NX Extrude Tutorial

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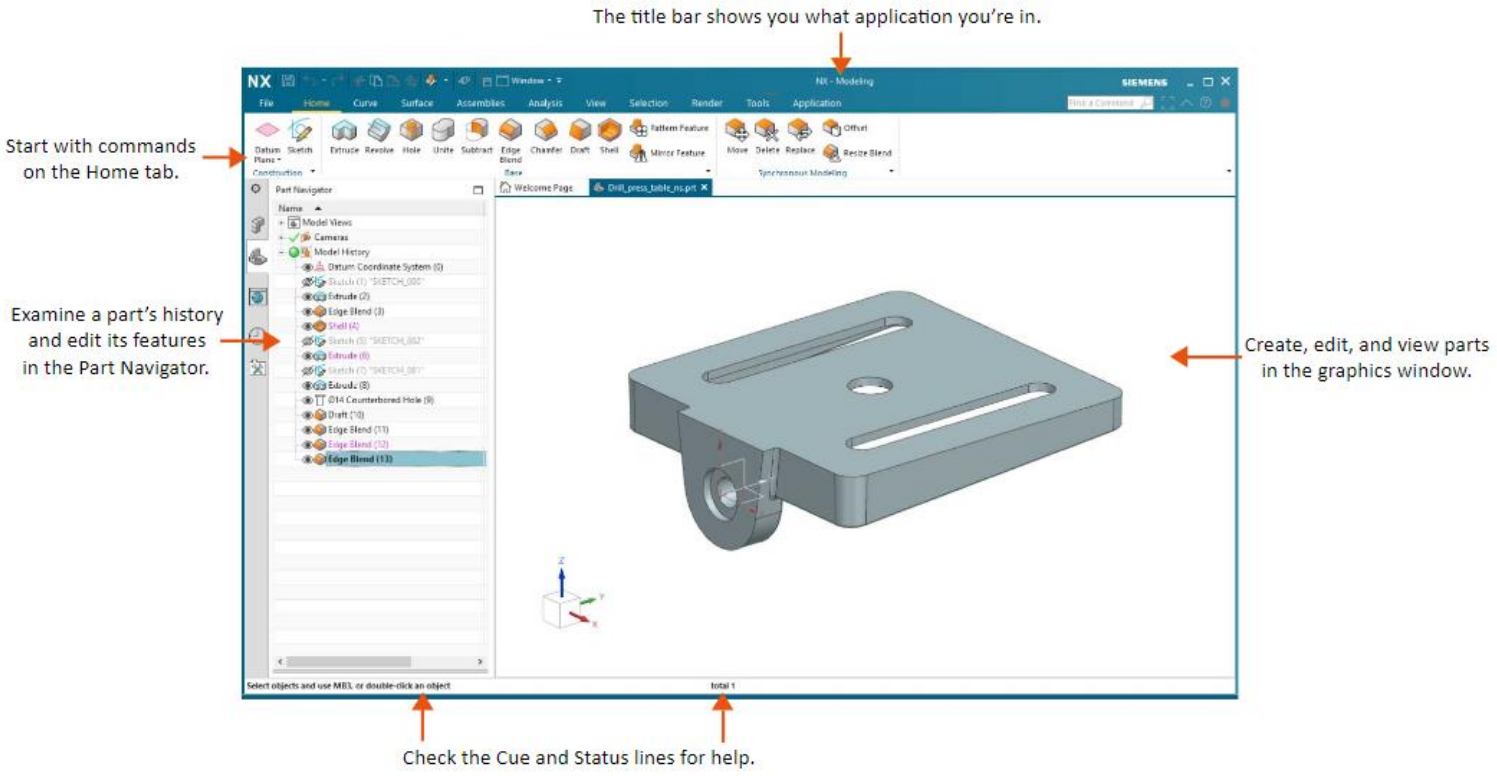
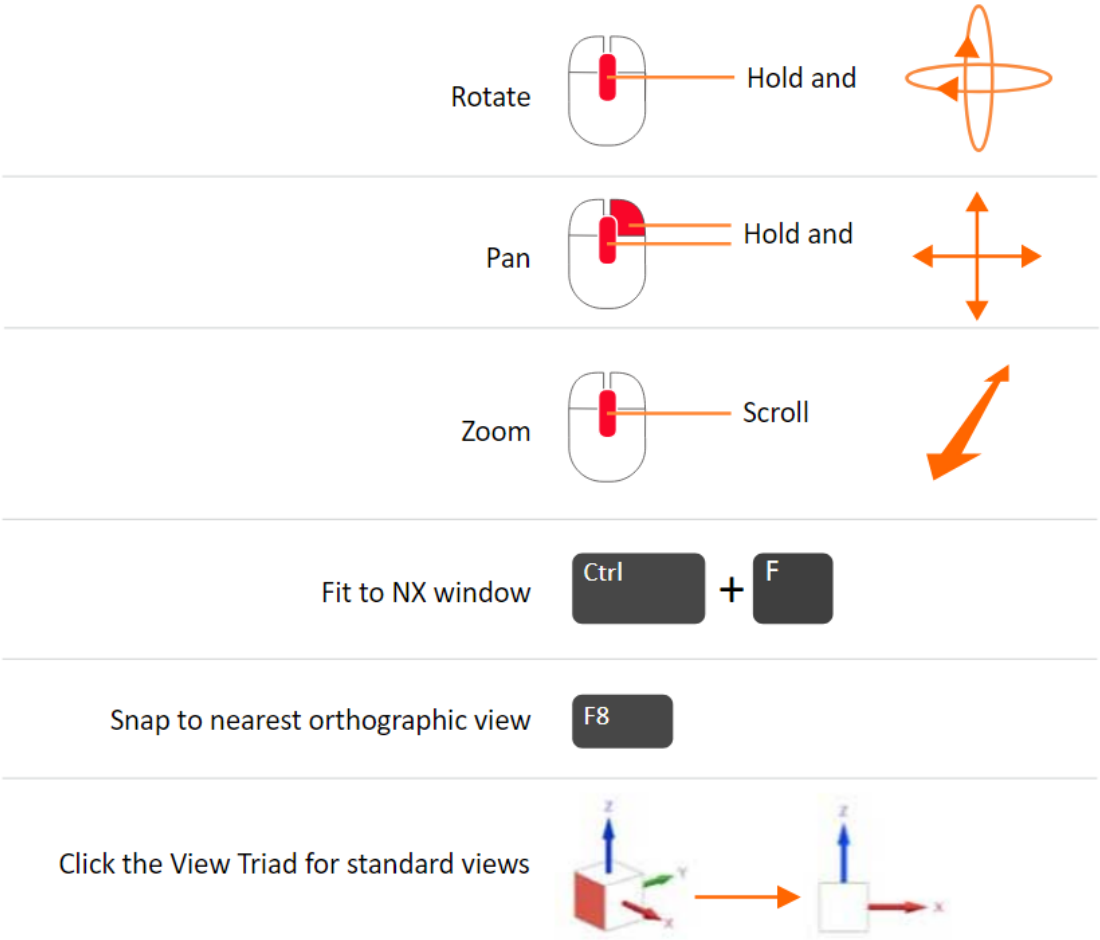
Screen Layout



Mouse and Keyboard Controls



Hot Keys

Isometric View: END

Snap to Nearest Orthographic View: F8

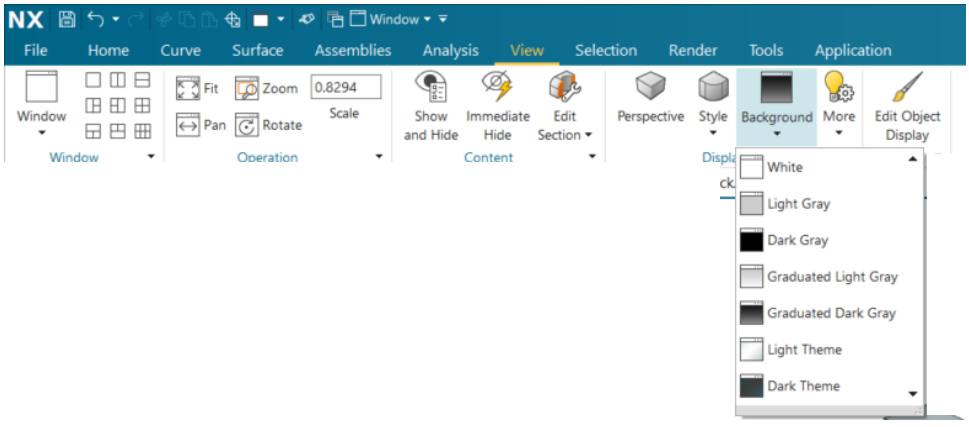
Snap to Selected Datum or Surface: Left Click Datum or Surface to View > Select F8

Fit to Window: Control (CTRL) + F

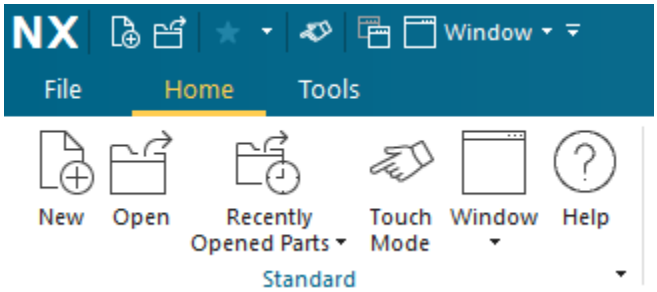
Refresh: F5

Background Colors

Select View Tab > Select Background Setup

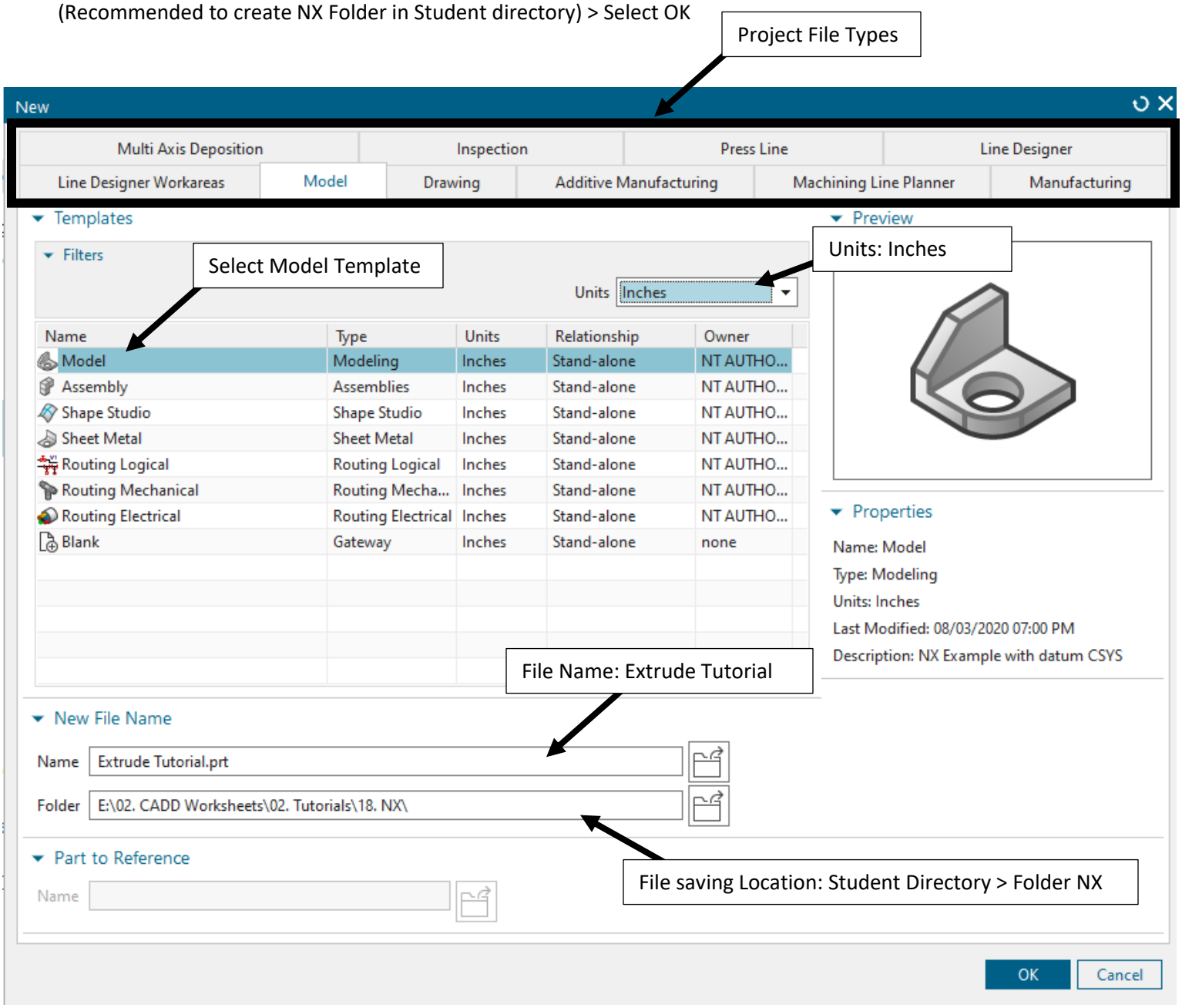


1. Select New from Home Tab

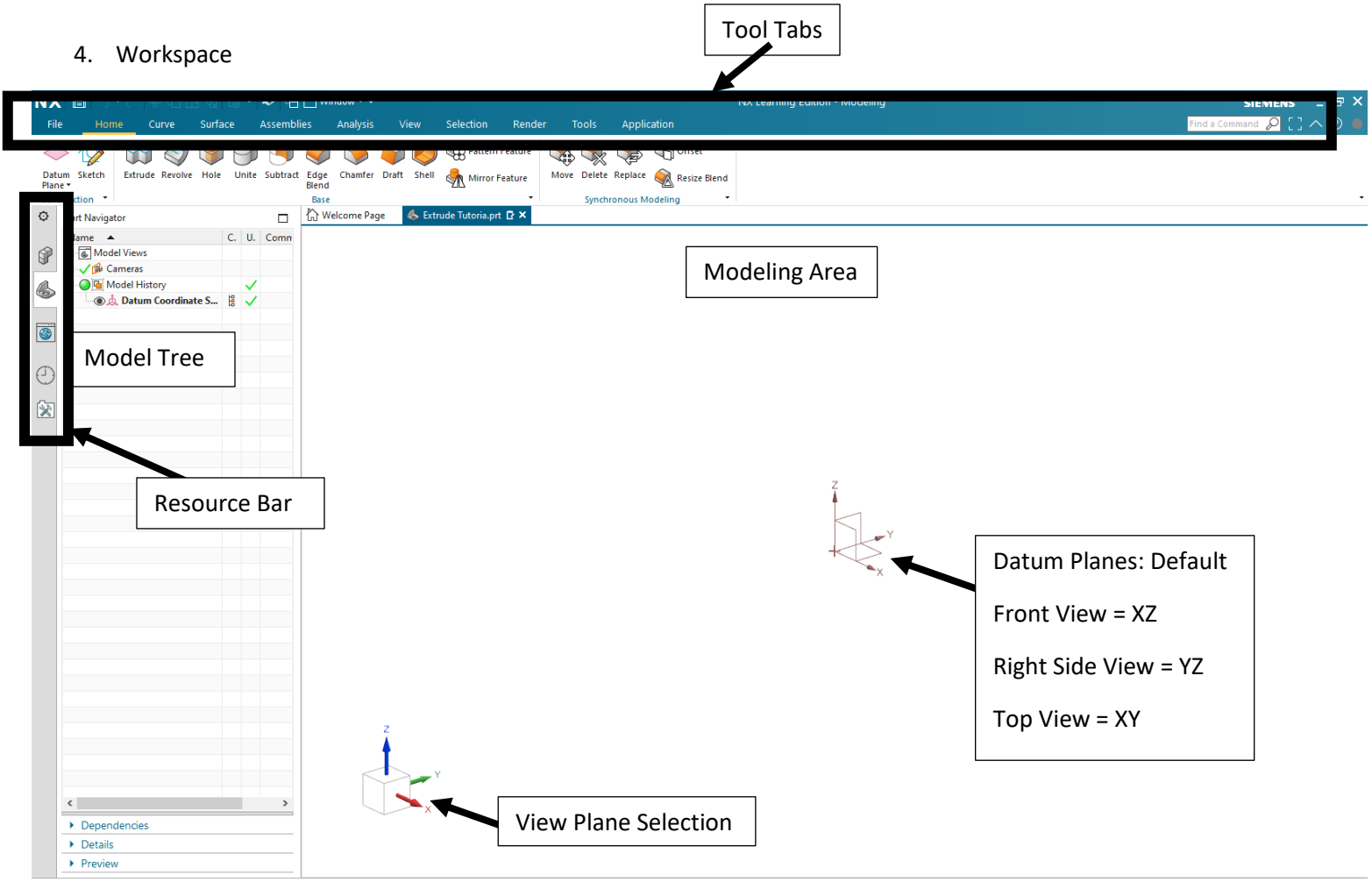


2. New Pop-Up Menu will appear. The pop up menu is setup to choose various types of project files (i.e. solid model, layout, assembly, machining/manufacturing, etc.)

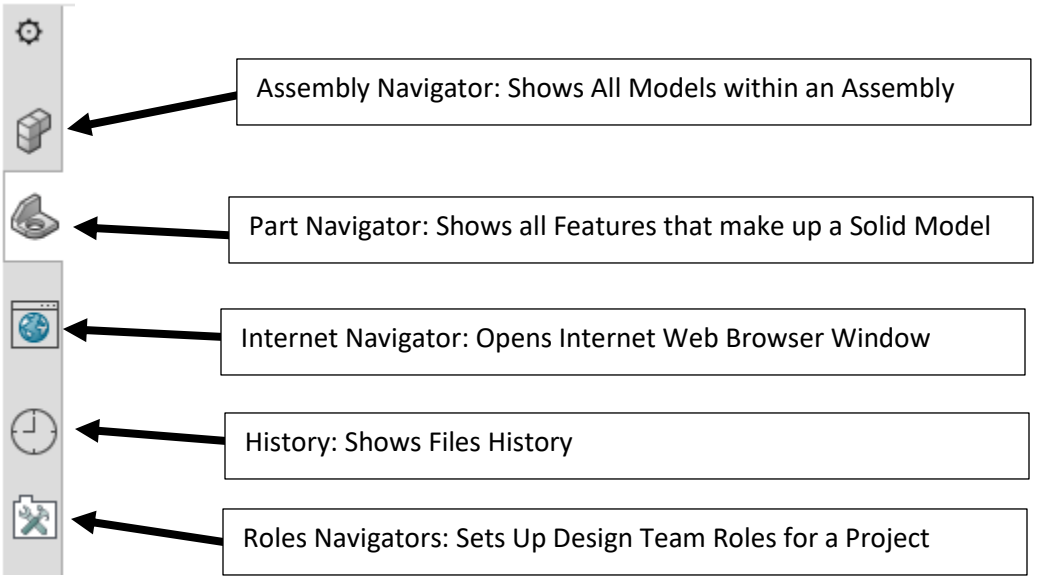
3. Select Model Tab > Select Model Tab > Select Model Template (This setup the work environment) > Change Units to Inches > Change File Name to Extrude Tutorial.prt > Select Folder > Navigate to Student Directory to Save work (Recommended to create NX Folder in Student directory) > Select OK



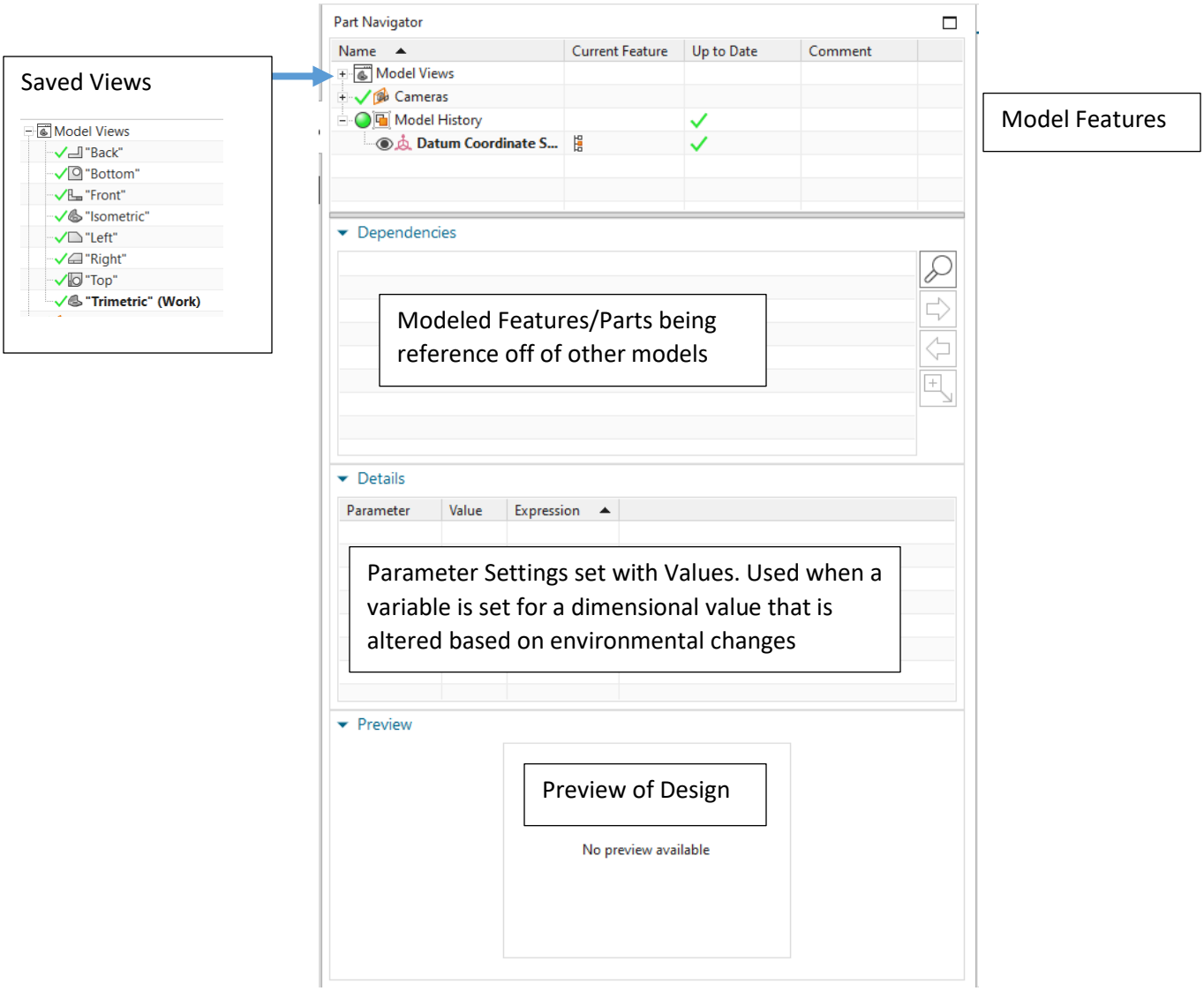
4. Workspace



a. Resource Bar: Each Option will Change the Model Tree to Show Selected Resource



b. Model Tree: Part Navigator



5. Model: Single Extrude

Two Options for creating an extruded feature

Option 1 External Sketch: This is where the user creates an independent sketch separate from the extrude. Typically used when the sketch maybe used multiple times with mutliple 3D features

Steps: Select Home Tab > Select Sketch > Select Datum Plane or Surface to Draw On > Draw Closed Profile Sketch and Set Dimensional Values > Accept Sketch > Select Extrude Icon > Set Extrude Dimensions > Except Extrude Feature

Option 2 Internal Sketch: This will embed the sketch as part of the extrude feature. This is used to help to minmize the length of the model tree and keep associated 2D and 3D features that are dependent on each other grouped together

Steps: Select Extrude Icon > Select Datum or surface to Sketch On > Draw Closed Profile Sketch > and Set Dimensional Values > Accept Sketch > Set Extrude Demensions > Except Extrude

a. L-Block feature (Internal Sketch)

i. Select Extrude Icon  > Pop-Up Menu will Appear

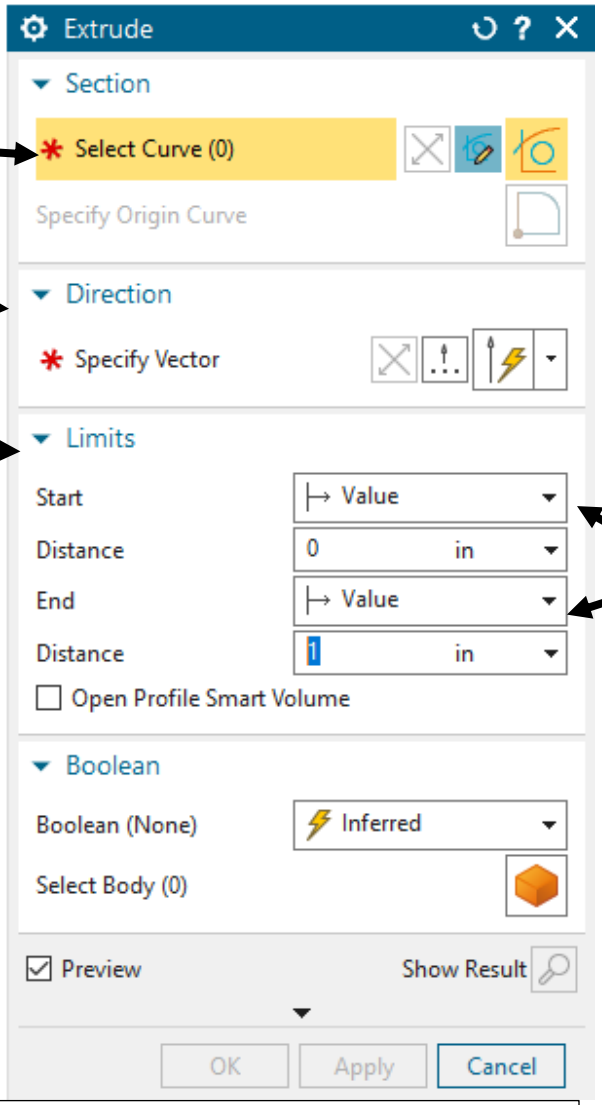
Sketch Menu: Allows the user to Select a pre-created sketch or embed a sketch by selecting a Datum or Surface to Draw on

Direction: Allows User to Set Direction of 3D Feature based on its start point (Default is Normal to Selected Datum>Face)

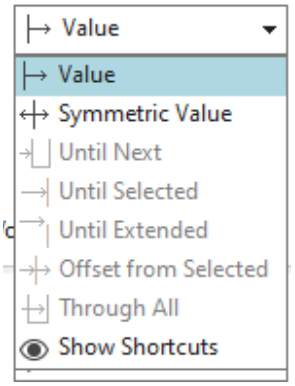
Limits: Sets Dimensional Values of the 3D Shape

Start: Sets Offset Distance from Sketch and Selected Surface (See Below)

End: Sets Total Extrude Distance of Sketch from Selected Datum or Surface (NOTE: IF Start Value is set to anything else, but 0 the extrude amount of solid material that will appear will be Start Distance – End Distance (See Below)



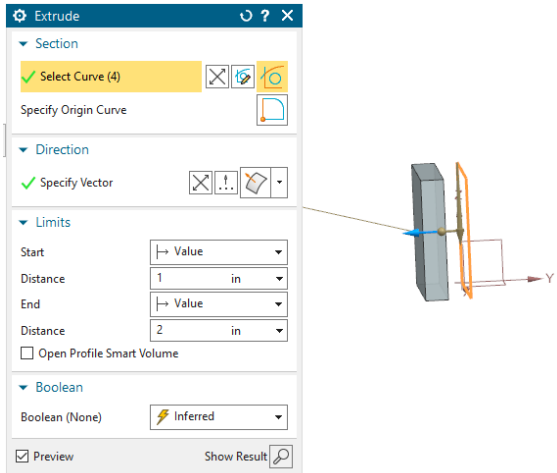
Extrude Options



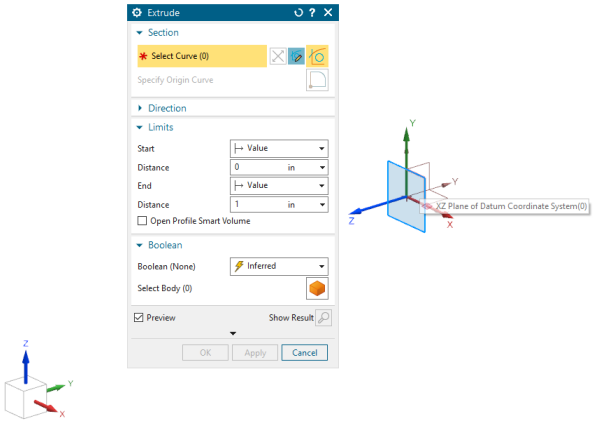
Extrude Options: Extrude with Start Distance

Start: Offsets Sketch from Selected Datum or Surface

- Start Value = 1”
- End Value = 2”
- Total Solid Material =1” (1” Sketch Offset from Start – 2” End Distance)



b. Select XZ Datum (Front View) > Work Space will Rotate Normal (Perpendicular) to Viewer

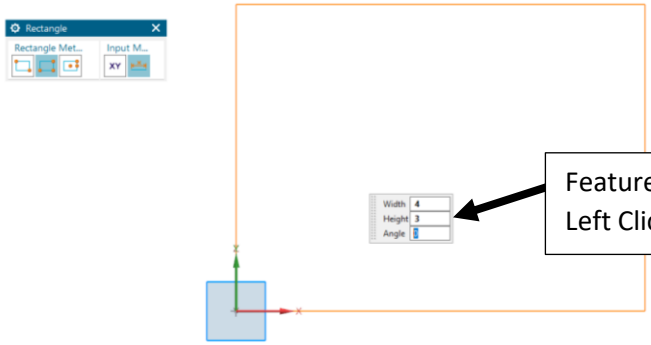


NOTE: If view is rotated out of orthographic view (2D) > Press F8 or Selected face on View Rotator (bottom left corner of workspace) to rotate back to 2D

NOTE: Default Profile 

is active by default. This a poly line tool, meaning user will need to create a closed profile before the tool will stop drawing lines. Press Escape or Select another tool to deactivate

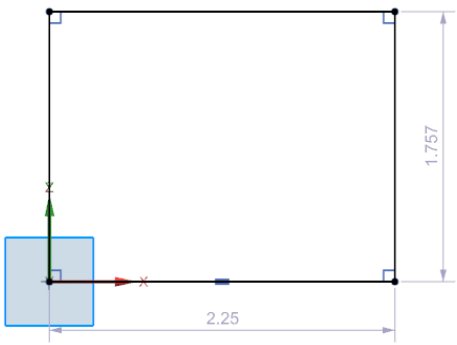
c. Draw the following Rectangle Profile
Dimensions Setting
Option 1: Set Dimension in the Tool Dialogue Box



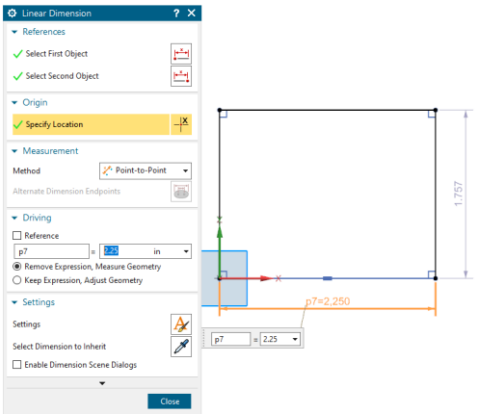
Feature Dialogue Box. Press Tab to switch between Settings > Left Click or Enter to Place

i.e.

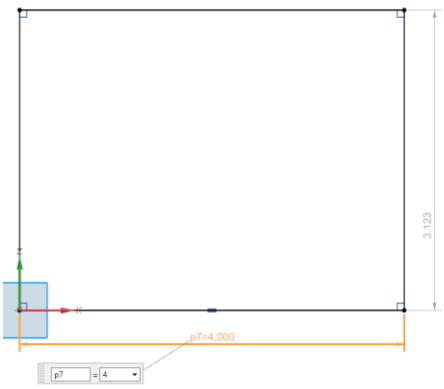
Option 2: Set Rectangle then Change Weak Dimensions



Set Sketch Outline

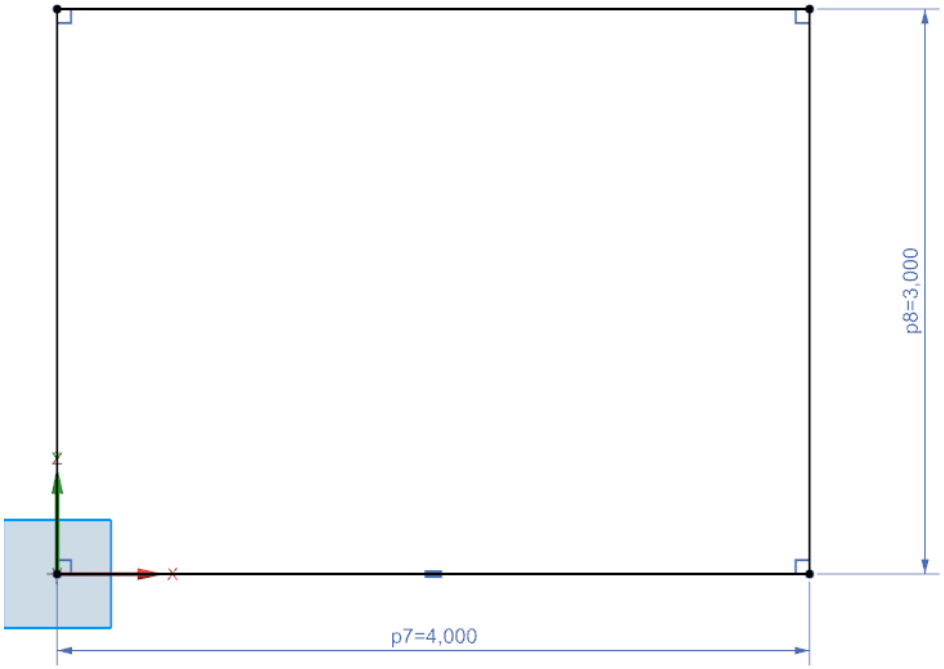


Double Click on Dimensions > Change Value > Press Enter



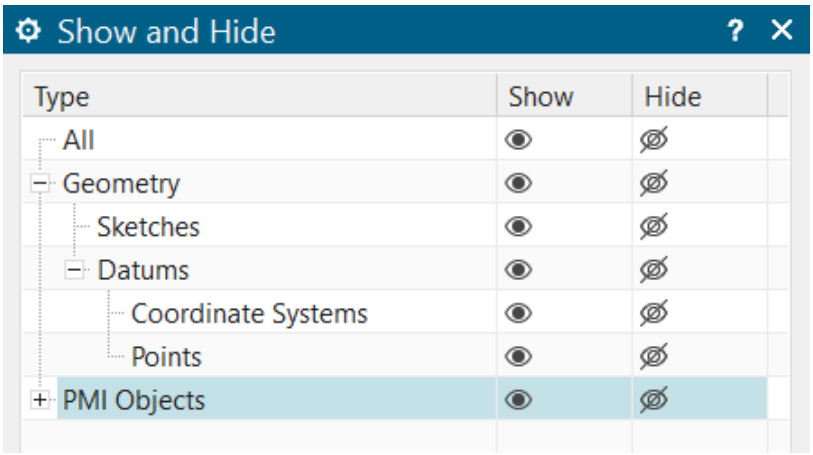
Profile will Re-Size

Final Size



NOTE: Dimensions can be moved by placing cursor on Dimension > Hold Left Mouse Key Down > Drag Cursor to desired Location

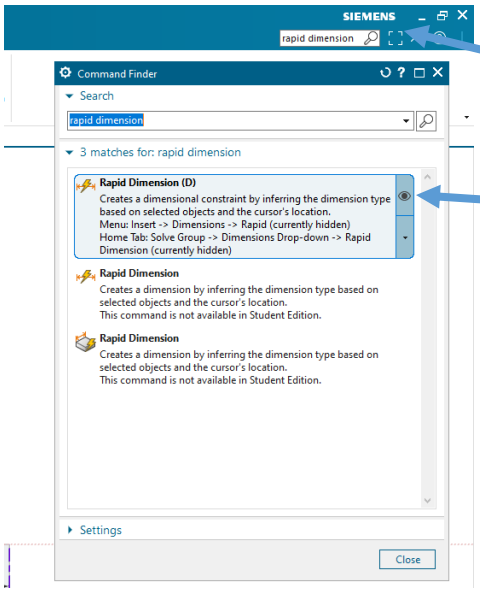
NOTE To Hide Axis and Datum Planes: Press CTRL-W > Show and Hide Pop-Up Menu will Appear > Press the Eye Ball to Hide or Show desired feature



d. Turn On Rapid Dimensioning if not on the Tool Bar

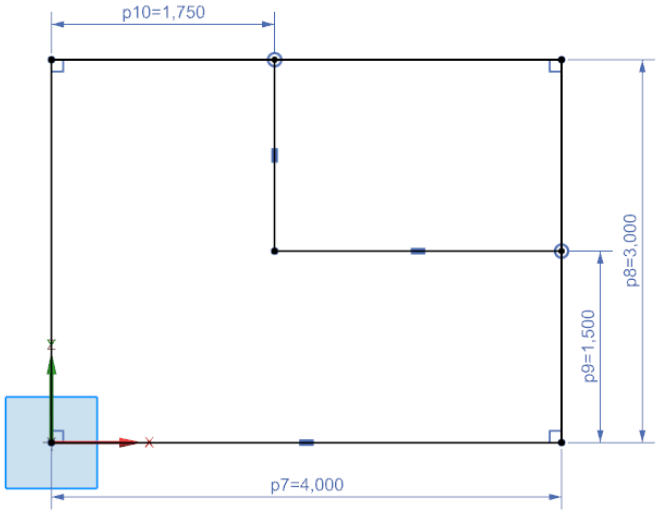
i. In the Search Menu Type “Rapid Dimension” > Press Enter (NOTE: DO NOT Click on the predicted search results) > Select the Eyeball to Add Rapid Dimension Icon to the Toolbar

ii.



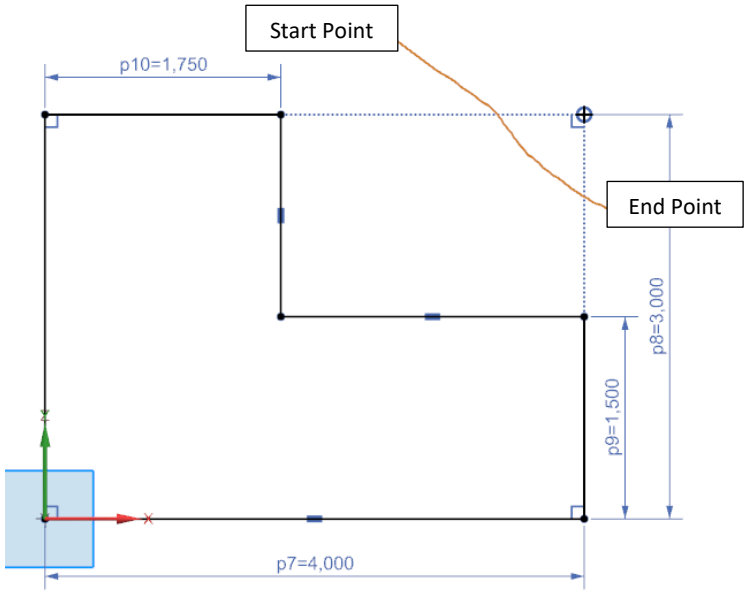
e. Draw the following lines > Set Dimensions as shown using Rapid Dimension Tool 

NOTE: To place Dimensions p9 and p10 Select the two parallel lines the dimension is measuring between



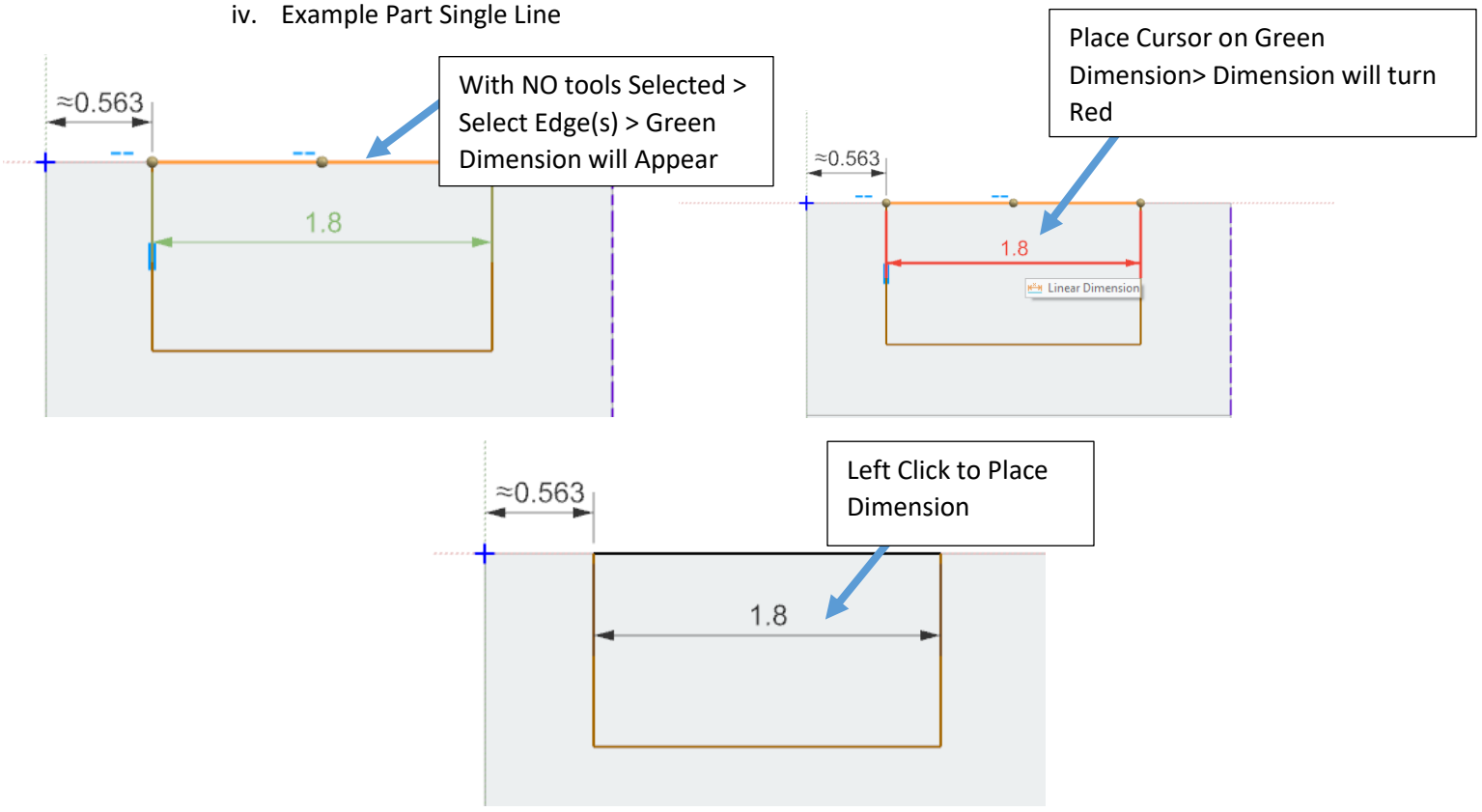
f. Trim: Select Trim Tool
Two Options to use this tool

i. Option 1: Select each Line to trim separately by left clicking on the line
ii. Option 2: Place cursor off the part but close the set of lines to trim > Hold Left Mouse Key Down > Drag Cursor through the lines to be trimmed > Let go of Left Mouse Key when Complete.

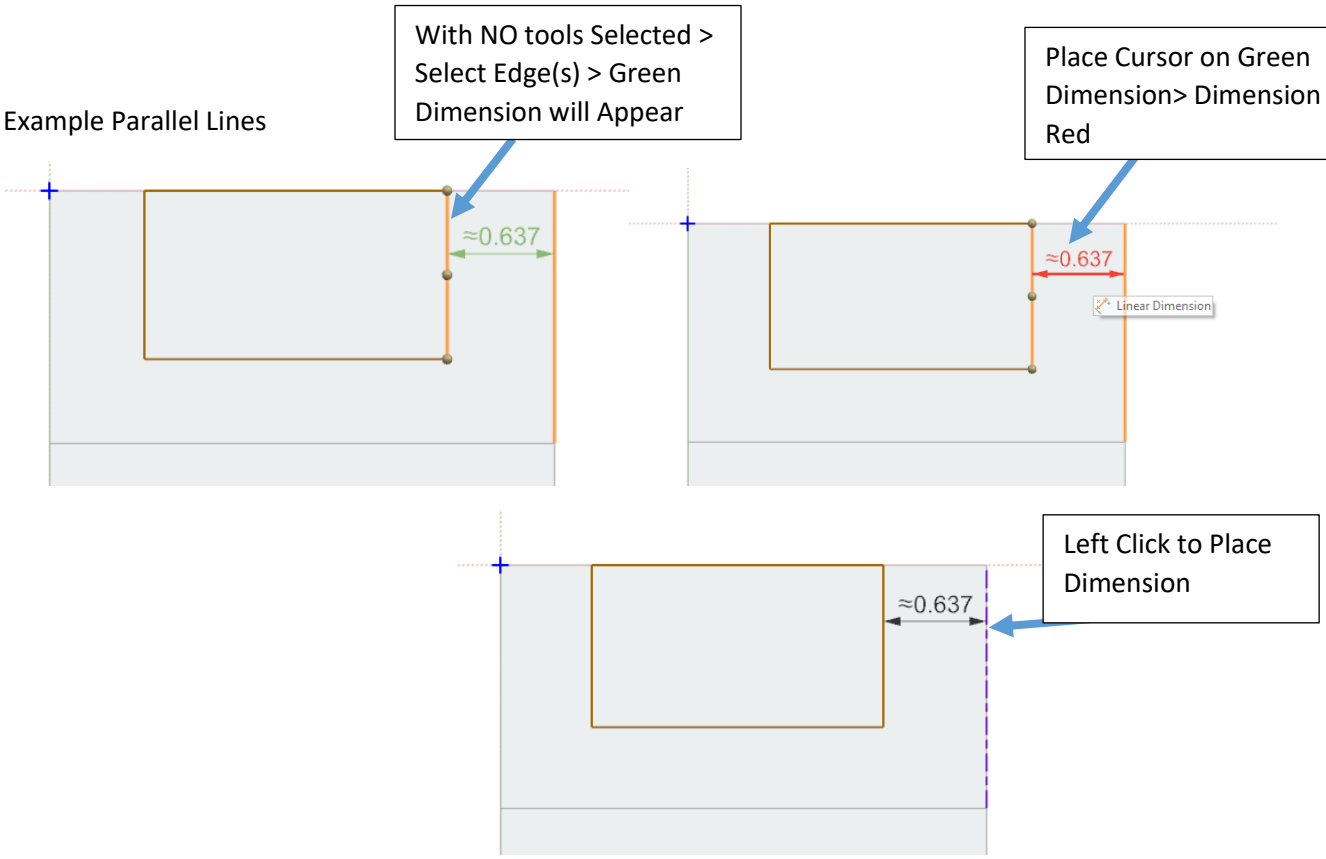


NOTE: there is a second way to set dimensions:

iii. With no tools selected > Click on the Edge(s) to Dimension > Dimension will Appear Green > Left Click on the Dimension Value to Place
iv. Example Part Single Line



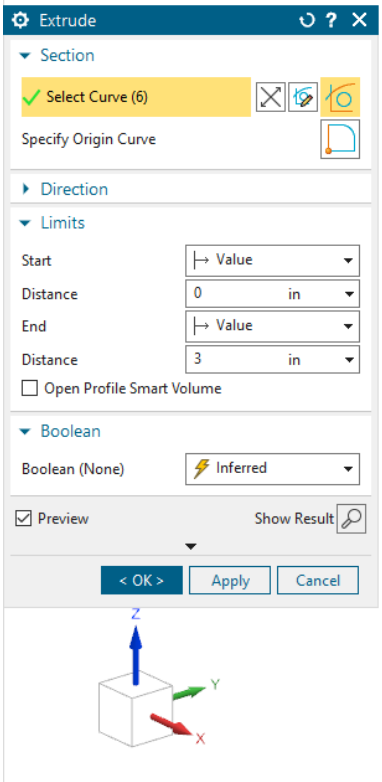
Example Parallel Lines

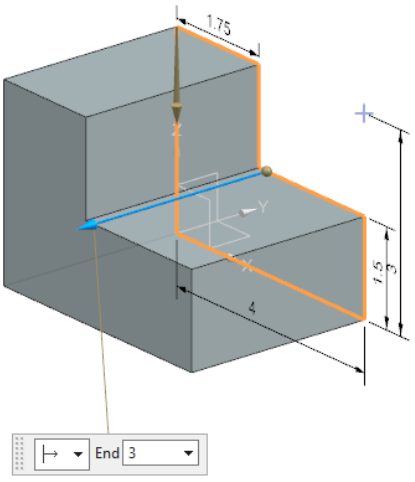


g. Select the Finish Icon on Sketch Toolbar 

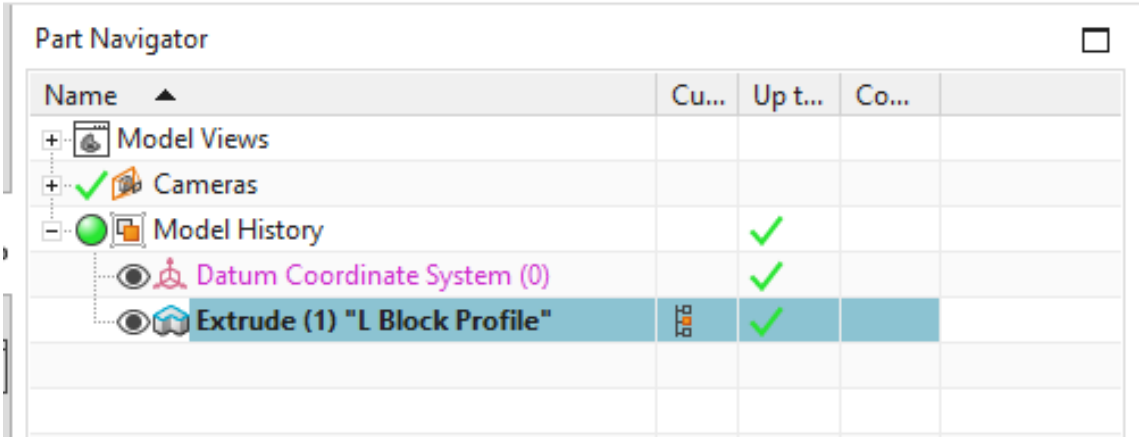
. This will accept the sketch.
NOTE: If Profile is not complete NX will turn the model into a Surface model, showing just the outline of the profile as a thickened line.

h. Change End Value Distance to 3 > Select Ok
NOTICE: The Strong Dimensions Visible on the part > User can double click on the values and change them from the preview





i. Rename Extrude: Select Extrude (1) from Model Tree > Select Rename > Change name to L-Block Profile

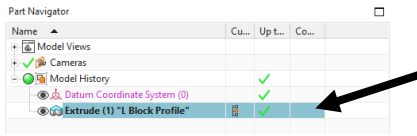


j. Edit Extrude and/or Sketch

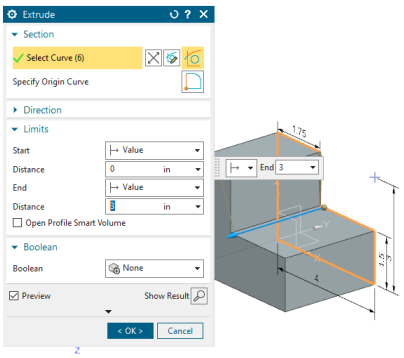
i. Edit Extrude: Double Click on the Extrude Feature in the Model Tree > Extrude Properties Menu will Appear > Adjust Values > Select Ok to Accept

ii. Edit Sketch: Three Options

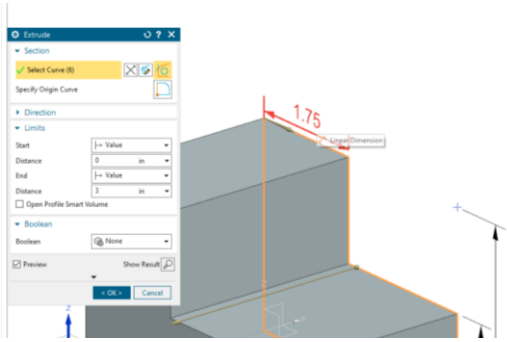
1. Option 1: Double Click on Extrude Feature > Extrude Properties Menu will with Sketch Dimensions > Double Click on Dimension to Change > Press Enter to Accept > Select Ok in Extrude Properties to complete



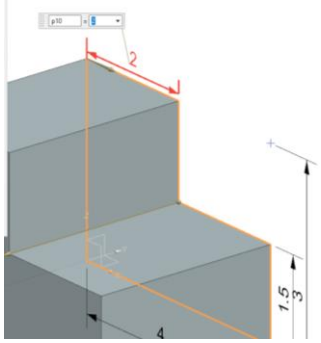
Double Click on Feature in Model



Extrude Properties Menu



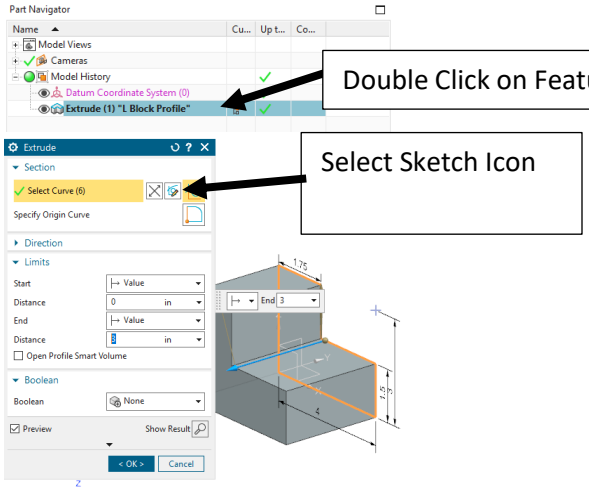
Double Click on Dimension Value



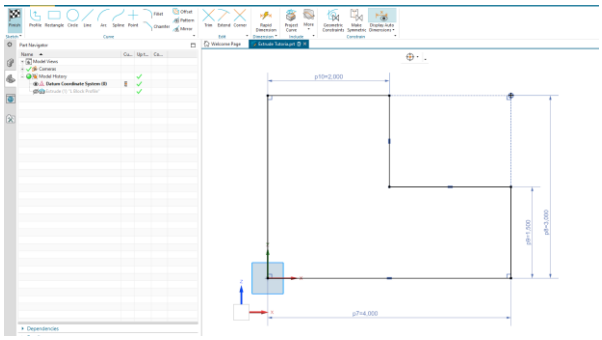
Change Value > Press Enter

NOTE: When Editing the Extrude the Sketch may become detached from the Extrude, this happens when the user selects in space making the software think the user is selecting another profile to extrude. To Correct Select an edge of the profile to reconnect

2. Option 2: Double Click on Extrude Feature > Extrude Properties Menu will with Sketch Dimensions > Select sketch Icon > Edit Sketch > Select Finish Icon > Select Ok in Extrude Properties to complete

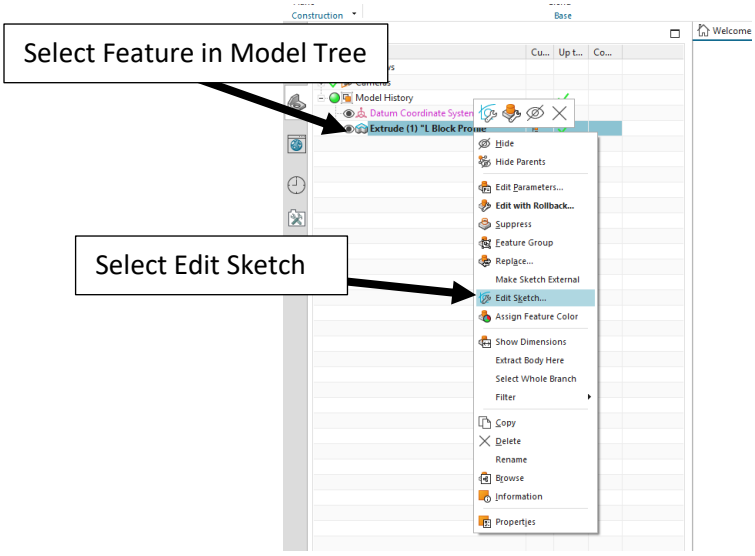


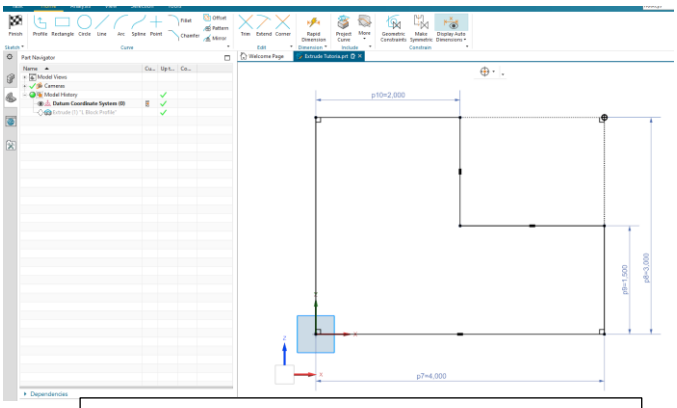
Extrude Properties Menu



Edit Sketch > Select Finish Icon when Edits are Completed

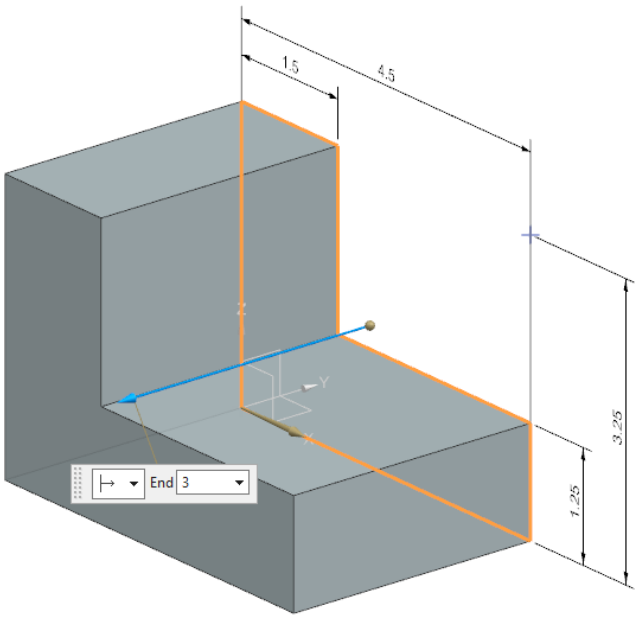
3. Option 3: Select Extrude Feature From Model Tree (Left Click Once) > Right Mouse Button > Select Edit Sketch



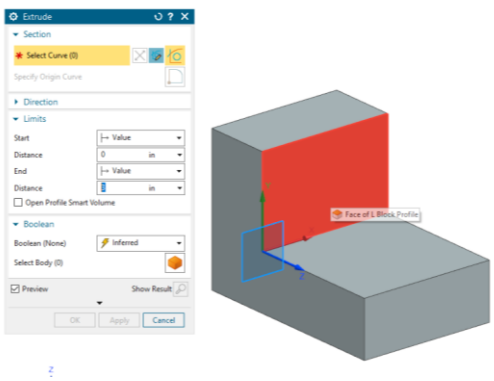


Edit Sketch > Select Finish when Complete

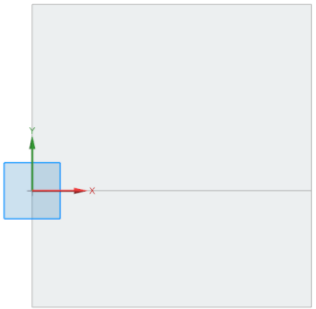
iii. Using one of the methods above > Edit the Sketch to the Following



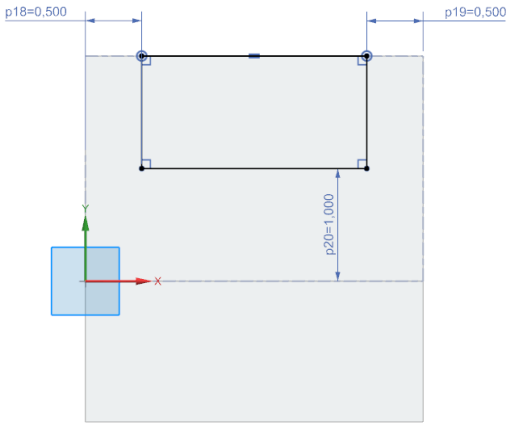
6. Add/Remove Material
    a. Top Cut
        i. Select Home Tab
        ii. Select the Extrude Icon
        iii. Select the Face Shown



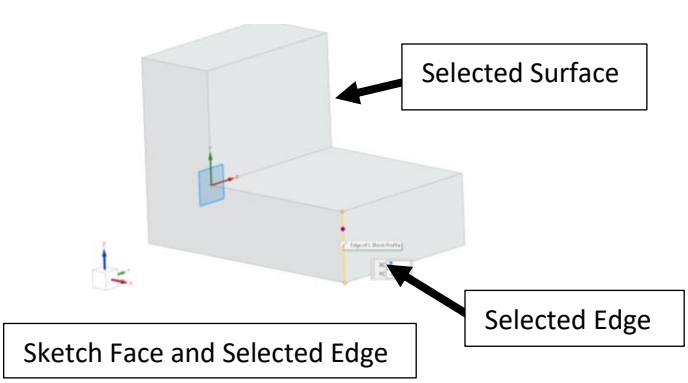
Notice: Object becomes Transparent when selecting. This allows the user to see all of edges of a part

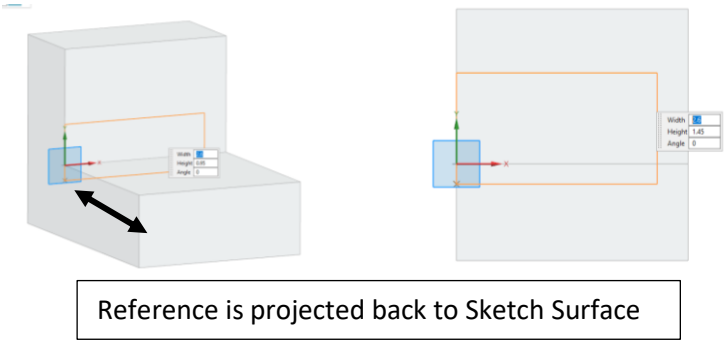


iv. Draw the Following Rectangle Profile and Set Dimensional Values > Select Finish Icon when Complete



Notice NX allows the user to snap to existing edges. NOTE: Edges DO NOT need to be at the same plane as the sketch to snap too
For Example

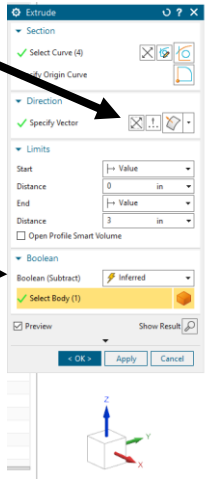


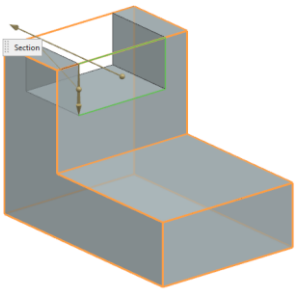


v. Extrusion Settings

1. Select Reverse Direction in the Direction Tab
   Notice: the body automatically switches the extrude from adding material to subtracting material. This is due to the Boolean Setting is set to Inferred. If a user wanted to add material in this direction then the Boolean would need to be changed to Added

Select Reverse Direction

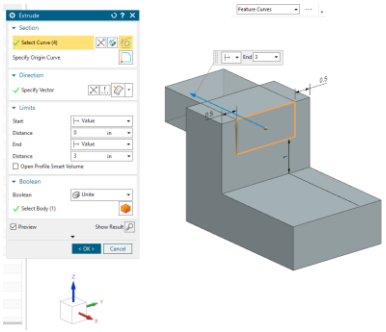




Boolean: Allows user to let the software to decide on adding/subtract material based on location or set by user

Option set to Inferred or Subtract

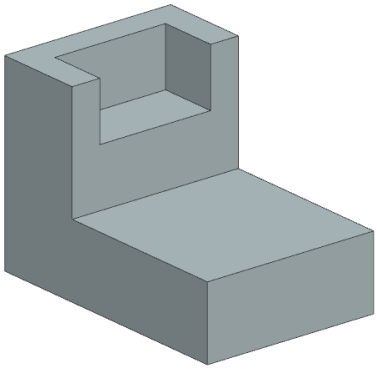
Example of Boolean set to Unite (Adding) thru material



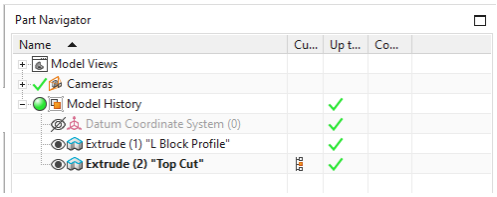
a. Cube Add
b. Arc on Back

2. Extrude Distance 1.00 > Select OK

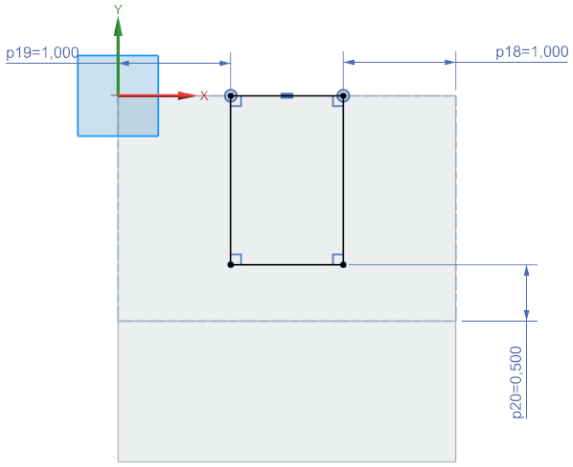
Part Will Look as Follows



vi. Change Extrude(2) Name to Top Cut

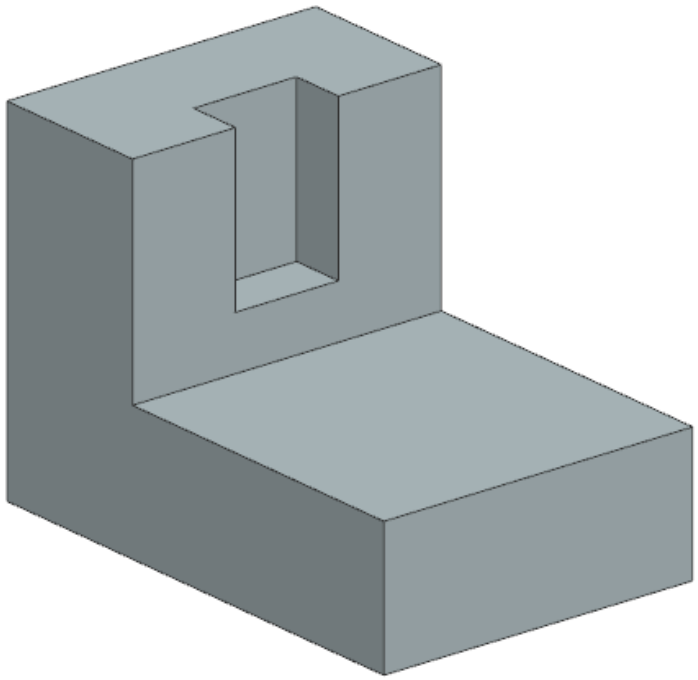


vii. Modify Sketch to the following

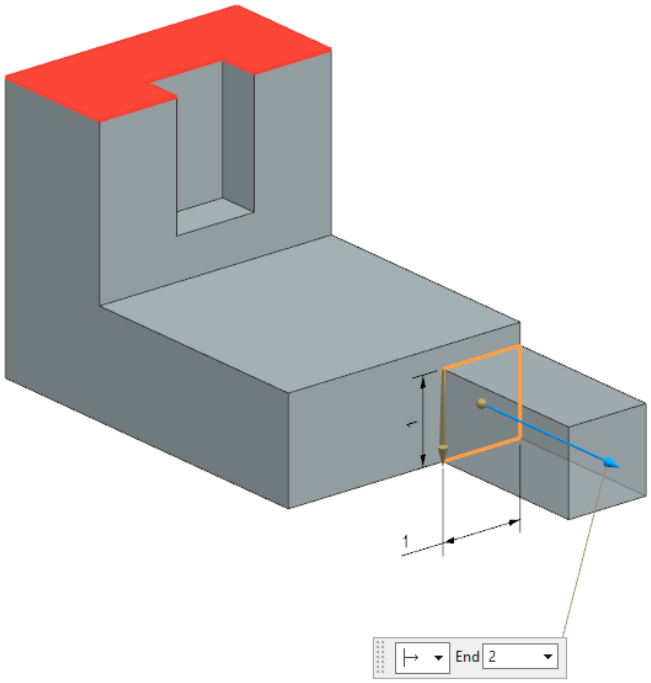


viii. Modify Extrude Distance to .500

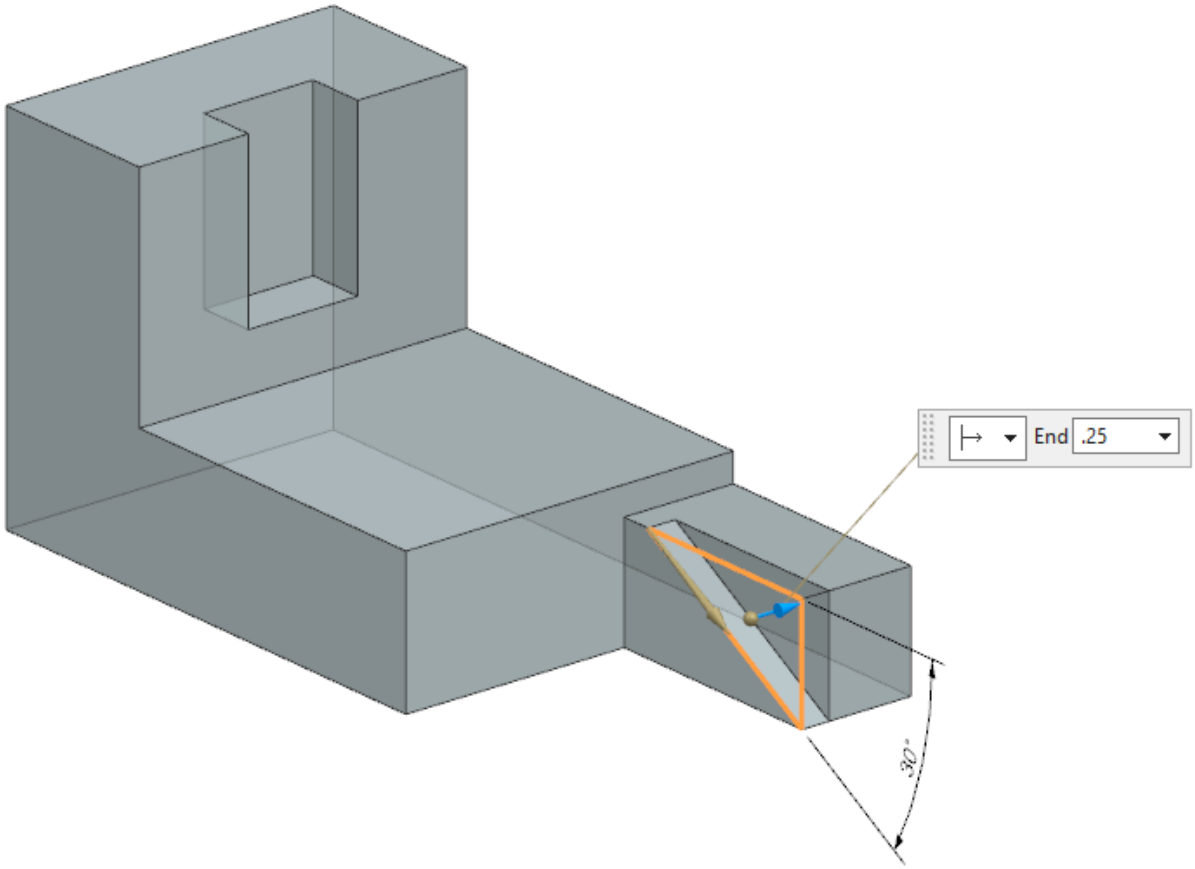
ix. Part Will Look as Follows



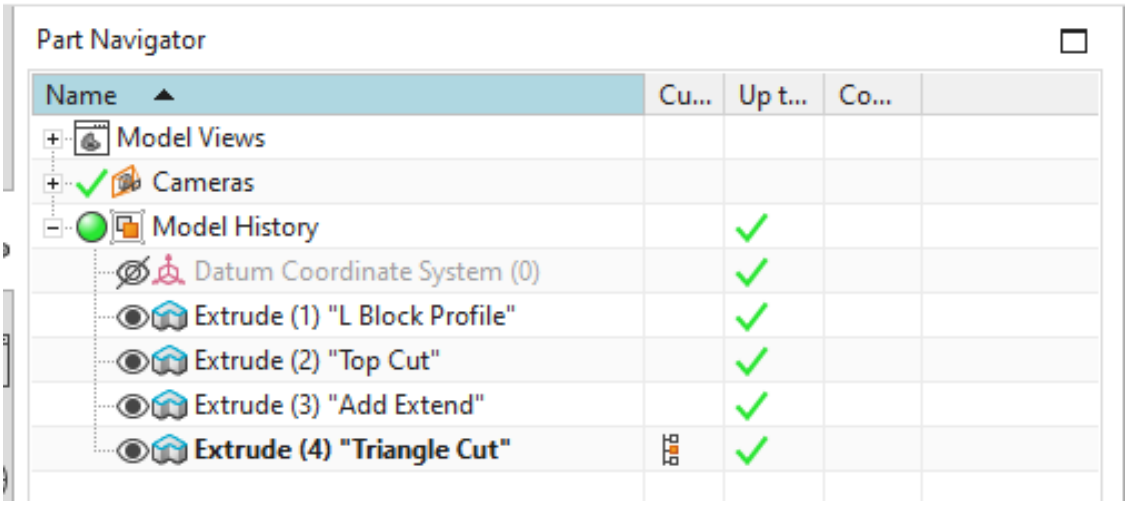
7. Add the Following 2 Features >
    1. Cube Extrude



    2. Triangle Cut



3. Rename Features in Model Tree

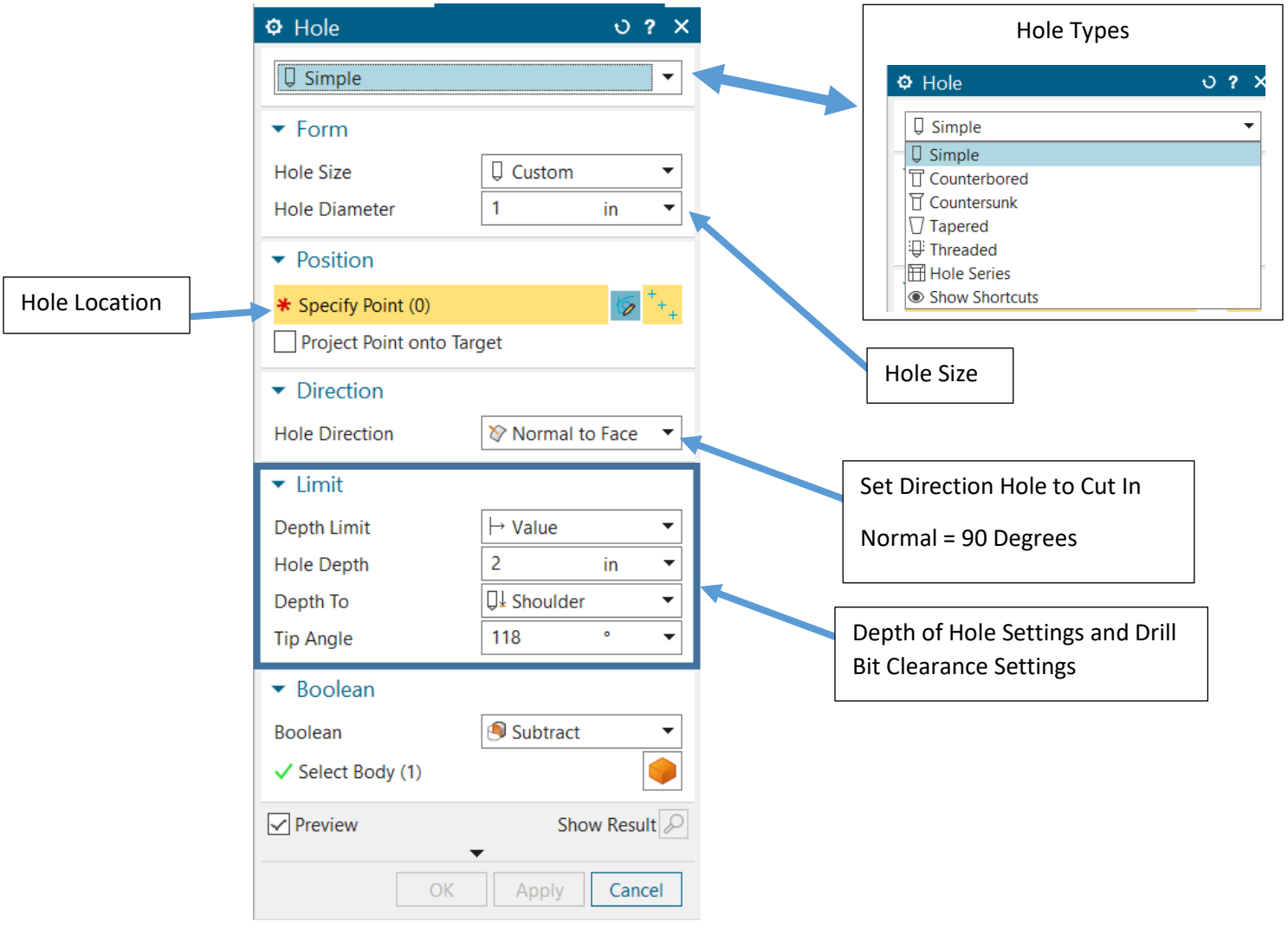


8. Holes

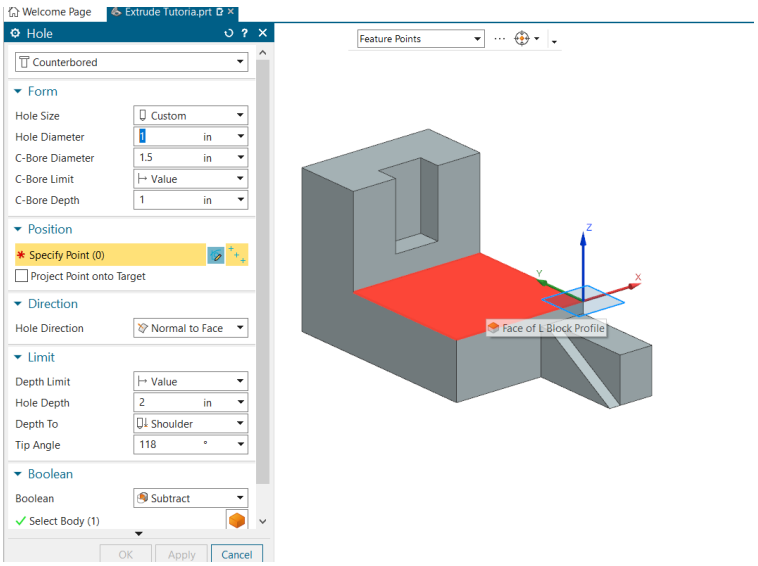
a. Hole Tool: Simple

i. Select the Hole Tool

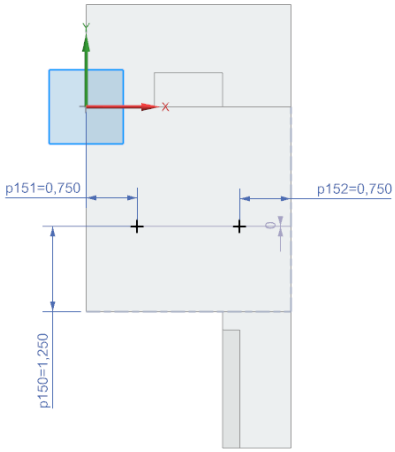
ii. Hole Tool Menu



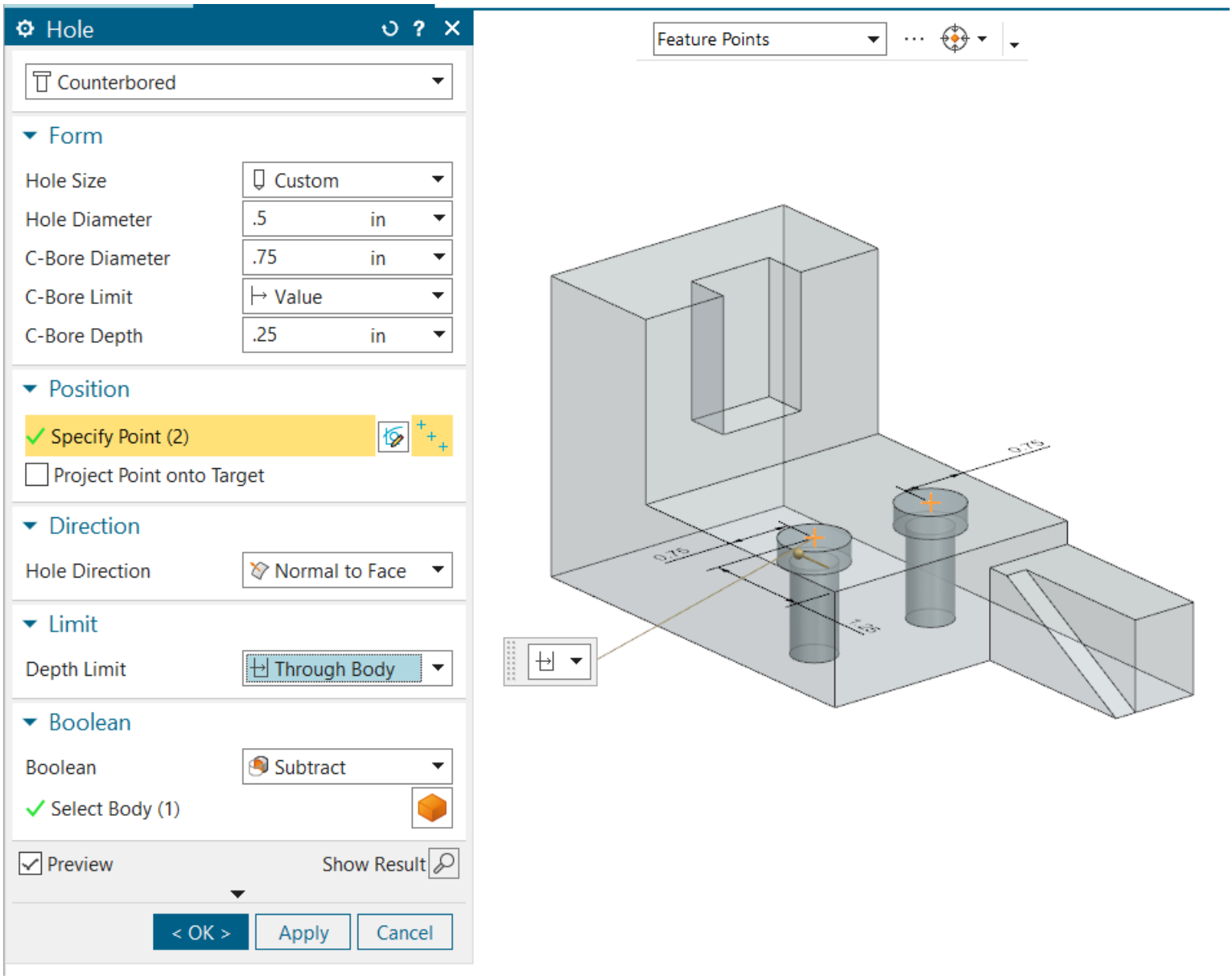
iii. Select the following surface



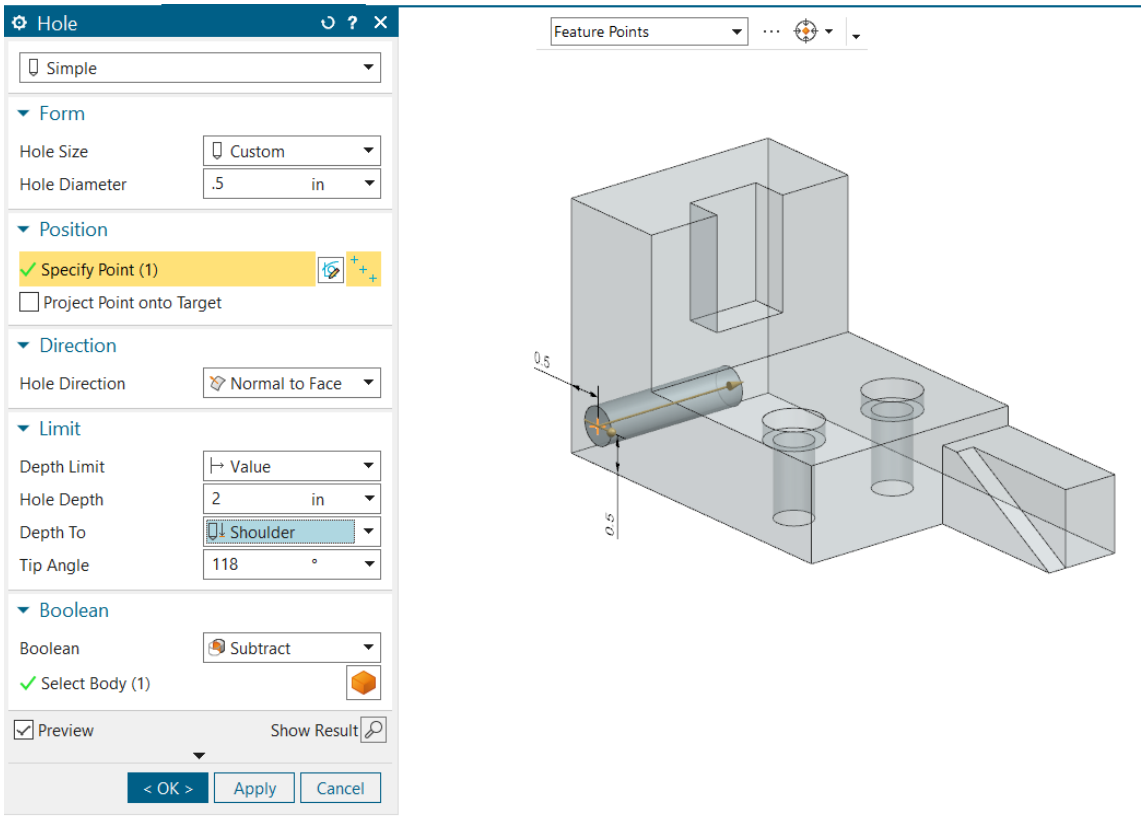
iv. Sketch View will open. User will set Center Points for 1 or more holes > Set the following 2 holes > Select Finish Icon to Exit Sketch
NOTE: Select Point Icon and Place Point on Surface > Set Locator Dimensions



v. Set Hole Type to Counterbore > Set the Following Dimensions > Select OK when Complete

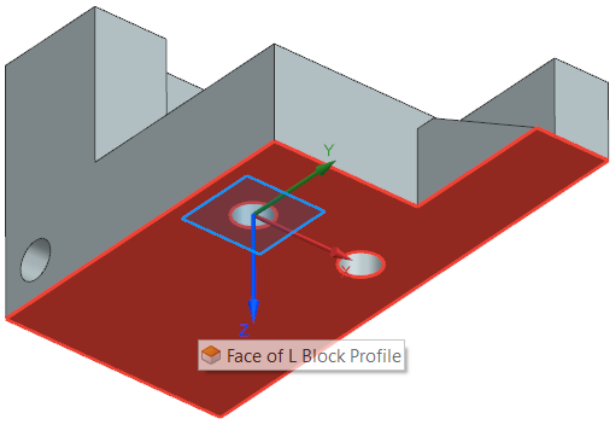


vi. Drill the following simple hole with the settings shown



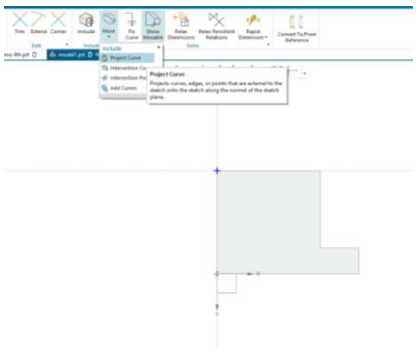
b. Arcs

i. Select Extrude Tool > Select the Bottom Surface of the Block

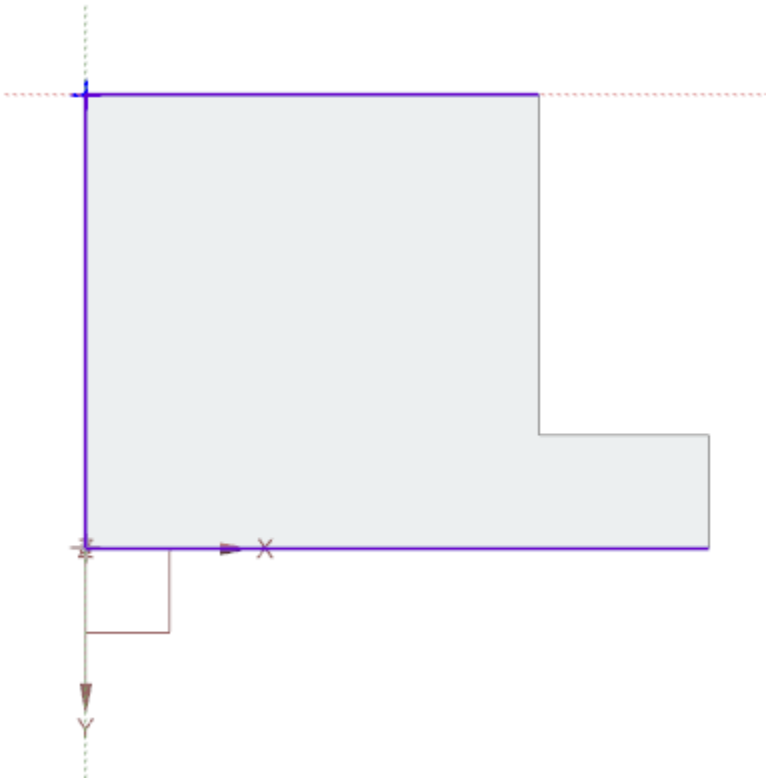


ii. NX Requires the user to reference existing edges by drawing constructions lines on the edges or projected lines

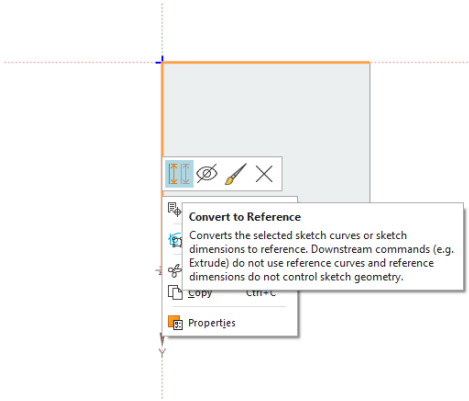
1. More Drop Down Menu > Select Project Curve



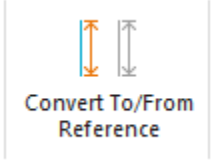
2. Select the Edges as Shown (NOTE Blue Lines are Projected)



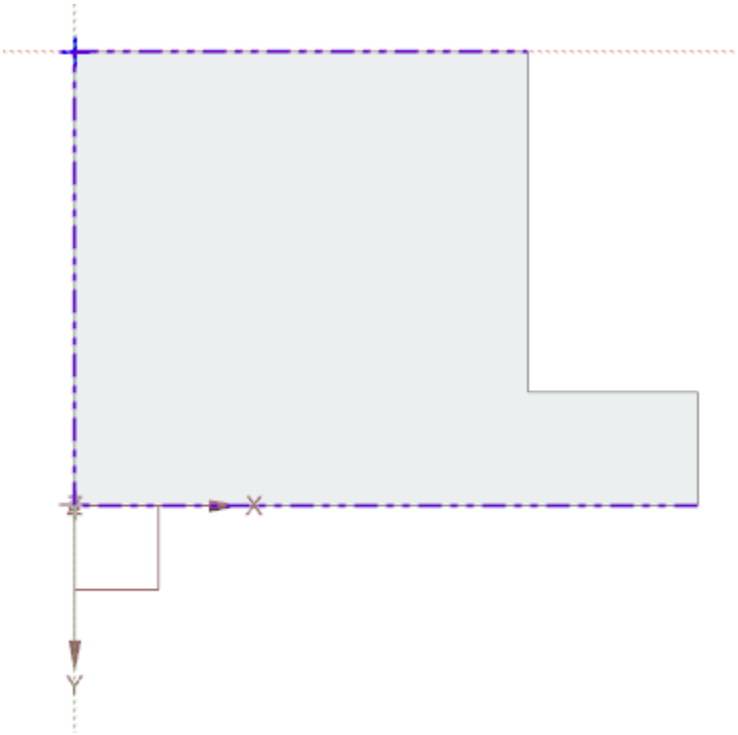
3. Select the Projected Edges > Select Convert To/From Reference
    a. Option 1: Right Click Select Convert To/From Reference



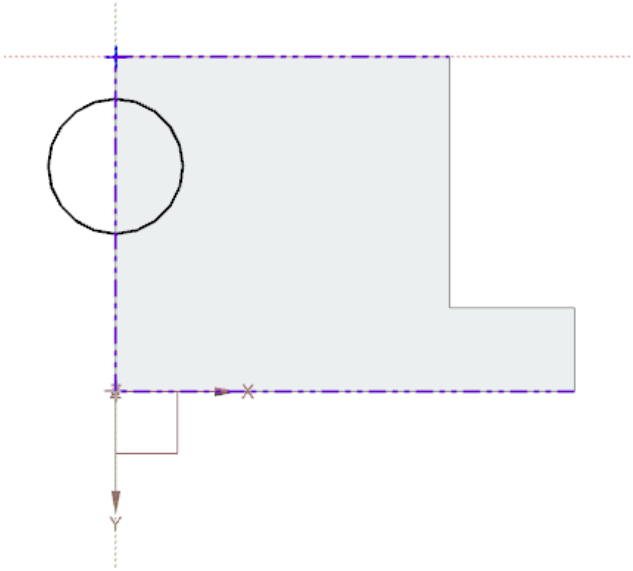
b. Option 2: Add Tool Icon to Tool bar



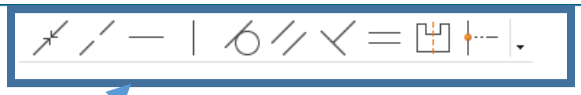
Final Construction Lines



iii. Draw a circle on the line edge shown
NOTE: Distance does not matter, a tangent constraint will be used to lock distance into place

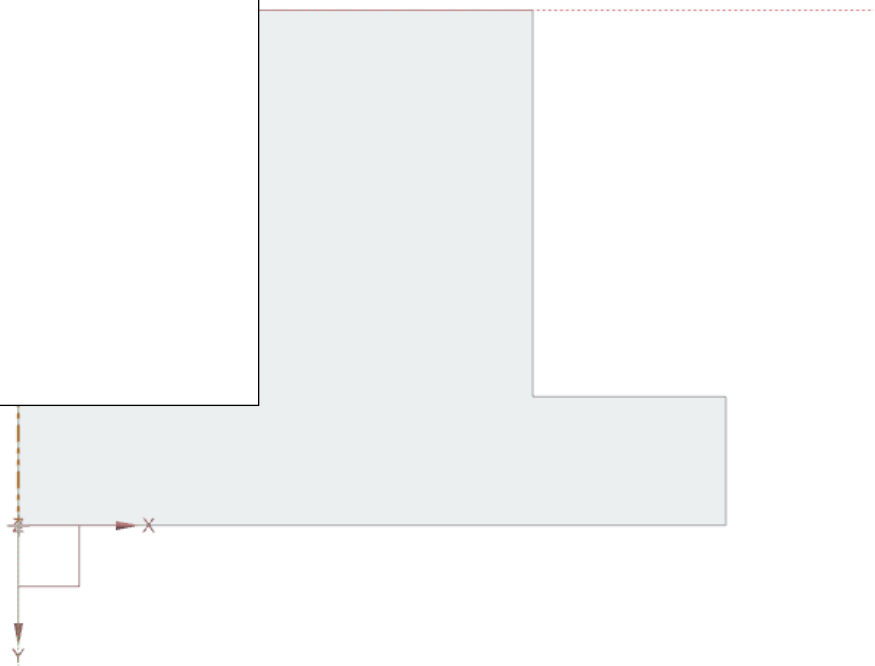


NOTE: Constraint Tools found at the Top of the Workspace when there are not any tools selected

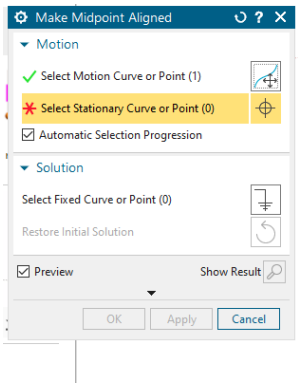


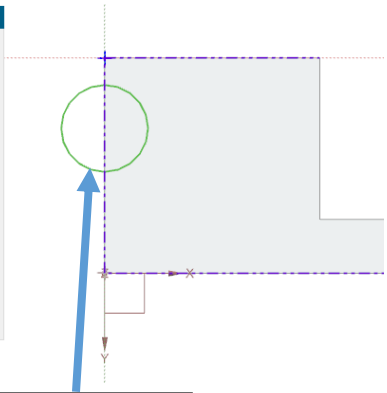
Constraint Bar (Tools Left to Right)

1. Coincident
2. Collinear
3. Horizontal
4. Vertical
5. Tangent
6. Parallel
7. Perpendicular
8. Equal Length
9. Symmetrical
10. Mid-Point

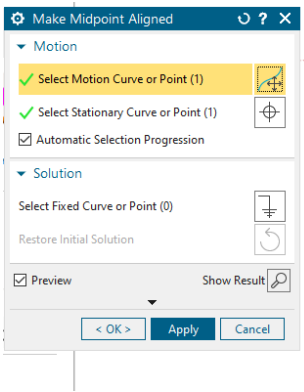


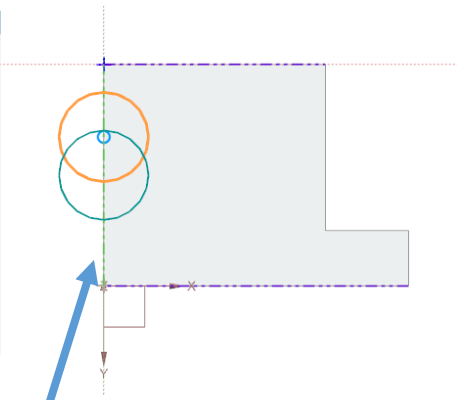
1. Select MidPoint Icon  > Select Select Edge Edge of Circle > Select Construction Line> Press Apply





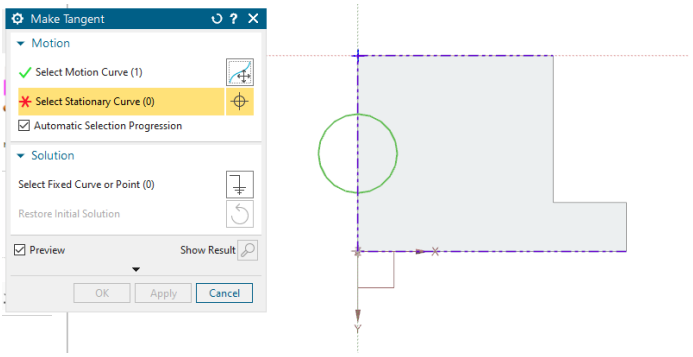
Select Circle 1st



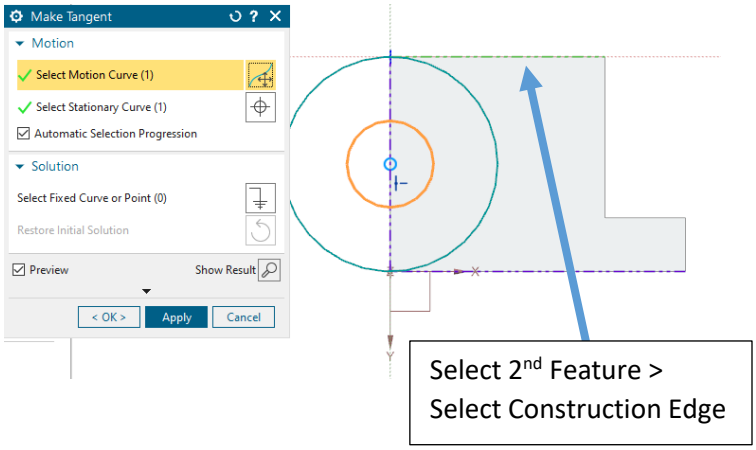


Select 2nd Feature Option First > Then Select Edge > Press Apply To Lock the Circle in place

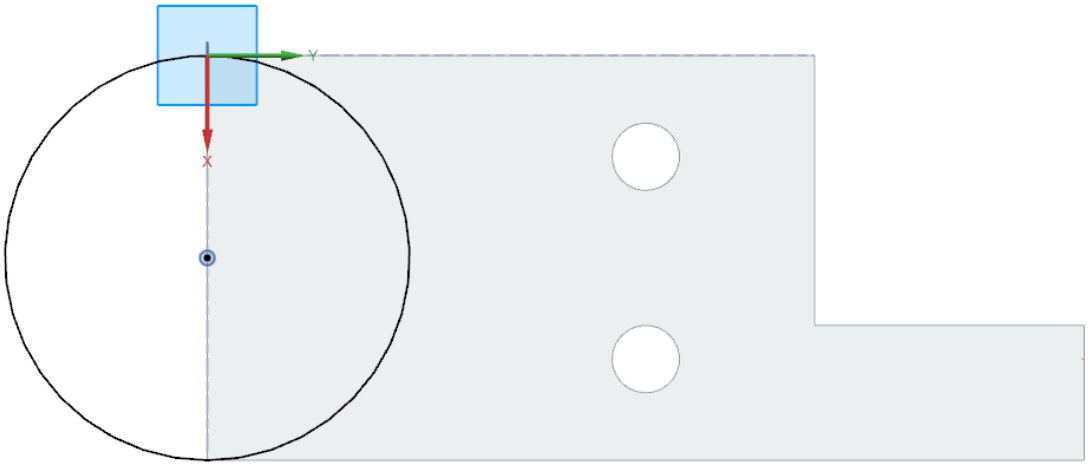
2. Select Tangent Icon  > 1st Select Circle > Select 2nd Feature Selection > Select Horizontal edge of part



Select Tangent Tool > Select Circle

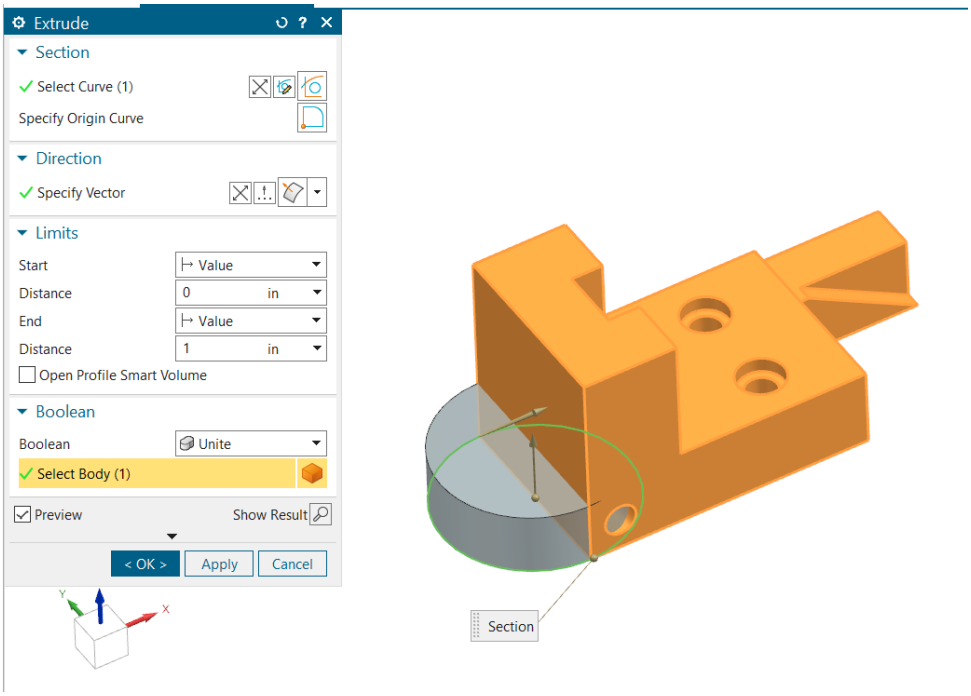


Select 2nd Feature > Select Construction Edge

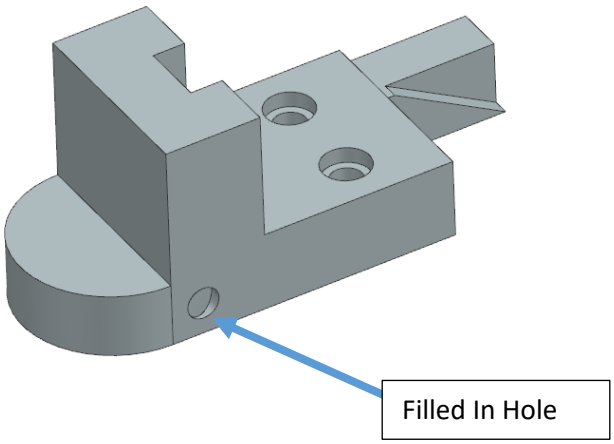


3. Select Finish Icon

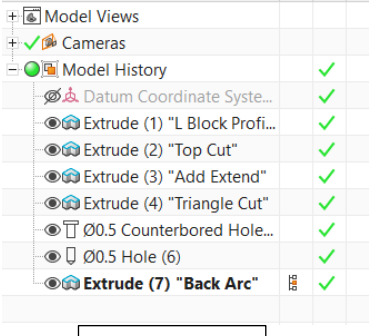
4. Extrude Distance 1”
   NOTE: The software set to “Inferred” which means the default state will make the feature cut into the part > Change the Boolean to Unite to make the feature merged with the part > Select OK to Accept Extrude



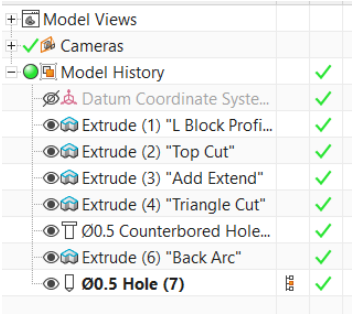
5. Model Tree > Rename Feature Back Arc
6. Notice the Side Hole is filled in



   a. 2 Options to Fix
      i. Option 1: Reorder the Model Tree: Place Cursor on Back Arc on Model Tree > Hold Left Mouse Key down > Drag Feature above Dia. .50 Hole

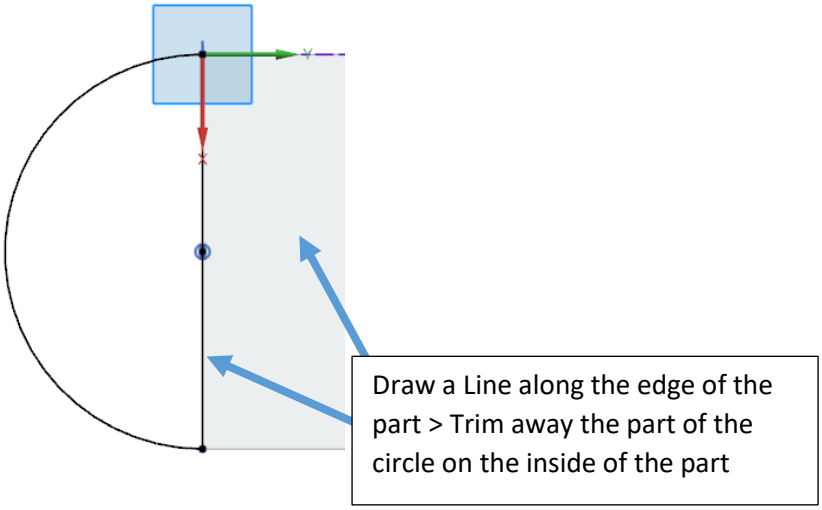


Before



After

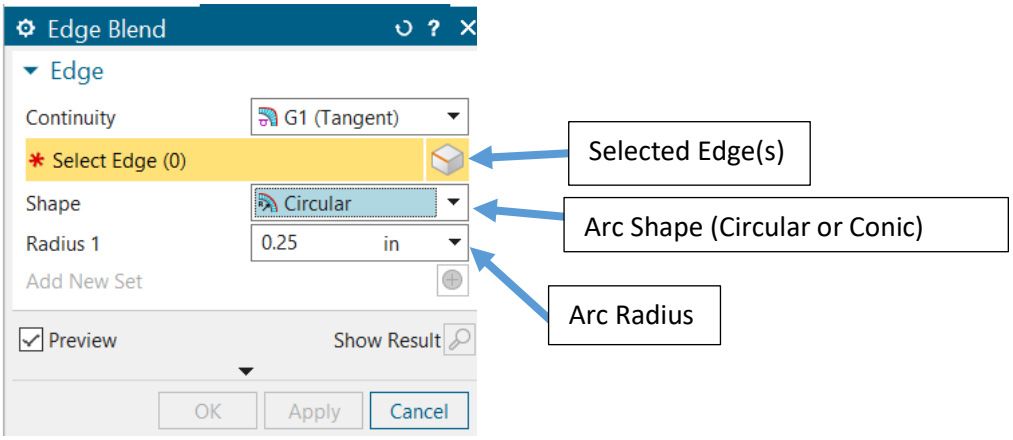
ii. Option 2: Edit Sketch Turning the Circle into an arc



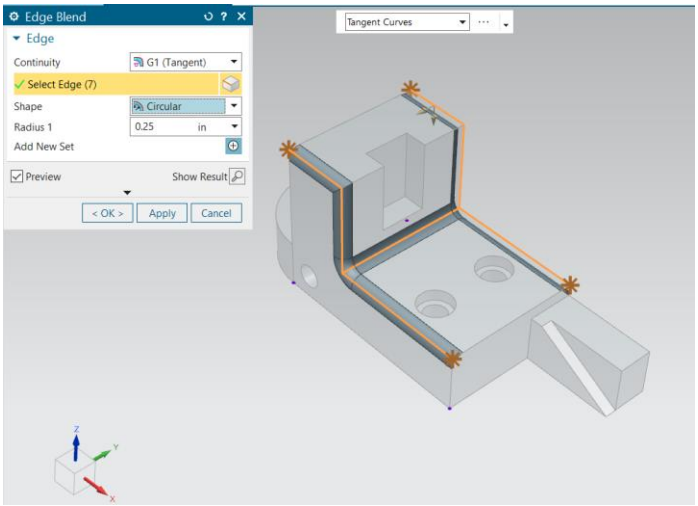
c. Fillets and Rounds

i. Select Edge Blend Tool 

(Edge Blend will Create Fillets and Rounds)

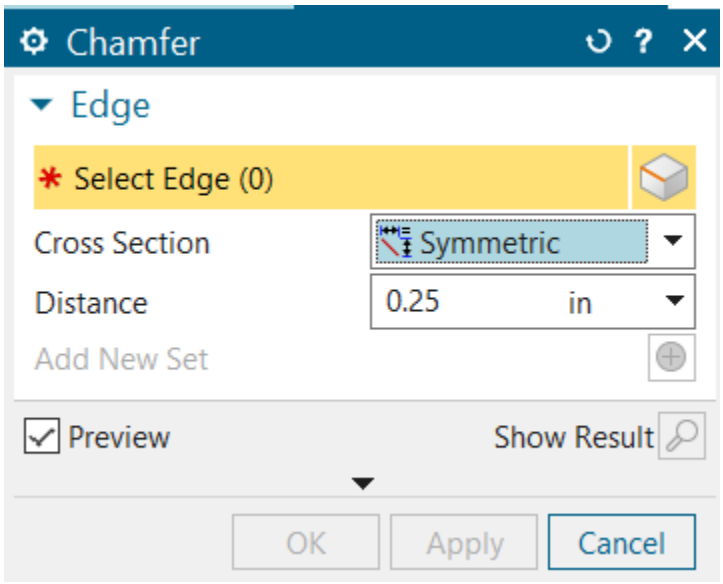


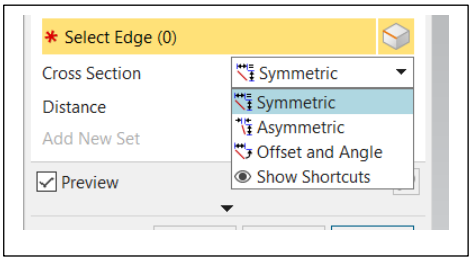
ii. Select the following Edges> Select OK



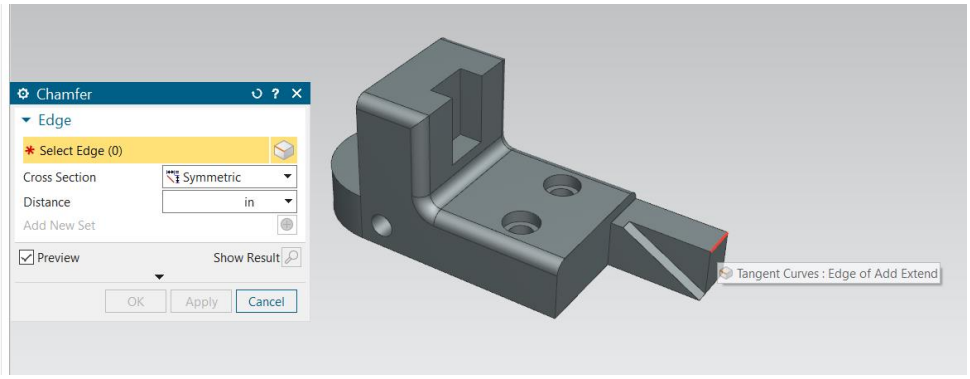
d. Chamfer

i. Select Home Tab > Select Chamfer Tool 



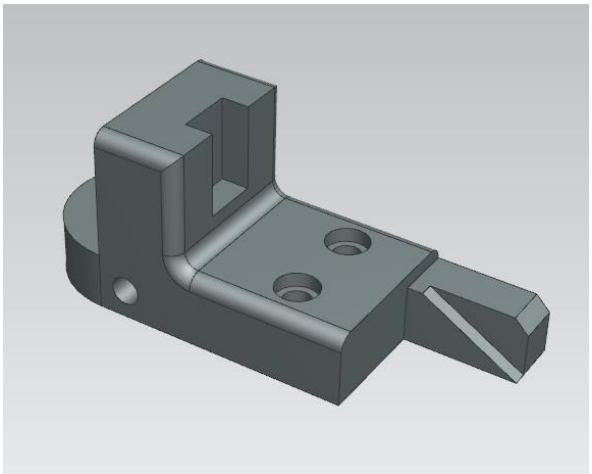


ii. Select the Following Edge



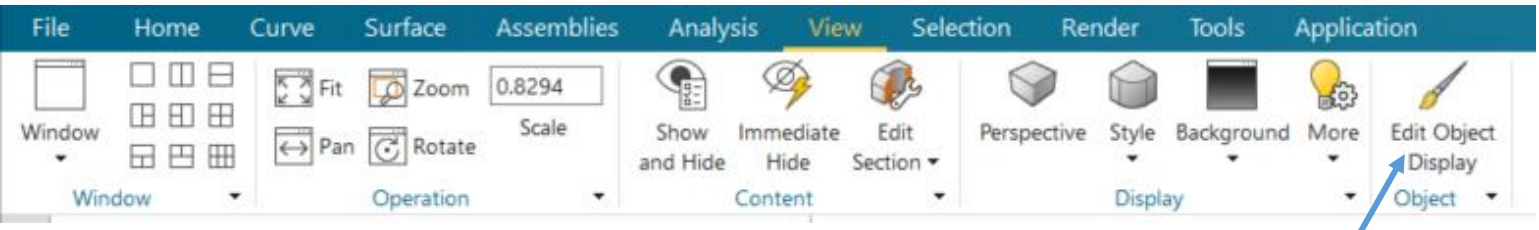
iii. Select Ok

Final Part



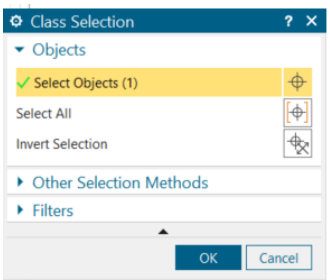
Optional: Change Part Color

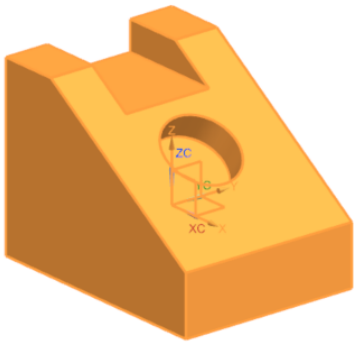
i. Select View Tab > Select Edit Object Display



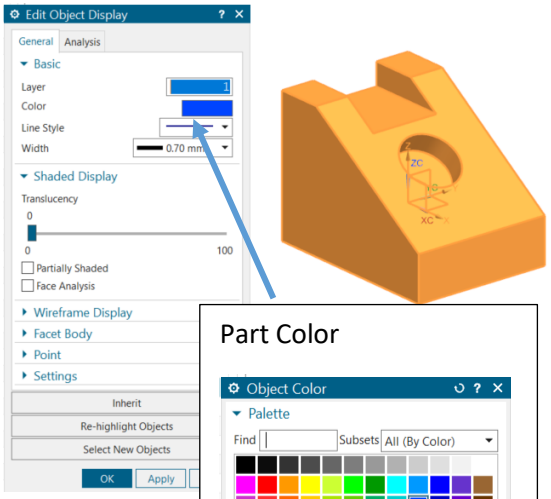
ii. Class Selection Menu Appears

1. Select Part(s) > Select Ok





2. Select Part Color to Change to > Select Apply When Complete
NOTE: Menu maybe used to alter many other features colors, line types, line weight, etc.



Part Color