FastCAM®
QUICK-GUIDE

CREATION OF AN INDIVIDUAL CUTTING PROGRAM

22.5°

16 EQUAL SPACES

Ø 36.32 TYP. ON
A 310.39 PCD

19.05

Ø 136.40

11.25°
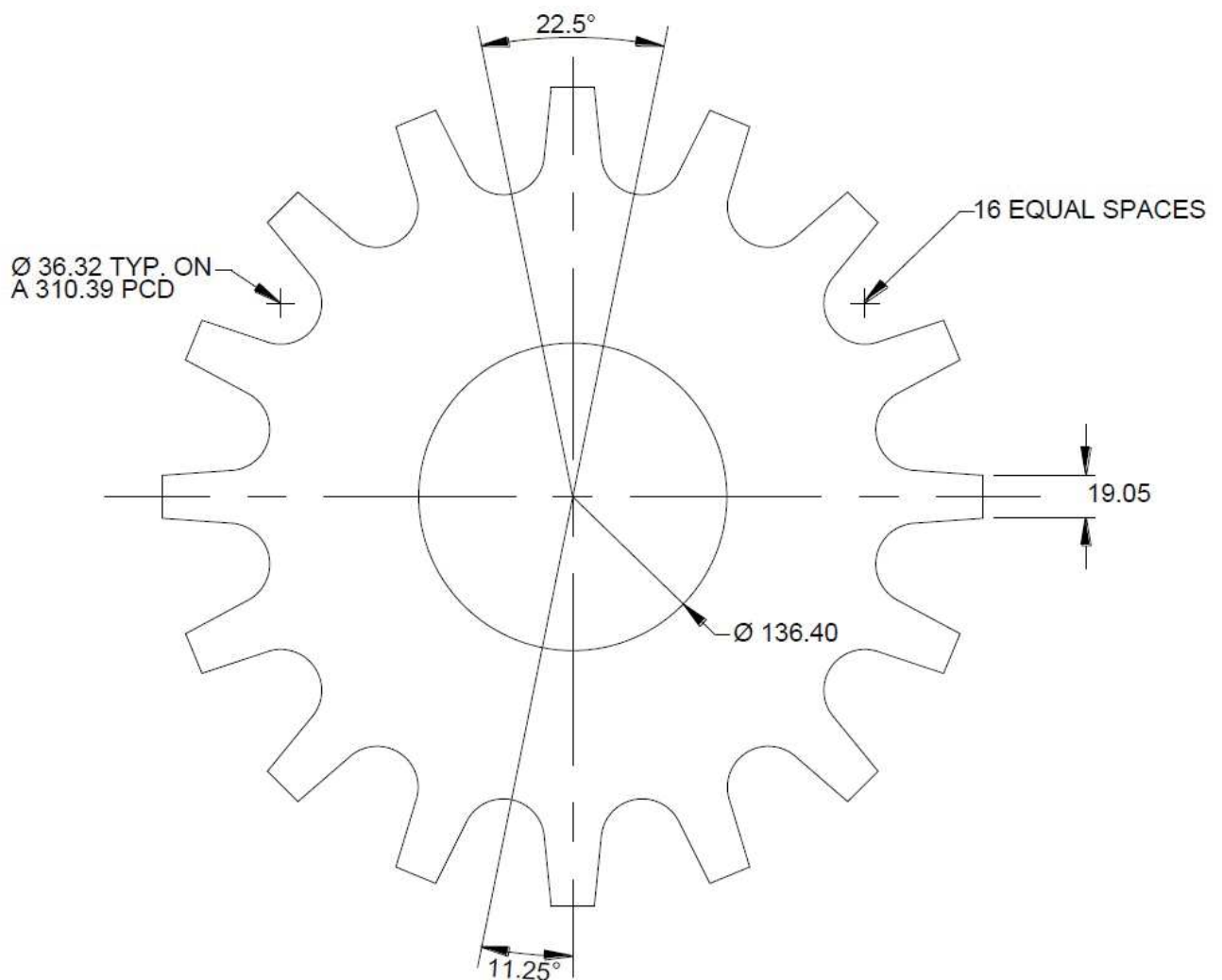
THERMACUT®

THE CUTTING COMPANY®

INDEX

SECTION 1.
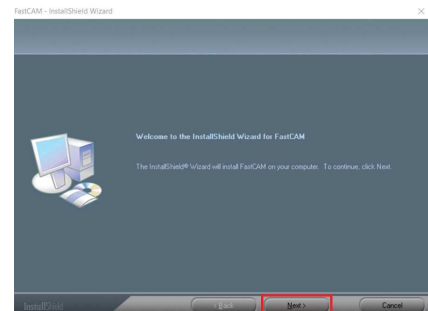
INSTALLING FASTCAM® SOFTWARE:

1. Insert CD.

2. Select „English". Preferred language can be changed later.
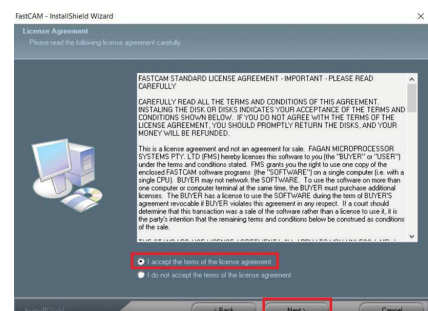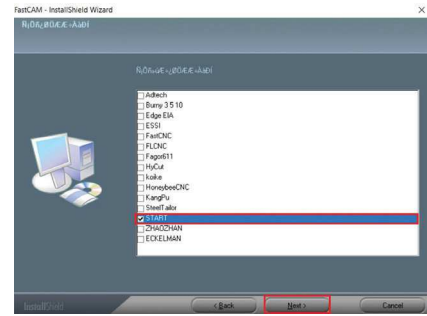
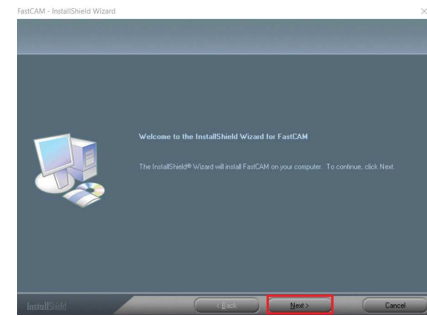3. Select „Install FastCam".

4. Follow the wizard „Next".

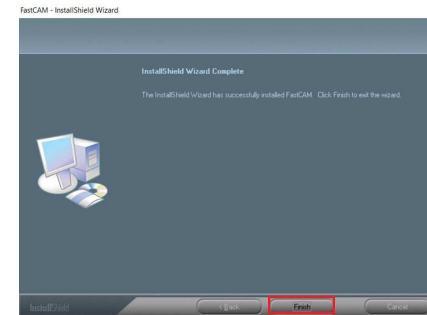5. Accept the „terms of the license agreement".

6. Select the „START" field.
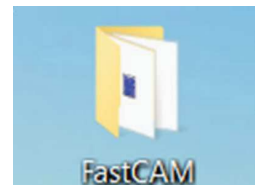

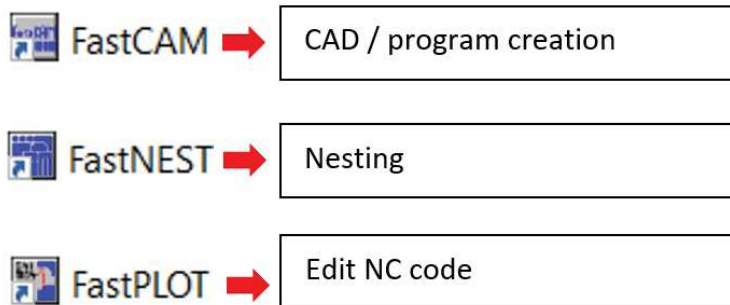7. Select „Install" in the next window.

8. Installation completed „Finish".

9. When the installation is complete, a new folder „FastCAM" will appear on the desktop.

10. Open folder.

 CAD / program creation

 Nesting

 Edit NC code

11. To open the software, the **Software protection USB-dongle** must be used. Then open the „FastCAM"software to change the national language.
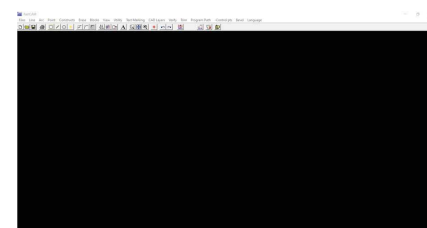
12. Click on „Language" in the upper bar and select the preferred language.
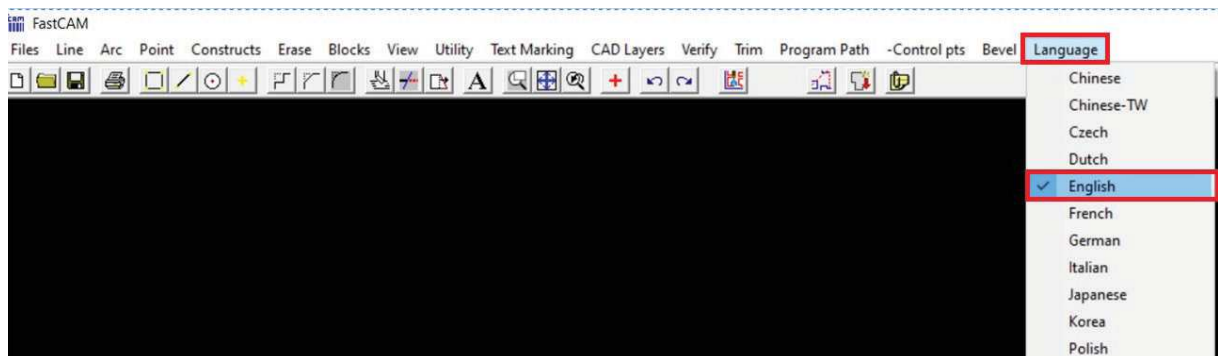
13. If necessary click on „Utility" in the upper bar, and change the measurement unit of Metric to Inch.
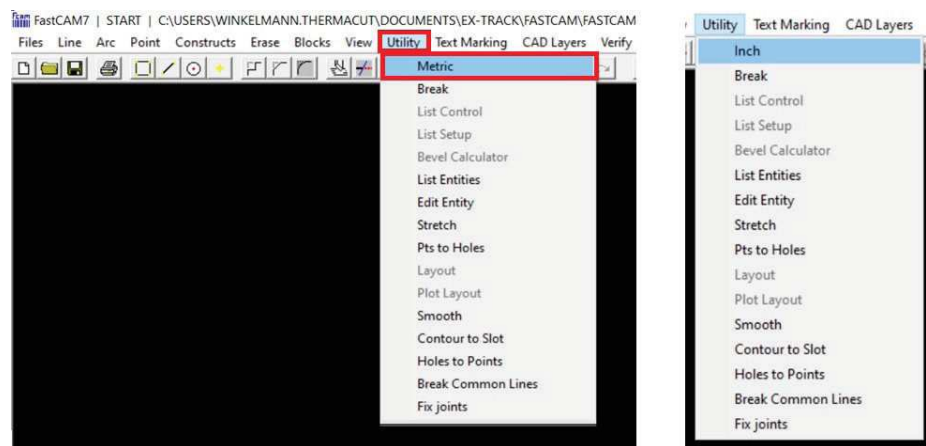
14. The installation is complete, after selecting the preferred language and changing of the measurement unit. When using the software, the delivery included **Software protection USB-dongle** must always be used.

SECTION 2

CREATING A SIMPLE CUTTING PROGRAM:

This quick guide is an example of how to create a cutting program easily and quickly. There could be also be other ways to create a cutting program.
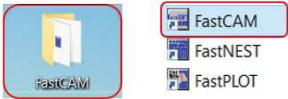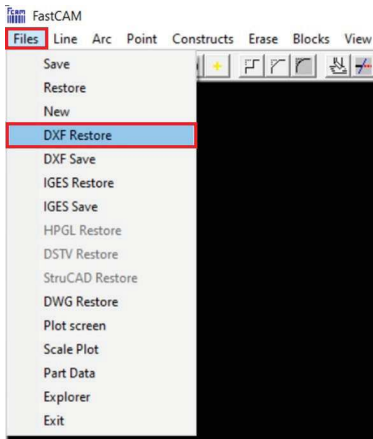
1. Open the software „FastCAM" (only with Software protection USB-dongle) on a PC.

2. Restore DXF/DWG.

1.   Files

2.   DXF or DWG Restore

3. Adjust DXF Option.
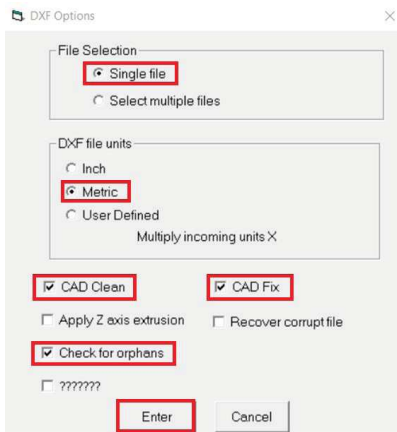
1. File Selection - Select „Single file"

2. DXF file unit - Select „Metric"

3. Select the options

    - CAD Clean

    - CAD Fix

    - Check for orphans

(These three options check the DXF/DWG after small errors,

such as lines that are too small.)

4. Confirm with „Enter"

IMPORTANT - Requirement regarding a DXF/DWG drawing:

- The drawing should be in scale 1:1.

- All contours must be closed.

- No orphans.

- No double lines.

- The smallest radius must not be smaller than the kerf setting for cutting.

4. Select DXF / DWG on the hard disk.
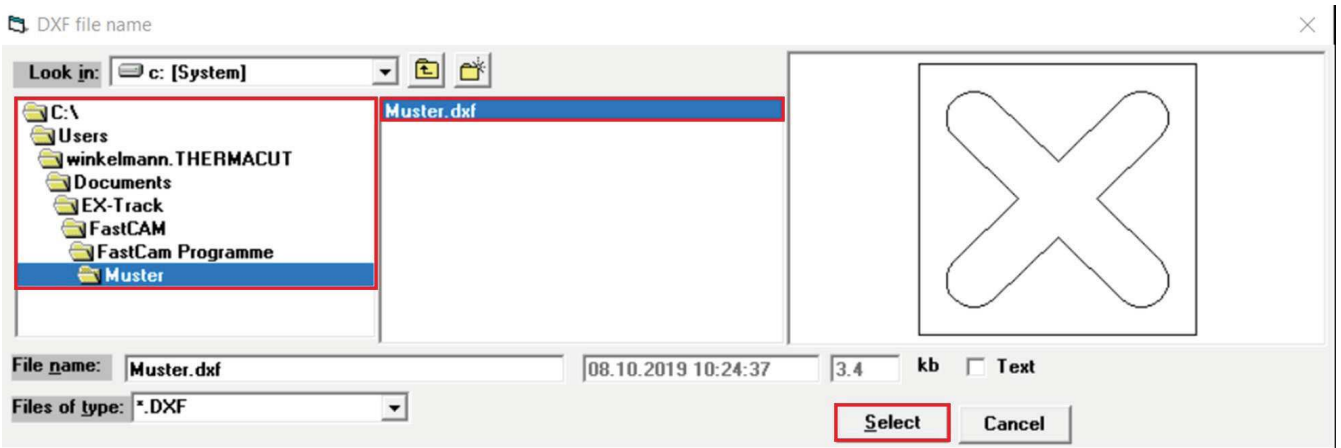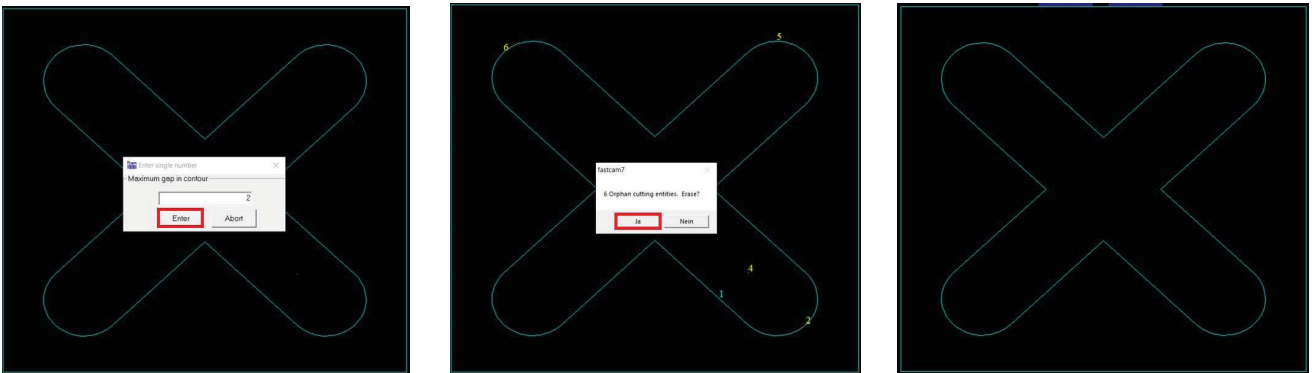

5. Import the DXF drawing and verify it by the previous „Option" selections.

Confirm the preset value with „Enter".

It ask for the maximum gap in the contour.
Everything smaller than 0,2mm is automatically deleted.

6 orphanes cutting units were found. Confirm the selection for erasure with "yes" (Ja).

When checking the drawing, the FastCAM found 6 small points that would have caused problems when cutting later.

The cleaned drawing is shown.

6. If necessary, save the clean DXF/DWG drawing on the PC, as seen below.
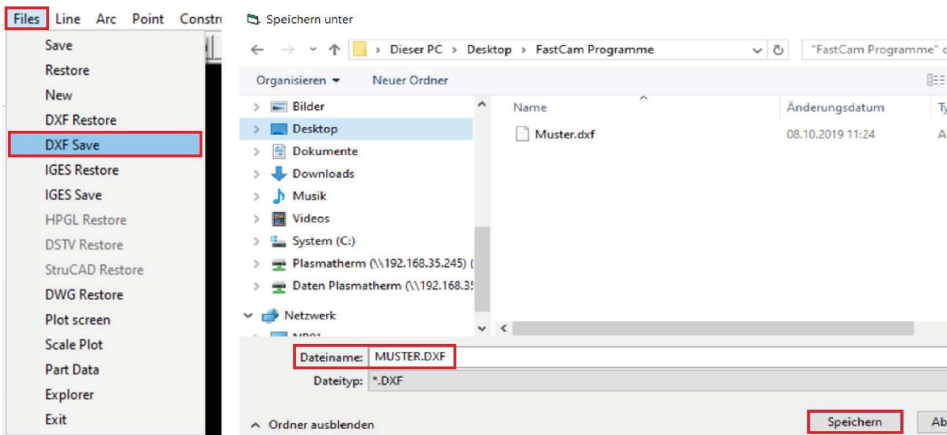
1. Files
2. DXF Save
3. Select the directory on the PC and save (Speichern).

7. Creating cutting technology.

Select the function „Set FastPATH"

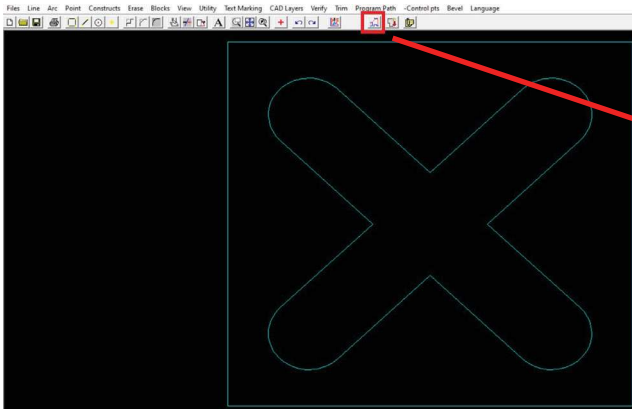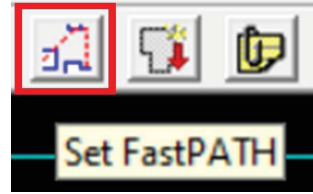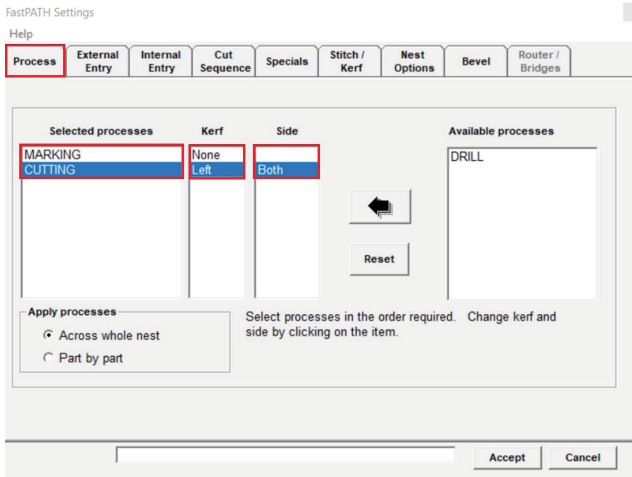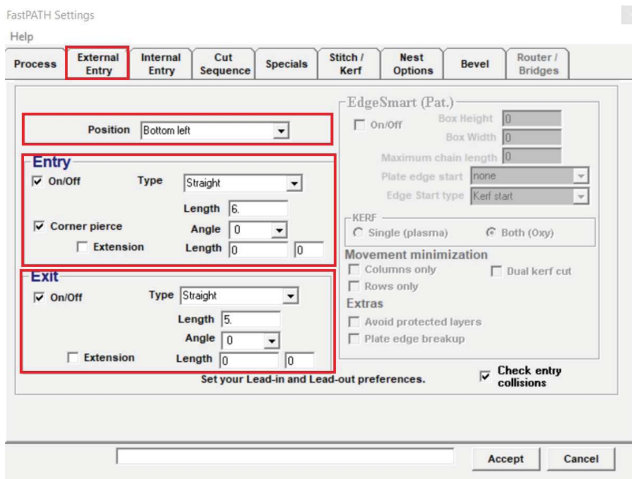This function is used to define

the lead-in and lead-out geometry.

8. FastPATH settings.

Select tab „Process" defines the process.

- Selected processes box = Select „Cutting".

- Kerf box = Select „Left".

- Side box = Select „Both".

Select tab „External Entry" defines the lead-in and lead-
out geometry for the outer contour.

„Position" box defines select the starting point.

„Entry" box defines the lead-in geometry.

- Type = Select the type of lead-in in Pull-Down menu
(straight ahead, quarter circle or semicircle).

- Length = length of the lead-in geometry in mm or inch.
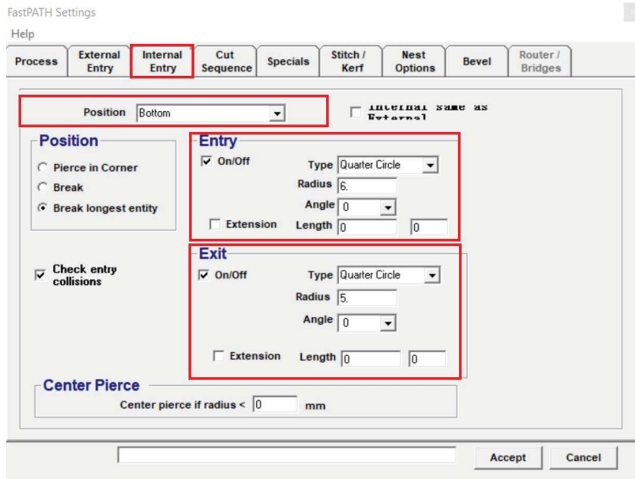
- Angle = 0°, 45° or 90° lead-in angle.

- Corner pierce = check box (If available the lead-in is
placed at a corner).

„Exit" box defines the lead-out geometry.

- Type = Select the type of lead-out (straight ahead,
quarter circle or semicircle).

- Length = length of the lead-out geometry in mm or
inch.

- Angle = 0°, 45° or 90° lead-out angle.

Select tab „Internal Entry" defines the lead-in and lead-out geometry for the inner contour.

„Position" box defines select the starting point.

„Entry" box defines the lead-in geometry. Select On or OFF then enter.

- Type = Select the type of lead-in (straight ahead, quarter circle or semicircle).
- Radius = Enter value.
- Angle = 0°, 45° or 90° lead-in angle.
- Length = length of the lead-in geometry in mm or inch.

„Exit" box defines the lead-out geometry

- Type = Select the type of lead-out (straight ahead, quarter circle or semicircle).
- Length = length of the lead-out geometry in mm or inch.
- Angle = 0°, 45° or 90° lead-out angle.

Tipp's for the lead-in and lead-out geometry

Inner contours:

You should try to move to the contour with a radius (quarter circle / half circle). The radius should be adapted to the material thickness or hole diameter. The starting point of the lead-in should be at least so far away from the contour that the contour is not damaged when piercing.

The lead-in and lead-out geometry may be different for materials and material thicknesses.
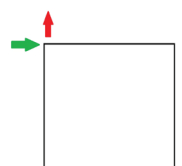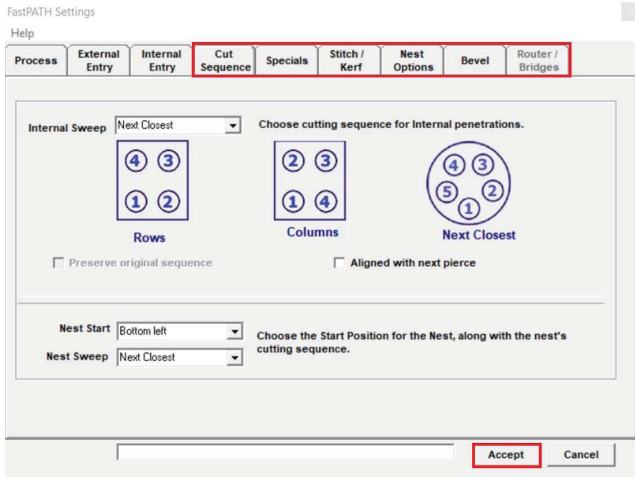
Outer contours:

If there is a "sharp" corner (90 degrees) on the outer contour, this is the ideal starting point. The preferred lead-in geometry is then to move "straight ahead" on the contour. Otherwise, as with the inner contours, you should move gently with a radius to the contour.

Example hole:

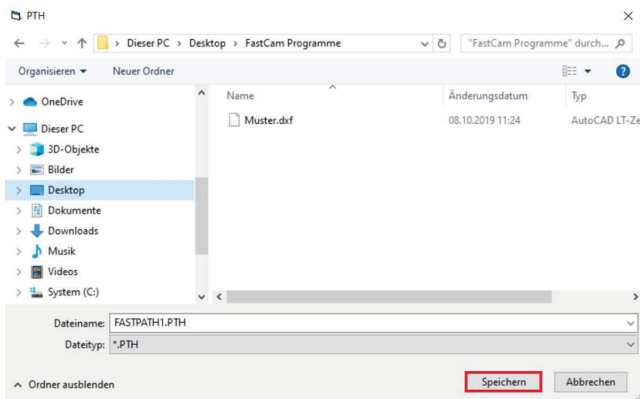Example outer contour with a sharp edge:

In „Cut Sequence" you can change the cutting sequence.

„Specials / Stitch, Kerf / Nest Options / Bevel" are for the time being not required for creating a cutting program.

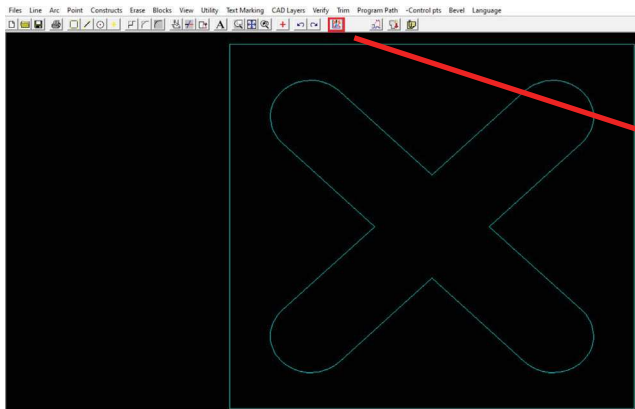As soon as the settings are completed, close the field with "Accept".

Then FastCAM wants to save the settings for lead-in and lead-out geometry.

- Select the required directory.
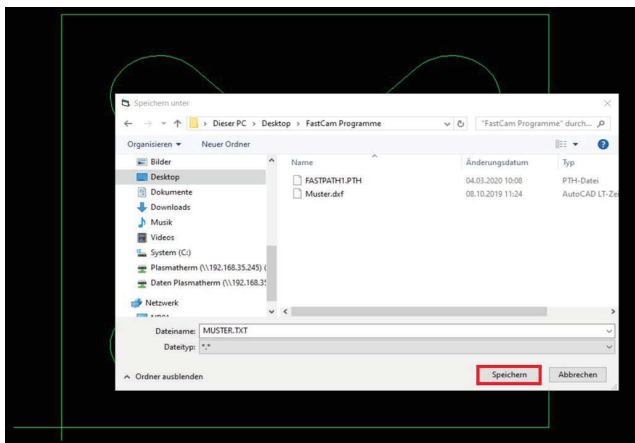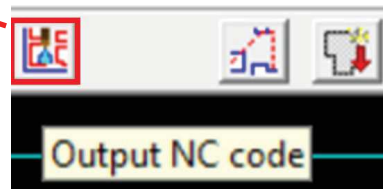- It will be saved as FASTPATH1.PTH (The name can be changed. It is important that the file is saved as .PTH).
- Close the field with „Save" (Speichern).

9. Assign lead-in and lead-out geometry / create cutting program.

Select function „Output NC Code".
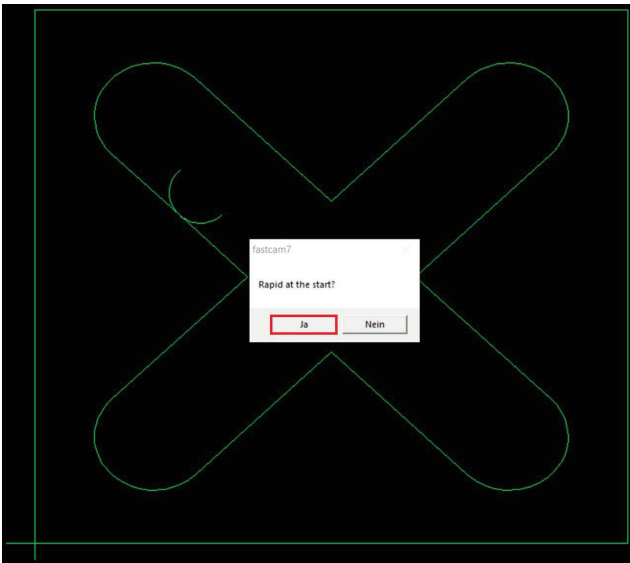With this function the setting of lead-in and lead-out geometry assigned.

Then FastCAM wants to save the cutting program

- Select the required directory.
- It will be saved as Muster.TXT (The name can be changed. It is important that the file is saved as.TXT).
- Close the field with „Save" (Speichern).

FastCAM asks „Rapid at the start"?
- Select these message with „yes" (Ja).

Yes (Ja) means:
It can be selected with the mouse by clicking in the field of the program starting point (see next page).

No (Nein) means:
The cutting program starts at the starting point of the outer contour. This can therefore be any position, depending on the starting point for the outer contour set before.
For pierce starting, when positioning the cutting torch, care should be taken to ensure that the lead-out is still in the material.

If the previous field was confirmed with "yes", you can now use the mouse to set the program starting point. For example, click with the left mouse button on the left below where the "red cross" is shown.
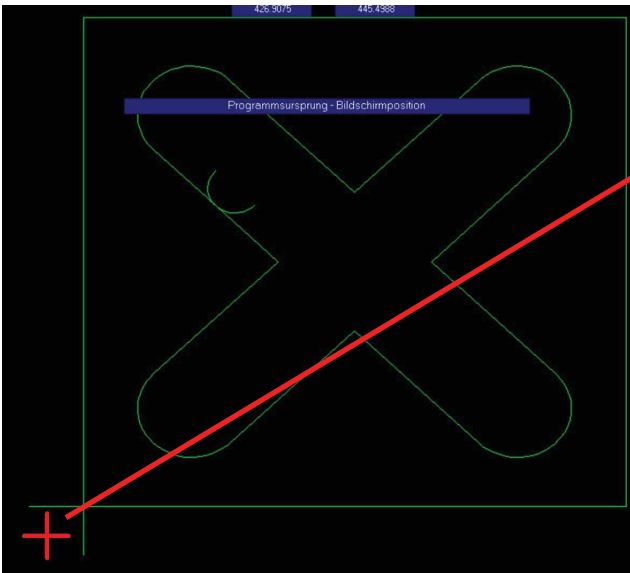
This method has the advantage that the later positioning of the cutting torch is simplified.

This means that when standing in front of the CNC control, the cutting torch can be positioned at the bottom left of the plate to be cut, without having to take care that the lead-out can still take place in the material.
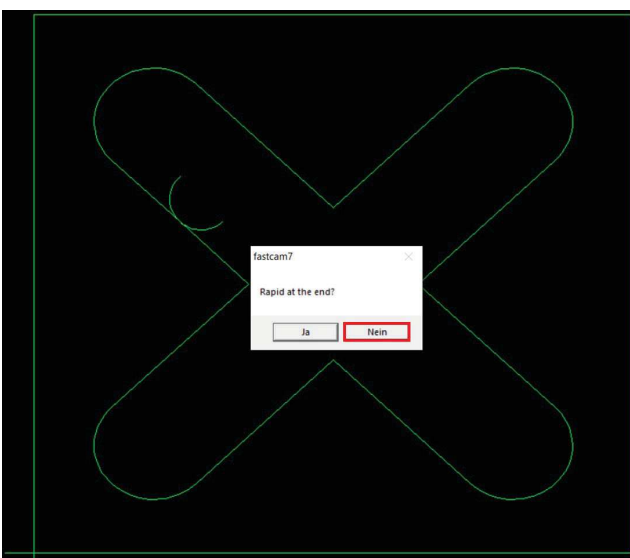
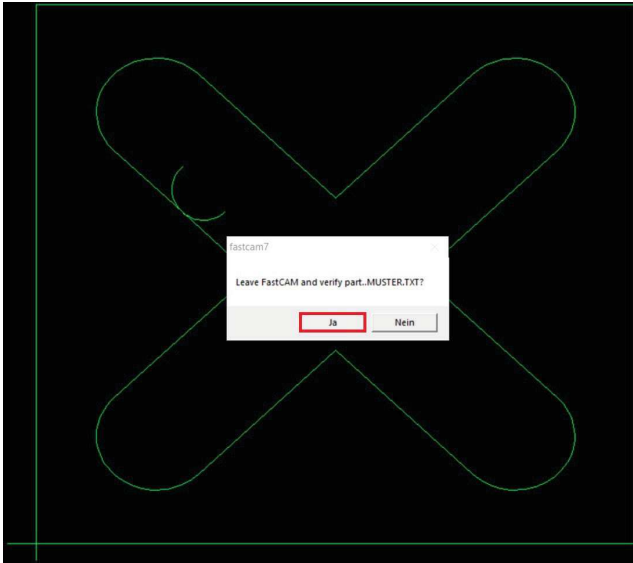After assigning the program start point, FastCAM asks „Rapid at the end?"

- Select this message with „no" (Nein).

No (Nein) means:
As soon as the last contour has been cut, the cutting torch stops at this contour.
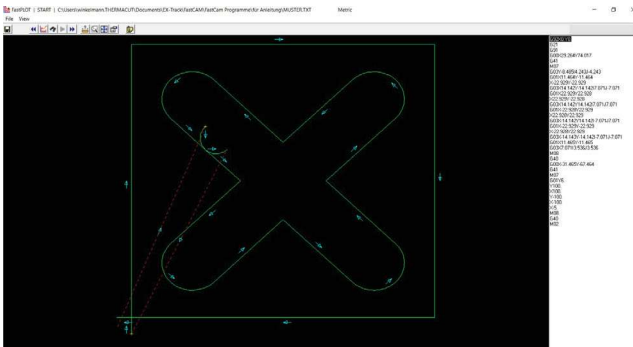
Yes (Ja) means:
As soon as the last contour has been cut, the cutting torch moves back to the program starting point.

FastCAM ask „Leave FastCAM and verify part. Muster TXT?"

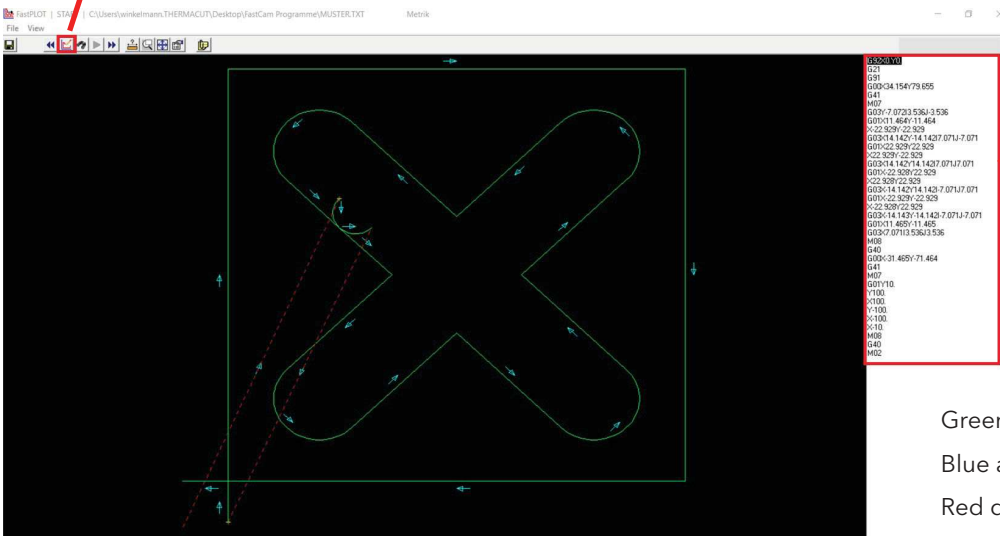- Select this message with „yes" (Ja).

After it, the FastPLOT program opens automatically. Here the program can be checked whether

- the cutting sequence is ok
- the rapid moves are ok
- starting point is ok
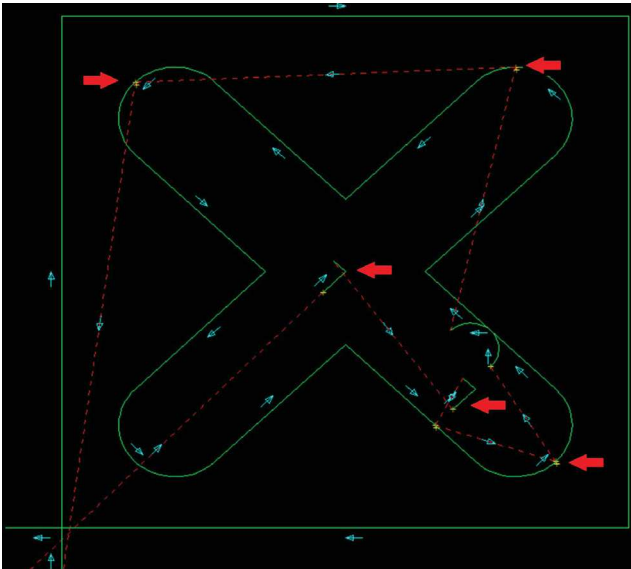- etc.

10. Check cutting program.

By pressing this field, the cutting program (NC code) can do a Dry-Run step by step with the cursor keys (up / down).


NC code

Green line = Cutting contour
Blue arrows= Cutting direction
Red dotted line = Rapid move

Example of a faulty DXF drawing.

Because the DXF drawing is faulty (e.g. by double lines or small points on a line), the FastCAM generates too many starting points, which can be easily recognized by too many rapid movements and repeated starts on a line (contour).

The red arrows show the wrong points and lines.
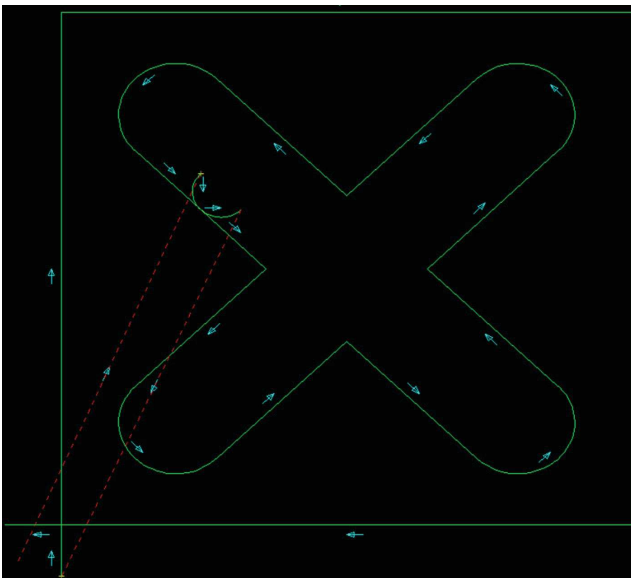
After cleaning the DXF drawing, FastCAM shows the correct cutting sequence.

It is clearly visible that there are less rapid movements and that each line (contour) is only cut once.

In the case of a line or contour, only a rapid movement should lead to the contour and away again at the end of the line (contour).

11. Cutting program.

The finished cutting program can now be saved from the PC to a USB flash drive and can be cut on the EX-TRACK.

The finished cutting programs are saved as .TXT.

USB Flash Drives or Memory Sticks MUST BE FORMATTED TO FAT (FAT16) or FAT32. Best to have only program files on dedicated USB flash drive, or CNC may lockup trying to read other formats. FAT32 will work for USB drives up to 32 GB in size. Files can be no larger than 4GB. If your USB drive is larger than 32 GB, it can be reset to 32.
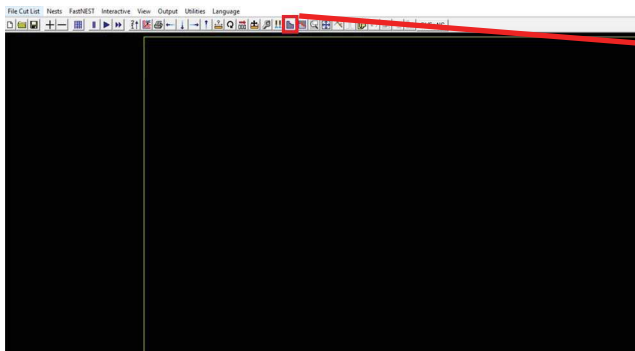
SECTION 3.

NESTING:

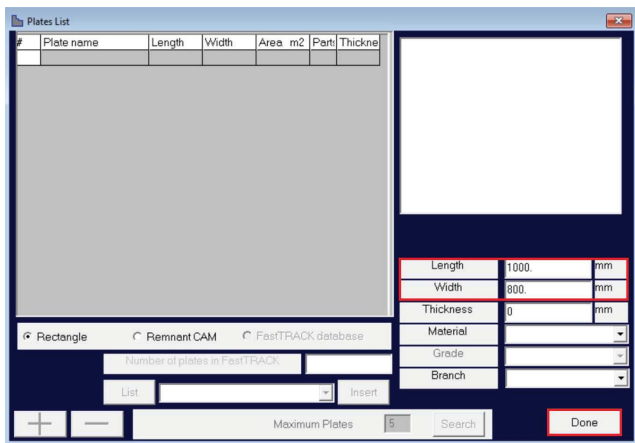When nesting, different contours can be nested in a define plate size.

1. Open the software „FastNEST" (only with USB-dongle).

2. Define plate size.
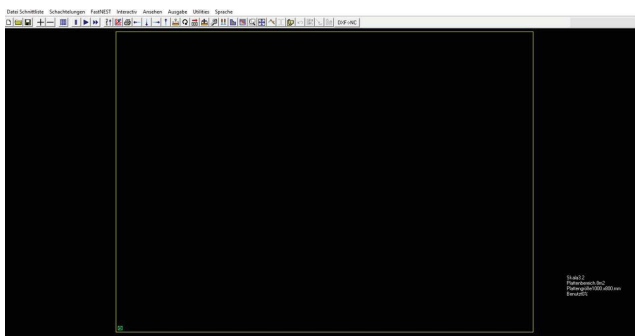

Select the function „change plate".

A new window opens where the plate size can be defined.
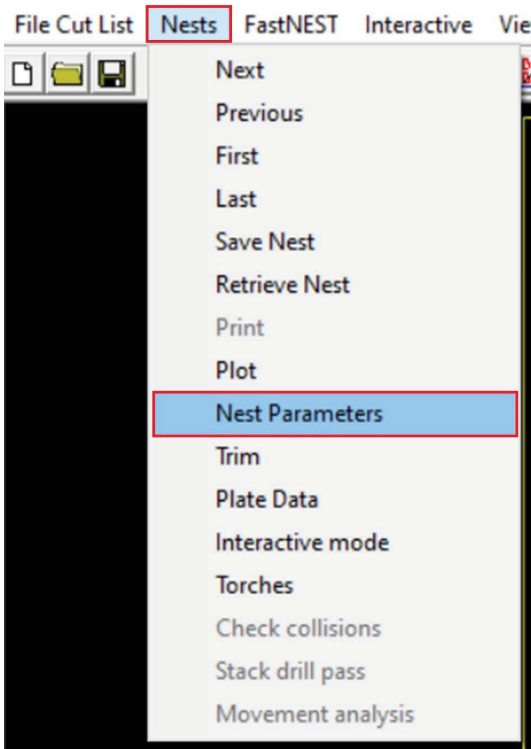- Enter the length and width.
- Close the field with „Done".

Do not need set the thickness and material, because the EX-TRACK® has no automatic cutting database.

Important - Maximum plate size is 3050 x 1525 mm (10 x 5 ft.). This is the maximum working area of the EX-TRACK® system.
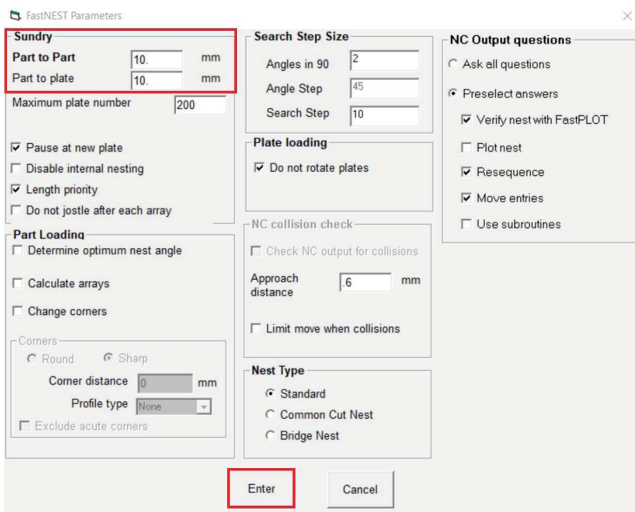
The yellow frame shows the adjusted plate size.

3. Set nesting distance.

Select the „Nests" menu and then „Nest Parameters".

In this menu, the nesting distance of the plate edge to the cutting part as well as between the individual cutting parts can be adjusted.

The distances can be set in the "Sundry" section.
- Part to part = Nesting distance between the parts to be cut.
- Part to plate = Nesting distance between plate edge and the part to be cut.

The nesting distance depends on the parts to be cut and the material thickness and can be different..
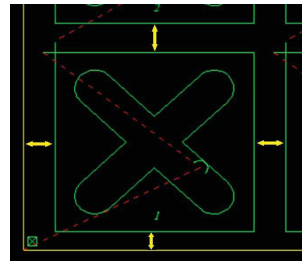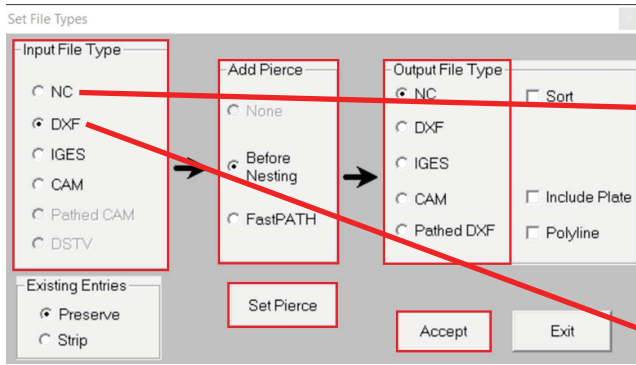

4. Set file type.

Select the function „DXF - NC".

Input File Type

First possibility = select „NC" and confirm "Accept"
You can directly continue with point 5 "Add cutting parts". The requirement is that the individual cutting programs to be nested have already been created in advance (xxx.TXT).

**Second possibility** = By selecting the „DXF", then continue with the following steps.

Add Pierce - Select „Before Nesting".
Output File Type - Select „NC".

Then click on „Set Pierce", to define the lead-in and lead-out geometry.

By selecting the field „ Set Pierce" the field „FastPATH Settings" opens.

- Repeat the procedure description „creation of a simple cutting program" under position 8 „FastPATH Settings".
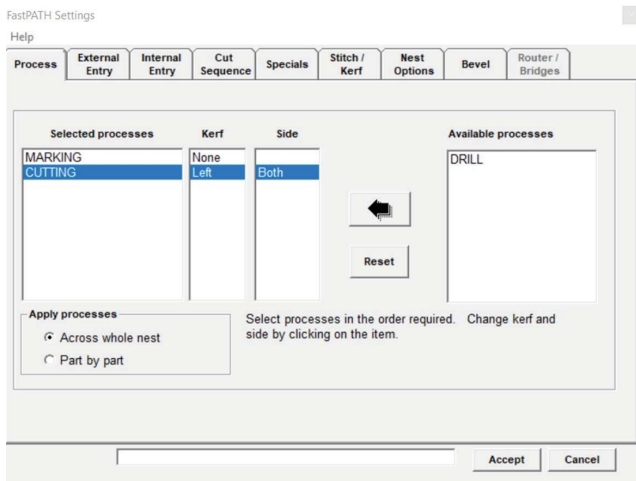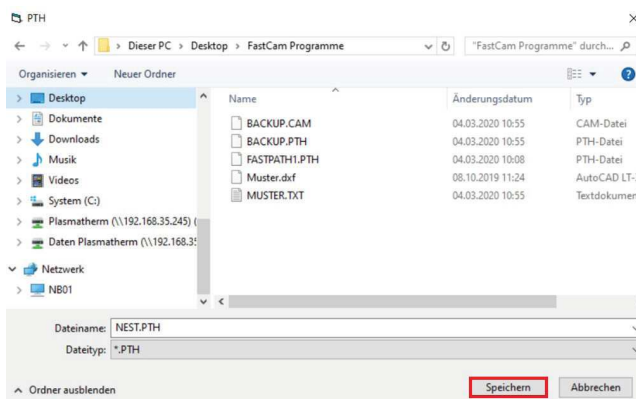

Then FastNEST wants to save the settings of the lead-in and lead-out geometry.

- Select the required directory.
- It will be saved as NEST.PTH.
(The name can be changed. It is important that the file is saved as .PTH)
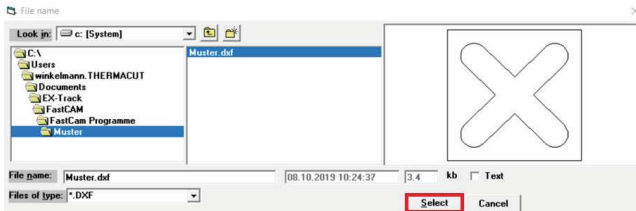- Close the field with „Save" (Speichern).

5. Add cutting parts.

Individual parts can be added and removed by pressing the „ + " and „ – " fields.
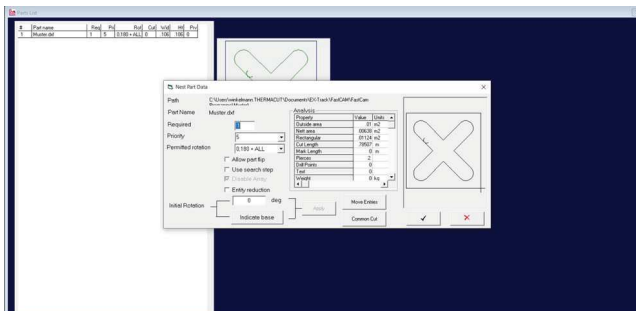
- Select „ + ".

Select the required DXF file to be cut
- Press „Select" to confirm.
Either select the DXF files here, or if the cutting programs have been created in advance, select the cutting programs.
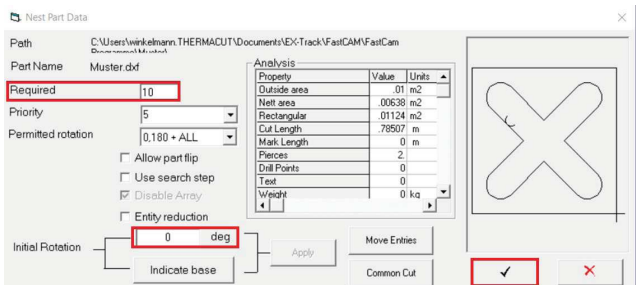
Depending on whether DXF or NC was selected in „Input – File type", the corresponding file type is displayed here.
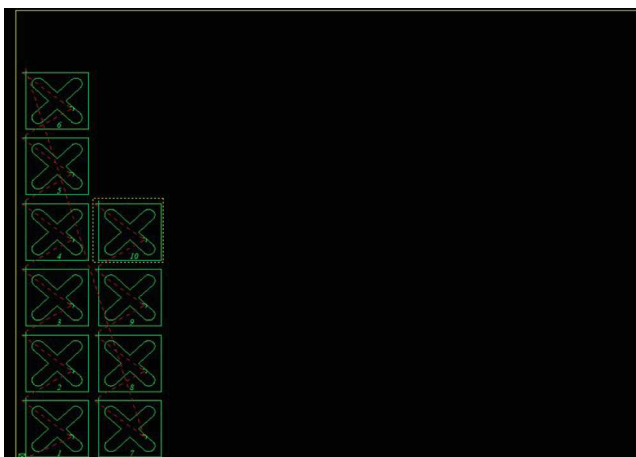
A new window opens, where the amounts can be set..

Enter the number of parts to nest in the "Required" field.
At „Initial Rotation", enter the rotation (if necessary).
Then confirm.

After confirming the parts are nested on the plate.

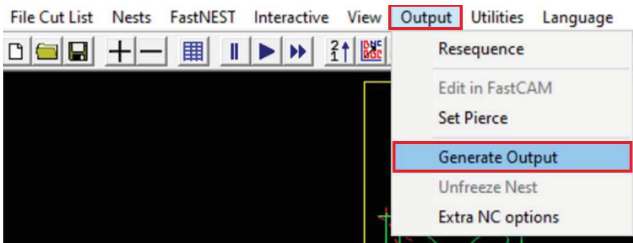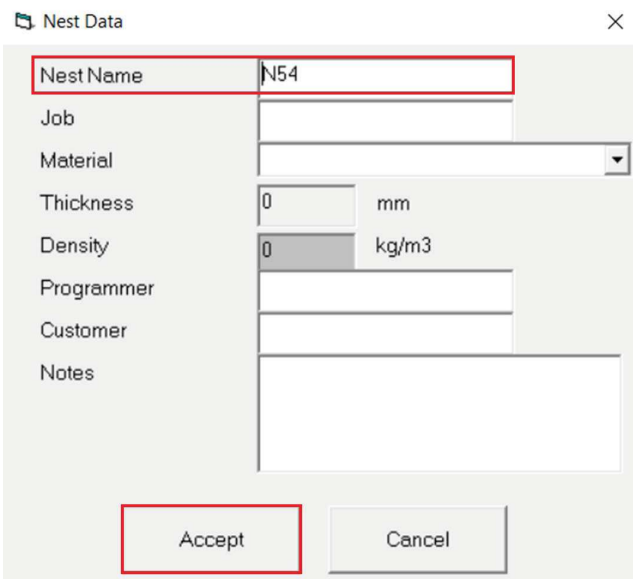## 6. Add more parts.

Starting at Position 4 „Set file type", further parts can be nested on the plate.

As an example, 10 more parts were added here.

7. Save nesting.

Choose „Output" in the menu and click on „Generate Output".

A new window „Nest Data" appears.

- „Nest Name" enter the name.
- Accept.

In the next window define one of the possible nesting sequences.
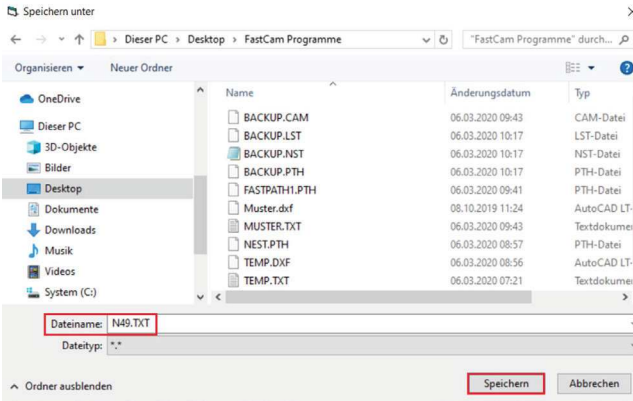
Examples for nesting sequences:
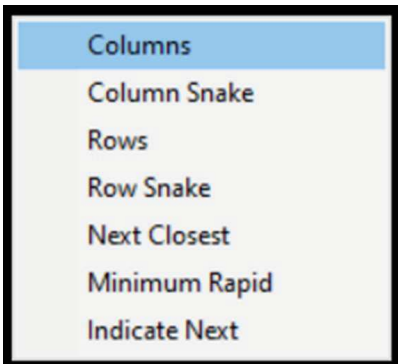
"Next Closest" = The closest part is cut next.

"Minimum Rapid" = Try to keep the rapid movements from part to part as short as possible.
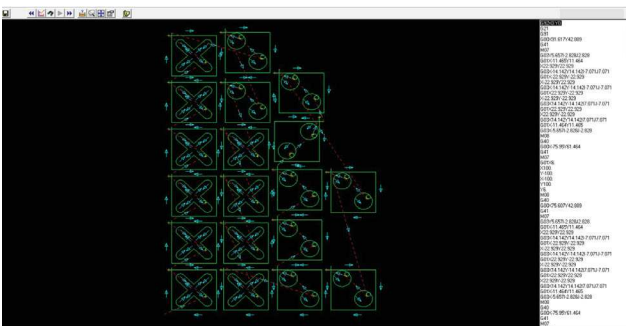
The menu changes.

Confirm by clicking the

A new window appears where the nested program can now be saved.
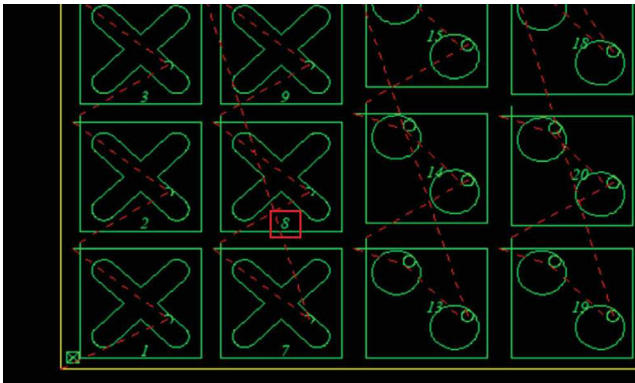
8. Check nesting.

After saving the nesting is automatically opened in FastPLOT.

As descriped in point 10 "Check cutting program", the nesting can also be checked and dry-run.

9. Nesting program.

The finished cutting program can now be saved from the PC to a USB flash drive and can be cut on the EX-TRACK®. The nesting program is also saved as .TXT.

Information about the Nesting

The numbers displayed in the parts indicate the sequence.

By double-clicking on a part, the sequence can also be changed. Red box below: Drag up to Sequence 8 / 2 the individual parts can also be rotated afterwards.
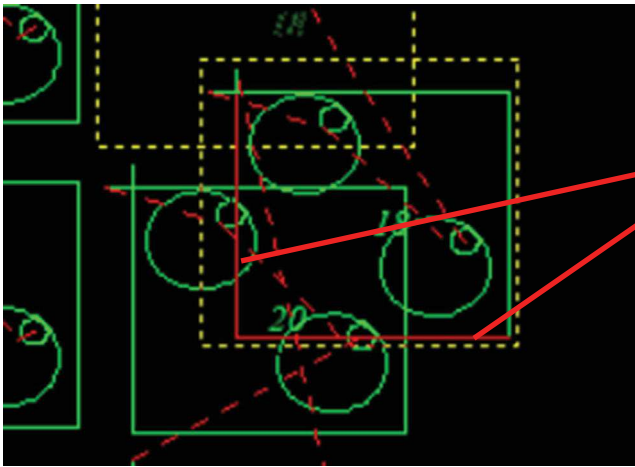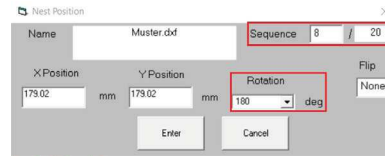
If you click on a part and keep the left mouse button pressed, the chosen part can be moved.

As you can see, the software also shows nesting errors as soon as parts placed on top of each other. This is shown with a red line.
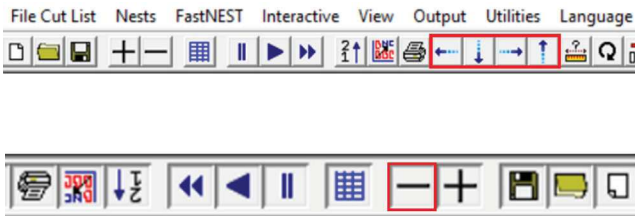
When a part is selected on the plate, it can be repositioned to the set nesting distances using these arrow keys.

If a nested part should be removed, select the required part and delete it with the „ - " button.

NOTES:

NOTES:

THERMACUT®

THE CUTTING COMPANY®