CENTROID™

CNC Cylinder Head Port Replication

REV 06/19/06

Outline

01 – Probe and Tool Setup

02 – Initial Port Digitizing (3+2, Radial and Automatic)

03 – Creating a Surface for 5 Axis Digitizing in MasterCAM

04 – Creating a Toolpath for 5 Axis Digitizing in MasterCAM

05 – Dealing with 5 axis Digitized Data in MasterCAM

06 – Creating Cutting Toolpaths in MasterCAM

CNC Cylinder Head Port Replication

01 – Probe and Tool Setup

Setting Up Reference Tool

This tells the control the offset and diameter of any tools/probe tips to be used

Reference Tool = 1/8” Probe Stylus, with no extensions

- Ensure TT-1 is located at normal place on machine
  - Often this is the top left rotary table on the TRT25-100
- Check that the DP-4 is plugged in, **NOT the TT-1**!!
  - If the TT-1 is plugged in, when the DP-4 is used, SERIOUS DAMAGE will be done to both units!
- Verify that B Axis is set to 0.
  - If needed, press ALT +M to bring up MDI, and type “**B0**”, Followed by “**Cycle Start**”.
- Center the DP-4 about 1” above the center of the TT-1

Setting Up Reference Tool

- From the Main Menu, press
  - **F1 (Setup) > F2 (Tool) > F1 (Offset Library)**
- If the control is Machine Coordinate System (check the upper left corner of the screen), switch to Work Coordinate System (WCS)
  - Press **ALT+D** to cycle between **MCS** and **WCS**
- Set the **Z Reference Height** by pressing **F1 (Zref Height)**
- Follow prompts, press **F3** to indicate you want to set the Zref Height, followed by **Cycle Start**

Setting up Probes

- Insert your new stylus into the probe
  - Don’t forget to indicate in the probe using a dial indicator!
- Scroll down the list to the tool number that this probe will be
- Select the **Height Offset** column
- Jog probe tip to approximately 1” above the TT-1
- Press **F3 (Auto Measure)** followed by **Cycle Start**
- The height offset is automatically filled in. Go to the newly entered value, and subtract the radius of the probe tip
  - This ensures that the control is reading from the center of the tip
- Repeat steps whenever you change probe styluses.

Setting up Probes

- Insert your new stylus into the probe
  - Don’t forget to indicate in the probe using a dial indicator!
- Scroll down the list to the tool number that this probe will be
- Select the **Height Offset** column
- Jog probe tip to approximately 1” above the **TT-1**
- Press **F3 (Auto Measure)** followed by **Cycle Start**
- The height offset is automatically filled in. Go to the newly entered value, and subtract the radius of the probe tip
  - This ensures that the control is reading from the center of the tip
- Repeat steps whenever you change probe styluses.

Setting up Tools

- Unplug the **DP-4**
- Mount the new tool into the machine
- Plug in the **TT-1**
- Select the **Height Offset** column
- Jog probe tip to approximately 1” above the **TT-1**
- Press **F3 (Auto Measure)**
- Remove the radius from the new value.

Follow these steps any time the tool/probe is removed from the tool holder, or the TRT25-100 is moved!

CNC Cylinder Head Port Replication

02 – Initial Port Digitizing

Introduction

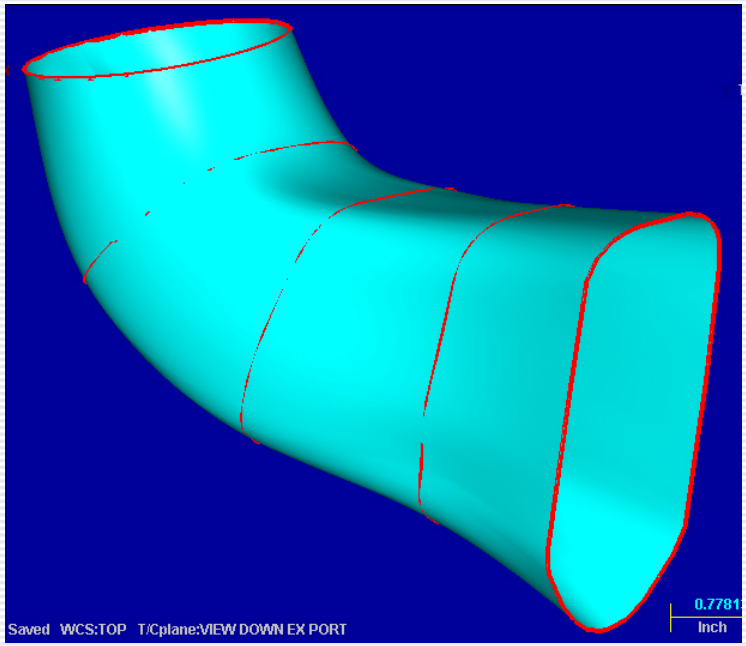
To start with, you must first begin with the rough shape of the port that you wish to digitize.

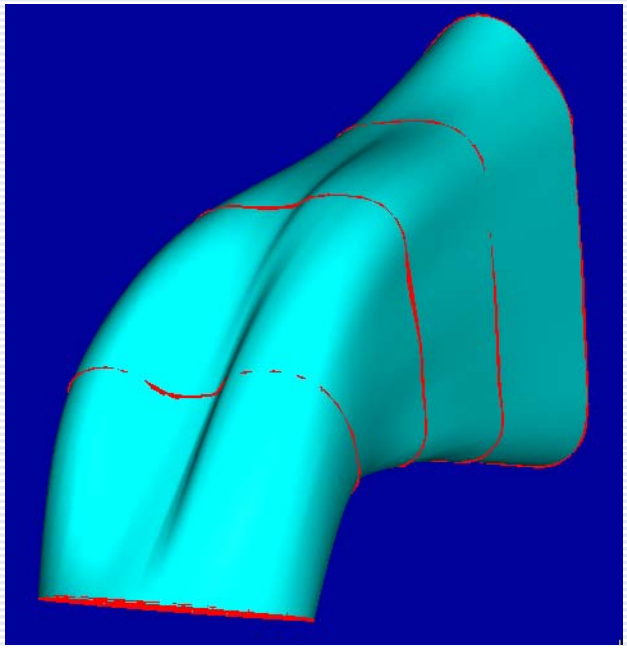
The way to make the rough shape is to do a few 3+2 axis digitized passes, and export them to MasterCAM to create a surface.

Generally you need 5 to 6 rough passes to create an acceptable surface. These passes create a spline, which is used to stretch a surface over.

Example of the Surface

There needs to be one spline at both of the openings, and then over any notable features (such as valve guides)





Rough surface splines (shown in red), rough surface (shown in light blue)

Level The Table

Digitizing the two open edges is done with a simple radial digitizing.

- Start by leveling the table so that the B axis is at 0°, and the A axis rotated so the valve seats are level.
  - Start by entering MDI (press **ALT+M**), then enter “B0” and press **Cycle Start.** Be sure that the B Axis will not hit anything when leveling!
  - Then take two readings on opposite sides of the valve seat, by using the **F9 (Digitize) > F4 (Probe) > F7 (Single Axis).**
- Rotate the A Axis by using MDI and entering “A#” where # is the number that you found above.
- Verify that you’re set to use the proper tool number for digitizing
  - Do this by going to MDI, and typing in **g43h<tool #>**. I’m using #14 tool for this, so I enter **g43h14** and press **Enter**.

Digitizing the Openings

- On the Main Menu, press **F9 (Digitize) > F2 (Radial)**
- Jog the probe tip to the center of the port, level with the valve seats then press **F1 (Center)** to define center position.
- Enter the settings as such:
  - **Containment Radius:** This is the maximum radius from the center that the probe will move. This port is approx 1.5” at the widest, so a radius of 1” is plenty.
  - **Z Patch Depth**: Controls how far down the probe will go. We want two radial patches, .25 apart, so I entered .25 here.
  - **Z Step:** Controls how far down the probe will step per pass. We only want two passes (the minimum), so enter .25 here
  - **Outer Stepover:** Controls what the step around is, in inches. For a rough surface, .2 will be sufficient. You can go finer as needed.
  - **Replay Pattern/Feedrate:** Controls speed/direction it would cut, if we were doing direct cut back. Ignore these settings.

Digitizing the Openings

- **Digitized File Name:** Press **<Enter>** and enter a file name for these splines.
- **Containment Angle:** Allows you to restrict the digitizing to a certain amount of degrees. Leave this set to "**Full**"
- **Multiple Patch:** This allows you to add on to an existing file. We will not be using this, so set to "**No"**.

- Repeat this procedure on the other side. Be sure to change the name of the file, so you do not overwrite the previous data!

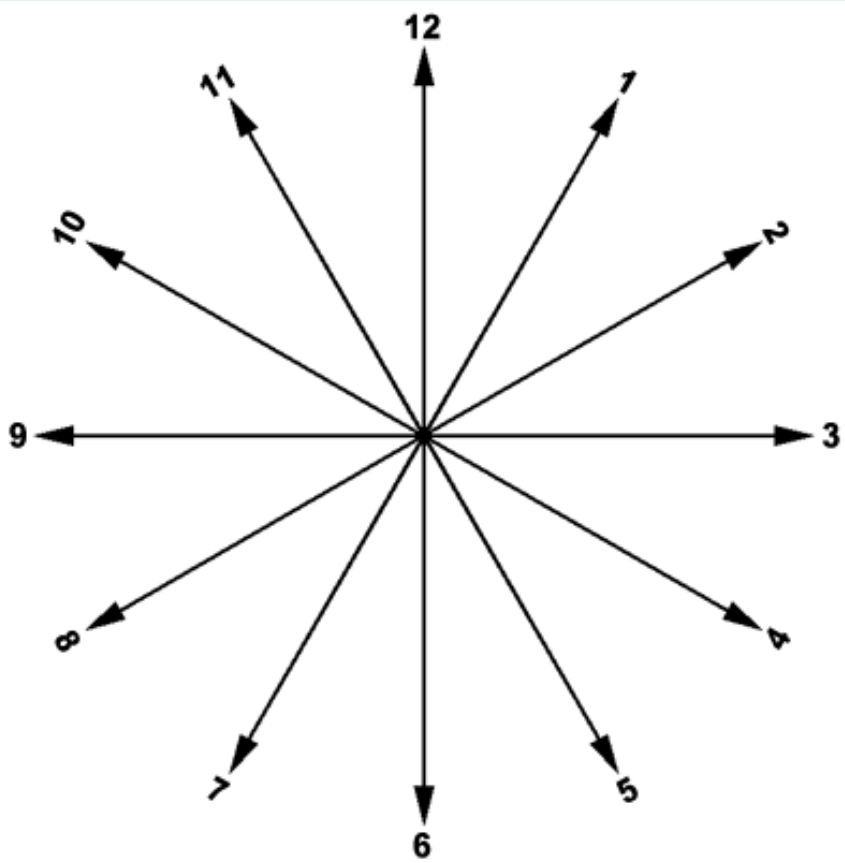
Manual Probing

- The areas around the valve guide and certain other features are too complex for a simple radial digitizing. For this a **Manual Probing** routine has been developed.

- Load the file by pressing **F2 (Load)** from the main menu, and choosing the file **"ManualProbing.nc"** Press **<Enter>**
- Press **F6 (Edit)** to edit the file
  - The first line after the comment lines (indicated by a "#" at the beginning) contains the output file name. This should be changed to something you will remember.
  - The file name must remain inside quote marks.
  - Press **ALT+F** to select the file menu, then choose **Exit**. Save changes.
- Press **Cycle Start** to begin the program.

Manual Digitizing

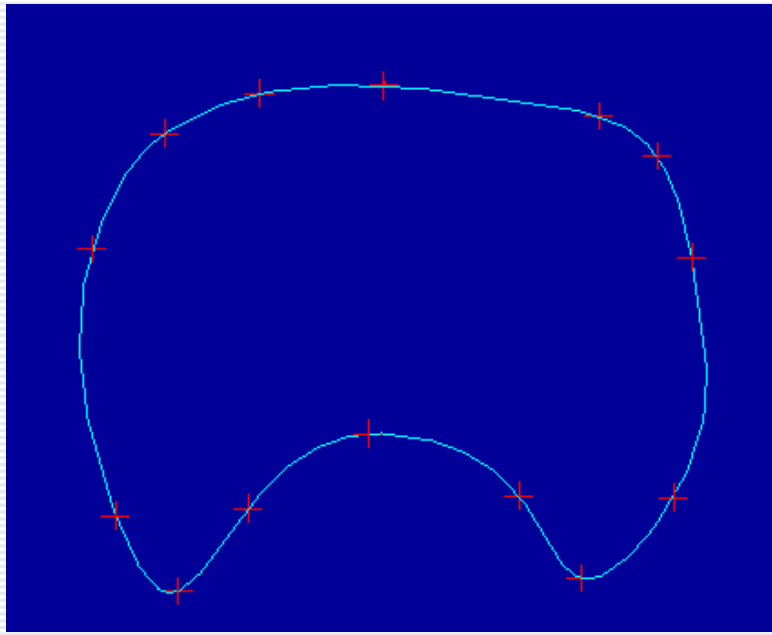
- Verify your settings displayed on the screen, then press **Cycle Start** to continue
- Use the MPG or direction keys to position the probe near the area to be digitized. Try to get to about ¼” away from the surface.
- Press **Cycle Start** to begin the acquisition process
- The prompt that pops up is asking for a direction in “Clock Coordinates”. For example, for 3 O'clock, enter “3”.



Manual Probing

- Press **Cycle Start** to acquire that point.
  - The probe will automatically travel up to 2” away from the start point, and always returns on the same line it moved on.
- Reposition if necessary and go through these steps again.
- Press the **ESC** key to exit the program at any time. If you reenter the program without changing the file name, the program will automatically append to the file for you. It will never overwrite data.

- You can see in this example, I digitized 15 points in this fashion. Shown in light blue is the spline that will connect them in MasterCAM



Exporting Data

The data that was collected will need to be rotated before it can be brought into MasterCAM. The **Patch Manager** that is built into the control will makes this process seamless.

- From the main menu, press **F9 (Digitize) > F5 (Patch Manager)**
- Press **F1 (New)** then enter a name for your project
  - The Patch manager was designed to work with many files for the same project, such as all of the digitize files for your cylinder head.
- Press **F2 (Add)**.
  - Use the arrow keys to navigate through the listed files. Press the **Space Bar** to select which files you wish to add. You can highlight as many as needed.
  - Press **F10 (Accept)**

Exporting Data

- Press **F6 (Xform Dig5)** to export the file.
  - The file will be saved as the project name.txt. For example, if you called your project “LS1”, your output file will be named “LS1.txt”.
- From this menu, you can also backplot the digitized data, and have it displayed properly.

CNC Cylinder Head Port Replication

03 – 5 Axis Digitizing with MasterCAM

Introduction

Using the 3+2 data, MasterCAM can be used to create a 5 axis digitizing routine. This will be much quicker and more accurate than doing 3+2 for the entire port.

The companion videos to this chapter are provided on the training DVD. There are two videos under the “01 - Creating Rough Surface” directory.

These videos are vital to understanding all of the subtleties associated with this process.

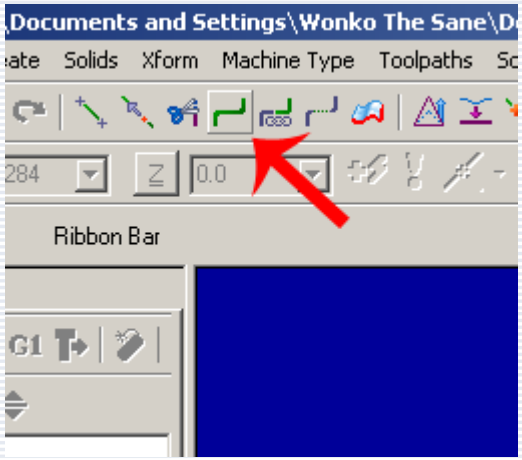
Opening Data

Mastercam has many settings for importing the data. It's important to use the correct one for your application.

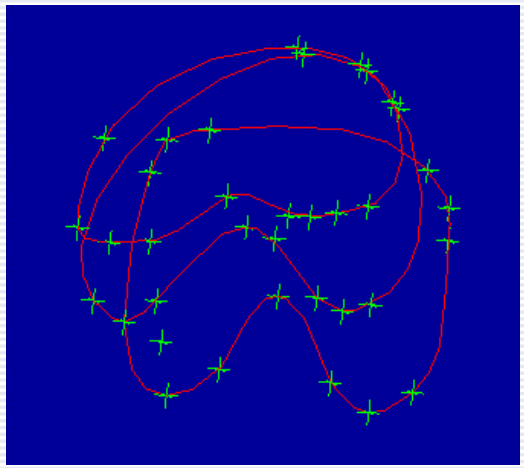
- Go to **Settings>Configuration** and choose **Converters**
- You have three options:
  - Points – Brings in raw points, with nothing connecting them. Use for Manually digitized points.
  - Lines – Same as points, but automatically connected by lines. No real use for us, as lines are not suitable for surface creation.
  - Splines – Creates one continuous spline. This is the optimum setting, however, it does not work well with the 3+2 Manually Digitized data. Use this with 3+2 Radial or 5 axis digitized data
- Choose Points, as we're working with Manually Digitized Data.
- Press the green check mark (OK)

Creating Splines

- Go to **File>Open** and browse to the directory with your file
- Be sure to change the "**Files of Type:**" to "**ASCII Files (*.TXT, *.CSV)**"
- Select your file and press "**Open**"



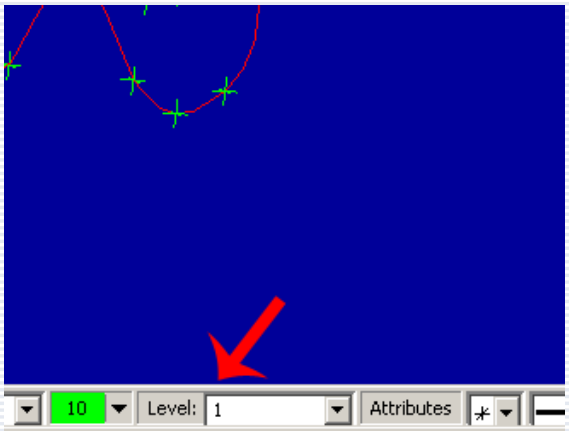
- Choose the "**Create Manual Spline**" either from the toolbar or from **Create> Spline> Create Manual Spline**



- Select every point in order, starting with the center of the valve guide.
- Repeat this for as many spline "slices" as you have.

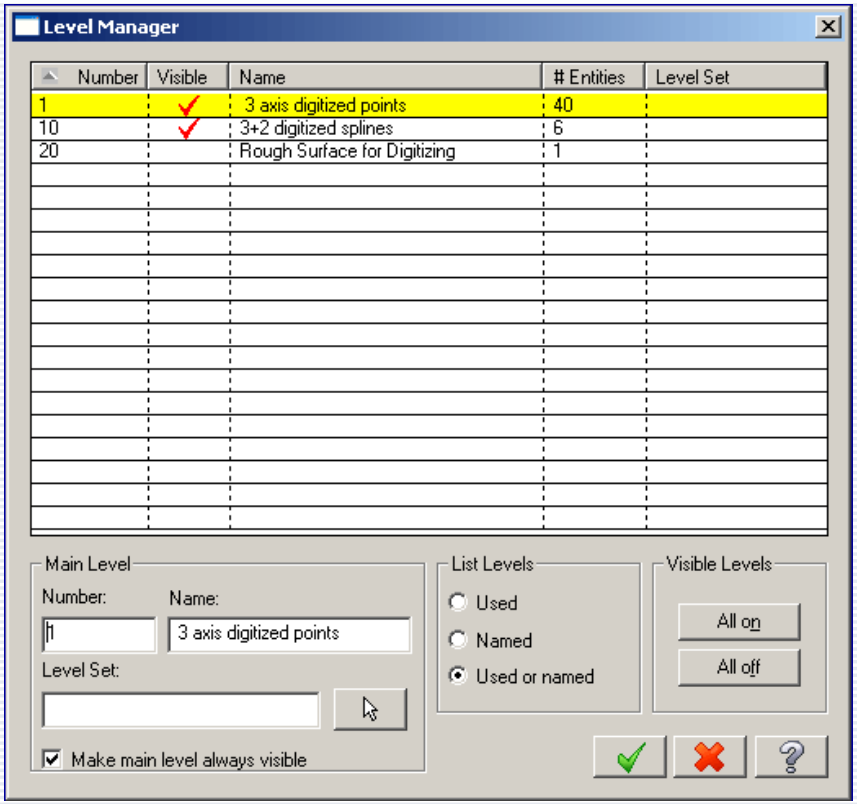
Level Manager

- Keep your data organized by grouping things together on “levels”. Levels will allow you to quickly turn objects on and off, keeping your workspace more organized.
- The level manager is accessed by clicking on the “Level” box, along the bottom of the screen.



  - Layers are named by number. You can define any number, from 1-2,000,000,000.
  - Change the level you’re working in by clicking in the number area and typing your new layer.

Level Manager



- **Number -** Refers to which level are already defined.
- **Visible –** Whether the level is visible or not.
- **Name –** The level name that is supplied by you.
- **# Entities -** Number of objects in that level.
- **Level Set -** Is equivalent to a “group”. You can have the same level set for multiple levels, so you can tell they’re all related.

- The level highlighted in yellow indicates which level is the master. This level cannot be turned off. To change the master, simply click on the number column of another level.

Editing Spline Endpoints

- With only the splines displayed, go under **Create Point Endpoints**  on the toolbar.
  - This will display all the endpoints on the current splines

- It is important to have all of the end points in a line before creating a surface. Unaligned end points will result in a twisted/warped surface.
- To align points, start by ensuring that you're in the Top view, press 
- Use the **Project Point**  to create a point on the area you want
  - Select the spline to create a point on, then drag the arrow to where the point will be placed.
- Put a point on any splines that need a new endpoint.

Breaking Splines

- Use the Trim At Point command to break up the spline.
- Select the "**TRIM**" command 
  - Select the "**Break**"  and "**To Point**"  buttons on the **Ribbon Bar**
- Click on the spline to break, and then the point.
- Repeat this for all the splines to be broken.

- The splines need be recreated into one spline after being broken. Select "**Create Curves Spline**" 
  - Click on the spline, as near the new end point as possible. Be sure that the green arrow starts at the new end point.

Fixing Splines

This same method can be used to fix irregularities in splines.

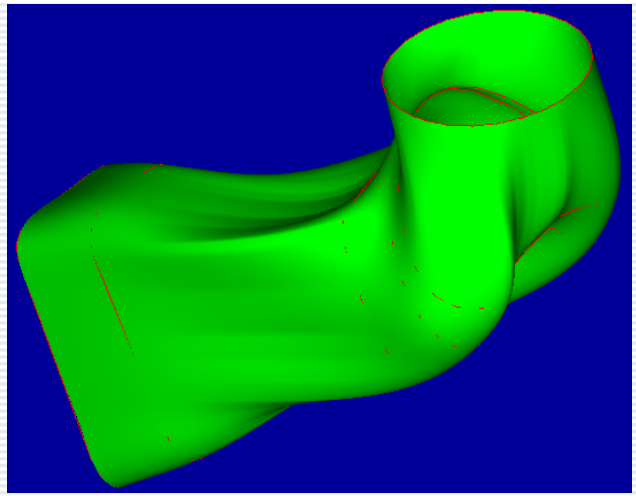
- Break the spline in the middle of the area to be fixed using the previous methods.
- Select "**Create Blended Spline**" 
  - Click on one side of the spline to be blended, and click the arrow on a good area of the spline.
  - Click on the other side, again clicking on a good point.
  - Use the "**Magnitude Adjustment**" on the ribbon bar to control how "powerful" the first and second points are.
  - Press the green check mark "**OK**", when you're satisfied with the curve.
- This process may need to be repeated to get the proper curve.

- Create a new curves spline when done.

Creating A Surface

Now that the splines are in order, create the surface across them.

- Select "**Create Ruled/Lofted Surface**" 
  - Select each spline in order. Always try to click on the same side of the spline.
  - Pay careful attention to the green arrow at the end point. Make sure that it points the same direction for every spline. If it does not, press the "**Reverse**" button  to reverse the direction.
  - Press the Green "**OK**" button to create the surface. It should look like a very rough version of your port.
  - Remember that absolute accuracy is not required.



CNC Cylinder Head Port Replication

04 – 5 Axis Digitizing toolpath with MasterCAM

Introduction

Using the rough surface, a 5 axis digitizing toolpath can be created.

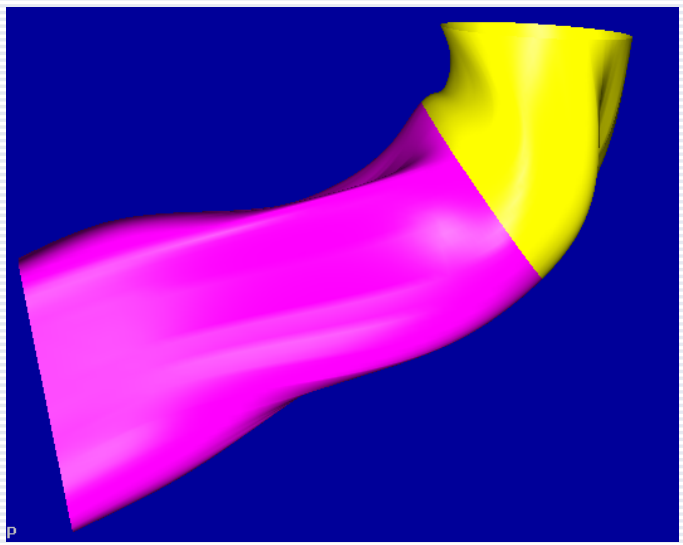
The companion video to this chapter are provided on the training DVD. There are two videos under the “02 - 5 axis digitizing toolpathing” directory.

These videos are vital to understanding all of the subtleties associated with this process.

Splitting The Surface

The surface must be split in two, so the toolpath can get half from the chamber side, and half from the port side.

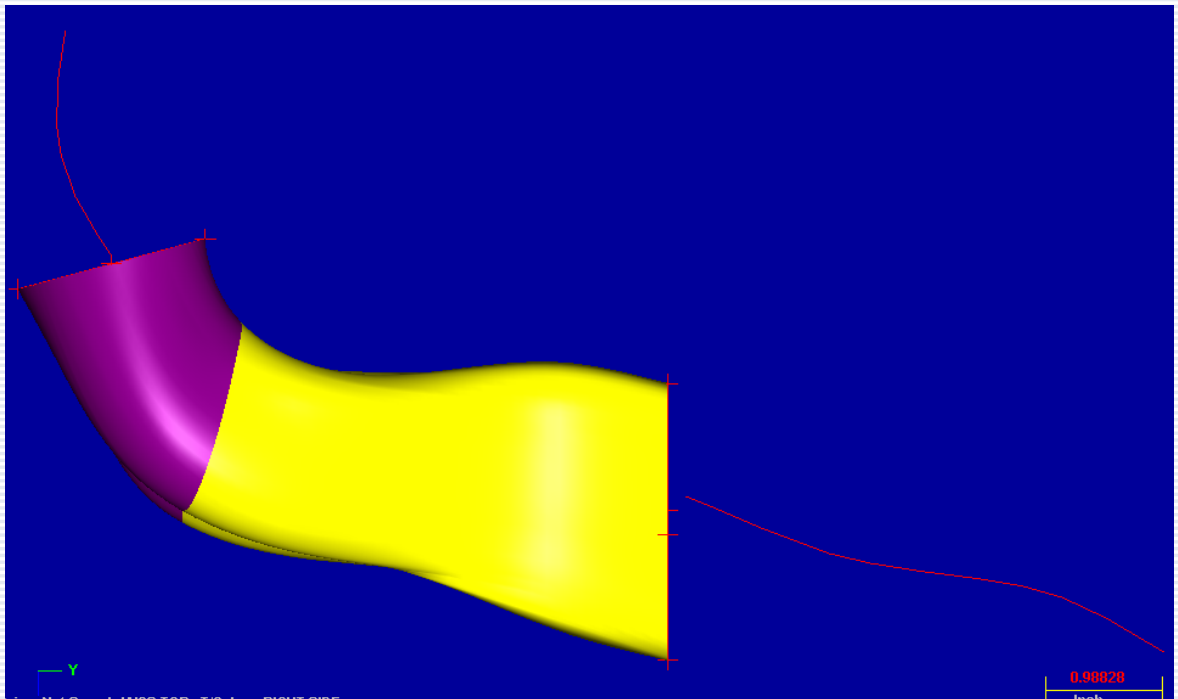
- Select "**Split Surface**"  and then select the surface to split
  - Position the arrow so it is pointing in the direction that you wish to split, and click.
  - By default, it will try to split it along the length, instead of the width. Press the "**Reverse**" button  to change that.
  - Click to finish the operation.



Tool Control Splines

In order to constrain the tool movement around a certain point, a tool control spline is created. Mastercam will ensure that the shaft of the tool is pivoting on that tool control spline at all times.

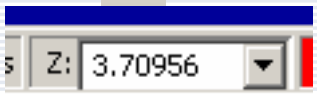
The spline should be roughly the same shape and length as the section of port you want to digitize.



Tool control splines shown in red

Tool Control Splines

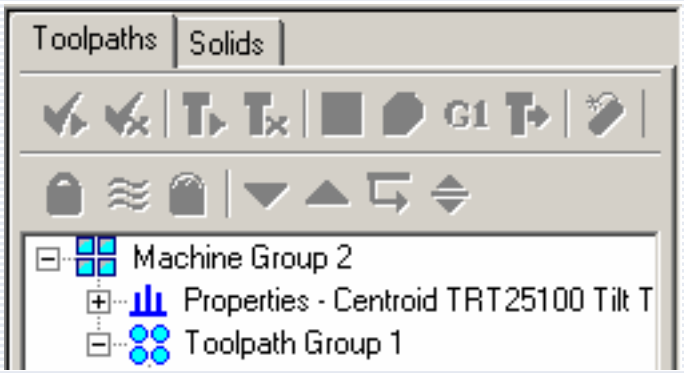
The tool control spline should be started at the center of the port.

- Create a spline at the edge of the port by using "**Create Curve on One Edge**"
- Use "**Create Point Segment**" to divide the curve into 4 points.
  - Mastercam starts counting at 1, so for 4 points enter "5".
- Draw a line between two points that divide the spline the cleanest
- From a "**Side View**" , change your **Z Depth** to half way across the line.
  - Click on the 

box at the bottom of Mastercam, then the center of the line.
- Use "**Create Manual Spline**" to create a spline that mimics the shape of the port. It should have no sharp corners to it.

Creating Tool Paths

- Change back to the top view  .
  - It is important to always deal with toolpaths from the top tool plane.
- Verify that your **Machine Group** is set to the CENTROID TRT25-100.



If there is no Machine Group, go to **Machine Type>Router>Centroid TRT25-100**

•If there is a generic machine there, expand the Properties group, then click on **Files**.

•Chose **Replace** under "**Machine – Toolpath Copy**" and choose the CENTROID.

The two primary types of toolpaths that we will be using are the **Flowline**  and the **Port**  toolpaths.

Creating Tool Paths

- Choose the **Flowline** toolpath for this one.
- Ensure that it's set to "**5 axis**" at the top.
- Click on "**Surface(s)**" under **"Cut Pattern**"
  - Click on your surface to use. After you hit okay, a rough flowline will be created over the surface. Ensure that this starts at the opening of the port, and not in the middle. Choose **Step Direction** to change the starting point. If it is on the outside, choose **Offset** to change it to the inside. Press **OK**.
- Click on **"Chain**" under "**Tool Axis Control**"
  - select the tool control spline. Be sure to click on it near the top of it, so the arrows are facing downwards, into the port. Press **Ok.**
  - **"Entire Chain Per Surface**" is fine for this, so press **OK.**
  - Press **"OK**" to finalize the selection part.

Tool Setup

- Right click in the "**Tool Parameters**" box and choose "create new tool"
  - The type is "**Lol. Mill**"
  - The "**Diameter"** is going to be .25, the "**Arbor diameter**" Is .16 and the **"Overall**" (Height) is 6.
  - Click on the "**Parameters**" tab. The **"Dia. Offset #**" and "**Length Offset** #" are whatever number you defined the probe as before.
    - "**Feed Rate**" is 20, **"Plunge**" and "**Retract**" rate is 10.
  - Press OK.
- Open the **Misc Values** box. Uncheck the "**Automatically set to post values**" if necessary. **Set** "**Use Probing**" to 1
  - Probing Approach Distance is .1
  - Probing Move Distance is .2

Multiaxis Parameters

- Click on the **Multiaxis Parameters** Tab.
- Change the "**Tip Comp**" to **Center**
- **Stock To Leave on Drive Surfaces** is set to the radius of the probe, so enter .125.

- Click on the **Flow 5ax Parameters** Tab.
- **Cut Control Distance** is the step down it will do after each pass. We find a setting of .05-.1 works well. Do a .5 for now, to ensure we get enough data around the valve guides.
- **Stepover Control Distance** is the length between the probed points. WE find that .06-.2 works well for this. Set it to .1 for now.
- Check to ensure that **Cutting Method** is set to **"One Way**".

- Click on **Gap Settings**. Gap controls how far it will "step" down without rapid moves. Change the **Gap Size Distance** to 2.

Axis Limits

- The **X Axis Angle Limits** control how much B Axis movement there will be. When the B is level (0), MasterCAM considers it @ 90 Degrees.
- Check the box "**Use Angle Limits**". We want to limit this to 10 degrees on either side, to speed up digitizing time.
  - Enter **80** in the box for Minimum.
  - Enter **100** in the box for Maximum.
- Ensure that **"Modify Tool Direction…"** is checked, **Delete** will cause major problems with your tool paths.

- Press OK, and wait for it to generate your toolpath.

BackPlotting and Posting

- Press the **Backplot** button the left side 
- Slide the bar to the right, or press the Play button to simulate the tool movement. Watch the motion of the tool, it should be consistent, with no rapid moves, and it should be following the tool control spline, starting at the top, and moving down it as the toolpath progresses.
- If editing is needed, close out of the backplot, and open your parameters up for that toolpath.

- If everything looks alright, press the **POST** button 

- If you have multiple toolpaths setup, it may ask you if you want to output them all. Select **No**.
- Save your file.

This file is now ready to be run on the machine.

Running the File

- On the machine, ensure that the DP-4 is plugged in and operational.
- Load the file up by pressing **F2** (**Load)** . Type in the drive letter that you are opening from.
- Browse to the file and press **Enter**
- Backplot (**F8)** the file on the control to ensure that there are no rapids or surprises.
- **Edit (F6)** the, and check its' initial A and B axis rotation, as well as the XYZ location.
- Rotate the table to the approximate A and B position by using **MDI.** For example, if the files say initial A is 33.433 and the B is 3.32, then in **MDI** enter A33B3 and press **Cycle Start**.
- Jog the probe down to near the starting point, ensuring that it won't encounter any problems in its initial movements.

When it's all clear, press **Cycle Start** to begin digitizing.

CNC Cylinder Head Port Replication

05 – Manipulating 5 Axis Digitized Data

Introduction

Now that the 5 axis toolpath has digitized its data, some manipulation will be necessary in MasterCAM to prepare it for the cutting toolpath.

The companion video to this chapter are provided on the training DVD. There are two videos under the “03 - Manipulating 5 axis Digitized Data” directory.

These videos are vital to understanding all of the subtleties associated with this process.

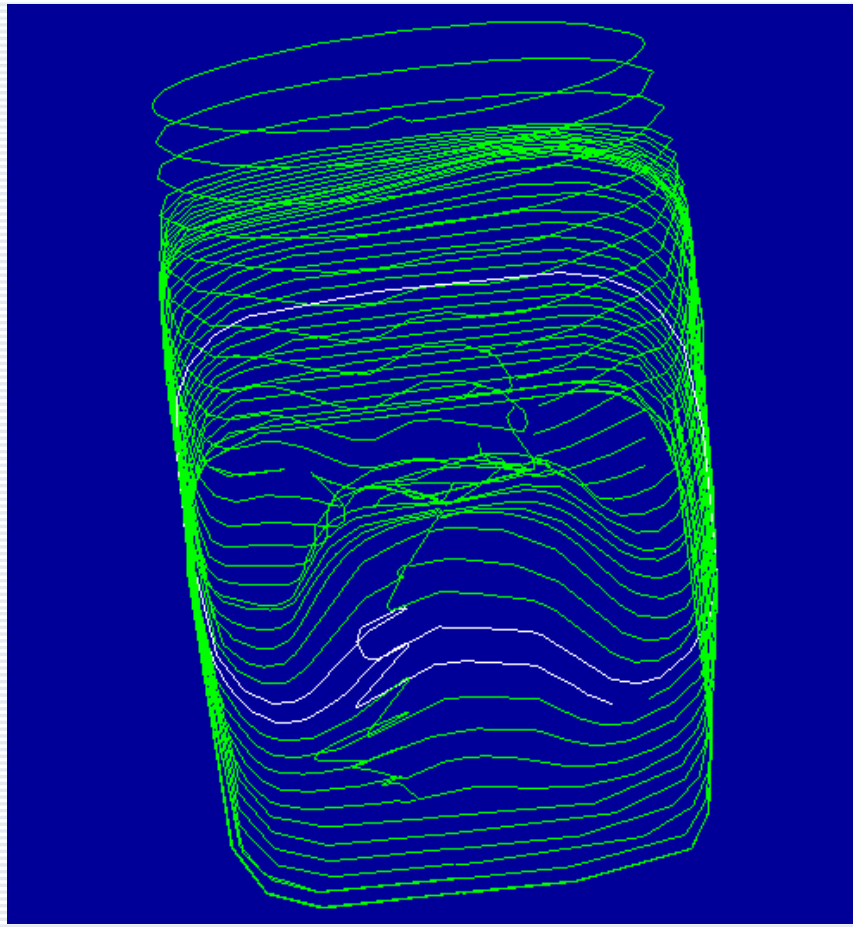
Export Data

The first step is to use the patch manager on the control to output all files for a port as one rotated file. Simply use the patch manager like was done before to load both files, and transform 5 axis on them.

This will again output the files as a single .txt file, which can then be loaded into MasterCAM.

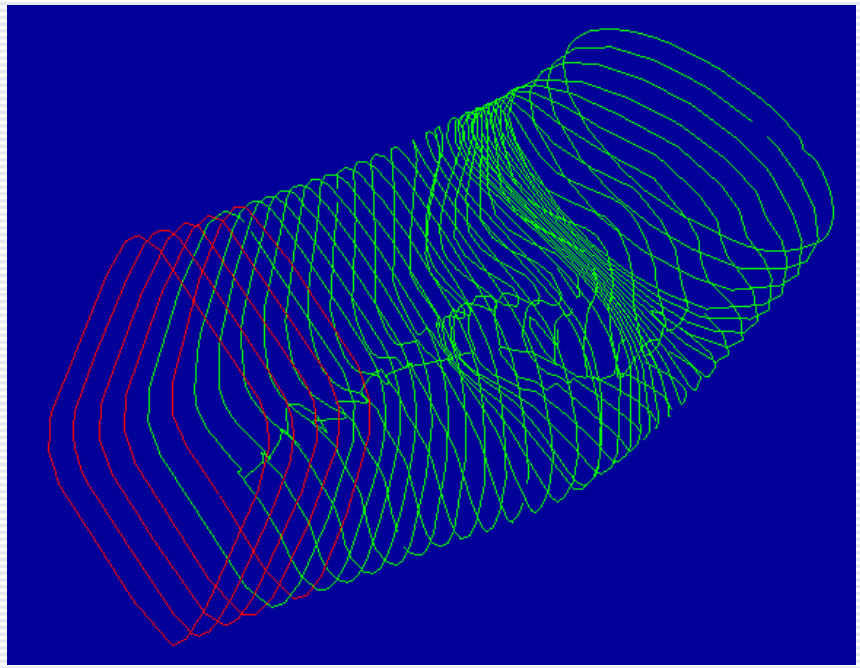
Opening Data

- Ensure that your convert settings (**Settings>Configuration>Converters**) is set to “Splines”
- Open the .txt file in MasterCAM.
- You may get a few warnings about incomplete spline data. Do not be concerned, some data doesn’t translate properly. Just press OK on the warnings. This will not affect the finished port.
- The splines will likely look joined together and disfigured around the valve guide like the example.

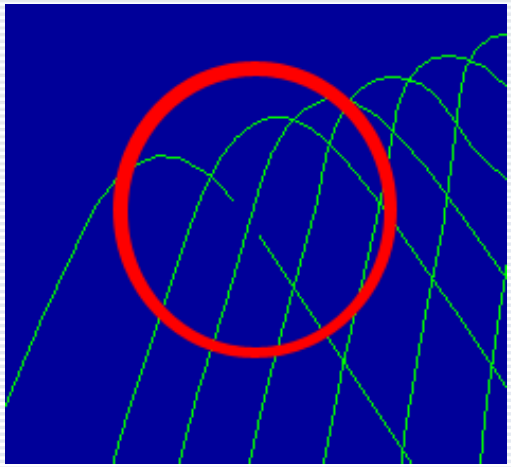


Data Manipulation

- Before manipulating the data, make another layer, with visibility turned off. This will be used to organize data that is done.
- Start by moving any good splines that do not require manipulation to the new layer, shown in red.
- All of the remaining splines will need to be modified in some way.



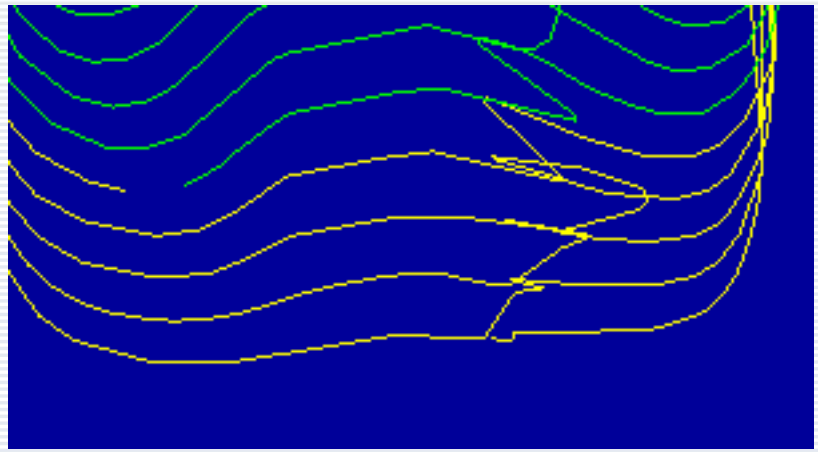
Blending Broken Splines

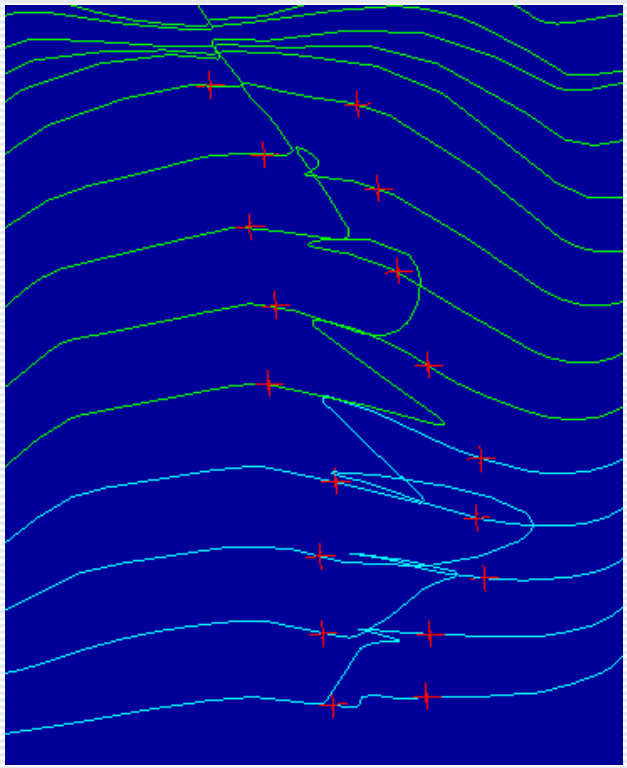


- The lowest left spline in my example simply has a break in the data.
- Use the **“Create Blended Spline**” command  to bridge this gap.
  - Click on one side of the gap. Click the arrow on the edge. Click on the other side of the gap.
- It should now be bridged. Create a **Curves Spline** to join all the pieces into one spline. Don’t forget to delete the previous curves.

Connected Splines

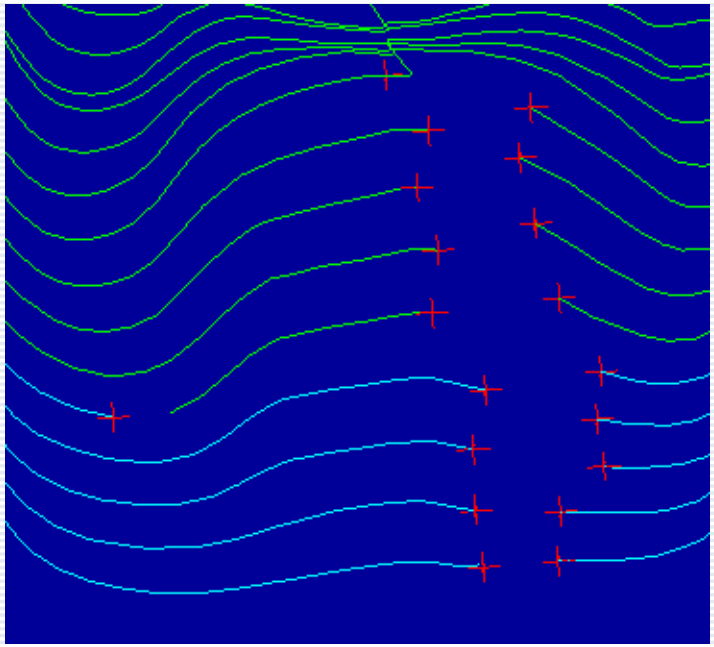
- The next few splines are all joined together at the end points. They will need to be separated.





- Use **Create Point Dynamic** to create points at the break points, like in my example.
- Use the **Break** command with the **At Point** setting to break all those splines at the new points.
- Delete the middle sections

Blended Splines



- After all the splines are broken, make a **Blended Spline** to bridge them together.
- Select **Create Curve Spline** to delete the old parts and make a new spline.

- Move any newly completed curves to the hidden level.

- Follow this procedure for any splines that are joined or split.
- Pay special attention to the “start/end” points of splines, as they are often not truly connected. You can often only see this when zoomed in.

Breaking The Splines

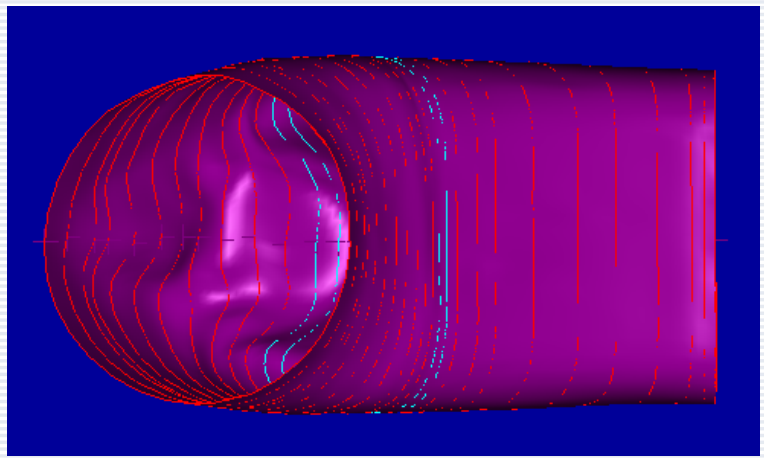
Just as with the 3+2 digitized data, it's important that all the splines are broken in the same place.

The procedure to follow is the same as before as well.

- **Create Dynamic Point>Trim At Point>Create Curves Spline**

Create a new surface in the same method as was done with the 3+2 manually digitized data.

- **Create Lofted Surface > Select Every Spline in order**
  - If you select a spline, and there is a red arrow in addition to the normal green arrow, that means the spline is not closed. Escape out of the surface command, and fix that spline.



Manipulating the Splines

Any minor imperfections in the surface can be corrected by breaking the spline in the middle of the problem, then blending using the options of **Create Blended Spline**.

- Use the "**Magnitude Adjustment**" on the ribbon bar to control how "powerful" the first and second points are.
- You may need to create a second **Blended Spline** to get the proper shape.
- After creating your newly modified spline, don't forget to **Create Curves Spline** to join it before creating your new surface.

This method should allow you to "massage" away minor trouble spots on your surface. If there is a major trouble spot, you will most likely have to re-digitize that section of the port. This time use a smaller step over and around.

CNC Cylinder Head Port Replication

06 – Cutting Toolpaths

Introduction

With your new surface created and split, the toolpathing is nearly identical to what was done in section 04:

- Create Surface
- Split Surface (It's a good idea to overlap your split surfaces!)
- Find the center of the port opening, set the Z depth
- Create a tool control spline
- Create a flowline or port toolpath (try flowline first!)
- Enter Settings
- Translate toolpaths to replicate them across the cylinder head

Prepare for Toolpath

- The surface should have been created/manipulated per Section 05. If not, create a **Ruled/Lofted Surface**
- Split the surface using the **Split Surface** command.
  - The surface should be split through the valve guide, just as before.
- Find the center of the opening
  - **Create Curve on One Edge > Create Point Segments > Create Line (**across two points)
- Set the Z Depth by clicking on the **"Z"** along the bottom toolbar
- Create a **Manual Spline** through the port, and **Xform Drag** into position to use as **Tool Control Spline**
- Create a **Flowline Toolpath** using your new surfaces and tool control splines

ToolPath Setup

Most of the settings are going to be the same as with digitizing except for a few:

- Under **Misc Settings** on the "**ToolPath Parameters**" tab, ensure that the first setting (Use Probing) is set to 1
- On the **MultiAxis Parameters** tab, make the number slightly larger on the "stock to leave on drive surface" if you're doing a roughing pass.
- On the **Flowline 5ax Parameters** tab, under **Cut Control**, uncheck "distance" and enter a cut tolerance of .0001 . This will ensure lots of vectors for a smoother toolpath.
- Change your "**Stepover Control**" distance to the appropriate amount. I'd recommend .040 for roughing, .020 for finish.
- That's all that needs changed.

Translating Toolpaths

When all of the necessary toolpaths have been created, it's a simple matter to translate them to other the other ports in the head. Ensure that you're using the top view before proceeding. Transformed toolpaths are associative. This means that if you update the original, all transformed toolpaths will update instantly.

- Use the **Transform Toolpath** 
- Select which toolpath you're transforming in the **Source Operations** box.
- Select the **Translate** tab up at the top.
- Be sure that the **Translate** settings is set to "**Between Points**".
- On the right, enter your original center point, and your new center point. The original center point is the center point for the cylinder that you digitized. Your new center point is the center for the cylinder head that you wish to copy the toolpath to.
- Press **OK**, repeat as necessary.
- Post the translated toolpath just as you would a regular toolpath.