

PS Moldmaker 7.0 Training Course

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1. Introduction

PS-Moldmaker 7.0

PS Moldmaker has some very significant changes from previous versions, in version 7.0. there are several key points that we will cover during this training course.

- Component Wizard.
Addition of Misumi & LKM catalogues
- Mold base Wizard
Addition of LKM catalogue
- Cavity/Core Wizard enhancements.
Allow General edit operations for Multi cavity tools
Solid Pocket feature incorporated
- Mold Lock Wizard
- Increased functionality in PS-Assembly.
Power Features to create “Fit’s” for standard and user defined components
Sharing Imported components
New Tree icons
- Further Drafting enhancements.
Angled section views
Scrap Section Views
Hidden Line feature removal

Pre Requisite's

The content in this training course is aimed at the advanced user and may not be suitable for users that are new to PS Moldmaker / PS-Assembly

The exercises build on existing functionality and are designed to show only the new features, no attempt is made to explain how to use the basics of the wizards or assembly.

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2. Make Watertight

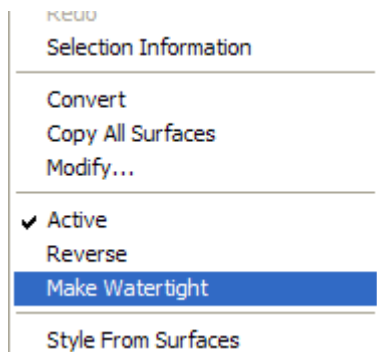
The **Make Watertight Wizard** has been restructured to improve usability. The major changes are:

- Medium Holes has replaced Large Gaps and are shown in green
- Repair options can be selected from drop down lists.
- Options to fill Large Holes and Medium Holes are on separate pages.
- You can either select options from the **Repair options** section on the first page of the wizard and **select Finish** to make a solid watertight, or use the **Advanced** options for additional control of the process.
- If a solid is watertight to a tolerance above the present tolerance and you select **Make Watertight** from the solid pop-up menu, the **Seventh page of Make Watertight Wizard** is displayed..

Open Clip.psmodel.

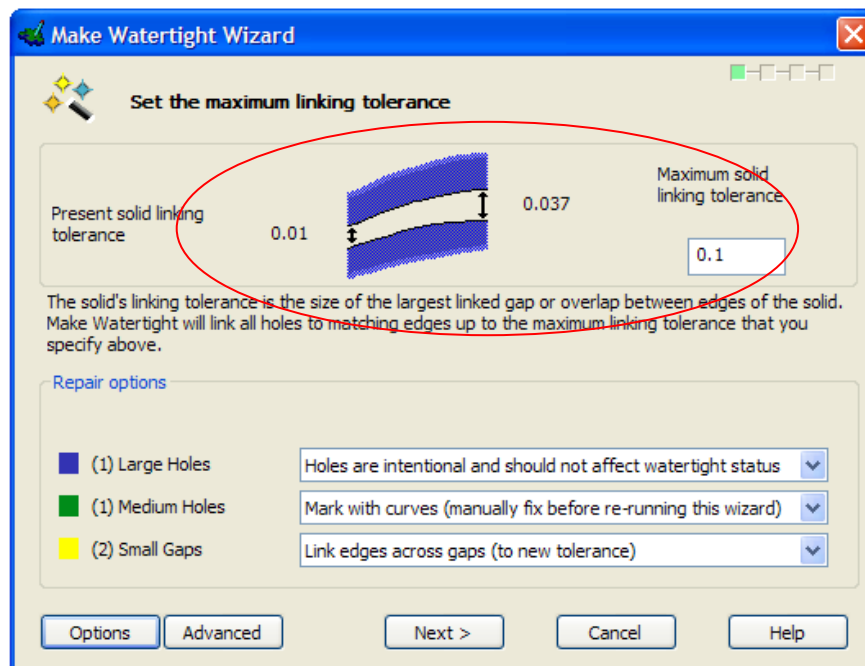


- To start the Make Watertight use the RMB menu whilst over the solid.



A prompt to Make watertight will be raised if you are creating a solid from surfaces and prior to running Die Wizard and Slide Core wizard.

The First page of the Wizard is displayed:-



The Wizard reports the problems found based on the current linking tolerance.

The Displayed NEW Solid linking tolerance is based on the distance required to close **small gaps ONLY**. In this case it is **0.037mm**

The **Maximum solid linking tolerance** is shown as **0.1mm**, this value taken from the options page. The default is 10 x present solid linking tolerance

Large Holes

One large hole is identified. The default option displayed in the wizard is making the assumption that the hole is intentional and should not affect the watertight status.

As this hole must be repaired the option should be changed to mark with a curve for manual fixing.

Medium Holes

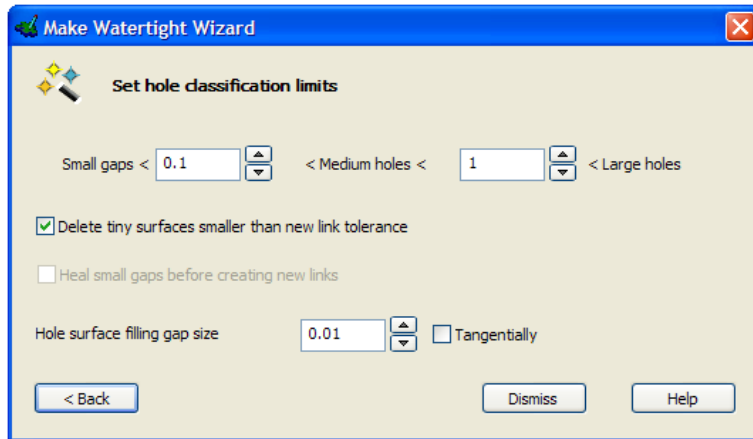
One medium hole is also identified

The default option shown by the wizard is to create curves for manual fixing. This option is correct in this case.

Small Gaps

There are 2 small gaps identified by the wizard at the current linking tolerance.

Options.



Hole classification limits.

Holes and gaps are classified by default as:-

Small gaps < 10 x tolerance
 Medium holes < 100 x tolerance
 Large Holes > 100 x tolerance

The options page allows the user to change the classification values for the limits

Delete tiny surfaces.

The definition of Tiny surfaces is any surfaces that are smaller than the link tolerance. Tiny surfaces were previously referred to as Sliver surfaces.

By default the link tolerance will be set to the maximum value that will not delete any tiny surfaces. By ticking the option to delete then the linking tolerance can be set to a greater value.

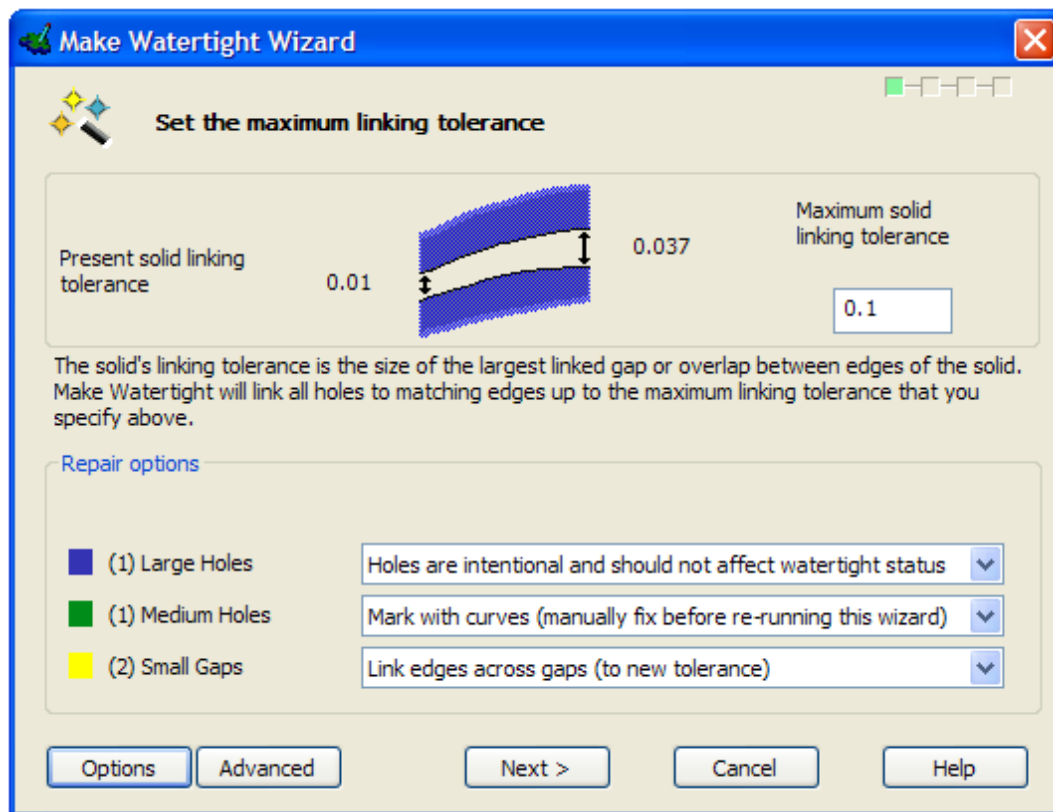
If surfaces are subsequently deleted they may leave a medium hole, the wizard will reassess the solid once any surfaces are deleted.

Hole surface filling gap size.

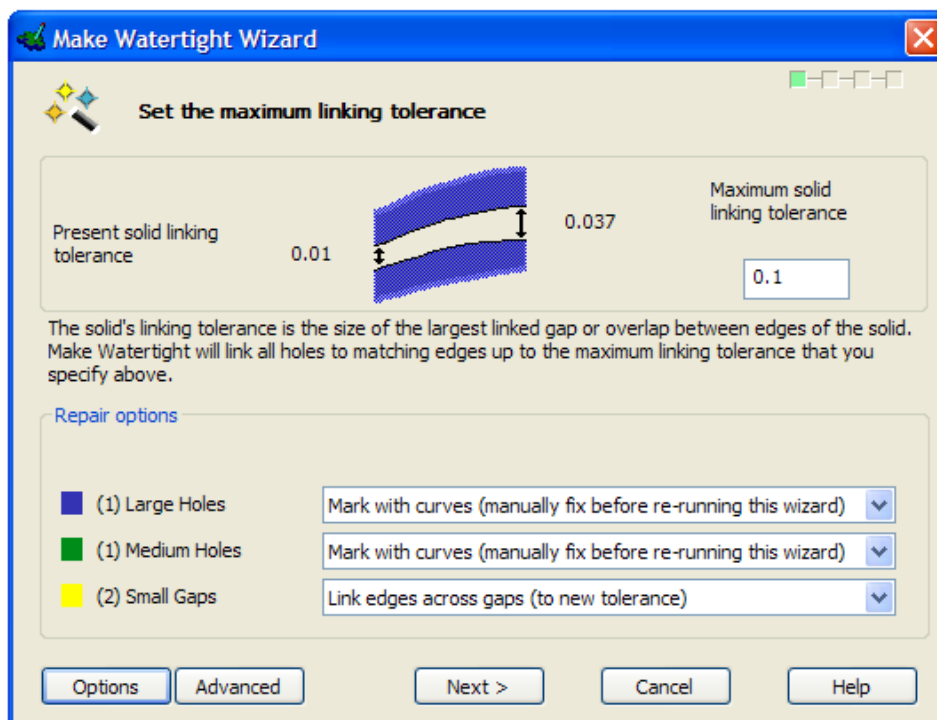
The maximum size of a gap in the boundary of any filled hole that is to be ignored.

The default setting is general tolerance

If this value is set higher there is more chance of small gaps being left when a surface is created



- Change the option on the large hole to **mark with curves**.

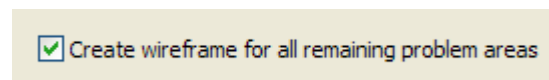


- Now press **Next** to progress



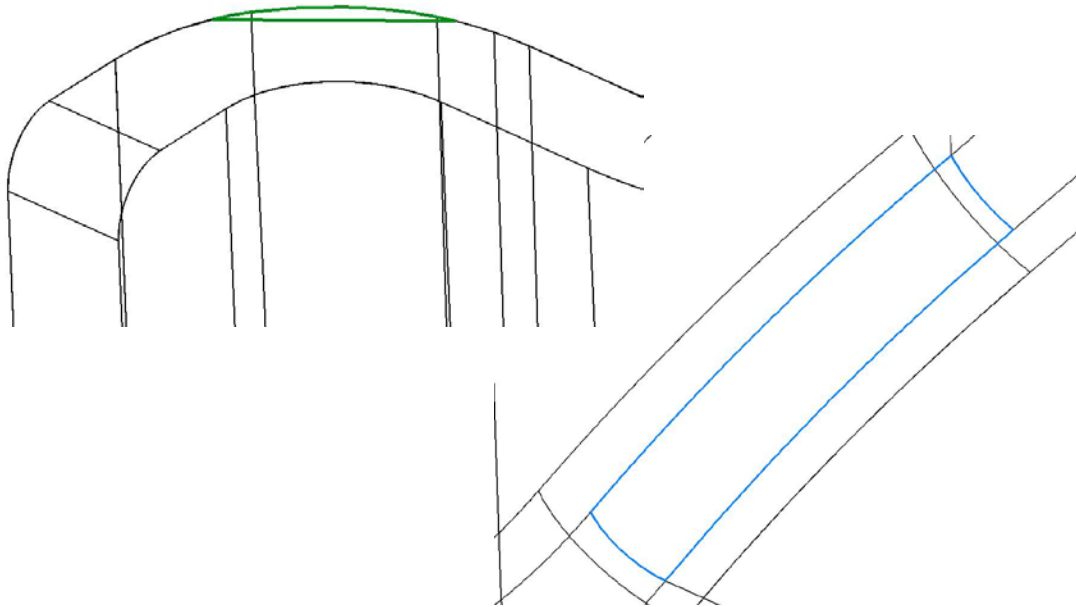
The wizard is showing what will happen when Finish is pressed.

Be sure to check that the create option wire frame is ticked.



- Hit **Finish**

The wizard will now close and create curves for the large and medium holes



the Wire frame curves that are created by the wizard are now coloured and thicker for easier recognition.

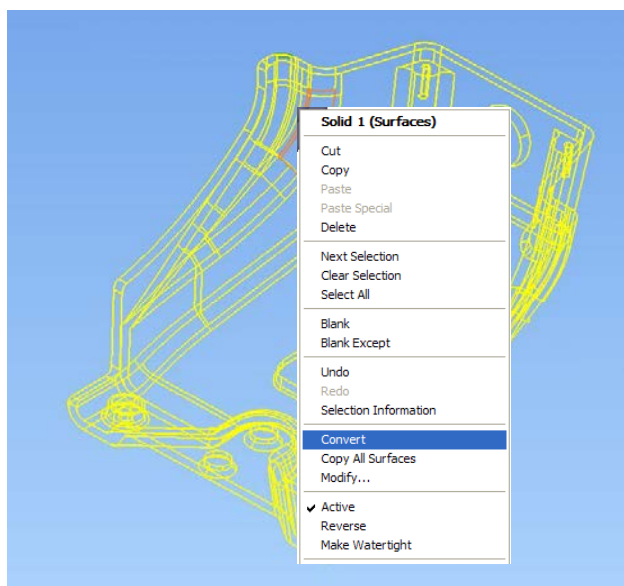
Curve colours

Large Holes are coloured Blue

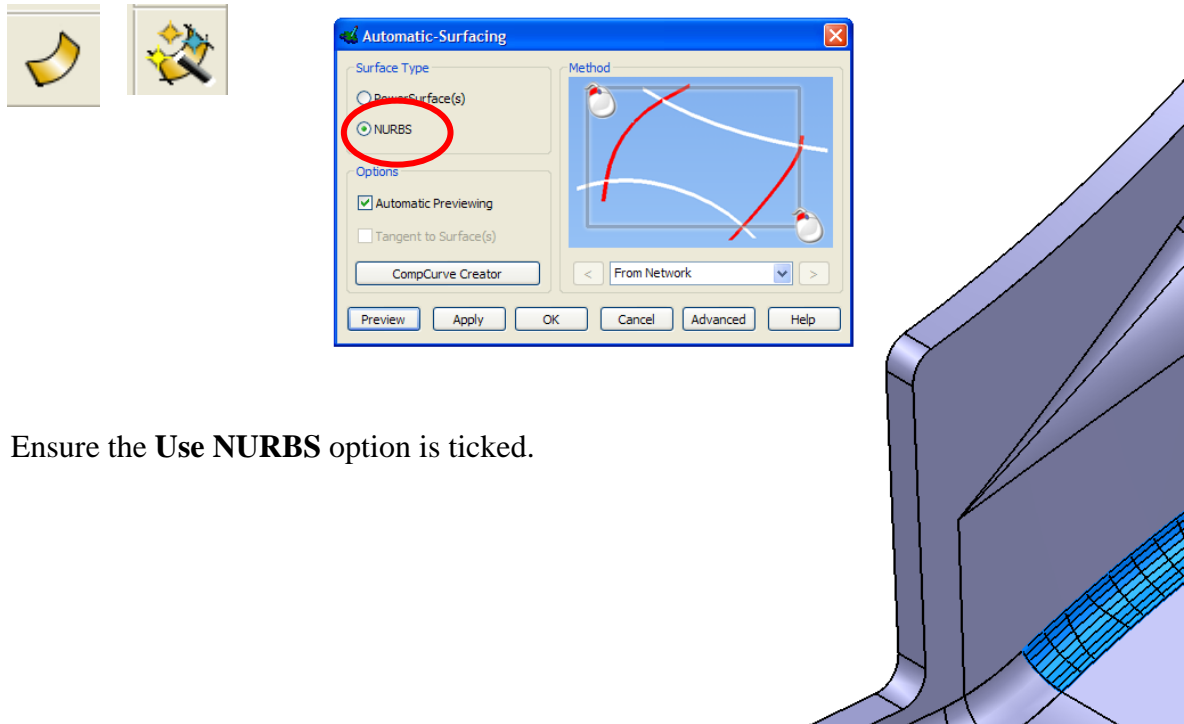
Medium holes are coloured Green

Small gaps are coloured yellow

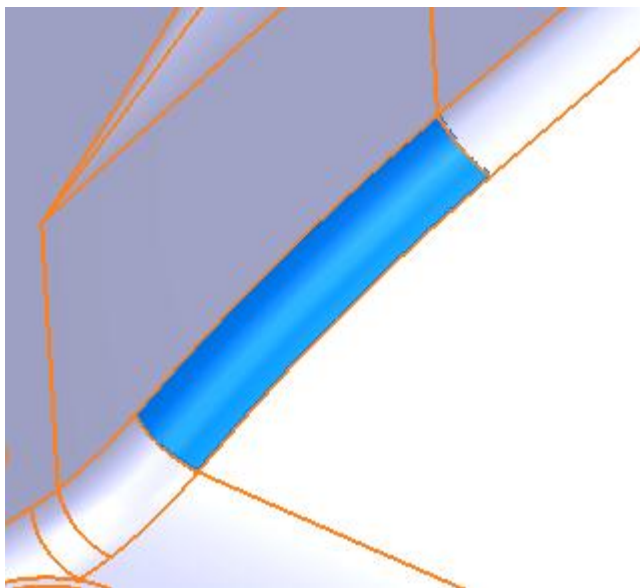
- Using the curves, **convert** the solid to surfaces and repair the **Large** hole.



- Select the **Blue Wireframe** and create a surface from **network of Wireframe**.

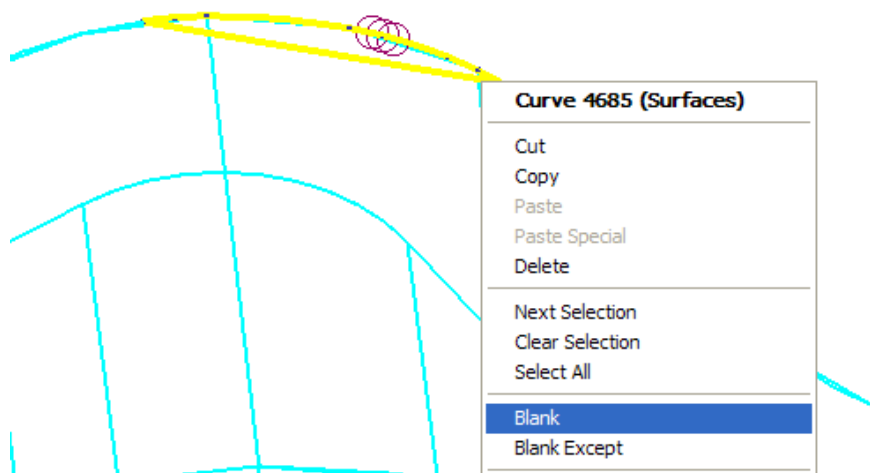


- Ensure the **Use NURBS** option is ticked.



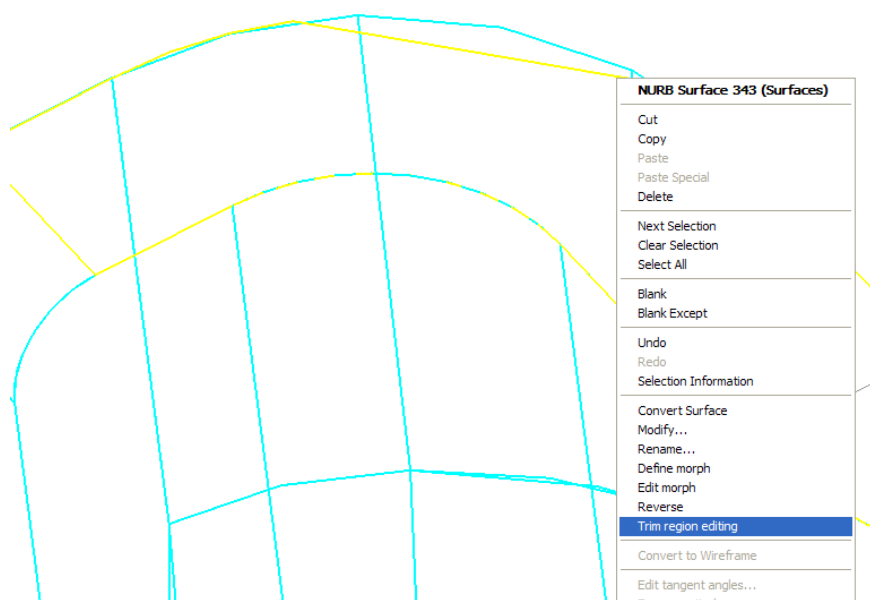
Now progress to fix the Medium Hole, this is marked with Green Wireframe.

- Select the **Green** Wireframe and **Blank**



In this case we have blanked the Wireframe as it is only used to indicate clearly the problem area. You do not need the curves to repair the hole

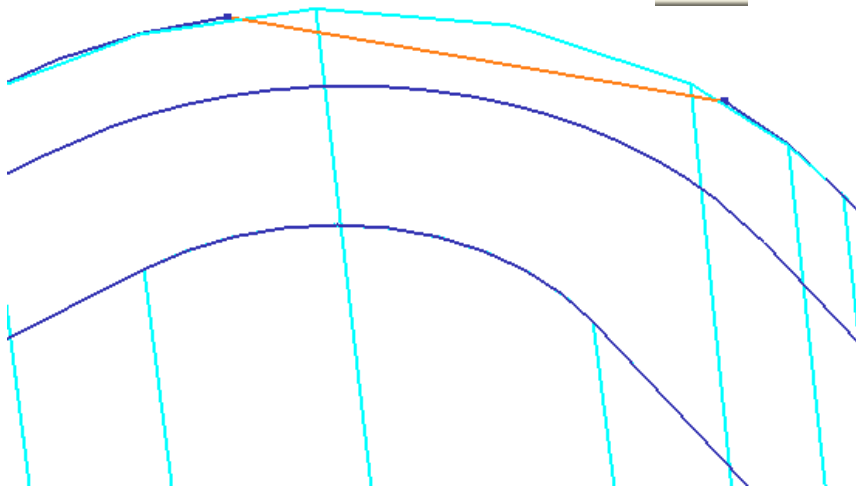
- Select the surface indicated and enter **Trim Region Editing** via RMB



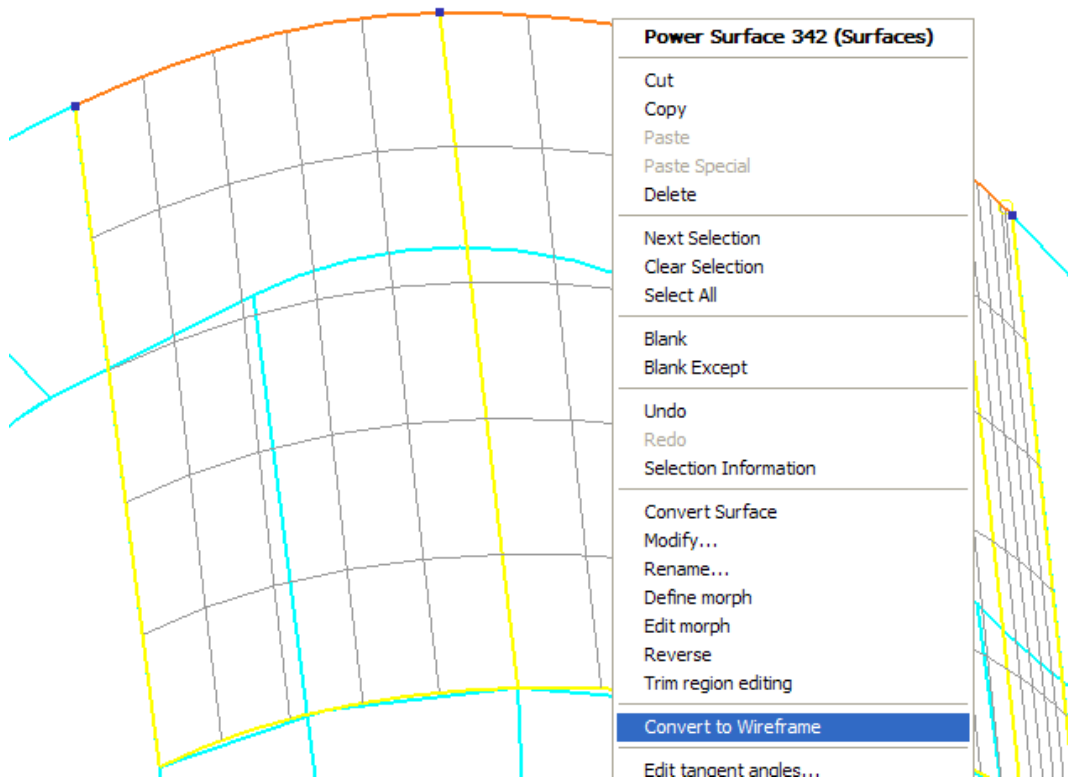
- Select **P-curve** Edits from the trim region editing toolbar



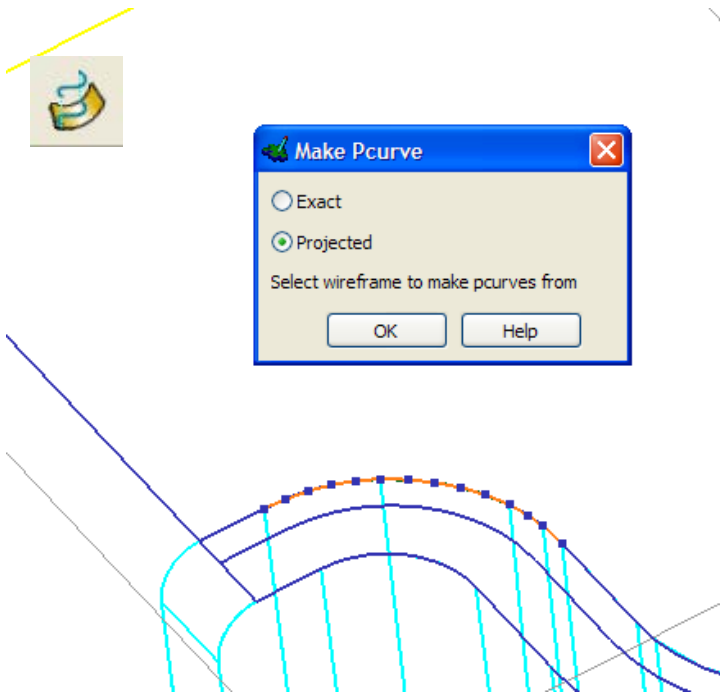
- Select the incorrect P-curve and **delete** it



- Select the edge indicated on the adjacent surface and **convert to wireframe**



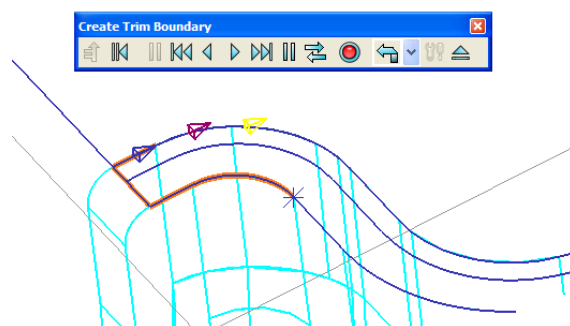
- From the wireframe create a P-curve using the **projected** option.



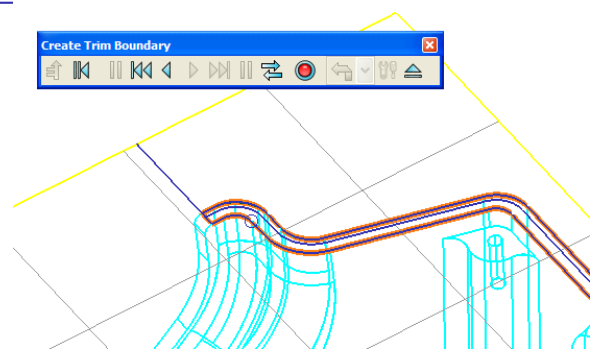
- Return to **Boundary Edits**



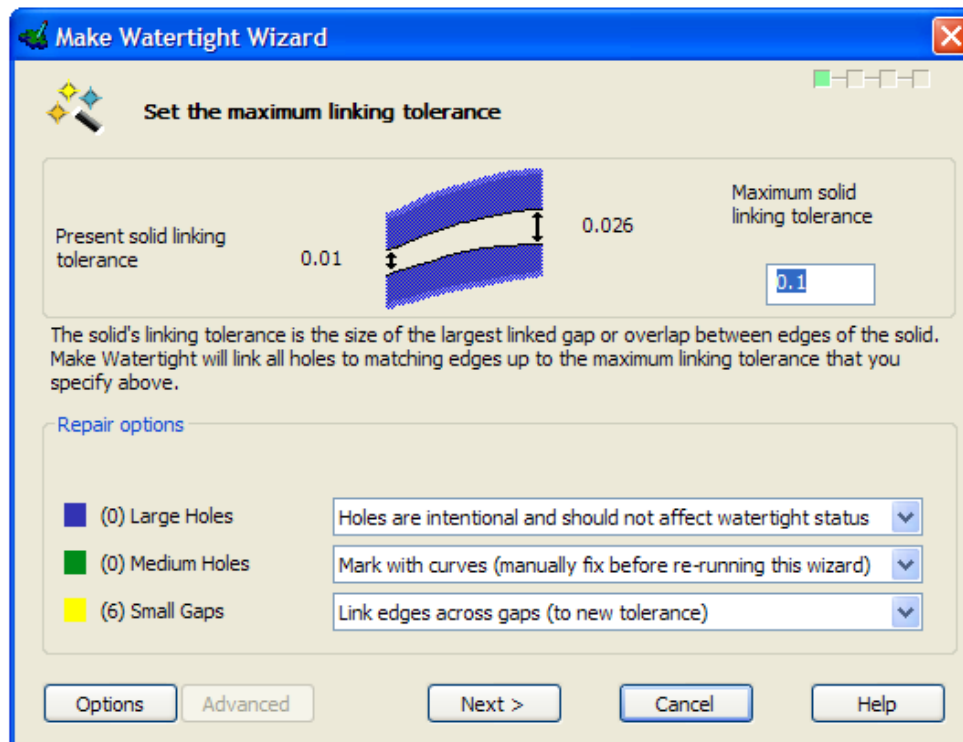
- Recreate** the Trim Boundary



Reverse the boundary if necessary



- Re-create the solid and run make watertight to verify the results

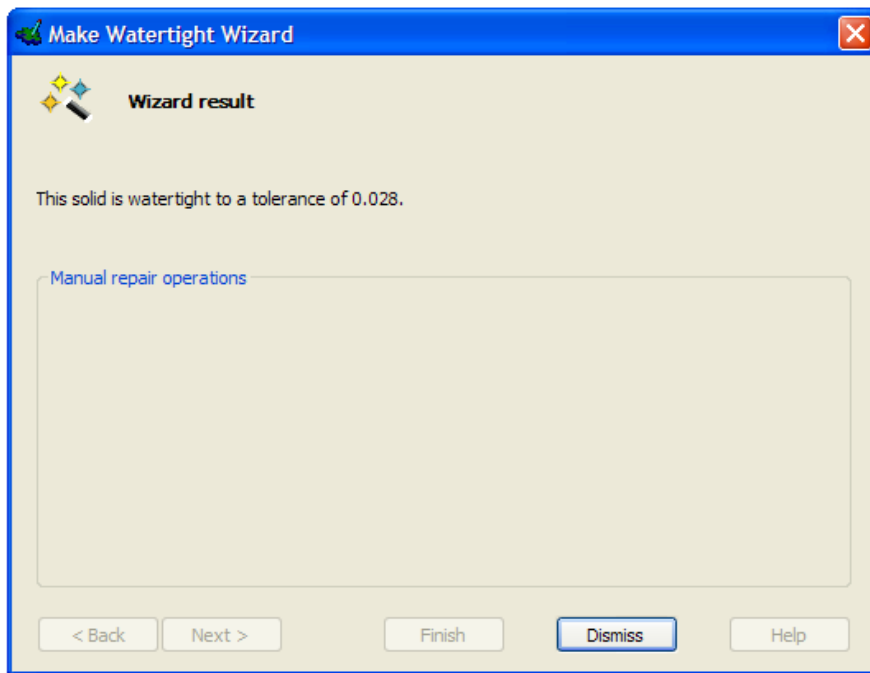


- **Next** to progress

The seventh page of the wizard is displayed with the option to heal the solid edges if possible or to leave the edges as they are



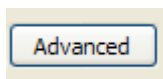
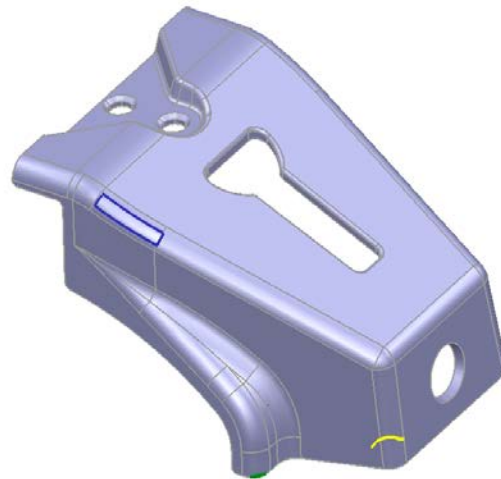
- **Finish**



The solid is Watertight to 0.028mm

Advanced Fixing in Version 7.0

In addition to the methods you have just used Version 7 contains some Advanced repair tools.



Select the advanced option

The Advanced option allows **Large** and **Medium** holes to be fixed within the wizard. When activated each hole will be highlighted and the graphics window will show the actual hole that you are going to fix.

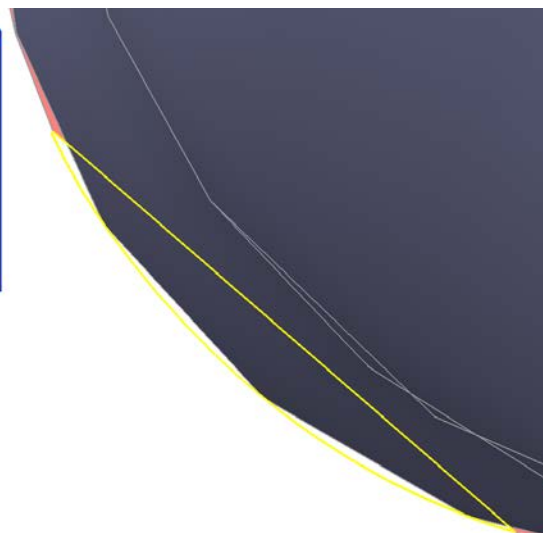
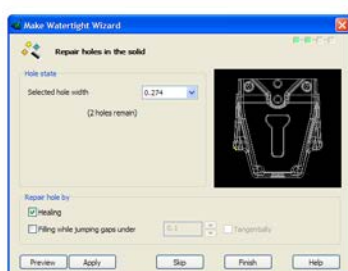
The options that can be used are:-

Healing

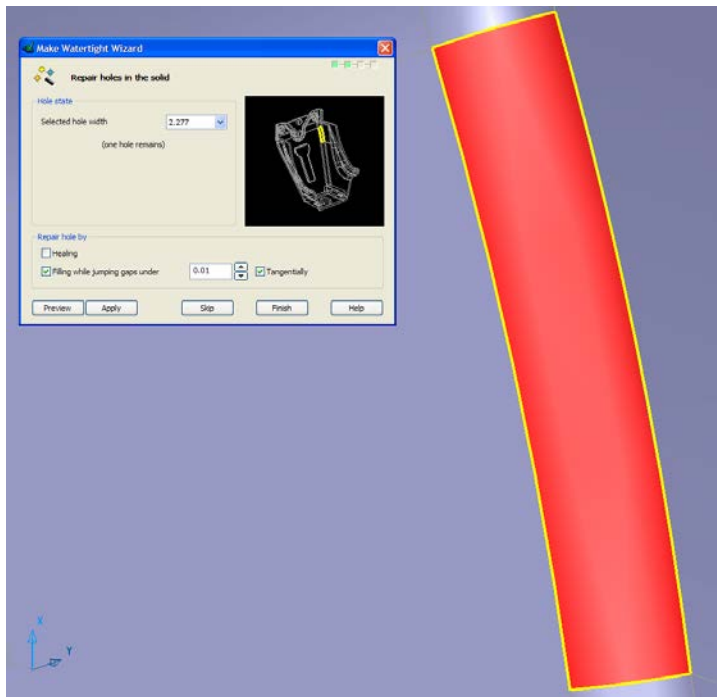
Fill with NON tangential surface

Fill with tangential surface

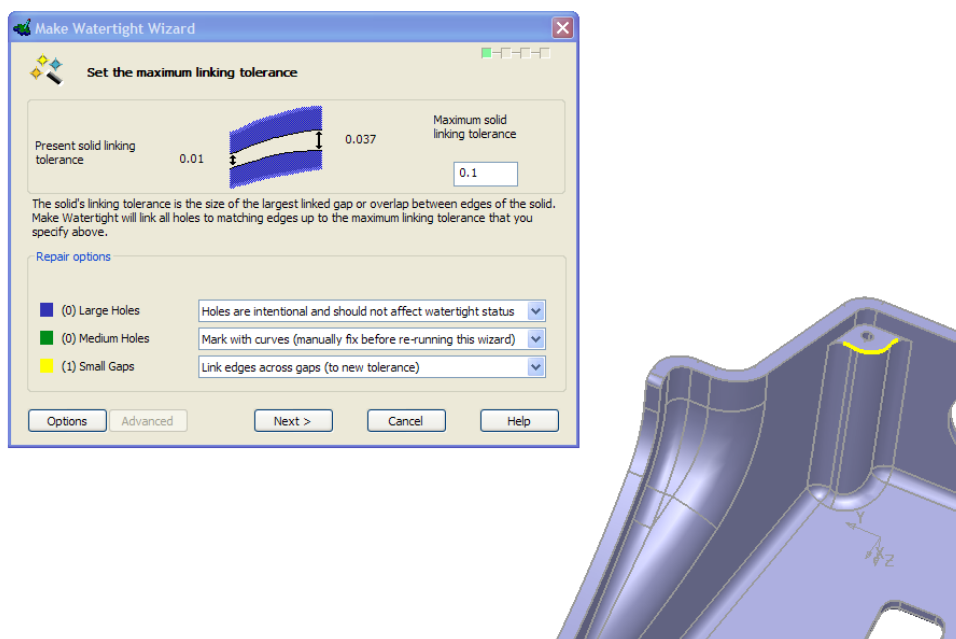
The first hole that is highlighted can be fixed by healing the surface boundary



- Fill the second hole with a tangential surface

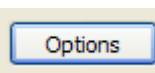
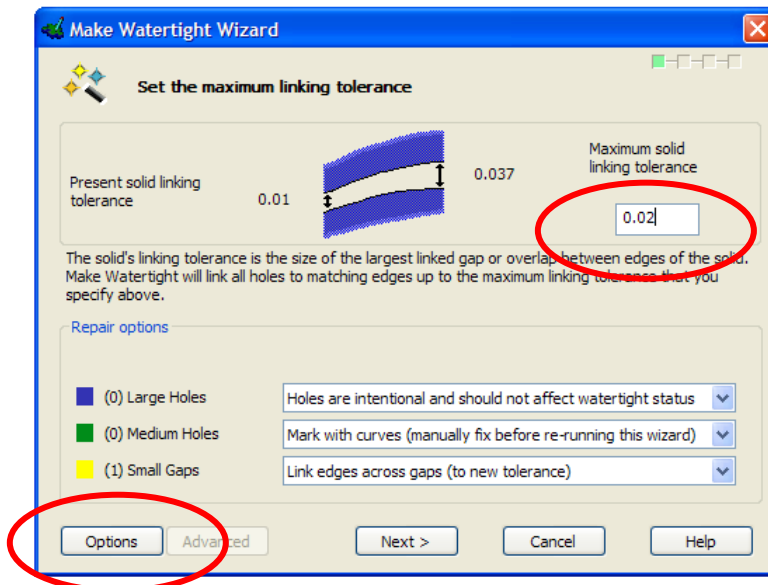


Once the Large and Medium holes are fixed then you are left with 1 small gap

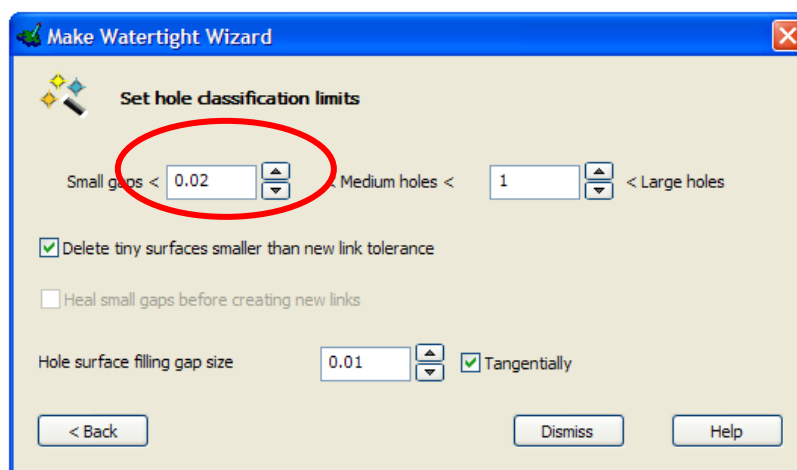


Change the Maximum solid linking tolerance to 0.02

This will set the maximum gap that you will allow the wizard to fix.



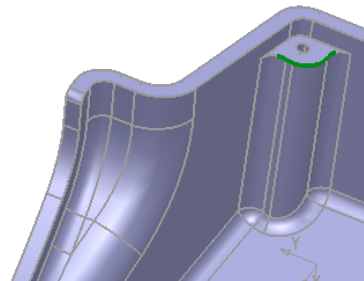
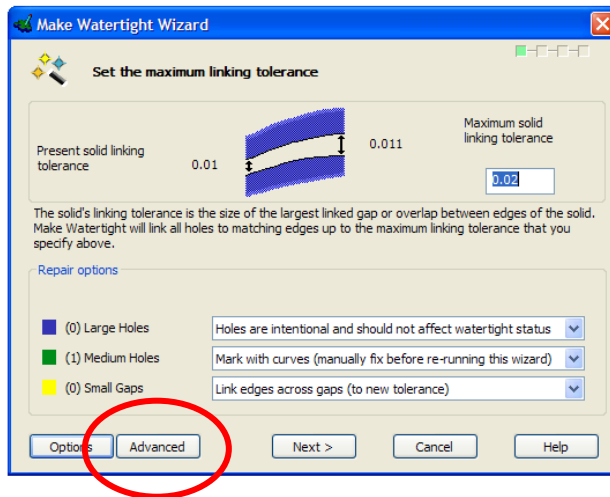
- Now Select the options button



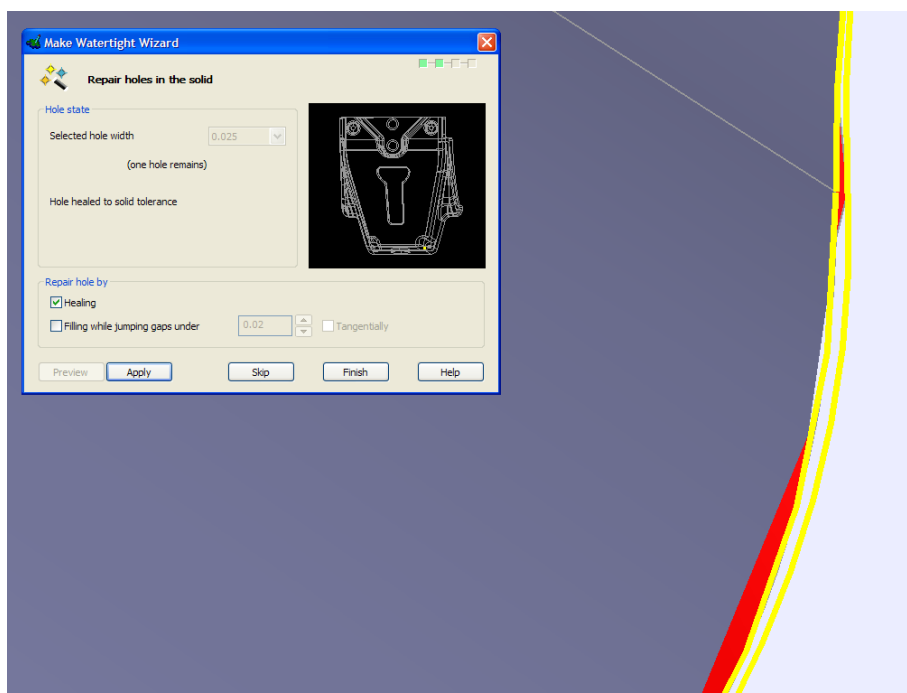
Reduce the Small Gap classification value to 0.02, this will change the existing small gap to a medium hole.

By changing the gap to a Medium hole you can now use the Advanced fixing to Heal the gap.

- Select the Advanced option.

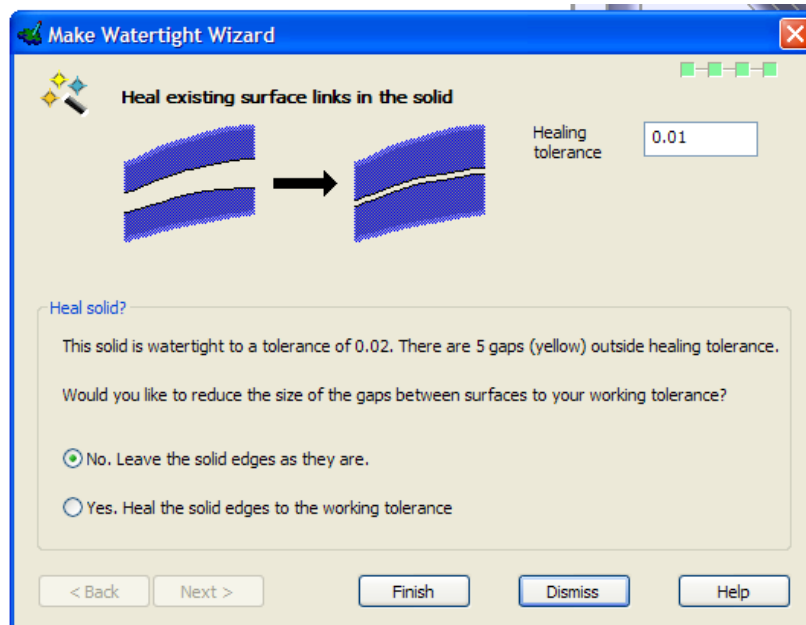


- Heal the Gap.



Now that the gaps / holes are fixed the wizard will report that the solid is watertight. There are two options available.

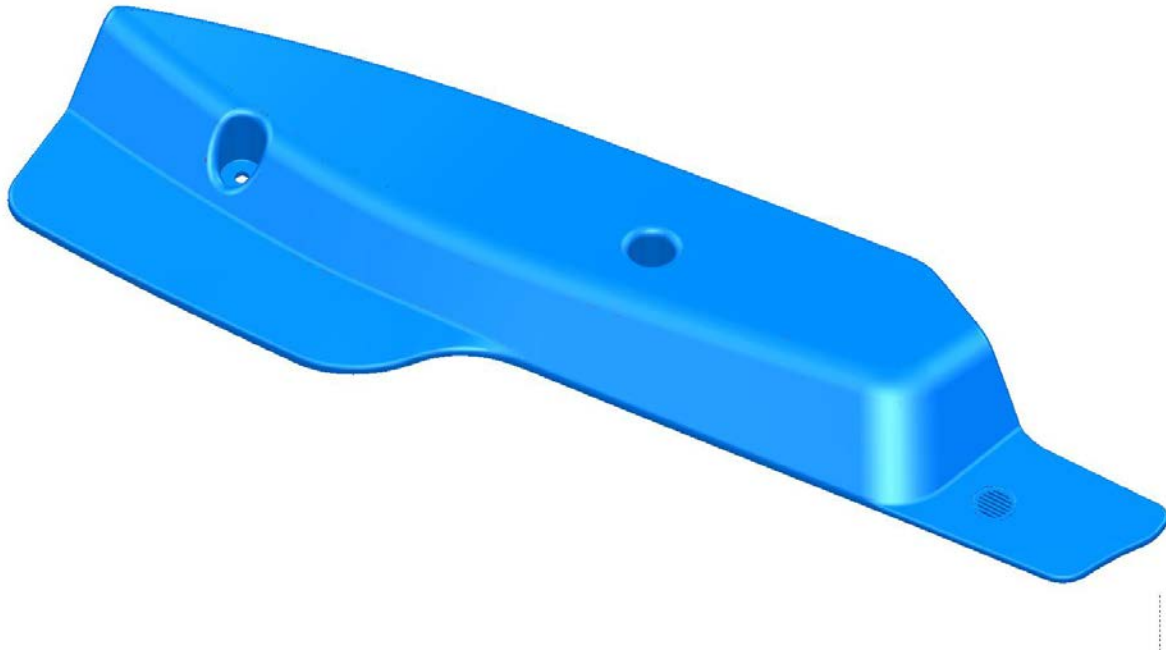
Generally the best option to take is the default and leave the edges as they are.



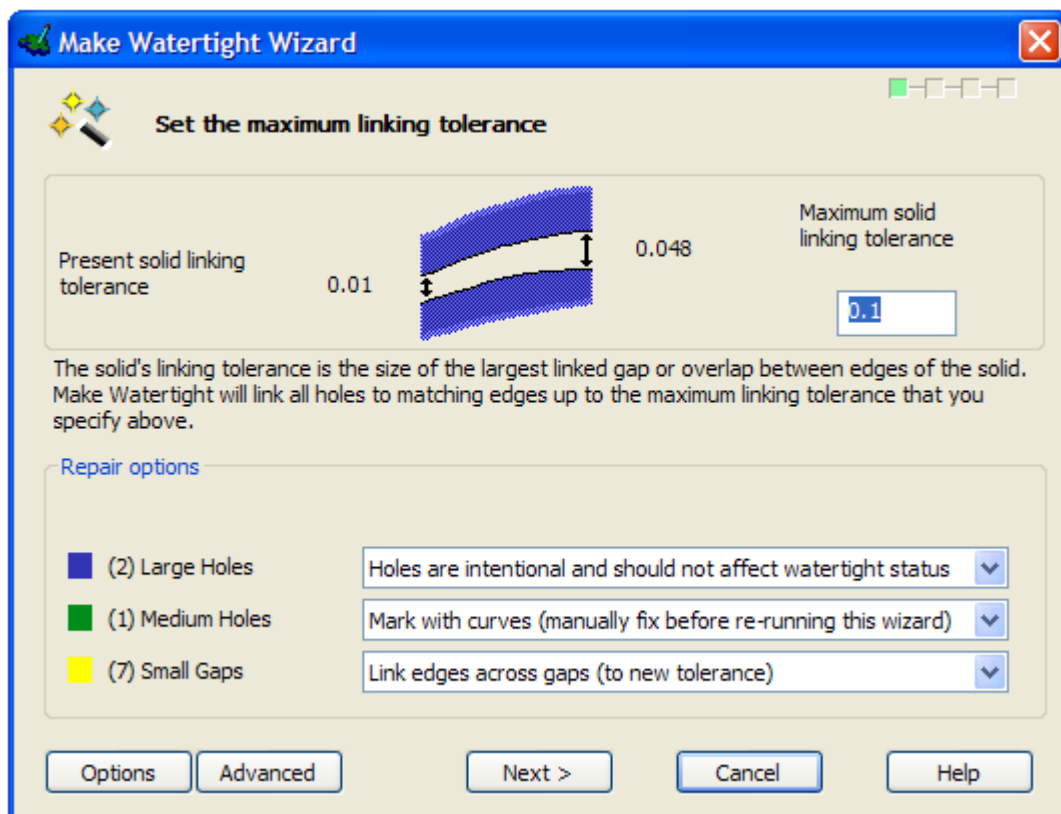
- Hit **Finish** then **Dismiss** to exit the wizard



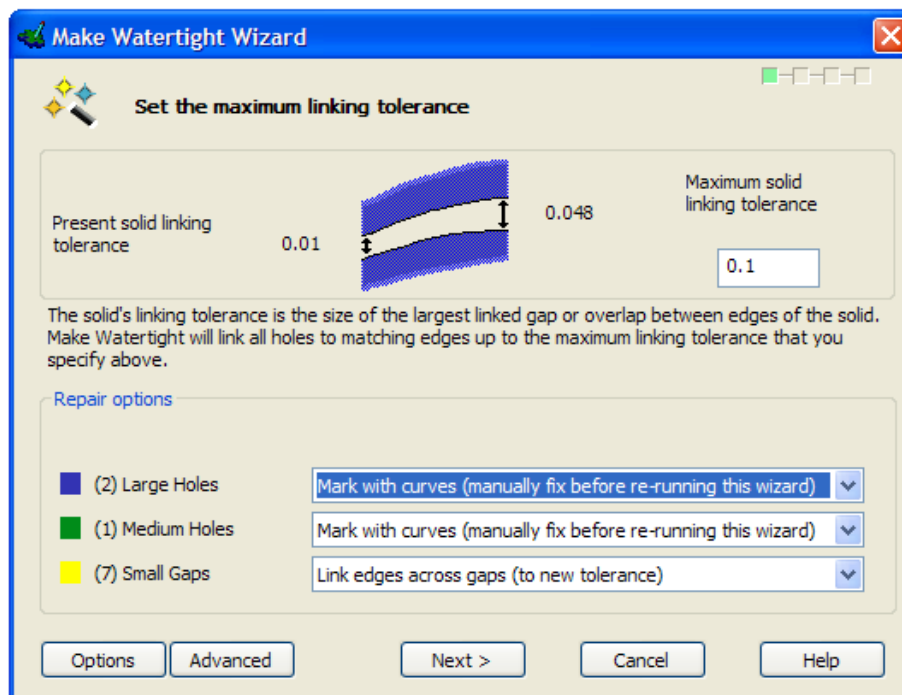
Open the Model **Trim.psmodel**



- Run **Make Watertight**



- Select the repair options to **Mark with curves** for large and medium holes



- Next



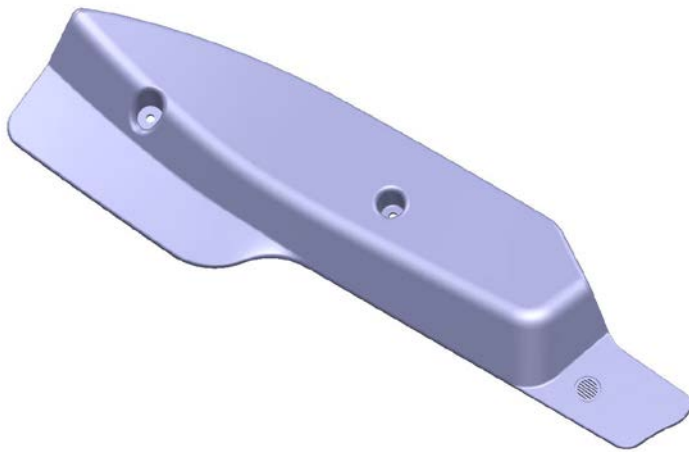
- **Finish**
- **Convert** the Solid back to surfaces and repair the holes.
- Re run Make Watertight.

3. Die Wizard

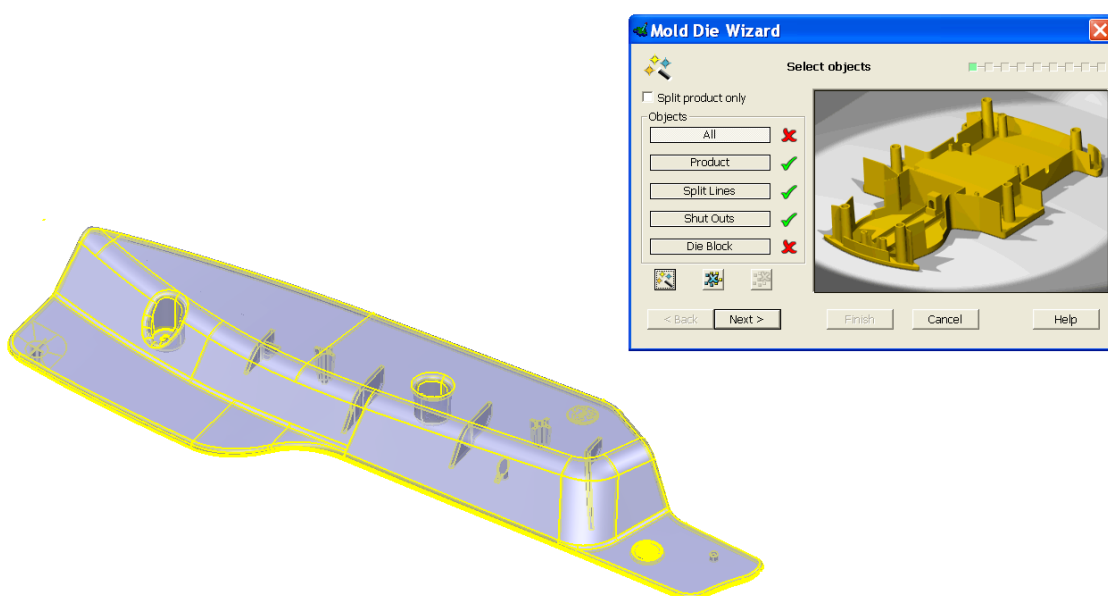
Stepped Split Surfaces

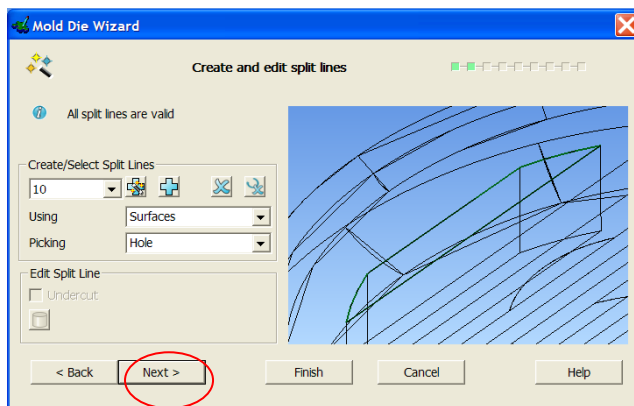
A number of enhancements have been made to Die Wizard; these include the creation of Stepped Split surfaces from within the wizard.

Open model – Stepped_split.psmodel

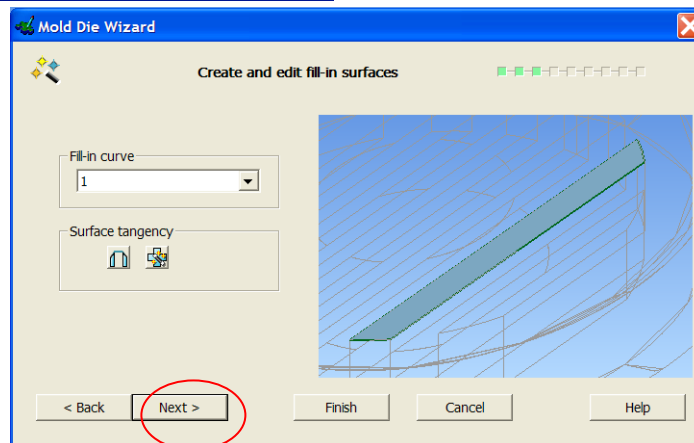


- Start the **Die wizard**, and box select all entities.



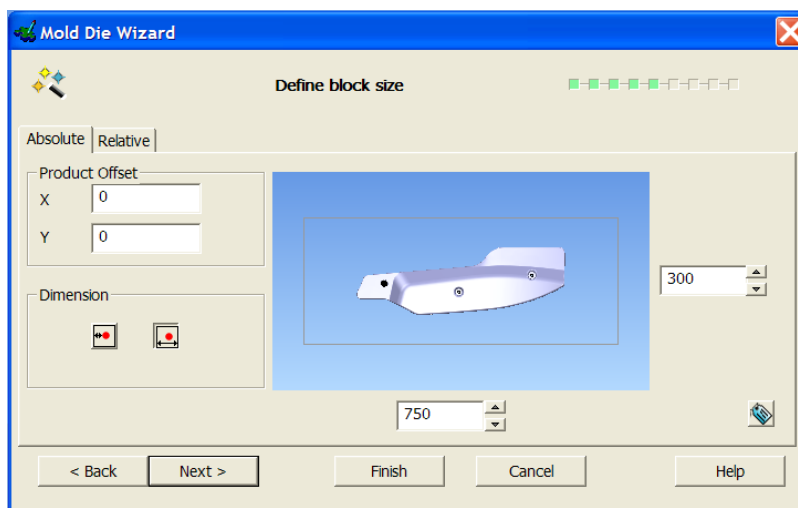
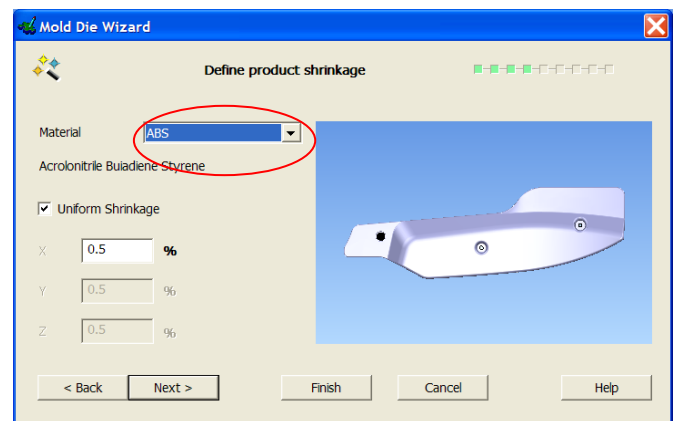


- Press Next

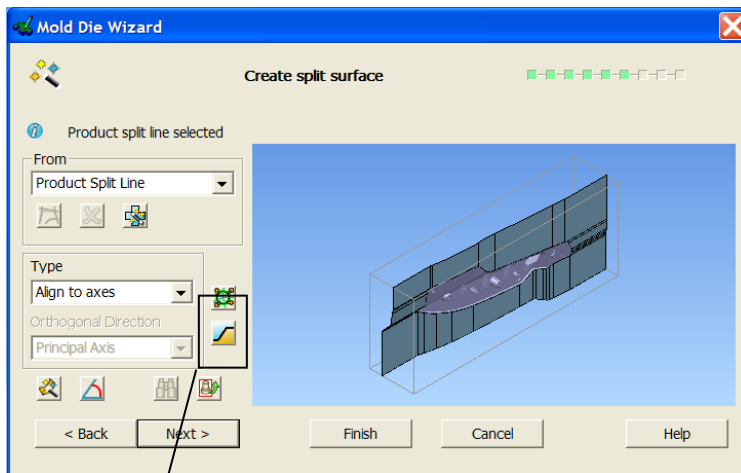


- Press Next

- Change the material to ABS to add shrinkage



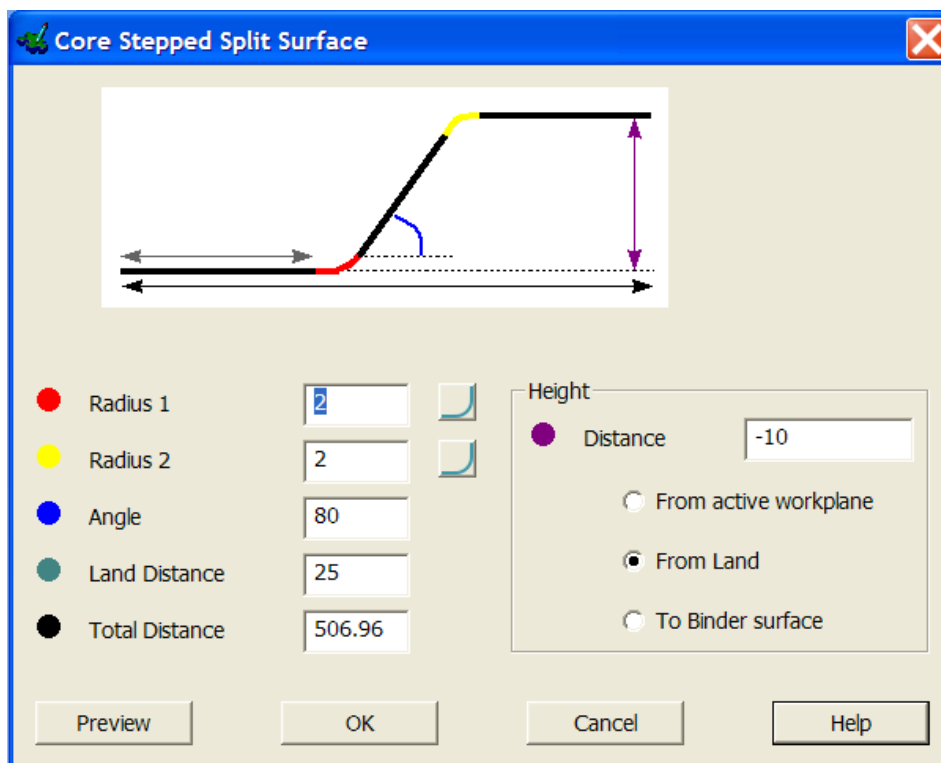
- Modify the block sizes to 750 x 300



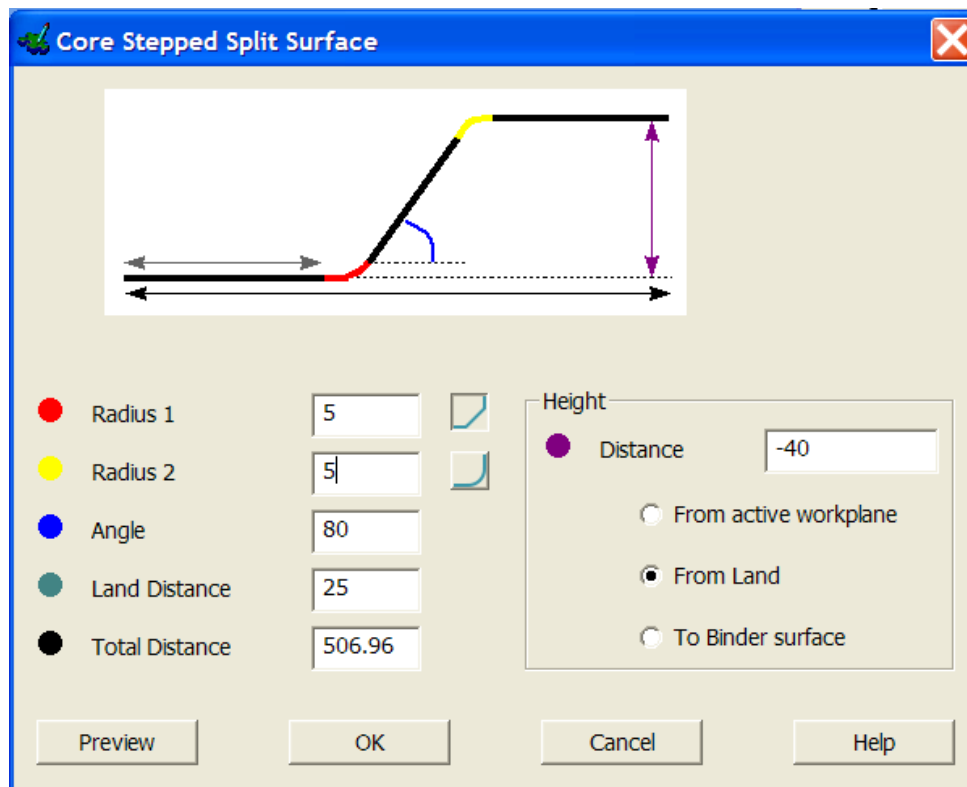
- Select the **stepped split** option

The Stepped Split surface form is raised; there are several options to set the place from which the height of the surface is measured.

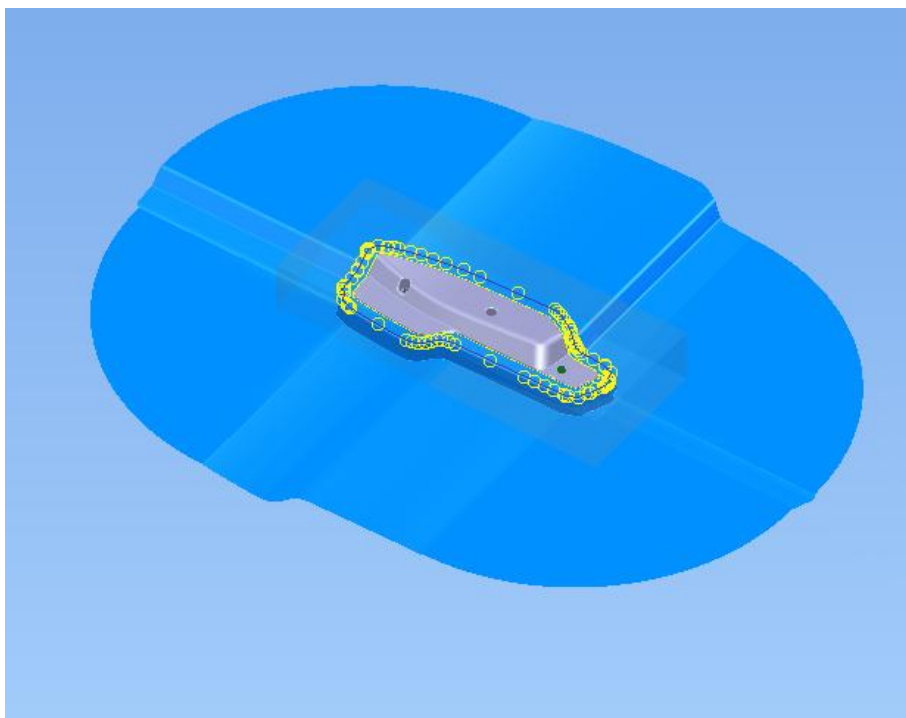
In this example we will keep the default of height 'from land'



- Change the **Height** distance to **-40**, **Radius 1** to **chamfer 5mm**, **radius 2** to **5mm**



- Hit **Preview** to see the surface.

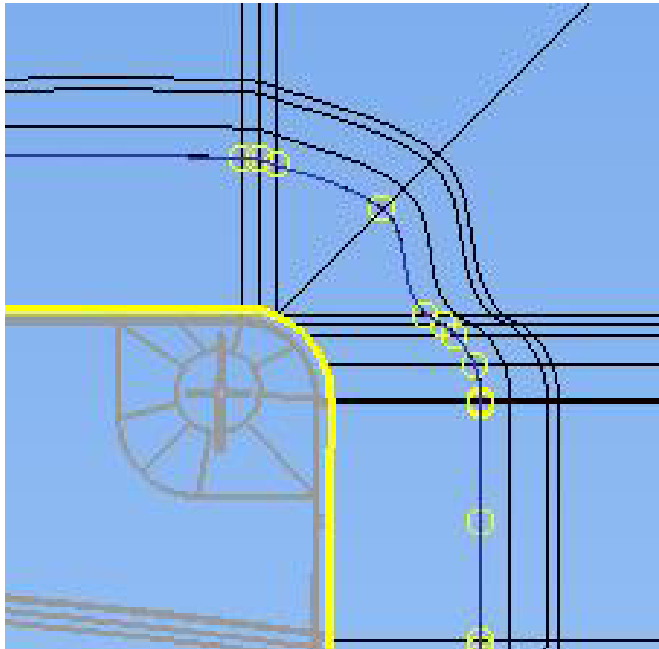


You can see that the surface has several 'Control Points' highlighted yellow.

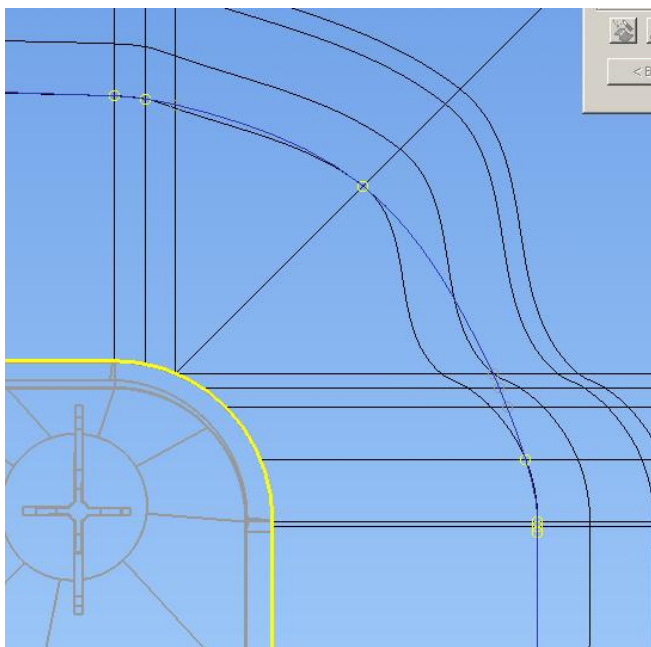
These control points can be toggled on or off to give a smoothing effect on the surface. This feature is very useful when the stepped surface does not need to follow the part geometry exactly.

This smoothing of the surface can produce a surface that is more suitable for machining.

- Select a **top view** and toggle ‘off’ some of the points to create a smoother surface. The preview will update as you go.

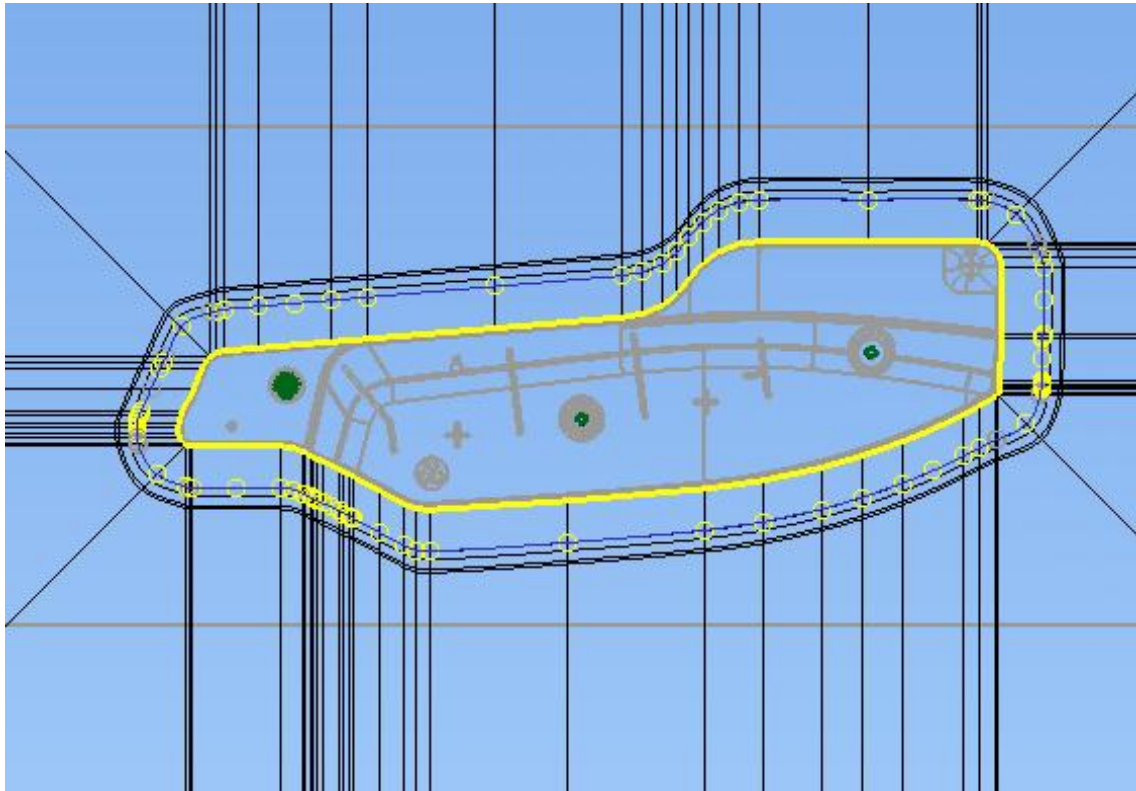


Before



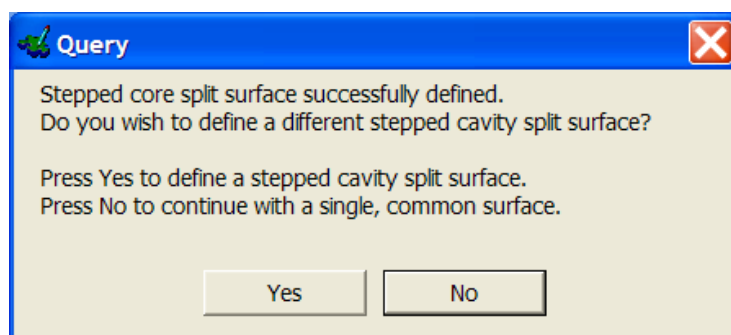
Preview of new profile

- Repeat the process of ‘toggling points’ until you have a smooth profile



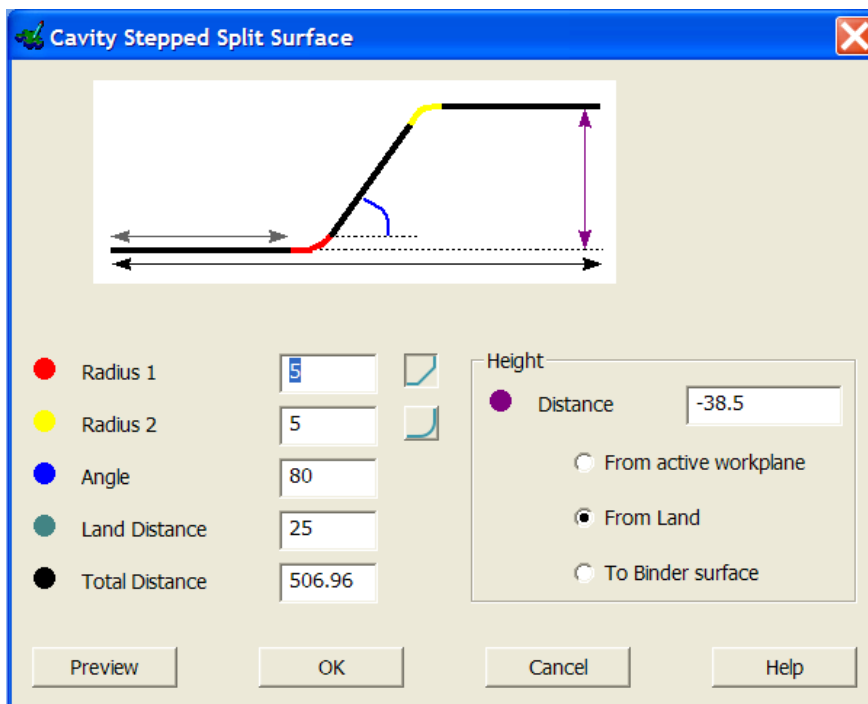
- Hit **OK** on the Form.

You will be asked if you wish to create a slightly different surface for the cavity block, this allows control over clearance between the Mold halves.

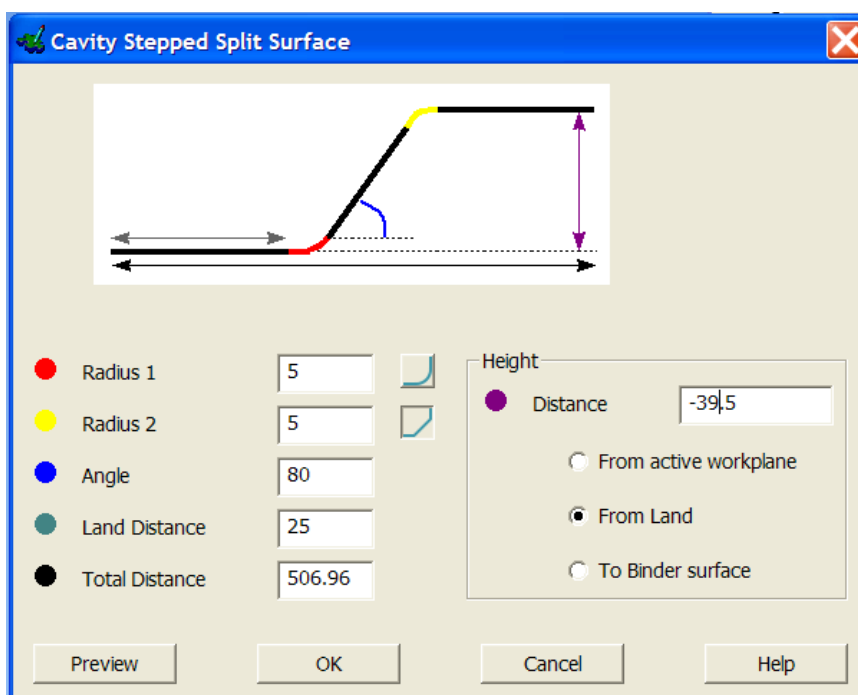


- Select **Yes**.

The Stepped Split surface form for the cavity will be raised

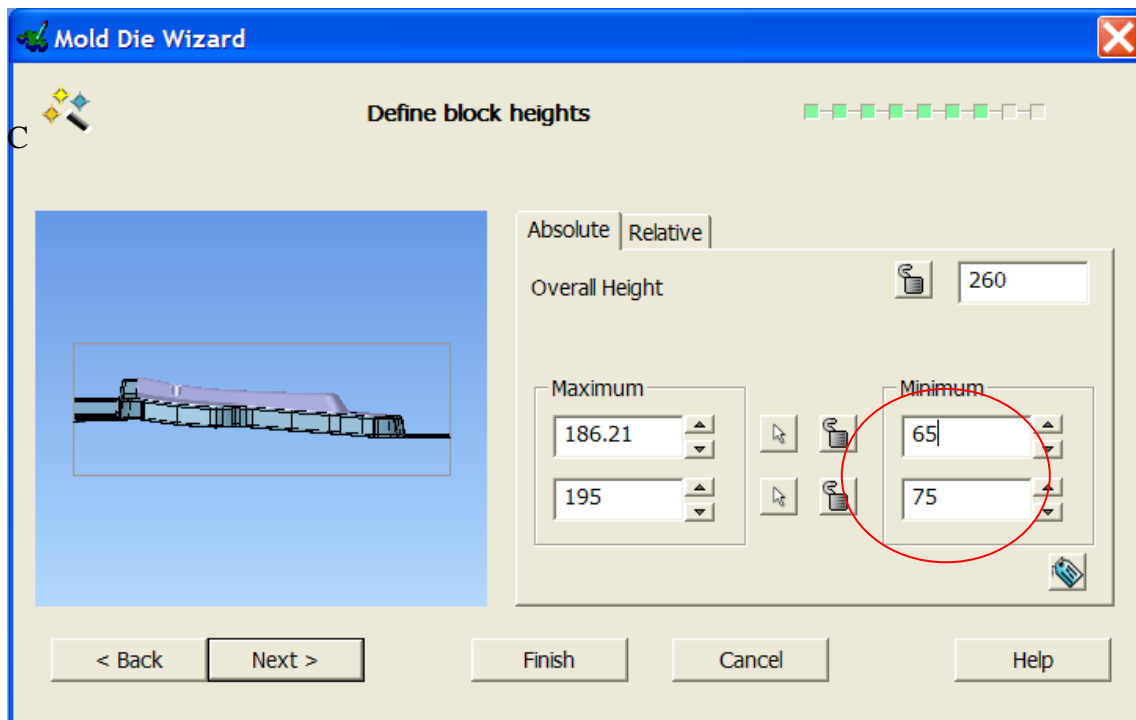


- Change the **Height** value to **-39.5mm**; this will create a 0.5mm gap on the outer flat area of the split surface. All other areas will match the core surface.
- Change **Radius 2** to a **chamfer 5** and **Radius 1** back to **radius 5**

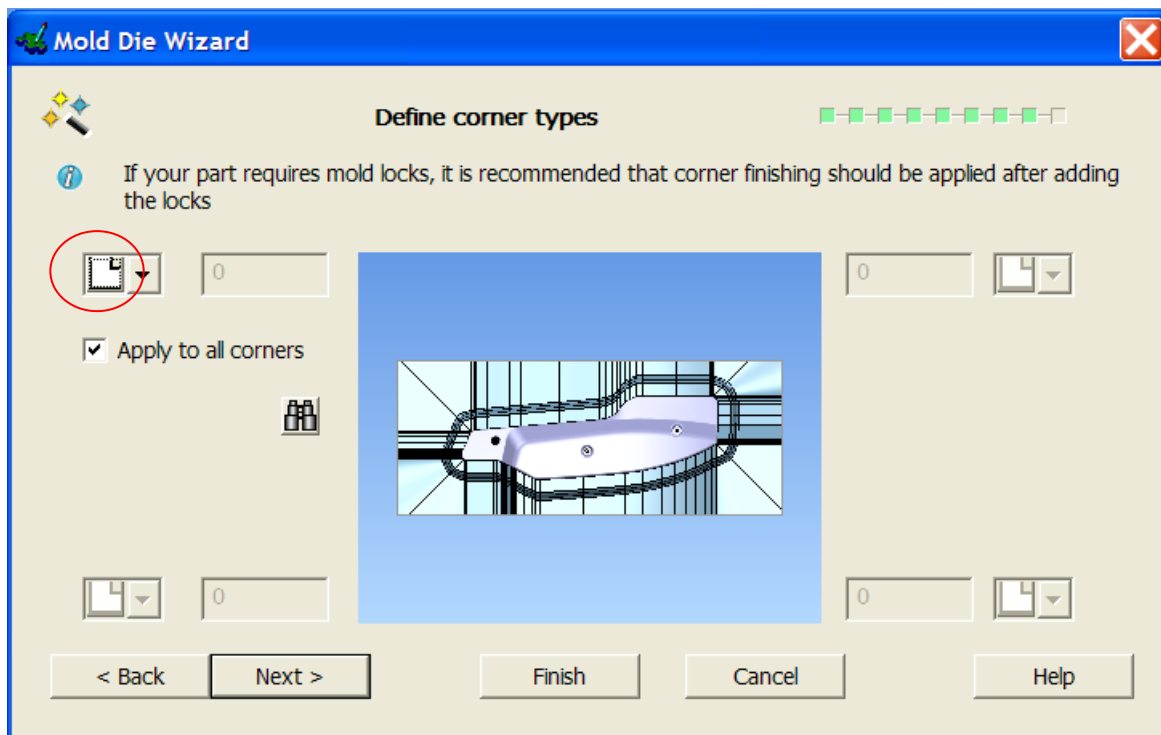


- **OK** the form.

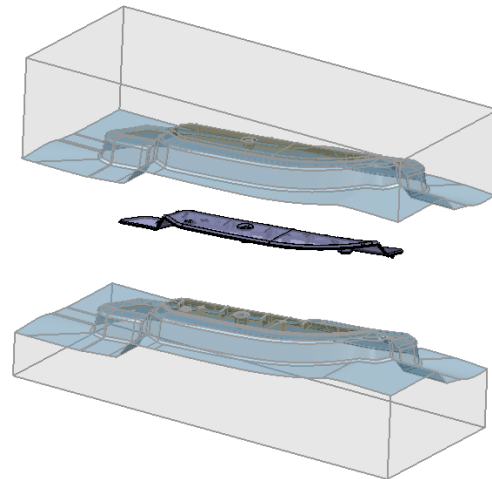
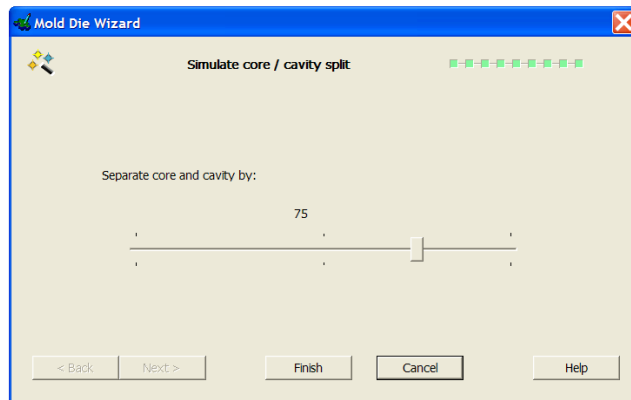
You are now back to the define block heights page in the die wizard.



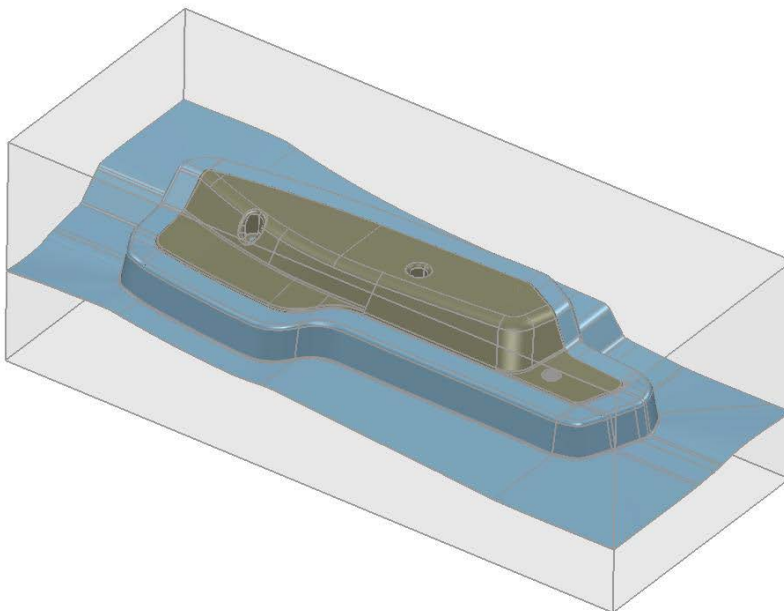
- Change the **absolute minimum** values as indicated above.



- Change the corner type setting to **square**. Hit **Next**



- **Preview** the results, and then hit **Finish**.



Die wizard has a new colour scheme to improve clarity; this scheme separates the split surfaces from the part geometry.

The block remains transparent so that internal features added later can be easily seen.

- **Close** the model.

User defined Split surfaces

Split surfaces that have different geometry can also now be used with Die Wizard.

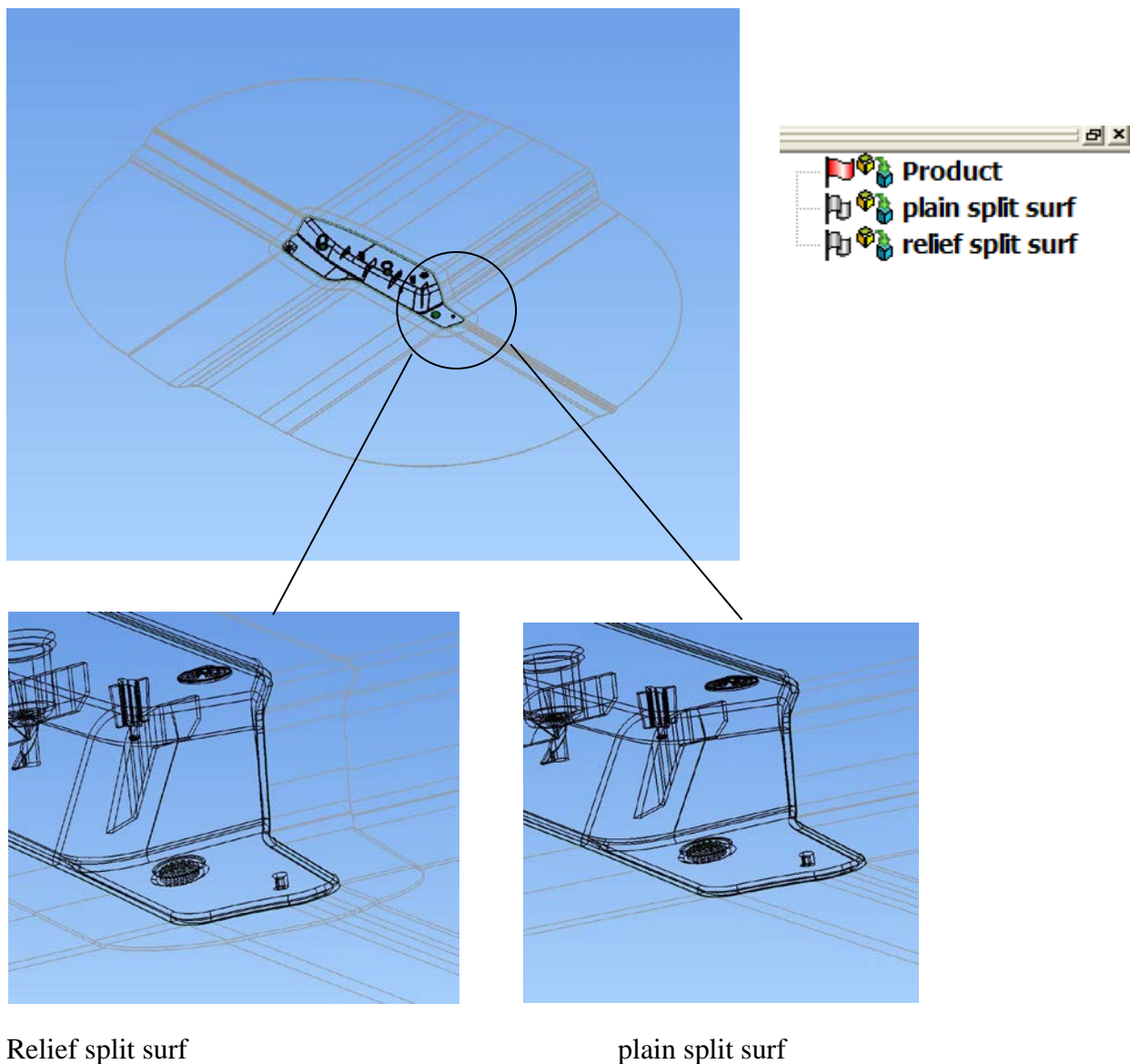
It is often the case that split surfaces are manually created to allow for die relief / die locks or slide cores. When this is done the split surfaces may be very different on the core to the cavity.

Die wizard will now allow you to supply these surfaces.

The split surfaces must be supplied as SOLIDS.

- Open the model – User_Defined_shut_outs.psmodel

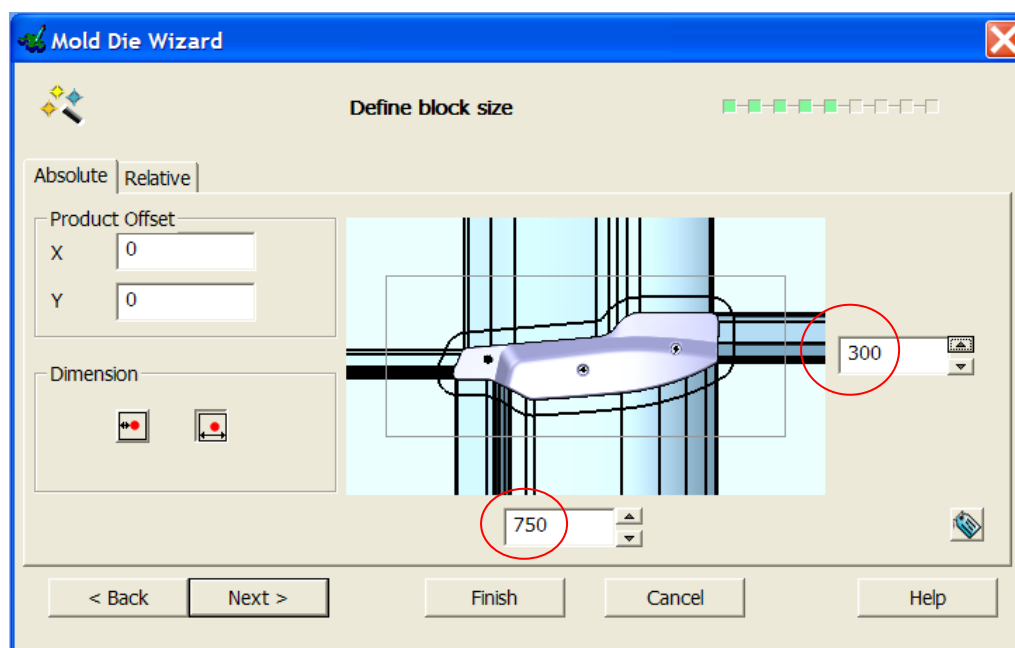
The model contains three solid, the product and the two different split surfaces. View the model with each split surface solid blanked to see the difference



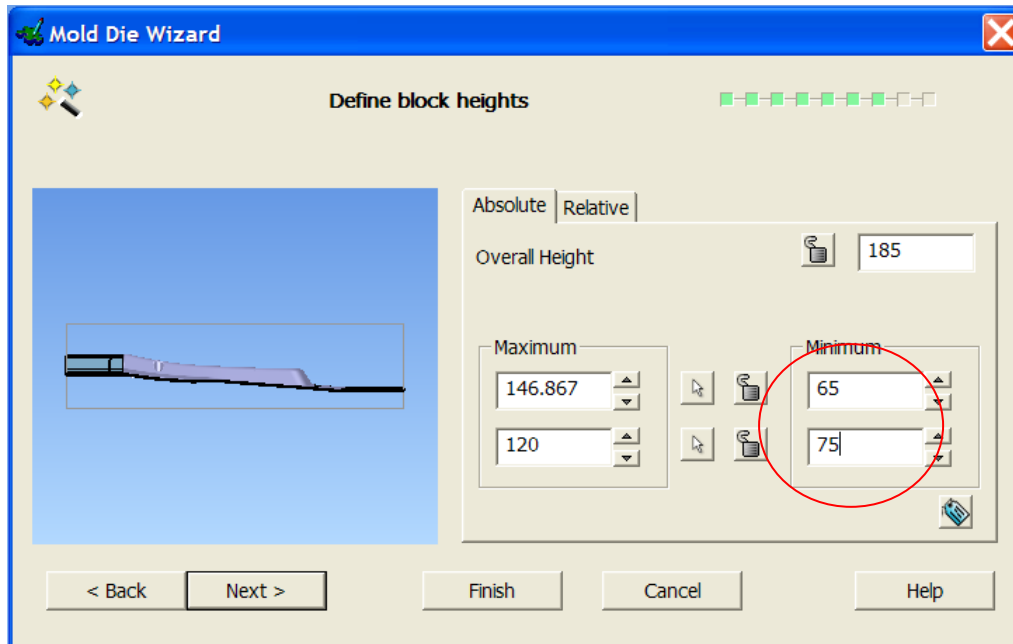
- Start **Die Wizard**,



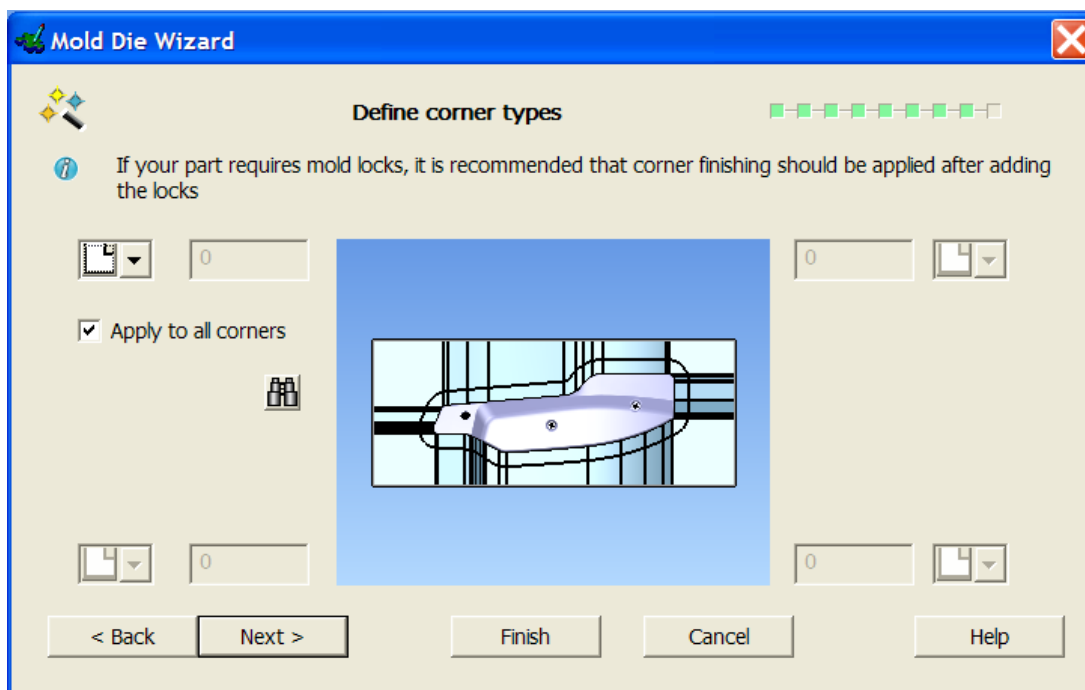
As everything is supplied you can press **Next** until the Define block size page.



- Set the block sizes to **750mm x 300mm**. Hit **Next**

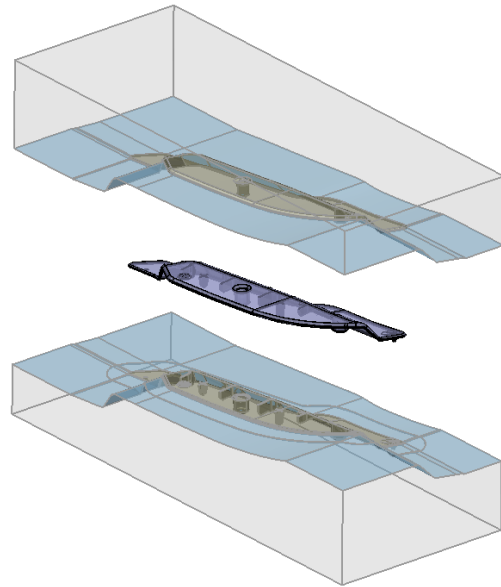
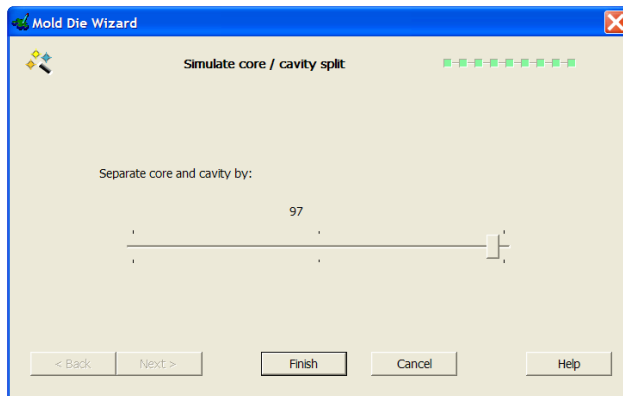


- Change the **minimum Heights** to **65mm** and **75mm**. press **Next**



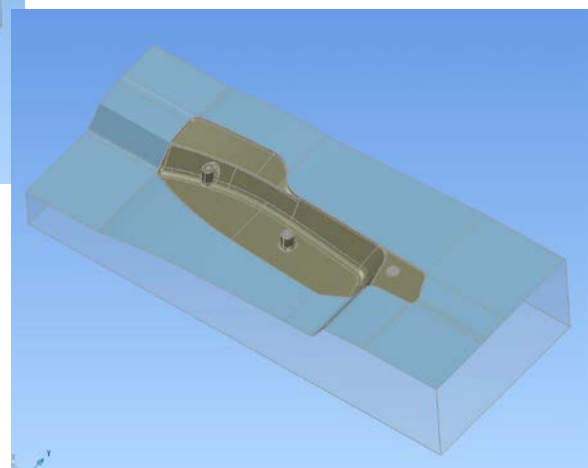
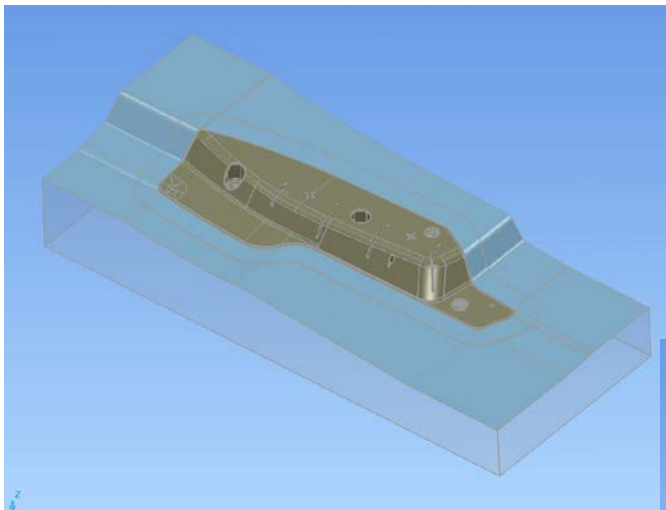
- Set all corners to **square**. press **Next**

- **Preview the results**



- **Press **Finish**.**

The core and cavity blocks can now be viewed to see the different split surfaces:-



Tip

When using die wizard to create core and cavities that will later have Mold locks added, it is advisable to leave square corners on the blocks.

If you add corner radii the Mold lock wizard will try to match the radius, if you add chamfers the Mold lock wizard will fail. Due to the automatic extension used to intersect the Mold lock with the block.

The warning displayed in Die Wizard:-

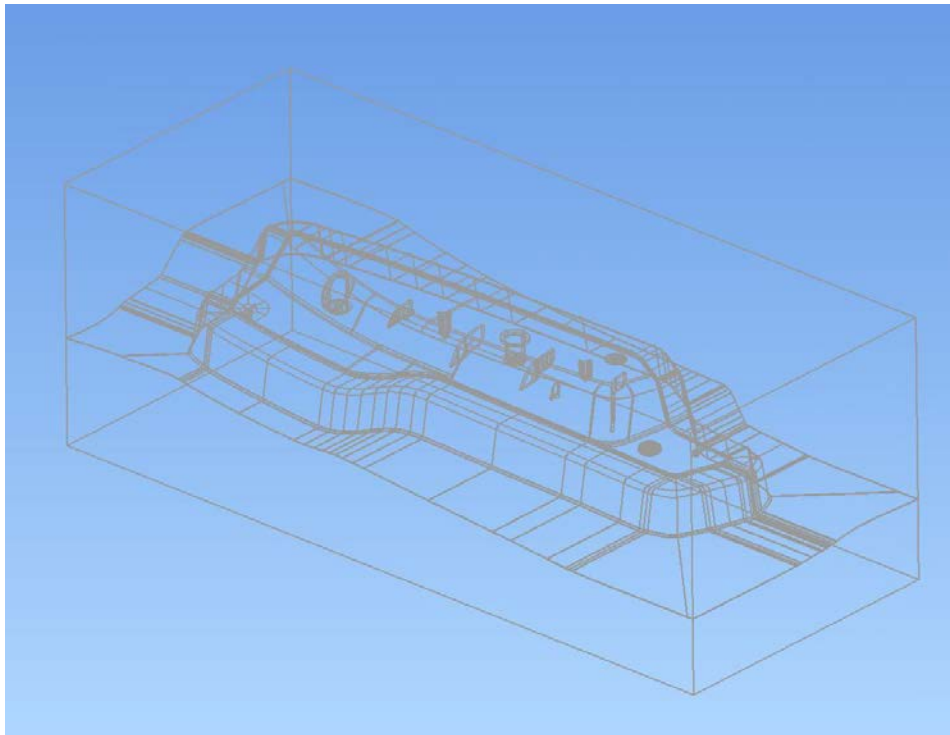


If your part requires mold locks, it is recommended that corner finishing should be applied after adding the locks

4. Mold Locks

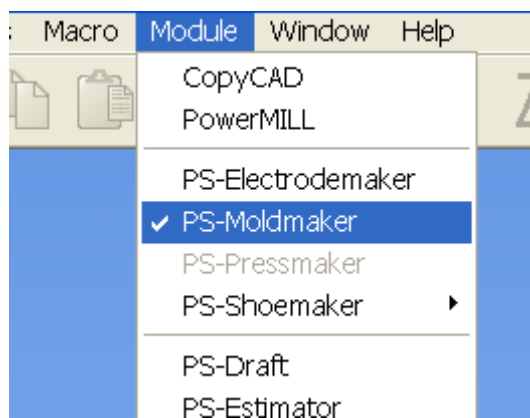
A new Mold lock Wizard features in version 6.0. The following exercise uses one type of lock configuration, there are several options.

Open the model – Mold_Locks.psmodel

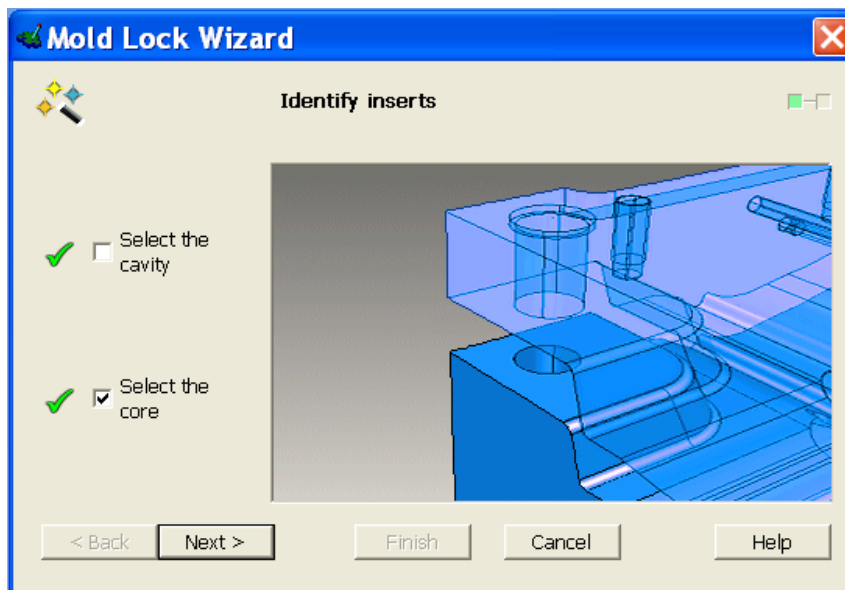


The Mold lock wizard works with both components and solids

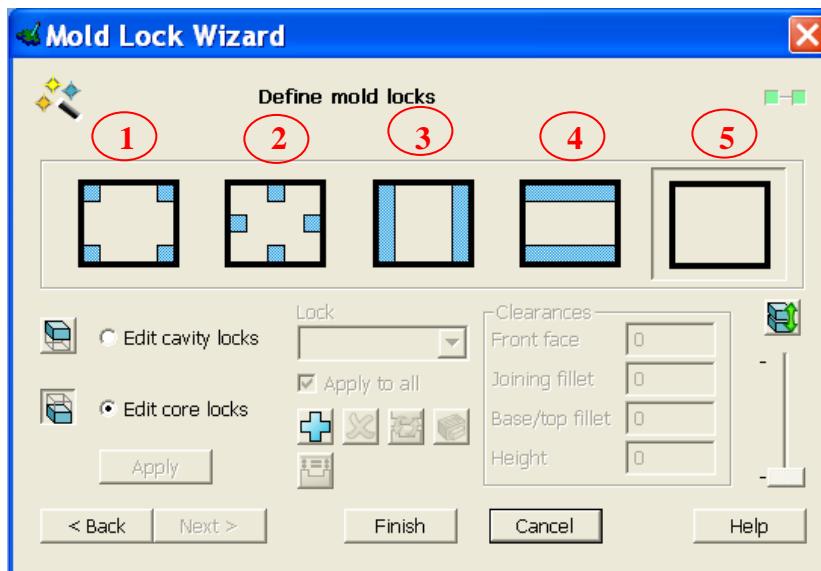
- Start **PS-Moldmaker** and select **Mold lock wizard** from the drop down menu



The first page will prompt you to identify the core and cavity



- **Select** each component and press **Next**.

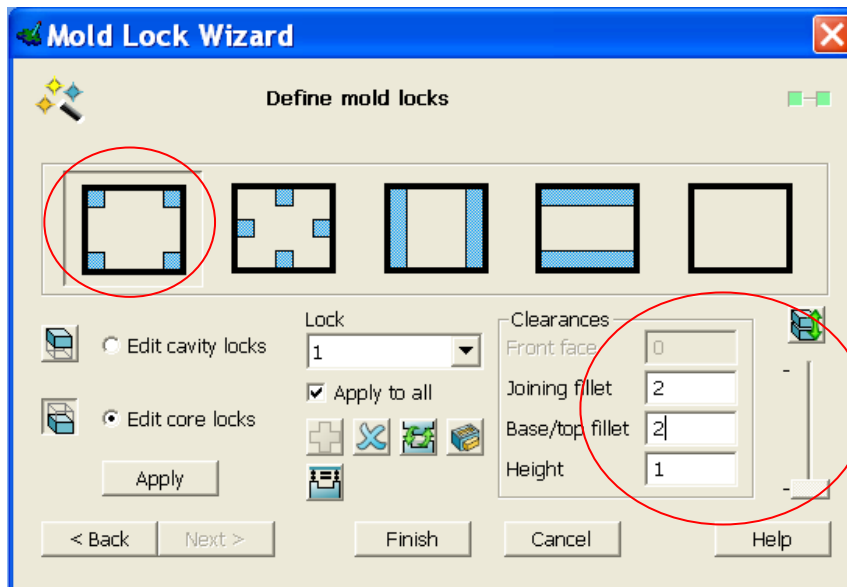


The main page of the wizard contains all of the options for creation of various types of Mold locks and clearance settings between core and cavity.

There are five options to create locks:-

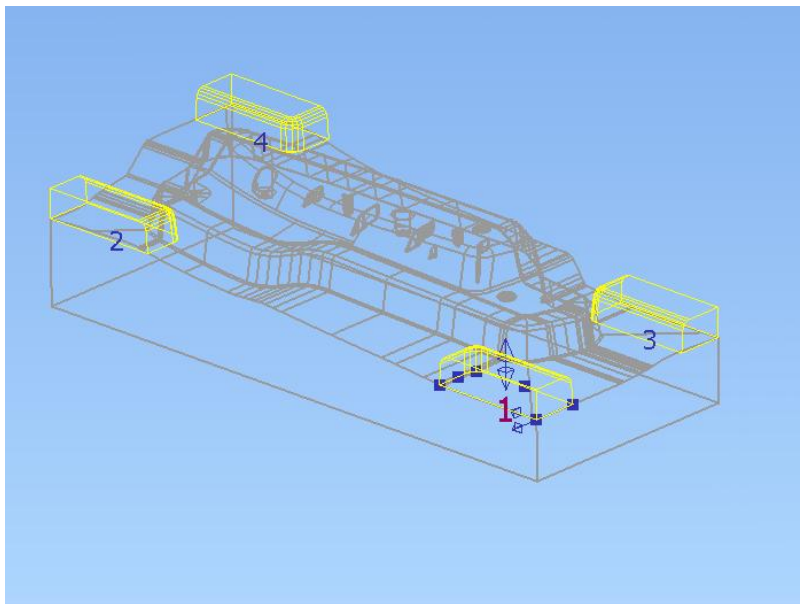
- 1) Locks in each corner of the block
- 2) Locks in the centre of each side of the block
- 3) Locks the full width of the block
- 4) Locks the full length of the block
- 5) User defined locks

- For this exercise **select the first option** to add **locks in each corner** of the block.

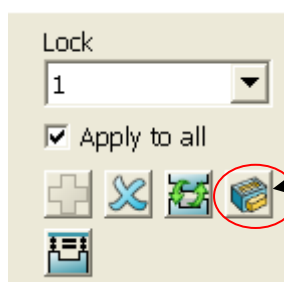


- Change the values

Once a creation option is selected the default locks are displayed on the blocks

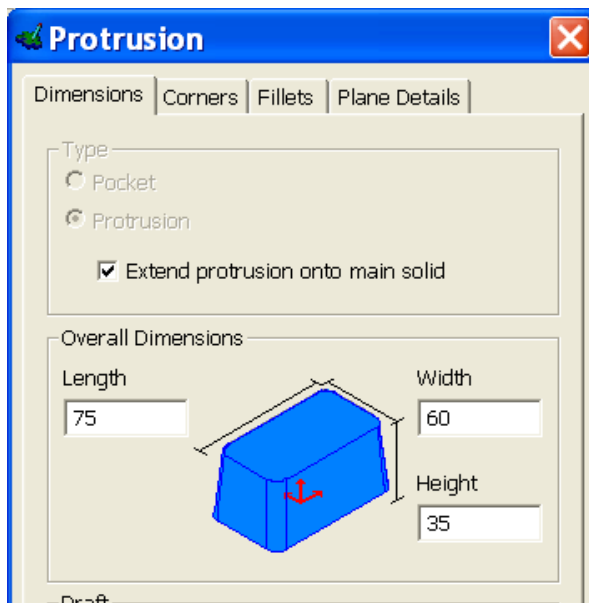


The default sizes can be modified, either as a group (apply to all) or individually

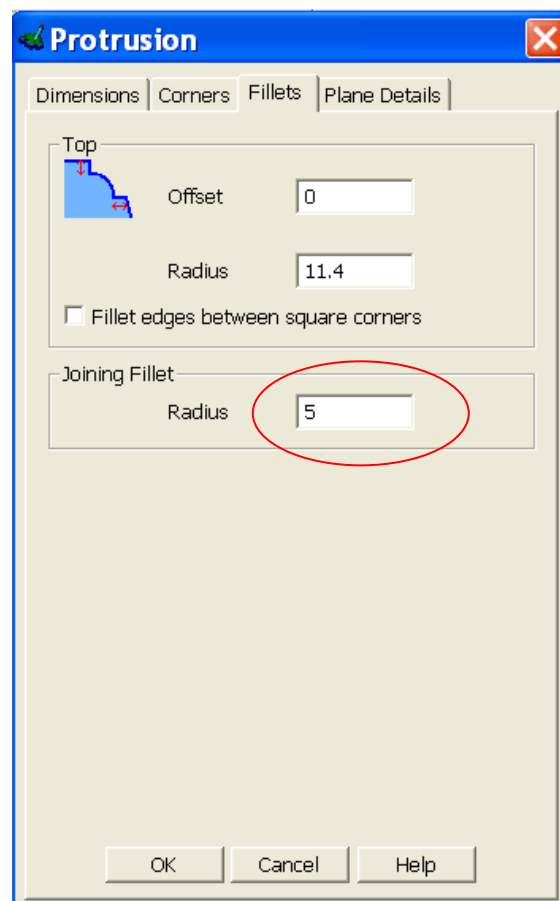


- Modify** protrusion / pocket dimensions

The dimensions for the male lock are displayed as the '**Edit core locks**' option is checked on the main form.



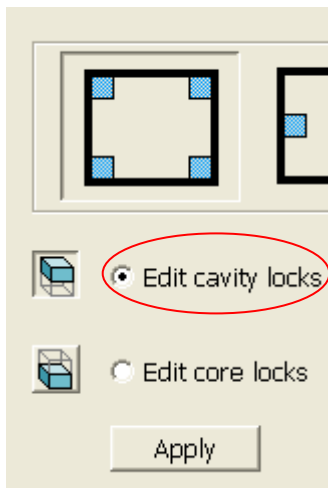
- Change the **Length / Width and Height**



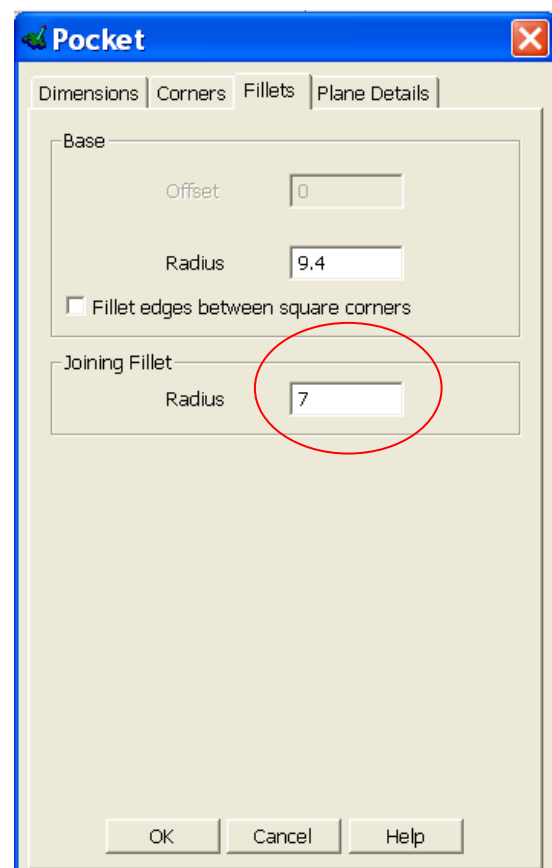
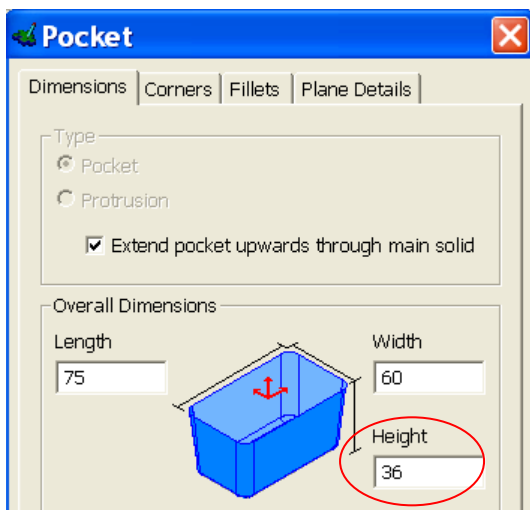
- Click on the **Fillets** tab to set the value

- **OK** the form.

- Select the option to **Edit cavity locks**.

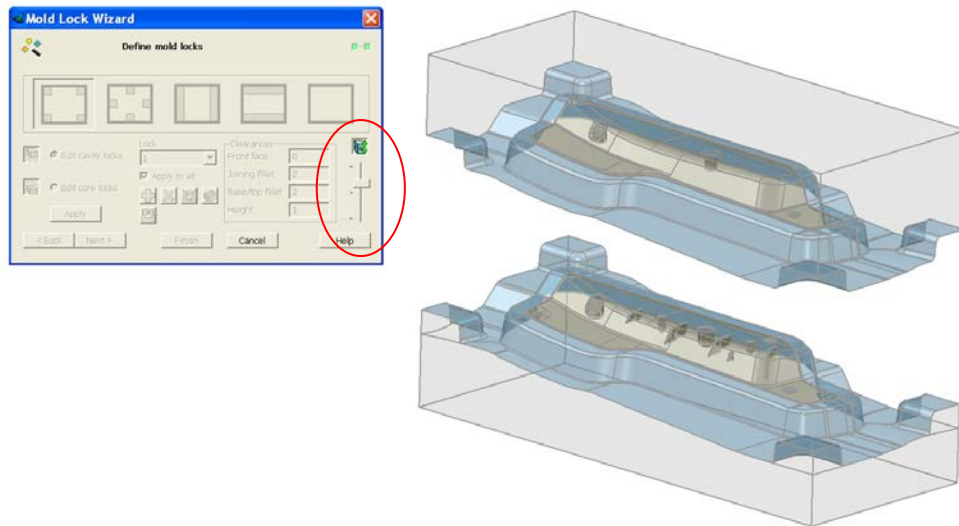


Although you will not change any values, the effect of the clearances can be seen.



- **OK** the forms

- The Mold lock wizard also has a **preview**, switch on any try it out.



- When all of the values are correct, either press '**Apply**' to create the locks and keep the form open for further editing or '**Finish**' to create and close the form.

Tip

There are two 'toggle' buttons on the main form, the button to set all Mold lock heights to the same Z value cannot be toggled back. If you hit this button and don't like the results then you must restart the wizard

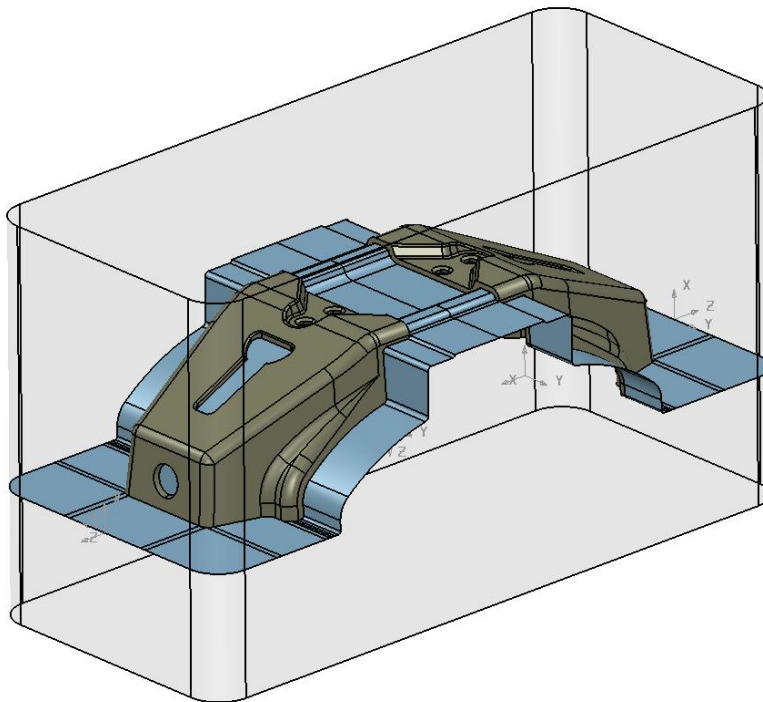


5. Slide Core

The Slide core Wizard now has the ability to modify existing slide cores by re running the Wizard.

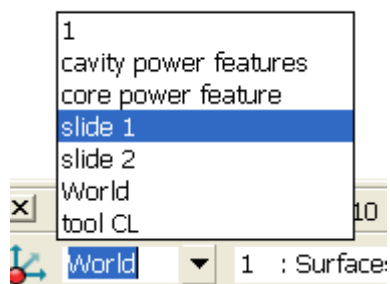
The slide core also created a feature in the core / cavity inserts where applicable that can be deleted if the core is no longer required.

Open Model – Slide Core.psmodel

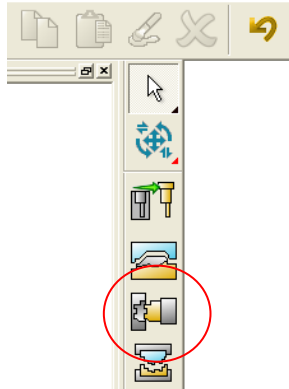


There are pre created Die blocks and Workplanes in the model.

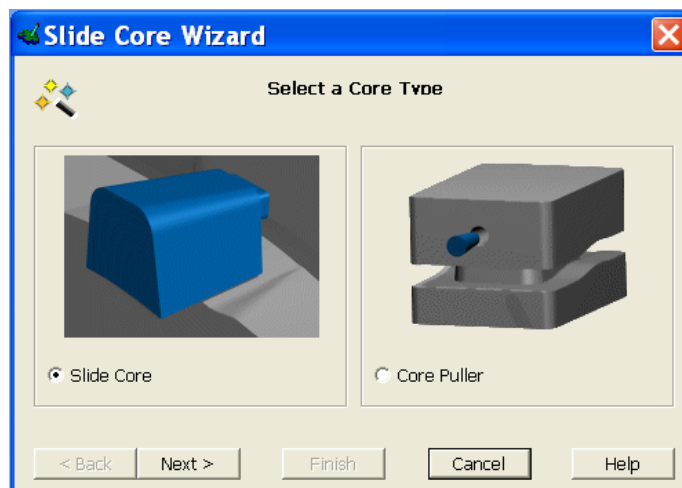
- Activate the Workplane – slide 1



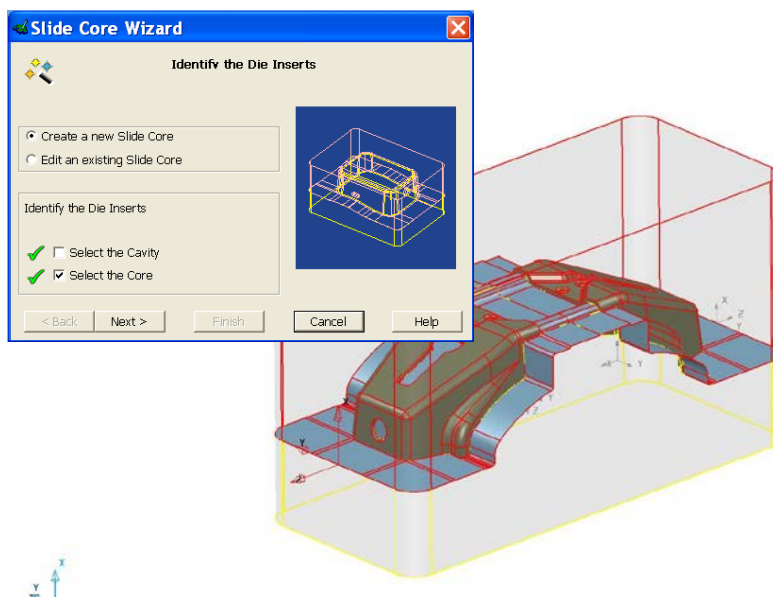
- Start the **Slide Core Wizard** from the Wizards **drop down menu**.



- Select the Option **Slide Core** then **Next**

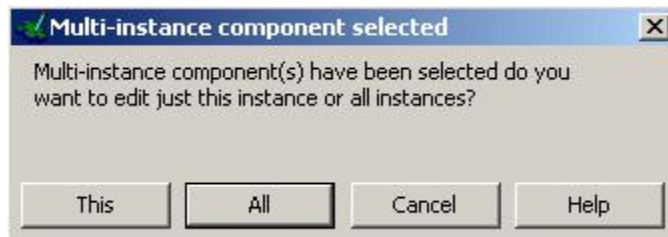


- The second page of the wizard automatically detects the **core and cavity inserts**

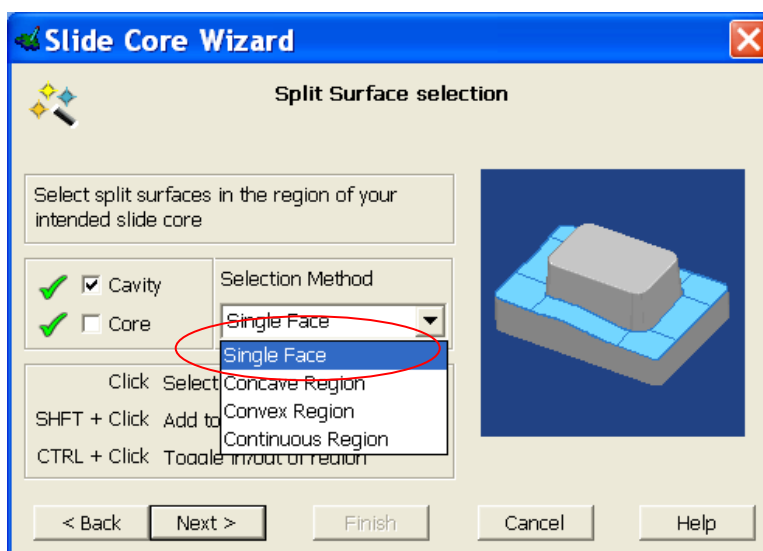


As all assembly components except plates and trimmed items are multiple instance, the wizard asks which edit option you want to perform.

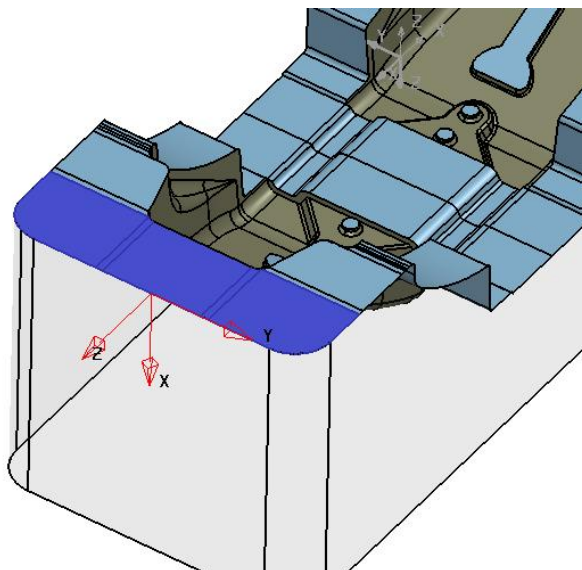
- Select **ALL** instances



- Next

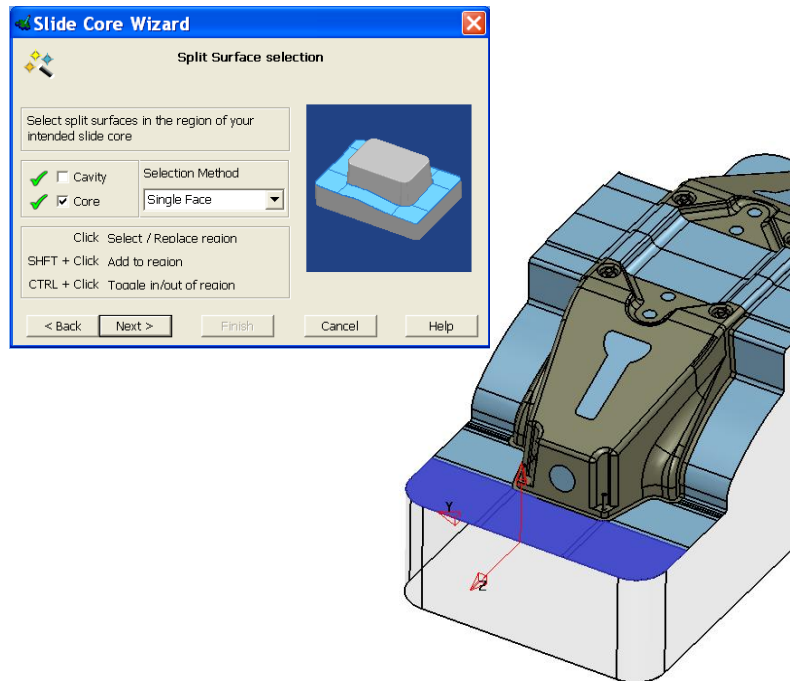


- Use **Single face** selection method

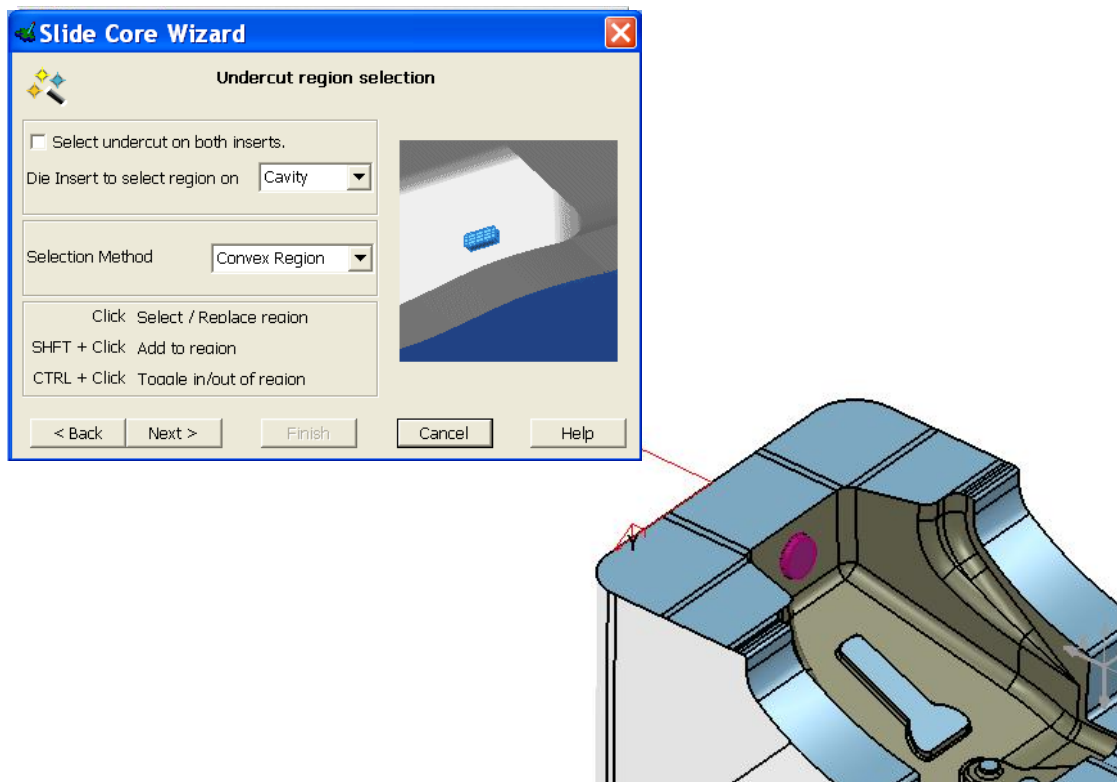


- **Select** the first face then **shift-select** the other two to identify the region adjacent to the slide core

- Repeat the **selection** for the **Core insert**

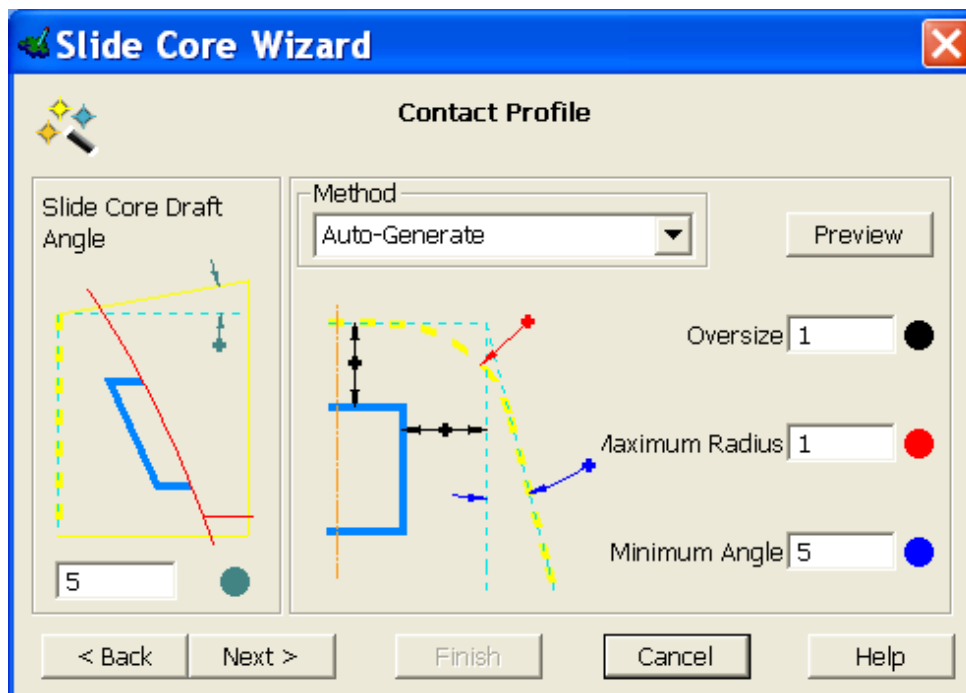


- Pick the **Undercut region** on the **Cavity insert**.

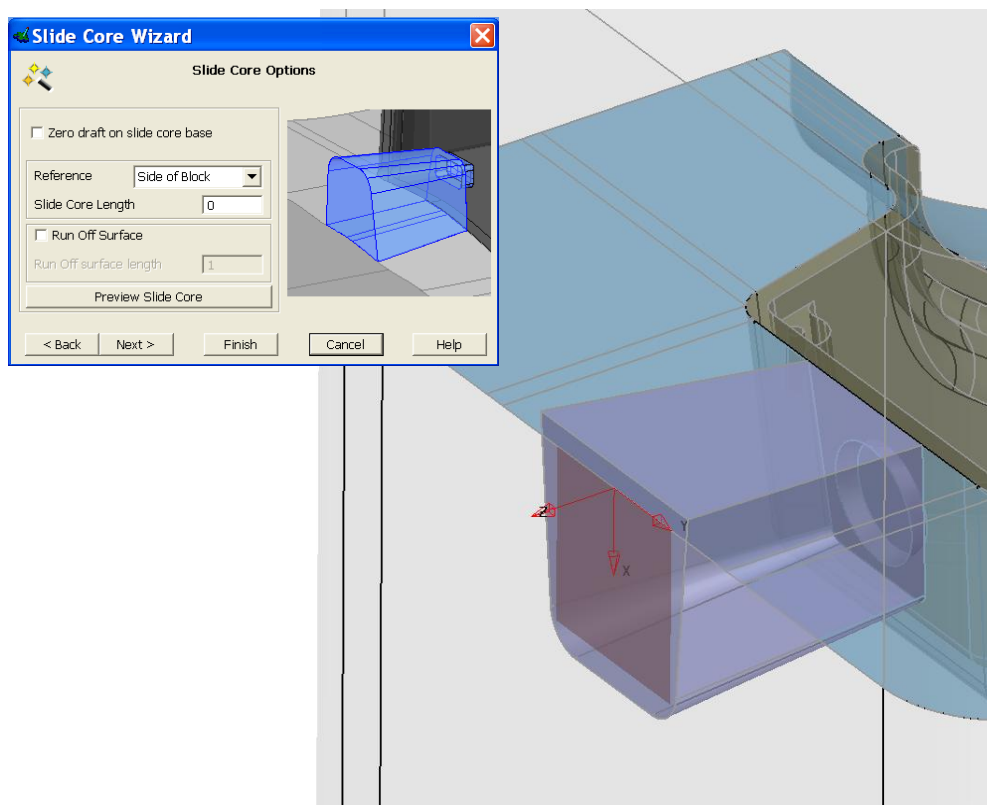


- **Next**

- Check that the Slide core draft angle is set to 5 degrees, leave all other default values

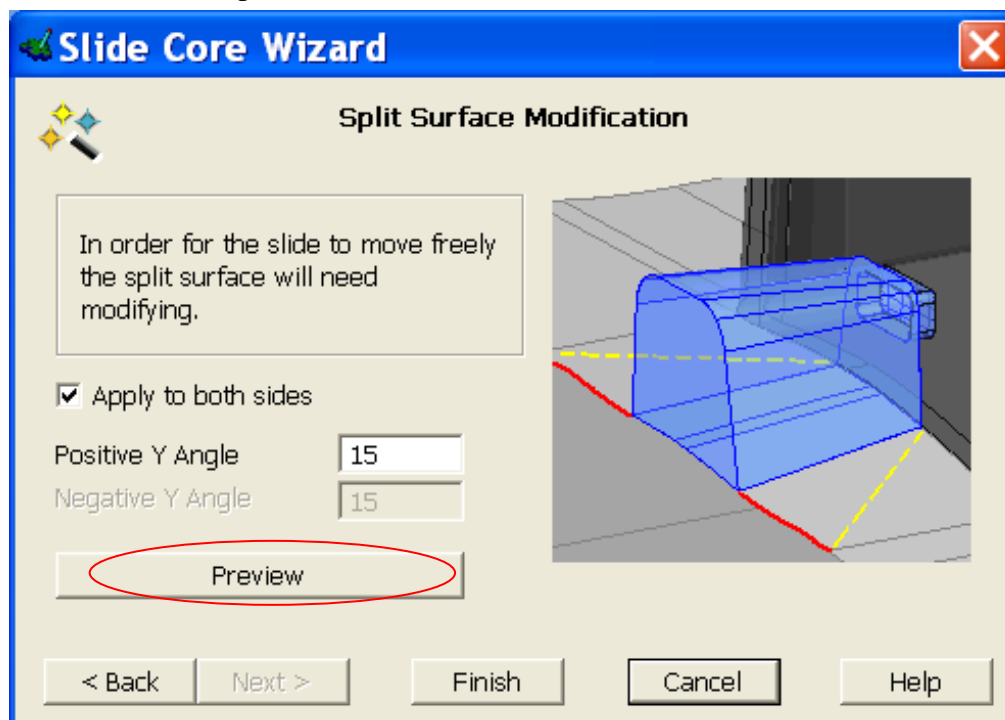


- Next
- Set the **Reference** to side of block, length zero

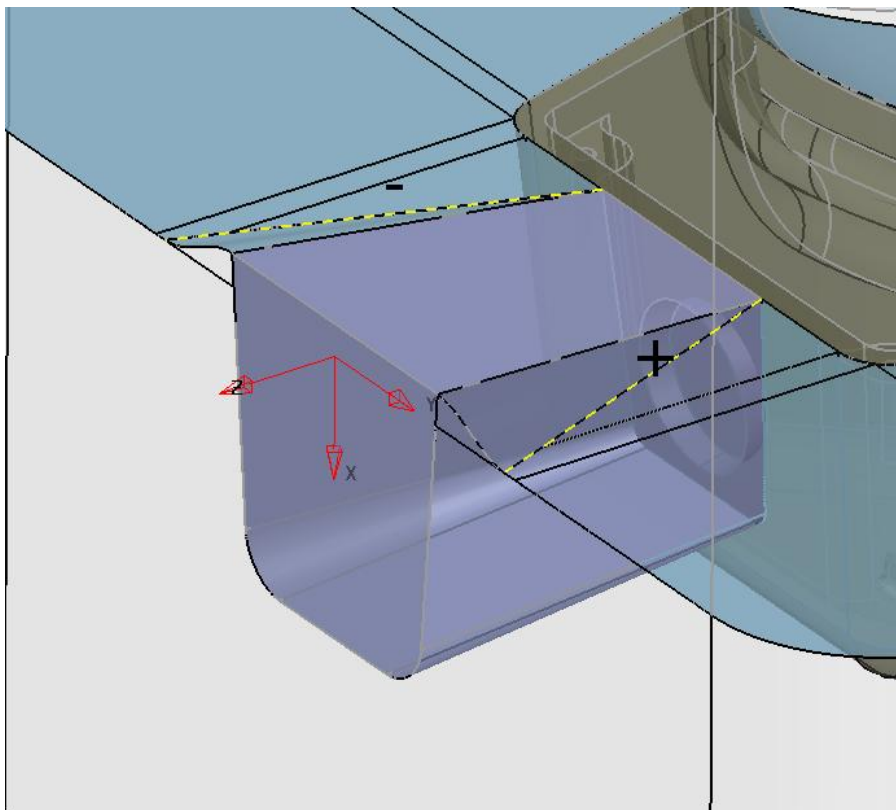


- Next

- **Preview** the split surface Modification



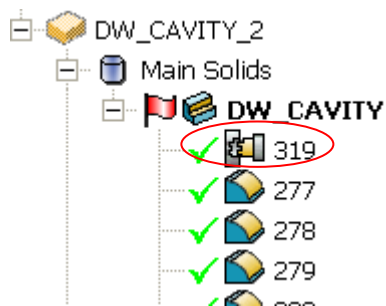
The changes that will be made to the split surfaces are shown



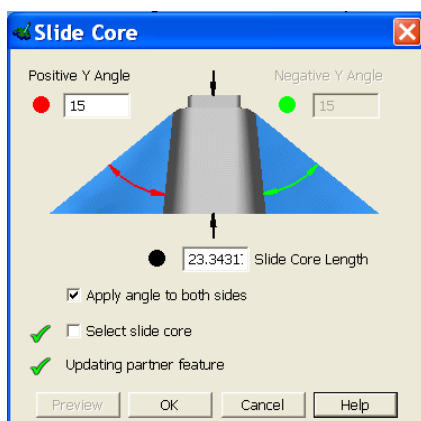
- **Finish** to create the Slide core and slide core features in the core and cavity inserts.

If you **Modify the Geometry** of the Cavity insert you can see the **Slide Core Feature**

- Turn On PS-Assembly



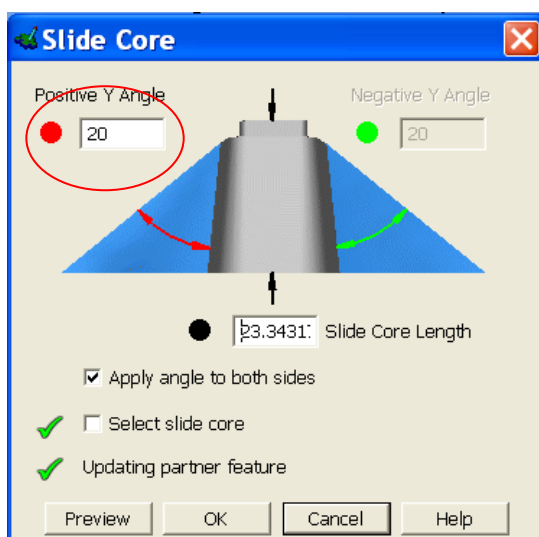
- Modify the Slide Core Feature



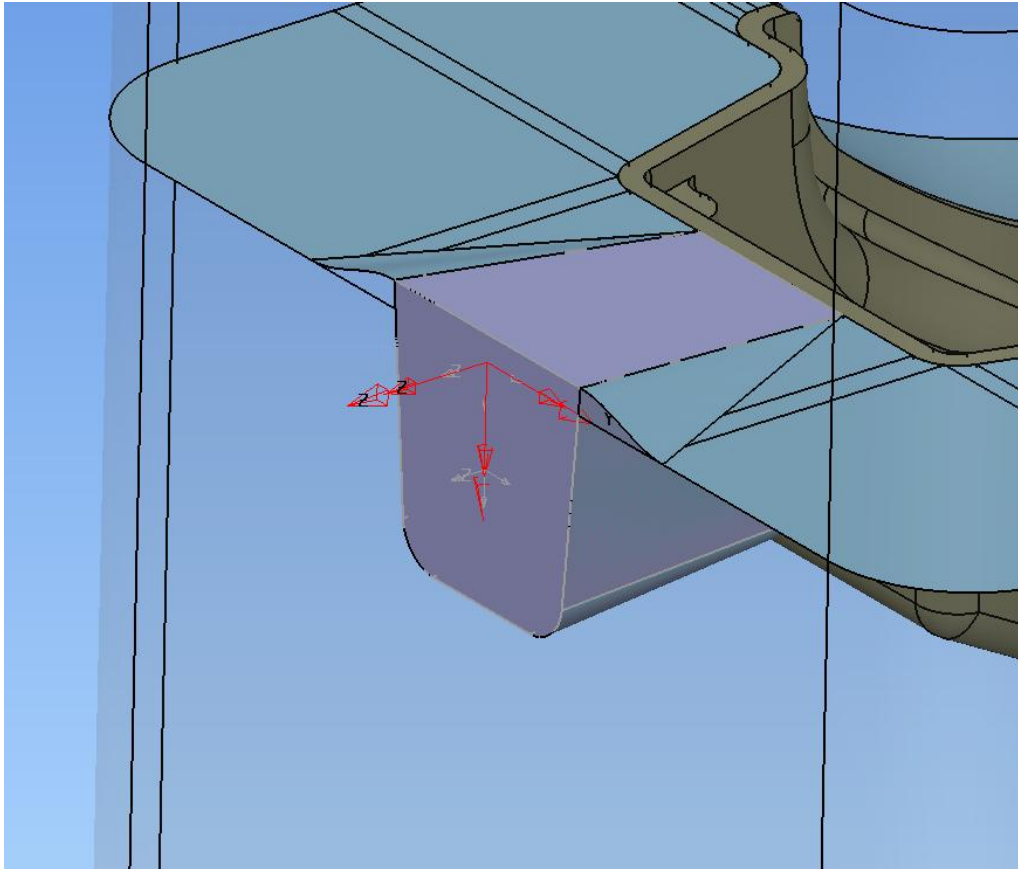
The available options when modifying the slide core feature are:-

Change either both angles together or individually
Change the length of the slide core

- Modify the angle to 20 degrees



- **Preview** then **OK** to change the feature

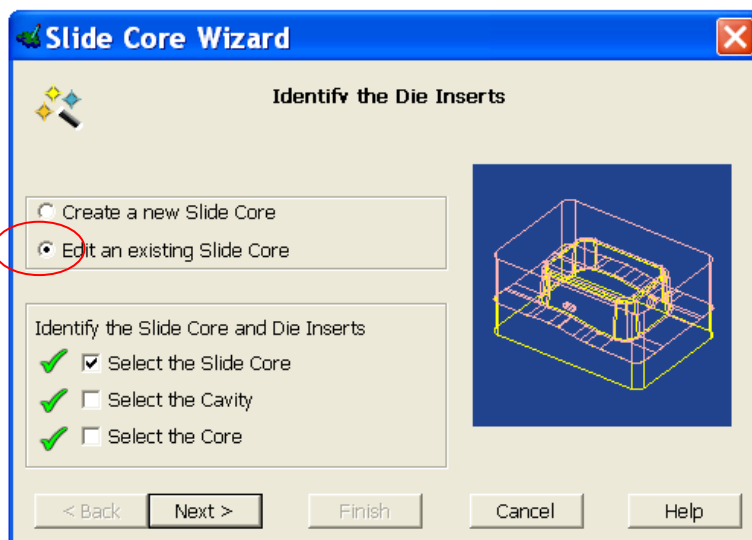


Modifying the Slide Core

To make changes to the Slide core you can now re enter the wizard.

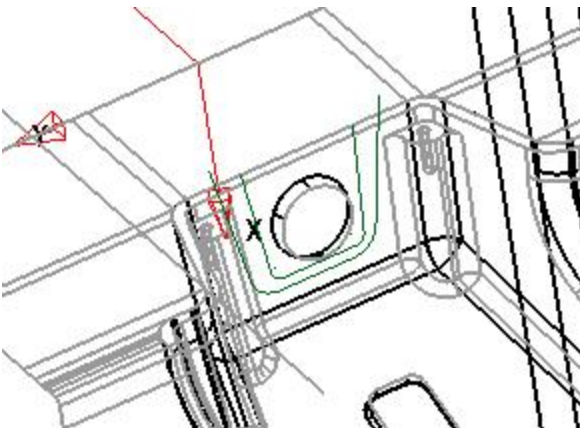


- Check the **Edit an existing Slide core** option and select the slide core solid



- **Turn on Level 11**

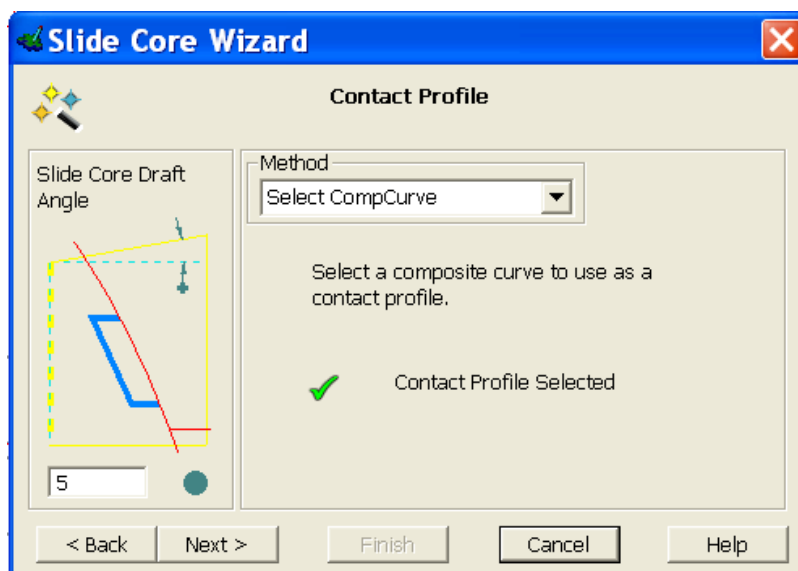
The Slide wizard has left a curve on the model from the Auto generated profile; to change the profile the larger comp curve will be used



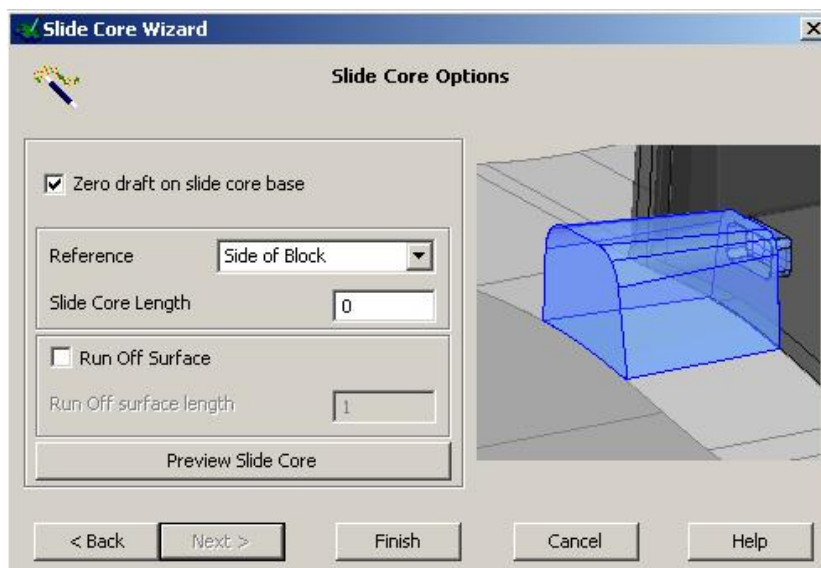
The wizard remembers the split faces selected previously so hit next until the undercut selection page.

As you are changing the contact profile using a supplied curve the undercut does not need to be selected, so hit Next on this page.

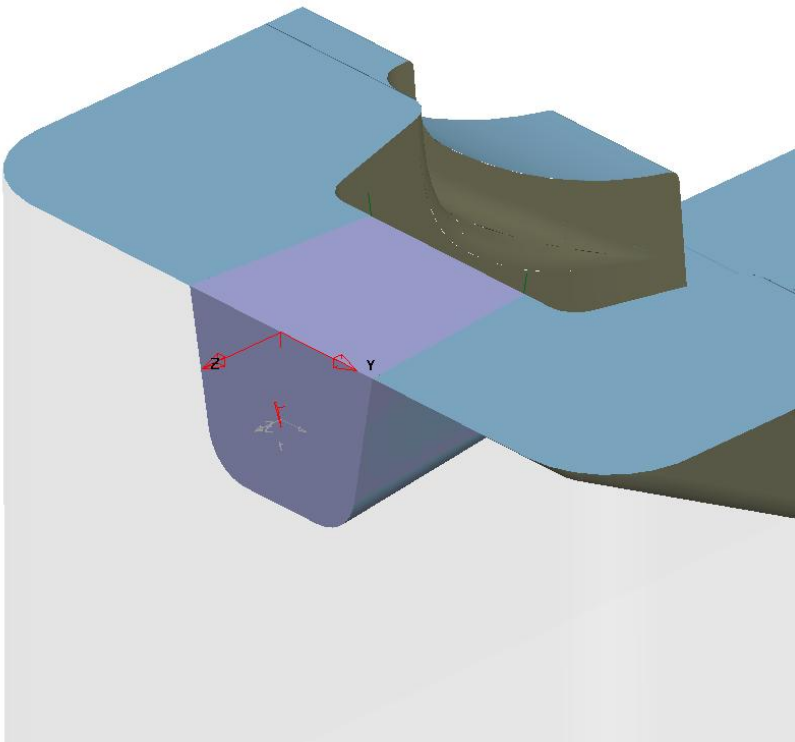
- Pick the **comp curve** on **level 11**



- Tick the option for **Zero draft on slide core base**



- **Preview** the Slide Core
- **Finish** to create the Modified core and split surfaces.



Repeat the above steps to create the Slide core on the opposite end of the inserts, the workplane required is **slide 2**

6. Defining Power Features.

PS-Assembly 6.0 incorporates some fundamental changes to the usage and definition of Power Features (formally named Tool Solids).

This can be broken down into the following areas:-

- Creation of solid features
- Setting the Advanced 'Type' of the feature
- Methods of generating Power Features
- Updating features after modifications

The following three chapters cover the different aspects of the changes using basic scenarios to describe the steps required and some rules that should be applied when using Power features.

The following terms will be used throughout the training course when referring to Power Features:-

Source - this describes a component containing solid features that will be copied.

Target – this describes a component that can accept the copied features.

Tool Solids

In earlier versions power features were known as tool solids, they allowed solid features to be included with a component and copied to any intersecting components.

The copying of the features was governed by where the source features intersected with any target components. This resulted in the features requiring modifications to suit individual applications.

The rule was – copy All features to All intersecting components.

Power Features

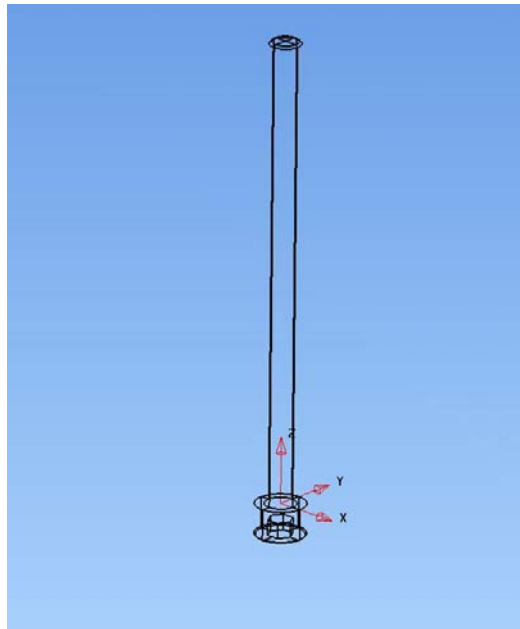
From this point on the flexibility for the copying of Power Features has been significantly improved. There are a new set of 'type' instructions that allow each source feature to be set to copy to specific intersecting Target components.

In the following exercise you will create features and set the rules for their application.

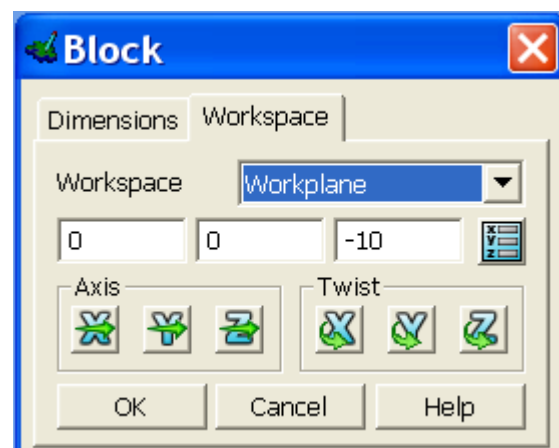
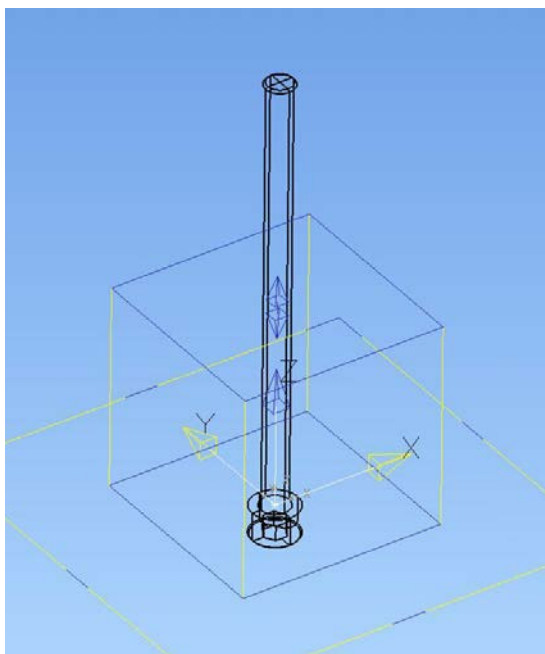
- **Open Model** - cap screw.psmodel

On level 20 there is a cap screw solid, this is a parametised part. The parameters contained within the screw will be used to drive the Power Features also.

- Select the **Work plane** on the screw and **activate**. The power features must be created relative to this work plane.



- Now create a **primitive block** with the plane centre on **0 0 -10**



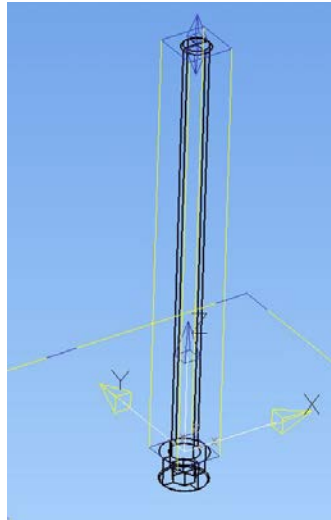
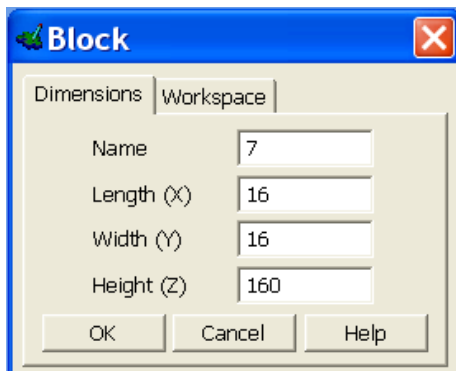
- **Edit** the block and **add parameters** from the screw.
- Use **RMB** in each **field** to raise the **Calculator** form, Select the **Parameter Tab** and add parameters as follows:-

Length = dk

Width = dk

Height = l+k

You should have these new values.

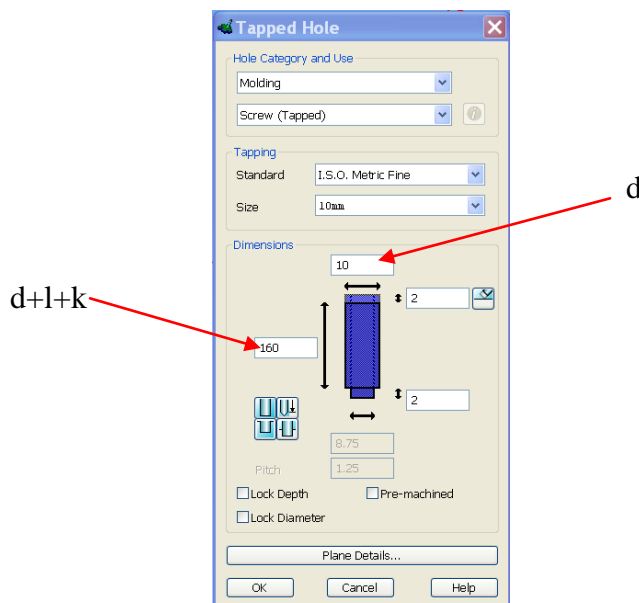


The primitive block will now adjust size to suit the cap screw size.

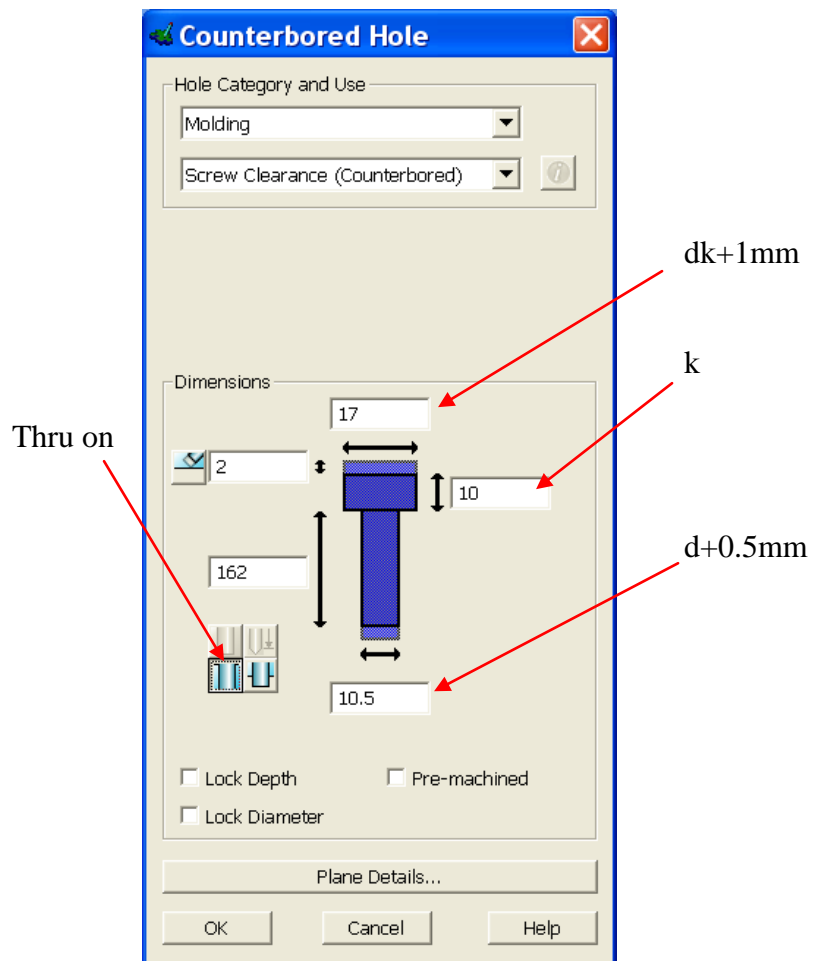
- Make the solid block **Active**

Now you can define the features needed for the screw, there are three holes. All are positioned at **0 0 -10**:-

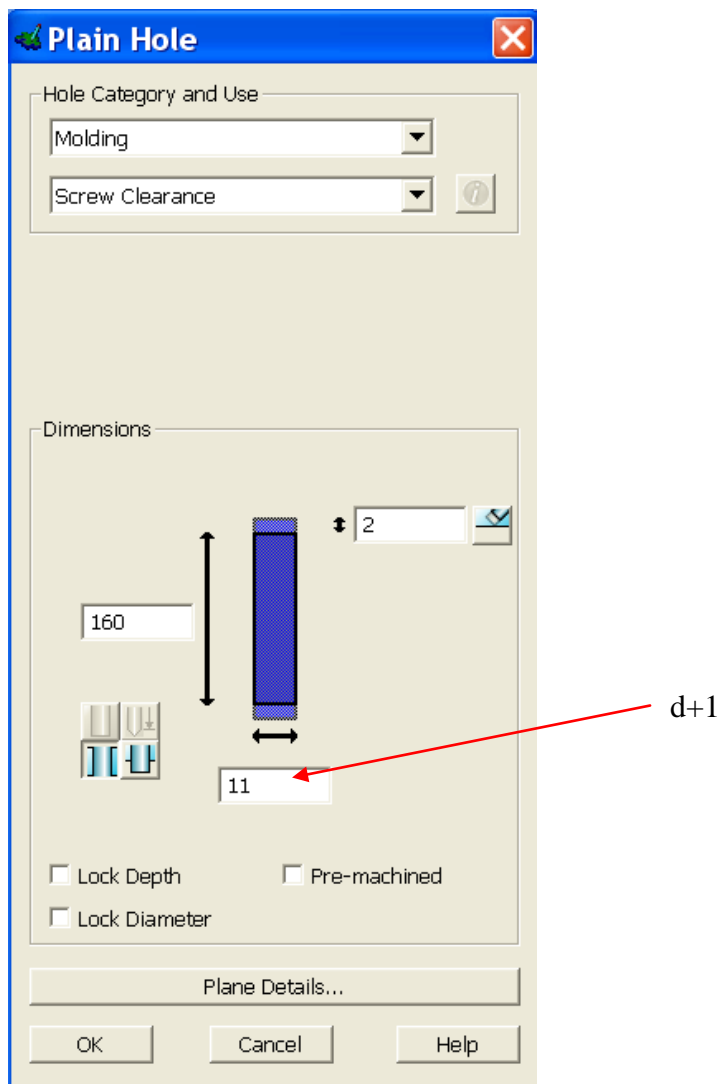
Tapped hole – change the values indicated to the **parameters** shown to match the screw



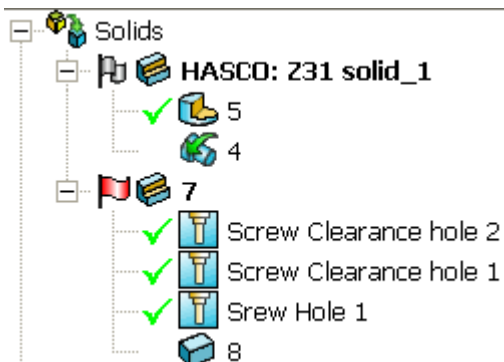
Counter Bored Hole – change the values to parameters where indicated and make the hole ‘Thru’



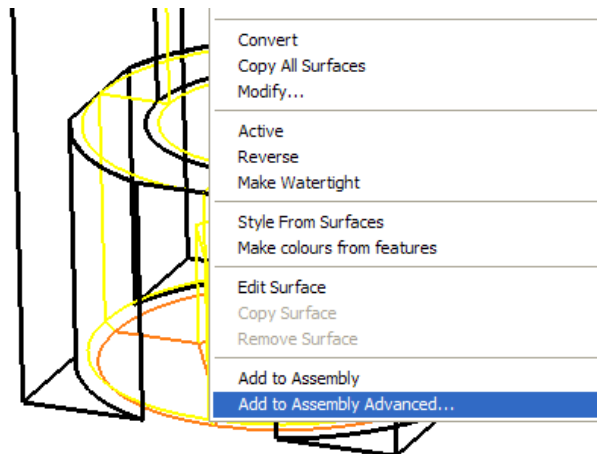
Clearance Hole



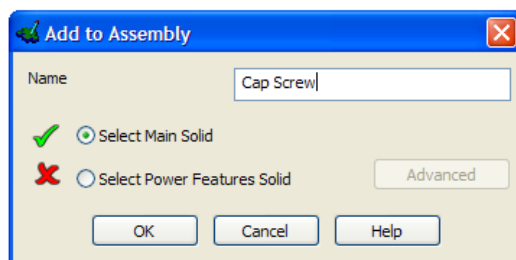
The solid history tree should now show two solids.



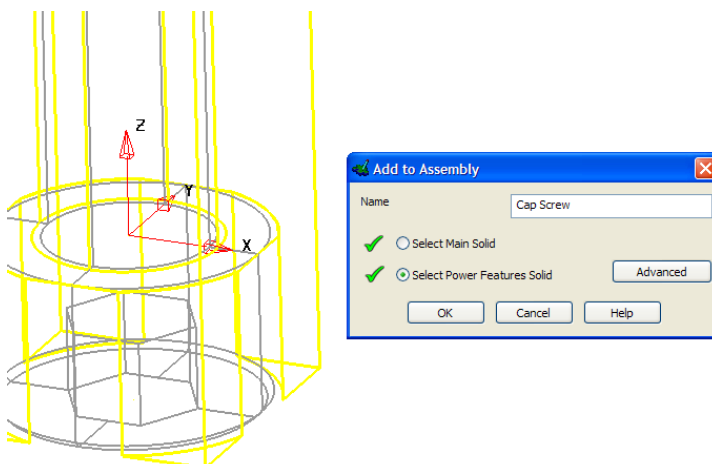
- **RMB** menu on the solid and pick **Add to Assembly Advanced...**



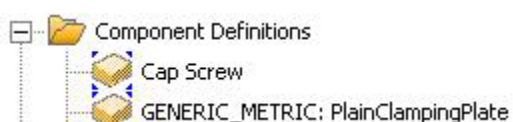
- Change the name to **Cap Screw**



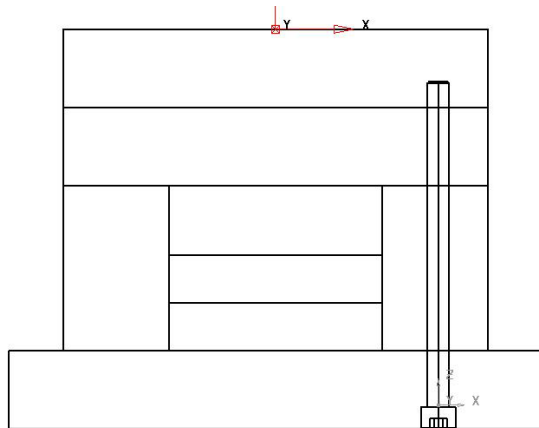
- Now select the **Power Feature Solid**



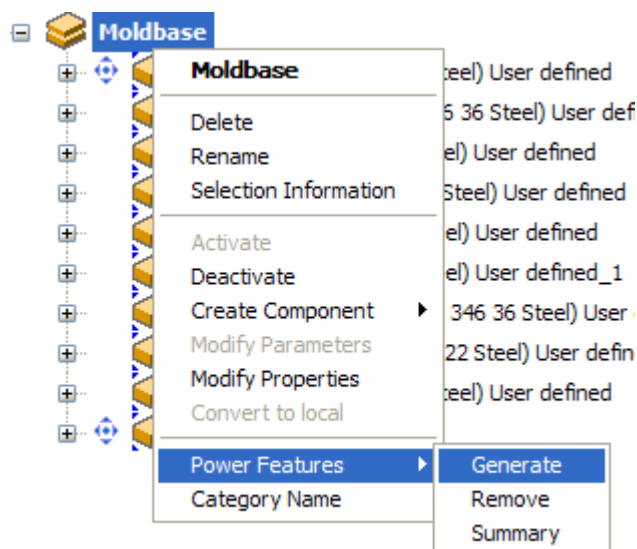
- **OK** the form, a component is created.



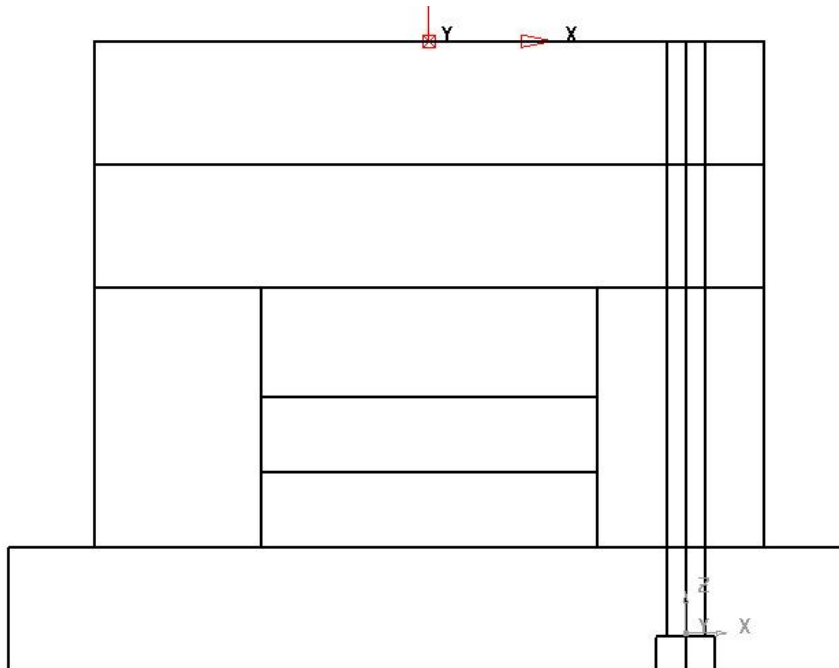
- Turn on the **Moving half level group**, activate **DW_WP**



- From the assembly tree RMB on **Moldbase** and **Generate Power Features**.

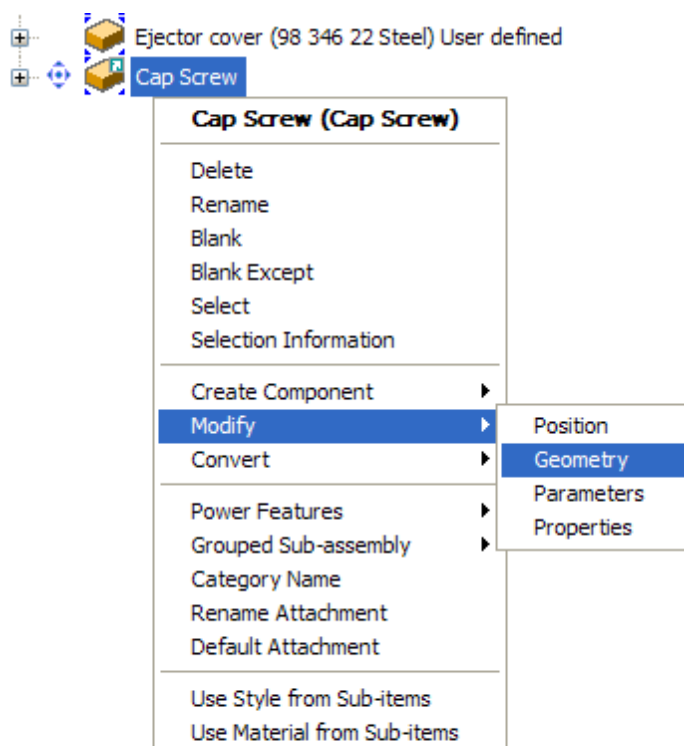


The results can be seen and it is clear that they are incorrect.

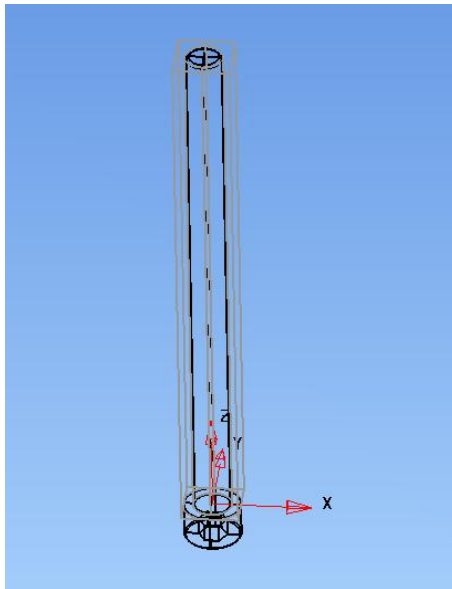


To correct the Power Features we must use the Advanced Option.

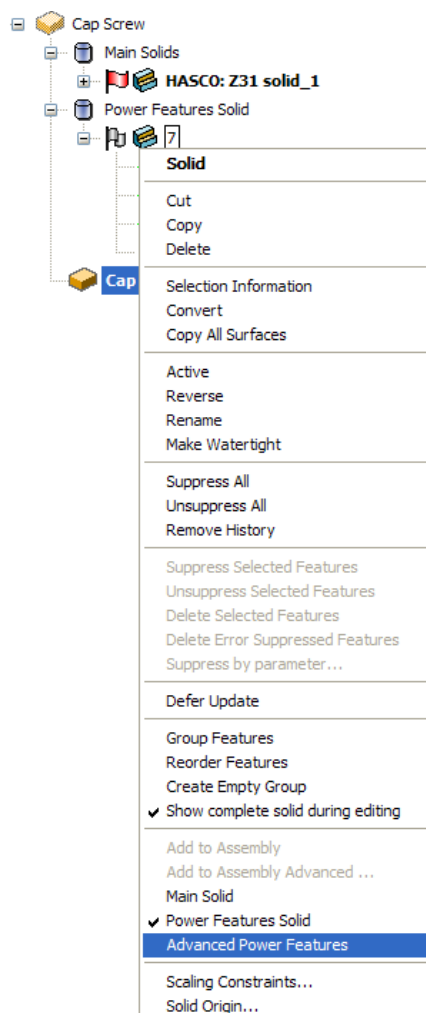
- First select the **component** and **modify the geometry**



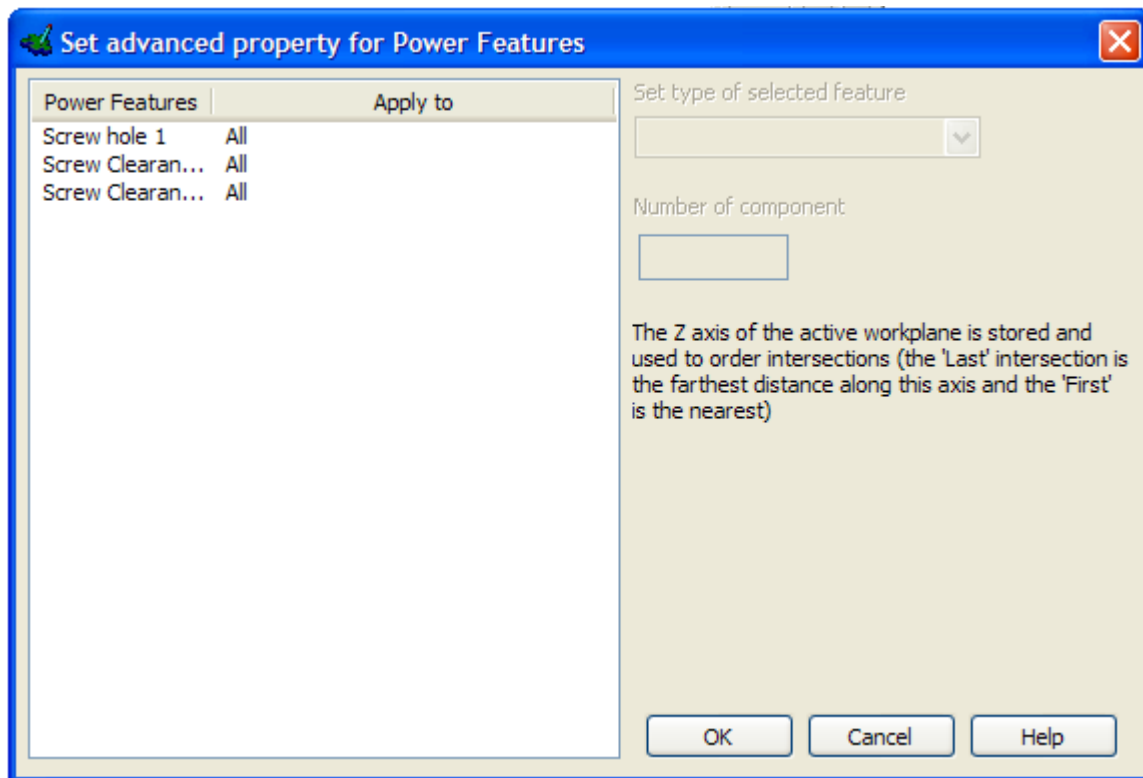
- **Activate the Work Plane** on the screw geometry.



- RMB on the power feature solid, select **Advance Power features**



The Set Advanced Power Features form is raised



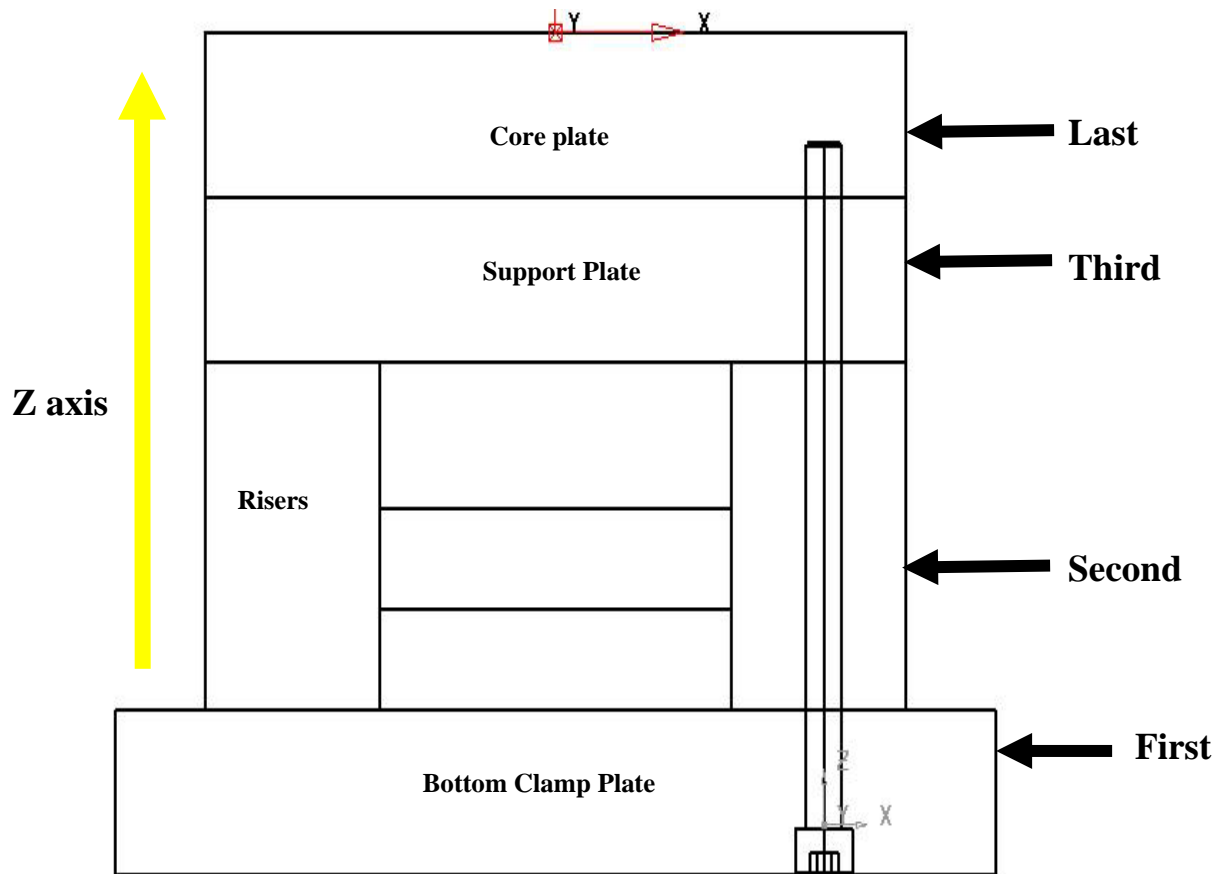
As you can see the features 'Apply to' is set to ALL. This means that any feature on the screw (Source) can be copied to any component that it intersects with (Target).

There are various options that you can set, they are:-

- First
- First and Second
- All but First
- Last
- Last and second Last
- All but last
- Number
- All but number
- All but first and last

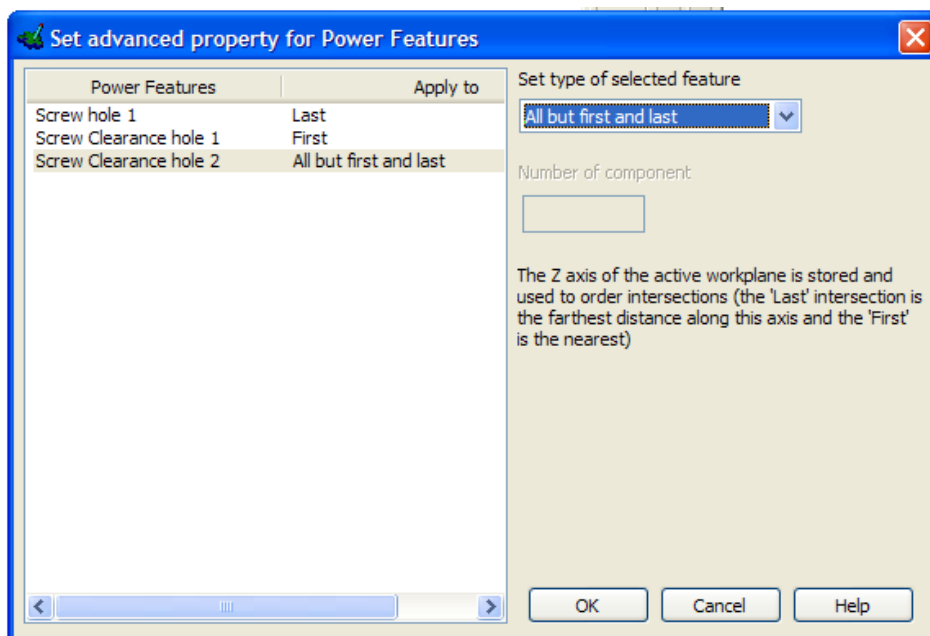
These options are based on the Z axis direction of the active Work plane when the form is accepted. This is an extremely important rule.

The following picture indicates how the options required are determined, you can see from the Z axis in relation to the mould plates that the Bottom clamp plate is the first, Risers are second, Support plate is third and Core plate is last

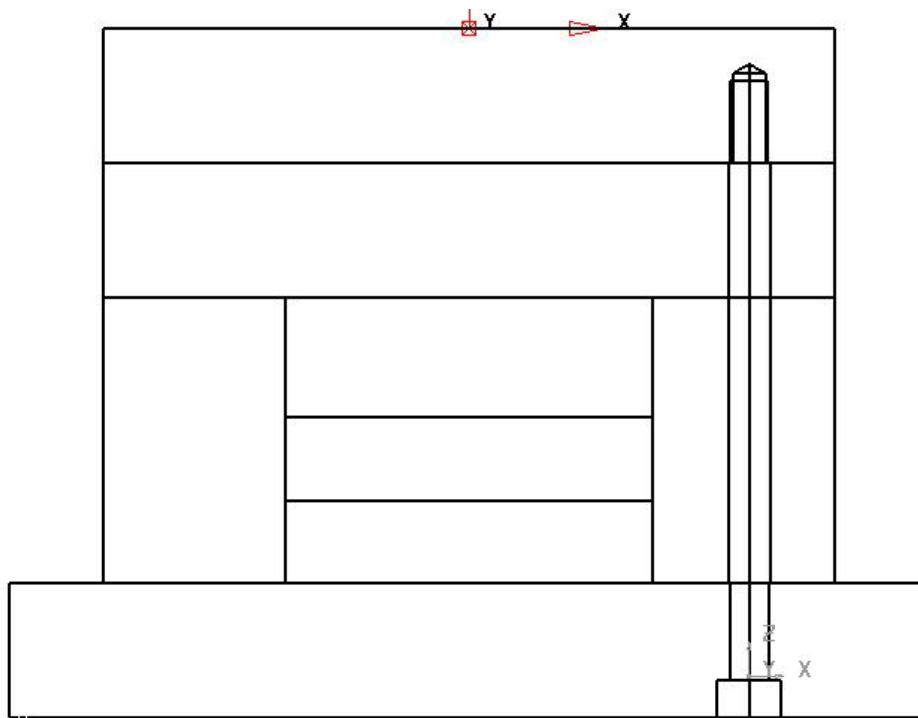


Based on the above intersections the Set Advanced Power Features options are changed to the correct 'type'

- **Modify** the settings by **picking each feature** and selecting the desired **option** from the **drop down list**.



- **OK** the form, **accept** the changes and **generate Power Features again**.



The holes are correctly generated.

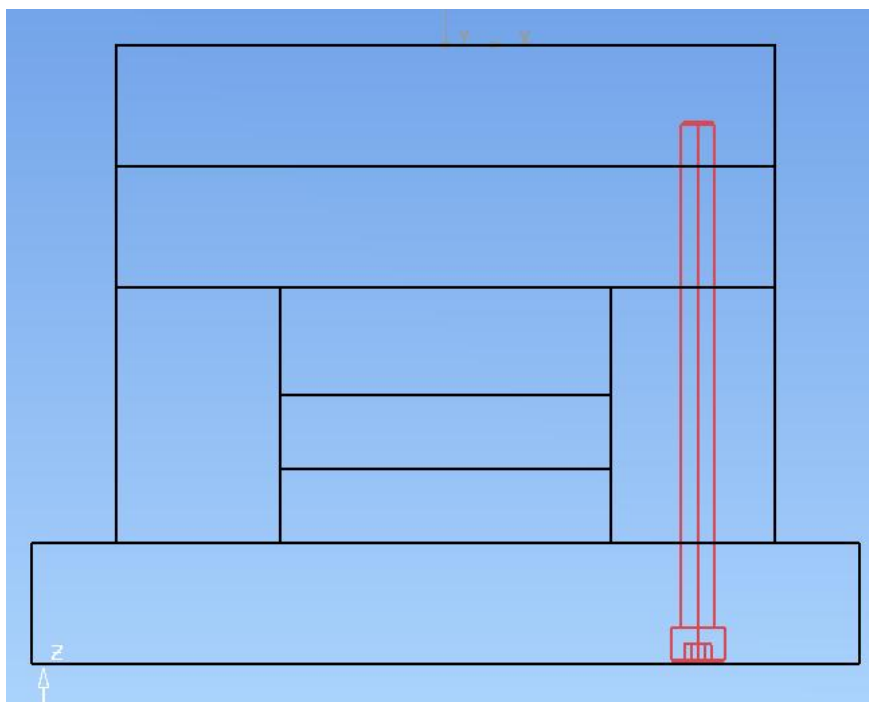
7. Generating Power Features.

Further enhancements to the generation of Power features have been added, this chapter looks at the changes and new functionality.

- **Open the model** – Cap Screw and Moldbase.psmodel and start **assembly** mode.

The model contains a PS-Moldmaker STD HASCO screw component positioned in a moldbase.

- Display only the **Moving half level group** and **view from Front (-Y)**



Generate Power Features

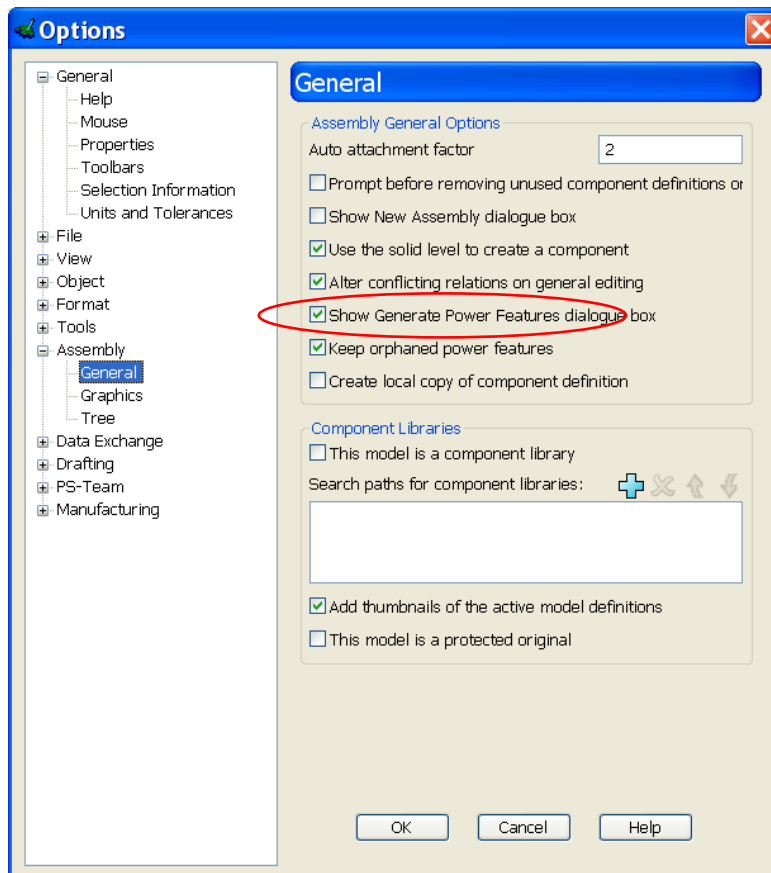
Although the methods to generate power features are similar to previous releases they have slightly different behaviour.

We have seen in the earlier exercise that using the RMB on the assembly tree will generate or remove power features for the whole assembly.



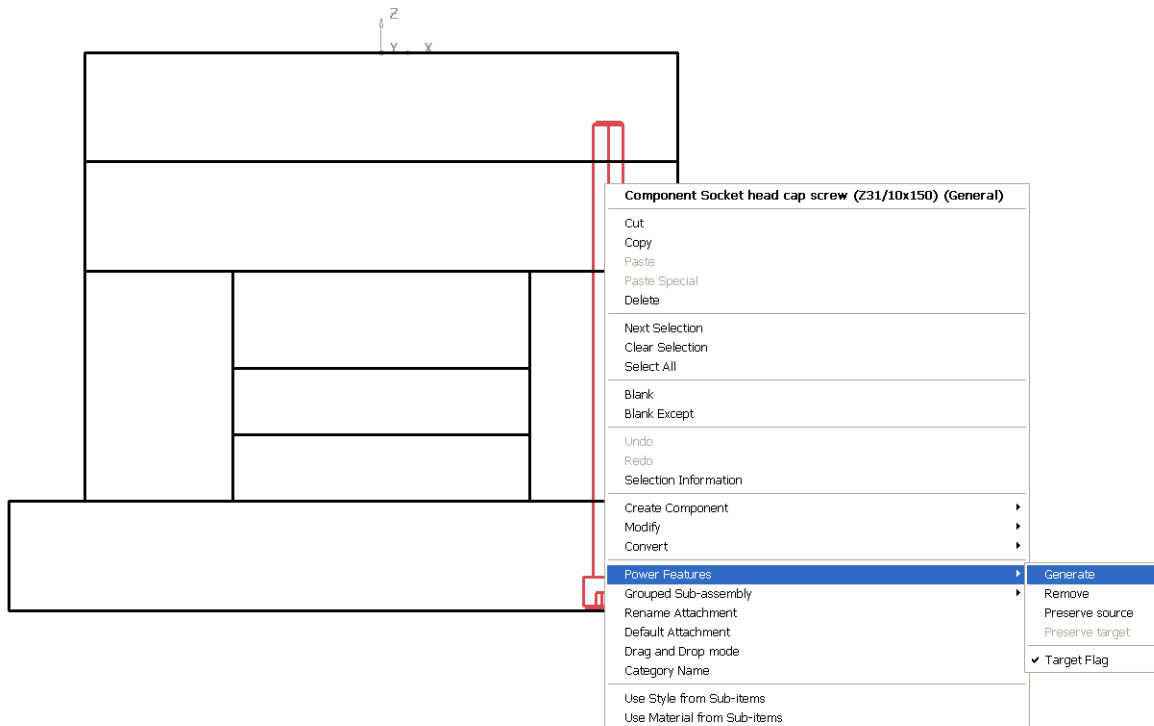
The only difference in the menu is the additional option of 'Power Features Summary'; we will look at this in more detail later in the exercise.

- Before progressing further we should look **Tools – options – assembly – general**

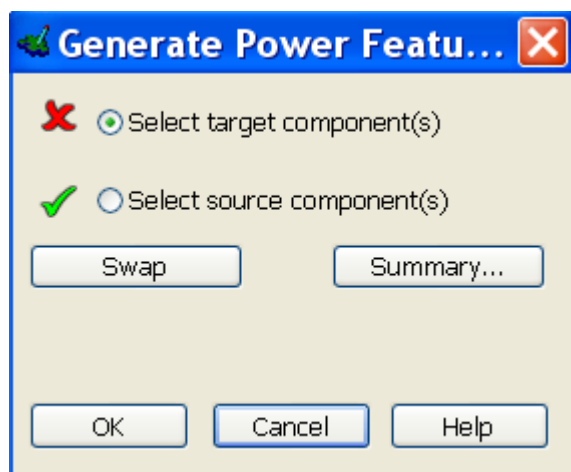


- The new option – **Show generate power features** dialog should be checked.

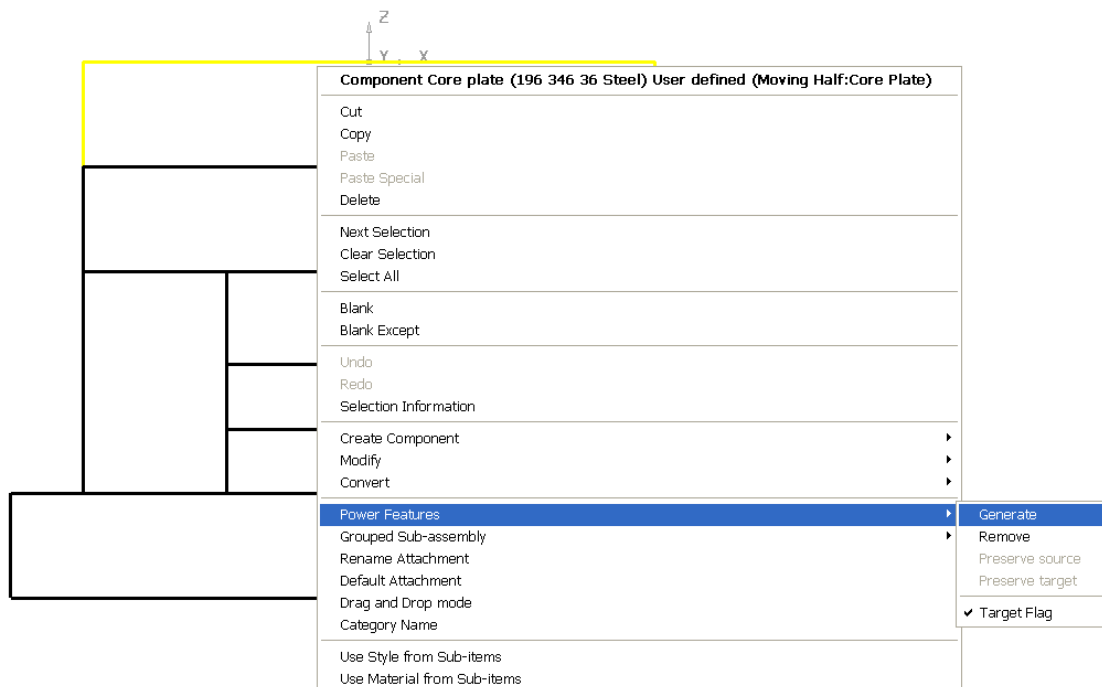
- Select the **screw component** and using RMB select **Generate Power Features**



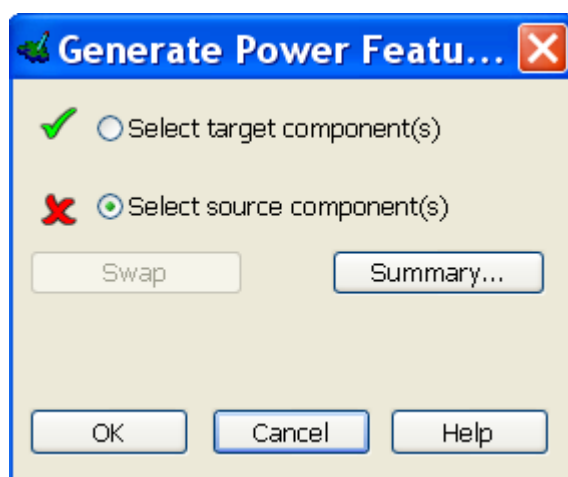
In this case the screw is recognised as a source component (due to the fact that it contains power features); if the form is accepted then any intersecting target component will have features generated ‘for’ the screw.



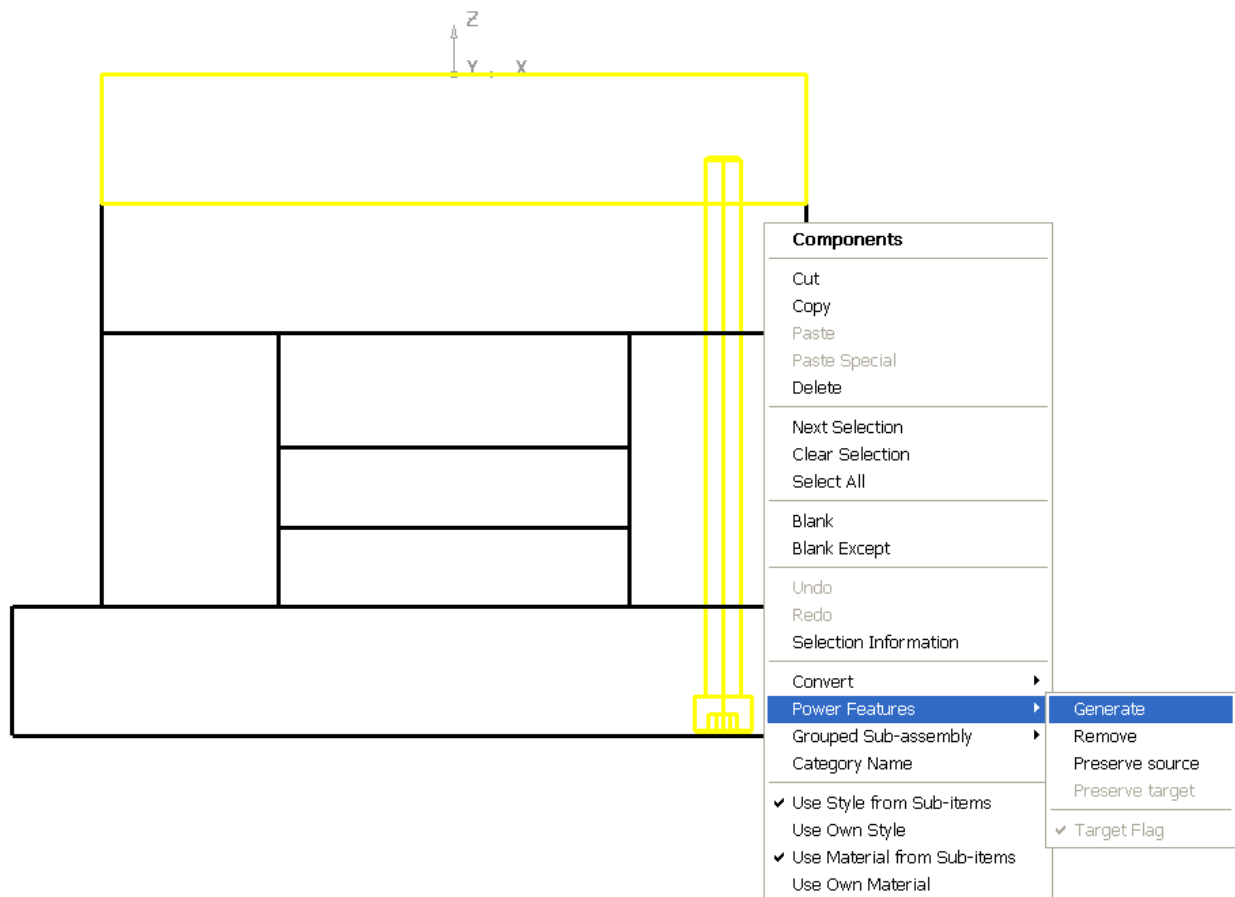
- Select the **core plate** and using RMB select **Generate Power Features**



In this case the plate is recognised as a target component (due to the fact that it does not contain any power features); if the form is accepted then any intersecting features from any source component will be generated 'in' the plate.



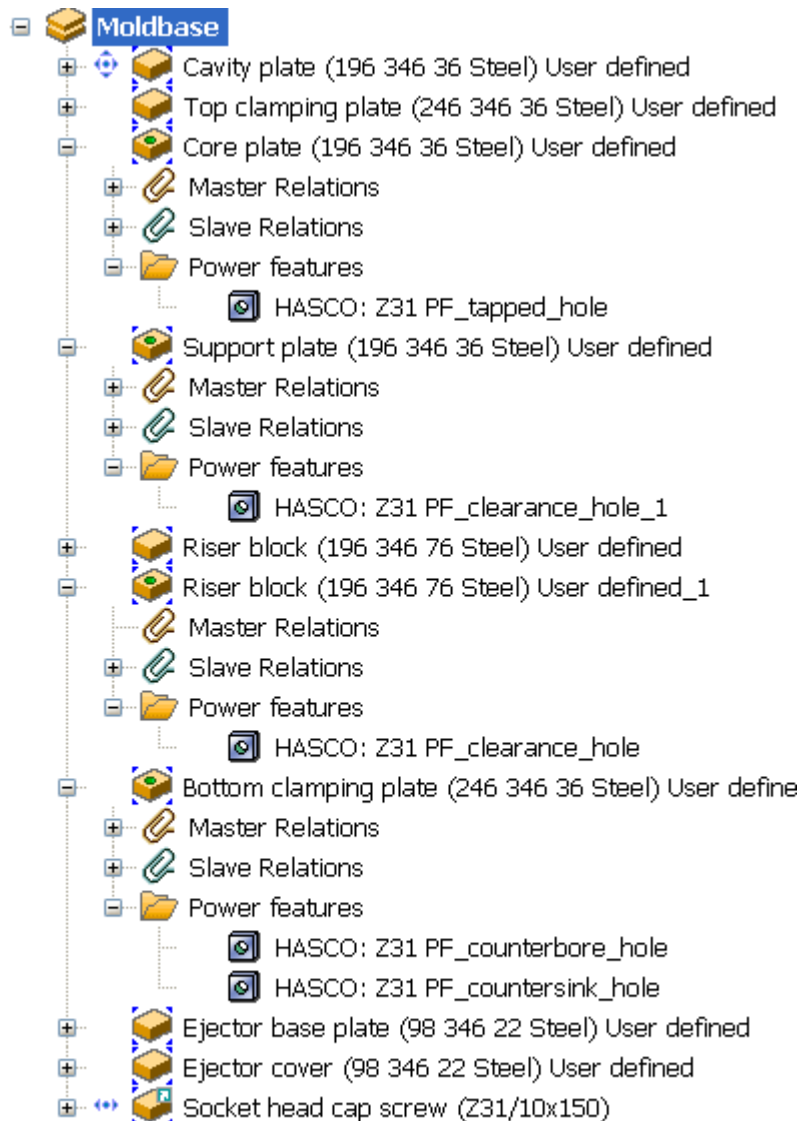
- Select both the **core plate** and **screw**, using RMB select **Generate Power Features**



If both source and target components are in the selection, then it is assumed that you have selected the specific components and power features that will be generated, in this case only the tapped hole for the screw is generated in the Core Plate



If we use any of the methods to generate power features the holes are generated in the correct plates and the assembly tree now displays information regarding the features.



There are several changes to the assembly tree icons relating to power feature, we can see some of them in the above picture.

If a component has power features copied into them by another (is a target) then the tree icon displays a hole and there is a new branch on the tree listing the features, naming their source and the type of hole.

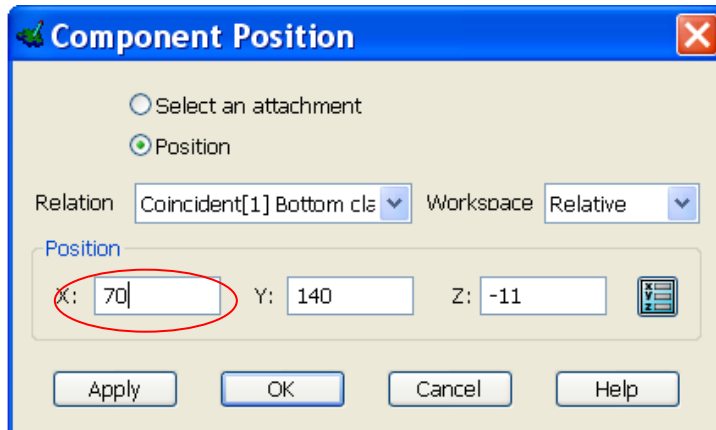


When a component contains power features (is a source) then it has a small green arrow in the top right hand corner.

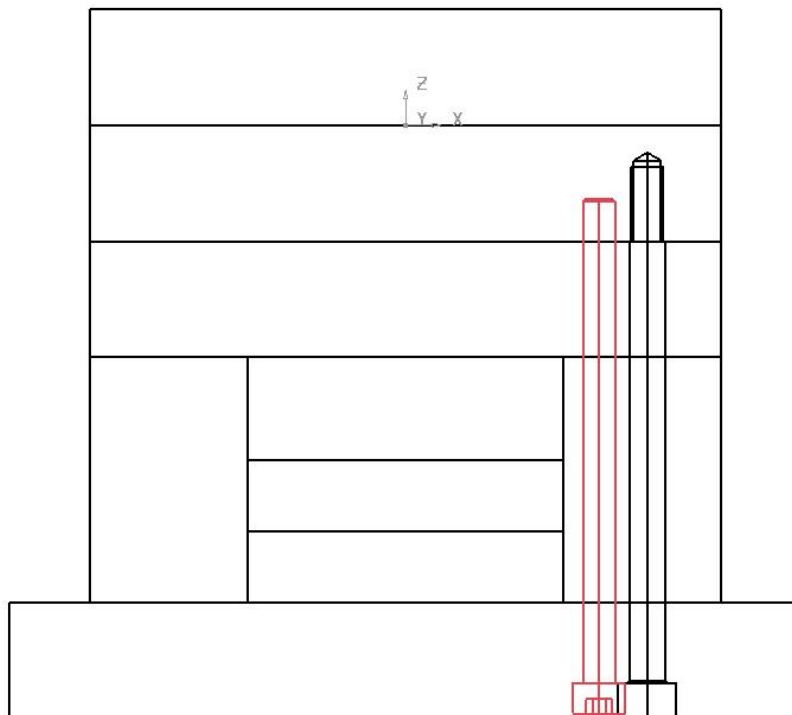


Once Power features are created the source and target components are 'linked' so that any change in the components is recognised.

- Select the **screw component** and double click to raise the **component position form**, change the **X** value.

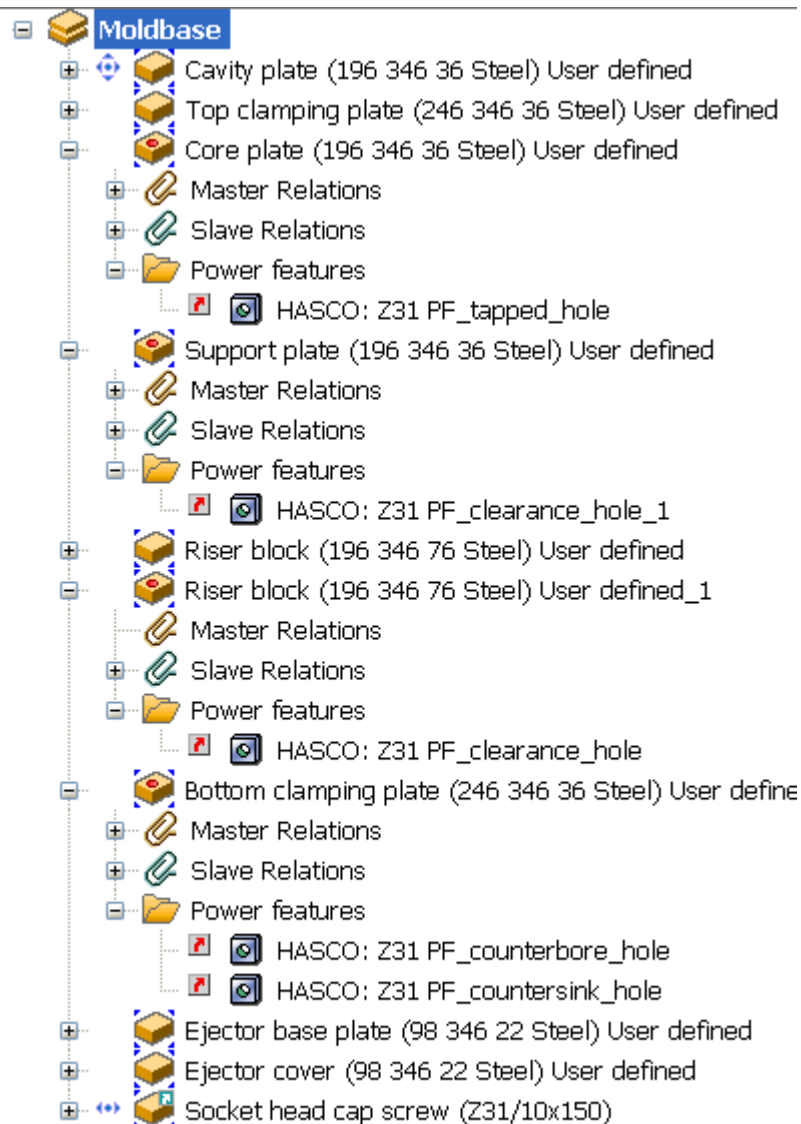


The screw will move away from the holes, therefore they are no longer correct



Note: The holes will not immediately update when the screw is moved

The assembly tree will update and display the following information:-



When a target component has features that have become out of date the tree icon's power feature indicator hole turns red.



Any features that are out of date are displayed in the tree branch with a boxed red arrow beside them.



The out of date status happens when:-

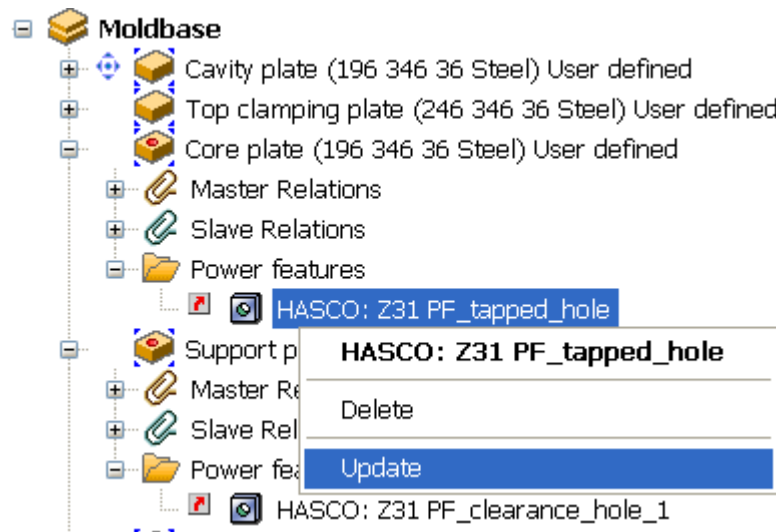
A source component is moved

A target component is moved

A source component has parameters changed

When features are recognised as out of date they can be corrected by simply repeating the generate power features method that was used to create them.

Alternatively individual features can be updated using RMB on the feature in the assembly tree.



8. Control of Power Features

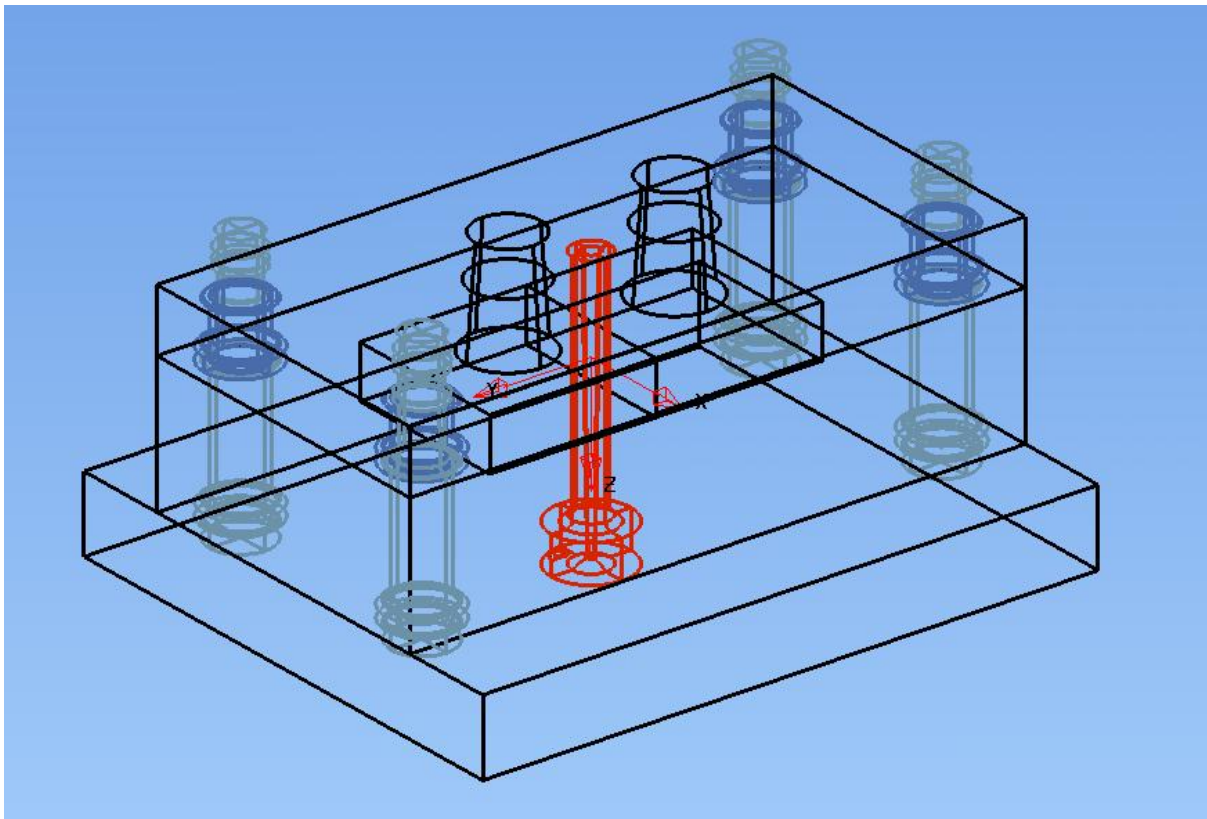
The components that are created using wizards have common default setting for their use with power features. However there are situations that the defaults will not give the correct result.

This chapter covers the methods and settings that can be used to control the application of power features to other components.

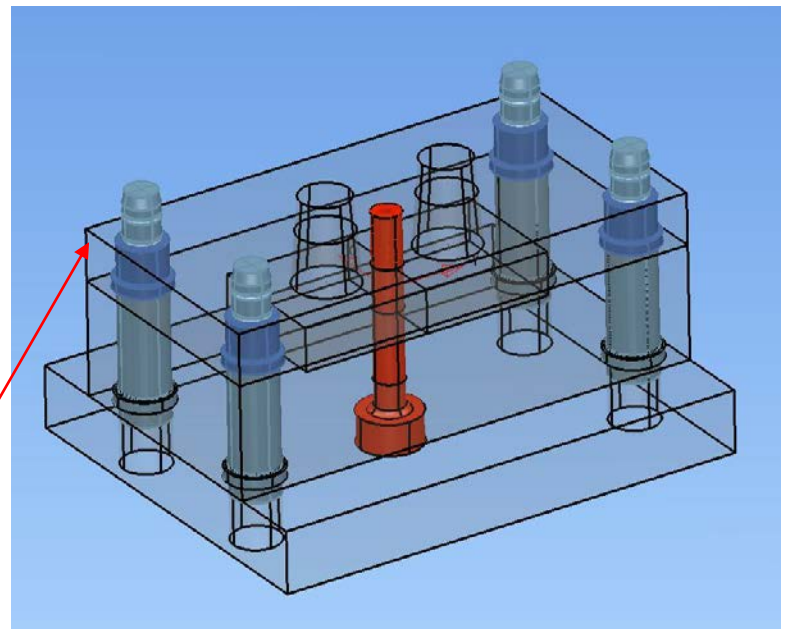
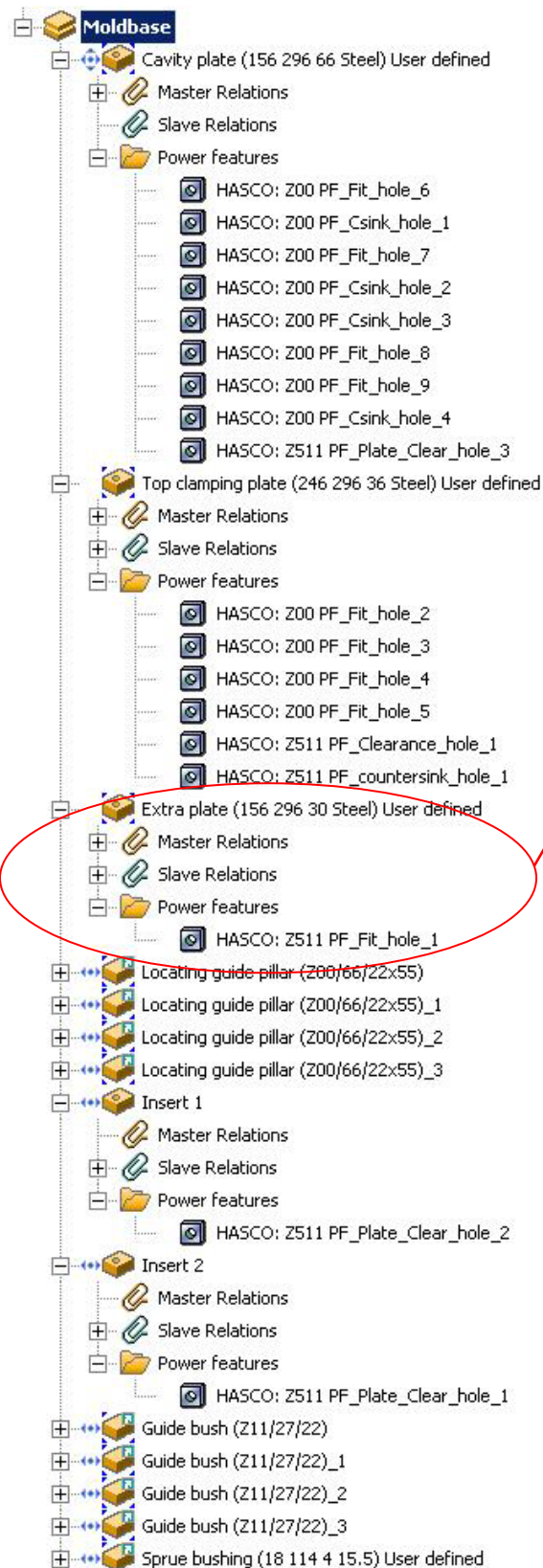
Open the model – Control Power Features.psmodel and start assembly mode.

The model contains a mixture of plates / components and inserts.

The purpose of this model is purely to explain the control methods that can be used with power features



- **Generate Power Features** from the **Assembly tree** for all of the components

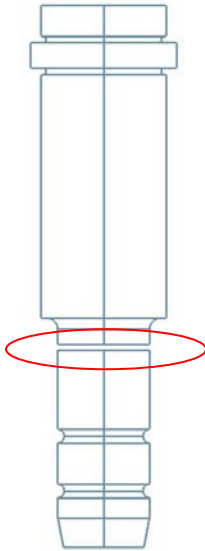


On examination, the assembly tree reveals that the Extra Plate does not contain any holes for the Guide Bushes. As the Bushes are located in this plate then you would expect to see the fit holes.

If you view the power features that have been applied to the Guide Pillar it can be seen that a hole from the Guide Bush has been copied incorrectly to the Pillar.

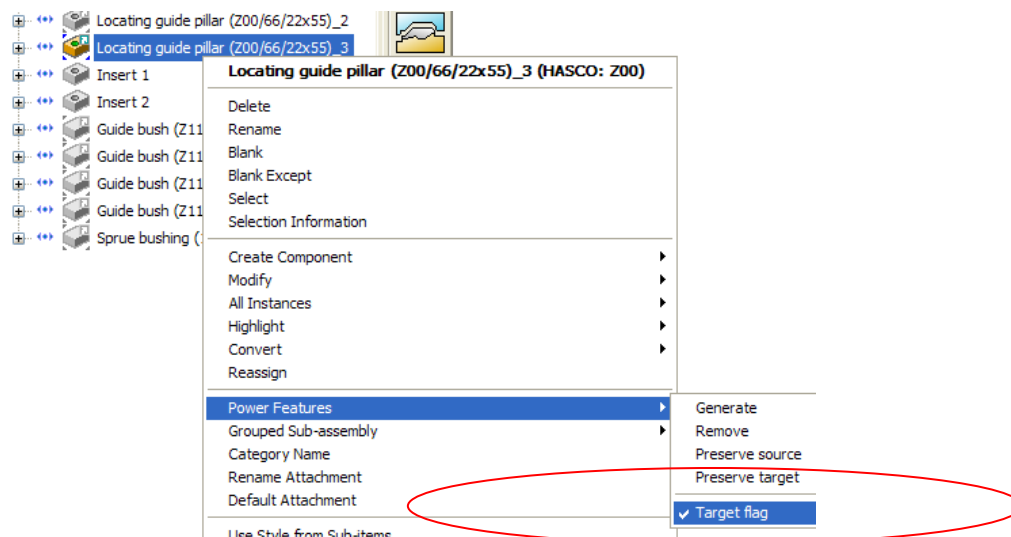


- Select a Guide Pillar and blank except to see the hole from the Bush has “cut the Pillar in Half”



- In the Assembly tree select a Pillar and using RMB view the **Attributes**, In this case the **Power Feature Target** attribute is turned ON

This Attribute means that when Power Feature intersections are calculated the component will be considered as a target.



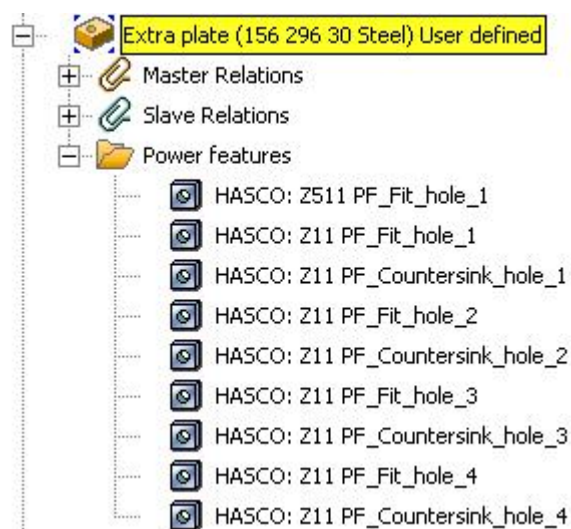
- Turn the Attribute **OFF** (un tick by picking the label)



By default all PS-Moldmaker components will have the Attribute set to the most appropriate state. Many components will never had Power Features applied to them, the example here is that a Pillar will never be cut by a mating Bush as they fit together.

- Set the Attribute for all Guide Pillars to be OFF
- Generate Power Features again

Now the Extra plate has the correct holes to suit the Guide Bush, the Pillar is no longer cut by the Bush.



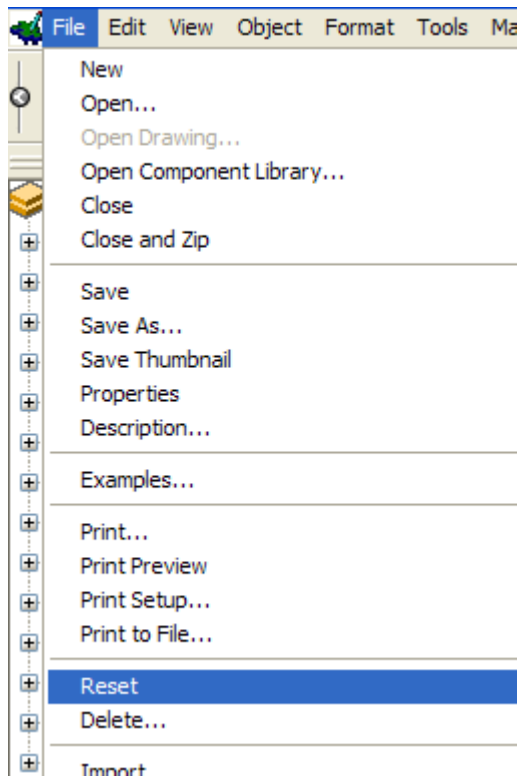
Note:

It should be noted that when a power features target 'flag' is turned off then the component will never be included in the intersection calculations.

So if you do not want a component to be cut but it needs to be included in the intersection calculation then changing the attribute will not give you the correct results.

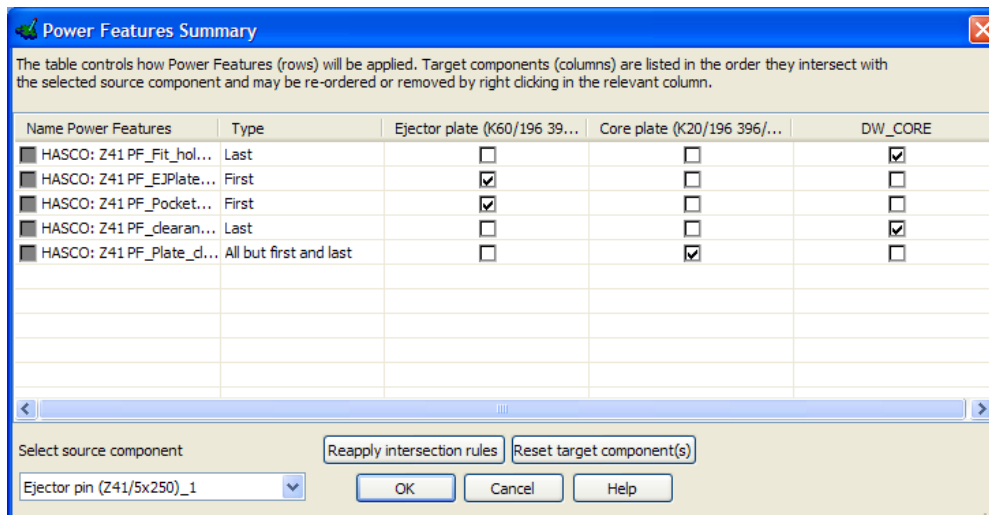
The next stage of this exercise requires the model to be at the original state.

- From the **File** menu **Reset** the model



Power Features Summary Sheet (Reference Guide)

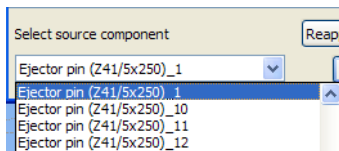
The next control option and possibly the most useful is the summary sheet...



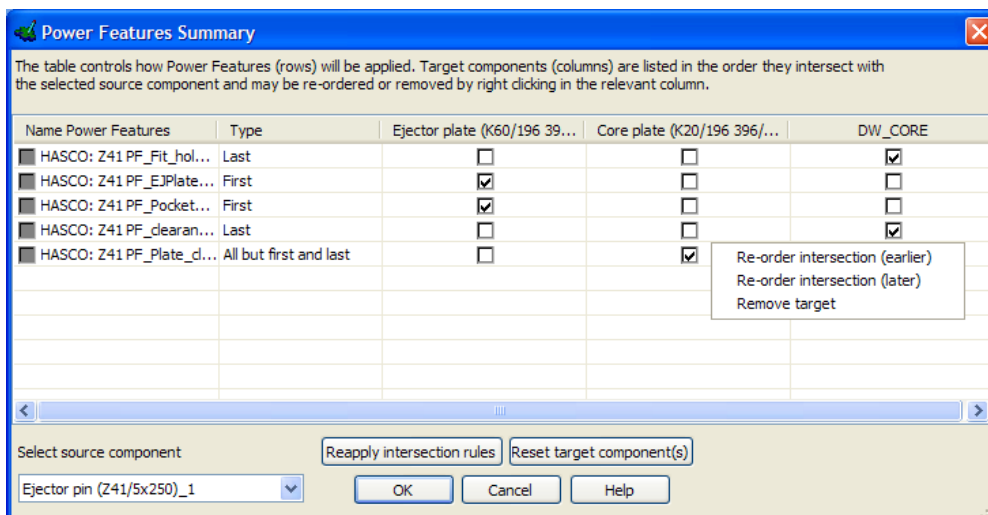
Name Power Features – states the power features that are contained in the selected source component

Type – displays the advanced power feature intersection setting.

Select source component - when you select a component from the list, you can set the options for the Power Features in the table for that component. So if there are 4 pins, there will be four options in the drop-down list



RMB menu - The menu can be used to 'move' the target components to a position earlier or later in the sheet. This will change the features that are applied to that target when the defaults are re applied. Targets can also be removed from the sheet.



You can change the application of a power feature to a particular target component by ticking or un-ticking the relevant box in the table.

This may not always give a result as all of the power features may not intersect with all of the targets

A Power feature can be applied to **all** targets by ticking the box next to the Name



| Name Power Features | Type | Ejector plate (K60/196 39... | Core plate (K20/196 396/... | DW_CORE |
|--|-------|-------------------------------------|-------------------------------------|-------------------------------------|
| <input checked="" type="checkbox"/> HASCO: Z41 PF_Fit_hol... | Last | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> |
| <input type="checkbox"/> HASCO: Z41 PF_EJPlate... | First | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| <input type="checkbox"/> HASCO: Z41 PF_Bodest... | First | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |

Reset target component(s)

If the sheet is re-ordered or targets are removed the **defaults** can be reset to the initial state.

Reapply intersection rules

After re-ordering or removal of targets the **default intersection rules** can be reapplied to the remaining targets.

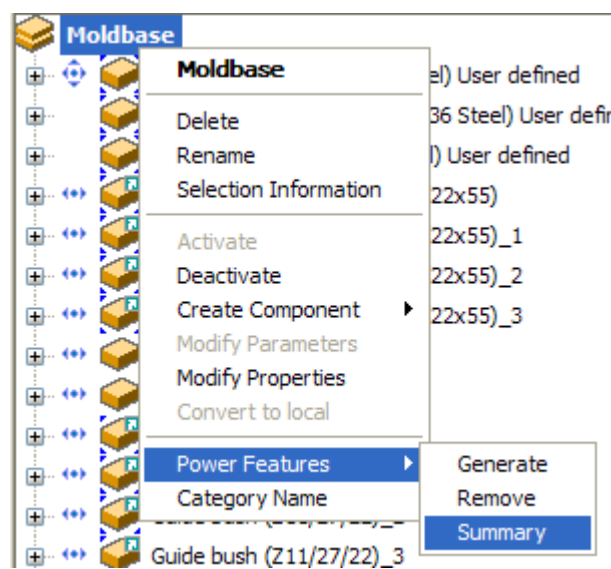
OK

OK will **save** ANY modifications that you have done and close the sheet

Cancel

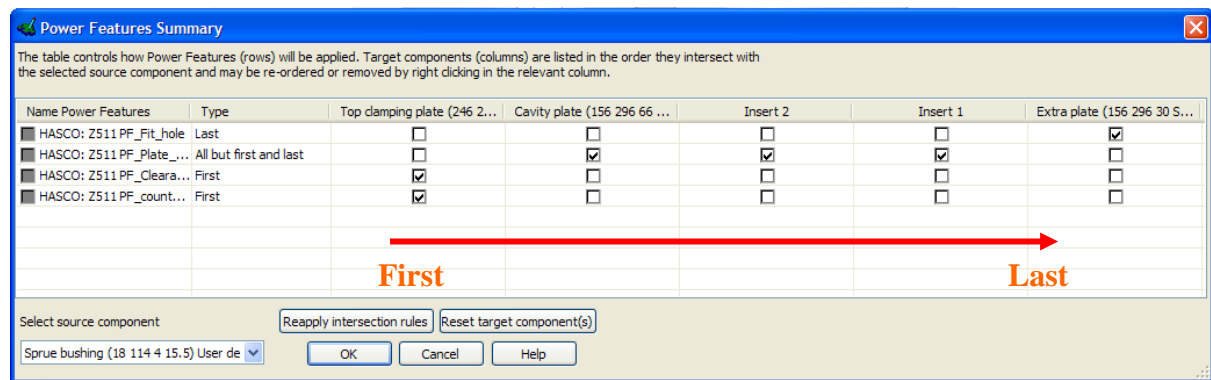
Cancel will close the sheet and **discard** any changes

- Select **Power Features Summary** from the assembly tree RMB menu



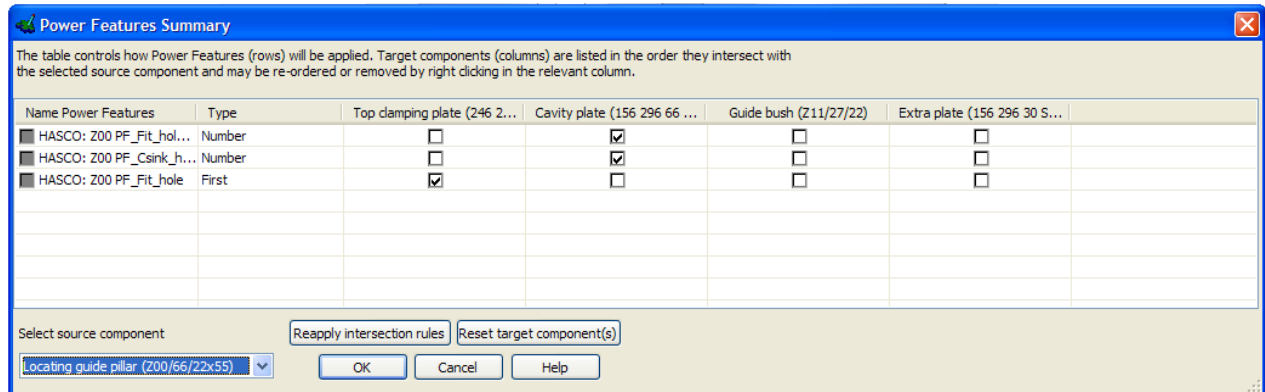
The Power Features Summary Sheet is raised.

When the Power Features Summary is raised the intersections are shown for the latest component added to the assembly, in this case it was the Sprue Bush. You can see that the features are applied from **left to right**, the **Top Clamping Plate** being the **First** and the **Extra Plate** being the **Last**.

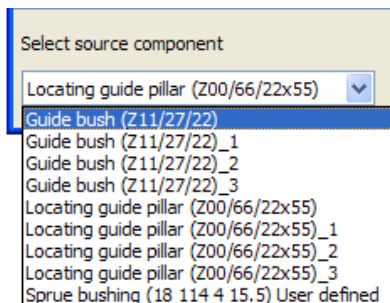


The intersecting Target components are all shown in the sheet; in this case the Guide Bush and Extra plate were identified as intersecting.

They do not have any features applied to them as the Advanced settings on the Hasco Z00 only specifies the first and second intersections to be used.

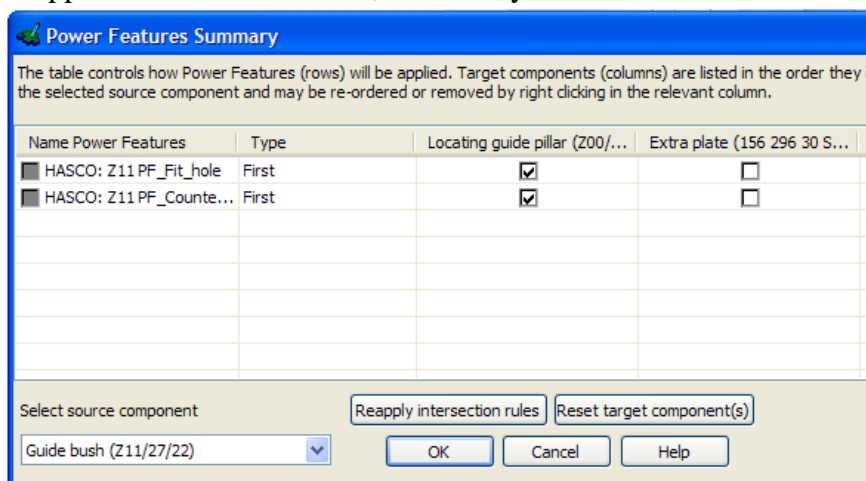


The Source component can be selected from the Drop down list. The intersections are shown for the current selection.

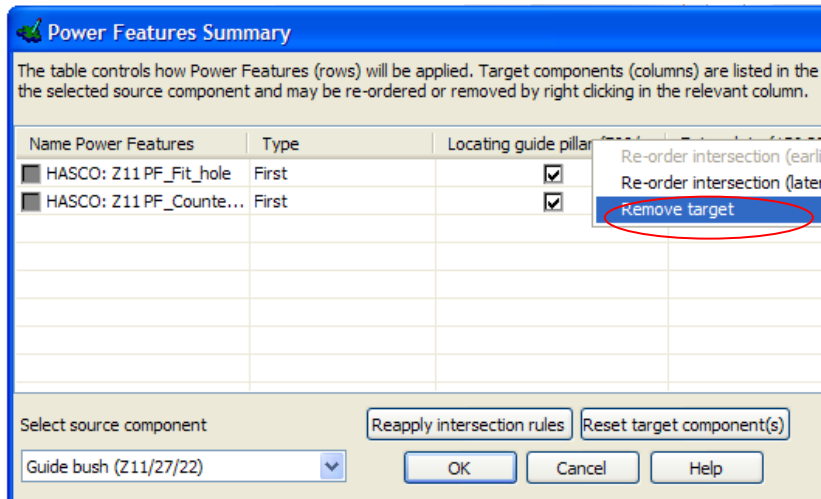


- Select the **Hasco Z11 Guide Bush** as the **Source** component.

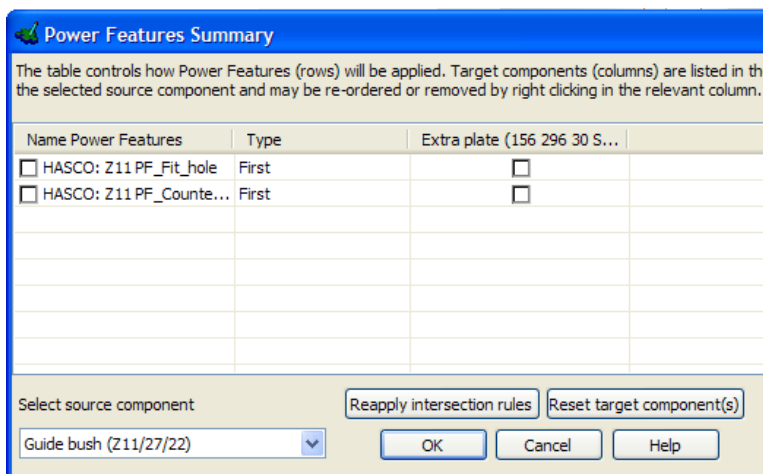
The Summary sheet indicates that the **Guide Pillar** is the **First** intersection and the **Extra Plate** is the **Last**. As the **Advanced** settings only apply features to the **First** both features will be applied to the Guide Pillar, incorrectly



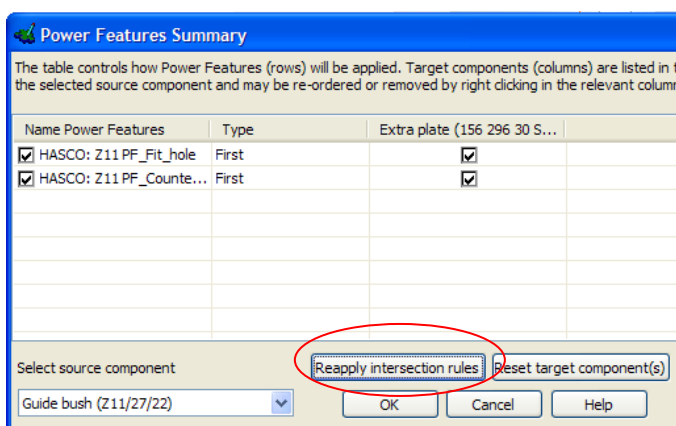
- Use the RMB menu on the **Guide Pillar Target** column, pick **Remove Target**



Now that the Guide Pillar has been removed as a Target the **Extra plate** remains as the only intersecting component, however as this was not the First component originally the summary sheet still does not show the features being applied.

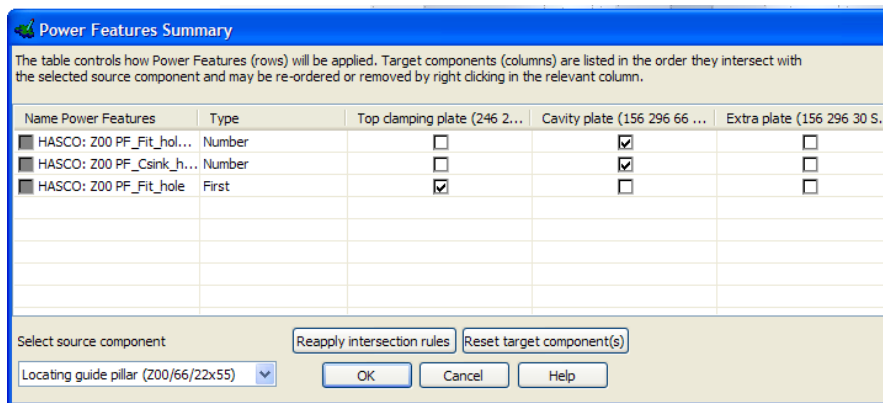


- **Reapply intersection rules** to update the sheet, this action confirms the Extra Plate as being the First intersection



- **Cancel** the Summary Sheet

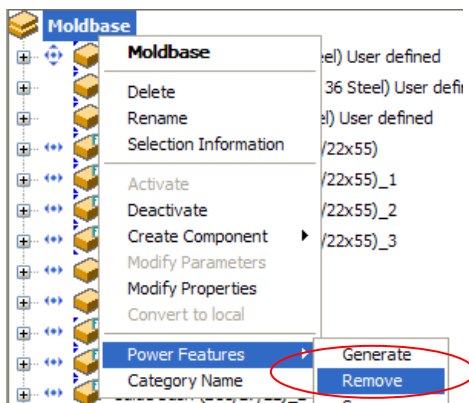
- Modify the **Attribute Power** feature **Target** for the **Guide Pillars** and **Bushes** to **OFF**



- Raise the **summary sheet** again

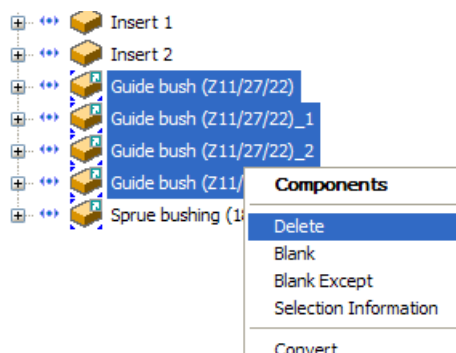
View the information for Z00 Guide Pillar; you can see that the Guide Bush component that was previously included is not longer shown.

- **Remove** all Power Features. (Use this option with Great care)



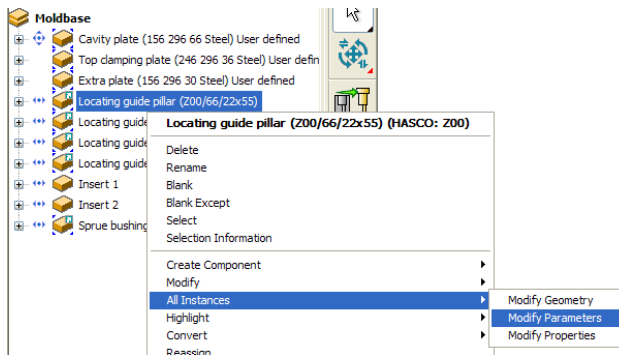
The next exercise looks at how you can change the applications of power features using the summary sheet.

- In the assembly tree select the **Guide Bush** components and **Delete** them

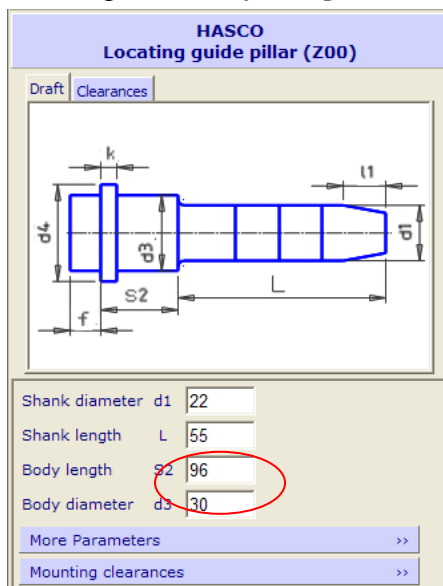


We are going to change the Extra plate that was in a 'Stripper plate' scenario to a fixed plate.

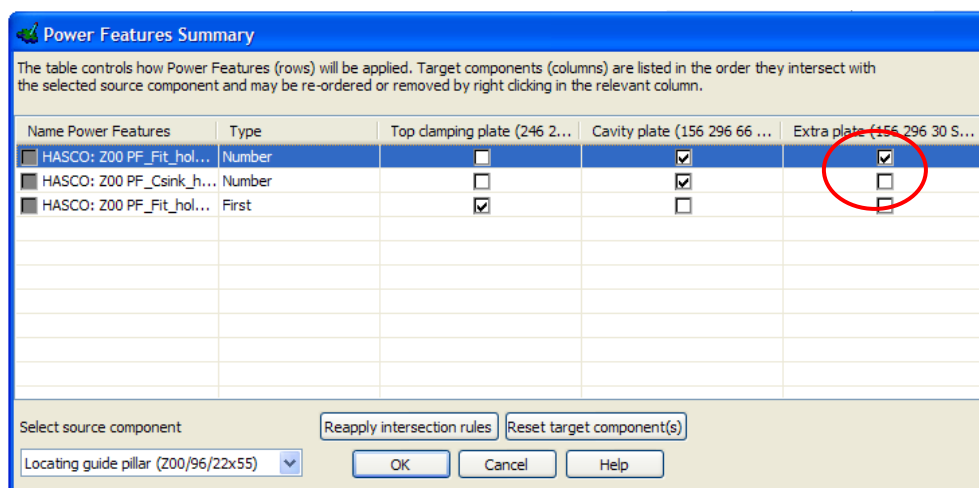
- Select a Guide pillar and Modify Parameters (All instances). This will modify all of the pillars in one go.



- Change the **Body Length** to **96mm**



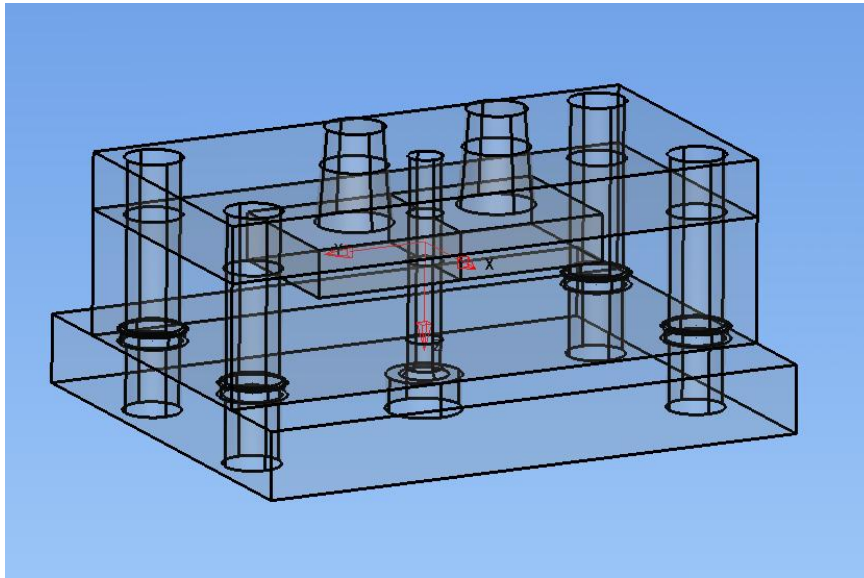
- Raise the **Summary Sheet** and view the **Guide Pillar** information.
- Tick the box to apply the **Fit hole** to the **Extra Plate** as shown below.



The Guide Pillar Power Features do not normally intersect with the Extra plate as the Advanced Power Features are set to component First and Number 2.

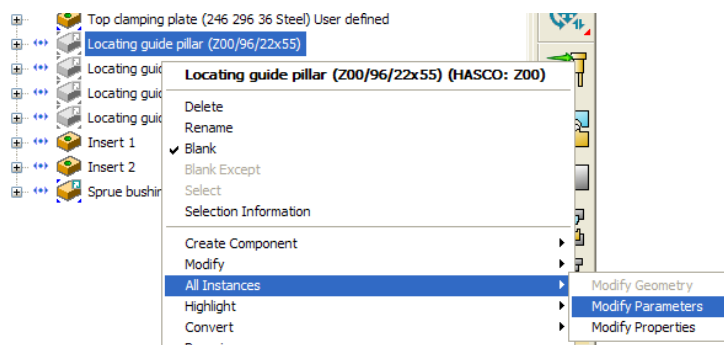
The Advanced settings are done this way as the most common applications for the Guide Pillar are with two plates i.e. Cavity plate and Top Clamping plate.

- **OK** to save the changes
- **Generate Power Features** for the whole assembly
- Select the **pillars** and **sprue bush** and **Blank** to see the holes

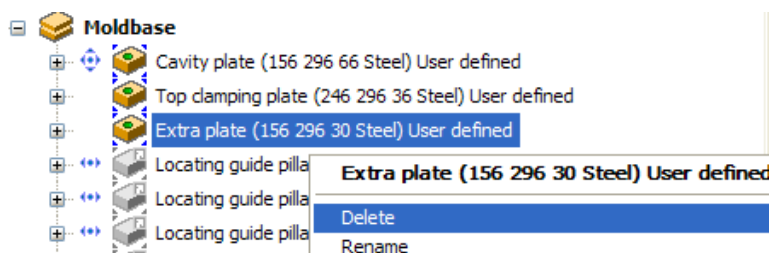


A further change is required to the model to progress.

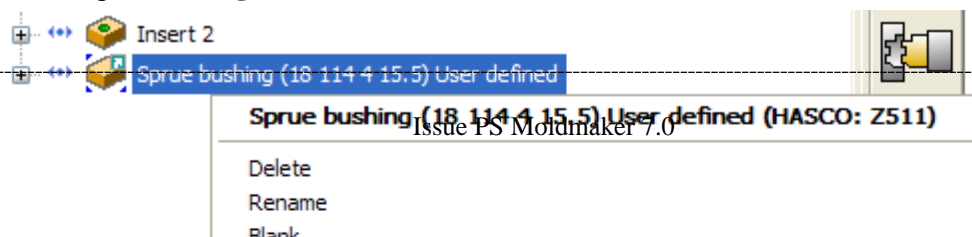
- **Delete the Extra Plate** component and set the **Body Length** parameter on ALL of the **Guide Pillars** back to **66mm**



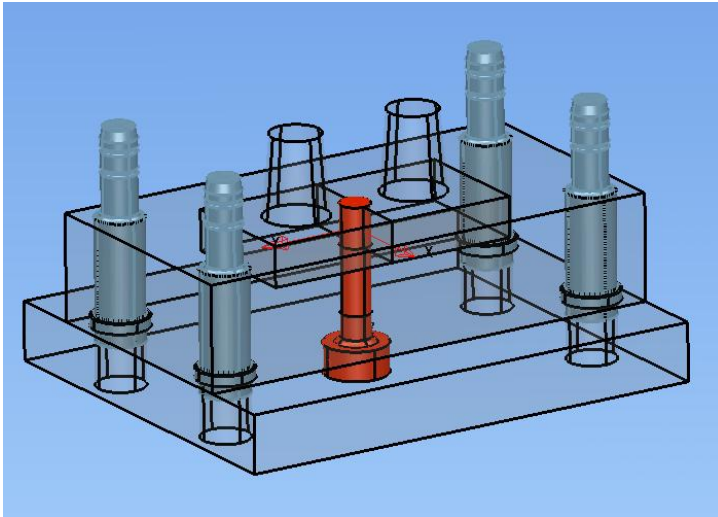
- **Modify Parameters (all instances)** on the **Sprue Bush**



- Change the **Length Parameter** to **84mm**



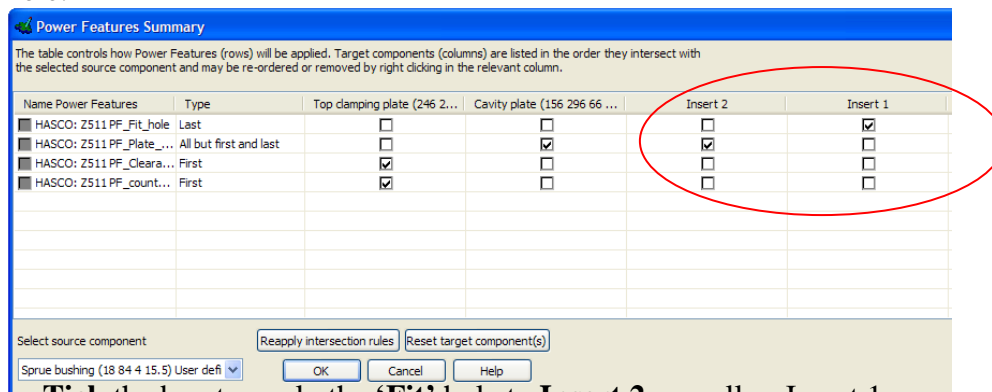
The model in its new state is shown below.



- **Raise the Power Features Summary Sheet**

The **Fit** hole is going to be applied to the **Insert 1** component only as this is identified as the **Last intersection**. This identification could have been **Insert 2** as they both occupy the same relative position to the **sprue bush** component.

Currently the **Insert 2** will have a Plate clearance hole applied where it should be a '**Fit**' hole.



- **Tick** the box to apply the '**Fit**' hole to **Insert 2** as well as Insert 1.
- **Un tick** the box that is applying the **Plate clearance** hole to **Insert 2**

Power Features Summary

The table controls how Power Features (rows) will be applied. Target components (columns) are listed in the order they intersect with the selected source component and may be re-ordered or removed by right clicking in the relevant column.

| Name Power Features | Type | Top clamping plate (246 2... | Cavity plate (156 296 66 ... | Insert 2 | Insert 1 |
|--------------------------|------------------------|-------------------------------------|-------------------------------------|-------------------------------------|-------------------------------------|
| HASCO: Z511 PF_Fit_hole | Last | <input type="checkbox"/> | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input checked="" type="checkbox"/> |
| HASCO: Z511 PF_Plate_... | All but first and last | <input type="checkbox"/> | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| HASCO: Z511 PF_Cleara... | First | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| HASCO: Z511 PF_count... | First | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> | <input type="checkbox"/> |
| | | | | | |
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| | | | | | |
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| | | | | | |

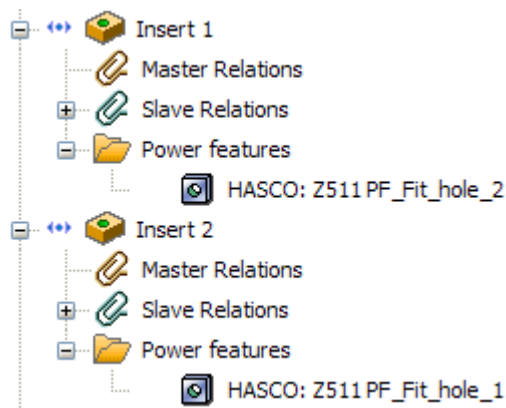
Select source component: Sprue bushing (18 84 4 15.5) User defi

Reapply intersection rules Reset target component(s)

OK Cancel Help

- **OK** the form and **Generate Power Features** for the assembly

You can see from the assembly tree that a 'Fit' hole is now applied to both Inserts



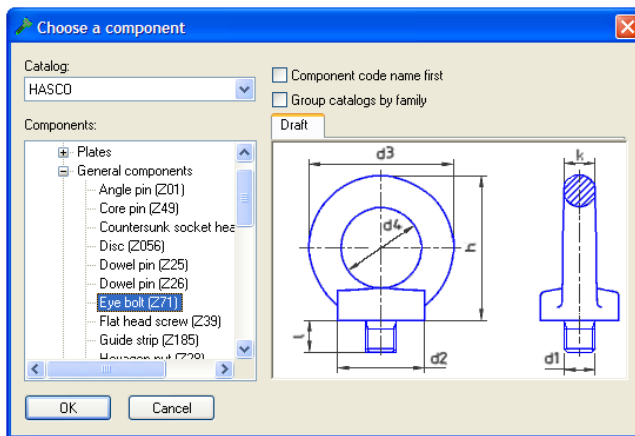
Preserve Power Features

The next option that we are going to use is the **Preserve Power Features**

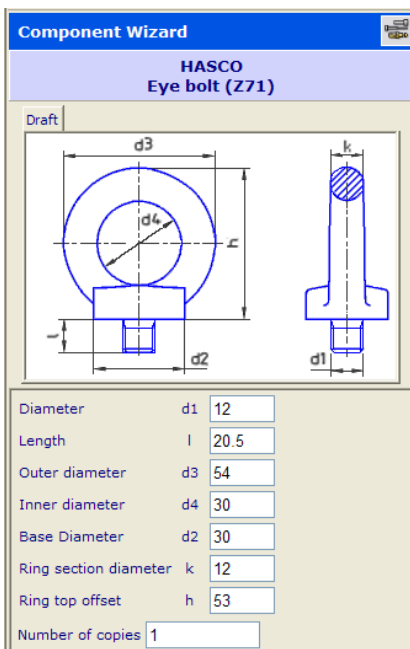
The word **Preserve** is used to describe the action of **permanently** copying the features to a target component and **breaking the link** with the **source** component.

The Preserve option can be used either for all “features” **contained** in a **Source** component or any “features” that have been **applied** to a **Target** component.

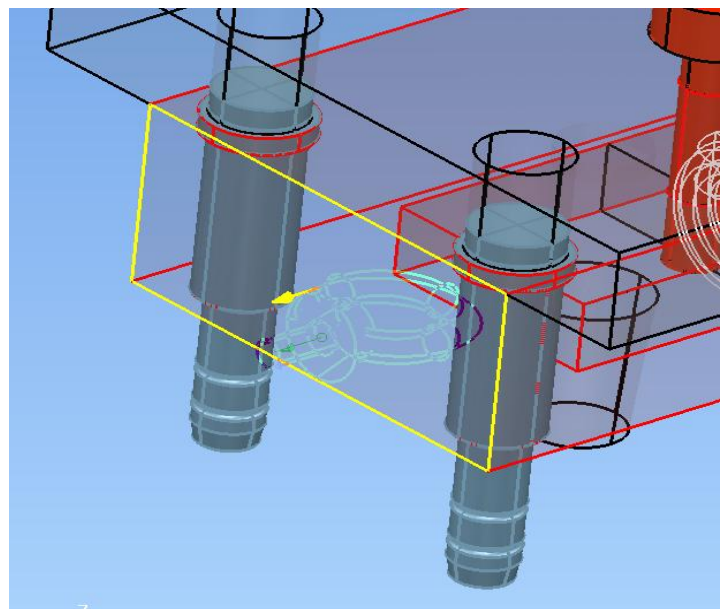
- Using **Component Wizard** create a **Hasco Z71** Eye Bolt



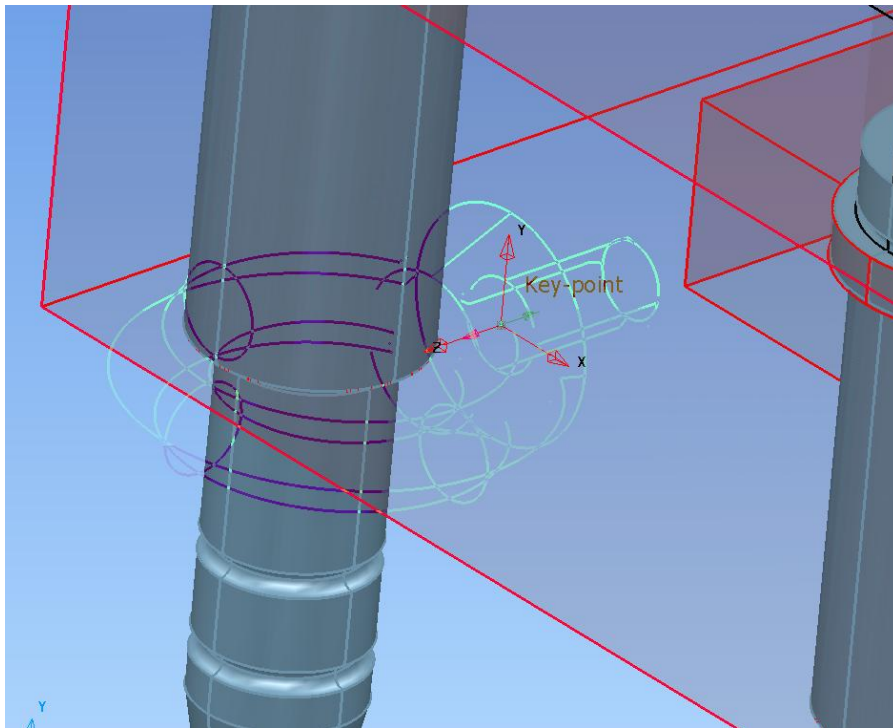
- Select the **M12** dimensions, select the **Cavity Plate** as **Master** component and **Apply**



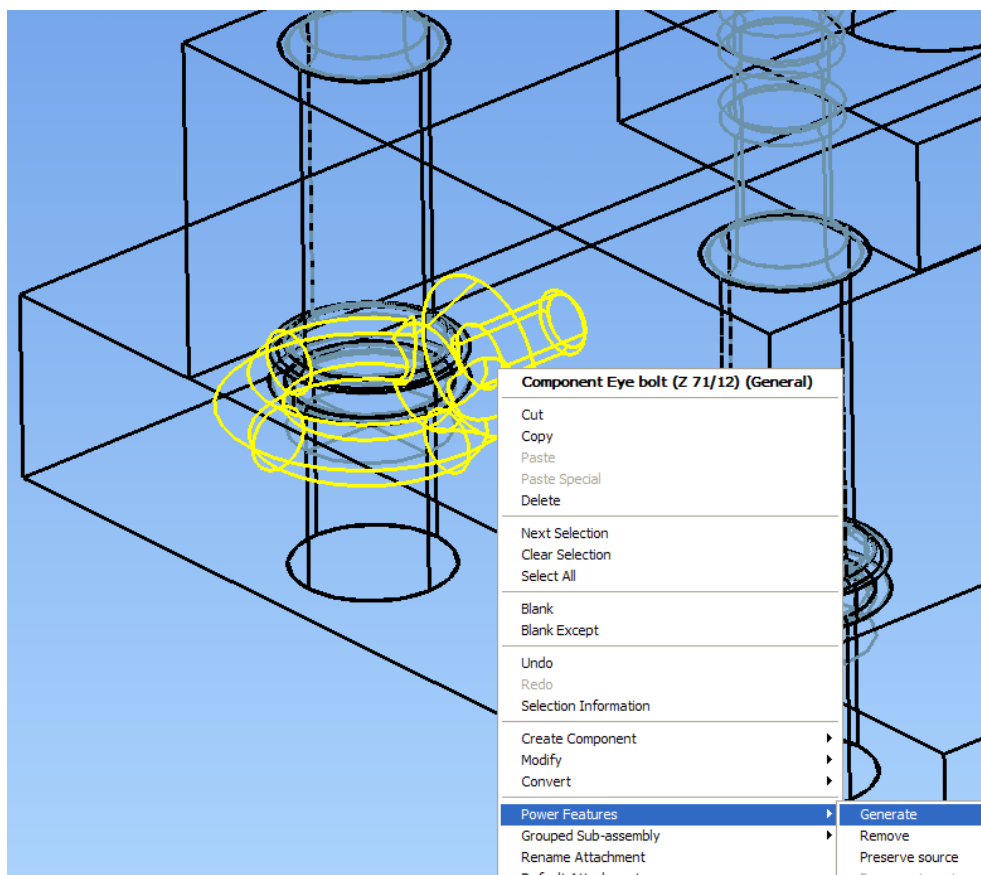
- Pick one end of the **Cavity plate** and position the **Eye Bolt**.
The **alignment** direction may need to be **reversed**



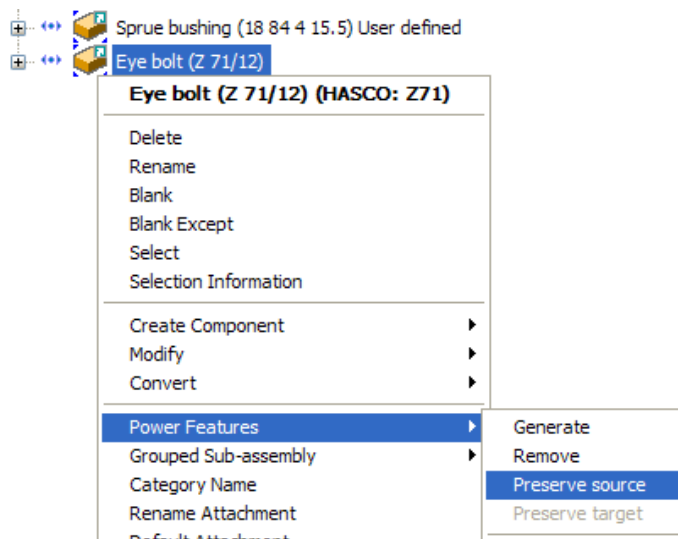
- Use the **Key Point** to snap on to



Select the **Eye Bolt** and using RMB menu **Generate Power Features only** for this component

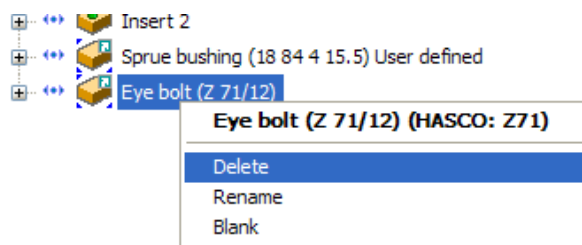


In the Assembly tree, **select the Eye Bolt** and using RMB menu Pick **Preserve PFs source**



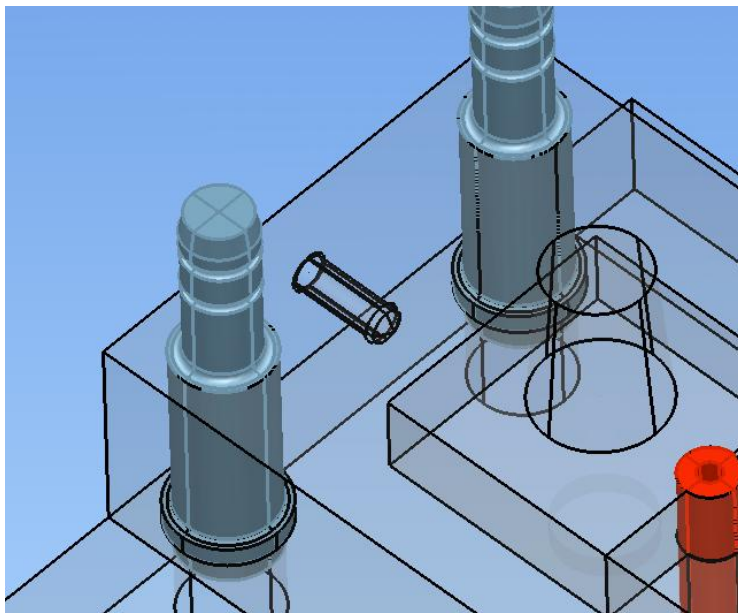
PFs is abbreviation for **Power Features**

- Select the **Eye Bolt** component and **Delete**



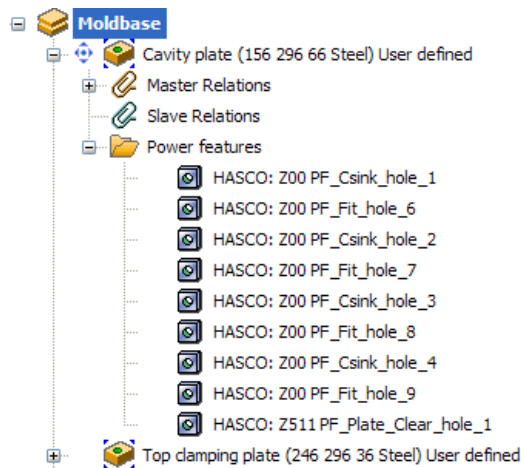
- **Generate Power Features** for the assembly

The Tapped hole remains in the plate after the Eye Bolt has been deleted

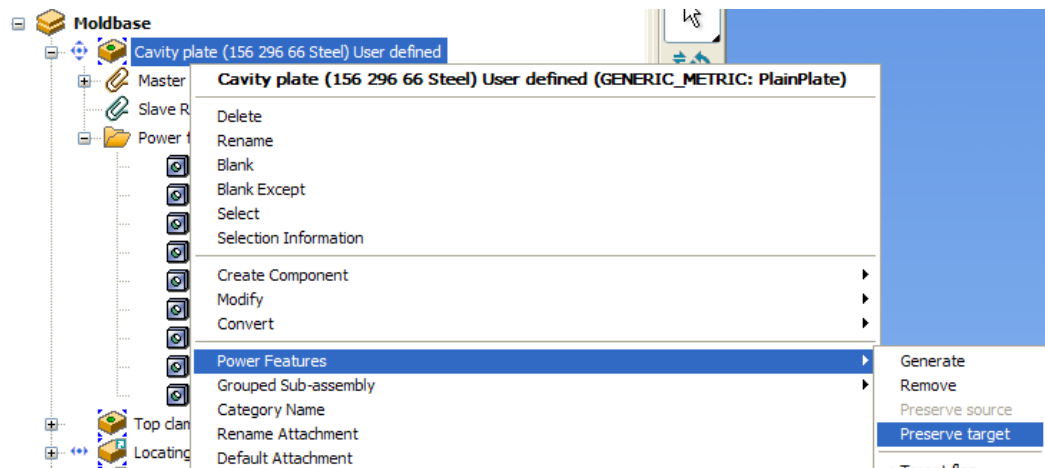


The same option can be used on the Cavity plate component

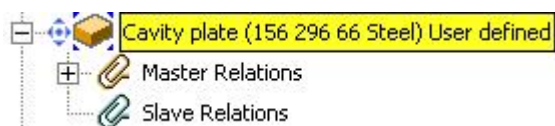
The assembly tree shows the Power Features on the Cavity Plate



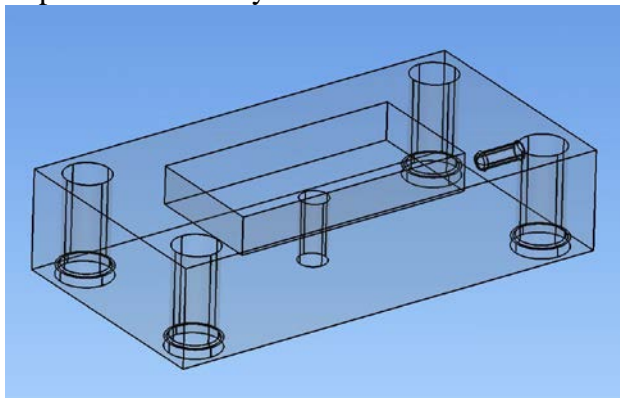
- Select the **Preserve PFs target**



The holes are no longer Power Features on the Cavity Plate



- Select the **Cavity plate** and **Blank Except**, to see the holes that are now permanently copied to the Cavity Plate



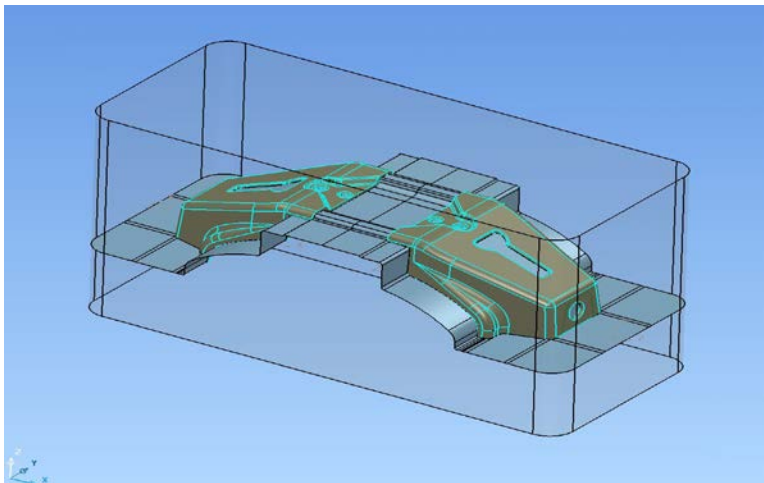
9. Moldbase Wizard - Standard.

When Creating a Moldbase with Moldbase Wizard there are now several methods that can be used to increase the flexibility when designing custom and very large tools.

The first method is basically the same as previous versions, the main improvement being that all moldbase components are now parametric.

In this exercise we will build a tool design based on a standard Hasco moldbase, adding any extra ejector pins / screws etc via Component Wizard.

- Open model – Moldbase 1.psmodel



In this model there are pre-created core and cavity components. Display only the **Die Inserts level group**

- Start **PS-assembly** and **PS-Moldmaker**.
- **Activate** the Work Plate **tool CL**
- Start the **Moldbase Wizard** from the Moldmaker wizard tool bar



The first page is displayed in the browser window; make the indicated selections and changes.

Mold Base Wizard

Base catalog: **HASCO**

Dimensions: ☒ Standard ☐ Custom
196x396

Base type: **Standard mold**

☐ Support plate
☒ Risers
☒ Ejection system

Guide pillars: ☒ Fixed half ☐ Moving half

Show the mold base

Rotate the mold base: ☐ **☒** ☐ ☐

Plates

Top clamping plate
 Cavity plate
 Core plate
 Ejector plate
 Ejector base plate
 Risers
 Bottom clamping plate

Type: **K10 (overhang in width, loc. rir)**

Thickness: **36**

Material: **1730**

Dimensions: Standard, mms
 Product code: K10/196x396x36/1.1730

Comments...
 Dimensions...
 Levels...

< Back Next > Cancel

- Select **HASCO**
- Select **196 x 396**
- **Remove the Support Plate**
- Select **Rotate 90**
- Modify **Cavity Plate thickness** to **96mm**
- Modify **Core Plate thickness** to **56mm**
- Modify **Riser thickness** to **76mm**

Mold Base Wizard

Components

- Guide pillars
- Guide bushes
- Centering sleeves
- Fixed half screws
- Moving half screws
- Ejector base plate screws
- Locating ring
- Seating washer discs
- Seating washer screws

Remove Add...

Show the component

Catalog HASCO

Type Z00

Shank diameter 22

Shank length 95

Dimensions: Standard, mms Dimensions...

Product code: Z00/96/22x95

Levels...

< Back Next > Cancel

In this case we will accept the standard components that are selected by the wizard

- Select **Next**

Mold Base Wizard

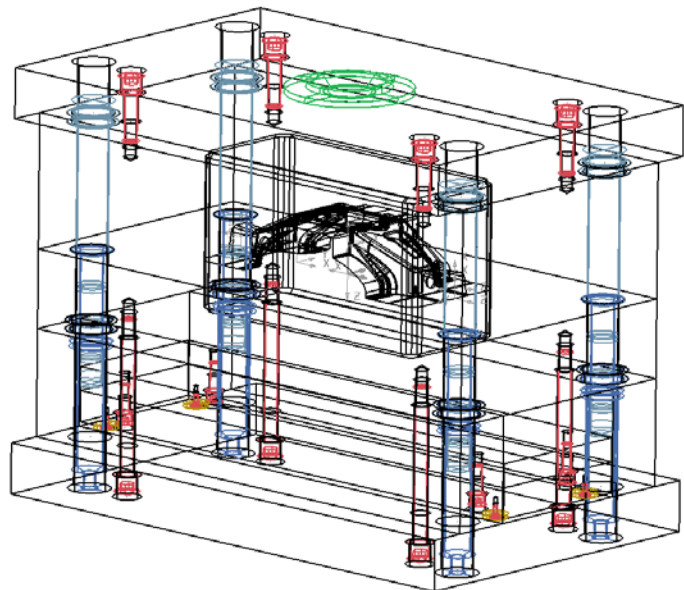
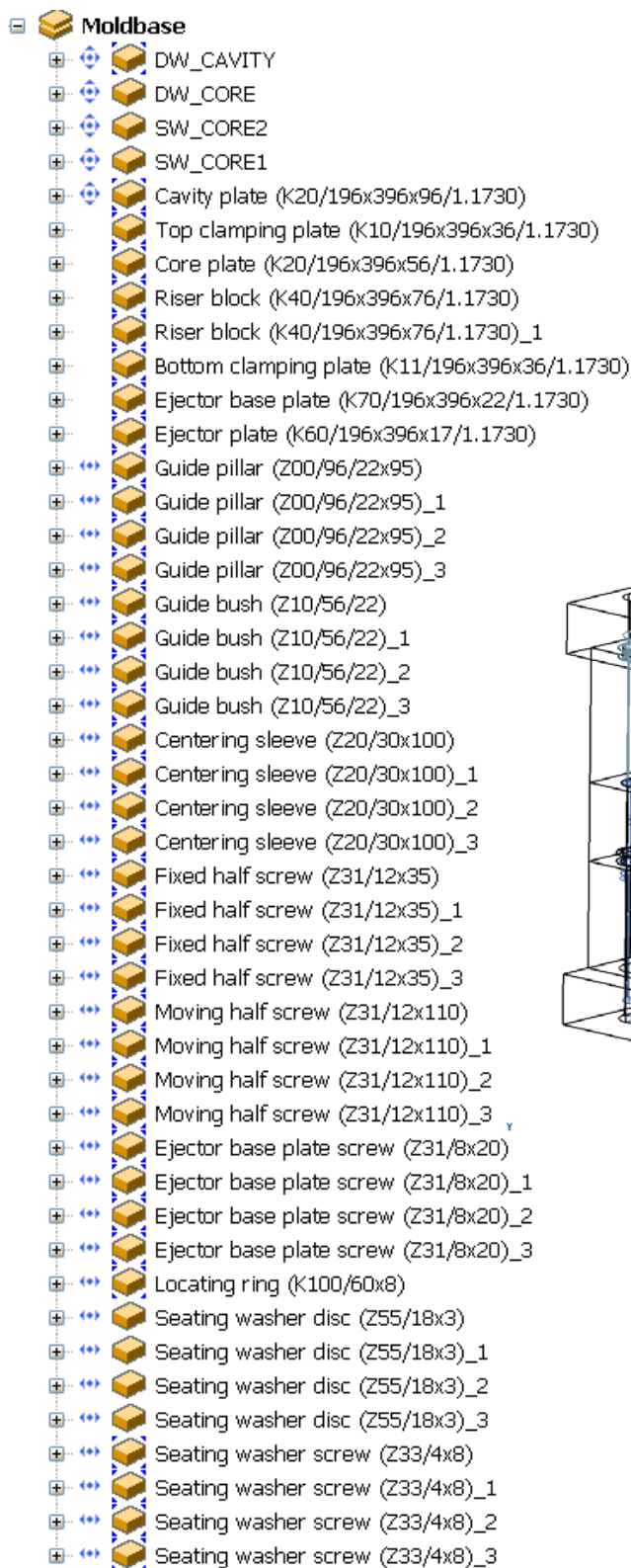
You have successfully completed the Mold Base Wizard

You specified the following parameters for the mold base:

| | |
|------------------------------|------------------------------------|
| Mold base | |
| Catalog | HASCO |
| Type | Standard mold |
| Guide pillars | Fixed half |
| Rotation angle | 90 |
| Width | 196 |
| Length | 396 |
| Dimensions | Standard |
| Cavity plate | |
| Type | K20 |
| Thickness | 96 |
| Material | 1730 |
| Dimensions | Standard |
| Top clamping plate | |
| Type | K10 (overhang in width, loc. ring) |
| Thickness | 36 |
| Material | 1730 |
| Dimensions | Standard |
| Core plate | |
| Type | K20 |
| Thickness | 56 |
| Material | 1730 |
| Dimensions | Standard |
| Risers | |
| Type | K40 (standard form) |
| Thickness | 76 |
| Material | 1730 |
| Dimensions | Standard |
| Bottom clamping plate | |
| Type | K11 (overhang in width) |
| Thickness | 36 |
| Material | 1730 |
| Dimensions | Standard |
| Ejector base plate | |
| Type | K70 (standard form) |
| Thickness | 22 |
| Material | 1730 |
| Dimensions | Standard |

< Back Finish Cancel

- Select **Finish** to build the moldbase



From the assembly tree we can see all of the components that have been created.

A very important point to note here is that the standard components do not have power features.

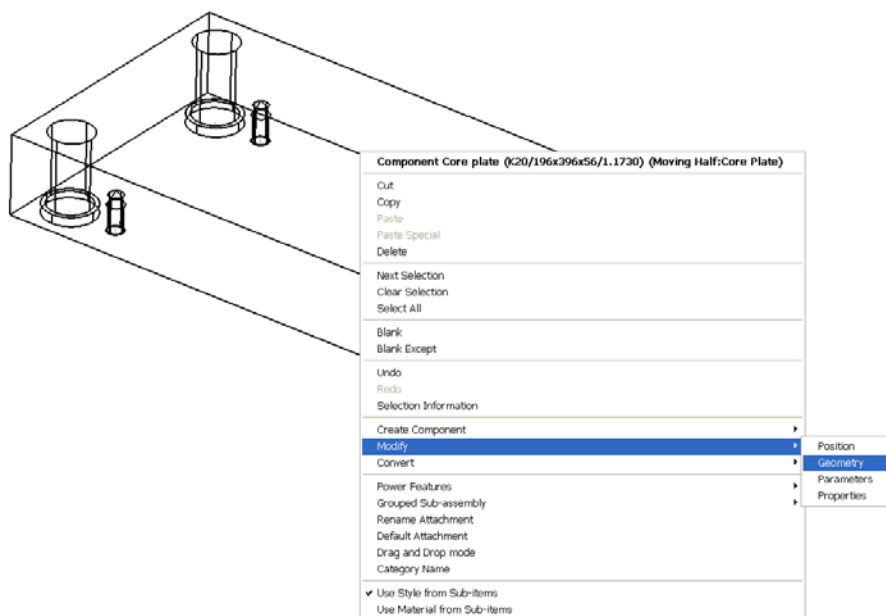
The Power features are omitted from components that are positioned in a standard moldbase

This is based on the fact that a Standard mold base already has the holes etc pre machined, it does not need to have them 'cut' by the standard components.

Add Pockets to core & cavity plates


To fit the core and cavity inserts into the moldbase we need to create pockets, we can do this manually.

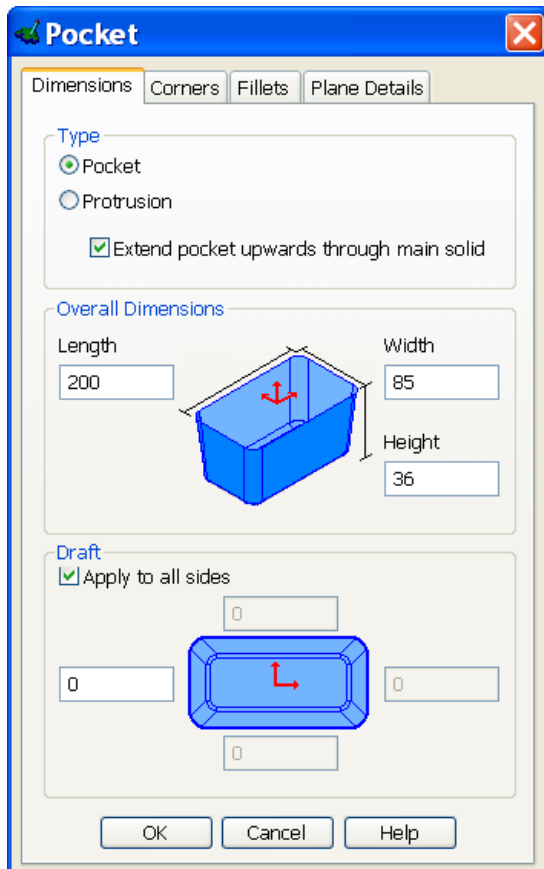
- Using the RMB, Modify geometry of the Core plate



PS-Moldmaker creates all moldbase plates as single instance component automatically; you can see from the RBM menu that there is not an option to modify 'this' or 'all' instances.

Any other items created by the component wizard will be multiple instance components.

- Select the Solid Pocket feature creation 
- Position the pocket in the centre of the plate i.e. XYZ 0

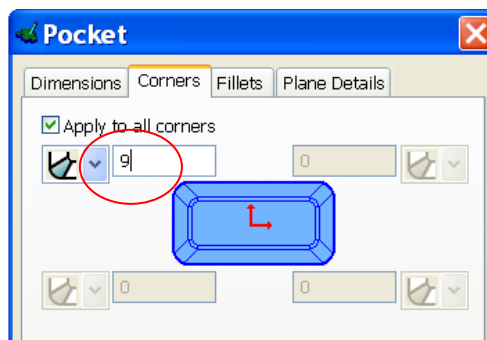


- Set the values of Length, Width and Height to match the Core insert.

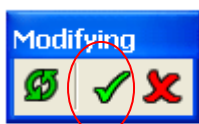
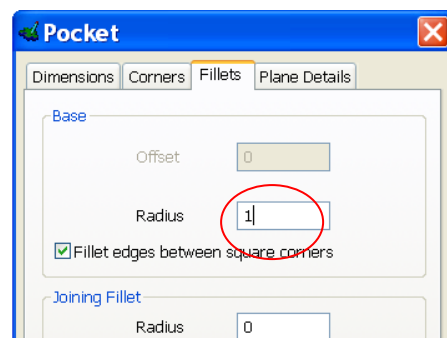
These values can be shared parameters so that the pocket will update if the insert sizes change.

The core / cavity inserts have a 10mm corner radii.

- Set the corners to 9mm to allow clearance

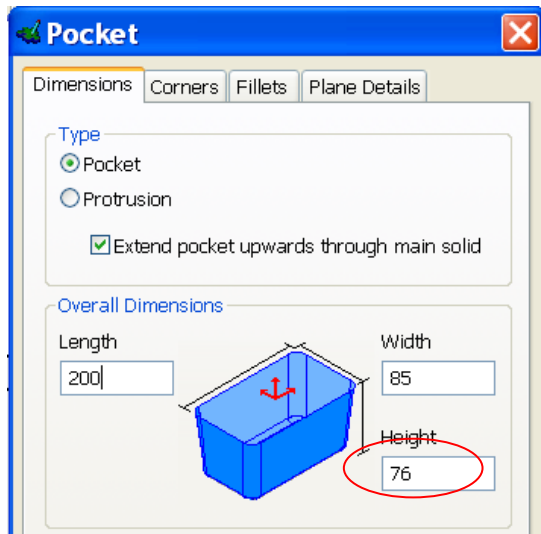


- Add a 1mm radius to the base of the pocket.



- **Accept** the changes

Repeat the operation on the Cavity plate, there are two changes that must be made to the Pocket form.



- Modify the **Height** to match the cavity insert.



- **Accept** the changes

Component Wizard.

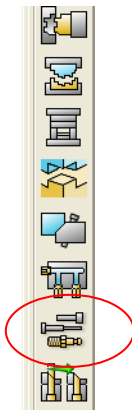
Now that the moldbase is complete the next step is to add any further components that are required for this tool design.

Many of the standard components in PS Moldmaker are fully parametric, they include the following:-

Ejector Pins
Screws
Pillars
Bushes
Support Pillars
Dowels
Sprue Bushes
Stop Pins
Locating Rings

The method for adding components has not changed from earlier versions; however there are some changes to the wizard to allow previewing of new and edited components and new mounting clearances on certain components.

- Start the **Component Wizard** from the Wizard toolbar.



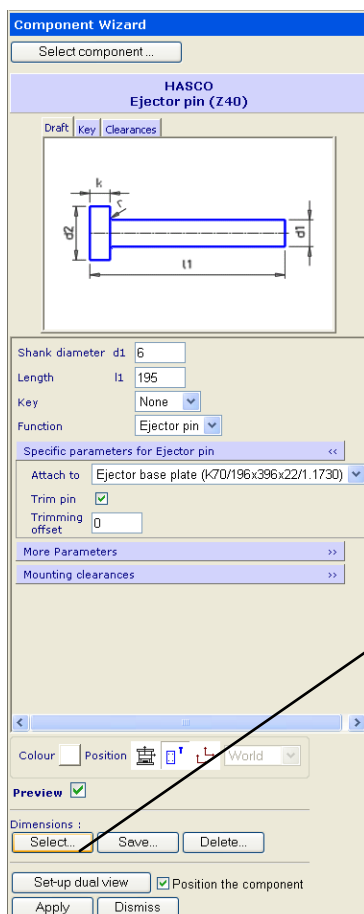
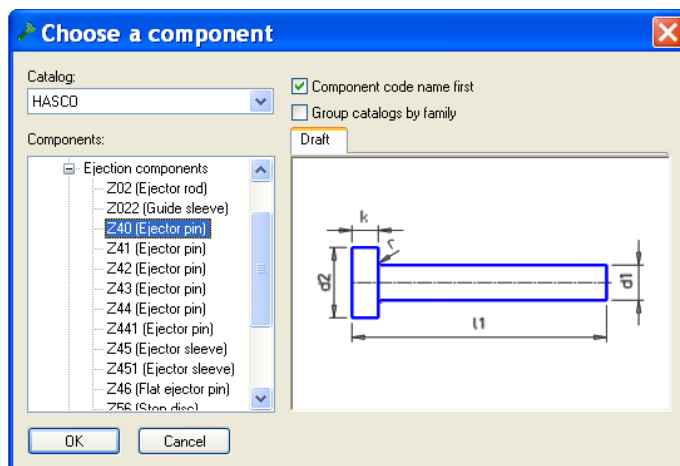
The first components we will add are core pins; there are two different ways of fitting core pins that are supported by the wizard.

Single plate: - When a pin is placed in one component such as an insert to create hole or detail features

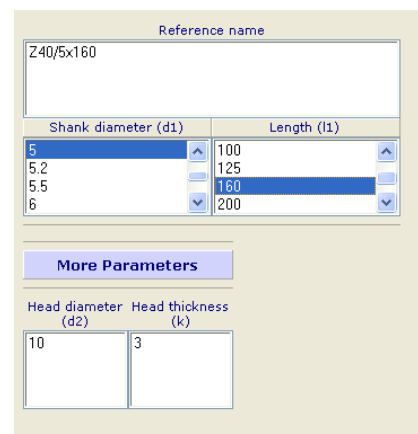
Multiple plate: - When a pin is used with a sleeve ejector for example

The power feature requirements are different so a **fitting** must be specified to attain the correct results

- Select **Hasco Z40 Ejector pin** from the component wizard

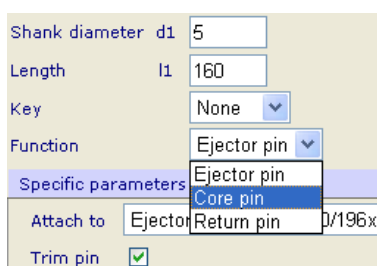


When the form is raised it contains by **default** the settings for an ejector pin.
We can change these settings to suit the **function / fitting** that we require.

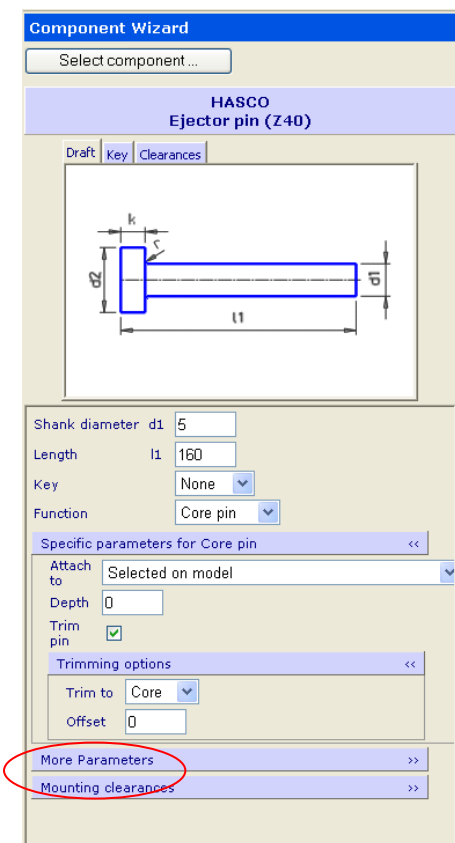


- Select the **standard parameters** as shown

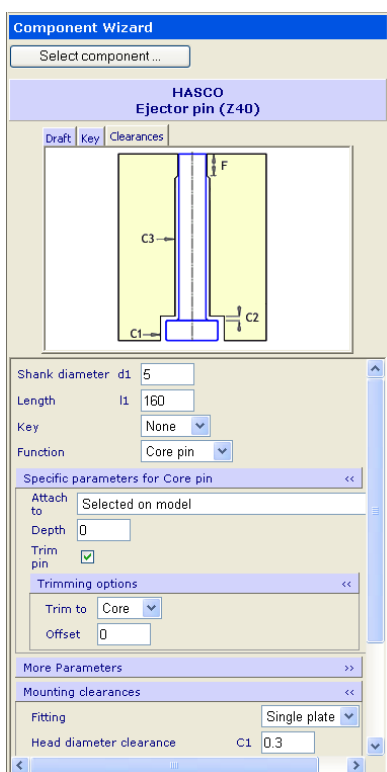
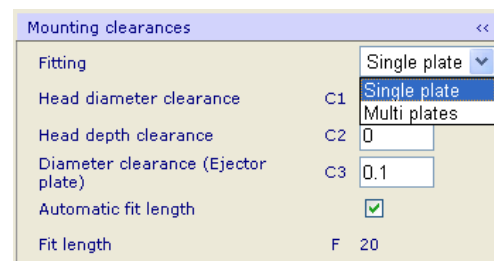
- Next Select the **Core Pin function** from the drop down menu



The options for core pins is now displayed



- Click on **mounting clearances** and check that **single plate fitting** is selected



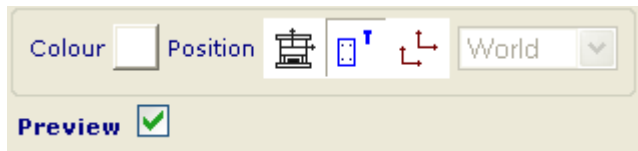
The draft image has changed to display the **fitting** that will be used for this core pin and the clearances that can be modified

Automatic Fit Length

| | |
|----------------------|-------------------------------------|
| Automatic fit length | <input checked="" type="checkbox"/> |
| Fit length | F 20 |

The fit length is calculated based on the diameter of the pin, by un-checking the automatic box the length can be user defined.

Preview Toolbar



A new preview toolbar is added to the component wizard. There are options for:-



Positioning of the preview image relative to a work plane which can be changed on the fly,



Positioning in free space

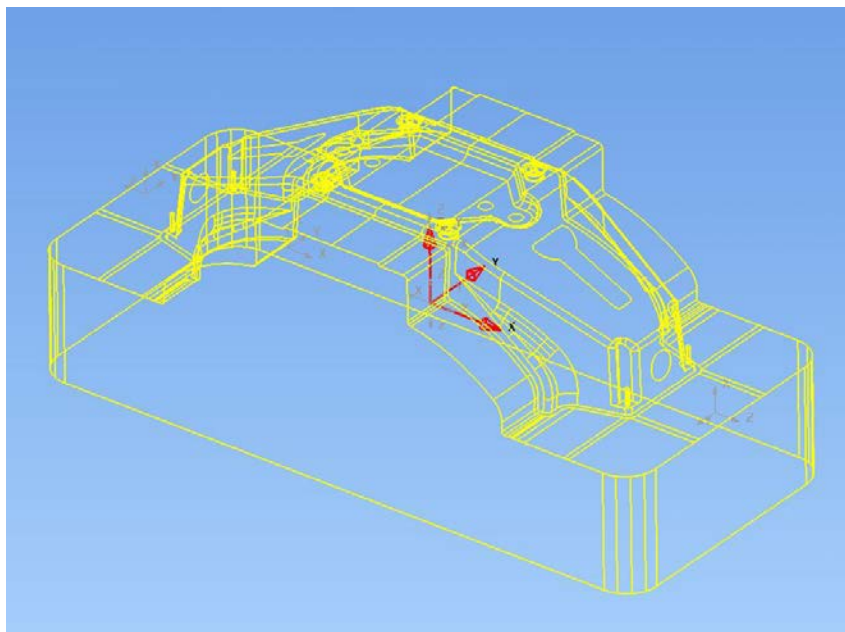


For existing components, the preview can be displayed in the current position.

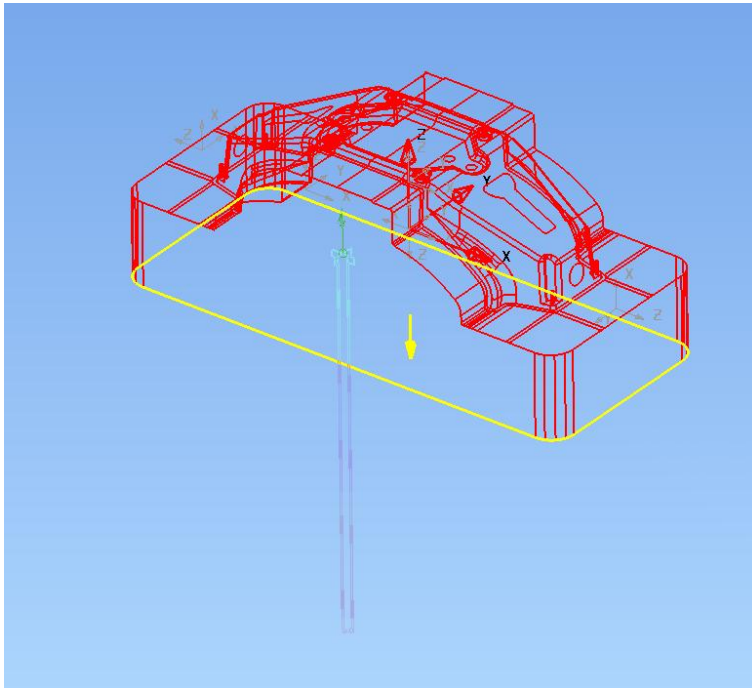
Colours can also be set for the preview images.

Preview can be turned off if not required

- Select the core insert component and hit **Apply**



- Select the **master attachment** on the base of the core insert



You can see that the pin is positioned in the wrong direction; this can be changed by using the toggle button on the bolt command toolbar.

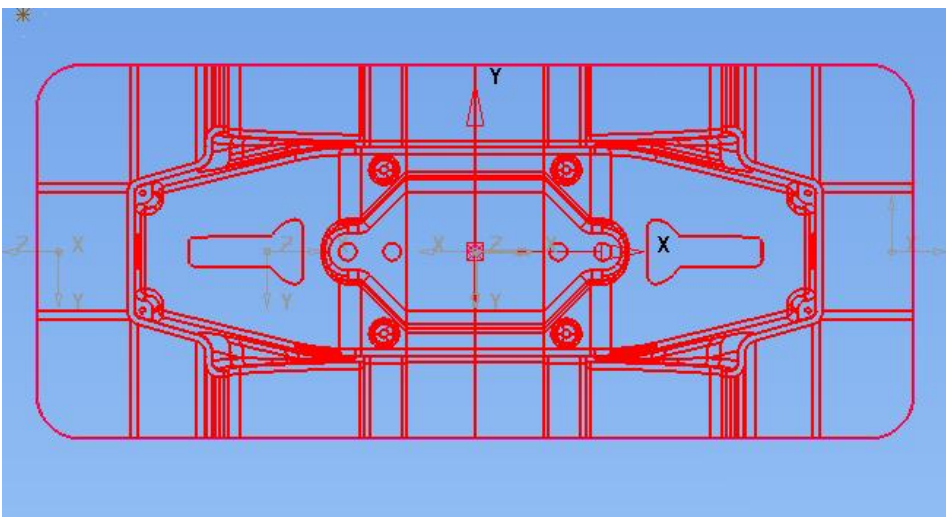


- Align

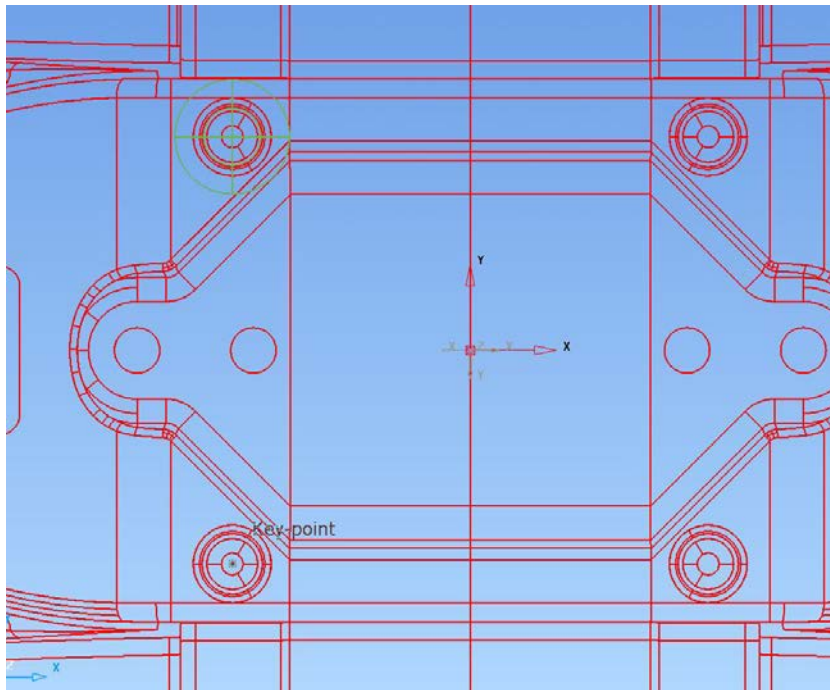


- Anti Align

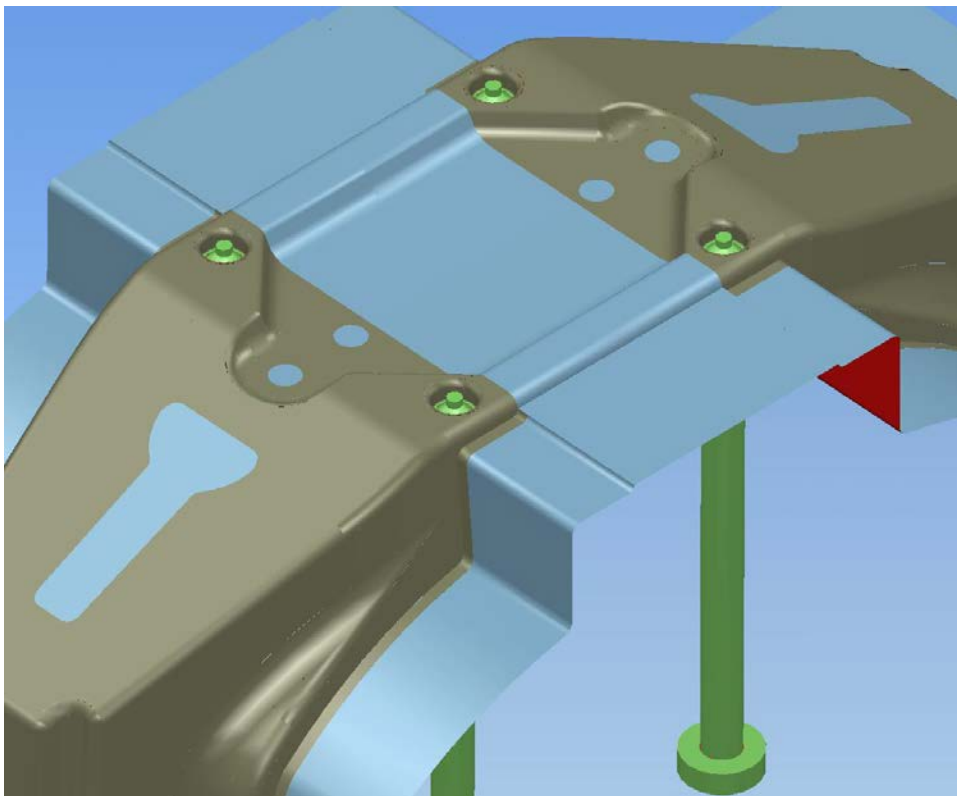
- View the model in **Z direction**



- Position the pins on the **Key points** that can be found on the round features



- **Dismiss** the form

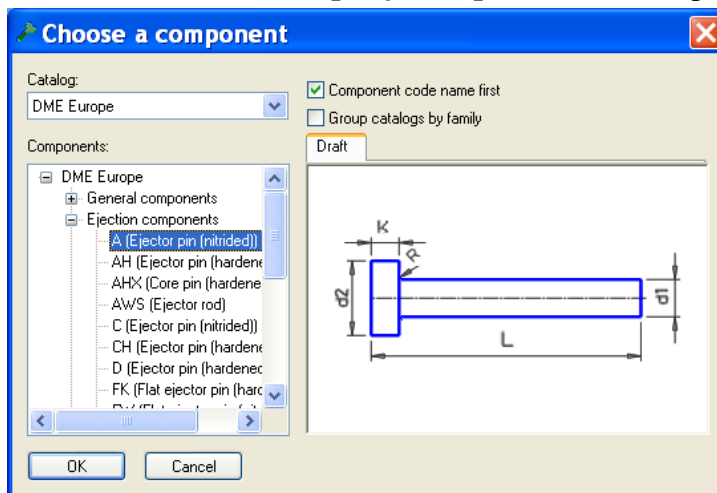


- Display the model in **shaded mode**

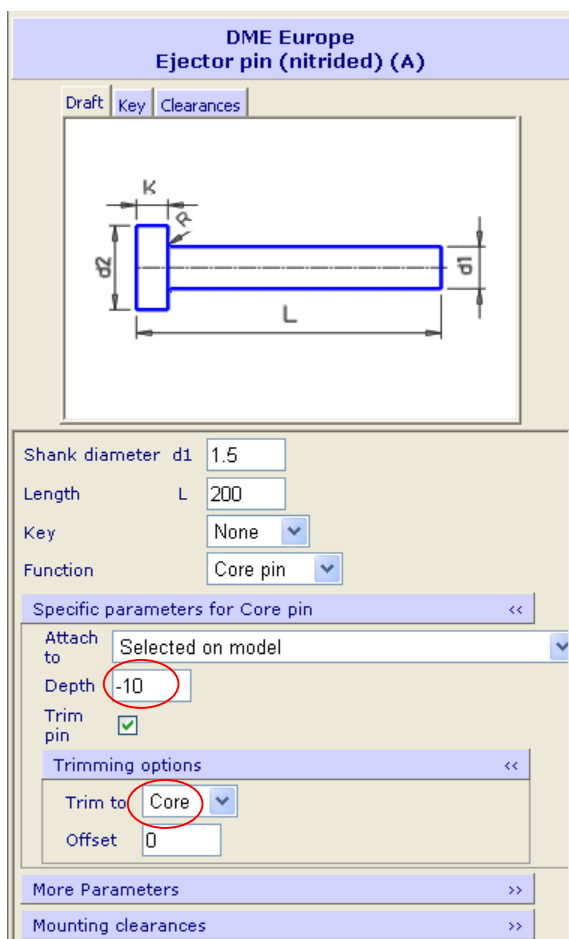
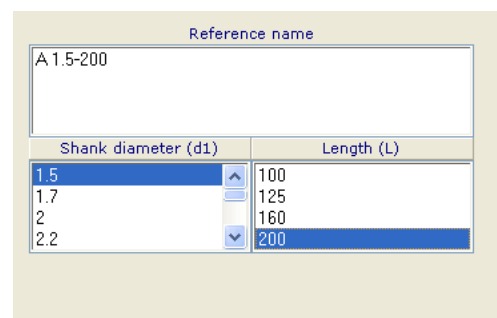


Next you will add the core pins that will form the internal holes for the screw bosses.

- Select **DME Europe ejector pin A** in the component wizard



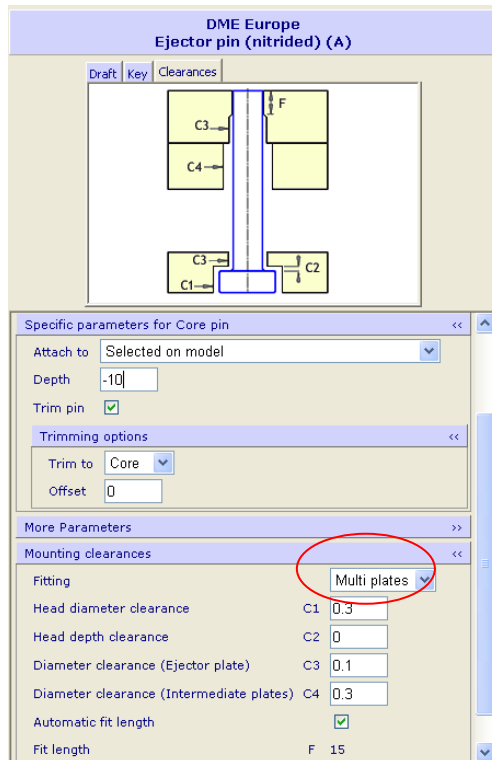
- Pick the **parameters** indicated



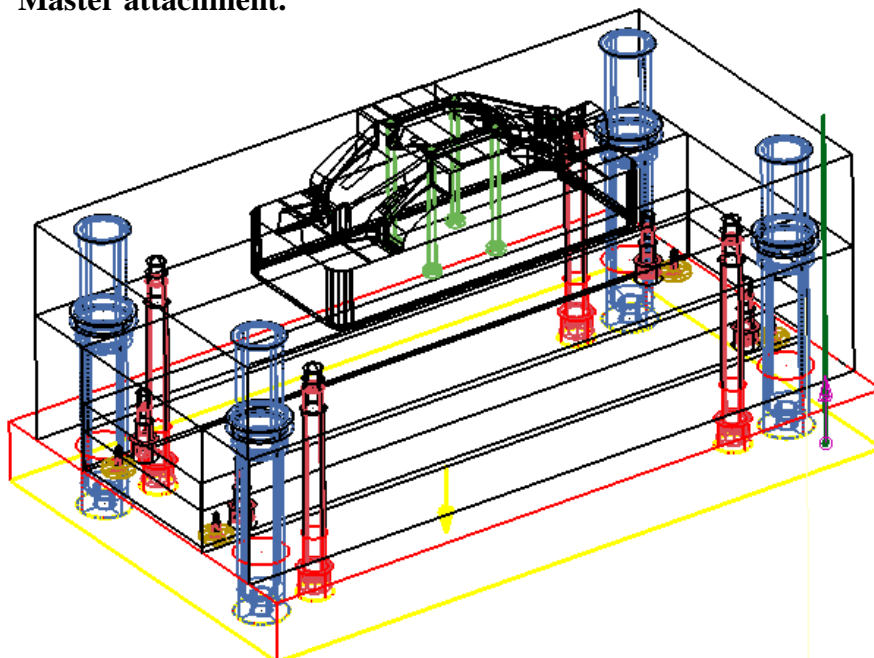
- Set the attachment **depth** to be -10mm
- Check the trimming option is set to **core**

- Open the **mounting clearance** parameter display and check that the **multi plates** option is selected

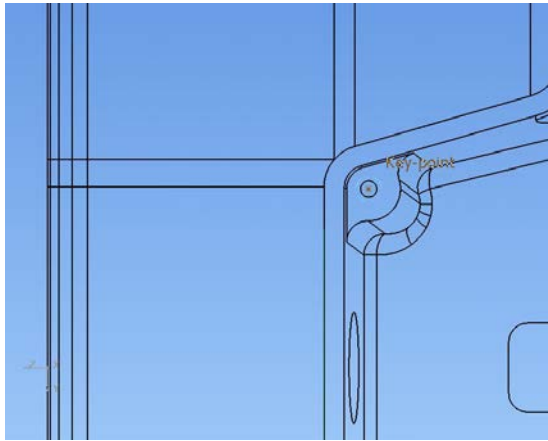
The draft indicates the clearances that will be used in each plate



- Select the **Bottom Clamping plate** and hit **Apply**, Pick the bottom face as the **Master attachment**.

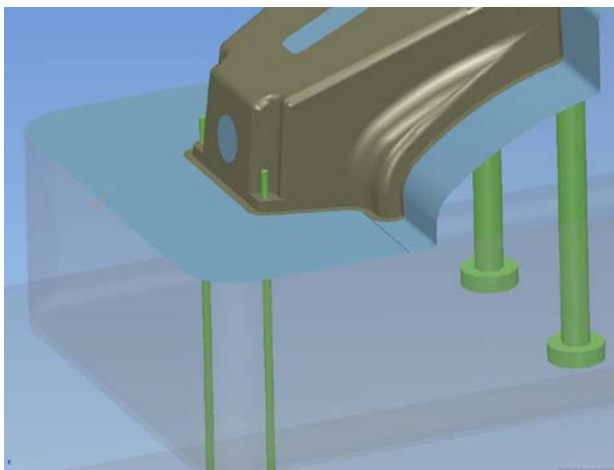


- Position the core pins using the **Key points** on the 'boss' features



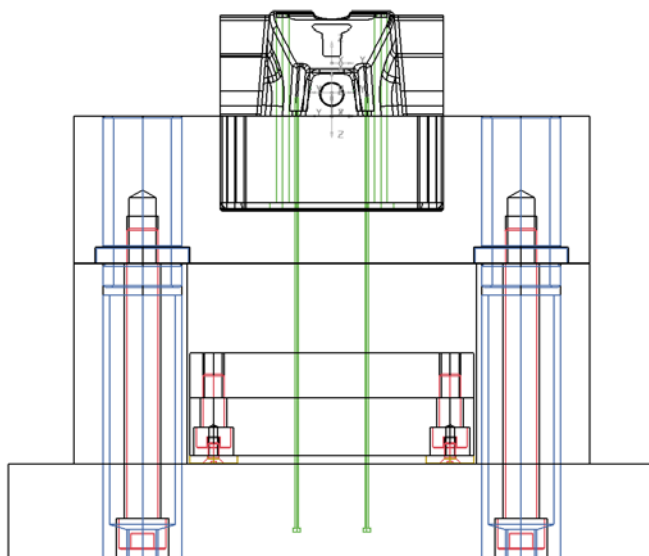
Be careful to snap to the correct Key Point

- **Dismiss** the form once to **Trim** the pins



A Shaded view clarifies the pin trimming

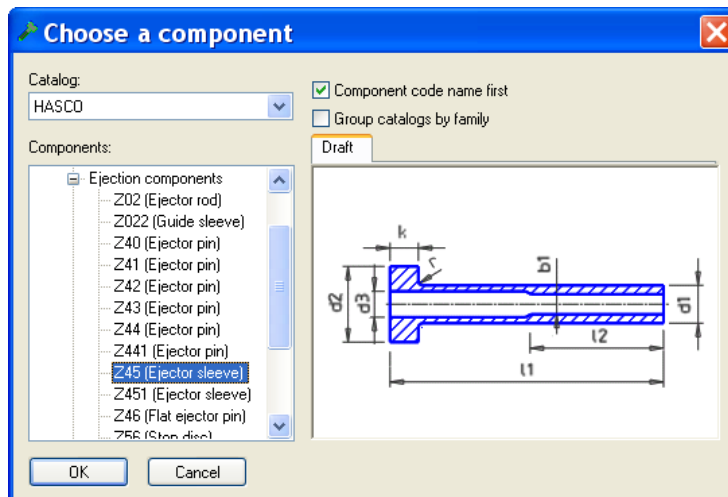
- **Dismiss** to close the form.



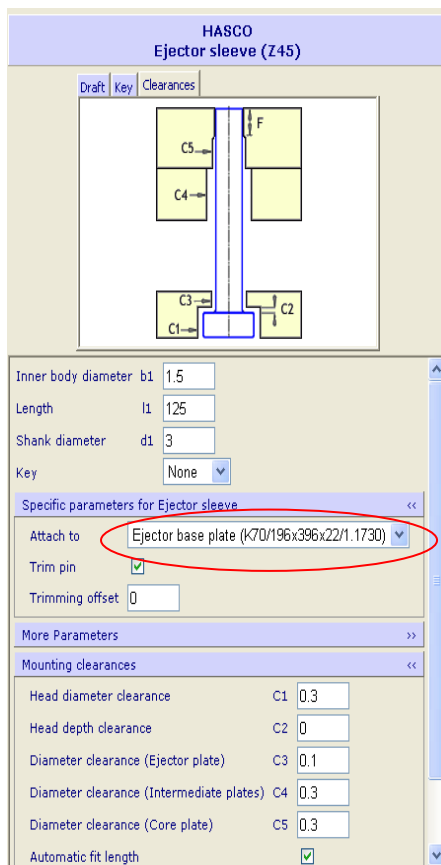
The pin positioning can be verified in an X axis view direction

Now that the core pins are in position you can add some Sleeve ejector pins over the top.

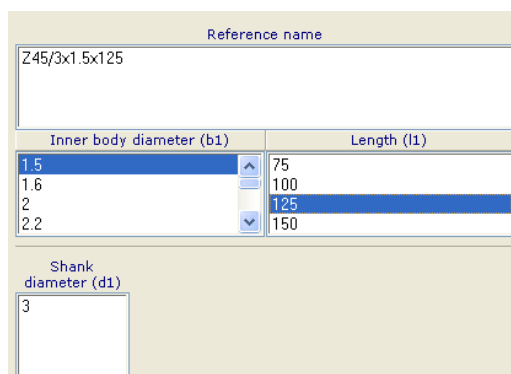
- Select the Hasco **Z45 Ejector Sleeve** from component wizard



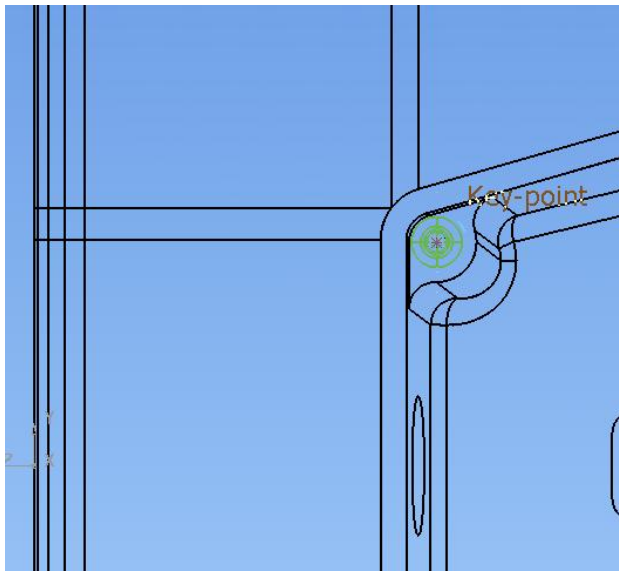
- Pick the **parameters** indicated



- Note that the ejector plate is selected **automatically**, but can be changed if a two stage ejector system is used



- Snap the pins to the **key pints** on the boss features



Be careful to snap to the correct Key Point

- **Dismiss** the form once to **Trim** the pins
- **Dismiss** again to close the form

Keyed Ejector pins

So far the ejection and core pins that have been added have not included any key / timing methods.

In PS Moldmaker 6.0 there are two options of keying pins, both methods are based on positioning a flat on the head of the pin and creating holes / features to suit in the ejector plates.

In the case of a core pin the holes / features are applied to the plate containing the pin head.

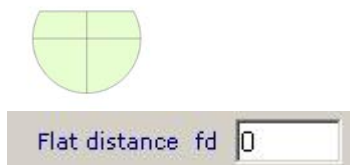
The keying methods are selected using a drop down menu in the component wizard. Until now you have left this set to the default setting of None.



The new options available are utilised by selection from a drop down menu



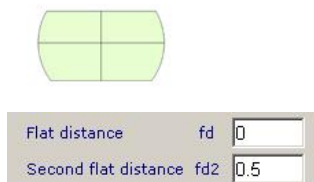
Single Option



The single option has one flat on the head that is positioned relative to the diameter of the pin.

A parameter named fd is used for the flat position.
fd signifying flat distance

Double Option

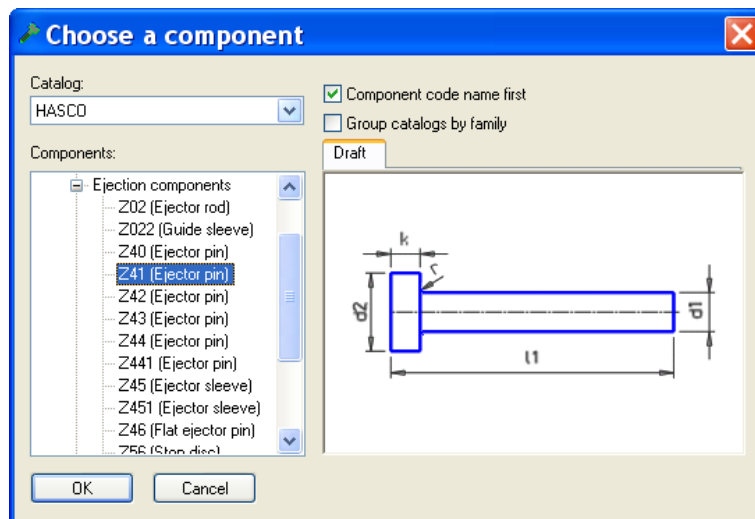


The double option has two flats on the head; both are positioned relative to the diameter of the pin. One must have a different value so that the pin cannot be inserted 180 degrees out of position.

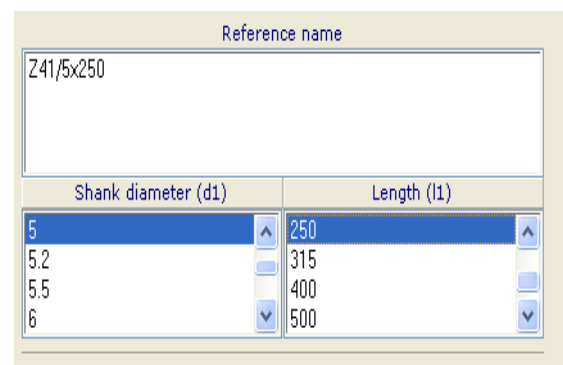
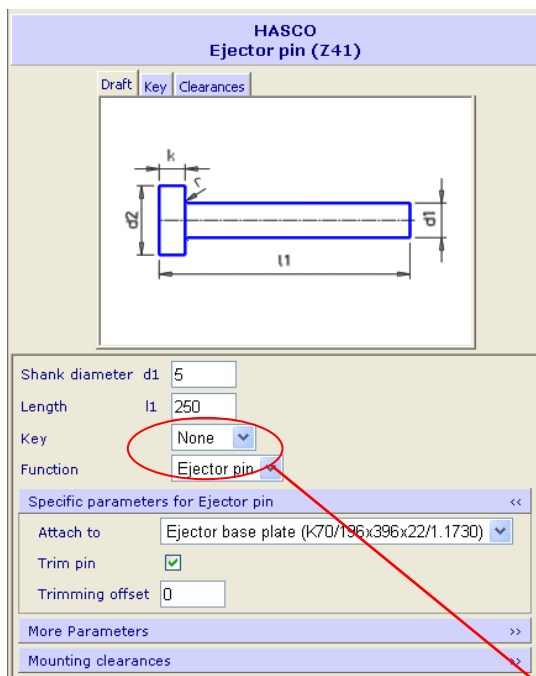
Parameters named fd and fd2 are used for the flat position.

The next section uses keying for the main ejector pins.

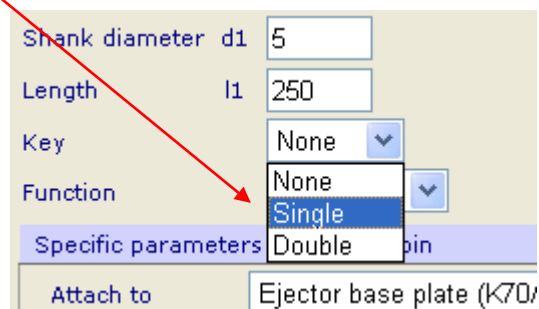
- Select **Hasco Z41 Ejector Pin** in the component wizard



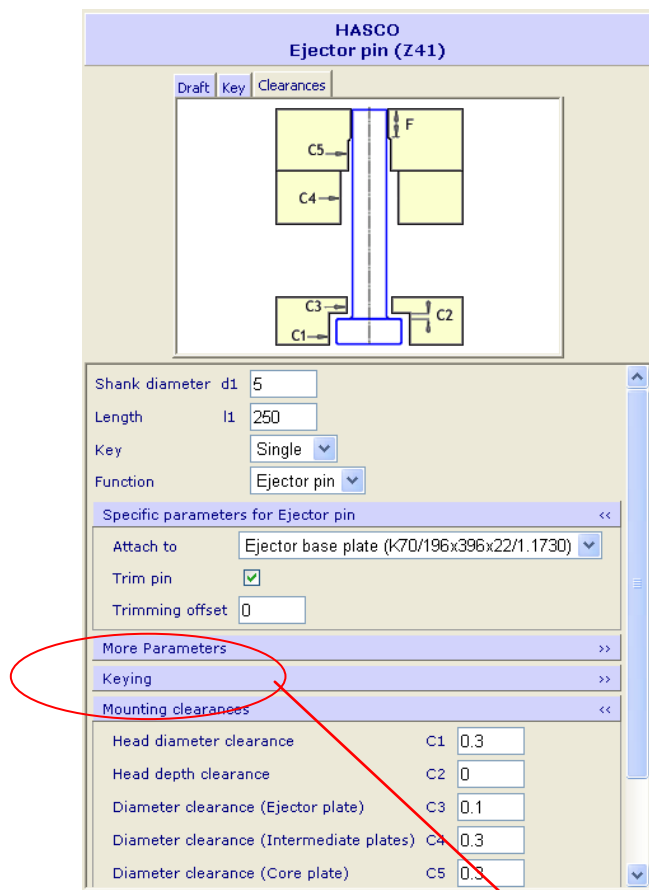
- Pick the **parameters** indicated



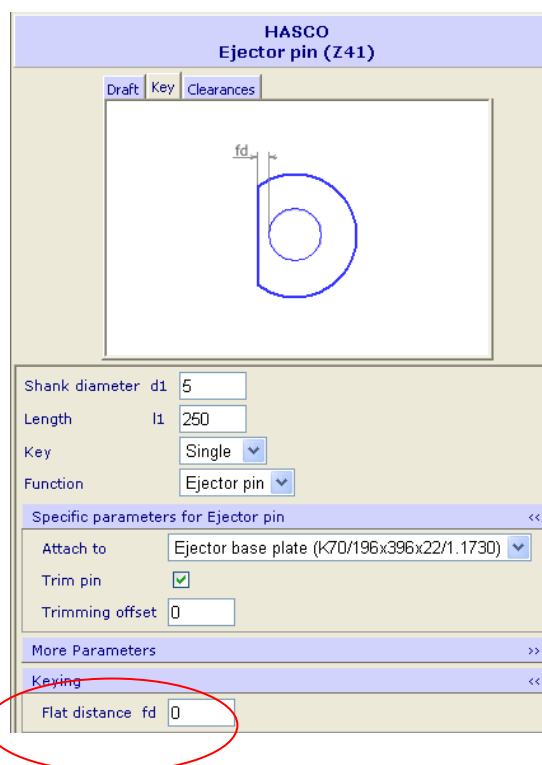
- Select the Key option **Single**



Once the Key option is chosen the mounting clearances can be checked and modified if required. There is no clearance parameter between the Flat on the ejector pin head and the pocket feature created in the ejector plate



- Click the **Keying** option and the draft image will change

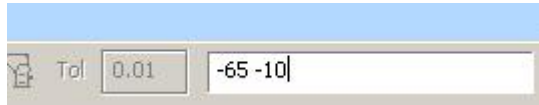


The flat distance (fd) can be changed, the default for single is zero

When all of the desired options / settings are correct; you can proceed to create the pins.

- Select **Apply**

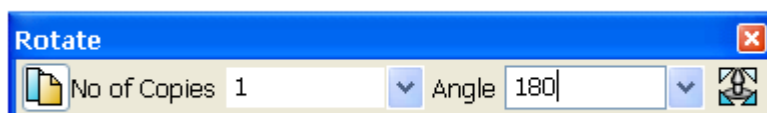
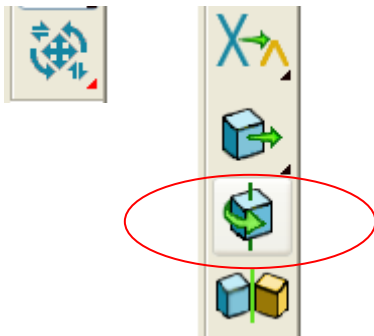
In this case we can use the input box to type coordinates for positioning the pins



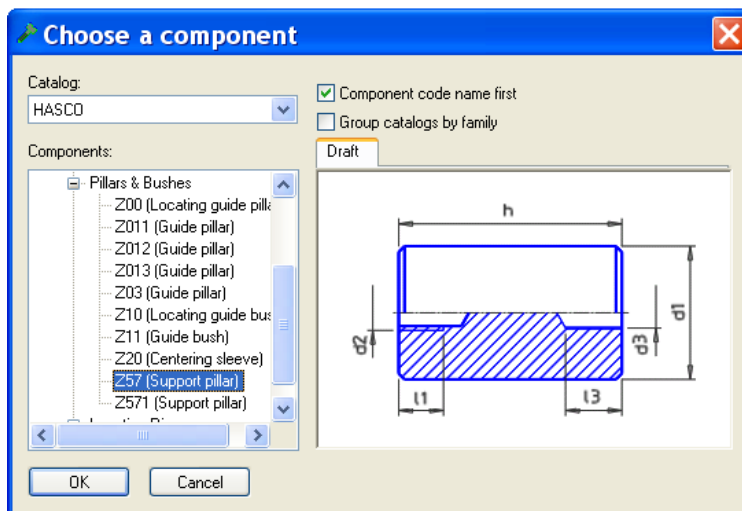
Use the following coordinates:-

| | |
|---------|---------|
| X -65.0 | Y -10.0 |
| X -65.0 | Y 10.0 |
| X -48.0 | Y 14.0 |
| X -48.0 | Y -14.0 |
| X -32.0 | Y 15.0 |
| X -32.0 | Y -15.0 |
| X -20.0 | Y 5.0 |
| X -20.0 | Y -5.0 |

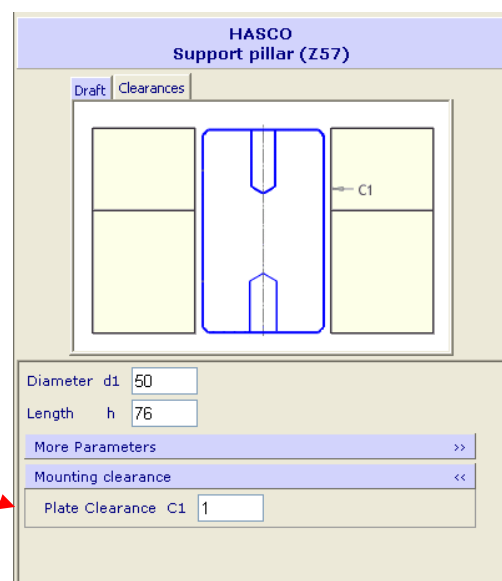
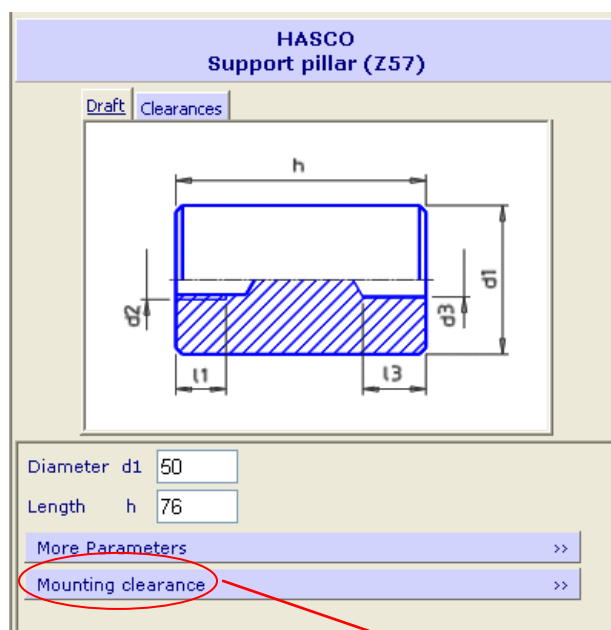
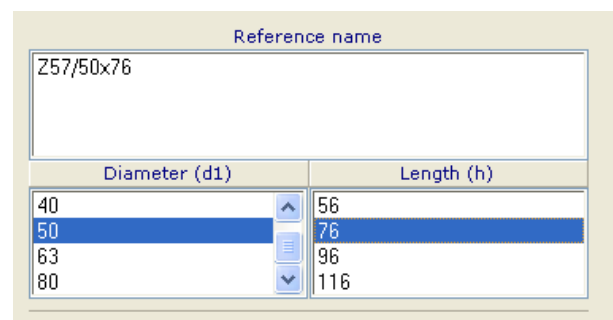
- Click **Dismiss** to trim the pins
- **Dismiss** the form to close
- Now Select the new pins and **Rotate with copy** around the **Z axis 180 degrees**



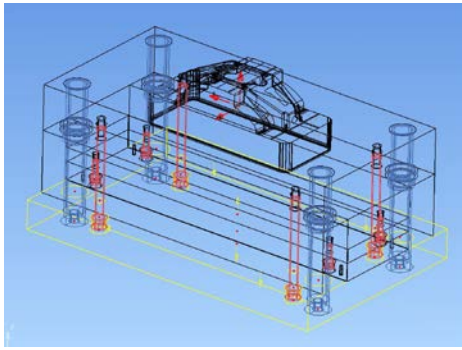
- Select Hasco **Z57 Support Pillar** in the component wizard



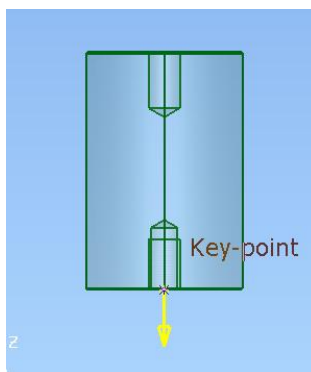
- Pick the **parameters** indicated



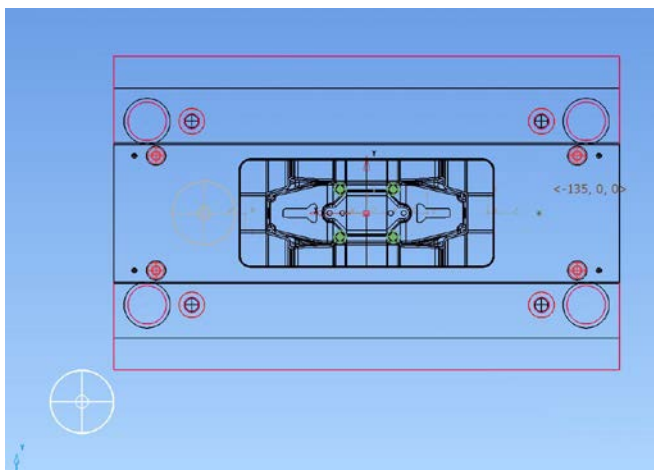
- Check the clearance parameter and modify if needed
- Select the **Bottom Clamping Plate** and hit **Apply**



- Select the **Slave attachment** on the **base of the pillar**

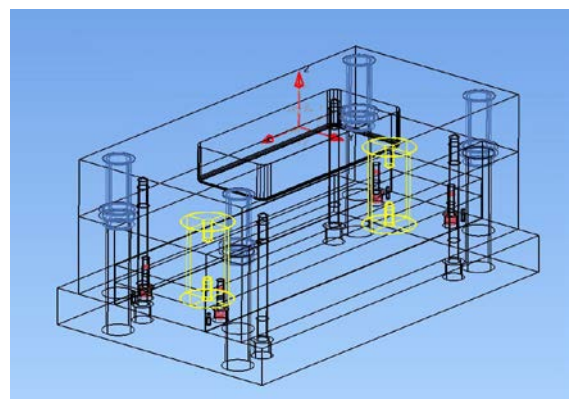


- Select the **master attachment** on the inside face of the **Bottom Clamping Plate**

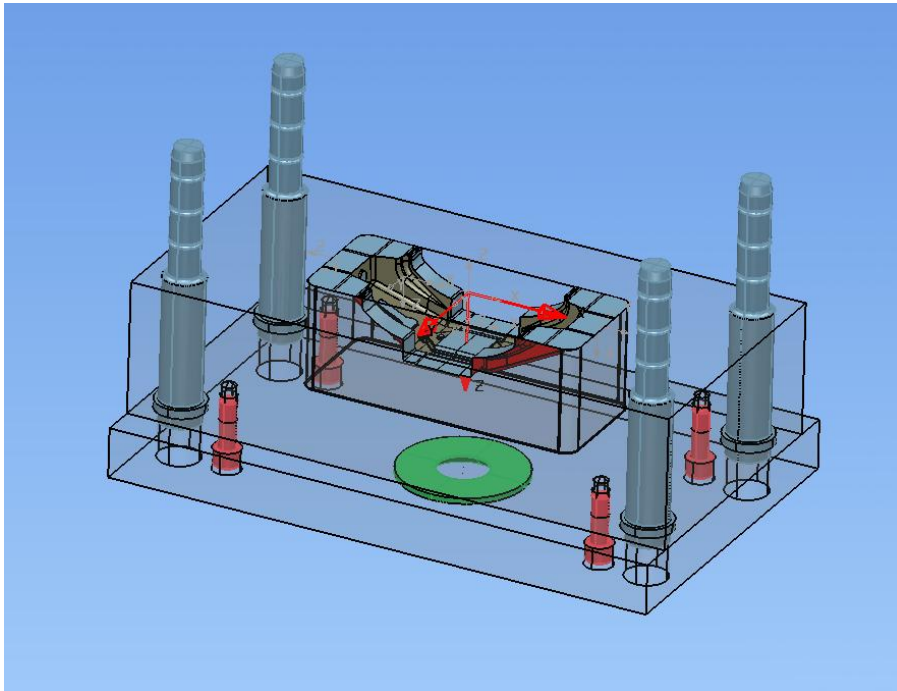


- Position the pillars X 135 Y 0
 X -135 Y 0

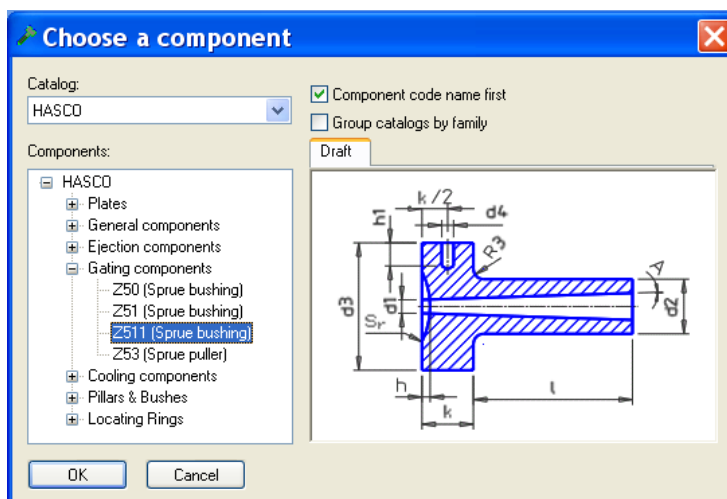
- **Dismiss** the form to close
Next you can add the sprue bush



- Display only the **fixed half levels group** and the **cavity insert**



- Select the **Hasco Z511 sprue bushing**



- Pick the **parameters** indicated

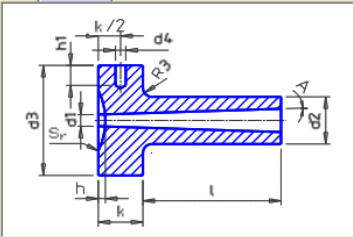
| Reference name | | |
|---------------------|------------------|-----------------------------|
| Z511/18x76/3/15.5 | | |
| Shank diameter (d2) | Shank length (l) | Sprue opening diameter (d1) |
| 18 | 56 | 3 |
| | 76 | 4 |
| | 96 | |
| | 116 | |
| Sphere radius (Sr) | | |
| 0 | | |
| 15.5 | | |
| 40 | | |

Component Wizard

Select component ...




HASCO Sprue bushing (Z511)

Draft | **Clearances**



Shank diameter d2 18
Shank length l 76
Sprue opening diameter d1 3
Sphere radius Sr 15.5

More Parameters >>
Mounting clearances >>
Number of copies 1

Colour Position    World

Preview ☒

Dimensions :
Select... Save... Delete...

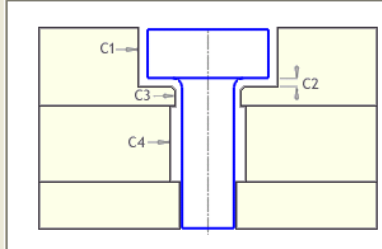
Set-up dual view ☐ Position the component
Apply Dismiss

The Wizard display the parameters and options

- Click on the **mounting clearances** tab

HASCO Sprue bushing (Z511)

Draft | **Clearances**



Shank diameter d2 18
Shank length l 76
Sprue opening diameter d1 3
Sphere radius Sr 15.5

More Parameters >>
Mounting clearances <<

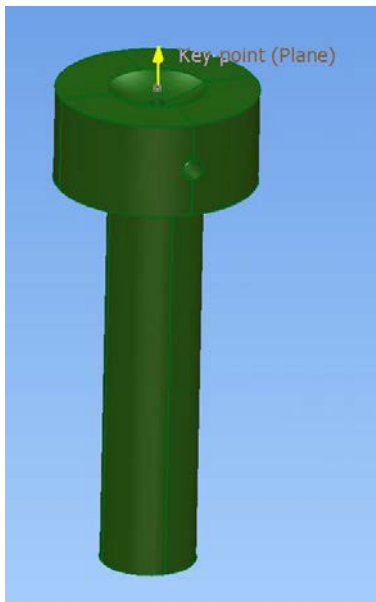
Head diameter clearance C1 1
Head depth clearance C2 0
Clamping plate clearance C3 0.3
Plate clearance C4 0.5

Number of copies 1

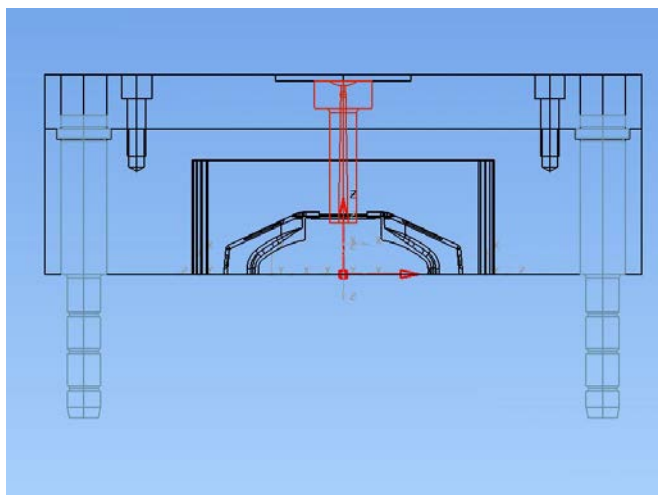
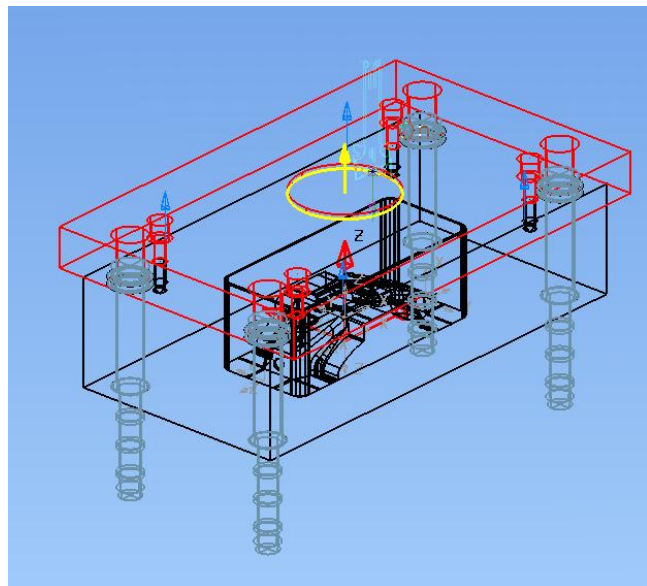
- Check the **mounting clearances** are correct
No adjustment is required in this case

- Select the **Top Clamping Plate** and hit **Apply**

- Select the **Slave attachment** on the top of the head of the sprue bush



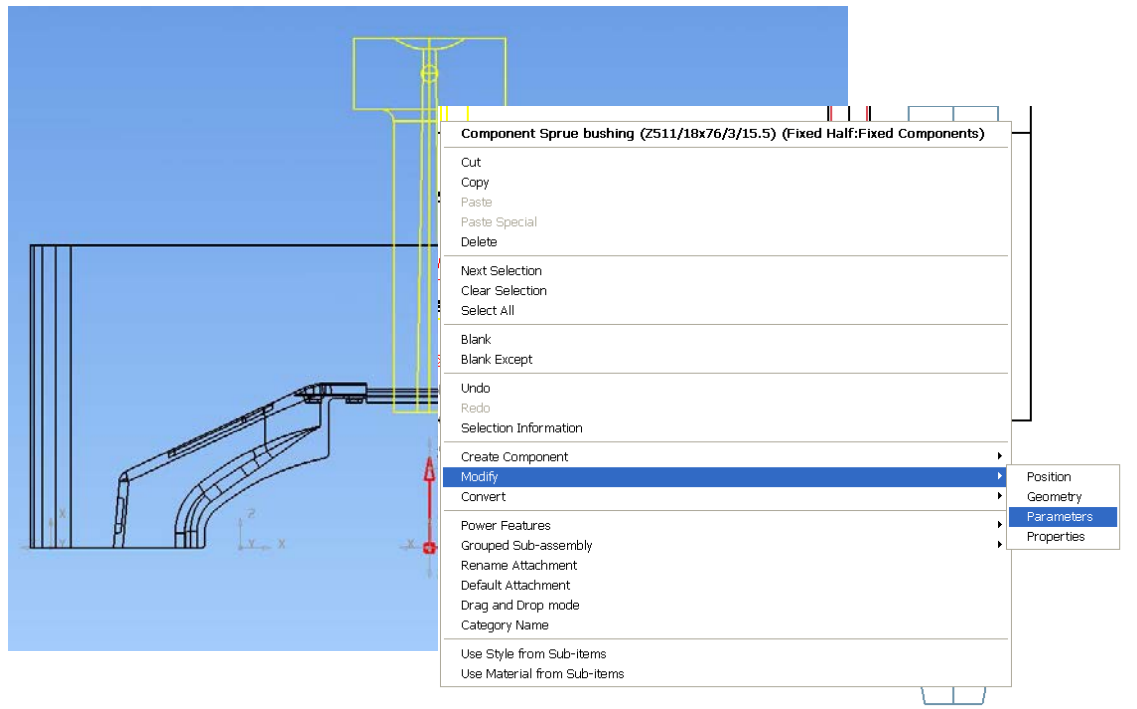
- Select the locating ring recess plane as the **master attachment**



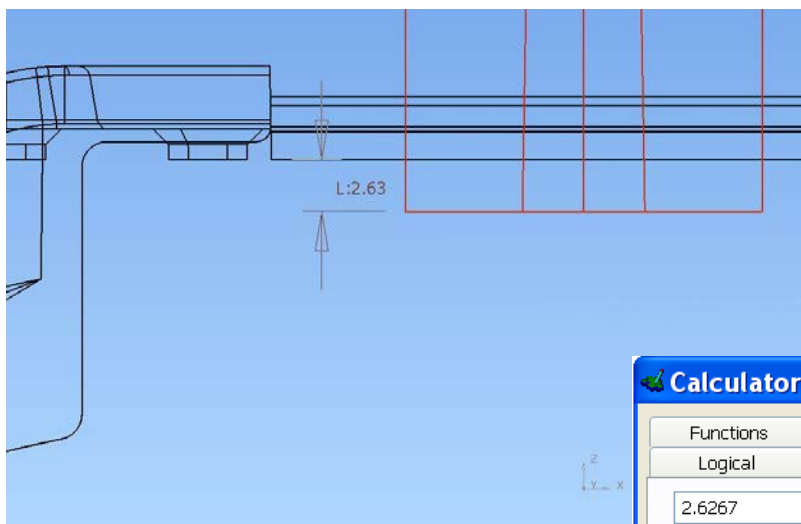
- View the model in the **Y axis** direction

The sprue bush is too long and further adjustment is required

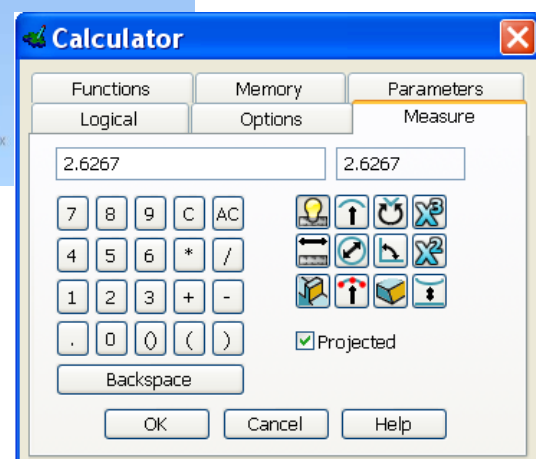
- From the **RMB** menu select **Modify Parameter ALL instances**



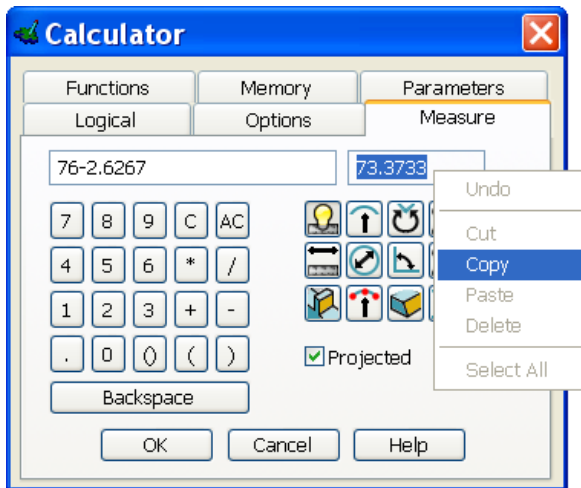
- Raise the **calculator** from the toolbar at the bottom of the window



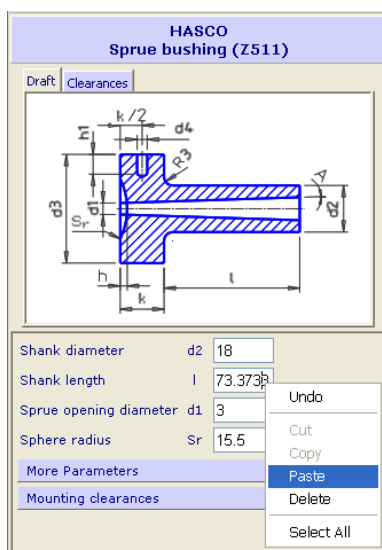
- Use the **measure function** to dimension the sprue bush excess



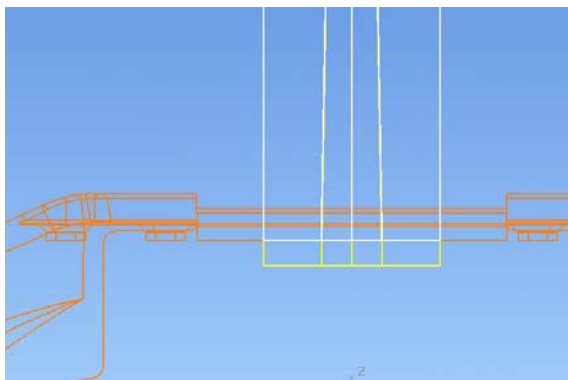
We know that the sprue bush has a shank length of 76mm from the parameter form; we can use this to calculate the new parameter value.



- Subtract the measured dimension from the shank length.
- **Copy** the resultant value using **RMB** menu



- **Paste** the value into the parameter field in the component wizard form

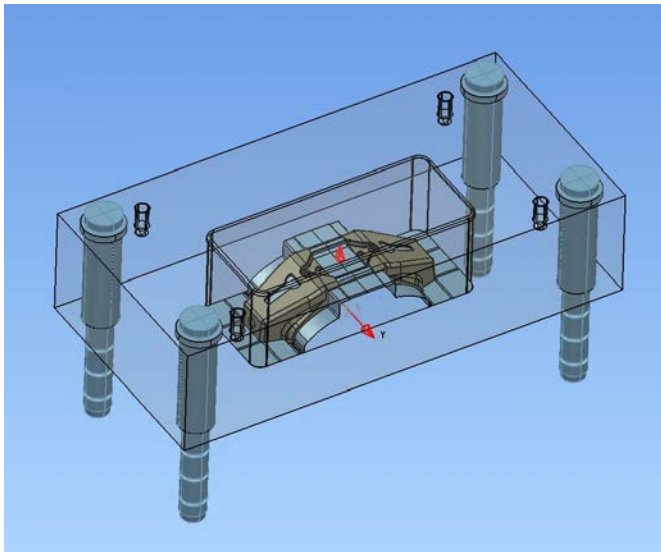


The new length is displayed in preview

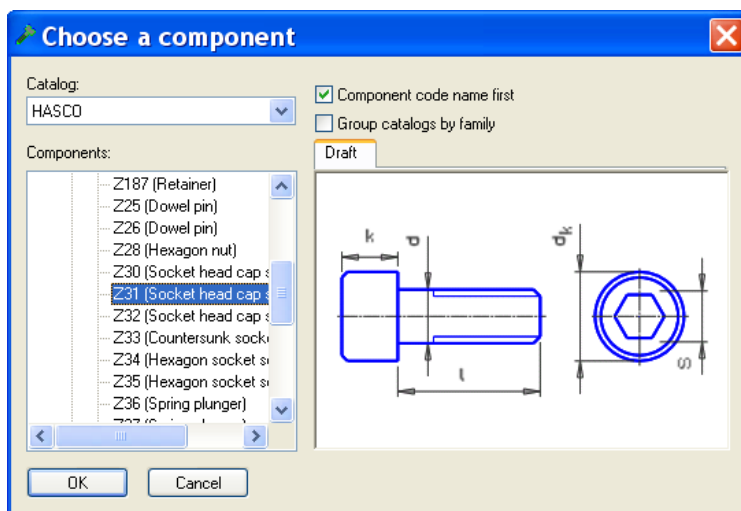
- **Apply** then **dismiss** the form.

To retain the core and cavity inserts in the pockets, cap screws will be used.

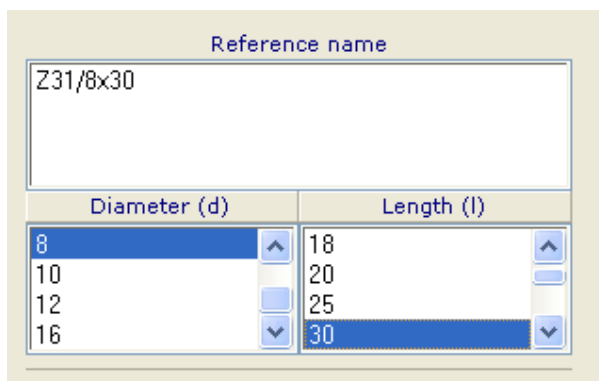
- Display only the **cavity plate** and **cavity insert**



- Select the **Hasco Z31 Socket Head Cap Screw**



- Pick the parameters for **M8 x 30** long



Set the Depth to 9mm, this positions the screw head 1mm below the surface of the plate to allow for any variation in the screw manufacturing process

Component Wizard

Select component ...

HASCO
Socket head cap screw (Z31)

Draft | **Clearances**

Diameter d 8
Length l 30

Specific parameters for Generic screw <<

Attach to Selected on model
Depth 9

Component Wizard

Select component ...

HASCO
Socket head cap screw (Z31)

Draft | **Clearances**

Diameter d 8
Length l 30

Specific parameters for Generic screw <<

Attach to Selected on model
Depth 9

More Parameters >>
Mounting clearances <<

Automatic clearances ☒

Head diameter clearance C1 1
Head depth clearance C2 0
Shank diameter clearance C3 0.8
Plate clearance C4 0.8

In the mounting clearances part of the form a check box can be seen for automatic clearances.

The automatic clearance values calculated are derived from a standard for clearance holes relating to screws.

If the Automatic clearances are unchecked then the user can input their own values

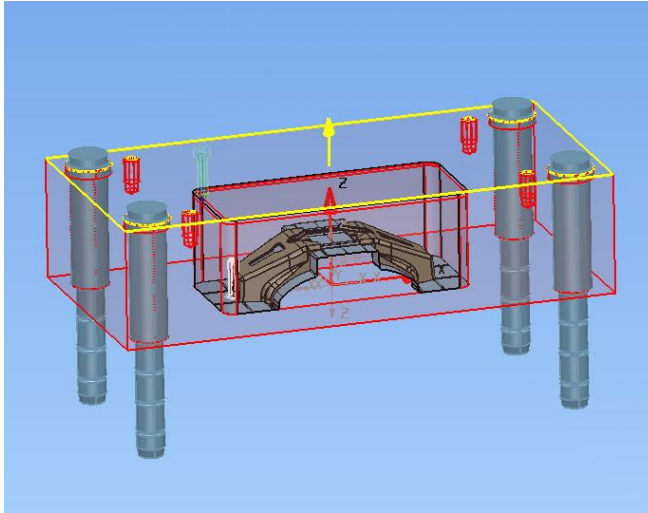
Different default values are displayed for Metric and Inch units

Mounting clearances <<

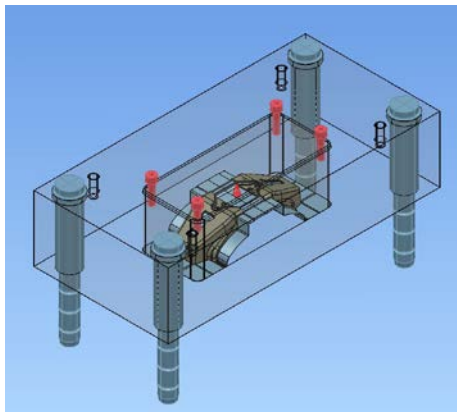
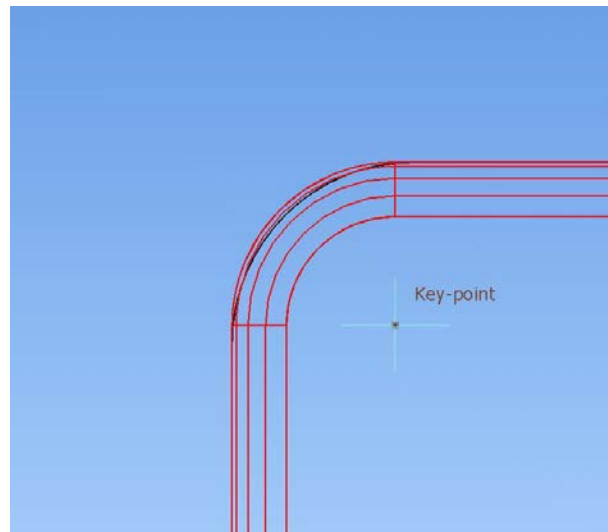
Automatic clearances ☐

Head diameter clearance C1 0.5
Head depth clearance C2 0
Shank diameter clearance C3 0.5
Plate clearance C4 0.5

- Select the **cavity plate** and **Apply**
- Pick the back face of the cavity plate as the **master attachment**



- Position the screws on the **Key point** relating to the corner radius of the **Cavity insert**.

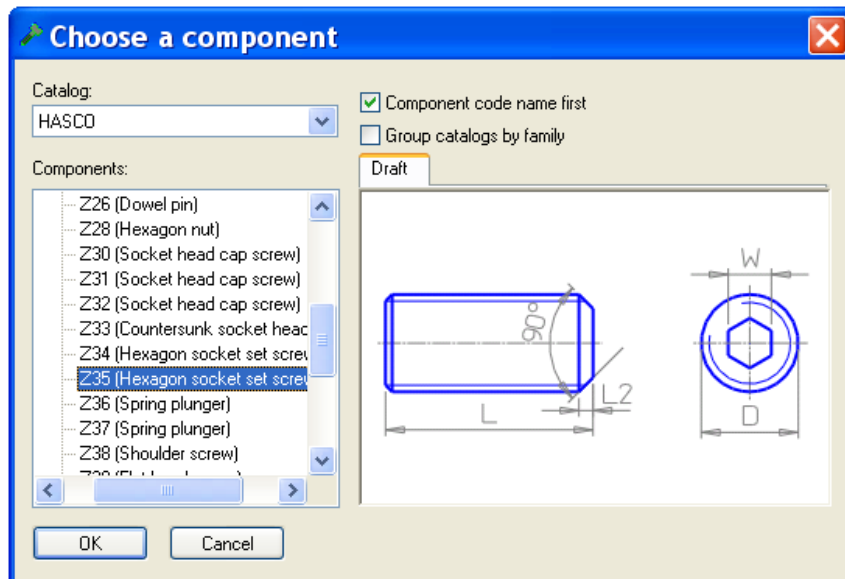


- Add screw to the core plate / insert and then add two M10x40 screws to the support pillars.

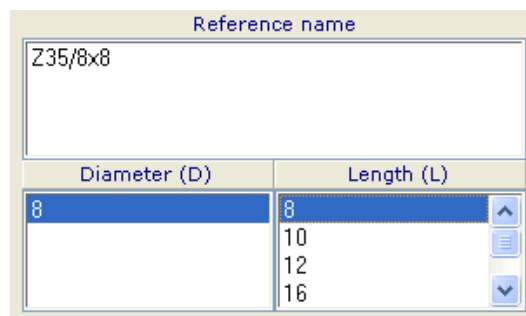
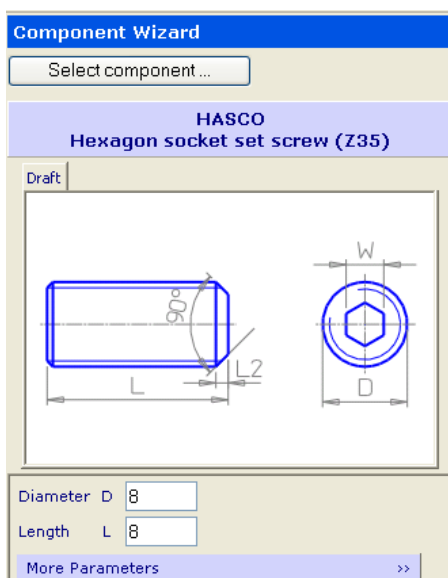
- **Generate power features** for the Assembly, use **RMB** on the Assembly Name **Moldbase**

To retain the core pins in the bottom clamping plate we will use hexagon socket set screws.

- Select the **Hasco Z35** component in the wizard

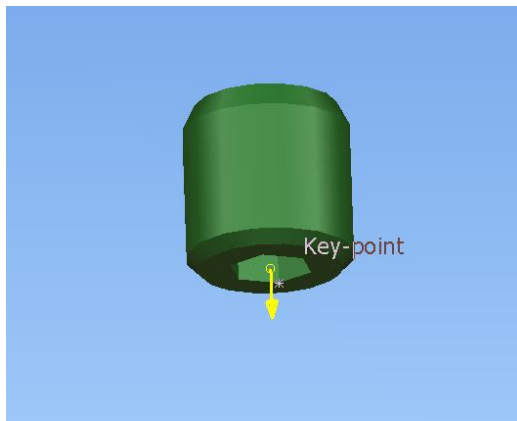


- Select the **parameters** indicated

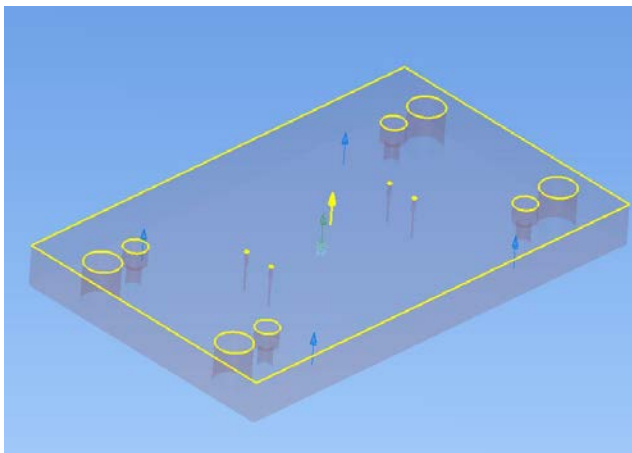


The set screw has only a screw tapped hole for it's power feature so there are not any clearance Parameters to display

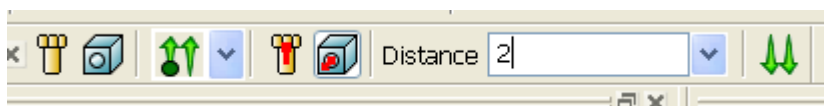
- Select the **Slave attachment** on the set screw as shown below



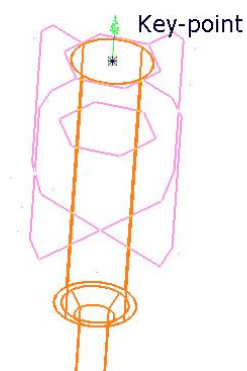
- Select the **Master attachment** on the Bottom Clamp Plate as shown below



- Set the **relation distance** to 2mm



- Position the screws using the **Key points** on the holes created by the **core pin power features**.



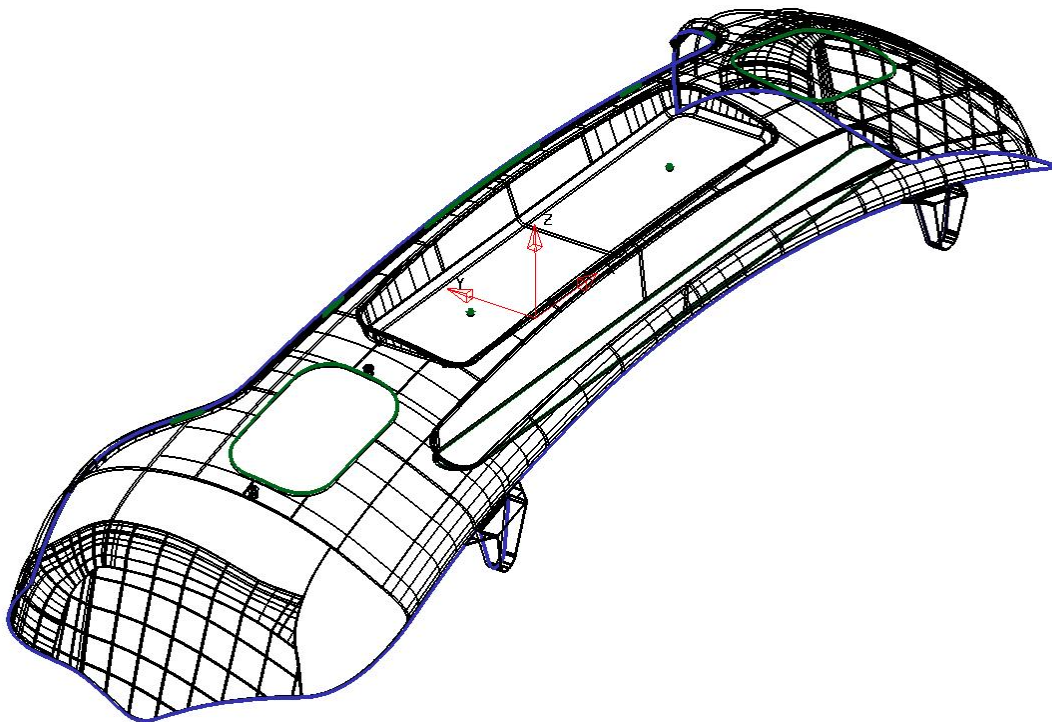
- Select the 4 Set screws and using **RMB generate power features**

10. Moldbase Wizard - Custom.

When Creating a Moldbase with Moldbase Wizard there are now several methods that can be used to increase the flexibility when designing custom and very large tools.

In this exercise we will build a tool design based on a Generic Metric Plain plate moldbase, adding any components via Component Wizard.

Open the model – Moldbase2.psmodel

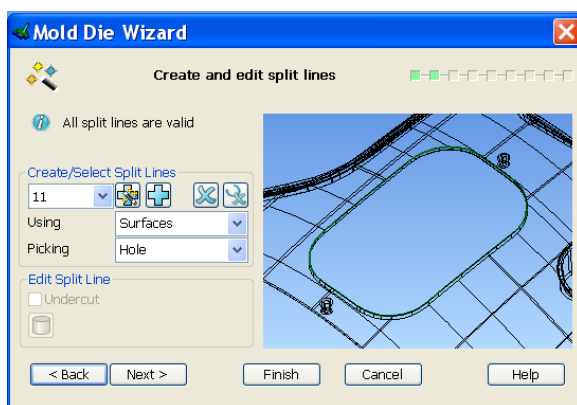
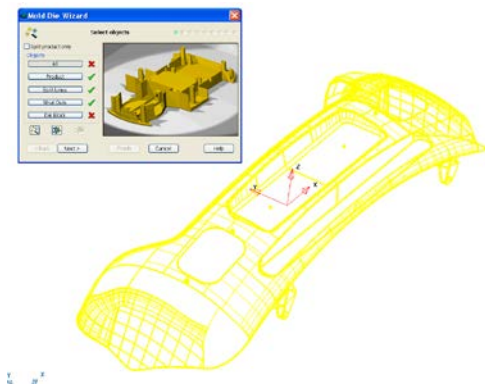


This exercise will use an alternative method of creating direct in plates die blocks. The Cavity/Core wizard can be used but in version 6.0 the plates are not parametric.

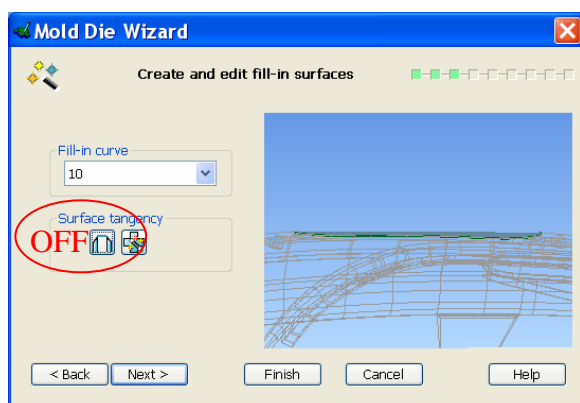
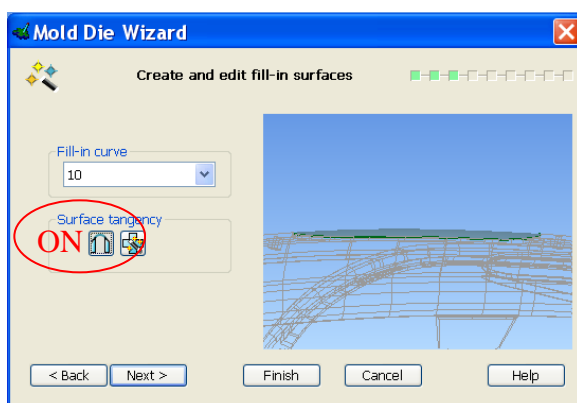
With some manual work we can make a moldbase with some parametric editing allowed on the core / cavity plates.

The first stage is to use Die Wizard to produce the Solid 'shells' that will be used in the moldbase.

- Select the **part / curves and split surfaces** and start the **Die Wizard**



Surface tangency toggle on fill in surfaces

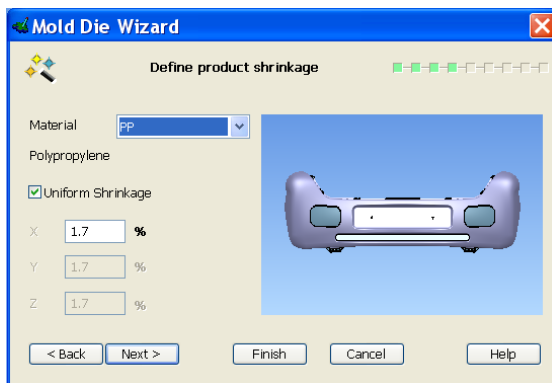


- Select the fill in curve No 10 and view from the front (X axis).
- Toggle the surface tangency button to see the effects on the surface

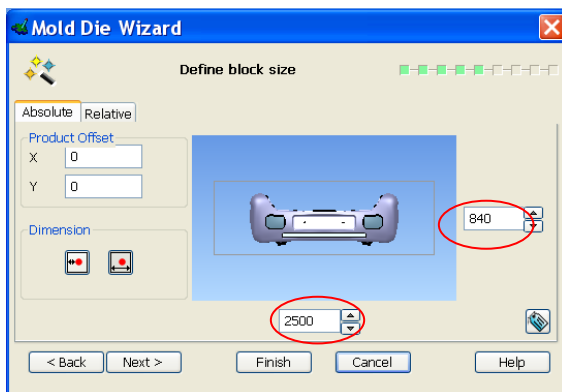
If user defined fill in surfaces are supplied without dependencies they can be recreated using the wizard.



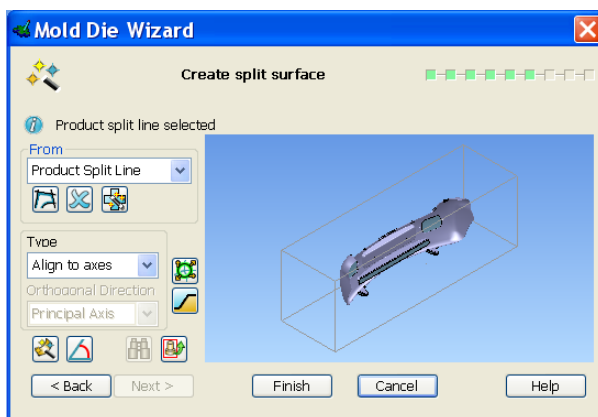
- Add shrinkage



- Define the block sizes



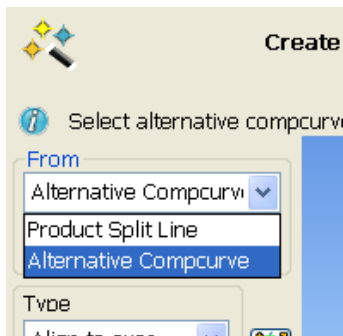
As the selection contains **shut out** surfaces the split line is not automatically displayed



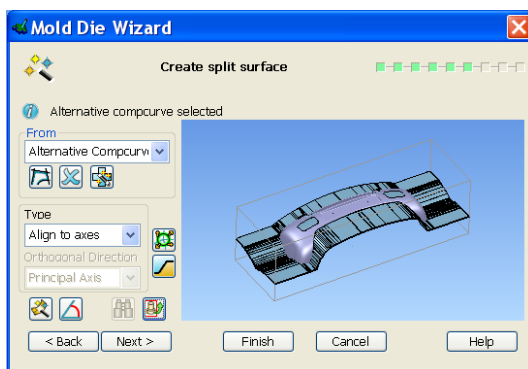
- Turn on **Level 70**, this level contains the alternate Compcurve



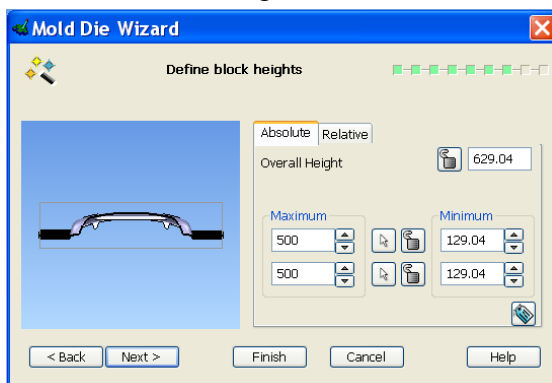
- Select the Alternative Comp Curve option from the drop down menu.



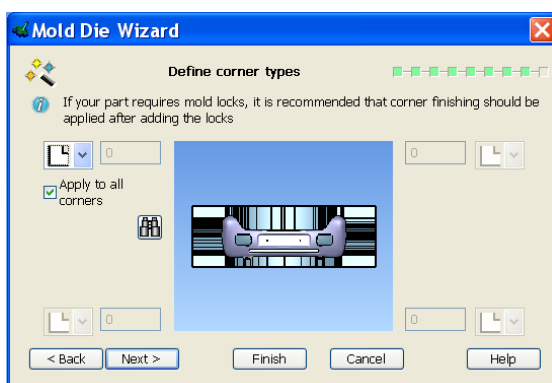
- Preview the results



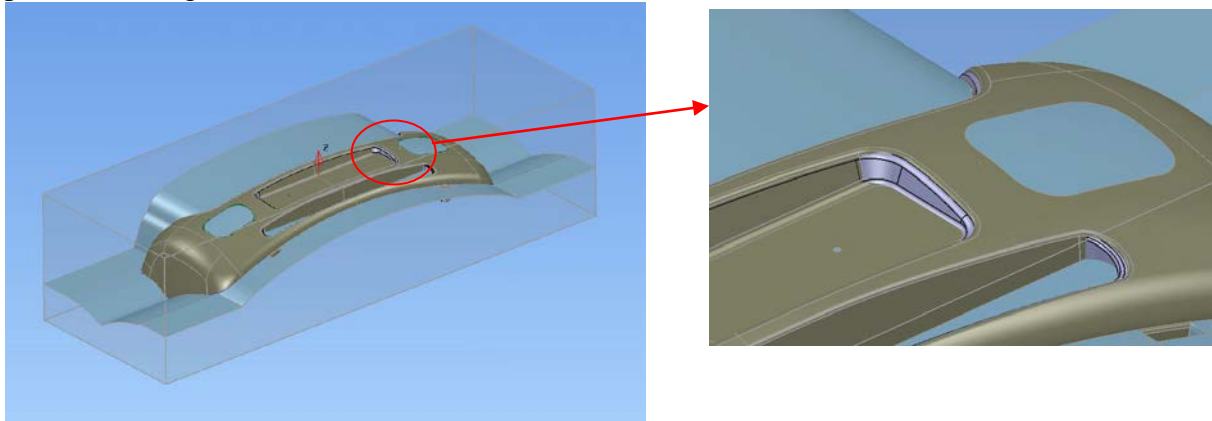
- Set the block heights.



- Change the corner settings to square

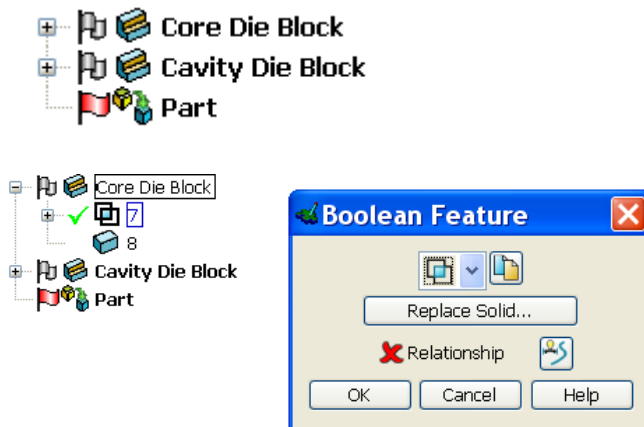


On the finished die blocks you can see that shrinkage has been applied to the dies only, the part is still original size.



The next stage is to **copy out the solid 'shells'**.

- **rename** the solids in the tree to the following:-



- **Copy out the solid 'shell' from the core die block**



- **Rename** the new solid to **"core shell"**
- Repeat the above operation for the **Cavity 'shell'**.
- Place the **core die block on level 40** and the **cavity die block on level 30**

Start PS-assembly and PS-Moldmaker.

Start the Moldbase Wizard from the Moldmaker wizard drop down menu



Mold Base Wizard

Base catalog: GENERIC (Metric) ▼

Dimensions: ☐ Standard ☒ Custom

2400 500

Base type: Standard mold ▼

☒ Support plate

☒ Risers

☒ Ejection system

Guide pillars: ☒ Fixed half ☐ Moving half

Show the mold base

Rotate the mold base:

Plates

Top clamping plate ☒

Cavity plate ☐

Core plate ☐

Support plate ☐

Ejector plate ☐

Ejector base plate ☐

Risers ☐

Bottom clamping plate ☐

Type: Plain plate ▼

Thickness: 246

Material: Steel

Dimensions: Custom, mms

Product: 500 2400 246

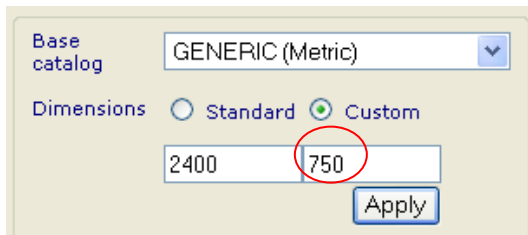
code: Steel

- Select Generic Metric catalogue
- Change dimensions to **Custom** and input values shown then **Apply**
- Rotate the Mold base 90 degrees
- Select **Make ALL plates plain**

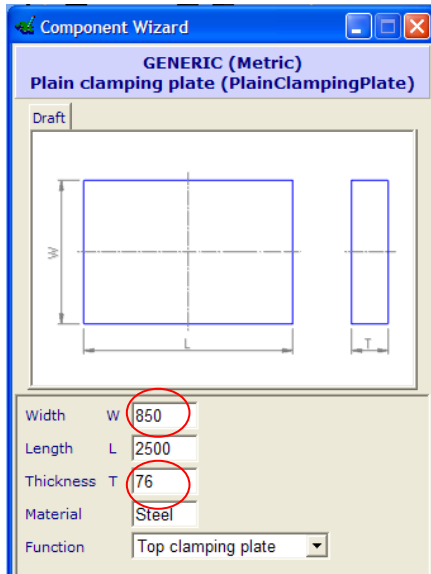
- **View the model in Plan (Z axis)** to check the moldbase size

The mold base is not wide enough.

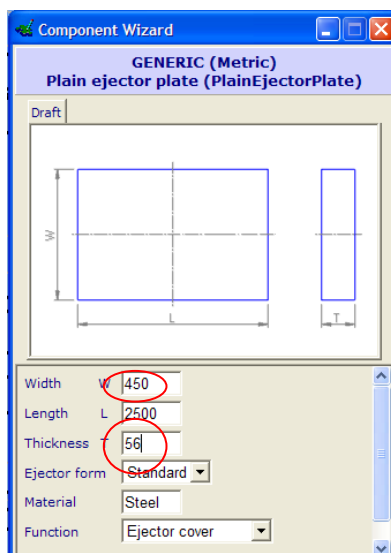
- Modify the width to 750mm



- **Modify both Top and Bottom clamping plates dimensions to these new values**

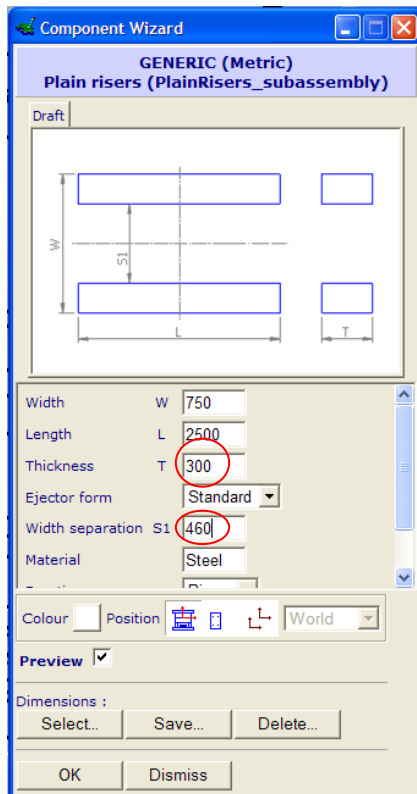


- **Modify both ejector plates dimensions to the new Width of 450mm**
the ejector base plate to **Thickness 96mm** and the ejector cover plate to **Thickness 56mm**



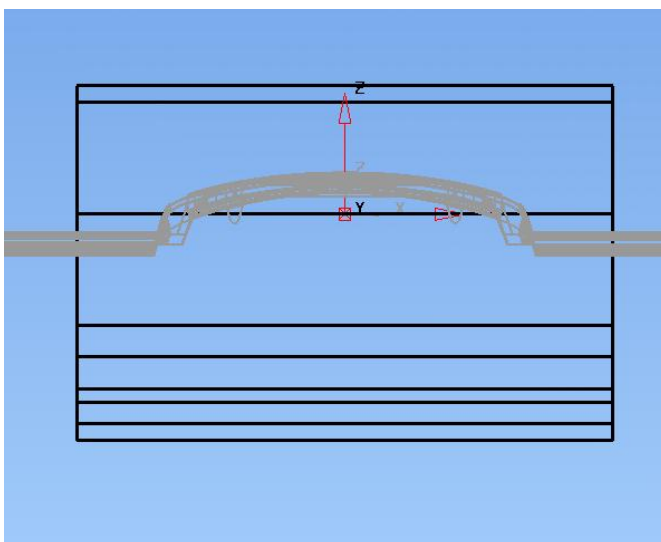
- Modify the Thickness of both the **Core and Cavity Plates to 500mm**

- Modify the Risers **Dimensions** to be **Thickness 300mm** with a **Width Separation of 460mm**

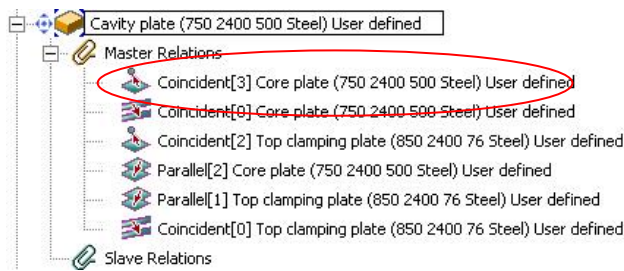


- **Next**
- **Next**
- **Finish** to create the Mold base

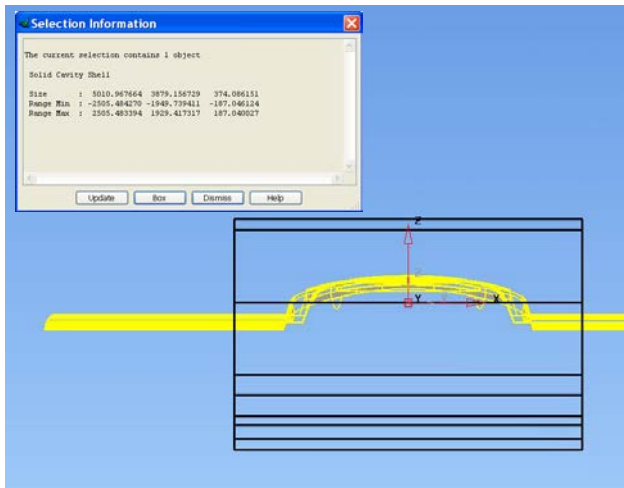
The result should look like this:-



- In the **assembly tree** select the **Cavity plate** component and using the RMB menu, **Delete** the **Point Plane relation** with the **Core plate**.



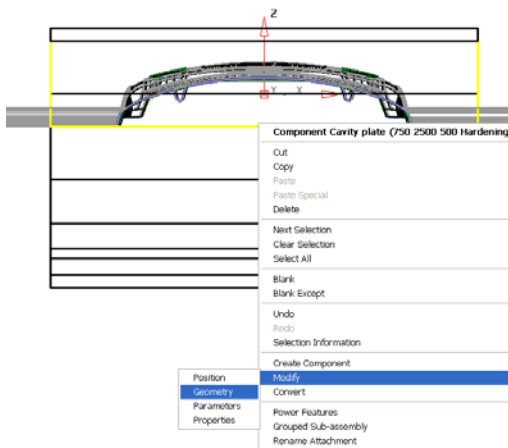
- Select the **cavity shell** solid and display the **Information**, take note of the **min Z** value.



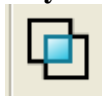
- Select the **cavity plate component** and using **general Edits - Move**

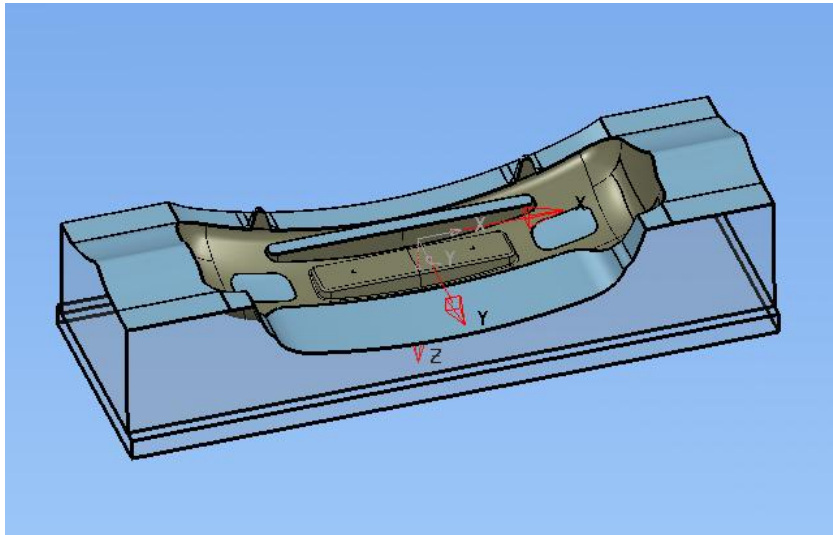


- With the **Cavity plate component** still selected, use **RMB** menu to **Modify Geometry**.

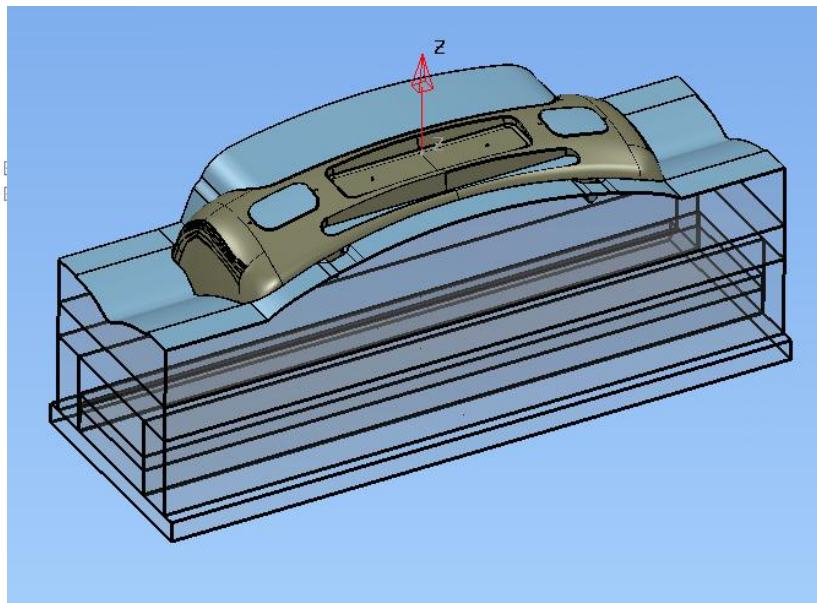


- Select the **“cavity shell”** solid and pick **solid intersect** from the **solids menu**



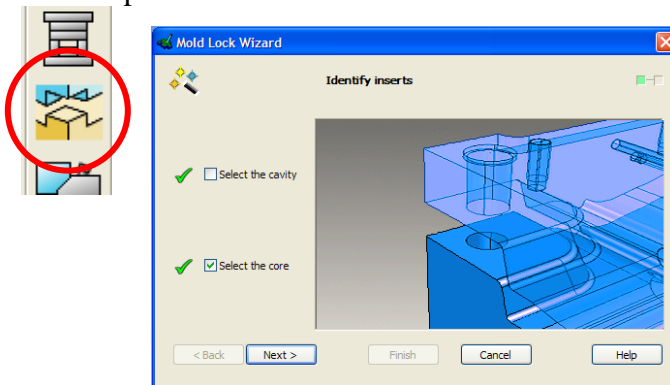


- Repeat the above steps for the **Core plate**, using the “**core shell**” solid.

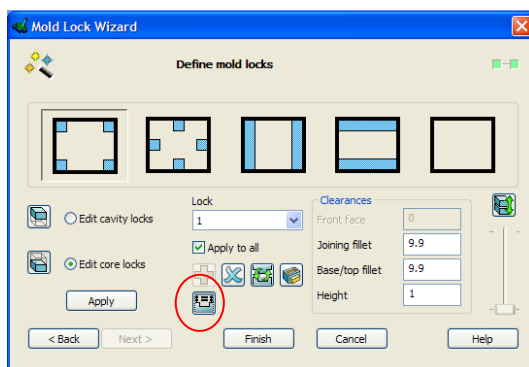



The Mold base is ready to have Mold Locks added

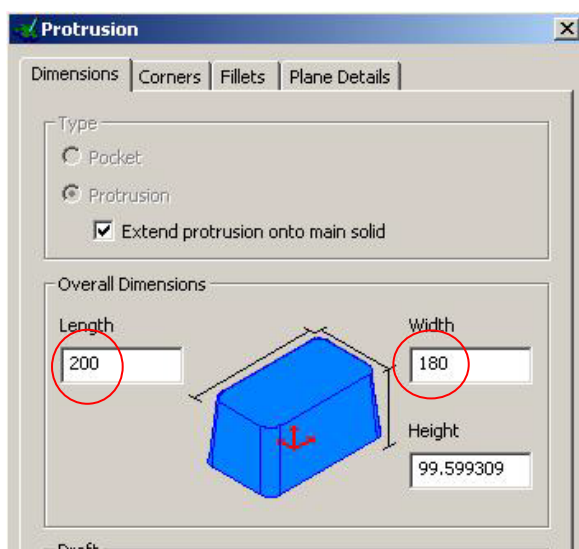
- Start the **Mold Lock Wizard** from the **Wizards** menu, identify the **cavity** and **core** components



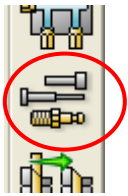
- Select the **four corner** option and set **all heights equal** using the button indicated.



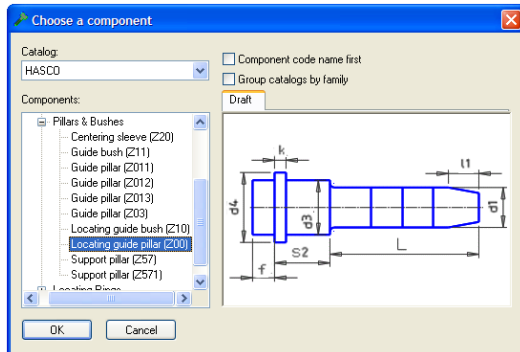
- Modify** the locks  and input new values for **Length** and **Width**.



- OK** the modifications
 - Finish** the Mold Lock Wizard to create the locks.
- Now you can add some components using **Component Wizard**.



- Select **Hasco Z00 Locating Guide Pillar**.



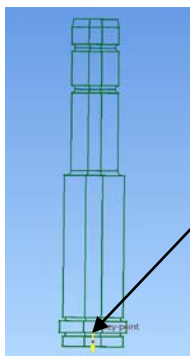
- Select the largest size pillar from the **standard dimensions**

Reference name
Z00/136/42x215

| Shank diameter (d1) | Shank length (L) | Body length (S2) |
|---------------------|------------------|------------------|
| 40 | 95 | 136 |
| 42 | 135 | 156 |
| | 215 | |

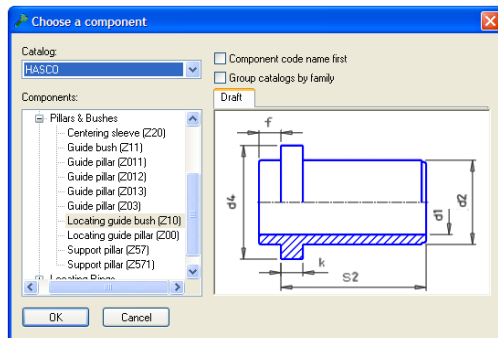
Body diameter (d3)
54

- Using the **cavity plate** as the **Master component**; add one pillar, in a corner of the plate.
Position X 1150 Y 305.



- The **slave attachment** should be selected on the bottom of the head not the base of the part

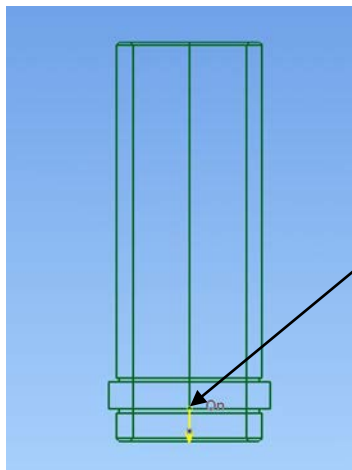
- Select **Hasco Z10 Locating Guide Bush**.



- Select the largest size from the **standard dimensions**

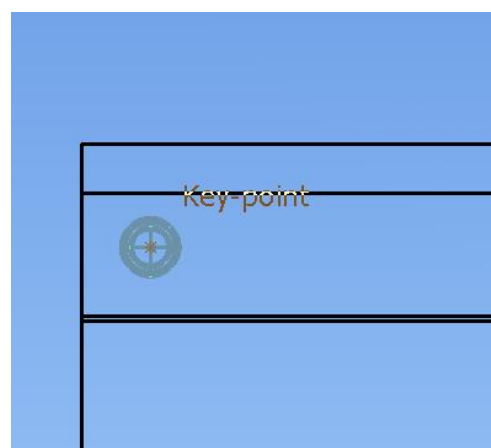
| Reference name | | |
|---------------------|-------------|---------------|
| Z10/136/42 | | |
| Inner diameter (d1) | Length (S2) | Diameter (d2) |
| 30 | 136 | 54 |
| 32 | 156 | |
| 40 | 196 | |
| 42 | 246 | |

- Using the **Core plate** as the **Master component** add one Bush, use the **key point** on the **pillar component** to position the bushes.

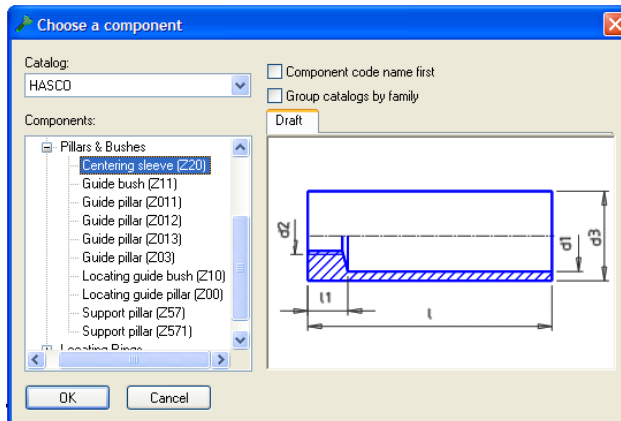


- The **slave attachment** should be selected on the bottom of the head not the base of the part

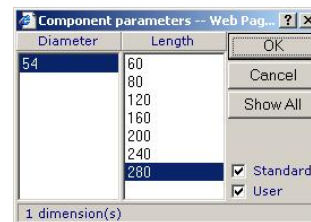
- Pick the **Key point**



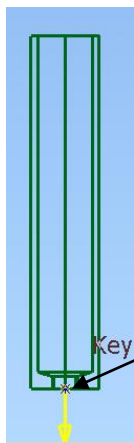
- Select **Hasco Z20 Centring Sleeve**.



- Select the largest size from the **standard dimensions**



- Using the **Bottom Clamping plate** as the **Master component** add one Sleeve, use the **key point** on the **pillar component** to position the bushes.



- The **slave attachment** should be selected on the bottom of the part



- Pick the **Key point**

As you can see all of the new components that we have added are far too small, this is due to using standard components that have a specific size range.

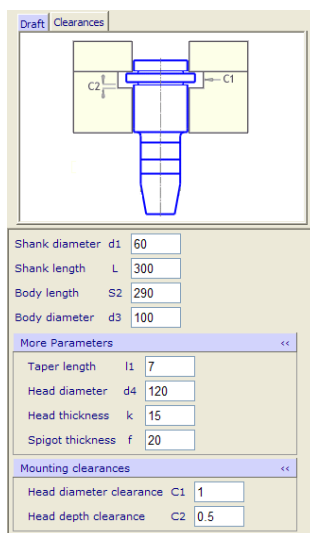
When designing this type of mold tool, very large components are required. Generally they are custom made to suit the design.

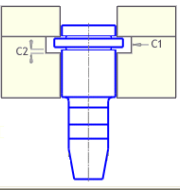
Now that there are parametric components we can start with a standard component and very easily modify to get our custom parts.

- **Select the Locating Guide Pillar** and using RMB menu pick **300Modify Parameters**.

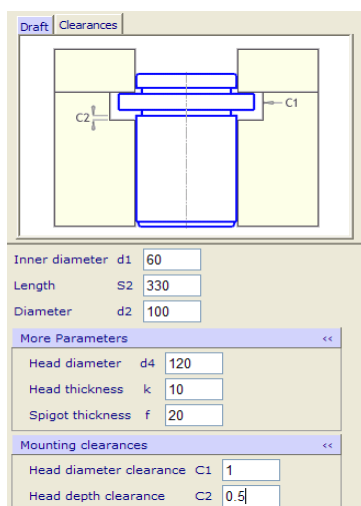
The component wizard is opened.

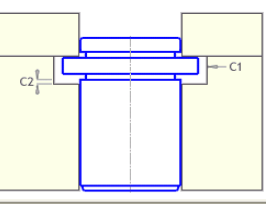
- **Modify** the parameters to the values indicated here:

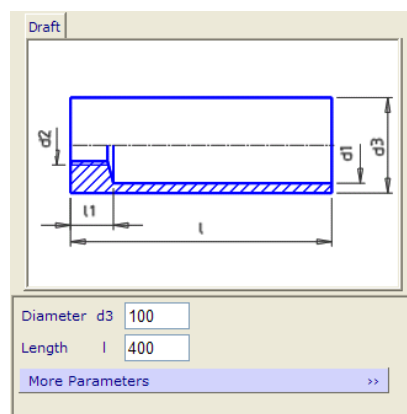


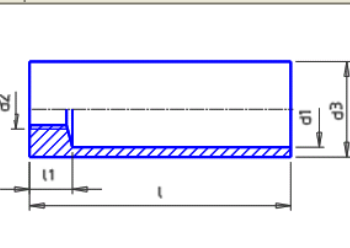
| Draft | |
|---|--------|
| Clearances | |
|  | |
| Shank diameter | d1 60 |
| Shank length | L 300 |
| Body length | S2 290 |
| Body diameter | d3 100 |
| More Parameters << | |
| Taper length | l1 7 |
| Head diameter | d4 120 |
| Head thickness | k 15 |
| Spigot thickness | f 20 |
| Mounting clearances << | |
| Head diameter clearance | C1 1 |
| Head depth clearance | C2 0.5 |

- **Apply** then **Dismiss** the form
- Repeat for the **Locating Guide Bush** and the **Centring Sleeve**

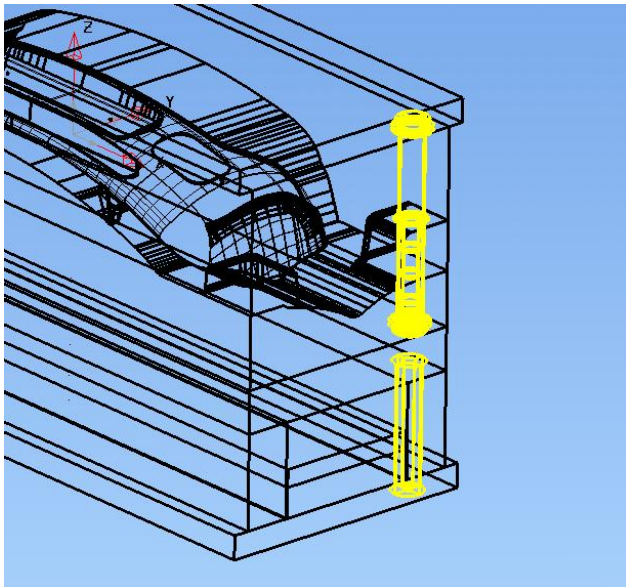


| Draft | |
|---|--------|
| Clearances | |
|  | |
| Inner diameter | d1 60 |
| Length | S2 330 |
| Diameter | d2 100 |
| More Parameters << | |
| Head diameter | d4 120 |
| Head thickness | k 10 |
| Spigot thickness | f 20 |
| Mounting clearances << | |
| Head diameter clearance | C1 1 |
| Head depth clearance | C2 0.5 |

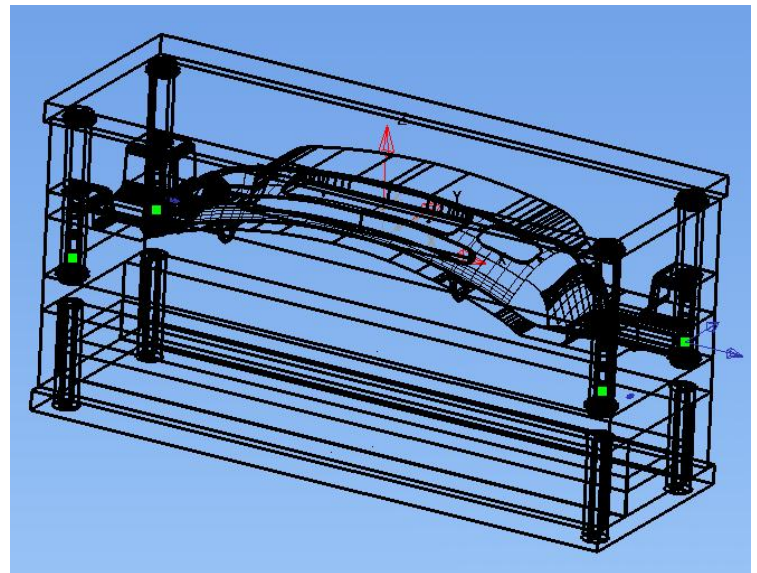
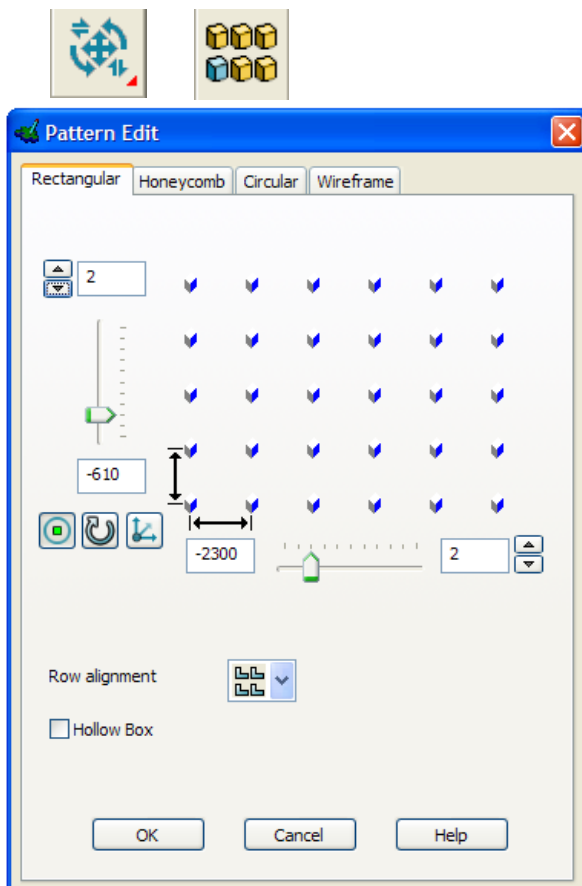


| Draft | |
|--|--------|
|  | |
| Diameter | d3 100 |
| Length | l 400 |
| More Parameters >> | |

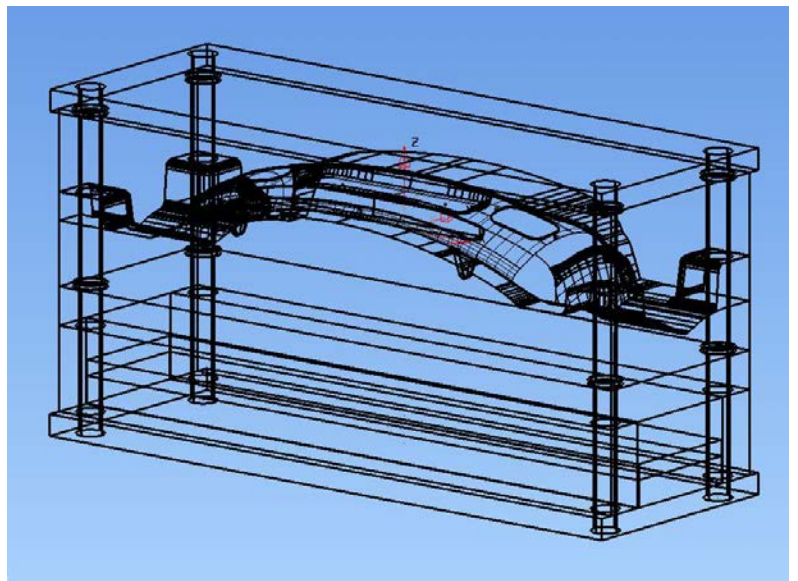
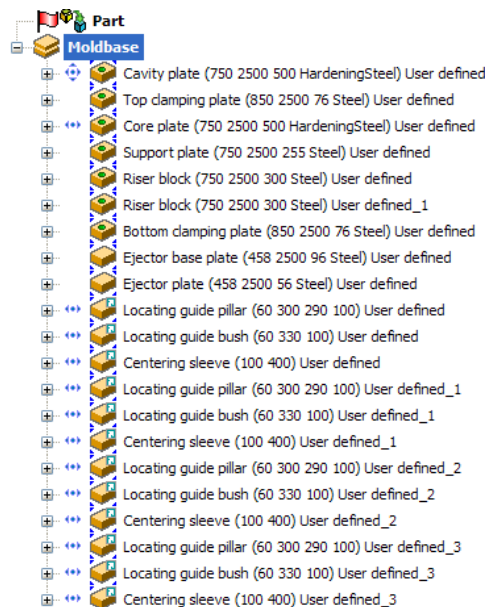
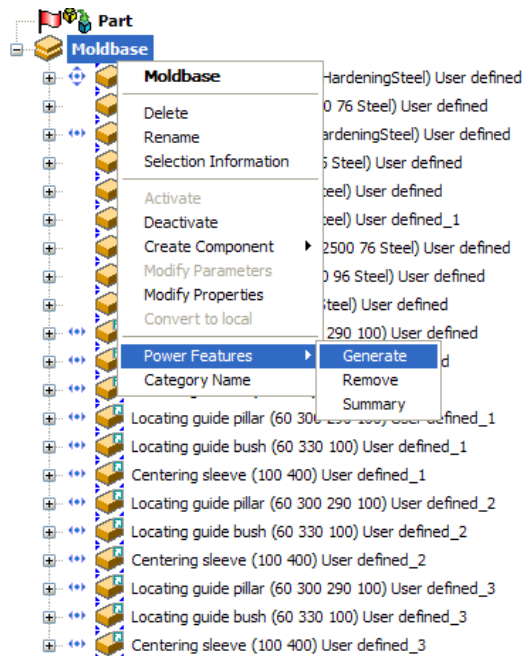
To populate the moldbase select the Pillar/Bush and Centring Sleeve and use the Pattern Command to produce the other three sets of components.



- Start the Pattern Command



- Hit **OK** to create the components and close the form

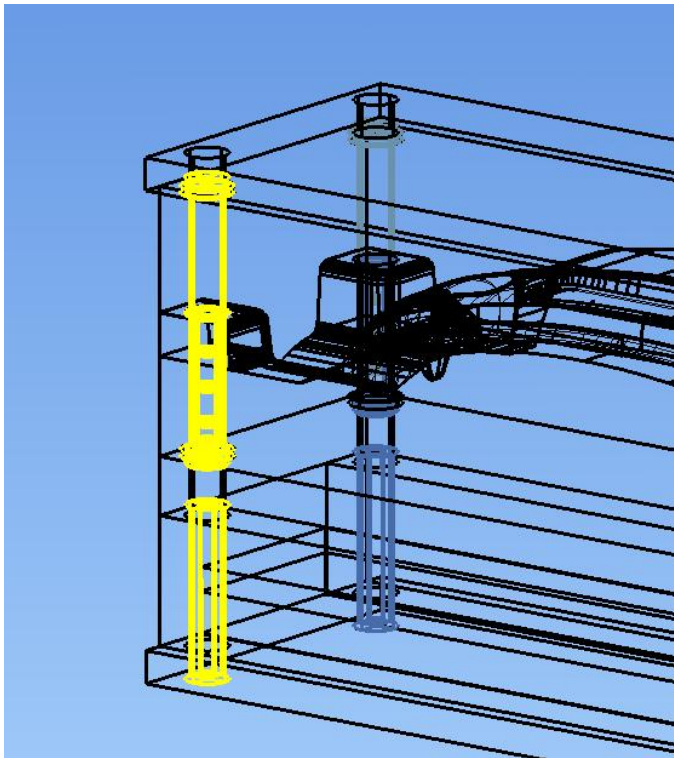


- Select and blank the additional components so that the power feature holes can be viewed clearly.

Tip

You can now use **shift click** to select multiple entities in the assembly tree

- Select a single set of components with the same XY position.



- Raise the General Edits toolbar

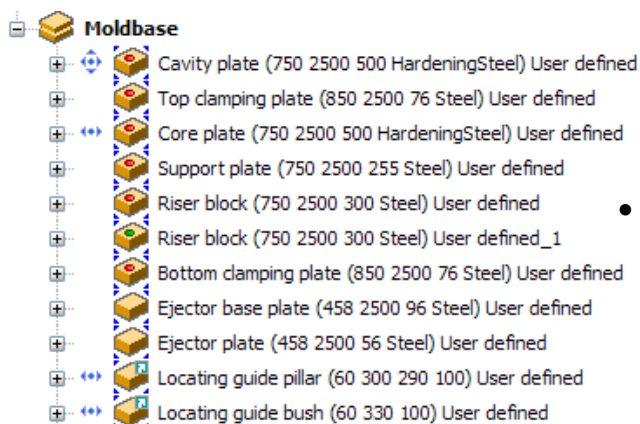


- Pick Move from the Toolbar



- Move the components 50mm in X axis

As a result of moving these three components the power features are highlighted as out of date.



- Re-generate the power features to update the holes

11. Moldbase Wizard - Stack.

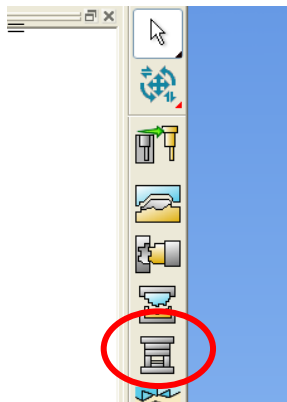
Moldbase Wizard has been enhanced with some new options to build custom Mold bases.

You can now start with a single plate and add or remove plates in any position to create any combination of plates that you require.









The option to add plates extends to the standard catalogues although you can not remove a standard plate, all extra plates added will be plain.

Note: If you add extra plates to a standard Mold base e.g. Hasco during initial creation in the wizard then Power features will be enabled for the standard components. This can cause problems as it will create holes where standard plate holes already exist.

- Start with a New Model and run the Moldbase Wizard from the Moldmaker wizard drop down menu



New Moldbase Options

- | | |
|---|---|
|  | Make the selected plate plain |
|  | Make ALL plates plain |
|  | Rename the selected plate |
|  | When inserting a plate move the existing plates where needed. |
|  | When inserting a plate do not move the existing plates, but 'Force' the plate into the stack (typical application is ejector plates) |
|  | Insert a plate above the selected plate |
|  | Insert a plate below the selected plate |
|  | Remove selected plate from moldbase |

Select **HASCO** standard **196x346** then **Stack of Plates** option from the drop down menu

The New option is displayed.

The default number of plates will always be at least one,

All plates created using this method will be plain.

The Master plate is always created and is used as a starting point when the assembly relations are built between subsequent plates

There are a number of new GENERIC plate 'types'

- Change the **Number of plates** to **8** and **Apply**

Mold Base Wizard

Base catalog: HASCO

Dimensions: ☒ Standard ☐ Custom

196x346

Base type: Stack of plates

Number of plates: 8 **Apply**

Show the mold base

Rotate the mold base: ☐ ☐ ☐ ☐

Plates

Extra plate
Extra plate
Extra plate
Master plate
Extra plate
Extra plate
Extra plate
Extra plate

Type: GENERIC plate

Extra plates are created above and below the master plate

Plates

Extra plate
Extra plate
Extra plate
Master plate
Extra plate
Extra plate
Extra plate
Extra plate

Type: GENERIC plate

Thickness: GENERIC plate

Material: GENERIC standard risers

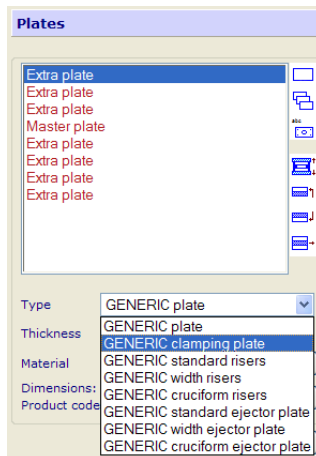
Dimensions: GENERIC width risers

Product code: GENERIC cruciform risers

GENERIC standard ejector plate
GENERIC width ejector plate
GENERIC cruciform ejector plate

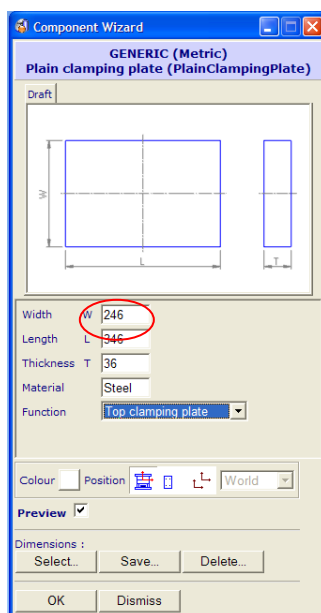
Set the type of plate from the drop down menu. Standard type are supported, any others can remain as Generic Plates

- Set the **type** of the top extra plate to be **GENERIC clamping plate**



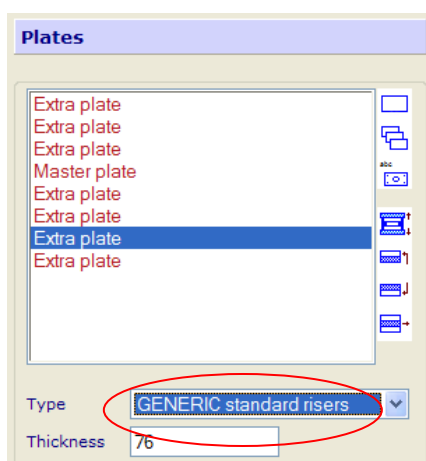
Note that although the type has changed the name has not.

- Modify the **Dimensions**

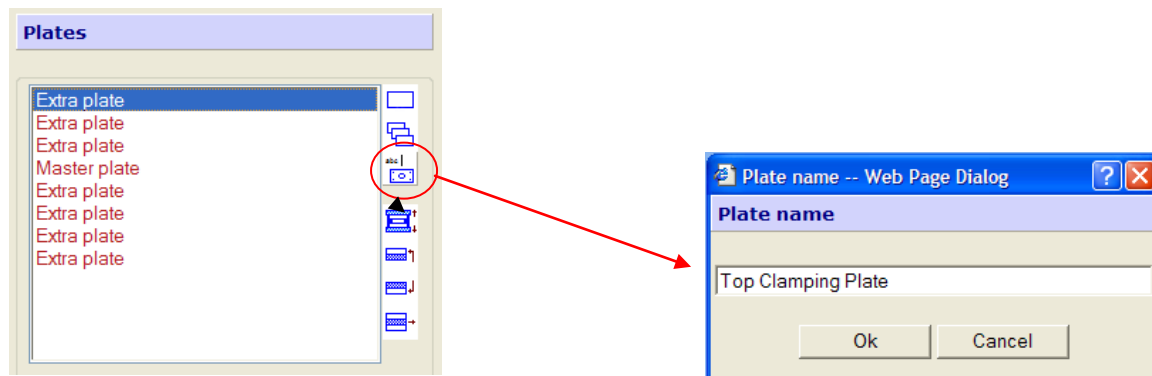


- Change the **Width to 246mm** to give an overhang, **OK** to close the form.

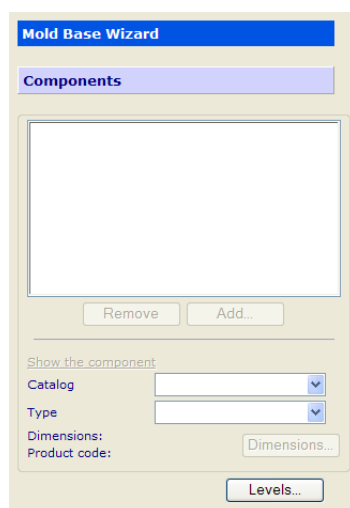
As all the plates are created as plain plates to create Risers we must change the type of the plate in the position where the risers should be. This could be any position in the moldbase.



- Select the **Top extra plate** and rename it to **Top Clamping Plate**



- Repeat for the **bottom extra plate** and rename it to **Bottom Clamping Plate**
- Next

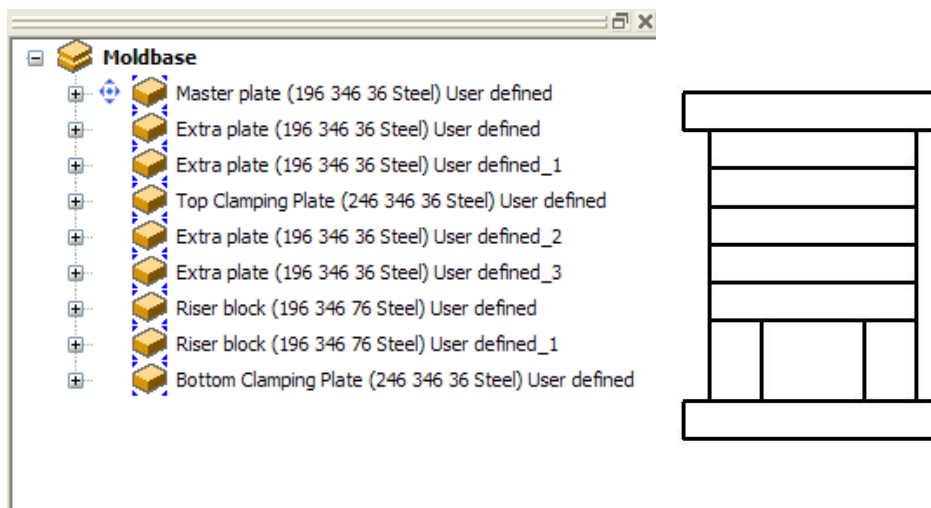


- Next



- **Finish** to create the Mold base

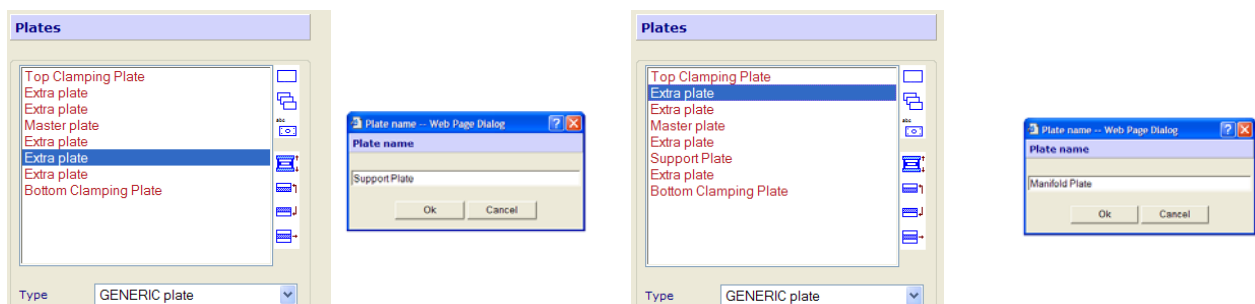
The assembly tree reflects the changes made so far.



- Re enter the **Mold base Wizard**

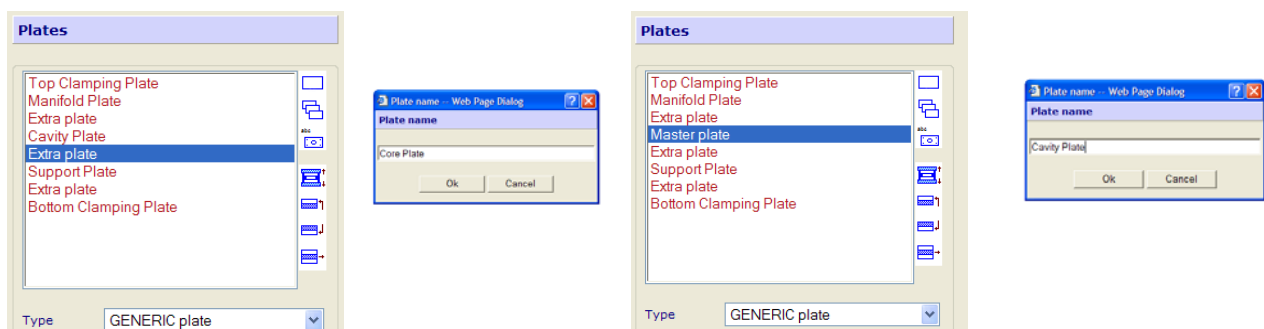


- **Rename the remaining Plates**



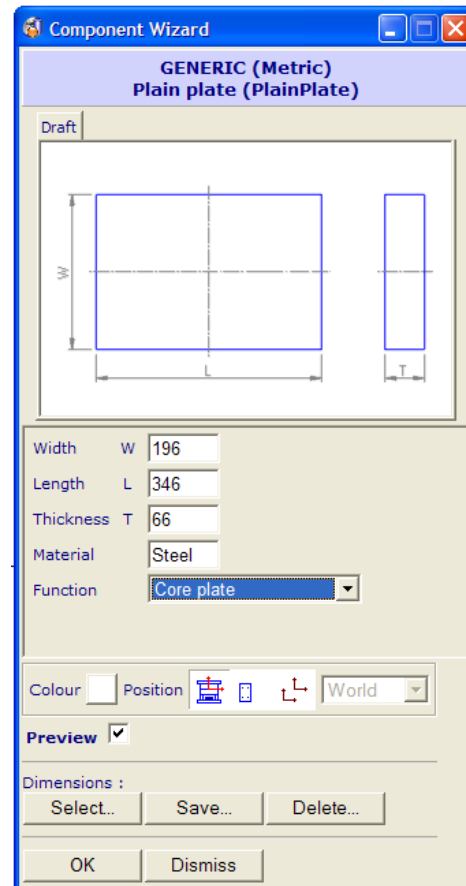
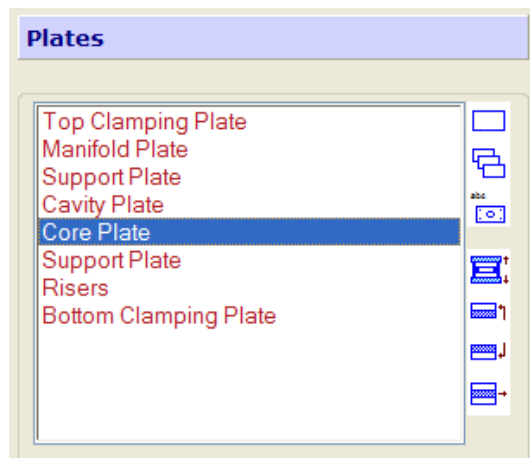
Support Plate

Manifold Plate



Core Plate

Cavity Plate



- Using the **dimensions form** change the thickness to **66mm** and the **function** of the **core plate**
- **Repeat** for the **cavity plate**

- **Next – Next - Finish**

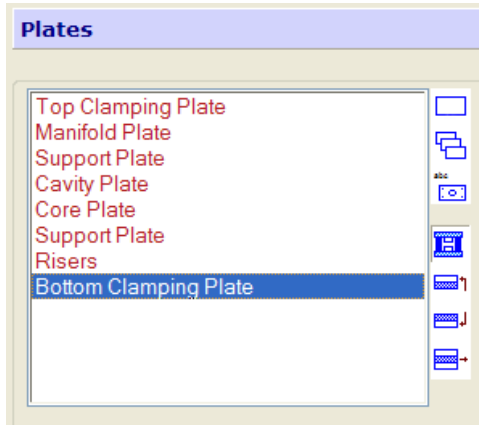


- Re enter the **Mold base Wizard**




Now you will create the ejector plates

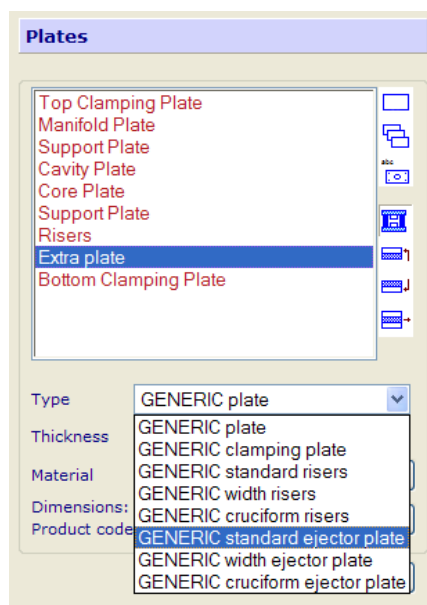
- Select the **Bottom Clamping Plate**



- Click  Toggle 'Force' extra plate

- Click  Extra plate above selected

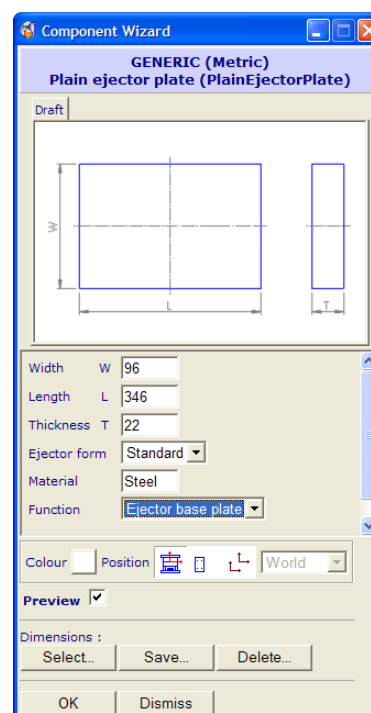
A new plate is created; it has the same name as the previously selected plate



- Change the 'type' to **GENERIC standard ejector plate**

- Using the dimensions form change
The **width of the ejector plate** to **96mm**

The default ejector plate does not have any clearance with the risers

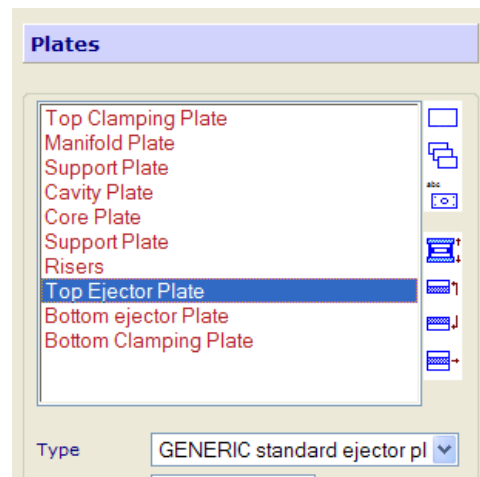


- Rename the plate to **Bottom Ejector Plate**

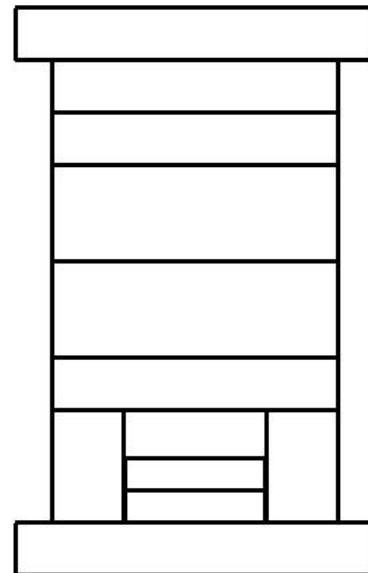
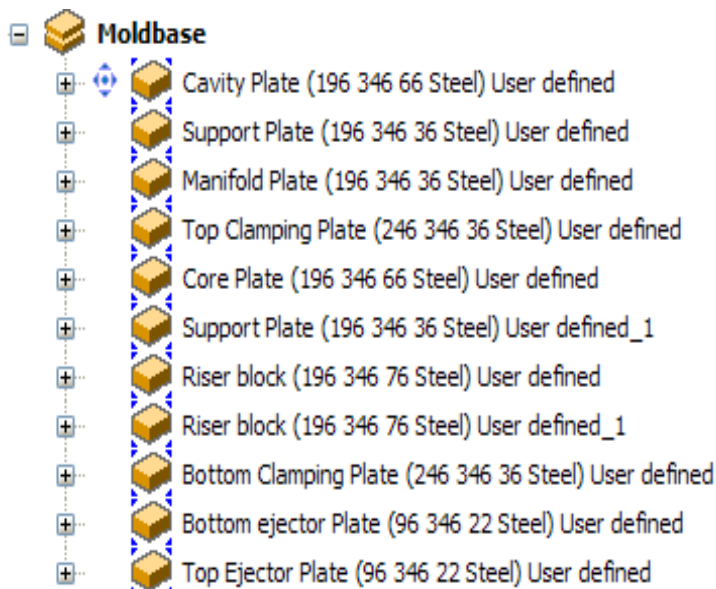
- Add another **plate above** and repeat
The steps to create the **Top Ejector Plate**



Extra plate above selected



- **Next – Next - Finish**



The Moldbase is now complete; all of the components are “User defined”. This description is used whenever a component cannot be sourced from a standard catalogue.

As you have seen the Mold base Wizard can be re entered to make changes at any time.

Exercise

Create levels and groups of levels for Fixed and Moving half sides of the tool

Add pillars / bushes and plate screws to the moldbase.

Important Notes:

If you re enter the moldbase wizard and change the “number of plates” this will result in a NEW moldbase being created with any changes lost

Also changing the “set size” i.e. to 246 396 will also result in a NEW Mold base being created.

Any changes to the moldbase configuration should be done using the add or remove plates options.

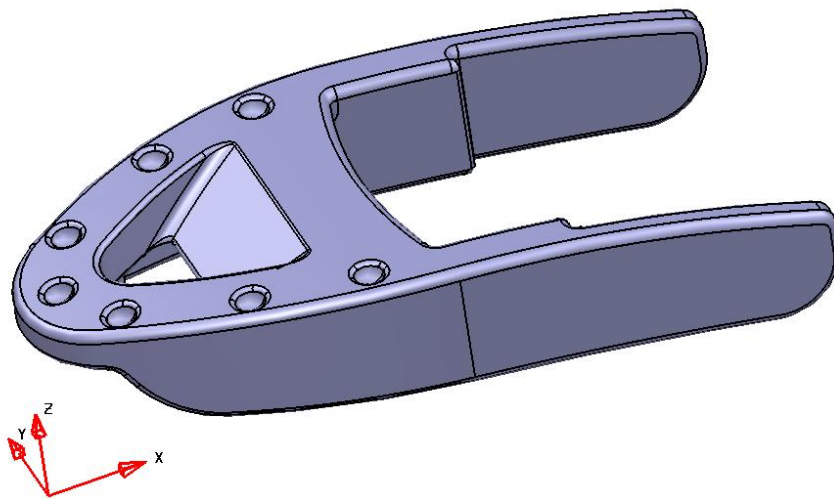
Any changes to plate sizes should be done in either the dimensions form in the wizard or by modifying the parameters on the individual plates via component wizard.

12. Sub Assembly

When using PS-Assembly one of the most useful ways of reducing work / model size is to use sub assemblies for components that are used in multiple places.

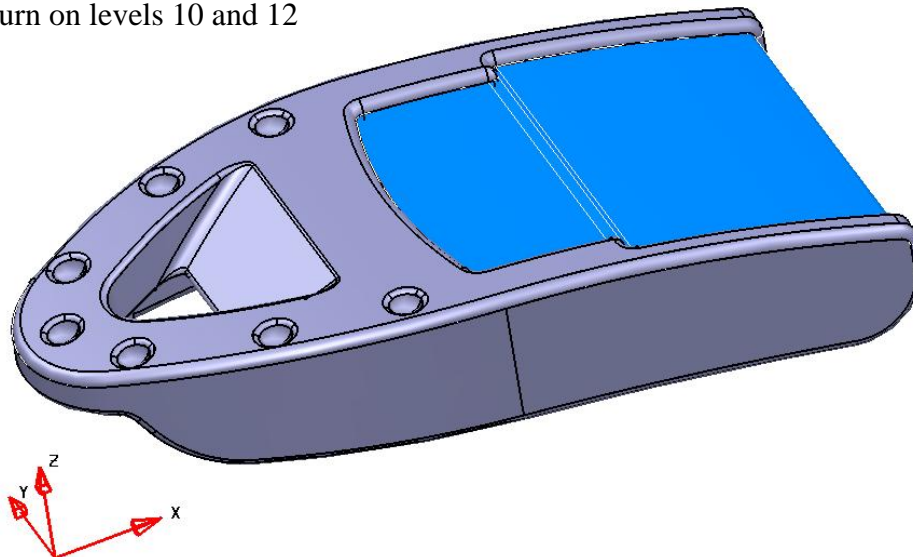
This option allows modifications to be performed more easily.

Open Model – Sub Assembly.psmodel

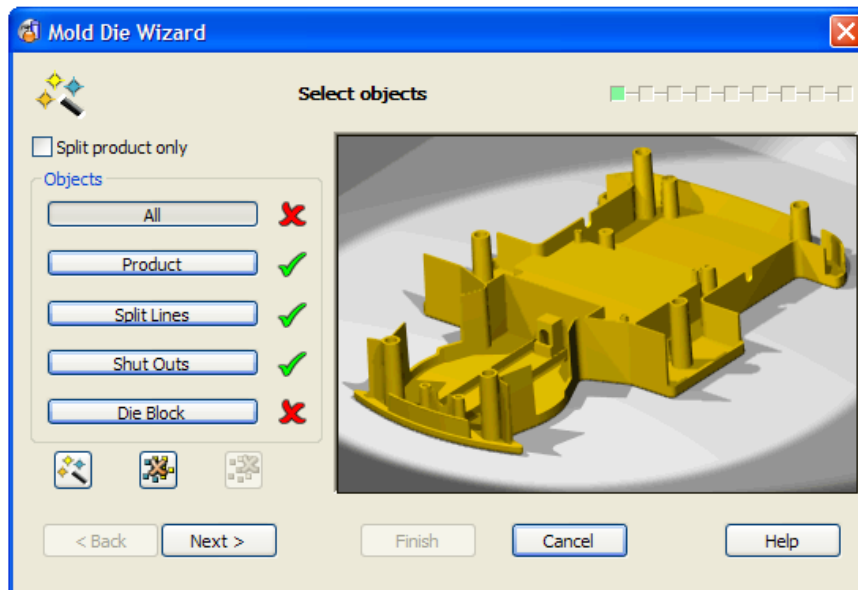


There are pre created fill-in and split curves in the model.

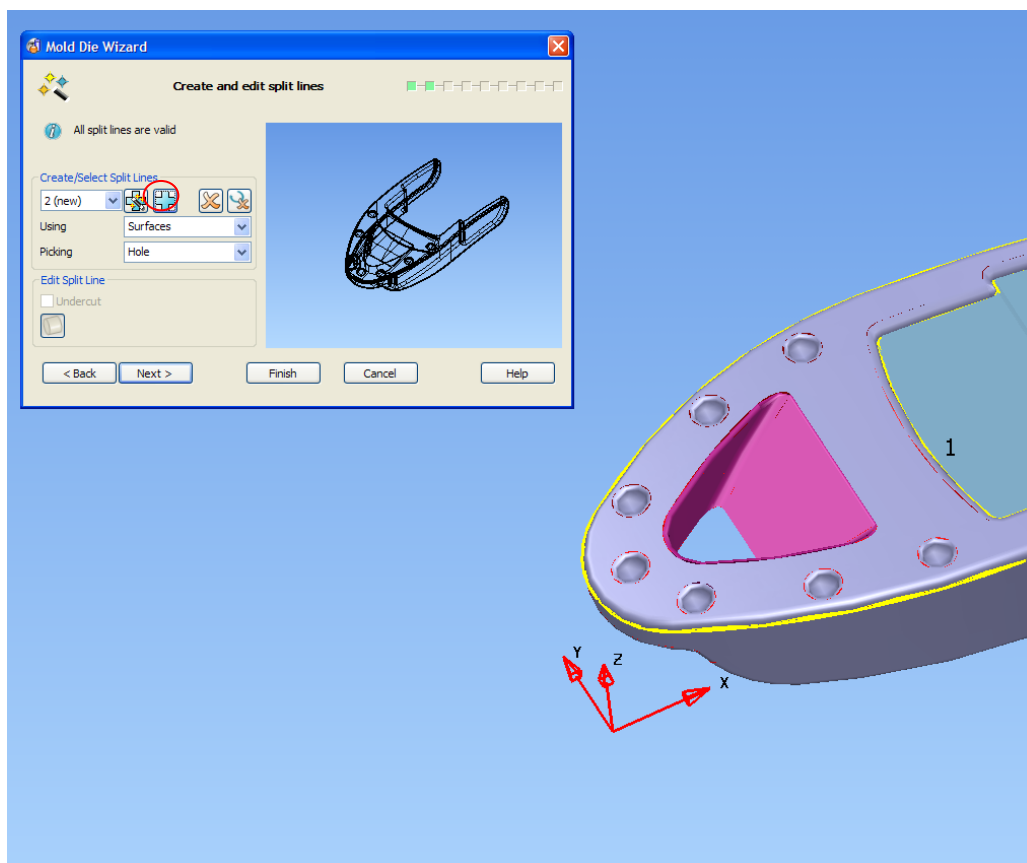
- Turn on levels 10 and 12



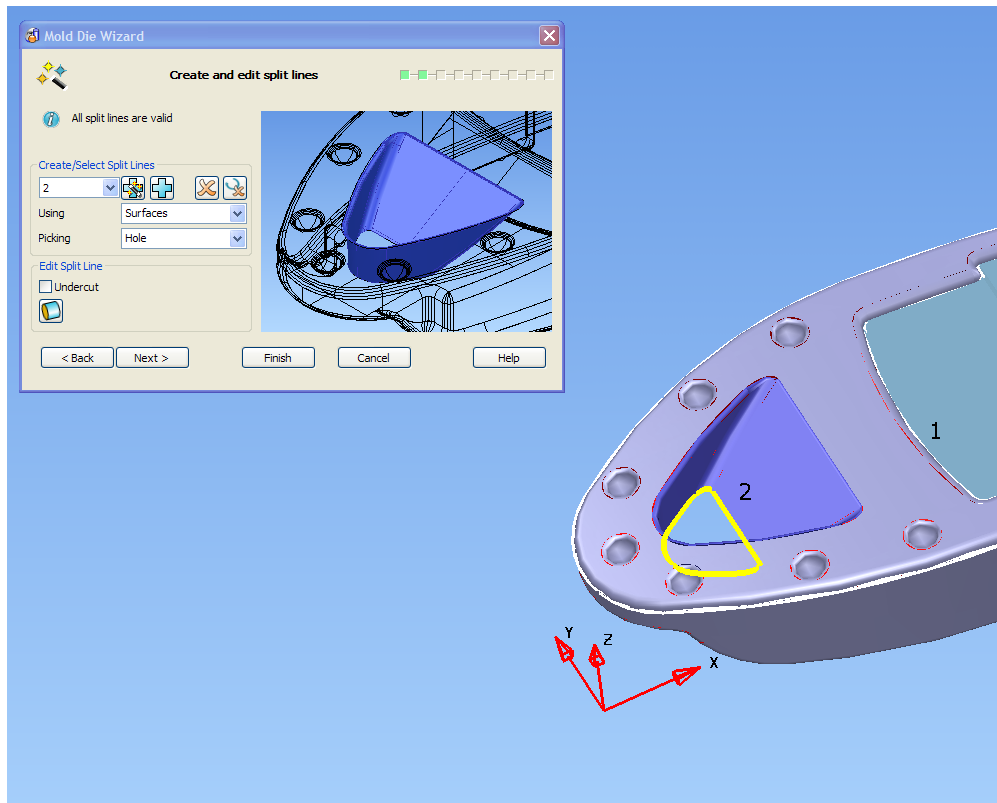
- **Select all** and start **Die Wizard**




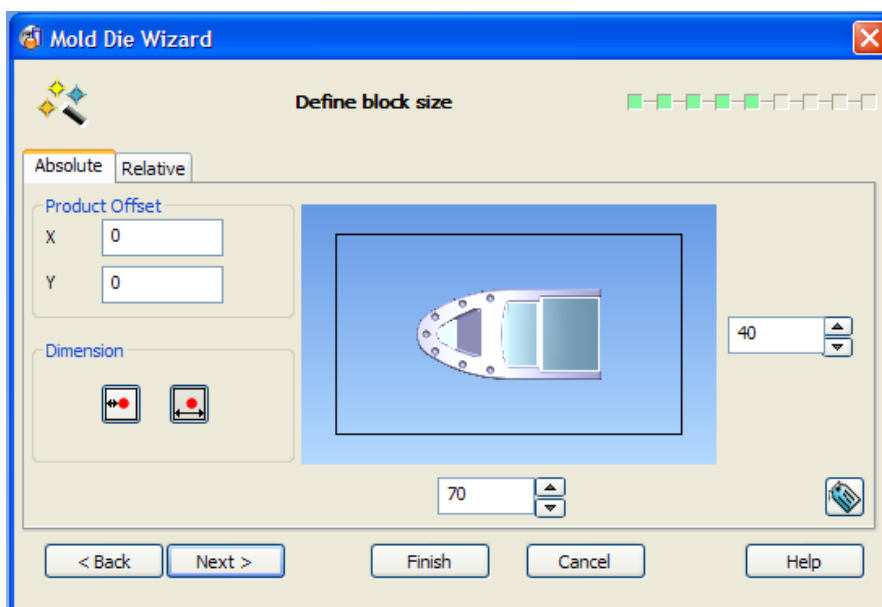
- Add a **new split curve** using **surface selection**



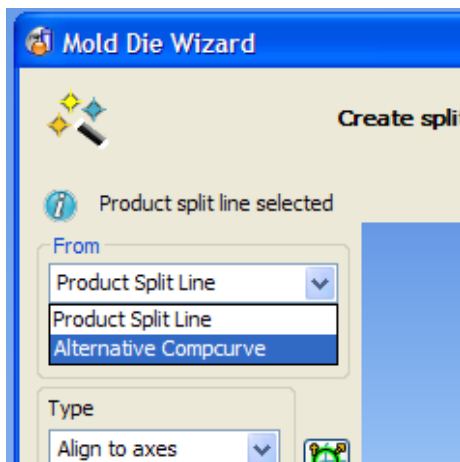
- **Next** with the split curve as created



- Turn **off** the fill in **surface tangency**  **Next**
- **Next** until the Define block size page
- Set the block sizes to **70 x 40**



- Select the **Alternative Comp curve** on the Create split surface page

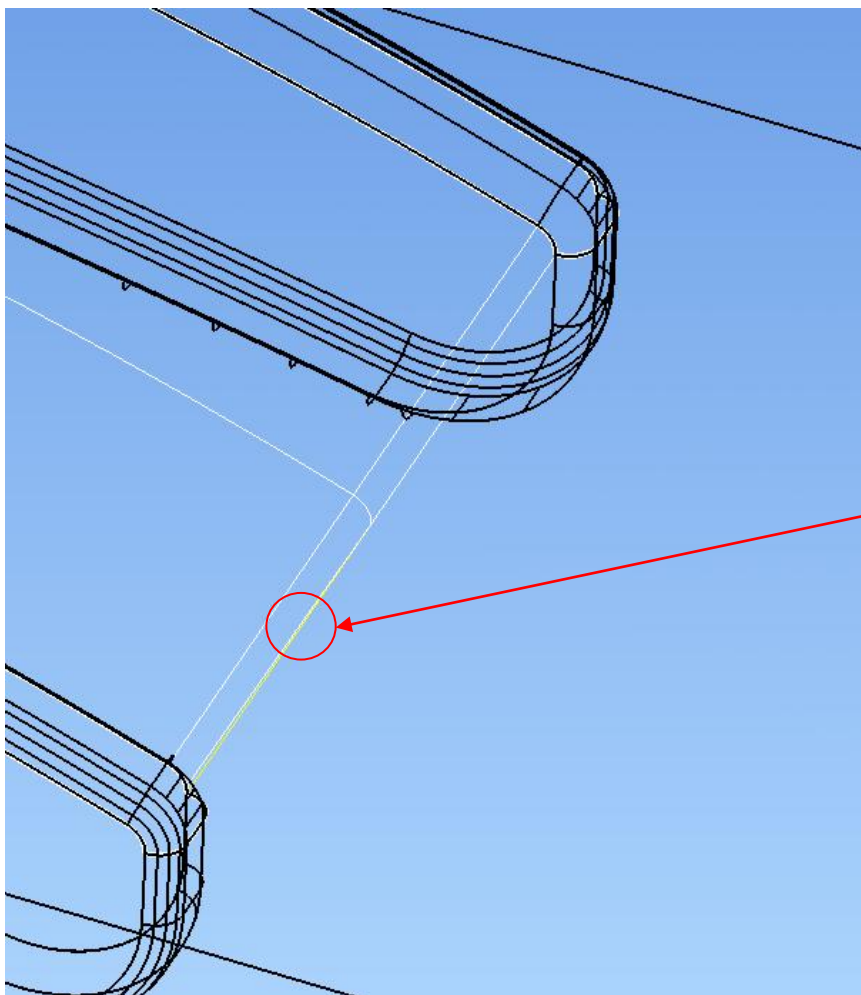


- Turn on **Level 11**

- View the model in **wireframe**

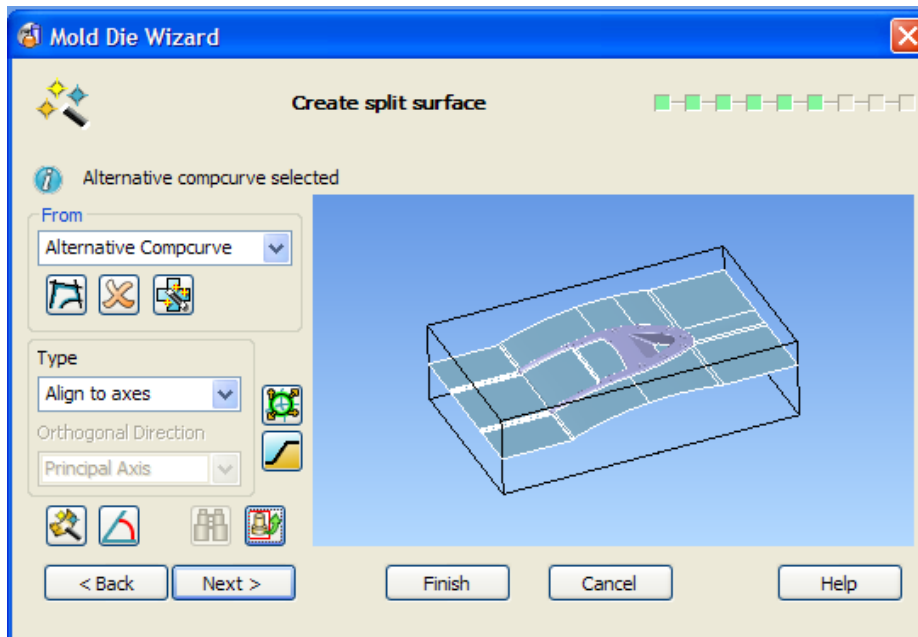


- **Pick** the **comp curve** on Level 11

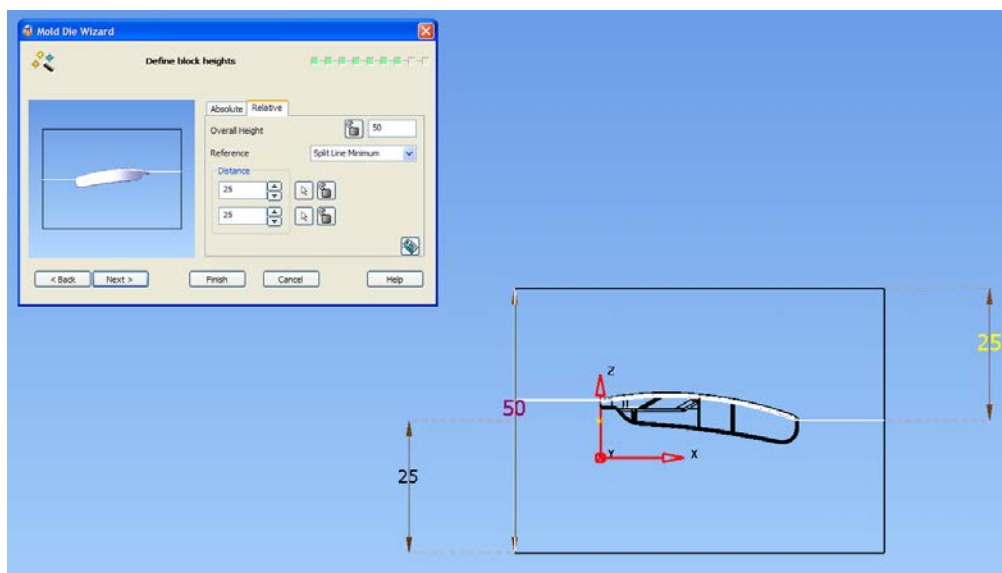


• Pick Here

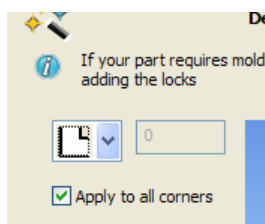
- **Preview** the split surface, **Next**



- Set the block heights **relative to the split line minimum**

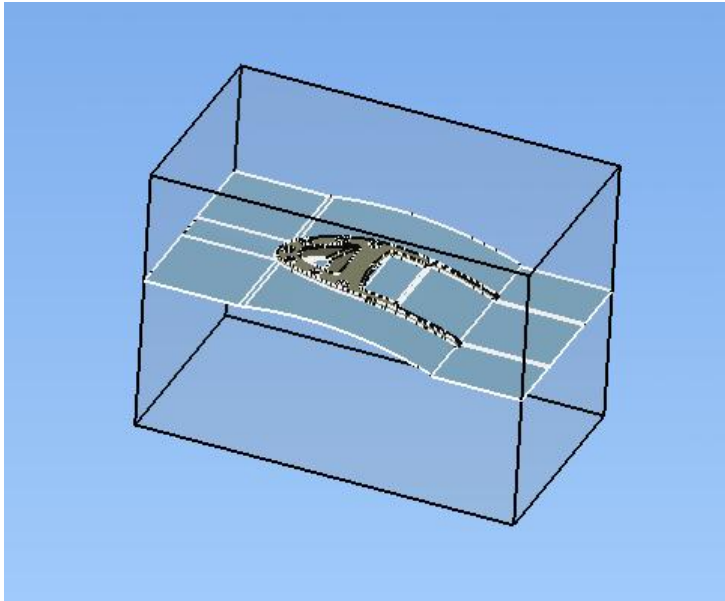


- **Next**

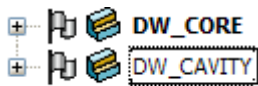


- **Apply** square corners

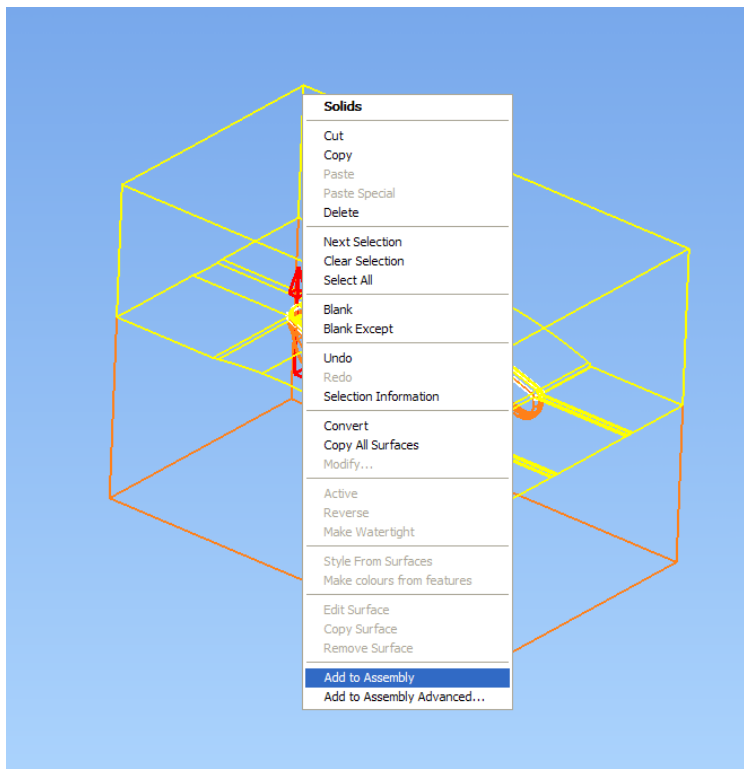
- **Finish** to create the die blocks



- **Rename** the solids to **DW_CORE** and **DW_CAVITY**

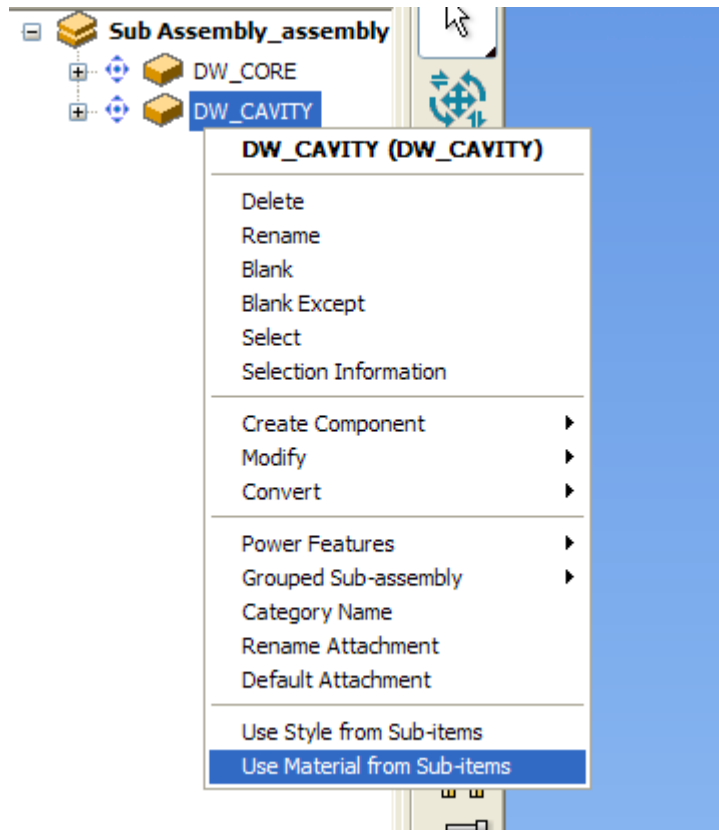


- Make **COMPONENTS** from **selected solids** by **Adding to Assembly**



When the solids are converted to components you will notice that the colour changes and they are solid shaded, this is due to the default material settings for components

- To maintain the material settings from the solids use RMB on the component and select **Use Material from Sub-items**

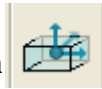


- Select both components

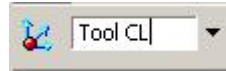
- Create a new **Work Plane**



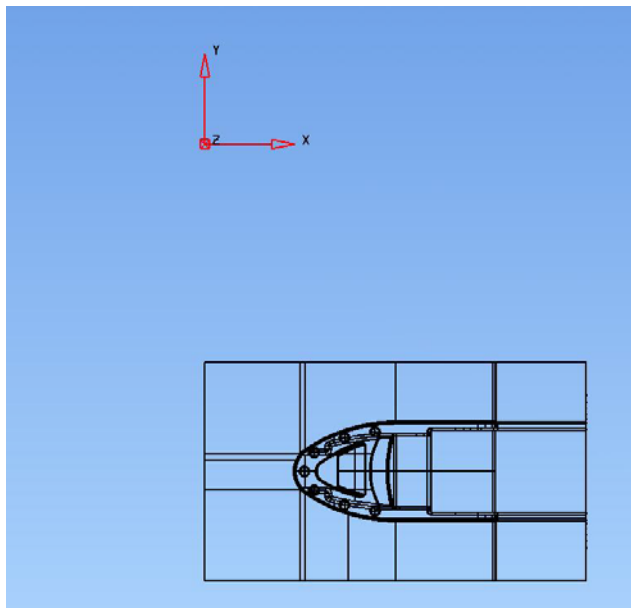
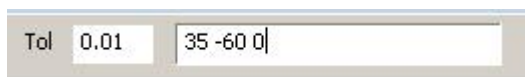
- Select a Work Plane in the **centre of selection**



- **Rename** the new Work Plane to **Tool CL**

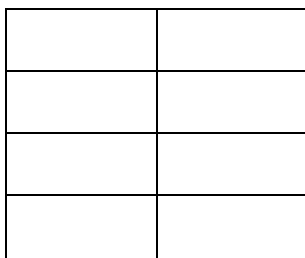


- Use **General Edits** to **MOVE** the die blocks.



The new location of the Die blocks is determined by the requirement to have a multi impression tool with eight sets of Core and Cavity inserts.

The layout will be as indicated below



- Start **PS-Moldmaker**



- Start **Mold Base Wizard**

Mold Base Wizard

Base catalog: HASCO

Dimensions: ☒ Standard ☐ Custom

246x296

Base type: Standard mold

☒ Support plate

☒ Risers

☒ Ejection system

Guide pillars: ☒ Fixed half ☐ Moving half

[Show the mold base](#)

Rotate the mold base:

Plates

- Top clamping plate
- Cavity plate
- Core plate
- Support plate
- Ejector plate
- Ejector base plate
- Risers
- Bottom clamping plate

Type: K10 (overhang in width, loc. rin)

Thickness: 27

Material: 1730 [Comments...](#)

Dimensions: Standard, mms

Product code: K10/246x296x27/1.1730 [Dimensions...](#)

[Levels...](#)

[< Back](#) [Next >](#) [Cancel](#)

- Create a **HASCO 246x296 Mold base**

- Leave all settings as default

- Modify the **Core and Cavity plate thickness** to 46mm

Plates

- Top clamping plate
- Cavity plate
- Core plate
- Support plate
- Ejector plate
- Ejector base plate
- Risers
- Bottom clamping plate

Type: K20

Thickness: 46

Material: 1730 [Comments...](#)

Dimensions: Standard, mms

Product code: K20/246x296x46/1.1730 [Dimensions...](#)

Mold Base Wizard

Components

- Guide pillars
- Guide bushes
- Centering sleeves
- Fixed half screws
- Moving half screws
- Ejector base plate screws
- Locating ring
- Seating washer discs
- Seating washer screws

Remove Add...

Show the component

Catalog HASCO

Type Z00

Shank diameter 22

Shank length 85

Dimensions: Standard, mms
Product code: Z00/46/22x85

Dimensions...

Levels...

- **Next**, accept the components selected by the Wizard

- **Finish** to create the Mold base

Mold Base Wizard

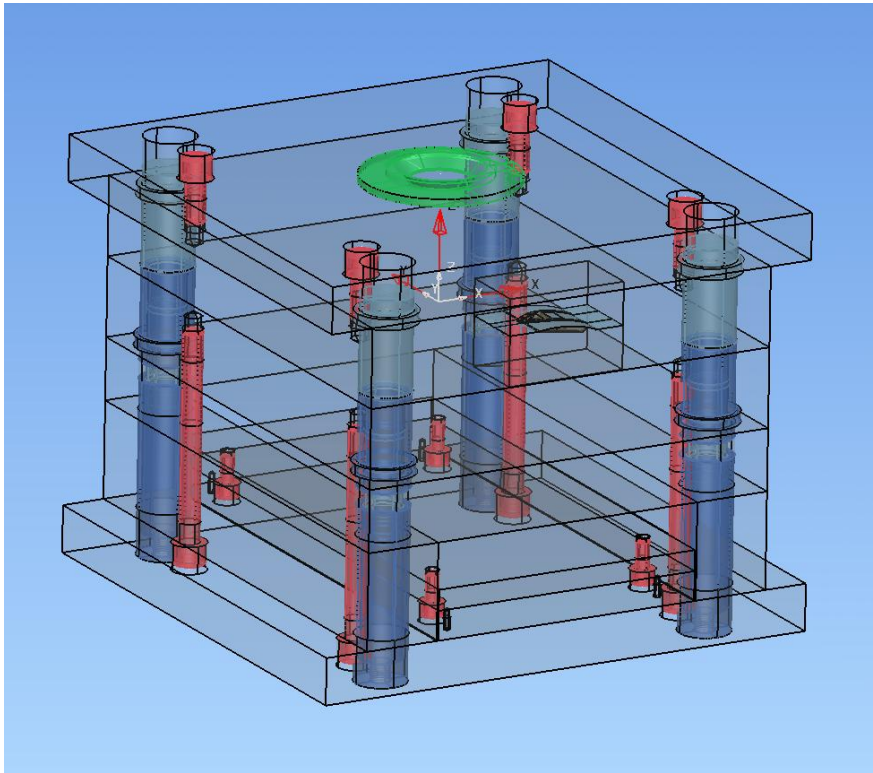
You have successfully completed the Mold Base Wizard

You specified the following parameters for the mold base:

| | |
|------------------------------|------------------------------------|
| Mold base | |
| Catalog | HASCO |
| Type | Standard mold |
| Guide pillars | Fixed half |
| Rotation angle | 0 |
| Width | 246 |
| Length | 296 |
| Dimensions | Standard |
| Cavity plate | |
| Type | K20 |
| Thickness | 46 |
| Material | 1730 |
| Dimensions | Standard |
| Top clamping plate | |
| Type | K10 (overhang in width, loc. ring) |
| Thickness | 27 |
| Material | 1730 |
| Dimensions | Standard |
| Core plate | |
| Type | K20 |
| Thickness | 46 |
| Material | 1730 |
| Dimensions | Standard |
| Support plate | |
| Type | K30 |
| Thickness | 36 |
| Material | 1730 |
| Dimensions | Standard |
| Risers | |
| Type | K40 (standard form) |
| Thickness | 56 |
| Material | 1730 |
| Dimensions | Standard |
| Bottom clamping plate | |
| Type | K11 (overhang in width) |
| Thickness | 27 |

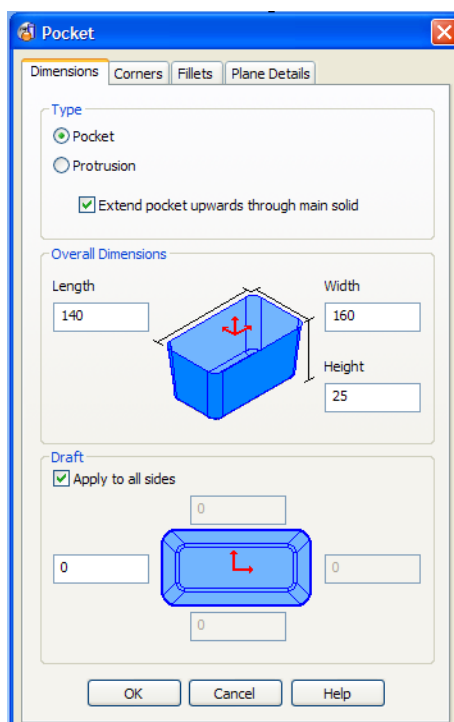
< Back Finish Cancel

The Moldbase is created based on the work plane **Tool CL**.



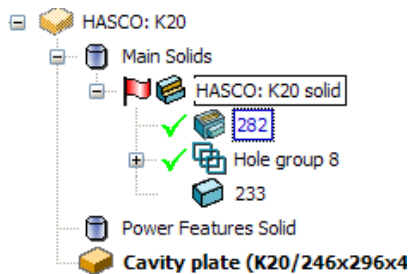
Both the Core and Cavity plates require pockets adding to create a void for the die inserts to fit.

- **Modify Geometry** on each plate and place the pockets at **0 0 0**, using the dimensions shown in the form below.

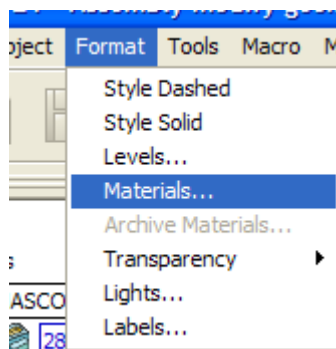


As each of the pockets is created you will see that the material is set to the solid default and does not match the rest of the plate. To rectify this you can set the material for the selected feature.

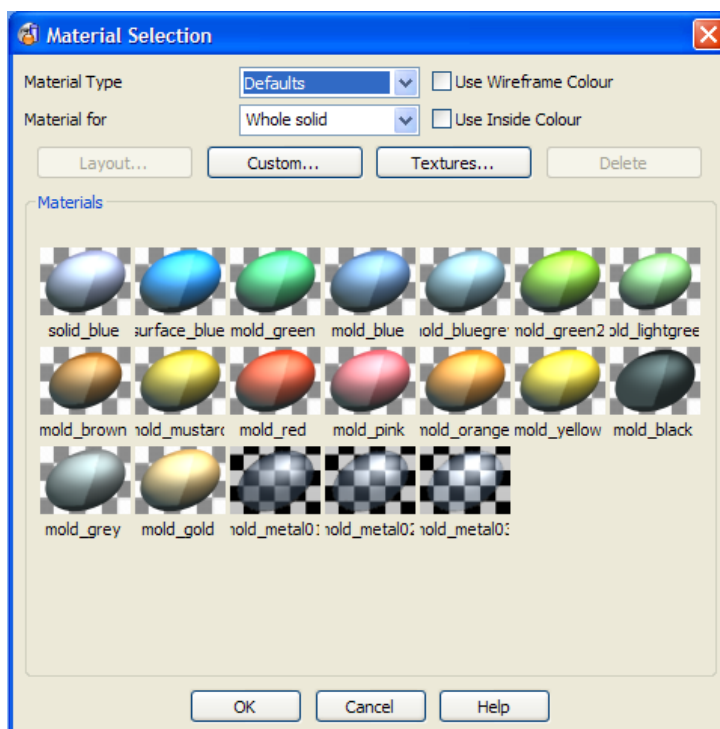
- Select the **Pocket feature** in the solid tree



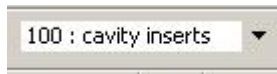
- On the **Main Toolbar** select **Format - Materials**



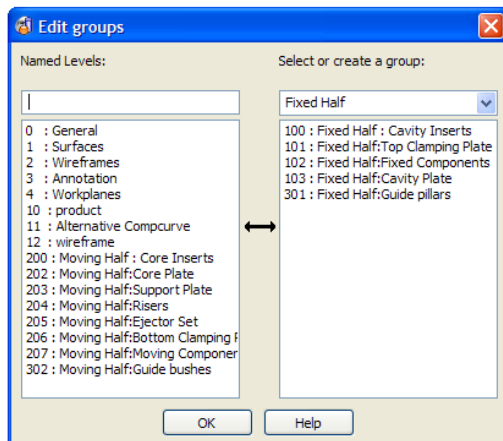
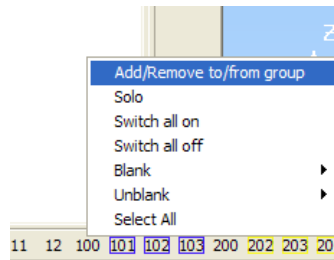
- Select the **Mold_metal03** from the **Defaults material type** and apply to the **selected feature**.



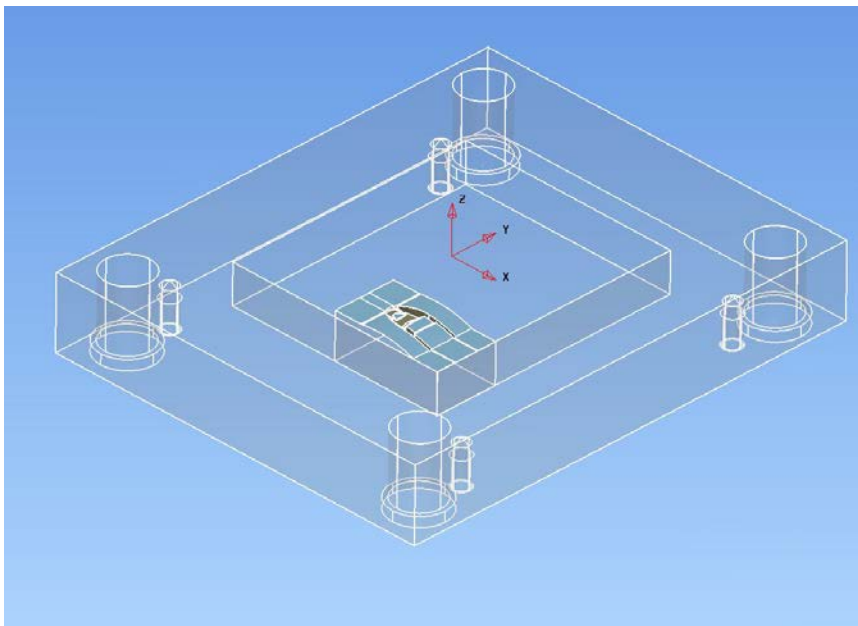
- Repeat for both **Core** and **Cavity plates**
- Create **Level 100** and place the **cavity insert** on this level.




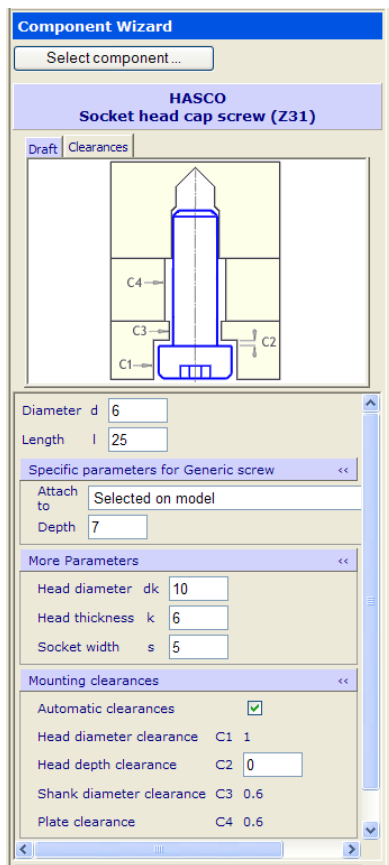
- Add **Level 100** to the **Fixed Half Levels Group**



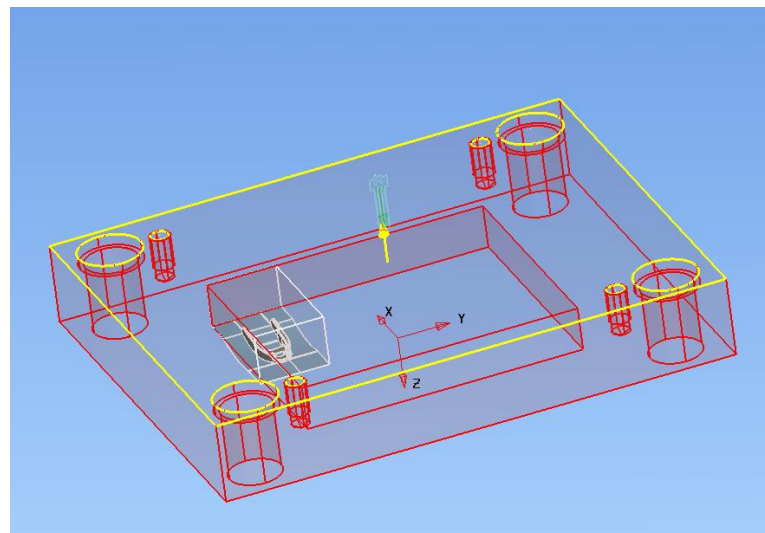
- Repeat for the **Core insert** using **Level 200**
- Display only the **Core plate** and **DW_CORE**



- Use **Component Wizard**  to create **Hasco Z31 Socket head cap screws**.

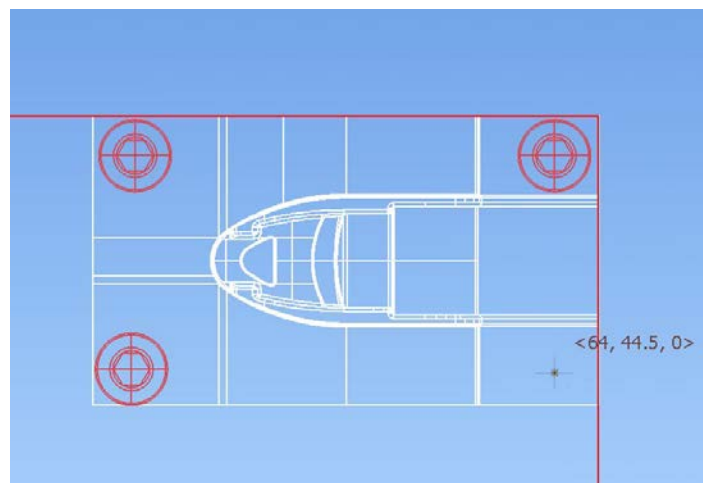


- Input the values shown and select the **core plate** as **master component**



- Place the screws on the **bottom of the core plate**

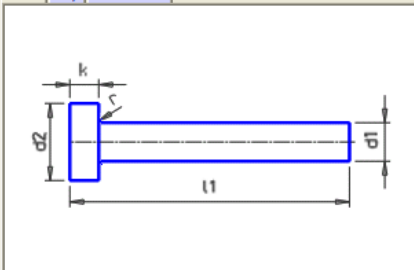
- Position the screws in the **four Corners** of the insert



- Select the **Round and Flat ejector pins** as shown and position in the **core insert**
- Turn on the **Ejector Set Level 205**

HASCO Ejector pin (Z41)

Draft Key Clearances



Shank diameter d1 2.5

Length l1 200

Key None

Function Ejector pin

Specific parameters for Ejector pin <<

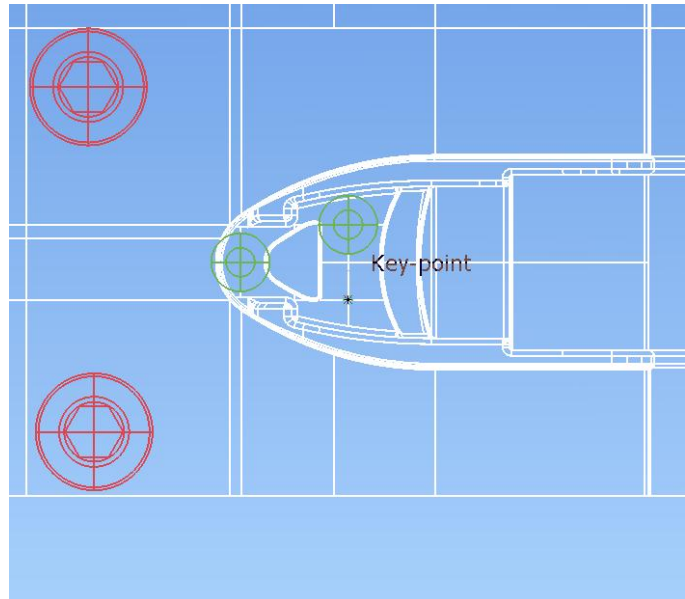
Attach to Ejector base plate (K70/246x296x17/1.17)

Trim pin ☒

Trimming offset 0

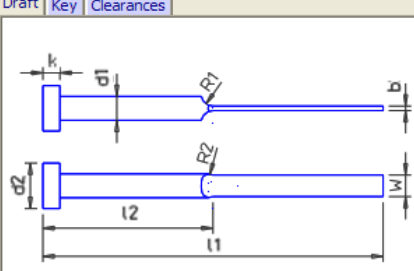
More Parameters >>

Mounting clearances >>



HASCO Flat ejector pin (Z46)

Draft Key Clearances



Body diameter d1 4.2

Length l1 125

Blade thickness b 0.8

Blade width w 3.8

Key None

Specific parameters for Ejector pin <<

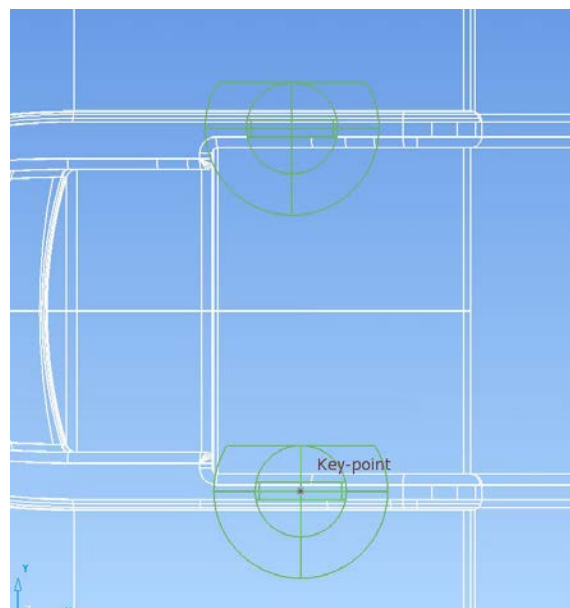
Attach to Ejector base plate (K70/246x296x17/1.17)

Trim pin ☒

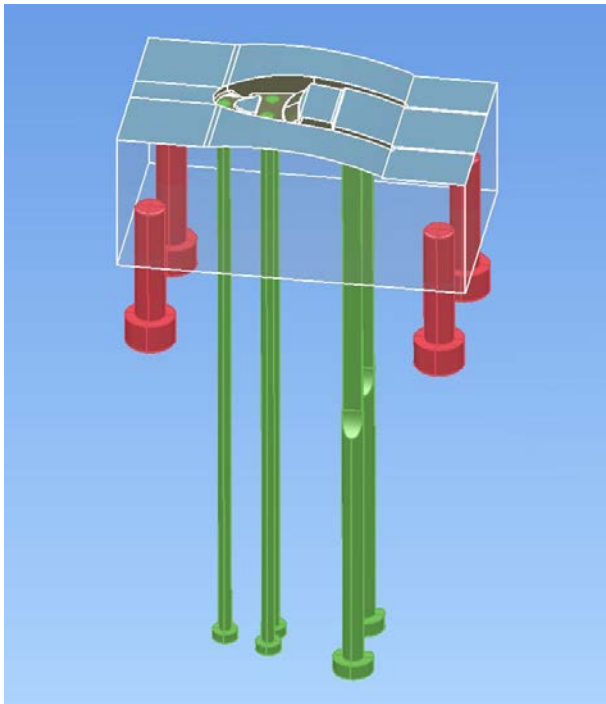
Trimming offset 0

More Parameters >>

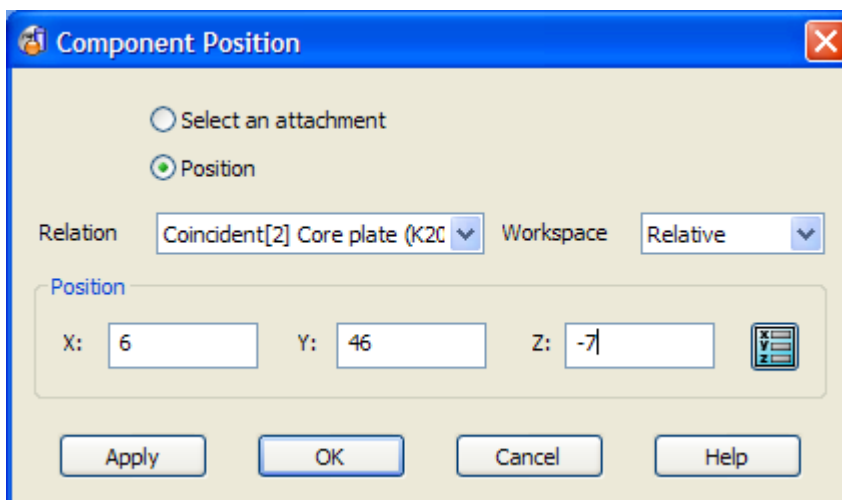
Mounting clearances >>



- You now have a **core inset** with **Screws and Ejector pins**

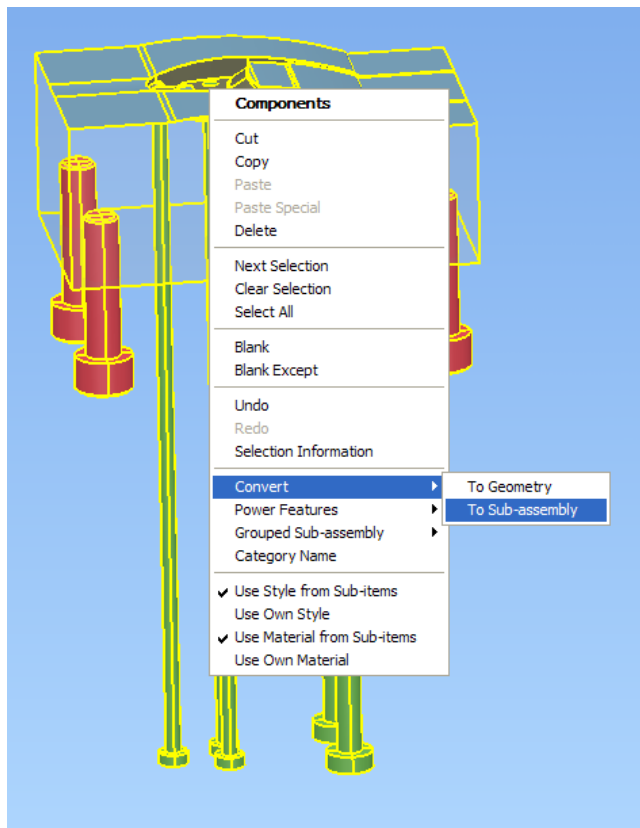


- Using the Assembly Positioning Form move the screws to the following locations:-



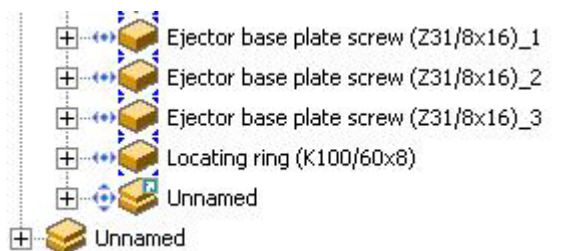
X 6.0 Y 46.0
X 6.0 Y 74.0
X 64.0 Y 46.0
X 64.0 Y 74.0

- **Select the Core Insert, Screws and Pins**, then using RMB menu **Convert to Sub Assembly**



In the Assembly tree you will see both the New Sub Assembly definition and the instance in the main assembly

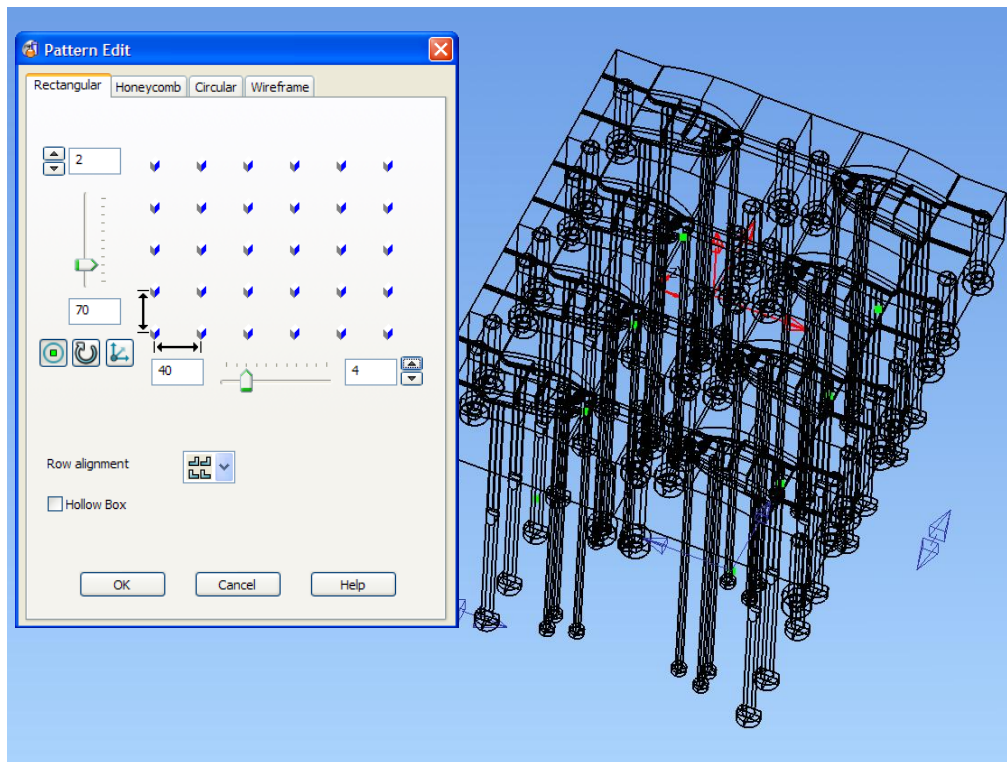
They both have the default name - unnamed



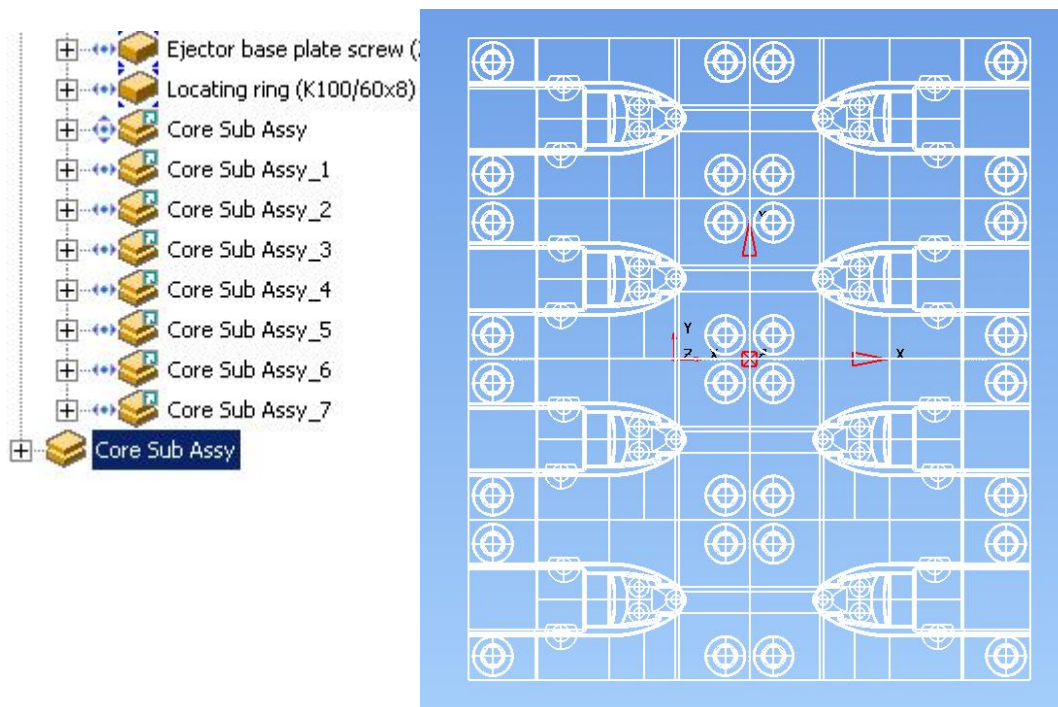
- **Rename both the Definition and the instance** to Core Sub Assy



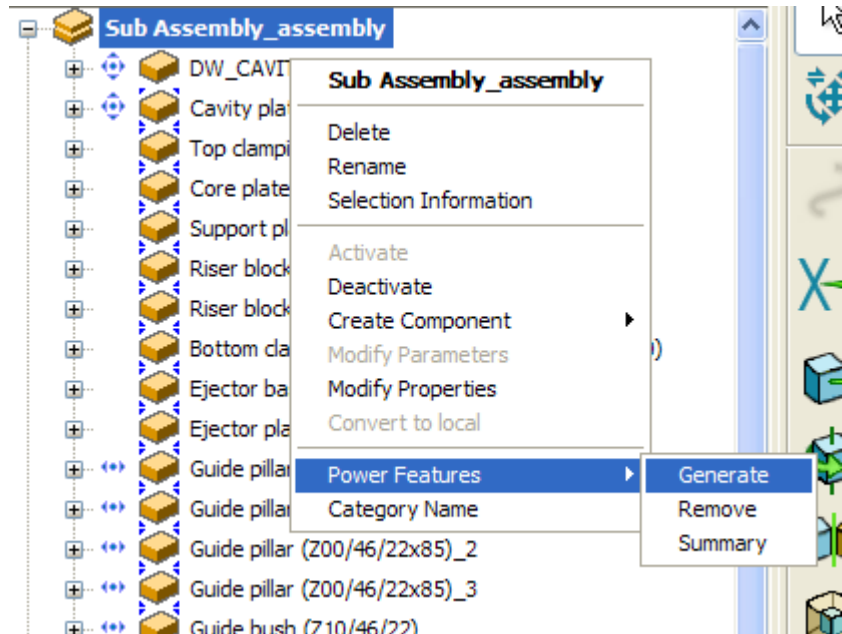
- Select the **Sub Assembly** instance and using the **pattern** operation; create pins and screws for all of the other inserts. Note the dimension below.



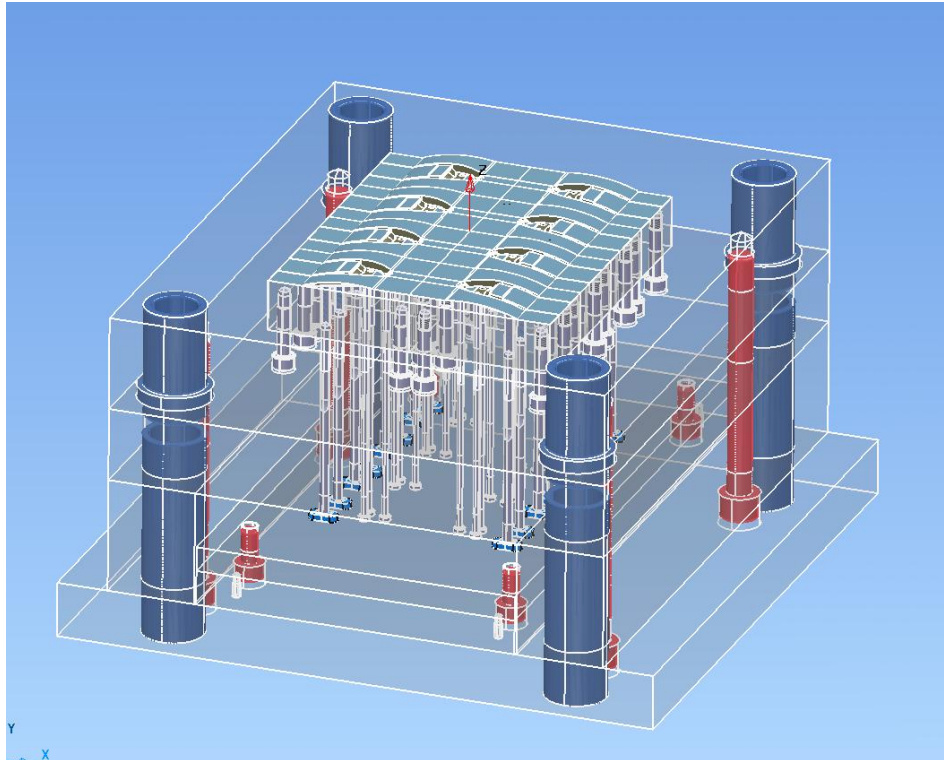
The Assembly tree now contains the multiple instances of the **Core components Sub Assembly**



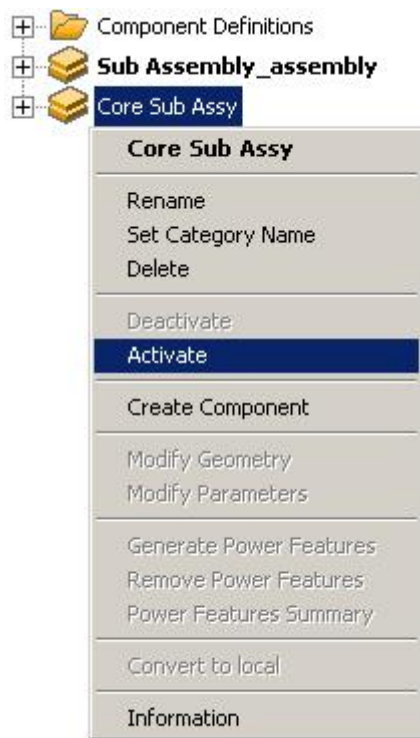
- Select the first Sub Assembly – **Core Sub Assy**, use RMB menu to **Generate Power Features**



The Moldbase and inserts contain the 'Fit' holes.

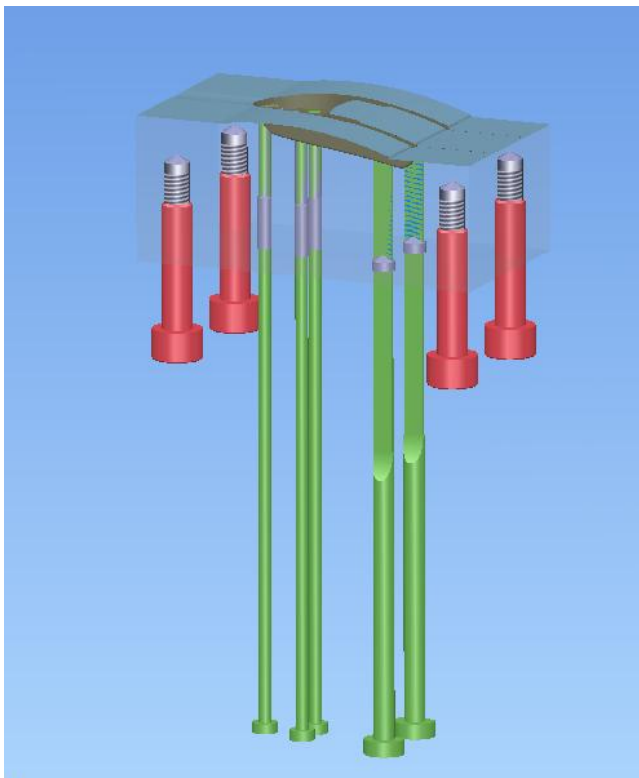


- In the Assembly Tree **Activate** the **Core Sub Assy**



Once the Sub Assembly is Active then the parts can be individually modified.

The effect of using the sub assembly means that a modification to one component is updated to all other instances in the main assembly.



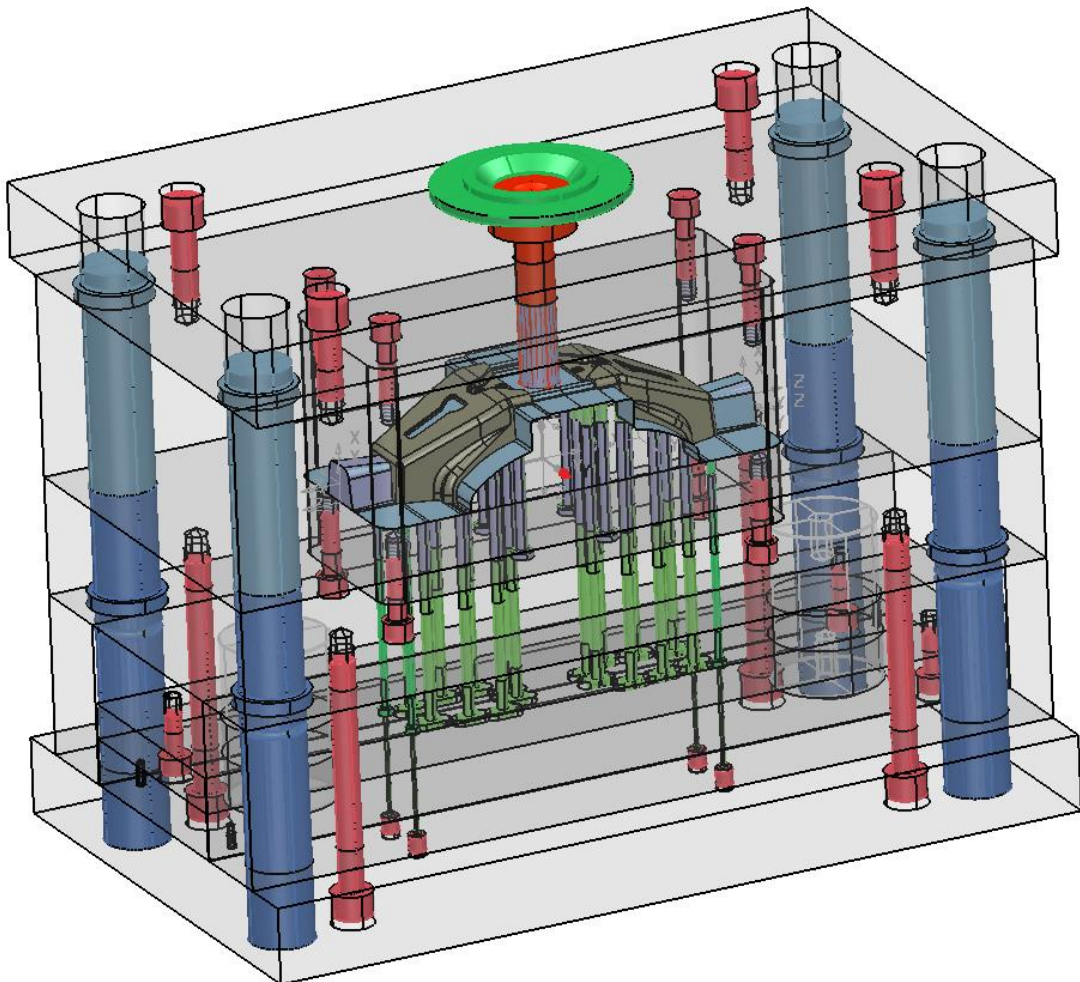
13. Drafting

PS-Draft has several new features:-

- Angled Stepped Sections
- Scrap Sections
- Hidden Line feature removal
- B.O.M. update
- Hole Schedules with Excel
- Automatic Creation of component drawings

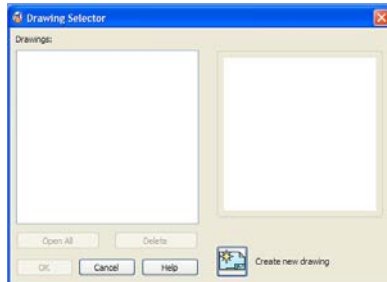
All of the above options will be used to create 2D drawings for tool design created in an earlier exercise.

Open Model – Drafting.psmodel

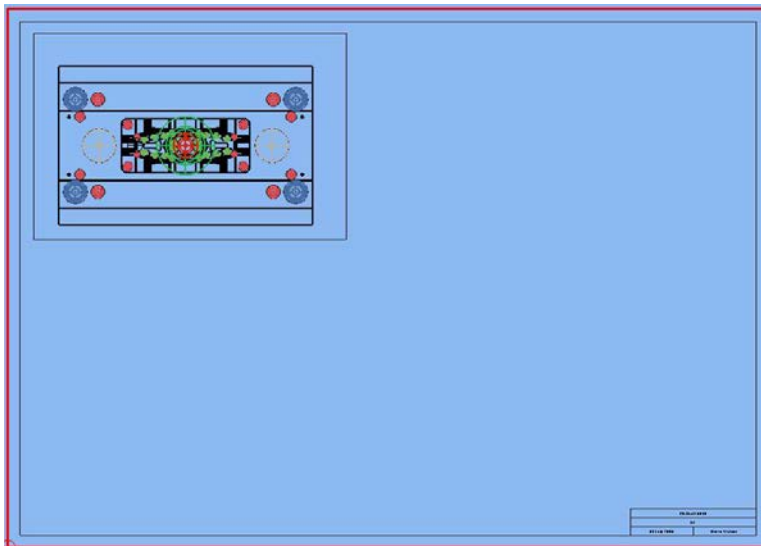


The first drawing will be the General Assembly or G.A.

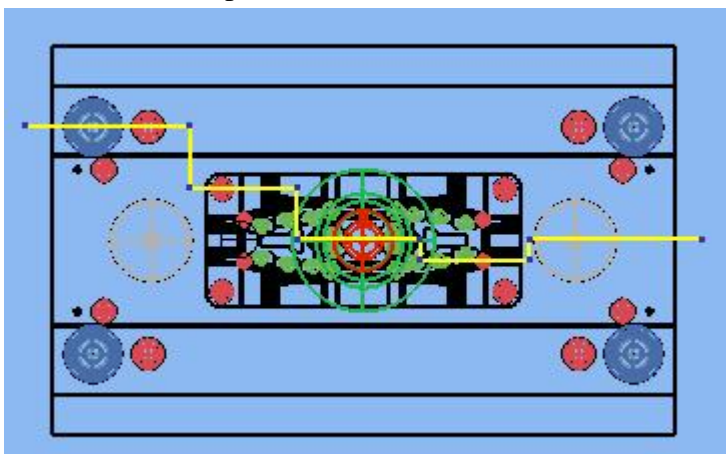
- Enter Drafting Mode,  then **Create new drawing**



- Ensure that both **Fixed** and **Moving half group levels** are turned on.
- Modify the drawing and make the **size A0**, Name the drawing **“GA”**
- Create a **Plan view** and place as indicted below.



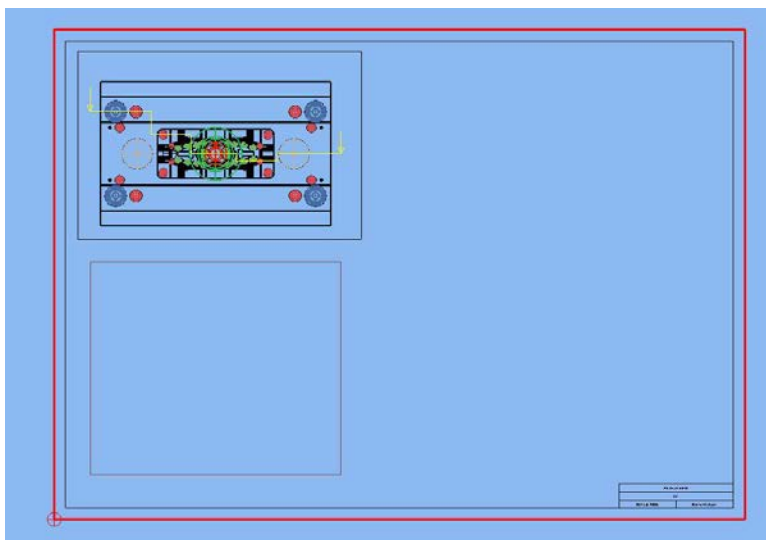
- Create the composite curve below.



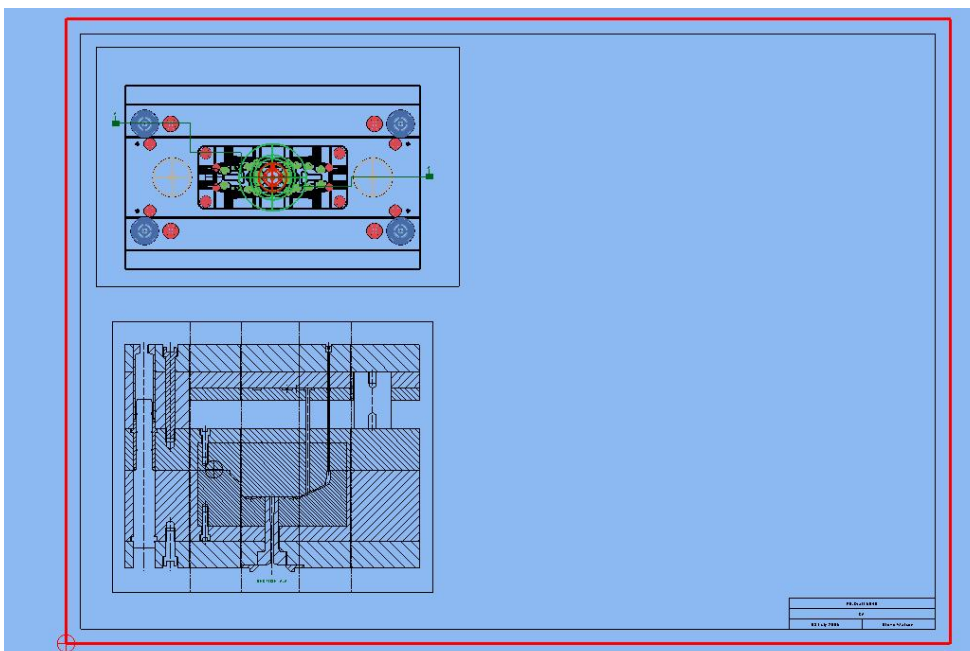
- Select the Comp curve and pick **Create section view with background**



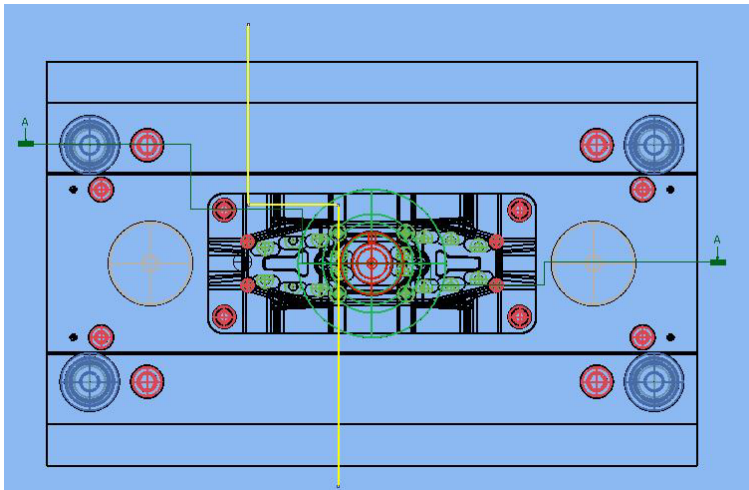
- Place the section view as indicated below



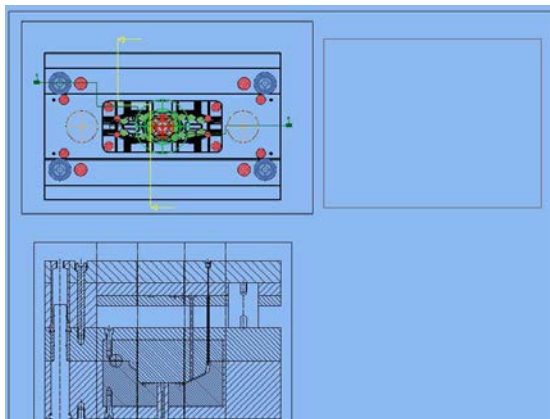
Section 'AA'



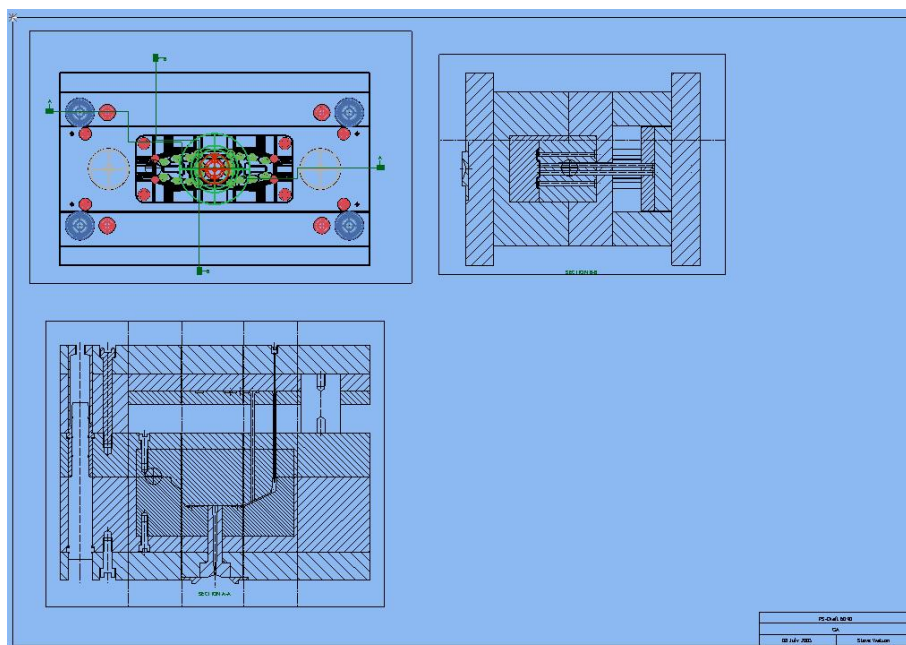
- Create a comp curve as indicated below, select the curve and create a **section view with background**



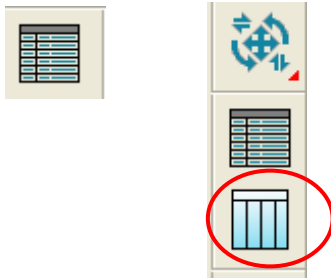
- Place the section view as indicated below.



Section 'BB'



- Next, Create a **Bill of Materials** from the Schedule toolbar



The B.O.M. form will be created.

| Views | Label | Part | Description | Catalogue | Material | Diameter | Length | Width | Height | Quantity |
|-------|-------|--|--------------------------|-----------|----------|----------|--------|--------|--------|----------|
| | | A 1.5-200 | Core pin | DME | | 3.00 | 164.97 | | | 4 |
| | | DW_CAVITY | | | | | 200.00 | 85.00 | 112.00 | 1 |
| | | DW_CORE | | | | | 200.00 | 85.00 | 112.00 | 1 |
| | | K10/196 396/36/1730 | Top clamping plate | HASCO | 1730 | | 396.00 | 246.00 | 36.00 | 1 |
| | | K100/60x8 | Locating ring | HASCO | | 90.00 | 8.00 | | | 1 |
| | | K11/196 396/36/1730 | Bottom clamping plate | HASCO | 1730 | | 396.00 | 246.00 | 36.00 | 1 |
| | | K20/196 396/56/1730 | Core plate | HASCO | 1730 | | 396.00 | 196.00 | 56.00 | 1 |
| | | K20/196 396/96/1730 | Cavity plate | HASCO | 1730 | | 396.00 | 196.00 | 96.00 | 1 |
| | | K40/196 396/76/1730 | Riser block | HASCO | 1730 | | 396.00 | 43.00 | 76.00 | 2 |
| | | K60/196 396/17/1730 | Ejector plate | HASCO | 1730 | | 396.00 | 108.00 | 17.00 | 1 |
| | | K70/196 396/22/1730 | Ejector base plate | HASCO | 1730 | | 396.00 | 108.00 | 22.00 | 1 |
| | | SW_CORE_Definition | SW_CORE1 | | | | 23.34 | 23.94 | 17.93 | 2 |
| | | Sprue bushing (18 73.3733 3 15.5) User defined | | | | 38.00 | 91.37 | | | 1 |
| | | Z00/96/22x95 | Guide pillar | HASCO | | 35.00 | 200.00 | | | 4 |
| | | Z10/56/22 | Guide bush | HASCO | | 35.00 | 65.00 | | | 4 |
| | | Z20/30x100 | Centering sleeve | HASCO | | 30.00 | 100.00 | | | 4 |
| | | Z31/10x40 | Socket head cap screw | HASCO | | 16.00 | 50.00 | | | 2 |
| | | Z31/12x110 | Moving half screw | HASCO | | 18.00 | 122.00 | | | 4 |
| | | Z31/12x35 | Fixed half screw | HASCO | | 18.00 | 47.00 | | | 4 |
| | | Z31/8x20 | Ejector base plate screw | HASCO | | 13.00 | 28.00 | | | 4 |
| | | Z31/8x30 | Socket head cap screw | HASCO | | 13.00 | 38.00 | | | 8 |
| | | Z35/8x8 | Hexagon socket set screw | HASCO | | 8.00 | 8.00 | | | 4 |
| | | Z40/5x160 | Core pin | HASCO | | 10.00 | 75.75 | | | 4 |
| | | Z41/5x250 | Ejector pin | HASCO | | 10.00 | 133.97 | | | 16 |
| | | Z45/3x1.5x125 | Ejector sleeve | HASCO | | 6.00 | 111.89 | | | 4 |
| | | Z57/50x76 | Support pillar | HASCO | | 50.00 | 76.00 | | | 2 |

1. Select a view to balloon components in GA - V1

2. Select the components to balloon by clicking in the list or on the drawing

3. Optionally, click to manually balloon any selected components that shouldn't be automatically ballooned

4. Return to stage 1 to select other views if necessary

5. Select a drawing to create the bill of materials on GA

6. Create the bill of materials on the drawing or save it to disk

OK Help

The B.O.M. is displayed with the default columns in view, there are many more columns of information that are currently hidden.

To change the columns that are displayed use the RMB on the column header, you can select the information that is displayed. Once selected the columns will be 'remembered'.

The label column is blank as none have been created yet.

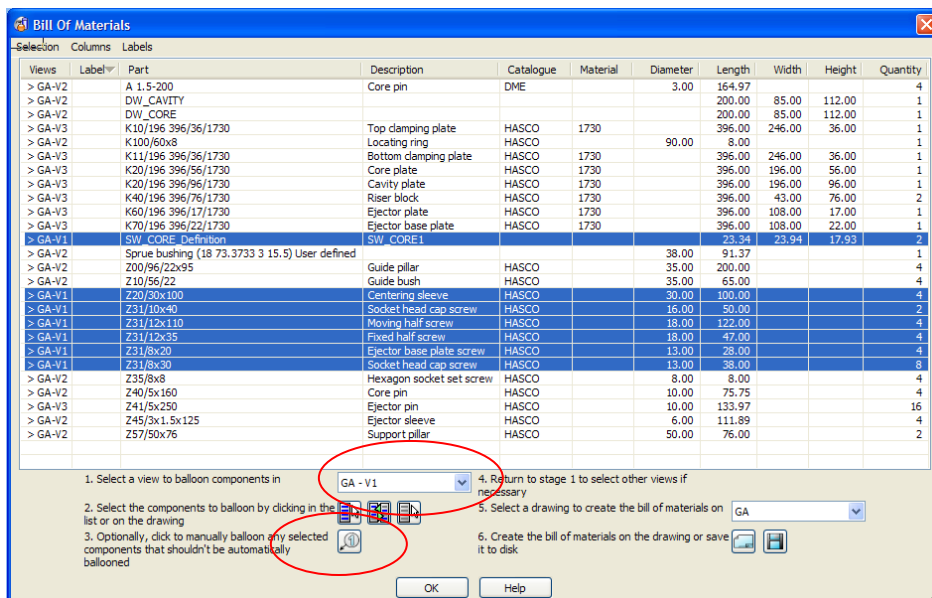
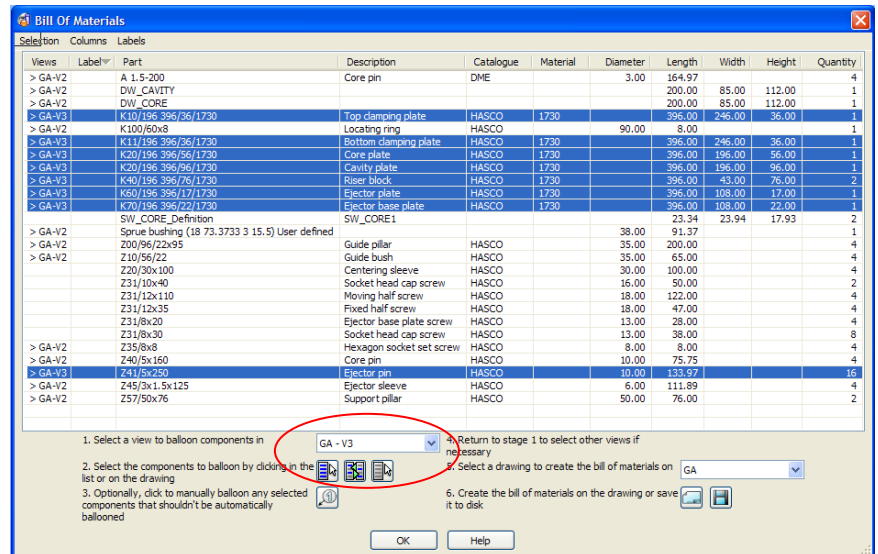
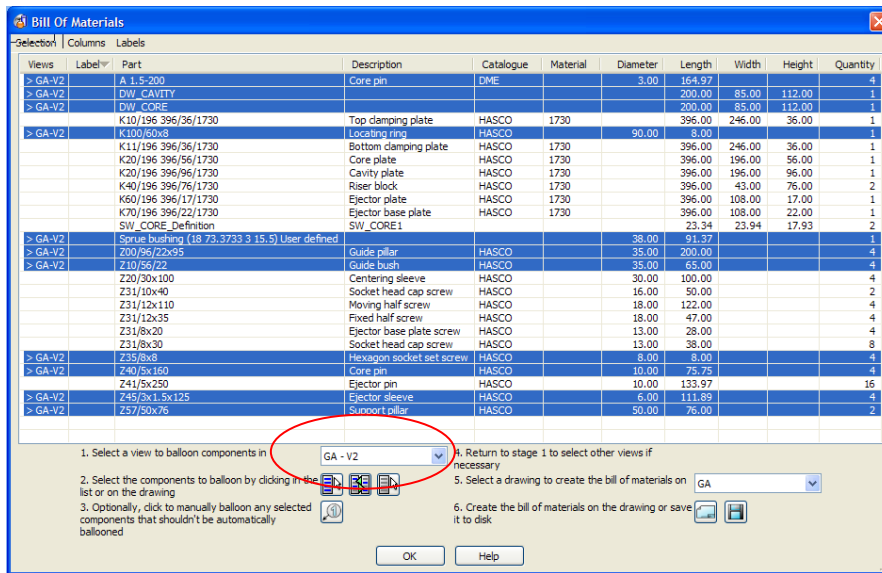
All of the labels can be created and placed on the drawing automatically. However the position of the automatic balloons is not usually desirable.

In this case the balloons will be created manually.

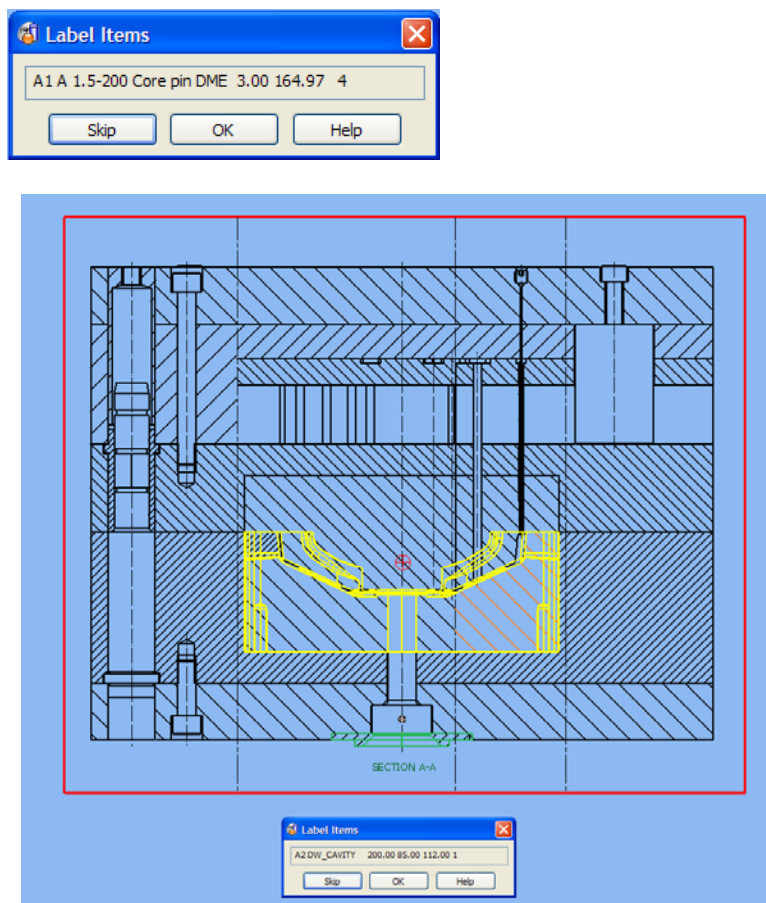
Note:

The balloons / labels will be created as per the settings in **tools – options- balloon**

- Select each **component** and the **view** in which the balloon will be created, then pick **Balloon Items**

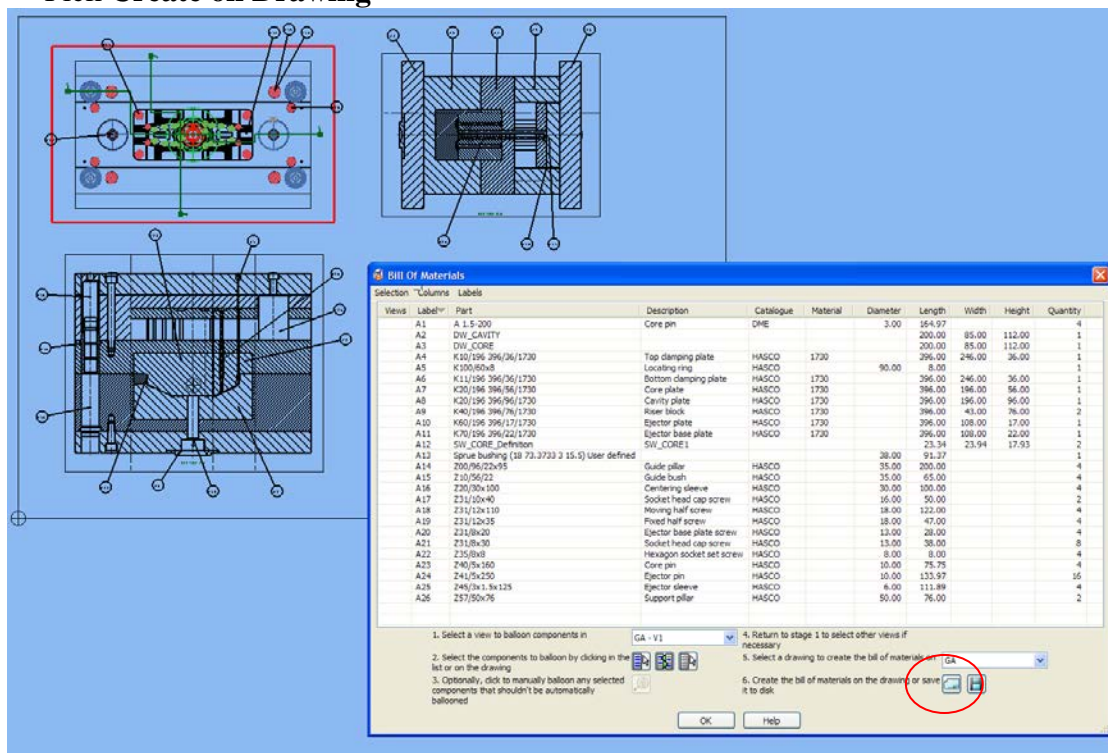


The Label Items form appears, the item that is shown will be highlighted to assist picking.



When all of the balloons are positioned the B.O.M. form reappears,

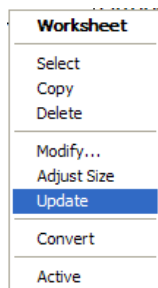
- **Pick Create on Drawing**



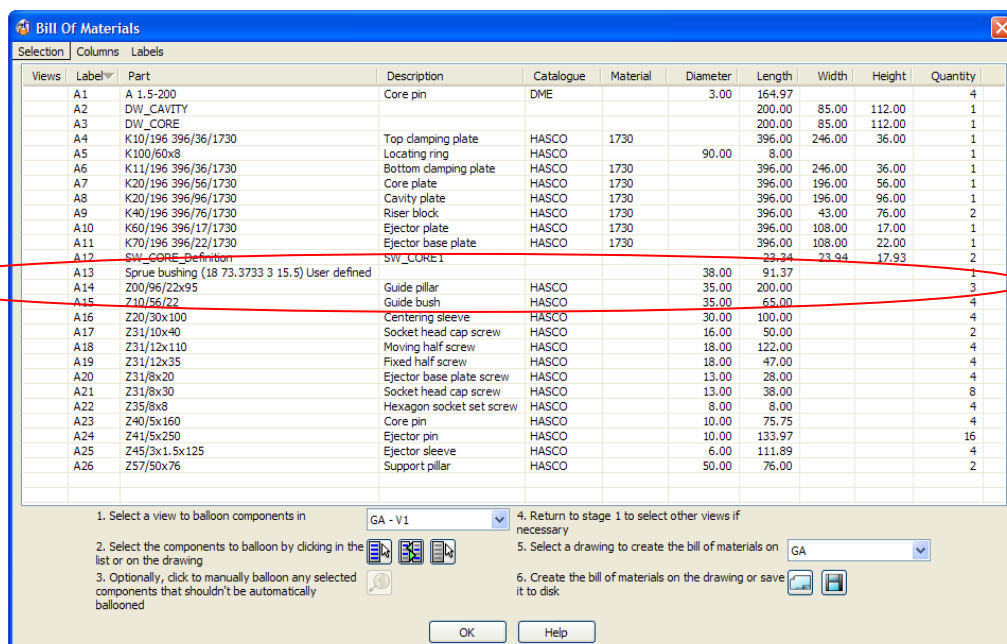
The B.O.M. is created on the drawing as a view; this is an OLE link to Microsoft Excel

| Label | Part | Description | Catalogue | Material | Diameter | Length | Width | Height | Quantity |
|-------|--|--------------------------|-----------|----------|----------|--------|--------|--------|----------|
| A1 | A 1.5-200 | Core pin | DME | | 3.00 | 164.97 | | | 4 |
| A2 | DW_CAVITY | | | | | 200.00 | 85.00 | 112.00 | 1 |
| A3 | DW_CORE | | | | | 200.00 | 85.00 | 112.00 | 1 |
| A4 | K10/196 396/36/1730 | Top clamping plate | HASCO | 1730 | | 396.00 | 246.00 | 36.00 | 1 |
| A5 | K100/60x8 | Locating ring | HASCO | | 90.00 | 8.00 | | | 1 |
| A6 | K11/196 396/36/1730 | Bottom clamping plate | HASCO | 1730 | | 396.00 | 246.00 | 36.00 | 1 |
| A7 | K20/196 396/56/1730 | Core plate | HASCO | 1730 | | 396.00 | 196.00 | 56.00 | 1 |
| A8 | K20/196 396/96/1730 | Cavity plate | HASCO | 1730 | | 396.00 | 196.00 | 96.00 | 1 |
| A9 | K40/196 396/76/1730 | Riser block | HASCO | 1730 | | 396.00 | 43.00 | 76.00 | 2 |
| A10 | K60/196 396/17/1730 | Ejector plate | HASCO | 1730 | | 396.00 | 108.00 | 17.00 | 1 |
| A11 | K70/196 396/22/1730 | Ejector base plate | HASCO | 1730 | | 396.00 | 108.00 | 22.00 | 1 |
| A12 | SW_CORE_Definition | SW_CORE1 | | | | 23.34 | 23.94 | 17.93 | 2 |
| A13 | Sprue bushing (18 73.3733 3 15.5) User defined | | | | 38.00 | 91.37 | | | 1 |
| A14 | Z00/96/22x95 | Guide pillar | HASCO | | 35.00 | 200.00 | | | 4 |
| A15 | Z10/56/22 | Guide bush | HASCO | | 35.00 | 65.00 | | | 4 |
| A16 | Z20/30x100 | Centering sleeve | HASCO | | 30.00 | 100.00 | | | 4 |
| A17 | Z31/10x40 | Socket head cap screw | HASCO | | 16.00 | 50.00 | | | 2 |
| A18 | Z31/12x110 | Moving half screw | HASCO | | 18.00 | 122.00 | | | 4 |
| A19 | Z31/12x35 | Fixed half screw | HASCO | | 18.00 | 47.00 | | | 4 |
| A20 | Z31/8x20 | Ejector base plate screw | HASCO | | 13.00 | 28.00 | | | 4 |
| A21 | Z31/8x30 | Socket head cap screw | HASCO | | 13.00 | 38.00 | | | 8 |
| A22 | Z35/8x8 | Hexagon socket set screw | HASCO | | 8.00 | 8.00 | | | 4 |
| A23 | Z40/5x160 | Core pin | HASCO | | 10.00 | 75.75 | | | 4 |
| A24 | Z41/5x250 | Ejector pin | HASCO | | 10.00 | 133.97 | | | 16 |
| A25 | Z45/3x1.5x125 | Ejector sleeve | HASCO | | 6.00 | 111.89 | | | 4 |
| A26 | Z57/50x76 | Support pillar | HASCO | | 50.00 | 76.00 | | | 2 |

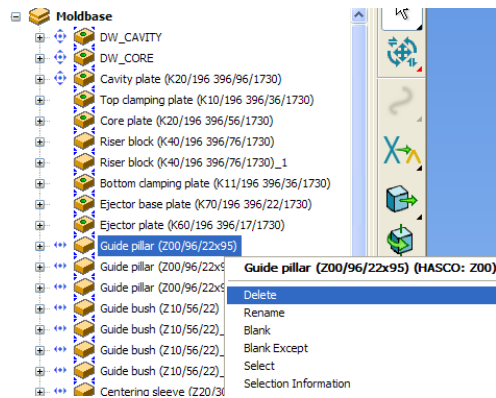
- Return to Assembly model and delete the component **Guide pillar (Z00/96/22x95)_2**
- Re open the drawing GA and in the B.O.M. worksheet select **update** from the RMB menu



Raise the B.O.M. form from the schedule toolbar, you can see that the label has not been removed; this is due to the Guide pillar that was deleted not being the first instance.



- Return again to the Assembly and delete the component **Guide pillar (Z00/96/22x95)**



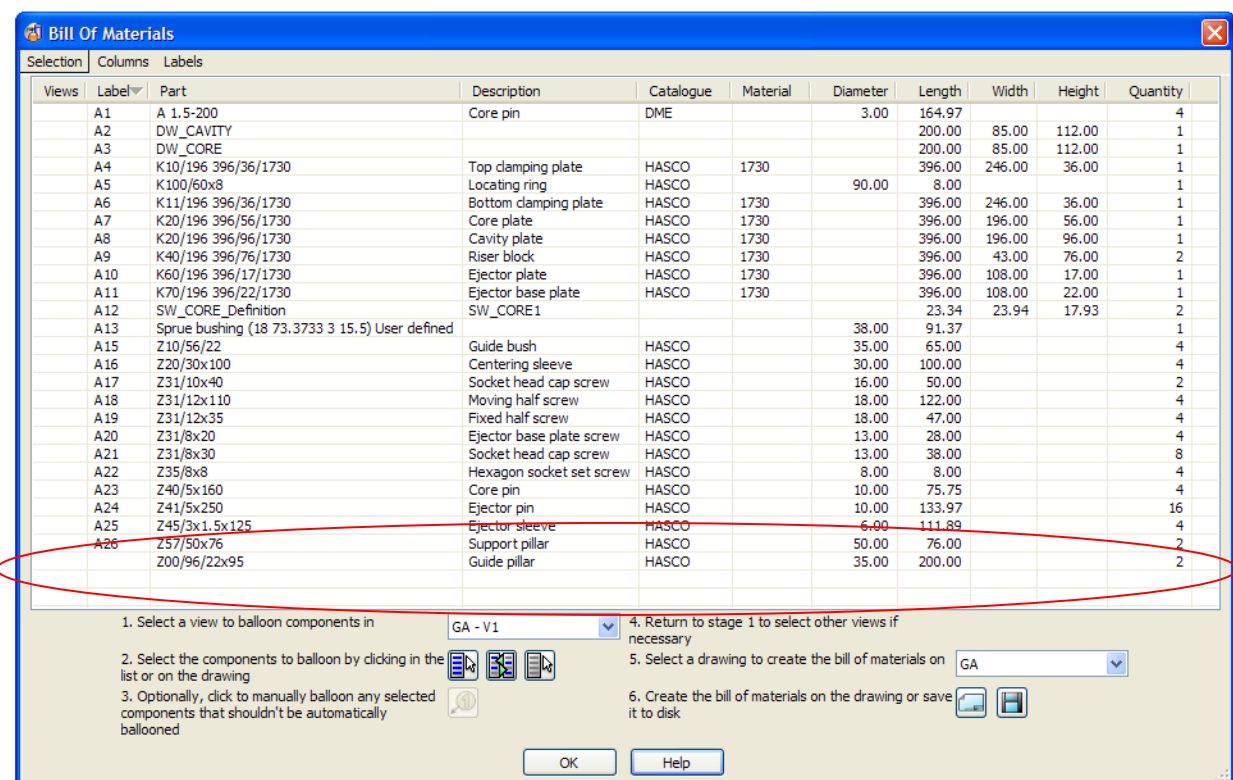
As this Guide Pillar is the first instance the label was attached to this component in the B.O.M.

Consequently the link is now broken between the label and component, this must currently be restored manually.

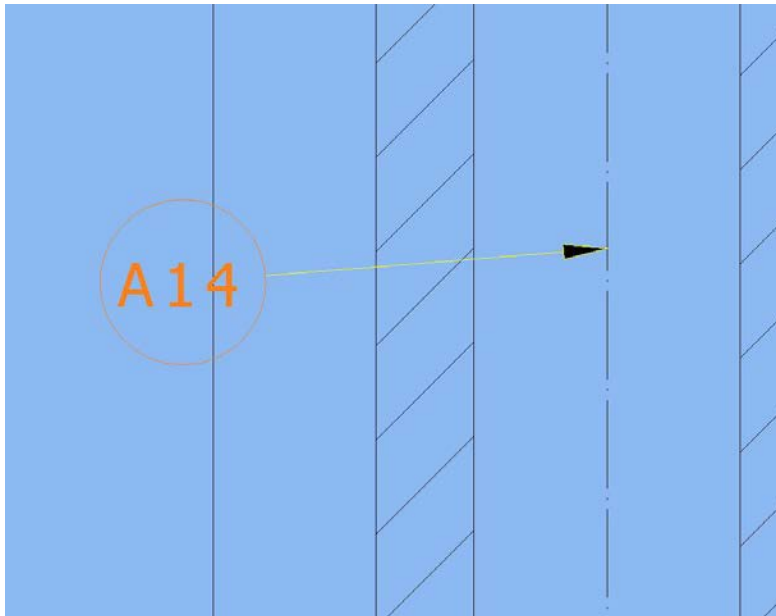
Future versions of PS-Draft will update automatically

- Raise the B.O.M. form from the schedule toolbar

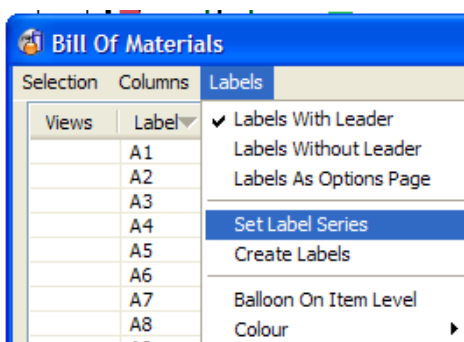
The guide pillar has now moved to the bottom of the form as the B.O.M. is sorted by Label at this stage.



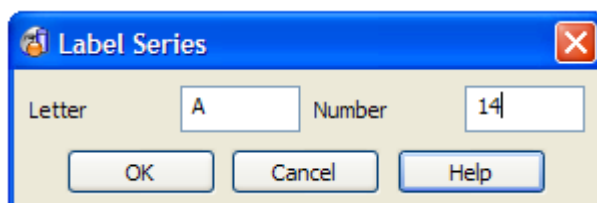
- **Dismiss** the form and **delete** the now unused **A14 label and leader line**



- Raise the B.O.M. form from the schedule toolbar, select the **Guide Pillar** and using RMB menu **Set Label Series**



- The label that is required is A14, so set the series accordingly.



- Manually create a new **A14** label and balloon. **Dismiss** the Form

| Views | Label | Part | Description | Catalogue | Material | Diameter | Length | Width | Height | Quantity |
|-------|-------|--|--------------------------|-----------|----------|----------|--------|--------|--------|----------|
| | A1 | A 1.5-200 | Core pin | DME | | 3.00 | 164.97 | | | 4 |
| | A2 | DW_CAVITY | | | | | 200.00 | 85.00 | 112.00 | 1 |
| | A3 | DW_CORE | | | | | 200.00 | 85.00 | 112.00 | 1 |
| | A4 | K10/196 396/36/1730 | Top clamping plate | HASCO | 1730 | | 396.00 | 246.00 | 36.00 | 1 |
| | A5 | K100/60x8 | Locating ring | HASCO | | 90.00 | 8.00 | | | 1 |
| | A6 | K11/196 396/36/1730 | Bottom clamping plate | HASCO | 1730 | | 396.00 | 246.00 | 36.00 | 1 |
| | A7 | K20/196 396/56/1730 | Core plate | HASCO | 1730 | | 396.00 | 196.00 | 56.00 | 1 |
| | A8 | K20/196 396/96/1730 | Cavity plate | HASCO | 1730 | | 396.00 | 196.00 | 96.00 | 1 |
| | A9 | K40/196 396/76/1730 | Riser block | HASCO | 1730 | | 396.00 | 43.00 | 76.00 | 2 |
| | A10 | K60/196 396/17/1730 | Ejector plate | HASCO | 1730 | | 396.00 | 108.00 | 17.00 | 1 |
| | A11 | K70/196 396/22/1730 | Ejector base plate | HASCO | 1730 | | 396.00 | 108.00 | 22.00 | 1 |
| | A12 | SW_CORE_Definition | SW_CORE1 | | | | 23.34 | 23.94 | 17.93 | 2 |
| | A13 | Sprue bushing (18 73.3733 3 15.5) User defined | | | | 38.00 | 91.37 | | | 1 |
| | A14 | Z00/96/22x95 | Guide pillar | HASCO | | 35.00 | 200.00 | | | 2 |
| | A15 | Z10/56/22 | Guide bush | HASCO | | 35.00 | 65.00 | | | 4 |
| | A16 | Z20/30x100 | Centering sleeve | HASCO | | 30.00 | 100.00 | | | 4 |
| | A17 | Z31/10x40 | Socket head cap screw | HASCO | | 16.00 | 50.00 | | | 2 |
| | A18 | Z31/12x110 | Moving half screw | HASCO | | 18.00 | 122.00 | | | 4 |
| | A19 | Z31/12x35 | Fixed half screw | HASCO | | 18.00 | 47.00 | | | 4 |
| | A20 | Z31/8x20 | Ejector base plate screw | HASCO | | 13.00 | 28.00 | | | 4 |
| | A21 | Z31/8x30 | Socket head cap screw | HASCO | | 13.00 | 38.00 | | | 8 |
| | A22 | Z35/8x8 | Hexagon socket set screw | HASCO | | 8.00 | 8.00 | | | 4 |
| | A23 | Z40/5x160 | Core pin | HASCO | | 10.00 | 75.75 | | | 4 |
| | A24 | Z41/5x250 | Ejector pin | HASCO | | 10.00 | 133.97 | | | 16 |
| | A25 | Z45/3x1.5x125 | Ejector sleeve | HASCO | | 6.00 | 111.89 | | | 4 |
| | A26 | Z57/50x76 | Support pillar | HASCO | | 50.00 | 76.00 | | | 2 |

1. Select a view to balloon components in GA - V2

2. Select the components to balloon by clicking in the list or on the drawing

3. Optionally, click to manually balloon any selected components that shouldn't be automatically ballooned

4. Return to stage 1 to select other views if necessary

5. Select a drawing to create the bill of materials on GA

6. Create the bill of materials on the drawing or save it to disk

OK Help

- Update then **Adjust the size** of the Worksheet on the drawing

| Label | Part | Description | C |
|-------|--|--------------------------|----|
| A1 | A 1.5-200 | Core pin | D |
| A2 | DW_CAVITY | | |
| A3 | DW_CORE | | |
| A4 | K10/196 396/36/1730 | Top clamping plate | H. |
| A5 | K100/60x8 | Locating ring | H. |
| A6 | K11/196 396/36/1730 | Bottom clamping plate | H. |
| A7 | K20/196 396/56/1730 | Core plate | H. |
| A8 | K20/196 396/96/1730 | Cavity plate | H. |
| A9 | K40/196 396/76/1730 | Riser block | H. |
| A10 | K60/196 396/17/1730 | Ejector plate | H. |
| A11 | K70/196 396/22/1730 | Ejector base plate | H. |
| A12 | SW_CORE_Definition | SW_CORE1 | |
| A13 | Sprue bushing (18 73.3733 3 15.5) User defined | | |
| A14 | Z00/96/22x95 | Guide pillar | H. |
| A15 | Z10/56/22 | Guide bush | H. |
| A16 | Z20/30x100 | Centering sleeve | H. |
| A17 | Z31/10x40 | Socket head cap screw | H. |
| A18 | Z31/12x110 | Moving half screw | H. |
| A19 | Z31/12x35 | Fixed half screw | H. |
| A20 | Z31/8x20 | Ejector base plate screw | H. |
| A21 | Z31/8x30 | Socket head cap screw | H. |
| A22 | Z35/8x8 | Hexagon socket set screw | H. |
| A23 | Z40/5x160 | Core pin | H. |
| A24 | Z41/5x250 | Ejector pin | H. |
| A25 | Z45/3x1.5x125 | Ejector sleeve | H. |
| A26 | Z57/50x76 | Support pillar | H. |

Worksheet

Select

Copy

Delete

Modify...

Adjust Size

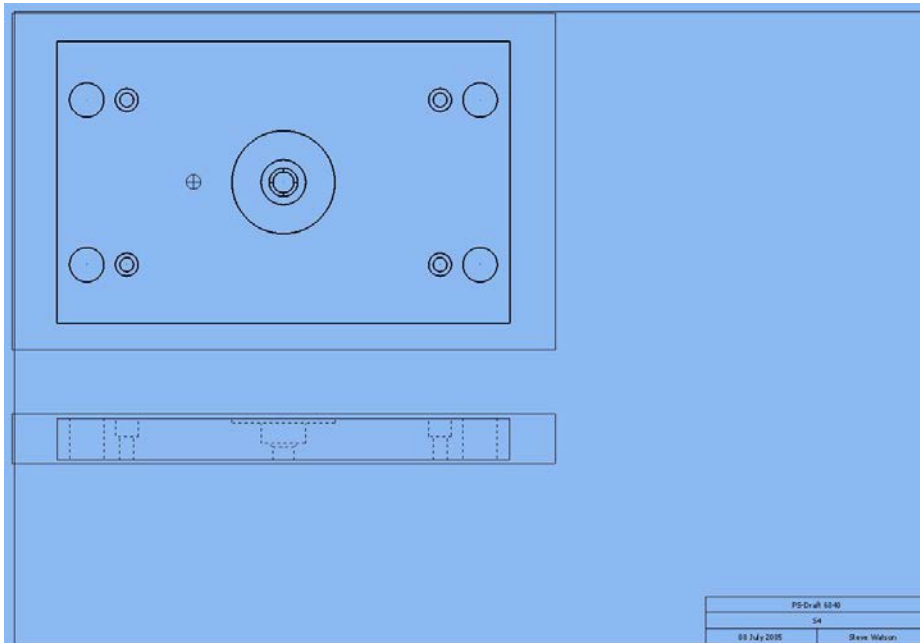
Update

Convert

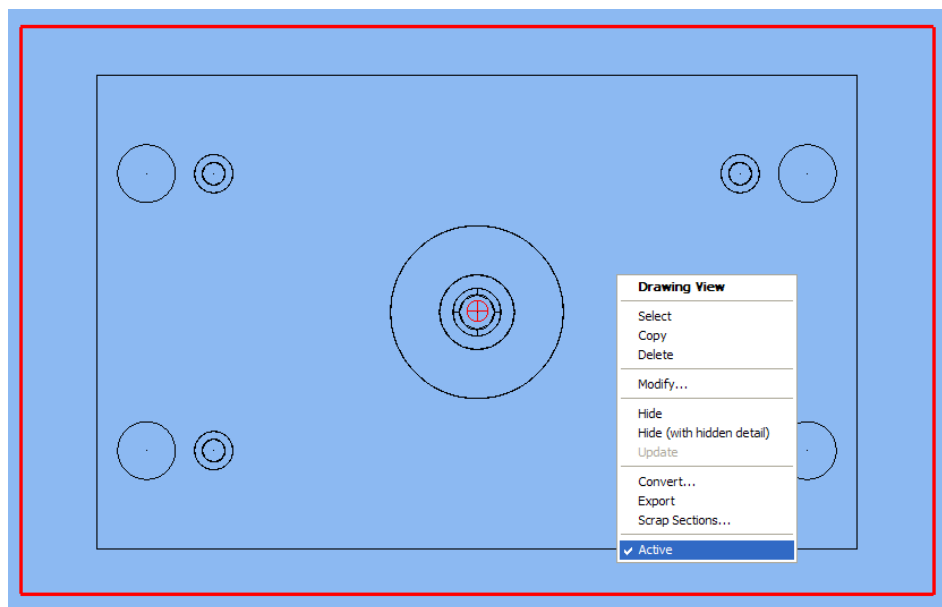
Active

Hole Schedules

- Create a drawing for the **Top Clamping plate**, with a plan and hidden line side view



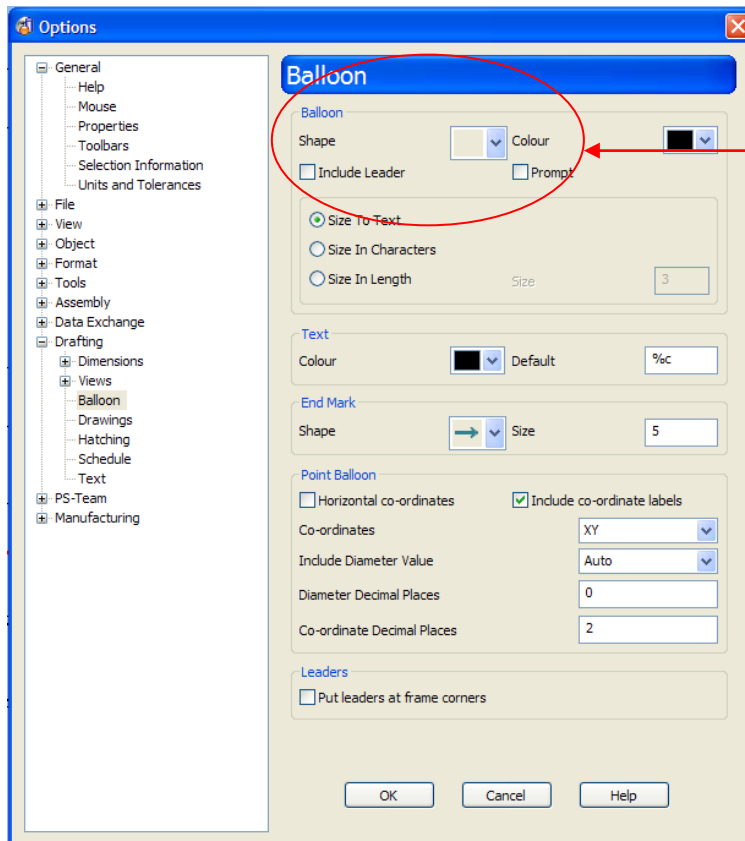
- **Activate** the Plan view



- Start the **Schedule** operation from the Drafting toolbar



Labels will be created for the schedule relative to the current balloon settings in Tools – Options – Drafting - Balloon

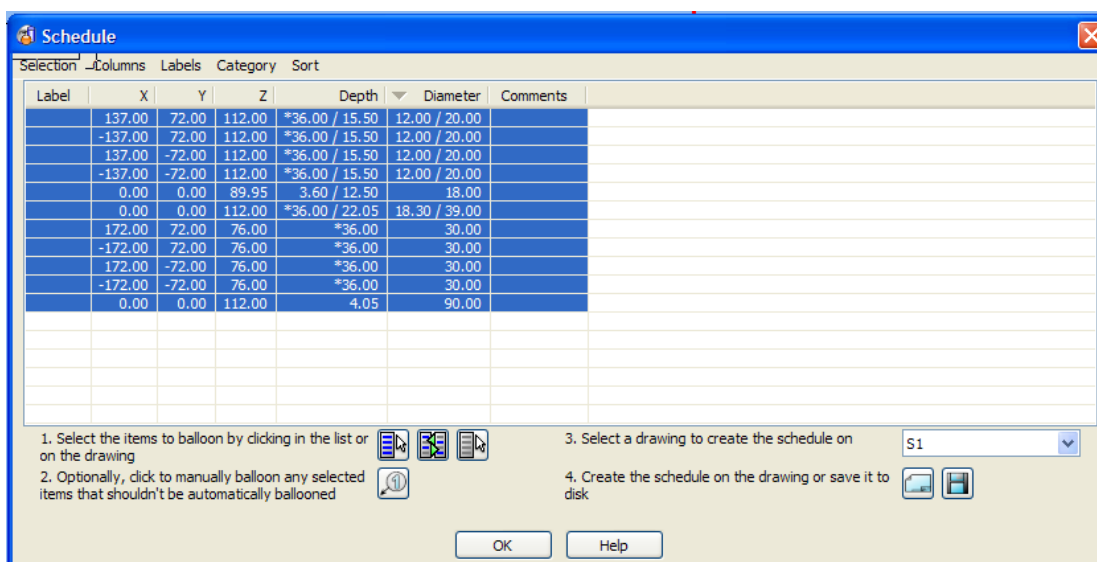


These settings create only a label, there is no balloon or leader line

The Schedule form contains all of the details for the holes in the Active view.

Using the RMB on the column header you can change the information that is displayed

- Select all of the Holes for labelling

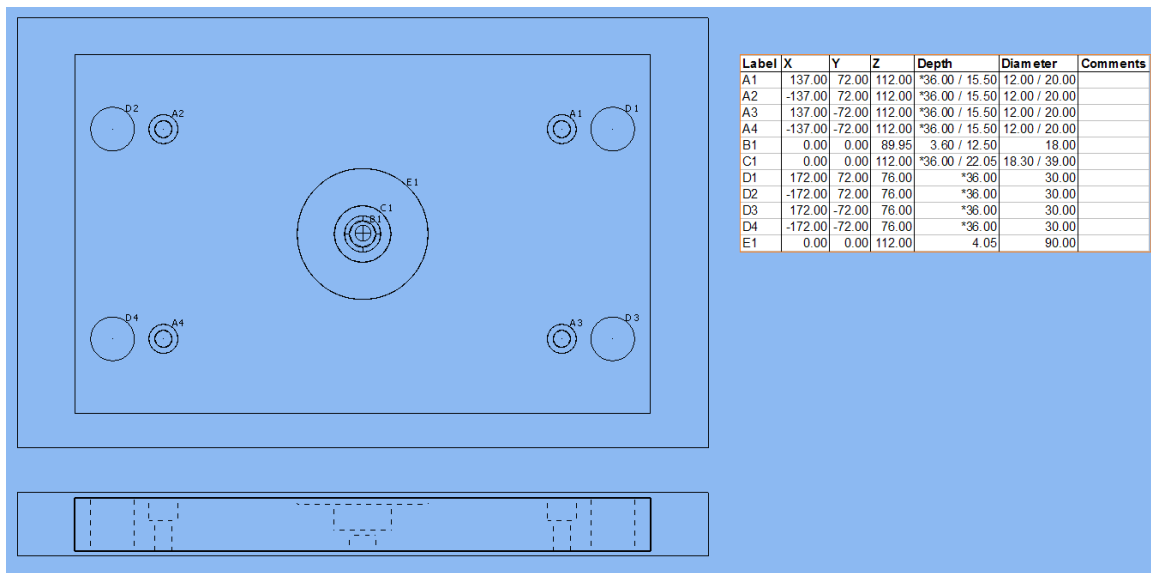


- The **Label Items** form will be raised, **Dismiss** this form to force the labels to be created automatically.

You could if you wish place the labels manually as per the B.O.M. exercise

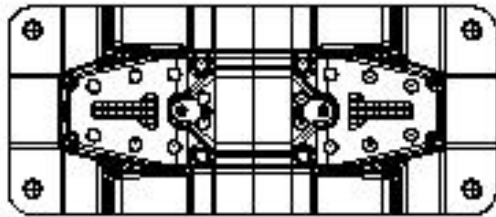
| Label | X | Y | Z | Direction | Depth | Diameter | Pre-drilled | Comments |
|-------|--------|--------|--------|-----------|----------------|---------------|-------------|----------|
| A1 | 216.26 | 72.00 | 132.00 | In | *36.00 / 15.50 | 12.00 / 20.00 | * | |
| A2 | -57.74 | 72.00 | 132.00 | In | *36.00 / 15.50 | 12.00 / 20.00 | * | |
| A3 | 216.26 | -72.00 | 132.00 | In | *36.00 / 15.50 | 12.00 / 20.00 | * | |
| A4 | -57.74 | -72.00 | 132.00 | In | *36.00 / 15.50 | 12.00 / 20.00 | * | |
| B1 | 79.26 | 0.00 | 109.95 | In | 3.60 / 12.50 | 18.00 | | |
| C1 | 79.26 | 0.00 | 132.00 | In | *36.00 / 22.05 | 18.30 / 39.00 | | |
| D1 | 251.26 | 72.00 | 96.00 | Out | *36.00 | 30.00 | * | |
| D2 | -92.74 | 72.00 | 96.00 | Out | *36.00 | 30.00 | * | |
| D3 | 251.26 | -72.00 | 96.00 | Out | *36.00 | 30.00 | * | |
| D4 | -92.74 | -72.00 | 96.00 | Out | *36.00 | 30.00 | * | |
| E1 | 79.26 | 0.00 | 132.00 | In | 4.05 | 90.00 | * | |

- Position the Hole Schedule on the drawing

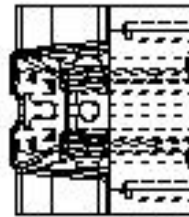


Scrap Sections / Hidden Line Feature removal

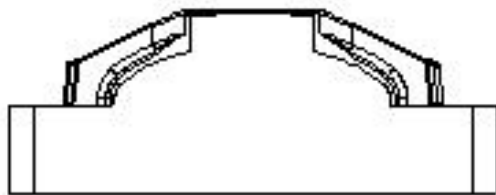
- Create a new drawing for the **Core insert**, the two side and end views are **relative** views.
- Make the **side** and **end** views **hidden**, the end showing hidden detail



Wire frame plan view

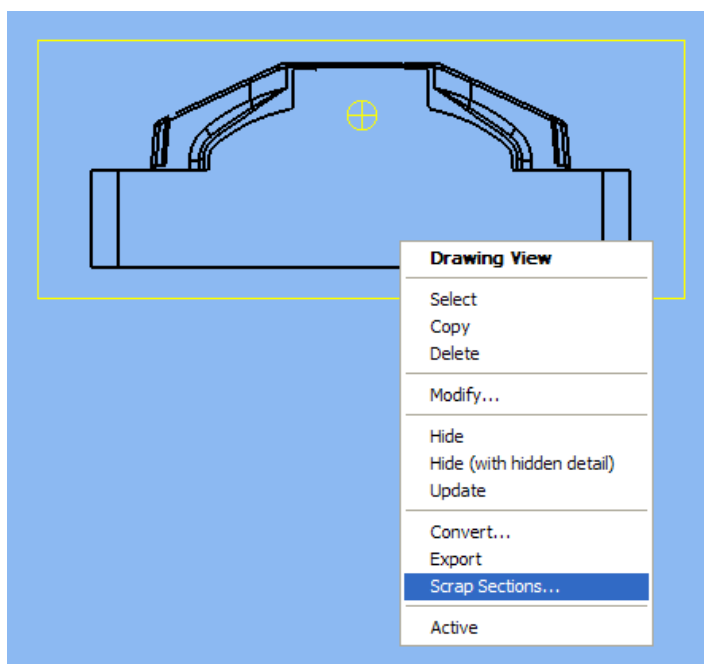


Hidden end view with detail shown

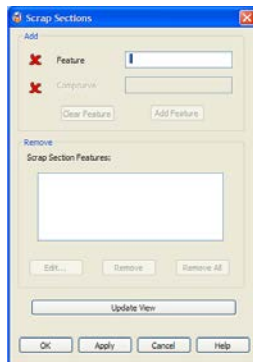


Hidden Side View

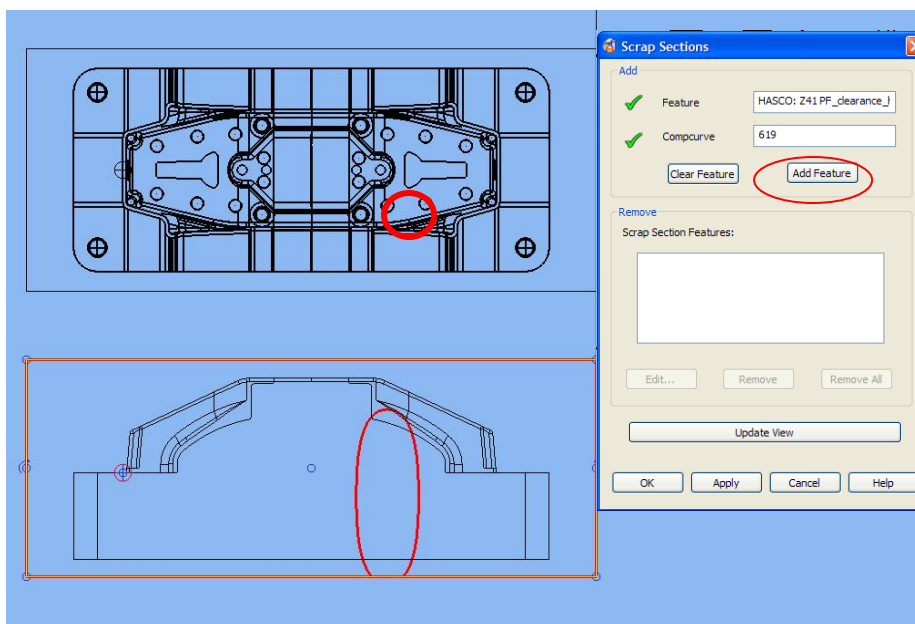
- In the hidden side view use RMB to create **Scrap sections**



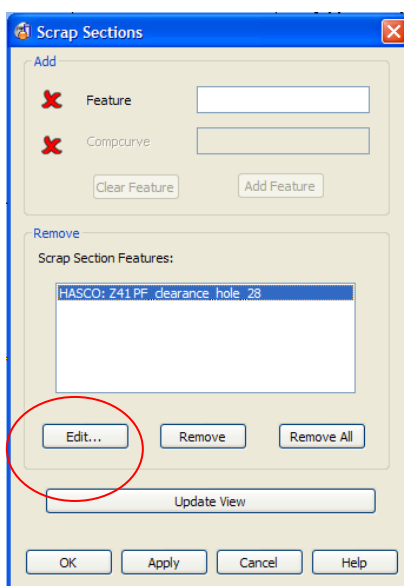
The Scrap sections only support **hole** features at present, you must select a hole and a comp curve will currently be created automatically



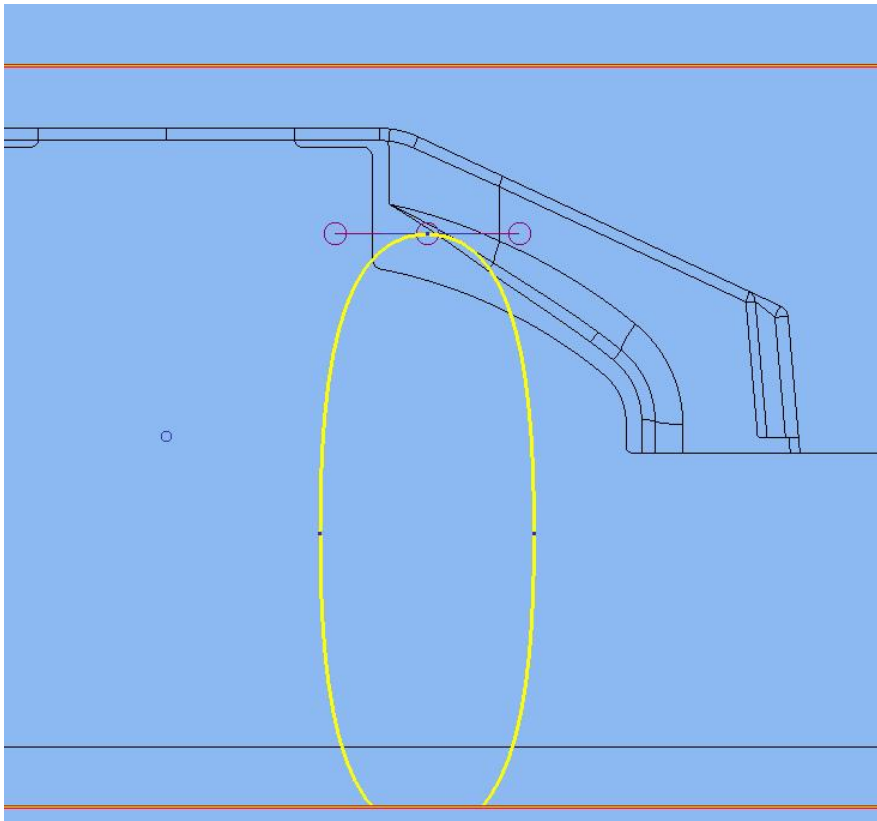
- Pick the hole feature indicated and **Add** to the list, this can be repeated for more hole features



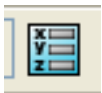
- Edit the **Scrap Section** feature



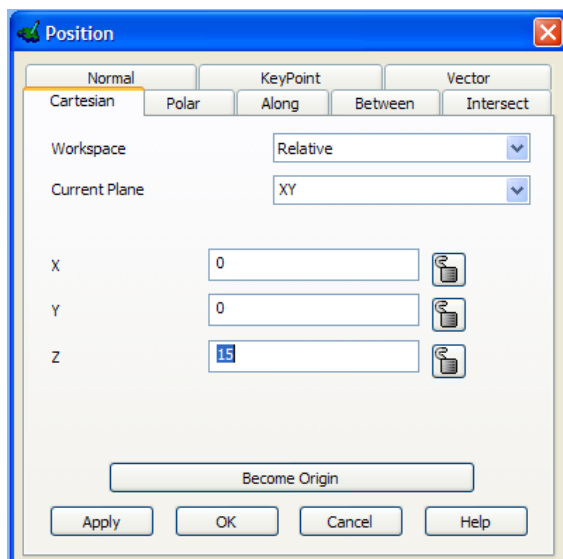
- **Select** the node on the top of the curve

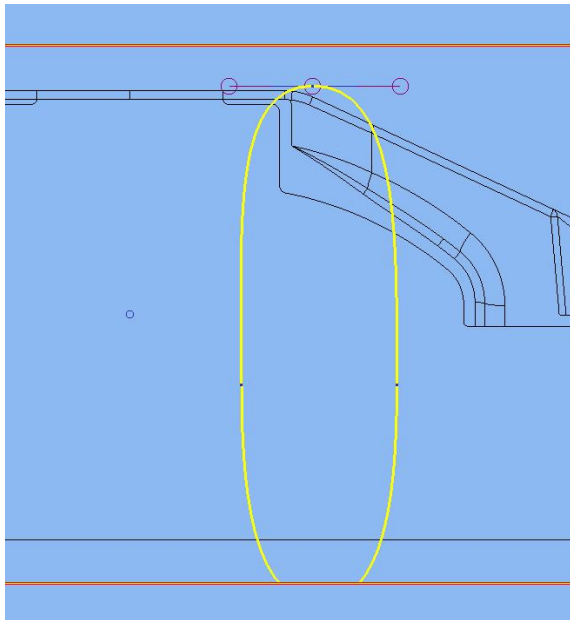


- Open the **Position Form** from the lower toolbar

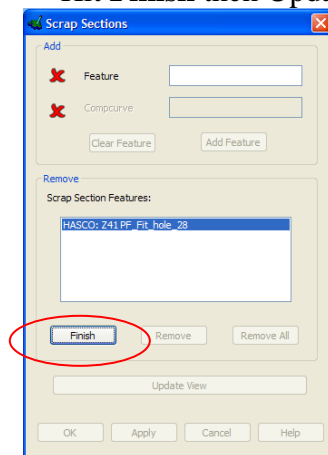


- Move the selected node in **Z** only 15mm

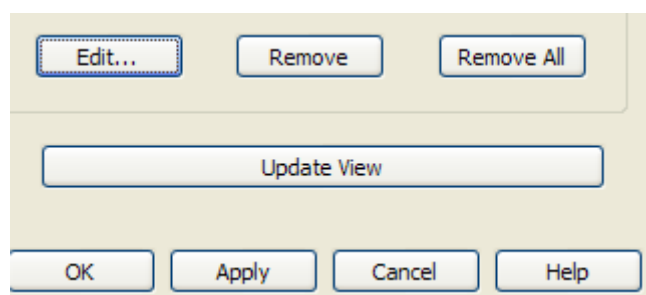




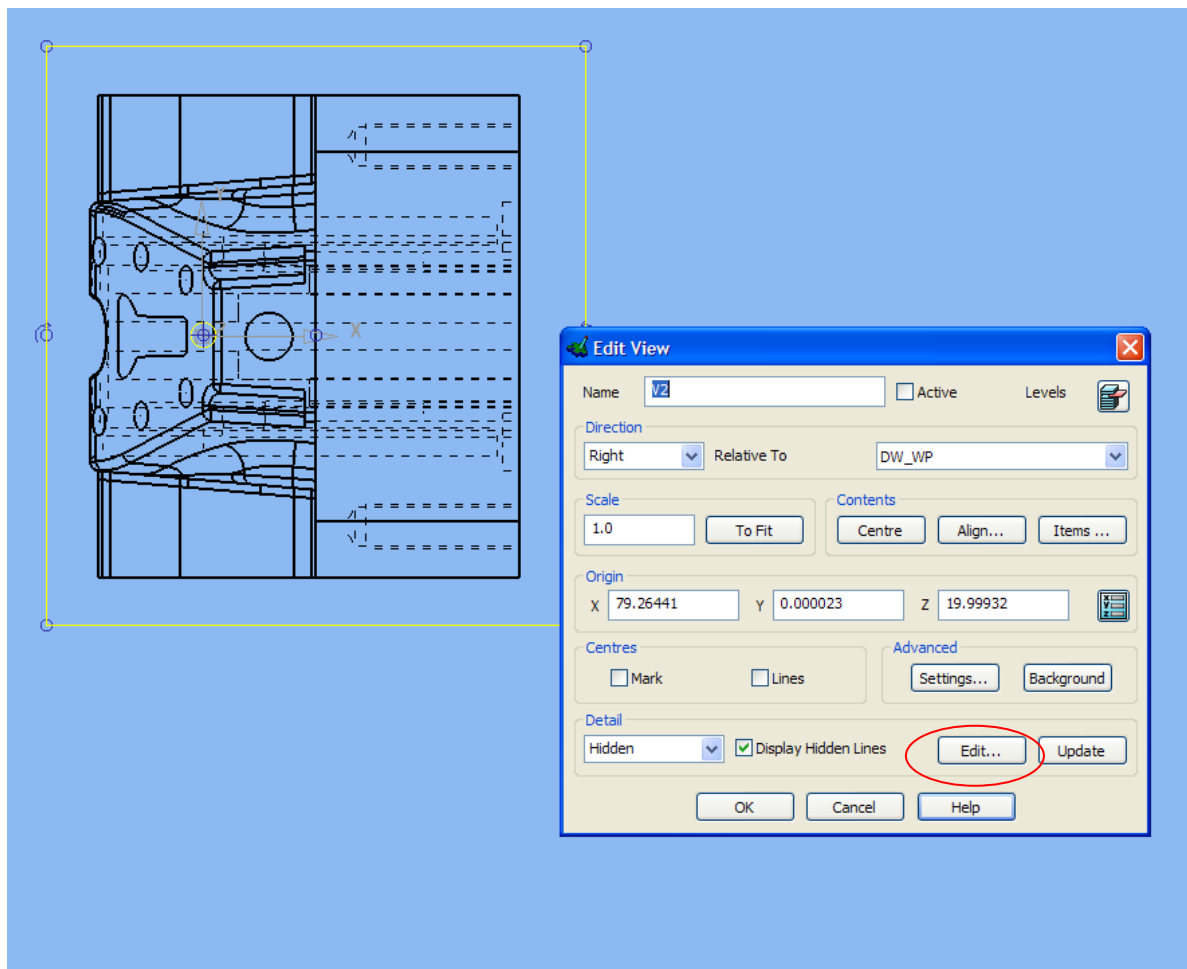
- Hit **Finish** then Update View to see the results.



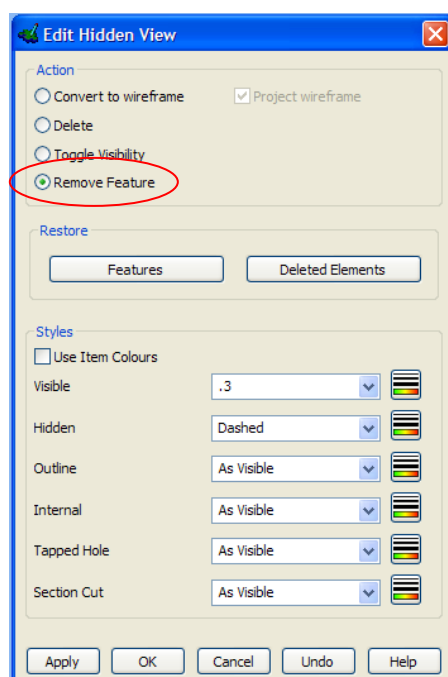
- **OK** to close the form



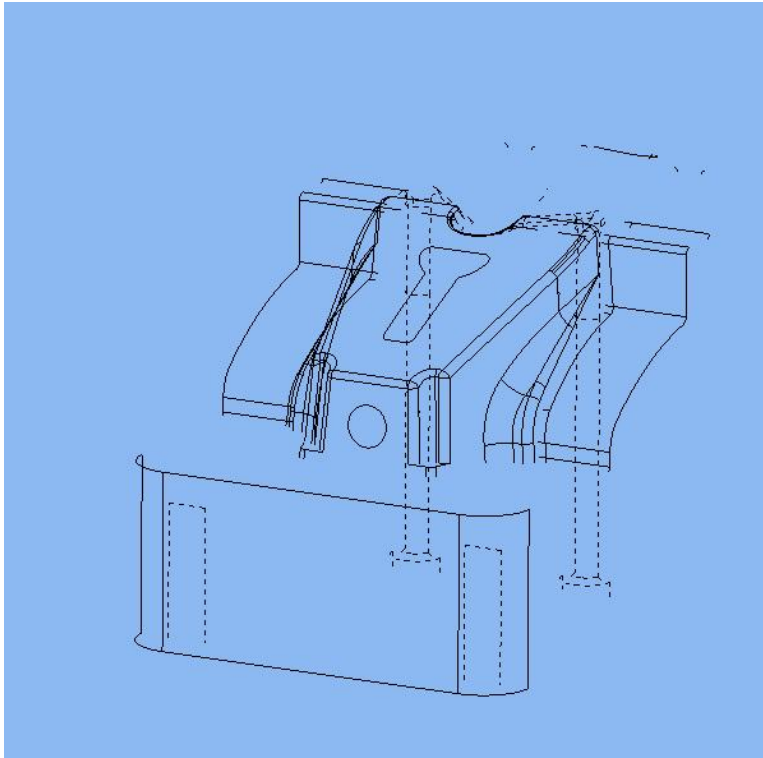
- **Modify** the Hidden Line view and enter the **Hidden Line Edit** mode



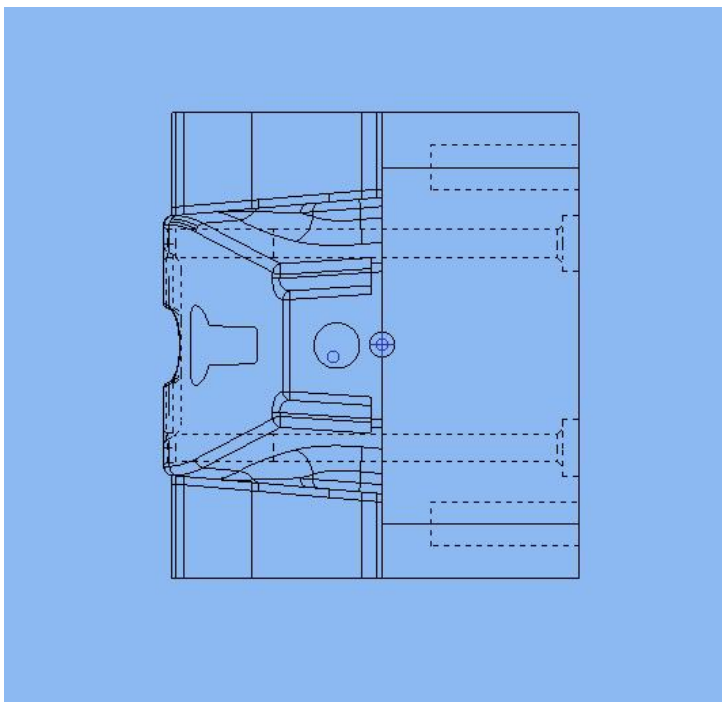
- Select the **Remove Feature** on the Edit Hidden View form



- Rotate the view to aid selection of features to be removed, once selected hit **Apply** on the from to remove



- Hit **OK** to close the form and **reset the view**



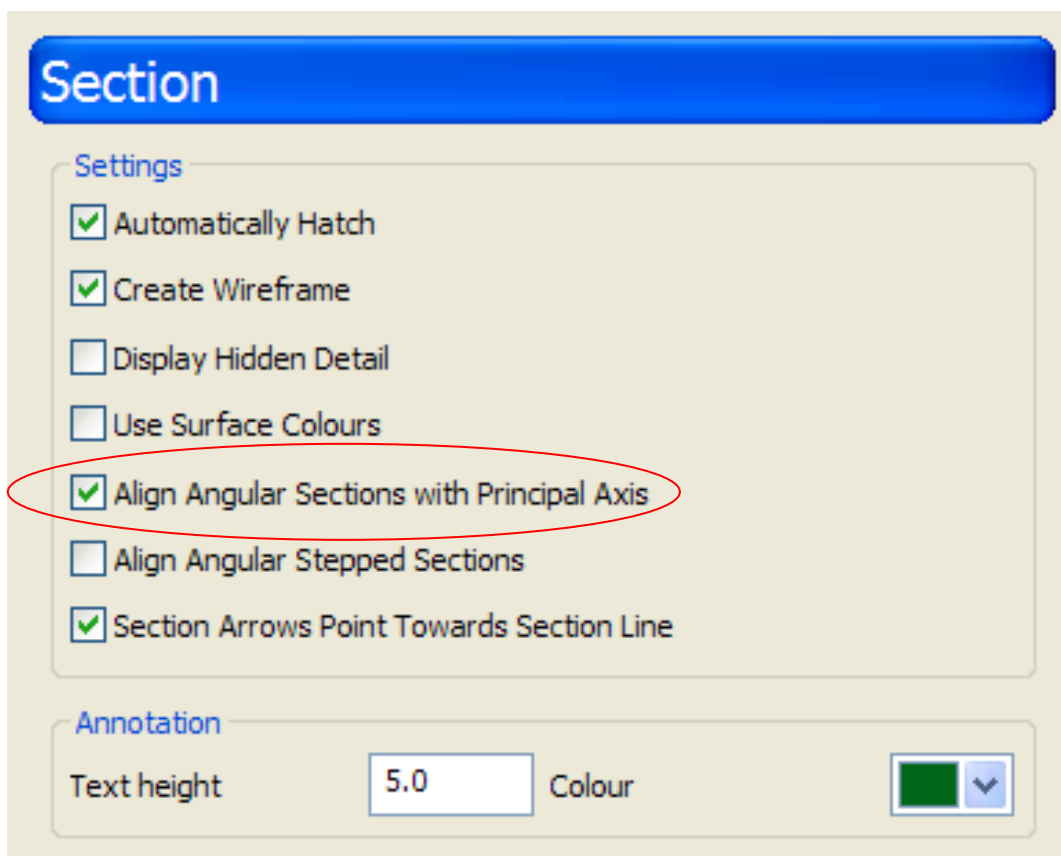
Angled Sections

Angled stepped sections can now be created, there are two options to control how the section view is created:-

Align Angular Sections to principle axis - This option will create a 'Projected' section View.

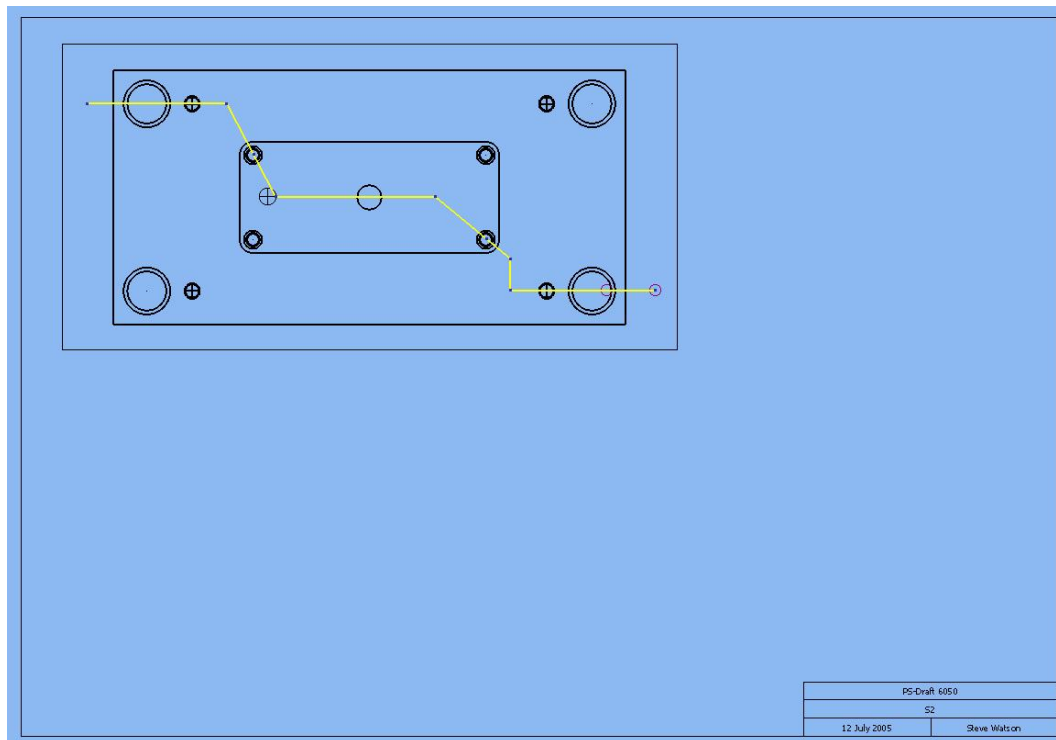
Align angular stepped sections - This option will create a section view that displays the 'True' length of the section line. This is useful when dimensions are added to the sectioned view

- Set the option to align with **principle plane**

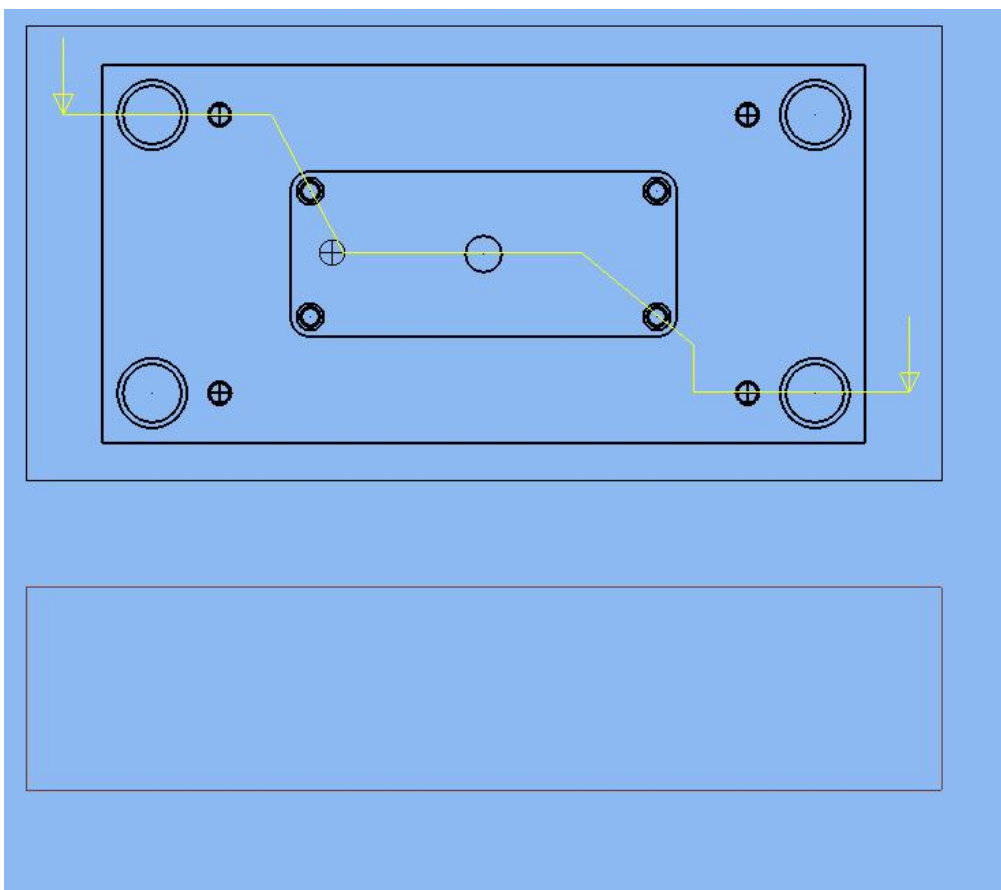


Create a new drawing for the Cavity Plate, Create only a plan view.

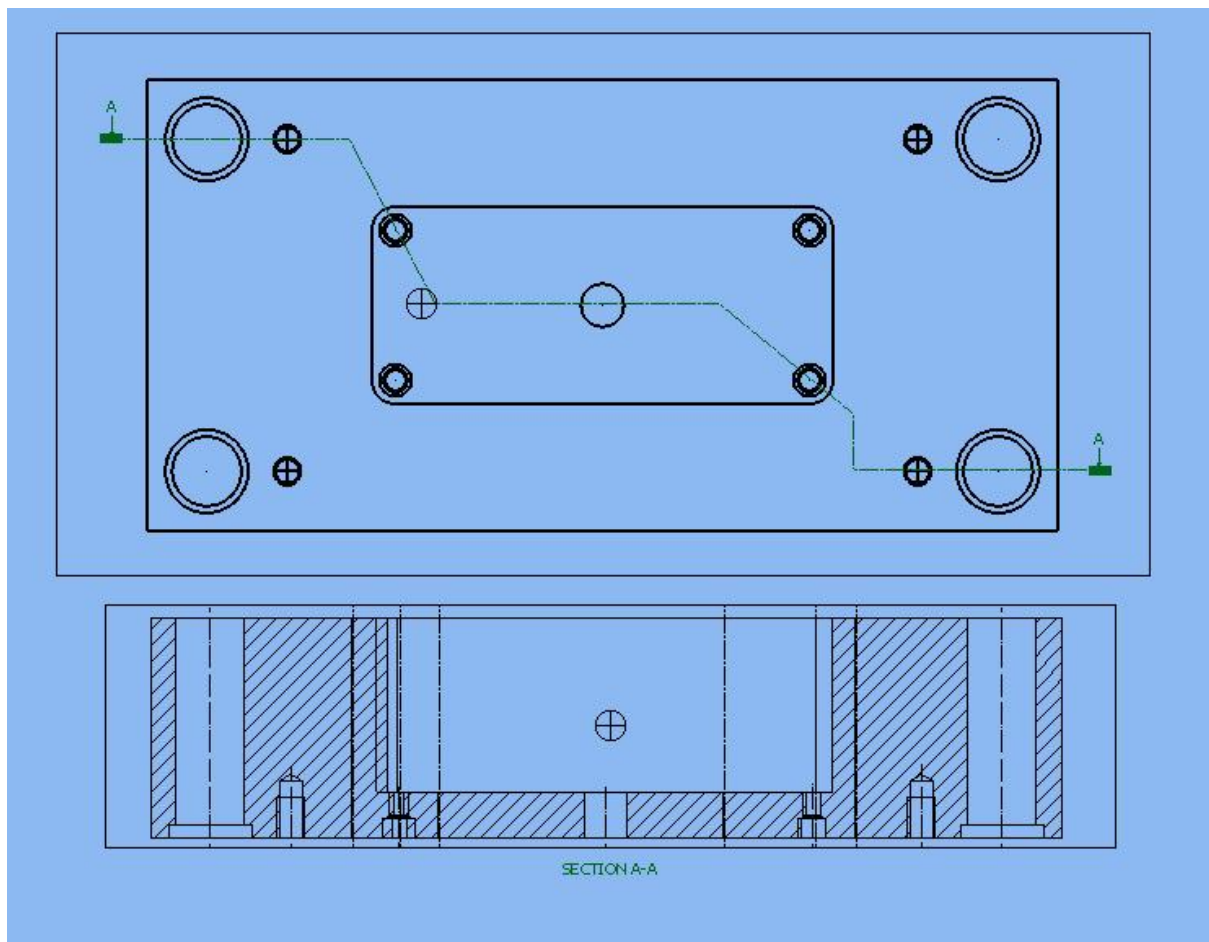
- Create a composite curve as indicated



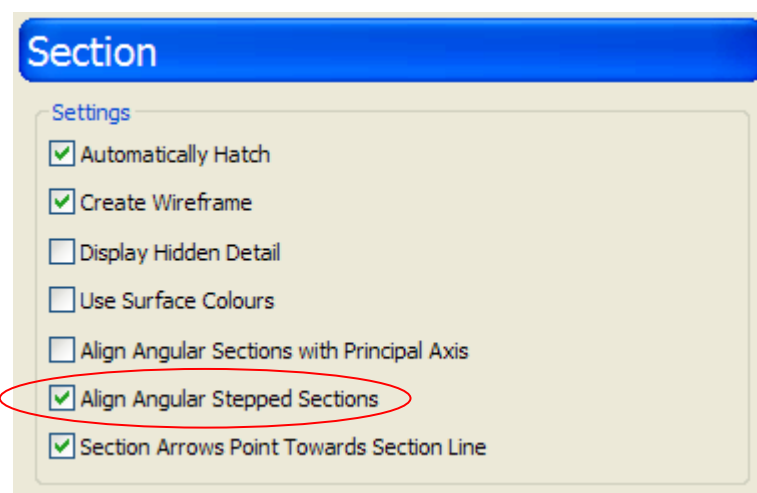
- Create a **section view with background**, position as indicated below



You can see that the section view is projected; the overall size is the same as the plan view



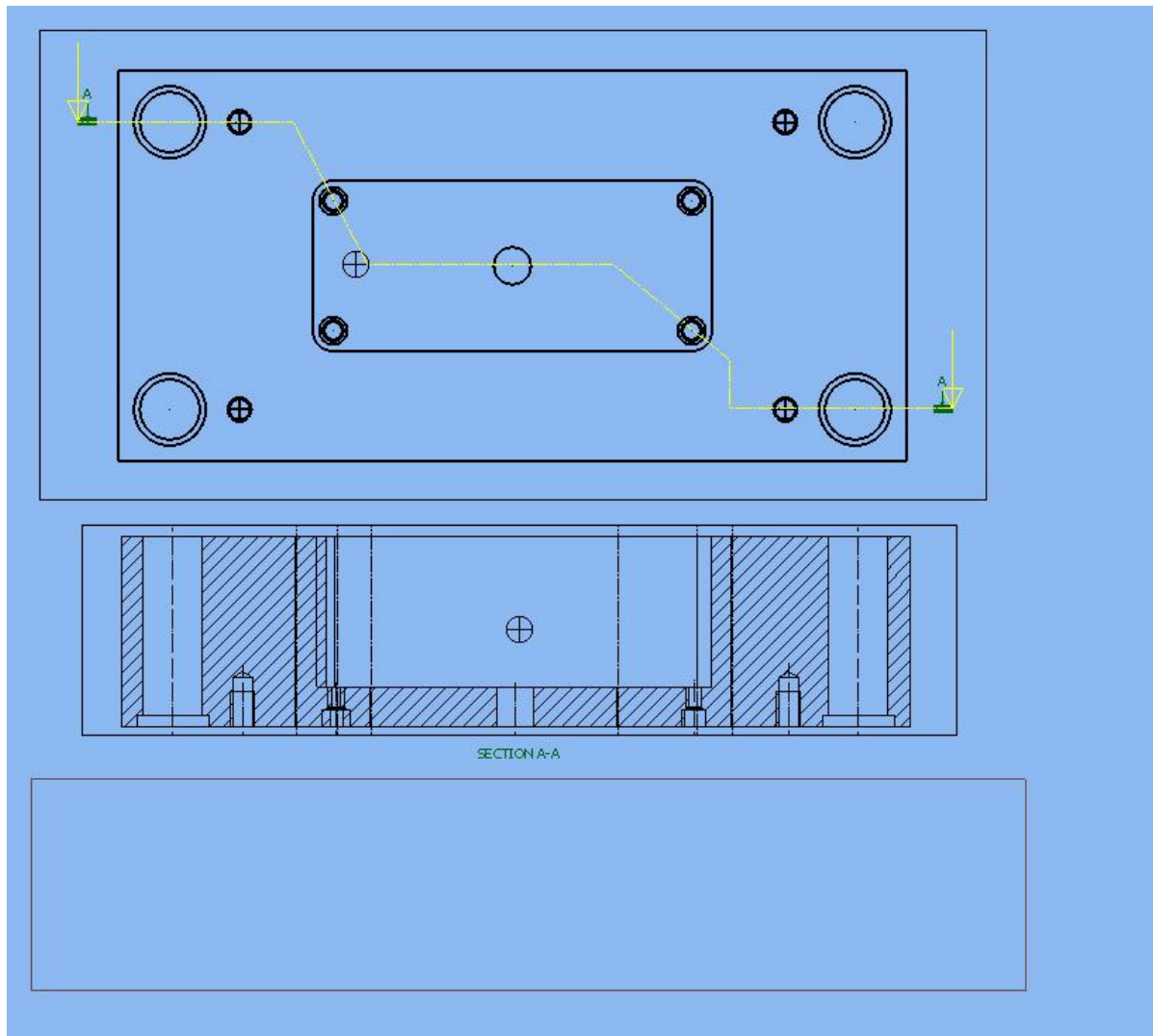
- Change the option to **Align Angular Stepped Sections**



- Using the **same composite curve** create another **section view with background**
- **Position** the section as indicated below.

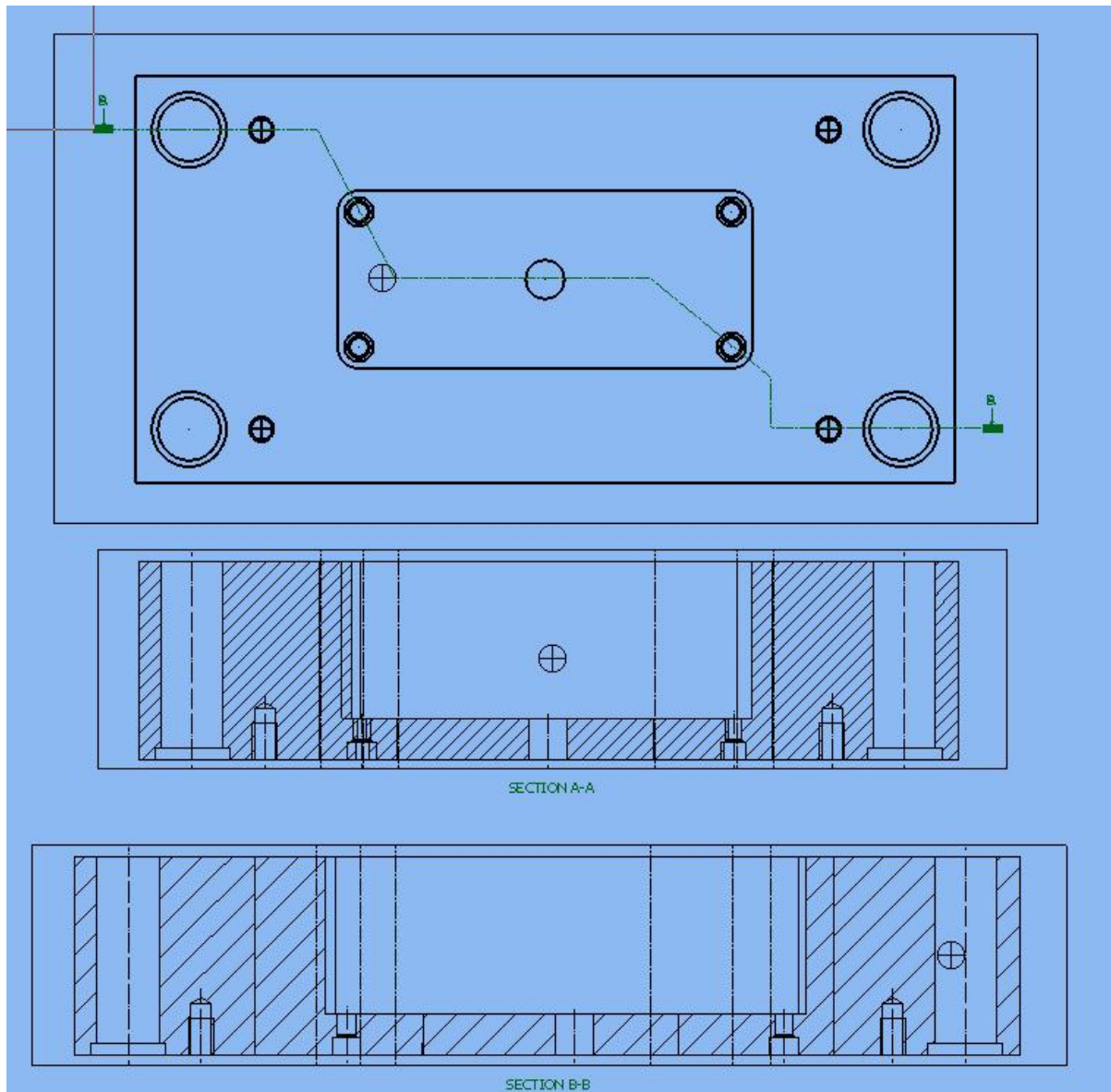
You will notice that the section does not have the same ‘snap’ alignment to the plan view as the first section view.

This section can be positioned anywhere, although it will be aligned to the section direction.



Now that the section view is created you can see that it is quite different to the projected section view.

This view shows the full length of the section line so that any features that the angle passes through can be dimensioned correctly



Automatic drawing creation

There are some new methods to create drawings automatically in PS-Draft.

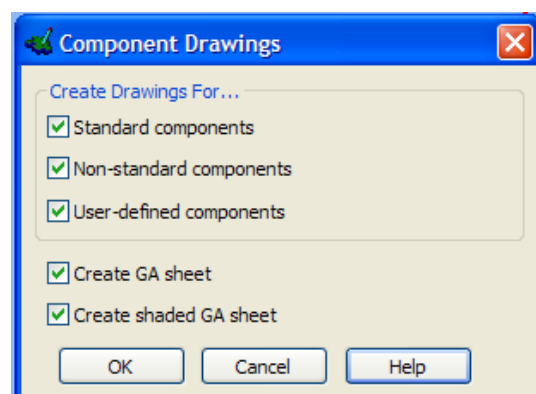
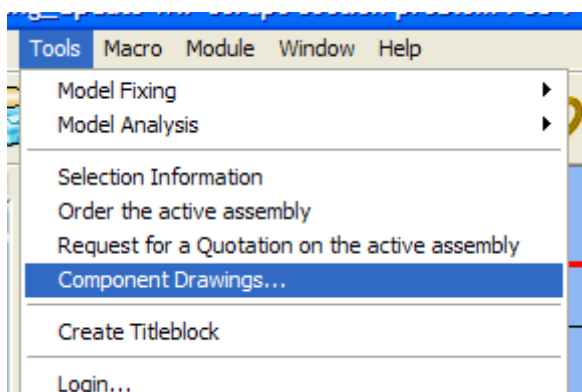
As the Moldmaker catalogues are parametric the parameter values can be extracted and placed on a drawing with a draft showing their use.

There are two new options to create drawings:-

Create Standard Component Drawings – Used when standard PSMM components that have not had any parameter changes

Create NON- Standard Component Drawings - Used when the parameters have been changed to make standard components NON standard

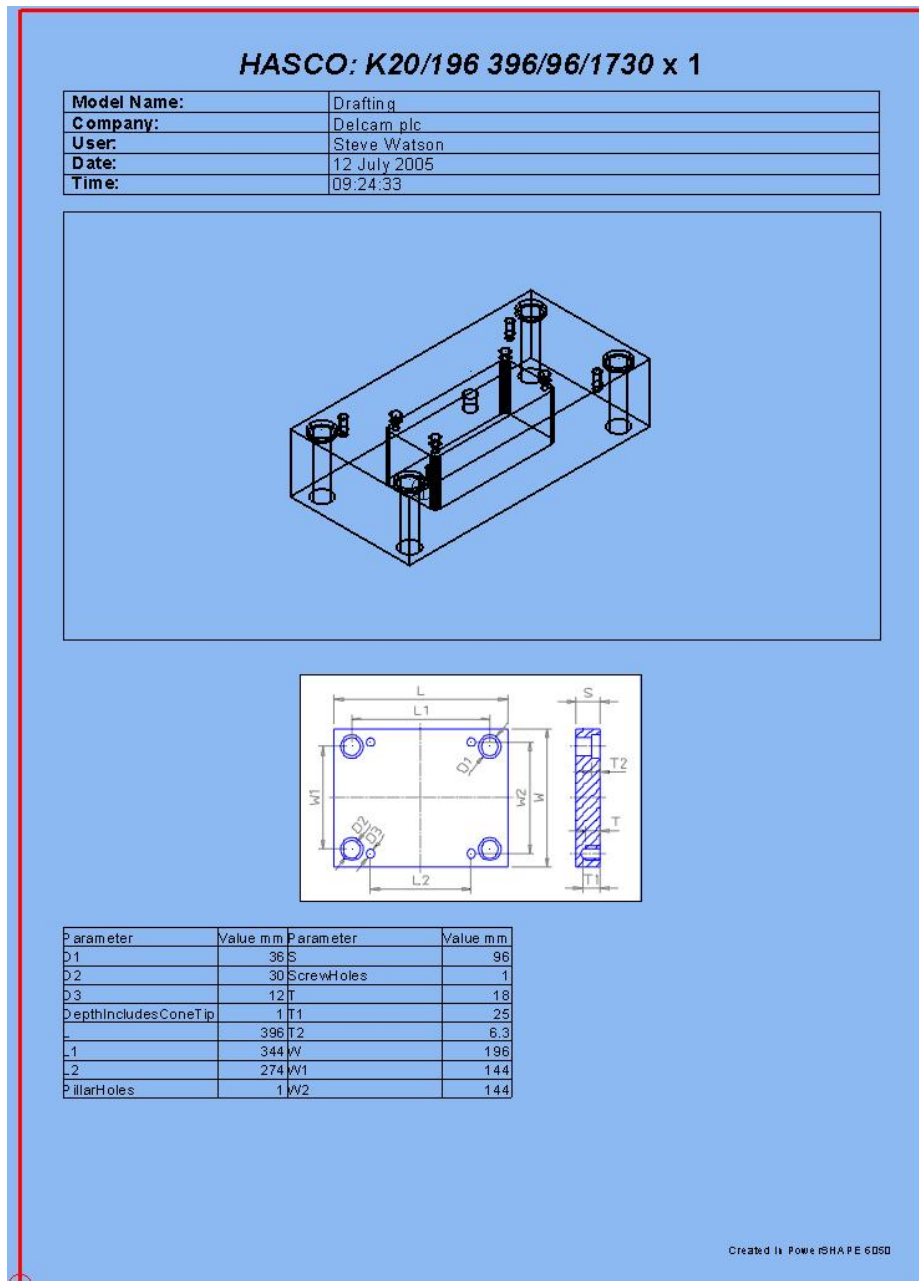
Both can be found in Tools – Component Drawings along with the old Assembly hole schedules.



- Using the Cavity plate, create a Standard Component drawing
- Select a random part or several parts and try out these drawings.

Note:

You must be in Drafting mode for these drawings to work



Template

An additional option has been added to the drawing templates. A hidden line view can now be used within a template.

The template must contain a part that has a hidden view applied.

Any other type of view can still be set up 'empty'

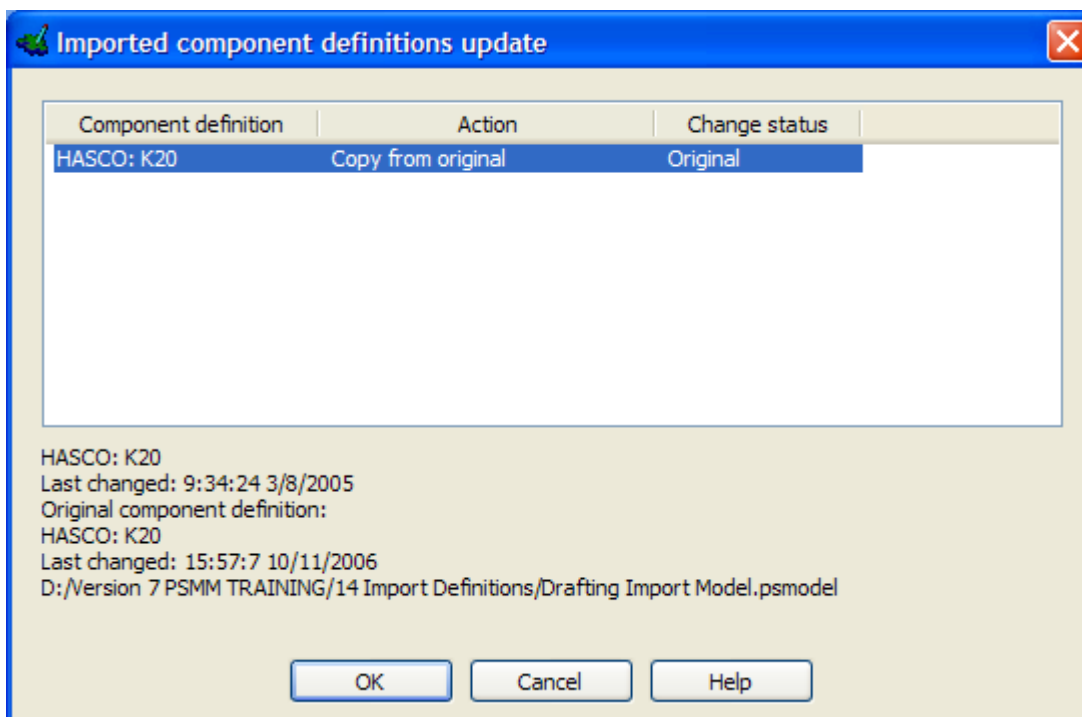
14. Imported Component Definitions

Updating Imported Component Definitions (Reference Guide)

When using imported component definitions there is a mechanism to update the definitions when there are changes either to the master or the imported definition.

This operation is performed when the model is opened / saved or closed and can be manually updated as required.

The update can be done by selecting the Refresh Imported Component Definitions button from the Assembly toolbar.

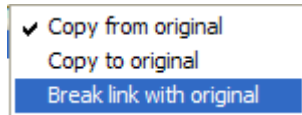


Component Definition - The name of the imported component definition that has changed. There will be an entry for each definition that has been updated.

Action- Select an option from the pop-up RMB menu to define how the definitions are synchronised.

| | |
|-----------------------|--|
| Change Status- | The status of change original and imported definitions relative to each other. This will be one of the following |
| Original - | The original definition has been changed |
| Imported - | The Imported definition has been changed |
| Both - | The original and the imported have both been changed |

The RMB will raise the Action pop-up menu



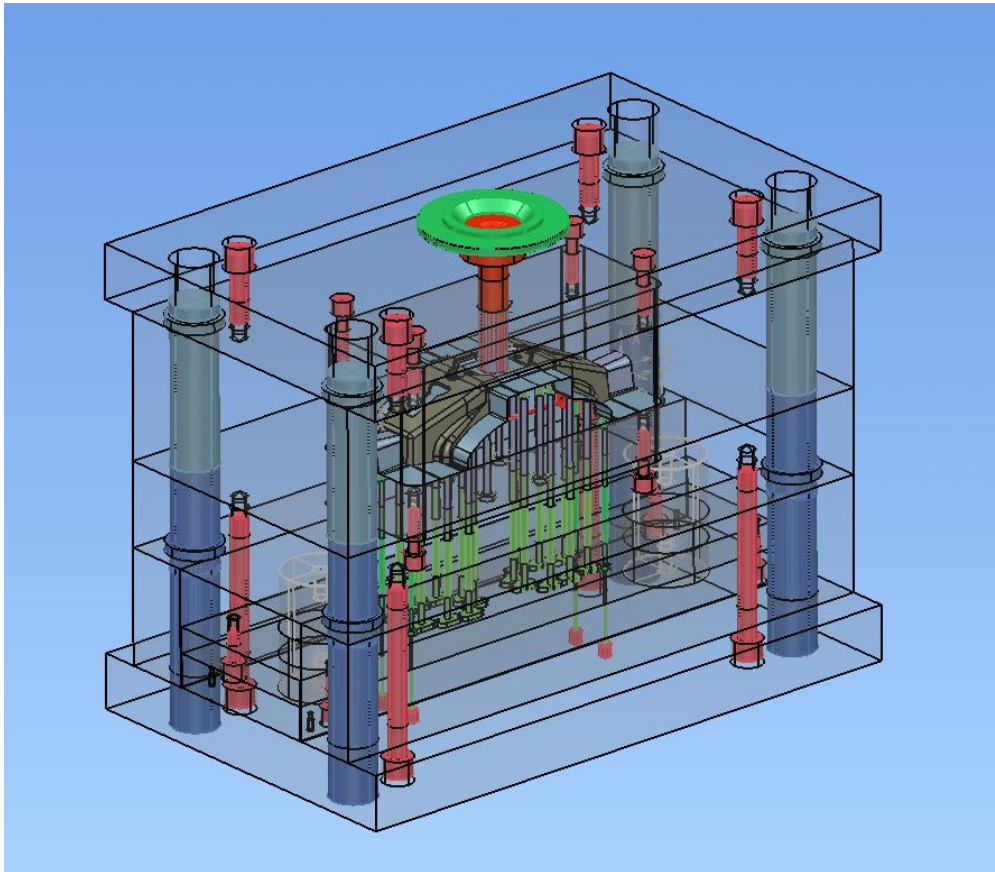
| | |
|-----------------------------------|--|
| Copy from Original - | replace the imported component definition with its original |
| Copy to Original - | Replace the original component definition with the imported one. |
| Break link with original - | Convert the imported component definition to Local. The definition will no longer be affected by any changes to the original definition. |

Additional Information is also displayed about the selected definition is displayed below the list of definitions.

Name of the current component definition
 Time of the last change to this definition
 Name of the Original component definition
 Time of last change to the original definition
 Path to the model that contains the original definition

OK - Synchronise the definitions to the action selected

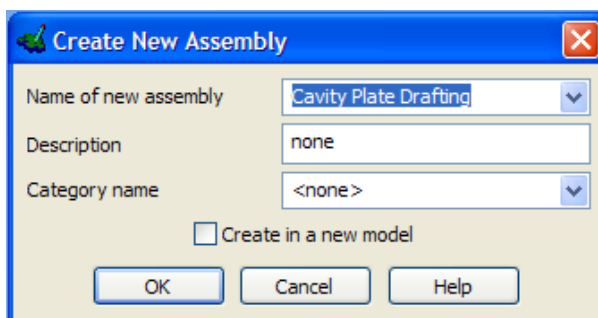
The Master Assembly contains the complete tool design. For this exercise you will be creating a 2D drawing for the cavity plate.



- Open a **NEW** model
- Turn On PS-Assembly and **create a new empty assembly**



- Name the new assembly **Cavity Plate Drafting**



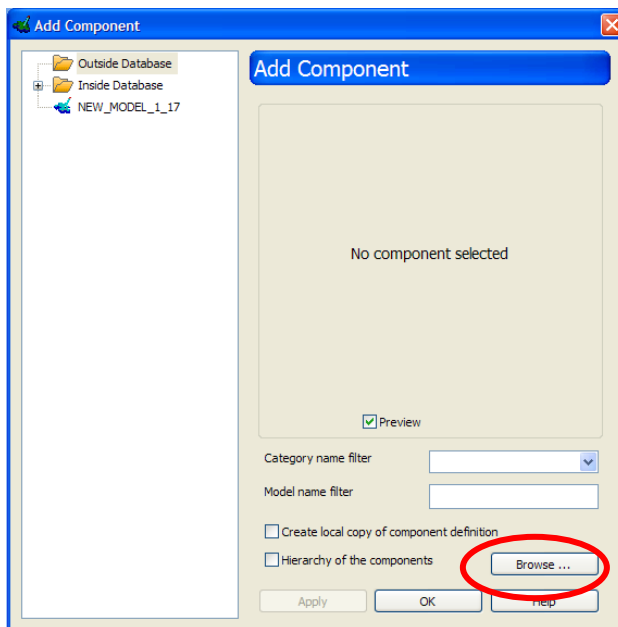
- Select **create component** from the Assembly toolbar



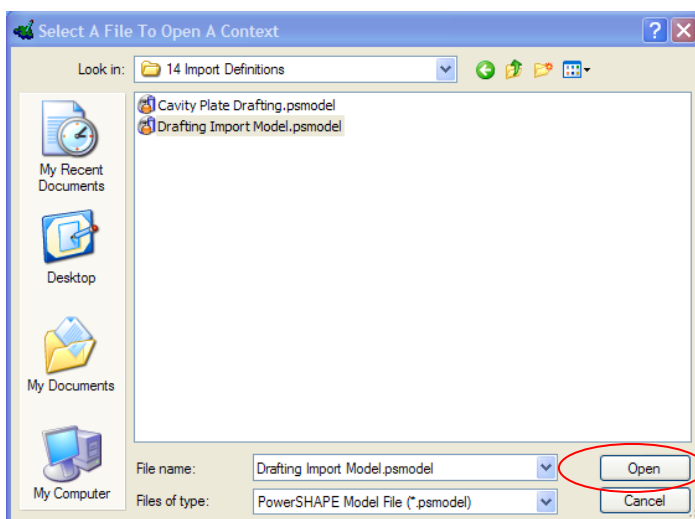
- Create component in free space

The Component form is opened, as the Master model is not in the database you must supply the location of the file

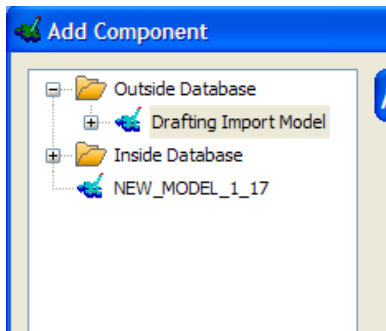
- Select **Browse**



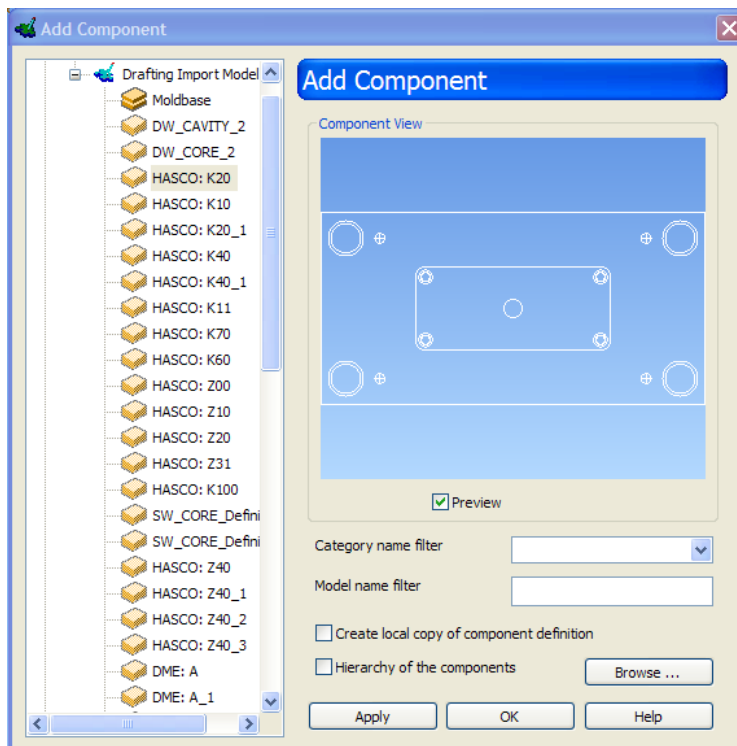
- select the model and **Open**



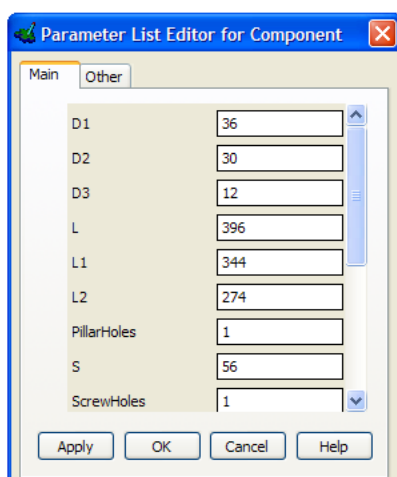
The Master Model is placed in the Add Component Tree



- Expand the tree and Pick the **HASCO K20** definition, the preview will confirm that it is the **Cavity Plate**. Press **Apply** to import the definition.



The Parameters of the Definition are displayed, **OK** to close this form

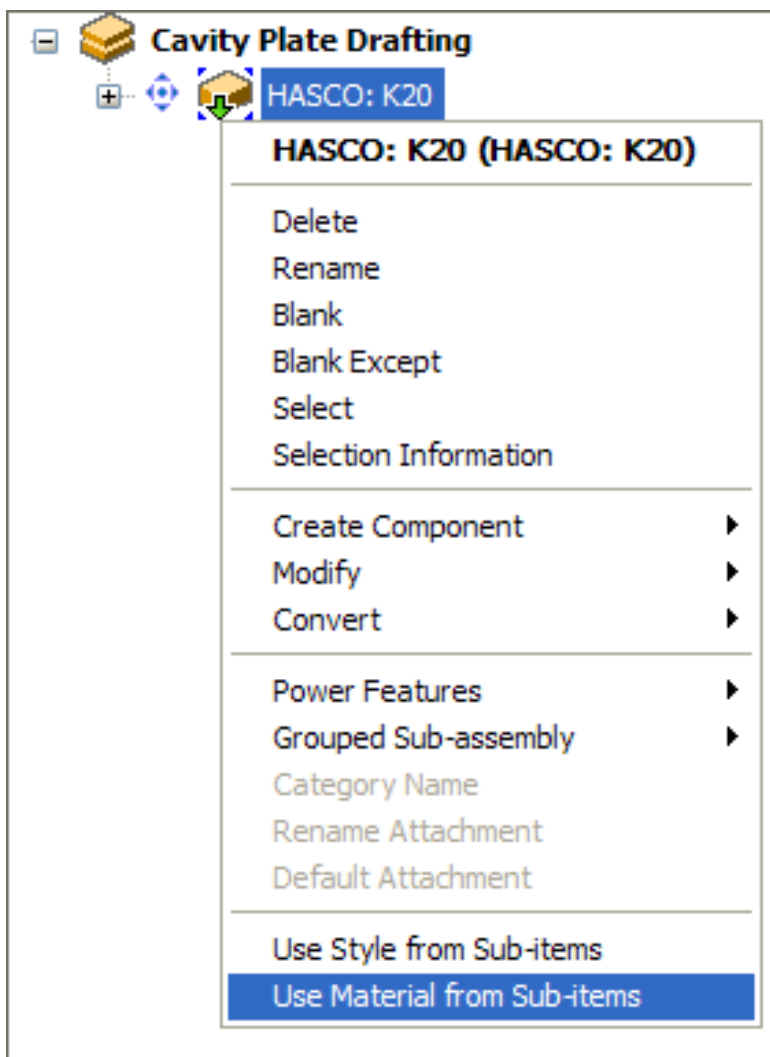


Expand the Component Definitions Tree branch, There is a dashed line thru the HASCO K20 definition. This indicates that the definition is **Linked** to another model



You will also notice that the component has been displayed in the default material for assembly components.

- With the RMB menu change to **Use Material from Sub-items**





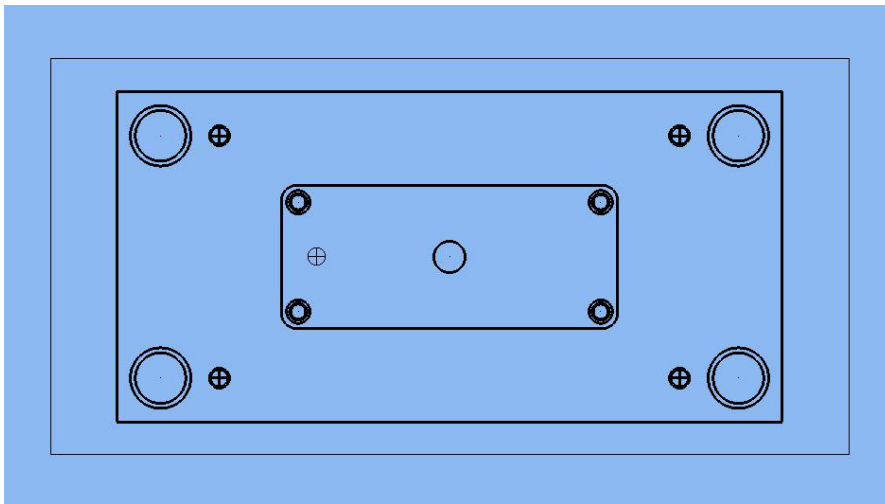
- Select **Drafting Mode** on the main toolbar



Create new drawing

Create a **new drawing**

- Create a **Plan view** of the plate

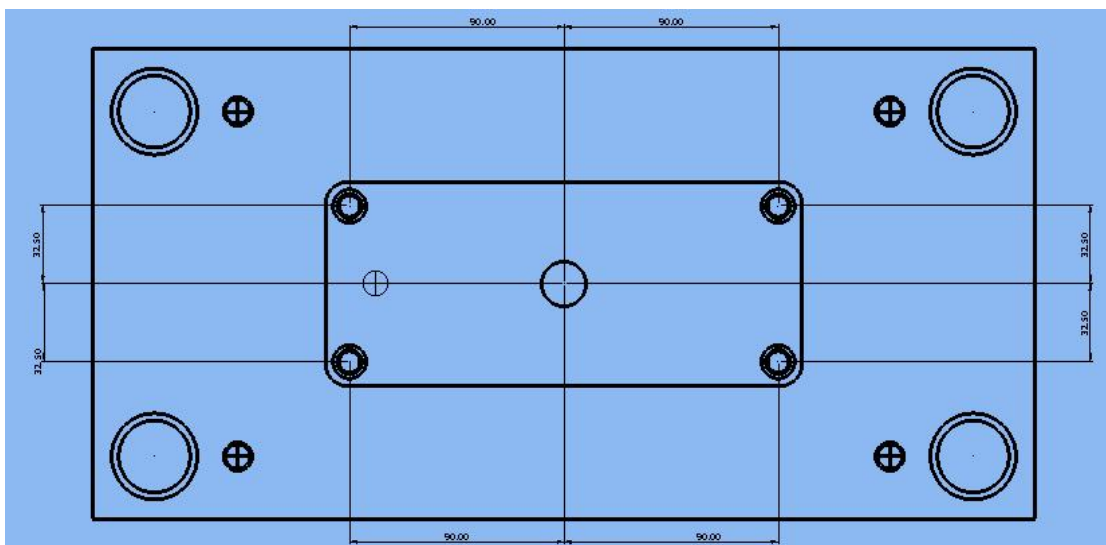


- Select **dimensioning** from the drafting toolbar

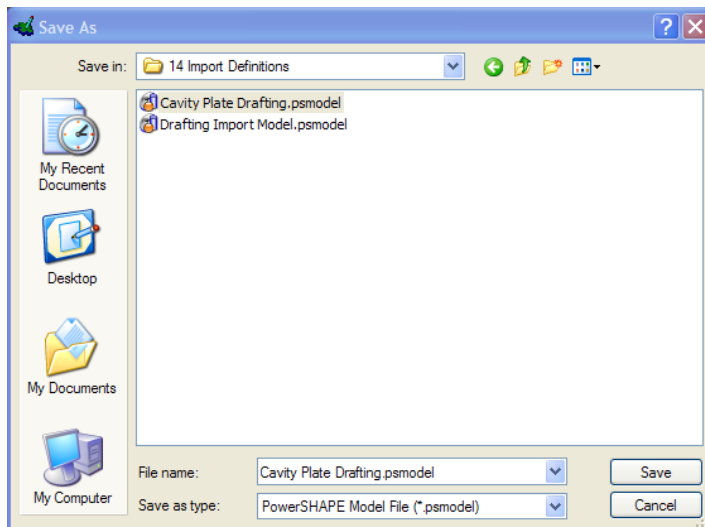
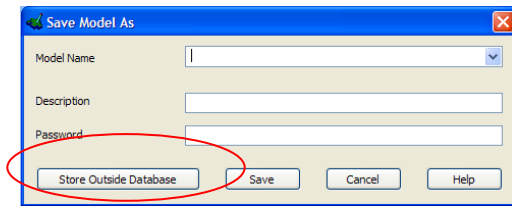


- Pick **Automatic Dimensions** from the drop down menu

- Dimension the holes as shown below

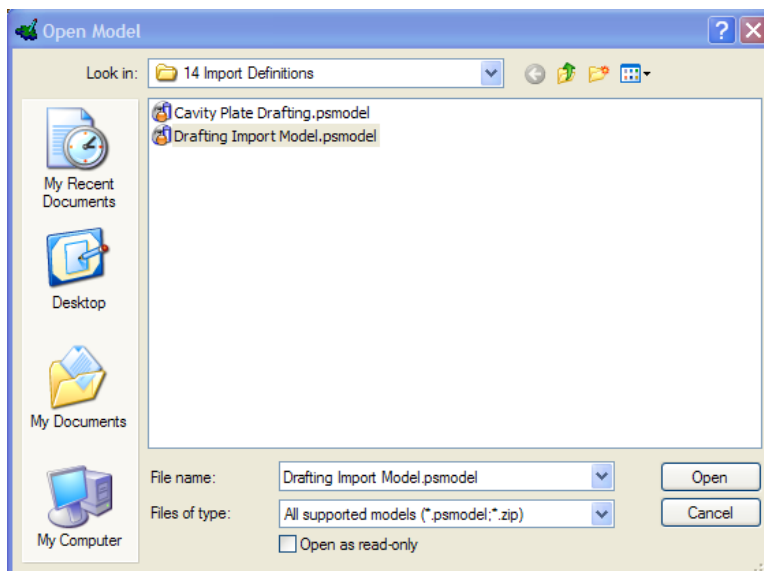


- Save the model as **Cavity Plate Drafting**, save outside the database

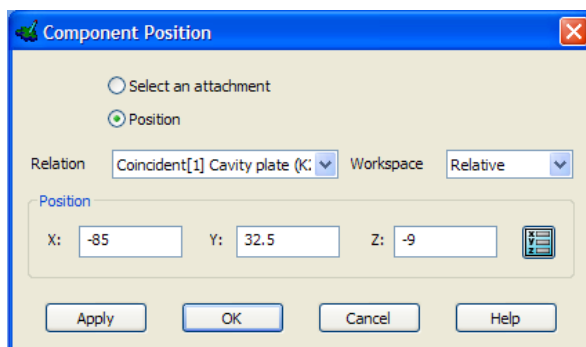
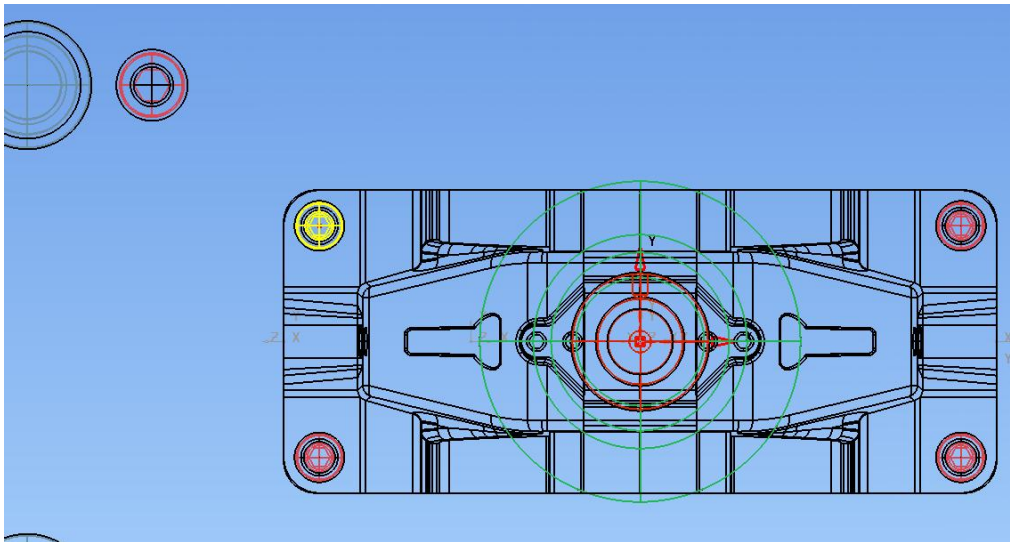


To action a modification you need to open the Master model and make a change.

- Open the **Drafting Import Model**

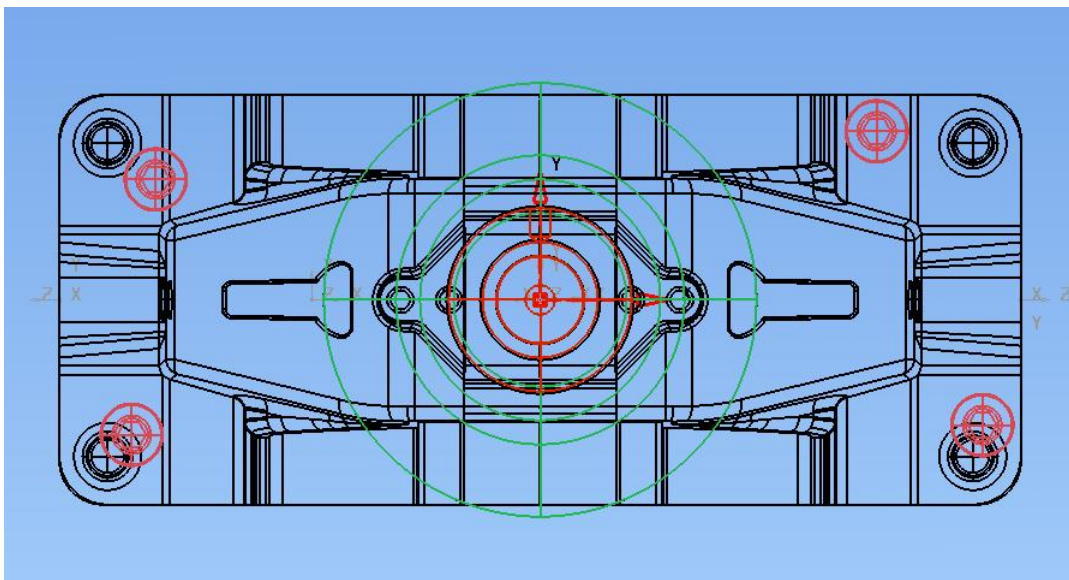


- Select the **Screws** that fix the cavity insert into the cavity plate and double click to raise the **Component position** form

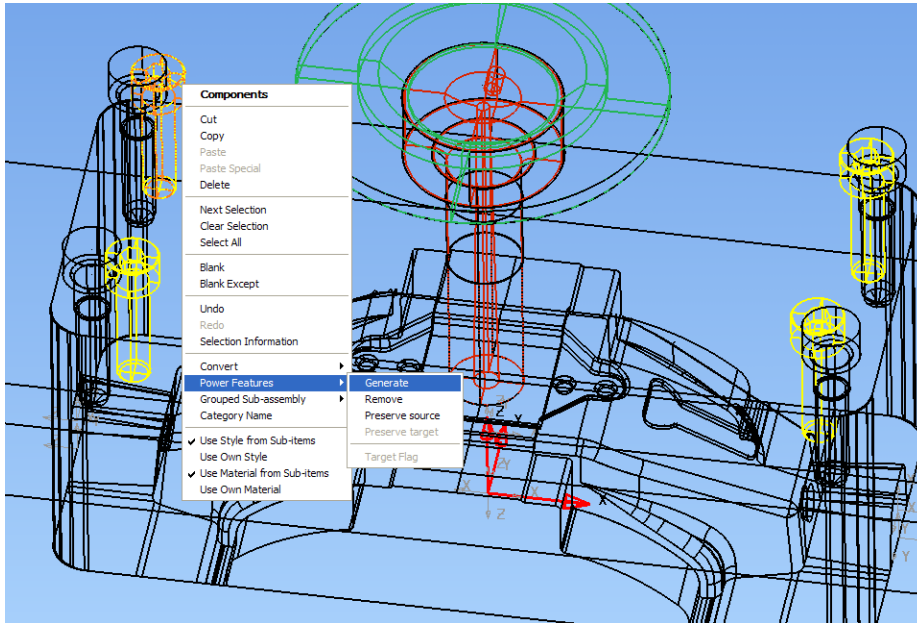


- Use the component position form to move each of the **four screws** it turn to a new position.

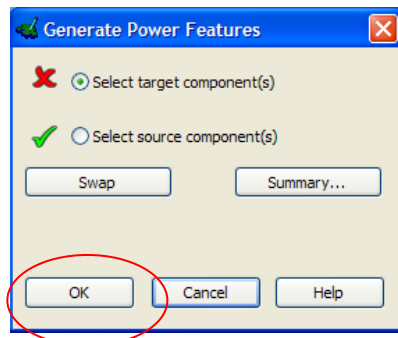
The actual position does not matter in this case.



- Select the four screws and RMB menu to **Generate Power Features**

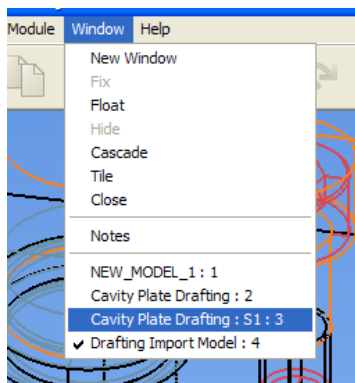


- **OK** the form to generate features



- Save the Model

- Change the View Window to the **Cavity Plate Drafting S1** drawing

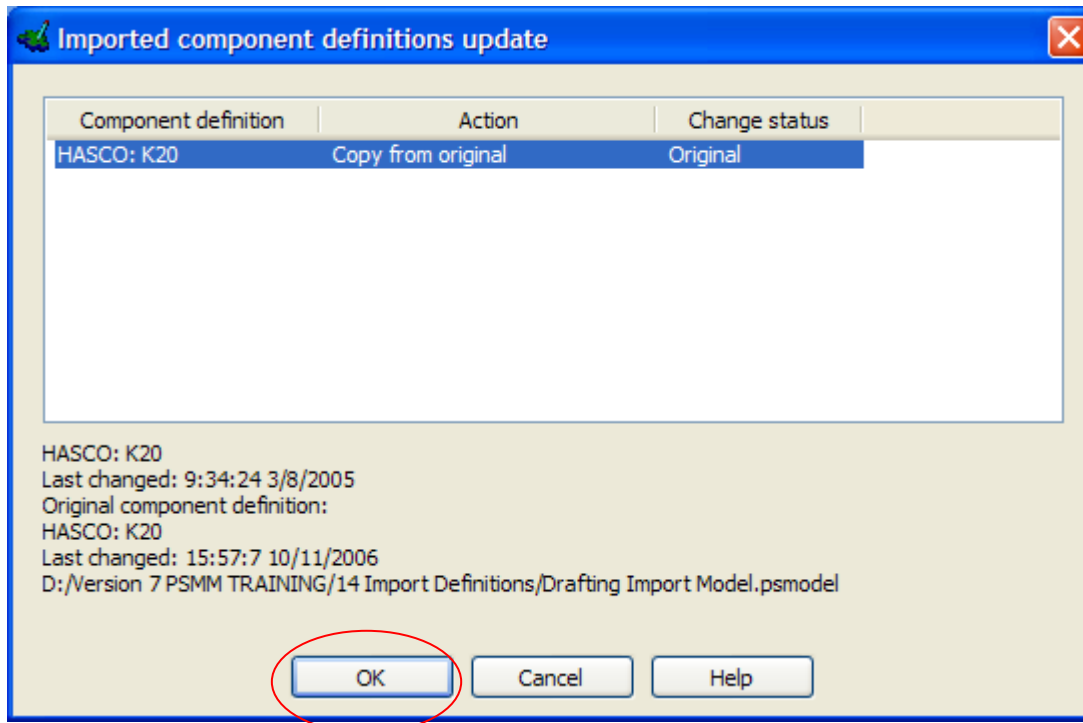




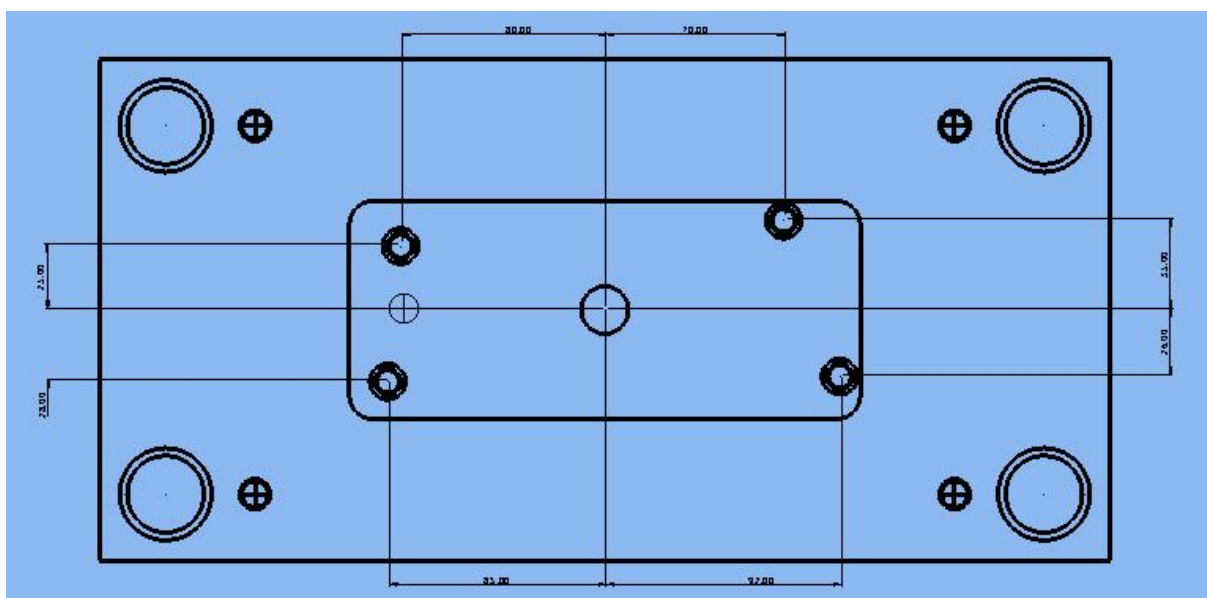
- Press the **Update Imported definition** Button on the Assembly toolbar

As the master model has changed the update mechanism reports the change information and the default Action is set to Copy from original

- **OK** to update the model



The drawing has now updated and the dimensions have changed to suit the new hole positions.



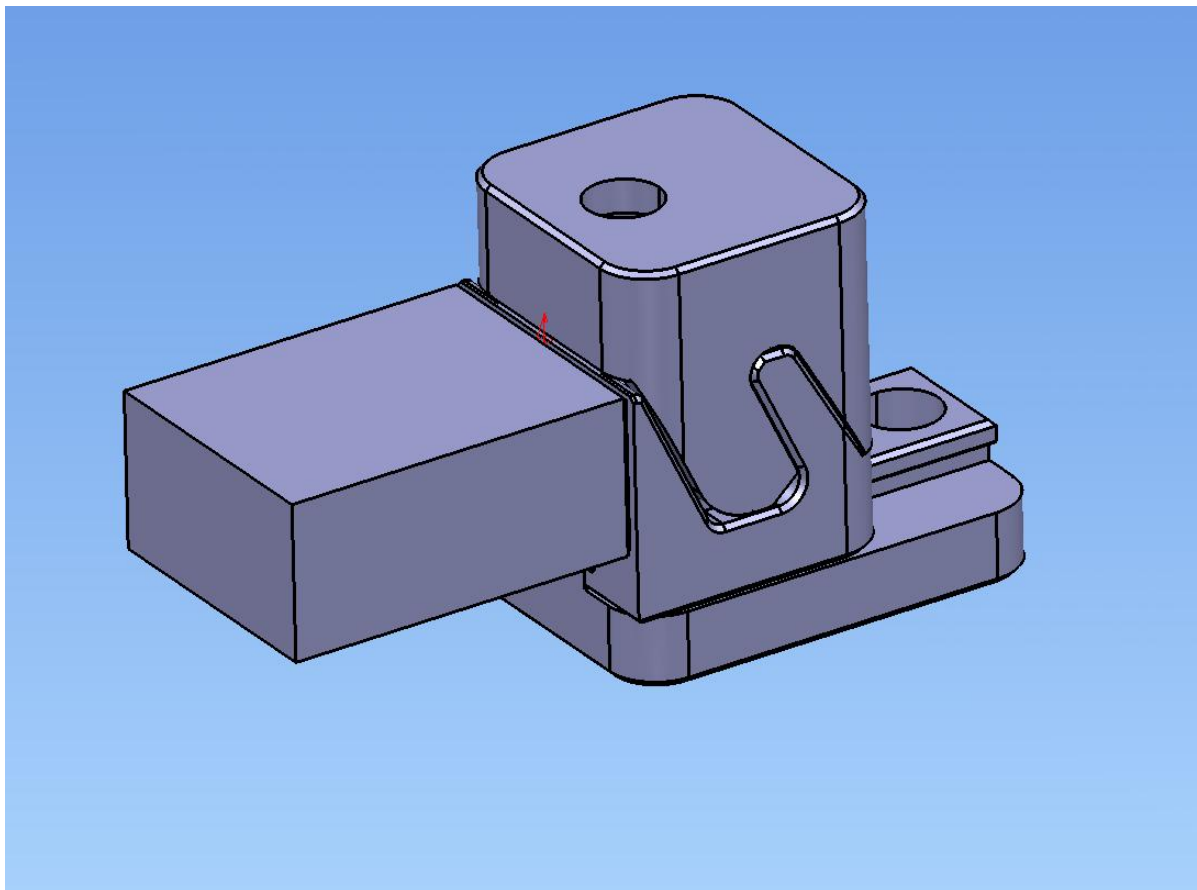
15. Power Feature Exercise

CAM 300 Slide Mechanism

As the use of standard slide mechanisms grows, it is more common to have component libraries containing standard parts that are not supported by PS-Moldmaker

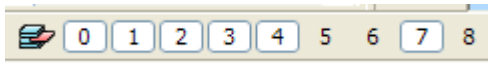
In this exercise you will add some Power Features to a slide assembly that was downloaded from a supplier web site.

- Open model – **CAM-300.psmodel**

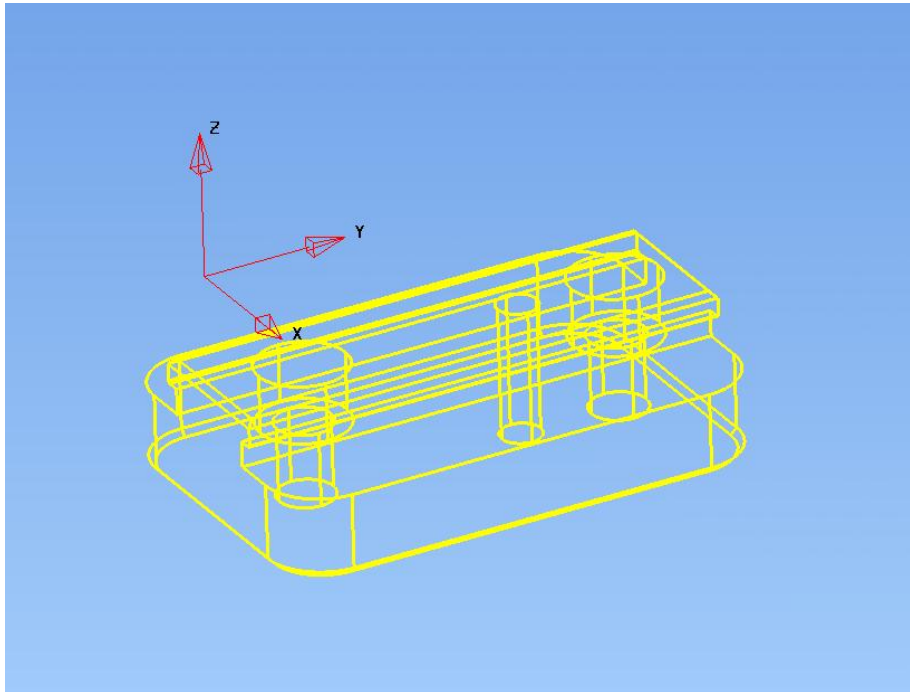


The model contains the slide mechanism and insert, they have been pre assembled.

- Turn OFF levels 5, 6 and 8.



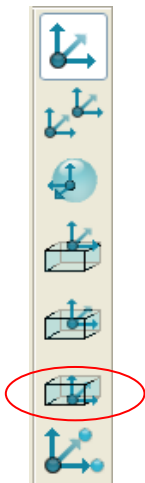
- Select the **CAM-300 GIB** component



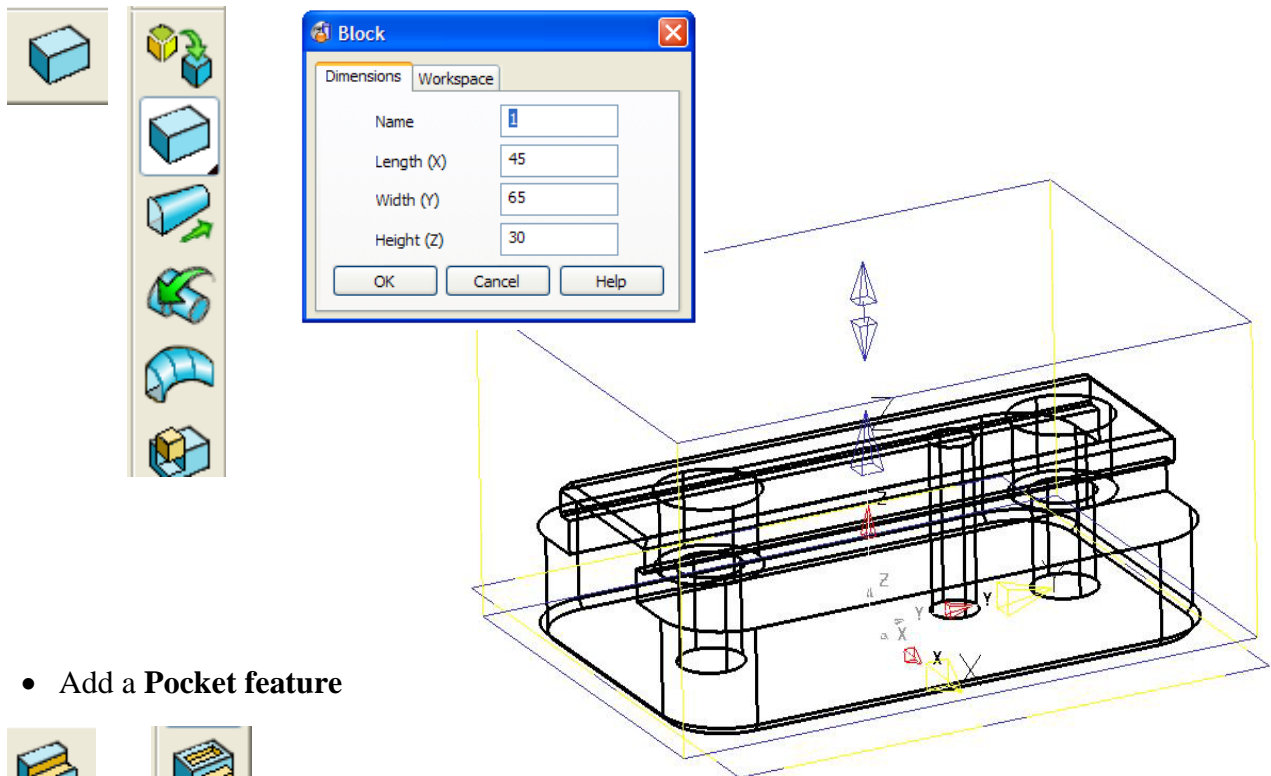
- Create a new **Work Plane**



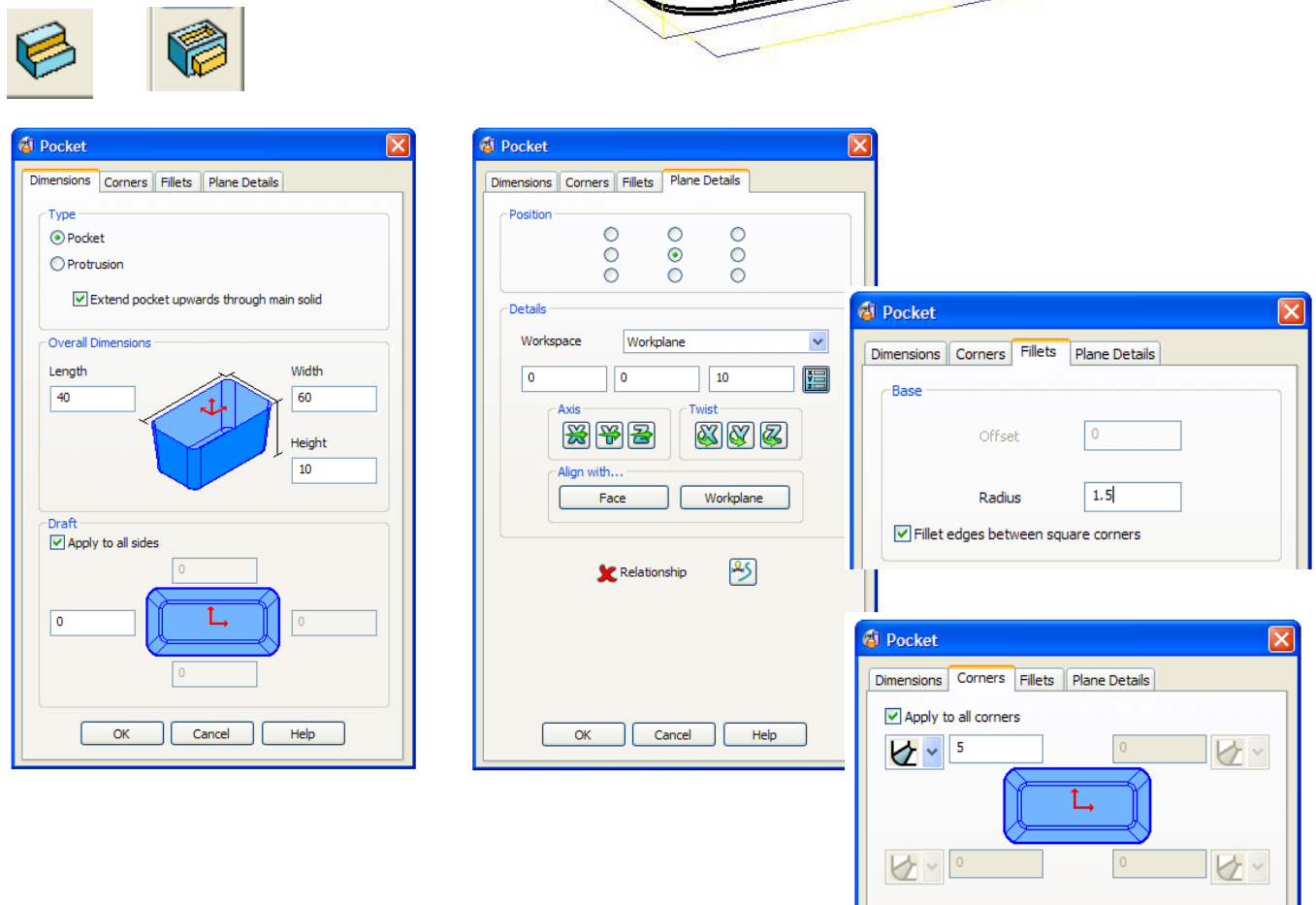
- Create the WP in the **Bottom centre** of the GIB

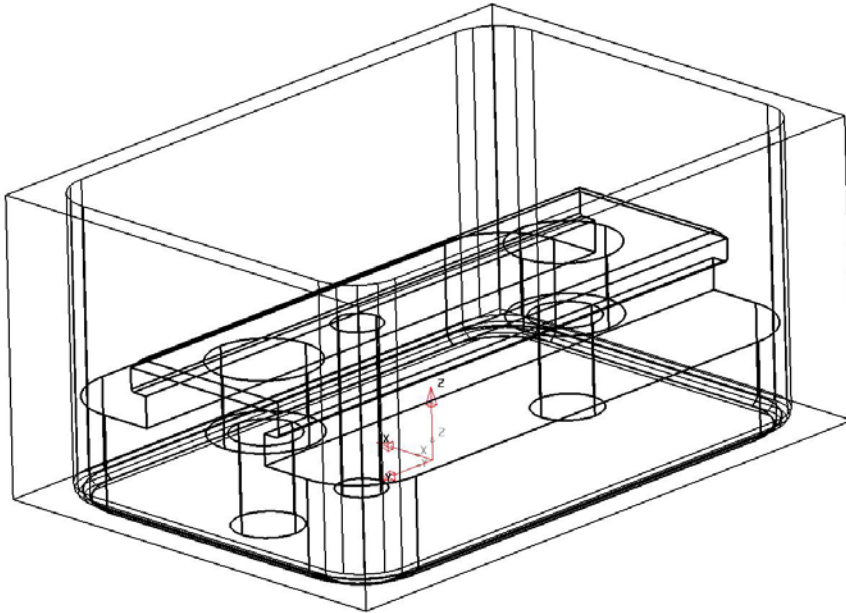


- Create a **Primitive solid block**. 45mm x 65mm x 30mm, centre 0 0 0

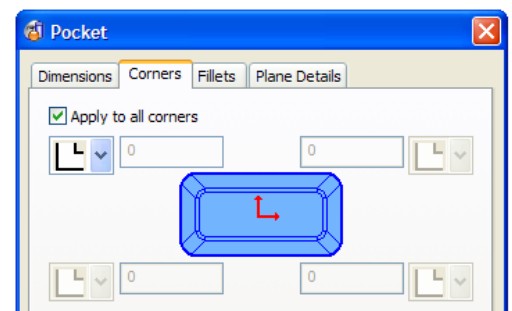
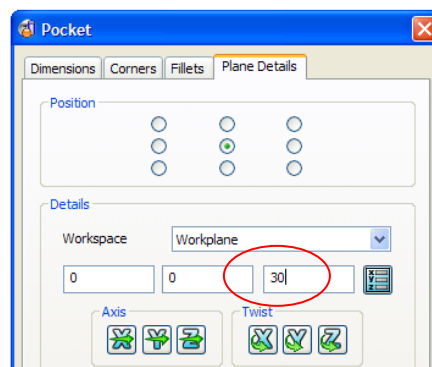
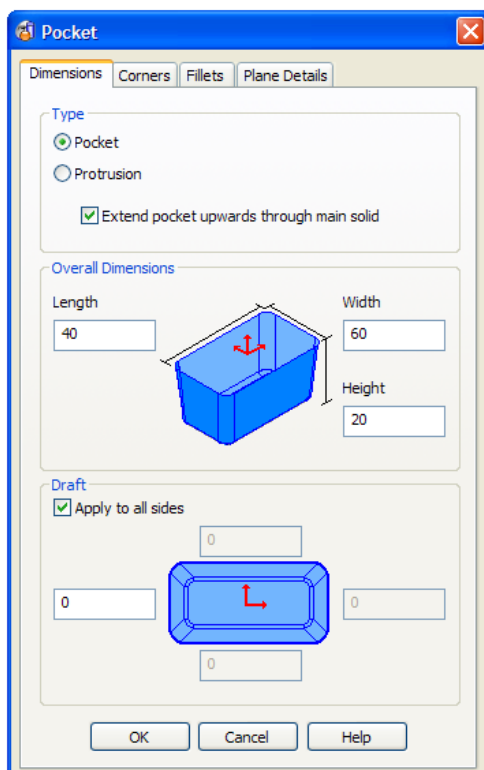


- Add a **Pocket** feature



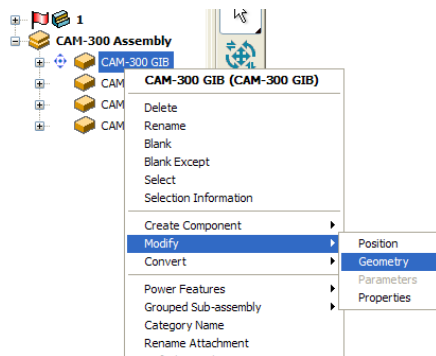


- Add another **Pocket** feature, note the plane Z height and the square corners

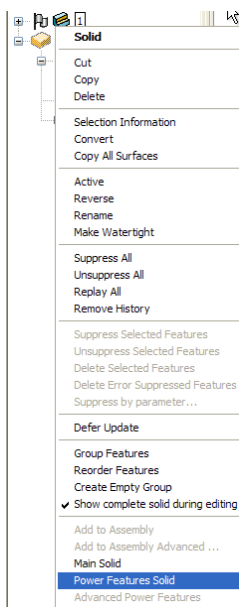


The power feature solid is now created for the GIB component; the next step adds the solid into the component so that it can be used.

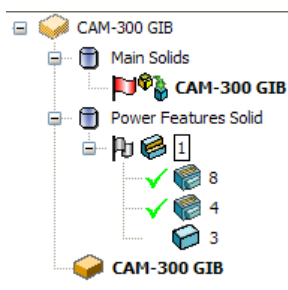
- **Modify the geometry** of the **CAM-300 GIB** component.



- Select the **solid** and pick **Power Feature Solid** from the RMB menu.



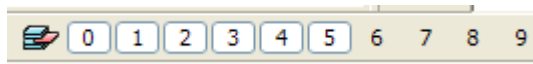
The assembly tree now shows the main and power feature solids in the CAM-300 GIB component



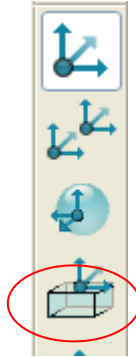
- **Accept the modifications.**



- Display only **Level 5**

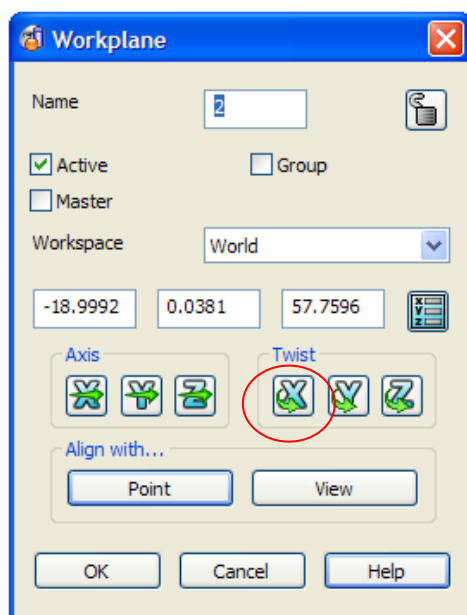
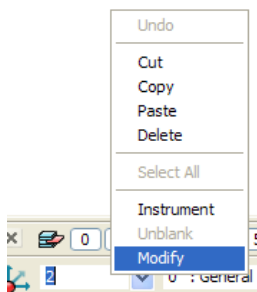


- Select the **CAM-300 HEEL** Component

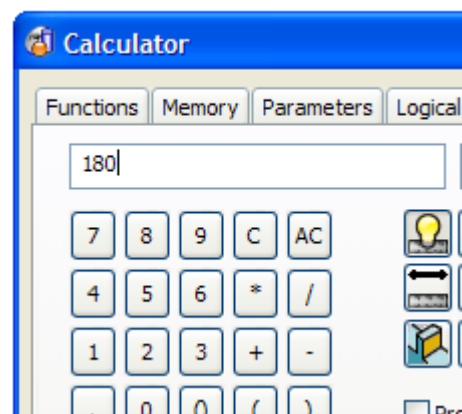


- Create a new Work plane at the **Top of the selection**

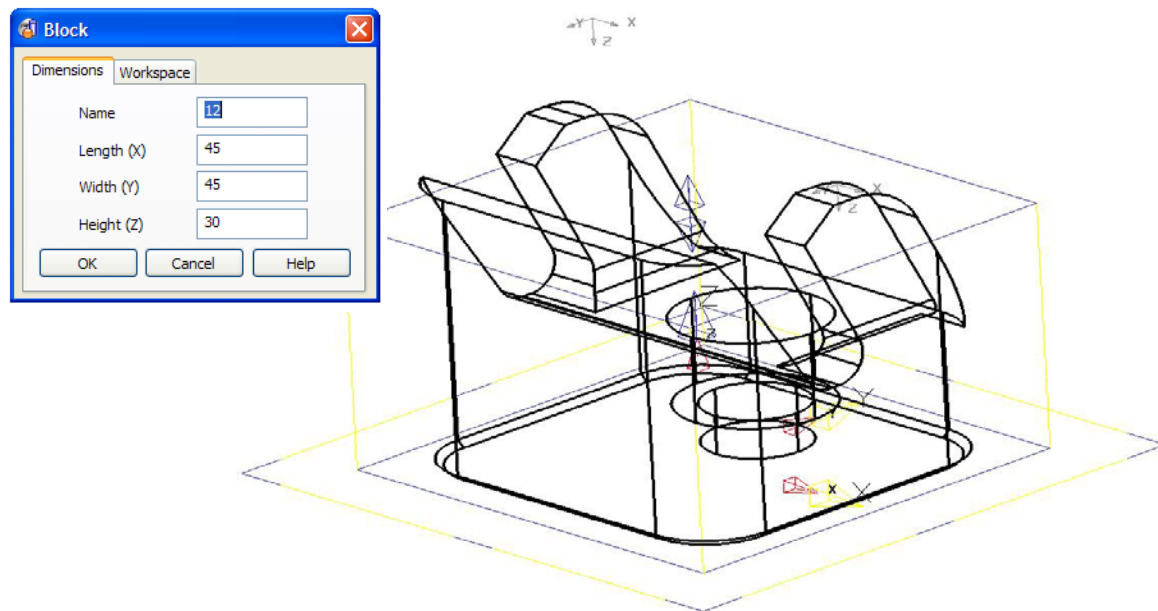
- Select the new Work Plane and **Modify**



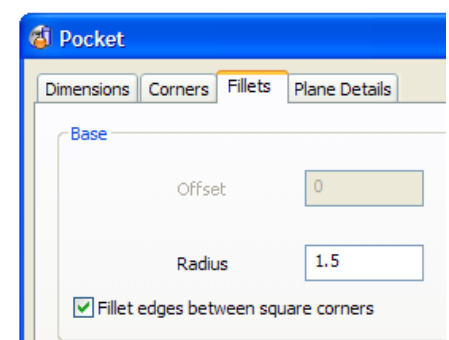
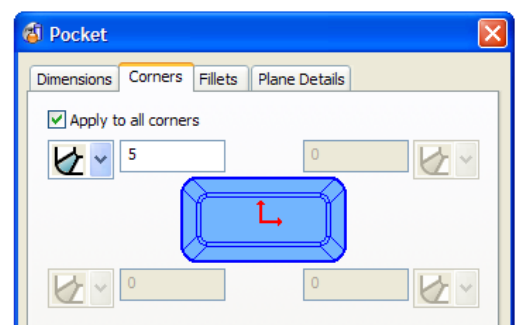
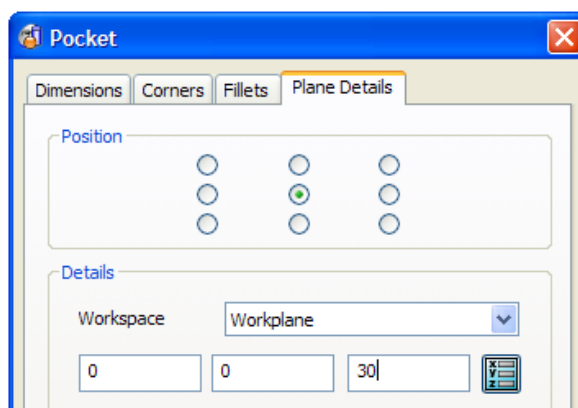
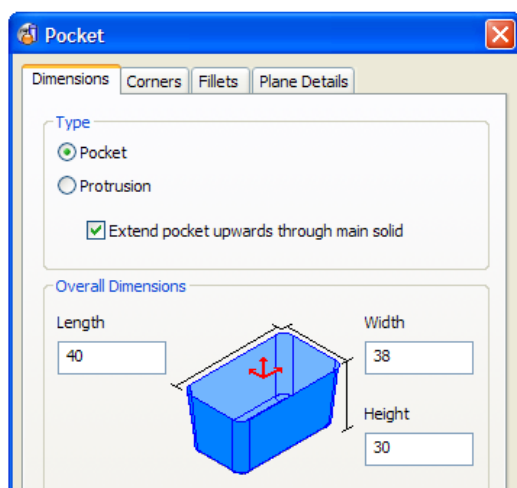
- **Twist the X axis** 180 degrees



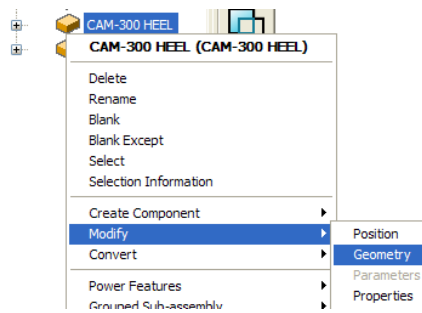
- Create a **primitive solid block** 45mm x 45mm x 30mm, centre 0 0 0



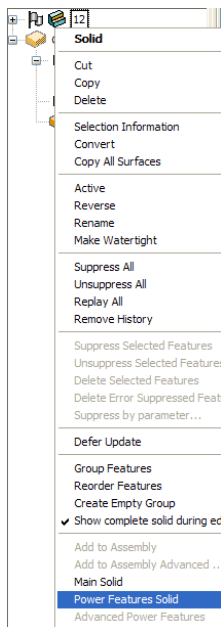
- Add a **Pocket** feature with the dimensions as stated below



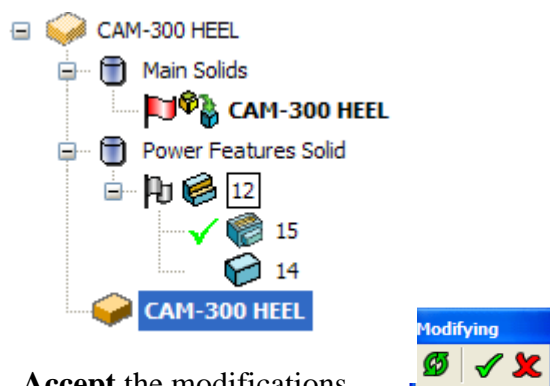
- **Modify the Geometry** of the **CAM-300 HEEL** component



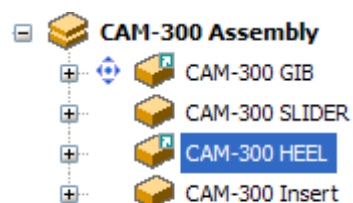
- Select the **Solid** and pick **Power Feature Solid** from the RMB menu



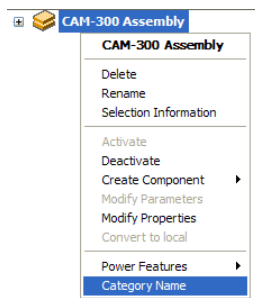
The solid is now the power feature solid for the HEEL component



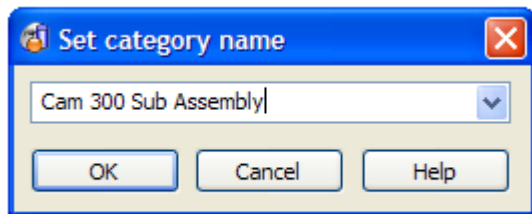
- **Accept the modifications**



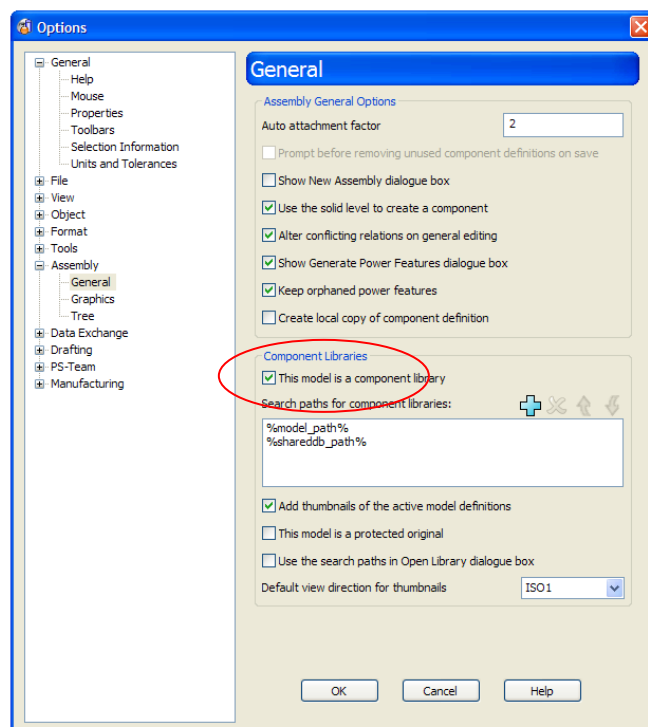
- In the Assembly tree use RMB menu to **Set Category Name**



- Input the **Category name** as shown below and **OK**

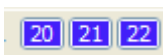


- Open the **Tools – Options – Assembly – General** page
- Set the option **The model is a Component Library**



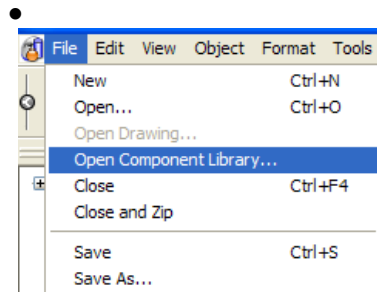
- **Save the Model as CAM-300 EX16**

- Open - **Tool.psmodel**

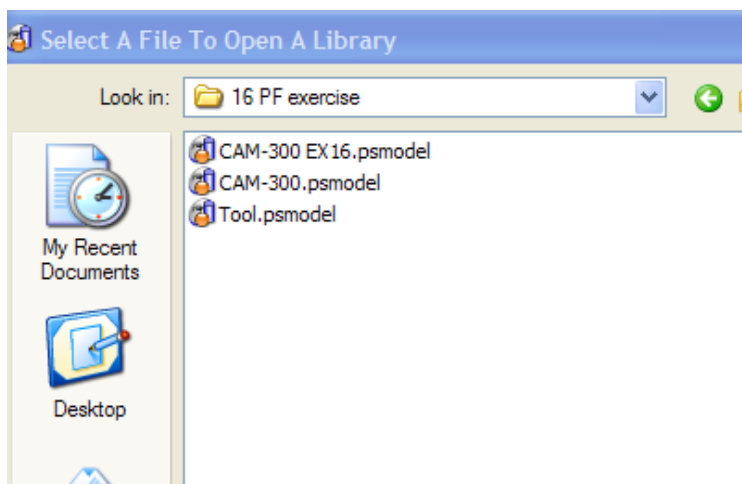


- Display only the **Die Inserts Level**

- From **File** menu select **Open Component Library**

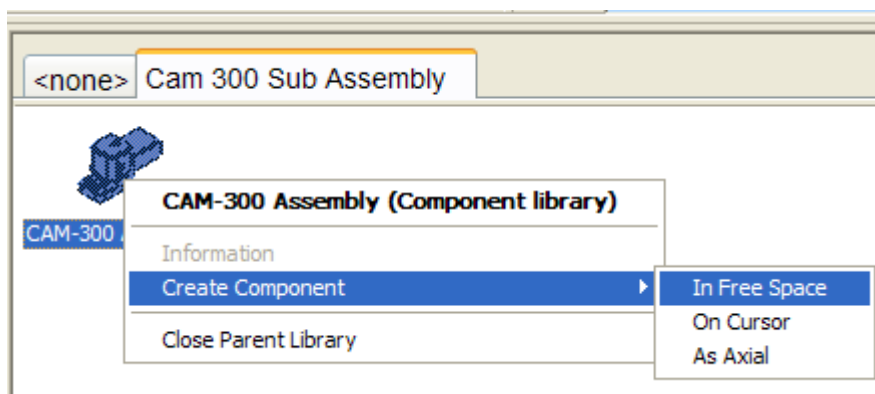


- Choose **CAM-300 EX16.psmodel**

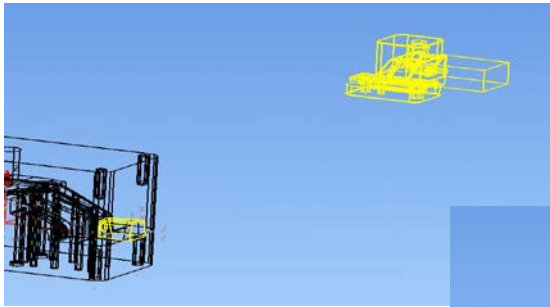


The Component library window will open the definitions will be displayed along the bottom of the window.

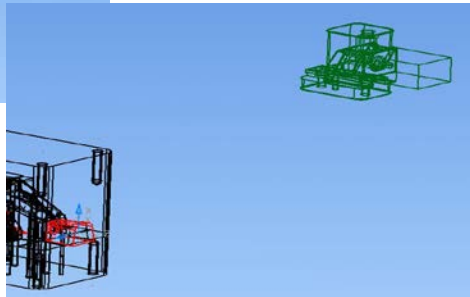
- Select the **CAM-300 Sub Assembly** and **Create Component** using RMB menu



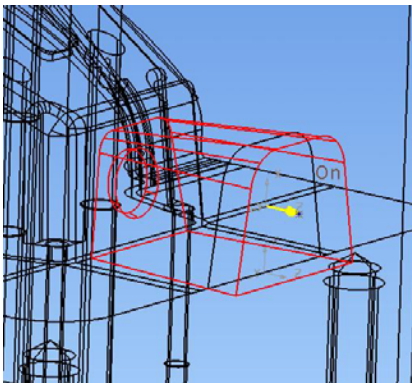
- Select the **Sub Assembly** and the **SW_CORE1**, select the **Create relations** from the assembly toolbar



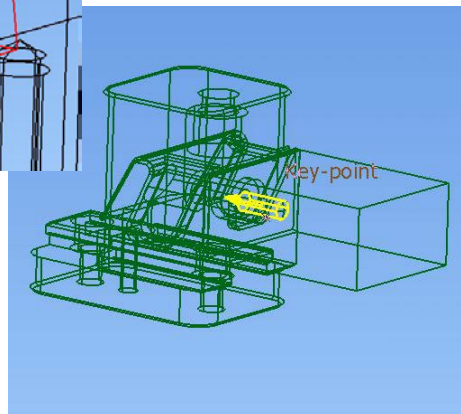
- Ensure that the **Slide Core** component is the **Master**



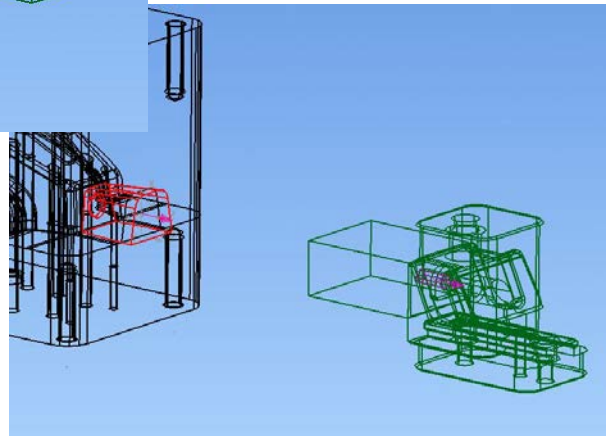
- Select **Line / Line relation** to create



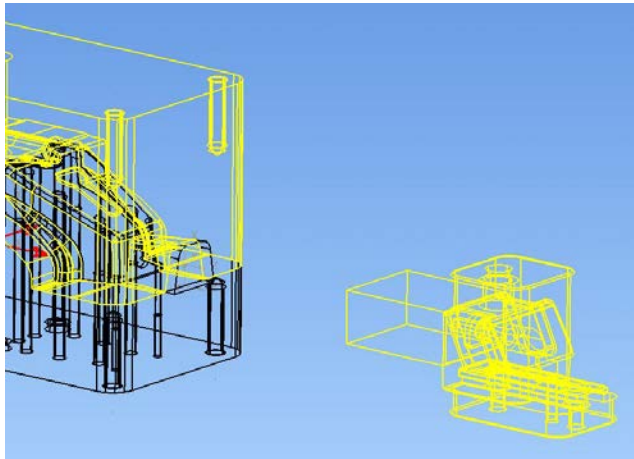
- Pick the **Axis** on the **Slide Core** as shown.



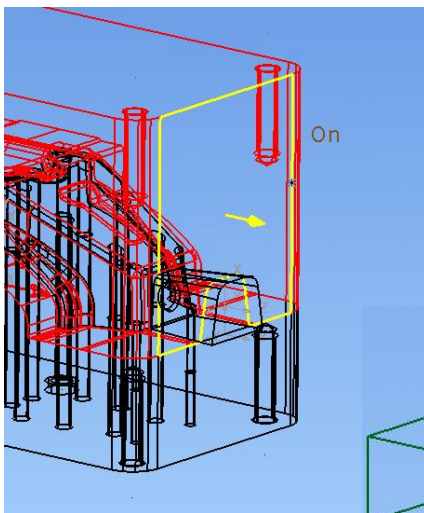
- Pick the **Axis** on the **CAM slide** as shown



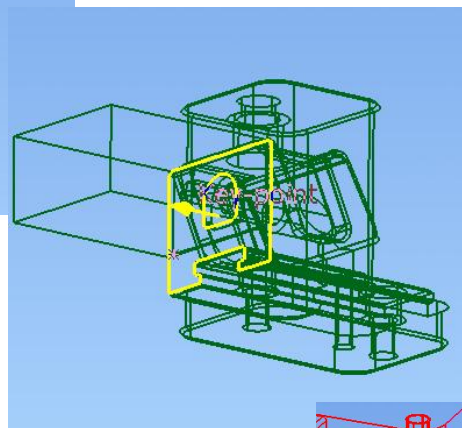
- Select the CAM-300 Slide and the Cavity Insert, **Create Assembly relations**



The type of relation used this time is **Plane / Plane**, Ensure the Cavity Insert is the **Master** component.

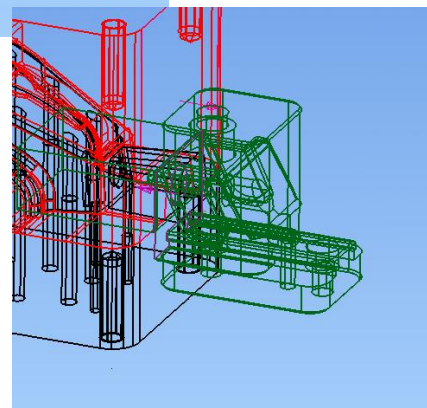


- Pick the end of the Cavity Insert



- Pick the Face on the CAM slide as indicated

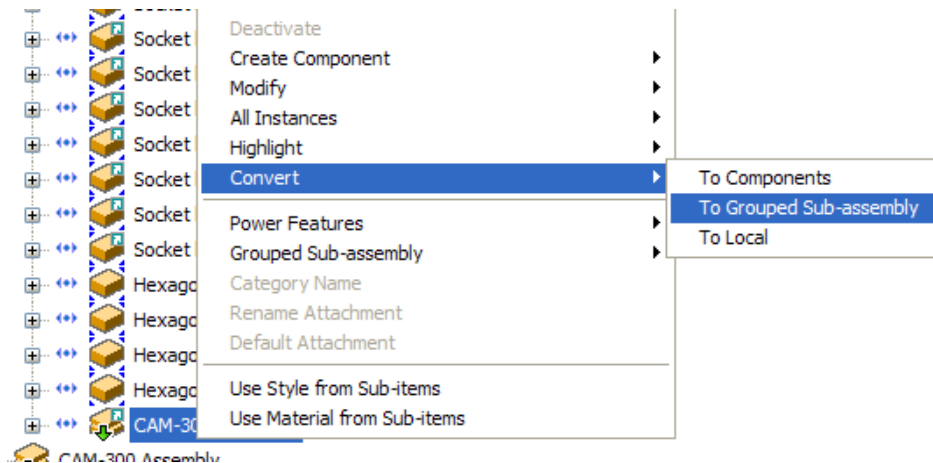
The assembled components should look like this



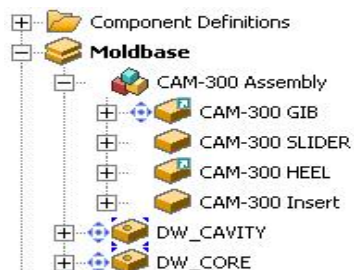
- Display only **Level 22**



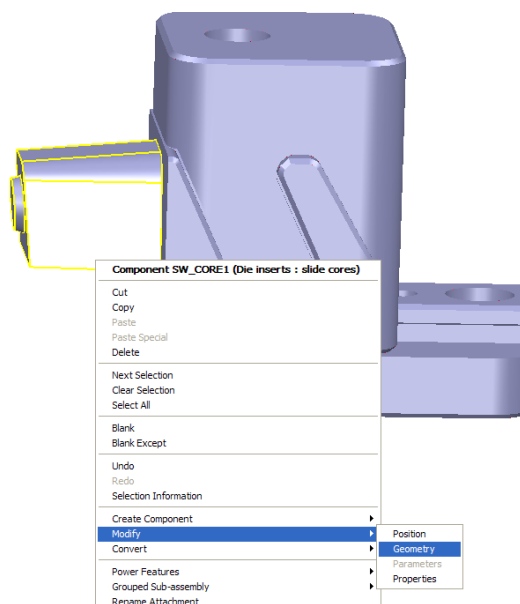
- Select the **CAM-300 Assembly** from the assembly tree and using RMB menu, **Convert into Grouped Sub-Assembly**



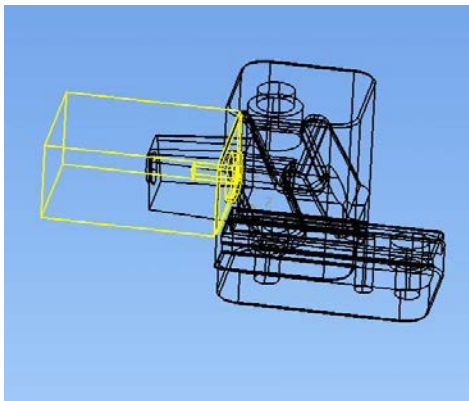
The Group will be created at the top of the assembly tree



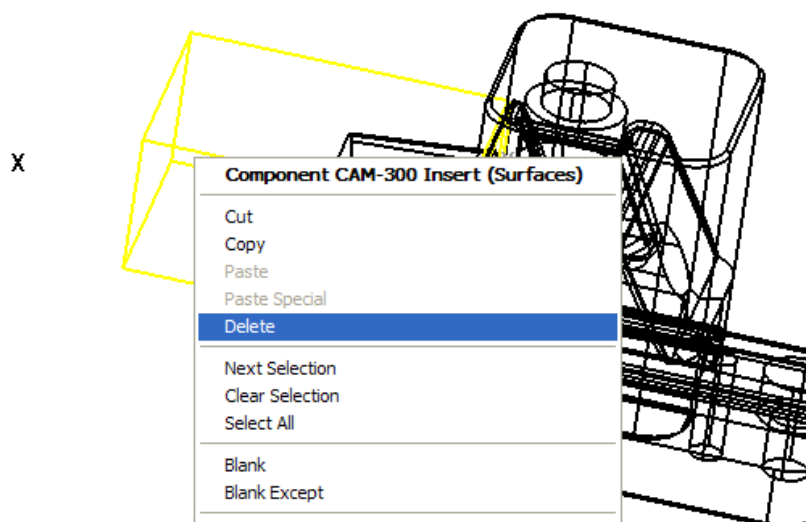
- Select **SW_CORE1** and **Modify Geometry**



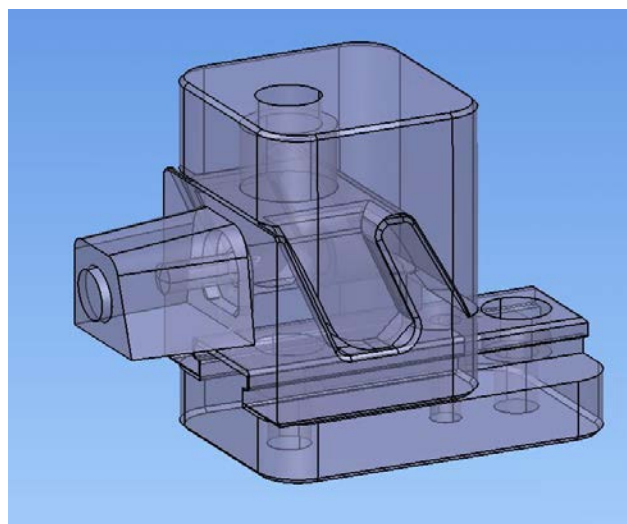
- Select the **CAM-300 Insert** component and perform a **Solid Intersection**.



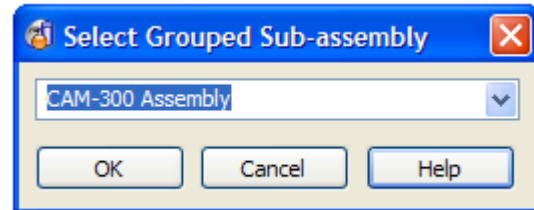
- **Delete** the **CAM-300 Insert** component



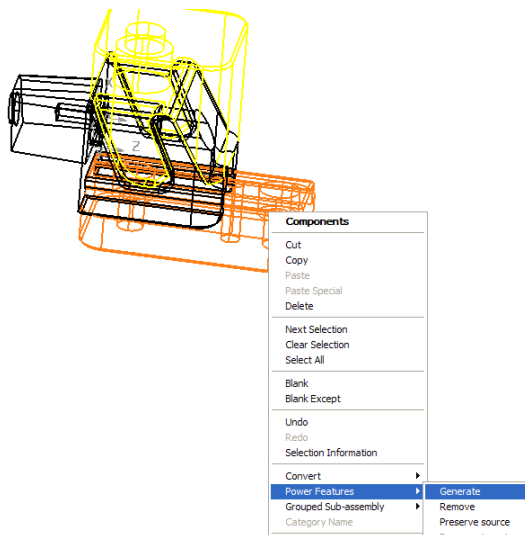
The SW_CORE1 component has now the boss feature that is needed for location into the CAM-300 Slide.



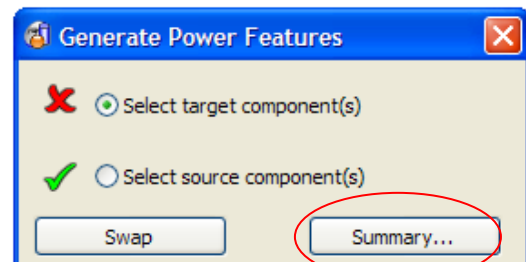
- Select the **SW_CORE1** component and **ADD** it to the **Grouped Sub Assembly** with the other CAM slide components.



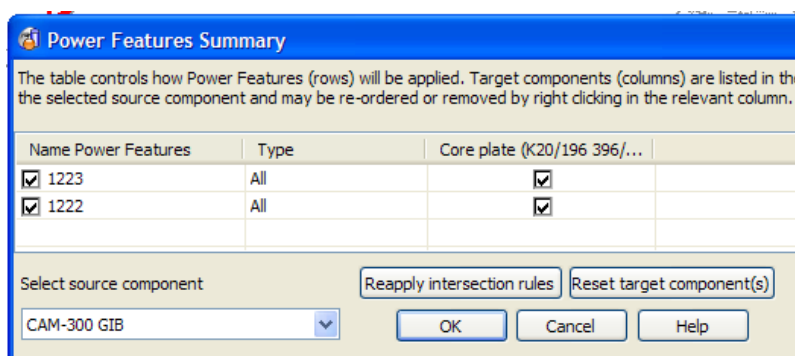
- Select the **CAM-300 GIB** and **CAM-300 HEEL** components and use RMB menu to **Generate Power Features**.



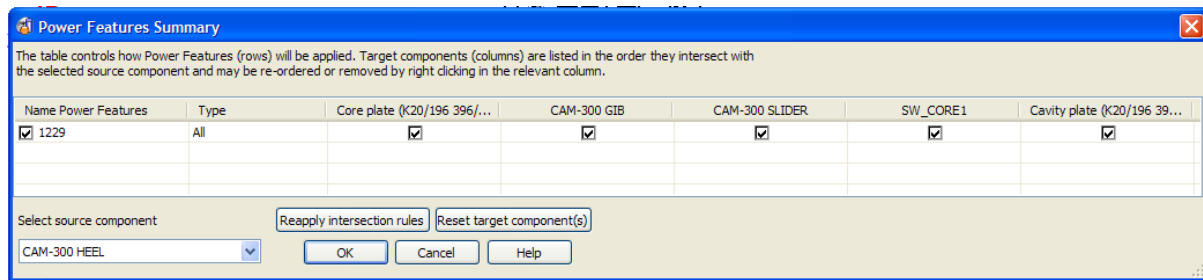
- Pick Summary on the Form



The summary sheet indicates that the Pocket feature on the **CAM-300 GIB** component will only be applied to the **Core Plate**, this is correct.

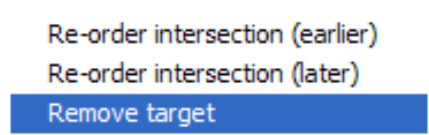


- Change the **Source component** to display the **CAM-300 HEEL**

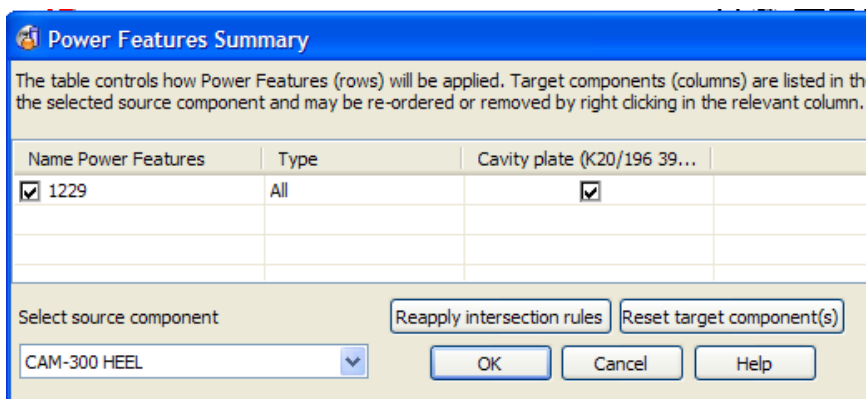


You can see that incorrectly the Pocket feature on the CAM-300 HEEL is going to be copied into many components.

- Use the RMB menu to **Remove all the Target components except the Cavity Plate**



The modified summary should look like this:-

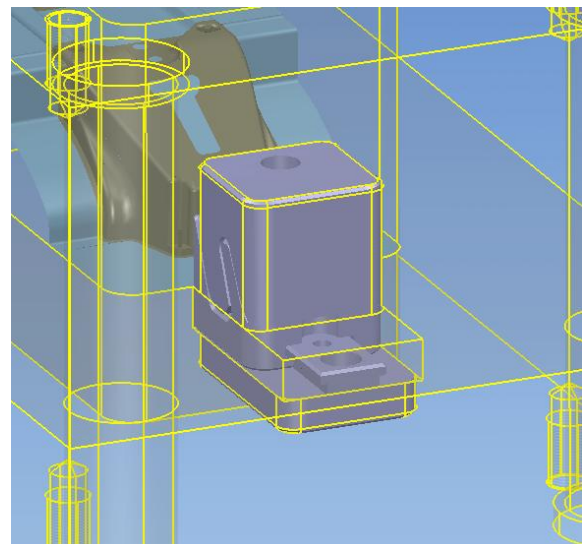


- OK** the Summary and **OK** the Generate Power Features form.

The Pocket Features are cut into the Core and Cavity Plates.

- Repeat** the Exercise for SW_CORE2

Use the definition that is already imported
DO NOT create another from the library



Now that the features have been generated they may not be exactly as required, for instance the corner radius on the pocket in the plates adjacent to the insert pockets should be removed to aid manufacture.

Any changes to the geometry of the power features can be made on the **DEFINITION** by modifying geometry

Also the clearance pocket in the core plate should be slightly larger as it is defined the same size as the slider

- Try some geometry modifications on the power features contained in the **DEFINITION** of the **GIB** component.