
PowerSHAPE 2015 R2

Reference Help

Delcam Toolmaker



PowerSHAPE

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Patent Information

Emboss functionality is subject to patent number GB 2389764 and patent applications US 10/174524 and GB 2410351.

Morphing functionality is subject to patent application GB 2401213.

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Delcam Toolmaker

Use the following sections to find information on Delcam Toolmaker:

- Introduction to Delcam Toolmaker (see page 3)
- Core/Cavity separation (see page 12)
- Cavity-Core Wizard (see page 15)
- Slides (see page 53)
- Mold Base Wizard
- Mold Lock Wizard (see page 112)
- Cooling channels (see page 120)
- Power Features in Delcam Toolmaker (see page 138)
- Mold simulator (see page 142)
- Bill of Material (BOM) (see page 146)
- Requesting quotations and ordering components
- Catalogs (see page 146)
- Drafting for Delcam Toolmaker (see page 160)
- Customising Delcam Toolmaker (see page 161)
- Toolmaker Options (see page 163)

Introduction to Delcam Toolmaker

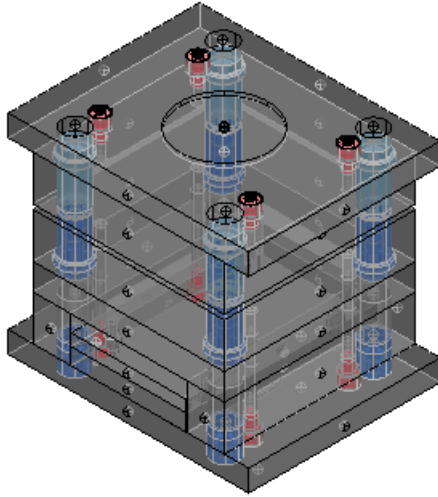
Delcam Toolmaker groups together manufacturing-oriented functionality. It includes

- Plastic injection tool design (previously known as PS-Moldmaker). For further details, see Using Delcam Toolmaker for designing molds (see page 4)
- Assembly modelling. For further details, see What is Assembly modelling?

- Press tool design.

Using Delcam Toolmaker for designing molds

Delcam Toolmaker runs within PowerSHAPE and enables you to design plastic injection mold tools.



There are a number of steps when you are constructing a mold tool. You can

- run a Wizard to build the die blocks - including generation of the parting surfaces
- run Wizards to generate a complete mold base
- pick mold components from standard catalogues.

The components are symbols of 3D PowerSHAPE solids. Therefore, you can use PowerSHAPE to customise a mold design.

When you create components, you can enter non-standard dimensions. You can even generate your own catalogues.

Parametric Components in Delcam Toolmaker

All major components and plates are now fully parameterised. You can easily change the size of any component by re-running one of the Wizards and selecting either standard or custom dimensions.

Any modifications that are made after creation are maintained.

Example:

Pockets and fixing screws have been added to a standard size cavity plate. When placing the cooling channels you realise that the plate needs to be thicker.

The plate can be resized to the next standard thickness, or you can define a custom thickness, whilst maintaining all of the pocket or hole features.

Using Delcam Toolmaker for designing press tools

You can design press tools by combining PowerSHAPE surface modelling techniques and specific Delcam Toolmaker functionality.

- 1 Orient the part using workplane alignment options. For details see [Creating a workplane](#)
- 2 Model the complex die and punch by using surface modelling techniques to create new surfaces and blending smoothly between existing surfaces. For details see [Surface modelling](#).
- 3 Make modifications to existing addendum surfaces using trim region editing and surface limiting tools. Results from formability simulation software will help you identify the changes you need to make.

For further details see:

Limit Selection

Trim region editing

- 4 Use additional press specific functionality to reduce the time take to design press tools. This functionality includes
 - Fill edge surface creation. For details see [Creating a fill edge surface](#)
 - Addendum surface creation
 - Trimline creation
 - Drawbead creation.
- 5 Use solid modelling and assembly modelling to add standard components such as pillars and posts to complete the press die tool. For further details see

[Solid modelling](#)

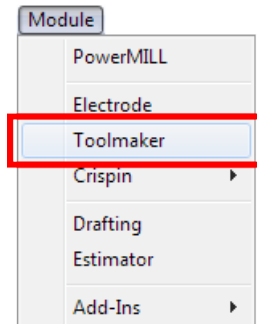
[Introduction to Assembly modelling](#)

[Adding components using the Component Wizard](#)

How do I run Delcam Toolmaker?

Delcam Toolmaker is run from within PowerSHAPE.

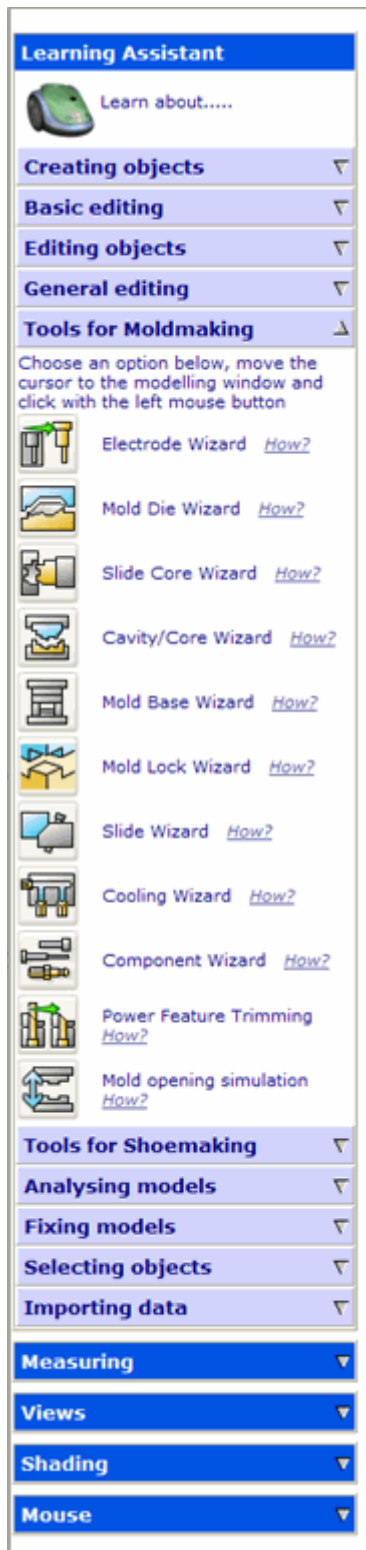
- 1 Start up PowerSHAPE.
- 2 From the **Module** menu, select **Toolmaker** to display the Toolmaker toolbar.



- 3 Select a Toolmaker wizard (see page 8) from the Toolmaker toolbar



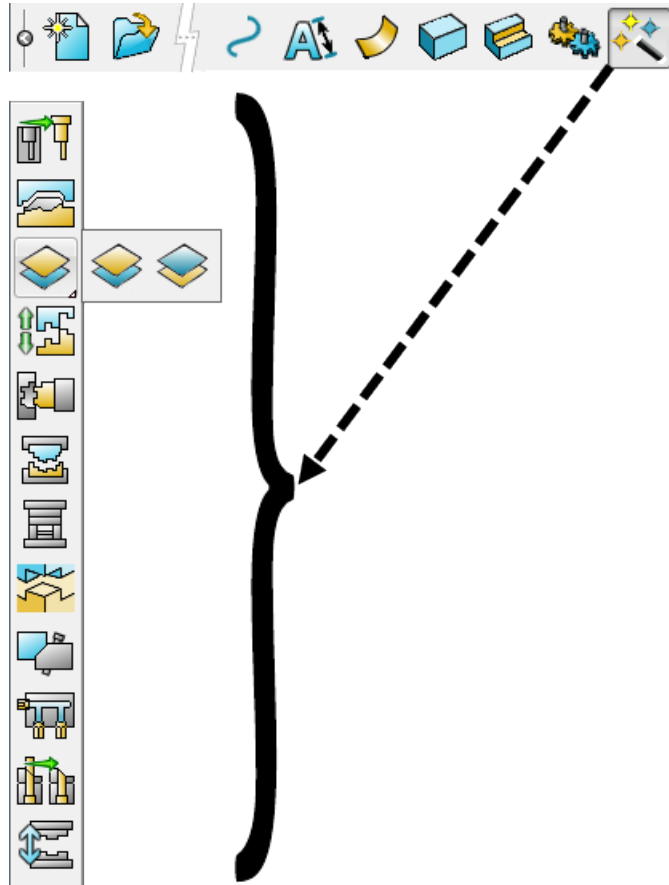
If the **Assistant** is running, the **Tools for Moldmaking** options are displayed in the Assistant window. Select one of the **Tools for Moldmaking** buttons to run a Delcam Toolmaker command.



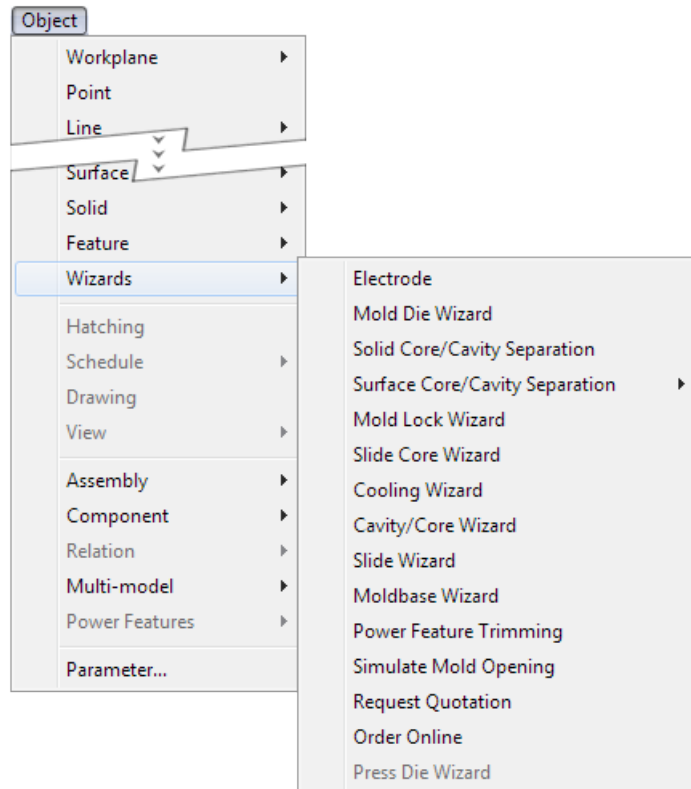
Accessing Toolmaker wizards

With **Delcam Toolmaker** selected from the **Module** menu, you can access the Toolmaker wizards in one of the following ways:

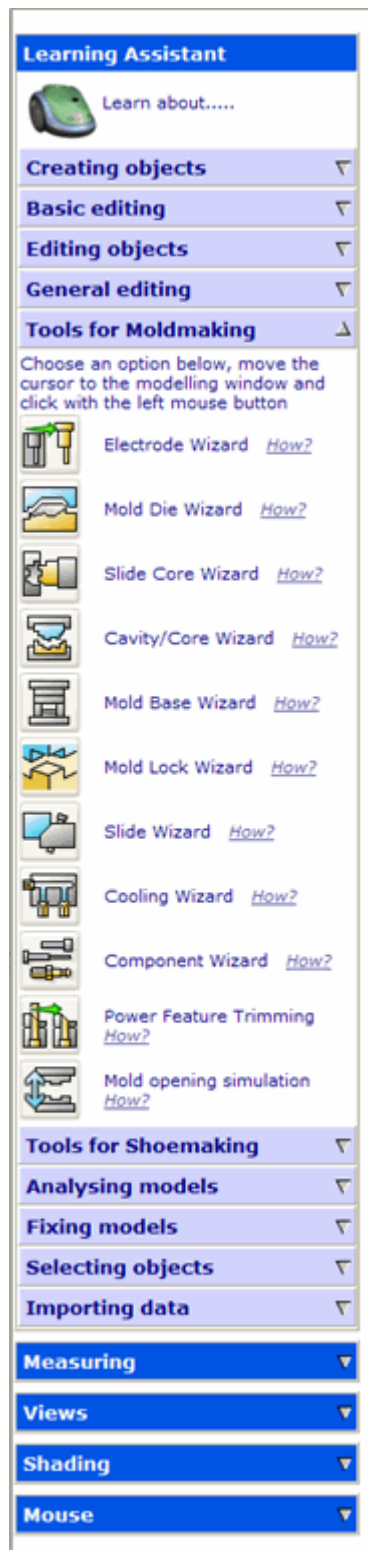
- Select **Wizard** and choose the appropriate wizard from the Toolmaker toolbar. (see page 10)



- Select **Object** from the main toolbar and choose the required wizard from the **Wizards** option.



- From the Learning Assistant, select the required wizard.



Toolmaker toolbar

The following wizards and options are available on the Toolmaker toolbar



Electrode wizard



Mold Die wizard (see page 24)



Core-cavity separation (see page 13)



Cavity-Core wizard (see page 15)



Slide Core wizard (see page 58)



Mold Base wizard (see page 101)



Mold Lock wizard (see page 112)



Slide wizard (see page 83)



Cooling wizard (see page 120)



Power Feature Trimming (see page 139)



Mold simulation (see page 142)

Running macros from Delcam Toolmaker wizards

You can run macros from the following:

- **Mold Lock Wizard.**
- **Slide Core Wizard.**
- **Cooling Wizard.**

Creating a macro

- 1 Create a macro to set some pre-defined values on the dialogs.



Details about creating macros are included in the Customising PowerSHAPE section of the on-line-help or the pdf of the Customising PowerSHAPE manual.

- 2 Create a button to run your macros and add them to a user-defined toolbar.



Details about user-defined toolbars are included in the Basic Modelling Concepts section of the on-line help or the pdf of the Modelling Concepts manual.

Running a macro

When you have created a macro, you can run the macro using one of the following methods:

- Select **Macro > Run** and selecting the required macro from the dialog.
- Click a button on a custom toolbar.

Updates to Delcam Toolmaker

Updates to Delcam Toolmaker appear on Delcam's web site:

<http://updates.delcam.com> (<http://updates.delcam.com>)

If you have valid PAF licenses and maintenance, you can download new versions of Delcam Toolmaker.

New features will be documented in either the Release Notes or the on-line help.

Preview geometry

When you select components within Delcam Toolmaker wizards, they are drawn on the screen as preview geometry. This displays the components as they will appear in the mold.

To adjust the line width of preview objects in Delcam Toolmaker, edit the following in your *powershape.con* file:

```
preview_linewidth: width_number
```

The width is in pixels. The default size is 2 pixels.



You must restart PowerSHAPE to use the new preview_linewidth.

Core / Cavity Separation

Surface Core / Cavity Separation

Surface Core / Cavity Separation (see page 13) lets you find either the upper (**Visible**) or lower (**Hidden**) surfaces of a model, where *up* is defined as the principal axis of the active workplane, or of **World** if there is no active workplane.

When importing a model (for example, using IGES), from which a mold tool will be designed, it is necessary to split the surfaces into those which form the outer face of the component (the cavity side of the mold) and those which form the inner face of the component (the core side of the mold). Some models (especially IGES) can contain thousands of surfaces, some of which will be in close proximity, but each describing different parts of the mold. Separating them manually would be a difficult and tedious task, but this option calculates the split quickly and easily.

Solid Core / Cavity Separation



When using **Solid Core/Cavity Separation**:

- solids are split based on the natural horizons from looking down the Z axis of the active workplane.
- parts are automatically split into core/cavity groups.
- you can move faces between different groups.
- any number of split directions can be used to define sliders.
- you can edit the split line by sketching new wireframe and dividing the relevant faces. The new faces can then be assigned to the relevant split group.

Using Core - Cavity Separation

This lets you find either the upper (**Visible**) or lower (**Hidden**) surfaces of a model.

Use the options as follows:

- 1 Open the model.
- 2 Add a workplane if the split is not going to be down the Z-axis of the global workspace. Position and orient the workplane accordingly.
- 3 Click  to display the Wizard toolbar. The buttons that are displayed on the toolbar will reflect the module you are running.
- 4 Click **Surface Core/Cavity**  flyout (*Wizard toolbar*)
- 5 Click the appropriate option on the flyout.



- **Visible Surfaces**. This finds and selects all the visible surfaces of a model.



- **Hidden Surfaces**. This finds and selects all the hidden surfaces of a model.

You can now move the selected surfaces to a new level.




When you have run the wizard and assigned the upper and lower parts to levels, you are left with parts that fall into one of the following categories:

- parallel to the point of view
- ambiguous (could be upper or lower)
- totally hidden inside the model

Surface Core / Cavity Separation - an example using the interface



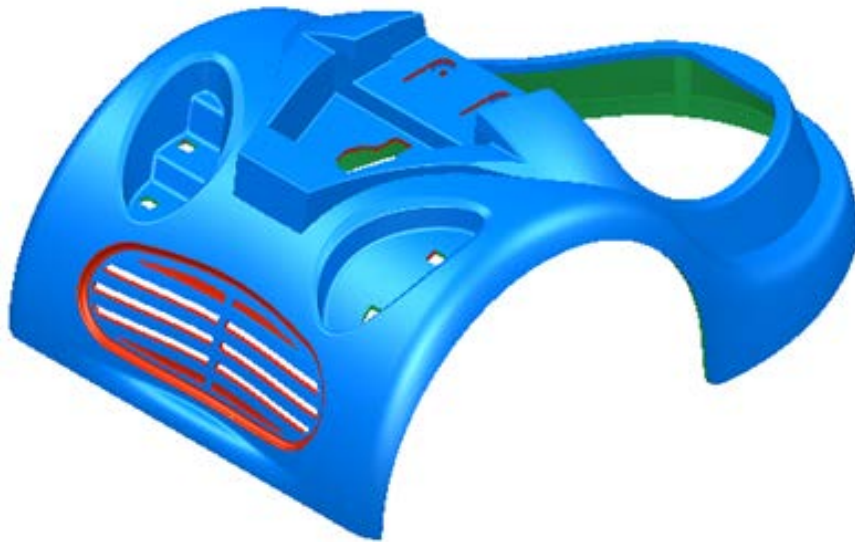
The model above can be used to illustrate the surface core/cavity separation functionality.

- 1 Click  to display the Wizard toolbar.
- 2 Click **Surface Core/Cavity**  flyout (*Wizard toolbar*)
- 3 Click **Visible Surfaces** . This finds and selects all the visible surfaces of the model.

The split model will have the following surfaces

- upper
- lower

ambiguous/vertical/totally hidden



Cavity-Core wizard

Using the Cavity-Core wizard

This wizard creates the cavity and core plates for the model. You can:

- Create non-inserted plates (see page 18)
- Create die inserts in the plates (see page 49)

Running the Cavity-Core wizard

- 1 Click  on the **Toolmaker** toolbar.

This displays the **Cavity-Core Wizard** in the top part of the browser window.

Cavity-Core Wizard

☐ Non-inserted core/cavity plates.
☒ Inserted core/cavity plates with inserts

Creating Inserts

Run Mold Die Wizard...

No insert defined yet!
Run Mold Die Wizard or register inserts.

Register Inserts

Register

☐ Display the names of the selected items.

☒ Select workplane ✗
☐ Select cavity solid ✗
☐ Select core solid ✗

< Back Next > Cancel Help

2 Choose one of the following options.

☐ Non-inserted core/cavity plates.
☒ Inserted core/cavity plates with inserts

As you select a different option, the rest of page changes to help you decide what to do next.

Non-inserted core/cavity plates - The wizard displays the **Register Plates** options.

You can do one of the following:

- Register solids as core and cavity plates and click **Next** to continue using the wizard. For further details on registering solids, see Registering solids as inserts/plates (see page 17)

- Click **Next** to continue using the wizard.

For further details, see Creating non-inserted plates (see page 18).

Inserted core/cavity plates with insert(s) - The wizard displays the **Creating Inserts and Register Inserts** options.

You can do the following:

- Register solids as core and cavity inserts. For further details, see Registering solids as inserts/plates (see page 17)
- Use the **Mold Die Wizard** to create core and cavity inserts. For further details, see Using the Mold Die Wizard (see page 24)

Once you have created or registered all the required inserts, click **Next** to create the plates. For further details, see Defining the plates (see page 22).

Registering solids as inserts/plates

To register the solids as inserts/plates,

- 1 Turn on **Display the names of the selected items** if you want to check the names of the objects as you select them.
- 2 Turn on **Select Workplane**.
- 3 Select the workplane.


To register the solids, you must also have a workplane at the centre of the plates. Its Z axis represents the main opening and closing direction of the mold.



- 4 Turn on **Select cavity solid**.
- 5 Select the solid which represents the cavity solid.
- 6 Turn on **Select core solid**.
- 7 Select the solid which represents the core solid.

If the objects are not suitable, an error message will appear in the wizard.

- 8 Click **Register**.

You can register multiple inserts by repeating the above steps.

Once you have either registered the solids as core and cavity inserts/plates, a table showing the sets of Cavity/Core/Inserts is displayed in the wizard. You can check the items by clicking on each  icon in turn.

	Cavity	Core	Inserts
1			

Creating non-inserted plates

The **Non-inserted core/cavity plates** option creates impressions directly into the core and cavity plates, without using separate inserts.

The dialog box is titled 'Mold Base' and contains three main sections: 'Mold Base', 'Cavity plate', and 'Core plate'. The 'Mold Base' section includes a 'Catalogue' dropdown menu with '<Choose a catalogue>' selected, a 'Type' dropdown menu, 'Guide pillars' radio buttons for 'Fixed half' (selected) and 'Moving half', a 'Show the mold base' link, and color-coded boxes for 'Fixed half' (red) and 'Moving half' (green). The 'Cavity plate' section has a 'Type' dropdown menu, a 'Material' dropdown menu, and a 'Comments...' button. The 'Core plate' section has a 'Type' dropdown menu, a 'Material' dropdown menu, a 'Comments...' button, and a 'Levels...' button. At the bottom are navigation buttons: '< Back', 'Next >', 'Cancel', and 'Help'.

- 1 Define the core and cavity plates by selecting the options you require. For further details, see Defining the plates (see page 22)

As you select the plates, they are drawn on the screen. The fixed half is red and the moving half is green (using the default colour scheme).

- 2 Click **Next**.



- 3 If no plates are registered, click **Run Mold die wizard** to start up the Mold die wizard and use it to create the required plates. It will automatically use the standard plates sizes. If you want to use custom plates, simply change the sizes of the plates in the wizard.

For more see Using the Mold die wizard (see page 24).

- 4 If you want to create slides, click **Insert slide mechanism** to start the **Slide** wizard. Use the wizard to create slides. For further details, see Creating the slide assembly using the Slide wizard (see page 82).
- 5 Click **Next** to display the following page.

Dimensions

Mold base

Dimensions

☒ Standard
☐ Custom

95x95

Ejection system type

Standard (1)

☒ Fixed half
☒ Moving half
☐ Ejection system

Rotate the mold base:

☐
☐
☐
☐

Cavity plate

Thickness

9

Material

2162

Dimensions: Standard, mms

Product code: K20/95x95x9/1.2162

Dimensions...

Core plate

Thickness

9

Material

2162

Dimensions: Standard, mms

Product code : K20/95x95x9/1.2162

Dimensions...

< Back

Next >

Cancel

Help

If you click the **Dimensions** button, you can see a preview of the ejector footprint and holes in the core and cavity plates in the graphics window,

- 6 Click **Next** to display the following page.

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Cavity-Core Wizard

You have successfully completed the Cavity/Core wizard

You specified the following parameters:

Mold base	
Catalogue	HASCO
Type	Standard mold
Guide pillars	Fixed half
Rotation angle	270
Width	95
Length	95
Dimensions	Standard

Cavity plate	
Type	K20
Thickness	9
Material	2162
Dimensions	Standard

Core plate	
Type	K20
Thickness	9
Material	2162
Dimensions	Standard

Press the "Finish" button to create the cavity and core plates.

☒ Run the Mold Base Wizard on exit

< Back Finish Cancel Help

- Use this page to check your parameters and then create the cavity and core plates.

Run the Mold Base Wizard on exit - This automatically runs the **Mold Base Wizard** after the cavity and core plates are created.

Back - If you want to change the values, click **Back** to go back to previous pages.

- Click **Finish** to create the cavity and core plates.

Each plate contains split surfaces and guide pillar/pin holes. The plates are created in an assembly. For further details on assemblies, see What is an assembly?.

If the **Run the Mold Base Wizard on exit** is selected, the **Mold Base Wizard** is run after the cavity and core plates are created. For further details, see Using the Mold Base Wizard (*see page 101*).

Defining the plates

Use this page to select the cavity and core plates from the catalogue.

Mold Base

Catalogue: HASCO

Type: Standard mold

Guide pillars: ☒ Fixed half ☐ Moving half

[Show the mold base](#)

■ Fixed half ■ Moving half

Cavity plate

Type: K20

Material: 2162 [Comments...](#)

Core plate

Type: K20

Material: 2162 [Comments...](#)

[Levels...](#)

< Back Next > Cancel Help

Catalogue - Choose a catalogue from the menu. Delcam Toolmaker then determines the best size for the mold base from the catalogue and displays the values.

Type - Select the type of mold base.

Guide pillars - This option selects which half contains the guide pillars. You can choose **Fixed half** or **Moving half**.

Cavity plate - Select the **Type** and **Material** of cavity plate which is available from the catalogue.

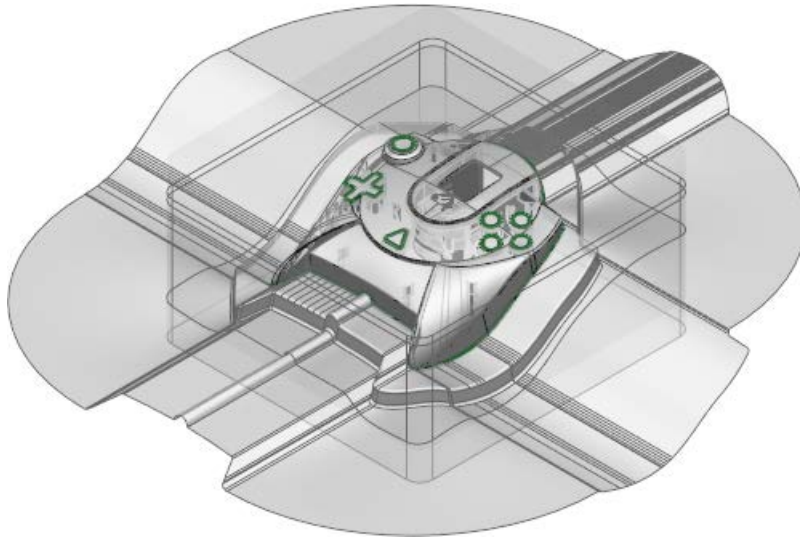
Core plate - Select the **Type** and **Material** of core plate which is available from the catalogue.



As you select the plates, they are drawn on the screen. The fixed half is red and the moving half is green (using the default colour scheme).

Levels - This allows you specify on which level to put each component of the mold base. For further details, see First page of the Mold Base Wizard (see page 103).

Next - This displays the page to define the position of the inserts in your cavity and core plates. For further details, see Defining the inserts/pockets (see page 49).

Using the Mold Die Wizard



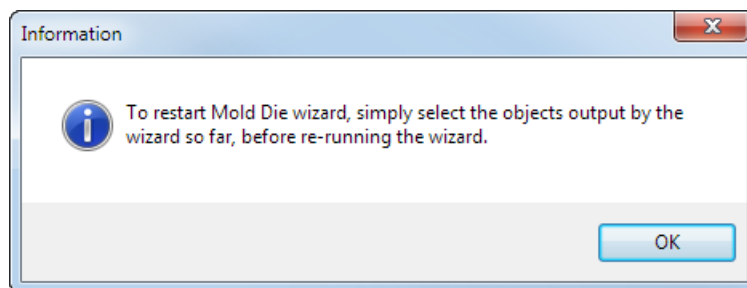
- 1 Start the Mold Die Wizard using one of the following methods:
 - Select the **Object** menu. Choose **Wizards** and then **Mold Die Wizard**.
 - Click the **Wizard** button  and from the wizards toolbar, select the **Mold Die Wizard** button .
- 2 Select the objects using the first page of the wizard, Mold Die Wizard Step 1 - Select objects (see page 25)
- 3 Create and edit the split lines using the second page of the wizard, Mold Die Wizard Step 2 - Creating and Editing split lines (see page 28)
- 4 Create and edit fill-in surfaces using the third page of the wizard, Mold Die Wizard Step 3 - Creating and editing fill-in surfaces (see page 30)
- 5 Define the product shrinkage using the fourth page of the wizard, Mold Die Wizard Step 4 - Defining Product Shrinkage (see page 32)
- 6 Define the block size using the fifth page of the wizard, Mold Die Wizard Step 5 - Defining the block size (see page 33)
- 7 Create the split surface using the sixth page of the wizard, Mold Die Wizard Step 6 - Creating the split surface (see page 36)

- 8 Define the block heights using the seventh page of the wizard, Mold Die Wizard Step 7 - Defining the block heights (see page 40)
- 9 Define the corner types using the eighth page of the wizard, Mold Die Wizard Step 8 - Defining the corner types (see page 43)
- 10 Simulate the opening of the core/cavity using the ninth page of the wizard, Mold Die Wizard Step 9 - Simulating the core/cavity split (see page 46)

Tips on using the Mold Die Wizard

When using the Mold Die Wizard, it will be helpful to remember that:

- You can leave the wizard at any time by clicking the **Finish** button on the wizard. You can modify your data manually and start the wizard again.
- If you exit Mold Die Wizard without splitting the mold, you can select the objects that the wizard has already created and re-run the wizard.



- You can click the **Back** button on any page of the Wizard (except the final page) to edit settings in previous steps.

Using split surfaces with the Mold Die Wizard

You can supply two sets of split surfaces to the wizard,

- one to be used for the core insert
- one for the cavity.

This allows you to build relief into regions of the mold so that pressure can be increased at vital points of the split surface. To use this functionality, the two sets of split surfaces must be supplied as two solids, otherwise the wizard will assume that you only have one set of surfaces.

If you supply a mixture of solids and surfaces, the wizard assumes that the surfaces are common to both inserts. Solids will be analysed to see if they form a cavity/core pair with another solid. If so, one of the solids will be part of the core split surface and the other solid will be part of the cavity split surface. If a solid split surface cannot be paired then it is assumed to be common to both inserts.

Exiting Mold Die Wizard without splitting the mold

If you exit the wizard without splitting the mold, the messages that are displayed reflect the functionality that you have been using.

For example,

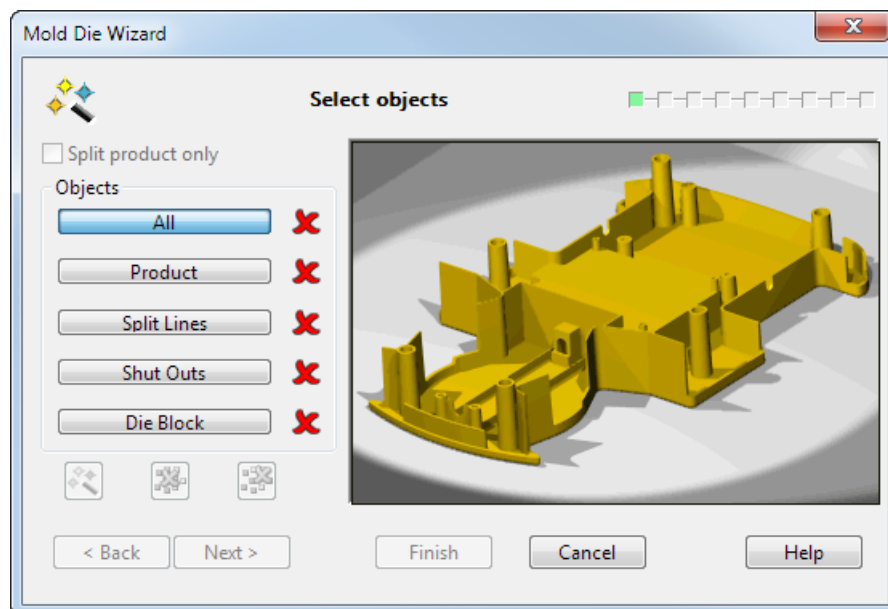
Shrinkage has not been applied - if you specified shrinkage.

Stepped surfaces are output as solids - if stepped surfaces were created.

Delcam Toolmaker will only recognise complete inserts named DW_CORE and DW_CAVITY - if the Mold Die Wizard was run from Delcam Toolmaker.

Mold Die Wizard Step 1 - Select objects

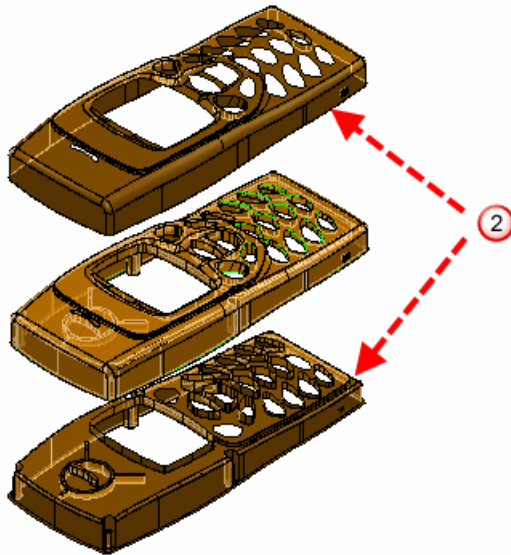
This lets you choose the objects to be used within the wizard.



The image on the right of the wizard shows an example part. The image changes to illustrate the types of objects you should select from your own part when you choose a selection option.

If you select objects before starting the wizard, they are automatically classified into categories. You may then alter this classification. You can use *CTRL-A* to select all the items that match the **Objects** selection on the **Mold Die Wizard Step 1 - Select Objects** page. To use this feature, you must make the background active by clicking on the window away from the wizard.

Split product only — Click this option if you only want to create the core and cavity surfaces. No die block or run off surfaces will be created.



Selecting this option will hide any buttons or options on the dialog that are not required. This option is not available if you are running the Mold Die Wizard with **Non-inserted core/cavity plates** selected.

All — Click this option and select all the objects you want to use.

You can then click the **Classify** button  to put the objects in their correct categories.

Product — Click this option and select the part you want to use.

A product can be:

- A single, connected solid.
- A set of surfaces as long as they can be converted into a single, connected solid. During the operation, the surfaces are automatically converted into a solid.

Split Lines — Click this option and select the split lines you want to use.

The split lines are composite curves. The main split line is used to create the split surface and the other split lines are used to create fill-in surfaces, which shut out holes in the product solid.

If more than one composite curve is selected, PowerSHAPE treats the outermost curve as the split line and uses the inner curves to create fill-in surfaces.

This page of the wizard allows you to select existing composite curves as split lines. Page 1 of the wizard allows you to generate split lines using methods available within the wizard.

Shut Outs — Click this option and select the split surfaces you want to use. You can select:

- Split surfaces. The wizard will later automatically trim these surfaces to the block.

These surfaces can be used to fill awkward regions where it is difficult to create a single split surface.

- Surfaces, which represent filled holes in the die inserts. Note that each group of fill-in surfaces must be enclosed by a composite curve.

Die Block — This option is not available if you are running the Mold Die Wizard with **Non-inserted core/cavity plates** selected.

Click this option and select the die block you want to use. You can select:

- A solid primitive block.
- Any other solid, but you can't change its size using the wizard. The solid must contain the part.

A die block is automatically created by the wizard if one is not supplied.



— If objects are selected using the **All** option, click this option to filter them into their correct groups.



— This clears all the selections.

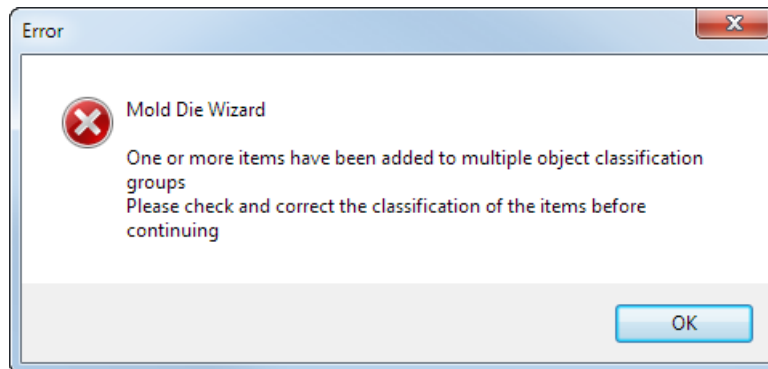


— This clears the current selection.

Next — Takes you to Step 2 of the wizard. If you entered the wizard with no split lines, the wizard will attempt to generate the split lines for you. For subsequent pages of the Mold Die Wizard, all surfaces are viewed trimmed.

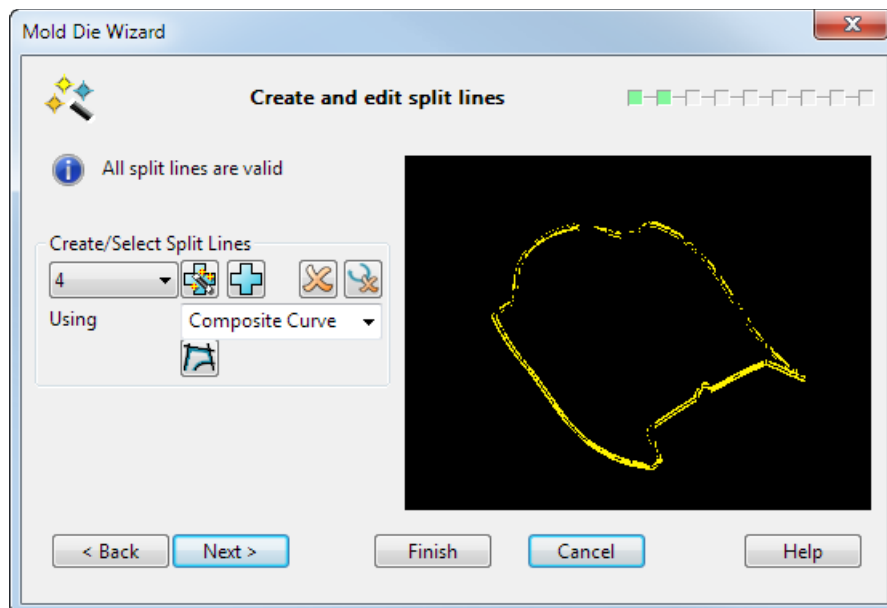



You are warned if you have selected the same item in two classification groups (for example both the Product and Die Block group)



Mold Die Wizard Step 2 - Creating and Editing split lines

This step lets you generate split lines.




 **All split lines are valid** — This information string displays the current overall status of the split lines. There are four possible messages:

All split lines are valid

No split lines created

Invalid split lines present

Create/Select Split Lines

 **4** — This selection box displays additional information for the user regarding the validity of the split line.

If a split line is invalid, **(invalid)** is displayed next to the number of the split line.

If a new selection is added that is yet to be defined, **(new)** is displayed next to the number of the split line.



— Attempt to generate split lines automatically. If you enter the wizard with no split lines, the wizard will auto-generate them for you when you select **Next** on the first page of the wizard.



— Add split line/surface selection. This adds another selection option to the **Create/Select Split Lines** drop down list.



— Delete split line/surface selection. Delete the split line selection that is currently shown in the **Create/Select Split Lines** drop down list. Other split line surface selections will be renumbered if necessary.



— Delete all invalid split lines. If a curve is not closed, clicking this button will delete all the open curves and the empty selections.

Using — Use this drop down list to choose the method to be used to create the split line.

Composite Curve - Create the split line using a composite curve.

You can manually create the composite curve by clicking

Surfaces - Create the split line by selecting surfaces.

The **Picking** and **Edit Split Line** options below are only available if **Surfaces** was chosen from the **Using** drop down list.

Picking — select **Hole** or **Single Face** from the drop down list.

Hole — automatically selects concave regions. You can automatically select holes in side walls by selecting **Surfaces**, then **Hole** and clicking the

Single Face — selects individual faces in the solid.

Edit Split Line — The **Undercut** and options have been grouped in this section. These options are only visible if you select the **Surfaces** option in the **Using** drop down box in the **Create/Select Split Lines** section.

Undercut — This option allows you to indicate if a hole is an undercut and controls how the split is simulated in **Mold Die Wizard Step 8**.

- If **Undercut** is selected, the surface will move outwards when simulating the slide movement **Mold Die Wizard Step 9**
- If **Undercut** is deselected, the surface does not move when the slide movement is simulated in **Mold Die Wizard Step 9**



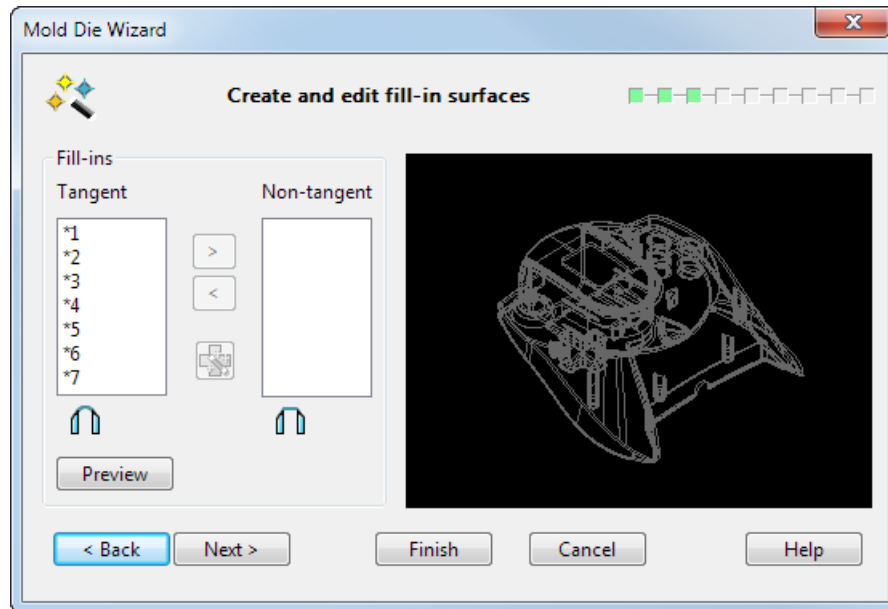
- Use this button to toggle the end of the hole that is selected.

Next — Displays Step 3 of the **Mold Die Wizard**.

Finish — If you select this option at this point and you previously selected the **Split product only** option on the previous page, the part will not be split. You will simply exit from the Mold Die Wizard. Splitting occurs after you have defined the product shrinkage.

Mold Die Wizard Step 3 - Creating and editing fill-in surfaces

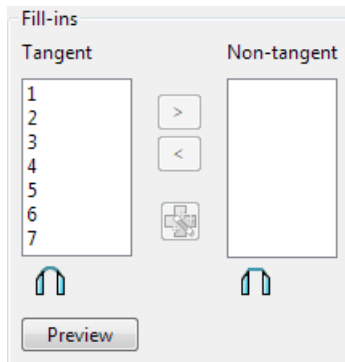
This page creates and controls the fill-in surfaces





Fill-ins — This section lets you select the fill-in lines that define the fill-in surfaces to be created and edited.







The fill-in lines on the model are numbered; these numbers correspond to the numbers that are displayed in the **Tangent** and **Non-tangent** lists




Tangent  — The fill-in lines in this list, have a fill-in surface that is tangent continuous to the surrounding surfaces.

Non-tangent  — The fill-in lines in this list, have a fill-in surface that is tangent non-continuous to the surrounding surfaces.

Fill-in lines can be moved between the two list using the  and  buttons. When a fill-in line is moved to another list it is marked with '*', which means that its current fill-in surface is not yet updated.

You can move multiple lines by using **Shift-click**  and **CTRL-click**  to select the required items.

 — Select this option to recalculate the surface tangency of the selected fill-in line

Preview — Preview the fill-in surfaces. Fill-in lines that were previously marked with * will then have the * removed.

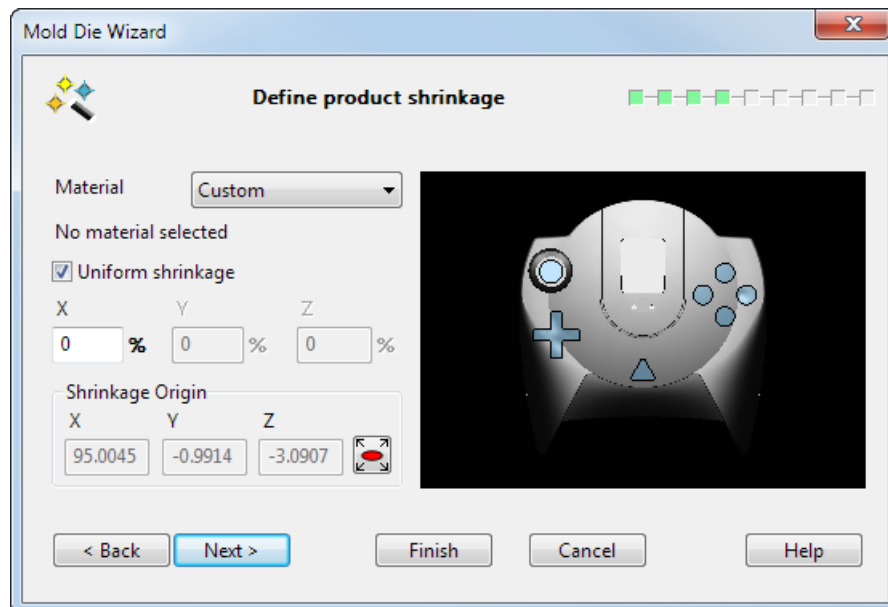
Back — Returns to the **Create and edit split lines** page of the Mold Die Wizard.

Next — The fill-in surfaces are applied and Step 4 of the wizard (**Define Product Shrinkage**) is displayed.

Finish — If you select this option at this point you will exit from the Mold Die Wizard, keeping any changes that you have made up so far.

Mold Die Wizard Step 4 - Defining Product Shrinkage


This step lets you define the product shrinkage factor and a shrinkage origin.




The shrinkage factor is the amount by which the mold cavity increases to allow for shrinkage.

- 1 Select a **Material** from the drop down list. The drop down list displays the abbreviated names of the materials; when a material is selected, the full name of the material is displayed below. A predefined value (dependent on the material selected) will be automatically inserted into X, Y and Z.
- 2 The materials are stored in a file called *material_shrinkage.csv* in ...[Delcam\powershapeXXXX\file\diewizard](#) (where XXXX is the version number of PowerSHAPE). You can edit this file as required.
- 3 Deselect **Uniform shrinkage** to set different shrinkage factors for X, Y and Z.

The current values for the X, Y and Z coordinates of the shrinkage origin are displayed in the input fields. The coordinates are given relatively the current active workplane.

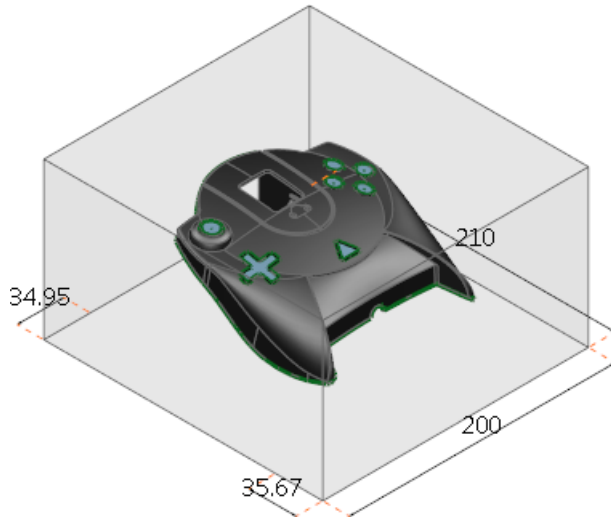
- 4 Click  to change the **Shrinkage Origin**.

Select a new shrinkage origin by clicking anywhere on the model or defining the point in the **Position**  dialog.

- 5 Click **Next** to display Step 5 of the **Mold Die Wizard**.

Mold Die Wizard Step 5 - Defining the block size

A block is automatically created around the selected object, unless you entered the Mold Die Wizard with a block already defined. The block size can be defined using either absolute or relative values. The wizard contains an image of the part as shaded and the block as wireframe. You can also rotate, zoom and pan this image using the mouse.

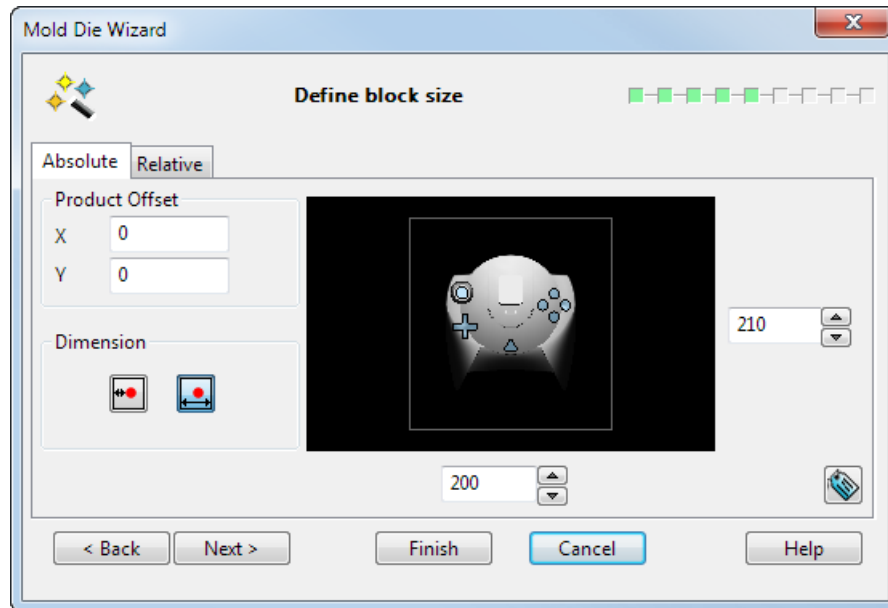


For details on how to change the view using the mouse, see Middle mouse button.

- 1 Define the block size using the absolute or relative values.
- 2 Click **Next** to display Step 6 to create the split surface. Any surfaces you provide that lie outside the die block will be removed automatically. If, at a later stage of the wizard, you return to Step 5 and change the width, length or origin, the original, untrimmed, surfaces are displayed.


Defining the block size using absolute values


The **Absolute** tab lets you define the height of the block using absolute values





Product Offset — Enter values in the X and Y text box to offset the product centre from the block centre.

Dimension

 - Select this option to set the minimum land dimensions when you enter values in the boxes. This is the minimum distance between the product and the block.

 - Select this option to set the overall width and length of the block when you enter values in the boxes.

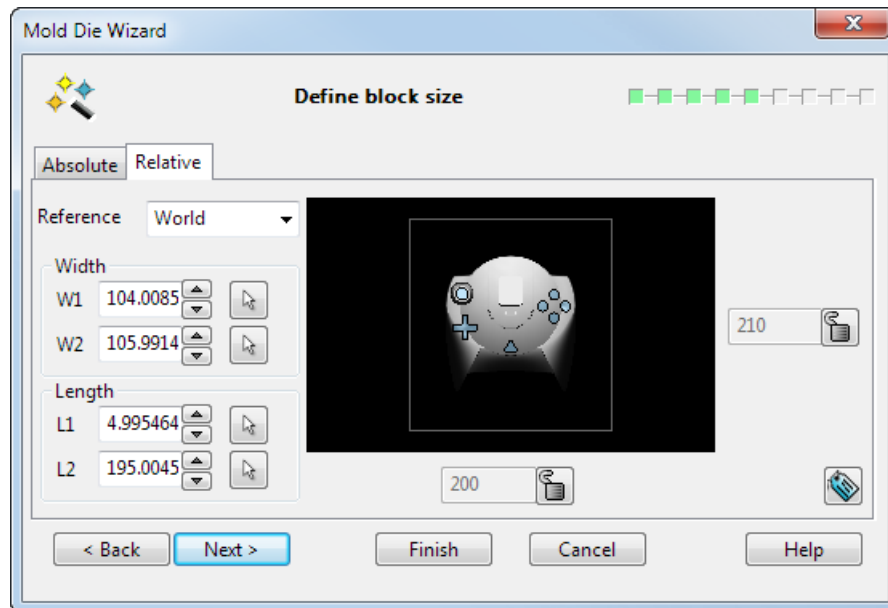
 - Use the arrows on these boxes to define the width and length of the die block. You can also type absolute values into the boxes

 — Display/hide dimension name labels on the model in the PowerSHAPE graphics window. The dynamic dimensions appear the same size on the screen, regardless of the zoom factor.

Next — displays Step 6 of the **Mold Die Wizard**.

Defining the block size using relative values

Using this page, you can specify the width and length of the block by altering the distance between a known reference point and a block edge.



Reference — The reference point is marked with a yellow star. The available reference points are:

World workplane

Active workplane

Product Centre.

Active is the default reference point (if there is one), otherwise **World** is used.

Width

W1 — The distance between the reference point and the nearest block edge.

W2 — The distance between the reference point and the furthest block edge



— **W1** and **W2** can both be set using the mouse. Click button and then use the mouse to snap to points on construction lines or other objects.



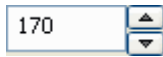
Length

L1 — The distance between the reference point and the nearest block edge.

L2 — The distance between the reference point and the furthest block edge



— **L1** and **L2** can both be set using the mouse. Click button and then use the mouse to snap to points on construction lines or other objects.



— Use the arrows on the two boxes to define the width and length of the die block.



— The block length or width is currently unlocked. Click this button to lock the block length or block width.



— The block length or width is currently locked, so the locked dimension will stay constant and you can alter the block position relative to the product. Click this button to unlock the block length or block width.

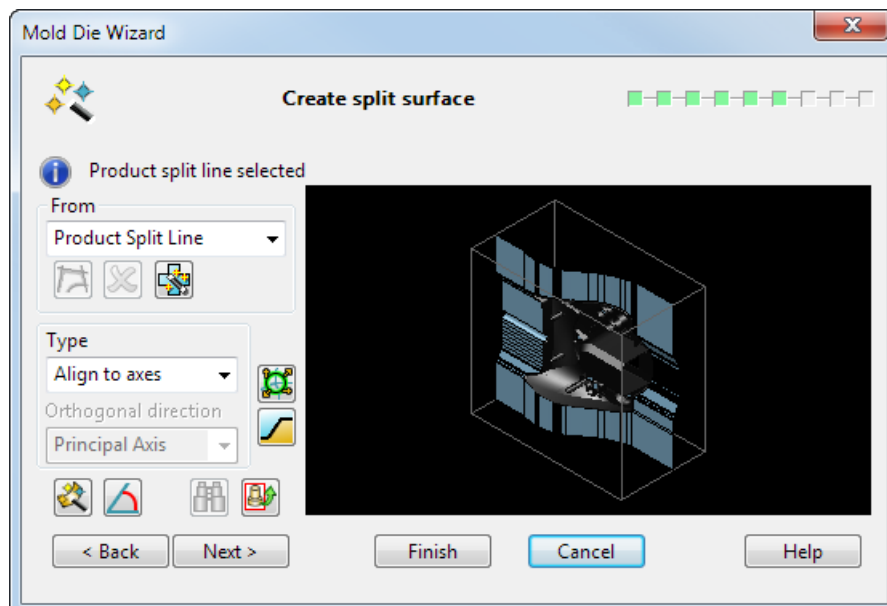


— Display/hide dimension name labels on the model in the PowerSHAPE graphics window. This replaces the **Show labels** option box from previous versions. The dynamic dimensions have been modified so that they appear the same size on the screen, regardless of the zoom factor.

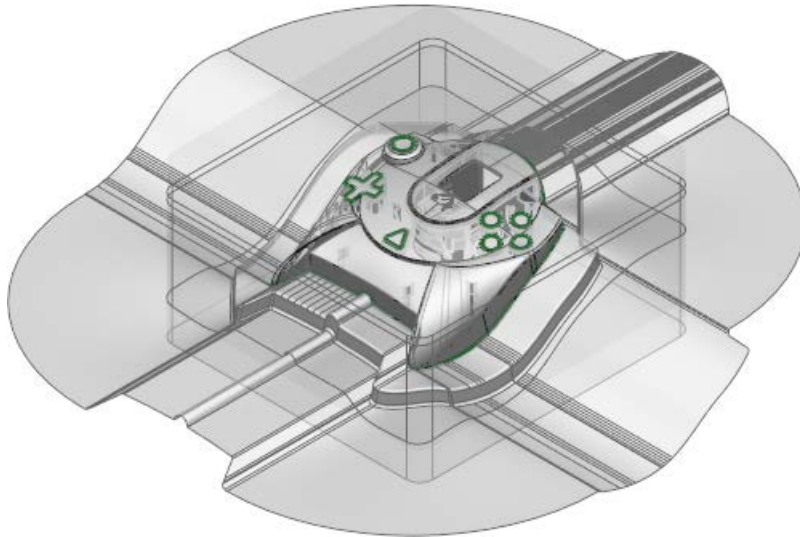
Next — displays Step 6 of the **Mold Die Wizard**.


Mold Die Wizard Step 6 - Creating the split surface


This step defines the split surface.




A split surface is created using the default settings.




Change the settings as necessary; you can use the reset  button to set the graphic in the wizard graphic window back to the original view.


 **Product split line selected** — This indicates the curve that is currently selected to be used to generate the split surface.

From


 **Product Split Line** — Use the product split line to generate the split surface. The **Alternative Compcurve** option lets you select an alternative composite curve from which to generate the split surface.

 — If **Alternative Compcurve** is selected from the drop down, this option is used to create an alternative curve to use when generating the split surface. This means that you can create a partial split surface and then use the wizard to generate the remaining portion. Selecting this option automatically hides the block and the product to make it easier to trace the curve.


The alternative composite curve should surround the partial split surface and the product.


 — The **Delete** button now deletes all composite curves created on this page. If one of the curves is currently selected, then it is also deleted and the wizard defaults to the product split line.

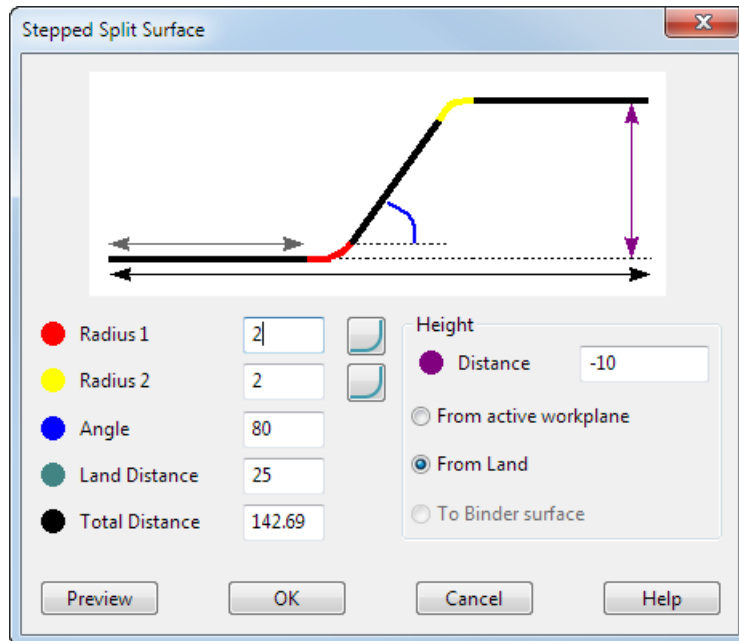
Type —

 **Align to axes** — The options on this drop down list define the split direction. For further details, see Split Surface dialog.

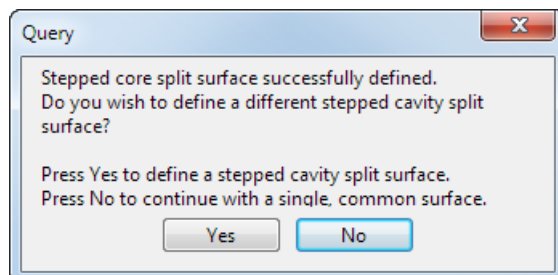
Orthogonal direction — Affects the direction of laterals of the split surface. For further details, see Split Surface dialog.

 — Advanced split surface options. The Split Segments dialog is displayed.

 — Create one or two stepped split surfaces. When you select this button, the Stepped Split Surface dialog for a core (**Core Stepped Split Surfaces**) or cavity (**Cavity Stepped Split Surface**) is displayed.



OK — A stepped split surface is created and another dialog is displayed.



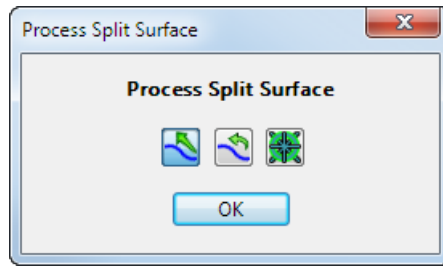
Yes — The stepped split surface form is displayed again and the user can create a second surface for the cavity insert.

No — Use the surface that has been created as the common surface.

If you create two stepped surfaces within the Mold Die Wizard and exit the wizard without splitting the mold, the two surfaces are output as solids.



— **Process Split Surface**. Click this to display the **Process Split Surface** dialog.



— **Retain tangents**. Select this to retain the tangency of the projected curve across the split surface.



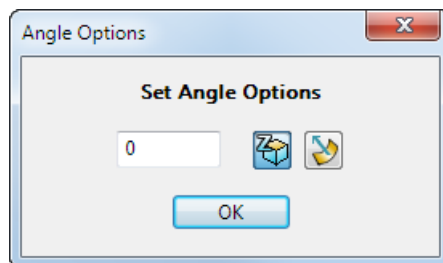
— **Smooth surface**. Attempt to smooth the split surface by adjusting the tangent direction of the projected curve.



— **Preserve symmetry**. Selecting this option preserves the symmetry of the split surface when **Align to Axes** has been selected from the **Type** drop down list.



— **Create angled split surface**. This displays the new **Angle Options** dialog.



— Enter the required taper angle in the text box.



— Create split surface in principal plane.

Note: You can access the principal plane options on the status bar of the PowerSHAPE window. You can use these options to select a different plane and then create a vertical split surface by using the **Angle Options** or the advanced split surface option.



— Create split surface normal to underlying surface.



— Generate the split surface.

Next — Displays Step 7 of the **Mold Die Wizard**.

Creating a split surface using a composite curve and the Mold Die Wizard

- 1 Create a partial split surface without using the Mold Die Wizard
- 2 Start the Mold Die Wizard, with the split surface selected.

- 3 On the **Creating split surface** page, trace a composite curve around the part and all the surfaces.

You can generate the composite curve around the part and surfaces before entering the Mold Die Wizard, but this should not be selected when you start the wizard. If it is, the wizard will try to use this as a split line and generate an alert that it doesn't completely lie on the part.

- 4 Pick the related composite curve.

- 5 Click  to generate the split surface.

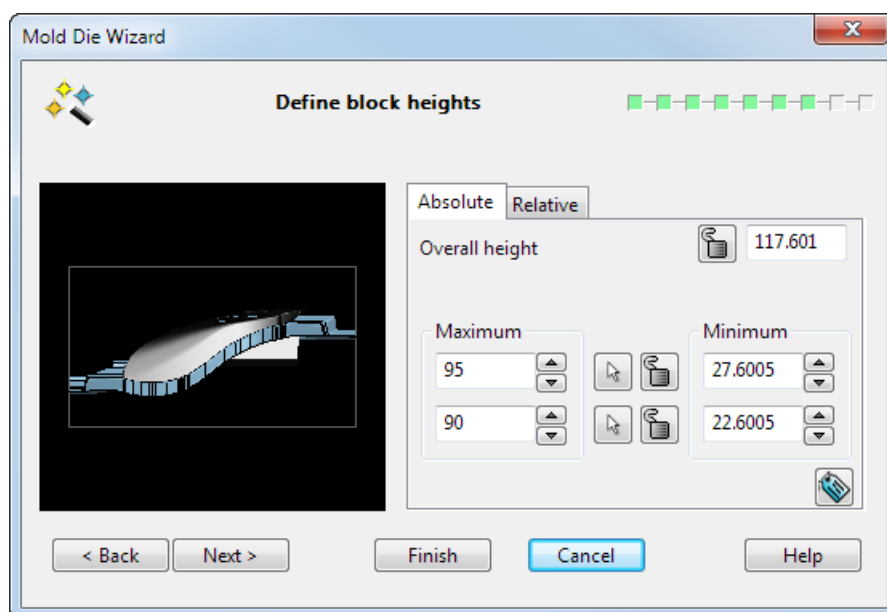
Mold Die Wizard Step 7 - Defining the block heights

This step defines the height of the two halves of the die block. Select the relevant tab on the dialog to define the height of the block using

- Absolute values (see page 40)
- Relative values (see page 42)


Defining the block height using absolute values



The **Absolute** tab lets you define the height of the die block using absolute values.



- 1 Change the two maximum values, if necessary.


The top value represents the maximum height of cavity and the bottom value the maximum height of the core. The overall height changes if you change the maximum values.

You can use  to use the mouse to set the height of the block. This can be used to snap to other objects or a key point. You can use this method to determine the height of either the cavity or core (not both). If you try to use this method to determine a height that is locked, the height will be unlocked.

You can use  and  to lock/unlock the appropriate value. At any one time, one of the following heights can be locked:

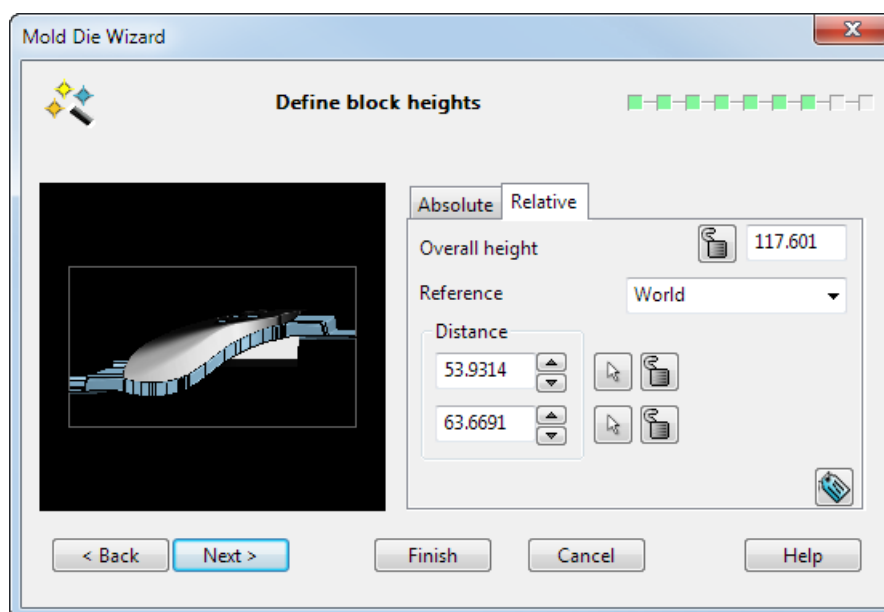
- overall height.
- height of the cavity.
- height of the core.

Locking the cavity or core heights will prevent that height from being changed.

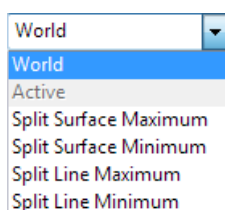
- 2 The minimum thickness of the two die inserts is automatically calculated and the values displayed in the wizard, but you can also set the height of the block using this option. The top value represents the minimum height of the cavity. The bottom value is the minimum height of the core.
- 3 You can also change the overall height.
If you change the overall height, the two maximum heights adjust proportionally. If **Overall Height** is locked, and the cavity height is adjusted, the core height will be automatically adjusted to maintain the correct **Overall Height**.
- 4 Dimension labels can be displayed by selecting the **Display/Hide labels**  button.
- 5 Click **Next** to go to **Step 8** where you can define the corner types on the block. If you select **Finish** at this point, defaults will be used where values are unspecified. The die block will not split into core and cavity parts.

Defining the block height using relative values


The **Relative** tab lets you define the height of the die block using a distance from a known reference point.





- 1 Change the **Overall height** of the die block if necessary.
- 2 Select the reference point from the drop down list. The height of the block is set using a distance from a known reference point. If an **Active** workplane exists, this is the default option; otherwise the **World** workplane option is displayed.



- 3 Change the two **Distance** values, if necessary. The two values displayed represent the distance between
 - the chosen reference point and the top of the block.
 - the chosen reference point and the bottom of the block .
 - The height of the block can be adjusted by changing these values.

You can use  to use the mouse to set the height of the block. This can be used to snap to other objects or a key point. You can use this method to determine the height of either the cavity or core (not both). If you try to use this method to determine a height that is locked, the height will be unlocked.

You can use  and  to lock/unlock the appropriate value. At any one time, one of the following heights can be locked:

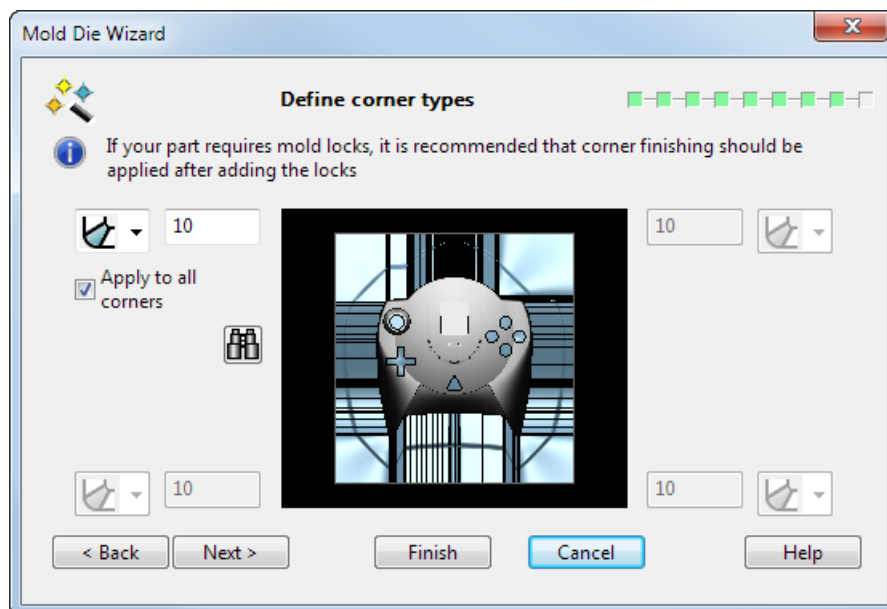
- overall height.
- distance between the reference point and the top of the block.
- distance between the reference point and the bottom of the block.

Locking the cavity or core heights will prevent that height from being changed. If **Overall Height** is locked, and the top height is adjusted, the bottom height will be automatically adjusted to maintain the correct **Overall Height**.

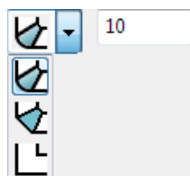
- 4 Dimension labels can be displayed by selecting the **Show Labels** option.
- 5 Click **Next** to go to **Step 8**, where you can define the corner types on the block. If you select **Finish** at this point, defaults will be used where values are unspecified. The die block will not split into core and cavity parts.

Mold Die Wizard Step 8 - Defining the corner types

This dialog lets you define the type of corner for each corner of the die block.



- 1 Use the drop down list to select the type of corner you want to add to the die inserts and enter the value for the corner finish.

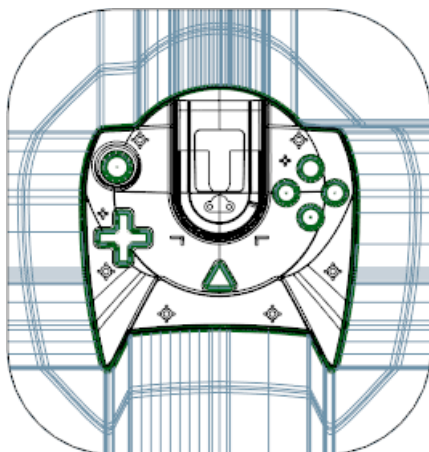


The default value for the value is 10mm, but if the block is so small that this may cause problems, a smaller default value may be used.

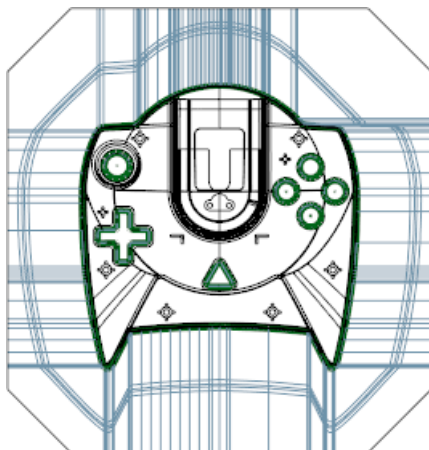
- 2 Select **Apply to all corners** if you want all four corners to be the type and size selected. If this option is *OFF*, the other three sets of options become active and the user can then choose a different corner finishing for each corner.



- Adds fillets. Input the corner radius value.



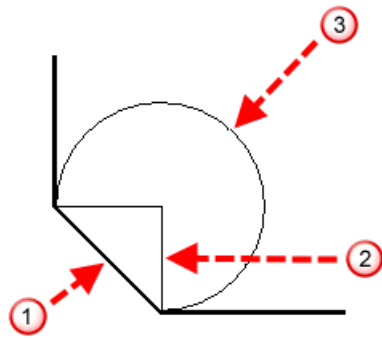
- Adds chamfers. Input the distance value.



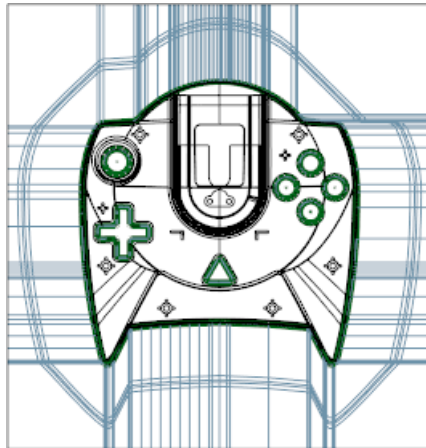
The distance used to create the chamfer is determined as follows:

- ① Chamfer
- ② Distance


- ③ Arc with radius equal to distance

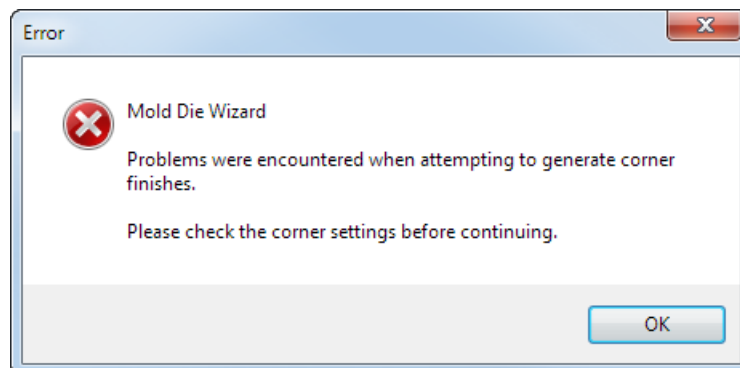


- Adds no corner finish.



The **Corner Type** options are not available when creating non-inserted plates.

- 3 Use **Preview**  to view the block with the selected corner finishes applied. If the settings cause a problem (for example, the product sticks out of the block), the **Error** dialog will be displayed. Click **OK** and adjust the settings to correct the problem.



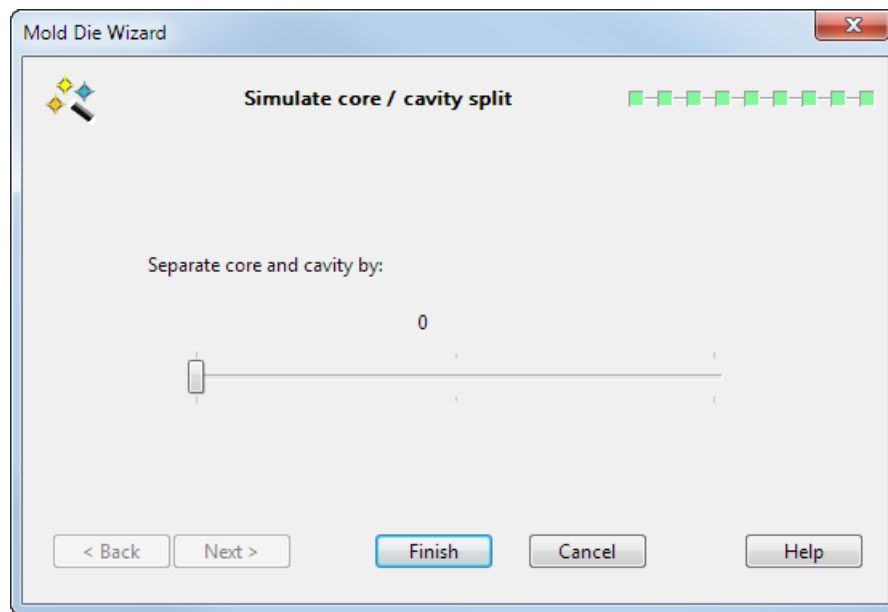
- 4 Click **Next** to display **Step 9** of the Mold Die Wizard, or **Finish** to complete the wizard operation using default values where necessary.



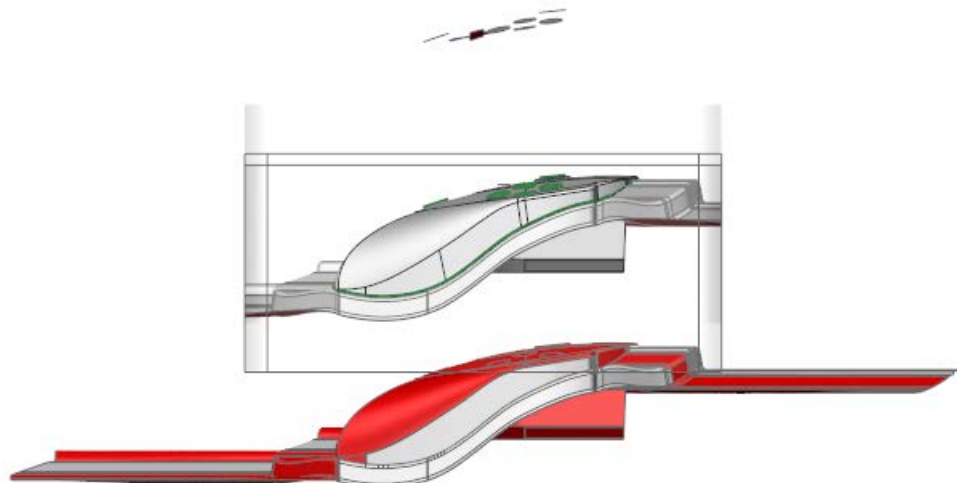
When you select **Next** or **Finish**, the wizard will only proceed if the selected corner settings do not cause a problem. If the settings cause a problem (for example, the product sticks out of the block), the wizard will display the **Error** dialog and remain on the current page so that you can see the problem and adjust the corner type or value accordingly.

Mold Die Wizard Step 9 - Simulating the core/cavity split

This page of the Mold Die Wizard shows the model in the current shading mode, rather than in wireframe mode.



- 1 Use the slider to split the die from the selected object. You can see the two halves moving in the graphics window. This is only a visual aid to examine the interior of the die.

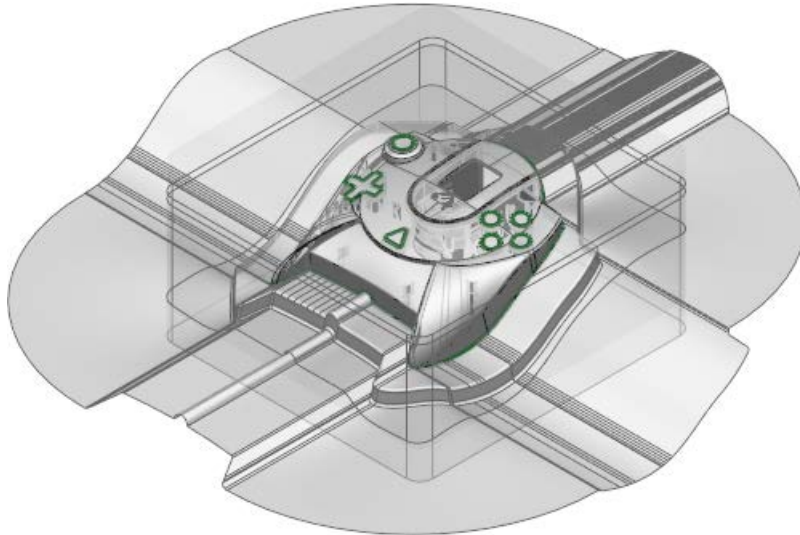



If you are only splitting the part, you will have two solids - one for the core and the other for the cavity.

2 Click **Finish** to save the die inserts. Inserts produced by Mold Die Wizard use the following materials:

- Surfaces in shut out regions use *PasteI10*.
Surfaces in the impression region use *PasteI08*.
All other surfaces in the die block use *mold_metal03*.

The original solid is encased in the two halves of the die.



A table showing the sets of Cavity/Core/Inserts is displayed in the **Creating Inserts** section of the browser window. You can check the items by clicking on each  icon in turn.

	Cavity	Core	Inserts
1			

3 You are now ready to create the cavity and core plates. Click **Next** in the browser window.

Mold Base

Catalogue<Choose a catalogue>

Type

Guide pillars:☒ Fixed half ☐ Moving half

Show the mold base

Fixed half

Moving half

Cavity plate

Type

Material

Comments...

Core plate

Type

Material

Comments...

Levels...

< Back

Next >

Cancel

Help

This displays the page from which you select the core and cavity plates from the catalogue. For further details, see Defining the plates (see page 22).

Defining the inserts/pockets

Use this page to define the inserts or pockets in the cavity and core plates.

Dimensions

Minimum pocket clearances

In X, MX In Y, MY In Z, MZ

[Show the clearances diagram](#)

Mold base | Inserts and pockets | Slides |

Dimensions ☒ Standard ☐ Custom

Ejection system type

☒ Adjust plate sizes automatically when changing parameters of the inserts

☒ Fixed half ☒ Moving half ☐ Ejection system

Rotate the mold base: ☐ ☐ ☐ ☐

Cavity plate

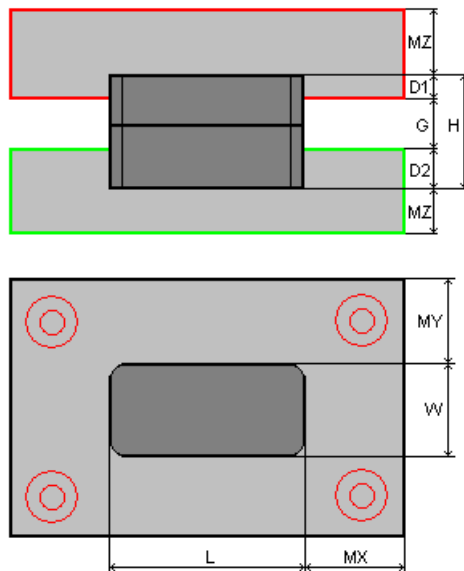
Thickness Pocket Z clearance
Material
Dimensions: Standard, mms
Product code: K20/156x156x66/1.2162

Core plate

Thickness Pocket Z clearance
Material
Dimensions: Standard, mms
Product code: K20/156x156x76/1.2162

Minimum pocket clearances — Defines the minimum clearance around the pockets.

Show the clearances diagram — This expands the window to display a diagram which shows how the dimensions of the inserts or pockets are measured.



You can hide it again by clicking the **Hide the clearances diagram** option.

The following tabs let you specify more detail about the inserts/pockets:

- Mold base (see page 51)

- Inserts and pockets (see page 52)

- Slides (see page 53)

Next — This displays the page to confirm your parameters.

Mold base

This section defines the mold base.

Mold base | Inserts and pockets | Slides

Dimensions ☒ Standard ☐ Custom
156x156

Ejection system type Standard (1)

☒ Adjust plate sizes automatically when changing parameters of the inserts

☒ Fixed half ☒ Moving half ☐ Ejection system

Rotate the mold base: ☒ ☐ ☐ ☐

Cavity plate

Thickness 66 Pocket Z clearance 11

Material 2162

Dimensions: Standard, mms
Product code: K20/156x156x66/1.2162 Dimensions...

Core plate

Thickness 76 Pocket Z clearance 16

Material 2162

Dimensions: Standard, mms
Product code : K20/156x156x76/1.2162 Dimensions...

Dimensions — The wizard has calculated the minimum size of the mold base to match the sizes of the inserts defined on the page. Check if the size is suitable and select a different size from the menu if required.

If you want to create a custom mold base, select the **Custom** option. Enter your dimensions and click **Apply**.

Dimensions ☐ Standard ☒ Custom

156 156

Apply

Ejector system type — Select the type of ejector system.

Adjust plate sizes automatically when changing parameters of the inserts — If on, the plates sizes will automatically change when you change the sizes of the inserts.

The fixed half is red, the moving half is green and the ejector system is white (using the default colour scheme).

Rotate the mold base — You can rotate the mold to be created by 0°, 90°, 180°, or 270°.

The default rotation is 0°. This conforms to the mold industry standard, that is, the width of the mold base is along the X axis and the length is along the Y axis.

Cavity/Core plate — Select the **Thickness** and **Material** of cavity/core plate. The Z clearance of the pocket is given on the right.

If you are using a standard mold base, values available from the catalogue can be selected from the drop down lists.

For a custom mold base, you can type in the values

Dimensions — This tells you if the selected plate is standard or custom and its units. Click this button to change the dimensions of the selected plate. This displays a dialog that is similar to the page for defining the dimensions for components. For further details see Defining a component using the Component Wizard dialog.

Inserts and pockets

This defines the position of the inserts or the pockets.

- 1 If you use the **Die Wizard** to create the inserts, the correct dimensions are automatically entered to match the inserts. If you are creating an empty pocket, you need to enter the attributes of the pocket.

Offsets of the pocket - Offsets the position of the pocket within the core and cavity plates.

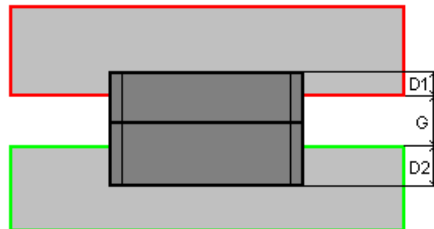
Through pockets - If selected, the pocket cuts all the way through the core and cavity plates.

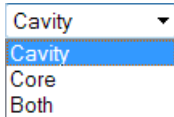
Define the vertical position of the inserts by - These are the vertical positions of the inserts using any of the two dimensions:

D1 and **D2**

D1 and **G**

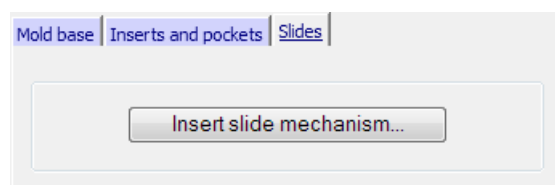
D2 and **G**



- 2 Click **Update the list** to update the list of inserts.
- 3 Select inserts from the list. Select multiple inserts using the standard **Shift + click** and **Ctrl + click** techniques to select items from the list.
- 4 Use the following options to define the pocket:
 - a **Define a pocket** - This option is selected by default.
 - b **Combined pocket** - If selected, this creates a combined pocket for all the inserts. If deselected, individual pockets are created for the selected insert.
 - c Use the drop-down list to choose whether to apply the pockets to the **Cavity**, **Core** or **Both**.

 - d Click **Pocket** to displays the Pocket dialog. You can use this to define the **Draft**, **Corners** and **Fillets** on the selected inserts

Slides

This allows you to create a slide mechanism in your mold.



Insert slide mechanism — Click this option to start the **Slide** wizard. For further details, Creating the slide assembly using the Slide wizard (see page 82).



Confirming your values

Use this page to check your parameters and then create the cavity and core plates.

Cavity-Core Wizard

You have successfully completed the Cavity/Core wizard

You specified the following parameters:

Insert parameters	
Width	60
Length	60
Thickness	120
Corner radius	8
Insert placement	
Offset in X	0
Offset in Y	0
Gap between the cavity plates	5
Depth of the pocket in the bottom cavity plate	60
Mold base	
Catalogue	HASCO
Type	Standard mold
Guide pillars	Fixed half
Rotation angle	0
Width	156
Length	156
Dimensions	Custom
 Cavity plate	
Type	K20
Thickness	66
Material	2162
Dimensions	Custom
 Core plate	
Type	K20
Thickness	76
Material	2162
Dimensions	Custom

Press the "Finish" button to create the cavity and core plates.

☒ Run the Mold Base Wizard on exit

< Back

Finish

Cancel

Help

Run the Mold Base Wizard on exit — This automatically runs the **Mold Base Wizard** after the cavity and core plates are created.

Back — If you want to change the values, click **Back** to go back to previous pages.

Finish — Creates the cavity and core plates. The plates are created in an assembly. For further details, see *What is an assembly?*

If the **Run the Mold Base Wizard on exit** is selected, the **Mold Base Wizard** is run after the cavity and core plates are created. For further details, see *Using the Mold Base Wizard* (see page 101).

Re-running the Cavity-Core Wizard

You can re-run the Cavity-Core Wizard. The parameters of any previously created plates are used. You can change the parameters of the cavity plates, core plates and the layout and number of inserts. On completing the wizard, all relevant components are redefined and any pockets updated.

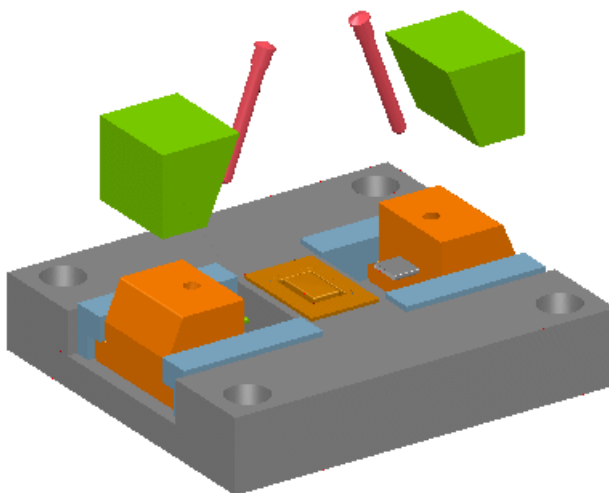
Slides

Adding a slide to a mold

You can create cores and pullers to fit undercut regions. The cores and pullers can move within the mold using slides, which are mechanical components.

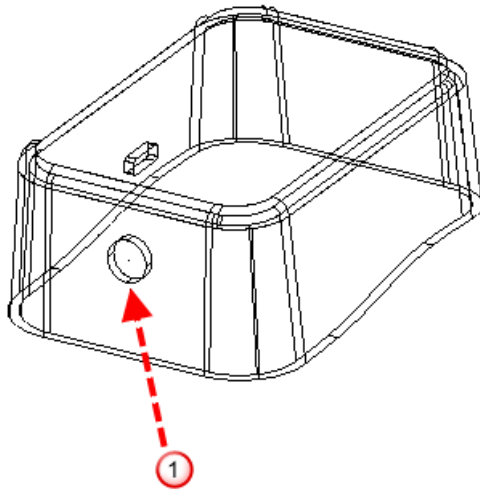
Slides are created using the **Slide** wizard.

Below, you can see the components in a slide mechanism.



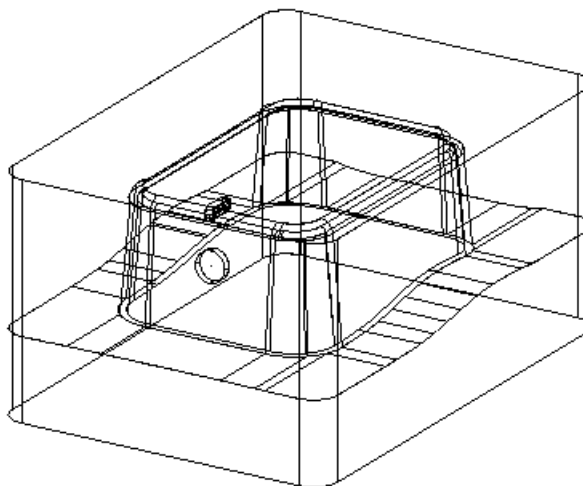
Creating a core using the Slide Core wizard

The following example will be used to show you how to use the **Slide Core** wizard. ① indicates the undercut region



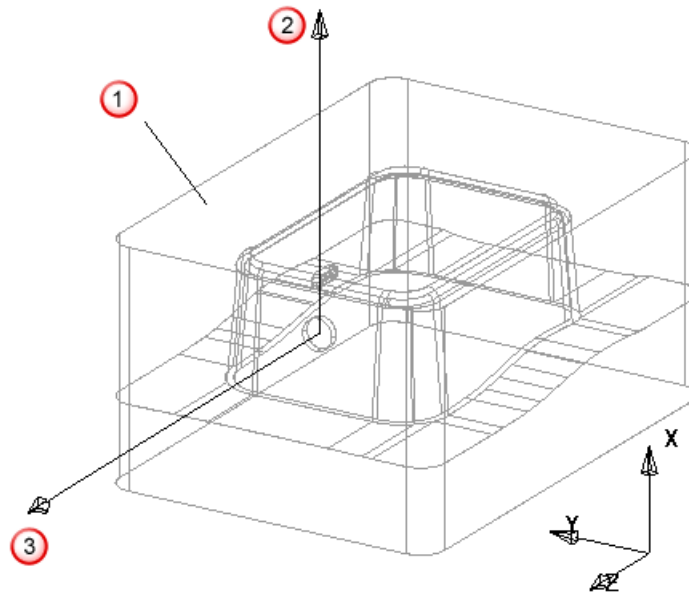
Before using the Slide Core wizard

- 1 Make sure that you are running Delcam Toolmaker.
- 2 If necessary, create the die inserts using the **Mold Die Wizard**.

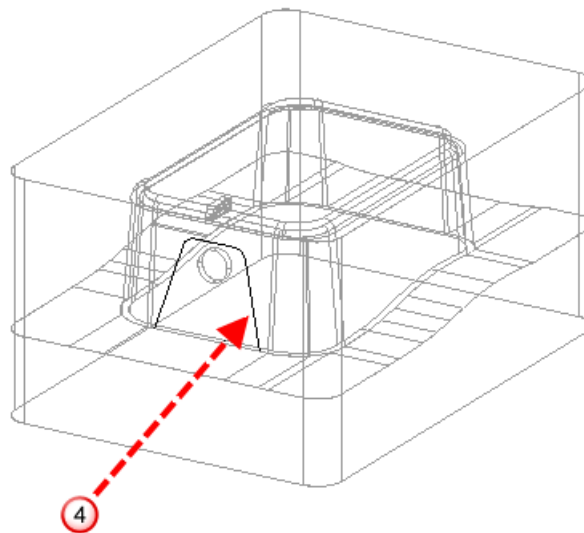


- 3 Make sure the die inserts are watertight. To check if the die inserts are watertight, right click the insert and select **Make watertight** from the pop up menu.
- 4 Orient the workplane by pointing the X axis along the draw direction so it points towards the cavity and away from the core and the Z axis points in the slide extraction direction.
 - ① Cavity is higher up the X axis than the core
 - ② Draw direction

③ Direction of slide extraction



- 5 Create the contact profile (not needed for a core puller). This is an open composite curve, whose ends touch the split line of the die inserts. It lies on the solid that contains the undercut area.



The contact profile ④ forms the outer walls of the core.

The composite curve doesn't need to lie on the die insert. You can create a 2D composite curve on the XY plane. The wizard will project it onto the die insert along the Z axis in order to create the contact profile.

The **Slide Core wizard** can also automatically generate a contact profile.

An easy way to generate a contact profile is:

- a) Create a 2D composite curve of the outline of the core.

- b) Position the composite curve so that it is close to the undercut region.
- c) Create an extruded solid from the composite curve so it intersects with the appropriate die insert.
- d) Create a composite curve from the intersection of the extruded solid and the die insert.
- e) Limit composite back to the split line.

6 You are now ready to use the **Slide core wizard**.

Running the Slide Core wizard

1 If necessary, create the die inserts using the **Mold Die Wizard**.

2 Click  and select  from the Toolmaker toolbar.

You can also start the **Slide Wizard** from within the **Cavity-Core Wizard** (see page 15). This is the recommended way to start the wizard, because you can interactively design slides and cavity and core plates together.

If you have multiple cavities and are running the Slide Wizard from within the Cavity Core Wizard, you work on only one core. The cores for the other cavities are created from this one.

Slide Wizard (1)

☒ Create a slide core...

☐ Register Workplanes and optional pre_modelled slide cores in order to be used to design slide mechanisms...

☐ Design slide mechanisms...

Slide Core Wizard

- Before starting the Slide Core Wizard
Activate a workplane where :
Z is the slide movement axis.
X is toward the top of the slide.

Start Slide Core Wizard...


Slide Core Component Creation

- Register a slide core solid as a component.


No Slide Core found ▼ Register

This functionality allows the creation of a slide core component without creating a full slide assembly. It allows the slide core component to be attached manually to any slide or another component created in the assembly.

< Back Next > Cancel Help

- 3 Ensure that the **Create a slide core** option is selected.
- 4 Click the **Start Slide Core Wizard** button to display the **Select a Core Type** page of the wizard. You can also start the Slide Core Wizard by selecting **Create slide core** .



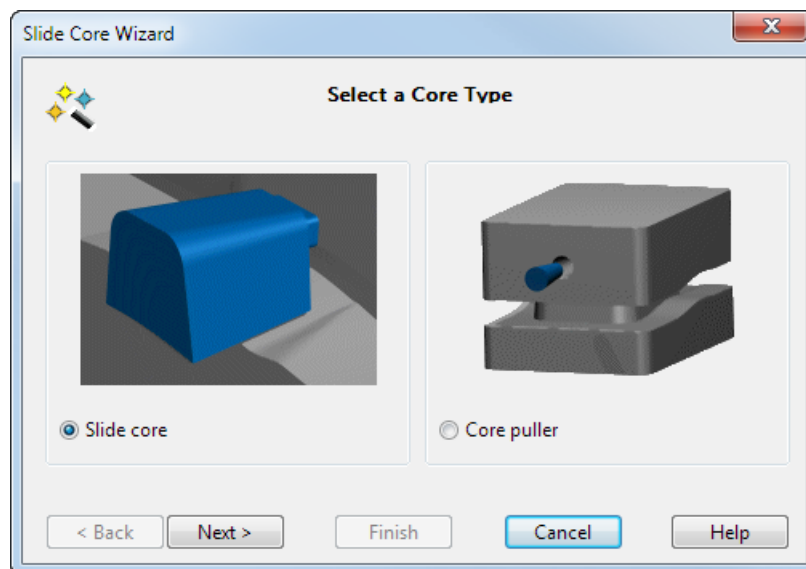
The workplane  button on the toolbar remains active while you are using the following pages of the Slide Core Wizard:

Slide Core Type
Puller Insert selection
Slide Insert selection
Split Surface selection
Undercut Surface selection

This means that you can create, edit, and activate workplanes while the Slide Core Wizard is running

Select a Core Type page

This selects the type of core to create.



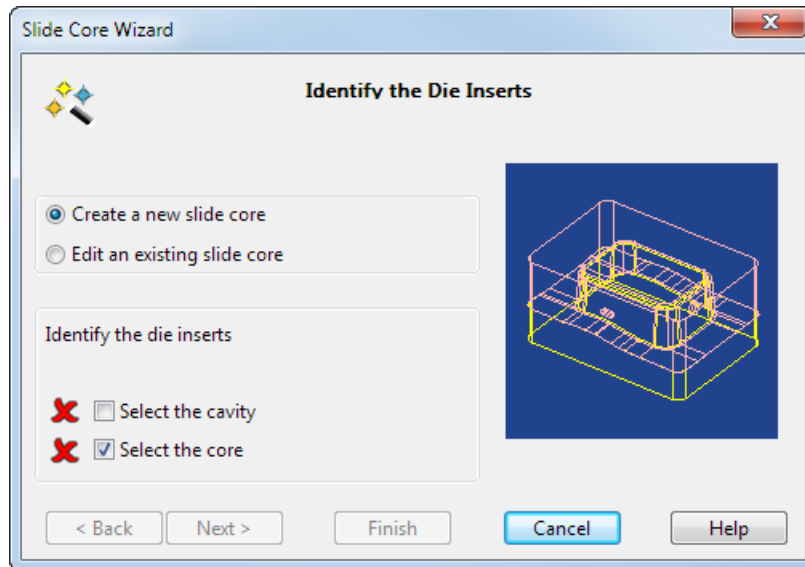
Select one of the options and then click **Next**.

If you select **Slide core**, you will go to the Identify the Die Inserts page (see page 61).

If you select **Core puller**, you will go to a different page, the Identify the Die Insert page (see page 78) for the core puller.

Identify the Die Inserts page

This selects the die inserts for the core and cavity plates.



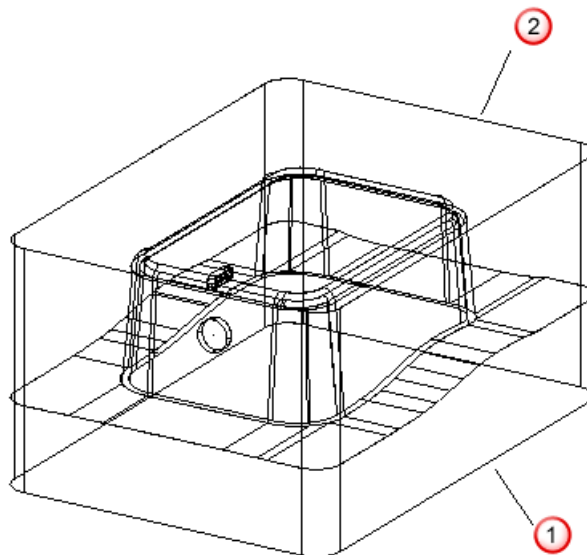
This page of the wizard is skipped in the following cases:

- The **Identify the Die Inserts** page is omitted from the wizard if the two inserts can be selected and there are no slide core features in the model.
- You have solids or components called DW_CORE and DW_CAVITY in your model. These solids are automatically selected, providing they are not blanked or on a level that is turned off. (Solids with these names are automatically produced by the Mold Die Wizard when run through Delcam Toolmaker.)

Creating a new slide core

- 1 To select a die insert for the core plate, click **Select the Core**.
- 2 Select the core on the solid. ①
- 3 Click Select the Cavity

- 4 Select the cavity on the solid ②.



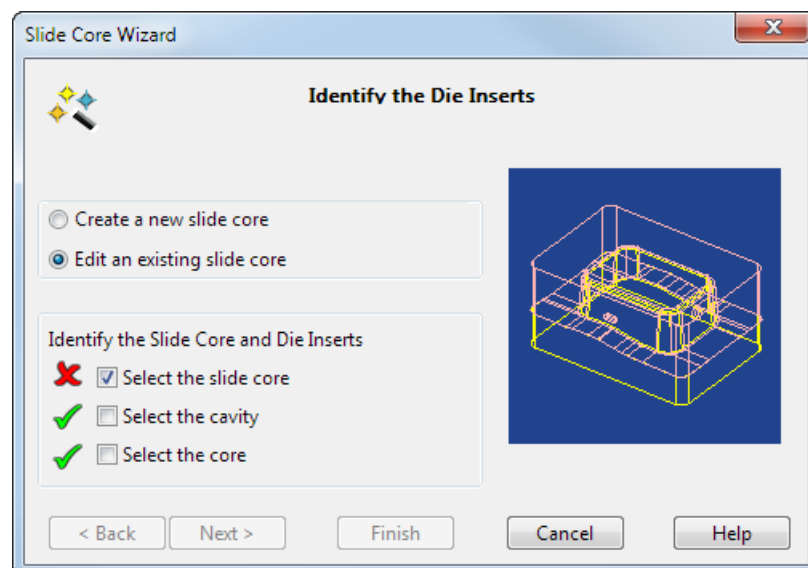
You can select any two solids or components, provided they are connected by a common face.

If you select non-single instance components, the Multi-instance component selected dialog (see page 63) is displayed.

- 5 Click **Next** to go to the Split Surface selection page (see page 64).

Editing an existing slide core

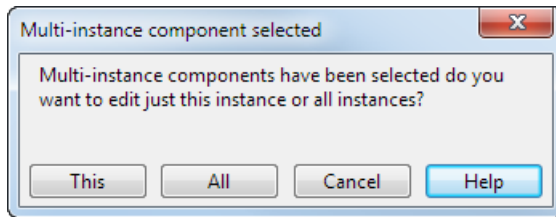
- 1 Select **Edit an existing Slide Core**. An additional option will appear on the dialog.



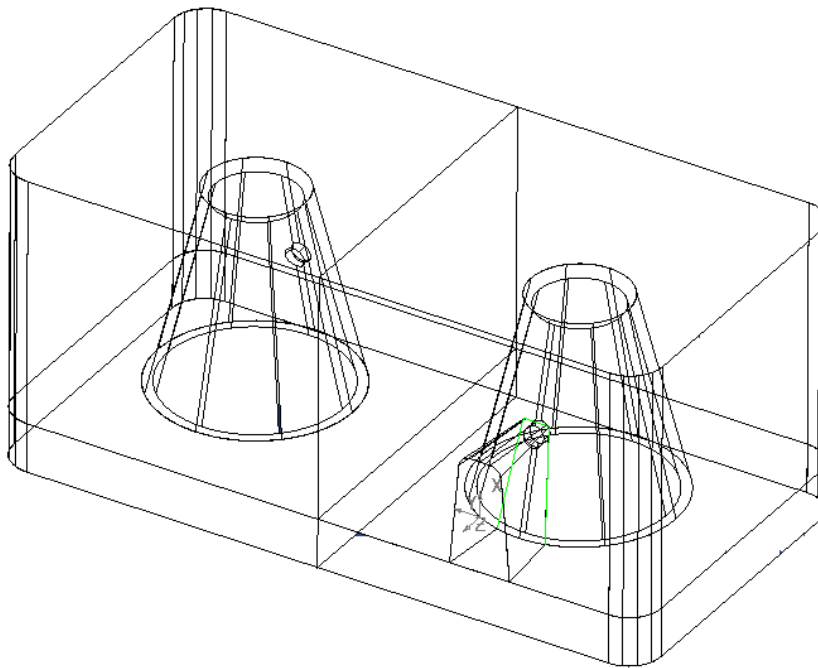
- 2 Choose **Select the Slide Core**. This option becomes available if you have selected Edit an existing Slide Core
- 3 Select a solid or component that is the existing slide core. When you select a slide core for editing, the **X** is changed to **✓**.

- 4 Click **Next** to go to the Split Surface selection page (see page 64).

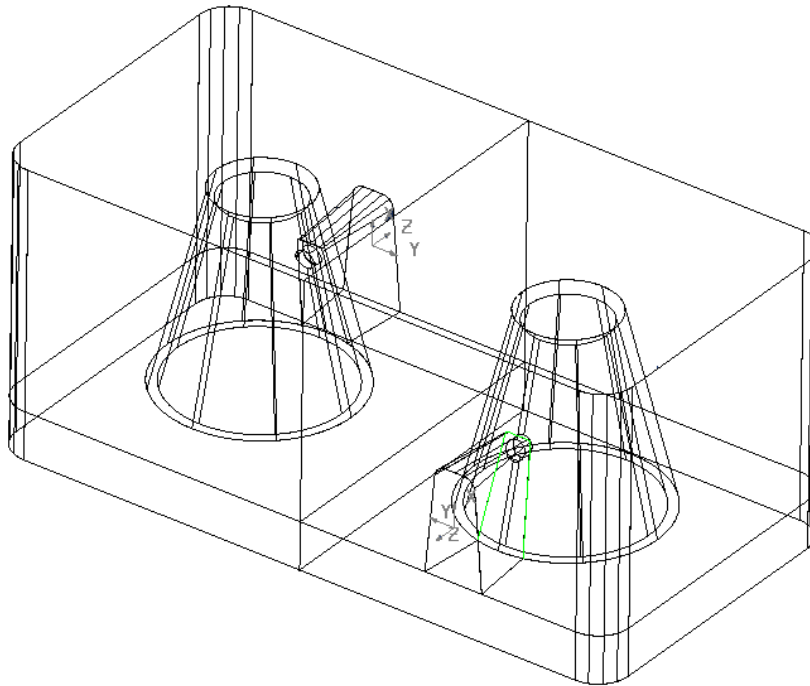
Multi-instance component selected dialog



This — The instance you have selected is turned into a single-instanced components. You can now edit the instance you have selected.



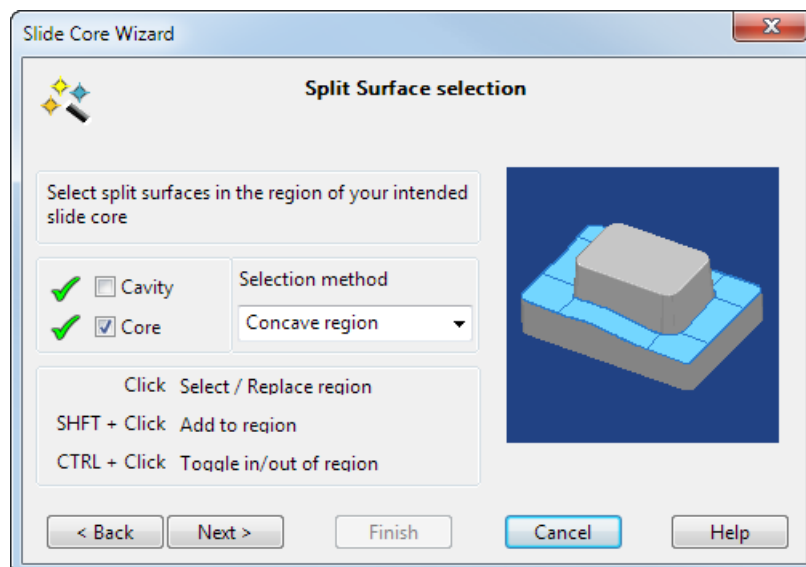
All — A workplane and slide core is created for all the instances of the insert.



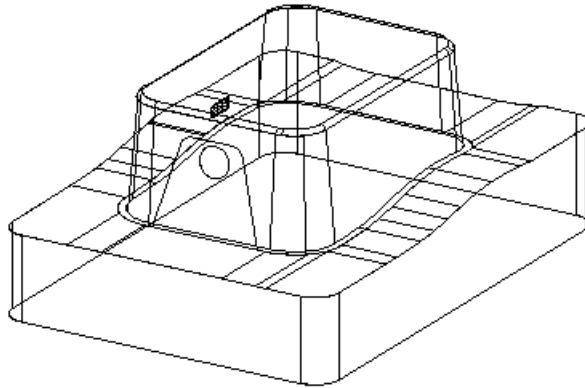
Cancel — Return to **Identify Die Inserts** page of Slide Core wizard.

Split Surface selection page

This selects the split surface on the die insert for the core and cavity plates.

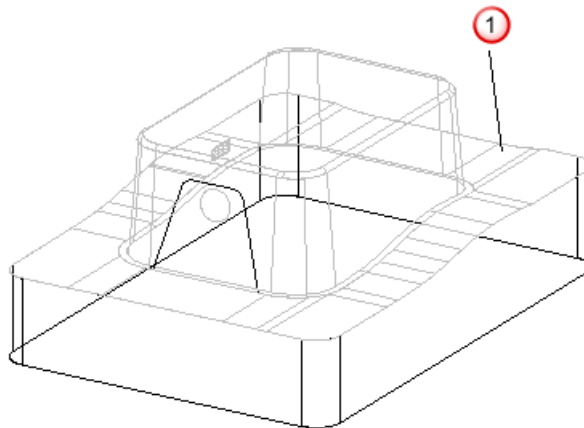


- 1 Select **Core**. The die insert for that plate is displayed on the screen and the other one is temporarily removed from the screen.



- 2 Select the split surfaces on the core plate.

As you move the cursor over the die insert, regions are highlighted ①. These are the regions where you can select the split surface.

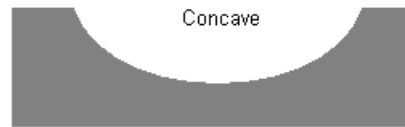


If you click a highlighted region, it is selected and changes colour. Any regions already selected are removed from the selection.

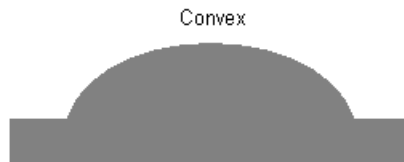
- It is easier to see selected regions in shaded mode.
- You can add and remove regions from the current selection using the **Shift** and **Control** keys as shown on the wizard.
- You can use box selection to select regions. Only whole faces within the box are selected, which makes it easier to select groups of faces. You can also use box selection with the **Shift** and **Control** keys to add and remove regions.

The four **Selection method** options determine what type of region is highlighted:

Concave region - Highlights concave regions on the die insert.



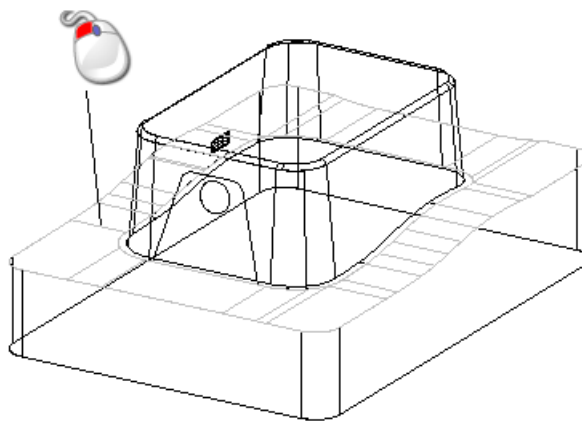
Convex region - Highlight convex regions on the die insert.



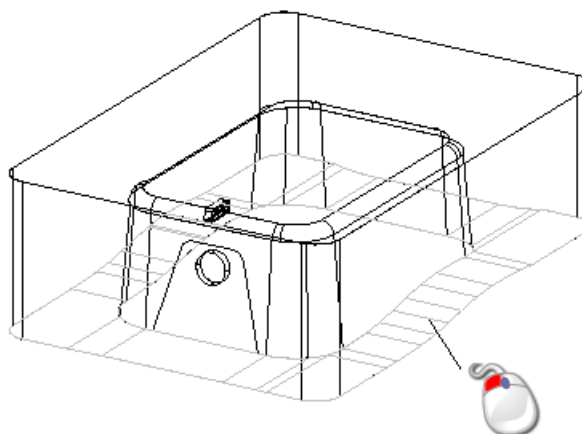
Continuous region - This selects a region of surfaces that are tangent continuous.

Single Face - Highlights only single faces.

- 3 Select the following face using the **Single face** option.



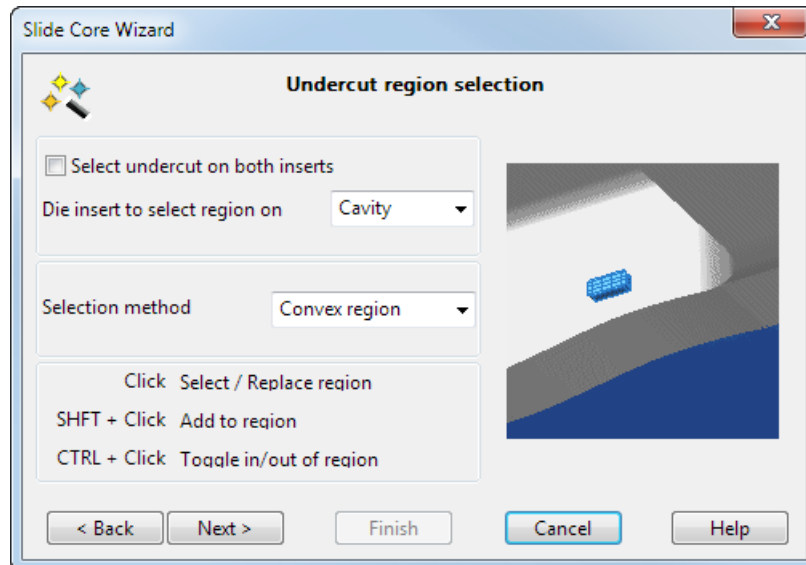
- 4 Select the **Cavity** and select the split surfaces on that die insert. You can now pick part of the split surfaces, rather than selecting all the split surfaces.



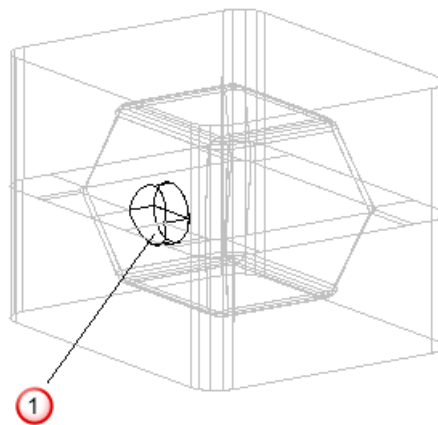
- 5 Click **Next** to go to the Undercut region selection page (see page 67).

Undercut region selection page

This selects the undercut region on the die inserts.



- 1 If the undercut region spans the split line as shown in ①, click **Select undercut on both inserts**.

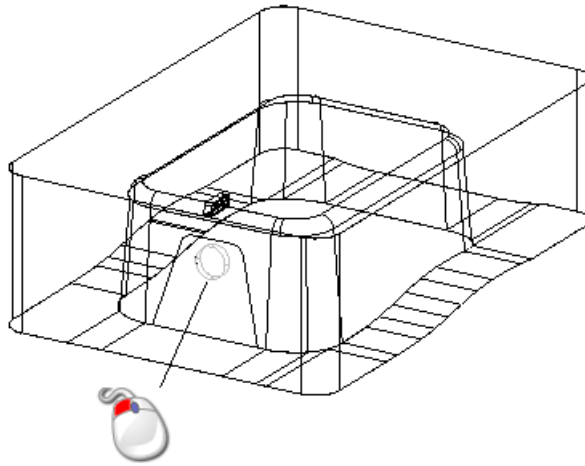


- 2 Use **Die insert to select region on** to select the die insert on which the undercut lies.

If the undercut spans the split line, you need to select the undercut region in both the core and cavity die inserts.

If you have an existing contact profile, you don't need to select an undercut region. The undercut region is automatically selected using the contact profile. However, if you select an undercut, you will know that the correct undercut is covered by the contact profile.

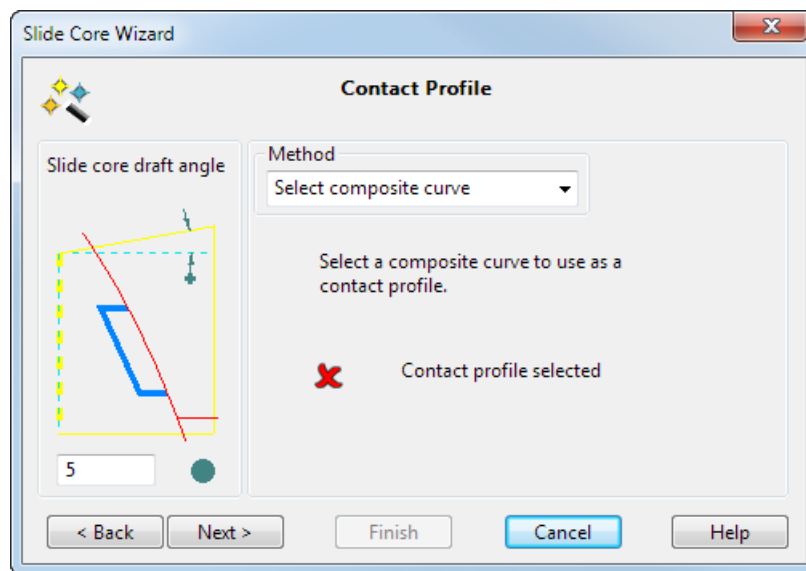
- 3 Select the undercut region on the die insert.



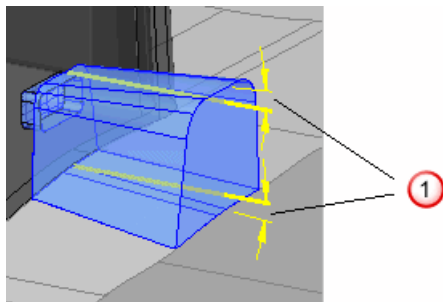
- 4 Click **Next** to go to the Contact Profile page (see page 68).

Contact Profile page

This is used to select or automatically generate the contact profile.



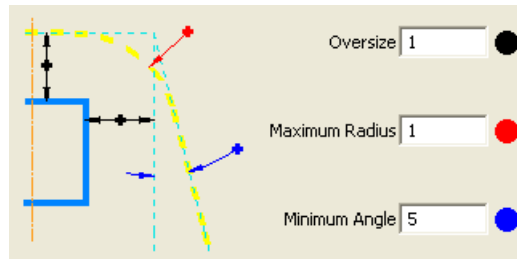
Slide core draft angle — Enter the draft angle ①:



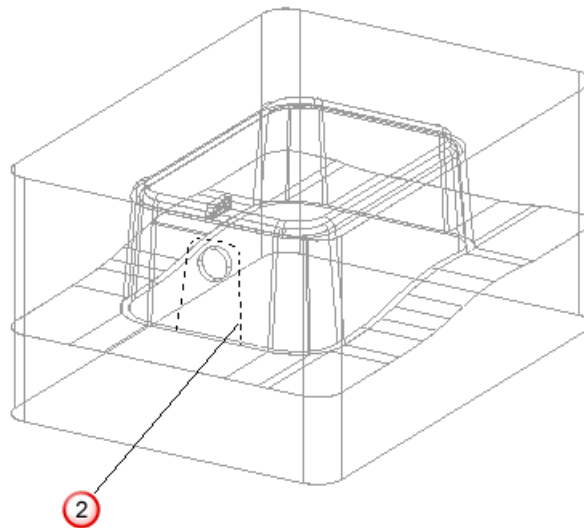
Method — When you select a method, the options change on the wizard page.

Auto-generate — This method is available only if an undercut region is selected.

When it is selected, the following options are available on the wizard.



A rough contact profile is drawn in dotted lines as a visual aid. This gives you an idea of what the actual contact profile will look like using the values on the wizard.



As you change the values, the rough contact profile ② changes size.

Oversize — Distance from the undercut region.

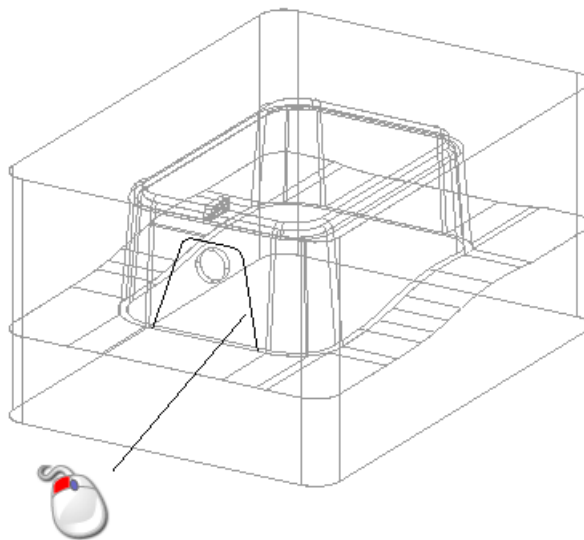
Maximum radius — Maximum fillet radius at the corners of the contact profile.

Minimum angle — Minimum angle at the sides of the contact profile.

A contact profile is automatically created from the values entered in the wizard.

Preview — Click this to see what the contact profile actually looks like in the model.

Select CompCurve — When this method is selected, you can select an existing composite curve.

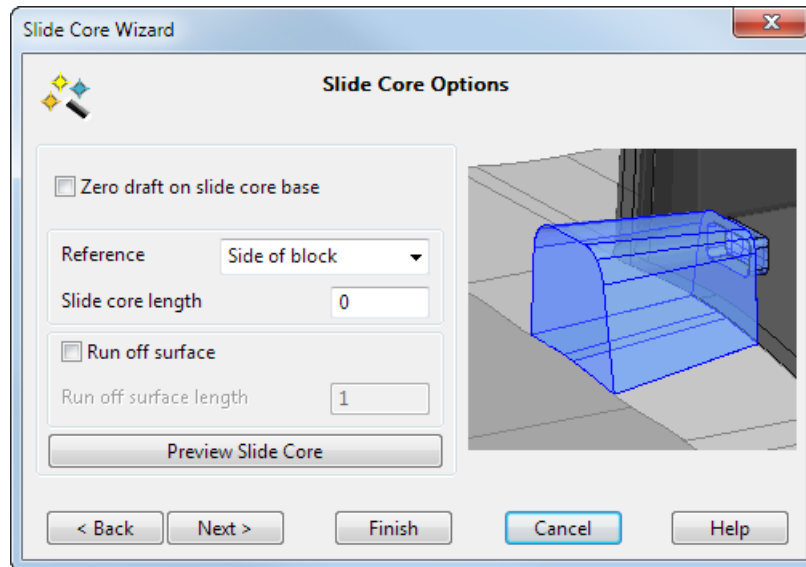


The composite curve does not need to lie on the die insert. The wizard projects it onto the die insert along the Z axis in order to create the contact profile.

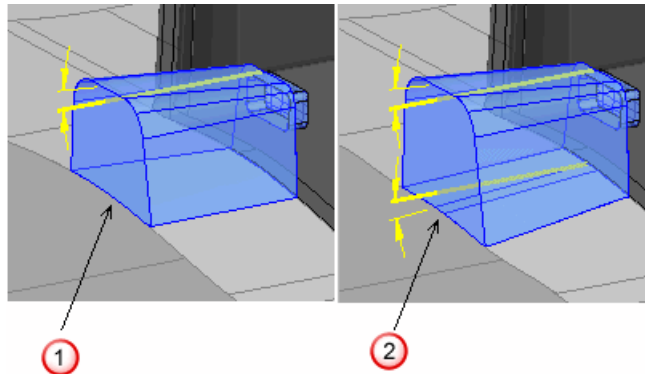
Next — Click the next button to go to the Slide Core Options page (see page 71).

Slide Core Options page

This sets the options for the slide core.



- 1 If you want a zero draft angle on the slide core base, select the **Zero draft on slide core base** option. The base of the solid core is trimmed to the split surface.

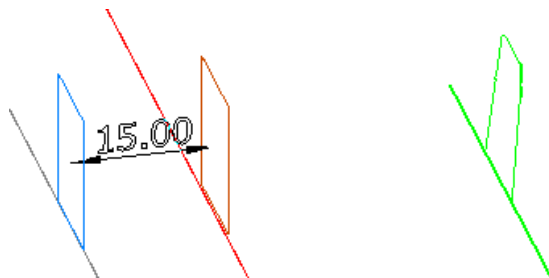


① — No draft angle at the base

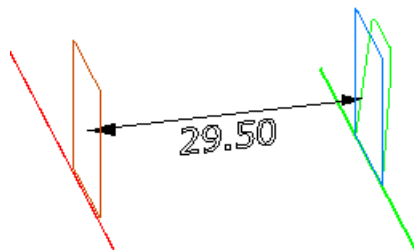
② — Draft angle at the base

- 2 Select the **Reference** point for measuring the slide core length from the following options.

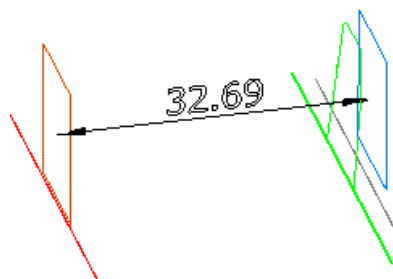
Side of block - Side of Cavity/Core insert.



Contact profile - Maximum Z of the contact profile.



Slide core front - Minimum Z of the undercut surfaces.



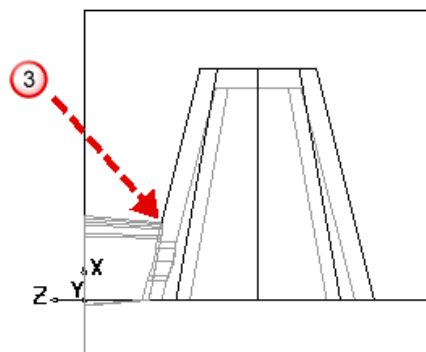
- 3 Set the **Slide core length** from the reference point.

The reference is shown as a blue plane and the other plane shows the extent of the slide core. The dimension on the model shows the distance between the two planes.

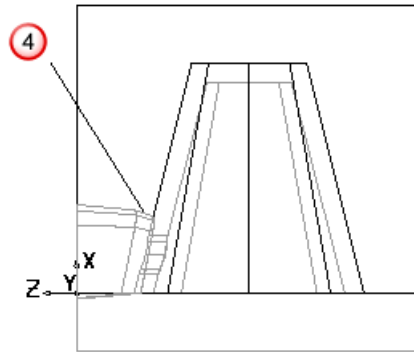
If the slide core does not extend to the edge of the insert, the **Split Surface Modification** page is not available.

- 4 If you want a run off surface on the slide core base, select the **Run Off Surface** option.

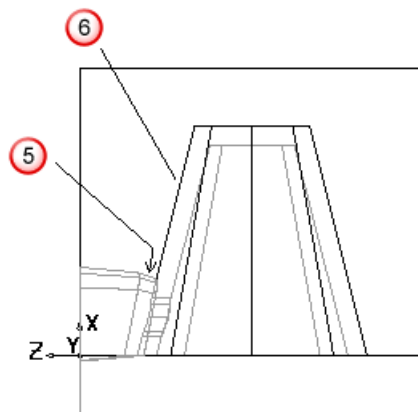
If you have a sharp edge ③ at the top of the base where it touches the die plate, you can add a run off surface to prevent it from wearing.



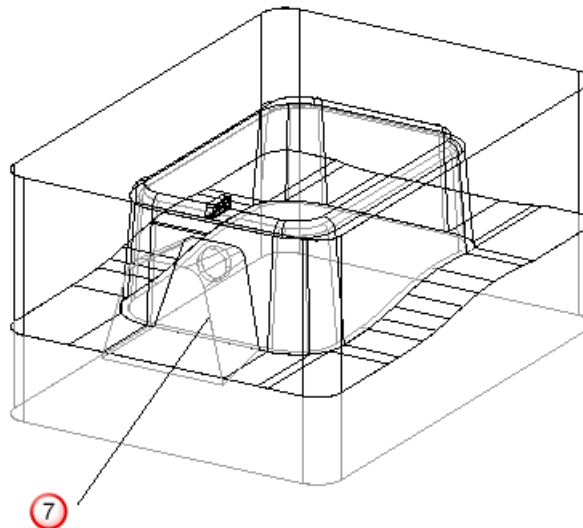
The run off surface **4** is added at the sharp edge.



Input a value for the length **5**. This length is measured along the normal of the die plate **6**.



5 Click **Preview Slide Core** to see the preview of the core. **7**



6 Click **Next** or **Finish**.

If you click **Next**, you go to the **Split Surface Modification** page of the **Slide core** wizard.

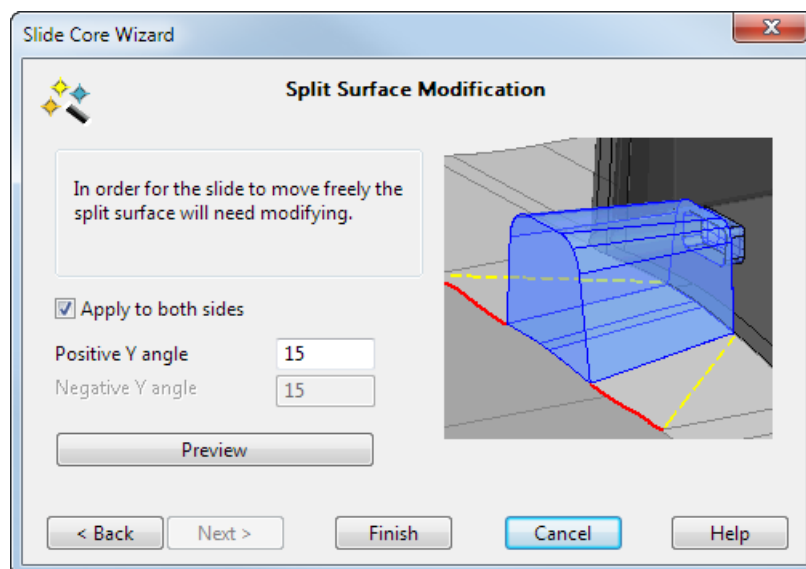
If you click **Finish** at this stage, the shape of the core will be removed from the die insert, on which the undercut was selected. The split surfaces will not be modified to allow the core to fit.

The split surfaces are modified on the next page of the **Slide core** wizard.

If you selected the **Zero draft on slide core base** option, you can only click **Finish** as there is no need to modify the split surface.

Split Surface Modification page

This allows you to automatically modify the split surfaces on the die inserts, so that the slide core can easily slide in and out of the inserts.

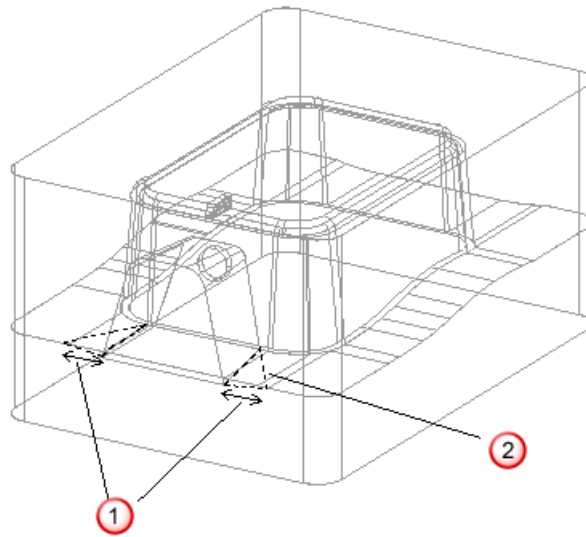


Split Surface Modification is a feature that combines split surface modification and Boolean functions so that the slide core can easily slide out from the inserts.

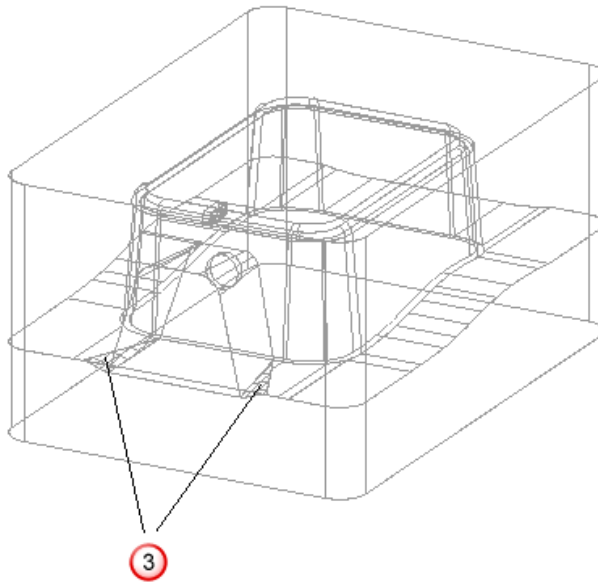
- 1 Input the angle **1**. This is the angle that the split modification extends beyond the side of the core.

Use the **Apply to both sides** option if you want the split surface on both sides of the slide core to be modified using the same angle.

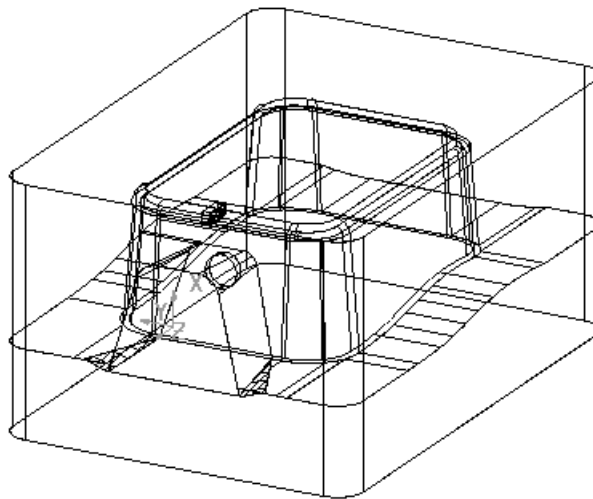
Dotted lines ② are drawn on the split surface to show where it will be modified. The bigger the angle, the gentler the change in the split surface.




2 Click **Preview** to see the modified split surface ③.



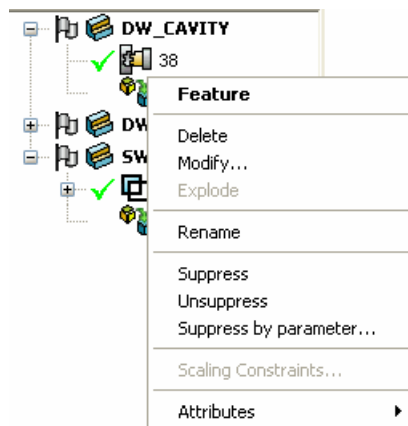
- 3 Click **Finish** to create the core and modify the split surfaces.



The new solid is named *SW_CORE n* , where n is a whole number. A workplane is also created with the name *SW_WP n* . These names are used by the Slide wizard to recognise the core. The Split Surface Modification feature is represented by  in the solid tree.

Editing a slide core feature

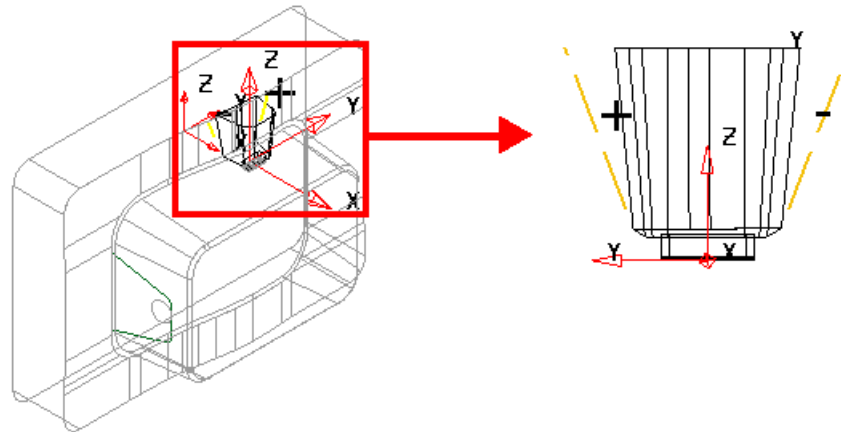
You can modify the slide core feature by selecting **Modify** from the **Feature** pop up menu and using the **Slide Core** dialog.



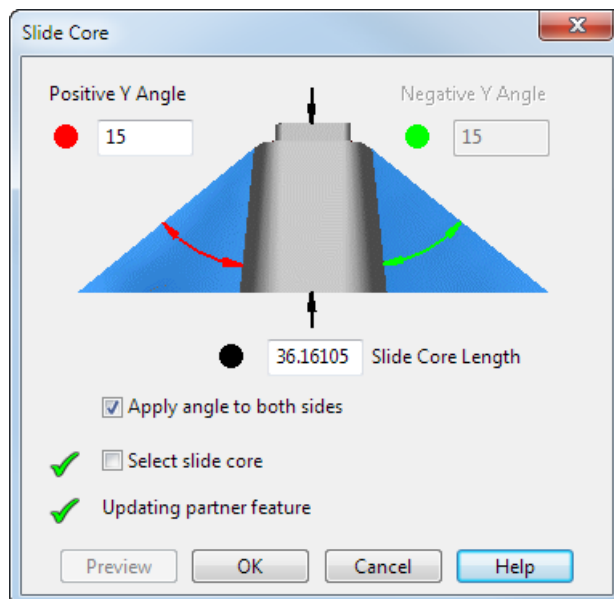
You can change the following:

- **The split surface modification angles.** You can change the positive and negative angles independently, or apply the same angle to both sides. This will also try to alter the matching feature on the other insert.

- **The length of slide core in the feature.** The other feature and the slide core will also be updated if Delcam Toolmaker identifies them.



Slide Core dialog



Positive Y Angle — Alter the angle on the positive Y side.

Negative Y Angle — Alter the angle on the negative Y side. This option is not available if **Apply angle to both sides** option is selected.

Apply angle to both sides — If this option is selected, both sides of the slide core use the modification angle that is specified in **Positive Y Angle**.

Slide Core Length — Alter the overall length of the slide core.

Select slide core — Delcam Toolmaker will try to select the slide core automatically when you choose **Modify** from the Feature pop up menu. If a slide core is found, a ✓ is displayed. If no slide core is found, a ✗ is displayed.

Updating partner feature — When you select **Modify** from the Feature pop up menu, Delcam Toolmaker tries to identify the partner feature, that is the feature that was created at the same time, but on the other insert. If a partner feature is found, a ✓ is displayed. If no partner feature is found, a ✗ is displayed.

Preview — Preview the changes.

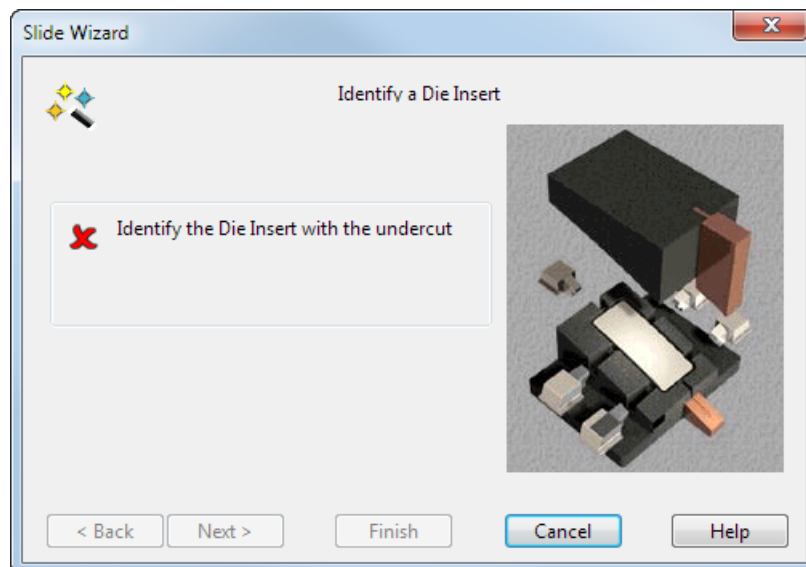
OK — Accept the edits and remove the dialog.



The options that are available on this dialog depends on the feature

Identify a Die Insert page - core puller

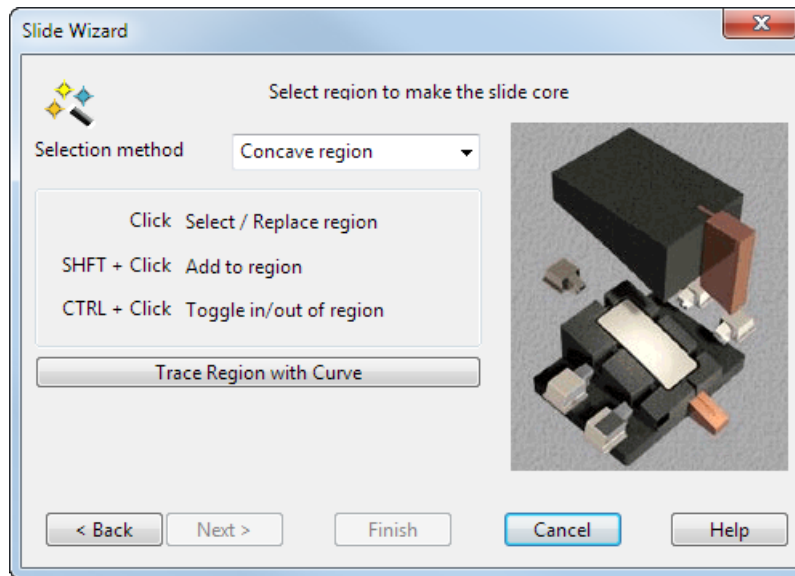
This is used to create the core puller.




- 1 Select the die insert with the undercut.
- 2 Select **Next** to go to the Select a Region page (see page 79).

Select a Region page

This is used to select the undercut region.



An arrow is drawn, pointing in the direction of the +Z axis. This shows the direction of puller.

If the wrong workplane is selected, you can select another one using the **Workplane** drop down list World  in the bottom left corner.

- 1 Select an undercut region on the solid where you want to create a core puller.

As you move the cursor over the solid, regions highlight. These are the regions you can select.

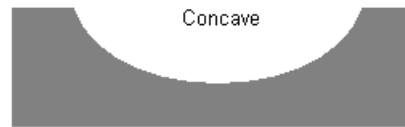
If you click a highlighted region, it is selected and changes colour. Note that any regions already selected are removed from the selection.

You can add and remove regions from the current selection using the **Shift** and **Ctrl** keys as shown on the wizard.

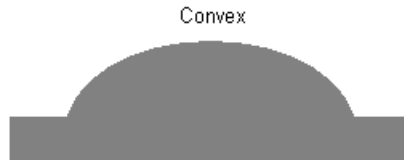
You can use box selection to select regions. Only whole faces within the box are selected, which makes it easier to select groups of faces. You can also use box selection with the **Shift** and **Ctrl** keys to add and remove regions.

Selection method - These three options determine what type of region is highlighted:

Concave region - Highlights concave regions on the die insert.



Convex region - Highlight convex regions on the die insert.



Single face - Highlights only single faces.

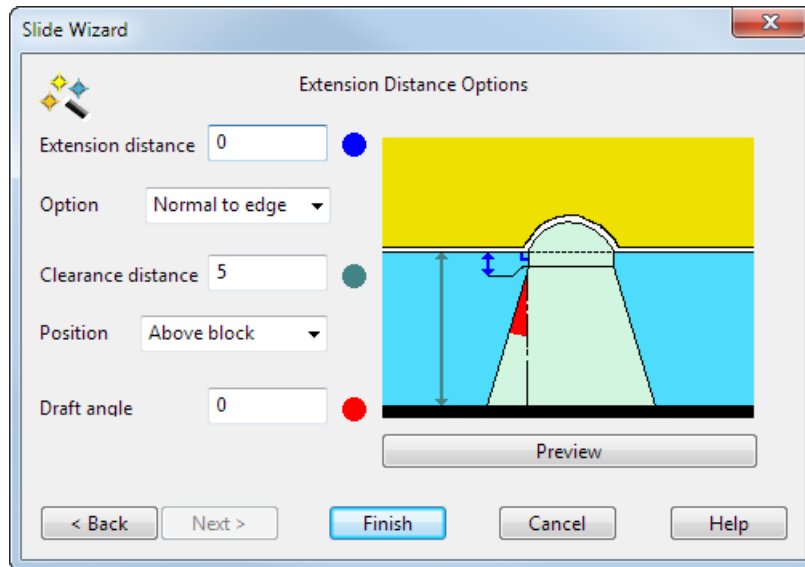
Trace Region with Curve - You can trace a closed curve on the active solid to define the region. Click **Trace Region with Curve** to display the **Create Composite Curve** control panel. Use the control panel to select a curve. The curve is coloured the same as regions selected using the automatic method. For further details, see *Creating a composite curve by tracing*.

The Trace Region method is intended to be used when the automatic selection fails because the solid is linked badly or the region is too complex. The Trace Region method takes longer to compute the undercut region so try the selection method first whenever possible. Extension Distance Options page (see page 81)

- 2 Click **Next** to display the Extension Distance Options page of the wizard.

Extension Distance Options page

Use this page to set the extension of the puller.



- 1 The graphic window on this page of the wizard shows where the three values are calculated using the blue, red and green colour codes.
- 2 Use the options on the page to define the extension of the puller.

Extension distance — This is the extension of the core puller from the undercut region.

Option — This gives three options, which determine how the core puller is extended. You may find that an option does not work for a particular core puller. If you are not sure which option to use, try one and click **Preview**. If you are not happy with the results, then try another option.

Normal to edge — The extension is perpendicular to the edge of the undercut region.

Surface internals — The extension is along the direction of the laterals/longitudinals on the underlying surface of the undercut region.

Vertical — The extension is aligned vertically along the Z axis. The internal curves of the extension are aligned with the Z axis, but the extension itself is still tangent continuous to the undercut region. For this reason, this option is not suitable for extending faces which lie flat in the XY plane.

Clearance distance — This is the distance of the core base from the selected **Position** option. It is measured along the Z axis.

Position — This is one of the following: **Above block** or **Above extension**.

Above block — This is the highest point of the solid from the XY plane.

Above extension — This is the highest point of the extension of the burning face from the XY plane.

Draft angle — This is the angle on the extension of the core puller, excluding the part which extends from the undercut region. It is measured from the Z axis.

Preview — This displays the core puller in the model window using the values from the dialog. Change the values if necessary and click **Preview** again.

- 3 Click **Finish** to create the core puller.

Creating the slide assembly using the Slide wizard

You can create a slide assembly using one of the following:

- A workplane. The workplane marks the origin of the slide within the mold.

Once you have created the slide, you can attach a core to the slide. The core can be created from any solid. For further details, see Registering a slide core solid as a component (see page 101)

- A slide core and its workplane.

You can create the slide core and workplane for the undercut region using the **Slide Core wizard**. For further details, see Creating a core using the Slide Core wizard (see page 56).

When you create a core using the Slide Core wizard, the solid core and workplane are automatically registered.

To use any of the above, they must be registered so that the wizard can recognise them.

Registering workplanes and solid cores

For each slide mechanism, you need to register the workplane and solids together as follows:

- 1 On the first page of the Slide wizard, select Register Workplanes and optional pre_modelled slide cores in order to be used to design slide mechanisms.



- 2 Select the workplane you want to use. The workplane button on the toolbar remains active while you are using the first page of the Slide Wizard, so you can create, edit and activate workplanes while the Slide Wizard is running.
- 3 If you want to attach solids or components as cores to the slide, select that option and select the objects.
- 4 Click **Register**.



Creating the slide assembly

To create the slide assembly,

- 1 Click  (Toolmaker toolbar)

Slide Wizard (1)

☒ Create a slide core...

☐ Register Workplanes and optional pre_modelled slide cores in order to be used to design slide mechanisms...

☐ Design slide mechanisms...

Slide Core Wizard

- Before starting the Slide Core Wizard
Activate a workplane where :
Z is the slide movement axis.
X is toward the top of the slide.

Start Slide Core Wizard...

Slide Core Component Creation

- Register a slide core solid as a component.

No Slide Core found ▼ Register

This functionality allows the creation of a slide core component without creating a full slide assembly. It allows the slide core component to be attached manually to any slide or another component created in the assembly.

< Back Next > Cancel Help

- 2 Make sure you have a registered workplane with/without a registered solid core. For further details, see Registering workplanes and solid cores (see page 82)

- 3 Select the **Design slide mechanisms** option.

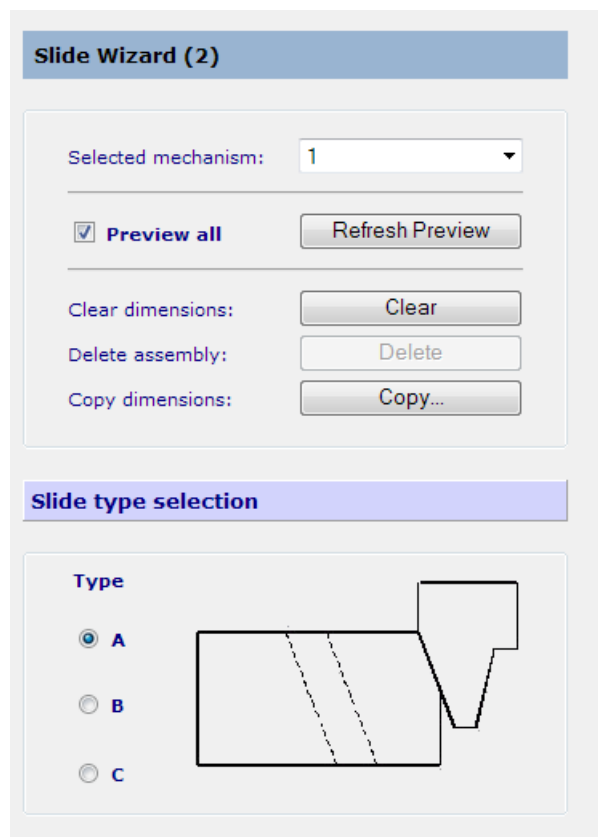


The 'Slide Wizard (1)' dialog box has a blue title bar. It contains three radio button options: 'Create a slide core...', 'Register Workplanes and optional pre_modelled slide cores in order to be used to design slide mechanisms...', and 'Design slide mechanisms...'. The third option is selected. Below these is a section titled 'Slide Assembly Creation' with a bullet point: 'Press Next to select components for the slide assembly.' At the bottom are three buttons: '< Back', 'Next >', and 'Cancel'.

- 4 Click **Next**.

Second page of the Slide wizard

This page lets you select the type of slide to create.



The 'Slide Wizard (2)' dialog box has a blue title bar. It features a 'Selected mechanism:' dropdown menu with '1' selected. Below this is a 'Preview all' checkbox (checked) and a 'Refresh Preview' button. Further down are three rows of controls: 'Clear dimensions:' with a 'Clear' button, 'Delete assembly:' with a 'Delete' button, and 'Copy dimensions:' with a 'Copy...' button. A section titled 'Slide type selection' contains a 'Type' label and three radio button options: 'A', 'B', and 'C'. Option 'A' is selected. To the right of the radio buttons is a diagram of a mechanical assembly with a rectangular base and a complex, multi-faceted top structure.

Selected mechanism — This tells you which slide core you are currently working on. This list contains all the slide cores that you need to create slides for. Delcam Toolmaker knows whether you have slide cores in your mold by their names.

If you have multiple cavities, only the cores for one cavity are given. When you create their slides, the slides for all the other cavities are created from them.

Preview all — If on, this draws a preview of all the slides you have defined. You may need to click **Refresh Preview**. If off, only the current slide is displayed.

The above three items appear on all the pages. You can switch from one slide to another at any time.

Refresh Preview — Updates the view depending on which slides you want to preview.

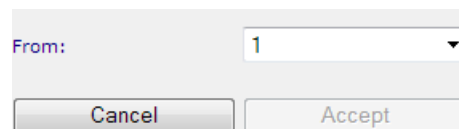
Clear dimensions — Click **Clear** to remove all the dimensions in the **Slide wizard** for the selected slide core.

Copy dimensions — This allows you to duplicate the components from an existing slide to the one currently selected.

To duplicate a slide:

- 1 Click **Copy**.

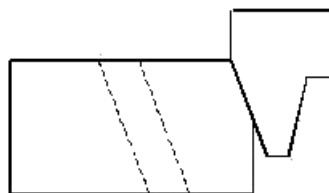
More options appear below it.



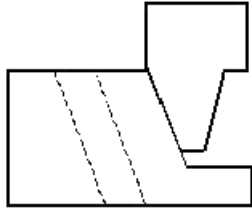
- 2 Select the name of the slide core you want to copy from.
- 3 Click **Copy**.

Slide type selection — Choose the slide type from one of the following:

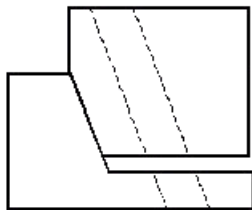
Type A — This is the most common slide configuration, which has the locking angle for the heel block machined on the outside edge of the slider.



Type B — This has the heel block location machined out of the slider to save space.



Type C — This is similar to Type B but the angle pin actually enters the heel block making the assembly slightly more compact.



Next — Displays the next page of the wizard.

Third page of the wizard

This page lets you define the basic dimensions of the three main components of the slide.

- Slide
- Heel block (see page 92)

- Angle pin (see page 94)

Slide Wizard (3)

Selected mechanism: 1

☒ Preview all
Refresh Preview

Click on component to display its dimensions.

Get slide dimensions...

Slide Bottom Length

Slide Top Length

Slide Height

Heel block angle

Slide Shoulder Width

Slide Shoulder Height

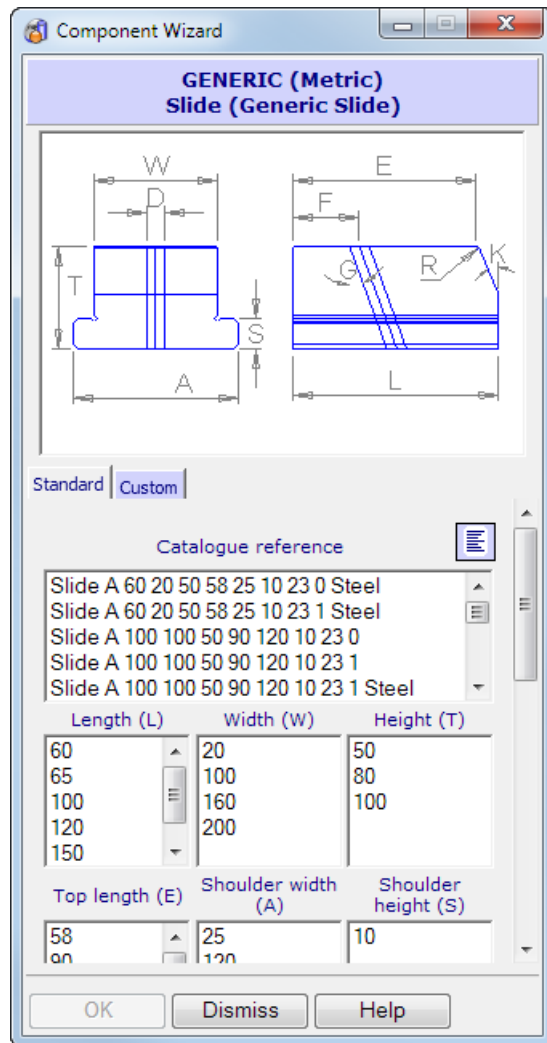
Slide Top Width

Horizontal Offset

Vertical Offset

Component details...

When this page of the wizard is displayed, the options are not available. When you first specify the components for the selected slide, the Component Wizard dialog is also displayed for you to enter details about the component.



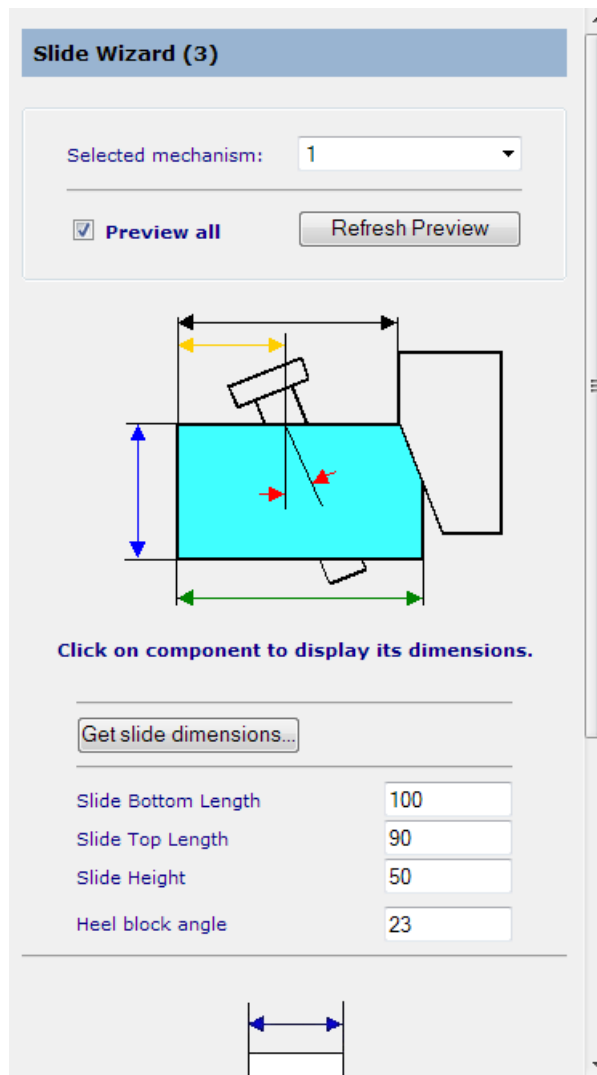
- 1 Use the Component wizard dialog to set the dimensions of the slider.

The image on this page depends on the type of slide you selected. We will give a general description of the page so that it applies to all types of slides.

You can select values in the dialog. Alternatively, click on **Custom** to enter your own values.

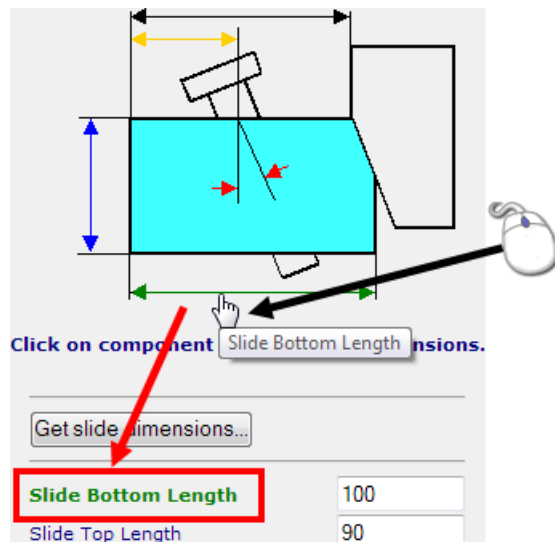
- 2 Select a combination

- 3 Click **OK** to close the Component wizard dialog and transfer the dimensions to the Slide Wizard page

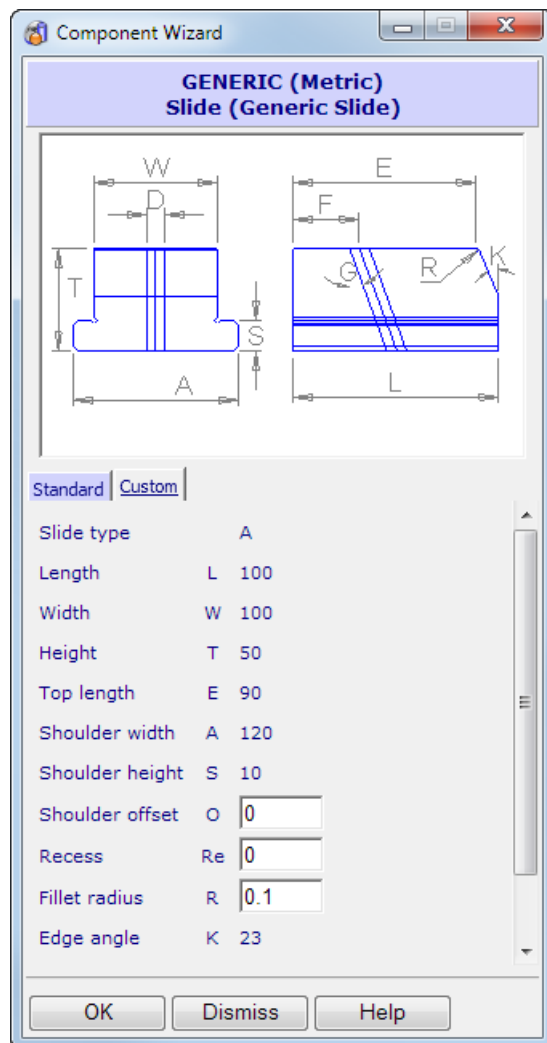


- 4 Edit the values to your requirements. If you want to display the **Component wizard** dialog again, click **Get slide dimensions**.

If you hover the mouse over one of the dimensions on the image, the text corresponding to that dimension highlights.



You can access other parameters of the slider by clicking the **Component details** button to display the component's dialog.

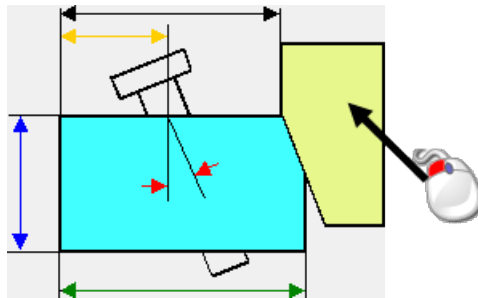


The values you entered in the wizard appear in this dialog. This dialog allows you to edit the smaller dimensions (such as radii and chamfers) for the component.

- 5 Fill in the dimensions. This dialog is very similar to the browser page used to define components. For further details see [Defining a component using the Component Wizard dialog](#)
- 6 Click **OK**.

Dimensioning the heel block

- 1 Move the mouse over the heel block in the image.

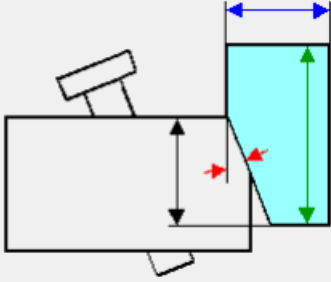


- 2 Click the heel block on the image in the wizard and the page changes.

Slide Wizard (3)

Selected mechanism: 1

☒ **Preview all**



Click on component to display its dimensions.

☐ **No Heel Block**
☒ **Heel Block Component**
☐ **Machined Directly Heel Block**

Heel Block Top Length	50
Heel Block Angle	23
Heel Block recess	0
Heel Block Height	50
Heel Block Edge Height	50
Heel Block Width	100

☐ **Insert a wear plate**

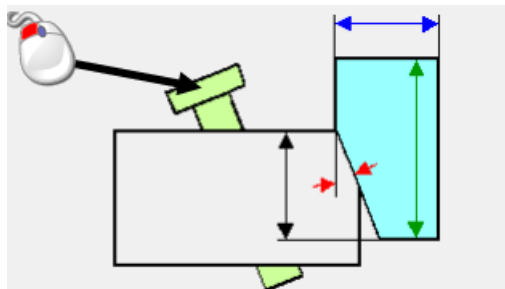
- 3 Set the options for the heel block.
- If you don't want a heel block, select **No Heel Block**.
- If you will machine the heel block into either the core or cavity plate, select **Machined Directly Heel Block**.
- If you want a heel block component, select **Heel Block Component**.
- 4 If **Heel Block Component** is selected, enter values to define the heel block.

- 5 If you want a wear plate in front of the heel block, select **Insert a wear plate** and enter the **Wear Plate Thickness**, **Wear Plate Height**, and **Wear Plate Width**.

<input checked="" type="checkbox"/> Insert a wear plate	
Wear Plate Thickness	<input type="text" value="3"/>
Wear Plate Height	<input type="text" value="50"/>
Wear Plate Width	<input type="text" value="100"/>

Dimensioning the angle pin

- 1 Move the mouse over the angle pin in the image.

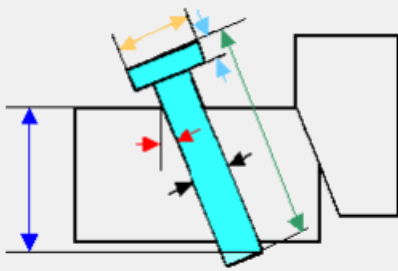


- 2 Click the angle pin on the image in the wizard and the page changes.

Slide Wizard (3)

Selected mechanism: 1

☒ **Preview all**



Click on component to display its dimensions.

Number of angle pins: ☐ 0 ☒ 1 ☐ 2

Angle Pin Length	100
Pin Angle	20
Angle Pin Diameter	19.9
Angle Pin Head Thickness	19.9
Angle Pin Head Diameter	39.8
Hole Diameter	20

- 3 Set the options for the pins.
- 4 Click **Next** to display the next page of the wizard.

Fourth page of the wizard

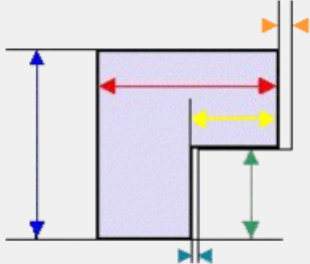
This sets the dimensions of the guide rail for the slider.

Slide Wizard (4)

Selected mechanism: 1

☒ Preview all Refresh Preview

☒ Insert guide rails



Length	143
Width	20.1
Height	20
Cut Width	10.1
Cut Height	10
Shoulder clearance	0.1
Contact clearance	0
Position offset	0

Component details...

☐ Insert a wear plate

< Back Next > Cancel Help

- 1 Enter the required dimensions. The dimensions options are only available if **Insert guide rails** is *ON*. The graphic shows you the dimensions that you are defining.

Shoulder clearance — Recess the position of the shoulders on the slide component by this amount.

Contact clearance — Clearance distance (gap) between the guide rail and the slider body.

Position offset — Offset the position of the guide rail relative to the shoulders of the slide by this amount.

- 2 Enter the dimensions for the wear plate. The options that are available depend on the settings of **Insert a wear plate** and **Insert guide rails**.

Insert a wear plate — If *ON*, the following options appear at the bottom of the browser page.

Slide Wizard (4)

Selected mechanism: 1

☒ **Preview all** [Refresh Preview](#)

☐ **Insert guide rails**

☒ **Insert a wear plate**

Wear Plate Length: 143

Wear Plate Thickness: 10

Wear Plate Width: 120.2

[Component details...](#)

Insert Guide Rails — If *ON*, the following options appear at the bottom of the browser page.

☒ **Insert a wear plate**

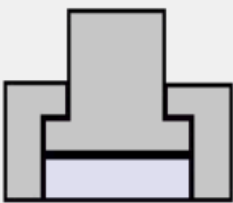
Wear Plate Length: 143

Wear Plate Thickness: 10

Wear Plate Width: 120.2

☒ Limit wearplate width to the guide rails

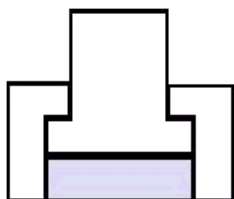
☐ Limit guide rails height to the top of the wearplate



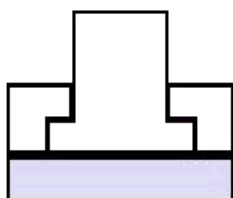
[Component details...](#)

You can select two size of wear plate:

- Limit wearplate width to the guide rails.



- Limit guide rails height to the top of the wearplate.



Next — When you have set the dimensions, click to display the Fifth page of the wizard (see page 98)

Fifth page of the wizard

This creates a pocket for the slide assembly.

Slide Wizard (5)

Selected mechanism: 1

☒ Preview all Refresh Preview

Slide Pocket

☒ Create pocket for slide assembly

Options : ☐ Blind ☒ Through

Side Clearance	0
Shoulder clearance	0.1
Contact clearance	0
Position offset	0

Create pocket for slide assembly — If *ON*, a panel of options is displayed. If *OFF*, all the options are suppressed.

Side Clearance — The clearance across the width of the Slide when the pocket is generated.

Shoulder Clearance — Recess the position of the shoulders on the slide component by the value in this box. If **Insert guide rails option** was *ON* in **Slide Wizard (4)**, this option is not available on this page.

Contact Clearance — Clearance distance (gap) between guide rail and slider body. If **Insert guide rails option** was *ON* in **Slide Wizard (4)**, this option is not available on this page.

Position Offset — Offset the position of the guide rail relative to the shoulders of the slide by the value in this box. If **Insert guide rails option** was *ON* in **Slide Wizard (4)**, this option is not available on this page.

Options — Choose the type of pocket to create in the core and cavity plates.

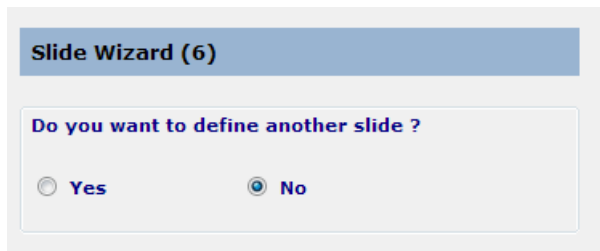
Through — The pocket cuts all the way through the core and cavity plates.

Blind — The pocket lies in the core and cavity plates. You can specify its length within the plates.

Next — Display next page of the wizard.

Sixth page of the wizard

This gives you the option of creating another slide.

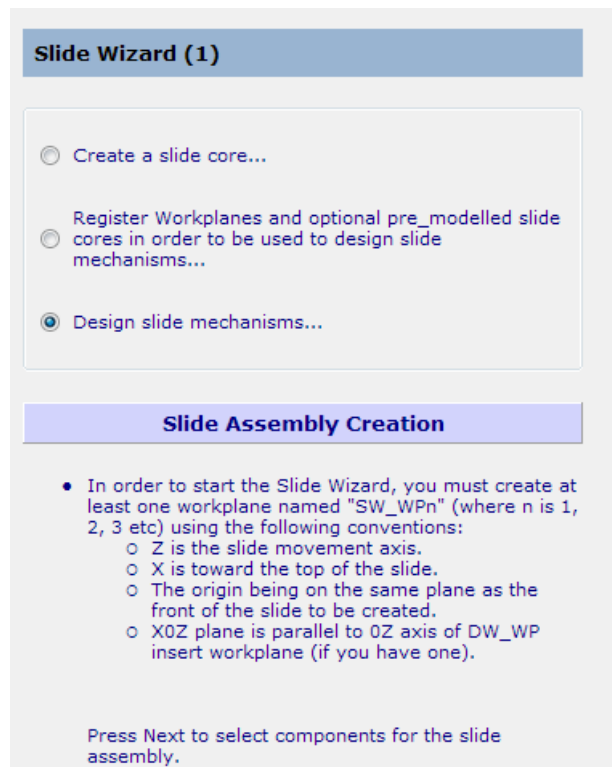


Slide Wizard (6)

Do you want to define another slide ?

☐ Yes ☒ No

Yes — Return to first page of the Slide wizard to define another slide.



No — Components in the preview slide assembly are created in the mold.

Next — Continue. The action will reflecting the selection you have made.

The components in the slides are named with the following string:

Component name+slide mechanism id+(C,R)

where **C** is the column number and **R** is the row number.

If there are multiple cavities in the model, the following naming convention is used:

GuideStrip(2)1(2,1)

Slide1(2,1)

If there is a single cavity in the model, the following naming convention is used:

GuideStrip(2)1

Slide1

The number of cavities is defined in the **Multiple cavities** section of the Core and Cavity Wizard.

Re-running the Slide Wizard

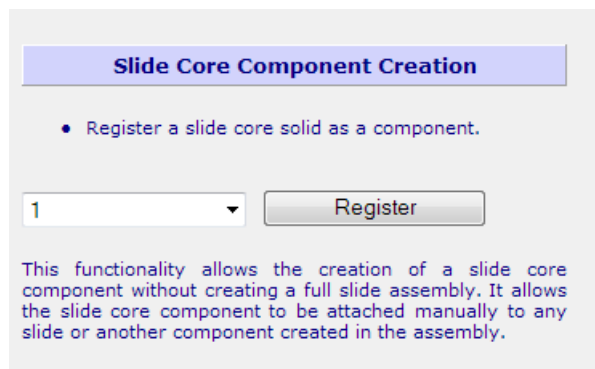
You can re-run the Slide Wizard. The parameters of any previously created slides are used. You may change any of the parameters; the slide assembly will redefine itself and pockets are updated.

Registering a slide core solid as a component

When you create a slide using only a workplane, you can use any solid as its core.

To do this you must register the solid as a component and then position it on the slide mechanism.


- 1 Create a slide using a workplane only.
- 2 Create a solid for the core.
- 3 Make sure the name of the solid is of the form *SW_CORE n* , where n is an integer.
- 4 Make sure the first page of the **Slide** wizard is displayed. For further details, see Creating a core using the Slide Core wizard (see page 56)
- 5 Scroll down the page to the **Slide Core Component Creation** section.



- 6 Select the name of the solid from the drop down list.
- 7 Click **Register**.
- 8 You can now position the component. For further details, see Repositioning a component

Using the Mold Base Wizard

You can run this wizard automatically after the **Cavity Core Wizard**, by selecting the **Run the Mold Base Wizard on exit** option on the final page of the Cavity Core Wizard. Alternatively, you can run the Mold Base Wizard as follows:

- 1 Click  (Toolmaker toolbar) to display the first page of the Mold Base Wizard (see page 103).
- 2 Use the first page of the Mold Base Wizard to select the mold base and the components from a catalogue.
- 3 Use the second page of the Mold Base Wizard (see page 107) to set the dimensions of the listed components used in the mold base.
- 4 Use the third page of the Mold Base Wizard (see page 108) to check your parameters.

The Mold Base Wizard produces parameterised components. Power Features are not included for components in a standard mold base, since all fits and clearances are already defined. However, if the mold base contains plain plates, Power

Features are included so that relevant features may be applied later.

You can edit an existing mold base (see page 112).

First page of the Mold Base Wizard

The first page of the **Mold Base Wizard** enables you to select the mold base and its components.

Mold Base Wizard

Base catalog: <Choose a catalog>

Dimensions: ☒ Standard ☐ Custom

Base type:

☒ Support plate

☒ Risers

☒ Ejection system

Guide pillars: ☒ Fixed half ☐ Moving half

Show the mold base

Rotate the mold base:

Plates

Type:

Thickness:

Material:

Dimensions:

Product code:

Comments...

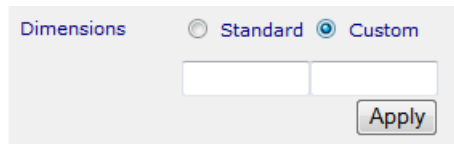
Dimensions...

Levels...

< Back Next > Cancel Help

- 1 Choose a catalogue from the **Catalog** menu. If you have already set it in the **Cavity-Core Wizard**, you cannot change it here.
- 2 Use the **Dimensions** options to select the size of the mold base. This may already have been defined in the **Cavity-Core Wizard**.

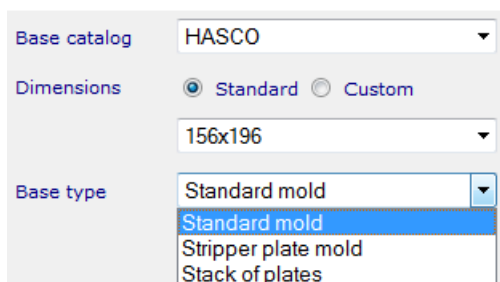
If you want to create a custom mold base, select the **Custom** option. Two boxes appear below the option for you to enter your dimensions.



3 Enter your values and click **Apply**.

4 Select the type of mold base from the **Base Type** options .

When you choose **Stack of plates** from the **Base type** list, the wizard creates a mold that consists of one special plate, the **Master plate**.



You can then do one of the following:

- add optional plates to the mold using the plate list buttons
- enter the number of plates in the **Number of Plates** box and select **Apply**.



1. *Stack of plates is not currently supported by the Cavity Core Wizard.*

2. *You cannot remove the master plate from the mold.*

- 1 A **Support plate** is included as part of the standard mold construction. If you don't want a support plate, deselect this option.
- 2 Ejector plates are included as part of the standard mold construction. If you don't want an ejector plate, deselect **Ejector system**.
- 3 Select which half contains the **Guide pillars**. Choose one of the following:
 - **Fixed half**
 - **Moving half**.
- 4 Select one of the **Rotate the mold base** options to rotate the mold to be created by 0°, 90°, 180°, or 270°.

The default rotation is 0°. This conforms to the mold industry standard, that is, the width of the mold base is along the X axis and the length is along the Y axis.

- 5 The Plates section displays a list of plates in the selected mold base.

Select a plate from the list. It is highlighted on the model and its **Type**, **Thickness**, and **Material** is displayed. You can set the **Type**, **Thickness**, and **Material** available for each plate from the menus.

The following buttons work on the plate that is currently selected in the **Plates** list.



— Make the selected plate plain.




— Make all the plates in the mold plain.




— Rename the plate.



— Attach an optional plate to the selected plate.

If the  option is selected, the inserted plate is attached unconditionally to the requested face of the selected plate. Positions of all other plates are unchanged. This option may be used to add optional ejector plates to the mold.

If the default  option is used, an optional plate is inserted on the requested face of the selected plate. All the other plates are moved to accommodate the optional plate.

On finishing the wizard, assembly modelling relations are used to attach the optional plate to either the selected plate or the plate above/below the selected plate. The position of the master cavity plate of the mold relative to the optional plate determines the final location of the optional plate.



— Insert an optional plate above the selected plate.



— Insert an optional plate below the selected plate.



— Remove the selected optional plate. You cannot use this option to remove a predefined plate.



1. You cannot insert an optional plate between the cavity and core plates if they have been created in the Cavity Core wizard.
2. A mold with an optional plate between the cavity and core plates will not be detected by the Cavity Core wizard.

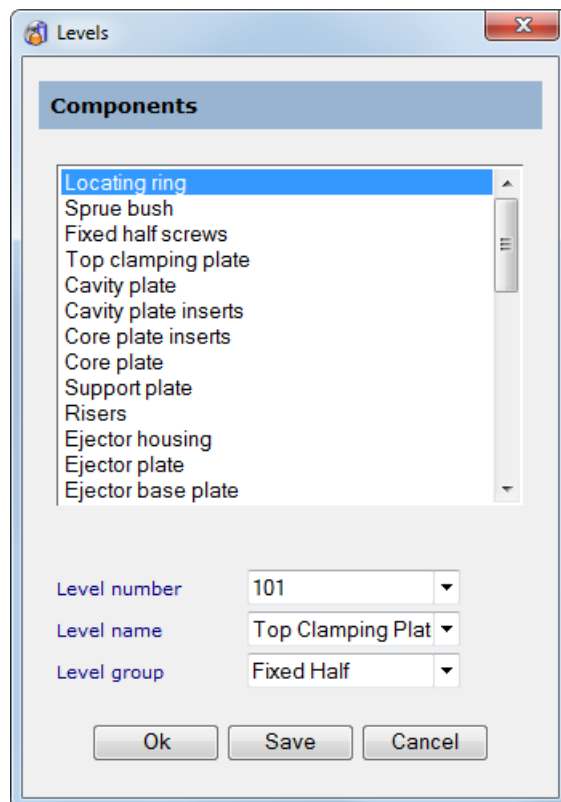
- 1 The **Dimensions** information that is displayed indicates the units and if the selected plate is standard or custom.

Click **Dimensions** to change the dimensions of the selected plate-. This displays a dialog for Defining a component.

If you used the **Cavity-Core Wizard**, the correct **Type**, **Thickness** and **Material** for the cavity and core plates are automatically entered. You may alter these values for inserted and empty pocket modes.

- 2 Click **Levels** to specify the level for each component of the mold base. By default, components are created on different levels. The levels for the components are put in two groups: moving half and fixed half.

When you click **Levels**, the following dialog is displayed.



- a Select a component from the list.
- b In the three **Level** boxes, you can see the details of the level it will be created in.

Level number — Edit the level number by either typing in a value or selecting a number from the list.

Level name and **Level group** — Use these to change the details about the level.

- c Click one of the following:

OK — Components will be created on the given levels.

Save — Details of the level for each component are stored. These details are then available for all future molds.

- 3 Click **Next** to display the second page of the **Mold Base Wizard**.

Second page of the Mold Base Wizard

The wizard searches the component catalogue for pillars, bushes, sleeves and screws whose length and diameter match your chosen mold base size and thickness of plates.

If the wizard cannot find matching pillars, bushes sleeves or screws, custom dimensions are used instead. If this happens, and you would like to use standard component dimensions, try one or both of the following:

- Click **Back** and change the thickness of plates, the dimension of the plates or, in some cases, the mold base size.
- Click **Dimensions** and modify the dimensions of the components accordingly.

This page sets the dimensions of the listed components used in the mold base.

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 14
Shank length 75
Dimensions: Standard, mms
Product code: Z00/17/14x75

Dimensions...
Levels...

< Back Next > Cancel Help

- 1 Select the required components from the list and set their sizes.
Remove - Removes the selected component from the list.
Add - Any removed components can be added back again by clicking the **Add** button and selecting the component from the dialog displayed.
- 2 Click **Show the component** to expand the window to display a diagram which shows an image of the component and how its dimensions are measured. You can hide the image again by clicking the **Hide the component** option.
- 3 Click **Levels** to specify the level for each component of the mold base (see page 103)e.
- 4 Click **Next** to go to the Third page of the Mold Base Wizard (see page 108).

Third page of the Mold Base Wizard

Use the final page of the Mold Base Wizard to

- save the current mold parameters as a mold configuration (see page 110). This configuration will include:
 - catalog.
 - mold size.
 - mold type.
 - guide pillar placement.
 - list of plates (including all blank plates that have been added manually).
 - auxiliary components and dimensions.
- load a saved mold configuration (see page 111). These parameters are applied to the new mold.
- load a recently used mold configuration (see page 111). These parameters are applied to the new mold.

- check your parameters before creating the mold base.

Mold Base Wizard

You have successfully completed the Mold Base Wizard

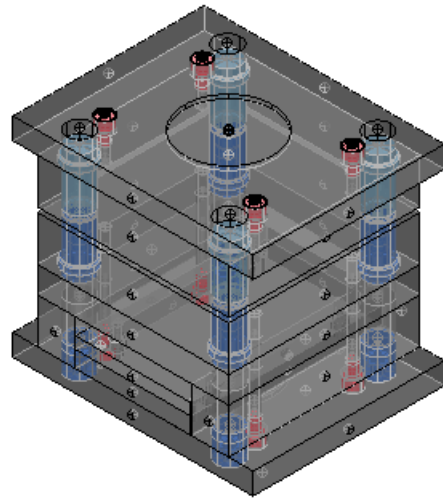
You specified the following parameters for the mold base:

Mold base	
Catalog	HASCO
Type	Standard mold
Guide pillars	Fixed half
Rotation angle	0
Width	296
Length	346
Dimensions	Standard
Cavity plate	
Type	K20
Thickness	22
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	22
Material	1730
Dimensions	Standard
Support plate	

< Back Finish Cancel Help

- 1 Change the view to *ISO1* so that you can see the components clearly as they are created.

- 2 Click **Finish**. Wait a short time for the mold base to be created.



When the mold base is finished, you can view it in different ways by selecting the following views

Wireframe 

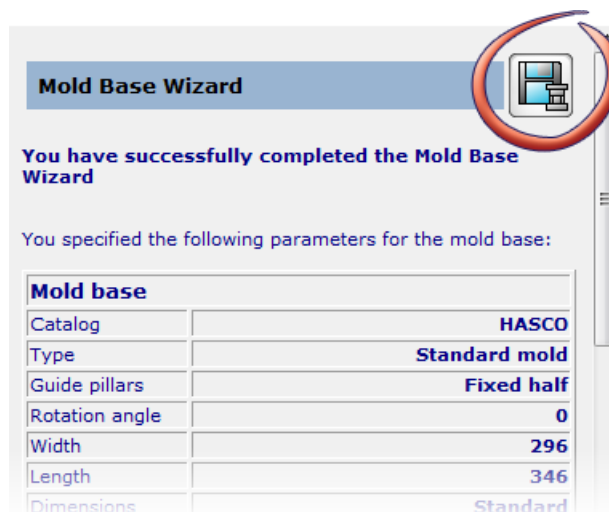
Shaded 

Transparent wire 

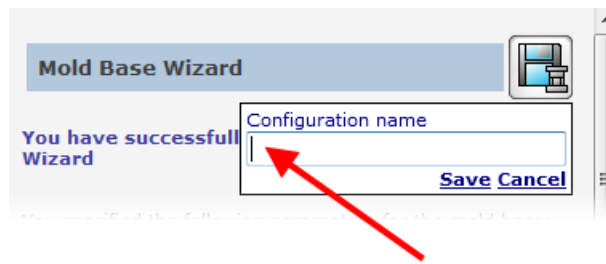
Shaded wire 

Saving a mold configuration

- 1 Click  on the final page of the **Mold Base Wizard**

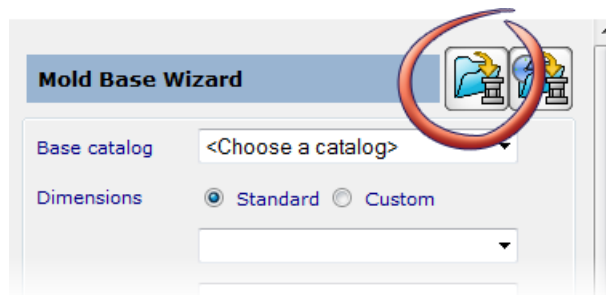


- 2 Enter a **Configuration name** and click **Save**.

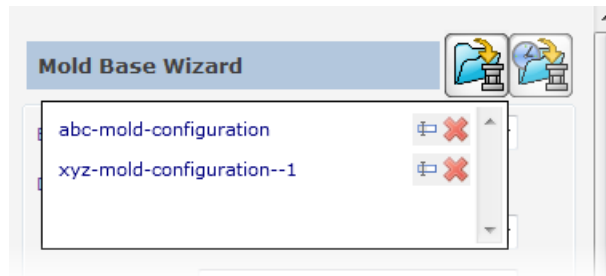




Loading a saved mold configuration

- 1 Click .




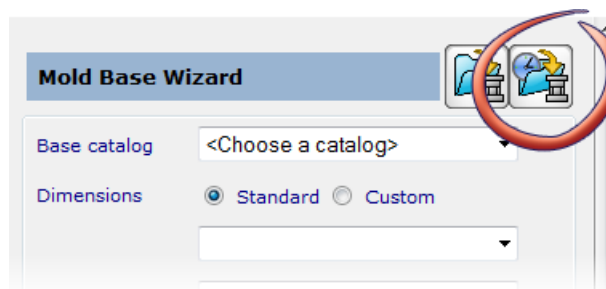
- 2 Select the name of the saved configuration from the list.



- 3 Use the following as required:
 - Click  to rename the mold configuration.
 - Click  to delete the mold configuration.

Loading a recent configuration

- 1 Click  to load a recent mold configuration.



- 2 Select the required mold configuration from the list



Editing the mold base


You can re-run the Mold Base Wizard to make relevant changes.

Each component in the mold base is registered as an Assembly component. You can edit the components using Assembly modelling.

Mold Lock Wizard

The Mold Lock Wizard lets you can define several locks on a mold at the same time. Without this functionality, you would need to create each half of each lock independently, ensuring that the dimensions and positioning of the male and female halves were complimentary.

Creating a Mold Lock

- 1 Select **Toolmaker** from the Module menu.
- 2 Click  (Toolmaker toolbar).
- 3 Select the cavity and core inserts using **Mold Lock Wizard - Page 1** dialog.
- 4 Define the mold locks using the options on the **Mold Lock Wizard - Page 2** dialog.
- 5 Use the simulation option on **Mold Lock Wizard - Page 2** to check the mold locks and modify as appropriate.







You can only run the Mold Lock wizard if you have a Delcam Toolmaker licence.

Adding additional Mold Locks

Once locks for a specific layout have been created you can not add any more locks to that layout. Selecting the layout from the layout options will simply add the already created locks to the list, allowing you to edit the created locks. The only exception to this is custom layout where preview locks can be created even when created custom locks already exist.

The following example illustrates the way selecting different layouts is handled.

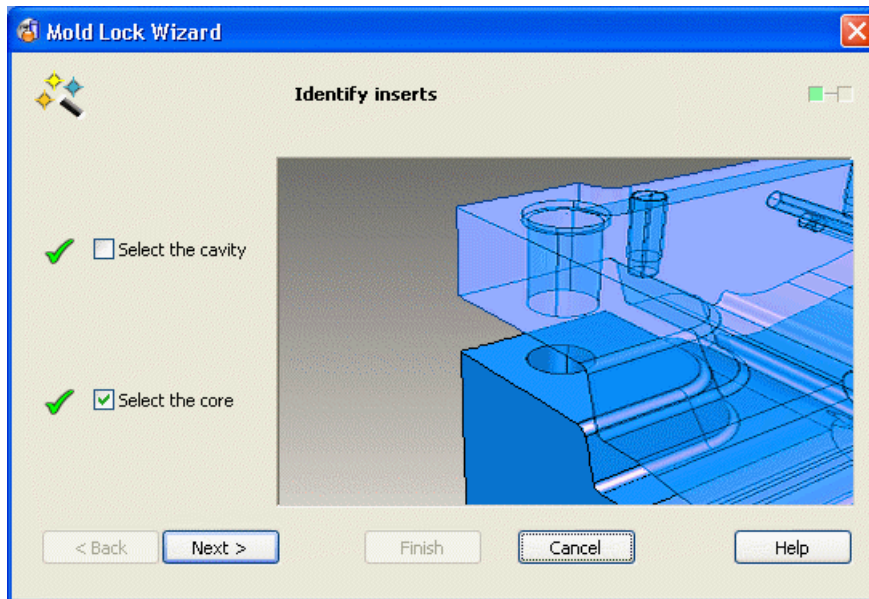
- 1 Select the cavity and core inserts using **Mold Lock Wizard - Page 1** dialog.
- 2 Select the first layout  and create preview locks. Click **Apply** to create the locks. These are now classed as created locks.
- 3 Every time you come back to this layout, these created locks will be displayed and can be edited but you can not add any further locks to this layout.
- 4 Now select the second layout . This creates preview locks for this layout.
- 5 Switch back to the first layout . The previously created locks are again displayed and the preview locks are hidden.
- 6 Selecting the second layout  will display the preview locks again.



Preview locks are not deleted when switching to a layout that contains created locks, they are merely undrawn. Preview locks are only deleted when you switch to a layout that contains no created locks or you manually delete them.

Mold Lock Wizard - Page 1

This page is simply used to select and identify the core and cavity inserts. The inserts can either be solids or components.



Select the cavity - Select the cavity insert. The cross **✗** icon changes to a tick **✓** icon..

Select the core - Select the core insert. The cross **✗** icon changes to a tick **✓** icon..

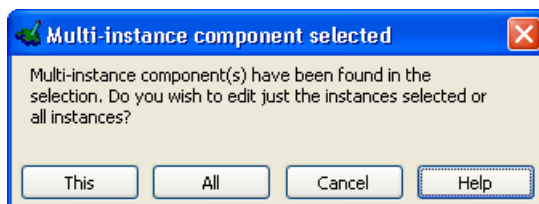
Next - If you select this option and you have selected any multi-instance components, the Multi-Instance Components dialog will be displayed. In all other cases, **Mold Lock Wizard - Page 2** will be displayed

If you make an incorrect selection an error message is displayed. Examples of incorrect selection are:

- The item selected is not a solid or component
- Multiple items have been selected
- The same item is selected for both core and cavity

Multi-Instance Components dialog

This dialog is displayed if you have selected any multi-instance components.



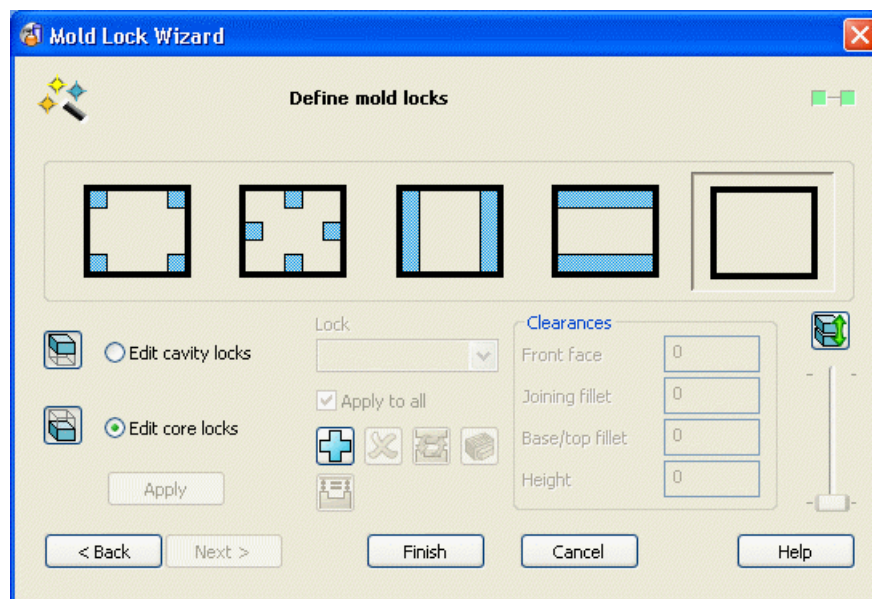
This - The selected multi-instance components are converted into single-instance components. Any change that you make in the wizard will only be made to this instance of the component.

All - Keeps the component(s) as multi-instance components and takes you to **Mold Lock Wizard - Page 2**. Any changes that you make in the wizard will be made to all instances.

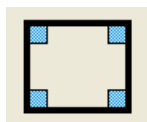
Cancel - Removes the dialog and returns you to the **Mold Lock Wizard - Page 1** so that you can exit the wizard or make another selection.

Mold Lock Wizard - Page 2

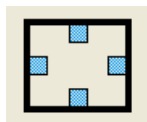
This page lets you define the layout and clearances of the mold locks.



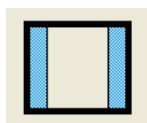
You can select a predefined layout for the mold locks or define your own layout by selecting the custom layout option and editing the locks accordingly.



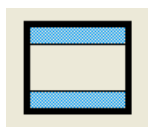
Position four mold locks on the corners of the inserts



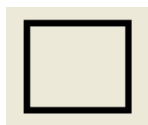
Position four mold locks so that they are at the midpoint of the edges of the insert.



Create a pair of mold locks running along the left and right edges of the inserts.

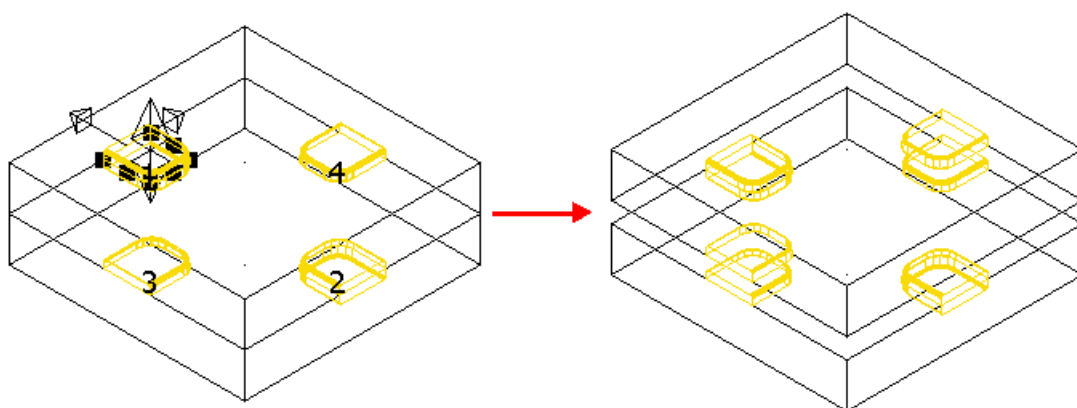


Create a pair of mold locks running along the top and bottom edges of the inserts.

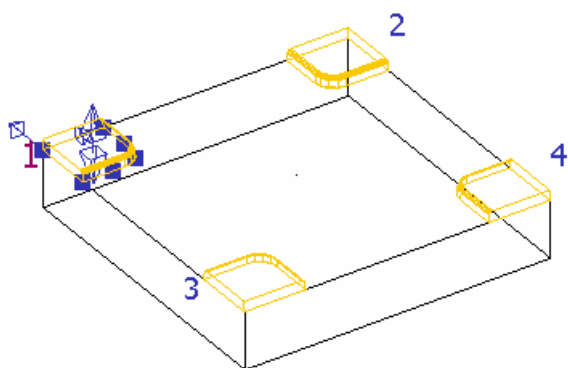


Custom layout, with no preview locks initially.

When using predefined layouts, the male lock half (the protrusion), is always added to the core insert and the female lock half (the pocket) on the cavity insert.



All the preview mold lock halves are displayed as instrumentation and the current active one has drag handles attached to it. The locks are labelled, using the same name that is displayed in the **Lock** drop down list. The colour of the label for the active lock is a different colour to the labels for the other locks.



- Show/hide the cavity insert.



- Show/hide the core insert.

Edit cavity locks - Edit the cavity locks, with the corresponding changes being made to the associated core locks.

Edit core locks - Edit the core locks, with the corresponding changes being made to the associated cavity locks.

When you select **Edit cavity locks** or **Edit core locks**, the insert that you are not using is hidden to help editing. On start up, **Edit core locks** is automatically selected and the cavity insert is hidden.

Lock - The name of the current active lock is displayed in the box. To change the active lock, select a lock from the drop down list.

If Mold Lock Wizard has already used the inserts to create some mold locks, these will be automatically identified and added to the **Lock** drop down list. Any mold lock in the drop down list can be edited using the options available on this page of the wizard.

Apply to all - If *ON*, the changes that are made to the current lock will be made to all the other locks on the same layout. The option is unavailable if you are using a Custom layout; in this case you must edit each lock individually. If *OFF*, any change will be made to the current lock only.




*The **Apply to all** option is only applicable to updated dimensions. So, if the option is ON and the length of one lock is updated, the length of all the core and the cavity locks will be updated. If the widths of these locks were different, they would retain their original dimensions.*



- Add a new lock to the inserts. This option is only available if you are using Custom layout.

To add a new preview lock,

- 1 Select the **Add a new lock**  button. The preview lock is attached to the cursor. If necessary, you can detach the lock from the cursor by clicking anywhere on the graphics window, away from the two inserts.
- 2 Position the male half of the preview lock on the insert and click the left mouse button.
- 3 You can position the male half of the lock on either the cavity or core insert; the corresponding female half of the lock is automatically positioned on the other insert.





- Delete a lock. This deletes both halves of the active lock. Another lock (if available) automatically becomes the active lock.



- Toggle male/female lock halves. When you select this button, existing male lock halves are converted to female lock halves.

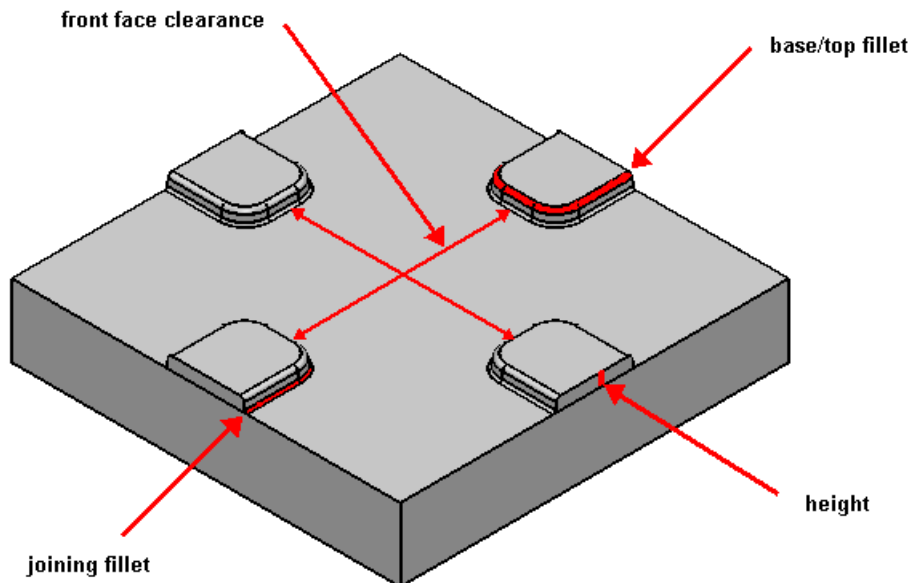
So if the male lock was attached to the core insert and the female lock half was attached to the cavity, selecting this option would attach the male lock to the cavity insert and the female lock to the core insert.

 - Display the **Protrusion** dialog for the currently active lock half. This lets you adjust the dimensions and positioning of the lock. Selecting **Accept** on the **Protrusion** dialog will apply the changes to the other half of the lock. If **Apply to all** is set, it will also apply the changes to the other locks.

 - If this option is selected, the tops of the male lock halves on the active insert will be at the same Z position. This will be set to the Z value of the lock with the highest Z position.

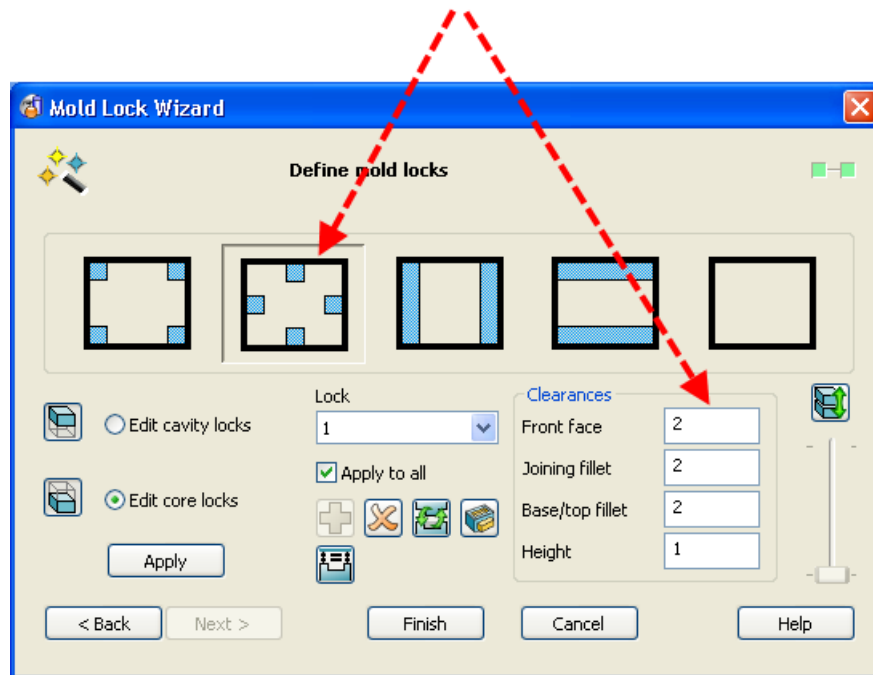
 1. This option overrides **Apply to all**.
2. Any male lock halves on the inactive insert are ignored.

Clearances - These options are used to specify the clearance between the male and female lock halves. There are four clearances that can be set at this point. If the **Apply to all option** is *ON*, any changes made in this section will be applied to all locks in the current layout.



Front face - Enter a value to change the clearance between the front surfaces for the lock halves.

This option is only available if you are using the second layout option



Joining fillet - Enter a value to change the joining fillet radius clearance between the halves. The joining fillet clearance is the one between the fillets that join the feature to the insert.

Base/top fillet - Enter a value to change the radius clearance between the female base fillet and the male top fillet.

Height - Enter a value to change the height clearance between the lock halves.

Any clearances that you set are applied to the inactive lock. So, if the height of the active lock half is 10 and you set the height clearance to be 1, the inactive lock half will be either 11 (if the inactive half is a pocket), or 9 (if the inactive half is a protrusion).



If you select the **Start/End opening simulation** button, the slider is active, but all other options on the page are disabled. The slider can then be used to separate the inserts. The slider is disabled unless you are in simulation mode.

Apply - Creates locks from any preview locks that currently exist. If any of the mold lock halves have an invalid definition, for example if a fillet radius is too large, the definition is marked by a series of dotted lines and a warning message is displayed.

The warning message tells you the lock half to adjust. You can either make an adjustment, or ignore the warning and create the locks.

Finish - Exit the Mold Lock wizard. If some preview locks exist, but you have not used **Apply** to create the locks, selecting **Finish** will cause these to be created.

Re-running the Mold Lock Wizard

It is possible to re-run the Mold Lock Wizard to edit previously created locks. The wizard will automatically detect these locks and identify them, based on the layout. If only one lock half is non-existent (if you deleted it or it failed to be created correctly), the wizard will automatically create a preview for the missing lock half. This can then be edited as normal. When you select **Apply** or **Finish**, the wizard will attempt to create this lock half.

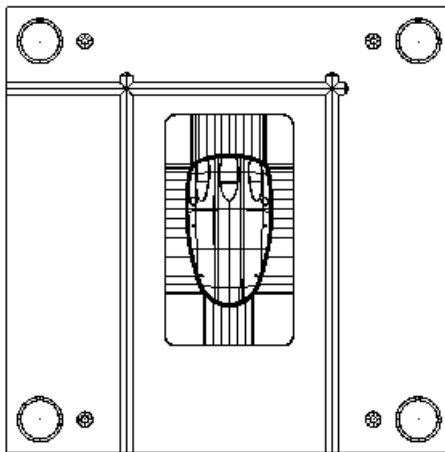


If you suppress either lock half, the corresponding lock will be ignored


Cooling channels

Creating cooling channels

You can use the **Cooling wizard** (see page 120) to create cooling channels in your mold.



Starting the Cooling Wizard

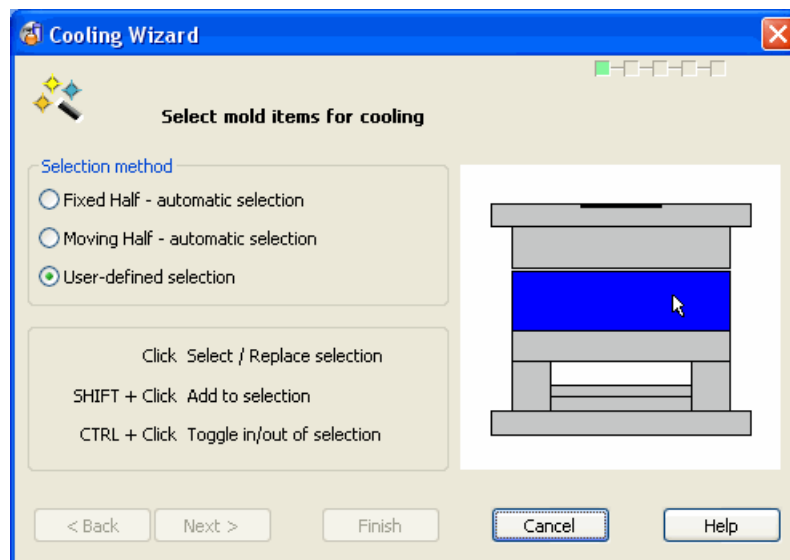
- 1 Make sure the Z axis of the current workspace runs vertically through the mold plates.
- 2 Click  (Toolmaker toolbar). The first page of the Cooling Wizard is displayed.

You can re-run the Cooling Wizard; any existing cooling holes in the selected mold components are re-read into the wizard. You can then modify them and they will be updated when the wizard closes. The layout wireframe will be re-created for the existing holes and new holes can be created.

- 3 Select the items for cooling using the **Select mold items** page of the wizard.
- 4 Set the default hole size using the **Set default hole sizes** page of the wizard.
- 5 Sketch the desired flow of the coolant using the **Sketch flow** page of the wizard.
- 6 Check the holes using the **Error checking** page.
- 7 Define the layout piece lines and error curves to be saved using the **Save options** page of the wizard.

Select mold items page

This lets you select the mold components.



Choose one of the following selection methods:

- **Fixed Half** - If you select this option, Delcam Toolmaker automatically selects all components with the string *cavity* in the name, for example, Cavity Plate, DW_CAVITY and Cavity die insert.
- **Moving Half** - If you select this option, Delcam Toolmaker automatically selects all components with the string *core* or *support* in the name, for example, Core Plate, DW_CORE, Support plate.
 - You can only select the **Fixed Half** and **Moving Half** options if the active assembly is called *Moldbase*.

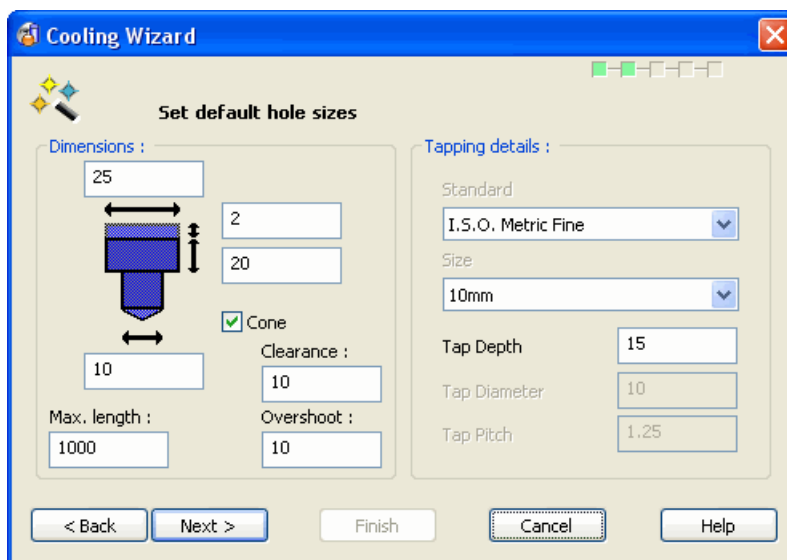
- **User-defined** - Click a component or a solid to add it to the selection. Any components or solids already selected are removed from the selection. You can add and remove components and solids from the current selection using the **Shift** and **Control** keys as shown on the wizard.

The image on the wizard changes when you click a different selection method. This shows what components will be selected.

Click **Next** to go to the Set default hole sizes page (see page 122) of the wizard.

Set default hole sizes page

This sets the default hole sizes.

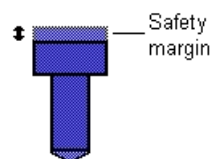


- 1 Set the dimensions of the default hole.

These values are used to create previews of the holes. You can edit individual holes later in the wizard before creating them.

Dimensions - You can tell what each value represents from the image on the wizard.

The safety margin is the length above the hole which ensures that the hole cuts the mold plate correctly when drilled.



The safety margin is added to the start of each hole automatically. If the hole cuts straight through the mold plate, the safety margin is also added to the end of the hole.

Clearance - The clearance depth to be used when creating a baffle.

Overshoot - Holes are extended by this length to ensure they intersect with each other within the mold plates. This value is not added to holes that cut straight through a mold plate.

If the distance between any end of a hole and the mold plate surface is less than the overshoot distance, then that end of the hole is extended to the surface.

Cone - This adds a cone to the tip of the hole.

Max. Length - This is the maximum length that can be drilled for a hole.

2 Set the tapping details of the default hole.

You can enter the tapping values from a standard or your own values.

Standard - Select the name of the standard. If you want enter your own values, select **User defined**.

Size - Select the size.

Tapping Depth - Input the depth of tapping required.

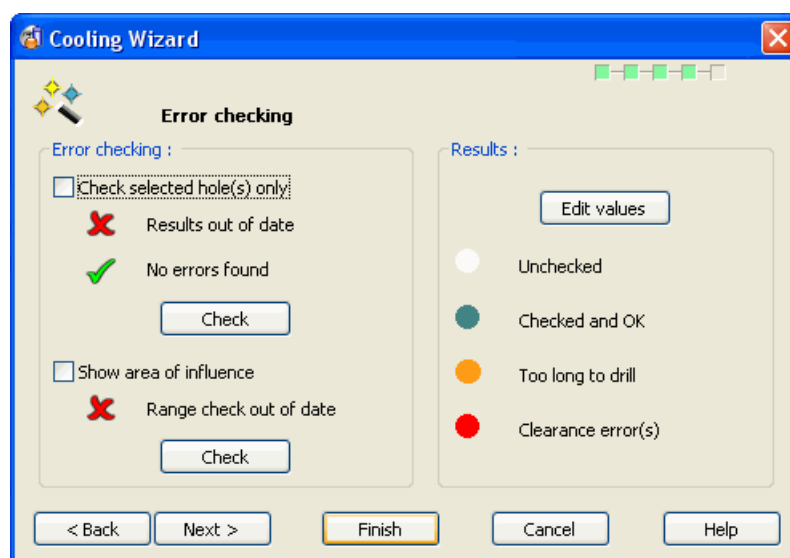
Tapping Diameter - (Only available for **User defined** tapping) Input a diameter value.

Tapping Pitch - (Only available for **User defined** tapping) Input a pitch value.

3 Click **Next** to go to the Sketch the flow page (see page 127) of the wizard.

Error checking page

This page lets you check the holes.

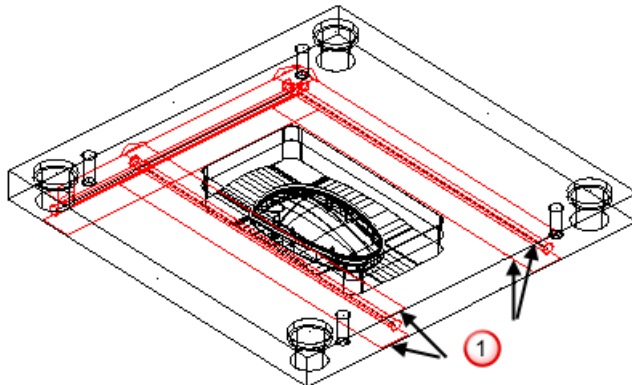


Check selected holes only - If *ON*, only selected channel holes are error checked. If *OFF*, all channel holes are checked.

Results out of date/Results not out of date - The results are out of date if any lines or holes have been edited since the last error check was done.

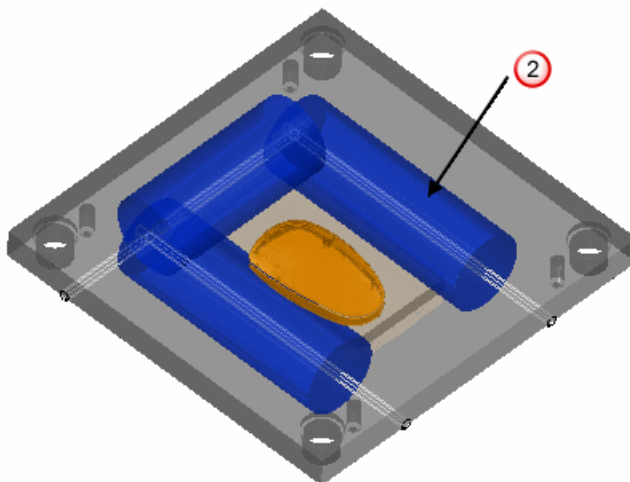
Errors found/No errors found - Indicates whether any errors have been found on the most recent check.

Check - Click this to check holes for errors. Holes are coloured red if they have errors and green if they don't. Red profiles, indicated by ① are also drawn to show where holes are too close to the part surface.



Show area of influence - If *ON*, and you have checked the range of influence, blue cylinders (using the default colour scheme) are displayed around the holes as indicated by ②. These give a rough indication of what parts of the mold plate would be cooled by the cooling holes.

If *OFF*, the blue cylinders are turned off and not displayed.



Shading is most effective in translucent shaded mode.

Range check out of date - The results are out of date if any lines or holes have been edited since the last range check was done.

Check - Click this to check the range of influence. You will only see the range of influence if you turn on **Show area of influence**.

Edit values - This displays the Clearance values dialog (see page 125), which you can use to set the distances for error checking.

Unchecked - Unchecked holes are coloured white (using the default colour scheme).

Checked and OK - Holes, which have been checked and are ok, are coloured green (using the default colour scheme).

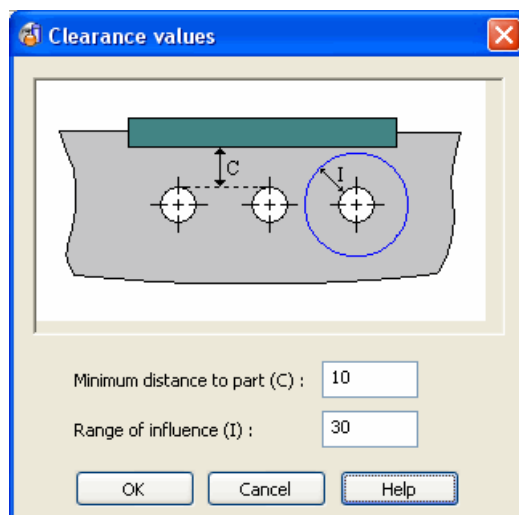
Too long to drill - Holes, which are too long to drill, are coloured orange (using the default colour scheme).

Clearance error(s) - Holes, which have clearance errors, are coloured red (using the default colour scheme).

Next - Takes you to the **Save options** page of the Cooling wizard.

Clearance values dialog

This sets the values used for checking the holes.



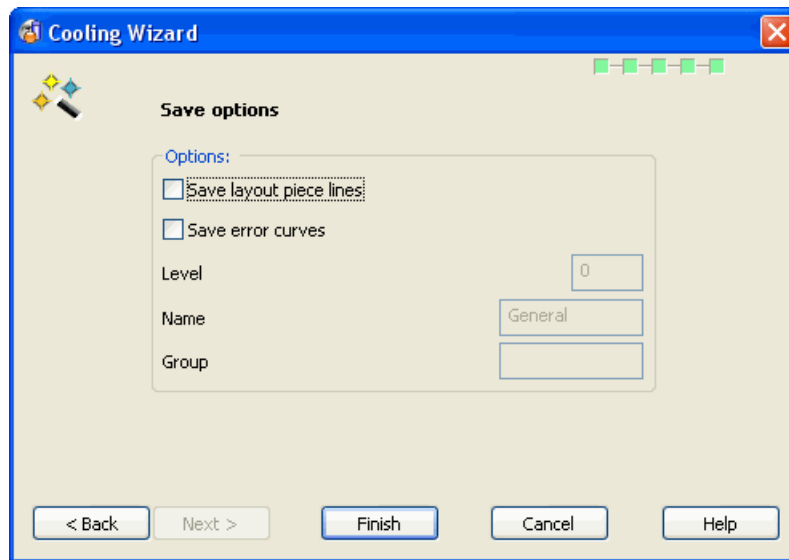
Minimum distance of part - This sets the minimum distance allowed between the hole and the part or a feature on the mold.

Range of influence - This sets the range which will be cooled by the hole.

OK - Stores the changes and removes the dialog from the screen.

Save option page

This allows you to save the layout piece lines and error curves.



Save layout piece lines - Turn this option *ON* if you want to save the layout piece lines when you exit the wizard.

Save error curves - Turn this option *ON* if you want to save the error curves when you exit the wizard.

The advantage of saving the error curves is that if the cooling hole cannot be moved and some other feature in the mold needs moving (for example, the ejector pin hole), the curves help you to remember where the problem lies.

Level - Enter the number of the level where you want to save the layout piece lines and the error curves.

Name - If the level is not named, you can enter its name here.

Group - If the level is named using the **Name** text box, you can put it in a level group.

Cancel - This exits the wizard at any time and deletes all layout pieces.

Finish - Creates the cooling holes in the mold plate. You can also click **Finish** from other pages if some layout pieces have been created.



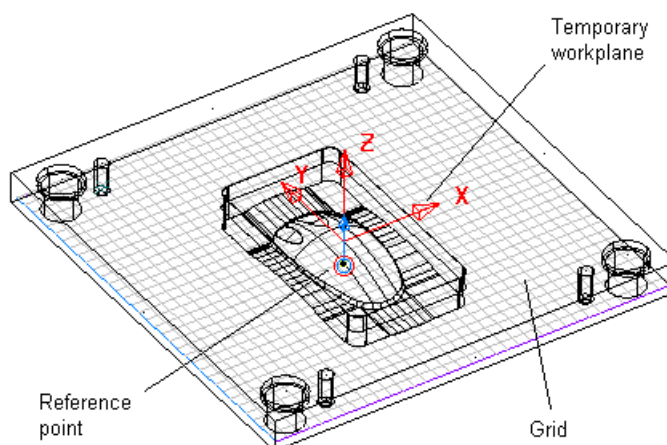
1. If you started the wizard from within Delcam Toolmaker and the mold plate contains components that are created from the same component definition, then a new component definition is created for each modified component.
2. Each component must have a definition. If the component changes, its definition must change too. If its definition is used by many components, that definition cannot change because all its other components will change too. So a new definition is created.
3. If a component is the only one created from a particular definition, no new definition is created, just the existing one is updated.

Sketch the flow page

This page of the Cooling Wizard is used to enter the layout of the cooling channels and edit it.

A grid is drawn and fitted within the mold plates. You can view the grid as either ruled or dotted lines by changing the **Grid type** option on the **View** page of the **Options** dialog.

This grid represents the plane on which the holes will lie. A temporary workplane is also created at the centre of the bounding box around the mold plates. This provides a local origin on the plane from which the holes are created.





A reference point is drawn too. By default, it touches the bottom plane of the bounding box around the mold plate and is directly below the origin of the temporary workplane.


Use the following grid controls in the **Status** bar to alter the grid.

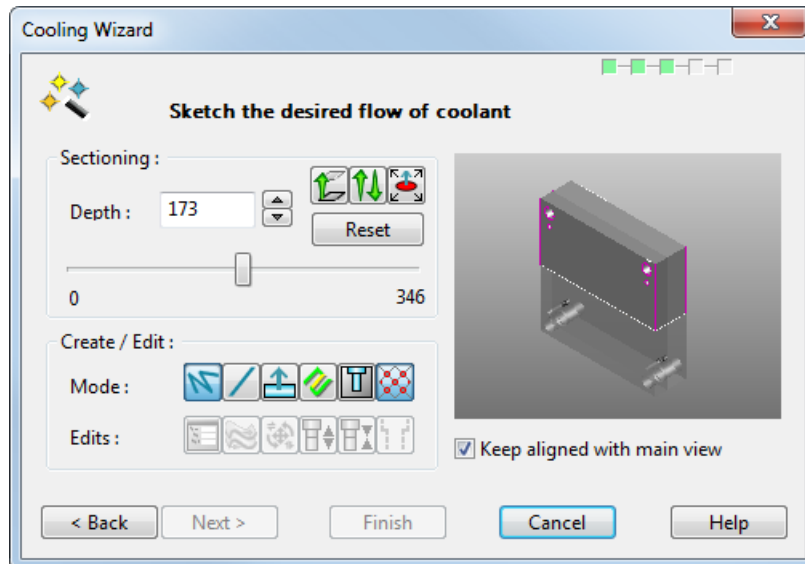


- **Grid on/off**. This draws/undraws the grid.

- **Grid Scale**. This displays the scale of the current grid.

 - Click to lock the scale of the grid. To unlock the scale, click the **Grid lock** button . You can now change the scale of the grid by zooming in and out. To increase the scale, zoom out, and to decrease it, zoom in.

To lock the scale of the grid again, click the **Grid lock**  button. To set your own grid scale, type your value into the grid scale text box. The grid scale is automatically locked to that value.



The graphic window in the dialog shows a shaded view of the selected mold plates.

There are two sections to this page of the wizard:

- **Sectioning** (see page 129)
- **Create / Edit** - The buttons in this section are used for Creating layout piece lines, (see page 129) Editing layout piece lines (see page 131) and Editing Holes (see page 132).


Keep aligned with main view - If on, the view in the wizard keeps the same orientation as the view in the graphics window. As you change the view in the graphics window, the view in the wizard changes too. You cannot change the view in the wizard.

If this option is off and you change the orientation of the view of the graphics window, the view in the wizard does not update. You can independently change the view in the wizard by zooming and panning.

Next - Takes you to the **Error checking** page of the wizard.

Sectioning

This positions the plane on which the holes are created. By default, the plane is parallel to the principal plane and lies midway through the bounding box around the selected objects.

Use the **Principal plane** buttons  on the **Status bar** to select the plane in which you want to create the holes in the mold plate. Each button displays the label of the axis which is normal to the plane. For example, the button with label Z represents the XY plane.

You can move the plane within the bounding box of the mold plate.

Depth - This is the depth of the plane. It is measured from the reference point.



- The slider changes the depth of the plane. If you move the slider, the plane moves through the mold plate. If you move the reference point along the principal axis, the values at the end of the slider change. The values on the slider are the minimum and maximum depths that the plane can move in the mold plate and are measured from the reference point.



- **Move plane to point.** Select this button and then input a position. The plane will move to that position. You can use the temporary workplane to create holes from particular points on the plane.



- **Reverse plane normal** reverses the direction of creation of the baffle.

Reset - This resets the reference point, the temporary workplane and the plane back to their default positions.



- Move the origin of the reference point. Click this button and then click a point within the selected mold components. This also moves the origin of the temporary workplane so that it is aligned with the reference point. You can move the point to a critical area and use it to measure the distance of the plane from that area.

Creating layout piece lines

Creates holes in the mold plate. Holes are positioned by creating layout piece lines.

- 1 Select either the **Continuous Line**  or the **Single**  line

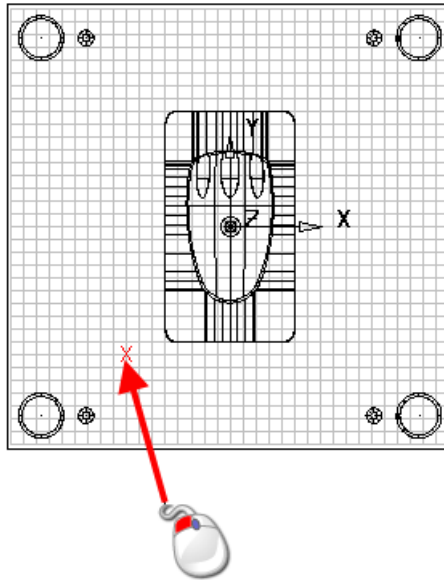


- Lets you create continuous lines on the mold plate. Continuous lines are ones where the end of the last line becomes the start of the next.



- Lets you create single lines. These lines are created between two distinct positions.

- 2 Enter a position on the grid.



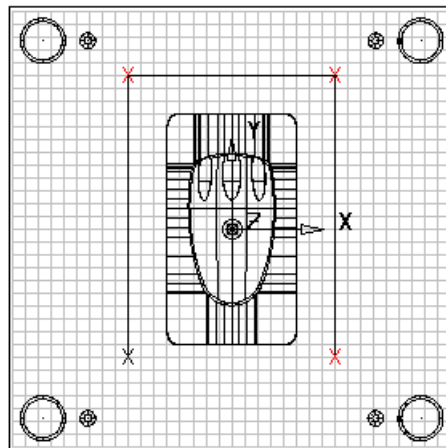
- 3 Enter all the positions for the lines. Use the following buttons on the Sketch the flow page (see page 127) to help you create the layout.



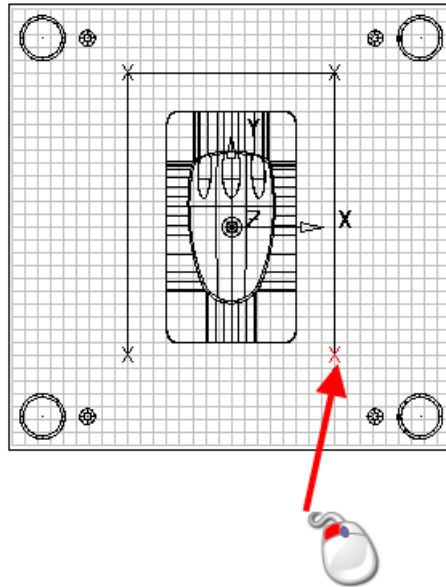
- Turns **Snap to grid** *ON* and *OFF*. This option is *ON* by default.



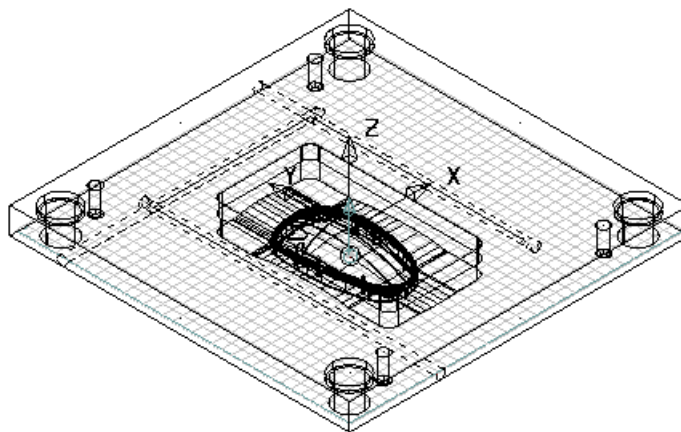
- **Show outline**. If *ON* the outlines of the hole is drawn. Shaded holes are drawn in the window of the wizard.




- 4 Create the end point by clicking the last point again. To delete a line, use the Delete key.






These layout pieces represent the layout of the cooling holes. As you enter the lines, the holes are displayed in the view on the wizard. You only need to enter the layout of the cooling circuit and not the positions of the holes. The positions of the holes are automatically calculated by the wizard.



Editing layout piece lines

- 1 Click **Edit layout** .
- 2 Select a line.
- 3 Edit the line as follows:
 - When you select a layout piece, the edit handles are displayed. You can use these to move the line.
 - The layout piece lines are edited in the same way as the lines created when modelling. For further details see Editing a line.

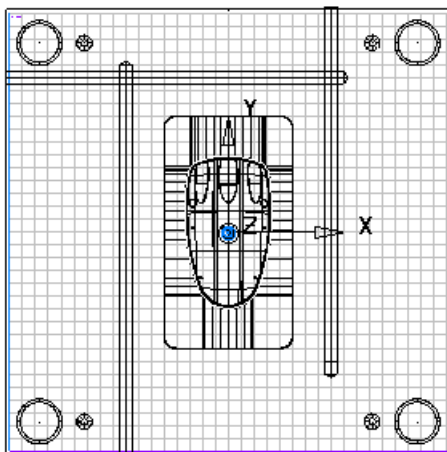
- Either click the **Edit modify** button  or double click the line to display the Line Editor dialog . Use this dialog to edit the line.
- To reverse the line, click the **Reverse** button .
- You can use the **Move, Rotate, Mirror** and **Offset** tools on the **General Edit** toolbar. Click **General Edit**  to display the General Edit toolbar. When you move, mirror or rotate a cooling hole, the hole data is copied with the layout line so for example a counterbored hole will remain counterbored.



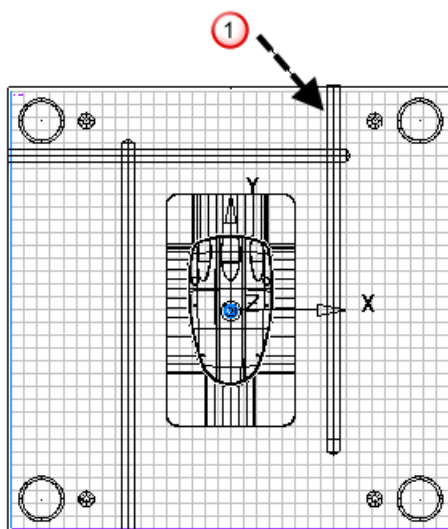
You can select lines previously created outside the cooling wizard, in addition to any lines created using the wizard.

Editing holes


- 1 To view the holes created, click **Edit holes** . This also takes you into edit hole mode. The holes are drawn on the mold plate.




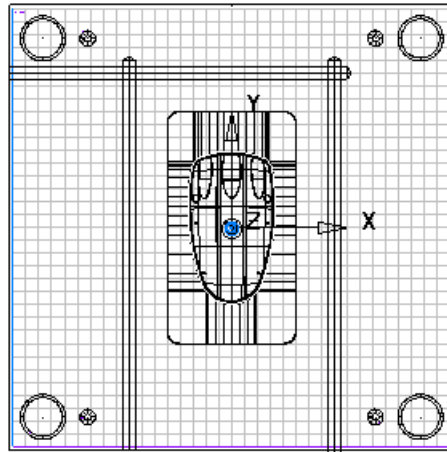
- 2 Click a hole to select it 1




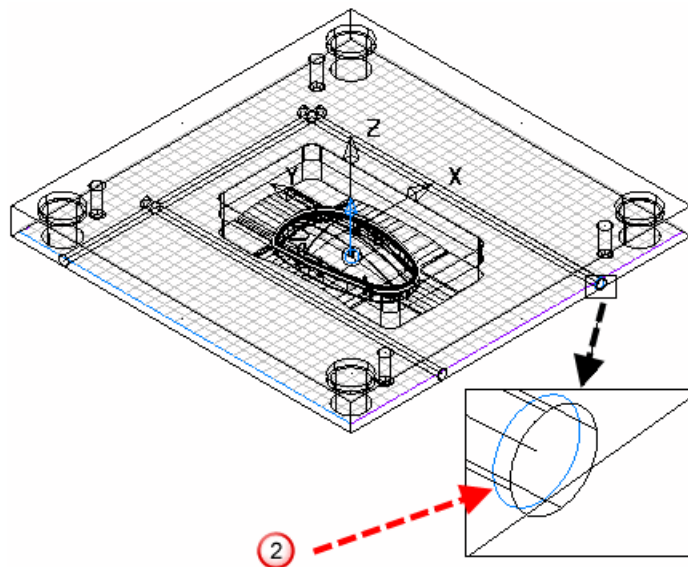
3 Edit the hole as follows:


- To reverse the drilling direction, click the **Reverse** button . This changes the drilling site to the opposite face of the mold plate.



In the example, you can see that the hole on the right should enter the plate from the bottom. You can click the **Reverse** button  to correct the direction of the hole.

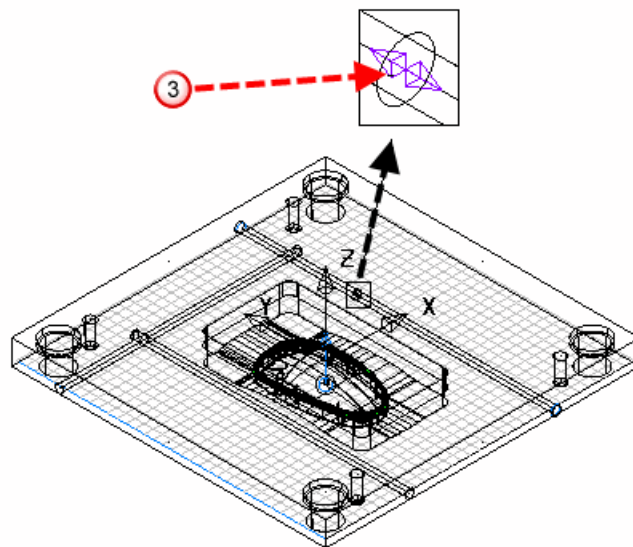



- You can change the size of the selected hole by dragging its edit handles. In the example, a drilled hole has been selected . You can edit its diameter by dragging the handles at the end of the hole.





- You can also edit one or more holes using Edit Selected Hole dialog (see page 135). To display this dialog, either click the **Edit modify** button  or double click the line at the centre of the hole.

- In edit hole mode, you cannot edit the positions of cooling holes or delete them. To edit a position of a hole or delete it, click the **Edit line** button  to go to line editing mode. The only way a cooling hole can be moved is by moving the layout piece that requires that hole. If you edit a layout piece line, its hole(s) are recreated using the default hole values and any edits you previously made to the hole(s) are lost.
- If a hole is too long to drill from one end, you can turn it into a through hole. A through hole is one which is drilled from both ends. It can be defined by two different types of holes. If you want to create a through hole, click the **Through hole** button . A handle is added where the two holes meet.

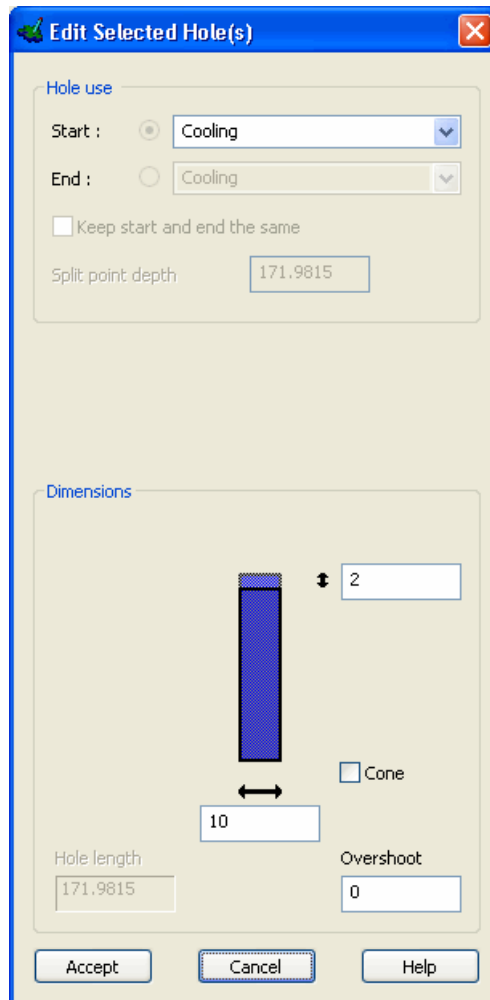


You can drag the handle  to edit the position where the two holes split.

- If you want to undo a through hole created using the **Through hole** button , click the **Non-through hole** button . If the through hole was reversed, the non-through hole will be the one before it was made into a through hole.

Edit Selected Hole dialog

This dialog defines the type of holes which are used by a cooling channel



If you are editing multiple holes, the values on the dialog are filled in for one of the holes in the selection list. If you change a value which is not valid for a particular hole, that hole will not update but the others will.

Hole type - Selects the type of hole used in the cooling channel. If the cooling channel is a through hole, you define the start and end holes by selecting **Start** and **End** options.

Start - This defines the type of hole for the start hole. Select the type of hole. The dimensions for the hole will display on the dialog.

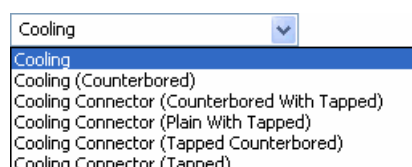
If the cooling channel is not a through hole, the **Start** option defines the hole.

To set the dimensions for the start hole, make sure the toggle button on the left is selected. The dimensions for the hole will display on the dialog.

End - (Only available for through holes) This defines the type of hole for the end of the through hole. Select the type of hole. The dimensions for the hole will display on the dialog.

To set the dimensions for the end hole, select the toggle button on the left. The dimensions for the hole will display on the dialog.

The Cooling Wizard uses all available, appropriate solid hole types, allowing you to define drill, tapping, tapping pre-drill and counterbore diameters and depths.



Keep start and end the same - (Only available for through holes) If this option is on and you change a dimension of any hole, the same dimension of the other hole automatically change too.

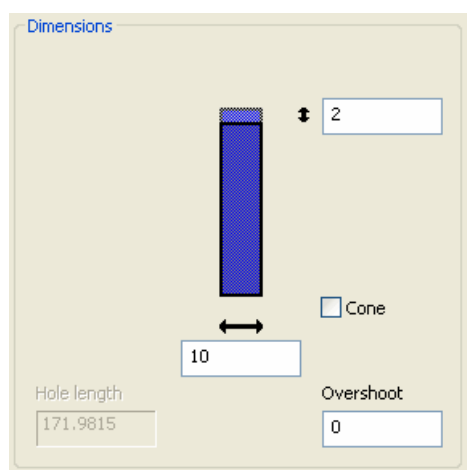
If this option is off, the changes will only apply to the end that is currently selected.

Split point depth - (Only available for through holes) This is the position along the start hole where the end hole joins it.

Dimensions - For hole type, an image of a hole is given with boxes to change particular dimensions.

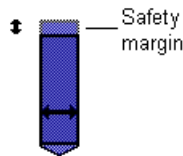
For each type of hole, you can set different dimensions. The explanation is very similar for each hole. Therefore, a general description is given below.

If you want to set the dimensions for the start hole, make sure the toggle next to **Start** is selected. Similarly, if you want to set the dimensions for the end hole, make sure the toggle next to **End** is selected.



You can change the size of the hole by entering values in the dimension boxes. For tapped holes, the sizes can also be changed by selecting different **Standard** and **Size** options.

The safety margin is the length above the hole which ensures that the hole cuts the mold plate correctly when drilled.



The safety margin is added to the start of each hole automatically. If the hole cuts straight through the mold plate, the safety margin is also added to the end of the hole.

Overshoot - Holes are extended by this length to ensure they intersect with each other within the mold plates. This value is not added to holes that cut straight through a mold plate.

Cone - This adds a cone to the tip of the hole.

Hole length - This is the length of the hole. You cannot edit this length here. To change it, you must change the length of the layout piece line.

Tapping details - These set the tapping details of the hole. You can set the standard and size of the hole using these options.

Standard - Select the name of the standard. If you want enter your own values, select **User defined**.


Size - Select the size.


Tapping Diameter - (Only available for **User defined** tapping) Input a diameter value.

Tapping Pitch - (Only available for **User defined** tapping) Input a pitch value.

OK - Modifies the hole as required.

Creating a baffle


- 1 Position the gridding plane.
- 2 Click the **Create baffle layout**  button.
- 3 Click on the grid to define where the baffle is to be created.

The baffle hole is created and the layout line representing the centre line of the baffle is displayed. You can see what the baffle looks like by selecting the **Edit Hole**  button.

- 4 To edit the hole, double-click on the hole to display the **Edit Selected Hole** dialog. You can change the dimensions and the **Clearance** can be adjusted in this dialog as well as selecting a different type of baffle. The following types of baffle are available

Cooling Baffle
Cooling Baffle (Counterbored)
Cooling Connector Baffle (Counterbored With Tapped)
Cooling Connector Baffle (Plain With Tapped)
Cooling Connector Baffle (Tapped Counterbored)
Cooling Connector Baffle (Tapped)

The clearance on cooling baffles is now recalculated if the position of the baffles has been edited.

- 5 You can see the outline of the baffle by selecting the **Show outline**  button.



If the baffle you created goes through multiple mold items and you delete the baffle in the top mold item, the baffle will be recreated in the remaining mold item(s). Any clearance is applied to the recreated baffle.

Power Features in Delcam Toolmaker

Introduction

You can use the **Power Feature Trimming** to:

- Trim ejector pins and ejector sleeves to their correct sizes.
- Generate trimming for pins in both sub and main assemblies. This feature set can then be patterned across the multiple cavities.

Automatically adding clearances and fits to components

Delcam Toolmaker uses Power Features to add fits and clearances into your model.

Parametric fit and clearance features are automatically defined for major mold components in all component catalogues. Parametric components that are available include pins, sleeves, blades, screws, pillars, bushes, location rings and dowels. You have complete control over when and how fit and clearance features are applied. For example, you can make changes to the size or position of multiple components before automatically updating the Power Features.

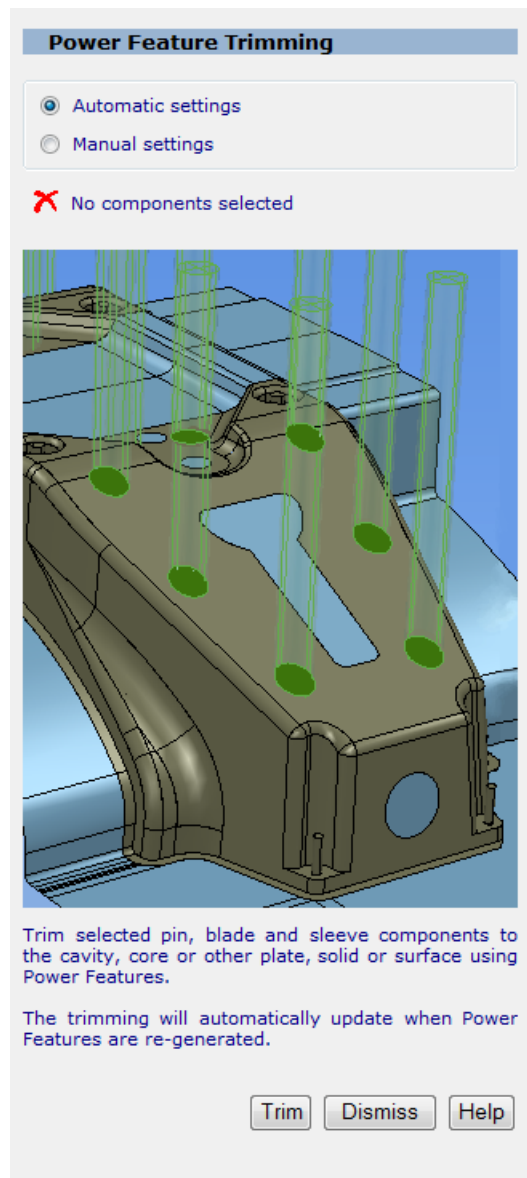
Ejector pins, sleeves and blades (and also core and return pins) may be trimmed to the cavity impression. When Power Features are updated, a component will automatically retrim if required. If you wish to retrim a component without updating all of its Power Features, re-run **Power Feature Trimming**.

For further details, see What is a Power Feature?

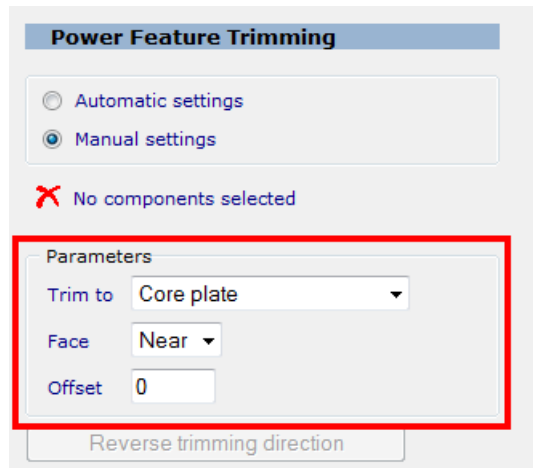
Power Feature Trimming

- 1 Click  (*Toolmaker toolbar*).

The **Power Feature Trimming** page is displayed.



- 2 If you select **Manual settings**, the **Parameters** section is displayed. Use this section to specify the parameters to be used for trimming the pins.

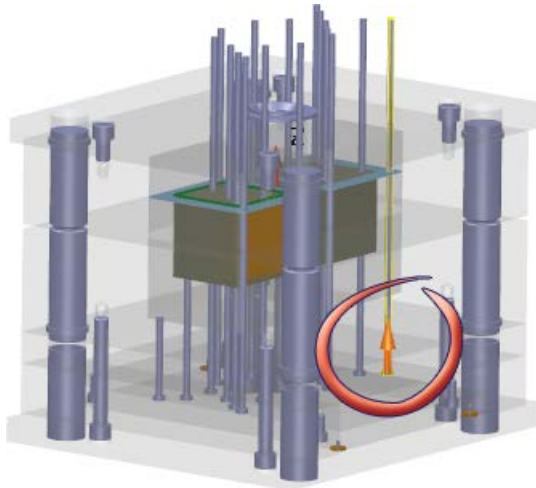


The **Parameters** section contains two drop-down menus:

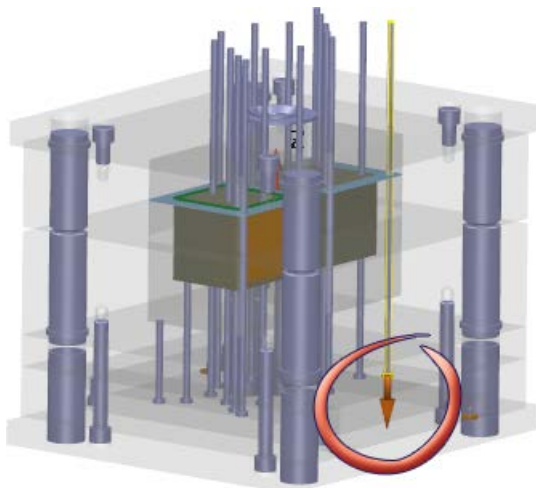
- **Trim to** defines the plate and contains these options:
 - Core plate** (available when core pin, ejector sleeve or ejector pin components are selected)
 - Cavity plate** (available when core pin or return pin components are selected)
 - Selected component**
 - Selected solid / surfaces**
- **Face** defines the face of the plate to trim to and contains these options:
 - Near**
 - Far**

If you select components with different functions (for example, an ejector pin and a return pin), the **Trim to** drop-down list will contain all options (Core plate, Cavity plate, Selected plate, Selected solid /surfaces). When you click **Trim**, a dialog will ask you to confirm if you want to apply trimming to all selected pins.

Reverse trimming direction lets you change the trimming direction for all selected components in the model. The trimming direction for individual selected components is indicated by an arrow.



You can change the trim direction for a component by clicking the arrow.



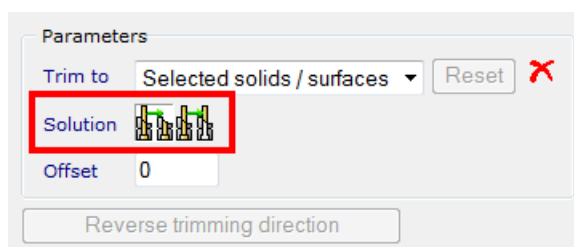
- 3 When you have specified the trim parameters, click **Trim** to apply the changes when you re-generate the Power Feature.



*If you are unhappy with the trimming operation, click **Undo** (Main toolbar) to undo the previously applied trimming operations.*

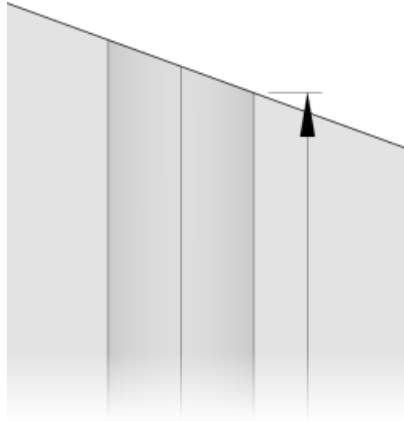


*When trimming to a surface, the **Solution** buttons ensure that the correct part of the pin is left in the model.*





When using ejector pin trimming, the depth of the fit hole is measured from the lowest point of the trimmed region of the ejector pin. This allows holes to have sufficient depth when they are trimmed to a steep 3D face.



Power Feature Trimming - choosing the correct solution

Mold simulator

You can simulate opening and closing of a mold tool using Delcam Toolmaker. For details see Opening and closing a mold (see page 142)

Opening and closing a mold

To open and close the mold,

- 1 Click  (Toolmaker toolbar).

- 2 Use the options on the **Mold tool opening** dialog to define the way the mold opens and closes.

Mold tool opening

Maximal distances

Press opening	226	Ejection stroke	27
Daylight	55		
Mold height	171		

Automatic

Number of steps	20	Step time, ms	100
Step	2.8		

Start Stop

Manual

Moving half

Ejection plates

Explode the components in Z

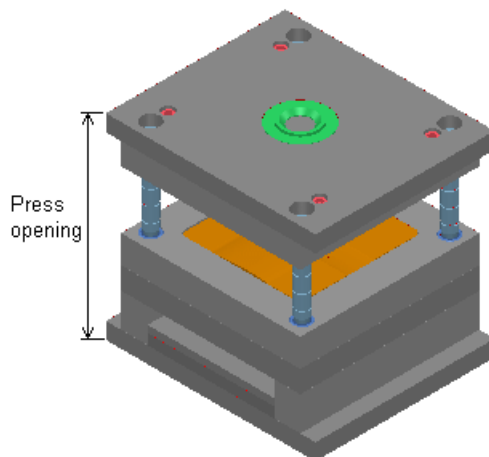
Distance

☐ Plates only

☐ Keep in open position Dismiss

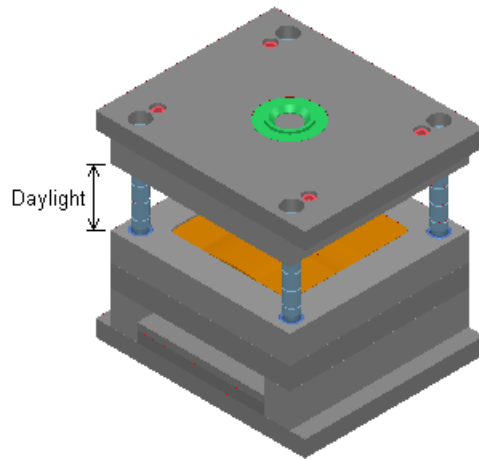
Maximal distances - Set the values for maximal distances. These are used to simulate the opening and closing of the mold tool.

Press opening - Maximum available space in which the mold can open.



The maximum daylight distance is calculated from this value. If you change this value, the maximum daylight distance changes too.

Daylight - Maximum opening between the core and cavity plates.



The part falls out of the mold from this opening. The opening is measured by subtracting the maximum press opening distance from the mold height. If you change this value, the maximum press opening distance changes too.

Ejector stroke - Maximum distance that the ejector plate will move.

mold height - Height of the mold.

Automatic - Sets the values to automatically open and close the mold tool.

Number of steps - Number of steps to take to reach the maximum daylight distance. If you change this, the step size also changes.

Step - Size of each step. If you adjust this value, **Number of steps** will update too.

Step time - Time for each step movement.

Start - Begins opening and closing the mold tool.

Stop - Stops the automatic opening and closing of the mold tool.

Manual opening - Sets the distances to manually open the mold tool.

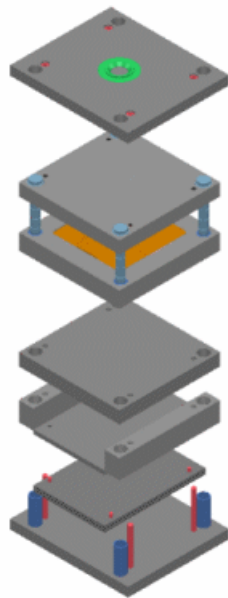
Moving half - The moving half is opened by the given distance. Input a value by either using the slider or entering a value. The maximum distance is set by the daylight distance.

Ejector plates - The ejector plates are moved to the given distance. Input a value by either using the slider or entering a value. The maximum distance is set by the daylight distance.

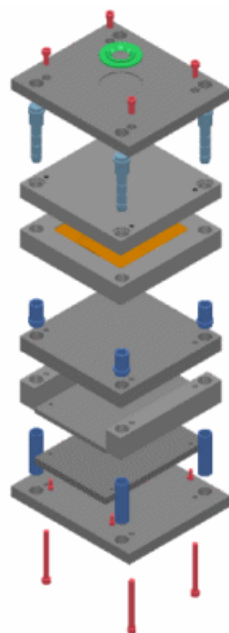
Explode the components in Z - The components of the mold tool are positioned from each other by the set distance.

Distance - Sets the distance.


Plates only - If *ON*, the plates are put in new positions and all the other components stay attached to their corresponding plates.



If *OFF*, all components are put in new positions.



Keep in open position - When you click **Dismiss**, the mold is left open with whatever distances were currently set. This is useful for producing drawings on open mold bases.

- 3 To close the mold again after leaving **Mold tool opening**, return to this page by clicking 

Bill of Material (BOM)

Creating a bill of material (BOM)

The bill of material is a list of components/sub-assemblies used by an assembly. Even more helpful is that the list not only displays the components/sub-assemblies but also some of their characteristics.

The bill of material can be output as:

- a drawing window
- a file
- an embedded Microsoft Excel spreadsheet

For further details on embedded bill of materials, see [Creating an embedded Bill of Materials](#).

Catalogs

Select components from supplier component catalogs (see page 146).

Use custom catalogs (see page 147) to:

- Group together frequently used components.
- Rename standard components
- Customise components

Component catalogs

Product specific catalogs are supported, so different component libraries are available with, for example, Delcam Designer, Delcam Toolmaker and Delcam Crispin SoleDesign.

Delcam Toolmaker supports mold components from the following catalog suppliers:

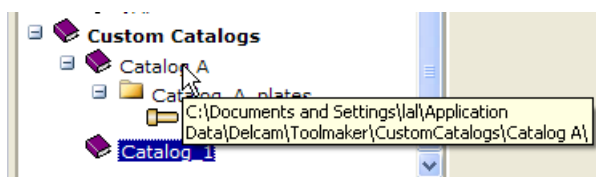
Crusteel	Danly	DME - Europe (both Euro and D-M-E Standards)
DME - North America	DMS	Futaba
Hales	Hasco	Kishin

LKM	Meusburger	MISUMI
PCS	Pedrotti	Polimold
Progressive	Rabourdin	VAP
Sideco	Strack	Unbrako (metric and imperial)

Locating and sharing custom catalogs

To locate a custom catalog:

- 1 Hover over the name of the catalog in **Custom Catalogs** to view the tooltip showing the path to the catalog.



- 2 Use **Browse to an existing custom catalog** (see page 159) to locate the required catalog.

To share a custom catalog:


- 1 Locate the required custom catalog, using **Browse to an existing custom catalog** (see page 159).
- 2 Using standard copy and paste techniques, copy the required custom catalog folder from the user's computer and paste into the Custom Catalogs folder on your computer.

Custom catalogs


A custom catalog lets you:

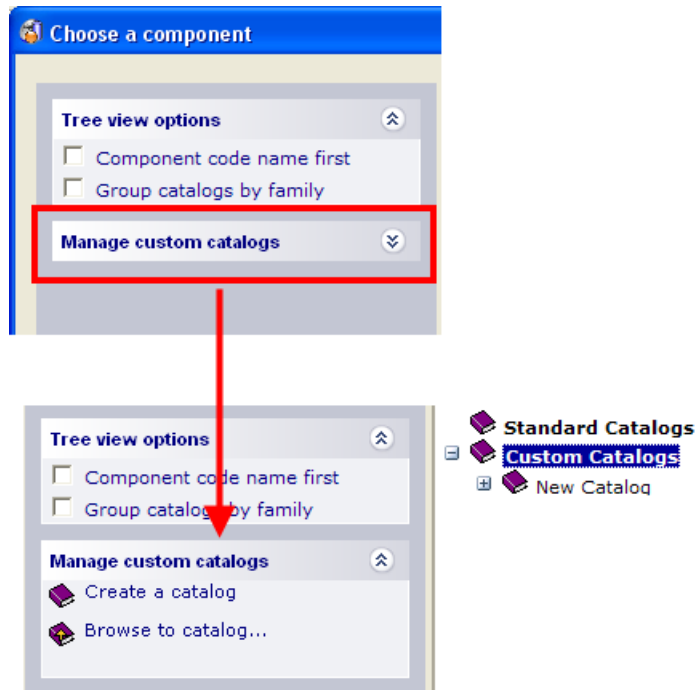
- Group together frequently used components. A custom catalog may contain components from several different manufacturers.
- Rename standard components. The underlying geometry of the component and link to the manufacturer is unaffected; the component inherits any change to the manufacturer catalog component.
- Modify components with your own dimensions, order codes and names. These components can then be used in the Component Wizard.



Before working with custom catalogs, click  to close the Tree view options on the Component Wizard dialog. This will minimise scrolling.

To manage custom catalogs:

- 1 Select **Module > Toolmaker** to enter Toolmaker mode.
- 2 Click **Component Wizard**  from the Assembly toolbar.
- 3 Click **Manage custom catalogs** to display the options.



- 4 Use the options to:
 - Browse to an existing catalog (see page 159).
 - Create a new custom catalog (see page 148).

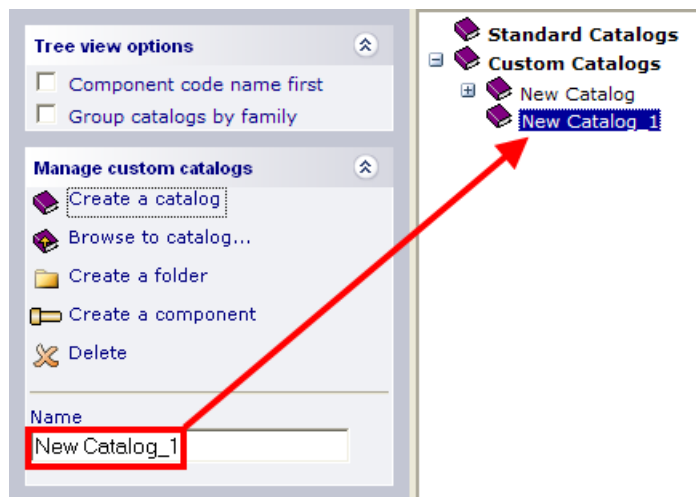
Creating a new custom catalog

- 1 Click **Create a catalog**.

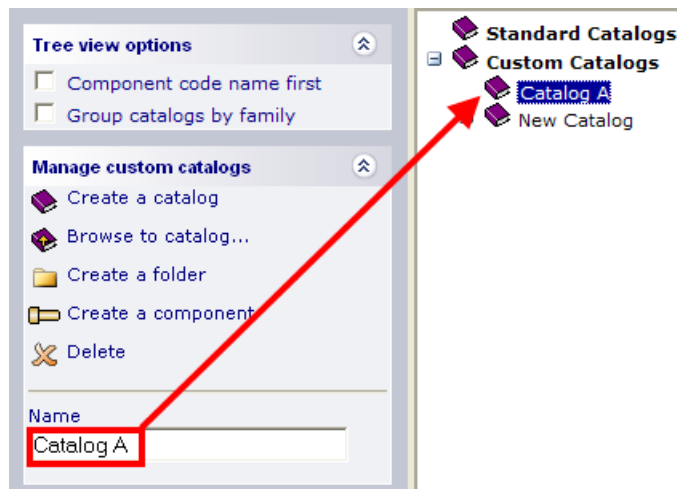
Additional options are displayed to let you:

- Create a folder (see page 149).
- Create a component (see page 150).
- Delete a catalog (see page 157).

- Name the new catalog or folder.



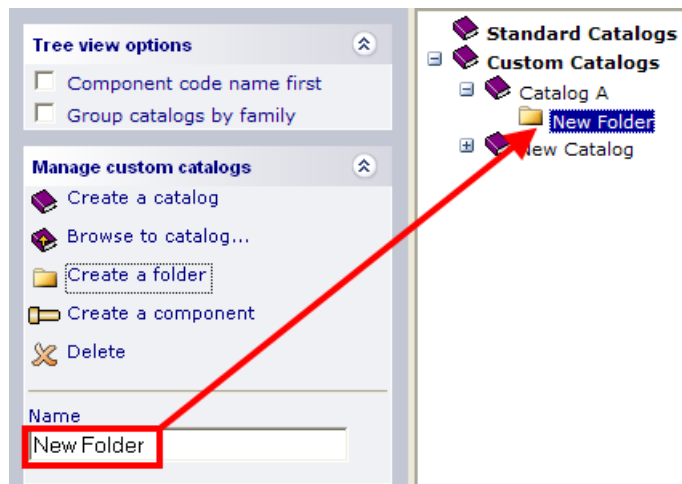
- 2 Enter a **Name** for the new catalog, for example **Catalog_A**. If you don't enter a name, the catalog uses the default name that was allocated when the catalog was created.



Custom catalog - Create a folder

- 1 With **Component Wizard** running, click **Manage custom catalogs** to display the options.

- 2 Click **Create a folder** to create a new folder to hold particular types of components.



- 3 Enter a **Name** for the folder. If you don't enter a name, the folder uses the default name that was allocated when the folder was created.

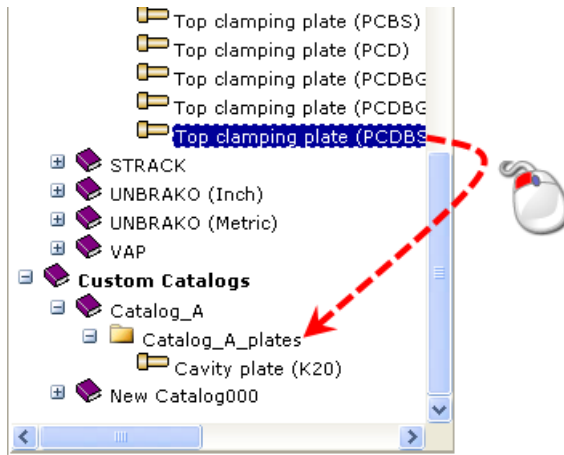
Custom catalog - Create a custom component

Use the following sections to create and modify custom components:

- Adding a component (see page 150).
- Adding user-modelled components (see page 153)
- Modifying component dimensions (see page 155)

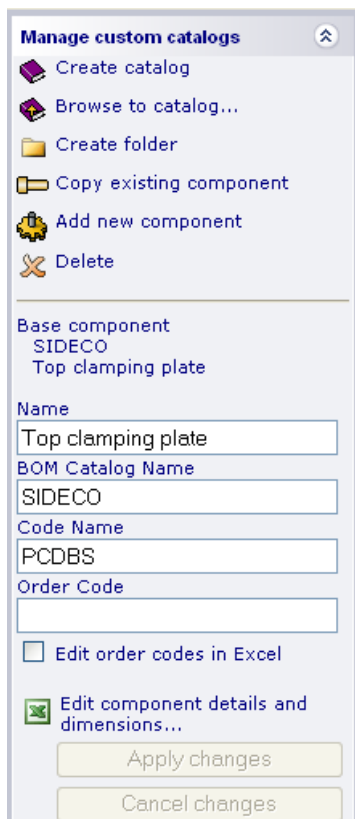
Custom catalog - Adding a component

- 1 With **Component Wizard** running, click **Manage custom catalogs** to display the options.
- 2 Locate the appropriate destination folder in the **Custom Catalogs**.
- 3 Select a component from a standard catalog, or an existing custom catalog.
- 4 Drag the selected component into the custom catalog as shown below:



When a custom component is dragged from one custom catalog to another, the component is moved. To copy a component and leave the original untouched, hold down the CTRL key and drag the component.

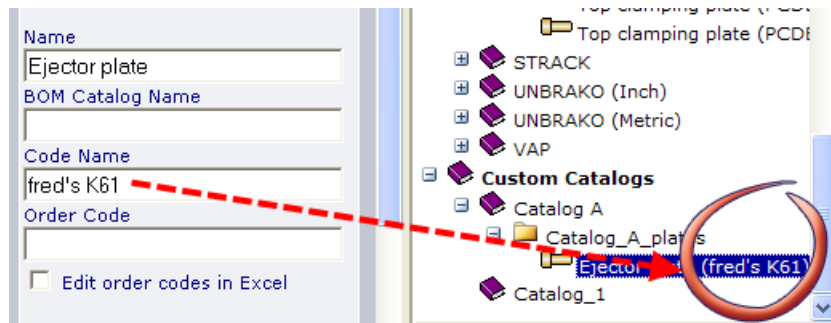
The details of the component are displayed under **Manage custom catalogs**.



*You can also select **Copy an existing component** and use the **Destination folder** and **Source component** options to create the component.*

- 5 Enter a **BOM Catalog Name** to be displayed for the custom component in the Bill of Materials.

- 6 Modify the **Code Name** as required. Any change you make appears in the **Custom Catalogs** tree.



- 7 Leave **Order Code** blank to display the default order code.

Alternatively, customise the **Order Code** in one of the following ways:

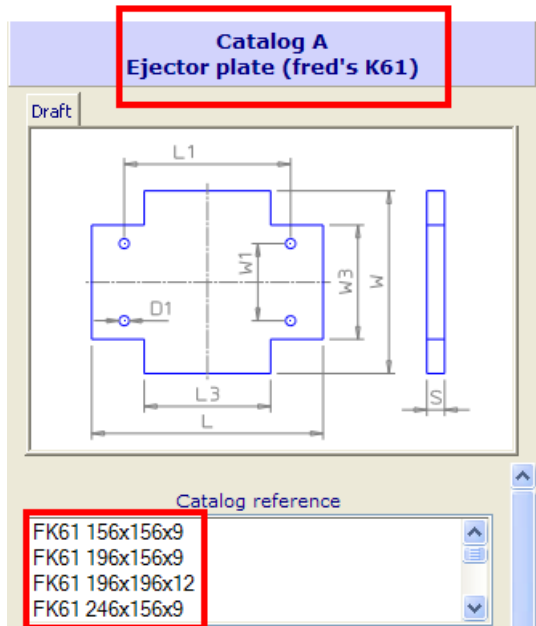
- Add a fixed **Code Name**. If you added **FK61**, all sizes of that component would have *FK61* as the order code.
- Add a combination of a fixed and variable code. The variable part of the code uses the dimension parameters to create a unique code for each row.

For example: **FK61 %L% x %M% x %S%** would produce the following order codes.

FK61 95x95x6
FK61 130x100x6
FK61 156x156x9
FK61 196x156x9
FK61 246x156x9
FK61 346x196x12
FK61 346x346x17
FK61 396x346x17
FK61 496x346x17
FK61 396x396x17
FK61 546x396x17

- Select **Edit order codes in Excel** to edit the order code from within a spreadsheet (see page 156).
- 8 Click **OK** to close the **Choose a component** dialog. The Component Wizard is opened in the normal way.

- 9 Click **Select** from the **Dimensions** options in the Component Wizard. The customised catalog and order codes are displayed.

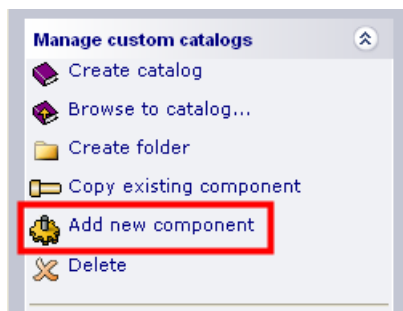


Custom component FK61 is still linked to the original manufacturer's catalog. Any changes that are made by the manufacturer (for example a change in the sizes of the plates) will be automatically inherited by the FK61

Adding user-modelled components

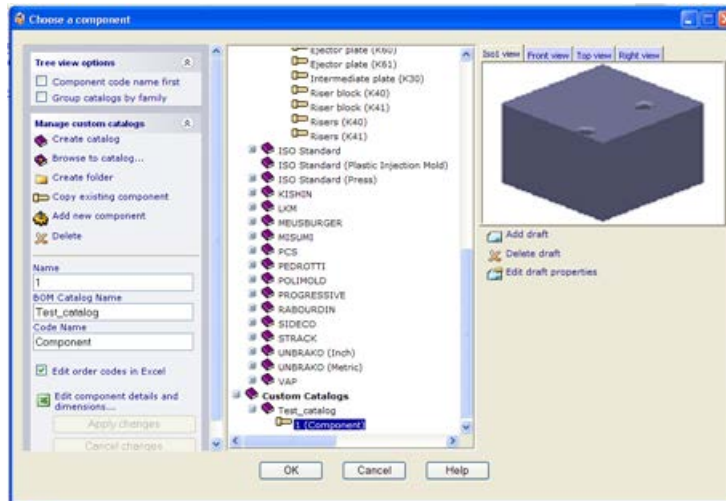
With the source component displayed:

- 1 Click **Add new component**.

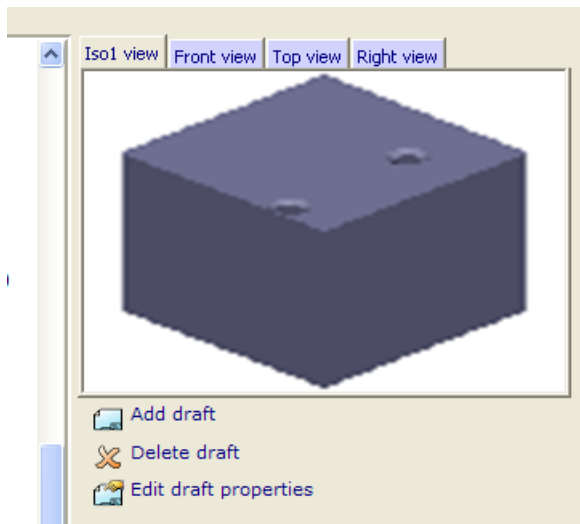


- 2 Select the following:
- Source component on the model.
 - **Destination folder**.
- 3 Click **Use selected**.


- 4 Click **Add** to add the component to the custom catalog. The dialog updates to display the component:





- 5 Click **Edit component details and dimensions** to display the MS Excel spreadsheet for the component.
- 6 Change the name and dimensions in the spreadsheet as required. Switch back to PowerSHAPE.
- 7 Click **Apply changes** to:
- Update the **Custom Catalogs** tree to reflect any component name change.
 - Close the spreadsheet.
 - Display the component in the graphic window on the dialog.



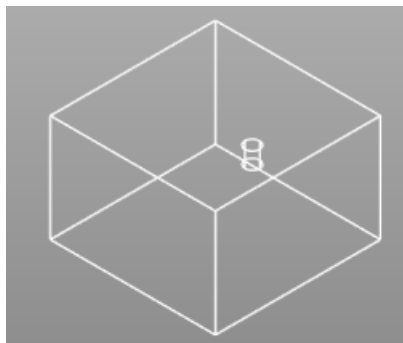
- 8 Use the following options as required:

- Click  to **Add draft**. Choose the required graphic from the **Select a Draft** dialog. The selected graphic is added to a **Draft** tab.

- To delete a tab, select the tab and click  to **Delete draft**.
 - To change the title of a tab, select the tab and click  **Edit draft properties**. Enter the new tab **Title** and click **OK** (just below the **Title**) to change the tab.
- 9** Add further user-defined components as required.
- 10** When all components have been added to the custom catalog, click **OK** to finish.

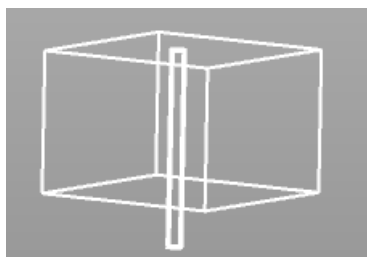
Preview of a user-modelled component

Any user-modelled custom component that is added to a custom catalog will display a preview when it is selected from the catalog.



The preview reflects the component geometry that is stored in the solid tree.

The following example shows the preview that is generated if you have created a custom component where the hole depth is greater than the dimension of the block.




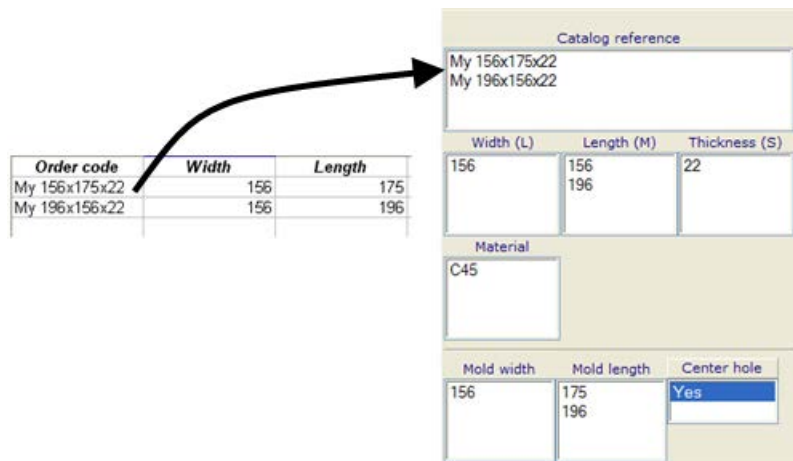
If required, you can modify the geometry of the component and create a new component from the modified geometry.

Custom catalogs - Modifying component dimensions

To modify the dimensions of a custom component:

- 1** Select the component from the custom catalog.
- 2** The cells that contain the parameter short names and order codes in the **Dimensions** worksheet will be protected by default. Select **Edit order codes in Excel** to allow editing of these cells.

- 3 Click  to open a MS-Excel spreadsheet containing the component details and dimensions. Using MS-Excel to edit the component details and dimensions breaks the link to the manufacturer's catalog (see page 156).
- 4 Make the required changes to the data on the **Dimensions** worksheet and return to **Choose a component** dialog.
- 5 Click **Accept changes**.
- 6 Click **OK** to close the dialog and return to the **Component Wizard**. Any changes that you have made to the dimensions will be reflected in the **Dimensions** options as shown below:



Custom catalogs - Linked and independent custom components

Linked custom components

By default, custom components have an underlying link to the manufacturer's catalog. You can edit any of the details on the **Component Wizard** and the link will be maintained. This means that if the underlying standard component is updated in a later version of PowerSHAPE, the custom component will be updated as well.

Independent custom components

The **Component Wizard** lets you use MS-Excel to:

- edit the order codes (by selecting **Edit order codes in Excel**).
- edit dimensions.

In both cases, the custom component becomes independent of the manufacturer's catalog and will be unaffected by any update to the standard component.

Custom catalogs - Delete a catalog, folder or component

- 1 With **Component Wizard** running, click **Manage custom catalogs** to display the options.
- 2 Select the catalog, folder or component to delete.
- 3 Select **Delete**.

Custom catalogs - Grouping

You can change the following:

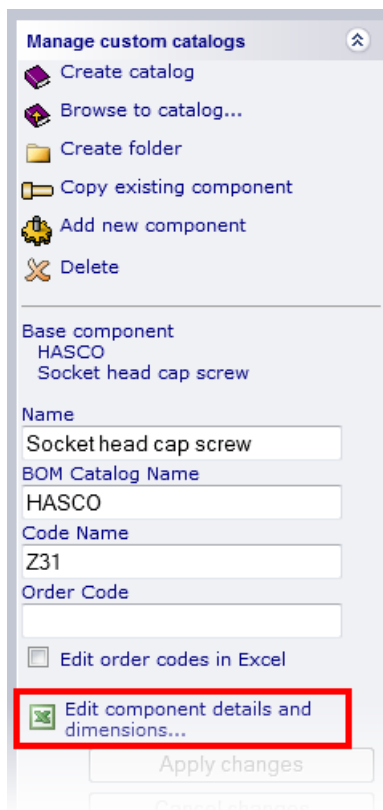
- Grouping of directions.
- Order in which dimensions and groups are displayed.

Example

- 1 Create a custom catalog called **XXXX** that contains the **Hasco Socket head cap screw (Z30)**.



- 2 Click **Edit component details and dimensions**.



- 3 MS-Excel will open automatically on your task toolbar.

- 4 Swap to MS-Excel to display the spreadsheet that has been created automatically.

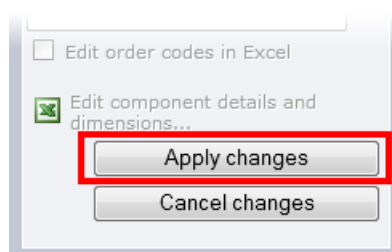
	A	B	C
1	Name	Socket head cap screw	
2	Code name	Z31	
3			
4			
5	Dimensions		
6	Short name	Name	Group
7	<i>d</i>	Diameter	
8	<i>l</i>	Length	
9	<i>dk</i>	Head diameter	More Parameters
10	<i>k</i>	Head thickness	More Parameters
11	<i>s</i>	Socket width	More Parameters

- 5 Enter the following in the spreadsheet:

- **Main Parameters** into the **Group** cells for **d** and **l**
- **Head Parameters** into the **Group** cells for **dk** and **k**

	A	B	C
1	Name	Socket head cap screw	
2	Code name	Z31	
3			
4			
5	Dimensions		
6	Short name	Name	Group
7	<i>d</i>	Diameter	Main Parameters
8	<i>l</i>	Length	Main Parameters
9	<i>dk</i>	Head diameter	Head Parameters
10	<i>k</i>	Head thickness	Head Parameters
11	<i>s</i>	Socket width	More Parameters

- 6 Save the changes to the spreadsheet
- 7 Swap back to PowerSHAPE and click **Apply changes**.



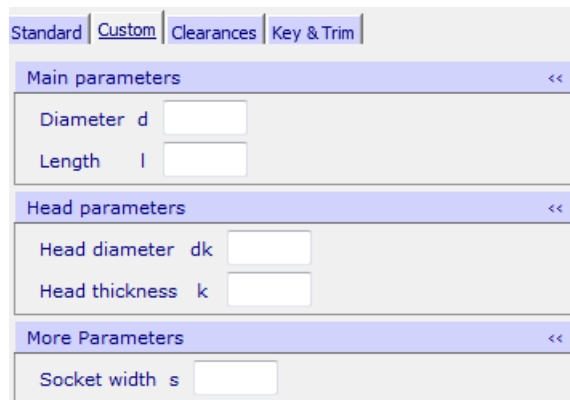
The component parameters are grouped as follows:

- Dimensions that have the same group name are combined into one group.
- If a group with the given name does not exist, a new group is created.
- Dimensions with no group name are included in the main group or parameters.

- 8 Double-click the component in the custom catalog tree to open the component in the **Component Wizard** window.
- 9 Click the **Custom** tab. **Main parameters** and **Head parameters** groups are displayed.



- 10 Click the groups to display the parameters.



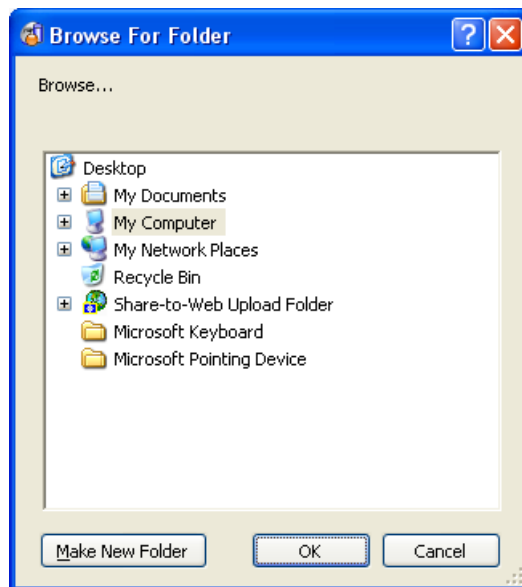
This technique is useful for a plate; you can group parameters relating to fixing screws separately from those relating to pillar holes.

Browse to an existing custom catalog

Browse to an existing custom catalog on your computer or another computer on the network.

- 1 With **Component Wizard** running, click **Manage custom catalogs** to display the options.

- 2 Click **Browse to catalog ...** to display the **Browse For Folder** dialog.



- 3 Navigate to the folder containing the required catalog.

In Microsoft XP, the default location for custom catalogs is one of the following:

C:\Documents and Settings\\Application
Data\Delcam\Toolmaker\CustomCatalogs

C:\Documents and Settings\All Users\Application
Data\Delcam\Toolmaker\CustomCatalogs

In Microsoft Vista, the custom catalog data is kept in one of the following locations:

C:\Users\name>\AppData\Roaming\Delcam\Toolmaker\CustomCatalogs
C:\ProgramData\Delcam\Toolmaker\CustomCatalogs

Drafting for Delcam Toolmaker

Drafting

Delcam Draft can produce your General Assembly, Detail drawings and Bill of Materials. You can find information on these as follows:

General Assembly drawings - For further details, see Creating a drawing

Detail drawings - For further details, see Creating component drawings

Bill of Materials - For further details, see Creating an embedded Bill of Materials.

Customising Delcam Toolmaker

You can change some Toolmaker options by changing values in the *GlobalOptions.xml* configuration file. Most of the options that are stored in the file are managed automatically from within PowerSHAPE and should not be changed manually. You can define options for the following within the configuration file:

- Bounding box to be used by Cavity Core Wizard.
- Action to be taken if guide pillars or bushes are moved.
- Preview colour for selected component in the Moldbase Wizard.

When PowerSHAPE is installed in the default location, the configuration file can be found in one of the following locations:

For XP installation, the path is `c:\Documents and Settings\All Users\psmm`

For Vista installations, the path is : `c:\ProgramData\psmm`

This configuration file is in XML format and can be edited using an XML editor (recommended) or a text editor.

Configuration options are grouped into sections within the file as follows:

<GlobalOptions>

<Section1Name>

<Option1Name

OptionType="string|number|boolean">some_value</Option1Name
>

<Option2Name

OptionType="string|number|boolean">some_value</Option2Name
>

</Section1Name>

<Section2Name>

<Option1Name

OptionType="string|number|boolean">some_value</Option1Name
>

<Option2Name

OptionType="string|number|boolean">some_value</Option2Name
>

</Section2Name>

</GlobalOptions>

The default options file is shown below. The options that are included in the default file are set automatically from within PowerSHAPE and should not be modified.

<GlobalOptions>

<SelectComponentDlg>

<ComponentCodeNameFirst

OptionType="string">0</ComponentCodeNameFirst>

<GroupCatalogsByFamily

OptionType="string">0</GroupCatalogsByFamily>

<ExpandedItems OptionType="string">DMS

DMS__5</ExpandedItems>

<SelectedItem

OptionType="string">DMS_GB1000GuideBush</SelectedItem>

</SelectComponentDlg>

</GlobalOptions>

You can add the sections to the options file. Although changes to the file are retained when a new version of PowerSHAPE is installed, it is recommended that you keep a 'safe' copy of the updated file. The following sections can be added to the configuration file:

<CCWizard>

<ForceActualBBoxForDWObjects

OptionType="string">0</ForceActualBBoxForDWObjects>

</CCWizard>

0 - When calculating the bounding boxes of plate/inserts solids, use the bounding box of the base primitive solid from the solid tree. This is the default setting.

1 - Calculate the actual bounding boxes of plate/inserts solids.

Users may want to switch this option on (**1**) when the Cavity Core Wizard fails to get correct bounding box using the default method.

<Common>

<fDoNotArrangeGPAndGBGroups

OptionType="string">0</fDoNotArrangeGPAndGBGroups>

</Common>

0 - When users change the placement of guide pillars/bushes (to the moving or fixed half), the guide pillars/bushes level is moved to the proper level group.(default).

1 - Do nothing when users change the placement of guide pillars/bushes.

<MBWizard>

<SelCompColor OptionType="string">#DDDD00</SelCompColor
OptionType>

</MBWizard>

This option defines the preview colour of the currently selected component in the Moldbase Wizard component lists. The format of the string is #RRGGBB, where *RR*, *GG*, *BB* are the hexadecimal values for red, green and blue. The default setting is #DDDD00 (yellow).

Toolmaker options

Use these **Options** pages to select default settings to be used in Toolmaker. There are two pages of Toolmaker options:

General options (see page 163)

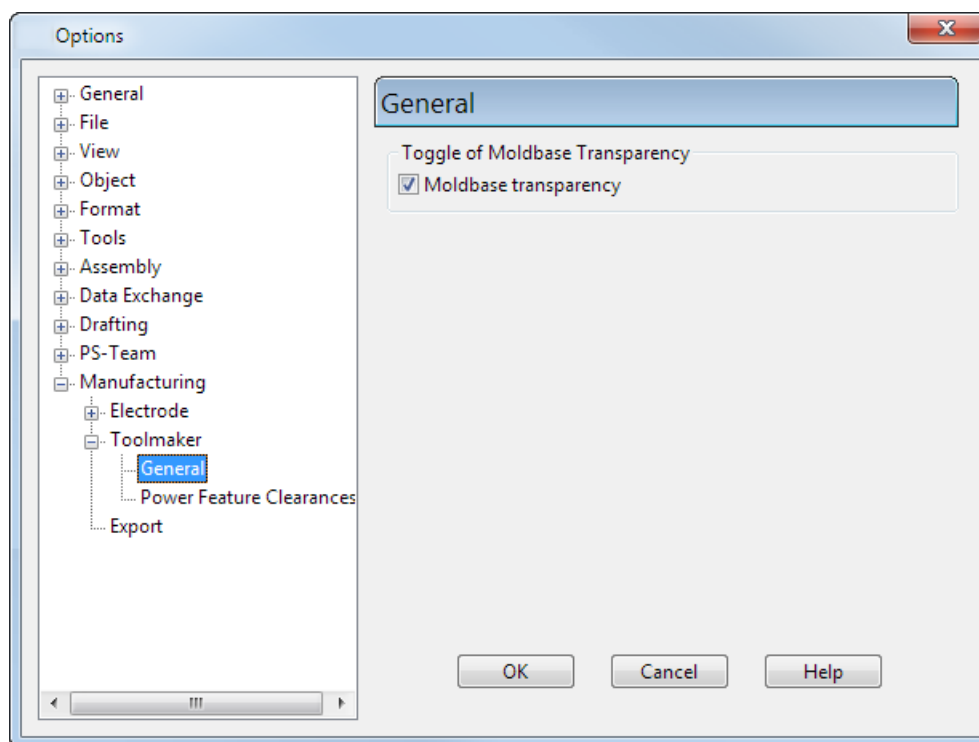
Custom Power Feature Clearances (see page 165)

General options

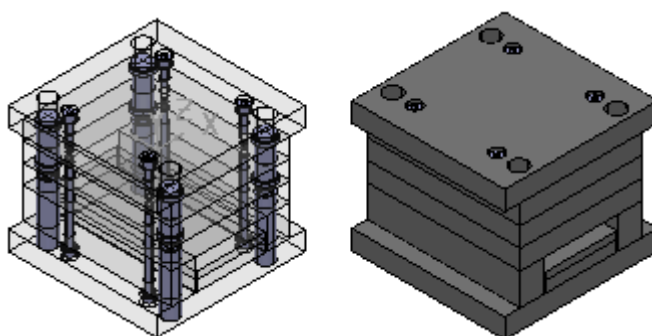
The Toolmaker options page lets you control whether the moldbase plates are transparent or opaque. More options will be added in future releases.

- 1 From the main menu, select **Tools > Options > Manufacturing > Toolmaker > General**. The **General** dialog is displayed.

- 2 Use this dialog to define Toolmaker options.

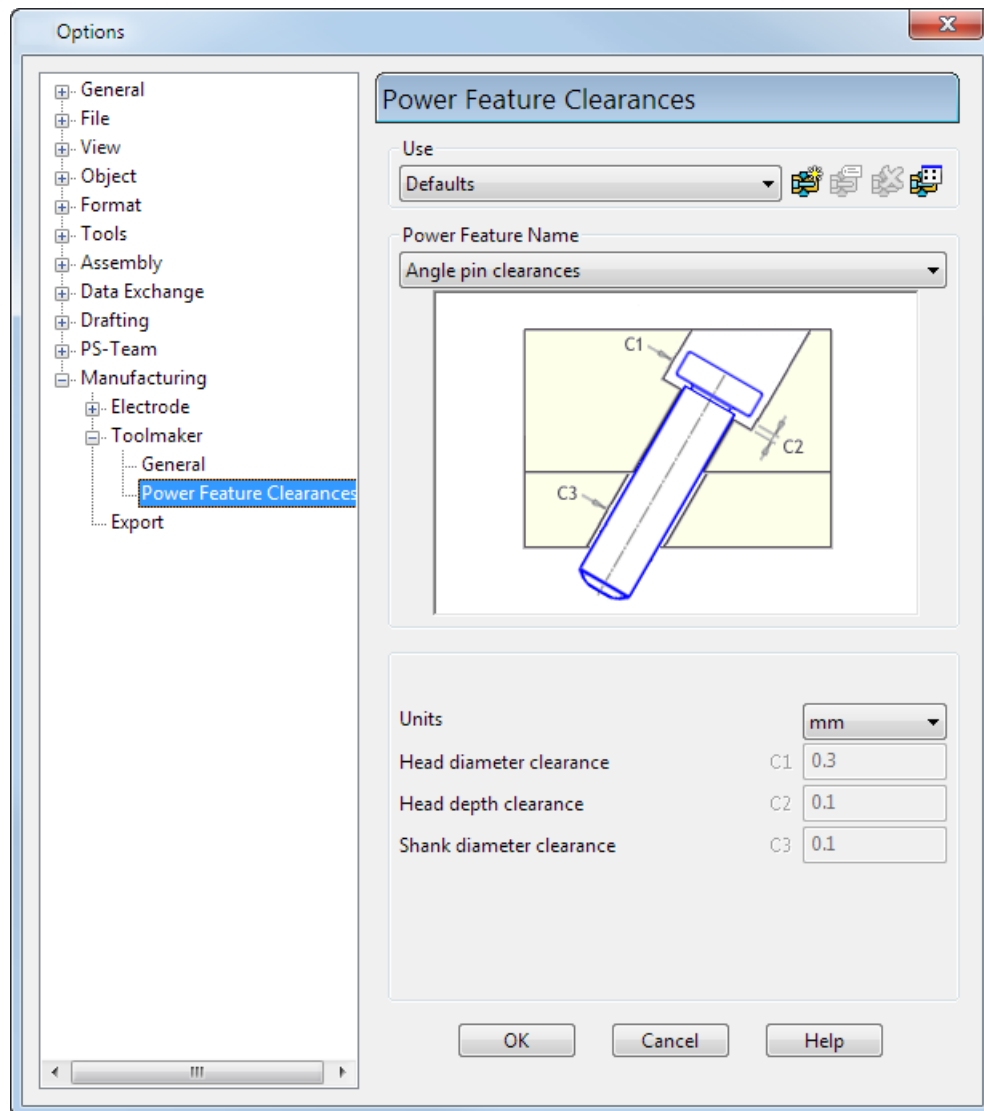


Moldbase transparency - If *ON*, the plates of the moldbase are transparent. If *OFF*, the moldbase plates are opaque.



Power Feature Clearance

Use the **Power Features Clearances Options** dialog to create a customised set of Power Features clearances that are applied across all the component catalogs. The clearance set that you select is then used by the Component Wizard.



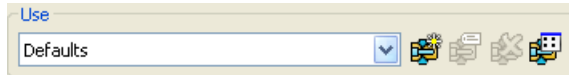
Use this dialog to:

- Create a customised set of Power Feature clearances (see page 166).
- Rename a customised set of Power Feature clearances (see page 166).
- Delete a customised set of Power Feature clearances (see page 167).
- Locating a customised set of Power Feature clearances (see page 167).

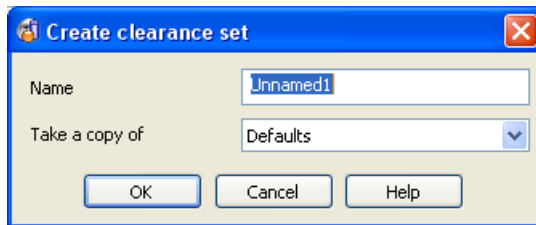
Create a customised set of Power Feature clearances

To create a customised set of Power Feature clearances:

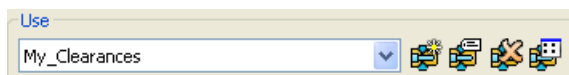
- 1 Select **Tool > Options > Manufacturing > Toolmaker > Power Features Clearances** to display Power Feature Clearances **Options** dialog. The first time the dialog is displayed, **Defaults** is automatically selected in the **Use** drop-down list.



- 2 Click  to display the **Create clearance set** dialog.



- 3 Enter a new **Name** for a new set of clearances; for example **My_Clearances**.
- 4 From the **Take a copy of** drop-down list, select the set of clearances to use as the template for the new set of clearances. Initially, *Defaults* is the only option.
- 5 Click **OK** to return to the **Options** dialog. **My_Clearances** is now automatically selected.




- 6 From the **Power Feature Name** drop-down list, select a Power Feature to be modified. The dimensions that are displayed depend on your selection
- 7 Enter a new value to modify a dimension.
- 8 Click **OK**. The new clearance set called **My_Clearances** is created. It is a customised version of the original *Defaults* clearance set.

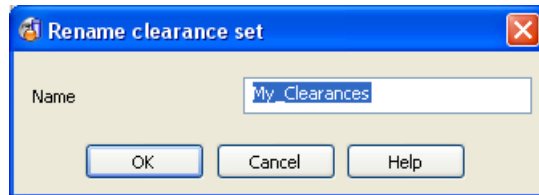
My_Clearances is saved to a file called *ClearanceSet_My_Clearances.xml*. Hover the mouse over the selection in the drop-down list to see the location of the file.

The Component Wizard now uses *ClearanceSet_My_Clearances* as the clearance set until you create a new set or select a different set from the drop-down list.

Rename a customised set of Power Feature clearances

To rename a clearance set:


- 1 Select **Tool > Options > Manufacturing > Toolmaker > Power Features Clearances** to display Power Feature Clearances **Options** dialog.
- 2 From the drop-down list, select the clearance set to be renamed.
- 3 Click  to display the **Rename clearance set** dialog.



- 4 Enter the new **Name**.
- 5 Click **OK**. The updated name is now displayed in the drop-down list.


Delete a customised set of Power Feature clearances

To delete a customised clearance set:

- 1 Select **Tool > Options > Manufacturing > Toolmaker > Power Features Clearances** to display Power Feature Clearances **Options** dialog.
- 2 From the drop-down list, select the clearance set to be deleted.
- 3 Click . The clearance set is removed from the drop-down list.

Locating a customised set of Power Feature clearances

To locate a clearance set in another location:

- 1 Select **Tool > Options > Manufacturing > Toolmaker > Power Features Clearances** to display Power Feature Clearances **Options** dialog.
- 2 Click  to display the **Select Clearance Set** dialog.
- 3 Navigate to the clearance set. This may be on your computer or held on a network drive so that it is available to several users.