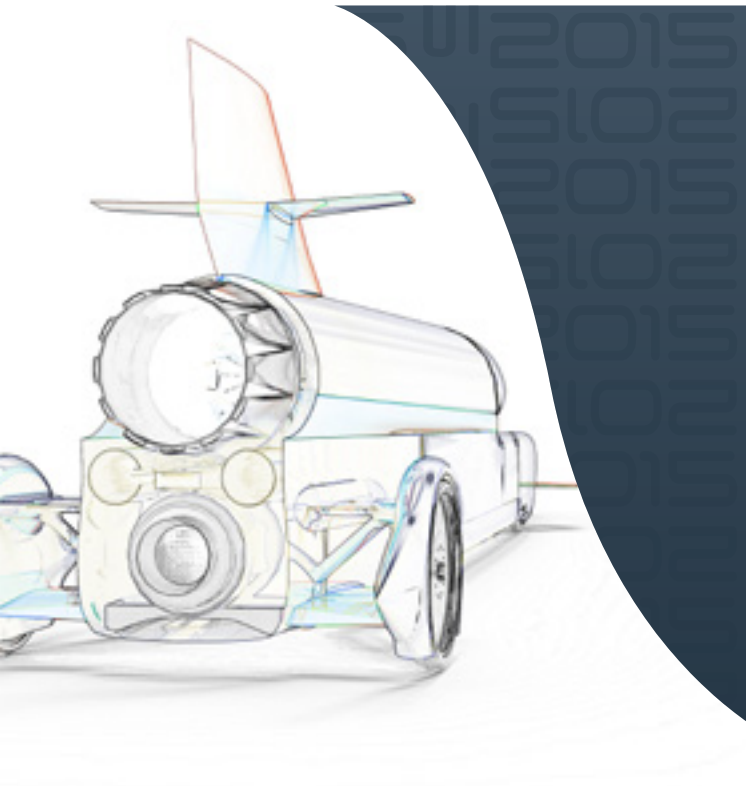


Delcam Advanced Manufacturing Solutions

PartMaker 2015

PartMaker WireEDM

Version 2015 or Higher

User Guide

Sequence of Programming Steps for PartMaker-EDM

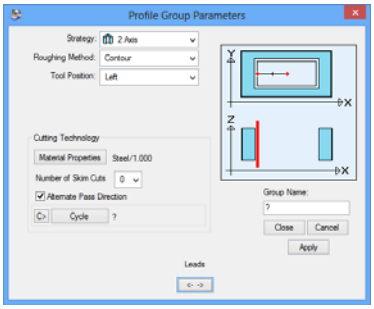
Note: *If you already have entered the required tools and cycles for a job proceed to step 3.*

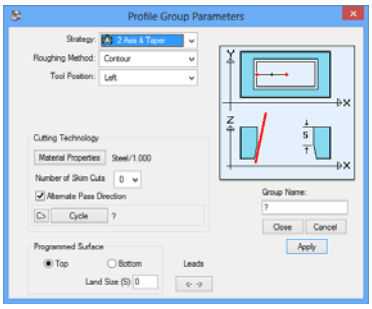
1. Choose the **Tools** command from the **ToolMinder** menu to add new wires.
2. Choose the **Cycles** command from the **ToolMinder** menu to add new cycles.
3. Choose the **Open Material File** command from the **File** menu to load the materials file.
4. Choose the **Defaults** command from the **Job Optimizer** menu to set up your defaults.
5. Choose the **Setup** command from the **View** menu to specify part boundaries and choose a wire.

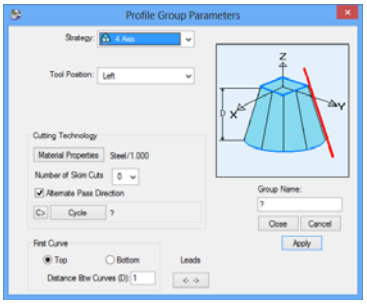
6. Create geometry in CAD mode or choose the **Import** button from the tool bar if you want to import geometry from a .dxf file or a solid model.
7. Switch back to CAM Mode clicking the CAD/CAM switch.

The CAD/CAM switch appears as in CAM Mode and as in CAD Mode

8. Choose the **New Profile Group** command from the **Part Features** menu to add a new profile group.







9. Create profiles using the following icons:

Profile Icon Chain Geometry 2 Point Chain

10. Set approach and escape points by double clicking on a tool path with the **Selection** icon.

11. Modify burning strategies and tapers using the **Profile Info Dialog** icon
12. Repeat steps 8 through 11 to create groups of all part features required for the job.

13. Choose the **Open Post Configuration File** command from the **Job Optimizer** menu and select the desired post processor for your Wire EDM machine.

14. Choose the **Generate Process Table** command from the **Job Optimizer** menu to generate a process table.

15. Choose **Simulation** from the main tool bar to watch the part the cut in 3D.

16. Choose the **Generate NC Program** command from the **Job Optimizer** menu to generate an NC Program.

PartMaker 2015

User Manual

User Guide/PartMaker WireEDM

Important User Notices

Copyright 1991-2014 Delcam Ltd. All rights reserved.

Delcam Ltd has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence. Delcam Ltd grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

Patents

PartMaker software is subject to the following patents:
Patent granted: US 6, 112, 133 Visual system and method for generating a CNC program for machining parts with planar and curvilinear surfaces
Patent granted: US 6, 741, 905 Visual system for programming of simultaneous and synchronous machining operations on lathes

PartMaker 2015. Published on 08 December 2014

Contents

Chapter 1: Getting Started with PartMaker 1

In Your Package 1
Technical Support 1
Before Using PartMaker... 2
System Requirements 2
Starting PartMaker 3
Exiting PartMaker 4
Using On-Line Help 4
Installing Updates 6
Working with Databases 6
Working With Files 6

Chapter 2: PartMaker Fundamentals 1

Introduction 1
Tools (Wire) Database 1
Material Database 2
Cycles Database 3
Working in a Face Window 10
Using the CAD/CAM Switch 11
Using the Color Bar 11
Using the Icon Toolbars 11

Main Toolbar 12
Face Window Toolbar – CAM Mode 12
Face Window Toolbar – CAD Mode 12

Creating Groups of Part Features 13
Using the Graphics Icons in PartMaker CAD mode 13

Geometry Drawing Icons 14
Dimensioning Icons 15
Snap Modes 16

Using the Graphics Icons in PartMaker CAM mode 17

Chapter 3: PartMaker WireEDM Tutorial 1

Introduction 1

How you will Create EDM_DEMO 1

Getting Started 3

Set Up Part Boundaries 4
Select a Material for the Part 7
Setting Defaults 9

Create Geometry 11

Creating the tutorial part in PartMaker CAM Mode 13

Define Group 1 Profiles for Inside Contour 13
Define Group 2 Profiles for Tapered Circle 19
Define Group 3 Profiles for Slot 24
Define Group 4 Profiles for Outside Contours 28

Generate a Process Table 32

Simulate the Process 34

Generate an NC Program 35

Chapter 4: PartMaker® Simulation 1

Introduction 1

Using PartMaker Simulation 1

The Simulation Toolbar 3

Simulation Features 4

Reset 4
Show Finished Part 4
Save Finished Part As STL file 4
Compare Finished Part to Solid Model 5
Comparison Options 5
View Setting Features 5
View Manipulation Features: Rotate, Move and Zoom 7
Sectioning Features 8

Simulation Options 9

Pause On 9
Display Control 10
Display Quality 10
Colors and Textures 10

Appendix A: Solids Import Tutorial for WireEDM 1

Introduction 1

Sequence of Steps for importing solid models and geometry extraction 1

Importing a Solid Model 2
Origin/Axis Transformations 3
Geometry Transfer and Part Interrogation 3

Edit Coordinate System Dialog 5

Copyright @2014 Delcam Ltd

Chapter 1: Getting Started with PartMaker

In Your Package

- PartMaker CD-ROMs
- PartMaker *Manuals*
- Hardware lock

Separately, if you have a stand-alone license of PartMaker, you will receive a Product Authorization File (PAF).

Technical Support

Please use the following information when you wish to get Technical Support or more information about PartMaker's products:

Phone: 215-643-5077

Fax: 215-653-0105

e-mail: support@partmaker.com

Web Site: http://www.PartMaker.com

A Customer PIN has also come included with your PartMaker software.

Before Using PartMaker...

Before you begin working with **PartMaker**, you need a basic knowledge of Microsoft Windows operations. You should understand mouse techniques such as pointing, clicking, double clicking, dragging, choosing menu commands, and making dialog selections. If you are unfamiliar with basic Microsoft Windows terms or techniques, see your Microsoft Windows documentation for details.

System Requirements

Please read the following sections to determine what you'll need to get started with **PartMaker**. To use PartMaker, you need a Windows-based PC with the following specifications:

	PartMaker Standard	**Parasolid Import Module (Solids)***	**Full Machine Simulation (FMS)**	**Advanced Surface Machining (ASM)**
Processor	Pentium 3 800 MHz or higher	Pentium 3 800 MHz or higher	Pentium 4 3 GHz or higher	Intel Core Duo processor or higher
Memory	512 MB or more	1 GB or more	2 GB or more	4 GB or more
Hard drive	40 GB	40 GB	40 GB	80 GB
Independent video card	128 MB NVIDIA	512 MB NVIDIA	512 MB NVIDIA	1 GB NVIDIA

64-bit and 32-bit versions of PartMaker are available and are supported on Windows 7. Only the 64-bit version of PartMaker is compatible with Windows 8 PCs.

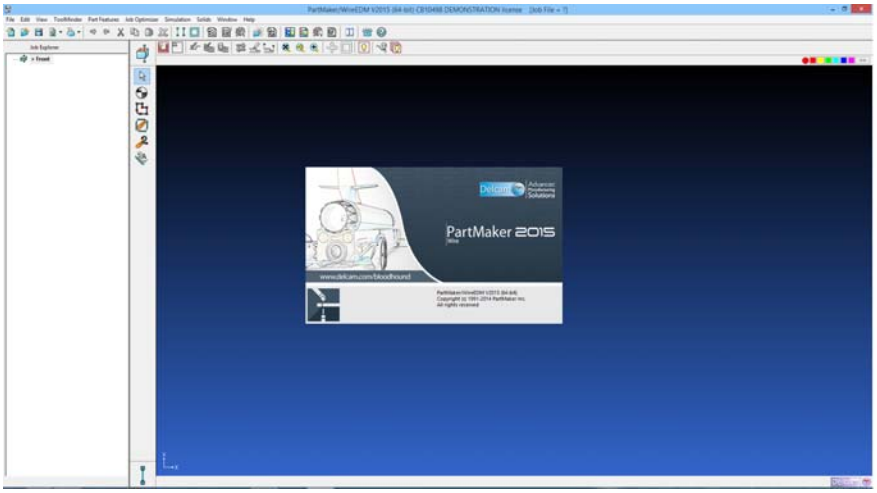
Please take note that Microsoft has discontinued support for the Windows XP and Vista operating systems.

Copyright @2014 Delcam Ltd

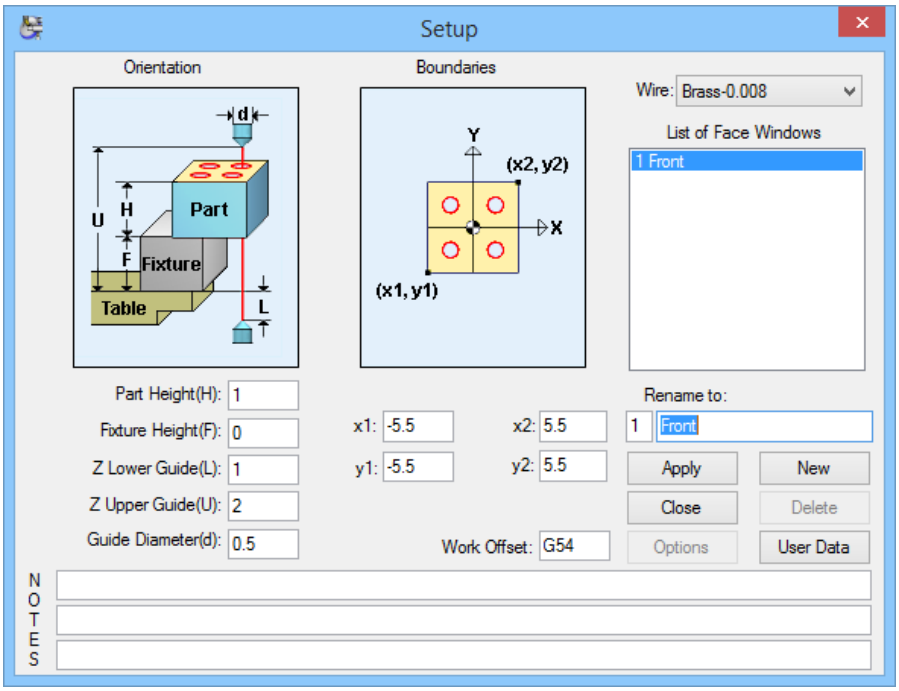
Starting PartMaker

From your desktop, double-click the PartMaker WireEDM icon.

A window containing information about **PartMaker** appears for a few moments.



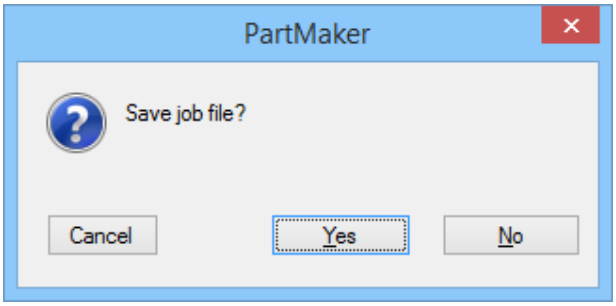
The **PartMaker** application window and the Setup dialog are displayed in the **Face Window**.



Exiting PartMaker

1 To exit **PartMaker**, choose **Exit** from the **File** menu.

If you changed an open job file, a dialog prompts you to save the current job file before exiting.



2 Do one of the following:

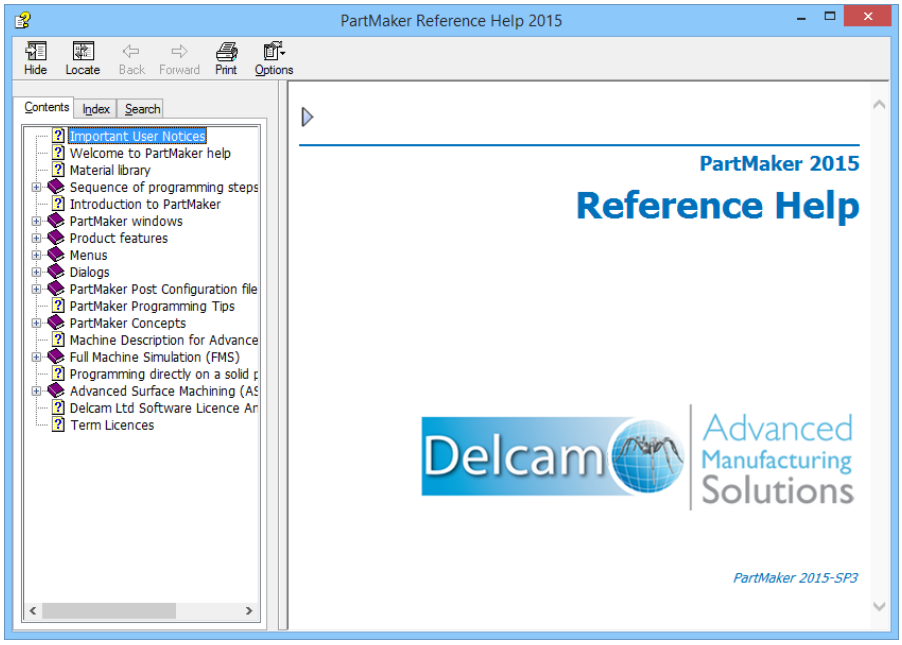
- Click <Yes> to save the changes.
- If the file is untitled, select a location and enter a file name in the dialog that appears, then click <OK>.
- Click <No> to discard the changes and exit.
- Click <Cancel> to return to the presentation window without exiting.

Using On-Line Help

On-line help provides fast access to information about the application's tools, commands, dialogs, and program features. Help commands are located in the Help menu in **PartMaker**:

1 Choose **Help Topics** from the **PartMaker Help** menu or press **<F1>.**

The Help window appears, displaying the **Main Index**.



2 Click an index to view a list of help topics.

Copyright @2014 Delcam Ltd

3 Click the topic you want to read about.

Information about the selected topic appears in the window.

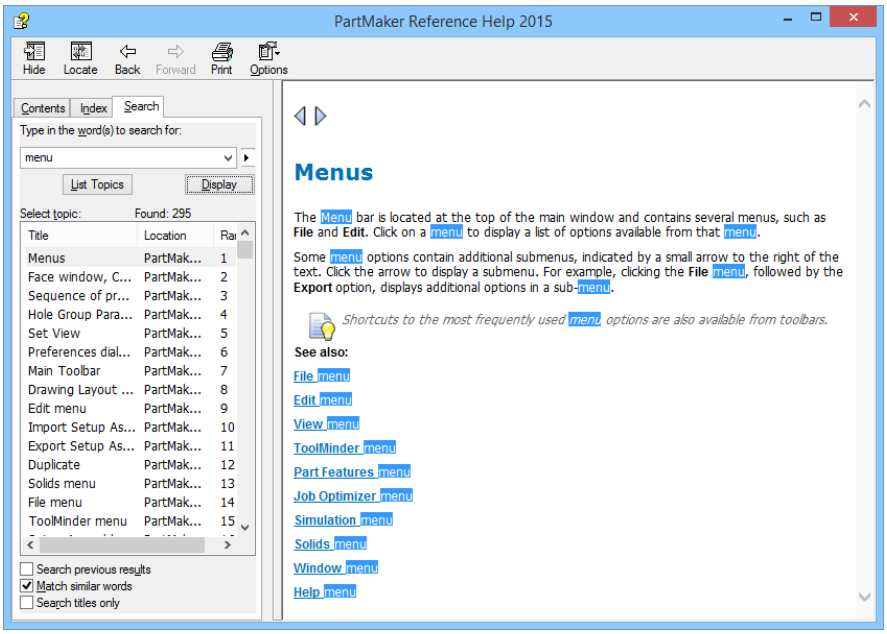
Note: *Some topic lists include sub-lists from which you can select a help topic.*

4 To return to the last screen you viewed, click the **<Back>** button.

5 To search for information, click the **<Search>** button.

6 Type in a key word to choose a topic and click the **<List Topics>** button.

7 Click the **<Display>** button to view the selected topic. Your help screen will appear as below:



8 When you are finished using help, choose Close from the help window's Control menu.

Using Context-Sensitive Help

To view context-sensitive help while viewing any **PartMaker** dialog:

1 Display any **PartMaker** dialog.

2 Press <F1> to display the help topic for the dialog.

Installing Updates

PartMaker Version 2010 and higher allows you to update to more recent versions and service packs of the software automatically from the PartMaker environment. You can check for and download the most recent version of PartMaker by selecting **Check for PartMaker Updates** from the **Help** menu.

Working with Databases

Databases are where you store tool, material, and cycle information for **PartMaker** jobs. **PartMaker** uses material, tools, and cycles databases to store the information needed for each job.

Material Database

A Materials Database allows you to store your shop's accumulated materials data. Materials data is used by PartMaker in the automatic calculation of feedrates and spindle speeds. In addition, you can utilize material data in the extensive material library provided with PartMaker.

Turn to Appendix A for a complete list of materials.

Tools (Wire) Database

A Tools Database allows you to keep track of your current wire inventory. PartMaker uses geometric and wire characteristics saved with each tool in the database for automatic cutting conditions.

Cycles Database

PartMaker lets you combine a number of repetitive operations (center drilling, drilling, tapping, boring, etc.) into a single entity called a Cycle. All cycles are saved in a Cycles Database. For each cycle operation, you can designate a tool from the Tools Database. Once a cycle is created, you can recall it at any time when you need to use it again.

Working With Files

PartMaker uses several kinds of files for storing tools, cycles, and material information as well as your jobs and the postprocessor files used to create NC programs for machining.

- Information about wires in the tool crib is saved in Tools files that have the file extension **.TDE**.
- Information about material characteristics needed to calculate machining strategies is saved in Material files that have the file extension **.MDE**.
- Information about user-created machining cycles is saved in Cycles files that have the file extension **.CDE** .
- Part geometry is saved in Job files that have the file extension **.JBE**.
- Postprocessor configuration information is saved in Post Configuration files that have the file extension **.PST**.

Note: *In PartMaker Wire EDM, notice that the Tools, Materials, Cycles and Job files all end with an* ***e****. This distinguishes Wire EDM files from files use in other PartMaker applications.*

Copyright @2014 Delcam Ltd

Chapter 2: PartMaker Fundamentals

Introduction

This chapter provides an overview of the **PartMaker** environment. You will learn basic concepts such as how to work in a face window to define part geometry in CAD Mode and locations of profiles and other icons in CAM mode. Also, this chapter will describe how to apply PartMaker's knowledge-based approach to programming Wire EDM.

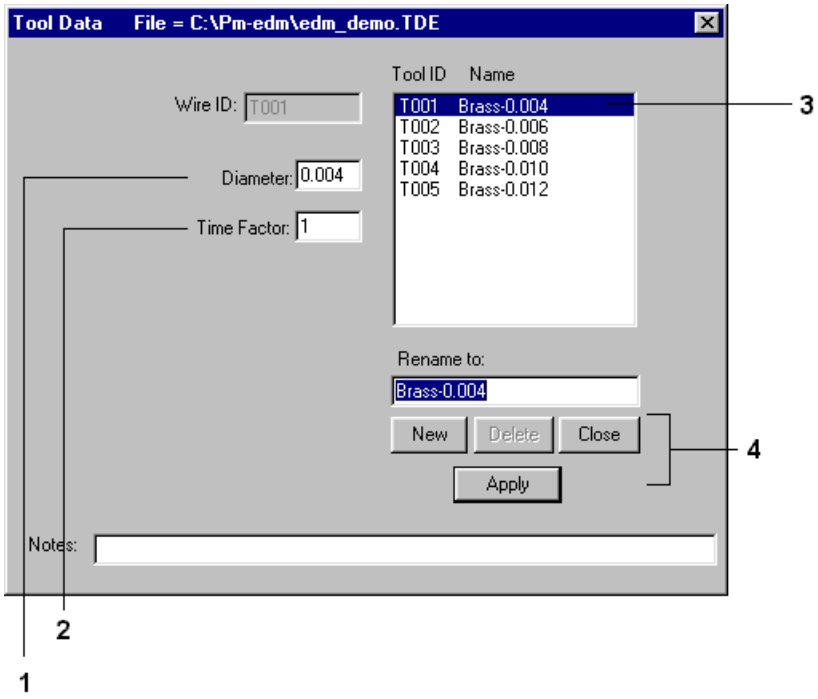
Tools (Wire) Database

A Tools Database allows you to keep track of your current wire inventory. PartMaker uses geometric and wire characteristics saved with each tool in the database for automatic cutting conditions.

Working with the Tools Database in PartMaker Wire EDM

You can access the Tools Database by choosing **Tools** from the **ToolMinder** menu.

When doing so you should see the dialog shown below:



1 **Diameter** represents the wire diameter being used

2 **Time Factor** allows the user to incorporate the adaptive feed rate control provided by some EDM controls in the time calculation displayed on PartMaker's Process Table.

3 This list specifies the current wire inventory stored in the **Tools Database**. The wires specified here are available for selection from the **Setup** dialog when starting PartMaker.

4 Clicking these function buttons with the left hand mouse button does the following:

<New> - Allows you to add additional wires to the Tools database.

<Delete> - Allows you to remove or delete an existing tool from the tools database.

<Close> - Accepts the changes you have made and closes the dialog.

<Apply> - Applies the changes you have made and leaves the dialog open.

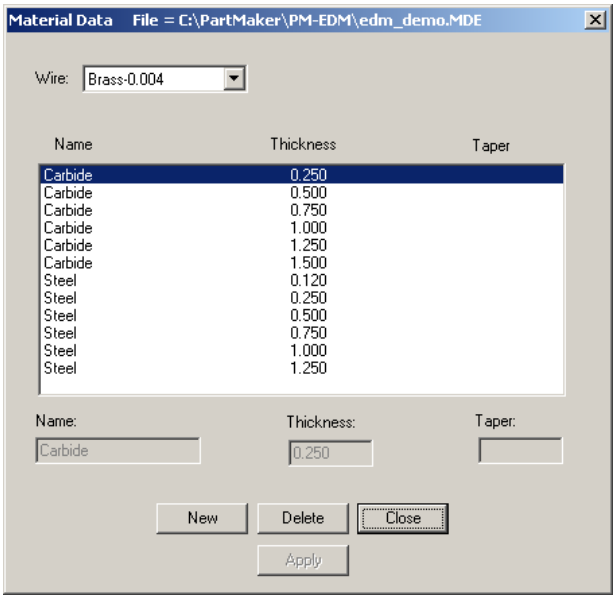
Material Database

A Materials Database allows you to store your shop's accumulated materials data. Materials data is used by **PartMaker** in the automatic creation of burning strategies. In addition, you can utilize material data in the extensive material library provided with **PartMaker**.

Working with the Material Database in PartMaker Wire EDM

You can access the Materials Database by choosing **Material** from the **ToolMinder** menu.

When doing so you should see the dialog shown below:



Wire represents the wire diameter as specified in the **Tools Database** for which the materials listed are available

The list of Materials represents the current inventory of materials available for the given the wire selected.

These fields specify the following information for a given material:

Name – Specifies the name of the designated material.

Thickness – Specifies the initial thickness of the designated material.

Taper - Specifies the taper of the cut for a given material. You may have a number of the same materials each specified with a different taper. This is important because various taper sizes will warrant different burning strategies and power settings.

Copyright @2014 Delcam Ltd

Cycles Database

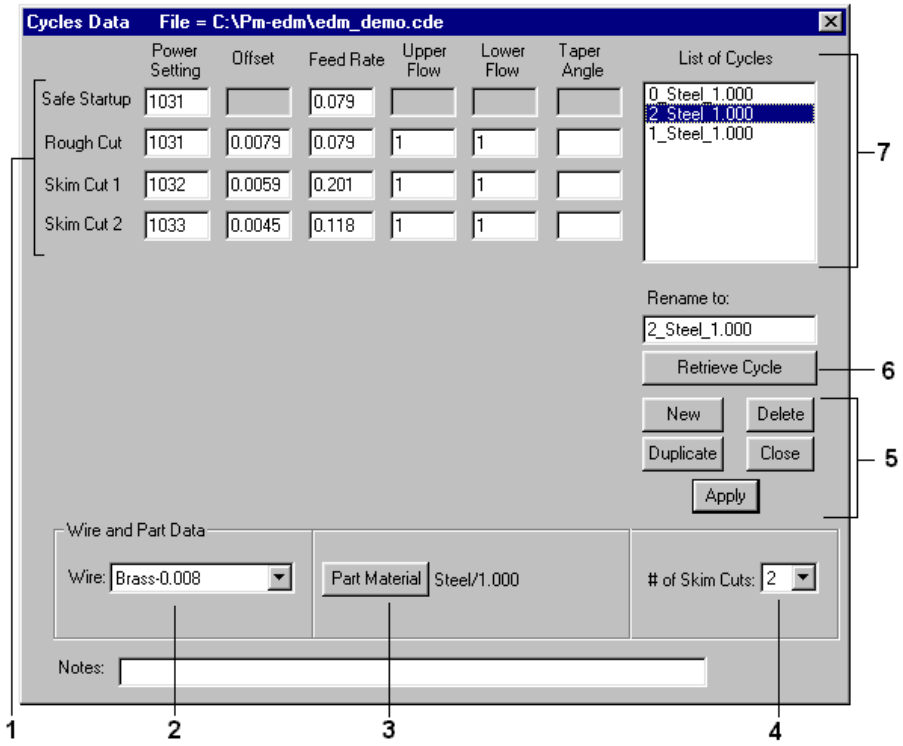
PartMaker lets you combine a number of repetitive operations (start up, rough cutting, finish cutting, etc.) into a single entity called a Cycle. All cycles are saved in a Cycles Database. Once a cycle is created, you can recall it at any time when you need to use it again.

Working with the Cycles Database in PartMaker Wire EDM

Cycles may only be added once the appropriate Wires and Materials have been created in the **Tools** and **Materials Databases** respectively.

You can access the Cycles Database by choosing **Cycles** from the **ToolMinder** menu.

When doing so you should see the dialog shown below:



1 These fields allow you to set the **Power Setting**, **Offset**, recommended **Feedrate**, **Upper Flow**, **Lower Flow** and **Taper Angle** for the Safe Startup, Rough Cut and each Skim Cut being defined for a given Cycle.

2 Specifies the wire from the **Tools Database** being used in the cycles.

3 Specifies the material from the **Material Library** being used in the current cycle.

4 Specifies the number of Skim Cuts being taken in this cycle.

5 Clicking these function buttons with the left hand mouse button does the following:

<New> - Allows you to add an additional Cycle to the Cycles database.

<Delete> - Allows you to remove or delete an existing Cycles from the Cycles database.

<Close> - Accepts the changes you have made and closes the dialog.

<Apply> - Applies the changes you have made and leaves the dialog open.

6 The <**Retrieve Cycle>** button allows you to select a cycle from an existing library of cycles created for a specific machine. Your PartMaker Wire EDM software comes with one such file standard as configured for a Mitsubishi FX-20 Wire EDM.

7 **The List of Cycles** text box lists all of the cycles currently stored in the Cycles database.

Below, find instruction for adding Cycles to the Cycles Database.

Adding Cycles to the Cycles Database in PartMaker Wire EDM

Cycles may be added to the Cycles Databases either by creating them one by one with advice of your Wire EDM manual or selecting them from a preexisting library of cycles for your machine. Most commonly you will be creating these cycles, though you can consult PartMaker Inc. or your authorized PartMaker reseller for more information on obtaining a preexisting library of cycles for your machine.

Method 1: Adding Cycles Individually

To create cycles individually, it is recommended you refer to the power recommendations that came with your Wire EDM's operations and/or programming manual.

On the page that follows, such information as taken from a Mitsubishi FX-20 Model Wire EDM (inches) has been inserted. Based upon this information, a typical inch cycle will be created.

Copyright @2014 Delcam Ltd

Mitsubishi
FX Submerged Extended View Data Sheet

Wire Diameter and Type	Material Type	Process	Material Thickness
.004 hard brass	Carbide	ST	0.25

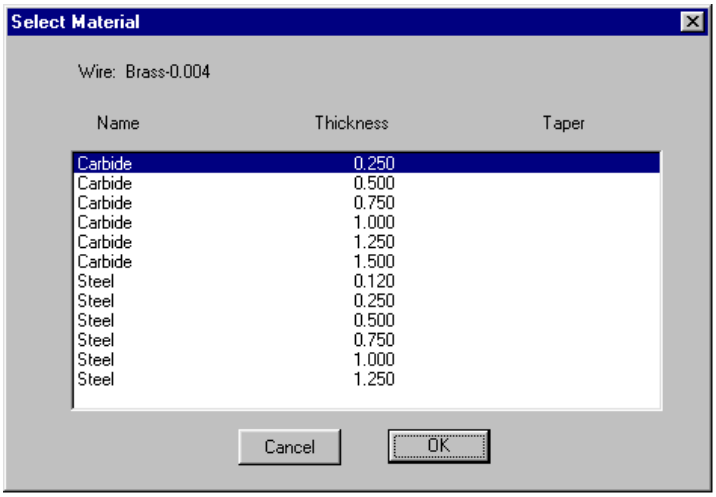
Cutting Process	Start-Up	Rough Cut	Skim 1	Skim 2	Skim 3	Skim 4	Skim 5
E-pack Number	5210	5211	5212	5213	5214	5215	5216
Voltage Open	14	14	4	3	4	3	4
Power Setting	4	4	4	4	3	3	1
Off Time	1	1	1	1	10	10	4
Stabilizer A	1	2	1	1	2	2	2
Stabilizer B	16	14	16	14	8	8	3
Wire Speed	12	14	14	12	12	12	12
Wire Tension	3	3	4	4	4	4	4
Pre-Tension	4	6	8	8	8	8	8
Liquid Quantity	2	2	1	1	1	1	1
Liquid Resistivity	7	7	7	7	7	7	7
Voltage Gap	56	46	46	50	65	60	35
Stabilizer C	1	1	1	1	1	1	1
AE Mode	21	21	21	21	11	11	11
Stabilizer E	1	2	1	1	1	1	1
Feedrate Address	0.015	0.035	0.1	0.09	0.14	0.14	0.14
Offset Value(s)	---------	**1**	**2**	**3**	**4**	**5**	**6**
Rough Cut	---------	0.00300	---------	---------	---------	---------	---------
Rough & 1 Skim	---------	0.00450	0.00250	---------	---------	---------	---------
Rough & 2 Skims	---------	0.00540	0.00340	0.00240	---------	---------	---------
Rough & 3 Skims	---------	0.00570	0.00370	0.00270	0.00230	---------	---------
Rough & 4 Skims	---------	0.00590	0.00390	0.00290	0.00250	0.00230	---------
Rough & 5 Skims	---------	0.00600	0.00400	0.00300	0.00260	0.00240	0.00250
Stepping Increment	---------	---------	0.00100	---------	---------	---------	---------
Upper Flow Rate	4.0	4.0	1.0	1.0	1.0	1.0	1.0
Lower Flow Rate	4.0	4.0	1.0	1.0	1.0	1.0	1.0

Copyright @2014 Delcam Ltd

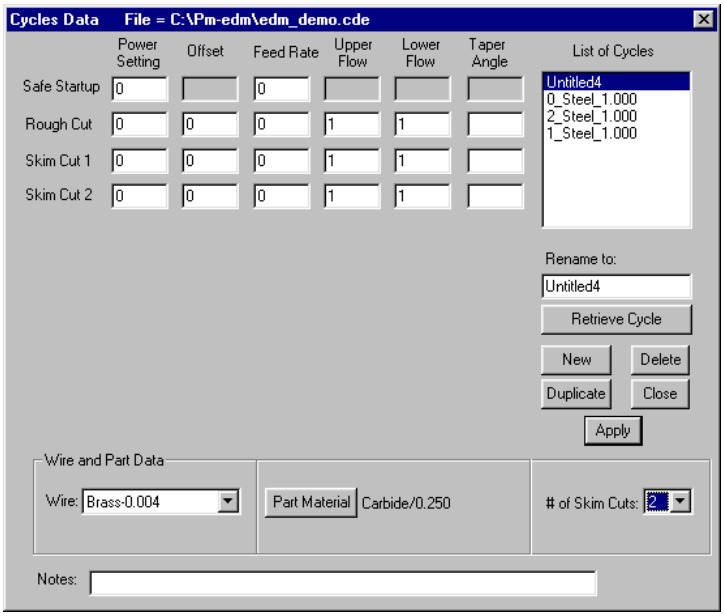
Method 1: Adding Cycles Individually (cont'd)

Referring to the information provided on the previous page, we will create a cycle for .250 inch thick carbide using 2 skim cuts with a .004 inch diameter wire. To do so:

1. Click the **<New>** button to create a new cycle that appears in the list of cycles as **Untitled 4**.
2. From the Wire drop down menu, choose **Brass .004**
3. Click the **Part Material** button and you will be prompted with dialog below:



4. From the list of Materials, choose Carbide 0.250 and click the **<OK>** button
5. From the **# of Skim Cuts** drop down menu, choose 2. At this point, your cycles dialog should appear as shown below:



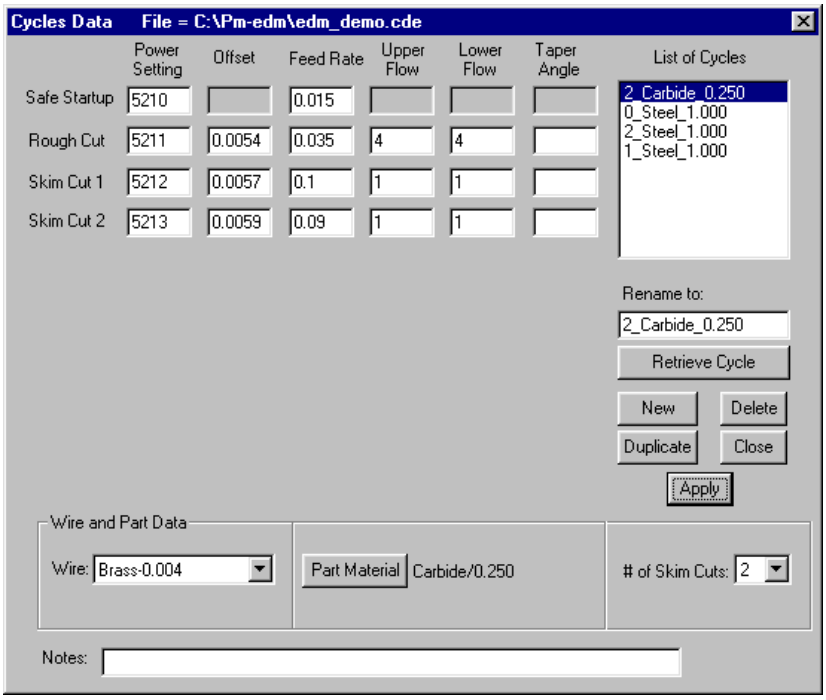
Copyright @2014 Delcam Ltd

6 Now, complete the Cycles page based upon the information provided on previous page. The relevant data you will enter is as follows:

	Power Setting	Offset	Feed Rate	Upper Flow	Lower Flow	Taper Angle
Safe Startup	5210		.015			N/A
Rough Cut	5211	.00540	.035	4.0	4.0	N/A
Skim Cut 1	5212	.00570	.100	1.0	1.0	N/A
Skim Cut 2	5213	.00590	.090	1.0	1.0	N/A

Make sure you understand where these values originated.

7 Once you entered the values in the table above, type 2_Carbide_0.250 in the Rename to: field, and click the <Apply> button. Your completed cycles dialog should appear as shown below:



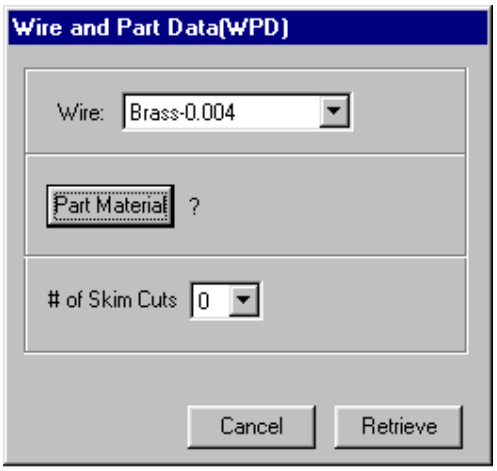
Method 2: Adding Cycles using the Retrieve Cycle Button

In certain cases PartMaker comes preloaded with Cycles for specific machine. These cycles can be accessed using the **Retrieve Cycle** button.

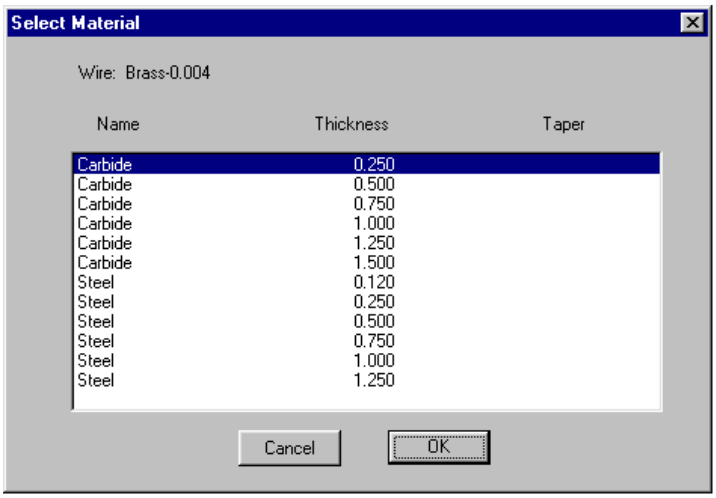
Important! *Before moving on, please be certain that cycles for your exact model of Wire EDM have been created.*

To retrieve the same Cycle created above from library for your machine:

1. Click the **<Retrieve Cycle>** button. After doing so, you will be prompted with **Wire and Part Data** dialog shown below:



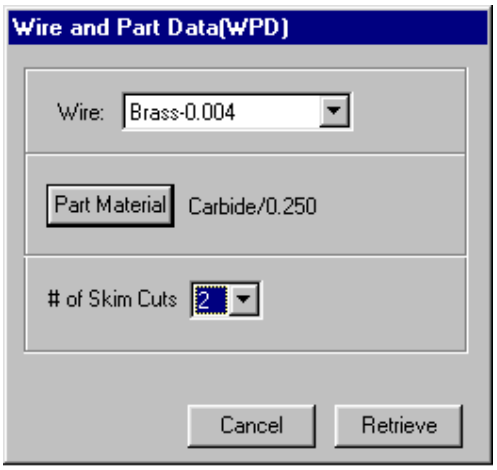
2. From the Wire drop down menu, choose **Brass .004**
3. Click the **Part Material** button and you will be prompted with dialog below:



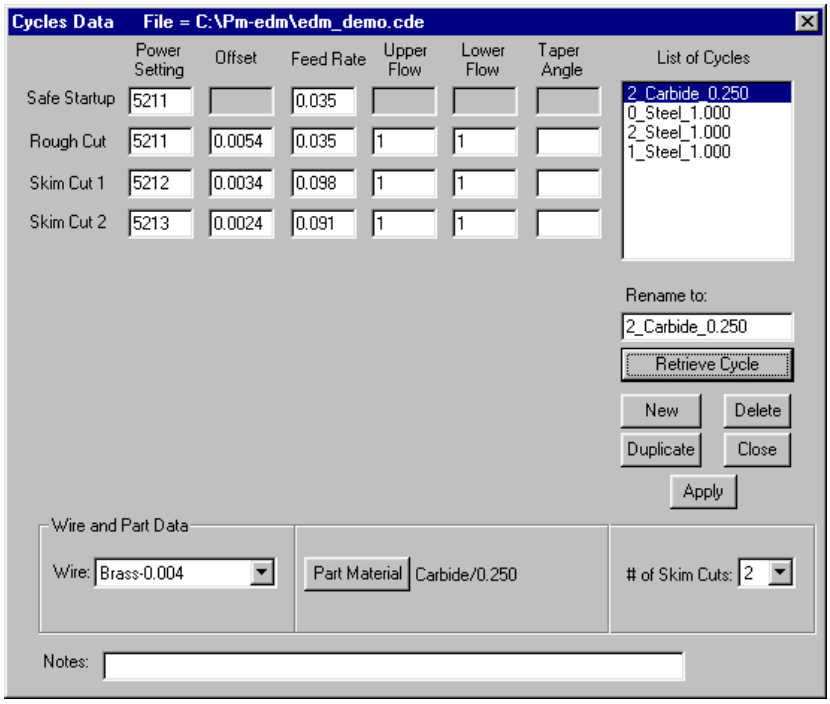
4. From the list of Materials, choose Carbide 0.250 and click the **<OK>** button
5. From the **# of Skim Cuts** drop down menu, choose 2.

Copyright @2014 Delcam Ltd

At this point, your **Wire and Part Data** dialog should appear as shown below:



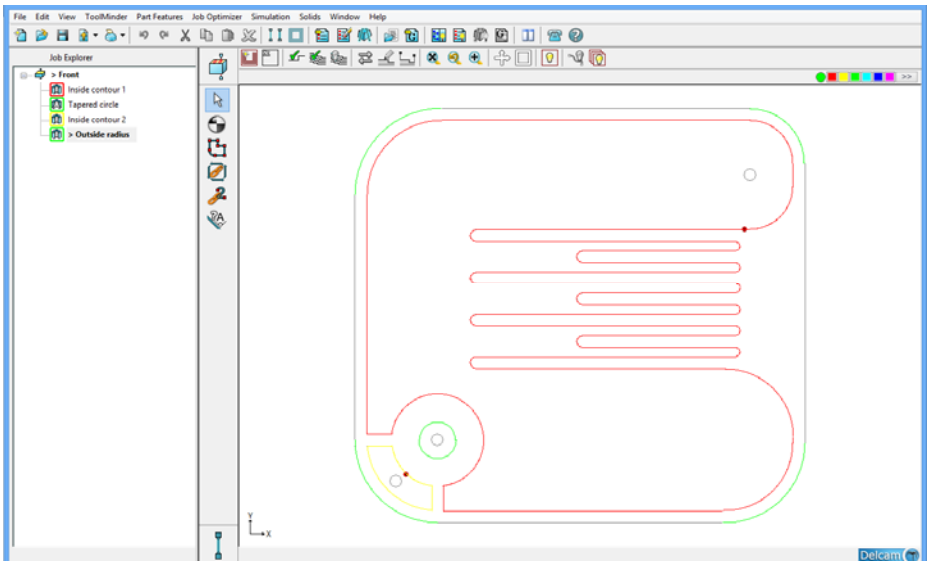
6 Click the retrieve button and your cycle will be retrieved from the master cycles database for your Wire EDM. Your completed cycles dialog should appear as shown below:



In Chapter 3, **Creating a Sample Part**, you will learn more about how to apply the uses of cycles to various burning strategies to create paths for the wire.

Working in a Face Window

A **Face Window** is the work area in which you define groups of holes and profiles that need to be machined. The main area of the window displays the part boundaries; the left side displays group icons, which have cycle name labels needed to machine the current part.



	Displaying the Axes	Choose Show Axes from the View menu.
	Displaying the Grid	Choose Show Grid from the View menu.
	Displaying the Boundaries	Choose Show Boundaries from the View menu.

Copyright @2014 Delcam Ltd

Using the CAD/CAM Switch

This icon represents CAM mode. Clicking it will switch the program to the CAD mode.

All tool path creation and process development is done in the CAM mode. All geometry creation and alteration is done in the CAD mode.

Using the Color Bar

You use the color bar in the upper-right corner of a **Face Window** to select a color for group symbols. You can also use it to change the color of geometry. The group symbol or geometric element will have the color of the currently chosen **Sample Color**.

Note: *If you want to change the color of a Group Symbol, select the Group Symbol in the* ***Face Window****, click a color on the color bar, and then click the Sample Color on the left side of the color bar.*

To change the color of a geometric element, select the geometry in the **Face Window**, click a color on the color bar, and then click the Sample Color on the left side of the color bar.

Using the Icon Toolbars

PartMaker Version 9 and higher provides Toolbars to speed up commonly used functions throughout the software. PartMaker supports four types of icon toolbars, a **Main Toolbar** which is available throughout a job file, a **Face Window Toolbar** which is associated with each **Face Window**, a **Solids Window Toolbar** which appears when working with an imported Solid Model and a **Simulation Toolbar** which appears when working in PartMaker's 3D simulation. Here, the **Main Toolbar** and **Face Window Toolbar** are explained. Chapter 6 of this guide will explain the use of the Simulation Toolbar while Appendix C to this manual will explain the use of the **Solids Window Toolbar**.

Main Toolbar

The **Main Toolbar** lets you perform many of the functions in PartMaker's menus with the click of a button. The **Main Toolbar** will look the same throughout your PartMaker session and appears as below:

Note: *When holding your mouse over any of these icons, a "Tool Tip" will appear showing the function performed by that icon. Each command is explained in greater detail in Chapter 4 of this manual.*

Face Window Toolbar – CAM Mode

The **Face Window Toolbar** lets you perform many of the functions in PartMaker's menu's with the click of a button. The **Face Window Toolbar** applies only to the active **Face Window** and may look differently depending on the **Machining Function** Face Window being used.

Face Window Toolbar – CAD Mode

Displaying Toolbar Icons

The display of the Toolbars can be manipulated by choosing **Toolbars** from the **View** menu. When a toolbar is checked, it will appear. When unchecked, it will not be visible.

Copyright @2014 Delcam Ltd

Creating Groups of Part Features

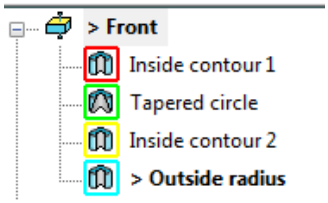
Groups of part features such as holes and profiles are created in **PartMaker**. The currently selected group icon or **Group Symbol** in a **Face Window** is the work group from which tool paths are being added.

To create a new Group Symbol:

1 Click the desired color on the color bar.

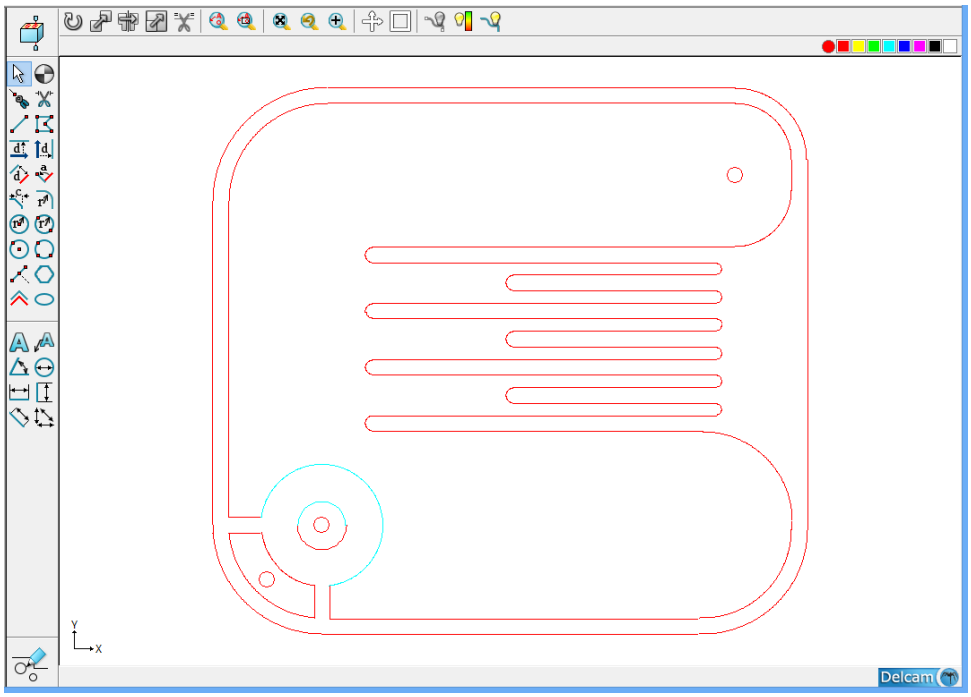
2 Choose New Profile from the Part Features menu to display the Group Parameters dialog.

When you exit the dialog, the new group is created using the color you specified.



Using the Graphics Icons in PartMaker CAD mode

You can create your part drawings in **PartMaker** in the CAD mode. Alternatively, you also import drawings from other CAD systems into **PartMaker** by using the **Import** command from the **File** menu. Imported geometry can be altered in **PartMaker**'s CAD mode.



Geometry is created in **PartMaker** CAD using the Graphics Icons that include Drawing Icons and Snap Modes.

Geometry Drawing Icons

Geometry Drawing icons allow you to create different types of geometric constructions such as lines and arcs. **PartMaker**'s geometry drawing icons are defined below.

Selection Icon indicates that the system is in the selection mode. In this mode you can select lines, arcs and circles in the **Face Window** either by clicking on them or dragging a selection rectangle around them.

Zero Icon allows you to define a new location for the zero point using Snap Modes.

Trim/Extend Icon allows you to stretch or shorten an existing line or arc.

Remove Icon allows you to remove portions of lines or arcs between intersection points.

Line Thru Two Points Icon allows you to create a line between two points specified using Snap Modes.

Connected Lines Icon allows you to create connected lines with end points specified using Snap Modes.

Line Parallel Horizontal axis Icon allows you to create a line parallel to the Horizontal axis by specifying positive or negative offset.

Line Parallel Vertical axis Icon allows you to create a line parallel to the Vertical axis by specifying positive or negative offset.

Parallel Element Icon allows you to create a line, arc, or circle parallel to the cursor selected entity specifying an unsigned distance.

Line on an Angle Icon allows you to create a line on an angle to a cursor-selected line passing through a point specified using Snap Modes.

Fillet Icon allows you to insert a fillet between two cursor-selected lines or arcs.

Chamfer Icon allows you to insert a chamfer between two cursor-selected lines or arcs.

Circle with a Known Radius and Center Icon allows you to define a circle with a center specified using Snap Modes.

Circle with a Known Radius and Two Points Icon allows you to define a circle with two points on the circumference specified using Snap Modes.

Circle with a Known Center and a Point Icon allows you to define a circle with a center and a point on the circumference specified using Snap Modes.

Circle Through Three Points Icon allows you to define a circle with three points specified using Snap Modes.

Polygon Icon allows you to create a various rectangles by entering a flat-to-flat distance.

Divide/Append Icon aids you in the "breaking" of segments at specific point within the geometry.

Multiple Offset Icon allows you to offset the selected geometry chain through a user-defined distance.

Ellipse Icon allows you to create an approximate ellipse in the face window, using tangential circular arcs.

Copyright @2014 Delcam Ltd

Dimensioning Icons

The Dimensioning Icons appear in the PartMaker CAD mode. These icons allow you to create dimension and annotations on your drawings in PartMaker.

Note Icon Allows you to create a note on a dimensioned drawing.

Note with Leader Icon allows you create a note with an arrow leader to point to a specific item on your drawing you wish to annotate.

Angular Dimension This icon allows you to insert an Angular Dimension between two lines..

Circular Dimension Icon This icon allows you to insert a Circular Dimension on arc or circle.

Horizontal Dimension Icon This icon allows you to insert a Horizontal Dimension between two points that are either end points of arcs and lines or centers of circles.

Vertical Dimension Icon This icon allows you to insert a Vertical Dimension between two points that are either end points of arcs and lines or centers of circles.

Linear Dimension Icon This icon allows you to insert a Linear Dimension between two points that are either end points of arcs and lines or centers of circles.

Flexible Dimension Icon This icon allows you to insert a “Flexible Dimension” that is automatically set by the software to either Horizontal Dimension or Vertical Dimension or Linear Dimension depending on the cursor position when the text location is specified.

Snap Modes

Snap modes are used in conjunction with geometry icons to define point location. **PartMaker**'s Snap Modes are defined below. Snap drawing modes are also used in the CAM mode to define tool path location.

XY Coordinates: Used for entering Cartesian coordinates.

Polar Coordinates: Used for entering Polar coordinates

Point on a Circle: Clicking a point on an existing arc or circle and entering an Angle

Screen: Clicking a cursor location anywhere within the face window

Grid: Clicking a grid location (grid can be turned on from the View Menu by choosing Show Grid. Grid size can be set in the Preferences dialog accessed from the View menu).

Circle Center: Clicking the center an existing circle

Closest Intersection: Clicking the closest intersection of geometric elements

End of an Element: Clicking the end of an existing line or arc

Middle of an Element: Clicking the middle of an existing line or arc

Tangency: Clicking near the tangency point that will be calculated by the system

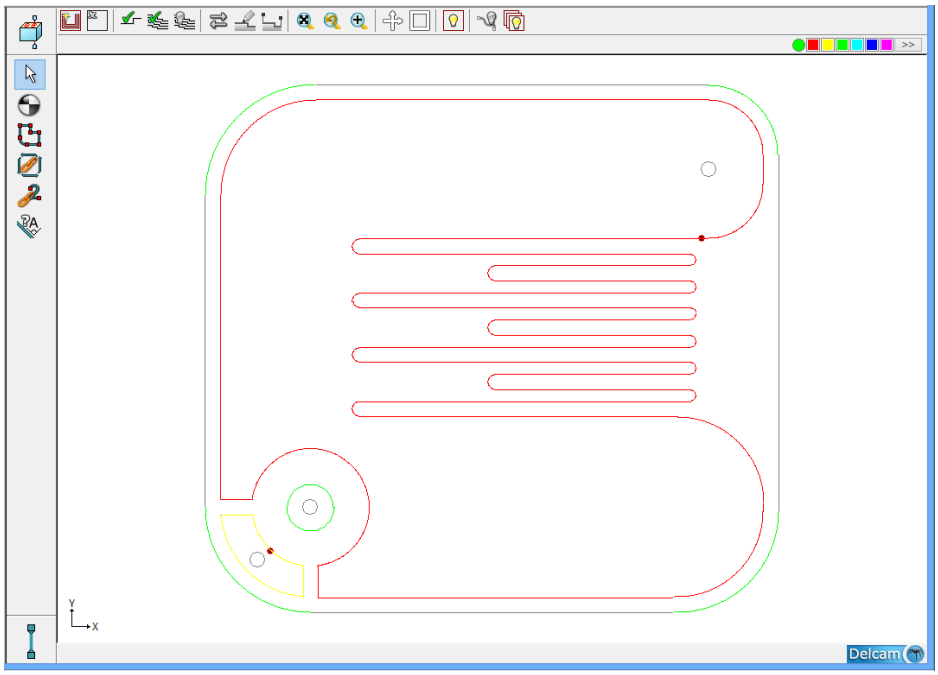
Horizontal Constraint allows you to specify a point whose vertical coordinate remains constant, allowing changes in horizontal coordinate only.

Vertical Constraint allows you to specify a point whose horizontal coordinate remains constant, allowing changes in vertical coordinate only.

Copyright @2014 Delcam Ltd

Using the Graphics Icons in PartMaker CAM mode

The Graphics Icons on the left side of a CAM **Face Window** allow you to define and manipulate tool paths. Certain Graphics Icons for use in assigning a particular part feature to your part (i.e. a Group Symbol) will have a different appearance and function depending on the Machining Function **Face Window** being used.



The two icons below will have the same appearance and usage regardless of which machining function **Face Window** you are using.

Selection Icon indicates that the system is in the selection mode. In this mode, you can select part features and tool paths in the Face Window either by clicking or dragging a selection rectangle around them.

Selected profiles in a Face Window are highlighted; unselected profiles are not highlighted. Selected holes in a Face Window are highlighted; unselected holes are hollow. You can delete selected elements by pressing the <Backspace> key on the keyboard or by choosing the Delete command from the Edit menu.

Zero Icon allows you to define a new location for the zero point or origin using Snap Modes.

Graphics icons on the left side of a face window allow you to define shapes for the EDM tool path.

Profile Icon enables you to specify that the currently active group symbol is to be applied to the profile described by either coordinate entry or by clicking on geometric elements.

Chain Icon enables you to specify that the currently active group symbol is to be applied to a profile that is created when PartMaker automatically connects all the elements that form an unbroken chain when an initial line or arc is clicked.

2-Point Chain icon The 2-Point Chain icon allows you to define a profile by the start point in the chain, and the end point of the chain.

Profile Info enables you to customize the properties of all or part of a tool path including changing a federate, offset, and adding text to individual finishing profile lines in contouring.

Copyright @2014 Delcam Ltd

Chapter 3: PartMaker WireEDM Tutorial

Introduction

The print on the following page represents a part that may be made on a Wire EDM. The name of this part is EDM_DEMO. You'll create the EDM_DEMO part in this chapter using **PartMaker**.

This tutorial is designed to help you learn some of the most common **PartMaker** commands and features you will use every day. Before you begin, we recommend you read Chapter 2 if you have not already done so to familiarize yourself with some basic **PartMaker** concepts and features.

Important! *The tutorial below is presented in both inch and metric units, depending on how you are working with PartMaker. The inch units of measure are given first, with the metric entries gives in parenthesis ( ).*

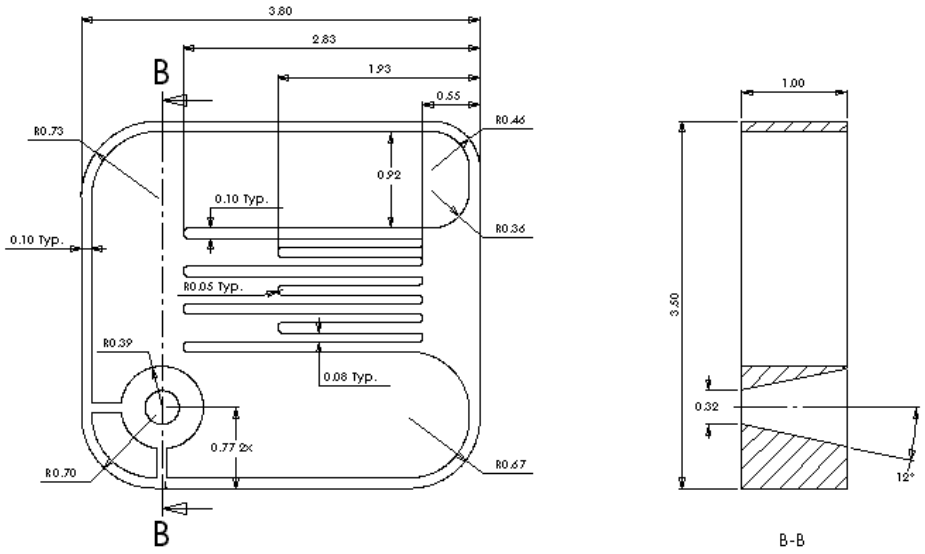
For example you may see instruction that says: Enter an X value 1 (25), where 1 is 1 inch and 25 is 25 mm.

How you will Create EDM_DEMO

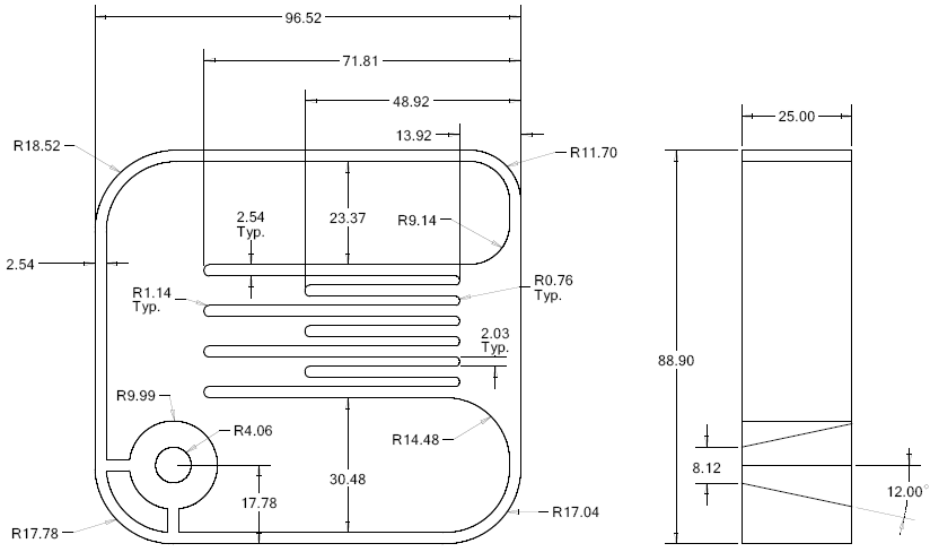
Here are the major steps you'll follow to create EDM_DEMO:

- Start **PartMaker** and open a new **Face Window**
- Specify part boundaries
- View tool information for the part
- View cycle information for the part
- Select a material for the part
- Set up **PartMaker** defaults
- Create Geometry
- Define profiles for Inside Curves
- Define profiles for Tapered circle
- Define profiles for Outside curves
- Select a post processor
- Generate a process table containing burning data
- Simulate burning of the part
- Generate an NC program for the part
- Print the NC program for the part

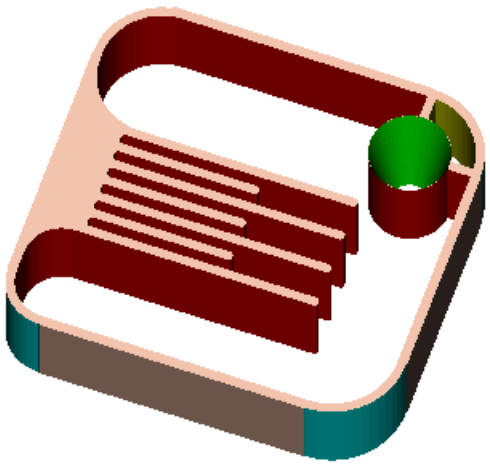
Inch Units



Metric Units



Finished Part



Copyright @2014 Delcam Ltd

Getting Started

The components of the completed tutorial part can be found in your PartMaker EDM directory under the following file names:

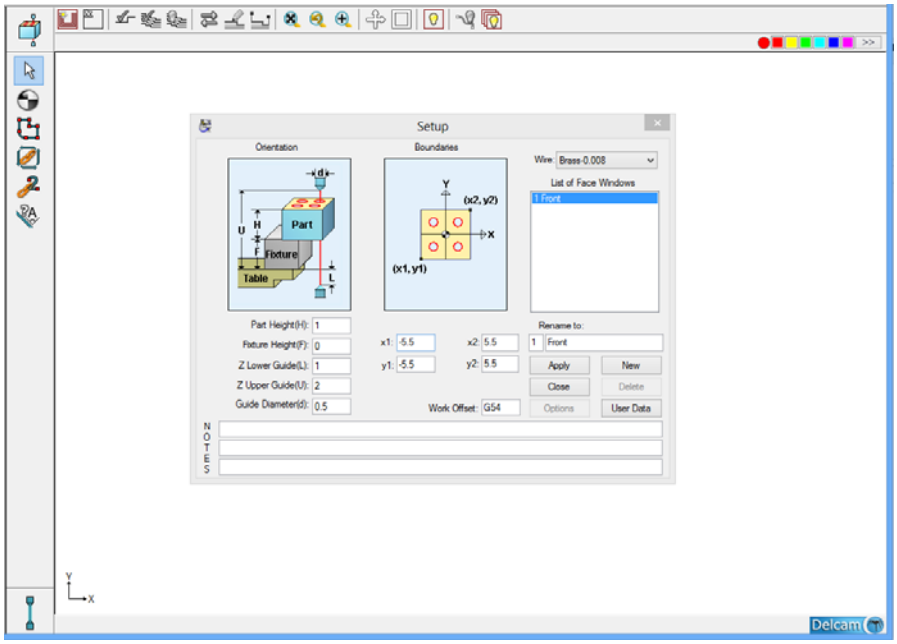
Completed Job File:	C:\PartMaker\PM-EDM\edm_demo.JBE
Completed Tools Data Base:	C:\PartMaker\PM-EDM\edm_demo.TDE
Completed Cycles Data Base:	C:\PartMaker\PM-EDM\edm_demo.CDE

EDM_DEMO has four groups of profiles to be burned. Each group has a cycle that records the burning strategies of each feature. Operations are shown for each group that is used for burning the part.

You may find it useful to refer to the print on the previous page as you create EDM_DEMO. We have provided the finished part.

Start PartMaker WireEDM

1. In the Windows Start button, locate the **PartMaker** Group and click the **PartMaker EDM** application icon or just double click the WireEDM on your Windows desktop.
2. PartMaker will open and by default the **Setup** window will be open.

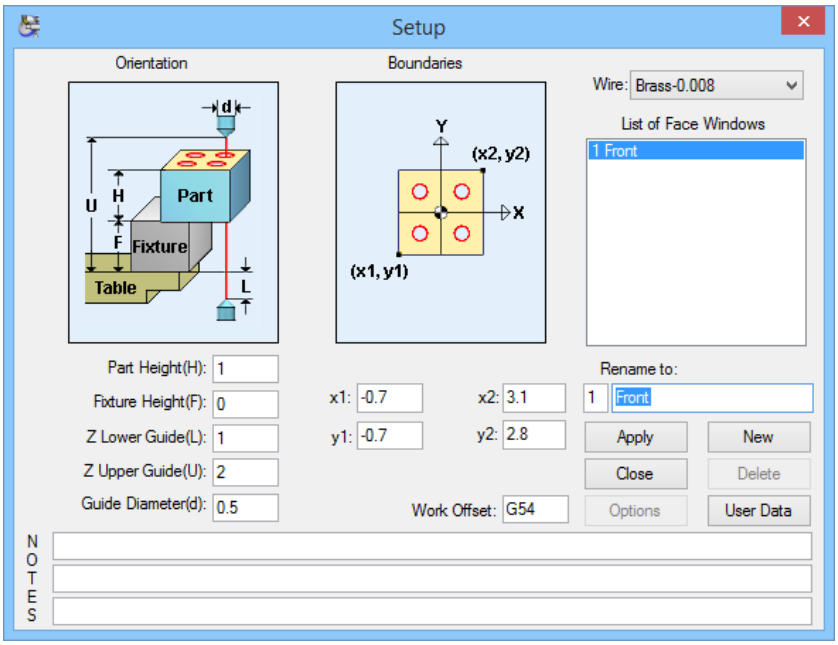


A **Face Window** is the work area in which you define geometry and part features. The main area of a **Face Window** displays the part boundaries; the left side of the window is used to display *group icons*, which show the cycles for machining a part.

Note: *If a dialog or window obscures the* ***Face Window*** *at any time during the tutorial, you can make the* ***Face Window*** *active again by clicking its title bar.*

Set Up Part Boundaries

1 To access the **Setup** dialog choose **Setup** from the **View** menu or click the Shortcut to the Setup Dialog Icon in the upper left hand corner of the Face Window. The **Setup** dialog (inch) appears on top of the **Face Window**.



Tip: *To advance from one numeric entry field to another in a dialog, press the* ***<Tab>*** *key. You can also press the* ***<Tab>*** *key to advance from the X coordinate field to the Y coordinate field when a numeric prompt appears at the bottom of a* ***Face Window****.*

2 Enter the following values next to these options in the dialog. Metric dimensions shown (metric):

Part Height (H)	1.0	(25.0)
Fixture Height (F)	0.0	(0.0)
Z Lower Guide (L)	1.0	(25.0)
Z Upper Guide (U)	1.1	(27.94)
Guide Diameter (d)	0.5	(12.5)
x1	-0.7	(-17.5)
x2	3.1	(78.0)
y1	-0.7	(-17.5)
y2	2.8	(70.0)
Work Offset	G54	

Copyright @2014 Delcam Ltd

The fields in the **Setup** dialog have the following meanings:

Part Height (H): The measured height of the part.

Fixture Height (F): There are cases where the part may be elevated from the table. Insert difference between table and part bottom, this will be especially important for taper cutting.

Z Lower Guide (L): Distance between the machine table and top of the lower guide representing the distance from the top of the table to lower guide.

Z Upper Guide (U): The distance from the table to the bottom of the upper guide, i.e. the position for upper head, measured from top of table to bottom of the upper guide.

Guide diameter (d): This x value is used to set the size of the guides when viewed in PartMaker Simulation.

Boundaries (x1, y1, x2, y2): Describe the size of material you will be working with.

Work Offset: Set this if your machine requires a work offset.

Wire: Select the diameter of wire you are going to use. The wires available in the Setup dialog correspond to those that currently exist in the Tool Database.

3 Click **<Apply>**.

4 Click the **<Close>** box in the lower-right corner of the dialog to return to the **Face Window**.

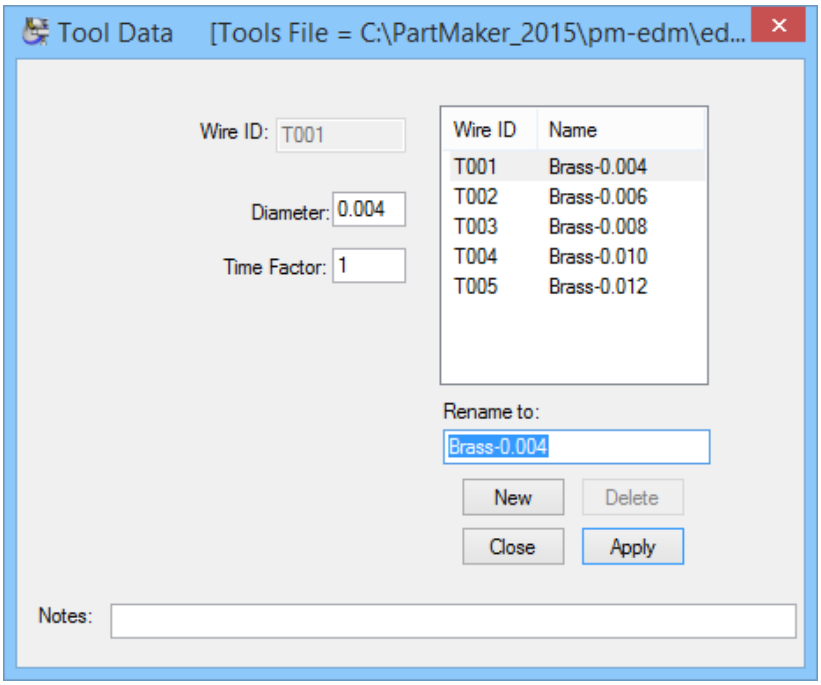
5 Click anywhere in the **Face Window** to make sure it is active.

View the Tool Information

The **Tools Database** allows you to keep track of your current wire inventory. **PartMaker** uses geometric and cutting characteristics saved with each wire in the database for automatic cutting condition creation and automatic cycle time calculation.

You can view the wires used for EDM_DEMO in the Tools dialog by choosing **Tools** from the **ToolMinder** menu. The *Tool ID* specifies the ID number of the currently selected wire in PartMaker's Tool Database.

Note: *The wires used in this chapter are for the tutorial only. As you begin to use* ***PartMaker****, you will add wires to the tools database.*



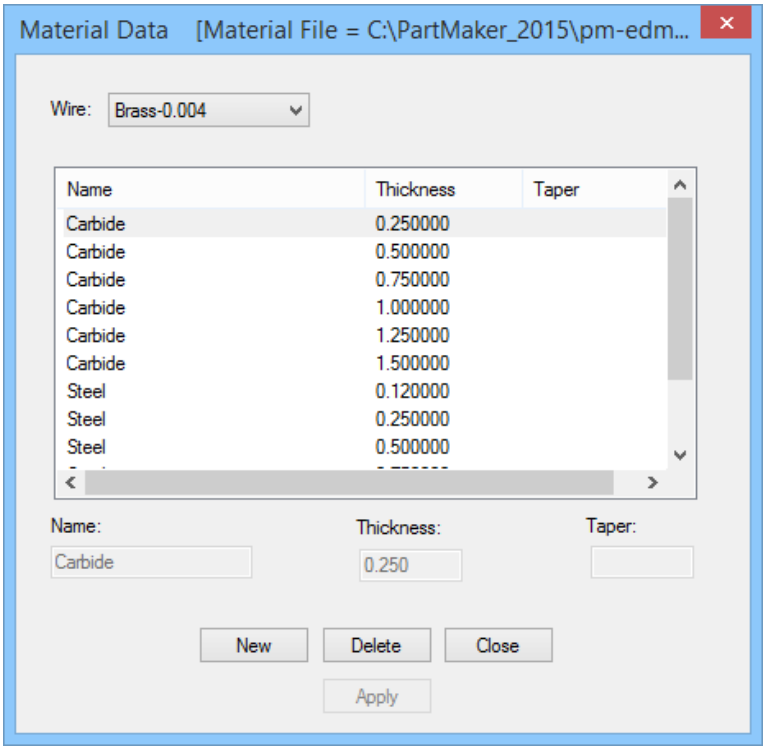
See Chapter 2 for more information using the Tools Database in PartMaker EDM.

Copyright @2014 Delcam Ltd

Select a Material for the Part

PartMaker has a materials database from which you can select a variety of material types compatible with various wire types and diameters. A material database allows you to store your shop's accumulated materials data. **PartMaker** uses this data for the automatic calculation of cycles and cutting technologies.

You can view information about the material used to create the EDM_DEMO part by choosing **Material** from the **ToolMinder** menu.



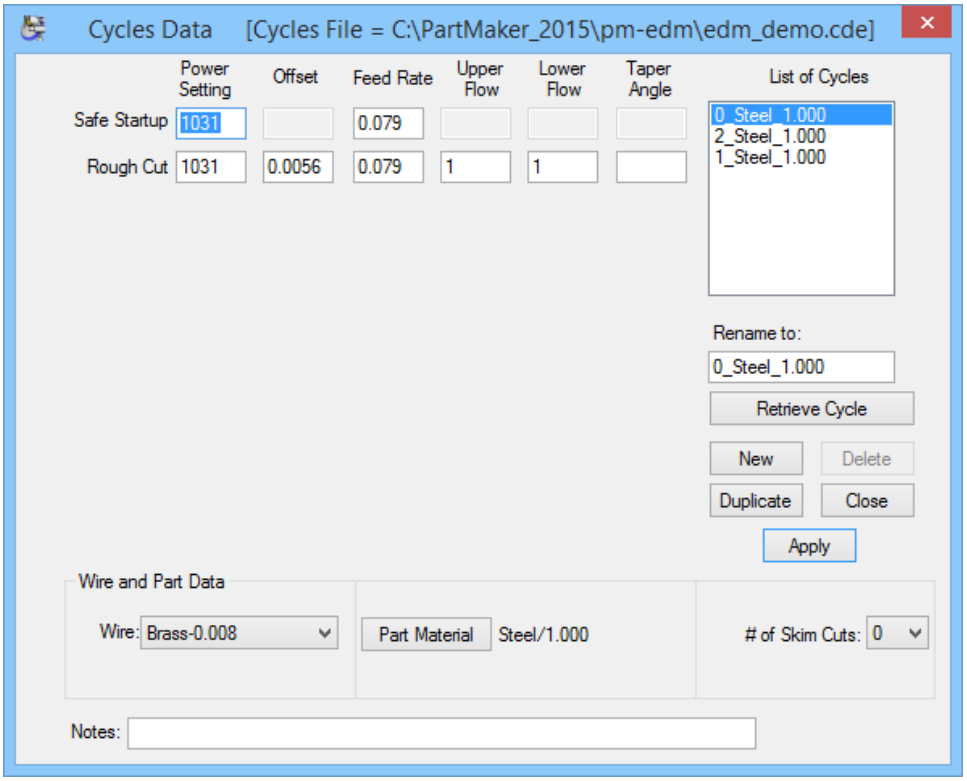
Note: *When exiting a dialog, the* ***Face Window*** *loses focus and its title bar becomes grey. You may find a number of menu commands being disabled (gray) if a front window is not active (not in focus). To make sure the Face Window is the active window, click its title bar.*

See Chapter 2 for more information using the Material Database in PartMaker EDM.

View the Cycle Information

PartMaker lets you combine a series of roughing and finishing operations into a single entity called a cycle. The Cycle information you create is saved in a *Cycles Database* for easy retrieval; reusing cycle information is as easy as opening a cycles file from the cycle's database.

You can view the cycles by choosing **Cycles** from the **ToolMinder** menu.



See Chapter 2 for more information using the Cycles Database in PartMaker EDM.

Copyright @2014 Delcam Ltd

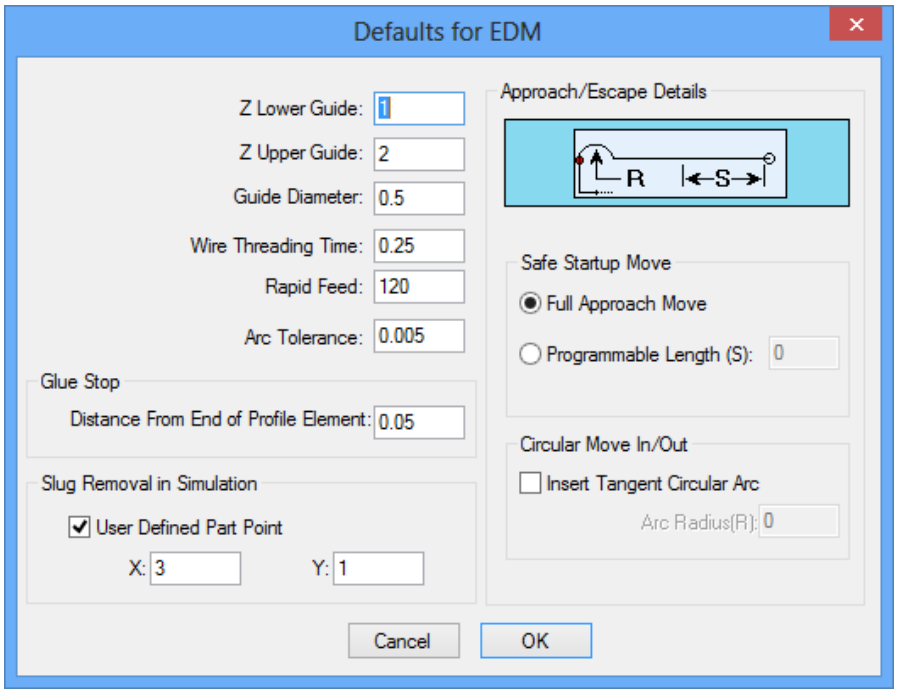
Setting Defaults

In this section, you'll set the Defaults dialog for your part. Since the tutorial is designed for a part using the inch mode, you must be sure that **PartMaker** is in the correct mode. To do so:

1. Choose **<Preferences>** from the **View** menu to display the Preferences dialog
2. Click the **<Inch>** button if it isn't already selected
3. Click **<OK>** to close and save your changes.

Default settings are used in the program generated for a part. These parameters are used for each new group (cycle) you create. You can change Process Parameters at any time in the Process Parameters dialog (**Process Parameters** command in the **Job Optimizer** menu); you can change Part Boundaries at any time in the Setup dialog (**Setup** command in the **View** menu).

Select **<Defaults>** from the **Job Optimizer** menu to display the Defaults dialog.



Z Lower Guide. Specifies the default distance between the top of the table and the lower guide.

Z Upper Guide. Specifies the default distance between the top of the table and the upper guide.

Guide Diameter. Specifies the default diameter of the guide in simulation.

Wire Threading Time (min). Specifies the default wire threading time in minutes to be used in time calculations.

Rapid Feed. Specifies the default rapid feed rate to be used in time calculations.

Arc Tolerance. Specifies the maximum distance between the arc and any line used to approximate this arc. Used in 4-axis jobs.

Glue Stop: Distance From End Of Profile Element. Specifies the default glue stop point distance from the end point of the Profile Element.

Slug Removal in Simulation: User Defined Part Point. Specifies a point that belongs to the finished part. Any slug created during simulation that does not contain the User Defined Part Point would be removed.

X. Specifies the horizontal coordinate of the User Defined Part Point.

Y. Specifies the vertical coordinate of the User Defined Part Point.

Safe Startup Move:

Full Approach Move: Specifies that the length of the safe startup move equals to the length of the first move of the tool path.

Programmable Length (S): Allows the user to specify the length of the safe startup move.

Tip: *The default button in all* ***PartMaker*** *dialogs is the heavily-outlined button (****<OK>*** *or* ***<Apply>****, for example). You can use the default button by clicking it or by pressing the* ***<Enter>*** *key.*

Copyright @2014 Delcam Ltd

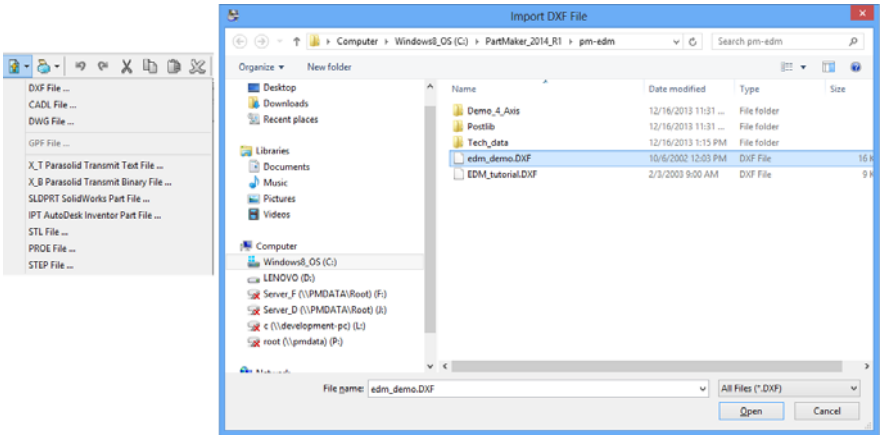
Create Geometry

There are two methods of creating Geometry in **PartMaker**. You can either import Geometry from a CAD file or define Geometry using **PartMaker's** CAD mode. In this tutorial, geometry will be imported from an existing DXF. For more information on creating part geometry, please see the section titled **Using Graphics Icons in PartMaker CAD Mode** in Chapter 2 of this guide.

Import Geometry from a CAD system via DXF file.

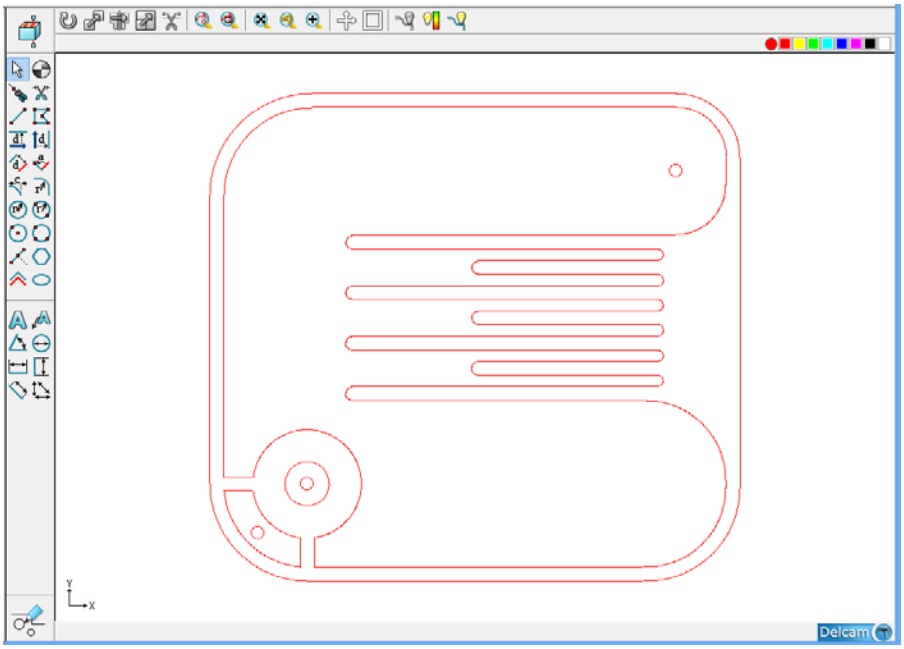
To import geometry for EDM_DEMO, do the following:

1. Click the CAD/CAM switch to go to CAD Mode
2. Choose **Import, DXF File** from the **File** menu to display Import DXF File dialog.



3. Locate and select **edm_demo.dxf**, then click **<OK>**.

Your screen should appear as shown below:



Now that you have a drawing in the **Face Window**, take a few minutes to explore some of the basic features of **PartMaker** that you will be using.

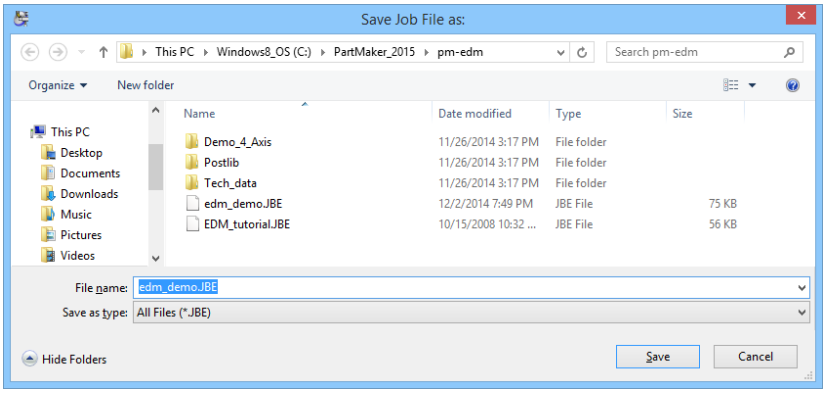
From the **View** menu, you have the following **Zoom** options available:

	Select **Full View** to expand the drawing to fill the screen.
	Select **Previous View** to return the screen display to the previous setting.
	Select **Zoom In** and use any mouse button to drag opposite corners of a rectangle that you want to magnify.
	Select **Zoom Out** to shrink the drawing on the screen
	Select **Continuous Zoom** to magnify the drawing at a steady rate. You can do this by pressing **<Ctrl>** and the **<+>** or **<–>** key
	Select **Continuous Pan** to move the drawing to the left or right at a steady rate. You can do this by pressing **<Ctrl>** and the directional keys.
	Select **Zoom to Fit** to create the smallest possible window that will contain all of the geometry, dimensions and notes in the current window.
	Select **Zoom All** to create the smallest possible window that will contain all of the geometry, dimensions, notes and the boundaries in the current window
	Select **Show Axes** to show the current X0 Z0 location. Select **Hide Axes** to turn them off again.
	Show and **Hide Boundaries** is used to show the boundaries of the rough stock being turned.

Depending on your needs, any or all of these display options may be activated at any time during the drawing (CAD) or programming (CAM) process.

Save Your Work

1. Choose **Save Job File As...** from the **File** menu to display the **Save Job File as:** dialog.
2. Enter the name **EDM_DEMO** in the space provided and click **<Save>**.

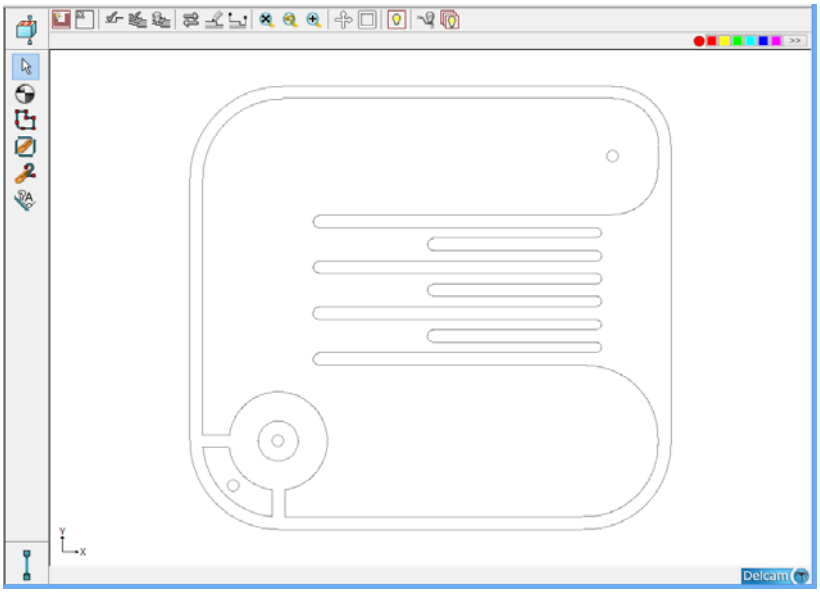


Copyright @2014 Delcam Ltd

Creating the tutorial part in PartMaker CAM Mode

Now that you have the geometry for TUTORIAL, you are ready to begin working in **PartMaker** CAM mode. To do so, you must first be in CAM mode. You are in CAM mode when the CAD/CAM switch in the lower left-hand corner of the screen appears as the one to the left.

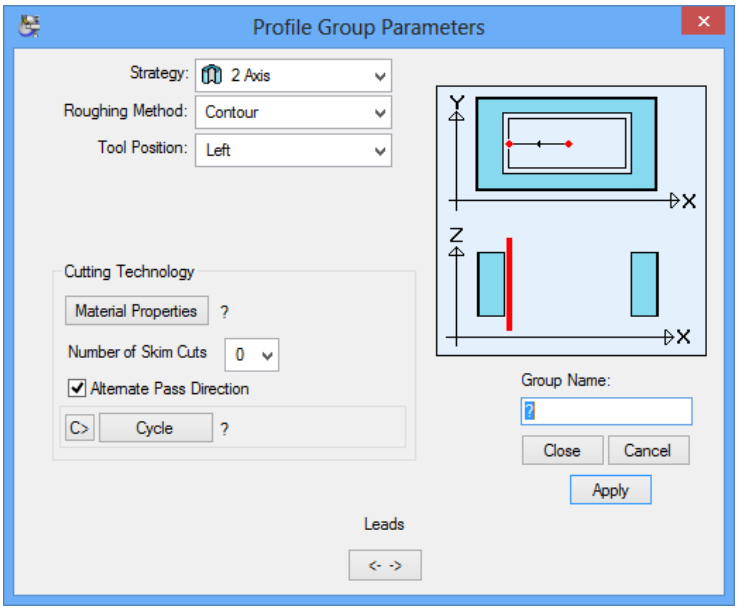
Before proceeding, your **Face Window** should appear as the one below, with your geometry appearing as a gray outline on the screen:



Define Group 1 Profiles for Inside Contour

The first operation in this tutorial is inside contour.

1. Make sure you are in the CAM Mode
2. Choose red from the color bar
3. Choose **New Profile Group** to display the Profile Group Parameters dialog.



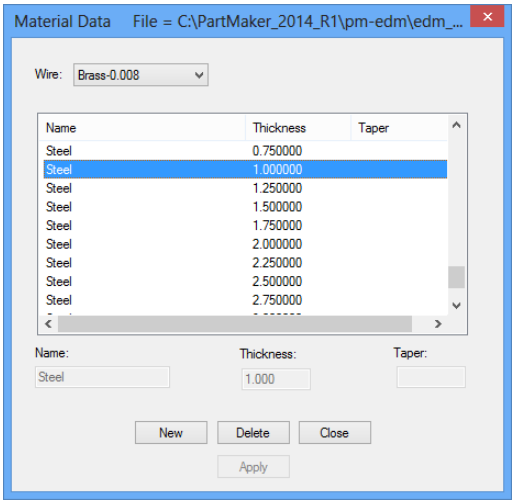
4. Choose **2 Axis** from the **Strategy** drop-down menu.

5 Choose **Contour** from the **Roughing Method** drop-down menu.

6 Choose **Left** from the **Tool Position** drop-down menu

Alternate Pass Direction: When your part requires multiple passes for accuracy or finish you can define wire motion in two different ways. All passes start at the start point and burn to the end point. You may also select Alternate Pass Direction. Doing so will enable you to start at the start point and burn to the end point and then from there, burn back to the start point. This will save you time in cutting the wire and repositioning to the start point.

7 Click the <Material Properties> button. Once you activate the Select Materials button it will take you to the Material Library. Browse through the library and select the material type and height you are going to machine.



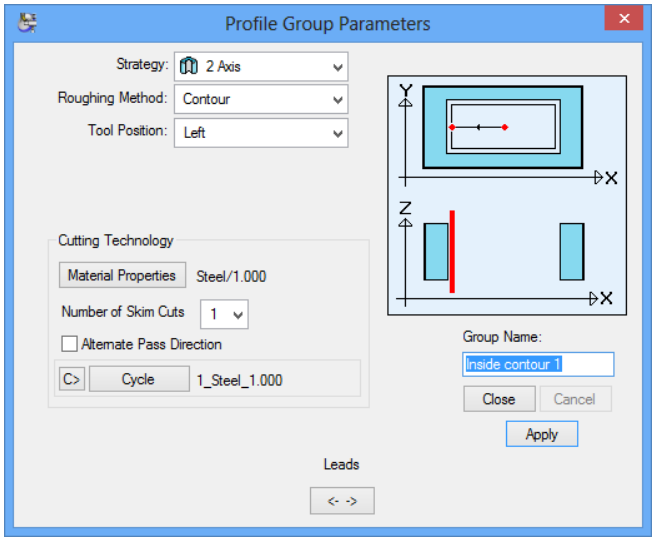
8 For this job select Steel 1.0 (25.00) and click **<OK>**.

9 Choose 1 from the **Number of Skim Cuts** drop-down menu.

10 Click the **Cycle** button to check the cycle database for all possible cycles that match your Profile Group criteria. You can click the **<C>** the see the Cycle that was selected.

11 In the Group Name box, enter: Inside Contour 1.

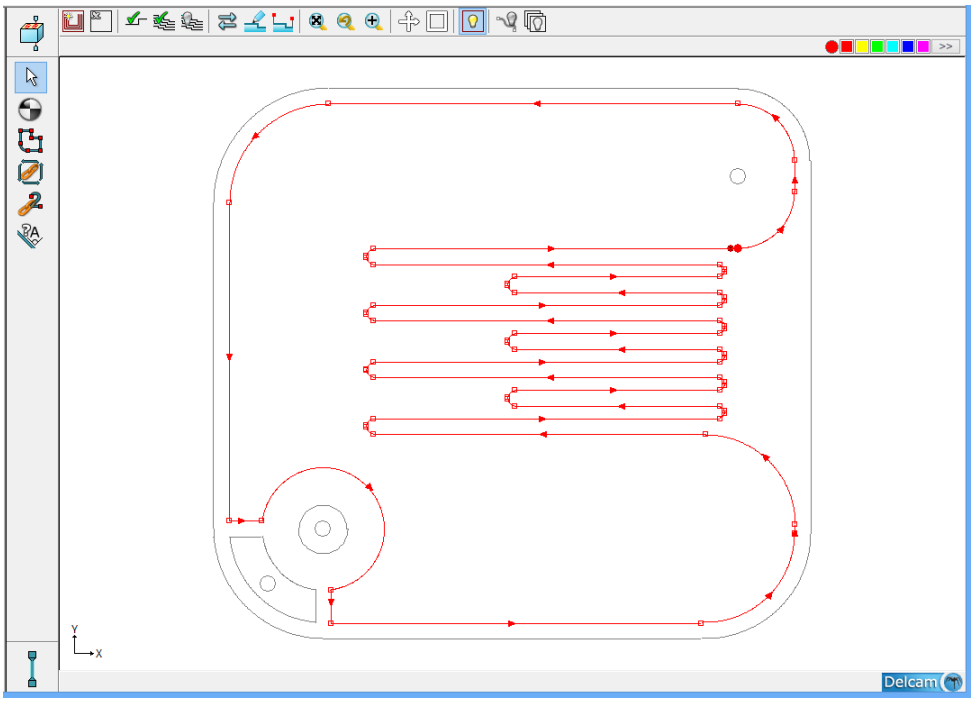
12 Once you are satisfied that your completed **Profile Group Parameters** dialog appears as shown below, click **<Close>**:



Copyright @2014 Delcam Ltd

Create Profiles Using the Chain Geometry Icon

1 Select the **Chain Geometry** icon on the left side of the Face Window. Click the cursor on the location marked by the arrow on the screen below.

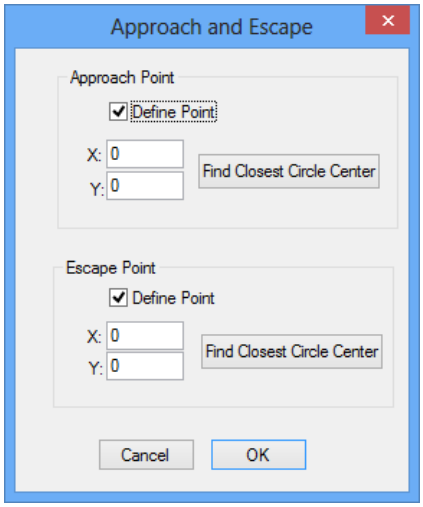


Approach and Escape

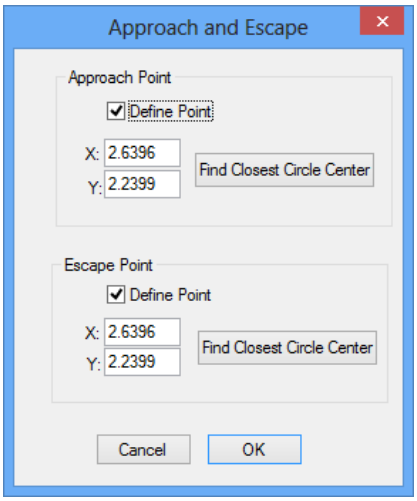
When starting or ending a profile within the work piece, there is usually a drilled hole or other feature through which to thread the wire. PartMaker will automatically select the closest circle center to start or end the tool path.

1 Click the **Selection** Icon and select any part of the tool path you just created.

2 Click on the **Approach and Escape** button and the dialog as shown below will open:



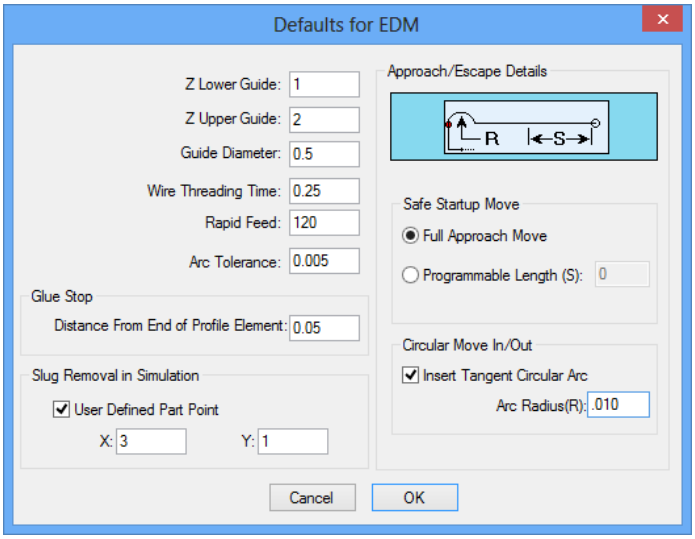
3 Click the **Find Closest Circle Center** button to automatically start the tool path at the center of the closest circle drawn in the CAD mode. If you have not created a circle, but you know the coordinates of the circle, you can enter them by clicking the **Define Point** box and entering the X and Y Coordinates in the respective fields. Once your completed **Approach and Escape** dialog appears as shown below, click **<OK>**.



Approach and Escape with Tangent Arc

In addition to the simple linear APPROACH / ESCAPE move, PartMaker provides the ability to add a small tangent arc to this move to eliminate any 'witness' marks that may be created by the contact of the wire to the finished profile.

This TANGENT ARC definition is accessed through the JOB OPTIMIZER > DEFAULTS menu item, which opens the dialog shown below:



In the lower right hand corner of this dialog are the parameters for the Circular Move In / Out utility.

By checking this box, you instruct PartMaker to add a small arc move at any location where an approach / escape move contacts the finished profile.

Note that the radius of this arc should be small enough that the wire will burn the entire arc material away and not leave a 'slug' to drop down into the lower guide.

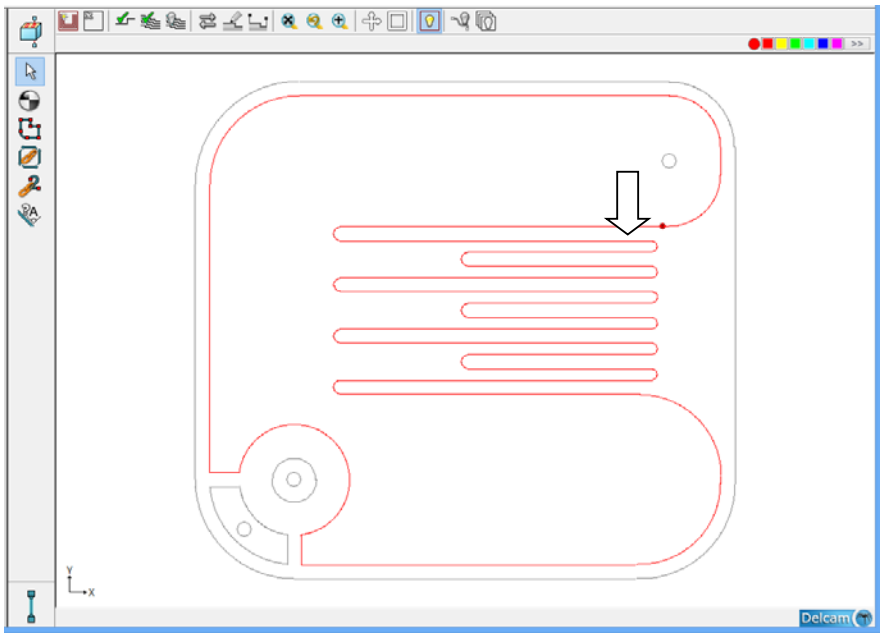
The arcs created by this utility will not be shown in verification, but will appear in the final output code.

Copyright @2014 Delcam Ltd

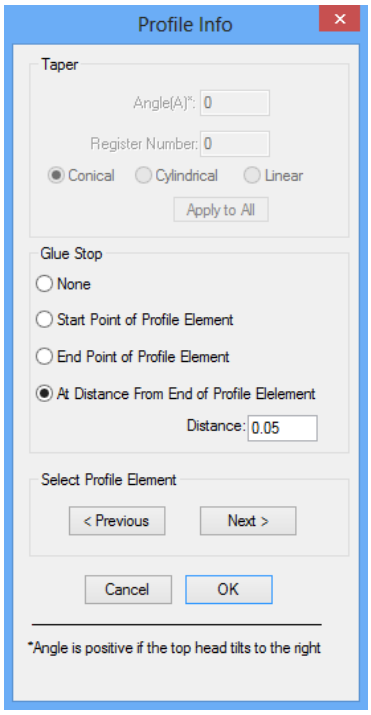
Glue Stop

1 Click the **Profile Info** Icon.

2 Click the cursor on the location marked by the arrow on the screen below.

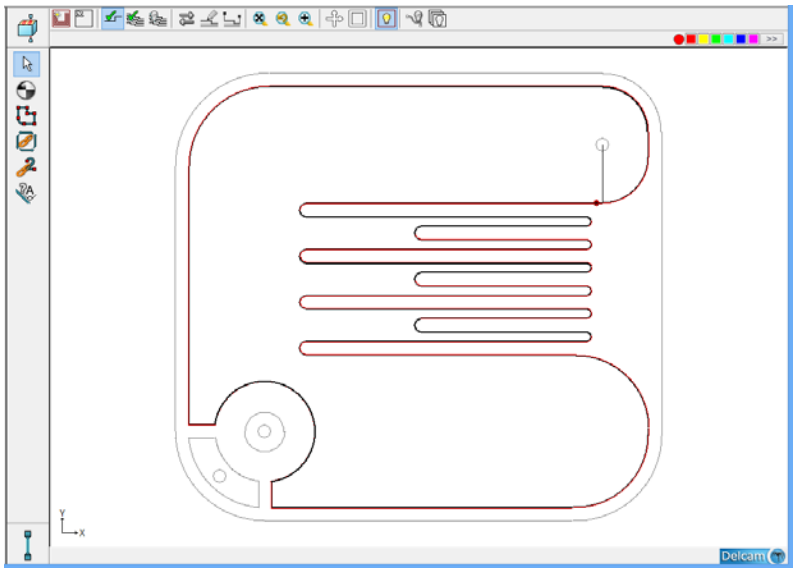


3 Now the Profile info dialog will appear. Select the radio button for **At a Distance From End of Profile Element.**



Verify Tool Path

1 Select the **Verify Work Group Tool Path** command from the **Part Features** menu. You will see a picture as below.



2 Select the Hide Every Tool Path command from the Part Features menu.

Copyright @2014 Delcam Ltd

Define Group 2 Profiles for Tapered Circle

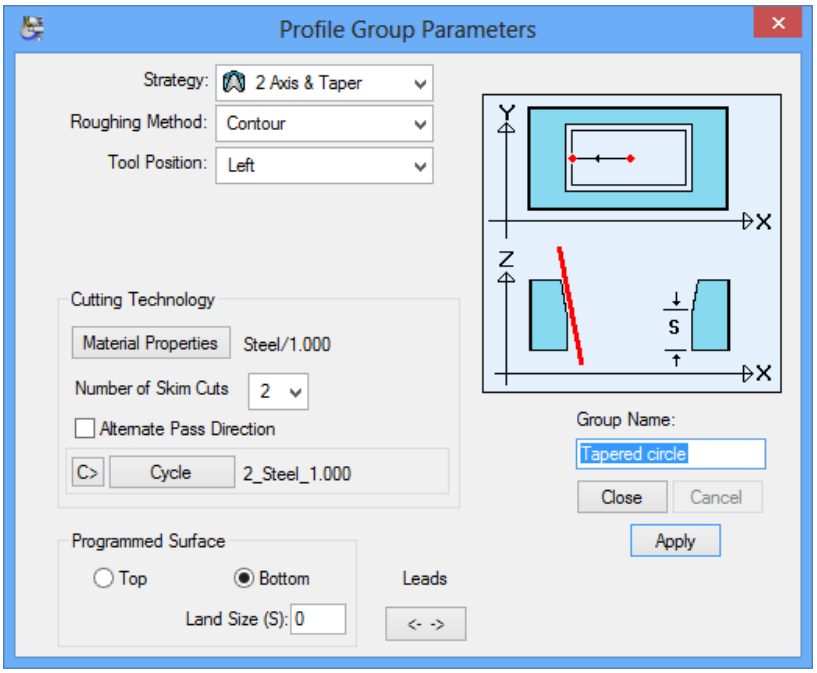
1. Choose green from the color bar such that green appears as the Sample Color.

2. Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog.
3. Choose **2 Axis & Taper** from the **Strategy** drop-down menu.
4. Choose **Contour** from the **Roughing Method** drop-down menu.
5. Choose **Left** from the **Tool Position** drop-down menu
6. Choose **2** from the **Number of Skim Cuts** drop-down menu.

Note: *The material has already been selected from creation of the previous part feature.*

7. Click the Cycle button to check the cycle database for all possible cycles that match your Profile Group criteria.
8. In the **Group Name** box, title this process **Tapered Circle**.

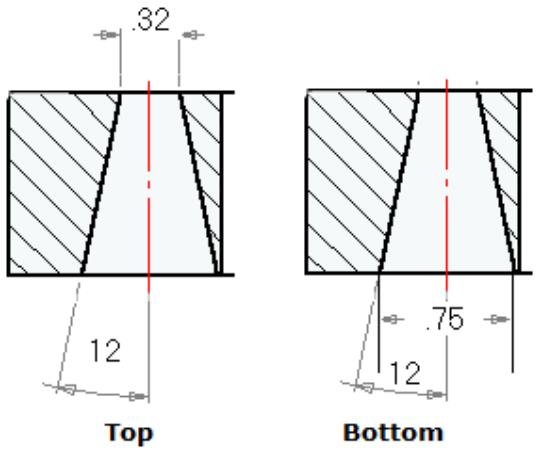
   Your completed dialog should appear as shown below:



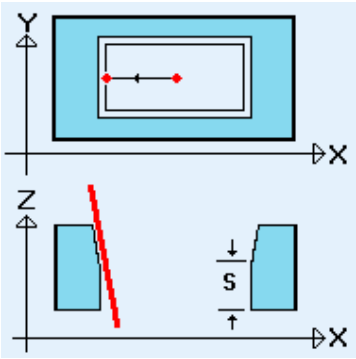
Before clicking **<Close>** please take a moment to learn more about 2-axis taper cutting with PartMaker.

Taper Cutting

Programmed Surface in the **Profile Group Parameters** dialog is the location of the geometry you have created. If you are creating a tapered cone as shown below, you will draw a circle. You must indicate whether that circle size corresponds with the geometry on the top of the part or on the bottom of the part.



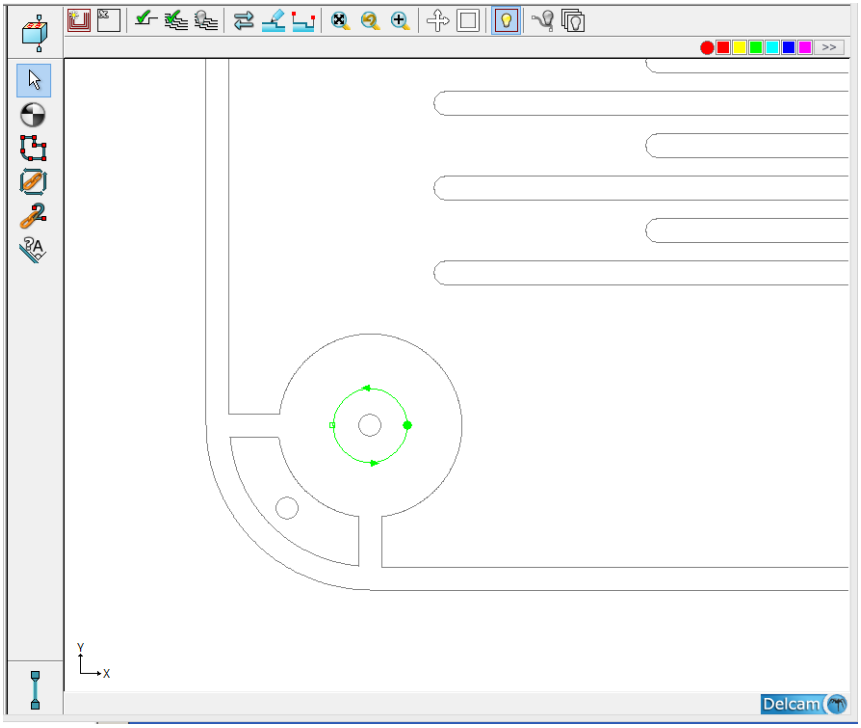
The last field in the **Profile Group Parameters** dialog is the **Land Size**. This is only used when you are cutting a taper and a land. A land is the result of a taper that does not continue through the height of the part.



Copyright @2014 Delcam Ltd

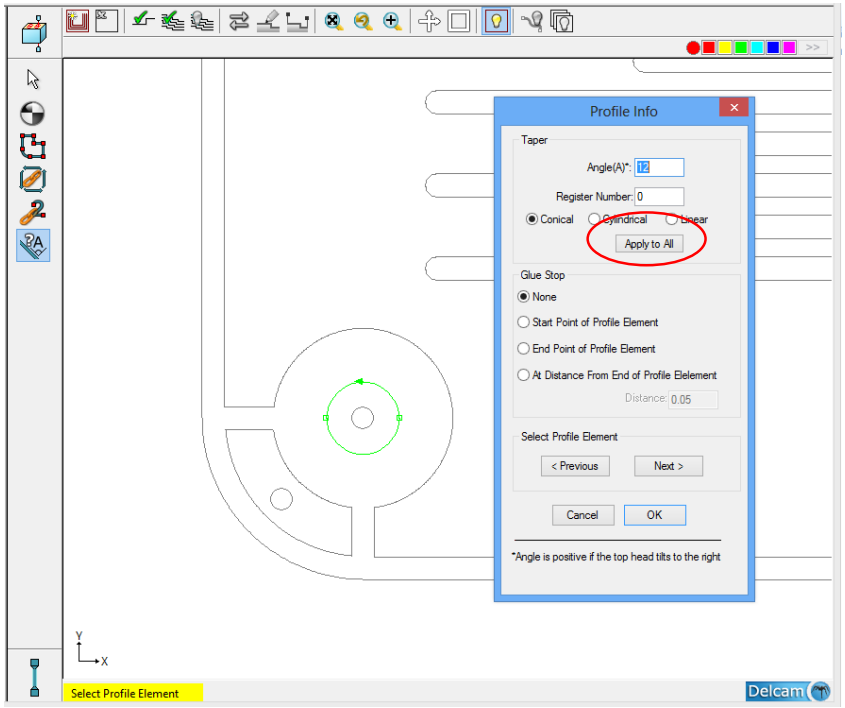
Creating Tapered Profiles Using the Chain Geometry Icon and Profile Element Data

1. Select the Chain Geometry icon on the left side of the **Face Window**.
2. Click your cursor to chain the profile that you want to taper, as shown below:



3. Select the **Check Profile Info Dialog Icon** from the left side of the window.
4. Double click the question mark on the tool path you have just created.
5. Complete the **Profile Info** dialog with the appropriate parameters for your taper.
   - **Angle (A)** - The angle of the taper. This could be positive or negative depending on the programmed surface.
   - **Register Number** – Allows a specific command to be added to a specific line of NC code.
   - **Conical** - Applies only to a circular arc. The center of the circular arc remains unchanged for the top and bottom profiles.
   - **Cylindrical** – Applies only to a circular move.
   - **Linear** – When two adjacent elements have a different angle, choosing a linear taper type will smooth the transition between them.
   - **Glue Stop** – Select Glue Stop if you would like to insert a stop at a specific place on the profile.
6. Change the Taper Angle to 12 degrees and select <**Apply to All**>

7 Leave Glue Stop set to **None**.



8 When your Profile Info dialog appears as shown above, click **<OK>**.

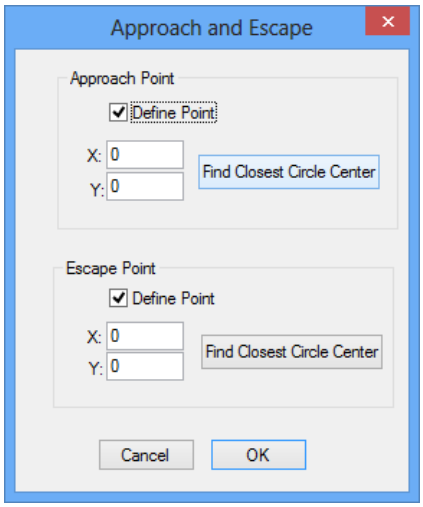
Note: *There will be Profile Info for each individual segment of your tool path. When PartMaker first prompts you for the Profile Info it is a universal prompt, applying to the entire tool path. This will apply to all segments of the tool path. If you have different angles within the same tool path you will go back later and change the Profile Info where appropriate by using the Profile Info icon.*

Approach and Escape

When starting or ending a profile within the work piece, there is usually a drilled hole or other feature through which to thread the wire. PartMaker will automatically select the closest circle center to start or end the tool path.

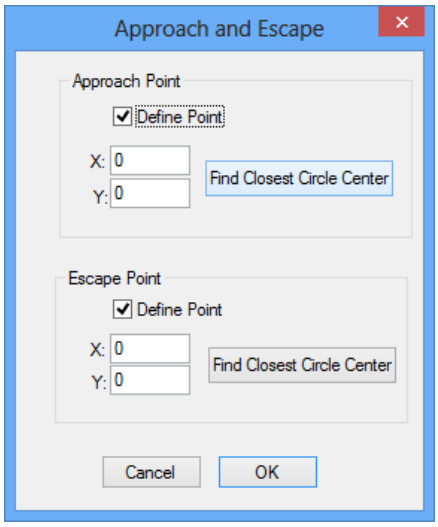
1 Click the **Selection** Icon

2 Double click on any part of the tool path you just created. You will see the Approach and Escape dialog open as shown below:



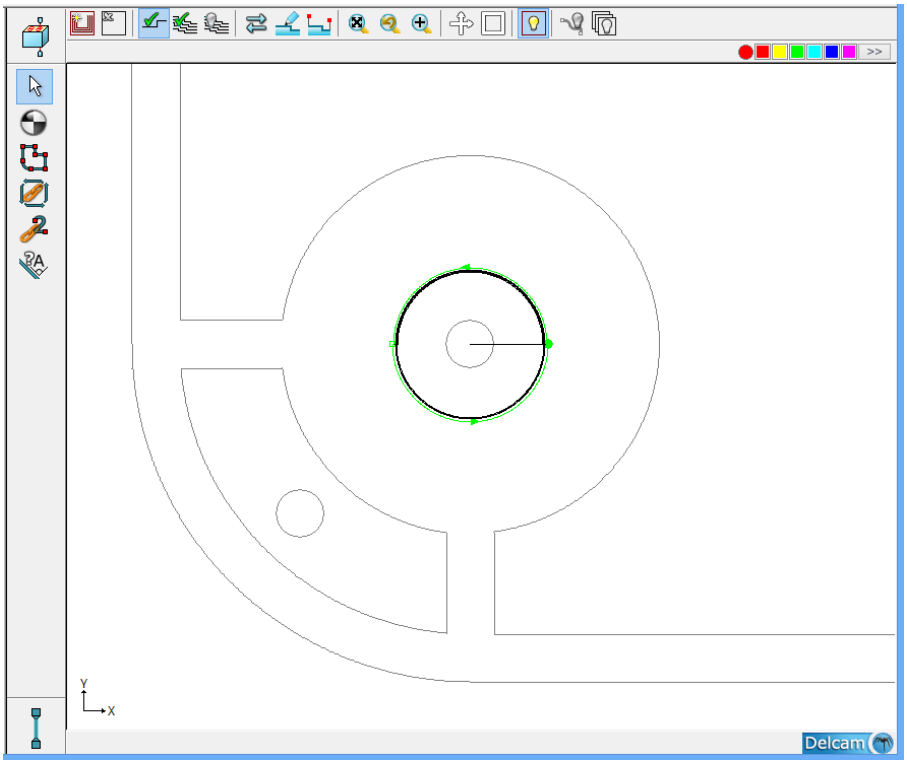
Copyright @2014 Delcam Ltd

3 Click the **Find Closest Circle Center** button to automatically start the tool path at the center of the closest circle drawn in the CAD mode. If you have not created a circle, but you know the coordinates of the circle, you can enter them by clicking the **Define Point** box and entering the X and Y Coordinates in the respective fields. Once your completed **Approach and Escape** dialog appears as shown below, click **<OK>**.



Verify Tool Path

1 Select the **Verify Work Group Tool Path** command from the **Part Features** menu. You will see a picture as below.



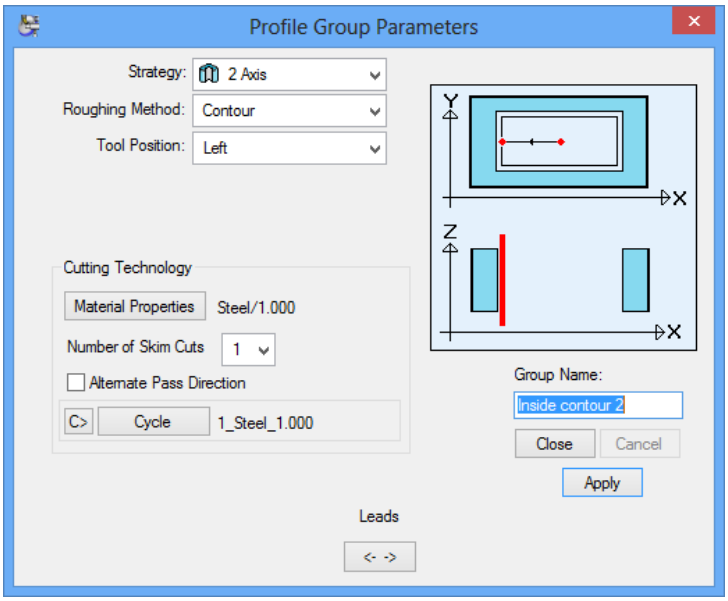
2 Select the Hide Every Tool Path command from the Part Features menu.

Define Group 3 Profiles for Slot

1. Choose Yellow from the color bar such that yellow appears as the Sample Color

2. Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog
3. Choose **2-axis** from the **Strategy:** drop-down menu
4. Choose **Contour** from the **Roughing Method** drop-down menu
5. Choose **Left** from the **Tool Position** drop-down menu
6. Choose **1** from the **Number of Skim Cuts** drop-down menu
7. Click the **Cycle** button to check the cycle database for all possible cycles that match your Profile Group criteria.
8. In the **Group Name** box, call this process **Inside Contour 2**.

   Your completed dialog should appear as shown below:

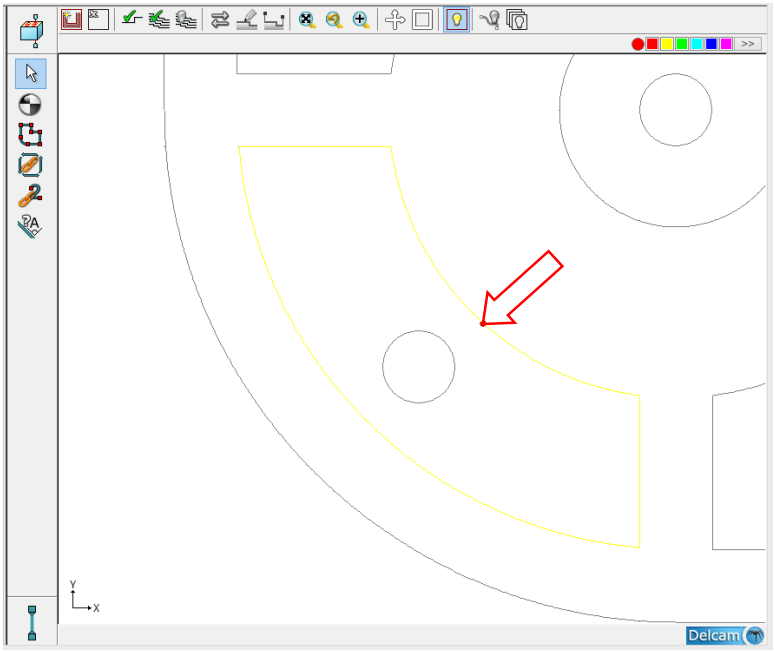


9. When your Profile Group Parameters appears as shown above, click **Close**.

Copyright @2014 Delcam Ltd

Create Profiles Using the Chain Geometry Icon

1 Select the Chain Geometry Icon on the left hand side of the Face Window. Click the cursor on the location marked by the arrow.



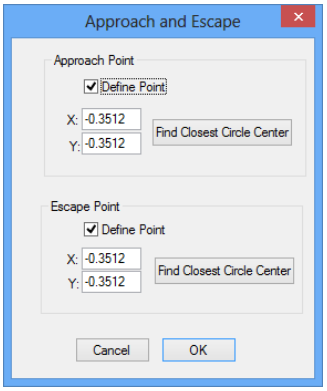
2 There will also be an approach and escape move similar to the first profile created. To do so

a Click the **Selection** Icon

b Double click on any part of the tool path you just created. You will see the Approach and Escape dialog open. Under **Approach Point**.

c Click the **Find Closest Circle Center** button.

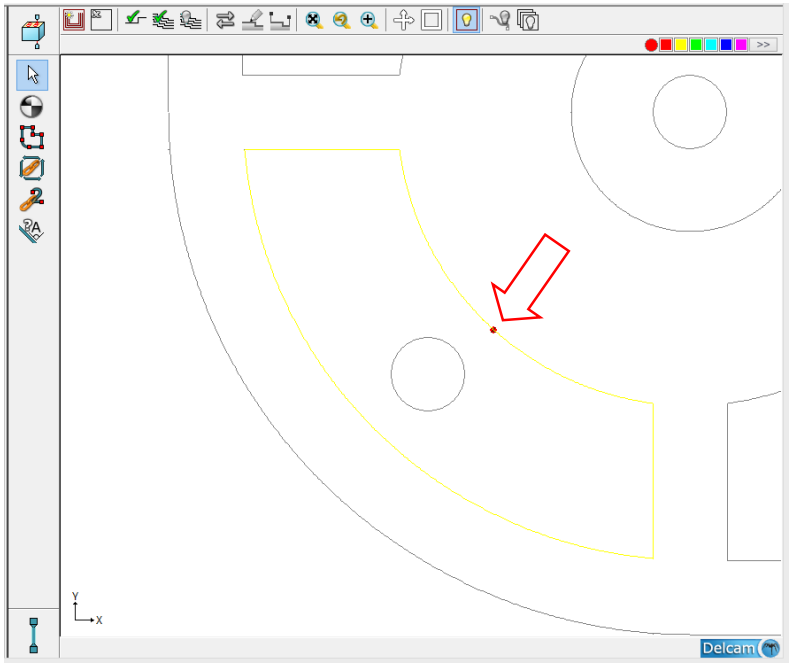
d Once your completed Approach and Escape dialog appears as shown below, click **<OK>:**



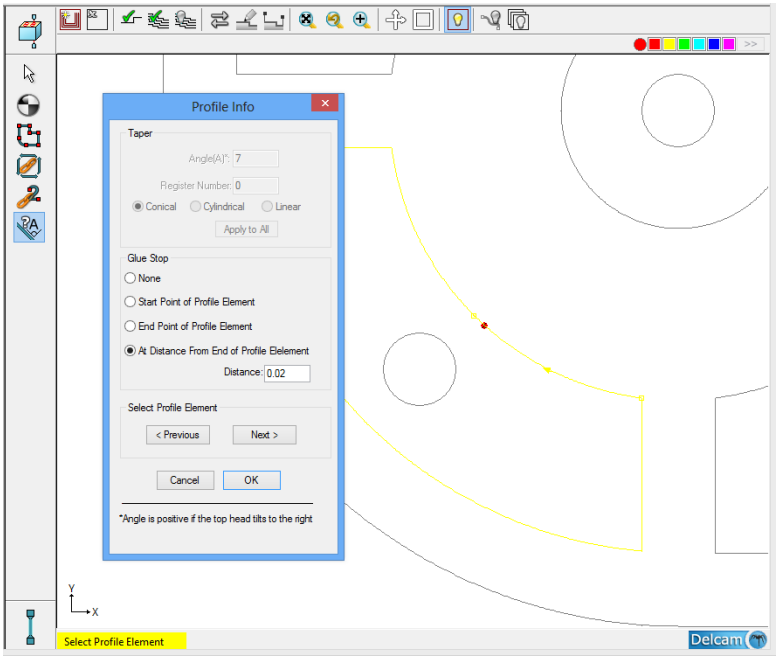
Glue Stop

1 Click the **Profile Info** Icon.

2 Click the cursor on the location marked by the arrow on the screen below.



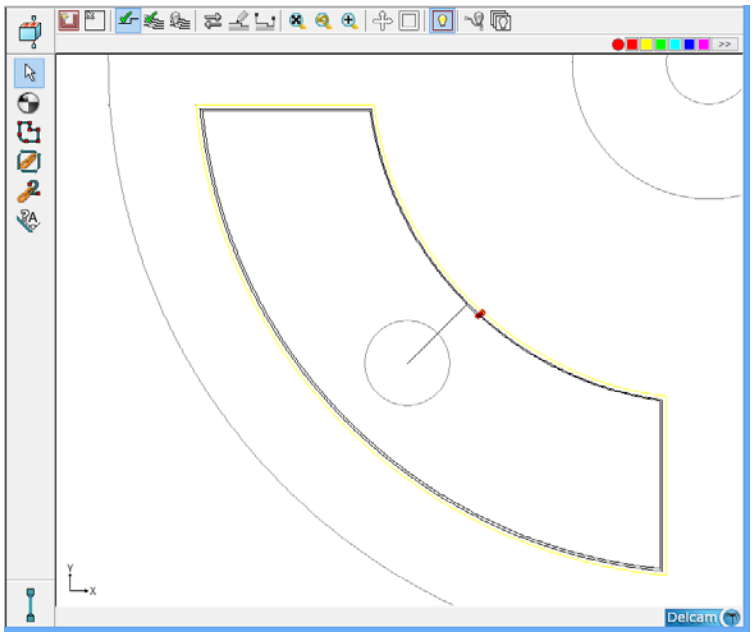
3 Now the Profile info dialog will appear. Select the radio button for "**At a Distance From End of Profile Element**"



Copyright @2014 Delcam Ltd

Verify Tool Path

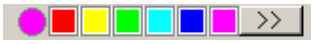
1 Select the **Verify Work Group Tool Path** command from the **Part Features** menu. You will see a picture as below.



2 Select the Hide Every Tool Path command from the Part Features menu.

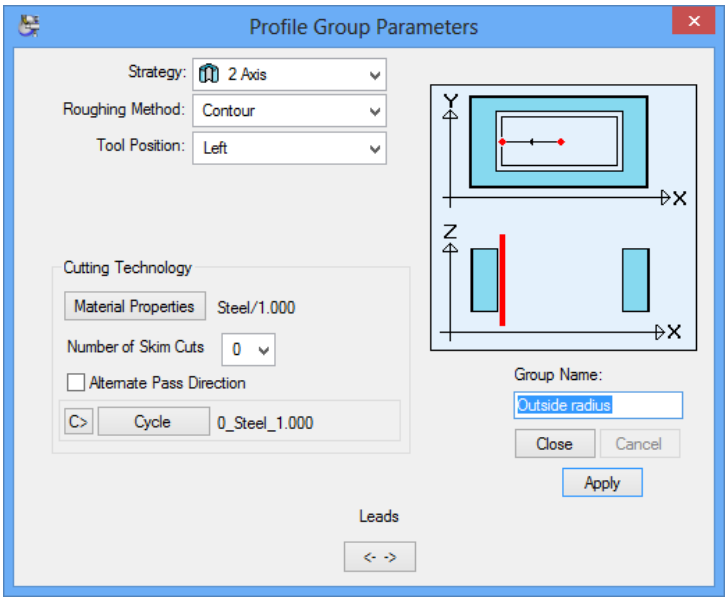
Define Group 4 Profiles for Outside Contours

1. Choose purple from the color bar such that purple appears as the Sample Color



2. Choose **New Profile Group** from the **Part Features** menu to display the Profile Group Parameters dialog
3. Choose **2-axis** from the **Strategy:** drop-down menu
4. Choose **Contour** from the **Roughing Method** drop-down menu
5. Choose **Left** from the **Tool Position** drop-down menu
6. Choose **0** from the **Number of Skim Cuts** drop-down menu
7. Click the Cycle button to check the cycle database for all possible cycles that match your Profile Group criteria.
8. In the Group Name box enter **Outside Radius**.

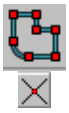
   Your completed dialog should appear as shown below:



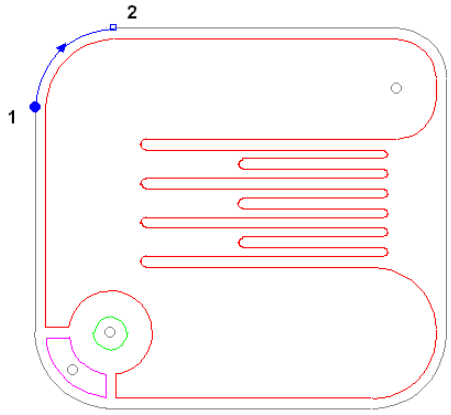
Copyright @2014 Delcam Ltd

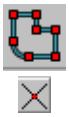
Create Profiles Using the Define Profile Icon

Using the **Define Profile** icon and a Snap Mode, you can define a profile by connecting points defined by X, Y coordinates, polar coordinates, a point on an arc or circle, a screen location, a grid location, a circle center location, the closest intersection and endpoint of a line, or a midpoint of a line.

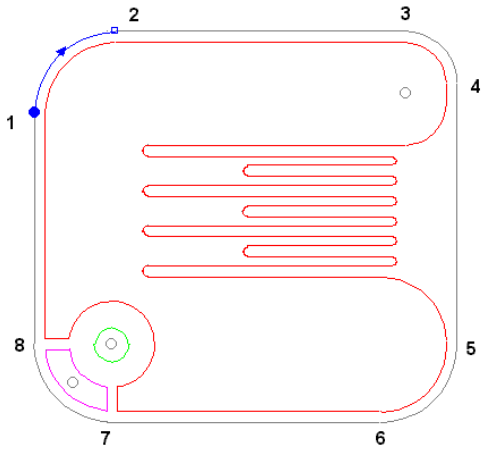


1. Select the **Define Profile** icon on the left hand side of the **Face Window**. The snap modes will appear at the top of the screen.
2. Select the **Closest Intersection** snap mode and then click the cursor on the locations marked by points 1 and 2.



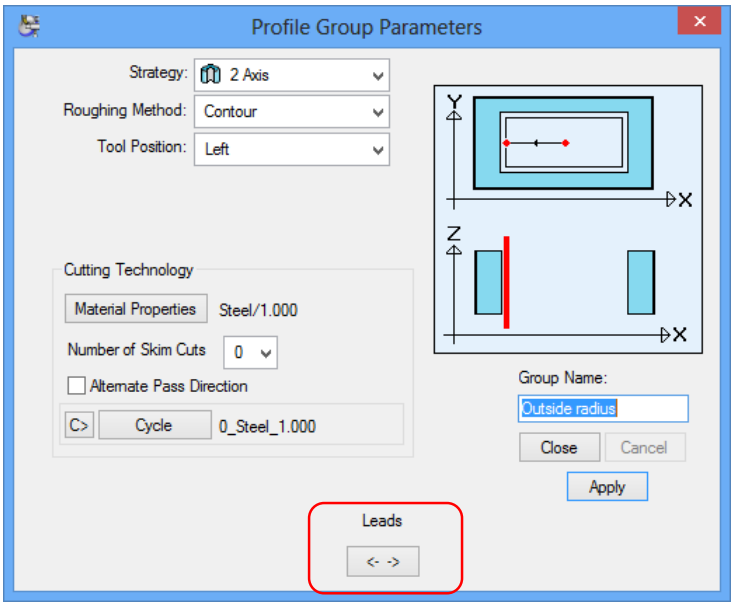


3. Once again, select the **Define Profile** icon on the left hand side of the Face Window. The snap modes will appear at the top of the screen. Select the closest intersection snap mode and click on the locations marked by 3 and 4. After doing so, select the **Define Profile** icon again to select the next corner.
4. Repeat these steps for locations 5 and 6, and 7 and 8.

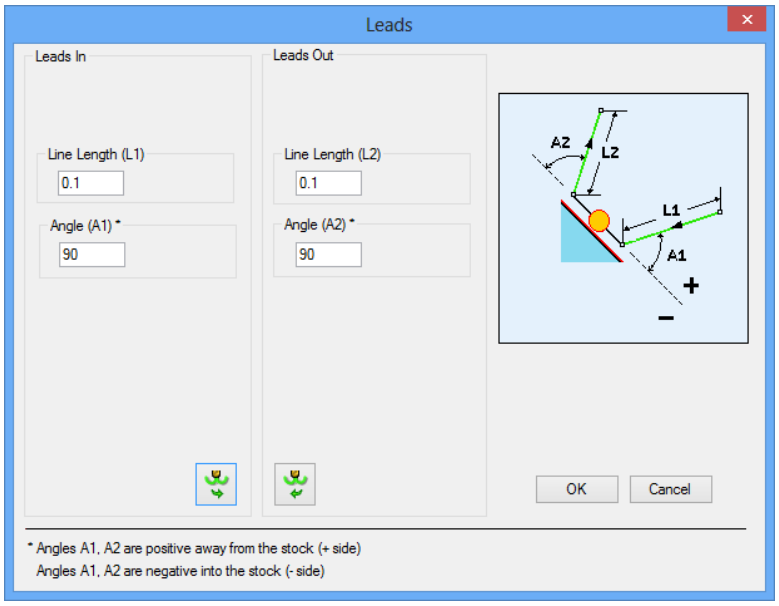


Leads

After selecting all four corners open the profile group by double clicking on the **Outside Radius** process on the right side of the CAM screen then select the **Leads** button.



1 Select Leads to .100 and the Angle to 90.

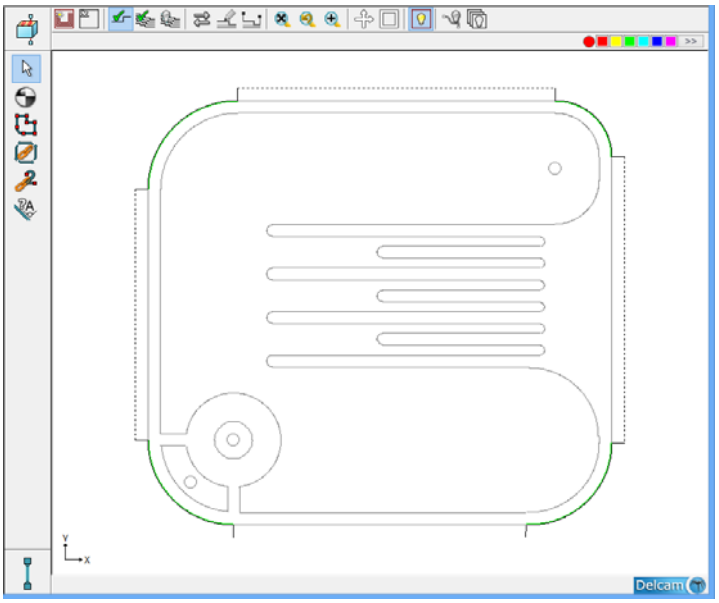


2 Select <**OK**>

Copyright @2014 Delcam Ltd

Verify Tool Path

1 Select the **Verify Work Group Tool Path** command from the **Part Features** menu. You will see a picture as below.



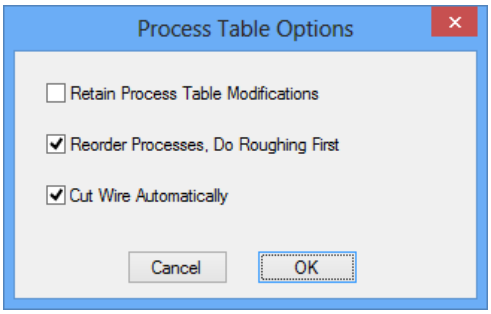
2 Select the Hide Every Tool Path command from the Part Features menu.

Generate a Process Table

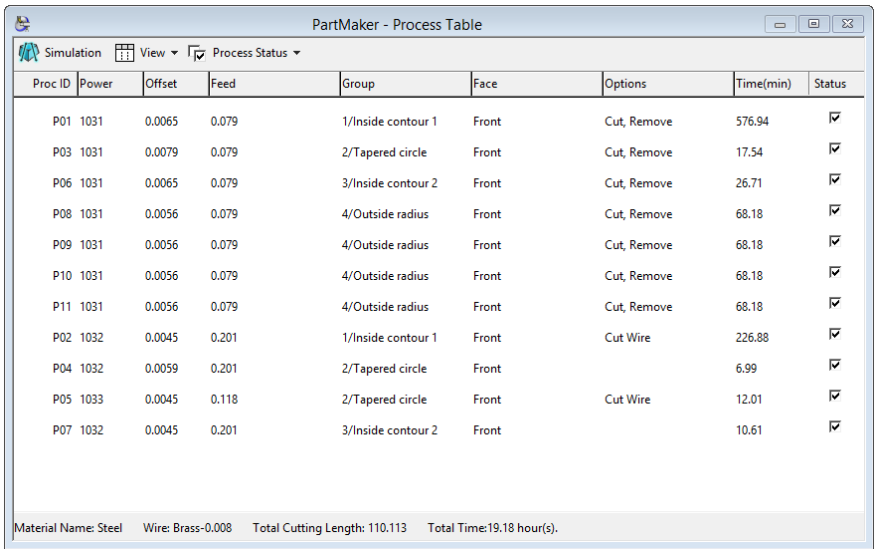
A *process table* shows you all of the processes for a part. Before generating a process table, you can choose to reorder processes to do all roughing operations first, and to cut wire automatically.

When **PartMaker** generates a **Process Table**, all cutting conditions such as feedrate burn time are calculated automatically based on the cycles and material information.

1. Choose **Generate Process Table** from the Job Optimizer menu to display the Process Table Options dialog.
2. Turn on the first check box in the dialog. The dialog box should appear as shown below:



3. Click **<OK>** to display the Process Table window.



Note: *Since we selected* ***Reorder Process, Do Roughing First****, you will see all of the roughing operations in sequence.* ***Cut Wire Automatically*** *was also checked so PartMaker will insert wire cuts after each roughing operation and in between different finish profiles.*

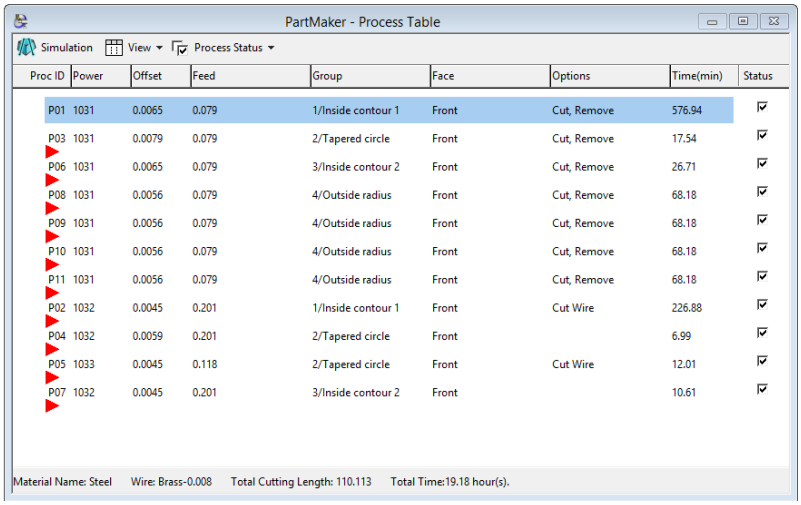
Copyright @2014 Delcam Ltd

Modify the Process Parameters

Process 1 is a roughing process for the ID contour in Edm_demo.jbe. In this section, you'll change the cutting settings, feed rate, and other parameters used for this process.

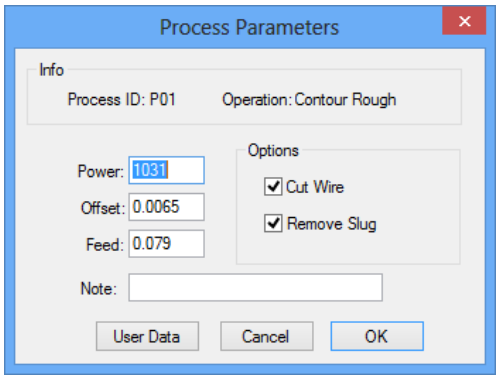
1 Click anywhere on the first process line in the process table window.

A colored rectangle highlights the entire line.



Note: *Colored triangles appear on the left side of the process table window. To move a selected process in the table, click a triangle—it will become the new location for the selected process. If you make a mistake when moving a process in the table, choose* ***Undo*** *from the* ***Edit*** *menu to move it back to its original position.*

2 Choose Process Parameters from the Job Optimizer menu to display the **Process Parameters** dialog.



3 The power field is used to indicate the cutting setting on your machine. This will vary from one machine manufacturer to another.

4 The offset field identifies the distance from the wire to the part for each process.

5 The feed rate is the distance the wire will travel in a certain time.

6 The Cut Wire option is used to identify a process where you will need to cut the wire before moving to the next process and rethreading.

7 Remove Slug is used to identify stop points in the program where the operator would remove the slug piece before a finishing process can occur.

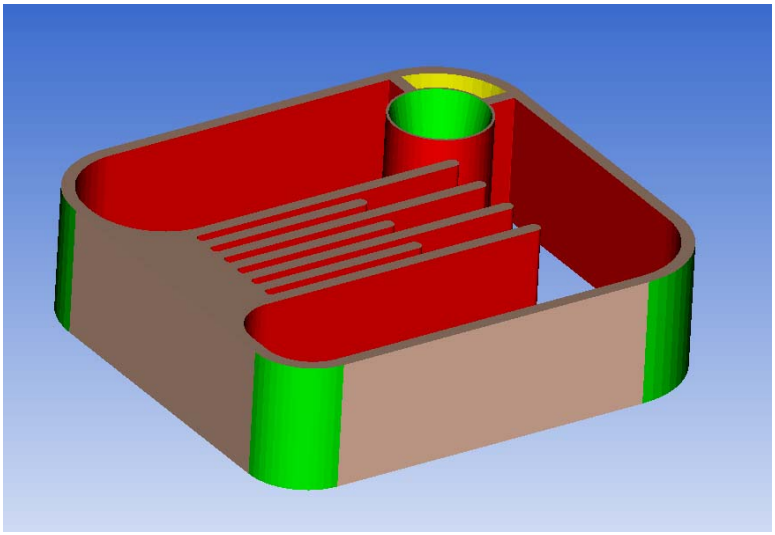
8 Click **<OK>** to return to the Process Table window.

Simulate the Process

Once you are satisfied with the appearance of your Process Table you can simulate cutting.

1 From the Process Table, click the Simulate in the lower left hand corner of the Process Table or just press the <Space Bar>

2 When the simulation Window appears, press the **<Space Bar>** or choose **Simulation** from the **Solids** menu to initiate 3D simulation. Your completed part should appear as shown below:

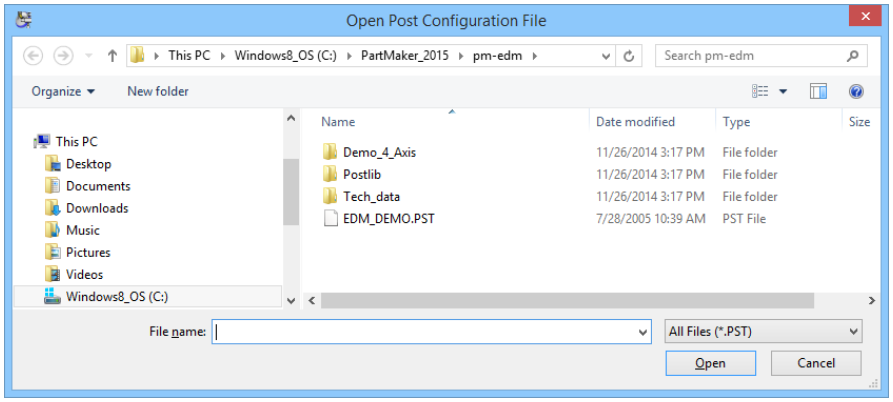


Copyright @2014 Delcam Ltd

Generate an NC Program

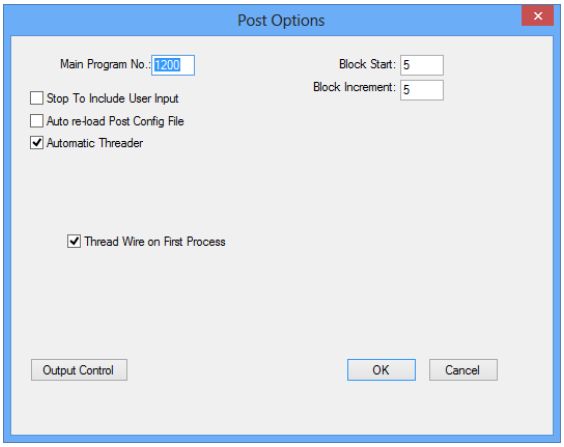
1 Choose Post Config File = ? from the Job Optimizer menu to display the Open Post Configuration File dialog.

2 Select the EDM_DEMO.pst postprocessor, then click <**Open**>.



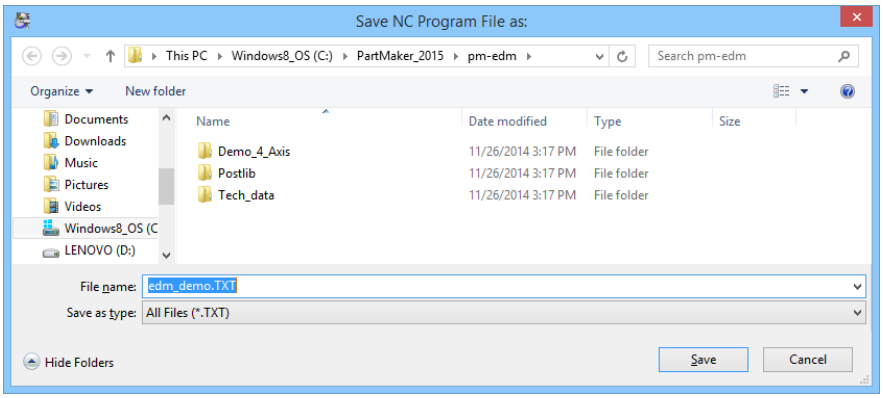
Tip: *If you use the same .PST file all the time, make sure it is the only one under the Pm-edm folder and it will be loaded automatically when you open the software.*

3 Choose **Generate NC Program** from the **Job Optimizer** menu to display the **Save NC Program** dialog. When you generate an NC Program for the first time during a programming session the Post Options dialog is displayed before **the Save NC Program** dialog.



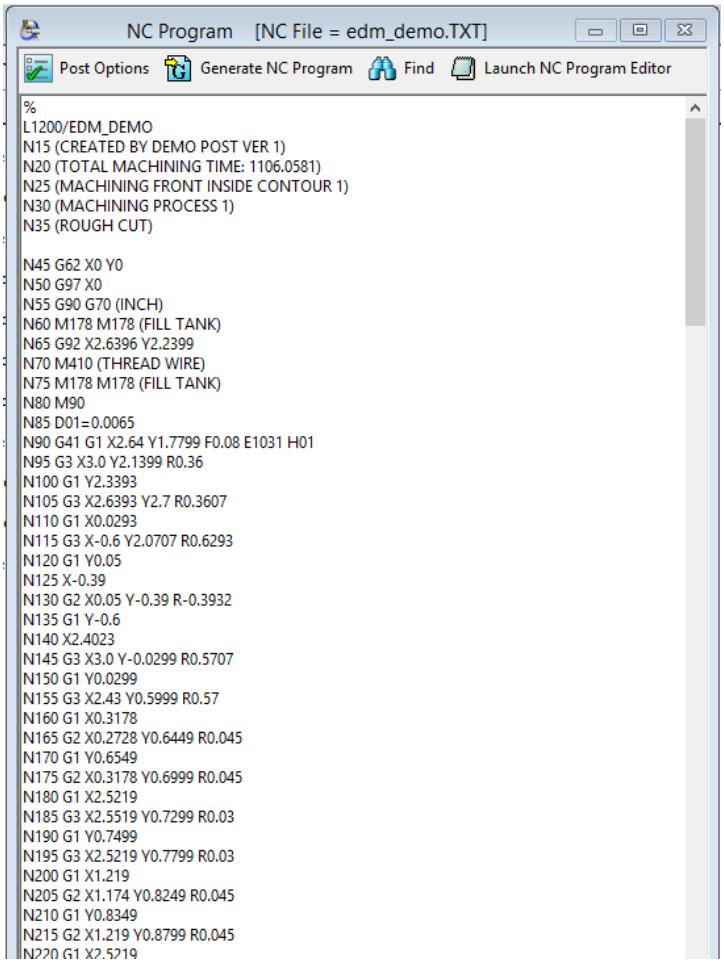
4 Choose the Post Options you prefer. If you leave Block Increment at zero there will be no sequence numbers in your part program.

5 Click **<OK>** to display the **Save NC Program** dialog.



6 Enter the name TUTORIAL.TXT for the NC program, then click **<Save>** to generate the NC program

Your completed NC program will be display below:



You have completed all of the steps necessary to program your Wire EDM.

Copyright @2014 Delcam Ltd

Chapter 4: PartMaker® Simulation

Introduction

PartMaker Versions 5.0 and higher come standard with an integrated 3D solid modeling process simulation and verification module. The purpose of simulation is to let you visualize all the processes as ordered on PartMaker's **Process Table**. You should run Simulation after the Process Table is arranged to your liking.

PartMaker's integrated process simulation module allows you to:

- Simulate all machining processes as they appear on the Process Table
- Observe the results of any process reordering
- Examine a solid model of the part through dynamic rotation and cross-sectioning

Using PartMaker Simulation

PartMaker Simulation can be viewed at any point in the programming process. You need not complete programming a part to see the results of your work.

Important! *To launch PartMaker Simulation you must have a Process Table generated. To generate the Process Table, choose* ***Generate Process Table*** *from the* ***Job Optimizer*** *menu. It is not necessary to complete synchronization of simultaneous operations before launching PartMaker Simulation.*

Launching Simulation

To launch PartMaker Simulation:

Press the **Simulation** button in on the Main tool bar

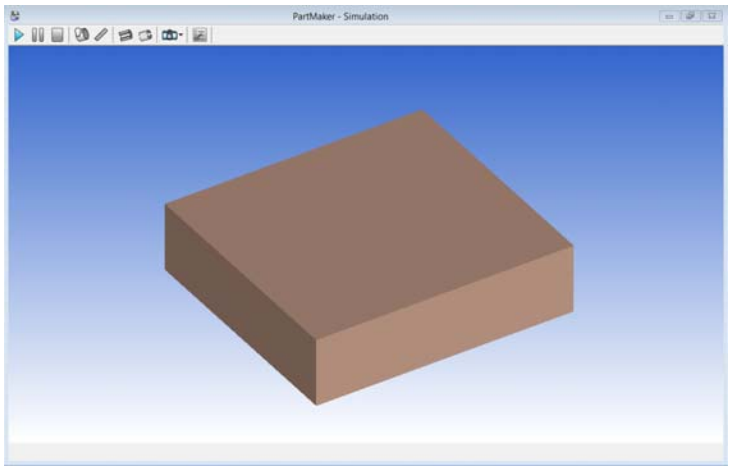
or

Press the Simulation button in the lower left hand corner of the Process Table

or

Choose **Simulation** from the **Simulation** menu

Upon launching simulation, you will see an uncut work piece and the machining table that has been defined. A newly launched simulation window is shown below:



Running Simulation

Once you have launched PartMaker Simulation, to start machining:

1 To start Simulation press the **Start Simulation** button.

or

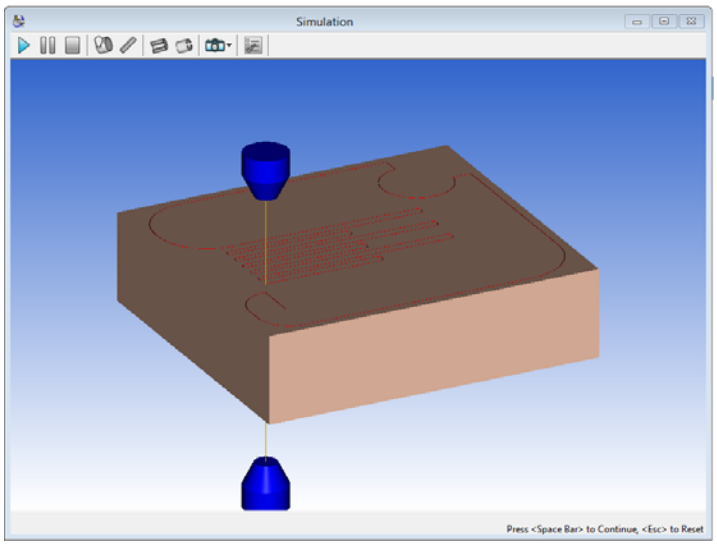
Choose Start Simulation from the Simulation menu

or

Press the **<Space Bar>** on your keyboard

2 Once simulation has started, you can suspend cutting be pressing the **Pause** or the **<Esc>** button on your keyboard.

Below, is an example of suspending simulation in progress:



Copyright @2014 Delcam Ltd

The Simulation Toolbar

Many of the common functions used in PartMaker's 3D simulation can be accessed via the Toolbar that appears at the top of the Simulation Window as shown here:

Each icon on the **Simulation Toolbar** is explained below. The features each of these icons enable will be explained in greater depth throughout this chapter.

Starts the simulation

Pauses the simulation

Resets the simulation

Shows the finished 3D model of the part

Allows you to measure the finished part

Allows you to take cross sectional views of the completed part

Allows you to show a full view of a part after sectioning

Allows you to manipulate the view of machined part

Allows you to see the Simulation Options dialog

Simulation Features

PartMaker Simulation has a number of process and part analysis features available both during and after process simulation is complete.

These features are described here: some of these features are not available in PartMaker EDM.

Reset

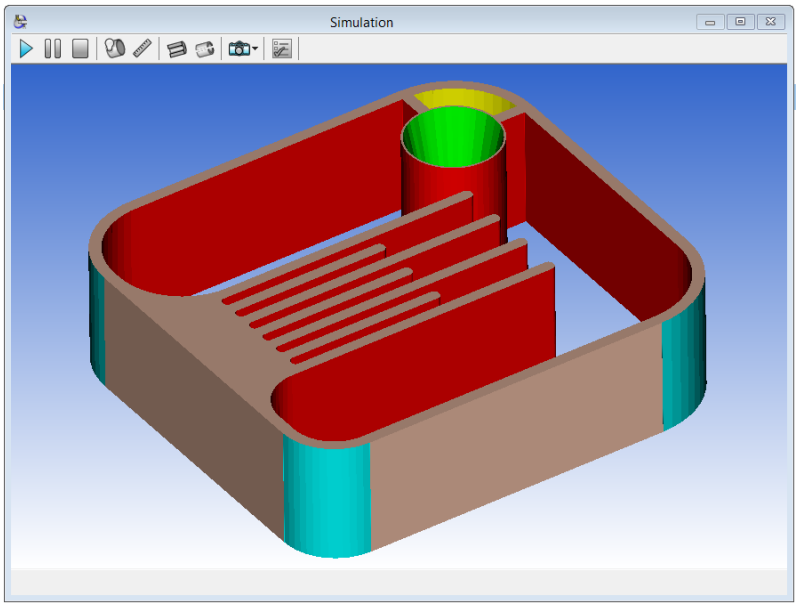
At any point you can reset the simulation to the initial screen by choosing **Reset** from the **Simulation** menu. Alternatively, if you have suspended **Simulation** by pressing **<Esc>**, you can press **<Esc>** again to reset the simulation.

The **Reset** command is also accessible by clicking the right mouse button when viewing the finished part (see below).

Show Finished Part

By choosing the **Show Finished Part** command from the **Simulation** menu, you can see a 3D model of your programmed part. You need not run the simulation prior to choosing **Show Finished Part**.

When choosing **Show Finished Part**, the verified part alone will appear in the **Simulation** window as shown here:



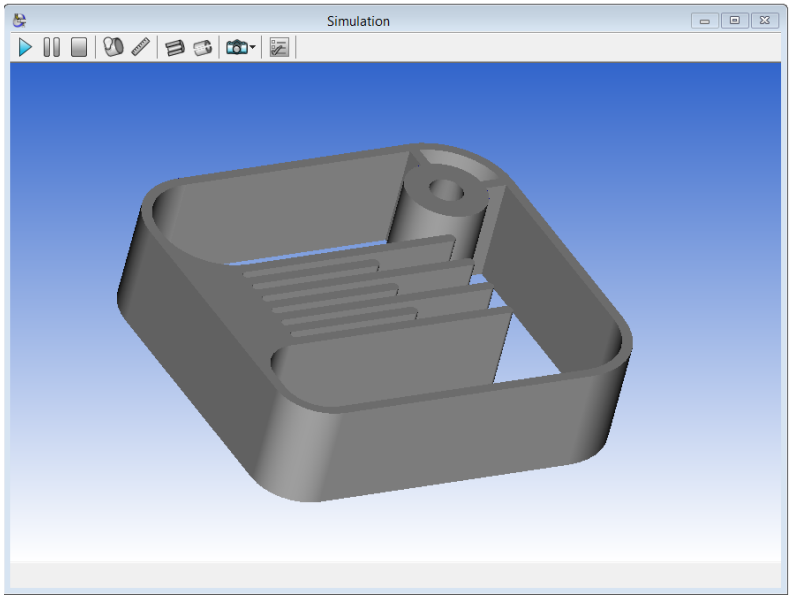
Save Finished Part As STL file

This will save the finished part to an STL file format. This can be then used again in PartMaker for Custom Stock or Surface Machining.

Copyright @2014 Delcam Ltd

Compare Finished Part to Solid Model

This will use the solid model specified in the **Comparison Options Dialog** to show a comparison of the part to the solid model.



Comparison Options

This will open the **Comparison Options** dialog. This allows you to specify the comparison models as well as the over and undercut display colors.

View Setting Features

PartMaker Basic Simulation allows you to automatically set a variety of views when simulation has been halted by using the **<Esc>** key or when using the **Show Finished Part** command. Various view orientations are available from the **View** sub-menu under the **Solids** menu.

Default

Choosing **Default** sets the view in the simulation window to the default view when **Simulation** was launched.

Best Fit

Choosing **Best Fit** sets the view in the simulation window to the best fit relative to the **Simulation** window as determined automatically by PartMaker.

XY View

Choosing the **XY View** command sets the view in the simulation window normal to the cross sectional plane formed by XY axes. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.

YZ View

Choosing the **YZ View** command sets the view in the simulation window normal to the cross sectional plane formed by YZ axes. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.

ZX View

Choosing the **ZX View** command sets the view in the simulation window normal to the cross sectional plane formed by ZX axes. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.

Isometric View

Choosing the **Isometric View** command sets the view in the simulation window aligned to the X, Y and Z axes. This view presents the part at a 45 degree angle.

Show Coordinate Axes

Checking this selection on allows the user to the axes symbol in lower left hand corner of PartMaker Simulation Window.

Show Finished part Origin

This will hide or show the 3-D axis on the finished part.

Show Coordinate axis

This will hide or show the 3-D coordinate system axes in the lower left hand side of the simulation window.

Background Color

Allows the user to choose the background color of the simulation. The background may be set to gradient by choosing different colors for the Top and Bottom Gradient Color respectively.

Datum Color

Sets the color of each axis of the datum, X, Y and Z respectively.

Copyright @2014 Delcam Ltd

View Manipulation Features: Rotate, Move and Zoom

PartMaker simulation allows you to manipulate the view of either the entire process simulation or the finished part (when using **Show Finished Part**). When using PartMaker, one of these commands will be activated, as denoted by a check next to it under the **Change View** sub menu under the **Simulation** menu.

Whenever the **Rotate, Move** or **Zoom** command is checked, it can be activated by holding down the left mouse key and moving your mouse.

It is advisable to always leave the **Rotate** command checked as both **Move** and **Zoom** can always be accessed using keyboard shortcut keys or the mouse. For example, if the **Rotate** command is checked under the **Change View** sub menu under the **Simulation** menu, **Rotate** can be activated by clicking the left mouse button and moving the mouse, **Move** can be activated by holding down the mouse's scroll wheel and moving the mouse and **Zoom** can be activated by simply scrolling the mouse's scroll wheel.

PartMaker provides fully dynamic simulation such that the **Rotate, Move** and **Zoom** commands can be accessed at any time during simulation.

Rotate

This command allows you to dynamically rotate the entire process simulation within the simulation window. When using **Show Finished Part**, **Rotate** allows you to dynamically rotate the solid model of the completed part.

When **Rotate** is active, you can rotate the process simulation or finished part by clicking the left hand mouse button and dragging your mouse in the orientation you would like to see the view.

Rotate may only be used when it is activated from the **Solids** menu. You can activate **Rotate** by choosing it from the **Solids** menu. Its activation is denoted by a check next to it under the Simulation window.

Move

This command allows you to dynamically move the entire process simulation around the simulation window. When using **Show Finished Part**, **Rotate** allows you to dynamically rotate the solid model of the completed part.

When **Move** is active, you can move the process simulation or finished part by clicking the left hand mouse button and dragging your mouse in the direction you want to move the view.

You can move the view if **Move** is NOT active by holding down the **<Ctrl> + <Arrow>** keys on your keyboard, with arrows denoting the direction in which the view will move. These are the same keys used when performing panning in PartMaker.

Additionally, **Move** can always be activated using the mouse by holding down the mouse wheel (if your mouse is so equipped) and dragging your mouse in the direction you want to move the view.

You can activate **Move** by choosing it from the **Solids** menu. Its activation is denoted by a check next to it under the Simulation window.

Zoom

This command allows you to dynamically size the entire process simulation within the simulation window when it has been halted by using the **<Esc>** key. When using **Show Finished Part**, **Zoom** allows you to dynamically size the solid model of the completed part.

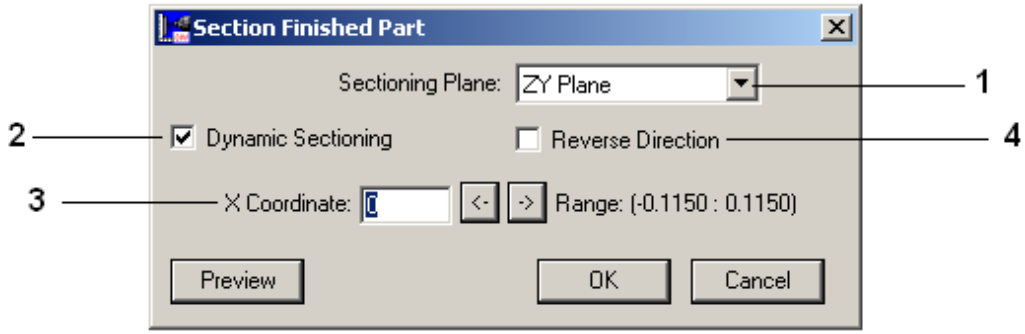
When **Zoom** is active, you can scale the process simulation or finished part by clicking the left hand mouse button and dragging it upward to increase the size of the part and dragging downward to decrease the size of the part.

You can scale the view if **Scale** is NOT active by pressing the **<F4>** key to decrease the size of the view or the **<F5>** key to increase the size of the view. These are the same keys used when performing **Continuous Zoom In** or **Zoom Out** in PartMaker.

Additionally, **Scale** can always be activated using the mouse by scrolling the mouse wheel (if your mouse is so equipped) upward to zoom in and downward to zoom out.

Sectioning Features

PartMaker simulation allows you to section the solid view to inspect the interior of the part. Various sectioning options by choosing **Sectional View** from the **Simulation** menu. Choosing **Sectional View** opens the **Section Finished Part** dialog as shown below:



1. **Sectioning Plane –** The Sectioning Plane section of this dialog allows you to determine the plane across which sectioning will occur normal to. The use can section a finished part normal to the XY, ZY and ZX plane. The axes along which the part is defined are constantly active in the lower left hand corner of the simulation window.
2. **Dynamic Sectioning** – Checking this box enables dynamic section, or the ability to see the part being sectioned as you adjust the location of the sectional plane. If Dynamic Sectioning is unchecked, , you will be able to see the result of your sectioning by clicking the <**Preview>** button or by exiting this dialog by clicking **<OK>**
3. **Plane Coordinate -** This area of the dialog allows you to adjust the level of the sectioning plane.
4. **Reverse Direction -** This allows you to reverse the direction of sectional plane shown.

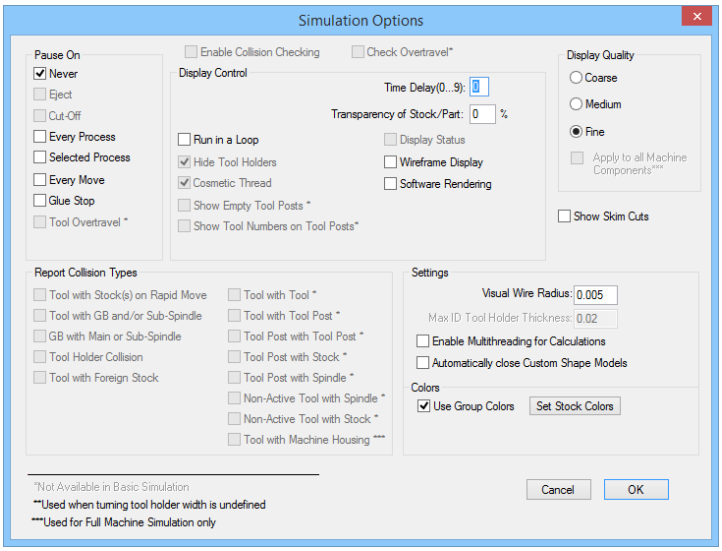
Copyright @2014 Delcam Ltd

Simulation Options

You can manipulate the appearance of simulation by choosing **Options** from the **Simulation** menu or by clicking the right mouse button in the **Simulation Window** and choosing **Options**.

PartMaker Simulation allows you to change the appearance of the view in the simulation window when simulation has been halted by using the **<Esc>** key or when using the **Show Finished Part** command.

The **Options** dialog as it appears in Basic Simulation for PartMaker Turn-Mill is displayed below:



Pause On

Never: Specifies that simulation should never pause during simulation

Every Process*:* Specifies that simulation should pause at the beginning of every process in the current Process Table

Selected Process*:* Specifies that simulation should pause at the beginning of a selected process in the current Process Table

Every Move*:* Specifies that simulation should pause before every tool move

Glue stop: Specifies that simulation should pause every time a glue stop occurs

Display Control

Time Delay (0...9): Specifies a time delay following every screen update for every tool motion

Transparency of Stock/Part*:* Specifies how transparent the part and/or stock should appear. 0% means solid part, 100% means "fully transparent" part, i.e. an invisible part.

Run in a Loop*:* Specifies that Simulation will run in a continuous loop

Software Rendering*:* Specifies that Open GL software graphics libraries should be used instead of the hardware implementation of the Open GL drivers that are dependent on the actual Graphics Video card used in every computer. If you use a high performance AGP video card you may achieve a significant simulation speed improvement if you do not use Software Rendering

Wireframe Display*:* Specifies that everything will be displayed using a wire frame representation instead of solid modeling

Display Quality

Coarse*:* Specifies that solids will be displayed using relatively large facets to assure the fastest simulation

Medium*:* Specifies that solids will be displayed using medium size facets that will result in lesser simulation speed

Fine*:* Specifies that solids will be displayed using very small size facets that will result in further decrease in simulation speed

Colors and Textures

Use Group Colors: Specifies that user-defined Group Colors will be used as cut colors. If this option is not checked a metallic gray color will be used as cut color for all tools

Set Stock Color*:* Allows a user to choose the color of initial stock using a Windows standard dialog.

Set Stock Texture: Opens the **Texture Dialog** for various texture options for the stock.

Copyright @2014 Delcam Ltd

Appendix A: Solids Import Tutorial for WireEDM

Introduction

This tutorial will teach you how to import 3D solid models into PartMaker/WireEDM for the purpose of extracting tool path geometry from them.

PartMaker Version **5.3.8** and higher features the ability to directly import 3D solid models in Parasolid™ transmit format (.X_T and .X_B files), as well as native SolidWorks™ format (.SLDPRT files) and Autodesk Inventor™ (.IPT files). Once imported, it is then possible to inspect the solid model and extract geometry from the solid model by taking sections, projected views and 'unwrapped' geometry. It is also possible to interrogate the model in order to obtain various plane angles, plane offsets, hole diameters, hole depths, etc.

The ability to import solids is available with every PartMaker product, namely, PartMaker/Turn, PartMaker/Mill, PartMaker/Turn-Mill, PartMaker/SwissCAM and PartMaker/WireEDM. The steps presented in this tutorial are carried out in PartMaker/WireEDM but can be used in any PartMaker application.

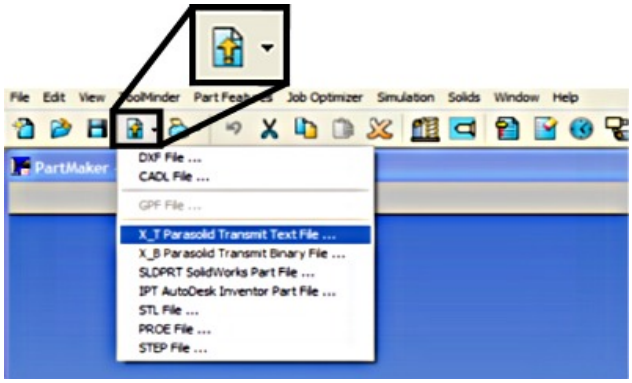
Sequence of Steps for importing solid models and geometry extraction

The steps outlined below describe the sequence in which you should approach in importing and manipulating solid models within PartMaker. This tutorial follows the sequence of steps below in importing and geometry extraction of a typical solid model.

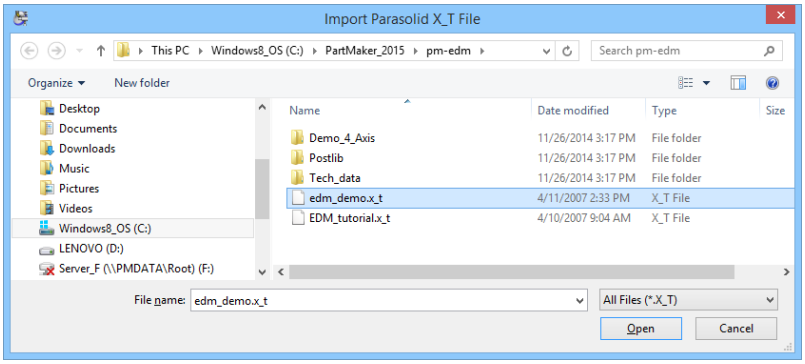
1. **Import Solid Model :** Read solid part file into Partmaker.
2. **Origin Axis Transformations:** Perform all the necessary origin and axis transformations to the part so that it is aligned correctly.
3. **Geometry Transfer and Part Interrogation:** Transfer geometry to the face window either by copying and pasting or doing a direct transfer. At the same time, view geometric information about the part by double clicking on various topological features like faces, edges and vertices.

Importing a Solid Model

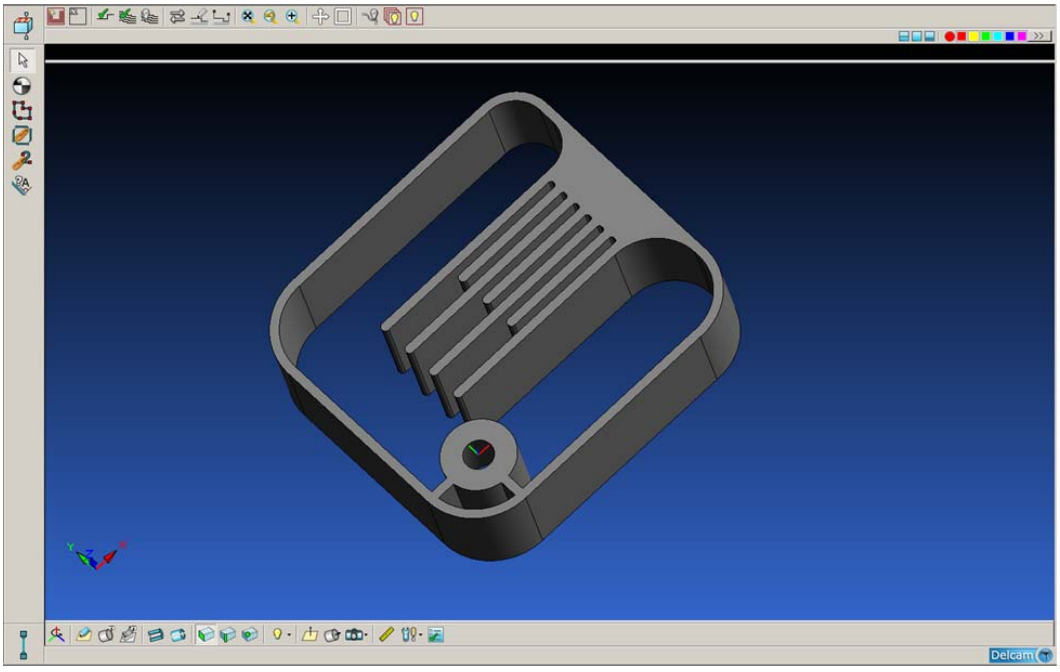
1 Select the **Import** icon and select the X_T Solid Model. The following file formats can currently be imported into PartMaker:



2 Choose the edm_demo.X_T file and click <Open>



3 When the file is read into PartMaker, it will appear in a separate Solids window. It can then be visually inspected by **rotation**, **translation** and **scaling**. There is a separate tool bar at the bottom of the Solids window for editing and manipulating the imported solid.

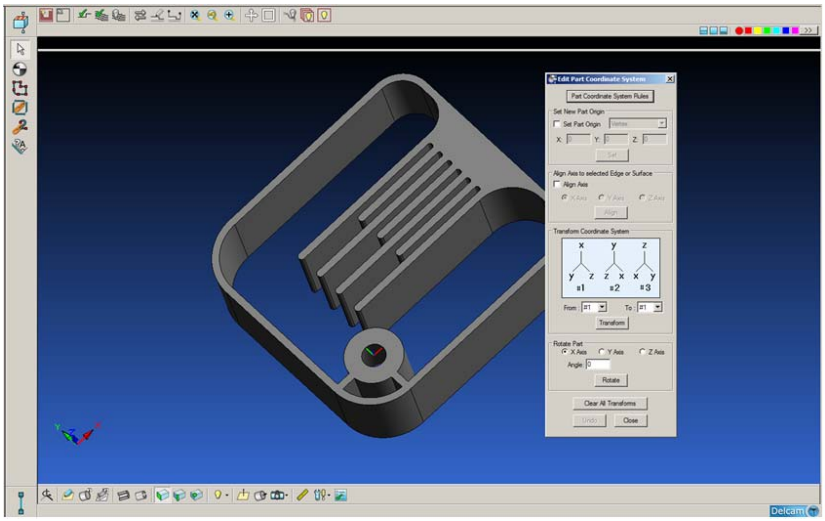


Copyright @2014 Delcam Ltd

Origin/Axis Transformations

The part has to be correctly aligned according to PartMaker alignment conventions for accurate and meaningful geometry extraction. This can be accomplished by selecting the **Edit Coordinate** button. You will then be able to:

- **Change the origin** by setting it at a new position.
- **Re-align the coordinate axes** in order to change the orientation of the part.
- **Rotate the part** about an axis through an angle in order to simplify geometry extraction.

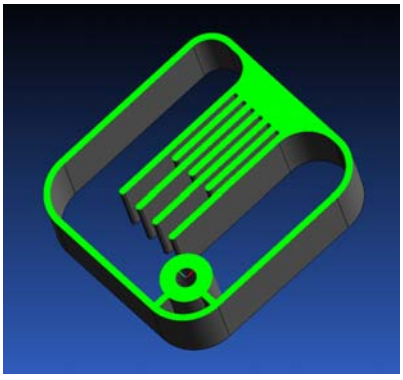


However, in this case, since the part is aligned correctly along the Z-axis, we will not have to carry out any transformation operations on the part. Please refer to the section explaining this dialog at the end of the tutorial for more information.

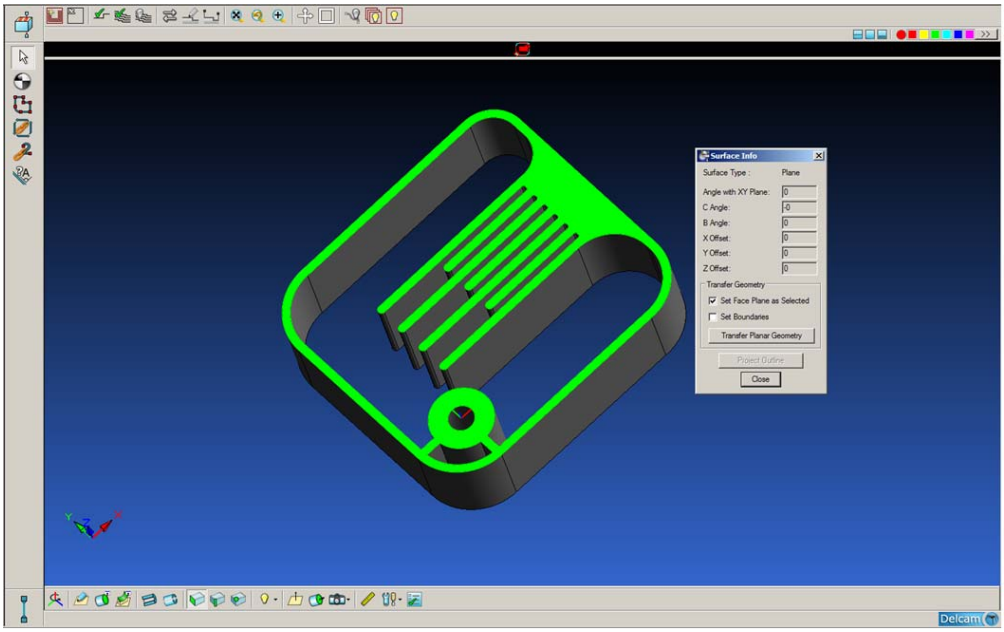
Geometry Transfer and Part Interrogation

First make sure that PartMaker is in Overlapping Windows Mode. This makes it easier for geometry transfer by keeping both the Solids import window as well as the face window visible simultaneously. To do so, choose **Preferences** from the **View Menu**. Enable **Overlapping Windows**. Then choose **Arrange Windows** from the **Window Menu**. You will observe tiling of the face window and the Solids import window. Next, we will program the job file for the part by extracting geometry from the solid model.

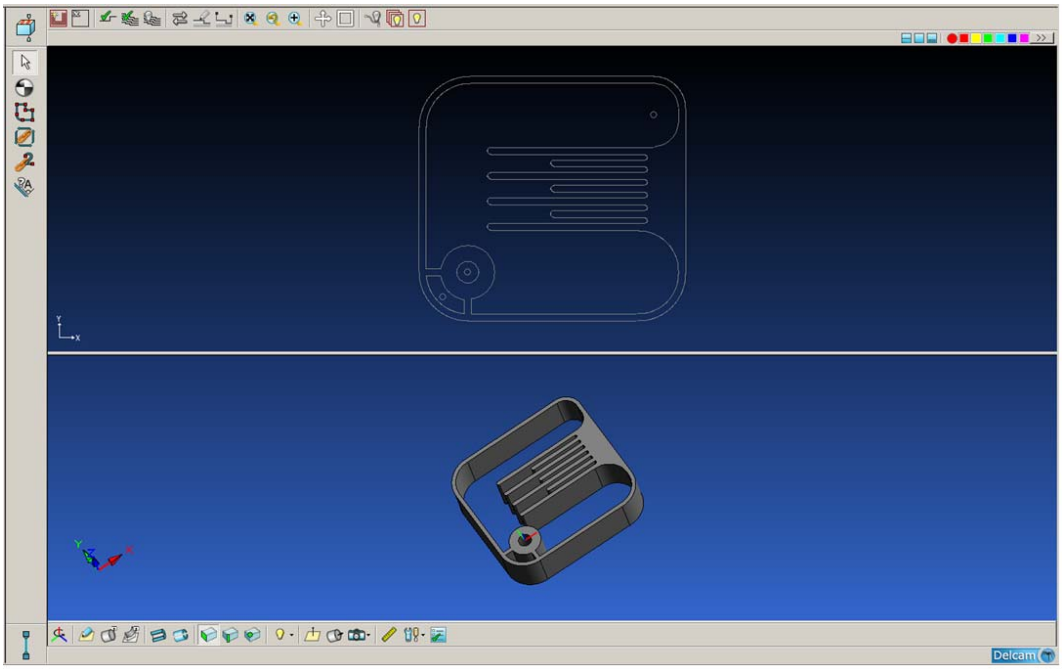
Double click on the face as shown.



This will bring up the **Surface Info Dialog,** displaying various face features like angles with the XY, XZ and YZ planes as well as the z, y and z offsets from the origin. Check the box for "**Set Boundaries"** and "**Set Face Plane as Selected"** and click the "**Transfer Geometry**" button. This button transfers the face geometry into the XY window.



You can now program the machining operations for the part as discussed in the WireEDM tutorial.



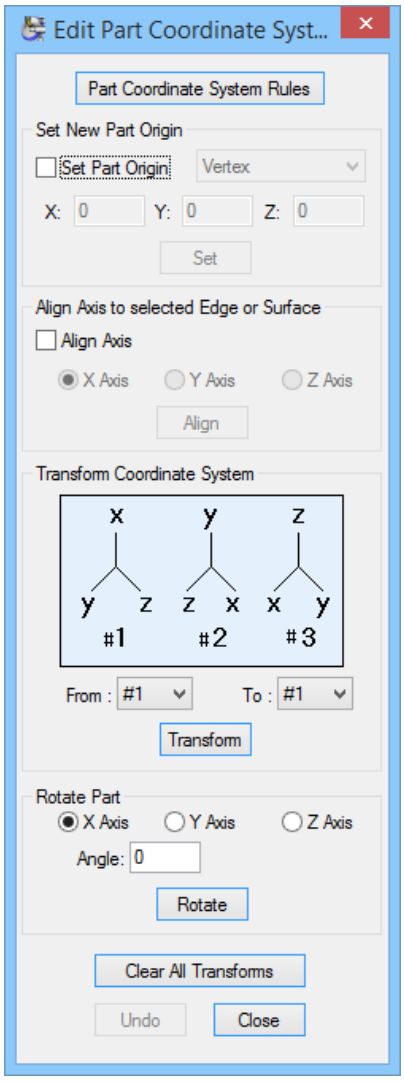
Copyright @2014 Delcam Ltd

Edit Coordinate System Dialog

In order to extract geometry correctly, the part has to be oriented along a specific axis with the origin at a particular point. Often the imported part needs to be transformed manually in order to orient it in the proper manner. The following kinds of transformations are possible:

- **Changing the origin** by setting it at a new position.
- **Re-aligning the coordinate axes** in order to change the orientation of the part.
- **Rotating the part** about an axis through an angle in order to simplify geometry extraction.

In order to transform a part, choose Edit Coordinate System from the Solids Menu. The following dialog box will be displayed.



Set New Part Origin

Set Part Origin checkbox: Allows you to select the method of choosing a new origin:

- ***Vertex****:* Sets the new origin at the selected vertex of the part.
- ***Arc/Circle Center*** *:* Sets the new origin at the center of the selected arc or circle.
- ***Face Center****:* Sets the new origin at the center of the selected face. The face can be a sphere, torus or a planar face.
- ***Edge MidPoint****:* Sets the new origin at the midpoint of the selected edge. The edge has to be a straight line.

X. The x coordinate of the new origin

Y. The y coordinate of the new origin.

Z. The z coordinate of the new origin.

These values can be manually set or automatically filled when a selection is made using one of the methods of selection discussed earlier.

<Set> button**:** Sets the new origin at the point described by the new coordinates.

Align Axis to selected Edge or Surface

Align Axis checkbox: Allows you to align the selected edge or surface from the solid model with the Part Coordinate System axis (**X Axis**, **Y Axis** or **Z Axis**) selected in this area of the dialog.

<Align> button: Aligns the edge or surface selected in the solid model with the selected axis of the Part Coordinate System.

Transform Coordinate System

From. Allows you to select the initial axis orientation to be transformed from.

To. Allows you to select the final axis orientation to be transformed to.

<Transform> button: Sets the selections for transforming the axis orientation from **'From'** to **'To'**.

Rotate Part

X Axis: If selected, rotates the part about the X axis

Y Axis: If selected, rotates the part about the Y axis

Z Axis: If selected, rotates the part about the Z axis

Angle: Angle about which to rotate the part in degrees. Negative values are allowed.

<Rotate> button: Rotates the part about one of the above selected axes.

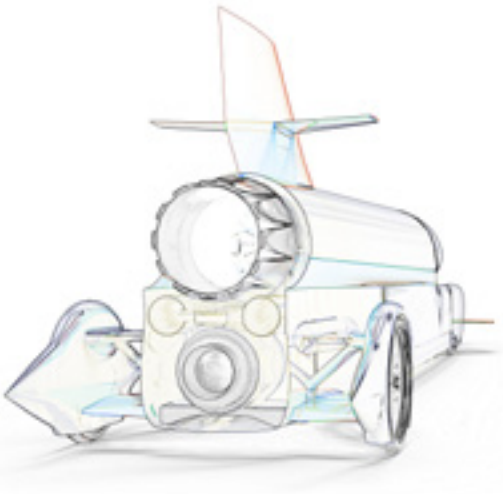
Buttons

<Clear All Transforms>: Click to clear all the transforms performed on the part since it was imported into the job file and return it back to its original coordinate system.

<Undo>: Click to reverse the last origin change, axis transform, or part rotation performed on the part.

<Close>: Click to close the dialog.

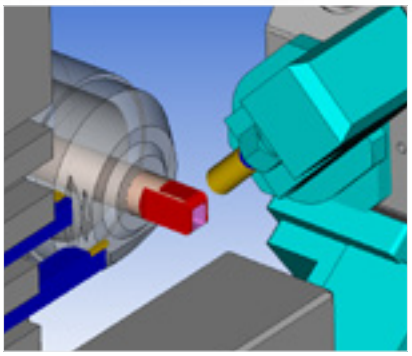
Copyright @2014 Delcam Ltd



PartMaker 2015

World-leading CAM software for production machining

www.partmaker.com

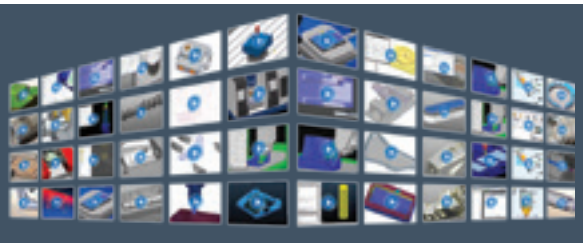


PartMaker 2015

www.partmaker.com

www.delcam.tv

Unlock the power of Delcam software



LEARNING ZONE

Visit **www.delcam.tv/lz** and see how Delcam software could benefit your business!

Delcam Advanced Manufacturing Solutions

Powering your productivity

Delcam
Small Heath Business Park, Birmingham, B10 0HJ, United Kingdom
T: +44 (0) 121 766 5544 | E: marketing@delcam.com
W: www.delcam.com

PartMaker Inc.
550 Pinetown Road, Suite 470, Ft. Washington PA 19034 USA
T: +215 643 5077 | E: info@partmaker.com
W: www.partmaker.com

To contact your local reseller, visit **www.delcam.com/resellers**

AUTODESK.

© Copyright Delcam Ltd 2014. All trademarks are the property of their respective owners.