



# VxCRAFT - QUICKSTARTGUIDE

This quick start guides you step by step through a simple project. Our goal is for you to quickly and easily understand VxCraft's workflow and get an overview of the software.

So, let's get started - with a simple, but impressive engraving!



**Tip:** You can also find this tutorial as a video on Youtube at:  
[www.youtube.com/@vxcraft](https://www.youtube.com/@vxcraft)

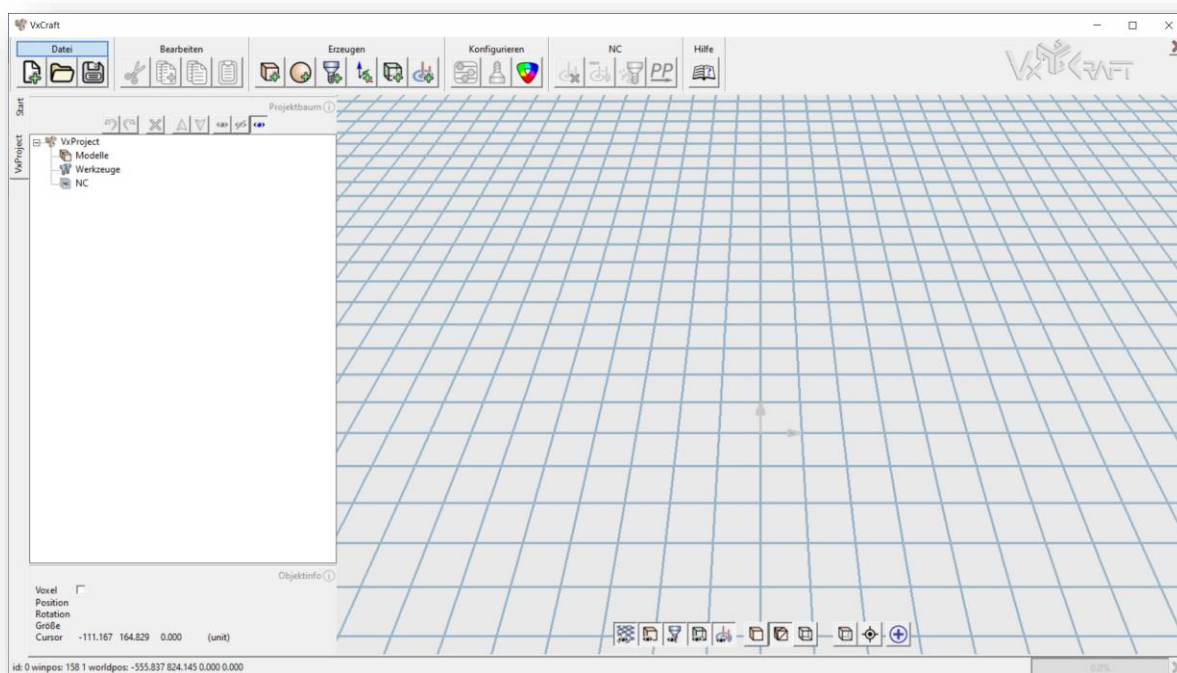




# A NEW PROJECT

To start a new project with VxCraft, simply go to the "File" menu category and select "New". This will start with an empty workspace and you are ready to create your first CNC wood art.

For a usual workflow it is recommended to save the project also at the beginning via "File/Save as".



On the left side of the program interface you will find the project tree.

We will go through this step by step from top to bottom, which is the usual workflow for creating an NC project.

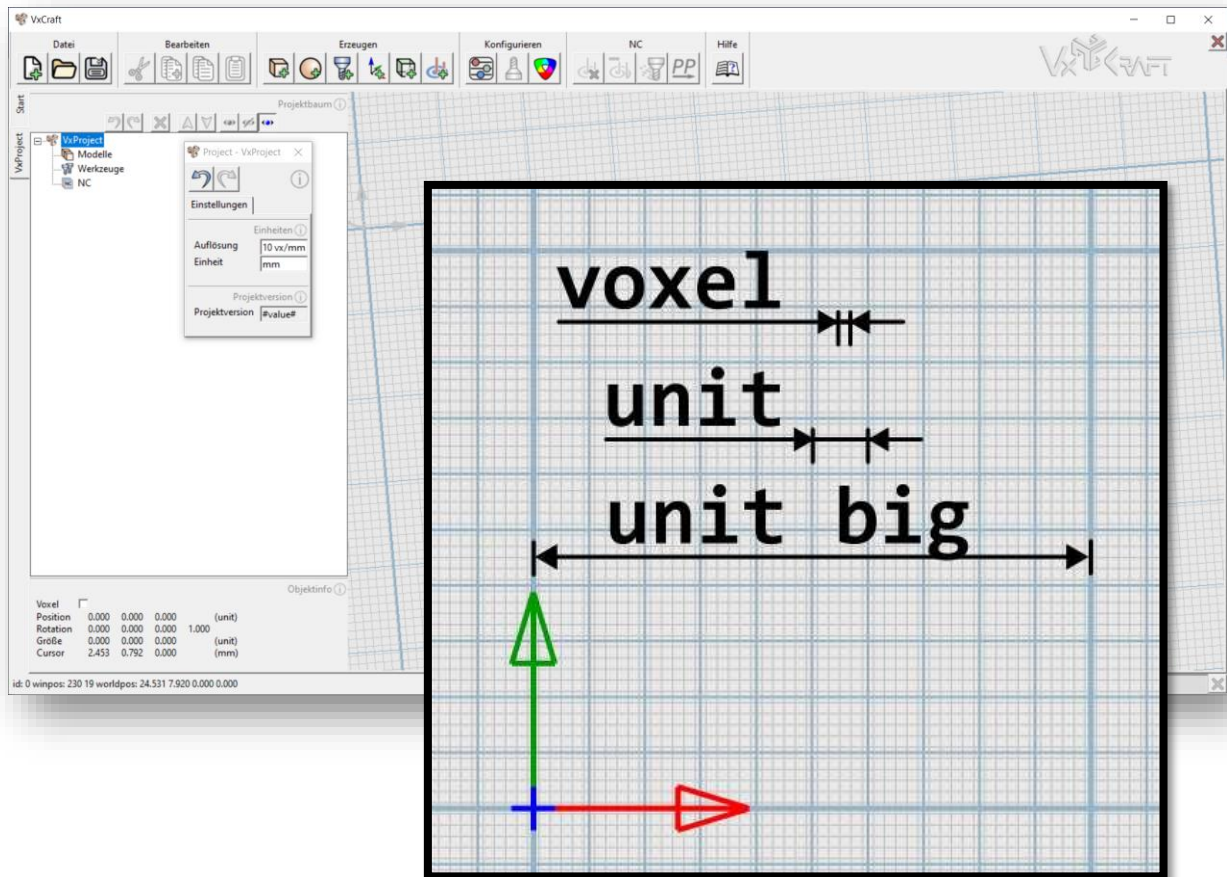




# PROJECT SETTINGS

Let's start with the top element - the project itself.

Double-clicking or using the context menu on this element opens the settings window. Here we set the resolution to 10 voxels per millimeter - an ideal setting for fine details in small logos.



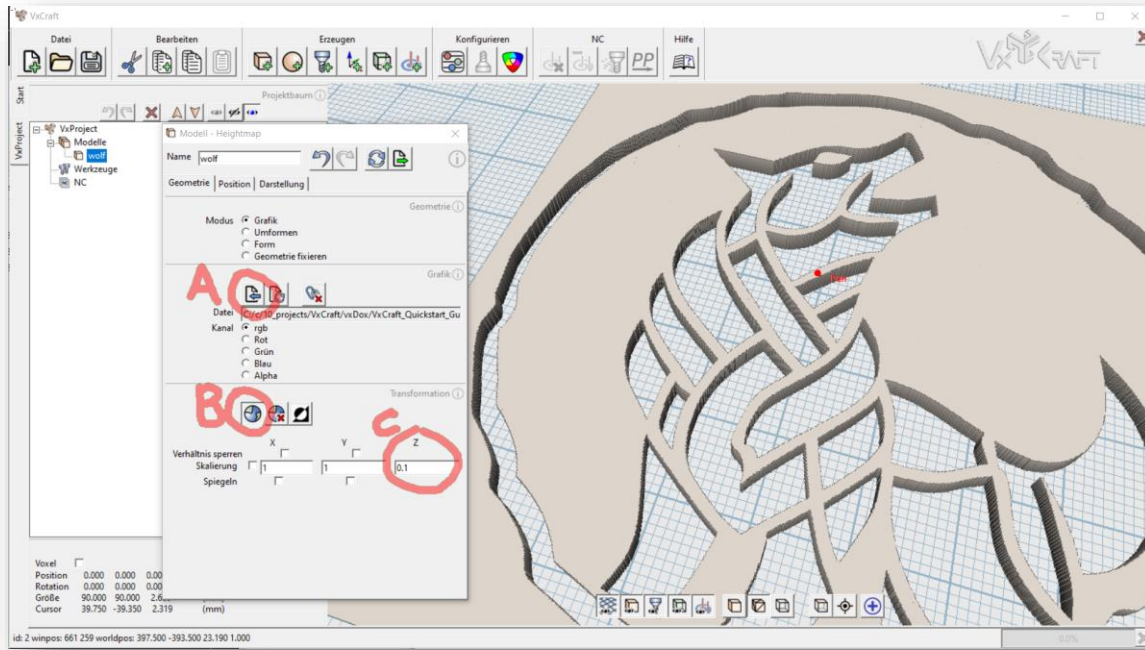
Once we do that, the working grid will adjust. The very fine lines show the size of a voxel or pixel. The slightly stronger lines show our resolution and the distance in our case is one millimeter. The thick lines are always ten times the distance of our unit, which is 10 millimeters. This gives us a feeling for the scale of our project.





# THE FIRST MODEL

Okay, now we can load the logo! Using the context menu in the project tree on "Models" we create a new heightmap model. In the configuration dialog that opens automatically, we assign our graphic to the model by importing the corresponding file (A).



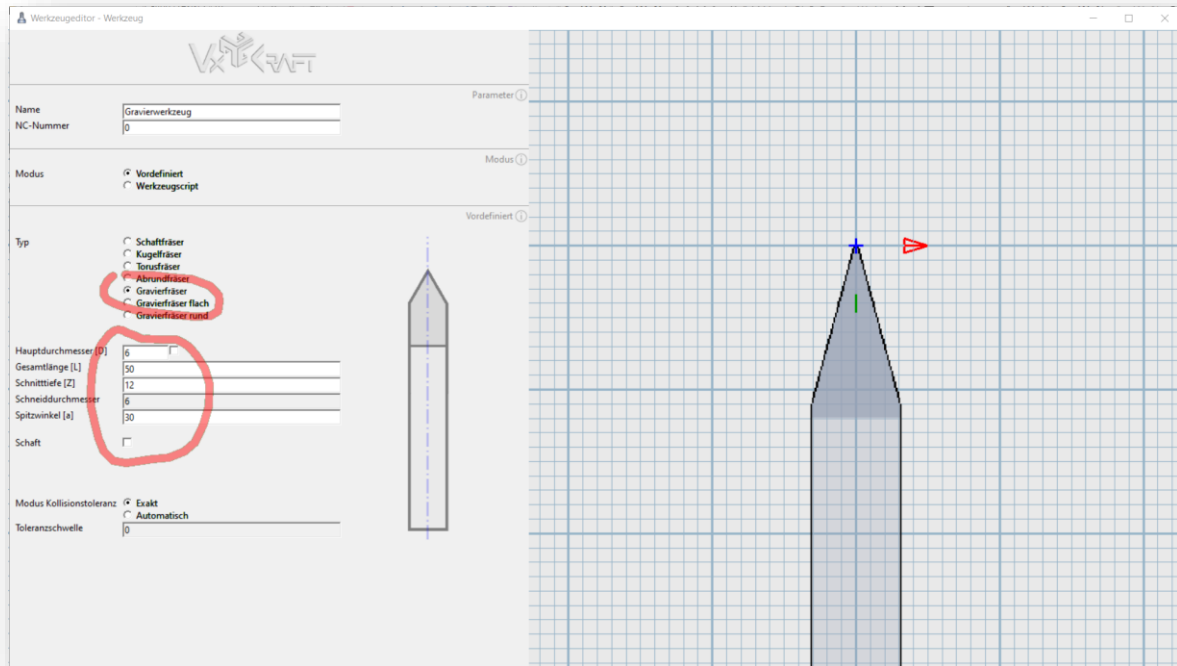
However, due to the high contrast of the graphics, the model is still too high. No problem, we just need to correct the height. To do this, we activate the transformation (B) and adjust the z-scaling (C) to make the height map flatter. It's not important to set the exact height - the exact engraving depth will be set later.





# THE GRAVER TOOL

The model is ready, but we still need to tell VxCraft which tool we are using. For this we create a new tool - again via context menu on "Tools" and "Create tool". This time, in addition to the tool settings window, the tool editor will open to define the geometry.



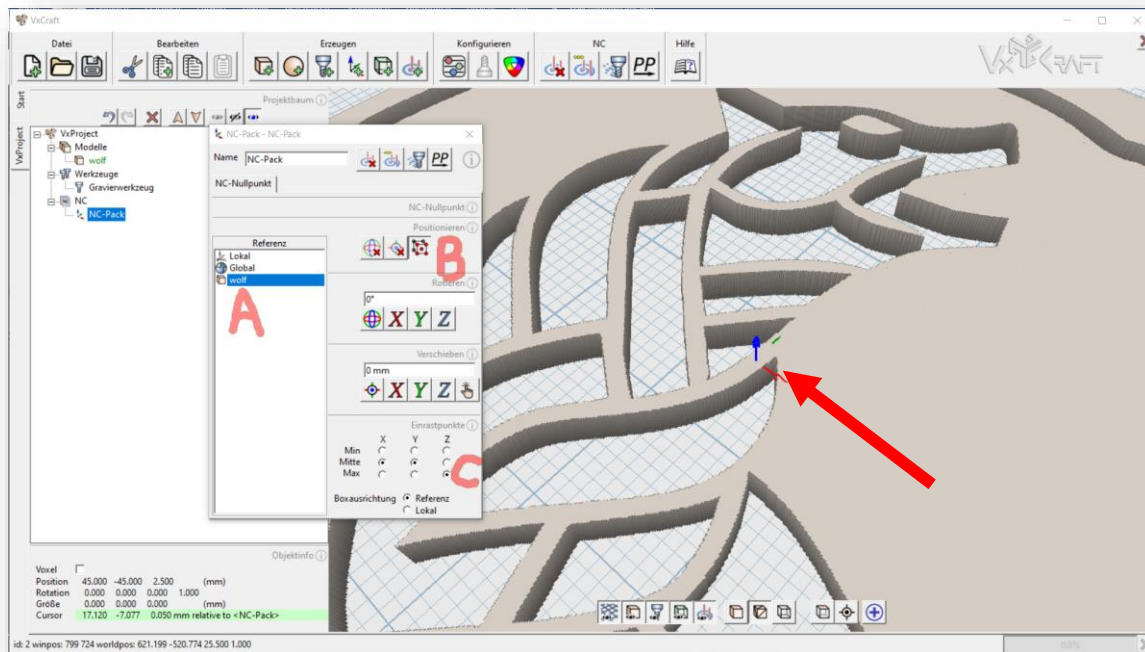
We select the predefined type of engraving cutter and define the exact geometry by entering the dimensions. We can close the window.



# NC-PACK / NC-ZERO POINT



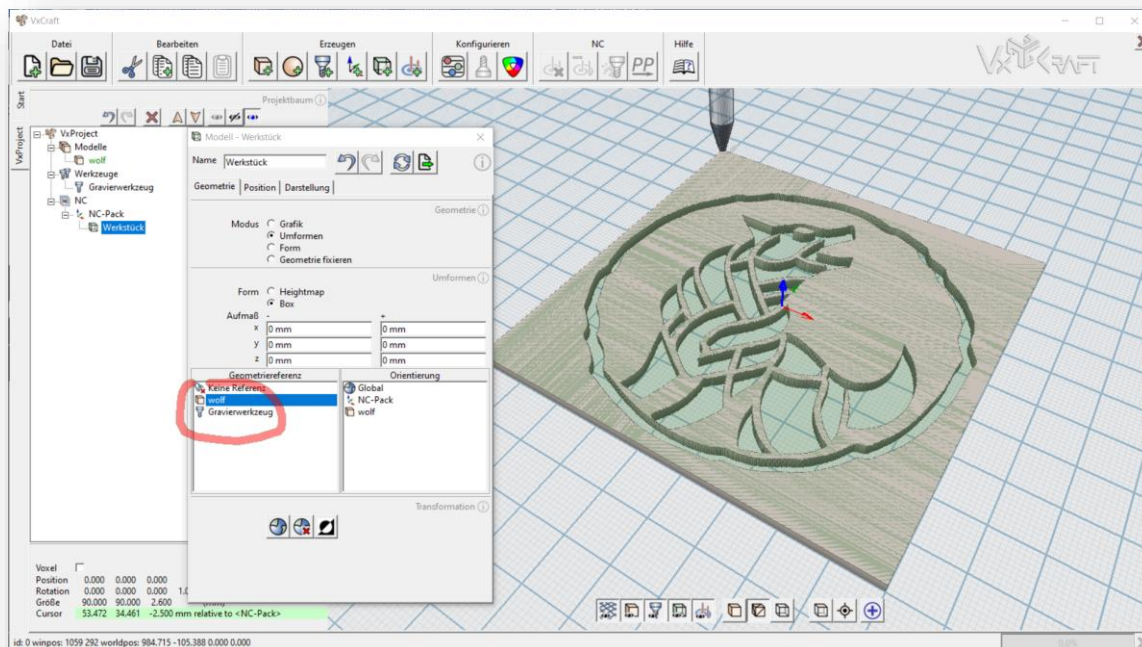
The next step is to define the machine origin. We create a new NC package via the context menu on "NC".



In our example, we want the NC origin to be at the top center of the model. So we select our logo as reference (A), activate the snap points (B) and select the desired position in the snap table (C).

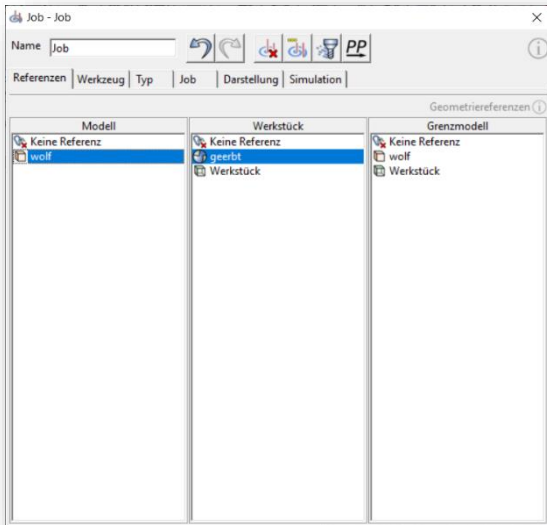


The next object we need is the stock. Using the context menu on our NC package, we can create our new workpiece.



The dimensions should simply match our model, so we simply select our logo as a reference and the stock immediately appears as a green/transparent object in the render view.

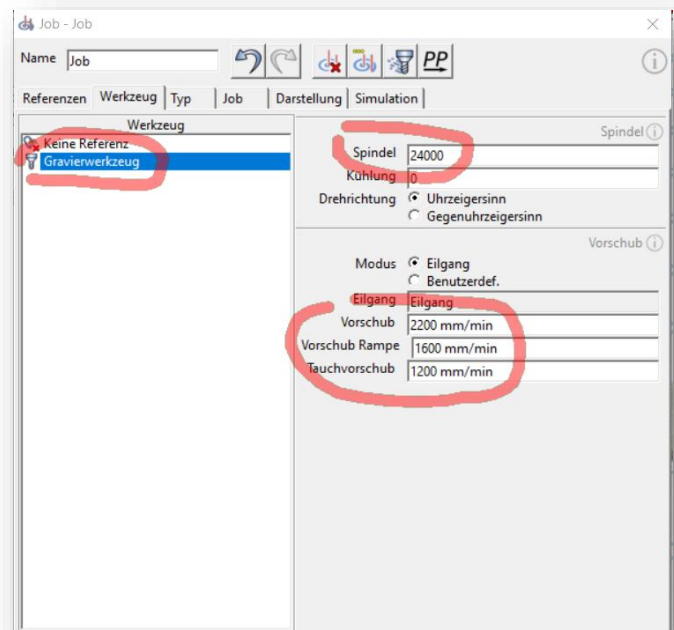
# ENGRAVING JOB / REFERENCES



Okay, all relevant geometric objects are now defined, so we can create our engraving job. Again via context menu on "NC-Pack" we create a new job.

In the opened settings dialog, we switch to the "References" tab and select our model.

In the next tab - "Tool" - we select our engraving tool and set the cutting data (spindle speed and feeds).



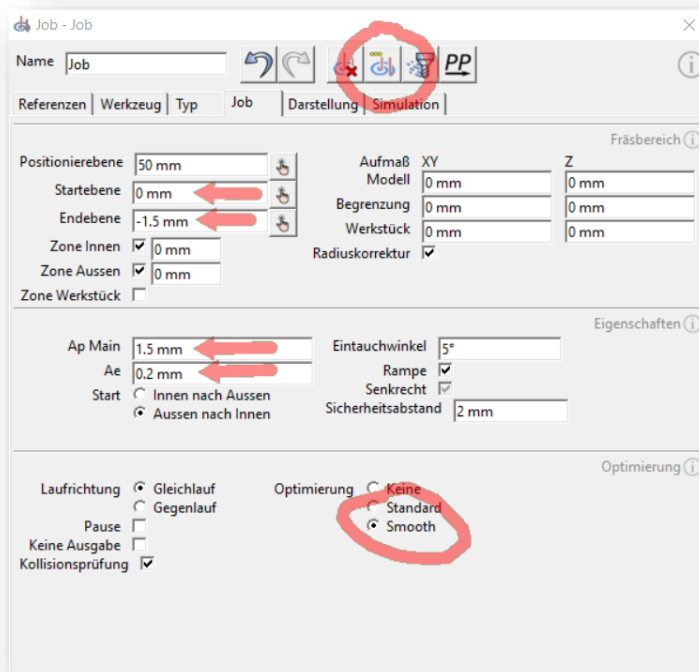
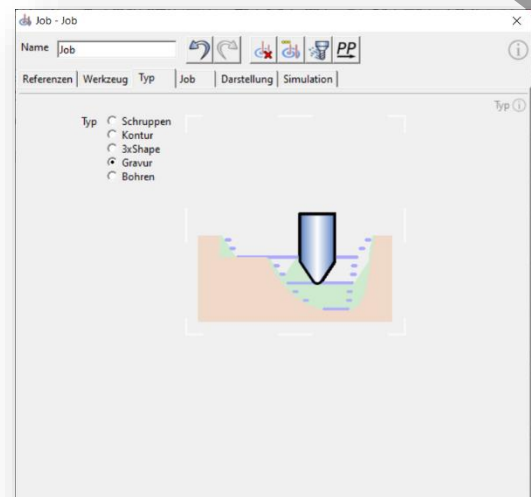




# JOBPARAMETER

We switch to the "Type" tab and select the "Engraving" milling strategy.

In the next tab - "Job" - there are some parameters, but don't worry - most of them can be left at their default settings. The most important ones are the Start Height, which we set to 0, and the End Plane, which sets the engraving depth to -1.5 mm.



With 'AP Main' we specify the maximum vertical infeed. Our engraving is not that deep, so we can feed everything at once. We choose the side feed 'ae' to be very small at 0.2 mm, so that our pointed cutter can also cleanly ream the bottom surface.

With a small tweak, we change the optimization mode to 'Smooth', which also

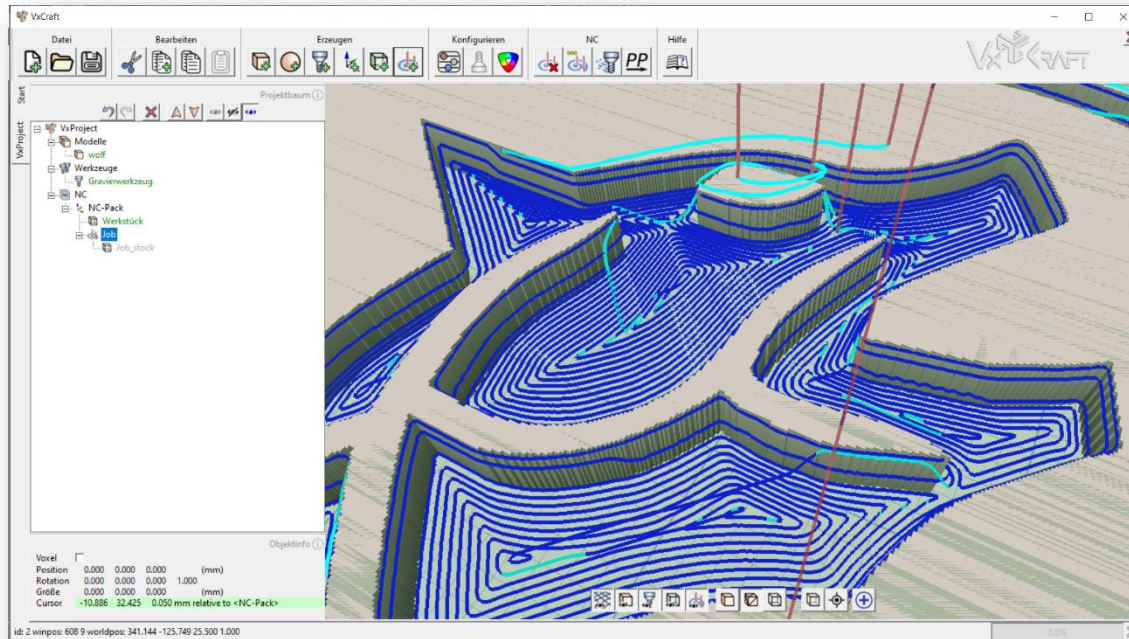
smoothes the calculated toolpaths.

That's it. Via the toolbar we execute the calculation of the job.

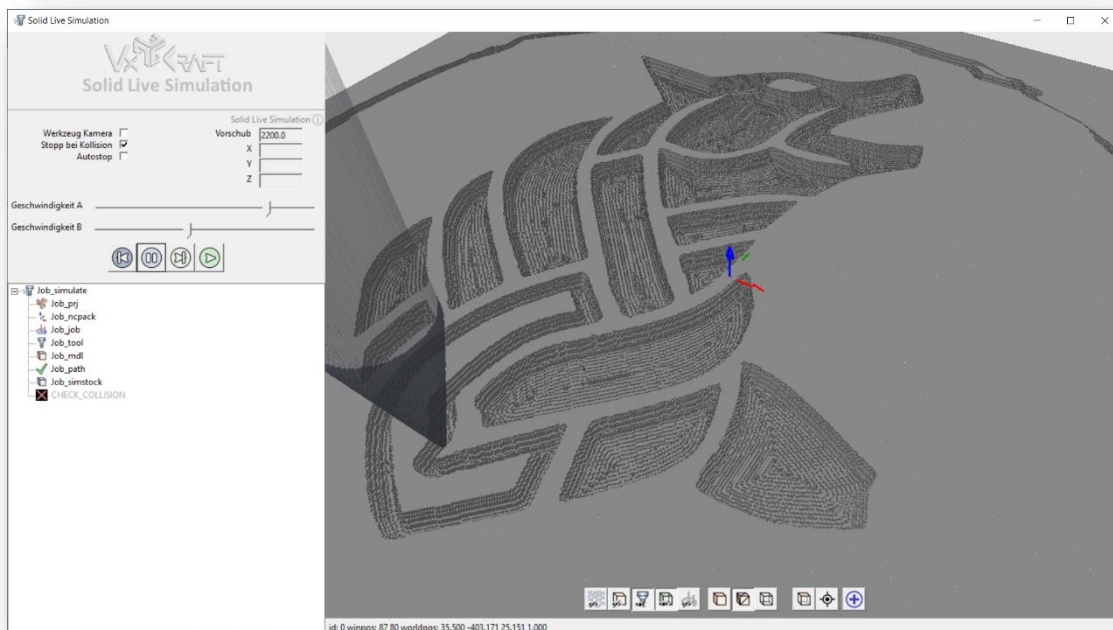




# TOOLPATHS / SIMULATION



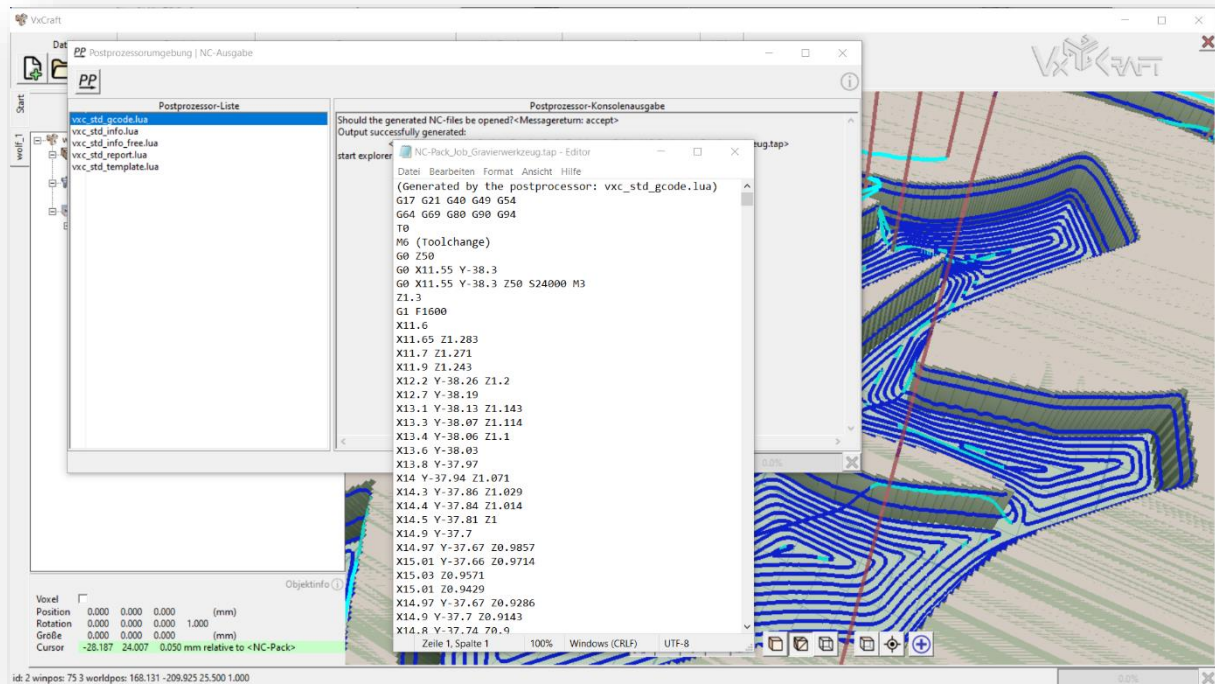
The toolpaths have been created cleanly and before we go to the last step, we visualize the material removal with the Solid Live simulation. The simulation environment can be started via context menu on the job.





# POSTPROCESS

To make the toolpaths usable for the CNC machine, we need to export the data. To do this, select our job and open the postprocessor environment using the context menu.



In our example, we will use the standard G-Code postprocessor. If you don't have a postprocessor installed, you can find a selection of free postprocessors to install at [www.vxcraft.com](http://www.vxcraft.com).



# BRING YOUR CNC VISIONS TO LIFE



We hope this QuickStart Guide helped you get started with VxCraft. You can find more help and information in the **VxCraft manual**, on our Youtube channel [www.youtube.com/@vxcraft](https://www.youtube.com/@vxcraft) and of course you are welcome to visit us on our website [www.vxcraft.com](https://www.vxcraft.com)!



WE WISH YOU SUCCESS WITH YOUR PROJECTS

—

BRING YOUR CNC VISIONS TO LIFE!



[www.vxcraft.com](https://www.vxcraft.com) | [info@vxcraft.com](mailto:info@vxcraft.com)

© 2023 VxCraft