ZW3D from Entry to Master Tutorial

Mold Design

ZW3D™ V2023 From Entry to Master Mold Design

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Foreword

In this tutorial, we provide various case studies, which are from easy to difficult and combine theory with practice. We hope to improve users' 3D CAD/CAM skills and techniques with ZW3D.

The tutorial bases on our technical engineers' years of experience in the industry and ZW3D, which is the fruit of a lot of efforts and wisdom. We sincerely hope that the tutorial will do help to you, and your precious advice on it is highly welcomed.

There are three series for this tutorial: ***Primary Tutorial***, ***From Entry to Master Tutorial***, and ***Advanced Tutorial***. From easy to difficult, they offer a step-by-step learning process that can meet different user needs.

Primary Tutorial series is for users who have little or no prior 3D CAD/CAM experience. If you are green hands of 3D CAD/CAM software, or if you are a new user of ZW3D, we recommend that you get started with this tutorial. Here you can learn the basic knowledge and concepts of ZW3D, rapidly master the simple operations and workflows of ZW3D, and practice simple cases.

From Entry to Master Tutorial series is for users with basic know-how of 3D CAD/CAM software. If you have experience in 3D CAD/CAM software and want to master common functions of ZW3D, we suggest that you start with this series. Here you can dig deeper into the functions and master more operations of ZW3D.

Advanced Tutorial series is for users with practical experience in 3D CAD/CAM software. If you hope to have a comprehensive command of ZW3D and get the complicated operations done independently, you can choose to learn this series. Here you can learn to use the software more flexibly and get rich experience to increase your efficiency.

What you are learning is ***ZW3D From Entry to Master Mold Design***, a master tutorial.

Thanks for being our user!

The ZW3D Team

Contents

1 Mold Design

Key Points:

- ✧ Mold Preparation
- ✧ Region Analysis and Definition
- ✧ Parting Line
- ✧ Patch and Parting Faces
- ✧ Insert, Slider and Lifter
- ✧ Mold Base and Standard Parts

Notes:

This tutorial is based on Metric system. All mold base and standard parts library will be in Metric.

1.1 Import and Healing Model

Mold design can start from ZW3D native model or other imported external data.

This tutorial starts from importing external data.

STEP 01 Create a new file ***Project.Z3***.

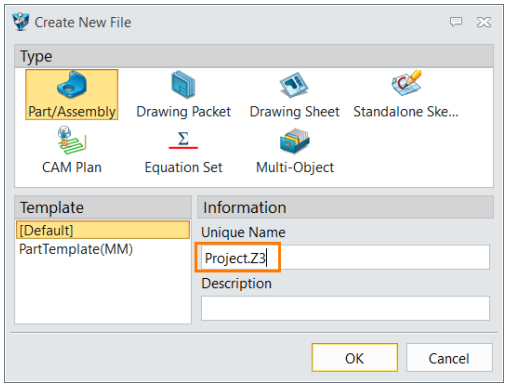


Figure 1 Create a New File

STEP 02 Exit to object manager, Rename the part as ***001_Part_Model***.

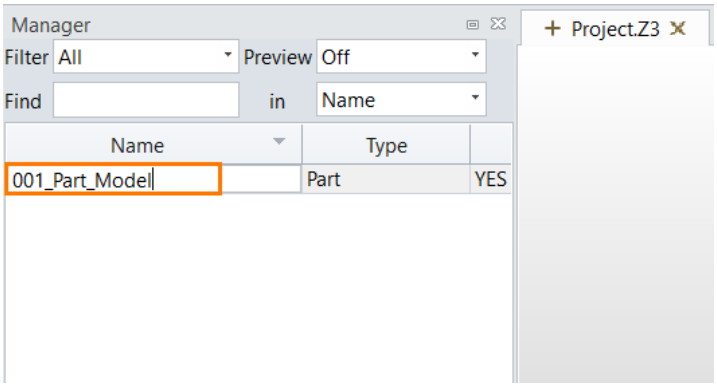


Figure 2 Rename the Part

STEP 03 Click ***Import***, search for the path "***…\ZWSOFT\ZW3D 2021Eng\training***", then select the file ***Back Cover***.

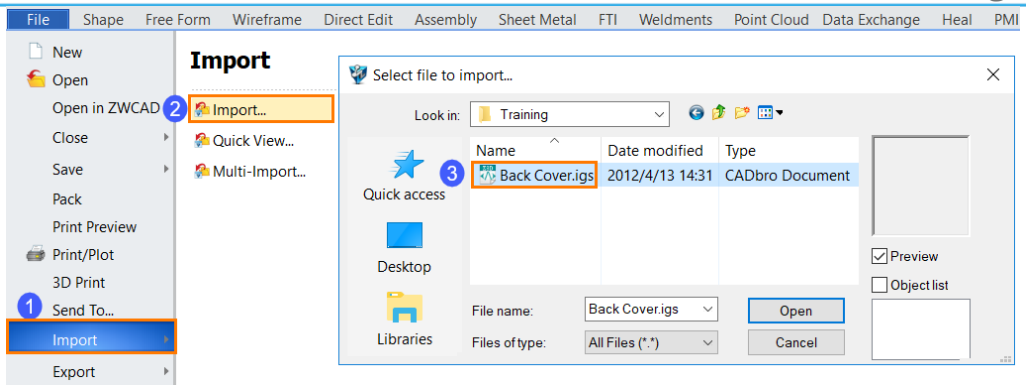


Figure 3 Import External Model

STEP 04 After importing, ***Ctrl+A*** to zoom all, then the imported model will show as Figure4.

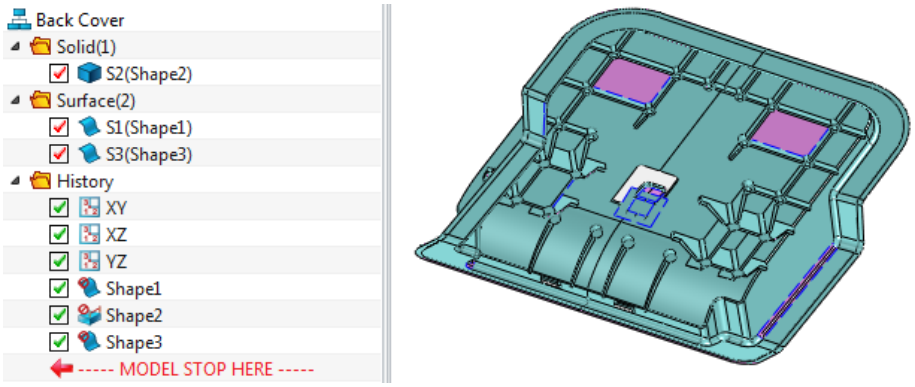


Figure 4 Imported Part Model

STEP 05 Use ***Show Open Edges*** in ***Heal*** tab, then you can find ***open edge number*** is shown as Figure5.

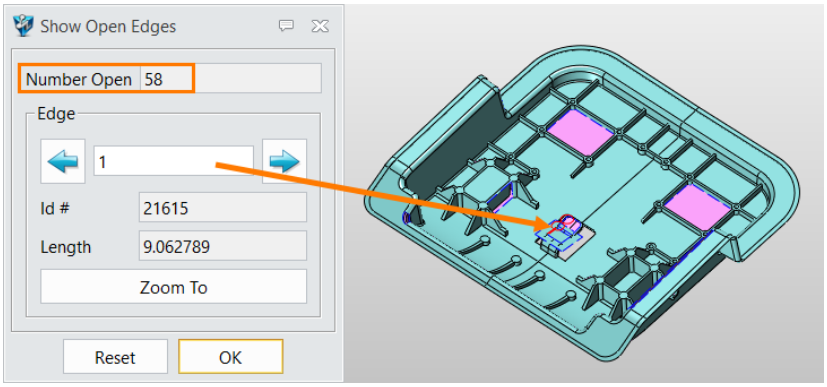


Figure 5 Show Open Edges

STEP 06 Click ***Zoom to***, and it will zoom to the highlighted open edge.

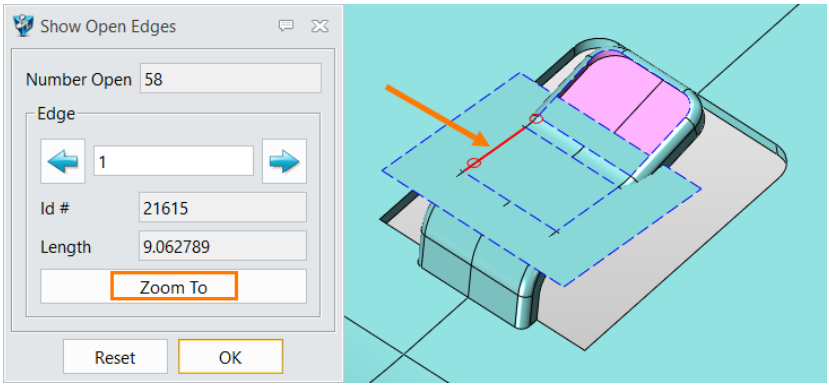


Figure 6 Zoom to highlighted open edge

STEP 07 Use ***Erase*** to delete needless faces.

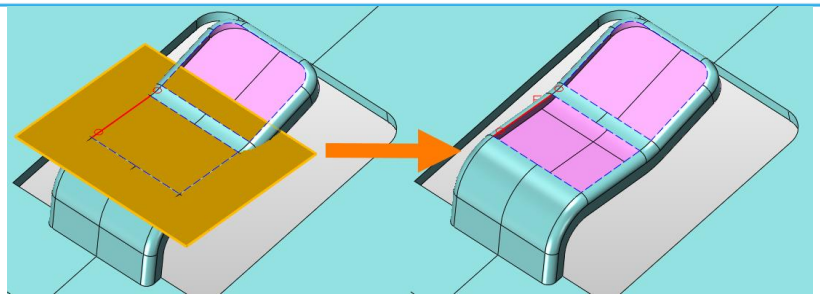


Figure 7 Erase Needless Faces

STEP 08 Use ***Fill gap*** in ***Heal*** tab, pick the highlighted edge. It will automaticlly pick the surrounded loop.

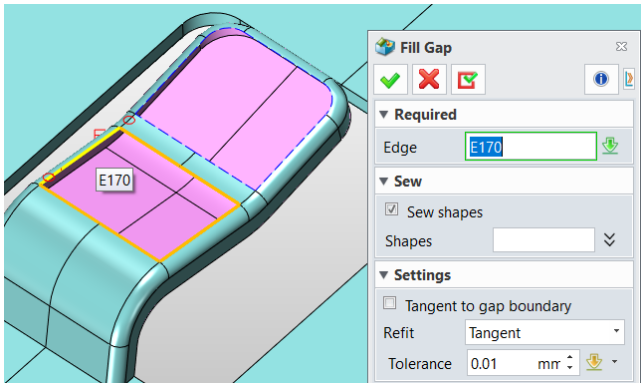


Figure 8 Fill Gap

STEP 09 Click ***OK***, the gap will be fixed, and the open edge number reduces.

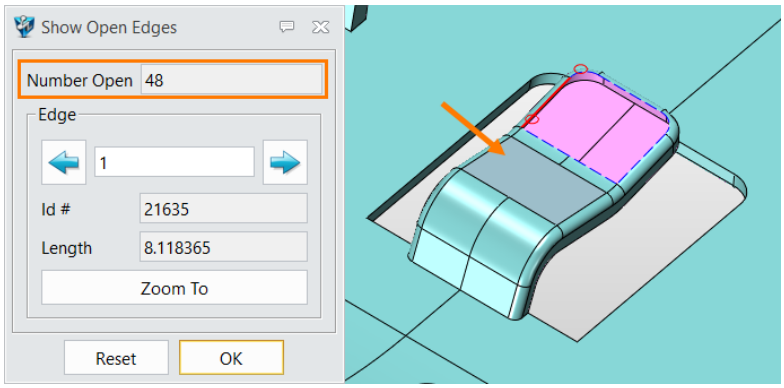


Figure 9 Fill Gap Result

STEP 10 Use the ***Fill Gap*** to fix another loop, the result will show as figure10.

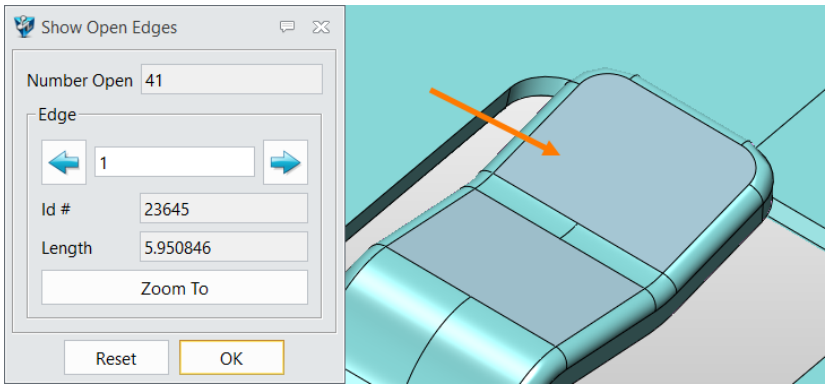


Figure 10 Fill Another Loop

STEP 11 Continue to ***Zoom to*** another open edge. Use ***Fill Gap*** command, then pick the open edge. Setting is shown as Figure11.

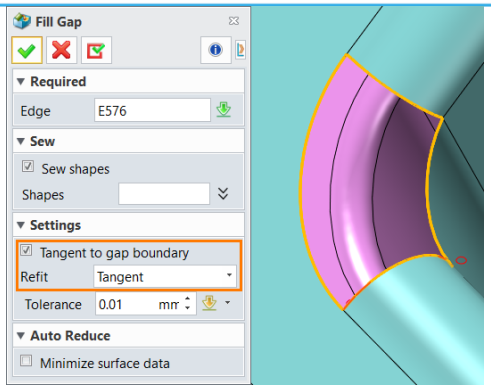


Figure 11 Fill Gap with Tangency to Gap Boundary

STEP 12 ***Zoom to*** another open edge. Use ***Fill Gap*** command, then pick the open edge. Setting is shown as Figure12.

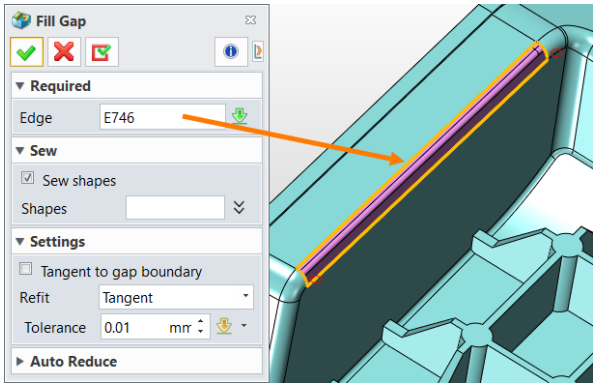


Figure 12 Continue to Fill Gap

STEP 13 Continue to ***Zoom to*** another open edge. Use ***Trimmed plane*** command in Free Form tab, then pick the open edge. Setting is shown as Figure13.

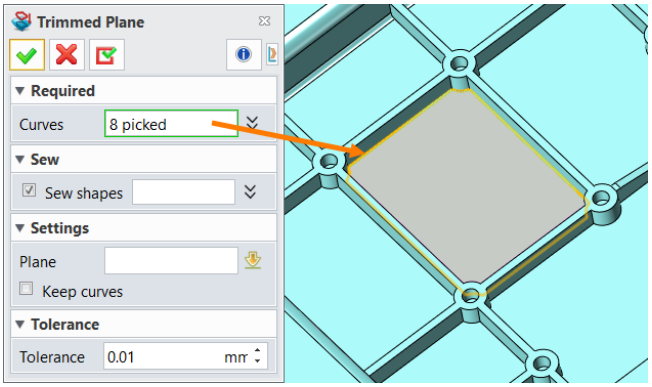


Figure 13 Trimmed Plane

STEP 14 Continue to ***Zoom to*** another open edge. Use ***Ruled*** command in ***Free Form*** tab, then pick the open edge. Setting is shown as Figure14.

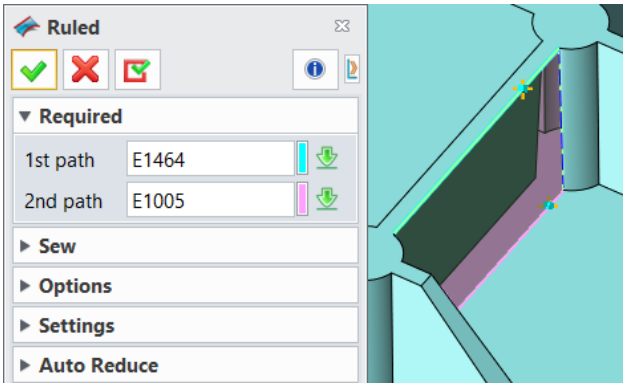


Figure 14 Create Ruled Face

STEP 15 ***Zoom to*** another open edge. Use ***Trim to curves*** command to trim the face. Setting is shown in Figure 15.

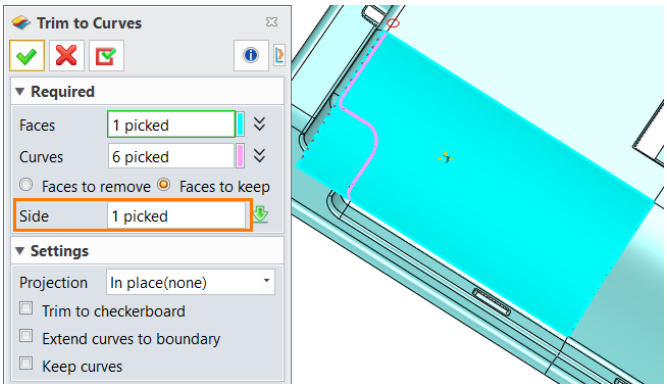


Figure 15 Trim to Curves

You can get the result like Figure 16.

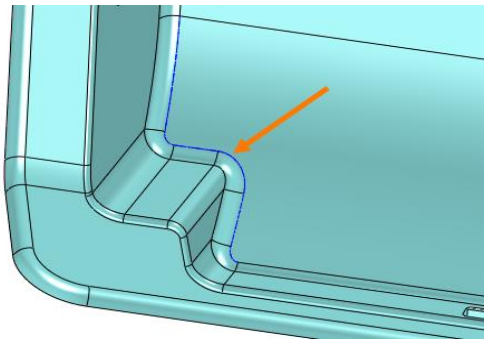


Figure 16 Result of “Trim to curves”

STEP 16 Then use ***Sew*** command in ***Heal*** module to sew the gap.

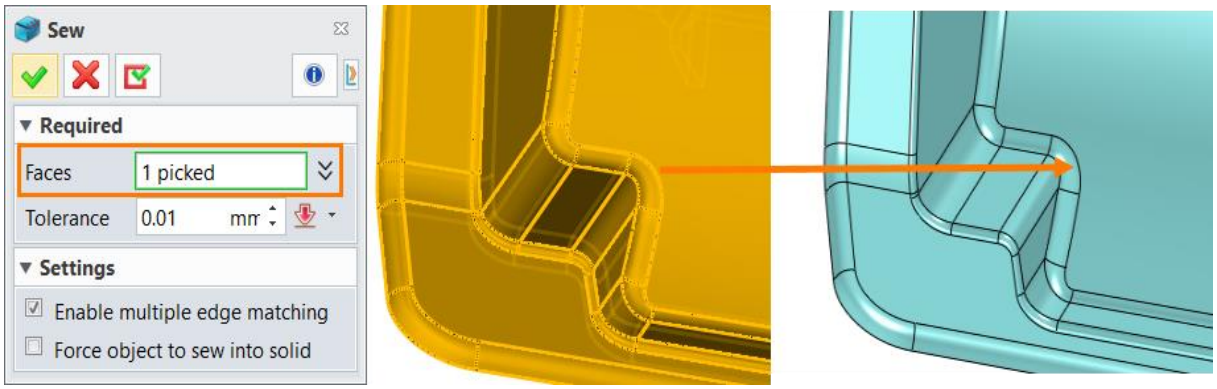


Figure 17 Sew the Gap

STEP 17 ***Zoom to*** another open edge. Then use ***Extend Face*** command in ***Free Form*** tab to extend the face till the boundary merges

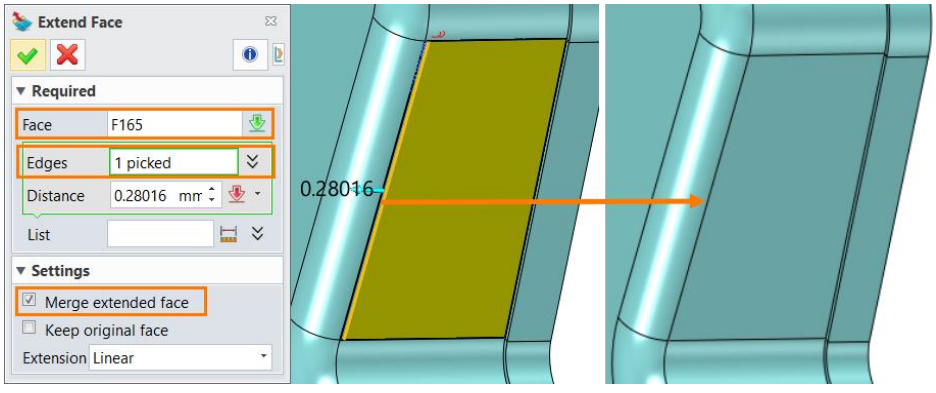


Figure 18 Extend Face

STEP 18 ***Zoom to*** another open edge. Repeat STEP 13 to ***Trimmed Plane***

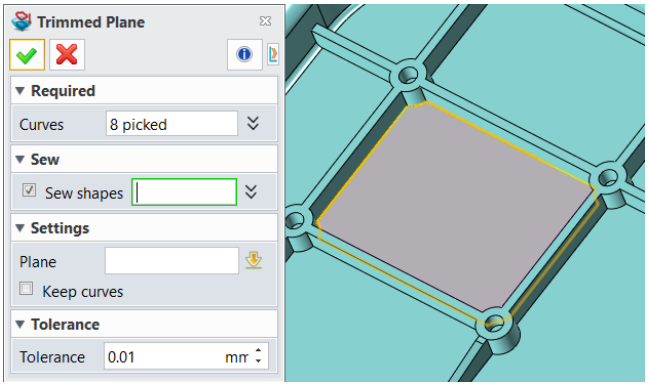


Figure 19 Trimmed Plane

STEP 19 After that, the number of open edges is 0, and the surface will change to a solid.

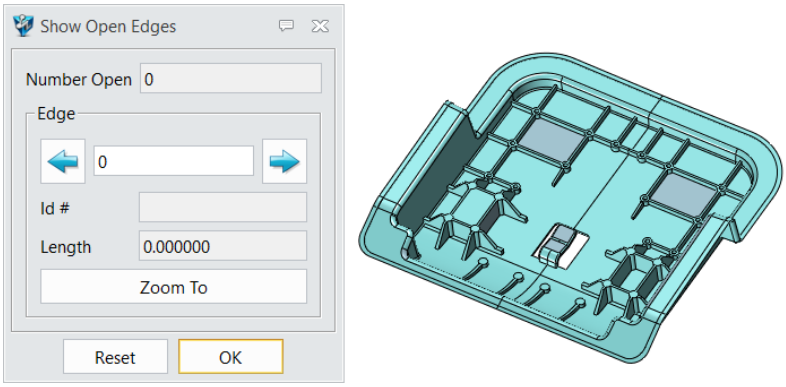


Figure 20 Close All Open Edges

STEP 20 Check the model again, there is an intersection with other shapes.

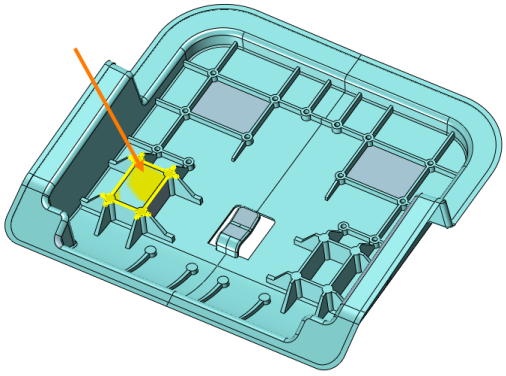


Figure 21 Check the intersection

STEP 21 Use ***Remove Shape*** command in ***Shape*** tab.

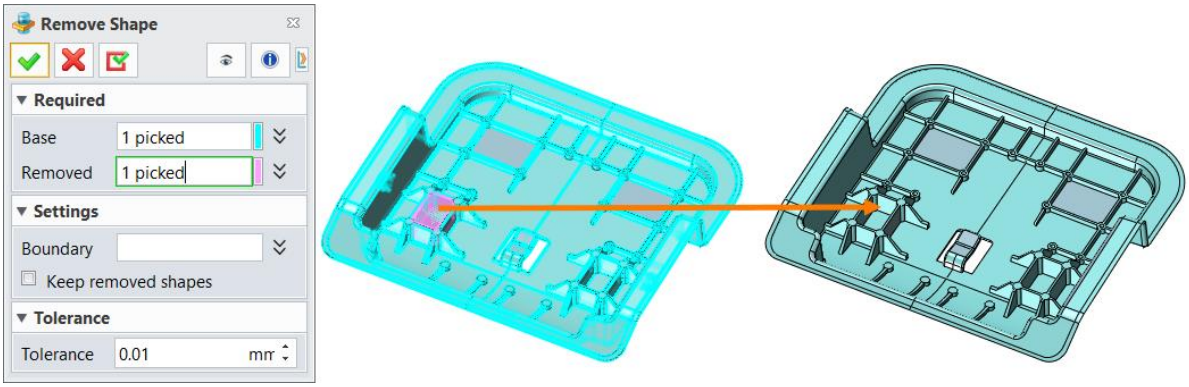


Figure 22 Remove Shape

Up to now, the model has been healed for mold design preparation.

STEP 22 Save the file.

1.2 Transfer, Analysis and Shrink

1.2.1 Transfer the Product

After healing the model, transfer the product to a suitable position and direction for easier demold.

STEP 1 Create a new object named ***002_Position_Shrink***

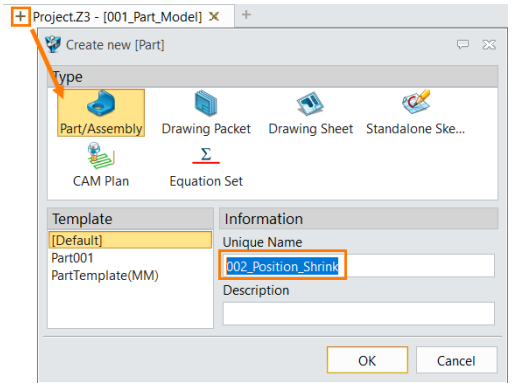


Figure 23 Create New Object for Position and Shrink

STEP 2 Use ***Import*** command in ***Mold*** tab to import ***001_Part_Model***

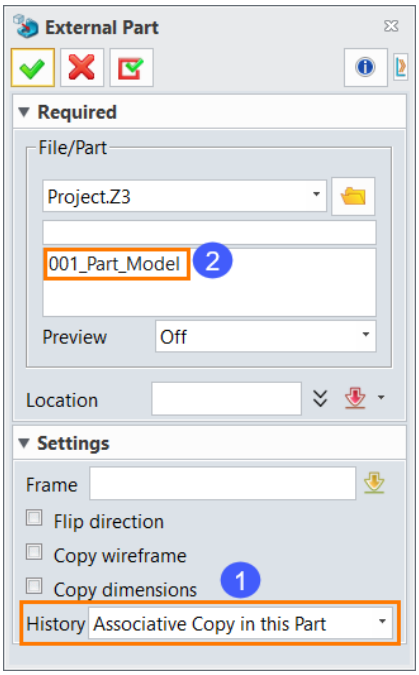


Figure 24 Import "001_Part_Model"

STEP 3 Use ***Align*** command to correct the product position correct to meet the request of mold design.

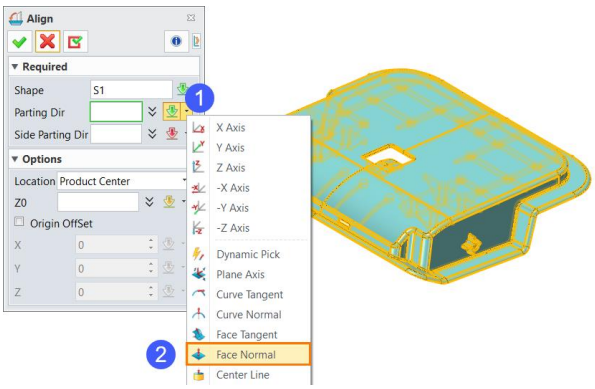


Figure 25 Align the Product

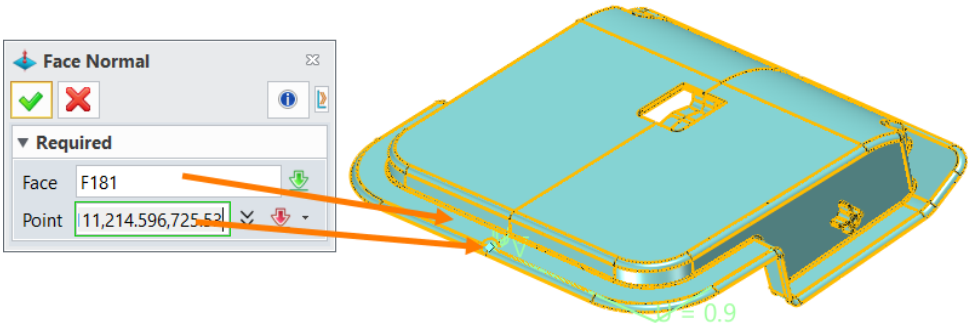


Figure 26 Pick Face Normal

Then pick side direction as Figure27.

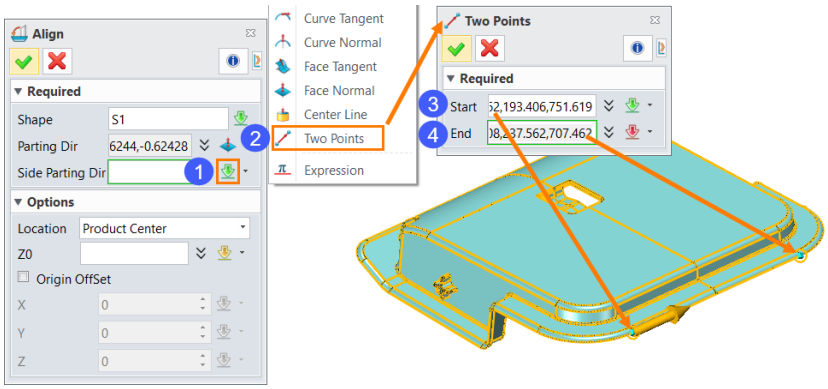


Figure 27 Pick Side Direction

Then pick Z0 as figure28.

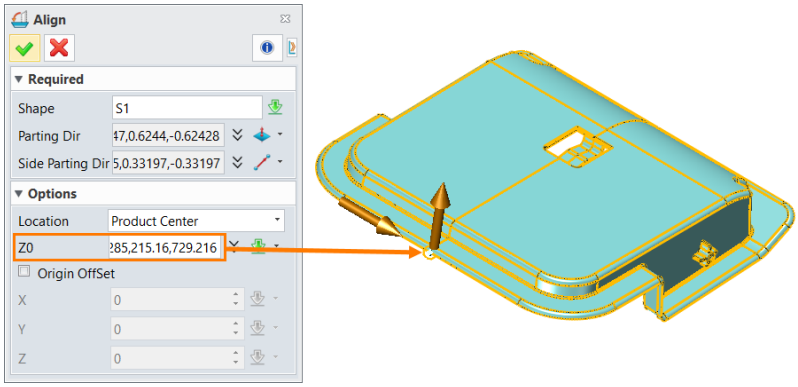


Figure 28 Pick Z0

STEP 4 Click ***OK***, then press ***Ctrl+A*** key to zoom all. The result shows as Figure 29.

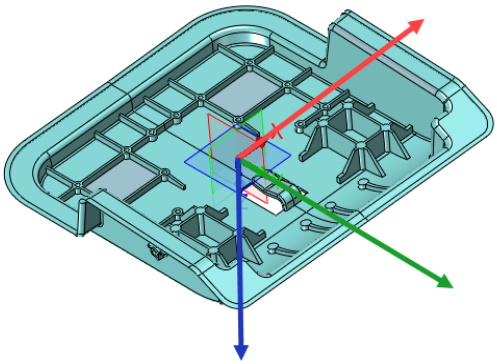


Figure 29 Result of Alignment

1.2.2 Draft Angle Analysis and Thickness Analysis

STEP 1 Use ***Draft*** command in ***Mold*** tab to verify the draft angle on the 3D model.

Set as below.

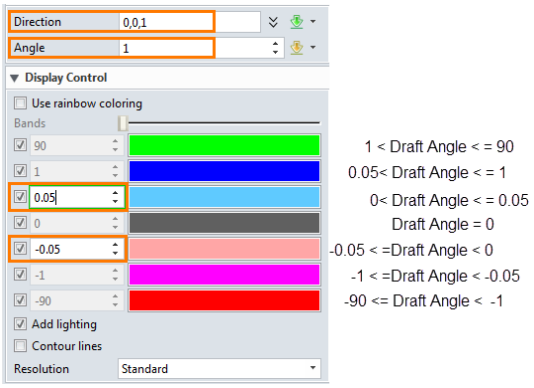


Figure 30 Draft Angle Analysis

Like the below result shows, there are some vertical faces and undercut faces.

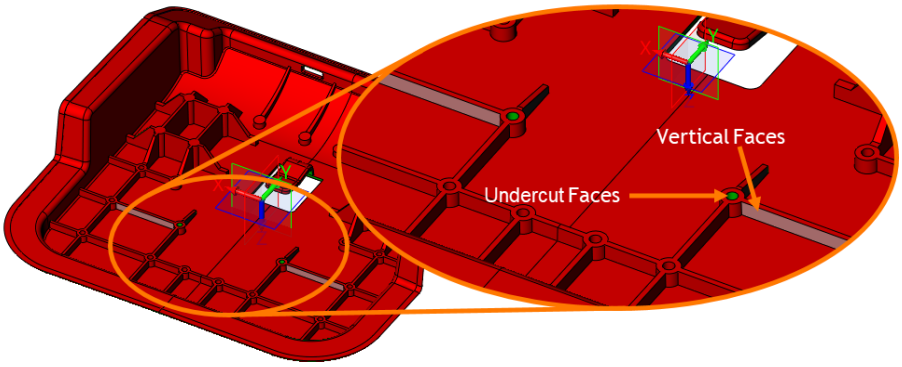


Figure 31 Result of Draft Angle Analysis

STEP 2 Use ***Draft*** command in ***Shape*** tab to adjust the draft angle to get a better productivity.

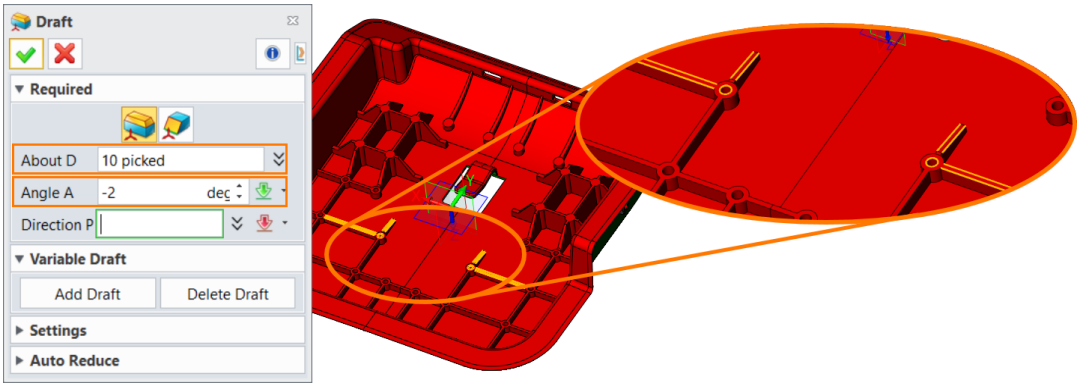


Figure 32 Adjust Draft Angle

STEP 3 Now change back to shape mode.

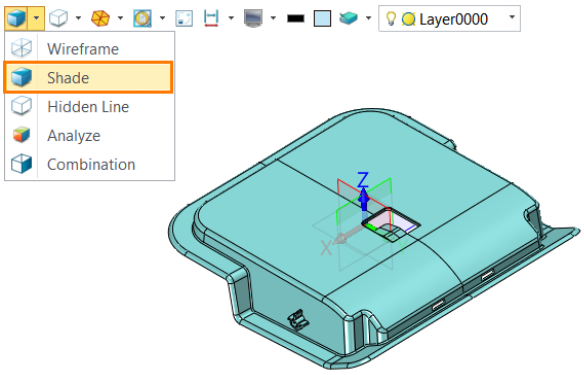


Figure 33 Display in Shape Mode

STEP 4 Use ***Shrink*** command in ***Mold*** tab to scale the model. Set shrink rate as ***1.006***.

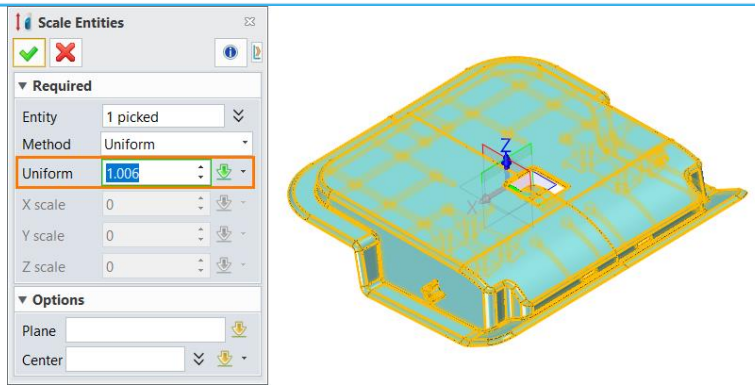


Figure 34 Set Shrink Rate

1.3 Mold Parting

1.3.1 Region Analysis and Definition

STEP 1 Create a new part named ***003_Parting.Z3***.

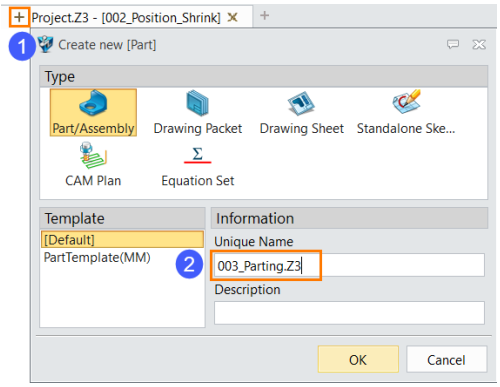


Figure 35 Create “003_Parting.Z3”

STEP2 Use ***Import*** command in ***Mold*** tab to import ***002_Position_Shrink***.

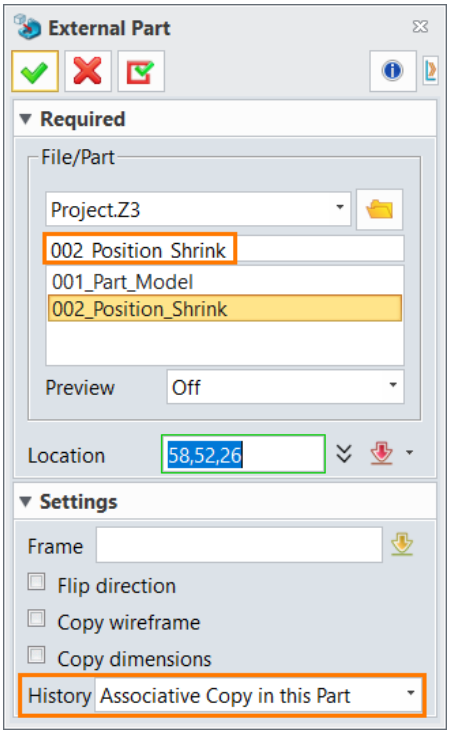


Figure 36 Import “002_Position_Shrink”

STEP3 Use ***Region Analysis*** command in ***Mold*** tab. Then click ***Calculate***.

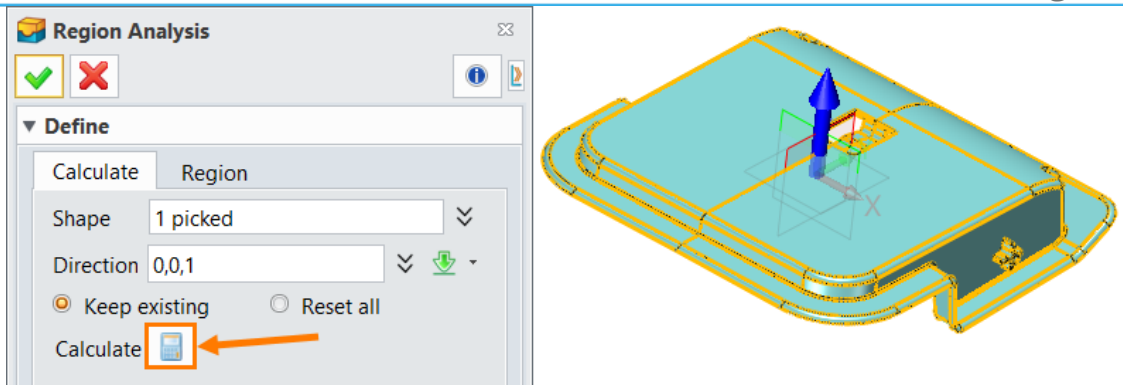


Figure 37 Calculate Regions

STEP4 The result shows as below. There are a lot of undefined faces. Click ***OK***.

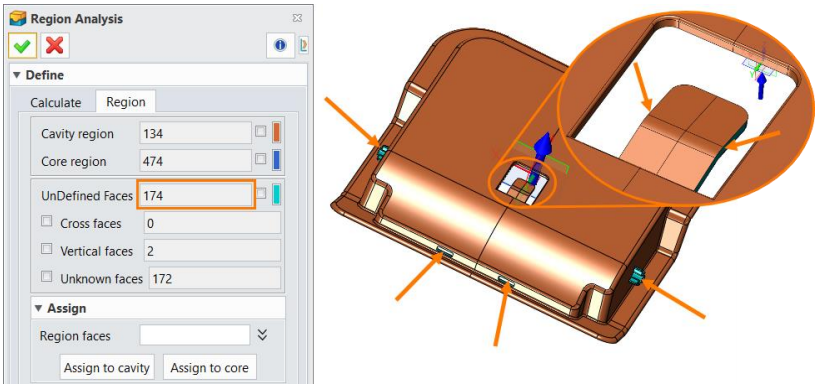


Figure 38 Figure 1 Region Analysis Result

STEP5 Pick ***Region faces*** as below, assign them to cavity

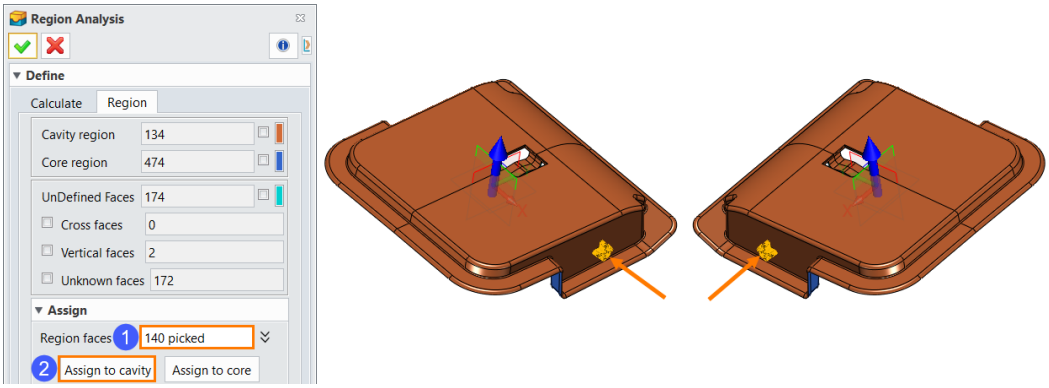


Figure 39 Assign Faces to Cavity

STEP6 Check ***Vertical faces*** and ***Unknown faces***, assign them to core. Click ***OK***.

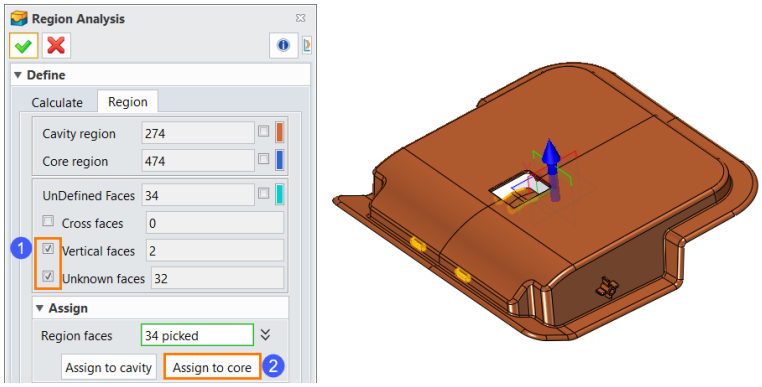


Figure 40 Assign Faces to Core

After that, all faces are assigned to cavity or core regions.

STEP7 Use ***Region Definition*** command in ***Mold*** tab.

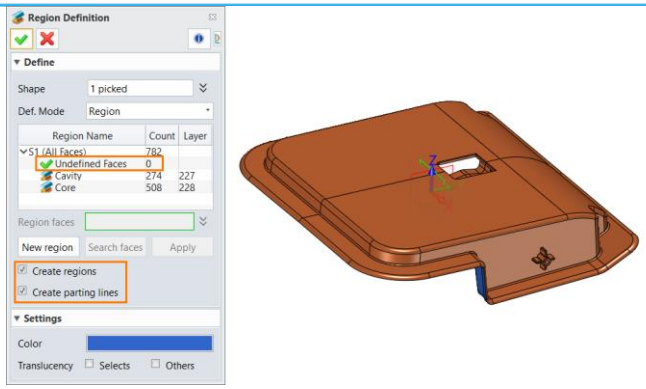


Figure 41 Region Definition

Make sure the count of undefined faces is 0. Then check ***Create regions*** and ***Create parting lines***. Click ***OK***, and you will get a result shown in Figure 42. The cavity & core surfaces and parting lines will be created in a folder of history tree.

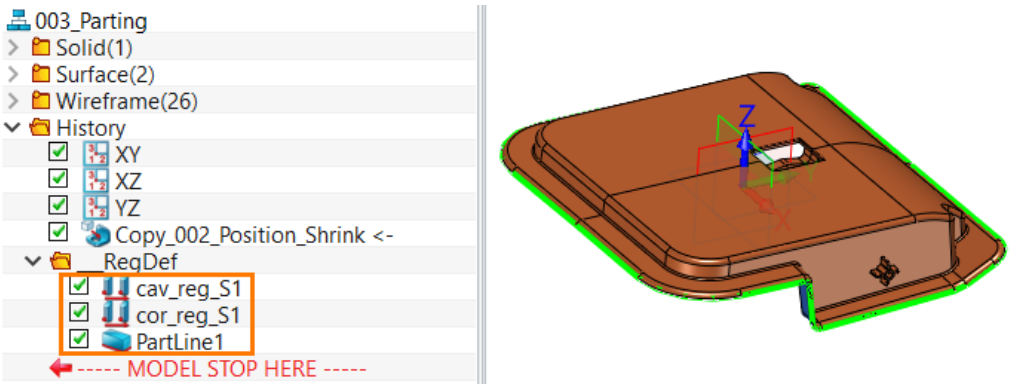


Figure 42 Create Cavity, Core and Parting Lines

1.3.2 Patch and Parting Faces

STEP1 Use ***Patch*** command in ***Mold*** tab to create patch faces for inner loops between cavity and core. Pick the Shape, then all loops will be shown in the list. Check ***Preview all patches***, all patches will be shown in preview.

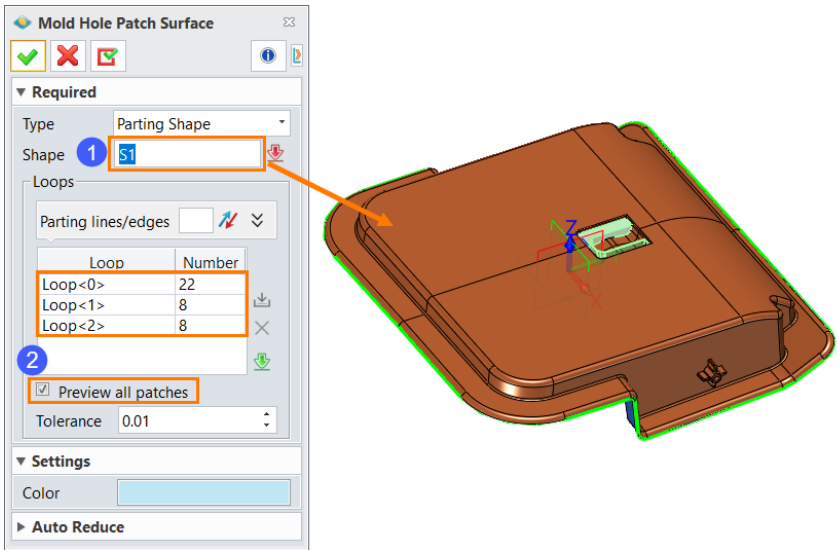


Figure 43 Create Patch Faces

Pick ***Loop<0>***, switch the side, the preview of ***Loop<0>*** is changed as Figure44.

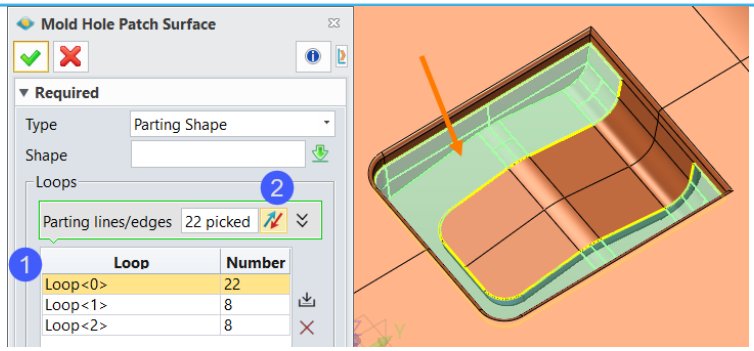


Figure 44 Switch Patch Face Side

STEP2 Click ***OK***. Result is shown in Figure45.

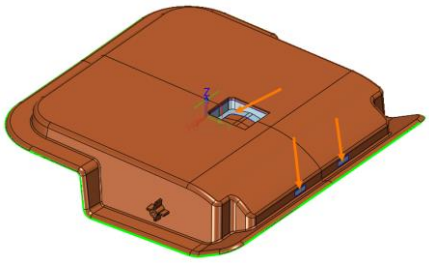


Figure 45 Result of Patch Faces

STEP3 Use ***Parting Face*** command to create parting faces.

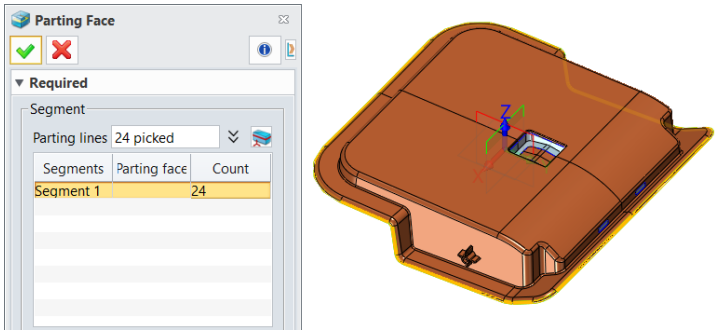


Figure 46 Create Parting Faces

There is the only one parting line segment. Next separate it into several segments for parting faces creation.

First click ***Edit guide line***.

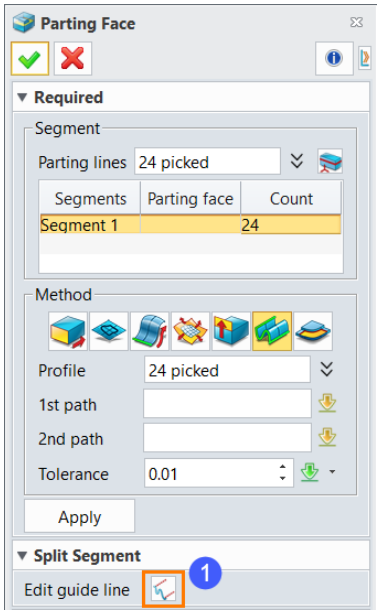


Figure 47 Split Segment by Guide Line

Then pick a guiding line, the direction and add in the list.

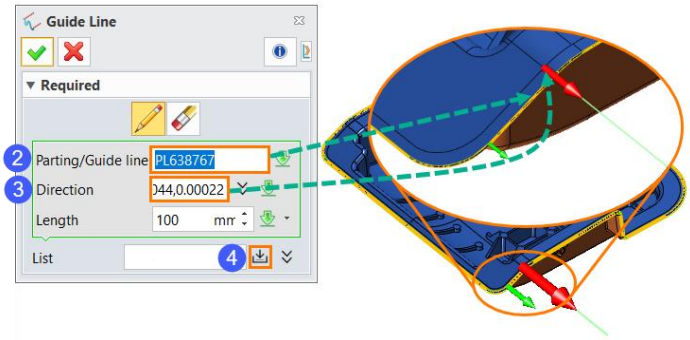


Figure 48 Define First Guide Line

Continue to create other guide lines.

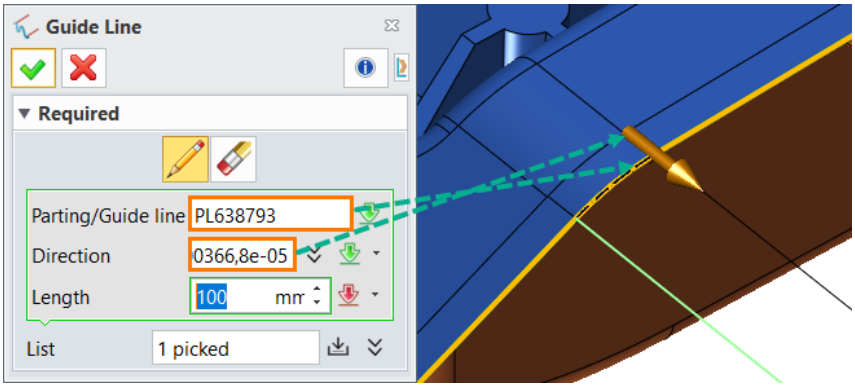


Figure 49 Define Second Guide Line

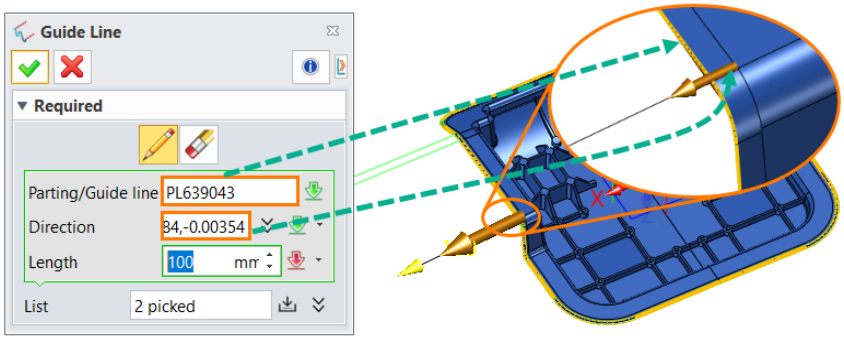


Figure 50 Define Third Guide Line

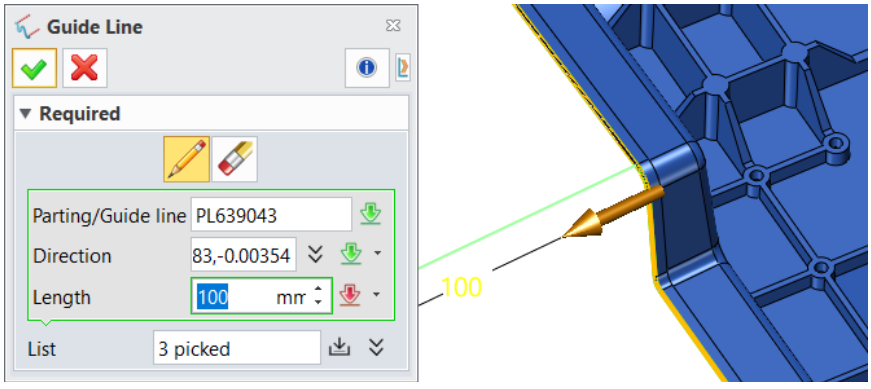


Figure 51 Define Fourth Guide Line

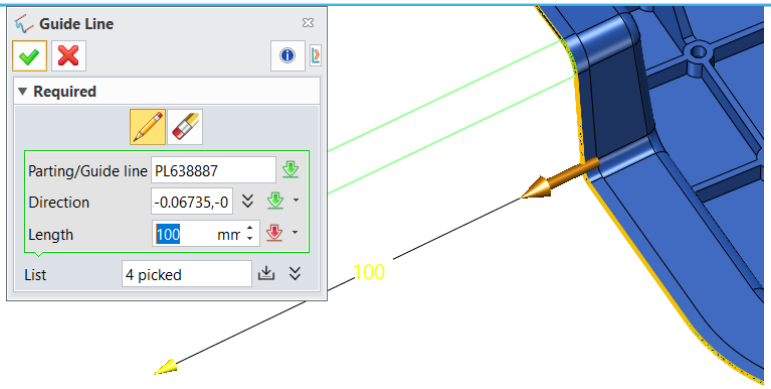


Figure 52 Define Fifth Guide Line

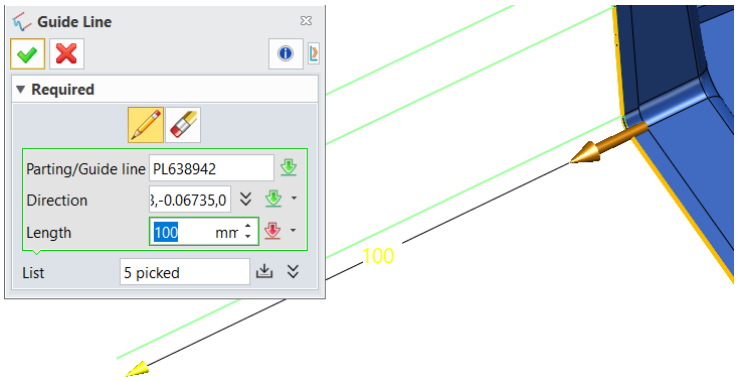


Figure 53 Define Sixth Guide Line

Now the preview shows as figure53.

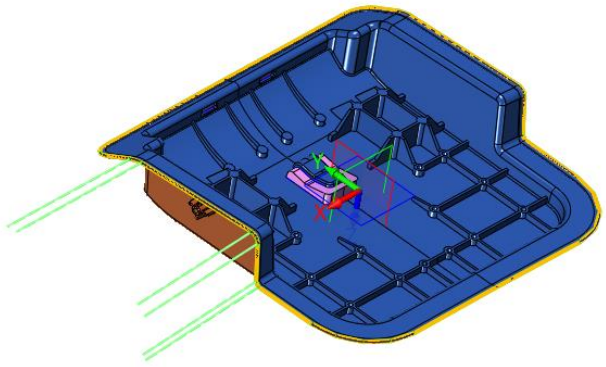


Figure 54 Guide Lines on Half of the Model

Create guide lines on the other half in the same way. There are 12 guide lines in total.

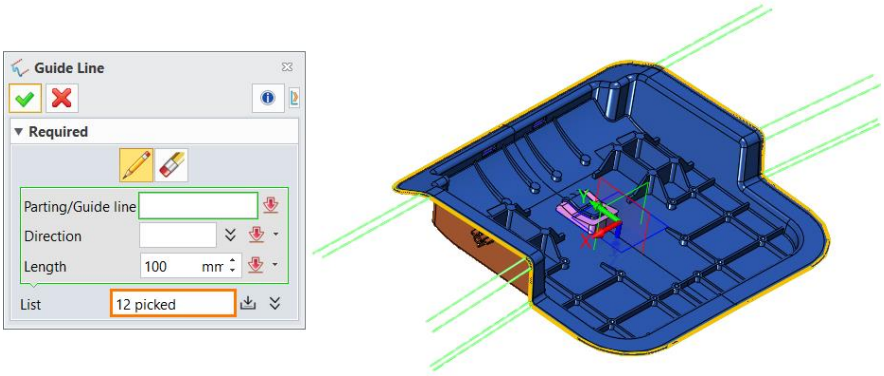


Figure 55 Finishing Guide Line Creation

STEP4 Click ***OK***. Now parting lines have been separated into 12 segments.

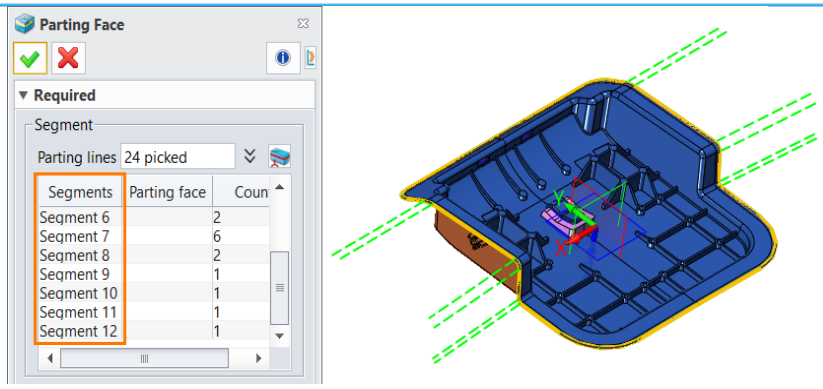


Figure 56 Segments of Parting Lines

STEP5 Now start to create parting face for each segment.

Select ***Segment 1***, use ***extend shape***, then click ***Apply***, then parting face for this segment will be created.

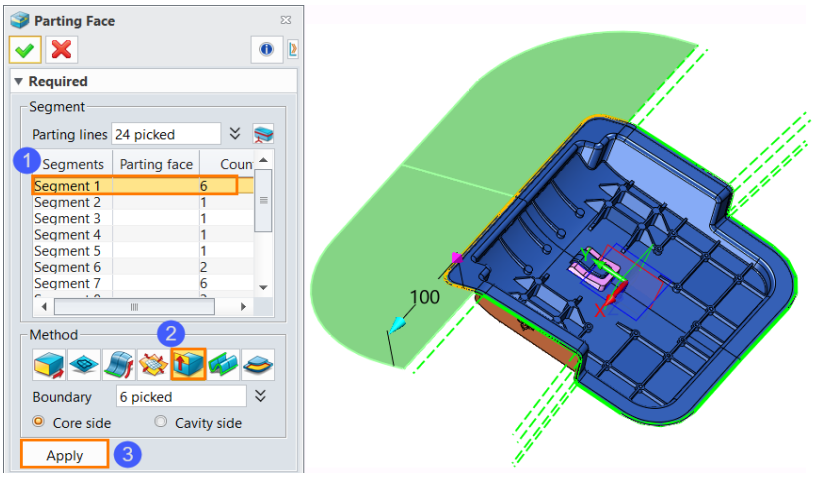


Figure 57 Create Parting Face for Segment 1

Select ***Segment 3***, use ***Trimmed Plane by Boundaries***, then click ***Apply***.

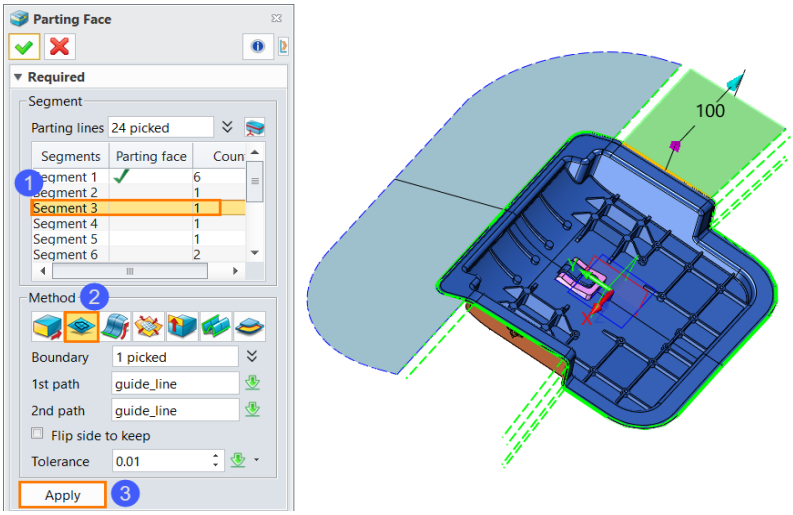


Figure 58 Create Parting Face for Segment 3

Select ***Segment 5***, use ***Trimmed Plane by Boundaries***, then click ***Apply***.

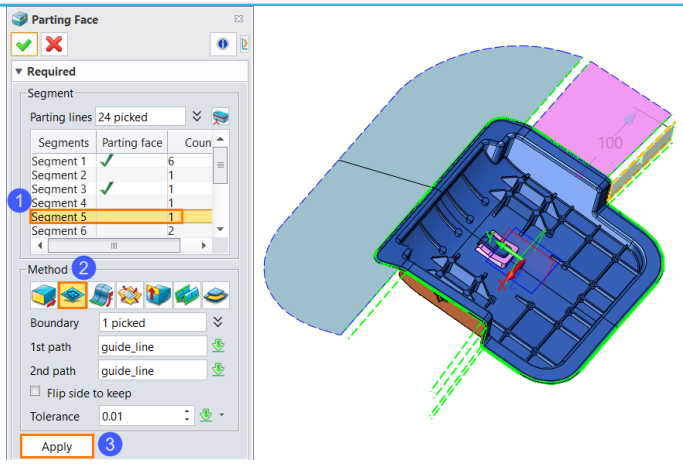


Figure 59 Create Parting Face for Segment 5

Select ***Segment 7***, use ***Trimmed Plane by Boundaries***, then click ***Apply***.

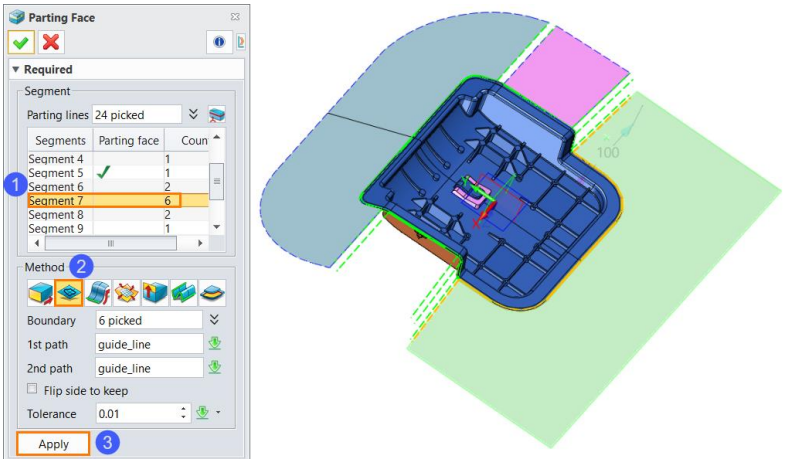


Figure 60 Create Parting Face for Segment 7

Select ***Segment 9***, use ***Trimmed Plane by Boundaries***, then click ***Apply***.

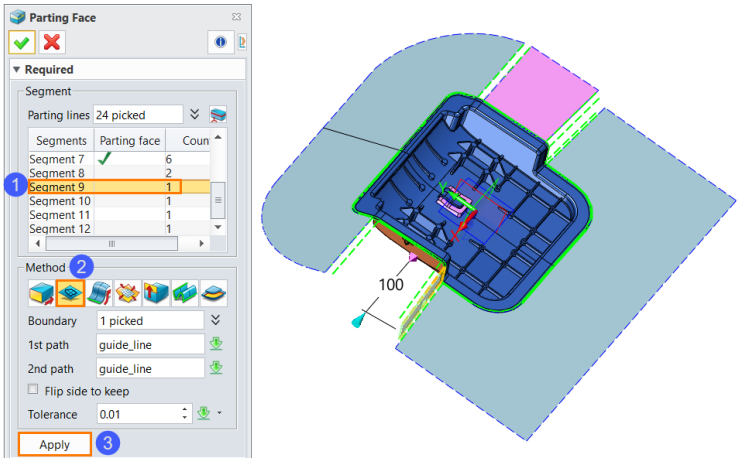


Figure 61 Create Parting Face for Segment 9

Select ***Segment 9***, use ***Trimmed Plane by Boundaries***, then click ***Apply***.

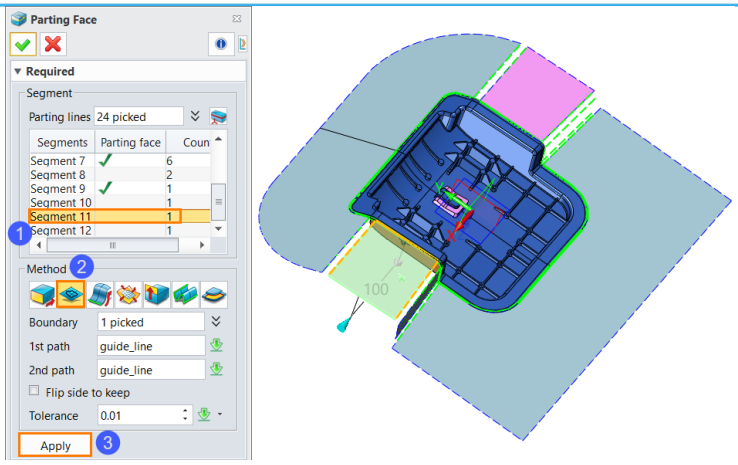


Figure 62 Create Parting Face for Segment 11

Select ***Segment 2***, use ***extend shape***, then click ***Apply***.

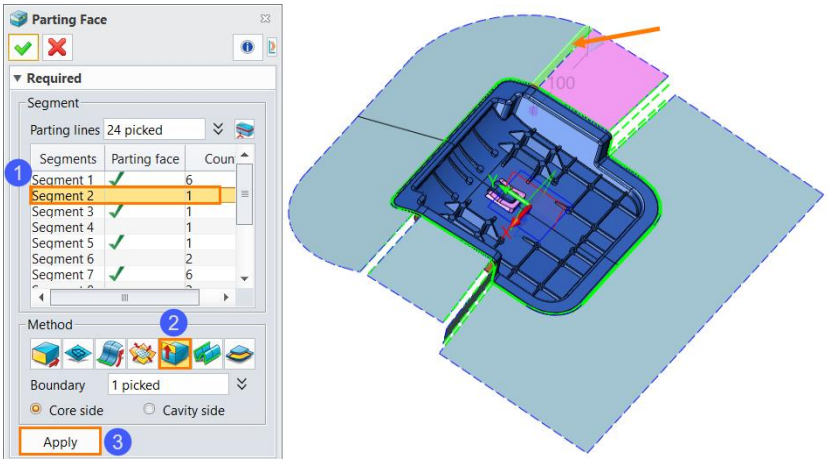


Figure 63 Create Parting Face for Segment 2

Select ***Segment 4***, use ***extend shape***, then click ***Apply***.

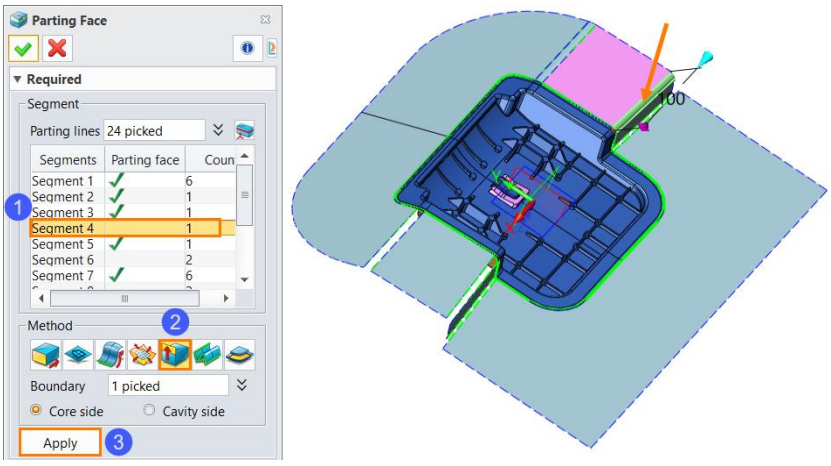


Figure 64 Create Parting Face for Segment 4

Select ***Segment 6***, use ***extend shape***, change the extend value to ***140***, then click ***Apply***.

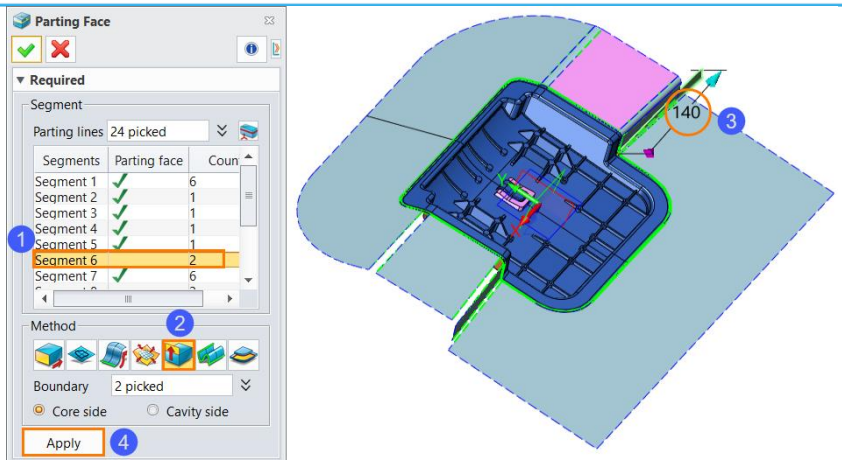


Figure 65 Create Parting Face for Segment 6

Select ***Segment 8***, use ***extend shape***, change the extend value to ***140***, then click ***Apply***.

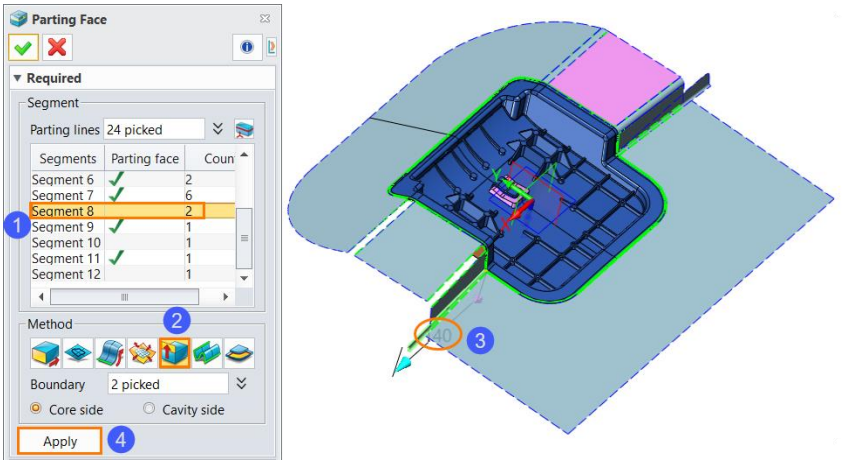


Figure 66 Create Parting Face for Segment 8

Select ***Segment 10***, use ***extend shape***, then click ***Apply***.

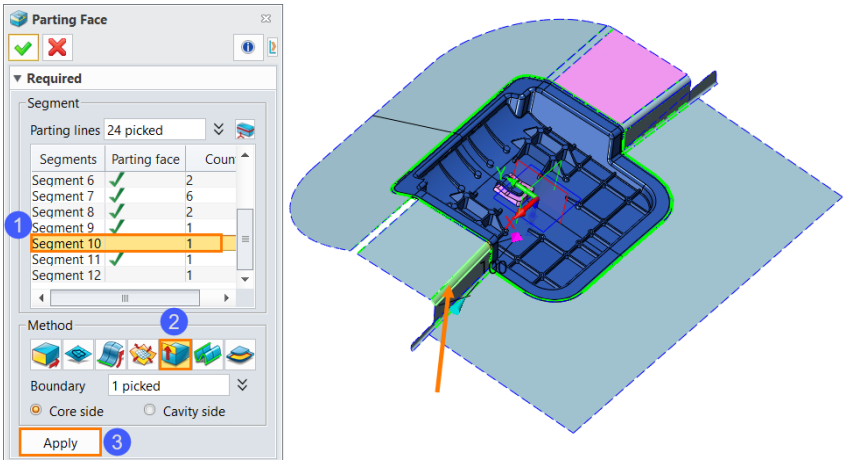


Figure 67 Create Parting Face for Segment 10

Select ***Segment 12***, use ***extend shape***, then click ***Apply***.

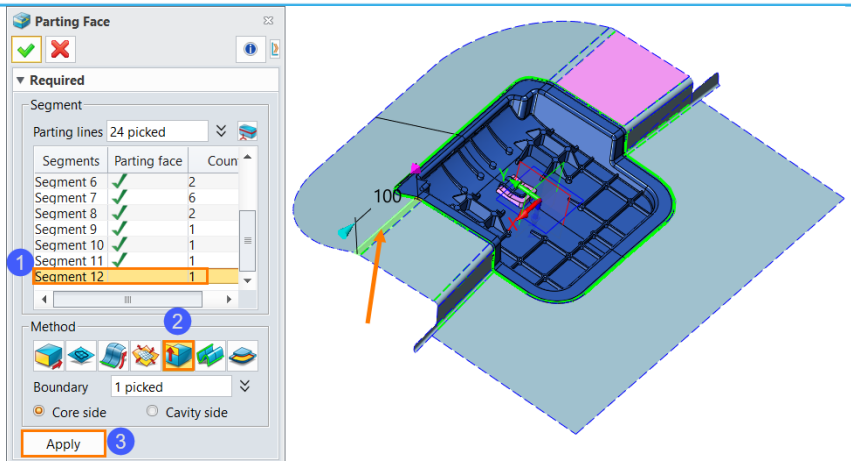


Figure 68 Create Parting Face for Segment 12

After clicking ***OK***, all parting faces are shown in history.

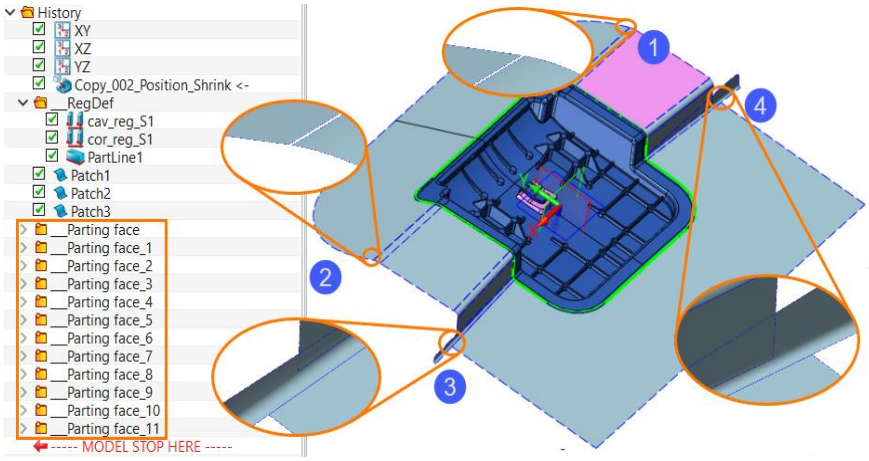


Figure 69 Parting Face Folders in History

As shown in figure69, there are some gaps or overlaps between different parting faces.

STEP6 Fill gaps and trim overlaps.

Use ***Blend Face*** command to create a blend face to fix gap 1.

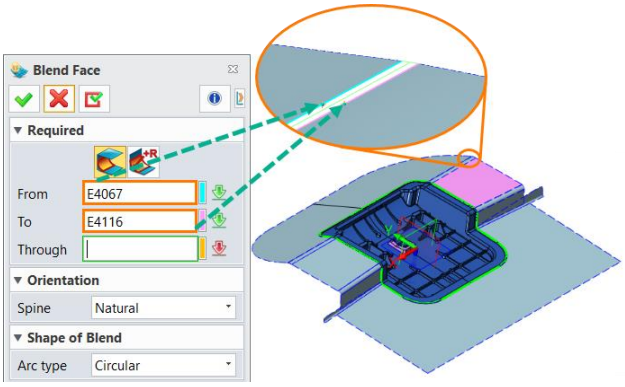


Figure 70 Fie Gap 1

Use ***Blend Face*** command to create a blend face to fix gap 2.

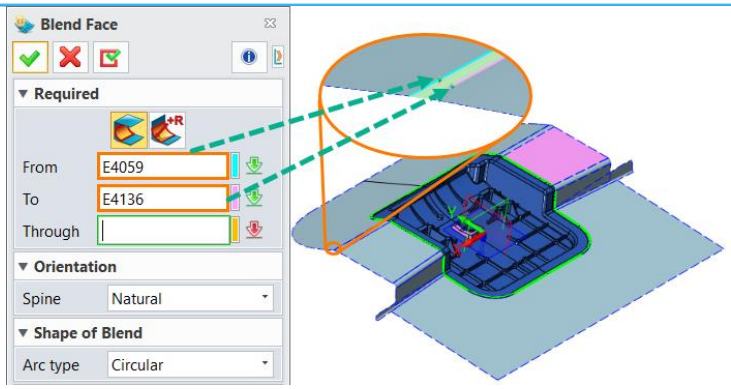


Figure 71 Fix Gap 2

Use ***Trim to Curves*** command to trim overlap 3.

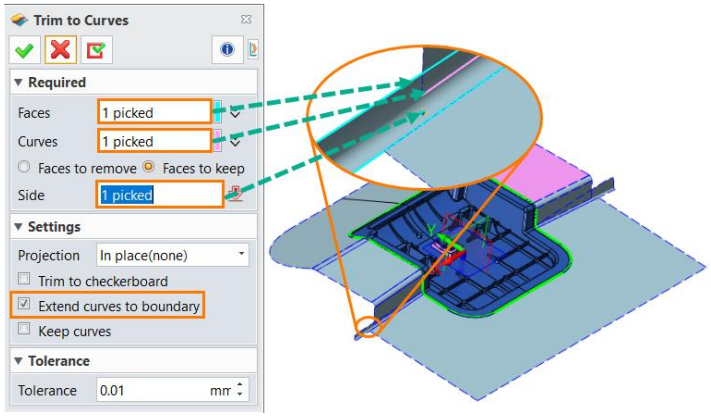


Figure 72 Trim Overlap 3

Use ***Trim to Curves*** command to trim overlap 4.

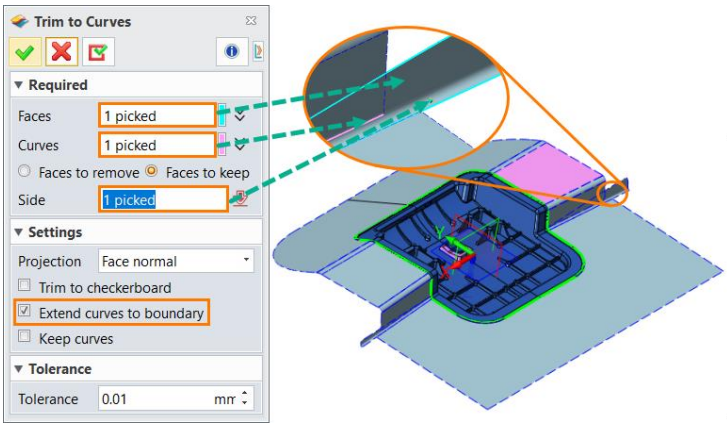


Figure 73 Trim Overlap 4

1.3.3 Workpiece and Trim

STEP1 Use ***Workpiece*** command in ***Mold*** tab to create a workpiece. Set parameters as below.

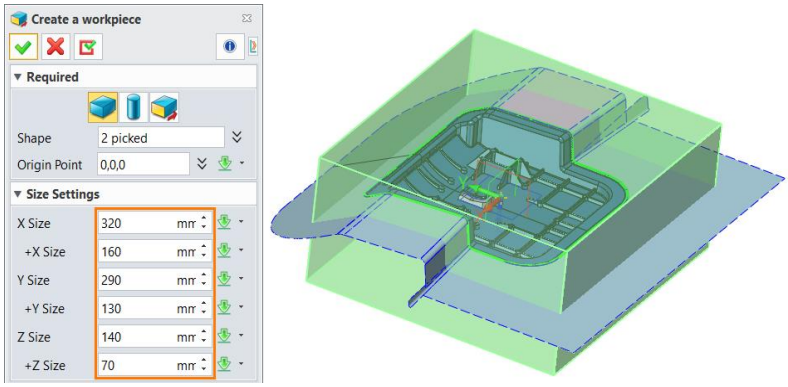


Figure 74 Create Workpiece

STEP2 Use ***Trim*** command in ***Mold*** tab to trim the model.

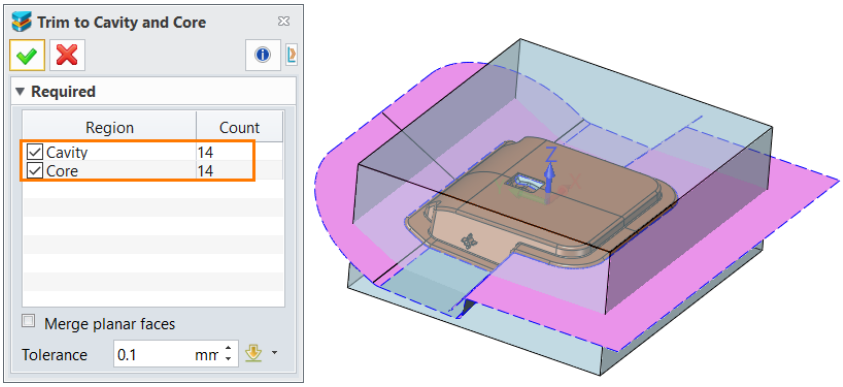


Figure 75 Create Workpiece

It will automatically recognize all regions including cavity and core.

Click OK, and the result shows as below.

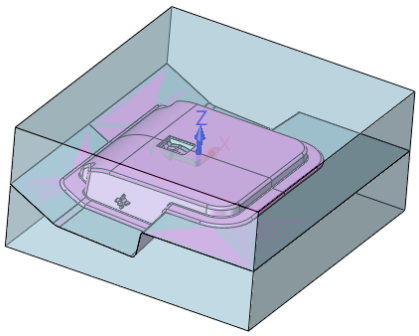


Figure 76 Trim Result

STEP3 Use ***Extrude*** command in ***Shape*** module to extrude a planar face for cavity. Set parameters as below.

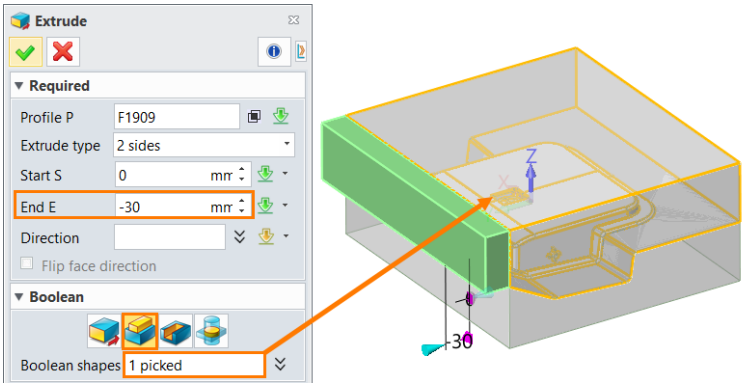


Figure 77 Extrude Planar Face for Cavity

STEP4 Use ***Extrude*** command in ***Shape*** module to extrude a planar face for core. Parameters show as below.

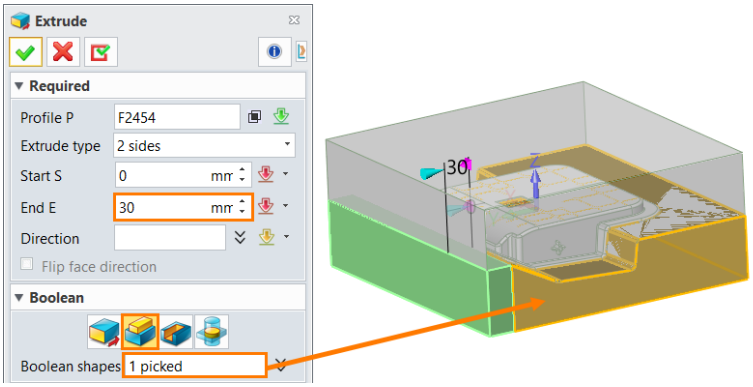


Figure 78 Extrude Planar Face for Core

STEP5 Use ***Fillet*** command in ***Shape*** tab to create fillets on vertical edges of cavity and core. Set the radius of fillet as ***30***

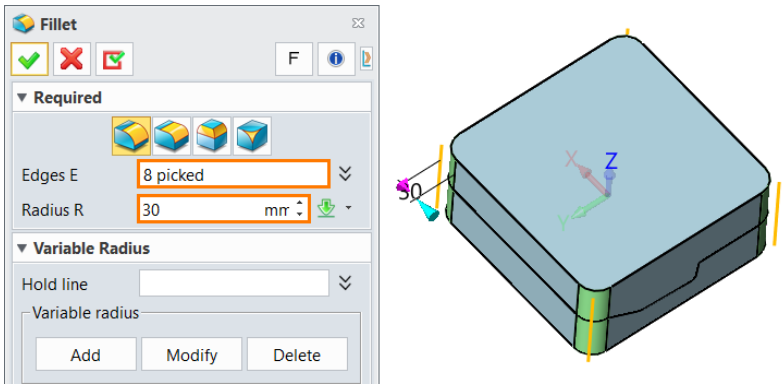


Figure 79 Add Fillets on Vertical Edges

Click ***OK***, the result shows as below.

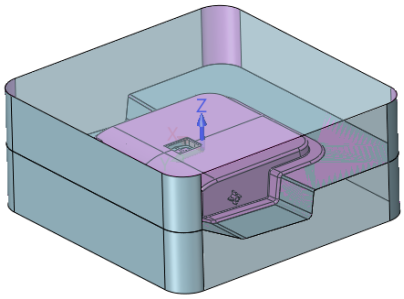


Figure 80 Fillet Result

STEP6 Pick the new faces of ***Cavity*** and ***Core***, right click then select ***Face Attribute***.

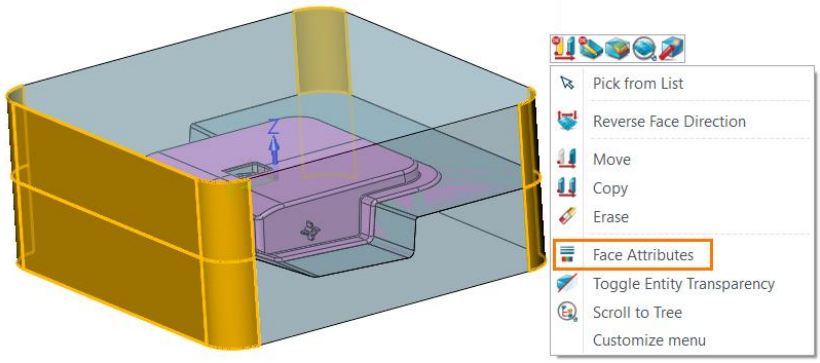


Figure 81 Set Face Attribute

Set transparency as 50%.

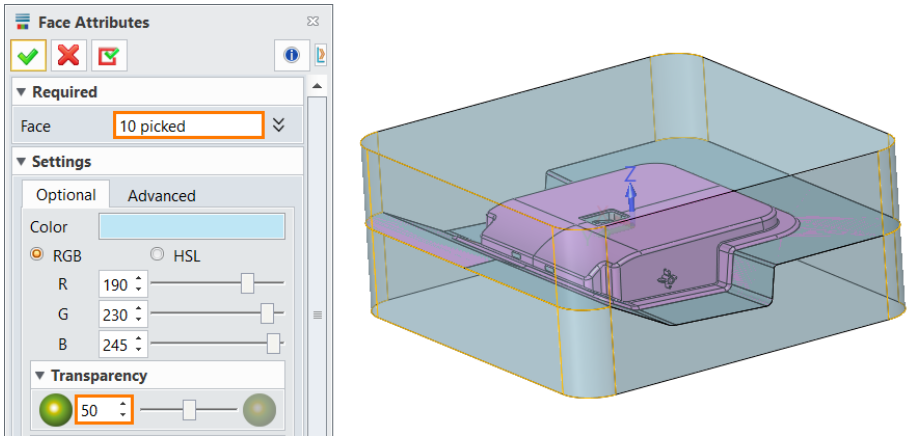


Figure 82 Set Transparency

So far, the model has been successfully parted

1.4 Other Molding Parts Creation

After parting the mold, let's create inserts, Sliders and lifters.

STEP1 Use ***Geom to Part*** command in ***Assembly***, select ***cavity*** shape, input ***004_Fixed_Half***, pick Extract mode as ***Associative extract***.

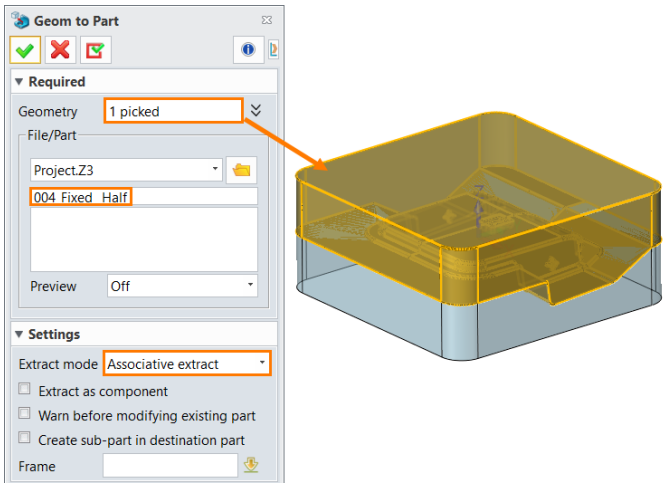


Figure 83 Extract "Cavity" to "004_Fixed_Half"

STEP2 Use ***Geom to Part*** command in ***Assembly***, select ***core*** shape, input ***005_Moving_Half***, pick Extract mode as ***Associative extract***.

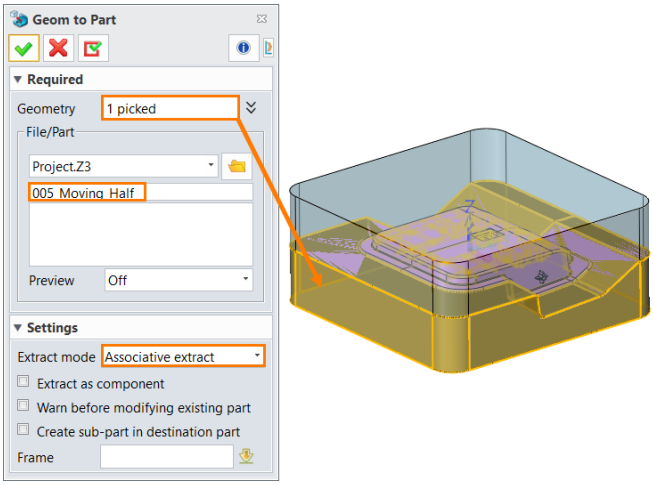


Figure 84 Extract "Core" to "005_Moving_Half"

STEP3 Use ***Geom to Part*** command in ***Assembly***, select ***core*** shape, input ***006_Epin_Cutter***, pick Extract mode as ***Associative extract***.

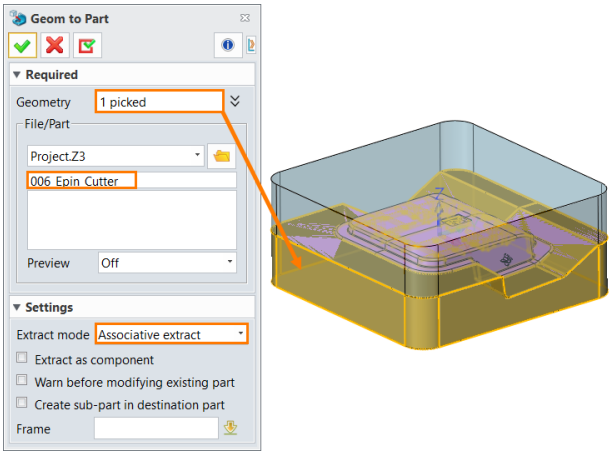


Figure 85 Extract "Core" to "006_Epin_Cutter"

STEP4 Use ***Exit*** to return to the object manager. Objects are listed as below.

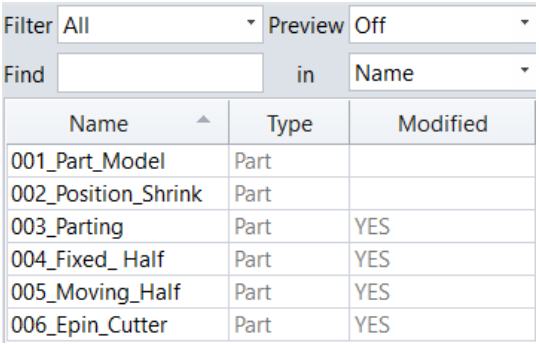


Figure 86 Objects Listed in Manager

STEP5 Go to ***006_Epin_Cutter***, use ***Erase*** command to create some faces.

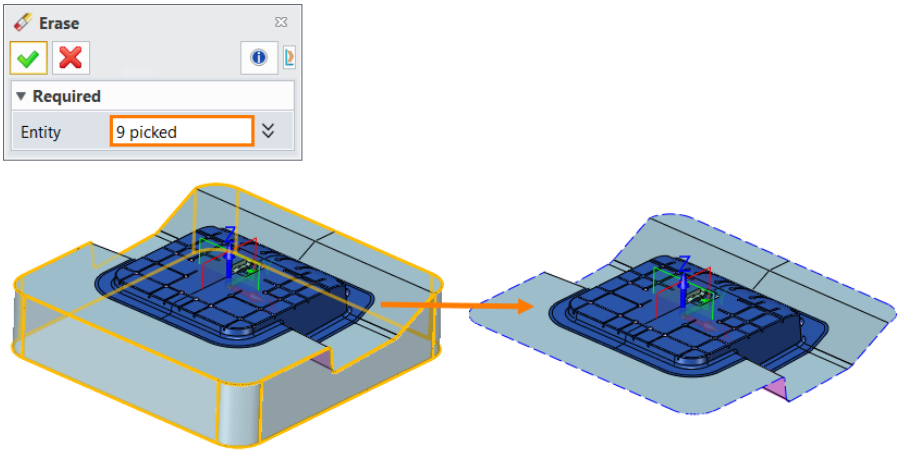


Figure 87 Erase Some Faces

STEP6 Use ***Reverse Direction*** command in ***Free Form*** tab to reverse the direction of the shape.

Switch the ***Filte***r to ***Shape***, then pick the shape.

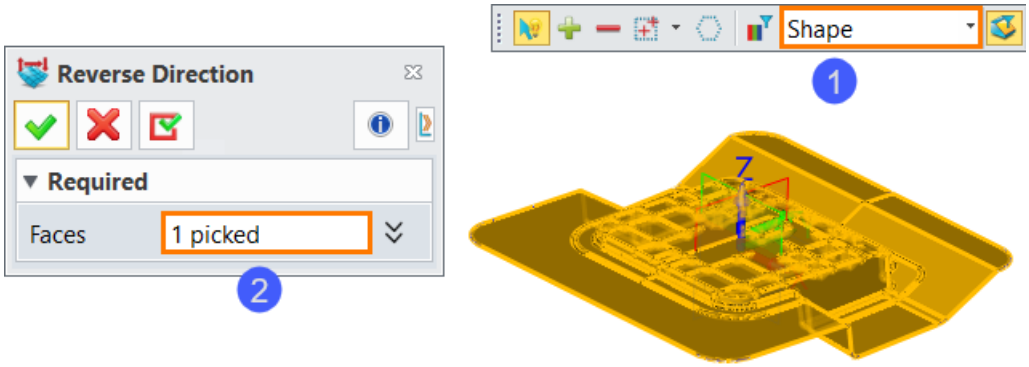


Figure 88 Figure 2 Reverse Direction

The result shows as below.

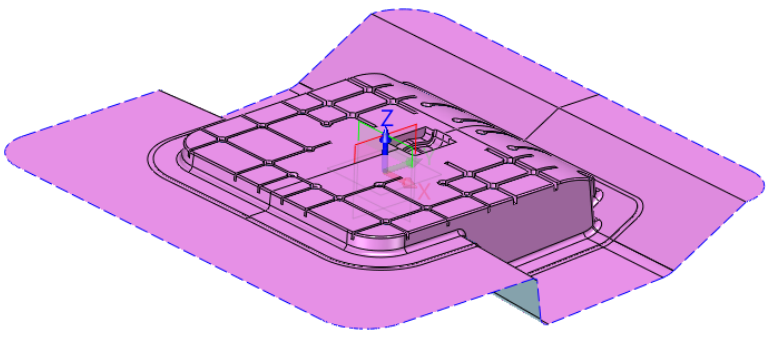


Figure 89 Result of “Reverse Direction”

Use ***Merge Faces*** command in ***Free Form*** tab to merge some faces as below.

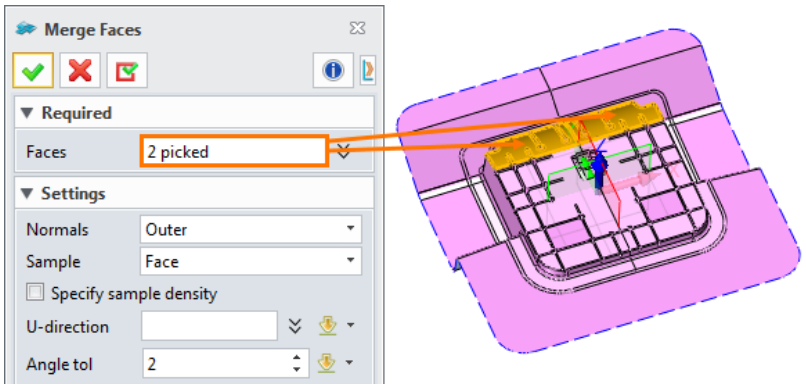


Figure 90 Merge Faces 1

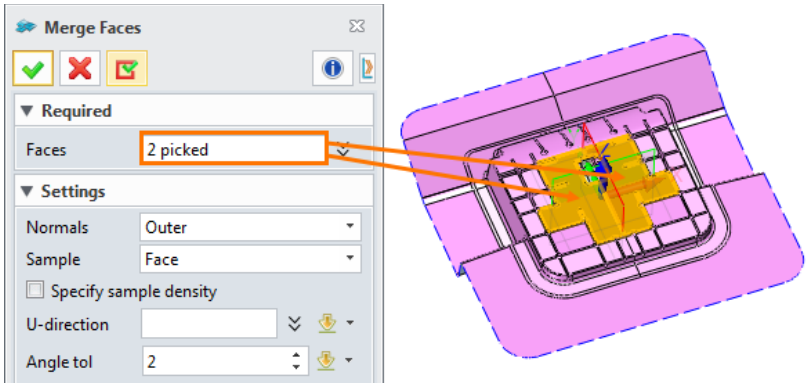


Figure 91 Merge Faces 2

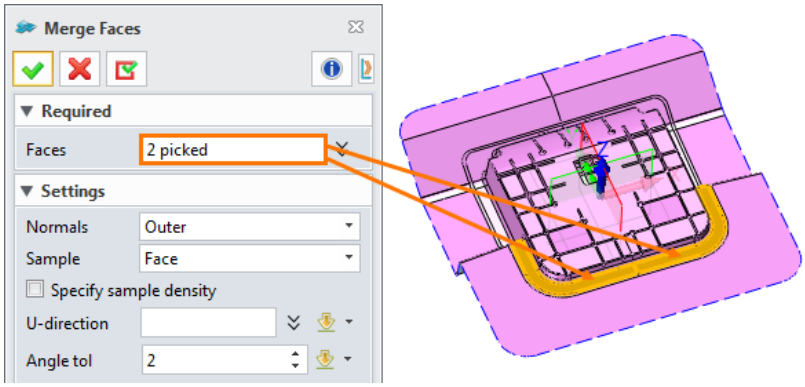


Figure 92 Merge Faces 3

1.4.1 Create Inserts

Now let's create inserts.

STEP 01 Go to ***005_Moving_Half***. Use ***Sketch*** command in ***Shape*** tab to create a sketch based on XY Plane.

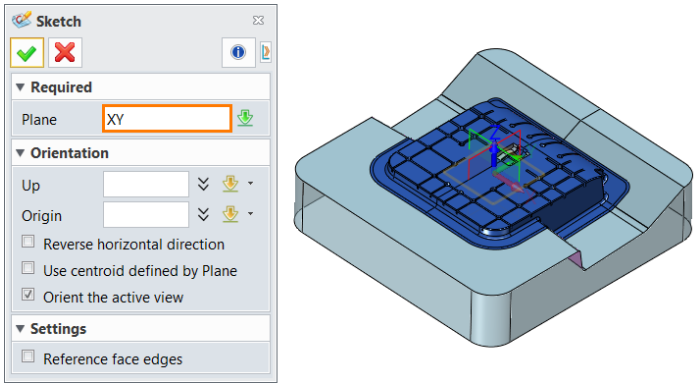


Figure 93 Create a Sketch Based on XY Datum

Draw a sketch as below.

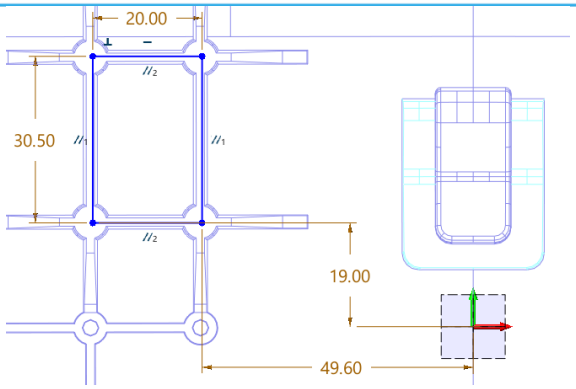


Figure 94 Well-defined Sketch

STEP 02 Use ***Extrude*** command in ***Shape*** tab to extrude the sketch.

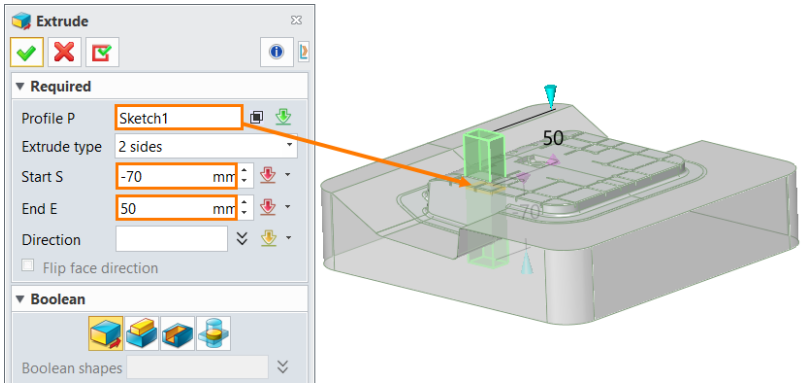


Figure 95 Extrude the sketch

STEP 03 Use ***Blank*** command to hide the core shape.

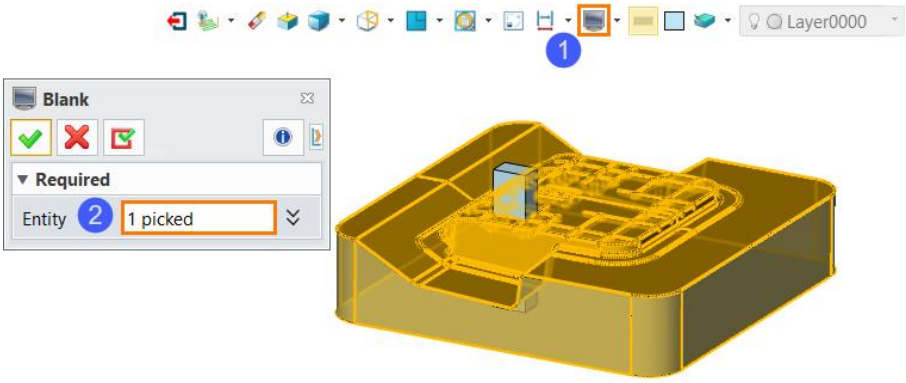


Figure 96 Blank the Core Shape

STEP 04 Use ***Extrude*** command again to create a base for the insert.

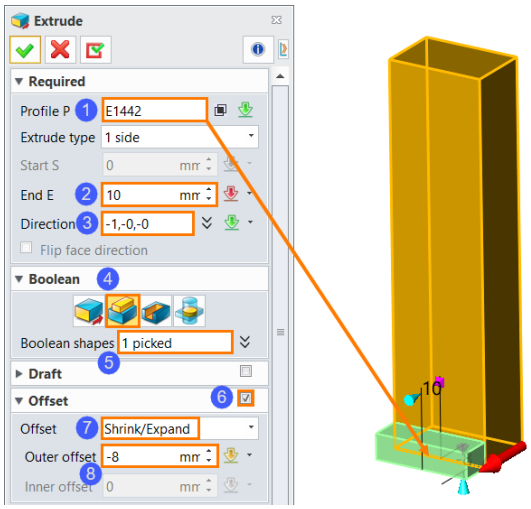


Figure 97 Extrude an Edge of the Bottom Face

STEP 05 Use ***Unblank All*** command to show all shapes

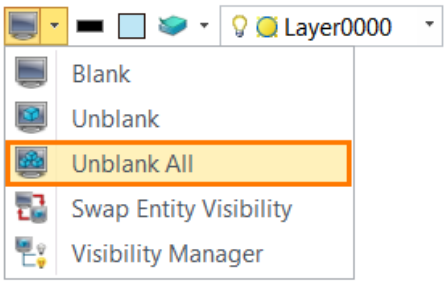


Figure 98 Unblank All Entities

STEP 06 Use ***Divide*** command in ***Shape*** tab to divide the insert shape from the core.

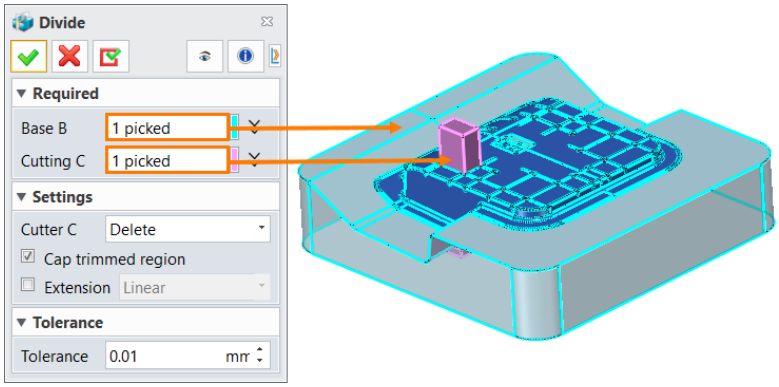


Figure 99 Divide the Shape from the Core

Result is shown as below.

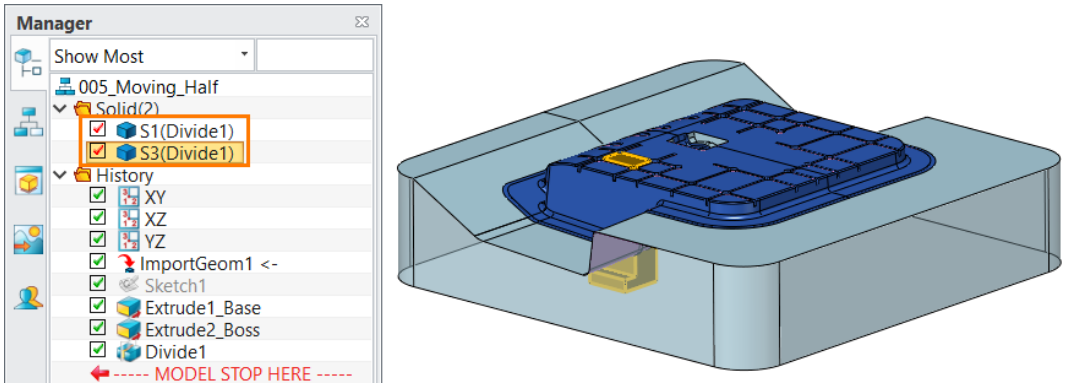


Figure 100 Result of Division

STEP 07 Right click on the insert shape, then select ***Face Attribute***.

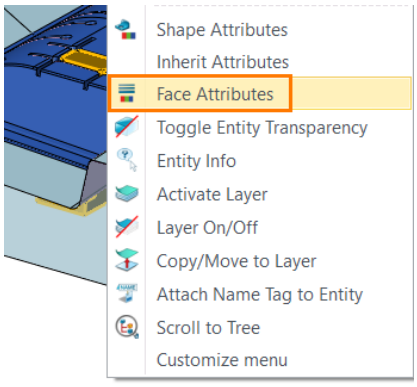


Figure 101 Set Face Attributes

STEP 08 Change the color as below.

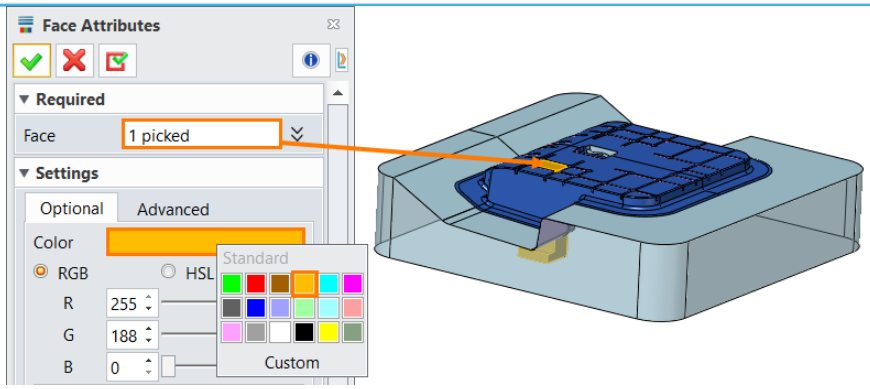


Figure 102 Change the Color of the Insert

STEP 09 Use ***Mirror Geometry*** command in ***Shape*** tab to mirror the insert by YZ datum.

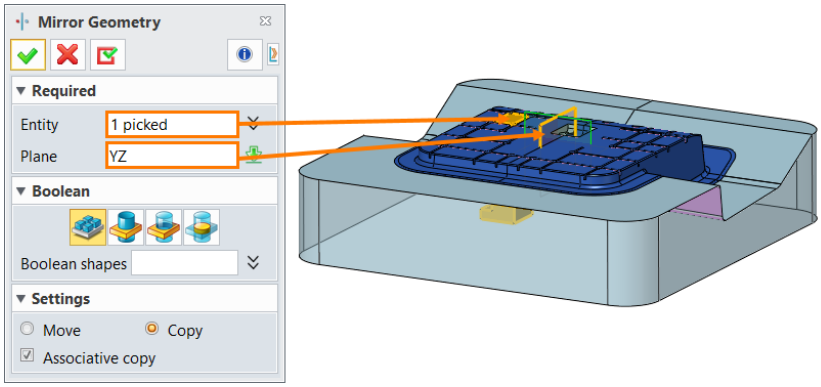


Figure 103 Mirror the Insert

Result is shown as below.

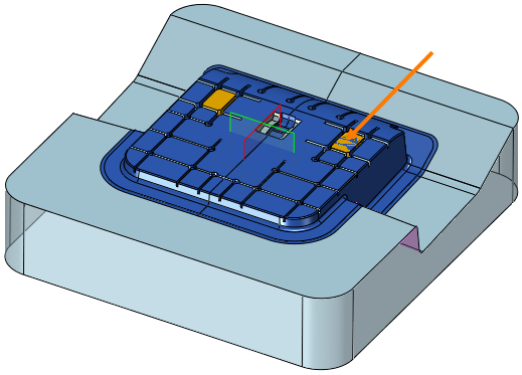


Figure 104 Mirror Result

STEP 10 Use ***Remove Shape*** command to create a pocket for the insert.

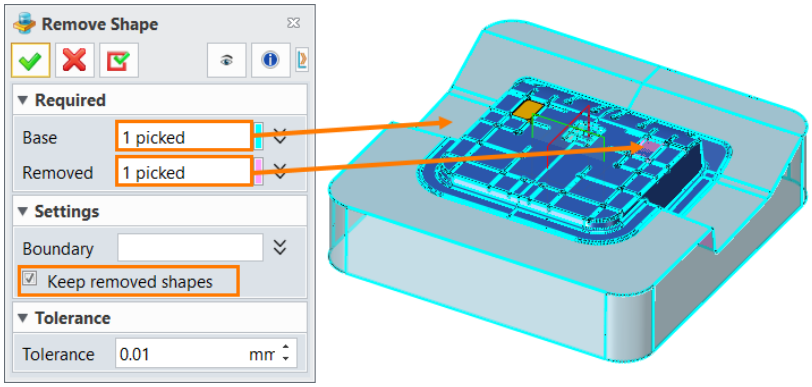


Figure 105 Create a Pocket for the Insert

At last, the inserts are finished as below.

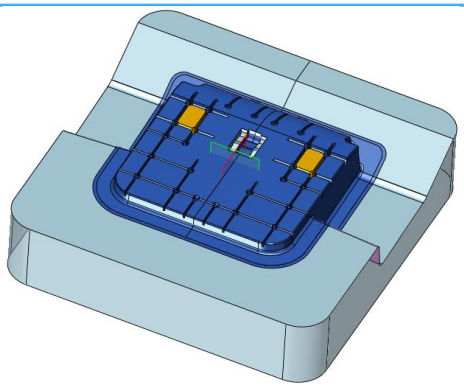


Figure 106 Pocket Result

1.4.2 Create Lifters

We need to create lifters for the lower undercut areas.

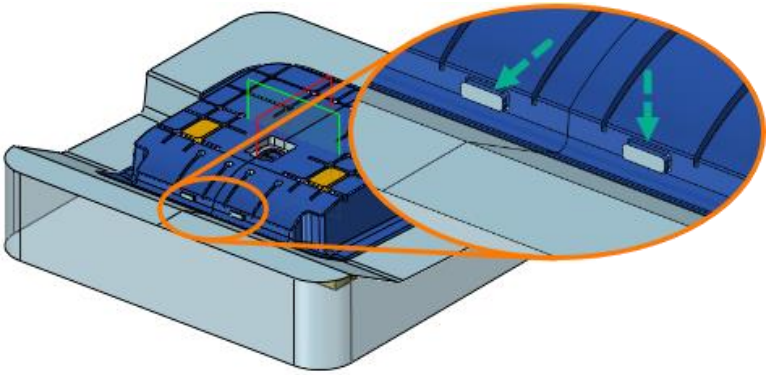


Figure 107 Inner Undercut Area

STEP 01 Use ***Datum*** command in ***Shape*** tab to create a new datum, then select the YZ plane, and set the origin point to the middle of line as below.

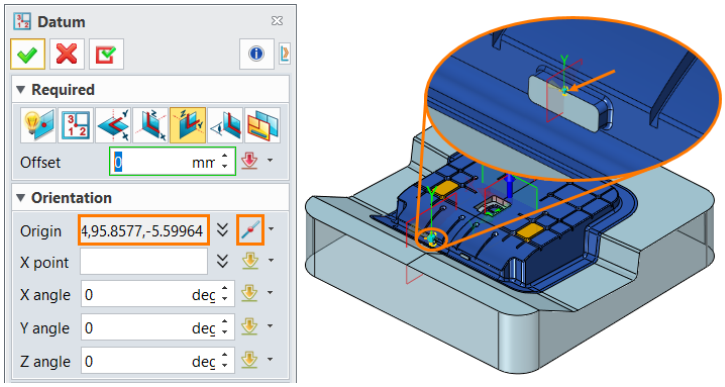


Figure 108 Create a New Datum

STEP 02 Create a sketch based on plane1.

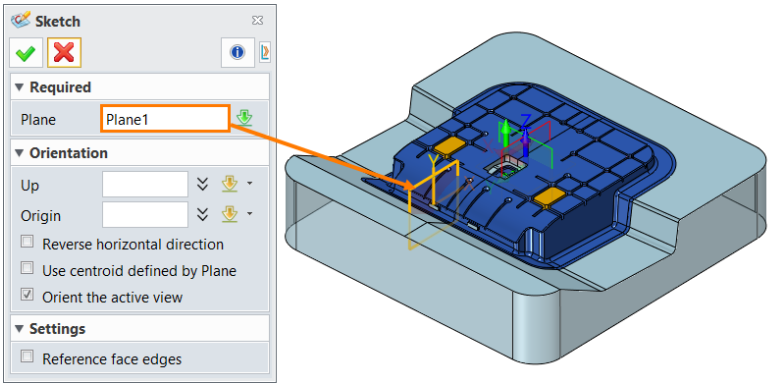


Figure 109 Create a Sketch Based on the New Plane

Draw the sketch as below.

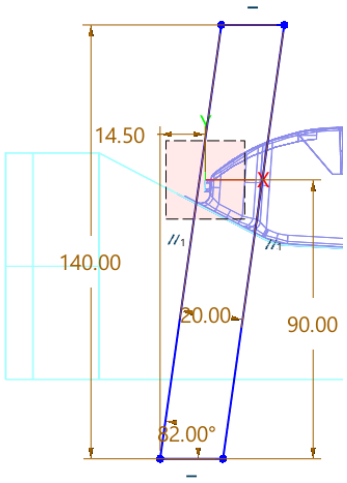


Figure 110 Well-defined Sketch for the Lifter

STEP 03 Exit the sketch, then use ***Extrude*** command to extrude the sketch. Set parameters as below.

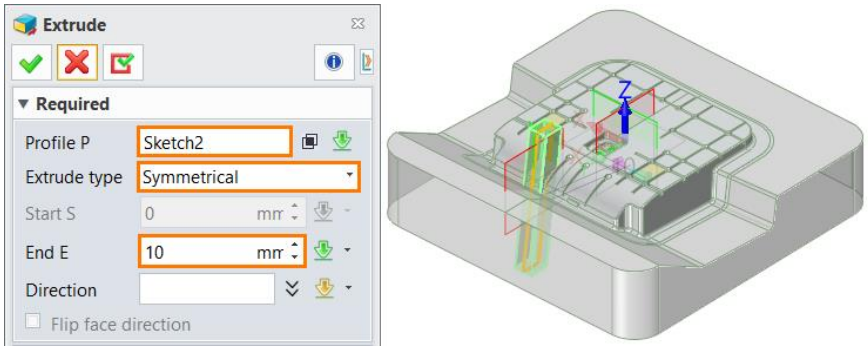


Figure 111 Extrude the Sketch

Change the color of the ***Lifter*** shape as below.

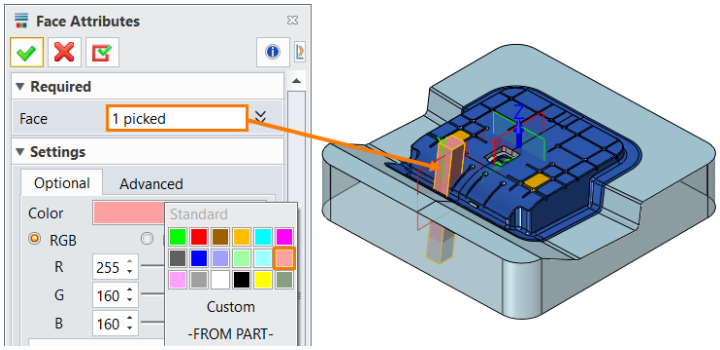


Figure 112 Set Face Color for the Lifter

Use ***Mirror Geometry*** command to mirror the “Lifter” shape based on YZ Datum.

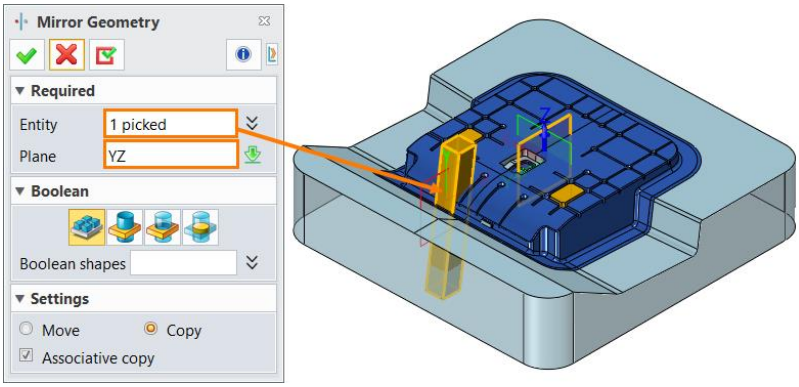


Figure 113 Mirror the Lifter

The result shows as below.

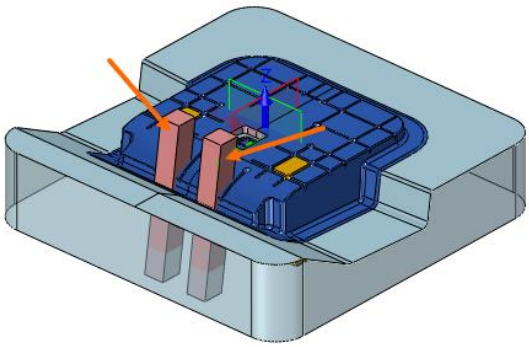
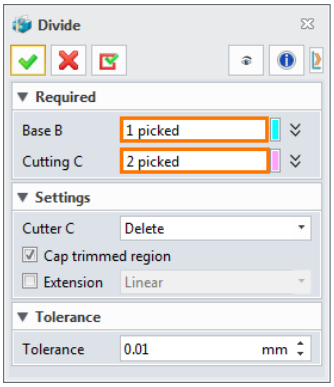


Figure 114 Mirror Result

STEP 04 Use ***Divide*** command to divide the lifters from the ***Core*** shape.



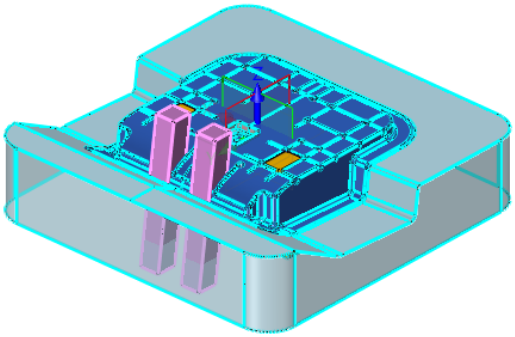
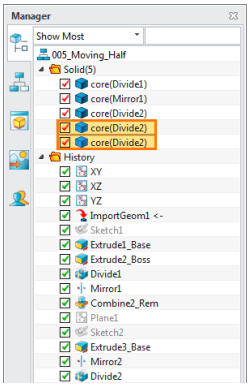


Figure 115 Divide Lifters from the Core

The result shows as below.



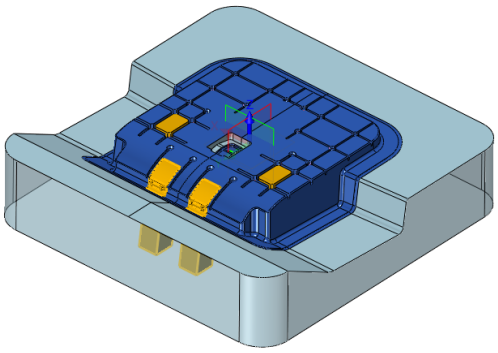
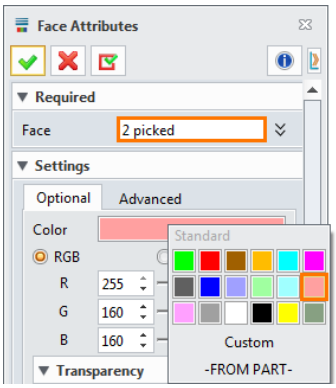


Figure 116 Division Result

Change the color of ***Lifter*** shapes.



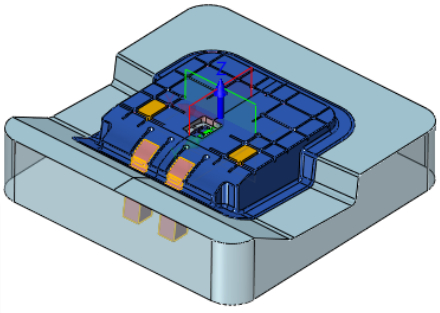


Figure 117 Set Faces Attributes for the Lifters

STEP 05 Use ***Geom to Part*** command in ***Assembly*** tab to extract the core as a part.

Pick the ***Core*** shape, create a new object named ***201_Core***, then select ***Associate extract*** mode.

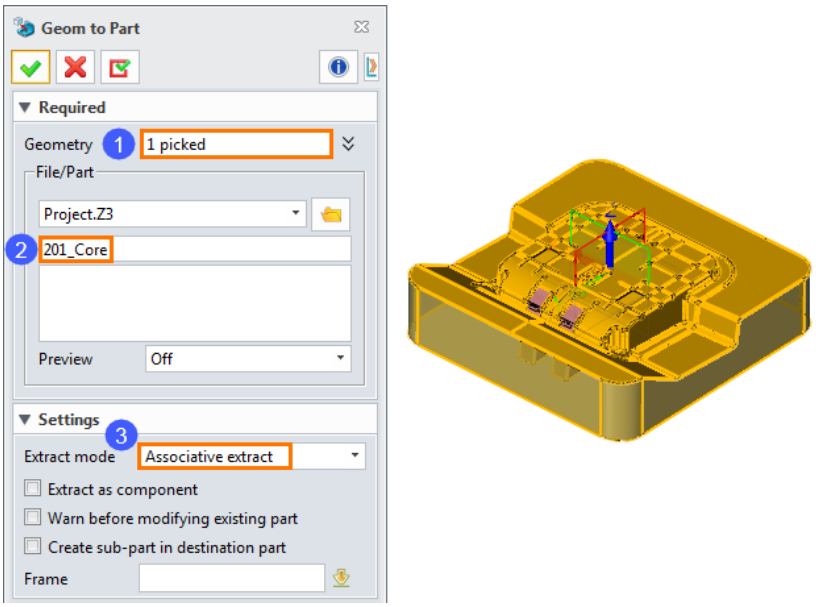


Figure 118 Extract the Core Shape to a New Part

Extract the two inserts and two lifters as ***202_Insert1***, ***203_Insert2***, ***204_Lifter1*** and ***205_Lifter2*** with the same operations.

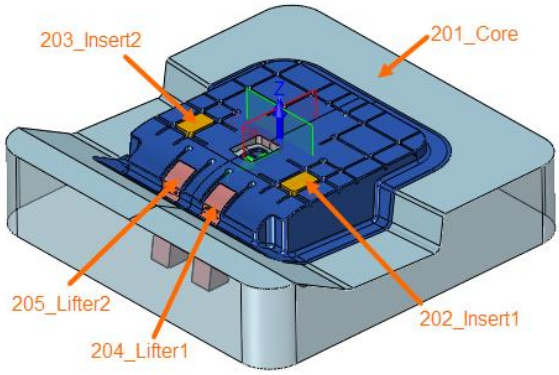


Figure 119 Extract All Shapes

After these operations, you can find them in the root manager.

Name	Type	Modified
001_Part_Model	Part	
002_Position_Shrink	Part	
003_Parting	Part	
004_Fixed_ Half	Part	
005_Moving_Half	Part	YES
006_Epin_Cutter	Part	
201_Core	Part	
202_Insert1	Part	YES
203_Insert2	Part	YES
204_Lifter1	Part	YES
205_Lifter2	Part	YES

Figure 120 New Parts Listed in the Root Manager

1.4.3 Create Sliders

STEP 01 Go to ***004_Fixed_ Half***, use ***Sketch*** command to create a sketch based on XY Datum.

The sketch shows as below.

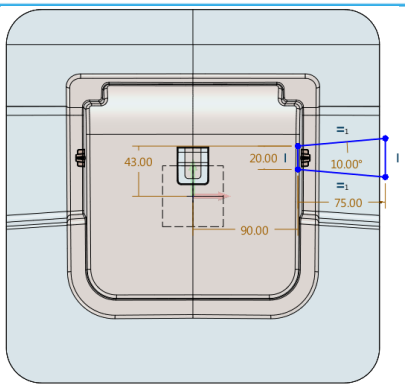
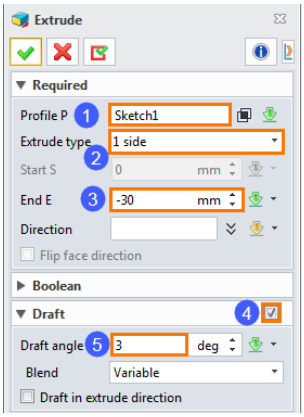


Figure 121 Create a Well-defined Sketch for the Slider

STEP 02 Extrude the sketch. Set parameters as below.



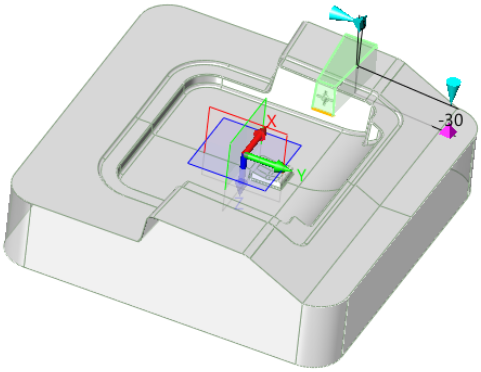
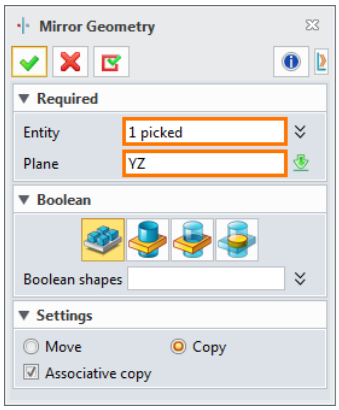


Figure 122 Extrude the Sketch

STEP 03 Use ***Mirror Geometry*** command to mirror the slider by YZ Datum.



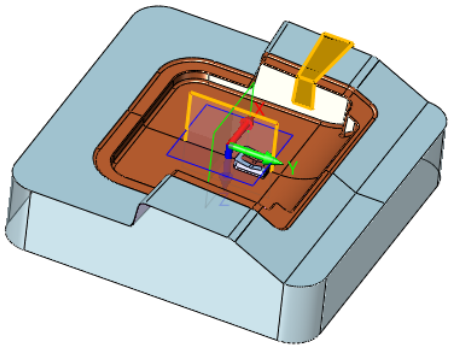
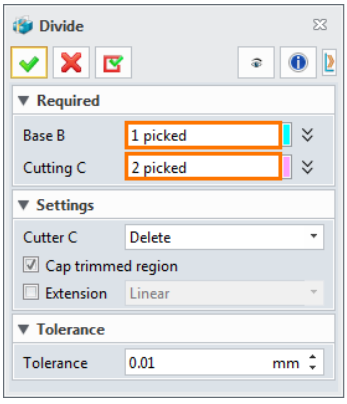


Figure 123 Mirror the Slider Tip

STEP 04 Use ***Divide*** command to divide sliders from the ***cavity*** shape.



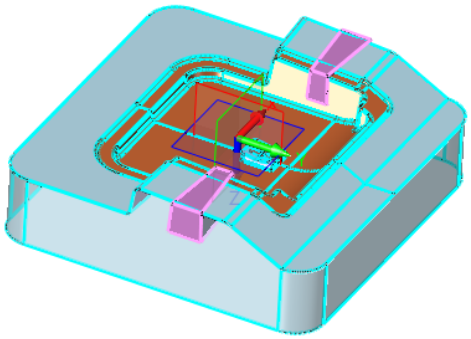


Figure 124 Divide Sliders from the Cavity Shape

The result shows as below.

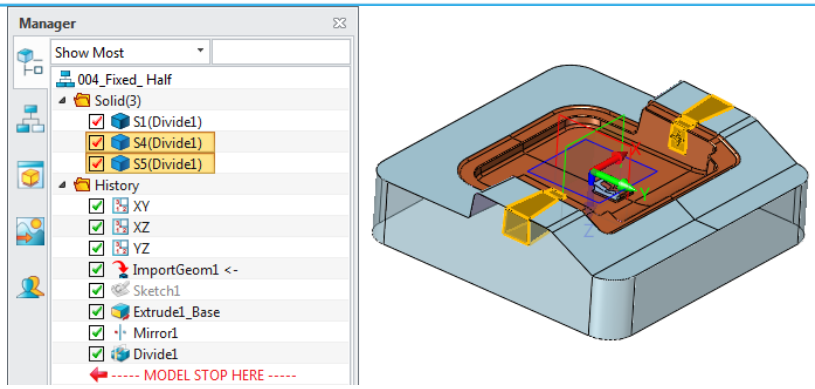


Figure 125 Division Result

Change the color of ***Slider*** shapes as below.

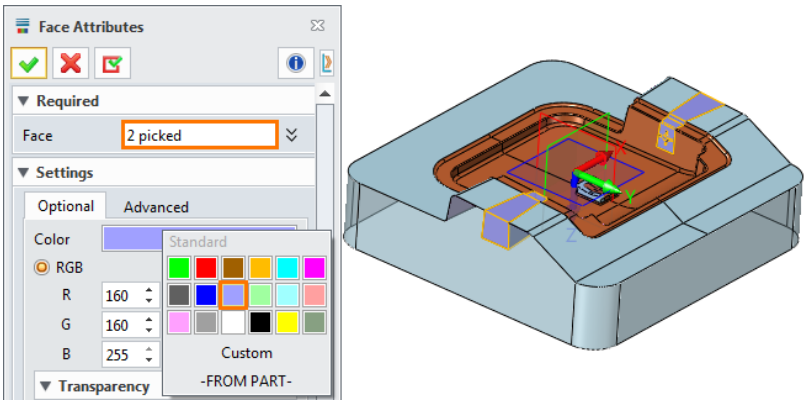


Figure 126 Change the Color of the Slider

STEP 05 Create a sketch based on the bottom face of the slider, then pick the center point of the hole as the original point.

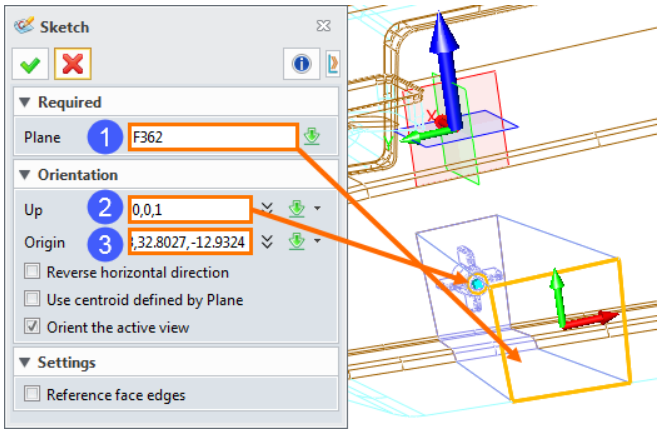


Figure 127 Create a Sketch for the Slider Base

Draw a sketch as below.

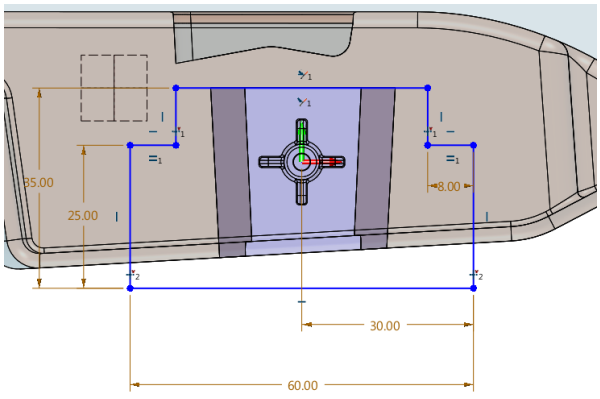


Figure 128 Draw a Well-defined Sketch

STEP 06 Use ***Extrude*** command in ***Shape*** tab to extrude a base for the Slider.

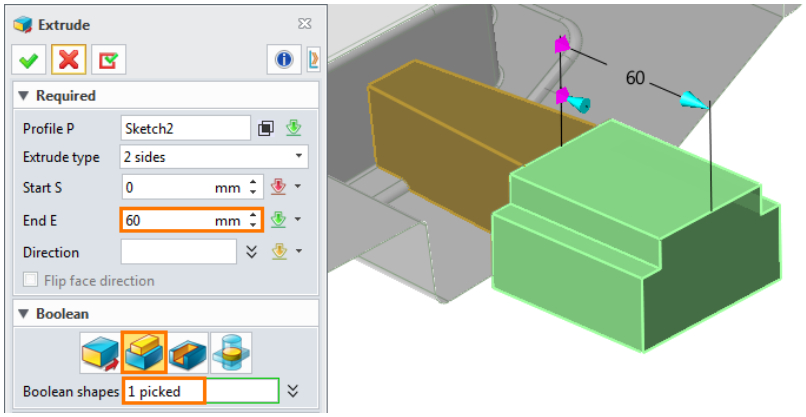


Figure 129 Extrude the Sketch

STEP 07 Create a ***Draft*** for the Slider.

First, create a sketch based on the side face of the slider.

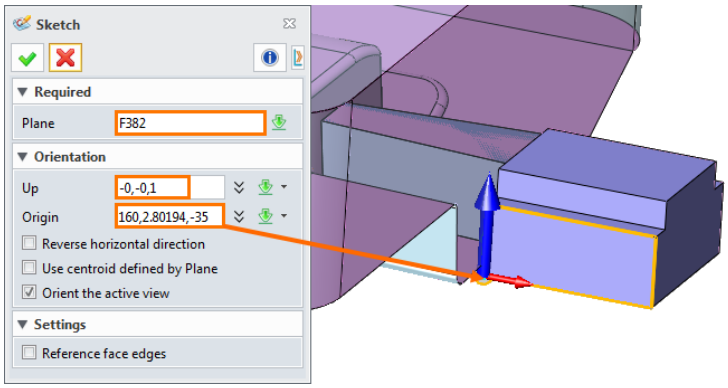


Figure 130 Create a Sketch for the Draft

Draw the sketch as below.

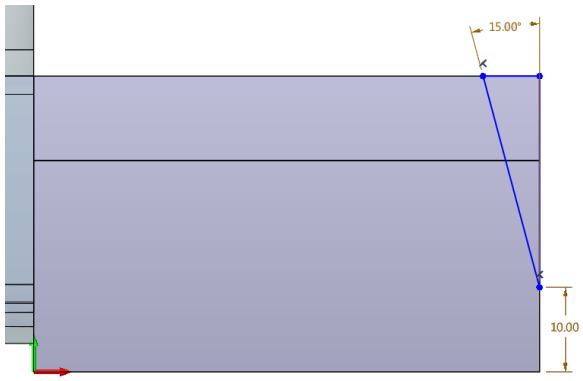


Figure 131 Draw a Well-defined Sketch for the Draft

STEP 08 Extrude the sketch to cut the slider.

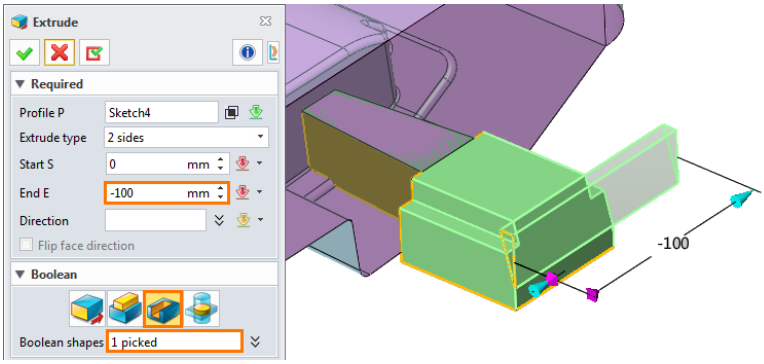


Figure 132 Extrude the Sketch to Cut the Slider

STEP 09 A add a base for the other slider with the same routes. The final result shows as below.

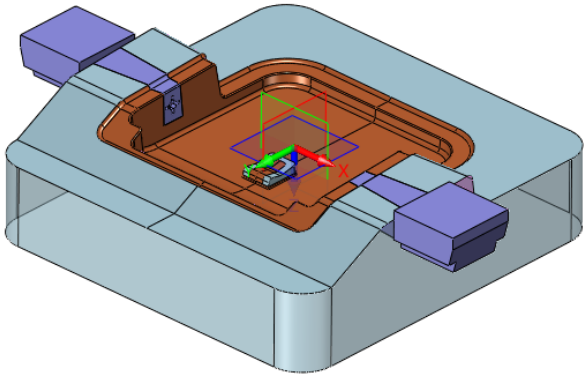


Figure 133 Result of Slider Creation

STEP 10 Use ***Geom to Part*** command in ***Assembly*** tab to extract the cavity to a new part ***101_Cavity***.

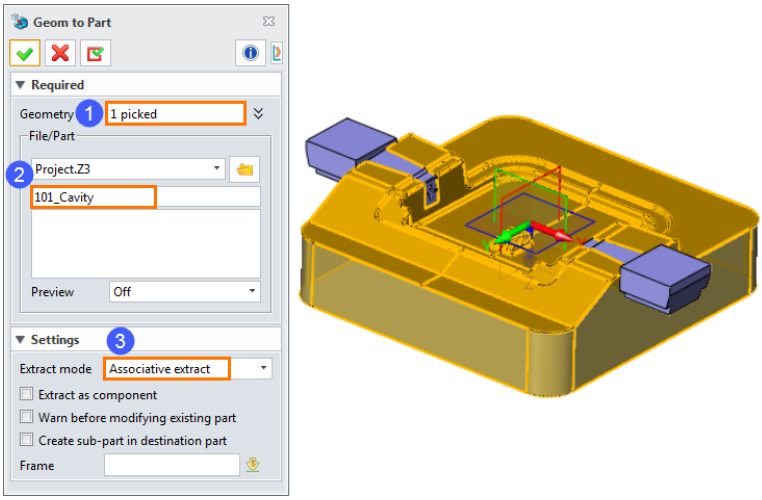


Figure 134 Extract the Cavity

STEP 11 Use the same operations to extract two sliders to part ***206_Slider1*** and ***207_Slider2***.

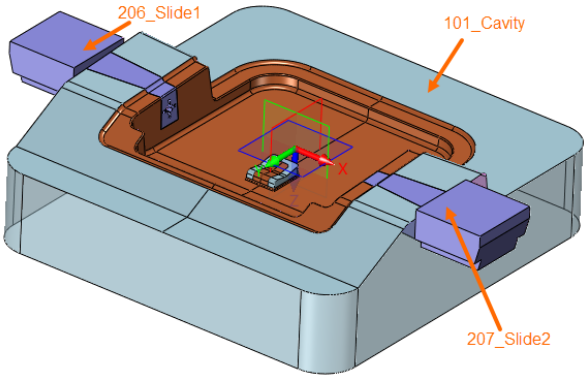


Figure 135 Extract the Sliders

Now the object manager shows as below.

Name	Type
000_Mold_Assembly	Part
001_Part_Model	Part
002_Position_Shrink	Part
003_Parting	Part
004_Fixed_ Half	Part
005_Moving_Half	Part
006_Epin_Cutter	Part
101_Cavity	Part
201_Core	Part
202_Insert1	Part
203_Insert2	Part
204_Lifter1	Part
205_Lifter2	Part
206_Slider1	Part
207_Slider2	Part

Figure 136 New Objects in Manager

1.5 Load Moldbase

STEP 01 Create a new part named ***000_Mold_Assembly***.

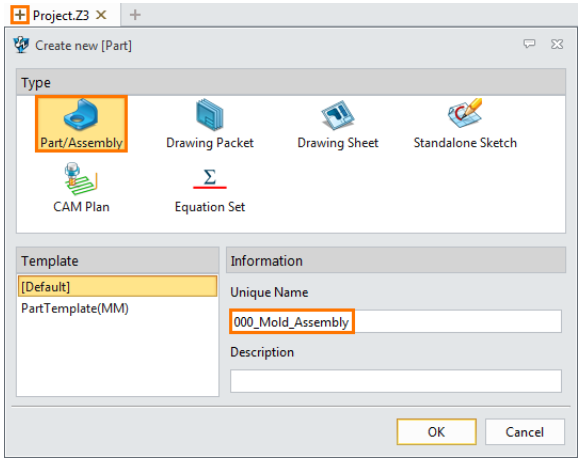


Figure 137 Create “000_Mold_Assembly”

STEP 02 Use ***Multi-Insert*** command in ***Assembly*** tab, pick ***101_Cavity, 201_Core, 202_Insert1*** ,***203_Insert2, 204_Lifter1*** and ***205_Lifter2***, ***206_Slider1***, ***207_Slider2*** from the list one by one.

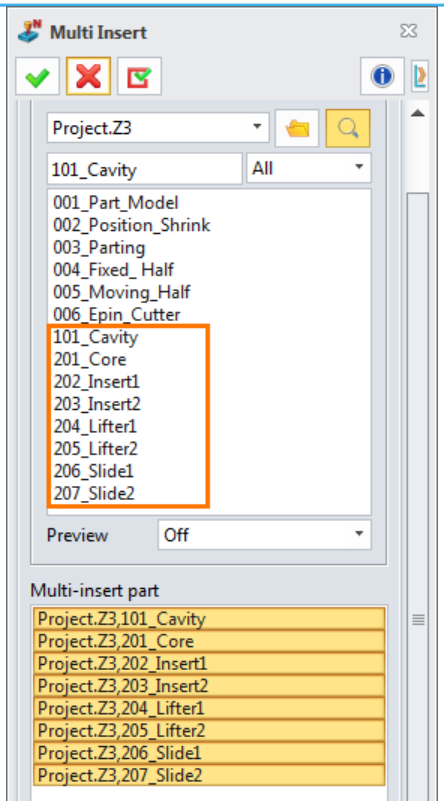


Figure 138 Insert Related Objects

Now there are 8 components in ***000_Mold_Assembly***.

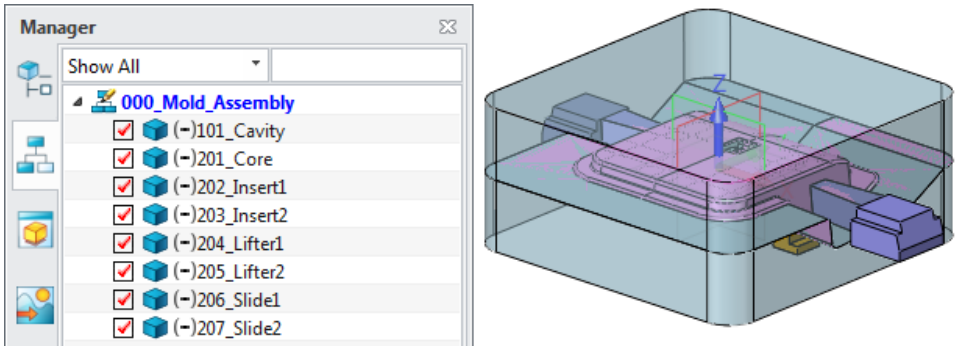


Figure 139 Components in “000_Mold_Assembly”

STEP 03 Use ***Layer*** command to open ***Layer Manager***. Create new layers ***Cavity*** and ***Core***.

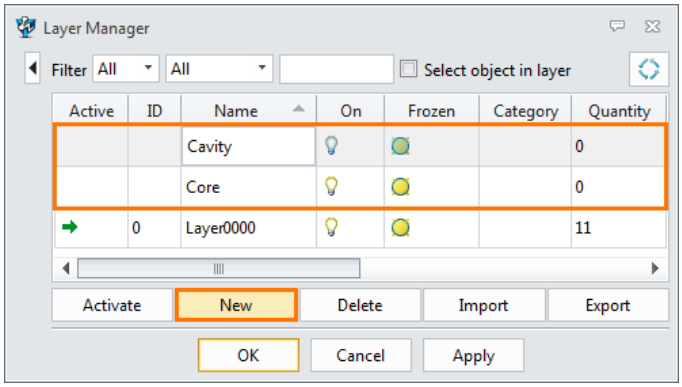


Figure 140 Create New Layers

Then assign the ***101_Cavity*** to Layer ***Cavity***

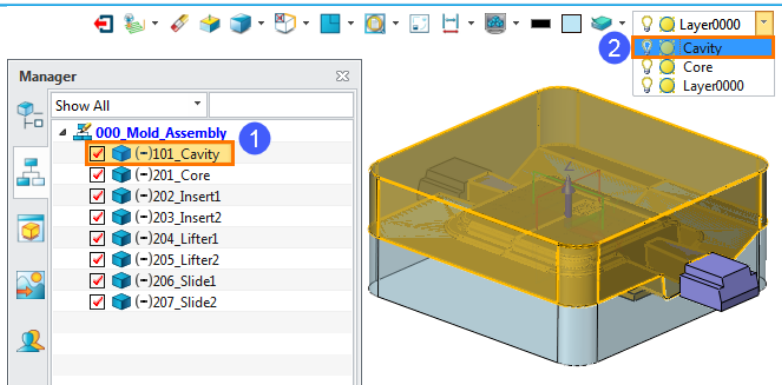


Figure 141 Assign "101_Cavity" to Layer "Cavity"

Use the same operation to assign "***201_Core" "202_Insert1" "203_Insert2" "204_Lifter1" and "205_Lifter2" "206_Slider1" "207_Slider2" t***o Layer "***Core***".

Finally, there are 1 object in Layer ***Cavity*** and 7 objects in Layer ***Core***.

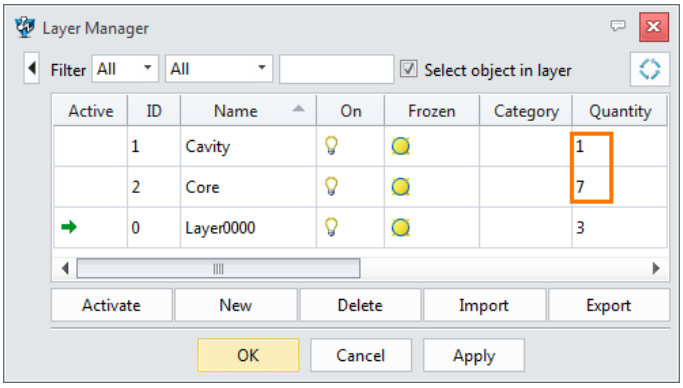


Figure 142 Objects Quantities in Different Layer

STEP 04 Use ***Moldbase*** command to insert moldbase. Pick the ***LKM*** supplier, SG type, CI Class, and XY Datum.

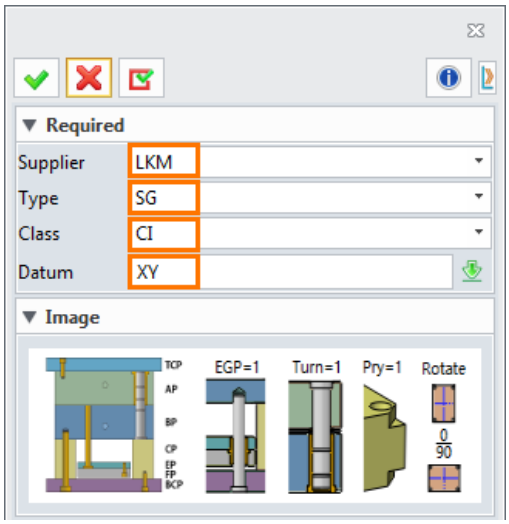


Figure 143 Insert MoldBase

Set main parameters as below.

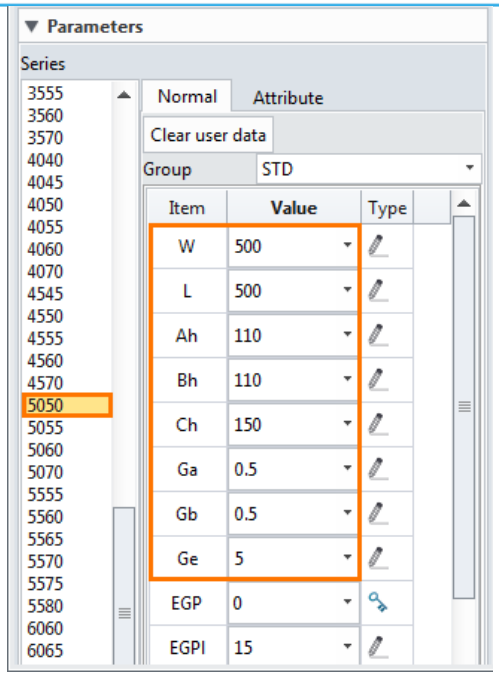


Figure 144 Set Parameters

The result shows as below.

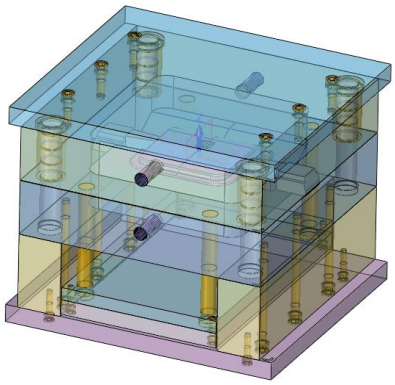


Figure 145 Insert Result

STEP 05 Create a pocket on ***AP*** for cavity.

Double click ***AP*** in Assembly manager to activate ***AP***.

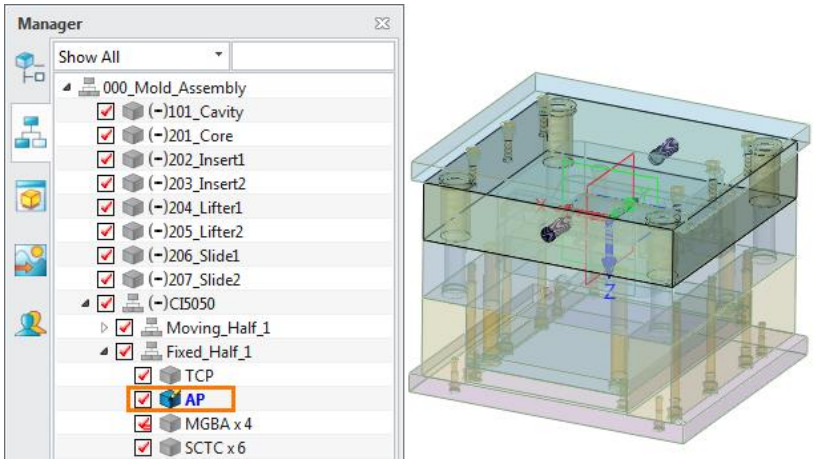


Figure 146 Activate “AP”

STEP 06 Use ***Curve list*** command in “Wireframe” tab to creat a curve list as below. (Use ***Ctrl + F*** on keyboard to switch to ***wireframe display mode***)

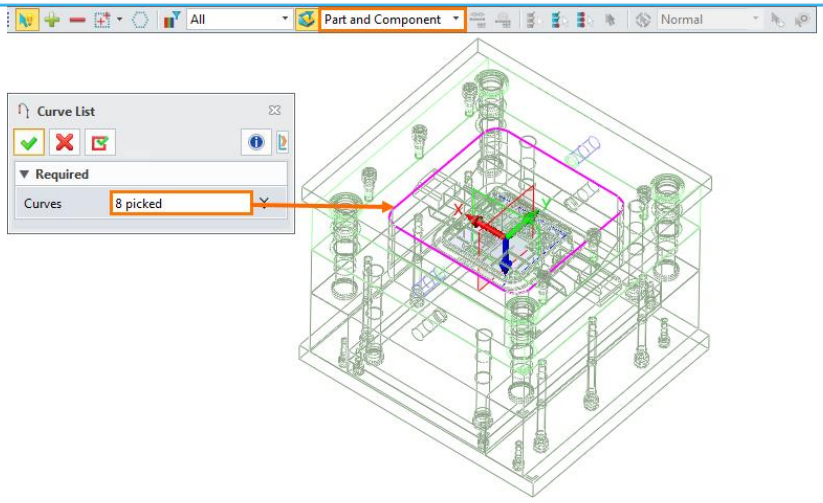


Figure 147 Create a “Curve List”

STEP 07 Click ***Show Target*** in toolbar. Extrude the curve list. Set parameters as below.

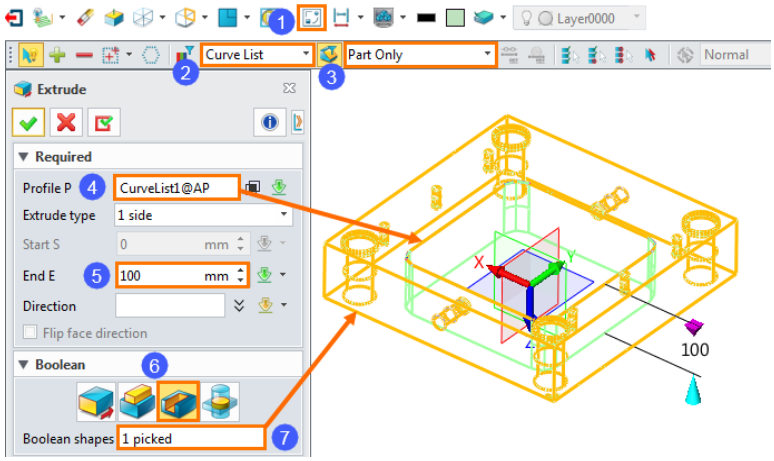


Figure 148 Extrude the Curve List the Create a Pocket

The result shows as below.

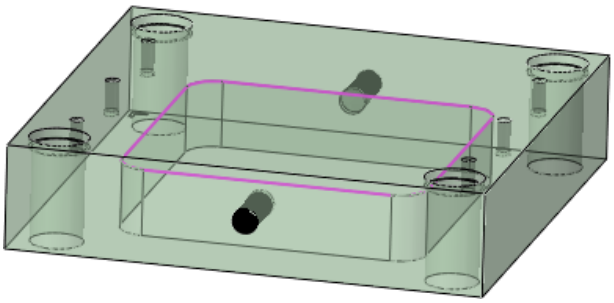


Figure 149 Pocket Result

STEP 08 Click ***Show all*** to show all components. Then double click on ***BP*** in assembly manager to activate ***BP***.

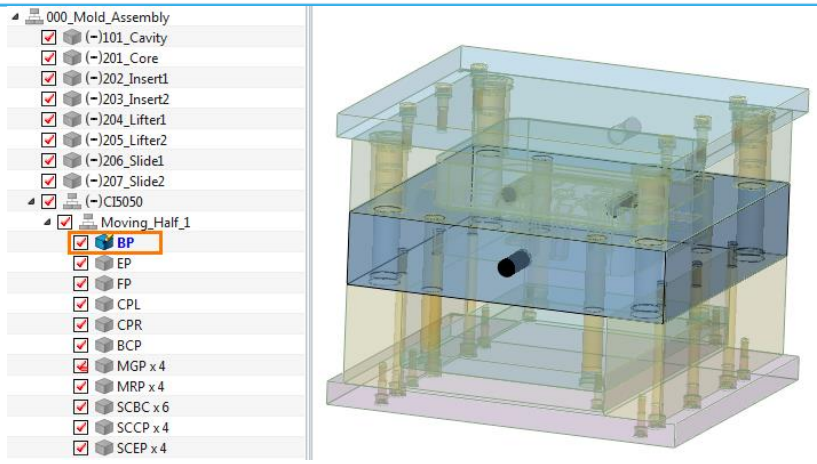


Figure 150 Activate “BP”

STEP 09 Use the same operations to create a pocket on ***BP*** for Core. The result shows as below.

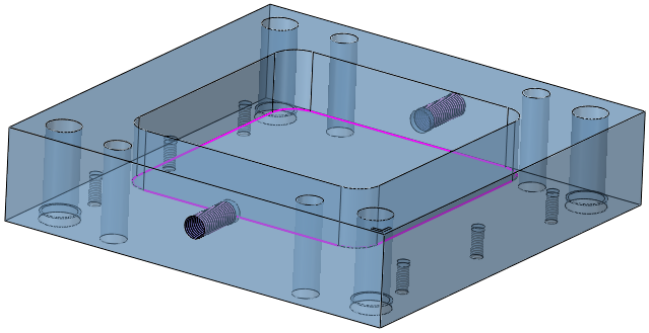


Figure 151 Create a Pocket on “BP”

You can get the overview of the whole assembly shows as below.

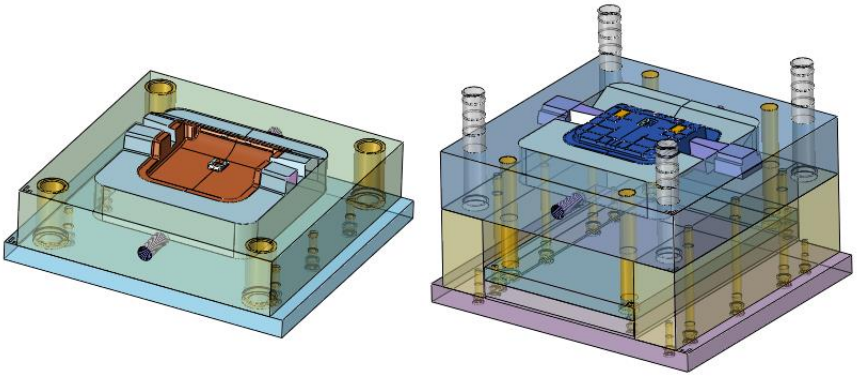


Figure 152 Overview of the Mold Assembly

1.6 Adjust Sliders and Lifters

1.6.1 Adjust Sliders

STEP 01 Double click on the ***206_ Slider1*** to activate it.

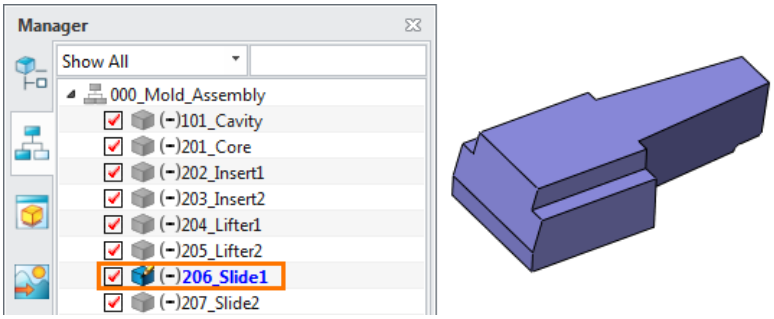


Figure 153 Activate the “206_ Slider1”

STEP 02 Use ***DE Face Offset*** command in ***Direct Edit*** tab to offset the faces as below.

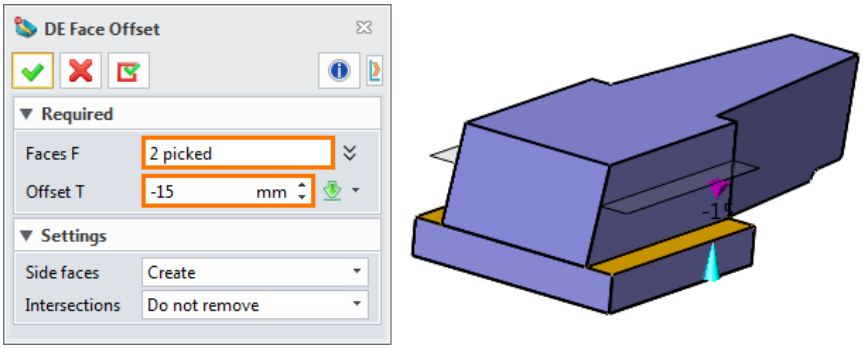


Figure 154 Offset Some Faces

Up to now, we have adjusted the ***206_Slider1***.

STEP 04 Use the same operations to offset the faces on ***207_ Slider2***

STEP 05 Activate the ***BP***, create a sketch based on the plane as below.

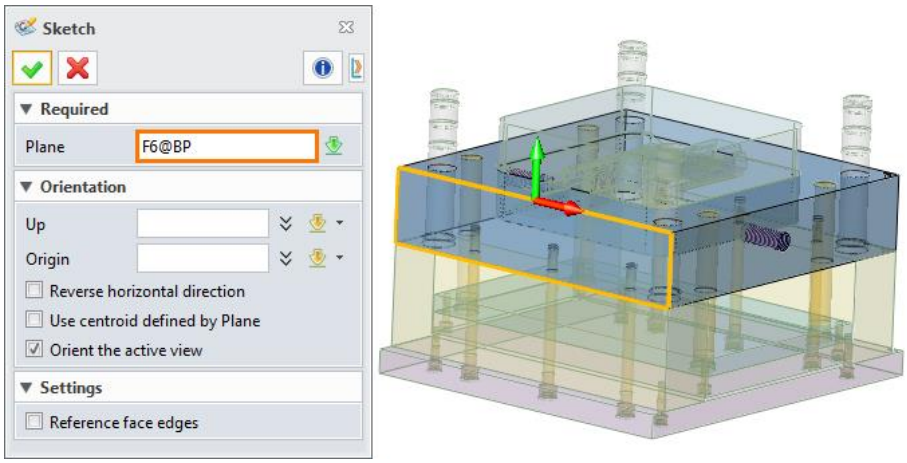


Figure 155 Create a Sketch on a Side Face of "BP"

STEP 06 Use ***Reference*** command in ***Sketch*** tab. Reference ten curves as below.

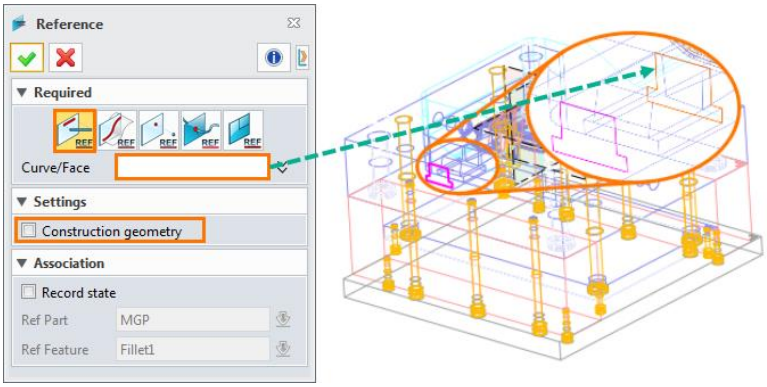


Figure 156 Reference Curves from the Slider.

STEP 07 Exit the sketch. Extrude the sketch to cut a pocket for the Slider. Set parameters as below.

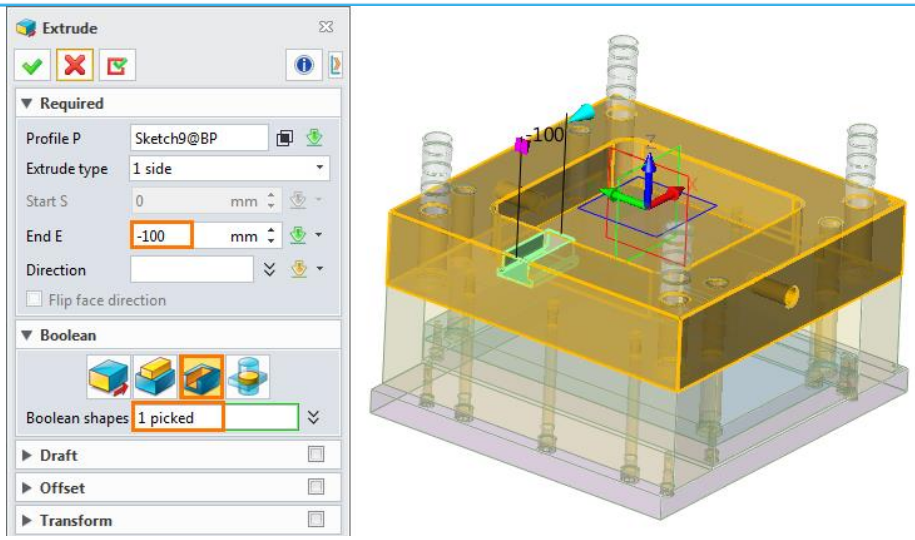


Figure 157 Extrude the Sketch to Cut the BP

STEP 08 Extrude the side face as below.

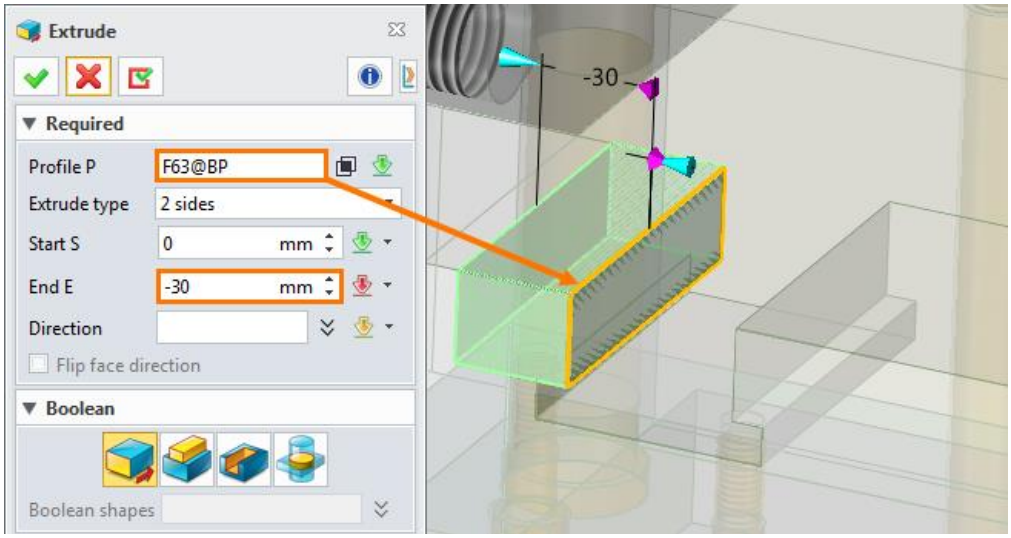


Figure 158 Extrude a Side Face of the Pocket

STEP 09 Extrude the another side face as below.

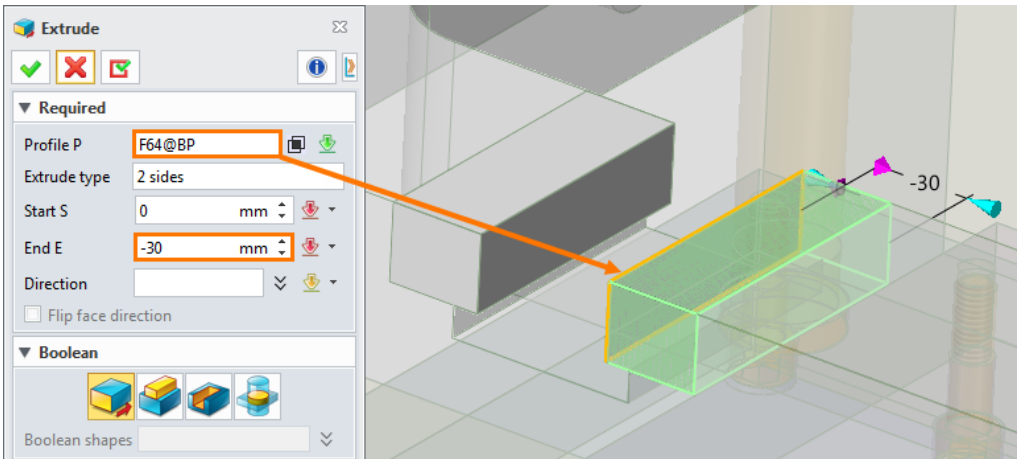


Figure 159 Extrude the Other Side Face of the Pocket

STEP 10 Use ***Divide*** command to divide these two shapes from ***BP*** shape.

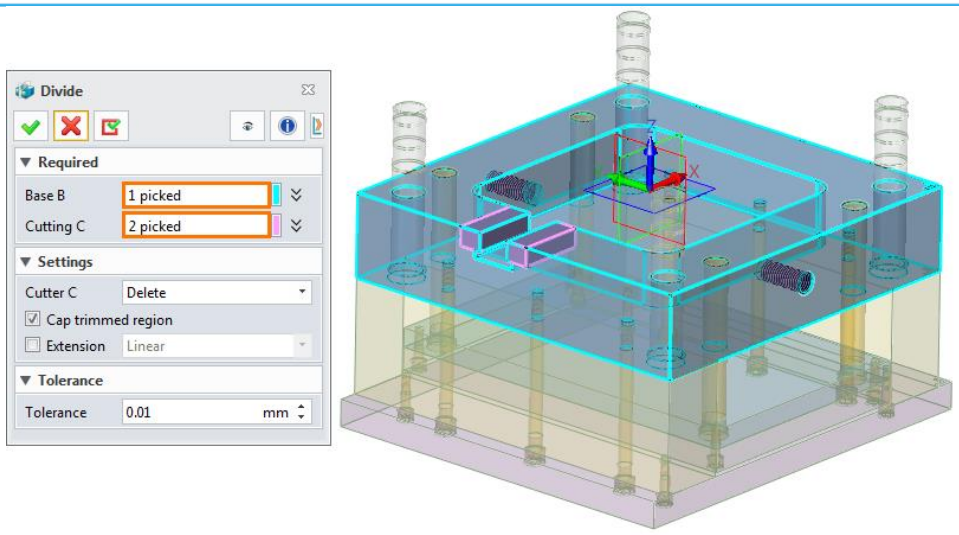


Figure 160 Divide the Blocks from BP

STEP 11 Use ***DE Face Offset*** command to offset the groove side faces.

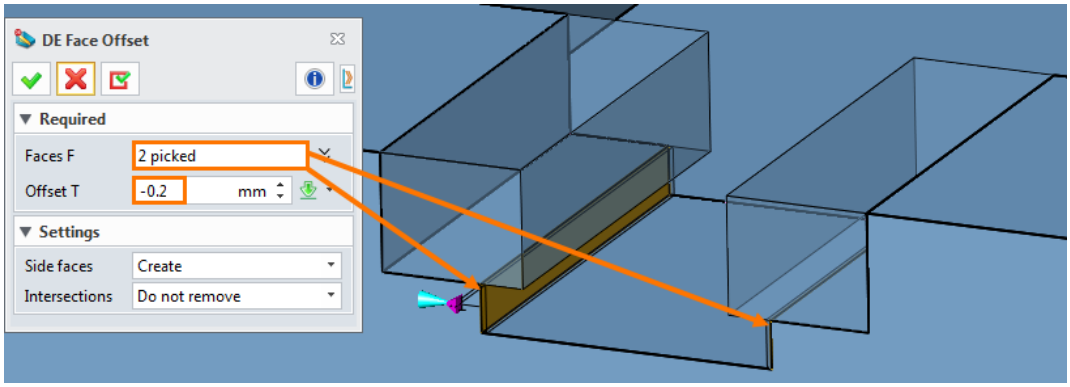


Figure 161 Set Tolerance

STEP 12 Adjust the other llider with the same operations and create a pocket for it.

The final shows as below. There are five shapes in the part, ***BP***.

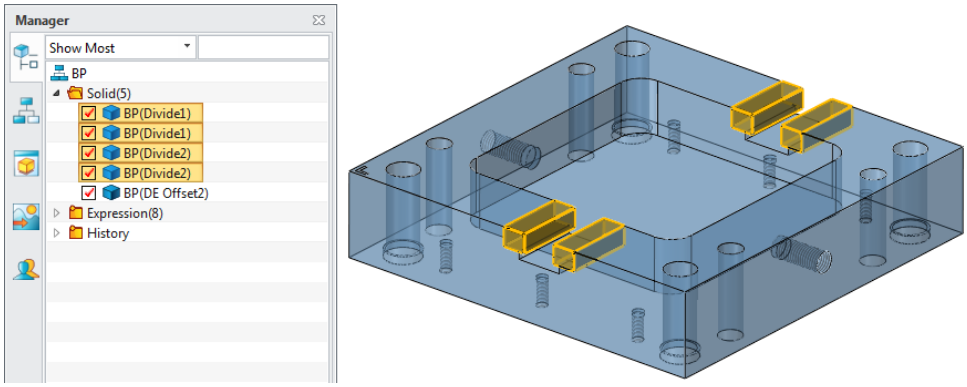


Figure 162 Division Result

STEP 13 Use ***Geom to Part*** commmand in ***Assembly*** tab to extract a clamping block as a new component of ***BP***.

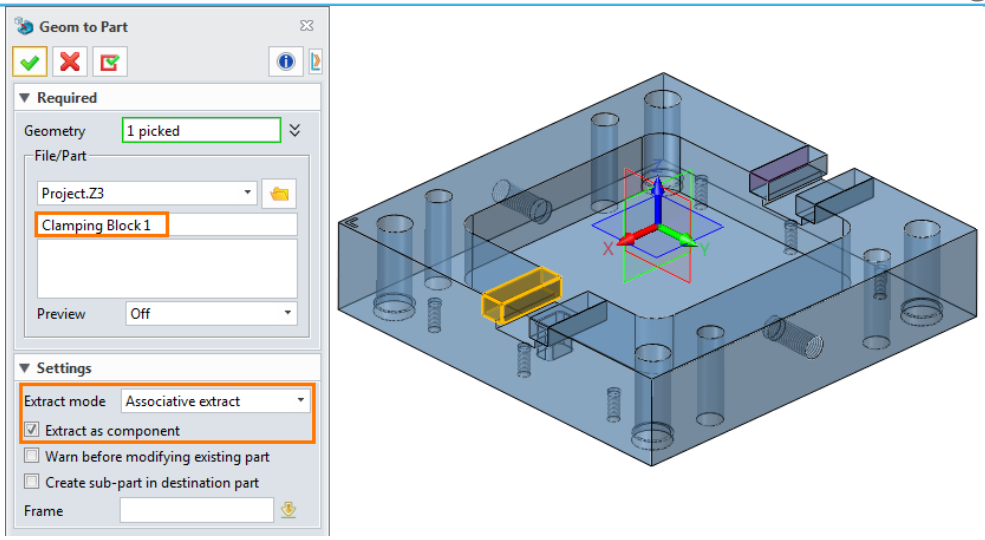


Figure 163 Extract the Block

STEP 14 Extract other three clamping blocks as other three components with the same operations. Final result shows as Figure 164.

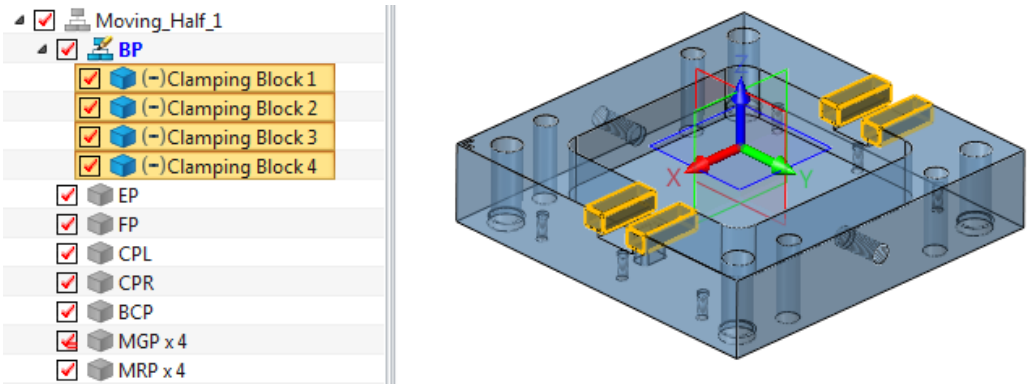


Figure 164 Extract Another Three Blocks

1.6.2 Adjust Lifters

STEP 1 Double click on ***000_Mold_Assembly*** to activate it. Pick ***204_Lifter1, 205_Lifter2, BP, EP*** and ***FP*** at the same time, then select ***Show Only*** in right-click menu.

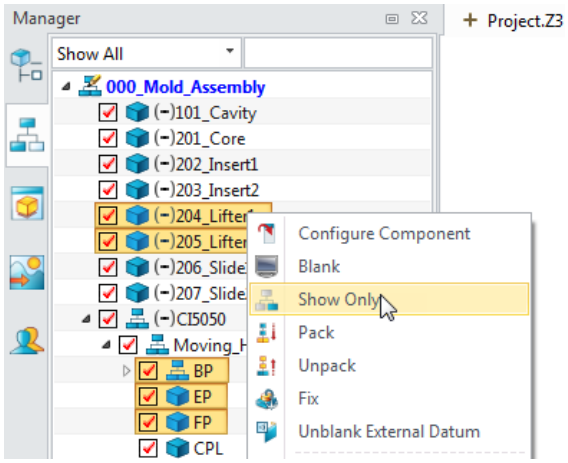


Figure 165 Only Show Some Components

The result shows as below.

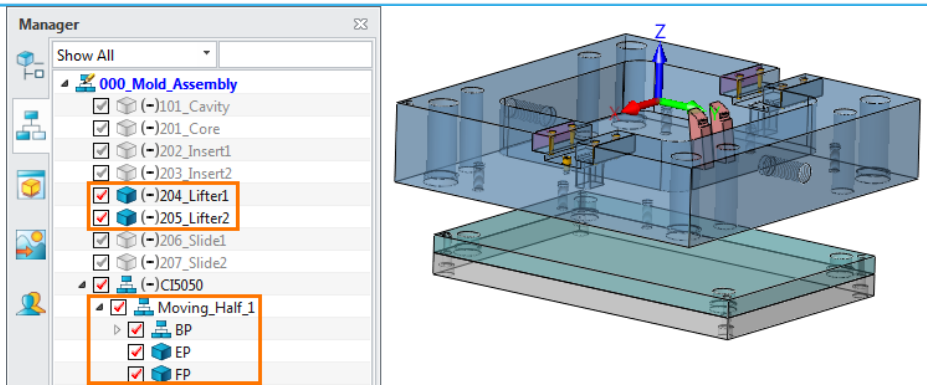


Figure 166 Result after "Show Only"

STEP 2 Double click on ***204_Lifter1*** to activate it.

Use ***DE Face Offset*** command in ***Direct Edit*** tab to offset the bottom face of the lifter.

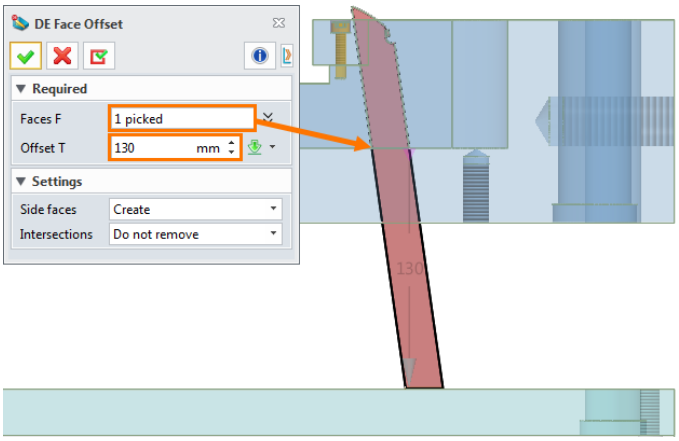


Figure 167 Offset the Bottom Face of the Lifter

Creat a sketch based on the side face of the lifter.

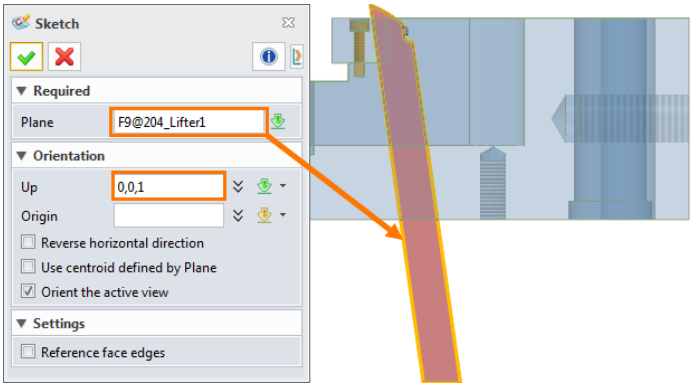


Figure 168 Creat a Sketch Based on the Side Face of the Lifter

Draw the sketch as below.

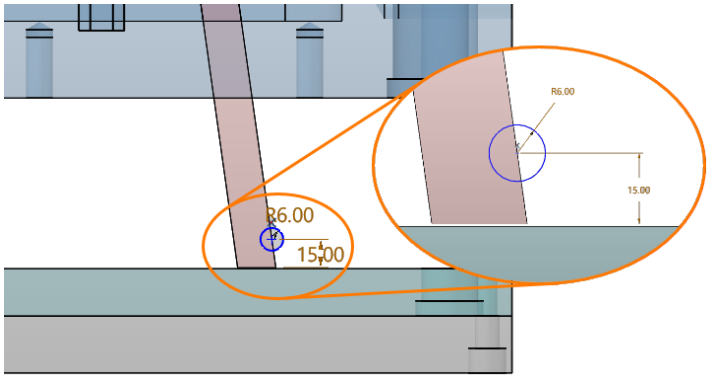


Figure 169 Draw a Well-defined Sketch

Extrude the sketch. Set parameters as below.

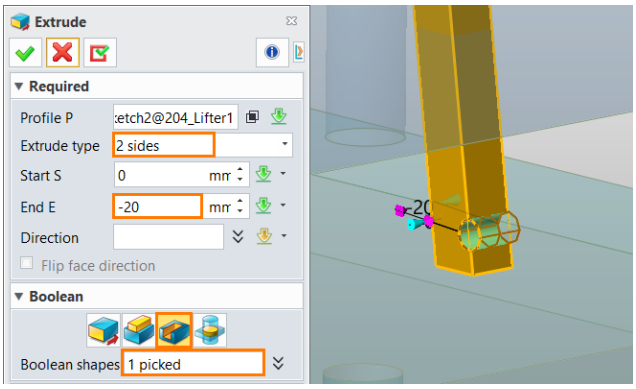


Figure 170 Extrude the Sketch

Create another sketch based on the same side face.

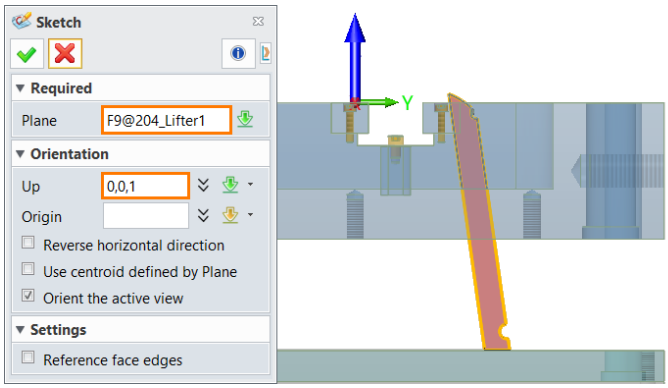


Figure 171 Create Another Sketch Based on the Same Side Face

Draw the sketch as below.

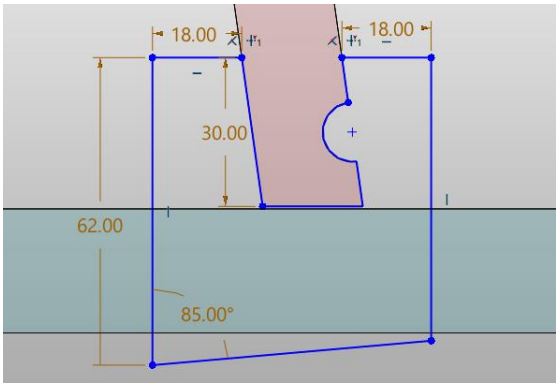


Figure 172 Draw a Well-define Sketch

Extrude the sketch as a block. Set paremeters as below.

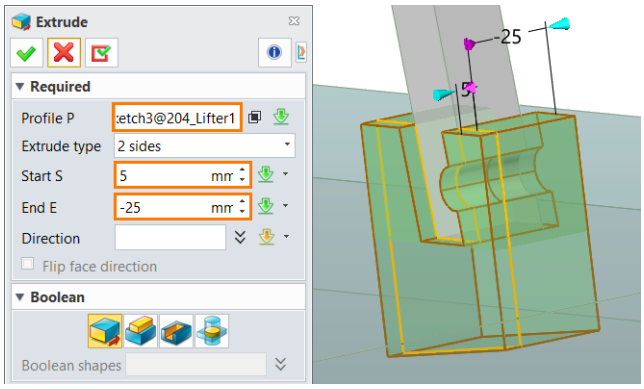


Figure 173 Extrude the Sketch

Create a sketch on the the side face of the block.

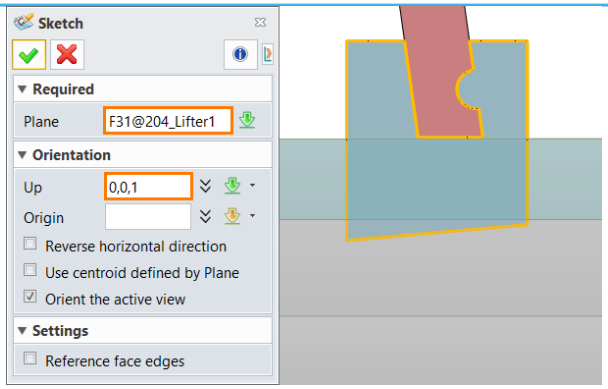


Figure 174 Create Another Sketch Based on the Side Face

Draw a sketch as below.

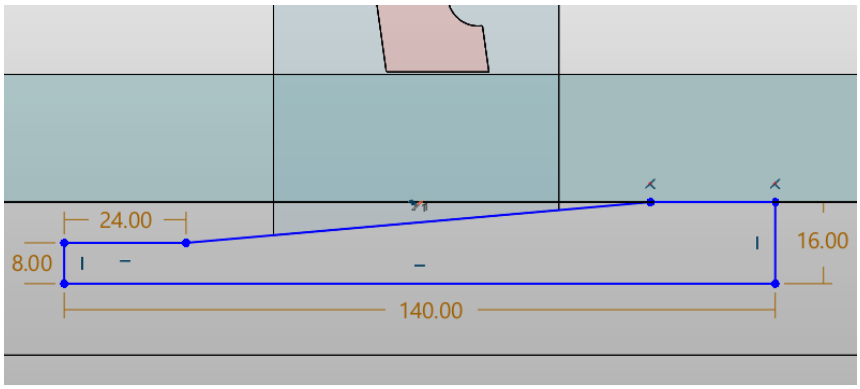


Figure 175 Draw a Well-defined Sketch

Extrude the sketch. Set parameters as below.

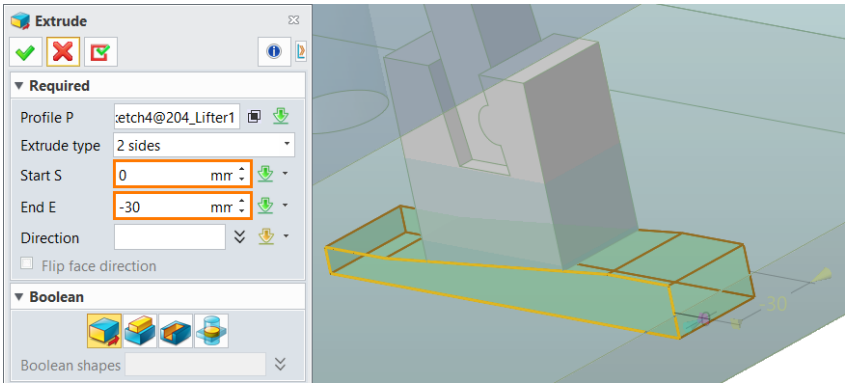


Figure 176 Extrude the Sketch

Use ***Geom to Part*** to extract these two shapes as components.

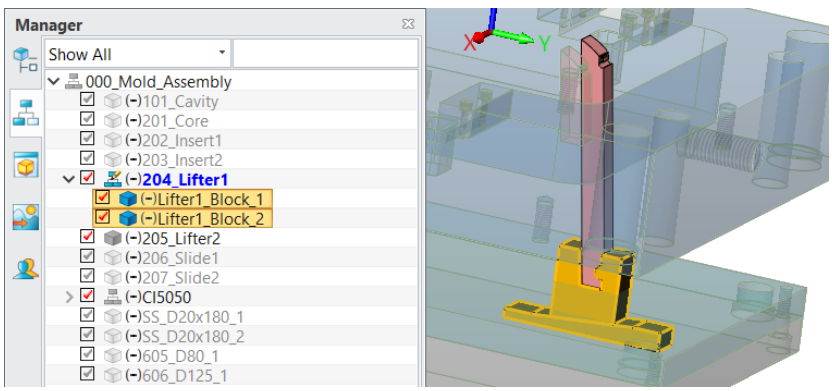


Figure 177 Extract the Two Blocks

Then, activate the ***EP***, use ***Reference*** command to reference the ***Lifter_block_1***.

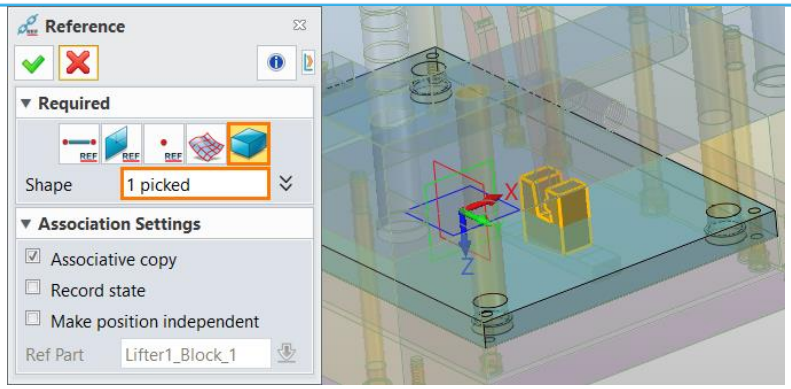


Figure 178 Reference “Lifter_block_1”

Use ***Remove Shape*** command to create a pocket.

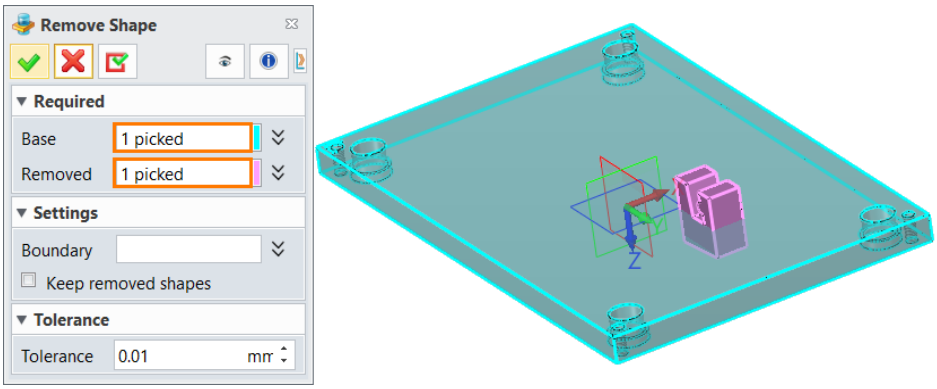


Figure 179 Create a Pocket for the Block

Activate ***FP***, then use ***Reference*** command to reference the bottom face of ***Lifter1_Block_2***.

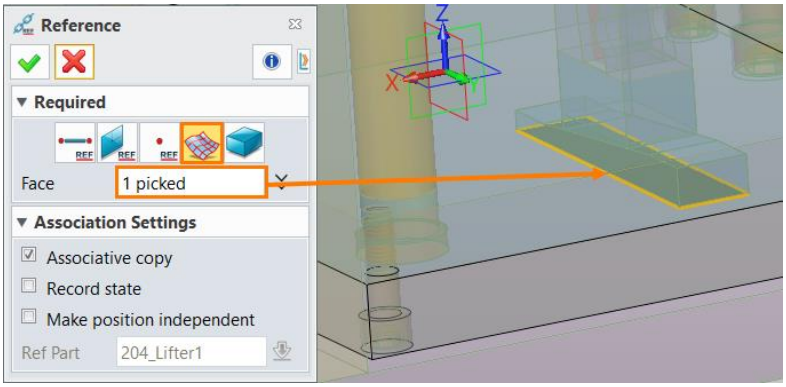


Figure 180 Reference the Bottom Face of “Lifter1_Block_2”

Extrude the sketch and remove it from the ***BP***.

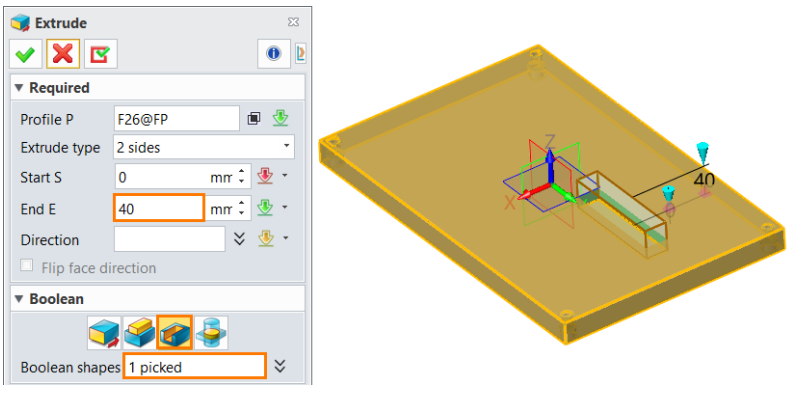


Figure 181 Extrude the Face to Cut the “BP”

Use the same operation to adjust the other lifter.

Final result shows as below.

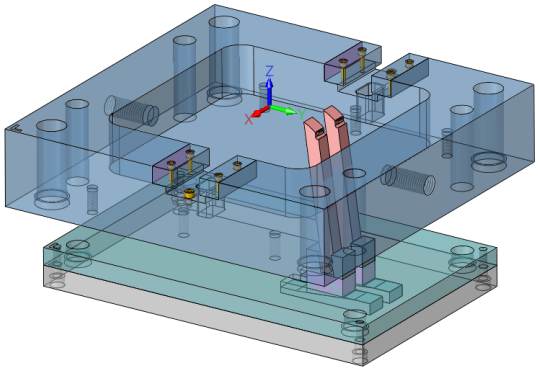


Figure 182 Final Result of the Lifters

1.7 Insert Standard Parts

1.7.1 Insert Angle Pins and Screws for Sliders

STEP 1 Return to ***000_Mold_Assembly***, then hide the ***TCP*** plate.

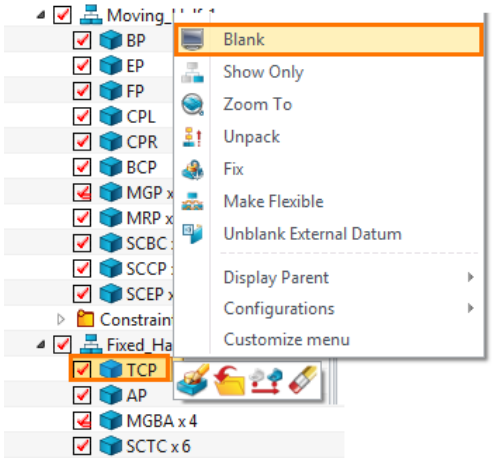


Figure 183 Blank the “TCP”

STEP 2 Use ***General*** command in ***Mold*** tab.

Pick the type as below.

Choose ***Between*** to pick ***Insert Point***.

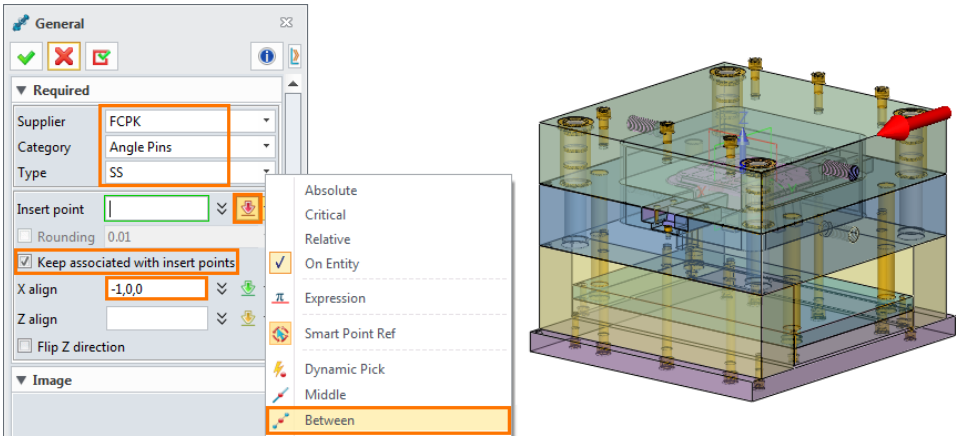


Figure 184 Insert an Angle Pin

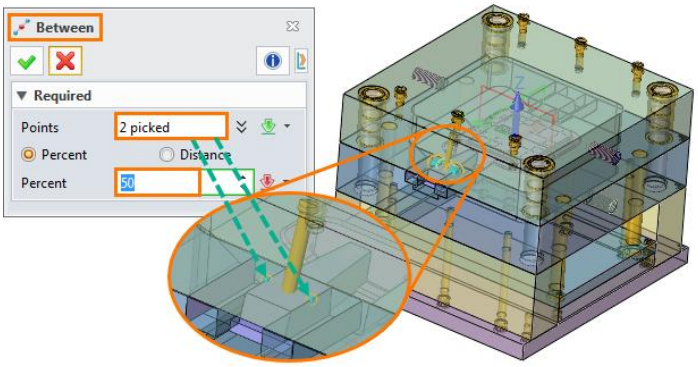


Figure 185 Choose “Between” to Pick the Location Point

Set other settings as below.

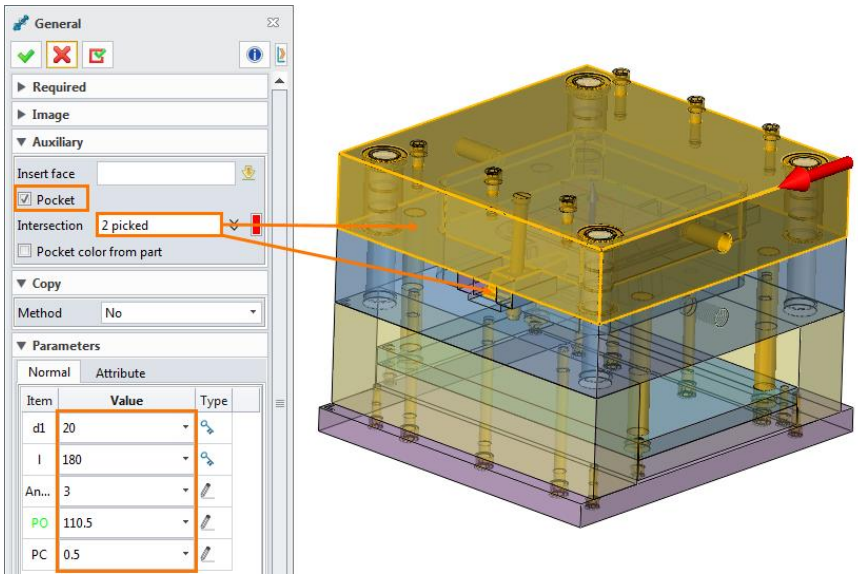


Figure 186 Other Settings and Parameters

Click ***OK***, it will automatically create pockets on the Slider and the ***AP*** for the angle pin.

STEP 3 Create a pocket on ***BP*** for the angle pin.

Double click on the ***BP*** to activate it.

Create a sketch on the plane as shown in figure 187.

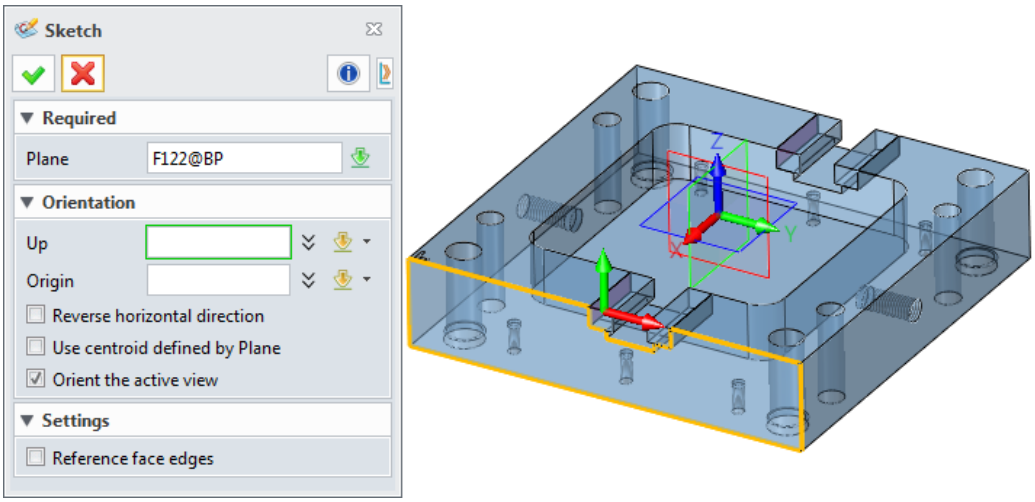


Figure 187 Create a Sketch on the Side Face

Draw the sketch as below.

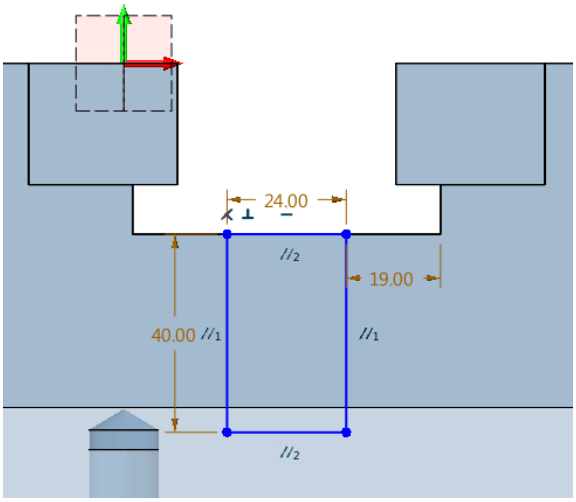


Figure 188 Draw a Well-Defined Sketch

Extrude the sketch.

Set parameters as below.

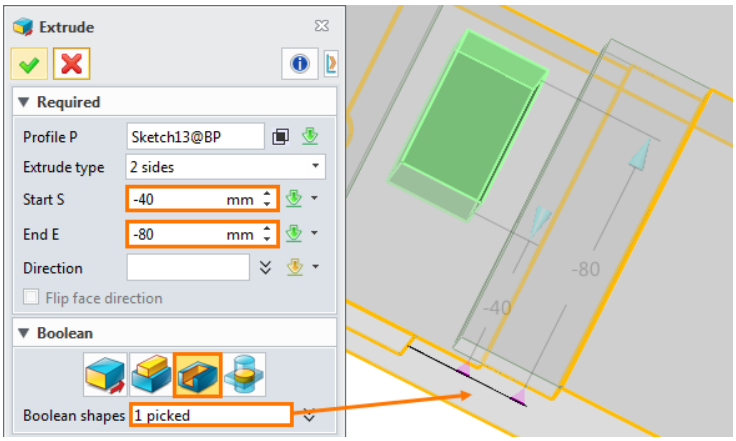


Figure 189 Extrude the Sketch

Add fillets on vertical edges.

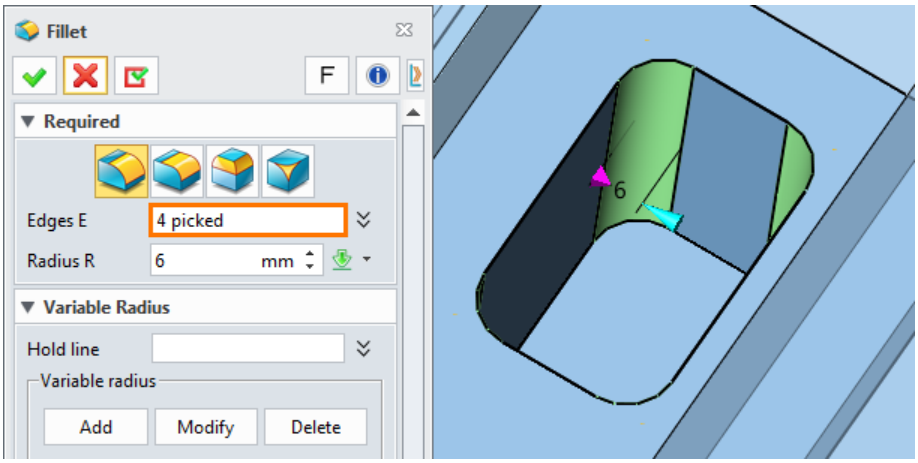


Figure 190 Add Fillets

Add chamfers on the top edges.

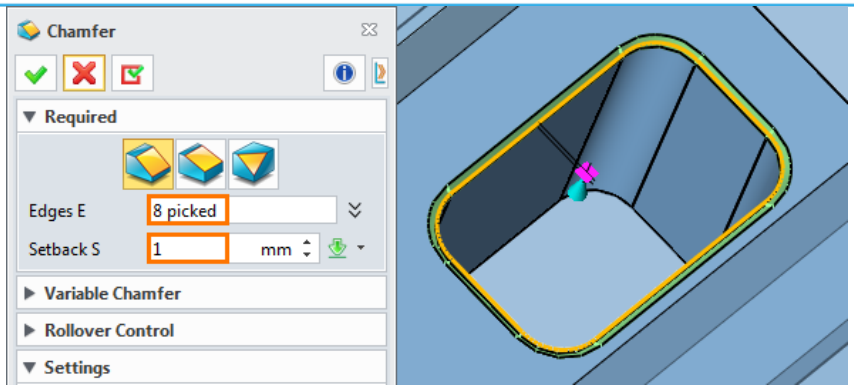


Figure 191 Add Chamfers

STEP 4 Add screws on sliding clamping blocks.

Use ***Screw*** command in ***Mold*** tab to insert screws.

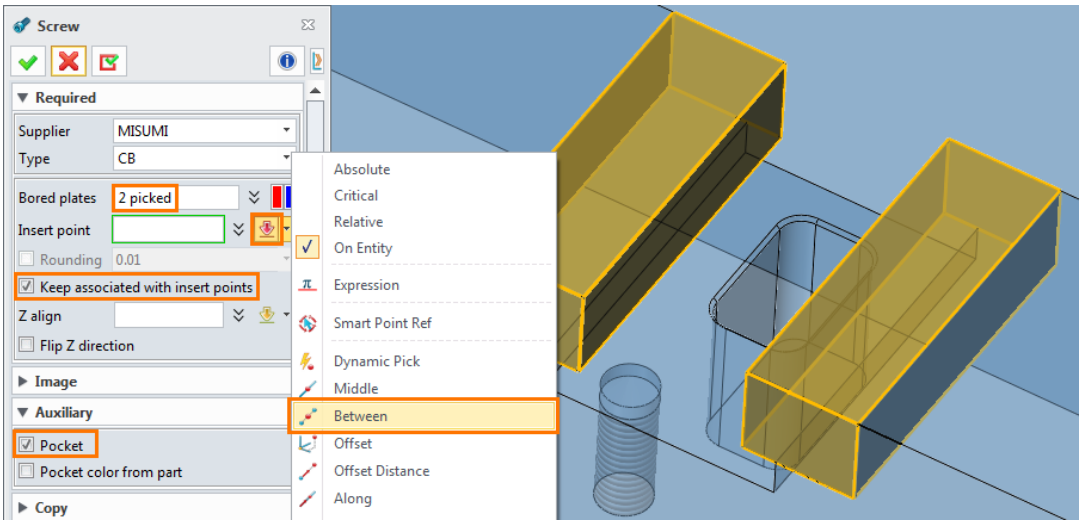


Figure 192 Insert Screws

Pick ***Between*** in the right menu, then select two points as Figure 193.

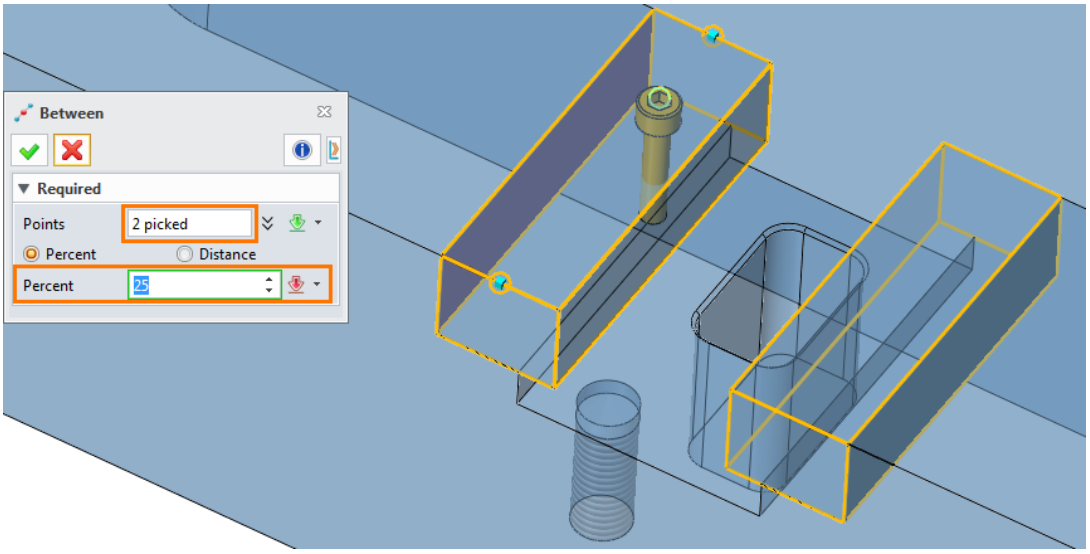


Figure 193 Choose “Between” to Pick the Location Point

Pick other 3 points in the same way. The result shows as Figure 194.

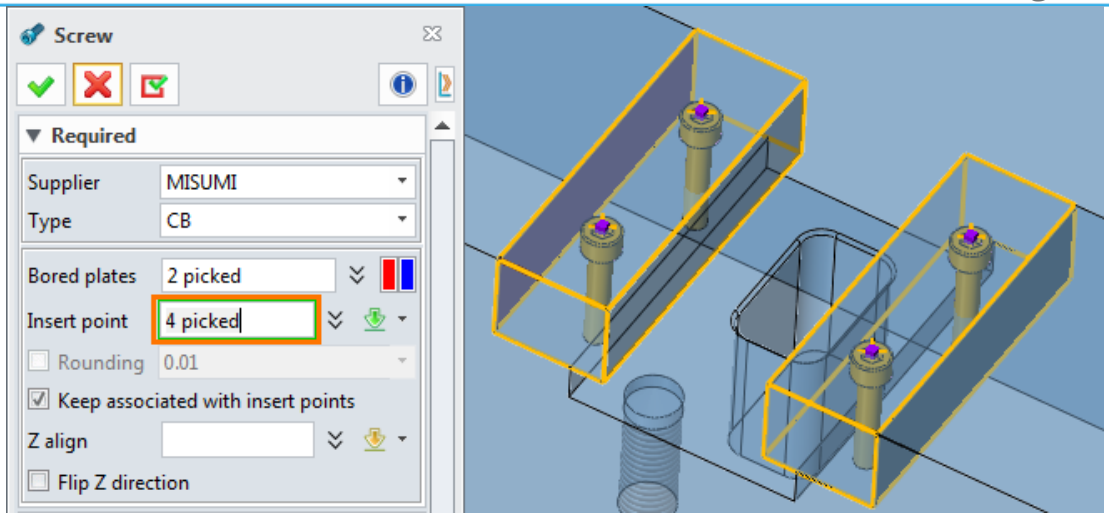


Figure 194 Insert Points

Set main parameters as below, and other parameters as default.

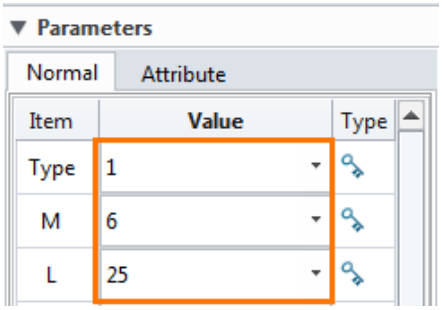


Figure 195 Main Parameters

The result shows as Figure 196.

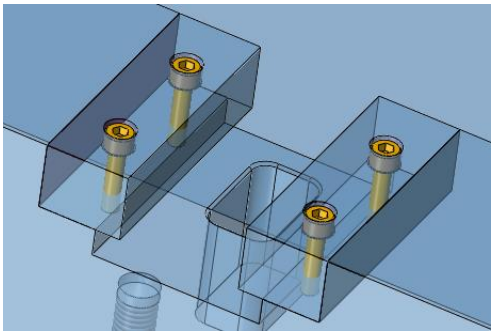


Figure 196 Insert Result

Use ***Geom to Part*** command in ***Assembly*** tab to extract a clamping block as a new component of ***BP***.

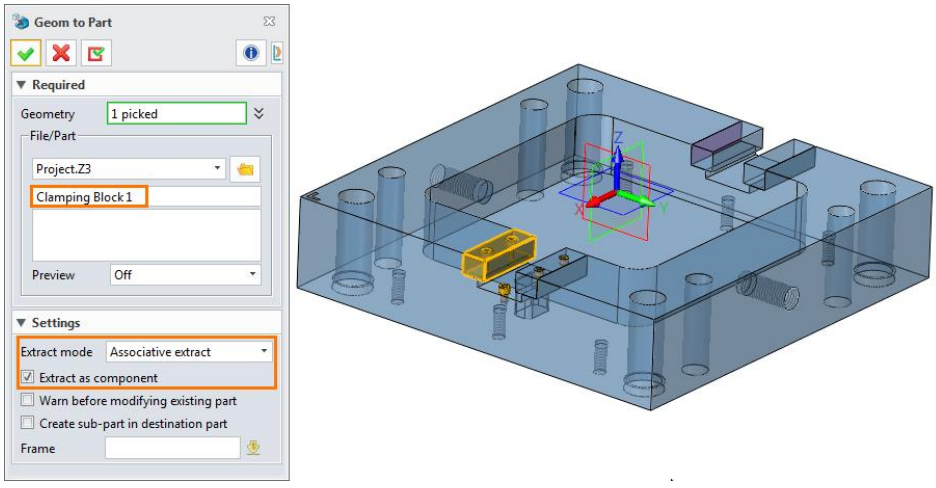


Figure 197 Extract the Clamping Block

Extract the other clamping block as the other component.

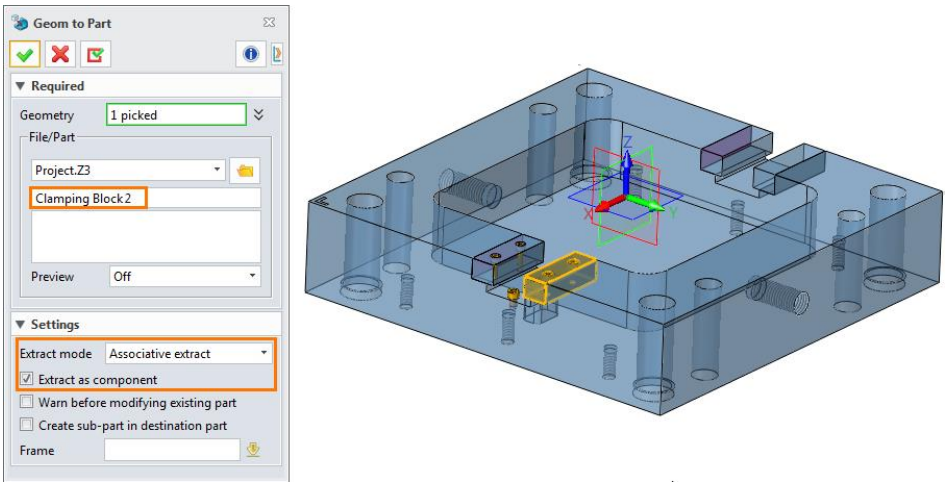


Figure 198 Extract the Other Clamping Block

STEP 5 Add a stop screw on ***BP***.

Use ***General*** command in ***Mold*** tab.

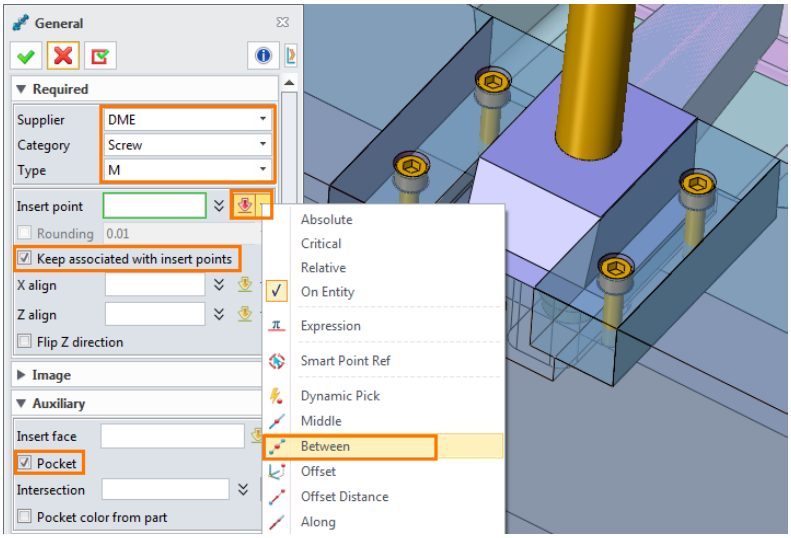


Figure 199 Insert a “Stop Screw”

Choose ***Between*** to pick the point as below.

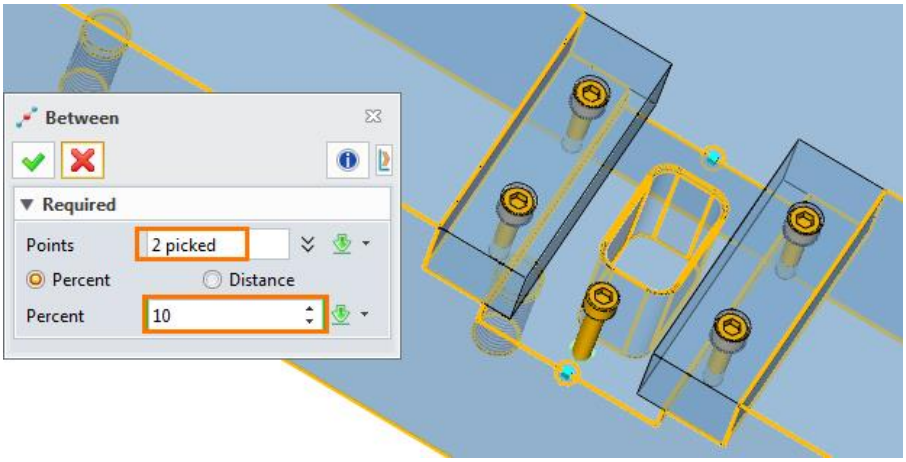


Figure 200 Choose “Between” to Pick a Point

Set main parameters as below, and other parameters as default.

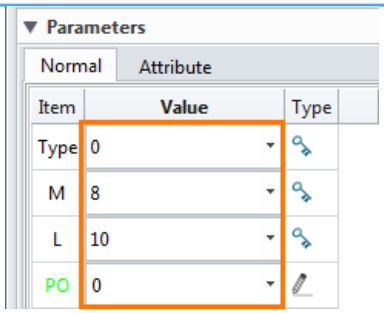


Figure 201 Main Parameters

The result shows as Figure 202.

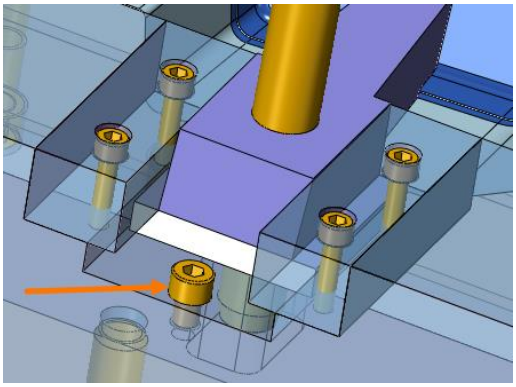


Figure 202 Result

STEP 06 Create a tight block for the Slider.

Double click on “AP” to activate it.

Use ***Reference*** command in ***Assembly*** tab.

Reference the Slider as below.

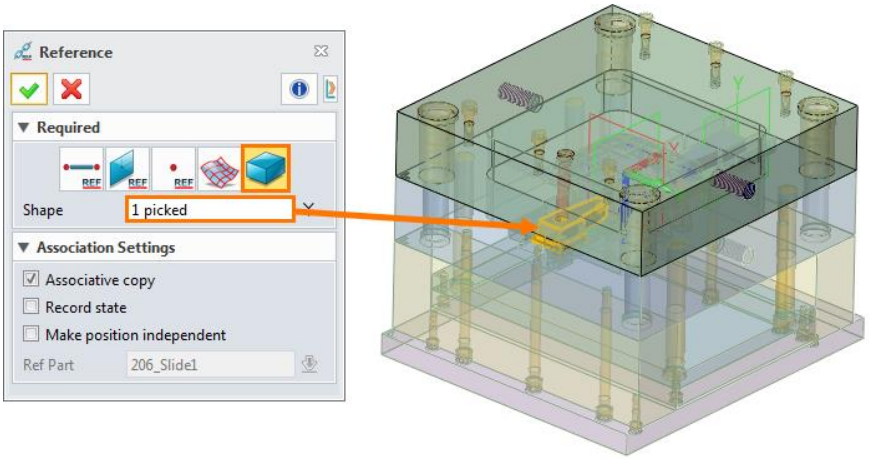


Figure 203 Reference the Slider

Use ***Datum*** command in ***Shape*** tab to create a datum as below.

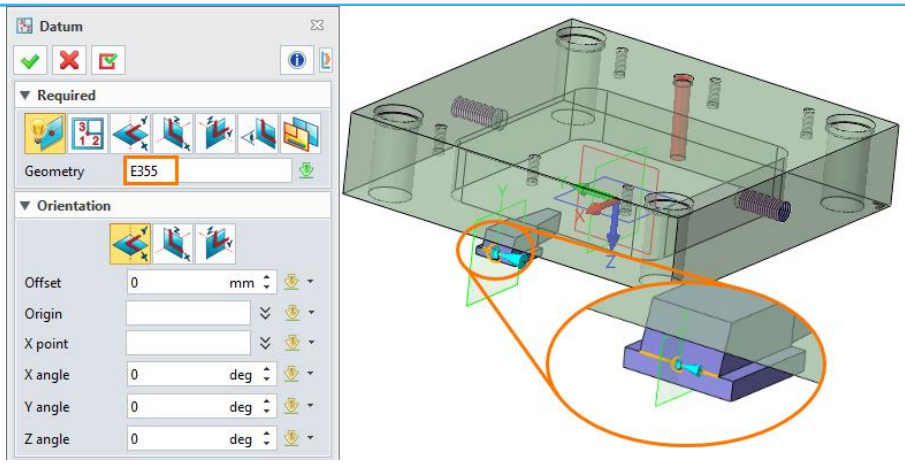


Figure 204 Create a New Datum

Create a sketch based on the new plane.

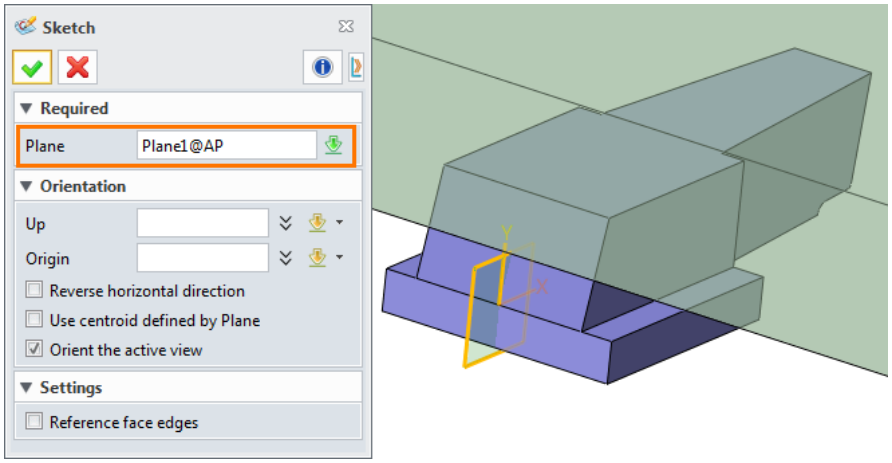


Figure 205 Create a Sketch on the New Datum

Draw the sketch as below.

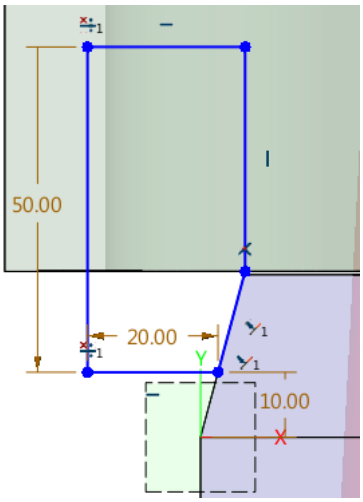


Figure 206 Draw a Well-defined Sketch

Extrude the sketch. Set parameters as Figure 207.

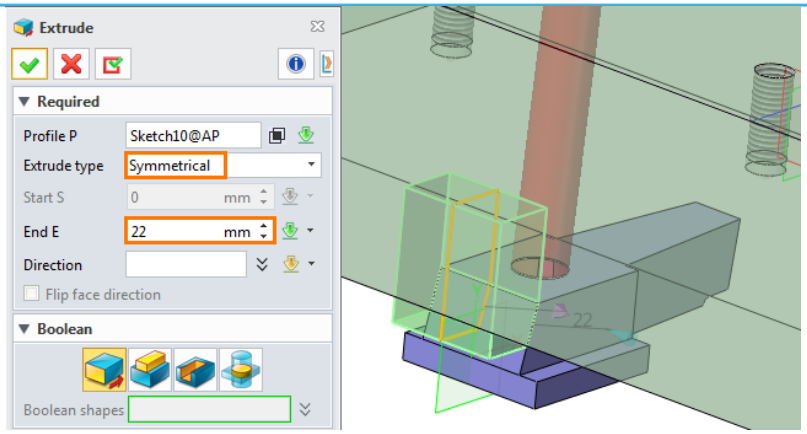


Figure 207 Extrude the Sketch

Add fillets on vertical edges of the block.

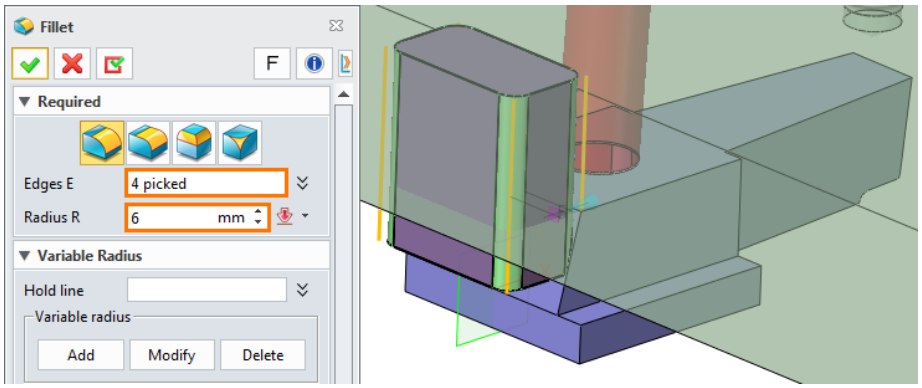


Figure 208 Add Fillets

Use ***Remove Shape*** command to remove the block from ***AP***.

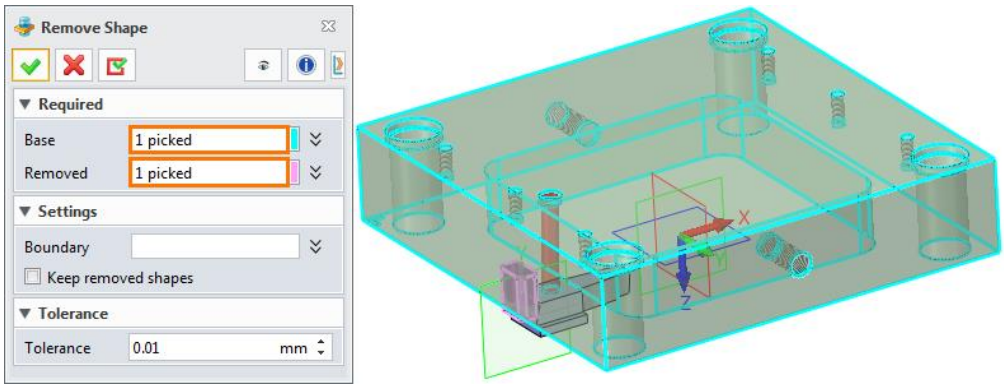


Figure 209 Use “Remove Shape” to Create a Pocket

Use ***Geom to Part*** command to extract the ***tight block*** as a component of ***AP***.

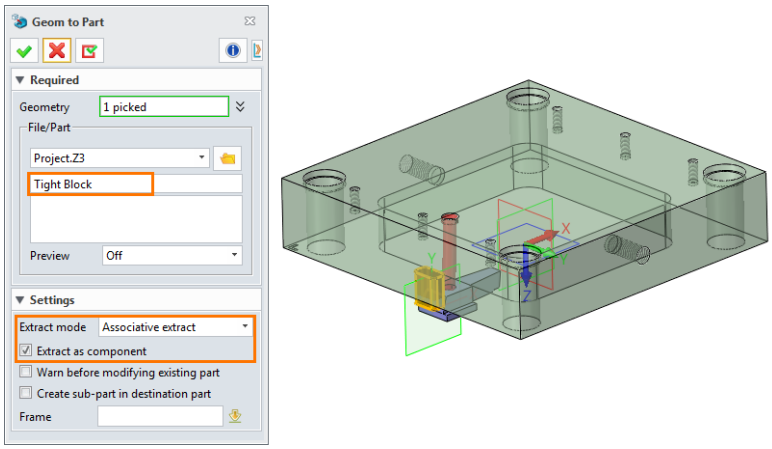


Figure 210 Extract the “Tight Block”

Use ***Screw*** command to insert a screw to lock the block.

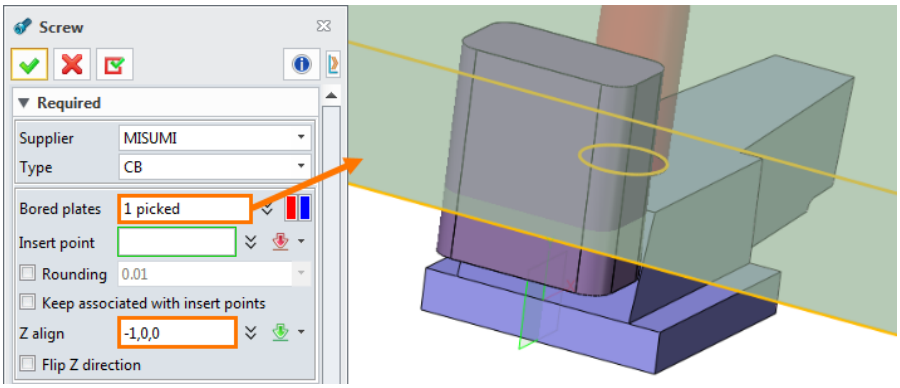


Figure 211 Insert a Screw

Pick ***Insert point*** by choosing ***Between***.

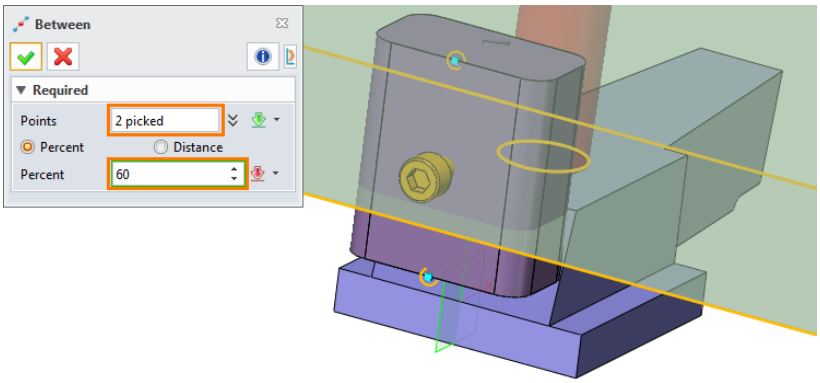


Figure 212 Pick “Insert point” by Using “Between”

Set main parameters as below, and other parameters as default.

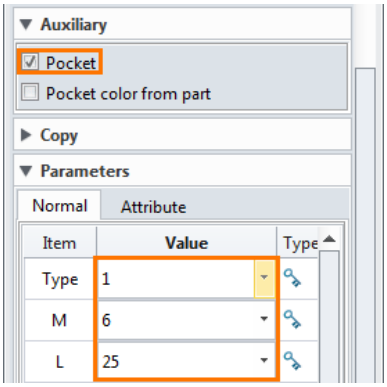


Figure 213 Main Parameters of the Screw

Use ***Erase*** command in toolbar to erase the referenced slider in ***AP***.

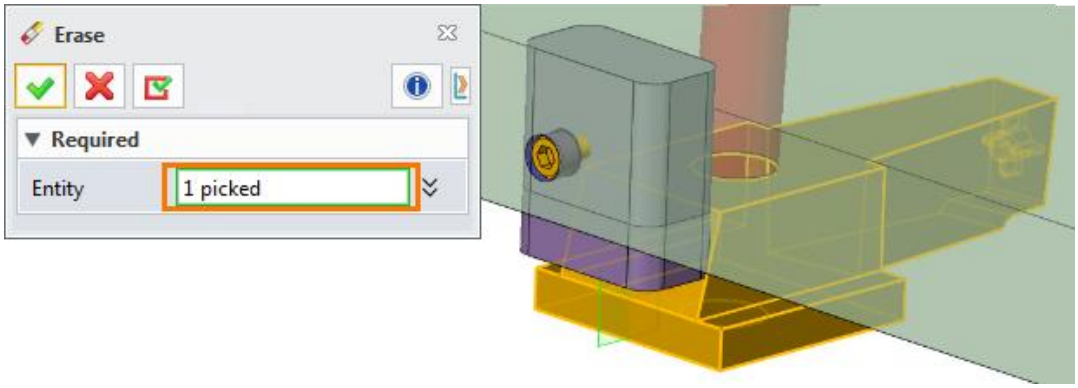


Figure 214 Erase the Referenced Slider

Return to ***000_Mold_Assembly***, ant you can get the result as Figure 215.

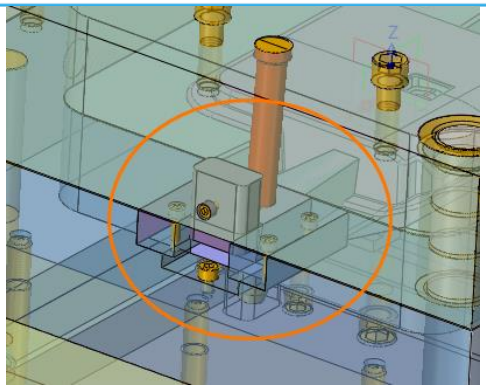


Figure 215 General Result of the Slider

STEP 07 Apply the same operations on the other Slider. The final result shows as below.

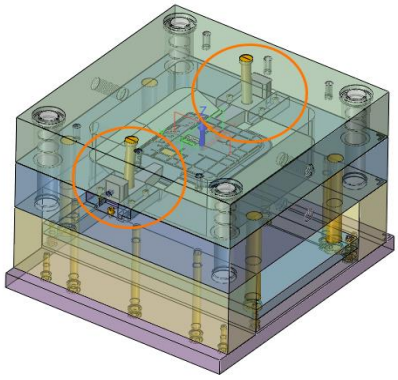


Figure 216 Final Result of Sliders

1.7.2 Insert Locating Ring and Sprue Bush

STEP 01 Use ***General*** command to insert a location ring on ***TCP***.

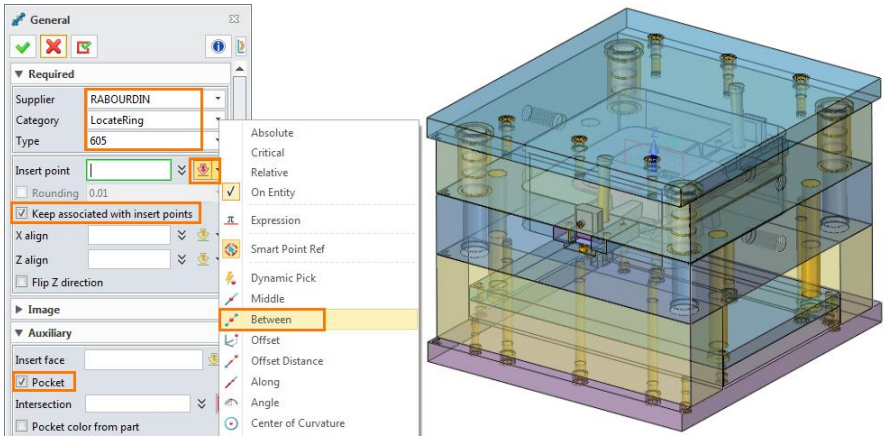


Figure 217 Insert a Location Ring

Choose “Between” to select the center point.

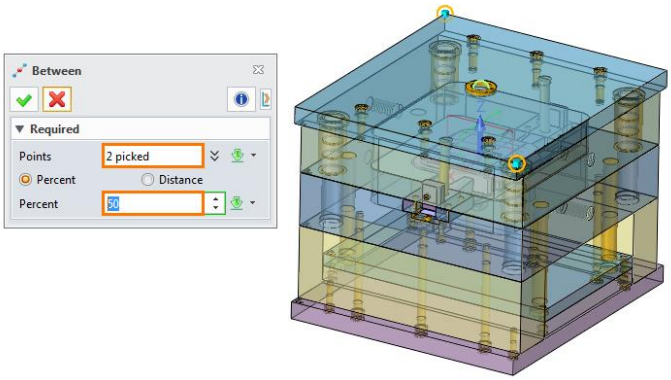


Figure 218 Pick the Center Point by “Between”

Set main parameters as below. Click ***OK***.

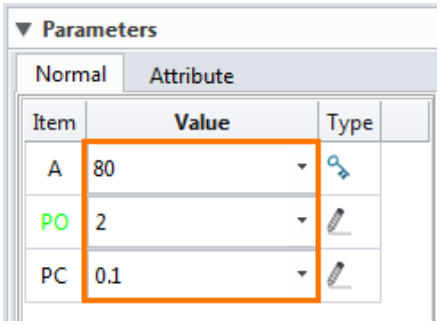


Figure 219 Main Parameters of the Location Ring

STEP 02 Continue to use ***General*** command to insert the sprue bush.

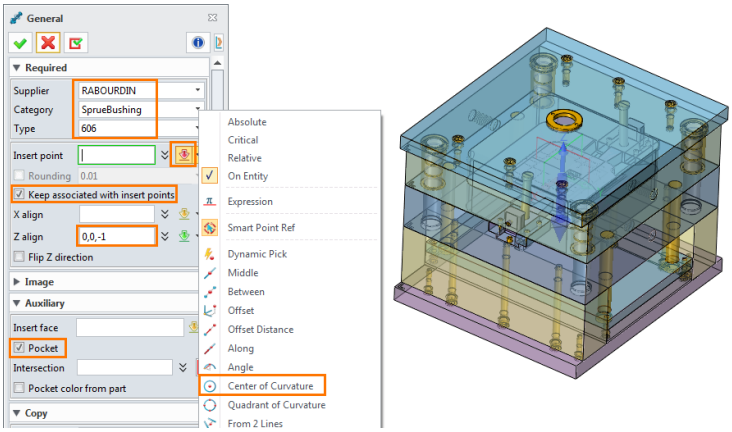


Figure 220 Insert the Sprue Bush

Use ***Center of Curvature*** in the right menu to pick the center point.

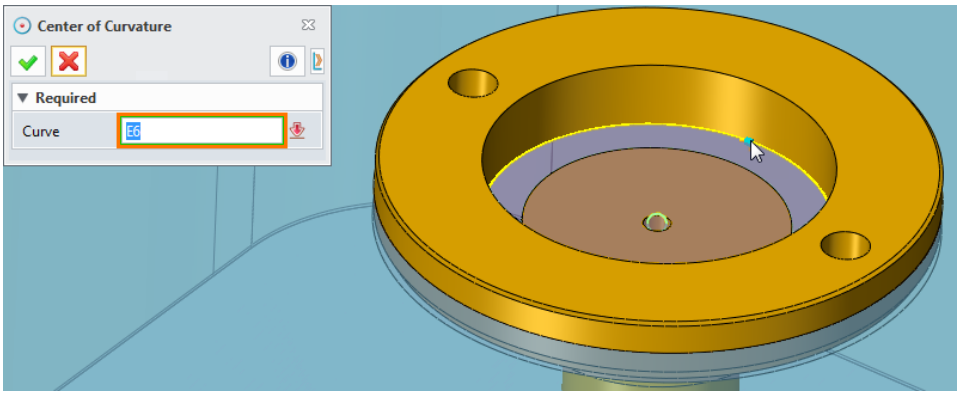


Figure 221 Pick the Center Point of the Location Ring

Set main parameters as below. Customize the parameter “P0” to ensure the spruebush can touch the cavity.

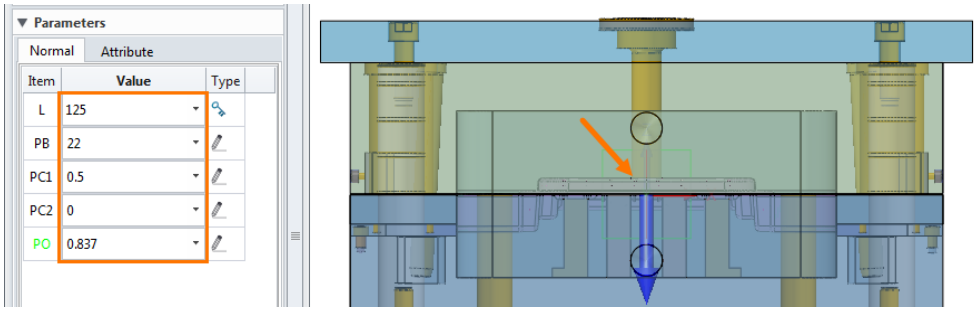


Figure 222 Main Parameters of the Sprue Bush

1.7.3 Insert Elector Pins

STEP 01 Double click on ***000_Mold_Assembly***. Hide the fixed half and the cavity.

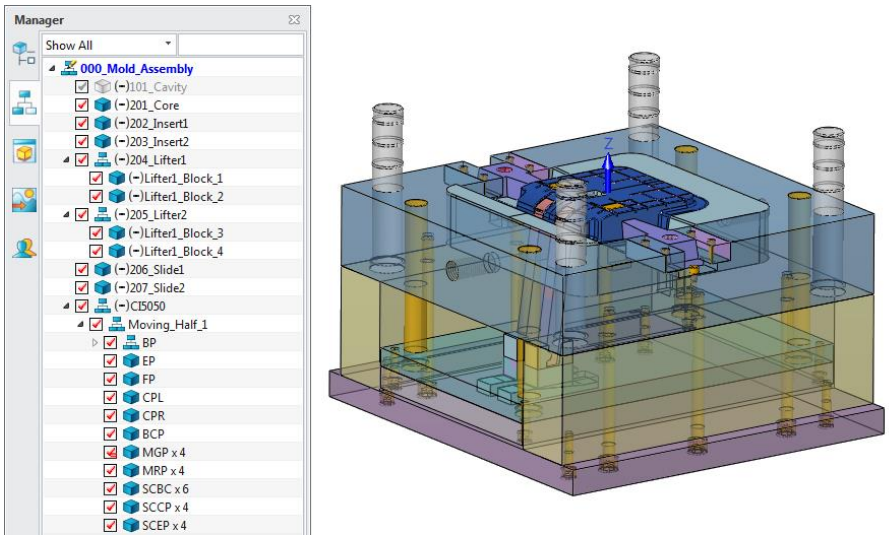


Figure 223 Activate "000_Mold_Assembly"

STEP 02 Use ***Sketch*** command in ***Mold*** tab to create a sketch on XY datum to mark the position of the elector pins.

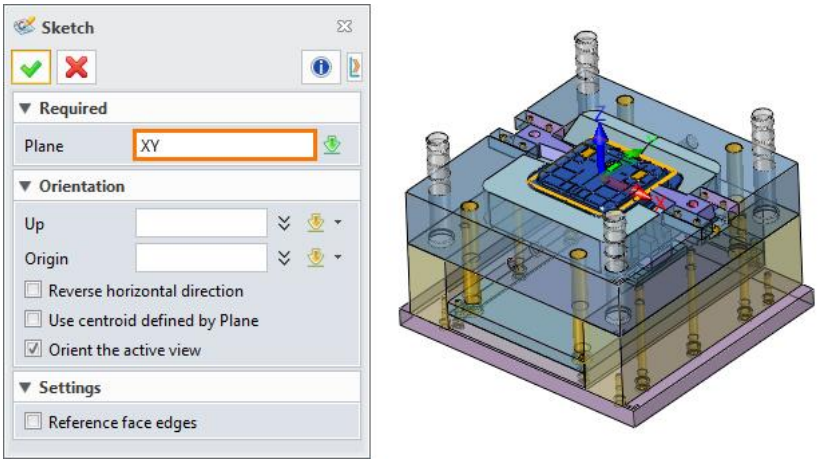


Figure 224 Create a Sketch on XY Datum

STEP 03 Draw the sketch as shown in Figure 225.

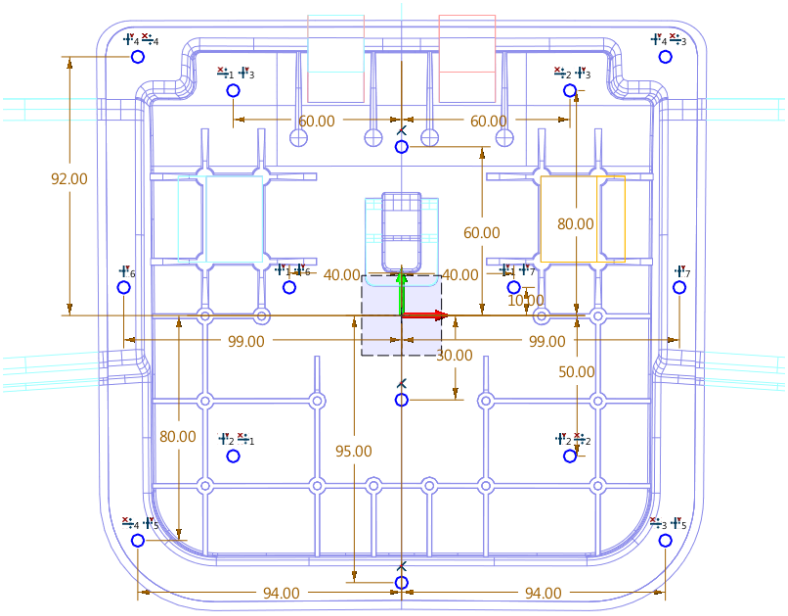


Figure 225 Well-Defined Points in Sketch

STEP 04 Use ***Epin*** command in ***Mold*** tab to inser ejector pins.

Pick the points of the sketch as ***Insert point***.

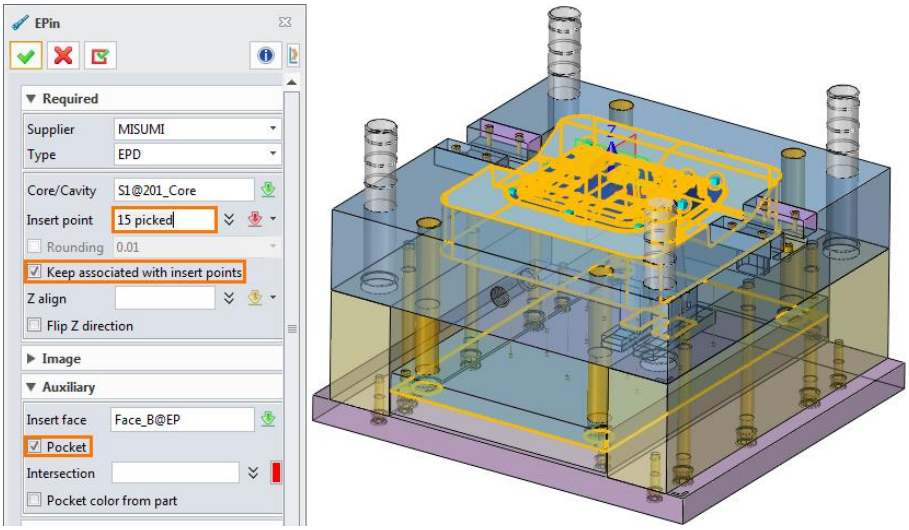


Figure 226 Insert Ejector Pins

Set parameters as below.

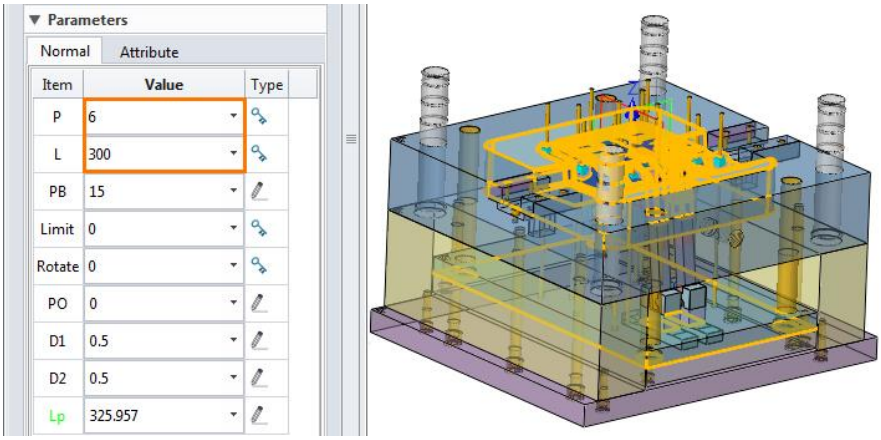


Figure 227 Parameters of Ejector Pins

STEP 05 Trim the elector pins.

Use ***Insert*** command in ***Assembly*** tab to insert the ***006_Epin_Cutter***.

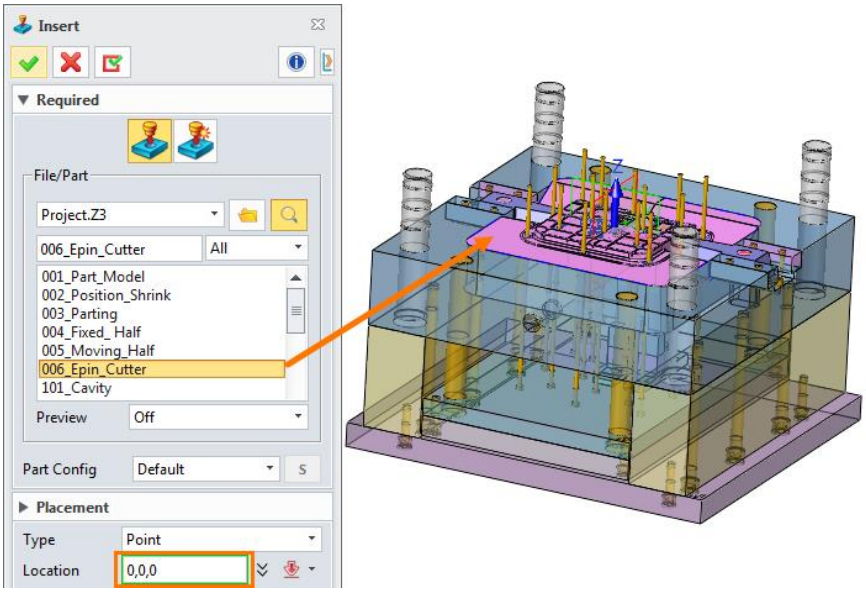


Figure 228 Insert the “006_Epin_Cutter”

STEP 06 Use ***Trim Pin*** command to trim the elector pins. Pick all elector pins, then choose the

006_Epin_Cutter as Cutter.

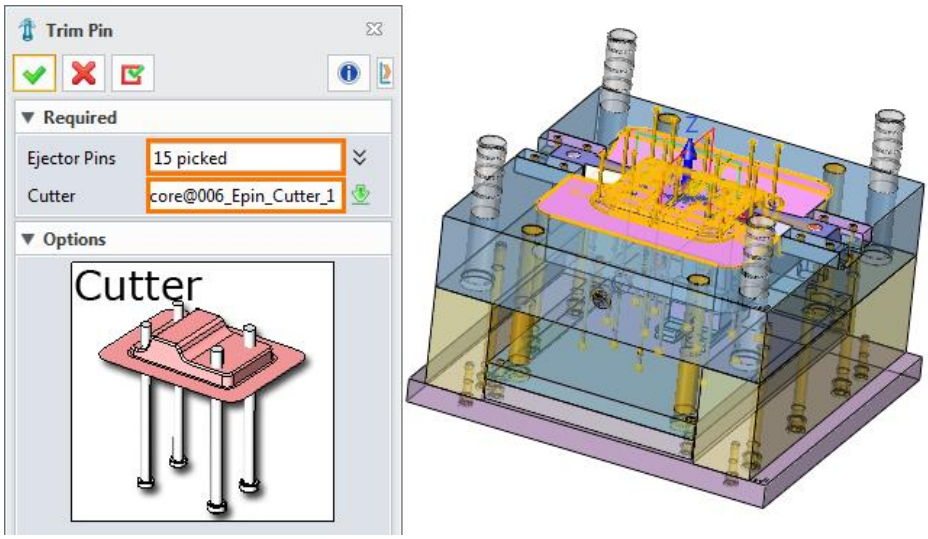


Figure 229 Trim Ejector Pins

The below image is the result.

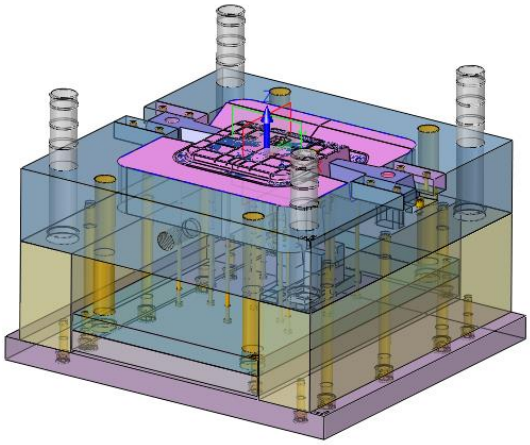


Figure 230 Trim Result

STEP 07 Hide the ***006_Epin_Cutter***.

1.7.4 Insert Pillars

STEP 01 Activate ***000_Mold_Assembly***.

Create a sketch on XY Datum.

Draw the sketch as below image.

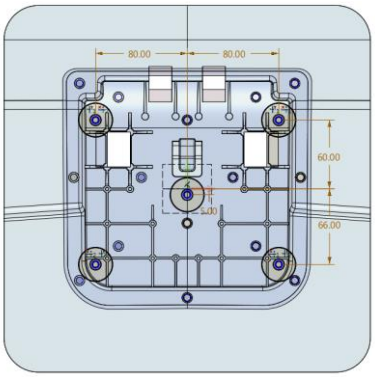


Figure 231 Well-defined Points in Sketch

STEP 02 Insert ***Support Pillar***.

Use ***S.P***. command in ***Mold*** Tab. Pick the points of the sketch.

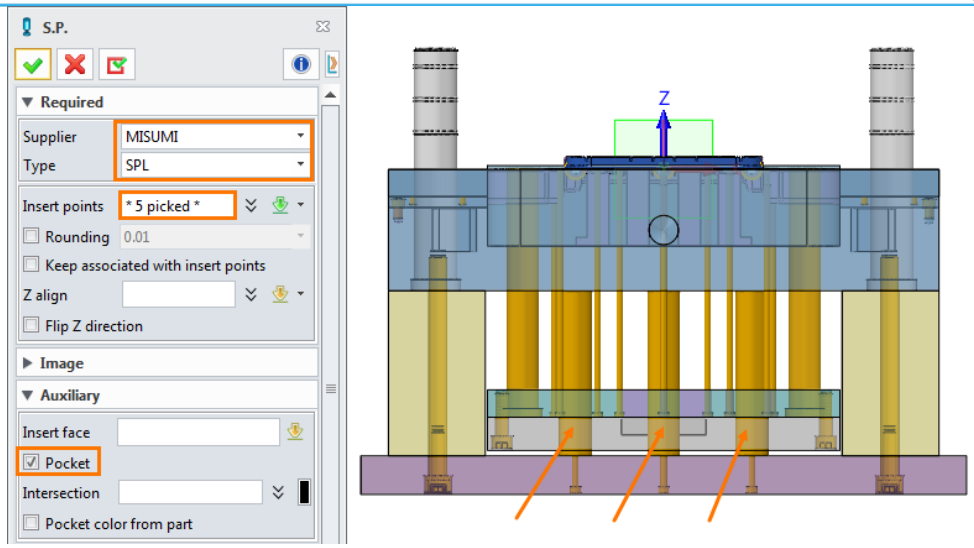


Figure 232 Insert Support Pillars

Up to now, we have finished inserting mold components. Finally the assembly shows as Figure 233.

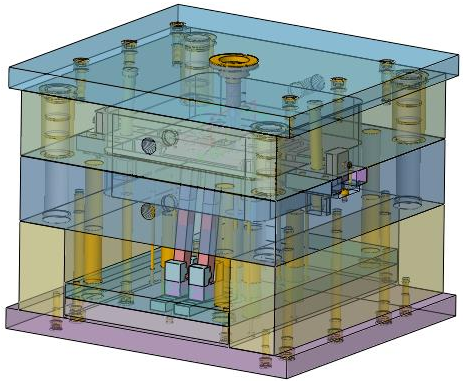


Figure 233 General View of Final Result

Notes: *So far, we have added screws, angle pins, location ring, sprue bush, ejector pins and support pillars in the library. However, we still need to add more libraries for the whole 3D mold design, including positioning, spring, insulation plate, wear plates, and so on.*

1.8 Bill of Materials

STEP 01 Use ***BOM*** command in "Mold" tab. It will automatically create a BOM table for the whole mold assembly.

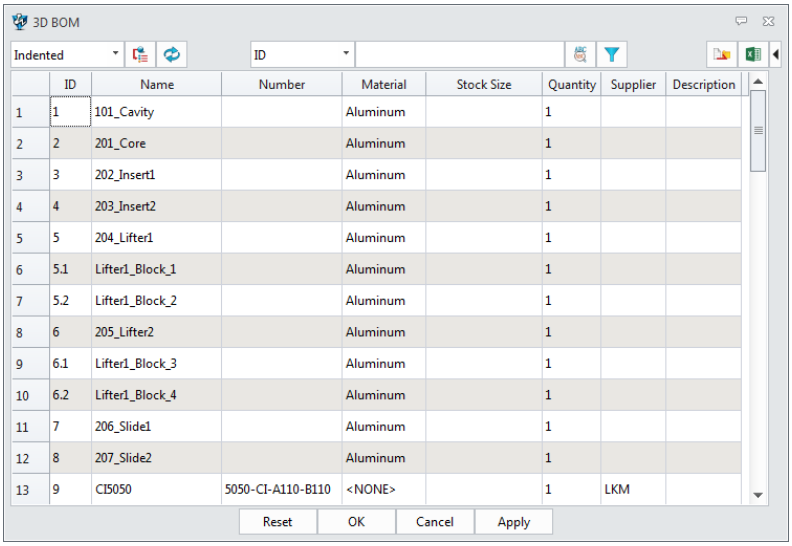


Figure 234 Bill of Materials

STEP 02 Stock Size calculation.

Right click the ***Stock Size*** tab in the table. Pick ***Auto Stock Size***.

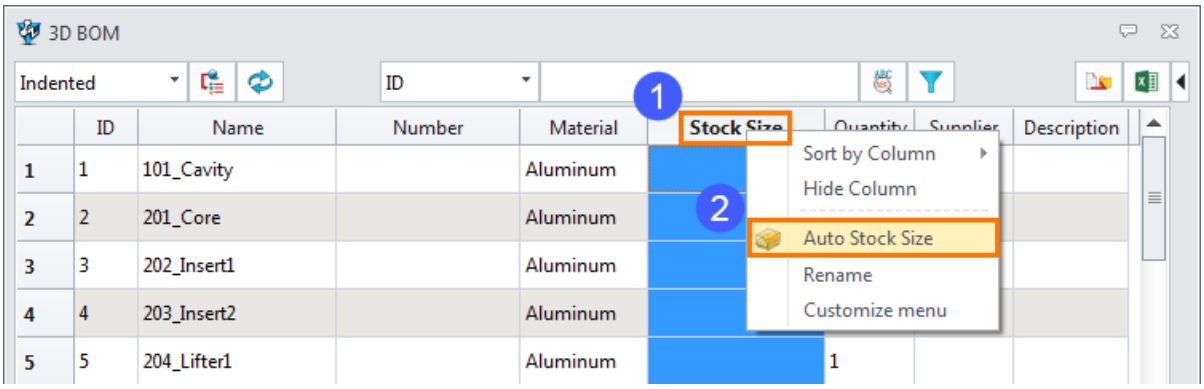


Figure 235 Auto Stock Size

Select the type as ***Block***, click ***OK***.

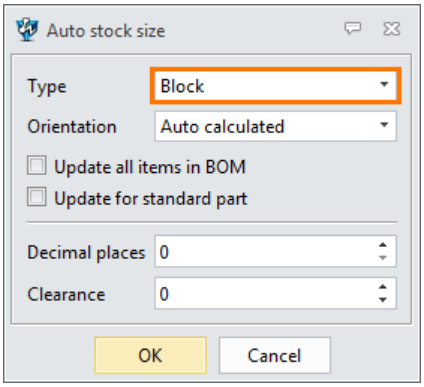


Figure 236 Setting in “Auto Stock Size”

After that, stocks of all components will be created.

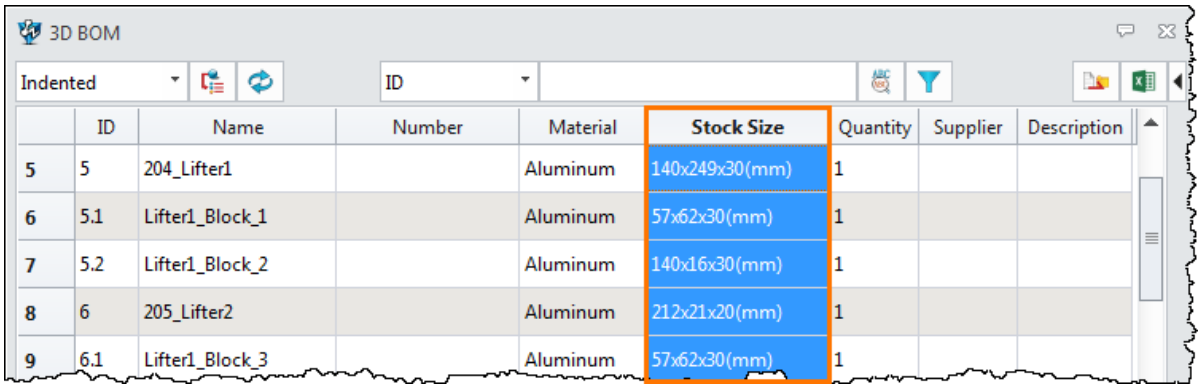
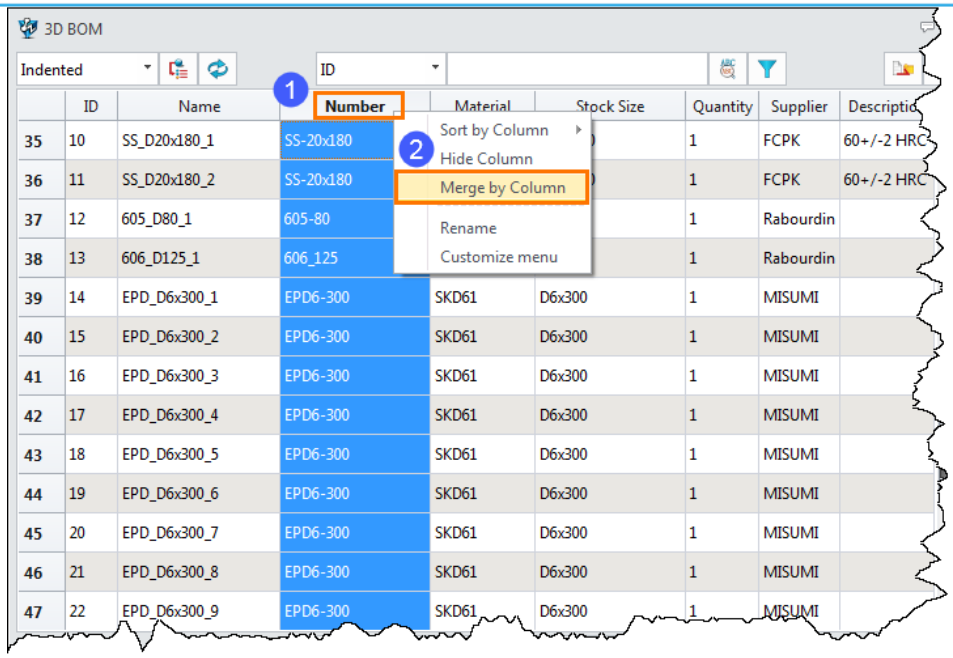


Figure 237 Result of “Auto Stock Size”

STEP 03 Merge components with the same ***Number***.

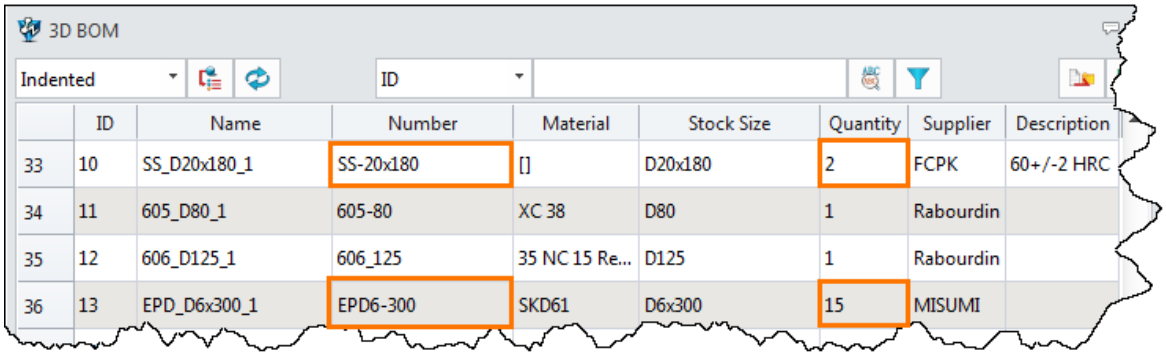
Right click the ***Number*** tab, then select ***Merge by Column***.



	ID	Name	Number	Material	Stock Size	Quantity	Supplier	Descriptic
35	10	SS_D20x180_1	SS-20x180			1	FCPK	60+/-2 HRC
36	11	SS_D20x180_2	SS-20x180			1	FCPK	60+/-2 HRC
37	12	605_D80_1	605-80			1	Rabourdin	
38	13	606_D125_1	606_125			1	Rabourdin	
39	14	EPD_D6x300_1	EPD6-300	SKD61	D6x300	1	MISUMI	
40	15	EPD_D6x300_2	EPD6-300	SKD61	D6x300	1	MISUMI	
41	16	EPD_D6x300_3	EPD6-300	SKD61	D6x300	1	MISUMI	
42	17	EPD_D6x300_4	EPD6-300	SKD61	D6x300	1	MISUMI	
43	18	EPD_D6x300_5	EPD6-300	SKD61	D6x300	1	MISUMI	
44	19	EPD_D6x300_6	EPD6-300	SKD61	D6x300	1	MISUMI	
45	20	EPD_D6x300_7	EPD6-300	SKD61	D6x300	1	MISUMI	
46	21	EPD_D6x300_8	EPD6-300	SKD61	D6x300	1	MISUMI	
47	22	EPD_D6x300_9	EPD6-300	SKD61	D6x300	1	MISUMI	

Figure 238 Merge Rows by Number

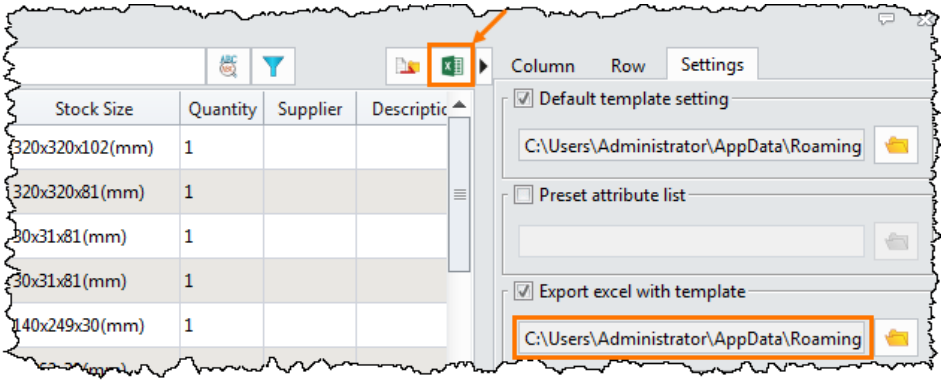
The components with the same ***Number*** will be merged together.



	ID	Name	Number	Material	Stock Size	Quantity	Supplier	Description
33	10	SS_D20x180_1	SS-20x180	[]	D20x180	2	FCPK	60+/-2 HRC
34	11	605_D80_1	605-80	XC 38	D80	1	Rabourdin	
35	12	606_D125_1	606_125	35 NC 15 Re...	D125	1	Rabourdin	
36	13	EPD_D6x300_1	EPD6-300	SKD61	D6x300	15	MISUMI	

Figure 239 Merge Result

STEP 04 Use ***Export data*** button to export the BOM table as a excel file with preset template.



Stock Size	Quantity	Supplier	Descriptic
320x320x102(mm)	1		
320x320x81(mm)	1		
30x31x81(mm)	1		
30x31x81(mm)	1		
140x249x30(mm)	1		

Figure 240 Export the BOM

The final result shows as Figure241.

2	ZWSOFT						
3	Bill of Material						
4	Part Number			Designer		Date	2018/12/21
5	ID	Number	Name	Material	Stock Size	Quantity	Remark
6	1		101_Cavity	Aluminum	320x320x102(mm)	1	
7	2		201_Core	Aluminum	320x320x81(mm)	1	
8	3		202_Insert1	Aluminum	30x31x81(mm)	1	
9	4		203_Insert2	Aluminum	30x31x81(mm)	1	
10	5		204_Lifter1	Aluminum	140x249x30(mm)	1	
11	5.1		Lifter1_Block_1	Aluminum	57x62x30(mm)	1	
12	5.2		Lifter1_Block_2	Aluminum	140x16x30(mm)	1	
13	6		205_Lifter2	Aluminum	212x21x20(mm)	1	
14	6.1		Lifter1_Block_3	Aluminum	57x62x30(mm)	1	
15	6.2		Lifter1_Block_4	Aluminum	140x16x30(mm)	1	
16	7		206_Slide1	Aluminum	129x60x35(mm)	1	
17	8		207_Slide2	Aluminum	129x60x35(mm)	1	

Figure 241 BOM with Template in Excel

1.9 2D Drafting

Use ***2D Sheet*** command in toolbar to create a general drafting for the mold. Pick the template ***A0_H(ANSI)***.

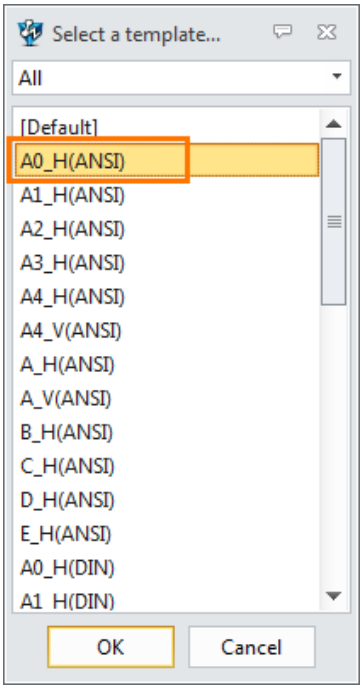


Figure 242 Create a 2D Sheet with a Template

Create views as below.

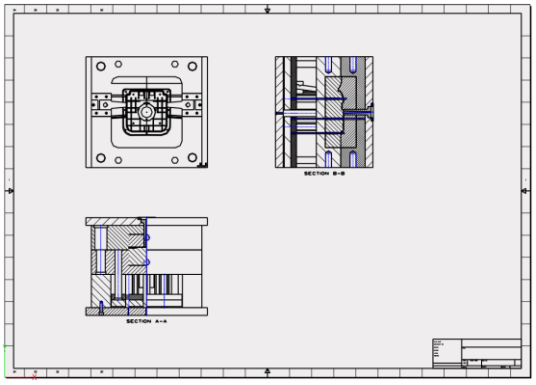


Figure 243 Views of the Mold Assembly

Use ***BOM*** command to create a BOM table.

Inherit 3D BOM.

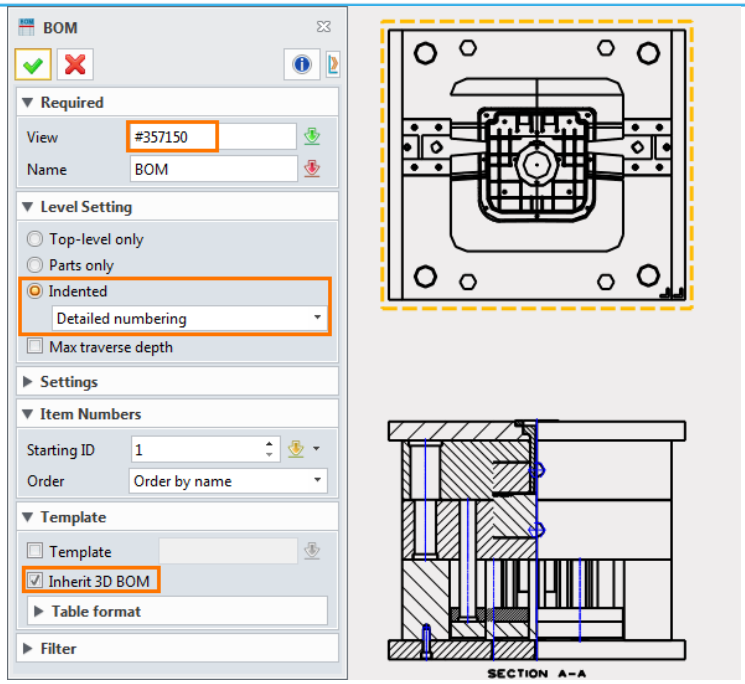


Figure 244 Inherit the 3D BOM

Pick the Bottom-Right, then pick the point as shown in figure 110.

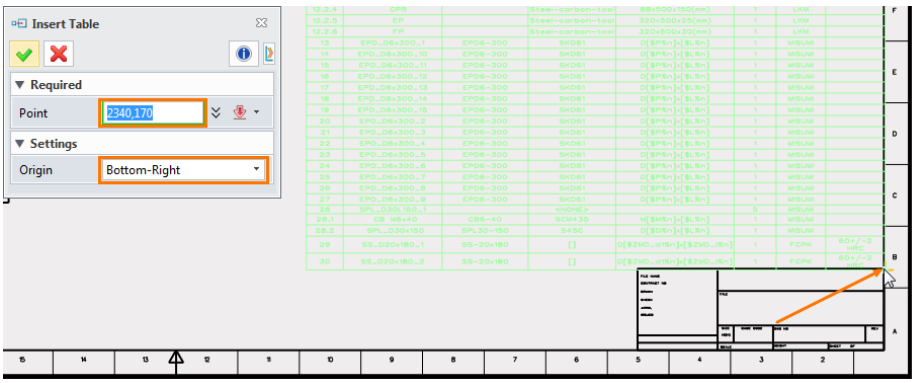


Figure 245 Locate the BOM Table

The final result shows as below.

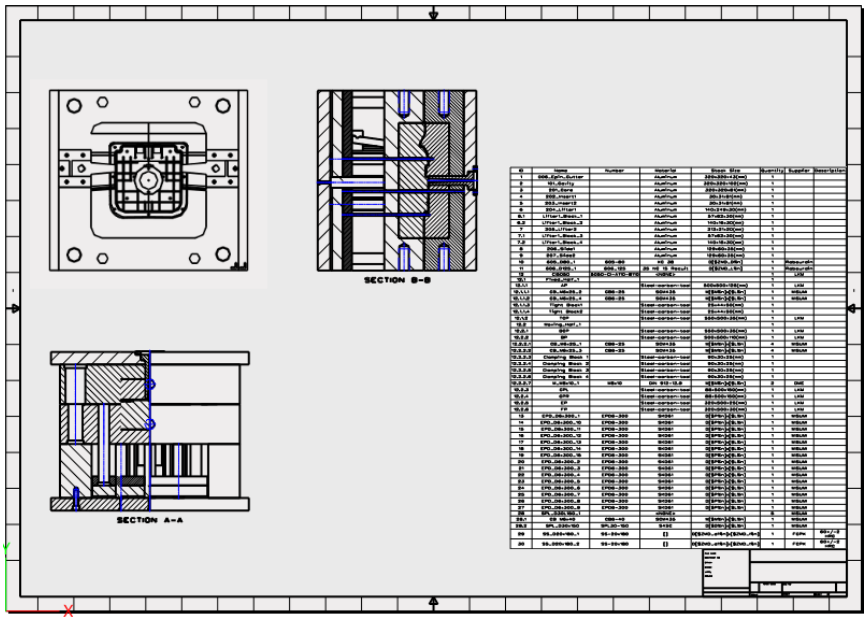


Figure 246 Final Result of the 2D Sheet

After that, user can continue to create 2D sheet for all components.

2 Epilogue

So far, we have finished this course, which shows most of functions of ZW Mold functions. Of course, we need to make more steps to finish the mold design and manufacture, such as coolant systems, mold features, mold flow analysis, electrode design, mold plate manufacturing and so on. At present ZW3D has its independent electrode design and powerful 2X and 3X milling for mold plate manufacturing. More mold features like new coolant system design, runner design, more powerful sliders and lifters library are coming soon.