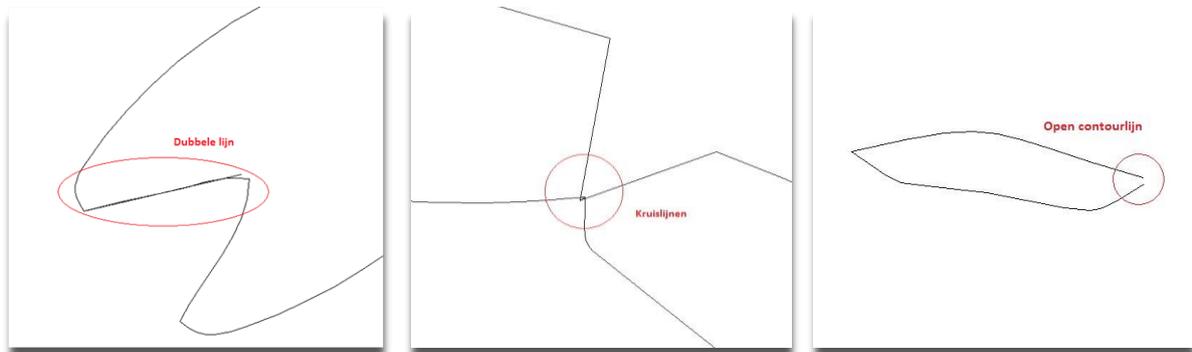




## Instruction Solidworks

IMPORTANT! BEFORE YOU UPLOAD A DRAWING, IT IS ESSENTIAL THAT THERE ARE NO DOUBLE LINES.

IF YOU WANT TO ENGRAVE, IT SHOULD NOT HAVE ANY OPEN CONTOUR LINES AND / OR CROSS LINES. FIRST ADJUST YOUR DRAWING BEFORE CONTINUING TO THE FOLLOWING STEPS.

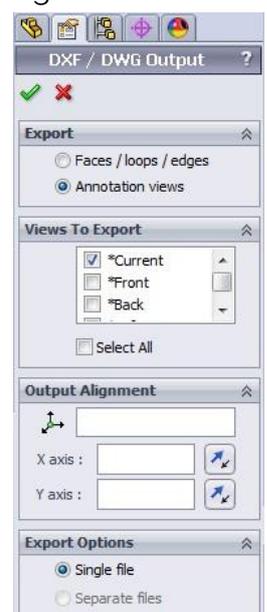


### Step 1: Save as

When the drawing is ready, make sure you have it in the front view. Go to *File > Save as* and select .Dxf. Click on the "options" button in the same popup. Now you will get a window with a number of options for exporting a DXF file. It is important that **Version R2000-2002** is selected here, and that export **splines as splines** are checked. Then press **OK** to close the window.

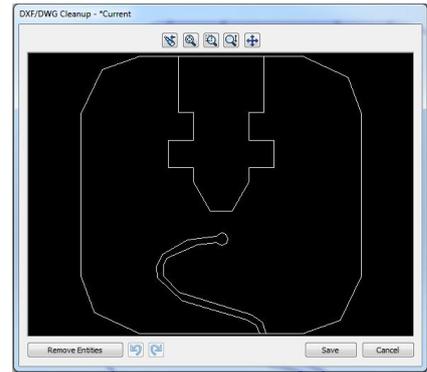
### Step 2: View

You have returned to the previous window, then click **Save**. Now the window which is shown on the right appears on the left. If your drawing is in the front view, then in Views to Export > Current is the correct option that should be checked. If your drawing is in a different plane as a front view, select a different view here. Then press the green check mark at the top left.



### Step 3: Check your .DXF file

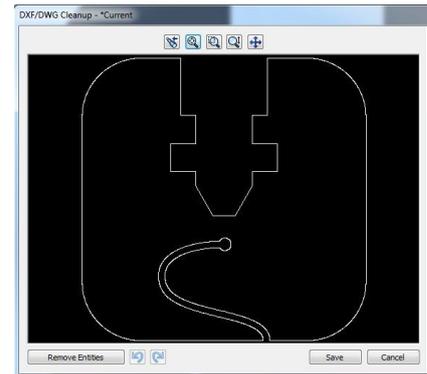
Now a new window appears where the newly created dxf is displayed. The dxf is usually a 1 on 1 copy of the drawing, but sometimes all arc lines (splines) become very angular. This is clearly shown in the example at the right. If this is not the case, press save and go to step 5. If this is the case, continue with step 4.



### Step 4: Extrude the shape

Click on cancel to return to your working drawing. Go to Features > Extruded Boss / Base in the top toolbar. Make a solid shape of the drawing and then export it again as .DXF, this prevents making the arc lines angular.

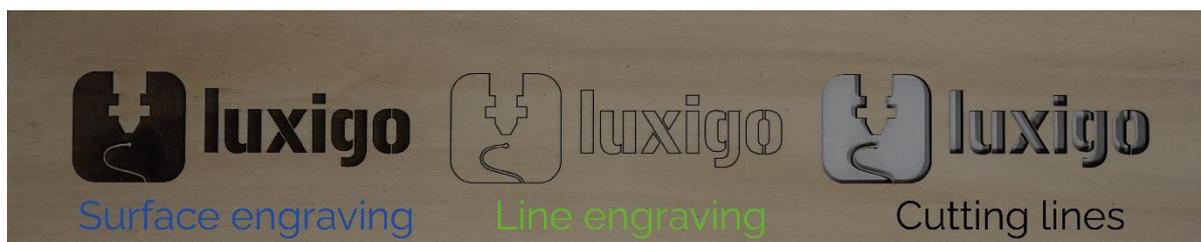
If the drawing has been extruded, click with the right mouse button on the plane from which a drawing is to be made, and click on the normal to arrow. Now the plane will be in the correct front view and you can repeat steps 2 and 3. If all goes well, the drawing must now have good arc lines such as the drawing at the right.



### Step 5: Choose your method

Unfortunately it is not possible to give colors in Solidworks to different parts within the drawings. You can open the .DXF drawings from Solidworks in Illustrator. It is then possible to provide the lines with colors in this program.

However, if you do not have Illustrator, please indicate in the comments of the upload form what should be engraved line and / or surface.



#### Black lines - Laser cutting

Leave the outer contour of the part to be cut black, so this is not the outer contour or working surface of the entire drawing.

#### Red lines - Internal cutting lines

Internal cutting lines are the surfaces to be cut that fall within the outer contour, for example, holes.

Make these lines red. The reason why internal cutting lines must be used is as follows.

When an "O" is to be cut, it is important that the hole is cut first and then the outer contour of this letter. When the outer contour is first cut, the part will be slightly skewed, when the hole is cut it will no longer be in the right place.

### Line engraving (green)

Instead of engraving a surface, which takes a lot of time, it is also possible to have the laser cut quickly and with low power. The material is then drawn as if it were. Make engraving lines green.

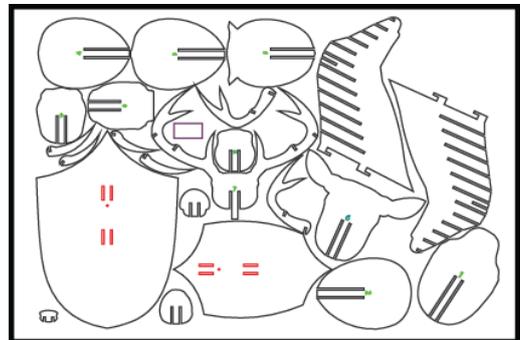
### Surface engraving (blue)

The laser moves like a printer, so that a layer is always burned. As a result, the surface to be engraved is slightly sunk into the material. Make surfaces engraving areas blue.

## Step 6: Nesting

Place as many parts as possible that should be cut from the same material on one work surface. For regular cutting jobs, the maximum size of the working surface is 1200x1200 mm, for surface engraving it is 1150 x 1200 mm.

Only the standard sizes 300x200mm, 600x400mm and 900x600mm are in stock. Larger dimensions up to 1200x900 mm can be supplied after consultation.



### Solidworks

Within Solidworks it is possible to put multiple parts in one sheet before export. Make a new assembly and put all parts in one plane.

Save the assembly as part and open the part afterwards.

It is possible to export the part as described in the previous steps.

### Illustrator

In Illustrator, multiple drawings can easily be moved to 1 file, in Solidworks this is rather difficult. You can make a new drawing in Illustrator. By pressing **shift + ctrl + p** it is easy to import multiple .dxf files.

## Step 7: Upload file

Submitting multiple files is no problem. To keep it clear we want to receive the files with the file name like: **"Your name – size – material + thickness"** (Luxigo – 600x400 – MDF2mm)

Upload your file on <https://www.luxigo.nl/en/upload-bestand/> and you will receive a quotation from us with a price and delivery indication as soon as possible.