

# ***Mastercam*** 2019

IMPERIAL – HANDBOOK VOLUME 2



Demo Software Download Instructions Included

# 3D MODELING & MACHINING

# ***Mastercam***<sup>®</sup> 2019

## **Handbook Volume 2**

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## **Mastercam 2019 Handbook Volume 2**

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# Legend

## Conventions

Key words and Mastercam menu items are shown in bold the first time they are used. Columns on the outside edges of each page and note pages at the end of each chapter provide ample space for taking notes.

Useful tips, recommended settings, best practices, and detailed instruction on the most important features are included when possible.

An accompanying CD ROM contains files needed to complete all exercises in this handbook.

**The following terms are used throughout this book.**

- **Left Click** means to click once on the left mouse button.
- **Click** means the same as left click.
- **Right Click** means to click once on the right mouse button.
- **Scroll** means to roll the mouse scroll wheel, or move the scroll index in a list.
- **Options** are Mastercam functions selected from the main menu.
- **Enter** means to select the <Enter> key on your computer keyboard.
- **Press** means to press on a keyboard key.
- **Choose** means to select a menu option or button.
- **Open/Close** means to open or close a dialog or information box.
- **Dialog Box/Panel** is a window that opens to allow input of information and setting of defaults.
- **Drop Down/Fly Out Menus** are menus that expand down, left, right, or up, to reveal more menu lists.
- A **Function** is the same as a menu option or command.
- **Help** means the Mastercam help files loaded with your software.

# 2 Splines

## OBJECTIVES

This chapter introduces spline modeling and machining. Upon completion of this chapter, you should be able to do the following:

- Know the definition of a spline.
- Know the meaning of the acronym **NURBS**.
- Understand the difference between **NURBS** and **Parametric splines** and explain the relative advantages of each.
- Create splines given a list of coordinates or text data file.
- Understand chordal deviation.
- Control the end conditions of a spline.
- Understand factors related to machining splines including cut tolerance and arc filtering.
- Understand machine performance issues related to machining very short lines and arcs at high feed rates.

## INTRODUCTION

A spline is a curve that passes smoothly through a set of construction points. Splines that lay flat in a construction plane are called 2D splines. Those that do not are called 3D splines.

Splines are used to define complex shapes such as aircraft wings, boat hulls, skateboards, and sculpted or organic shapes common to consumer goods. They are used extensively in surface modeling.

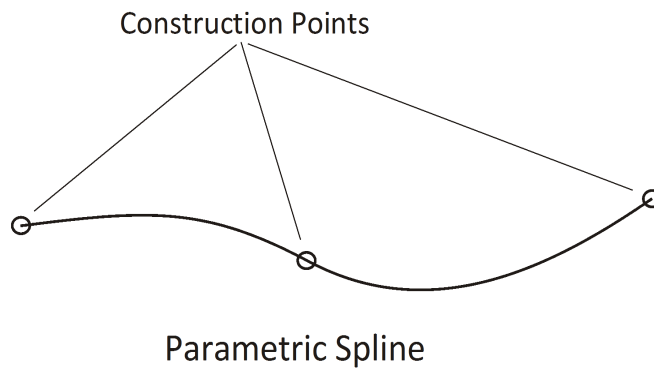
Many of the issues related to spline modeling and machining are closely related to surface modeling and machining. Therefore, a thorough understanding of how to create and machine splines forms a foundation for your understanding of surface modeling and machining.

Mastercam allows the creation of two different types of splines, **Parametric** and **NURBS**. While both are created using the same steps, they use different mathematic equations. The type of spline created is set in the Mastercam Configuration, CAD Settings dialog box. **NURBS** is the default setting.

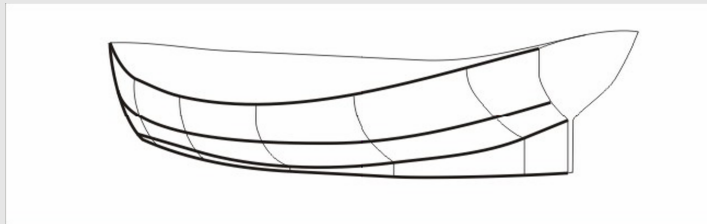
## PARAMETRIC SPLINES

**Parametric Splines** are based on polynomial mathematics. The shape of a parametric spline is determined by a set of coefficients for each construction point.

One advantage of a parametric spline is that after it is created it is easy to precisely change the shape of the spline by moving construction points.



Splines were developed in the ship building industry to define the shape of boat hulls. In wood boat construction, long planks of wood are fastened to bulkheads, called lofting sections. These planks bend smoothly between these sections to give the boat its shape. In fact, the mathematics of splines is based on the bending properties of wood.





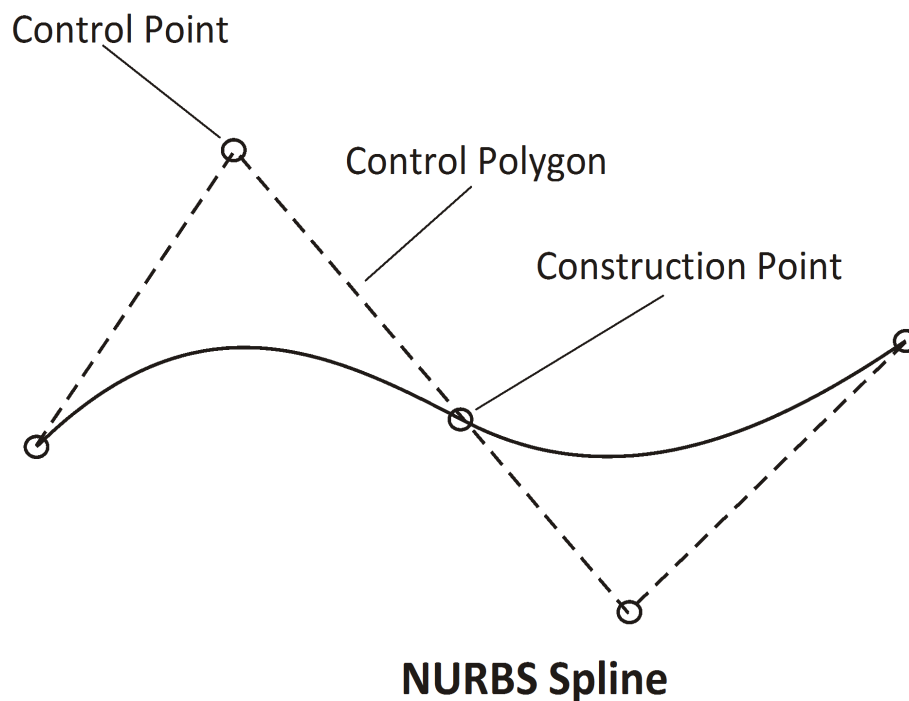
## NURBS SPLINES

**NURBS** stands for **Non-Uniform Rational B-Spline**. **NURBS** are the most advanced method for defining splines and they offer several advantages over parametric splines:

- They are more stable.
- Changes in shape are more localized. Moving a NURBS control point has less effect on the spline far away from the point moved.
- They are more efficient and use less memory.
- Arcs, ellipses, and other conic sections can be represented exactly.

**NURBS splines** pass through a set of construction points. The shape of the spline can be controlled by moving control points located at the vertices of the control polygon.

One disadvantage of NURBS is that only the control points, including the endpoints of the spline, can be directly manipulated after the curve is created. Do not use NURBS if construction points between the endpoints need to be precisely moved after the spline is created.



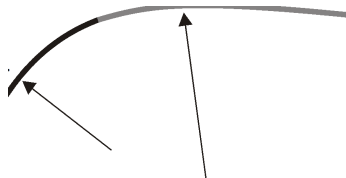
**Continuity** describes how two entities blend together. There are three types of continuity: C0, C1, and C2. As the illustrations show below, C0 continuity can result in a visible seam on the part.

**C0 continuity** means the entities share endpoints, but are not tangent to each other.



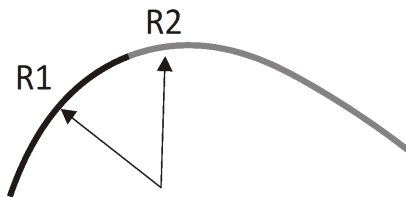
C0: Continuous

**C1 continuity** means the entities are connected and tangent.



1: Continuous & Tangent

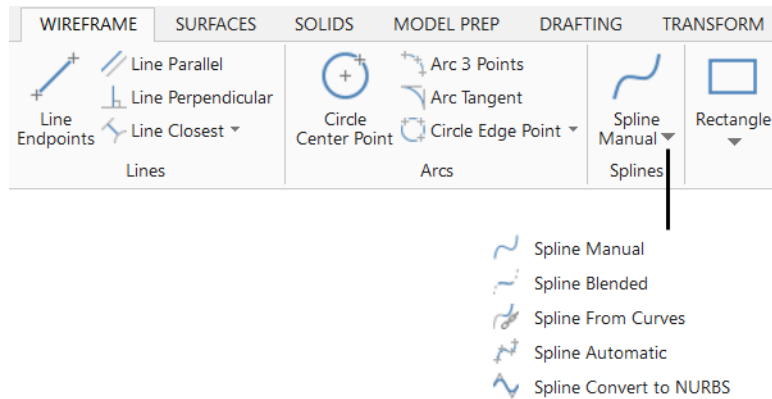
**C2 continuity** means the entities are connected, tangent and share the same initial radius of curvature.



C2: Continuous, Tangent & Same  
Initial Radius of Curvature

## SPLINE CREATION

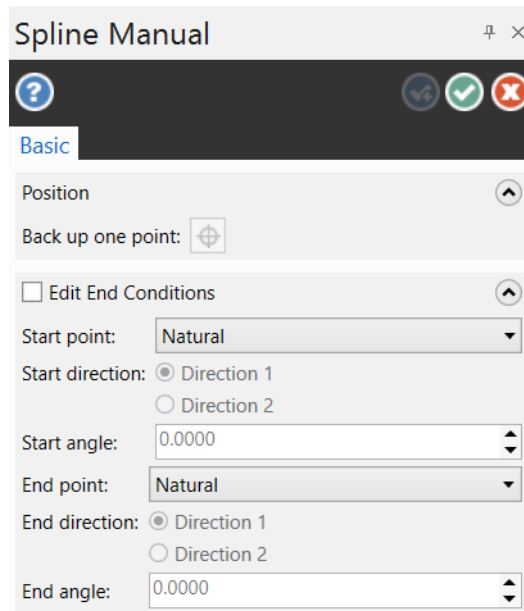
Create splines by selecting the **WIREFRAME** tab, and then the drop down below **Spline Manual** from the **Spline** group.



Item	Definition
<b>Spline Manual</b>	Create a spline by manually selecting construction points.
<b>Spline Blended</b>	Create a spline that blends smoothly between two existing entities.
<b>Spline From Curves</b>	Create a spline from a chain of wireframe entities.
<b>Spline Automatic</b>	Automatically select construction points by picking first, second and last point.
<b>Spline Convert to NURBS</b>	Converts lines, arcs and parametric splines to NURBS splines. Converts also curve-generated and parametric surfaces to NURBS surfaces. NURBS (Non-Uniform Rational Basis Spline) entities require less data storage than parametric entities and they are not associated with their generating curves.

## Spline Manual Panel

The **Spline Manual** panel controls construction point selection and spline End Conditions.



Item	Definition
<b>Back up one point</b>	Allows you to edit the last point you entered.
<b>Edit End Conditions</b>	Allows you to control the start and end points of the spline after all the control points are selected. You can choose between: <b>Natural</b> - sets the point's tangent vector to the optimal tangency condition for minimal curve length. <b>3 Point</b> - sets the point's tangent vector to the endpoint of an arc calculated from the first three points of the spline. <b>To Entity</b> - sets the point's tangent vector based on the tangent vector of a curve at the point you select it. <b>To End</b> - sets the point's tangent vector to the endpoint of the selected curve. <b>Angle</b> - sets the point's tangent vector based on a specific angle.
<b>Start direction</b>	Sets the direction of the spline according to the selection order of the points.
<b>OK and Create New Operation</b>	Create a spline and remain in the Spline panel.
<b>OK</b>	Create a spline and exit the Spline panel.

Select the control points in the order, use the **Backup One Point** button to unselect that last point if needed.

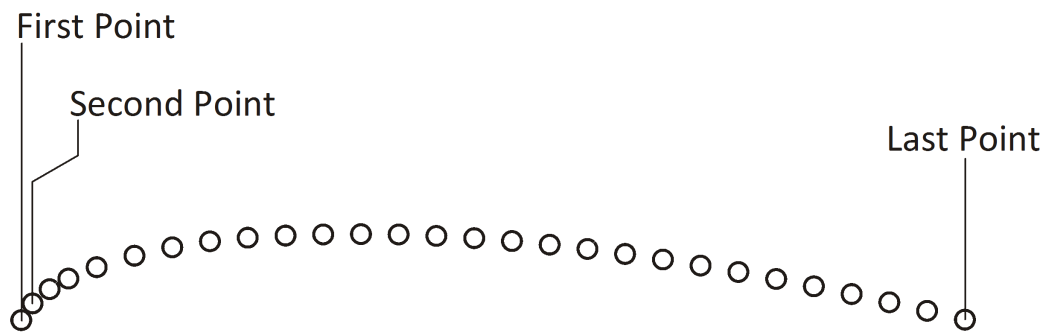
Once all points have been picked, press <ENTER> or click on the **OK and Create New Operation** or **OK** button on the spline toolbar to complete the spline.

You can also complete the spline by clicking twice on the last control point.

## Spline Automatic

Splines that have many construction points that are aligned in a well-ordered path can be selected using the **Create Automatic Spline** function.

Automatic prompts to select the first, second, and last construction points for the spline. It then seeks out all points on the display, picking the next closest until all points have been picked.



Well-Ordered Construction Points



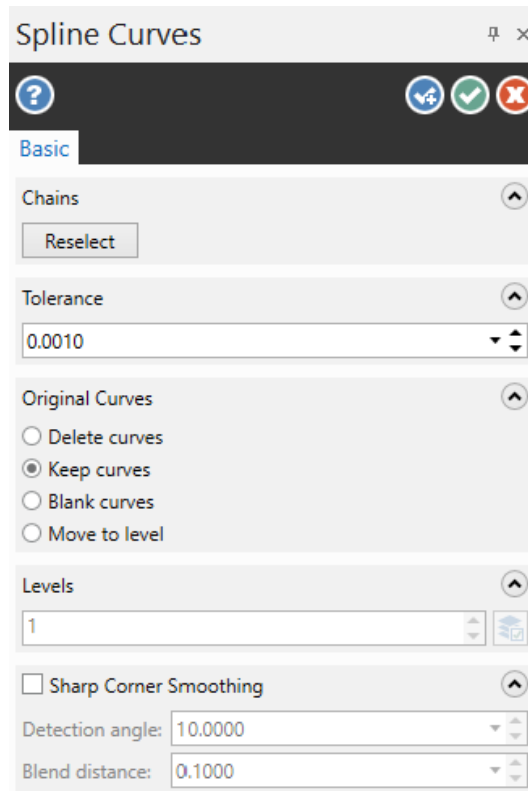
Resulting Spline

If the points do not form a well-ordered path, this function may be unable to determine the correct construction point sequence. In this case, you will have to use manual selection.

## Spline From Curves

**Spline From Curves** creates a spline from a chain of entities. The chain can be made up of lines, arcs, or splines.

This function has applications in both surface and solid modeling. For example, a certain profile of individual entities may simply not work to create a solid extrusion or surface, yet the same chain converted to a NURBS spline will. Converting curves to a spline makes profiles easier to select, since you pick a single entity rather than chaining many.



Item	Definition
<b>Chain Reselect</b>	Reselect the entities used to create the spline.
<b>Tolerance</b>	Chordal deviation sets how closely the resulting spline follows the original curves.
<b>Original Curves Options</b>	<p>Determines what to do with the original curves.</p> <ul style="list-style-type: none"> <li>• Delete curves: deletes the curves</li> <li>• Keep curves: retains curves on the original level</li> <li>• Blank curves: blanks the curves. Use blank/un-blank to recover the curves.</li> <li>• Move to level: moves curves to the level specified in the Level box.</li> </ul>
<b>Levels</b>	<p>Sets the level where the original curves can be moved.</p> <p>Allows you to open the Level Manager and create a new level or select an existing level where the spline will be created.</p>
<b>Sharp Corner Smoothing</b>	Detects and blends sharp corners automatically for a smooth spline.



Item	Definition
<b>OK and Create New Operation</b>	Create a spline and remain in the Spline Curves panel.
<b>OK</b>	Create a spline and exit the spline ribbon bar.

Exercise caution when using this function. Avoid sharp corners, and consider adding a very small fillet radius between non-tangent curves. While **NURBS splines** do a good job of following sharp corners, they can cause problems when used to drive surfaces.

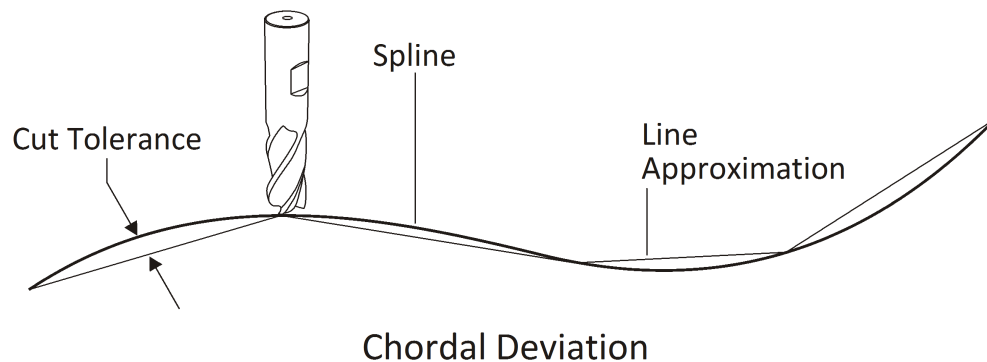
To recover the original curves, use **Level**. While you can use **Edit, Trim/Break, Break Many Pieces** to decompose the spline into lines and arcs, the result will usually consist of more entities than the original curves used to define the spline.

## Tolerance

**The Tolerance or Chordal deviation** is a measure of how accurately lines and arcs are used to approximate the shape of a spline follow the original path. Curve creation tolerance, linearization tolerance, max depth variance, and filter tolerance settings are all based on chordal deviation.

One way to understand chordal deviation is to consider how a spline is machined. Most CNC machines cannot process splines, only lines (G01), and arcs (G02/G03). Mastercam approximates the spline using lines and arcs based on a tolerance setting.

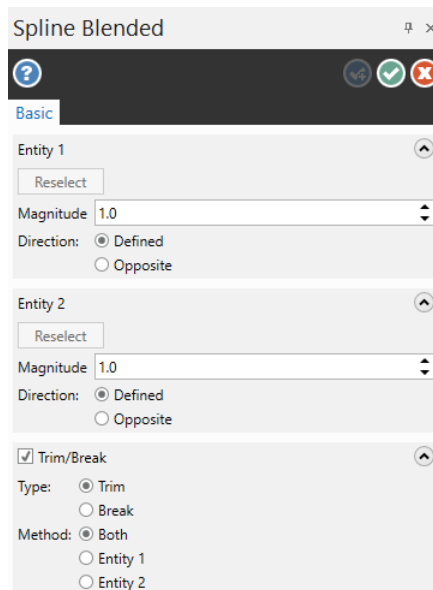
Smaller tolerances result in a path that follow the original spline shape more accurately, but often result in a larger G-code file. There is always a balance between choosing chordal deviation values that meet the part requirements but that are practical in real-world applications.



All tolerance settings in Mastercam are Mid-Tol values, meaning that a value of 0.001 will allow the path to vary up to 0.001 to either side of the spline. Thus, the total deviation is twice the tolerance value. For example, a tolerance of 0.001 means the path can vary as much as 0.002.

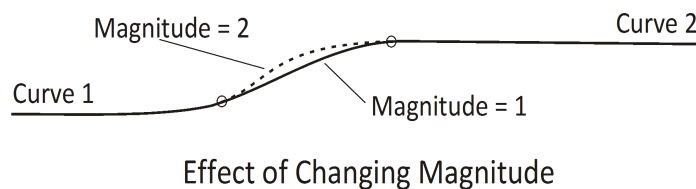
## Spline Blended

Spline blended is used to create a spline between two entities. The result is a spline that transitions smoothly between the two entities and is tangent to both.



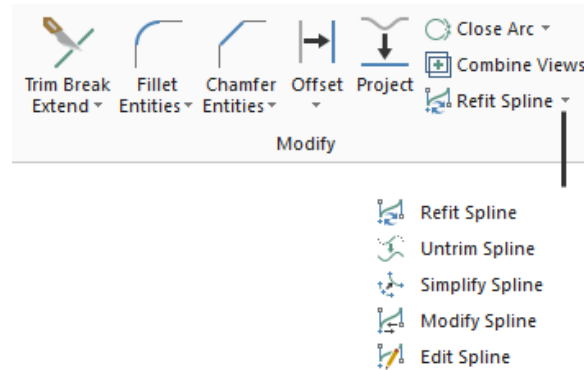
Item	Definition
<b>Entity 1 Magnitude</b>	Controls the initial strength, or "velocity" of the spline at its start point. A larger magnitude creates a greater curvature.
<b>Entity 2 Magnitude</b>	Controls the initial velocity of the spline at its end point.
<b>Trim</b>	Controls what happens to the selected curves. <ul style="list-style-type: none"> <li>• <b>None</b> leaves the curves intact.</li> <li>• <b>Both</b> trims both curves back to the point where the spline.</li> <li>• <b>Entity 1</b> trims back only the first curve.</li> <li>• <b>Entity 2</b> trims back the second curve.</li> </ul>

Mastercam prompts to **Select curve 1**. Pick the first curve to blend and use the mouse to slide the arrow on the curve to where you want the new spline to begin. Repeat for curve 2. Set the magnitude for both until the spline follows the path desired, and then select **OK** and **Create New Operation** or **OK**.



## Alter or reshape splines

You can alter an existing spline using one of the commands from the **WIREFRAME** tab **Modify** group.

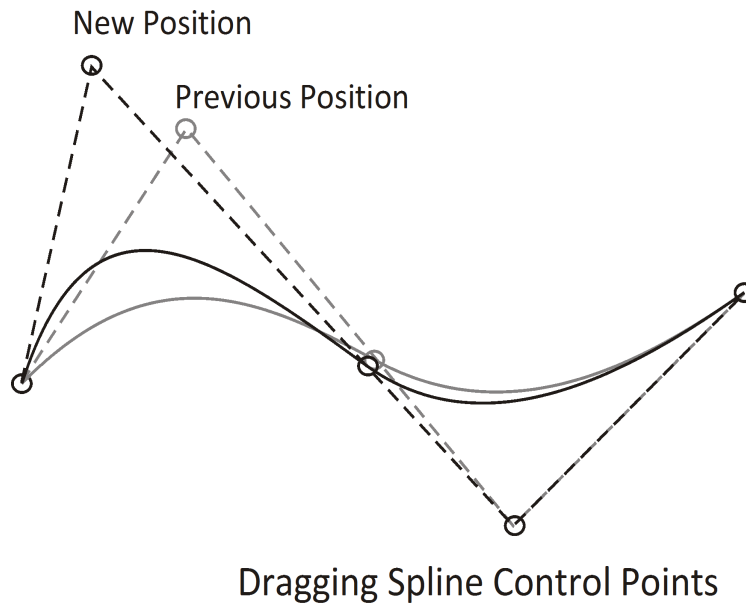


Item	Definition
<b>Refit Spline</b>	Recreates splines that have too many control points or sharp corners into a smoother spline with less control points.
<b>Untrim Spline</b>	Untrims the selected spline to its original extents.
<b>Simplify Spline</b>	Converts circular shaped splines to arc entities.
<b>Modify Spline</b>	Changes the shape of a NURBS or Parametric spline.
<b>Edit Spline</b>	Allows you to refine the spline and to convert lines, arcs and Parametric splines to a NURBS spline. You can move a node point, rotate the tangent vector, or change the length of the tangent vector to edit the curve at the node. You can add, remove or increase the number of control points.

### Modify Spline

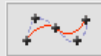
Modify the shape of a NURBS spline by altering the control points of the control polygons or the end points of the spline using **Edit**, **Modify Spline**.

Mastercam will prompt to **select a spline or surface**. Pick the Spline to modify. The control polygon and control points are displayed. Click on a control point and move the mouse to change the shape of the entity. Click the left mouse button to anchor the control point in the new position, and press <Enter> when done.



This function is used to move the endpoints of the spline to connect it to another entity. It can also be used to alter the shape of the NURBS entity to improve the aesthetics of the shape if you have the authority to do so.

The **Modify NURBS** function can also be selected from the **Analyze Entity Properties** dialog box.



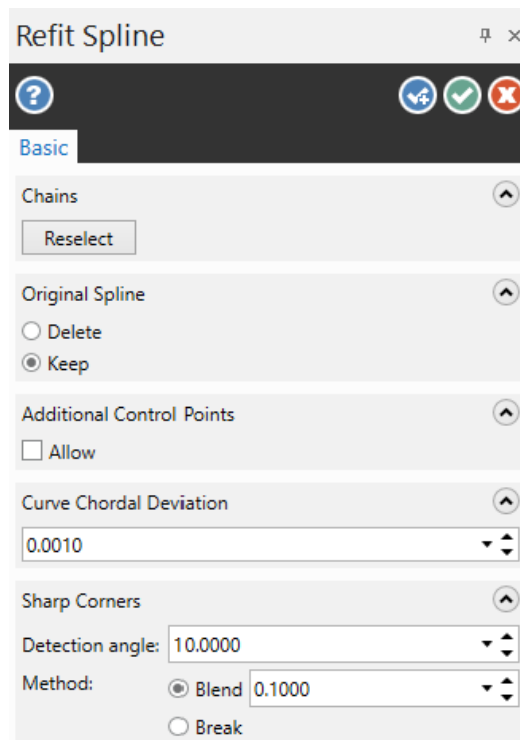
## Refit Spline

While most splines have less than 100 control points, complex splines can have hundreds or thousands. Splines with very large numbers of control points can be cumbersome or nearly impossible to work with. Surfaces created from these splines will also be difficult or impossible to work with, or will require very long toolpath calculation times.

As you will learn later in this chapter, overly complex splines may result in serious problems at the machine as it struggles to process very large G-code files made up of very short moves. Thus, splines with too many control points may produce a poor quality finish.

Use **Analyze, Entity Properties** to check the number of control points in a spline. You may be able to eliminate some of these points and still maintain the shape of the spline within tolerance.

This process is referred to by many terms, including refit, reducing, filtering and removing nodes. Use the **WIREFRAME** tab and from the **Modify** group, select the **Refit Spline** to remove excessive control points.

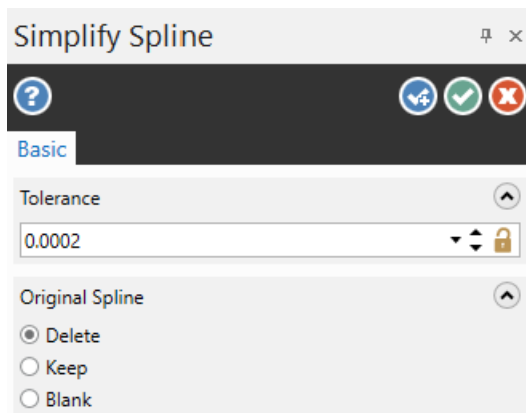


**Undo** does not work for this function, so be sure to save your work first. Enter the **Curve Chordal Deviation** and then select **OK** button.

Use this function cautiously. Be sure to consider all the tolerances that affect the total tolerance of the part and choose a value that is within this allowable error. In rare cases, removing nodes can cause a spline to change shape dramatically. If this happens, read the saved model back in and try using a smaller **Curve Chordal Deviation** value.

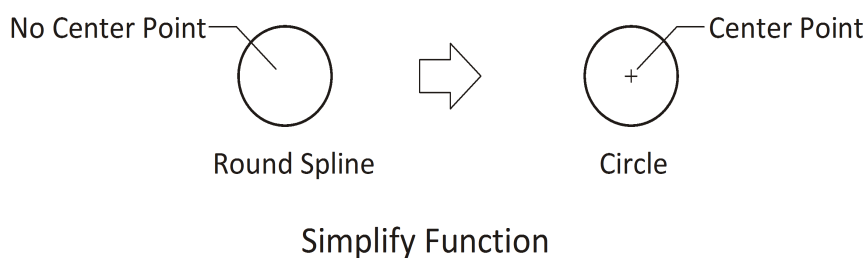
## Simplify Spline

**Simplify Spline** is used to decompose splines into simpler geometric entities. For example, some CAD systems output arcs as splines. While these splines appear perfectly round, they have no center, so they cannot be used for drill locations.



Item	Definition
<b>Tolerance</b>	How much the circular spline can be out of round and still be converted to an arc.
<b>Original Spline Options</b>	Determines what happens to the original splines selected.

Select the **WIREFRAME** tab, and from the **Modify** group select **Simplify Spline** to convert round splines to arcs. Pick the round splines, enter a **Tolerance** value, set what to do with the original geometry, and then select **OK**.

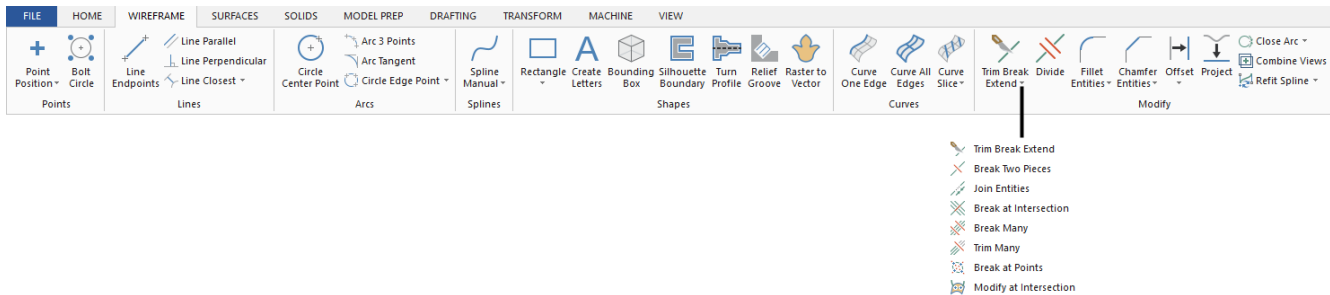


If the error **No splines were converted** appears in prompt line, decrease the tolerance and try again until the message displays the number of splines converted.

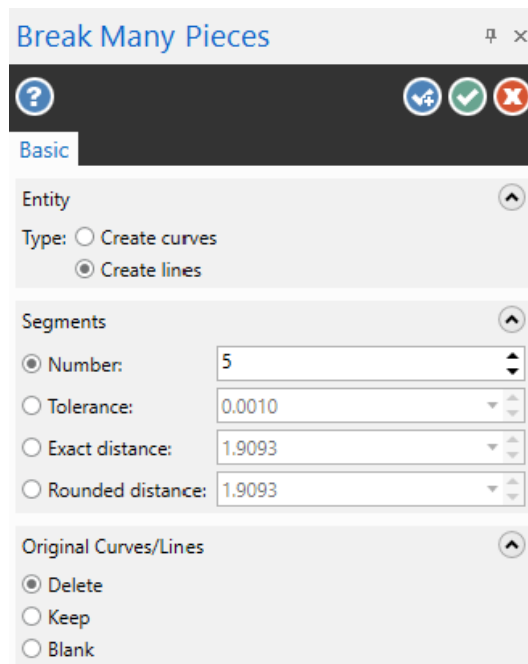


## BREAK MANY PIECES

Use **WIREFRAME** tab, in the **Modify** group, select the drop down arrow below the **Trim Break Extend** and select **Break Many** to convert a spline to tangent lines and curves. This process is sometimes referred to as "exploding" the spline.



Once you select the function the following **Break Many Pieces** panel appears on the screen.



Item	Definition
<b>Entity Type</b>	Determines if the resulting entities will be all lines or tangent curves.
<b>Number</b>	Specifies the number of resulting entities.
<b>Tolerance</b>	Specify a chordal deviation value.
<b>Exact distance</b>	Creates segments of the exact length specified in the distance field.

Item	Definition
<b>Rounded distance</b>	Changes the entered number and/or Distance values so that the function creates segments of all equal lengths.
<b>Original Curves/Lines Options</b>	Determines what happens to the original geometry.

Select a method and value for the result: number, distance, or tolerance. Select the option for handling the original spline. Select whether you want lines or tangent curves as the result, and then select OK.

Tangent curves usually produce far fewer entities than lines. Curves with an arc length of less than .0002 inches (Set in the Configuration, Tolerances dialog box) will be converted to lines even if the curve option is selected.

## TROUBLESHOOTING SPLINES

Most problems with splines are the result of excess construction points. Use **Refit Spline** to reduce the number of node points.

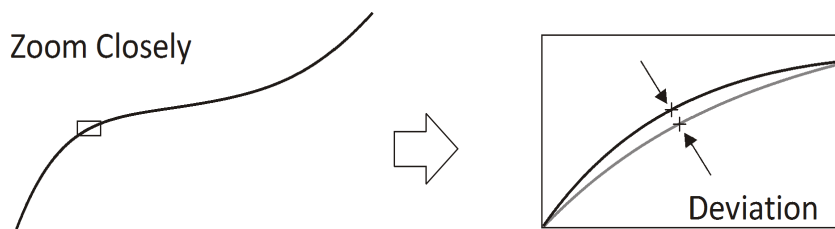
A spline may have a sudden change in direction, sometimes at the extreme ends of the spline. This may be caused by a control point out of sequence or sharp reversal at a control point that causes the spline to double back on itself.

Find these problems using the **Analyze, Analyze Dynamic** function. Drag the mouse along the spline and watch for sudden changes in the direction of the green tangent vector arrow, particularly at the end points of the spline. The spline may be repaired using **Refit Spline** or by completely reconstructing the spline.

In some exceptional cases, you may need to completely recreate a spline. Do this by first using **Create Point Segment** to create points along the spline. Put these points on a different color or layer so they will be easy to delete.

Inspect the points generated to check for excessive points and dense areas of points where the spline may reverse direction. Delete any of these unnecessary or problematic points.

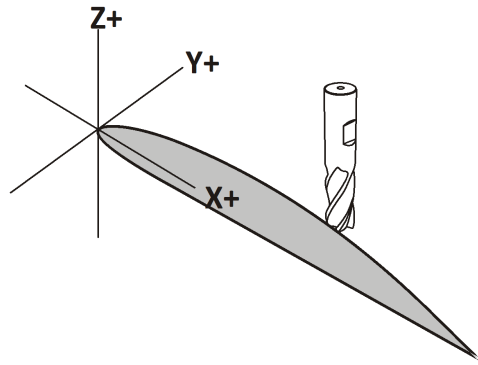
Then use **Create Spline Automatic** to remake the new spline. Make a quick check of the results by zooming very close at regular intervals along the spline. Use **Analyze Distance** to measure the distance between the splines.



Manually Measuring Deviation

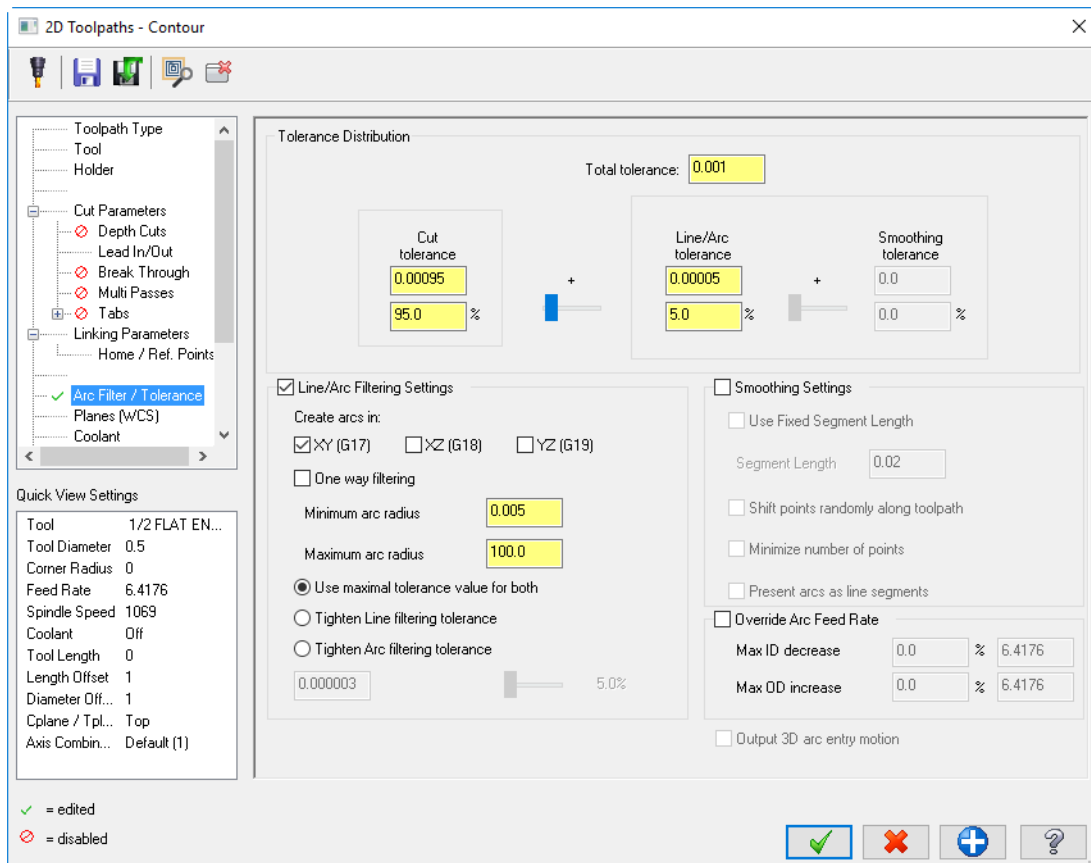
## MACHINING 2D SPLINES

2D Splines lay flat in a construction plane normal to the machine spindle.



Airfoil Rib 2D Spline Example

Machine a 2D spline like any other profile by using **Contour** or **Pocket** toolpaths. Be sure to set the **Tolerance Distribution** settings.



## Tolerance

**Tolerance Distribution** allows you to dynamically adjust the toolpath's total tolerance. Total tolerance is the sum of the cut tolerance and the line/arc and smoothing tolerances. Tolerance distribution can significantly reduce the size of the NC code file and improve machine performance and surface finish.

The screenshot shows the 'Tolerance Distribution' dialog box. At the top, 'Total tolerance' is set to 0.001. Below this, three main sections are shown: 'Cut tolerance' (0.00095, 95.0%), 'Line/Arc tolerance' (0.00005, 5.0%), and 'Smoothing tolerance' (0.0, 0.0%). The 'Line/Arc Filtering Settings' section is checked, showing options for 'Create arcs in' (XY, XZ, YZ), 'One way filtering' (unchecked), 'Minimum arc radius' (0.005), 'Maximum arc radius' (100.0), and radio buttons for 'Use maximal tolerance value for both' (selected), 'Tighten Line filtering tolerance', and 'Tighten Arc filtering tolerance'. The 'Smoothing Settings' section is unchecked, showing options for 'Use Fixed Segment Length' (unchecked), 'Segment Length' (0.02), 'Shift points randomly along toolpath' (unchecked), 'Minimize number of points' (unchecked), 'Present arcs as line segments' (unchecked), 'Override Arc Feed Rate' (unchecked), 'Max ID decrease' (0.0, 6.4176%), 'Max OD increase' (0.0, 6.4176%), and 'Output 3D arc entry motion' (unchecked).

Item	Definition
<b>Total Tolerance</b>	The amount entered in this field, the cut, line/arc and smoothing tolerances are redistributed relative to their current percentages.
<b>One-Way Filtering</b>	Causes the toolpath to be filtered in one direction. This can result in a better surface finish than zigzag filtering for surface machining. It is not relevant for spline machining.
<b>Create Arcs In XY</b>	Determines if arcs are created when possible. You can choose to create arcs in the X-Y (G17), X-Z (G18), or Y-Z (G19) work planes. If your machine is not able to handle arcs in all three planes, de-select the XZ and YZ options.
<b>Minimum Arc Radius</b>	Arcs less than the minimum arc radius are converted to lines. Leave these settings to the defaults unless they are outside your machine parameters.
<b>Maximum Arc Radius</b>	Arcs greater than this value are converted to line segments.

## Applying Tolerances

There are three tolerance settings that affect spline toolpaths.

Item	Definition
<b>Curve Chordal Deviation</b>	This is the tolerance used in defining the spline and is set in the Settings, Configuration, Tolerances dialog box. It is recommended that you do not change the default 0.0002 value.
<b>Total Tolerance</b>	This tolerance is the sum of the cut tolerance and the line/arc and smoothing tolerances.
<b>Smoothing Tolerance</b>	Smoothing improves the quality of the machined surfaces and also creates cutting conditions for the tools, decreasing tool wear.

Very close spline and surface creation tolerances can also cause problems. In extreme cases, the Mastercam MCAM file can become so large that it consumes all available RAM in the computer. If this happens, a small hard drive icon will appear in the lower right corner of the graphics screen to alert you. Once the system starts paging to the hard drive, it becomes much slower.

While a faster computer or more RAM can help, some files can become so bloated that no PC computer is powerful enough to deal with them effectively.

Since a small total tolerance produces a toolpath that closely follows the profile, it is tempting to set all tolerances to the smallest value possible thinking this will produce the best quality part. This is not always the case.

Small tolerances result in larger G-code files. Very large G-code files, particularly those involving very short moves, can cause numerous problems at the machine, including data starvation and acceleration/deceleration effects.

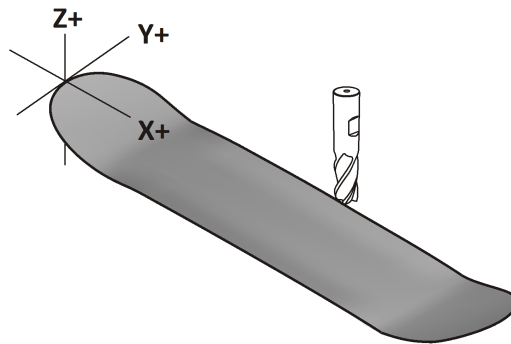
To help guide the decision making process, consider the following rules:

- Use the tolerances on the part print, including surface finish, to help determine the overall tolerance.
- Tolerances are additive. For example, a spline created with a .001 tolerance, machined with a .001 tolerance, and then filtered with a .001 tolerance, has a total deviation of up to .006. This assumes zero deflection of the cutter.
- When rough machining, it is acceptable, use larger tolerances, since all the material left will be machined away by finish operations anyway. A good rule of thumb is 10-20% of stock allowance.
- Use smaller tolerances for finishing operations; typically 5-10 times the surface tolerance.
- When finishing, it is better to err on the side of a slightly smaller tolerance. A total tolerance .001 is clearly visible as facets on many parts. While the finish may be well within the tolerance called out on the print, the part may require a significant amount of polishing to achieve an acceptable visual appearance.
- Surface finish depends on many factors. The quality of CAD data, machine tool performance, tooling, speeds, feeds, work rigidity, and even coolant can greatly affect part finish and precision.

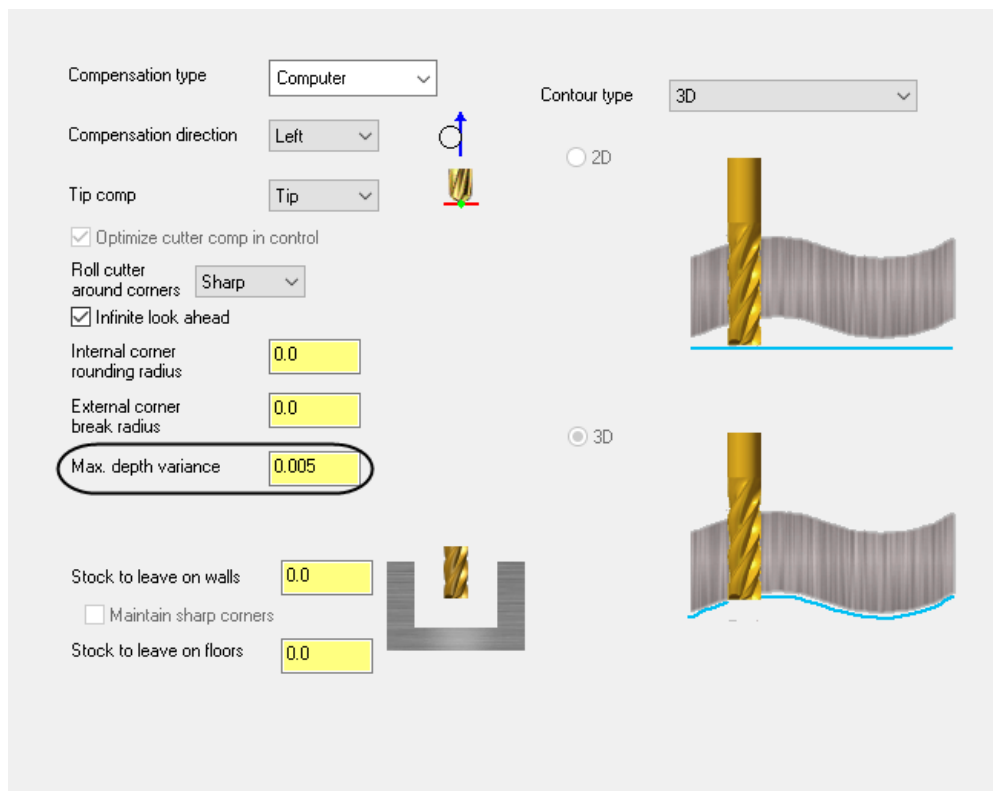


## MACHINING 3D SPLINES

**3D splines** are splines with changing Z-values relative to the machine spindle. Common applications of 3D spline machining include trim cuts of composite molds and O-ring grooves for vacuum fixtures for complex 3D parts.



Skateboard Trim Cut 3D Spline Example



Item	Definition
<b>Max. Depth Variance</b>	This is the tolerance along the Z-axis used in creating the toolpath.
<b>Tip Comp</b>	For a ball nose cutter this sets whether the tip or center of the ball is controlled. In almost every case, this is set to tip, since it is easier for the operator to set the tool height compensation to the tip rather than tool center.

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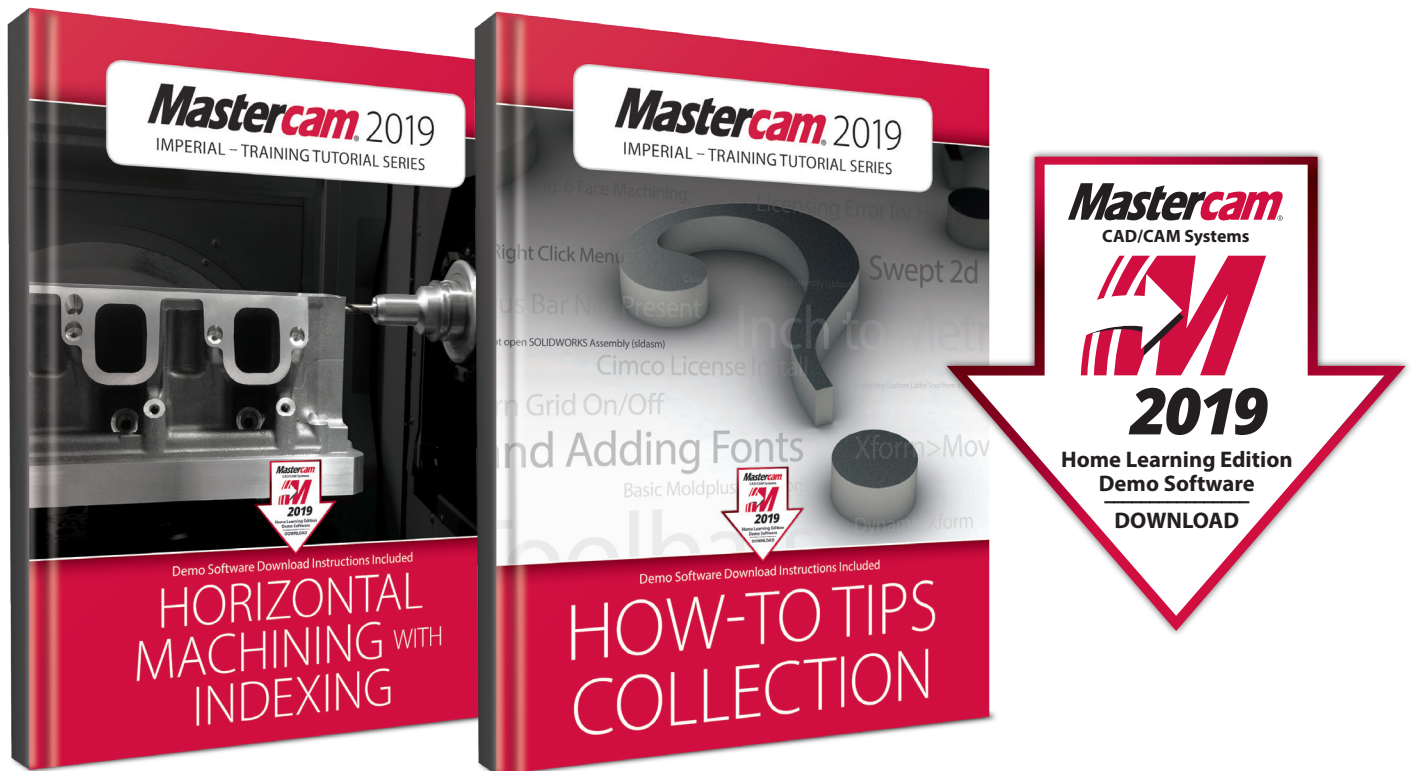
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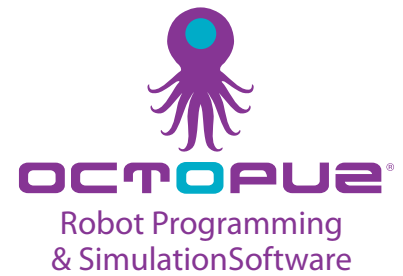
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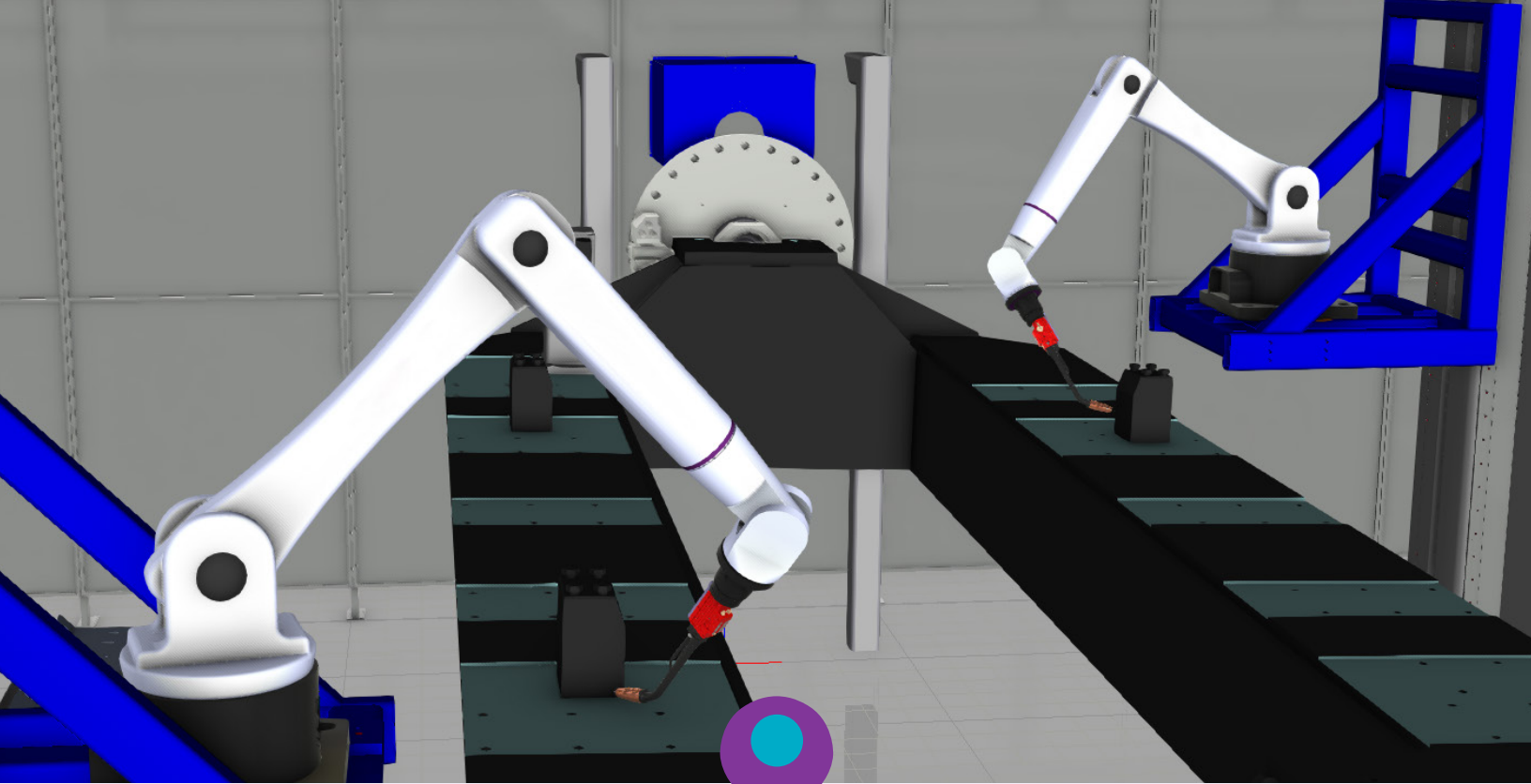
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