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PartMaker 2015

Modeling

World-leading CAM software for production machining

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Getting Started

PartMaker Modeling 2015

Getting Started

PartMaker
Modeling

PartMaker Modeling

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Acknowledgements

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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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Getting Started with PartMaker Modeling 2015

The following sections include information to enable you to install and start using PartMaker Modeling:

Installing PartMaker Modeling (see page 4)

Starting PartMaker Modeling (see page 14)

Using the Browser window (see page 16)

Getting help with PartMaker Modeling (see page 18)

Basic modeling concepts (see page 103)

Getting Started

The following sections include information to enable you to install and start using PartMaker Modeling:

What is PartMaker Modeling? (see page 2)

PartMaker Modeling functionality (see page 4)

Installing PartMaker Modeling (see page 4)

Starting PartMaker Modeling

What is the Browser? (see page 16)

Getting Help with PartMaker Modeling (see page 18)

Example data (see page 21)

Further information (see page 21)

What is PartMaker Modeling?

PartMaker Modeling uses PowerSHAPE to provide a powerful hybrid solid, surface and wireframe modeller for the design of products with complex shapes.



You can use **PartMaker Modeling** to design products from scratch; particularly those with sculptured shapes. It is also ideally suited to preparing existing product models for manufacture — by importing a model from another CAD system and adding features for manufacture such as fillets, split, and draft surfaces.

PartMaker Modeling releases

PartMaker Modeling releases use a year-based title in the form PartMaker Modeling 2015. This title is used in the **About** dialog and the Main toolbar.

For example:

Delcam PartMaker Modeling for the first customer release after July 1st.

Delcam PartMaker Modeling R2 for the second customer release after July 1st.

64-bit PartMaker Modeling

PartMaker Modeling is available in two versions:

- PartMaker Modeling 64-bit runs only on a 64-bit operating system and has no limit on the amount of available memory it can use.
- PartMaker Modeling 32-bit runs on 32-bit and 64-bit operating systems but has a limit on the amount of memory it can use.

The installation procedure enables you to select the version you want to install.

Benefits of the 64-bit version

Using a 64-bit version of PartMaker Modeling:

- Enables you to handle much larger models because there is no limit on the amount of available memory it can access.
- Improves calculation times.
- Avoids "out of memory" errors.

Using the 64-bit version

The 64-bit version of PartMaker Modeling requires:

- 64-bit hardware (Intel/AMD) and operating system (Vista or Windows 7).
- More memory on start-up and for operations like rendering (approximately 20% depending on usage). We recommend a minimum of 8 GB memory.

PCs bought in the last few years should be capable of running a 64-bit operating system. However, 32-bit Windows is still common.

Using the 32-bit version

It is possible to run the 32-bit version on a PC with a 64-bit operating system but you will only use up to 4 GB of memory. If you have a 64-bit PC you will need a 64-bit operating system and appropriate drivers for your hardware.

If you have a 32-bit PC, the only option is to run the 32-bit version of PartMaker Modeling. This uses up to 2 GB of memory.

Compatibility

All data files are interchangeable between 32-bit and 64-bit versions.

Computer issues

Use one of the following links to help you find out if your PC and Operating System are 64-bit:

<http://support.microsoft.com/kb/827218>

<http://www.lytebyte.com/2009/08/04/how-to-check-if-my-processor-is-x86-32-bit-or-x64-64-bit-in-windows>

To find out if you are running a 64-bit version of PartMaker Modeling look in the title bar or the dialog displayed from the **Help > About** menu.

There are separate installers for the 64-bit and 32-bit versions of PartMaker Modeling. It is possible to have both versions installed on the same computer; there is a separate desktop shortcut for each version.

64-bit issues

The 64-bit version of PartMaker Modeling:

- Supports dongle or paf licencing.
- Does not support the flex network licencing system.

PartMaker Modeling functionality

PartMaker Modeling provides PartMaker users with functionality for creating and repairing 3D solid models.

PartMaker Modeling includes the following functionality:

- complete solid, surface, and wireframe modelling.
- full 2D drafting capability using Delcam Drafting.
- surface and solid morphing.
- Direct Modelling.
- assembly modelling.
- import and export of a comprehensive range of file types. The PartMaker Solids Import module is also available.

Installing PartMaker Modeling

Use the following sections to install PartMaker Modeling:

- Before you begin (see page 5).
- Using the Installation Wizard (see page 5).
- Installing your Product Authorisation File (PAF) (see page 9).
- Installation options for advanced users (see page 9):
- Changing the drawing standard (see page 13).
- Upgrading an existing version of the software (see page 13).
- Installation problems (see page 13).

- General installation information (see page 14).
- Model version (see page 14)
- Uninstalling PartMaker Modeling (see page 14).

Before you begin

- 1 To start PartMaker Modeling, you will require a **Product Authorisation File (PAF)**. Please contact your Sales Agent for more details.
- 2 By default, the installer will set up each user with their own private centralised shared data area. If you are happy with the default setup, proceed with installing PartMaker Modeling. For further details, see *Installing PartMaker Modeling*.

However, if you are a system administrator and want to control where centralised shared data files are installed on your network, you need to set up registry entries **before installing PartMaker Modeling**. For further details, see Installation options for advanced users (see page 9).



*In PartMaker Modeling, the centralised shared data files are called **the shared database (DB)** or **shareddb** for short.*

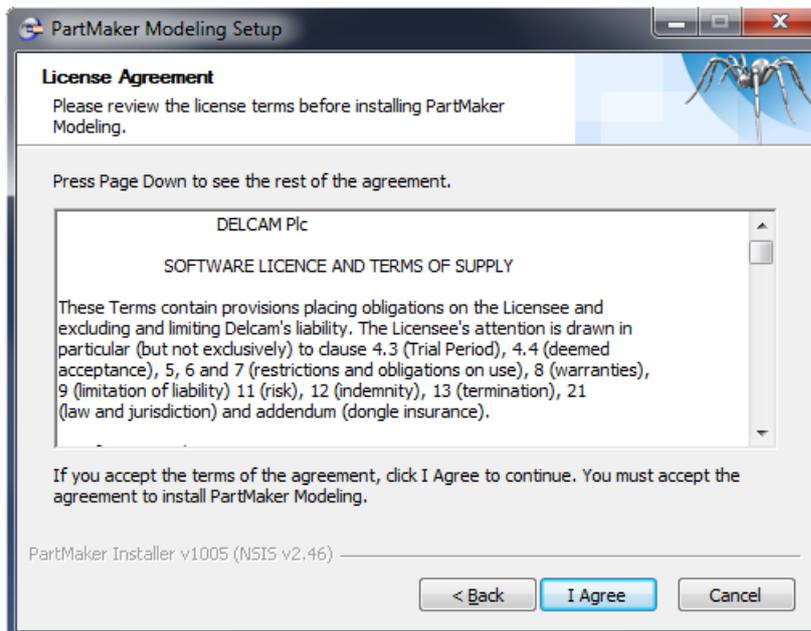
Using the Install Wizard

- 1 Log in to your system as an administrator, insert the DVD in the drive and wait a few seconds. The Install Wizard is displayed.
- 2 Use the following steps of the Install Wizard to install PartMaker Modeling:
 - a Use the drop-down list to select **English** as the installation Language.

- b Click **OK** to display the **Welcome** page of the **PartMaker Modeling Setup Wizard**.

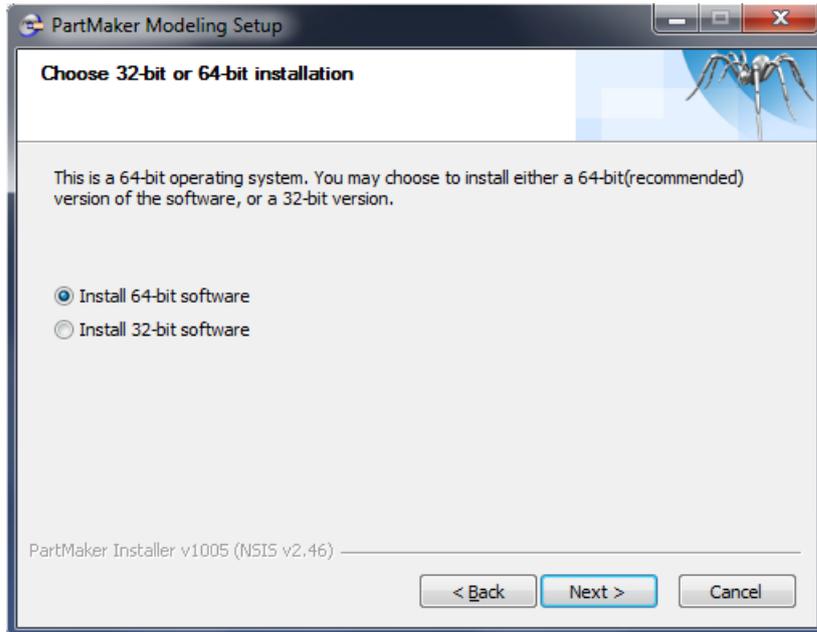


- c Click **Next** to display the **License Agreement** page of the **PartMaker Modeling Setup Wizard**.

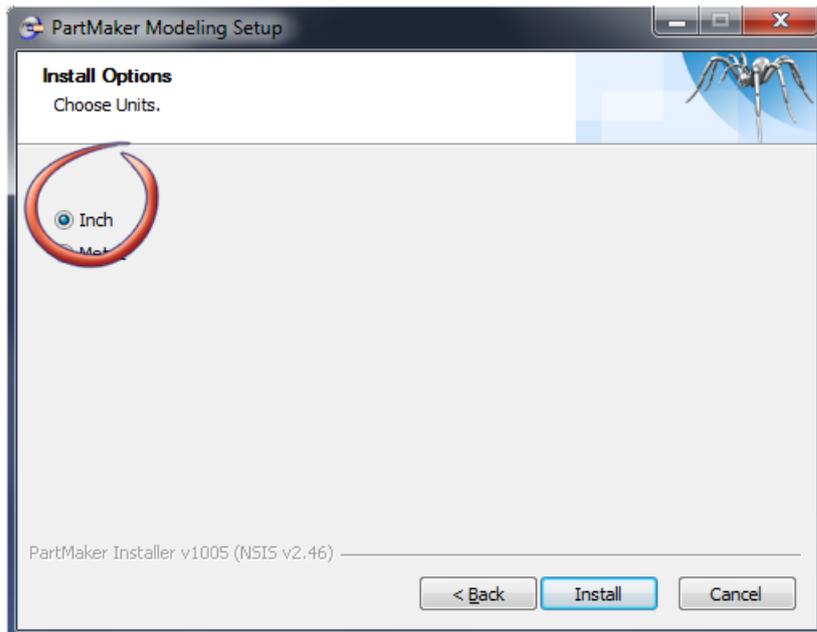


- d Click **I Agree**.

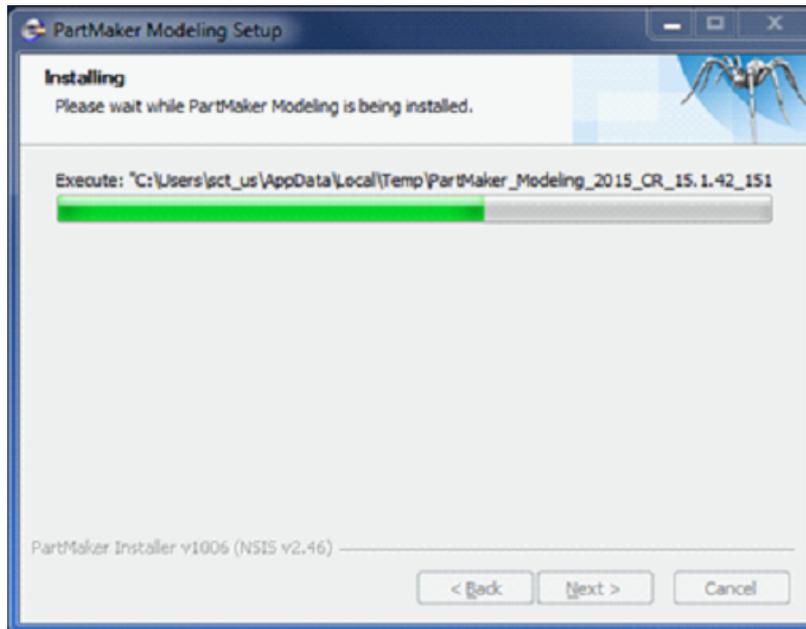
- e Select the appropriate option for the version of the software you wish to install. If you have a 64-bit operating system, it is recommended that you select **Install 64-bit software**.



- f Click **Next** to display the **Install Options** page of the **PartMaker Modeling Setup Wizard**.



- g If you normally work in inches, leave the **Inch** option selected and click **Install** to display the **Installing** page of the **PartMaker Modeling Setup Wizard**. This shows progress through the installation.



When installation is complete, the final page of the **PartMaker Modeling Setup Wizard** is displayed.



- h Click **Finish** to complete the installation.

- 3 When installation is complete, restart your computer.



For details on default installation paths for your operating system, see: Windows 7 default installation paths (see page 9)

Windows 7 default installation paths

If your operating system is Windows 7 or later, by default, the installation wizard installs folders in the following locations:

C:\Program Files\Delcam\XXXXX, (where XXXX is the product and product version)

c:\Users\Public\Documents\Delcam\shareddb

or displayed as c:\Users\Public\Public Documents\Delcam\shareddb

c:\Users\Public\Documents\Delcam\parts

or displayed as c:\Users\Public\Public Documents\Delcam\parts

For 32-bit installations on a 64-bit operating system, the path is:

C:\Program Files (x86)\Delcam\XXXXX, (where XXXXX is the product and product version)

If you want to install the *shared database* and parts directory onto a different drive or directory, see *Setting the shared database on a different drive using the Registry Editor* (see page 11).

Installing your Product Authorisation File (PAF)

After you have installed PartMaker Modeling:

- 1 Restart your computer.
- 2 Locate the **Product Authorisation File (PAF)** provided by your sales agent. You may have received this file as an attachment to an email, on a USB drive, or on a CD/DVD disk.
- 3 Copy the .paf file into the folder indicated in the email.
- 4 When you have installed your PAF, start PartMaker Modeling.
- 5 When PartMaker Modeling is running, click **Help > Check for PAF updates** to check for a new PAF to download.



You need to have a valid licence file and be connected to the Internet to use this option.

Installation options for advanced users

The following information is provided for advanced users with specific installation requirements.

- Installing the shared database (see page 10):

The shared database is installed by default during the installation process. Skip this section unless you are a system administrator, wanting to control the location of shared data files on a network.

- Setting the shared database on a different drive using the Registry Editor (see page 11):

The location of the shared database does not need to be changed for the program to work. Skip this section unless you are familiar with using the regedit command to edit the registry and you want to set the *shared database* database or parts directory on a different drive.

- Location of user-specific files (see page 12):
- Specify the location of user-specific files such as custom toolbars and shortcuts, so that the same location is used for future versions.
- When you install PartMaker Modeling, you are asked to select your unit of working. If you select **Inch** (the default setting), the ANSI drawing standard is applied. If required, you can change the drawing standard that is used (see page 13).

Installing the shared database



Important: *The shared database is installed by default during install. Skip this section unless you are a system administrator wanting to control the location of shared data files on a network.*

PartMaker Modeling allows a group of users to share information that is applicable to everyone in that group. For example, the index of models being created or company standards. This information is stored in a shared database (DB) area called shareddb. When you install PartMaker Modeling, by default, everyone has their own private shared database area and their own parts areas where models are stored by default.

If you want to use these default paths and folders, proceed with installing the program. For further details, see Installing PartMaker Modeling.

For details on the default installation paths for your operating system, see Windows 7 default installation paths (see page 9).

If you want to install the *shared database* and parts directory onto a different drive or directory, you need to make sure the paths have been set in the registry before installing. For further details, see Setting the shared database on a different drive using the Registry Editor (see page 11).

Setting the shared database on a different drive using the Registry Editor

WARNING



This section should only be attempted if you are familiar with using the regedit command to edit the registry.

Incorrectly editing the registry may severely damage your system. At the very least, you should back up any valued data on the computer before making changes to the registry.



This operation should be completed before installing PartMaker Modeling.

If you already have PartMaker Modeling installed, this version should be uninstalled before completing the steps shown in the next section. You will also need to manually delete the shareddb and parts directories.

Setting the shareddb on a different drive

This is a guide to setting the *shared database* database or parts directory on a different drive using the **Registry Editor**.



The location of the shareddb does not need to be changed for PartMaker Modeling to work. Skip this section unless you specifically want to set the shared database database or parts directory on a different drive and you are familiar with the regedit command.

The installation path examples given use **D:** as the installation drive. The drive on your system may be different.

If the shared database is intended to be shared, for example on a network, the path should include the following:

`\\name of machine\location of parts directory or shareddb\`

The following are some example paths:

`\\server1\d\dcam\parts\`

`\\server1\d\dcam\shareddb\`

- 1** In the **Registry Editor**, open the following directory to see if a **Delcam** branch exists:

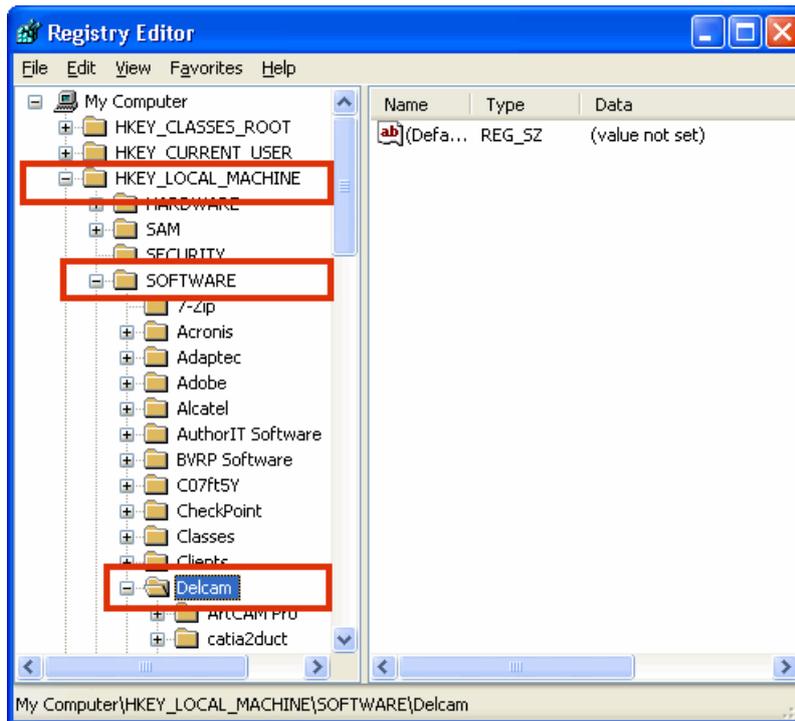
`HKEY_LOCAL_MACHINE\SOFTWARE`

If you are using a 64-bit machine, open the following directory:

`HKEY_LOCAL_MACHINE\SOFTWARE\WOW6432NODE`

If a Delcam branch exists continue with the following instructions.

If a Delcam branch does **not** exist, create a new **key** in HKEY_LOCAL_MACHINE\SOFTWARE and call it **Delcam**.



Continue with the following instructions:

- 2 Open HKEY_LOCAL_MACHINE\SOFTWARE\Delcam and create a **key** called *config*.
- 3 Open HKEY_LOCAL_MACHINE\SOFTWARE\Delcam\config and create a **key** called *node*.
- 4 Open HKEY_LOCAL_MACHINE\SOFTWARE\Delcam\config\node and create a new **string** called *parts*.
- 5 Set the **Value data** of *parts* to be, for example \\server1\d\dcam\parts\
- 6 Open HKEY_LOCAL_MACHINE\SOFTWARE\Delcam\config\node and create a new **string** called *shareddb*.
- 7 Set the **Value data** of *shareddb* to be, for example \\server1\d\dcam\shareddb\
- 8 Exit the **Registry Editor** and proceed with installing the program. For further details, see Installing PartMaker Modeling.

Location of user-specific files

The location of the user-specific files (for example custom toolbars and shortcuts) is structured so that the same location will be used for all future versions. These files are stored in xxxx\PowerSHAPE where xxxx is:

C:\Users\[username]\AppData\Roaming

the location that is indicated by your **HOME** variable.

Changing the drawing standard

The shared database contains details of all the drawing standards supported by PartMaker Modeling. One of these is set up as standard, so that every CAD model created will have this standard set by default. The standard contains the units that you are working in. So, for example, if you work to ANSI standard in inches it is important to choose the correct standard.

When you install PartMaker Modeling, you are asked to select your unit of working. If you select **Inch** (the default setting), the ANSI drawing standard is applied.

To change the default drawing standard on installation

You can change the default drawing standard when you are installing PartMaker Modeling using the Install Wizard (see page 5).

To change the default drawing standard on an existing installation

- a Run the **Install Wizard** (see page 5) to change the default standard for an existing installation or computer that has had a previous version of PartMaker Modeling installed.

Upgrading an existing version of PartMaker Modeling

If you have an older version of the product and wish to install Delcam PartMaker Modeling 2015, insert the DVD into the disk drive and follow the installation instructions. The installation wizard will guide you through installing the new version.

Installation problems

Installation problems usually only happen if you attempt to bypass the installation process: For example, copying files manually rather than using the installation wizard. The installation wizard does extra work, such as setting up registry entries and should always be used in preference to manual copying.

If you experience problems with your installation try uninstalling Delcam PartMaker Modeling and re-installing it using the installation wizard.

If you still experience problems contact your sales agent.

General Installation Information

- All programs appear as separate products in **Start > Control Panel > Add or Remove Programs**.
- Shortcuts are created on the desktop and the other options are added to the **Start** menu.

Model version

All models opened in PartMaker Modeling 2015 use model version 41. When PartMaker Modeling 2015 is installed and run for the first time, the initialisation work will open the latest NEW_MODEL_MASTER_XX you have (where XX <41) and save it as NEW_MODEL_MASTER_41.

Uninstalling PartMaker Modeling

- 1 To uninstall PartMaker Modeling select one of the following:
 - **Start > Control Panel > Add or Remove programs** (XP systems)
 - **Computer > Uninstall or change a program** (Vista or Windows 7 systems)
 - From the **Start** menu select **All Programs** and use the sub-menus to. select **Delcam > PowerSHAPE PowerSHAPE 2015 > Uninstall PowerSHAPE2015**.
- 2 Select Delcam PowerSHAPEXXXXX (where XXXXX is the product version) and select **Remove**.

This will remove Delcam PowerSHAPE only. Any sub-products will need to be removed separately.



The shareddb directory and the parts directory are NOT removed. They are left permanently on the machine.

The PAF file directory will also be unaffected.

Starting PartMaker Modeling

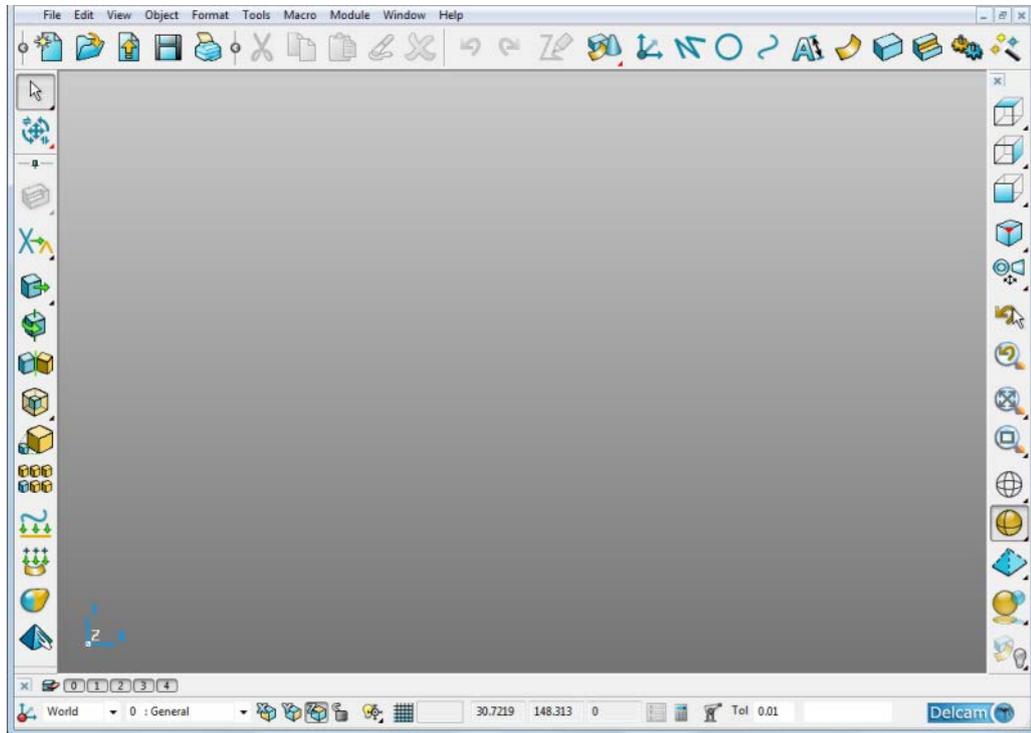
To start PartMaker Modeling:

- 1 From the **Start** menu select **All Programs** and use the sub-menus to.
 - a Select **Delcam**.
 - b Select **PowerSHAPE**.
 - c Select the **PowerSHAPE 2015**.
 - d Select **Modeling 2015**.

Alternatively, double-click the **PartMaker Modeling** icon on your desktop 

After a brief wait, the program starts up. A new large window appears on the screen.

This is **PartMaker Modeling**.

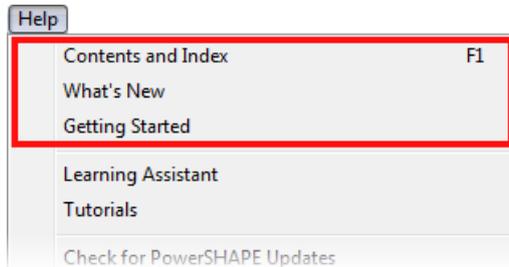


What is the Browser?

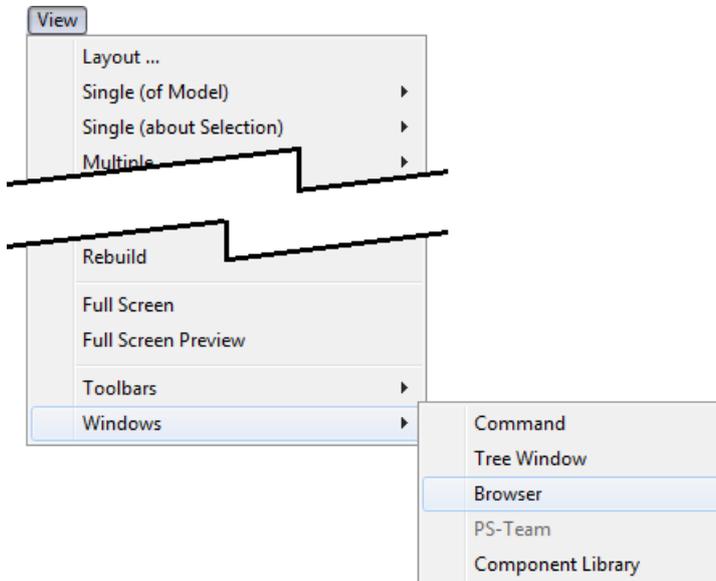
The **Browser** is an additional support window in PartMaker Modeling where the HTML help, interactive help, and tutorials are displayed.

Opening the Browser

To open the Browser select one of the options from the **Help** menu, shown below.



You can also select **Browser** from the **Windows** option on the **View** menu.



Using the Browser

Navigation buttons are provided to assist you find the help you need.



Click on a tab in the Browser to display the information:



 **Home** — This displays a welcome information page for the PowerSHAPE product you are using.

 **Learning Assistant** — This tab is inactive if a Delcam Toolmaker wizard is running. For further information on the Learning Assistant see Getting Help with PartMaker Modeling (see page 18).

 **Tutorial** — For further information see Getting Help with PartMaker Modeling (see page 18).

 **Reference help** — For further information see Getting Help with PartMaker Modeling (see page 18).

 **Delcam.com browser.**

 **Delcam PS-Team** — This is currently in development.

 **Delcam-Toolmaker** — This tab is active only when a Delcam Toolmaker wizard is running in the Browser.

Closing the Browser

To close the Browser click the cross shown below:



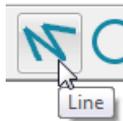
Getting Help with PartMaker Modeling

You can get help in one of the following ways:

- Tooltips (see page 18)
- Context-sensitive help (see page 18)
- Status bar help (see page 18)
- Reference help (see page 19)
- Tutorials
- Learning assistant (see page 19)

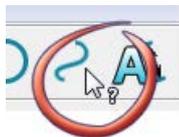
Tooltips

Placing the cursor over a button displays a brief description of the function of the button.



Context-sensitive help

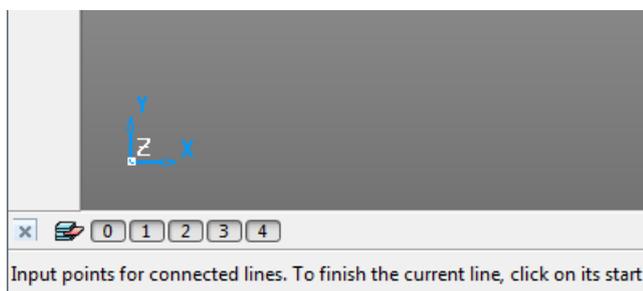
Use **Shift** + **F1** to access context-sensitive help on buttons. When you hold down the **Shift** key and press **F1**,  is displayed. Move this to the relevant button and click, for example:



Context-sensitive help is available on toolbar buttons and buttons on a flyout, but not on the flyout button itself.

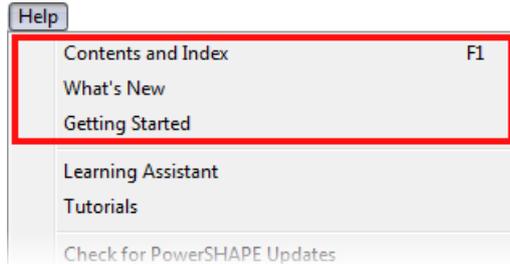
Status bar help

The status bar displays information when you place the cursor over a button.



Reference help

- 1 Select **Help** to display the options.



- 2 Select **Contents And Index** to display the full Reference Help in the Browser.
- 3 You can change the display by selecting the following:



Hide

Displays the help in the full width of the Browser.



Show

Displays the **Contents** or **Index**.

Learning Assistant

Start the **Learning Assistant** in one of the following ways to display the options in the Browser:

- Select **Learning Assistant** from the **Help** menu.
- Select  tab in the Browser.



The Learning Assistant explains the steps required to use each main function. Click on the steps to display drop-down interactive help to guide you through the tasks.

The help is context-sensitive and updates automatically as you use the software.



Full information is available in the **Getting Started** section of the on-line **Reference Help**.

Using the Learning Assistant to create a single line

- 1 Select **Creating Objects** to display the menu.



- 2 Click **Lines** .

3 Click **Single Line** . You can get help on creating a single line by clicking **How?**

4 Create the line in the graphics window.

5 Click **Exit line creation**  to end line creation.

The Learning Assistant displays **Editing Lines** options.

Example data

You can import sample data to use in PartMaker Modeling. The examples that are supplied are all deliberately simple models to make it easy for you to experiment with them.

1 From the **File** menu, select **Examples**.

2 Select a file and the data is imported.

In some cases, a dialog is displayed to use with the data.

The following example data is available:

- Concept aircraft
- Perfume bottle
- Jigsaw piece
- Milk bottle
- Shoe sole

Further information

For details of the complete functionality, refer to the complete user documentation that is available in PartMaker Modeling. You can access these from the **Help** menu in PartMaker Modeling.

Training courses

Training courses are available. Please contact your Delcam Sales Partner for further details.

Tutorial - Designing a plate

Use this tutorial to create a solid model of a plate by:

- checking the drawing standards of the model
- creating and editing lines
- creating a composite curve
- creating a solid extrusion from the composite curve
- creating solid cut features
- creating solid hole features
- creating solid chamfers
- changing the material of the model

1. Check the tolerances of the models

This tutorial is going to be completed using the ANSI drawing standard.

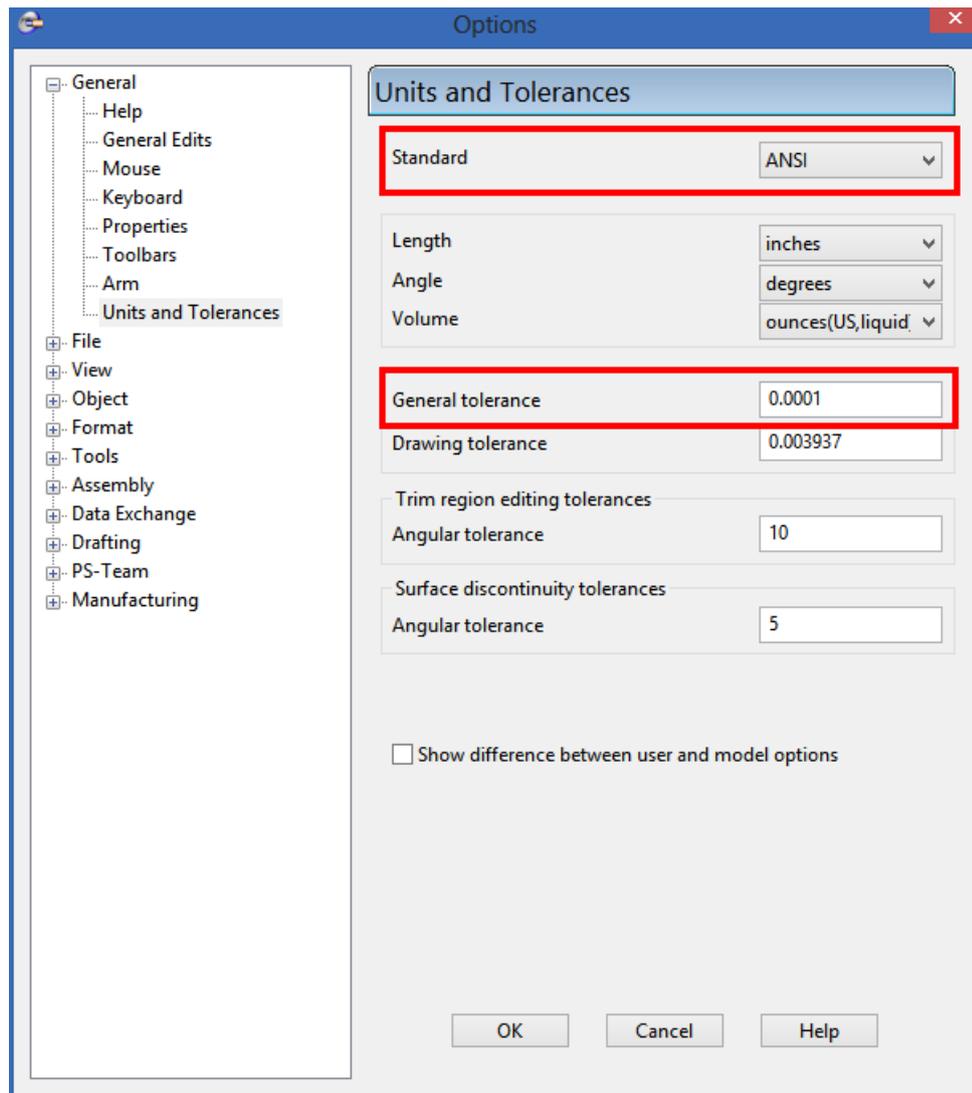
- Standard = ANSI
- Length = inches
- Angle = degrees
- Volume = ounces(US, liquid)

- 1** Click **Open new model** .

An empty model is opened and the **Standard** should be set to **ANSI**.

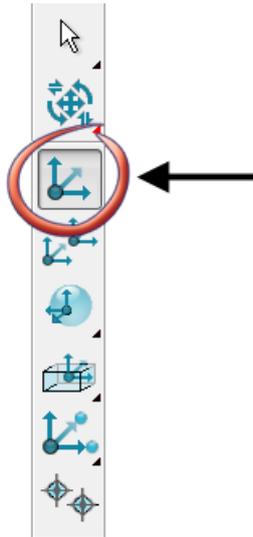
- 2** Click **Tools > Options** from the Main menu.
- 3** Click **Units and Tolerances**.

The values should be as displayed below:



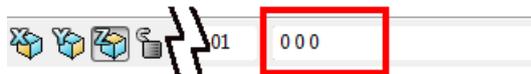
- 4 If the values aren't correct change to match the above image.
General tolerance = 0.0001.
- 5 Click **View > Shading**.

- 2 Click **Single workplane** .

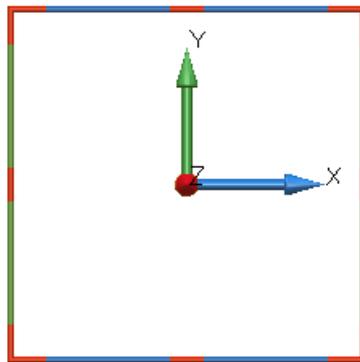


- 3 Click in the **Command Input** box in the **Status Bar** and type:

0 0 0



- 4 Press **Enter**.



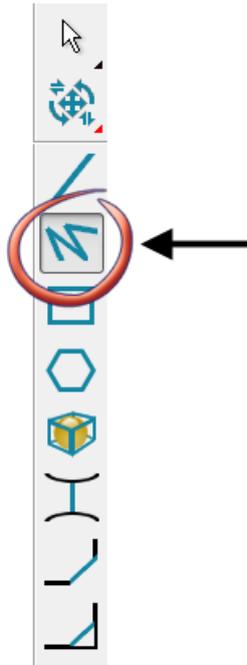
- 5 Click anywhere in the graphics window to deselect the model.

3. Create lines

- 1 Click **Line** .



2 Click **Continuous Lines** .



3 Click in the **Command Input** box in the **Status Bar** and type:

4 0 0

4 Press **Enter**.

4 3 0

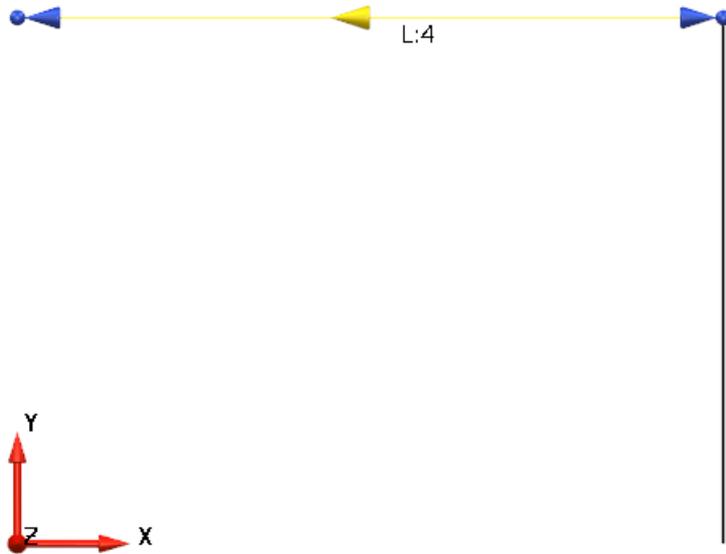
5 Press **Enter**.

0 3 0

6 Press **Enter**.

7 Click **Select** .

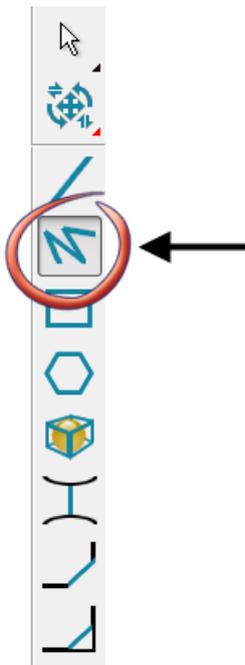
8 Click **Resize to Fit** .



9 Click **Line** .



10 Click **Continuous Lines** .



11 Click in the **Command Input** box in the **Status Bar** and type:
0.5 0 0

12 Press **Enter**.

0.5 2 0

13 Press **Enter**.

3 2 0

14 Press **Enter**.

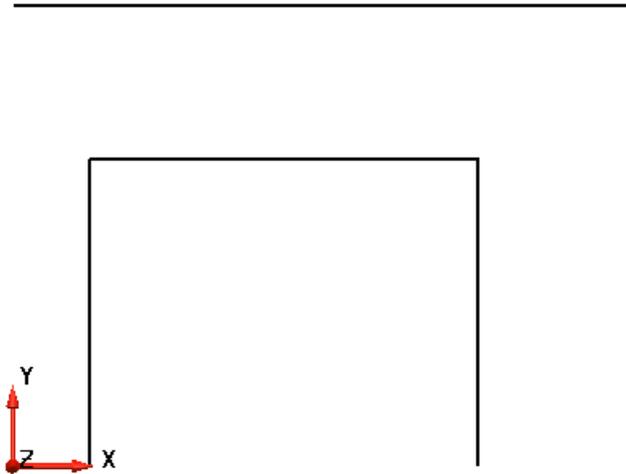
3 0 0

15 Press **Enter**.

16 Click **Select** .

17 Click anywhere in the graphics window to deselect the model.

18 Your model should look like the one shown below:

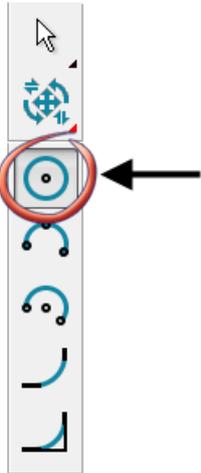


4. Create arcs

1 Click **Arc** .



- 2 Click **Create a Full Arc** .



- 3 Click in the **Command Input** box in the **Status Bar** and type:
r 1

- 4 Press **Enter**.

3 2 0

- 5 Press **Enter**.

r 0.5

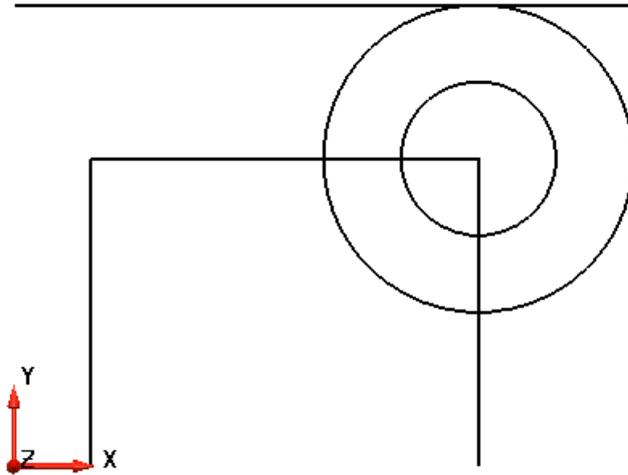
- 6 Press **Enter**.

3 2 0

- 7 Click **Select** .

- 8 Click anywhere in the graphics window to deselect the model.

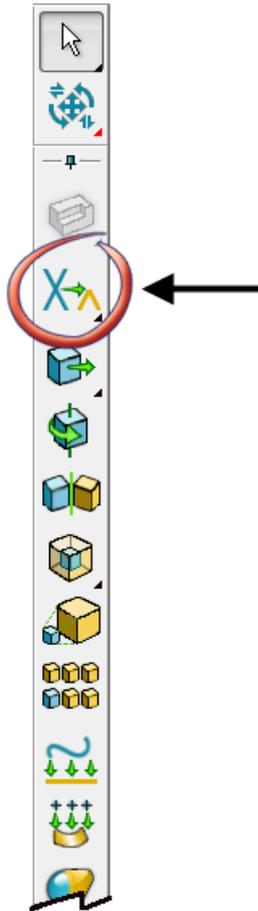
Your model should look like the one shown below:



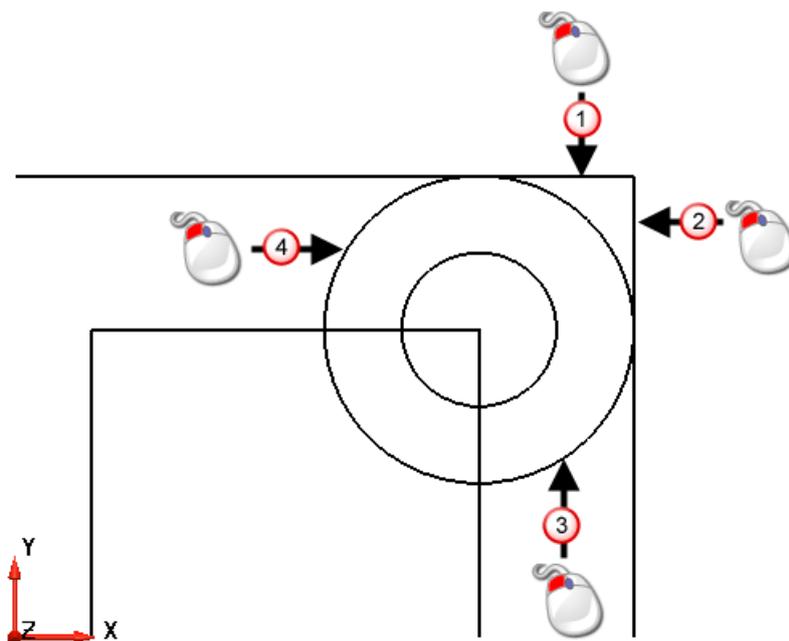
5. Limit wireframes - 1

- 1 Click **Show General Edits Options** .

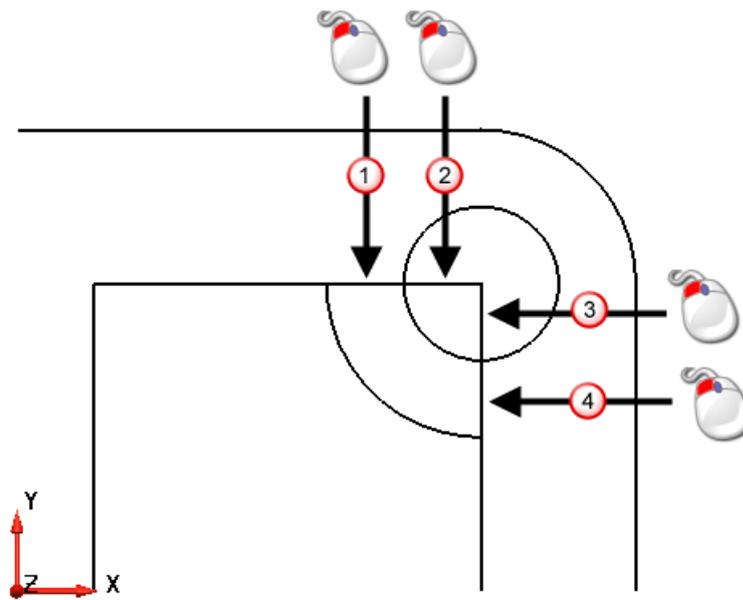
2 Click **Interactively Limit Wireframes** .



3 Click the positions shown below:

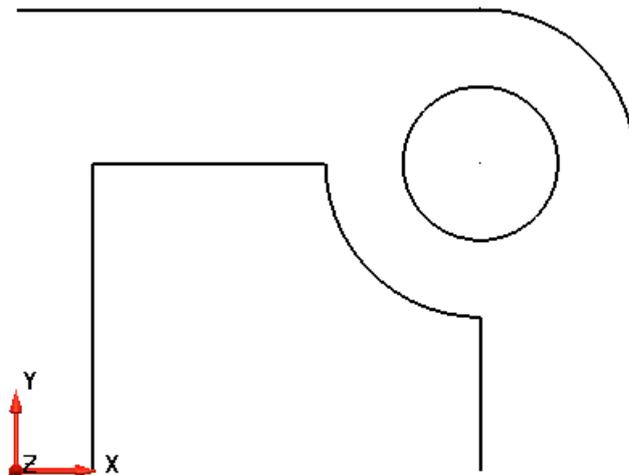


4 Click the positions shown below:



5 Click **Select** .

The lines and arcs are trimmed.

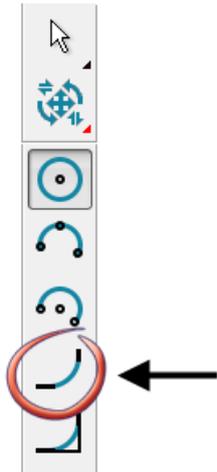


6. Create fillet arcs - 1

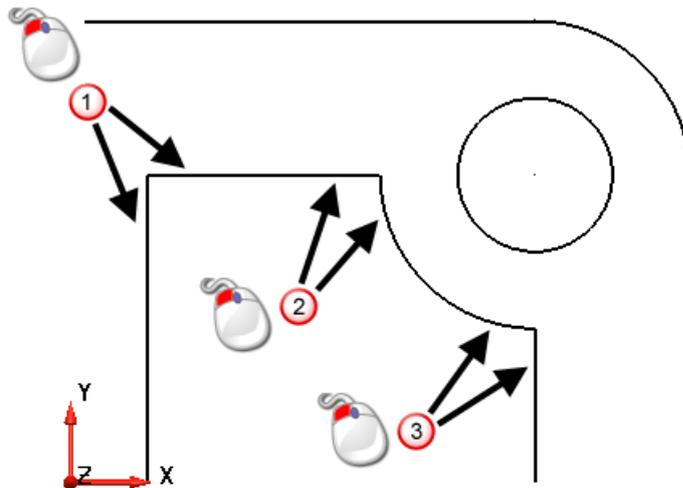
1 Click **Arc** .



- 2 Click **Create a fillet arc** .

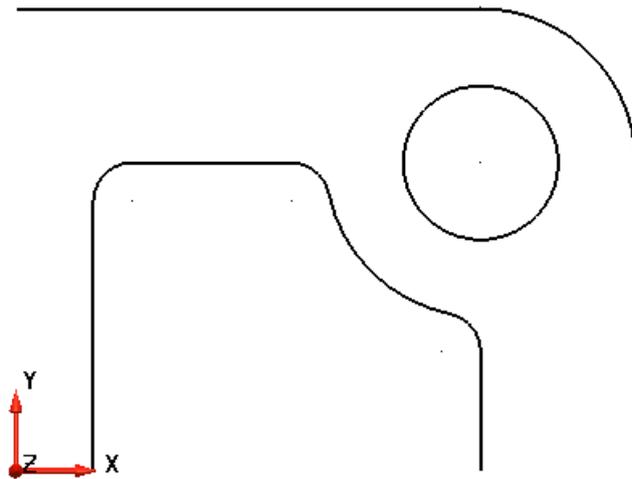


- 3 Click in the **Command Input** box in the **Status Bar** and type:
r 0.25
- 4 Press **Enter**.
- 5 Click the pairs of lines and arcs shown below:



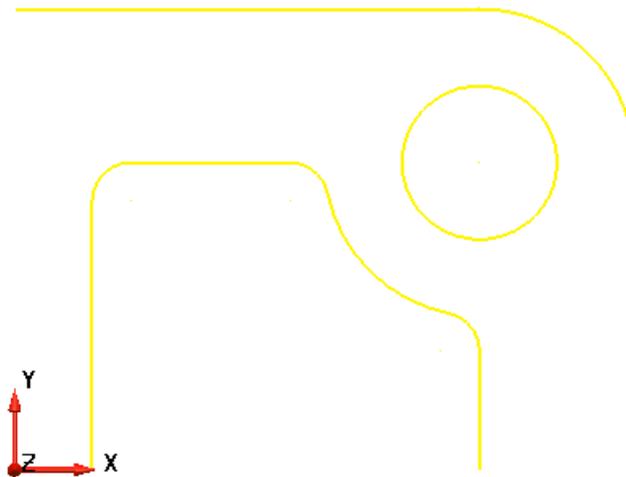
- 6 Click **Select** .
- 7 Click anywhere in the graphics window to deselect the model.

Your model should look like the one shown below:



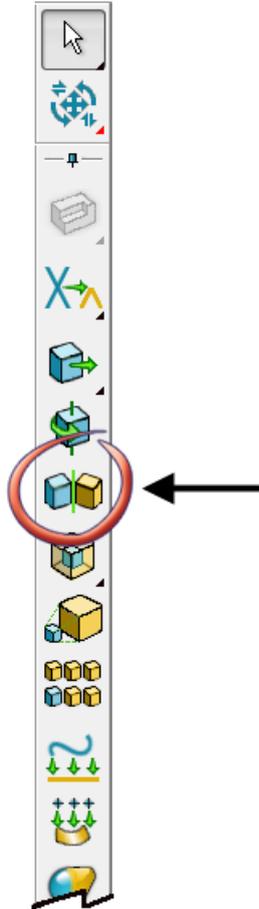
7. Mirror the wireframes

- 1 Click **Quick select all wireframes** .

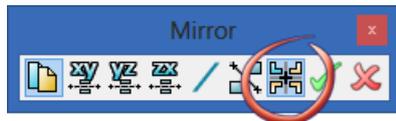


- 2 Click **Show General Edits Options** .

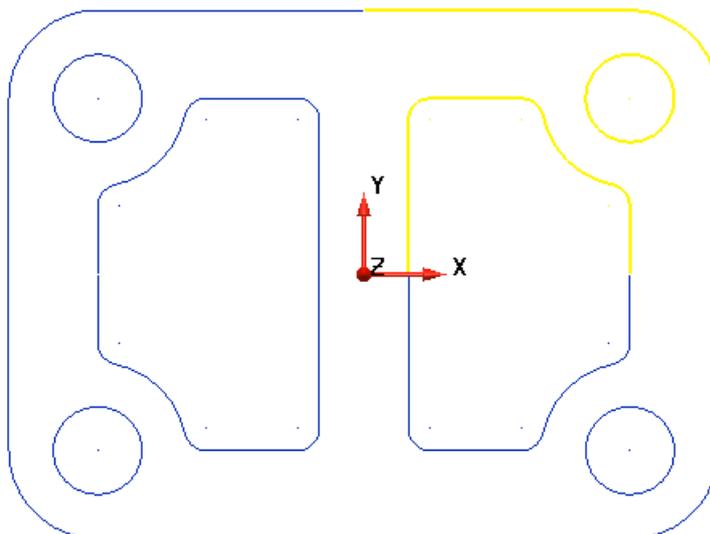
- 3 Click **Mirror/symmetrise object** .



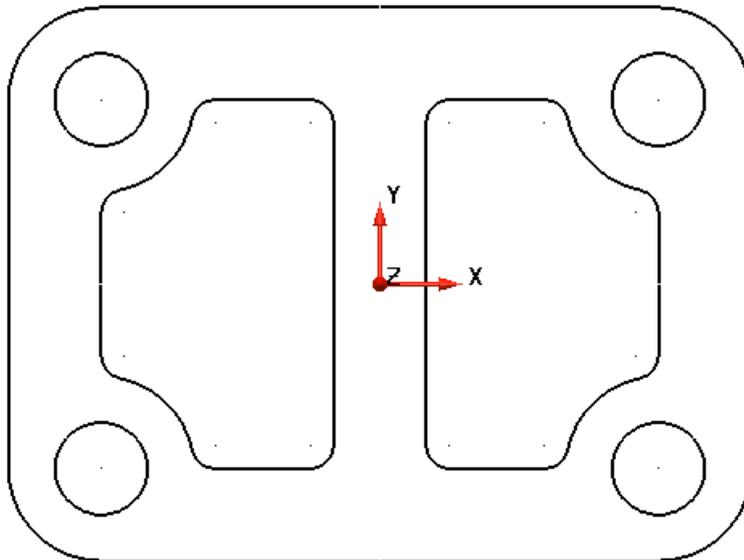
- 4 Click **Create multiple copies of the selected items** .



A preview of the mirror is drawn on the model.



- 5 Click **Apply** .
- 6 Click **Dismiss** .
- 7 Click **Resize to Fit** .
- 8 Click anywhere in the graphics window to deselect the model.
The geometry is mirrored across the 3 planes.

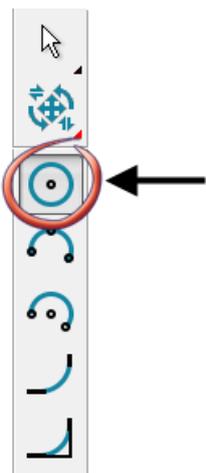


8. Create a full arc - 1

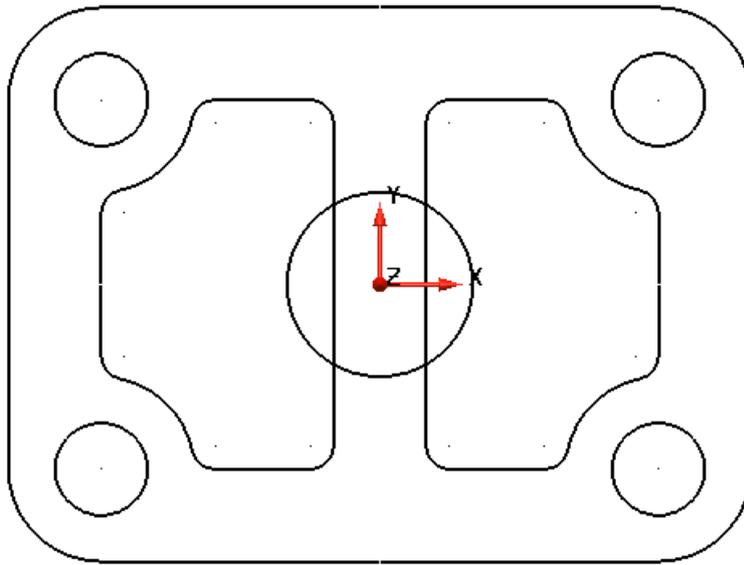
- 1 Click **Arc** .



- 2 Click **Create a Full Arc** .



- 3 Click in the **Command Input** box in the **Status Bar** and type:
r 1
- 4 Press **Enter**.
0 0 0
- 5 Press **Enter**.
- 6 Click **Select** .
- 7 Click anywhere in the graphics window to deselect the model.
Your model should look like the one shown below:

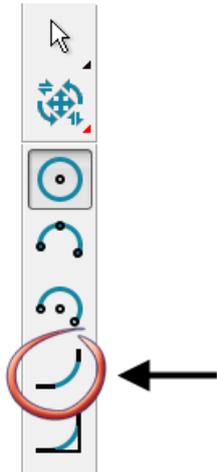


9. Create fillet arcs - 2

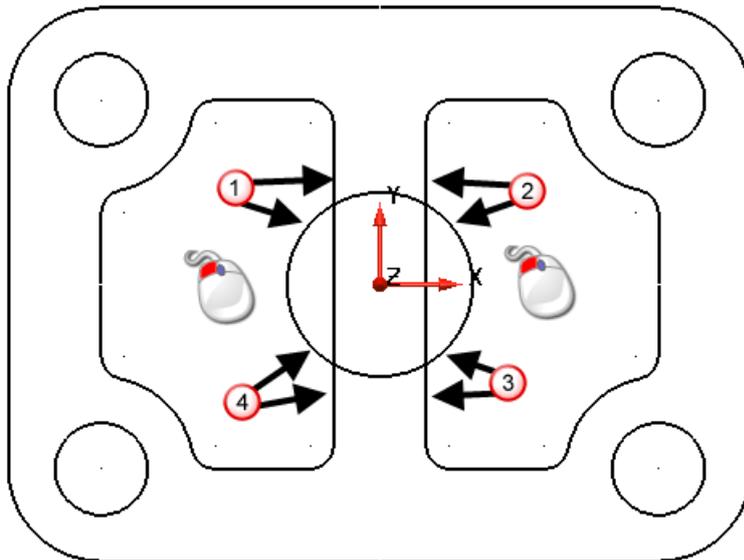
- 1 Click **Arc** .



- 2 Click **Create a fillet arc** .



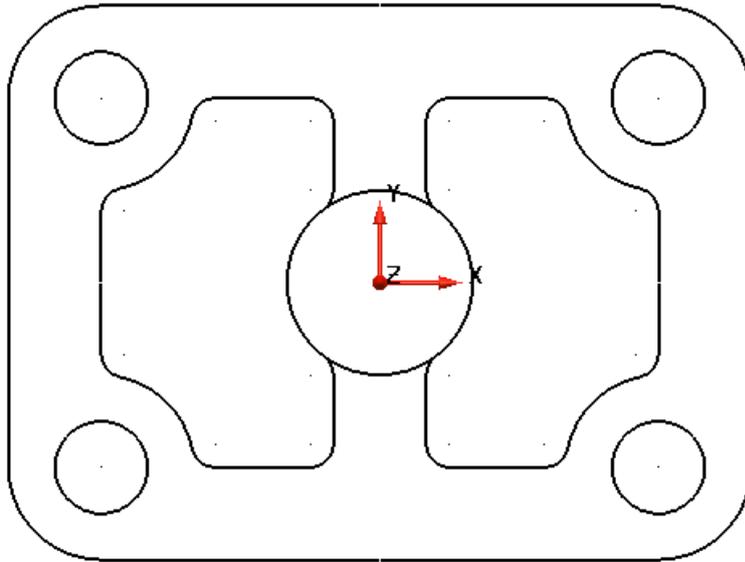
- 3 Click in the **Command Input** box in the **Status Bar** and type:
r 0.25
- 4 Press **Enter**.
- 5 Click the pairs of lines and arcs shown below:



- 6 Click **Select** .

- 7 Click anywhere in the graphics window to deselect the model.

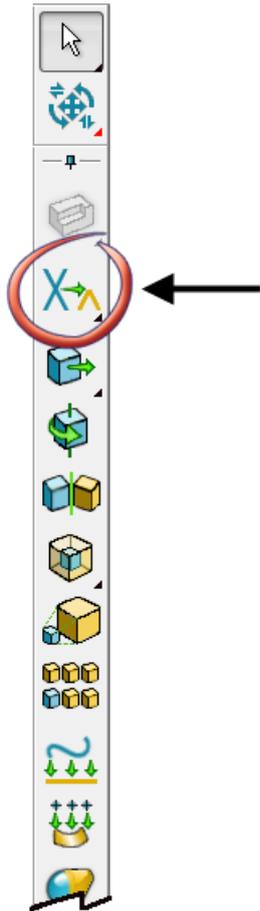
Your model should look like the one shown below:



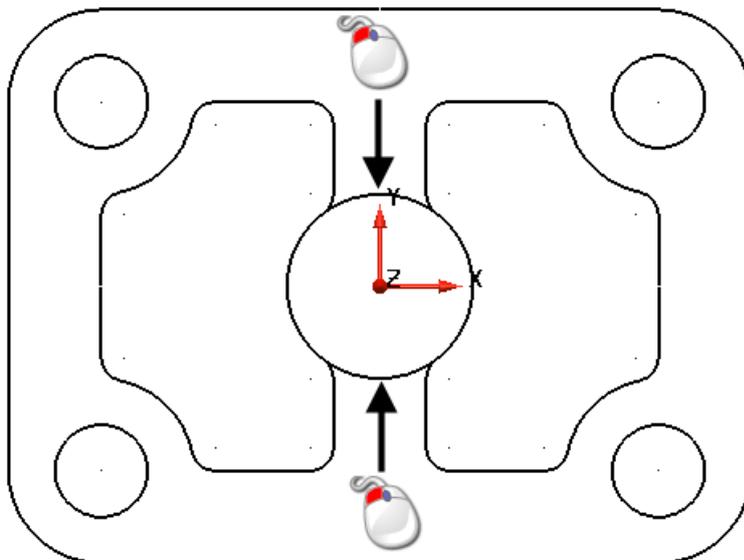
10. Limit wireframes - 2

- 1 Click **Show General Edits Options** .

2 Click **Interactively Limit Wireframes** .



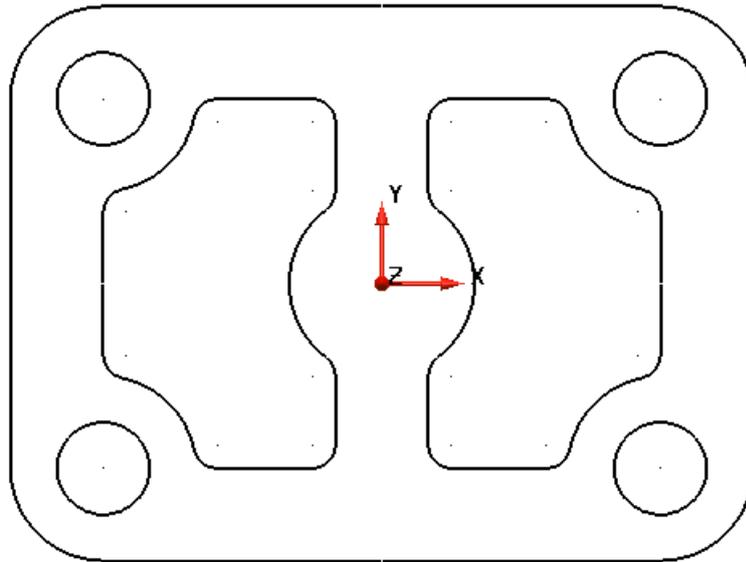
3 Click the positions shown below:



4 Click **Select** .

5 Click anywhere in the graphics window to deselect the model.

Your drawing should look as follows.

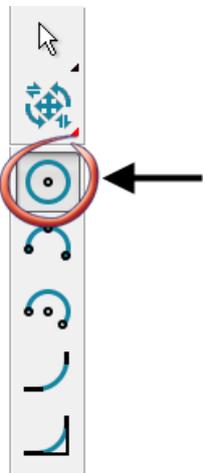


11. Create a full arc - 2

- 1 Click **Arc** .



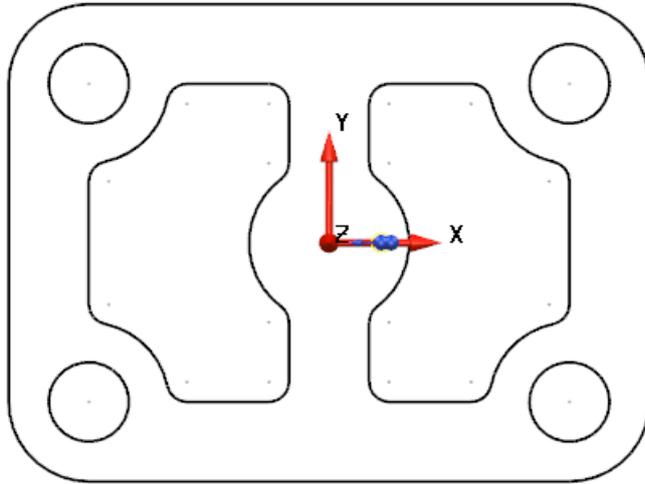
- 2 Click **Create a Full Arc** .



- 3 Click in the **Command Input** box in the **Status Bar** and type:
r 0.125
- 4 Press **Enter**.
0.650 0 0

5 Press **Enter**.

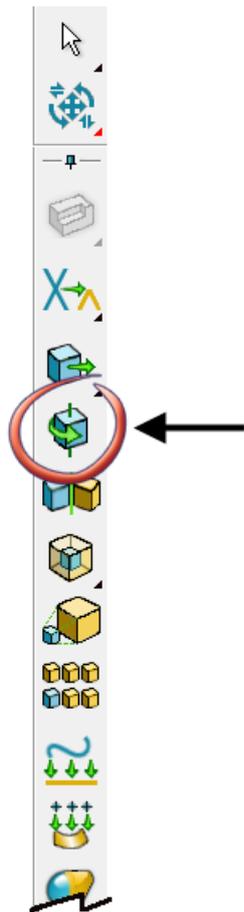
6 Click **Select** .



12. Copy and rotate the arc

1 Click **Show General Edits Options** .

- 2 Click **Rotate items** .

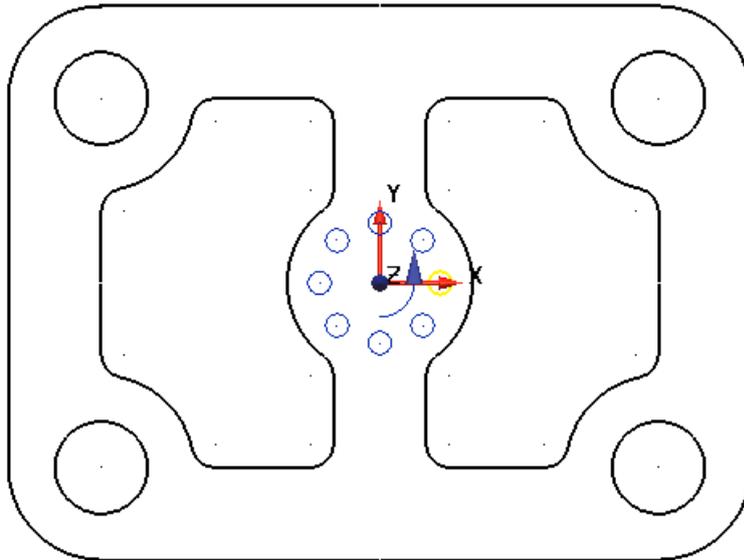


The **Rotate** toolbar is displayed.

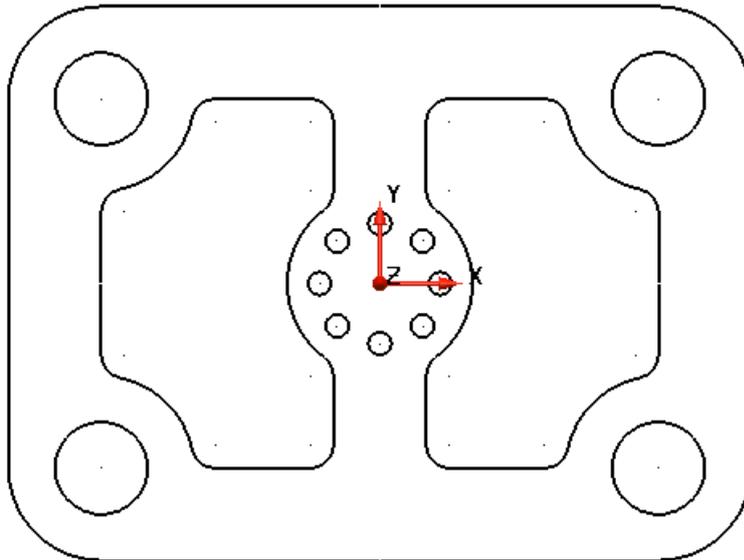


- 3 Click **Copy the selected items** .
- 4 Enter **7** in **Copies**.
- 5 From the **Angle** drop-down list, click **45**.

A preview of the rotation is drawn on the model.

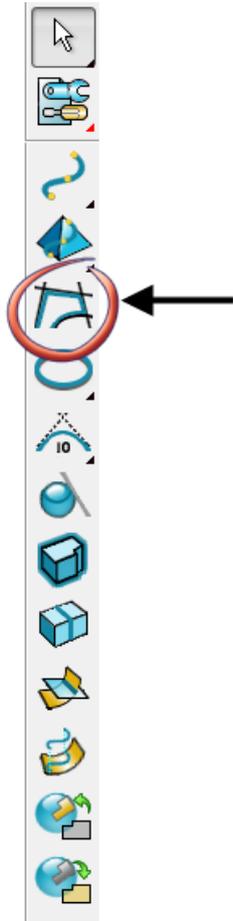


- 6 Click **Apply** .
- 7 Click **Dismiss** .
- 8 Click **Select** .
- 9 Click anywhere in the graphics window to deselect the model.



13. Create composite curves

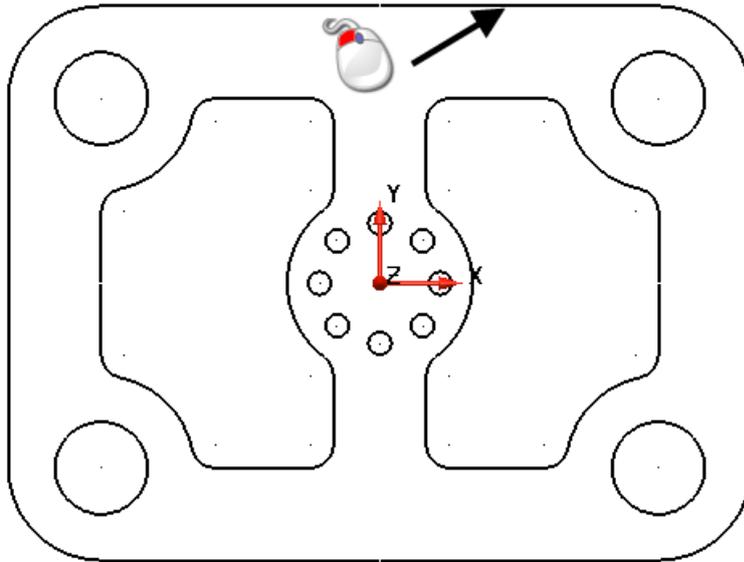
- 1 Click **Create a Composite Curve by tracing** .



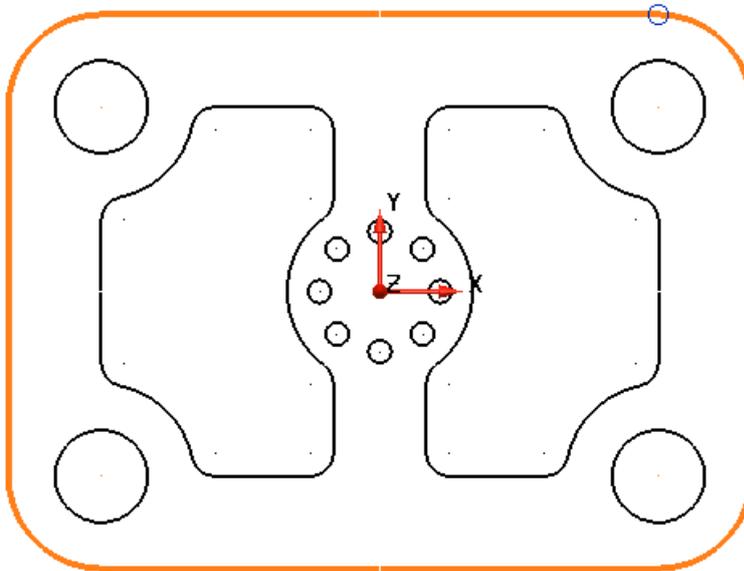
The **Create Composite Curve** toolbar is displayed.



2 Click the line shown below:

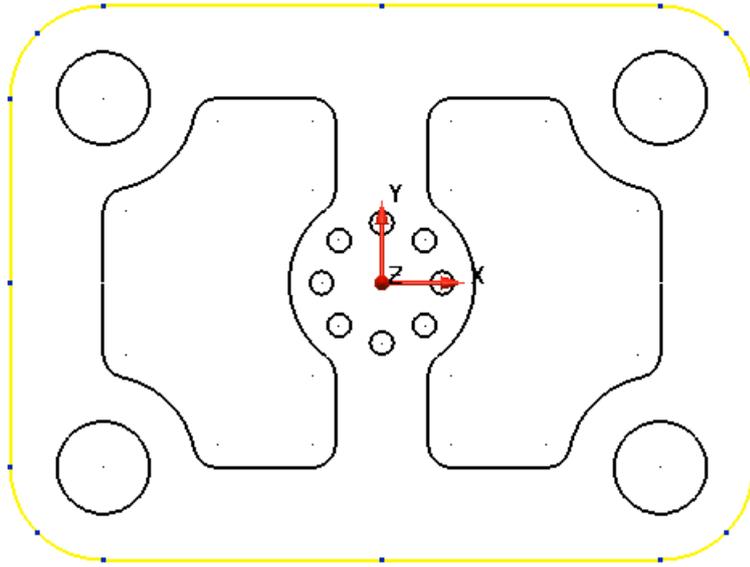


A composite curve is traced around the closed outline of the model.



3 Click **Save** .

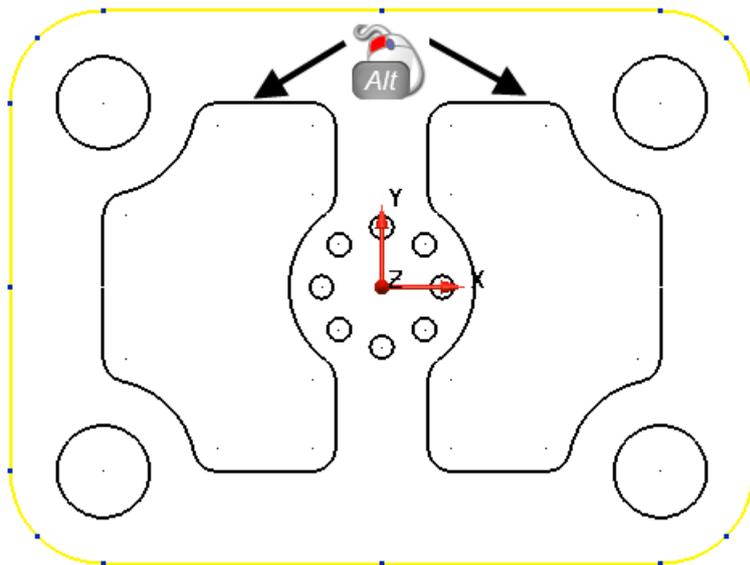
- 4 Click **Eject** .



- 5 Hold down the **ALT** key and click the left mouse button, as shown below.



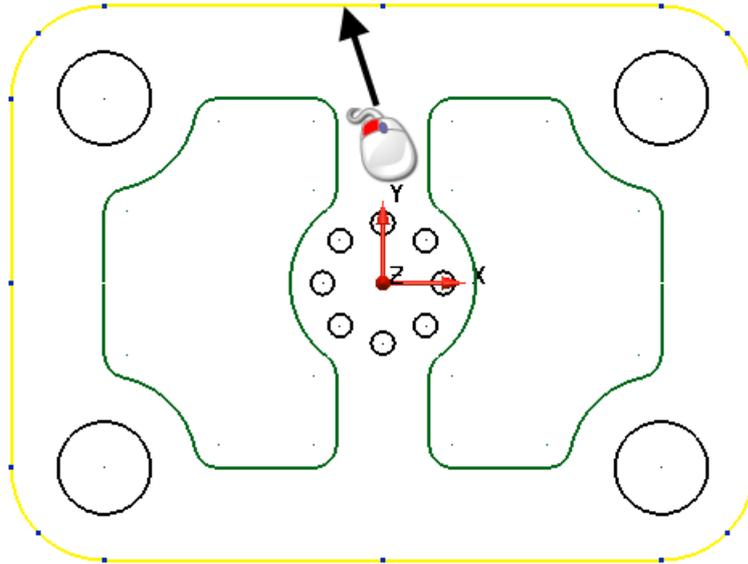
This is the shortcut for creating a composite curve.



- 6 Click anywhere in the graphics window to deselect the model.

14. Create a solid extrusion

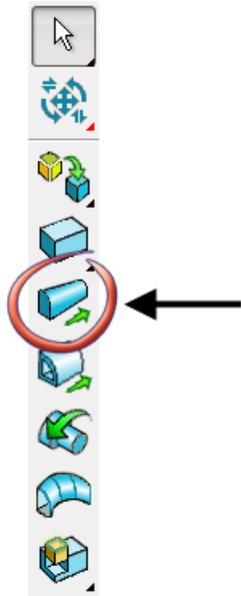
- 1 Click the curve:



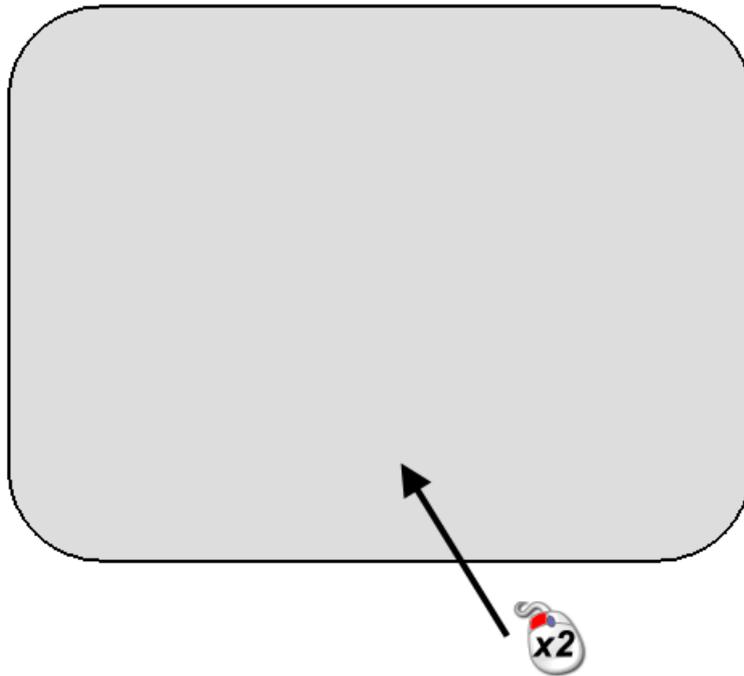
- 2 Click **Solid** 



- 3 Click **Solid Extrusion** 

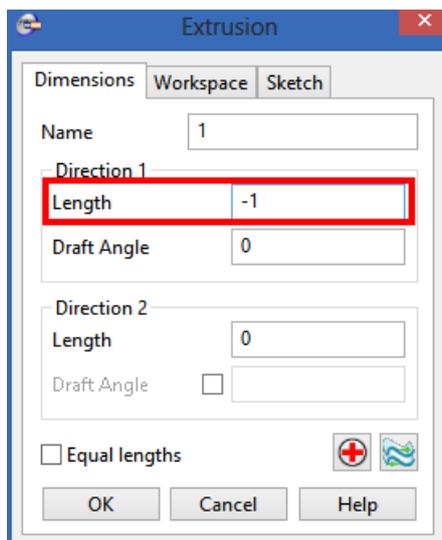


- 4 Double click the extruded solid.



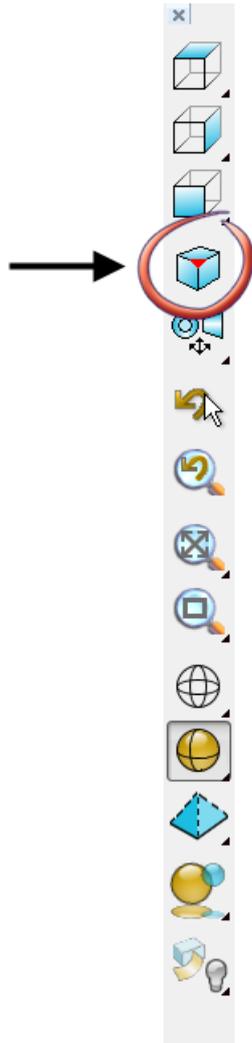
The Extrusion dialog is displayed.

- 5 Enter **-1** in **Length** of Direction 1.

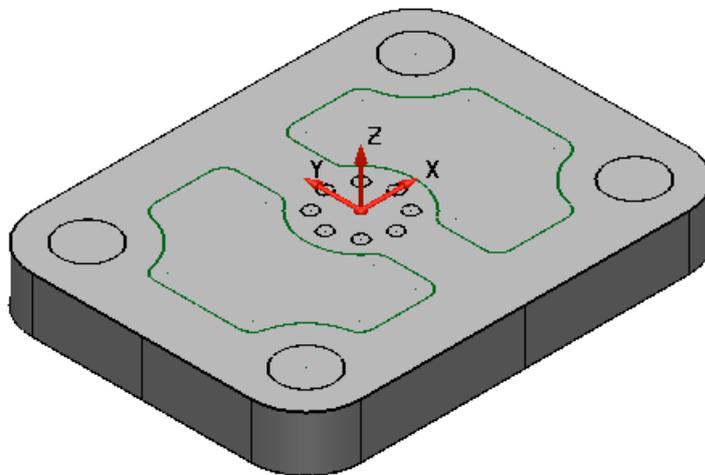


- 6 Click **OK**.

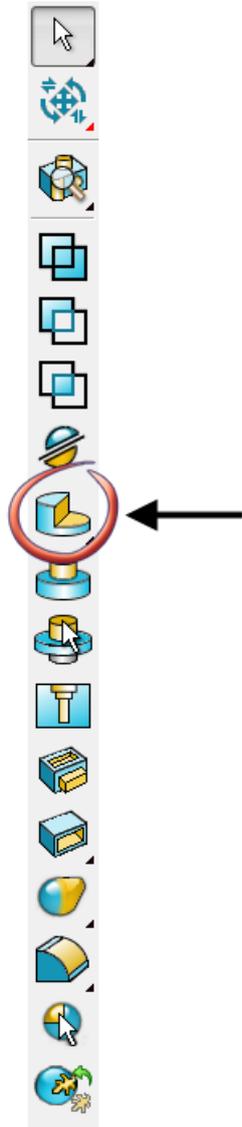
7 Click **View ISO1** .



8 Click anywhere in the graphics window to deselect the model.

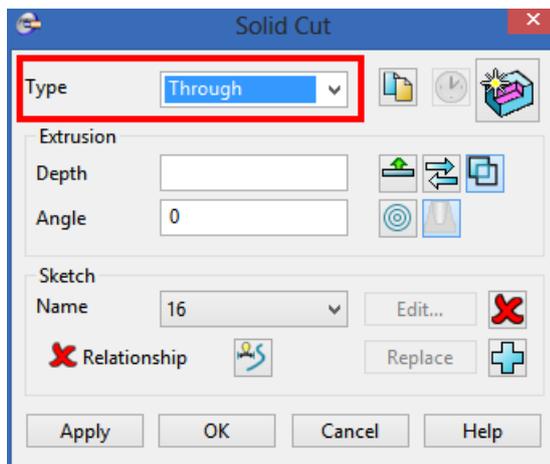


- 4 Click **Create Solid Cut Feature** .

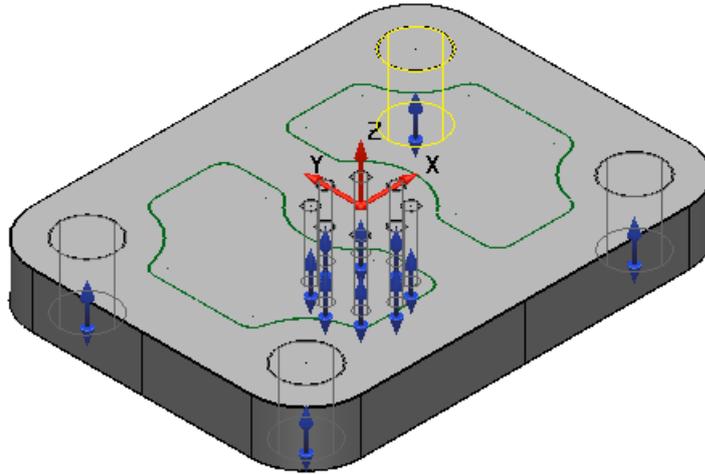


The Solid cut dialog is displayed.

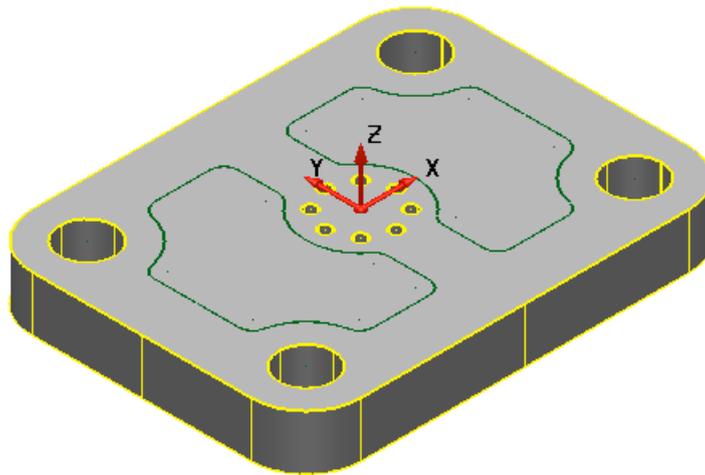
- 5 From the **Type** drop-down list, click **Through**.



Preview graphics are drawn on the model.

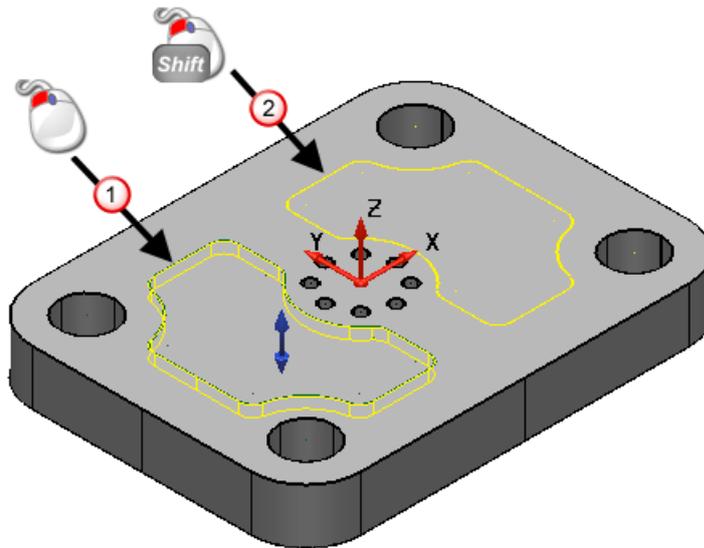


6 Click **Apply**.

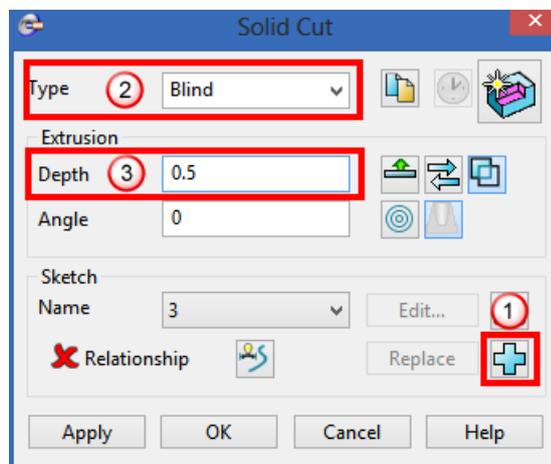


7 Click the composite curve.

- 8 Hold down the **Shift** key and click the composite curve.

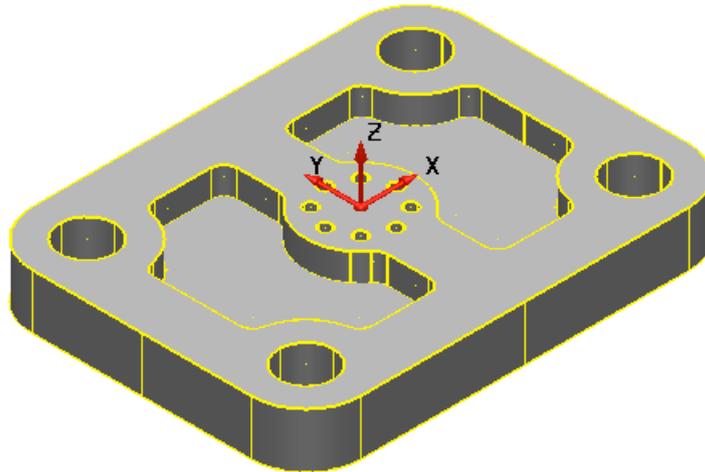


- 9 Click **Add the currently selected sketch** .
- 10 From the **Type** drop-down list, click **Blind**.
- 11 Enter a **Depth** of **0.5**.



- 12 Click **OK**.

Pockets have been cut into the solid.



13 Click anywhere in the graphics window to deselect the model.

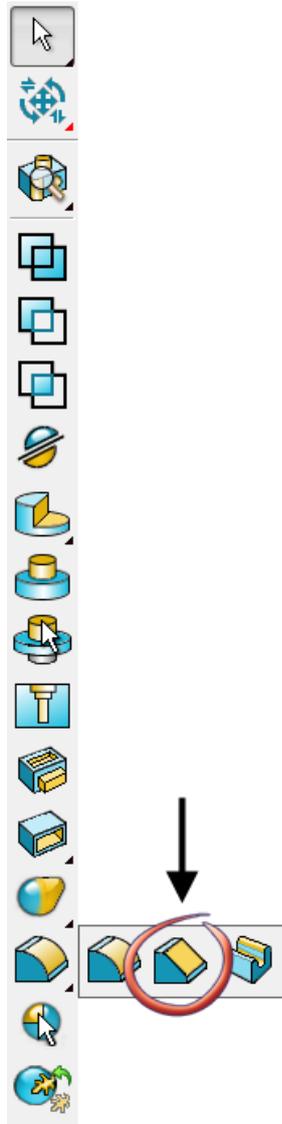
16. Create solid chamfer features

1 Click **Feature** .

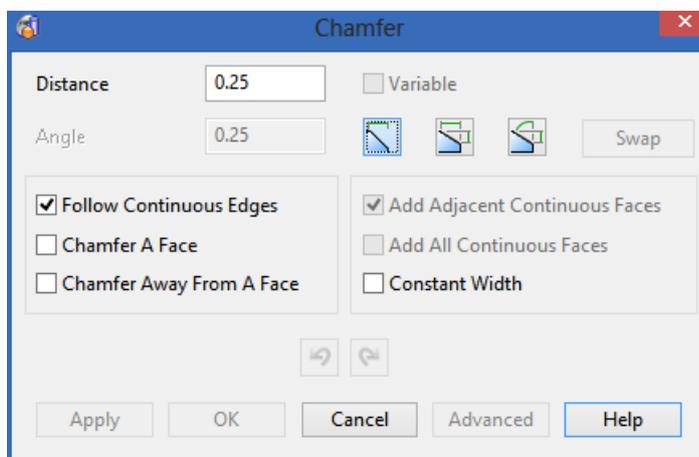


2 Click **Create solid fillet**  using the right mouse button

- 3 From the flyout click **Create solid chamfer** .

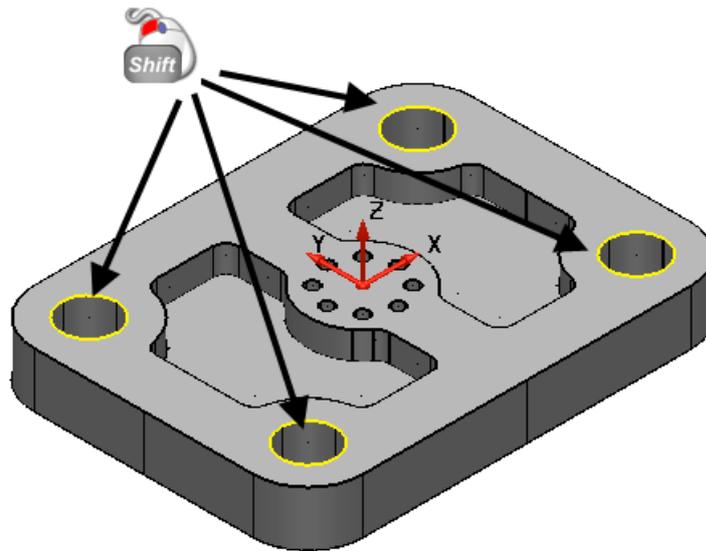


The Chamfer dialog is displayed.

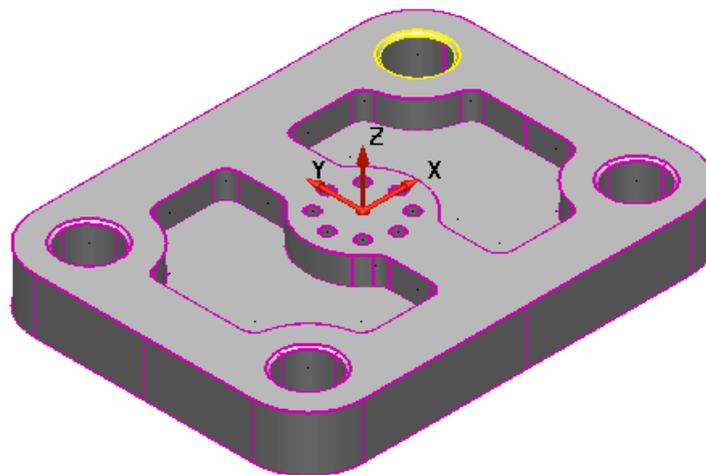


- 4 Enter a **Distance** of **0.050**.

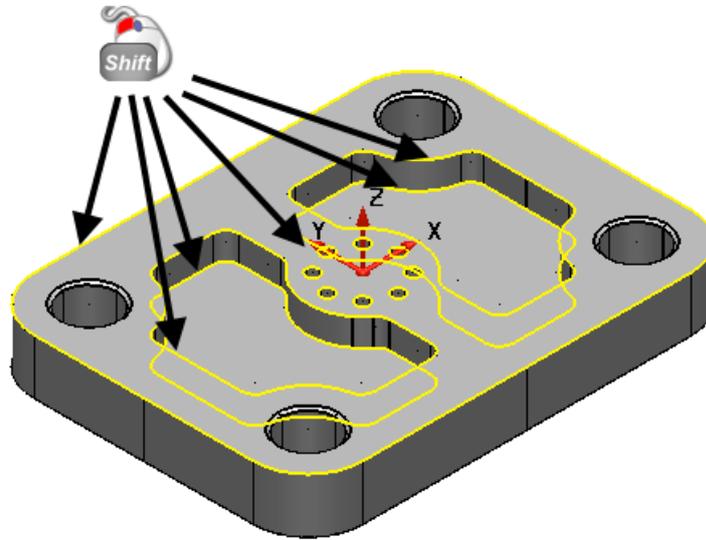
- 5 Hold down the **Shift** key and click the 1 diameter cuts.



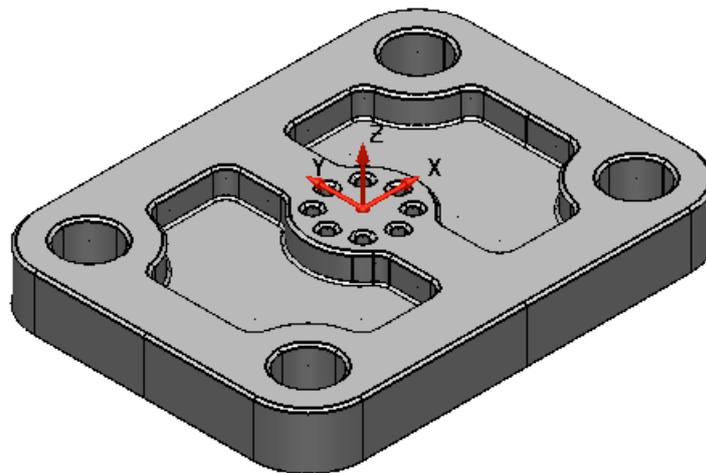
- 6 Click **Apply**.



- 7 Hold down the **Shift** key, click the edges of the other cuts and the edge of the extrusion.



- 8 Click **OK**.
- 9 Click anywhere in the graphics window to deselect the model.

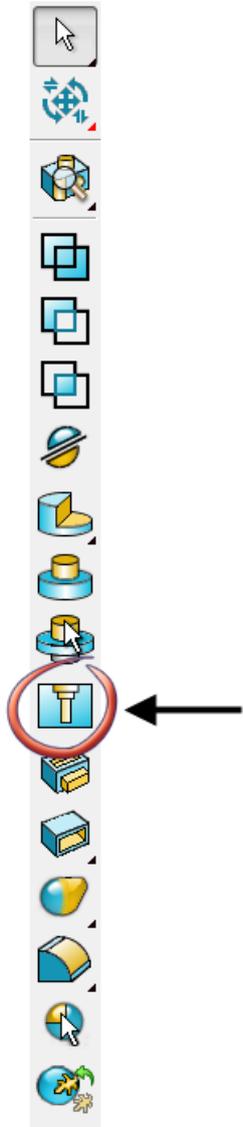


17. Create solid tapped holes

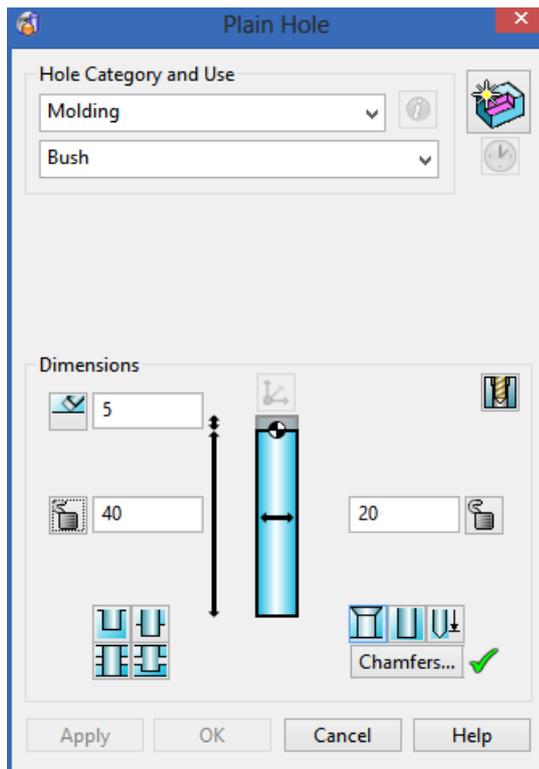
- 1 Click **Feature** .



2 Click **Create a Hole** .

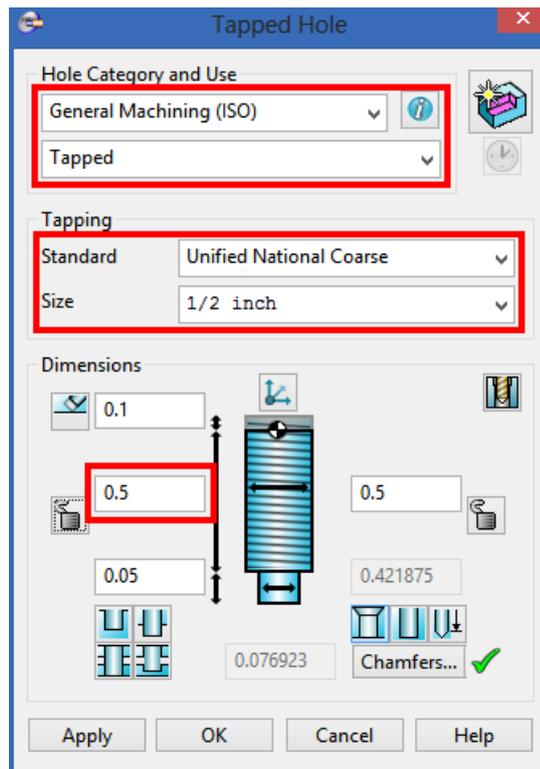


The Hole dialog is displayed.



- 3 Click in the **Command Input** box in the **Status Bar** and type:
0 2.5 0
- 4 Press **Enter**.
- 5 Click **Hole category** of **General Machining (ISO)** from the drop-down list.
- 6 Click **Use of Tapped** from the drop-down list.
- 7 Click **Standard** of **Unified National Course** from the drop-down list.
- 8 Click **Size** of **1/2 inch** from the drop-down list.

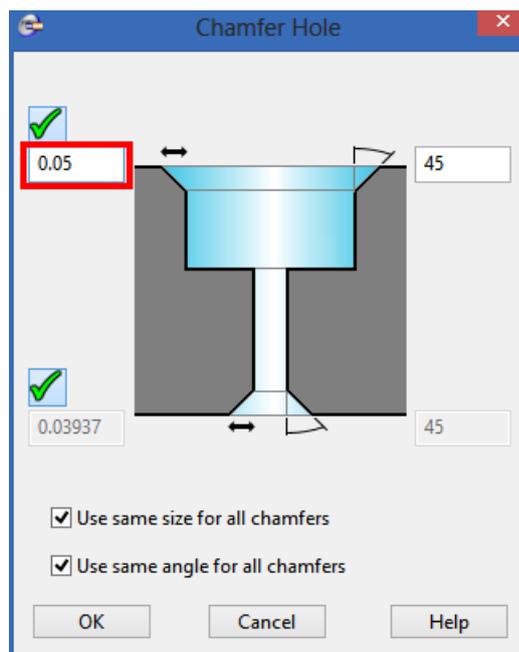
9 Enter **0.5** in **Set tapping depth**.



10 Click **Chamfers**.

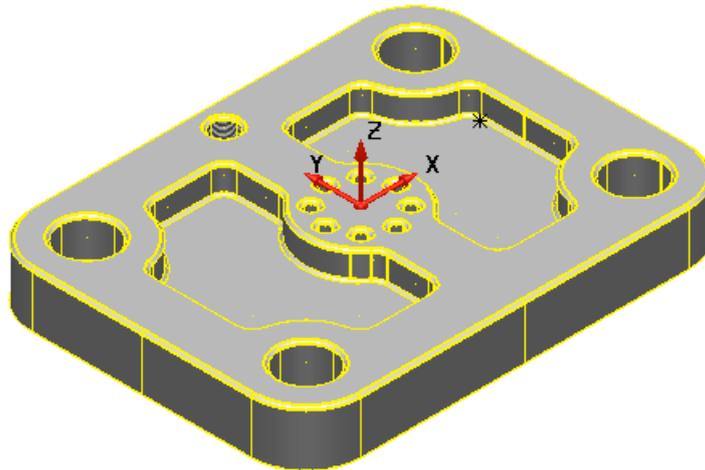
The Chamfer dialog is displayed.

11 Enter **0.05** to **Set the hole base chamfer size**.



12 Click **OK**.

13 Click **Apply**.



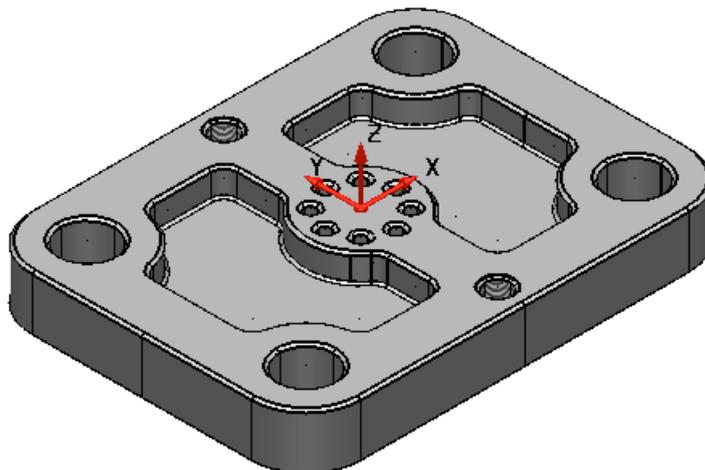
14 Click in the **Command Input** box in the **Status Bar** and type:

0 -2.5 0

15 Press **Enter**.

16 Click **OK**.

17 Click anywhere in the graphics window to deselect the model.

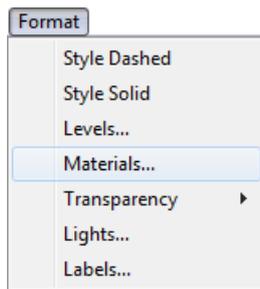


18. Change the material

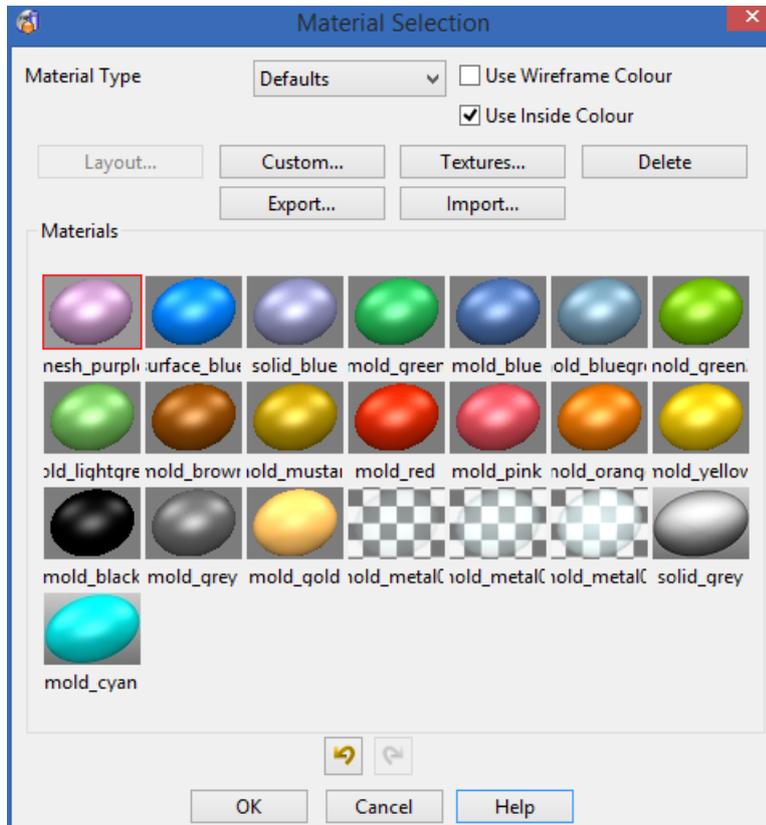
1 Click the solid.

2 From the main menu, click **Format**.

3 Click **Materials...**



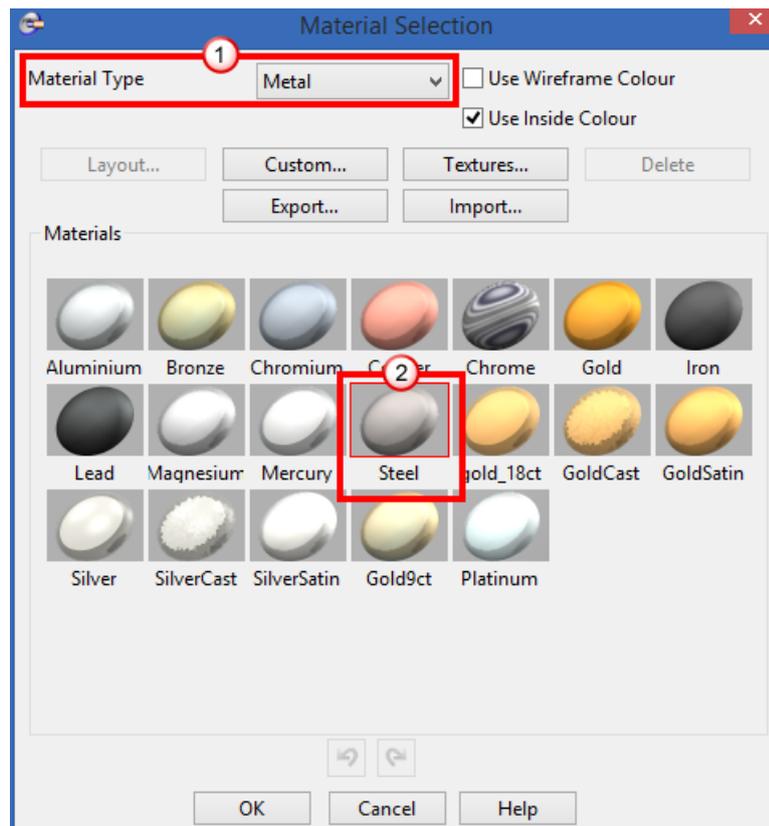
The **Material Selection** dialog is displayed.



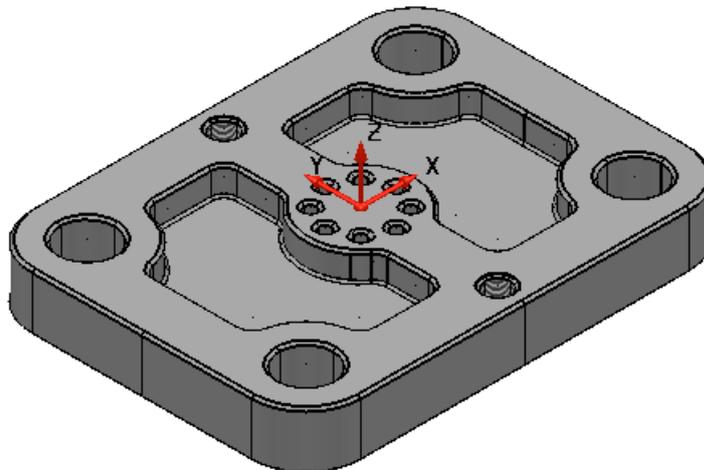
4 Do the following:

- 1 From the **Material type** drop-down list, click **Metal**.

- 2 Click **Steel**.

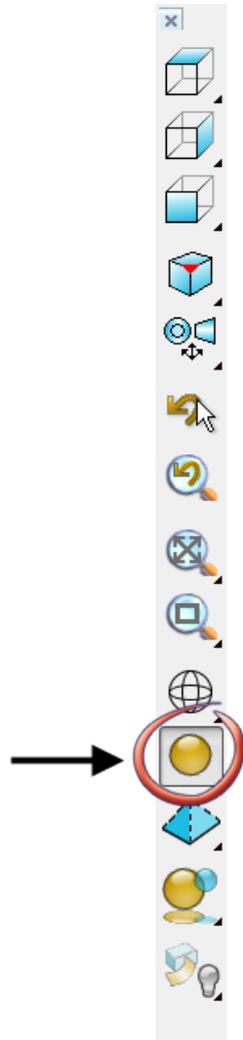


- 5 Click **OK**.
- 6 Click anywhere in the graphics window to deselect the model.



19. Change the shading option

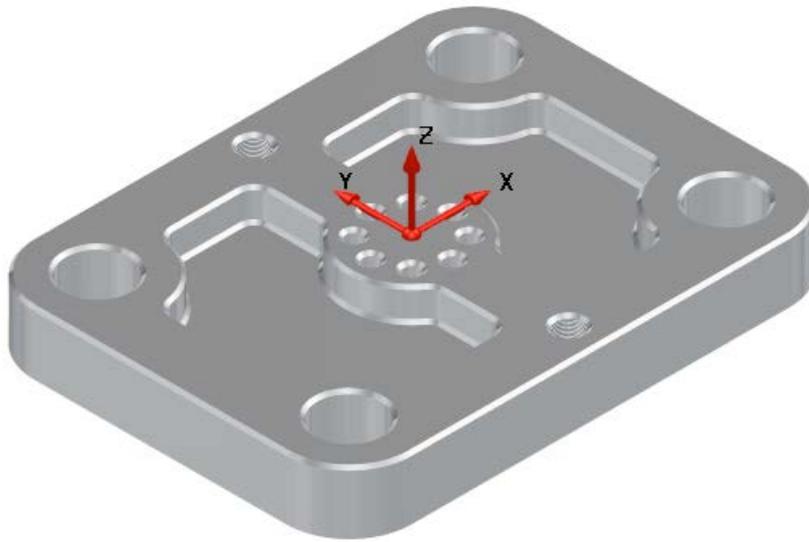
- 1 Click **Shaded View** , using the right mouse button.



- 2 From the flyout, click **Enhanced Shading** 



3 Your model should look like the one shown below:



Tutorial - Designing a pin

Use this tutorial to create a solid model of a pin using by:

- checking the drawing standards of the model
- creating and editing lines
- creating a composite curve
- creating a solid revolution from the composite curve
- creating a wireframe polygon
- creating a solid boss feature
- creating a solid hole feature
- creating solid chamfers
- changing the material of the model

1. Check the tolerances of the models

This tutorial is going to be completed using the ANSI drawing standard.

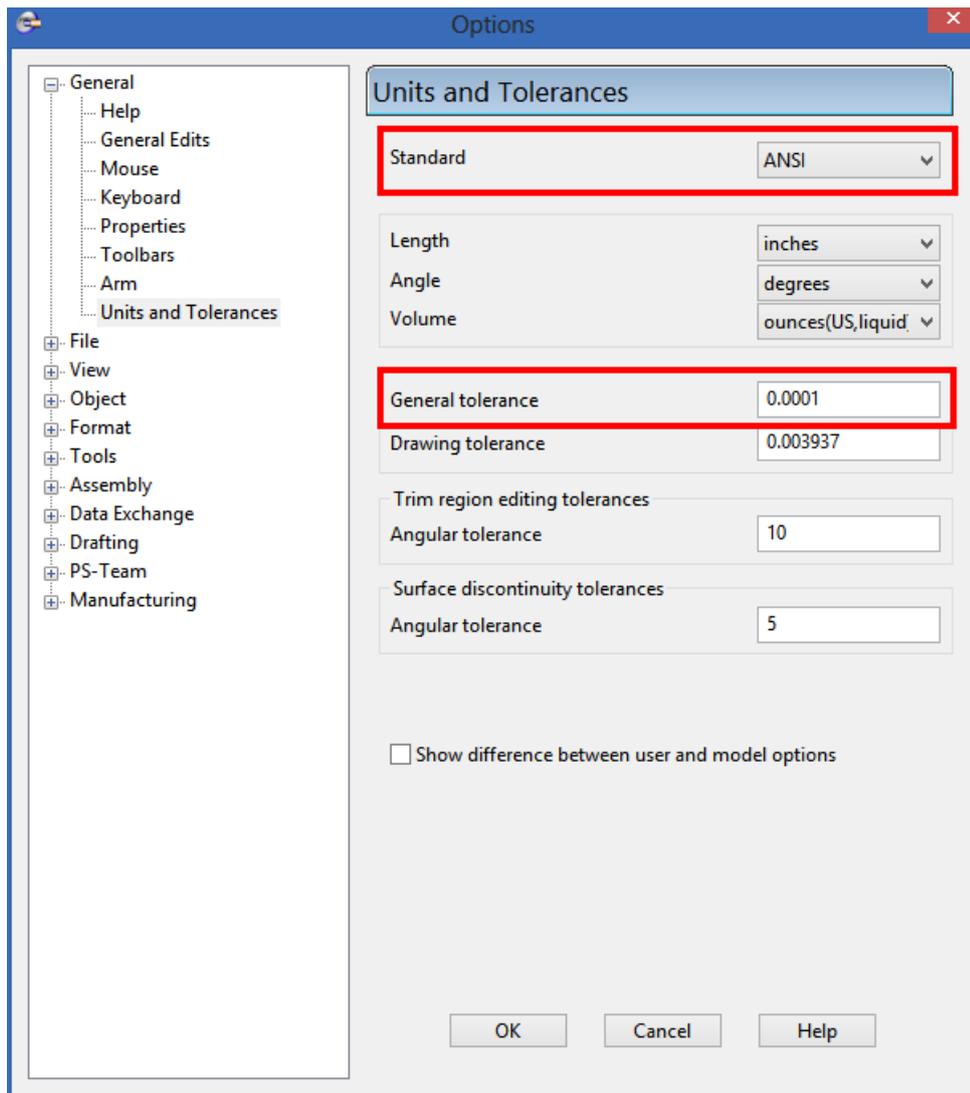
- Standard = ANSI
- Length = inches
- Angle = degrees
- Volume = ounces(US, liquid)

- 1** Click **Open new model** .

An empty model is opened and the **Standard** should be set to **ANSI**.

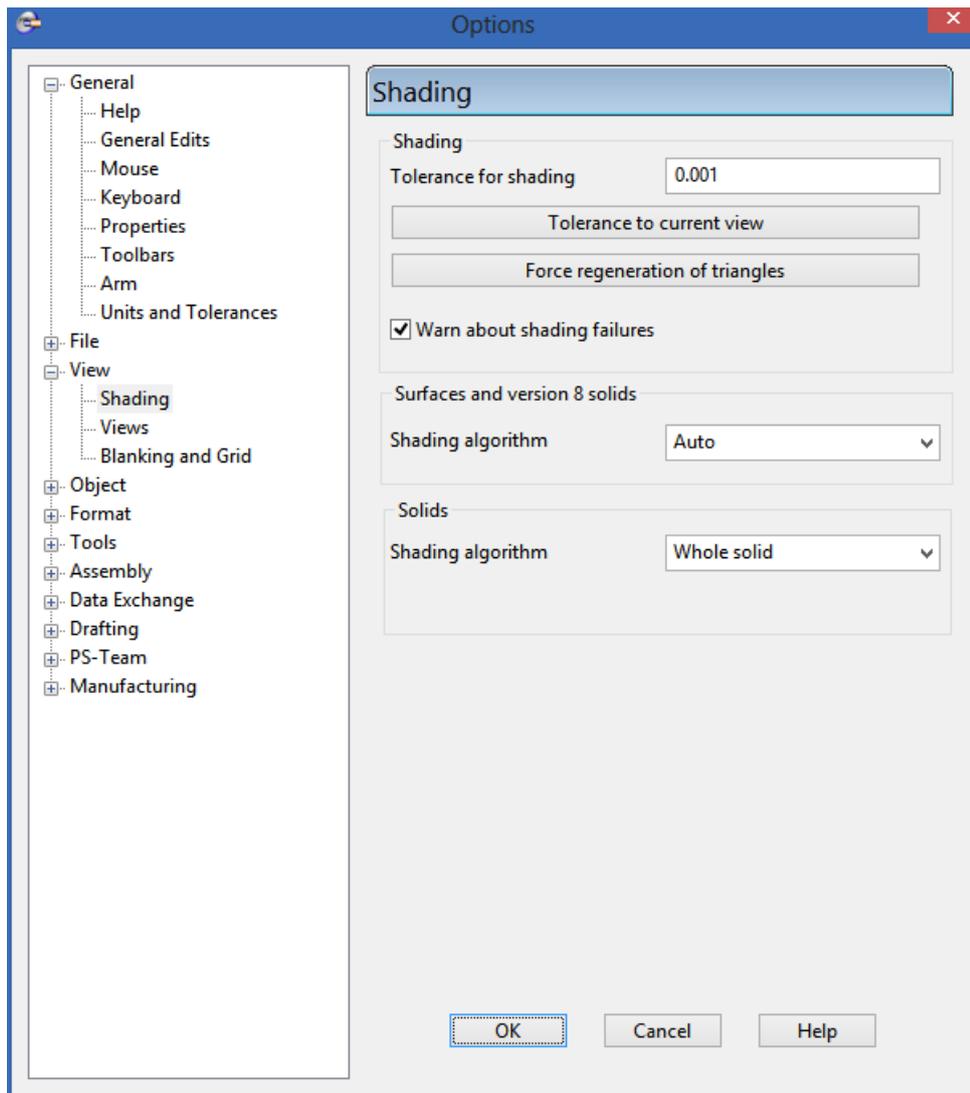
- 2** Click **Tools > Options** from the Main menu.
- 3** Click **Units and Tolerances**.

The values should be as displayed below:



- 4 If the values aren't correct change to match the above image.
General tolerance = 0.0001.
- 5 Click **View > Shading**.

The values should be as displayed below:



- 6 If the values aren't correct change to match the above image.
Tolerance for shading = 0.001.
- 7 Click **Force Regeneration of Triangles**.
- 8 Click **OK**.

2. Create a workplane

- 1 Click **Workplane** .

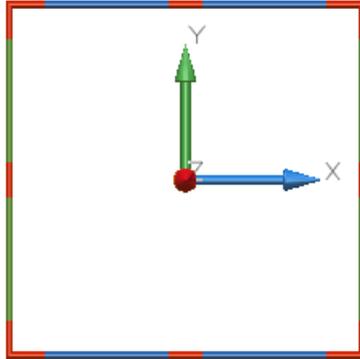


- 2 Click in the **Command Input** box in the **Status Bar** and type:

0 0 0



- 3 Press **Enter**.



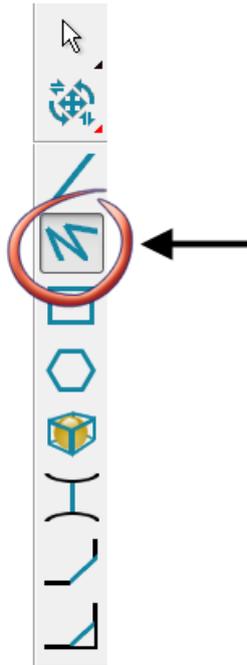
- 4 Click anywhere in the graphics window to deselect the model.

3. Create lines

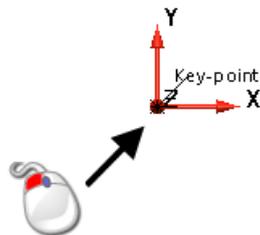
- 1 Click **Line** .



- 2 Click **Continuous Lines** .



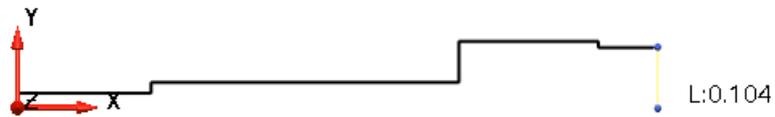
- 3 Move the cursor over the axis of the workplane.
4 When *Key-point* displays, click the left mouse button.



This creates the first point of the line.

- 5 Click in the **Command Input** box in the **Status Bar** and type:
0 0.025 0
- 6 Press **Enter**.
0.23 0.025 0
- 7 Press **Enter**.
0.23 0.0437 0
- 8 Press **Enter**.

Your model should look like the one shown below:



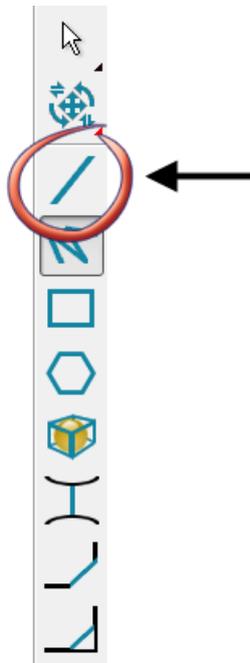
18 Click anywhere in the graphics window to deselect the model.

4. Create a chamfer

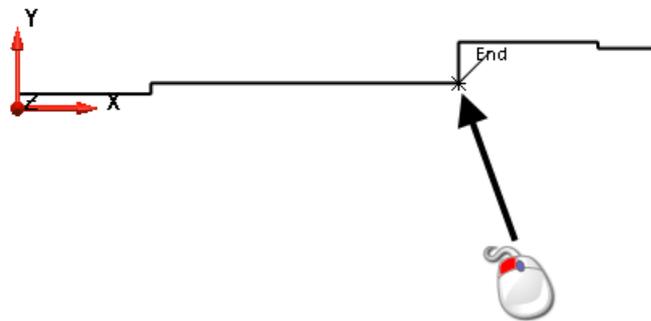
1 Click **Line** .



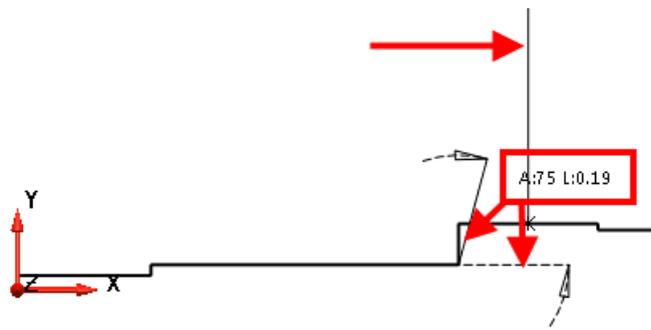
2 Click **Create a single line** .



- 3 Move the cursor over the end of the line. When *End* displays, click the left mouse button.



- 4 Move the cursor around the model. Construction lines and dimensions activate and display.



- 5 When an angle of 60 is displayed and the line crosses the existing line, click the left mouse button.



- 6 Click **Select** .

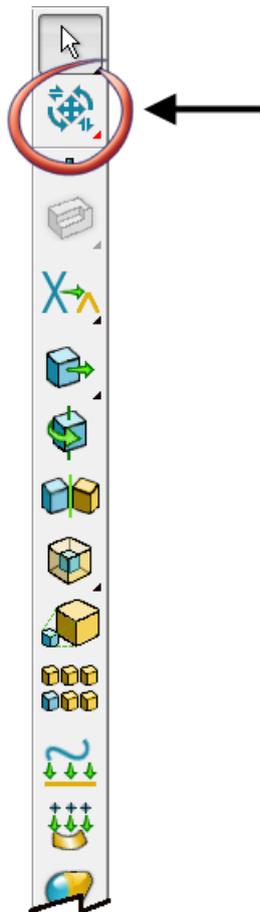
- 7 Click anywhere in the graphics window to deselect the model.



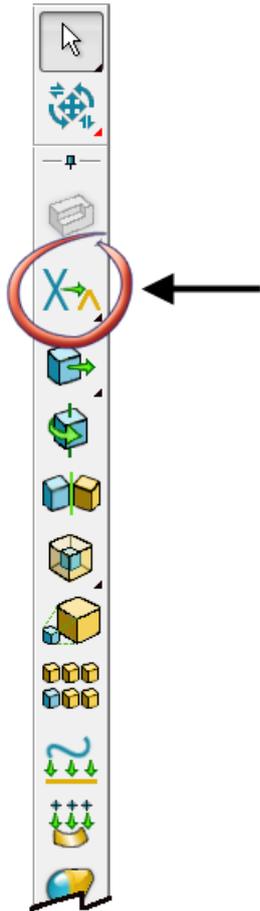
5. Limit the lines

- 1 Click **Show General Edits Options** .

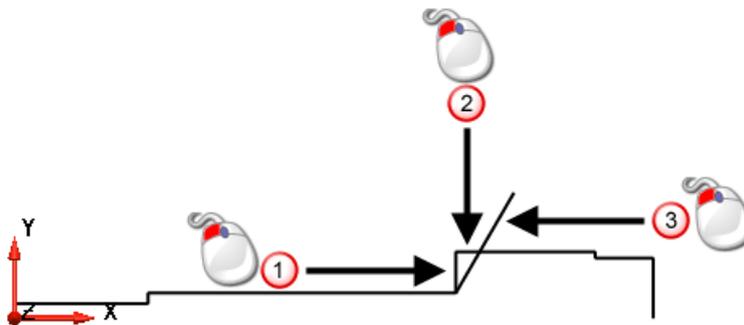
 *Keep clicking the button until the toolbar shown below is displayed.*



2 Click **Interactively Limit Wireframes** .



3 Click the three positions shown below:

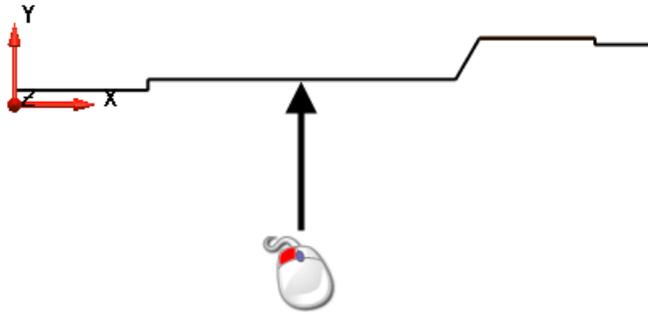


4 Click **Select** .

The **Create Composite Curve** toolbar is displayed.



- 3 Click the line shown below:



- 4 Click **Save** .

- 5 Click **Eject** .

The lines are combined into one composite curve.



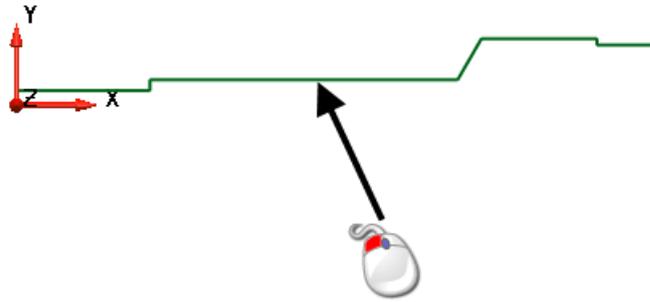
- 6 Click anywhere in the graphics window to deselect the model.

7. Create a solid revolution

- 1 From the **Status Bar** at the bottom of the screen, click the **X** principal plane.



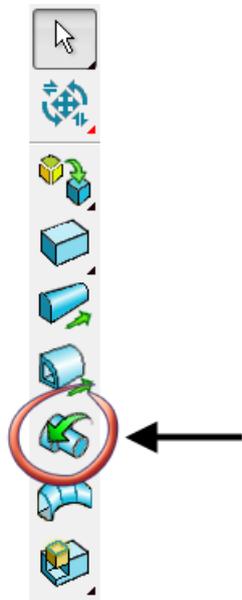
2 Click the composite curve.



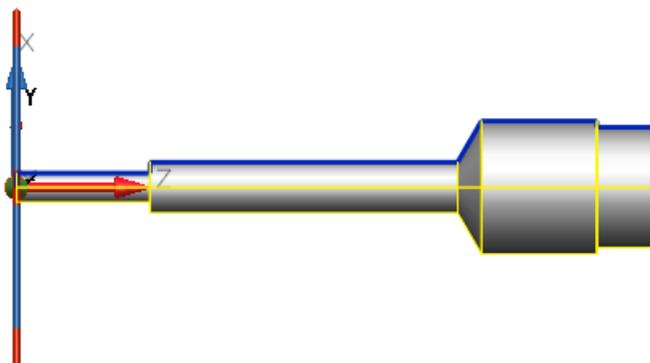
3 Click **Solid** .



4 Click **Solid Revolution** .



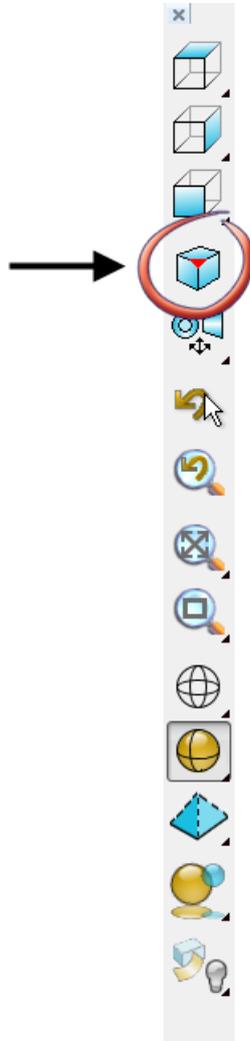
A solid revolution is created from the composite curve.



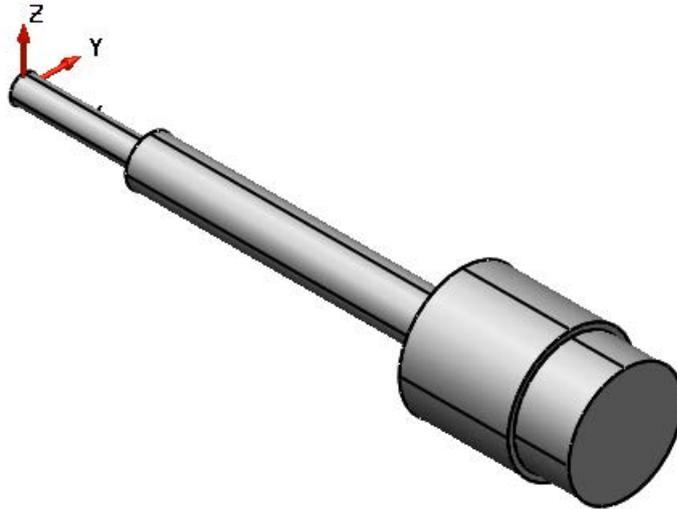
- 5 Click anywhere in the graphics window to deselect the model.

8. Create a wireframe polygon

- 1 Click **View ISO1** using the right mouse button .



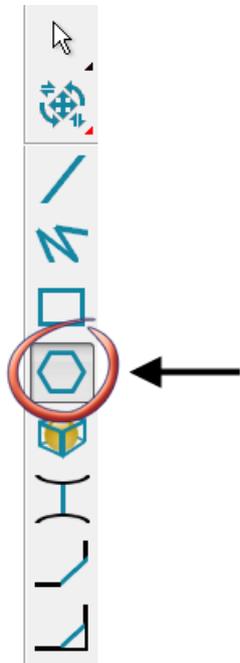
2 From the flyout, click **View ISO2** .



3 Click **Line** .

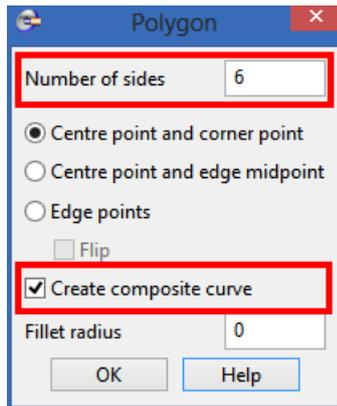


4 Click **Create a polygon** .

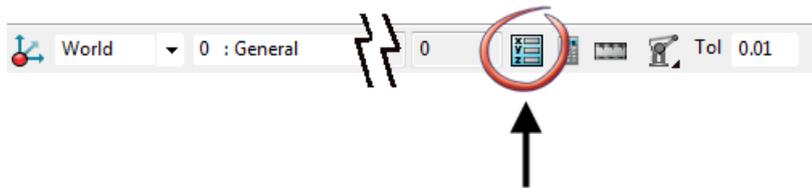


5 Enter **6** in **Number of sides**.

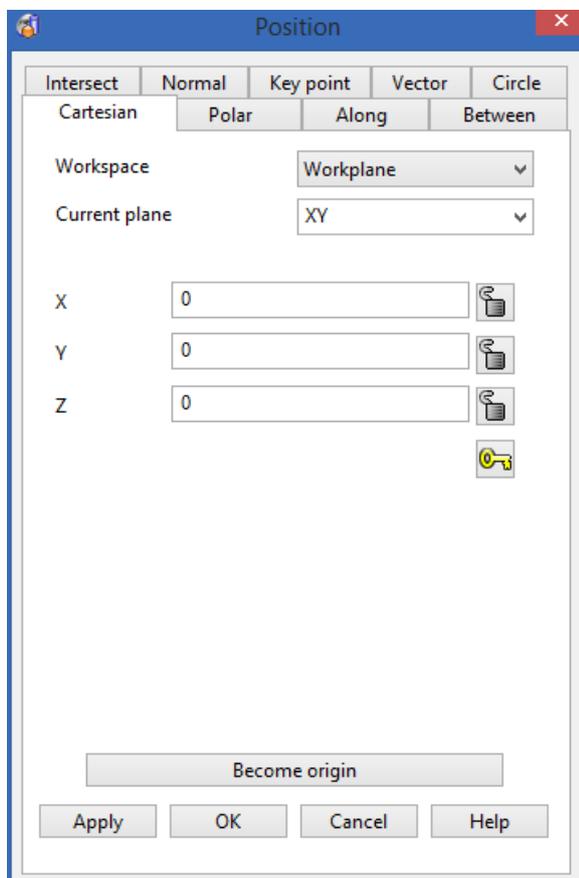
6 Click **Create composite curve**.



7 From the status bar, click **Open the Position dialog** .

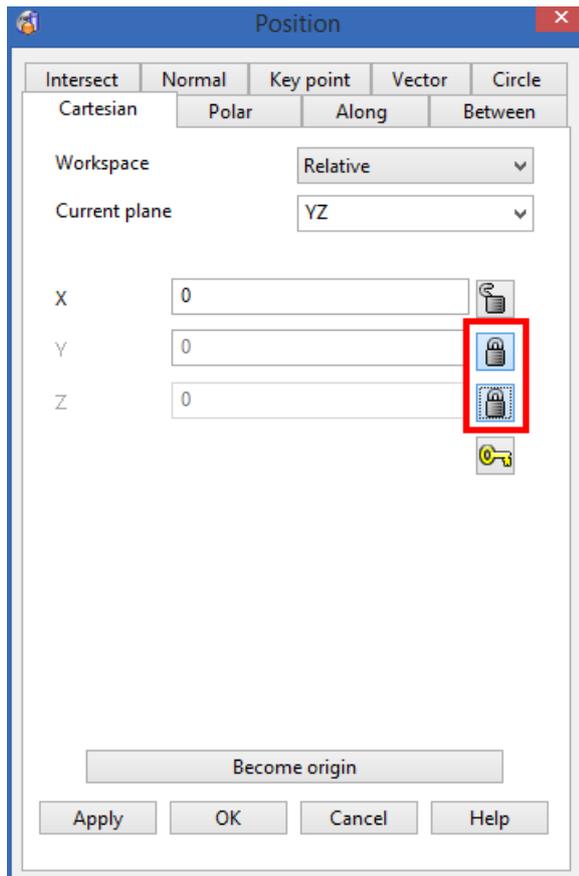


The **Position** dialog is displayed.

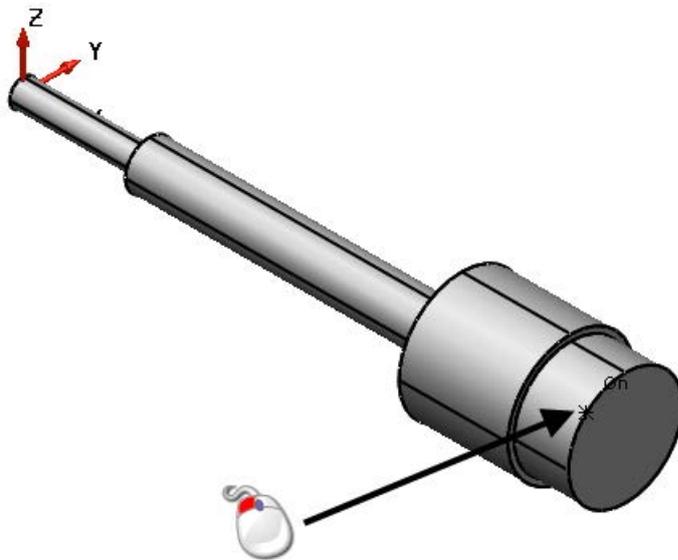


8 Click **Lock Y** .

9 Click **Lock Z** .

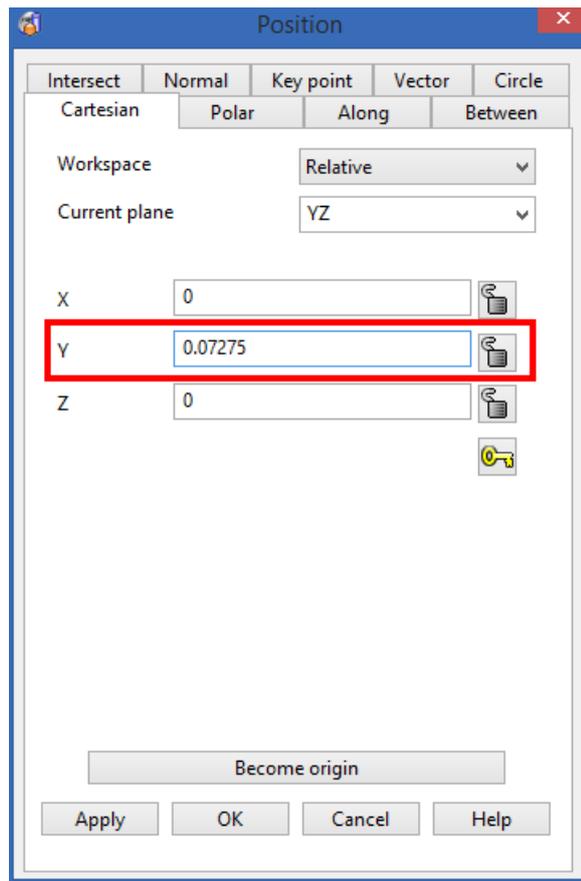


10 Click the position shown below;



11 Click **Apply**.

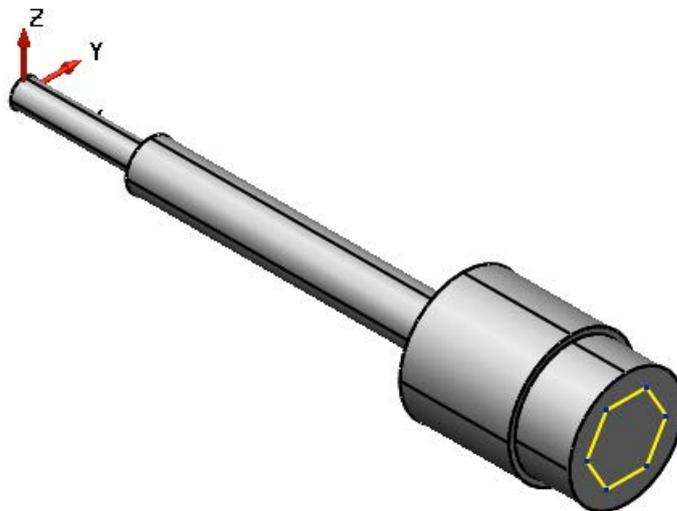
12 Enter **0.07275** in Y.



13 Click **OK**.

14 Click **OK**.

A polygon is created on the end face of the revolution.

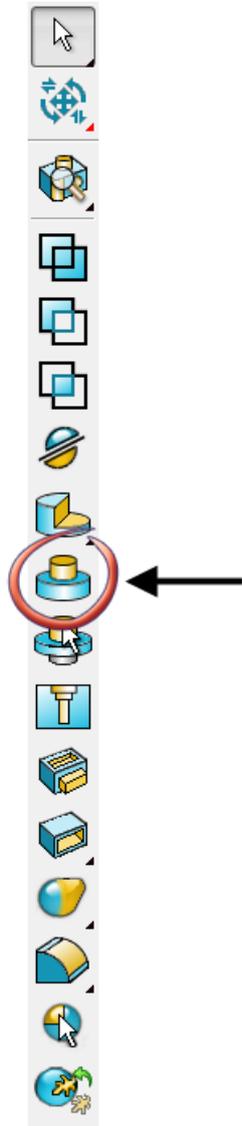


9. Create a solid boss feature

- 1 Click **Feature** .

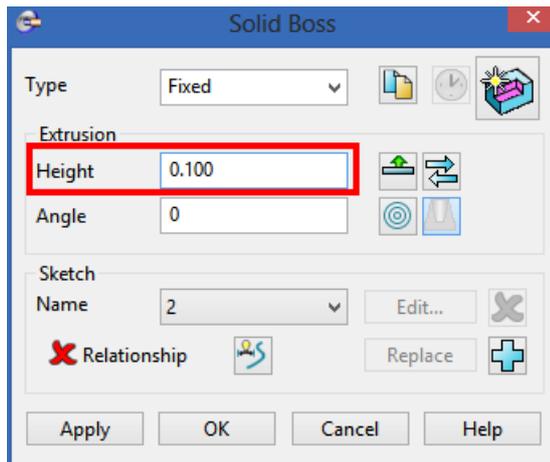


- 2 Click **Create solid boss feature** .



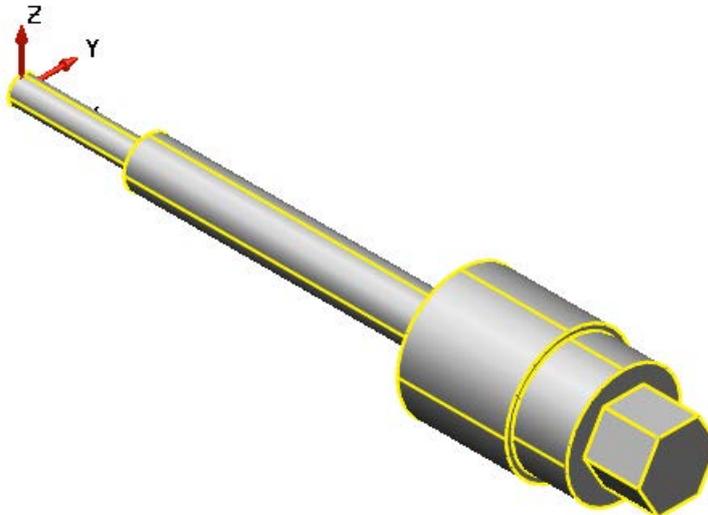
The Boss dialog is displayed.

- 3 Enter a **Height** of **0.100**.



- 4 Click **OK**.

- 5 Click **Select** .



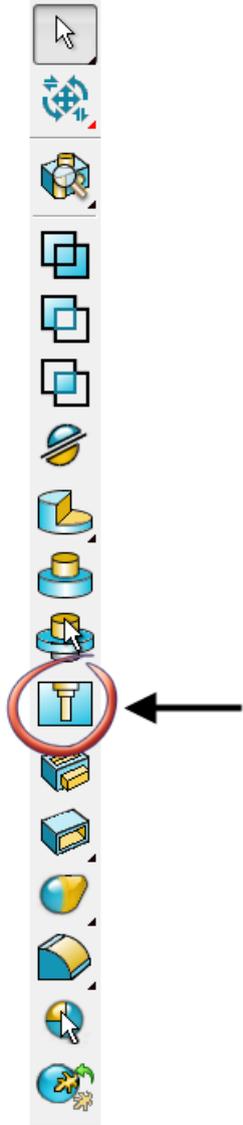
- 6 Click anywhere in the graphics window to deselect the model.

10. Create a solid tapped hole feature

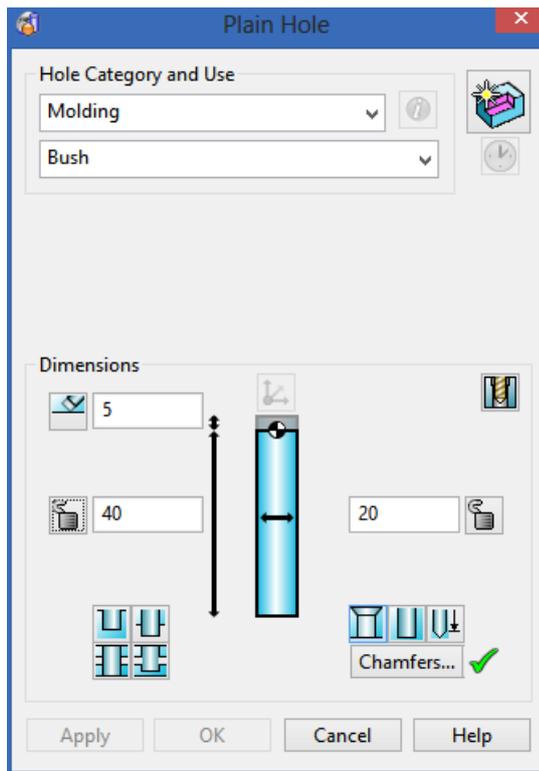
- 1 Click **Feature** .



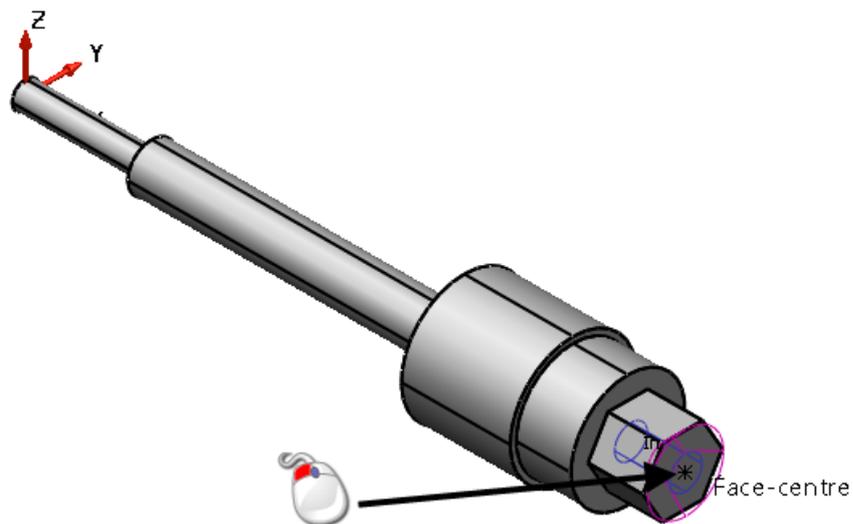
2 Click **Create a Hole** .



The Hole dialog is displayed.

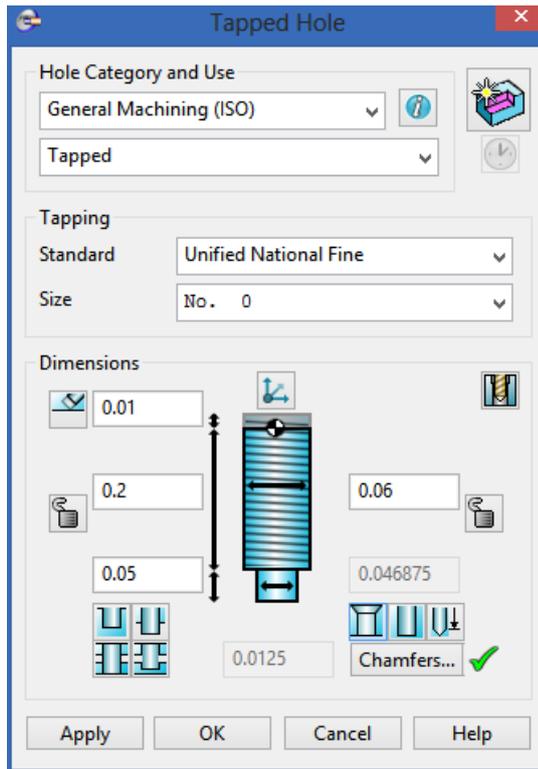


- 3 When *Face -centre* displays click the left mouse button.



- 4 Click **Hole category** of **General Machining (ISO)** from the drop-down list.
- 5 Click **Use** of **Tapped** from the drop-down list.
- 6 Click **Standard** of **Unified National Fine** from the drop-down list.

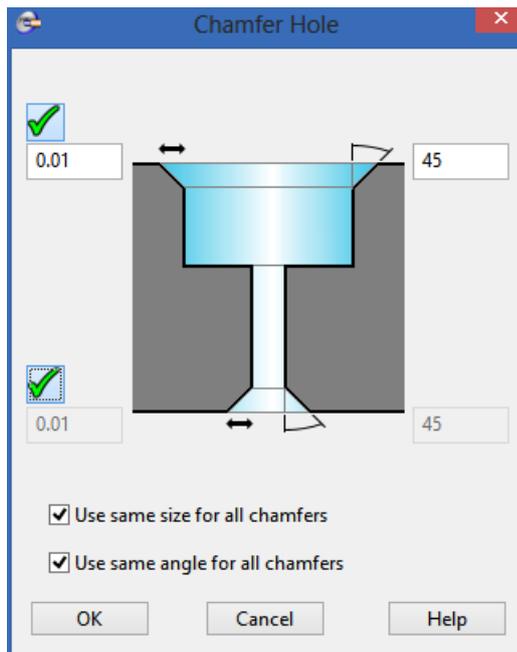
- 7 Click **Size** of **No. 0** from the drop-down list.



- 8 Click **Chamfers**.

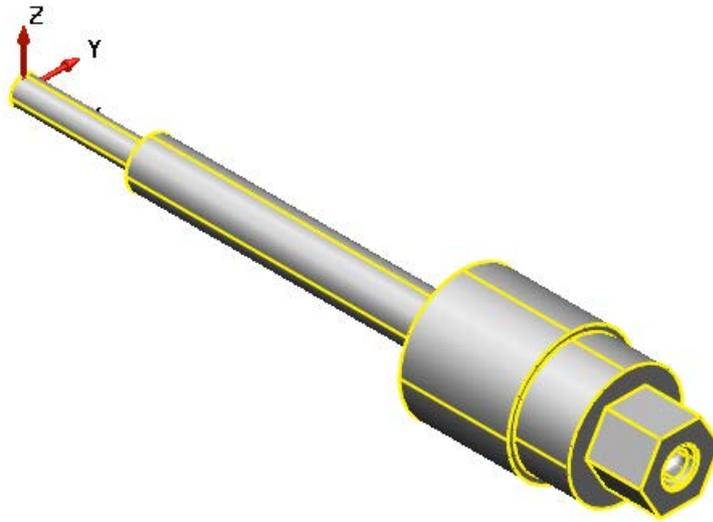
The Chamfer dialog is displayed.

- 9 Enter **0.01** to **Set the hole base chamfer size**.



- 10 Click **OK**.

11 Click **OK**.



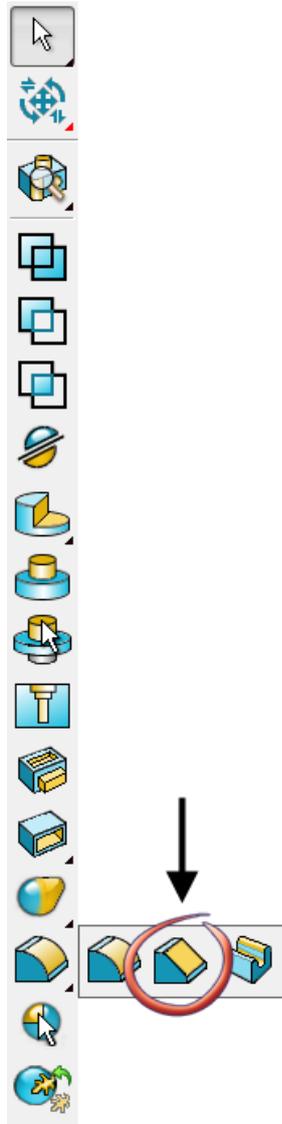
11. Create solid chamfer features

1 Click **Feature** .

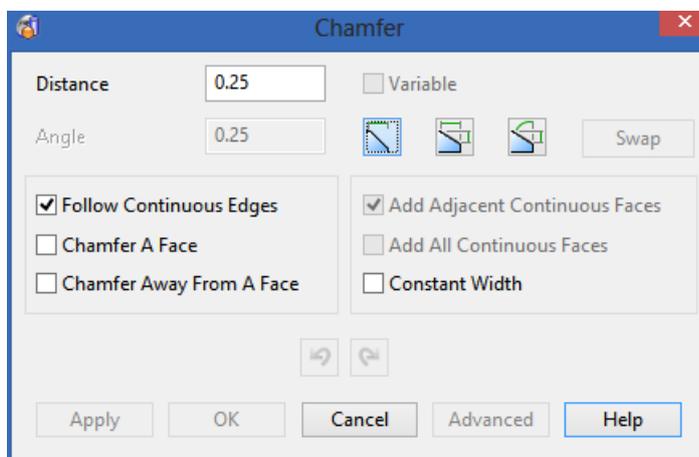


2 Click **Create solid fillet**  using the right mouse button

- 3 From the flyout click **Create solid chamfer** .

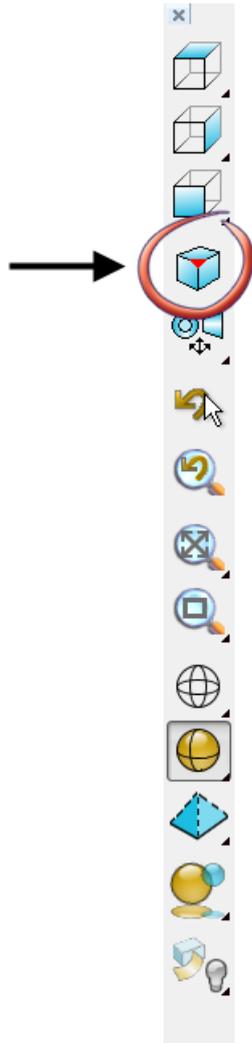


The Chamfer dialog is displayed.

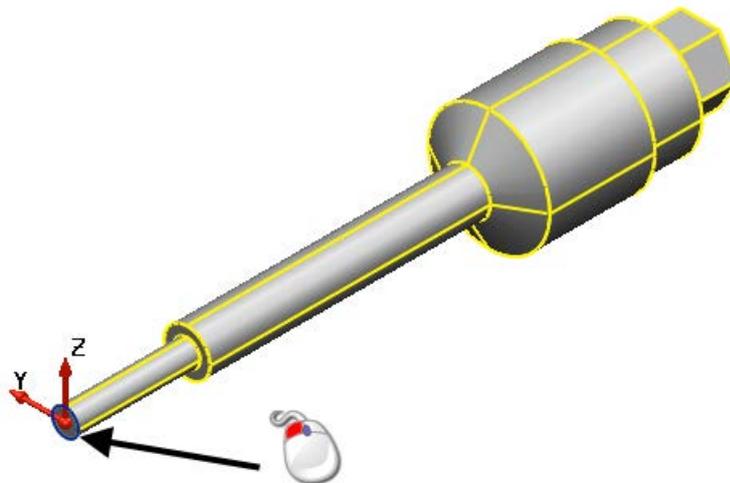


- 4 Enter a **Distance** of **0.003**.

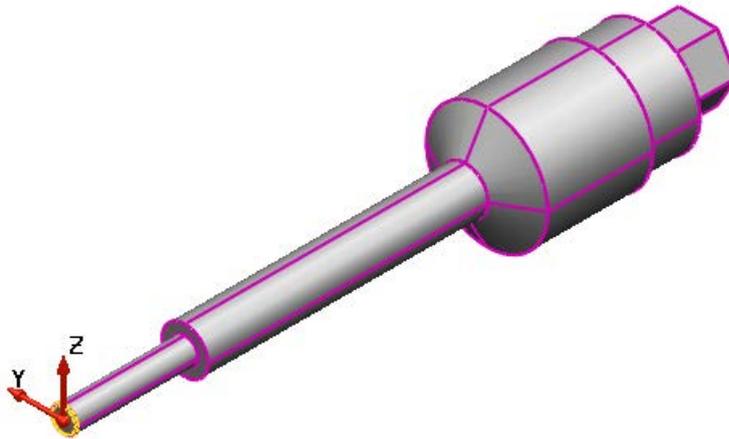
5 Click **View ISO1** .



6 Click the .050 diameter face.



7 Click **Apply**.

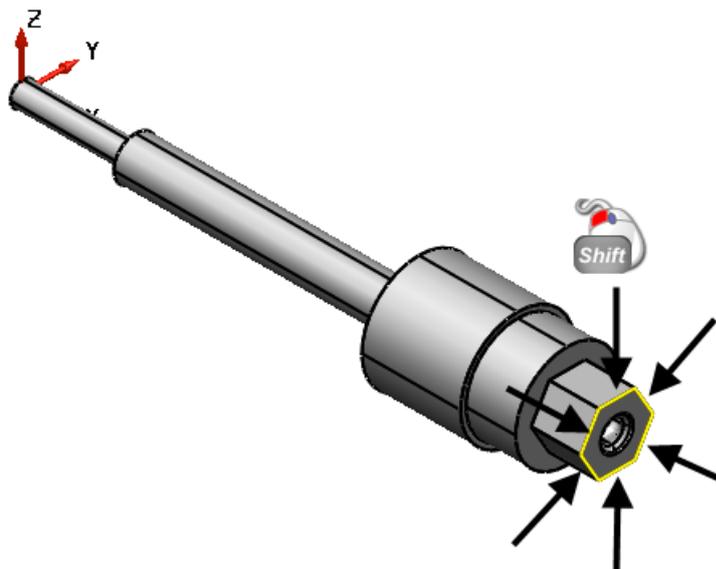


8 Enter a **Distance** of **0.005**.

9 From the flyout, click **View ISO2** .

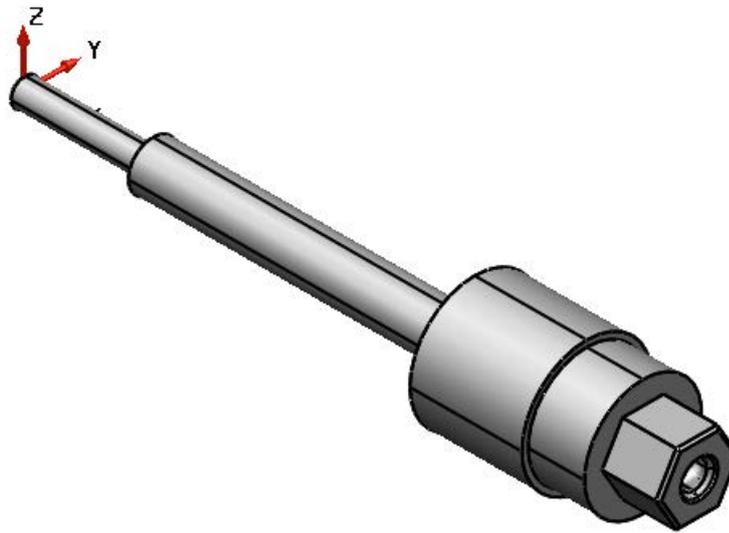


10 Hold down the **Shift** key and click all the edges of the hexagon face, shown below;



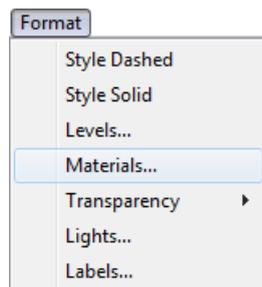
11 Click **OK**.

12 Click anywhere in the graphics window to deselect the model.

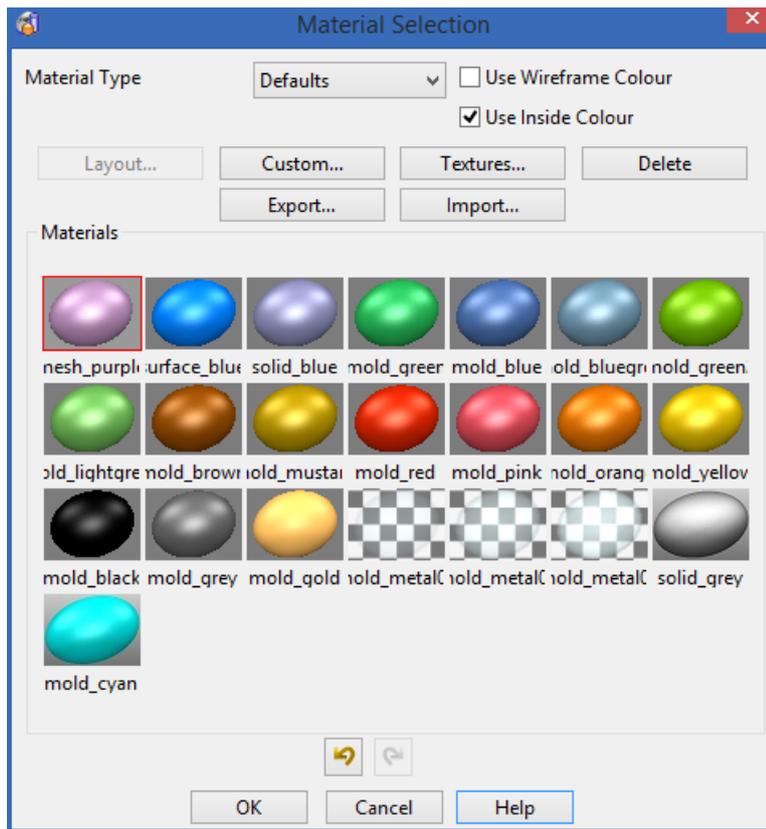


12. Change the material

- 1 Click the solid.
- 2 From the main menu, click **Format**.
- 3 Click **Materials...**



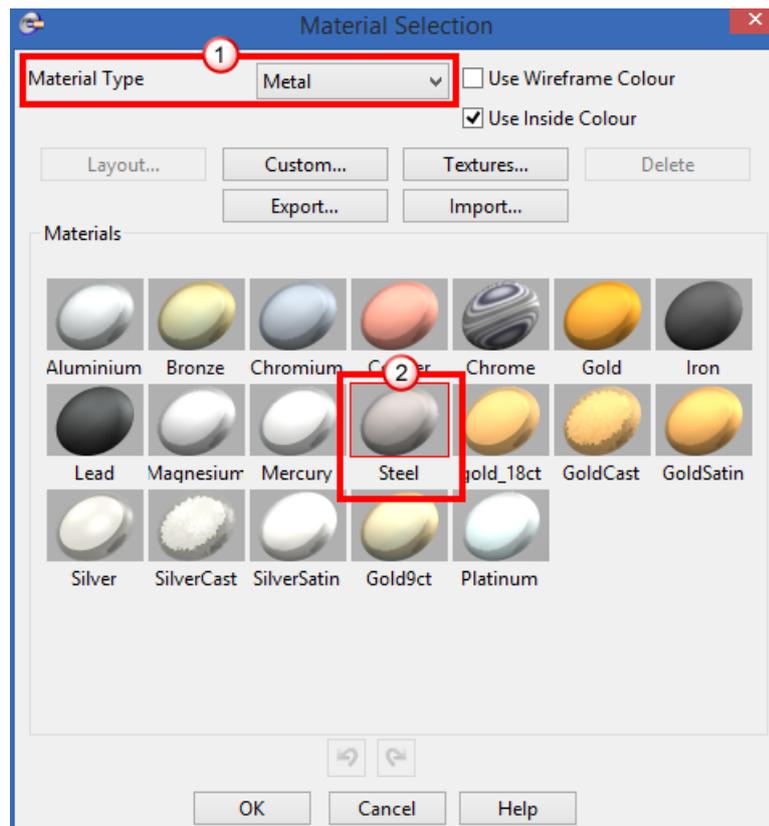
The **Material Selection** dialog is displayed.



4 Do the following:

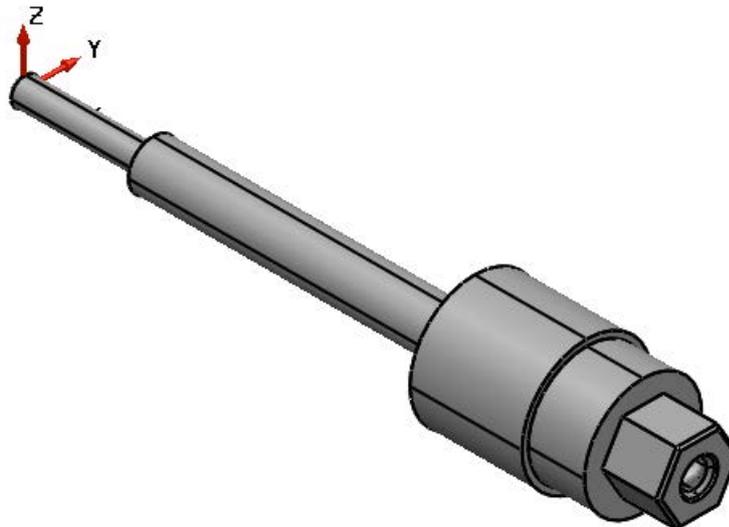
① From the **Material type** drop-down list, click **Metal**.

2 Click **Steel**.



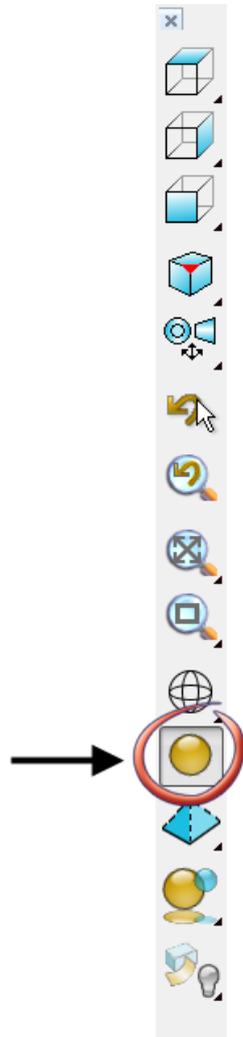
5 Click **OK**.

6 Click anywhere in the graphics window to deselect the model.



13. Change the shading option

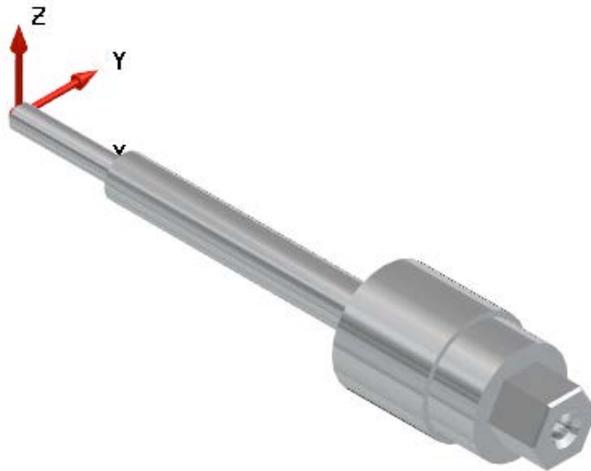
- 1 Click **Shaded View** , using the right mouse button.



- 2 From the flyout, click **Enhanced Shading** 



3 Your model should look like the one shown below:



Basic modelling concepts

Use the following sections to find information on basic modelling concepts:

PartMaker Modeling (see page 103)

User interface (see page 106)

Intelligent Cursor (see page 147)

Models (see page 161)

The coordinate system (see page 200)

Calculator and measuring

Objects

Parameters

PartMaker Modeling

PartMaker Modeling enables you to design complex shapes using the powerful 3D modelling tools in PowerSHAPE. It is ideally suited to preparing models for manufacture; either from scratch or by editing a part-complete model, imported from another CAD system.

PartMaker Modeling supports standard Microsoft Windows™ functionality such as cut, copy and paste, drag-and-drop, and rubber-band editing of objects. You can also edit any modelling object (such as line, arc, curve) by right-clicking and using the context menus. The menus contain all the main operations available for the selected object.

Ease-of-use

Ease-of-use is central to PartMaker Modeling:

- You can create most objects with one or two mouse clicks.
- You can dynamically drag dimensions using an object's editing handles to create the desired size and shape.
- You can also enter discrete dimension values using dialogs.
- The Intelligent Cursor™ generates construction lines and highlights objects' snap points and tangencies as the cursor passes over an object.

Modelling for manufacture

PartMaker Modeling simplifies the addition of manufacturing features such as:

- chamfers.
- fillets.
- split surfaces.
- draft surfaces.

The software is designed to handle a wide variety of parts and to deal with issues of modeling for manufacture. This ensures that the part designs that you receive, are easy and productive to manufacture, irrespective of the quality and complexity of the design.

Surfacing tools

PartMaker Modeling offers sophisticated surfacing tools, including surface creation from curves and dynamic surface manipulation, allowing quick and easy creation and subsequent modification of the CAD surfaces.

Starting PartMaker Modeling

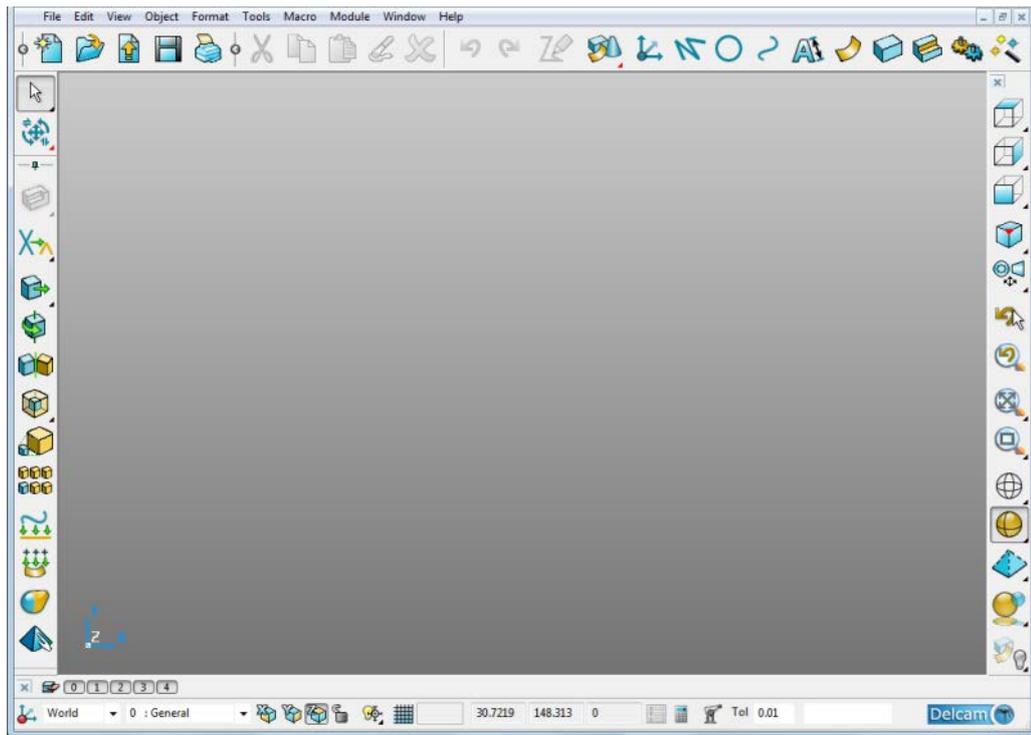
To start PartMaker Modeling:

- 1 From the **Start** menu select **All Programs** and use the sub-menus to.
 - a Select **Delcam**.
 - b Select **PowerSHAPE**.
 - c Select the **PowerSHAPE 2015**.
 - d Select **Modeling 2015**.

Alternatively, double-click the **PartMaker Modeling** icon on your desktop 

After a brief wait, the program starts up. A new large window appears on the screen.

This is **PartMaker Modeling**.



User interface

Use the following sections to find information on the user interface:

Screen layout (see page 106)

Menus overview (see page 107)

Using the menus (see page 107)

Toolbars overview (see page 110)

Using the toolbars (see page 111)

Status bar overview (see page 125)

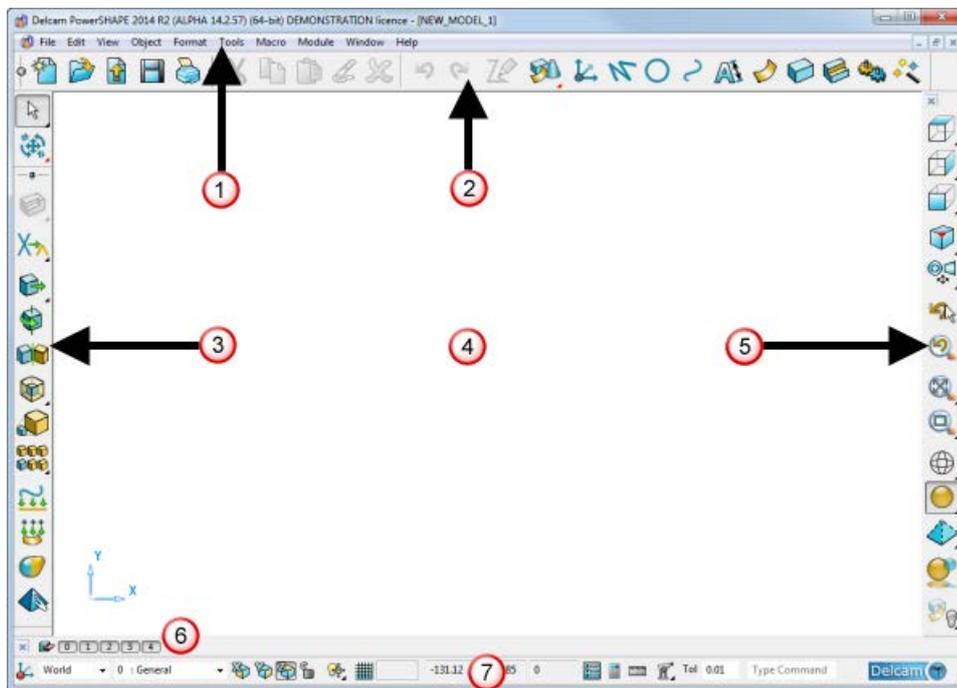
Dialog overview (see page 128)

Using the mouse (see page 131)

Using the keyboard (see page 143)

PartMaker Modeling Graphics area (see page 143)

Screen layout



PartMaker Modeling uses a Windows-style interface with menus and toolbars. The areas of the screen are identified as follows:

- ① Menu
- ② Main toolbar
- ③ Sub-toolbars — When you select a button on the Main toolbar, in most cases an associated sub-toolbar is displayed.

- ④ Graphics area
- ⑤ Views toolbar
- ⑥ Layers toolbar
- ⑦ Status bar

Using PartMaker Modeling menus

You can control most functionality in PartMaker Modeling on the menus. These expand from the menu bar that is permanently displayed across the top of the software. In most cases, functionality that is available on the menus is also available from one of the toolbars.

Clicking on a menu option opens the selected menu (see page 107). This contains commands and further sub menus. Options are dimmed when not available.

An arrow ▶ icon shows that a menu option has a sub-menu.

Using the keyboard

- Navigate around the menus with the arrow keys to pre-select items.
- Press **Enter** to select an item.
- Press **Esc** to close the menus.
- Use keyboard shortcuts to open menus. Hold down the **Alt** key and press an appropriate underlined character for the menu. For example, **Alt+E** opens the **Edit** menu. Combinations of keys are indicated using the + key. For example **Alt+F+S** is the same as selecting **File > Save**.

PartMaker Modeling menus

Use the following links for details of the tools and functions available from the menus.

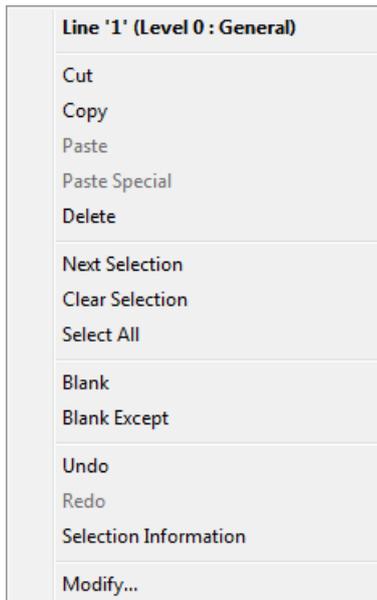
File	Controls model file related tasks such as Open , Save , Close , and Delete .
Edit	Provides the Undo/Redo commands and tools for handling the clipboard such as Cut , Copy , and Paste . It also provides general editing tools applicable to most object types, such as Mirror and Rotate .

View	Controls how the models are displayed in the graphics windows, providing a range of pre-set views. Select View Layout to display the View Edit dialog, which enables you to configure the view. You can display toolbars using View Toolbars .
Object	Provides tools for creating all the available object types, such as Workplane , Surface , and Line . The menu option for each type has a submenu showing the available creation methods for each one.
Format	Provides options to change line types, colours, and to assign display levels. Use levels to reduce screen clutter on complex models.
Tools	Provides miscellaneous tools including model fixing and model analysis. You can also set your preferences on how the program operates using Options .
Macro	Provides the controls for creating a macro of your modelling actions. The macros can be played back using Run .
Module	Use this menu to view and open the modules that you can run while using PartMaker Modeling.
Window	Lists all the open graphics windows and enables you to manage them.
Help	Enables you to access help and the Delcam websites, and check for updates. You can also view the software version and release details in the About dialog.
Popup menus (see page 108)	Right-click an object to display a menu. This menu includes the commands appropriate for editing the object. Right clicking within a graphics window, but away from any objects displays a View menu.

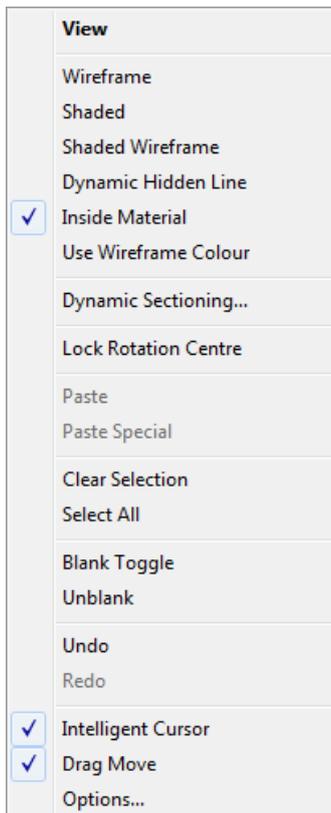
Popup menus

Display popup menus by right-clicking in the graphics window. The menu options displayed correspond with what you click on.

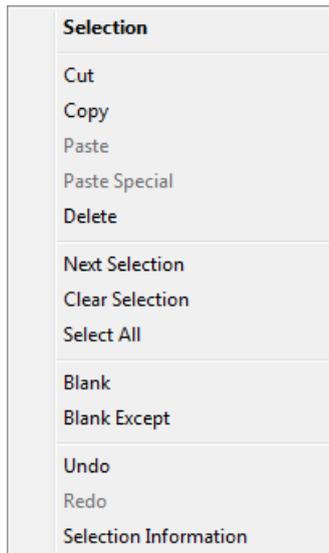
If you right-click an object, a menu displays the common edit options for that object. At the top of the menu, the type, name of the object, and the level are displayed.



Right-click in an empty area of the graphics window to display the **View** menu. This menu provides the shading options from the **View** menu and other common options.



If a mixture of objects is selected and you right-click one of these selected objects, the **Selection** menu is displayed. This menu provides some common edit commands.

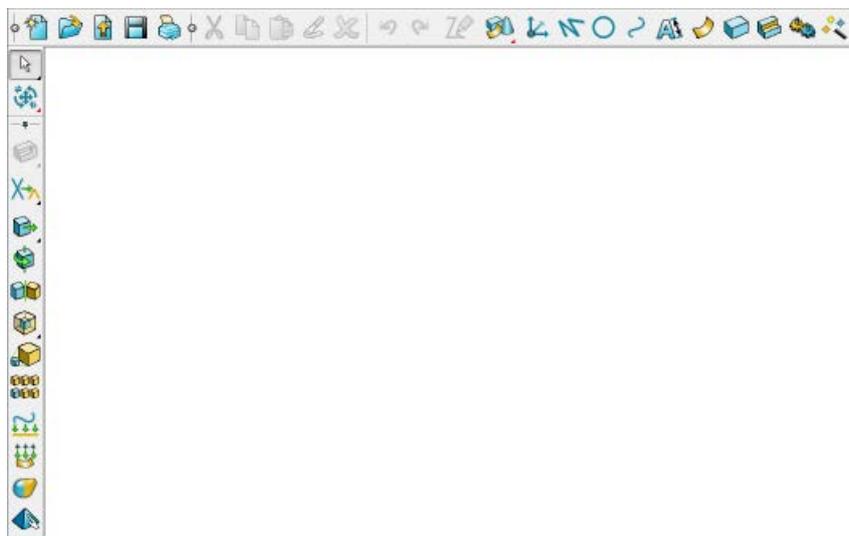


Toolbars overview

Toolbars are areas of the screen that contain buttons. Click on buttons using the mouse to activate the functionality. In most cases, toolbar buttons provide shortcuts to options on the menus. The **Main** toolbar is always visible and is displayed across the top.



The **General Edits** toolbar is displayed down the side by default.



This is also where the following toolbars are displayed when you select a corresponding object, menu item, or button.

- **General Edits**
- **Model Analysis**

- **Model Fixing**
- **Object Creation**

You can:

- Display or hide some toolbars according to the functions in use. For example the **Surface Edit** toolbar and the **Curve Edit** toolbar, shown in the image below, are displayed automatically when corresponding objects are selected.



- Expand some toolbars using the flyout (see page 124) button to display similar items or functions grouped together.
- Pin (see page 112) some toolbars anywhere on the screen, or dock them onto another toolbar using their *pins* to keep them visible when you display another toolbar.
- Create your own toolbars (see page 114) to contain your favourite buttons, menu items, or functions.
- Display a toolbar as follows:
 - 1 Right-click on an empty space on any toolbar to display a menu of available toolbars.
 - 2 Select the toolbar that you wish to display from the menu.
- Display large or small buttons on the toolbars using the **Toolbars** options page of the **Options** dialog.
- Reverse the display of the (view and editing) toolbars using the **Toolbars** options page of the **Options** dialog.
- Remove some toolbars from the screen by clicking the Close  button on the toolbar.

Using the toolbars

To use a toolbar:

- 1 Position the cursor over a button.
- 2 Left-click the button to activate the function.

The buttons on toolbars can include:

- an icon to represent the function
- a text description
- a shortcut to a Main menu item
- flyouts (see page 124) to secondary toolbars.

Some toolbars change according to the function selected, such as the **General Edit** toolbar. These have specific purposes and do not necessarily duplicate main menu options.

Repositioning a pinned toolbar

To reposition a pinned toolbar in the graphics area:

- 1 Click the pin  on the toolbar you wish to move, for example the **General Edits** toolbar. The whole pin is displayed only when the toolbar is in its default position.



The pin icon changes to pinned , the toolbar undocks and displays in the graphics area ready to be repositioned.

- 2 Click on the title bar of the toolbar and drag it to the required position in the graphics area.
- 3 Click the pin again to return the toolbar automatically to its default location. The pin changes back to unpinned .

The next time you click the pin on the toolbar, the pinned toolbar is displayed automatically in the last pinned position.

Docking a pinned toolbar

To dock a pinned toolbar with another toolbar:

- 1 Click the pin  button on the toolbar you wish to move, for example the **General Edits** toolbar. The whole pin is displayed only when the toolbar is in its default position.



The pin icon changes to pinned , the toolbar undocks and displays in the graphics area ready to be repositioned.

- 2 Click on the title bar of the toolbar and drag it over the edge of another toolbar. It snaps automatically into the toolbar area. You can also dock a toolbar at the edge of the screen.
- 3 Click the pin again to return the toolbar automatically to its default location. The pin changes back to the unpinned  icon.

The next time you click the pin on the toolbar, the pinned toolbar is displayed automatically in the last pinned position.

User-defined toolbars

A user-defined toolbar is a toolbar that you create to contain your favourite buttons, menu items, or functions.

- Your customised toolbar can contain both menu and toolbar items as well as command lines.
- You can send your user-defined toolbar to other users.

- You can group or separate options on your toolbar using the **Customise** dialog (see page 116).
- You can add your own icons and you can customise buttons to be added to your toolbar using the Customise Picture dialog (see page 121).

Using your personal toolbars

Use the following sections to create and use your own toolbars:

Create a user-defined toolbar (see page 115)

Edit a user-defined toolbar (see page 118)

Display an existing user-defined toolbar (see page 118)

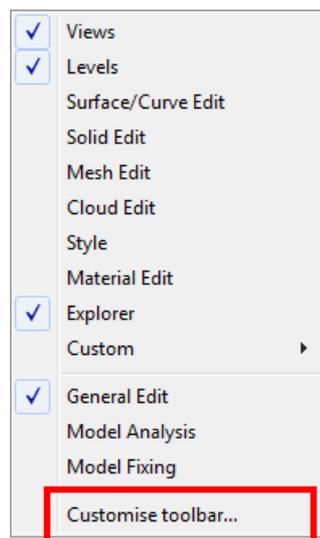
Customise buttons for user-defined toolbars (see page 118)

Sharing a user-defined toolbar and custom buttons (see page 123)

Creating a toolbar

To create a toolbar:

- 1 Right-click on an empty section of any toolbar and select **Customise toolbar** from the menu.

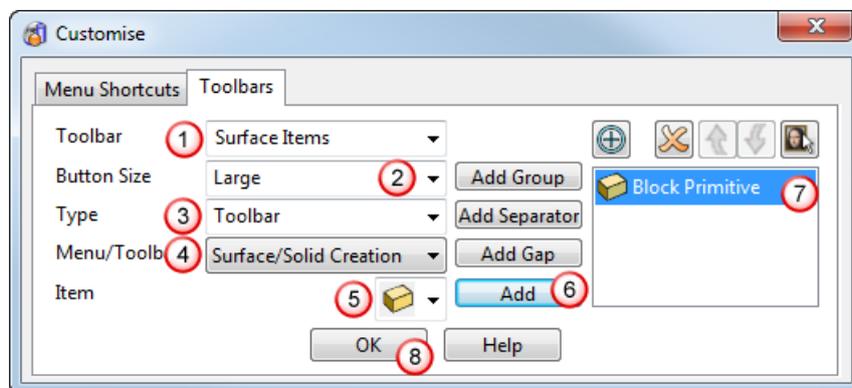


The **Customise** dialog (see page 116) is displayed.

- 2 On the **Toolbars** tab define your toolbar and select the items you want to add.
- 3 Click **Add** to add each new item to the toolbar.
- 4 Use the buttons on the dialog to order and organise the toolbar items.

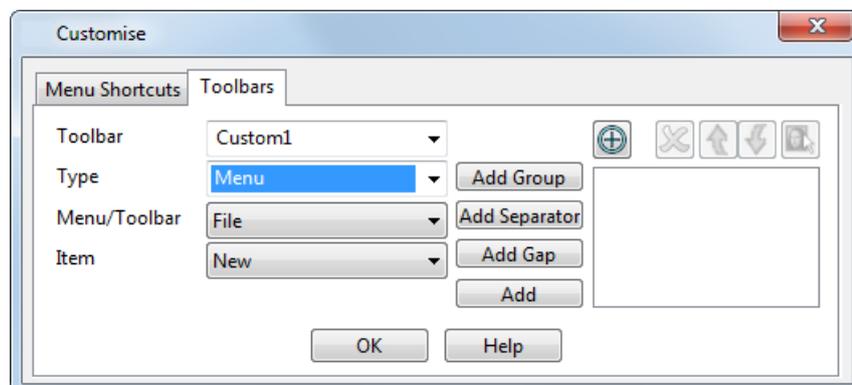
The example below describes adding a block primitive from the surface creation toolbar to a new custom toolbar.

- ① Select a toolbar and enter a new name, such as **Surface items**.
- ② Select the **Button Size** size from:
 - **Large**
 - **Small**
 - **As Options Page**
- ③ For the **Type**, select **Toolbar**.
- ④ Select the **Menu/Toolbar** that contains the item you want to add to your toolbar, such as **Surface/Solid Creation**.
- ⑤ Select the **Item** to add, such as the **Block Primitive**  button.
- ⑥ Click **Add** to add the block to your toolbar.
- ⑦ Added items, for example the **Block Primitive** button, are displayed in the added items list.
- ⑧ Click **OK**.



Customise dialog - Toolbars

Use this dialog to define up to four customised toolbars.



Toolbar — Enter a new name for a new toolbar or select an existing user-defined toolbar from the list. You can define up to four toolbars.

Type — Select from the following:

- **Menu** — Select this option to add a menu item to your toolbar and select the menu from the **Menu/Toolbar** list.
- **Toolbar** — Select this option to add a toolbar item to your toolbar and select the toolbar from the **Menu/Toolbar** list.
- **Command** — Select this option to enter your own **Command** and **Description** to be added to your toolbar.

Menu/Toolbar — Select the menu or toolbar that contains the option you want to add to your new toolbar.

Item — Select the menu option or toolbar button that you want to add to your new toolbar.

Add Group — Click this button to add a group marker. The group marker  icon is displayed in the list of added items at the right of the dialog. A group marker shows the beginning of a group of buttons, menu items, or commands. On the toolbar, the group marker is a toggle button:



Hide Group — Click this button to contract the items in the group.



Show Group — Click this button to expand the items in the group.

Add Separator — Click this button to add a separator marker to divide up your toolbar items. The separator marker  icon is displayed in the list of added items at the right of the dialog.

Add Gap — Click this button to add a small space between items on the toolbar. A gap is displayed in the list of added items at the right of the dialog.

Add — Click this button to add the selected **Item** to the toolbar. The item is displayed in the list of added items at the right of the dialog.



Drag Crosshairs — Click and hold the mouse button and drag the crosshairs over a toolbar button. The button is added automatically to custom toolbar and is displayed on the dialog.



Remove — Click this button to remove the item selected in the added items list from your toolbar.



Move the item up the menu — Click this button to move the selected item up the order of items on the toolbar.



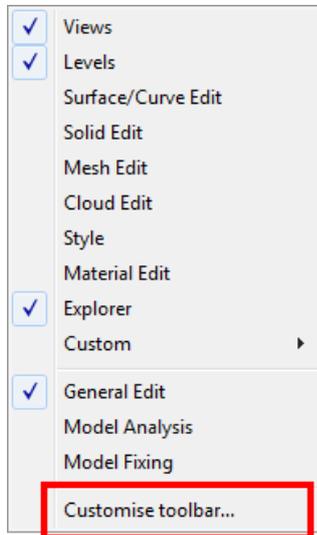
Move the item down the menu — Click this option to move it down the order of items on the toolbar.



Change button icon — Click this button to change the icon for the button selected in the added items list. The **Customise Button** dialog (see page 120) is displayed.

Edit a user-defined toolbar

- 1 Select **Customise toolbar** from the popup menu, displayed when you right-click on an empty section of any toolbar.



- 2 Select the **Toolbars** tab on the Customise dialog (see page 116).
- 3 Select the toolbar you want to edit from the **Toolbar** drop-down list.
- 4 Make your changes.
- 5 Click **OK**.

Displaying an existing user-defined toolbar

- 1 Select **View > Toolbars > Custom**.
- 2 Select the toolbar you want to display.

Customising buttons for user-defined toolbars

When you are creating or editing a user-defined toolbar, you can edit and customise buttons. You can:

- customise an existing button on a user-defined toolbar (see page 119).
- create a new button to be added to the toolbar (see page 119).
- select a new button on the **Customise Button** dialog (see page 120).
- use the bitmap editor on the **Customise Picture** dialog (see page 121).

Customising a button on a user-defined toolbar

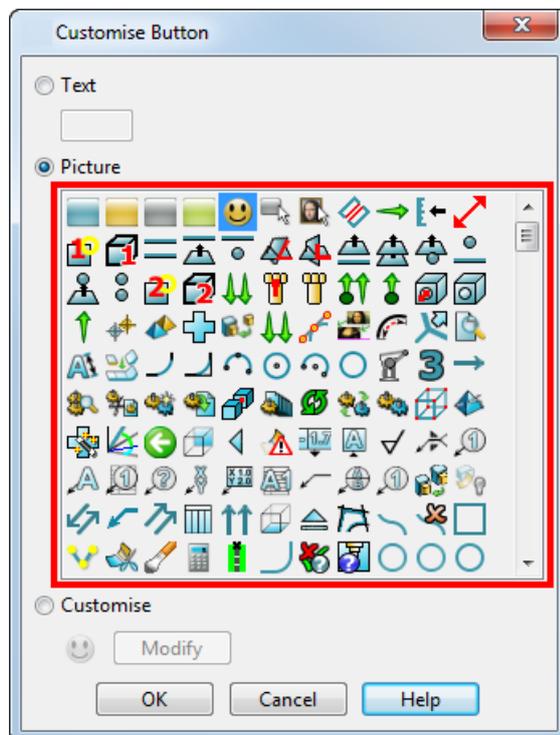
To customise a button on an existing user-defined toolbar:

- 1 On the **Customise** dialog (see page 116), select the button you want to customise in the *added* list.
- 2 Click the **Change Button**  button to display the **Customise Button** dialog (see page 120).
- 3 Select **Customise**. The **Modify** option becomes available and a small thumbnail of the selected button is displayed.
- 4 Click **Modify**.
- 5 Use the bitmap editing functions on the **Customise Picture** dialog (see page 121) to edit the bitmap picture.

Creating a new button for a user-defined toolbar

To create a new button to be added to a user-defined toolbar:

- 1 Display the **Customise Button** dialog (see page 120).
- 2 Select the **Picture** option.
- 3 Select a picture or blank background.



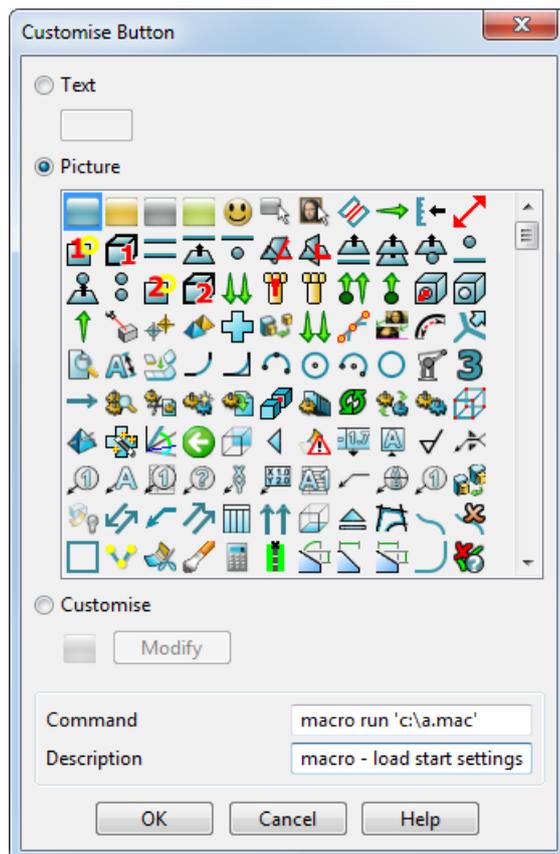
- 4 Select the **Customise** option. The **Modify** option activates and a small thumbnail of the selected picture is displayed.
- 5 Click **Modify**.
The **Customise Picture** dialog (see page 121) is displayed.

- 6 Use the bitmap editing functions to edit or customise your selected button.

Customise Button dialog

Use this dialog to:

- enter text to display as the item's button.
- select a picture to be displayed as the item's button.
- select a picture to edit or customise using the bitmap editing functionality on the **Customise Picture** dialog (see page 121)
- change the description of a command. This option is displayed only if you have selected **Command** from the **Type** list.



Text — Select this option and enter text to be displayed as a button.

Picture — Select this option and select an icon to be displayed as a button.

Customise — If selected, the **Modify** button is displayed.

Modify — Click this button to display the selected picture in the bitmap editor on the **Customise Picture** dialog (see page 121) to customise the picture before adding it to a user-defined toolbar.

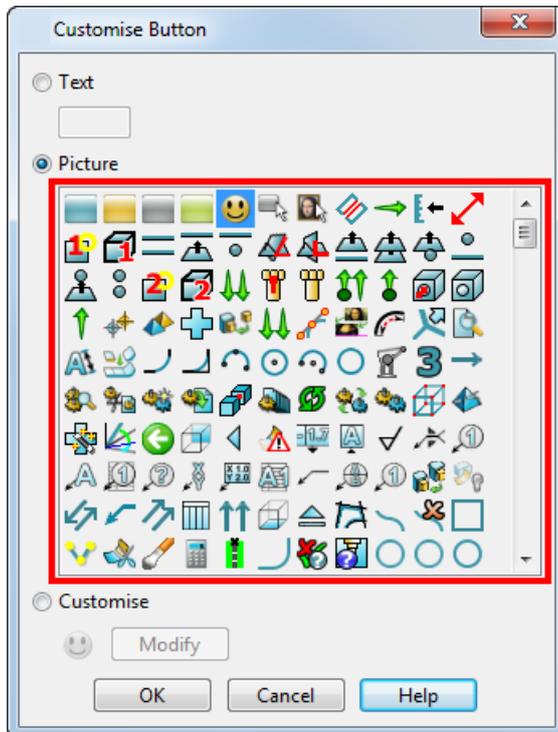
Command — The command is displayed and you can edit it.

Description — The description for the command is displayed and you can edit it.

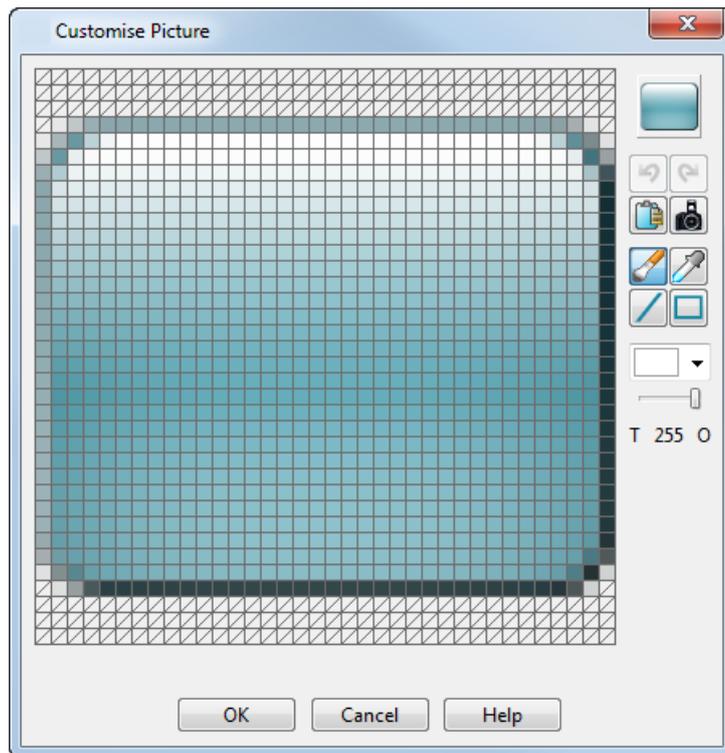
Customise Picture dialog

Use the bitmap editing functions on this dialog to customise the button selected on the **Customise Button** dialog (see page 120). You can:

- customise a button on an existing user-defined toolbar (see page 119).
- create a new button (see page 119) by selecting a picture or background button from those provided.



The selected button is displayed as a bitmap on a grid showing each pixel, ready for editing.



At the top right of the dialog there is a preview of the button, which is updated as you make changes.

 **Undo** — Undo the last command.

 **Redo** — Reverse the last **Undo** command.

 **Paste** — Click this button to paste the clipboard contents. You can copy an image from an image editor (for example MS-Paint), copy the image to the clipboard and then paste it this button.

 **Thumbnail** — Click this button to use a thumbnail of the current model.

 **Brush** — Select this option to change the colour of the pixels that you drag your mouse over to the selected **Colour** and **Transparency**.

 **Dropper** — Select this option and click on the bitmap image to match the **Colour** setting to an existing colour.

 **Line** — Select this tool to draw a straight line with your cursor. Set the **Colour** and **Transparency** of the line.

 **Box** — Use this tool to draw a box. Set the **Colour** and **Transparency** of the box.

 **Colour** — Use the colour chart to select a colour.



T 255 O Transparency — Drag the slider between **T** (transparent) and **O** (opaque).

OK — Click this button to save the edited image.

Cancel — Click this button to close the dialog without saving your changes.

Sharing a user-defined toolbar and custom buttons

User-defined toolbars have a name in the format `CustomToolbarX`. Custom buttons have a name in the format `ToolbarX_PictureY.bmp`.

To send a user-defined toolbar and its buttons to another user:

- 1 Find the user-specific files. They are stored in `xxxx\PowerSHAPE` where `xxxx` is one of the following:
 - `C:\Documents and Settings\[username]\Application Data` (XP installation)
 - `C:\Users\[username]\AppData\Roaming\` (Windows 7 and Windows 8 installation)
 - the location that is indicated by your **HOME** variable.

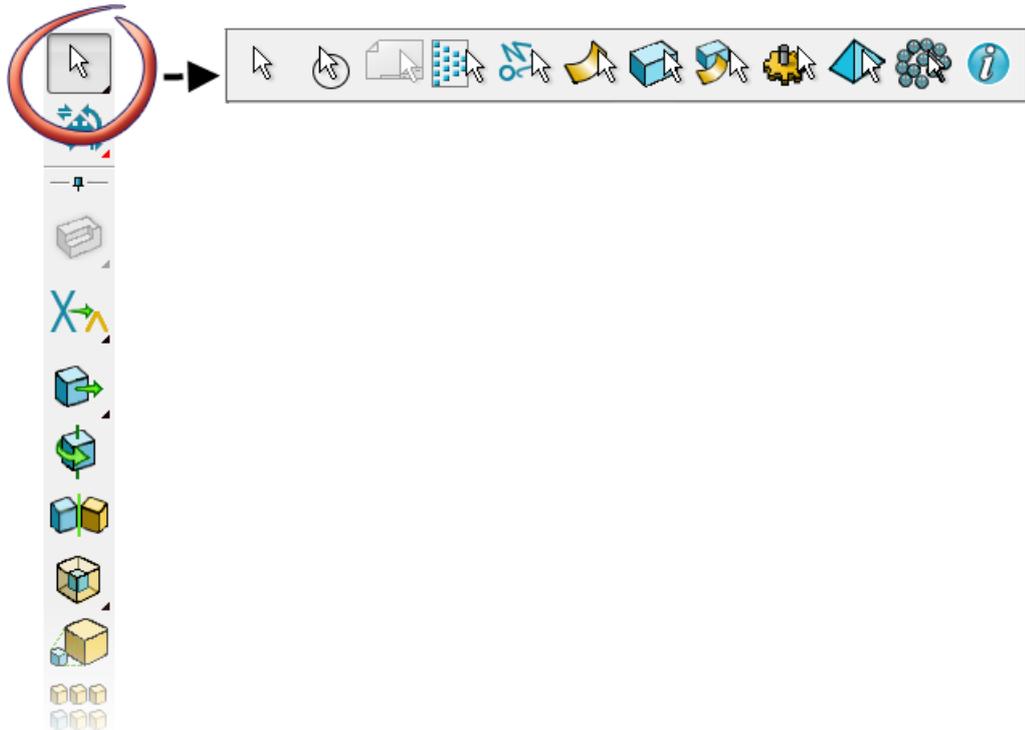


*To find the **Local config** folder (PowerSHAPE), enter **print app.paths** in the command window.*

- 2 Select the required files. They include:
 - a file that contains the toolbar, for example `CustomToolbar1`.
 - one or more button files from the toolbar, for example `Toolbar1_Picture1.bmp`.
- 3 Email the files to another user, with instructions to copy the files into the folder containing their user-specific files.

Flyouts overview

A flyout toolbar is an extended part of a toolbar (see page 110).



Access a flyout toolbar using a flyout button. Flyout buttons have a small triangle icon in the bottom right corner. There are two types of flyout buttons:

- Red ▲ icon — Flyouts with a red icon contain mode buttons. Right-click on the flyout button to display the flyout toolbar. Select a mode button to display the buttons for the mode on the toolbar.

For example, if you select the **Model Fixing**  mode button from the **General Edits**  flyout, the toolbar displays the model-fixing buttons.

 *You can cycle through the available modes by repeatedly clicking the flyout button.*

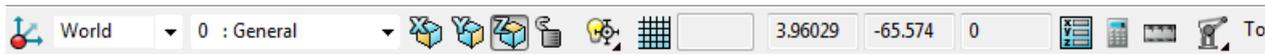
- Black ▲ icon — Flyouts with a black icon group similar buttons, for example the **Offset**  flyout. Right-click on the flyout button to show the flyout toolbar and click the button you want. The most recently used button from the flyout toolbar is displayed as the flyout button.

You can display a flyout toolbar in one of these ways:

- Hover your cursor over the flyout button. The flyout toolbar is hidden when you move your cursor away.
- Right-click on the flyout button. The flyout toolbar is displayed until you click.

Status bar overview

The Status bar is displayed at the bottom of the screen.



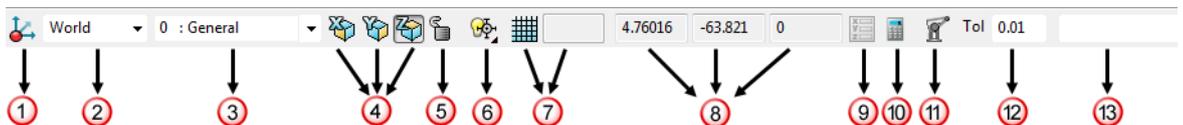
Use the Status bar to:

- Set the default drawing plane (see page 200).
- Set point-locking (see page 225).
- Display the Intelligent Cursor flyout (see page 151).
- Display the Grid (see page 224).
- Enter point co-ordinates (see page 205).
- Display the Position dialog (see page 208).
- Display the Calculator.
- Display the Measure dialog.
- Connect/Disconnect the Arm.

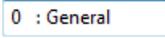
For further details, see Using the status bar (see page 125)

Using the status bar

The **Status bar** is fixed at the bottom of the screen.



Key	Object	Name	Description
①		Temporary workplane	Click this button to create or remove a temporary workplane. For further details see, Creating a temporary workplane in the Wireframe modelling manual.

- 2  **Workplane** Use this list to activate and de-activate workplanes. You can also use it to change the name of the selected workplane. For further details, see Workplane list.
- 3  **Current Level** This displays the level of the selected item or the level you are working on. Use this to change the level of the selected items. Only named levels that appear in the **Level** dialog are displayed in this list. For further details, see **View > Toolbars > Levels**
- 4  **Principal plane** Select a button to set the principal plane of the current workspace. Each button displays the label of the axis that is normal to the principal plane. For example, the button with label **Z** represents the **XY** principal plane.
- 5  **Point lock** Toggle this button to enable or disable point-locking when entering positions using the mouse. For further details, see Using point-locking (see page 225).
- 6  **Intelligent Cursor** Right-click this flyout button to display the Intelligent Cursor flyout toolbar (see page 151).
- 7  **Grid on/off** Toggle this button to enable or disable the grid. The cursor snaps to the grid lines Grid when positioning objects.

7		Grid scale	View and edit the scale of the grid (see page 224) when it is enabled.
8		Coordinates	The X, Y, and Z coordinates of the cursor are displayed respectively.
9		Position	When you are creating objects, click this button to display the Position dialog (see page 208), where there are various tools for entering positions.
10		Calculator	Click this button to open the Calculator .
11		Connect/Disconnect the Arm	Use this flyout to connect or disconnect an arm. To connect an arm you need an appropriate licence file and drivers installed on your computer.
12		Tol	The general tolerance variable is displayed. You can edit the value.
13		Command input	Use this to: <ul style="list-style-type: none"> ▪ enter data (see page 127). ▪ view previously entered data (see page 128). ▪ display the Command window (see page 128).

Using the Status bar to enter data

There are two ways to enter data.

- Click in the **Command input** box. Enter the data and press **Enter**.
- Move the cursor into the graphics window and enter the data. The data displays in the **Command input** box. Press **Enter** to enter the data.

Using the Status bar to view previously entered data

Click in the **Data entry box**.

- 1 Use the *up* and *down* arrow keys on the keyboard to display the previously entered data.
 - The *up* arrow key displays previously entered data upwards from the last entry.
 - The *down* arrow key displays previously entered data downwards from the first entry.
- 2 If you want to enter any of the data again, press **Enter** when it is displayed in the **Data entry box**.

Using the Status bar to display the command window

Move the cursor into the **Data entry box** and double-click in the box. The **Command** window is displayed.

Dialog overview

- A dialog is a movable box that is displayed when you need to provide information to the software. Dialogs may contain:
 - data entry boxes
 - expandable lists
 - buttons
- Most operations in a dialog are controlled by using the mouse or by using the menus. Some options and functions display other dialogs. For example, when you open a model a dialog is displayed for you to select a model from a list.
- Options that are followed by , display dialogs for you to make choices and enter details.
- Click **OK** to use the new information and close the box.

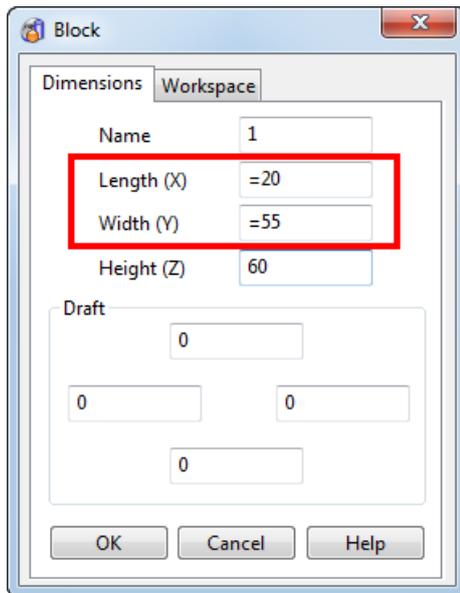
Text boxes and data boxes



Use these boxes to enter text or numbers. Click in the box, keeping the mouse cursor inside the box, then type the characters. The data is entered when you press *Enter*.

A value that is defined with a parameter or expression is indicated by the use of = in addition to the value.

For example, if you enter **Length** as **a=20** and **Width** values as **a+35**, the length and width values will be displayed in the dialog as **=20** and **=55** respectively.



Drop-down lists



Enter your data in one of the following ways:

- Click in the box and type in your data.
- Click the arrow  to expand the box showing any previous entries. You can then click on the one you want.

Option menus

These appear as drop-down lists that expand to show a menu list.



Click the arrow  to expand the list. Click on an option in the list to select it. The menu collapses to display the selected option.

Value box

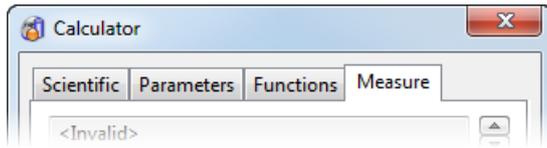


Increase or decrease the value by clicking the up or down arrow. You can also type a new value into the box.

Tabs

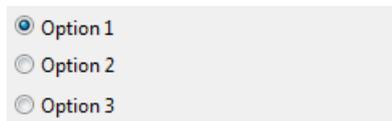
Tabs are used in dialogs to show a different work page for each tab.

The example below shows the tabs on the **Calculator** dialog.

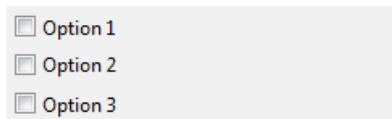


Option selection buttons

There are two types of option selection buttons.



These buttons are small and placed next to their options. When one button is clicked, a dot appears in the button and the other previously clicked button pops out. Only one option button can be *ON* at any time and one option button must always be selected. It is not possible to have them all *OFF*.



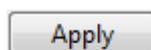
These options buttons are fully independent. A group of these can be any combination of *ON* and *OFF*. Click a button to set *ON* and click again to set *OFF*.

Buttons



Each button displays a unique image or text to represent a command. When clicked, the button appears to press-in and the associated command is activated. When the command is not available, the icon image or text appears grey and the button will not press-in when clicked.

Apply button



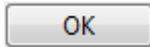
Click to make the changes required, but to leave the dialog displayed.

When a dialog is displayed, you can click the right-mouse-button in the graphics area as a shortcut for **Apply** or **Accept**.



Apply has precedence over **Accept**, so when a dialog has both **Apply** and **Accept** buttons available, clicking the right mouse button is the same as clicking **Apply**.

OK button

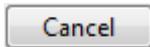


Click to make the changes required and close the dialog.



Pressing **Enter** is the same as clicking **OK** on a dialog.

Cancel button



This button closes the dialog, discarding any edits carried out on the model while it was displayed.

Using the mouse

The mouse is the main device for controlling the software. You can click buttons or objects with or without keyboard “modifier” keys (*Alt*, *Shift*, *Ctrl*) to alter the way you can create, select, move, edit or view models on the screen. The following terms are used in the documentation:

- **Click** — Press and release the left mouse button (see page 134).
- **Double-click** — Rapidly click the left mouse button twice.
- **Drag** — Hold down the left mouse button and move the mouse while holding the button down.
- **Move** — Move the mouse without selecting any of the mouse buttons.

When other mouse buttons are to be used the required button is specified; for example

- **Middle-click**
- **Middle-drag**
- **Right-click**

The mouse controls such as *double-click speed* and *hit radius* are set to default values, but can be specified on the **Mouse** page of the **Options** dialog.

The cursor that is displayed when you use the mouse indicates the operation that you are doing.

Context-sensitive cursors

The cursor that is displayed when you use the mouse reflects the operation that is being carried out. Context-sensitive cursors are available in the following areas:

- Annotation (see page 132)
- Curve toolbar (see page 132)
- Drafting (see page 132)
- General edits toolbar (see page 133)
- View toolbar (see page 133)
- Other (see page 133)

Context sensitive cursors — Annotation toolbar

-  — Add text along a curve.
-  — Add text (horizontally or vertically).

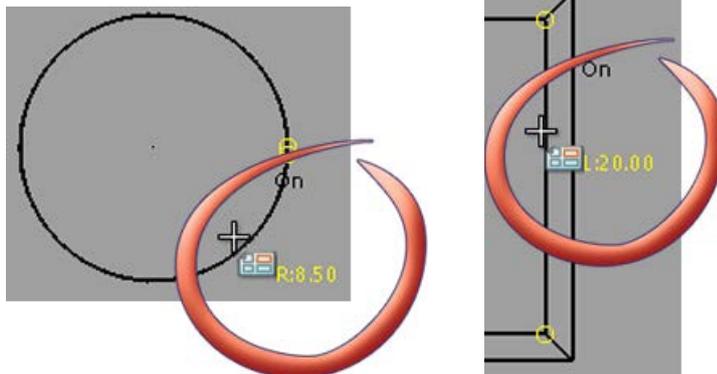
Context sensitive cursors — Curve toolbar

-  — Stop curve.
-  — Close curve.
-  — Sketch curve.
-  — Select start and end point (composite curve).

Context sensitive cursors — Drafting

-  — Activate view.
-  — View selection.
-  — Drawing selection.

Arc and line lengths are displayed on the cursor when creating dimensions.



Context sensitive cursors — General edits

-  — Limit cut.
-  — Limit point.
-  — Move/copy/stretch (also used for drag-move).
-  — Rotate items.
-  — Mirror items.
-  — Mirror items across a user defined plane.
-  — Mirror items in a wireframe line.
-  — Offset items.
-  — Scale items.

Context sensitive cursors — View toolbar

-  — Zoom to box.
-  — Zoom.
-  — Rotate.
-  — Pan.

Context sensitive cursors — Other

-  — Warning/error.
-  — Cursor selection/edge-pick hand.
-  — Add component.
-  — Principal plane-lock toggled.
-  — Action not allowed.
-  — Action not allowed, plane.
-  — Action not allowed, arc.
-  — Paste attributes.
-  — Selection filter.

Using the left mouse button



This is the main mouse button for selecting menu options, clicking buttons on toolbars and dialogs and for editing model objects. It has different effects according to where the cursor is positioned.

You can use the left mouse button:

- over a menu (see page 134)
- over a non-active graphics window (see page 135)
- over an active graphics window (see page 135)
- on an object (see page 138)
- in a dialog (see page 136)
- after a Cut or Copy (see page 142).

Over a menu

	Single Click highlights and selects menu options.
	Double Click has no particular effect over menus. PartMaker Modeling assumes you intended to single click.
	Drag by holding down the left mouse button and moving the mouse to move the cursor. You can: <ul style="list-style-type: none">▪ Drag the cursor over objects in the graphics window to highlight them for selection.▪ Edit and create dimensions by dragging them to the required position.▪ Edit objects by dragging their handles.
	Move the cursor over the different options, highlighting the ones that are available. You can: <ul style="list-style-type: none">▪ Move the cursor over menus to display submenus where they are available.▪ Move the cursor over toolbars to display flyouts where they are available.▪ Activate construction lines when the intelligent cursor is active.▪ Move the cursor along construction lines.

Over a non-active graphics window



Single-click in a non-active graphics window activates the window. The title of the active window is highlighted.

Over an active graphics window



Single Click in an active graphics window usually signifies an attempt to select an object or to enter a position. Click away from any object to clear the current selection.

When entering positions, click away from any object to input a position on the principal plane of the active workplane.



Double-click has no effect unless you are pointing at an object.



Drag (hold down the **Left** mouse button while dragging) to draw a rubber band box. Objects within the box are selected, indicated by their yellow highlight colour and handles.

Note:

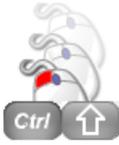
- Any other objects already selected and not within the box become deselected.
- If the box contains no objects, the current selection is cleared.
- When more than one object is selected, the graphical handles are not displayed.



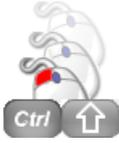
Shift-Drag (hold down the **Shift** key while dragging) draws a rubber band box. Objects within the box are added to the current selection.



Ctrl-Drag (hold down the **Ctrl** key while dragging) draws a rubber band box. Objects within the box are added to the current selection or deselected if they are already selected.



Ctrl-Shift-Drag (hold down both the **Ctrl** and **Shift** keys while dragging) draws a rubber band box. Objects within the box are deselected from the current selection.



Shift-Alt-Drag (hold down the **Shift** and **Alt** keys while dragging) rotates the view around the X axis.

In a dialog



Single click to select options and press buttons. On slider controls, clicks on the track cause the slider to step towards the cursor.



Double-click is not effective on all dialogs. Double click in a text box to select all the text. If you then type new text, the selected text is replaced by the new text.

Double click on an item in a list to select the item and accept the dialog. For example, double clicking a file name in the **Open Model** dialog, accepts the dialog and opens the model.



Shift-click (hold down the **Shift** key while clicking) to select multiple items in a list. All items between the last item selected and the current item clicked become selected.



Drag slider controls to move the slider button.

Using the middle mouse button

Use this mouse button with keyboard “modifier” keys to control the view of your model. It is not used within the dialog and does not alter a model’s design.



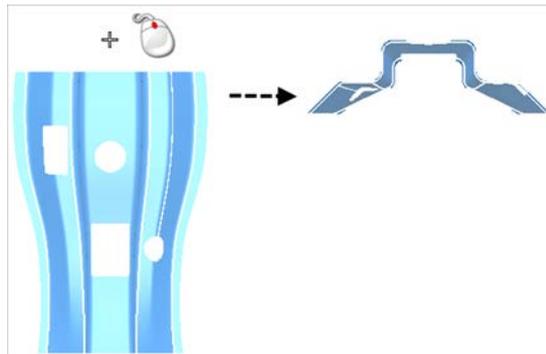
If you have a two button mouse, operations using the middle mouse button are available using the other two buttons. For further details, see [Right-hand mouse button](#) (see page 140) and [Left and right mouse buttons](#) (see page 142).



Middle-click to **Rotate** the view, hold down the middle button and move the mouse. The model turns as if the cursor is pushing and pulling on the outer surface of a ball and a ball graphic appears over the centre of rotation to show what is happening.



Middle-click to change **Orientation**. Position the cursor to the side of the model and click the middle mouse button. The model will be rotated so that you are viewing the model from the position of the cursor.



Shift+Middle-click to **Pan** a particular point to the centre of the view, hold down the **Shift** key and click the point in the model you want to appear in the middle of the view.



Ctrl+Middle-click to **Pan** a particular point to the centre of the view, hold down the **Control** key and click the point in the model you want to appear in the middle of the view.



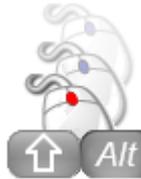
Shift+Middle-click+drag to **Pan** the view, hold down the **Shift** key and drag the middle button. The view follows the mouse movements.



Ctrl+Middle-click+drag to **Zoom** the view, hold down the **Control** key and drag the middle button. Moving up the screen magnifies the view. Moving down the screen reduces the view.



Ctrl+Shift+Middle-click+Drag to **Zoom to a box**, hold down the **Shift** and **Ctrl** keys and drag the middle button. A rubber-band box appears. When you release the mouse button, the view zooms to the box size.



Shift+Alt+Middle-click+drag (holding down both the **Shift** and **Alt** keys while dragging) to rotate the view around the Y axis.

On an object

When selected, objects are displayed with additional edit handles. For example:

- Lines have key points at each end.
- Curves have key points on arms extending from the selected control point on the curve.
- Arcs have key points at the centre and at the start and end of the arc sector.
- Workplanes have a “control rectangle” and origin.

If you drag a key point, only the aspect of the object controlled by the point will move.

To move the whole object, be careful to click on the object itself, but away from any of its *key points*.



Single-click over an unselected object to select it. The object turns yellow with its key points shown as graphical “handles”. The selected object’s key point closest to the cursor when selected becomes its anchor point.

Any other objects already selected become deselected.

If the Intelligent Cursor is *ON*, the cursor changes as you move over objects and current selections.



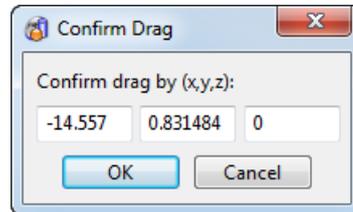
Double-click to select the object, but also to send the command to display a dialog or toolbar allowing you to edit the object’s properties. For further details, see *Modifying an object in the Menus and Toolbars manual*.

When entering a series of points (for example, for a series of lines or a curve), double click to finish the series. A subsequent single click starts a new series.



Drag a selected object to move it. While the button is pressed, the object follows the mouse. When you release the button, the **Confirm Drag** dialog is displayed for you to confirm the **x, y, z** coordinates for the move. Although the object appears to continue to move with the cursor, the coordinates are fixed where you released the mouse button.

Click **OK** to confirm the coordinate



The move is relative to the active workplane.



Shift+click (hold down the **Shift** key while clicking) to add objects to the current selection.



Ctrl+click (hold down the **Ctrl** key while clicking) to add objects to the current selection or deselect those already selected.



Alt+click (hold down the **Alt** key while clicking) to create composite curves. For further details, see *Creating a composite curve by tracing in the Wireframe modelling manual*.



Ctrl+Shift+click (hold down both the **Ctrl** and **Shift** keys while clicking) to deselect objects from the current selection.

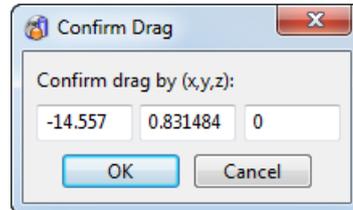


Ctrl+Drag (hold down the **Ctrl** key while dragging) to copy the selected objects. The copy is dropped when you release the mouse button.

This copy does not overwrite data already on the Cut/Copy clipboard. The **Confirm Drag** dialog is displayed where you confirm the **x, y, z** coordinates for the move.



The move is relative to the active workplane.



Using the right mouse button

You can use the right mouse button:

- Over an object (see page 141)
- In a text box (see page 141)
- Over an active graphics window (see page 141)
- When a dialog is displayed, you can click the right-mouse button in the graphics area as a shortcut for **Apply** or **Accept**. If **Apply** and **Accept** are unavailable, clicking the right mouse button will close the dialog without making any changes.



Apply has precedence over **Accept**, so when a dialog has both **Apply** and **Accept** buttons available, clicking the right mouse button is the same as clicking **Apply**.



Click with this mouse button to display popup menus and the calculator. You can also use this button to pan, zoom and drag the view.

Over an object



Single right click in the active model window to display popup menus. These contain various options including **Cut, Copy, Paste** and **Delete**. Click over different object types or a blank space to display options on the popup menu that are relevant to the object type. For further details, see popup menus (see page 108).

In a text box



Single right-click in a numeric text box to display the **calculator**. The calculation results are displayed automatically in the text box when you press the *Enter* key or move the mouse outside the numeric text box.

In a text box, this displays a popup menu with options **Undo, Cut, Copy, Paste, Delete** and **Select All**. You can use these options on the data in the box.



Shift+right-click in a numeric text box to display a popup menu with options **Undo, Cut, Copy, Paste, Delete**, and **Select All**. You can use these options on the data in the box.

Over an active graphics window

	Shift+right-click to Pan a particular point to the centre of the view, hold down the Shift key and click the point in the model you want to appear in the middle of the view.
	Ctrl+right-click to Pan a particular point to the centre of the view, hold down the Control key and click the point in the model you want to appear in the middle of the view.
	Shift+right-click+Drag to Pan the view, hold down the Shift key and drag the right button. The view follows the mouse movements.

	<p>Ctrl+right-click+Drag to Zoom the view, hold down the Ctrl key and drag the right button. Moving up the screen magnifies the view. Moving down the screen reduces the view</p>
	<p>Alt+right-click-Drag to Zoom to a box, hold down the Alt key and drag the right button. A rubber-band box appears. When you release the mouse button, the view zooms to the box size.</p>
	<p>Ctrl+Shift+right-click+Drag (hold down both the Ctrl and Shift keys while dragging the right mouse) to rotate the view. The model turns as if the cursor is pushing and pulling on the outer surface of a ball and a ball graphic appears over the centre of rotation to show what is happening.</p>
	<p>Shift+Alt+right-click+Drag (holding down the Shift and Alt keys while dragging) to rotate the view around the Z axis.</p>

After a Cut or Copy operation



Ctrl+Alt+Click (hold down the **Ctrl** and **Alt** keys while clicking) to enter an anchor point for **Paste Special**.

Left and right mouse buttons



Shift+Alt+Drag (hold down the **Shift** and **Alt** keys while dragging) to rotate the view around the Y axis.



Left-click+right-click to rotate the view, hold down both the left and right buttons and move the mouse. The model turns as if the cursor is pushing and pulling on the outer surface of a ball and a ball graphic appears over the centre of rotation to show what is happening.

Using the keyboard

The keyboard is usually used to enter information into a dialog. This might be when you want to create or edit an object to a specific size and position.

Certain "modifier" keys are also used with the mouse to change the way the mouse behaves. For further details, see [Using the mouse](#) (see page 131).

There are also a number of keyboard shortcuts that allow you to do a variety of operations such as open and close models. For further details, see [Shortcuts to menu options](#).

PartMaker Modeling graphics area

PartMaker Modeling runs within its own window containing:

- Menus (see page 107)
- Toolbars (see page 110)
- Status bar (see page 125)
- Graphics area (see page 143)

For further details, see [Screen layout](#) (see page 106) and [The coordinate system](#) (see page 200)

The graphics area may be:

- a single modelling window that fills the graphics area.
- several modelling windows that overlay each other in the graphics area. These windows can be placed anywhere on the screen, not necessarily within the graphics area. For further details, see [Window menu overview](#).

Each modelling window displays views of a single model. You can open as many modelling windows for each model as you like, but closing a model's last window also closes the model.

Modelling windows in the graphics area

Use the Window menu to control the content displayed in the graphics area:

- Create new modelling windows.
- Swap between modelling windows.
- Make a different modelling window active. To do this click on the window or select it from the menu. Only one modelling window is active at any time and this is indicated by the window's highlighted title.
- Change the display of modelling windows.

Use the View menu to:

- Change the view shown in modelling windows.
- Increase the visible modelling area by selecting Full Screen or Full Screen Preview.

Displaying modelling windows

- Use Single window mode (see page 144) to display a single working modelling window that fills the whole area.
- Use Multi-window mode (see page 145) to display several overlapping modelling windows.
- Display several tiled model windows that are tiled vertically or tiled horizontally.

Closing modelling windows

Click  at the top corner of the window.

Resizing modelling windows

Windows have resize handles at the corners and borders. Drag them to change the window size. For more information, see your operating system documentation.

Full screen mode

Use the following typed commands to make PartMaker Modeling stretch to fill the screen:

GUI FULLSCREEN ON/OFF removes the taskbar and puts menu off the top of the screen.

GUI PREVIEWSCREEN ON/OFF removes the taskbar, puts menu off the top of the screen, and removes all toolbars.

The menus can still be accessed off the top of the screen.

Single window mode

Click **Windows fix/float**  to fill the graphics area (see page 143) with the active modelling window. To display a different modelling window, select it from the list on the **Window** menu.

For further details, see Window menu overview.

Multi-window mode

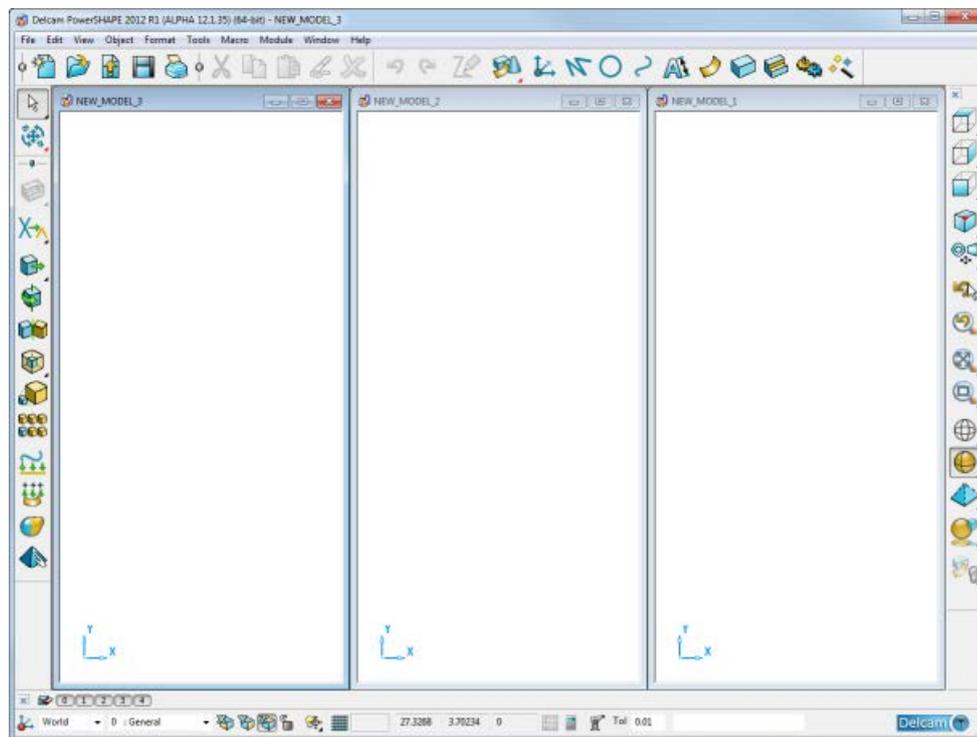
When the modelling window fills the graphics area, click **Windows fix/float**  to display each window in their own frames. These modelling windows are moveable and resizable. Each has a banner across the top showing the window name.

Click on the banner (or the border) of a window to bring the window to the front of other windows. Drag the banner to move the window around the screen.

For further details, see Window menu overview in the Menus and Toolbars manual

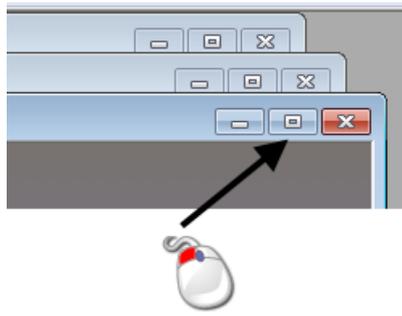
Tile vertically

This displays open modelling windows vertically so they are all visible within the graphics area. Their individual sizes depend on the number of windows open.



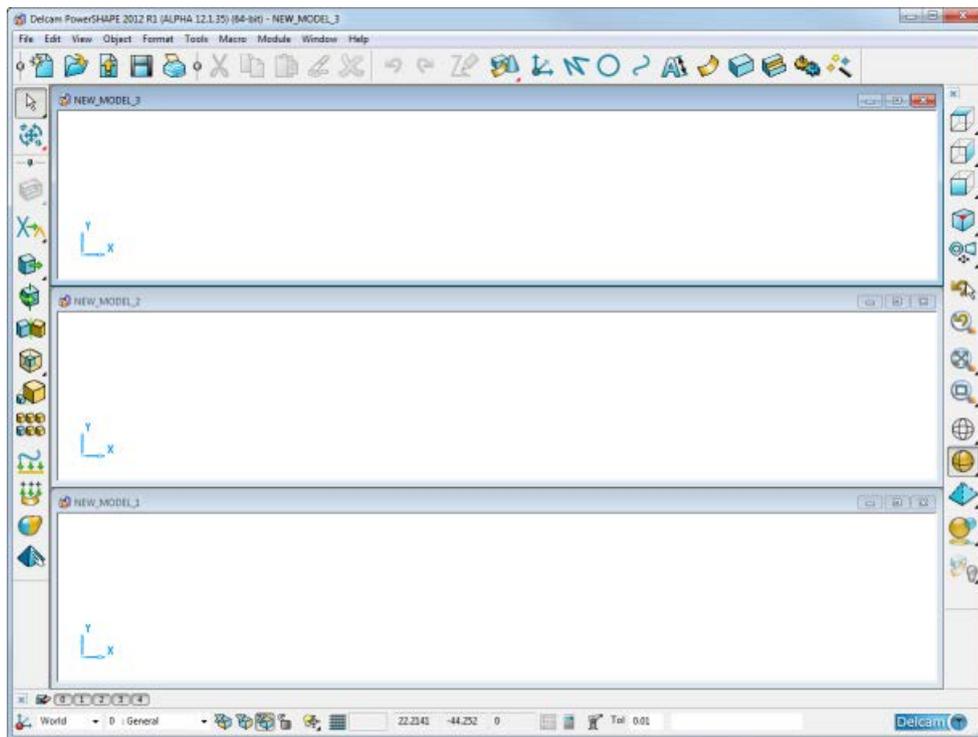
Tiled modelling windows can be resized and moved in the same way as any other window controlled by the operating system. However, moving one will almost certainly cause it to overlap some other windows.

Click on the **Maximise** button on a *Tiled* window to revert to a single window display.



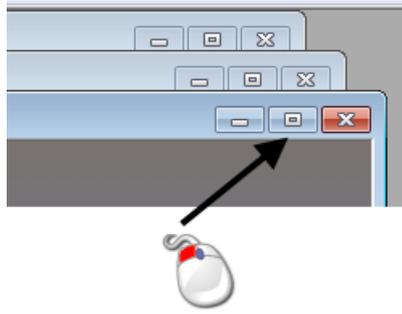
Tile horizontally

This displays open modelling windows horizontally so they are all visible within the graphics area. Their individual sizes depend on the number of windows open.



Tiled modelling windows can be resized and moved in the same way as any other window controlled by the operating system. However, moving one will almost certainly cause it to overlap other windows.

Click on the **Maximise** button on a *Tiled* window to revert to a single window display.



Intelligent Cursor

Use the following sections for information on using the Intelligent Cursor:

- What is the Intelligent Cursor? (see page 147)
- Construction lines (see page 149)
- Intelligent Cursor flyout (see page 151)
- Entering positions (see page 153)
- Selection (see page 155)
- Using a locked axis (see page 155)
- Editing objects (see page 159)

What is the Intelligent Cursor?

The Intelligent Cursor provides dynamic assistance for its two primary functions:

- **Point input** — It activates construction lines and labels for dragging, snapping and for point creation.
- **Selection** — It highlights the object under the cursor ready for selection. For further details, see Selection using the Intelligent Cursor (see page 155)

The cursor icon changes to assist you when selecting and editing objects. For further details, see Intelligent Cursor construction labels (see page 153)

Using the Intelligent Cursor

The Intelligent Cursor is enabled by default. You can enable and disable the Intelligent Cursor in one of these ways:

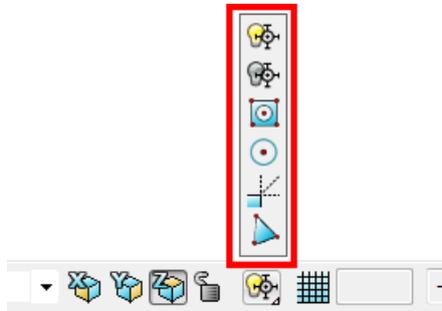
- Turning the Intelligent Cursor off or on (see page 148)
- Popup menu (see page 148)

- The Mouse page of the Options dialog (see page 149)

Turning the Intelligent cursor on or off

The Intelligent Cursor is enabled by default. You can enable and disable the Intelligent Cursor using the options on the Intelligent Cursor flyout (see page 151) on the status bar.

- 1 Right-click the Intelligent Cursor  button on the status bar to display the **Intelligent Cursor** flyout.



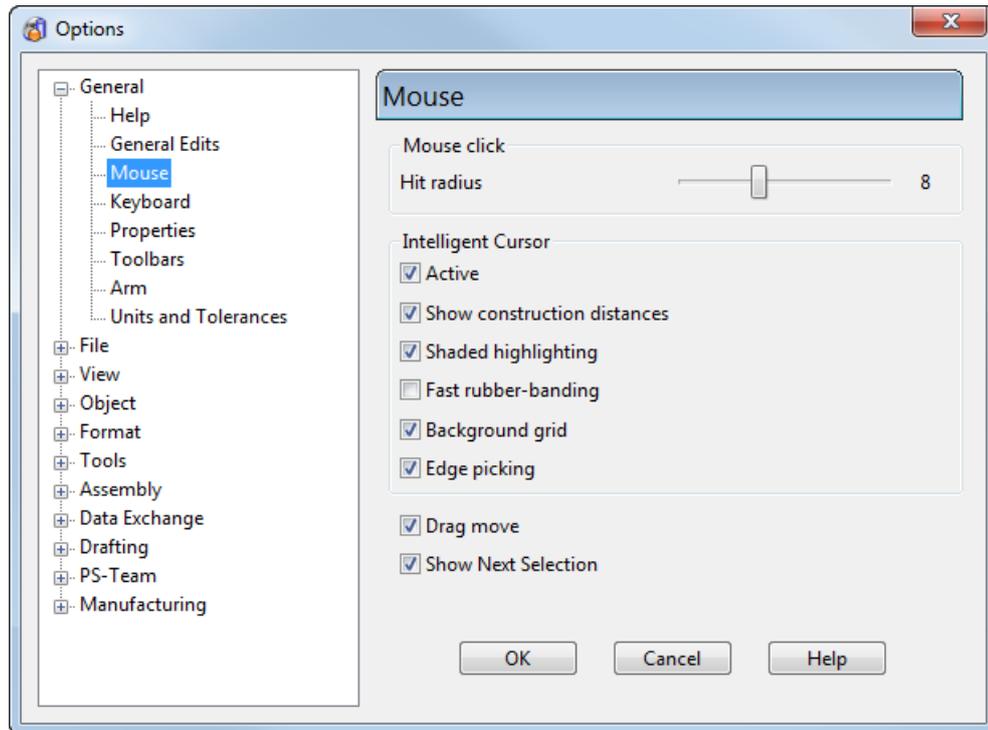
- 2 Click the **Intelligent Cursor Off**  button to disable the Intelligent Cursor.
- 3 Click the **Intelligent Cursor On**  button to enable the Intelligent Cursor.

View popup menu

- 1 In modelling mode, right-click in an empty space in the graphics window to display the View menu (see page 108).
- 2 Deselect the **Intelligent Cursor** option to disable it.
- 3 Select it again to enable the Intelligent Cursor.

Mouse page of the Options dialog

Use the **Active** option to toggle the **Intelligent Cursor**. This option is selected by default.



The **Intelligent Cursor** displays in the graphics window:

- various cursor types
- construction lines
- construction labels
- construction distances

When you click **OK**, PartMaker Modeling uses the new Intelligent Cursor settings. For further details on Intelligent Cursor options, see Mouse options.

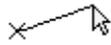
Construction lines

When you hover the Intelligent Cursor over an important key point (one that displays any of the construction labels (see page 153)), construction lines are created from that key point. You can now drag the cursor along a construction line and see the appropriate labels display. Clicking causes the position to snap to that point.

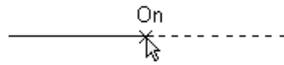
The following example draws a box:

- 1 On the Main toolbar, select the **Line**  button.
- 2 On the Line toolbar, select the **Continuous**  button.

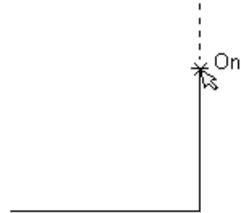
- 3 Click to start a line.



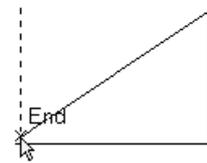
Drag along a construction line and click.



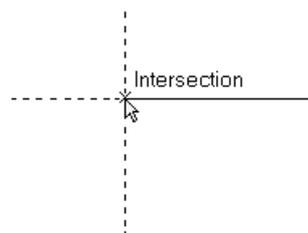
- 4 Drag along a 90° construction line and click. The line snaps to give a square intersection.



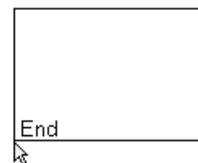
- 5 Drag the cursor down to the original start point and hover. The cursor displays **End** and the construction lines display.



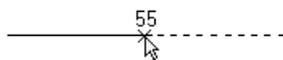
- 6 Drag along the vertical construction line until **Intersection** is shown.



- 7 Snap to the start and finish with one more click when **End** is displayed.



If the Intelligent Cursor Gridding is enabled, the label **On** in the above diagrams is replaced by a distance from the last point selected. For example:



This provides a convenient way to enter accurate dimensions.



To disable construction lines temporarily, press and hold the **Shift** key.

Turning construction lines on or off

The Intelligent Cursor's construction lines are enabled by default. To disable construction lines, click the Construction Lines  button from the Intelligent Cursor flyout (see page 151) on the Status bar. The construction lines are not displayed until they are turned back on. All functionality is still available.

Intelligent Cursor flyout

- 1 Click  (*Status bar*) using the right mouse button to display the **Intelligent Cursor** flyout.
- 2 Select the Intelligent Cursor option you require.



The Intelligent Cursor options are for point entry only (not selection).



Intelligent Cursor on (default) (see page 148).



Intelligent Cursor off (see page 148).



Snap to keypoints (see page 152).



Snap to centre keypoints (see page 152).



Construction lines on/off (see page 151).



Click this button to snap the cursor to the nearest node instead of the exact point on the mesh.

- 3 Click  (*Status bar*) to return to the default flyout setting when you have finished using the Intelligent Cursor options.

Snap to keypoints

Use **Snap to keypoints** to detect only keypoints in your model when entering points. Anything that is not a keypoint is ignored by the functionality in use. On wireframe items the cursor will snap to the nearest keypoint.

When this option is selected, the cursor changes to  as a reminder that point entry is restricted to keypoints.

To activate **Snap to keypoints**:

- 1 Click  (*Status bar*) using the right mouse button to display the Intelligent Cursor flyout (see page 151).
- 2 Click  **Snap to keypoints**.

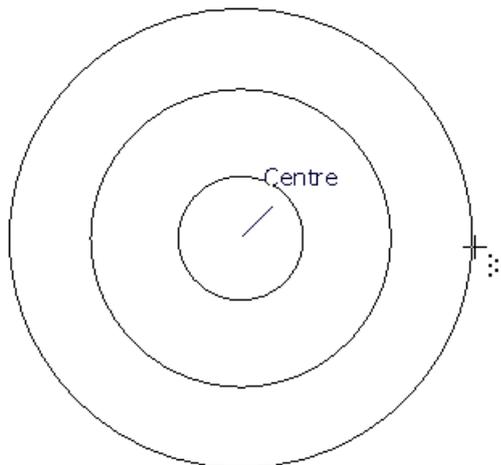
Snap to centre keypoints

Snap to centre keypoints is a restrictive option that filters out all point input except circle and hole centres. This is particularly useful when dimensioning a large number of holes.

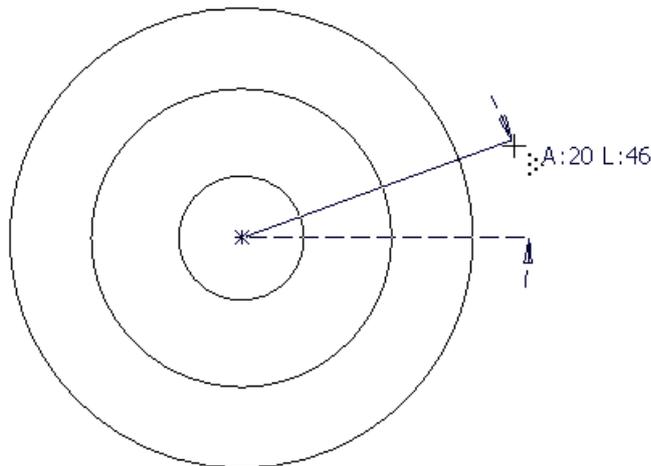
When the option is selected, the cursor changes to  as a reminder that point entry is restricted to centre keypoints.

Any point on the geometry of circles will snap to the centre of that arc. For example,

- 1 Hover the cursor over an arc



- 2 Click to snap the centre of that arc.



It is not necessary to select the centre of the arc directly.



This is a useful technique when working with overlapping or concentric circles where the centre is not easily identifiable.

To activate **Snap to centre keypoints**:

- 1 Click  (status bar) using the right mouse button to display the Intelligent Cursor flyout (see page 152).
- 2 Click  to activate **Snap to centre keypoints**.

Entering positions using the Intelligent Cursor

The Intelligent Cursor shows additional information to help you to define the position you want when entering positions.

When you move the cursor over an object, construction labels (see page 153) are displayed by the Intelligent Cursor. These construction labels:

- are shown and hidden as you move the cursor over them.
- indicate what happens if you click while the label is displayed.
- make snapping with the cursor more accurate. The labels display when key points are within snap range of key positions in the model. You can click when a label (see page 153) is displayed to snap to the cursor at the labelled position.



The Intelligent Cursor (see page 148) is enabled by default.

Intelligent Cursor construction labels

The labels you may see displayed by the Intelligent Cursor include:

- **End** — Shows that the position will snap to the end point of a line, arc, curve, or profile.
- **Mid-point** — Shows that the position will snap to a point mid way between the start and end points of a line or arc.
- **Centre** — Shows that the position will snap to the centre of an arc or circle, or hole feature. The name of the feature is also displayed.
- **Key-point** — Shows the keypoints on curves and surfaces where they can be manipulated. The position will snap to the nearest keypoint on the object under the cursor (for example, a surface patch corner or a curve key point). This label can also show the origin of a workplane.
- **On** — Shows that the cursor is on a line and the position will snap to a point that lies exactly on the object, but not at one of its key points (for example, end, corner, centre points).

If the **Background Grid** option is *ON*, a number representing a distance along the object from the last key point visited by the cursor is displayed. The values are rounded to the nearest sensible value. Zoom out to see coarser increments. Zoom in to see finer increments.

- **In**
Shows that the position will snap to a point on the inside of a surface patch or a solid face (not one of the keypoints or positions along a curve).
- **Intersection**
Shows where two objects cross and that the position will snap to the intersection between the two objects under the cursor.
- **Tangent**
Displays when a tangent to an arc can be made. The position will be calculated so that the line or arc being created is tangential to the line or arc under the cursor. The position will not necessarily be under the cursor. This works whether you are placing a line to an arc, an arc to a line or an arc to an arc.
- **End sign** 
This displays when clicking will complete a curve or continuous series of lines.
- **Closing a curve** 
This displays when clicking will create a closed curve.



*We strongly recommend that you normally work with the **Construction Level** switched OFF. For further details, see [Format > Levels](#).*

Selection using the Intelligent Cursor

When the Intelligent Cursor is enabled, objects are highlighted as you move the cursor over them. This shows which objects can be selected if you click at that point. For further details, see Turning the Intelligent cursor on or off (see page 148).

When several objects are under the cursor and one of them is selected, the Intelligent Cursor highlights the next object from the database at that position. A click now deselects the selected object, selects the highlighted object, and highlights the next one. This enables you to select the correct object from the group.

You can also right-click to display the object's menu and click **Next Selection** to select the next object.

The Intelligent Cursor option is displayed on the popup menu. Select or Deselect to toggle the Intelligent Cursor on and off. Selecting this option is the same as selecting the Intelligent Cursor **Active** option from the Mouse options page of the **Options** dialog.

Using a locked axis with the Intelligent Cursor

The Intelligent Cursor can be an effective tool when used with a locked axis.

- Intelligent Cursor modification keys (see page 155)
- Creation using a locked axis (see page 156)
- Dragging on a locked axis (see page 157)
- Moving points on a locked axis (see page 157)
- Snapping to points using a locked axis (see page 158)

Intelligent Cursor modification keys

The following keys on the keyboard can be used to modify the use of the Intelligent Cursor.

X — Press and hold the **X** key on the keyboard to lock the X axis whilst in any construction mode, dragging objects and moving points.

Y — Press and hold the **Y** key on the keyboard to lock the Y axis whilst in any construction mode, dragging objects and moving points.

Z — Press and hold the **Z** key on the keyboard to lock the Z axis whilst in any construction mode, dragging objects and moving points.

N — Press and hold the **N** key on the keyboard to move points locked on the axis of the normal of the surface.

Creation using a locked axis

As an alternative to using construction lines during line creation, you can use the **X**, **Y**, and **Z** keyboard modifier keys to lock the corresponding orthogonal axis. This can be useful for additional precision in complex models to control the direction of item construction.

When you are creating in free space, the cursor grids the distance along the locked axis as you move the mouse.

- 1 Select **Line** .
- 2 Enter a point.
- 3 Press and hold down the required **X**, **Y** or **Z** key to lock the **X**, **Y**, or **Z** axis.



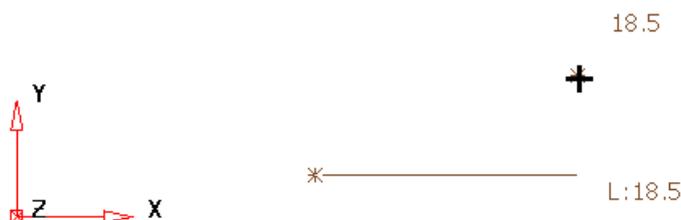
You may need to change the plane you are working in to lock the required axis. For example, to lock the Z axis you need to be entering points in the XZ or YZ plane (not XY).

- 4 Move the cursor to create the line to the required length.

The example below shows the construction line displaying along the X axis, whilst the mouse cursor is above. The potential line creation result is displayed at an angle of 25°.



The example below shows a similar mouse position above the x axis, but the X key has been held down to force the line creation to be locked onto the X axis, despite the cursor position located above the X axis.



Dragging on a locked axis

You can drag existing objects along a locked axis, by pressing and holding the corresponding keyboard modifier key X, Y, or Z when you start to drag. The intelligent cursor remembers where the drag started and locks that axis.

All objects in the selection are dragged along the corresponding locked axis, allowing multiple items to be moved together.

Moving points on a locked axis

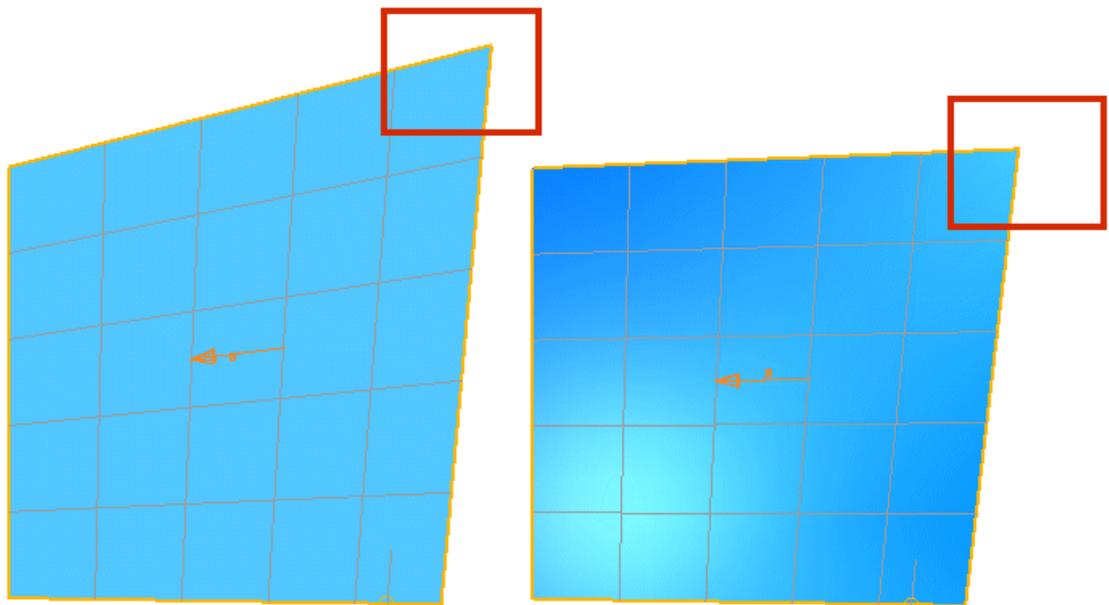
When moving points on a power surface, you can use the **N** keyboard key to lock the axis to guarantee the point is being moved in a perpendicular direction from the surface. The locked axis is the original point's surface normal.

When you use the **N** key, the drag point is locked on the axis, normal to the surface from where you picked the original point.

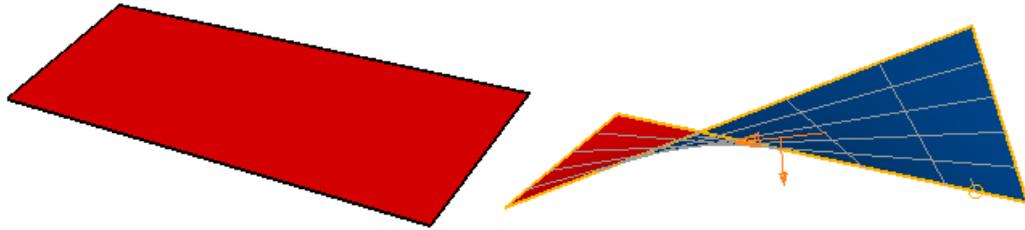


This does not work if you are using a view that looks down the surface normal. Change views to see the point moving perpendicular to the surface.

In these examples, both corner points on the surfaces have been dragged to change their shape from a standard plane. The **N** keyboard key was used for the image on the right to move the surface in a perpendicular direction to the surface normal.



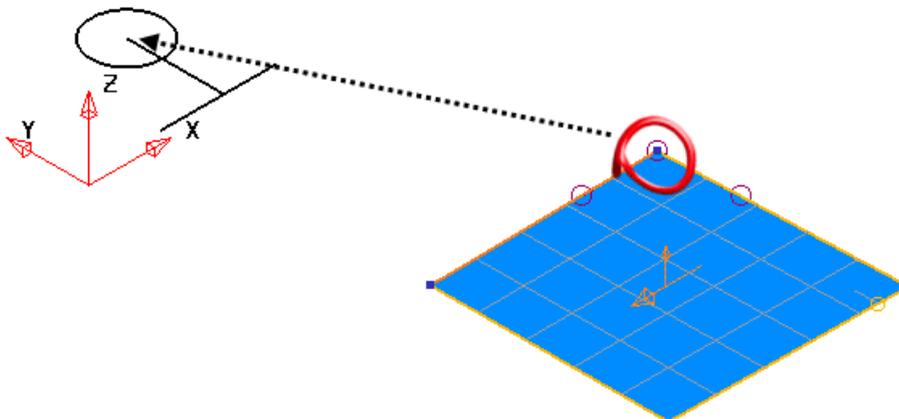
The effect is not truly visible until the surfaces are rotated, seen in the following rotated examples.



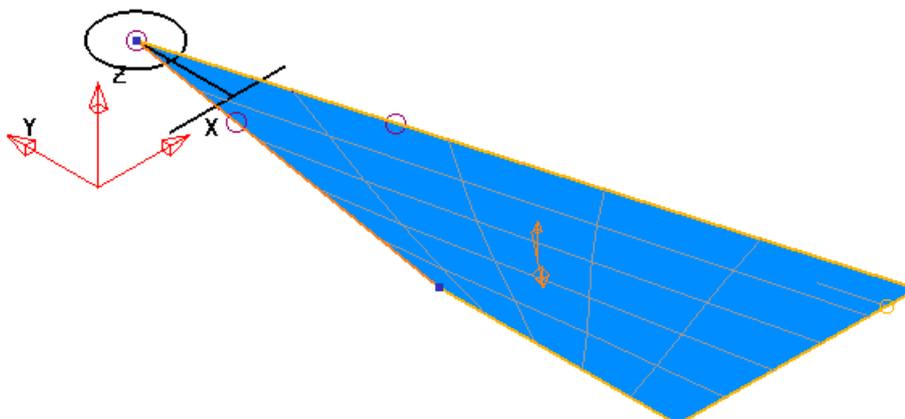
Snapping to points using a locked axis

When pressing and holding one of the keyboard keys **X**, **Y**, or **Z** to lock the axis during line construction, drag selected objects or move points, you can still move the cursor to a nearby point to snap to it. The axis remains locked as the intelligent cursor projects the resulting point from the snapped point to the locked axis.

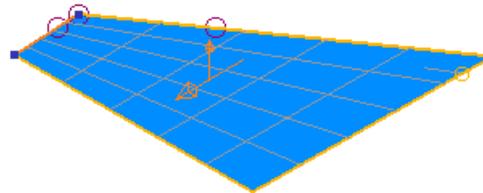
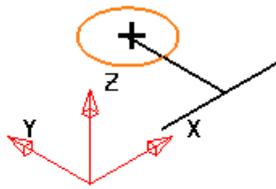
The example below shows a plane with a selected corner, ready to snap to the centre of the circle in the direction shown by the arrow.



The example below shows the corner point dragged and snapped to the circle centre.



The example below shows the corner point dragged to the circle centre, but using the **X** key to lock the X axis.



Editing objects using the Intelligent Cursor

When you start to drag an object, a group of objects or an editing handle of an object, the cursor feeds back information about points you can snap to.

The Intelligent Cursor:

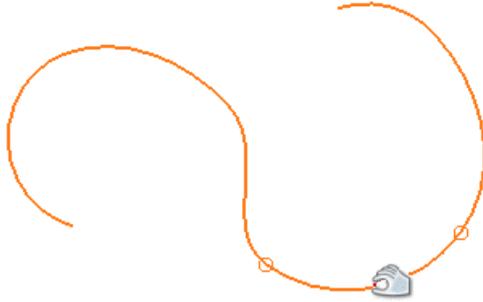
- finds intersections between wireframe items and a surface or solid.
- snaps to the projected intersection of wireframe items and surfaces.
- snaps bezier curves to triangles in a symbol imported as a solid. For further details, see **STL/DMT** options in the Menus and Toolbars manual.
- snaps to triangles in symbols (not applicable to components).

For further details, see Using the Intelligent Cursor to enter positions (see page 153) and Selection using the Intelligent Cursor (see page 155).

Object highlighting

When the intelligent cursor is enabled, objects that you can select are highlighted when you move the cursor over them.

The two keypoints nearest the cursor on a curve are highlighted and are displayed as small circles when you move the cursor over them. This helps you find them more easily and quickly. The keypoint markers display in both creation and edit modes.



For further details, see Selection using the Intelligent Cursor (see page 155).

Models

A model is a representation of a “real life” (usually engineering) component, built on the computer. You build models from a set of geometric objects (lines, arcs, curves, and surfaces) which together represent the components.

You can view a model from any angle and shade it to look very similar to the real component. You can also model other properties such as weight and volume.

Creating a model

- 1 Select **File > New** from the menu.

This creates a new model file and opens a new graphics window. The new window automatically becomes the active window. The window title shows the model name (in this case New_Model) and a sequential number.

- 2 The first time you save this model, the **Save Model As** dialog is displayed, which prompts you to enter a name for your model. For further details, see Saving a model with a new name (see page 161).

Saving a model

When you want to keep your work,

- 1 Select **File > Save**.

If a model was originally opened with write access, it is automatically saved, overwriting the stored version.

- 2 For a new or read-only model, you must use **File > Save As** to save the model with a new name. (see page 161)

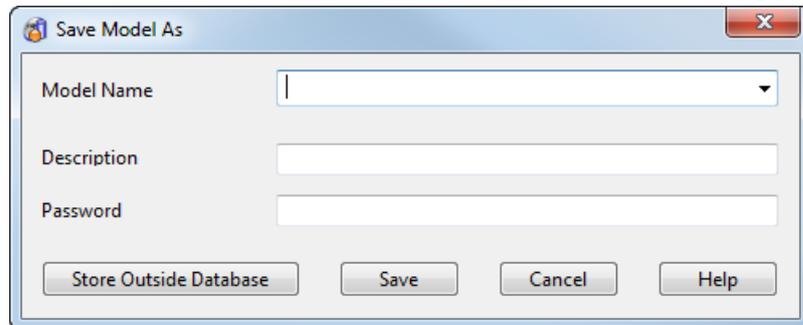
Saving a model with a new name

When you want to keep your work but don't want to overwrite an existing model, you can save the model using a new name. The location of the model you save is defined in the Tools > Options > File > Model dialog. As well as the path to the model, the **Always save and open from outside the database** option defines if the model is saved inside or outside the database.

- 1 Select **File > Save As** from the menu.
- 2 The dialog that is displayed depends on the setting of the **Always save and open from outside the database** option:

- When you first use the program, the **Always save and open from outside the database** option is selected by default and you automatically use the standard Windows **Save As** dialog to save models outside the database.
- To save models in the `shareddb` database, select **Tools > Options > File > Model** and deselect the **Always save and open from outside the database** option.

Selecting **Save As** from the **File** menu now displays the Save Model As dialog.



- 3 Use the dialog to save your model using a new name.



*If your models are single files and stored outside the database, you can display the **Save As** dialog instead of the **Save Model As** dialog. To do this, select the **Always save and open from outside the database** option.*

Checking for corruption

To spot any corruption as soon as possible after it has occurred, the File Doctor is run whenever you save a model. For further details, see **Tools > Model Fixing**.

If File Doctor finds any problems, you are asked to save the model under a new name or run File Doctor in fix mode to correct the problem.

To stop running File Doctor on every save, deselect the **Check Model Before Save** option on the **Model** page of the **Options** dialog.

A set of options is available from the **Model** page of the **Options** dialog. You can select which ones to use when using File Doctor. For further details, see **Model options**.

Closing a model

- 1 If you have more than one model open, make active the model you want to close by clicking in its graphics window or selecting the model from the **Window** menu.
- 2 Select **File > Close** from the menu.

The active graphics window closes.

- 3 You will be prompted to save changes you have made to the model:
 - Click **Yes** to save the changes to the original file name.
 - Click **No** to close the file and lose any changes that you have not saved.
- 4 The model is checked to see if the data can be compressed. If the program decides that compression is possible, a dialog is displayed:
 - Click **Yes** to compress the model data and close the file.
 - Click **No** to close without compressing the model data.



To save an edited file to a new filename you need to use File > Save As (see page 161).

Other ways to close a model

A model is also closed if you do one of the following:

- Close the last open graphics window of the model (by selecting **Close** from the **Window** menu).
- Exit the program.
- Using the **File > Close and Zip** command.

In all cases, you are prompted to save any unsaved edits.

Opening a model

Each model you open is displayed in a separate graphics window on the screen. You can also show each model in a number of windows and can have up to 20 models open at the same time.

You can open a model and select its read/write status.

- 1 Select **File > Open** from the menu.

The **Open Model** dialog is displayed

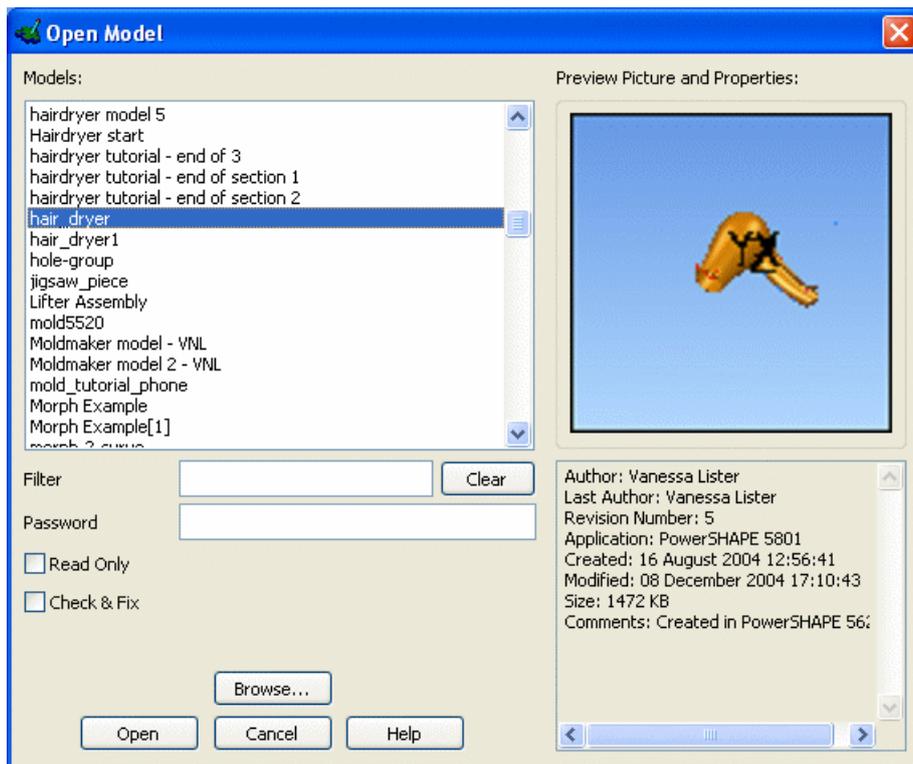
- 2 Use the dialog to open a model.

For further details, see Open Model dialog (see page 164).

If your models are single files and stored outside the database, you can always display the standard Windows **Open Model** dialog. To do this, turn on the **Always save and open from outside the database** option on the **Model** page of the **Options** dialog. For further details, see Model options.

Open Model dialog

Use this dialog to open models.



Models - Select a model name from the list.

If you select any item, you can use the keyboard arrow keys   to move to the next model.



*Models of the form NEW_MODEL_<number> are created when **File - New** is selected. These only appear in the **Models** list if you have system administrator security.*

Filter - When the list of model names becomes very long, it can be difficult to find the model you want. Entering a filter reduces the list to only those models which match the filter pattern, for example, entering a filter of *.vl will list only those models which have the extension .vl.

We suggest that you use a designer's initials for model name extensions. For example, **mymodel.vl**.

The filter is not case sensitive. Entering a filter of *.VL will list models that have the extension .VL as well as those with the extension .vl. For further details, see Filter options (*Menus and Toolbars*).

Clear - This clears the text in the **Filter** box.

Password - If the model is protected by a password, enter it in the text box.

Read Only - If you want to open the model, but prevent it from being overwritten, click the **Read Only** box until it shows a tick.

If you decide to save any changes, you will need to save the model under a new name using **File - Save As**. For further details, see Saving a model with a new name (see page 161).

Check & Fix - When ON, File Doctor examines the selected model and corrects any errors before opening the model. For further details, see Tools - Model Fixing - File doctor (*Menus and Toolbars*).

Preview Picture and Properties - If a thumbnail of the model has been saved, it is displayed in the top window. Information about the model appears in the bottom window, including file dates, size and description.

Browse - This opens models stored as single files and not in the database. Click the **Browse** button to display the standard Windows **Open Model** dialog. Use this dialog to select and open a model.

Models stored in the directory format and not in the database must be imported.

Search - This allows you to create a filter based on the fields used for temporary and vault models. It is only available if you have a naming_convention.mac file in your shareddb folder.

The naming_convention.mac file allows you to save vault and temporary models. For further details, see Defining your naming scheme (Managing your PowerSHAPE system).

Click **Search** to display the Search for files dialog.

Recover - This button only appears if you select a model that was not closed properly, for example, if there was a power failure while working on the model. **Recover** opens the model and restores it back to its state before the failure.

Open - The selected model is opened. If the model you are opening is shaded, you can open the model in wireframe mode by pressing the  button next to the progress bar on the status bar.

Cancel - Removes the dialog from the screen without opening a model.

Starting up PartMaker Modeling and opening a model

When you start up PartMaker Modeling at the command prompt, you can open an existing model at the same time by typing:

powershape model_name

For further details, see Other ways of opening psmodels in PartMaker Modeling (see page 166).



For the above commands to work, you must have `FILE OPEN IFNONE` in your login macro.

Other ways of opening psmodels in PartMaker Modeling

As well as using the **File Open** command to open models in the single file format (`.psmodel`), you can also:

- Double-click a `.psmodel` file. If the program is not running, it will start up and then open the model.
- Right-click the `.psmodel` file in Windows Explorer and select **Open**. You can open a selection of models in this way.
- Drag and drop one or more `.psmodel` files into the program window.
- Double-click a PartMaker Modeling.exe shortcut.
- Double-click a `.psmodel` shortcut.
- In a web browser, click on a URL to a `.psmodel`; for example, file://pshape/models/bottle_example.psmodel. You can either open the model immediately or download it and open it later.

Recently opened models

Select the **Recent Files** option at the bottom of the **File** menu to see the last four models listed. The *read* or *write* status when they were opened is also shown.

To open a recently opened model, select the **File** menu and click the model name you want. It will open with the same read or write status as last time.

If the model is not listed or you want a different read/write status, use the **File > Open** option. For further details, see *Opening a model* (see page 163)

Closing and opening zipped models

Files can be compressed to make them take less disk space. This is known as *zipping* a file. By zipping a model, you can save about 60% of disk space, but it takes longer to open as it needs time to unzip.

To close and zip a model,

- 1 Save the model if you haven't already done so.
- 2 Select **File > Close and Zip** from the menu.

The model file is stored in a zip file and the original file of the model is deleted.

To open a zipped model,

- 1 Select **File > Open** from the menu.
The **Open Model** dialog is displayed.
- 2 Select the model using the dialog.
- 3 If the model is stored outside the model database, click the **Browse** button to display the **Open** dialog. Use this dialog to select and open the zipped model. You may need to select **Zip** from the **Files of type** menu.
- 4 Click **Open**.

A copy of the model is extracted from the zip file and stored in the same folder as the zip file. While the model is open, any changes are saved to the copy.



*Zippered directory models are opened using **File Import**.*

When you finish working on the model, you can close it using:

- **File > Close** — This saves the copy you are working on and deletes the zip file.
- **File > Close and Zip** — This adds the copy to the zip file and then deletes the copy.

Changing the view of a model

Each window can contain:

- a single view of the model.
- two views of the model splitting the model window vertically.
- four views of the model splitting the model window vertically and horizontally.

Each of these views can be from any direction, at any scale, and can be parallel, isometric or perspective projections. Surfaces can be drawn as wireframe or shaded and objects can be blanked from specific views to reduce clutter.

To change the view of a model:

- Use the **Window** menu to create, delete and select windows.
- Use the **View** menu commands to alter the view of the model in the active window.

Select **View > Layout** to display the **View Edit** dialog which allows you to set up customised view arrangements. For further details, see View Edit dialog (*Menus and Toolbars*).

Select **View > Toolbars > Views** to display the **Views** toolbar, which provides many options to change the view. For further details, see View > Toolbars > Views (*Menus and Toolbars*).

- Use the middle mouse button (used with certain keyboard modifier keys) to dynamically alter the view by rotating, zooming and panning. For further details, see Middle mouse button (see page 136).
- **Quick Rotation** allows rotation of the selected object when you hold down the **R** on the keyboard and scroll the mouse-wheel. The object is rotated about its principal axis by **1** degree for each notch on the mouse-wheel.

Resetting a model

You can delete all the changes you have made to a model since you last saved it.



Reset permanently removes all unsaved edits from the active model. A warning is displayed, but once you click **Yes**, it cannot be undone.

- 1 From the **File** menu, select **Reset**. A warning is displayed.
- 2 Click **Yes** to lose the edits. The model is reset.
Click **No** to keep the edits. The reset command is ignored.

Changing the password of a model

You can change the password of a model as follows:

- 1 Open the model with read-write permissions.
- 2 In a graphics window of the model, click in an empty space to make sure nothing is selected.
- 3 From the **File** menu, select **Properties** to display the File Properties dialog
- 4 Use the dialog to edit the details and password of the model.

Finding out what a model contains

You can find out how many objects are stored, details of current levels and levels of objects in the selected model. You can edit details and information about a model and you can change the password of a model.

From the **File** menu, click **Properties** to display the **File Properties** dialog.

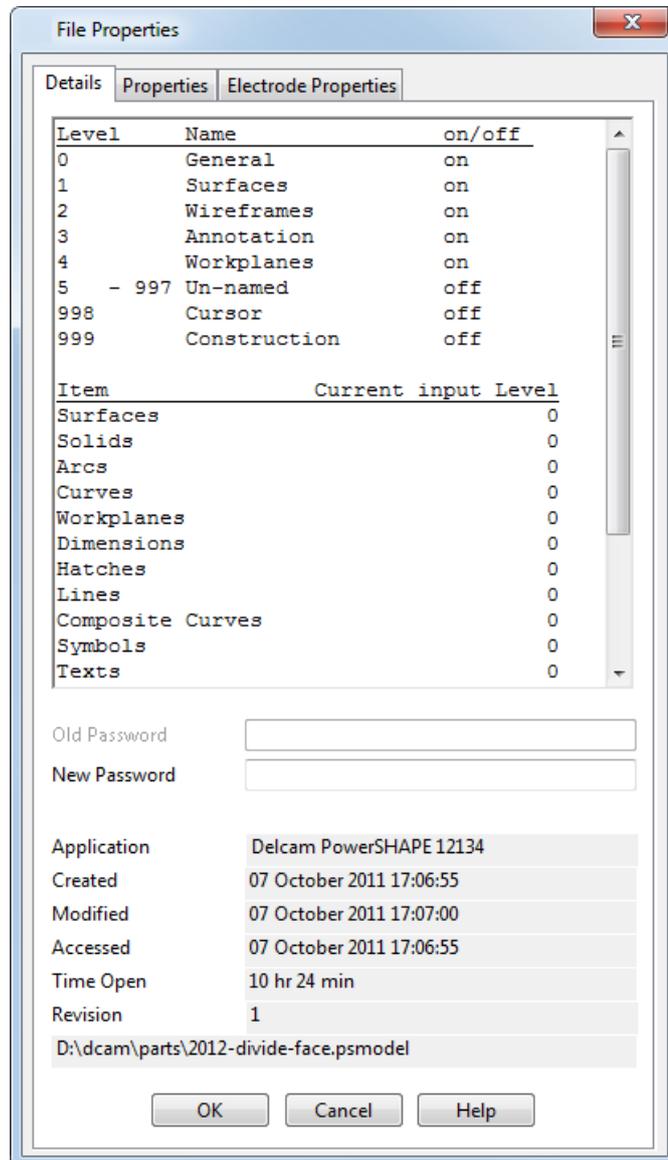
This contains the following tabs:

- **Details**
- **Properties** (see page 170)

- **Custom** (see page 171)

File Properties dialog - Details

The **Details** page of the **File Properties** dialog contains information about levels and objects for the current model in PartMaker Modeling.



The **Details** page tells you the following:

- Which levels are turned on and off.
- The current creation levels for objects.
- The number of objects of a particular type in the model.
- The levels on which particular types of objects lie.
- Totals for objects if they lie on more than one level.

Some of the information is automatically generated by the program. This includes:

- Details about when the model was created, last modified and last accessed. The length of time that the model has been open is also included.
- Path of the model.

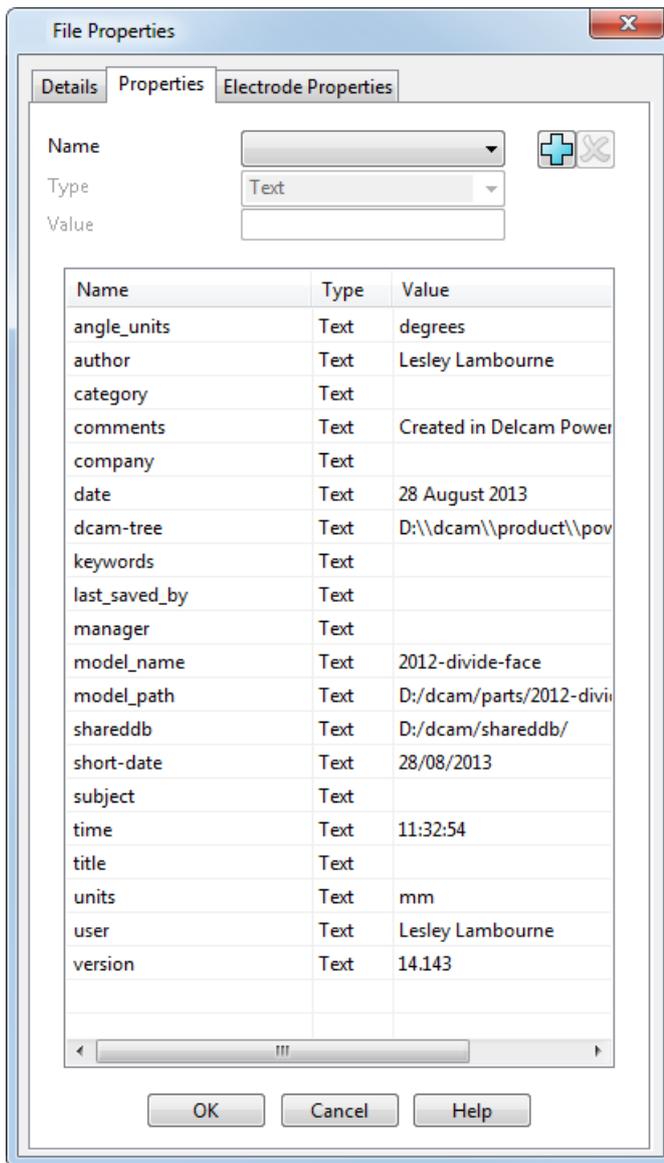
OK - This removes the dialog from the screen.



To print the information that is displayed, right click in the dialog and select **Print** from the popup.
To save the information that is displayed, right click in the dialog and select **Save To File**.

File properties dialog - Properties

The **Properties** page of the File Properties dialog displays information about the model.



Name - Enter the name of the custom property and click **Enter**. Other parameters now become available and the name is added to the custom property list.



-Click to add a property.



- Highlight a name in the list and click this button to remove a name from the list.

Type - The default property type is *Text*, but you can also choose *Date*, *Number*, *Yes or No* from the drop down list. The custom property list is updated to reflect your selection.

Value - Enter an appropriate value. The custom property list is updated accordingly.

OK - When you have entered all your custom properties, click **OK**.

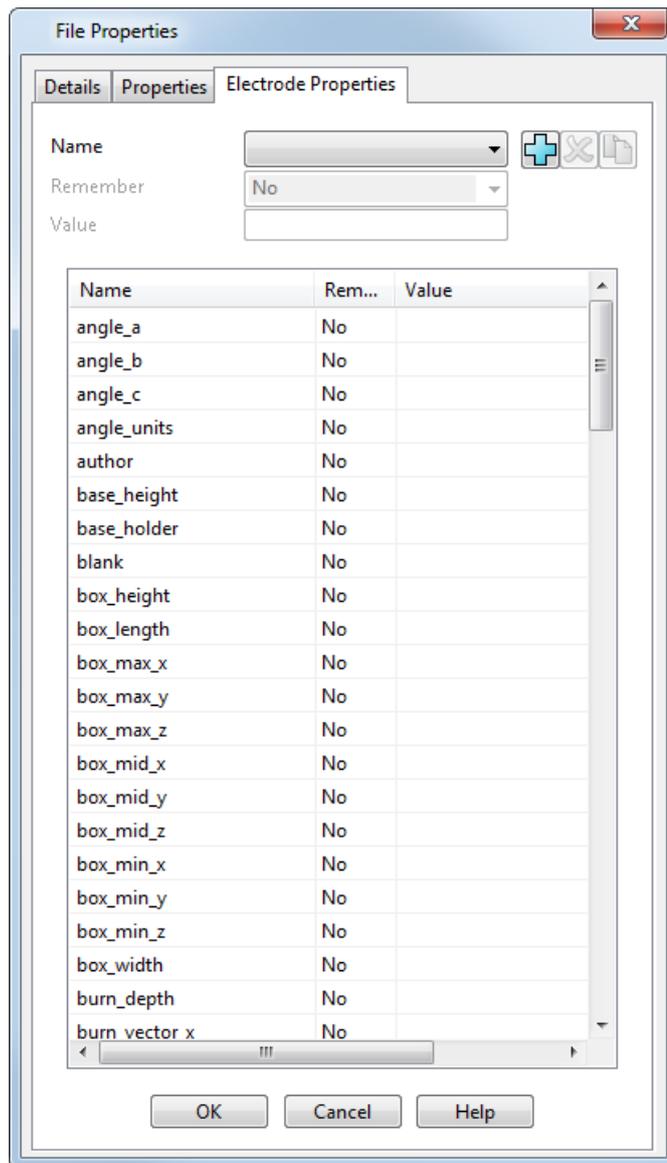
You can enter information in this dialog so that it can be included in text on your model or drawing. In addition, you can also use some information that is automatically generated by the program when you save a model.

File properties dialog - Electrode Properties

Use **Electrode Properties** to create setup strings that can be used to add custom electrode text to a drawing.

You can enter information in this dialog so that it can be incorporated into text on your electrode model or drawing. In addition, you can also use some information that is automatically generated by the program when you save a model.

Information in this dialog is used when adding an electrode substitution string to a drawing template.



Name - Enter the name of the property and click **Enter**. Other parameters now become available and the name is added to the property list.



- Click to add a property.



- Highlight a name in the list and click this button to remove a name from the list.



- Create a copy of the substitution string. on the Windows clipboard so it is ready to paste elsewhere.

Remember - Select **Yes** or **No** from the drop down list. If you select **Yes**, then the value for that property is automatically entered in the **Additional Electrode Details** dialog. The custom property list is updated to reflect your selection.

Value - Enter an appropriate value. The property list is updated accordingly.

OK - When you have entered all your properties, click **OK**.

Deleting a model

You can delete unwanted models that are stored in the database. Models stored outside the database can be deleted like any other file using Windows Explorer.

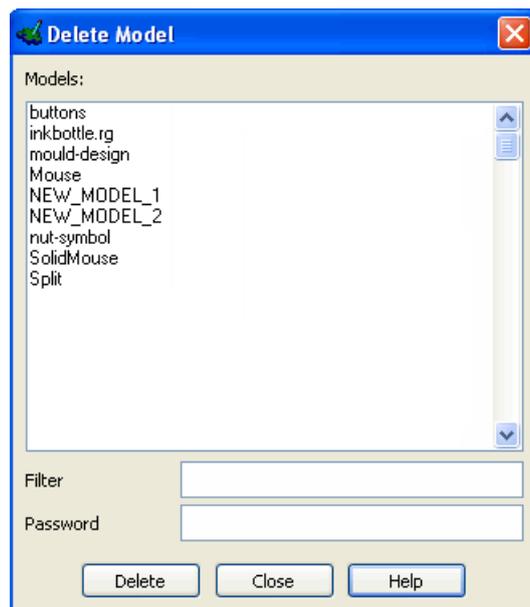
- 1 From the **File** menu, select **Delete** to display the Delete Model dialog (see page 173).
- 2 Use the dialog to delete the model.



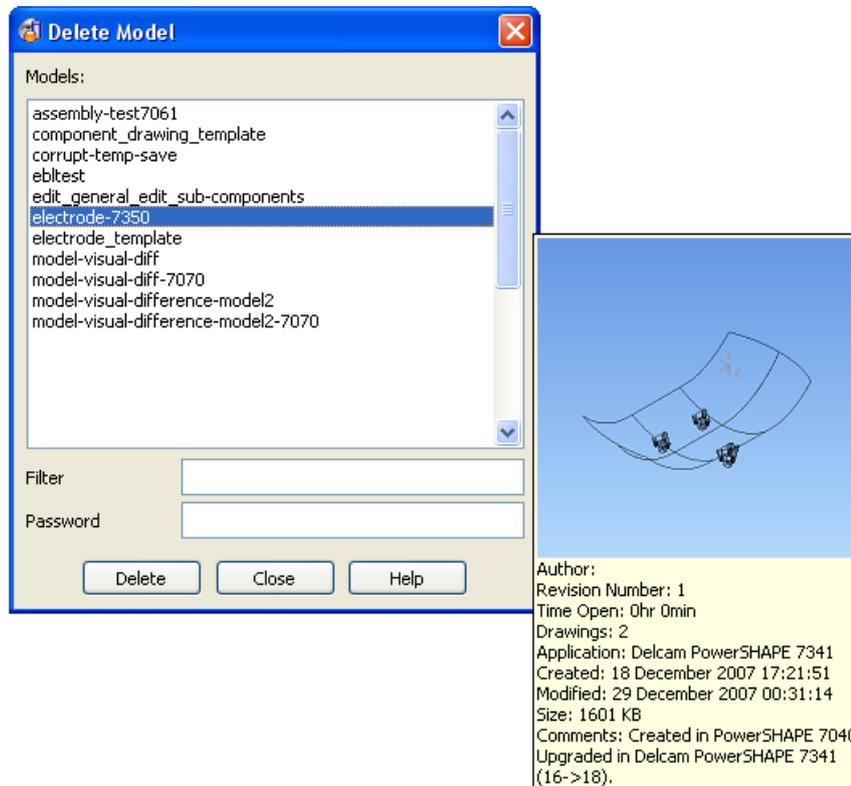
WARNING: Deleted models are permanently removed and cannot be recovered.

Delete Model dialog

Use this dialog to delete models.



Models — Click on a listed model name (to select it) and the item is highlighted. If you select any item, you can use the keyboard arrow keys   to move the highlight and scroll the list. The graphical tooltip displays a thumbnail of the model and file information.



*Models of the form NEW_MODEL_<number> are created when you select **File > New**. These are displayed in the Models list only if you have system administrator security.*

Select multiple listed model names to be deleted using the standard



SHIFT-Click or



CTRL-Click technique to select them. Move the cursor over the selection to see the individual thumbnails.

Filter — When the list of model names becomes very long, it can be difficult to find the model you want. Entering a filter reduces the list to only those models which match the filter pattern, for example, entering a filter of *.v/ will list only those models which have the extension .v/.

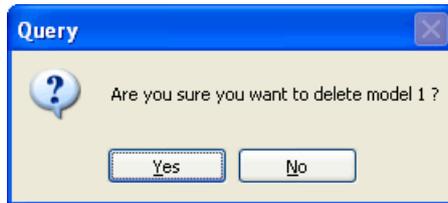
The filter is not case sensitive. Entering a filter of *.v/ will list models that have the extension .VL as well as those with the extension .v/.

For further details, see Filter options (*Menus and Toolbars*).

We suggest that you use a designer's initials for model name extensions. For example, `mymodel.vn1`.

Password — If the model is protected by a password, enter it in the text box.

Delete — When you click this button, a confirmation message is displayed:



Click **Yes** to remove the model from the database and return to the **Delete Model** dialog. If the **Use Recycle Bin** option is selected on the **Model** page of the **Options** dialog, the model is sent to the Recycle Bin (providing the bin is not full). Otherwise, the model is permanently removed from the system.

Click **No** to do nothing and return to the dialog.

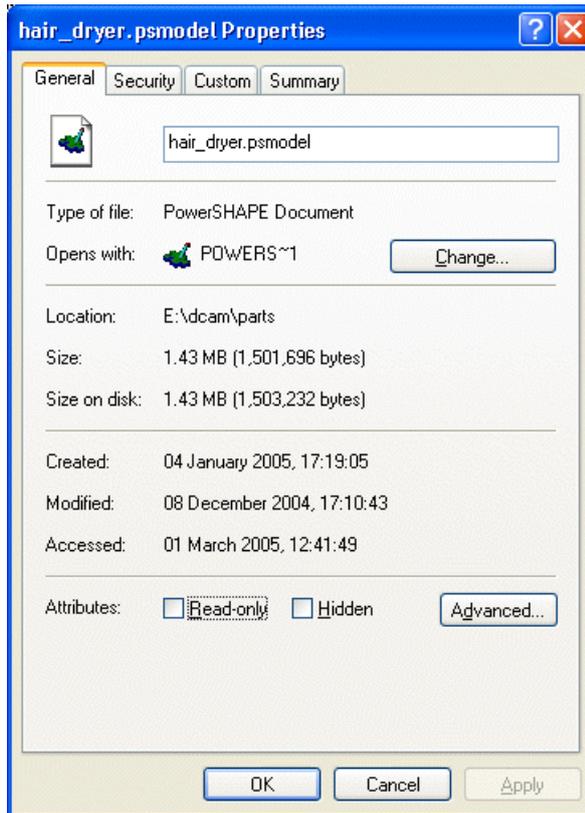
Close — When you have finished deleting models, click **Close** to remove the **Delete Model** dialog from the screen.

Finding out information about a model

You can find certain information about a model, without opening it.

- 1 In Windows Explorer, select a psmodel.

- 2 From the **File** menu, select **Properties** to display the File Properties dialog for the model.



The different tabs display information that is stored whenever a model is saved as a psmodel. For example:

- Date of creation
- Application name
- The name of the model and its description
- Current user as the Author (on the **Summary** page)
- Version of PartMaker Modeling in the **Comments** section (on the **Summary** page)

When opening a psmodel, the following will be read from the Windows Explorer's **File > Properties** dialog and stored by PartMaker Modeling:

- title
- subject
- author
- keywords
- comments
- last saved by
- revision number

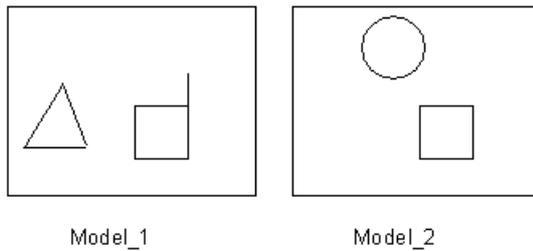
- manager
- company information

This information can be used when creating text.

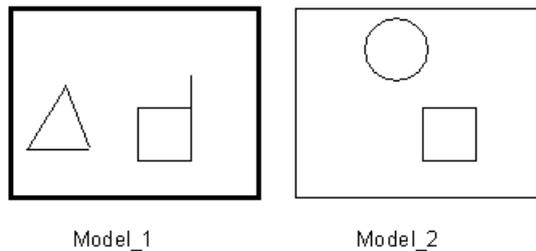
Comparing two models

You may have two models of the same component and want to see what the differences are between the two models.

- 1 Open the two models you wish to compare.
- 2 Suppose we have the following two models.



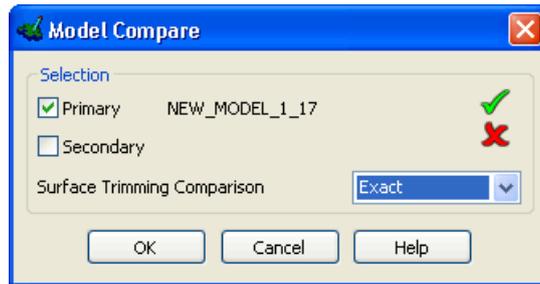
- 3 Click in the graphics window of the model in which you want to display the differences. This makes the graphics window active.
- 4 In our example, we made Model_1 the active model.



- 5 Select **Model Analysis**  on the **General edits**  flyout.
- 6 Click **Model Compare** (model analysis toolbar).
- 7 Use the Model Compare dialogue box (see page 178) to complete the comparison.

Model Compare dialog

Use this dialog to compare models.



Primary selection - This is used to choose the model in which the differences are shown. On the right of this, the name of the model is displayed. When you first display this dialog, the **Primary** selection is the model of the active graphics window. You can change the **Primary** selection if you want. Do this by pressing down the button on the left of **Primary**, if it not already pressed down, and then clicking in another graphics window. The name of the model of the new selection is displayed. We will call this model the **primary model**.

Secondary selection - This is used to choose the second model in the comparison. On the right of this, you can see a cross. This indicates that no model is selected yet. To select the second model, press the button to the left of the **Secondary** button and click in the graphics window of the model you want to compare. We will call this model the **secondary model**.

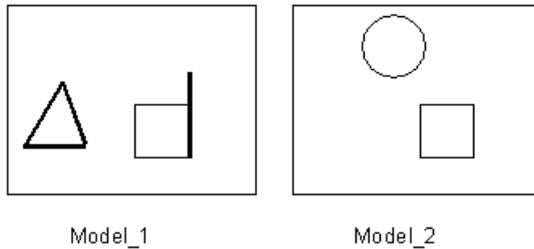
Surface Trimming Comparison - You can choose how to compare surfaces with trim boundaries and p-curves. There are three options:

Exact - This option finds surfaces with exactly the same pcurves and trim boundaries.

Boundaries - This option finds surfaces with exactly the same trim boundaries. Any p-curve not part of a trim boundary is ignored in the comparison.

None - This option treats two surfaces as the same if the untrimmed surfaces are the same - even if they have different trimmed regions.

OK - Compares the two models. All objects, which are in the primary model and not in the secondary model, are highlighted as selected in the primary model.



Objects that are in the secondary model but not in the primary model are not shown. To show these objects, simply carry out the above steps with the secondary model as the primary model and the primary model as the secondary.



Warning: Models are compared on the basis of positional information (XYZ coordinates) only. Differences, for example, in tangent direction, colour and level, are not picked up. This is done for reasons of speed. This command can be a useful tool to aid comparison of models, but you must use it with care and check manually for small differences that the command may not pick up.

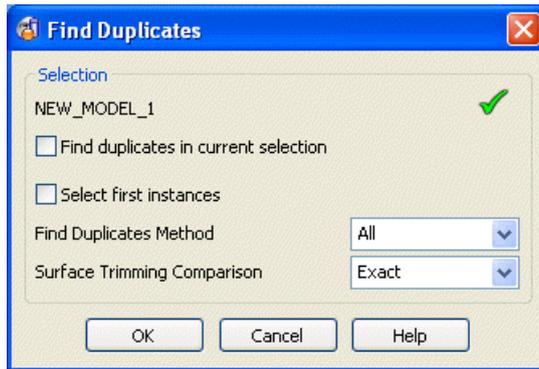
Finding duplicate objects in a model

Sometimes, you may have a copy of an object on top of itself. Use this command to highlight any duplicate objects in the model.

- 1 Click in the graphics window of the model in which you wish to find any duplicate objects. This makes the graphics window active.
- 2 Select objects if you want to find duplicates in the current selection.
- 3 Click **Model Fixing**  on the **General edits**  flyout.
- 4 Click  (model fixing toolbar) to display the Find Duplicates dialogue box. (see page 180)
- 5 Use the dialogue box to help you check if you have any duplicate objects.

Find Duplicates dialog

Use this dialog to find duplicate objects within a model.



Selection - This displays the name of the model to find duplicate objects in. By default, it is the active model. You can choose another open model by either clicking in its graphics window or selecting it from the **Window** menu to make it active.

Find duplicates in the current selection - If on, duplicates are found only in the objects which are currently selected. Otherwise, the whole model is checked.

Select first instances - If on, any instance of the object before the last is considered to be a duplicate. If off, any instance of the object after the first is considered to be a duplicated.

Find Duplicates Method - Duplicate surfaces are selected depending on the two options below.

All - This selects duplicate surfaces depending on the **Surface Trimming Comparison** option.

Untrimmed - If all duplicate surfaces are untrimmed, this option selects all the duplicates. If some of the duplicate surfaces are trimmed, it selects all of those duplicates which are untrimmed.

In some imported models, each surface is supplied untrimmed and trimmed. This command allows you to select all the untrimmed surfaces.

Surface Trimming Comparison - You can choose how to compare surfaces with trim boundaries and p-curves. There are three options:

Exact - This option finds surfaces with exactly the same pcurves and trim boundaries.

Boundaries - This option finds surfaces with exactly the same trim boundaries. Any p-curve not part of a trim boundary is ignored in the comparison.

None - This option treats two surfaces as the same if the untrimmed surfaces are the same - even if they have different trimmed regions.

OK- The dialog is removed from the screen. All duplicated objects are highlighted as selected and an information box is displayed showing how many items are duplicates.



For each set of duplicate objects, one is not selected. This allows you to operate on all duplicates within a model. For

*example, you can now delete them all by clicking **Delete*** .



Warning: *Duplicates are detected on the basis of positional information (XYZ coordinates) only. Objects may be selected as duplicates even if they differ in level, colour, tangent direction, or other non-positional ways.*

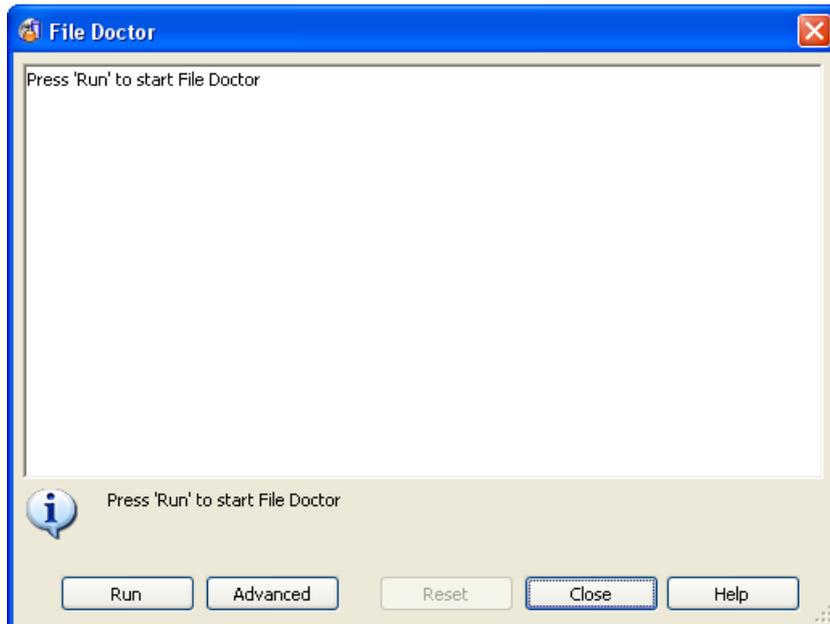
Tools > File doctor

This option provides tools to check and repair model files and the shared database. **File Doctor** is automatically opened if there have been errors in the session and you try to save an model using **Save As** or **Save**. This gives you the opportunity to correct errors and will help prevent you working with a model that is already corrupt.



File Doctor contains powerful tools. If it is used incorrectly, serious damage can be caused to your model or to your shared database. Consult Delcam Customer Support before using **File Doctor** on the shared database and always make a Full system backup before proceeding. Ensure you have back-up copies or duplicate models before using **File Doctor** on models.

- 1 From the **Tools** menu, select **File Doctor** to display the **File Doctor** dialog.



- 2 Use the File Doctor dialog (see page 185) or File Doctor (Advanced Mode) dialog (see page 187) to check the model.

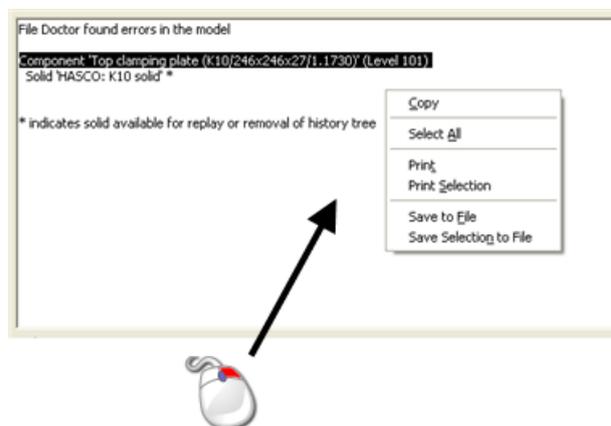


When a model is recovered a copy of the original is kept. The settings that control the action and the location of the copy can be changed by editing powershape.con. The default settings keep a copy of the original in the temp directory.

Using the File Doctor - step by step

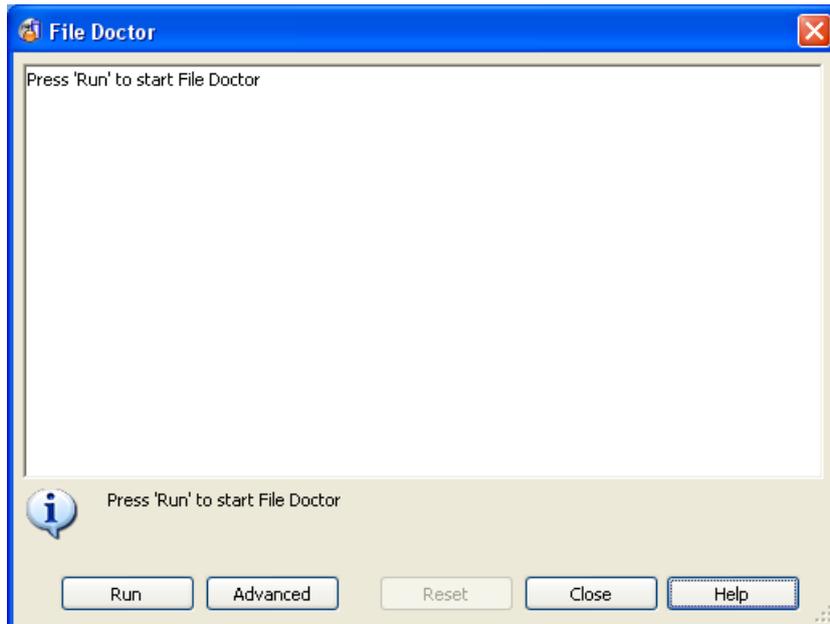


If you position the mouse in the information window of the File Doctor dialog and click the right mouse button, the popup menu that is displayed includes printing and saving options.

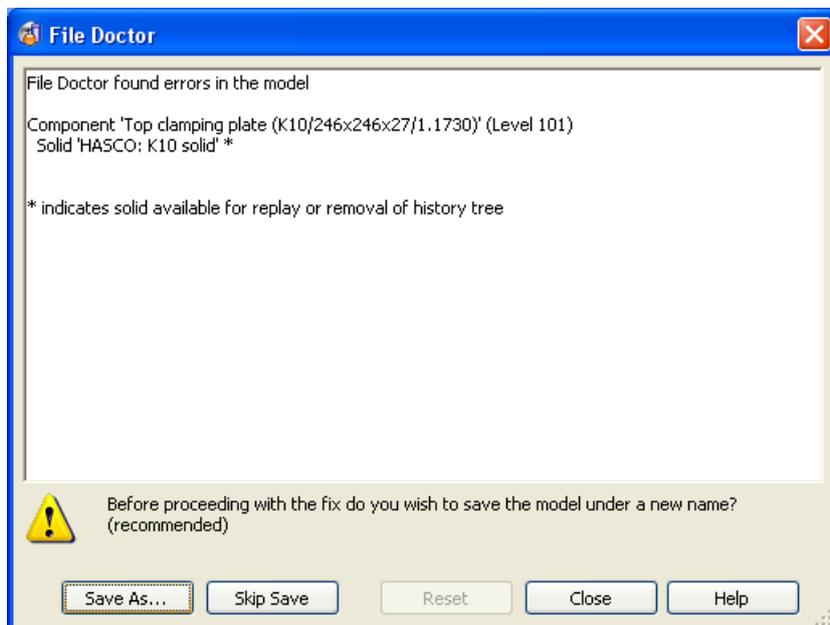


- 1 Ensure that the corrupt model is open. We recommend that this is the only model open.

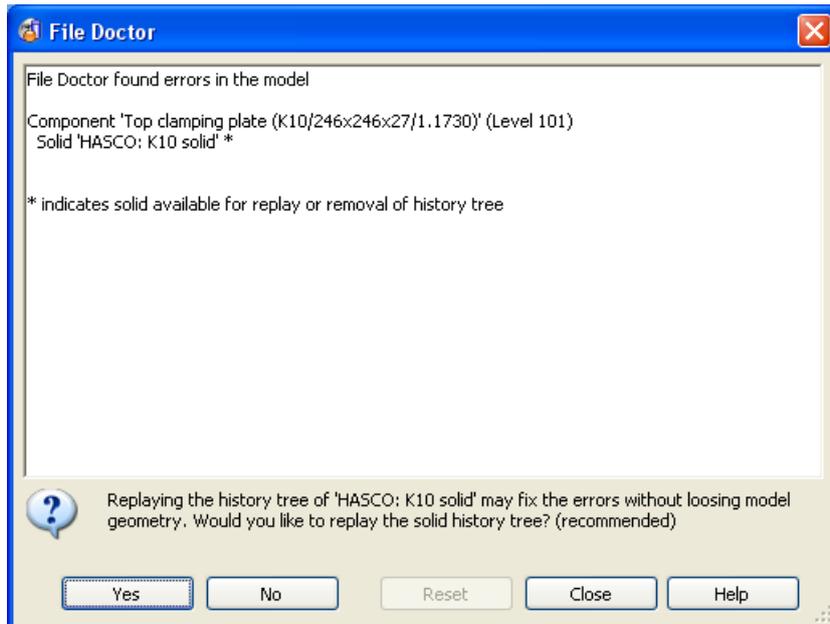
- 2 Select **Tools - File Doctor** to display the standard File Doctor dialog (see page 185).



- 3 Click **Run**. The File Doctor tries to identify the problem and uses the dialog to tell you what is wrong with the model.



- 4 Click **Save As**. This is the recommended selection so that you have a 'safe' copy.



- 5 The dialog that is displayed includes a suggestion of a fix that could be applied.

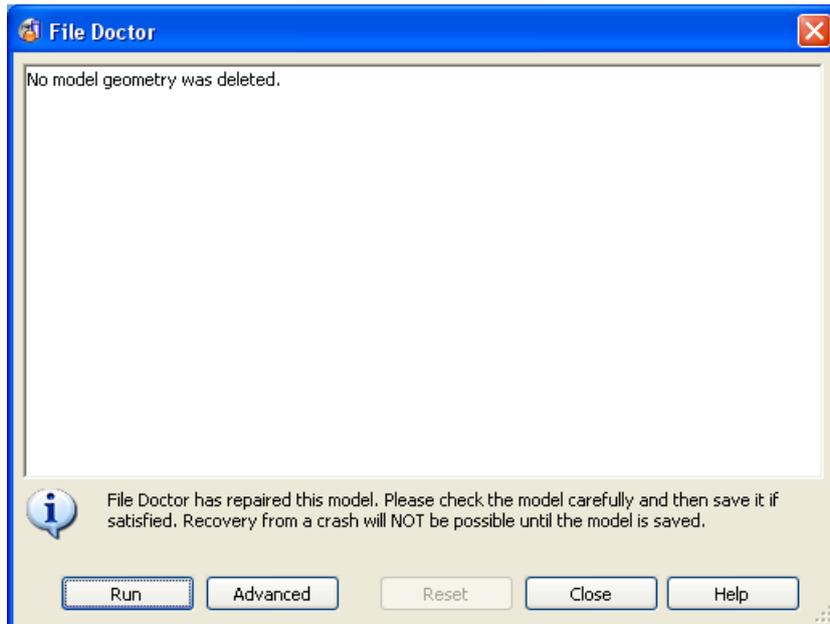
The type of fix offered by the File Doctor depends on the errors that are detected in the model. If possible, File Doctor will offer a fix that does not delete model geometry, such as replaying or removing the solid history tree.



Whilst replaying a solid history tree can fix some problems, it is possible that certain errors could cause PowerSHAPE to hang or crash whilst replaying. If this should happen,

- 1. Restart PowerSHAPE.*
- 2. Reload the model that was saved before the fix (step 4).*
- 3. Run File Doctor and when offered to replay the solid(s), choose No.*
- 4. You are now offered the choice of keeping the surface geometry (but losing the solid history tree) or keeping as much of the history tree as possible (but losing some features from the tree).*

- At the end of the process, File Doctor will tell you the geometry (if any) that had to be deleted to fix the model.



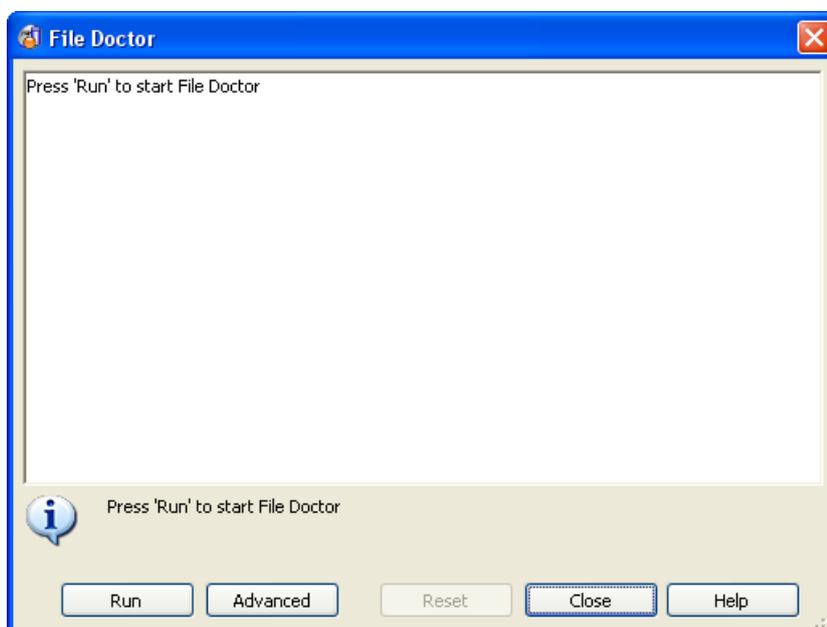
- Click **Close** to exit File Doctor.
- Check the model carefully and save it if you are satisfied with the fix.



Recovery from a crash is not possible until you have saved the fixed model.

File Doctor dialog

The standard version of the **File Doctor** dialog displays problems that are found with the model and suggested fixes.



The initial layout of buttons is shown. The buttons that are initially shown as **Run** and **Advanced** will change depending on the stage of the process.

The following buttons may be displayed at the bottom of the dialog:

Run - Start File Doctor check on the current model.

Advanced - Switch to advanced version of the File Doctor and display the File Doctor (Advanced Mode) dialog (see page 187). This mode gives full control over the checks and fixes that are run by the File Doctor. *Only experienced users should use this option.*

Save As - Save the model under a new name before proceeding with the File Doctor fixes.

Skip Save - Skip saving the model under a new name and move onto the fixing stage of the File Doctor.

Yes - Apply the possible fix.

No - Skip the fix.

Keep geometry - Fix a solid by maintaining surface geometry but removing the solid feature tree.

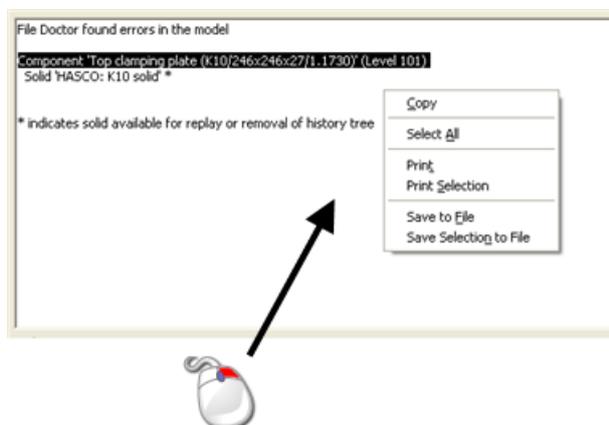
Keep history - Fix a solid by removing features with errors from the feature tree.

Reset - Reload the model that was saved before the File Doctor fix. This option is only available if the model was saved before the fix.

Close - Finish File Doctor.

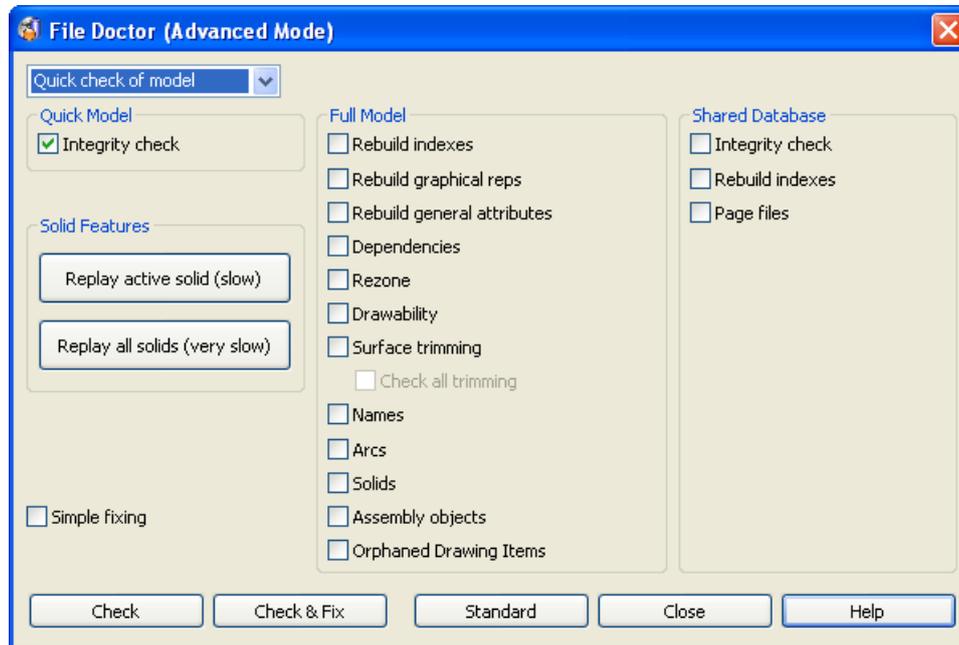


If you position the mouse in the information window of the File Doctor dialog and click the right mouse button, the popup menu that is displayed includes printing and saving options.



File Doctor (Advanced Mode) dialog

File doctor allows you to recover damaged models which would otherwise be unusable. If you would prefer to use the standard **File Doctor** dialog, type **Standard** in the command input box on the **Status** bar.



Quick Check of Model - Options from this menu automatically turn on the checks on the dialog. Use this menu to select one of the following:

Quick Check of Model - This selects only the **Integrity check** in the **Quick Model** section.

Full Check of Model - This selects the **Integrity check** and all the checks in the **Full Model** section.

Check Shared Database - This selects the **Integrity check** and all the checks in the **Shared Database** section.

User-defined - This allows you to select any check. It is also selected if you change the checks after selecting one of the three options above.

Quick Model - This section contains the following check:

Integrity check - This checks if any objects within the model refer to missing objects and explicit indexes.

If you select **Check**, an information window is displayed showing a list of errors. You will be alerted to any missing indexes.

If you select **Check & Fix**, the objects containing errors are deleted from the model. File Doctor displays an information window, showing the list of faults (if any) within the model that have been cured and any missing indexes that have been created. However, under some circumstances (such as where many objects refers to a missing object), automatic fixing can cause considerable damage. We therefore recommend that **Check & Fix** is used only when you have made a backup copy of the model.

Replay active solid (slow) and **Replay all solids (very slow)** allow you replay the solid feature trees.

Full Model - This section contains the following checks:

- **Rebuild indexes** - All models contain index files. This option clears the model indexes and builds new ones, but only when **Check & Fix** is clicked. There is no **Check** option for indexes.
- **Rebuild Graphical Reprs** - ensures that each object geometry definition has a matching and valid graphical representation. The option is only available under the **Check & Fix** option. To explain what it does:
Each object in PartMaker Modeling (such as lines and arcs) has two main components. These are the **geometry definition** and the **graphical representation**. The latter describes how the object is currently displayed in the various views on the screen. It can sometimes become corrupted (e.g. missing or incorrect) while the geometry definition is correct. Typical problems are that the object displays incorrectly or not at all or is not selectable.
- **Rebuild General Attributes** - If you choose **Check**, you are told if there are any errors in the styles of the objects. If you select **Check & Fix**, the styles with errors are replaced with a solid, foreground colour and width 0.3 style.
- **Dependencies** - If you choose **Check**, you are told if there are any errors in surface and vector dependencies. Dependencies are relationships between geometric objects. For example, a solid block or cylinder has a relationship between the main surface and the two capping surfaces. Surface primitives have dependencies between the surface and the set of parameters (for example, length and diameter) that define them.

If you select **Check & Fix**, File Doctor deletes any bad dependencies.



*We recommend you first run **Rebuild Indexes** from this section as database table indexes are often corrupt when surface dependencies are corrupt.*

With **Check & Fix**, editing dependencies of some objects may alter the dependencies of other objects. We advise you to run **Check & Fix** repeatedly until no further dependency errors are reported.

- **Rezone** - The model is divided into zones. When an object is picked, only the zones beneath the cursor are searched, which considerably speeds-up the picking operation. If an object becomes incorrectly zoned (as can happen occasionally), it displays correctly, but cannot be picked. This option regenerates these zones, curing most of the problems associated with unpickable objects. It is only available with **Check & Fix**.
- **Check Drawability** - PartMaker Modeling should be able to draw (on the screen) every object within a model. This option checks each object and deletes any that cannot be drawn. This option also checks for items that are undisplayable. If the File Doctor is run in **Check & Fix** mode, the undisplayable items are deleted.



Drawability in this context has no connection with "drawing" as used in Drafting.

- **Check Surface Trimming** - If you choose **Check**, you are told if there are any trimming problems on the surfaces. If you select **Check & Fix**, then PartMaker Modeling attempts to repair the surfaces. If some boundaries can't be repaired, they are deleted and you are told which ones are deleted.

Select the **Check all trimming option** on the **Advanced File Doctor** dialog to ignore any previous surface trimming checks and re-check and fix every surface.
- **Check Names** - If you choose **Check**, you are told if objects have duplicate or invalid names. If you select **Check & Fix**, PartMaker Modeling renames objects with duplicate or invalid names.
- **Check Arcs** - A valid arc has a start, end, mid and centre point and each point has a different ID and is owned by the arc. If you choose **Check**, you are told if any arc in the model is invalid. If you select **Check & Fix**, PartMaker Modeling deletes invalid arcs.
- **Check Solids** - This checks solids and their feature trees for corruption. If you select **Check & Fix**, the corruption is repaired. If the corruption is caused by one of the features in the solid tree, PartMaker Modeling tries to fix this feature. If this is not possible, the feature tree for the solid is deleted.
- **Check Assembly Objects** - This checks components and component definitions for corruption. If you select **Check & Fix**, the corruption is repaired. If this is not possible, data that cannot be repaired will be deleted.

- **Check Orphaned Drawing Items** - Some model items that are displayed in a drawing view but not displayed in model windows, such as centrelines, can be orphaned if the drawing view is deleted. This option checks for and removes any orphaned drawing items.

Shared Database - This section contains the following checks:

- **Integrity check** - This does a full integrity check/fix on the shared database.
- **Rebuild indexes** - This deletes and rebuilds the shared database indexes. These can sometimes become corrupted. For example, a hardware or software problem (such as insufficient disk space) occurring while creating, saving or deleting a model can cause damage. Affected models disappear from the menu list (as if deleted) or can appear twice. Regenerating usually solves these problems. The option is only available with **Check & Fix**.

- **Page Files** - rebuilds the page files, ensuring all are correct. It is available only with **Check & Fix**. To explain the purpose of the files:

The database tables keep a temporary copy of their current state in page files, from which data is paged in and out from core memory. These files can be damaged by a hardware or software problem occurring during a database update. The damage will affect all Delcam programs which use that database.



*We recommend that all users stop working with Delcam products before any option from the **Shared Database** section is used.*

Simple Fixing - If *ON*, simple fixes will be carried out when running File Doctor checks. Any corrupted data will be deleted. If *OFF*, the File Doctor will attempt to work round the corrupted data. For example, it will attempt to remove a single corrupt feature from the solid tree instead of removing the entire tree. Using **Simple Fixing** is more reliable, but it may delete a lot more of the model data.

Check - This checks and displays a report of any faults found by the **File Doctor** routine(s).



*The **Check** only option is more useful to Delcam Technical Support staff than to regular users. You may find the errors reported rather unintelligible.*

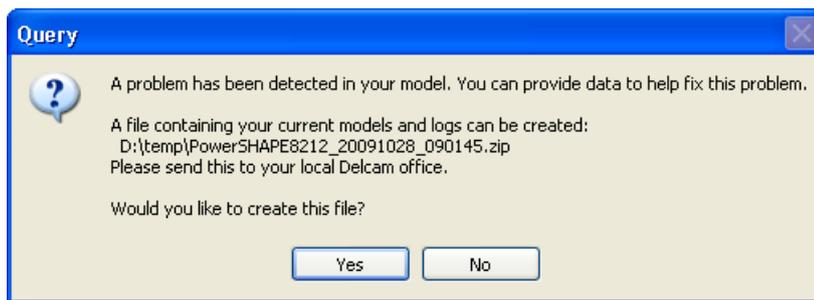
Check & Fix - This checks and repairs the data within the model.



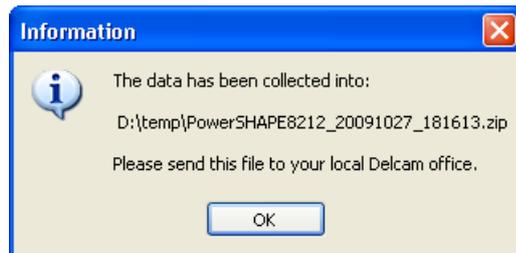
WARNING: All errors found by **File Doctor** are potentially serious and could cause the program to crash or malfunction (perhaps causing further corruption). You should always remove the errors with **Check & Fix** before continuing. In many cases, it works by deleting faulty objects. If this deletes large portions of your model, contact Delcam Technical Support, who may be able to retrieve some of the lost data. However, do **NOT** continue working on a corrupt model or database.

Submitting model information

When a database error is detected a dialog is displayed with a request to submit your model and macros to Delcam.



- 1 Click **Yes** to create the zip file. It is recommended that you create as requested.



- 2 Click **OK** to complete the operation.
- 3 Send the file to your local Delcam supplier.



Note that the date and the time are included in the filename.

Importing and Exporting a model

Use the following sections to import and export data:

Exporting data from a model (see page 192)

Importing data into a model (see page 197)

Exporting data from a model

You can export models to formats other than PartMaker Modeling.

- 1 Select the model or the specific objects you want to export.
- 2 Select **File** > **Export** from the menu to display the first page of the **Export** Wizard.
- 3 Use the **Export** Wizard to select the target system and settings for exporting files. Alternatively, click **Finish** to export the data using the following defaults:
 - export selected items, or the entire model if nothing is selected.
 - if a workplane is active, export the geometry to active, otherwise, export to World.
 - if a drawing window is active, export the converted drawings.



*The default for .dgc is to export to World. You can use the **Export** Wizard to export .dgc to the active workplane.*

For advanced modelling concepts, see:

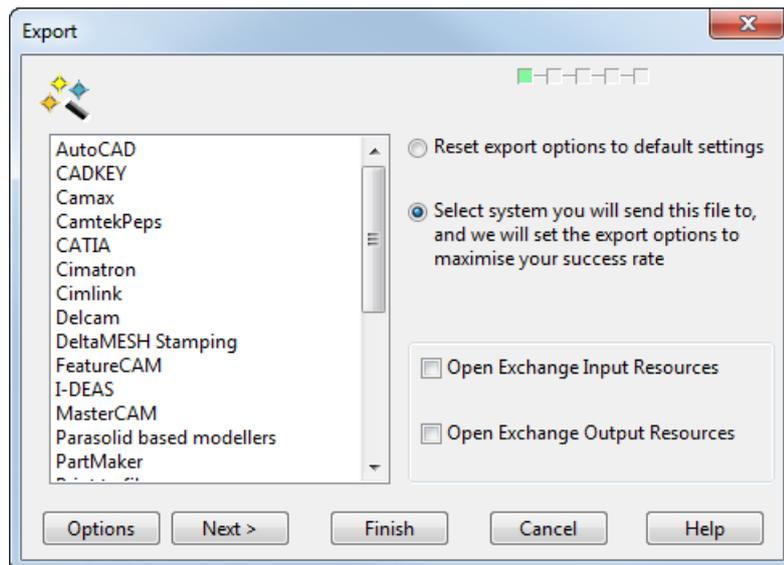
- Using Delcam Exchange to translate data
- Exporting data without a licence
- Exporting materials

Using the Export Wizard

Use the **Export** Wizard to:

- export selected model items, visible items or all items.
- export relative to an active workplane or World.
- export drawings.

- 1 Select **File > Export** to display the **Export Wizard**.



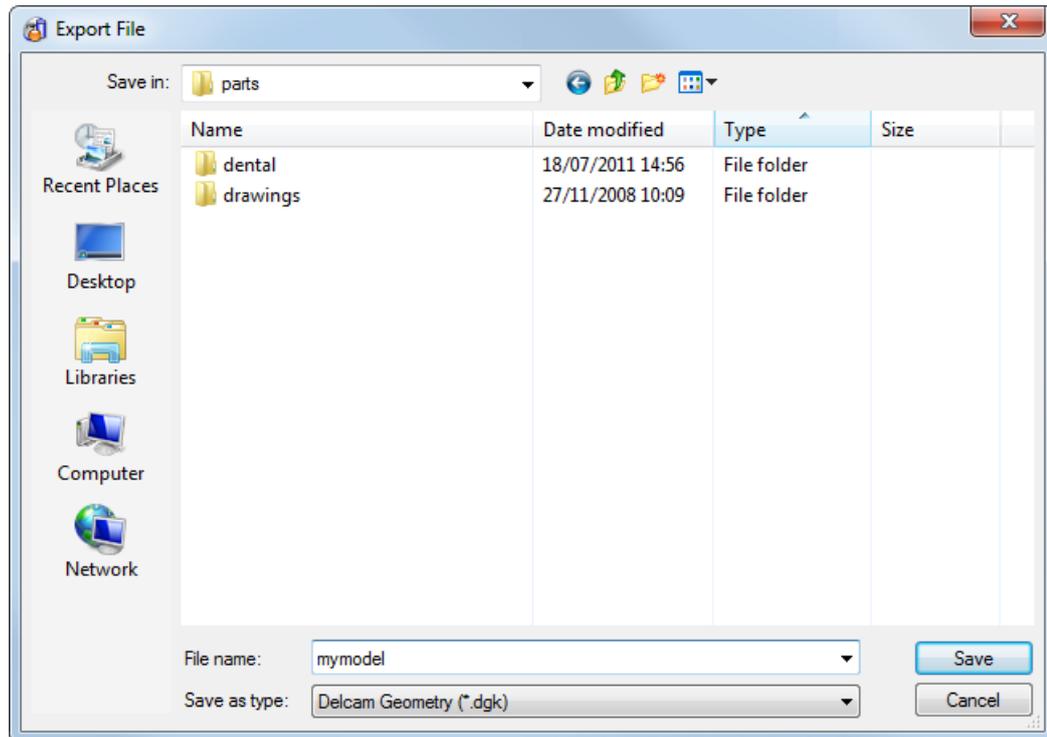
- 2 Select the target system for export from the list.
- 3 Click **Next** to display the **Export File** dialog.

The file name of your model is automatically displayed as the filename. Change the **File name** as required and select the file type from the **Save as type** list.

- 4 Click **Save**. The subsequent pages that are displayed depends on your export:
 - if you are exporting a model that includes at least one drawing, page 2 is displayed (see page 195).
 - if you are exporting a model that does not include drawings, page 3 is displayed (see page 195).
 - if your model includes workplanes, page 4 is displayed (see page 196).
 - if you are exporting a model that includes multiple drawings, page 5 is displayed (see page 197).
- 5 Select the appropriate options on the export pages that are displayed.
- 6 Click **Finish** to complete the export.

Export File dialog

Use this dialog to export files in the format selected on the Export dialog.



- 1 Select the correct **Save in** folder. If necessary, use the following to locate the required folder:
 - Click  to go up one level in the folder structure.
 - Click  to create a new folder.
 - Click  to display a menu containing options on how to display the files in the dialog.
- 2 Enter the **File name**. If you enter a file name with no extension, the **Save as type** extension is automatically added.
If you enter a filename with a valid export extension, the **Save as type** extension type overrides the type entered in the file's name.
- 3 Select the required export file type from the **Save as type** list.
- 4 Click one of the following:
 - **Save** to export data as specified.
 - **Cancel** to remove the dialog from the screen without exporting any files.



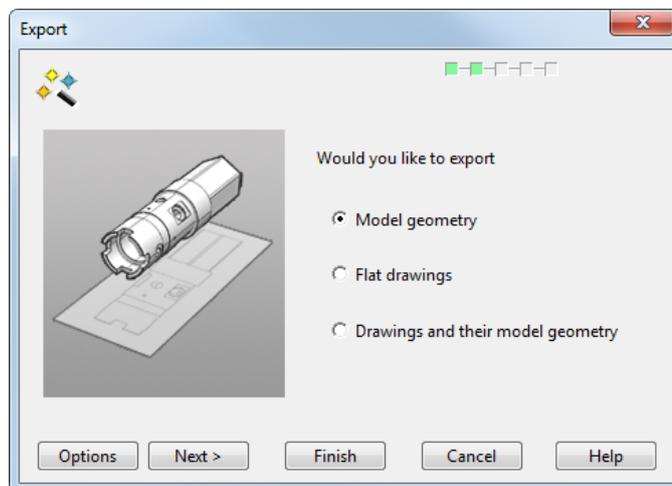
You can use either the PartMaker Modeling internal translators or Delcam Exchange to export data.

You cannot export badly trimmed surfaces. PartMaker Modeling tells you if badly trimmed surfaces are detected on export and gives you two options:

- Export the selection with the offending surfaces untrimmed.
- Abandon the export. This allows you to fix the surfaces using the trim region editing tools.

Page 2 - Exporting a model that includes drawings

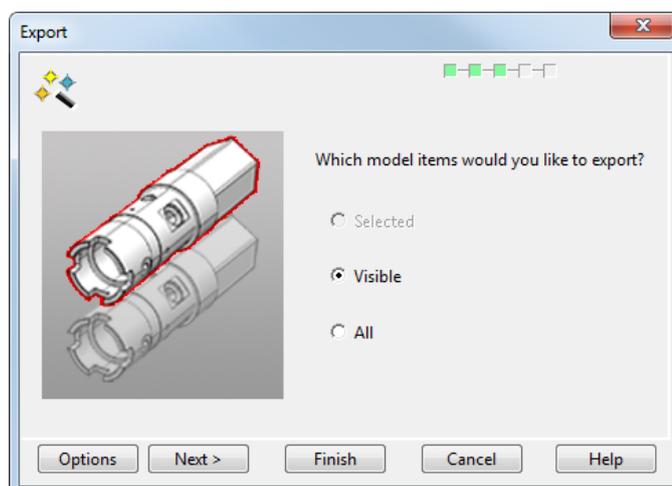
Use this page to export a model that includes drawings.



- 1 Select one of the options to specify the items to be exported:
 - model geometry only
 - drawings
 - drawings and associated model geometry
- 2 Click **Next** or **Finish**.

Page 3 - Exporting a model

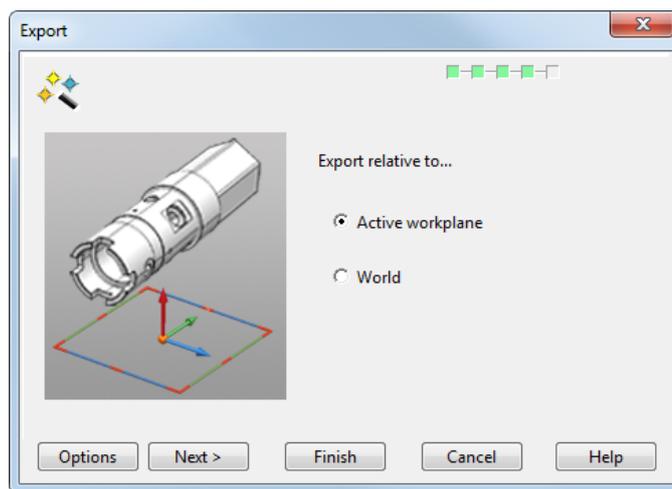
Use this page to specify the model items that you want to export.



- 1 Select one of the options to specify the items to be exported:
 - If you pre-selected some items, then **Selected** is the default option.
 - **Visible** exports items that are visible on the screen; if an item is blanked it is not exported. This option is useful for exporting the workplanes with a model.
 - **All** is the default option and is automatically selected if there are no pre-selected items.
- 2 Click **Next** or **Finish**.

Page 4 - Exporting workplanes

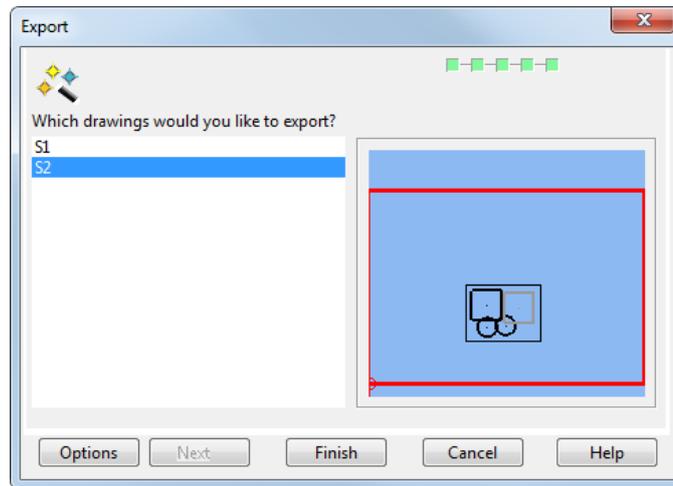
Use this page to specify the workplane to be used at export.



- 1 Select the option to be used when exporting:
 - Export relative to **Active workplane**.
 - Export relative to **World**.
- 2 Click **Next** or **Finish**.

Page 5 - Selecting a drawing to export

Use this page to select the drawings to be exported. This page is displayed only if there are multiple drawings associated with the model.



- 1 Select the drawings to be exported.
- 2 Click **Finish**.

Importing data into a model

When you import a model, it is automatically opened as a new model. However if you import any other file, it is opened in the current model window.



When you import a file, the model is zoomed to full, so the geometry is visible. This is particularly useful if the items are not close to the origin.

You can use one of the following methods to import data:

- Select **File > Import**.
- Click Import (see page 197) on the **Main** toolbar.
- Use the Advanced Import options to set up the data.
- Drag and drop files into the graphics area to import them.

For advanced modelling concepts, see:

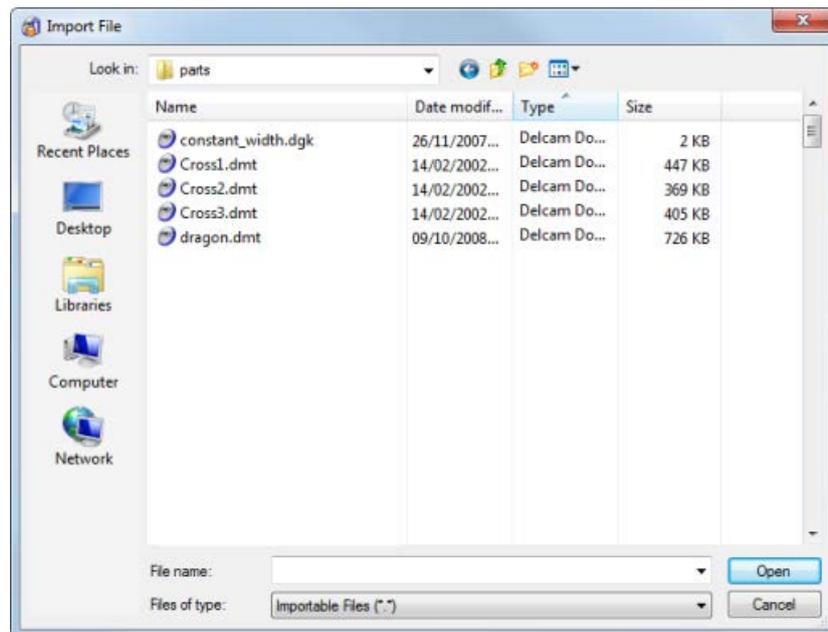
- Advanced Import options
- Importing data without a licence
- Importing Materials

Using the Import option

- 1 Select **File > Import**.

This is the same as clicking **Import**  on the **Main** toolbar.

- 2 Browse to the folder containing the file you want to import.



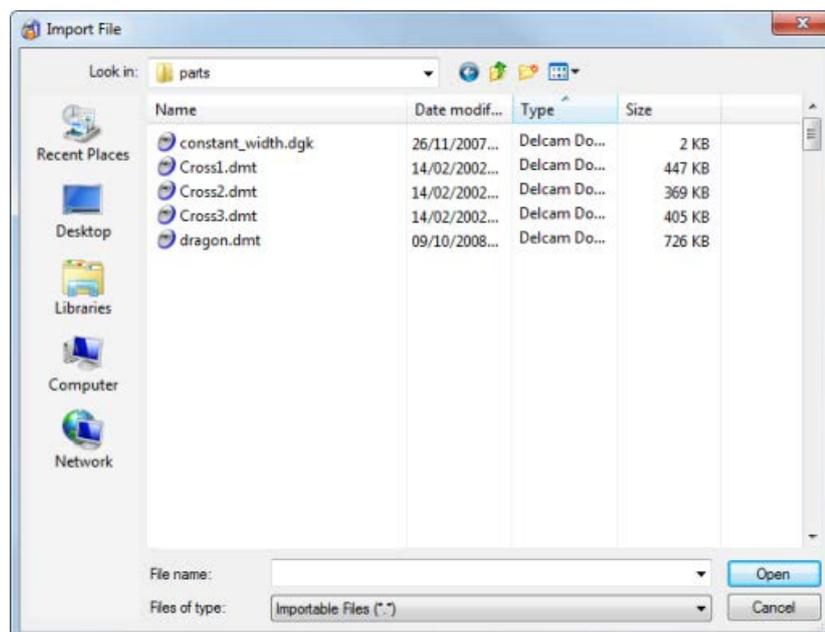
- 3 Click **Open**.

The file is imported and opened.

For further details, see *Importing data into a model* (see page 197).

Import dialog

Use this dialog to select files to import.



- 1 Select the correct **Look in** folder. If necessary, use the following to locate the required folder:

- Click  to go up one level in the folder structure.
- Click  to create a new folder.
- Click  to display a menu containing options on how to display the files in the dialog.

2 Enter the **File name**.

3 Select the required import file type from the **Files of type** list.

This displays the filter pattern which filter the file names of the current directory. By default, the pattern is *.* which displays all files. This selector lists the default filters for the file types that can be imported. You can select a filter from the list. When a filter is selected, files satisfying that filter are displayed.



***Files of type** is used only to help find files more easily. It does not determine the type of file to be imported. When a file is imported, the program looks at the contents of the file to determine its type and not its name. For example, if **Files of type** is set to *.pic, you can still import an IGES file by typing in the name in the **File name** text box. If an IGES file is called file.pic, this filter will help find it, but the file will still be imported as an IGES file.*

When importing IGES files, the program uses its own default tolerance values. You can change the defaults on the **Options** dialog before importing. The new values affect only the current model. Remember, large tolerances produce poor quality surfaces and small tolerances make the application run slowly. You may need to experiment to find the best balance for your models.

4 Click one of the following:

- **Open** to import data as specified.

When you import certain files, a log file is generated containing details about the import. This log file is added to either the directory containing the imported file or the temp directory.

If you import a model that has the same name as one already in the model database, its name is appended with [n] where n is an integer. The integer starts at 1 and increases as other models with the same name are imported. When such a model is imported, you need to open it using the Open Model dialog. (see page 164)

Generally, models are automatically opened in the program when they are imported.

- **Cancel** to remove the dialog from the screen without importing any files.



If you import files into a new model, the name of the last imported file (minus its extension) is automatically displayed as the **Model Name** in the **Save Model As** dialog when you save the model. For example, if the name of the last file imported is *handle.igs*, the **Model Name** is *handle* in the **Save Model As** dialog. This name is also displayed in the banner of the model's window, next to the new model's name. For example, NEW_MODEL_1(handle).



You can use either PartMaker Modeling's internal translators or Delcam Exchange to import data. For further details, see *Using Delcam Exchange to translate data*.

The coordinate system

The coordinate system defines the position of every object within the global workspace. The coordinate system contains the three axes **X**, **Y**, and **Z** which meet in the centre of the workspace, at a position called the origin.

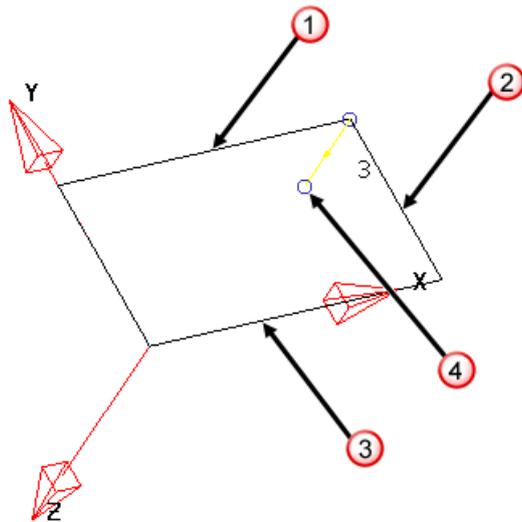
All models are created within the global workspace. Every position in a model is defined as three coordinates. For example, a position of 9,5,3 means **9** along the **X** axis, **5** along the **Y** axis and **3** along the **Z** axis. These coordinates are shown in the image below:

① **X** = 9

② **Y** = 5

③ **Z** = 3

④ Position 9,5,3

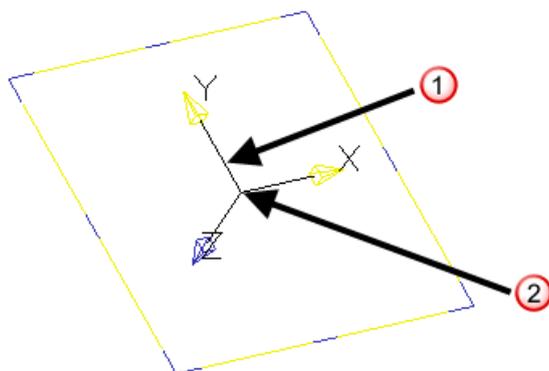


You create a model while working entirely within the global workspace, selecting the default drawing plane (also known as the principal plane) and entering coordinate values when necessary.

However, models can contain many angled faces and it is often easier to specify coordinates relative to a face rather than to the model as a whole. You can do this using workplanes (see page 201), shown in the image below:

① Dimensional axes **X, Y, Z**

② Origin



For further details, see *Creating a workplane in the Wireframe modelling manual*.

Workplanes and the coordinate system

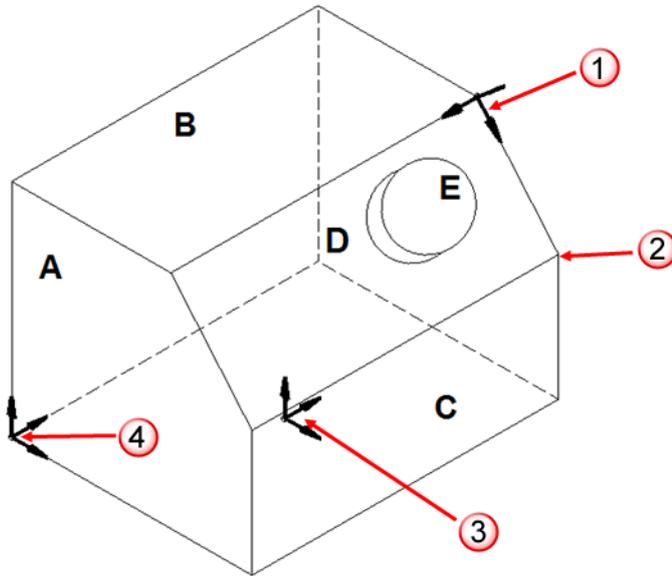
A workplane is a graphical object defining a *local coordinate system*.

You can:

- have as many workplanes as you like.

- place workplanes anywhere in the model.
- rotate the axes to align conveniently with features of the model.

The image below shows example of a *local coordinate system*:



① Local coordinate system (Workplane). Convenient for:

- Face **D**.
- Angled feature **E**.

② Local coordinates 20,0,0. Global coordinates 437,62.8,28.7

③ Global workspace coordinate system for faces A, B, C.

④ Local coordinate system (Workplane). Ideal for faces A,B,C

Only one workplane can be active at a time, which is known as the *active workplane*. It is shown in red in the default colour scheme. Inactive workplanes are grey.

Any specified coordinates apply to this workplane, not to the global workspace. PartMaker Modeling converts the local values to global values automatically and invisibly.

The three axes of a workspace listed below define three planes. These are known as the principal planes (see page 203).

- the XY plane lies through the X and Y axes
- the ZX plane lies through the X and Z axes
- the YZ plane lies through the Y and Z axes

Principal planes and the coordinate system

The **Principal plane** buttons  on the Status bar (see page 125) define which plane is the principal plane of the workspace. You can see this graphically as the axis of the rotation tracker updates in the modelling window. When clicking the mouse to enter positions in an empty space, the positions lie on the principal plane.



Each button displays the label of the axis which is normal to the principal plane.

Entering positions

To create an object, you need to specify its position in the workspace. Entering positions is one of the most common and important operations use the following methods for entering positions:

Entering positions using the cursor (see page 203)

Entering positions using the Intelligent Cursor (see page 153)

Entering positions using the status bar (see page 205)

Entering positions using dialogs (see page 207)

You can also create point objects to mark locations in your model.

Entering positions using the cursor

As the cursor moves across the principal plane (see page 203) (for example XY of the active workplane or global workspace), its coordinates with respect to the active workplane are shown in the status bar (see page 125). To enter a new position, move the cursor and click.

If the cursor is clicked within a certain radius of an existing object, the cursor snaps to the closest *key point* that exists on the object or snaps to a point of *intersection* between objects. This radius is known as the *hit radius*.

Entering positions using the Intelligent Cursor

The Intelligent Cursor shows additional information to help you to define the position you want when entering positions.

When you move the cursor over an object, construction labels (see page 153) are displayed by the Intelligent Cursor. These construction labels:

- are shown and hidden as you move the cursor over them.
- indicate what happens if you click while the label is displayed.

- make snapping with the cursor more accurate. The labels display when key points are within snap range of key positions in the model. You can click when a label (see page 153) is displayed to snap to the cursor at the labelled position.



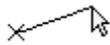
The Intelligent Cursor (see page 148) is enabled by default.

Construction lines

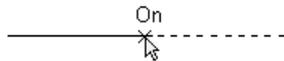
When you hover the Intelligent Cursor over an important key point (one that displays any of the construction labels (see page 153)), construction lines are created from that key point. You can now drag the cursor along a construction line and see the appropriate labels display. Clicking causes the position to snap to that point.

The following example draws a box:

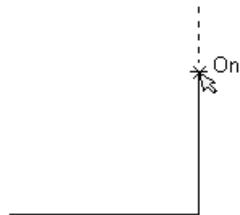
- 1 On the Main toolbar, select the **Line**  button.
- 2 On the Line toolbar, select the **Continuous**  button.
- 3 Click to start a line.



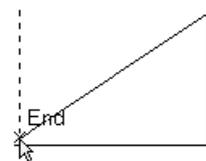
Drag along a construction line and click.



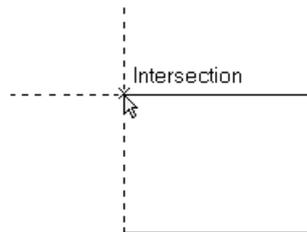
- 4 Drag along a 90° construction line and click. The line snaps to give a square intersection.



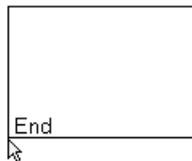
- 5 Drag the cursor down to the original start point and hover. The cursor displays **End** and the construction lines display.



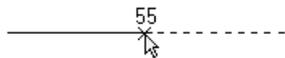
- 6 Drag along the vertical construction line until **Intersection** is shown.



- 7 Snap to the start and finish with one more click when **End** is displayed.



If the Intelligent Cursor Gridding is enabled, the label **On** in the above diagrams is replaced by a distance from the last point selected. For example:



This provides a convenient way to enter accurate dimensions.



*To disable construction lines temporarily, press and hold the **Shift** key.*

Entering positions using the status bar

You can enter Cartesian (**see page 205**) and polar (**see page 206**) positions in the **Data entry box** on the Status Bar (see page 125).

You can also type the coordinates in the graphics window and they are entered automatically into the **Data entry box** on the status bar. When you press the **Enter** key, coordinates are entered into PartMaker Modeling.

For more complex constructions click the **Position**  button on the Status Bar, to display the **Position** dialog (see page 208).

Entering Cartesian positions

The syntax for entering Cartesian positions (using the status bar) is as follows:

[workspace] X [Y [Z]] [units]

Square brackets [] show elements that are optional.

workspace specifies the origin from which to measure the coordinates.

You can use the following options:

- **world** = world workspace. You can abbreviate *world* to **w**.
- **absolute** or **@** = absolute workspace. This uses the origin of the current workspace. You can abbreviate *absolute* to **abs**.
- **relative** = relative workspace. This uses the last point entered as the origin. You can abbreviate *relative* to **re**.

X [Y [Z]] - These are the X Y and Z coordinates and are entered in this order by default. If the second and third values are not given, they are treated as zeros.

The order of the X Y and Z coordinates is determined by the **Local point input** option on the **Workplane** page of the **Options** dialog. For further details, see Workplane options (*Menus and Toolbars*).

units - You can enter the type of units, for example, **mm** or **inches**. Default units are used if none are specified.

Examples of Cartesian coordinates are given below.

20 30 40

w 15

@ 25 60.8 mm

abs 50 mm

re -30.7 0 90

You can also enter polar positions. For further details, see Entering polar positions (see page 206).

Entering polar positions

The syntax for entering polar positions (using the status bar) is as follows:

> angle [angle_units] length [length_units]

Square brackets [] show elements which are optional.

> indicates that polar coordinates are being entered.

angle is the value of the angle.

angle_units is the unit type, for example, *degrees*.

length is the value of the length.

length_units is the unit type, for example, *mm*.

Examples of polar coordinates are given below.

> 45 30

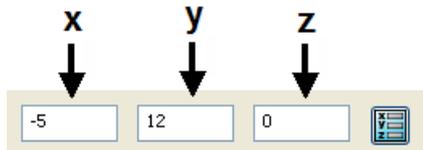
> 45 degrees 30 mm

> 30 2.5 inches

You can also enter Cartesian positions (see page 205).

Entering positions using dialogs

Where a dialog expects a position to be entered, separate text boxes for the X, Y, and Z axes are provided. Click the **Position**  button to display the **Position** dialog (see page 208) for more complex point entry tools.



To enter a position as three coordinate values directly into a dialog:

- 1 Click in the **X** axis box to select it and enter the value
- 2 Press the **tab** key to select the **Y** axis box, or click in the box.
- 3 Enter the value and repeat to enter the value for the **Z** axis box.
- 4 Click **OK** on the dialog to store the values.

Position dialog overview

When it is not sufficient to enter positions using the cursor, the status bar, or dialogs, use the tools on the Position dialog (see page 208) to assist with more complex constructions. For example, you may want to specify a point at a particular proportion along an object.

the values for the X, Y, Z axes

Display the Position dialog (see page 208) using one of the following methods:

- Click **Position**  on the status bar (see page 125)
- Click  on a dialog.

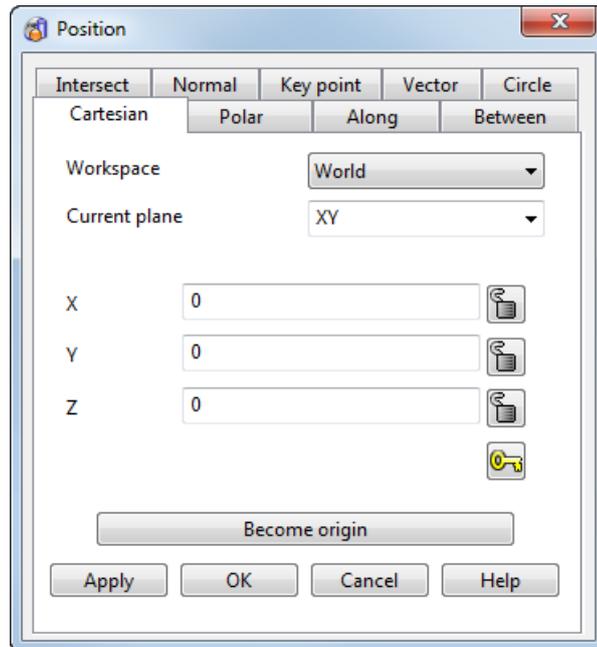
For further details, see Entering positions using dialogs (see page 207).



*The **Position**  button is active only when positions are required by the program. For example, when in object creation mode or when editing objects.*

Position dialog

Use the tools on the **Position** dialog to specify positions and constructions (see page 203).



Click a tab on the dialog to display the relevant page of position entry tools:

Cartesian (see page 208)

Polar (see page 210)

Along (see page 212)

Between (see page 214)

Intersect (see page 215)

Normal (see page 217)

KeyPoint (see page 219)

Vector (see page 221)

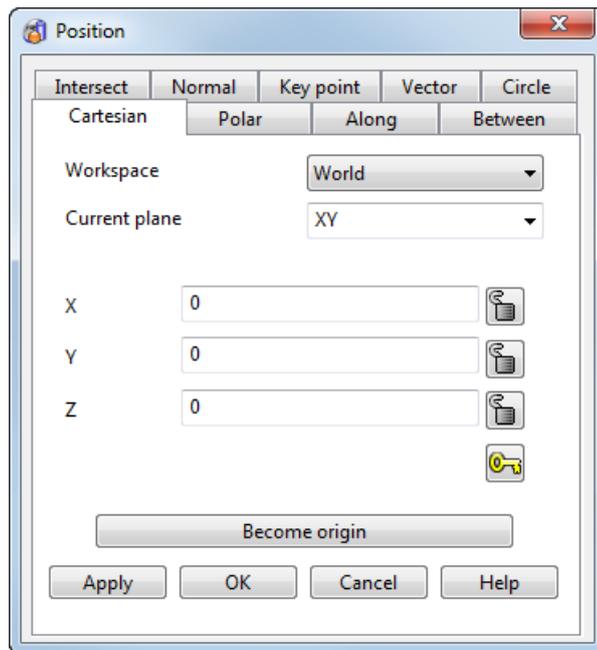
Circle

Cartesian tab

Use the **Cartesian** tab of the **Position** dialog (see page 208) to enter positions in the separate entry boxes provided for each axis. You can lock each individually by clicking .

Usually you can enter a value in a box without affecting the other values. However if you have locked some components on the **Polar** tab (see page 210), then the values of the other fields may change to comply with the constraints created by the locks.

If you click a position in the graphics window, its Cartesian coordinates are displayed in the dialog.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Current Plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

 **Locked** and  **Unlocked** — Click to toggle between **Locked** and **Unlocked**. Locking a value ensures that it remains fixed regardless of what happens to other components. The lock remains ON until you click again to unlock it. When an editing box is affected by a lock, it is dimmed to show that it is unavailable.

 **Unlock all** — Click this button to unlock all of the values above.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

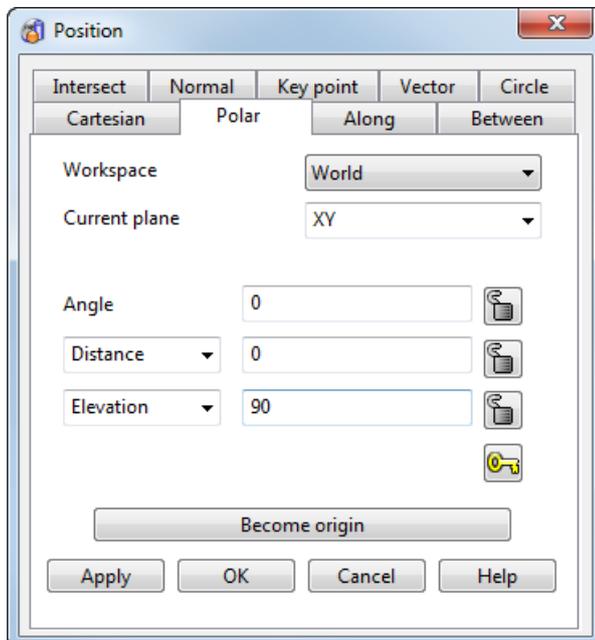
OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Polar tab

The **Polar** tab of the **Position** dialog (see page 208) is affected by the **Current plane** setting, which determines how the coordinates are calculated from the **Angle** and **Elevation/Height** fields.

Click a position in the graphics window to enter **Polar** coordinates into the dialog.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Current Plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

Angle — Enter the polar angle from the origin in the currently selected plane in the selected workspace.

Distance — Select one of the following options from the drop-down list:

- **Distance** — Enter the distance from the workplane origin.
- **X, Y, or Z** — Enter a distance on the axis on the current active plane.

Elevation — Select one of the following options from the drop-down list:

- **Elevation** — Enter the angle of elevation above the active plane from the origin of the workspace.
- **Height** — Enter the distance perpendicular to the selected plane. For example, the **Z** component when the **XY** plane is selected.

 **Locked** and  **Unlocked** — Click to toggle between **Locked** and **Unlocked**. Locking a value ensures that it remains fixed regardless of what happens to other components. The lock remains ON until you click again to unlock it. When an editing box is affected by a lock, it is dimmed to show that it is unavailable.

 **Unlock all** — Click this button to unlock all of the values above.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

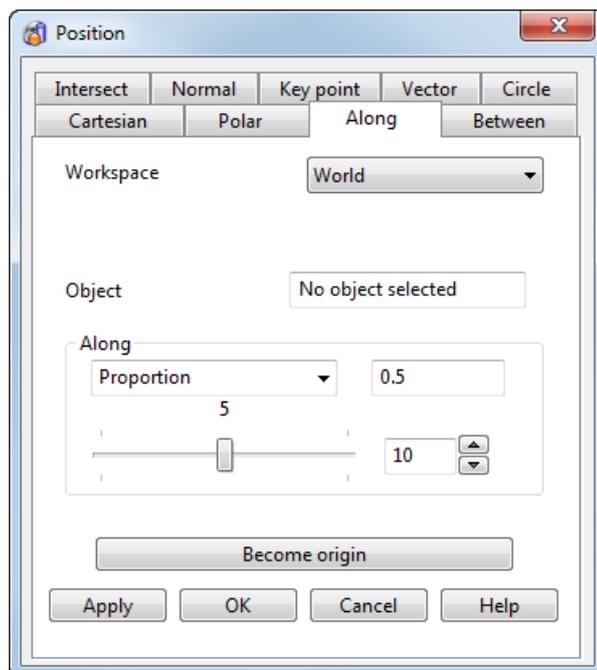
Cancel — Aborts the position entry.

Along tab

Use the **Along** tab of the **Position** dialog (see page 208) to create a position at a chosen parameter along a selected object.



*You can select composite curves when you are using the **Along** tab of the **Position** dialog:*



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Object — Select an object using the mouse. When an object is selected, its name is displayed in the text box.

Select one of the following options and enter a value, or use the slider:

Proportion — Enter a measurement for the proportional distance from the start of the object. For example, enter **0.5** for half way along or enter **0.25** for a quarter (or 25%) of the way along from the start of the object.

Distance — Enter a measurement from the start of the object.

The **slider** indicates the portion from the start of the object. By default, the slider has 10 divisions. You can change the number of graduations on the slider by using  . Use the arrows to select a value or type a value into the box.

 **Locked** and  **Unlocked** — Click to toggle between **Locked** and **Unlocked**. Locking a value ensures that it remains fixed regardless of what happens to other components. The lock remains ON until you click again to unlock it. When an editing box is affected by a lock, it is dimmed to show that it is unavailable.

 **Unlock all** — Click this button to unlock all of the values above.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

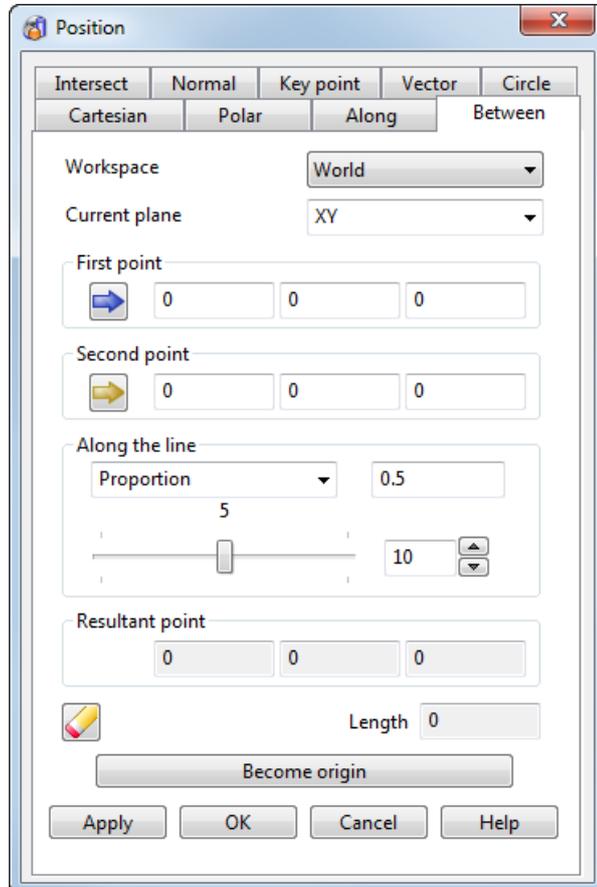
Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Between tab

Use the **Between** tab of the **Position** dialog (see page 208) in the same way as the **Along** tab (see page 212), but instead of using points at the ends of an object, select the start and end points which need not be on the same object. This allows accurate placement of a position between two connected or unconnected positions.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Current Plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

Click two positions in the model to enter the values for the following options:

First point — This displays the coordinates of the first click.

Second point — This displays the coordinates of the second click.

Select one of the following options and enter a value, or use the slider:

Proportion — Enter a measurement for the proportional distance from the start of the object. For example, enter **0.5** for half way along or enter **0.25** for a quarter (or 25%) of the way along from the start of the object.

Distance — Enter a measurement from the start of the object.

The **slider** indicates the portion from the start of the object. By default, the slider has 10 divisions. You can change the number

of graduations on the slider by using  . Use the arrows to select a value or type a value into the box.

Resultant point — This displays the coordinates of the *between* value as defined by the dialog.



Reset — Click this button to reset the values on the page.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

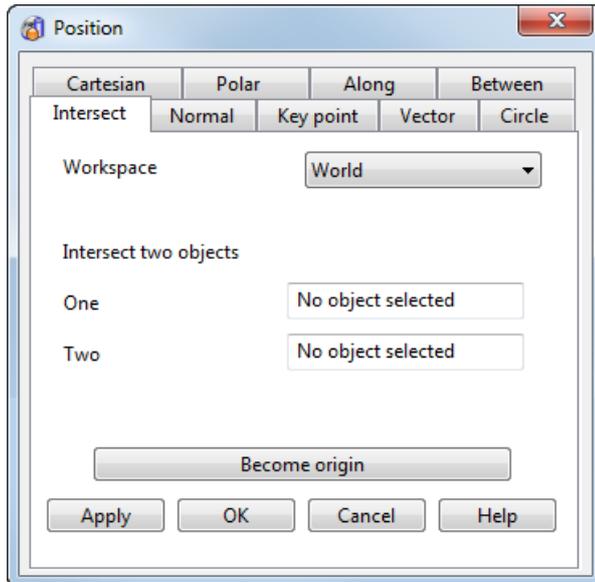
Cancel — Aborts the position entry.

Intersect tab

Use the **Intersect** tab of the **Position** dialog (see page 208) to find the position where two objects intersect.



You can select composite curves when you are using the **Intersect** page of the **Position** dialog:



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Select two objects in the model to find the intersection. The names of the selected objects are displayed in the following options:

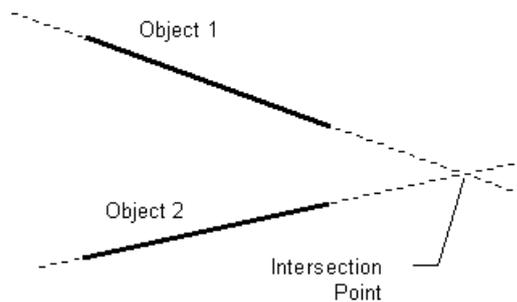
One — The name of one of the objects selected is displayed.

Two — The name of the other object selected is displayed.

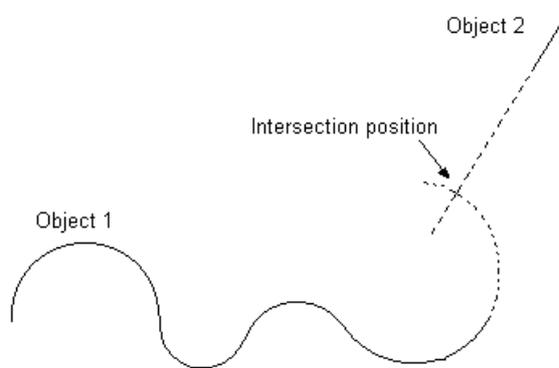
In order to find intersections, PartMaker Modeling does the following:

- If the position does not lie on either object, the objects are extended.

- Lines are treated as if they are of infinite length.
The intersection point of two lines that don't cross is shown below.



- Arcs are treated as full circles.
- If there are two possible intersections, the intersection position is the one closest to the last mouse click.
- Curves are extended by maintaining the curvature at the end point.



Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

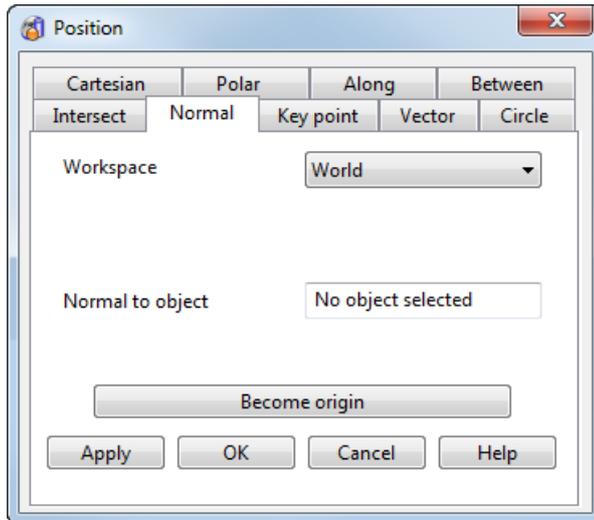
Cancel — Aborts the position entry.

Normal tab

Use the normal tab of the **Position** dialog (see page 208) to enter a position which is perpendicular to an object from the origin of the workspace displayed on the dialog.



You can select composite curves when you are using the **Normal** tab of the **Position** dialog:



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Normal to object — Click the object you want the position to be *normal* to. The name of the object is displayed in the box and the cursor moves to the position *normal* to the object.

To allow positions to be found that do not lie on the object, PartMaker Modeling does the following:

- Lines are treated as if they are of infinite length
- Arcs are treated as full circles.



An error dialog is displayed if a position cannot be found normal to the selected object.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

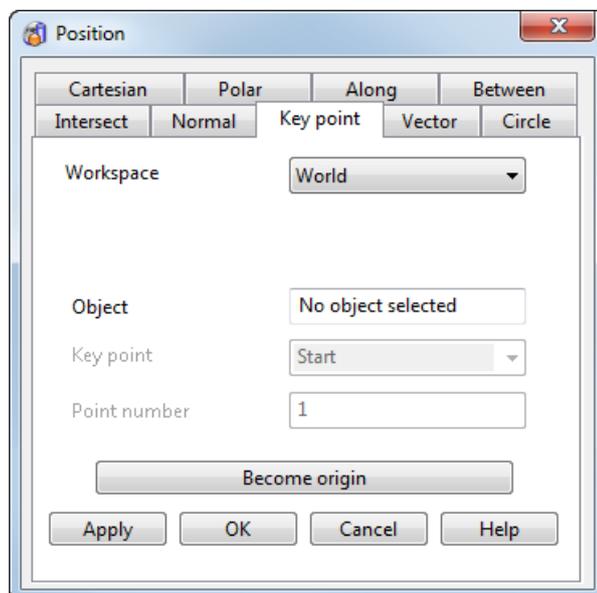
Cancel — Aborts the position entry.

Key point tab

Use the **Key point** tab of the **Position** dialog (see page 208) to select a position at a specific key point on the selected object. This is useful where more than one object has key points at the same position and (for example) you want to snap to a hidden key point on a specific object. Key points are a quick and convenient method for selecting specific objects.



*You can select composite curves when you are using the **Key point** page of the **Position** dialog:*



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Object — Select an object. The name of the object is displayed.

Key point — This determines which point on the object is selected. Select from the following options:

Centre — This applies to the centre point of arcs and circles.

Start and End — This can apply to any object as they all have start and end positions.

Point Number — This is intended for use with curves that can have many positions along their length. The **Point Number** box activates where you enter a number.

Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

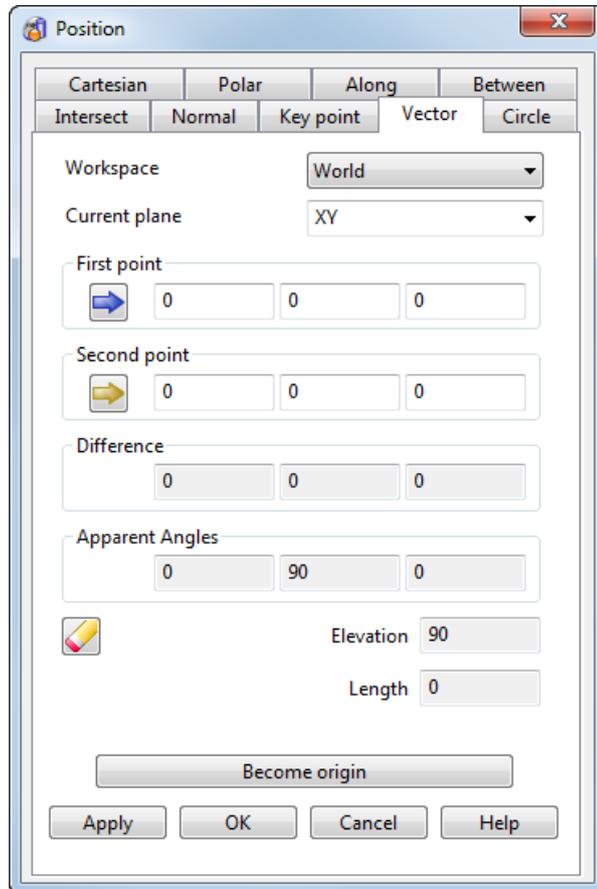
Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Vector tab

Use the **Vector** tab of the **Position** dialog (see page 208) to measure a vector between two points and use it to define a position. The offsets, angles, length, and elevation between the two points are displayed.



Workspace — Select the workspace you are working in. The available options are:

Relative — The position being defined is relative to another position which becomes the origin of the workspace. The positions are aligned with the active workplane or the global workspace if no workplane is active.

Relative mode is not always available since there may not be a suitable relative co-ordinate. In this case, select a suitable position (such as the intersection of two lines) and make it the origin of a relative workspace with the **Become Origin** option.

Workplane — The position being defined is relative to the active workplane if one exists.

World — The position being defined is relative to the global coordinate system.



If any items are locked, the locks are discarded when you select a different workspace.

Current Plane — Use this drop down list to select one of the three principal planes. The current plane is relative to the selected **Workspace** option.

You must already have a starting position from which the new position is entered.

First point — Enter the first point of the vector or click on a point on the model to enter its coordinates. This point is displayed in pink on the screen (using the default colour scheme).

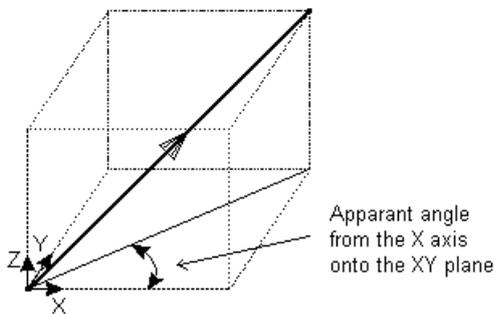
Second point — Enter the second point of the vector or click on a point on the model to enter its coordinates. This point is displayed in red on the screen (using the default colour scheme).

When you click **OK** or **Apply**, a new position is entered by going along the vector from the start position.

Information from the two positions of the vector is displayed automatically in the boxes for the following options:

Difference — These are the differences in the X, Y, and Z values of the first and second points of the vector.

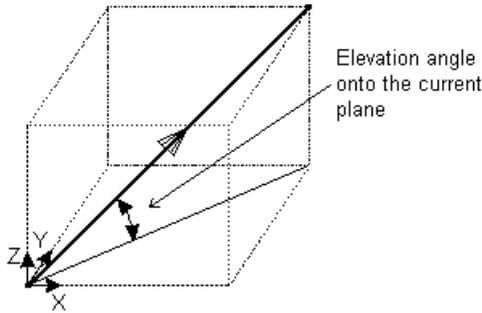
Apparent Angles — The *apparent angle* is measured between the projection of the vector onto a plane and the selected axis. If the axis is X, the plane is XY. For Y the plane is YZ and for Z it is ZX. An example is shown below of the apparent angle for the X axis:



Reset — Click this button to reset the values on the page.

Length — This is the true length (in 3D) between the points.

Elevation — This is the elevation angle of the new vector from the current plane.



Become origin — Click this button for the coordinates of the position defined by the dialog to become the origin. Any relative coordinates are measured from this origin.

Apply — Inputs the position. If the current operation allows multiple points, the dialog remains displayed ready for you to enter more points. Otherwise, the dialog is removed from the screen.

OK — Inputs the position and removes the dialog.

Cancel — Aborts the position entry.

Entering angles

Enter an angle using degrees, minutes, and seconds in any dialog that expects an angle. The format is one of the following:

> **d;m;s**

> **-d;m;s** (for negative angle)

> **-(d;m;s)** (for negative angle)

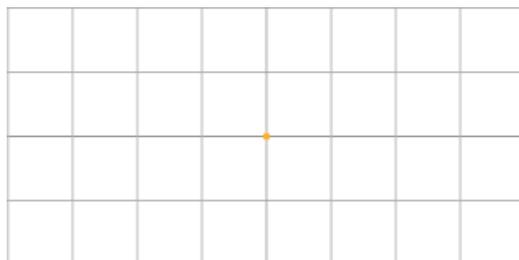
where **d**, **m**, and **s** are numbers.



*This syntax does not work for the **Calculator** dialog, or for the calculator that is embedded in the **Parameter** dialog.*

The grid

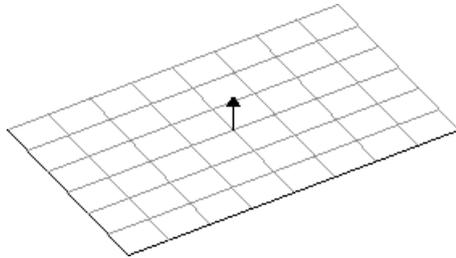
A grid is an array of points joined by lines. Use the grid as a guide to quickly enter positions by snapping to its points.



Using the grid

To use the grid turn the grid ON (see page 224).

The grid is drawn on the principal plane. If you change the principal plane, the grid automatically redraws on the new plane. The principal plane is displayed on the grid. Also displayed is the arrow at the origin of the current workspace. This shows the axis *normal* to the principal plane.



You can:

- Snap to the intersections of grid lines as if they are key points.

View the grid as either ruled or dotted lines by changing the Grid type option on the View page of the Options dialogs

The scale of the current grid is displayed in the Status bar (see page 125). The scale can be locked (see page 125) in the status bar to fix the scale. The scale is locked by default to a value of 10. You can set your own grid scale by typing your value into the grid scale text box in the Status bar (see page 125). The grid scale is locked automatically to that value.

To zoom in and out on a locked grid scale:

- Click **Zoom in**  to increase the scale.
- Click **Zoom out**  to decrease the scale.

To unlock the scale, deselect the **user defined scale** option on the **Blanking and Grid** page of the **Options** dialog. For further details, see Blanking and Grid in the Menus and Toolbars manual.

Turning the grid On and Off

To turn ON the grid

From the Status bar (see page 125), click **Grid on/off**  to draw the grid on the screen.

To turn OFF the grid

From the Status bar (see page 125), click **Grid on/off**  to remove the grid from the screen.

Using point-locking

When point-locking is ON, every position you enter using the mouse is locked onto the principal plane. For example, if the principal plane is XY , every point you enter using the mouse will have a Z coordinate of zero.

You can:

- Turn point-locking on (see page 225)
- Turn point-locking off (see page 226)

Turning point-locking on

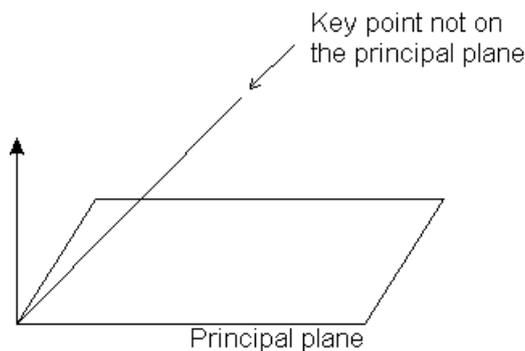
From the **Status bar**, click the **Point lock** button . (It changes to a shut lock .

Point-locking affects only points entered using the mouse.

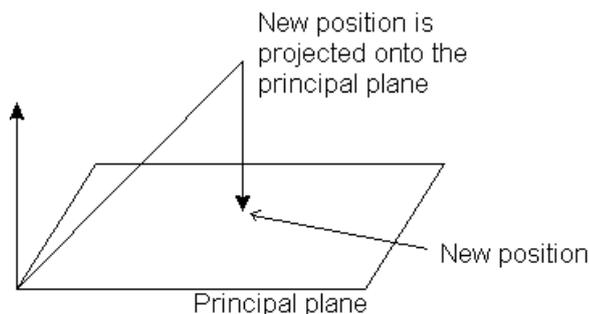
To enter a position that is not on the principal plane, you can:

- Use the Position dialog (see page 208).
- Type in the coordinates.

If you snap to a key point which does not lie on the principal plane (as shown below),



then the new position is projected onto the principal plane.



When you hover the intelligent cursor (see page 153) over such a key point, it changes to show that clicking the key point will project any new position onto the principal plane.

Turning point-locking off

From the Status bar, click the **Point lock** button . (It changes to an open lock .

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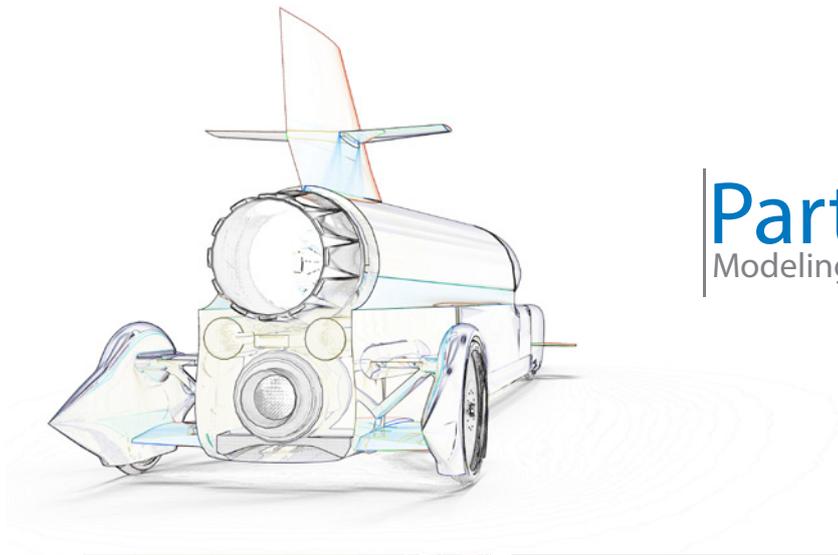
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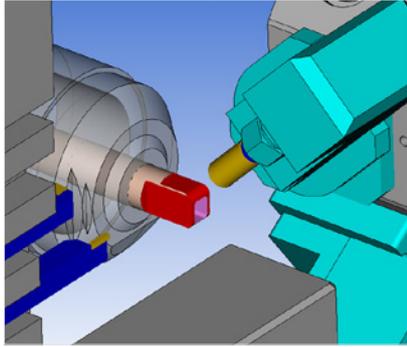
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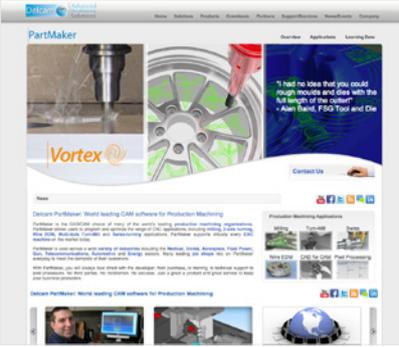
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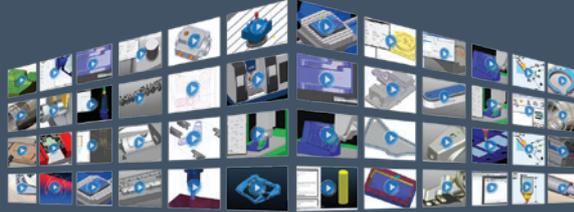


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