

About STEPCRAFT

The STEPCRAFT GmbH & Co. KG based in Menden, Germany, was established in 2012. The company produces and sells multifunctional CNC machines and accessories for private use and small-business applications and is one of the market leaders in this area. With the STEPCRAFT 210, 300, 420, 600 and 840, STEPCRAFT currently offers five different types of machines that can be equipped with a broad variety of system-guided tools such the HF-Spindle, the 3D Print Head, the Hot Wire Cutter or the Drag Knife.

The CAD, CAM & CNC Workbook



STEPCRAFT.

FOREWORD	1
CONCEPT	3
M01 INTRODUCTION	
1 CNC-TECHNOLOGY	
1.1 History	6
1.2 Manufacturing Processes	7
1.3 Coordinates and Reference Points	8
1.4 The Process	9
2 CNC-MILLING	
2.1 Milling	10
2.2 Up- and Downcut Milling	11
2.3 Peripheral and Face Milling	11
2.4 Types of Machines	12
2.5 Milling Tools	12
2.6 Formulae	13
└ 2.6.1 Cutting Speed	13
└ 2.6.2 Feed per Tooth	13
└ 2.6.3 Feed Rate	13
└ 2.6.4 Stepdown	13
3 CAD	
3.1 The History of CAD	14
3.2 Design	15
4 CAM	
4.1 Using CAM to Define Objects	16
4.2 Working with CAM Systems	17
5 MACHINE CONTROL	
5.1 Preparing the G-code	18-19
6 COMPREHENSION	
6.1 Questions	20-21

7 CREATING CAD MODELS WITH AUTODESK FUSION 360

7.1	The Workspace	24-25
7.2	Display Options	26-27
7.3	Using the Grid	28

8 SKETCHING

8.1	Sample Sketch	29-34
-----	---------------	-------

9 2D-TOOLS

9.1	Geometrical Shapes	35-39
9.2	Tools for Manipulating 2D Shapes	39
9.3	Trim	40
9.4	Offset	41
9.5	Sketch Skale	41
9.6	Mirror	42
9.7	Circular Pattern	42
9.8	Rectangular Pattern	43

10 CONSTRAINTS

10.1	Sketch Elements	44-46
------	-----------------	-------

11 3D-TOOLS

11.1	3D Objects	47
11.2	Fillet and Chamfer	48
11.3	Holes	49
11.4	Split and Combine	50-51
11.5	Multiplying	52-53
11.6	Sculpting	54-55

12 PRACTICAL USE

12.1	In-depth Example	56-57
12.2	The Frame	58-68
12.3	Bearing Blocks	69-72
12.4	Wheels	73-77

13 COMPREHENSION

13.1	Questions	78-79
------	-----------	-------

M03 CAM

14 CAM IN AUTODESK FUSION 360

14.1	An Introduction to CAM	82
14.2	Setup	83-86
14.3	Calculating Toolpaths	87-88
14.4	Tool Definitions	89-91
14.5	Milling Strategies	92-109
14.6	G-code Output	110-113

15 PRACTICAL USE

15.1	Sample CAM Process	114
15.2	Creating a Setup	115-116
15.3	Creating the Tool	117-118
15.4	Defining the Toolpaths	119-126
15.5	Post Process	127-128

16 G-CODE PROGRAMMING

16.1	Introduction and Basic Structure	129
------	----------------------------------	-----

17 G-CODE GLOSSARY

17.1	EIA-274-D Standard: Supported G-codes	130
------	---------------------------------------	-----

18 COMPREHENSION

18.1	Questions	131-135
------	-----------	---------

M04 MACHINE CONTROL

19 MACHINE CONTROL WITH UCCNC

19.1	Introducing the Software	138-139
19.2	Preparing the CNC System	140-141
19.3	Preparing for Milling	142-143
19.4	Loading a G-code File	144-145

19.5	Setting the Machine Zero Point	145
19.6	Setting the Workpiece Zero Point	146-147
19.7	Setting Spindle Speed	148-150
19.8	Safety Checks	151
19.9	Milling	151

20 PRACTICAL USE

20.1	Sample Milling Job	152-153
20.2	Clamping the Stock	153-155
20.3	Setting up the Tool	156-158
20.4	Loading the G-code and Setting the Machine Zero Point	159-160
20.5	Setting Workpiece Zero	161-163
20.6	Safety Checks	164
20.7	Milling	165-166

21 COMPREHENSION

21.1	Questions	168-169
------	-----------	---------

1 CNC TECHNOLOGY

1.1 HISTORY

CNC (**Computerized Numerical Control**) is a technology that uses computers to control machine tools. It is an integral and indispensable part of today's complex, customizable world of products and production techniques. The vision of the future incorporated in the move toward Industry 4.0—which promises completely new ways to organize and control the entire supply chain—is unthinkable without the use of CNC. As recently as fifteen years ago, this technology was the preserve of the automobile, space travel, and other innovative industries, while today, a local carpentry shop may have its own CNC machine and can easily produce custom hinges and other bespoke parts. One of the most important aspects of CNC technology is its ability to automate movements that used to be performed manually, such as the positioning of cranks and hand wheels.

Visual checks and embossed scales make it simple to adjust a manually-operated machine to achieve the desired result. The motors used in a CNC machine are not capable of such checks. They also have to be precisely instructed regarding the number of revolutions per minute (rpm) and the direction of rotation required by the job at hand, and the point at which the current movement needs to end.

Developments in electronics and the introduction of transistors in the middle of the twentieth century led to the creation of the NC (**Numerical Control**) technology, which enabled the first steps toward machine tool automation. Early NC techniques used punched-card storage to program binary switching operations. Each hole in a card altered the binary state of a switch, and altering the time lapse between two holes made it possible to control the duration of an operation. Parallel lines of holes on a single card facilitated the programming of multiple, simultaneous operations.

CNC technology replaced punched cards with magnetic memory, which increased both the number of operations a machine could perform and their precision. Simple binary switching was replaced by sequences of complex commands known as G-codes. These consist of sequences of letters and numbers that enable the definition of three-dimensional spatial coordinates as well as rpm settings and other important machining parameters. G-code has to be converted into a form that allows the machine to correctly interpret the nature and timing of the switching operations involved. Control of the motors and other components of the machine itself is still largely performed using the rules established with NC technology. One major advantage of using computers to control machining processes is the ability to automatically prevent collisions between the tool and the workpiece—a procedure that used to involve complex manual calculations.

1 CNC TECHNOLOGY

1.2 MANUFACTURING PROCESSES

Today, CNC technology is used in all manufacturing processes that involve machine tools. Manufacturing processes are used to create all manner of components and are defined in six major classes by the DIN 8580 standard (see figure 1). The aim of such processes is to shape components and alter the properties of the their materials—processes that can be approached and executed in many different ways.

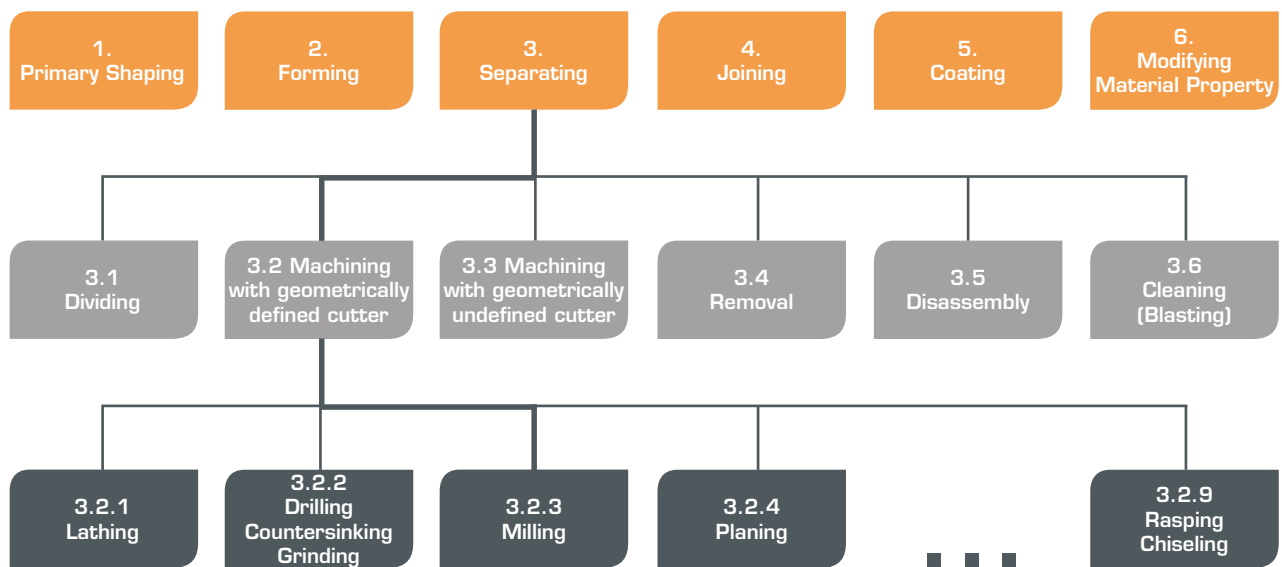


Fig. 1

- **Primary Shaping:** The basic shape of a component is created from scratch using the selected material. 3D printing is a contemporary example of this type of process.
- **Forming:** The shape of a solid blank is permanently altered but the material remains the same. Forging is an example of this type of process.
- **Cutting:** Parts of a blank are removed to create the desired shape. Machining techniques, such as sawing, filing, drilling, milling, and lathing, are examples of this type of process.
- **Joining:** Individual components or subcomponents are joined—for example using glue, welding techniques, bolts, or screws.
- **Coating:** The surface of a component is altered permanently using an amorphous material—for example, paint, varnish, or chrome-plating. This type of process is usually used for protection purposes or to provide a high-grade finish.
- **Modifying Material Properties:** Improvement of the base material for a specific purpose. For example, shock hardening metal to create a specific atomic structure in its surface. Metal hardened this way wears out more slowly and has an increased lifespan.

7 CREATING CAD MODELS WITH AUTODESK FUSION 360

7.3 USING THE GRID

The grid is one of Fusion’s most important design aids. It is switched on by default, and enables precise orientation and positioning of objects on the canvas. Most of the grid settings can be made using the button in the navigation bar at the bottom of the application window.



Fig. 34: The Grid and Snaps menu

The grid can be set to either fixed or adaptive mode. In fixed mode, the major grid spacings and minor subdivisions remain the same. For example, if you set major spacings of 100 mm with 10 subdivisions of 10 mm each, the spacings you set are retained in zoomed views. In adaptive mode, the spacings adapt automatically to the current resolution. In other words, the greater the zoom factor, the smaller the division displayed in the grid. In most cases, the flexibility provided by adaptive mode is preferable. The sketch palette is automatically displayed when you create a new sketch, and includes some frequently-used basic settings for the grid.

However, the grid does not just aid orientation. The **Snap** function automatically selects the nearest grid point for the current drawing tool, thus ensuring precise placement of the start and end points of lines. For example, if you are working with a 1mm grid, the precision this provides means you can often do without defining dimensions for an object, making the entire drawing process quicker and more effective.

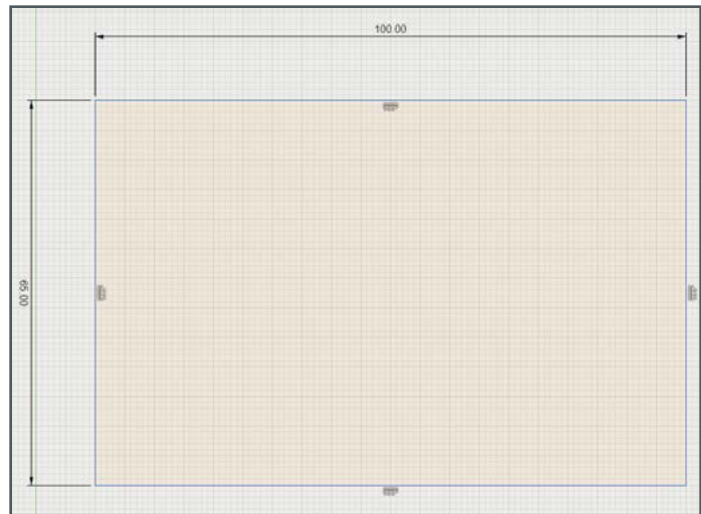


Fig. 35: Using the Snap option

8 SKETCHING

8.1 SAMPLE SKETCH

CAD models are often put together using multiple sketches. A sketch uses simple devices, such as lines and circles, to make a 2D drawing of a surface. Additional steps using various tools are then applied to assemble a 3D drawing from the 2D sketches. This way, you can create models of surfaces, solid bodies, and cavities.

To help you understand how this works, the following example shows you how to sketch a 40 x 80 x 50 mm cuboid (called a “box” in Fusion) with a 20 mm hole in it.

Step 1: Drawing the Basic Box

Navigate to the Sketch menu in the toolbar and start a new sketch using the **Sketch » Create Sketch** command.

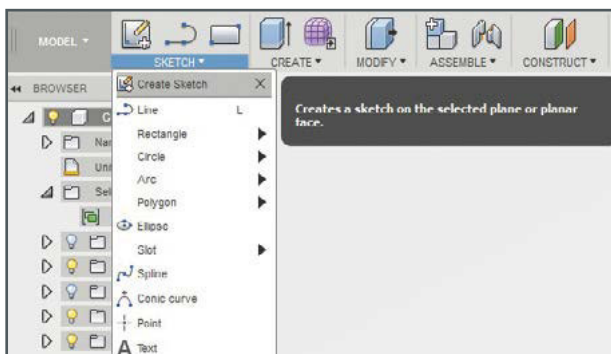


Fig. 36: Creating a new sketch

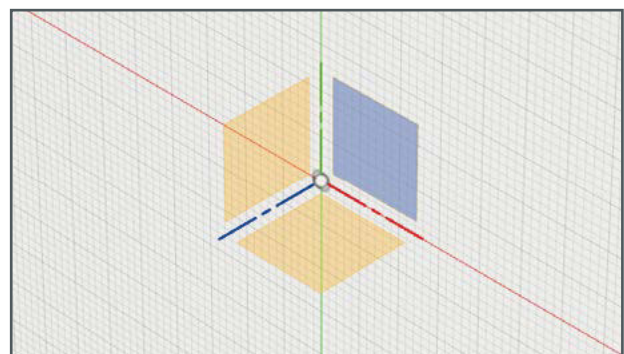


Fig. 37: Selecting a plane

Now set the plane (X-Y, X-Z, or Y-Z) on which to draw your shape. At this point, the application displays a 3D system of coordinates with three trapezoidal surfaces that represent the three planes. Click one of these to select it and convert the drawing to a 2D canvas.

The sketch palette, for setting options and constraints in the drawing, appears at the right-hand edge of the screen, and the sketching tool menu is revealed by clicking the word **Sketch** in the toolbar.

12 PRACTICAL USE

Begin by drawing the axis of symmetry for the mirroring action. This line is 250 mm long and is parallel to the horizontal axis at a distance of 75 mm. Select this line and convert it to a construction line in the **Sketch** palette. The canvas for the base plate is thus bounded by the two major axes and its axis of symmetry. This setup is defined as the default position for the Home view. Now draw the outer, lower border of the base plate.



Fig. 81: Defining the axis of symmetry

The dimensions of 75 mm, 65 mm, and 20 mm are part of the specifications, and the diagonal line results automatically from the others. Use the Sketch palette to add a parallel constraint to the 75 mm and 20 mm lines.



Fig. 82: Sketching the outer borders of the base plate

12 PRACTICAL USE

The cutout along the center line is also part of the given specifications. The 25 mm, 27 mm, and 40 mm lines, and the 163.5° angle are prescribed, as is the 15 mm distance between the left-hand edge and the cutout. The cutout itself is created as follows:

- Create a 40 mm line perpendicular to and to the right of the axis of symmetry, but not starting or ending at the start or end points of the axis. The 40 mm dimension is set as a constraint.
- Create a vertical 25 mm line downward from the left-hand end of the 40 mm line and set its length as a constraint
- Create a horizontal 27 mm line from the lower end of the 25 mm line and set its length as a constraint
- Create a non-dimensioned line about 20 mm long stretching vertically downward from the right-hand end of the 40 mm line. It is important that this line does not have a fixed length, as this would make it impossible to give it a constraining angle.
- Connect the 20 mm line to the 27 mm without entering fixed dimensions
- Right-click the 27 mm line and use the **Fix/UnFix** command to fix it (the line will be highlighted in a different color). This prevents you from unintentionally altering the line during the next step.
- Activate the **Sketch Dimension** tool and select the fixed 27 mm line and the non-dimensioned line on its right. Moving the cursor over the space between the two lines displays the angle between them. Click the box with the angle display and enter 163.5°.
- Use **Fix/UnFix** to unfix the lower (27 mm) line.
- Dimension the left-hand end of the axis of symmetry and the top-left end of the cutout to 15 mm. This moves the cutout to the correct position.

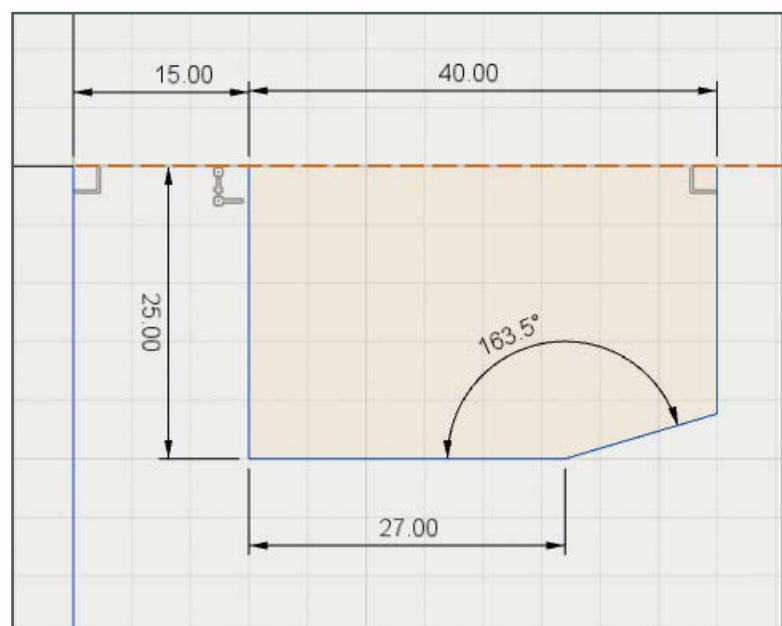


Fig. 83: Dimensioning the sketch

14 CAM IN AUTODESK FUSION 360

14.1 AN INTRODUCTION TO CAM

One of the distinguishing features of Autodesk Fusion 360 is its built-in CAM workspace, with which you can convert CAD designs into the G-code needed to control the machine tools used to manufacture real-world parts. G-code is machine-specific and requires the use of a CAM processor (called a post processor) that is compatible with your machine's controller. Fusion supports a large number of post processors for CNC milling machines. To switch to the CAM workspace, click the **Model » CAM** option in the workspace drop-down menu.

Before you begin the CAM process, you have to arrange the appropriate parts in the model workspace. If you need to create a specific part multiple times, you have to duplicate it in the CAD view. Arrange the parts as close as possible to one another to save material and maximize the number of parts you produce during a single manufacturing run. Remember, though, to leave sufficient space between parts to clamp the workpiece and allow the tool to move freely. CAM software does not automatically “know” what kind of tool you will use and how you wish to apply it. Once you have selected a post processor, you have to define the individual steps in the CAM process. You have to know in advance whether each step you define can actually be executed by the machine you are planning to use.

The steps involved in the Fusion CAM process are setup, toolpath creation, and G-code output.

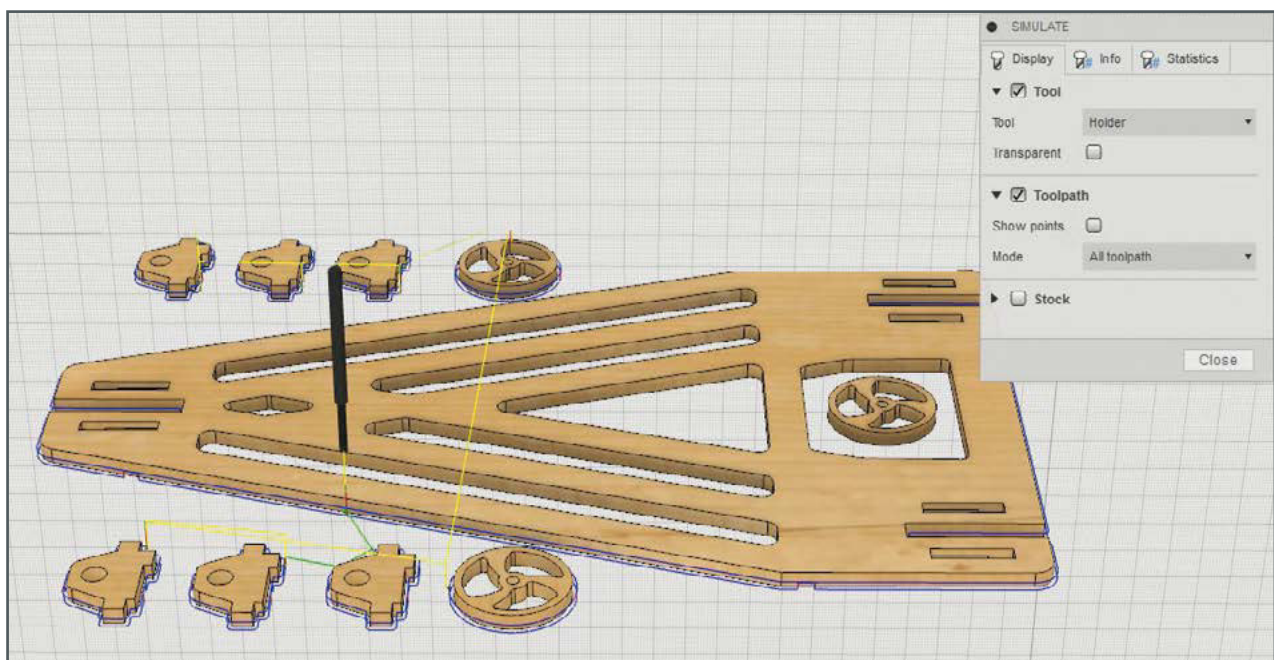


Fig. 109: The Fusion CAM workspace

14 CAM IN AUTODESK FUSION 360

14.2 SETUP

Before you can actually process a workpiece, you have to perform some basic setup steps. These include defining the work coordinate system (WCS) and the type of raw material, called the “stock”. The CAM module uses this data to define the orientation of the parts relative to the machine’s coordinate system.

Clicking the **New Setup** option in the **Setup** menu opens the Setup palette shown in figure 110. The first step is to define the type of operation: in this case, milling.

Now define the WCS. This determines the orientation of the X, Y, and Z axes relative to the workpiece. Because CNC technology uses a right-handed system of coordinates, you only need to define two axes. The orientation of the third axis is derived from the other two. The most important thing to look out for is that the top of the workpiece (i.e., which way is “up”) matches “up” in the machine’s system of coordinates. You can now define the individual parts and the attributes of the raw material.

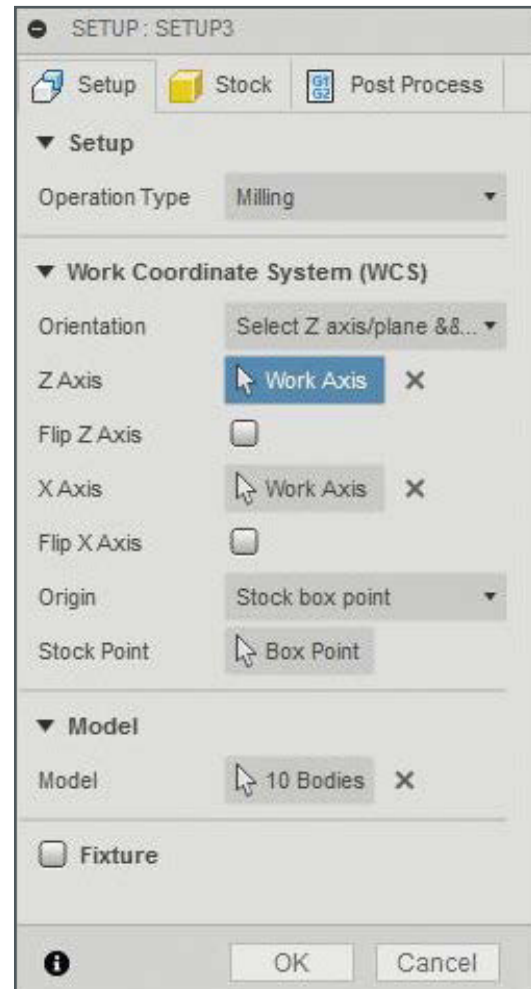


Fig. 110: The Setup palette

Step 1: Orienting the Axes

This is where we define the spatial axes and the workpiece origin. The Orientation drop-down menu contains the following options:

- Model orientation
- Select Z axis/plane & X axis
- Select Z axis/plane & Y axis
- Select X & Y axes
- Select coordinate system

19 MACHINE CONTROL WITH UCCNC

19.4 LOADING A G-CODE FILE

Click the **LOAD FILE** button to load a G-code file and display the code in the output window, where the first command is highlighted. The active code line is displayed in the status window to the right of output window, where it can be altered if necessary. The job can be started from any point within the file using the **RUN FROM HERE** button. However, we don't recommend that you use this function in the course of everyday milling, as incorrect use can damage the machine and the stock. Figure 172 shows the toolpaths described by the currently loaded file.

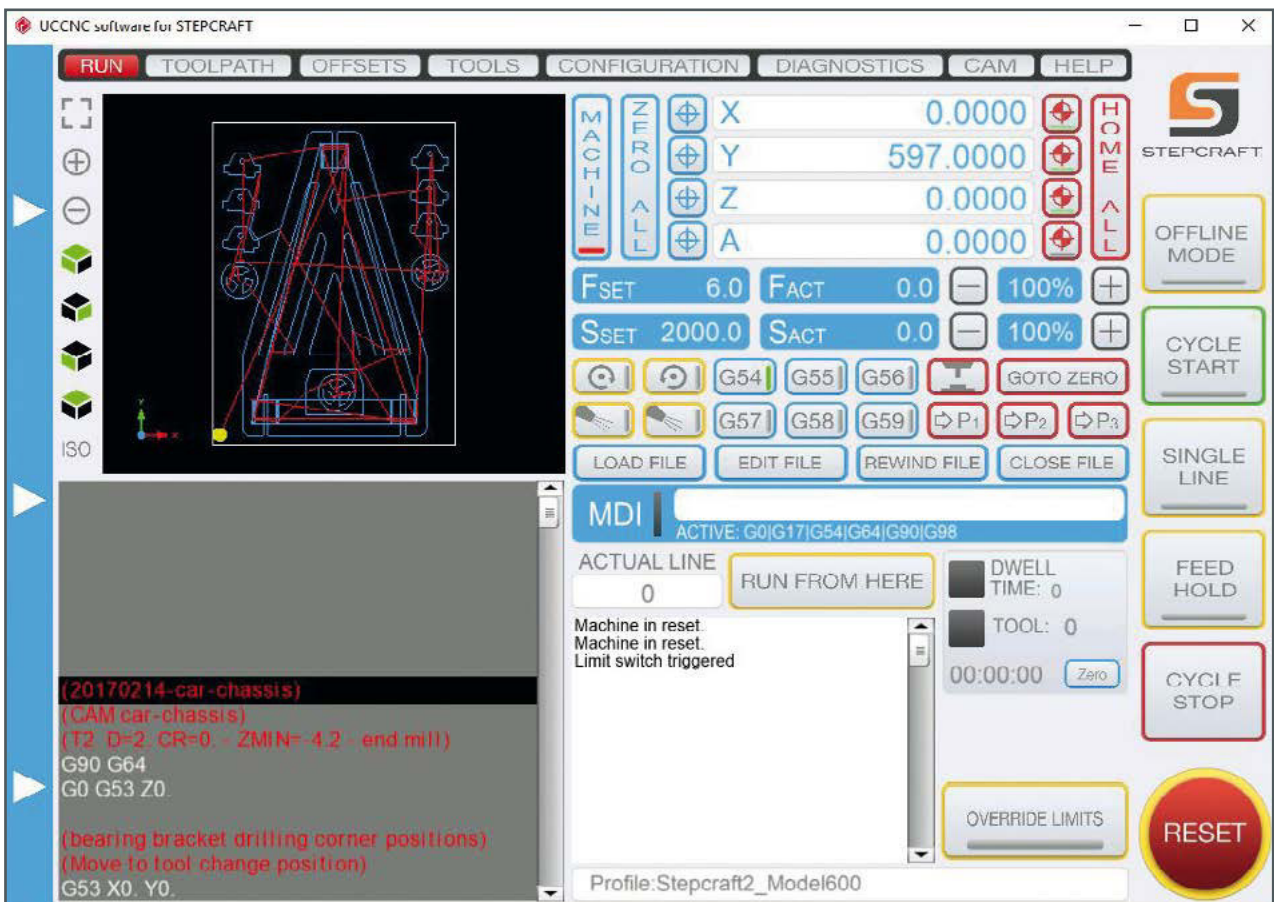


Fig. 172: The toolpath display

The display at top-left shows all the toolpaths included in the currently loaded G-code file, where you can zoom in and out to check the details of individual paths. The bar on the left of the toolpath display contains the zoom in/out buttons and buttons for switching between views. The ISO view is three-dimensional. Left-click-drag the orthogonal views to move them within the window, or right-click-drag the ISO view to rotate the model.

19 MACHINE CONTROL WITH UCCNC

During milling, the **TOOLPATH** view displays the current tool position along with the current spindle speed and feed rate values. These values can be altered proportionally for the entire job using the +/- buttons.

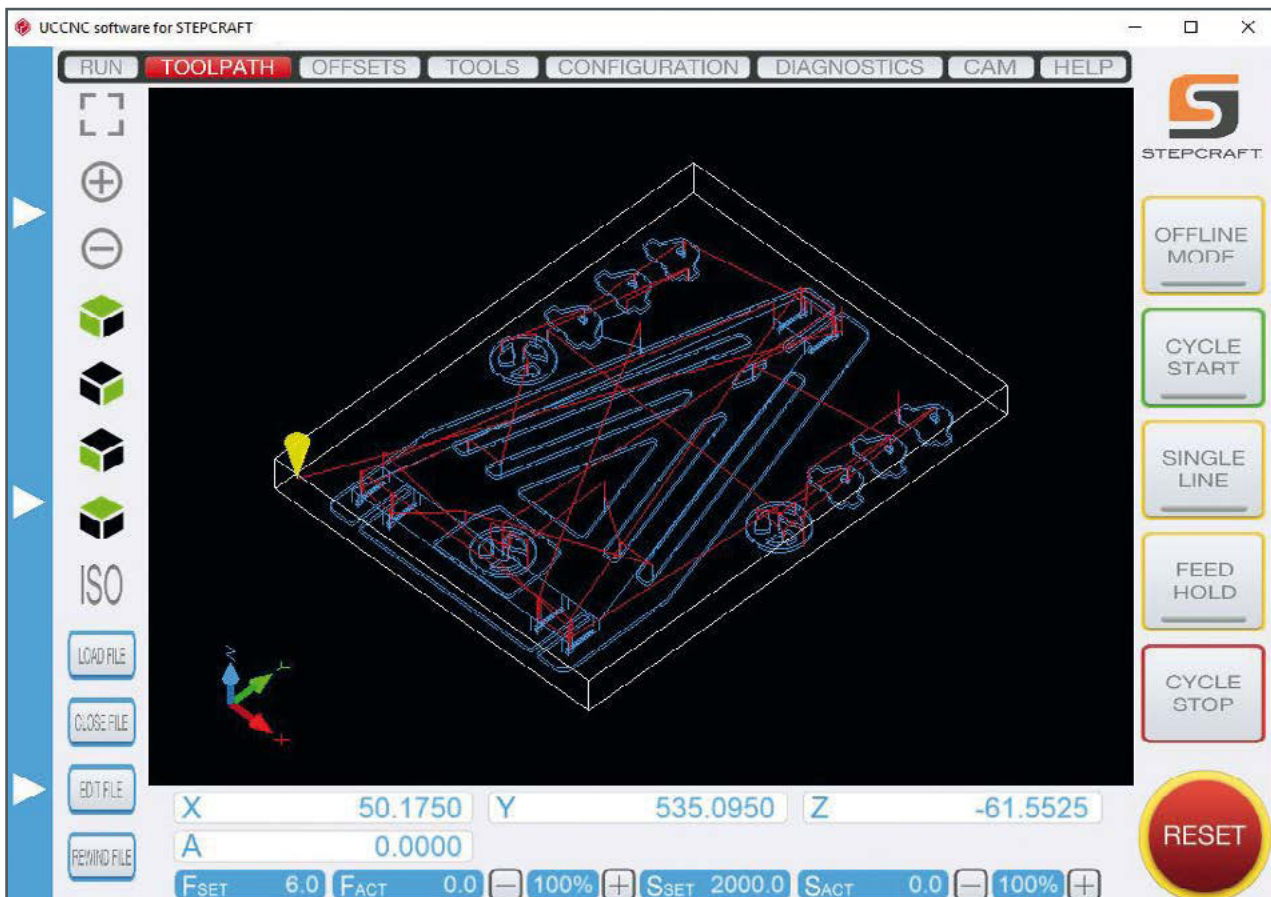


Fig. 173: The Toolpath view

19.5 SETTING THE MACHINE ZERO POINT

We now have to move the tool to the machine zero point using the **HOME ALL** button (see figure 174). This moves the tool toward the zero point of all three axes until the corresponding end switches are triggered. This ensures that the zero point in the machine's coordinate system remains constant and that all other points are defined absolutely in relation to it. This is particularly important if you are using a tool changer, as the position of the changer has to be established relative to the machine zero point to ensure accurate milling after each tool change. The machine zero point can also be used to establish the workpiece zero point.

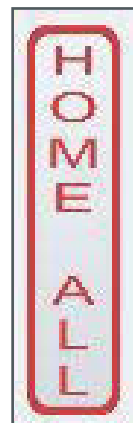


Fig. 174: The HOME ALL button