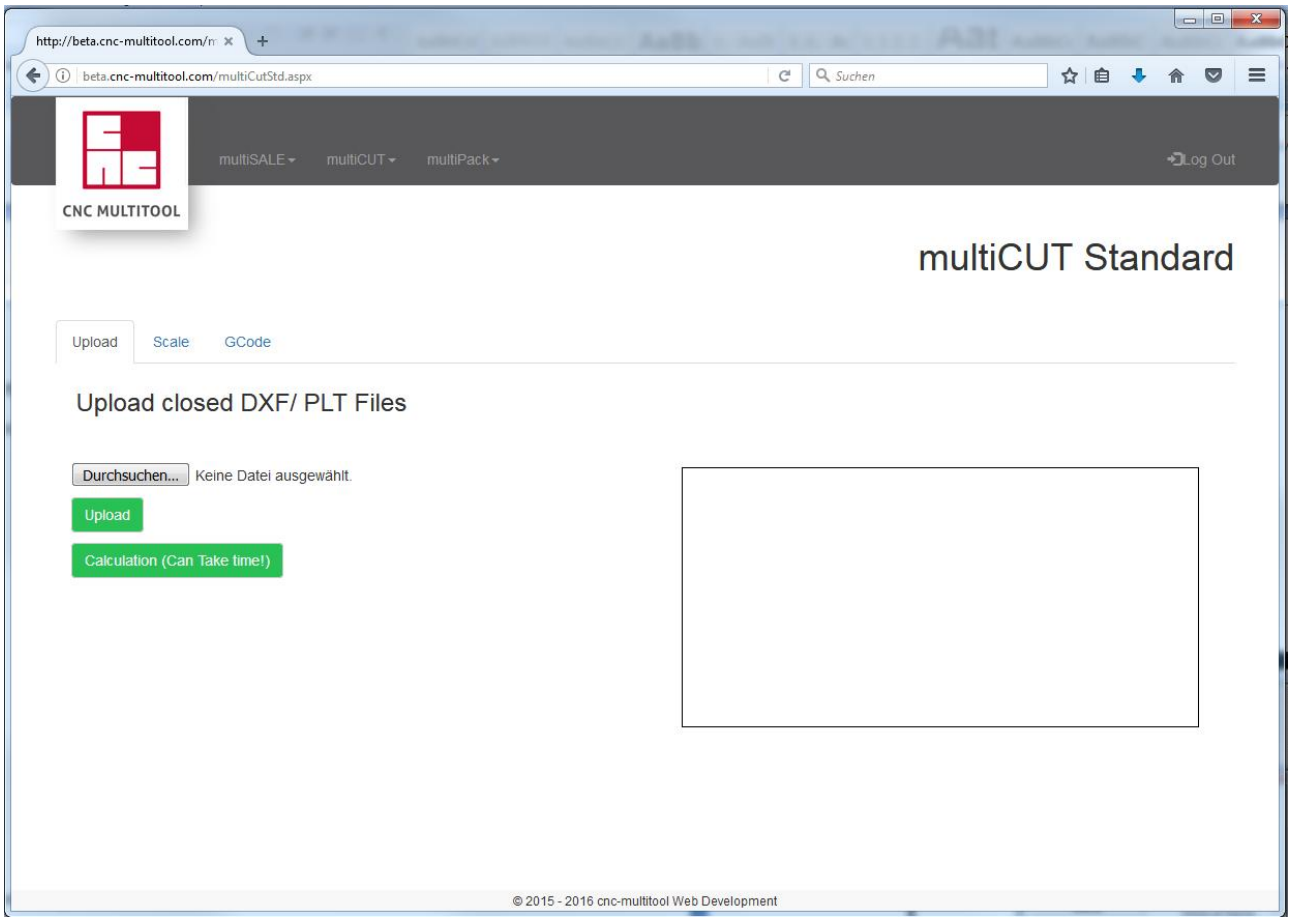
cnc-multitool.com

multiCUT - standard

NOTE: This software is still in beta test status. Possible errors or short-term failures are possible.



Introduction

multiCUT is " Software as a Service " (SaS) and comes out of the cloud.
It is a web-based program that runs in modern browsers such as Internet Explorer, Firefox, or Chrome.

The program is operating system independent and works on any Windows or Mac computer.

cnc-multitool.com

Feature

With multiCUT standard you can create your 2D cutting data quickly and easily.
The software supports individual closed contours as well as multiple closed contours.

In addition, it is possible to create the cutting data for whole plates and blocks.

1. Import of single or multiple closed 2D contours in PLT / DXF or DWG format.
2. Display contour data in the browser window incl. Zoom and move
3. Proportional scaling over the height or length of the 2D object
4. Calculate the cutting data as IsoGCode
5. Save the editing data locally on your computer.

cnc-multitool.com

This tutorial explains the cutting of 2D contours.

The procedure and the sequence are always the same, irrespective of whether this is a single 2D contour or several contours, e.g. Circles or letters.

to create a drawing

The drawing is the basis for working with a CNC-styrofoam machine.

We recommend to create drawings using the following programs:

a. Technical drawings
CAD program for 2D contours of all kinds (e.g., AutoCad)

b. Letters, logos
CorelDraw, Adobe Illustrator (AI), etc.

In general, all vector-based programs that can create DXF / PLT or DWG files can be used.

Note!
Draw drawings always on a scale of 1: 1 in the correct sizes.
So if you want to cut a square in the size 100x100m, please draw this also in the size 100x100mm!

cnc-multitool.com

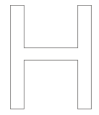
What can you cut?

Your drawing can consist of a simple single contour, a contour with interior cut-outs or variants thereof.

IMPORTANT!
All contours must be closed. Open contours are only divisible with the module *multiCUT open*.

Possible contours and contour variants

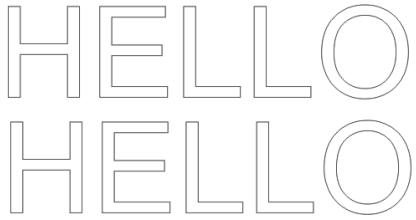
1. a single contour



2. a single ***closed*** contour with an inner contour (s)

3. several ***closed*** contours with and without inner contour (s)

4. several identical rows with ***closed*** contours with and without inner contour (s)

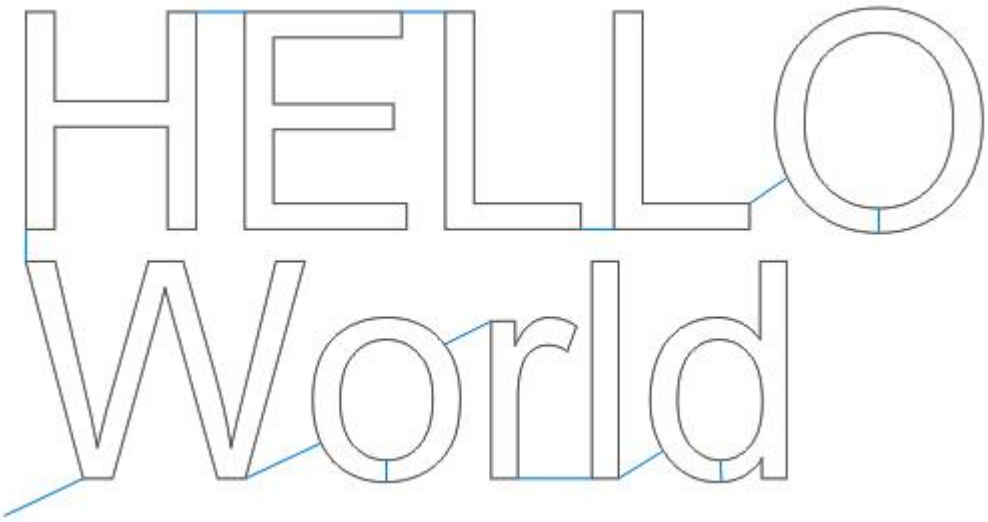


5. any ***closed*** contours with and without internal contour (s)

How should the data be prepared?

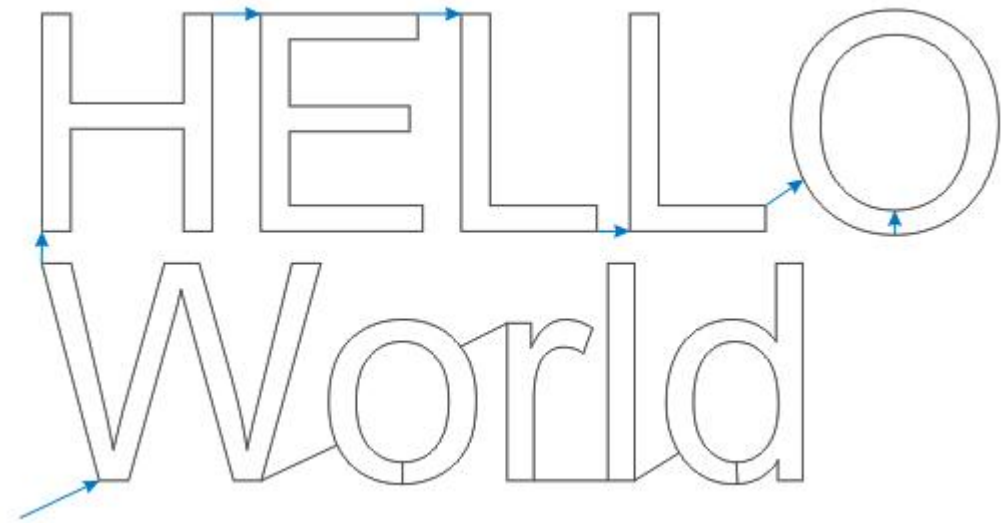
In the case of a CNC Styrofoam cutting machine, the starting point is always also the end point of the cut. For this to work, the individual contours must be connected with lines.

Here's an example:



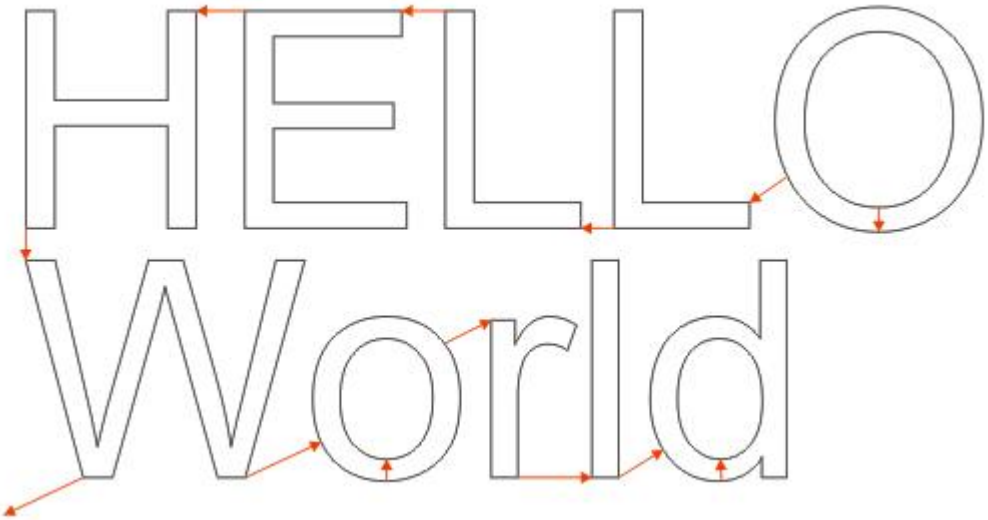
The starting point in most cases is at the bottom left.
Always cut clockwise (done automatically in the software)
Always cut the top first and then the bottom.

Info!
For illustration, we have made arrows out of the connecting lines.
Follow the blue arrows clockwise all the way to the letter “O”.



START

And then the red arrows for the return path and the rest back to END



END

Export drawing

After you have created the drawing, you must export or save the drawing in the DXF / PLT or DWG format.

We recommend to export the files the following:

DWG -Files: AutoCAD 2004-2007 format
DXF -Files: AutoCAD 2004 format

Some programs have difficulty creating a "clean" DXF file, which cannot be detected by multiCUT software. In this case, please experiment with the DXF export functions of your software. Sometimes double line above each other can cause problems too.

Corel Draw Note:
CorelDraw versions like X3 have difficulties exporting to the DXF format. Please use the PLT format. Please ensure that the curve resolution is as low a s possible at e.g. 0.0mm when exporting to the PLT format

Adobe Illustrator Note:
Most if not all Illustrator versions have difficulties exporting to the DXF format. Please either get a second party plugin to export to DXF format or export to EPS format and use Corel Draw to export to PLT format.

cnc-multitool.com

What is the Iso-GCode?

Our styroCAM software is used to create the machine program from your exported DXF / PLT / DWG drawings.

This machine program is then loaded into the machine software (e.g., Mach3). It includes the Iso-GCode with the traveling instructions for your machine.

In general, the Iso-GCode for our CNC Styrofoam machines consists of the following instructions:

G01 X Y A B C F instruction for defined cutting at a speed F
G00 X Y A B C - instruction for traveling at maximum speed

X / Y / A / B / C are the axes, F is the speed.

X = forward / backward left unit
Y = high / lower left unit
A = forward / backward right unit
B = high / lower right unit
C = axis of rotation (optional)

G01 X100 Y150 A100 B150 F450

Your machine drives the left (X / Y) and right (A / B) machine unit at a speed of 450mm / min to the same coordinates and your wire is parallel.

G01 X100 Y150 A200 B200 F450

Your machine moves the left (X / Y) and right (A / B) machine unit at a speed of 450mm / min to different coordinates and the wire is then oblique.

G00 X0 Y0 A0 B0

Your machine moves the left (X / Y) and right (A / B) machine unit back to the zero point at maximum speed

Wire firing / wire compensation

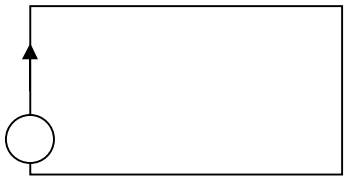
Styrofoam cutting is forceless cutting. The wire cuts / melts the material away without touching it.

This produces a cutting gap which is larger than the wire. This cutting gap has to be taken into account when cutting, means burn-off and must be compensated, since the object otherwise becomes too small.

The cutting speed and wire temperature influence the size of the cutting gap. We therefore recommend working with a 2mm cutting gap.

a. Blank without compensation for burn-off

Wire cuts on the contour in the direction of the arrow



Cut is too small, about 1.5 to 2mm around is missing.

b. Cutting with compensation

Wire cut to the left of the contour in the direction of the arrow

Cut is correct, about 1.5 to 2mm around was compensated.

Why 2mm? It is easier to define the cutting gap and adjust the machine accordingly.

Make a pre-cut. Draw a rectangle in the size of e.g. 200x200mm. Calculate the cutting file and cut it at a fixed speed F450 (450mm / min)

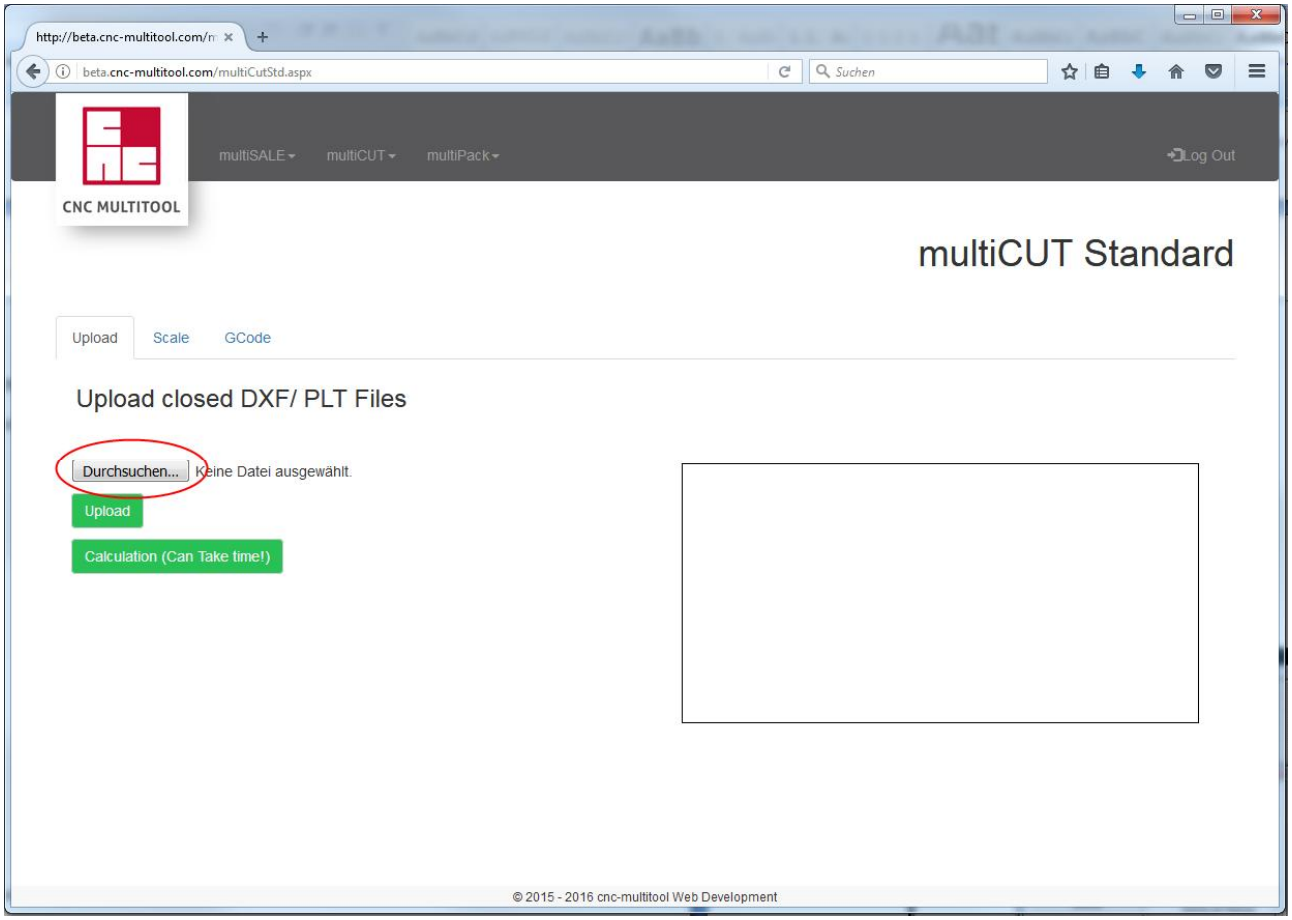
When cutting, adjust your power supply so that the cutting gap is 2mm.

Now you have the basic settings of speed, temperature and cutting gab. You can now easily change the speed at any time and adjust the power supply accordingly.

multiCUT - standard

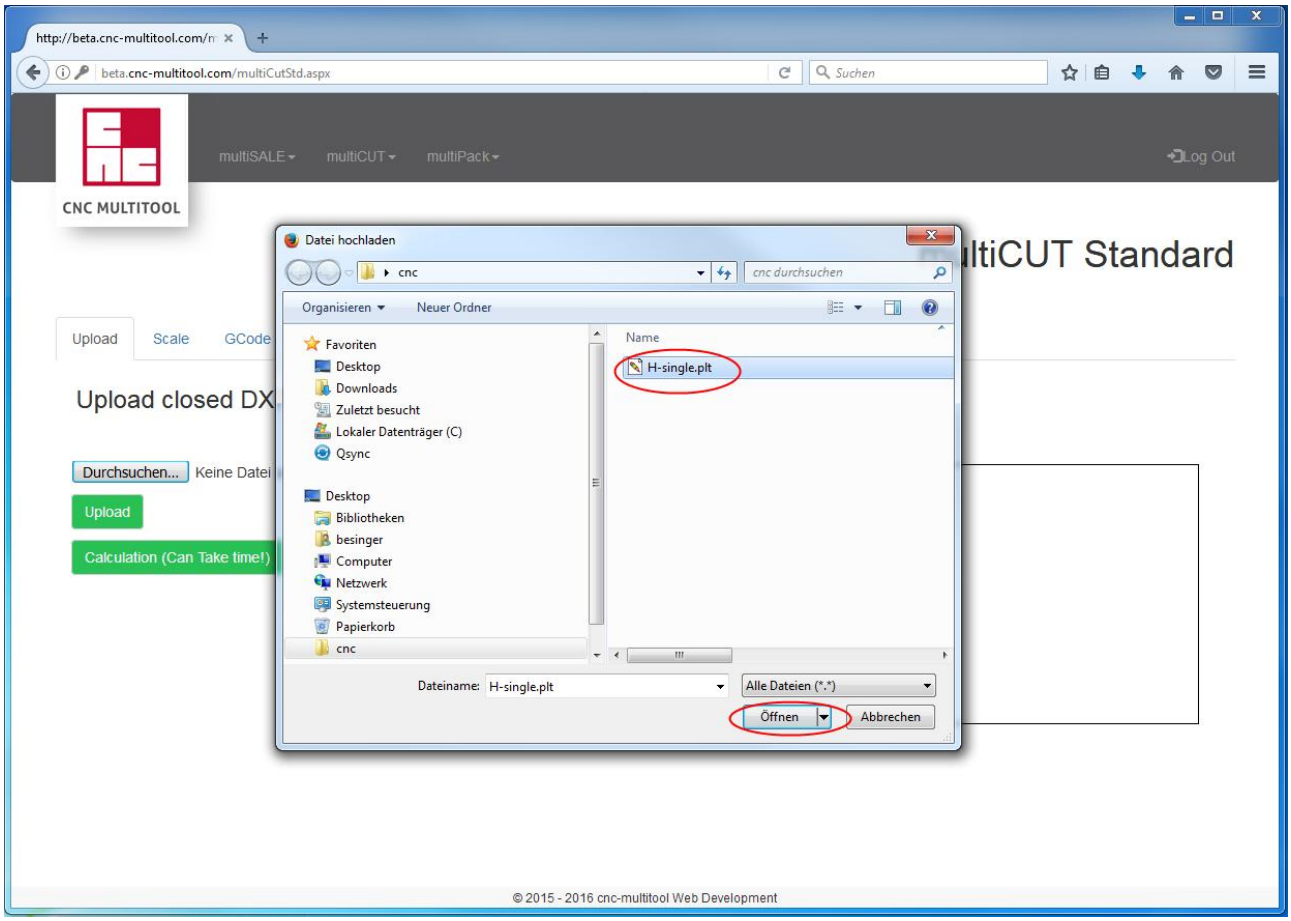
1. Closed outlines with entrances / exits

Click "multiCUT / Standard Contour"



Click "Durchsuchen"

Select your DXF / PLT or DWG file.



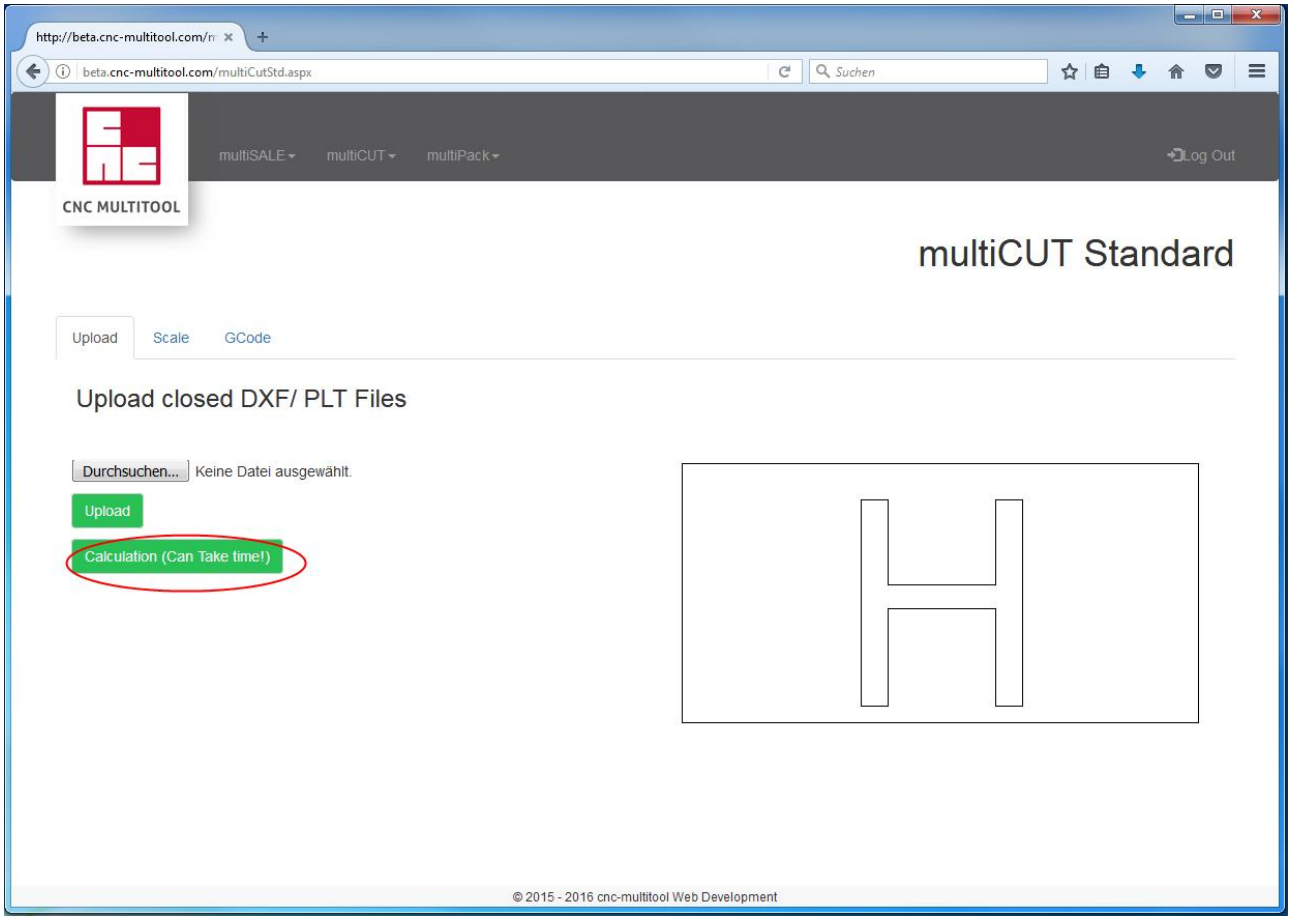
Important:
The DXF, DWG or PLT file you selected must not have open contours. For DWG and DXF files, be sure to save them in the correct AutoCAD format / version. It's best to test them.

Unfortunately, this is different from program to program.

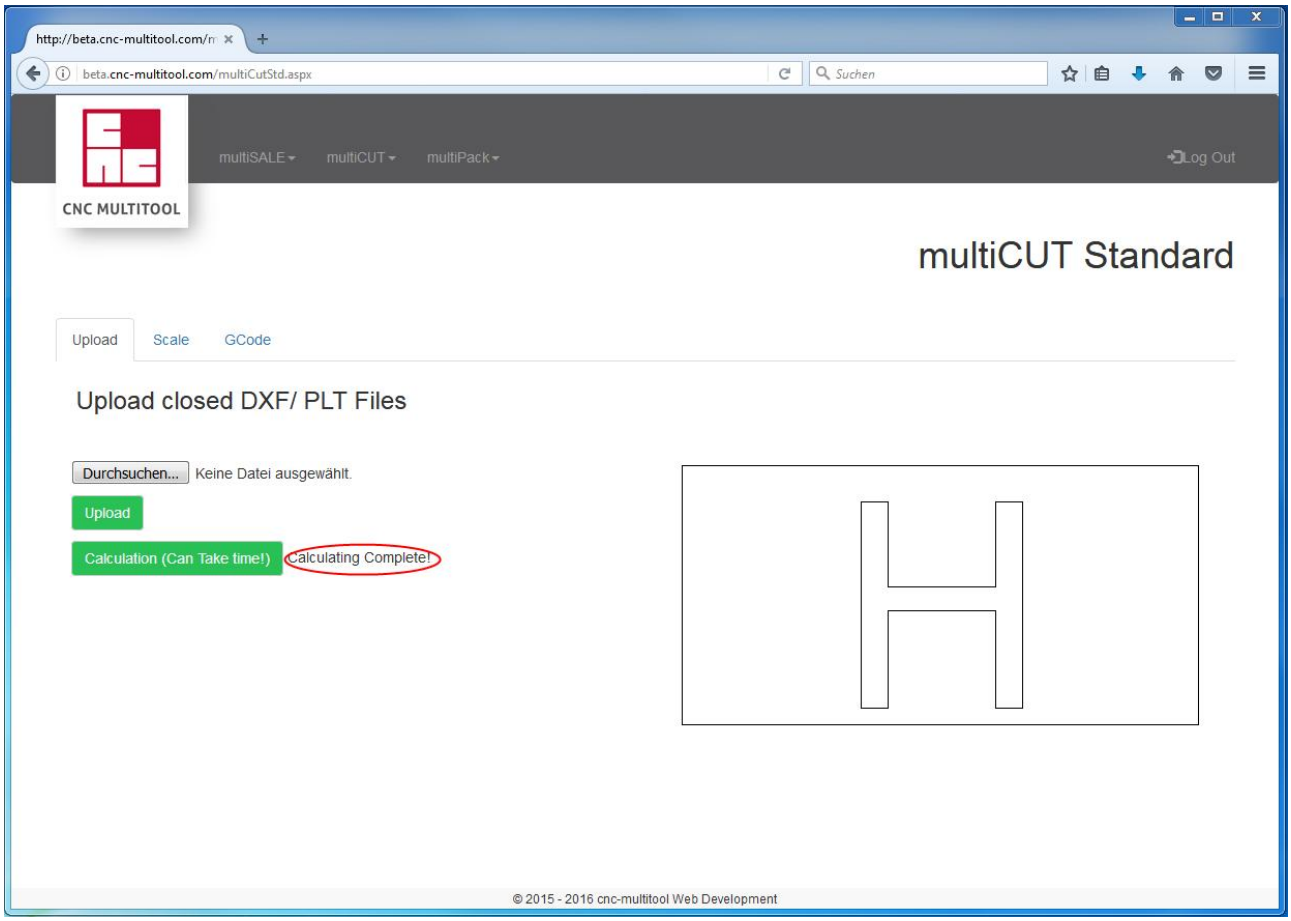
Note:
Corel Draw cannot create "reasonable" DXF files. We recommend to go use PLT files. When exporting as PLT, please set the curve resolution to zero.

Click on "Upload"

Click on "Calculation" (as soon as you can see the contour)



After successful calculation, you will get the message **"Calculation complete"**



If "**No open contours**" appears instead, please check the contours in your drawing.

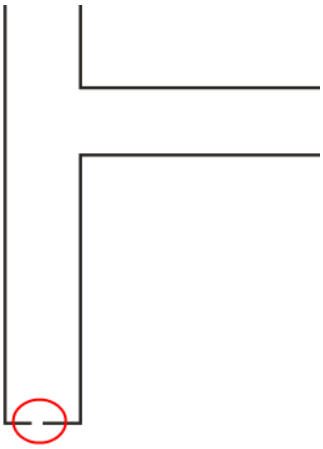
Note!
We've never seen the software being wrong so far. If "No open Contours" appears, your contour is open and not closed.

This error often occurs when users do not work with Line snap.

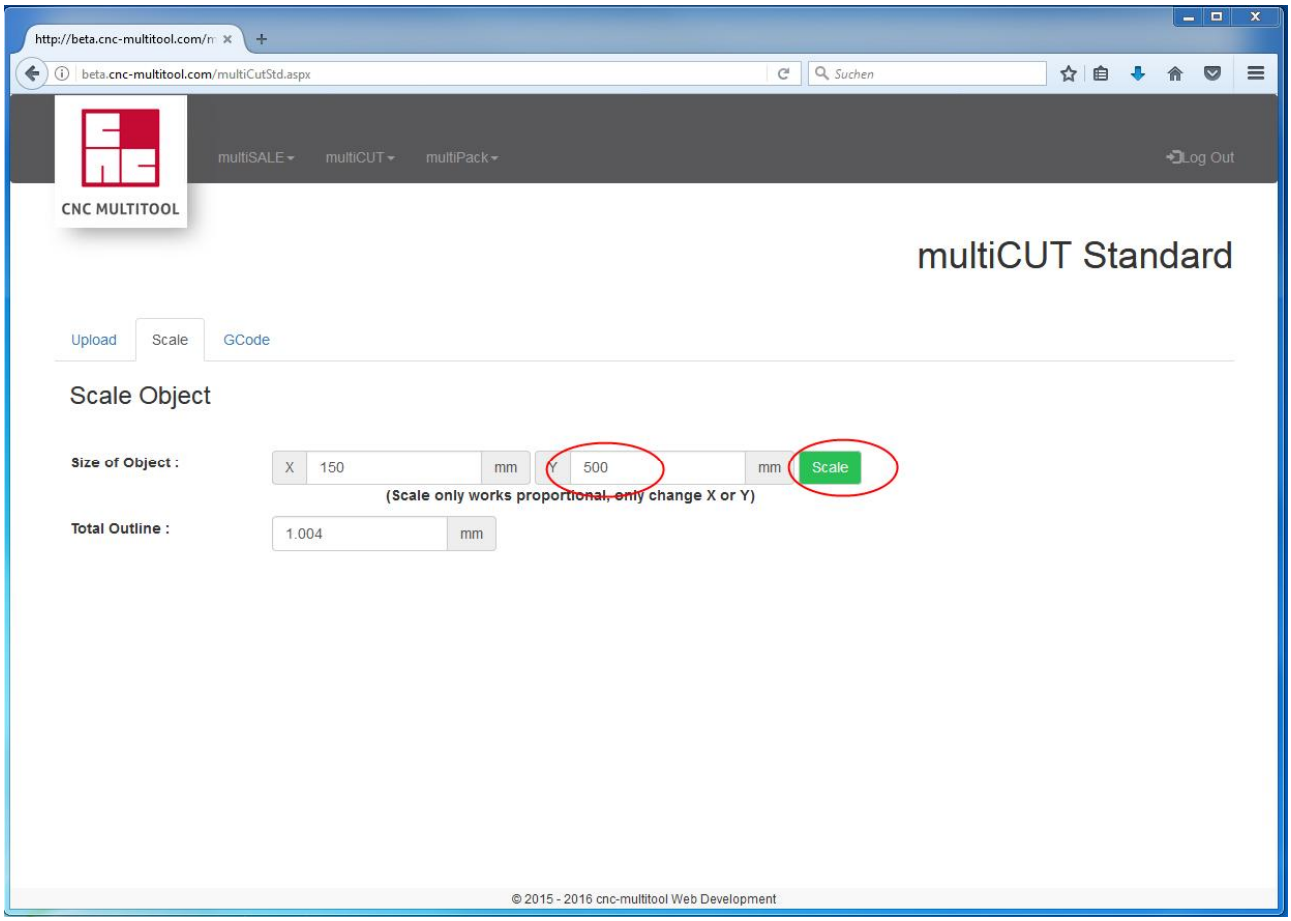
Possible cause of such an error?

Often a small carelessness is responsible for this mistake. These include in particular the following three examples:

Open Contour line



Click on the Tab "Scale"

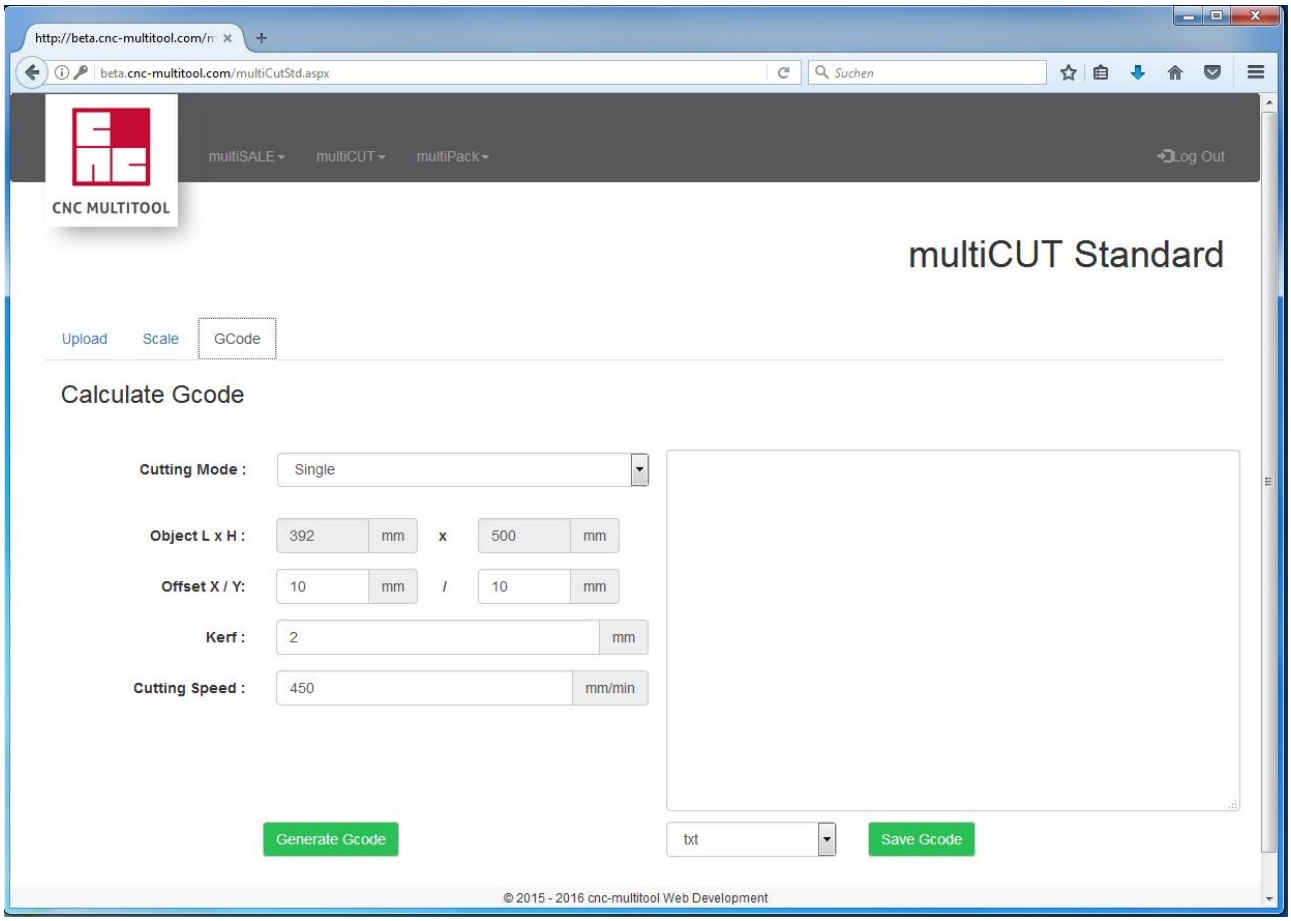


Check or adjust the size

Note:
The size can only be adjusted proportionally, by changing the X value or the Y value.
Then click on "Scale".

In this example, the height of the letter is 500mm.

Change to Tab “GCode”



Here you can define the basic parameters of your cut.

Cutting Mode: Single = single contour
multiple rows = multiple rows
complete Block = fill up a block with your block size

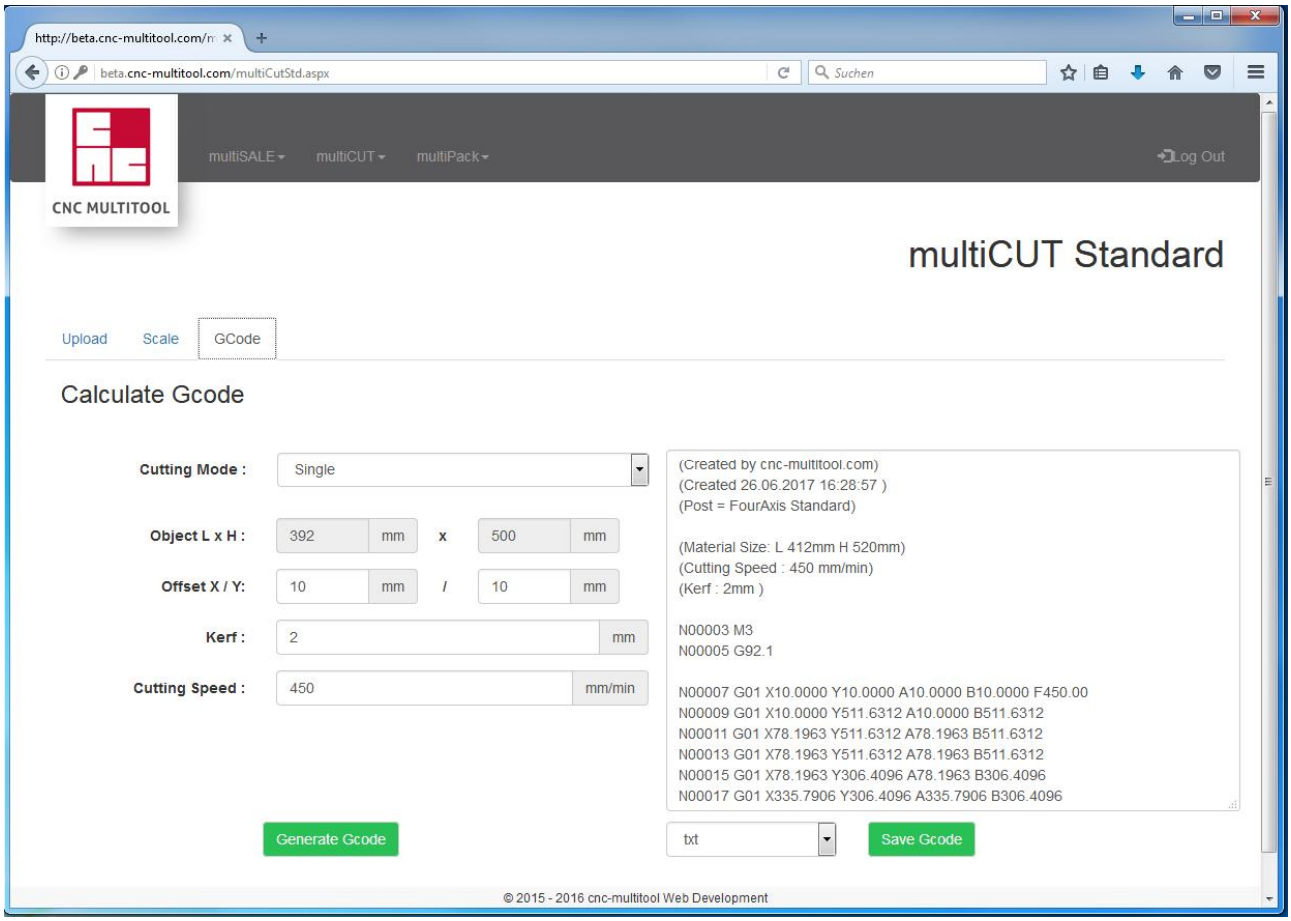
Object L x H: Corresponds to the size of your drawing
Offset X / Y: Enter the wire horizontally and vertically into the material
we recommend 10 / 10mm

Note:
You should always cut out of the full. Therefore, the wire must first travel into the material before it performs the actual cut. This is the offset.

Kerf: Enter your burn off (2mm recommended)
Cutting speed: Desired cutting speed. Depends on the material.
Begin slowly, 450mm / min is a good start.

Then click on "Generate Gcode".

Save the interface file locally on the computer / Lan drive or USB stick.



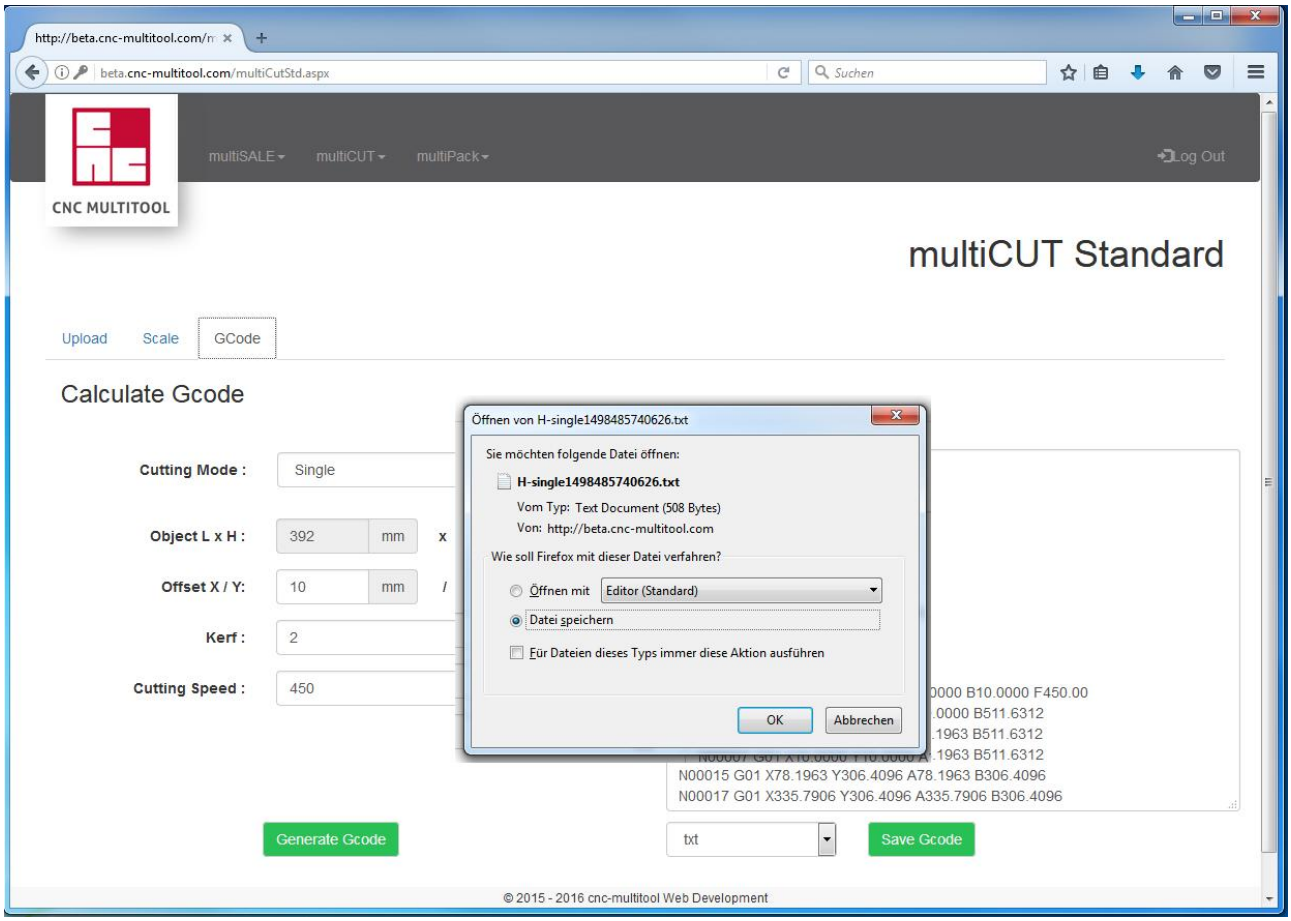
After the Gcode was created it must be saved.

Click "Save Gcode".

You can select the file extension yourself (nc, txt, dnc). We recommend txt, as these files can be opened in any standard text editor.

The file can now be saved locally on your computer, intranet or on a USB stick and then opened at your machine in the machine program (for example, Mach3).

Note:
In case you are using a USB stick, always copy the file to one of the local drives on your machine computer. Sometimes USB sticks get faulty and the machine program cannot read the file and cause the machine to stop.



Select "Save File" and then click "OK" - done.

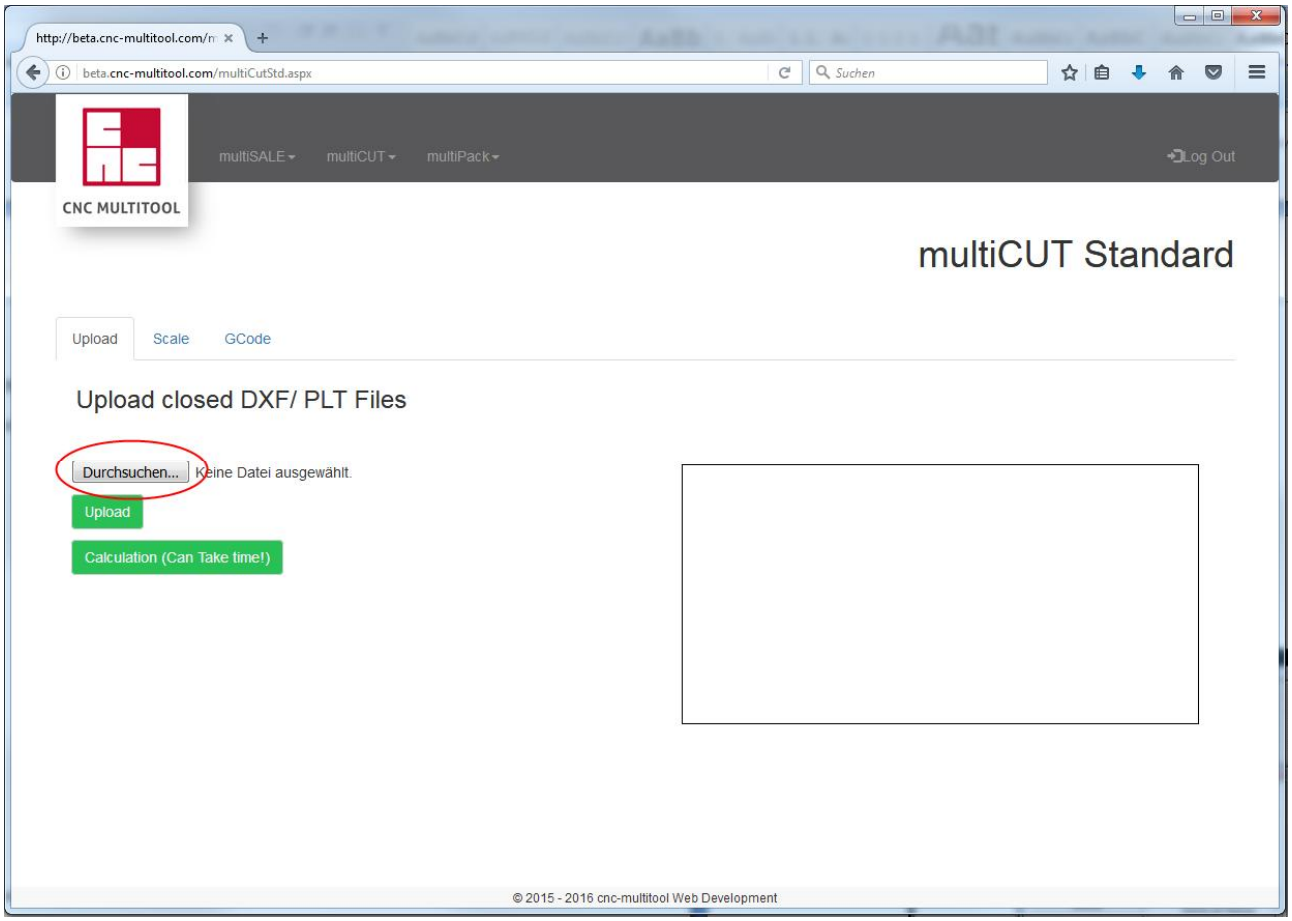
Load your cutting file in Mach3 or any other machine program and start the cut.

For further information please refer to the operating instructions of your machine.

If there is a problem, please contact the manufacturer.

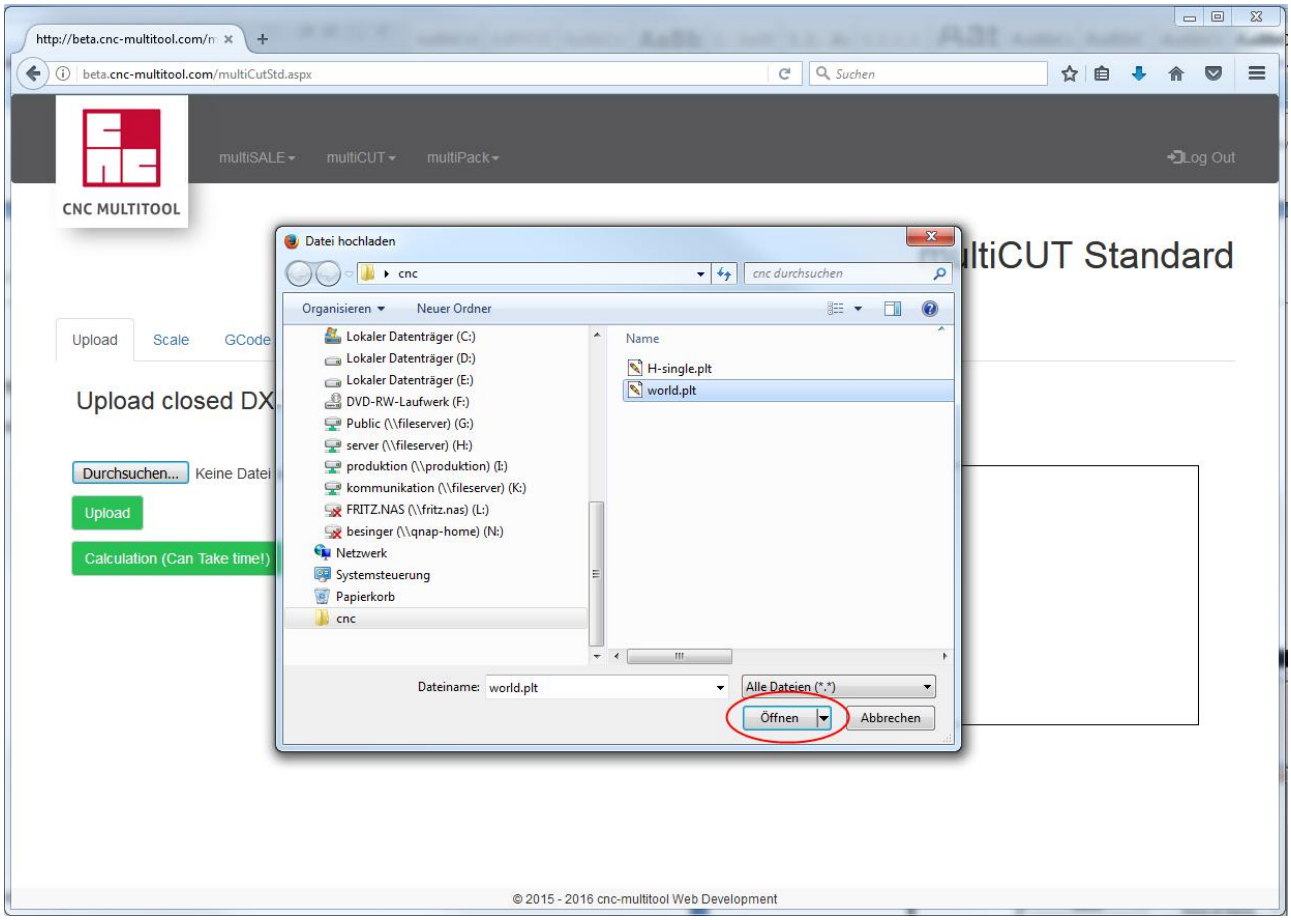
2. Multiple closed outlines with entrances / exits

Click "multiCUT / Standard Contour"



Click "Durchsuchen"

Select your DXF / PLT or DWG file.



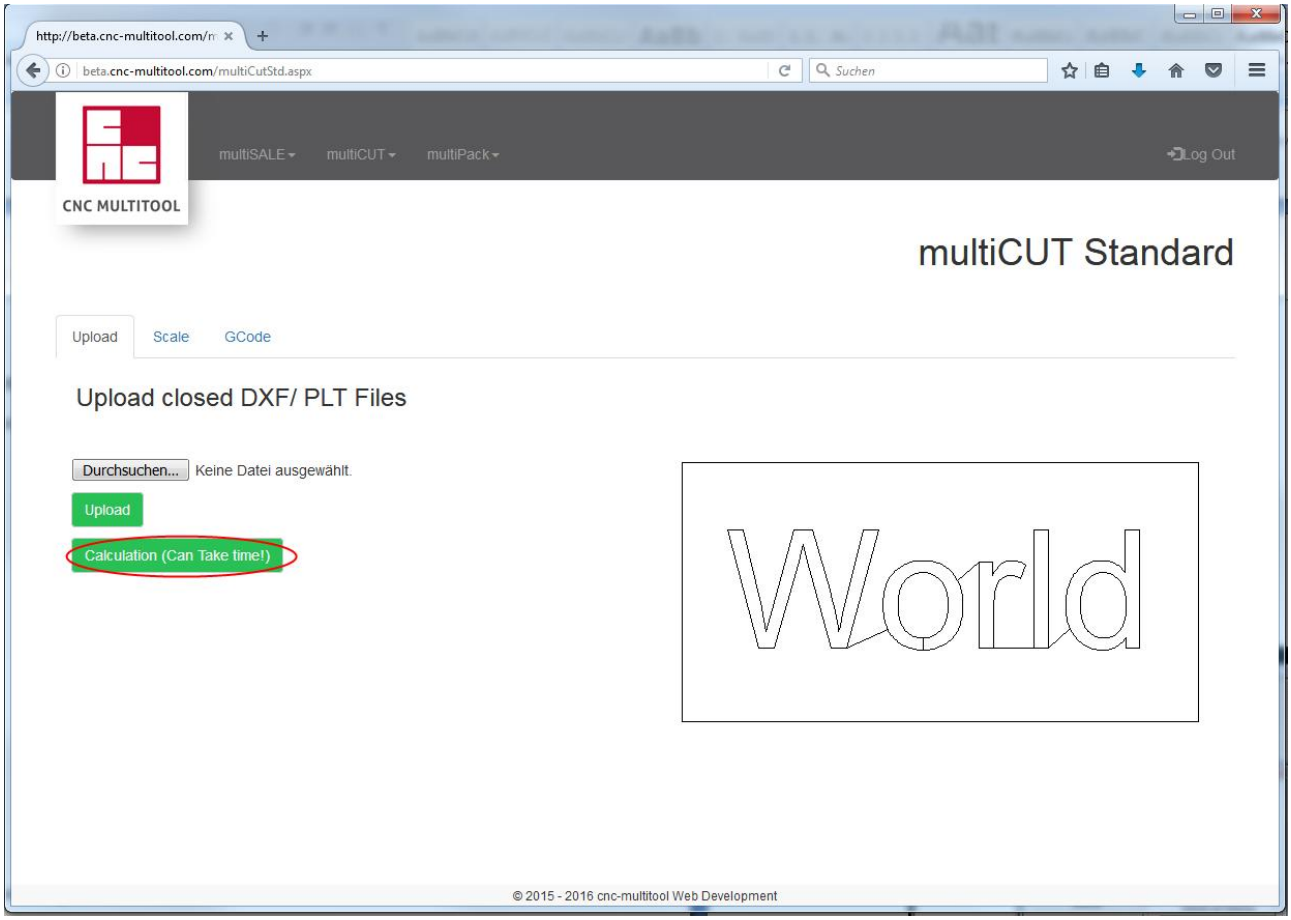
Important:
The DXF, DWG or PLT file you selected must not have open contours. For DWG and DXF files, be sure to save them in the correct AutoCAD format / version. It's best to test them.

Unfortunately, this is different from program to program.

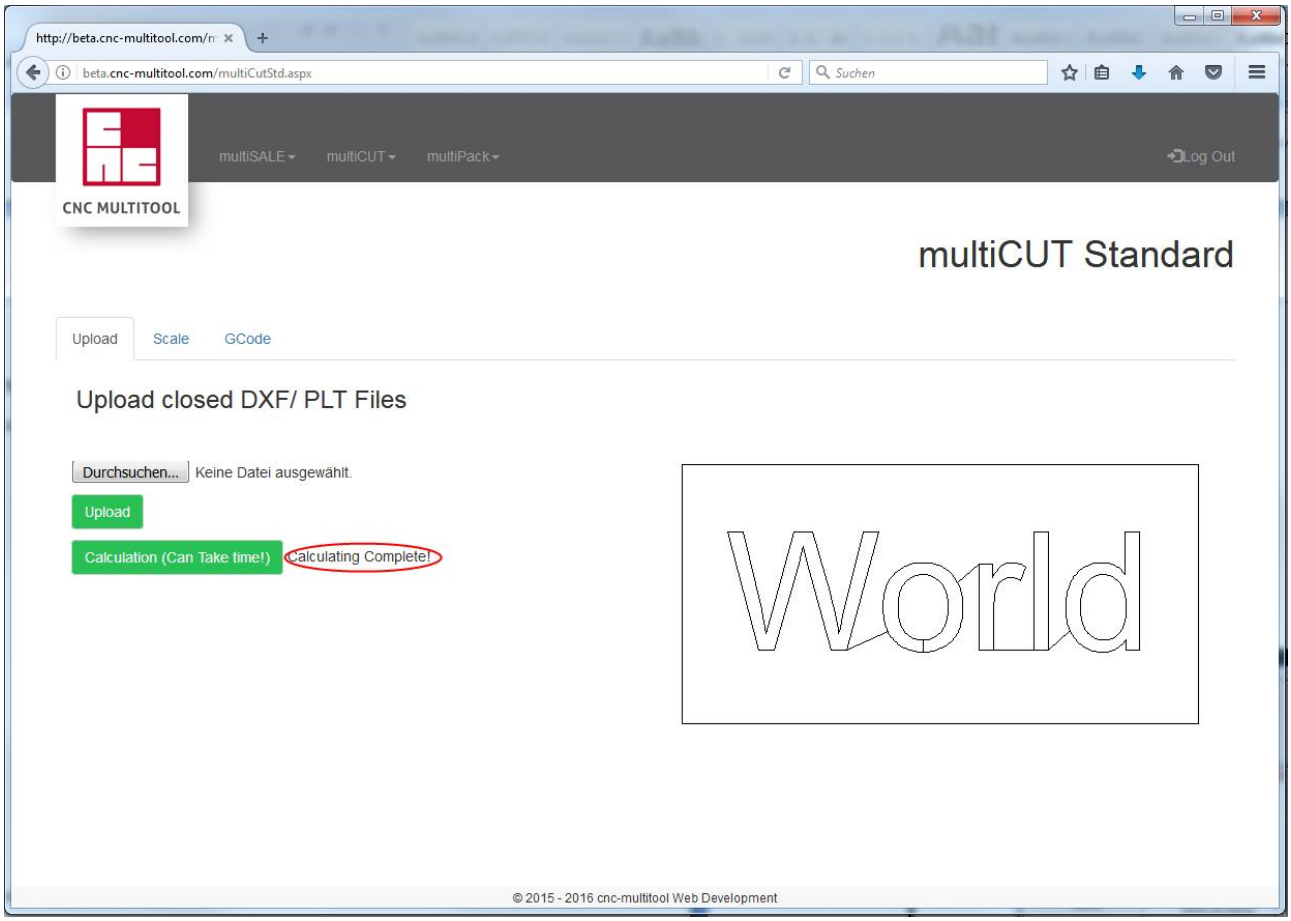
Note:
Corel Draw cannot create "reasonable" DXF files. We recommend to go use PLT files. When exporting as PLT, please set the curve resolution to zero.

Click on "Upload"

Click on "Calculation" (as soon as you can see the contour)



After successful calculation, you will get the message **"Calculation complete"**



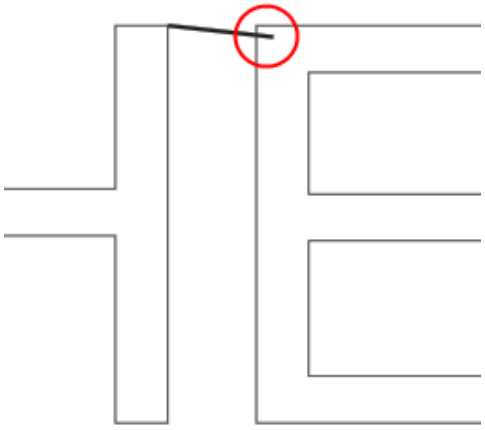
If "**No open contours**" appears instead, please check the contours in your drawing.

Note!
We've never seen the software being wrong so far. If "No open Contours" appears, your contour is open and not closed.

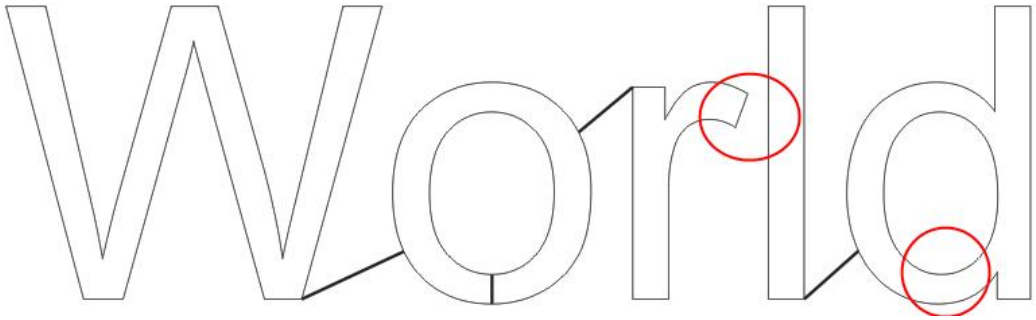
This error often occurs when the user does not work with the snap feature.
Please be sure to use it.

Please also ensure that the connecting lines between the contours have been set correctly.

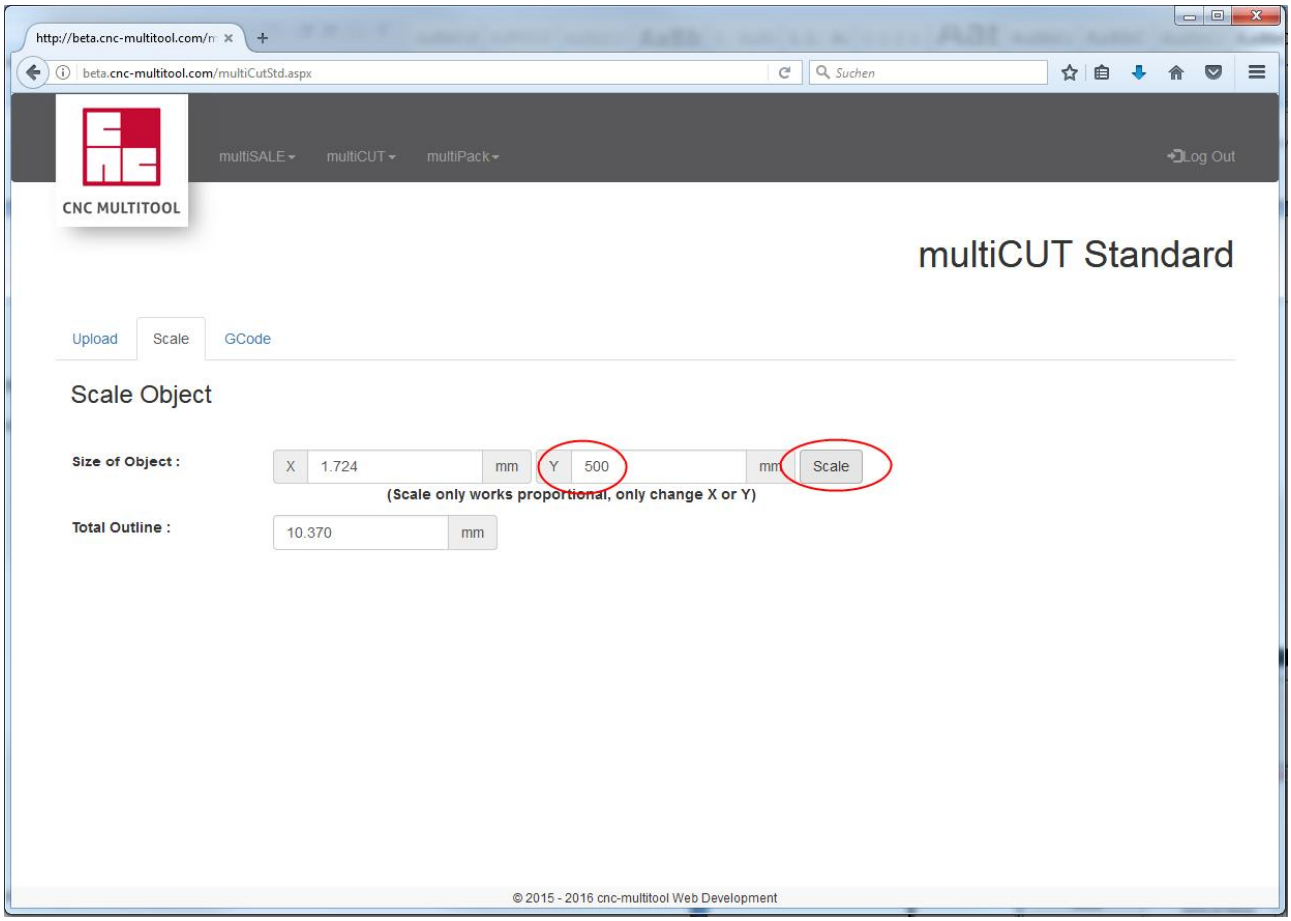
Wrong connecting lines without using the snap function (in your CAD / Corel/ Illustrator program)



Missing connecting lines



Click on the Tab "Scale"

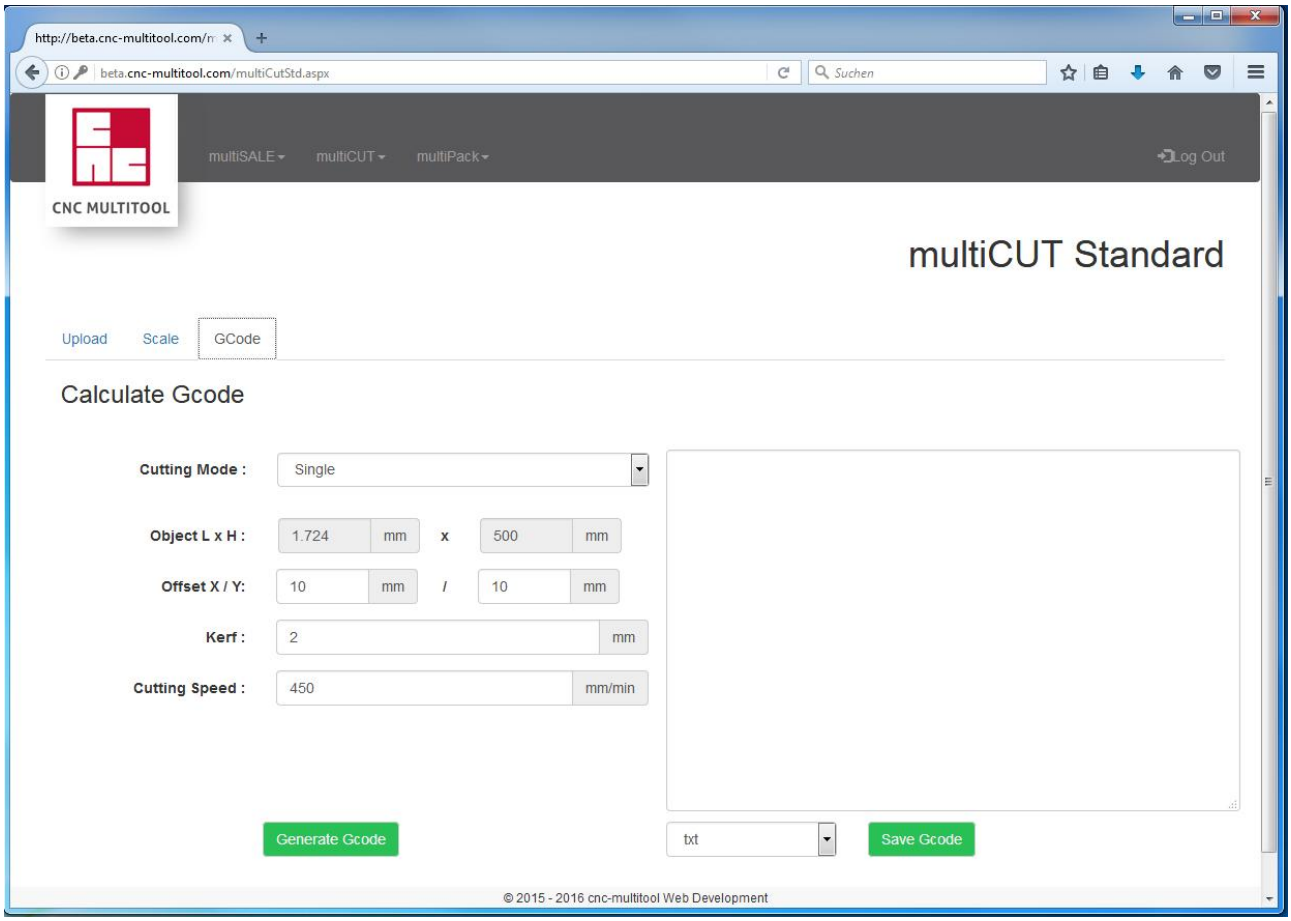


Check or adjust the size

Note:
The size can only be adjusted proportionally, by changing the X value or the Y value.
Then click on "Scale".

In this example, the height of the letter is 500mm.

Change to Tab “GCode”



Here you can define the basic parameters of your cut.

Cutting Mode: Single = single contour
multiple rows = multiple rows
complete Block = fill up a block with your block size

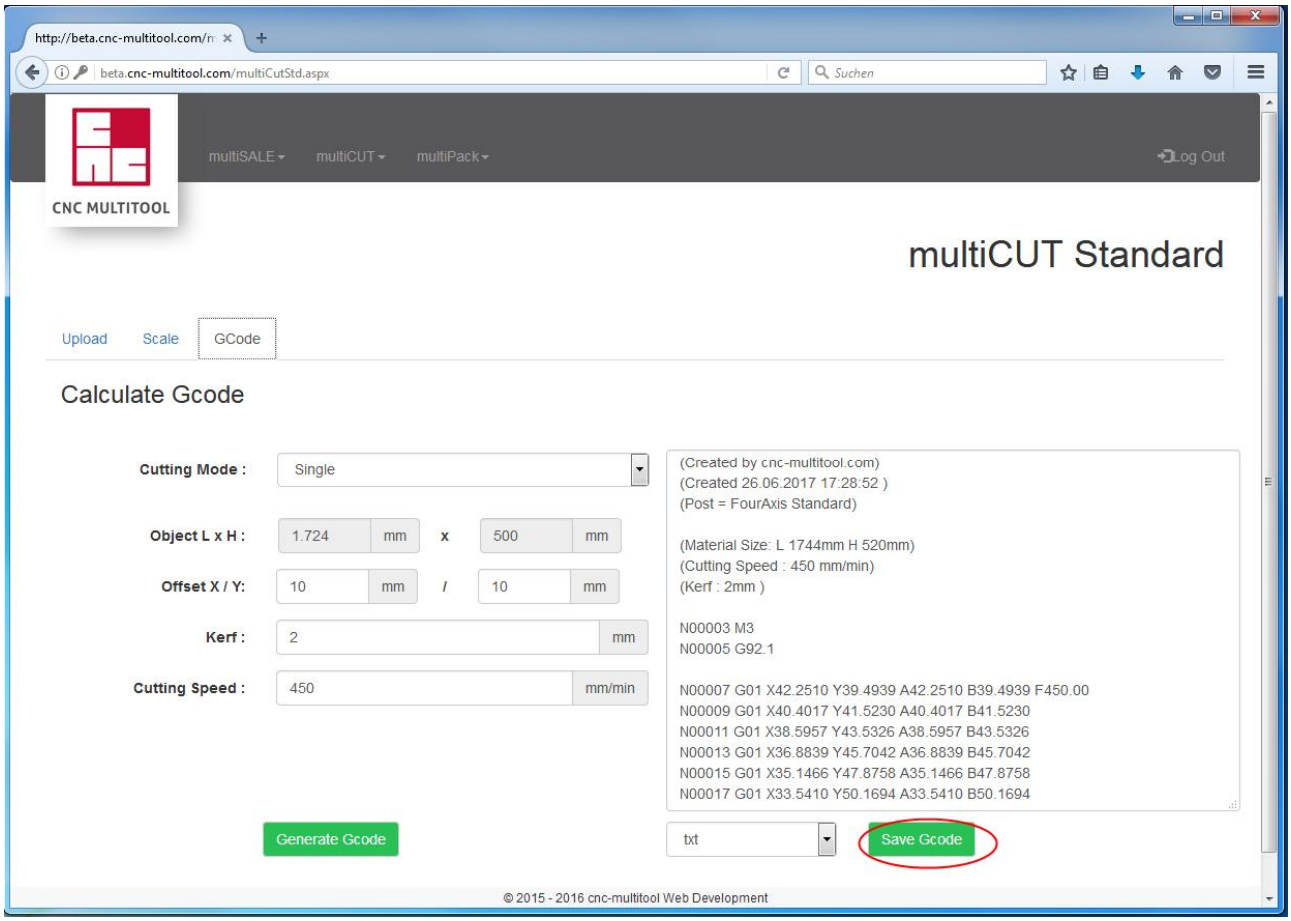
Object L x H: Corresponds to the size of your drawing
Offset X / Y: Enter the wire horizontally and vertically into the material
we recommend 10 / 10mm

Note:
You should always cut out of the full. Therefore, the wire must first travel into the material before it performs the actual cut. This is the offset.

Kerf: Enter your burn off (2mm recommended)
Cutting speed: Desired cutting speed. Depends on the material.
Begin slowly, 450mm / min is a good start.

Then click on "Generate Gcode".

Save the interface file locally on the computer / Lan drive or USB stick.



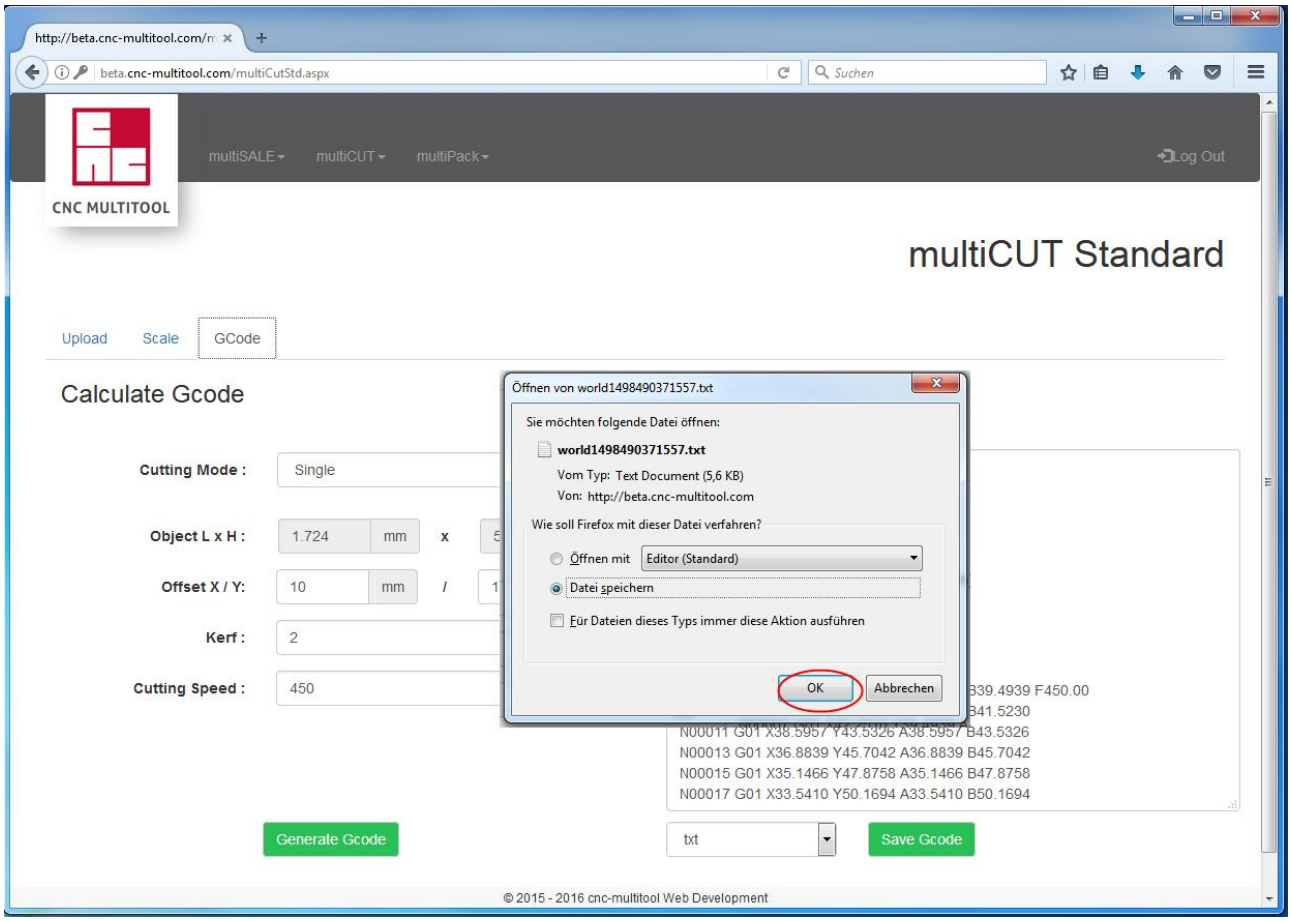
After the Gcode was created it must be saved.

Click "Save Gcode".

You can select the file extension yourself (nc, txt, dnc). We recommend txt, as these files can be opened in any standard text editor.

The file can now be saved locally on your computer, intranet or on a USB stick and then opened at your machine in the machine program (for example, Mach3).

Note:
In case you are using a USB stick, always copy the file to one of the local drives on your machine computer. Sometimes USB sticks get faulty and the machine program cannot read the file and cause the machine to stop.



Select "Save File" and then click "OK" - done.

Load your cutting file in Mach3 or any other machine program and start the cut.

For further information please refer to the operating instructions of your machine.

If there is a problem, please contact the manufacturer.