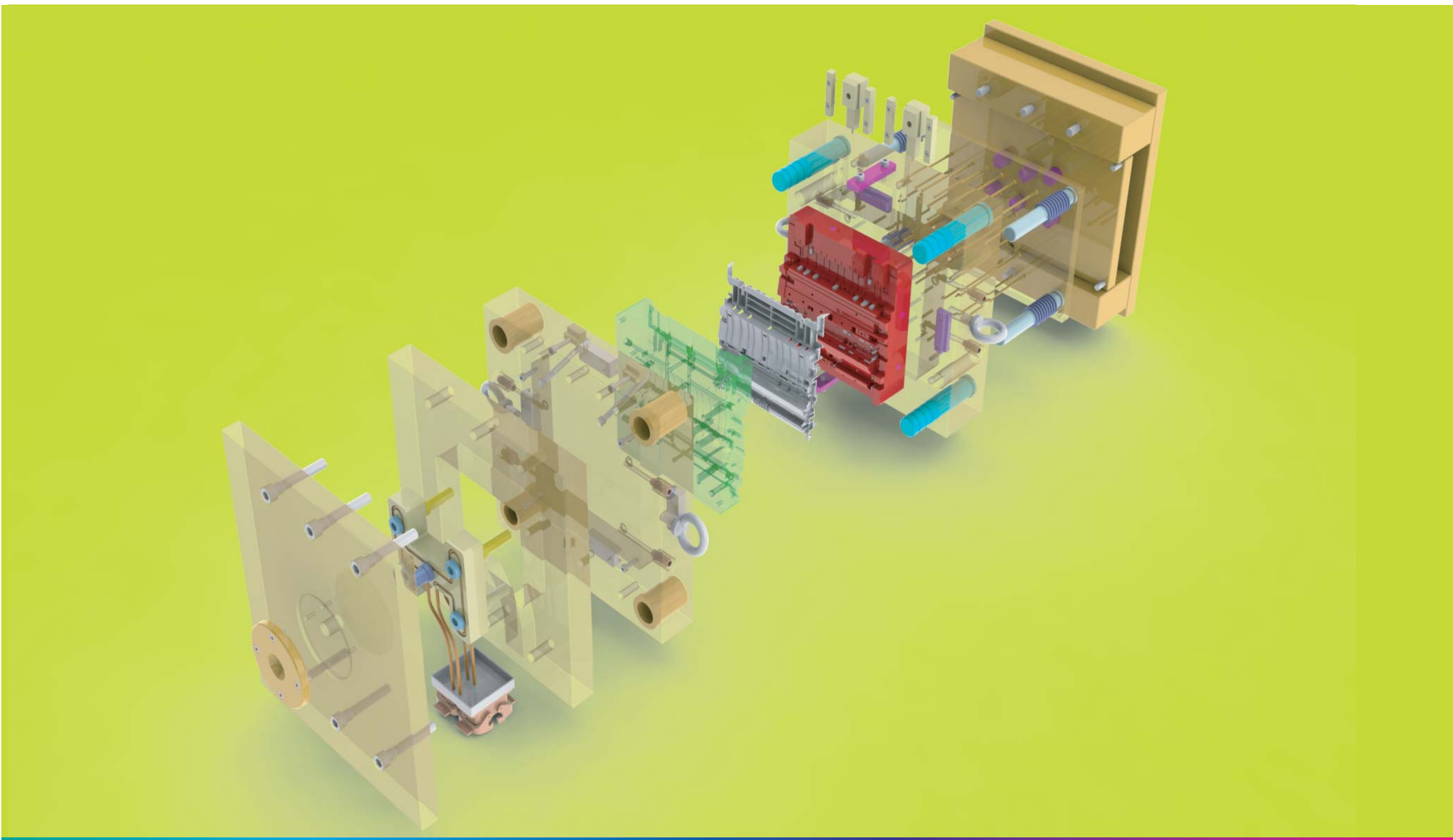
3DQuickMold Training Manual



www.3dquicktools.com

Online full color manual is located at:

http://3dquicktools.com/white_paper/3DQMTrainingManual.pdf

Contents

Preparation

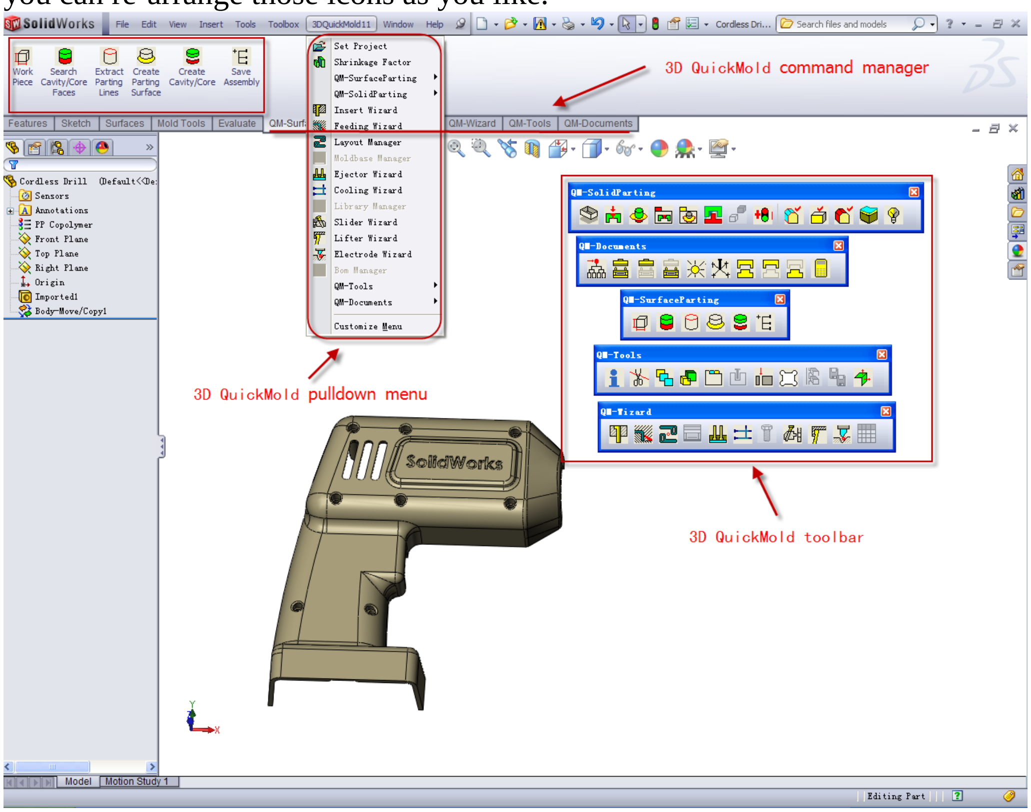
3DQuickMold is a professional plastic mold design software seamlessly integrated with **Solidworks**. This manual is based on the Solidworks 2011.

Please do the following after the installation:

1. Click Tools -> Options->Add-Ins, make sure 3DQuickMold2011 is checked
2. 3DQuickMold advises the user to start project with new SolidWorks part. To do this, you can open a new part, insert the original plastic model file. This can retain the completeness and independency of data of the original part. Adjust a suitable origin and position. The default **mould open direction** of 3DQuickMold is the Z-axis.

Software Interface

The new user interface looks like the following picture shown. Similar to other SolidWorks UIs, you can re-arrange those icons as you like.



1. 3DQuickMold pull-down menu

Click 3DQuickMold11 in the menu bar, pull down menu pops out as follows:

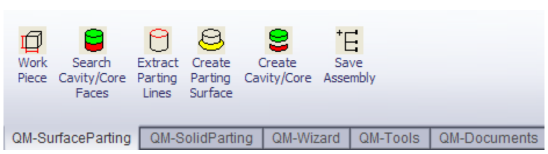
Each mold design module is listed below:

- Set Project： Set the current working models
- Shrinkage Factor： Scale the part to allow for shrinkage
- QM-SurfaceParting： Surface-based parting
- QM-SolidParting： Solid-based parting
- Insert Wizard： Sub-insert design
- Feeding Wizard： Runner and Gate design
- Layout Manager： Arrange the core/cavity layout
- Moldbase Manager： Load and edit mold base
- Ejector Wizard： Design ejectors
- Cooling Wizard： Cooling channel
- Library Manager： Standard libraries for mold design
- Slider Wizard： Slide design
- Lifter Wizard： Lifter design
- Electrode Wizard： Electrode design
- Bom Manager： Bill of materials

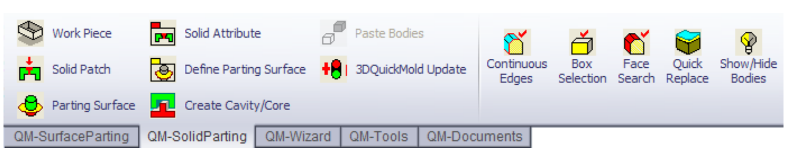
- QM-Tools： Effective tools for mold design
- QM-Documents： Activate working models

2. 3DQuickMold Command Managers

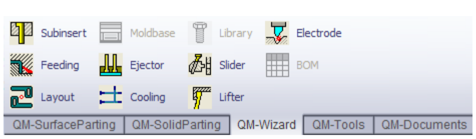
The corresponding command groups of the above modules are listed below:



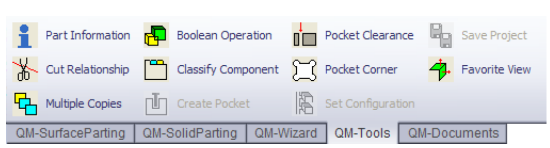
Surface parting group



Solid parting group



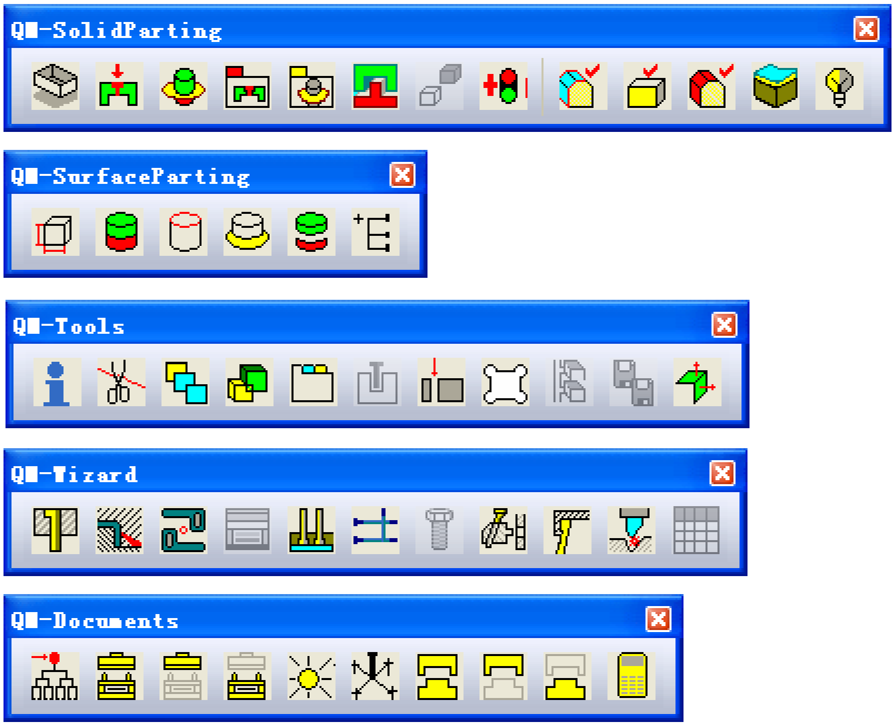
Mold design module group



Tools group for mold design

3. 3DQuickMold Toolbars

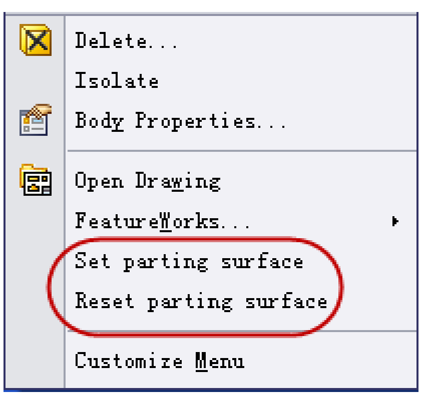
Toolbars are available as well, they can be shown up or hidden as needed.



They are QM-SolidParting, QM-SurfaceParting, QM-Tools, QM-Wiards and QM-Documents respectively.

4. 3DQuickMold Pop-up Menu

If face is selected, right-click the mouse button, a pop up menu is displayed as following.



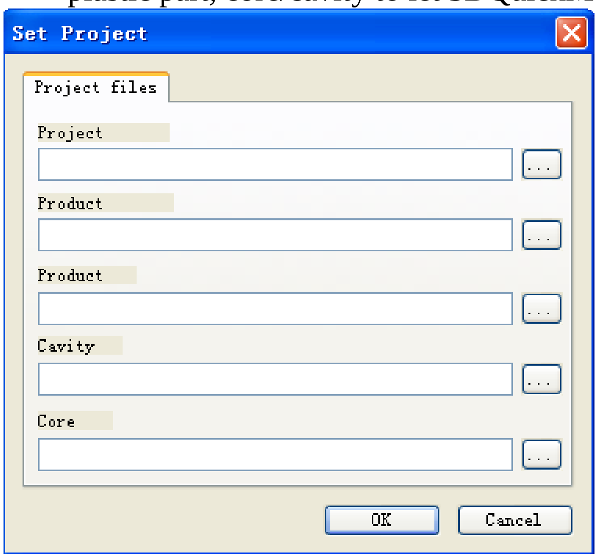
It is used to set or reset the parting surface.

Chapter 1. Set Project

This is not a mandatory step for 3DQuickMold now. Without setting the working project, you still can use most functions provided in the system, it is mainly used in some particular cases.

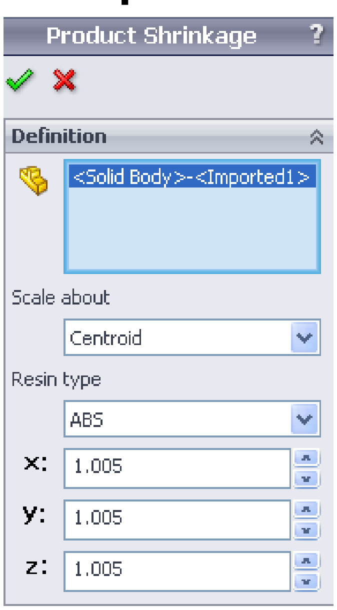
For the following purposes, you may set your project using the function：

- If the core/cavity was done by other CAD system and imported into SolidWorks for downstream design or even done by SolidWorks alone, you can use this UI to set your plastic part, core/cavity to let 3DQuickMold to recognize them.



- For mutilple mold design projects at the same SolidWorks envioroment, you need set your current working models such as plastic part, core, cavity, product assembly, Project assembly to tell the system which project you are working on.

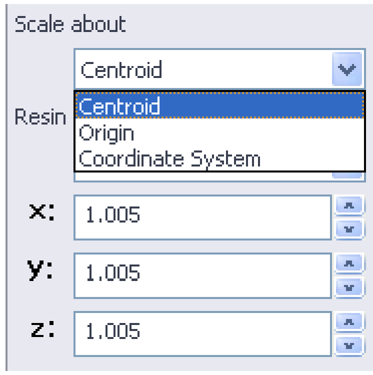
Chapter 2. Shrinkage Factor



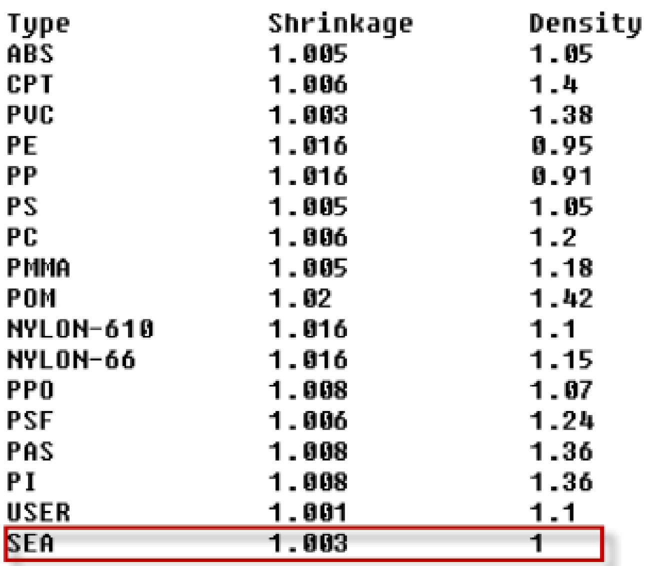
Every plastic material has a shrinkage factor assigned to it, the part has to be enlarged proportionally to compensate the contraction。 Click on 

, the following page pops out.

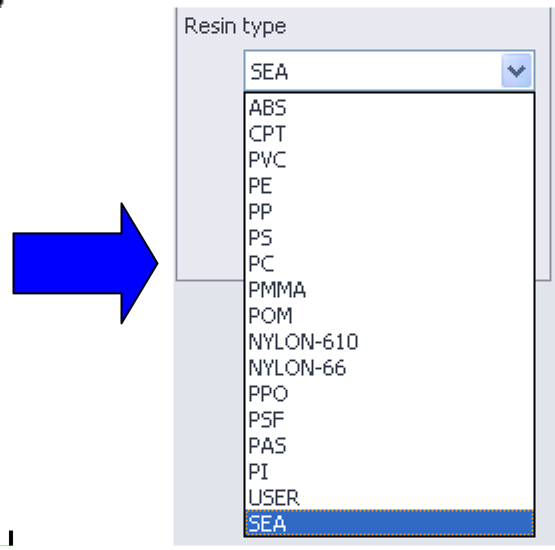
Different scale factor can be set to the product.
The product can be scaled along x, y, z direction.
The product can be scaled about the Centroid, the Origin, or a coordinate system selected.



Different plastic material is available under Resin type. The material database is located at the installation folder\ res \ shrinkage factor.txt and it can be edited. Below shows the adding of a new material in the shrinkage factor.txt that is highlighted in red.



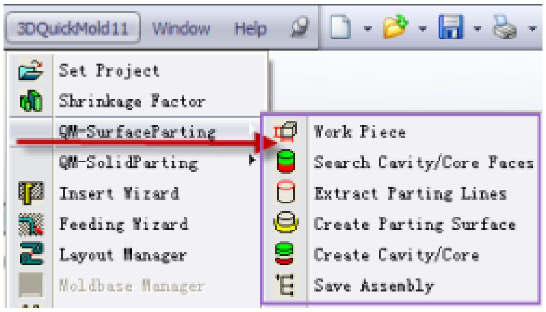
Type	Shrinkage	Density
ABS	1.005	1.05
CPT	1.006	1.4
PVC	1.003	1.38
PE	1.016	0.95
PP	1.016	0.91
PS	1.005	1.05
PC	1.006	1.2
PMMA	1.005	1.18
POM	1.02	1.42
NYLON-610	1.016	1.1
NYLON-66	1.016	1.15
PPO	1.008	1.07
PSF	1.006	1.24
PAS	1.008	1.36
PI	1.008	1.36
USER	1.001	1.1
SEA	1.003	1

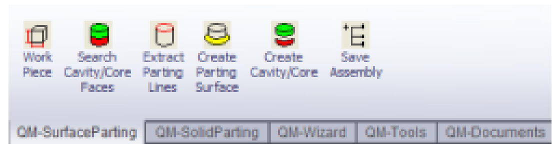


On the “Product Shrinkage” dialog, under resin type, the newly added resin can be found.

Chapter 3.QM-SurfaceParting

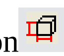
This surface-based parting module basically aims to simplify a lot of native SolidWorks parting process such as parting-line and shut-off creations.

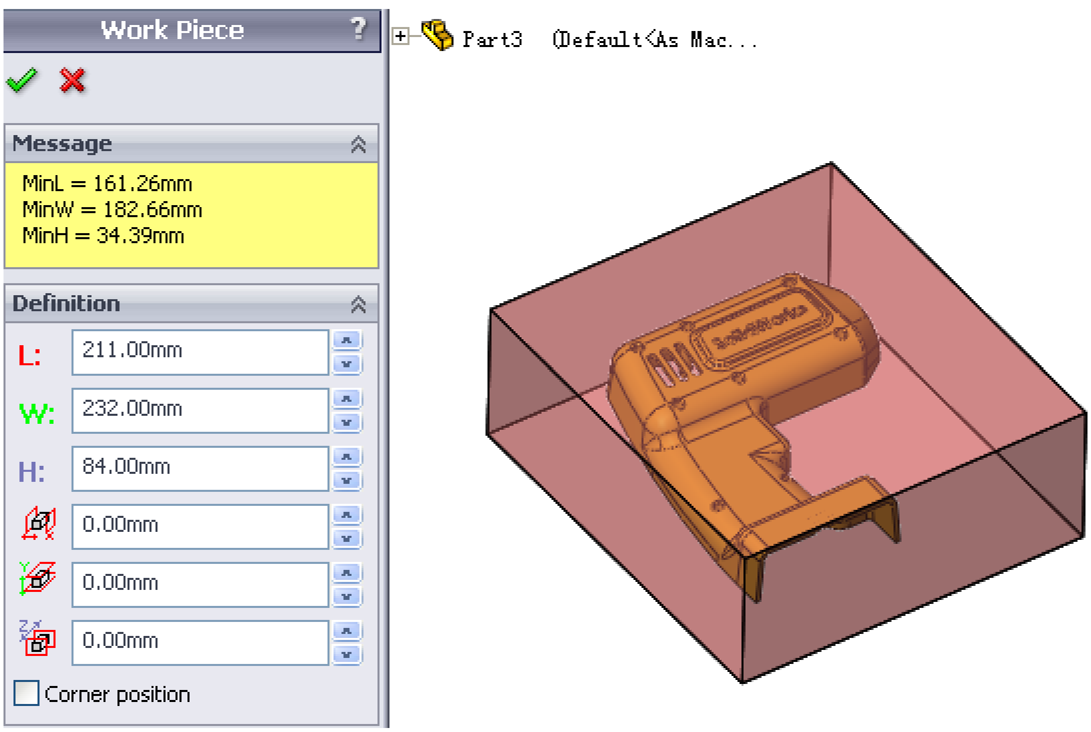




3.1 Workpiece

Workpiece is used to determine the core/cavity size, it is advised to be done before parting surface。

Click on  Work Piece, 3DQuickMold will show the minimum size of the current part, the default Length, Width and Height are set automatically, the preview as follows.



At Definition group, size of work piece and its relative position could be edited.

L: : Length of work piece

W: : Width of work piece

H: : Height of work piece

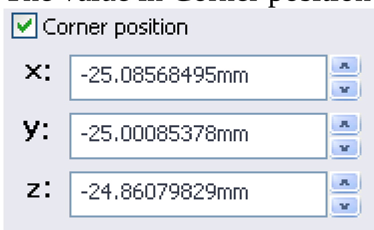
: Offset value along X direction

: Offset value along Y direction

: Offset value along Z direction

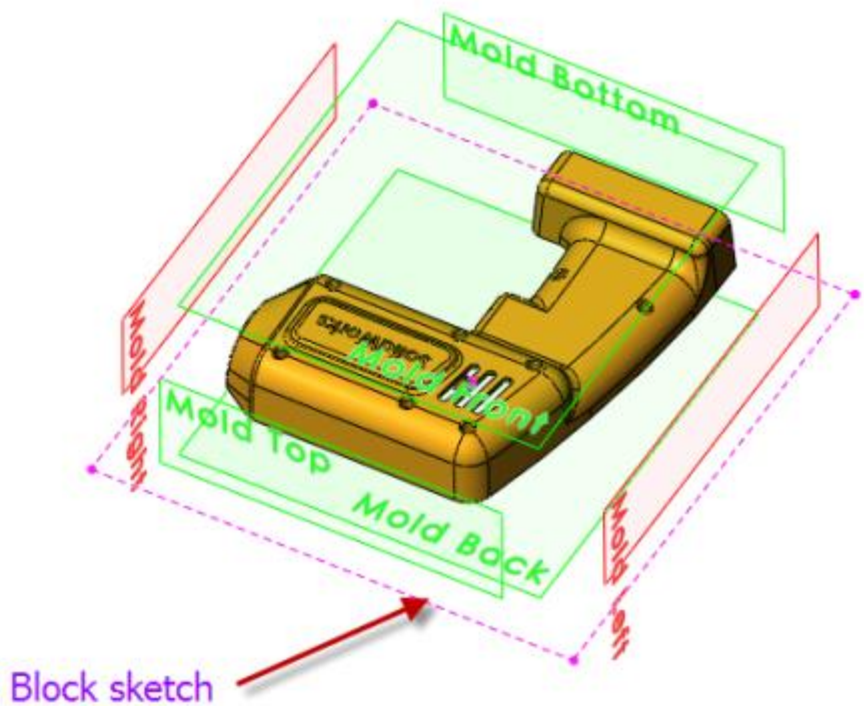
If Corner position option is checked, the position of the workpiece relative to the coordinate system is shown. The workpiece can be offset by changing the value of x, y, z.

The value in Corner position is the minimum value of the workpiece in the x, y, z direction.



After setting is completed, click OK.

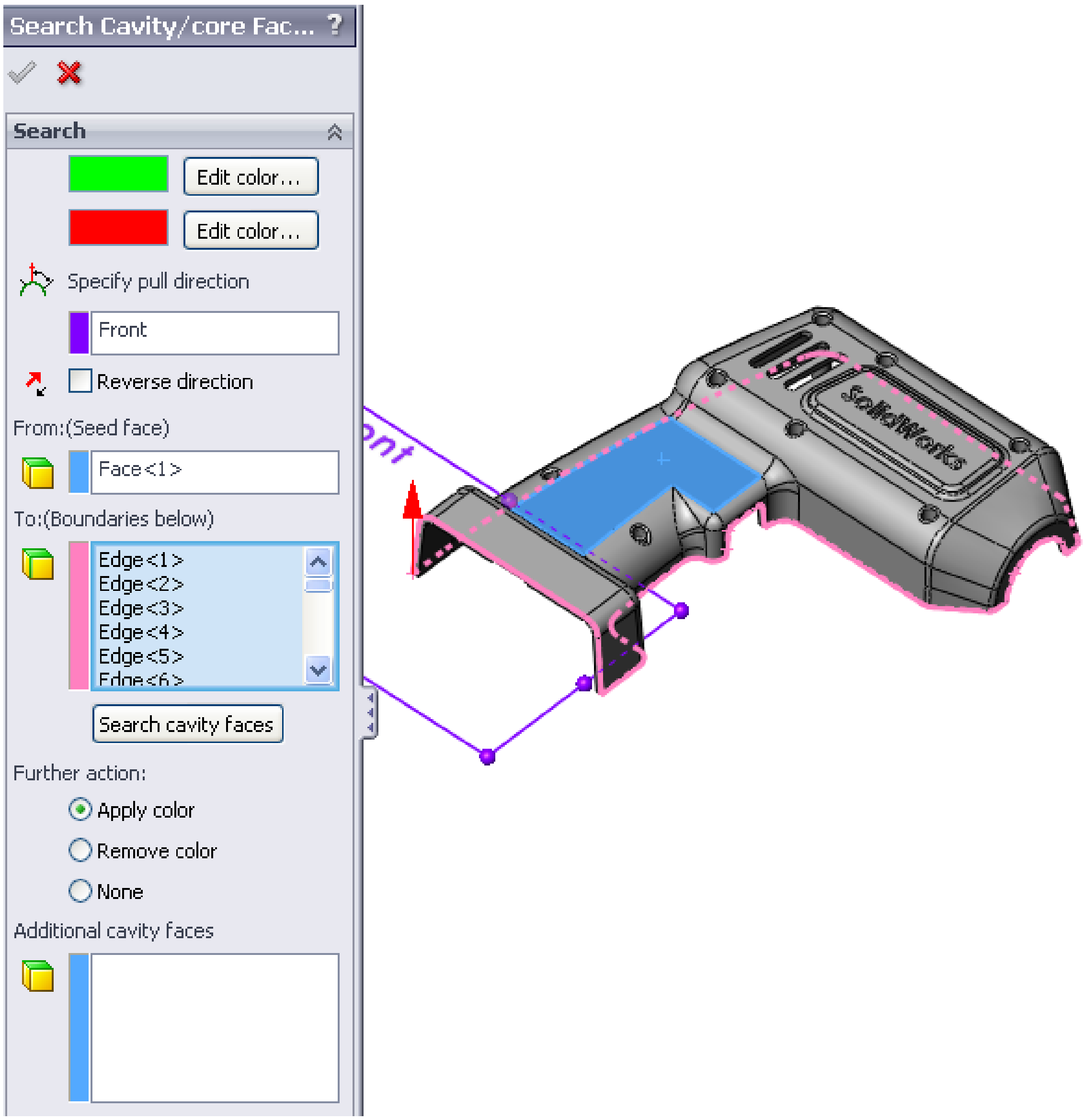
6 reference planes with the workpiece as border are created.（the first five reference planes are hidden，Mold Front is shown; the distance between Mold Front and sketch is the height of the workpiece, the size of the whole work piece is visualized.



A sketch of Block Sketch with Mold Back as reference is also created，This block sketch determine the size of the final size of the core and cavity, the dimension of the sketch is the dimension of the projected image of the workpiece on the xy plane.

3.2 Search Cavity/Core faces

This is a tool for searching faces, by selecting boundary condition and a seed face to propagate the search of faces on the part surface.
This function is similar to the SolidWorks **Draft analysis**, but additional boundary condition is used to control the researching result.



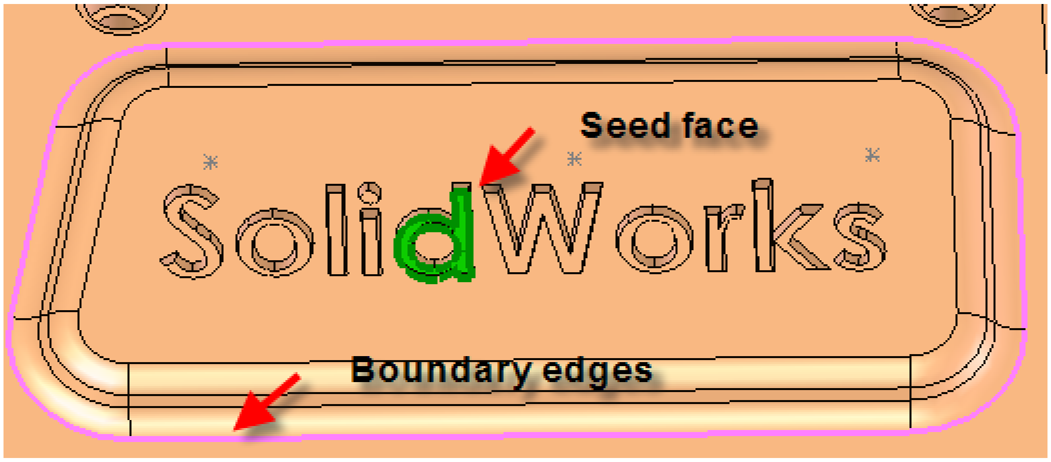
- Edit color： Specify the core/cavity color， by default, green for cavity, red for core.

Specify pull direction： If the pull direction is specified, only the faces that satisfy the mold draft condition will be selected.

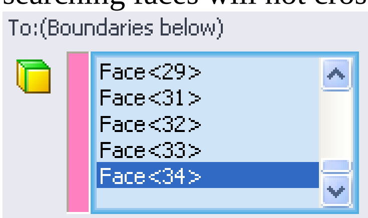
Using this function, only the connected faces are selected and can be knitted together.

Reverse selection：Reverse the mold open direction

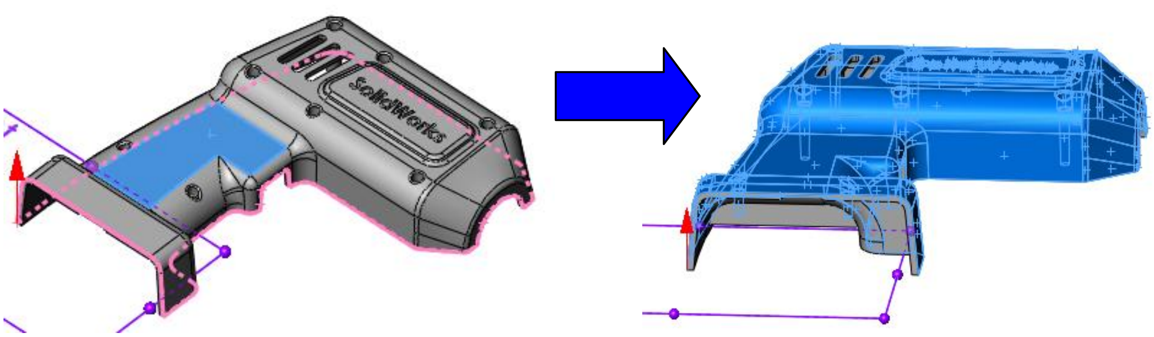
From： (seed face)



To:(Boundaries below): Boundaries could be edge or face type, by selecting the boundaries, the searching faces will not cross them.



In the following sample， specify direction and seed face, click Search cavity faces, result as below.



Further action:

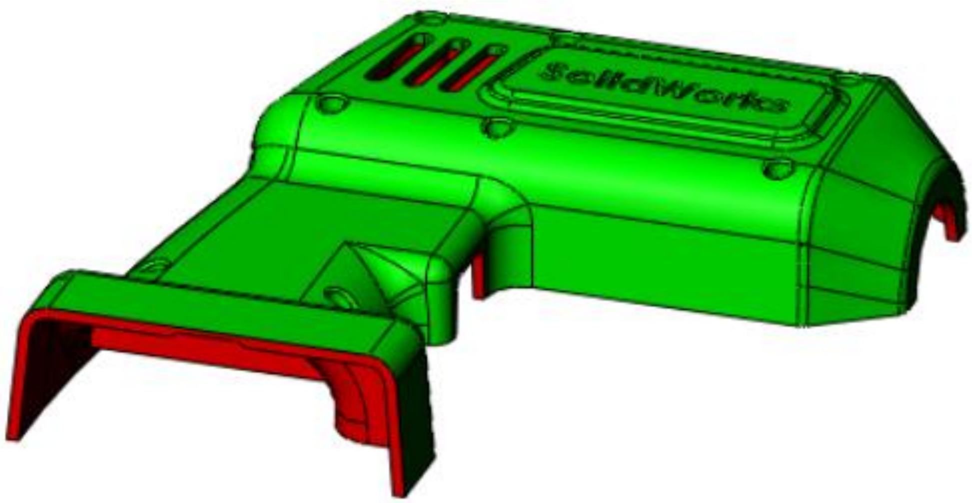
Apply color： Add color on the selected faces

Remove color： Remove face color on the selected faces

None： Select the faces alone, user can do any further operation by theirselves.

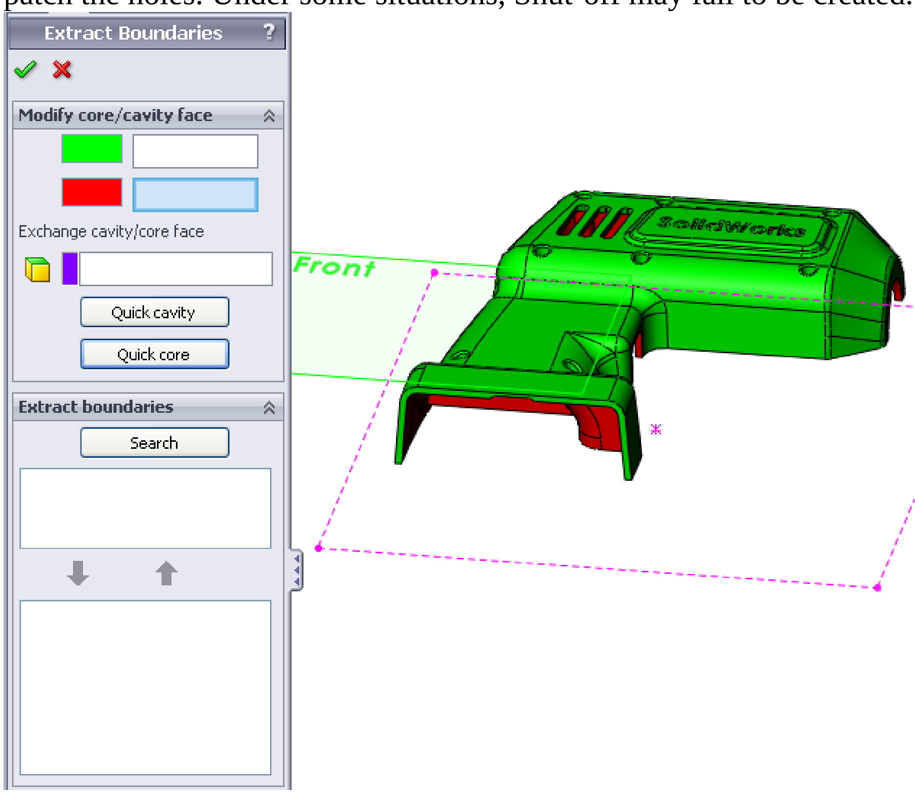
Additional cavity faces： Basides the faces selected automatically, can pick up more faces.

Click ， all faces on the model are classfied as core or cavity faces by different color.



3.3 Extract parting lines

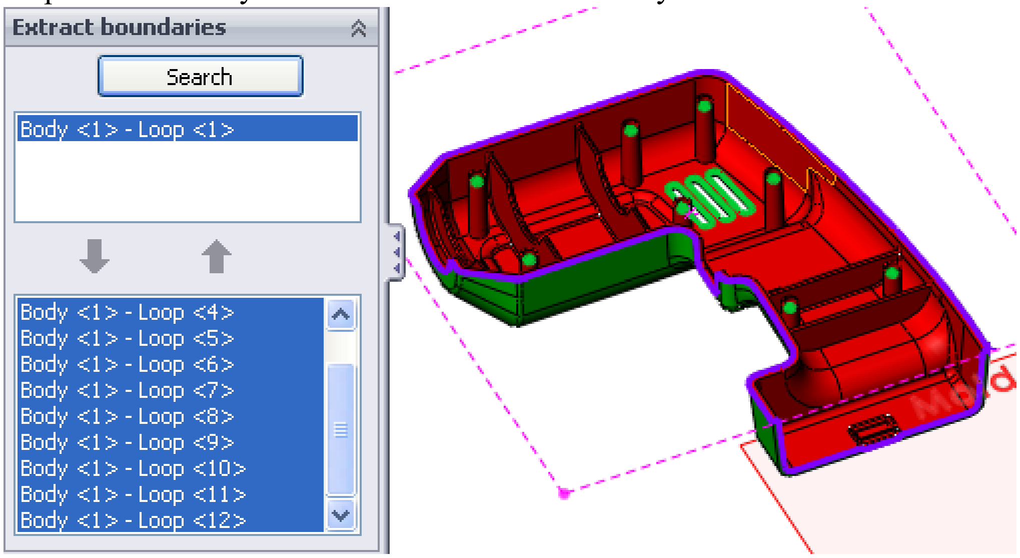
Once the core/cavity faces are fully defined, based on the face color, system can use this function to extract the parting lines and hole loops. Then, this function will try to use Shut-off surface to patch the holes. Under some situations, Shut-off may fail to be created.

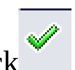


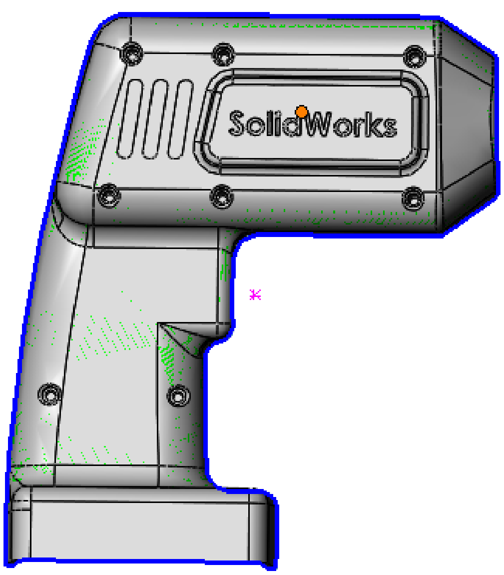
Exchange cavity/core face: Exchange the face color between cavity and core.
Quick cavity: All undefined faces are specified as cavity faces, color changed accordingly.
Qucik core: All undefined faces are specified as core faces, color changed accordingly.
Extract boundaries: Extract the parting lines and hole loops for shut-off surface.

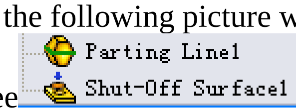
Click 

, All boundary edges will be extracted and classify into outer loop and inner loops automatically based on the current core/cavity faces' color information:



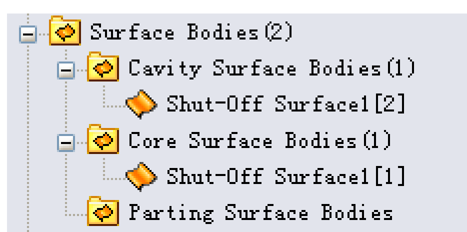
Click , The outer loops will be set as parting line and inner loops will be used to shut-off the holes on the plastic part。



If successful, the following picture will be shown on the feature tree 

.

Whether Shut-off surface created or not, greatly depends on the complexity of the holes on the part, do check the feature tree and the Surface Bodies folder before you proceed to the next step.

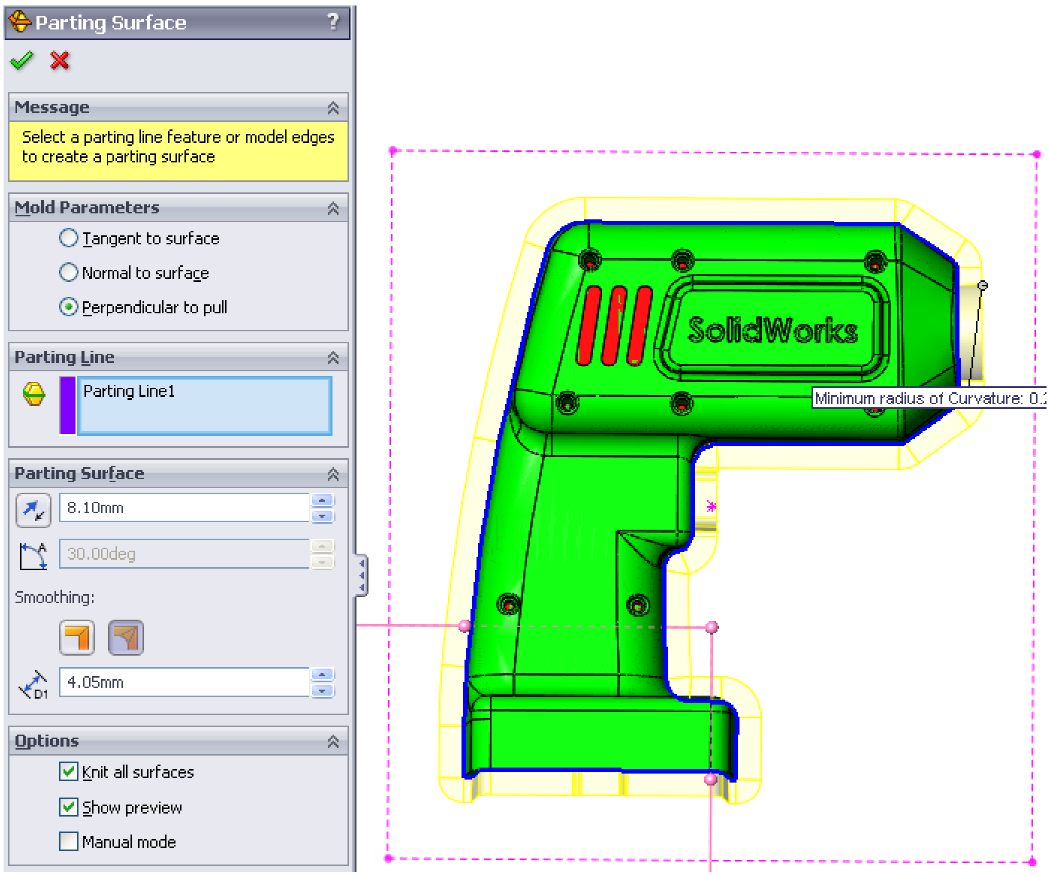


By right, the **Cavity Surface Bodies** and **Core Surface Bodies** should be under the **Surface Bodies** folder. If cannot find them, please do the shut-off manually in SolidWorks. For the complex hole, you can choose the no-fill option rather than contact or tangent.

Note: The Shut-off surface feature can be used once only in SolidWorks. For the complex part, to split the core/cavity using SolidWorks way, user can create their own surface bodies to patch the holes, and manually move those surface bodies to **Parting Surface Bodies** folder.

3.4 Create parting surface

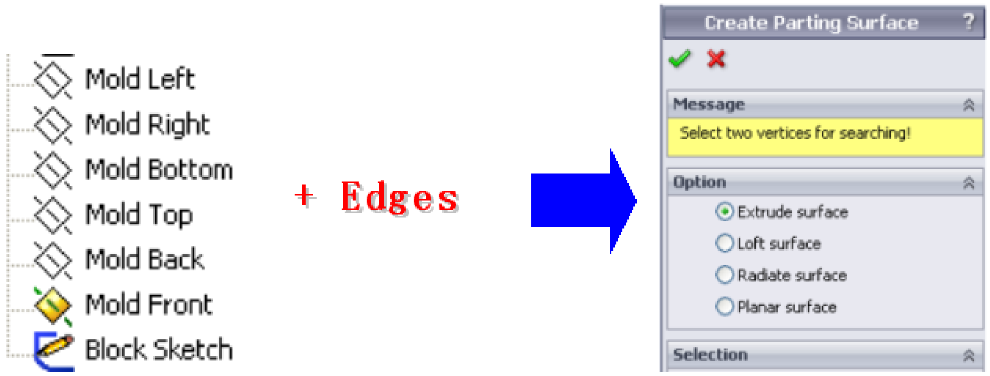
This command is used to create parting surface，if nothing is pre-selected, the SolidWorks Parting Surface interface will pops up.



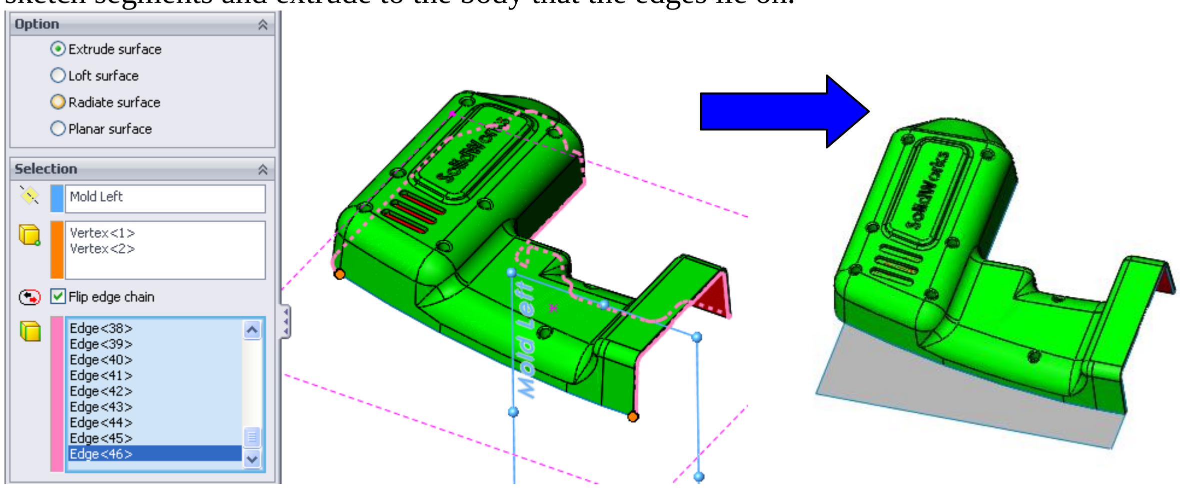
If parting lines exist, it is pre-selected automatically.

All surface-creating features in SolidWorks can be used to generate parting surface, the most common ones are Parting Surface, Ruled Surface, Extrude Surface, Loft Surface and Planar Surface.

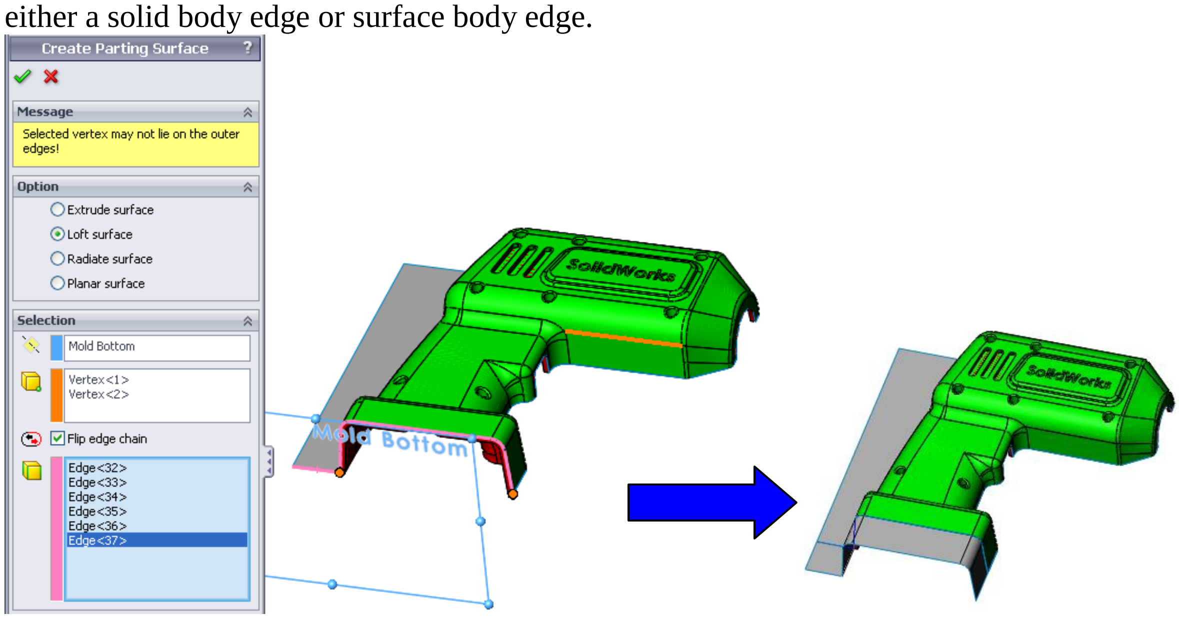
If a reference plane is pre-selected, the following dialog will pop up.



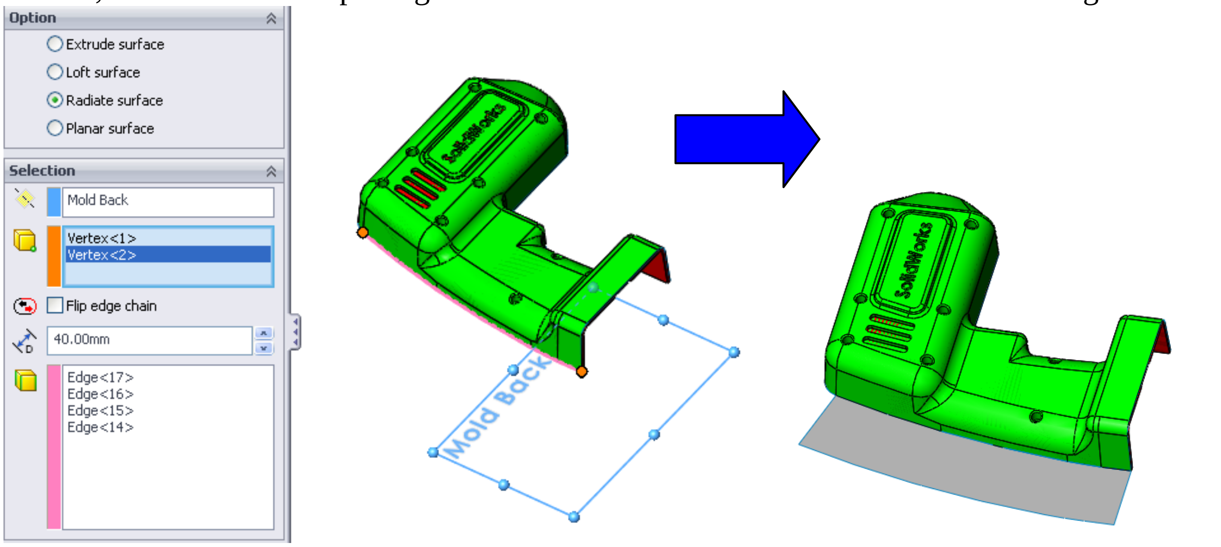
Extrude surface: Select the reference plane and some edges. The edges will be converted to sketch segments and extrude to the body that the edges lie on.



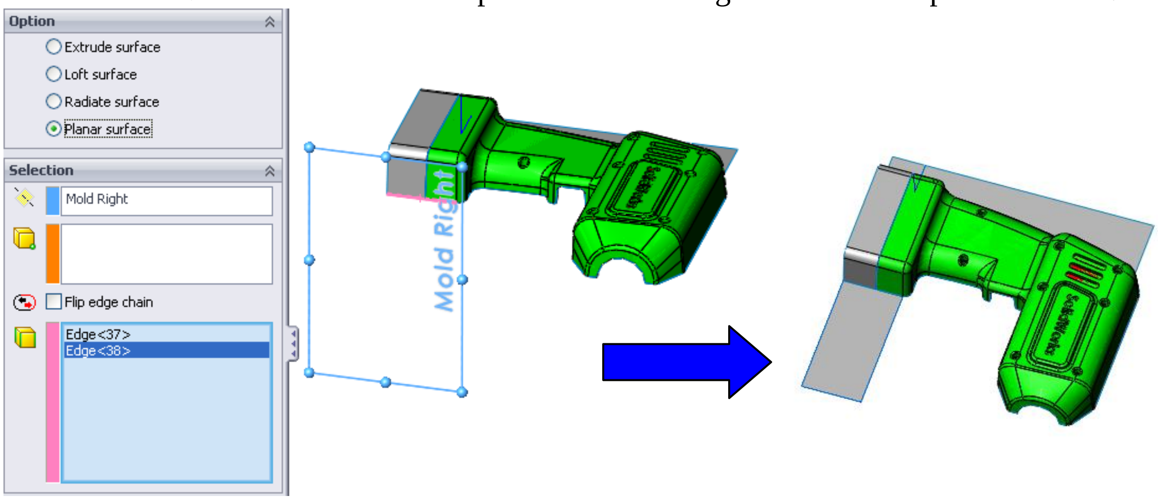
Loft surface: Select the reference plane and some edges. For loft surface, the edge could be either a solid body edge or surface body edge.



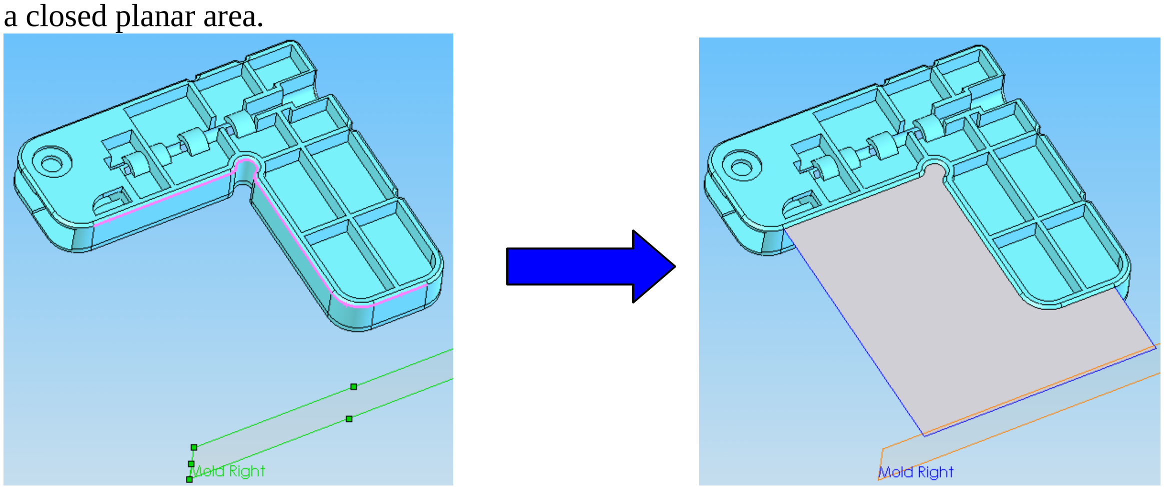
Radiate surface: Select the reference plane and some edges to create the radiate surface. Most of time, ruled surface and parting surface are better than radiate surface in mold design.



Planar surface: Select the reference plane and some edges to create the planar surface.

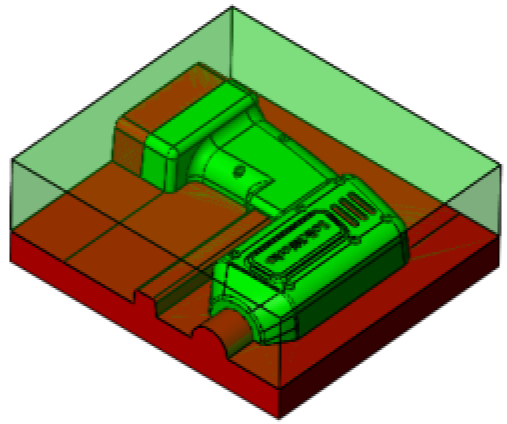


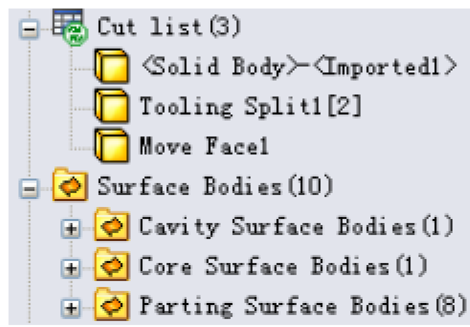
This option can simplify the planar surface creation. Some addtional edges are crated to enclose a closed planar area.



3.5 Create Cavity/Core

Supposing the parting line, parting surface and core/cavity surface bodies are all created successfully, click Create Cavity/Core will create the core/cavity automatically.

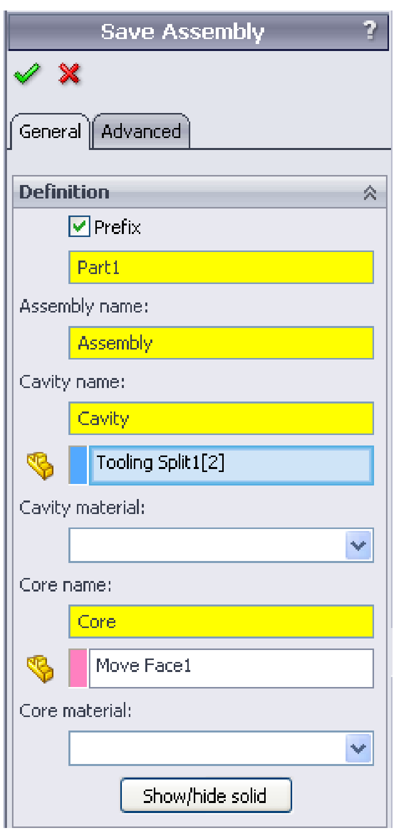




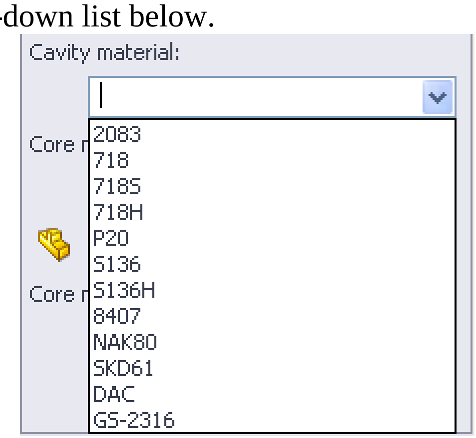
So far, all surface parting related functions are basically SolidWorks features but many steps are simplified a lot to improve the software performance.

3.6 Save Assembly

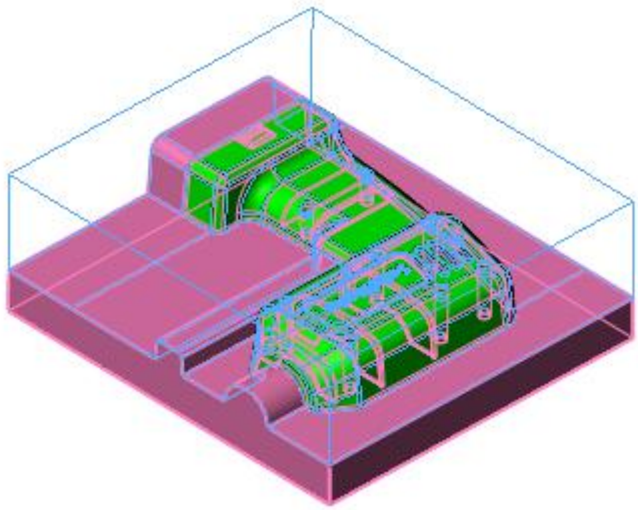
Save the core/cavity bodies into an assembly called product assembly, under this assembly, the original plastic part, the core and cavity are separated components.



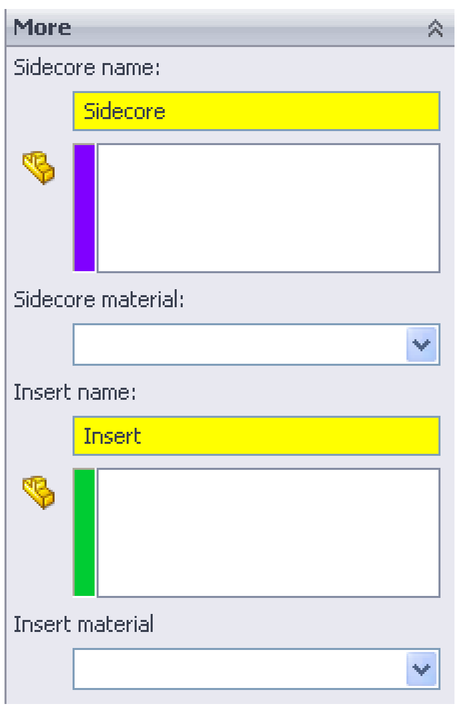
- **Prefix**: By default, it is the part name
- **Assembly name**: Assembly name for the product assembly
- **Cavity name**: Cavity name
- **Cavity material**: Can input or select a material name from the drop-down list below.



- **Core name**: Core name
- **Core material**: Can input or select a material name from the drop-down list below.
- **Show/hide solid**: Show or hide the selected bodies

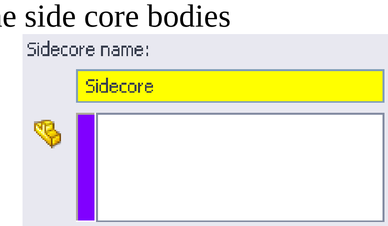


Please ensure the pre-selected core/cavity bodies are correct, if not, please pick up them manually in the graphic area or body folder.

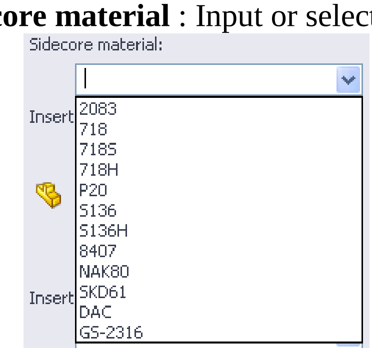


- **Sidecore name：** Name for side core

Select the side core bodies



- **Sidecore material** : Input or select

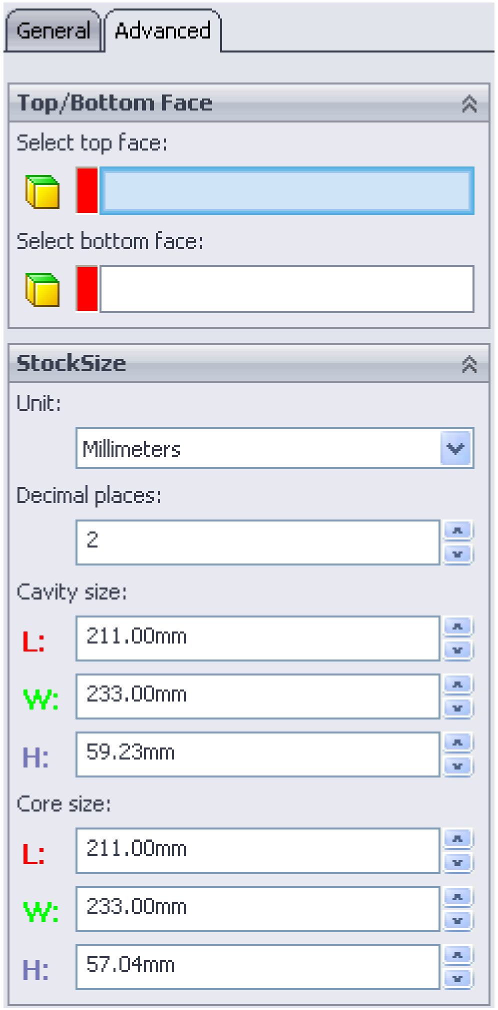


- **Insert name：** Name for insert component

Similar to the side core selection, pick up the insert bodies manually.

- **Insert material：** Input or select.

Advanced option：



Top/Bobbom Face : Specify the top or bottom face to create the CoreSheet for ejector trimming

- **Select top face：** Top face on cavity
- **Select bottom face：** Bottom face on core

StockSize : Those information will be savec into the file properties for future use such as BOM.

- Unit： Unit for the following size.
- Decimal places： Precision

- Cavity size

L: ： Length of the cavity workpiece

W: ： Width of the cavity workpiece

H: ： Height of the cavity workpiece

- Core size

L: ： Length of the core workpiece

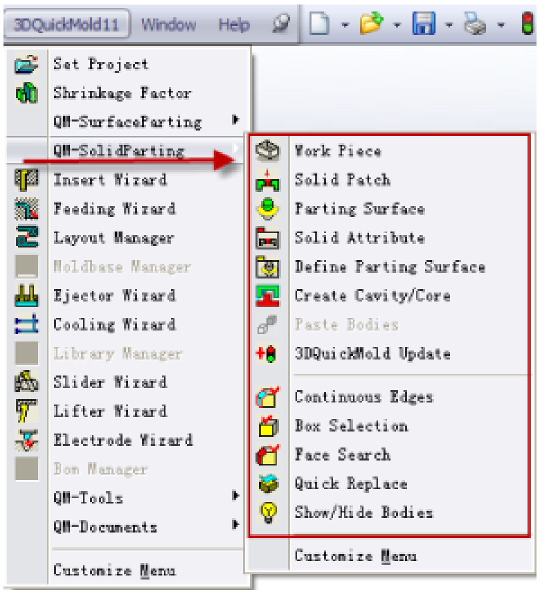
W: ： Width of the core workpiece

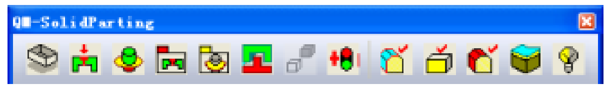
H: ： Height of the core workpiece

The system will calculate the minimum workpiece for the core/cavity based on the split core/cavity bodies and show them as default values for L, W, H. However, users can take the allowance into consideration and input their own values.

Chapter 4. QM-SolidParting

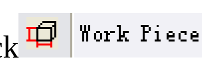
Solid Parting is a different parting concept from the Surface parting. Here, we will use solid bodies to patch the holes on part, use solid bodies to simplify the parting surface sometimes.

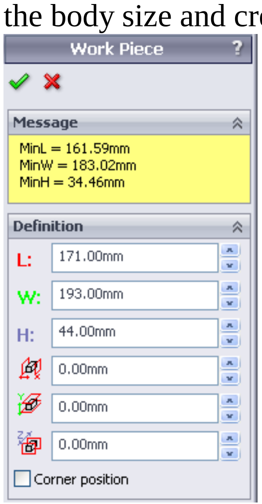


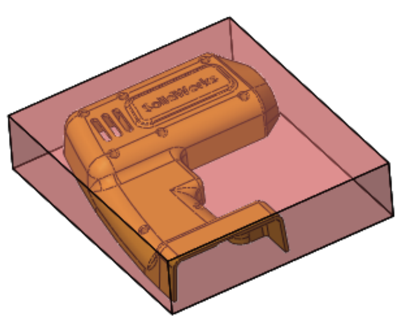


4.1 Workpiece

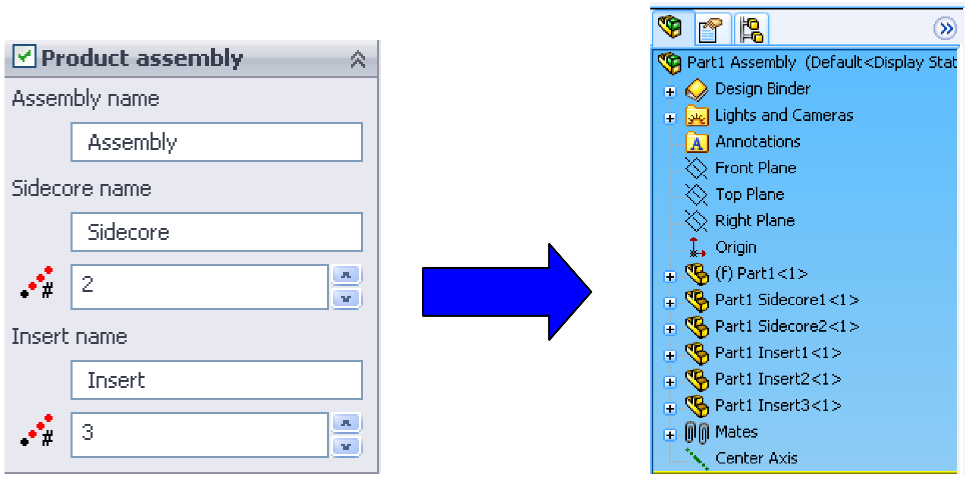
Workpiece function is used to define the work piece for machine core/cavity insert.

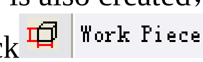
Click , 3DQuickMold will define the default dimension of the workpiece based on the body size and create a preview.





Check Product assembly, 3DQuickMold will create a * Assembly.sldasm assembly file (* is the name of the plastic part). The number of Sidecore and Insert can be defined here. It can be defined in the Product Assembly which will be mentioned later.

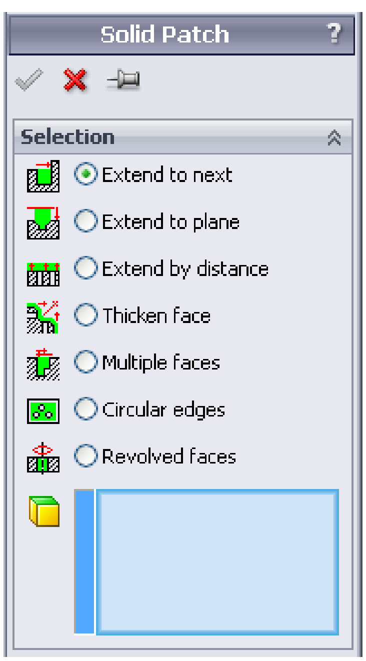


After setting is completed, click OK， 6 reference planes with the workpiece as border are created.（the first five reference planes are hidden， Mold Front is shown; the distance between Mold Front and sketch is the height of the workpiece), A sketch of Block Sketch with Mold Back as reference is also created， This block sketch determine the size of the final size of the core and cavity. Click 

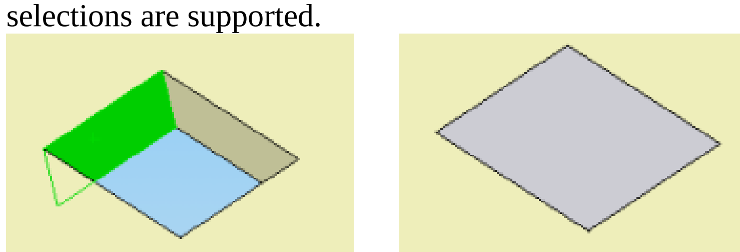
again can edit the existing work piece.

4.2 Solid Patch

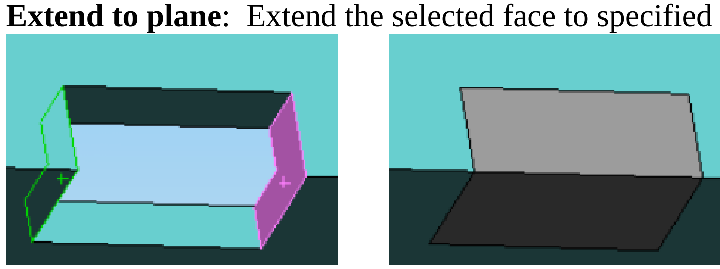
This function is used to create the solid bodies to patch holes on part.

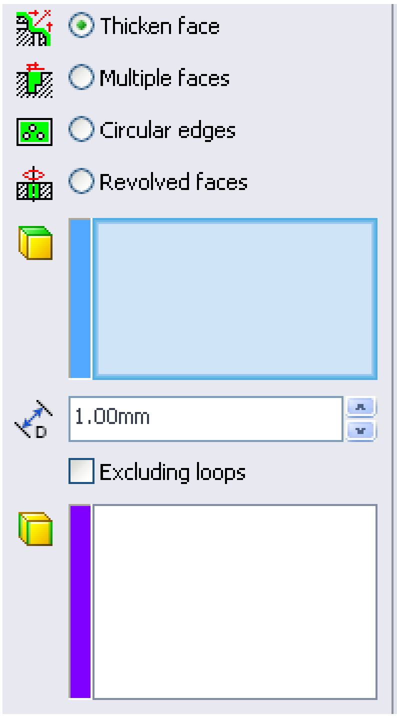


Extend to next: Extend the selected planar face and terminate onto the part. This function equals the Extrude feature but multiple selections are supported.

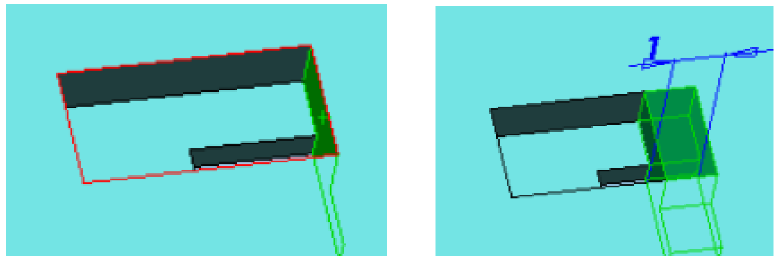


Extend to plane: Extend the selected face to specified plane





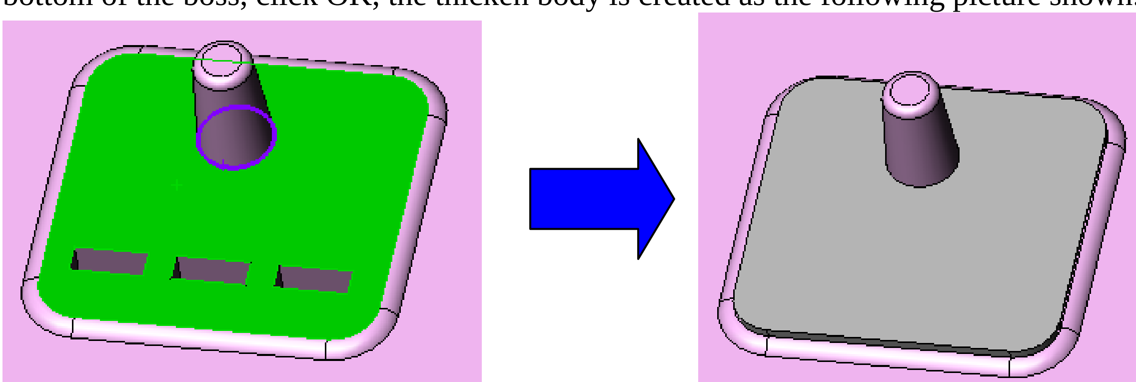
Extend by distance: Extend the selected planar to a certain distance.

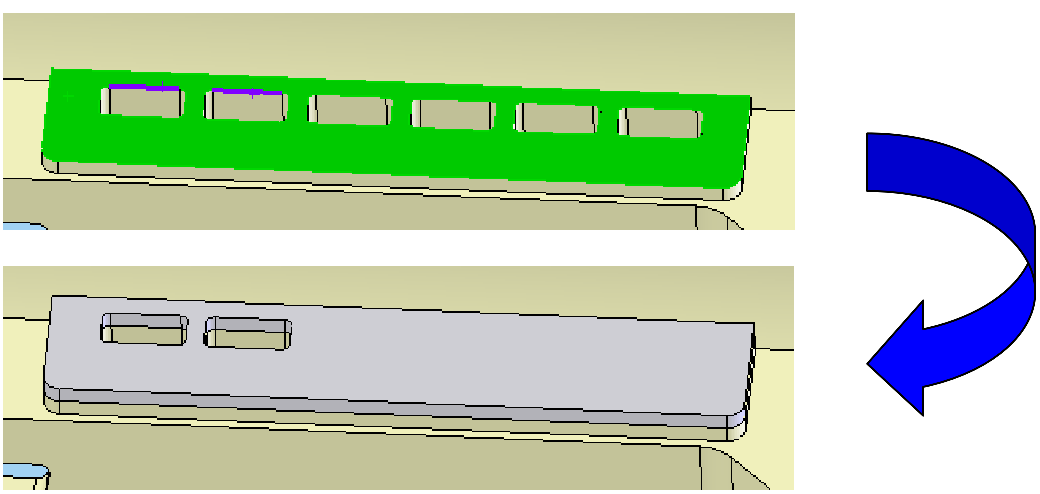


Thicken face: A thicken body is created on the selected faces

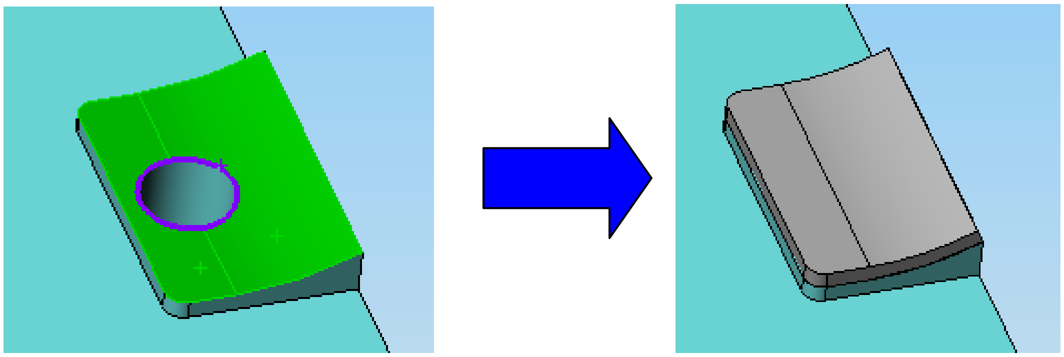
By default, all holes on the selected face are removed, for some exceptions, using Excluding loops, select the edge of the holes, those selected holes won’t be removed and remain on the thicken body when completed.

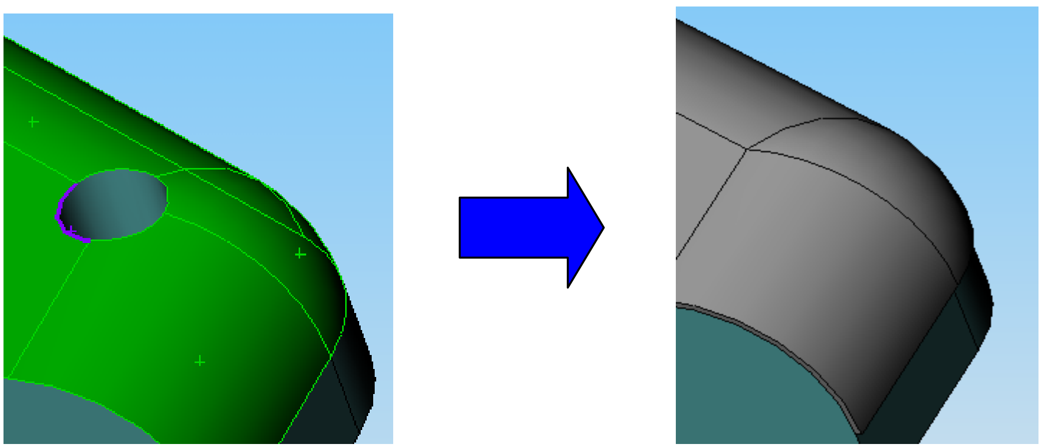
A sample shown below, there are holes on the selected face in green, selects the edge on the bottom of the boss, click OK, the thicken body is created as the following picture shown.



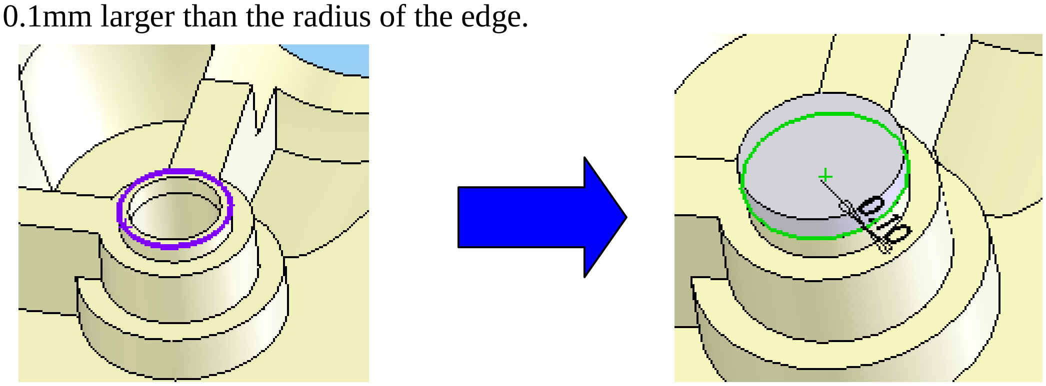


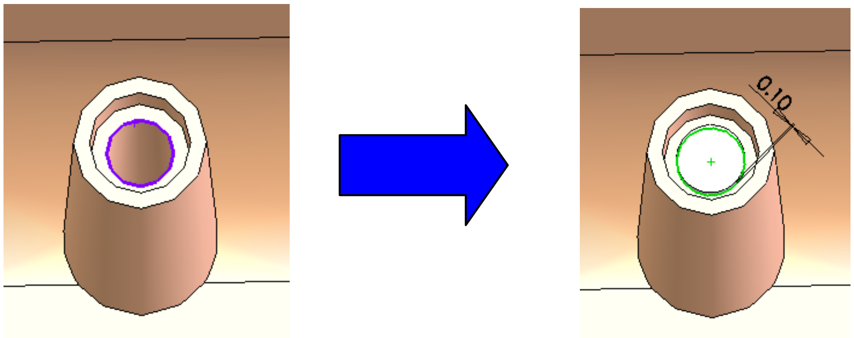
Multiple face: Fill the hole forming by multiple faces. For example, the hole shown below.



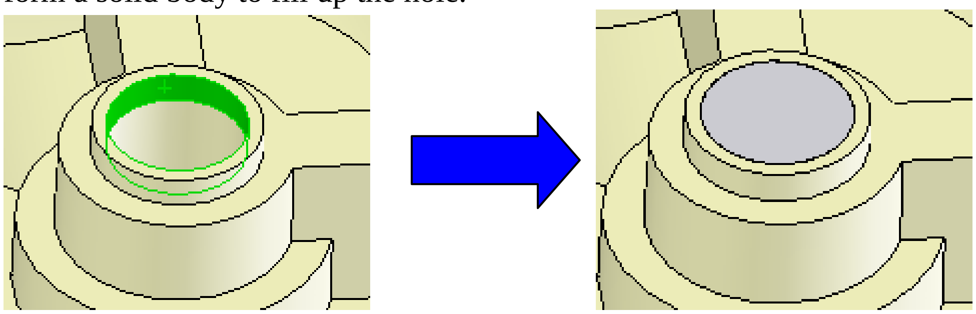


Circular edges: select the circular edges on the body, a cylindrical body is built with radius 0.1mm larger than the radius of the edge.

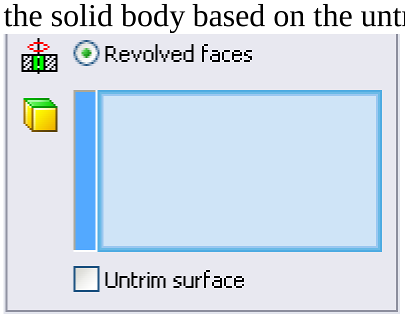


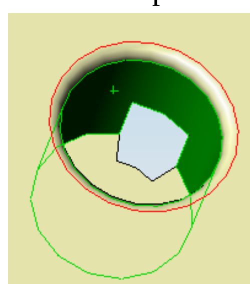


Revolved faces: Select revolved faces, this function combines a serial of Solidworks features to form a solid body to fill up the hole.



Untrim surface: Some revolved face should untrim the underlying surface first, and then create the solid body based on the untrimmed one. The below picture shows a very common situation.

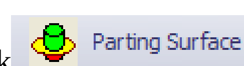




4.3 Parting Surface

This function is normally followed by the **Workpiece** creation. The six reference planes are used as references to build the parting surface.

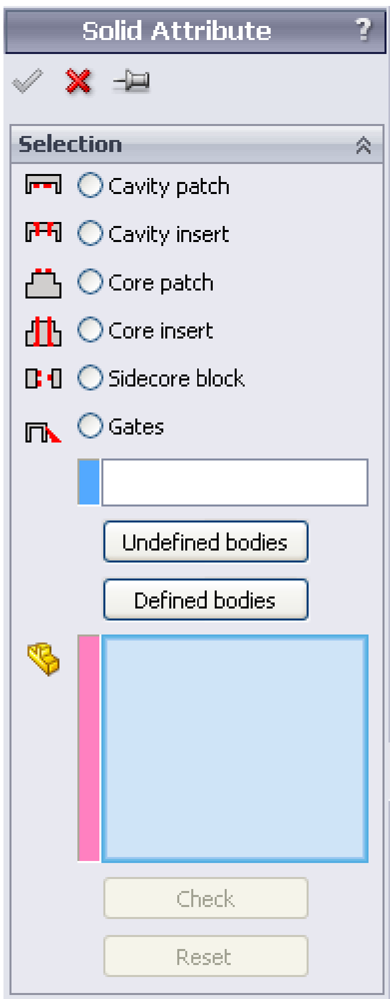
User can use any Solidworks surface feature to create the surface and specify them as parting surface later on. Using this function, the created surface will change to yellow color and set as parting surface automatically.

Click 

, the following dialog pops up, please refer to the function in Surface parting for details.

4.4 Solid Attribute

This tool is used together with "Paste body" for Solid parting approach in 3DQuickMold. Running this function, system will assign the attributes on the selected bodies.



Cavity patch: Specify the selected solid body as cavity patch body, a patch body normally will be pasted to the cavity side later on.

Cavity insert: Insert body on cavity side

Core patch: Patch body on core side

Core insert: Insert body on core side

Sidecore block: Sidecore body

Gates: Gate body

Undefined bodies: Select all bodies undefined so far

Defined bodies: Select all bodies defined already

Feature: When a feature consists of multiple bodies, pick up this feature on the feature tree, all bodies created by this feature could be selected automatically.

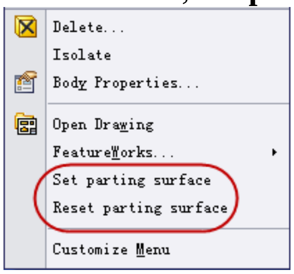
Check: Check the existing bodies for specified type.

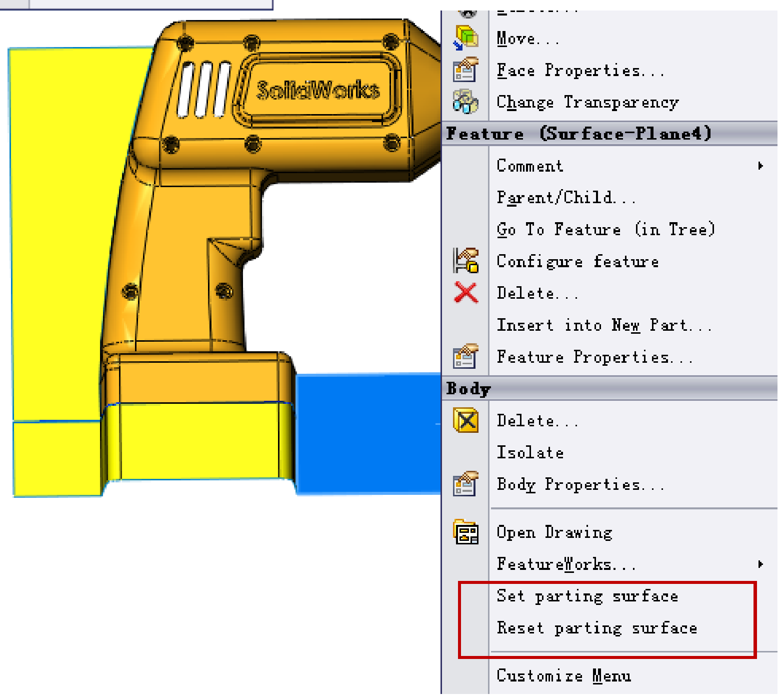
Reset: Reset the attribute on the body. If wrong attribute is assigned on the body, you need to reset it first, and then assign the new attribute again.

4.5 Define Parting Surface

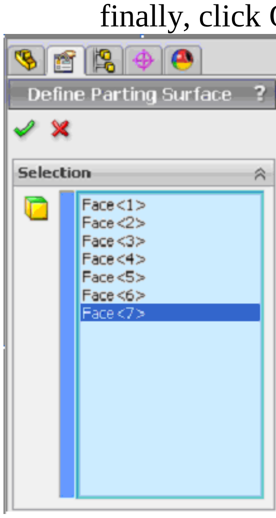
There are two ways to define the parting surface:

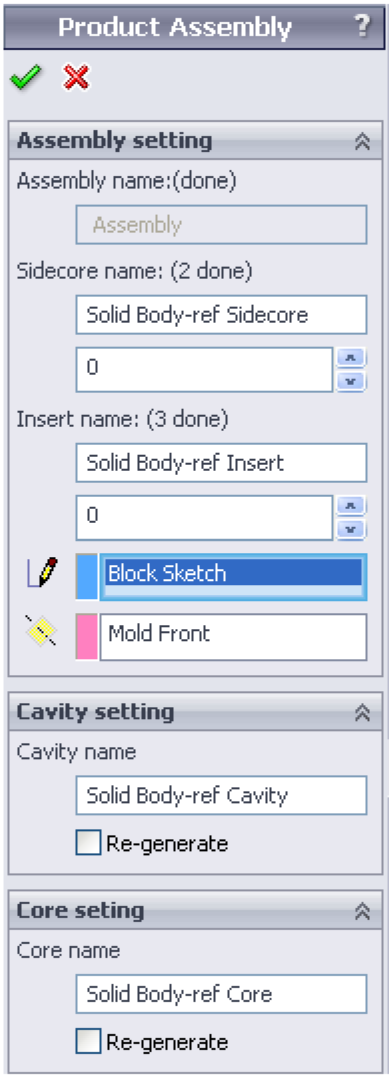
- Select face in the graphic or surface body under the body folder and right-click the mouse button, **Set parting surface** will do this.





- Click  , the following dialog pops out, pick up the face one by one, finally, click OK to confirm your selection, the selected face will become yellow.





4.6 Create Cavity/Core

This function is mainly used:

- as the final stage of the automatic split core/cavity
- to regenerate the core/cavity when the parting surface is changed.
- to add additional side cores or sub-insert parts to the product assemlby

Click 

, the property manager pop out. If the number of Sidecore and Insert of the workpiece is not specified before， specify it in the Product assembly。

Assembly name： Input name of the product assembly. If that assembly is already generated in Workpiece, the name field will be greyed out.

Sidecore name： Input the name of the side core, the number inside the bracket indicate the existing number of sidecore. If the number of sidecore is not sufficient, add by selecting a suitable number.

Insert name： The configuration of Insert is the same as sidecore

Sketch： Select sketch of the workpiece， **3DQuickMold** will select the sketch of Block Sketch generated in workpiece.

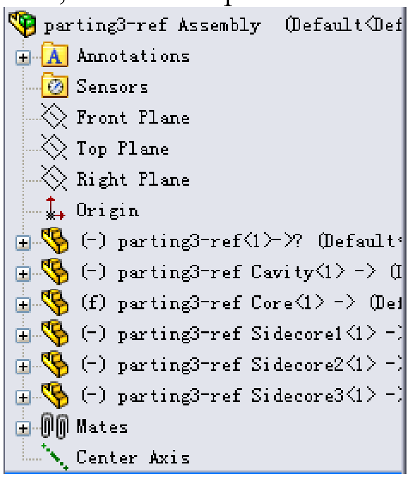
Plane： **3DQuickMold** will automatically select the Mold Front generated in Workpiece as the reference plane； This plane will be used to define the end condition of extrude feature.

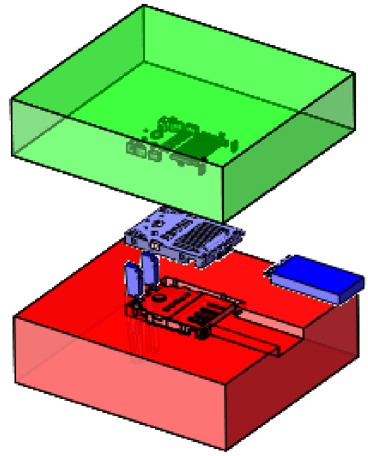
Custom sketch and plane can also be used. It is recommended to use the default selection.

Cavity setting:
Cavity name： Input the name of the cavity
Re-generate: When the parting surfaces are changed, you need to re-generate the cavity.

Core setting: Similar to the Cavity setting but
Tips: After Re-generate, the newly generated part will replace the previous core/cavity, any special feature on the previous core/cavity will not pass to the new one.

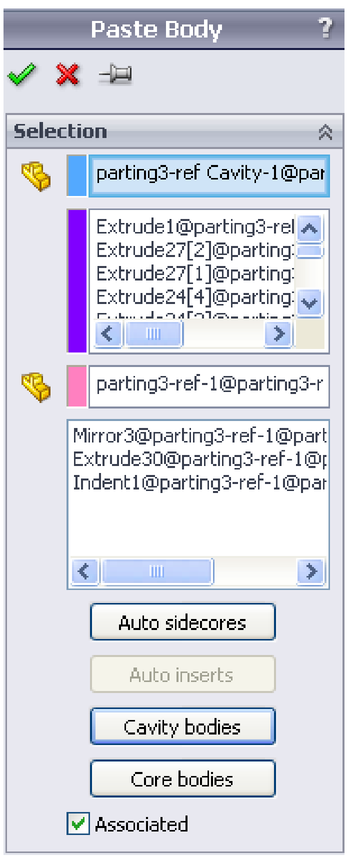
After the setting is completed, click OK，**3DQuickMold** will generate * core.sldprt，* cavity.sldprt and re-build * assembly.sldasm， *side core1.sldprt, *side core2.sldprt…for side core, *Insert1.sldprt and *Insert2.sldprt…for Insert component.





4.7 Paste Bodies

Normally, this function is used together with **Solid Attribute**
The bodies with attribute defined in Solid Attribute can be pasted from the plastic part to target components such as core/cavity, sub-insert or sidecores in batch.



Select component: Select the target component where the bodies will be pasted.

Select bodies: Select the bodies to paste.

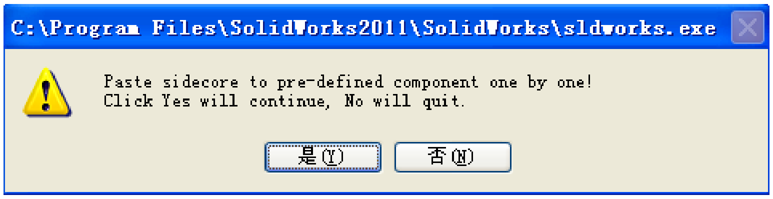
Select product: By default, the plastic part is automatically selected, bodies defined as side cores will be listed out for your selection.

Sidecore list: All Side core bodies defined in the product are listed out here.

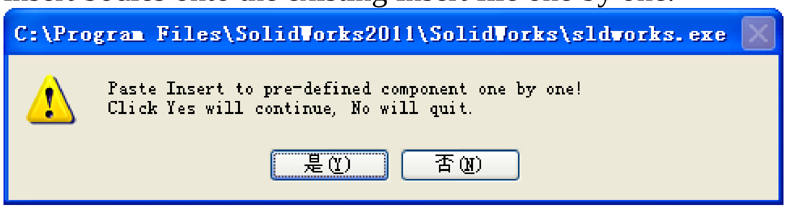
Cavity bodies: Click this button, the cavity blocks will be selected， they will be pasted to the cavity part.;

Core bodies: Click this button, the core blocks will be selected， they will be pasted to the core part

Associated: Turn on and turn off the associativity.
Auto Sidecores: Click this icon, the following dialogue box pop out, click **Yes** to paste defined sidecore bodies onto the existing Sidecore file one by one.

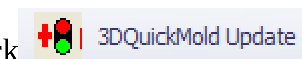


Auto Inserts: Click this icon, the following dialogue box pop out, click **Yes** to paste defined insert bodies onto the existing Insert file one by one.

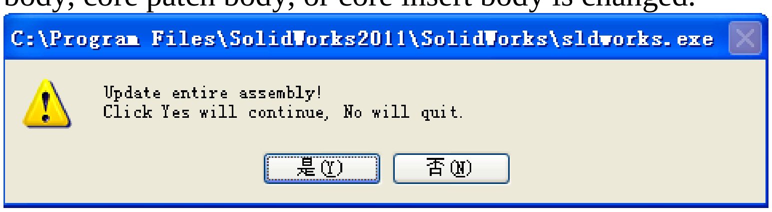


Finally, Click OK， the bodies will then be pasted to the selected target component.

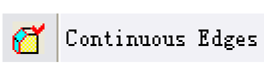
4.8 3DQuickMold Update

Click 

to update the part or assembly if the cavity patch body, cavity insert body, core patch body, or core insert body is changed.



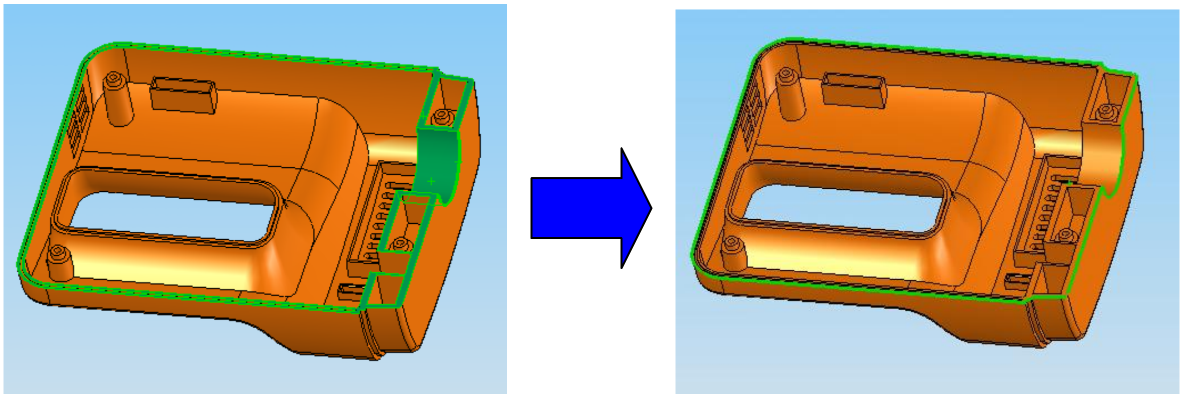
4.9 Continuous Edges



This is a tool for searching of edges, through difference combination of selected faces and edges（there are three combination， three corresponding usage），quick searches of all desired edges, select face first then edges， click **Continuous Edges** to proceed with the search.

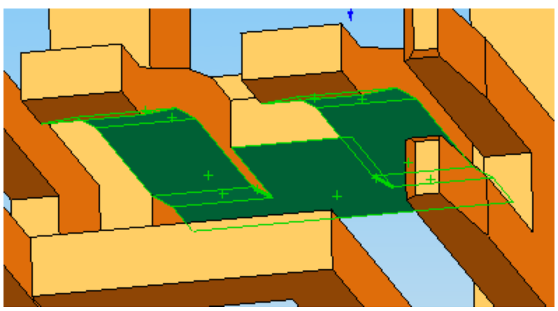
Usage 1： select one face (or more than one continuous face), on this faces select an edge， click ***Continuous Edges***， 3DQuickMold will search for a close loop among multiple faces. Normally, the parting lines are many but they belong to a few faces only, in this case, to avoid missing the small edge selection, this tool could be used.

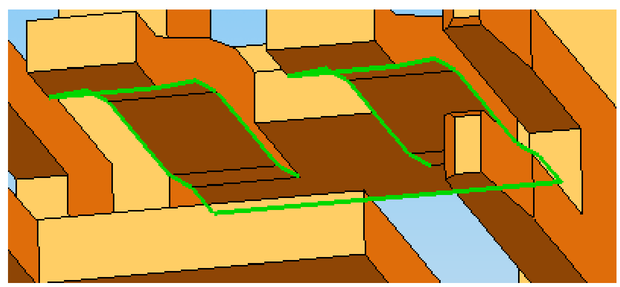
Select two faces and one edge as the following picture shown. Click **Continuous Edges,** the resulting edges are highlighted as below. It is a complete loop.



Usage 2： select one or more adjacent faces, on this faces select any two edges, click **Continuous Edges**， 3DQuickMold will search a partial open loop, the direction of the open loop will follow the first picking up point on the first edges.

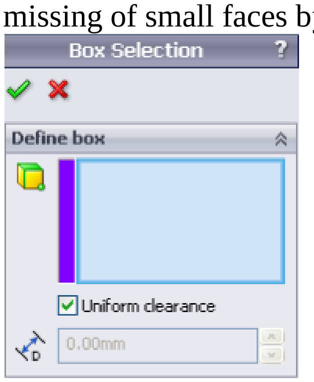
If the picking up point on the first selected edge is near to the top end, the partial open loop is selected as the below picture shown.

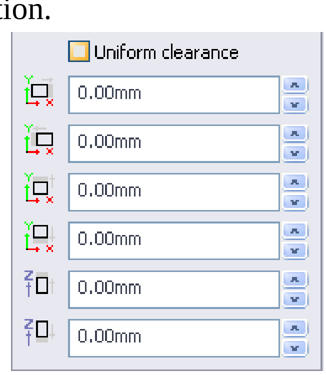


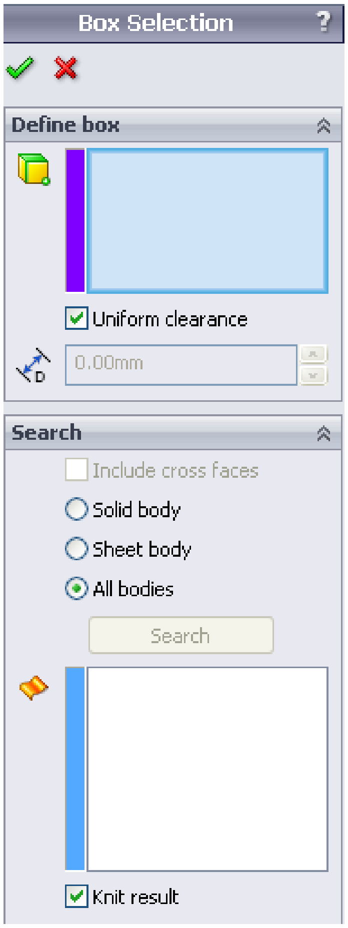


4.10 Box Selection

This is a tool for searching of faces， through a space box defined by some entities, all the faces fully enclosed by the box will be selected， it is commonly used in searching faces for the side core, inserts and electrode. Using these tools, users can avoid the missing of small faces by mouse click selection.







Define box: define the box dimension, entities can be vertices, edges and face.

Uniform clearance: the Uniform clearance will be added in the dimension of the box in the six direction of X, -X, Y, -Y, Z, -Z, non-uniform clearance could also be added.

 : Clearance value

Include cross faces: Including faces that are crossing the selection box.

Solid body: ct faces on solid bodies only.

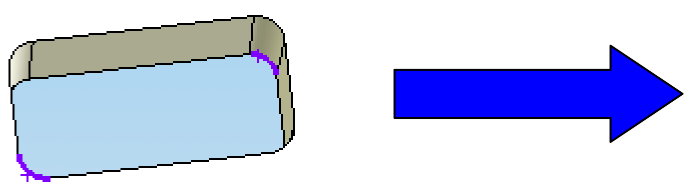
Sheet body: Select faces on sheet bodies only.

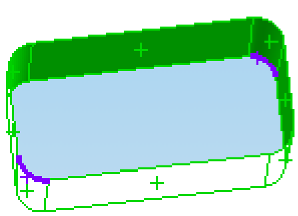
All bodies: Select faces both on sheet bodies and solid bodies

Search: Start searching the faces under the current conditions.

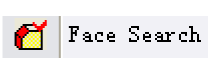
Knit result: If this option is checked, the selected faces will be knitted after OK

When set up is done, click OK, 3DQuickMold will search and highlight all the faces that fulfill the search criteria.

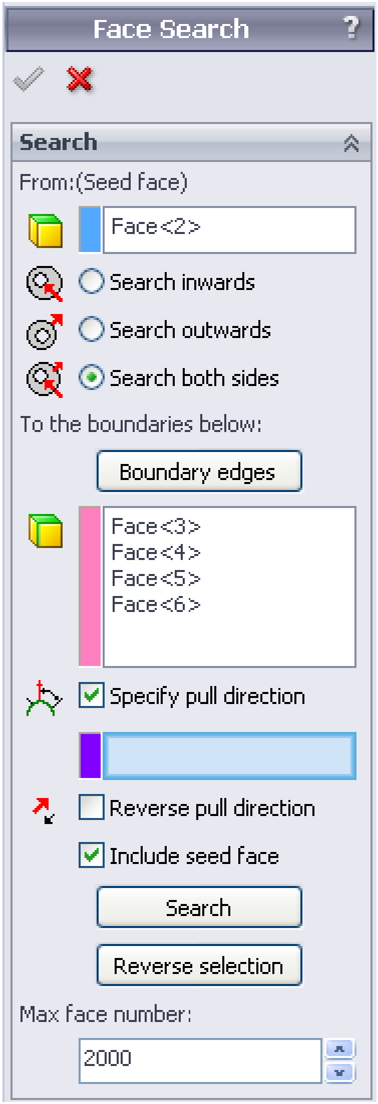




4.11 Face Search



This is a tool for searching faces, by selecting boundary condition and a seed face to propagate the search of faces on the part surface.



From: (seed face)

Search inwards: Search inward from the seed face's inner loop until it is bounded by the boundary edges.

Search outwards: Search outward from the outer edge of the seed face until it is bounded by the boundary edges.

Search both sides: Search both inward and.

To be boundaries below:

Boundary edges: Click this button, the parting lines that have been set will be selected automatically into the list box. The boundary edges can also be selected manually. Boundary can either be edges or faces.

Specify pull direction: If the pull direction is specified, only the faces that satisfy the mold draft condition will be selected.

Reverse pull direction: If the pull direction shown is not correct, click this button.

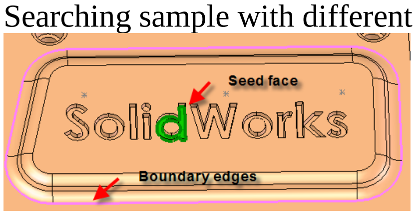
Include seed face: Include the seed face in the result list.

Search: Start searching faces that fulfill the search criteria.

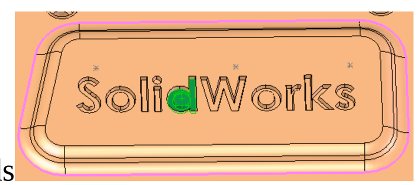
Reverse selection: Reverse the search list, this means all unselected faces on the part will be selected while the highlighted ones will be removed from the list.

Max face number: Specify the maximum face number to be selected. When the boundary edges are not defined correctly to enclose a regional area, search faces will result in all faces on the part selected, in this case, max face number can help user to find out the open edges among the boundary edges.

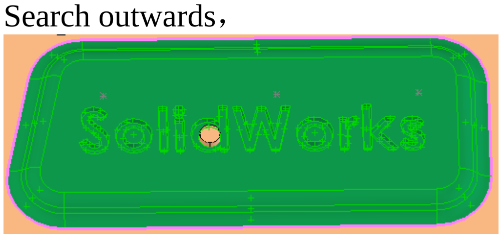
Searching sample with different option



Search inwards

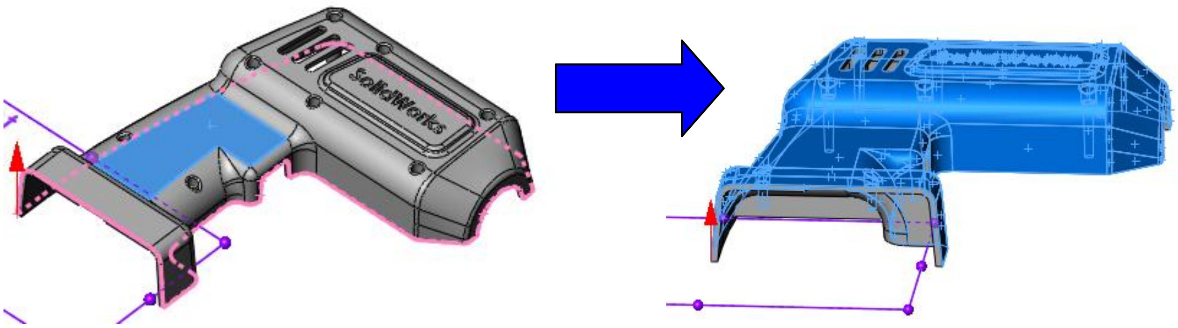


Search outwards,



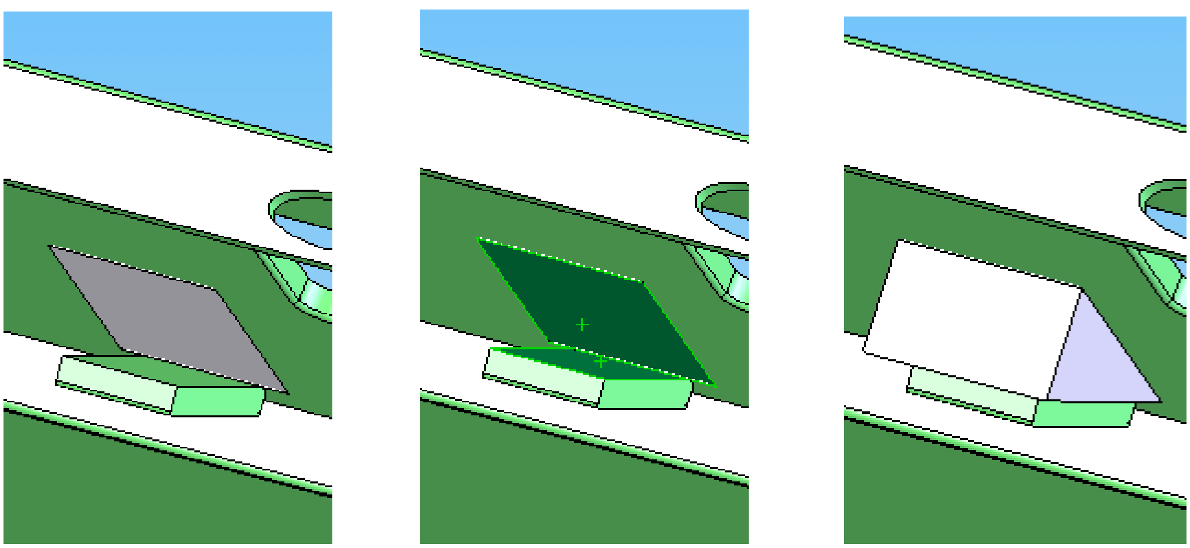
Search both sides

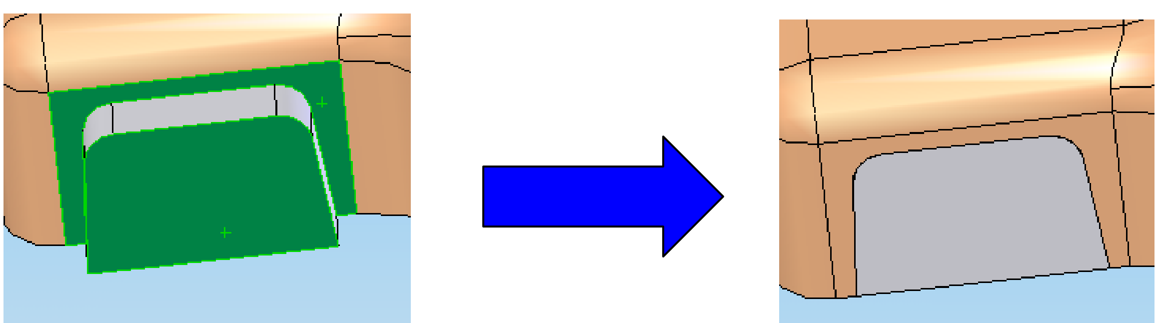
Select a seed face and specify the pull direction, the face searching can be carried out.



4.12 Quick Replace

To use “Quick replace”, two faces on different solid bodies could be selected, the system will always use the face on the plastic part to replace the one on other bodies. This means the original part won’t be changed whatever the order users pick up the faces.

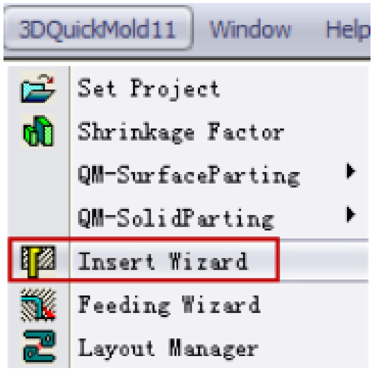


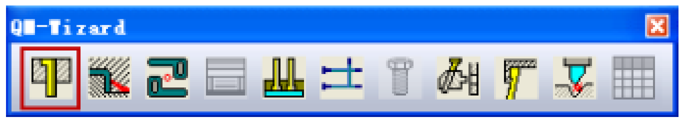


4.13 Show/Hide Bodies

Select a face or edge on body, click this function, the underlying body will be hidden, if nothing selected, click this function again, all hidden bodies will be shown again.

Chapter 5. SubInsert Manager

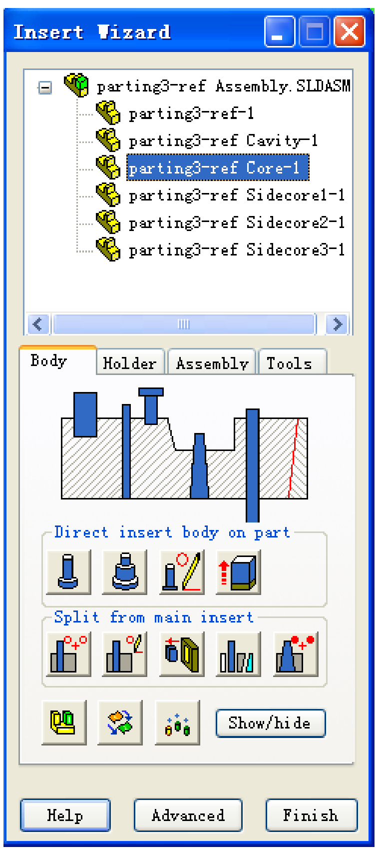




Usuallly, there are two methods for sub-insert desgin

- Design sub-insert on product, this step is done before mold spliting
- Design sub-insert on core or cavity, it is done after mold spliting

Click on  to access Subinsert design dialog.



Typical design procedures as follows:

Define sub-insert body => Create holder => Save sub-insert

On each wizard page, there are some utilities for users to do some basic design jobs quickly. By pre-selecting some edge, face or sketch, those functions can be performaned directly without further dialog shown up.

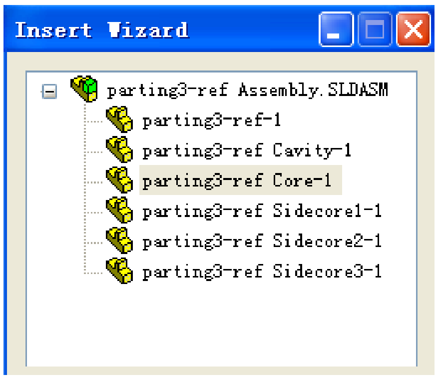
If a design job needs a few parameters to key in and preview is required, you can click “Advanced” button on the bottom of the wizard page.

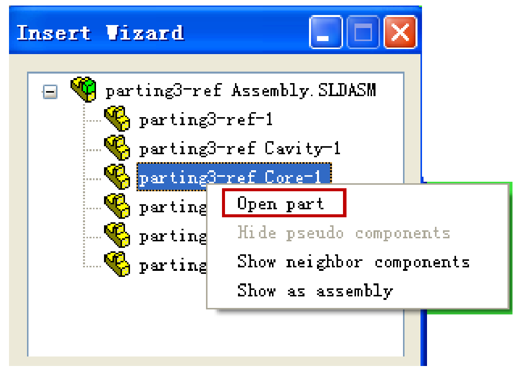
Normally, when you hover the mouse cursor over the bitmap button, there is a tooltips to show you what this button means.

Most of time, if don’t know how to use those functions, simply click on the button without any selection, there is an information dialog pops out to show you how to use it.

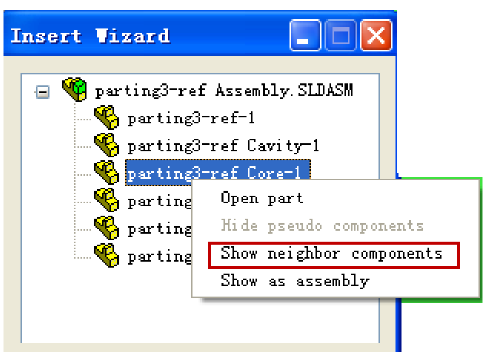
5.1 Components navigator

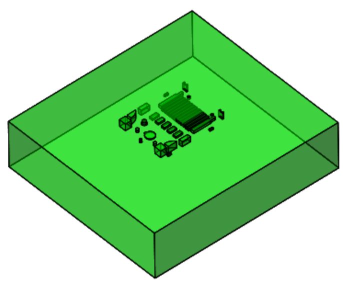
If working at assembly model, the assembly and all children components are listed as a tree to help user to navigate those models conveniently. Right-click mouse at selected item, there are different Open mode for you to choose. Open part is to activate the selected model.

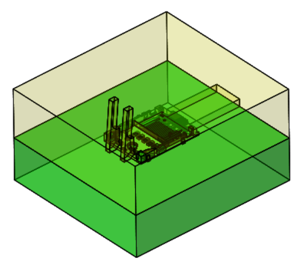




If select Show neighbor components， while the part opens, its neighbor components under the assembly will be displayed as temporary bodies.



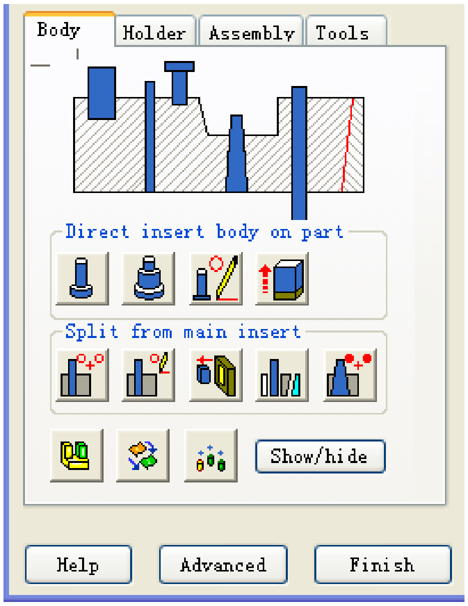




Different from the Show as neighbor components, if Show as assembly is selected, all children components are shown as temporary bodies instead of neighbor components alone.

Hide pseudo components will hide all temporary bodies if any.

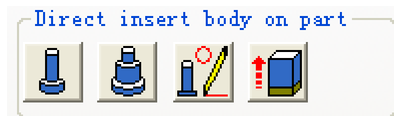
5.2 body



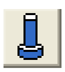
This functions works on part model only

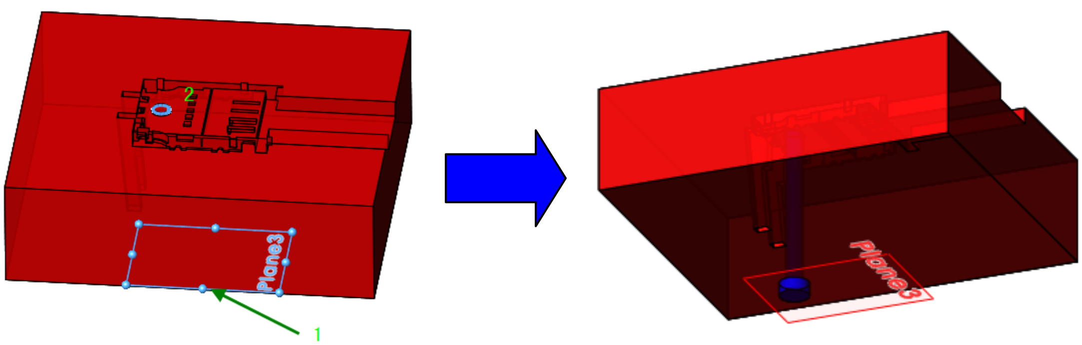
On this page, you can design inserts on part directly for the simple case, create drated insert or cut the insert from cavity/core model directly.

Direct insert body on part

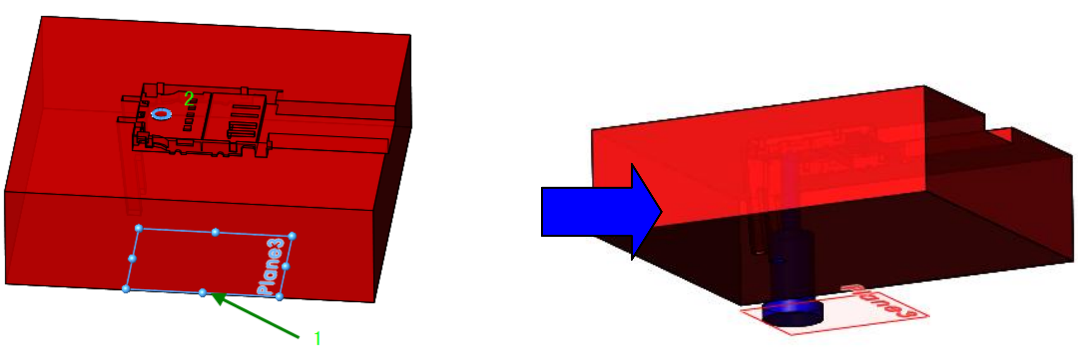


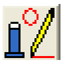
Those functions are all required pre-selections of edge, face or sketch.

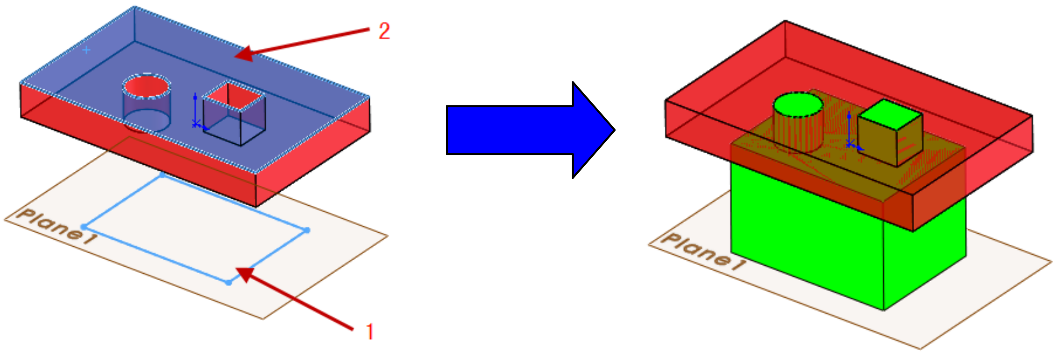
: Select a reference plane and a circular edge to create the round insert body directly on part. This function is used to design a core pin for the mold on core or cavity side.



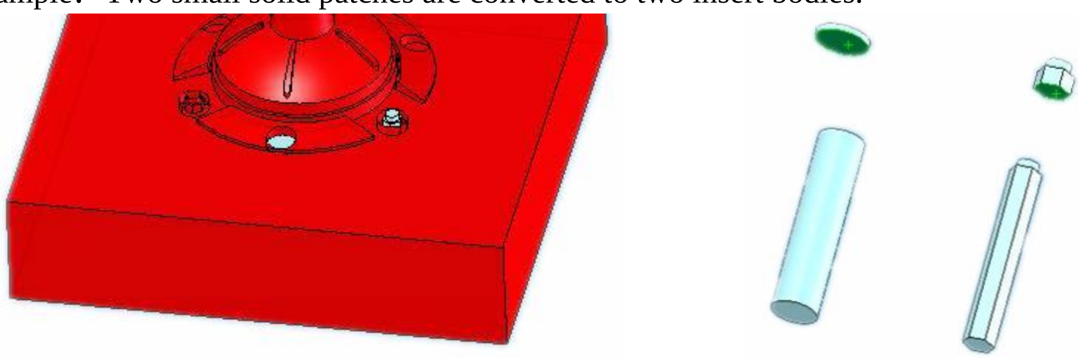
: Same selection as above, but a stepped core pin is created once the button is clicked.



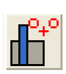
: Select a reference plane and a face on the part, an extruded body feature is created to be considered as insert, normally, this function is used to patch the holes on the face as well.



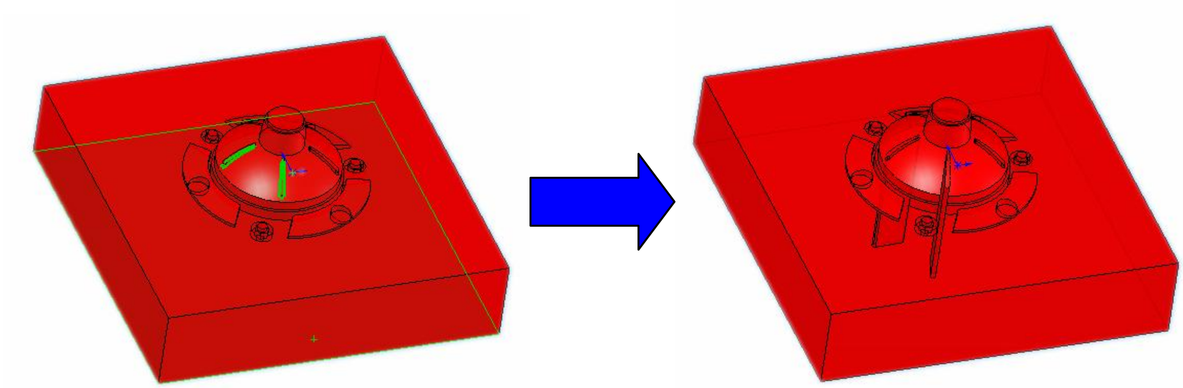
: Extend the selected body face to the bottom face of core or cavity, so it could be taken as an insert body. It is normally used to convert a solid patch to insert body.
Sample: Two small solid patches are converted to two insert bodies.

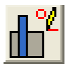


Split from main insert

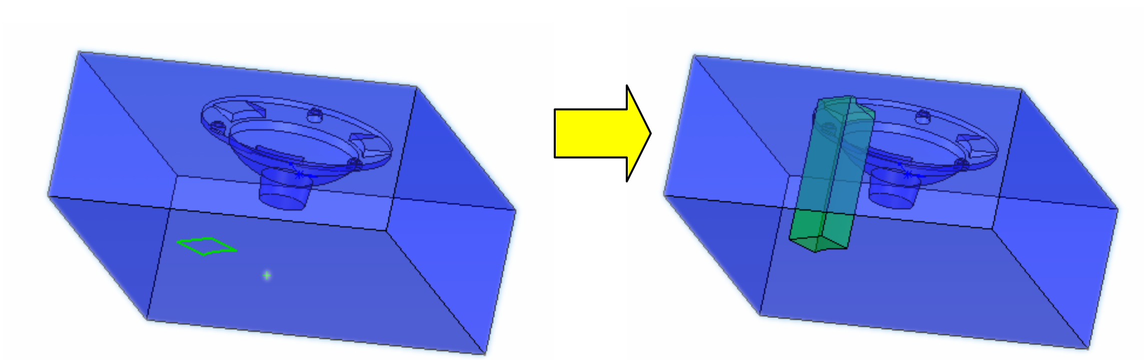
: Select in sequence the bottom face of the mold core or cavity and a closed chain edges to define the sub-insert body.

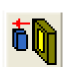
Continuous edge tool is normally used to help user to pick up the chain edges, this way, a closed loop is ensured.



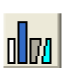
: Using sketch to define the insert body. Select a sketch, click this ion, the insert body is cut from the core/cavity immediately.

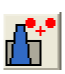
This is the most flexible way to define an insert body. User can use SolidWorks sketch tools to create the insert profile.

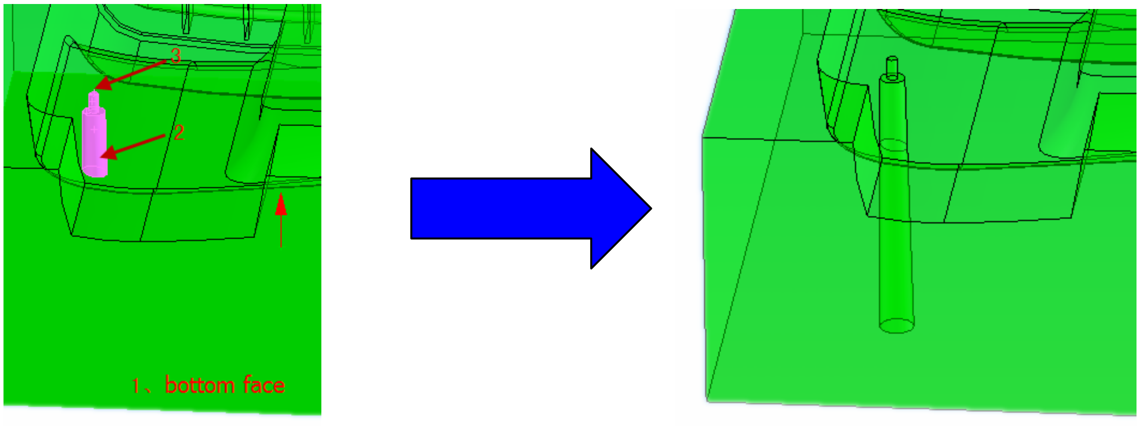


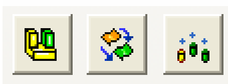
: Same as SolidWorks core function. To use this function, select a sketch and click this button, the SolidWorks “Core” page will pops out.

Compare to the sketch way mentioned above, the core function is good at creating a blind insert.

: Same as SolidWorks split function. The splitted bodies normally don’t need the pockets and holders as the typical rectangular ones or circular ones.

: Create a drafted insert body. Select the bottom face and some faces with draft to form the insert body. An example is shown as the following picture.



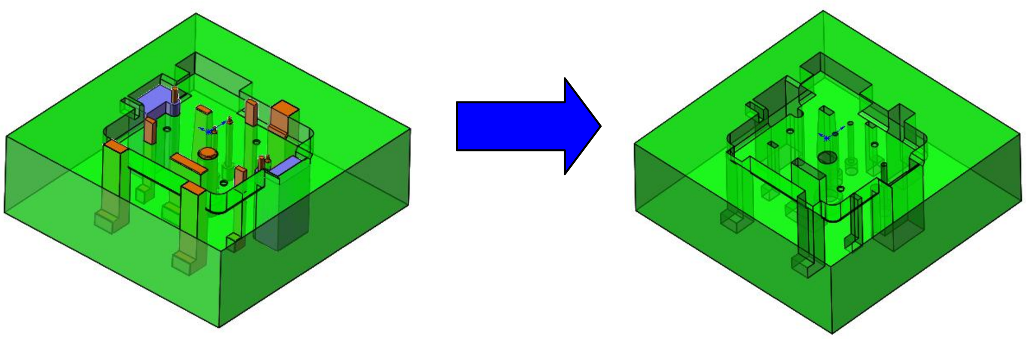
: Some SolidWorks functions are grouped here for user's convenience

: Mirror

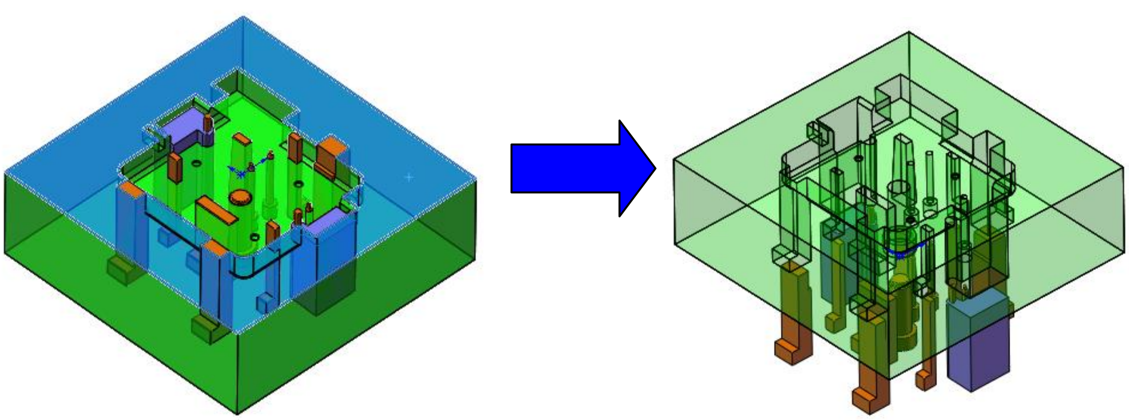
: Move/Copy

: Pattern

Show/hide: Show or hide the created sub-insert bodies, if nothing pre-selected, click this button will hide all existing bodies except the original model.

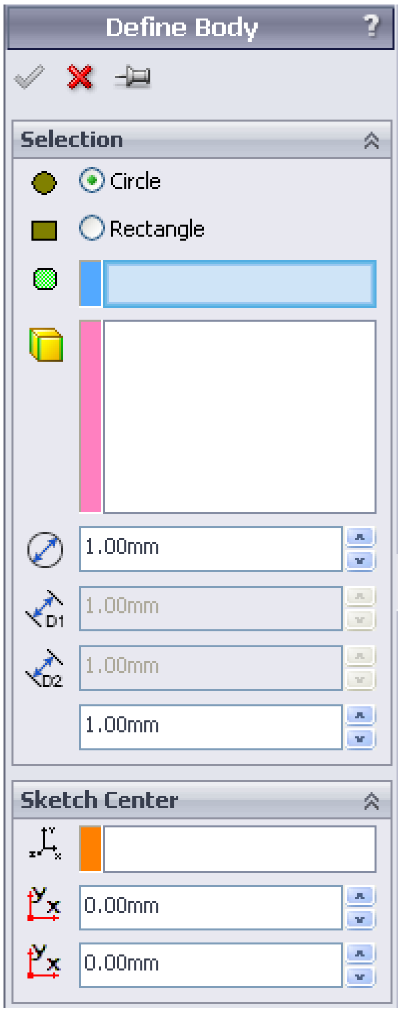


If a planar face is selected, its underlying body is displayed in transparent mode and move away a certain distance from the original position. Similar to explode view in assembly.



Advanced

Click the Advanced button on the bottom of the dialog, the Define Body page pops out.



: For circular sub-insert

: Bottom plane on core/cavity

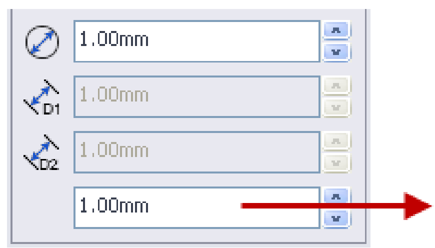
This plane is actually the sketch plane to create the insert's profile.

: Edge or vertices are selected as reference to define the circular insert.

If a vertex is selected, a circle centralized at this vertex is created.

If an edge is selected, a circle centralized at the middle point of the edge is created.

For multiple vertex and edge selection, the selected entities are used to define the minimum circle profile and its position.



: Diameter

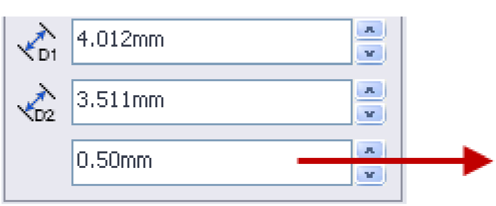
Clearance value

The default circle position and diameter are calculated based on the selected, however, both position and dimension of the circle could be changed.

Rectangle: For rectangular sub-insert

: Edge or vertices are selected as reference to define the rectangular insert.

If a vertex is selected, a rectangle centralized at this vertex is created.
If an edge is selected, a rectangle centralized at the middle point of the edge is created.
For multiple vertex and edge selection, the selected entities are used to define the minimum rectangle profile and its position.



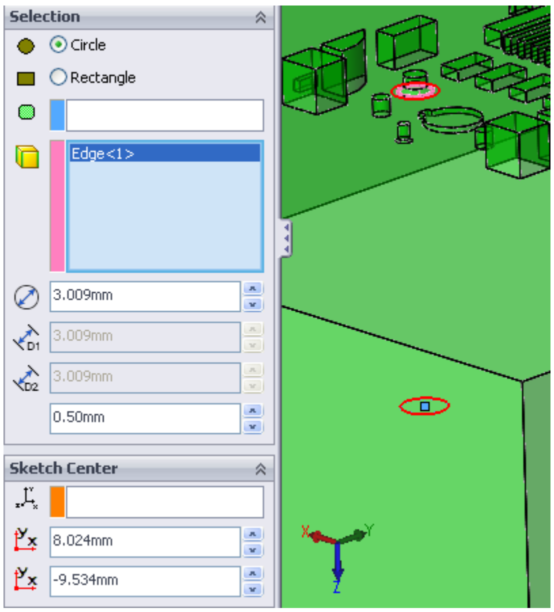
D1: Length of the rectangle
D2: Width of the rectangle

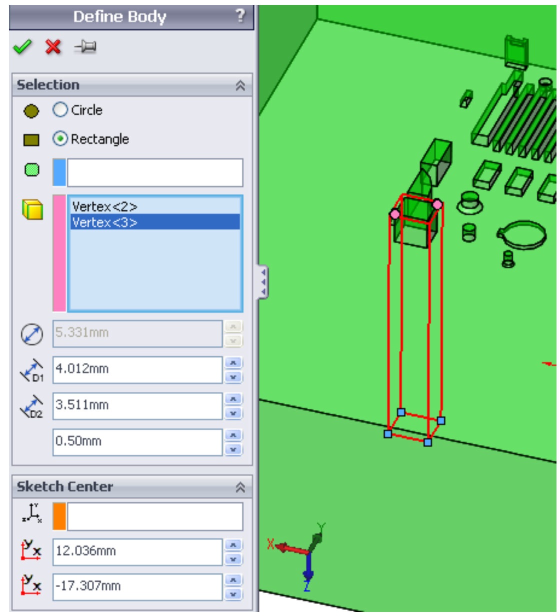
Clearance value

Sketch Center

: Reference coordinate system to define the insert's center.

: Display the position of the center of the Sub-insert relative to the coordinate system. If the coordinate system is not selected, the part coordiate system is selected. The center of the sub-insert can be changed by changing the setting here.

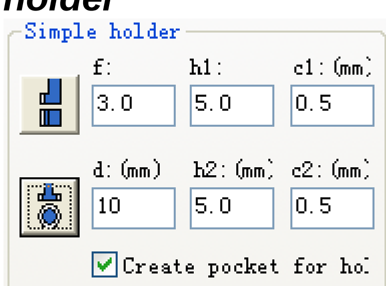


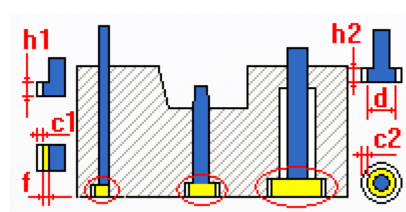


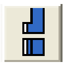
5.3 holder

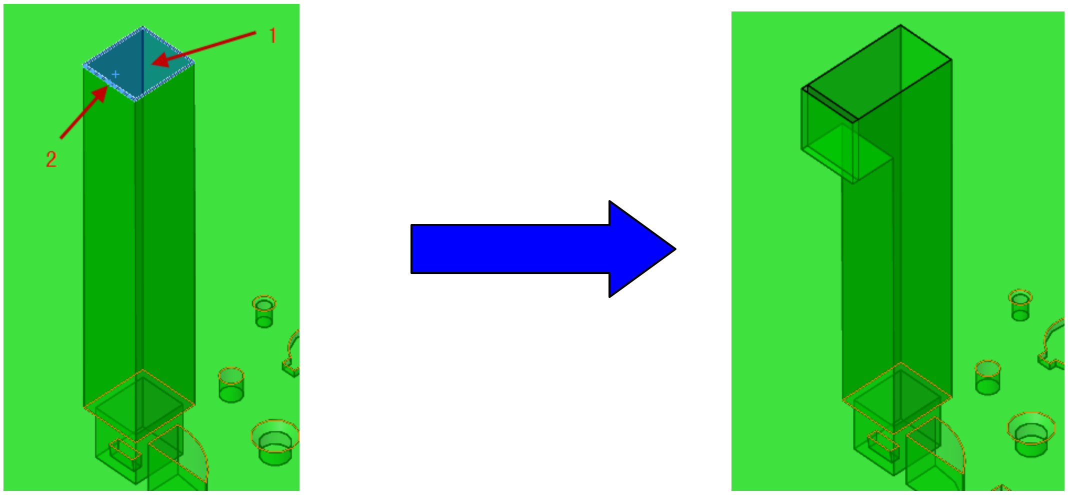
When activate the holder tab, you will find two buttons for simple holder creation, the circular one the rectangular one. If **Create pocket for holder** is checked, a pocket will be created as well to fit the created subinsert.

Simple holder

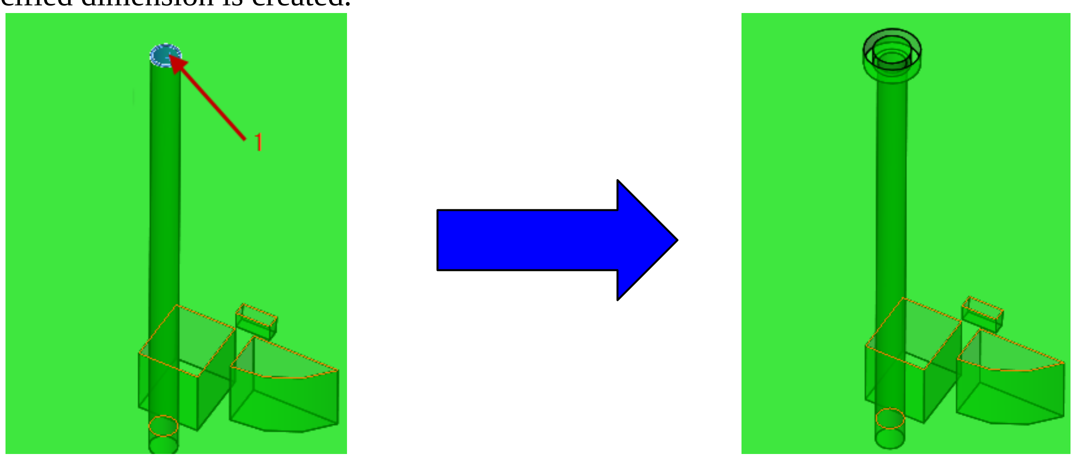




: Select the rectangular face and an edge on it, click this button, a rectangular holder is created. The holder size is decided by f, h1 and c1 as the picture shown. If **Create pocket for holder** is checked, the corresponding pocket will be created as well.



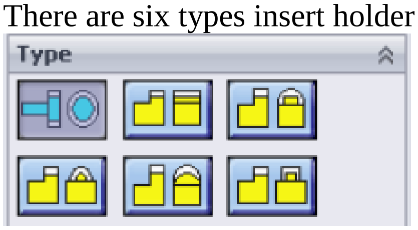
: Select the bottom face on the insert body, click this button, the circular holder with the specified dimension is created.

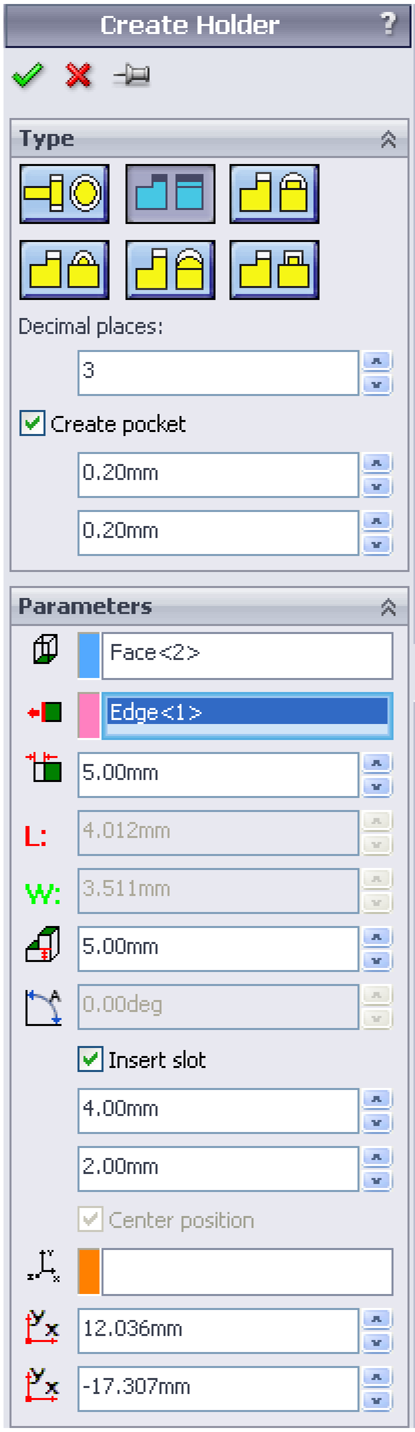


Advanced

Some kind of insert holders need more parameters to control its profile and size, those can be completed in Advanced.

There are six types insert holder provided in this page.





Create Pocket： If it is checked, the pocket for the insert holder will be created.

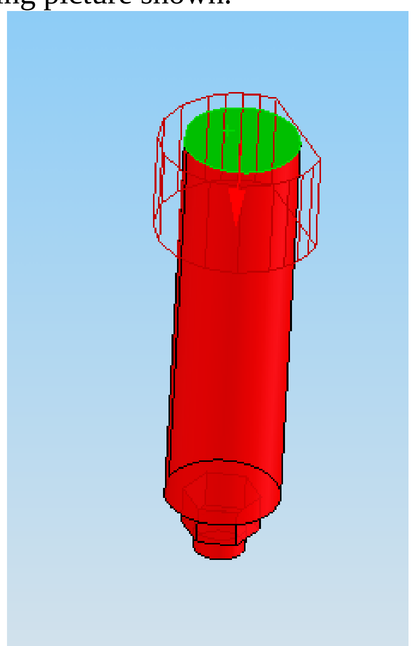
: Circular holder

： Bottom face on the insert body

： Diameter for the holder

： If the ejector head need to cut at single or both sides, this value should be set to be smaller than the diameter.

Single side: If checked, ejector head is cut at single side as the following picture shown.



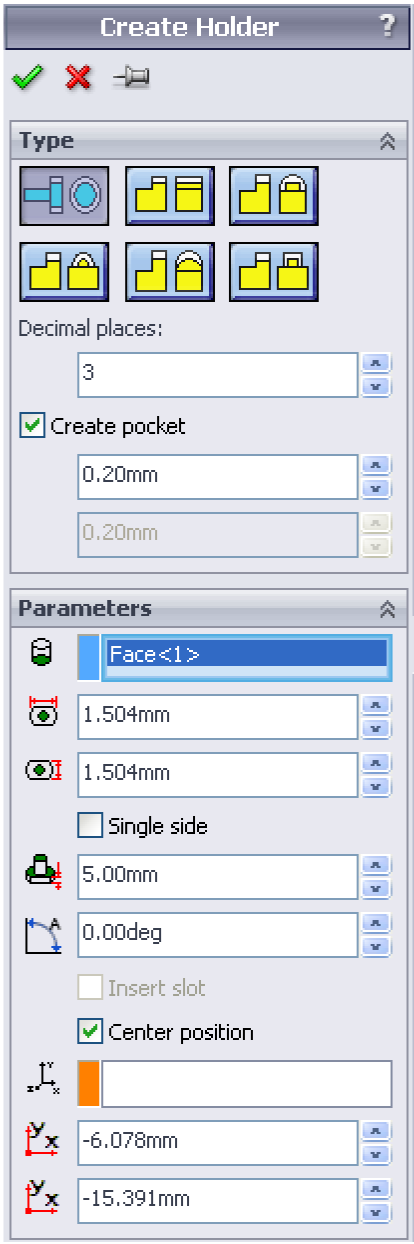
: Height of the holder

: Rotation angle of the holder

Center Position

: Select the reference coordinate system. If not selected, the part coordinate system.

: Value for the insert center relative to the selected coordinate system.

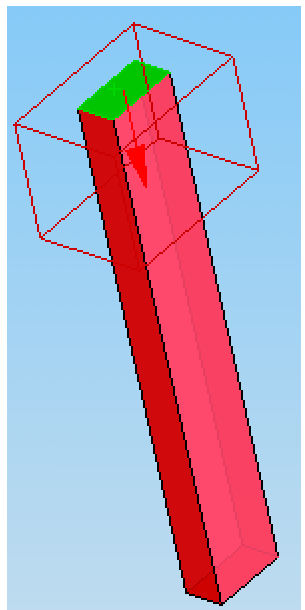


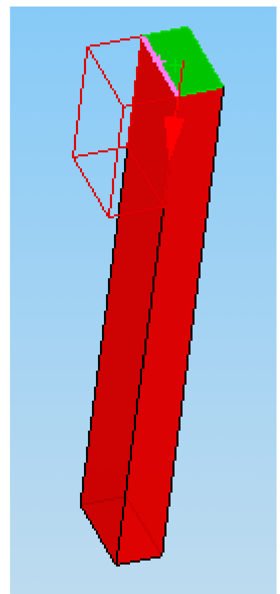
: Rectangular insert holder.

: Bottom face on the insert body

: Select a linear edge as reference to offset the flange, if nothing selected, the rectangular holder can be defined by length and width.

The pictures for those two situations are shown below. The design type depends on the selections.





If reference edge is not specified, user can input the length and width values directly to form the rectangular holder. By default, the holder center is located at the face center.

L: : Holder length

W: : Holder width

: Holder height

: Rotation angle

If a reference edge is selected

: Linear reference edge

: Offset distance

: Holder height

: Rectangular flange and circular pocket for the flange. Small flange type.

: Bottom face on the insert body
: Reference edge
: Pocket diameter for the flange
: Flange Height

: Circular flange and circular pocket for the flange. Small flange type.

: Bottom face on the insert body
: Reference edge
: Pocket diameter for the flange
: Flange Height

: Circular flange and circular pocket for the flange. The flange is tangent to the pocket, clearance at one side.

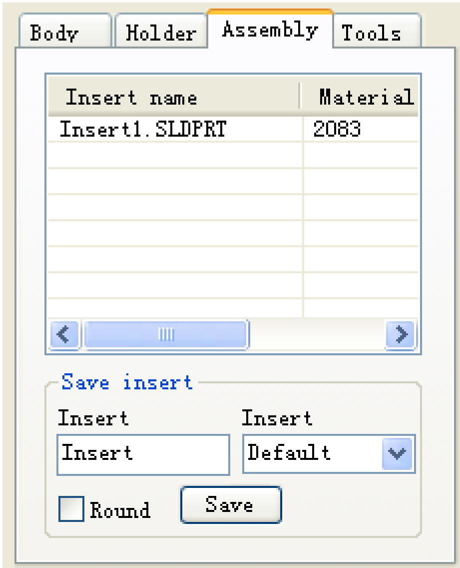
: Bottom face on the insert body
: Reference edge
: Flange Height

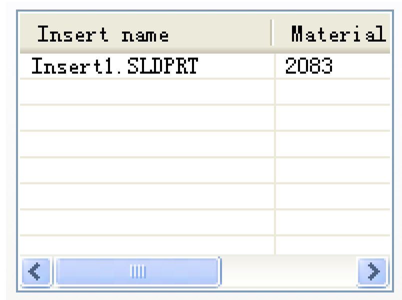
: Rectangular flange and rectangular pocket for this flange.

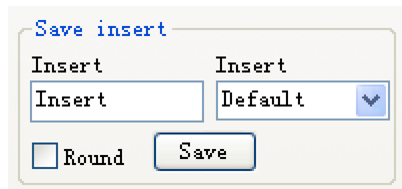
: Bottom face on the insert body
: Reference edge
: Size for the pocket
: Flange length
: Flange width
: Flange Height

5.4 Assembly

Save the insert body as separate component. By doing this, a component is created and insert into the current assembly model.

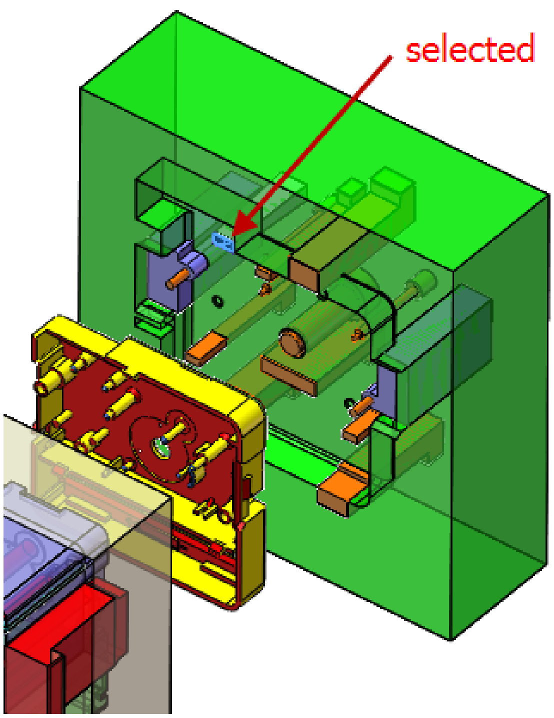




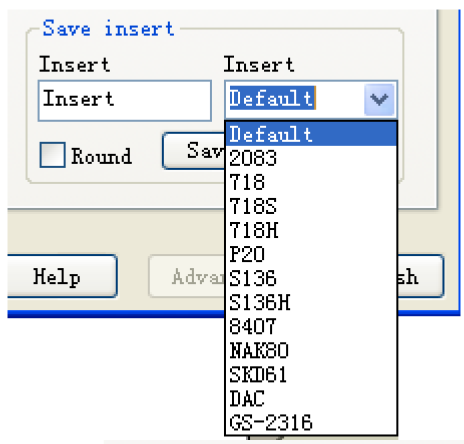


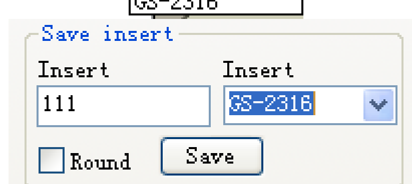
Insert name and material can be specified while saving the insert component.

Under the assembly model, user can pick up one face on the insert body and click  button to complete this function.



Insert name can be input in the text field.





Material can be chosen from the drop down list or input your own material name.

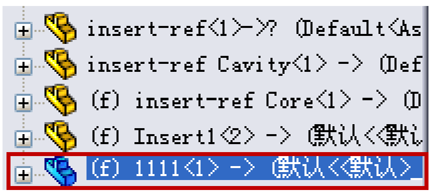
: By default, the insert is marked as rectangle size if this option is not checked.

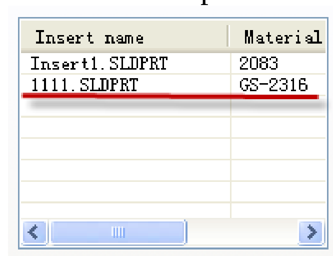
If it is checked, the insert is marked as round shape, its raw material size can be calculated by diameter and length.

For rectangular size, the insert's raw material will be calculated by length, width and height. Those information will be used in BOM.

Click 

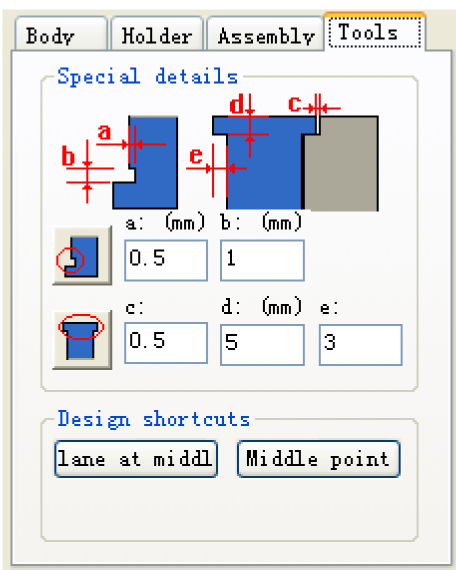
, the system will save the selected insert as a new component.

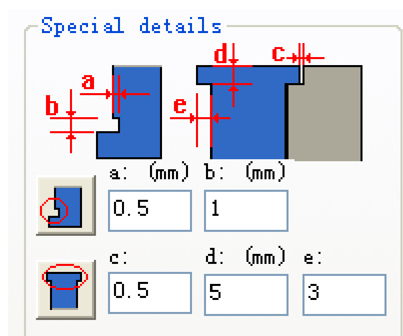




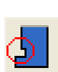
You can see the newly added insert component in the list and tree.

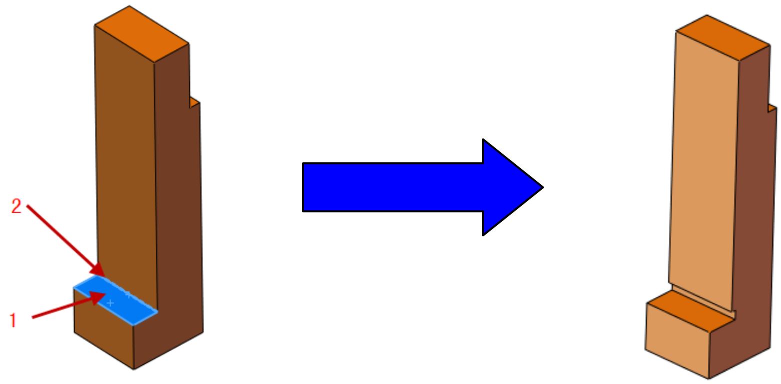
5.5 Tools



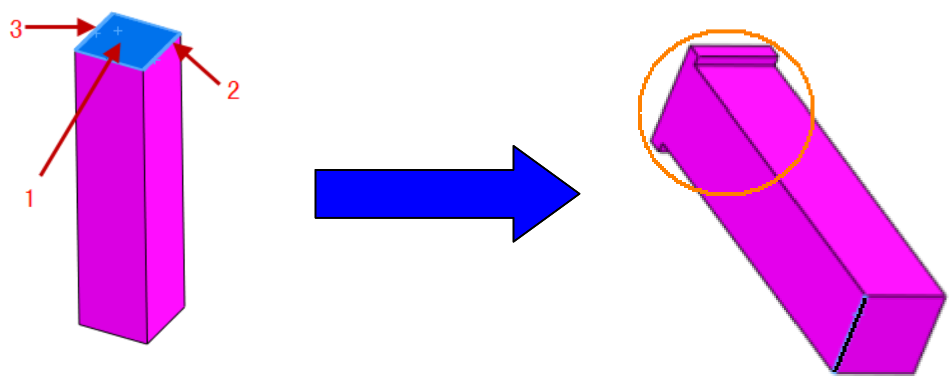


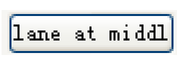
Some tools for special needs in insert design.

: Select a planar face and a linear edge to create the slot. For precision insert, this is a typical detail.

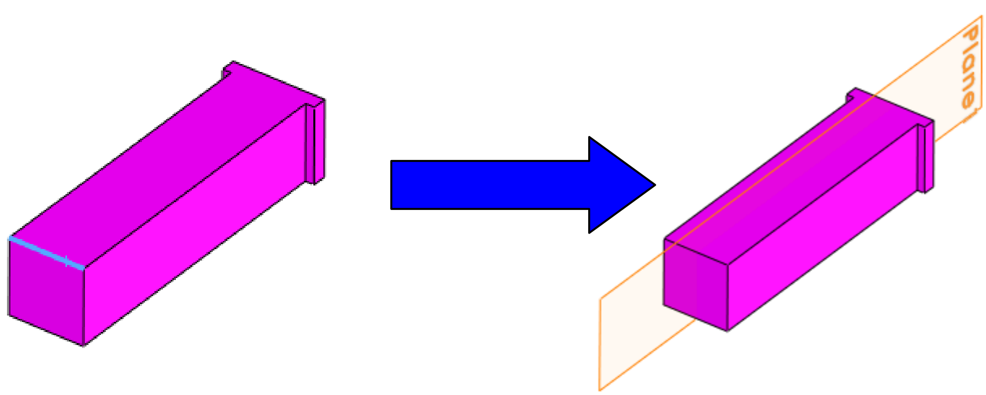


: Double flange on the insert. Select the bottom face and two opposite edges to design the insert flanges.





: Select a linear edge, a reference plane is created passing throght the middle point and perpendicular to the selected edge.



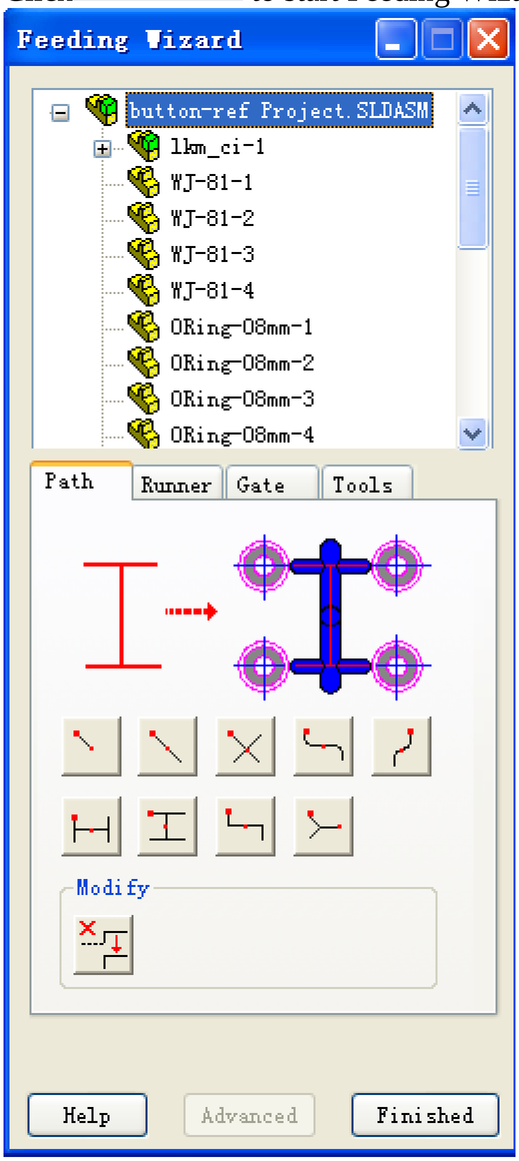
: Select two vertices to create a 3D Sketch point between them.

Chapter 6. Feeding Wizard

Feeding wizard is for runner, gate design, usually it is used after Layout. In the option of Layout, creation of a * Runner.sldprt file in * Project.sldasm is set as default, and this file is used for runner design.

Click 

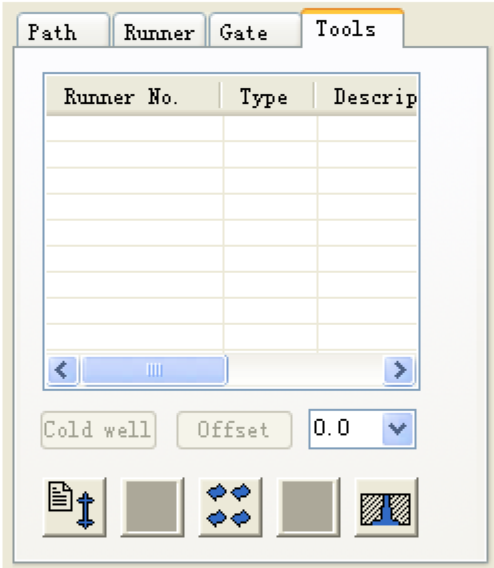
to start Feeding Wizard.

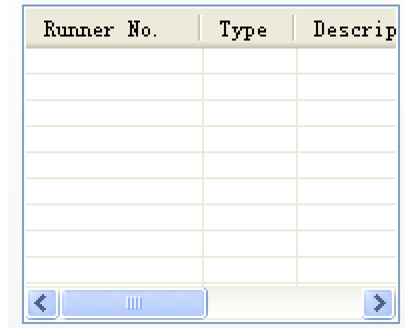


6.1 Component Navigator

Please refer to the Component Navigator in the insert wizard.

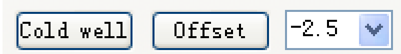
6.2 Tools





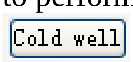
List the existing runners with name, type and size information.

Quick tools used to do some modifications.

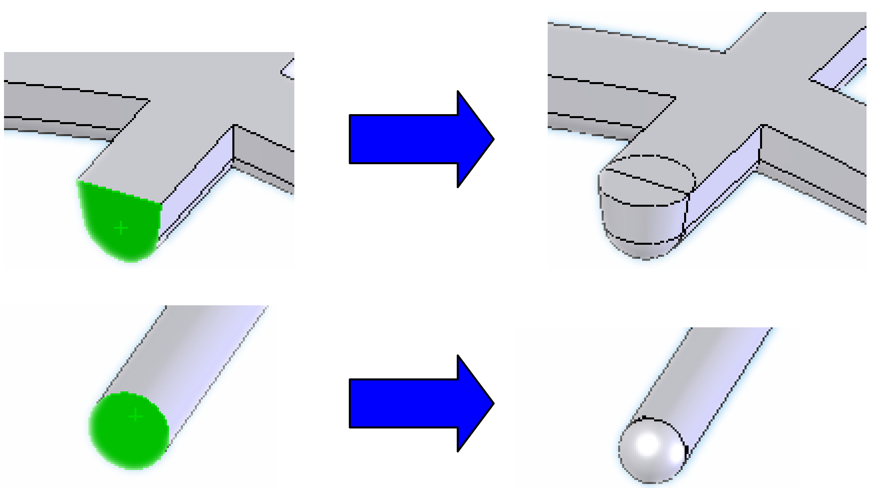


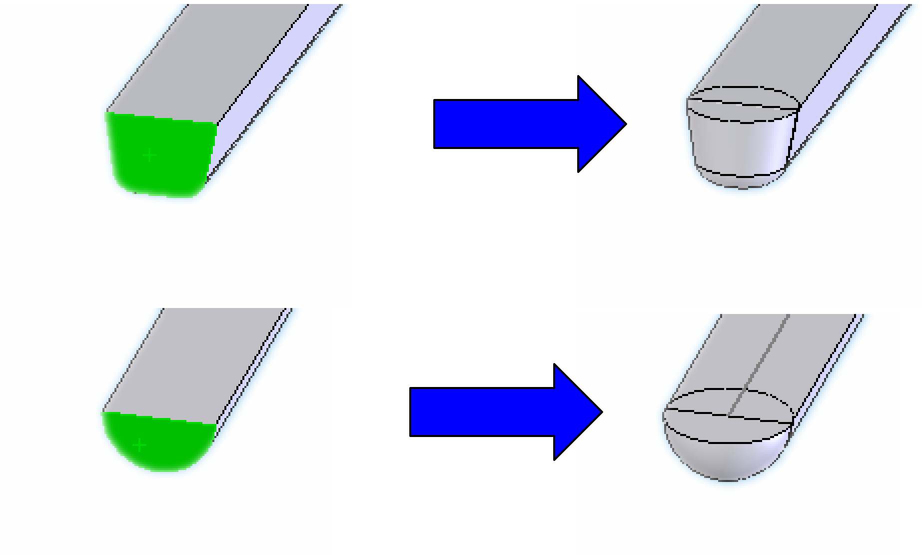
Used on part model only

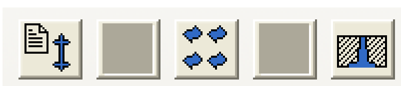
： Select the face required to extend or shorten, set the offset distance, click this icon to perform.



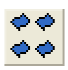
： Select the end faces on runner that will add cold well on them, click this icon to perform. Cold wells of Runner with different cross-section are shown as follows.

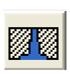


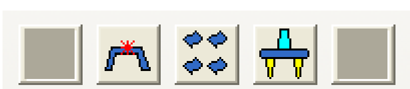


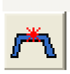


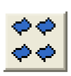
:

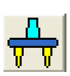
:

:



:

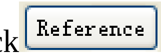
:

:

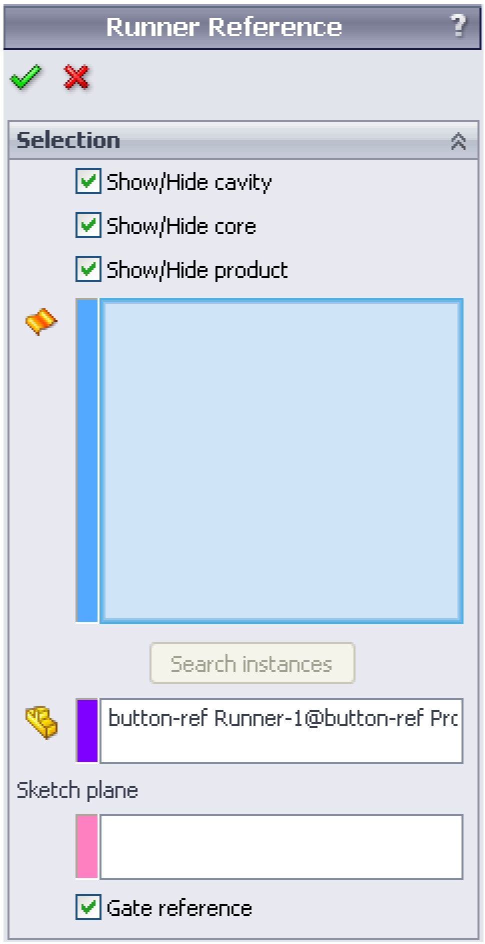
References

3DQuickMold doesn’t create feed system at the assembly envioroment directly, instead, it designs the runner and gate on part file which is prepared when the layout was created.

Reference helps to add the faces from the assembly file that is related to define the position of the gate to the runner file. The runner is designed then. This method helps to convert the work done from assembly to part, it can improve the design performance greatly.

Activate the Tools page, click 

, the following page is displayed.



Show/hide cavity: Show or hide cavity in the current assembly.

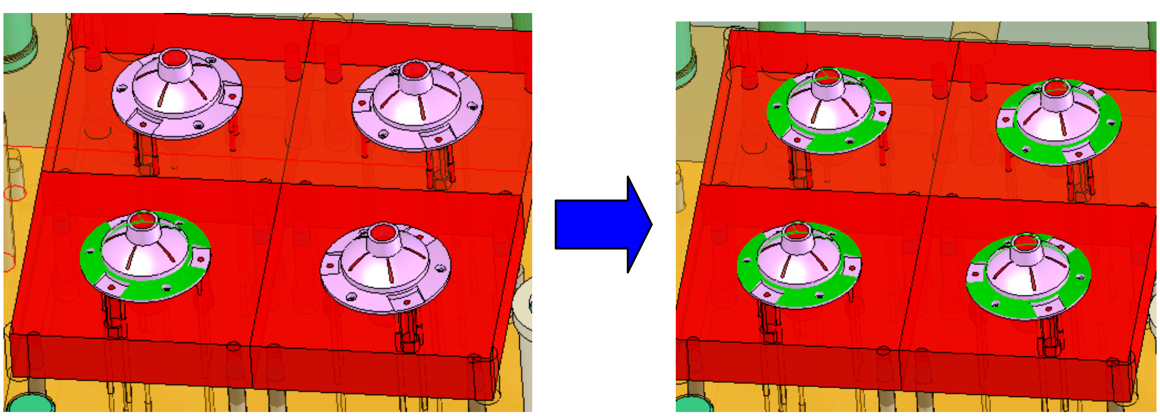
Show/hide core: Show or hide core in the current assembly.

Show/hide product: Show or hide product in the current assembly.

Reference faces: select the face to be the reference face for the runner design

Search instances: for multi-cavity layout, select a face on product/core/cavity, click this icon, 3DQuickMold will automatically add the same face from the other cavities to Reference faces

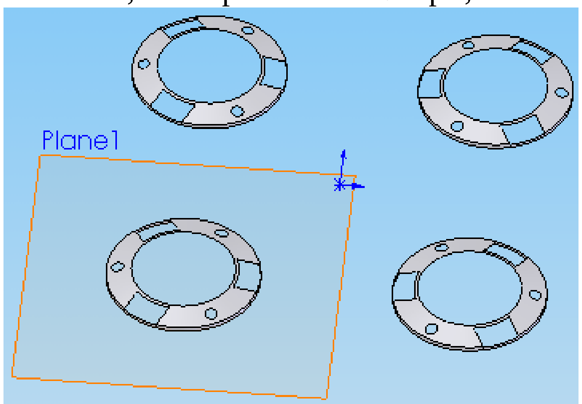
Select a face on the product, click **Search instances,** the corresponding faces on other instances are selected.



Runner part: Runner will be saved as a separate part, if the runner part is created before, the runner component is selected automatically, otherwise, user can select a new one.

Sketch plane: Select a plane as reference for the main runner creation.

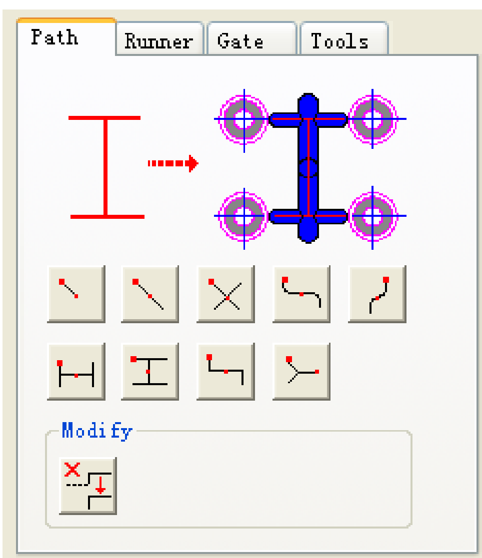
Click OK, then open * runner.sldprt, the result as follows

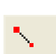


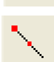
6.3 Path

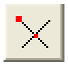
Some effective tools to define or modify the runner path.

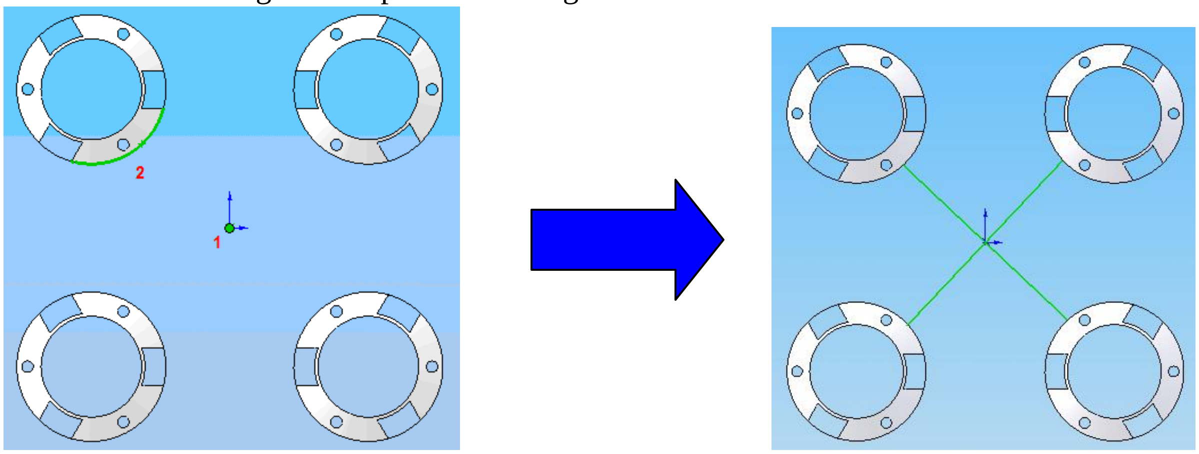
The red dot on the icon button represents the selections, two red dots means two selection required. Small red dot should be selected first, followed by the bigger one.



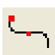
: Select the orgin and a point on an edge to create one sketch line.

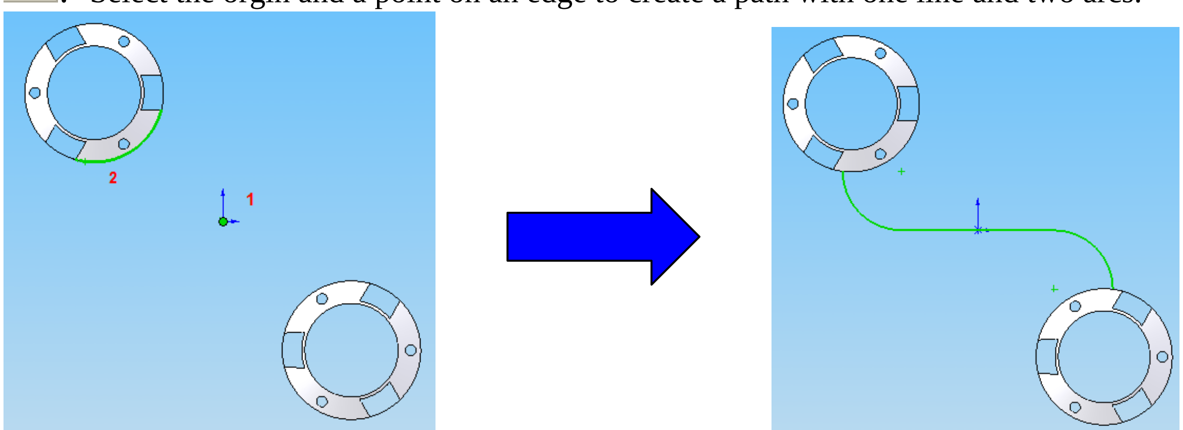
: Select the orgin and a point on an edge to create one sketch line, The selected origin is located at the middle point of the line created.

: Select the orgin and a point on an edge to create two sketch lines.

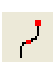


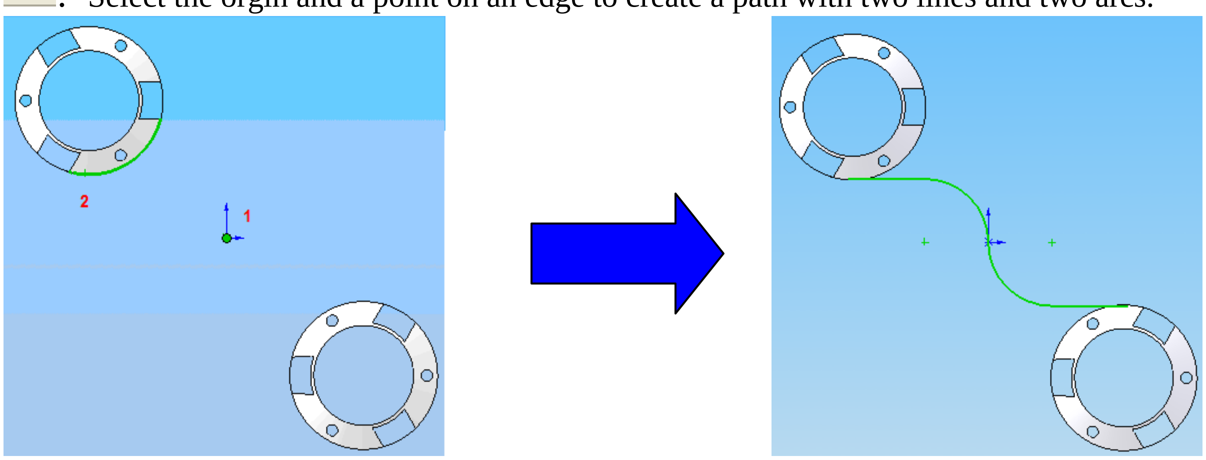
The selected origin is located at the middle point of the line created.

: Select the orgin and a point on an edge to create a path with one line and two arcs.

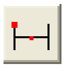


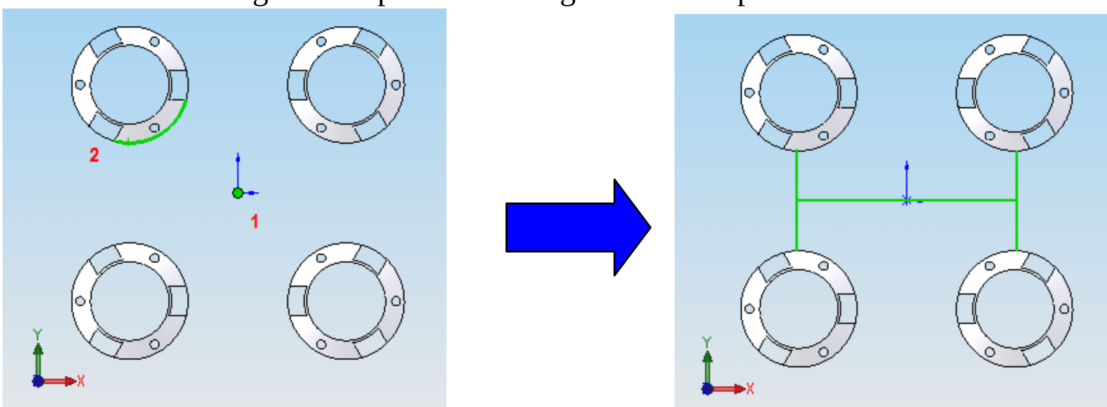
The line's middle point is located at the selected origin. Two arcs are tangent to the line.

: Select the orgin and a point on an edge to create a path with two lines and two arcs.

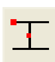


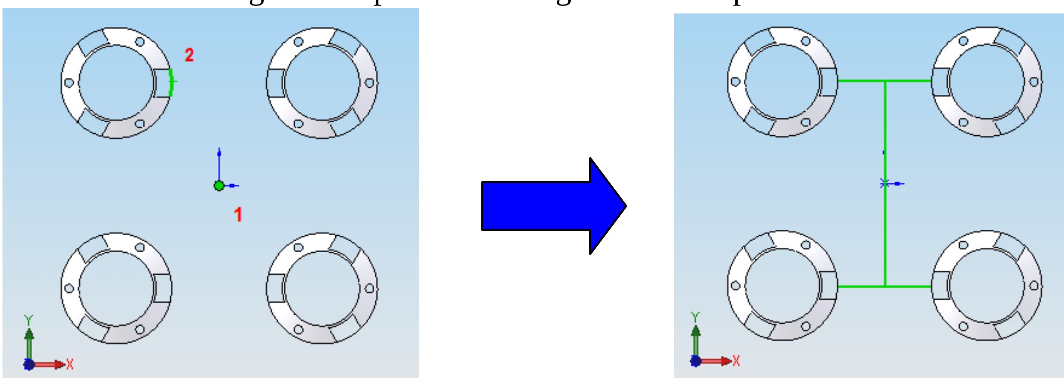
The two arcs are tangent to each other and tangent to the two lines created.

: Select the orgin and a point on an edge to create a path with three lines.

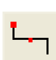


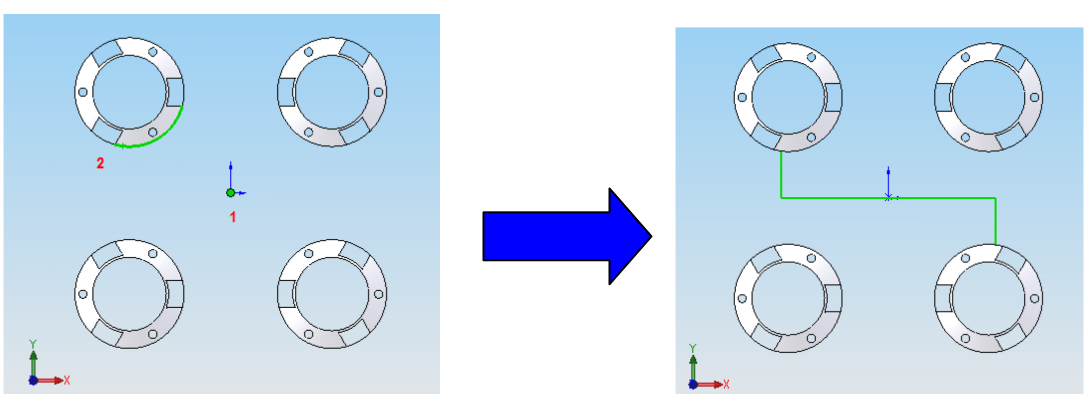
The line passing through the origin is perpendicular to the other two lines created.

: Select the orgin and a point on an edge to create a path with three lines.

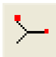


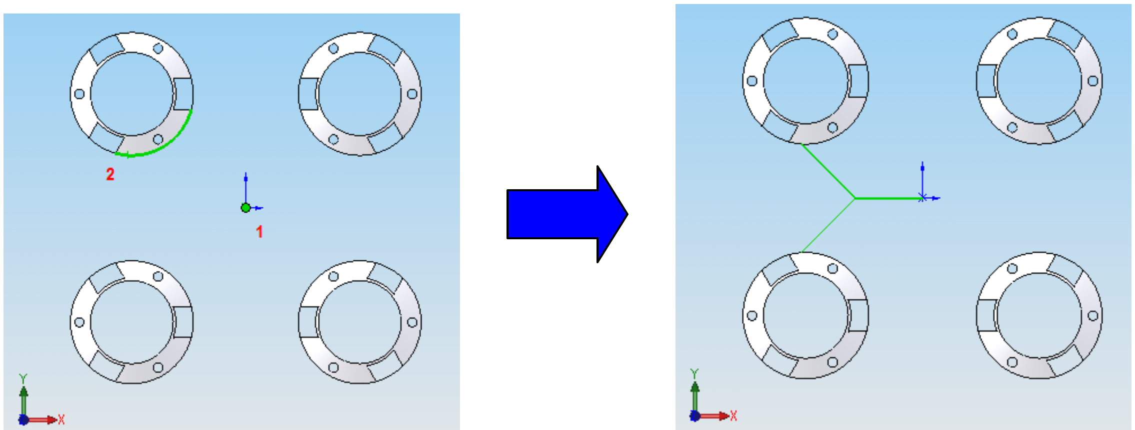
Similar to the previous one, but different orientation.

: Select the orgin and a point on an edge to create a path with three lines.



The sketch consists of three lines, and symmetrical about the Origin.

: Select the orgin and a point on an edge to create a path with three lines.

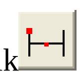


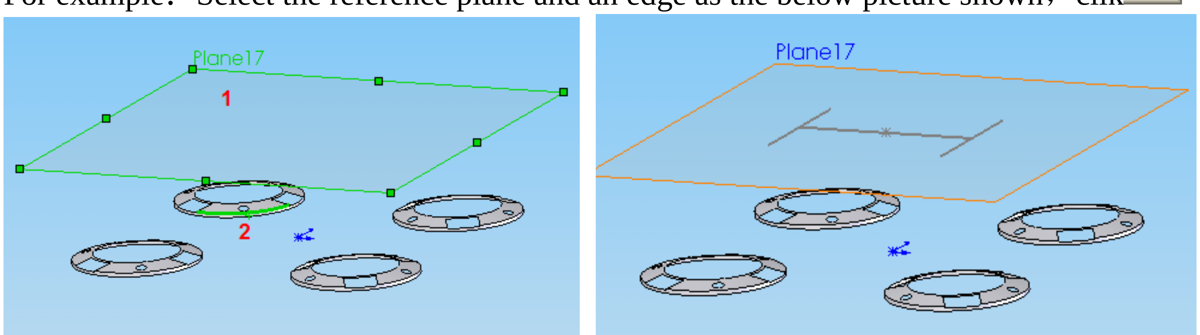
Additional notes:

If the first selection is origin, the sketch will be created on the front plane.

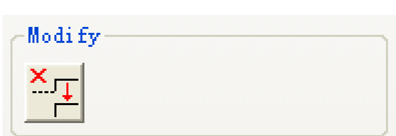
If the first selection is a sketch point, the newly created runner path will be added onto the existing sketch that owns the sketch point.

If the first selection is a reference plane, it would be the sketch plane for the runner path.

For example: Select the reference plane and an edge as the below picture shown, clik

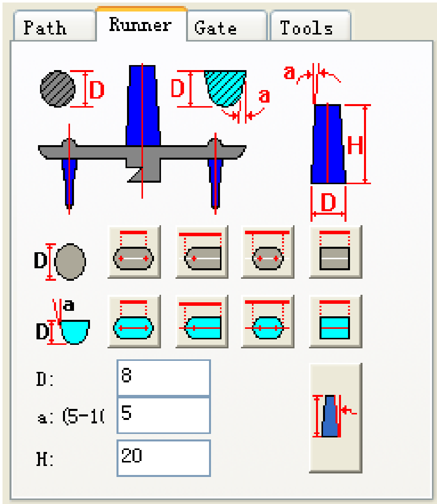


A runner path sketch is created on the selected reference plane.



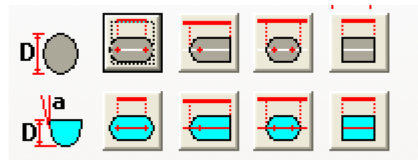
Select a sketch line, click this button to delete it quickly. Using this function, no need to edit the sketch.

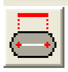
6.4 Runner

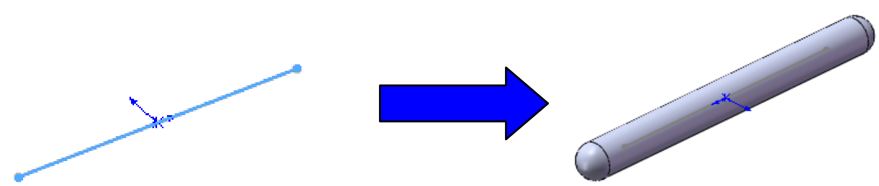


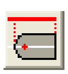
Horizontal runner

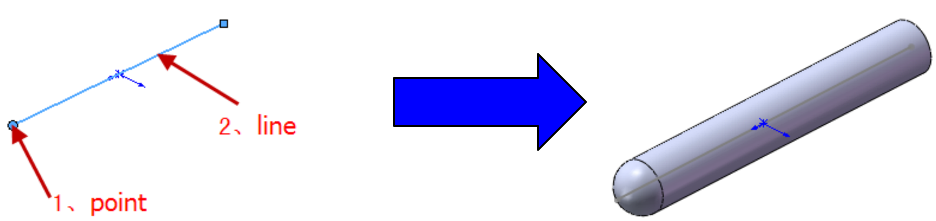
Some tools used for runner creation.

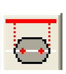


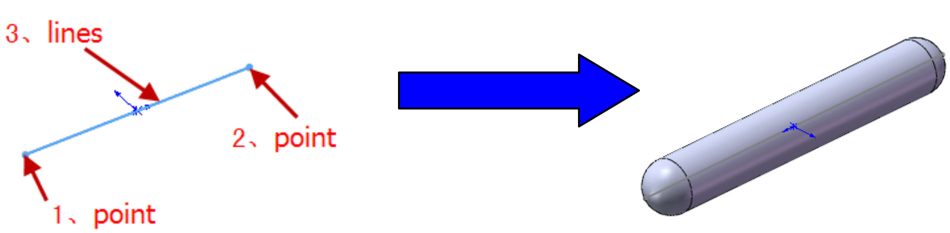
: Select a sketch line or connected sketch lines, circular runner and cold well at two ends are created.



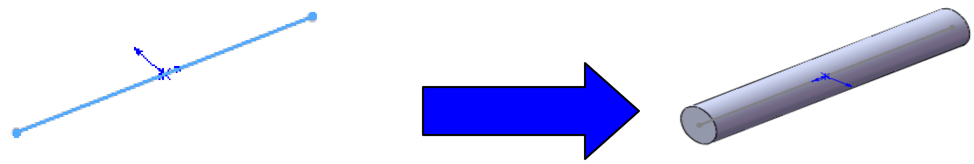
: Select a sketch point and a sketch line or connected sketch lines, circular runner and cold well at the selected point are created.



: Select two sketch points and a sketch line or connected sketch lines, circular runner and cold wells at two selected points are created.



: Select a sketch line or connected sketch lines, circular runner is created.



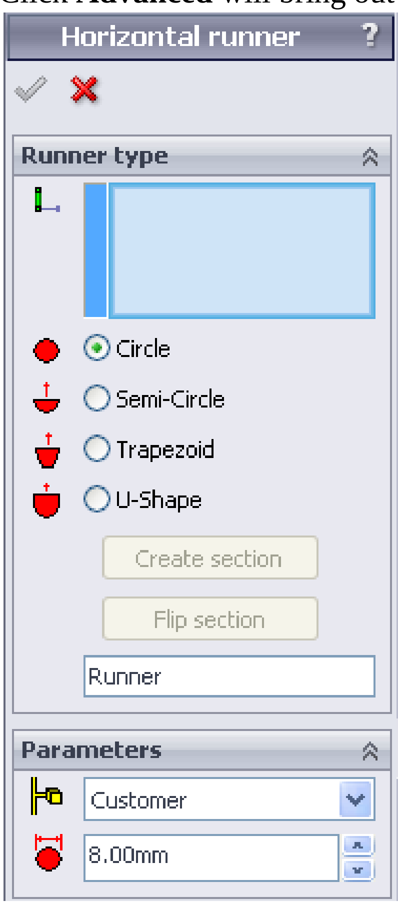
: Select a sketch line or connected sketch lines, U-shaped runner and cold well at two ends are created using the parameter D and a.

: Select a sketch point and a sketch line or connected sketch lines, U-shaped runner and cold well at the selected point are created using the parameter D and a.

: Select two sketch points and a sketch line or connected sketch lines, U-shaped runner and cold wells at two selected points are created using the parameter D and a.

: Select a sketch line or connected sketch lines, U-shaped runner is created.

Click **Advanced** will bring out the Horizontal runner page.



: Select sketch segments to define the runner path, segments selected should not be crossed and must be continuous.

The shapes of runner are categorized as follows::

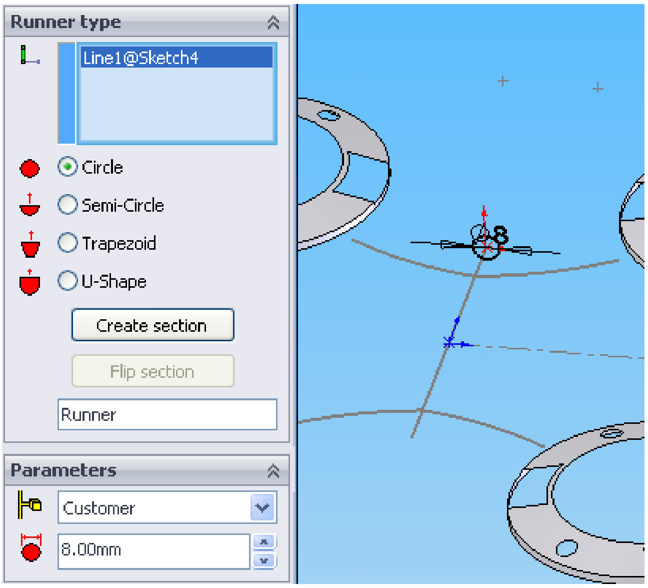
1. : Select circle as cross-section of the runner.
Parameters : define the diameter for the runner

2. : Select semi-circle as the cross-section of the runner .
Parameters : define the radius for the runner

3. : Select trapezium as the cross-section of the runner.
: define the width of the top face of the trapezium
: define the depth of the trapezium
: define the taper angle of the trapezoid
: define the corner radius for the bottom of the trapezium

4. : Select U-shaped as the cross-section of the runner
: define the radius for the U-shape runner
: define the taper angle of the U-shape

Create section: After selecting the runner path and the cross-section of the horizontal runner, click this button, a sketch will be created. The  selection will be hidden, "Flip section" button will be activated.



Flip section: Adjust the direction of the cross-section of runner. Click the function and the sketch will rotate by 180°.

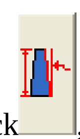
Runner naming prefix: Name the runner

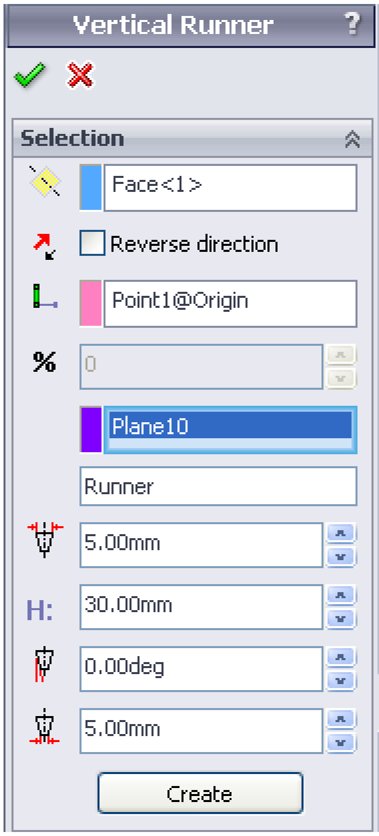
Click Ok, a runner using sweep feature is created.

By pre-selecting an existing runner as reference, when the page pops out, the reference runner type and dimension are used as default.

Vertical runner

Used to design the runner that is perpendicular to the PL surface, for example main runner

Click , the following dialog pops out.



: Select a reference plane to be the sketch plane of the vertical runner, an arrow will appear and indicate the direction for the vertical runner.

: If the direction is not correct, click this function to flip the direction.

: Pick up a point to be the centre of the vertical runner sketch, sketch point or sketch segment can be selected, if an segment is selected, % will be active.

%: Adjust the the position ratio of a point on the selected segment.

End Reference: The ending plane of the runner.

Runner naming prefix: Name the runner

: Define the diameter of the vertical runner

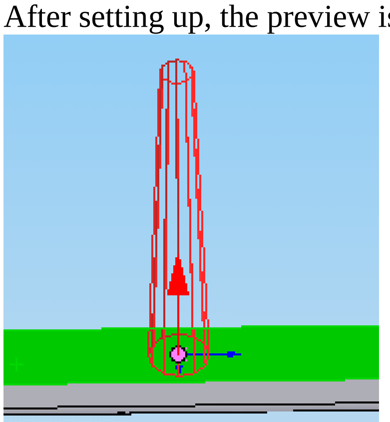
H: : Define the height of the vertical runner

: Define the taper angle of the vertical runner

: Define the diameter of the other side of the runner

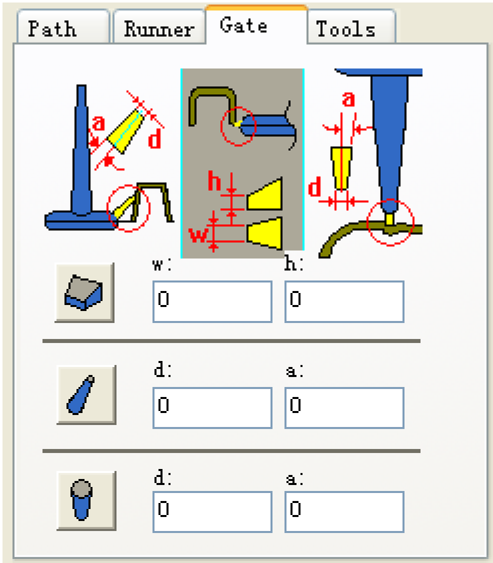
For the latter four setting, complete any 3, the remaining one will be computed automatically.

After setting up, the preview is shown.



Click **Create** and the feature tree will appear an extrude feature.

6.5 Gate design



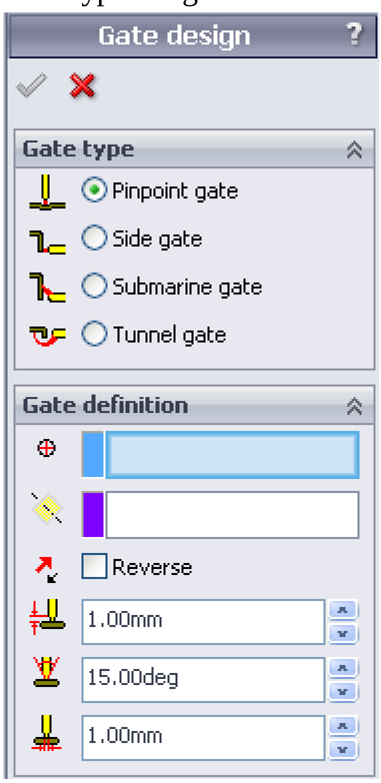
:

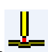
:

:

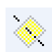
Click **Advanced**, the gate design dialog pops out as the following picture shown.

Four types of gate are availiavle here : Pinpoint gate, Side gate, Submarine gate and Tunnel gate

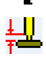


1.  Pinpoint gate

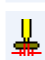
 : Select line or face to define the position of the gate

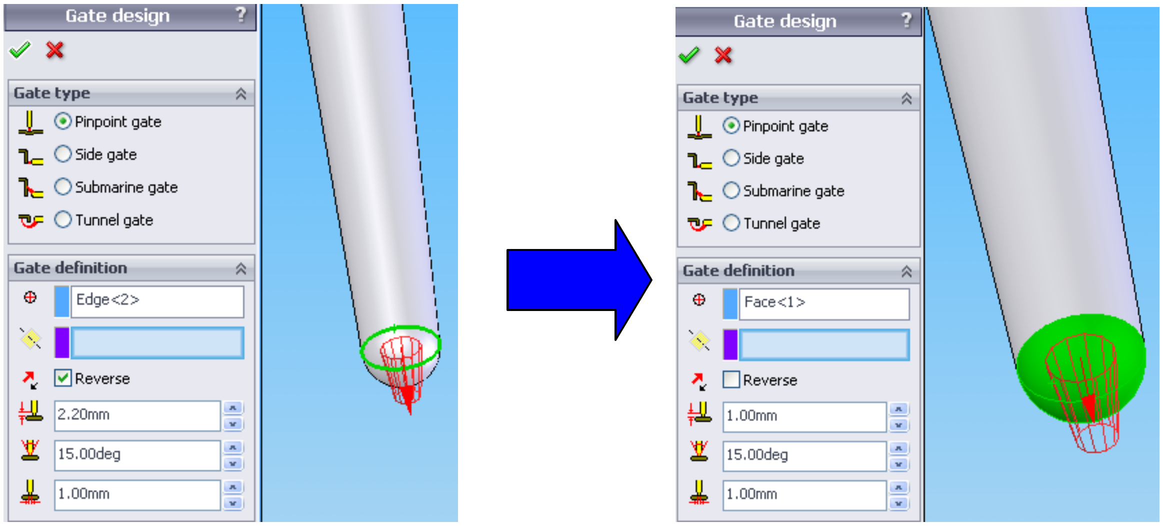
 : Select a plane or a line to define the direction

 : Flip the direction of the gate

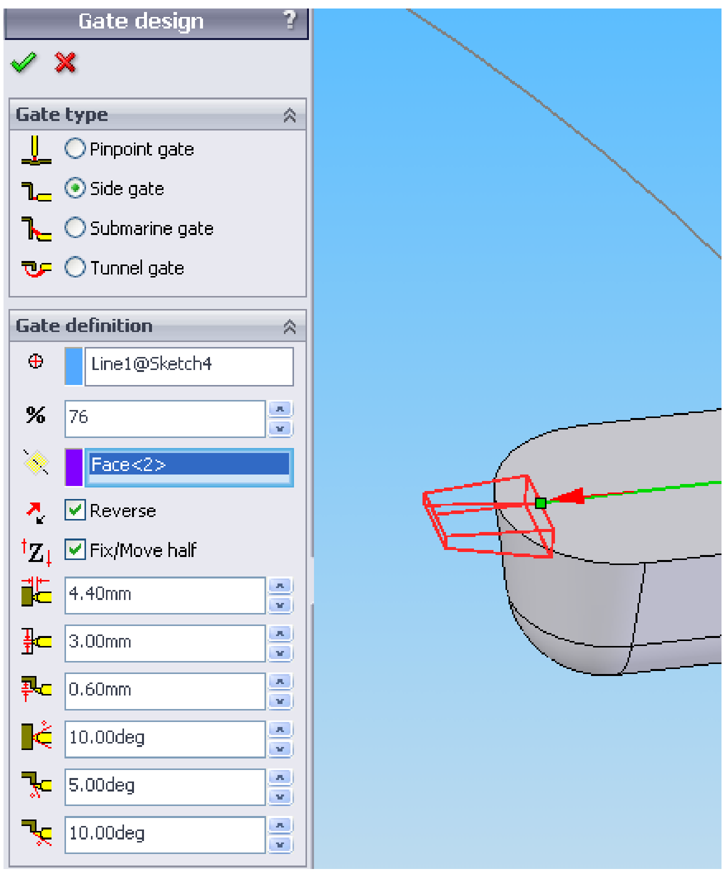
: Define the length of the gate. If a circular edge or planar face is selected, this meaure is the extension from the plane or circle center. If a spherical face is selected, the measure is the extension from the top of the curved surface.

 : Define the taper angle of the gate

 : Diameter of the gate



2. Side gate



:Select a sketch point or segment

: Percentage position

: Plane to define the direction

: Flip the direction if needed

: Fix/Move half: Change the gate location, it could be on fixed half or moving half.

: Define the lenght of gate

: Define the width of gate

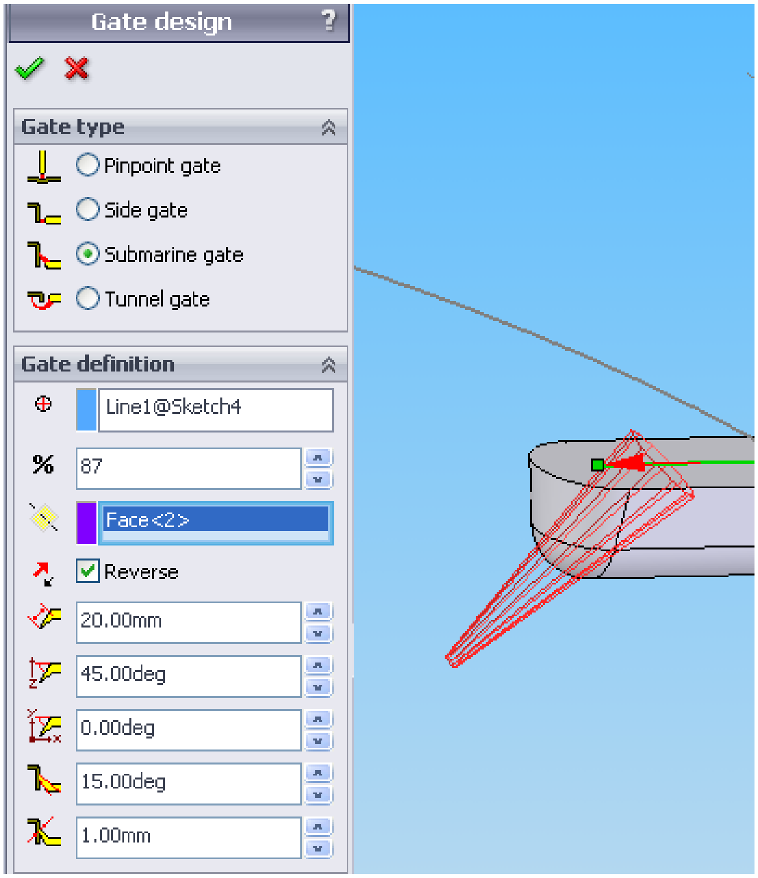
: Define the thickness of gate

: Approaching angle A

: Approaching angle B

: Approaching angle C

3. Submarine gate



: Select gate location

: Adjust the the position ratio of a point on the sketch segment. This is activated when a sketch segment is selected.

: Select a reference plane or a linear edge to define the direction

: Flip the direction of the gate

: Define the length of the gate

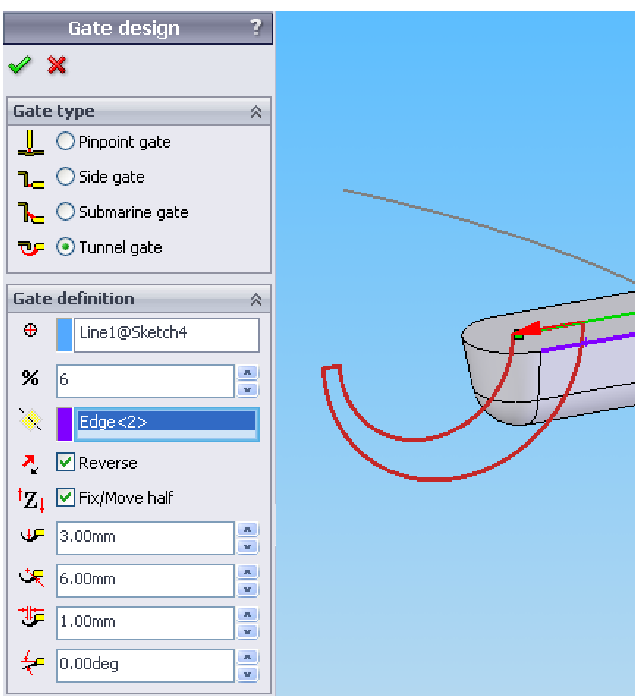
: Define the angle between the gate and XY plane

: Define the angle between gate and runner projected in Z direction

: Approaching angle

: Diameter of submarine gate

4. : Tunnel gate



: Select a sketch point or sketch segment

: Adjust the the position ratio of a point on the sketch segment. This is activated when a sketch segment is selected.

: Select a reference plane or a linear edge to define the direction.

: Flip the direction of the gate

Fix/Move half : Change the gate location, it could be on fixed half or moving half.

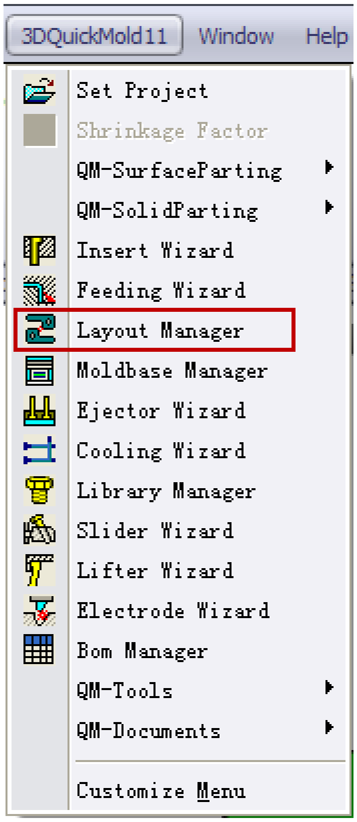
: Inner radius

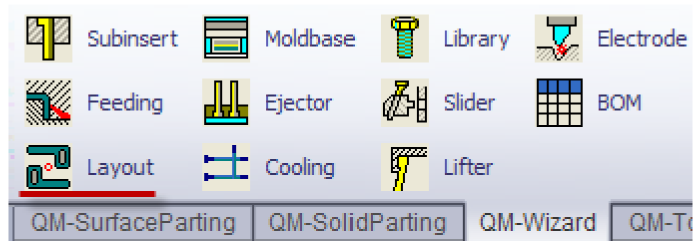
: Outer radius

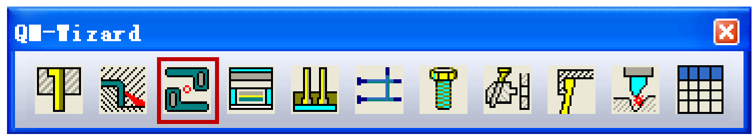
: Offset between two circular center

: Define the angle between gate and runner projected in Z direction

Chapter 7. Layout Manager





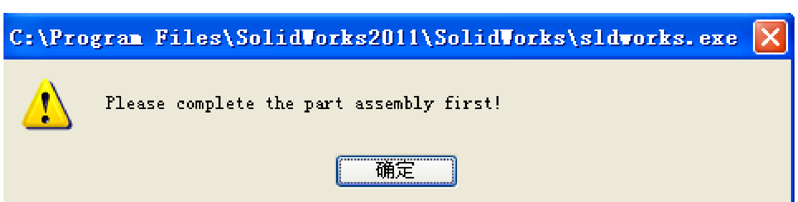


Layout Manager are mainly used for:

- It can arrange the position of cavities of a muti-cavity mold, a * Project .sldasm (top assembly file) is built. This function can be used after the * Assembly.sldasm (product assembly) is built. To enable this function, the plastic part must be activated.
- Edit the number and position of cavities of a *Project.sldasm (top assembly file) with finished layout.

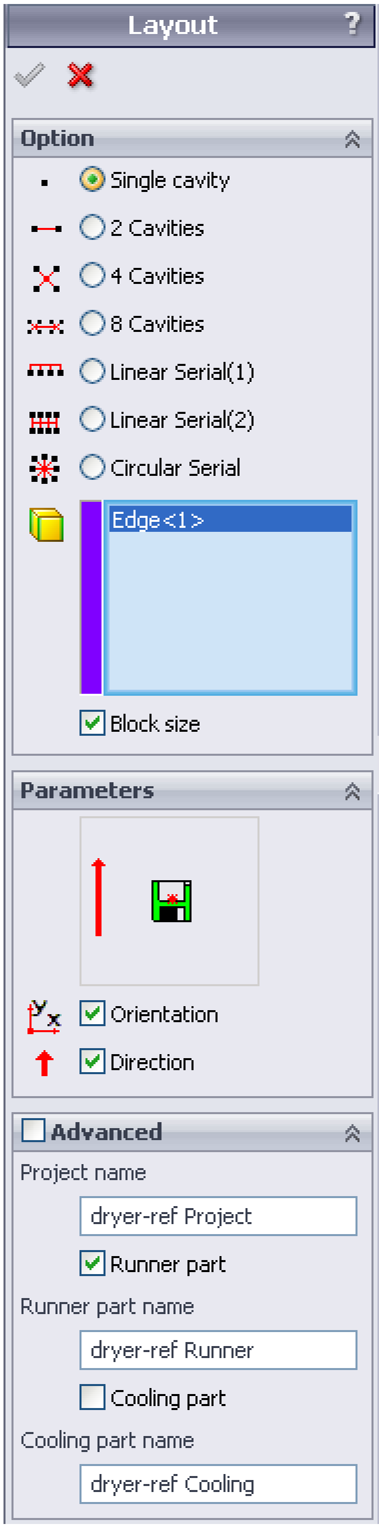
Note: for single-cavity mold, it also requires Layout, otherwise the * Project.sldasm cannot be generated.

If there is not any * Assembly.sldasm file, a warning message will pop out.



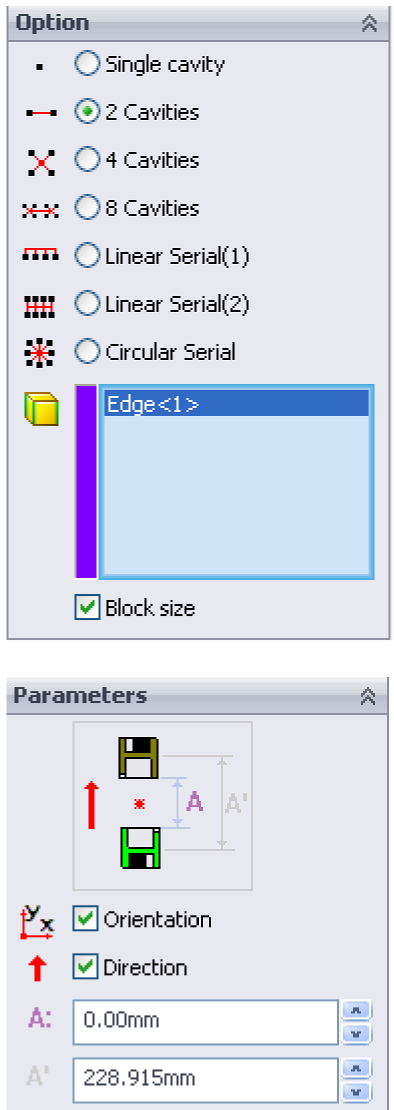
Click 

the Layout Manager pops out. Several patterns are available for selection.



Available patterns as follows:

Single cavity, 2 cavities, 4 cavities, 6 cavities, 8 cavities, Linear Serial (1), Linear Serial (2), and Circular Serial



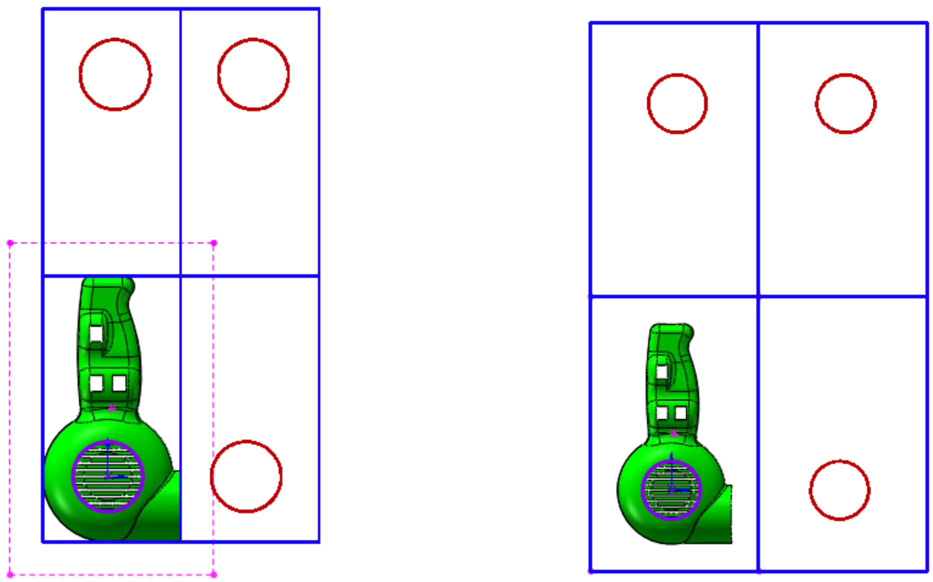
Note： for more cavities layout such as 16, 32, 64, 128 cavities， user can create 8-cavities layout. Then in Edit pattern, use Mirror Copy to finish

Reference edges： They are used for preview purpose only.

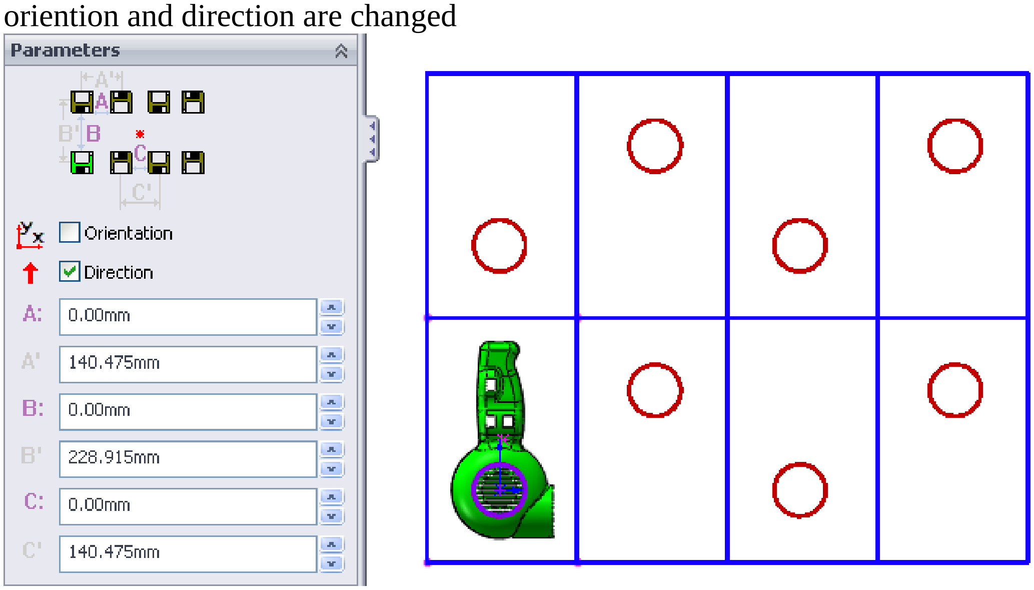
If nothing is pre-selected, 3DQuickMold will select the circular edges on the part as the reference edge. Edge can be added or deleted manually. The quantity of edge enhances the convinence of viewing the orientation of cavities inside the mold but it does not affect the cavity layout.

Block size: By default, the Block size is checked, maximum boundaries of core/cavity block is used to calculate the distance between each cavity.

If Block size is unchecked, maximum boundaries of the plastic part are used to calculate the distance between each cavity.

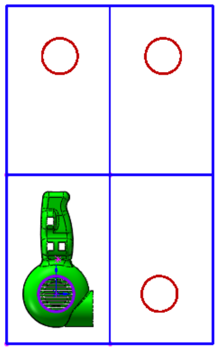


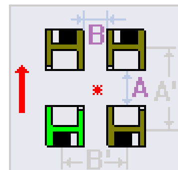
Orientation: Set the relative orientation of the other cavities to the original starting cavity The position of sidecore and the setting of runner should be taken into consideration when the oriention and direction are changed

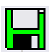


By check or uncheck **Orientation** and **Direction**, the other cavities can be positioned on any of the position in the top, down, left and right of the original cavity

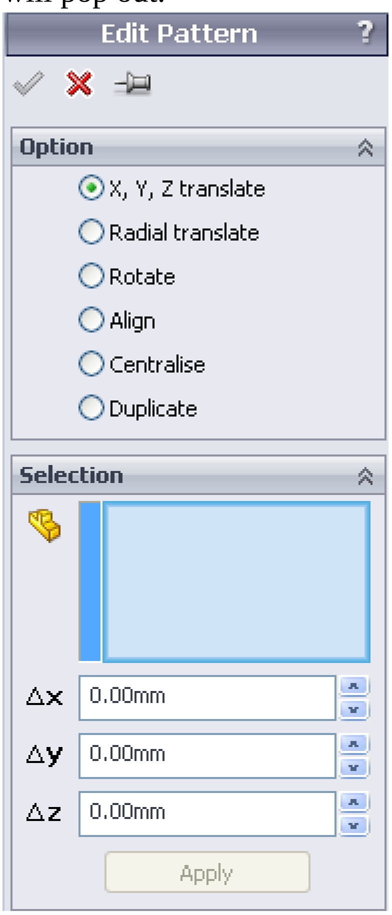
A: （There may be also B, C, E）Parameters to control the layout dimension, please refer to the picture shown on the dialog page.





Tips: change the orientation of the preview to Front view, the original cavity will look like the icon  on the page, you can easily figure out the final layout of multiple cavities.

For product which have been already ***Layout***, if ***Layout*** is click again, the ***Edit Pattern Manager*** will pop out.



X，Y，Z translate： Translate the selected cavity along the X, Y, Z direction

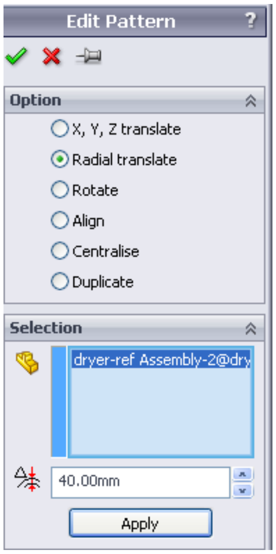
A simple to use is when the mold base is inserted, Edit Z to alight the parting surface on the mold base.

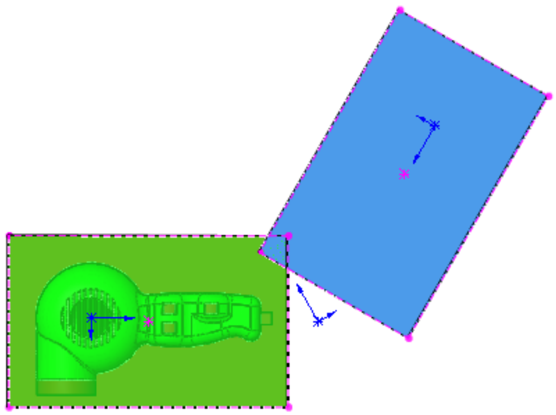
Radial translate： Move the selected cavity along the line joining the center of the selected cavity and the center of the layout.

Select the required cavity, enter the translation distance.

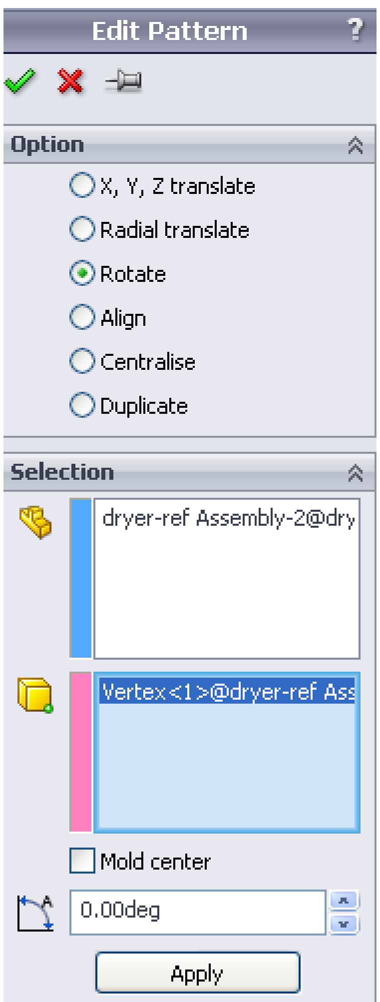
Note: The value entered in the field will return to 0 after clicking ***Apply*** as the value is the relative value. When the value has returned to 0, no change will appear if ***Apply*** is clicked.

If the value entered is not suitable, enter the negative value to undo the change.





Click **Apply** once the setting is done, the cavity position will update.



Rotate: Rotate the selected cavity. Different situations will depend on Mold Center's state.

- **Mold Center** is unchecked

1. Rotate the selected cavity about a selected point and perpendicular to the Z-axis.
2. If several points are selected, the rotation center will be the center of the polygon formed by the points, the cavity is rotated perpendicular to the Z-axis.
3. If no point is selected, the cavity is rotated about its center

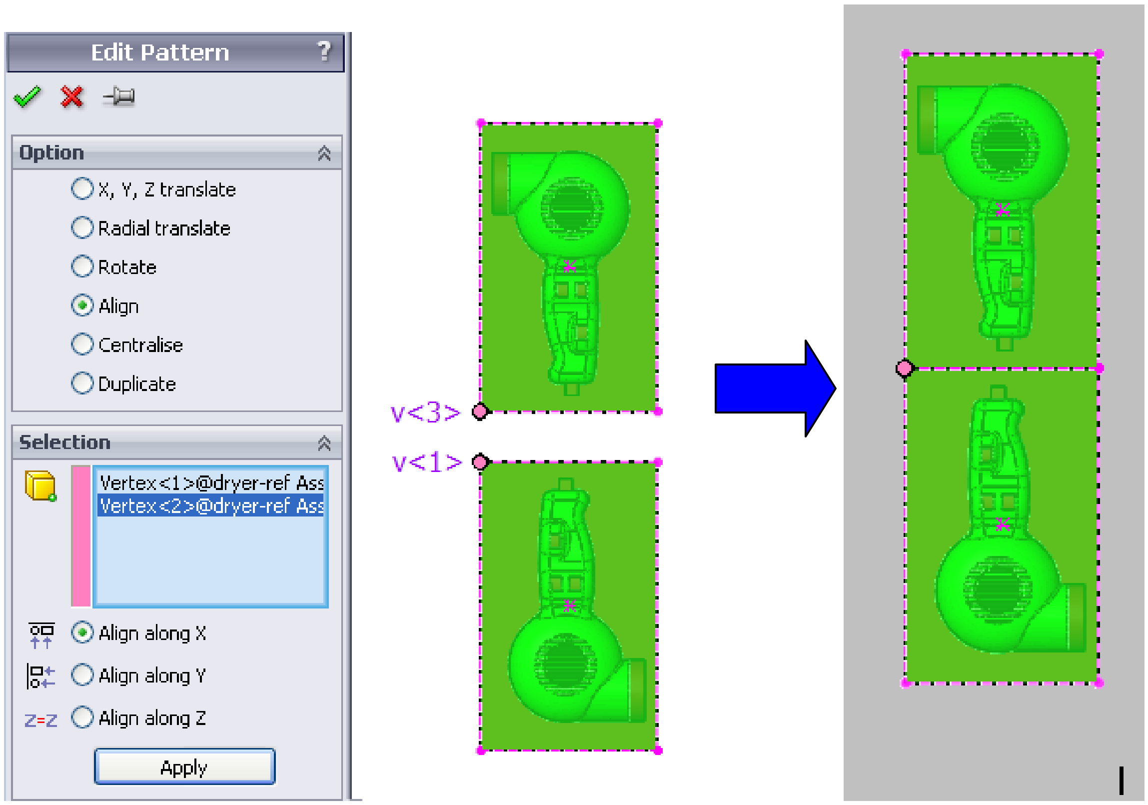
- **Mold Center** is checked

The cavity is rotated perpendicular to the Z-axis about the Origin of the generated * Project.sldasm.

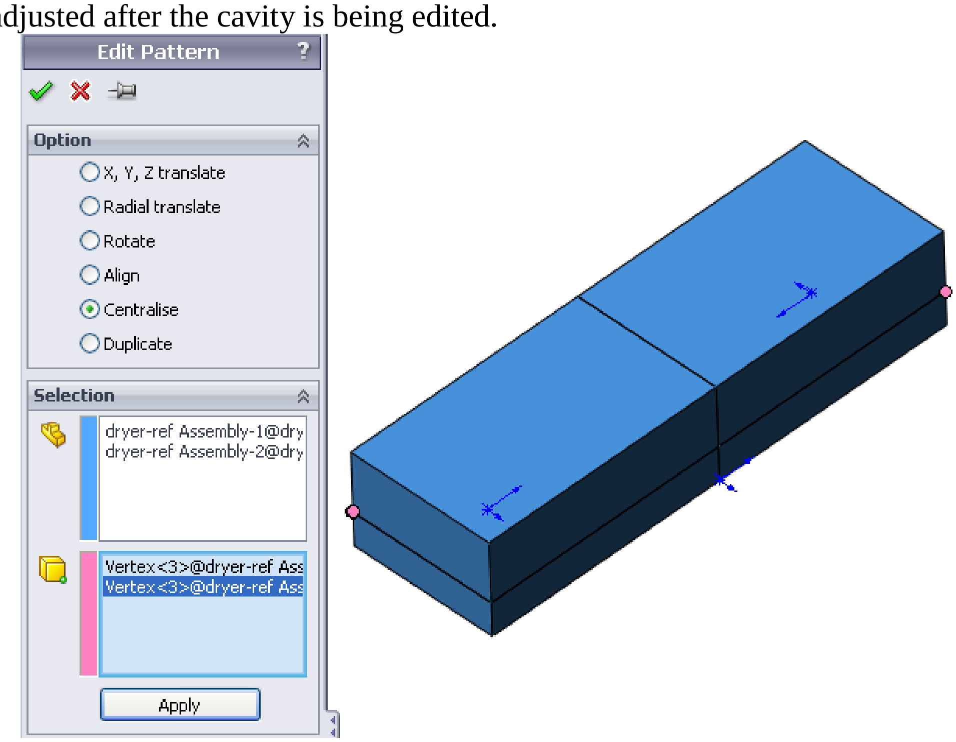
Align: alight horizontally or vertically to the selected point

Alight is in sequence, the first selected point or few vertex （moves） alights to the last selected vertex (stationary)

For example, selected the vertex of the cavity to be translated and the destination vertices in sequence.

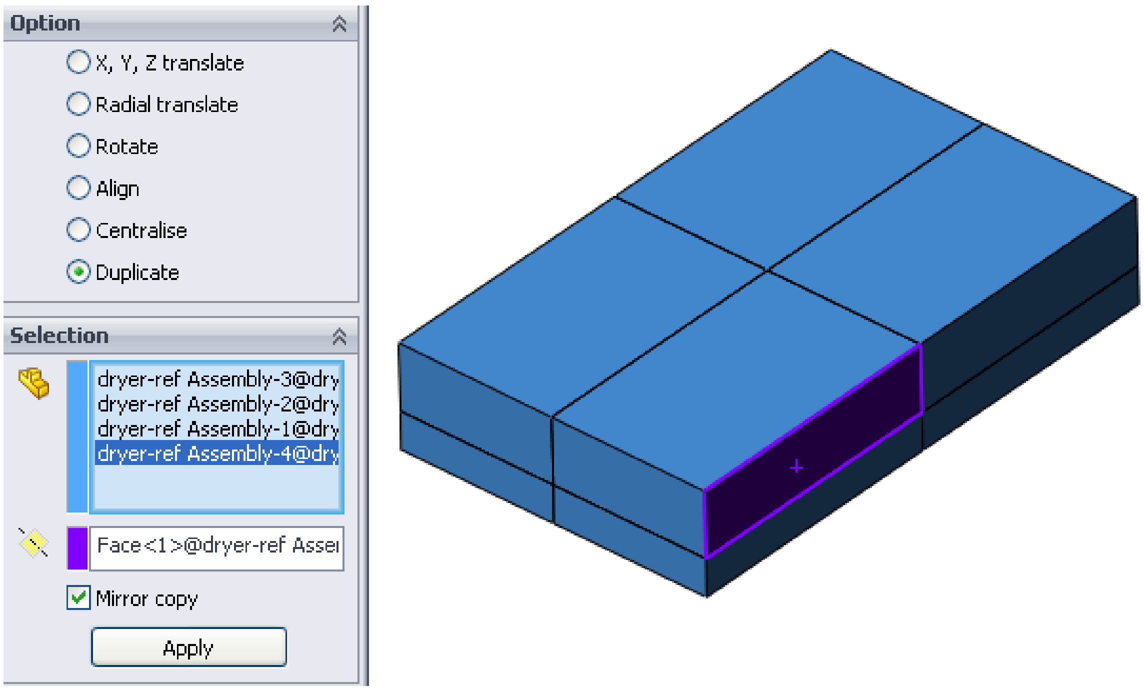


Centralise: Adjust the centre of the layout. The center point of the polygon formed by the selected points is alighted to the Origin. This is an important step as the center point has to be adjusted after the cavity is being edited.

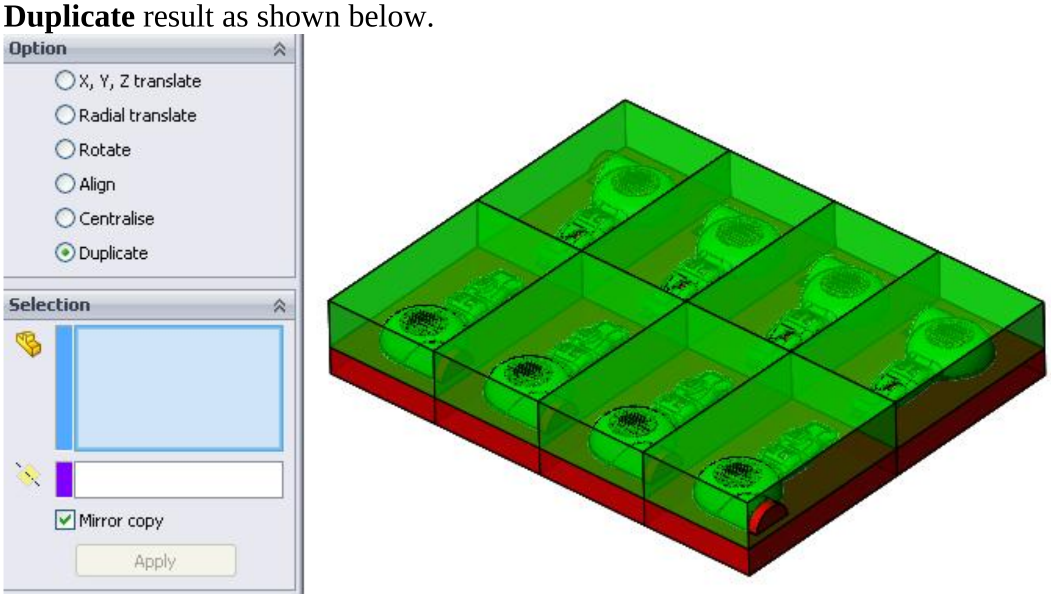


Duplicate: Duplicate the selected cavity, Different situations will appear if Mirror copy is checked or unchecked

- Mirror copy is unchecked
  Select the assembly to be duplicated, click **Apply** to proceed. The duplicate is overlapped with the original assembly.
- Mirror Copy is checked
  Perform mirror copy to the selected cavity, any planar surface can also be selected as mirror face, this can quickly produce 32-cavities, 64-cavities mold, etc.



Duplicate result as shown below.



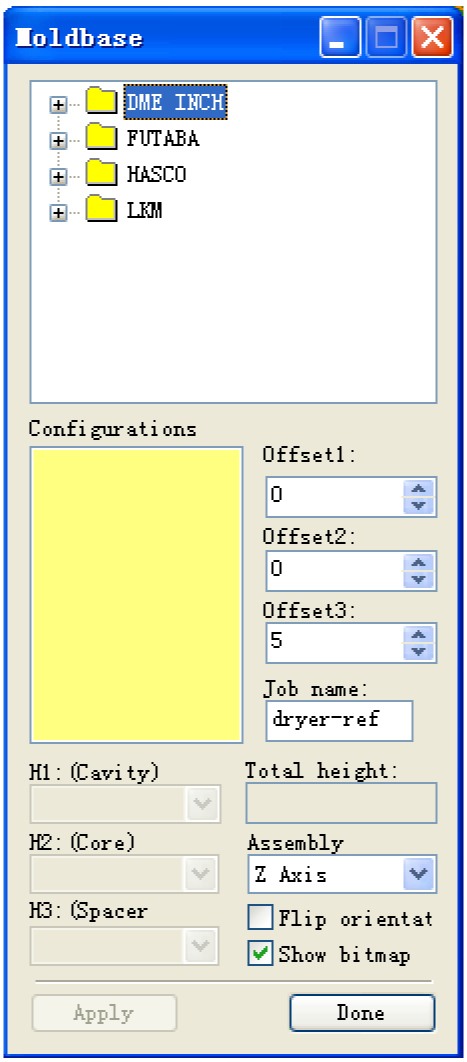
Chapter 8. Moldbase Manager

The moldbase manager can:

1. Insert moldbase into the top assembly after Layout
2. Edit moldbase after the moldbase was inserted

After Layout, *Project.sldasm (top assembly file) is generated, Click , the Moldbase dialogue box pops out.

The complete moldbase can be designed using the Moldbase manager, including standard and customized moldbase. The center of mold layout coincides with the moldbase center.



Standard moldbase is available in this dialogue box, the data listed is provided by the suppliers based on the published catalog.

Typical design process as follows:

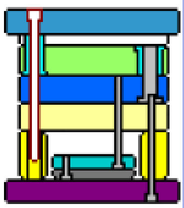
Moldbase supplier —> Mold base type —> General dimension (L & W) —> H1, H2 and H3 —>Gaps between fixed and movable half

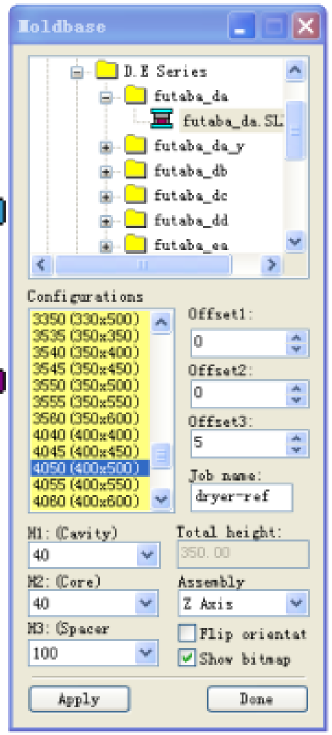
In 3DQuickMold, standard Moldbase from FUTABA, DME, HASCO and LKM is available for selection, more standards will be provided in future. Customized moldbase could be added.

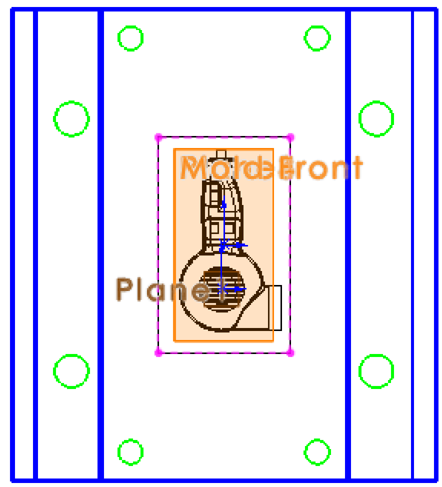
Select a type of moldbase, the preview of the selected moldbase appears near the dialog (check Show bitmap at the bottom right corner of the dialogue box to preview). As shown below.

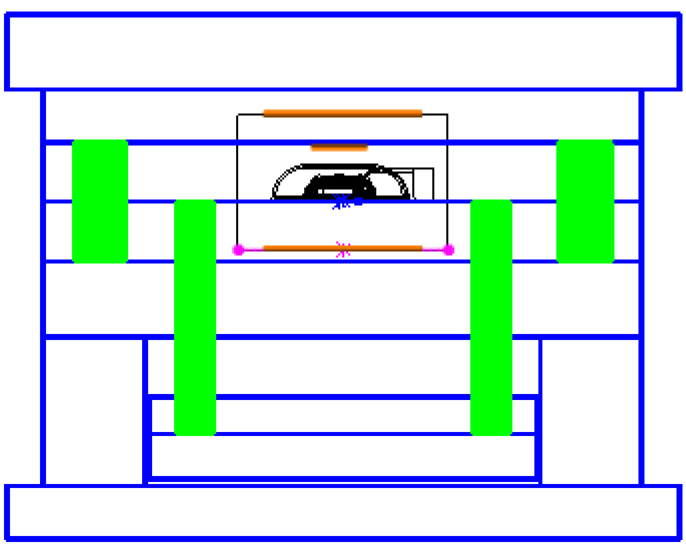
Under Configurations, different dimensions of the moldbase are shown. Select one of them, a 3D preview appears at the graphic area. This can be view in different orientation.

Use the Wireframe option to show a clearer view.









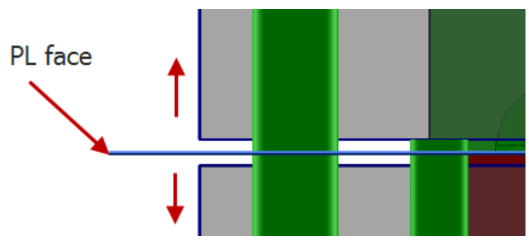
H1 (cavity): Define the thickness of A plate.
Standard thickness is available in the pull down menu.
All the value is standardized, custom input is not allowed at this time.

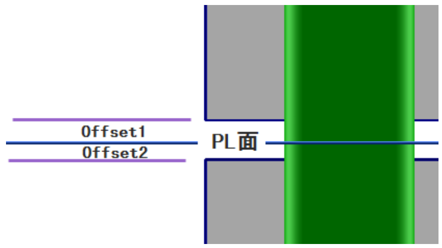
If custom input is required, first arbitrarily select a value to generate the moldbase, then click Moldbase to enter the edit mode to edit.

H2(core): for B plate
H3(space block)： for C plate

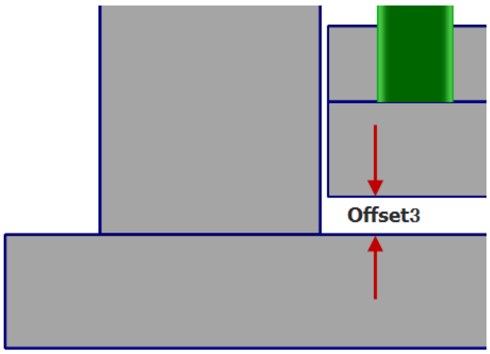
To edit the thickness of other plates, first generate the moldbase, and then click Moldbase again to perform editing.

Offset1：The spacing from cavity plate to moldbase center
Offset2：The spacing from core plate to moldbase center





Offset3：The spacing between ejector plate and bottom plate

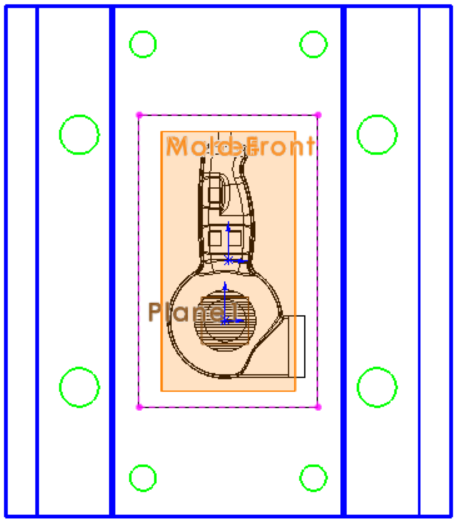


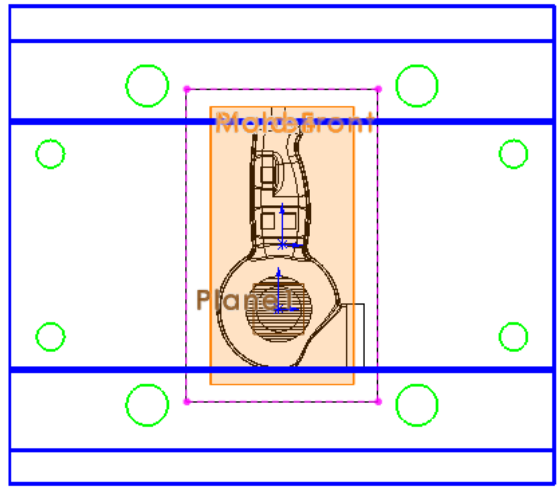
Job name: Prefix for all components in the mold base. Assembly will not be affected.
Total: Height of the mold base, this value for reference only, it cannot be edited directly.

Assembly: The relation between the mold layout and the mold base
There are altogether 6 directions, the X、Y、Z、-X、-Y、-Z direction for selection
The preview can be seen in the working window
This option is used for the part not using +Z direction for assembly.

Flip orientation: Adjust the relative direction of the moldbase and mold layout.
The following pictures show the two different situations.

Flip Orientation is checked or unchecked.

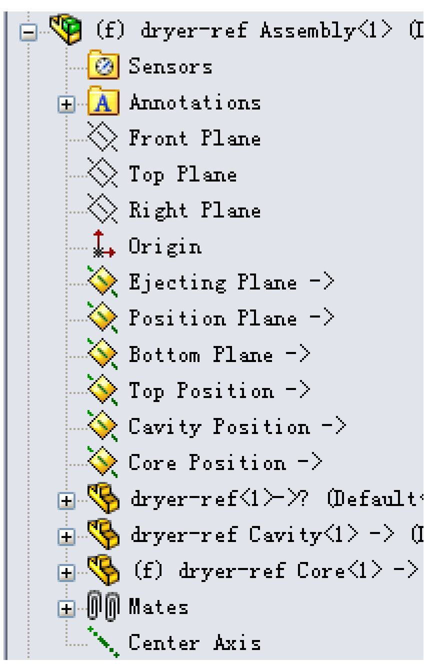


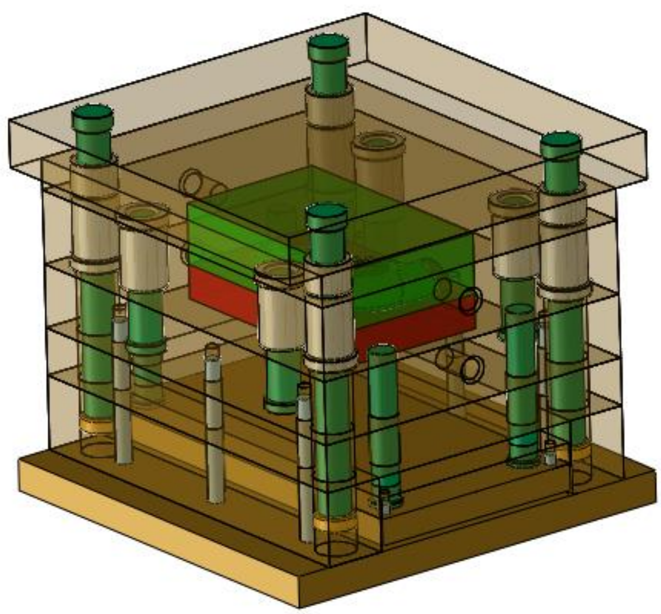


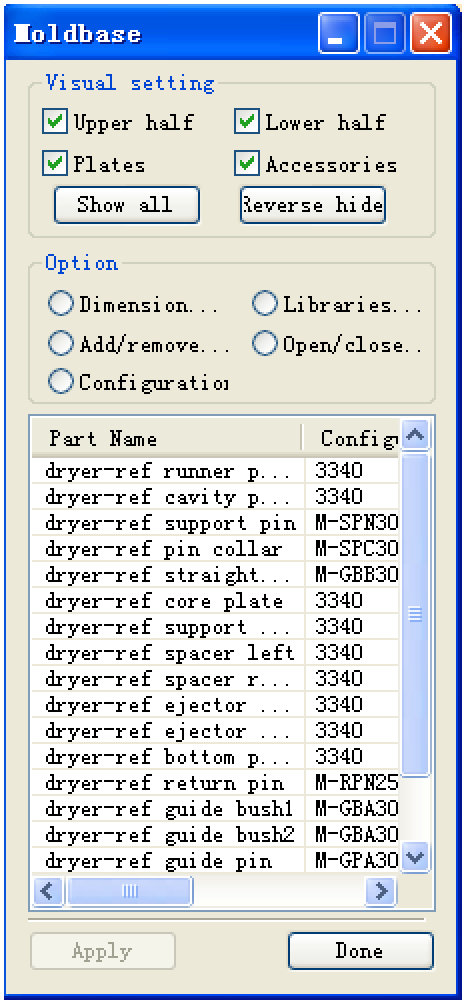
Show bitmap: Show bitmap near the dialog.

After setting up, click Apply, 3DQuickMold will generate the moldbase
The moldbase generated will be stored in the current working directory but not the default directory of the 3DQuickMold. It may take time to finish as there are many parts. Click Done to quit from the dialog.

A moldbase assembly appears under the * Project.sldasm tree. All reference plane related to the moldbase will be copied into * Assembly.sldasm. Those planes will be used for place some components such as ejectors and mounting screws.







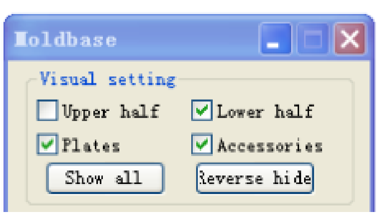
After the mold base is added, click the ***Moldbase Manager*** to enter the Moldbase editing function, current working window will automatically switch to the Moldbase assembly.;

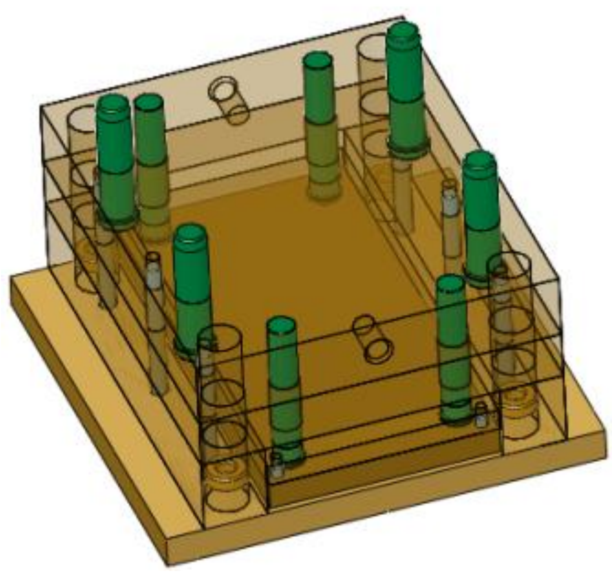
To construct non-standard moldbase, first select a similar standard one, and then do the custom editing. The following situation can be carried out in the edit mode.

- Changing the dimension of the mold plate and the position of the standard parts.
- Add or remove extra mold plates. (Such as manifold plate, double ejecting mold)
- Insert additional standard parts.

Visual setting: set the visibility of different mold base component;

Upper half: Show/hide the Upper half
Lower half: Show/hide the Lower half
Plates: Show/hide the mold plates
Accessories: Show/hide the accessories (guide pins、guide bushings、 screws)



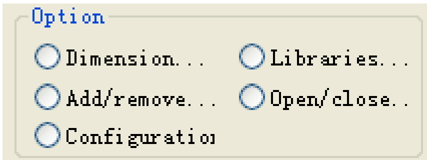


Show all: Show all components.

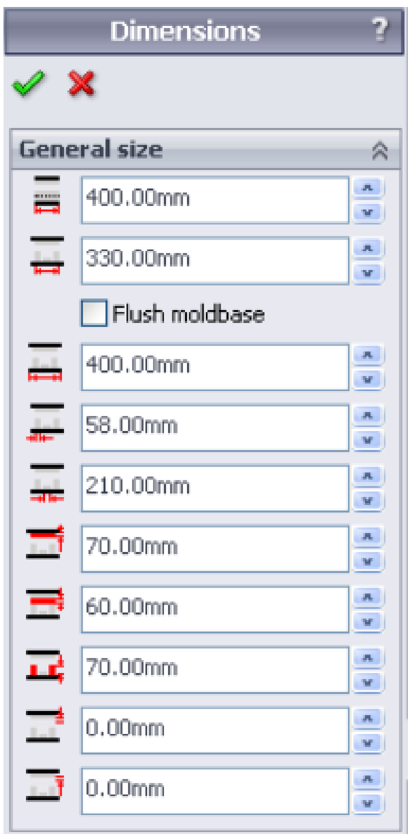
Reverse hide: Show only the selected parts. Select a few components and reverse hide the unselected.

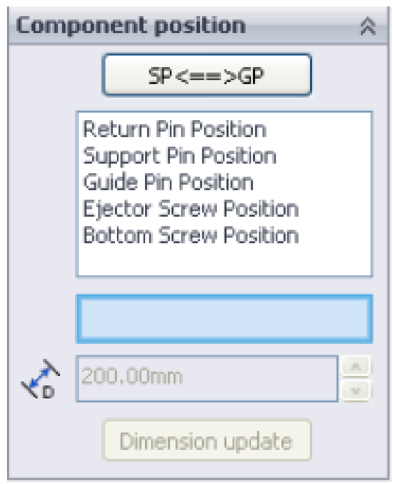
Under Option, there are 5 options for moldbase customization

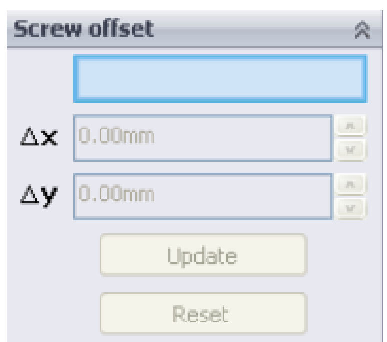
They are Dimensions, Libraries, Add/Remove, Open/Close, Configuration



Dimensions







Dimension of all the plates of the moldbase can be edited

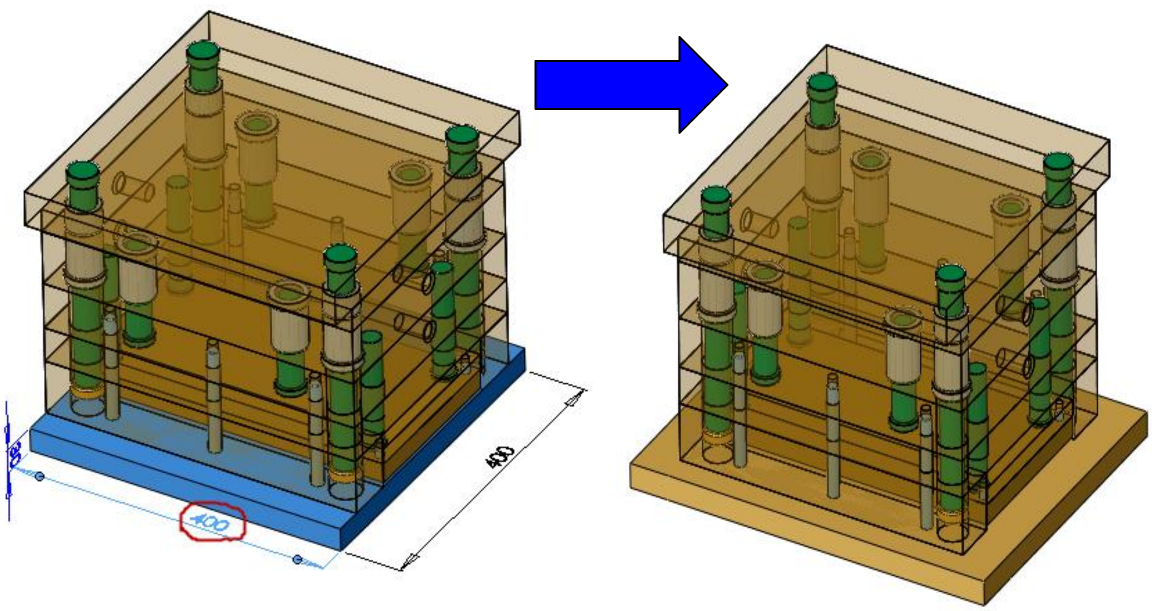
: length of moldbase
: width of moldbase

Flush moldbase: Check this option, An I type moldbase will become H type.

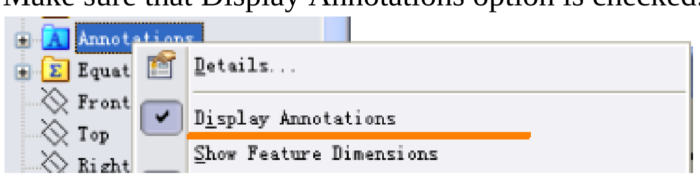
: width of bottom plate
: thickness of spacer block
: width of ejector plate
: thickness of plate A
: thickness of plate B
: height of the spacer block

Notes: For some dimensions not listed in the dialog, user can double click the component in the graphic area to display its standard dimensions and do the editing. After editing, clicks rebuild or go to the property tree to perform the edit dimension.

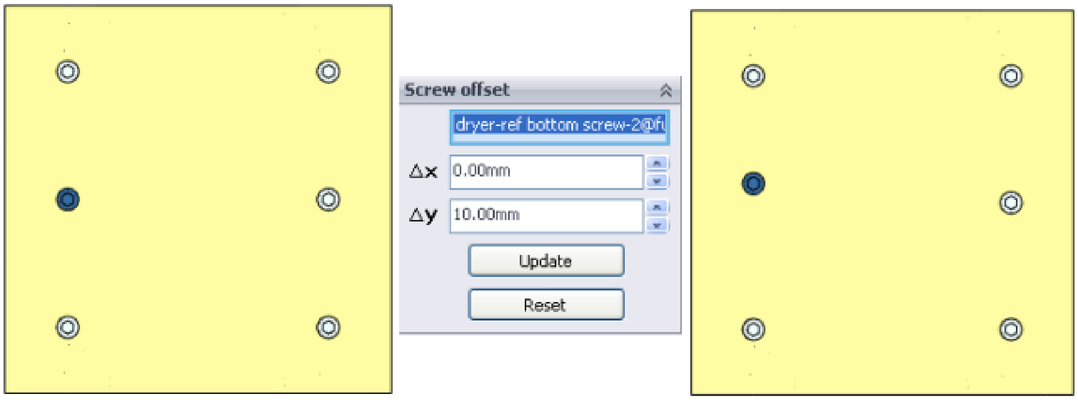
For example: double click the bottom plate to display all dimensions, change the dimension from 400 to 500.



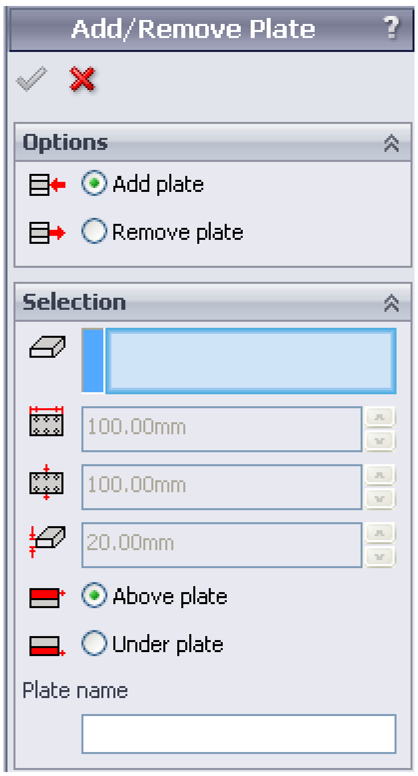
Component position: Edit the position and dimension of the accessories.
Make sure that Display Annotations option is checked.



Screw offset: Edit the screw position.
This function can only edit the screw position on the line up direction（Although there are both X and Y direction for translation, only the direction along the length of the space block could be edit）
Click ***Apply*** to translate the screw, click ***Reset*** button to move back to the original postion



Add/Remove: Add or remove selected mold plate (this function can produce multiple nonstandard mold base)



: add mold plate in the moldbase

: remove the added mold plate in the moldbase

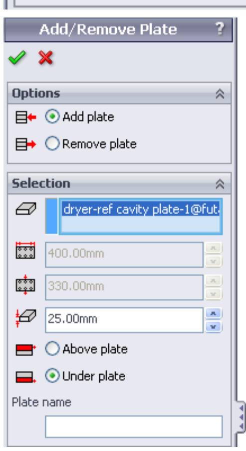
: reference plate, when adding mold plate, the mold plate will be added above or below the reference plate. Click to show the preview. If mold plate has to be removed， this would be the plate to be removed. the basic plate for the mold base could not be removed.

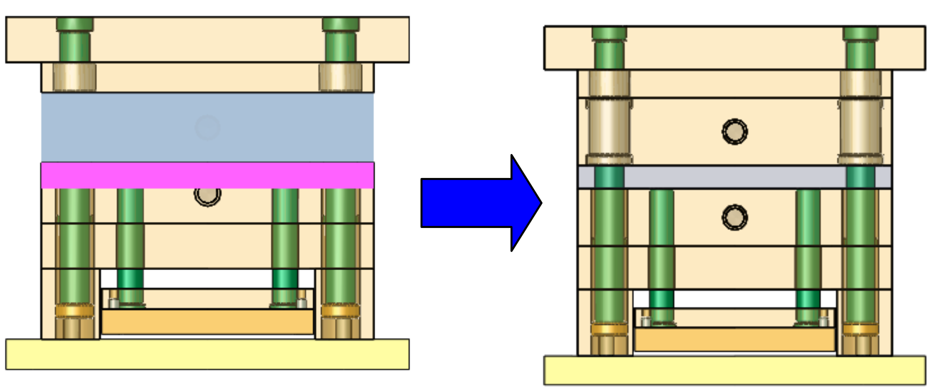
Note: when a mold plate is selected, 3DQuickMold decides whether a plate can be added above or below the reference plate. There are some rules for adding a plate:

1. Top plate can only be added above it, so it the ejector plate.
2. Bottom plate can only be added under it.
3. Spacer block is not allowed to be reference plate.

Other plate can be added either above or below the reference plate

Result after a plate is added to the current moldbase.





: length of the mold plate to be added
As the moldbase is a standard moldbase, the length of the mold plate cannot be edited here, it is controlled by 3DQuickMold. The length here is for reference only.
The length here is defined by the reference plate.

: width of the mold plate to be added, it cannot be changed and for reference only

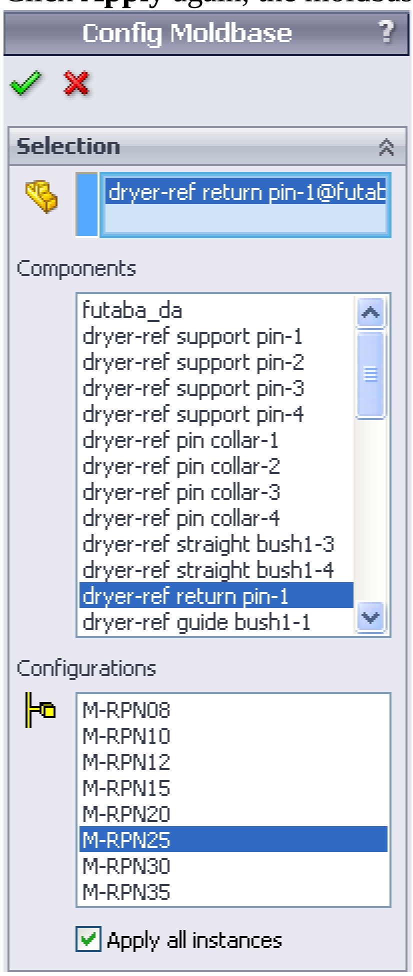
: thickness of the mold plate to be added

: add a plate above the reference plate, some types of plate cannot be added above the reference plate, this option is disabled for these types of plate

: add a plate under the reference plate,some types of plate cannot be added under the reference plate, this option is disabled for these types of plate

Plate name: Name the mold plate to be added

Open/close. Show the mold Open/Close condition.
click **Apply** , the upper mold will move up, the upper mold and the ejector plate can be dragged, but only in the Z direction. The lower half is fixed and cannot be dragged.
Click **Apply** again, the moldbase returns to the previous condition, and it cannot be moved.

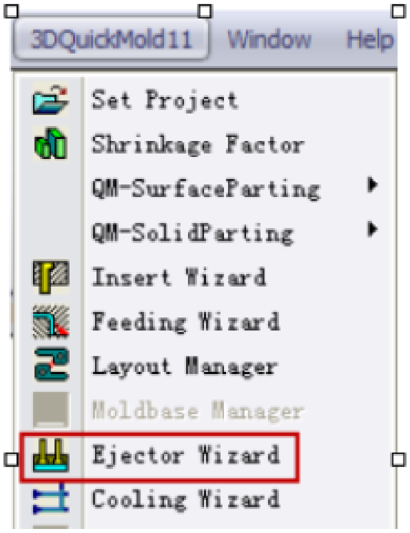


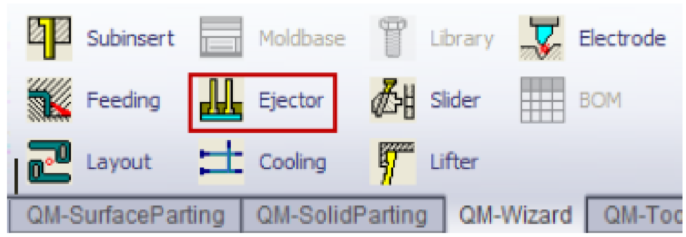
Configuration: Change the configuration of a mold component. All the configuration of all the standard parts is listed. For example, if the mold base is enlarged, screw, pin, etc., have to be enlarged

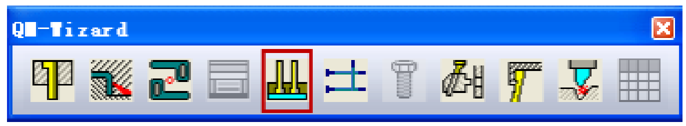
Selection: select the component to be edited
Components: select component form the list. The component is displayed in Selection
Configurations: relative configuration, if the configuration has to be changed, select and change the relative configuration
Apply all instances: Decide whether to apply the changes to the parts with same configuration.

Note: after editing, some parts may not be changed. Click Rebuild to apply the changes.

Chapter 9. Ejector Wizard

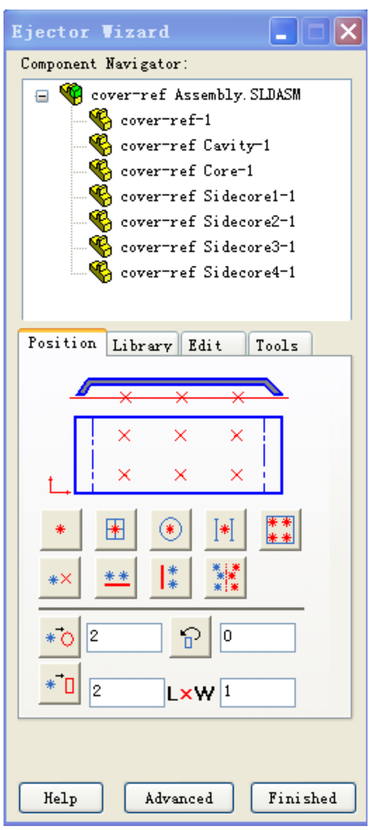






Ejector Wizard is used to add, edit, and adjust the ejector pin. Ejector pin can be added after the mold base has already been loaded

Click  to start the design of the ejector. The steps are as follows ,
Define the position of the ejector pin->Add the ejector pin->Trim the ejector pin->Create pockets on the core and mold plates that the ejector pins are passing through->Adjust ejector pin if it is not suitable.



9.1 Component navigator

Please refer to the Insert Wizard regarding how to use it.

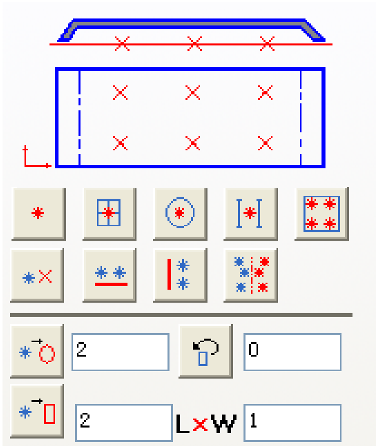
9.2 Position

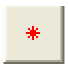
Those functions work on part model only. They are used to specify the ejector’s position in the form of 2D sketch points.

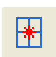
Actually, the sketch points for ejector’s position could be created on the Core or Product Assembly file. For example, the ejector that is used to eject the runner and gate may not locate on the core or Product, it is usually created on the Product Assembly.

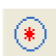
The system advises the user to create the sketch on Product file.

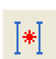
The function on the position page can assist user to create the sketch points quickly by using simple selections and one click.

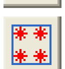


: Select a face, the sketch point at the pick up point

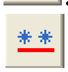
: Select a face, the sketch point at the middle of face

: Select a circle, the sketch point at the circle center.

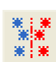
: Select two edges， the sketch point is located between the two pick up positions.

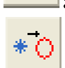
: Select a face， four sketch points are created on the face.

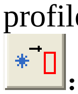
: Select a sketch point to remove

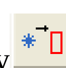
: Align two selected sketch points in horonzental direction.

: Align two selecged sketch points in vertical direction.

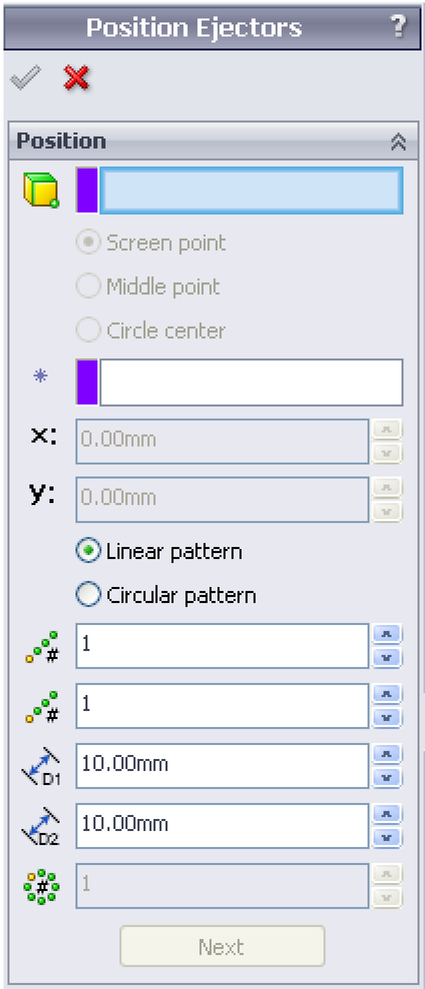
: Mirror the selected sketch points with a reference central line.

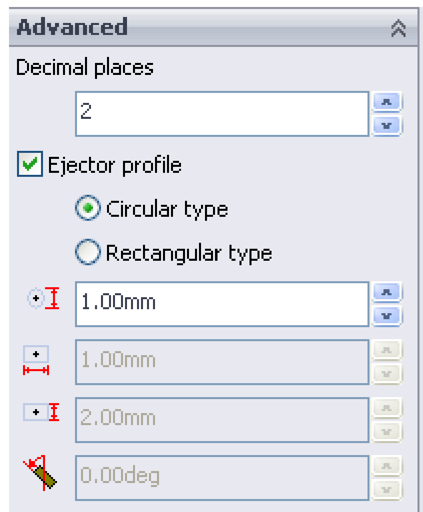
: Draw a circle profile at the selected sketch points to represent the regular ejector, stepped ejector or ejector sleeve. Sometimes, in order to simplify the ejector design, only the ejector profile is drawn to represent the real 3D ejector model.

: Draw a rectangle profile at the selected sketch points to represent the ejector blade. Same reason as above for simplification purpose.

: Rotate the rectangle profile drawn by , select a sketch line of the rectangle， click this icon to rotate the rectangle sketch.

Click Advanced, the following page pops out.





Select point, edge or face to define the center of the ejector pin here. The center can be defined by the 3 methods below.

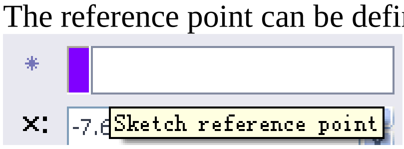
Screen point: select the point on the screen;

Middle point: select the middle point of the edge or face;

Circle point: Select the center of circle

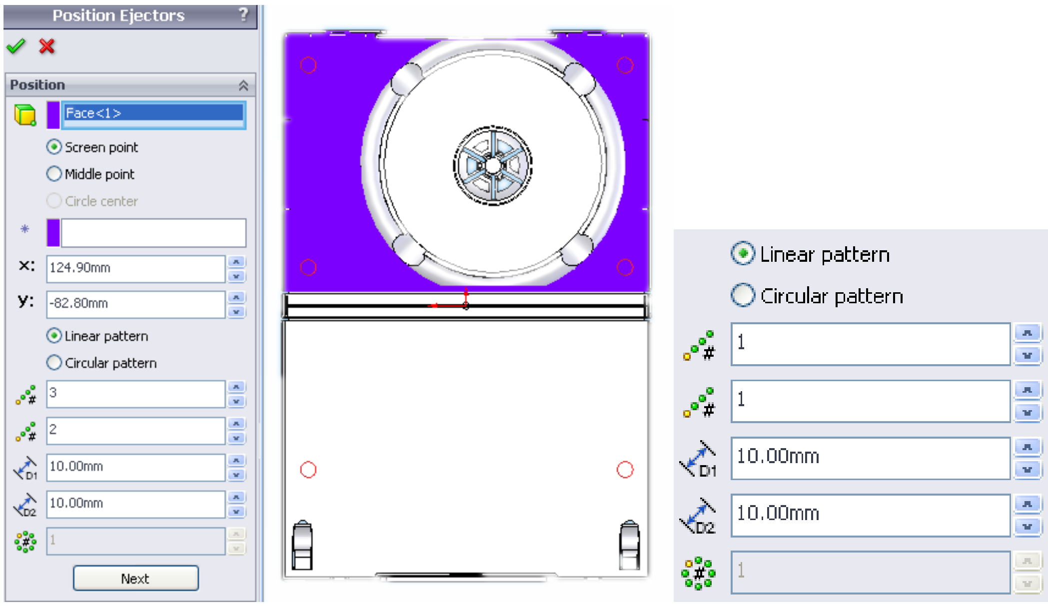
x, y coordinate system can also be used to define the position (origin as the default reference point)

The default reference point of the x, y coordinate is the part origin.
The reference point can be defined in the Sketch reference point

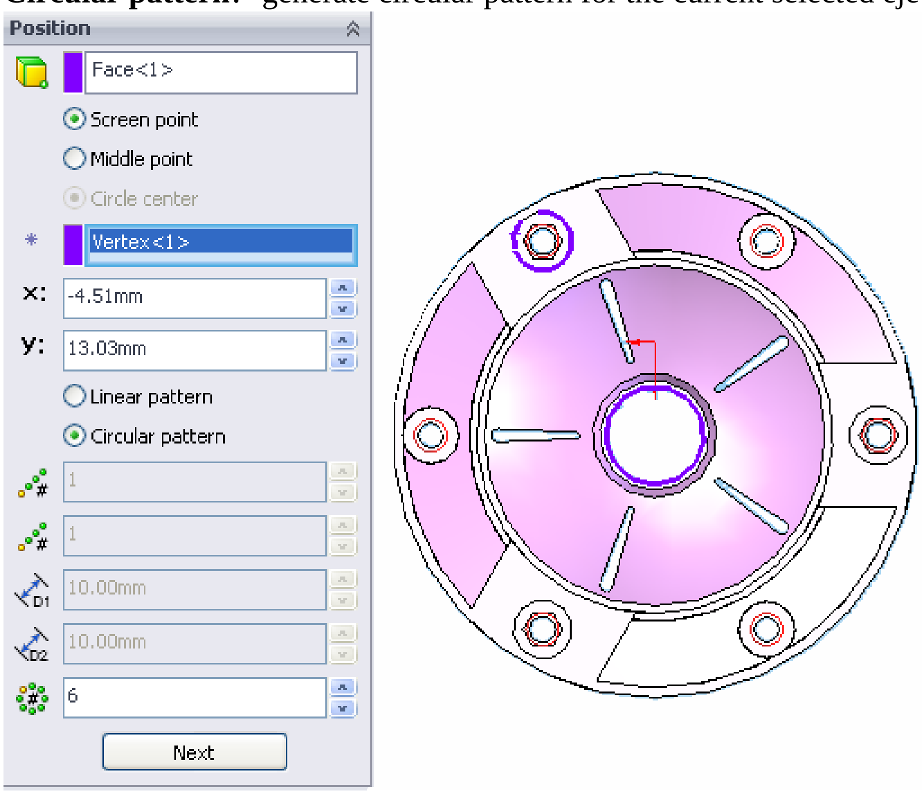


When the position of the point is defined
Click **Next** to create a sketch point and clear the previous ejector location

Linear pattern: generate linear pattern for the current selected ejector location

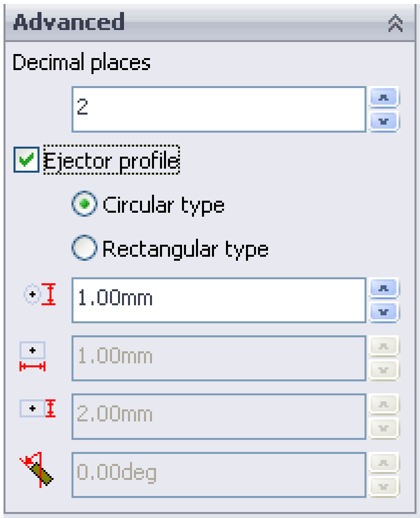


Circular pattern: generate circular pattern for the current selected ejector location



Different data row will be activated for different pattern. Click **Next** to apply.

On Advanced group, the profile of the ejector could be drawn to represent more details.



Decimal places: From 0 to 4, user can select from the list

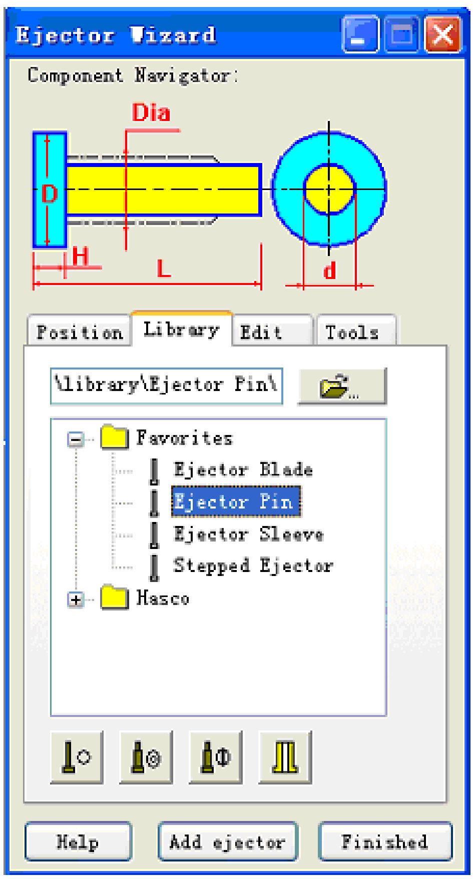
Ejector profile: Instead of using sketch point to represent the ejector's position, sometime, the ejector's profile such as circle or rectangle are used to represent the ejector.

Circular type: Need to specify diametere

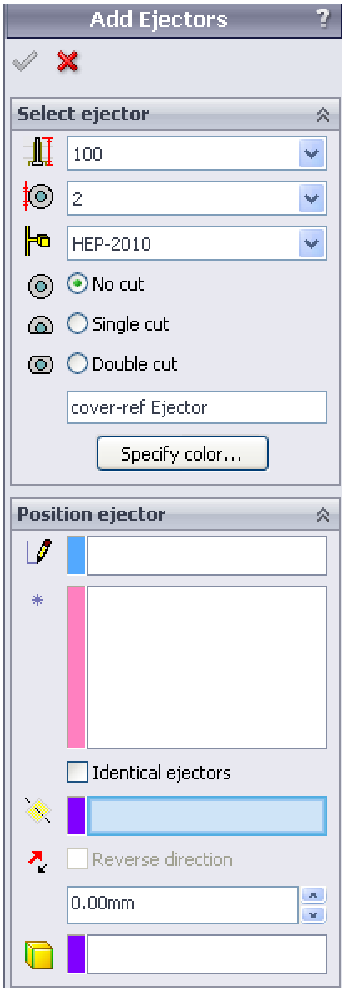
Rectangular type: Need to set the length , width and rotation angle

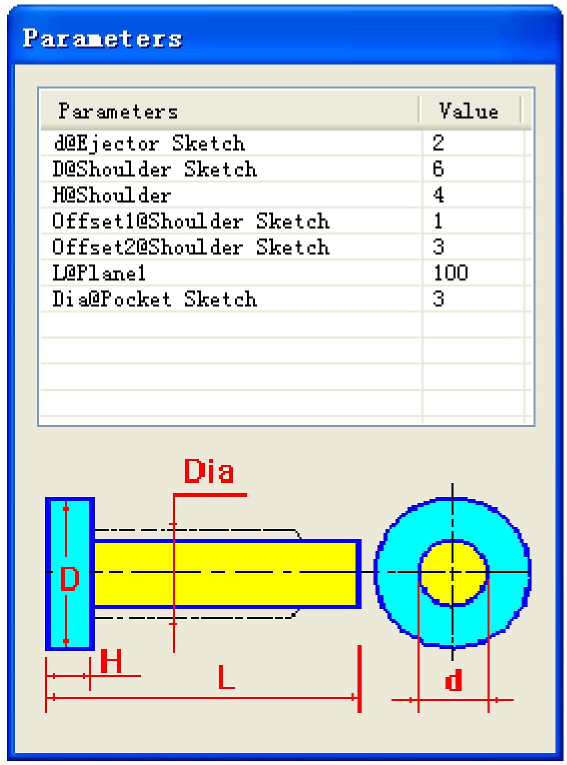
If ejector position sketch is not created before, a 2D sketch is created after press OK. If the position sketch exists, newly created sketch points will be added to the sketch automatically.

9.3 Library



Add the ejector to the assembly, generate the ejector part and save it to the working directory. Add ejector is only for assembly file. Select a type of ejector in the library, Click **Add ejectors**, the **Add Ejectors** page and the **Parameters** dialog pop out.





In parameter, the dimension of each ejector can be edited with reference to the picture at the bottom of the dialog. For example, to define the clearance between the ejector and the mold plate, edit Dia and d as shown in above.

Length: Select the length of the ejector. The length of the ejector is associated with the configuration. If the length is changed, the configuration will change accordingly.

Diameter: select the diameter of the ejector. The diameter of the ejector is associated with the configuration. If the diameter is changed, the configuration will change accordingly.

Configuration: Select configuration for the ejectors. If the Configuration is changed, the length and the diameter will change accordingly.

No cut: No cut on the ejector head
Single cut: Cut the ejector head on single side
Double cut： Cut the ejector head on double sides
Naming ejector： Name for the ejector component

Position ejector

: Select the sketch generated in **Position,** all the selected ejector locations will be listed in sketch point list, and this is a quick selection of all ejector locations.

: Select the location to add the ejector. If the sketch is selected in the previous step, the points in the sketch will be added automatically to here.
The points of ejectors of the same type and same configuration can be selected together.

Identical ejectors: Select whether to add the same ejectors for the above selected points. It is especially for interchanging ejector, reduces the number of different parts in the assembly, as a result easier to manage

Reference plane: The reference plane that the ejector will mate to.
In 3DQuickMold, the Ejecting plane is the default reference plane. Other plane such as Bottom plane will be used to place ejector sometimes.

Reverse direction: Reverse the direction of the ejector

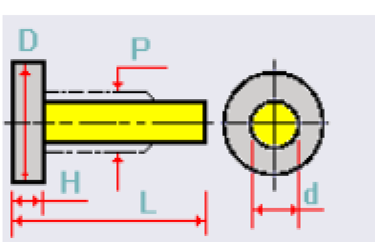
Select a circle: It can determine the diameter of an arc. Select an arc, a message showing the diameter of the arc will appear at the lower left hand corner. It helps to determine the size of ejector easier. Particularlly, for ejector sleeve

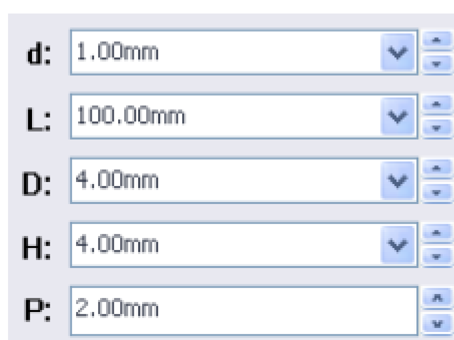
After the setting is done, click the OK button, 3DQuickMold will load the ejectors automatically, the ejector generated will be saved in the current working directory, it take a moment to generate all ejectors if the Number of ejector is too many.

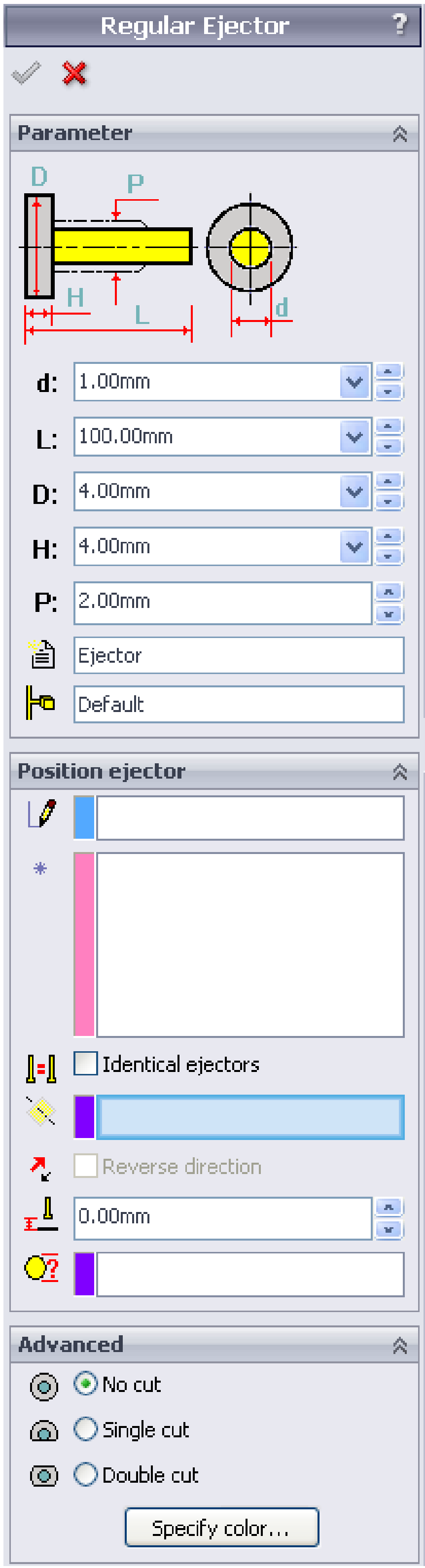
: Used to add non-standard ejectors

: Regular ejector

All parameters to define the regular ejector are listed below, preview is provided to help user to key in the right value.







Naming ejector: Name for this kind of ejector

Configuration: Input the ejector configuration such as catalog name for BOM reference

Position ejector

: Sketch to define the ejector positions

: Sketch points to define the ejector positions

Identical ejectors: select whether to add the same ejectors for the above selected points. It is especially for interchanging ejector, reduces the number of different parts in the assembly, as a result easier to manage.

Reference plane: The reference plane that the ejector will mate to.
In 3DQuickMold, the Ejecting plane is the default reference plane. Other plane such as Bottom plane will be used to place ejector sometimes.

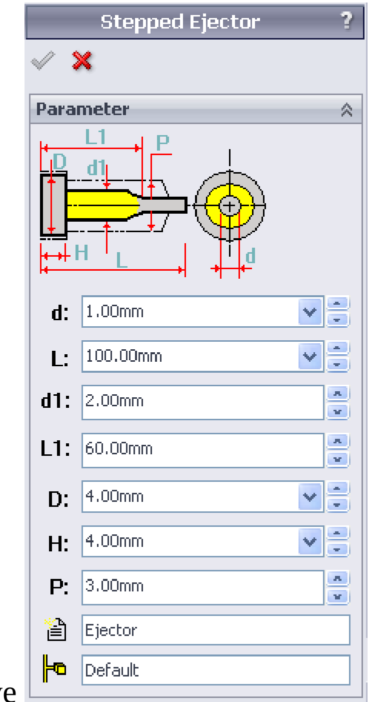
Reverse direction: Reverse direction

Select a circle: It can determine the diameter of an arc. Select an arc, a message box showing the diameter of the arc will appear at the lower left hand corner. It helps to determine the size of ejector easier. Particularlly, for ejector sleeve

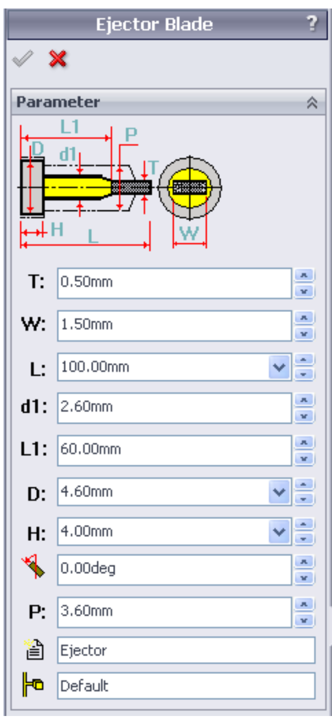
Advanced group
No cut: No cut on ejector head
Single cut: Cut ejector head on single side
Double cut: Cut ejector head on double sides

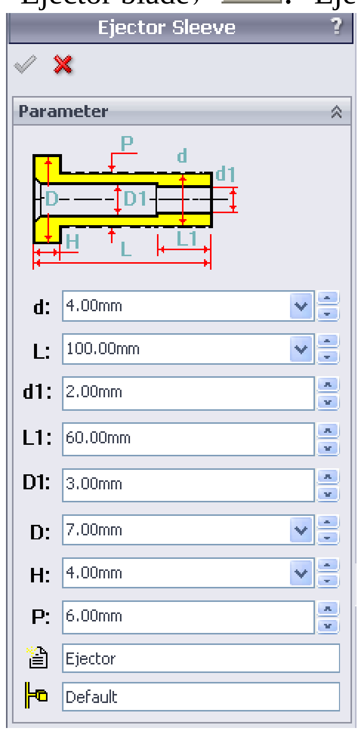
: Specify the ejector color

After the setting is done, click the OK button, 3DQuickMold will load the ejectors automatically, the ejector generated will be saved in the current working directory.



: Stepped ejector, : Ejector blade, : Ejector sleeve

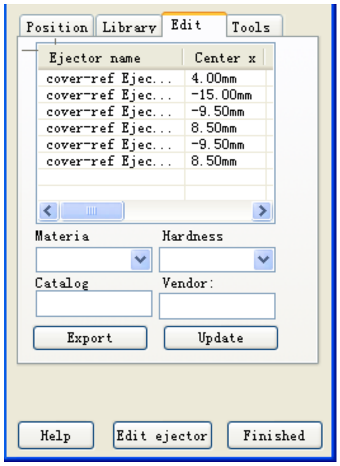


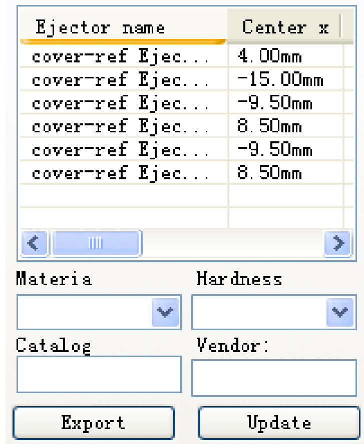


In addition:

- When the ejector is added, it is not yet fully constrained. Still can rotate, for ejector blade, additional mate is necessary to align its orientation.
- Ejector sleeve will take two steps to complete the design. Put an ejector on the Bottom Plane and Ejector sleeve on the Ejector Plane.

9.4 Edit



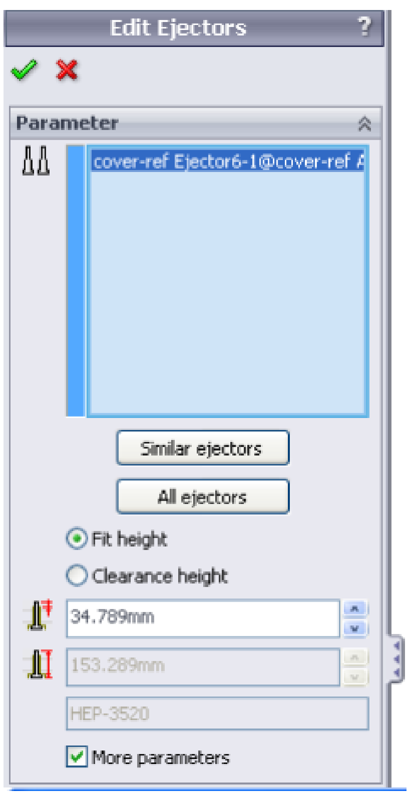


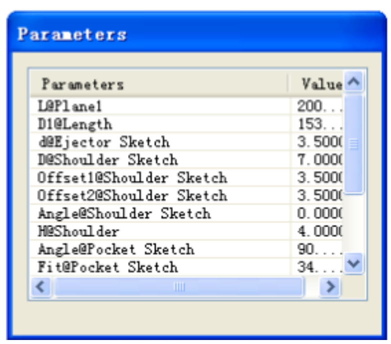
The existing ejectors are listed in this page showing the ejector name, position and catalog.

Export: Ejector information such as name, position, and catalog can be exported to a text file under the current working folder. This text file could be edited as part of BOM for ejectors

Update: Update the existing ejector information such as material, hardness and catalog.

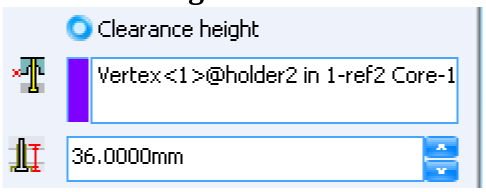
Edit ejector: To change the fit height and clearance height of ejector and core. Different parameter can be changed, especially the clearance.





Selector ejector: Select ejector to edit
Similar ejectors: Select ejectors with same type and same configuration
All ejectors: Select all ejectors and add to the ejector selection list.
Fit height: The fit height of the ejector and Core is displayed below

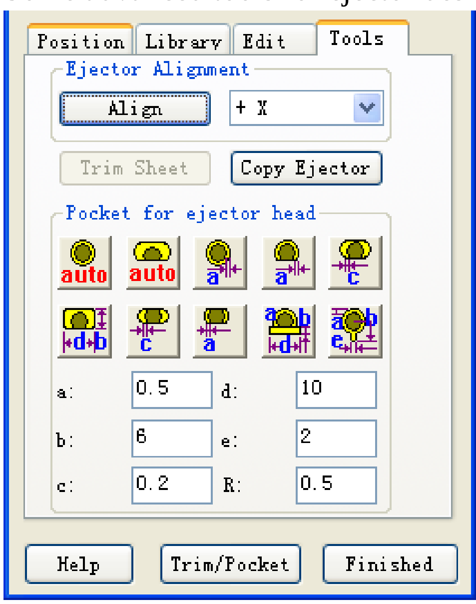
Clearance height: The Clearance height of the ejector and Core is displayed below

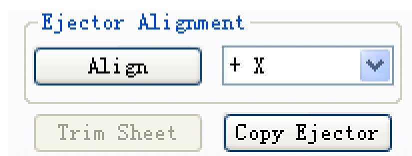


Original length: Total length of the ejector. For reference only, it cannot be edited.
Ejectors type: Ejector type. For reference only, it cannot be edited
More parameters: After selection, a dialogue box appears, it displays all the detailed dimensions of the selected ejectors.

9.5 Tools

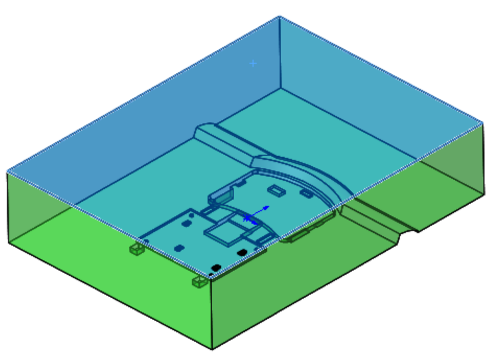
Some advanced tools for ejector design such as pocketing the ejector head.

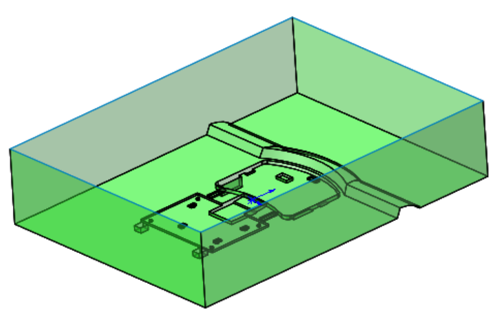




Align: By default, if the ejector head is cut on single side or both sides, the default orintation is Axis-x. Using this function, user can re-align the ejector to – X or + Y, -Y.

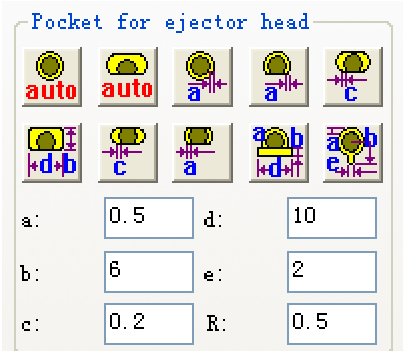
Trim Sheet: If the core/cavity are done by 3DquickMold successfully, by right, there is a sheet body feature named as CoreSheet in the core was created as well. However, if the core/cavity are imported from other CAD system or obtained manually, user can create this sheet body by selecting the bottom face and click this button.





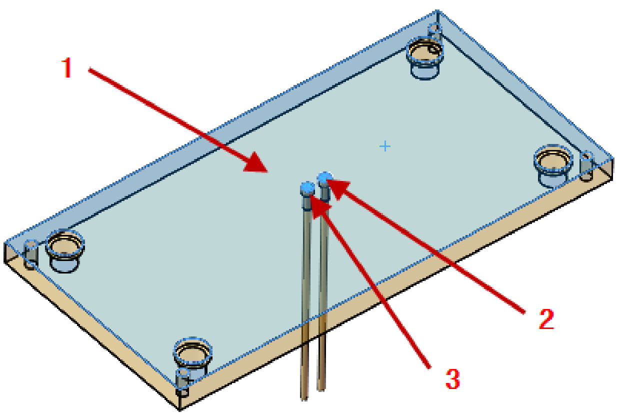
Copy Ejector: Copy ejector from one position to another.

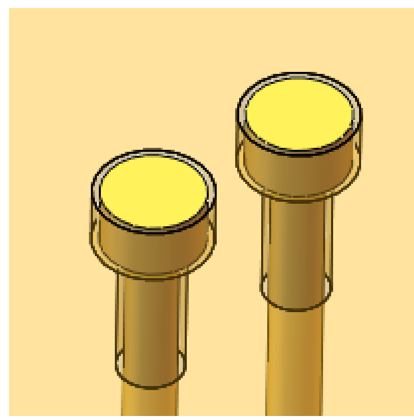
The following tools are used to create the different pockets for ejector head.



The icon on the button basically shows the pocket type.

: Select the planar face where the ejector is placed and some bottom faces on the target ejectors, click this button, the corresponding pockets are created. This function is used to create pocket for the ejector that is not cut. Dimensions are decided by internal settings.



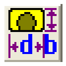


: Same selection as above but a different type of pocoket. This one is particular for the ejector that has single or double cuts on the ejector head.

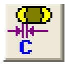
: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “a” as the bitmap shown.

: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “a” as the bitmap shown.

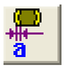
: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “c” as the bitmap shown.



: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “b” and “d” as the bitmap shown.

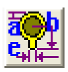


: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “c” as the bitmap shown.

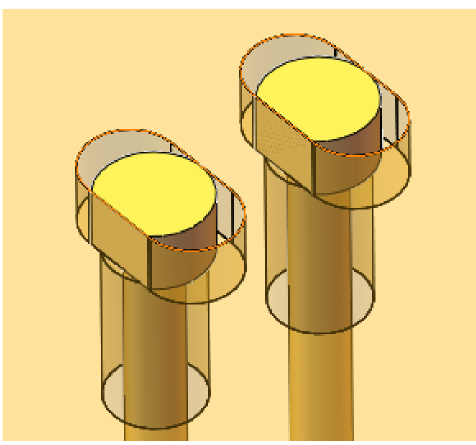


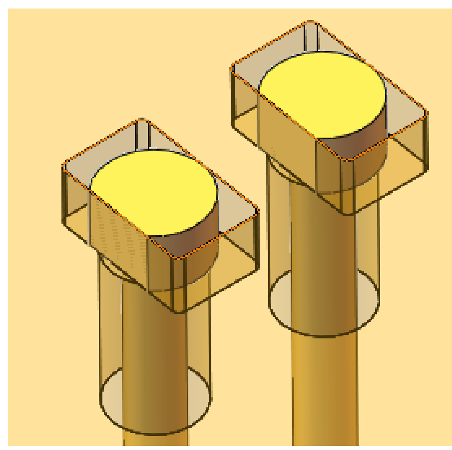
: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “a” as the bitmap shown.

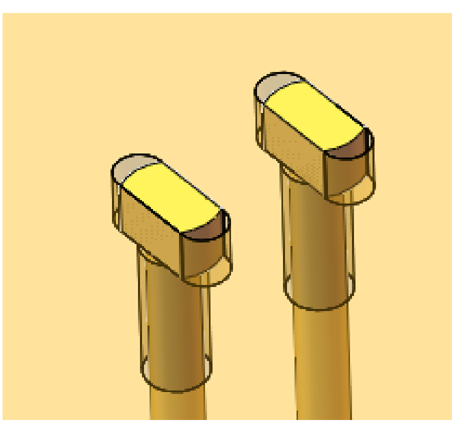
: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “a”, “b” and “d” as the bitmap shown.

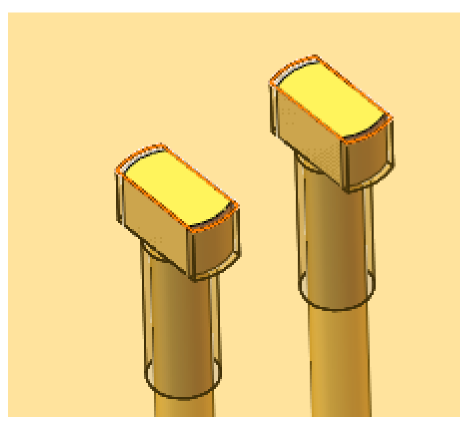


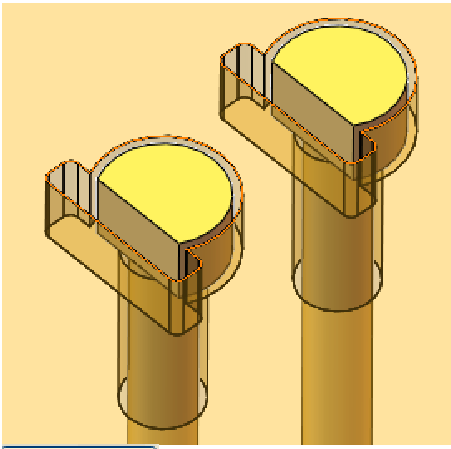
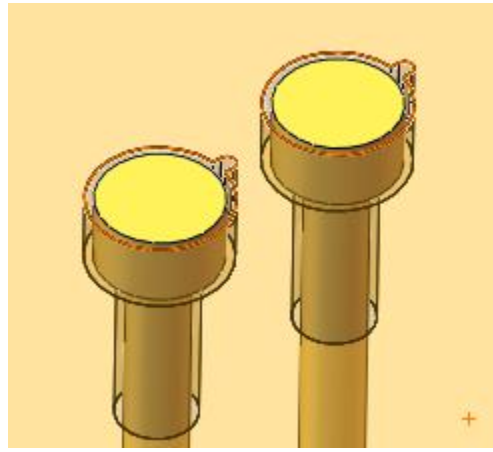
: Select the planar face where the ejector is placed and ejector bottom face. Pocket dimension is controlled by “a”, “b” and “e” as the bitmap shown.

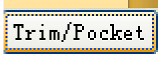








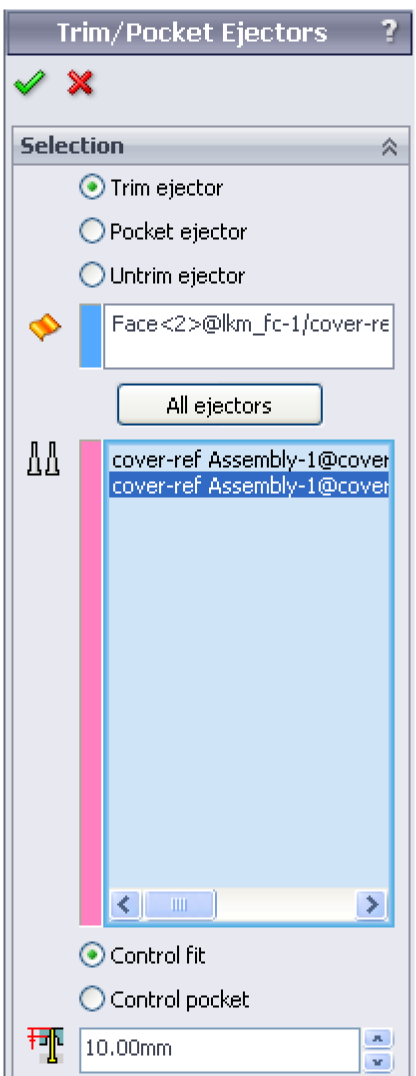


:

This function is mainly used for

1. Trim ejector
2. Create pockets for ejectors
3. Undo the trimming

To trim the ejectors automatically, there must be a sheet body named as CoreSheet in the core part that is actually the entire core faces. When the core/cavity is separated successfully, the **CoreSheet** should be generated.



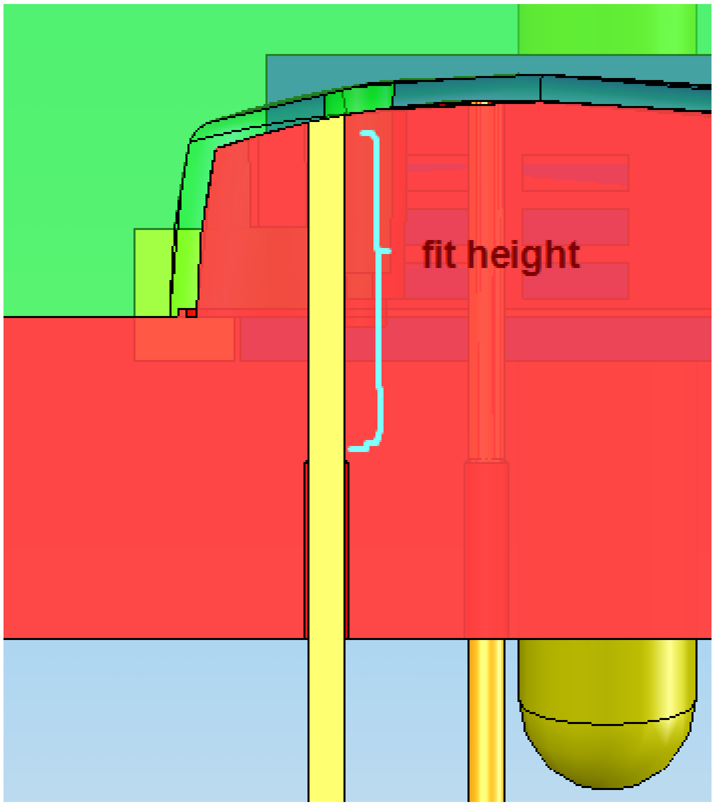
Trim ejector: Trim ejector so that the end surface of the ejector match the core profile.

Tool surface: Cutting tool surface, use this surface to trim all the ejector end surface, as default, 3DQuickMold will select the CoreSheet in the *core（* represent the file name of the plastic part）part for the tool surface, user can define their own tool surface to trim the ejectors.

All ejectors: Select all ejectors
Selector ejector: Select ejector manually

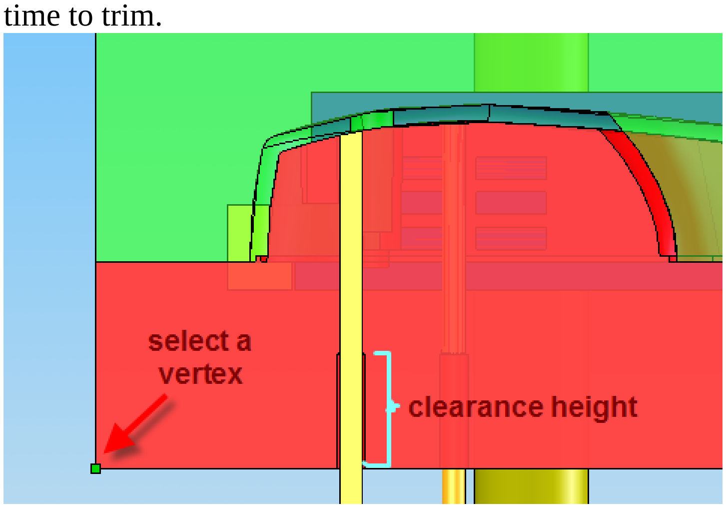
Fit height: the height of the ejector fit with the core
Value of the fit height: Select the fit height of the ejector pocket on the core insert. The value is measured down from Tool surface.

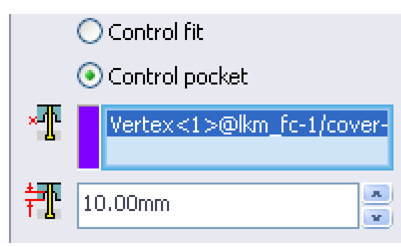
Clearance height: Select this will influence the pocketing behaviour of the core insert, it is the height of the clearance pocket measured up from the bottom surface of the core plate or from a vertex selected

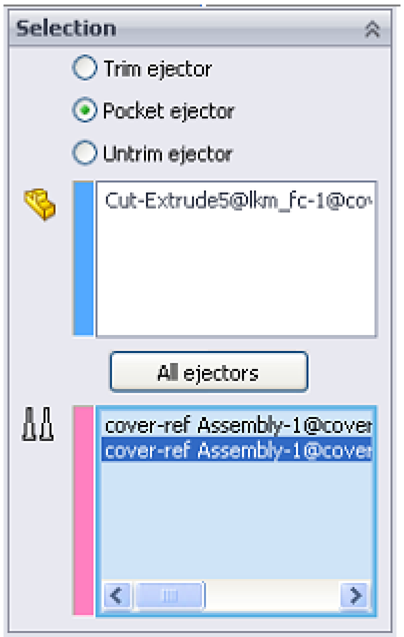


Vertex: Select a vertex as the starting point to measure the clearance
Clearance: Select the clearance height. The value is measured from the vertex in downward direction.

After the setting is completed, click OK, 3DQuickMold will trim the ejector automatically, the trimmed ejectors will then be rebuilt. If there are many selected ejectors, it may take a longer time to trim.







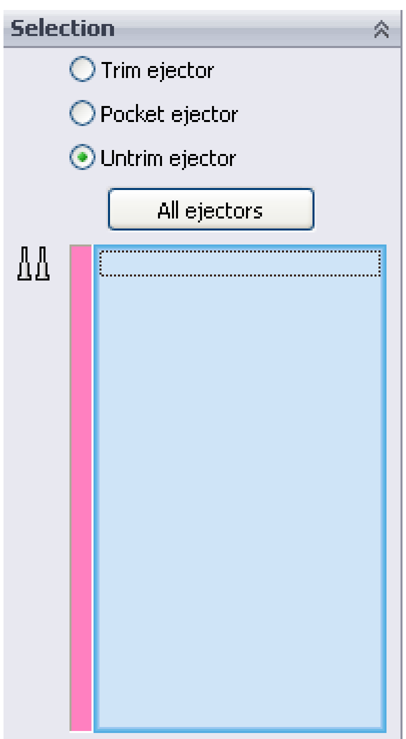
Pocket ejector: Pocket the mold core and mold plate which the ejector passing through.

Tool component: Here are the ejectors to create the pockets

All ejectors: select the ejectors under the current assembly

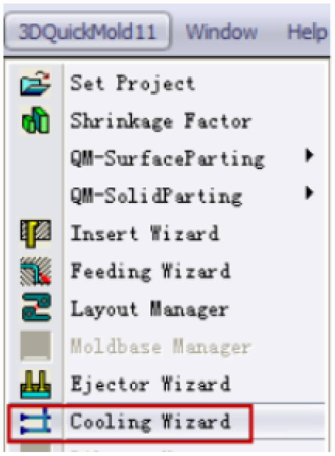
Selector ejector: Select the ejector as pocketing tool

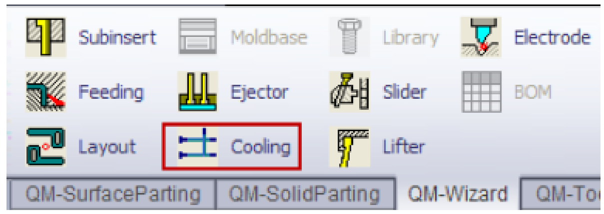
After set up is completed, click OK, 3DQuickMold will pocket the selected mold component, the pocketed component will be rebuilt.

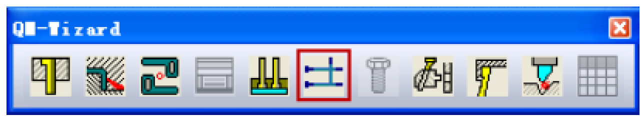


Untrim ejector: Restore the trimmed ejectors. When performing Trimming, if there is error on selecting the CoreSheet or the Core requires a great change, use this function to restore the ejector.

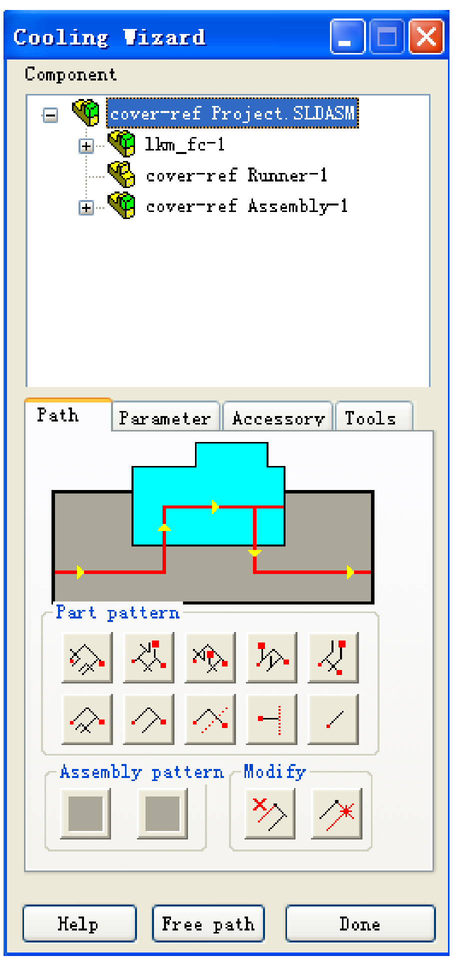
Chapter 10. Cooling Wizard







The uses of Cooling Wizard are to build, edit water channel, and add common standard parts of cooling to the mold. Normally, the channels are built on the core plate, cavity plate, sidecore and larger insert such as core/cavity.



All the design process of cooling can be completed in this module. Four options are provided which are path, parameters, accessory and Tools.

10.1 Component Navigator

Please refer to the chapter Insert wizard

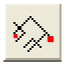
10.2 Path

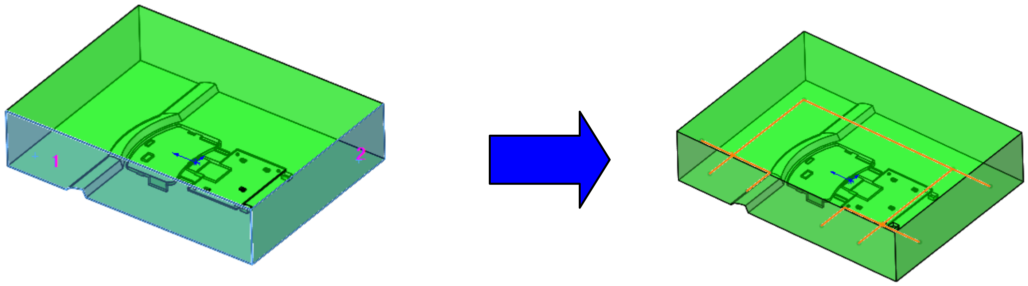
3DQuickMold provides some short cuts on this page

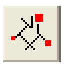
1. **Part pattern**: Works for part model only, normally used on core/cavity part.
2. **Assembly pattern**: Works for assembly only, normally used on moldbase plate.
3. **Modify**: Delete or make two sketch lines cross
4. **Free path**: A dialog will pop out to create the cooling path more flexible.

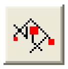
Actually, user can use SolidWorks sketch tools to create their own cooling path.

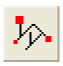
There are totally 10 icons in Part pattern, we will explain those functions one by one.

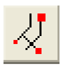
 : Pick up two faces in sequence to create a cooling path with 7 lines.

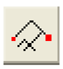


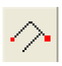
 : Pick up three faces in sequence to create a cooling path with 7 lines.

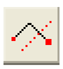
 : Pick up three faces in sequence to create a cooling path with 7 lines.

 : Pick up three faces in sequence to create a cooling path with 5 lines.

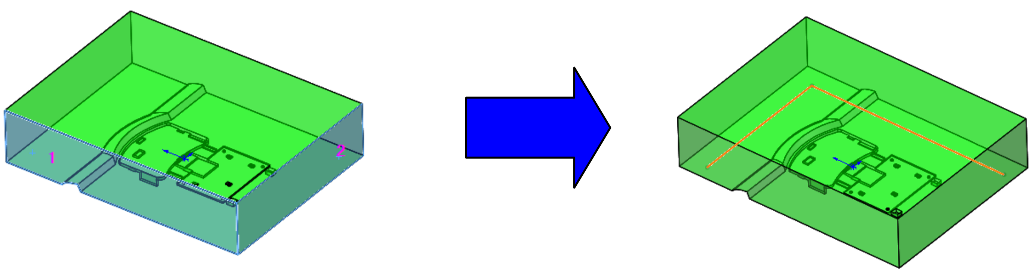
 : Pick up three faces in sequence to create a cooling path with 5 lines.

 : Pick up two faces in sequence to create a cooling path with 5 lines.

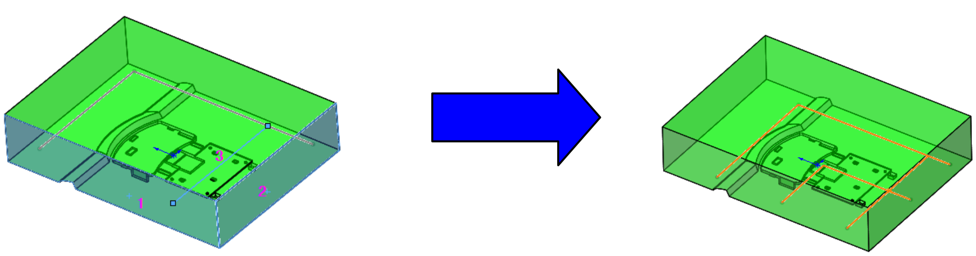
 : Pick up two faces in sequence to create a cooling path with 3 lines.

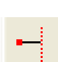
 : This function provides two methods of combination.

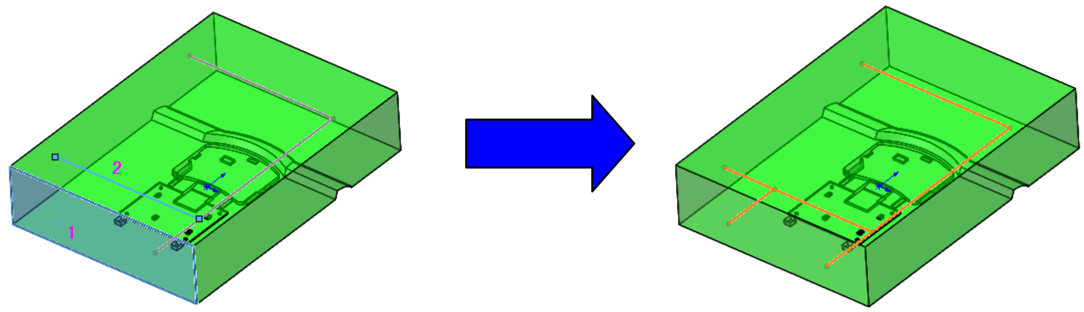
Method 1: Pick up two faces in sequence to create a cooling path with 2 lines

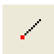


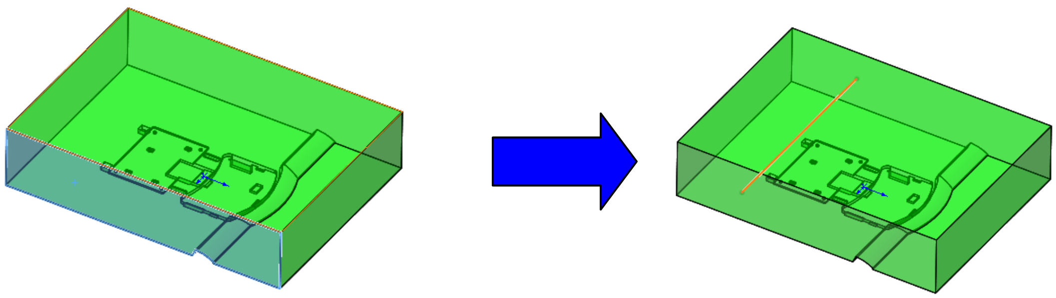
Method 2: Pick up two faces in sequence and one more sketch line as reference to create a cooling path with 2 lines. The created sketch line will cross the reference sketch line.



 : Pick up one face and one sketch line, a single connected sketch line is created.



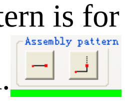
: Pick up one face, a sketch line ended at the opposite face is created.

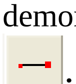


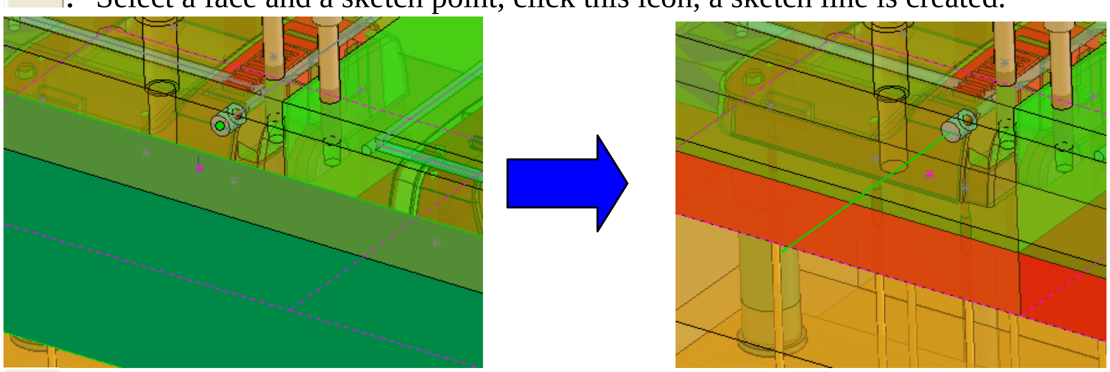
The selected points and line above is corresponding to the points and line in red on the icon and form the cooling path corresponding to the black line on the icon. Besides, the order of point and line selection affects the feasibility of constructing the path and the outcome.

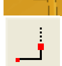
For the first time to create the cooling path, a 3Dsketch will be created in the feature tree, If positioning the cooling path is requird, edit the sketch.

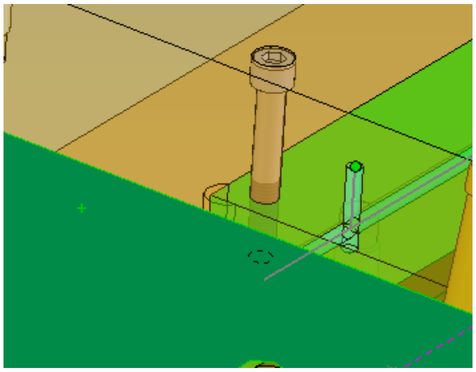
Note: the construction of complex cooling path may require several methods.

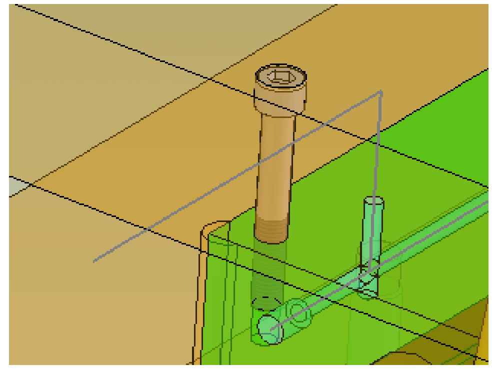
Assembly pattern: The assembly pattern is for the assembly file only. The following demonstrates the use of each commond. 

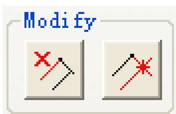
: Select a face and a sketch point, click this icon, a sketch line is created.

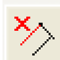


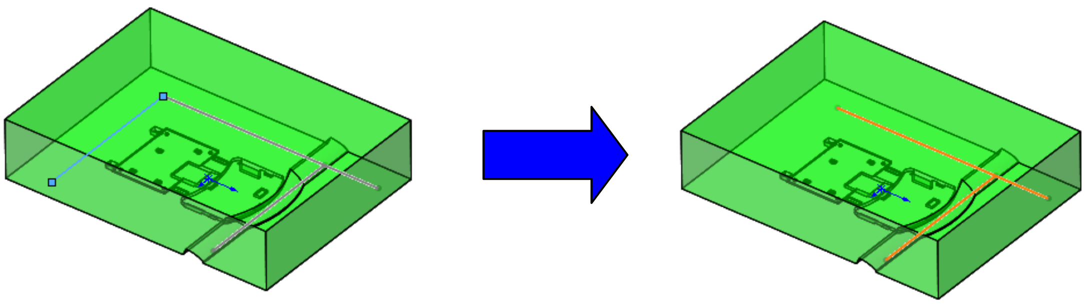
: Select a face and a sketch point, click this icon, two sketch lines are created.

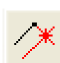


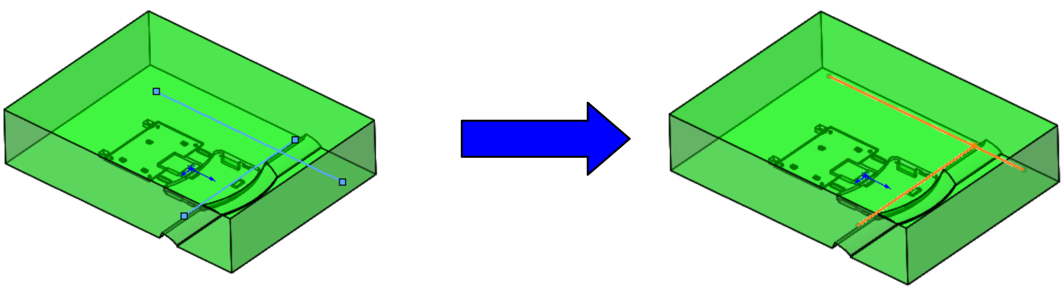




: Select a sketch line to remove

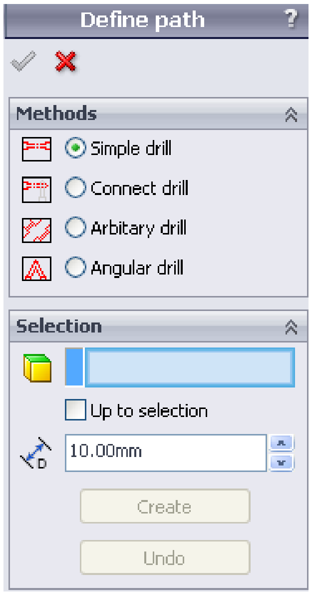


: Select two sketch line in sequence, this function will make the second one intersect with the first one.



For more flexible one, you can click **Free path** button to create it.

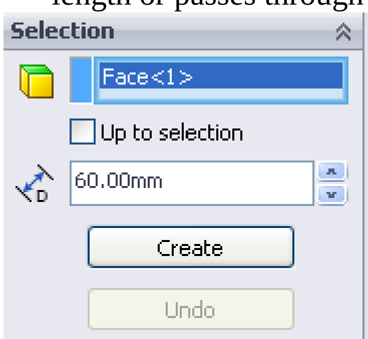
The following dialog will pop out.



Methods available for creating the cooling path：

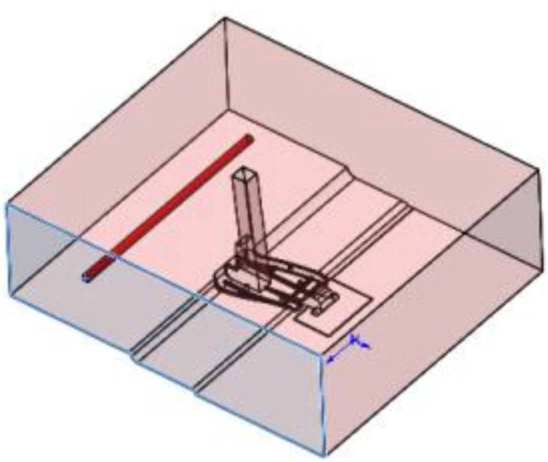
- Simple drill：
- Connect drill：
- Arbitary drill：
- Angular drill：

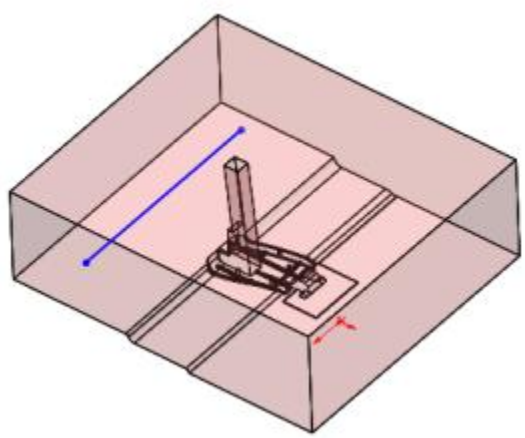
Simple drill： The cooling path is perpendicular to the selected face and extends to a certain length or passes through a certain face.



Drill from： Select a face as reference to construct the cooling path. The preview of the sketch line is displayed.

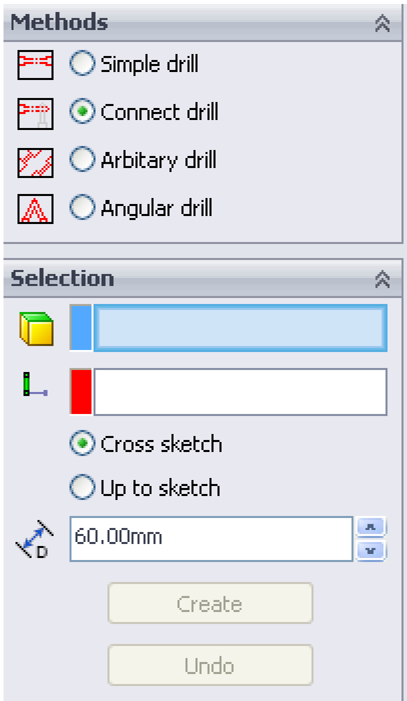
Drill length： Define the length of the sketch line.



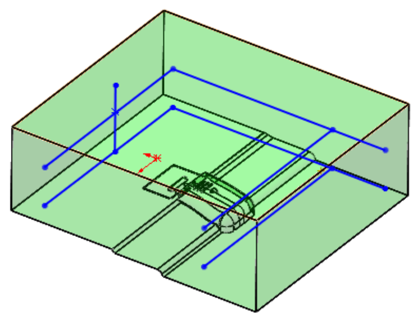


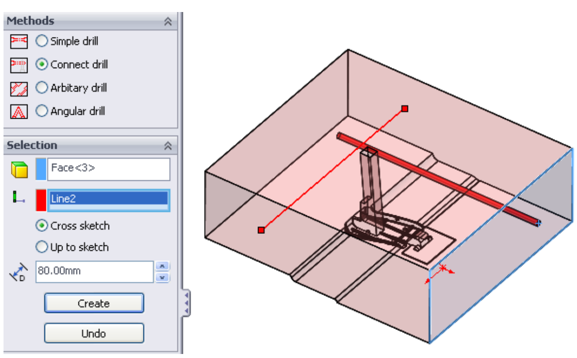
Click **Create** to create the sketch. Click **Undo** to cancel the last created sketch line.
Check **Select Up to selection** to build path between 2 faces.

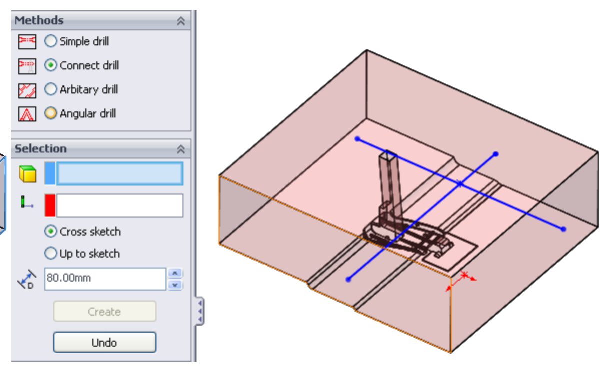
Connect drill: Create a sketch line that is connected to another sketch line



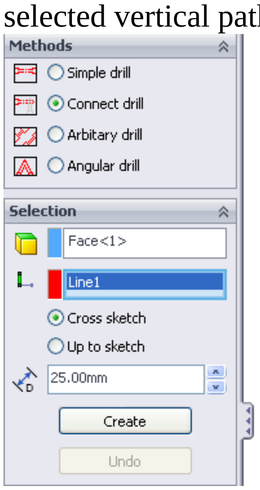
Drill from: Select a face as the starting face of a sketch line, the preview is displayed.
Select sketch segment: Select another sketch line as the connection.
Cross sketch: the newly generated sketch line pass through the connection path, but the length is defined by dimension.
Drill length: Define the length of the sketch line.

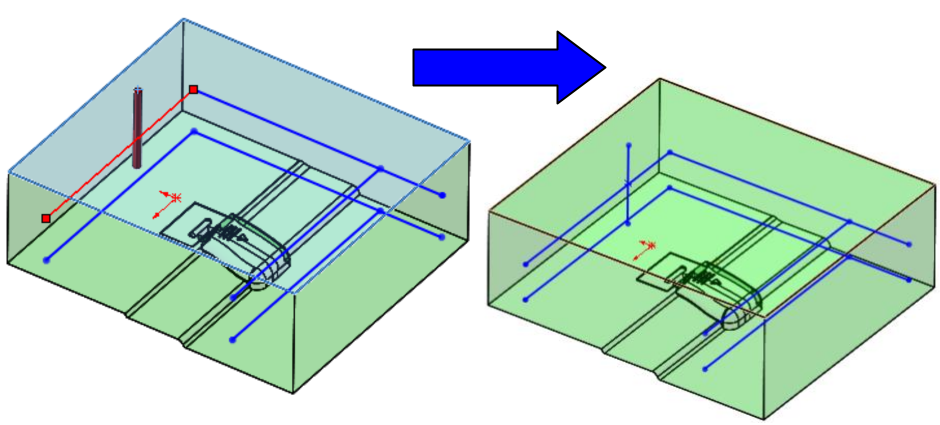




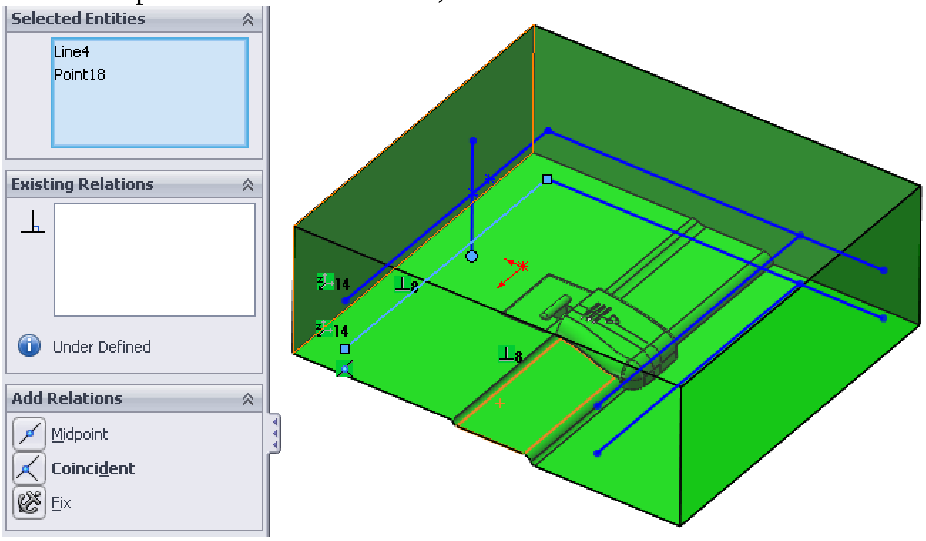


For a cooling path with many layers, Cross sketch is a useful tool。 For example: construct the selected vertical path passing through the two layers of horizontal cooling path.



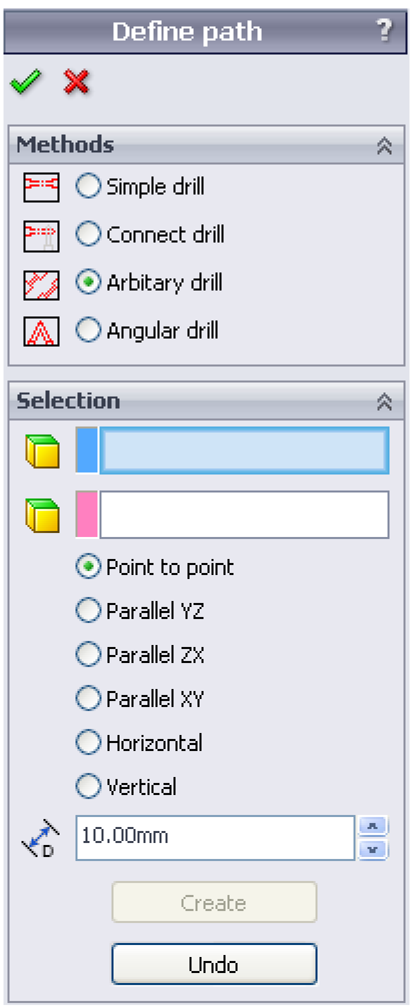


Use Solidworks sketch utilities to put more constrain. For example, edit the cooling sketch, Select the point and line as shown, select Concident in relation.

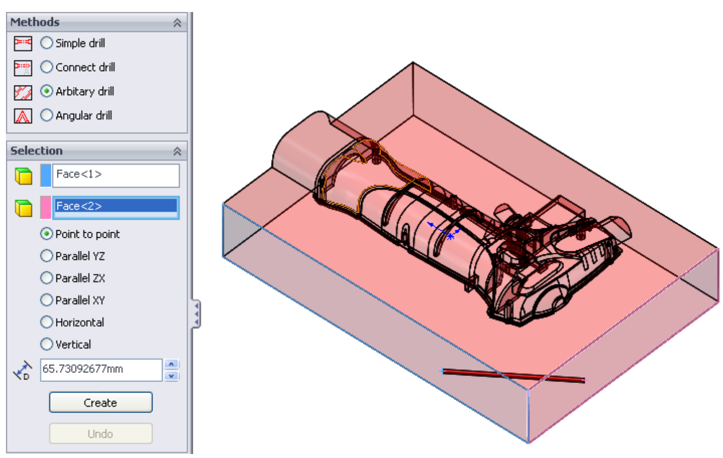


Up to sketch: The newly generated sketch line will end at the connection path.

Arbitrary dill: For the case that sketch line is not perpendicular to the selected face.



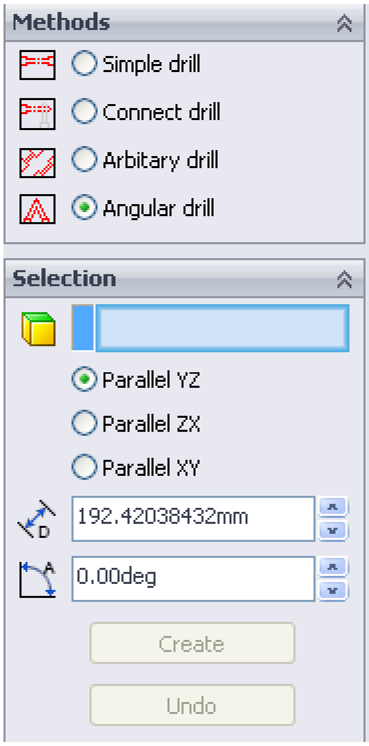
Drill from: Starting face of the sketch line to be created.
Drill to: Select a face as the ending face of the cooling path, the preview is displayed.
Point to point: Create the sketch line by connecting 2 selection points on the 2 faces.



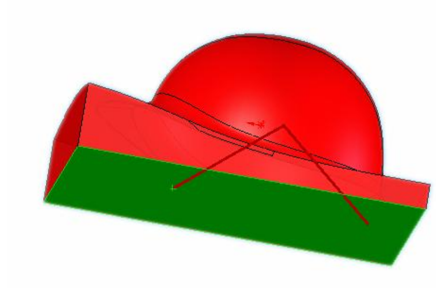
Parallel YZ: The sketch line is parallel to YZ plane
Parallel ZX: The sketch line is parallel to ZX plane
Parallel XY: The sketch line is parallel to XY plane
Horizontal: The path is parallel to the x-axis
Vertical: Created the path is parallel to the y-axis
Drill length: Define the length of the sketch line.

Note: If the selection is not appropriate, there may be contradiction of constrains and the sketch will be over defined.

Angular dill: V shaped connection

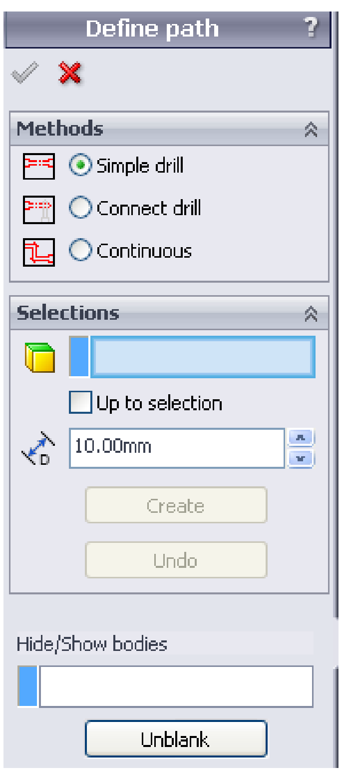


Drill from: Select a starting face of the sketch line
Parallel YZ: The generated path is parallel to the YZ plane
Parallel ZX: The generated path is parallel to the ZX plane
Parallel XY: The generated path is parallel to the XY plane
Drill length: Define the length of the sketch line
Drill angle: define the angle of the V-shaped connection.



Click **Create** to create the path.
After the setting of path is completed, click OK. The path will be built in a 3D sketch. If the accurate position of the path has to be defined, edit the 3D sketch to make it fully defined.

For assembly file, click **Free path** on the cooling wizard dialog, the following dialog pops out



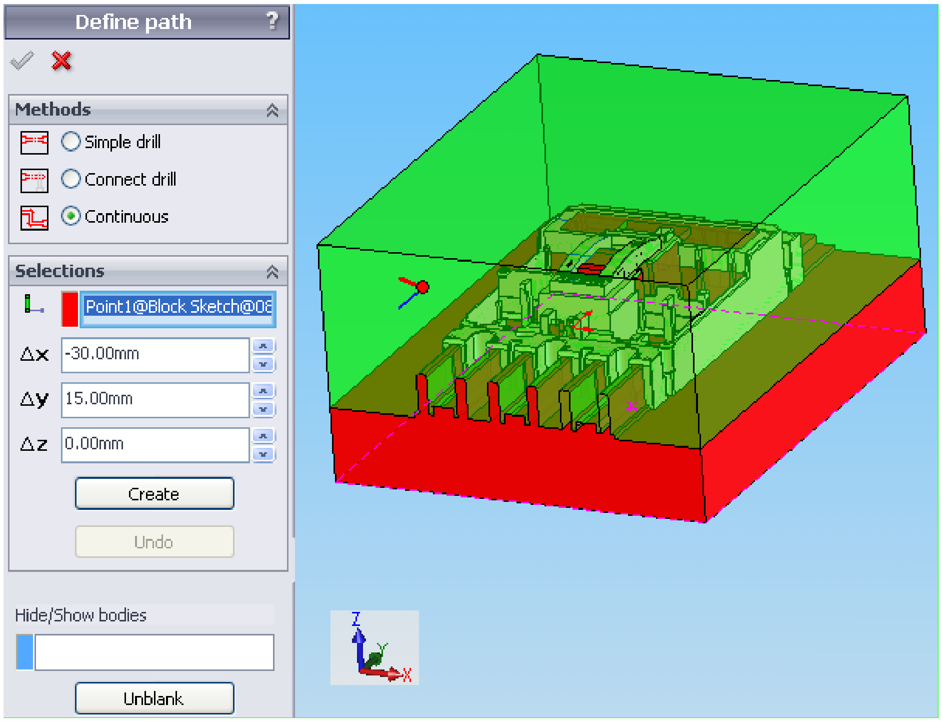
Methods available for creating cooling path on assembly

- Simple drill:
- Connect drill:
- Continuous:

The function of Simple drill and Connect drill for assembly file and part file is nearly the same. The difference is that for assembly file, the **Hide/Show** function is available hide or show the solid bodies.

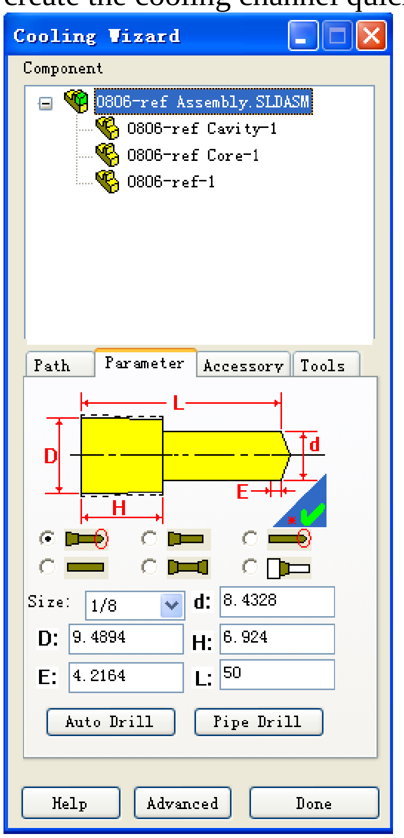
Continuous: Build the sketch of the path continuously.

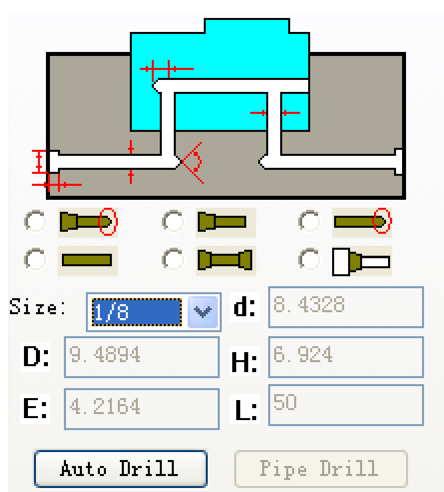
As shown below, select the point on a sketch line, input the value of extension in △X, △Y, △Z. translation in 2 or 3 directions is accepted. Change the value to its negative to reverse direction. The created line is the line joining the selected point and the translated point.



10.3 Parameters

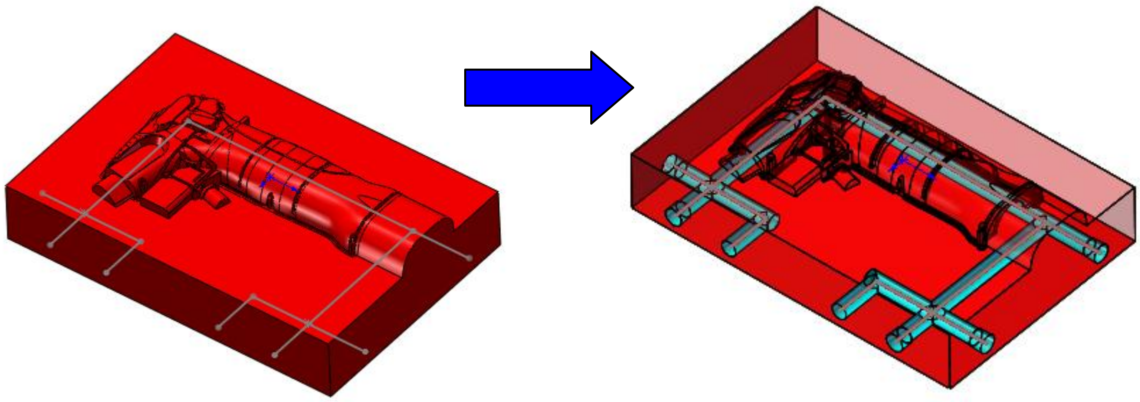
It is for building the cooling channels. On the **Parameters** page, **Auto Drill** and **Pipe Drill** can create the cooling channel quickly, for more complex cooling channel design, click **Advanced**.



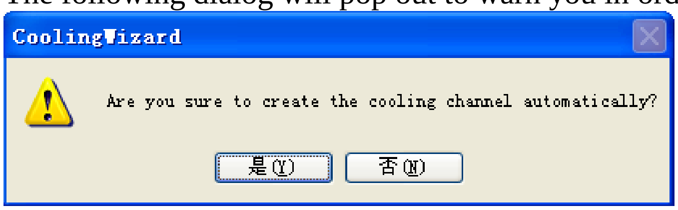


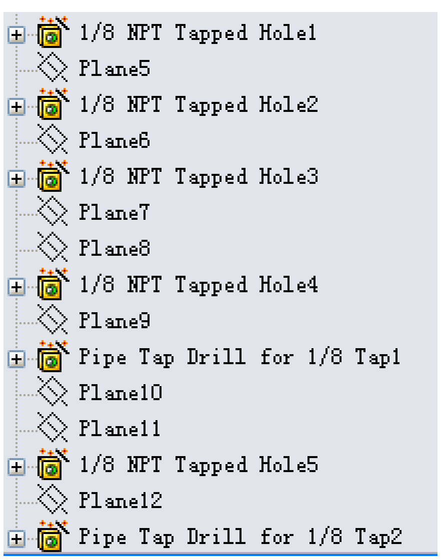
Auto Drill and **Pipe Drill** are used to create the cooling channels quickly.

Auto Drill: Based on the standard pattern created on the **Path** page, select the size, the system will use the internal rule to create the cooling channel automatically.



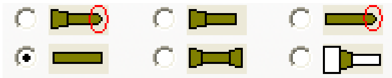
The following dialog will pop out to warn you in order to avoid the wrong operation.

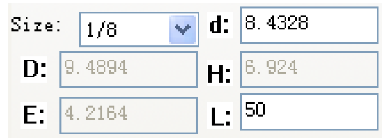




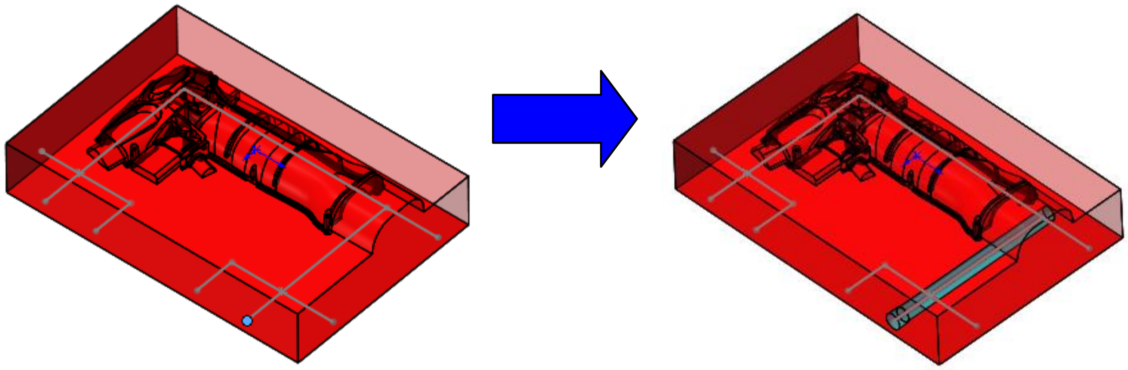
Pipe Drill: Create the cooling channel according the type selected.

There are six options for user to decide the channel type and parameters.
Parameters are available for customization.

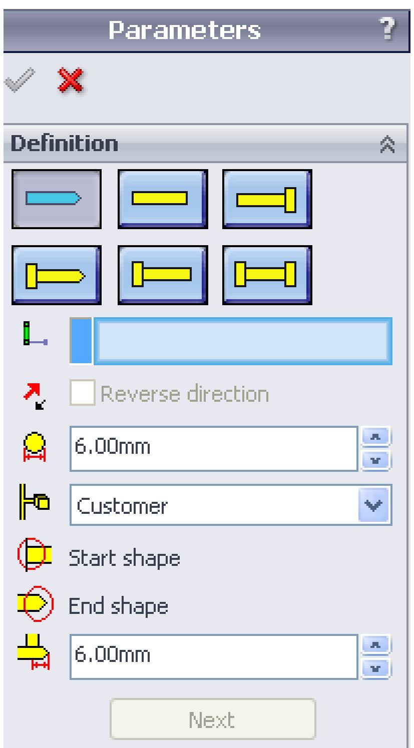


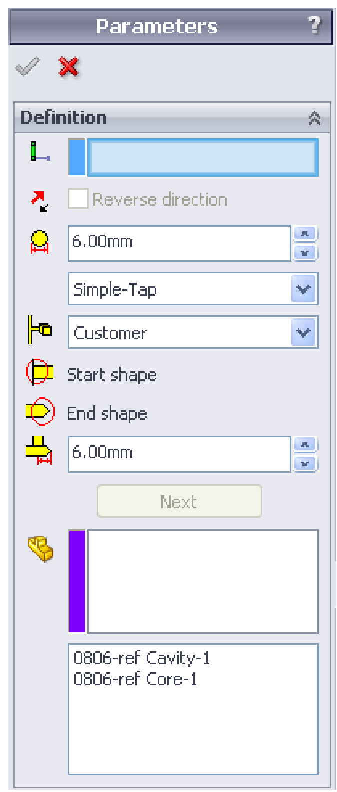


Select a sketch point on the cooling path, click this button, the system will create the cooling path.



Click **Advanced**， the following dialog will pop up, for part file and assembly file, the interface is slighty difference.





Working on the part model, the channel type as the six bitmaps shown on the dialog, corresponding parameters can be customized on the page as well.

Working on the assembly model, there are six options for cooling channel creation.
Simple-Tap： Simple hole and blind end
Simple-Simple： Both sides are simple hole
Simple-Counterbore： Simple hole and counterbore hole at another end
Counterbore-Tap： Counterbore hole with blind hole end
Counterbore-Simple： Counterbore hole with simple hole end
Counterbore-Counterbore： Both sides are counterbore holes

： Select the path on the screen， one path can only be selected every time
： Reverse the direction
： Define the Diameter of the water channel
: Select the configuration of the channel diameter. Select a type, other parameters will change corresponding to the type. User can check the data of different type of cooling channel in the installation directory res \ Cooling Parameters.txt, configuration can be added and changed also.

	A	B	C	D	E	F	G	H
1		Diameter	Start_diameter	Start_depth	End_diameter	End_depth	Extension	Angle
2	Customer	6	12	15	12	15	10	118
3	6mm	6	12	15	12	15	10	118
4	10mm	10	20	15	20	15	15	118
5	12mm	12	25	15	25	15	20	118

Start shape： Type of starting shape of cooling channel, which includes Simple 、 and Counterbore
When Counterbore is selected

： Counterbore diameter
： Counterbore depth

End shape： Three end types, there are Tap, Simple and Counterbore

When **Tap** is selected for the End shape：

： Over drill length
： Angle at bottom

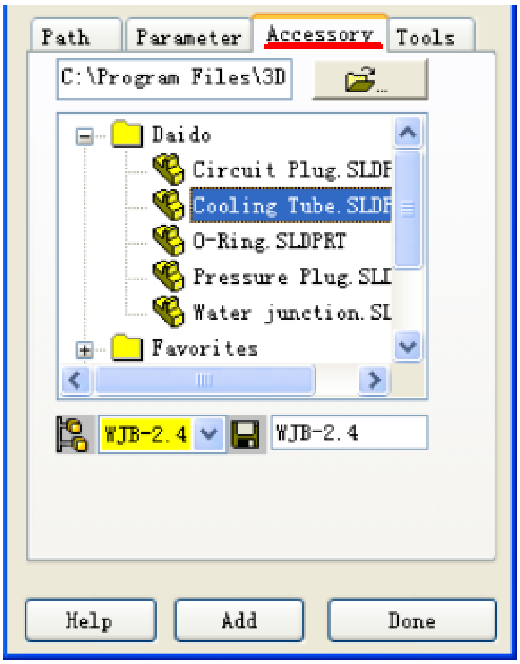
When **Counterbore** is selected for the End shape

： Counterbore diameter
： Counterbore depth

After setting a segment, click **Next** to proceed to set the other segment.

10.4 Accessory

Some typical standard parts for cooling such as End plug and O-ring are provided. The adding of these parts is required to perform in assembly files.



: Use this command, you can change your default library folder such as share folder in the network.

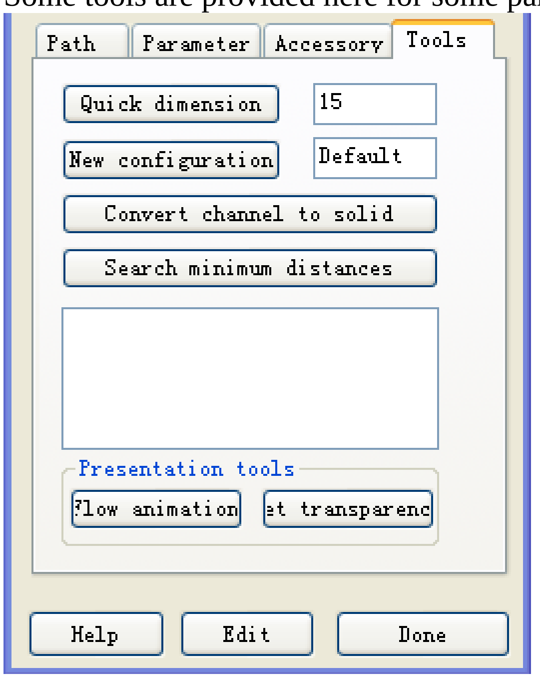
To add standard cooling accessories quickly, you need pre-select a circular edge for most parts.

When a part is selected, the corresponding bitmap is shown on the top of the dialog. Configuration list is updated accordingly.

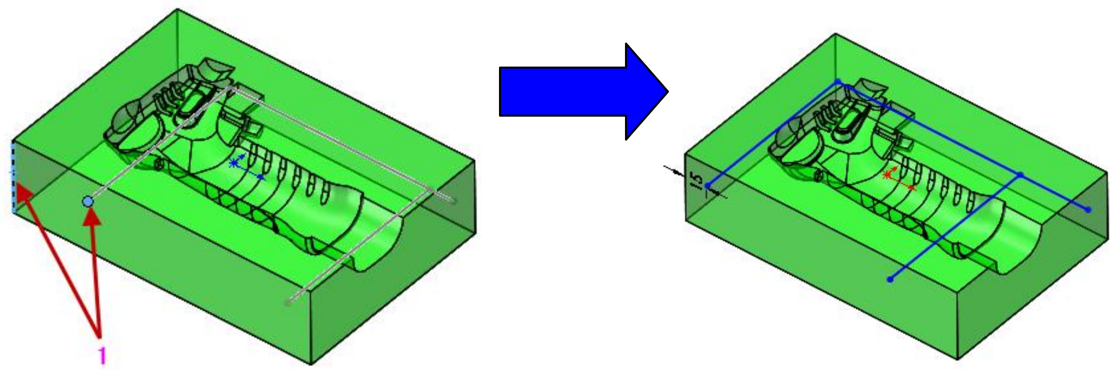
Select your desired configuration from the drop-down list, and key in your own name if needed, by default, the configuration name is the file name to save.

10.5 Tools

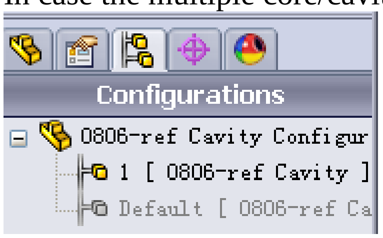
Some tools are provided here for some particular purpose use.



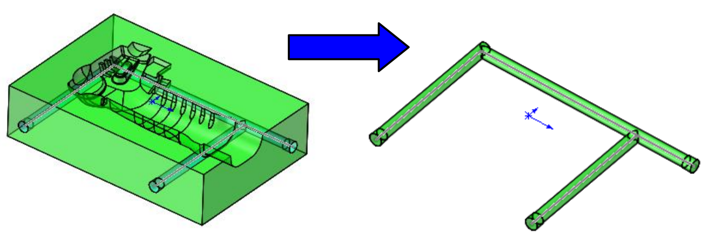
Quick dimension: Select a sketch point and a linear edge, add dimension quickly.



New configuration: Create a new configuration on the working part.
In case the multiple core/cavities have different cooling channels, this function is useful.

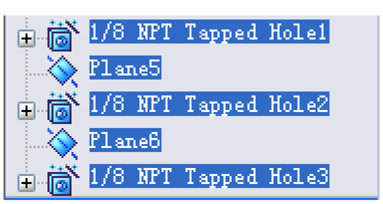


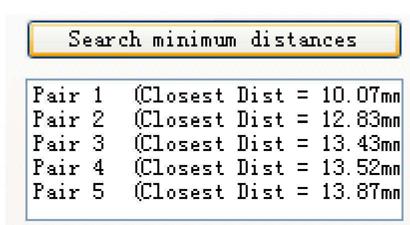
Convert channel to solid: Select all cooling channels in the feature tree, click this function, a solid body is created. This function is normally used to help to do the CAE analysis or do some interference checking at down stream process.

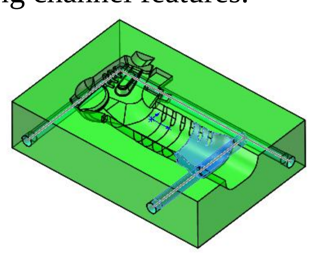


Search minimum distance:

This function is mainly used to detect if there is any interference or potential interference between the cooling channels and the sub-inserts and ejectors. The minimum 5 distances are liste on the page. To perform this function, you need select the cooling channel features.

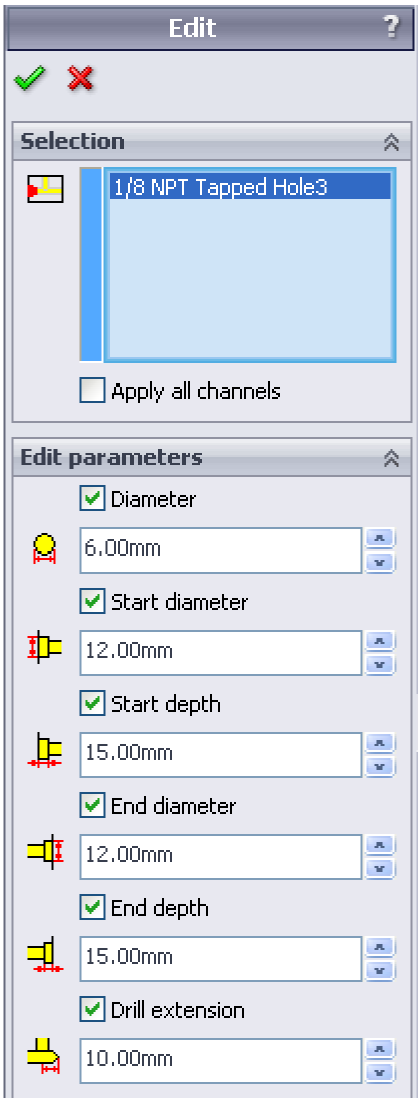






Pick up one of the listed five pairs of faces, two faces are highlighted to show where the interference or potential interference area is.

After cooling channel is created, user can click  to edit the Parameter. The dimension of the cooling channel can be edited here. The position and the relation can be edited using the sketch function of Solidworks.

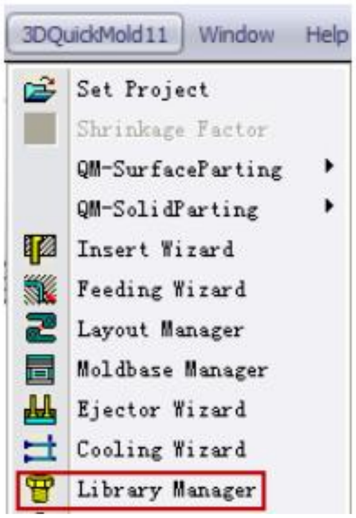


Apply all channels: Apply the changes to all the cooling channels.

Selection: Select the cooling channel in the feature tree or pick up the faces on the graphic.

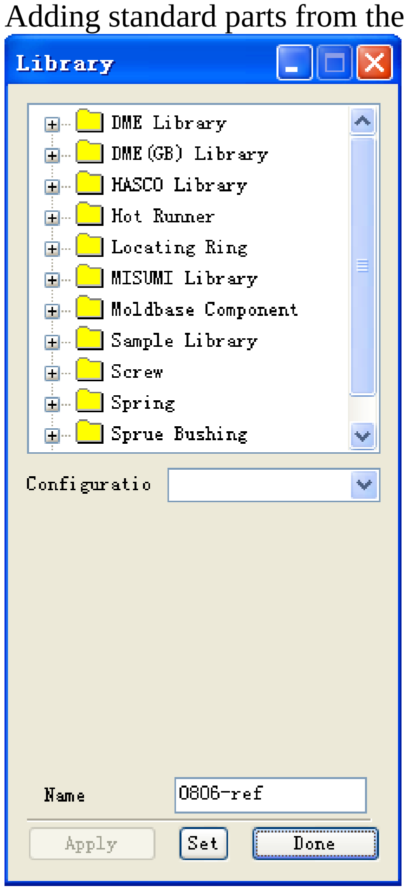
Set transparence: Set the selected face as transparence.

Chapter 11. Library Manager



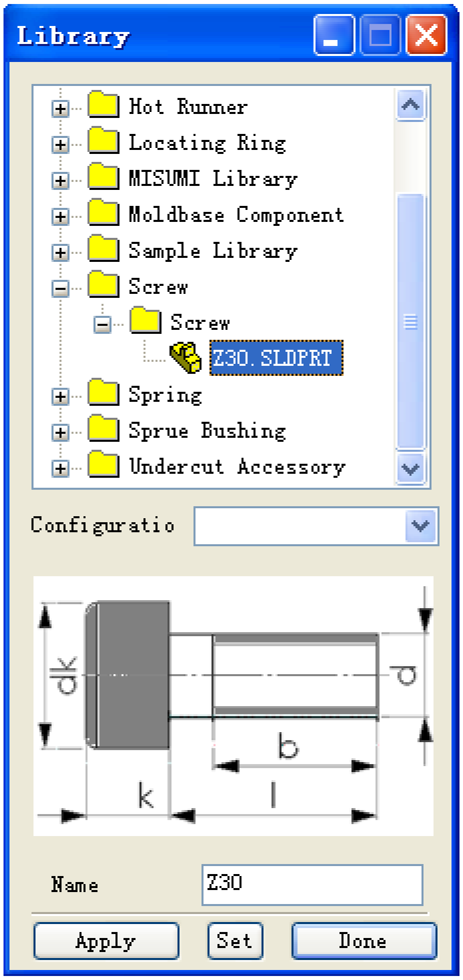
To increase the design efficency and reduce the duplicating procedure, 3DQuickMold provides a standard library manager. Those libraries include locating ring, sprue bushing, screw, spring and parts for slide and lifter.

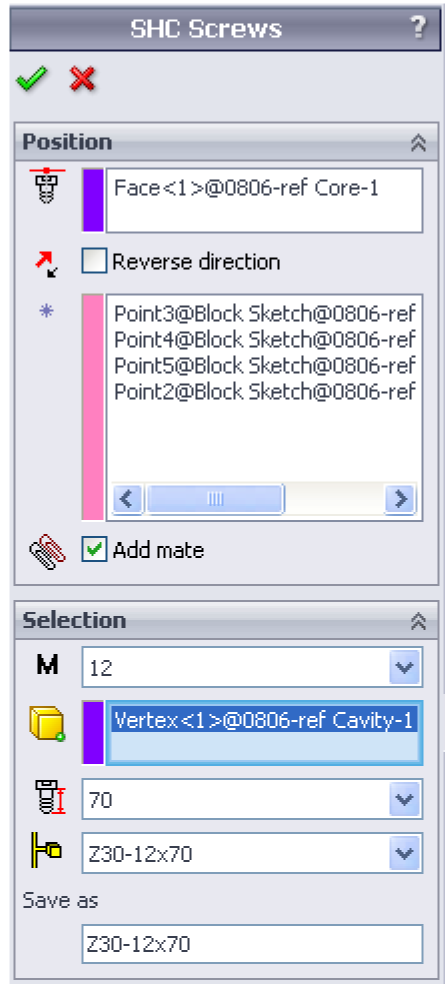
Adding standard parts from the Library to mold assembly can only be used in Assembly file.



11.1 Adding Screw

To add a screw, select a type, a part bitmap is shown. Click **Apply**, the property manager of the screw pops out, the size, position, plane and relation can be defined.





: Screw mate reference, please select a planar face

: Reverse the screw direction

: the position of the screw, this should be a sketch point.

: mate point: When this option is checked, a concentric condition is applied to the the screw and the selected point

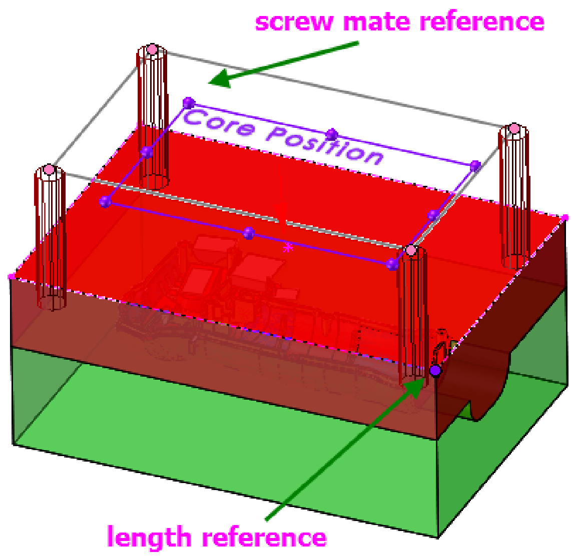
M : The screw diameter

: A reference vertex to determine the screw length

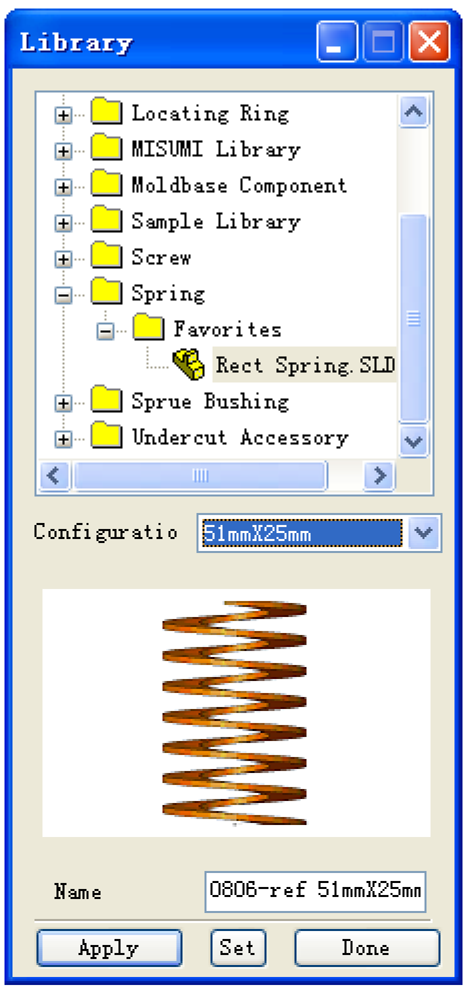
: length of the screw

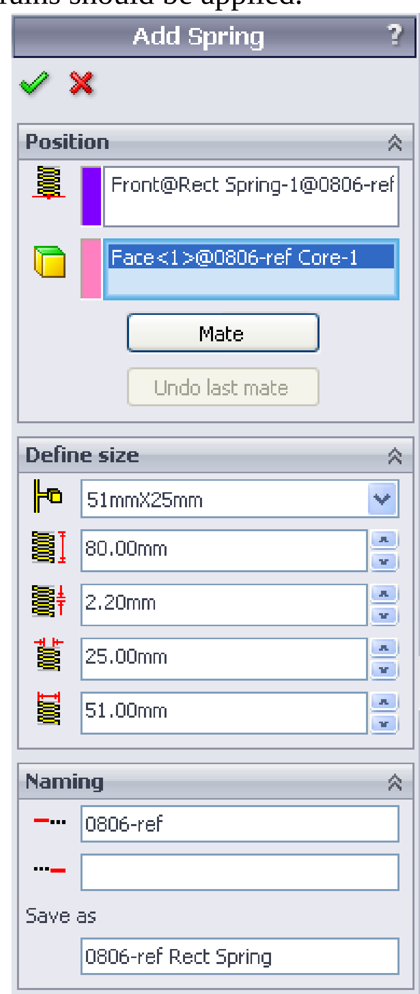
: Screw configuration : can be obtained by selecting the length and diameter, once a new configuration is selected, the screw length and diameter will be updated accordingly.

Save as: Screw name to save



After the screw is generated, corresponding pocket can be done by using Pocketing in Mold tools.
2. Adding spring， To add a spring, at least, two mate constrains should be applied.





Position

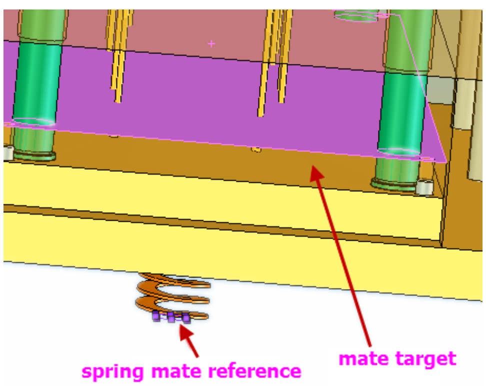
：mate reference. It could be the spring origin or the XY plane (Front) in the spring model.
：target mate. Mate reference on the assembly model. If the above selection is the spring origin, this face should be a cylindrical face. If the above selection is a plane, accordingly, a planar face is required to be picked up here.

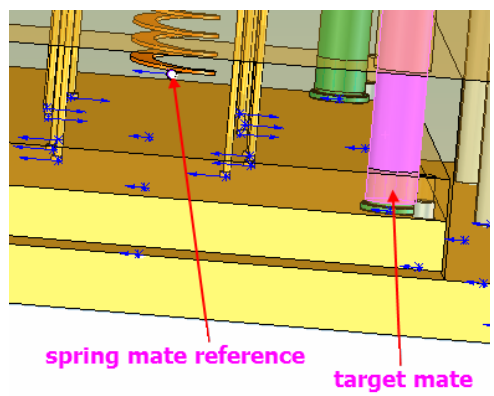
Size
：spring configuration
：length of spring
：Thickness of spring
：inner diameter of the spiring
：outer diameter of thr spring

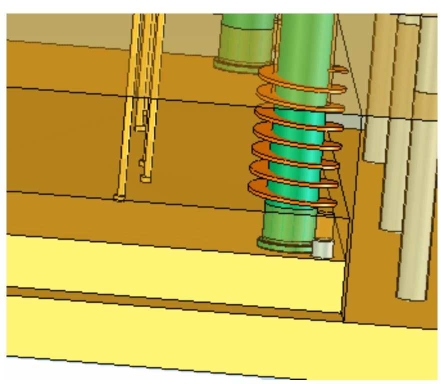
Naming
：Prefix
：Suffix

Save as：spring name

Example：Add spring on the return pin
First mate: Reference plane + Planar face
Second mate: Spring origin + Cylindrical face



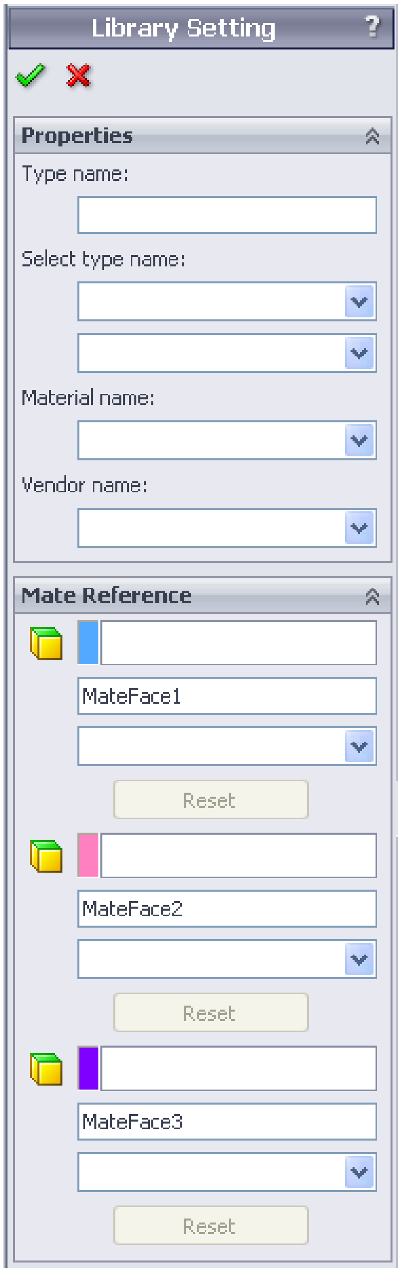




3. Set standard components

Users can customize their own standard components by this utility.
SolidWorks feature can be used to build standard components, make sure the models meet two basic requirements:

- All key dimensions are easy to be modified
- Configuration way to handle different standard sizes



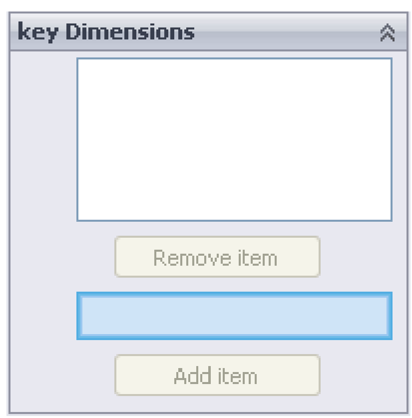
Type name： Select or key in user-defined type name
Select type name： Type name that will appear in BOM
Material name： Select from the drop down list or input
Vendor name： Select from the list or input

Mate Reference1： Reference plane, face for mate reference
Name1: Name for the first mate reference

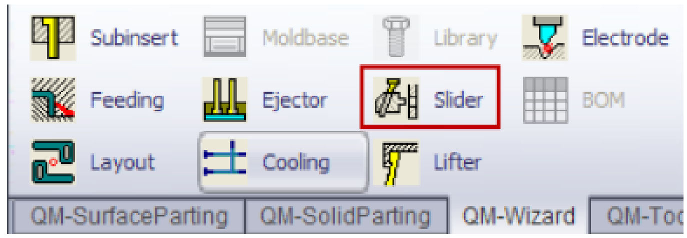
Mate Reference2： Reference plane, face for mate reference
Name2: Name for the second mate reference

Mate Reference3： Reference plane, face for mate reference
Name3: Name for the third mate reference

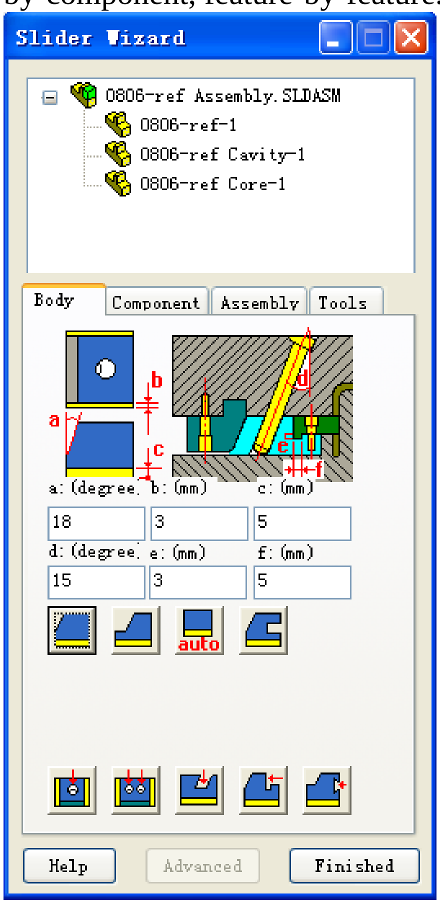
Key Dimensions： Select the key dimension for standard components, click Add item to add.



Chapter 12. Slider Wizard



Customer can chose the 3DQuickMold built-in slide, or build their own slide design component-by-component, feature-by-feature.



There are two different methods to design slide

One is to use the whole set of standard parts, where the angular pin, guide rail and slide body are all imported to the assembly. Standard slide are of this method, this approach could be seen in many other mold design solutions, it looks quite effective, but less flexiability.

Another method is to add parts one by one which enhances the flexibility. It is implemented on the Component page.

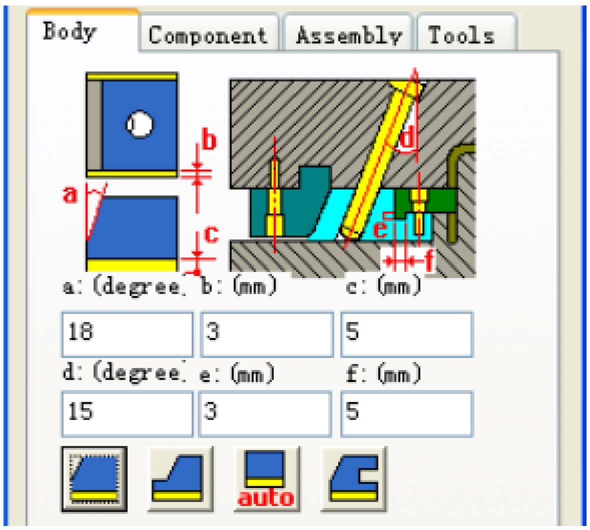
These two methods both have their own advantages, users can choose between them according to the need.

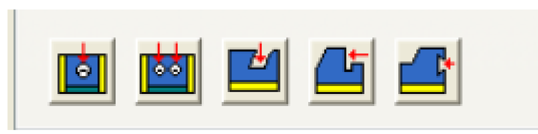
12. 1 Component Navigator

Please refer to the chapter in Insert wizard

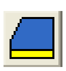
12.2 Body

Create the slide body by feature, detailed dimensions could be modified by SolidWorks way.



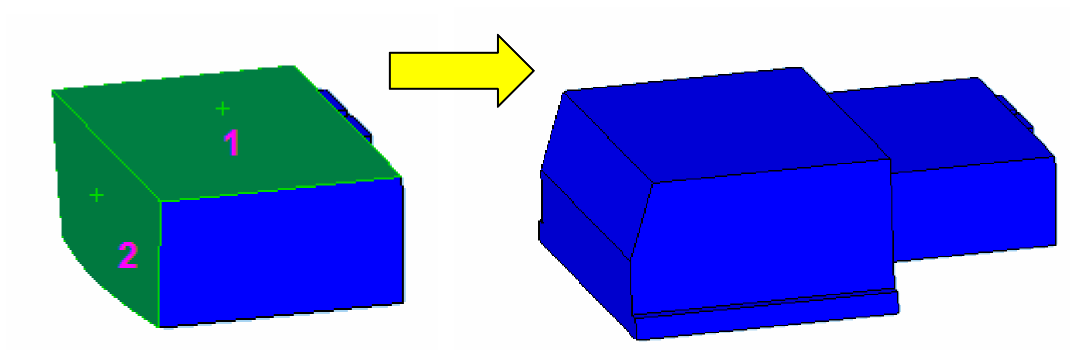


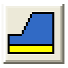
This function applies to part only

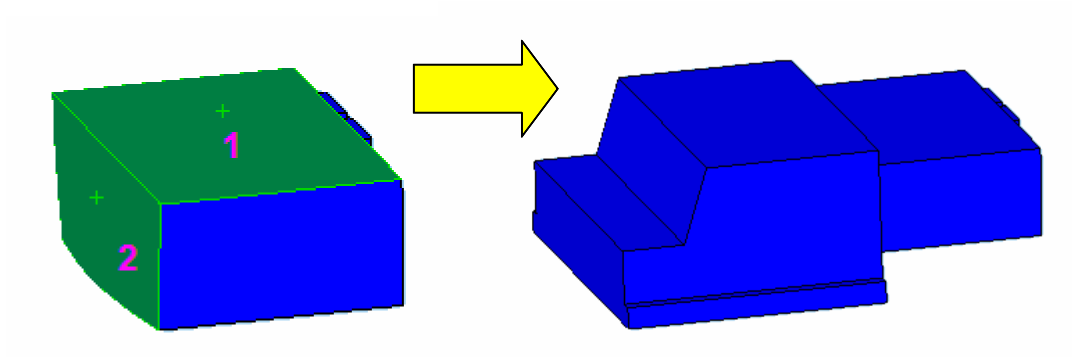
: Slider body type 1

Method: select two planes, click the icon, the first plane determines the orientation of slider body, the second one is the face that the slider body will attach to.

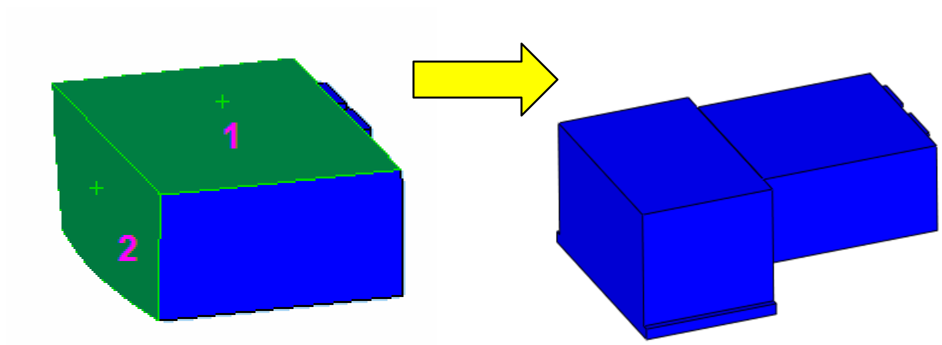
Selections as follows in order, Click the icon, the slider body will be created as the following picture shows.

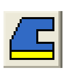


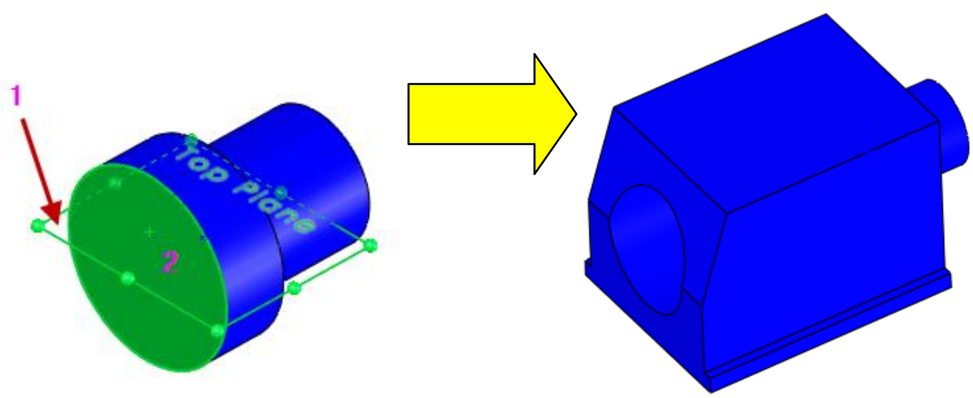
: Slider body type 2
Method: same as type 1, A different slider body will be created

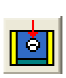


: Slider body type 3
Method: same as type 1, A different slider body will be created

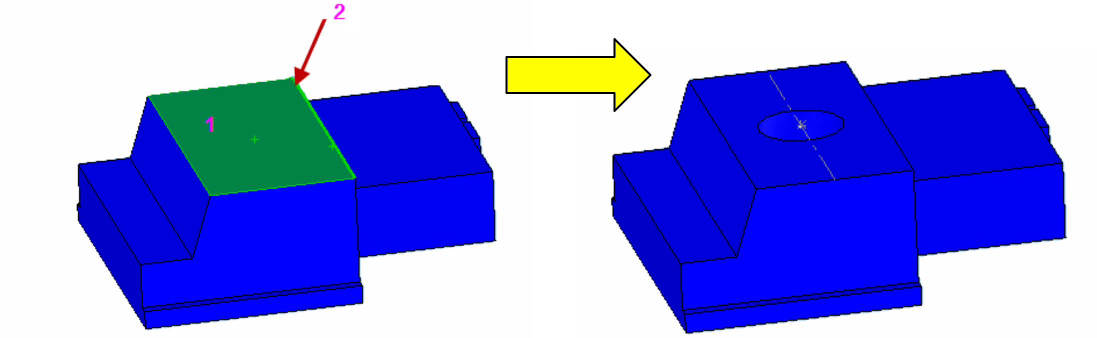


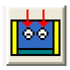
: Slider body type 4
Method: same as type 1, A different slider body will be created

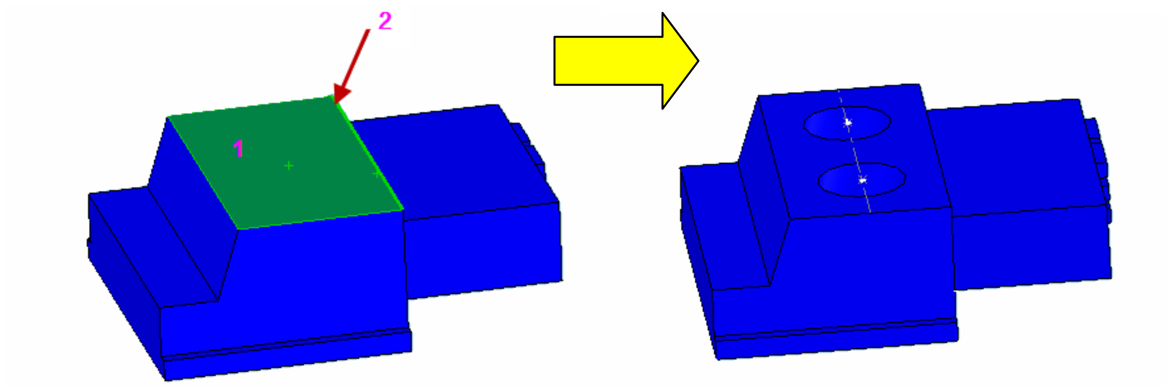


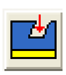
: Create a hole on slider body for angular pin.

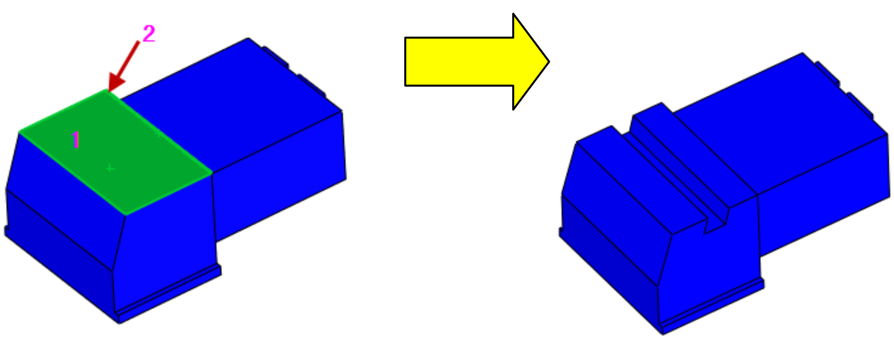
Method: select a face and an edge in sequence as follows, click the icon.

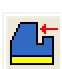


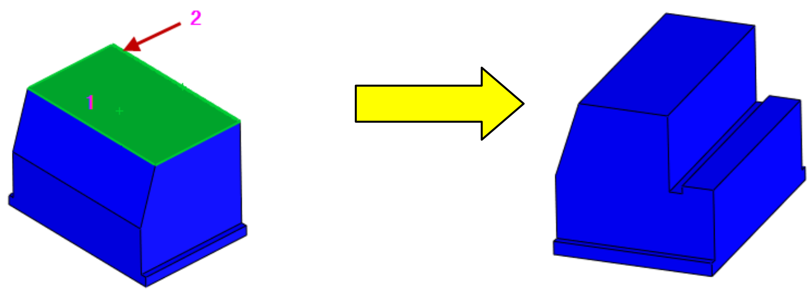
: Create two holes on slider body for angular pins.
Method: Select a face and an edge in sequence as follows, click the icon.

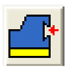


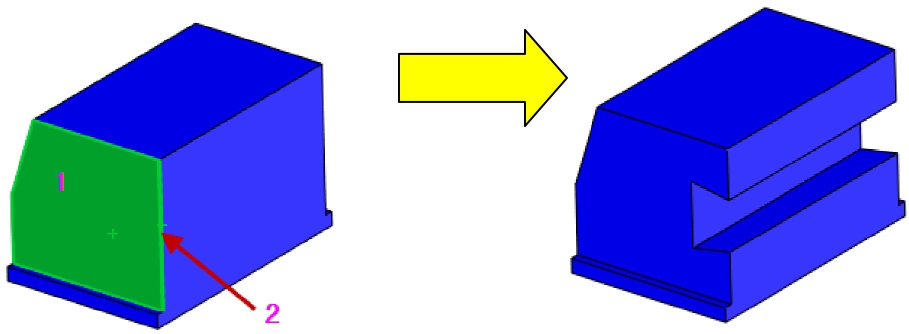
: Slot type 1 for finger cam
Method: Select a face and an edge in sequence as follows, click the icon.



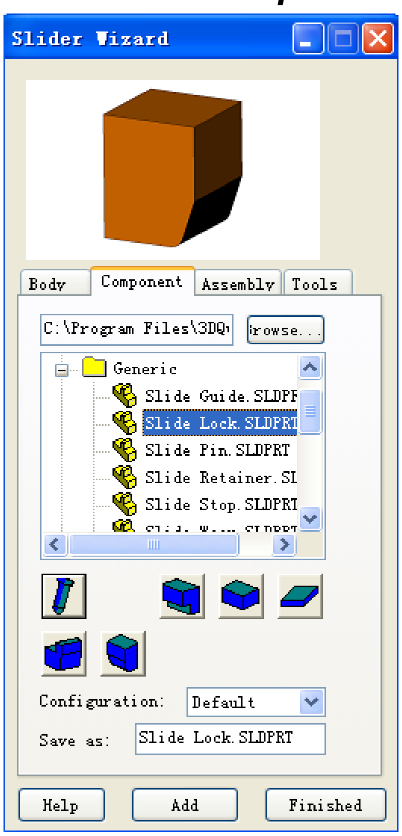
: Slot type 2 for connection of slider body and slider head
Method: Select a face and an edge in sequence as follows, click the icon.



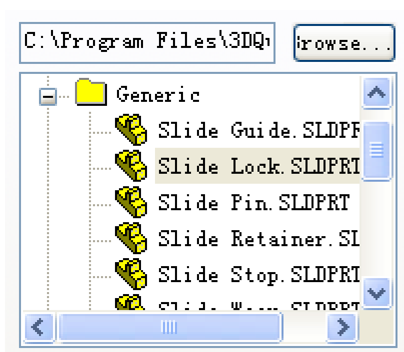
: Slot type 3 for connection of slider body and slider head
Method: Select a face and an edge in sequence as follows, click the icon



12.3 Component



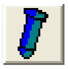
This page contains functions to add standard and non-standard slider components



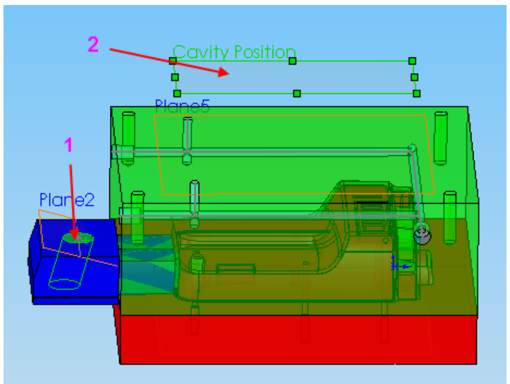
Click browse button to change your library folder if you have your own standard slider components apart from the installation folder

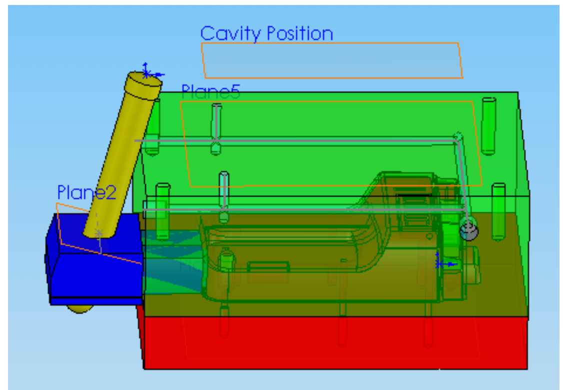
Select a component on the tree, the part is pre-viewed on the top of the dialog, click Add, a dialog will pop up to guide you to mate the component to the current assembly.

Configuration: All available configurations is listed for your option
Save as: Name to save

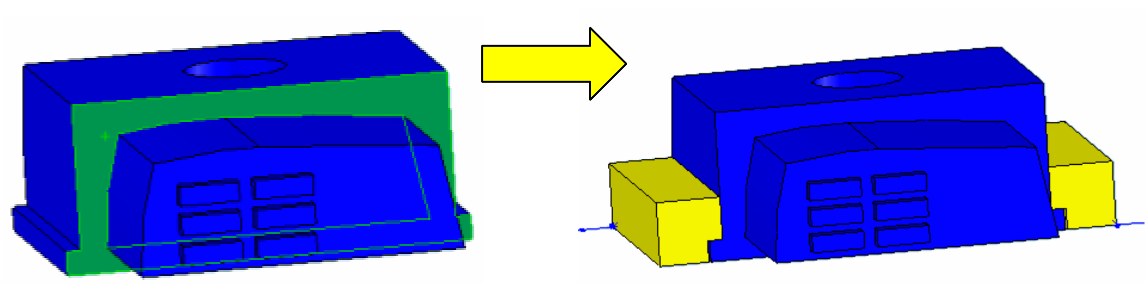
: Add angular pin for slider

Method: select the cylindrical face and a reference plane in sequence as follows, click the icon.

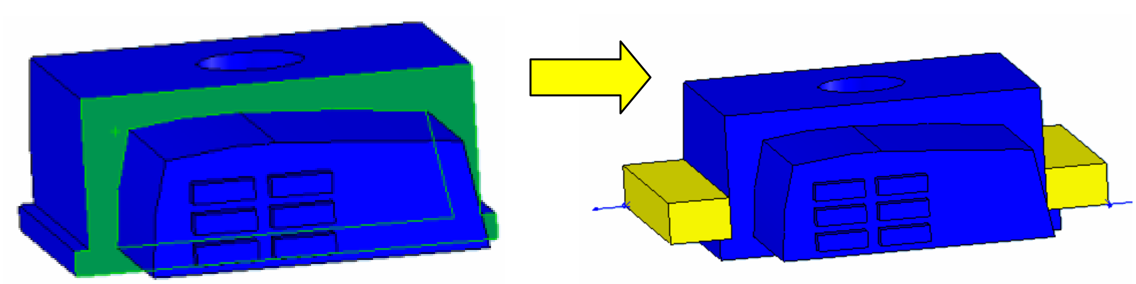


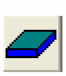


: Add guide rail type 1
Method: select the face as shown below, click the icon.

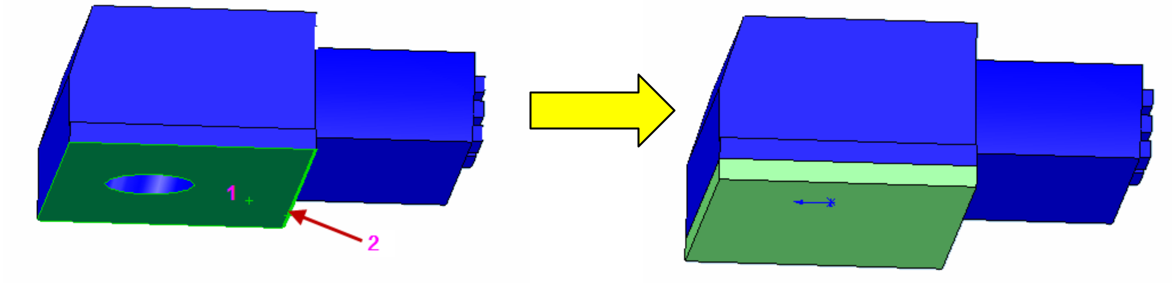


: Add guide rail type 2
Method: Same as the above



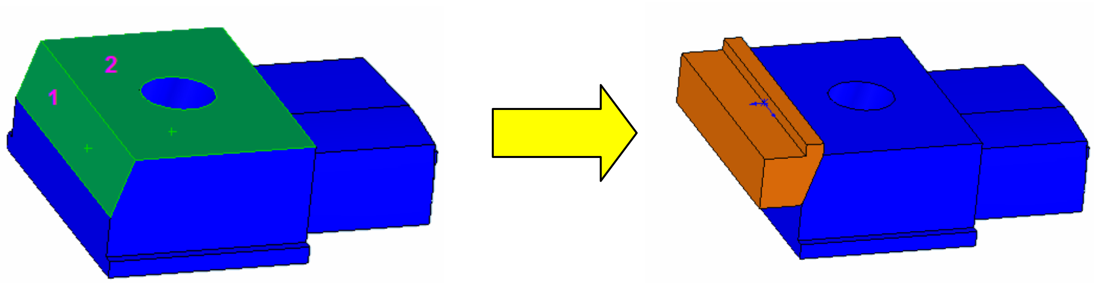
: Add wear plate

Method: select a face and an edge as follows, click the icon.



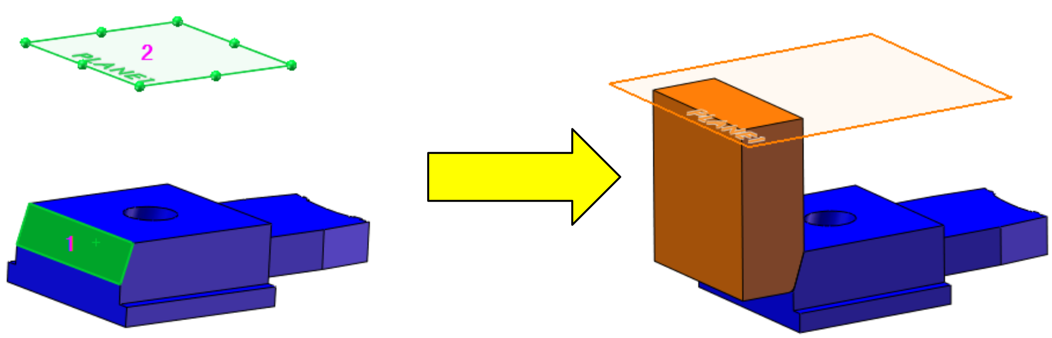
: Add locking block type 1

Method: Select two faces in sequence as follows.



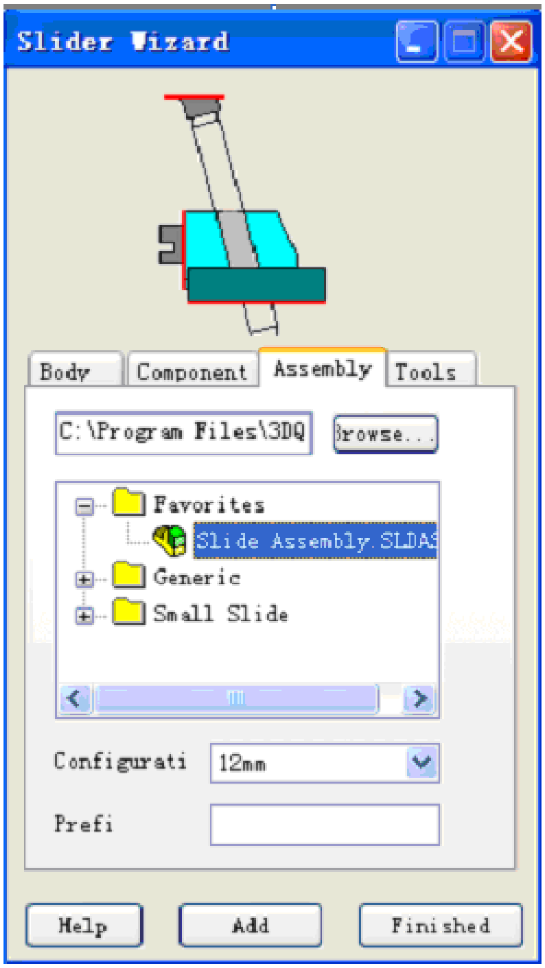
: Add locking block type 2

Method: Select a face and a reference plane in sequence as follows.



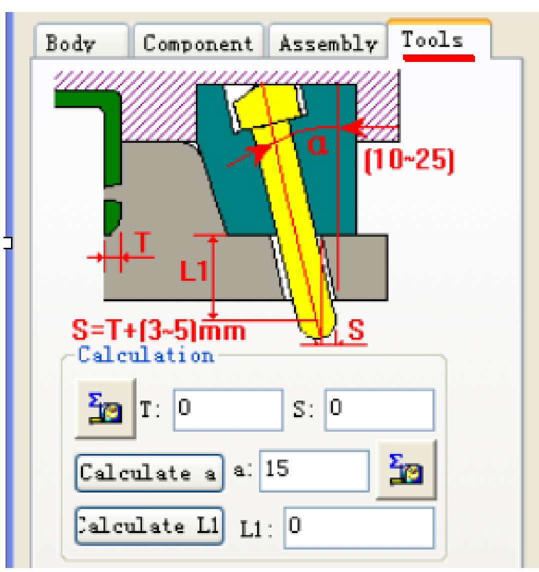
12.4 Assembly

Some pre-defined standard slide structures for use.



12.5 Tools

Some calculation tools for slider design.

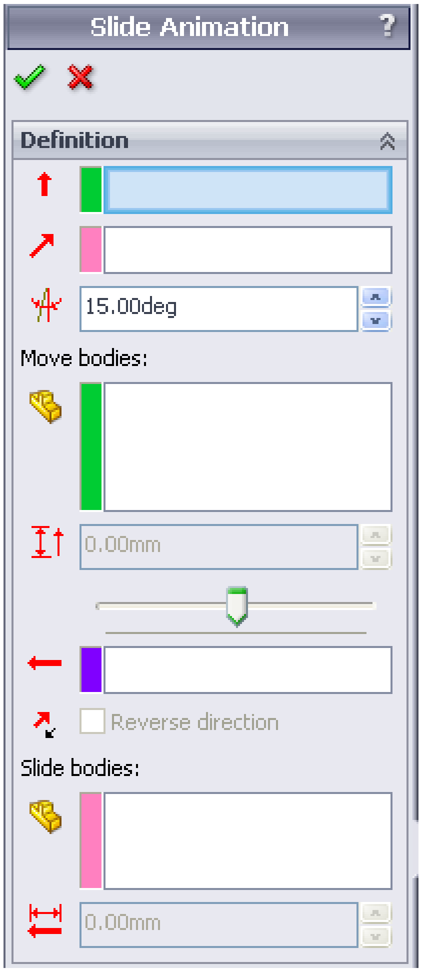


T: Stroke distance, obtained by measure

S: Obtained based on the T value

L1: By calculation
a: By calculation

Click the Simulation to the animation dialog



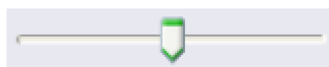
:Open direction, select planar face to define

:Pin direction

: Pin angle, the angle between the pin direction and open direction

: Bodies to move along the mold open direction, normally, it includes the angular pin, cavity and cavity plate.

: Open distance, drag the below slider control will change this value dynamatically.

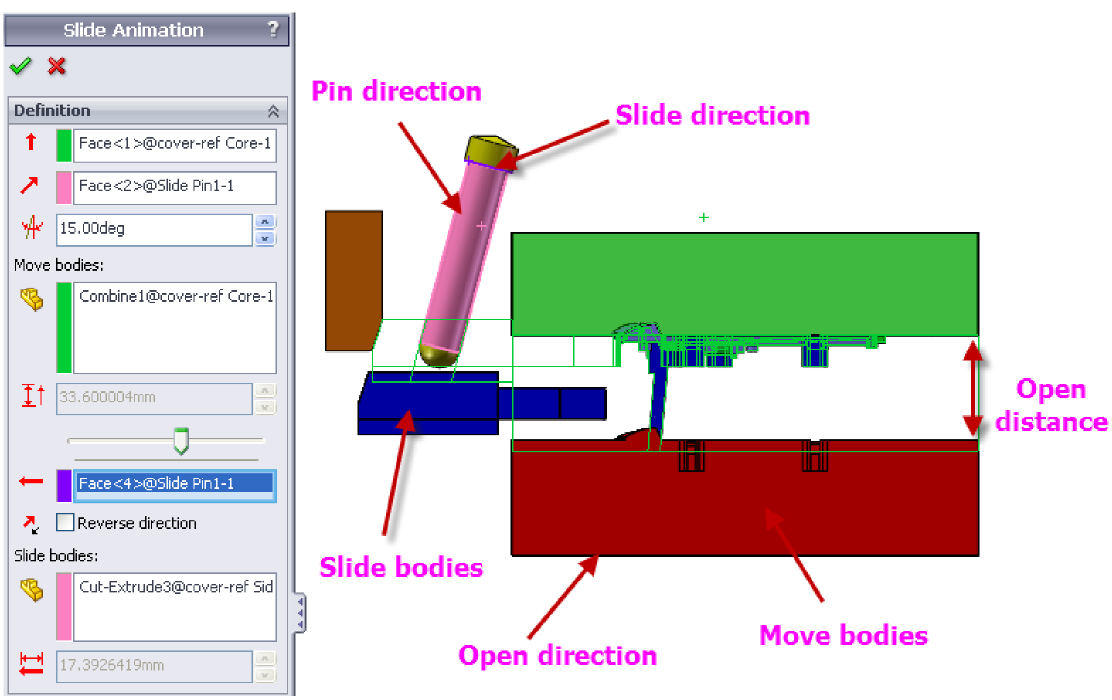


: Slide direction, select a planar face to define

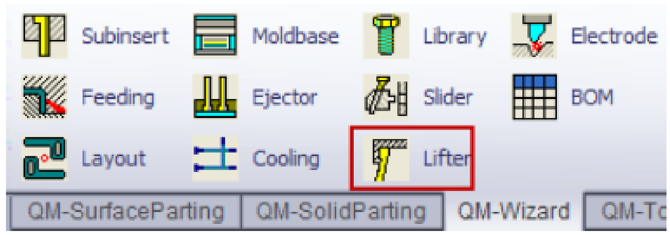
: Reverse direction if needed

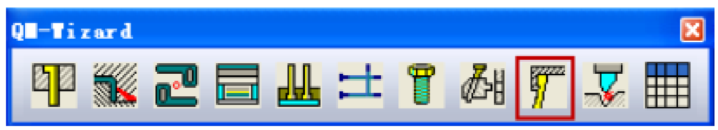
: Slide bodies to along the slide direction

:Release distance

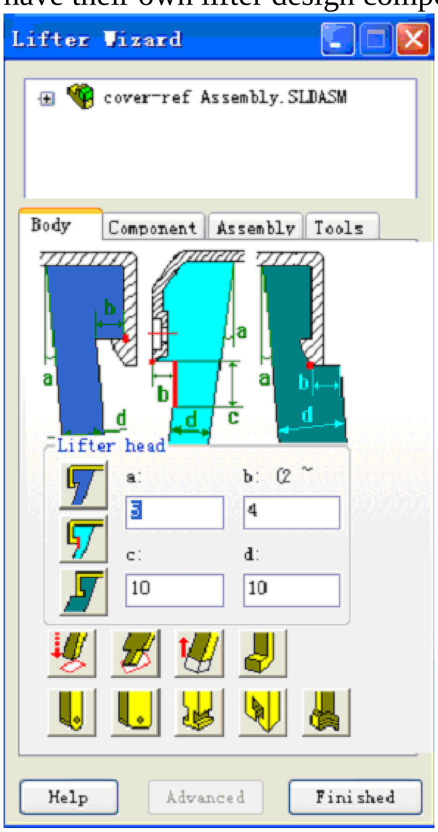


Chapter 13. Lifter Wizard





Customer can chose the 3DQuickMold built-in undercut mold release mechanism, user can also have their own lifter design component-by-component, feature-by-feature.



There are two different methods to design lifter

One is to use the whole set of standard parts, where the lifter pin, lifter house and wear plate are all imported to the assembly. Standard lifter is of this method, this approach could be seen in many other mold design solutions, it looks quite effective, but less flexiability.

Another method is to add parts one by one which enhances the flexibility. It is implemented on the Component page.

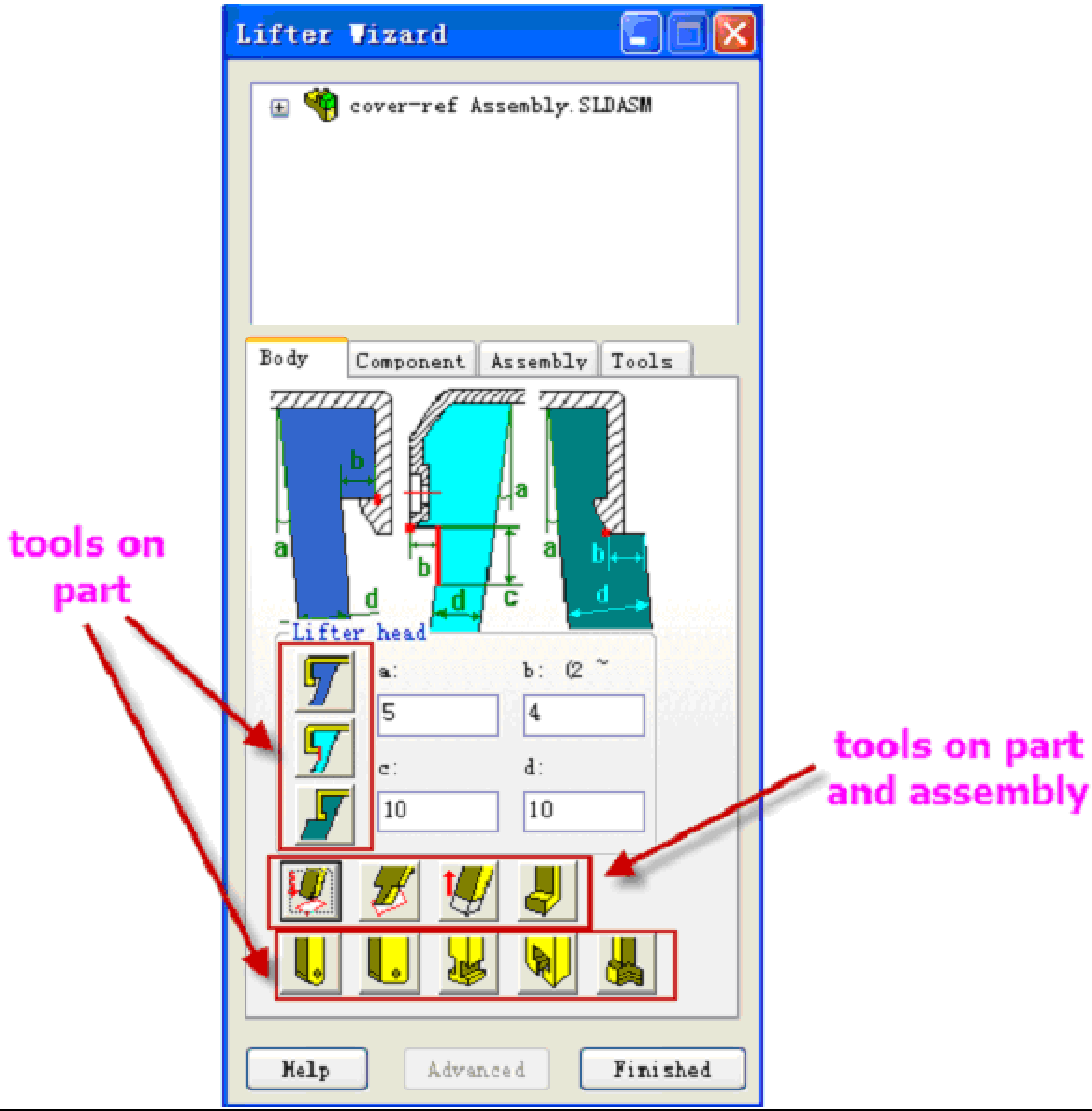
These two methods both have their own advantages, users can choose between them according to the need.

13.1 Component navigator

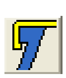
Please refer to the chapter in Insert wizard.

13.2 Body

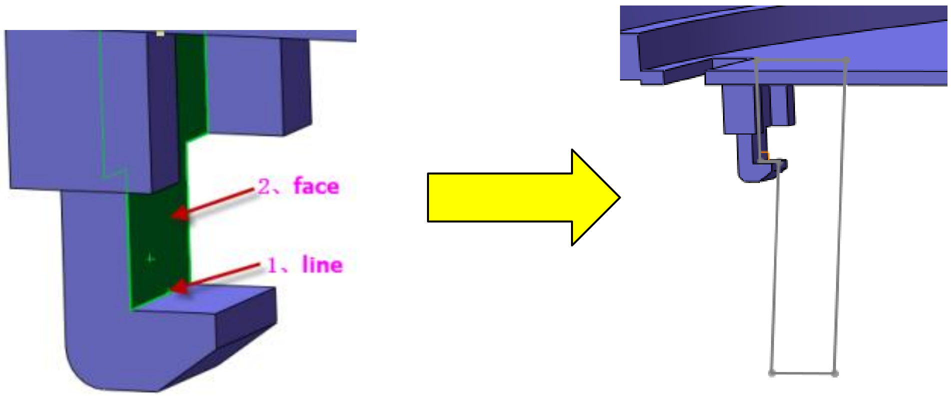
There are some tools for creating the profile of lifter head and tools for lifter body modification. Users can use SolidWorks function to do detailed dimensions.

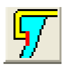


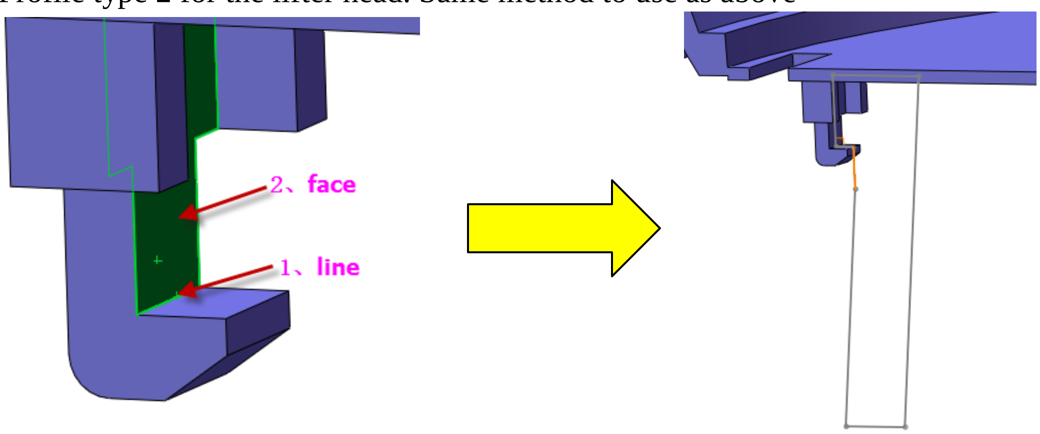
Most of the above functions applies to part only.

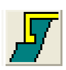
: Profile type 1 for the lifter head

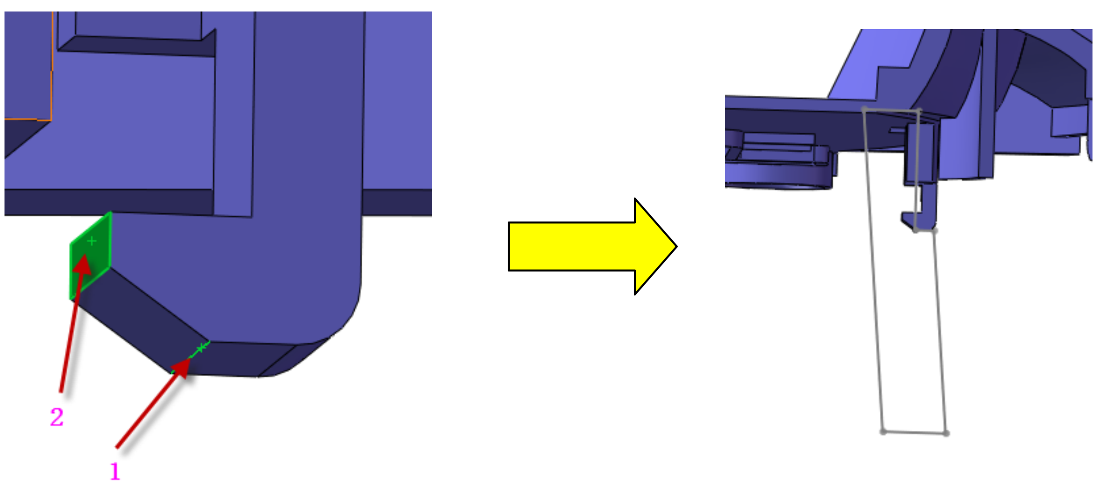
Method: Select one linear edge and a planar face, click this icon
Selection and result as the following picture shown.



: Profile type 2 for the lifter head. Same method to use as above



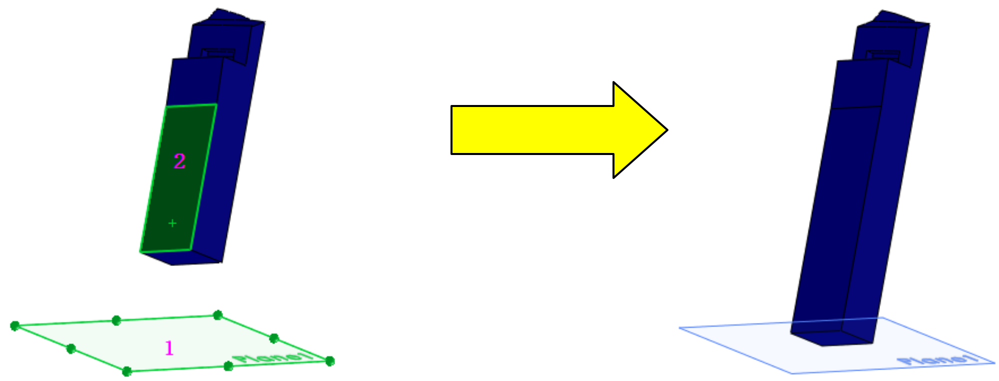
: Profile type 3 for the lifter head. Same method to use as above



The following 4 icons can be used on the assembly or part.

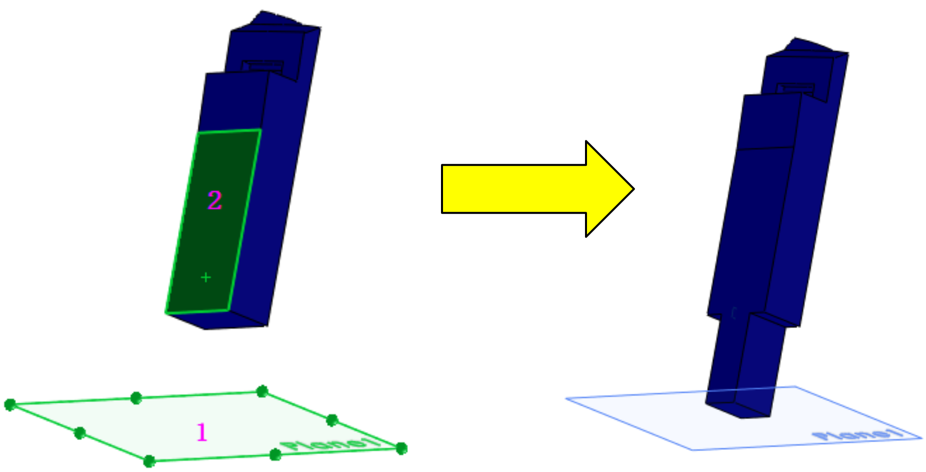
: Extend lifter body to the selected plane

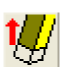
Method: select a reference plane and the inclined face on lifter body in order, click the icon. As shown below.



: Extend lifter body to the selected plane but using a narrower profile.

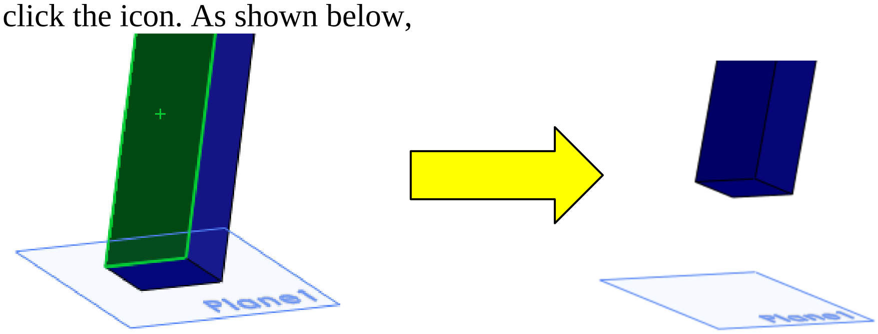
Method: Select a plane and an inclined face on the lifter body in order, click the icon. As shown below.



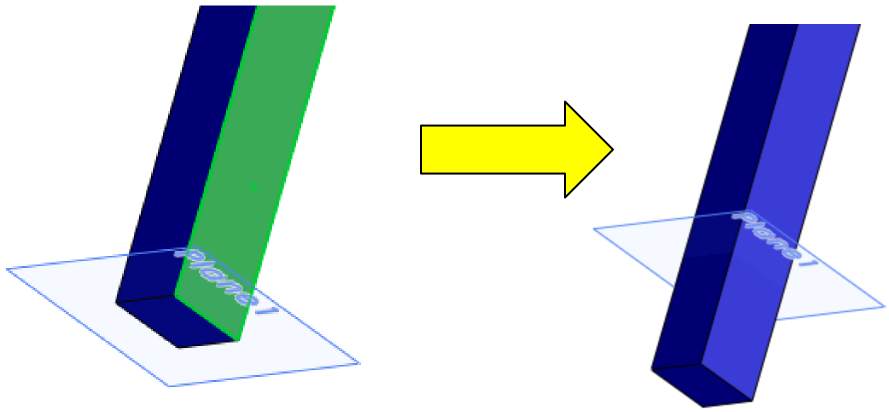
: Extend or shorten the length of lifter body.

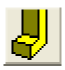
Different result depends on the selection point.

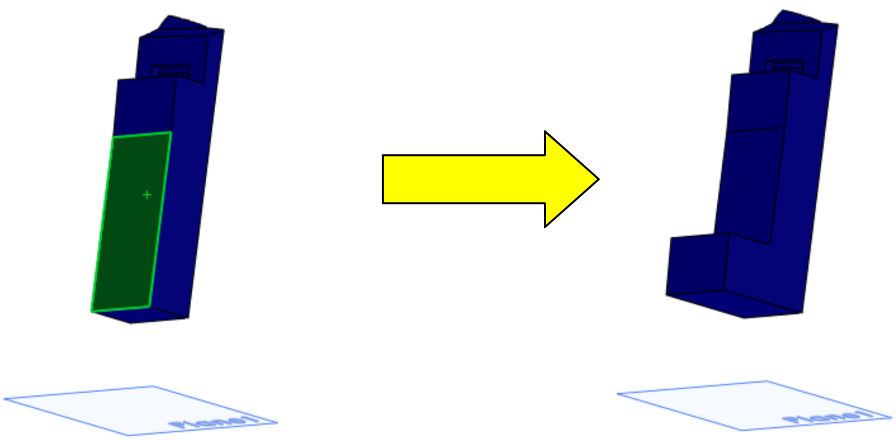
1. To shorten the length, Select the inclined face that is at acute angle with the lifter bottom face, click the icon. As shown below,

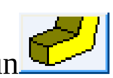


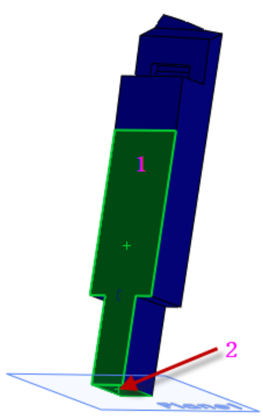
2. To extend the length, select the inclined face that is at obtuse angle with the lifter bottom base, click the icon. As shown below,



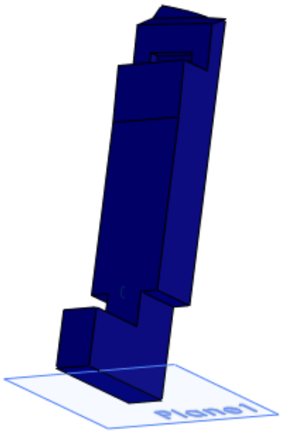
: Used to create a feature on a special lifter body as the picture shows.
To do this, select the inclined face that is at acute angle with the lifter bottom face, click the icon. As shown below,



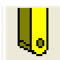
Note: For the following case, please select two faces to run 



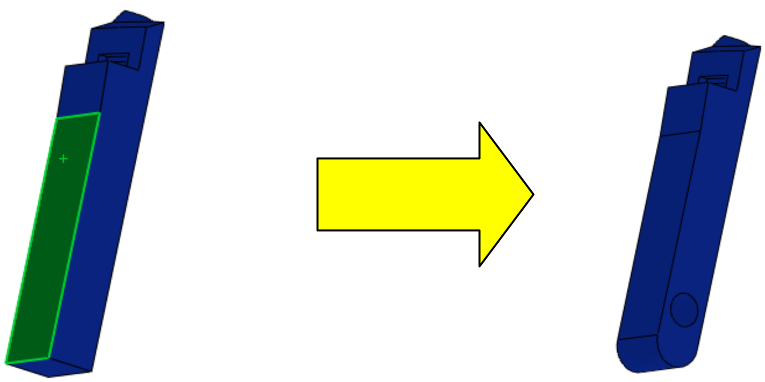
The result may vary depends on what you select

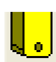


The following five icons are applicable on part file only, they are used to modify the lifter bottom area to suit different lifter assembly structure.

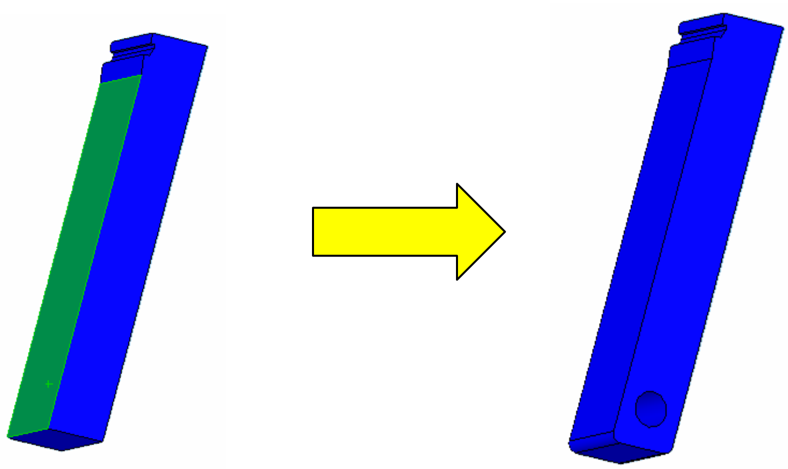
: Type 1

Method: select the inclined face, click the icon.



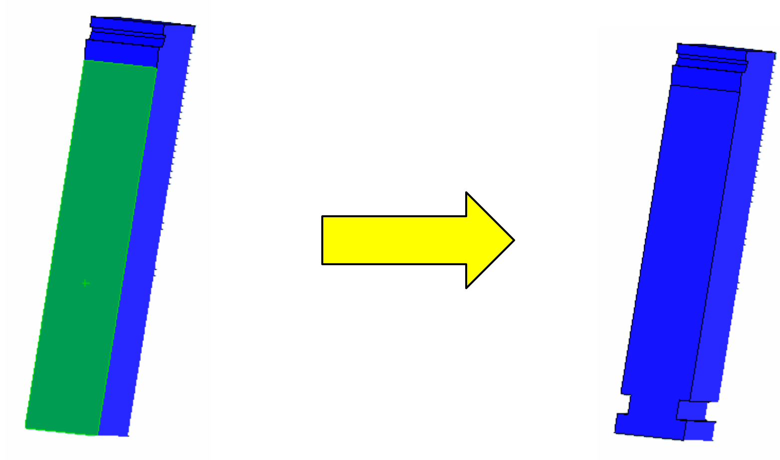
: Type 2

Method: select the inclined face, click the icon. As shown below.



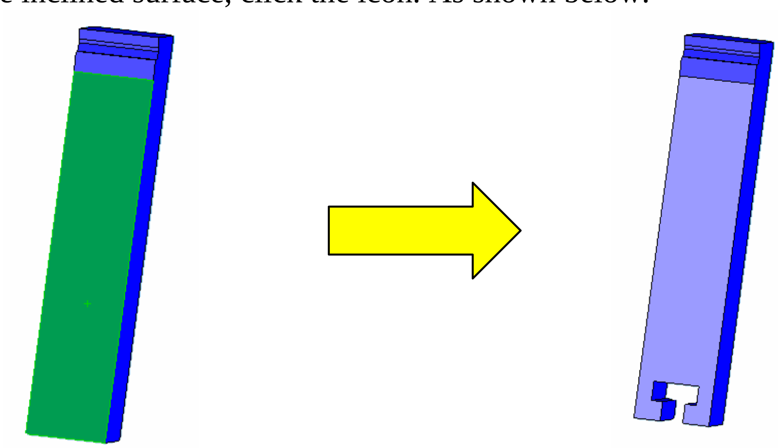
: Type 3

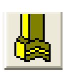
Method: select the inclined surface, click the icon. As shown below.



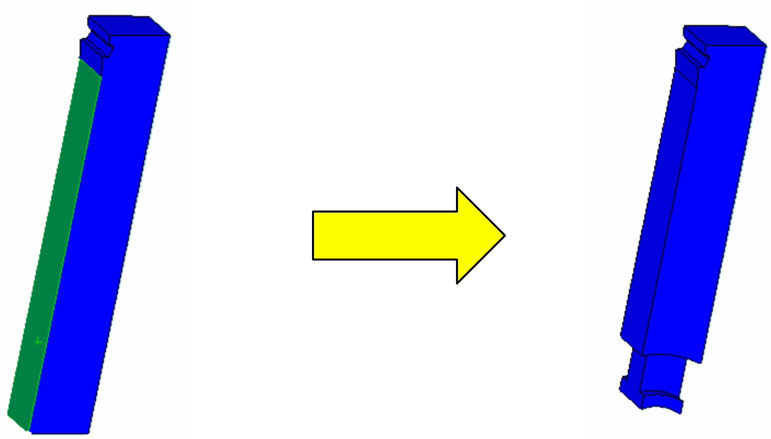
: Type 4

Method: select the inclined surface, click the icon. As shown below.

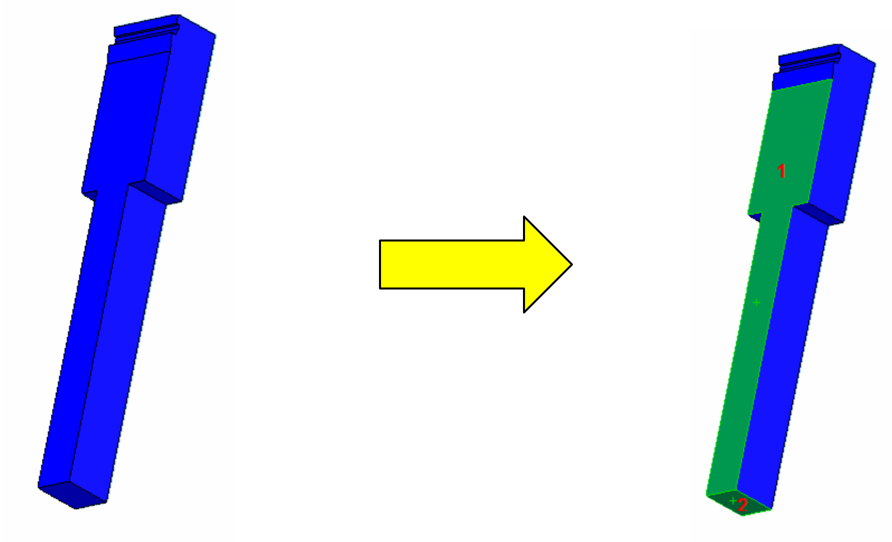


: Type 5

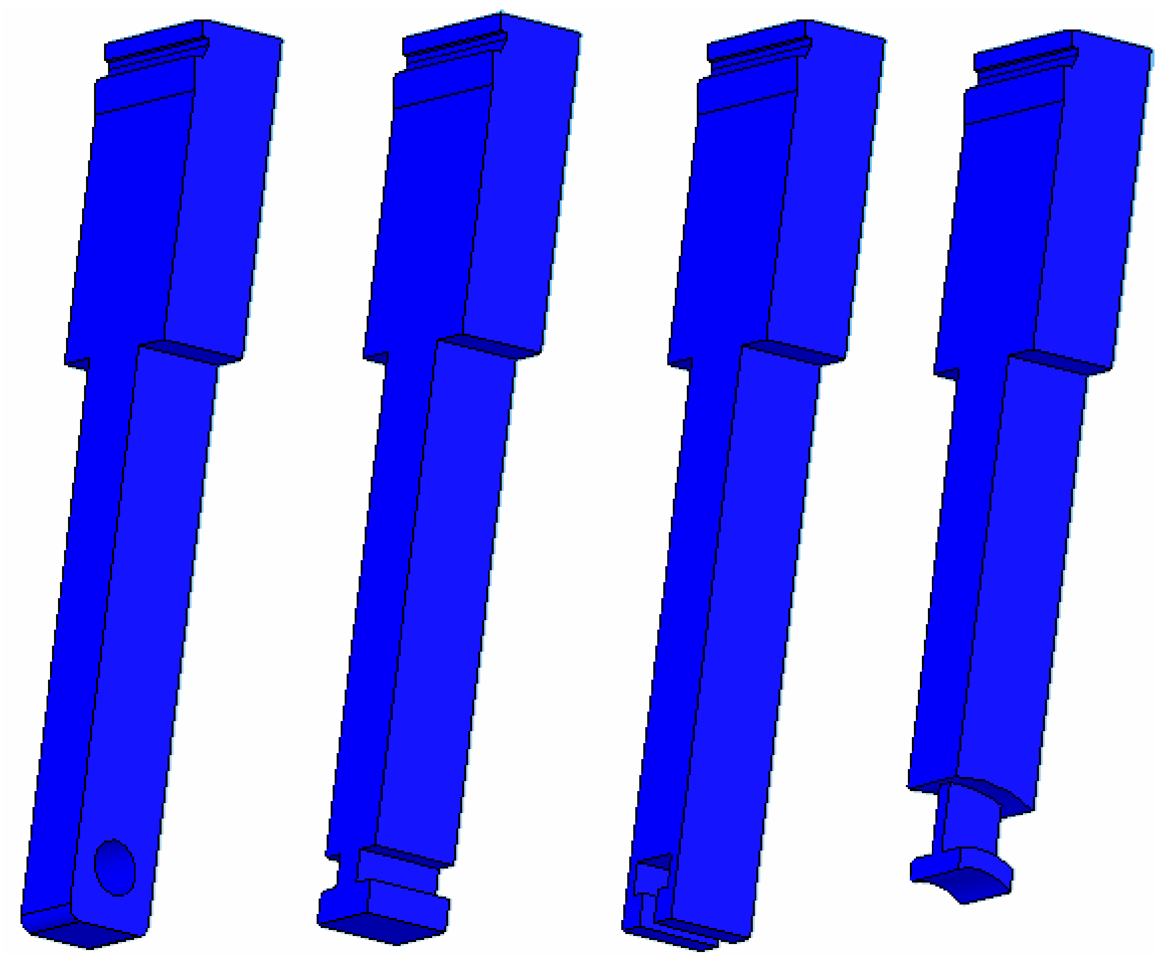
Method: select the inclined surface, click the icon. As shown below.



Note: for the following type, select an inclined face and a bottom face in order, click the icon to finish.

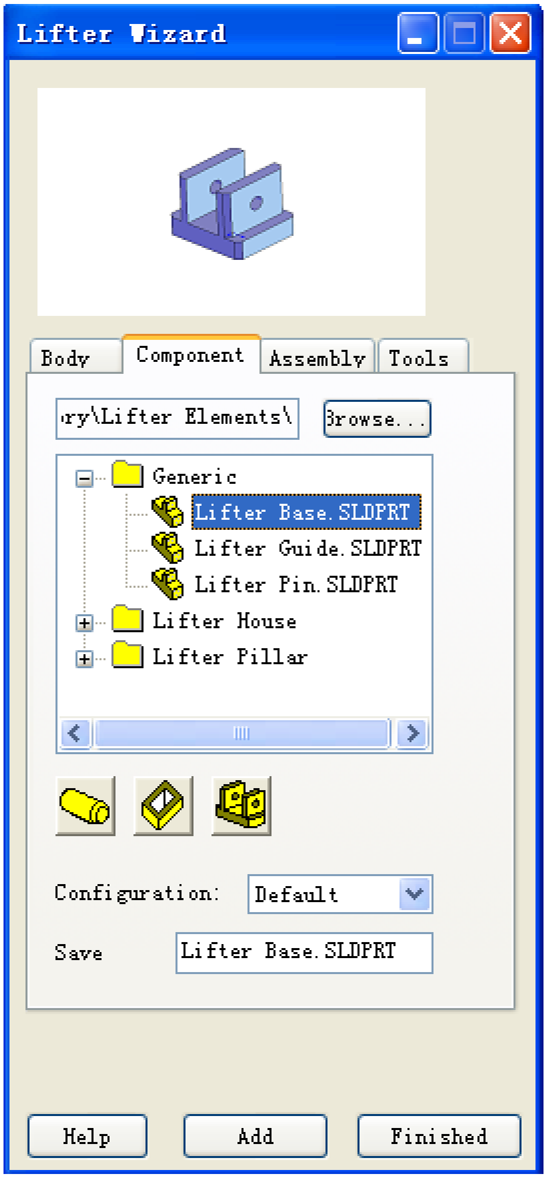


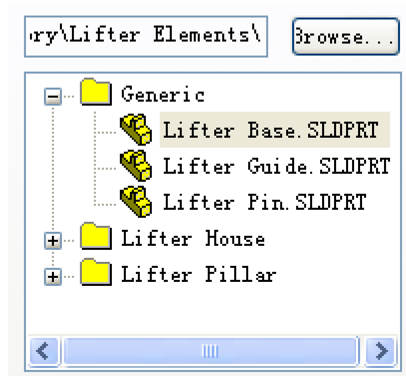
After selecting two faces, click each icon respectively and the result are shown below.



13.3 Component

On this page, user can add the standard component or non-standard lifter component on the assembly model.





By default, the tree shows the lifter standard components available in the system, click **Browse**, can select your own library folder.

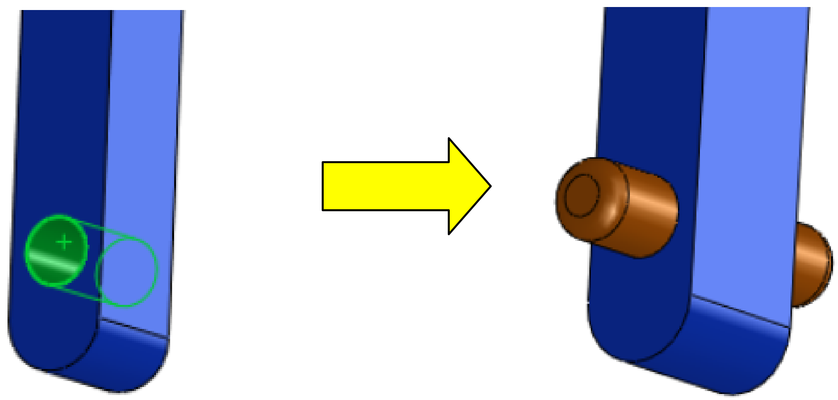
To add component to the assembly, you can select the corresponding entities on the assembly for mating reference, click **Add** to insert it to the working assembly model using the specified configuration and save it as the name in the text field.

When the component is selected in the tree, its bitmap is previewed.

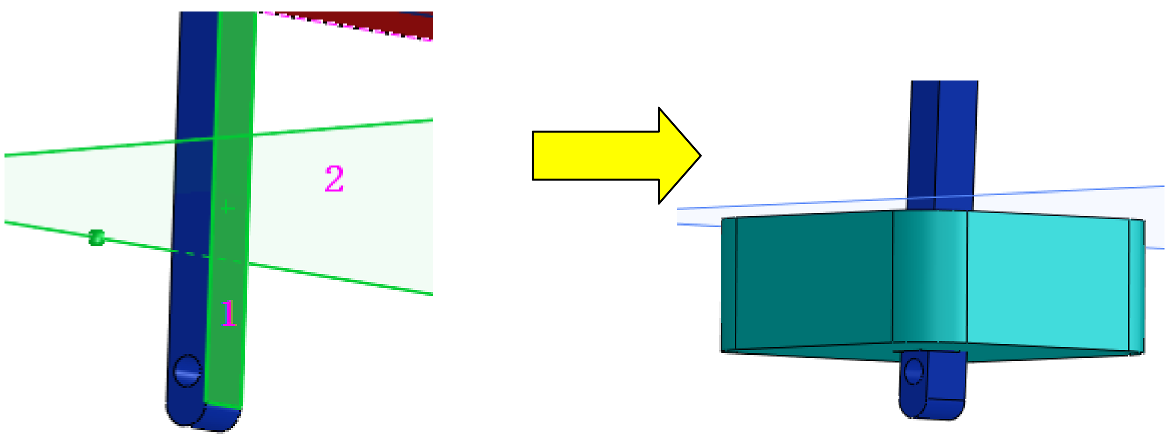
Configuration: All configurations available
Save: Name to save

The following three icons are used on assembly file, using those functions will add non-standard component to the assembly.

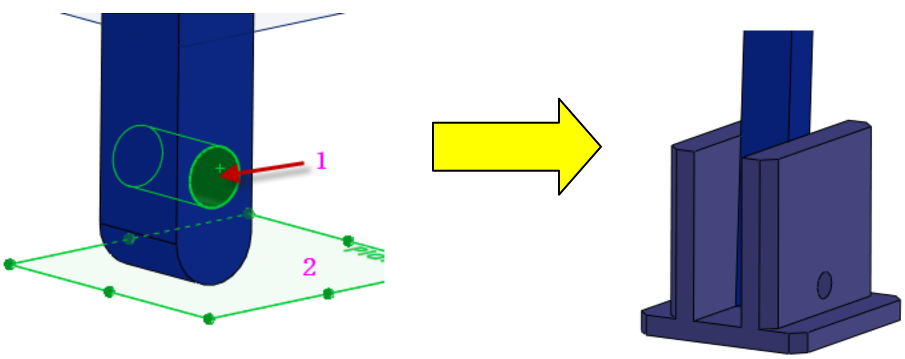
: Pin. Select the cyclindric face (The hole face).



: Guide plate. Select a inclined face and a reference plane

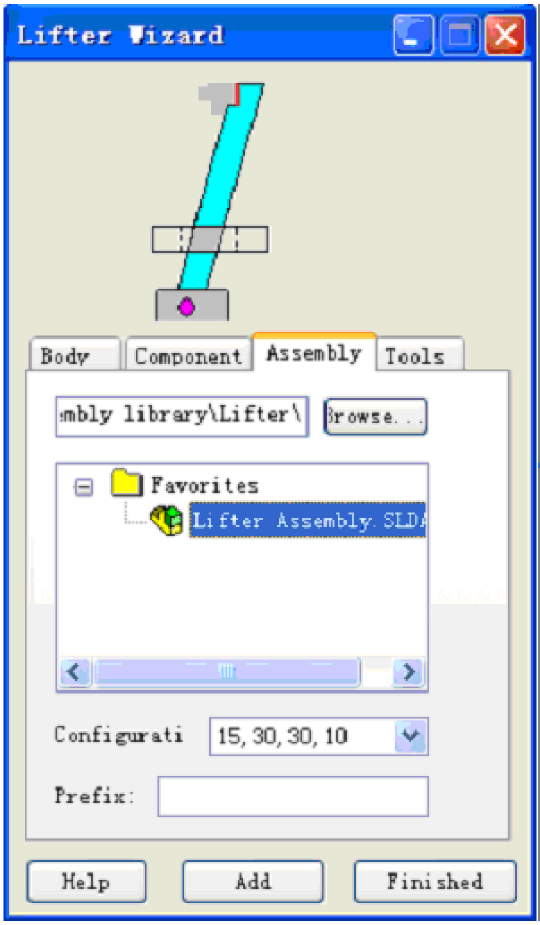


: Lifter house. Select a cylindrical face and a reference plane



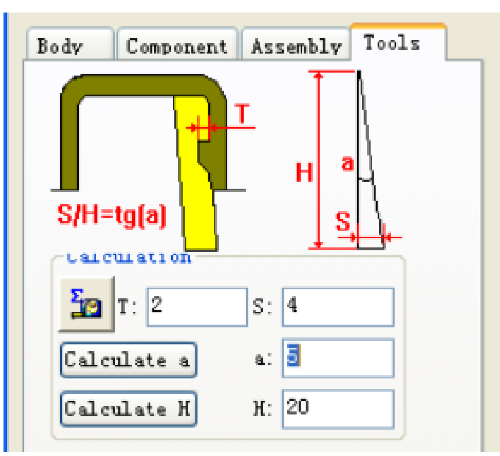
13.4 Assembly

Some standard lifter structures are pre-defined and ready to be customized.



13.5 Tools

Functions here are mainly used to calculate the left design parameters.

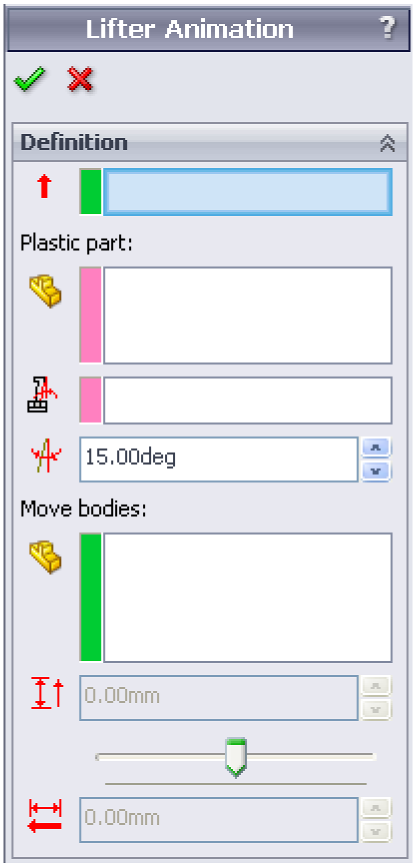


Available for part model only.

T: Can be obtained by measure

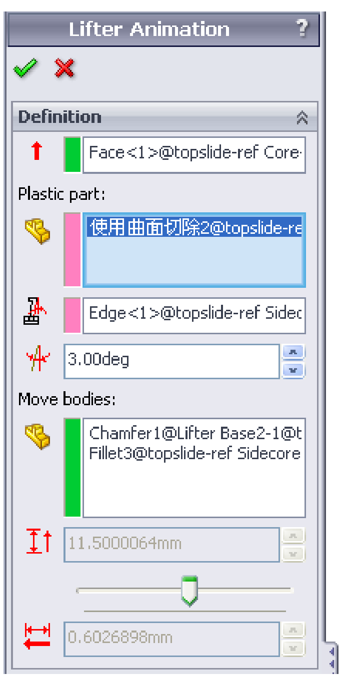
a: Input or by Calculate

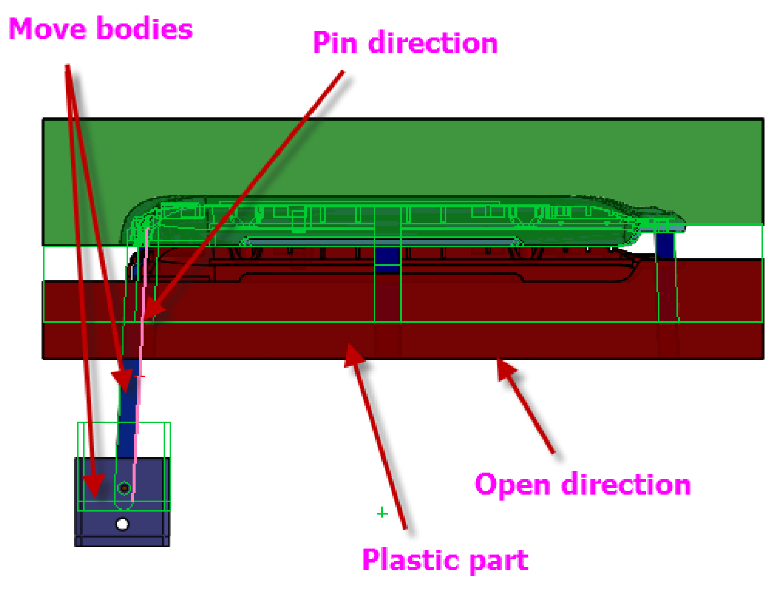
Simulate: Lifter animation function as the following picture shown.



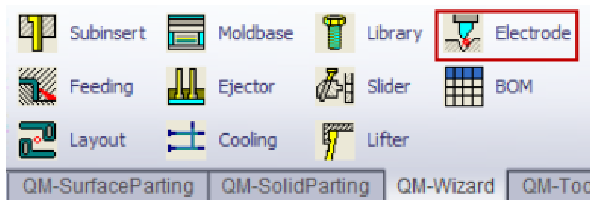
Open direction: Select a planar face to specify the mold direction.
Plastic part: Plastic part
Pin direction: Lifting direction along the lifter pin
Pin angle:
Move bodies: All bodies move together with the lifter pin
Open distance: When drag the slide bar along the slide, the open distance is updated dynamically.

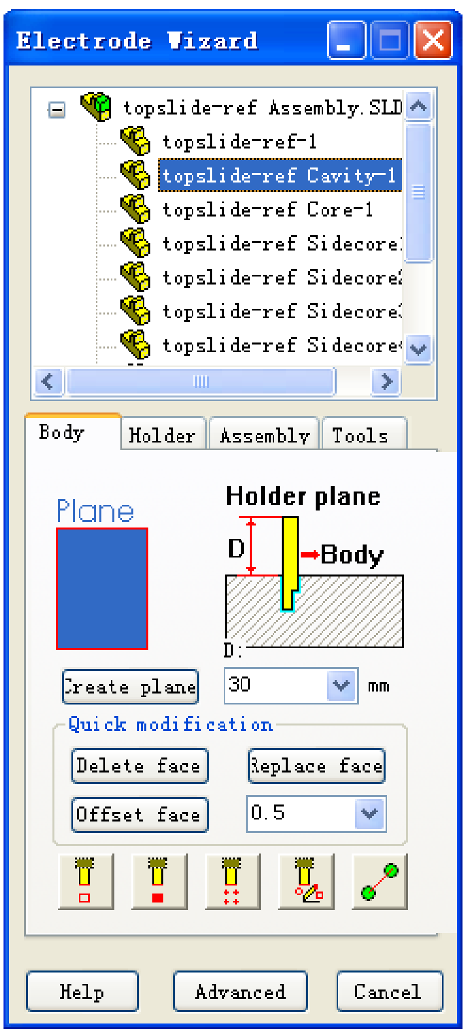
Release distance: Releasing distance value is displayed here. User can check if the undercut is completely released.





Chapter 14. Electrode Manager





Electrode Wizard is usually used after mold split. It is for the design of electrode.

The electrode design flow

Derive part from core, cavity or side core (it could be any kind of part)

Create the electrode body

Create the holder

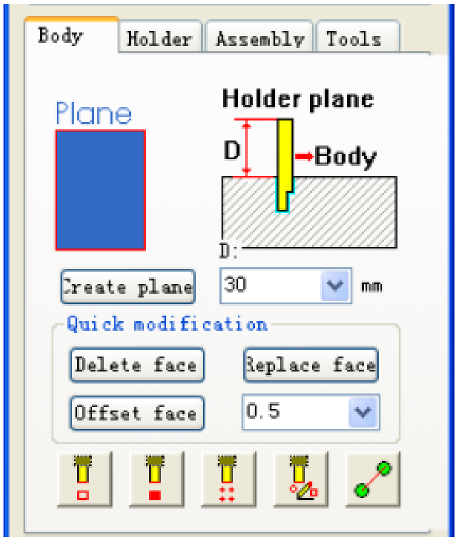
Create an electrode assembly

14.1 Component Navigator

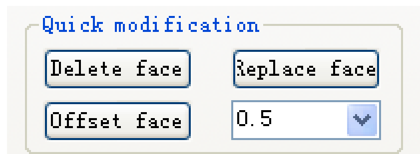
Please refer to the chapter Insert wizard.

14.2 Body

The functions on this page is used to define the electrode body

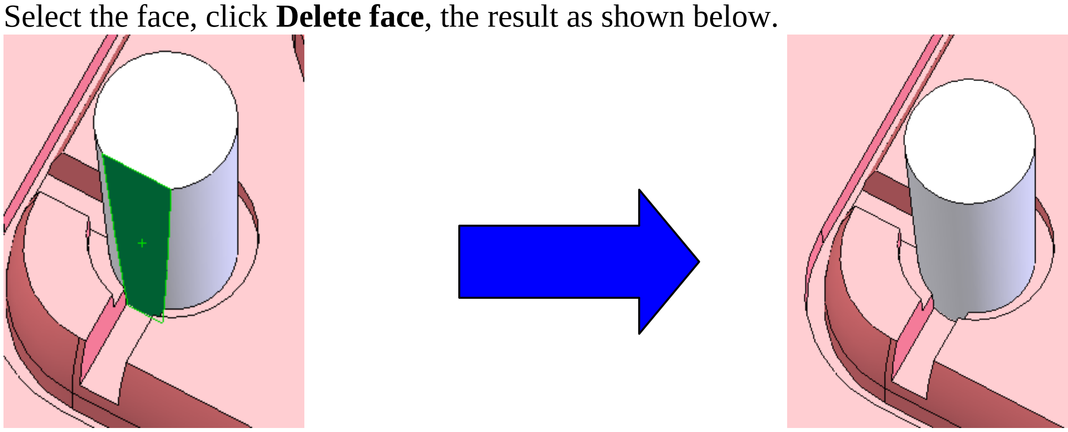


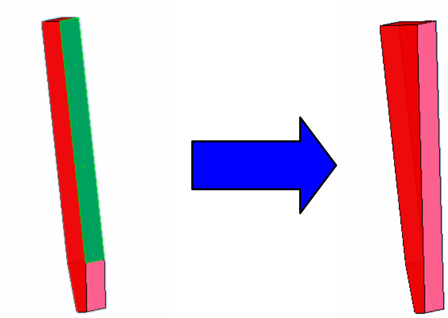
Create plane: Create reference plane as electrode position's reference of electrde holder. Select a planar face, the offset value in the edit box will be used.



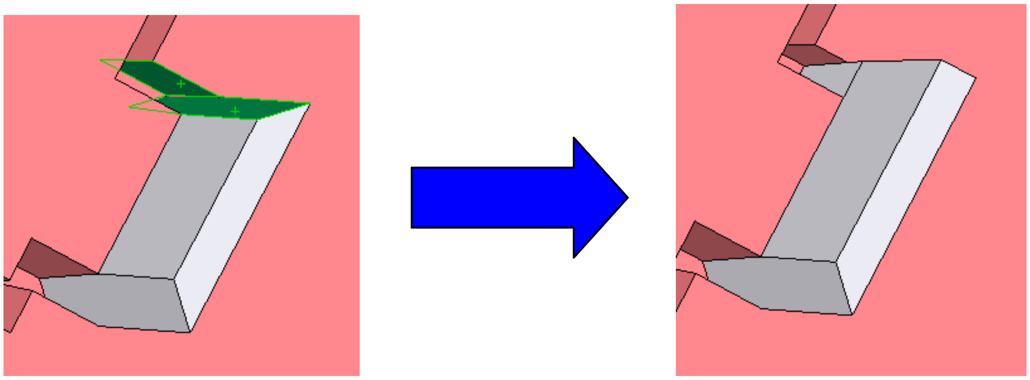
Delete face: A similar function like the Solidworks one. It is put here as this function is offen used in electrode design.

Select the face, click **Delete face**, the result as shown below.

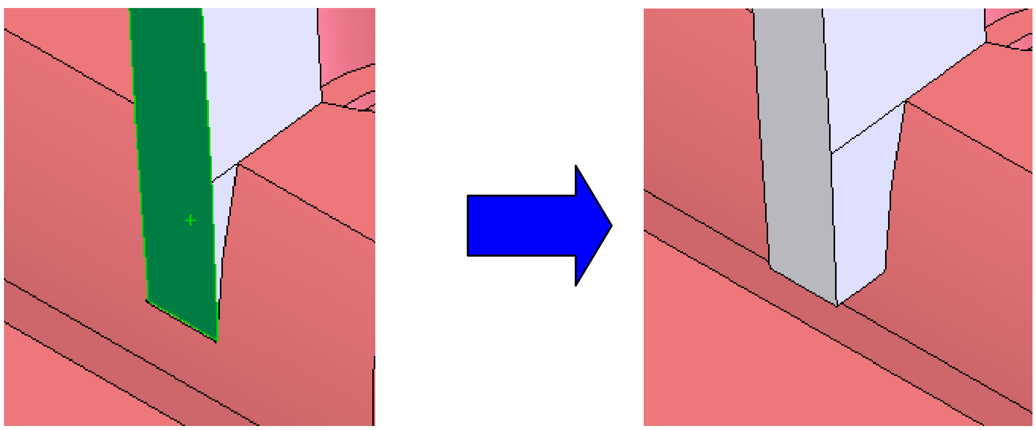


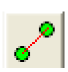


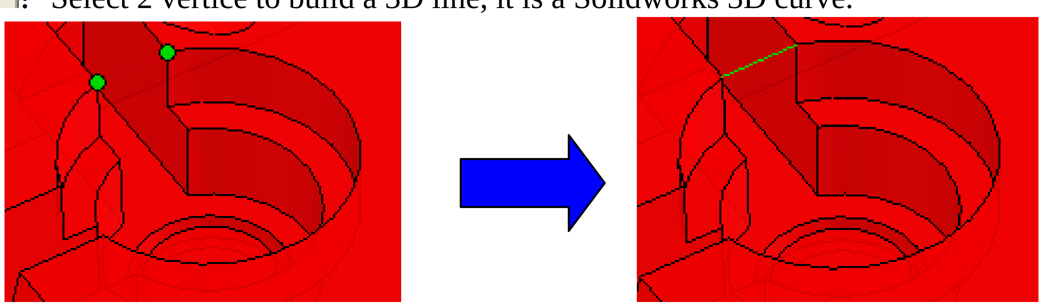
Replace face: Select the face on the electrode to be changed and a face on the core or cavity, click this button the face on the electrode is replaced by the selected face on the core or cavity.

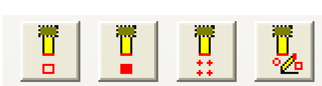


Offset face: Extend the selected face, Offset value could be set.

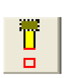


: Select 2 vertice to build a 3D line, it is a Solidworks 3D curve.

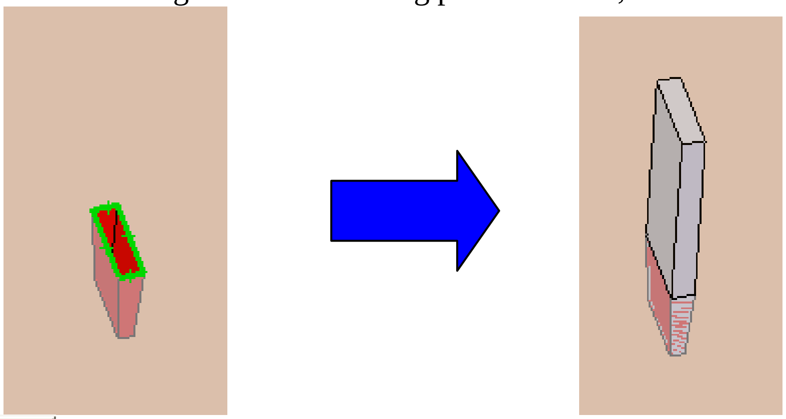


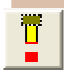


Four short-cuts used to create electrode body quickly. The above-mentioned reference plane is used as the sketch plane to create the electrode body.

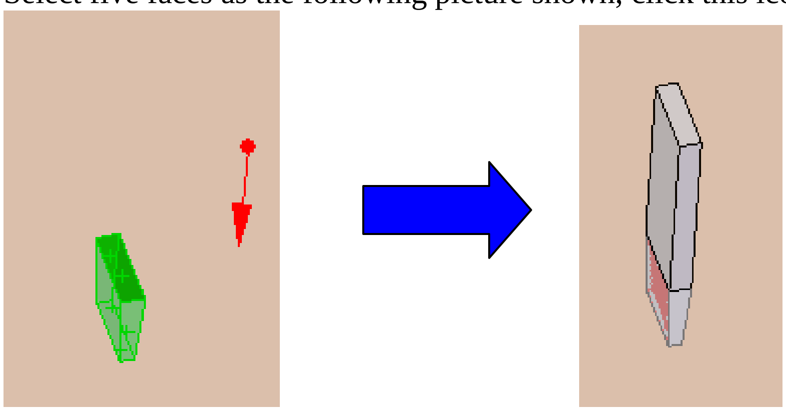
: Select closed chain edges, they are used to define the electrode body's profile.

Select four edges as the following picture shown, click this icon to create the electrode body



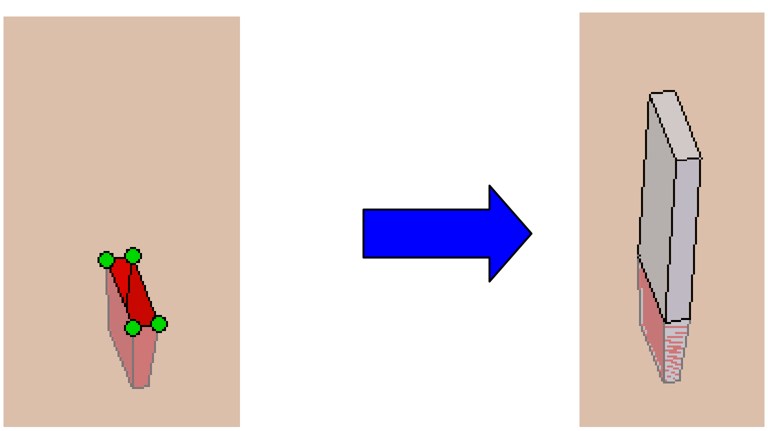
: Select some faces, using their outer edges to define the electrode body's profile

Select five faces as the following picture shown, click this icon to create the electrode body.

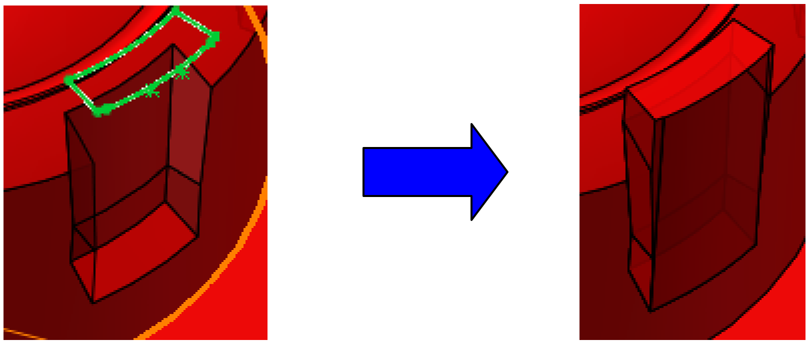


: Select some vertices on the part to define the electrode's profile

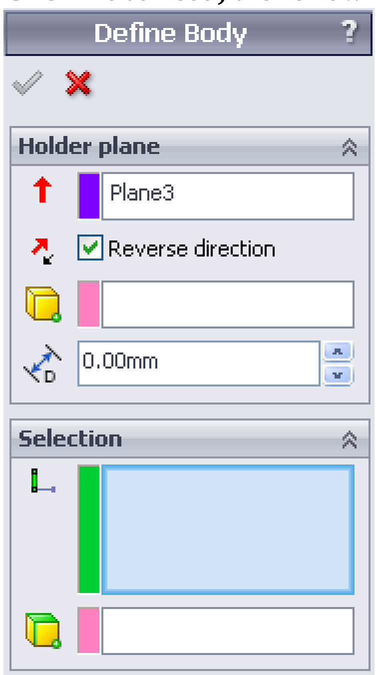
Select four vertices as the following picture shown, click this icon to create the electrode body.



: Select sketch on the part to define electrode's profile directly.



Click **Advanced,** the following page pops out.



Holder plane: Create sketch plane to define the electrode body, There are two methods to obtain the sketch plane:

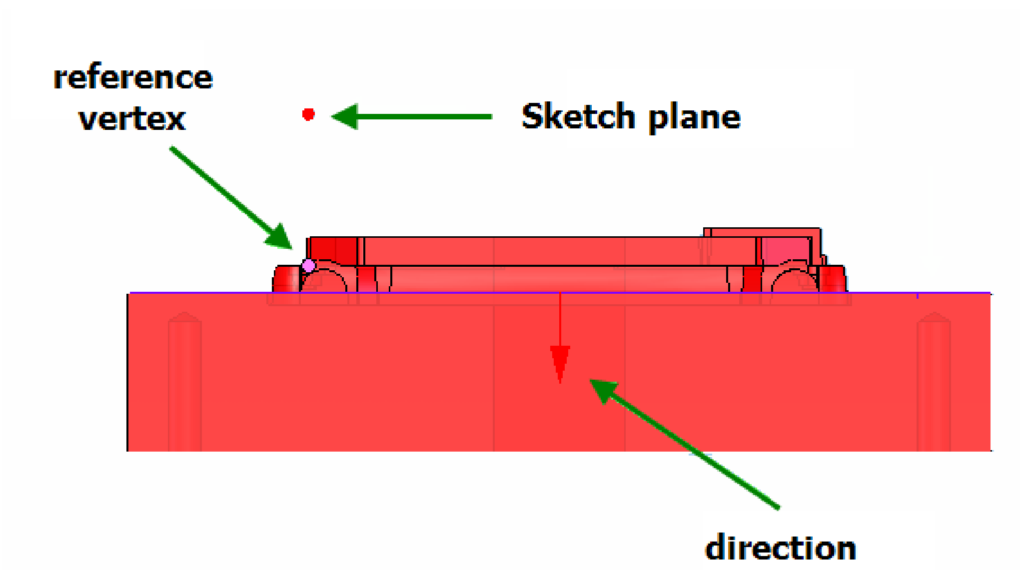
1. Pass through a vertex and parallel to a reference plane or planar face
2. Parallel to a face or reference plane at a defined distance

Direction: Select a planar face or reference plane to define the EDM direction.

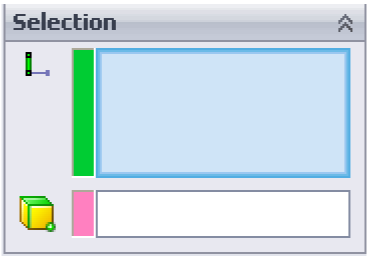
Reverse direction: Reverse the direction, if the EDM direction is not the default one.

Select reference vertex: Select reference vertex to define the offset value of the holder plane

Distance: Input a distance to define the sketch plane, there is a red dot on the screen to represent the holder position, this position must ensure that the electrode's holder won't be interfer with the part surface.



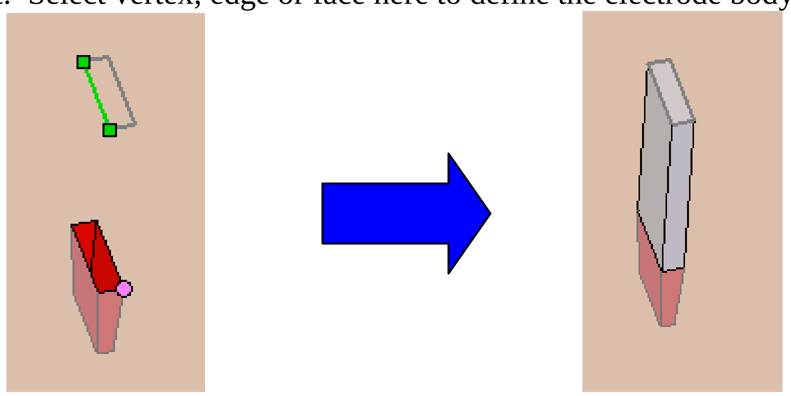
Selection:



Select sketch on the part to define electrode's profile directly.

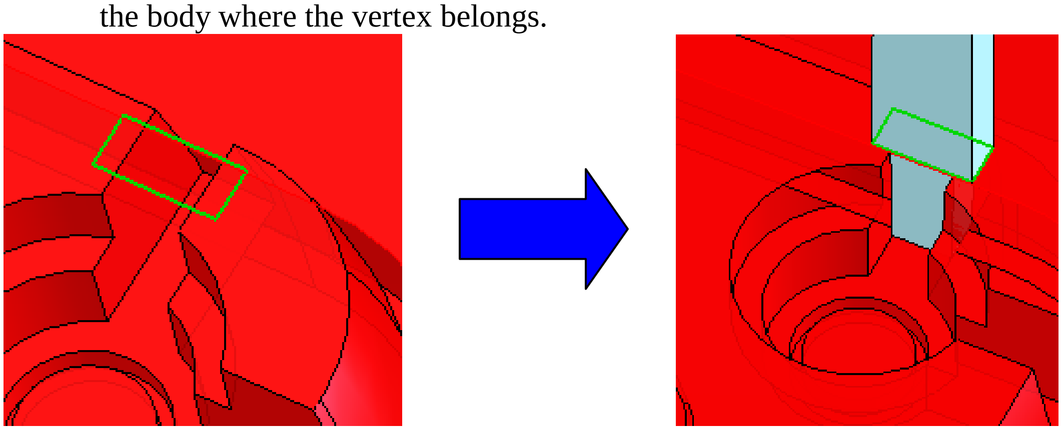
 : Select the sketch segment, they should lie on a closed loop

 Middle point: Select vertex, edge or face here to define the electrode body



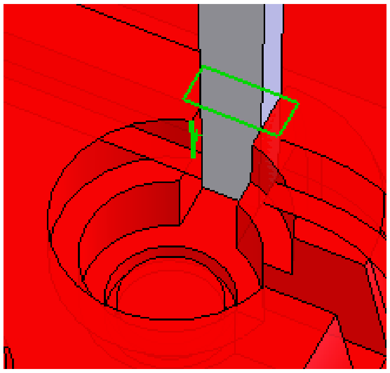
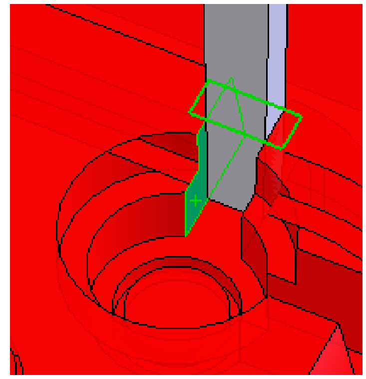
Example to explain the difference in selecting vertex, edge or face.

- Vertex: Select a vertex on the part, Extrude feature will extend from the sketch plane to the body where the vertex belongs.



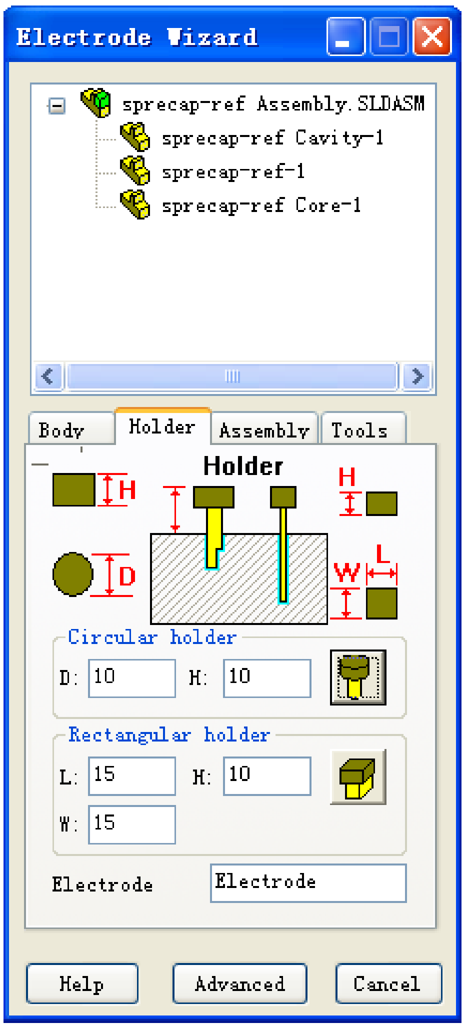
- Edge: The initial extrude body will be cut upto the middle point of the edge, then extend to the sketch plane.

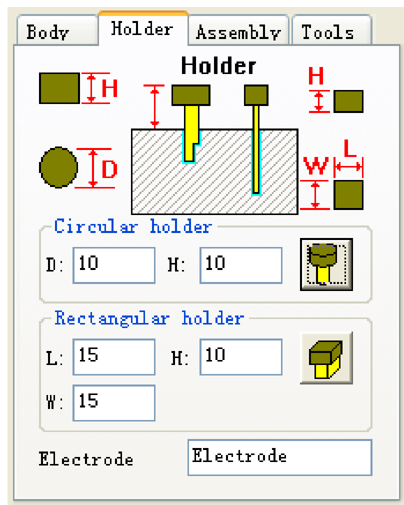
- Face: The initial extrude body will be cut upto the central point of the face, then extend to the sketch plane.

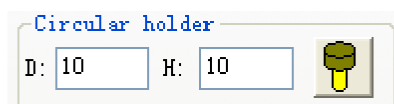
 

14.3 Holder

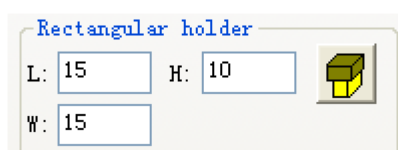
Create the holder of the electrode， the holder could be circular or rectangular type



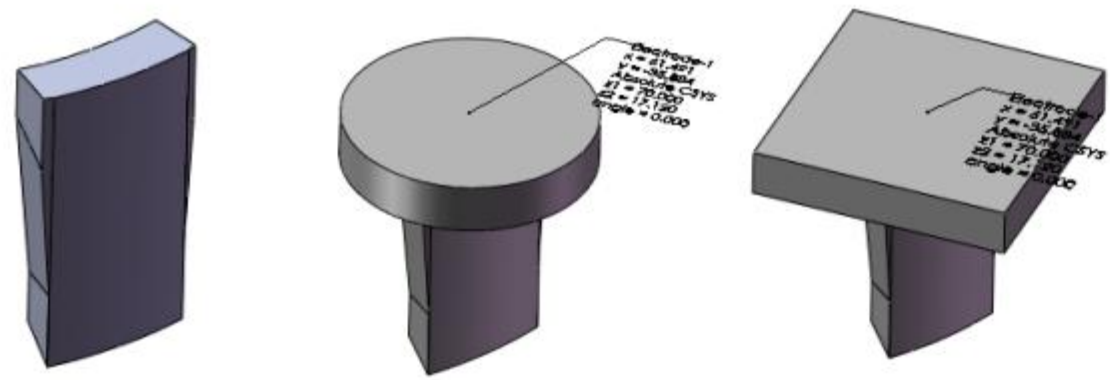


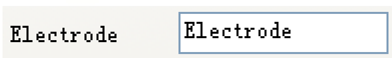


Circular holder： Select a face, the system will use the values D and H to create the holder, an annotation is attached to the holder as well.



Rectangular holder： Select a face, the system will use the values D and H to create the holder, an annotation is attached to the holder as well.





: Prefix for the electrode body

Click **Advanced**, the following picture pops out

Rectangle: Create a holder with rectanglar shape.
Select base face: Select the base face to be the reference plane to create the holder. Its minimum size is detemined and displayed automatically.

Reverse direction: Reverse the direction of the holder if necessary

Favorite size: Select the standard or customized holder size available in the system .

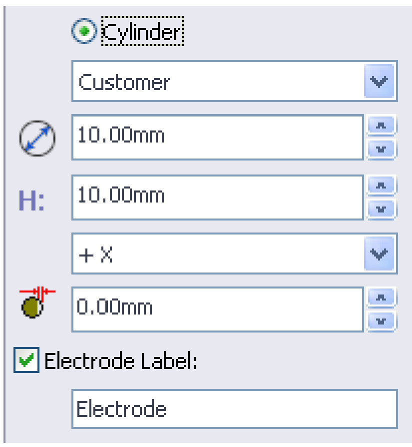
Block Electrode.txt could be found under the installation folder, this file could be customized.

	A	B	C	D
1		H	V	L
2	Customer	10	10	10
3	EHB-5-5	5	5	10
4	EHB-5-10	10	5	10
5	EHB-5-15	15	5	10
6	EHB-5-20	20	5	10
7	EHB-5-25	25	5	10
8	EHB-5-30	30	5	10

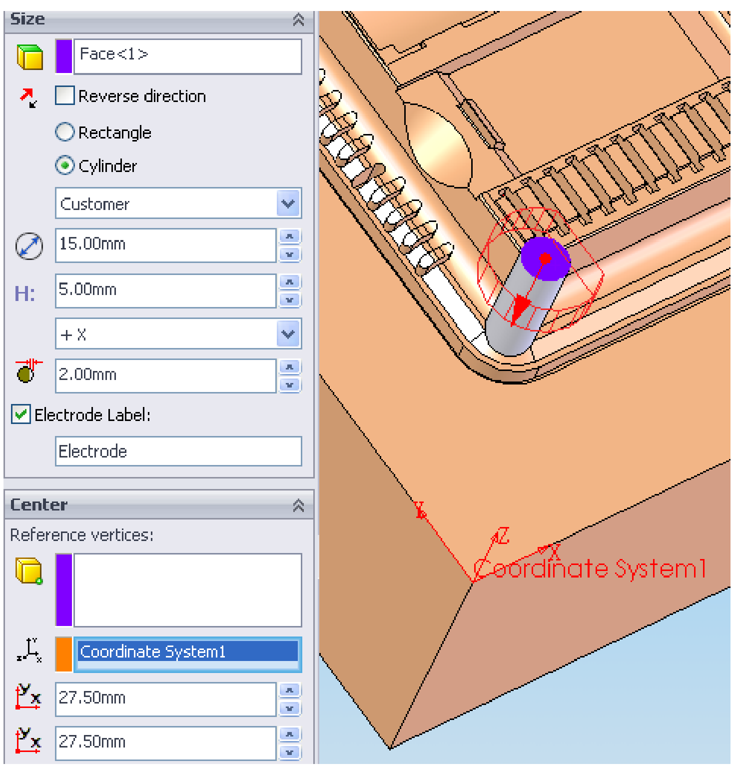
L: Define the length of holder
W: Define the width of holder
H: Define the height of holder
: Define the angle of rotation of the holder
Electrode Label: Electrode label name
Electrode Name: Name for electrode
Reference vertices: The default holder center is the center

of the selected face. However, it can be defined by specifying a reference vertex and its relative position.

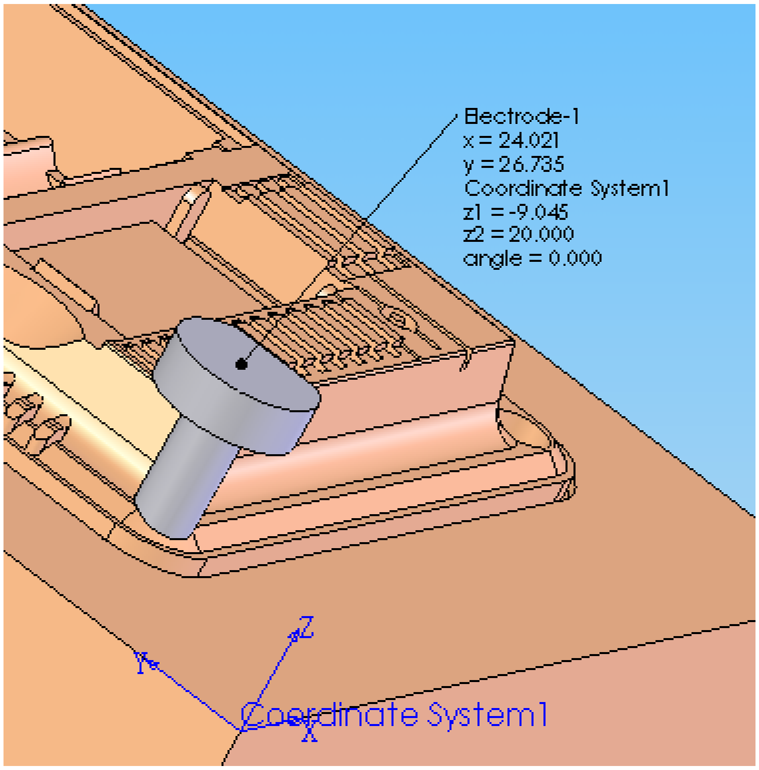
: Select a coordinate system for the position of the electrode.
The default center is the face center of the selected planar face
: the X coordinate of the centre of the electrode holder relative to the above selected coordinate system
: the Y coordinate of the centre of the electrode holder relative to the above selected coordinate system
Cylinder: Create a cylindrical holder



: Define the diameter of the holder.
H: Define the height of the holder
Mark Side: The orientation of the mark area, four options +X, -Y, -X, +Y for use to choose.
: Offset distance



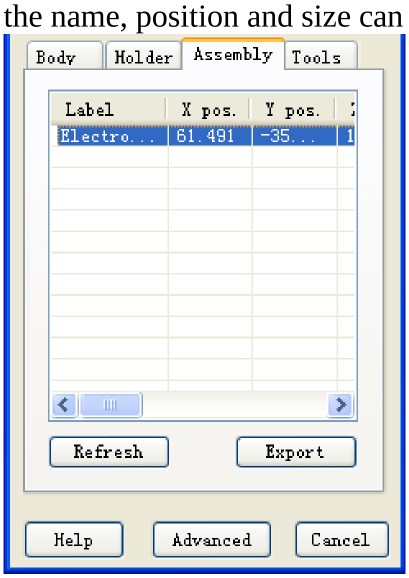
Click OK, an electrode holder is created. There is an annotation attached to the holder, major electrode's information is displayed on the screen such as its name, coordinate system and positions.

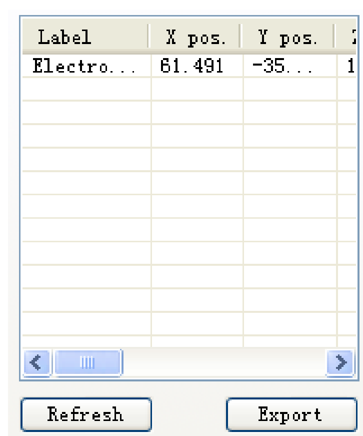


Z1 and Z2 are the electrode's lowest and highest position value in the selected coordinate system.

14.4 Assembly

Save the electrode part with multiple bodies into an assembly. The electrode information such as the name, position and size can be exported for reference.



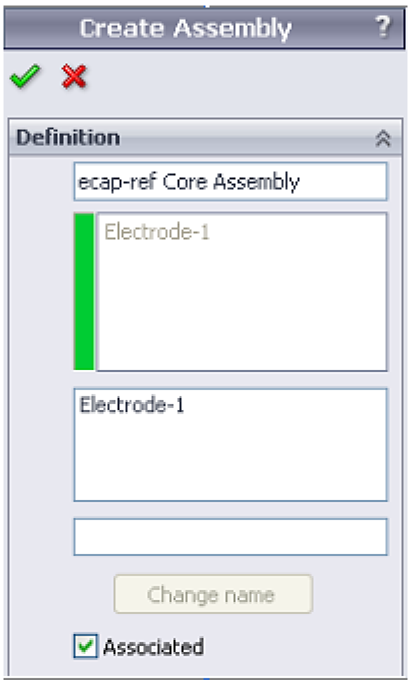


Refresh: Update the electrode information such as the name, position and so on.

Export: Export the current electrode information to an external text file under working folder.

Click **Advanced** will bring out the following dialog.

This dialog will define the electrode component name and its assembly name.



Assembly name: Name of the assembly to be created
Select electrode: Select a solid from the shown body list
Body list: List out all shown solid bodies in the part

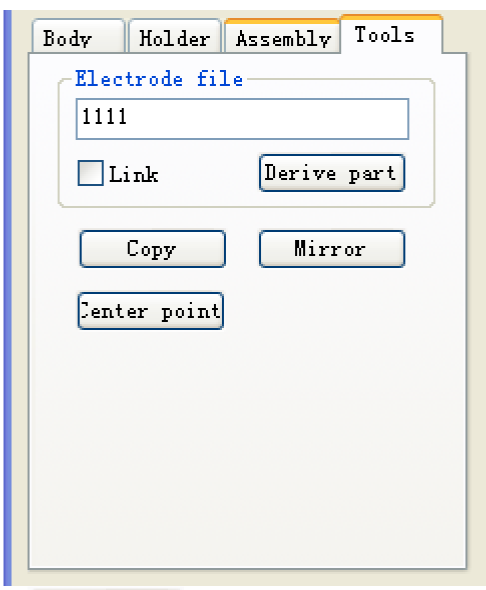
Part name: Input the component name. By default, the component's name is the solid body's name in the electrode part.

Change name: Change the selected body name to the input one below.

Associated: If this option is unchecked, the imported body will be created in the assembly component. In a certain situation, the imported bodies could achive a better performance.

14.5 Tools

Some effective tools for electrode design

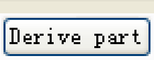


Electrode file
To avoid creating the electrodes on the core/cavity part directly, we need to create a new Solidworks part with the core or cavity inserted. This way, the associativity could be maintained and the total mold assembly won't become very complex.

Name: Name for the the new part
Link: Specify if the external reference of the new created part is locked or not.

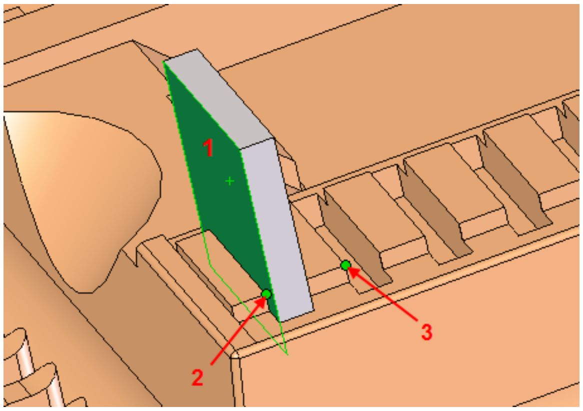
- If Checked , the feature in the tree indicate that the external reference is unlocked. 

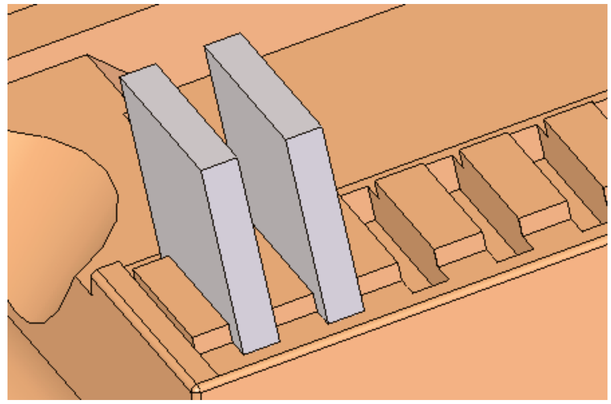
- If Unchecked, The feature in the tree indicate that the external reference is locked. 



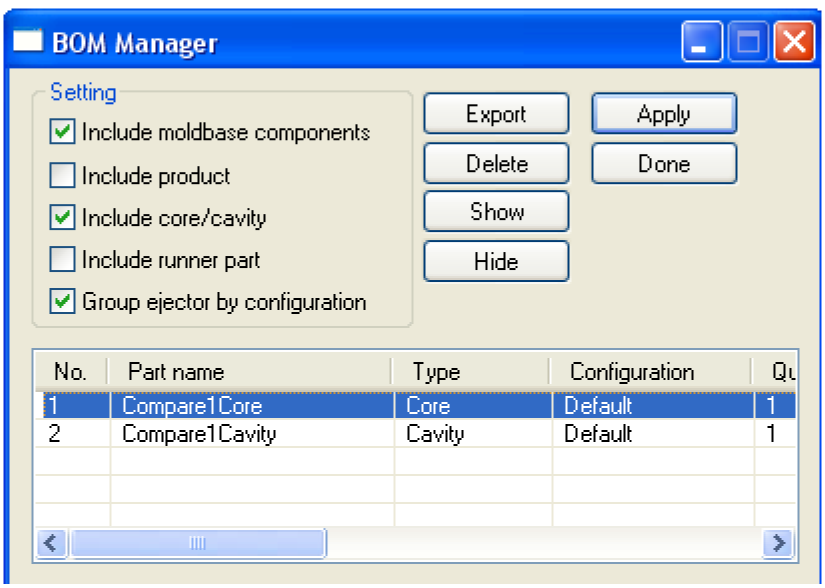
: Derive Part from the current one

: Copy body, select a face on the existing electrode body and two vertices on the electrode part in order, click this button, electrode body will be copied from the first vertex to the second one



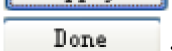


Chapter 15. BOM Manager



This function works on assembly models only.

:Update the BOM information on the list window

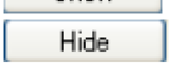


: Exit the BOM dialog

:Create a text file for the BOM information under the current working folder

: Delete the selected items on the list windows.

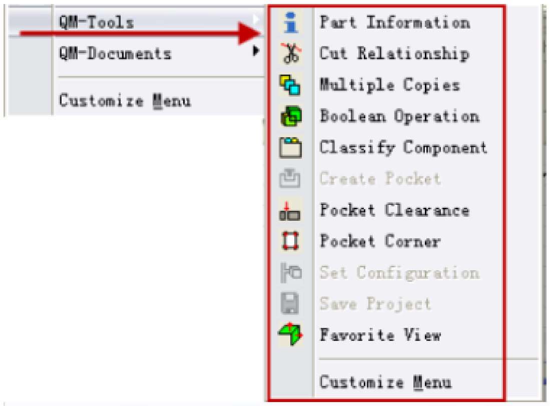
: Show the selected component items on the list window

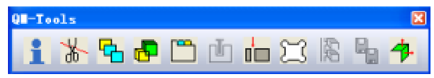


: Hide the selected component items on the list window.

As needed, sometimes, some components will be included or excluded from the BOM, please the settings accordingly.

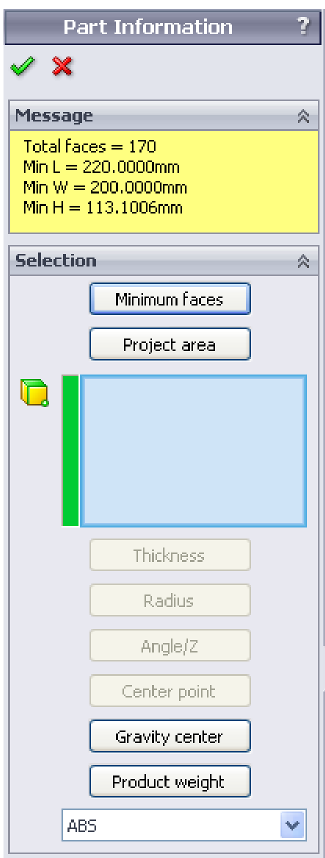
Chapter 16. QM- Tools





16.1 Part Information

Useful information on the selected solid body can be checked such as part dimension, minimum faces, project area and so on. They are very important information for mold design but difficult to obtain by Solidworks tools.



Message: Report on the selected body' geometric information such as number of faces, minimum enclose dimension, radius of curvature, center of gravity, etc.

Selection: Entities on the part, it could be vertex, edge and face type. Depending on the different selection combination, buttons below this selection box will become active or inactive accordingly.

Mininum faces: Search for the 10 smallest faces on the part, this function can help user to identify the problematic area on the part. Most of time, invalid faces on the part happens on the small faces, in manufacturing sense, some small faces should be avoided if possible to simplify the tool path for machining.

Project area: Project area on the Z direction, this value could be used to calculate the molding force and select the right molding machine accordingly.

Thickness: When one face is selected in the selection box, this button becomes active, click on it, the thickness value on the picking up position will be shown on the message box.

Radius: When there is one face selected in the ***Selection*** box, click this button, radius value on the picking up position is displayed on the message box.

Angle/Z: Angle value between the normal direction on the selection position and Axis Z.
Center point: It is used to create a center point between too selections, this point is shown in 3D sketch point.

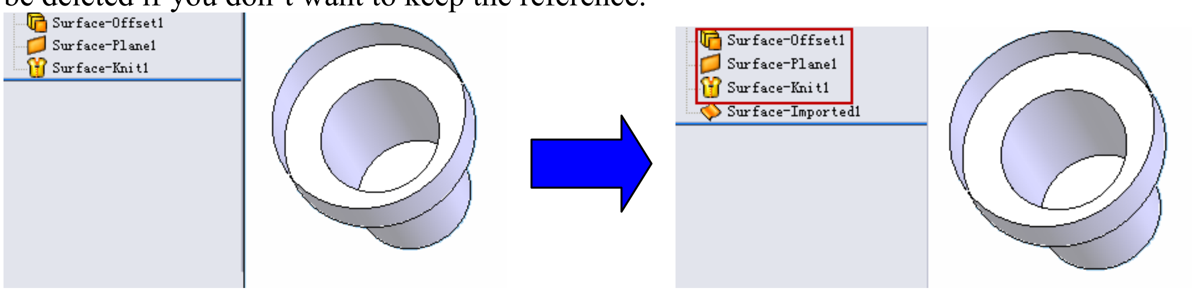
- When two vertices are selected, center point is the middle point between two vertices.
- When two edges are selected, the middle point of edge is taken as a vertex, those two vertices will form a center point as the above situation.
- When one edge and one vertex are selected, the middle point of edge is calculated as a vertex, together with another selected one, they will form a center point.

Gravity center: Display and create the gravity center.
Product weight: Based on the body volume and selected material, the part weight will be calculated and display in the message box.

16.2 Cut Relationship

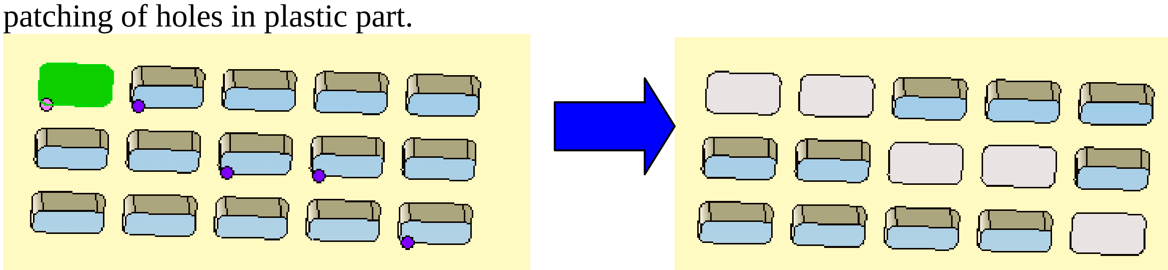
The face or surface could be selected on graphic area directly, or on the feature tree or surface body folder.

After selection, click **Cut Relationship**, a Surface-Imported is created, the circled features can be deleted if you don't want to keep the reference.



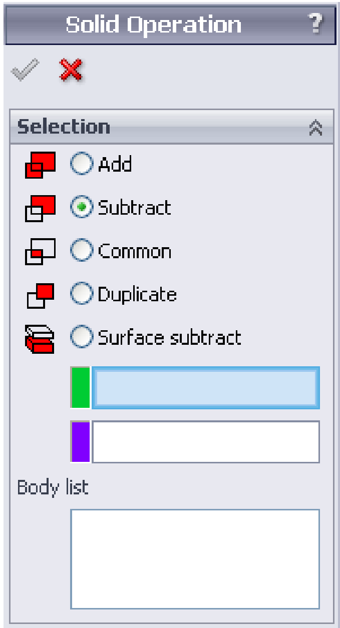
16.3 Multiple copy

Copy a surface to multiple destinations defined by vertices, commonly used in the surface patching of holes in plastic part.



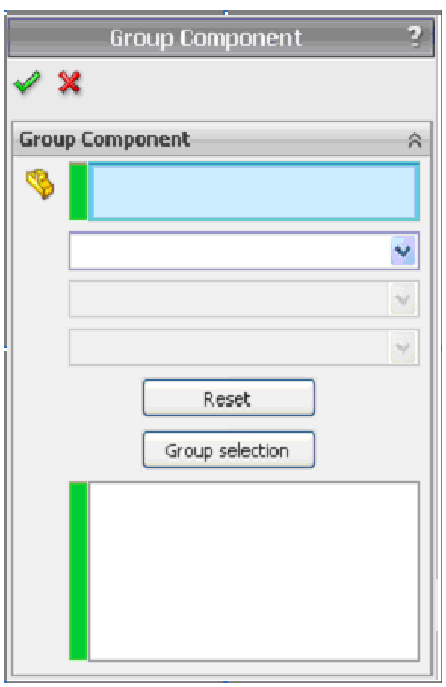
16.4 Boolean Operation

To improve the performance issue, sometime, full associativity is not necessary to be maintained throughout the mold design, particularly, if the original part is not a Solidworks native one, in this case, we can use imported body to handle solid operation such as Boolean operation.



16.5 Classify Component

For external component， define the group of the component， for the recognition and internal management of 3DQuickMold, this group setting will influence the activate working model function immediately.



Component type： Select the group that the component belongs to, in the pull down menu there listed all the group name of the mold, select here for the desired group of the component.

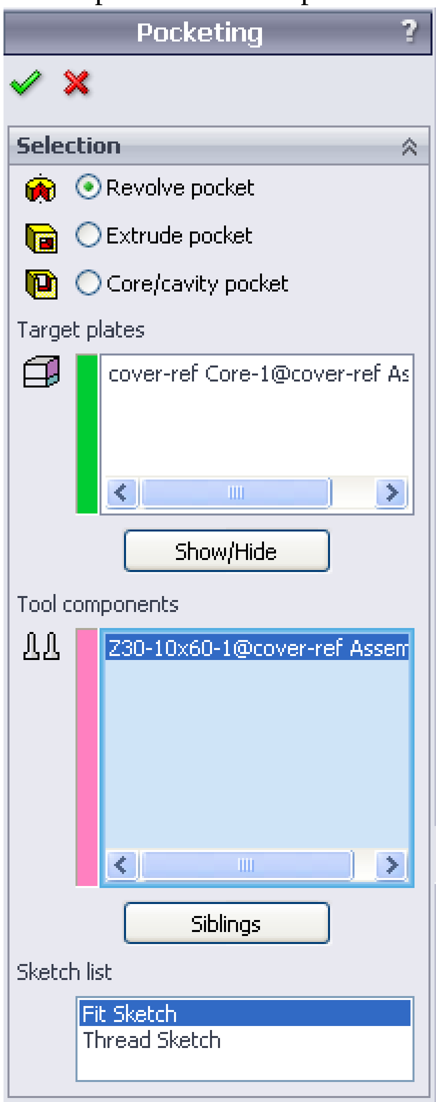
Component subtype： Some ccomponents can be further classified, in this case, the subtype will be available
Reset： Reset the component group.

Group selection： if there is more than one component belong to the same group, click this button and all the component with the same group will be display in the lower dialogue box and ready to be specified a new group.

If the mold is completely designed within 3DQuickMold, the system will automatically group the relevant component internally. So, the system can recognize which is the core/cavity and moldbase assembly no matter how you name them.

16.6 Create Pocket

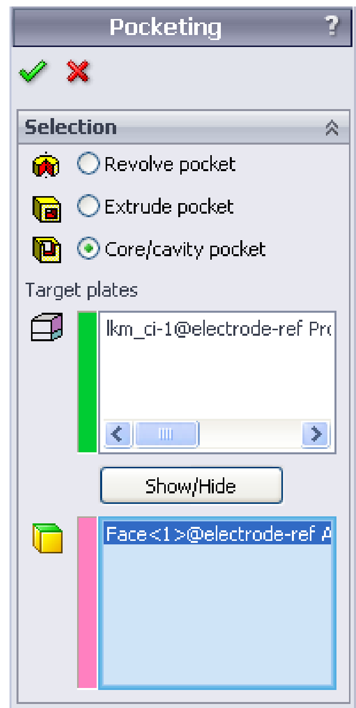
Create pockets for components on the mold plates.



Revolve pocket : For the round pocket, please refer to the description in the mold base section for details.

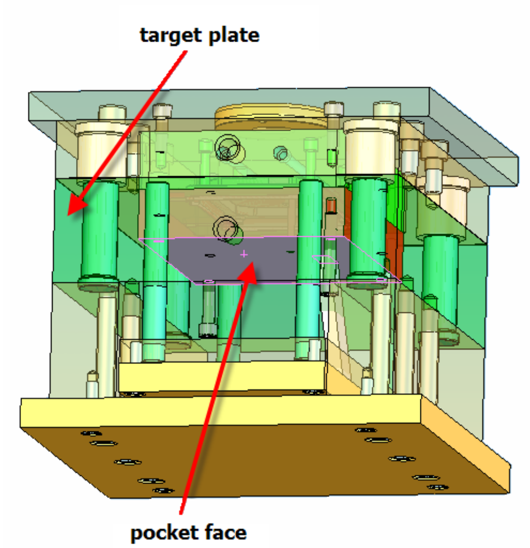
Extrude pocket : Please refer to the description in the mold base section.

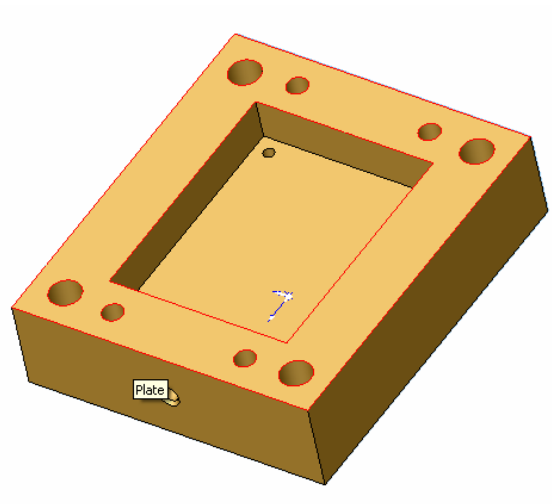
Core/cavity pocket: Particular for the pocket on the cavity plate and core plane for the main inserts that are known as core and cavity.



Target plates: Plate to be pocketed.
Show/Hide: Show or Hide the targeted plate.
Face: Select a face on the cavity top (if it is the A plate) or a face at the core bottom (if it is for the B plate)

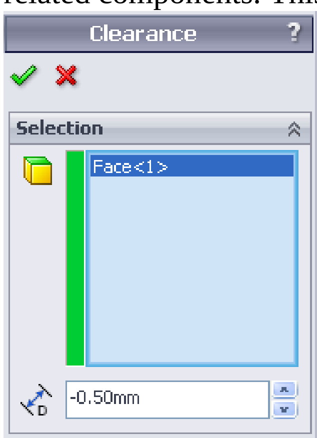
Open target plate after pocketing

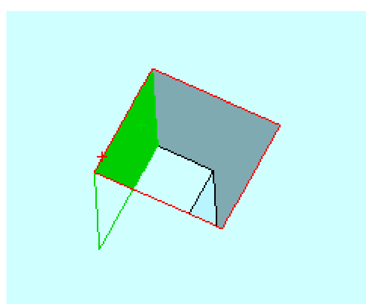
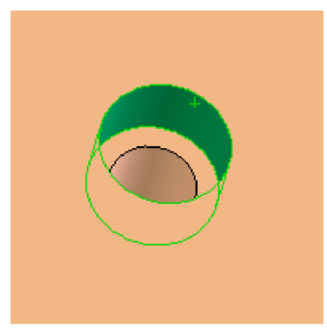




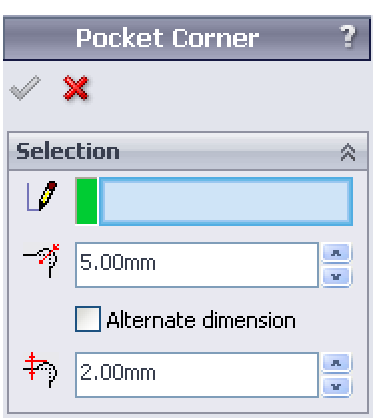
16.7 Pocket Clearance

If the pocket is done by Ins**ert Cavity**, by default, there is no clearance existing between the related components. This tool can help user to apply clearance.



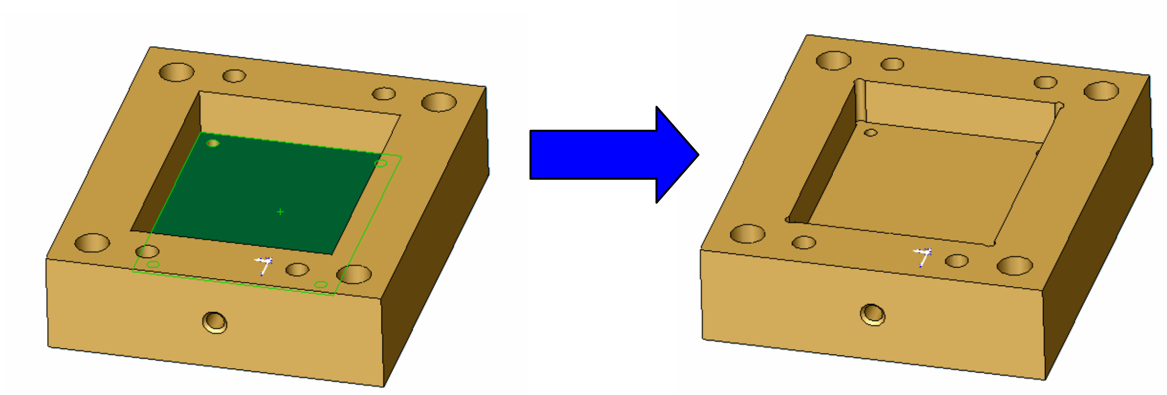
16.8 Pocket corner



To create the corner clearance for the pocket on the mold plates

Selection: Select the area to pocket corner
Corner radius: Set the Corner radius
Offset distance: Offset value from the arc center
Alternate dimension: Another way to represent the center offset.

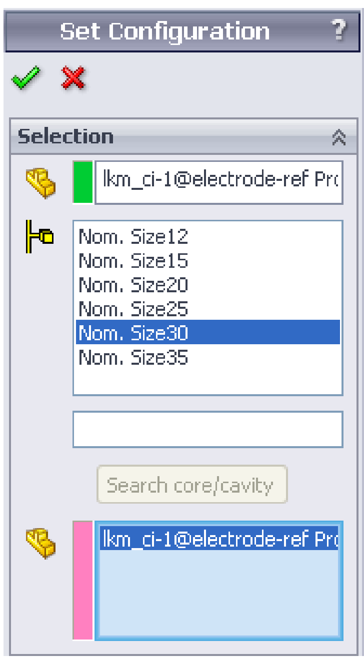
Select a face to specify the area to pocket corner, Click Ok.



16.9 Set configuration

In a multiple cavities layout, if one instance is changed, other instances may follow the change as they all have the same configuration by default. However, sometimes, we want the multiple cavities have some slight difference such as the cooling channels's inlet and outlet position may be diffirent. In this situation, we need to use different configuration to handle it.

Set configuration is used to add a new configuration to existing assembly model and its children components.



Selection

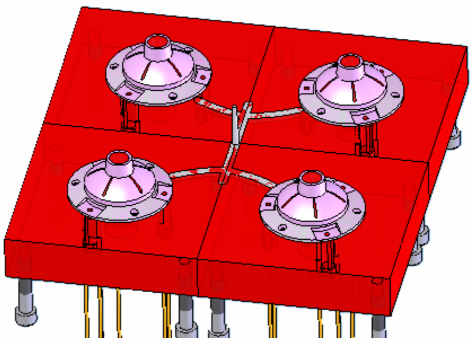
： Select an assembly to add a new configuration

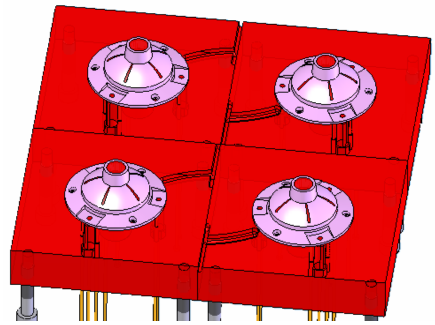
： List all existing configurations

Configuration name： Input the new configuration name to add

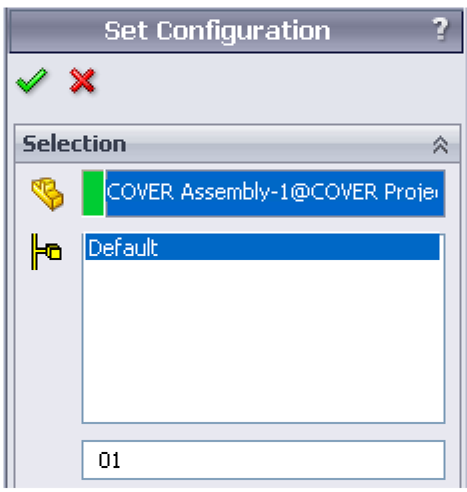
Search core/cavity： Search the core/cavity under the selected assembly.

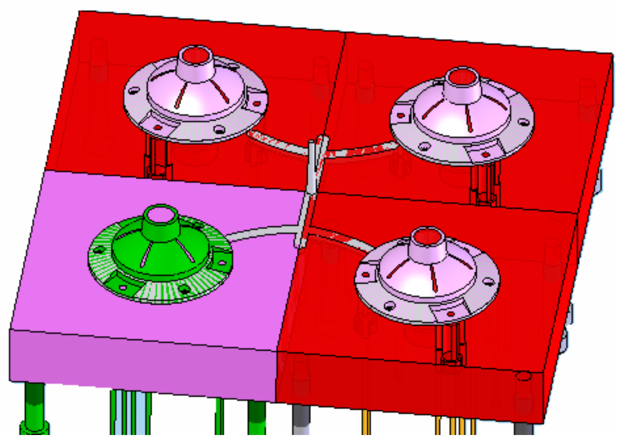
An example below showing you how to use this tool.
An assembly has 4 cavities, if we try to substract the runner from one of the cavity component, you will find that other 3 cavities are updated accordingly. This is not we want, to fix this issue, we will use the Set configuration function.

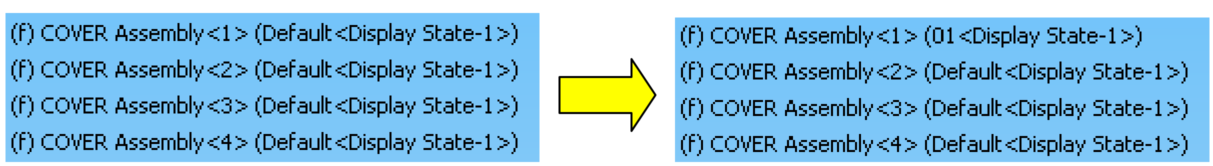




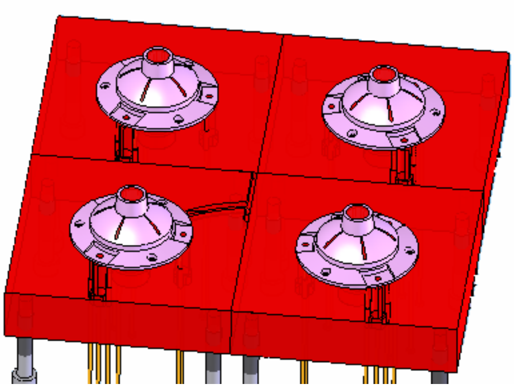
Add one new configuration named as 01 to one of four product assembly instances.







Do the “Cavity” again to subtract the runner from the core, you will find that only one instance with the configuration 01 activated affected, others remain unchanged as before.

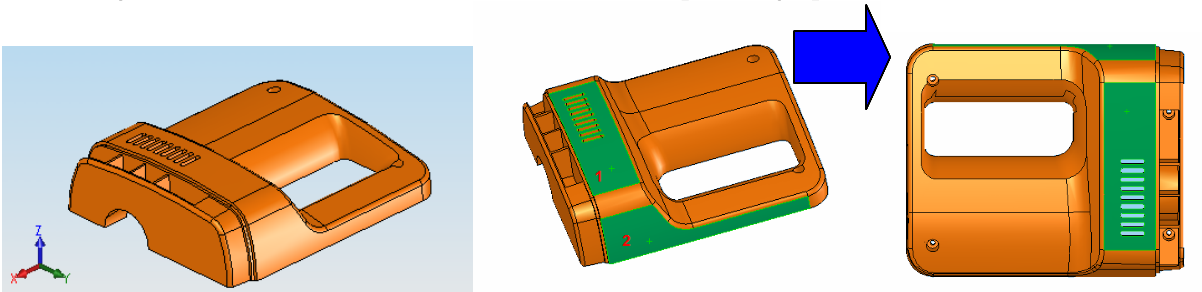


16.10 Save project

Save the entire assembly files including all children components one by one.

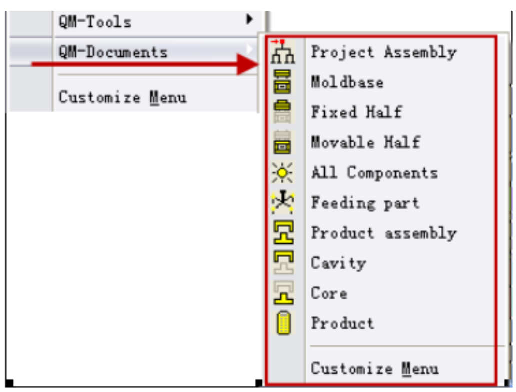
16.11 Favorite View

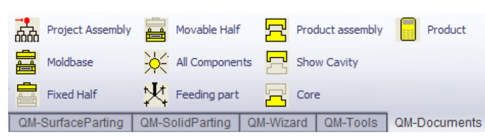
If nothing selected, click this button, a view with +Z pointing upwards is used

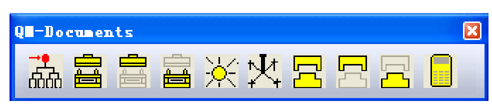


If two faces are selected, their normal directions are used to define a custom view as the above picture shown.

Chapter 17. QM-Document







1. Project assembly	/The top project assembly model
2. Moldbase	/Moldbase assembly model
3. Fix Half	/ Sometimes, it is called Upper half
4. Movable Half	/Sometimes, it is called Lower half
5. All Components	/Show all components
6. Feeding part	/The part has all runners and gates
7. Product assembly	/Assembly with core, cavity and product
8. Cavity	
9. Core	
10. Product	

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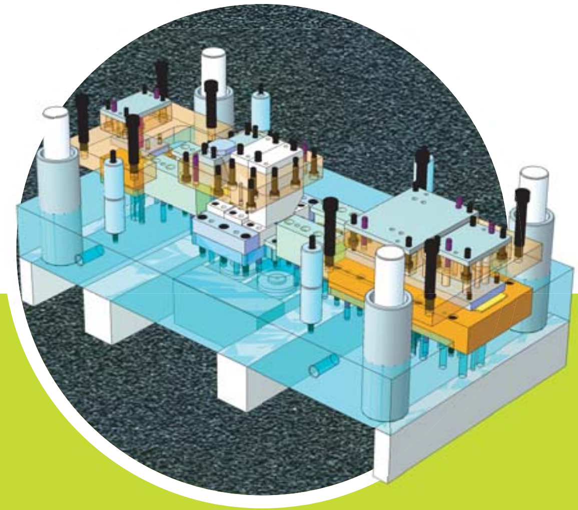

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