SIMmachina Tools 2025

User's manual

SIMmachina Tools 2025 Program documentation, distributed by:

HORUS Robotics GmbH

Trotzenburger Weg 3

D-51465 Bergisch Gladbach

info@horusrobotics.com

support@SIMmachina.com

The latest version of this manual can be obtained from

https://www. horusrobotics.com/en/downloads.html

Contents

Introduction

SIMmachina Tools is designed as a support tool to be used alongside CAD/CAM systems, enabling quick calculations that would otherwise require extensive manual work, books, or lengthy online searches.

Errors in workpiece setup (see parallel block selection), for example, sometimes only appear in the final machining step and can then be very difficult and/or costly to correct. With SIMmachina Tools, such errors can be identified before the first cut, allowing proactive adjustments.

No internet connection is required for this, which is important for matters like IT security, data protection, etc., and calculations can therefore also be performed on PCs installed directly at machine tool workstations.

For better readability, this manual uses terms like 'user' and 'operator,' but they are intended to include all people, regardless of gender.

‘Italicized, bold, and underlined words (e.g., *Copy*) represent hyperlinks, which, when clicked, take you directly to the corresponding chapter in this manual.

Important Usage Instructions

All functions of SIMmachina Tools have been developed according to the state of the art and extensively tested. The calculation results have been verified in practice in workshops and on machines. Great emphasis has been placed on preventing erroneous or implausible inputs from the user from the outset.

However, it is inherent to IT and computers that not all configurations with every PC hardware and software component available on the market, along with all possible versions of supported operating systems and drivers, can be verified.

For this reason, it is essential to view calculation results (essentially from all programs available on the market) with common sense and, if necessary, to question them.

Regarding SIMmachina Tools, this means that for calculation results (e.g., in the parallel block height calculation of the 'clearance') that are in the range of 1 mm or smaller, one should verify their inputs multiple times.

It is also advisable, during the first use of the values in the machine, to operate the machine at reduced speed and to ensure that you can stop the machine's movement at any time.

SIMmachina Tools has been developed for and tested extensively on Microsoft® Windows® 11 in the 64-bit version.

The program's functionality under Windows® 7 (32 and 64-bit) and Windows® 10 (32 and 64-bit) has been verified; however, there is only limited support for these operating systems.

Known issues across the different operating system versions are documented at the end of this documentation.

Structure of the Program

SIMmachina Tools is designed so that the desired calculation function is always in focus, and the user is not distracted by program components that are not needed for the task at hand.’

The desired function is selected through the main menu. Here, a distinction is made between machine functions, design functions, and general (auxiliary) calculations.

In the submenu 'Machine Functions,' you will find all the functions primarily needed directly at the machine tool or in the workshop, while the submenu 'Design Functions' contains the functions required in production planning or design.

Under the menu item 'Calculations,' you will find conversion and auxiliary functions to support the machine and design functions, as well as calculations directly at the machine tool, such as determining the center point of a circle.

For input fields that require specific units, these are indicated in square brackets following the description of the respective input field, e.g., 'Cutting Speed Vc [m/min]' in the cutting data calculator.

If the same units apply to various input fields, they are indicated in the border of the input fields, e.g. 'Inputs [l]' in the coolant calculation.

In all input fields, radio buttons, and combo boxes, tooltips facilitate usage and provide additional background information at relevant points. Tooltips appear when the mouse pointer hovers over an input or command field for a short duration (usually about 1-2 seconds).

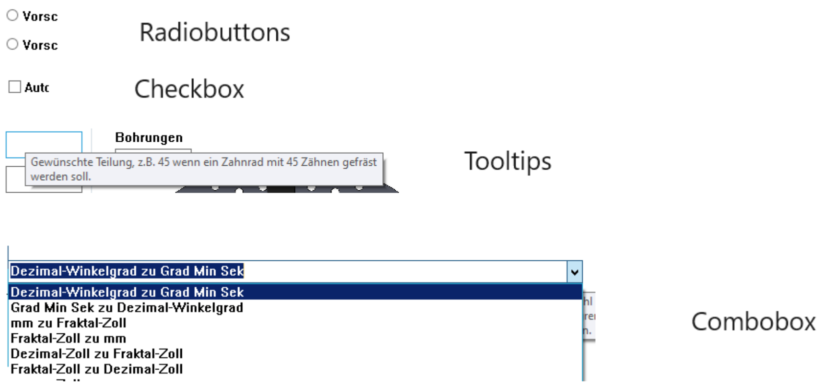

Fig 1: Explanation of the control elements.

Installation, Activation, and Setup

System Requirements

- Intel®-compatible processor with Microsoft® Windows® 7, 10, or 11 Professional or higher.
- RAM size depends on the requirements for the corresponding Windows® installation; however, 4 GB is generally recommended for smooth operation.
- SIMmachina Tools can be installed as either a 32-bit or 64-bit version. Both versions offer the same functionality to ensure compatibility with older PCs.
- There are no special requirements for the graphics card for the 2025 version.
- An internet connection/email is required for the one-time activation. However, activation can also be done through another PC, allowing the PC on which SIMmachina Tools is installed to remain permanently offline.
- Serial RS232 COM interface for data transmission to the machine.

Installation

The installation of SIMmachina Tools is described in the separate guide “SIMmachinaTools _Installation_Instructions.pdf”.

The latest version of this guide is available at:

https://www.horusrobotics.com/en/downloads.html

Please note: After installation, SIMmachina Tools must be run as an administrator during the first start. This can be done by right-clicking on the SIMmachina Tools program icon in the SIMmachina 2025 program group in the Start menu, then selecting 'More' and 'Run as administrator.'

If the user is not logged in as an administrator in Windows®, the administrator password may need to be entered.

Activation/Unlocking

The process of activating SIMmachina Tools is described in the separate guide 'SIMmachina_Tools_2025_ActivationGuide_English_Email.pdf'
The latest version of this guide can be found at

https://www. horusrobotics.com/en/downloads.html

Setting Up the Program for First Use

After installation and the first start, SIMmachina Tools is ready to use immediately. If a different language is desired, it can be ***changed*** through the options window.

However, for certain functions (e.g., serial data transmission via RS232), it is necessary to specify the relevant parameters of your PC in the options window.

Functions that require parameters from the machine tool being used, such as available spindle speeds or available hole plates for parting, can only be effectively used once your own data has been entered or updated in the machine database.

Menus and Machine Database

Main Menu

Through the main menu (Fig 2), all functions of SIMmachina Tools can be selected, which are the same for all function windows.

Individual function windows, such as the window for serial communication (Fig 3), may have additional specific menus that apply only to that function window.

All function windows that work directly with machine data (e.g., the NC counting format for the specific control of this machine) include the 'Machine Database' selection function in the main menu to choose the corresponding machine (Fig. 6), Function windows that operate without reference to specific machine data do not include the 'Machine Database' selection function for clarity.

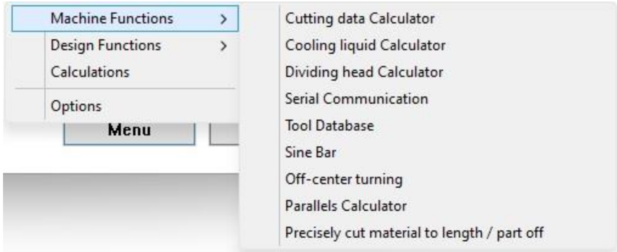

Fig 2: Main Menu

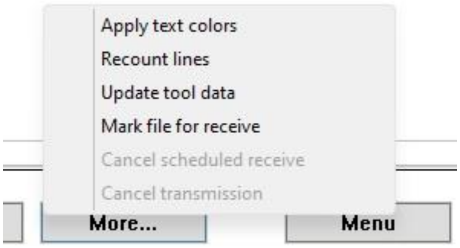

Fig 3: The specific menu in the 'Serial Communication' window.

Context Menu

The context menu (Fig 4) is opened by right-clicking in an input field.

Field-specific functions (e.g., Copy, Paste) can be accessed quickly, and after changing a value in the input field, the calculation function of the function window can be executed quickly without having to search for the 'Calculate' button.

The context menu also allows for quick deletion of the input field.

In some function windows (e.g., in the flange creation window), the context menu includes the special function 'Delete This Field and Corresponding Fields" (Fig 5).

When executed, this will delete not only the input field currently being edited but also all input fields whose values would no longer be valid after the change in the active input field or could lead to calculation problems. An example of this situation would be changing the number of holes in the flange creation window, which would no longer correspond to the old angle between the holes. If this function is selectable, it also indicates that the value in this input field is dependent on other fields/values.

Fig 4: The context menu.

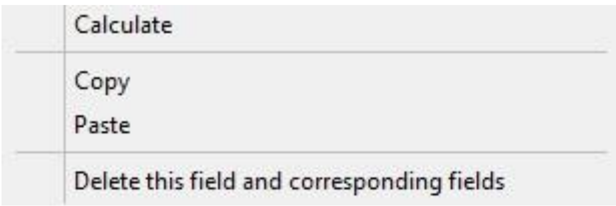

Fig 5: The context menu in the flange window.

Machine Database

The machine database is the storage location for data that is repeatedly used from machines and / or the workstation where the machine is located.

Examples include:

- The spindle speeds of machines with geometrically stepped spindle speeds.
- The selection of parallel supports available at this machine workstation.
- The jaw height of the 'favorite' machine vice used here.
- The selection of indexing plates available at this machine workstation.
- The number of indexing handle turns for one full rotation of the spindle.
- The RS232 interface settings.
- NC code data.

A special case is the coolant data. This storage area includes additional sub-storage areas, as it is assumed that each machine may have multiple coolant tanks (e.g., one for standard coolant and another for minimal quantity lubrication).

Up to 10 different coolant tanks or setups can be stored here.

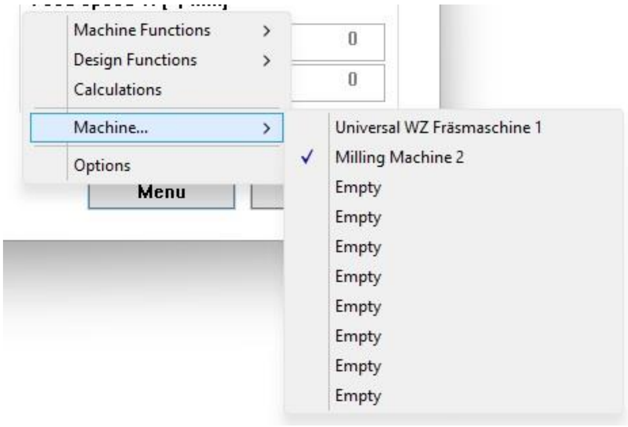

Fig. 6: The machine database menu.

Reports - Documentation of calculation results

In all main functions of SIMmachina Tools, except for the 'Calculations' and 'Serial Data Transfer' function windows, there is an option to generate reports.

The reports save the key results of each respective function as a .TXT file. The files are saved in the corresponding function subfolder within the 'ProgramData' directory.

So, for example, for the coolant calculation window, this would be ‘C:\ProgramData\SIMmachina Tools\KSM’.

The reports are automatically saved with the date and time in the file name, so there is no need to specify a file name each time or confirm overwriting. Older reports are thus retained until they are manually deleted.

In the individual function windows, the most recent report can be opened via the 'Report' button (Fig. 7) or in the interface for cutting data determination through the 'More' button followed by 'Open Report' (Fig. 8) in the system's standard text editor. If the checkbox for 'Automatically open' is checked, the report will be opened automatically after each calculation process.

Unneeded reports can be ***automatically deleted*** in the options window.

Fig. 7: Opening a report.

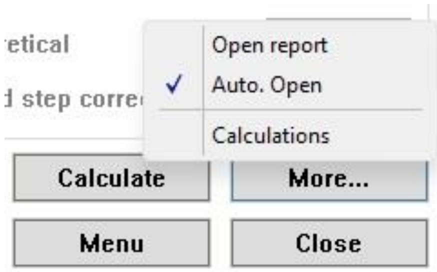

Fig. 8: Opening a report in the cutting data calculation.

Machine Functions

Cutting Data Calculation

Description

The cutting data calculation (Fig. 9) is used for the quick determination of the spindle speed and feed values, as well as the parameters 'dwell time' and 'rotational feed'.

The theoretical cutting data is determined, and for machines with geometrically stepped spindle speeds, the corresponding nearest speed level is selected, with all other determined parameters recalculated based on this speed level. The values can be calculated with adjustments for one level higher or lower.

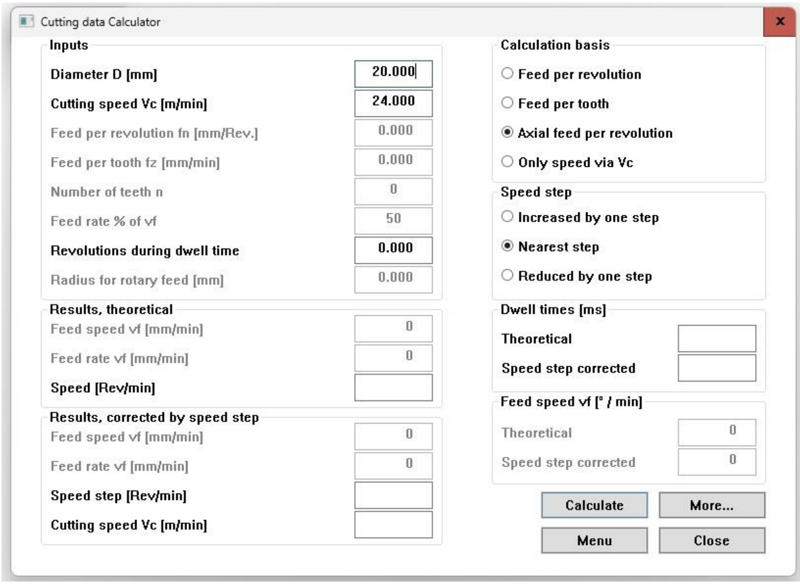

Fig. 9: The cutting data calculation.

General Calculation of Machining Parameters

Using the radio buttons 'Calculation Basis,' one first selects the fundamental data from which the machining values should be determined. (Fig. 10).

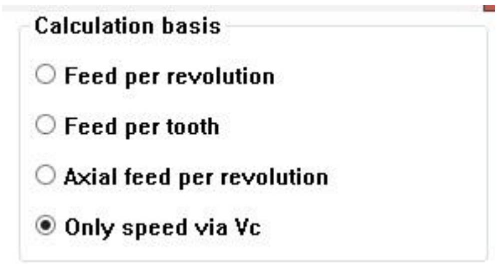

Fig. 10: Selection of the calculation basis.

The following values are required for:

Feed per revolution	Diameter of the tool, cutting speed, and feed for one revolution of the tool. Additionally, the plunge feed, the revolutions during dwell time, and the radius for the rotational feed can be specified.
Feed per tooth	Diameter of the tool, cutting speed, feed per tooth for one revolution of the tool, and number of teeth. Additionally, the plunge feed, the revolutions during dwell time, and the radius for the rotational feed can be specified.
Axial feed per revolution	Diameter of the tool, cutting speed, axial feed per revolution in the axial direction of the tool (e.g. drilling, reaming). Additionally, the revolutions during dwell time can be specified.
Speed only via Vc	Quick determination of speed based on the diameter of the tool and the cutting speed. Additionally, the revolutions during dwell time can be specified.

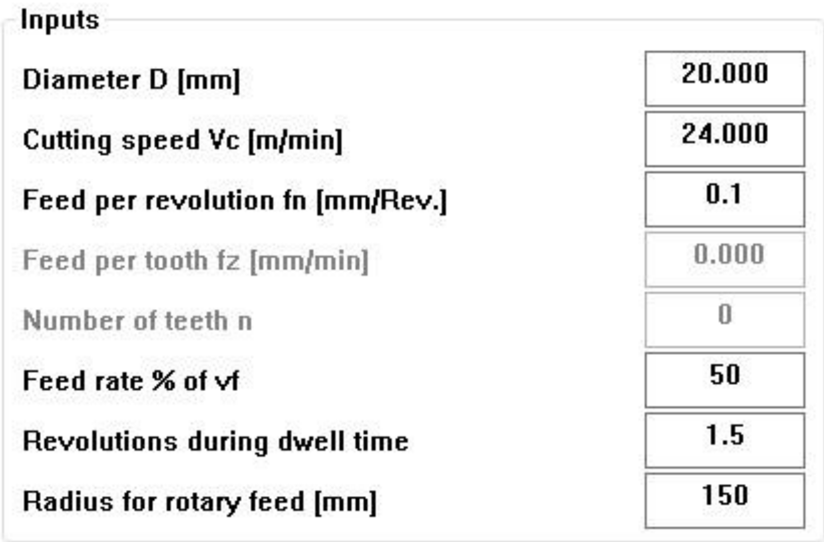

Fig. 11: Entering the key values.

After entering the data and pressing the 'Calc' button, the theoretical parameters as well as the corrected parameters of the nearest spindle speed level of the machine tool are determined. (Fig. 13).

Using the radio buttons 'Speed step' (Fig. 12), the speed level can be adjusted one level up (faster) or one level down (slower). All speed-level-corrected results are recalculated for the newly selected level.

This function requires that the ***speed levels have been entered*** in the options window..

Fig. 12: Step selection.

Fig. 13: Calculation results.

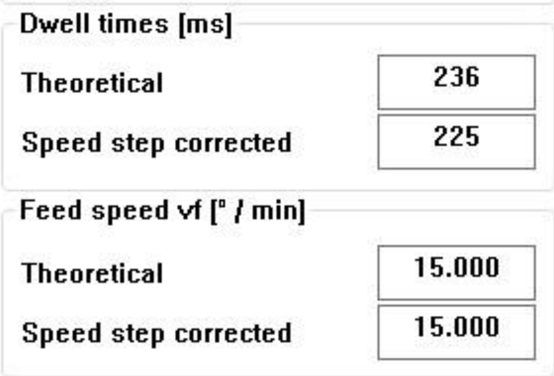

Fig. 14: Dwell time and rotational feed.

Plunge feed

By entering 'Feed rate % of vf,' you can specify the percentage value you want to use for the plunge into the workpiece based on the calculated feed speed. The plunge feed is then theoretically calculated for the entered parameters and speed-corrected for the nearest/selected speed level.

Dwell time

For blind hole operations or, for example, counterbores, it is necessary for the tool to perform a certain number of revolutions after the depth has been reached. Too few revolutions result in a poor surface finish of the bottom (leading to issues such as poor sealing at sealing surfaces), while too many revolutions lead to increased tool wear, heat generation, and longer processing times.

To determine an optimal value, there is the dwell time calculation. By entering the desired number of revolutions in the field 'Revolutions during dwell time,' the time in milliseconds [ms] is calculated that the axial feed movement must pause to achieve the desired number of revolutions. The calculated time can then be entered in the NC code at the location that defines the feed pause.

The dwell time is again theoretically calculated for the entered parameters and speed-corrected for the nearest/selected speed level.

If only the spindle speed and the desired number of revolutions during dwell time are available, it is easier to determine the dwell time in the 'Calculations' function window using the 'Dwell Time' function.

Rotational feed

When milling along a circular/spiral path, for example, on a rotary table, the feed speed vf can be converted from [mm/min] to [degrees/min].

To do this, enter the milling radius in the input field 'Radius for rotational feed'.

The rotational feed is again theoretically calculated for the entered parameters and speed-corrected for the nearest/selected speed level.

The context menu of the cutting data calculator

The context menu is opened via the 'More...' button and offers the following options:

Open report	Once the cutting data has been determined, the report can be opened here.
Auto. open	After each determination of the cutting data, the report will be opened automatically. A checkbox indicates this.
Calculations	Switch to the calculation function window to, for example, recalculate values.

Using the cutting data calculator from other functions

If you want to conveniently calculate cutting data in other function windows (e.g., in tool management), you can call up the cutting data calculator from these windows. Any values already entered there will lead to a preset mode in the cutting data calculator, and the corresponding values will already be populated in the input fields.

After the calculation is completed, you can return to the original function window via the context menu by selecting 'Back,' followed by choosing the value you want to adopt. The adopted values will then be directly inserted in the original function window at the corresponding locations..

In this context menu, the sub-menus of 'Back' mean:

Back with theoretical value	Theoretical determined data is adopted.
Back with step value	Stepped data is adopted from the currently selected speed level (nearest, increased by one level, or decreased by one level).
Back without data	No data is adopted.

Calculation of Coolant

Description

The coolant calculation is used to quickly determine the parameters for the preparation and maintenance of coolant emulsions in machine tools, coolant tanks, and containers (Figure 15).

It allows for the calculation of values for a completely new emulsion or the adjustment of existing emulsions by increasing or reducing their concentration (Figure 16).

For increasing concentration, there are options to either maintain the emulsion volume (i.e., a portion of the emulsion is replaced with coolant concentrate) or increase the overall volume.

For reducing concentration, the same two options are available. Additionally, you can choose whether to dilute with water or with a 1% emulsion.

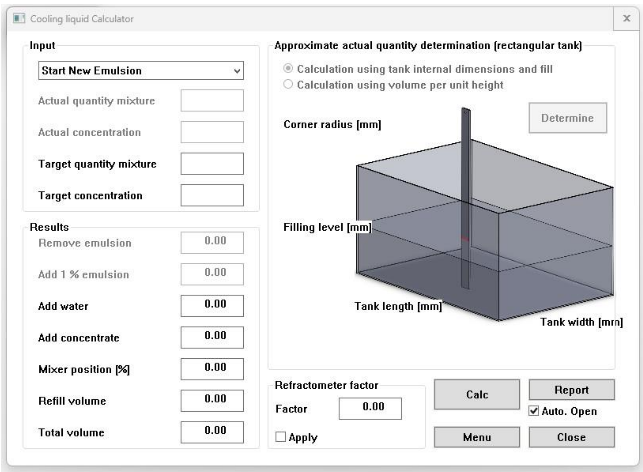

Fig. 15: The coolant calculation.

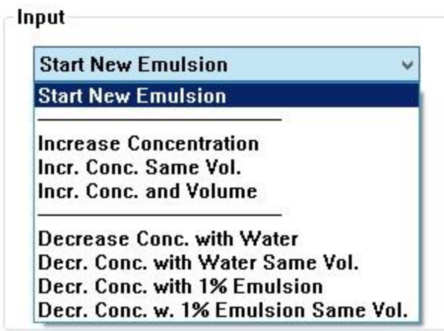

Fig. 16: Coolant calculation options.

How do you use this function?

In the input combo box, select the type of coolant calculation to be performed (Figure 16). Enter all required data in the ‘Input’ section, then click ‘Calculate.’ Input fields that are not needed will be disabled, preventing data entry.

The unit for lengths is indicated in the ‘Approximate Current Volume Determination’ section, appended to the respective labels, e.g., Fill Height [mm]. It does not matter which units are used for volume, as long as the same unit is consistently applied to all inputs.

For concentration inputs, you can either work with standard concentrations (Figure 17) or directly include the refractometer factor for a specific coolant concentrate (Figure 18). This helps avoid errors by working directly with the values corresponding to the refractometer reading, eliminating the need for manual conversions.

The refractometer factor can either be entered manually (Figure 19) or retrieved from a stored value in the machine database. To use the refractometer factor, check the box labeled ‘Apply.’

By toggling this checkbox on and off, you also have the option to directly convert concentration values, for example, for documentation purposes.

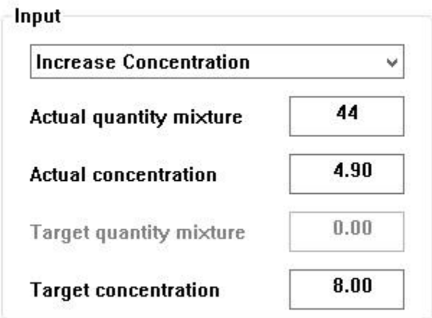

Fig. 17: Inputs in standard mode.

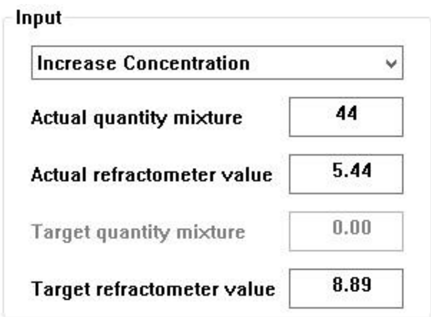

Fig. 18: Inputs with refractometer factor.

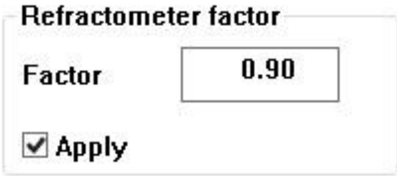

Fig. 19: Refractometer factor.

In all coolant calculations, except for the 'Prepare New Emulsion' calculation type, it is possible to perform an approximate current volume determination of a tank. A special feature of this function is that it assumes a rectangular, regularly shaped tank (Figure 20).

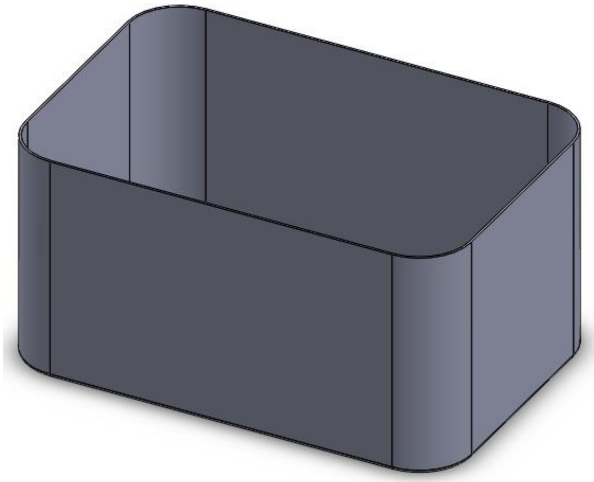

Fig. 20: Tank shape.

A uniform corner radius for all four sides can be included in the calculation. However, radii or chamfers at the tank bottom are not considered. Additionally, elements such as (baffle) plates, pump mounts, or pump intake nozzles that may be present in the tank are not factored into the calculation. This is why the method is referred to as an approximate current volume determination.

There are two calculation methods available:

Method 1: Using the tank's internal dimensions and the fill height.

Proceed as follows:

- Enter the internal dimensions of the tank and the corner radius into the corresponding input fields (***or import them from the machinedatabase***).
- Measure the current fill level (e.g., using a folding ruler) and enter it into the 'Fill Level' input field.

- Click ‘Determine’ The result will appear in the ‘Actual Mixture Quantity’ input field.

Method 2: Using the volume per height unit and the number of height units.

This calculation method requires prior preparation: for example, pour 10 liters of KSM emulsion into the empty tank and measure the current fill level (e.g., using a folding ruler), which might read 4.5 mm.

The 4.5 mm then represent 1 height unit (HU), and the 10 liters in this example determine the volume per height unit.

Please note: This method assumes that the volume per height unit remains consistent throughout the height of the tank. Variations in the volume per height unit, for example, due to the pump intake pipe, are not considered in this approach.

To determine the volume of a partially filled tank, proceed as follows:

- Enter the volume per height unit and the height unit into the corresponding input fields (***or import them from the machinedatabase*****).**
- Measure the current fill level (e.g., using a folding ruler) and enter it into the ‘Fill Level’ input field.
- Click ‘Determine’ The result will appear in the ‘Actual Mixture Quantity’ input field.

Serial Communication

Description

The Serial Communication Function Window (Fig. 21:) allows NC files to be edited and sent to or received from machine tools via the RS232 serial interface. For this, the PC running SIMmachina Tools must have a fully functional serial port. Support for USB<->RS232 and LAN<->RS232 adapters is planned but not yet implemented.

The correct settings for the RS232 interface must be entered in the options window.

Im Optionsfenster müssen die richtigen ***Daten für die RS232 Schnittstelle eingegeben*** worden sein.

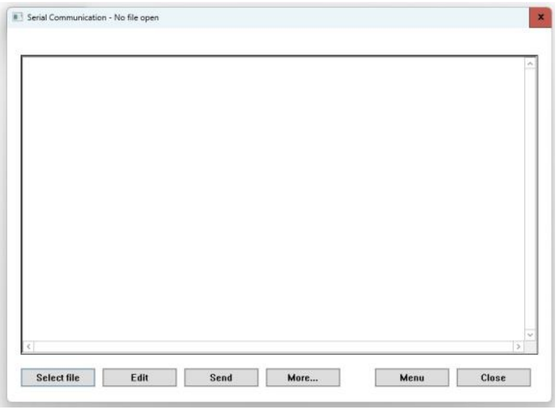

Fig. 21: The serial communication function window.

How to Use This Function

Opening an NC File

- Select the file using the **'Choose File'** button in the **'Open'** window (you may need to adjust the file extension setting at the bottom right) and confirm with **'Open'**, or drag and drop the file into the serial communication function window from Explorer by selecting it with the mouse, holding down the left mouse button, and then releasing it inside the window.
- The file is now open, and the file path is displayed at the top next to the window name.
- Any NC text file in ASCII format can be opened, regardless of the file extension.
- The opened file is color-coded: certain keywords are highlighted in different colors, e.g., all **'G00'** commands appear in red, while **'M08'** and **'M09'** are shown in blue. Commented-out lines and multi-line comment sections are also color-highlighted when opening the file (or when manually applying color formatting).
- If the file contains more than **10,000 lines**, the program will prompt whether to apply color formatting, as this may take longer depending on the file size.

Editing an NC File

- In the **Options** window, select where the NC file should be edited.
- Click inside the NC text with the mouse to start editing, or click the **‘Edit’** button. If an external editor is selected, it will open, allowing you to modify the file (**after editing, save the file and close the editor; the file will be automatically reloaded in SIMmachina Tools**). If no external editor is used, you can edit the NC text directly. To save, click the **‘Save’** button, which becomes active as soon as changes are made.
- After editing, pressing **‘Enter’**, **‘Arrow Up’**, or **‘Arrow Down’** will automatically apply color formatting to the edited line. However, if a comment spans multiple lines, color formatting is only applied when manually triggered via the menu.

Manually Applying Text Color Formatting

- In the **‘More’** menu, click **‘Apply Text Colors’**. The text will be color-formatted, and a progress bar window will appear until the formatting process is complete.
- To ***customize text colors***, open the **options menu** for the currently selected machine.

Renumbering an NC File

- In the **‘More’** menu, click **‘Renumber Lines’** to open the **‘Renumber Lines’** window.
- Position the window so that the NC code is clearly visible.
- Verify the **line numbering format**: check whether the NC lines in the NC code match the format shown in the **‘Current Format’** input field (e.g., **N0000**). The preferred numbering format is retrieved from the machine database for the currently selected machine.
- If you want to keep this format, ensure that the **‘New Format’** input field contains the same format as the **‘Current Format’** field.
- In the **‘Start Value’** input field, specify the number for the first line. For example, if using **N0000** as the format, entering **5** would result in the first line being **N0005**.
- In the **‘Line Increment’** field, define the numbering step. For example, with format **N0000**:

  - A step of **1** results in **N0000, N0001, N0002, ...**
  - A step of **3** results in **N0000, N0003, N0006, ...**
  - A step of **5** results in **N0000, N0005, N0010, ...**
  - The maximum increment allowed is **20**.

- Click **‘Renumber’** to apply the new numbering. For large files, the process may take some time. A **progress bar** indicates the ongoing process. Editing and other operations are disabled during renumbering.
- To **cancel** renumbering, click **‘Cancel’**—the NC file will remain unchanged.
- Close the window with **‘Close’** when done.

Updating Tool Data in the NC File

NC files that store **tool offsets** directly in the file can be updated using the **tool database**. This ensures that the latest tool offsets are always sent to the machine.

Currently, the following tool offset formats are supported:

D01 +119907	**Manufacturer 1:** Dialog format with the letter **D**, followed by the **tool number** and the **offset value**.
T2 R A L4.5 A (Comment)	**Manufacturer 1:** Dialog format with the letter **T**, followed by the **tool number**, **offset values**, and a **comment**.
TOOL DEF 1 L10 R5	**Manufacturer 2:** String format with **'TOOL DEF'**, followed by the **tool number** and **offset values**.

Updating Tool Offsets in the NC File:

To update the tool offsets, follow these steps:

- Ensure that the ***correct machine is loaded from the machine database*,** that the **latest tool data** is available in the database, and that the **tool format** in the NC file matches the tool database format for the selected machine.
- In the **'More'** menu, click **'Update Tool Data'**. The tool data will be updated, but **no validity check** is performed on the data.
- Before sending the NC file to the machine, **manually verify** the updated tool data for accuracy in the **edit area** or **editor window**.

Sending an NC File to a Machine

- Connect the machine and the PC running SIMmachina Tools using a **serial cable**, according to the specifications of the machine/control manufacturer.
- In the **Options** window, verify that the ***correct settings for the RS232 interface*** are configured.
- The **Event field** should display **'Connected'** (see Figure 22).
- Set the machine to **receive mode**.
- In SIMmachina Tools, click the **'Send'** button.
- The **Event field** will show **'Sending'** while the data is being transmitted.
- Once all data is transmitted, a dialog box will appear, informing you of how many bytes were transferred.
- After the transfer is complete, verify on the machine that the data was correctly received.

Fig. 22: Active Serial Connection

Receiving an NC File from a Machine:

- Connect the machine and the PC running SIMmachina Tools using a **serial cable**, according to the specifications of the machine/control manufacturer.
- In the **Options** window, verify the correct settings for the **RS232 interface**.
- The **Event field** should display **'Connected'** (see Figure 22).
- In the **'More'** menu, click **'Prepare to Receive File'**. SIMmachina Tools is now ready to receive and is waiting for the data. This will be indicated in the **Event field** as **'Receive Prepared'**.
- Switch the machine to 'Send.' The data will now be transmitted. The event field will display 'Receiving,' and the center of the event field will show the number of bytes already transferred.
- Once all data has been transmitted, a dialog box will appear, indicating the total number of bytes transferred.
- If this dialog box does not appear immediately or only after a long delay, the receive timeout in the options is set too high. If the displayed byte count stops increasing, you can cancel data reception by pressing the 'ESC' key.
- A dialog box will prompt you to save the data. This should be done if further processing of the data is required.
- The received data will be displayed in the NC editor.
- To cancel the reception process or the 'Receiving Pending' status, open the 'More' menu and select 'Stop Waiting for Reception.'

Notes on This Function Window

Depending on the machine/control type, the **Event field** may either **not function** at all or may only work **partially**.

Sinebar

Description

The Sinebar Function Window (see Figure 23) allows the calculation of two values:

- The height of the shim for a given angle, or
- The angle for a given shim height.

Only positive values can be entered, and the maximum angle is 90 degrees.

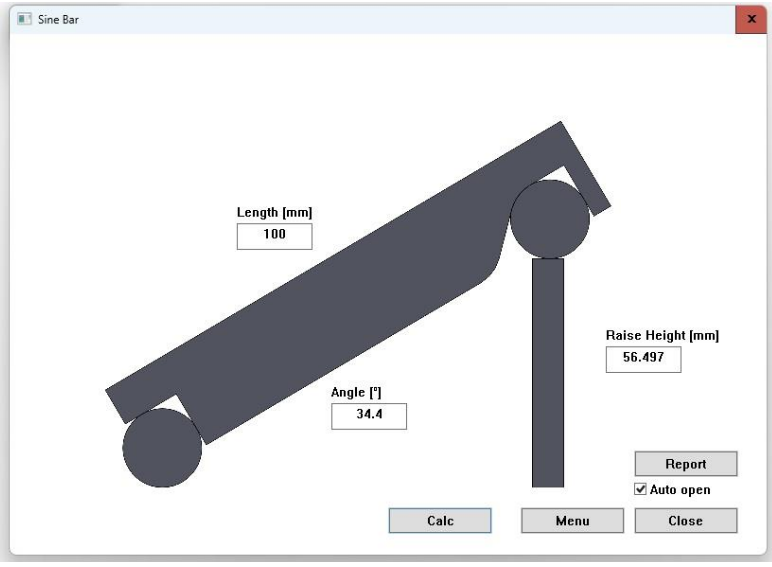

Fig. 23: The Sinebar

How to Use This Function

To perform the calculation, follow these steps:

- Enter the **length of the sinebar.**
- Depending on which value you want to calculate, enter the corresponding counterpart. For example, if you want to calculate the **raise height**, enter the **angle** and leave the **raise height input field** blank. To calculate the **angle**, enter the **raise height** and leave the **angle input field** blank.
- Click the **'Calc'** button.
- If you want to enter different angles (or different heights) consecutively, just enter the new values.
- If you wish to change the type of calculation (e.g., in the first step you calculated the shim height, and now you want to determine the angle for a specific shim height), it is necessary to **clear the input field** (or set its value to 0) for the value you want to calculate.

Dividing Head Calculations

Description

With the indexing function window (Figure 24), indirect divisions can be calculated on an indexing head/dividing attachment or rotary table. Precise angular divisions can also be determined.

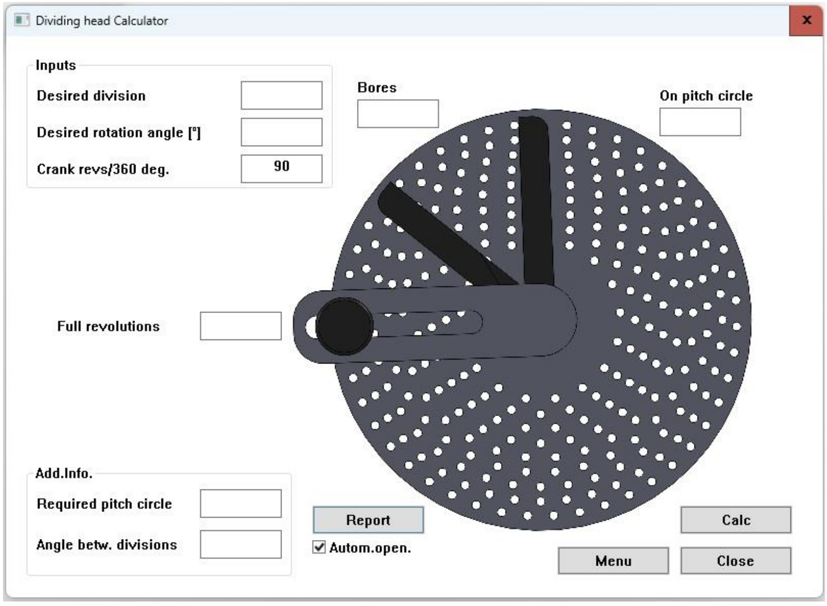

Fig. 24: Dividing head function window

The available indexing plates of the dividing attachment are taken from the machine database (this list ***can be modified via the options menu***). The number of crank rotations required for a full revolution of the indexing spindle is also retrieved from the machine database. However, this value can be adjusted by entering a new value in the input field 'Crank Rotations/360 Degrees.

How to Use This Function

1. Select the machine from the machine database whose stored indexing plates will be used.
2. Enter the desired division in the input field 'Desired Division' to calculate based on divisions, OR enter the angle in the input field 'Desired Rotation Angle' to calculate based on angles.

   Note: If calculating with divisions, the 'Desired Rotation Angle' field must be empty or set to 0, and vice versa.

3. Enter the number of crank rotations required for a full revolution of the indexing spindle in the input field 'Crank Rotations/360 Degrees.'
4. Press the 'Calc' button.

Calculation Results

The output fields display the following results:

- 'Full Rotations'**:** The number of complete crank rotations required per indexing step.
- 'Holes': The number of additional holes that need to be counted after the full rotations, based on the selected hole circle.
- 'On Pitch Circle': Indicates which hole circle is used.

If the required hole circle is available, a checkmark appears in the output field 'Required Pitch Circle' under 'Additional Info'**.** If the hole circle is not available, its number appears in the output field instead. In this case, the output fields for 'Full Rotations,' 'Holes,' and 'On Pitch Circle' will be marked with an X.

The angle between divisions (in degrees) is displayed in the output field **'**Angle Between Divisions**'.** This value can be used, for example, for indexing on a CNC dividing head or an indexing head/rotary table with a digital degree display.

Notes on This Function

When calculating using the **'**Desired Rotation Angle' field, only specific decimal places can be computed:

- With 40 crank rotations per full spindle revolution**,** only increments of 0.5 can be calculated.
- With 90 crank rotations per full spindle revolution, increments of 0.25, 0.5, and 0.75 are possible.
- For other crank rotation values, these increments must be determined experimentally.

Off-center turning

Description

In a three-jaw chuck, it is possible to clamp the workpiece in such a way that it can be machined eccentrically. To achieve this, a defined spacer must be placed under one of the jaws. The thickness of the spacer and the size of the eccentricity are not the same.

This method allows for radial eccentric turning, and it also makes it possible to drill axially off-center into the workpiece, for example, using the tailstock.

The ‘Off-Center Turning’ function window has two main modes, which can be selected using radio buttons:

Creatable Diameter as Input	Input for the workpiece diameter and the target diameter of the eccentric feature to be created. The maximum possible eccentric offset and the required spacer thickness are calculated.
Creatable Diameter for Information Only	Input of the workpiece diameter and the desired offset. This provides the thickness of the spacer to be used, as well as the maximum eccentric diameter that can be created. OR Input of the workpiece diameter and the thickness of the spacer. This provides the offset and the maximum eccentric diameter that can be created.

Only positive values can be entered. Mathematically, the offset must be smaller than the radius of the workpiece, i.e., offset < (diameter / 2.0). In practice, however, only much smaller offset values in the range of 5% to 20% of the diameter are practical.

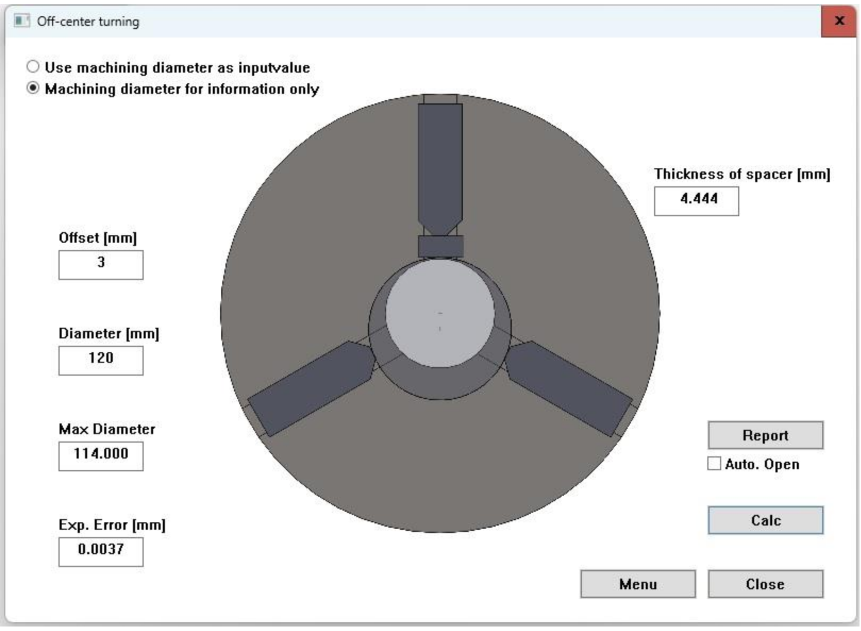

Fig. 25: Off-Center Turning

How do you use this function?

For the calculation of the offset or spacer thickness, proceed as follows:

- Enter the diameter of the workpiece.
- Depending on which value you want to calculate, enter the corresponding value. For example, if you want to calculate the spacer thickness, enter the offset value and leave the ‘Spacer Thickness’ input field blank. For the offset calculation, enter the spacer thickness and leave the offset field blank.
- Press the ‘Cal’ button.
- The estimated error of the eccentricity relative to the calculated spacer thickness can be read in the output field ‘Exp.Error’ The formula used has an error of approximately 1/800 or 0.00125.
- If you want to enter different spacer thicknesses (or different offsets) sequentially, simply enter the new values. However, if you want to change the calculation method (for instance, if you first calculated the spacer thickness and now wish to calculate the offset for a specific spacer thickness), you must clear (or set to 0) the input field for the value you want to calculate.

For the calculation of the maximum offset for a given eccentricity or spacer thickness, proceed as follows:

- Enter the diameter of the workpiece.
- Depending on which value you have (given eccentricity or spacer thickness), enter the corresponding value and leave the opposite field empty (or set it to 0).

- Press the ‘Calc’ button.

Notes on this feature window

Due to the offset caused by the thickness of the spacer, the two opposite jaws are used more at their edges. Using thin spacer plates between the jaws and the workpiece is a way to reduce marks and inaccuracies on the finished workpiece. This is particularly useful when using hard jaws and soft materials like aluminum. The thickness of the spacer plates must be added to the calculation as either 1x the workpiece radius or 2x the workpiece diameter.

At high speeds and/or larger eccentricities, it is advisable to consider the use of counterweights.

Please observe the machine's safety precautions:

- The spacer plates and the calculated spacer are only mechanically connected to the jaws and the workpiece and can be ejected from the clamping setup under unfavorable conditions. As a result, the workpiece could also be released from the clamping setup.
- Avoid high cutting values and potentially assess separately, as the clamping setup with additional components is not as stable as with the jaws alone.
- Consider using a center point.

Determine parallel block heights

Description

When clamping workpieces in a machine vise with smooth jaws (i.e. without a vertical positioning edge), parallel blocks or parallels are required to ensure the workpiece is aligned parallel to the milling table.

The goal is to clamp the workpiece as low as possible to maximize jaw coverage and minimize overhang above the vise jaws. The height of the parallel blocks depends on several factors:

- Vise jaw height
- Workpiece height
- Depth of side machining (from a top view) when the workpiece is approached from the outside
- Vertical tool overhang (from a top view), e.g., when using a chamfer or radius cutter

In prototype and one-off manufacturing, it is easy to select parallel blocks that allow for correct side milling (Figure 30) but later realize that the chamfer cutter would collide with the vise jaws during chamfering (Figure 31).

The result is a time-consuming re-clamping process using higher parallel blocks, which can lead to inferior results due to an additional setup. If the workpiece zero point was in an area that was milled away in previous steps, this could even result in total scrap.

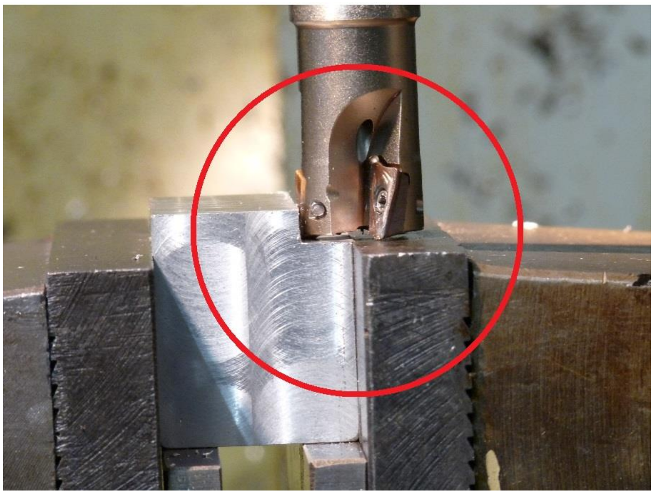

Fig. 26: Lateral machining possible

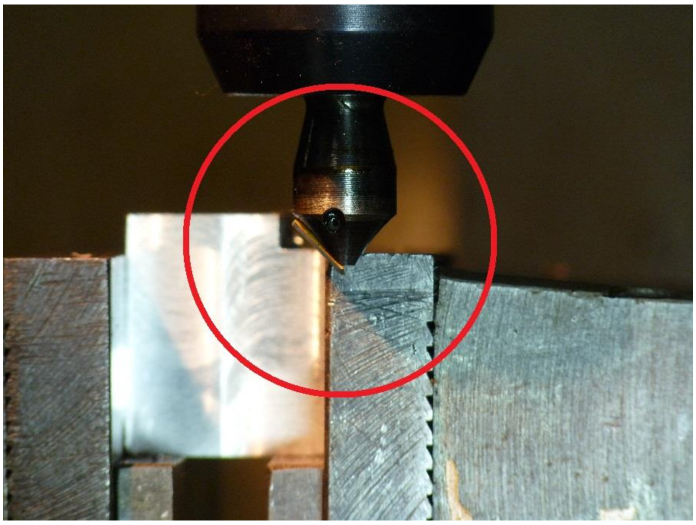

Fig. 27: Chamfering not possible due to excessive clamping depth.

The general use of excessively high parallel blocks results in a less stable clamping setup, reducing the rigidity of the vise-to-workpiece system. This increases the risk of the workpiece vibrating under unfavorable cutting parameters or, in the worst case, being 'pulled out' of the vise due to cutter engagement.

Potential consequences include tool breakage, total workpiece scrap, spindle damage, or even personal injury.

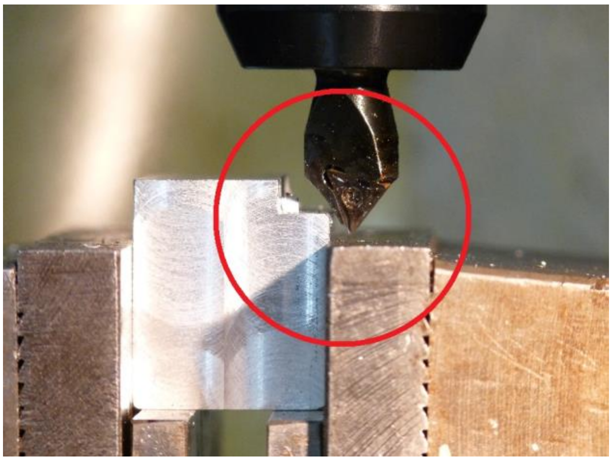

Fig. 28: Workpiece correctly clamped, chamfering possible.

While the images here represent an extreme example to illustrate the issue, even a missing clearance of just 0.1 mm to the hardened vise jaw can cause damage to both the jaw and the cutter or insert.

How do you use this function?

In the 'Parallels Calculator' function window, enter the workpiece height and lateral machining depth.

Side machining in SIMmachina Tools is defined as a lateral milling operation that passes through the outer contour of the workpiece. This milling can be performed either from the top or the side. The side machining depth is measured from the top surface of the workpiece.

To ensure the correct input for side machining depth, use the deepest level from a top-down perspective (if multiple lateral milling operations exist). Refer to Figure 33, where C and A represent different side machining depths, with A being the relevant depth.

If a tool is used during machining that extends deeper in the vertical direction than the milling depth of the side machining operation—such as a chamfer cutter—this overhang must be entered in the 'Tool Overhang Depth' input field.

The tool overhang depth is the distance from the milling base to the lowest point of the tool (see Figure 33 B). The critical factor is the overhang depth of the cutter used at the deepest side milling operation, regardless of the cutter type.

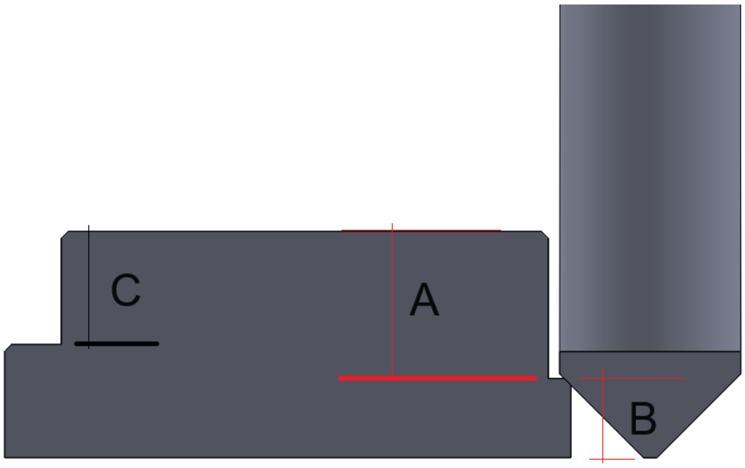

Fig. 29: Side machining depth and overhang depth

Additional required inputs include the depth (or width, depending on perspective) of the workpiece and the height of the vise jaws.

The jaw height is initially retrieved from the machine database using the stored values for the selected machine but can also be adjusted manually in the input field. When switching to a different machine, the jaw height value stored for that machine is automatically applied.

The combo box contains the available parallels for the selected machine (this list can be modified via the options menu). Selecting a parallel updates the graphical representation of the clamping setup. This update also occurs when pressing the 'Calc' button, for example, after changing individual values in the input fields.

The 'Recommend' button selects the optimal parallel height that is a) available and b) best matches the entered parameters to achieve the deepest possible clamping within the vise jaws.

The recommended parallel height and the absolute minimum required height (including the 'safety margin for calculations' from 'General Options') are displayed in the corresponding output fields. Additionally, the key values for the height of the workpiece above the jaws (i.e., the free height in space), below the jaws (i.e., the portion of the workpiece gripped by the jaws), the percentage of the workpiece held by the jaws, and the clearance between the tool's overhang depth and the top of the jaws are also shown.

If the required parallel height cannot be achieved with a single parallel, **SIMmachina Tools** attempts to reach the required height by combining two of the available parallels from the list, ensuring that the workpiece can still be clamped properly.

Notes on this feature window

The graphical output provides a general, qualitative representation of the clamping setup. Absolute measurements are not displayed here, as this would not be feasible in certain areas due to pixel size limitations. Only the numerical values in the output fields and reports are authoritative—**not** any measurements taken from the graphical output, such as with a ruler.

Additional notes regarding the clamping setup—for example, if the depth of the workpiece is smaller than the depth of the two parallels (see Figure 34)—are displayed as text within the graphical area.

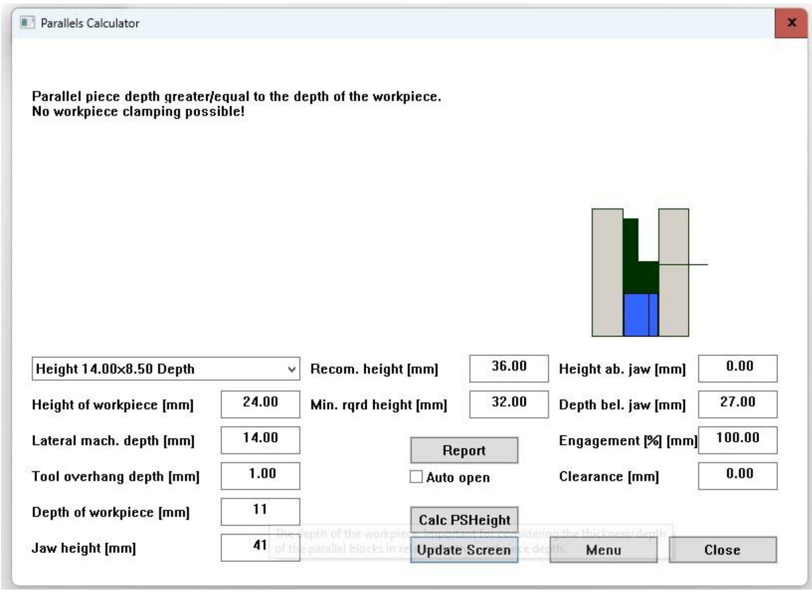

Fig. 30: Error messages

Tool Database

Description

The tool database function window is used for storing and managing tool data and for quickly retrieving cutting parameters.

In the ‘Serial Communication’ function window, tool offsets from an NC file can be updated in the tool database before being sent to the machine.

The tool database can store up to 499 tools per machine.

How do you use this function?

The tool database (Figure 35) can be accessed via ‘Menu → Machine Functions → Tool Database’.

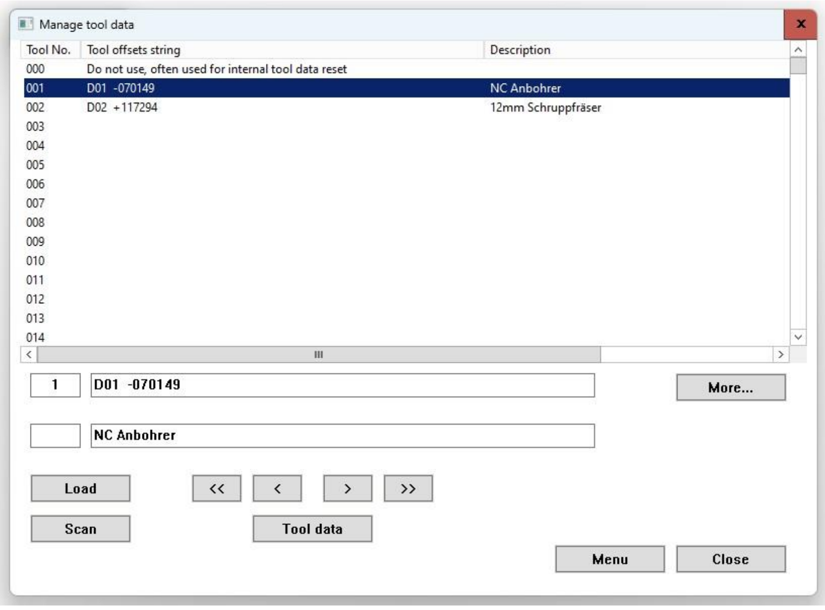

Fig. 31: The tool database

The table consists of 3 columns:

Column 1: ‘Tool No.’	The tool number in the table, ranging from 000 to 499.
Column 2: ‘Tool Offset String’	The tool offset text that is inserted into an NC file for the corresponding tool.
Column 3: ‘Description’	A custom description of the tool to make it easier to find in the table.

Individual tools can be selected as follows:

- Double-click the desired tool number in Column 1: ‘Tool No.’
- Enter the desired tool number directly into the input field below the table (left side), then press the ‘Load’ button.
- Using a barcode scanner: Press the ‘Scan’ button and scan the barcode on the tool holder. Once successfully scanned, the tool will be loaded.
- Navigation buttons (‘<<‘, ‘<‘, ‘>‘, ‘>>‘) allow browsing through the tool database. The selected tool is automatically loaded when a button is pressed.
- The ‘<<“ and ‘>>‘ buttons jump to the beginning or the last entry in the tool list, respectively.

Once a tool is selected, the ‘Tool Offset String’ and ‘Description’ fields are displayed in the input fields below the table for better readability.

To get more details about a tool, press the ‘Tool Data’ button to open the tool data window.

Pressing the ‘More’ button provides various options for editing table entries.

Copy Entry	Marks an entry for copying after it has been selected. The number of the marked entry is displayed in the menu after the word ‘copy’**.**
Cut Entry to move	Marks an entry for cutting/moving after it has been selected. The number of the marked entry is displayed in the menu after the word ‘Move’**.**
Insert Entry	Inserts a previously marked entry that was cut, moved, or copied. Before pasting, a new position in the table must be selected.

	If the entry was cut/moved, it is removed from the original position and inserted at the new location. If the entry was copied, only a duplicate is inserted at the new position. Copied entries contain **'!!!'** in the 'Tool Offset String' as an indicator that they still need to be edited (e.g., because the tool numbers in the string are identical).
Insert Range from to, Start Position	Copies a previously selected entry into a range of multiple entries. The start of the range is selected first, and its position is then displayed after the word 'Start Position'. Copied entries contain **'!!!'** in the 'Tool Offset String' as an indicator that they still need to be edited (e.g., because the tool numbers in the string are identical).
Insert Range from to, End Position	Copies a previously selected entry into a range of multiple entries. Here, the end of the range is selected. The end position must be greater than the start position (e.g., Start: 008, End: 010 and not Start: 010, End: 008).
Delete Entry	Permanently deletes the selected entry.
Delete Range from to, Start Position	Deletes a range of multiple entries. The **start** of the range is selected first, and its position is then displayed after the word 'Start Position'.
Delete Range from to, End Position	Deletes a range of multiple entries. The end of the range is selected, and all entries within this range are permanently deleted.
Cancel All Range Operations	Cancels all range operations and clears any selections.

Import	Imports the entire tool database from a file in SMWZ format. **Warning:** This will overwrite the current database. It is recommended to import any saved tool images as well.
Export	Exports the entire tool database to a file in SMWZ format. It is recommended to export any saved tool images as well.

The Tool Data Window

In the tool data window (**Figure 36**), additional information, such as cutting data, for the selected tool can be found and edited.

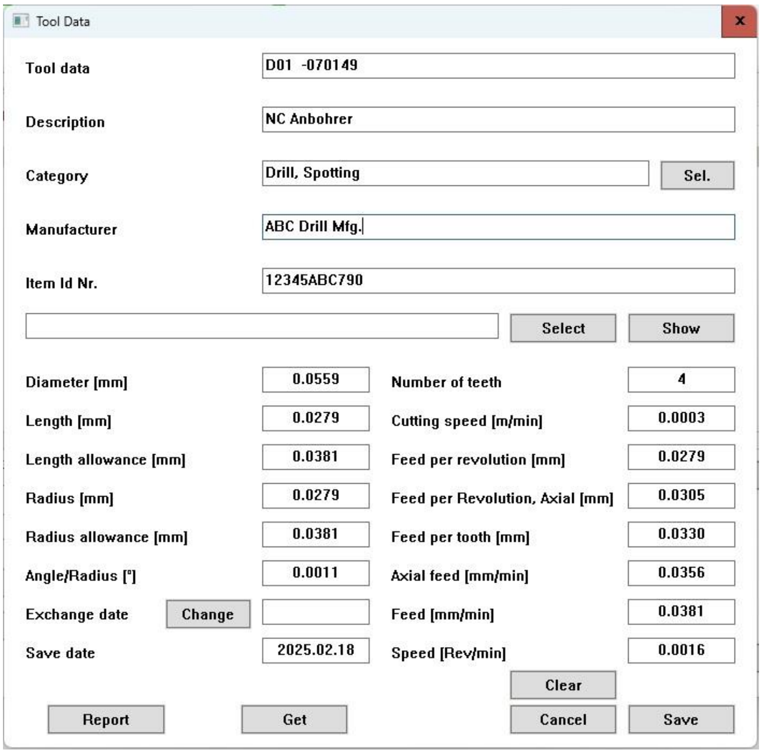

Fig. 32: Tool Data of the Selected Tool

The input options are organized as follows:

NC Data	The 'Tool Offset String', which contains the offset data for the tool. It must be manually created and verified for the used NC control system.
Description	Additional information can be entered here to help quickly locate the tool in the table.
Category	A custom tool category can be entered here, or a suggestion can be generated using the category finder (button 'Sel.').
Button 'Sel.'	Category Finder. After pressing the button, a tool category can be created by selecting an option. This category can then be manually edited and adjusted to personal requirements.
Manufacturer	Enter the name of the tool's manufacturer.
Item Id. No.	Enter the order or part number of the tool. Letters, numbers, and special characters are allowed.
Button 'Select'	Select a .jpg image of the tool. The image can then be added to the machine database directory (recommended, Figure 37). The path to the selected image will be displayed in the input field to the left of the button.
Button 'Show'	Expands the window to the right and displays the selected tool image. Pressing the button again will shrink the window and hide the image.

Button ‘Rotate’ in the expanded window	Rotates the image in 90-degree increments clockwise.

Fig. 33: Copy Image to Database

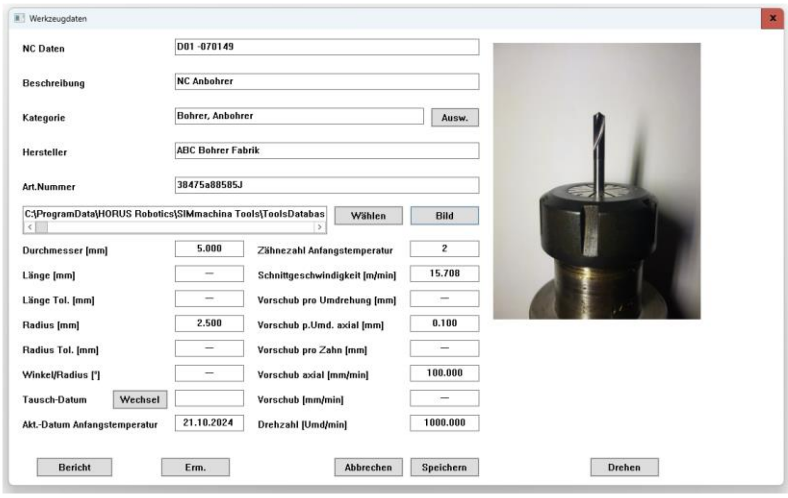

Fig. 34: Tool Picture

The lower half of the window contains the tool-specific input options:

Diameter	The diameter of the tool.
Length	The length of the tool. Informational only in this version of the program.

Length allowance	The length tolerance of the tool. Informational only in this version of the program.
Radius	The radius of the tool. It is automatically calculated after entering the diameter when saving.
Radius allowance.	The radius tolerance of the tool. Informational only in this version of the program.
Angle / Radius	For tools defined by a radius (e.g., radius cutter, quarter-circle cutter) or an angle (e.g., chamfer cutter), the value can be entered here. The angle can also be stored here for tools like spiral drills. Informational only in this version of the program.
Exchange date	By pressing the 'Change' button, the current date (as set on the PC) is inserted into the input field. This allows for easy logging of a tool change in the tool holder or the replacement of an indexable insert.
Button 'Change'	Inserts the current date (as set on the PC) into the 'Exchange Date' input field.
Save date	When saving, the current date (as set on the PC) is automatically inserted and saved here.
Number of teeth	Number of teeth / number of cutting edges of the tool.
Cutting speed	Cutting speed to be used for the tool. It may be updated by the cutting data calculator if cutting data has been calculated using the 'Calc.' button.
Feed per revolution	The feed per revolution (in the radial direction) to be used for the tool. It may be updated by the cutting data calculator if cutting data has been calculated using the 'Calc.' button.

Feed per revolution, Axial	The feed per revolution (in the axial direction, e.g., for drilling, reaming, etc.) to be used for the tool. It may be updated by the cutting data calculator if cutting data has been calculated using the 'Calc.' button.
Feed per tooth	The feed per tooth to be used for the tool. It may be updated by the cutting data calculator if cutting data has been calculated using the 'Calc.' button.
Axial feed	The feed rate (in the axial direction, e.g., for drilling, reaming, etc.) to be used for the tool. It may be updated by the cutting data calculator if cutting data has been calculated using the 'Calc.' button.
Feed	The feed rate (in the radial direction) to be used for the tool. It may be updated by the cutting data calculator if cutting data has been calculated using the 'Calc.' button.
Speed	The spindle speed to be used for the tool. It may be updated by the cutting data calculator if cutting data has been calculated using the 'Calc.' button.
Button 'Report'	Creates and opens the report for the current tool. Here, you can find all the data and generate a tool card containing the essential information for machining.
Button 'Get'	Opens the cutting data calculator. If data (e.g., cutting speed, feed per revolution) is already entered in the tool data window, the calculation mode of the cutting data calculator will be pre-selected. However, this can be changed separately. Diameter, radius, and number of teeth must be entered before pressing the button. If you want to calculate using axial feed values, leave the fields for radial values empty, and vice versa. The fields will then be filled with the result values upon returning from the cutting data calculator.

Button ‘Cancel’	Discards all changes and closes the tool data window without saving.
Button ‘Clear’	Clears all cutting data in the input fields.
Button ‘Save’	Saves all data and closes the tool data window.

Precision cut material to length/ Parting off

Description

The Precision cut material to length / Parting off function window is used for quickly cutting flat and round material. When used on lathes, round material can be comfortably parted off, while on milling machines, both round and flat materials can be cut to length.
This process occurs without changes to the coordinate system, meaning any coordinates can be used without, for example, having to zero the axis at a specific point.
For simplicity, the axis designation is 'Z' in turning mode and 'X' in milling modes, but you can adopt the axis names of your machine, as these are only for description purposes.
In milling modes, multi-axis cutting is also possible, which is done by entering and calculating individual values for each axis. Each axis must be calculated separately.
Note: Multi-axis cutting requires a specific type of workpiece clamping that allows all sides of the workpiece to be reached by the cutter. A normal clamping in a vise, as used for single-axis cutting, is generally not suitable here.

How do you use this function?

Select the cutting operation you want to perform in the combo box.

Parting off on the lathe	Standard parting off, starting from the faced right side.
Cutting to length on the milling machine Starting point on the left.	Cutting to length on the milling machine. The starting point for the operation is the left side when standing in front of the machine and looking toward the vise (Figure 39).
Cutting to length on the milling machine Starting point on the right.	Cutting to length on the milling machine. The starting point for the operation is the right side when standing in front of the machine and looking toward the vise (Figure 40).

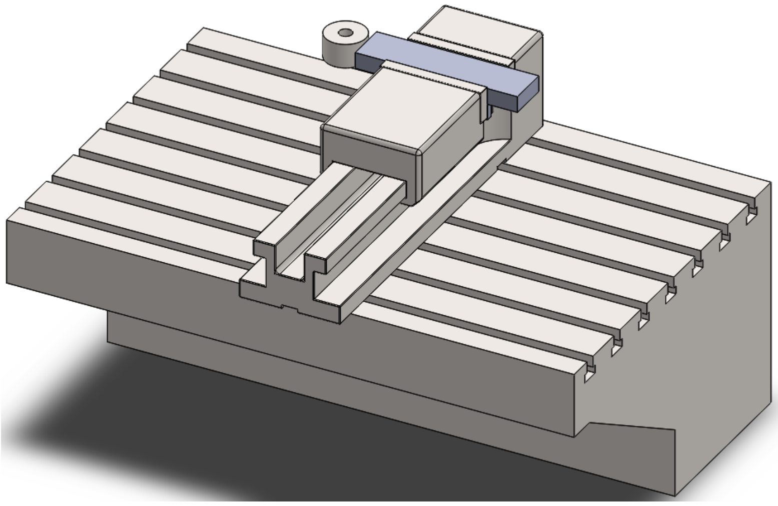

Fig. 35: Cutting to length, starting point on the left.

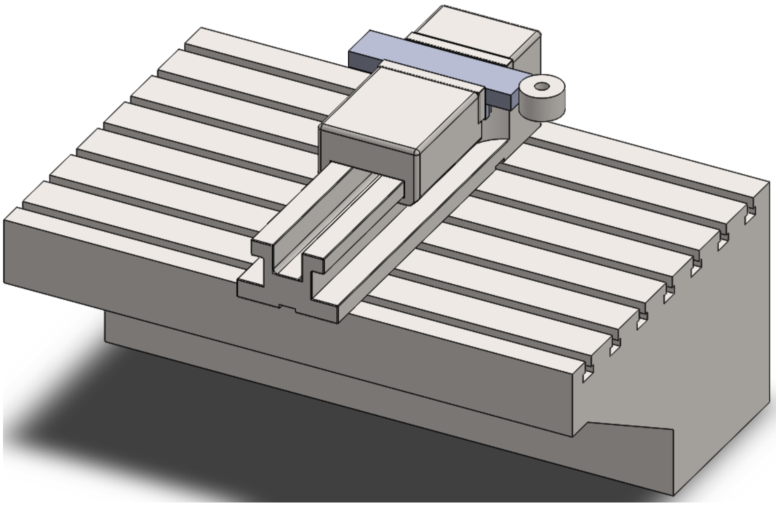

Fig. 36: Cutting to length, starting point on the right.

The following work steps are described according to the application:

All operations:

Select the axis coordinate origin that matches the machine being used. This step is crucial for the calculation.

Axis coordinate origin on the left	The zero point of the axis is on the left side, and the axis values increase as you move to the right along the axis.
Axis coordinate origin on the right:	The zero point of the axis is on the right side, and the axis values increase as you move to the left along the axis.

Parting off on the lathe

Target length A	Here, you enter the length to which the material should be cut. **Note**: Be mindful of obstacles, such as chuck jaws, and ensure there is enough clearance.
Starting point coordinate B	The axis coordinate where the parting tool or parting blade is positioned at point B, when the flat surface of the workpiece and the side surface of the cutting edge are just touching.
Tool width	The width of the cutting edge of the parting tool or the width of the indexable insert of the parting blade.

The result is the ‘cutting length coordinate.’ This represents the position on the axis (here ‘Z’) that must be reached to cut the workpiece to the ‘target length A.’

Again, be sure to avoid obstacles such as the chuck, chuck jaws, etc.

Cutting to length on the milling machine

Pre-consideration:
Which mode should be used for the length cutting?

Side B: Raw surface, must still be milled	This mode is intended for raw parts that have been cut to their rough length at both ends, for example, by sawing. Both ends may have an angled cut due to incorrect saw settings and need to be milled square.
Side B: Usable surface, zero point on edge	This mode is intended for raw parts that have already been milled square on one side, and whose flat surface is of sufficient quality that no further processing is required. Point B is positioned exactly on the edge (e.g., measured using a 3D probe).

Once the mode has been correctly selected for the raw part, you can proceed with entering the additional parameters:

Target length A	Here, you enter the length to which the material should be cut. **Note:** Be mindful of obstacles such as vise jaws, parallel pieces, clamping screws, clamping pads, and ensure there is enough clearance.
Raw part length C after first cut at B:	The length of the raw part after the first milling pass at point C. If the second, yet-to-be-processed side of the workpiece is slightly angled due to sawing, the largest value should be entered here.
Starting point coordinate B	The axis coordinate (here ‘X’) where the cutter is located after completing the milling of side B, OR (if

	the side did not need to be milled) the spindle center point contacting side B (measured using an edge sensor or 3D probe).
Finish allowance	If the raw part length was too large, the side A may need to be milled to length in several passes. For the final cut (finish cut), the oversize can be specified here, which must be cut first so that the final dimension is achieved with the finishing pass. **Note:** The coordinate of the finish undersize is only shown in the report and is **not** displayed in the window.
Tool diameter	The diameter of the cutter. The more accurately this is measured, the more precise the length of the cut workpiece will be.

Once all parameters have been selected and entered, pressing the 'Calc' button will provide the coordinates of the point on the axis (here 'X') where the raw part must be cut to achieve the dimension A in the 'Results' field.

Note: The coordinate of any finish undersize (rough milling coordinate A) is only shown in the report. Additionally, the cutter's engagement/feed width (Ae) will be displayed, which is required to reduce the raw part length to the target length. Any finish undersize is not considered here.

Design functions

Drill Depth Calculations

Description

Drill Depth Calculation Window

The drill depth calculation window allows you to determine the required drill depths for blind holes and through holes. It supports the machining operations drilling, reaming, and tapping core holes**.**

Due to the drill tip angle, as well as the chamfer/lead-in of reamers and taps, it is necessary to drill (and ream/tap) deeper than the functional cylindrical depth of the hole.

For through holes, the focus is on:

- The additional cutting depth needed after breaking through the workpiece to complete the drilling/reaming/tapping process.
- Determining when the feed rate should be reduced as the drill breaks through the material on the underside.

How to Use This Function?

Drilling Mode (Figure 25)

1. Select 'Drilling' under 'Input.'
2. Enter the following values:

    - Diameter
    - Desired cylindrical (usable) hole depth
    - Drill tip angle
    - Optional approach distance La

3. (Optional) Round up the drill depth

    - Activate the checkbox **'**Round up drilling to next**'** and enter a value.
    - Example: If the calculated depth is 22.9 mm, and you enter 0.5 mm, the depth is rounded up to 23.0 mm.

4. Enter a safety allowance under 'Safety Value.'

    - This value is added to the required drill depth and ensures a sufficient cylindrical drilling area.

5. Click **'**Calc**'** to determine the required drilling depth.

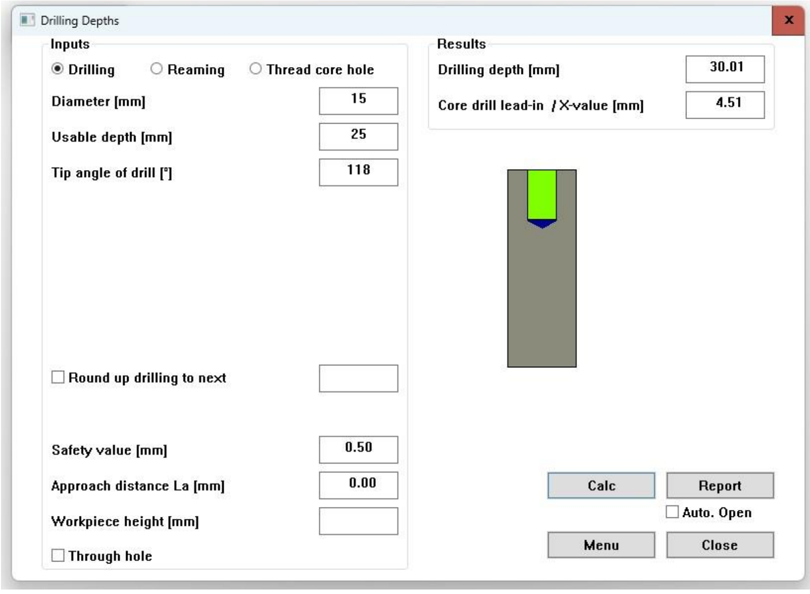

Fig. 37: Drill depth calculation

Reaming Mode (Figure 26)

The reaming function allows precise calculation of the required bore depth for reamed holes. Since reaming tools have a lead-in chamfer, it is necessary to drill deeper than the required cylindrical hole depth.

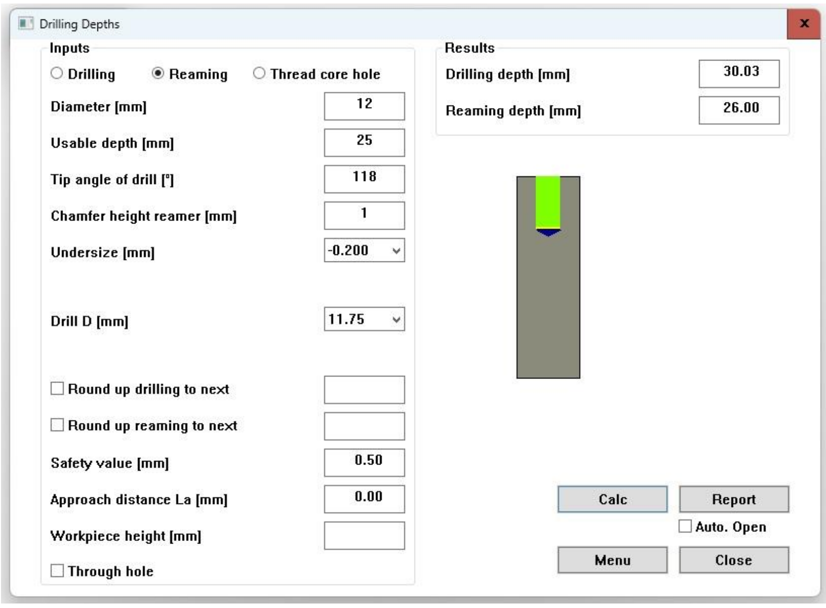

Fig. 38: Reaming

How to Use This Function

1. Select ‘Reaming’ under ‘Input.’
2. Enter the following values:

    - Diameter of the reamed hole (e.g., 8H7 → 8.0 mm).

- Desired usable cylindrical depth of the reamed section.
- Drill tip angle.
- Chamfer/lead-in height of the reamer.
- Optional approach distance La

3. Select the pre-drill undersize from the dropdown menu 'Undersize.'

    - Choose from standard values or select 'Custom' to enter a specific undersize.
    - The sign (+ or -) is not required; only the absolute value matters.

4. Select the drill diameter from the dropdown menu 'Drill D' or enter a custom value.
5. (Optional) Round up the drilling depth

    - Enable **'**Round up drilling to nex**t'** and specify a rounding value.
    - Example: A calculated depth of 22.9 mm is rounded up to 23.0 mm **if** 0.5 mm is selected.
    - **Note:** The reaming depth remains unchanged.

6. (Optional) Round up the reaming depth

    - Enable **'**Round up reaming to next' and specify a rounding value.
    - Example: A calculated reaming depth of 24.5 mm is rounded to 24.75 mm if 0.75 mm is selected.
    - The drill depth is automatically adjusted to accommodate the increased reaming depth.

7. Enter a safety allowance under 'Safety Value.'

    - This value is added to the required drill depth.
    - Example: If 5.0 mm is entered, the drill depth increases by 5 mm (useful for chip space, etc.).
    - **Note:** The reaming depth remains unaffected.

8. Click **'**Calc**'** to determine the required drilling and reaming depths.

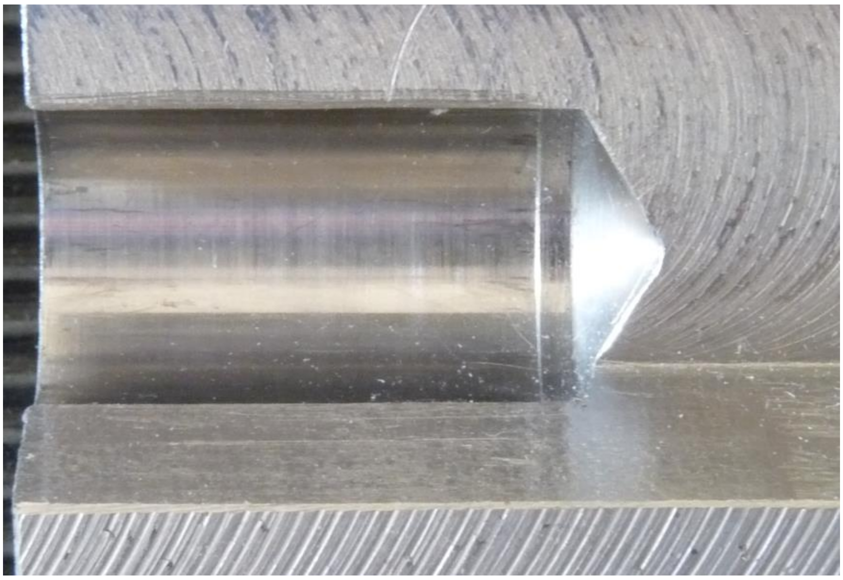

Fig. 39: Reaming a blindhole

Thread Core Hole Mode (Figure 28)

The thread core hole function calculates the required drilling depth for threading operations. Since taps have **a** lead-in chamfer, the hole must be drilled deeper than the usable thread depth.

How to Use This Function

1. Select 'Thread Core Hole' under 'Input.'
2. Choose the desired thread type from the dropdown menu.
3. Enter the following values:

    - Usable depth (cylindrical section of the thread).
    - Drill tip angle.
    - Chamfer/lead-in height of the tap.
    - Optional approach distance La
    - The tap's lead-in can be specified as:

        ▪ A length value (e.g., 3 mm).
        ▪ A factor according to DIN 2197 (e.g., 'pitch × factor').
        ▪ Select the appropriate radio button to define the method.
4. Drill diameter is pre-selected according to the chosen thread type.
5. (Optional) Round up the drilling depth

    - Enable 'Round up drilling depth to next' and specify a rounding value.
    - Example: A calculated depth of 22.9 mm is rounded up to 23.0 mm if 0.5 mm is selected.
    - **Note:** The thread depth remains unchanged.

6. (Optional) Round up the tapping depth

    - Enable 'Round up tapping depth to the next' and specify a rounding value.

- Example: A calculated thread depth of 24.5 mm is rounded to 24.75 mm if 0.75 mm is selected.
- The drill depth is automatically adjusted to accommodate the increased thread depth.

7. Enter a safety allowance under 'Safety Value.'

    - This value is added to the required drill depth.
    - Example: If 5.0 mm is entered, the drill depth increases by 5 mm (useful for chip space, etc.).
    - **Note:** The thread depth remains unaffected.

8. Click 'Calc' to determine the required drilling and threading depths.

Fig. 40: Threading

Additional Notes on the Depth Calculation Window

Workpiece Height Input

- The 'Workpiece Height' field is used for all three functions (Drilling, Reaming, and Thread Core Hole).
- It is essential for through-hole calculations and enhanced graphical representation.

Through-Hole Calculation

- To calculate **a** through-hole, enable the 'Through-Hole' checkbox.
- The software will determine the necessary depth for complete penetration and any required additional depth for chip clearance and full tool engagement.

Results Display

- On the right side, under 'Results', the calculated depths are displayed for reference.

Graphical Representation

- The colors in the graphic indicate different depth values:
  - Blue → Drilling depth
  - Yellow → Depth required for reamer or tap to achieve the intended function depth
  - Green → The final functional depth, representing the cylindrical drilled section, reamed section, or usable thread depth.

Dowel Pin Calculator

Description

In the 'Dowel Pin Calculator ' function window (Figure 41), the optimal pinning of two components using a cylindrical pin can be considered, and the drilling and reaming depths can be determined for different drill point angles. The focus is on blind and through-hole drilling.

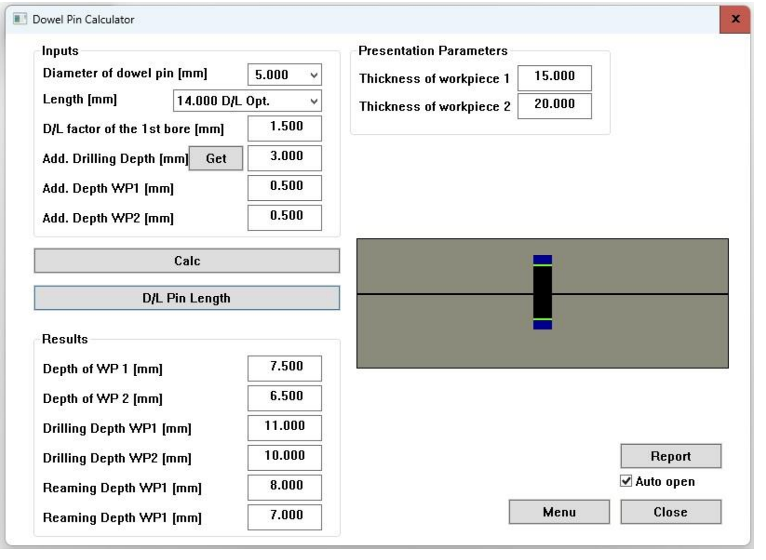

Fig. 41: The Dowel Pin Calculator Window

How do you use this function?

As the first step, it is recommended to enter the thickness of the components/plates to be pinned in the 'Display Parameters' input area. 'Thickness Workpiece 1' refers to the upper plate in the graphical representation, and 'Thickness Workpiece 2' refers to the lower plate. Instead of the total thickness of a workpiece, the corresponding material thickness within the workpiece that is available for the cylindrical pin can also be entered, as a cavity filled with liquid may start above it, and a through-hole should not be drilled there.

The subsequent procedure is as follows:

- In the input area, select the desired diameter of the cylindrical pin from the combo box.
- In the 'Length' combo box, the most common standard lengths listed in DIN EN ISO 2338 or ANSI/ASME B18.8.2 for the selected cylindrical pin diameter are available. You can make a preliminary selection here.
- In the input field 'D/L Factor of the 1st bore' enter the diameter-to-length factor for the cylindrical pin. A value of 1.5, for example, means that the cylindrical pin will have a depth in Workpiece 1 (upper plate in the graphic) equal to 1.5 times its diameter. A value of 1.0 means the pin depth corresponds to its diameter, while a value of 2.0 means it will be twice its diameter, and so on.

- Determine the specific drilling depth of the drill to be used by clicking the 'Get' button (the specific drilling depth is a combination of reaming depth, reamer lead-in, drill X value, etc.). After clicking the 'Get' button, you will switch to the drilling depth calculation window. Enter all relevant parameters, click the 'Calculate' button, and after clicking 'Back,' the calculated values will be transferred to the pinning function window. It is not possible to enter a single numeric value in the 'Add. Drilling Depth' field.
- After changing the cylindrical pin diameter, the drilling depth must be recalculated, as the drill and reamer diameters also change.
- Optionally, you can add an additional depth for the reaming depth in both workpieces in the 'Add. Depth WP1' and 'Add. Depth WP2' input fields. The reamed areas will be made deeper by these values, giving the cylindrical pin more clearance in depth. If these values are set to 0.0, the reaming will be done such that the entire length of the reamed area (sum of holes WS1 and WS2) only has a safety clearance of 0.1mm, meaning the cylindrical pin will have 0.1mm axial play. The safety clearance can be changed in the options (see general tab, 'Safety Factor for Calculations').
- Click 'Calc.'
- In the 'Results' area, all calculated depths will be displayed.
- If desired, you can select different pin lengths in the 'Length' combo box, and the calculation results will be automatically updated. The change in pin length will only affect Workpiece 2. To change Workpiece 1, select a different 'D/L Factor of the 1st Hole.'

The determination of the optimal pin length

- By pressing the 'D/L Pin Length' button, the optimal pin length can be determined from the available lengths. The algorithm tries to find the length that most closely matches the entered D/L factor in both workpieces. In workpiece 1, the entered D/L factor is considered, and the pin length is then calculated to match the D/L factor as closely as possible in workpiece 2. The shorter value is always selected automatically (i.e., the depth in workpiece 2 matches the depth in workpiece 1 but is never deeper). If a longer value is desired, it can be manually selected in the 'Length' combobox, with the optimal lengths marked with the label 'D/L Opt.' after the length.

Notes on this feature window

In the graphical output, the workpieces are shown in gray. The cylinder pins are in black, depth allowances are in green, and the absolute drilling depths are in blue. The graphical output serves as a general qualitative representation of the cylinder pins, drillings, and reamed areas. Absolute measurements are not displayed here, as this would not be possible in certain areas due to the pixel size. Only the numerical values in the output fields and reports are decisive, NOT any measurements, for example, using a ruler, in the graphical output. A correct graphical output is only possible once all required values have been entered in the respective input fields.

Flange Calculator

Description

In the ‘Flanges’ function window, flanges with 1 and 2 bolt circles and circular segments can be quickly created. The flanges can be round or square, and the center hole can be provided with a centering shoulder if needed.

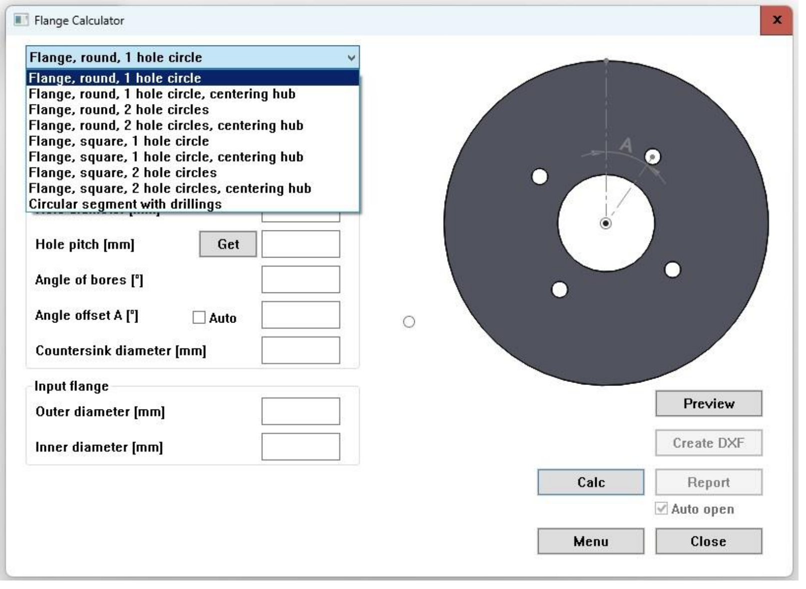

Fig. 42: Flange Type selection

How do you use this function?

Creating Flanges with One or Two Bolt Circles

- Select the type of flange you want to create from the combobox (Figure 42).
- For flanges with two bolt circles, choose which bolt circle you want to edit.
- In the ‘Input pitch circle’ section, enter the following in the corresponding input fields: the number of bores, the hole diameter, and the countersink diameter if you want to add a countersink around the hole. If no countersink is needed, leave the input field blank or enter 0.
- For the distance between the holes, you can either enter the hole pitch (if known) or determine the pitch between two adjacent holes by clicking ‘Get.’ This opens the calculation window where the pitch can be computed. By clicking ‘Back,’ the calculated pitch will return to the ‘Flanges’ function window.

- In the 'Angle Offset' input field, enter the angle by which the hole pattern should be rotated from the vertical axis, or check the 'Auto' box to ensure the upper two holes form a horizontal line.
- In the 'Input Flange' section, enter the following in the corresponding input fields: outer diameter/side length, inner diameter, and optionally, the centering diameter.
- Click 'Calc'.

In the graphic, you can also select the location of the zero point for all holes. For a round flange, set the radio button (Figure 43) either at the center or outside. For a square flange, the zero point can additionally be positioned at one of the four corners.

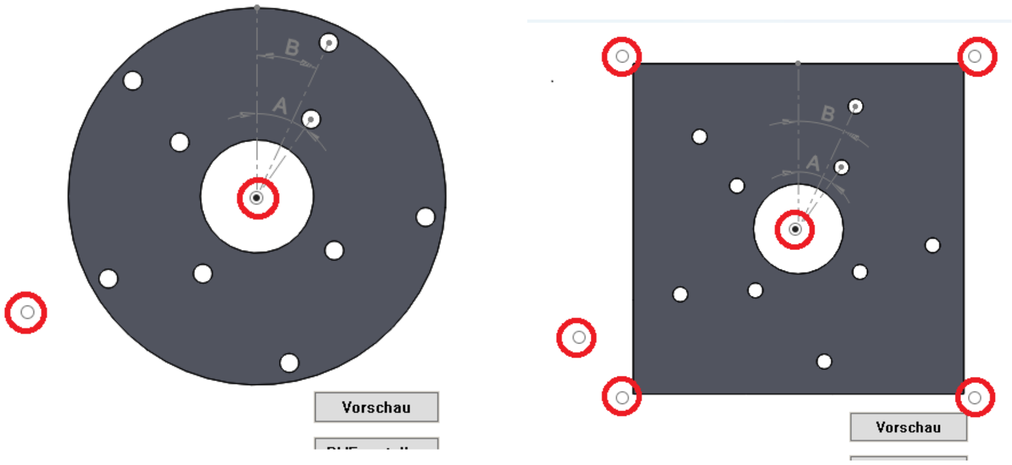

Fig. 43: Possible Zero Points Highlighted in Red

If the Zero Point is outside, the Zero Point in the X-axis and Y-axis can be entered in the 'Input Flange' section. For example, if you enter -45.0 for 'Zero Point X' and 50.55 for 'Zero Point Y', the center of the flange's center hole will be at X=-45.0 and Y=50.55, and all other holes will be dimensioned from this point.

Creating Circular Segments

- Select the function 'Circular Segment with drillings' from the combobox (Figure 42).
- In the 'Input pitch circle' section, enter the following in the corresponding input fields: the number of bores, the hole diameter, and the countersink diameter if you want to add a countersink around the hole. If no countersink is needed, leave the input field blank or enter 0.
- The angle between two adjacent holes is entered in the 'Angle between 2 drillings X' input field.
- In the 'Angle Offset A' input field, enter the angle by which the hole pattern should be rotated from the vertical axis. A positive value will rotate the pattern clockwise, and a negative value will rotate it counterclockwise.
- The center of the imaginary bolt circle for the circular segment is entered in the fields 'Center P Bolt Circle in X' and 'Center P Bolt Circle in Y.' X and Y are the axes, and if different axis labels are used, interpolate accordingly.

Flange Preview

By clicking the 'Preview' button, the preview window opens (clicking the button again closes it). Here, you can visually check the calculated flange (Figure 44). Use the zoom buttons at the bottom to enlarge, shrink, or reset the view to the default.

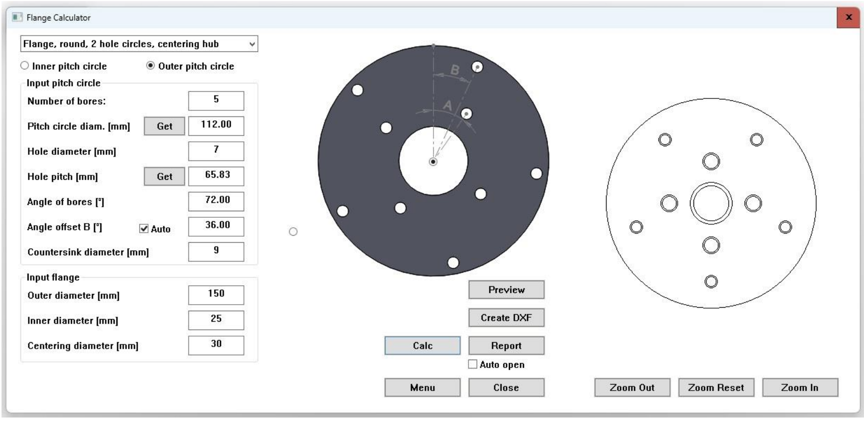

Fig. 44: The opened preview window

Output of the flange as DXF or report

Once all the inputs have been entered, you can create a DXF file (DXF Version 12) by clicking the 'Create DXF' button. After clicking the button, select the location where the file should be saved and enter a file name. The DXF file can then be imported into a CAD system and used as a basis for creating a 3D model of the flange.

If you want to drill the flange holes directly, you can view all the coordinates in the report. This allows for quick marking, center punching, and drilling.

Notes on this feature window

Using the right-click context menu, you can delete related or dependent values simultaneously, making it easy to see where new values need to be entered.

For example, if you want to change the number of holes afterward, click on the input field 'Number of Holes,' right-click to open the context menu, and select 'Delete this field and corresponding fields.' This resets all values that depend on the number of holes to zero, requiring new entries.

Calculations

Description

The ‘Calculations’ function window contains a collection of common calculations and conversions used in the workshop, at the machine, or in production planning.

How do you use this function?

- Select the desired calculation type from the ‘Function’ combo box.
- OR enter a keyword related to the desired calculation (e.g., ‘pitch diameter for holes’) in the ‘Search’ input field. The ‘Function’ combo box will then display only the calculation types that contain the search term. (To reset the list, clear all entries in the search field or press the ‘ESC’ key.) Select the calculation type from the list.
- On the left side, you will find the input fields required for the calculation. Enter the necessary values here.
- Depending on the complexity of the calculation, a drawing may be displayed to help visualize the input values.
- Press the ‘Calc’ button to perform the calculation.
- The results will be displayed on the right side.
- Click the ‘Copy’ button to copy the results to the clipboard for use elsewhere.

Notes on this feature window

Reports are not available in the ‘Calculations’ function window, but results can be easily copied. In the options, you can specify whether only the numerical values of the results are copied to the clipboard or if the values are copied along with their units.

We also offer custom function libraries. These libraries are tailored to run exclusively at the customer's site with the ordered functionality, ensuring that other customers do not have access to these features.

For inquiries about custom function libraries, please feel free to contact us via email.

Options

The options menu of SIMmachina Tools is divided into a general section and a machine-specific section.

The general section applies to all function windows, while the machine-specific section only applies to the currently ***selected machine in the machine*** database.

General

In the general options, the following settings can be adjusted:

Program Settings:

- 'Startup Page': The function window that is opened when SIMmachina Tools starts. If 'Last used function window' is selected, the last used function window (at the time the program was closed) will be opened at the next startup.
- 'Language': The language for the menus and the language of the 'Manual' that can be accessed via the 'Manual' button. *
- 'Unit for Lengths': The unit for user input of length measurements.
- 'Unit for Volumes': The unit for user input of volume measurements.
- Safety factor for calculations: A safety buffer that is incorporated into the result of some calculations.
- 'Open Reports by Default': Default setting for the 'Auto Open' option under the 'Report' button in various function windows. After each calculation, the report will automatically open in the default TXT file editor.
- 'Copy Units to Clipboard': ***See explanation here***.
- 'Level for calculations': Explanation below.
- 'User's Manual' Button: Opens this manual.
- 'About' Button: Information about SIMmachina Tools and checking for updates, if an internet connection is available.
- 'Data Directory' Button: Opens the data directory of SIMmachina Tools.
- 'License' Button: Important for upgrading to a higher version of SIMmachina Tools.
- Delete Reports: Delete all reports of a certain age.

Calculation Level

Here you can select which version of the calculation function window will be started when opened.

Standard	The standard function library provided with SIMmachina Tools for the calculation function window. A ***list of the current functionality*** is included in the appendix of this manual.

Custom	It is possible to obtain custom function libraries through us. The advantage of these custom libraries is that they run at the customer's site with the ordered functionality, but this functionality is not available to other customers. By selecting ‘Custom,’ the functionality of the custom function libraries will be loaded when the calculation function window is opened, instead of the standard function library. It is possible to switch between both libraries, but this always requires reopening the calculation function window. For proper operation, however, the ***correct installation of the custom function library*** is necessary.

Machine functions 1

In the machine functions 1 tab, the following settings can be adjusted:

Mechanical Data:

- 'Machine Name': The name of the machine or workstation, which will be displayed in the machine selection menu.
- 'Available Speeds in Steps': If the machine has a stepped gearbox, the speeds can be entered here.
- 'Available Parallel Pieces': All parallels available at this machine/workstation. ***See here.***
- 'Available Dividing Head Plates': All hole circles available at this machine/workstation. ***See here.***
- 'Rev. crank for 1 dividing dead rev.': How many times the crank of the dividing head must be turned for the spindle to complete one full revolution (360 degrees).
- 'Jaw Height of Vise': The jaw height of the preferred vise on this machine/workstation.
- 'Delete Machine' Button: Deletes the current machine. This will affect the settings in both Machine Functions 1 and Machine Functions 2. The machine will only be fully deleted after the button is pressed AND 'OK' or 'Apply' is clicked. If 'Cancel' is pressed, the data will not be deleted.

RS232 Settings and Port Status:

Data for the serial interface***, See here.***

Machine functions 2

NC Code Data:

- 'NC Line Numbering Format': ***<u>See here.</u>***
- 'NC Tool Length/ Diameter Format': ***<u>See here</u>.***
- 'Edit NC Data in External Editor' and 'Use Notepad'***: <u>See here.</u>***
- 'Revs. During Dwell Time': The standard revolutions to be performed when the axial feed is paused during dwell time.
- 'Mem': The program memory that is available in the machine controller.

Coolant Data:

- Combo Box: Select the tank to be edited.
- 'Machine/Tank Name': The name of the tank.
- 'KSS Tank Length' and 'KSS Tank Width': Internal dimensions: length and width of the tank.
- 'KSS Tank Capacity': The capacity of the tank.
- 'Tank Volume/Unit' and '1 Unit'***: <u>See here.</u>***
- 'KSS Tank 4x Corner Radius': Preferred corner radius. ***<u>See here.</u>***
- 'KSS Refractometer Factor': Preferred refractometer factor.
- 'KSS Target Concentration': Preferred target concentration.
- 'KSS Product Name': Preferred KSS concentrate.
- 'Delete Entry' Button: Deletes the selected tank from the combo box.
- 'Save Entry' Button: Saves the current data under a new tank if a changed name is entered under 'Machine/Tank Name.'
- 'Use Entry' Button: Use the current data in the 'Cooling liquid calculator' window.

Appendix

FAQ

How to Change the Language

Open the options window via the 'Menu' and 'Options.' Under the 'General Options' tab, you can select the desired language on the right side under 'Language.'

How to Add or Change Speed Steps for My Machine

- Select the machine for which the speed steps should be added or changed from a function window that has access to the machine database.
- Open the options window via 'Menu' and 'Options.'
- Select the 'Machine Functions 1' page.
- In the input field 'Available Speeds in Steps,' enter, remove, or add the desired speeds. **Note:** After each speed step, a semicolon must be used as a separator, e.g., '80.0; 100.0;'. There is no need for a space after the semicolon, and a comma or period can be used for the decimal separator.
  The program will convert everything into the correct format when saving.
- Confirm with 'Apply' or 'OK' and close the window.

How to Add or Change Parallels for My Machine

- Select the machine for which the parallel parts should be added or changed from a function window that has access to the machine database.
- Open the options window via 'Menu' and 'Options.'
- Select the 'Machine Functions 1' page.
- In the input field 'Available Parallel Parts,' enter, remove, or add the desired parallel parts. The input format is height x depth, e.g., '36.0x8.5.' The length of the parallel parts is not used here.
  Note: After each parallel part, a semicolon must be used as a separator, e.g., '36.0x8.5;'. There is no need for a space after the semicolon, and a comma or period can be used for the decimal separator.
  The program will convert everything into the correct format when saving. However, it is required to enter a decimal place after the period.
- Confirm with 'Apply' or 'OK' and close the window.

How to Add or Change Dividing Head Plates for My Machine

- Select the machine for which the dividing head plates should be added or changed from a function window that has access to the machine database.
- Open the options window via 'Menu' and 'Options.'
- Select the 'Machine Functions 1' page.

- In the input field ‘Available Dividing Head Plates,’ enter, remove, or add the desired hole plates. The input format is the number of holes as an integer. Note: After each hole count, a semicolon must be used as a separator, e.g., ‘19; 20; 21;’.
- Confirm with ‘Apply’ or ‘OK’ and close the window.

How to Change the RS232 Settings for My Machine

- In a function window with access to the machine database, select the machine tool for which the settings need to be changed.
- Open the options window via ‘Menu’ → ‘Options.’
- Select the ‘Machine Functions 1’ tab.
- In the ‘RS232 Settings’ section, choose whether to modify the settings for sending or receiving.
- Edit the serial interface parameters according to the requirements of the connected machine tool's control system.
- Save the settings by clicking ‘Apply’ or ‘OK.’
- The ‘Test’ button in the ‘Port Status’ section can be used to check whether the correct serial interface settings have been entered when a machine is connected. However, this does not work with all control systems.

The ‘Timeout’ input field in ‘Receive’ mode is used to automatically detect when all data has been successfully transferred from the machine to the PC, allowing the system to stop waiting for new data. If issues arise with complete data reception, this value can be gradually increased. The higher the value, the longer the system waits for new data. However, this value does not represent a unit of time (such as seconds) but rather refers to program loops or iterations.

This setting is particularly relevant for reception issues at very low baud rates. A value of 20 has proven effective across a wide range of baud rates, from high speeds down to 110 baud.

The ‘RS232 T.out’ input fields define the minimum timeout value (in minutes) for both sending and receiving. These determine how long the send and receive functions remain active before being terminated by the operating system. This termination can occur, for example, due to a transmission error or if the receive function waits too long for a ‘Xon’ signal.

In addition to the user-defined values, the system also adds the default Windows® values, which are determined when SIMmachina Tools starts. If the value is set to 0, the system will rely solely on the Windows® default values.

Flowcontrol ‘Xon/Xoff’

In the ‘Xon/Xoff’ flow control settings, the lower input fields can be adjusted separately for ‘Send’ and ‘Receive’ modes.

XOn Char:	The character for the Xon command. Input is in HEX format, e.g., enter '0x11' for the standard character '^Q' with the decimal value '17.'
XOff Char:	The character for the Xoff command. Input is in HEX format, e.g., enter '0x13' for the standard character '^S' with the decimal value '19.'

For 'Receive' mode:

XOn Buff:	The buffer size value at which XOn is sent. If set to 0, the system uses the Windows® default values.
XOff Buff:	The buffer size value at which XOff is sent. If set to 0, the system uses the Windows® default values.

How to Set Where an NC File Should Be Edited

- Select the machine for which the settings should be changed from a function window that has access to the machine database.
- Open the options window via 'Menu' and 'Options.'
- Select the 'Machine Functions 2' page.
- To edit the NC files directly in SIMmachina Tools, uncheck the 'Edit NC data in external editor' box.
- If you want to edit the NC files in an external editor, check the 'Edit NC data in external editor' box.
- For the Windows® Editor, check the 'Use Notepad' box.
- If you want to use another external editor, uncheck the 'Use Notepad' box and click 'Select' to choose the program file of the desired editor, then click 'Open.'

For an external editor to work, ensure it opens by double-clicking the NC file in the Explorer. This must be confirmed for SIMmachina Tools to use it. Also, ensure there are no spaces in the path or filename of the NC file.

How to Change the Preferred Line Numbering Format

- Select the machine for which the settings should be changed from a function window that has access to the machine database.
- Open the options window via 'Menu' and 'Options.'
- Select the 'Machine Functions 2' page.
- In the 'NC Line Numbering Format' input field, enter the desired numbering format. This could be a letter followed by n digits, like N0000 or N00000, or just digits, like 0000. The number of zeros determines the number of digits (e.g., N000: N000 Line 1, N001 Line 2, N003 Line 3, etc.). Leading zeros improve the readability of line numbers.
- Confirm with 'OK' or 'Apply.'

How to Change the Tool Offsets in the Tool Database

- Select the machine for which the tool offsets should be added or changed from a function window that has access to the machine database.
- Open the 'Tool Database' function window.
- Select the tool number to be edited from the list view (double-click on the number in the 'Tool No.' column). Then click the 'Tool Data' button to open the tool data window.
- Click on the 'NC Data' field and edit the tool data. The tool offset string is not checked by SIMmachina Tools at the moment, so the user must manually verify its accuracy.
- Click 'Save' to apply the new data or click 'Cancel' to keep the original data.

How to Change the Tool Format in the Tool Database

- Select the machine for which the tool format should be added or changed from a function window that has access to the machine database.
- Open the options window via 'Menu' and 'Options.'
- Select the 'Machine Functions 2' page.
- In the 'NC Tool Length/Diameter Format' input field, enter the desired format with the highest possible tool number. For example, D99 could be used for a tool format starting with D00 and having over 100 tools.

How to Set Whether Units Should Be Copied with the Values in the Calculation Window

- Open the options window via 'Menu' and 'Options.'
- Select the 'General' page.
- Check the box for 'Copy units to clipboard' if you want the units to be copied along with the values.
- Confirm with 'OK' or 'Apply.'

How to Copy a Text Block

Select the text to copy with the mouse and press Ctrl + C, or right-click and choose 'Copy' from the context menu.

How to Paste a Text Block

Click in the input field or the location where you want to paste the text and press Ctrl + V, or right-click and choose 'Paste' from the context menu.

How to Insert Coolant Data from the Machine Database

- Open the options window in the Coolant calculation window via 'Menu' and 'Options.'
- In the 'Machine Functions 2' tab, in the 'Coolant Data' section, select the tank whose data should be used from the combo box.
- Click 'Apply.'
- Close the menu with 'OK' or 'Cancel.' The tank data and the refractometer factor will now be available in the current calculation.

How to Select a Machine

In function windows that access the machine database, select the desired machine by going to 'Menu,' then 'Machine,' and clicking on the name of the machine you want to choose. A checkmark will appear next to the name, indicating that it is currently selected.

How to Add a Machine

- In function windows that access the machine database, select an empty slot by going to 'Menu,' then 'Machine,' and clicking on a field labeled 'Empty.' You can then add the machine data in the options.

How to Delete Older Reports

- Back up important reports first. You can open the corresponding data directory via the 'Data Dir' button in 'General Options.' Reports are located in the subfolders.
- Open the options window via 'Menu' and 'Options.'
- Select the 'General' page.
- In the lower part of the window, select the time range for which reports should be deleted, e.g., 'Older than 30 days.'
- Click 'Delete.' All reports older than the selected time period (e.g., older than 30 days from the current system date) will be permanently deleted and cannot be recovered.

How to Change the Colors for NC Code Syntax Highlighting

- Select the machine for which the colors should be added or changed from a function window that has access to the machine database.
- Open the options window via 'Menu' and 'Options.'
- Select the 'Machine Functions 2' page.
- Colors 1 to 3 are used for specific words, such as M30 or G00. Color 4 is used for comments with a start and end word, e.g., (and) or /* and */.
- For colors 1 to 3, enter the words separated by spaces, e.g., G00 G01 G02. For color 4, the format is: Start word | End word, e.g., [|] (|) ///.
- Use the 'Color1' to 'Color4' buttons to open the color selection window and pick the desired color.
- Close the options window with 'OK' or 'Apply.'

How to Install a Custom Function Library

- The custom function library comes with at least two DLL files: one is the actual library, and the other is the language library in German for the functions. If additional languages were ordered, the number of files will increase by one per language.
- All provided files must be copied into the directory 'C:\ProgramData\HORUS Robotics\SIMmachina Tools\CustomCalculations.' You can open the directory via the 'Data Dir' button in 'General Options.' You may need to navigate to the 'CustomCalculations' folder.
- Then, go into the subfolder for either 64 or 32, depending on whether you are copying the 64-bit or 32-bit version.
- Ensure that only one version of a custom function library is present in these directories for the calculation function window to work correctly.

Known Issues

Windows 7®:

Windows 10®:

Issue:

On some systems, it has occurred that multiple radio buttons are selected at the same time in the graphic. Functionally, the radio button that was last selected with the mouse is chosen, but visually, this creates an undesirable behavior.

Solution:

Now, if you hover the mouse over the additionally selected radio button (without clicking it), it will be visually corrected to its proper state.

Windows 11®:

Decimal Degrees to Degrees Minutes Seconds	Converts angle degrees from decimal form to degrees, minutes, and seconds, e.g. 13.510° to 13° 30' 36’.
Degrees Minutes Seconds to Decimal Degrees	Converts degrees, minutes, and seconds to decimal form, e.g. 13° 30' 36’ to 13.51°.
mm to Fractal Inches	Converts millimeter values to fractal inches, e.g. 26.000 mm to 1 3/128 inches. For input, 3 decimal places are required. An approximation may be calculated, and the result of the approximation will be displayed in the ‘Fractal Calculated’ input field..
Fractal Inches to mm	Converts fractal inch values to millimeters, e.g. 1 3/128 inches to 25.98 mm.
Decimal Inches to Fractal Inches	Converts decimal inch values to fractal inches, e.g. 1.500 inches to 1 1/2 inches. For input, 3 decimal places are required. An approximation may be calculated, and the result of the approximation will be displayed in the ‘Fractal Calculated’ input field.
Fractal Inches to Decimal Inches	Converts fractal inch values to decimal inches, e.g. 1 1/2 inches to 1.5 inches.
mm to Inches	Converts millimeter values to inches, e.g. 33 mm to 1.3 inches.

Inches to mm	Converts inch values to millimeters, e.g. 1.3 inches to 33.02 mm.
Dwell Time	Calculates the required time in milliseconds for the desired number of revolutions at a specific speed.
Torque Offset with Pivot in Drive	Calculates the changed torque value, e.g. DM, that results from the use of adapters due to the change in the lever arm. This calculation method is intended for torque tools where the pivot is exactly in the drive (e.g. the socket). A possible application is the use of torque adapters for ER collets.
Torque Offset with Pivot Outside Drive	Calculates the changed torque value, e.g. for insert tools, that results from the use of adapters due to the change in the lever arm. This calculation method is intended for torque tools where the pivot is offset from the drive (e.g. the socket). A possible application is the use of torque adapters for ER collets.
Temperature-Dependent Linear Expansion	Calculates the change in length based on temperature change and the material's coefficient of expansion. For example, this can be used to calculate the shrinkage for fitting bearing rings using dry ice.
Nm to Foot Pounds	Converts Nm values to foot pounds, e.g. 120 Nm to 88.51 ft.lbs.
Foot Pounds to Nm	Converts foot pound values to Nm, e.g. 80 ft.lbs to 108.46 Nm.

Pitch of Holes	Calculates the pitch (center-to-center distance) of two holes based on their diameter and the smallest inside or largest outside dimension.
US Drill Bit Size to mm	Determines the diameter in mm corresponding to the US drill bit number. Numbers 1-104 and letters a-z are allowed.
US Drill Bit Size to Inches	Determines the diameter in inches corresponding to the US drill bit number. Numbers 1-104 and letters a-z are allowed.
Drill Center Calculation	Calculates the center of a hole or cylinder along one axis between two values or along two axes.

List of Figures