The curve will be split at the intersection



- Click Split Edge and select the curve
- You can either enter a count number to split it to ‘n’ equal curves or enter a distance or a percentage



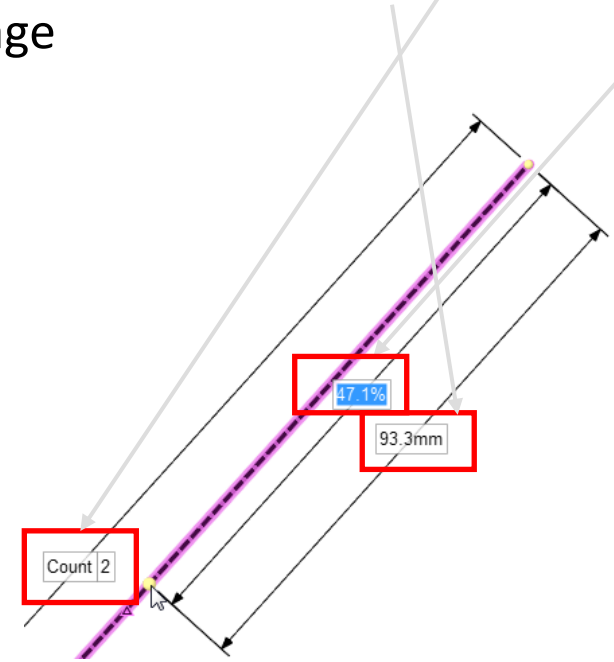
Modeling Tools – Split (4)

Split Beam

2 ways: split beam in the Prepare Tab or split edge

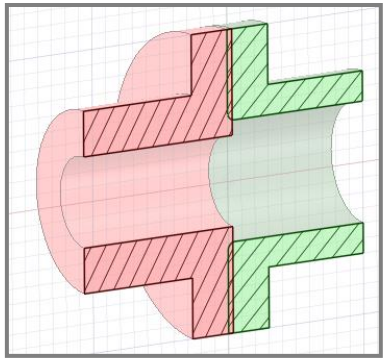
- If beams are currently displayed as Wire Beams, change them to Solid Beams
- Click Split in the Beams group on the Prepare tab
- Any connections that can be Split are highlighted
- Use the tool guides to Split connections:
  - Complete splits all of the highlighted connections
  - Select Problem splits only those connections that you select
    - After making your selections, simply exit the tool, do not click Complete
  - Exclude Problem removes connections that you select
    - After making your selections, click Complete to split the remaining connections

- Click Split Edge and select the Beam
- You can either enter a count number to split it to ‘n’ equal curves or enter a distance or a percentage



Modeling Tools – Working in Section Mode (1)

- Section mode helps to edit solids and surfaces by working with their edges in a cross section
- Useful for complex models that requires repair/cleanup of internal details



Problems with geometry:
Overlap
Small feature (fillet)
Inconsistent inner hole diameters

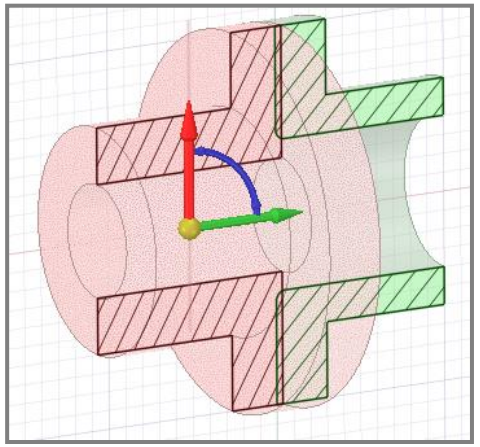
1. Remove overlap using Move tool

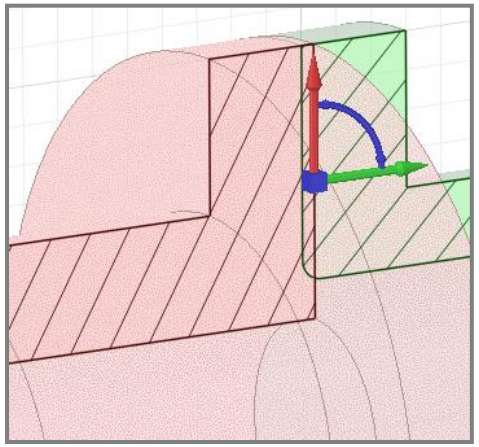
Enable “Select Component” tool guide and select any edge to select the solid body

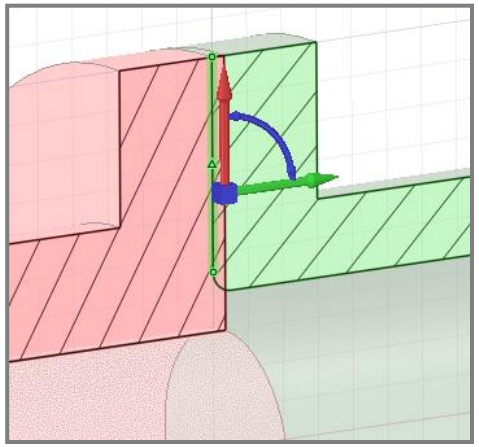
Select “Anchor“ tool guide and position move handle on outermost edge

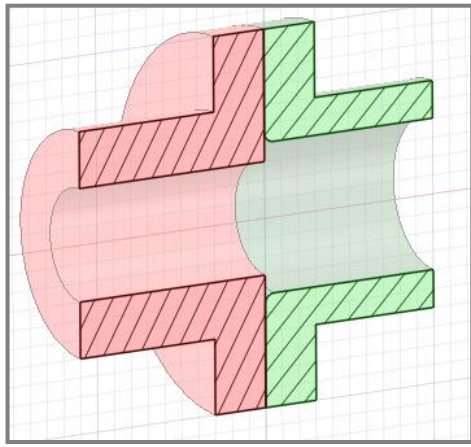
Select “Up To” Tool guide and select green highlighted edge

Body is now touching adjacent body (no overlap)



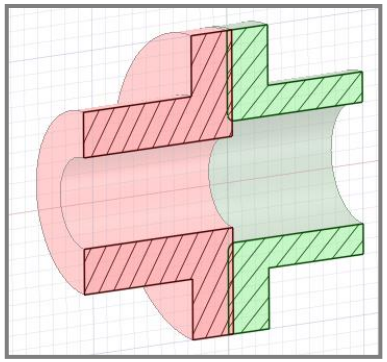






Modeling Tools – Working in Section Mode (2)

- Section mode helps to edit solids and surfaces by working with their edges in a cross section
- Useful for complex models that requires repair/cleanup of internal details



Problems with geometry:

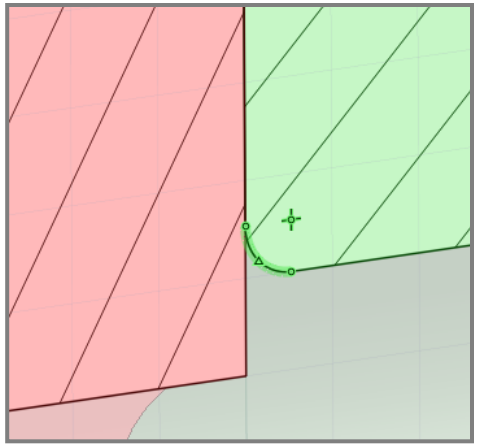
Overlap

Small feature (fillet)

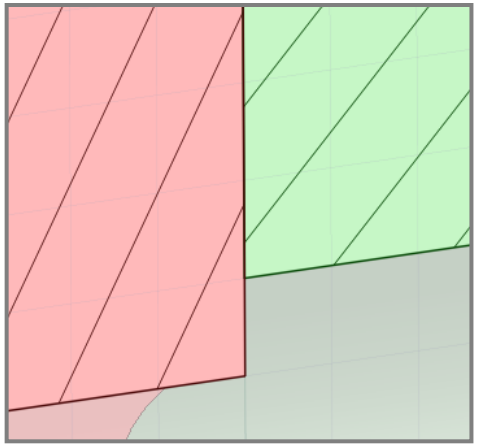
Inconsistent inner hole diameters

2. Remove small feature (fillet) using Fill tool

Select edge representing fillet

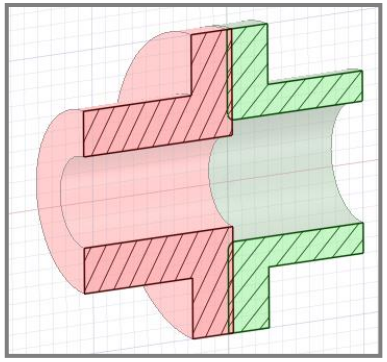


Fillet is now removed



Modeling Tools – Working in Section Mode (3)

- Section mode helps to edit solids and surfaces by working with their edges in a cross section
- Useful for complex models that requires repair/cleanup of internal details



Problems with geometry:

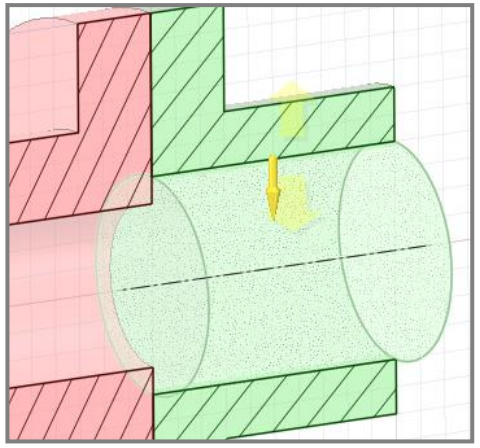
Overlap

Small feature (fillet)

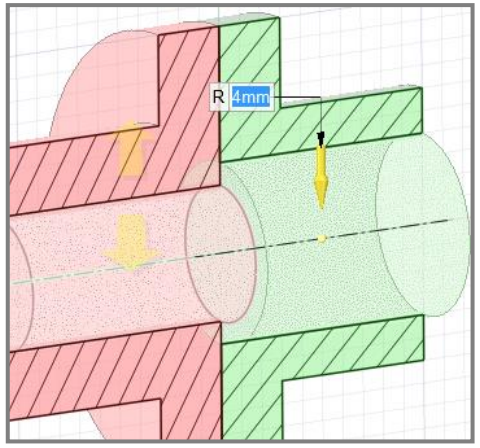
Inconsistent inner hole diameters

3. Make inner holes diameter consistent using Pull tool

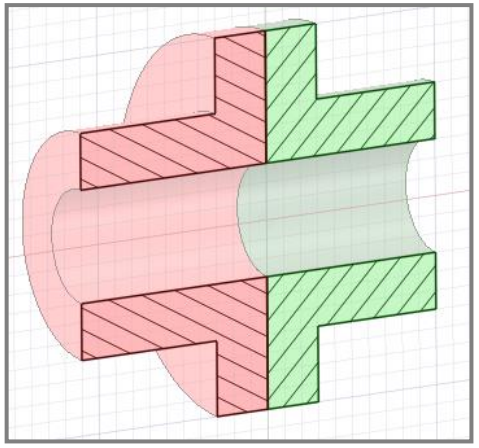
Select edge representing hole



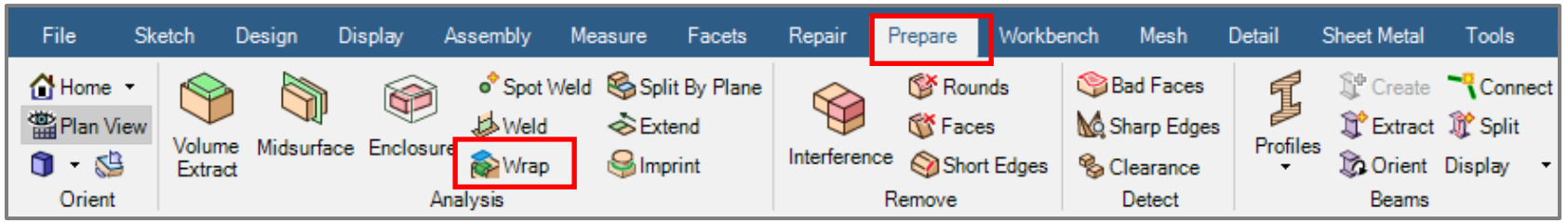
Using “Up To” tool guide, select inner edge of adjacent body



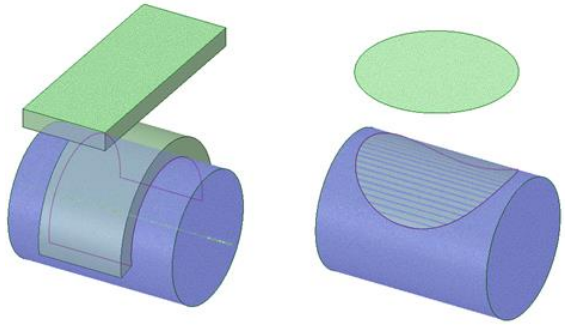
Final geometry



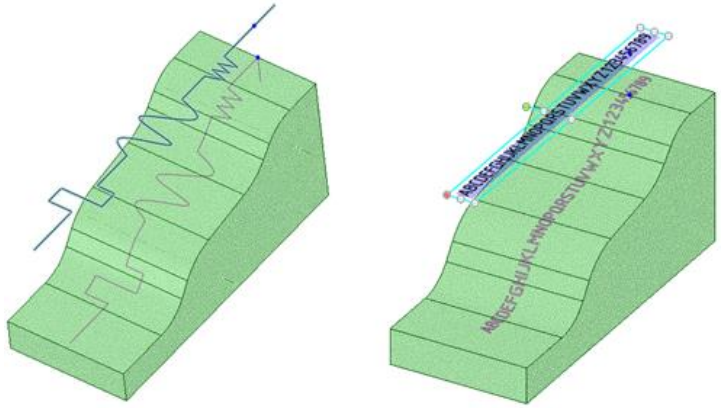
Wrapping geometry



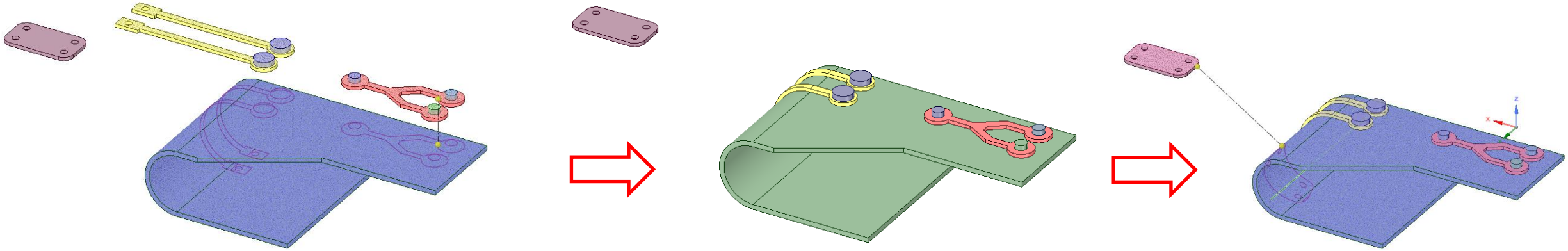
- This tool is used to wrap geometry around a target model



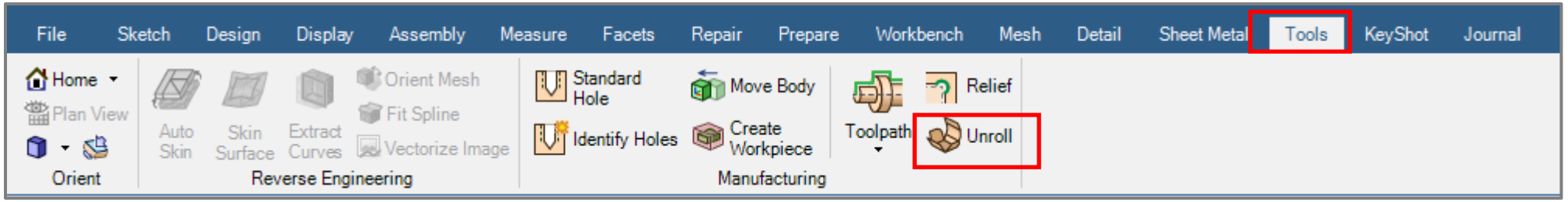
Solid block and surface



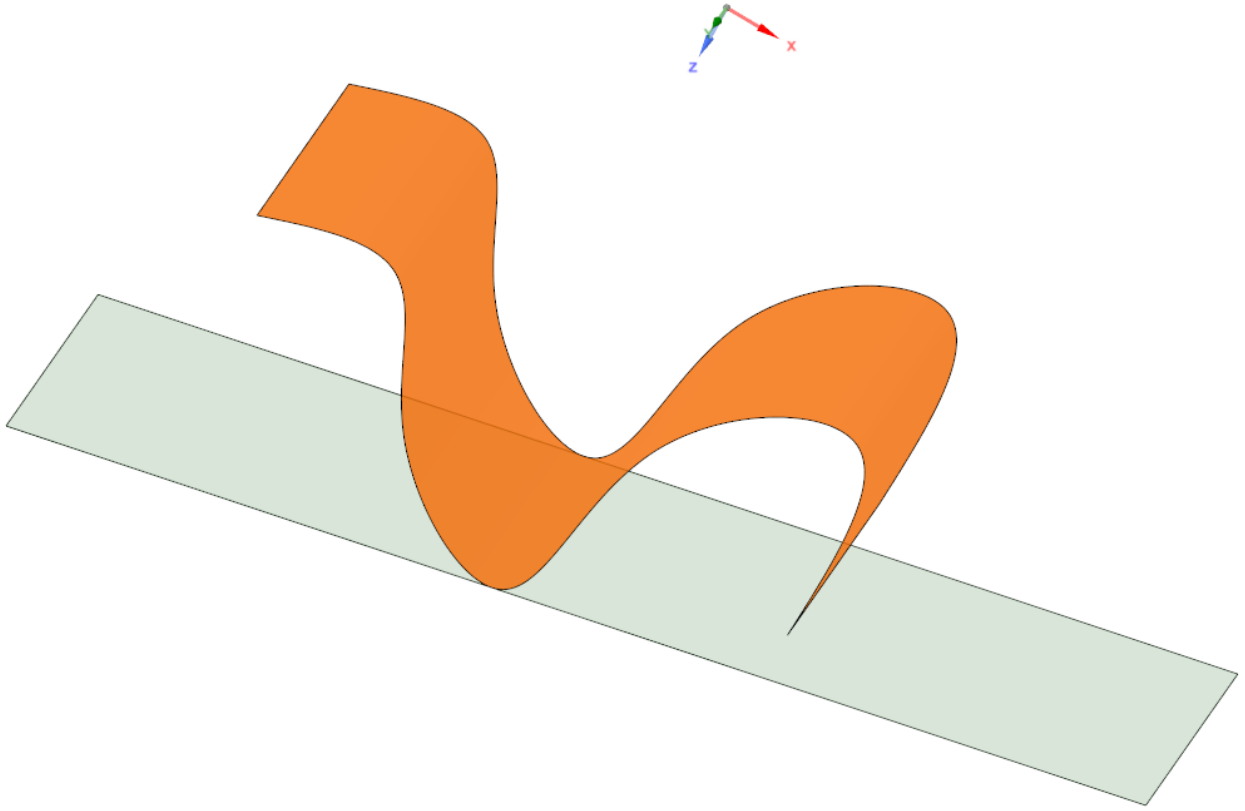
Sketched curves and notes



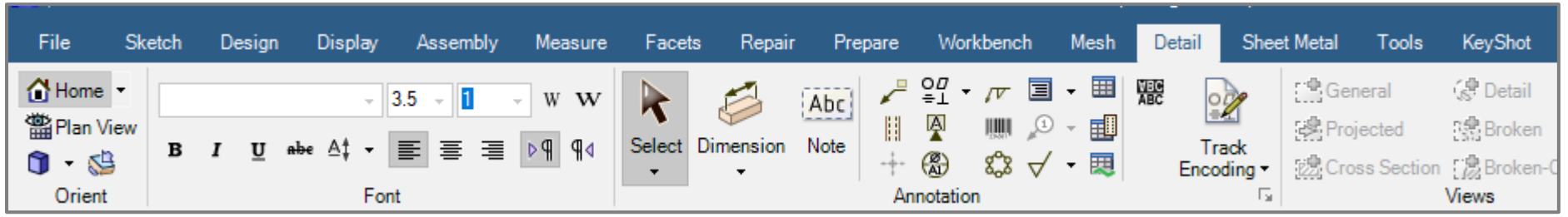
Unrolling surfaces



- It works on surface bodies with planar or curved faces

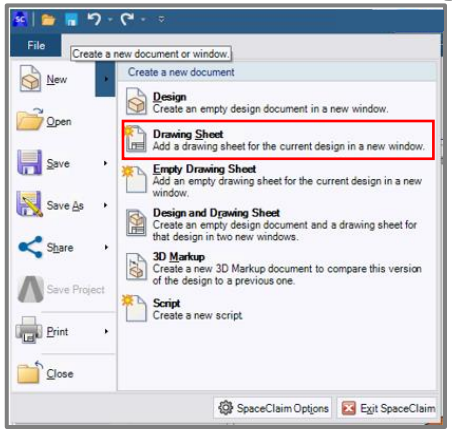


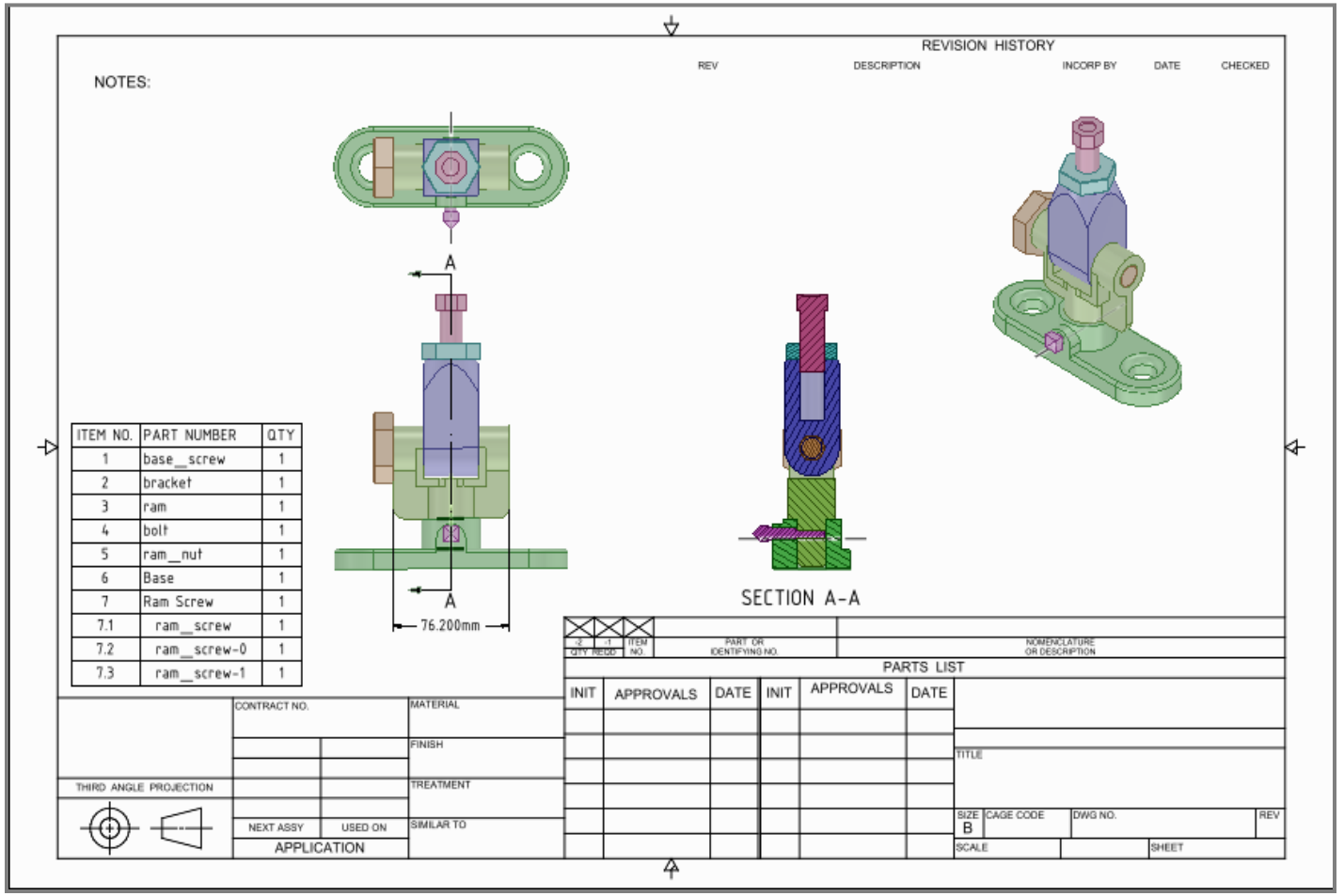
Detailing (1)



Detail tab

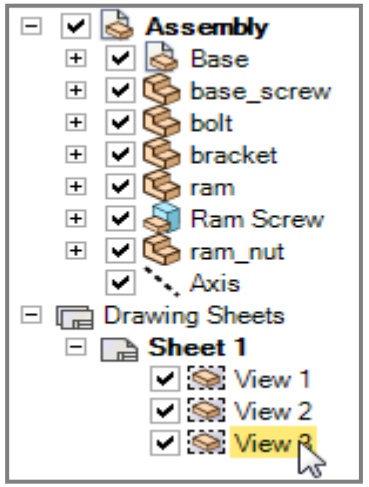
- Create drawing sheets for the current model. Views of the design are created automatically which can then be edited or moved on the sheet.
- Drawing sheets are saved within your design.
- To insert a new drawing sheet, go to File > New > Drawing Sheet

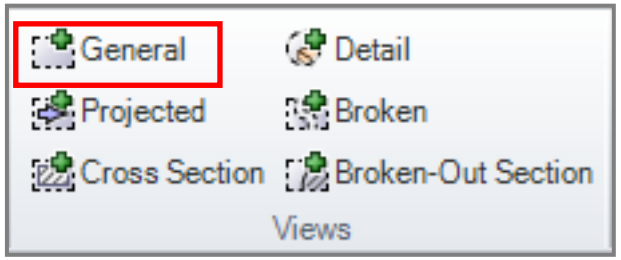


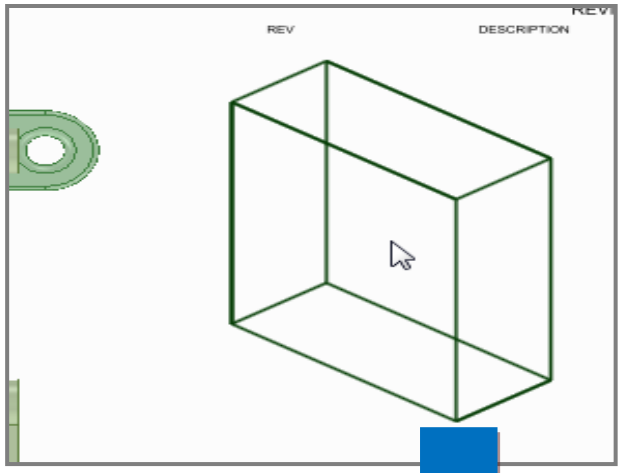


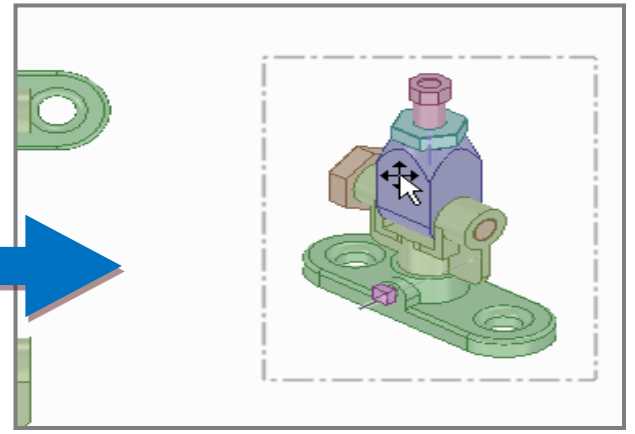
Detailing (2)

- You can add Dimensions, Cross Sections, General Views, and Annotations
- In the Structure tree, expand ‘Drawing Sheets’ and then ‘Sheet 1’ to show the Views on the sheet
- To insert a new General view
  - Click on ‘General’ button in the Views Ribbon group.
  - Place the preview box in the empty area on the sheet.
  - Note: Select a View (inside the dashed grey outline) and drag to move it on the Drawing Sheet.



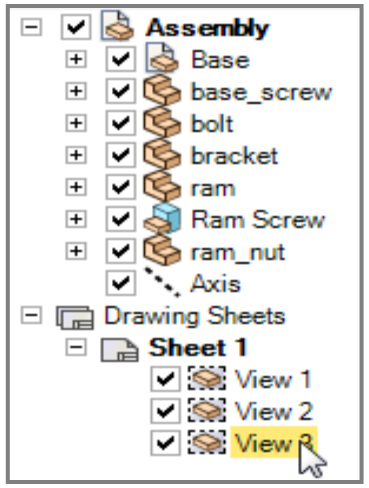


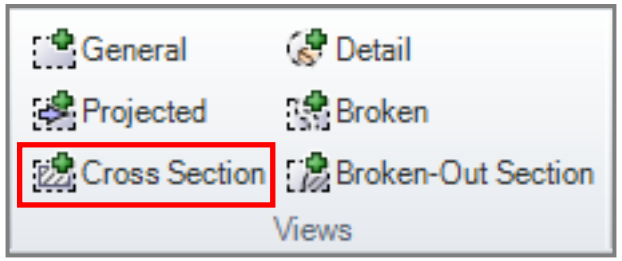


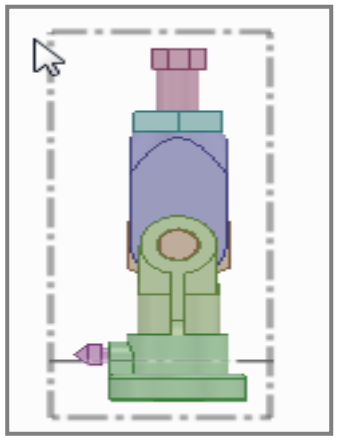


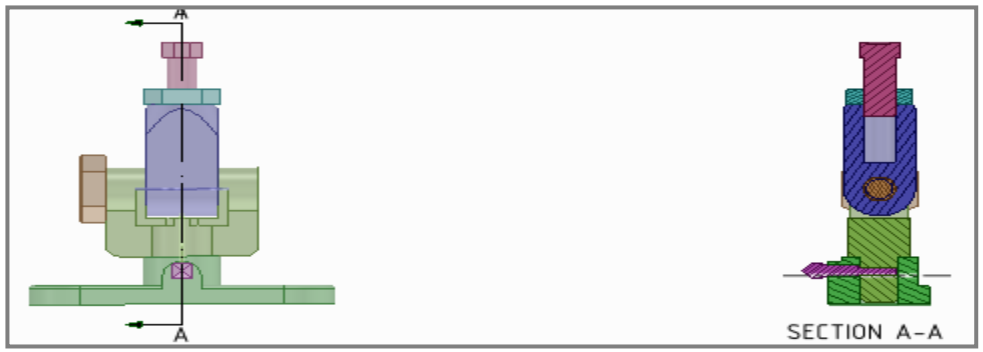
Detailing (3)

- To view the cross-section
  - Select the ‘Cross Section’ button.
  - Select View 3 (the Side view).
  - Drag the mouse over “view 1” (the Front view). As you move the mouse, View 3 should appear as a cross section.
  - Click to place the section plane.



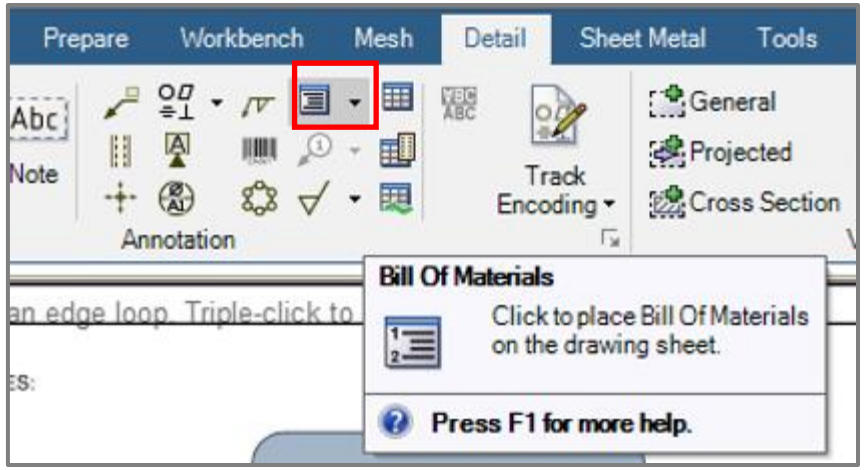


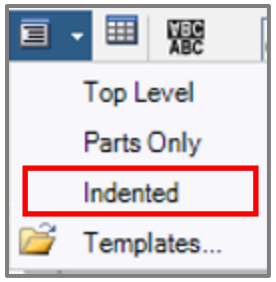


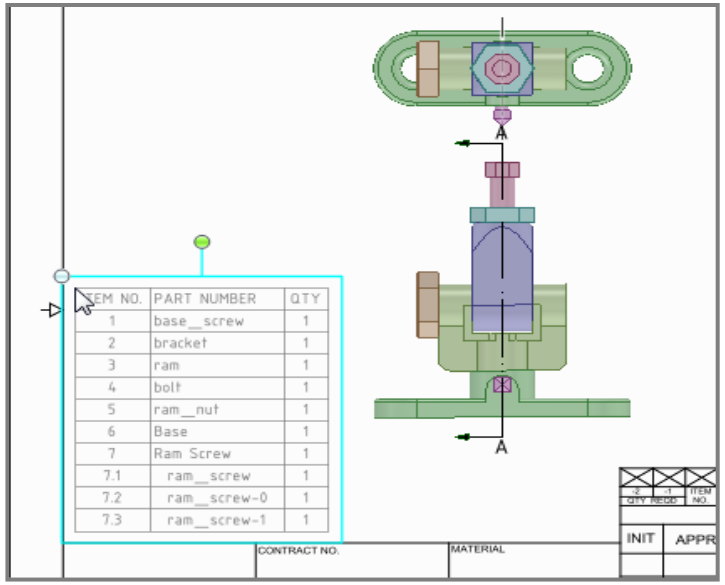


Detailing (4)

- Insert Bill of Materials
  - Select the Bill of Materials button in the Annotation Ribbon group.
  - Select Indented from the dropdown.
  - Place the BOM in the corner of the Drawing Sheet.

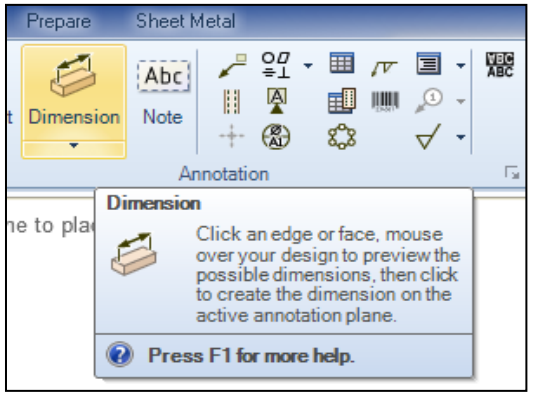


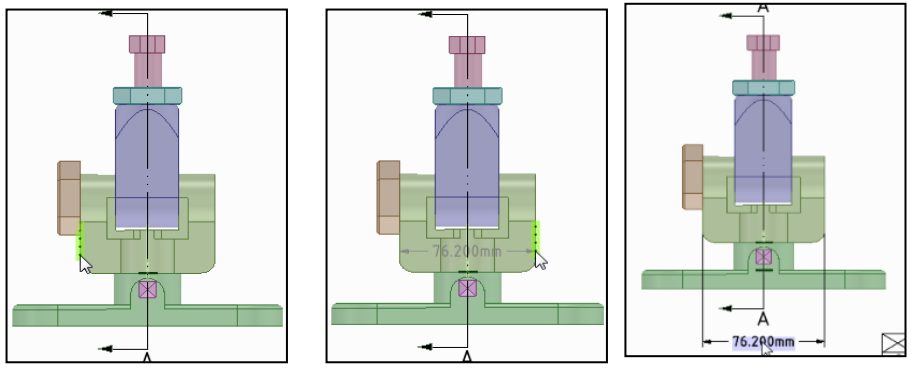




Detailing (5)

- Add Annotation
  - You can add Notes, Dimensions, Geometric Tolerances, Surface Finish, Datum Symbols, Center Marks, Center Lines, and Threads.
  - Use the ‘Select’ tool to select the highlighted object
  - Use the ‘Dimension’ tool to create a measured dimension.
    - Select one side of the ‘bracket,’ then the other.
    - Note: Placing a dimension with one reference will display the length of the edge, and with two references will display the distance or angle between them.
  - Use the Note tool to enter text onto the plane

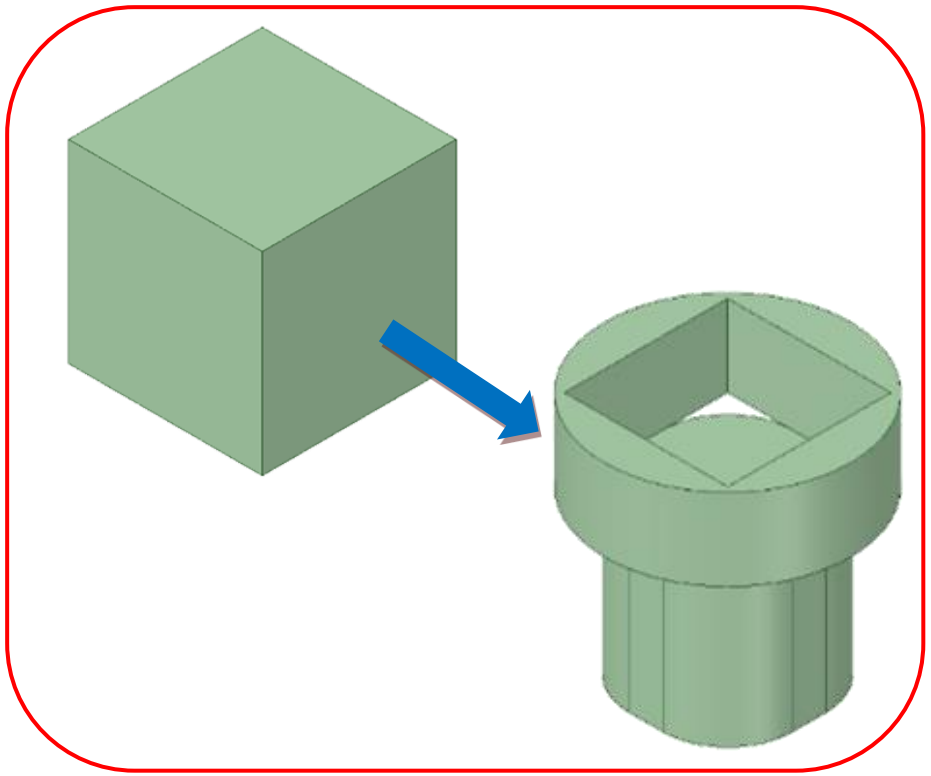


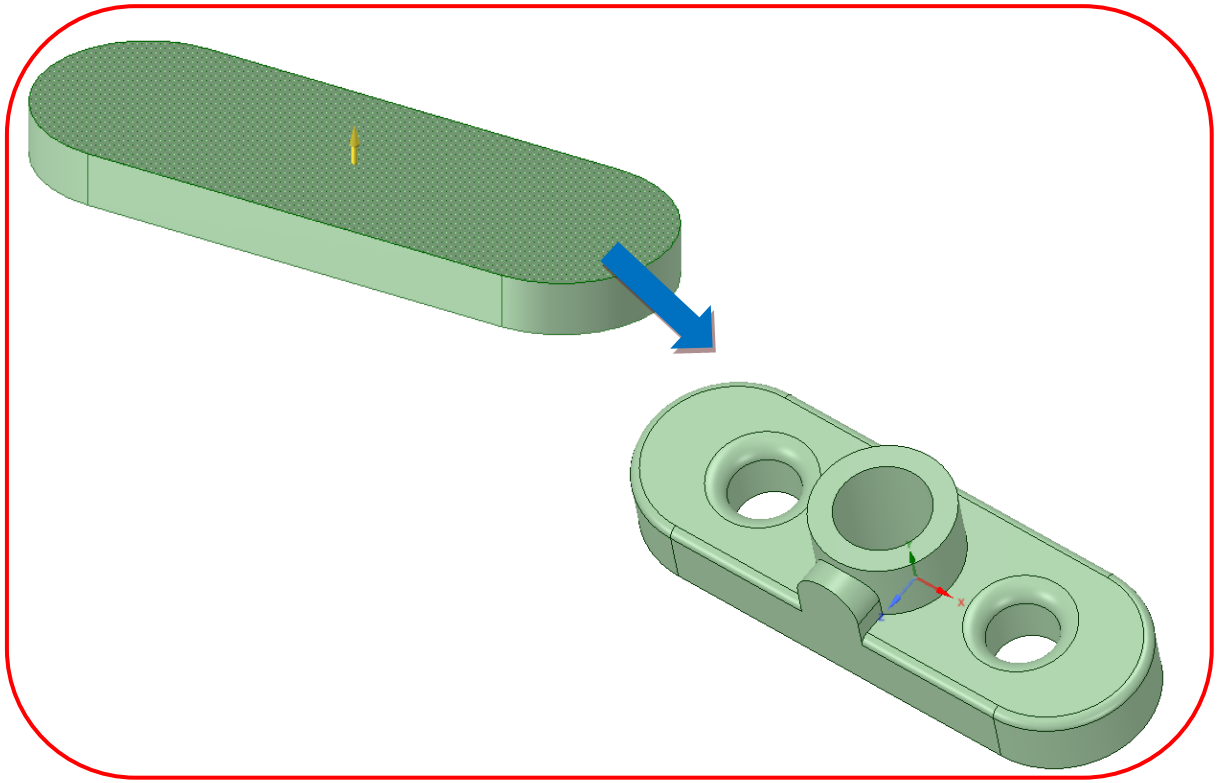


Summary

- In the Sketch tab we can create, change and constraint a sketch from scratch
- Constraints can be dimensional or geometrical
- All the Pull, Move and Combine options have been shown
- Detail tab will allow you to create a drawing sheet with adding dimensions, annotations and notes

Workshop 2.1 Creating Geometry





End of presentation