

ZW3D
Mold Advanced Tutorial
(Suitable for 2014 SP1)

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ZW3D™ V2014 Mold Advanced Tutorial

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Forward

The full version of the ZW3D documentation is available by selecting **Help Browser** from the ZW3D **Help Menu**. Go to the "**\ZWSOFT\ZW3D2014Eng\doc**" directory on the ZW3D program directory and open the file "**ZW3D.chm**".

This **Mold tutorial** is available as a printed manual and in PDF format. To view or print the PDF version, install the Adobe Acrobat Reader and open the corresponding files in the "**\ZWSOFT\ZW3D2014Eng\PDF**" folder in the program directory. Refer to the **HELP Manual** for information relating to specific tasks.

NOTES:

1. For best results, view and print the PDF version of the document using the Adobe Acrobat Reader.
2. Since the **Mold tutorial** is only a part of the complete manual, it is highly recommended that you utilize the **HELP Manual**. You will find additional topics, more information on command options, input options, additional command notes, and cross-links to related tasks and topics.
3. It should be noted that the **HELP Manual** will always be more up-to-date than any printed or PDF material. If you find any discrepancies between the two, always refer to the **HELP Manual**.
4. While you are working with ZW3D, simply press **F1** to view information about the command you selected.

To help you customize your CAD/CAM requirements, please contact our Customer Support department on our website <http://www.zwsoft.com> or call at (8620)38289780 ext 565. You can also contact your local sales representative or application engineer to discuss your requirements.

Thanks for being our customer!

The ZW3D Team

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Chapter 1 Importing and Creating Geometry

NOTE: This exercise is done in **MM** Units. It would simplify things if you were to edit the configuration, set your default units to MM and add a default part template called **PartTemplate(MM)**. You should see this template on the Create New (Part) menu, but you wouldn't have to pick it each time we create a new part object.

Select **Configuration** from the **Utilities** pull down menu or direct click the Configuration button  from Ribbon tab.

In the General tab, set the **Default linear units** to “mm”, and the **Max undo steps** to “200”.

Pick the Files tab, Click the folder icon for the **Default Part template**, and choose the **PartTemplate(MM)** from the list.

Let's create  a **Multi-Object** file  called “**Project_554.Z3**”.

Create  a **Part**  object called “**001_Model_Part**”

Now that we are in the part, we can verify what units we will be working in. To do that, we will select **Edit**, pull down menu on the **Utilities** toolbar and then choose the **Preferences** option.

The default system tolerance is 0.01 and we will keep that value for this exercise.

The Part settings menu allows us to configure the following:

Units – Option that allows the user to define work units.

Alternate Display of Free Edges – Option that allows visualization of openings and “**Gaps**” in the 3D model.

*******NOTE:** This option is the same as **View manager->Open Edge->Display**, Turn it on on the following exercise

Auto Sew On Face Creation– Option that indicates if the new faces created by the user are automatically sewn to the surrounding shapes.

*******NOTE:** Since 2014 version “**Auto Sew**” option have been controled inside each command, this option is invaild andonly used to keeping compatibility with the old version. So here you can ingnore this option.

Now, let’s import an IGES file. To do that, select, **File** and go to **Import**.

This opens the **IGES Import** dialog so that you can define settings for the file that you are importing. We will select the IGES option for importing.

Pick **OK**, then we can go to ‘IGES Import Settings’ where it is possible to define import parameters.

Import from - Select the file “**Backcover_554_Rev_04.igs**”, which is located in the “**\ZWSOFT\ZW3D 2014Engltraining**” installation directory.

Auto Sew Geometry – Indicates if the geometry is to be automatically united.

Auto Activate Part – Indicates that after importing, the geometry is automatically activated. This command is only necessary if the importing is done at the object level of the work session.

Rename Layer Collisions – Renames the layers if they already exist with the same name.

Remove Duplicate Surfaces – If there are duplicate trimmed surfaces, the duplicate is removed.

Break Trim Edges - Will analyze the edges of the surface and break the edges on the tangent points if needed.

Reset Dependencies - It is not uncommon for third party IGES files to contain incorrect dependency flags causing entities to not import, or to import more than once. Use this option to flush the dependencies reported in the file, and recreate them based on the entity references from the parameter section of the IGES file.

Backup import data– If this option is turned on, the import data will be stored in **Sub Part**

Force close - Use this option to force the closure of the imported part if possible. Some healing techniques may be used, such as trimming dangling edges or spanning missing surfaces.

Note: This option is not recommended unless you are sure the geometry should be a closed solid.

Drawings – Import the 2D lay-outs included in the file for the 3D environment.

Bounded Planes – Import planes with limits.

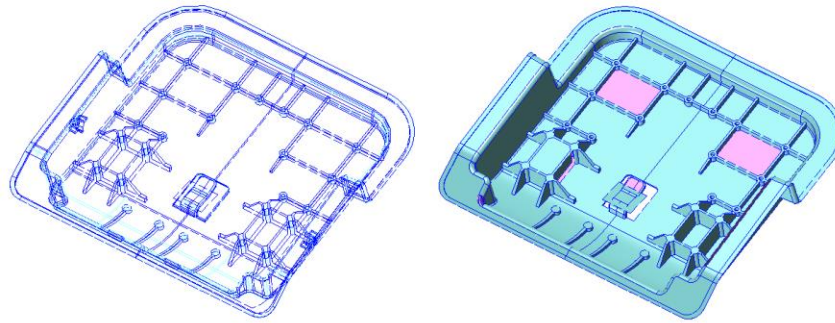
Unbounded Planes – Import planes without limits.

Read hidden entities – Import the hidden entities

*******NOTE:** For this exercise, deactivate the **Auto Sew Geometry** option

Select **OK**.

After you have finished importing the IGES file, the object should look like this:



Your model has all the edges of the surfaces dashed, because they were not sewn during the import.

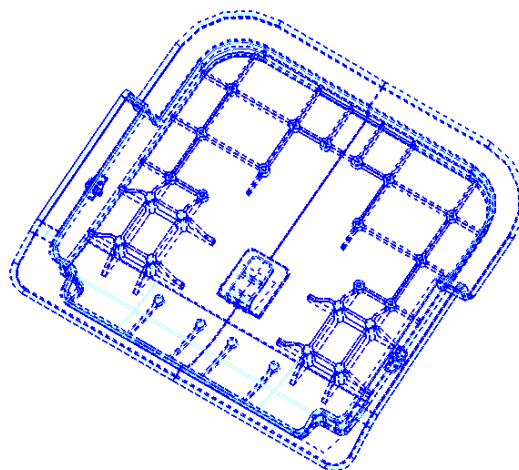
The quality of the imported geometry can be improved by “healing”.

Select the **Heal** tab, then choose the **Heal/Analyze** command.

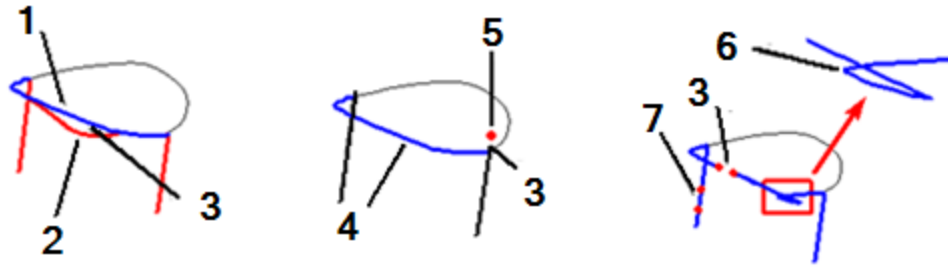


This command defines the tolerance for correction. Keep the **Tolerance** at **0.01** for now.

The result should be “Vertices not changed. Edges not changed. Loops not changed.”



The image below shows some examples of problem geometry.



- 1: Edge Curve 2: Adjacent Face 3: Gap 4: Incident Edge 5: Vertex Point 6: Self-intersecting Loop
7: Tiny Edge

During the import we deactivated the auto sew command. Now let's join the faces using **Sew**. To do this we need to display the **Free Form** tab as shown below. The icons on this tab are the tools for building and manipulating individual surfaces. Refer to the help document for more about each command.

A few commands are described below.



N-Sided Patch: Use this command to create a face by patching across 3 or more profiles. The profiles can be wireframe geometry, sketches, or face edges. Required inputs include the profiles to define the patch.



Intelligent Blend: Use this command to create an intelligent blend face. Required inputs include the curves, edges, or faces that the new face will start from, go to and pass through. Optional inputs include using curves, faces or a datum for spine control, circular and conic cross-sections, sewing, capping and the ability to give the face a unique name if desired.



Modify Numbers of Iso lines: Use this command to set the number of iso lines displayed in the U and V directions of a face. You can also set iso lines with the Modify Face command.



Explode Faces: Use this command to separate faces from a shape. Use the **Group** option to maintain the connectivity of the explode faces (i.e., they become a separate shape). Use the **Sew** command to join faces back into a shape.



Trim to Faces: Use this command to trim away part of a face or shape where it intersects with other faces, shapes, and/or datum planes.



Extend a Face: Use this command to extend one or more edges of a face. First select the face. Then select the edges to extend and enter the distance to extend.

Select the **Sew** command. 

Click **OK** to select all faces. You can change the value of the tolerance to be different from that of the Part Tolerance. **Don't change it at this time.**

Suggestions to help determine the scope of the needed repairs:

1. Put your **Pick Filter** on **Shape**. Window pick the entire screen.

The message window will report how many separate **shapes** have been imported.

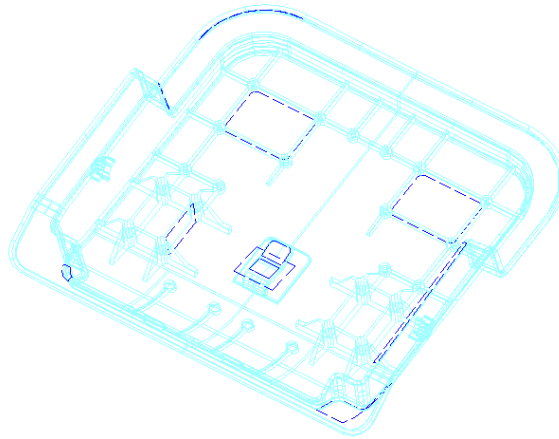
Or you can directly check the history manager, the Entity folder will show the quantity of Surface or Solid. You can use this to judge when you will get the perfect solid.

2. Right-click on the biggest shape that highlights and pick **Attributes**. Change the **face color** to the darker of the two light blue colors. This way you can visually see the other shapes and it also may give you an immediate idea of how to approach the repairs. You could also temporarily blank this shape to see what's left.

Your model should have most of the edges sewn. Do you notice the difference in appearance between the sewn edges and the open edges?

Whenever the surfaces are missing or have “Gaps”, the edges will be **blue and dashed**.

The result should be “ Number of unmatched edges is [61]. Maximum gap between unmatched edges is [0.011831mm].”



After sewing, the system will give you information about the largest **Gap** existing on the geometry. The user can refer to that value to redefine the **Sew** command using it as the new tolerance. You can use the **Heal** command again to correct the geometry with the intended tolerance. We will do this after we are finished repairing the model.

Use the command **Concatenate**  from **Heal** tab

Keep the **Vertex** input field empty, and pick **OK**.

*******HINT:** Most entity selection situations in ZW3D will pick all as the default setting if you keep the input field empty.

This will find colinear edges and combine them into one. **Make this command a regular part of your everyday Healing**

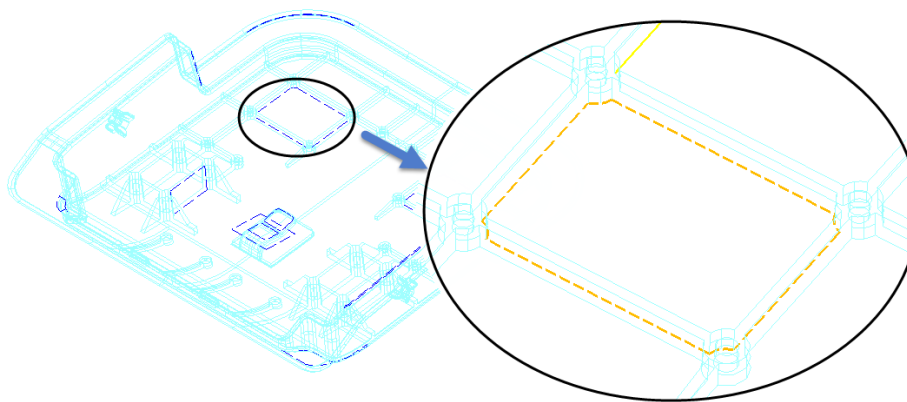
Creating the missing surfaces is the next step.

For this exercise let's use the “**Trimmed Plane**” command  from **Free Form** tab. This command permits the construction of surfaces from 2 or more edges and/or curves.

Select edges shown in the image. <**middle-click**> to “accept” the edge selections.
<**middle-click**> again to “finish” the command.

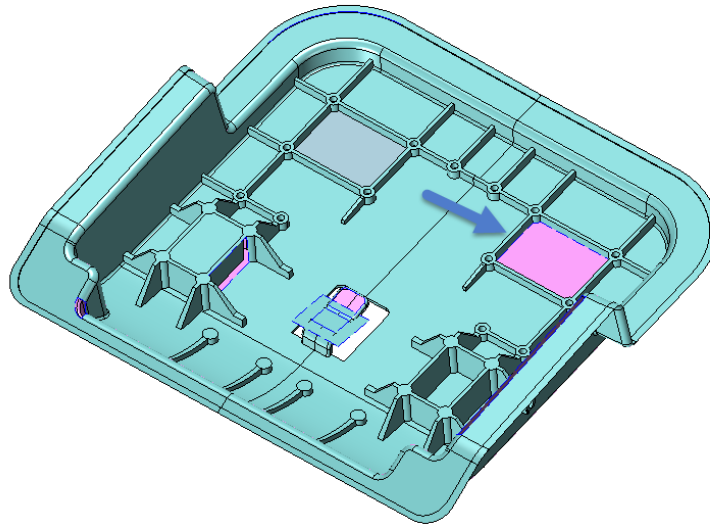
*******NOTE: The bottom faces of the pockets are missing. When the part is shaded, the red face you see is actually the inside of the back face of the part.**

To verify that a face is missing as opposed to a face that has its normals reversed, zoom in close to one of the edges and window pick the edge and look at the message window. Your **Filter** must be set to **Edge**. This will give you a count of the edges found. One edge means a face is missing.

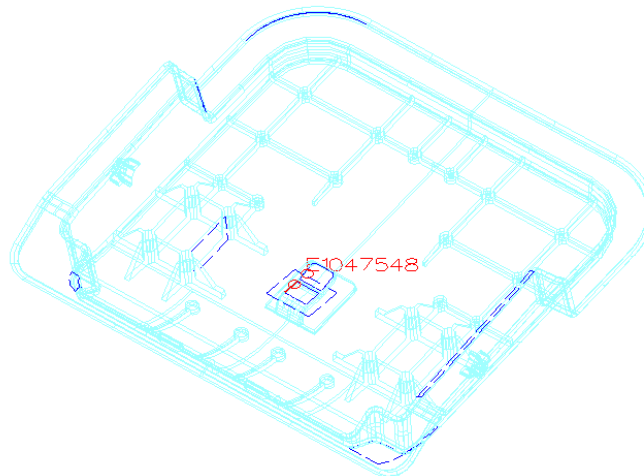


For the next surface we will use another command which will also allow us to create a surface on the free edges. These tools are found on the “**Heal**” Tab.

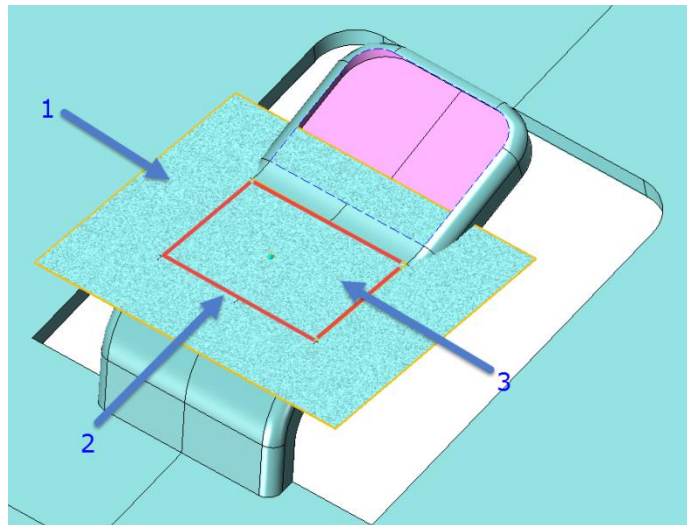
Select the “**Fill Gap**” command . Select an open edge as shown below and then <middle-click> to fill and confirm.



This menu also contains a command which helps us detect problems with the geometry. Select the command “**Show Open Edges**” . This command lists the number of free edges. It also displays a form, which allows you to “**Zoom To**”  to better identify the problem area.

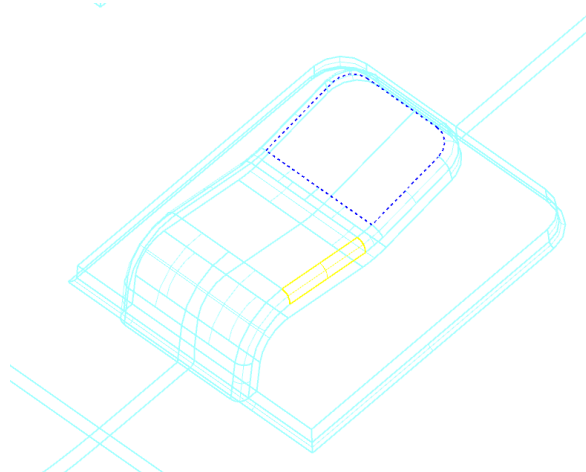


Now we will show you how to solve the problem. In this situation we can use the command **"Trim to curves"**  (Free Form Tab). To use this command, first select the surface to be trimmed. Next select the edges/curves for that trim. (Use the edges shown in the image). Finally, indicate the side you wish to keep.

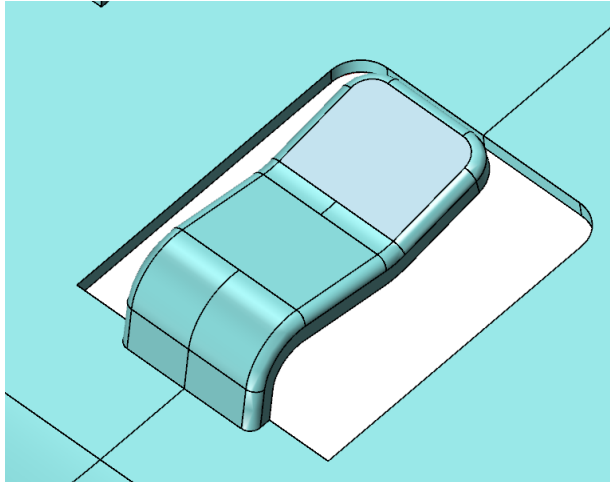


1:Face 2:Edges – Pick 6 3:Keep area

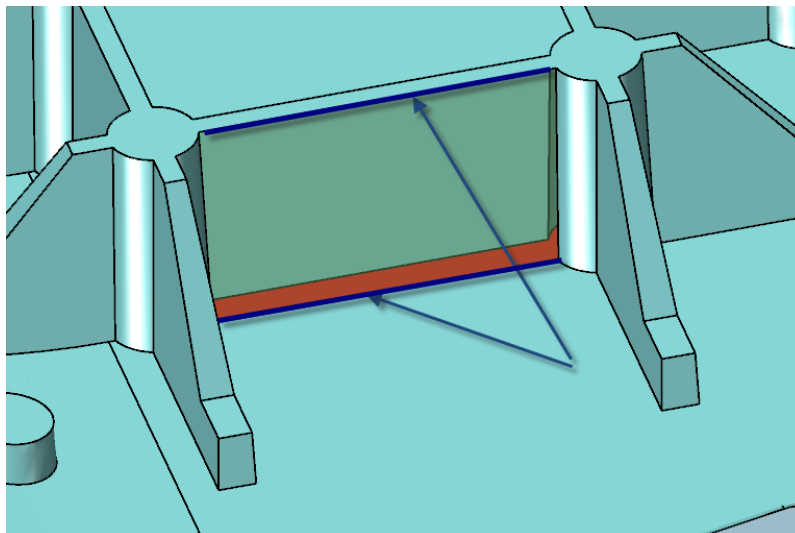
Trimming a surface does not automatically sew. Select the **Sew** command  (Heal Tab), and choose **OK**.



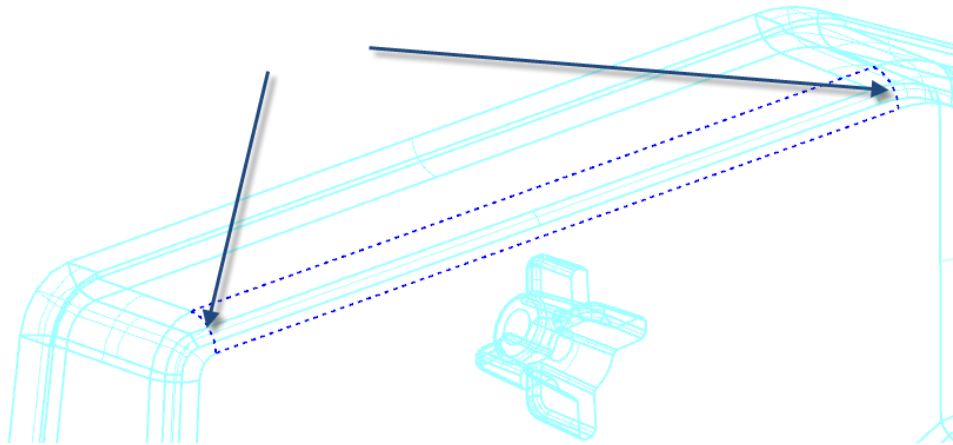
Use the **Fill Gap** command  to create the next surface and select an edge. This is shown in the image below.



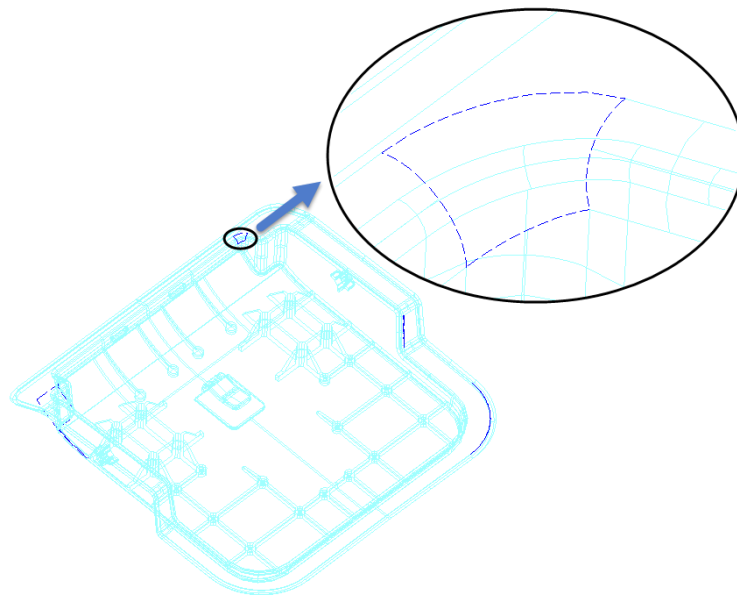
Find the problem area shown below. Select the **Ruled** command  (Free Form Tab), for the construction of this surface. Just select the two edges as shown in the image. Select both edges on the same end to avoid creating a twisted surface. (generally referred to as a “bow-tie” effect)



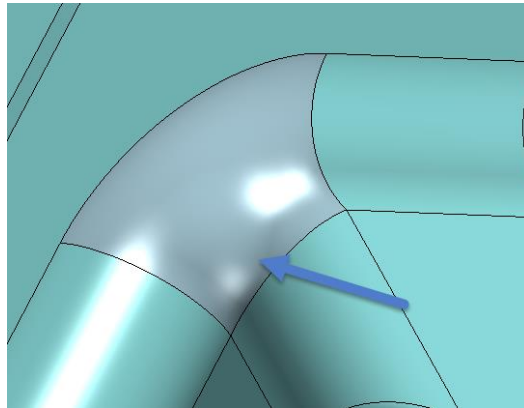
The **Ruled** command will not only create flat surfaces, but will also create surfaces supported by two curved edges. Select the arcs as shown in the image below. This type of surface creates straight line rulings between each point on the two curves selected.



Once again use the command **Inquire Open Edges**  to locate the open edge shown on this image.



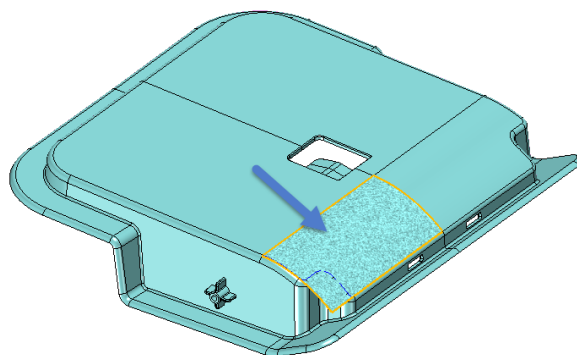
After locating the problem, use the **Fill Gap** command. Activate “**Tangent to gap boundary**” and set “**Refit**” to Tangent. Select one of the edges. You will now see a new surface that is tangent to the surfaces from which it originated as shown below.

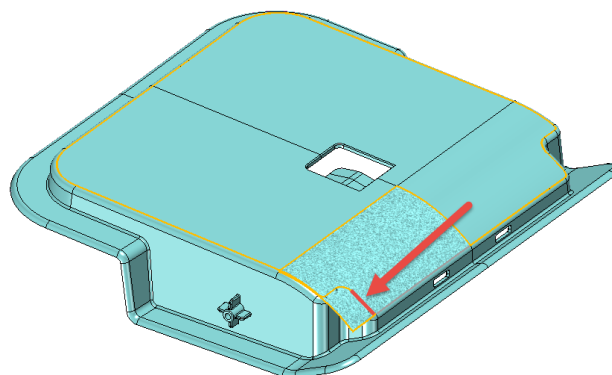


Now we are going to repair the surface shown in this image by using the **Trim to Curves** command  (**Free FormTab**).

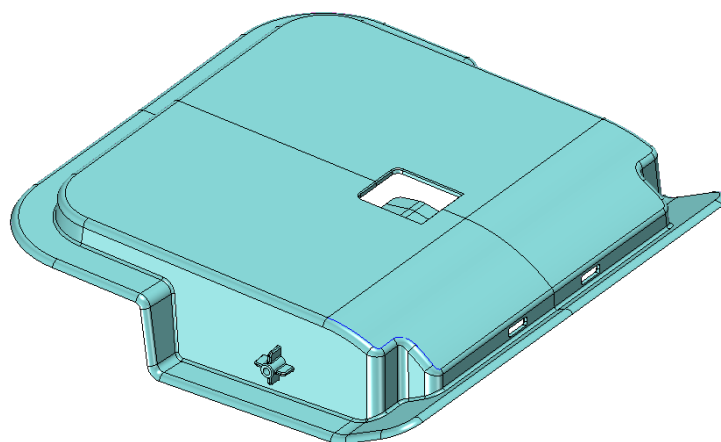
Select the surface to be trimmed (as shown in yellow below).

(You may have to change the **Filter** to **Face**)





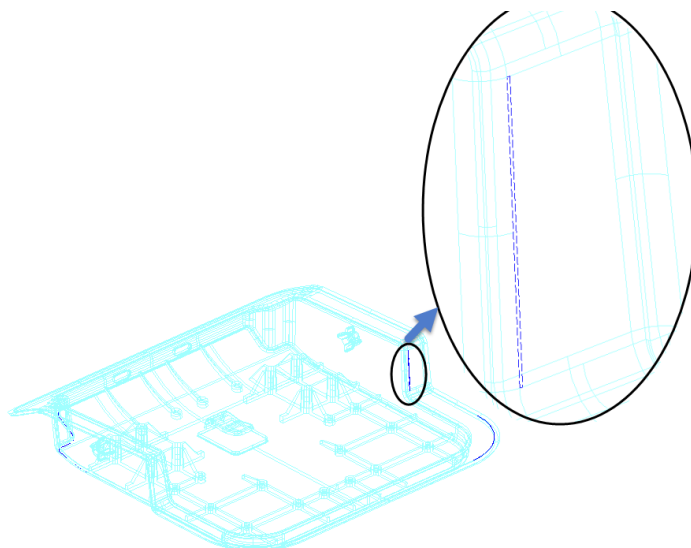
Select the boundary edge (use <shift-pick> to chain select the edges), as shown in red above. Finally select the side of the face to keep, as shown by the mouse figure above. See the result below.



Again this surface was trimmed.

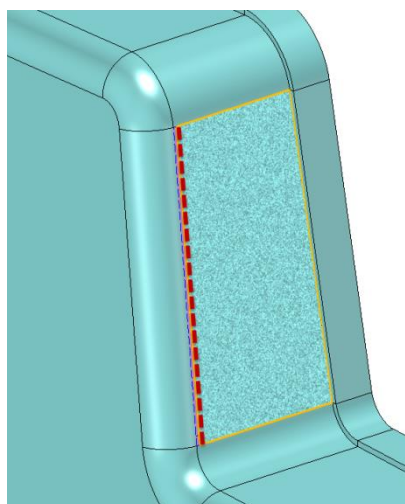
Sew and select **OK**.

The next surface will be repaired by extending and trimming.

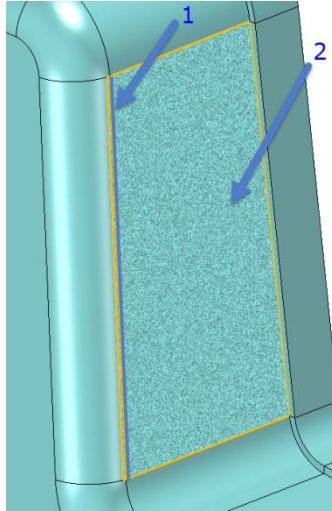


Use the **Extend Face** command  on the **Free Form** Tab. Select the open edge (as shown in red dashed) of the planar surface(indicated by the arrow) to extend.

Select an extension value of “**0.5**”.



Now trim the surface to the edge which will be the boundary.(**Trim to curves** command)

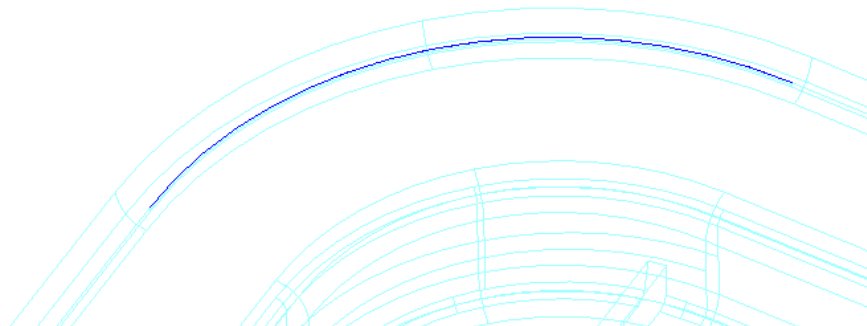


1: trim curve 2: Side of face to keep

Sew the part again.

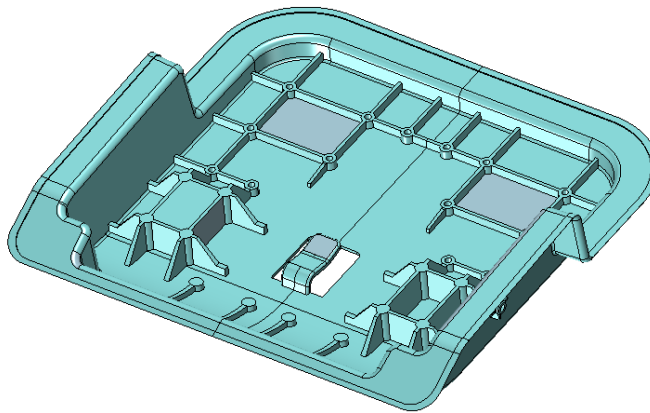
If you verify the openings on the model again, you will see one of two possible results. If after the initial sew command, you sewed again with the tolerance equal to the largest gap, you will have a solid with zero open edges.....you're done.

If you did not select that command you will have two edges left that do not match (use **Show open edges** command to inquire).



To easily repair the fault in the geometry use the **Close edge's gap** command . Just select two of the edges that need repair. The software will only select open edges so you don't have to be really accurate picking.

If you analyze the model again, you can verify that it is completely closed (use **Show open edges** command to inquire). However, the model does not have to be completely closed to continue the project in ZW3D.



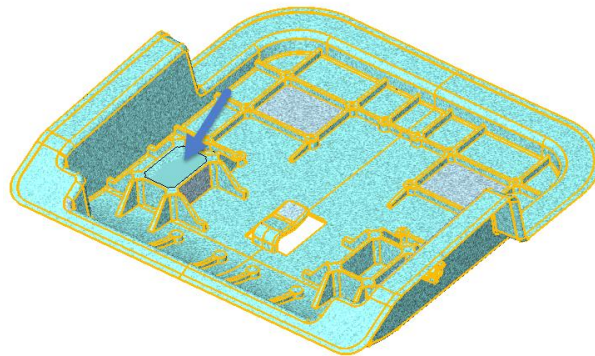
Turn on **Shade** . Change the **Pick filter** to **Shape**. Run the cursor over the edge of the part.

Right-Click and select **Attributes**. Change the **Face color**, or select **OK** for the default.

Run the cursor over the edge again. This will show us that we are still missing something.

Using **Blank** , **Unblank**  and **Swap entity visibility**  you will see that a solid was left in the model that needs to be subtracted.

We can use **Combine Shapes with the Remove Option** as suggested in this image.

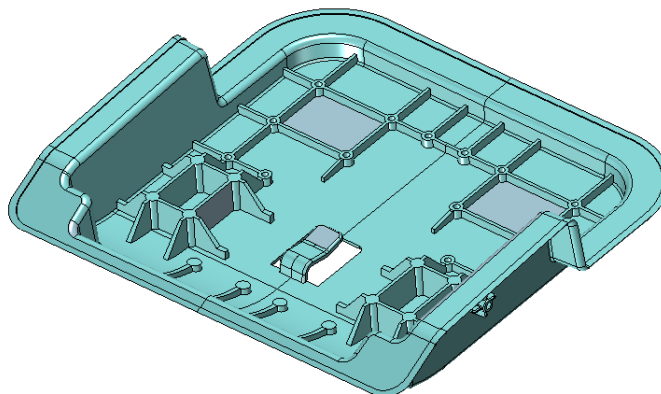


Use the **Combine shapes** command , and choose **Remove** method .

Select the mold part as your base. Select the object shown in the image as the object to be removed.

To complete the repair on the model we just need to heal by selecting **Heal/Analyze**  with a tolerance of **0.005**.

This will complete the import and repair of your model.



Save your file.**Exit** the part. 

Chapter 2 Translation, Analysis, and Scaling

After finishing the repairs on the imported part, the next step is to analyze it for draft angle violations and reposition the model.

Create  a **Part**  object and give it the name "**002_Part_Scale_&_Position**".

Use the command **Import Part Geometry**  or the icon as indicated on the image.

This command, import ageometry from another part, allows us to define the level of relations.

The **History** is the important option use to control the associative relationship

Sub-Part with Associative Copy - This option creates a sub-part with a copy of the geometry of the external part selected by the user. The history of the sub-part has an "Associative Copy" operation that imports the external part geometry.

Sub-Part with History - This option creates a sub-part with a copy of the full history of the external part selected by the user. Once the external part's geometry and history is copied into the sub-part, there is no longer any associativity between the parent/sub-part and the external part.

History copied into this part - This option copies the history of the external part into the active part, appending the external part's history to the end of the active part's history. Imported history operations are renamed as needed so they do not conflict with pre-existing operations in the parent part. It also incurs the overhead of an extra copy of the imported geometry (one copy in the sub-part, another copy in the parent part).

Associative Copy in this part - This option adds an "Associative Copy" operation to the history of the parent part that imports the geometry of the external part selected by the user.

Each time the parent part is regenerated, a new copy of the external part will be imported into the parent part.

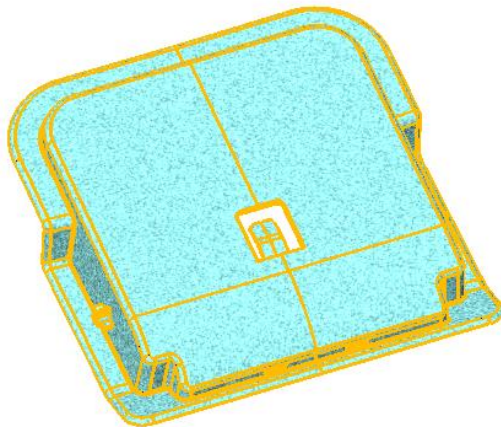
For this exercise, we will select the option “**Associative Copy in this part**”.

After selecting the object relation type, select the part “**001_Model_Part**” and locate to **origin**

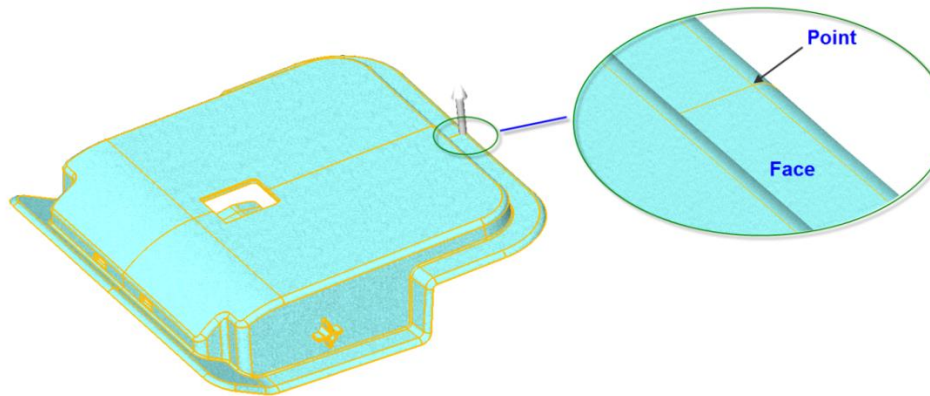
The next exercise will correct the product position to suit to the request of mold design.

Use the **Align Geometry** Command 

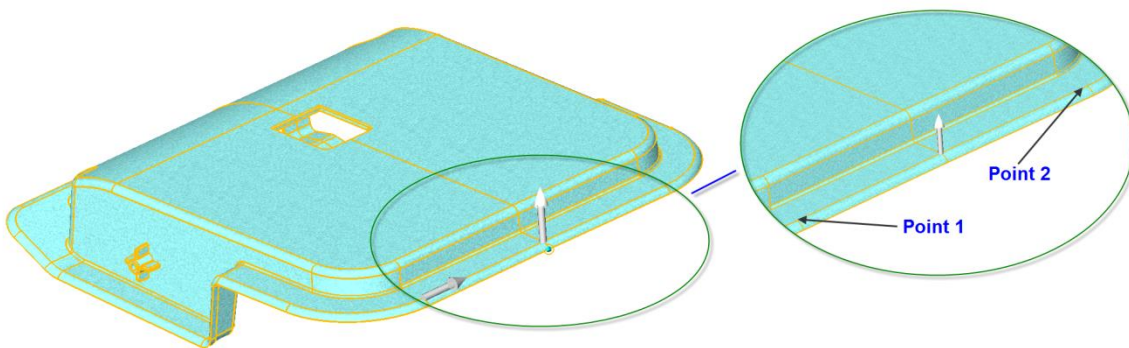
Select the **Align** command from the **Mold** tab, then choose the filter option **shape**. Select the product model shown as below .



Parting Direction : Right Click the empty window area and use “**Face Normal**” in the pull down menu to define the direction.

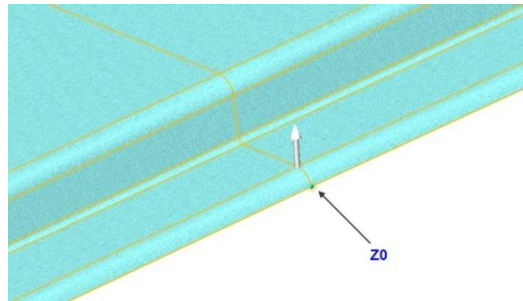


Side Parting Direction: Right Click the empty window area and use “Two Points” in the pull down menu to define the direction.

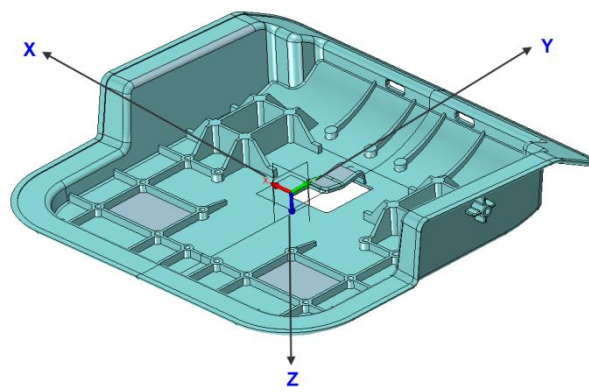


Select the “**Product Center**” as the **Location**.

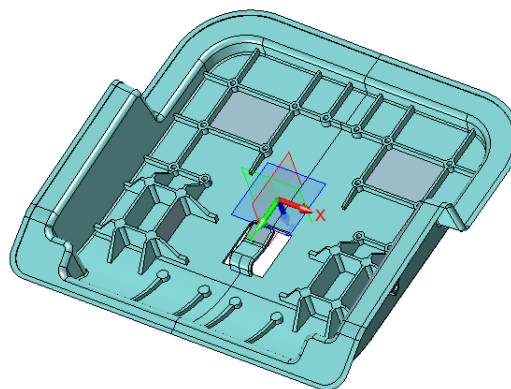
Z0: Select the **Point** as shown below.



Click **OK** after aligning the product. The model will be shown as below.



Next let's analyze the draft angles.

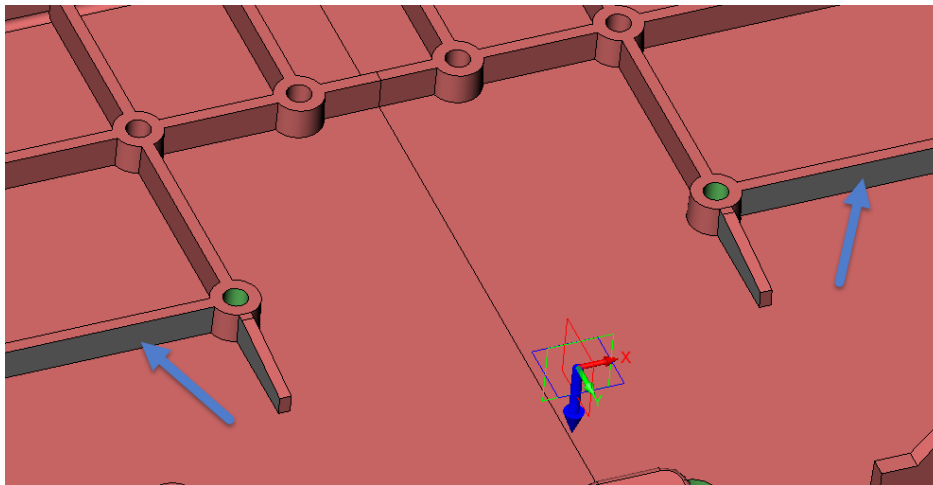


By using “**Draft**” , it can verify the existing draft on the 3D model. It also allows us to more clearly see which faces belong to the ‘Core’ and those that belong to the ‘Cavity.’

Select the “**Direction**” option. Now right-click on the graphics window and select the option **Z Axis**

For the draft **Angle** enter **-1** or use the slider provided.

Turn off the “**Use rainbow coloring**” option, then drag the “**Bands**” slider to the far left.



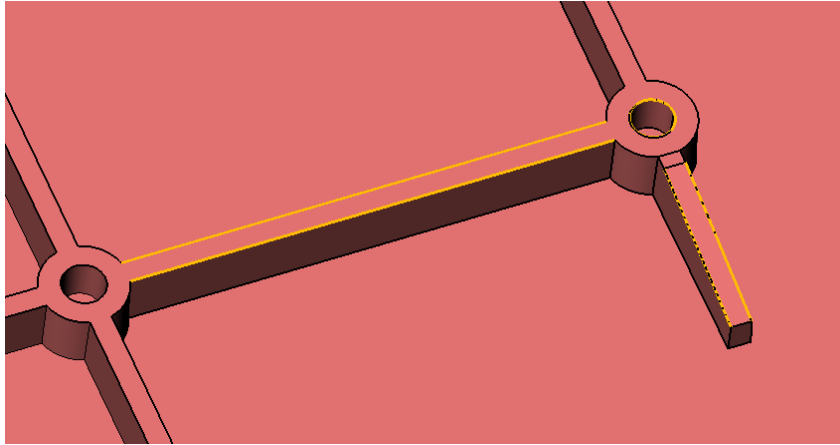
As the previous image showed, the **Blue** color faces means that they are vertical faces, this is not good for injection moulding.

Based on this analysis, we will correct the draft angle of these faces. Select **OK** at the bottom of the options form.

*******NOTE:** The display has been set to **Analyze** mode. You can correct the draft while still in this mode. To change it back to **Shade** mode select **View > Shade** from the **Document Aware Toolbars** menu.

You will correct the face angles.

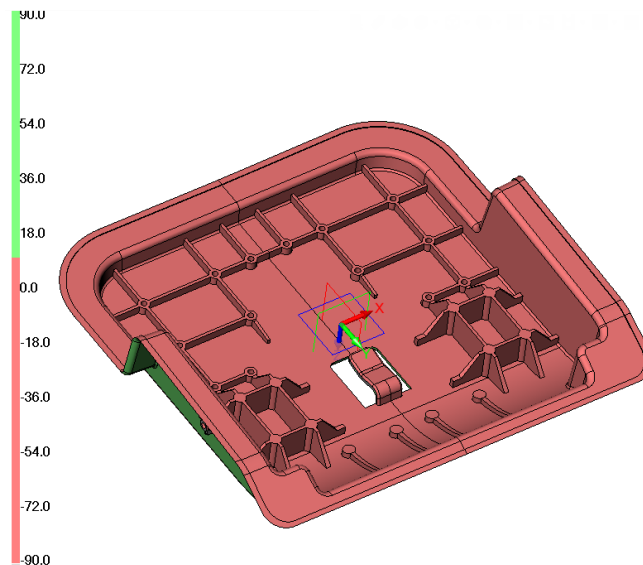
Select the **Draft** command  from the **ShapeTab**. Select the filter option **Edge**. Select the edges shown on the image below on both sides of the part.



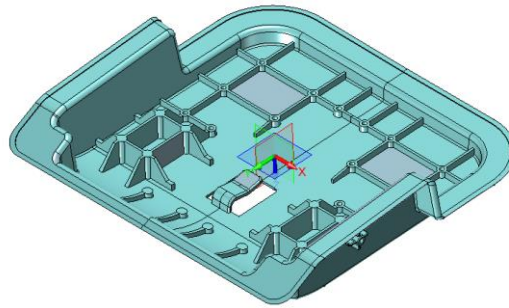
Set the **Draft angle** value to “-2”. **Right Click** and select **Z Axis** as **Direction**. Set **Intersection** to Remove all.

After that, do the same operation for another side.

If you analyze the model **Angle** again, it will verify that the faces have been corrected.



Now change to regular **shaded** mode. 

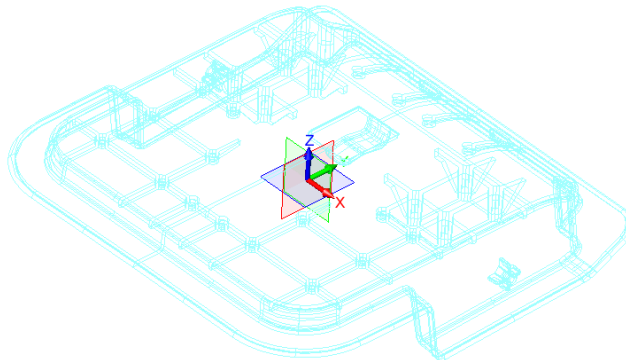


Save your file.

In the next exercise you will scale the 3D model to compensate for shrinkage in the mold. A scale of 0.6% is suggested. Scale the model using the command “**Shrink**”  that is found in the “**Mold**” Tab.

It will select the model as **Entity** automatically.

Indicate that the scale is **Uniform** and then set the scale factor to **1.006**.



Click **OK**.

Save your file. You are now finished with the editing of this “Part”. Exit this section by using the command “**Exit part**”. 

Chapter 3 Creation of Core, Cavity and PartingFaces

Create  a **Part**  object named “003_Core_Cavity”.

This new part will be used for core and cavity separation, and will be used in the construction of the parting faces.

After you have activated the new part, insert a component by using the **Insert Part Geometry** command  found on the **Mold** Tab.

Select “002_Part_Scale_&_Position” from list. Select the option, “No Backup” and locate it at the coordinates “0,0,0”.

Next, we will learn two different Parting methods in ZW3D. With each different method, we will use different functions for splitting products and create shut off surfaces.

Method 1: Parting lines method

Right Click the **Line color** command from **Document Aware** toolbars and configure the thickness of the 3D lines that will be created by the **Parting Line** command.



*******NOTE:**

1. The colour will have no affect here because the colour of parting lines is defined in the ZW3D Configuration. The default is green.

2. It has a different dialog box when you **Left click or Right click** the **Line colour** button, Left click will just call the simple interface to change the line colour, and Right click will call the complete interface to set the line attributes.

Parting lines will need to be created along the exterior edges forming a closed loop. You will also need to create parting lines around any holes in your part.

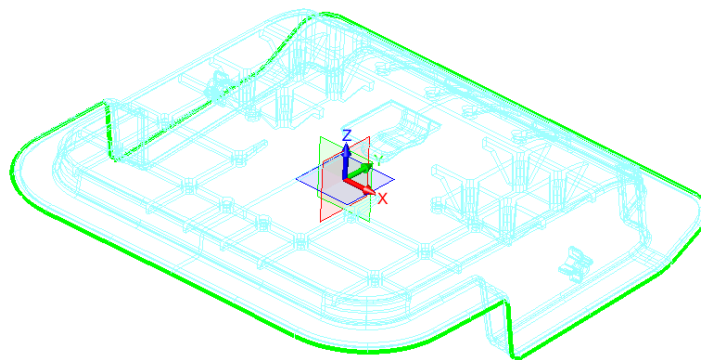
After finishing the preference, Click the **Parting lines** command , then select the **Parting line from silhouettes** command  from **Parting Lines Tools**

For the **Direction**, select the “**-Z axis**” as the axis of extraction, from the right-click menu.

For the **Faces**, <middle-click> to accept all of the faces.

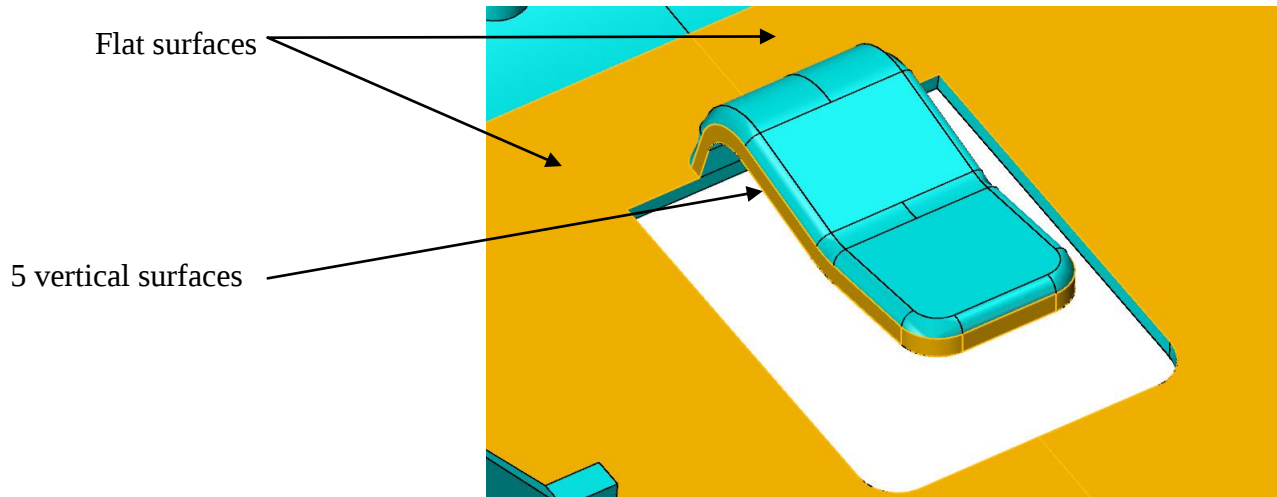
For the **Curve type**, select the “**Parting edges**”

<middle-click> to finish the command, the result should look like the image below.

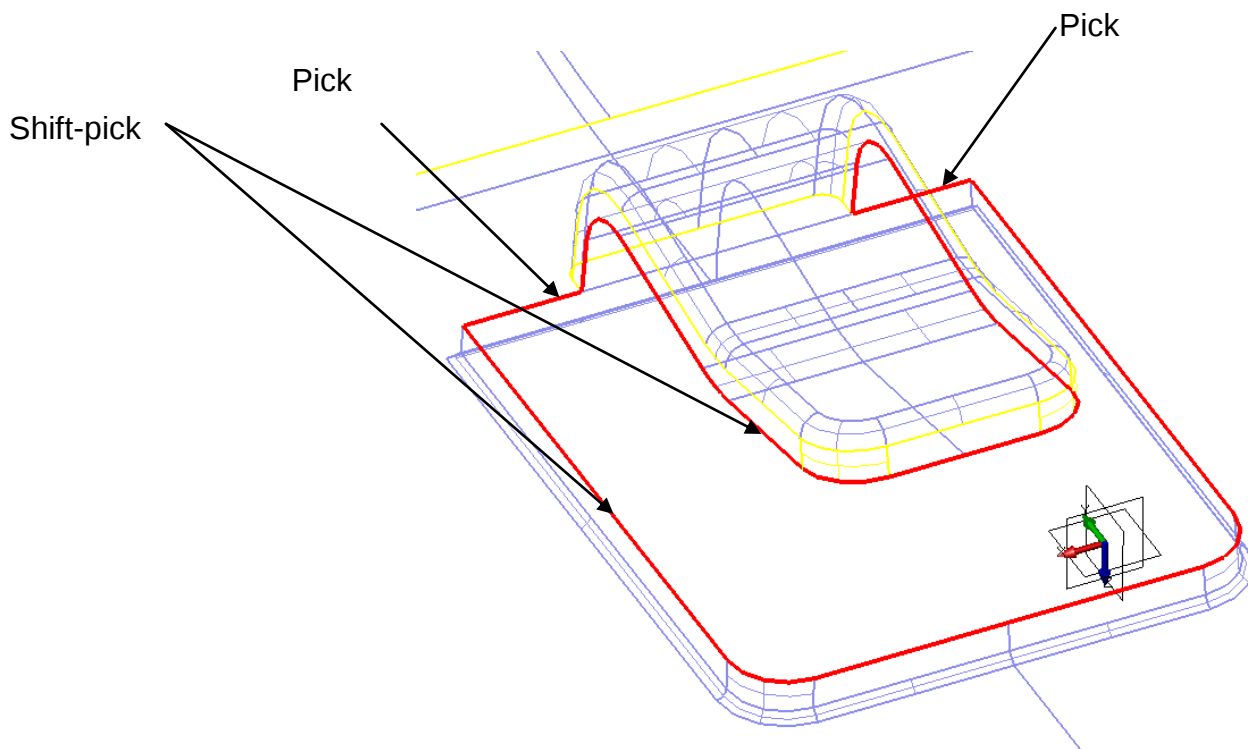


Create a new parting line around the holes in your part by using the **Parting lines from face edges** command. Using the command, **Parting lines from face edges**,  indicate all the of

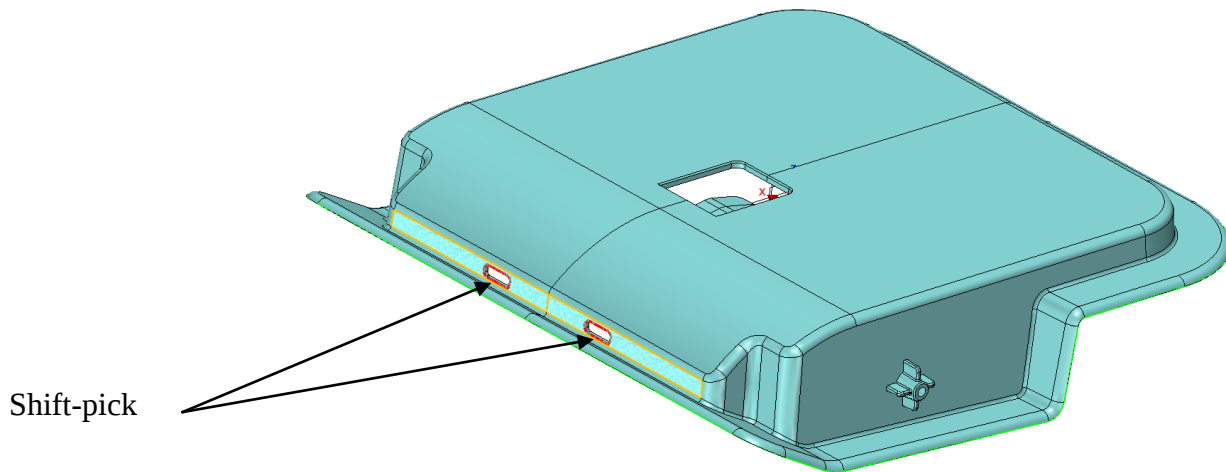
the faces you wish to calculate by picking the two large flat surfaces and the 5 vertical surfaces on the tab.



Select only the edges as shown in the image. If you get too many, <Ctrl-pick> to delete them. Again we should have a closed loop.



Repeat the previous command by <middle-clicking> and create parting lines on the edges as shown in the image. Pick the two faces and <shift-pick> the two edge loops.



Again we should have two closed loops.

To check for future problems in the parting lines, select the **Parting lines** command.



There should be no gaps or overlaps in order for the part to split in the next step.

ZW3D should prompt an **END POINT CHECK: 0 unmatched, 0 over-matched**.

With this **Splitting** method, after creating all the parting lines, we will use **Divide by parting line** to split the product to the **Cavity and Core region**.

Click the **Split**  command

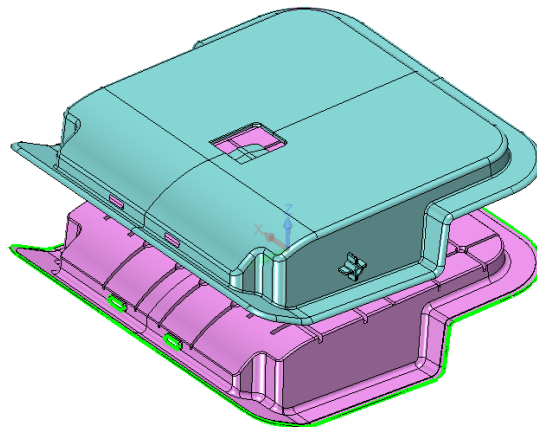
Select the second option, **By parting lines**  which is only used when **Parting lines** exist.

Select all of the lines necessary for the division of the model. If you wish to separate the object into two **Shapes**, select the **Separate Shape at Parting Line** option. Otherwise the model will be divided only on the surface and will not separate into two **Shapes**.

Make sure the option **Separate Shape at Parting Line** is checked. <right-click> pick all, <middle-click> twice to finish.

ZW3D will show the hint with “**Total 2 new created regions shape**” that you can know how many regions have been separated.

You now have two separate shapes. To temporarily see the results **blank – shape** or **Right click→Move →Along a direction (Z axis)** select one of the shapes. When you are done, click **Undo** to restore



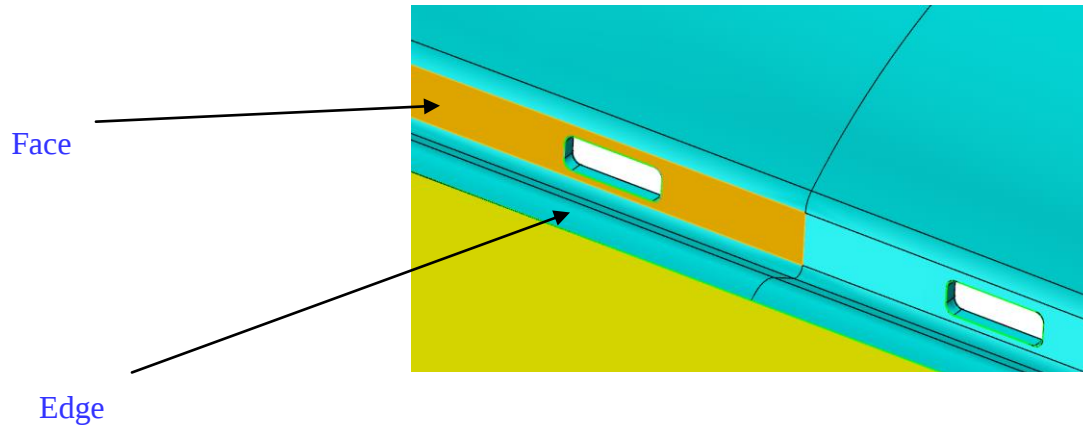
Next you will need to create three inner **shut off** surfaces.

Firstly, let's define a new color for the new faces that will be created.

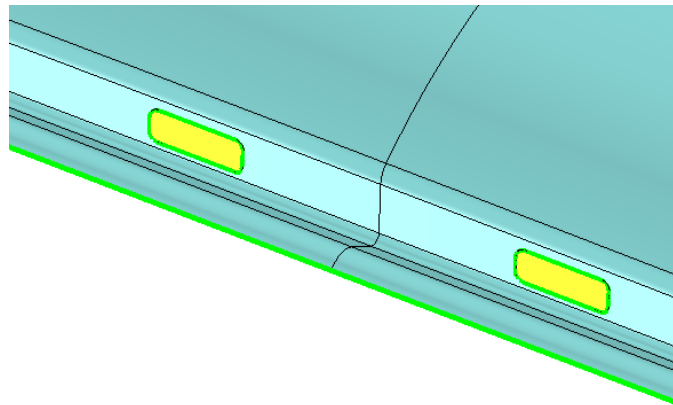
To create the new face color select the command **Face color** from **Document Aware Toolbar**. Define a color as suggested in the image  and apply this as the color.

Click the **Parting faces** command  , then select **Create faces by inverting loops**  from **Parting Face Tool**

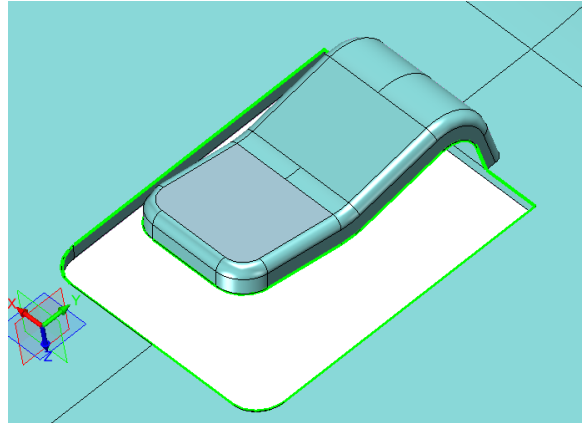
This will create missing faces on the areas shown in the image.



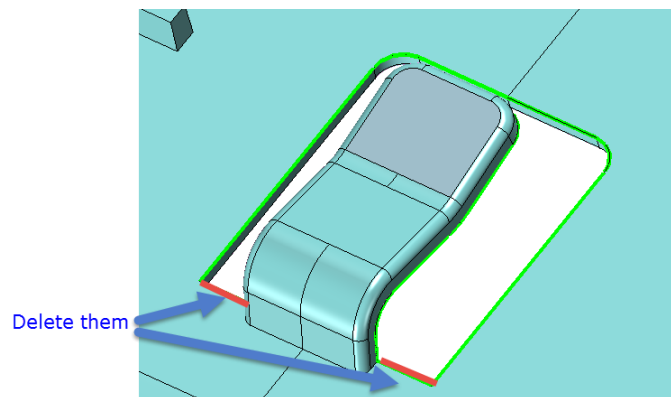
Select one face and one of the edges. Repeat for the other shutoff surface.



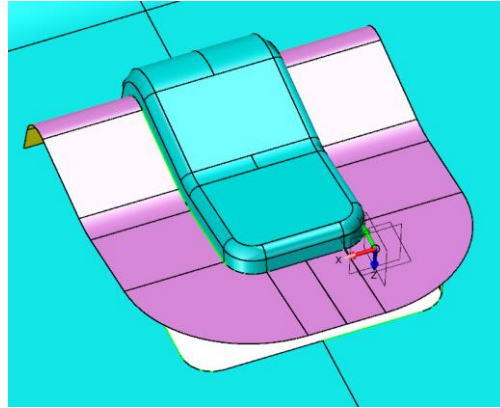
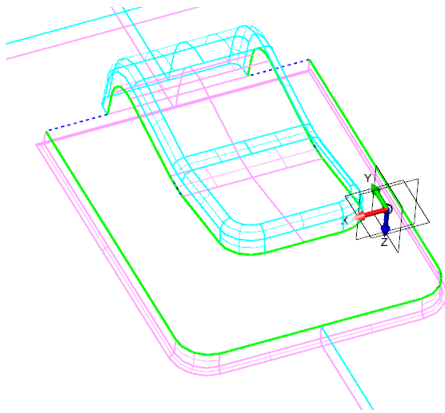
The last area to close off is the image shown below.



Use the **Erase**  command from **Document Aware Tool bar** to delete the two parting lines segments shown below. (You can set the **Filters** to **Parting Line**)

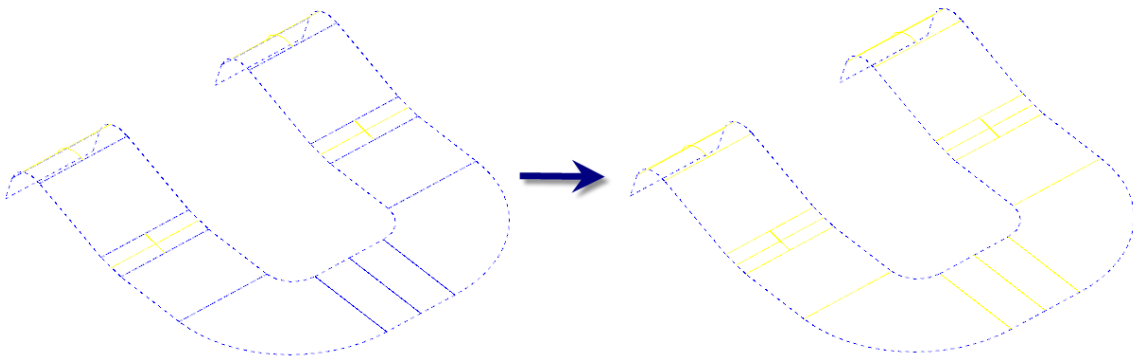


Select **Angel parting face**  from **Parting Face Tool**, and select the parting lines on the raised tab. **Shift-pick** the parting lines on the tab.

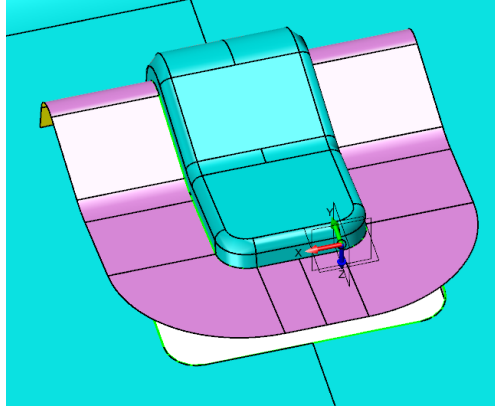


Indicate the value “10” for the **Distance** of the surfaces and “0” for the **Angle**.

Select **Sew**  and pick previous parting faces that you have just created, so that it becomes a single sewn parting face.

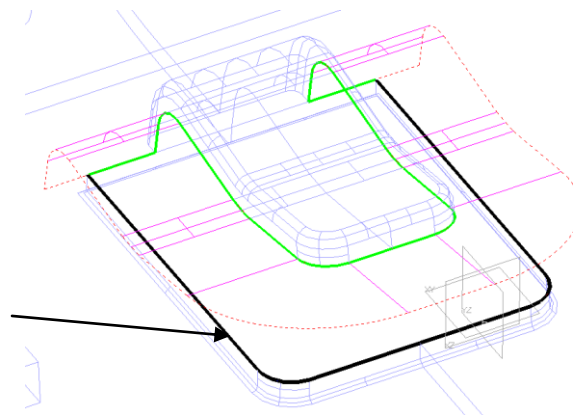


In order to keep the direction of surface consistent. Use **Reverse face direction** from the Free Form Tab if necessary.  (probably not)

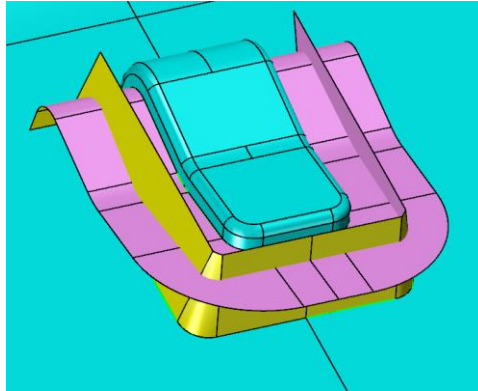


Select **Create faces at angles to surface** again and **shift-pick** the other parting lines (total 6) that lie on the flat surface.

Change the **Angle** to **10** degrees.

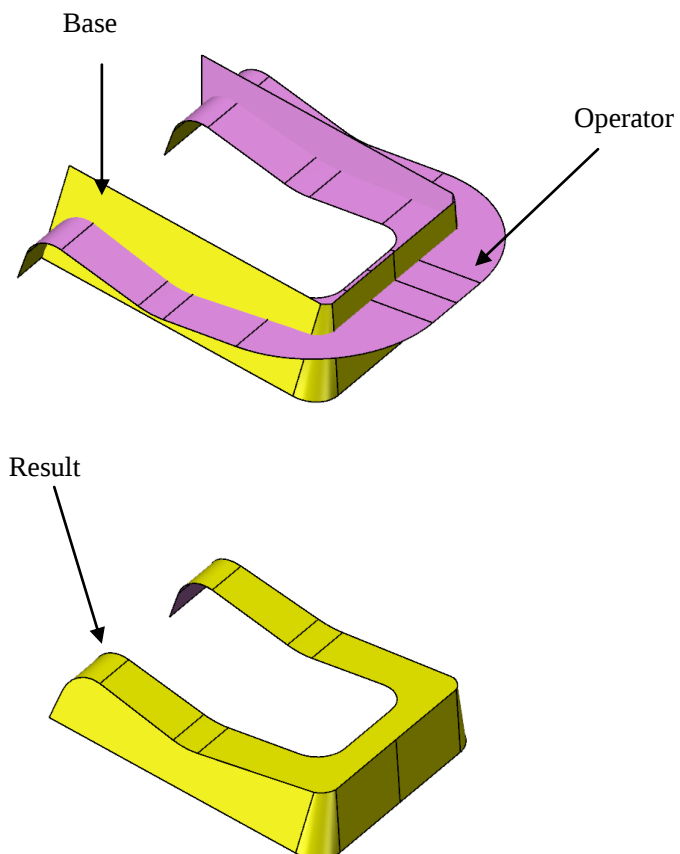


See the result below.(Notice:If the face direction isn't correct as shown below, please use **Reverse facedirection** to change it,as last step said).



Select **Sew** and pick one of the the parting faces just created. **OK**.

Use the **Combine Shape**  **>Remove**  operation as suggested in the image.



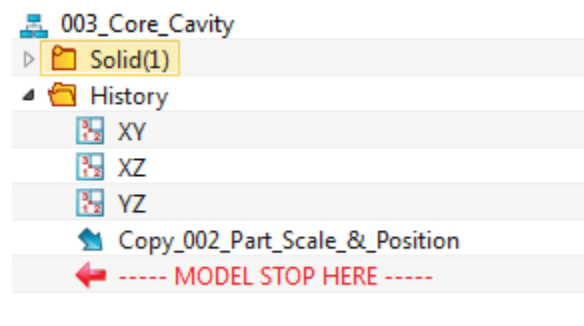
Save

Above is the first method to split the product to the Core and Cavity, create shut off surfaces, we will learn the second parting method with **Color region**

*******NOTE: In real work, you can decide which one you want to use in different situations.**

Method 2: Color Region

If you have finished the first method, go back to **Import the Part Geometry** step, to do that, you can drag the history bar to specify position.-

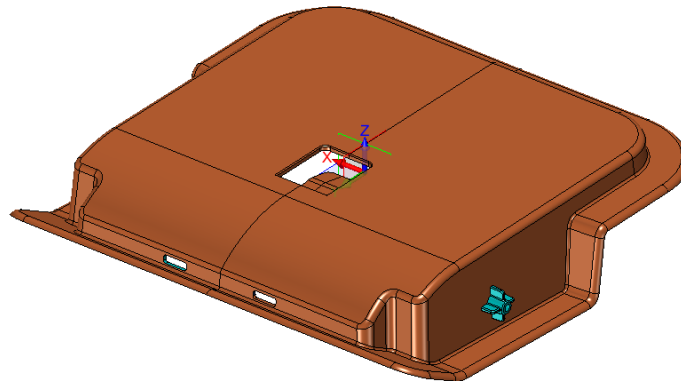


Use the **Region** command 

Pick **Calculate**, when you use **Region** command the first time, it can recognize the shape and use **Auto define** as the default option.

Right click the graphic windows then select the **Z** axis as the direction.

Pick **OK**, after analysing the product, the model will be shown as below.



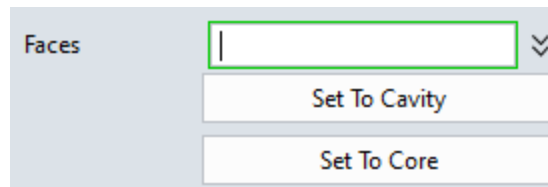
The region information:

Region Information	
Cavity Number	137
Core Number	479
UnDefined Faces	166
☐ UnCalculate Faces	0
☐ Cross Faces	0
☐ Vertical Faces	2
☐ Unknown Faces	164

After using the auto method to define the Cavity and Core face, there are still some undefined faces.

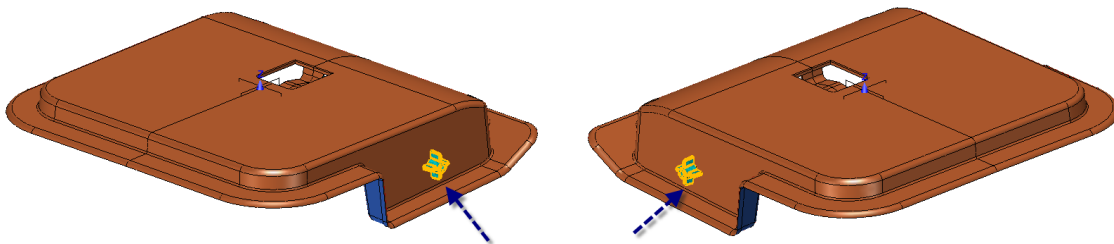
So next we will use the manual method to define the Cavity faces and Core faces.

Move the mouse to the **Face** input field



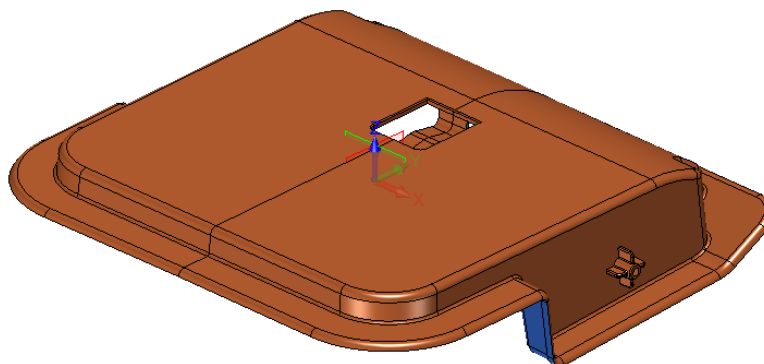
Select the face of both sides from the graphic window as shown below.

*****HINT: Holding the Shiftkey can help you select it easier.

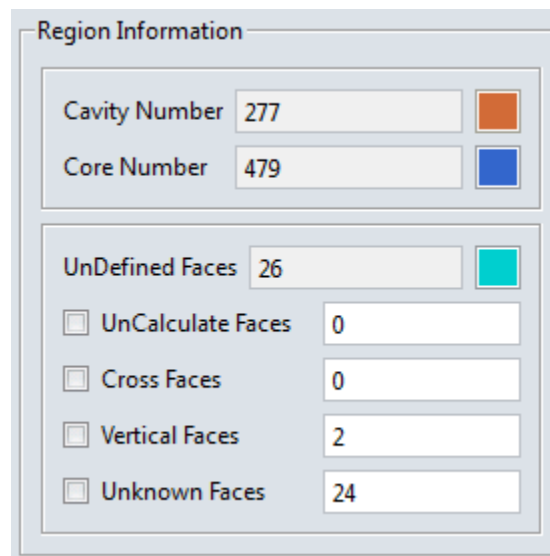


Click **Set To Cavity** Button

The final result:



The region information:



Region Information

Cavity Number 277

Core Number 479

UnDefined Faces 26

☐ UnCalculate Faces 0

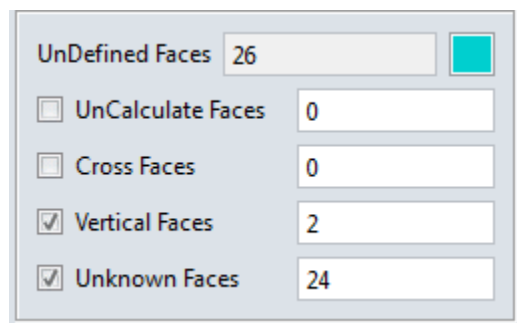
☐ Cross Faces 0

☐ Vertical Faces 2

☐ Unknown Faces 24

After defining all of the Cavity faces, let's define the Core faces.

Check the **Vertical Faces** and **UnknownFaces** option, those faces will be selected directly.



UnDefined Faces 26

☐ UnCalculate Faces 0

☐ Cross Faces 0

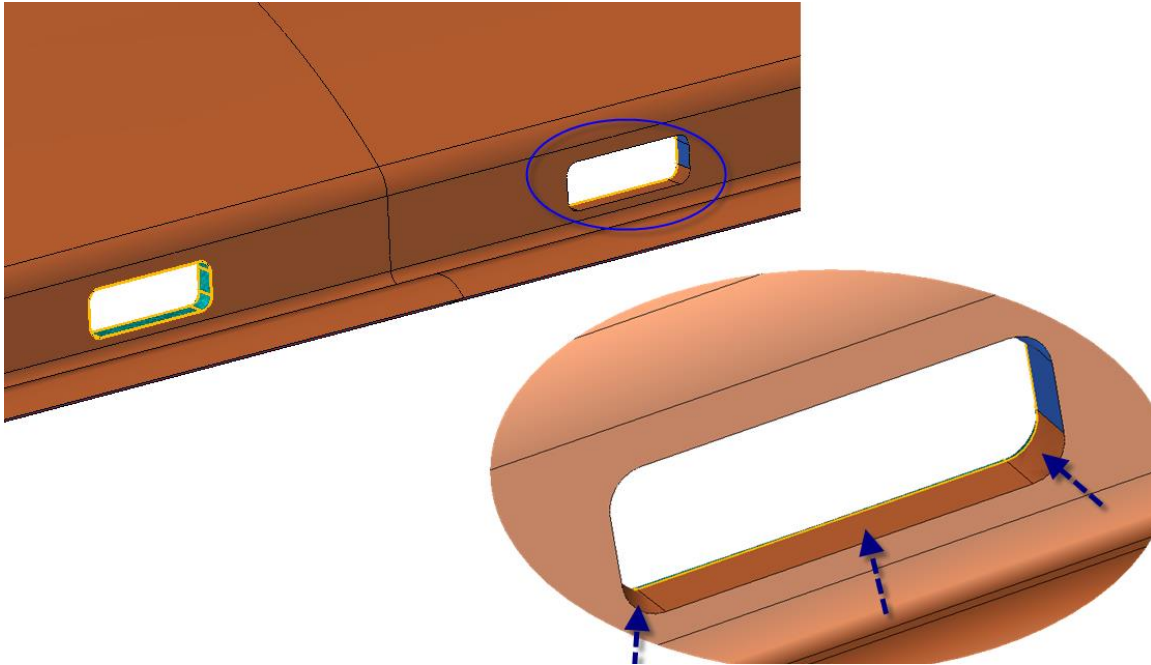
☒ Vertical Faces 2

☒ Unknown Faces 24

Rotate the model and select other face from operated window.

*******NOTE: Here what you selected will include some faces that belong to Cavity by Auto Define, so Manual Define will be used to enhance the functions of Auto Define**

Rotate the model and select 3 faces as shown below



Click **Set To Core** Button

Make sure all faces have been defined to the correct region, and all **undefined faces** option values show “0”. The correct **Region Information** should be shown as below.

Region Information	
Cavity Number	274
Core Number	508
UnDefined Faces	0
☐ UnCalculate Faces	0
☐ Cross Faces	0
☐ Vertical Faces	0
☐ Unknown Faces	0

Pick **OK** to finish the command

So far, we have defined the Cavity and Core color regions successfully.

*******NOTE: With this parting method, the process has a slight difference that we will create a shut off surface in advance of splitting the product.**

This is the same as the first method of creating shut off surfaces. Firstly, let's define a new color for the new faces that will be created.

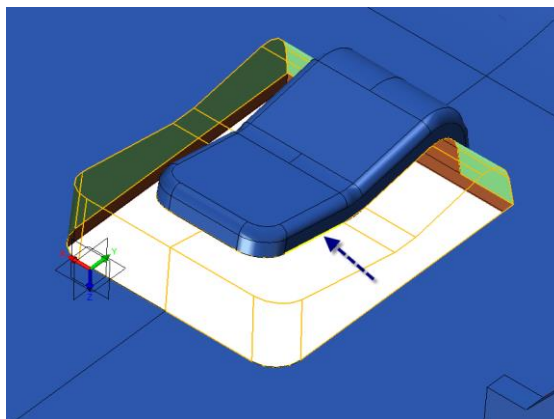
To create the new face color select the command **Face color** from **Document Aware Toolbar**. Define a color as suggested in the image  and apply this as the color.

Pick the **Patch**  command

*******HINT: This command is more useful to create shut off surfaces**

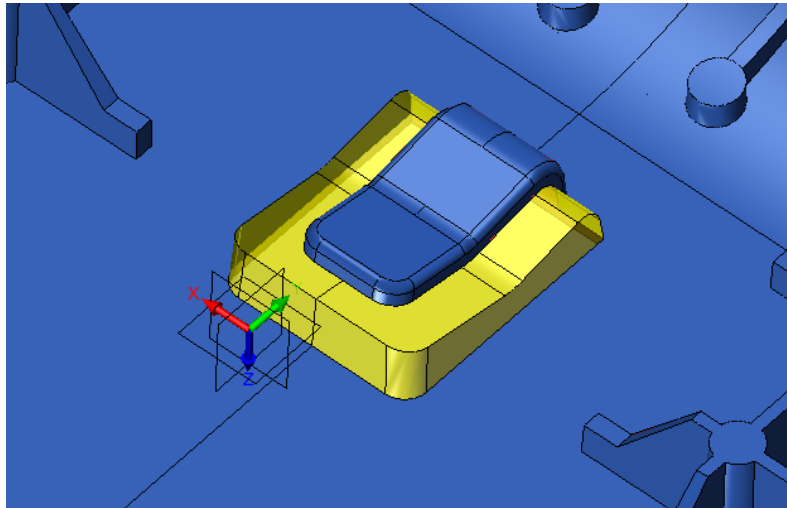
Change the **Type** to **Inner Edge**

Select the edge as shown as below, you will get the preview result.

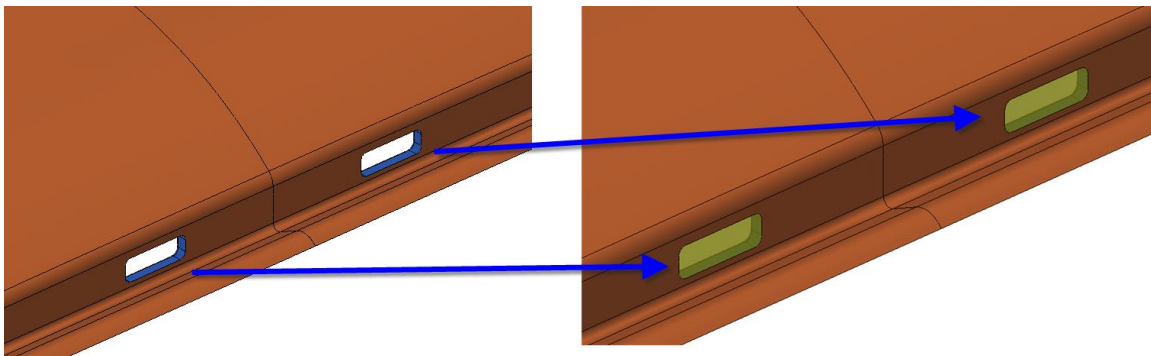


Use **Cavity/Left side**

Pick **OK**, the final result is show below



Repeat the last command with the samemethod to create another inner patch.(You can finish this step as a personal exercise)



Next, splitting the product model to cavity and core

Pick the **Split**  command.

Select the first option, **Byregion faces**  which is only used when the **Faces region** has been defined.

Shape : Pick the shape from graphic window

Keep the original shape: Turning on this option will let you keep the original shape after splitting Core and Cavity regions. The shape will be hidden since it was moved to the off layer which is named "Moldpart_org".

Create parting edges: Turning on this option can let you create the outer parting lines from the boundary of the Core and Cavity region.

Create interior loops: Turning on this option can let you create the inner parting lines

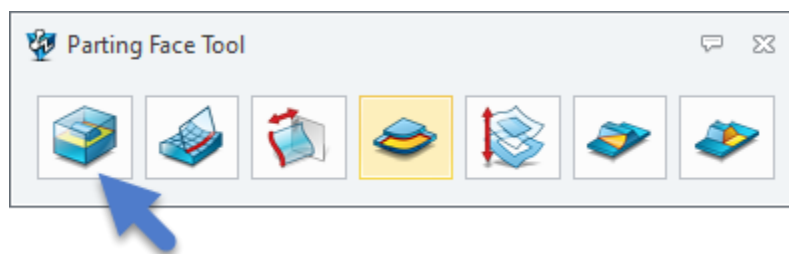
Force split with multi-regions: Turning on this option can let you to split the product model to more than two regions.

Select **"OK"**.

As in the first splitting method, you now have two separate shapes. To temporarily see the results **blank – shape** or **Right click–Move – Along a direction (Z axis)** select one of the shapes. When you're done, click **Undo** to restore.

So far we have finished the second method to get the Core and Cavity region and shut off surfaces, you can decide to use one of them. Next we will create outer parting faces.

Click the **Parting faces** command  , then select the first command on the toolbar



Here we can also use the command **Facecolor** from **Document Aware Toolbar** to preset a new color for parting faces.

With the **Parting Face** command,  ZW3D will support the use of Extrude, Sweep and Trim plane to create the outer parting face.

As long as you are entering this command, ZW3D will select all the outer parting lines automatically.

Set the **Distance** to 80

Don't change any thing for **Optional**, we will introduce that later.

Click **OK** to finish this command

[illegible]

Next we will go on to create the **Workpiece**

First, let's define a new color for the **Workpiece**

To create the new face color select the command **Face color** from **Document Aware Toolbar**.

Define a color as suggested in the image  and apply this as the color. Also you can set the color transparency so that we can easily see the internal Parting faces and Core&Cavity region.

Select **Workpiece** command  From **Mold** table

Here ZW3D offers three ways to create the workpiece

Block  : Create the workpiece with Basic block by defining the **Length, Width and Height**

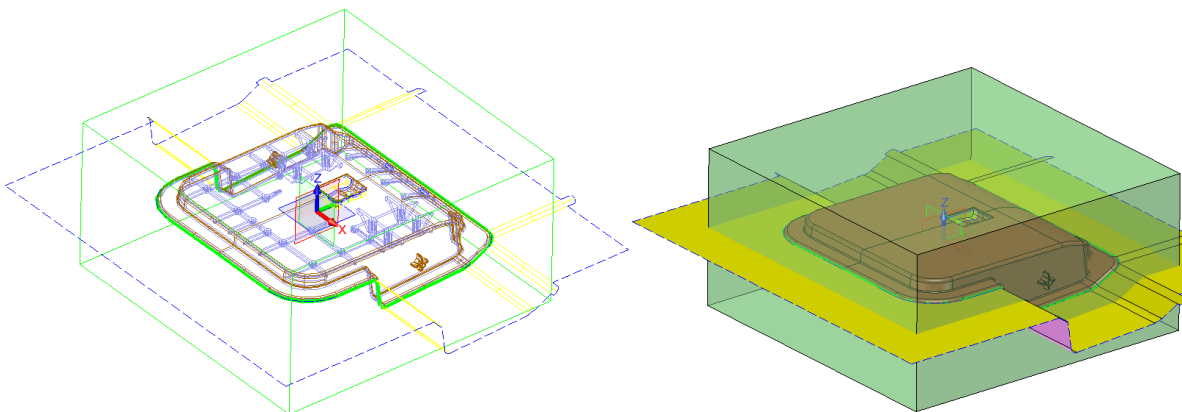
Cylinder  : Create the workpiece with Basic cylinder by defining the **Radius** and **Height**

Extrude  : Creating the workpiece by extruding from **2Dsketch**

Use the default method and set the parameter as shown below.

▼ Size Settings	
X Size	276
+X Size	138
Y Size	276
+Y Size	138
Z Size	140
+Z Size	70

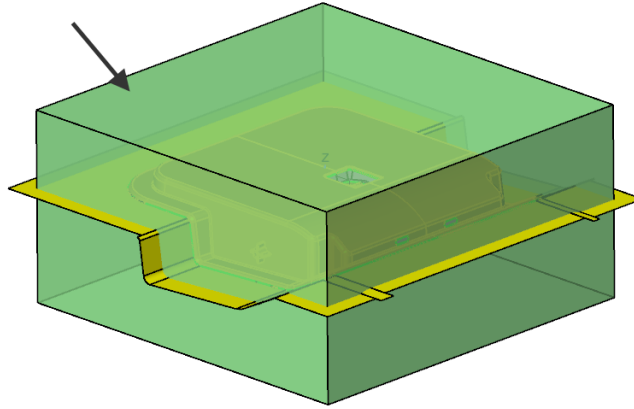
Click **OK**, we will see the result shown below



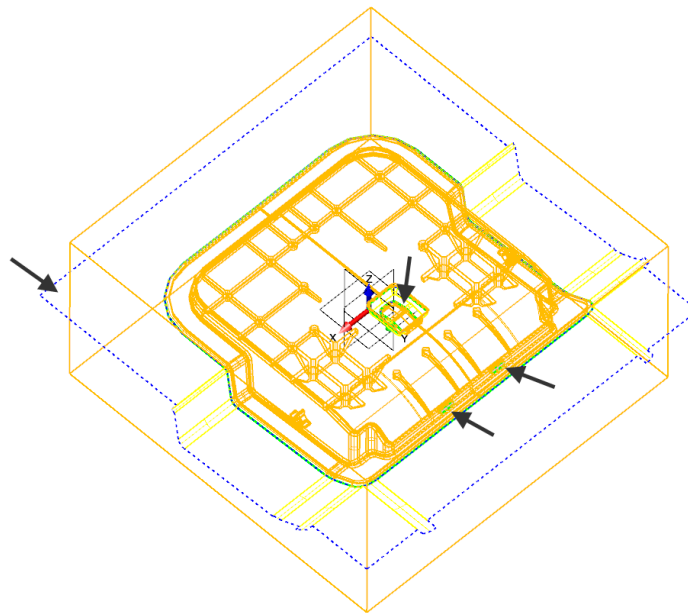
Our next step will be to split it to Core and Cavity shape.

Select the command **Trim**  from **Mold** tab

Plate: Select the Workpiece we just created

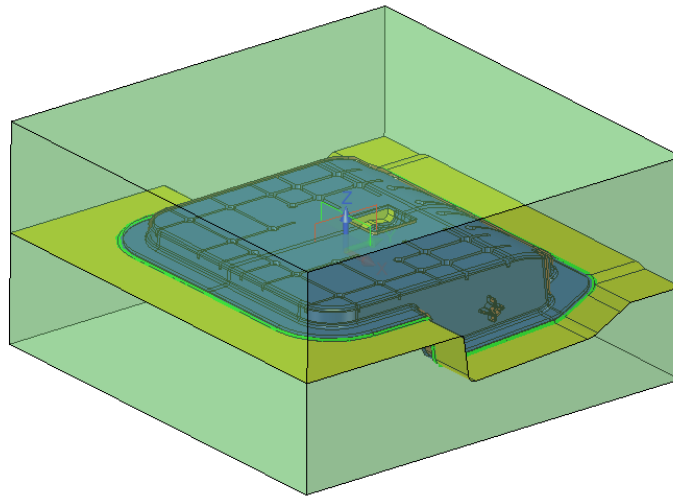


Parting: Select the all the parting face which include one outer parting face and three inner parting faces(If there are no other surface geometries, we can **Right Click>Pick All**)

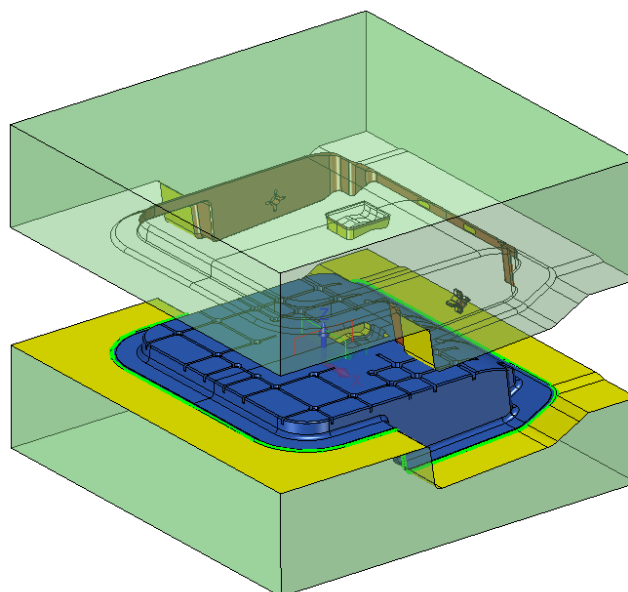


Core and Cavity Input Option: Select the Core and Cavity region (**Notice:** If you use the second splitting method, the input can recognize Core and Cavity automatically)

Pick OK, and you will get the result as shown below.



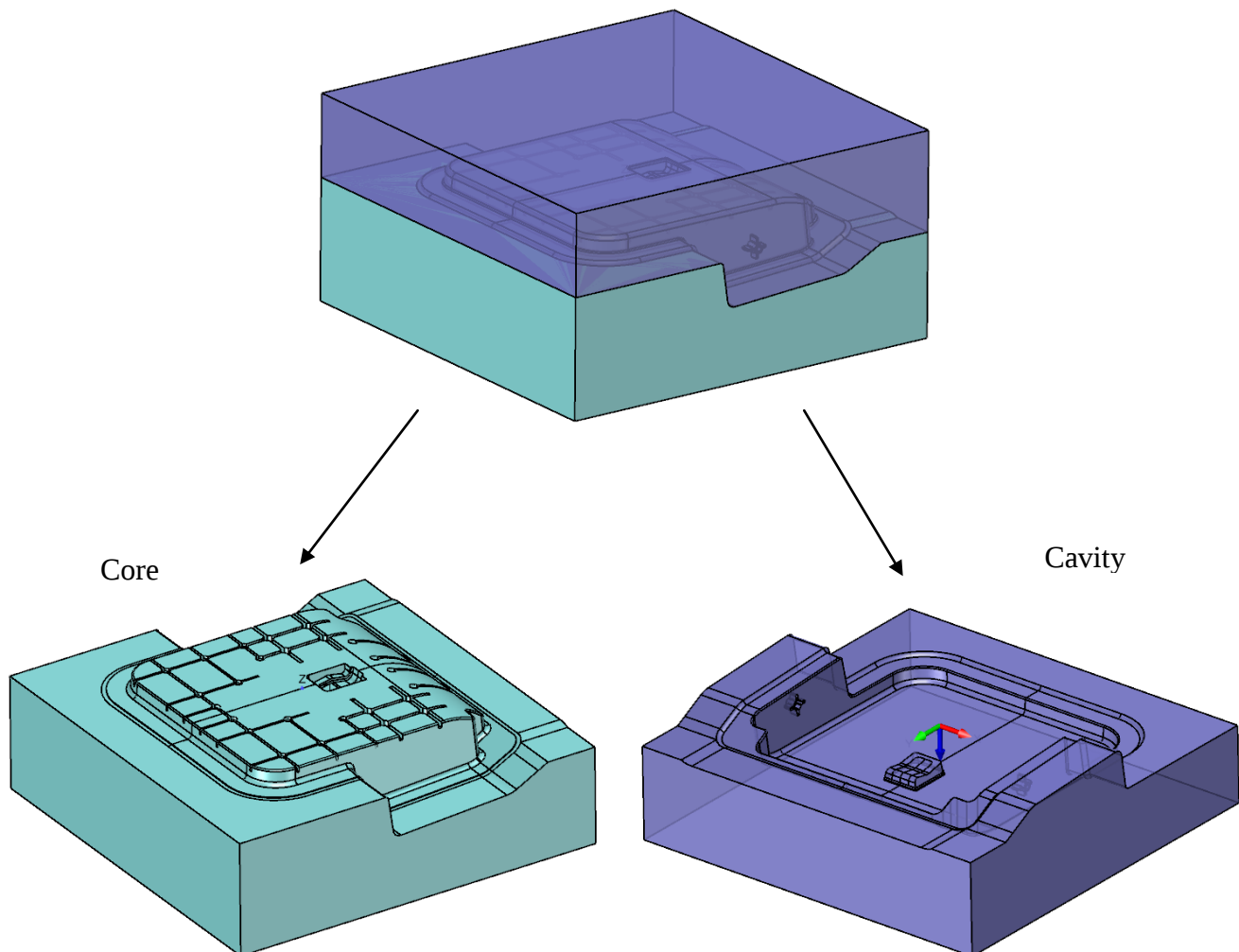
You now have two separate shapes. To temporarily see the results **Blank – shape** or **Right click–Move – Along a direction (Z axis)** select one of the shapes. When you are done, click **Undo**.



Before the next step, please assign different colors to the core and cavity shapes so that they can be easily differentiated (You can use any color you like).

Change the filter to **Shape**, Right Click the shape then pick **Attribute** then you can make it.

The final result could be like the one shown as below.



Save

Mold

Next, we are going to use command **Geometry to Part**  to export the geometries.

Assembly -> Geometry to Part 

Geometry: Select the cavity shape.

Create a new object named **“004_Ref_Cavity”**

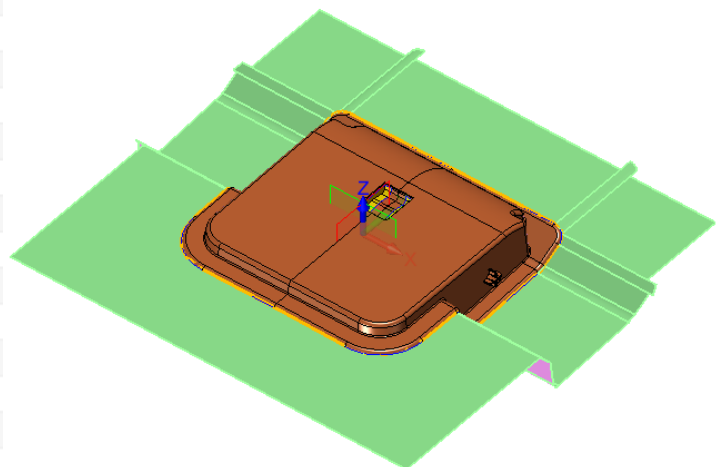
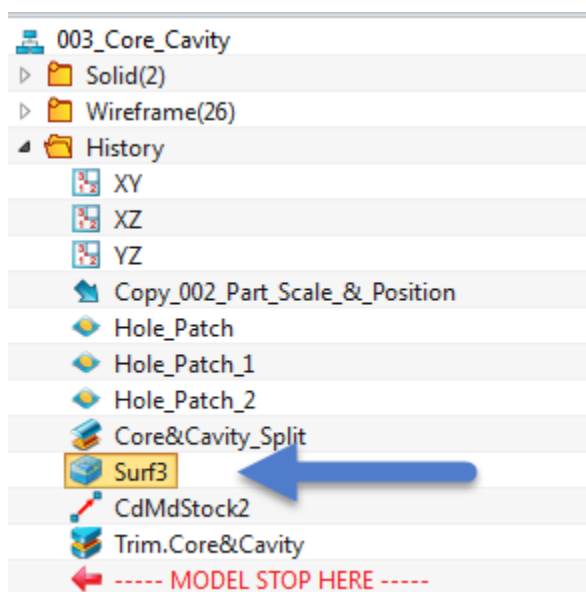
Uncheck all the options on the Setting, this can extract the geometries with associated relationship.

Repeat the operation, Select the core shape, give it a new object name **“005_Ref_Core”**

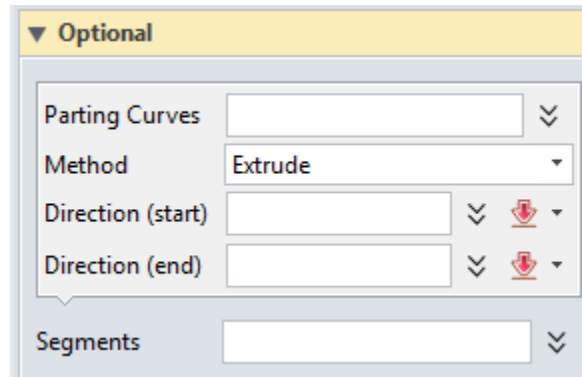
The next part of the exercise will demonstrate how to associate parts together so if a change is made to one it is propagated automatically to the others.

Exit to **Object manager** and select the **“003_Core_Cavity”** part to edit.

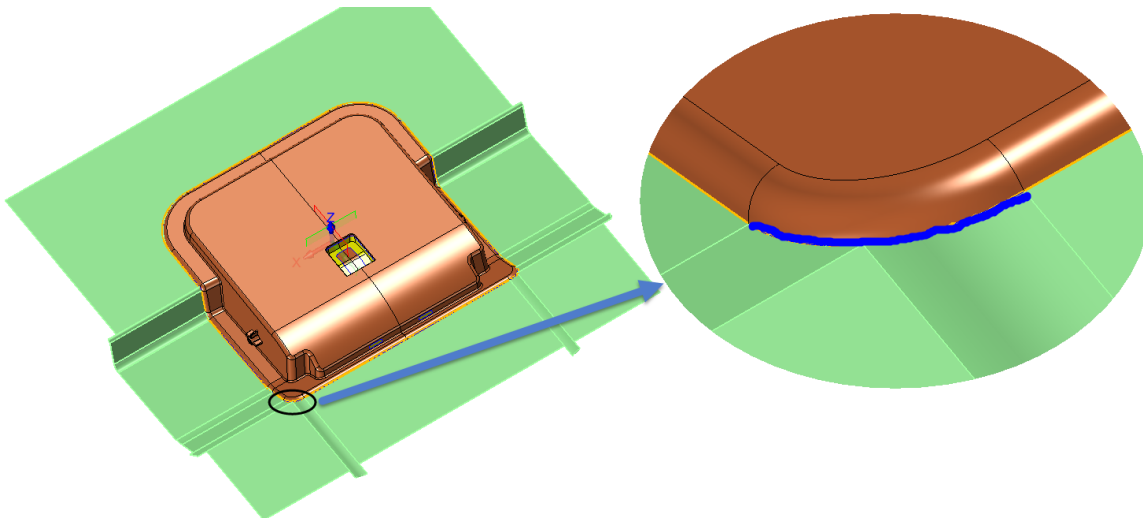
Right click on the **Parting face** feature and use **Redefine**  as shown as below



Next, we will learn how to use **Optional** function on **Parting face** command to modify a parting face.



Move the mouse cursor to the **Parting Curves** input field, select the parting line as shown as a blue color below.

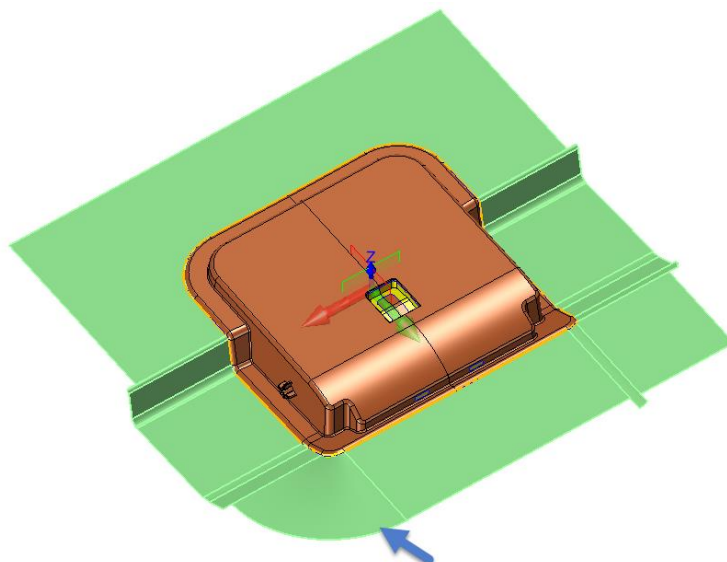


Change the **Method** to **Sweep**

Direction(start) **X axis**

Direction(end) **Y axis**

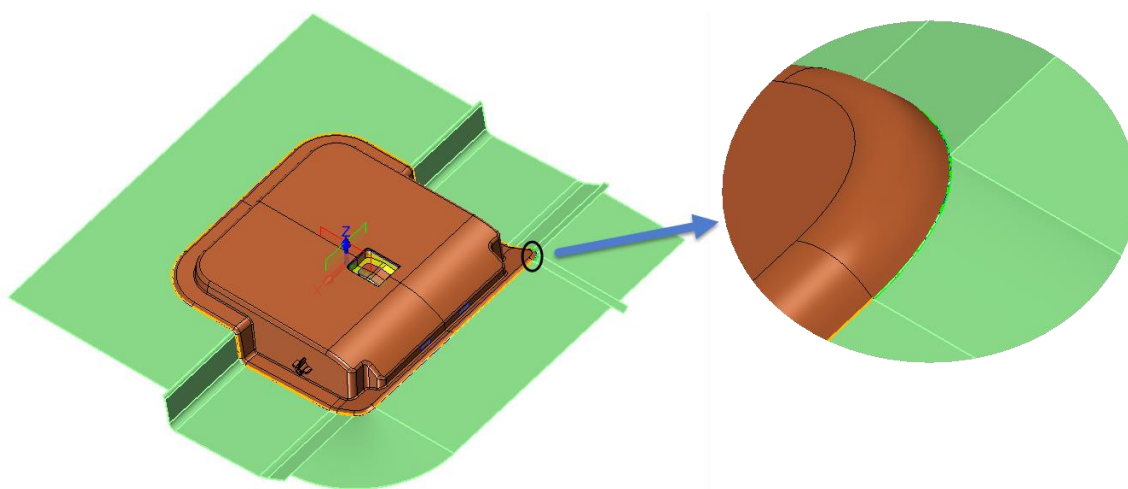
Then you have to Middle click to add this to **Segments** list, the final result is shown below.



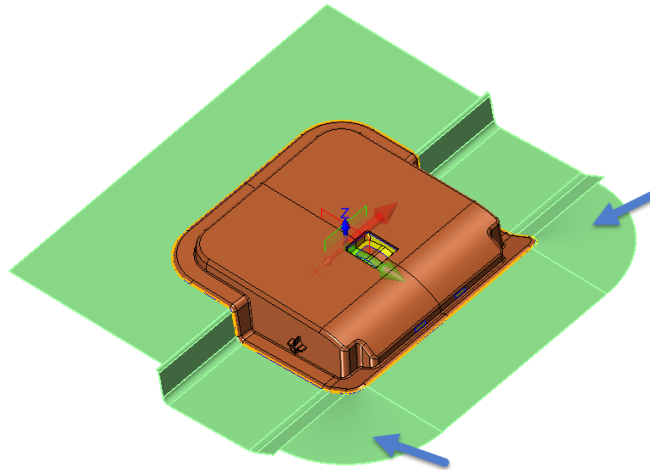
*******HINT:**

1. When defining the direction for the **Start** and **End**, you can right click the empty graphic window to select different axis.
2. If you find the result is not correct, please try to exchange the **Start** and **End** direction.

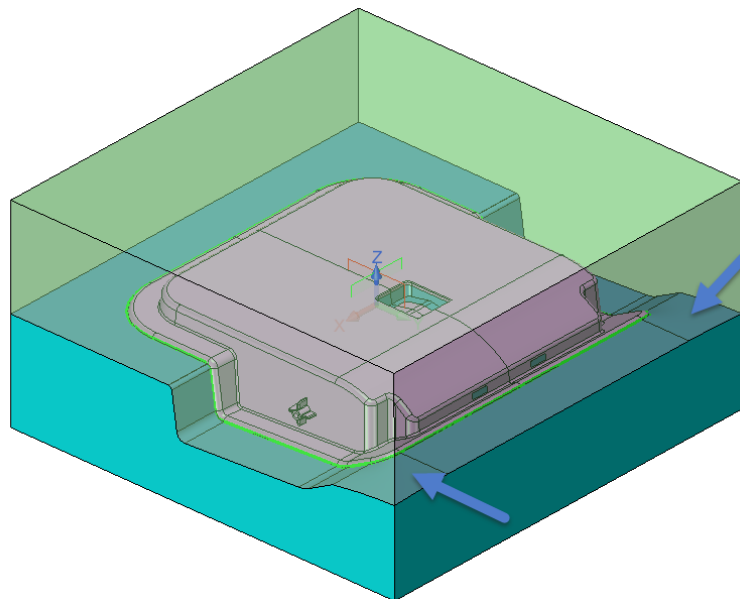
Do the same operation for another parting line as above (The sweep direction should start from the **Y** axis and end at the **-X** axis)



After making modifications as above, the result should be shown as below

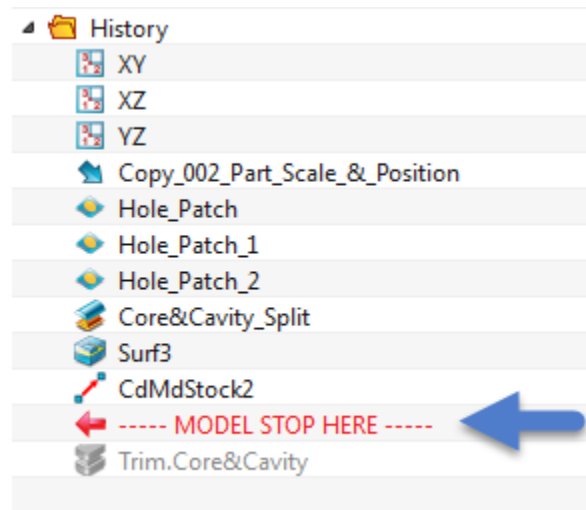


Click **OK** to confirm the modification and exit **Parting face** command, the history will go through to the end automatically, and the final result should be shown as below.

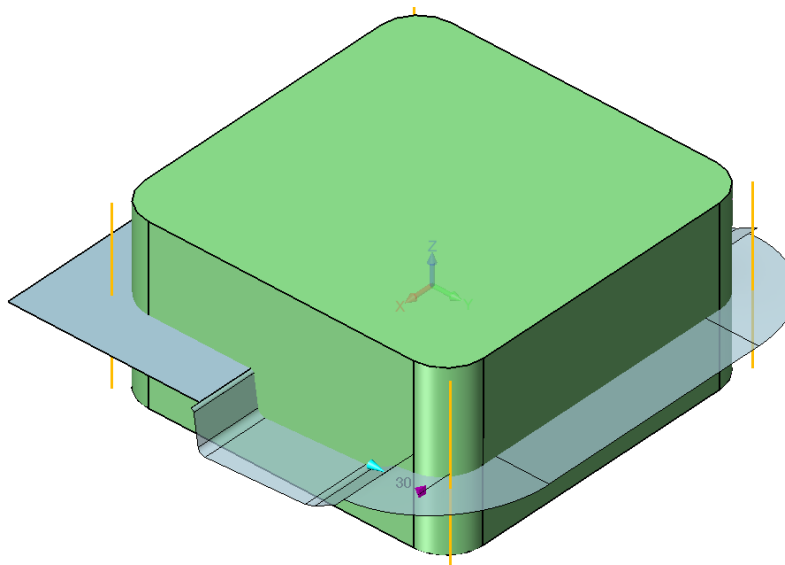


Next, we will use the same redefinition as above to make a little modification for **Workpiece**

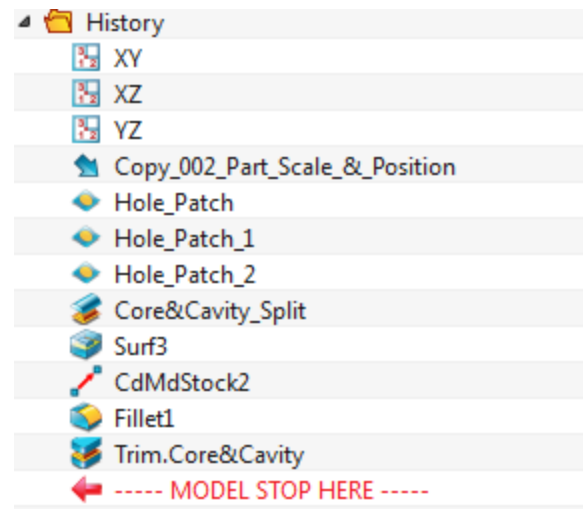
Drag the **history bar** to the position behind **Workpiece** feature



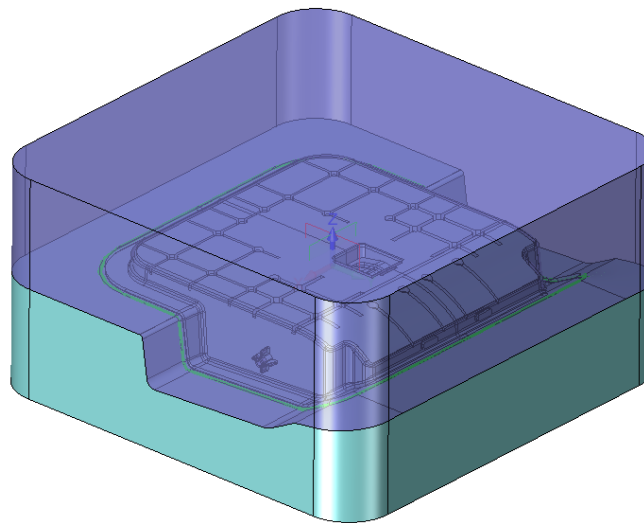
Apply **Fillets**  with a radius of “**30 mm**” on the edges as shown in the image.



Complete the history replay by clicking the **Replay all operations** button , or you can drag the history bar directly to the end.



After making the above modifications, the result should be shown as below:



*******NOTE:** If you find the shape color doesn't match the display image, you can define them by yourself, you can use any color you like.

Save the part.

We need a method for automatically updating the **004_Ref_Cavity** and **005_Ref_Core** to reflect these and any changes made to the **003_Core_Cavity** object.

The next step will be to update all the parts that are associated to the parting faces.

[illegible]

Link Manager



Select the ZW3D objects **LinkManager** found on the **Toolstab**.

Select **Insert** and select the objects to be updated. In this case, select the parts “**004_Ref_Cavity**” and “**005_Ref_Core**”.

Select **Select All**.

Check the option **Auto Regen** to turn it on.

If the user sets up the **link manager** to do an automatic regen of the **Reference** parts, each time the “**003_Core_Cavity**” part gets regen’d, the “**004_Ref_Cavity**” and “**005_Ref_Core**” part will be regen’d together.

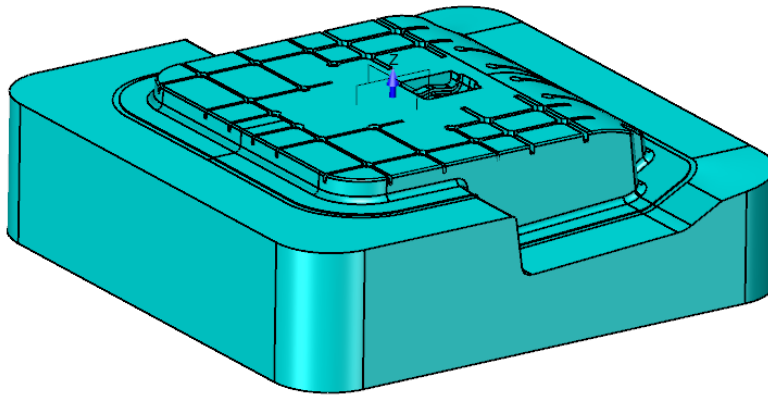
Select **Dismiss**.



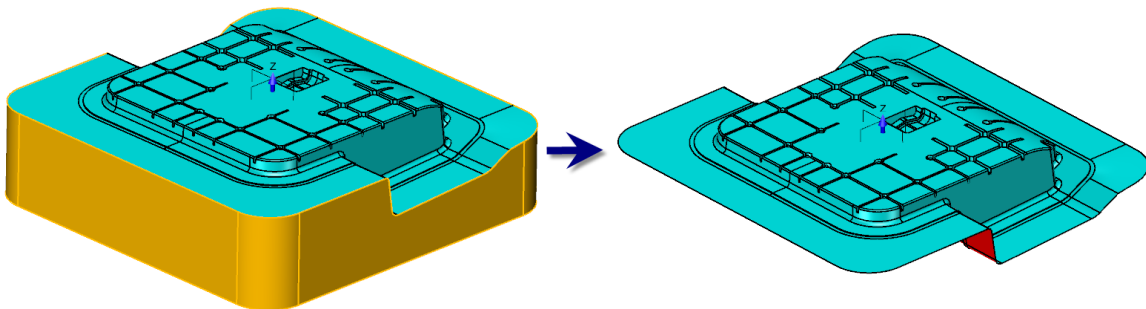
You can now run the **Regen** command . Watch the message window and you will see that the two reference parts are also regen.

The **'004_Ref_Cavity'** and **'005_Ref_Core'** are now updated according to the modifications that we created on the parting faces and workpiece.

[illegible]

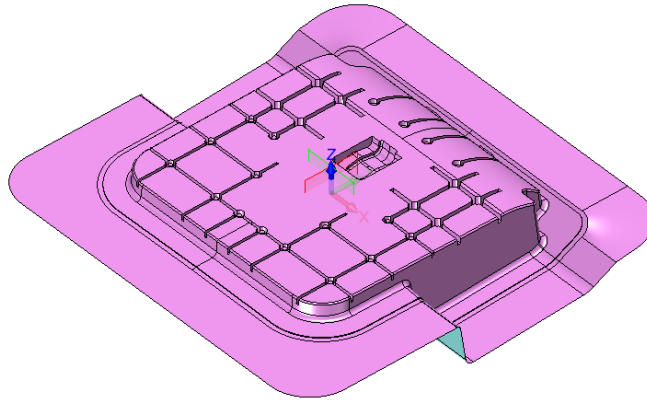


Use **Erase** command  to delete the faces as shown below.



The positive negative side of shape/faces should be oriented to Z+.

If necessary we can use **Reverse Face Direction** command  from the tab “**Free Form**” to orientate the shape/faces.



Create a new object with the name (Ex. **010_**). It will be used as a separator.

This separator is nothing more than a ZW3Dobject with a visually unique name. This helps us visualize and separate the different steps of the project in the **Objects** window.

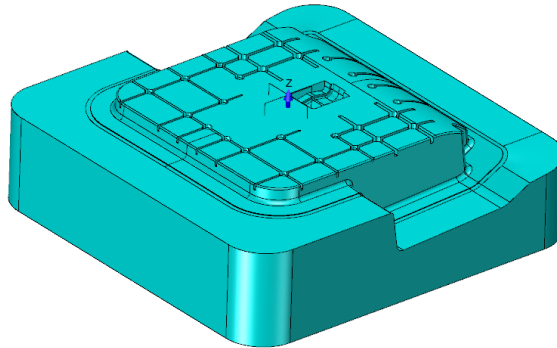
We suggest you follow this type of naming scheme. Since each part is based on the previous one, using numbers will help to organize your models in the object list.

Exit  the part.

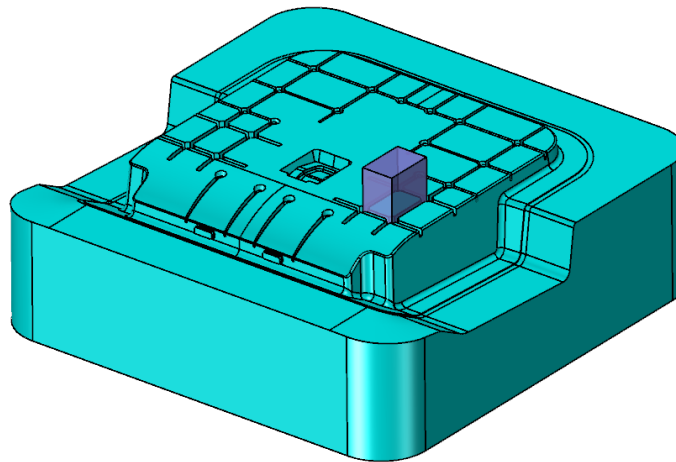
Save  the files.

Chapter 4 Creating Inserts

Exit to the **Object manager**, select the name “005_Ref_Core” from the object list to **edit**.



This cavity no longer contains any geometry. You will be creating an insert shape first.



Let's create an **Extrusion**.



Use the **Base** as a boolean option

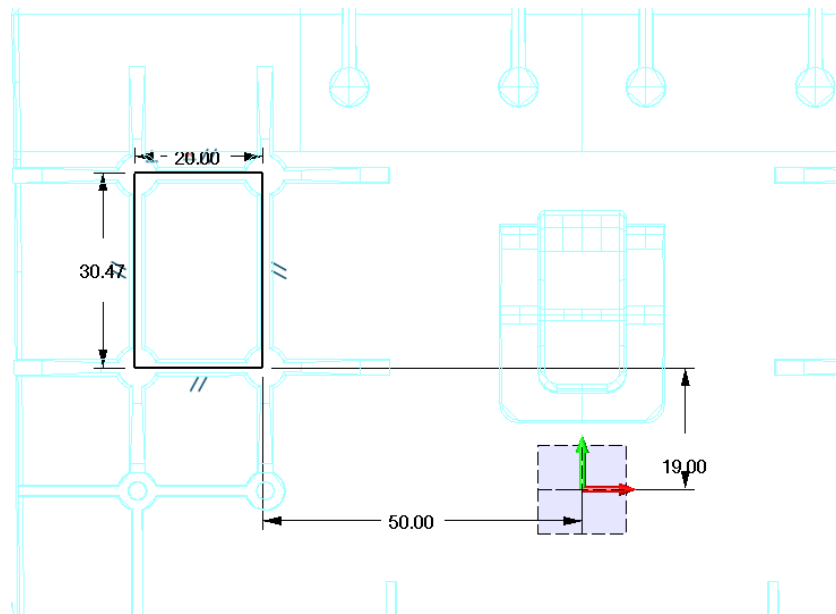


Profile P: Right click the graphic window **Insert Sketch**, Create a **rectangle**(corner option



) and dimension it from the point located at (0,0).

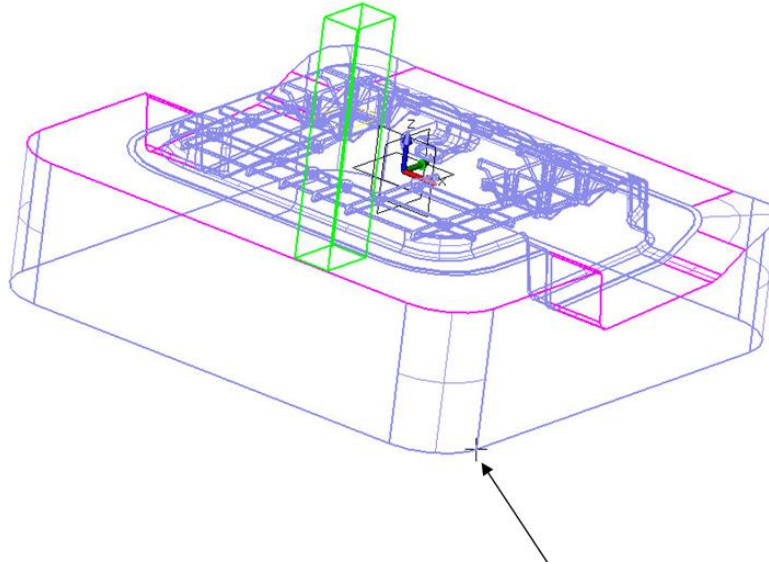
For this “Sketch” use the **dimensions** shown in the image below.



Exit Sketch by clicking Running man 

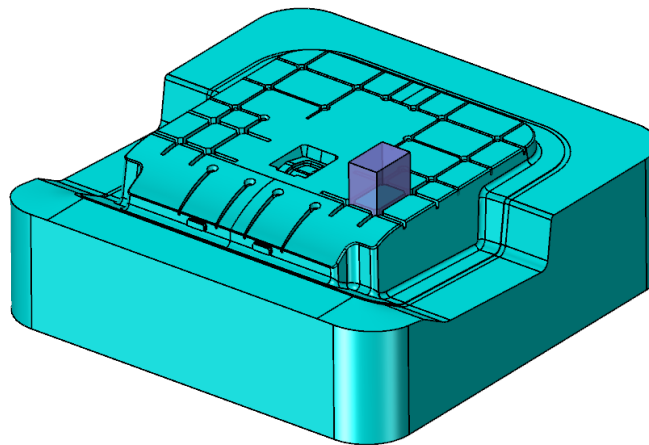
The **start** value of your extrusion will be the point shown in the image below. Right click on the graphical area and select **To Point**. Pick the point shown in the image below.

End value of “50”.



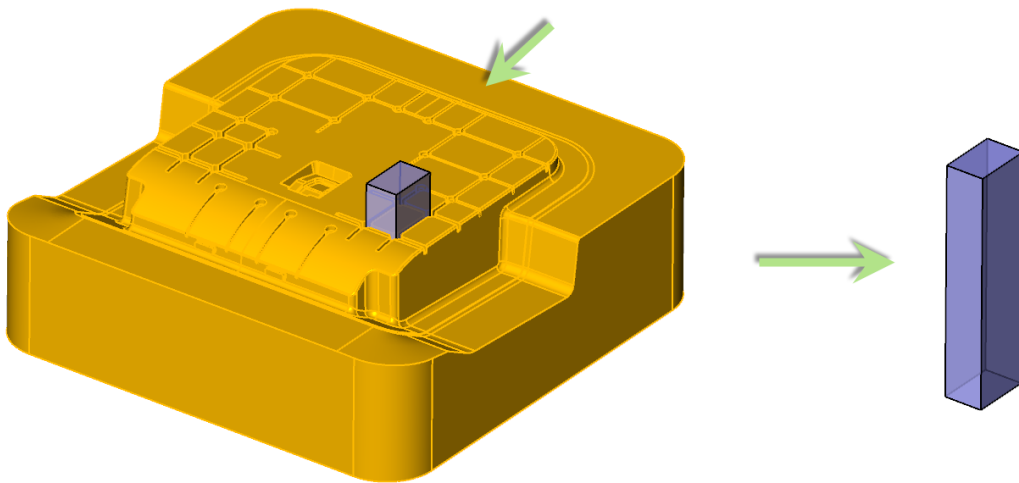
After creating the extrusion, select the **Shade** button. Your object should now look like this.

*******HINT:** You can set your favorite face color for the newly created shape at anytime.



Save the part.

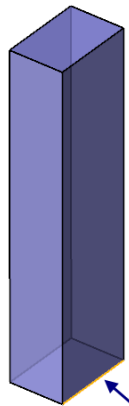
Hide the **Core shape**, by using the **Blank Entities** command icon.



Create an **extruded boss** on the new component.

Add an extrusion. Use **Extrude** command , choose the **Add** option. .

For the profile, we will use the edge from the extrude shape directly.

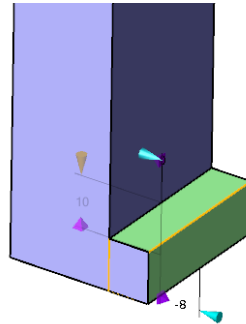


Extrude type: 1 side

End E: 10

Offset : Shrink/Expand

1st offset: 8

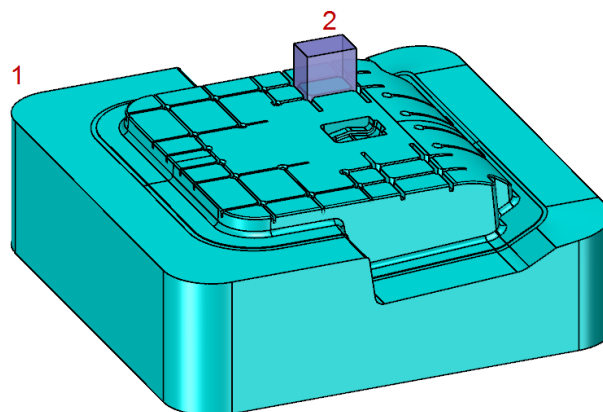


Save the part.

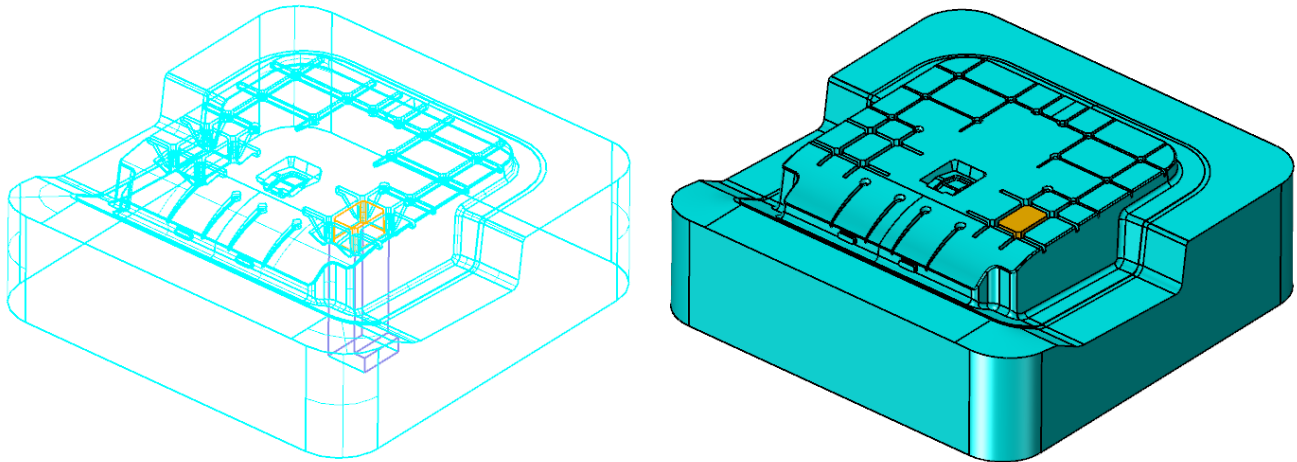
“Unblank all entites” so the core shapewill be visible.

Create a boolean operationbetween the two entities by using the **DivideShapes**command. 

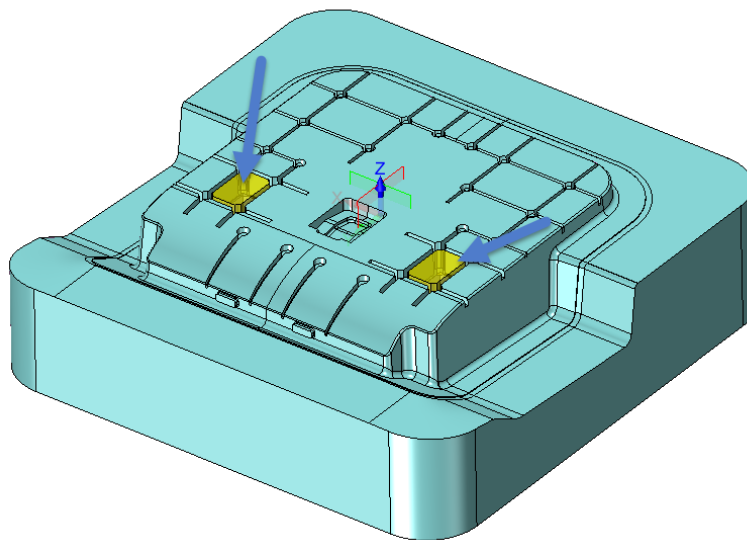
In this command, the **Core shape (1)** will be the object for the base and the object for cutting **(2)**will be the **previously created solid**.



When you have completed the command it should look like the image shown. Here we can change the insert shape to another color so that it can easily distinguished.



Now, there are two solid geometries existing in “005_Ref_Core”, optionally, let’s use the techniques previously employed in the construction of the first insert shape to construct the second insert shape.



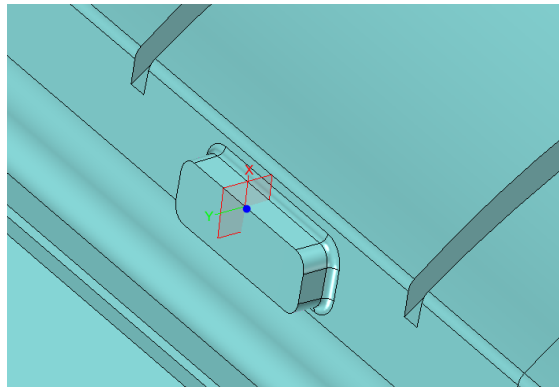
Chapter 5 Creating Lifters

For the next step of this exercise, let's create a '**lifter shape**' in the 'core'.

To create the shape of the 'lifter', you must first create an **Extrusion** base on a new datum plane.

Use the **Datum**  command from the **Shape** tab, then select the YZ plane, and set the origin point to the middle of line as shown below.

Hint: You can right click the graphic window then select the **Middle** option to snap the origin point.

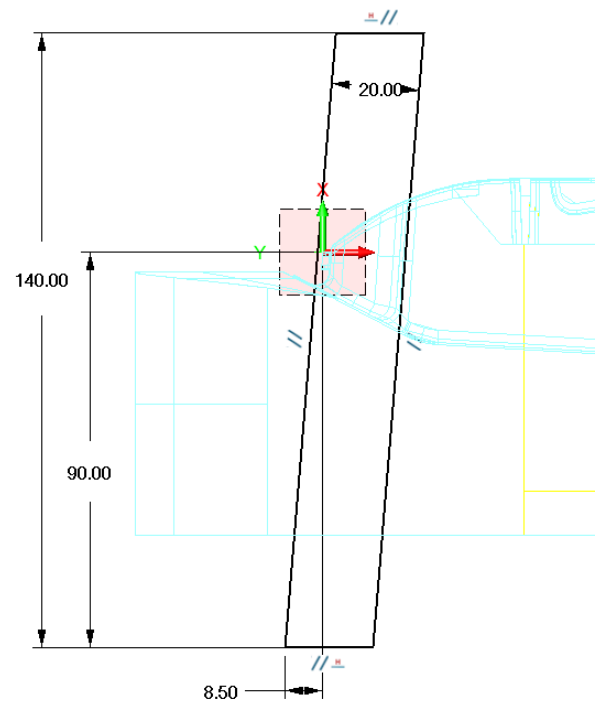


Use the **Extrude** command , then choose the **Base** option.

For the **Profile** option, Middle click to insert a sketch and go to **Up** input field , **use Z axis as the Up direction, after that** select the previous datum plane as the sketch plane.

(Using the sketch during Extrude command, please define the **Up direction** first, otherwise, after selecting the plane, ZW3D will use the default direction and enter the sketch environment directly)

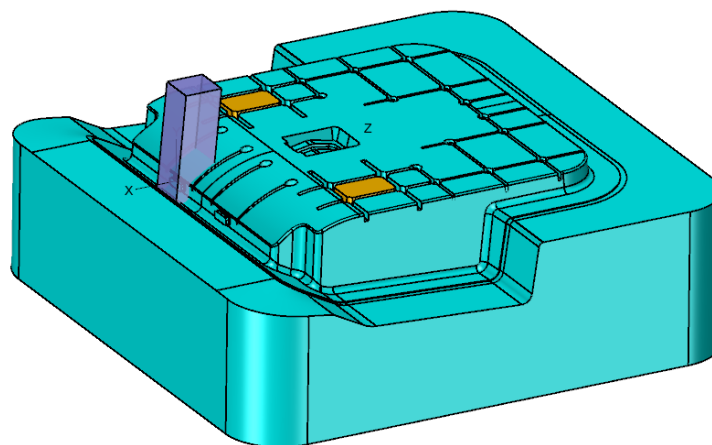
Create a profile as shown and set dimensions to "**0,0**".



Exit the “**Sketch**” and complete the extrusion with the values below.

Extrude type: **Symmetrical**

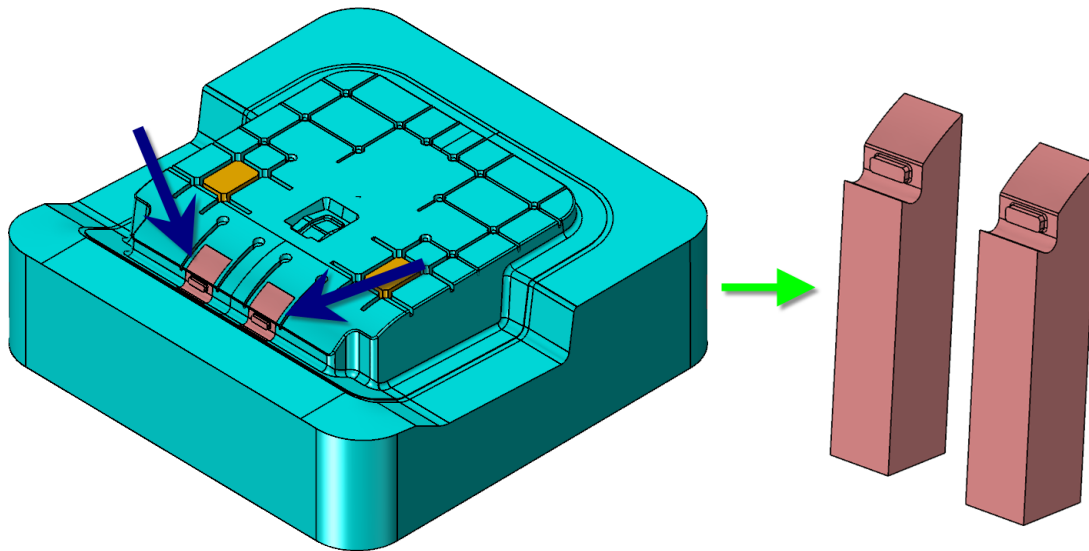
End = 10.



As an optional exercise, create another 'lifter shape'

Use the same operation as dividing Insert in previous chapter to divide the core shape.

After the final construction of the two 'lifter shapes', they should look like the image below.



After creating **insert shape** and **lifter shape**, they are still visualized as solid shapes on

005_Ref_Core, so next we will command **Geometry to Part**  to export them as independent parts.

Assembly -> Geometry to Part 

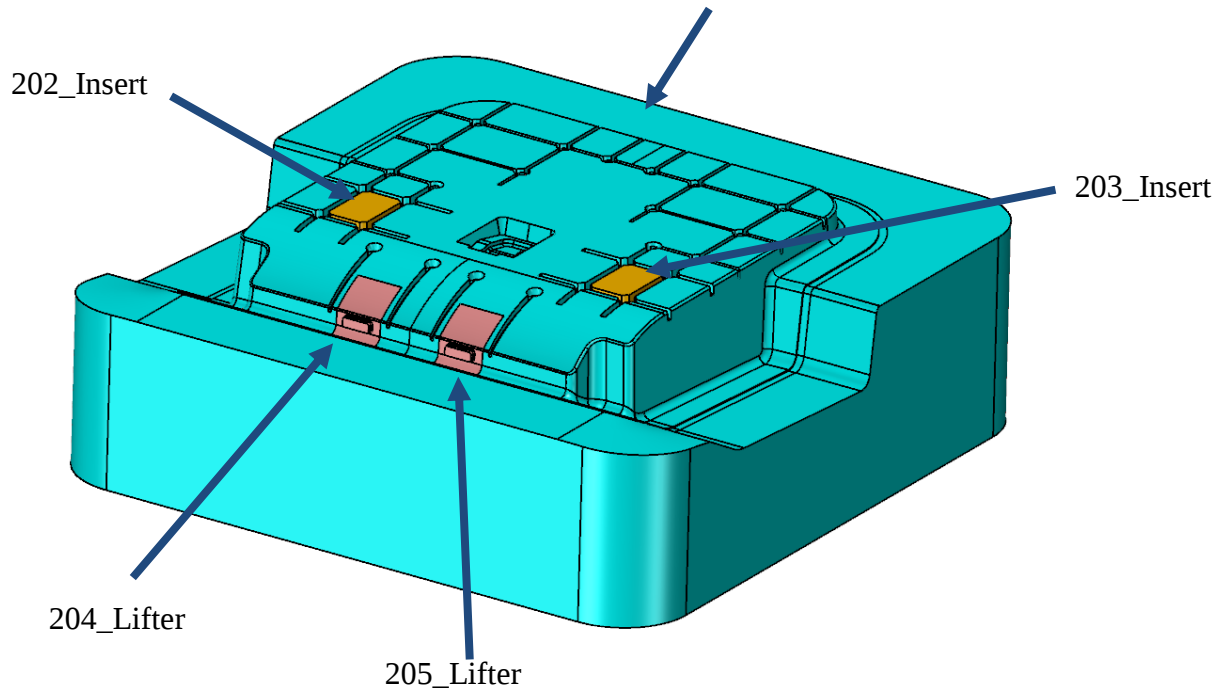
Geometry: Select the original core shape.

Create a new object name "**201_Core**"

Turn off all the options on **Setting**

Repeat the operation, Select insert shapes and lifter shapes respectively, as name for each part use "**202_Insert**", "**203_Insert**", "**204_Lifter**", "**205_Lifter**"

201_Core



After finishing, the object manager should look like:

Name	Type
001_Model_Part	Part
002_Part_Scale_&_Position	Part
003_Core_Cavity	Part
004_Ref_Cavity	Part
005_Ref_Core	Part
006_Epin_Cut	Part
010_____	Part
201_Core	Part
202_Insert	Part
203_Insert	Part
204_Lifter	Part
205_Lifter	Part

Chapter 6 Creating Sliders

To create **sliders** that belong to the cavity, we should **edit** the “004_Ref_Cavity”.

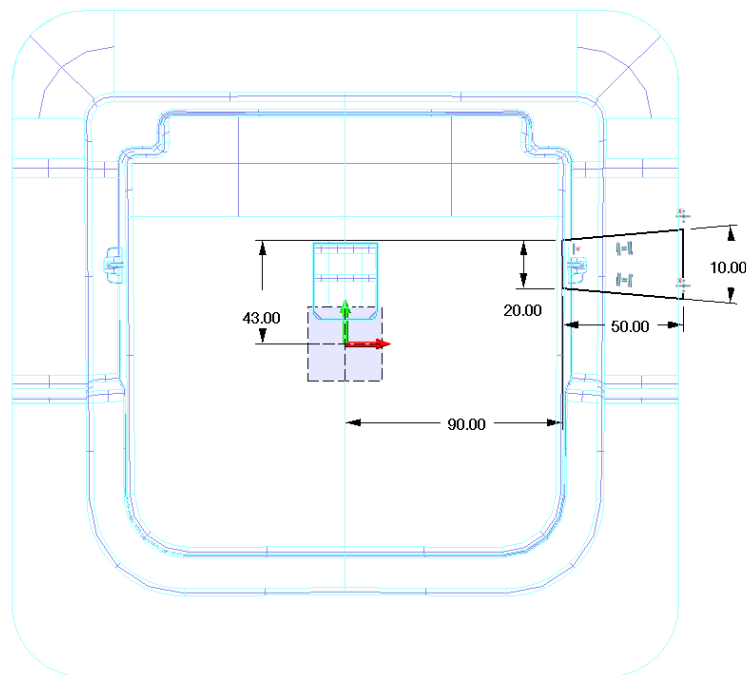
Exit to the object manager, then open “004_Ref_Cavity”.

Create an **Extrusion**.

Use the **Extrude** command , choose the **Base** option.

For the **Profile** option, middle click on the graphical area to insert a sketch .The sketch should be on the **XY** datum plane.

Create a shape just like the one shown in the image. Set the dimension to “0,0”.

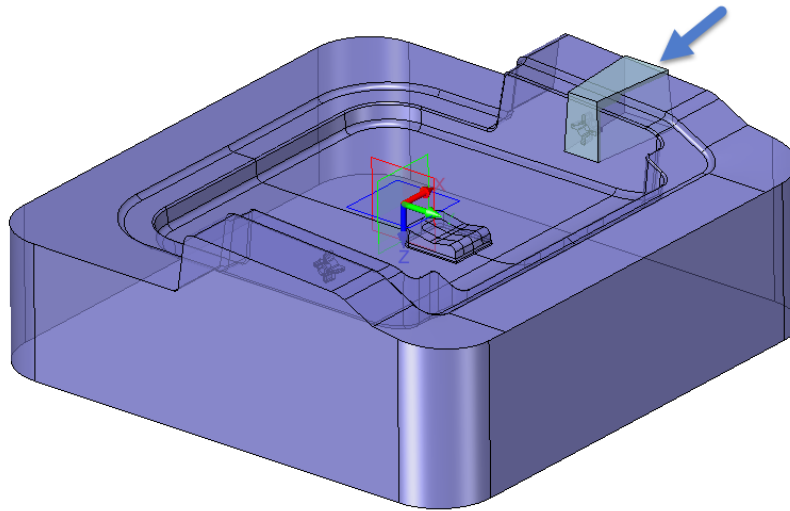


ExitSketch.

Start = 0. End = -30.

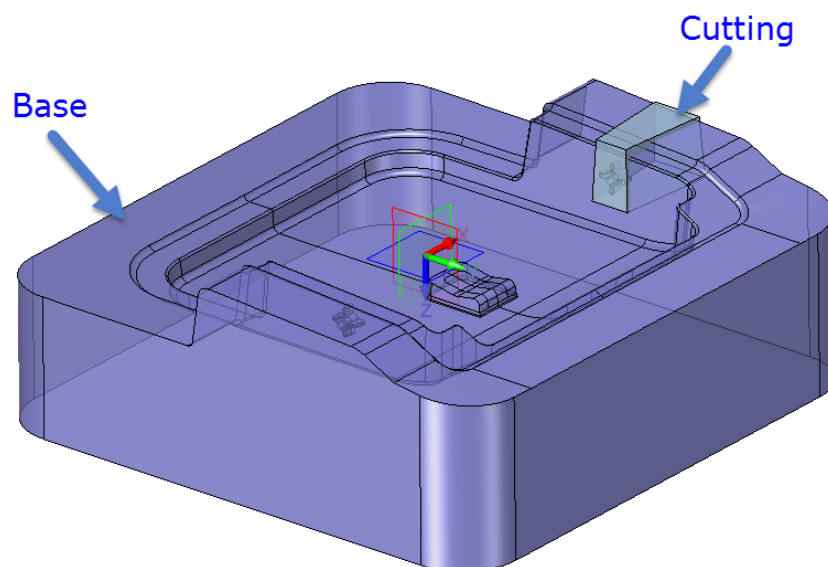
Draft angle = -3.

See the result.

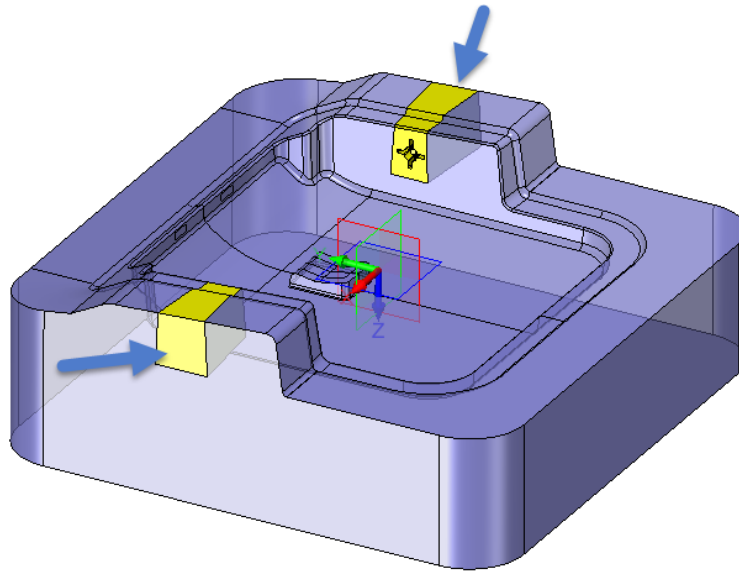


Use the **DivideShape**  command to create a trim operation between the 2 entities.

The base is the '**cavity shape**' and the solid previously created is the trim object.



Use the same process to create an extruded shape on the other side then use it to divide the original cavity shape again, the result should be as shown below.



Next we still need to export those two slider shapes to independent parts.

Edit -> Copy -> Geometry to Part 

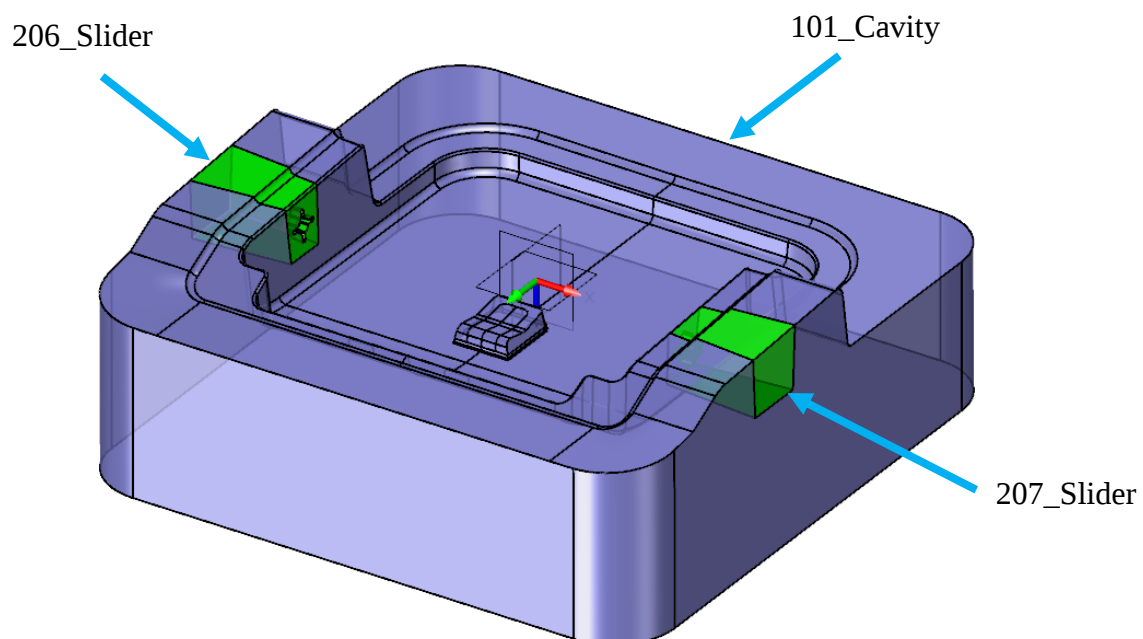
Geometry: Select original core shape.

Create a new object name **“101_ Cavity”**

Turn off all the options on **Setting**

Repeat the operation, select the slider shapes, and name each part respectively **“206_Slider”**, **“207_Slider”**

*****NOTE: Although those sliders were divided by “004_Ref_Cavity”, they should belong to a core group, so we mark it’s name with “2” letters.



Name	Type
001_Model_Part	Part
002_Part_Scale_&_Position	Part
003_Core_Cavity	Part
004_Ref_Cavity	Part
005_Ref_Core	Part
006_Epin_Cut	Part
010_____	Part
101_Cavity	Part
201_Core	Part
202_Insert	Part
203_Insert	Part
204_Lifter	Part
205_Lifter	Part
206_Slider	Part
207_Slider	Part

We are still not finished with creating the slider, next let's insert the slider part to add some geometry.

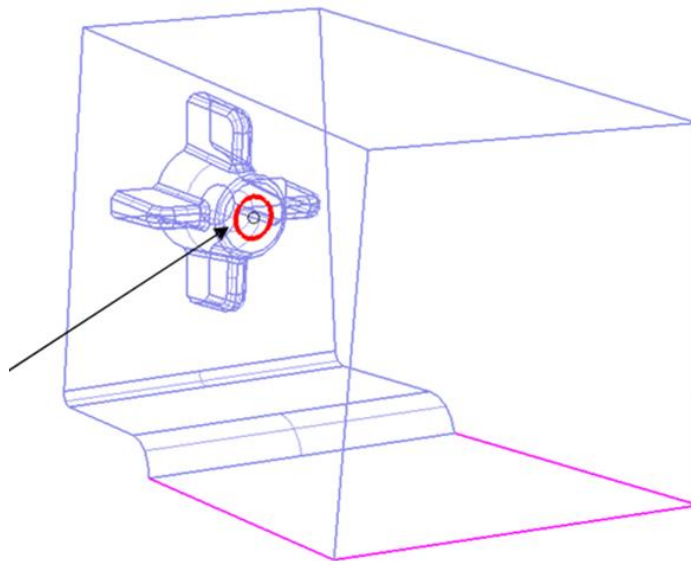
Exit to object manager. 

Double click "**206_Slider**" to insert.

In order to create an extension for your 'slider', you will need to create a reference point.

Open the **Wireframe** tab and select the **Point** command. 

Select the options **Critical + Center of Curvature** from the right mouse button menu. Select the edge on which to locate your point as shown in the image.



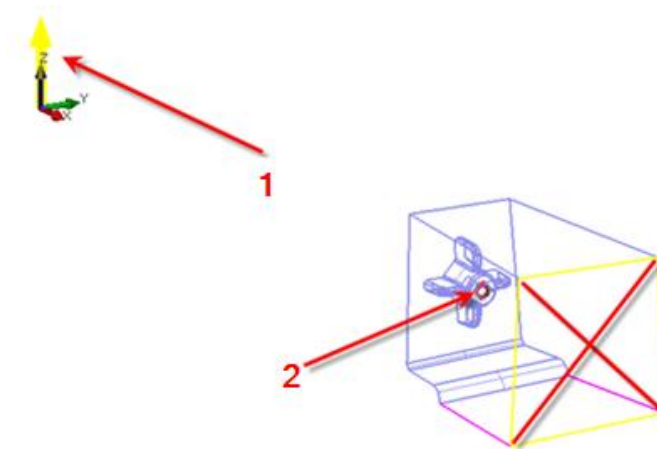
Insert Sketch as shown in the image.

Up until now, we were not concerned where the origin of a sketch was when the sketching was on a face.

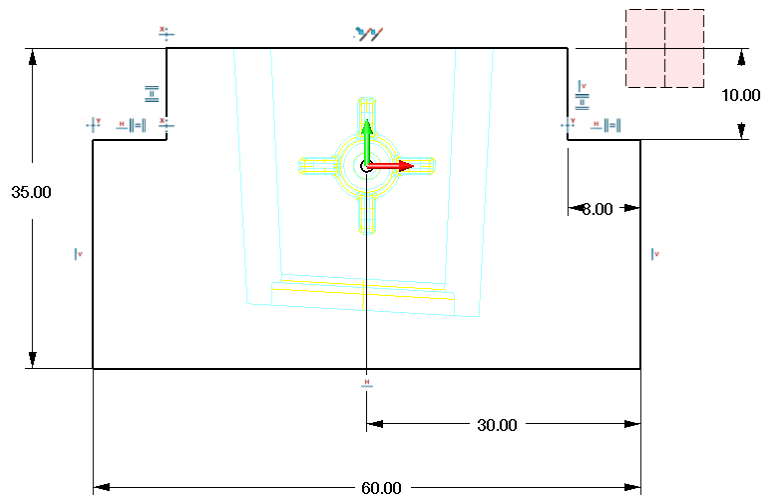
Now we want the origin to be projected from the point we've just inserted.

Use the **Origin** option and pick the point entity that we inserted in the last step (as indicated by arrow 2 below).

Pick the line indicated by the arrow for the **UP** direction (as indicated by arrow 1 below).

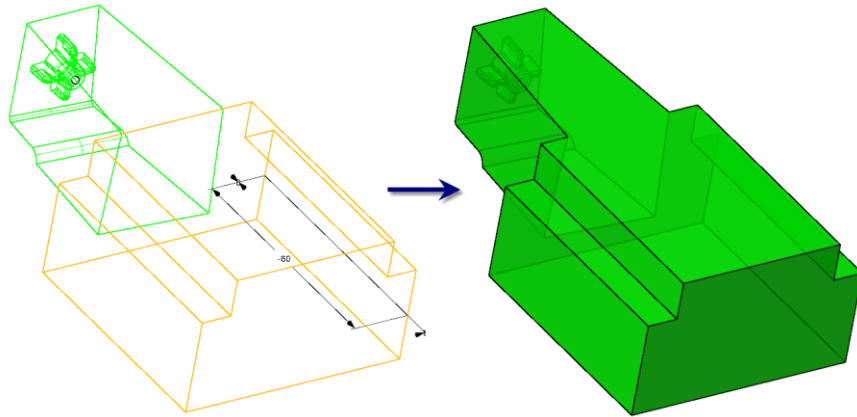


Draw the sketch shown below. (Start where the arrow points and draw clockwise)



Exit Sketch.

Extrude using the **Add** option, with a start value of “0” and an end value of “60”.
If your sketch was drawn correctly this seam will disappear.



Create a **draft angle** for your slide.

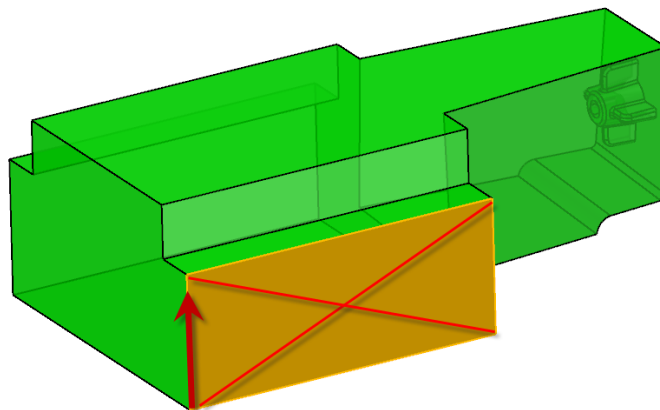
You'll need to create an **Extrusion** using the **Remove** option.



<**Middle-click**> to create a new sketch.

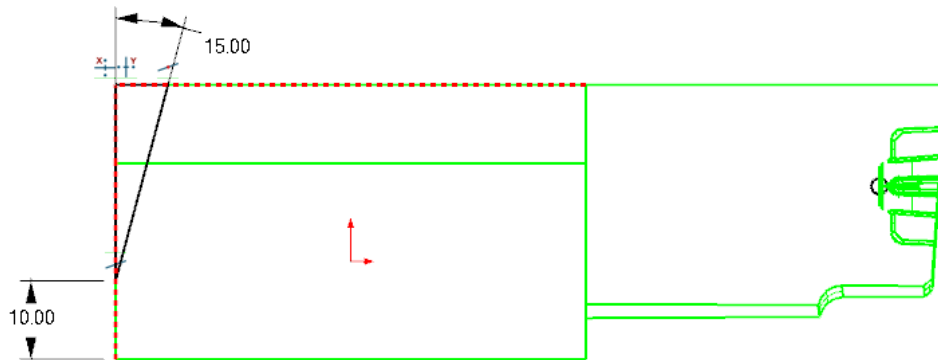
Select the **Up** option on the command's dialog menu and select the edge where the red arrow is located in the figure below.

The sketch should be on the face marked as shown below.



Look at the following figure. Create a **reference curve** on the three edges in the upper left corner of the part. (One horizontal and two vertical)

Create the lines as shown. Use the suggested **dimensions**.

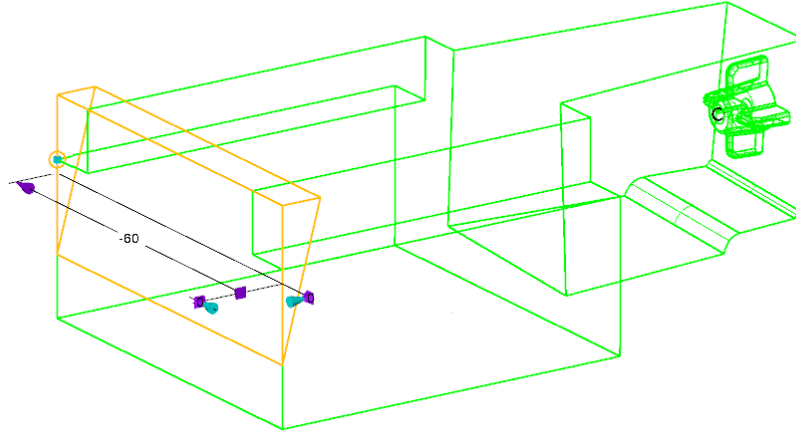


Exit your sketch.

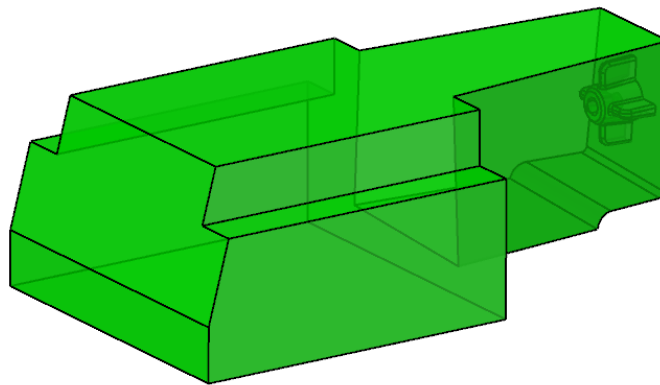
Complete an **extrude** using the **remove** option.

Start= 0.

For the **End** option, right click on the graphical area and select **To Point**, and then pick the point in the red circle shown in the image below. (The value of the **End** option should be -60)



See the result below.



As your final (optional) exercise, use the same process to add geometries to another slider
“207_Slider”

Chapter 7 Creating a Moldbase

Create  a **Part**  for the 3D Mold Assembly, which uses the name “000_Mold_Assembly”

For the next step, we will use the **Multi insert** command from the **Tools** tab 

Multi Component Insert

This command will insert all components in one step, while it is also possible to apply the constraint “Anchor”. Using this function we will simplify the assembly process for the core, cavity, inserts and so on.

From this function dialog box, select the **XY** plane as **Datum** plane.

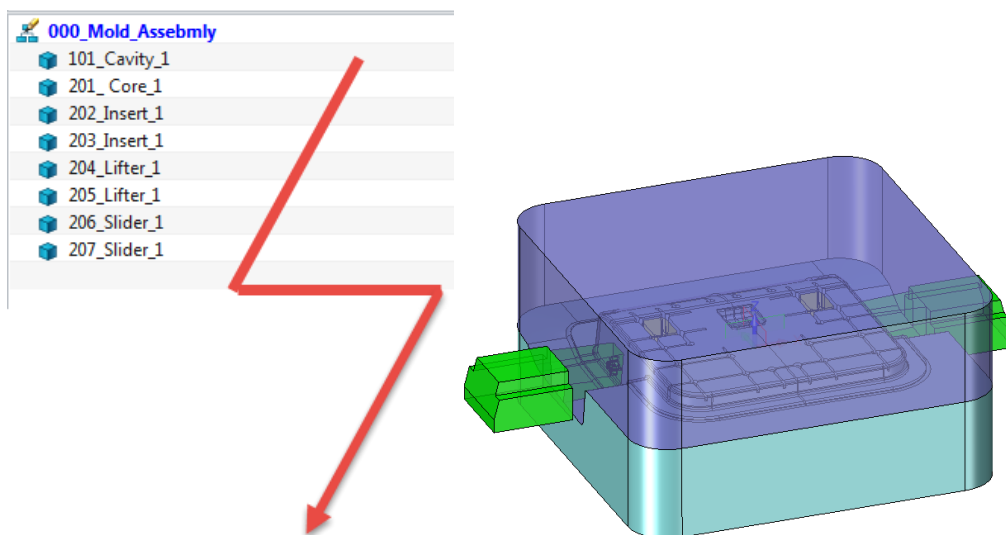
Pick the part names from the **List Components** and add them to **Components to Insert**, including:

“101_Cavity”, “201_Core”,

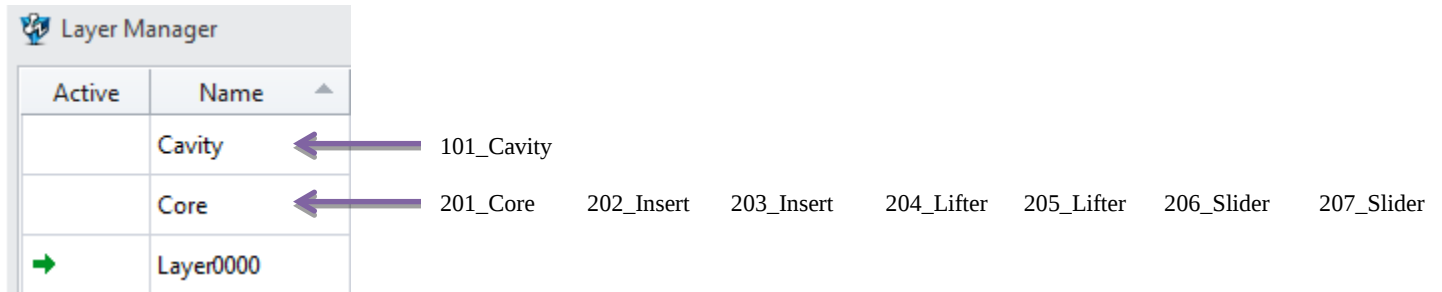
“202_Insert”, “203_Insert”, “204_Lifter”, “205_Lifter”, “206_Slider”, “207_Slider”

Turn on the **Anchor Components** option.

Press **OK** to finish this command, now we have inserted 8 components to “000_Mold_Assembly”.



Create two Layer call Cavity and Core, then assign the part to a specific layer as shown below.



Now we will insert a mold base, using the command **MoldBase** 

This command will insert the standard mold base. Here we will use **LKM** as the Supplier

As the mold type we will select **SG**

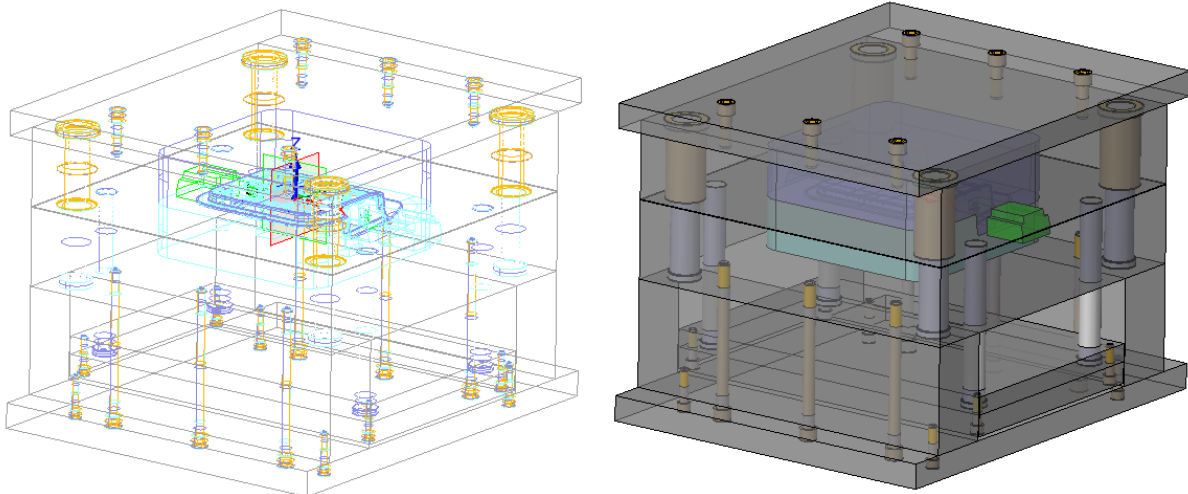
Select the **XY** as Datum plane or keep it empty (Empty means it will use XY as default)

For **Mould Series**, select “**500x500**”.

Set the parameter as below for **Normal** and **Advanced** options

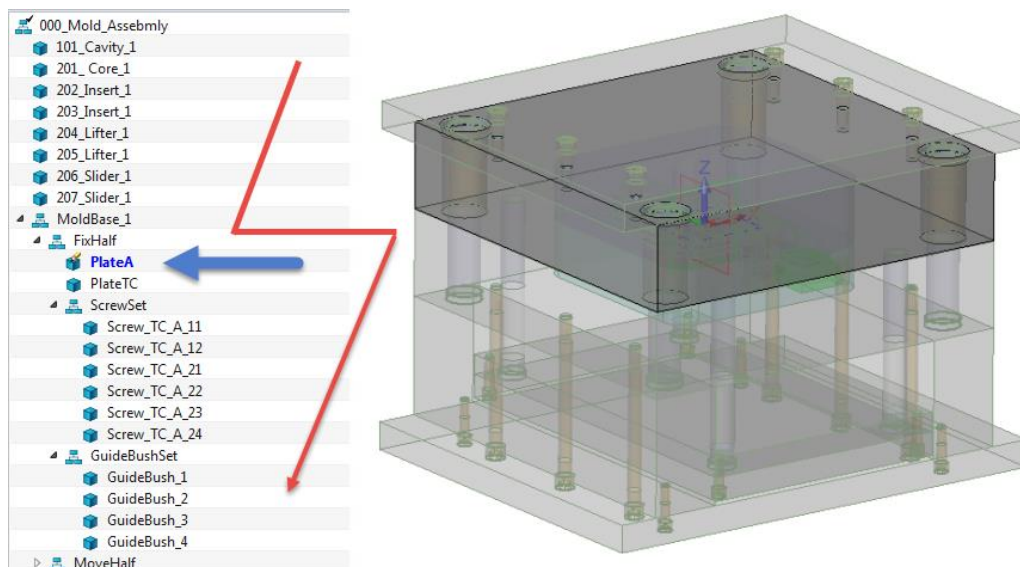
Normal		Advanced		BOM	Normal		Advanced		BOM
Item	Value				Item	Value			
Class	C				Rotate	90			
Type	I				Cavity Gap	0.5			
A	110				Core Gap	0.5			
B	110				Ejector Gap	5			
C	150				Plate Chamfer	0			

Click **OK**, you will see the result as below

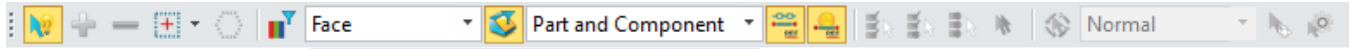


Now we will edit the core and cavity plates to open the pocket for the core and cavity components.

Double click **Plate A** to activate it in **Assembly manager**, the result will be as below.

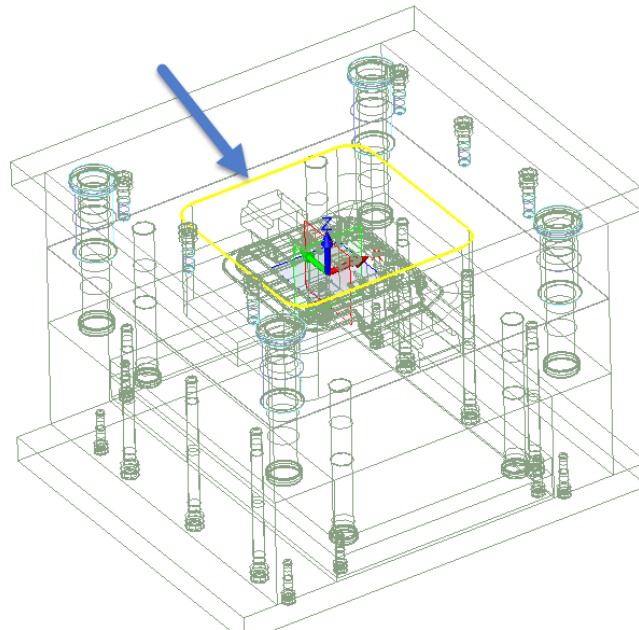


Use **Extrude** command then set the filter to **Face** from **Part and Component**



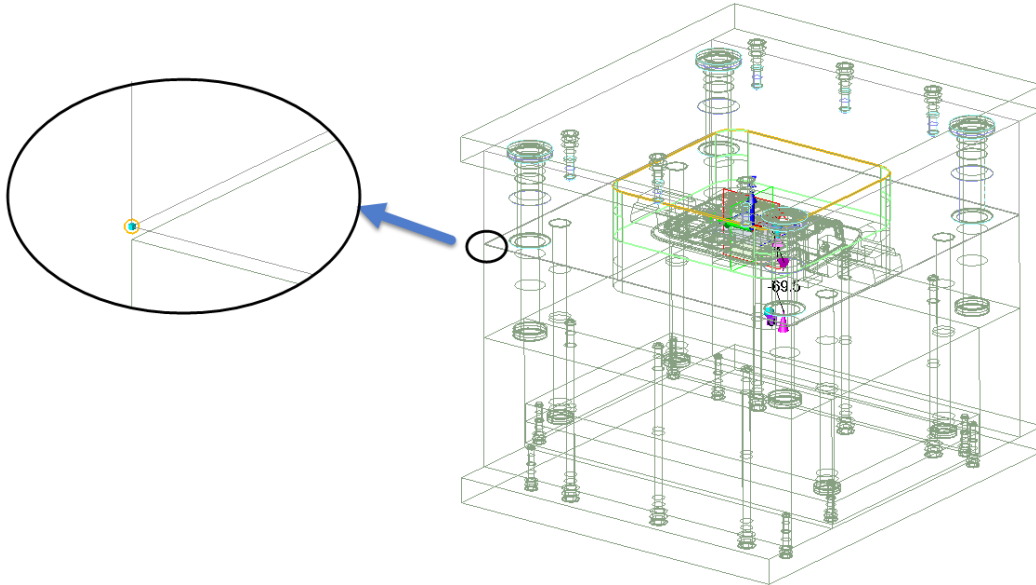
Then we need to select the bottom face of the cavity as shown below

*******NOTE:** In order to select the back entities on the graphic windows, there are some tips we can master. Changing the display mode to **Wireframe mode**, Right click using the **Pick from list** function.



Start = 0.

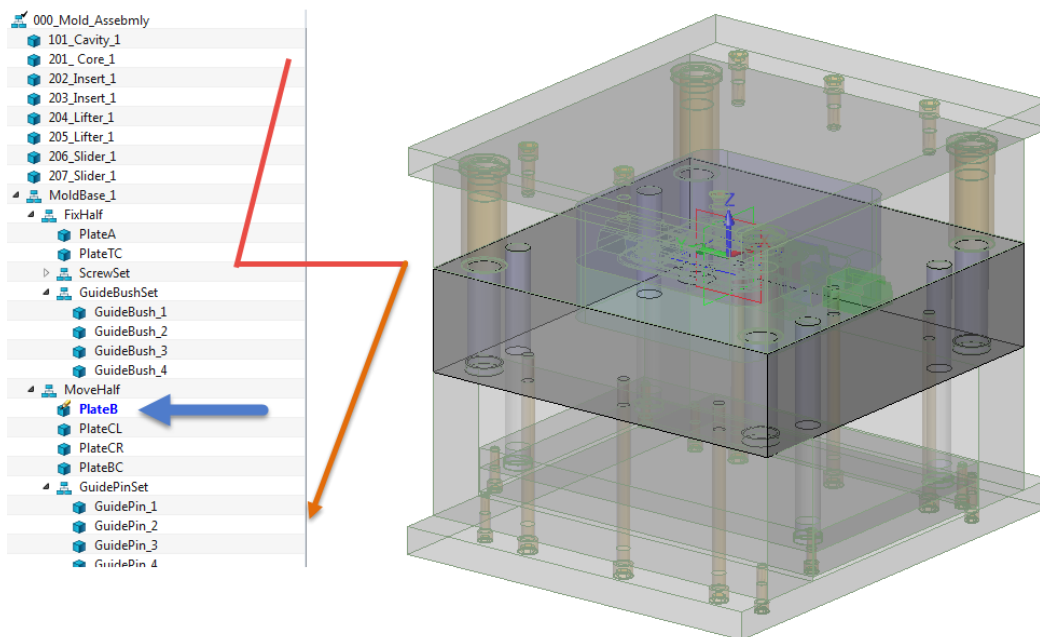
End = Right click the graphic window then use **To point**, select the point as shown below



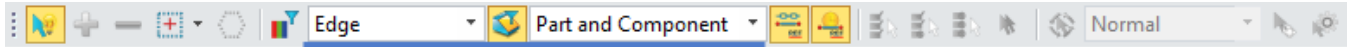
Set the Boolean option to **Remove**.



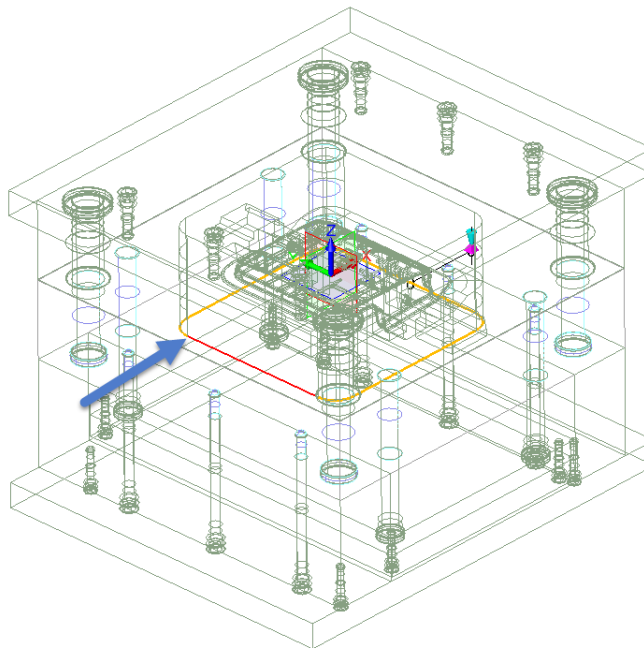
After that, double click **Plate B** to activate it in **Assembly manager**, the result will be as below.



Use **Extrude** command then set the filter to **Edge** from **Part and Component**



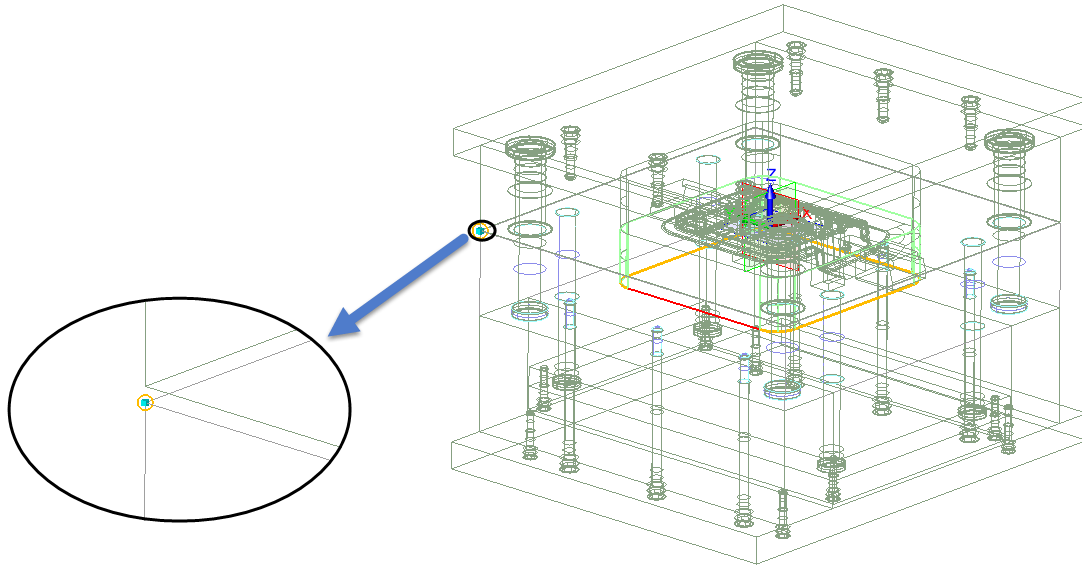
Then we need to select the edges of the bottom face in the core part as shown below



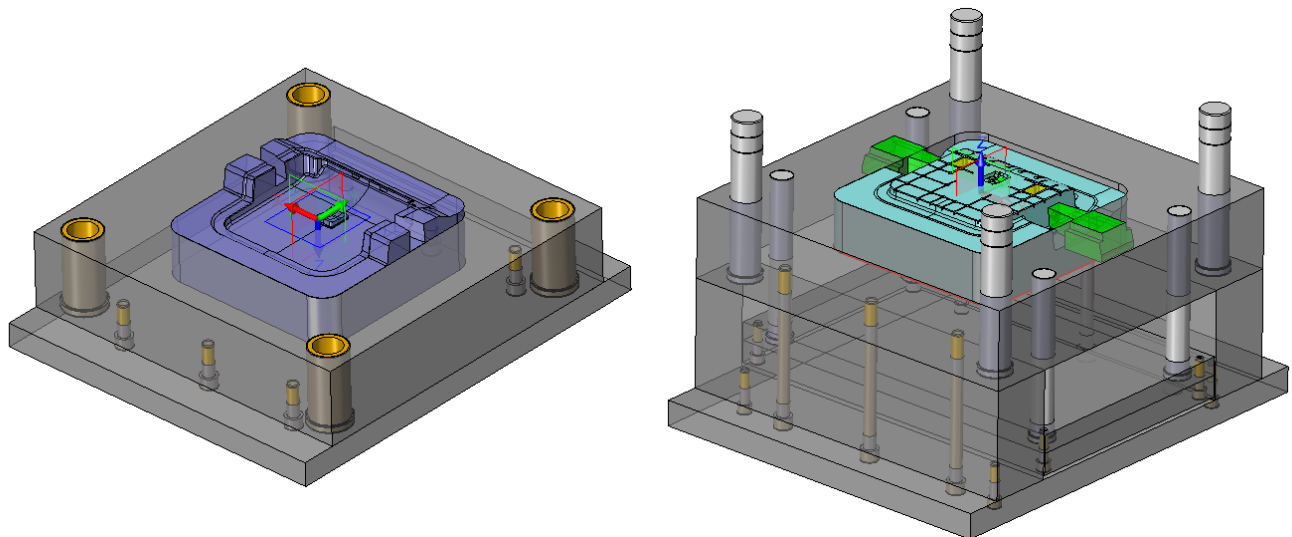
*******NOTE:** When we select the edges, except setting the filter, we need to right click then use Curve list to select all of them.

Start = 0.

End = Right click the graphic window then use **To point**, select the point as shown below



After finishing the process, the final result of the cavity and core component should be like the one below. You can go to the **Layer Manger** and use **Turn on/off layer** to control visibility.

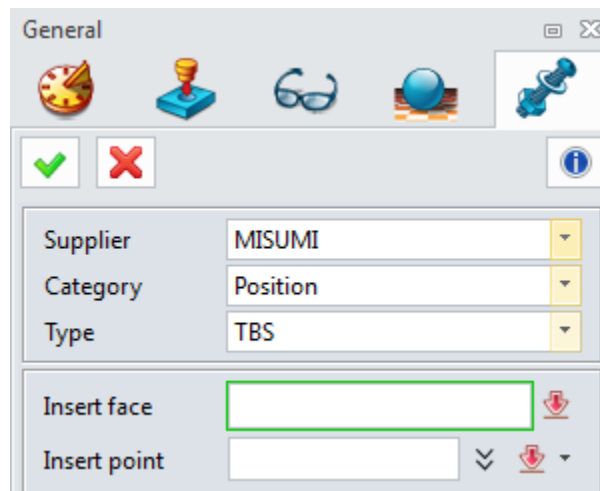


Chapter 8 Creating a Standard Library

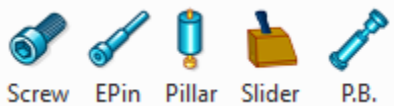
Standard Component Insertion

General standard component

This kind of standard component uses indicated **Face** and **Point** to insert the standard components



Specialized standard component



Those specialized standard components have more opportunity to be controlled, what's more, some input fields can be inputted automatically.

As an example, we will use a screw to show most of the functionalities.

To place components, we recommend using sketch. Based on the sketch points for the standard components, the location is faster and also much easier to use than any modifications of component locations.

For this exercise, we will use the sketch wizard to define standard component locations. This command is a tool that can quickly create a different way to sketch and place components.

Now, we will create the sketch for the location of screws for the Core and Cavity inserts. Select the command **Sketch**  from **Tools** tab.

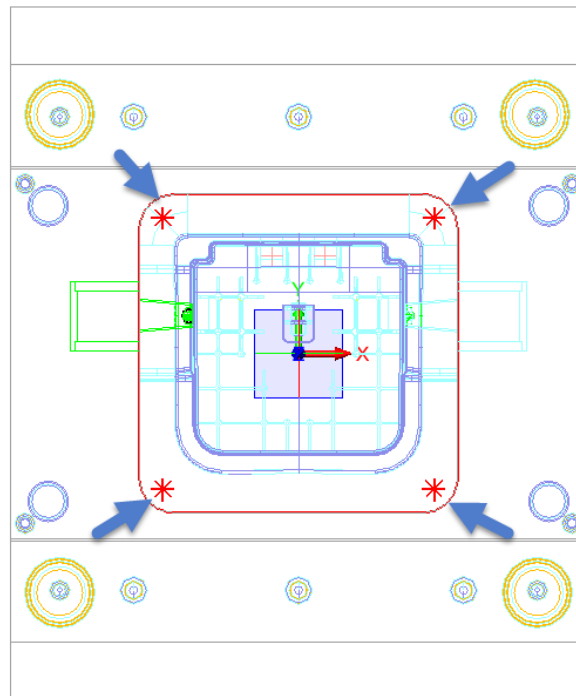
Datum use the **XY** plane

Sketch type use **4A**

D1=235

D2=235

The final result should be as shown below



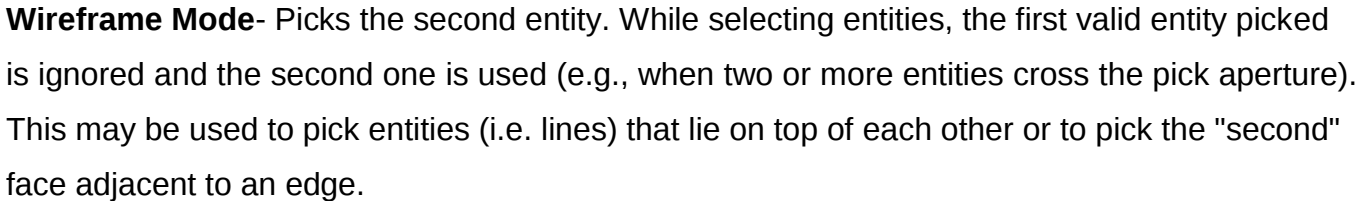
We will add screws based on the previous sketch.

Select the Command **Screw** 

Start Plate: For the start plate select “**PlateA**”

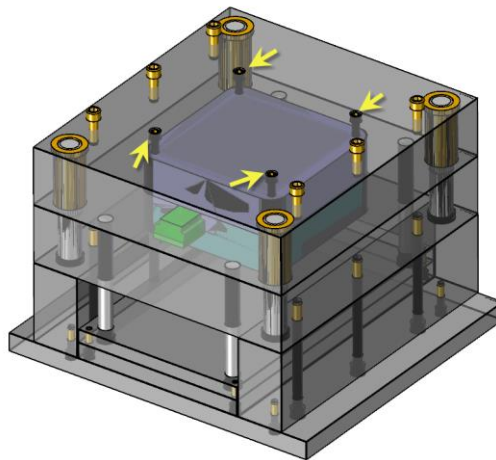
Start Face: For the start face, select the Top face of **PlateA**. This input field can only select a face that belongs to the **Start Plate**

*******HINT:** Sometimes to make the selection of a face between plates easier, we need to hide an object, or use ZW3D filter using the Alt key for selection in a different level.

[illegible]

Length (L): The system will automatically select the length, but the user can specify any length manually (65).

Click **OK** to finish, blank the **PlateTC**, and the image is shown below.

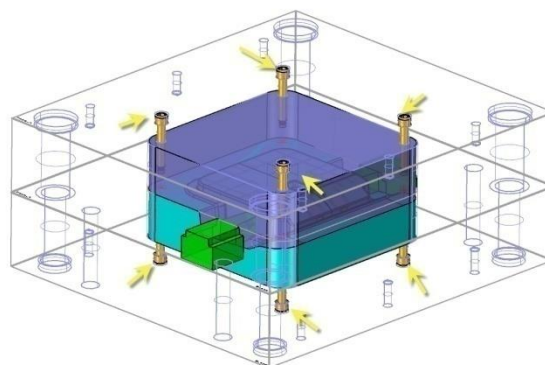


Now, we have inserted 4 screws for anchoring the cavity part. Optionally, we can use the same process to insert four other screws for anchoring the core part.

There are no detailed steps for this process, you can try to do it as personal exercise.

After that, you can use **Blank on** some other components or right click the component > Display >

Wireframe, to set the visibility.



In the next step we will place the **Locating ring** using **General standard component** 

Supplier: MISUMI

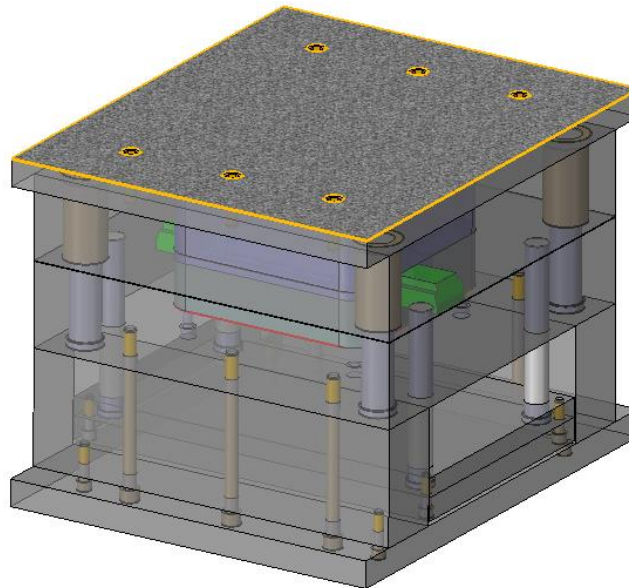
Category: LocatingRing

Type: LRBS

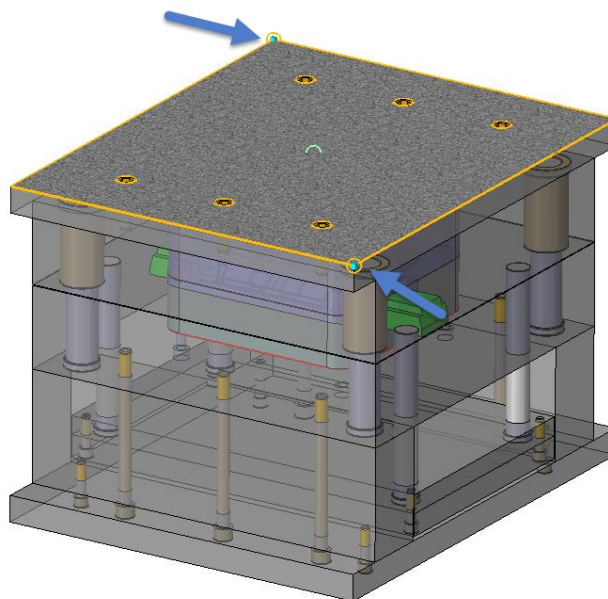
D: 120

T: 15

Insert face: Select the Top face of **PlateTC** as shown below

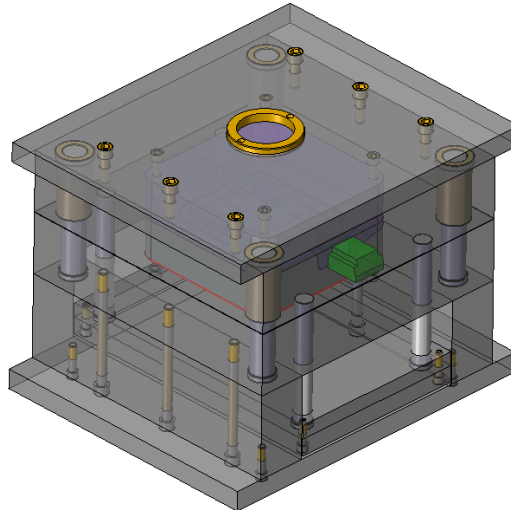


Insert Point: Right click the graphic window then select **Between** as a way to define the points, then select those two points as shown below.



*******NOTE:** You can go to **Advanced** option to define creating pockets by selecting yes or no.

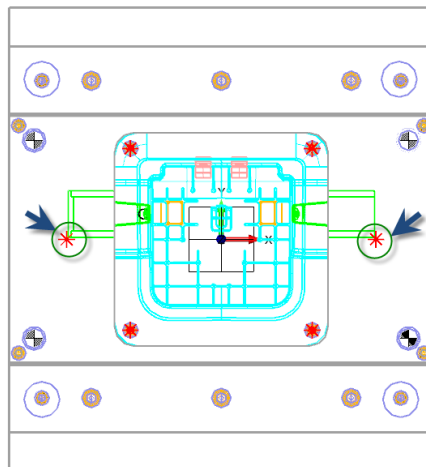
After that you can see the result as shown below.



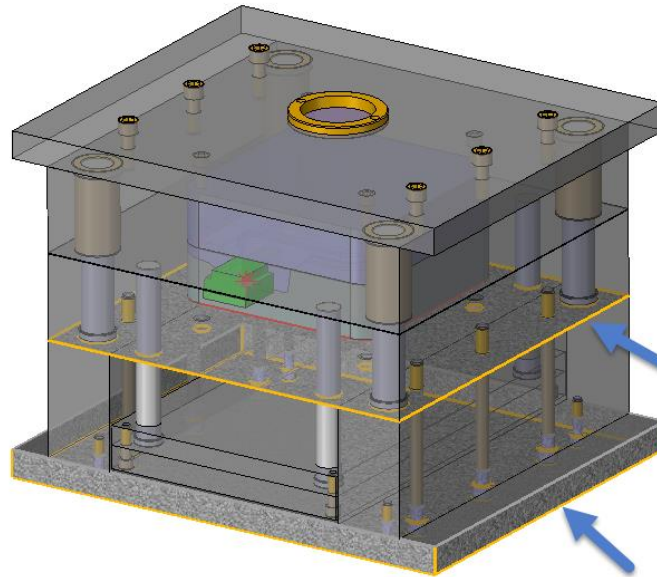
The next step, we will introduce support pillars. Before that we will also use the sketch to place the components.

Select the command **Sketch**  from **Tools** tab.

Use the sketch wizard to define the standard component location. For the type, choose the **Sketch 2C**, and for **Dim** (d1) use 400.



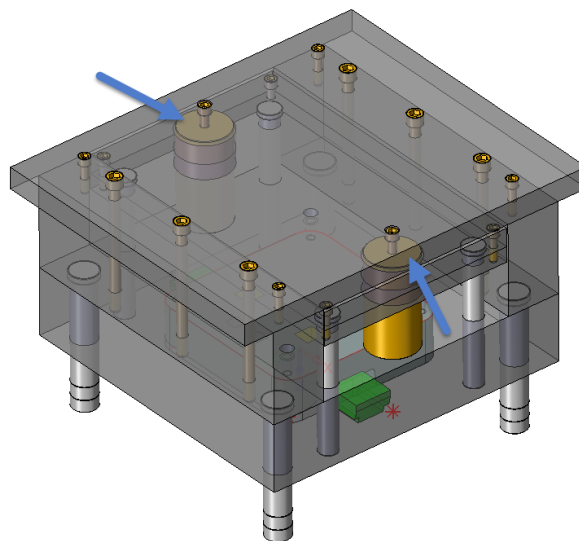
We will use the **Support Pillar**  command to insert the standard component. Once you enter this command, the locating face and plates have been selected automatically. You can see the result as below.



Here we will keep all options as default except **Insert point** and **Diameter(D)**

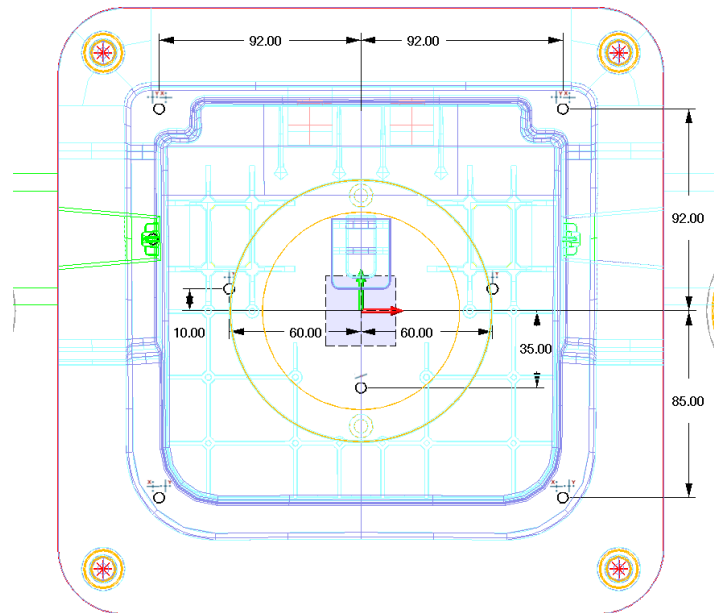
Insert point: Right click the graphic window and select the sketch what we created just now.

Diameter (D): 80



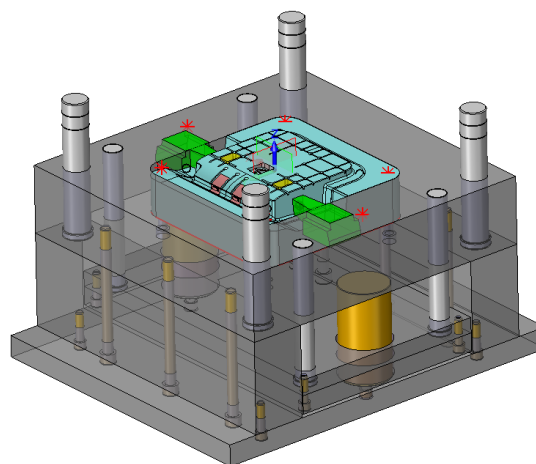
Next, we are going to add **Ejector pins** to this mold.

To create the ejector pins, make a new sketch on the **XY** datum, and create points as show on the image.



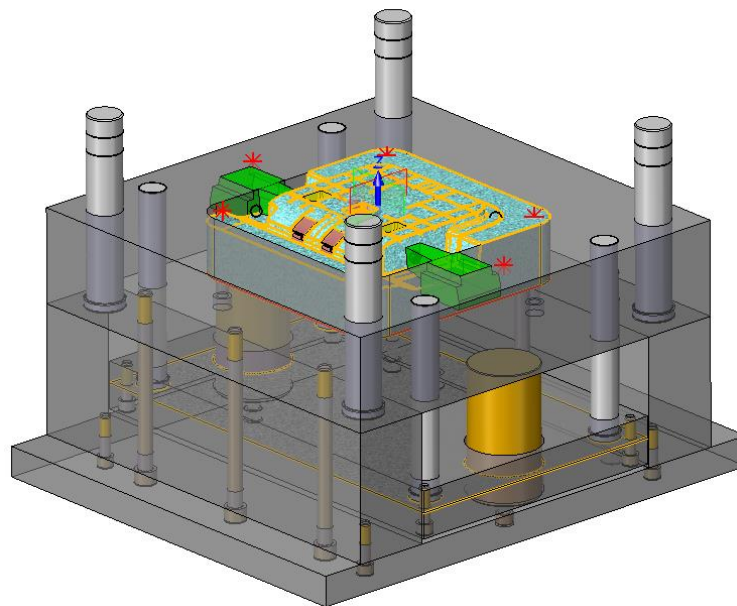
To finish the sketch, select **Exit** 

Before moving on to next step, please turn off the **Cavity** layer and use Visibility command to hide the Fix half of the Moldbase.



Now we will insert **Ejector Pins** using the previous sketch. Select the **Ejector Pin (E pin)**  command from the **Library** tab, the assembly part will be shown as below.

*******NOTE:** Because **E pin**  is the command of Specialized standard component, the Insert face and Core shape will be selected automatically, you can still select them by your self if you found they haven't selected.



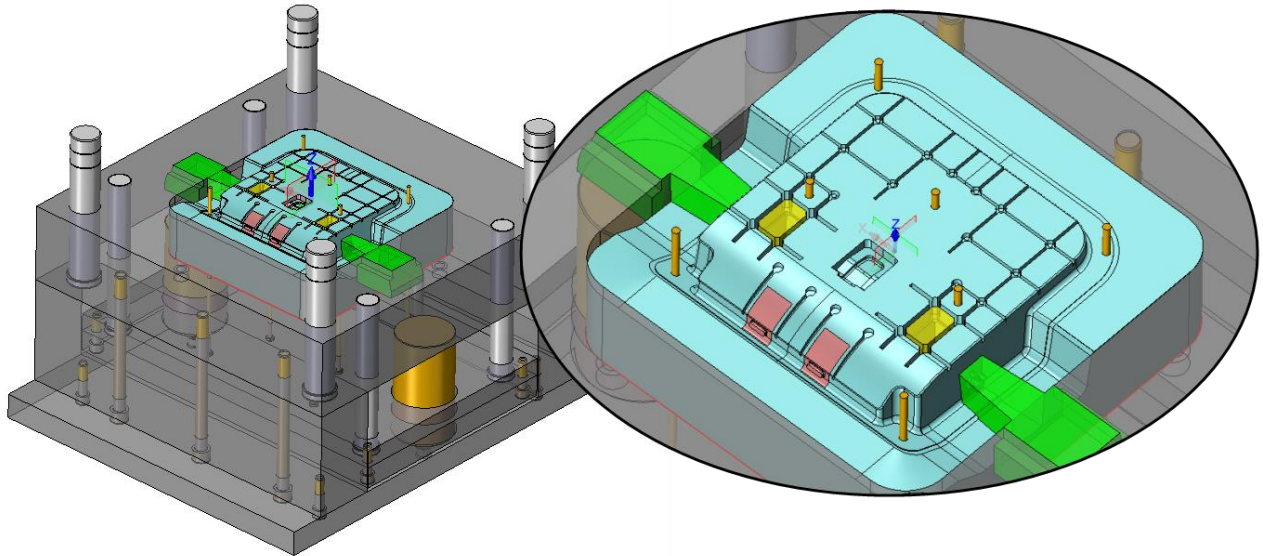
Supplier: DME

Type: A

Diameter: 5

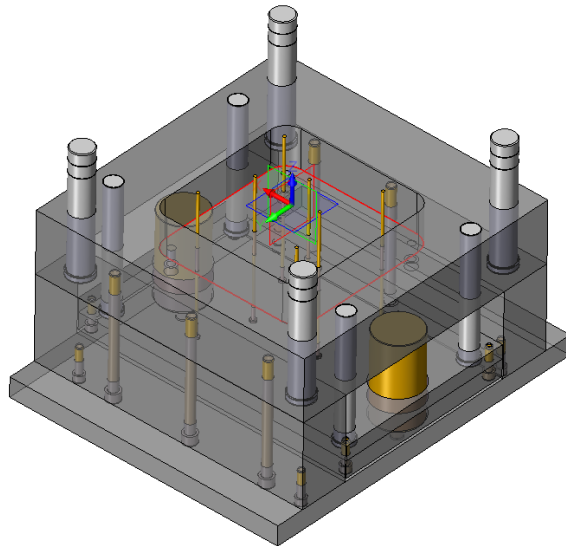
Length: 250

Choose to finish this command. The result should be as shown below.

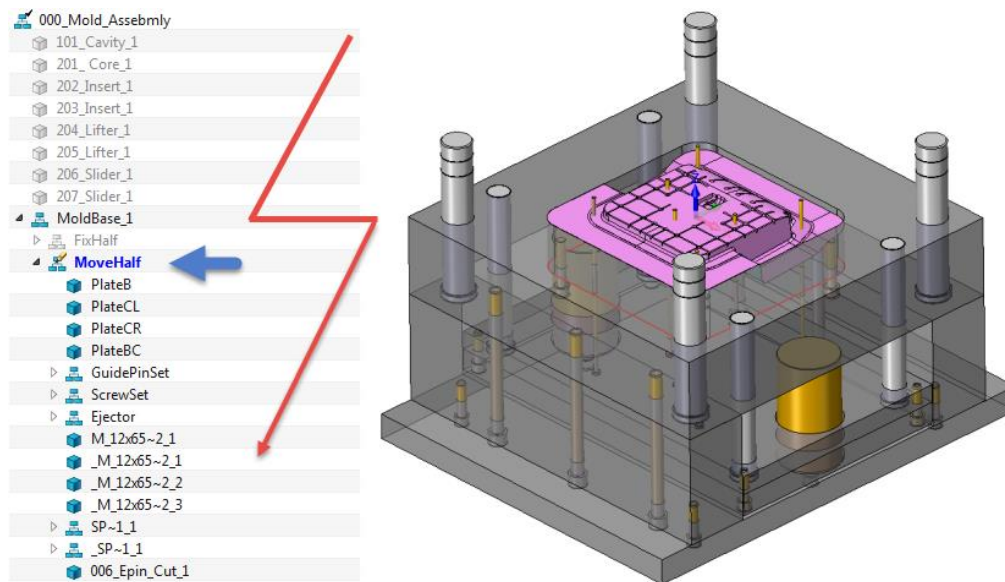


To cut the **Ejector Pins**, we will need to insert a component.

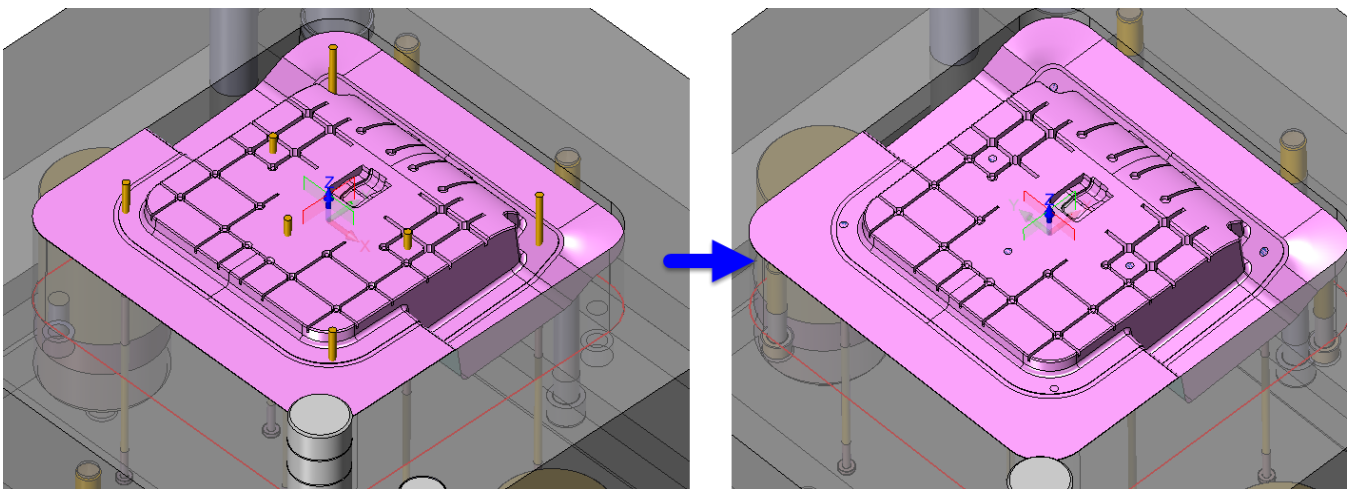
Before that, let's go to **Layer Manager** to turn off the **Core** layer, so that the ejector pins can be seen more clearly.



Activate the Part **MoveHalf**, then use the command **Insert Component** from the tool bar or mouse menu. We will insert the component named **006_Epin_Cut** and will locate it on coordinate "**0, 0, 0**" with the option "Anchor Component" active.

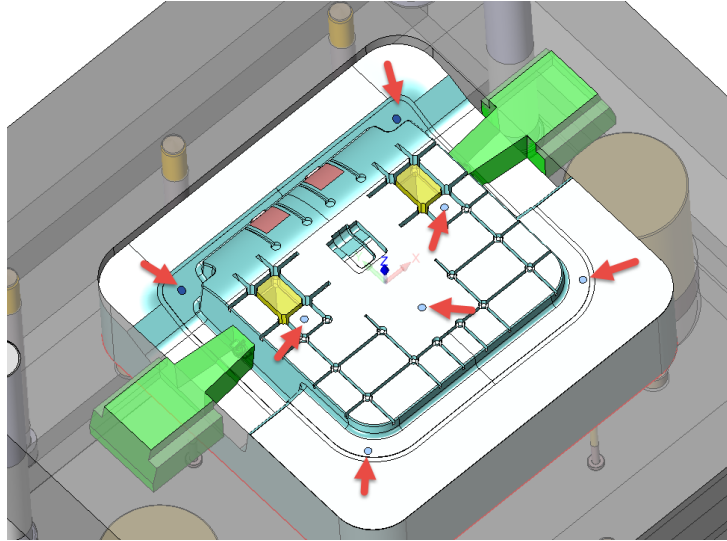


Use the command **Trim Ejector Pin**  to trim the pins by inserting the previous component. We can trim all the ejector pins in one step.



After trimming pins, we don't need "006_Epin_Cut" anymore. Here we can use the **Erase**  command to erase it.

Double click the part **000_Mold_Assembly** to activate it, before turning on the **Core** layer again. The final result should be like below.



*******Note:** So far, we have learned to add screws, and support pillars and ejector pins in the standard library. For the whole 3D mold design, however, we still need to add more libraries such as the Location Ring, Sprue Bushing and so on. We also need to finish the inner cooling, push-out structure design, which includes the slider and lifter structure. You can finish adding these things as a personal exercise.

Chapter 9 Assign Attributes for Mold Parts

Next, we will learn how to assign attributes to mold components.

Since all standard parts have the same properties, we just need to edit the non-standard parts.

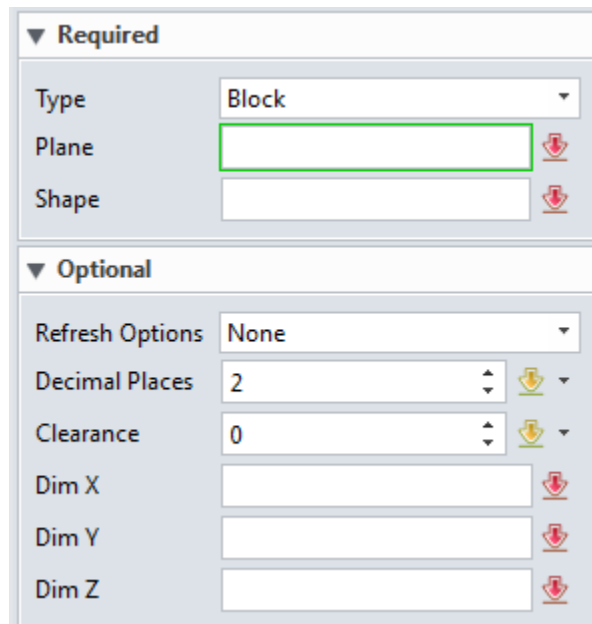
Using the **BOM**  command from the **Mold** tab, working on the **BOM list**, you can input different attributes for specified parts.

Hide list: The components will not show in the **BOM list**, you can use **Hide component** and **Display component** button to replace.

Column Edit: Edit the attributes and the marshalling sequence

Stock Edit: Calculate the stock size for selected part

For example, select the part “**101_Cavity**”, click **Stock Edit**, then the part “**101_Cavity**” has been activated and we already enter the interface of **Stock Edit**.



The screenshot shows the 'Stock Edit' dialog box with two main sections: 'Required' and 'Optional'.

Required Section:

- Type: Block (dropdown menu)
- Plane: (empty text box with a green border and a red download icon)
- Shape: (empty text box with a red download icon)

Optional Section:

- Refresh Options: None (dropdown menu)
- Decimal Places: 2 (spin box with a yellow download icon)
- Clearance: 0 (spin box with a yellow download icon)
- Dim X: (empty text box with a red download icon)
- Dim Y: (empty text box with a red download icon)
- Dim Z: (empty text box with a red download icon)

For the **Stock Edit** dialog box

Type: Use Block as the stock type

Plane: Use XY datum plane

Shape: Select the cavity shape

Refresh Options: None

You can set the **Decimal Place** and **Clearance** by yourself, which are the options to control the value of stock size. You can also input the value directly for X,Y,Z Dimension.

Here we set the **Decimal Place** to 0, and set the **Clearance** to 2.

Click **OK** to finish the **Stock Edit** command. When you check the attribute from the **BOM list** again, you can see that we have added **280x105x280** to the **Description**.

Repeat these steps to apply the information to the other non-standard objects.

Export BOM tables to Excel files

After defining all the attributes for non-standard components, we can export this BOM table to an Excel file where we can do further editing and exchange with the purchase department

Click the **Export BOM command**, ZW3D will allow you to select the directory to save the Excel file with a new name.

Open this Excel file, the **BOM** table can be shown as below.

ID	Name	Part No.	Stock size	Material	Quantity	Description
1	101_Cavity	101_Cavity	280x105x280	P20	1	
2	201_Core	201_Core	278.00x278.00x83.26	P20	1	
3	202_Insert	202_Insert	34x85x32	P20	1	
4	203_Insert	203_Insert	32x85x34	P20	1	
5	204_Lifter	204_Lifter	83x24x24	P20	1	
6	205_Lifter	205_Lifter		P20	1	
7	206_Slider	206_		P20	1	
8	A_5x250~1_1	A 5 x 250	10.0x10.0x210.9	WAS pre-hardened	7	Ejector pin
9	LRBS_120x15~1	LRBS 120-15		S45C	1	
10	M_12x65~1	M12x65		DIN 912-12.9	8	Socket head cap screw
11	Pillars~1	80-150		Aluminum	2	
12	Screws~1	Screws 12-45		Aluminum	2	

Chapter 10 Creating 2D Drafting for Mold

Next we are going to learn how to create 2D drafting for assembled mold

Standard parts automatically have the attribute information. For non-standard parts, we have to add the information manually using the methods we learnt in last chapter.

First, we need to insert a **2D Sheet** and create a standard view. There are two different ways to insert a **2D Sheet**:

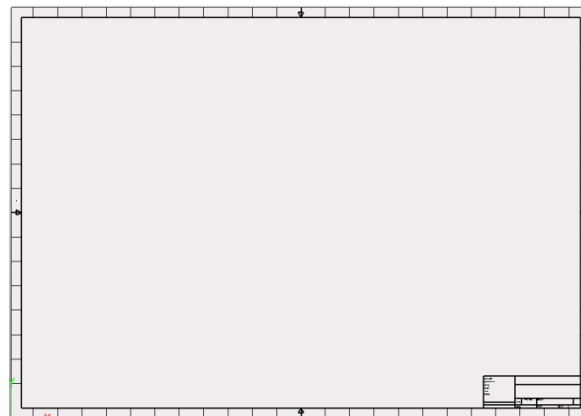
1. Activate the top assembly **000_Mold_Assembly**, **Right click** the graphic windows, and then pick **2D Sheet**.

2. **Exit** to the **Object manager** and create a **NEW - Drawing Sheet object**.



Select the drawing sheet template. Here we will use the **A0_H(ANSI)** templates.

*******NOTE:** These sheets can be configured by the user. The **Template** file that holds all of the definitions is located in the “resource” folder under the program directory. **Templates_IN.Z3** and **Templates_MM.Z3** are provided. Depending on how you answered the question when you entered ZW3D the first time, you are using either the inch or metric template file. See the Files tab of the ZW3D Configuration.



Before beginning to draw, look at the ZW3D configuration.

Click on the **Configuration**  button from Ribbon interface

Look at the **Color** tab and the **Display** tab. Don't change anything yet. Review the color settings for anything related to 2D geometry. The display tab defines global settings for line thickness.

Now click **Cancel**.

Select the **Style Manager** command  from the **Tools** tab, using this function, you can manage all attributes for entities, which includes **View**, **Line**, **text**, **Hatch**, **Dimension** and so on.

*******HINT:**Want to know more about **Style Manager** in ZW3D 2014, please look at the post below. <http://www.zwsoft.com/forum/thread-3461-1-1.html>

These configurations in **Style Manager** can be changed in the **Template_IN.Z3** and **Templates_MM.Z3** files as opposed to a single drawing sheet, and since every new project is based on the "template" the software will assume the new configurations that were previously defined.

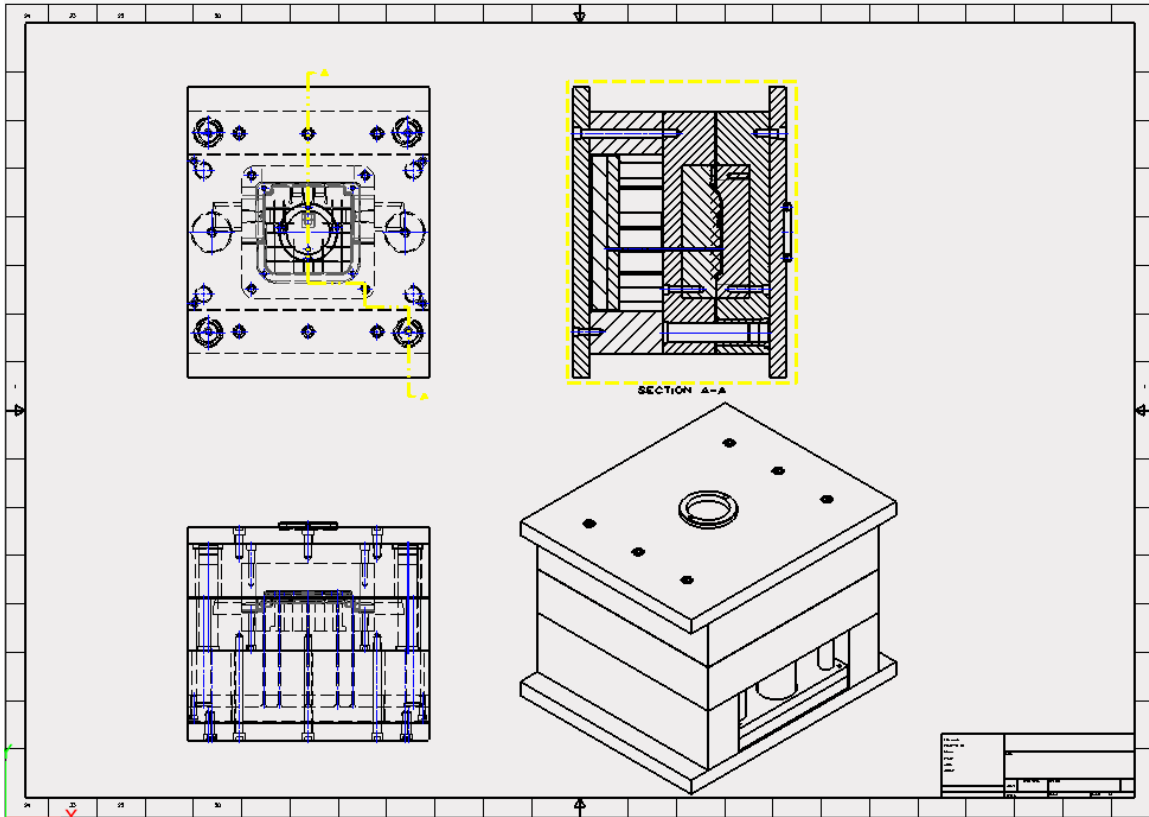
For the first exercise, select the **Standard View** command  and **Section View** command



under the **Layout** Tab to finish the main views.

*******NOTE:**Here we don't go over the detailed content. You can try this as an extra personal exercise.

The final drawing should look like this:



After creating the views, it is possible to change their position on the drawing. Just select the desired view and hold the left mouse button whilst dragging the view to your desired location.

Now that we have created the main views, we can move on to the key content that you're going to use to create a Bill of Materials (**BOM**).

*******NOTE:** Because we have already organized the BOM content and exported it to an Excel file in the last chapter, there are two ways you can generate the **BOM** table in 2D drafting.

Method 1.

Select the **Bom**  command from the **Table** tab.

View : Select the left section view from window.

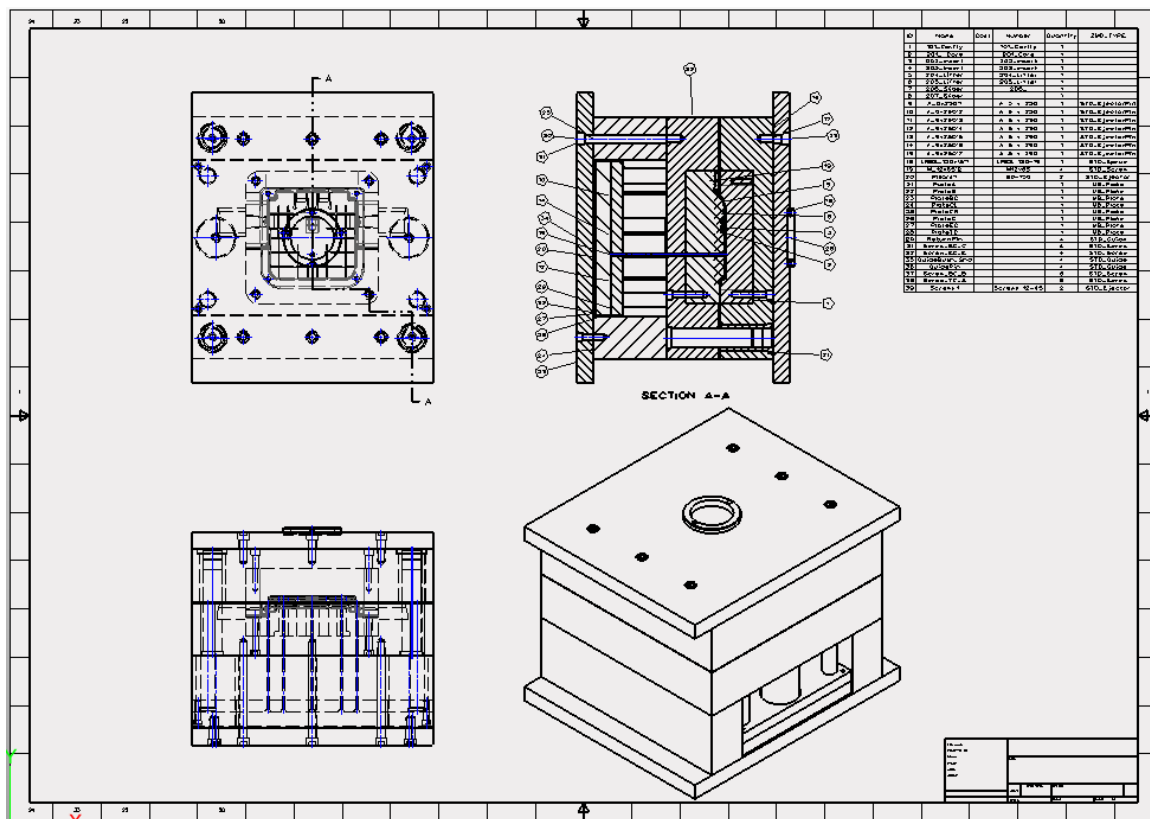
Name: Middle click to jump it.

Add **Description** to **Selected** columns.

Click **OK**, then it opens the dialog box to select the located point of the Bom table. Set the **Origin** as **Top-Right**.

Select the point of the left corner, to place the **Bom** table.

The final result should look like this:



Method 2.

Use the **Import Table** command , and change the file type to **Excel file (xls, xlsx)**. Here we can use the **Excel file** which we exported in the last chapter.

Click **OK**, The **Excel file** has been imported to 2D drafting as a table, and we can drag it anywhere.

*******NOTE:** Using this method to import the **BOM** table, it has not associated with the assembled mold, even it can't generate the **Balloon** on the view. So this is a special method that we recommend only for special situations.

Export BOM tabel

Apart from importing the excel file to become a **BOM** table, the BOM table also can be exported to an excel file when working in a 2D sheet enviroment.

Right click the **BOM** table

Select **Export Table Command**  from pull down menu or find the command from **Layout-**

>Edit Table->  directly

Define the **Save path** and the **format** as you wish.

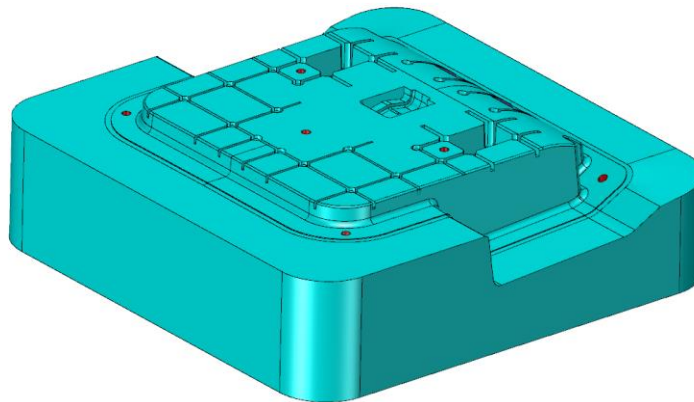
Chapter 11 Creating Electrodes

Next we will learn how to create electrodes.

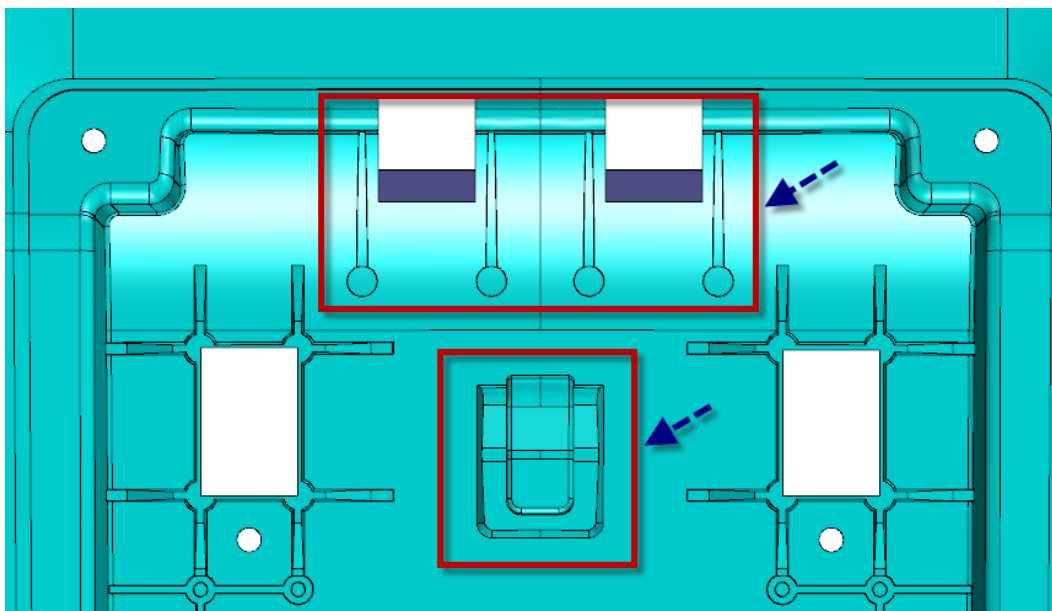
Create a new file called “**201_Core_E**”. This part will be used to create the sub electrode and will be associated with the core part.

Use the **Insert Part Geometry** command . Import the “**201_Core**” by selecting it from the object list.

Select the option, “**No Backup**” and locate it at the coordinates “**0,0,0**”.



We will create the electrode for the areas shown below.

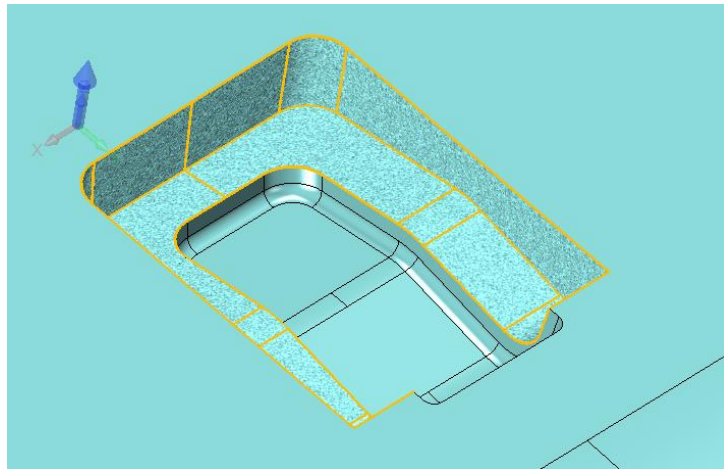


To create your first **Electrode**, go to **Mold> Tools > Electrode** 

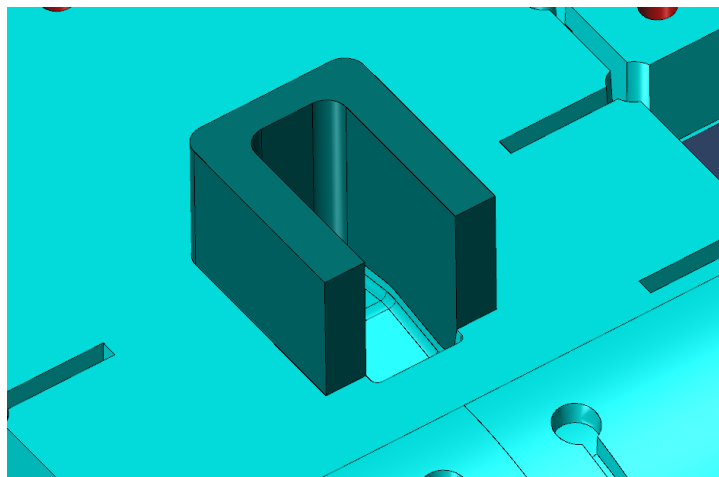
In the **Electrode** command dialog box, we need to define the main options.

Right click the graphic window and pick the **Z** axis as **Direction**.

Select the **faces** as shown below. You can hold down **Shift** to easily select all of them at the same time. (**15** faces)



We can keep the other options at their default values. We do not need to change any of them. Click **OK** to get the result as shown below.

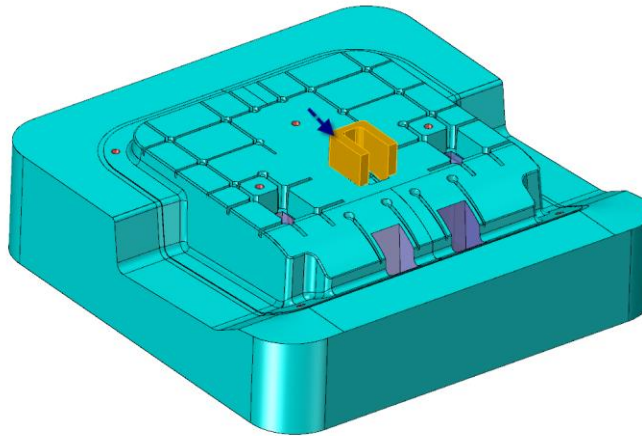


*******HINT:**While creating an electrode, you can right click on the **Face color**  command from the **Document aware toolbar** and change the Face attribute of the electrode shape.

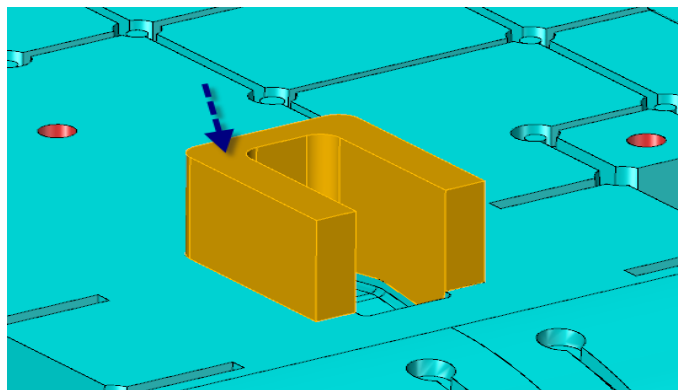
Select the **Electrode Tool Blank Command** 

Shape: Use **Rectangular** as the default value.

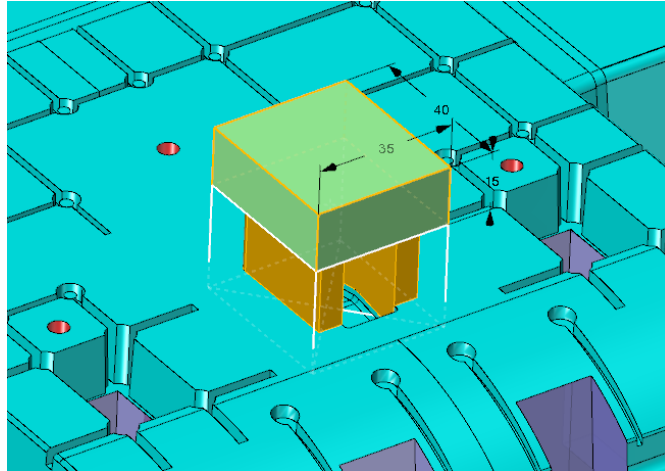
Electrodes: Select the shape we just created (Sometimes we need to set the **Filter** to **Shape** so that we can do this)



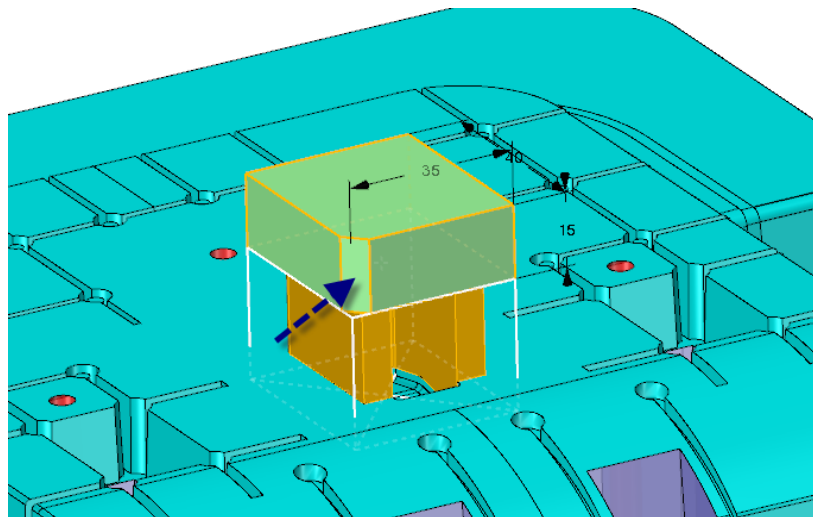
Datum: Pick the top face of the electrode shape



Set the **Length** to 40, **Width** to 35, and **Thickness** to 15.

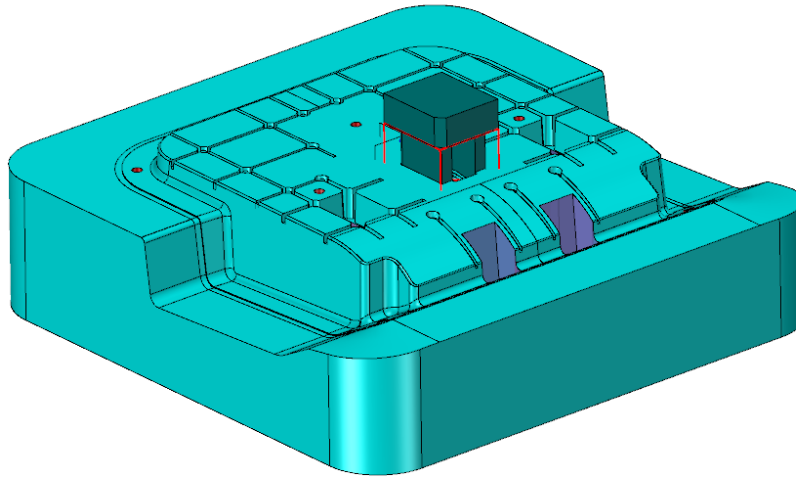


Go to the **Mark** option to select **Back-Top** as the position.



For other options, you can go to the help document, or try to change the effects as personal exercises.

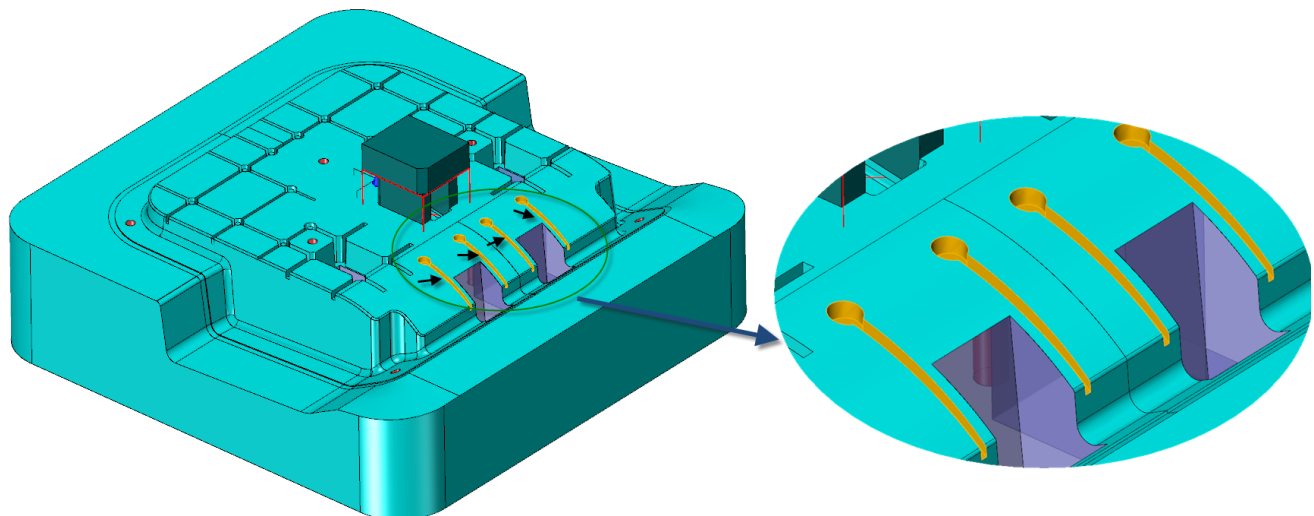
Click **OK** to finish this command, the final result should look like this:



Next we will create the second electrode, using the same method as the first one, but this electrode will be created with multi area faces.

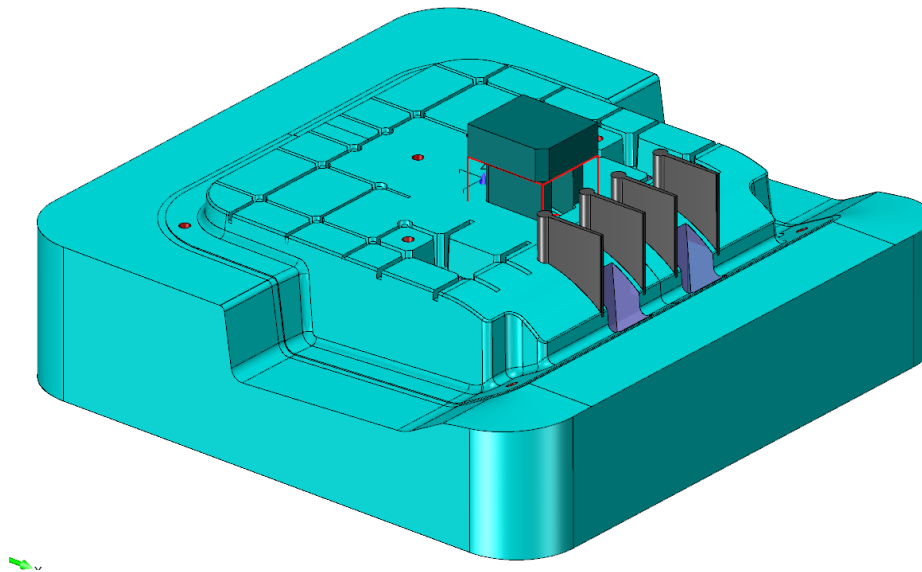
Use the **Electrode Command** 

For the **Face** option, select the faces as shown in the below image. There should be 20 faces to select in total.



For another option, you can keep the same as the previous electrode.

The final result should look like below.

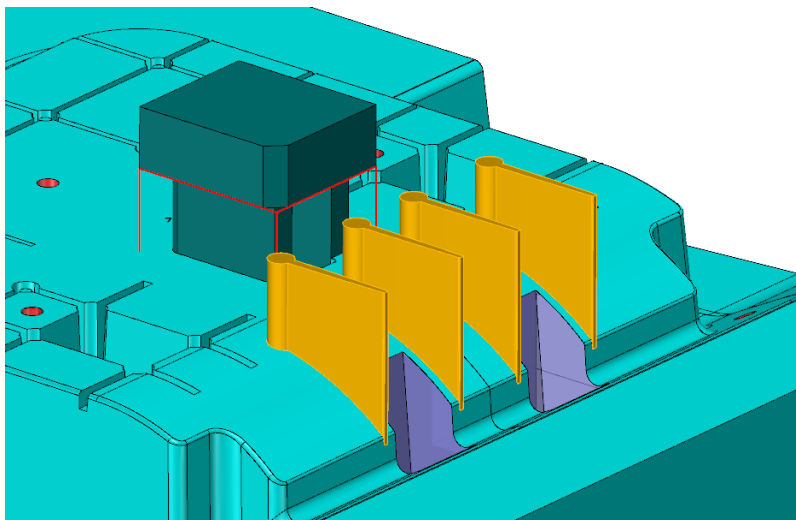


Select the **Electrode Tool Blank Command**

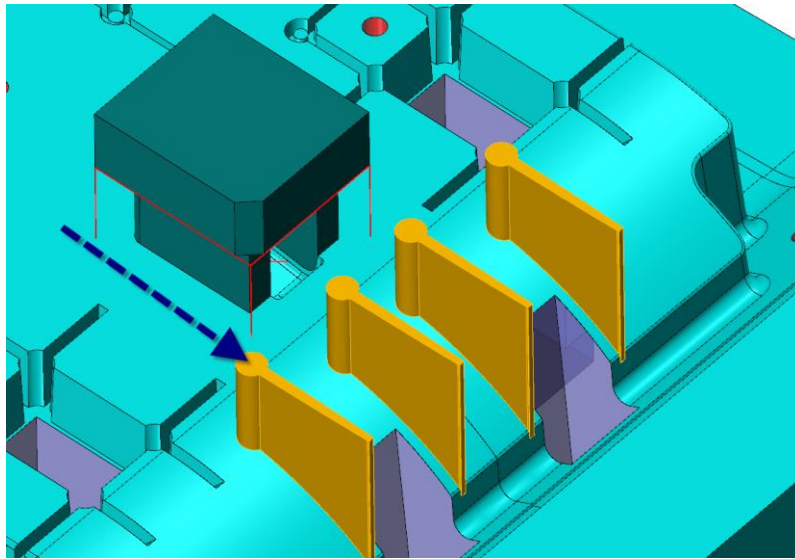


Shape: Use **Rectangular** as the default value.

Electrodes: Select the shapes we just created (Sometimes we need to set the **Filter** to **Shape** so that we can do this)

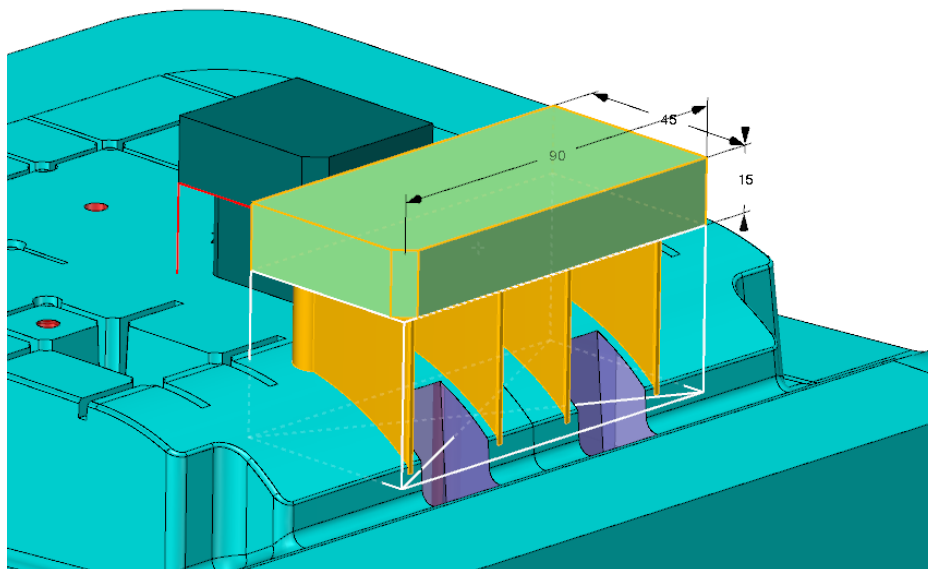


Datum: Pick the top face of any electrode shape

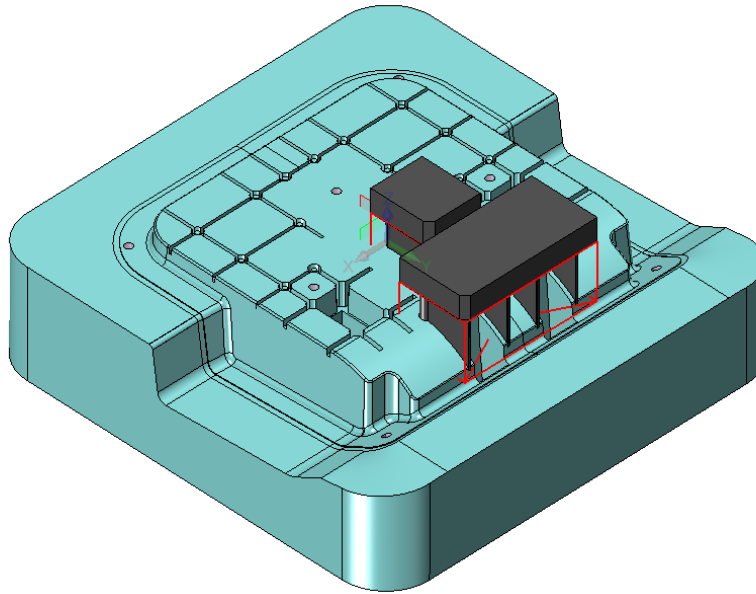


Set the **Length** to **45**, **Width** to **90**, **Thickness** to **15**.

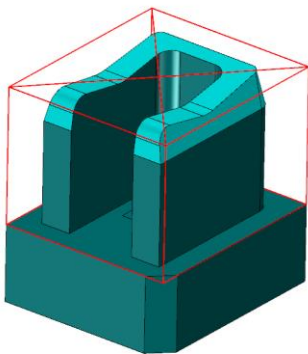
Go to the **Mark** option to select **Back-Top** as the position.



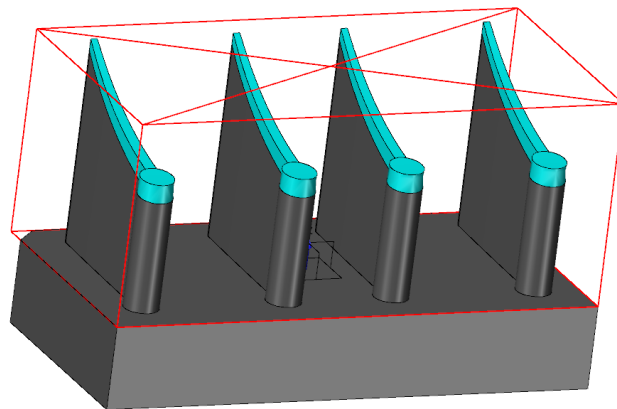
Choose **OK** to finish the command, and the final result should look like this:



If you exit to the **Object Manager**, you can see that there are 2 new objects named **201_Core_E_Blank_Electrode.1**, and **201_Core_E_Blank_Electrode.2** .



201_Core_E_Blank_Electrod



201_Core_E_Blank_Electrod

Save the files

Chapter 12 Creating an Electrode Table

Next we are going to learn how to create an electrode table.

Open the part **201_Core_E**

Then, **Right click** the Graphic windows and choose the **2D Sheet**

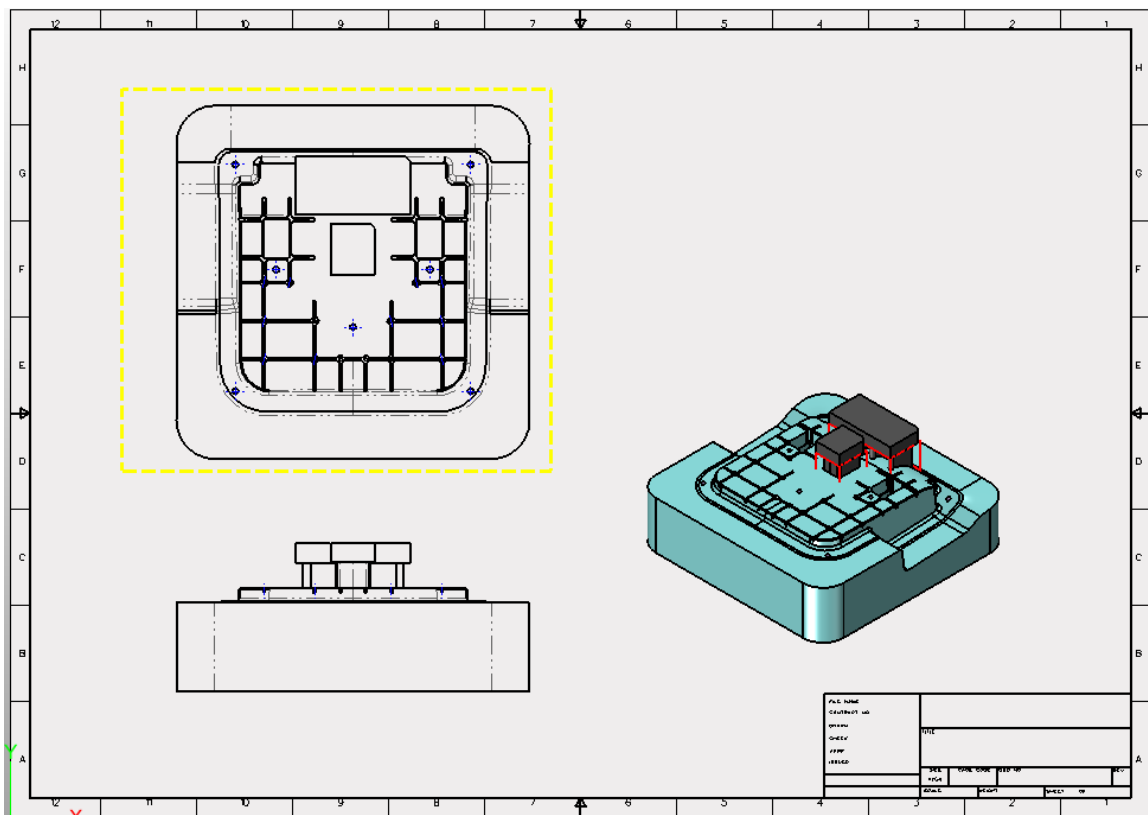


Select **A2 _H (ANSI)** as the template.

After inserting the **2D Sheet** environment, it will create the view as shown as below.

*******NOTE:** Here we don't go over the detailed content. You can try this as an extra personal exercise.

The final drawing should look like this:



Now that we have created the main views, we can move on to the key content that you're going to use to create an electrode table.

Select the **Electrode BOM** command  from the **Table** tab.

View: Select the left section view from the window.

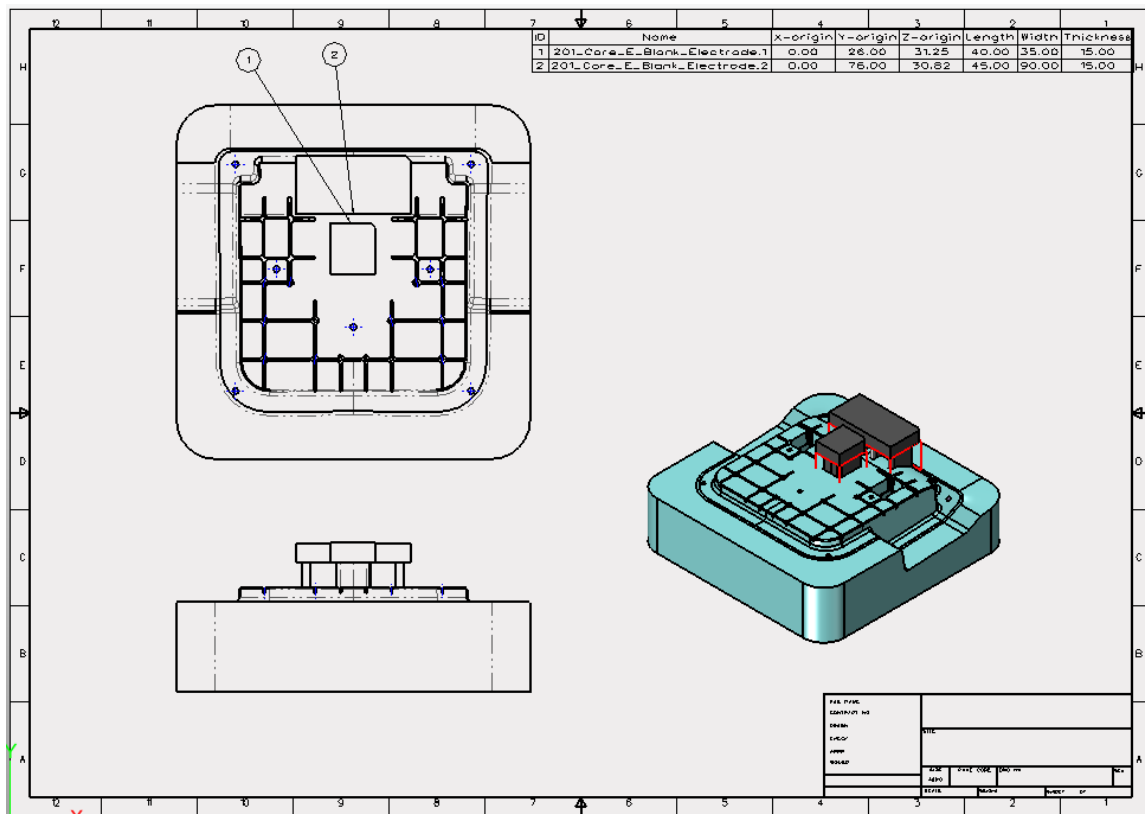
Name: Middle click to skip it.

Add the **Length, Width, and Thickness** to the **Selected** columns.

Click **OK**, then use the dialogue box to select the location point on the Bom table. Set the **Origin** as **Top-Right**.

Select the point on the left corner to place the **BOM** table.

The final result should look like this:



For the Bom or Electrode table, we can **Import/Export** them to **Excel** or **Txt** files. Next we will use the Electrode table as a learning example.

ID	Name	X-origin	Y-origin	Z-origin	Length	Width	Thickness
1	201_Core_E_Blank_Electrode.1	0.00	26.00	31.25	40.00	35.00	15.00
2	201_Core_E_Blank_Electrode.2	0.00	76.00	30.82	45.00	90.00	15.00

Right click the Electrode table

Select **Export Table Command**  from pull down menu

Define the **Save path** and the **format** as you wish.

EDM Table0.TXT

ID	Name	X-origin	Y-origin	Z-origin	Length	Width	Thickness
1	201_Core_E_Blank_Electrode.1	0.00	26.00	31.25	40.00	35.00	15.00
2	201_Core_E_Blank_Electrode.2	0.00	76.00	30.82	45.00	90.00	15.00

EDM Table0.XLS

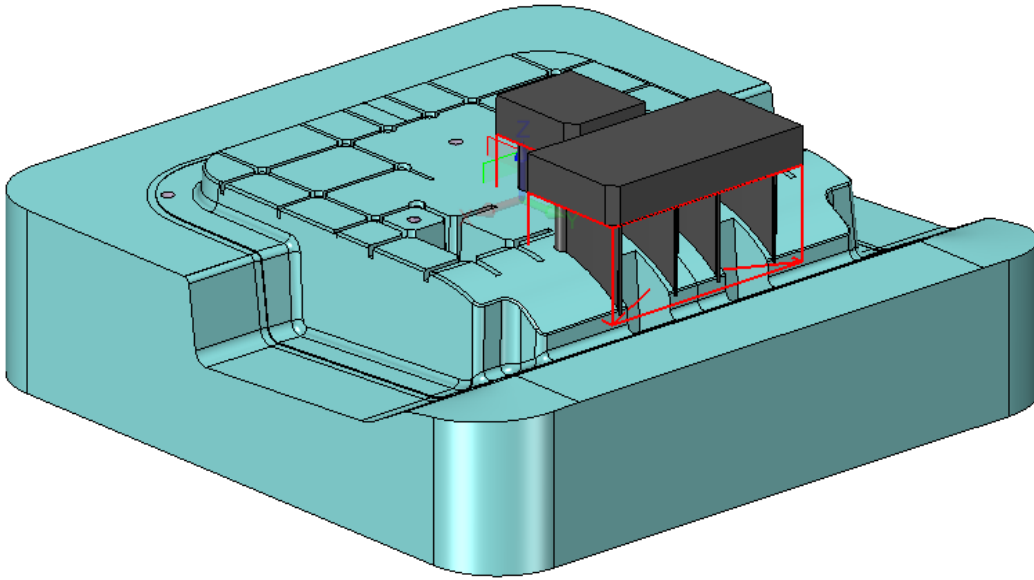
ID	Name	X-origin	Y-origin	Z-origin	Length	Width	Thickness
1	201_Core_E_Blank_Electrode.1	0	26	31.25	40	35	15
2	201_Core_E_Blank_Electrode.2	0	76	30.82	45	90	15

[illegible]

There is another way to create the 2d drafting and Electrode table automatically using

Electrode Sparking Drawcommand

Open the part **201_Core_E**



Click the **Electrode Sparking Draw** command from **Mold->Tools** tab

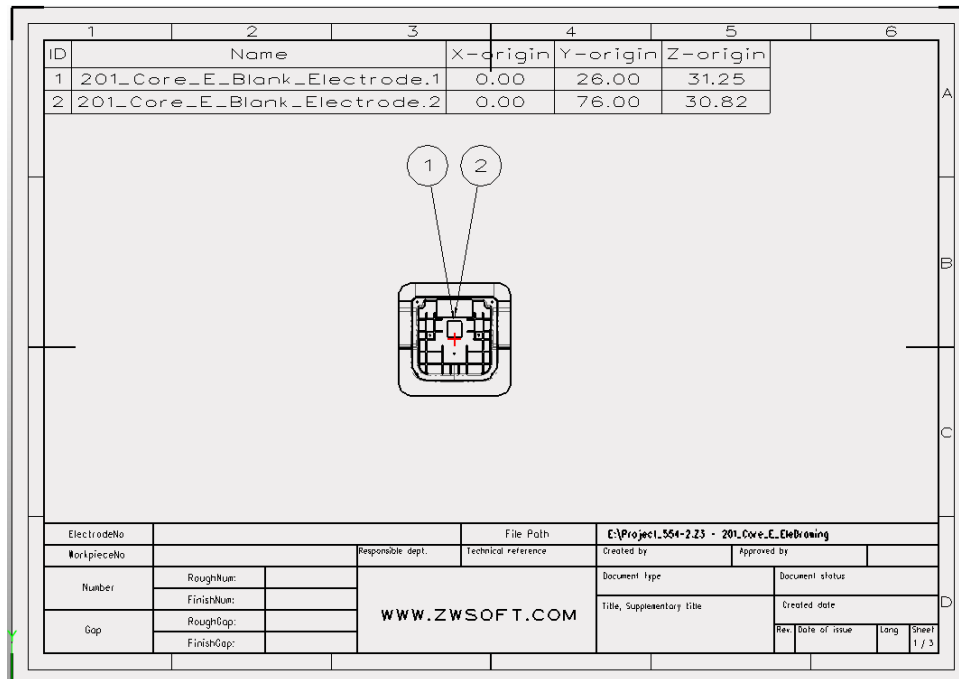
Workpiece: Select the Core shape as workpiece

Electrode : You can select the electrode to generate the 2D drafting, here we select both

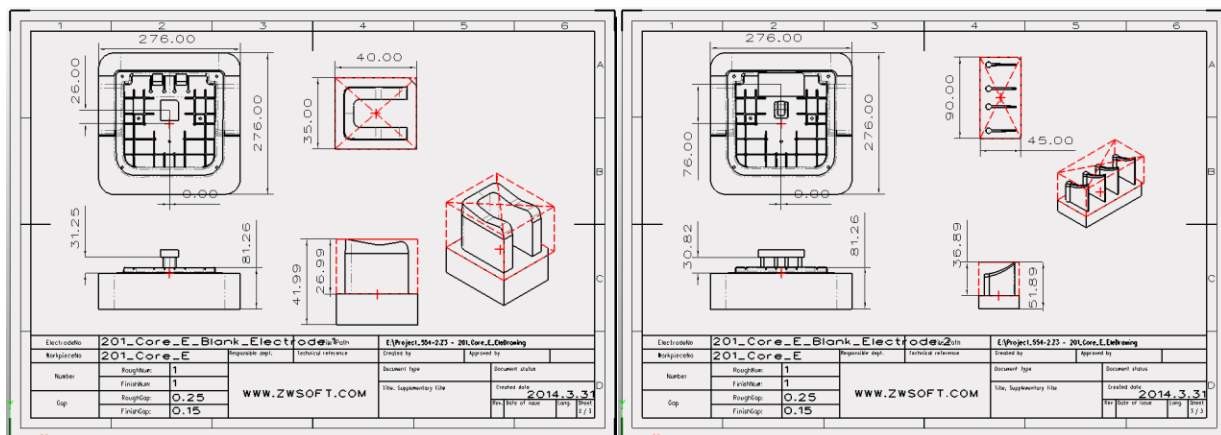
Set Electrode Attributes  : Using this button can call the Attributes list of the electrode to set more properties

Click **OK**, ZW3D will create the 2D Sheet called **201_Core_E_EleDrawing**, which includes 3 sheets

The first sheet generates the drafting for Electrode assembly with a table.



The second and the third sheets generate the drafting for the electrode part, and it has created some auto dimensions.



*******NOTE:**

1. After generating the 2d drafting automatically, we should adjust it manually to make it meet the output request.
2. If you want to customize the Eletrode drafting template, please go to the installed directory "ZWSOFT\ZW3D 2014 Eng\ZWMold\Model\EleTemplate.Z3"

3. If you want to customize the electrode view position and dimension style, please go to the installed directory “**ZWSOFT\ZW3D 2014
Eng\ZWMold\Resource\Templates_ZMD_ELEPositionConfig.cfg**”

Appendix-Introduction of Project Manager in Mold Design

Concept Explanation

Non-Project Manager - All the content that we learnt from Chapter 1 to 12 involved using the **Non-Project Manager** - The parting process was created as a single object, with the need to extract shapes to specific parts manually.

When undertaking the parting process, we need to create the objection manually and use the command **Import Part Geometry**  to copy the geometry from the external part to current part.

Project Manager: The parting process was created in the initial project automatically. There is no need to create each object manually

Next, we will learn a little bit about this mold design method in ZW3D.

Let's create  a **Multi-Object** file  called "**Product model**".

*******NOTE:** This file will not be used to create the mold project but just to save the product model.

Create  a **Part**  object called "**001_Model_Part**"

Using the same method as we did in Chapter 1

Import the iges "**Backcover_554_Rev_04.igs**", which is located in the "**\ZWSOFT\ZW3D 2014Eng\training**" installation directory.

Heal the model after importing using the same method as we did in Chapter 1.

NOTE:There is nothing different when healing the imported model.We will not introduce it again.You can do it by following the content in Chapter 1 or you can use the healed object to undertake the next step.

Save  the file

Next we will create the Mold project.

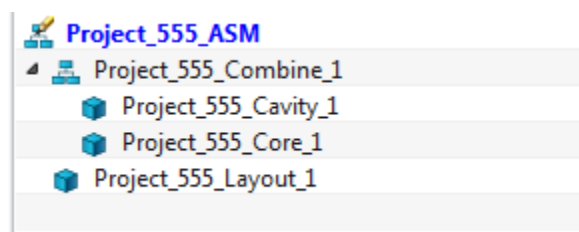
Use the **Project** command  from **Mold->Product** tab

Cavity Type: **Single-Cavity**

Project name: Project_555

Shrink: 1.005

Click **OK**, we can see the assembly structure has been created as shown below.



Next, we are going to insert the Product model

Use **Layout** command 

File: Click the **Open** button , find the file **Product model.z3** that we just created

Part:Select the object **001_Model_Part** from the list

Switch:  This command is used to switch the product model when you are doing family mold design that has more than one product model in the mold project. We are doing single cavity mold design with one product model, so it's not necessary to use this command.

Rename:  This command is used to rename all the part names in the Mold project

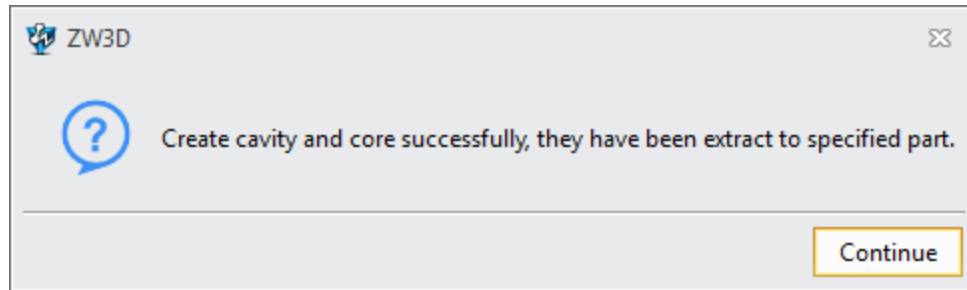
*******CAUTION:** Working on the Mold project environment, remember to not rename any part name in **Object manager** or through Right clicking the component on Assembly manager and renaming. Please use the Rename command to rename any part when using **Project manager** for mold design. Otherwise, it will destroy the associated relationship.

Combine:  This command only works when doing Multi cavity design, which is used to combine the multi cavity layout parts and place them in a single “combined” component. Doing multi cavity or family mold design, this command needs to be carried out before going to create a workpiece  or undertake any trimming. 

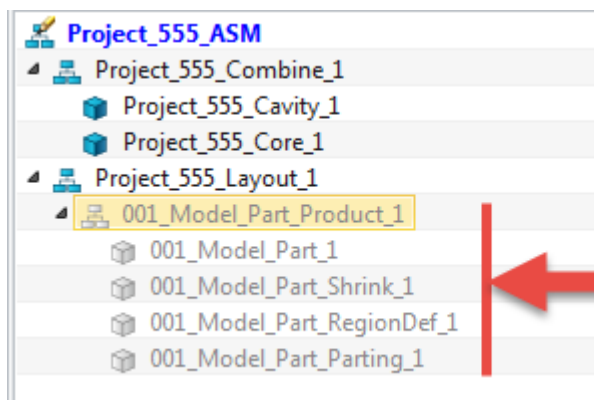
Next we can follow the operations in **Chapter 3** to do parting for this product model. The steps are the same.

*******NOTE:** We will not explain the detailed Parting operations again, you can follow **Chapter 3** to do it as a personal exercise.

Trim:  When we use this command in the Project manager environment, the only thing different to **Non-Project** mold design is that the Core and Cavity shapes will be extracted to a specified part. When you finish this operation, you will see the hint as shown in the image below.

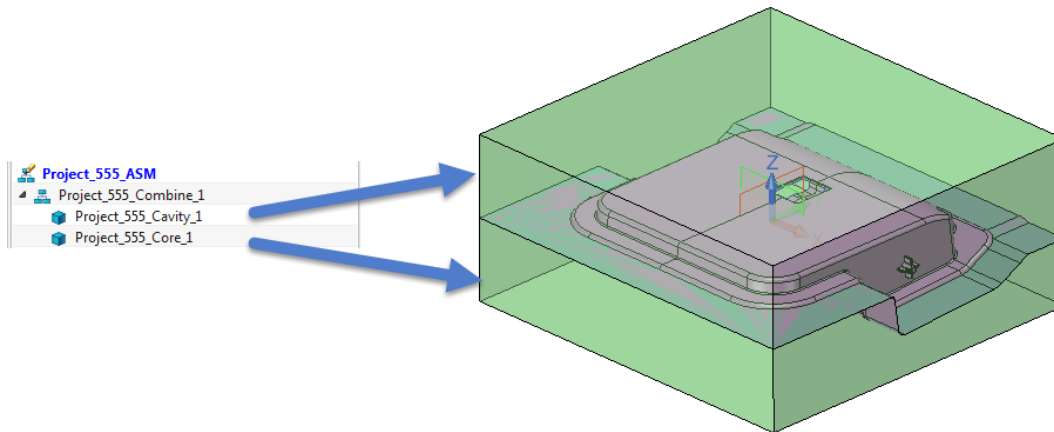


If you were doing the operation correctly, check the Assembly manager. The tree structure will be shown as below.



All the parts used for parting operations have been hidden, meaning they has finished their job and there is no need for them to be shown on the graphic window.

If you are checking the model from the graphic window, as we have learnt above, the cavity and core shape has been extracted to a specific part as shown below.



After that, we can continue to create **Inserts**, Lifters and Sliders, insert the **Moldbase** and **Standard component**.

This is a brief introduction to **Project Manager** in single cavity mold design.

Epilogue

Thank you for your valuable time. For this mold training guide, we used **Non-project** to complete the process from importing products to exporting BOM. What's more, we simply introduced the **Project manager** design method and some of its functions. Regarding other mould structure designs in detail, you can do it as personal practice. Please pay attention to our official website and forum to get the latest information.