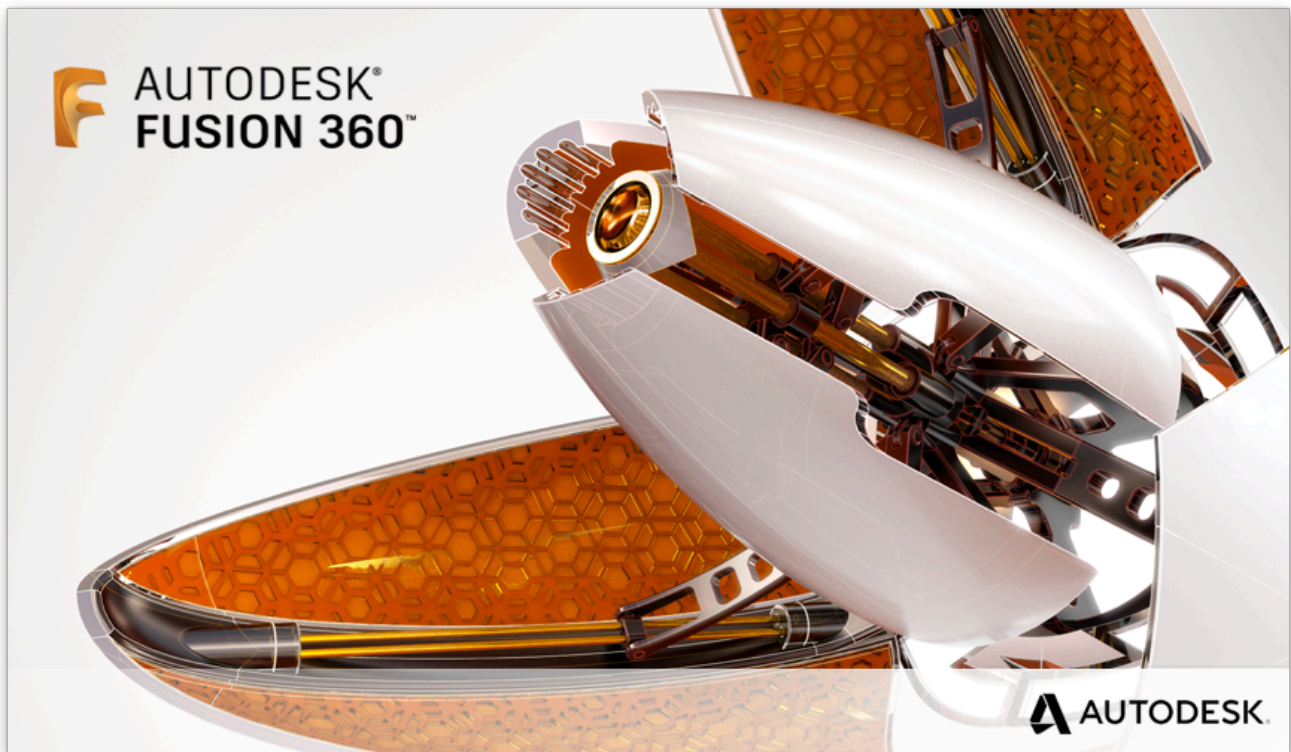


DragKnife Add-On Best Practices with Fusion360



Vin Faraci, Peter Böker

March 2019

Prerequisite	4
Set up.....	5
Tool Setup	7
Geometry Tab.....	8
Heights Tab	9
Passes Tab.....	10
Linking tab	11
Drag Knife tool path generation	12
Inspect the result.....	14
Blade Offset Tuning.....	15
Drag Knife Best Practices	15
Additional tips.....	16
How to swivel a spline.....	18
Supported post processors	22

Prerequisite

You need Fusion360 and be familiar with the manufacture workspace. Base knowledge of CAM and G-Code are also needed to follow each step.

The information in this guide is specific to the Donek Tools drag knife, but it can be used with knives from other vendors too.

More information about the Donek Tools can be found here:

<https://donektools.com>

DONEK TOOLS.com



Set up

We want to cut these simple Letters.

Steps to get the letters in Fusion360.

1. Make a Sketch and use the Text-Tool to write the text.
2. Extrude the text .055in / 1.4mm.

Your text should shown as below.



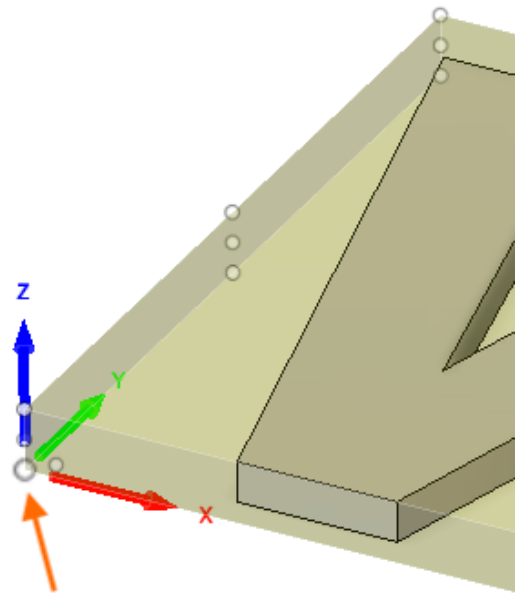
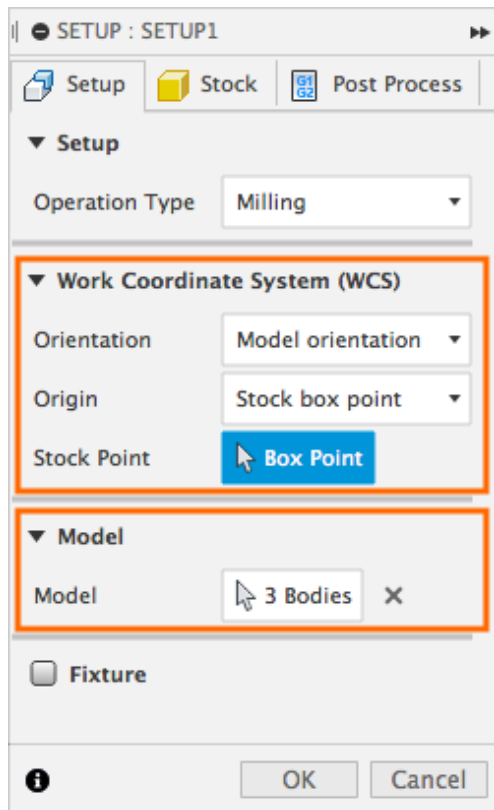
Now switch to the „Manufacture“ workspace and create a new setup.

If you don't know what we want, please read the Fusion360 learning documents that can be found here:

<https://www.autodesk.com/products/fusion-360/get-started>

Make sure you have all three letters in the **Model** section selected!

Pick up the right **Stock Point**! This should be the bottom of your working material.



Really important here is to select the table surface as the Z0 and make sure your stock height is exactly the height of your tool cutting material. Don't rely on the manufacturer data, use a caliper or micrometer and check it yourself.

Tool Setup

You can use any flat end mill tool or make one with appropriate feed rate.

Here some feed rates that recommended by Donek Tools for their DragKnives.

Use these feed rates as a suggestion, it depends on the material you want to cut and what feedrates the machine can handle!

Feedrates		
in/sec	in/min	mm/min
3	180	4600
4	240	6100
5	300	7600

Warning, always turn the spindle off by the machine control or the FU!

2D CONTOUR : 2D CONTOUR2

Tool Geometry Heights Passes Linking

▼ Tool

Tool
#9 - Ø2mm flat (Sorotec Holzfräser)

Coolant

▼ Feed & Speed

Spindle Speed

Surface Speed

Ramp Spindle Speed

Cutting Feedrate

Feed per Tooth

Lead-In Feedrate

Lead-Out Feedrate

Ramp Feedrate

Plunge Feedrate

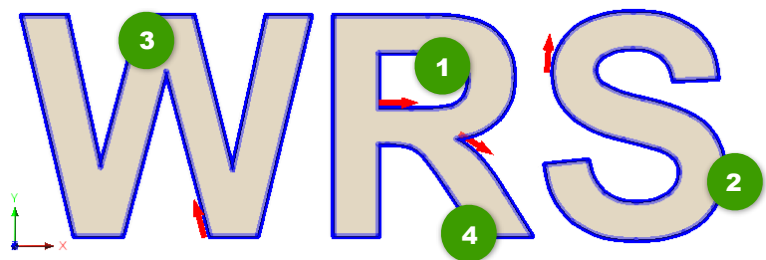
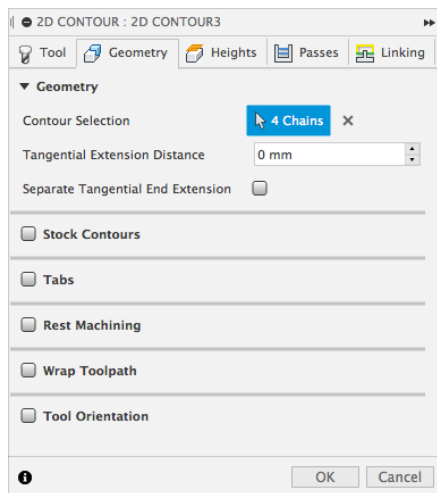
Feed per Revolution

In this dialog you can not turn your spindle off. But for safety reasons lower the **Spindle Speed** to the lowest value your spindle can handle it.

Here, only **Cutting Feedrate** and **Plunge Feedrate** matters.

Geometry Tab

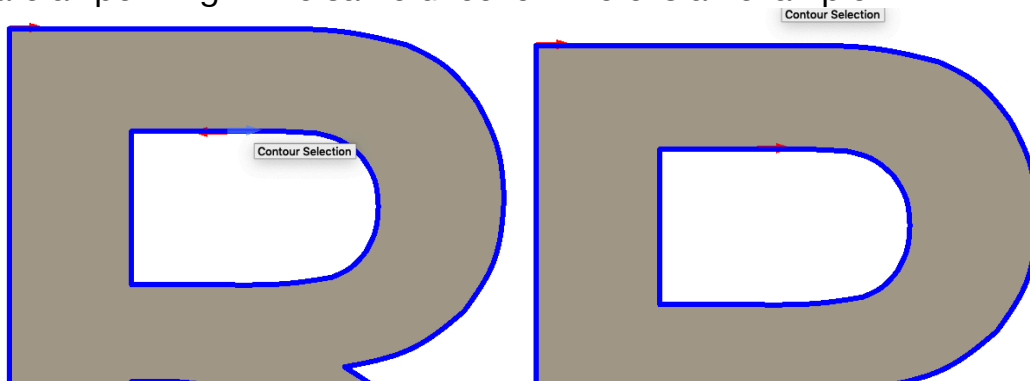
- Select the contours to be cut.
- Be sure to cut interior parts (ie. The center of a letter like A,B,D,O,P, or R) before cutting the outside line.
- Select contours in the order they should be cut.
- Be sure to turn on **Preserve Order** in the **Passes** tab.



You see the selection order that was used for that case. The **Preserve Order** in the **Passes** tab will use this selection order to calculate the tool path.

Keep the rule in mind, inside contours first!

When you select the contours, you must be sure to make sure the red arrows are all pointing in the same direction. Here is an example.



Incorrect.

Correct.

Double check that your direction of cut is the same for all contours. Failure to do this may cause damage to your tool, your machine or yourself!

Heights Tab

- **Top height** (model top 0mm)
- **Bottom Height** (model bottom 0mm)
- The bottom height must be set to zero or the code will not generate correctly.
- If you do not want to fully cut through your material you can adjust the depth in this screen or adjust your offset in Mach 3 to micro adjust your depth of cut. This can be done in the Mach3 offset tab.

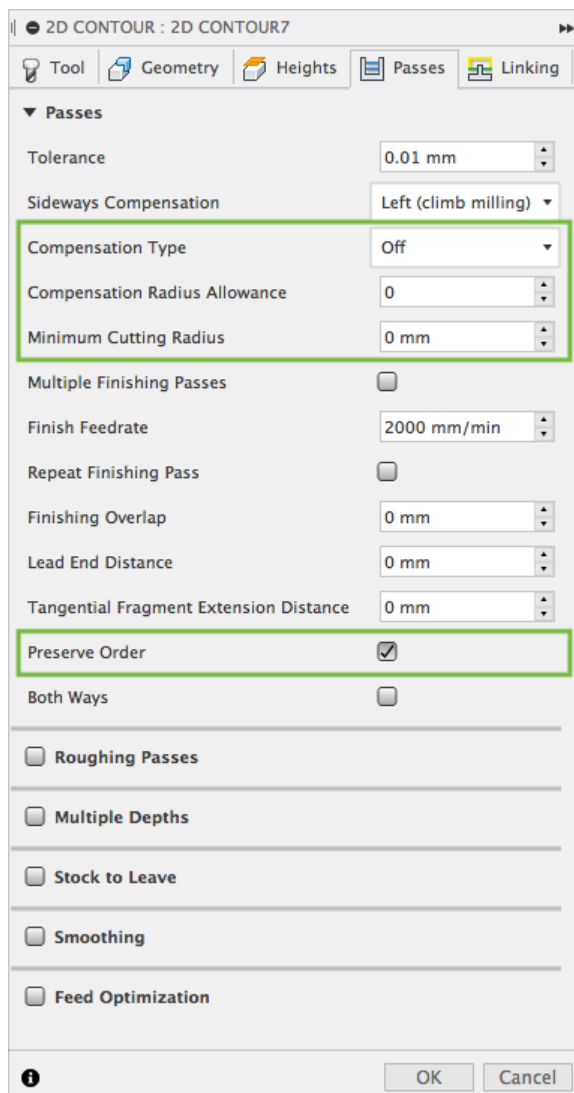
A screenshot of the '2D CONTOUR : 2D CONTOUR7' dialog box. The 'Heights' tab is selected. The settings are as follows:

- Clearance Height:** From: Retract height, Offset: 5 mm
- Retract Height:** From: Stock top, Offset: 3 mm
- Feed Height:** From: Top height, Offset: 2 mm
- Top Height:** From: Model top, Offset: 0 mm
- Bottom Height:** From: Model bottom, Offset: 0 mm

The 'Top Height' and 'Bottom Height' sections are highlighted with blue and purple boxes respectively. The dialog box has 'OK' and 'Cancel' buttons at the bottom right.

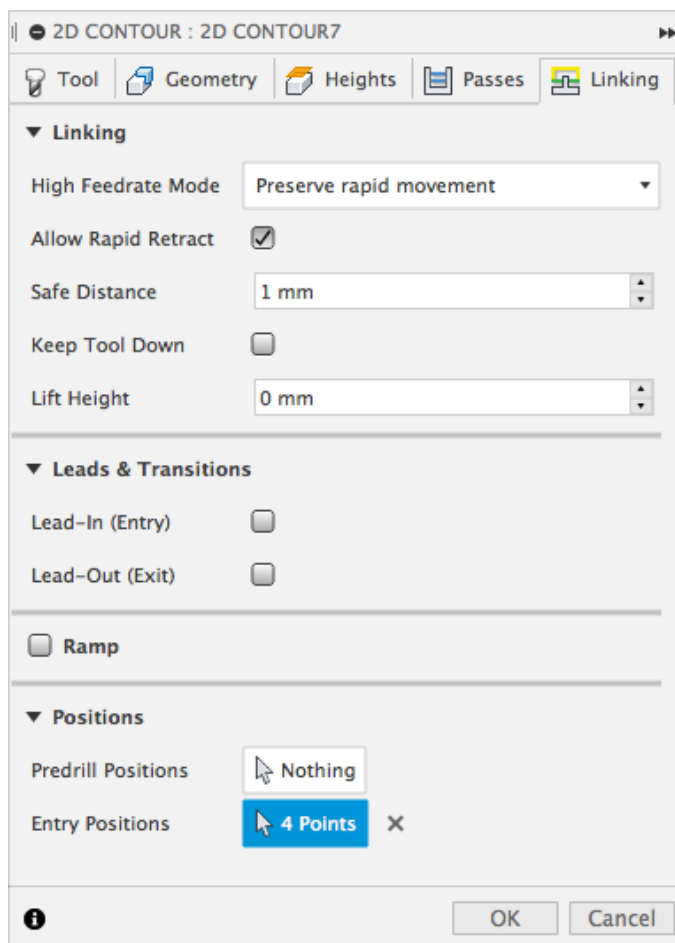
Passes Tab

- Turn **Compensation Type** off so the tool follows the center of the tool path.
- Be sure to cut interior parts (ie. The center of a letter like A,B,D,O,P, or R) before cutting the outside line.
- Be sure to turn on **Preserve Order** in the **Passes** tab if you have interior contours to cut.
- **Smoothing** off.
- **Feed Optimization** off.
- **Multiple Depths**: Multiple depth can be programmed with this add-on. This may be useful when cutting fragile or thick materials, This works in much the same way as regular 2d contour tool path if the entry point is set on a straight line and not at a corner.



Linking tab

- Turn off **Keep Tool Down**.
- Turn off lead in/out in **Leads & Transitions**.
- Turn off **Ramp**.
- Select your **Entry Positions** in the linking tab to start and end your cuts along the center of a straight line to orient your blade correctly.
- For best results your **Entry Positions** should be located in the center of a straight line.
- Adjust the **Entry Positions** on all contours such that they begin and end in the same place. This reduces the need to re-orient the blade at the beginning of a cut and makes your tool paths begin and end on top of each other.

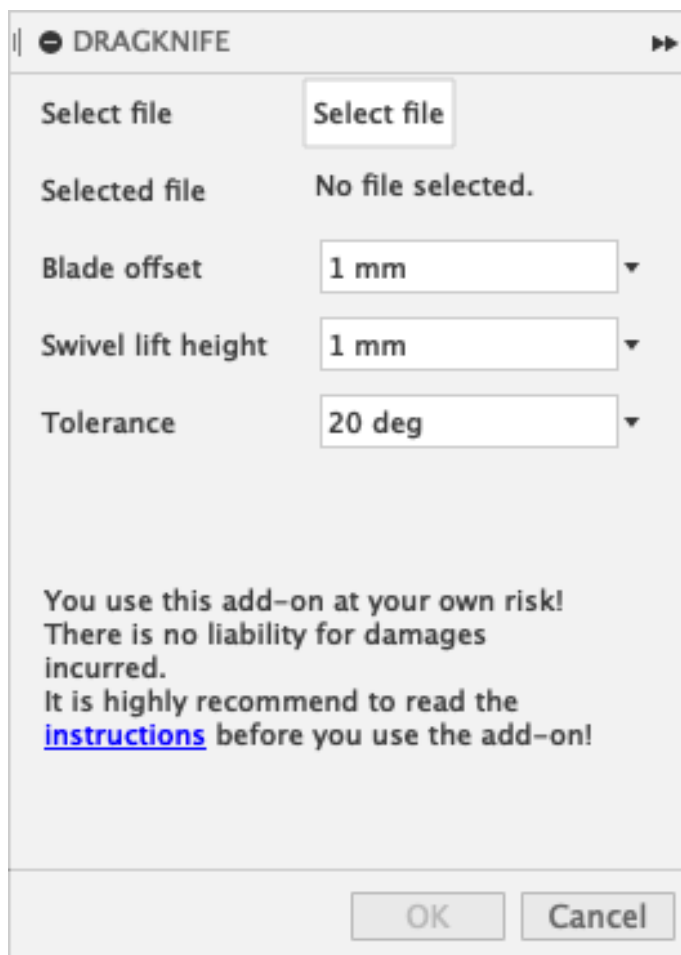
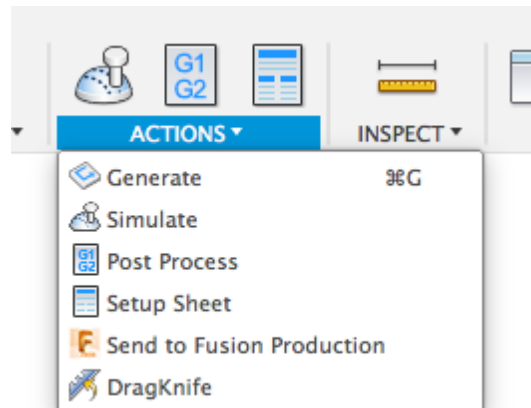


In this case the first cut will be the inner contour of the letter „R“ then, if the cut was done the knife is oriented left to right.

The machine retracts and moves to point 2. While the knife is correctly oriented it is able to cut the letter "W". The same on point 3 and 4.

Drag Knife tool path generation

- After you have input your 2D contour tool path parameters as shown in the example above click OK to generate the tool path
- Post process the file, name and save it. Take note of where this file is saved for use in the following steps.
- In the **ACTIONS** tab on the Fusion360 toolbar select **DragKnife**. This will bring up a new window with the drag knife path parameters.



Select File

Open your 2d contour tool path from the location you saved it.

Blade Offset

Enter your known blade offset value (see below on how to tune your blade offset).

Advice on determining blade offset for your tool and machine.

(refers to Donek drag knives only, but may work for other vendors too.)

See below for info on how to use the blade offset.

- The best starting point for a **Donek D1 or D3 tool** is 0.065in (1.65mm)
- The best starting point for a **Donek D2 or D4 tool** is
 - Standard blade 0.140in or (3.56mm)
 - Safety Blade 0.09 in or(2.29mm)

Swivel Height

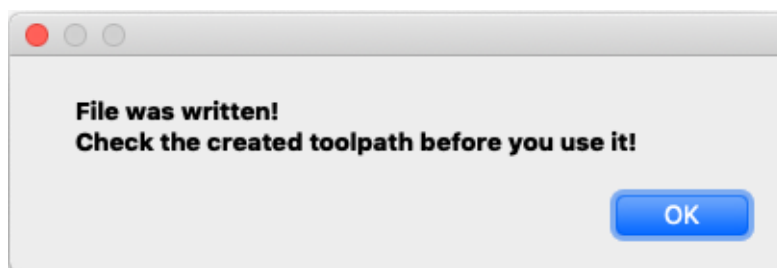
This should be set approximately .05mm below your top height. Be sure to measure your material thickness with a caliper. Do not rely on the manufacturer's specs.

Tolerance

By default this is set to 20 degrees. This is the angle recommended by Donek Tools. You may adjust this as needed but you can probably just leave it as is.

Click **Ok** to generate the drag knife tool path.

You should see this pop up.



Your new file should be save in the same folder and have the extension `_dragknife.tap`.

Inspect the result

To inspect the resulting tool path we recommend NCViewer, a G-Code back plotting tool.

This tool can be downloaded from the Autodesk App store or used online.

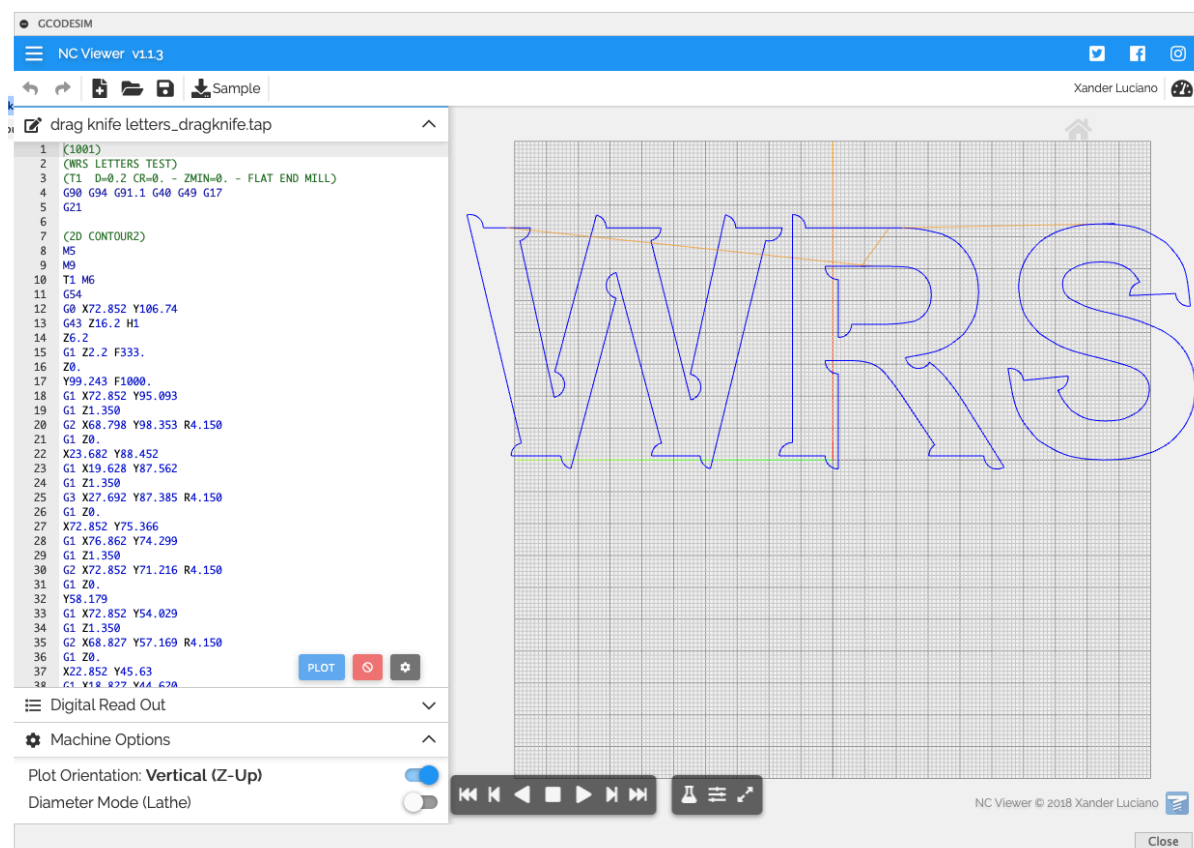
Online version:

<https://ncviewer.com>

Autodesk AppStore:

<https://apps.autodesk.com/en>

Open the resulting code with the ncviewer tool and verify that your corner actions look correct and simulate your cuts.



If your file looks correct you are now ready to load your file to your CNC control program and create some drag knife cuts!

Blade Offset Tuning

Depending on backlash and other factors in your machine, these values may not be accurate enough. The best way to fine tune this value is to cut test pieces. I recommend a 1.5in / 38mm square.

- Observe the tool in action
- If the tool turns too much at a corner, then the offset value is too large
- If the tool does not turn far enough at a corner, then the offset value is too small
- Typically adjustments of 0.005in or 0.12mm are recommended.
- Repeat your test cut until your desired result is met.

Drag Knife Best Practices

- **BEST PRACTICES** (posted by Donek Tool in CNCZone forum)
- Fly cut your table to ensure it is totally flat and true to your machine.
- Zero your cutter to the table surface, not the top of your material.
be sure to model and enter an accurate value for material thickness.

Never rely on the manufacturers stated value, measure the material with a caliper or micrometer.

Additional tips

- If you sketch text within Fusion360 you must extrude it to create bodies or you will not be able to select proper entry points.
- This add-on will not create corner actions on very sharp curves that have been created with a spline tool.

Use caution when creating sharp curves with splines.

You may cause: damage to your material, your blade to break, damage your drag knife or damage to CNC machine.

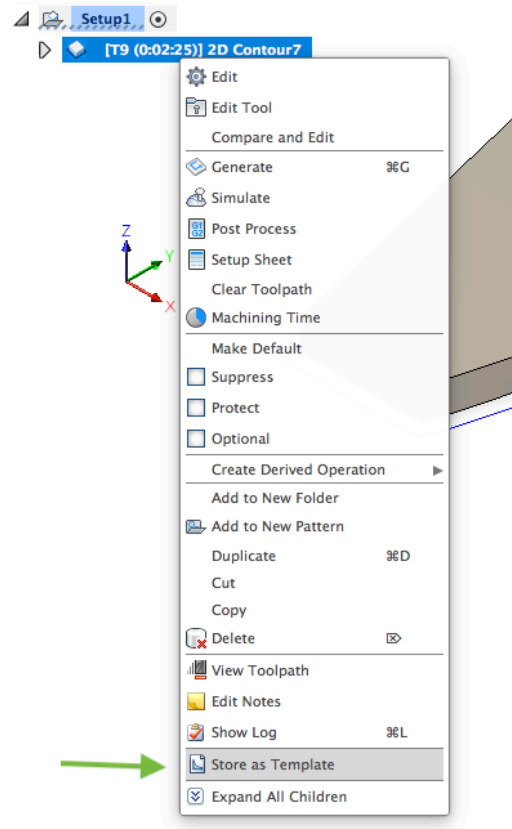
See „How to swivel a spline“ to understand how to modify and use sharp spline curves.

- Be sure your blade is seated properly in the holder or this may alter your blade offset and create unexpected results in your cut.
- If you change brands of utility blade, you may need to re-calibrate your blade offset.
- This Add-on has deleted the G-code to start your router/spindle but you should turn off your router/spindle for safety!

Double check that your router / spindle is off!

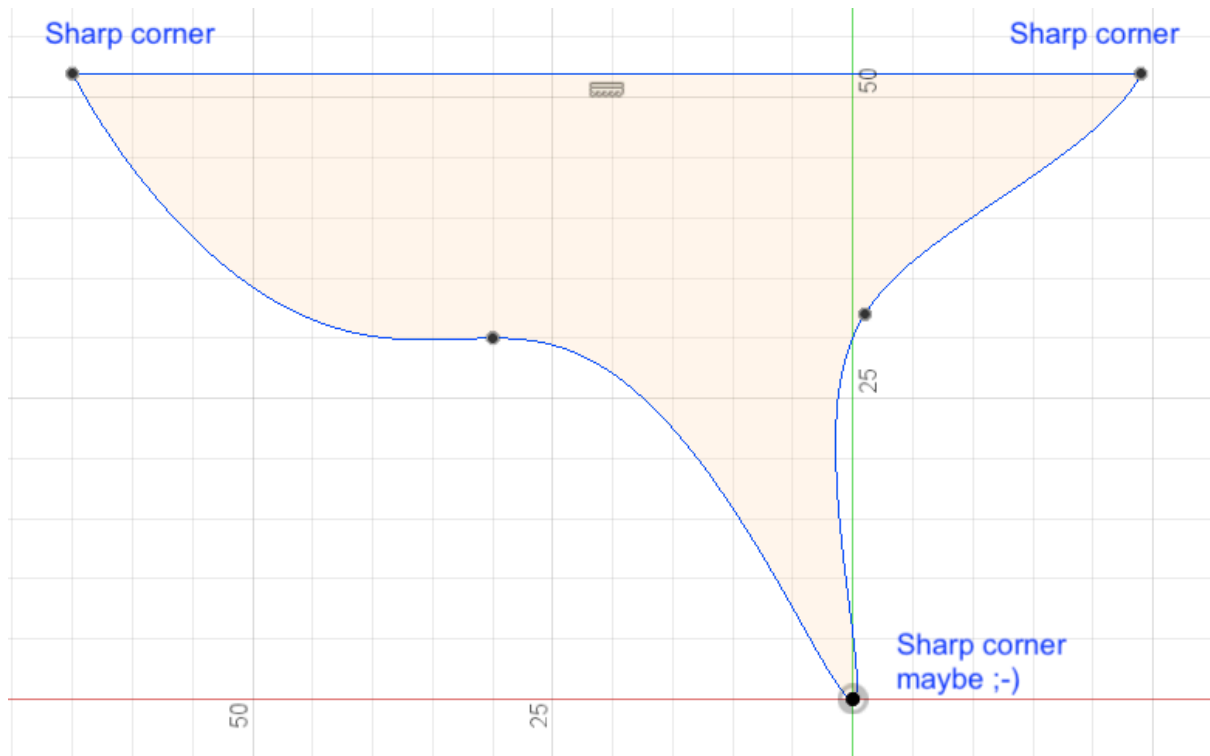
- When cutting your material place the surface that will not be visible in the final product face up on your cutting surface. This will hide any small marks created by the swivel action of the drag knife in your final product.
- Save your contour operation as a template to save time on your next drag knife project.

To save it as a template right click on the contour operation and in the context menu click on „**Store as Template**“



How to swivel a spline

Here we would like to discuss how to create a swivel action when using splines. The shape is simple and overall not a problem, but it has a section that seems to have a sharp corner.

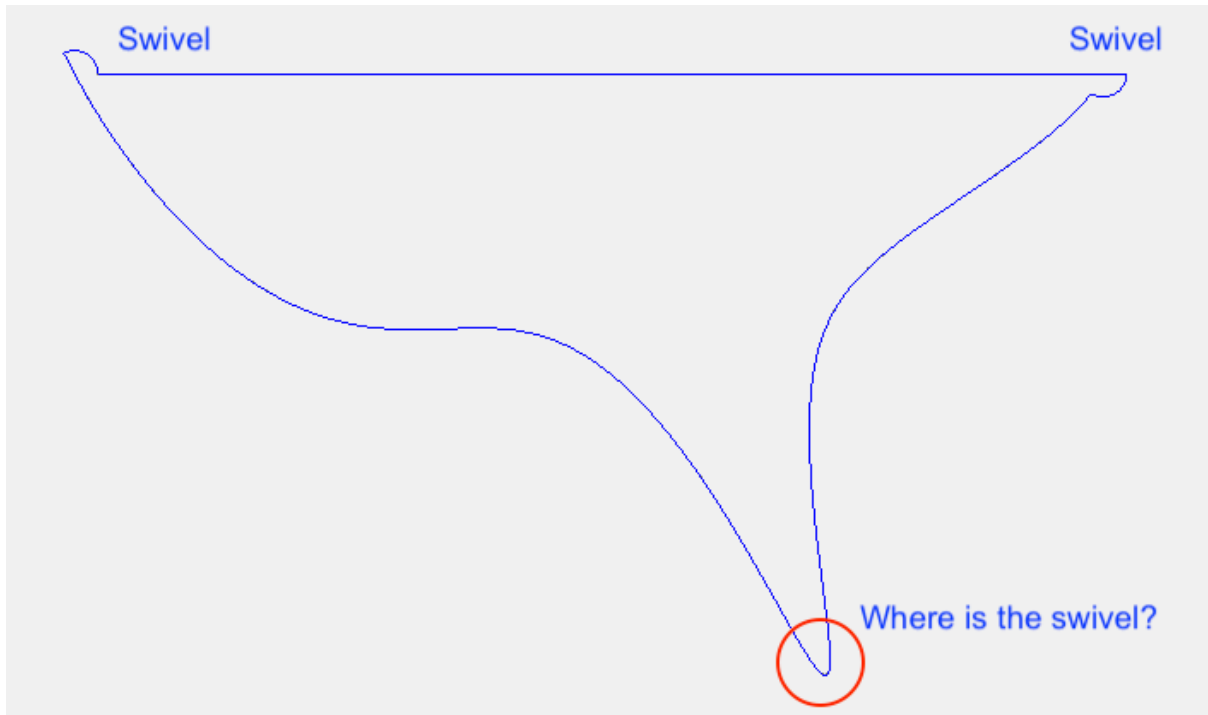


Customer shape.

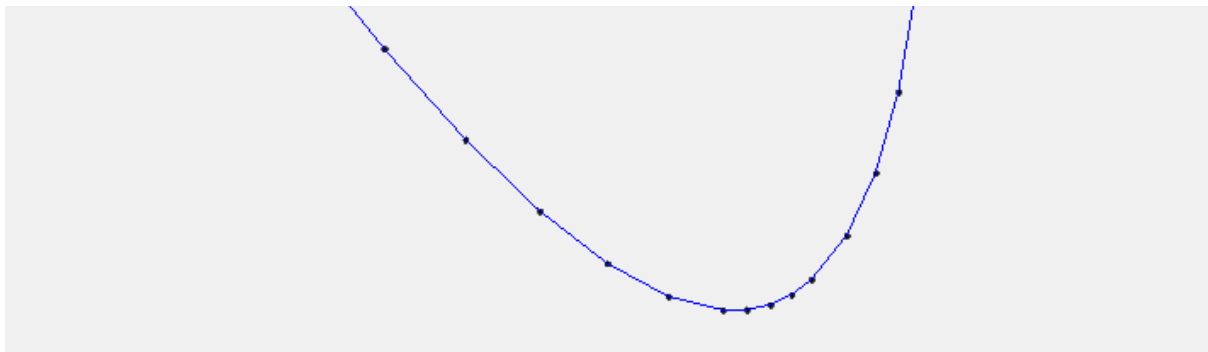
So far you created the setup and the operation (Hopefully you have done this from a template, this save a lot of time) and you post it.

Next open the DragKnife Add-On and prepare the DragKnife compatible tool path with it.

While inspecting the tool path with ncviewer.com you notice a corner that did not produce a swivel. Why the heck is the sharp looking corner not swiveled?



If we look deeper into the problem and turn on the points in ncviewer we can see the reason for that behavior.



Tiny segmented lines!

Through the post process the splines are converted into tiny little line segments. The sharper the »corner« the more tiny segmented lines are produced by the post-processing. This is the only way a cnc machine can follow such splines. But why is no swivel created? The simple answer is there is no corner here that exceeds the tolerance setting in the DragKnife Add-On.

Could we reduce the tolerance? Yes of course, but tolerance setting is a fine tuning tool.

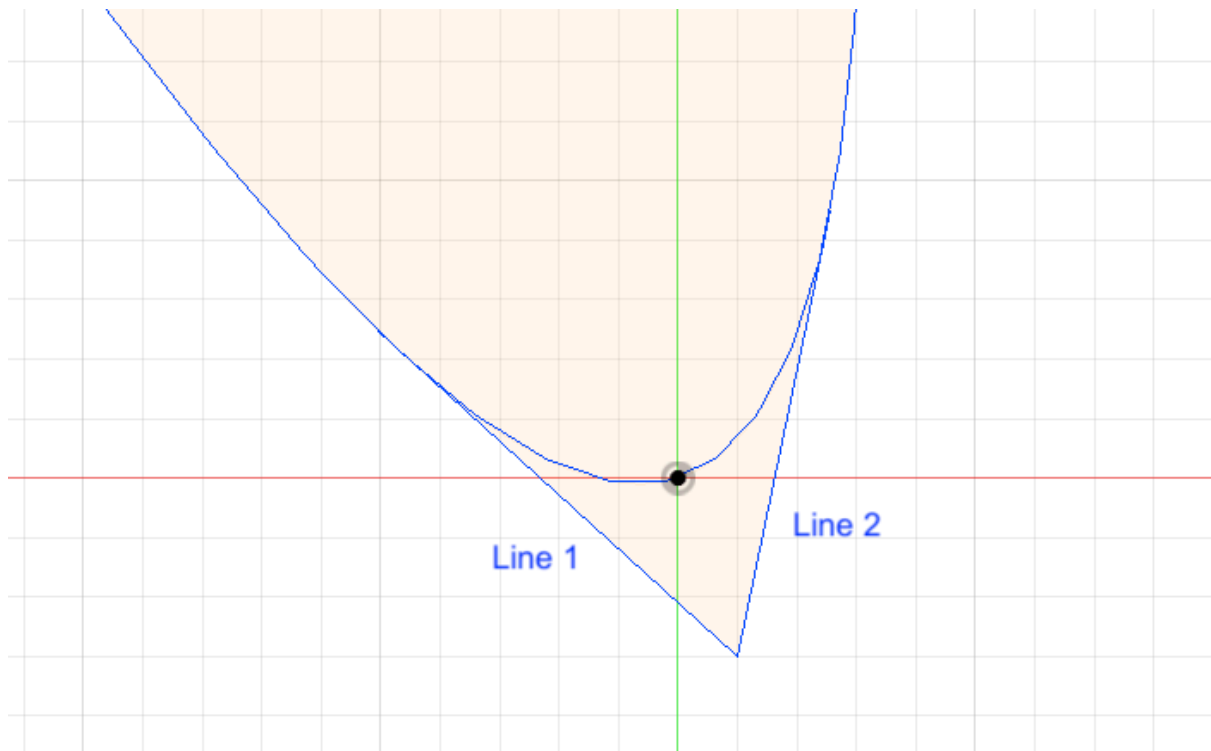
If you lower the tolerance too much this will create too many swivel actions. This is fun to see on a cnc but less productive.

There are two ways to get a productive solution.

- A. Edit the sketch to get a real corner that overreaches the tolerance.
(Recommend)
- B. Edit the extruded body to get a real corner.

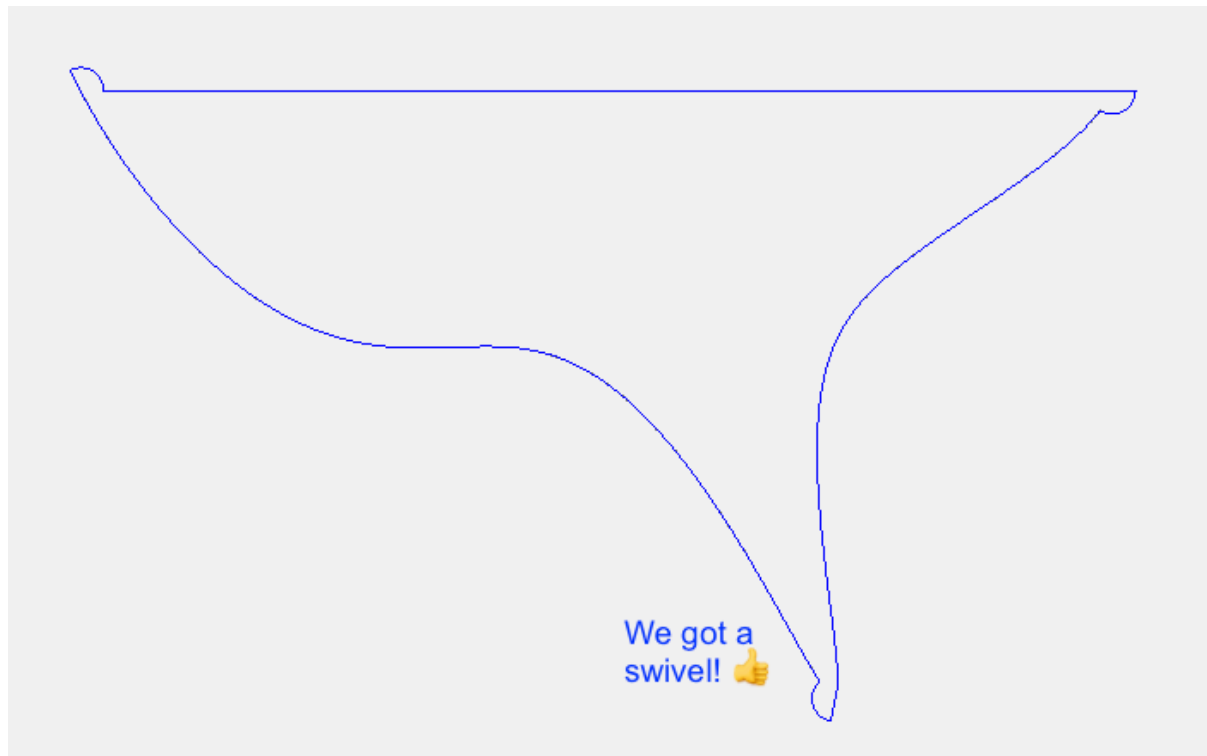
In our case, two lines are added to get a sharp corner.

We prefer »A«.



Don't forget to trim the rest of the spline between the two lines.

After post processing and preparing the code with the Add-On we have the following result.



This example shows that it is possible to get productive results with splines. It takes more time to get a convenient result but in the end you have proper G-Code that can't wait to run on your machine!

Keep always in mind »Safety first!«

Supported post processors

A list of fusion360 post processors that can be used with the DragKnife add-on.

- Biemmepi
- CNC Router Parts
- Eding CNC
- FlexiCAM
- Grbl
- Intelitek
- LinuxCNC (EMC2)
- Mach3Mill
- MultiCam ISO
- Opticut
- Thermwood 3-Axis
- Tormach PathPilot
- WinCNC