CIMCO Edit v7 User Guide

Contents

License Information

Information in this document is subject to change without notice and does not represent a commitment on the part of CIMCO A/S. The software described in this document may be used or copied only in accordance with the terms of the license. The purchaser may make one copy of the software for a backup, but no part of this user manual may be reproduced, stored in a retrieval system, or transmitted in any form or by any means electronically or mechanically, including photocopying and recording for any purpose other than the purchaser's personal use, without prior written permission from CIMCO A/S.

TERMS OF USE FOR:

> Software: CIMCO Edit v7
> Version: 7.x.x
> Date: March 2014
> Copyright © 1991-2014 by CIMCO A/S

Notice

CIMCO A/S reserves the right to make changes to the CIMCO Edit v7 Software at any time and without notice.

Software License

You have the right to use the number of licenses of the enclosed program, which you have bought from CIMCO A/S. You may not distribute copies of the program or related documentation to any persons or companies. You may not modify the program or related documentation without the prior written consent of CIMCO A/S.

Disclaimer of all Warranties and Liability

CIMCO A/S makes no warranties, either express or implied, with respect to the software, its quality, performance, merchantability, or fitness for any particular purpose. The entire risk as to its quality and performance is with the buyer. Should the CIMCO Edit v7 software prove defective following its purchase, the buyer (and not CIMCO A/S, its distributor, or its retailer) assumes the entire cost of all necessary servicing, repair, of correction and any incidental or consequential damages. In no event will CIMCO A/S be liable for direct, indirect, or consequential damages resulting from any defect in the software, even if CIMCO A/S has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Notice:

The accompanying software is confidential and proprietary to CIMCO A/S. No use or disclosure is permitted other than as expressly set forth by written license with CIMCO A/S.

CIMCO Edit, DNC-Max, and the CIMCO Logo are trademarks of CIMCO A/S.

Microsoft, Windows, Win32, and Windows NT are trademarks or registered trademarks of Microsoft Corporation.

Other brand and product names are trademarks or registered trademarks of their respective holders.

Introduction

CIMCO Edit v7 is the latest version of the most popular CNC program editor on the market. With over 100,000 licenses distributed in the past few years, CIMCO Edit is the editor-of-choice for professional CNC programmers who demand a full-featured and reliable, cost-effective professional editing and communication tool.

Every aspect of CIMCO Edit v7 is new, from the multi-pane tabbed layout to the dynamic toolbars and menus. CIMCO Edit v7 also includes new and powerful tools such as an enhanced file compare utility, a reengineered graphical backplotter, and the new NC-Assistant programming tool.

- **Full-featured CNC editor**

CIMCO Edit v7 provides a comprehensive set of essential editing tools necessary for meeting the demands of modern CNC program editing.

It has no program size limitations and includes CNC code specific options such as line numbering / renumbering, character handling and XYZ range finder. It also features math functions including basic math, rotate, mirror, tool compensation, and translate. CIMCO Edit v7 offers all the functions expected from an editor including drag-and-drop text editing. Best of all CIMCO Edit v7 is completely configurable and is easily adapted to any existing CNC program editing environment.

- **Faster editing with NC-Assistant**

The NC-Assistant makes editing NC code faster and easier than ever. Point at any M or G code and the NC-Assistant will identify the code allowing you to modify values using an interactive interface linked to the CNC code. Input the desired values for any register and the NC-Assistant updates the CNC code automatically.

The NC-Assistant in CIMCO Edit v7 allows you to quickly insert and edit complex cycles and operations. CIMCO Edit v7 includes built-in cycles and macros for the most common operations such as program start, program stop and tool change. You can also record or create custom cycles and macros for the operations most common to your own specific setups and applications.

- **Graphical Backplotter**

The 3D Mill / 2D Lathe backplotter handles your 3-axis Mill and 2-axis lathe CNC programs with step and continuous forward and reverse plotting. Edit the CNC program and the update is automatically reflected in the plot. Analyze the plot with dynamic zoom, pan, rotate and measuring functions. CIMCO Edit v7 supports solid visualization of NC code with tool holder collision check and gouge detection. The Solid Animation function allows you to see the material being removed.

- **Intelligent File Compare**

CIMCO Edit v7 features a fast and fully configurable side-by-side file compare, allowing the user to quickly identify CNC program changes. The file compare identifies changed and deleted / inserted lines, but ignores trivial format changes like block renumbering and spacing. Differences are displayed one line at a time, all at once or printed side-by-side for offline review

- **CNC Communications and DNC**

CIMCO Edit v7 includes DNC capabilities for reliable RS-232 communications with a variety of CNC controls. With the DNC option you can send and receive CNC programs to multiple machines simultaneously from inside CIMCO Edit v7.

- **Support for Mazatrol Files**

View Mazatrol Program files directly in CIMCO Edit v7 instead of on the Mazak Control in the workshop. Quickly verify and review program changes with Mazatrol file-compare.

- **CNC-Calc 7 - 2D CAD add-in for CIMCO Edit v7**

CNC-Calc is a fully featured 2D CAD solution that works inside CIMCO Edit v7. This add-on is a fast and effective solution for solving problems with complex 2D geometry. Users can draw or import (DXF) 2D geometry, specify cut depths, lead-in, lead-out, and other tool path variables and quickly generate CNC code in ISO and other conversational formats for contours and drilling.

What's New

This version of CIMCO Edit includes a number of improvements over previous versions.

- **New user interface.**
  The new user interface in CIMCO Edit v7 follows the latest standards for Windows user interface design.

- **Comprehensive help system.**
  Now everything you need to know about functions that was previously contained in the reference manuals is included in the in-product HTML help and at your fingertips. Stumped on a function? The **F1** key or the **Help** button will take you right to the information you need.

- **Optimized editor engine.**
  The new optimized editor engine loads large files quickly, and with minimal memory usage.

- **Flexible configuration.**
  Flexible configuration of colors, NC commands etc.

- **Advanced standard protocol.**
  Advanced standard protocol allows you to communicate with multiple CNC's simultaneously.

- **Solid simulation with tool animation.**
  The advanced graphical backplotter is now extended to include a dynamic stock animation which allows you to see the material being removed by the tool. You can select solid visualization for 3D milling and turning or 2D turning only.

- **Automatic tool scanning.**
  The advanced tool scanning feature allows you to automatically scan tool dimensions from the NC program file. Changes in the NC program are immediately reflected in the Tool Setup.

- **Automatic stock scanning.**
  CIMCO Edit can now automatically scan stock dimensions from the program file. Changes in the NC program are immediately reflected in the Solid Setup.

- **Load/Save tool libraries.**
  This feature allows you to load and save tools in the library.

- **Support for Fagor.**
  Among many other control types CIMCO Edit now supports Fagor simulation for milling operations.

- **Enhanced Search option.**
  The Search/Replace function now allows you to simultaneously search and replace multiple program lines.

Overview

- **Purpose**

You can use CIMCO Edit v7 to make changes and check CNC programs for your CNC machines. If you have a serial communication line from the PC to the CNC machine, you can send and receive programs directly, even when working with other CNC programs.

CIMCO Edit v7 is designed for editing CNC-programs. CIMCO Edit v7 is able to distinguish between several different formats including ISO, APT and Heidenhain to name some.

- **User interface**

CIMCO Edit v7 has a standard Windows user interface, having one or more windows for every open file. The functions in CIMCO Edit v7 can be activated through the Ribbon Bar, which helps you to quickly find the commands needed for a task. The commands are organized into logical groups under tabs.
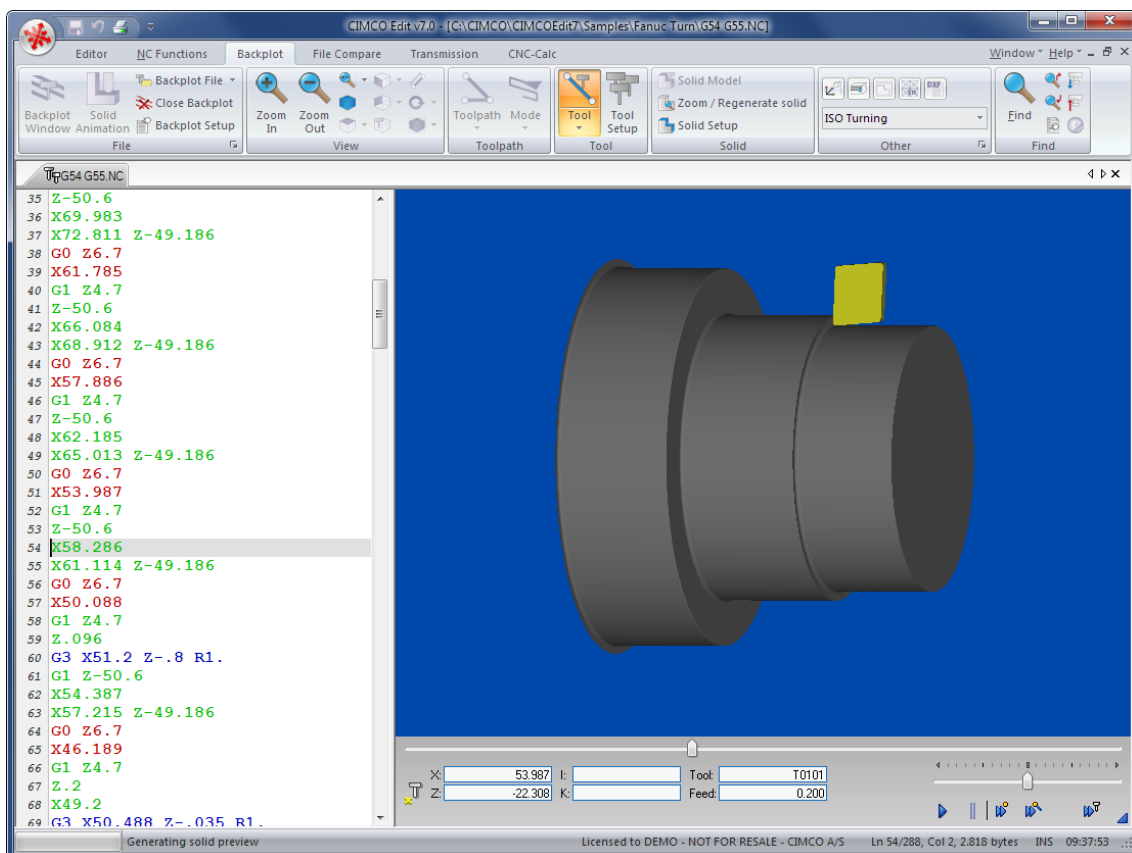
CIMCO Edit v7 User interface.

- **Functions**

You can undo and redo (almost) all operations you have done since the file was opened. You can find the position of a tool change, a spindle speed change or feed rate change, or display the range of the X/Y/Z axes.

If you are not sure if two CNC-programs are equal, the File Compare functions of CIMCO Edit v7 help you to pinpoint the differences. File Compare lets you choose two files that are put into a two-frame window. The differences are shown by coloring the line. You can step through the files, stopping at each difference.

You can customize the colors used to display coordinates and NC commands, renumber block numbers, and add/remove spaces to increase readability, or decrease program size/transmission time.

The DNC part of CIMCO Edit v7 lets you send and receive NC programs to your CNC machines in the workshop. The file can be sent from a window in CIMCO Edit v7 or directly from the hard disk. The file transmission is done while you are working with other CNC programs.

- **Setup**

CIMCO Edit v7 offers you a lot of possibilities to control the behaviour of the editor. The Ribbon tabs let you select functions and configure settings. You can access the configuration dialogs using the icon .

Using the **Editor** tab, you can configure the Global settings for *General*, *Editor*, and *Printing*.

In the *General* dialog, the appearance of the windows can be modified. You can choose whether a file should be displayed in more than one window, or if alterations to a file inside CIMCO Edit v7 through another program should be indicated. Further you can choose whether to share settings between users, or protect read-only files. You can specify the tab width, as well as the size of the recent files list. The language used for viewing Editor menus and a Color theme can also be selected.

In the *Editor* dialog, you can set the speed of the keyboard and choose if you want scrollbars or not.

The appearance of the printed CNC programs is controlled in the *Printing* dialog. Here, you can define a header and/or a footer. If you have a color printer, you might mark the color option.

CIMCO Edit v7 has 5 different templates for different CNC machines. You can set up how the NC functions behave and how the NC codes appear in the editor window. The **Find** function in the Editor menu needs a definition of *Tool change*, *Feed rate change*, etc. This definition is made here.

- **Further information**

In the section Program Tabs, all CIMCO Edit v7's functions are described. A lot of functions can also be accessed through one of the icons.

The setup of **Machine Types** is described in File Types.

If you need help on how to set up a serial communication line, you can refer to DNC Settings and DNC-Max User Guide.

The help system is accessible through the **?** icon, the **F1** key or the **Help** button. The section Using Editor Help gives you an introduction to the help in CIMCO Edit.

Installation

To install CIMCO Edit v7, follow the instructions below.

1. Locate installation files

If you receive CIMCO Edit v7 on a CD-ROM, open the CIMCO Edit v7 directory on the CD. If you receive CIMCO Edit v7 via e-mail, save the file in a temporary directory.

2. Run installation program

Start the installation program by clicking on CIMCO Edit v7 setup program:

CIMCO Edit v7 setup program.

3. Copying your Key file

If you do not have a key file but wish to install an evaluation version, just proceed to the next step.

If you have received a key file from CIMCO, it should be installed by double-clicking on the file "(name)-license.key" after the installation is completed. This opens the CIMCO License Key Manager. Click on the button **Install Keyfile** to install the key file.

4. Starting the editor

When the installation is completed and you have copied your key file, you are ready to start the editor for the first time. Click on the program icon on your desktop.

CIMCO Edit v7.

After starting the editor, please verify that your company name is shown in the **Licensed to** field in the *About* box. If not, the key file was not installed correctly. Please repeat the point no. 3 in order to install the key file correctly.

> ⓘ   If you are running an evaluation version, the license name is *Demo version*.

Program Tabs

The functions in CIMCO Edit v7 are activated using the appropriate icons on the Ribbon bar. The commands are organized into logical groups under tabs.

If the function can be activated through the Ribbon bar or a keyboard shortcut, the icon or the shortcut is displayed next to the command name.

Clicking the small down arrow next to an icon gives you access to additional options and features.

1. Editor

This section describes the functions in the **Editor** tab.

1.1 File

This section describes the commands in the function **File**. The menu contains the functions described below for file editing. You can access the file configuration using the icon 🔲.

For details about the File settings, click on the corresponding icon or see Editor Setup.

ℹ️ The **File** functions described below can also be reached through the icon 🌸.

The File menu.

New / *Ctrl+N*

An empty window opens, ready for input.

ℹ️ Configure the machine type for the program by clicking the setup icon.

Open / *Ctrl+O*

The open file dialog window lets you open an existing program.

Recent

A list of recently used files. Click the down arrow next to the Open icon to open recently used files.

Close / *Ctrl+F4*

Closes the current window. You will be prompted to save changes if the file has been modified.

Close All

Closes all open windows. You will be prompted to save changes for all modified files. Click the down arrow of the Close icon to access this option.

Save / *Ctrl+S*

Saves the current CNC program.

Save All

Saves all open CNC programs under their current names. Click the down arrow of the Save icon to access this function.

Save As

Saves the current CNC program under a new name.

Print / *Ctrl+P*

Prints the current CNC program. If you want to print a part of a program, mark the area of interest and choose **Print**.

Printer Setup

The dialog window for printout configuration is opened. Click the down arrow of the Print icon to open this option.

Global setup

This option lets you configure the general program settings of CIMCO Edit, also accessible through the icon. Please refer to section Editor Setup for more details.

1.2 File Type

This section describes how to configure **File Type** settings. The icon ▣ lets you access the global configuration of File Type.

The **File Type** settings determine how to perform renumbering, look for tool changes, configure backplot settings, colors, where to save files, etc.
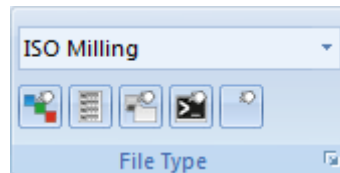
The File Type menu.

File Type

Use this drop-down list to select the file type which determines the coloring, syntax, etc.

Colors

Open the setup dialog to configure color highlighting.

Configure Block Numbering

Open the setup dialog to configure block numbering.

Directories / Extensions

Open the setup dialog to configure directories and extensions.

Machine Templates

Open the setup dialog to configure machine template specific parameters.

Other Settings

Open the setup dialog to configure other machine settings.

Setup File Type

Here you can configure the file type settings. The file type determines the colors, syntax, etc.

1.3 Edit

In this section, the ordinary text editing functions are described.

The Edit menu.

> You can select/mark a part of the text by dragging the cursor from start point to end point, while pressing the left mouse button. The background color on the marked text is changed to indicate the selected area (see Editor Setup / File Types / Colors).

Copy / *Ctrl+C*

Copies the selected text to the clipboard.

Cut / *Ctrl+X*

Deletes the selected text and saves it on the clipboard.

To paste the cut-out selection at another location in the file, place the cursor there and choose the command *Paste*.

Paste / *Ctrl+V*

Inserts the clipboard contents - after *Copy* or *Cut* - at the current cursor position.

Select All / *Ctrl+A*

Selects entire file content.

Undo / *Ctrl+Z*

Undo the last edit operation.

Redo / *Ctrl+Y*

Redo the last undo operation. You can redo all undo actions performed since the file was last opened.
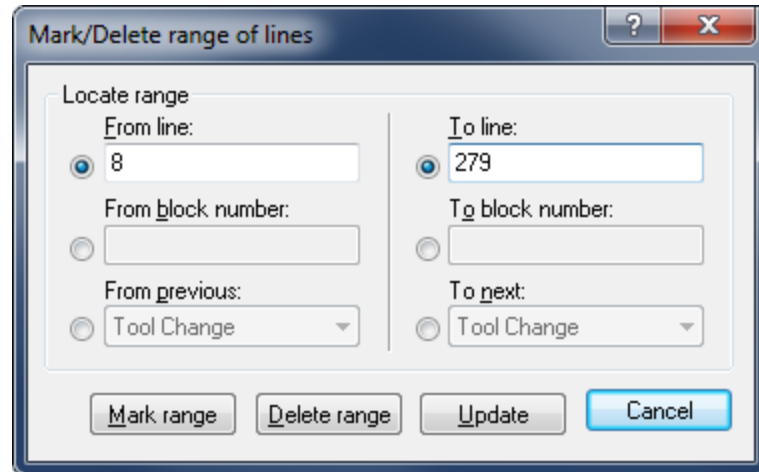
Del / *Del*

Deletes the selected part of the CNC program, or the character to the right of the cursor when no selection is done.

Mark/Delete range / *Ctrl+M*

When selecting the **Mark/Delete range** option under the *Del* drop-down menu, the following dialog appears:

Mark/Delete dialog box.

Here, you can specify a range of lines or block numbers to be marked or deleted. In addition, a special selection from a tool change, feed rate or spindle speed value can be marked up to the next or deleted.

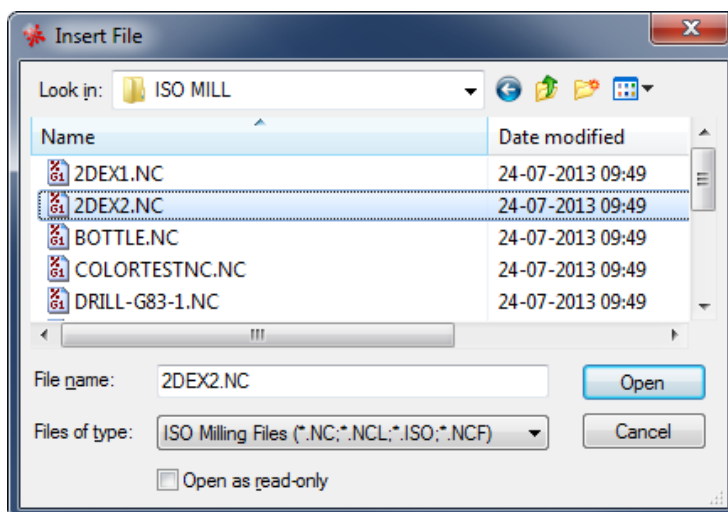> You can also access this function from the *Insert/Remove* menu of the NC Functions tab.

Append File

Appends a file to the end of the current CNC program.
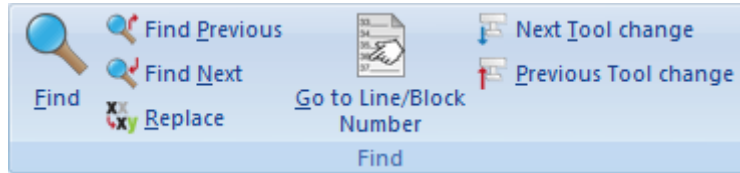
Insert File

Inserts a file into the current window at the cursor position.

Dialog Insert File.

1.4 Find

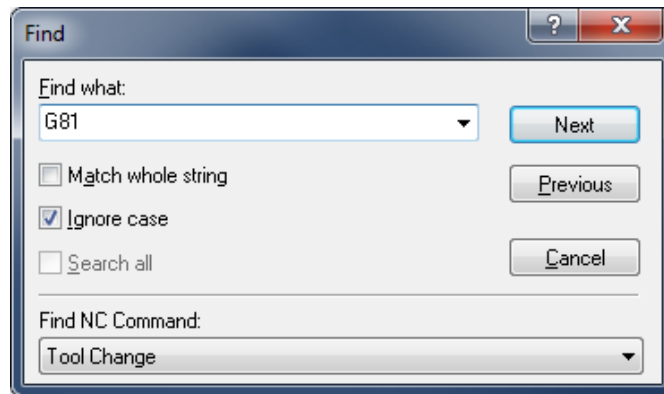This section describes the search for data through the file.

The Find menu.

🔍 Find / *Ctrl+F*

In the appearing dialog box, you can specify a text string and search for it through the CNC program. You can choose one of the defined tool operations instead of a text string.

Search dialog.

🔍 Find Previous / *Shift+F3*

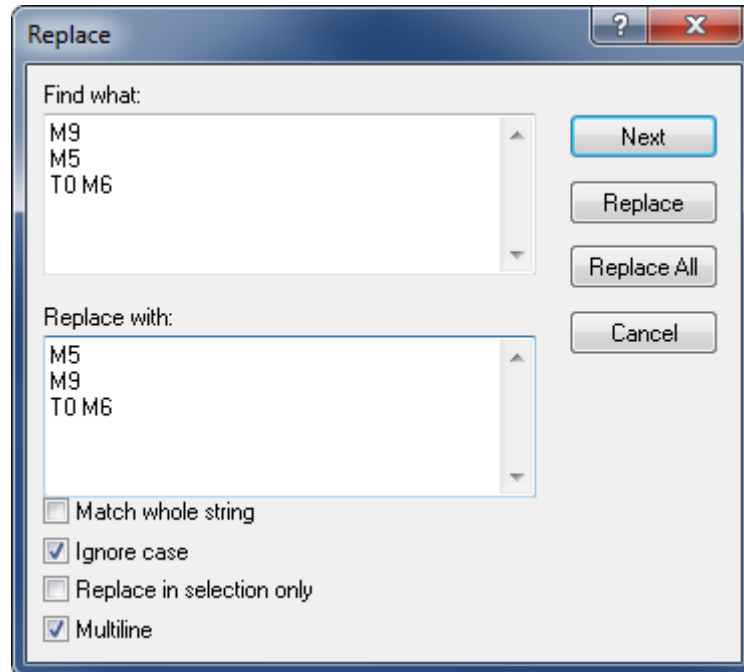Finds the previous occurrence of the specified search string in the CNC program.

🔍 Find Next / *F3*

Finds the next occurrence of the specified search string in the CNC program.

Replace / *Ctrl+H*

Replaces a specified string in the CNC program with another string.

Replace dialog with option *Multiline* enabled.

Go to Line/Block Number / *Ctrl+J*

Locate line or block number in current window.
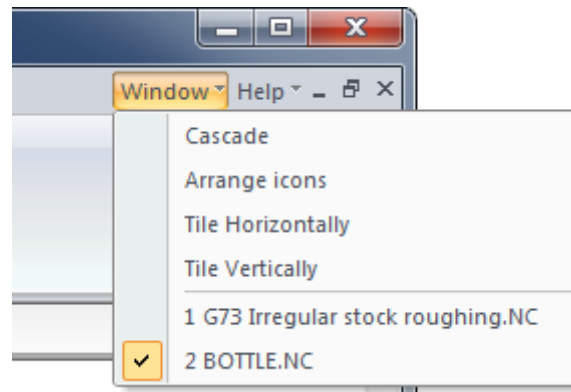
Next Tool Change

Goes to next tool change.

Previous Tool Change

Goes to previous tool change.

1.5 Window

This section describes how the sub-windows containing the CNC programs can be arranged.

Click on **Window** on the top right corner of the Ribbon. This will open a drop-down menu with four display modes of the programs.

The Window menu.

Cascade

Places the open files in an overlapping cascade in the window.

Arrange icons

Assembles the icons of the minimized CNC programs in the lower left corner of the window.

Tile Horizontally

Places the open files in horizontal editor windows in the program window.

Tile Vertically

Places the open files in vertical editor windows from left to right.

The names of the open files are listed in the lower part of the drop-down menu. When you click on a file to select it, the file will be marked with a check mark at the beginning of the line. The top of the dialog box is highlighted and the corresponding tab is activated in the editor window. This allows you to see, which window you have activated.

1.6 Help

If you need help with CIMCO Edit v7, click on the **Help** menu on the top right corner of the Ribbon and you receive the following selection:
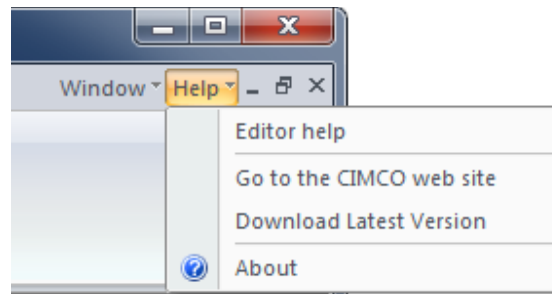
The Help menu.

The help system of CIMCO Edit v7 is explained in Using Editor Help.

Editor help

Opens the help file for CIMCO Edit.

> If the help file is not available in your language, it will open the default English help file.

Go to the CIMCO website

Takes you directly to CIMCO website if your computer is connected to the Internet.

Download Latest Version

Click this option to download the latest version of the editor CIMCO Edit.

> This command will not be available if option *Hide "Download Latest Version"* is checked in the general program settings.

About

Shows information about CIMCO Edit v7 and the license conditions.

2. NC Functions

This section describes the **NC functions** in the CIMCO Edit Ribbon. You can access the configuration of these functions using the icon ⊡. With the NC functions, CNC programs can be built or changed quickly.

2.1 Block Numbers

This section describes the functions concerning the appearance of the CNC program. For instance the commands and the block numbers can be changed.
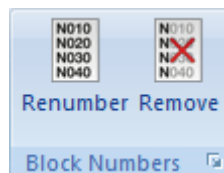
The Block Numbers menu.

Renumber / *Ctrl+T*

Inserts block numbers in the CNC program (or renumber the block numbers if some already exist in the CNC program).

Remove

Deletes all block numbers from the CNC program.

Configure Block Numbering

Opens the Block numbers configuration dialog. Please refer to Setup Block Numbers for more details.

2.2 Insert / Remove

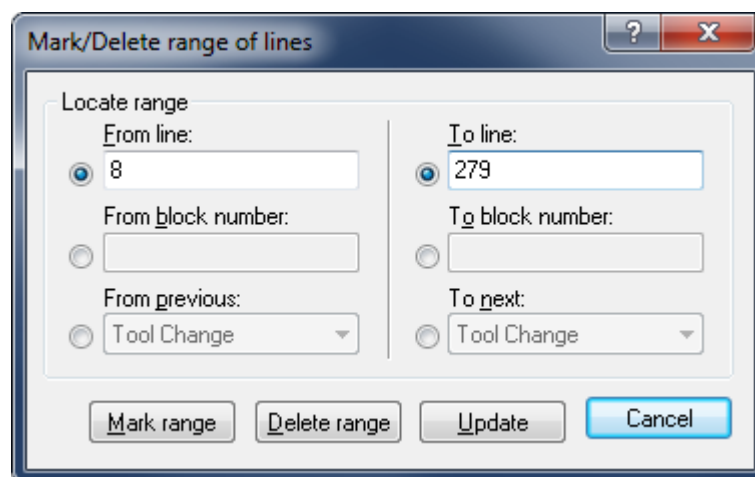This section describes the menu **Insert / Remove**. These functions allow you to quickly write and edit NC programs.

The Insert/Remove menu.

Mark/Delete Range / *Ctrl+M*

When clicking on **Mark/Delete Range** in the *Insert/Remove* tab, the following dialog appears:

Mark/Delete dialog box.

Here, you can specify a range of lines or block numbers to be marked or deleted. In addition, a selection from a tool change, feed rate or spindle speed value to the next value can be marked up or deleted.

Block Skips

Adds a skip-character to the beginning of a NC block. For this purpose, select the desired NC blocks and then click on **Block Skips** .

The block skip character can be defined in the Machine configuration dialog.

Remove Block Skips

All the block skip characters will be removed from the current CNC program.

> If you only want to remove the characters of a given NC block, select this NC block (completely or at any point) and click on the icon **Remove Block Skips** ✕.

Insert Spaces

Inserts space characters between the commands in the CNC program, where there is no space character between two commands. Existing space characters are left as they are.

Remove Spaces

Deletes all the space characters and tab signs from the CNC program (whitespace characters).

Remove Empty Lines
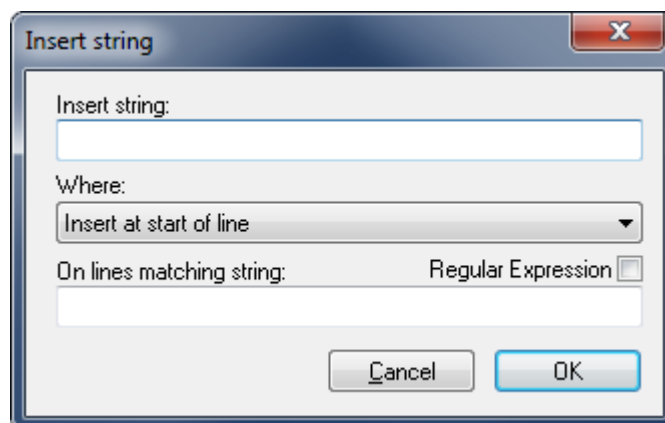
Removes all the empty lines from the NC program.

Remove Comments

Removes all the comments from the CN program - written in parentheses or with the *Comment start/end* characters defined in Machine.

Insert Strings

Inserts a user-defined string in the CNC program at the place defined in the dialog below.

Insert string dialog.

Remove Strings

Removes strings from the NC program according to the specifications entered in the *Remove String* dialog (similar to the one above).

DPR **Insert Monitoring Macros**

Inserts monitoring macros in the NC program.

When you click on this icon, the commands and values for program start, feedrate (F), spindle speed (S), and tool change (T) are found in the NC program and inserted into lines starting with **DPRNT ...** right after the individual command.

```
(AUßENKONTUR FLANSCH 123.50.12)
(AUSFÜHRUNG 2)
%
O1010
(ERSTELLUNGSDATUM:02.08.2005 16:15:34)
(PROGRAMMIERER:DK)
N0100 G00 G40 G49 G80 G90
N0110 T10 M6
N0120 S4000 M03
N0130 G00 X0.0 Y0.0
N0140 G00 Z100.0 G43 H12
N0150 G0 X-67.500 Y60.000
N0160 G00 Z2.0
N0170 G1 X-67.500 Y60.000 Z-5.000 F150 M08
N0180 G1 G41 X-67.500 Y55.000 F300
N0190 G3 X-62.500 Y50.000 I-62.500 J55.000
```

NC program without monitoring macros.

```
(AUßENKONTUR FLANSCH 123.50.12)
(AUSFÜHRUNG 2)
%
O1010
POPEN
DPRNT [MACO1010]
(ERSTELLUNGSDATUM:02.08.2005 16:15:34)
(PROGRAMMIERER:DK)
N0100 G00 G40 G49 G80 G90
N0110 T10 M6
DPRNT [MACT10]
N0120 S4000 M03
DPRNT [MACS4000.0]
N0130 G00 X0.0 Y0.0
N0140 G00 Z100.0 G43 H12
N0150 G0 X-67.500 Y60.000
N0160 G00 Z2.0
N0170 G1 X-67.500 Y60.000 Z-5.000 F150 M08
DPRNT [MACF150.0]
N0180 G1 G41 X-67.500 Y55.000 F300
DPRNT [MACF300.0]
N0190 G3 X-62.500 Y50.000 I-62.500 J55.000
```

CNC program with monitoring macros.

> (i) The observed values for the macros can be read via the serial interface for further processing.

✕ Remove Monitoring Macros

Removes all DPRNT lines that were added using the *Insert Monitoring Macros* function and restores the NC program to its original contents.

_X Auto Insert Spaces

When this option is activated, a space is automatically inserted before each NC-word if the program is written "in a row" (without spaces).

XY Uppercase / *Ctrl+U*

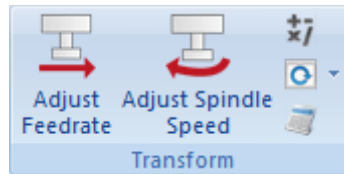Changes the case of the text to 'UPPERCASE' - including the comments.

xy Lowercase / *Ctrl+Shift+U*

Changes the case of the text to 'lowercase' - including the comments.

2.3 Transform

This section describes the functions under the menu **Transform**, which allows you to quickly write and change NC programs.
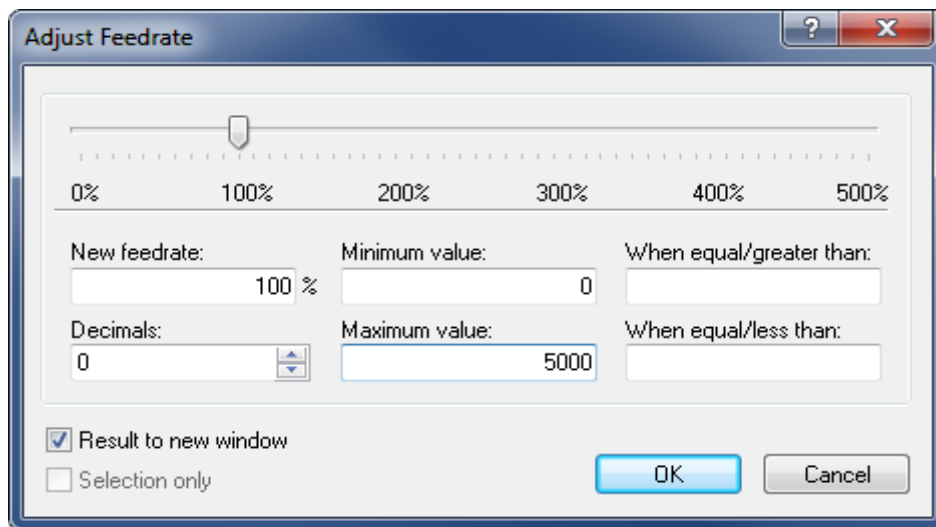

The Transform menu.

Adjust Feedrate

In this dialog, it is possible to adjust the feedrate of the active program. To do this, click and drag the pointer or click on the sliding bar of the pointer, *or* enter the new feedrate in the field **New feedrate**. Click **OK** to apply the command or **Cancel** to close the window without any changes taking effect.

Adjust feedrate dialog.

New feedrate

Enter into this field, the percentage with which the feedrate should be changed.

Minimum value

Enter the minimum feedrate into this field (in mm/min).

Maximum value

Enter the maximum feedrate into this field (in mm/min).

123 ⊟ Decimals

Enter the number of decimals to the right of the decimal point into this field.

abc When equal/greater than

Only values greater than or equal to the value specified here will be modified.

abc When equal/less than

Only values less than or equal to the value specified here will be modified.

☑ Result to new window

Check this field if the result of the changes made to the feedrate should be shown in a new window. If the *Selection only* option is checked, only the selection is put into the new window.

☑ Selection only
Check this field to change the feedrate in the selected program blocks only.

⊥ Adjust spindle speed

In this dialog, the spindle speed of the active program is adjusted. To do this, click and drag the pointer or click on the sliding bar of the pointer, *or* enter the new spindle speed in the field **New spindle speed**. Click **OK** to apply the command or **Cancel** to close the window without any changes taking effect.

Adjust spindle speed dialog.

abc   **New spindle speed**

Enter into this field, the percentage with which the spindle speed should be changed.

abc   **Minimum value**

Enter the minimum spindle speed here (in rev/min).

abc   **Maximum value**

Enter the maximum spindle speed here (in rev/min).

123   **Decimals**

Enter the number of decimals to the right of the decimal point here.

abc   **When equal/greater than**

Only values greater than or equal to the value specified here will be modified.

abc   **When equal/less than**

Only values less than or equal to the value specified here will be modified.
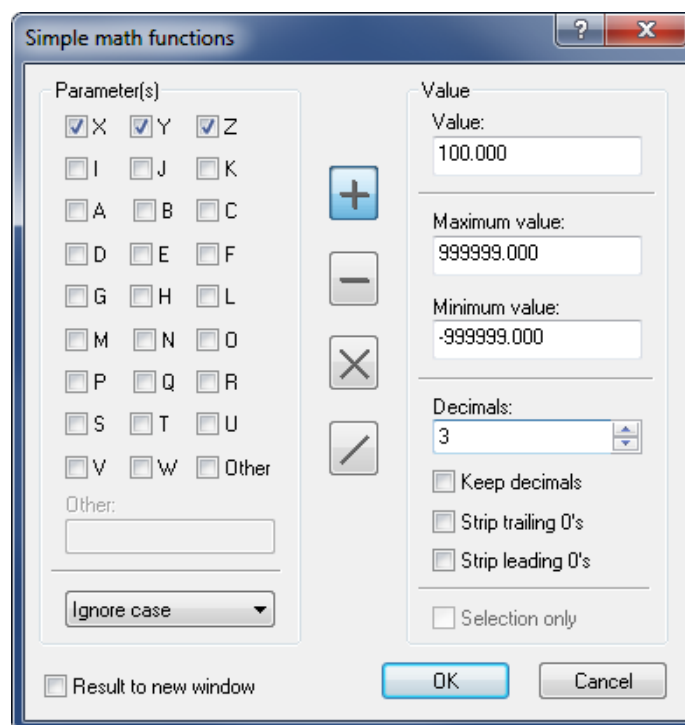
☑   **Result to new window**

Check this field if the result of the spindle speed changes should be shown in a new window. If the *Selection only* option is checked, only the selection is put into the new window.

☑   **Selection only**

Check this field to change the spindle speed in the selected program blocks only.

🗘 Simple math functions

Performs simple mathematical functions on the chosen NC file parameters.

Simple math functions dialog.

☑ Parameter(s)

Check the fields of the parameters to be changed.

abc Other

Check this option to enter parameters with more than one character. This enables an input field where special characters (Ó, Ö, Ø...) or multi-digit (ABC) parameter names can be entered.

abc ▾ UPPER/lower case

Use this field to specify UPPER and/or lower case of the parameters to be changed.

☑ Result to new window

Check this field if the result of the calculations should be shown in a new window. If the *Selection only* option is checked, only the selection is put into the new window.

Addition

Adds the specified value to all selected parameters.

Subtraction

Subtracts the specified value from all selected parameters.

Multiplication

Multiplies all selected parameters by the specified value.

Division

Divides all selected parameters by the specified value.

Value

Use this field to enter the value that will be inserted in the mathematical function (Addition / Subtraction / Multiplication / Division).

Maximum value

Enter the maximum value in this field. Parameters with values higher than specified here after performing the mathematical function will be set to this maximum value.

Minimum value

Enter the minimum value in this field. Parameters with values lower than specified here after performing the mathematical function will be set to this minimum value.

Decimals

In this field, enter the number of decimals to the right of the decimal point.

Keep decimals

Check this field to provide the new value with the same number of decimals as the original value.

Strip trailing 0's

Check this field to remove all zeros from the end of the number.

Strip leading 0's

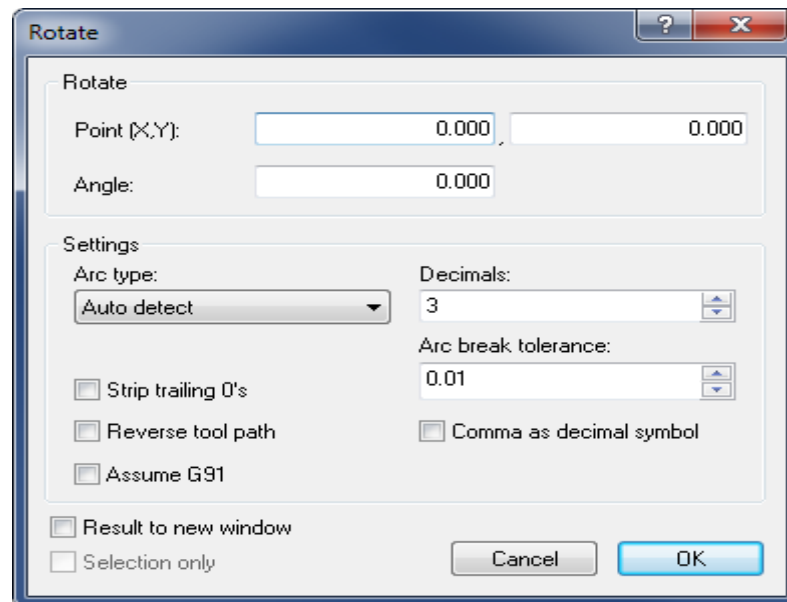Check this field to remove all zeros from the beginning of the number.

Selection only

Check this field to change the value of the selected program blocks only.

 Rotate

In this dialog you can rotate the outline around a specified point and with a specified angle. Click **OK** to apply the command or **Cancel** to close the window without any changes taking effect.

Rotate dialog.

 Point (X,Y)

Enter the coordinates of the center of the rotation in these fields.

 Angle

Enter the angle of rotation by degrees in this field.

 Arc type

Here you can specify the arc type by selecting one of the four different types:

- **Auto detect**: Default.
- **Relative to start**: Relative to the initial coordinates of the contour.
- **Relative to end**: Relative to the end coordinates of the contour.
- **Absolute arc center**: Relative to the absolute center of the arc.

 Decimals

Enter into this field the number of decimals to the right of the decimal point.

123 ⊟ **Arc break tolerance**

Use this field to specify the tolerance of each arc break when linearizing the tool path. A low value gives more line segments, while a high value gives less line segments.

☑ **Comma as decimal symbol**

Check this field to use comma as decimal separator.

☑ **Strip trailing 0's**

Check this field to remove all zeros from the end of the number.

☑ **Reverse tool path**

Check this field to reverse the trajectory of the tool path.

☑ **Assume G91**

Check this field when the control is preset for relative coordinates (G91).

☑ **Result to new window**

Check this field if the result of the settings should be shown in a new window. If the *Selection only* option is checked, only the selection is put into the new window.

☑ **Selection only**

Check this field to rotate the selected program blocks only.

Mirror

In this dialog you can mirror the outline around a specified line. Click **OK** to apply the command or **Cancel** to close the window without any changes taking effect.
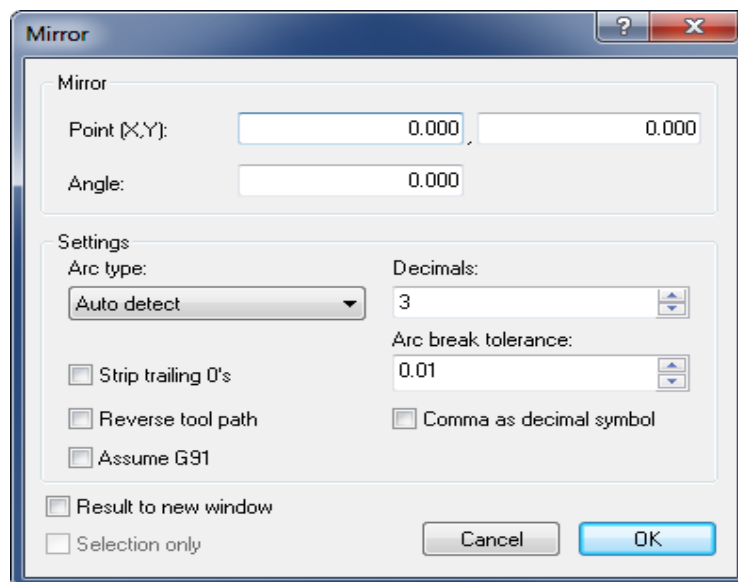
Mirror dialog.

Point (X,Y)

Enter into these fields the coordinates of the point of the mirror axis.

Angle

Enter into this field the angle of rotation by degrees of the mirror axis.

Arc type

Here you can specify the arc type by selecting one of the four different types:

- **Auto detect**: Default.
- **Relative to start**: Relative to the initial coordinates of the contour.
- **Relative to end**: Relative to the end coordinates of the contour.
- **Absolute arc center**: Relative to the absolute center of the arc.

Decimals

Enter into this field the number of decimals to the right of the decimal point.

Arc break tolerance

Use this field to specify the tolerance of each arc break when linearizing the tool path. A low value gives more line segments, while a high value gives less line segments.

☑ **Comma as decimal symbol**

Check this field to use comma as decimal separator.

☑ **Strip trailing 0's**

Check this field to remove all zeros from the end of the number.

☑ **Reverse tool path**

Check this field to reverse the trajectory of the tool path.

☑ **Assume G91**

Check this field when the control is preset for relative coordinates (G91).

☑ **Result to new window**

Check this field if the result of the mirror settings should be shown in a new window. If the *Selection only* option is checked, only the selection is put into the new window.

☑ **Selection only**

Check this field to mirror the selected program blocks only.

Tool compensation

In this dialog you can define a toolpath compensation (offset) for the active program, from the programmed workpiece contour to the left (G41), or to the right (G42) - based on the direction of feed -, in order to compensate different tool sizes.
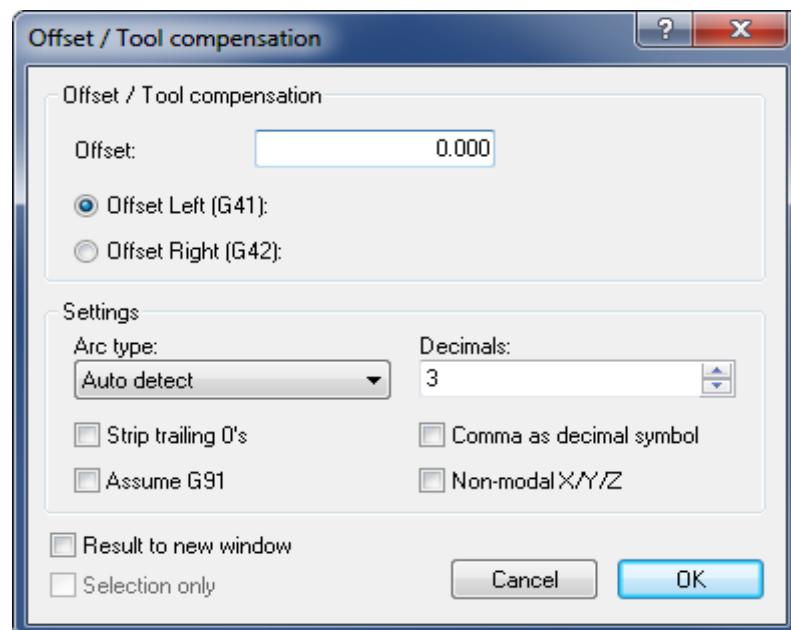
Offset/Tool compensation dialog.

abc **Offset**

Enter the offset value into this field.

◉ **Offset Left (G41)**

Enable compensation of the milling path, offset left (G41).

◉ **Offset Right (G42)**

Enable compensation of the milling path, offset right (G42).

abc ▾ **Arc type**

Here you can specify the arc type by selecting one of the four different types:

- **Auto detect**: Default.
- **Relative to start**: Relative to the initial coordinates of the contour.
- **Relative to end**: Relative to the end coordinates of the contour.
- **Absolute arc center**: Relative to the absolute center of the arc.

123 ⇕ **Decimals**

Enter into this field the number of decimals to the right of the decimal point.

☑ **Strip trailing 0's**

Check this field to remove all zeros from the end of the number.

☑ **Assume G91**

Check this field when the control is preset for relative coordinates (G91).

☑ **Comma as decimal symbol**

Check this field to use comma as decimal separator.

☑ **Non-modal X/Y/Z**

Check this field to use non-modal X/Y/Z values.

☑ **Result to new window**

Check this field if the result of the tool compensation settings should be shown in a new window. If the *Selection only* option is checked, only the selection is put into the new window.
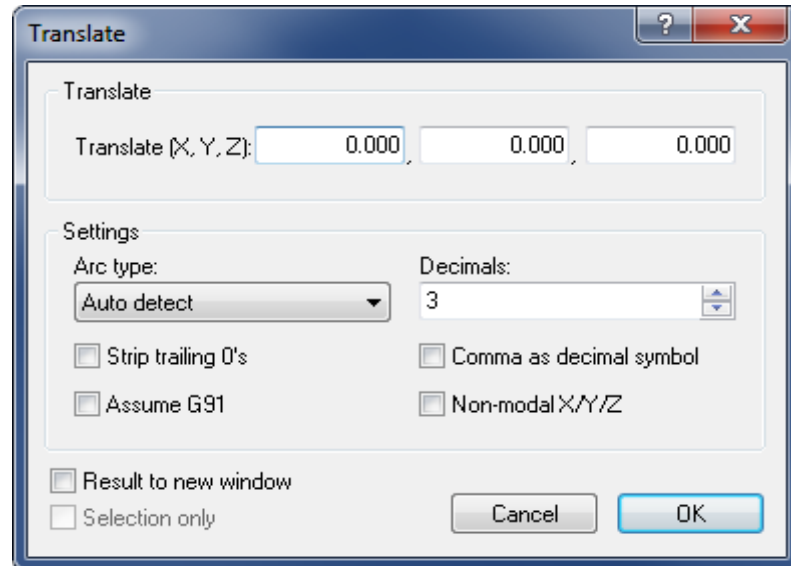
☑ **Selection only**

Check this field to offset the selected program blocks only.

🎨 **Translate**

This dialog translates the outline. The outline is moved to a position offset by the values entered in the **Translate (X,Y,Z)** fields.

Translate dialog.

abc **Translate (X,Y,Z)**

Enter into these fields the X, Y, and Z values to offset the outline.

abc ▾ **Arc type**

Here you can specify the arc type by selecting one of the four different types:

- **Auto detect**: Default.
- **Relative to start**: Relative to the initial coordinates of the contour.
- **Relative to end**: Relative to the end coordinates of the contour.
- **Absolute arc center**: Relative to the absolute center of the arc.

123 ⬍ **Decimals**

Enter into this field the number of decimals to the right of the decimal point.

☑ **Strip trailing 0's**

Check this field to remove all zeros from the end of the number.

☑ **Assume G91**

Check this field when the control is preset for relative coordinates (G91).

☑ **Comma as decimal symbol**

Check this field to use comma as decimal separator.

☑ **Non-modal X/Y/Z**

Check this field to use non-modal X/Y/Z values.

☑ **Result to new window**

Check this field if the result of the translate settings should be shown in a new window. If the *Selection only* option is checked, only the selection is put into the new window.

☑ **Selection only**

Check this field to translate the selected program blocks only.
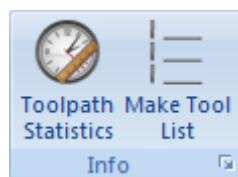
Heidenhain/ISO Converter

Use this function to convert Heidenhain CNC Programs to ISO Programs.

Show Windows Calculator

Click this option to open the Windows calculator.

2.4 Info

The menu **Info** contains functions for calculating Toolpaths Statistics and generating Tool List. You can access the configuration by using the icon ▣.

The Info menu.

Toolpath Statistics

Find range of X,Y,Z, machining time, tool change time, toolpath length, etc.

Make Tool List

Create a tool list for the current window.

Configure Tool List

Configure how the tool list is generated.

2.5 Macros

This section describes the function **Macros**. You can access the configuration of this function using the icon ⬛. CIMCO Edit v7 includes built-in cycles and macros for common operations like program start, program stop and tool change. You can also record or create custom cycles and macros for the operations most common to your own specific setups and applications.
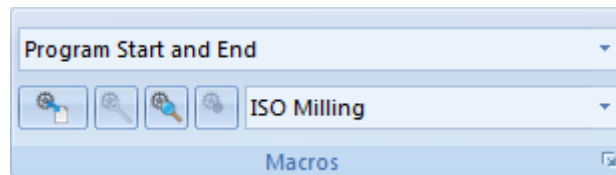
The Macros menu.

Macro Name

Shows the current macro for the selected file type. Click on the arrow and select a macro from the drop-down list.
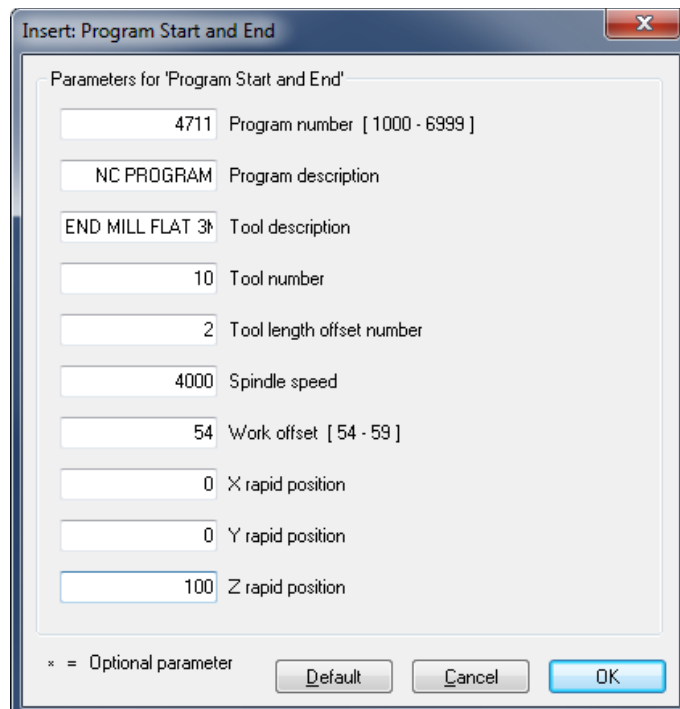
Insert Macro

Insert the selected macro.

Before inserting a macro, you must first define two types of parameters:

a) The required parameters, and b) The optional parameters (marked with *).

Click on the **Insert Macro** icon 🔧 to open the parameter input box. Enter the parameters and click OK.
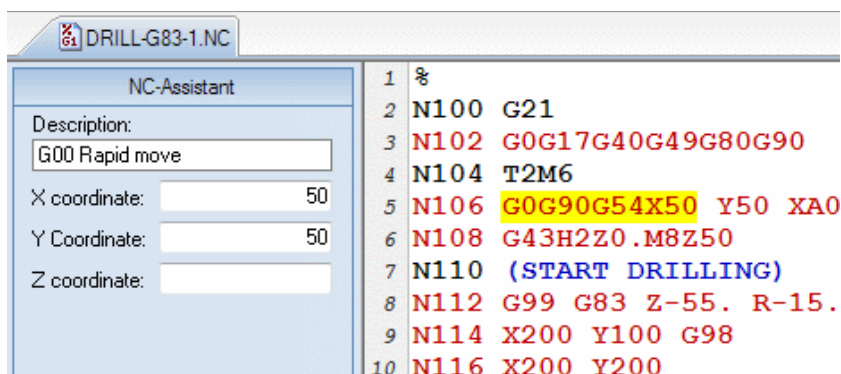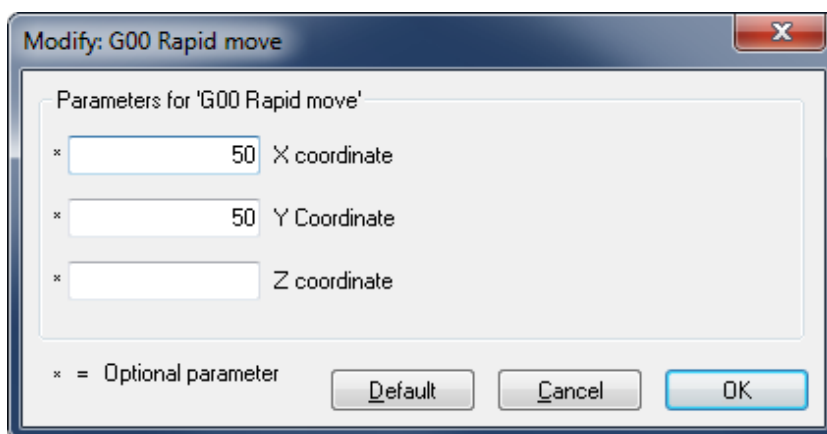

Program Start/End dialog.

Modify Macro

Modify the selected macro.

To modify an NC code in an inserted macro (e.g. change the Z value of the macro 'Program Start/Program End' from 100 to 80 mm), highlight the NC code and click on the **Modify Macro** icon . Enter the new Z value and then click **OK**.

NC Code selection in an inserted macro.

The Modify Macro dialog.

Find Macro

Click this icon to search for the specified macro.

You can search for a macro in an NC program in two ways:

- In the field *Cycles/Macros*, select the macro to search for in the NC program and click the button **Find Macro**.
- Select the macro to search for from the drop-down list and click on the icon *Find Macro*.

> If you receive the message: *Cannot find the macro ...* , it is because the desired macro was not specified as a macro. The NC text has been copied or created manually.

Record Macro

Creates a new macro based on the selected text.

To record a specific or frequently recurring NC block, select the block in the NC program and then click on **Record Macro**. The dialog *Add Macro* appears. Enter a name for your NC block (macro) and click **OK** to save it. The name is visible in the field *Macro name* of the *Macro Setup* dialog.

> If macro specific NC codes are variables, you must replace them with user-defined parameters in the *Macro Definition* field of the Macro Setup dialog.

File Type

Shows the current file type. Click on the arrow to select another file type.

Macro Setup

Here you can configure, add, and modify macros for the file types.

2.6 Macro Setup

CIMCO Edit v7 includes built-in cycles and macros for common operations like program start, program stop and tool change. You can create custom cycles and macros for the operations most common to your own specific setups and applications.
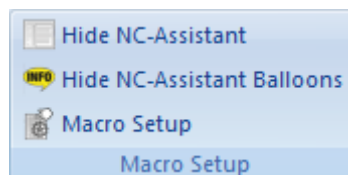
The Macro Setup menu.

Hide NC-Assistant / *Ctrl+Shift+A*
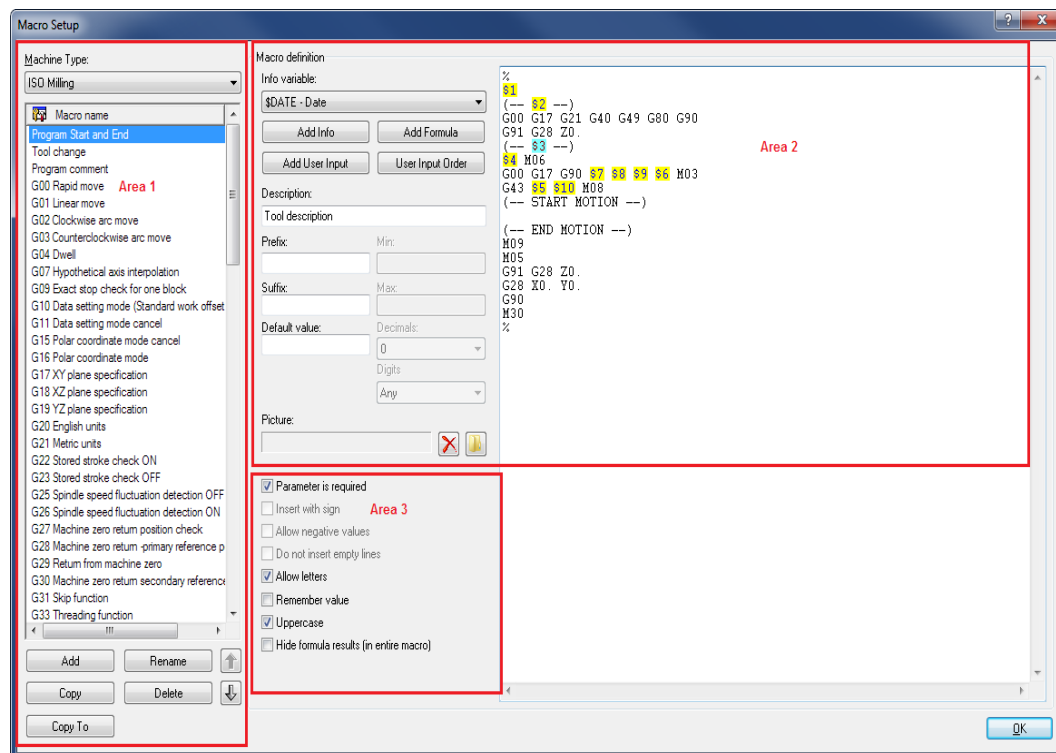
Click this icon to show or hide the NC-Assistant.

Hide NC-Assistant Balloons

Shows or hides the NC-Assistant Balloons.

Macro Setup

The dialogue Macro Setup is divided into three functional areas that are described in details in the following section.

Macro Setup dialog

Area 1: List of defined macros

Machine Type

This drop-down menu provides a list of defined machine types you can select from (e.g. ISO Milling, ISO Turning, Heidenhain, etc.).

Macro name

Here you can see all available macros for the selected machine type.

Clicking on the **Add** button will open the dialog *Add macro*. Enter the name of the new macro and click **OK**. You will find this name at the bottom of the *Macro name* list.

Add

Click this button to open the dialog *Add macro*. Enter the name of the new macro and click **OK**.

Button   **Rename**

Click this button to rename the selected macro.

Button   **Copy**

Click this button to make a copy of the selected macro. You can make a few changes in the name and the contents of the macro to create a similar macro and save it.

Button   **Delete**

Clicking this button will display a warning message asking if you really want to delete the selected macro. If you answer **Yes**, the macro name and its contents will be deleted.

Button   **Copy To**

Click this button to copy the selected macro to a different configuration.

Up / Down arrows

Use these buttons to move the selected macro up or down the list.

Area 2: Macro structure

abc   **Macro definition**

Use the field to the right to specify the contents of the macro.

A macro can contain both user and system variables. User variables are specified with a '$' character followed by a number, and system variables are specified with a '$' followed by the variable name.

In the field *Macro definition*, the macro block sequence is shown as it will appear in the NC program. NC code variables are replaced by parameters in ascending order. These parameters will be added to the macro program using the button **Add** (data input via the keyboard is not permitted and will not be accepted!). You can use the same variable multiple times in a macro.

To add a block number to a line, specify a '#' at the beginning of the line. If you need to insert a character right after a variable, you must insert a '|' (vertical bar) after the variable.

Example: To insert the user variable 2 followed by 100, specify $2|100'.

If this macro must start on a new line, make the first line of the macro definition blank.

The macro *Tool change* consists of 9 NC program blocks with 8 parameters ($1 ... $8). The block number format (e.g. N0100) and interval (e.g. 10) can be configured and modified in the dialog Setup Block Numbers.
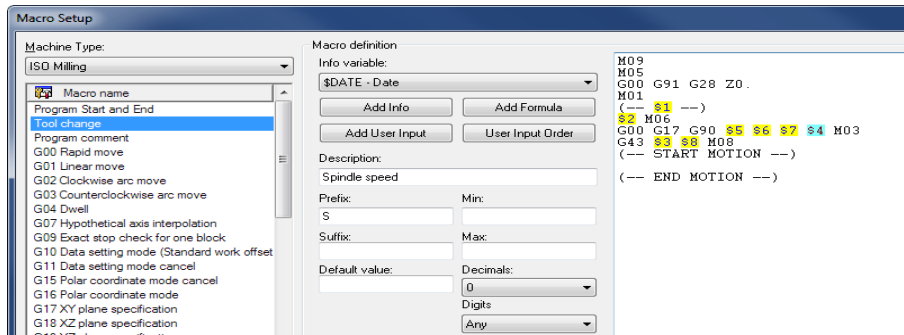
Example of *Macro definition* with 8 parameters.

Info variable

Use this drop-down menu to add additional information to the macro (for example: Date, time, date and time, filename, etc.).

Add Info

Click this button to insert a new information variable.

Add Formula

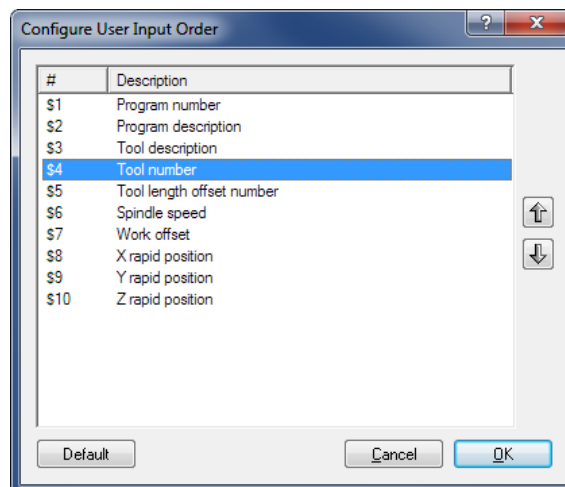Click this button to insert a new formula.

Add User Input

Click this button to insert a new user variable.

User Input Order

Click this button to arrange the order of user input variables when adding the macro. This does not change the order of the variables when the macro is inserted in the NC file.

When you click this button, the following window appears:

The User input order dialog.

Use the up/down arrows to change the order of the parameters.

[abc]   **Description**

Use this field to specify a description of the selected parameter.

> ⓘ   When you insert the macro into the CNC program by clicking the icon
> , a parameter input box appears to let you enter all macro
> parameters. The description of the parameter is right next to the
> parameter field.

Macro parameter input fields (example).

[abc]   **Prefix**

Use this field to specify a prefix string for the selected parameter, e.g. S for spindle speed.
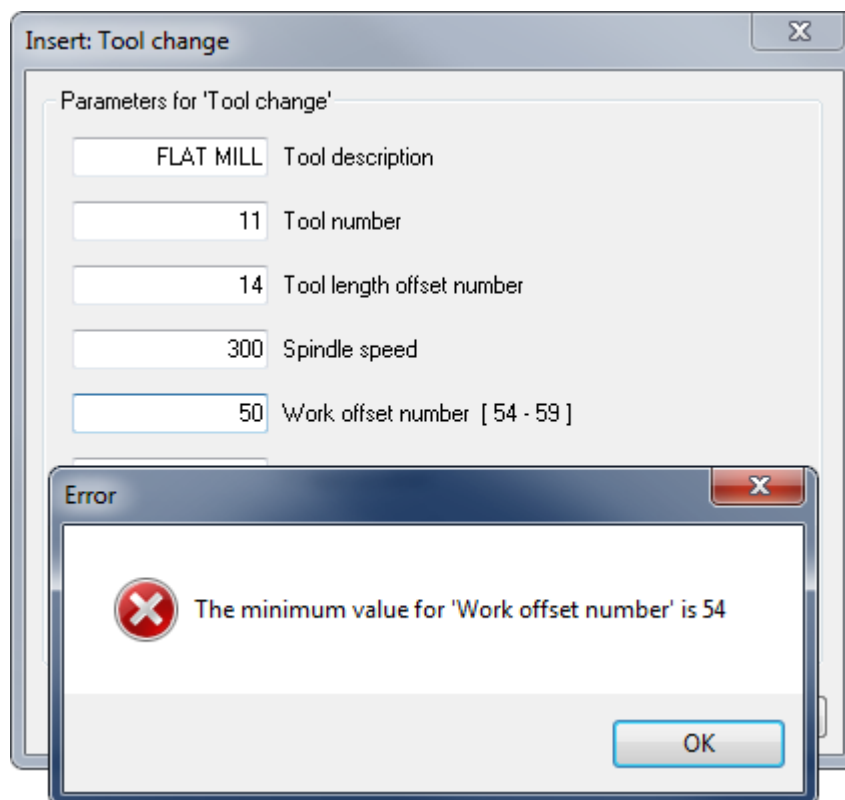
[abc]   **Suffix**

Use this field to specify a suffix string for the selected parameter.

abc **Min**

Use this field to specify a minimum value for the selected parameter. If you enter a value lower than the **Min** value, you will immediately be prompted for correction.

Error message for value lower than *Min*.

abc **Max**

Use this field to specify a maximum value for the selected parameter. If you enter a value higher than the **Max** value, you will immediately be prompted for change.

abc **Default value**

Use this field to specify a default value for the selected parameter.

abc ▾ **Decimals**

Use this field to specify the number of decimal digits.

abc ▾ **Digits**

Use this field to specify the number of digits. E.g. when inserting an O-number with 4 digits, the result becomes 'O0007'.

Area 3: Parameter monitoring / Additional options

Use the check boxes to enable or disable additional options.

> ⓘ The additional options must be selected carefully, as they affect the final result.

☑   **Parameter is required**

Check this field if the user is required to specify a value for the selected parameter.

If you do not check this box, the sign * will appear in the parameter input field, indicating that this parameter is *optional*. In this case, you do not need to enter a value in the field. If you do it anyway, you will change the (modal) value for this NC code.

Optional Parameters can for example be found at the 3 axes (X, Y, Z) of linear move (G01).

☑   **Insert with sign**

Check this field if the parameter value should be inserted with a sign.

Example: The value 123.4 will be inserted as +123.4.

☑   **Allow negative values**

Check this field to allow negative values for the current parameter.

If you leave this field unchecked, only positive values will be accepted!

> ⓘ If you (accidentally) enter a negative value into the macro fields, the (-) sign will be ignored. The entered value will be considered as the absolute value of the parameter and will output positive NC codes in the CNC program.

☑   **Do not insert empty lines**

Check this field if no blank line should be inserted when this parameter is not specified.
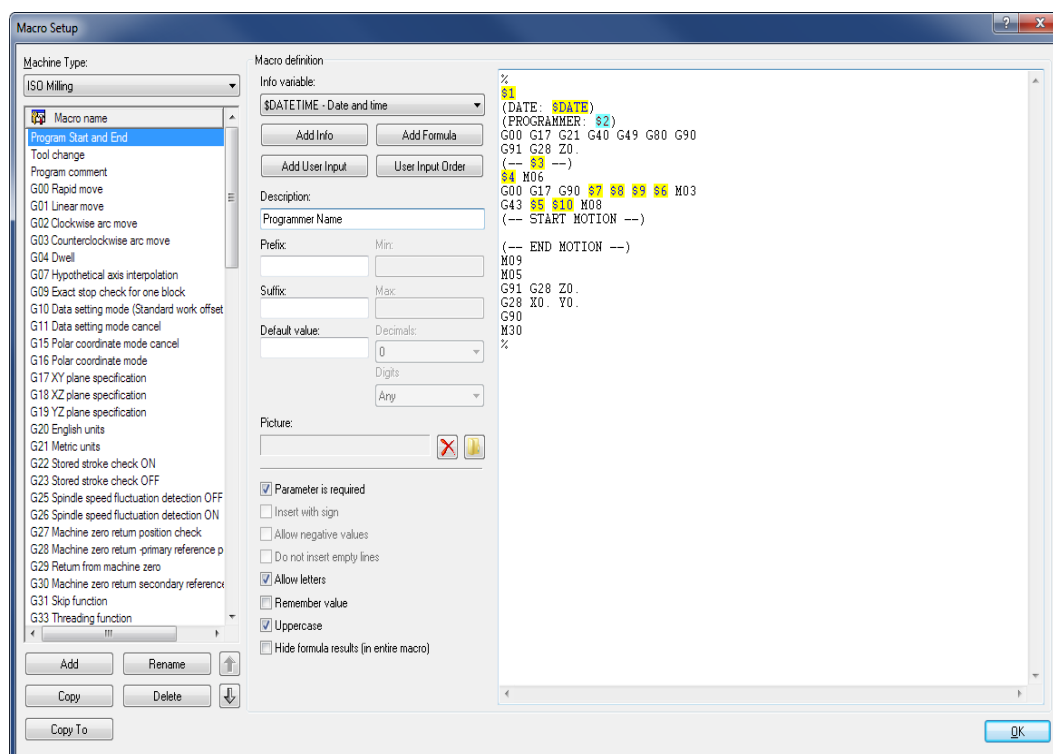
☑ **Allow letters**

Check this field to allow the user to enter text string.

According to ISO Convention, texts in the CNC program must be written in parentheses so they can be read on CNC machines. To prevent forgetting the brackets, these are generated automatically in CIMCO Edit v7 when the field Allow letters is checked. Besides, the input fields *Min, Max, Decimals, Digits* and the box *Allow negative values* are grayed out.

The parameter $2 in the macro *Program Start and End* can include programmers' initials, text strings, and special characters in parentheses.
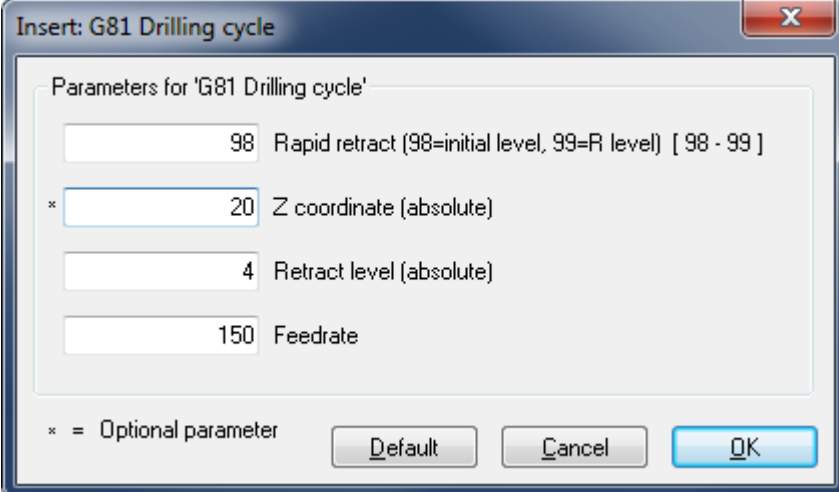

Allow letters.

☑ **Remember value**

Check this field if the last value entered for this parameter should be used the next time the macro is called.

Example: Let's assume you need to drill a blind hole into a special steel with three depths Z10, Z16 and Z20 at a defined location (X,Y). To achieve this, use the macro 'G81 Drilling cycle' and check the box *Remember value* for the parameters $1, $3, and $4. This way, those parameters will be automatically filled out with the previous settings for the blind hole machining. As we want to drill at different depths, we do not select *Remember value* for the parameter $2, which is the modal Z value of the operation.

Since *Remember value* is not activated, the field $2 will be marked with an * and is initially empty.

If the drilling program was previously run and you now call the macro 'G81 Drilling cycle' by clicking the icon , you just need to enter the first desired Z-value 10 into the field *Z coordinate (absolute)* and click on OK. Repeat the macro command with the Z-values 16 and 20, all other fields will be automatically filled out with the previously used values. Then you will have your blind hole.

Remember value in macro G81.

☑ **Uppercase**

If the option *Allow letters* is enabled and you want to write with capital letters, check this field to make the user input uppercase.

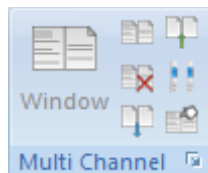> You must check this box if you want to use both lower and uppercase in the text.

☑ **Hide formula results (in entire macro)**

When this field is checked, the results of NC-Assistant macro formulas are hidden.

2.7 Multi Channel

This section describes the NC Function command **Multi Channel**. You can access the configuration of this function using the icon .

The Multi Channel menu.

Window

This option enables you to view the current file in multi channel mode.

Files

Click this icon to view multiple files in multi channel mode.

Close

Closes the multi channel view.

Next / *Ctrl+Down Arrow*

Jumps to the next synchronization point.

Previous / *Ctrl+Up Arrow*

Jumps to the previous synchronization point.

Synchronize Scrolling

Use this option to scroll all channels at the same time.

Setup Multi Channel

This option lets you configure the multi channel view.

3. Backplot

This section describes the **Backplot** functions in the Editor Ribbon.

The optional 3D/2D backplot in CIMCO Edit v7 is used for the simulation of CNC programs. The Backplot configuration is accessible through the icons 🖺or 🔲.

The graphical backplotter shows the toolpath for milling and turning operations. You can simulate NC programs for Milling operations using wireframe or solid view, as well as solid animation. For Turning operations, you can use wireframe simulation and solid animation.

The solid animation visualizes the manufacturing process of the piece. The simulation allows you to observe the stock being removed by the tool, and thus get a complete and intuitive visualization of the toolpath.

Within the Backplot tab, you can select either *Backplot Window* or *Solid Animation*. In the Backplot Window the toolpath is visualized as a wireframe, and a Solid Model of the workpiece can be created. In Solid Animation, the operations are visualized on a material block and show the manufacturing process. You cannot toggle between Backplot Window and Solid Animation during the dynamic simulation.

In Backplot mode, the NC program is displayed in the left pane of the window, while the plot is shown in the right pane. You can stop and restart the simulation, control the speed and direction of the tool movement, etc.

During the dynamic simulation, a gray bar shows in the CN program, which NC block is being processed. You can use Full Screen to make the graphical simulation cover the whole screen without showing the program. You can also skip through the NC codes with the options *Jump to next tool, Jump to next cutting pass* and *Jump to next move*.
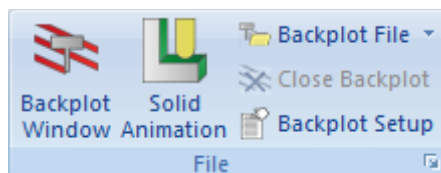
Dynamic zoom, pan, rotate and measuring functions can be used to analyze the plot.

> ℹ️ You need a Backplot enabled license key to use this function.

3.1 File

In this section, the backplot function **File** is described. The file menu contains the below described functions for file simulation and general backplot settings.

The File menu.

Backplot Window / *Ctrl+Shift+P*

Shows backplot of current window.

The window splits in two, allowing the lines of the program file and their toolpath simulation to be shown simultaneously.

The icon is active only when a program file is already open. Clicking this icon disables the *Solid Animation* icon.

Solid Animation

Select this option to show the solid animation of the current NC program.

The window splits in two, allowing the lines of the program file and the toolpath simulation to be shown simultaneously.

The icon is active only when a program file is already open. Clicking this icon disables the *Backplot Window* icon.

Backplot File

Enables you to backplot file directly from disk, without opening the file.

When you click this icon, a window opens to let you select a file to plot. A click on the down arrow next to the icon will display a list of recently simulated programs. In both cases, you get a complete plot of the selected program without showing the CNC program.

Close Backplot / *Ctrl+Shift+Q*

Closes the backplot window.

Backplot Setup

This option lets you configure the backplot.
Before starting the graphical simulation, you must first configure the file type for the program you have already opened. To do this, click on **Backplot Setup** to open the configuration dialog. In the Setup dialog, select the *Control Type* (for example 'ISO Milling') and configure the desired parameters. See Backplot Setting for more information.

3.2 View

This section describes the **View** menu of the optional 3D/2D Backplot command. These functions define how the current file is shown during the simulation.

> After selecting a view, you can restart the graphical tool path simulation.
>
> It is possible to change to a different view during the simulation.

The View menu.

Zoom In / *Mouse Wheel*

Click on this icon to enlarge the view by "zooming in" the entire plot for a detailed view.

Zoom Out / *Mouse Wheel*

Click on this icon to reduce the view by "zooming out".

Fit To Window / *Mouse Middle Button*

Click this icon to adjust the plot view to the size of the window.

Zoom Selection

Click this icon to zoom in on the selected area.

Zoom To Highlighted

Click on this icon to zoom to the highlighted element.

View Reset

Click this icon for the default perspective view (initial size and position).

View from Top (XY/G17)

Click this icon to view the simulation from the top.

View from Bottom (XY)

Click this icon to view the simulation from the bottom.

View from Front (XZ/G18)

Click this icon to view the simulation from the front.

View from Back (XZ)

Click on this icon to view the simulation from the back.

View from Left (YZ/G19)

Click on this icon to view the simulation from the left.

View from Right (YZ)

Click this icon to view the simulation from the right.

View from Tool

Click this icon to view the simulation from the tool.

Measure Distance

Click this icon to measure the distance between 2 points.

If you want to measure the distance between two points of the generated toolpath, click this icon to get an arrow with a ruler. Move the mouse to the first point and press the left mouse button to select it. Point 1 becomes black. Drag the mouse pointer to point 2 (or point 3, 4, 5, etc.) while holding the left mouse button down to obtain the absolute distance (in space) and the distance in the XY plane (G17) and the ZX plane (G18) between the two points.
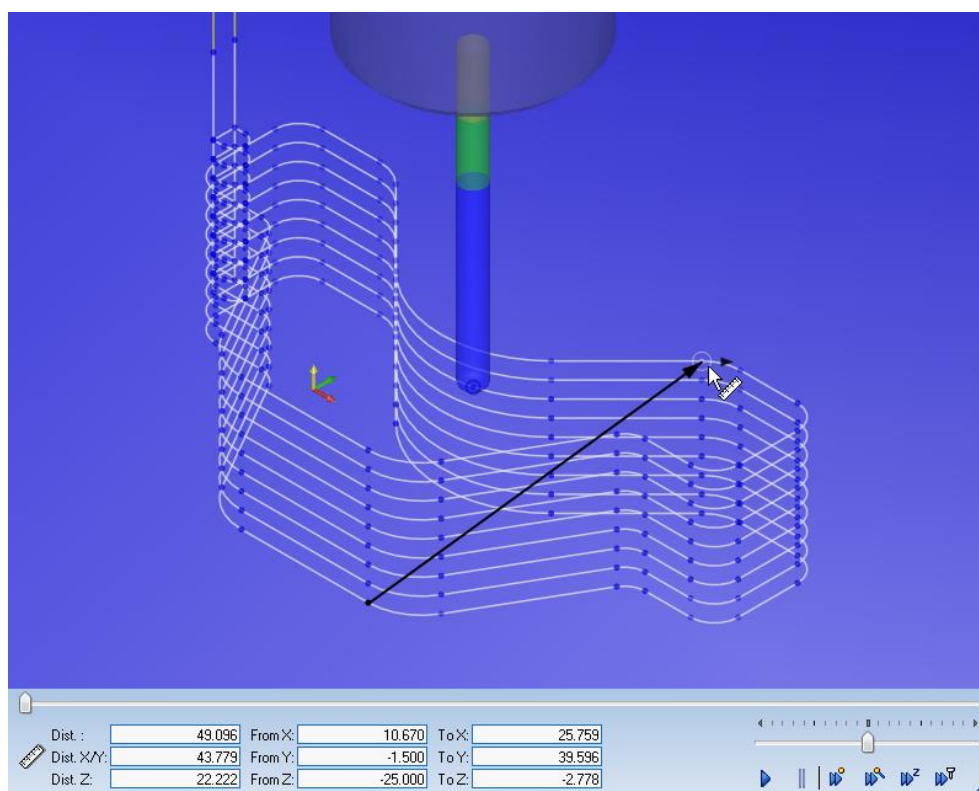
Measure distance.

Rotate View

Select this item to rotate the view.

Zoom View

Select this item to zoom the view.

Pan View

Select this item to pan the view.

Show Bounding Box

Click on this icon to show the toolpath bounding box.

Set Origin Location

Select this item to set the origin location.

Show Plot Control

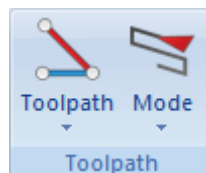Select this item to show/hide the plot control.

3.3 Toolpath

This section describes the **Toolpath** function in the Backplot menu. This is the representation of the current file relative to the toolpath in the simulation.

> After selecting a view, you can restart the graphical simulation of toolpath.
>
> It is possible to change to a different view during the simulation.

The Toolpath menu.
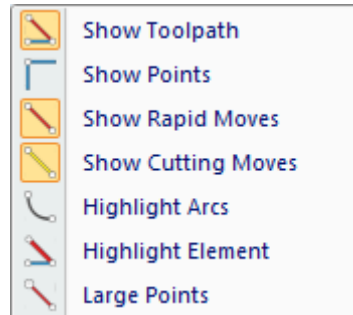
Toolpath

Show/hide the paths of the tool.

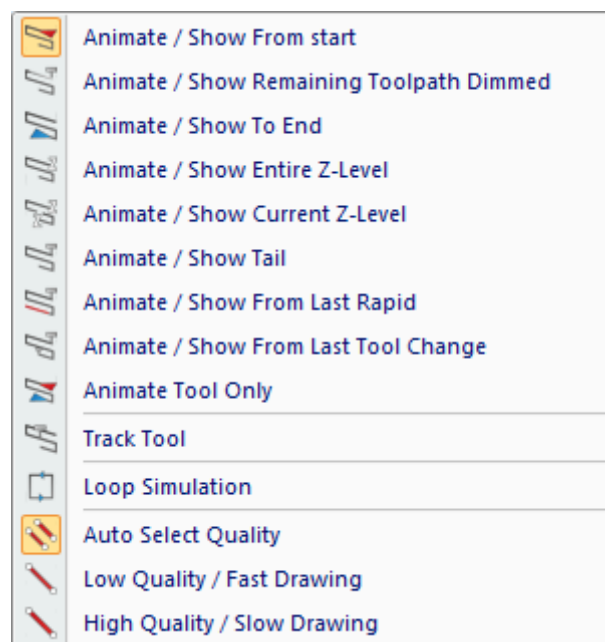Click the down arrow under this function to activate or deactivate toolpath options.

Show/hide toolpaths.

Mode

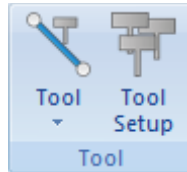Show toolpath from start until current position.

Click on the down arrow to open a drop-down menu, from which you can select options and additional functions for toolpath simulation.

Options for toolpath simulation.

3.4 Tool

This section describes the Backplot function **Tool**. The functions in this menu control the presentation of the tools in the simulation.

The Tool menu.

Show/Hide Tool

Select this item to show or hide the simulation tool.

Show Tool Holder

Select this item to show the tool holder.

Show Tool Transparent

Select this item to show the tool transparent.

Show Tool Colors

Select this item to show the tool colors.

Show Tool Vector

Select this item to show the tool vector.

Tool Setup

Click this icon to configure tool types, colors, and tool library.

Clicking on this icon will open the tool library. Here you can select a tool in the tool list of the bottom half of the window with one click and double-click to get the tool dimensions. You can also change one or more dimensions of the tool, enter a tool name and then assign it to the opened CNC program. Click OK to save tool information in the tool library.

In this dialog you can also load and save tool libraries. Tool libraries are especially useful on machines with fixed tool position.

- **Milling Tools**

    In milling operations, clicking the Tool Setup icon opens the following window:
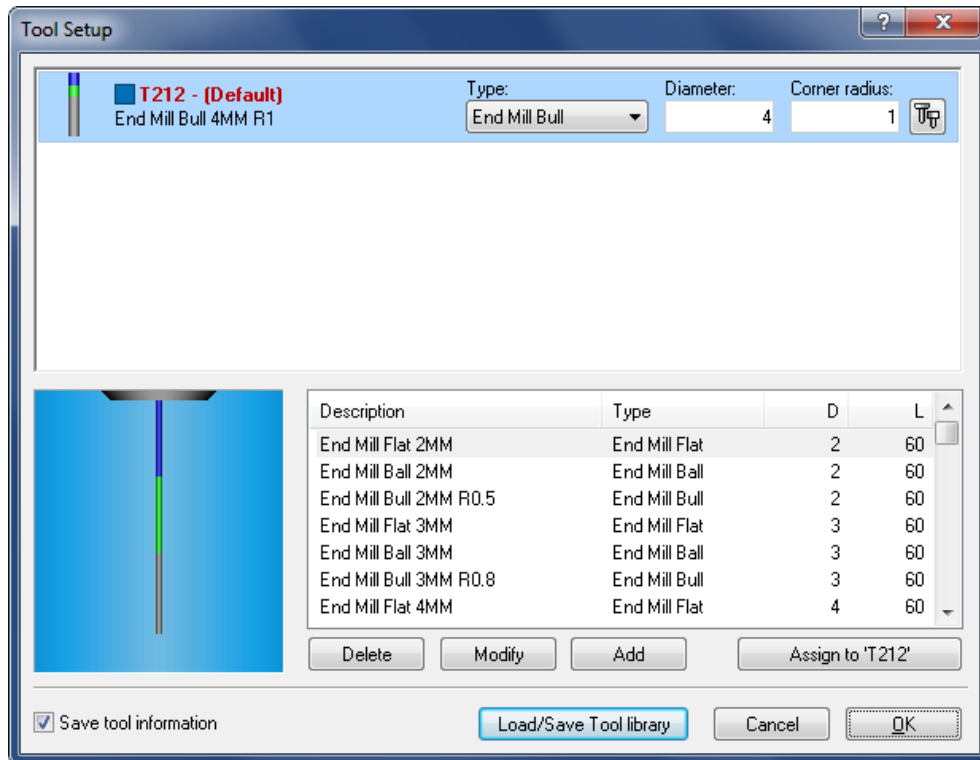
Milling tool library.

The Backplotter can automatically detect the tools specified in the NC program, or you can choose to configure them manually. The upper half of the window shows the tools currently used in the NC program. For every tool in the list, you can click on the icon at the top right of the **Setup Tool** window to open the configuration dialog of the selected tool. In this dialog, you can determine the dimensions of the tool and the color for the tool path.
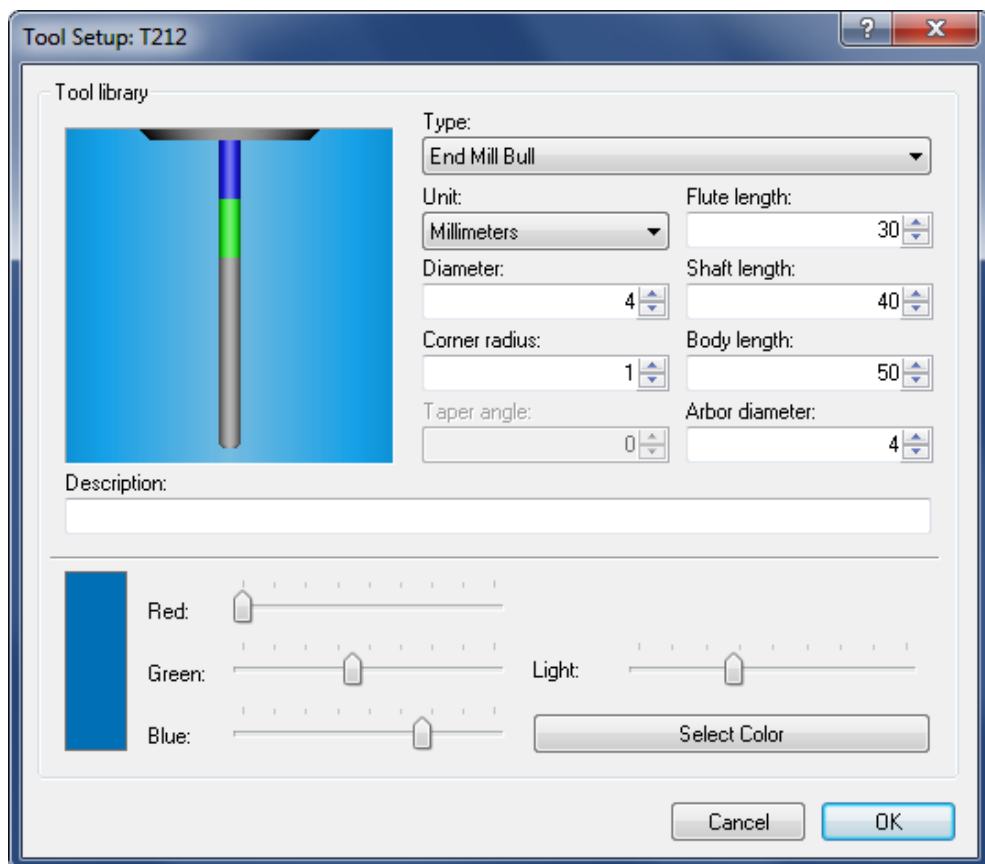
Tool setup and Toolpath color.

Enter the tool parameters and description, select a color for the tool path, and then click **OK**.

The tool path color cannot be set for Solid Animation.

- **Turning Tools**

  In turning operations, the following window opens when you click on the **Tool Setup** icon:
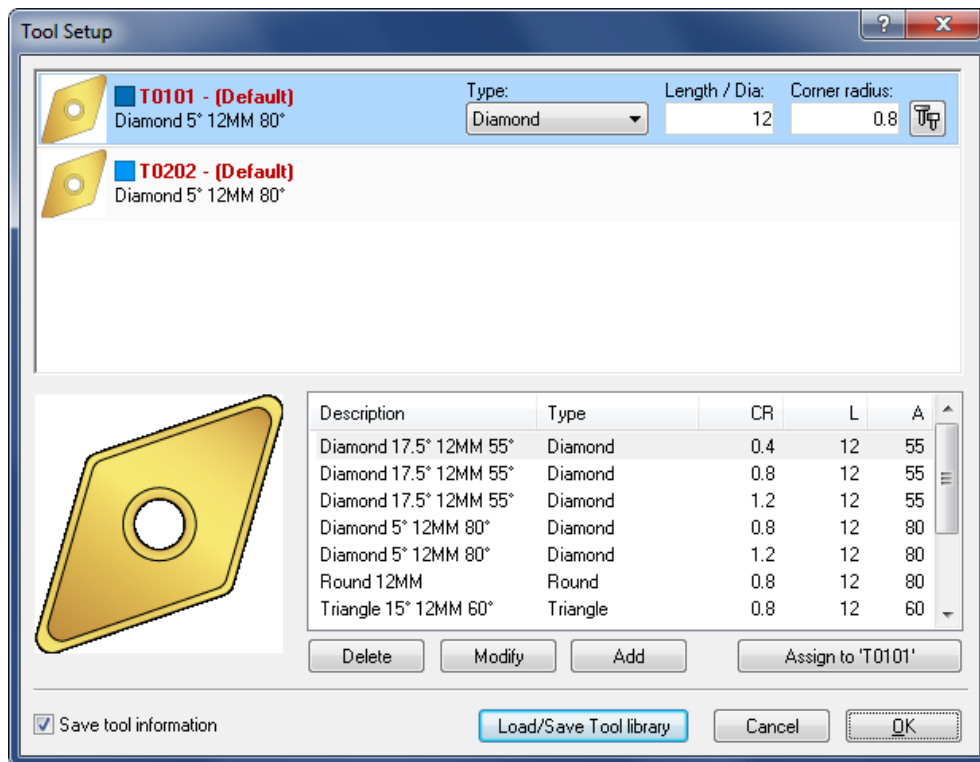
Turning tool library.

For every tool, you can click on the ⊡ icon at the right end of the Turning tool setup window to open the **Tool Setup** dialog of the selected tool.
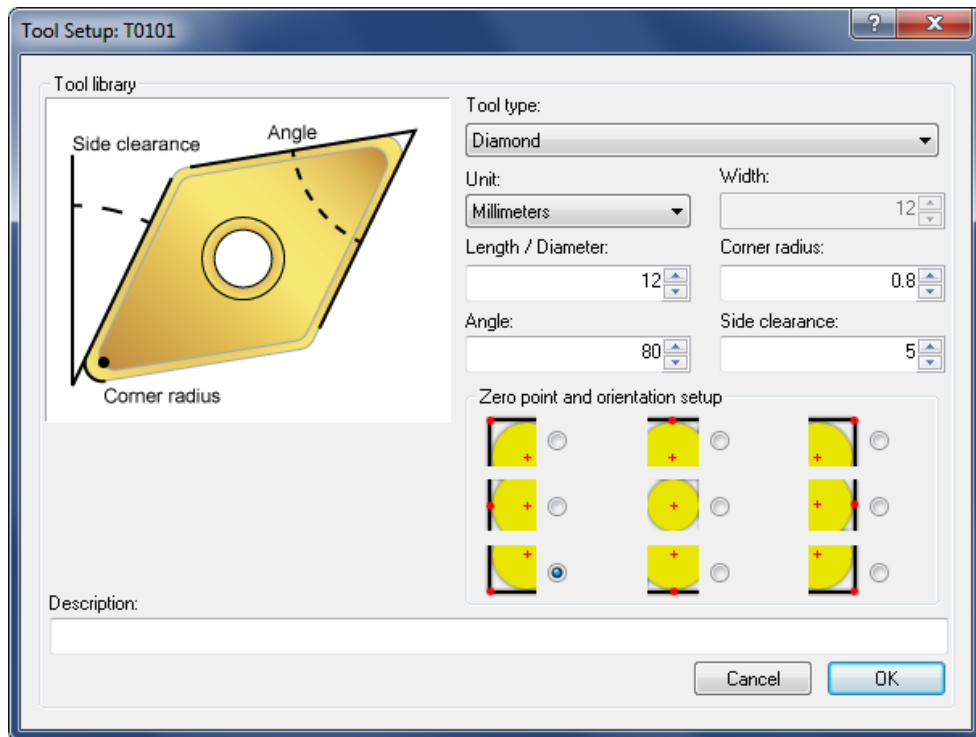
Turning tool setup dialog.

Enter the tool parameters and description, select the tool zero point and orientation, and then click **OK**.

When you click **Load/Save Tool library** in the tool setup window (both Milling and Turning), the following dialog appears:
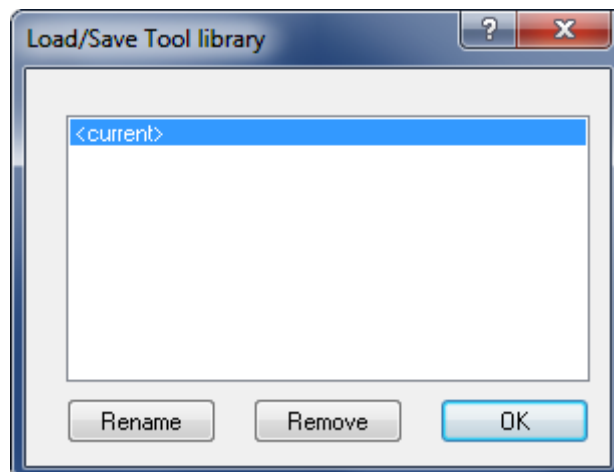
Load/Save Tool library.

Save the current tool library by clicking **Rename**, and enter a new name. You load a tool library by selecting the desired tool library and clicking **OK**. If you wish to remove a tool library, select the desired tool library and click **Remove**.

3.5 Solid

In this section, the Backplot function **Solid** and the simulation of the Solid Model are described.

> ⓘ The Solid Model is only available for Milling operations when using the *Backplot Window* option.

The Solid menu.

Solid Model

Shows / Hides the solid model.

Click on this icon to toggle between the toolpath view and the Solid visualization with toolpath.

Zoom / Regenerate solid

Creates a solid from the current view.

Click this icon to regenerate a solid model of the workpiece, based on the settings specified in the **Solid Setup**. You can select if the toolpath should be shown or hidden when a solid model is displayed.

When this option is enabled, the program will be checked for collision during dynamic simulation. The collision check options are activated in the **Solid Setup** dialog.
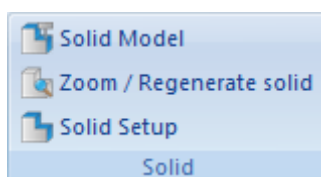
When a collision is detected, the simulation stops one block before the collision - both in the graphical simulation and the NC program.

Solid Setup for Milling operations

Click on this icon to configure the **Solid Model** and the options for the solid simulation. The specified stock dimensions are the minimum and maximum X, Y, Z values in your NC program. If you want to increase or decrease the stock dimensions to get a much more even toolpath, then click one or several times the corresponding button (+5% or -5%) to obtain the desired dimensions. Use the button **Round** to round off stock dimensions.
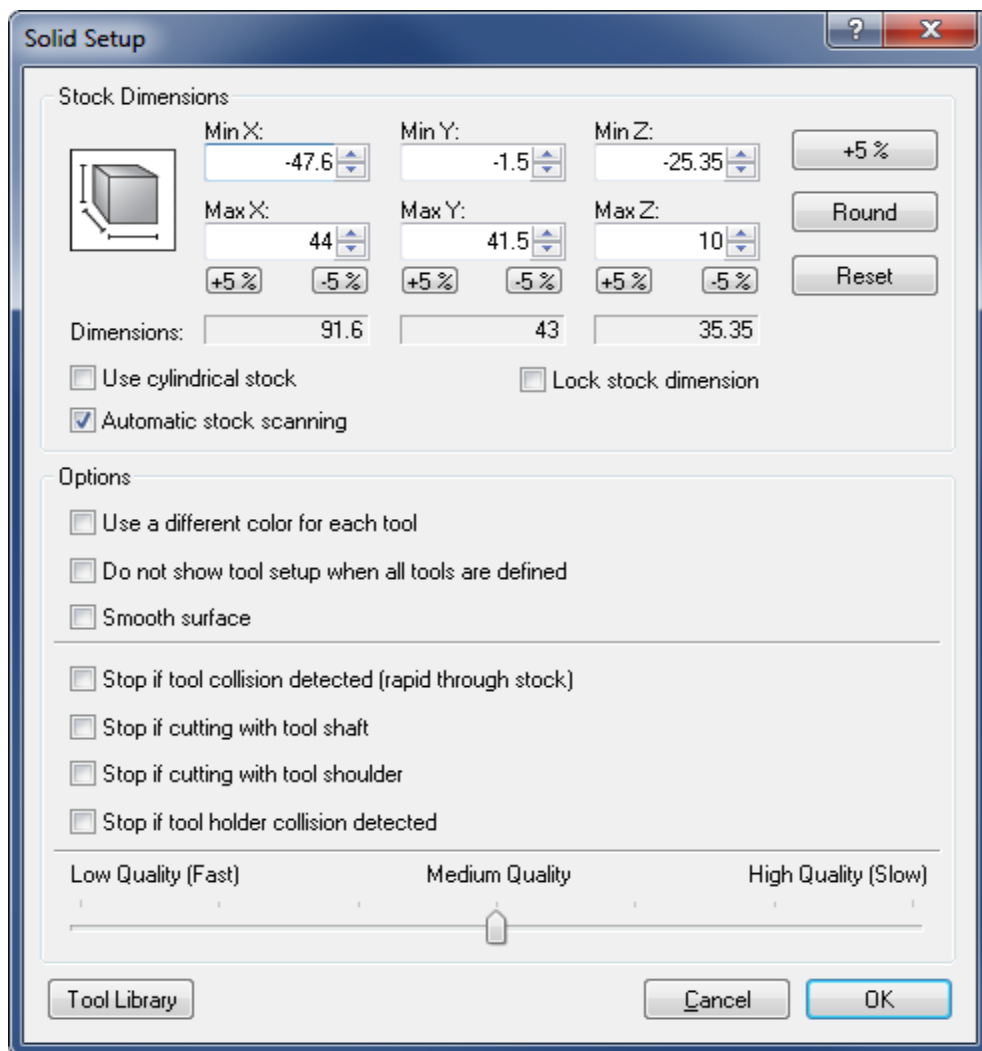
Solid Setup dialog for milling.

If the **Automatic stock scanning** option is enabled, stock dimensions will automatically be scanned from the NC program file. To specify stock dimensions within an NC program, add a comment with the following layouts:

For Heidenhain machines the line which defines the stock dimensions should have the following format:

BLK FORM 0.1 Z X-116.383 Y-50. Z-80.
BLK FORM 0.2 X+33.617 Y+50. Z+0

For ISO machines, insert the stock dimensions as comments within the first 100 lines of program, and with the following layout:

(STOCK Z X-25 Y-2 Z-35)
(STOCK X+25 Y+45 Z+0)

> Decimal numbers are supported. The +/- sign is required to be in front of each value.
>
> Input fields are disabled when Stock dimensions are found and Automatic stock scanning is enabled.

The **Lock stock dimension** option allows you to lock the stock dimensions to a machine. The stock values will be saved in the machine settings to make them available for used in other NC programs. This is especially useful when testing different NC programs on the machine.

Solid Setup for Turning operations

Click this icon to configure the **Solid Model** for the Solid Animation. The specified stock dimensions are the maximum values in your NC program, and the *Position on Z* is calculated on basis of these values. Use the button **Round** to round off stock dimensions, click **Reset** to revert to the stock original values.
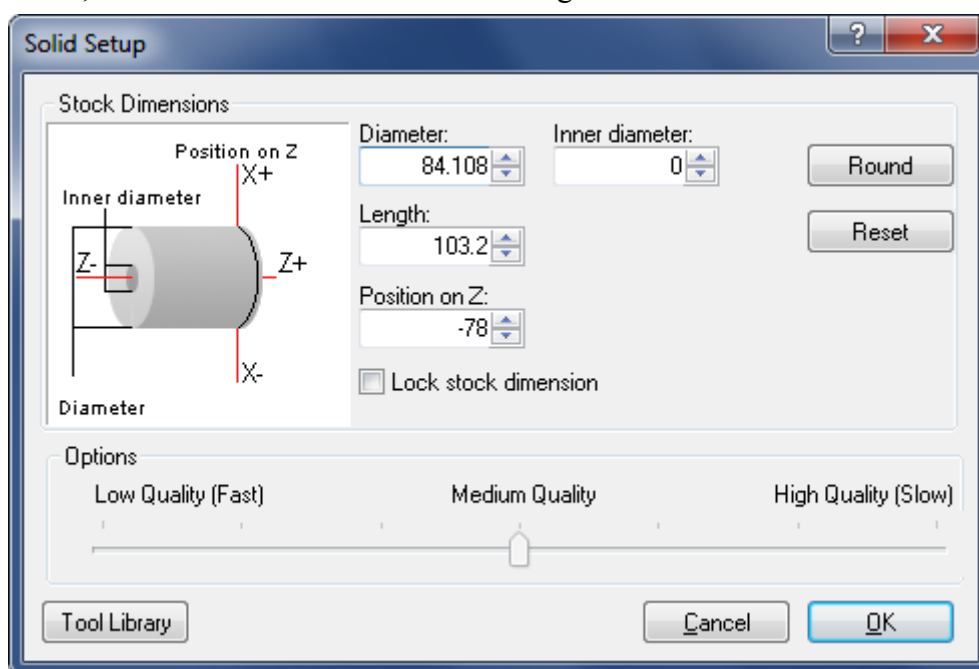
Solid Setup dialog for turning.

The **Lock stock dimension** option allows you to lock the stock dimensions to a machine. Checking this field saves the stock values in the machine settings, so they are available for used in other NC programs. This is especially useful when testing different NC programs on the machine.

> The stock dimensions are locked temporary and will not be available when CIMCO Edit v7 is restarted.

3.6 Other

This section describes the Backplot function **Other**. This function is concerned with simulation in full screen, workpiece and tool offsets, 5-axis machine setup, load of STL files, DXF files export, and the Simulation control type.
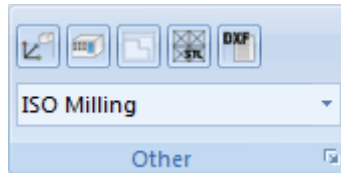
The Other menu.

Full Screen

Shows the backplot in full screen.

Set Workpiece/Tool Offsets

This option enables you to set workpiece and tool offsets.

When you click on this icon, the Workpiece/Tool Offset dialog appears for entering the X, Y, Z coordinates of Machine Offset and Workpiece Offset, as well as the Tool Length Compensation.

Workpiece and tool offsets.

5-Axis Machine Setup

This function lets you configure 5-axis machines.
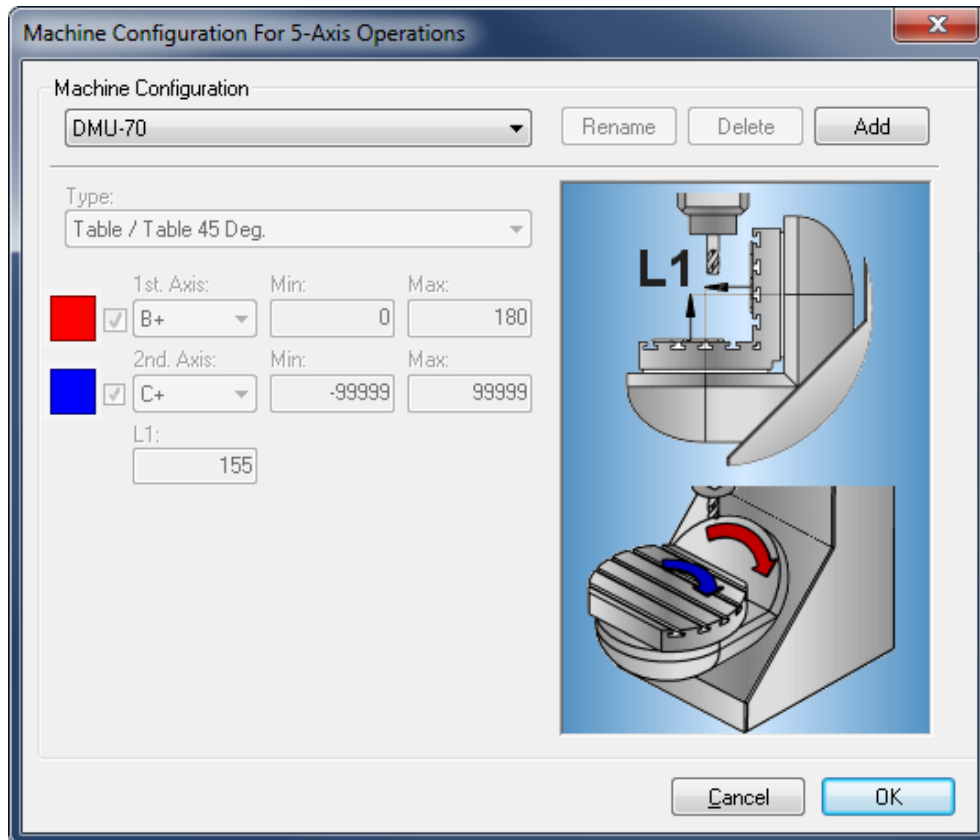
5-Axis Machine Setup dialog.

Load STL File

Loads an STL file (stereolithography).

Export as DXF File

Exports toolpath as a DXF file.

Control Type

Select the control type from this list.

Backplot Setup

This option lets you configure the backplot settings. Please refer to File Types Settings for further information.

3.7 Find

This section describes the search for data through the file.

The Find menu.

🔍 **Find /** *Ctrl+F*

Finds the specified text string (or one of the defined tool operations) in the CNC-program.

🔍 **Find Previous /** *Shift+F3*

Finds the previous occurrence of the specified search string in the CNC program.

🔍 **Find Next /** *F3*

Finds the next occurrence of the specified search string in the CNC program.

📑 **Go to Line/Block Number /** *Ctrl+J*

Locate line or block number in current window.

📑 **Next Tool Change**

Goes to next tool change.

📑 **Previous Tool Change**

Goes to previous tool change.

⊘ **Toolpath Statistics**

Find range of X,Y,Z, machining time, tool change time, toolpath length, etc.

3.8 Information Bar

This section describes the Backplot **Information Bar**. The Information Bar is used to control the simulation and show information about NC codes.

Backplot Information Bar (Milling)

The Information Bar is shown in the lower part of the Backplot window and contains the following elements:
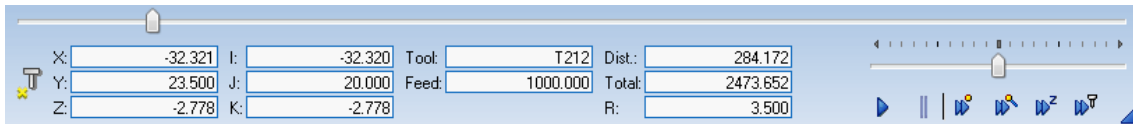
The Backplot Information Bar (Milling).

Dynamic Simulation Progress Bar (0 - 100%)

Speed and Direction Slider

From midpoint to the right = forward, slow ... fast.
From midpoint to the left = backward, slow ... fast.

Current Tool Coordinates

Shows the current tool coordinates within the defined preparatory functions (G00 - G03).
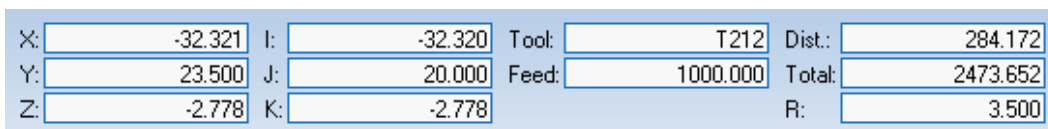
 Start / Stop simulation

 Pause simulation

 Jump to next move

 Jump to next cutting pass

 Jump to next Z-level

 Jump to next tool

◢   **Show / hide information bar**

Backplot Information Bar (Turning)

The Information Bar is shown in the lower part of the Backplot window and contains the following elements:

X:	2.870	I:	2.870	Tool:	T0101	Dist.:	1.931
Z:	-0.871	K:	-0.746	Feed:	0.014	Total:	16.746
						R:	0.125
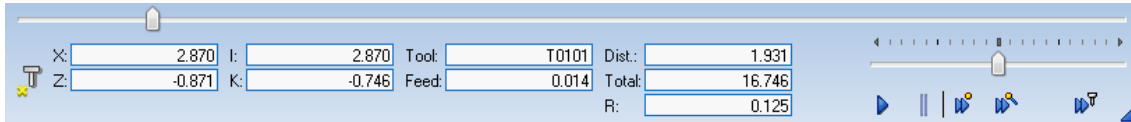
The Backplot Information Bar (Turning).

Dynamic Simulation Progress Bar (0 - 100%)

Speed and Direction Slider

From midpoint to the right = forward, slow ... fast.
From midpoint to the left = backward, slow ... fast.

Current Tool Coordinates

Shows the current tool coordinates within the defined preparatory functions (G00 - G03).

X:	2.870	I:	2.870	Tool:	T0101	Dist.:	1.931
Z:	-0.871	K:	-0.746	Feed:	0.014	Total:	16.746
						R:	0.125
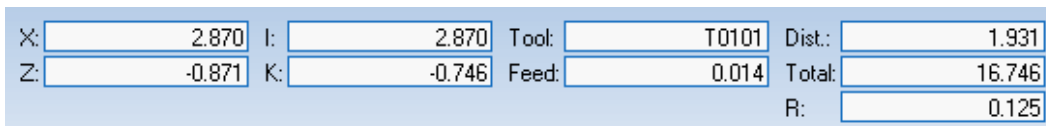
▶   **Start / Stop simulation**

‖   **Pause simulation**

⏭   **Jump to next move**

⏭   **Jump to next cutting pass**

⏭   **Jump to next tool**

◢   **Show / hide information bar**

4. File Compare

This section describes the **File Compare** menu in the CIMCO Edit Ribbon. You can access the configuration of the function using the icon ▣.

4.1 File Compare

In this section, the functions used to compare two files are described. You start a comparison of two files by choosing the files. The files are put into a double framed window, and you can start the search for differences. To exit the file compare window, use the **End File Compare** ▣ menu item.
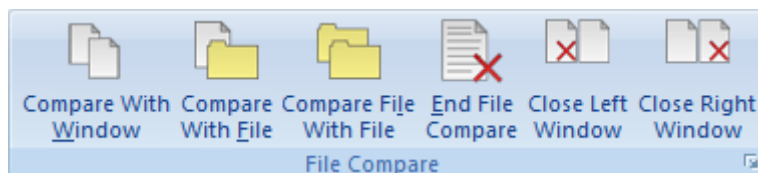
The File Compare menu.

Compare with Window

Compares the file in the current window with a file in another.

Compare with File

Compares the file in the current window with a file. The other file is opened through the *Open file* dialog.

Compare File with File

Compares two files which are not already in a window. The files are opened using the *Open file* dialog.

End File Compare

The file compare session is stopped and both files are left open.

Close Left Window

Closes left file compare window.

Close Right Window

Closes right file compare window.

Setup

Use this icon to set up file compare settings.

4.2 Go To

This section describes the functions used to move through the differences found during file compare.
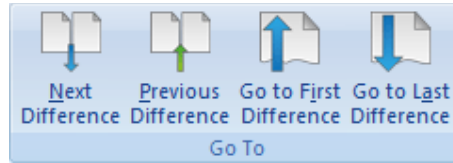
The Go To menu.

Next Difference / *Ctrl+Down Arrow*

The cursor jumps to the next difference.

Previous Difference / *Ctrl+Up Arrow*

The cursor jumps to the previous difference.

Go to First Difference

The cursor moves to the line containing the first difference.

Go to Last Difference

The cursor moves down to the line containing the last difference.

4.3 Sync

This section describes the features used to synchronize differences.

When you synchronize a difference, the selected difference is copied from one window to another. The icons described below determine the synchronization direction (from right to left or from left to right).

When the text is moved from one to another window, the line is no longer highlighted. This is because the texts in both windows are identical after synchronization.
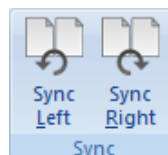
The Sync menu.

Sync Left / *Ctrl+Left Arrow*

Synchronizes current difference from right to left window.

Sync Right / *Ctrl+Right Arrow*

Synchronizes current difference from left to right window.

4.4 Other

This section describes other functions used for file compare.
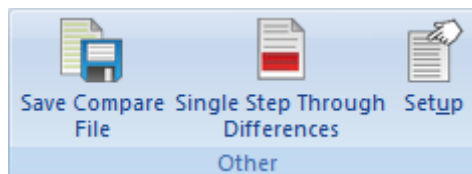
The Other menu.

Save Compare File

Saves the result of file compare into a file.

Single Step Through Differences

By activating this function, any change possibility is limited to the line containing the difference, i.e. the line where the cursor is positioned.

Example: If the function ▬ is enabled and the cursor is positioned on the second line of a three-line red block, then the text of this line can be synchronized from left to right or from right to left. The first and the third lines of the block are not affected by the action and continue to show the differences in red.

Click on the icon ▬ or open the *File compare settings* dialog to activate the **Single line step** function.

Setup

Configure the settings of the File Compare function. You can set up the different parameters for File Compare in the **File Compare Settings** dialog. The behavior of the File Compare functions is changed through the Machine Type settings.

If you want to change the configuration during a file compare operation, you must first end the current operation with the icon ✖, and then start a new operation with the icon ▯.

ℹ️ In the 'File Compare Settings' dialog below, different File compare options can be selected by checking the fields.
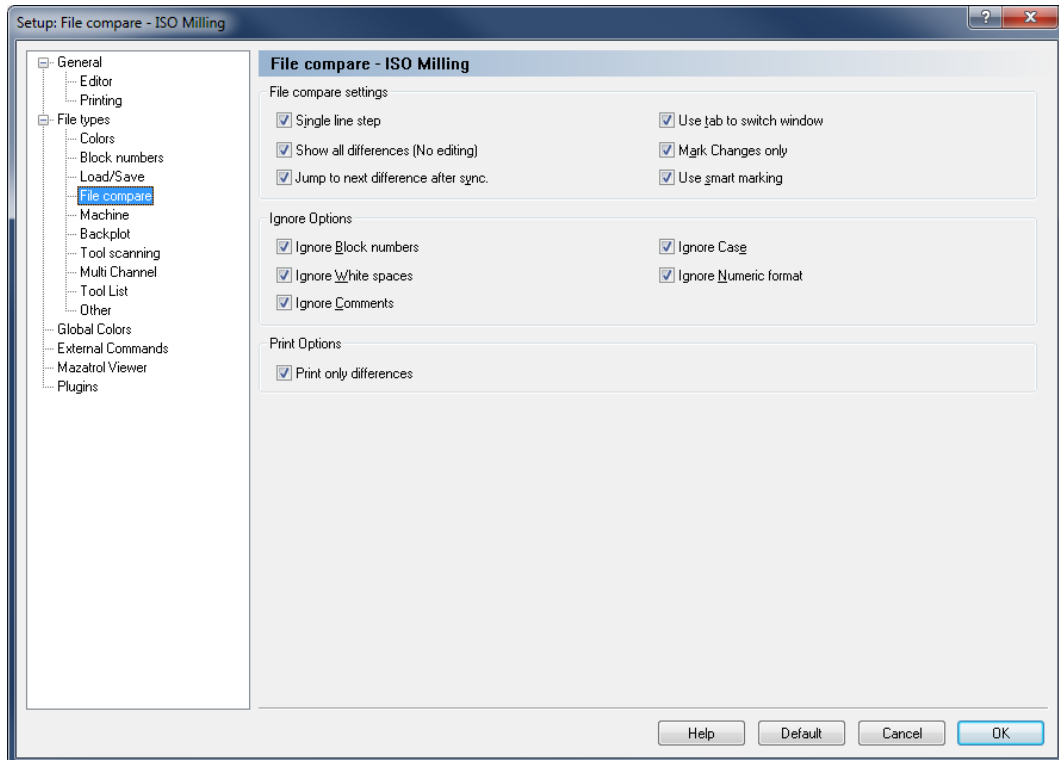
File Compare Settings.

File Compare Example

The file compare results of two settings *Show all differences* and *Mark changes only* are shown as example.

Show all differences

When this option is enabled, all lines in both windows containing at least one difference are gray, and the difference is highlighted with red.

If you move the cursor with the icon to a line with a difference, the whole line is colored in red. If the following lines also contain differences, the block is marked red. You can use the icon to synchronize one or more highlighted line(s) from left to right, or from right to left with the icon .
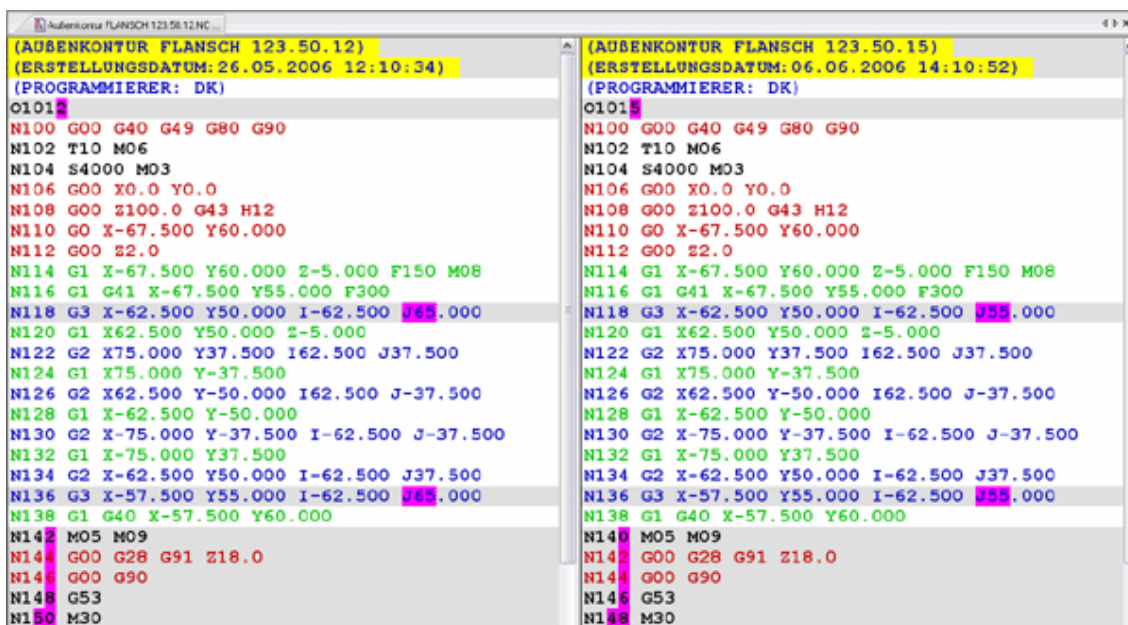
When text is moved from one window to another, no line is highlighted. This is because the texts in both windows are identical.

Mark changes only
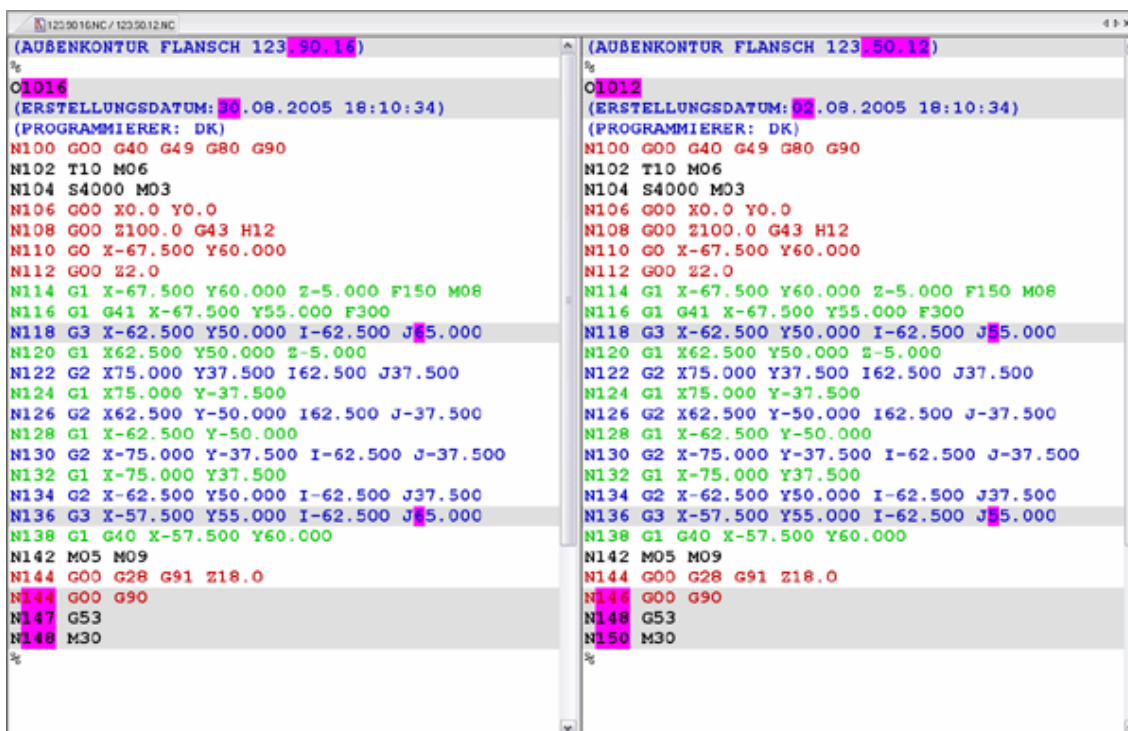
When this option is enabled, all lines in both windows containing at least one difference are gray, but only the difference is highlighted with red.

If there are differences in successive lines, and the cursor is moved to the first line with the icon 🔼, all lines are gray and the texts are marked red. You can synchronize the highlighted block from left to right with the icon 🔄, or from right to the left with the icon 🔄.

Further information about the File Compare settings is found under Setup File Compare.

File compare results with the option *Show all differences*.

File Compare results with the option *Mark changes only*.

5. Transmission

In this chapter, the **Transmission** menu in the CIMCO Edit v7 Ribbon is described.

Transmission is used to send NC programs from a PC to CNC machines or receive programs from the CNC machines. You can access the configuration of this function using the icon 🔲.

> ℹ️   The Editor must be licensed for DNC operations to use this option.

5.1 Transmission

In this section, the functions related to sending files to and from CNC machines are described. The functions are reachable through the **Transmission** tab. The toolbar is shown below, and every icon is showed next to the command name.

With the *DNC option*, the CIMCO Edit v7 user is able to send CNC programs from a PC to a CNC machine or receive programs from it. It is also possible to transfer CNC programs to / from multiple CNC machines at the same time.
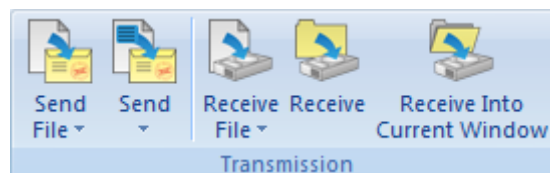
The Transmission menu.

Send File

Transmits a file to a selected machine.

Click this icon to open the search dialog for the file to send. After opening the selected file, the *Transmit status* window appears, showing the transfer data and the progress of the current transmission.

Click on the arrow below the icon to get a drop-down list with the last recently sent files to choose from.

Send

Transmits the current CNC program to a selected machine.

Send Selected Lines

Transmits the highlighted CNC program blocks to a selected machine.

Receive File

Receives a CNC program from a selected machine, saving it as a file without opening it.

Click on this icon to open the dialog *Receive file* for the file to be received. After entering a name for the expected CNC program and clicking **Save**, the *Receive status* dialog will appear showing the transfer status.

Click on the arrow below the icon to get a drop-down list with the last recently received files.

Receive

Receives a CNC program from a selected machine, opening it in a new window.

Receive Into Current Window

Receives a CNC program into the current window.

5.2 Machine

This section describes the functions related to file transmission on existing machine types. You can access the configuration of the selected machine through the icon. Read more about setting parameters for DNC communication under DNC Settings.
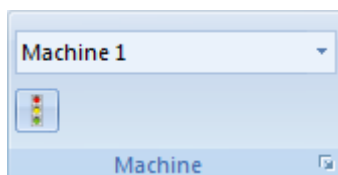
The Machine menu.

Machine Type

Select the CNC machine for the data transmission from the drop-down list. Click the field to show a drop-down list with the machines available for selection.

> The machine type determines the COM port and DNC settings.

Status

Click this option to open the status window of current transmissions and/or receptions.

If the five icons on the *Transmission* menu are grayed out, and it seems that a CNC program transfer can or will be in progress, then you can click on the icon to display the transmission state window (sending or receiving):
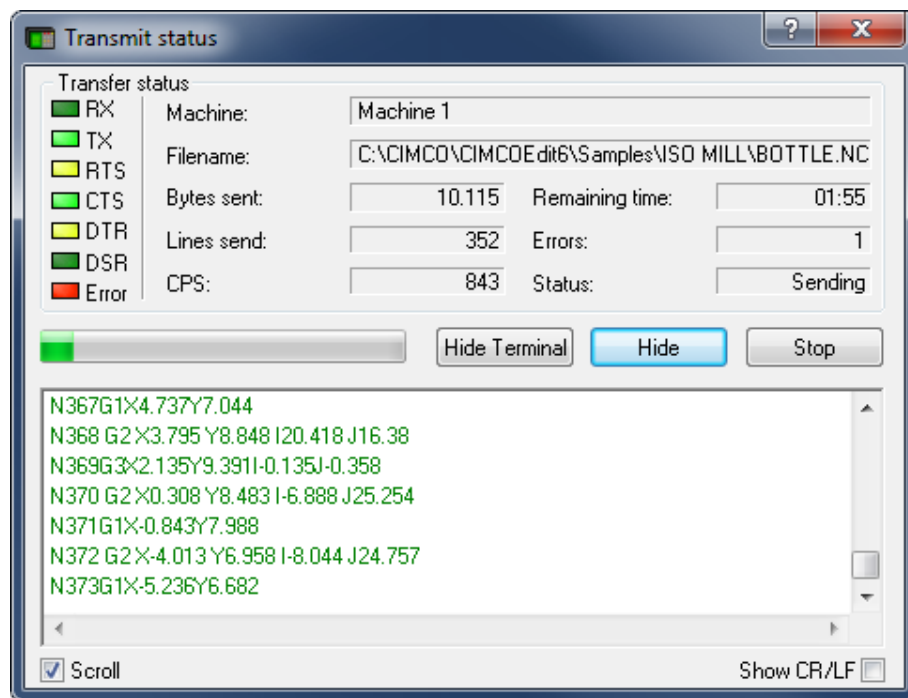
'Transmit status' window.

The top left of the status window shows the signals of the serial data transmission according to EIA RS-232-C, which light up when activated. The top right part shows information of the current transmission (Machine, Filename, Bytes send, etc.). Below the transfer status information there is a progress bar, three buttons to Start/Stop and Cancel the transmission and Show/Hide the terminal box, which shows the NC program being transmitted in plain text. Options to *Scroll* the text in the terminal box and to *Show CR/LF* (Carriage Return / Line Feed characters) are also available at the bottom of the window.

If you click the **Stop** button when a transmission is in progress, you will get the following dialog:
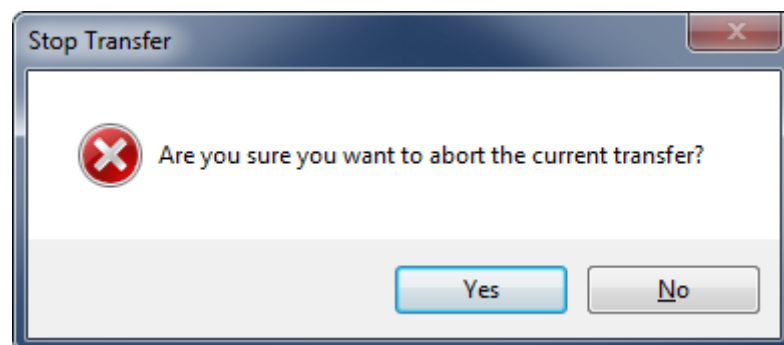
'Stop transfer' dialog.

Here you have the option to cancel the transfer (answering **Yes**) or continue (answering **No**).

When receiving a file from the machine, the transmission status window will look like this:
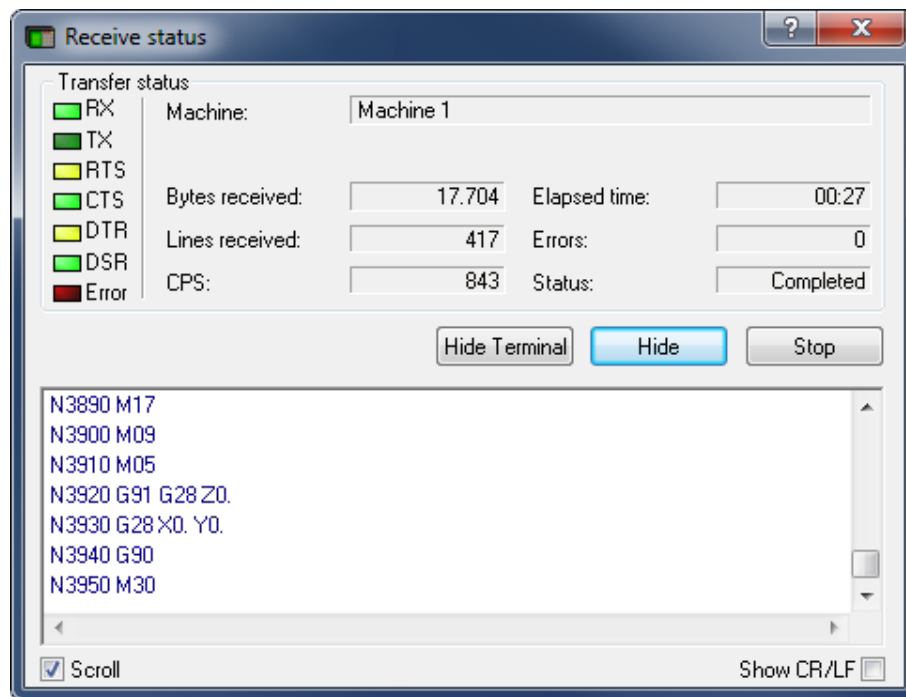
'Receive status' window.

When the CNC program (with M30) was completely received, is shown the following message:
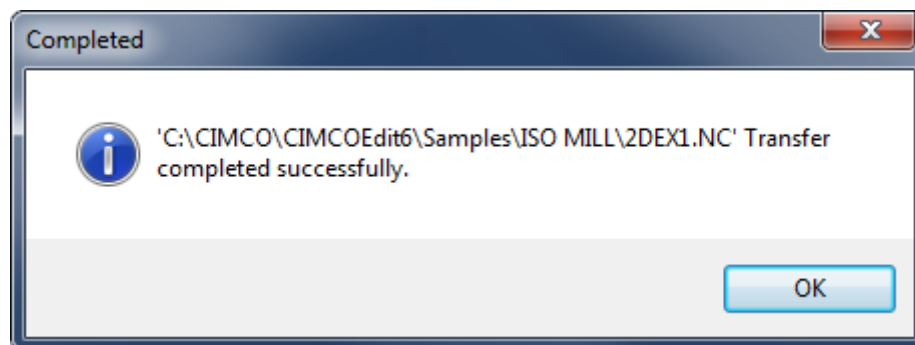
Report 'Transfer completed'.

Click **OK** to confirm the receipt of this message.

5.3 Setup

This section describes the functions related to the DNC settings for existing machine types. You can also configure and/or add new CNC machines.

First, determine what CNC machines are installed and check the transmission parameters. Open the CIMCO Edit v7 tab *Transmission* and click on the icon **DNC settings** to open the configuration dialog.

In this dialog you can see a list of the existing machines, and select one to rename it, delete it or add a new one.

If you want to change the transmission parameters of the selected machine, you can click on the **Setup** button in the lower right part of the dialog. The machine settings dialog will appear, and you can review and modify the interface parameters. From the tree menu on the left side, you can access the *Transmit* and *Receive* dialogs to check/change the transmission parameters.

Set in the *Directories* dialog your default folders (send and receive folders) from which you want to send and receive CNC programs to/from the machine. Use the folder icon to the right of the entry fields to navigate or create a new folder, where to store the transmission files.
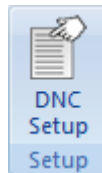
The Setup menu.

DNC Setup

Click this icon to add new machines and configure the DNC settings in CIMCO Edit. Read more about setting parameters for DNC communication under DNC Settings.

6. Editor Setup

This section describes how to configure the CIMCO Edit settings.

6.1 General Program Settings

If you wish to change the program settings concerning the program in general, enter the Setup menu and click on General. The general configuration dialog is shown below.
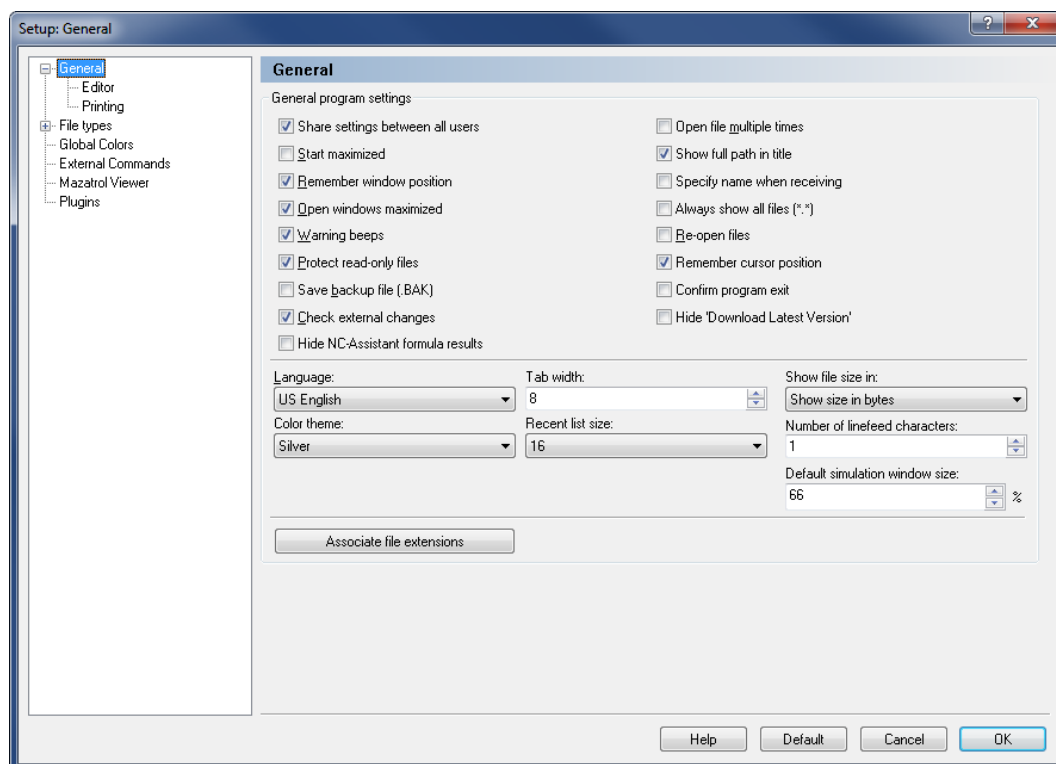
General program settings.

General Program Settings

☑ **Share settings between all users**

Check this field to share the editor settings between all users. All users must have read/write access to the program directory.

If this option is not checked, each user can define its user-specific settings, i.e.:

- User A uses language 'US English', tab width 10, etc.

- User B uses language 'Deutsch', tab width 8, etc.

These settings are stored in Windows (under USERNAME), so every user gets "his" Editor after logging in.

> ⓘ   If a user does not have read/write access to the program directory, this option is disabled.

☑ **Start maximized**

When this field is checked, CIMCO Edit v7 will always start maximized, i.e. filling up the whole screen.

☑ **Remember window position**

When this field is checked, the editor will start with the same window size and position as the last time the editor was used.

☑ **Open windows maximized**

When this field is checked, Edit windows will be opened maximized.

☑ **Warning beeps**

Check this field to use warnings beeps (switch this off for silent operation).

☑ **Protect read-only files**

Check this field to prevent read-only files from being modified or edited.

☑ **Save backup file (.BAK)**

Check this field to back up the previous version when saving a file. The backup file is named *filename.BAK*.

☑ **Check external changes**

When this field is checked, the editor will check if a file is modified outside the editor.

If a file is modified outside the editor, you will be notified and you will be asked if you want to reload the file. If you choose to do that, any changes made since the last save will be lost.

☑ **Hide NC-Assistant formula results**

Check this field if the result of NC-Assistant macro formulas should be hidden.

☑ **Open file multiple times**

Check this field if you want to be able to open the same file in more than one window in the editor.

☑ **Show full path in title**

Select this option to show the full path of files in the window title.

☑ **Specify name when receiving**

Check this field if you wish to specify a filename when using the Transmission tab function *Receive file and open in editor*.

☑ **Always show all files (*.*)**

Check this field if the file dialog should always show all files.

☑ **Re-open files**

Select this option to re-open files when the program is started.

☑ **Remember cursor position**

Check this field to place the cursor at the position where it was when the file was last opened.

☑ **Confirm program exit**

Check this field to confirm when closing the program.

☑ **Hide "Download Latest Version"**

Check this field to hide the *Download Latest Version* link from the Help menu.

> ⓘ   You must restart the editor after changing this option.

abc ▾ **Language**

Select the language to be used in menus and dialogs.

The following languages are currently available: Chinese, Czech, Danish, Dutch, English, Estonian, Finnish, French, German, Hungarian, Italian, Japanese, Korean, Polish, Portuguese, Romanian, Russian, Spanish, Swedish, Taiwan, and Thai

> ⓘ   Changing language will not have effect until the program is restarted.

123 ↕ **Tab width**

Use this field to specify the tab width in characters.

Show file size in

Select from the list, how the file size should be displayed in the status bar.

> The file size can be shown in bytes, in meter, or in feet.

Color theme

Select the color theme to be used in CIMCO Edit.

> Changing color theme will not have effect until the program is restarted.

Recent list size

Use this field to specify the number of files shown in the *Recent List*.

Number of linefeed characters

Use this field to specify how many line feed characters that should be used to calculate the file size shown in the status bar.

Example: If you specify a value of 2, and the file contains 12 lines, 24 bytes will be added to the file size shown in the status bar.

Default simulation window size

Use this field to change the default size of the simulation window in percent of the screen size. The default value is 66 %. Decrease this value to show NC files with long program lines, or increase the value to enlarge the simulation window.

> You can resize the Backplot window between 10-90% of default value 66%.

Associate file extensions

Click this button to associate file extensions (*.NC,*.H, etc.) with CIMCO Edit, overwriting the current association.

When you click this button, CIMCO Edit re-associates all the extensions defined for each file type under *File types - Load/Save*.

6.1.1   Editor Settings

By clicking on **Editor** in the left hand list of the General Setup menu, the Editor window will be displayed (see below). In this window, you can configure the settings for the editor.
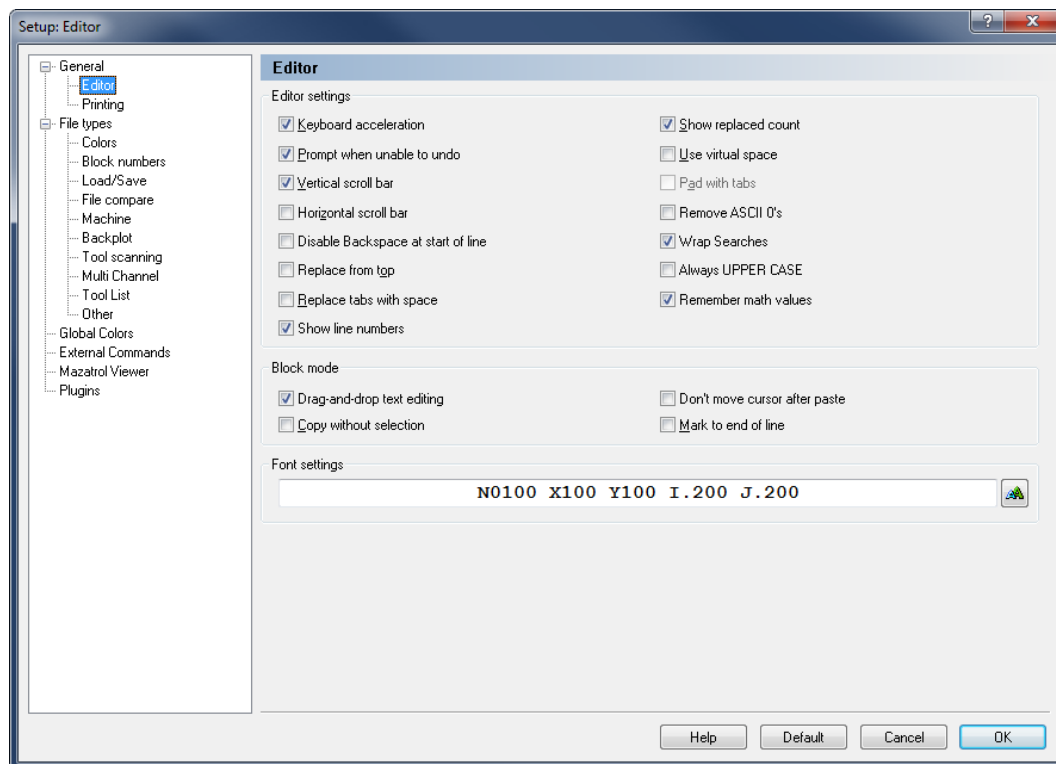
Editor setup dialog.

Editor settings

☑   **Keyboard acceleration**

Enables your cursor to move faster.

☑   **Prompt when unable to undo**

When this is active, you will be told when you are about to do something (such as renumbering a large file) which cannot be undone. If this is switched off, no such warning will appear.

☑   **Vertical scroll bar**

Enables a vertical scroll bar.

☑   **Horizontal scroll bar**

Enables a horizontal scroll bar.

☑ **Disable Backspace at start of line**

Prevents that lines are joined by using the backspace key.

☑ **Replace from top**

Any search and replace options will automatically start from the beginning of the file.

☑ **Replace tabs with space**

Inserts spaces when the tab key is used, and replaces existing tabs with spaces when a file is opened.

☑ **Show line numbers**

Check this field to show line numbers.

☑ **Show replaced count**

Shows the number of strings that were replaced after the *Replace all* function has been used.

☑ **Use virtual space**

Allows the cursor to move into space that currently contains no text.

☑ **Pad with tabs**

Use tabs to fill in large empty spaces in virtual space. (Please refer to *Use virtual space* above.)

☑ **Remove ASCII 0's**

Check this field to remove ASCII 0's from the file. If you do not check this option, ASCII 0's are replaced with ASCII 128.

> ⓘ   If you transmit files with 7 data bits, ASCII 128 becomes ASCII 0.

☑ **Wrap Searches**

Check this field if searches should continue searching from the top of the file if no match is found before the end of the file.

☑ **Always UPPER CASE**

Check this field if inserted text should always be UPPER CASE (all CAPS).

☑ **Remember math values**

Remembers the last values used in the Simple Math Functions.

Block Mode

☑ **Drag-and-drop text editing**

When this option is enabled, you can drag any selected text to a new location.

☑ **Copy without selection**

Enables you to copy the line at the cursor, without marking it as a selection.

☑ **Don't move cursor after paste**

When this option is checked, the cursor will not move to the end of the pasted text, but remain at the beginning of the text.

☑ **Mark to end of line**

If a whole line is selected, the indication continues across virtual space to the side of the window.

Font settings

abc　　This field shows an example of the selected font. Click the 🏔 **Select font** icon to the right to change the font used in the editor window.

6.1.2   Printing

To change printer settings, choose **Printing** from the left hand list of the General Setup menu. The printer configuration dialog is shown below.
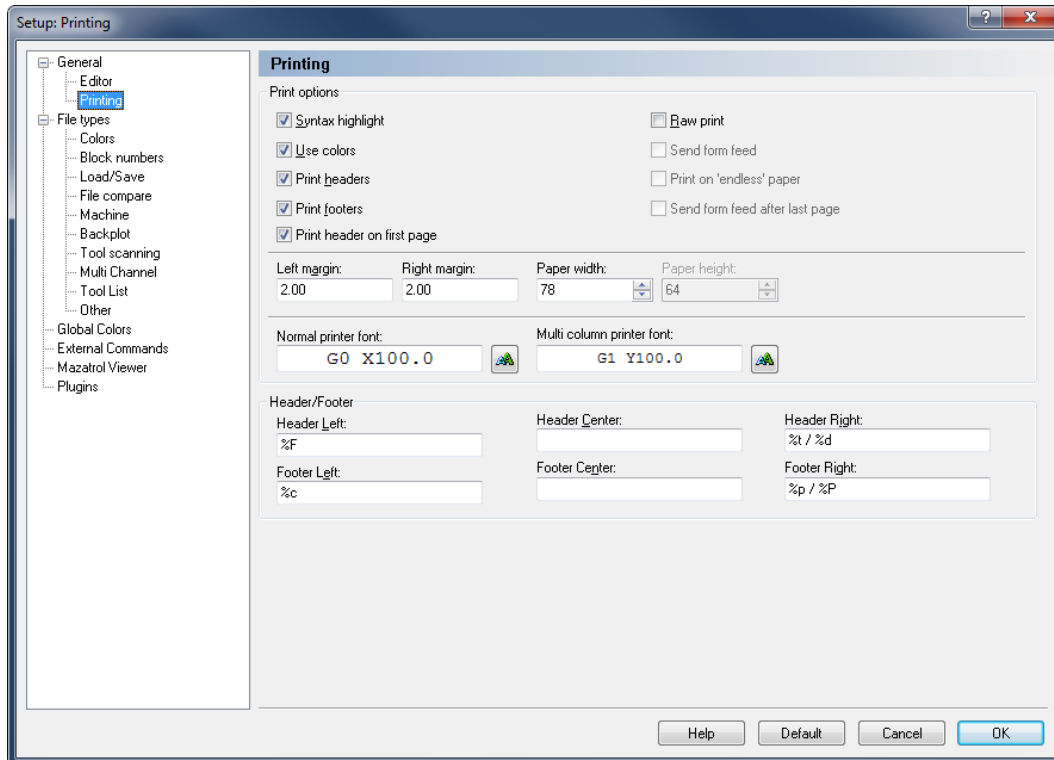
Printer setup.

Print options

☑ **Syntax highlight**

Select this option to apply syntax highlighting to the printed text. Use Bold and Italic to highlight NC commands and comments.

☑ **Use colors**

Enables color printouts. This requires a color printer.

☑ **Print headers**

Prints headers on every page.

☑ **Print footers**

Prints footers on every page.

Print header on first page

Select this option to print a header on the first page.

Raw print

Produces simple text output to the printer. This is useful when printing large programs to matrix printers. When raw print is selected, the options *Syntax highlight*, *Use colors* and *Print header on first page* are disabled. The options *Send form feed*, *Print on endless paper*, *Send form feed after last page* and *Paper height* will be activated to setup the raw print.

Send form feed

When this option is checked, a form feed character (FF) will be sent after reaching the *Paper height* counter, and the next line will be printed on the next page. No form feed character is needed after the last page, so the option *Send form feed after last page* will be grayed out.

Print on "endless" paper

Check this field to print on endless paper. When this option is selected, no header and/or footer is printed.

Send form feed after last page

If you check this option, a form feed character will be sent at the end of a raw print on continuous paper to indicate the last printed page.

Left margin / Right margin

Specify the size of the margins in centimeters. If the margins are in inches add "in"

Example: To specify a one-inch margin, enter "1in".

Paper width / Paper height

Specify the size of the paper in width and height of the paper in characters.

If for example in multi-column printing the font size is set to 11 points and the *Paper Width* is set to 35 (characters/line), a CNC program of 140 NC blocks will be printed legibly in two columns on an A4 page.

Normal printer font

Click the icon to the right to select the font size and type for the printout.

Multi column printer font

Click the icon to the right to select the font used when printing in 2 or more columns.

Header/Footer

abc   Use these fields to specify the contents of the header and footer. The following options
             are available:

- %p : Page number

- %P : Page count

- %f : File name

- %F : File name with path

- %t : Time

- %d : Date

- %c : Company name

- %U : Windows user name

6.2 File Types

This section describes how to configure file type settings.

File type settings determine how to perform renumbering, look for tool changes, compare
files, backplot settings, set up colors, where to save files, etc.

> Colors, block numbers, backplot, file compare, etc. are configured individually for each
> file type.

File type is a macro file, which contains specific properties for the machining type (e.g.
Turning, Milling, etc.), program codes (ISO G-code, Heidenhain, plain text), and commands
(renumbering, tool changes, colors, backplot, etc.).

The File types *ISO Turning*, *ISO Milling*, *Heidenhain TNC*, and *Text file* are integrated in
CIMCO Edit v7. You can add new file types e.g. *Turning* by modifying existing macros or
inserting your own macros. Information on how to proceed is found in the section Macro
Setup.

To configure File types settings, select 🗐 **Global Setup** in the *Editor* tab, or click on the setup
dialog launch icon ▣. This will open the Editor Setup window. Click on **File types** in the left
tree of the setup dialog.

The following dialog appears:
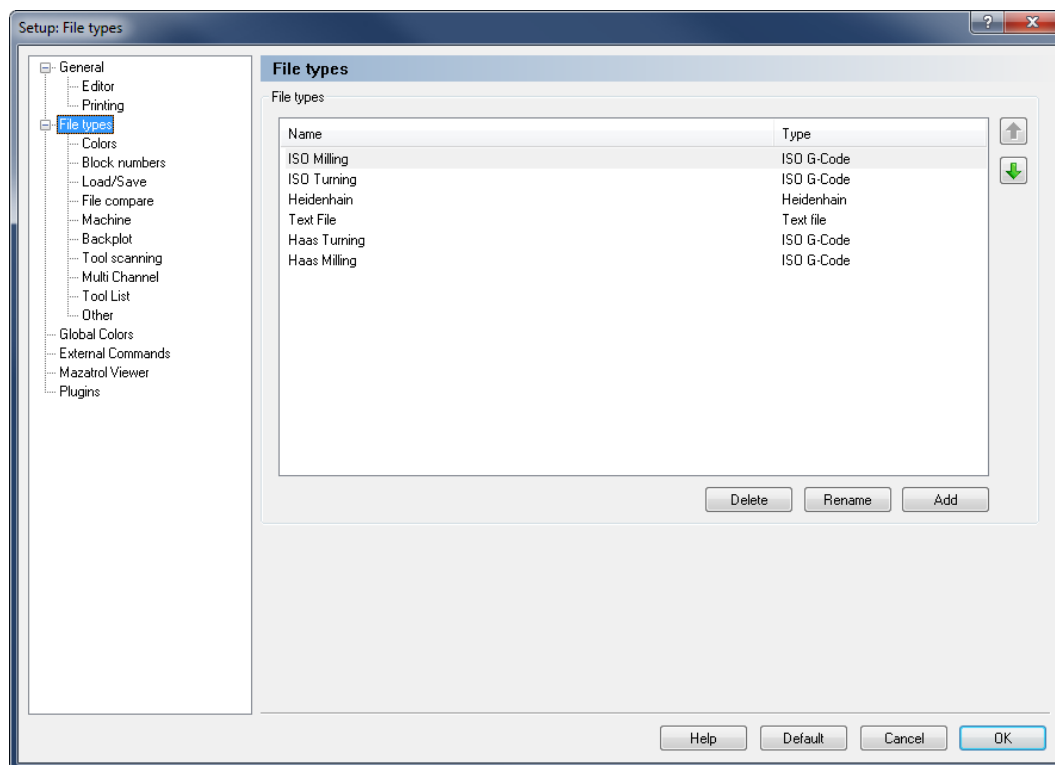
File type overview.

File types

This field shows a list of defined file types. Use the Up/Down arrows ⬆⬇ to move the selected file type up or down on the list.

Delete

To delete a file type, highlight it and click on the button **Delete**. You will be prompted to confirm the delete operation.

Rename

Click this button to rename the selected file type. To rename a file type, highlight it and click on the button **Rename**.

Add

Click this button to add a new file type

To add a new file type, click on the button **Add**.

The following window appears:
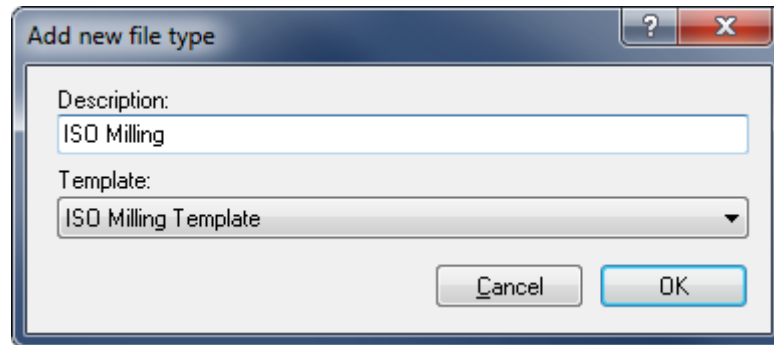
Add a new file type.

Enter the name of the new file type, select a file type template from the drop-down list and click **OK**. Click **Cancel** if you wish to abort the operation.

6.2.1   Colors

Use this dialog to configure color settings for the selected file type.

To define the colors used for highlighting within a certain file type, select **Colors** from the **File types** menu in the left tree of the Editor Setup dialog, or click the **Colors** icon in the *File Type* menu within the Editor tab. The color configuration dialog is shown below.
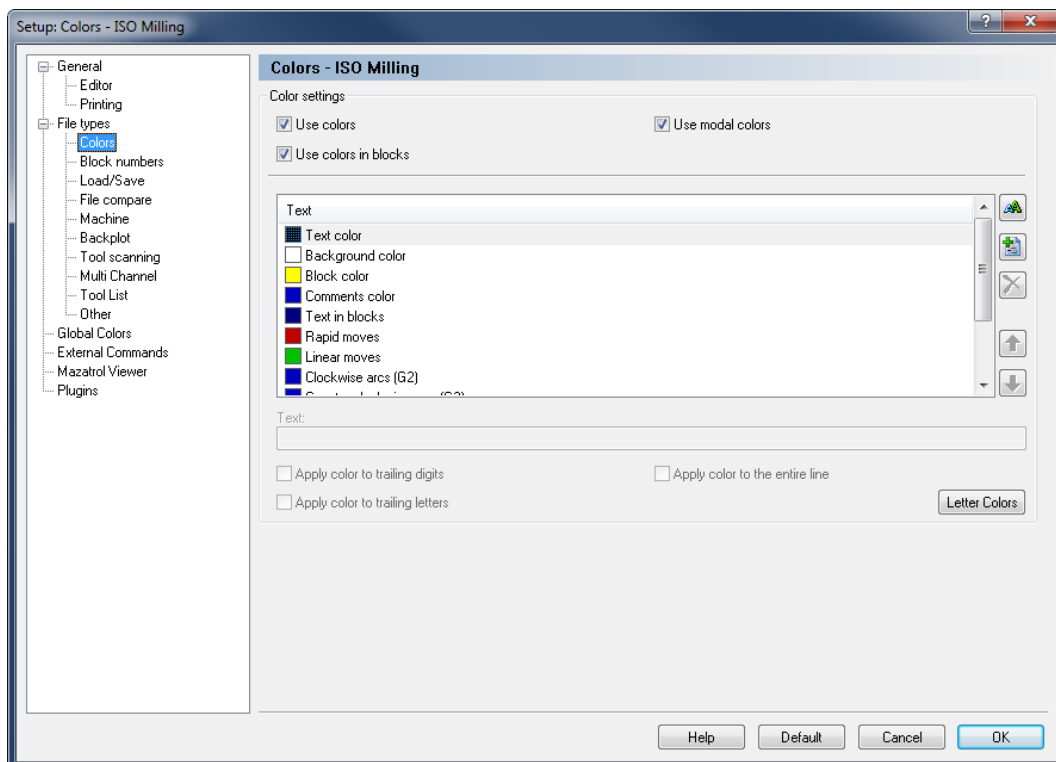
Color setup.

The following defined color settings apply for the file type "ISO Milling".

Color Settings

☑ **Use colors**

Select this option to use color highlighting.

☑ **Use colors in blocks**

Check this field to use color highlighting inside a selected block. When this field is unchecked, the default color for *Text in blocks* is used.

☑ **Use modal colors**

Check this field to use 'modal' colors. When this option is selected, blocks containing rapid moves, linear moves, and arcs will be assigned different colors.

> ℹ️ If all three options above are unchecked, the entire NC text - including comments - is set to the color defined in *Text color*.

abc **List of defined colors**

By clicking the button 🖼️ or double-clicking on the desired label, you can set the color for the selected item.

- **Text color**
  The color used for text, when *Use colors* is not checked.

- **Background color**
  The background color used for editor windows.

- **Block color**
  The background color within a selection.

- **Comments color**
  The color used for comments, as defined by the Comment start and Comment end, or Block Skip character on the Machine template configuration tab.

- **Text in blocks**
  The color used in one or more selected program blocks when the field *Use color in blocks* is not checked.

- **Rapid moves**
  The color used for rapid moves.

- **Linear moves**
  The color used for linear moves.

- **Clockwise arcs (G2)**
  The color used for clockwise arcs.

- **Counter clockwise arcs (G3)**
  The color used for counter clockwise arcs.

- **File compare background for different lines**
  The color used for the background for different lines.

- **File compare background for different text**
  The color used for the background for different text.

- **File compare color for missing lines**
  The color used for the background for missing lines.

Select color

Select a new color. To select a new color, highlight a color in the list and click on this button (you can also double-click in the list).

Add new color

Click this button to add a new color. Enter the text that triggers the new color in the field *Text*, and select the desired options to define what applies the new color to.

Remove selected color

Click this button to remove the selected color.

> You can only remove new added colors.

Up/Down arrows

Use these buttons to move the selected color up and/or down on the list. Entries higher on the list have higher priority.

Text

Use this field to specify a string that will be highlighted using the selected color.

Apply color to trailing digits

If this option is checked, the selected color will be applied to any digits following the selected letter.

Apply color to trailing letters

If this option is used, the selected color will be applied to any letters following the selected letter.

Apply color to the entire line

Check this field if the selected color should be applied to the entire line.

Letter Colors

Click this button to create a different color for each letter. This will set the coloring similar to the default in CIMCO Edit V4.

> The color settings will only affect the selected file type.

6.2.2   Block Numbers

The block numbers in a CNC control program are an important guide and are always at the beginning of an NC block. The major part of a CNC program consists of successive NC blocks where the tool paths for machining a workpiece are defined.

Because often changes need to be carried out, i.e. new NC blocks are inserted between two old ones, the definition of 'interval' between two NC blocks according to the expected number of NC blocks is an important decision that affects the size of the block numbers, and therefore their readability for the user. If the block number sequence is disturbed by many changes, a 'renumbering' might be required.

To configure renumbering and/or automatic block numbering, select **Block numbers** from the **File types** menu in the left tree of the Editor Setup dialog, or click the **Configure Block Numbering** icon 📋 in the *File Type* menu within the Editor tab. The block number configuration dialog is shown below.
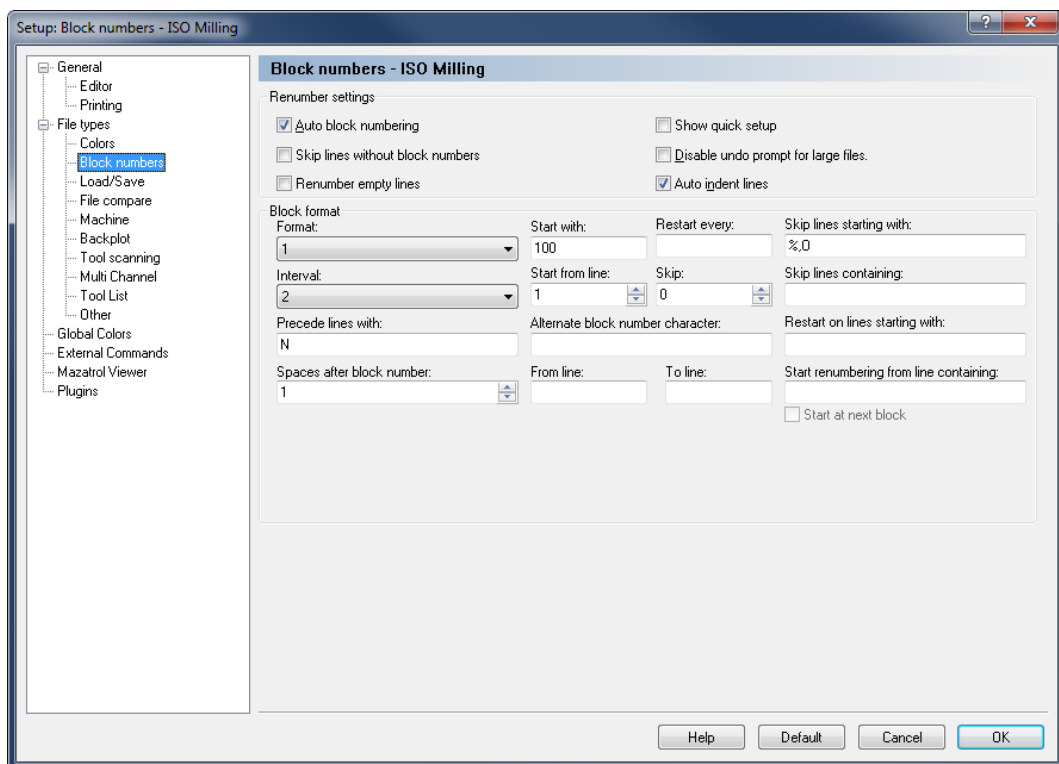
Block number configuration dialog.

Renumber Settings

☑ **Auto block numbering**

Inserts line numbers automatically, according to the format described. If you make a new line (after the existing lines), the new line number will be governed by the preceding line number and the setting for *Interval*. If you insert a line in an existing program, the new line number will be midway between the preceding line and the following line.

☑ **Skip lines without block numbers**

Check this field to skip lines without a block number when renumbering. The block number sequence is continued on the next line with block number.

☑ **Renumber empty lines**

Check this field to renumber empty lines - a single empty line or several consecutive empty lines.

☑ **Show quick setup**

Check this field to show a 'Quick Setup' dialog when renumbering a program.
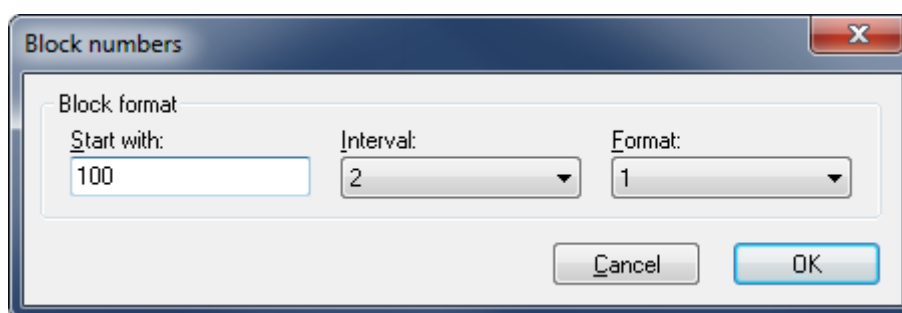
Quick block numbers setup dialog.

You can change the default values shown in the dialog (defined under 'Block format' in the Block numbers setup dialog) to fit your renumbering requirements and click OK.

☑ **Disable undo prompt for large files**

When this field is not checked, you will be told when you are about to renumber a large file, which cannot be undone. If this is switched on, no such warning will appear.

☑ **Auto indent lines**

Indents new lines. The indention is set at the same level as the line above.

Block Format

abc ▾ **Format**

This field provides a list of available block number formats. Select a format from the drop-down list. I.e. for 4 digit block numbers and max. 9999, select format "0001".

ⓘ   This will not limit the number of digits in the line number. If you wish to limit the number of digits, you have to specify a maximum line number in the *Restart every* field.

Start with

Makes it possible to specify the starting value of the block number.

Restart every

Specify when the line number must restart. This number is equal to the maximum line number.

Example: To use line numbers from 0-9999, specify 9999 in this field. If no value is specified, the line number will increment until it reaches 100000000, before it restarts.

Interval

Use this drop-down list to select the interval by which the block number must increase for each block. Preset intervals are 1, 2, 5, 10, 20, 100, 1000 and Auto.

If you select one of the *Auto ...* options, the number selected is the maximum block number, and the interval will be selected to distribute the block numbers as evenly as feasible, within this maximum block number.

Example: To renumber a CNC program with 482 NC blocks, select the option 'Auto 1000' for the interval. When you the click on the icon **Renumber** in the NC Functions toolbar *Block Numbers*, block number will be N0964 (which is the highest multiple of 482 that is less than 1000). From this, the program will calculate the most reasonable interval to be 2. This is the highest possible spacing between the block numbers if 964 is the highest possible number and 482 blocks need to be numbered.

If we want to renumber the same program of 482 NC blocks, and this time we select 'Auto 10000', the program will recognize the highest possible block number to be N9640, which divided by the number of blocks yields an optimum interval of 20.

> In this example, we assume that the value of the field *start with* is set to 0.

Start from line

Specify the first line in the program that will be given a line number.

Skip

Use this field to specify if block numbers should skip lines.

Example: If you specify 2 in this field, only every 3rd line will be numbered.

Precede lines with

Here, you can enter a string that will be inserted before the line number. For ISO type machines, this is normally the letter 'N'.

Alternate block number character

Use this field to specify an alternate block number character.

Example: If the control uses both 'N' and ':' for block numbers, specify ':' in this field.

Spaces after block number

Use this field to specify the number of spaces to insert after the block number. To insert block numbers with a specific column width, specify a negative value.

From line

Specify the number of lines that should be skipped before the renumbering begins.

To line

Use this field to specify the maximum number of lines that should be renumbered.

If a positive number is written here, it specifies the last line to be renumbered, counted from the beginning of the program.

Example 1: If your CNC program starts with 4 comment lines and a program start line (with program start trigger and number, but without 'N'), only the first 10 NC blocks will be renumbered if you specify 15 in this field.

If a negative value (-N) is specified, the last N lines (counted from the end of the program) will not be renumbered.

Example 2: If you specify the value -2, and the last line of your CNC program (line containing the program end code M30) is followed by another line containing a special character (e.g. %), then only the last line is not renumbered.

Skip lines starting with

Defines that lines starting with a particular string and/or characters should be omitted, i.e. '%', 'O', '(', etc. Multiple strings can be specified using a comma ',' as separator. If you wish to omit lines beginning with a comma ',', specify ',,'.

Restart on lines starting with

Use this field to specify when the block numbers should be restarted.

Example: To renumber several programs in the same file, specify 'O' in this field, then the block numbers will be reset every time an O-Number is encountered.

Skip lines containing

Here you can specify that lines containing a string and/or character should be omitted. Multiple strings can be specified using a comma ',' as separator.

abc   **Start renumbering from line containing**

Use this field to specify a renumbering start trigger. If you specify a renumbering start trigger, renumbering will start from the first block containing the specified string. To start from the first block after the block containing the string, check the *Start at next block* option below.

☑   **Start at next block**

Check this field to start renumbering from the block after the line containing the renumbering start trigger.

> ⓘ   The Block numbers settings will only affect the selected file type.

6.2.3   Load / Save

To configure Load/Save parameters, select **File Type** from the left tree menu in the Setup window and then click **Load/Save**, or click the **Directories / Extensions** icon 🖼 in the *File Type* menu within the Editor tab. The Load/Save configuration dialog is shown below.
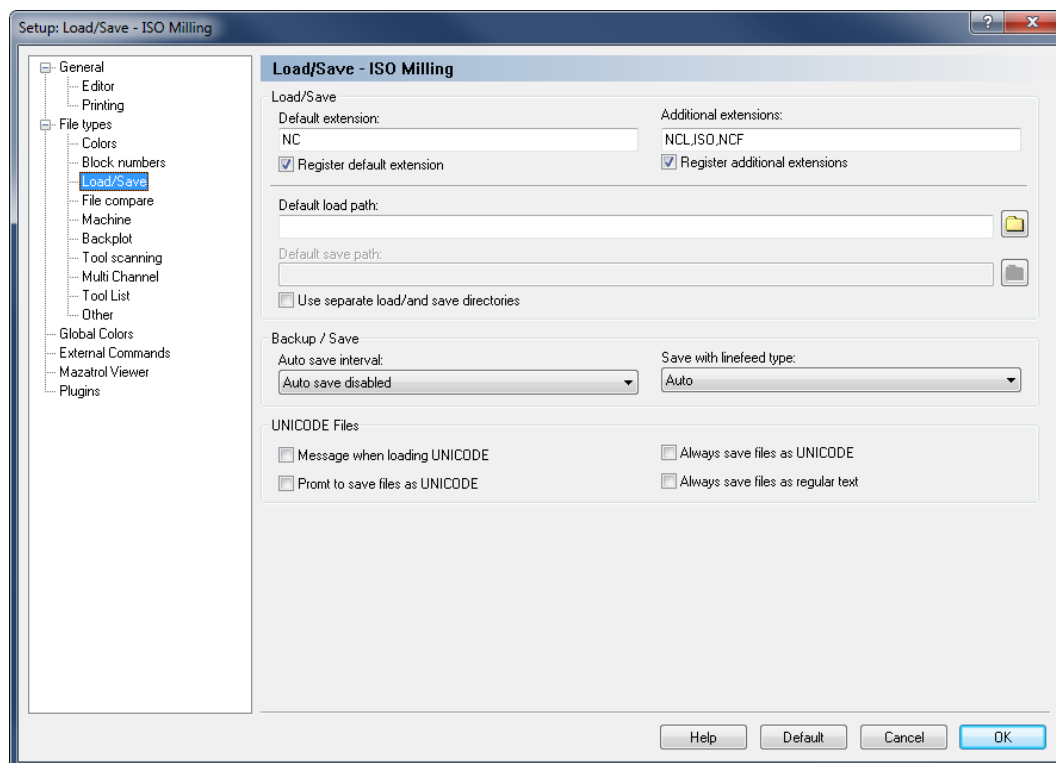
Load/Save configuration dialog.

Load/Save

abc **Default extension**

Specify a default extension to use for the selected file type. This extension will be appended to any new files saved, unless an extension is specified in the Save as window.

☑ **Register default extension**

Check this field to register the specified default extensions. When an extension is registered in Windows, clicking on a file with the specified extension in the Windows Explorer will automatically open the file in the editor.

abc **Additional extensions**

Define additional file extensions associated with your machine type. Extensions determine the type(s) of file(s) you wish to open (only files with the extension(s) defined here will be displayed in an Open file window). You can define more than one extension using a comma ',' as separator.

Example: To associate the extensions .ISO and .NC, specify ISO, NC.

☑ **Register additional extensions**

Check this field to register the specified Additional extensions. When an extension is registered in Windows, clicking on a file with the specified extension in the Windows Explorer will automatically open the file in the editor.

abc **Default load path**

This function specifies the default load path, which is where the file Open dialog will first point to. If no path is defined, the last directory you loaded a file from will be used. Click on the folder button 📁 to browse for an adequate load path.

abc **Default save path**

This function specifies the default save path, which is where the file Save dialog will first point to, when you wish to save a new file of the selected type or when you use the file Save as menu. If no path is defined, the last directory you saved a file in will be used. Click on the folder button 📁 to browse for an adequate save path.

☑ **Use separate load/and save directories**

When this field is checked, the editor will remember the last load and save position independently.

Backup / Save

abc ▾   **Auto save interval**

Select the length of the interval between auto saves from this list. Auto save saves your file at a regular interval, overwriting the original file.

abc ▾   **Save with linefeed type**

Use this field to specify the linefeed type used when saving files.

- Auto: Do not change linefeed type. When saving new files, the DOS/Windows linefeed type is used.
- DOS/Windows: CR LF (ASCII 13 10)
- Unix: LF (ASCII 10)
- Mac: CR (ASCII 13)

UNICODE Files

☑   **Message when loading UNICODE**

Displays a message when a UNICODE file is loaded.

☑   **Prompt to save files as UNICODE**

Check this field if the file should be saved as UNICODE.

☑   **Always save files as UNICODE**

Check this field if files always should be saved as UNICODE.

☑   **Always save files as regular text**

Check this field if files should never be saved as UNICODE.

ⓘ   The Load/Save settings will only affect the selected file type.

6.2.4   File Compare

To configure file compare settings, select **File compare** from the **File types** menu in the left window of the Editor Setup dialog, or click the **Setup file compare** icon ⬚ in the *Other* menu within the File Compare tab. The file compare configuration dialog is shown below:
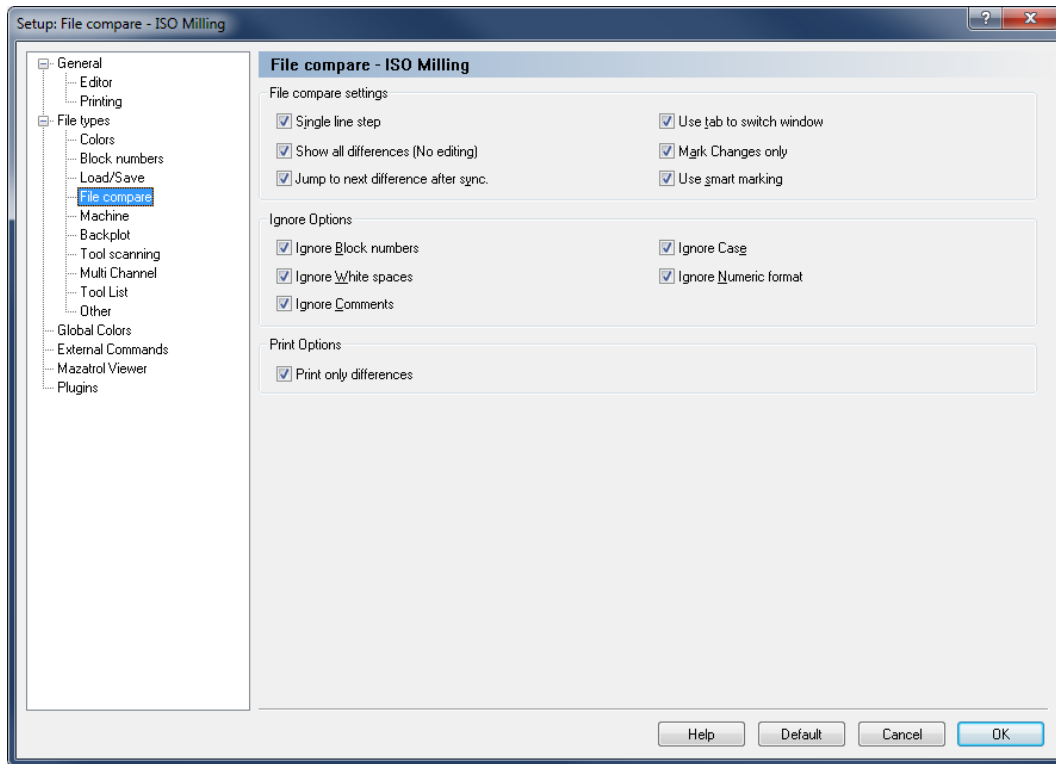
File compare configuration dialog.

File Compare Settings

☑ **Single line step**

Use this option to single step through multiple consecutive differences. Only the difference on the current line is highlighted.

☑ **Show all differences (No editing)**

Check this field to show all lines with at least one difference. All the differences are highlighted at the same time.

☑ **Jump to next difference after sync.**

When you select this option, the editor will jump to the next difference after you have synchronized two differences.

☑ **Use tab to switch window**

Select this option if you want to use the tab key to switch between the two files being compared. You will not be able to insert tabs in any of the files when this option is enabled.

Mark Changes only

When you select this option, all the lines with at least one difference will be activated, but only the difference will be highlighted, not the entire line.

Use smart marking

When using this option, not only the different part of the command will be marked, but the complete command if it contains at least one difference.

Ignore Options

Ignore Block numbers

The values of block numbers will be ignored, as will the presence/absence of block numbers (i.e. the fact that there are block numbers in one file and not in the other will be ignored).

Ignore White spaces

White space characters will be ignored. White space characters are space (ASCII 32) and tab (ASCII 9).

Ignore Comments

Anything inside a comment will be ignored. For this to work, you must have specified the correct Comment start e.g. '(' and Comment end e.g. ')' characters for this machine template under the Machine templates tab.

Ignore Case

If this option is checked, the letter case will be ignored, i.e. if one program contains lowercase and the other UPPERCASE characters for the same command.

Ignore Numeric format

Leading and trailing zeros will be ignored, as will optional '+' characters, i.e. 'X+14.10' matches 'X14.1', if this option is used.

Print Options

Print only differences

Check this field to print only the lines with differences between two files.

> ⓘ   The File Compare settings will only affect the selected file type.

6.2.5   Machine

To configure machine specific characters and commands, select **Machine** from the **File types** menu in the left hand list of the Editor Setup dialog, or click the **Machine Templates** icon ![icon] in the *File Type* menu within the Editor tab. The machine configuration dialog is shown below:
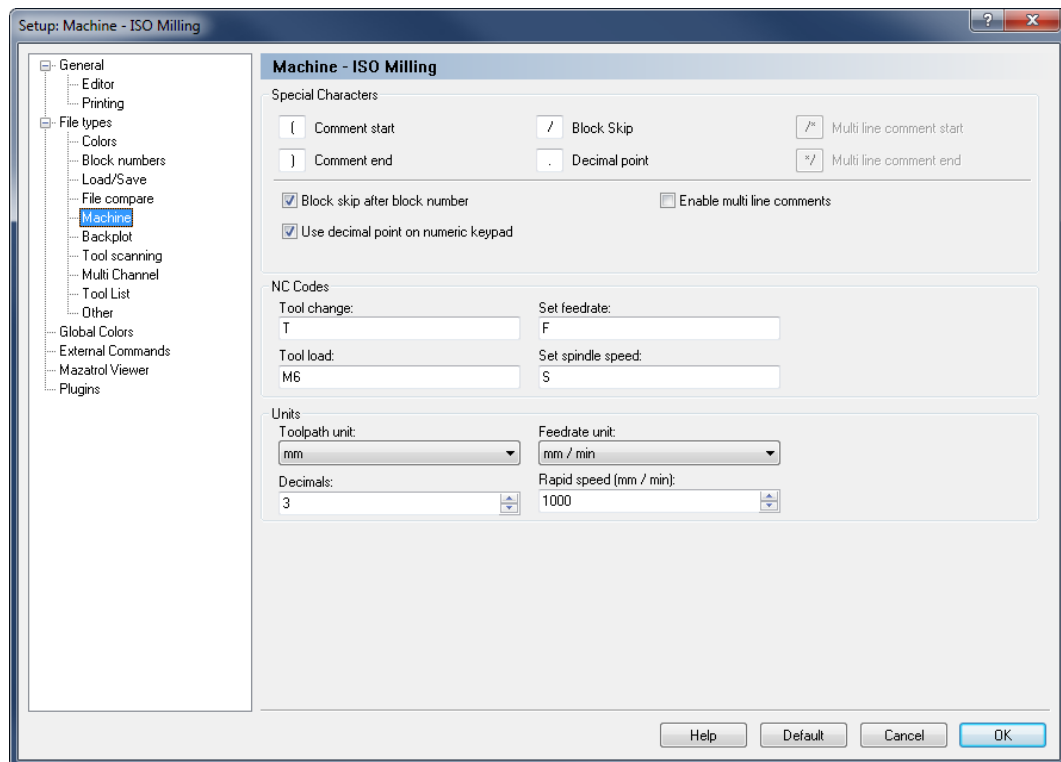
Machine type configuration dialog.

Spceial Characters

abc   **Comment start**

Enter the comment start character here, for example (.

abc   **Comment end**

Enter the comment end character here, for example).

abc   **Block skip**

Enter the block skip character here. If you don't want a block skip character, just leave the field empty.

abc   **Decimal point**

Enter the decimal separator here (decimal point or comma).

abc   **Multi-line comment start**

Use this field to specify the multi-line comment start string.

abc   **Multi-line comment end**

Use this field to specify the multi-line comment end string.

☑   **Block skip after block number**

Check this field if a block skip should be inserted after the block number. If you leave this field unchecked, the block skip will be just before the block number.

> ⓘ   You can specify another block skip character in the field *Block Skip*, if necessary.

☑   **Use decimal point on numeric keypad**

Check this field if the decimal key on the numeric keypad should insert the decimal point specified above.

☑   **Enable multi line comments**

Check this field to enable coloring of multi line comments.

NC Codes

abc   **Tool Change**

Use this field to specify the tool change prefix.

Example: If the tool number is specified as T010101, specify T in this field.

abc   **Tool load**

Use this field to specify the tool load command. If the tool load command is found on the same line as the tool change command, the tool is loaded immediately.

Example: N1230 T020202 M6.

abc   **Set feedrate**

Use this field to specify the feedrate prefix.

Example: If the feedrate is specified as F1000, specify F in this field.

Set spindle speed

Use this field to specify the spindle speed prefix.

Example: If the spindle speed is specified as S1000, specify S in this field.

Units

Toopath unit

Select the path unit from this drop-down list.

Feedrate unit

Select the feedrate unit from the drop-down list.

Decimals

Use this field to specify the number of decimals shown in the backplot etc.

Rapid speed (mm / min)

Use this field to specify the rapid feedrate.

The machine settings will only affect the selected file type.

6.2.6   Backplot

Use this dialog to configure general backplot settings.

To configure the Backplot settings, select **Backplot** from the **File types** menu in the left window of the Editor Setup dialog, or click the **Backplot Setup** icon ▊ in the *File* menu within the Backplot tab. The following dialog appears:
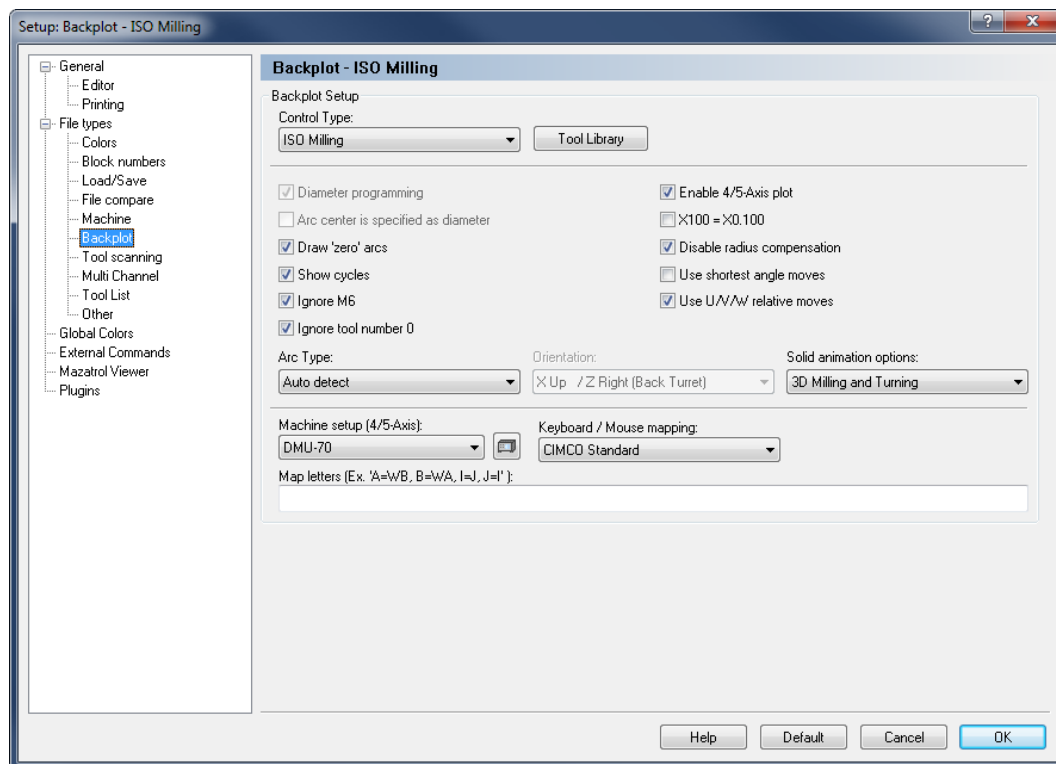
Backplot Setup dialog.

ℹ   Only the backplot options for the selected file type are active. All other options are grayed out.

Backplot Setup

`abc ▾` **Control Type**

Select the control type for the backplot from the list.

`Button` **Tool Library**

Use this button to set up the tool library for the selected control type. Information on how to set up tools is found under Tool.

☑ **Diameter programming**

Check this field to select diameter programming.

☑ **Arc center is specified as diameter**

Check this field to select diameter programming for arc centers also (I coordinate).

☑ **Draw 'zero' arcs**

Check this field if a full circle should be drawn on arcs with the same start and end-point.

☑ **Show cycles**

Check this field to plot cycles.

☑ **Ignore M6**

Check this field to ignore tool load commands (like M6).

ⓘ   You can specify the tool load command on the Machine setup page.

☑ **Ignore tool number 0**

Check this field to ignore tool number 0. This is useful if T0/T00/T0000 is used to cancel the length offset.

☑ **Enable 4/5-Axis plot**

Check this field to enable 4/5-Axis backplotting.

☑ **X100 = X0.100**

When this option is selected, the program will assume that a value without a decimal separator is 1/1000 of a regular move.

☑ **Disable radius compensation**

Check this field to disable tool radius compensation.

☑ **Use shortest angle moves**

Check this field to use the shortest angle.

Example: When moving from 359 degrees to 1 degree, the simulation will normally move 358 degrees back. When this option is checked, only a 2 degrees move is performed.

☑ **Use U/V/W relative moves**

Check this field if U/V/W should be used as relative moves.

`abc ▾` **Arc type**

Select the type of arc that is used from this drop-down list.

`abc ▾` **Orientation**

Use this field to specify the orientation of the plot for turning operations.

`Button` **Solid animation options**

Use this field to select if the solid animation should be performed as a normal or New OpenGL three dimensional visualization (3D milling and turning) or as a two dimensional simulation (2D turning).

> ⓘ New OpenGL is only enabled if the supported OpenGL version is equal or higher than 3.3 and the Graphics card is NVidia: GeForce, Quadro, NVS or ATI: Radeon, FirePro. All other graphics cards will by default use "Solid Animation" which supports OpenGL version 1.1 or higher.

> ⓘ The two-dimensional simulation is only available for turning routines.

`abc ▾` **Machine Setup (4/5-Axis)**

Use this field to specify the machine type. The machine type/geometry is important for 5-Axis simulation. Click on the icon 🖳 next to the field to edit the machine configuration.

`abc ▾` **Keyboard / Mouse mapping**

Select the keyboard mapping from this list.

`abc` **Map letters (Ex. 'A=WB, B=WA, I=J, J=I')**

Use this field to map the default letters to a different string.

Example: If a CNC uses XA for A-axis rotation, specify A=XA in this field.

Example: If a machine has switched the X and Y axis, specify X=Y, Y=X.

6.2.7 Tool Scanning

This section describes the Advanced Tool scan feature in CIMCO Edit v7. All commands are case insensitive, but the values are kept in all caps for compatibility reasons.

The Advanced Tool scanning feature allows you to automatically scan tool dimensions from the NC program file. With this function, the tool number, type, and dimensions will be automatically read from the NC program.

Only milling tools are supported.

When using **Automatic Tool scanning**, the tool scanner searches for lines in the NC file that contain the tool line trigger. The default trigger keyword is *TOOL*, and if necessary the user can change the tool line trigger.

The tool line is expected to be inside comments, and it uses the *Comment end* character from the machine settings if available.

When the tool is scanned, the order of the dimension keywords is irrelevant. The user is able to change the keywords.

Some words can significantly slow down the performance if they occur often in the NC file.

The Tool Scanning configuration dialog is shown below:
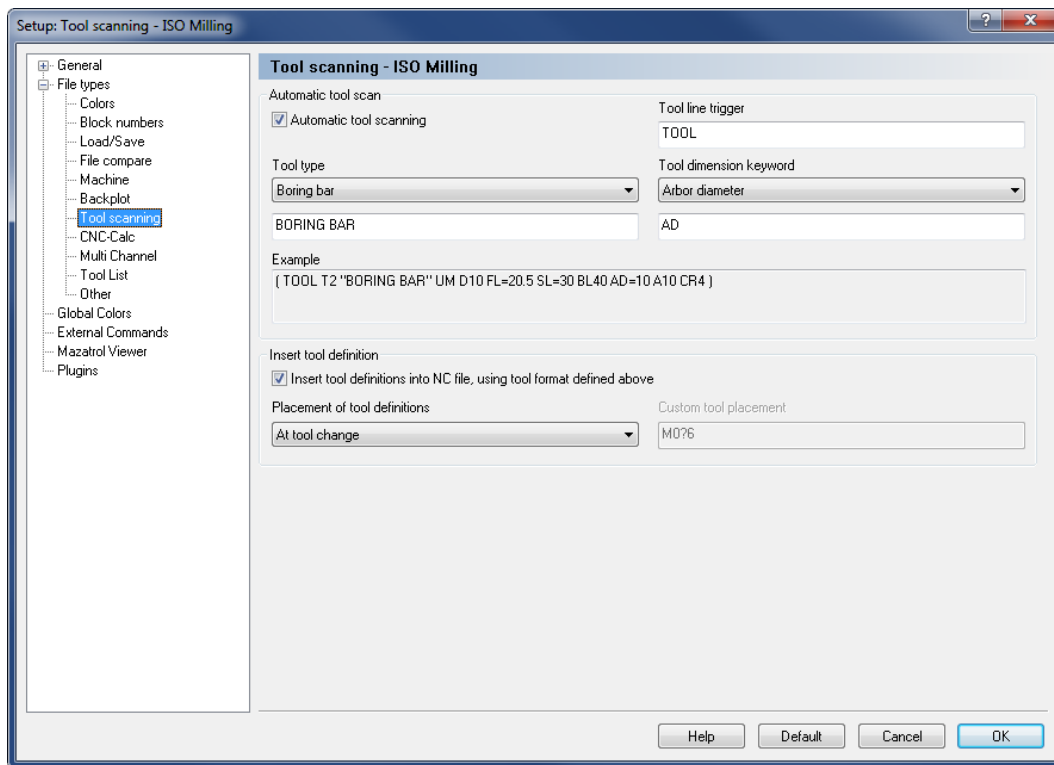
Tool scanning setup window.

Automatic tool scan

☑ Automatic tool scanning

Check this field to automatically scan tool dimensions from the NC program.

abc Tool line trigger

Enter in this field the word used as tool scanning trigger. Only normal characters A-Z and a-z are accepted. The default word is *TOOL*.

abc ▾ Tool type

Use this field to select the type of tool used for the operation. In the field below, you can edit the name of the tool, and the changes will automatically appear in the *Example* field. All characters are supported.

abc ▾ Tool dimension keyword

Use this field to select the keyword that should correlate to a tool dimension value. In the field below, you can edit the keyword of the selected tool dimension, and the changes will automatically appear in the *Example* field. Only normal characters A-Z and a-z are supported.

Example

This field shows an example of the selected tool with all its parameters.

> The fields "Tool Trigger", "Tool Type" and "Tool dimension Keyword" are only used to insert synonyms for standard keywords in the configuration. It is recommended to use the default keywords, as a change may cause problems.

Insert tool definition

Insert tool definitions into NC file, using tool format defined above

Check this option to enable to insertion of tool definitions into NC file, using the tool format defined above. The insertion is done after *Tool Setup*.

Placement of tool definitions

Use this field to select where the tool definitions should be placed in the file. You can select *Top of file*, *At tool change* or *Custom* from the drop-down list. If the selected placement fails, the tool definition is placed at line 1.

Custom tool placement

Use this field to specify a keyword or regular expression (Perl) to search for the line, the tool definition should be placed after. If the search fails, the tool definition is placed at line 1.

The custom tool placement can be specified in two ways:

Simple: In the simple mode you use a keyword such as M6, G2 to specify the tool placement. Then all the tools are placed after the first occurrence of M6 or G2.

Advanced: The advanced mode lets you use a regular expression (Perl) to specify the tool placement.

Examples:
M0?6 which matches M6 or M06
M0+6 which matches M06 or M006, but not M6.

You can use the following wildcards and repetition characters to specify the advanced custom tool placement.

Wildcards and repetition of string occurrences

ˆ	Beginning of line or string
$	End of line or string
.	Any character except newline
*	Match previous character 0 or more times
+	Match previous character 1 or more times
?	Match previous character 0 or 1 time
|	Alternative, either....or
( )	Grouping
[ ]	Set of characters
{ }	Repetition modifier
\\	Quote or special
{n}	Match previous character exactly n times
{n,}	Match previous character at least n times
{n,m}	Match previous character at least n but not more than m times

Please refer to Perl regular expression documentation for more information.

This field is only activated if **Custom** is selected in the *Placement of tool definitions* field.

6.2.7.1   Defining tools in NC-programs

> All tool lines can be grouped at the start of the program. In ISO codes, the tool lines are expected to be inside comments i.e. between parentheses. For Heidenhain programs, tool lines start with a semicolon.

Tool lines start with the trigger word **TOOL** followed by the tool number, and the tool type in inverted commas.

The default tool types are named as follows:

End mill flat:	"END MILL FLAT"
End mill sphere:	"END MILL SPHERE"
End mill bull:	"END MILL BULL"
Taper mill:	"TAPER MILL"
Drill:	"DRILL"
Center drill:	"CENTER DRILL"
Spot drill:	"SPOT DRILL"
Reamer:	"REAMER"
Boring bar:	"BORING BAR"
Counter bore:	"COUNTER BORE"
Counter sink:	"COUNTER SINK"
Chamfer mill:	"CHAMFER MILL"
Face mill:	"FACE MILL"
Slot mill:	"SLOT MILL"
Rad mill:	"RAD MILL"

> If a tool line contains an unknown tool type, End mill flat is used.

The tool type is then followed by dimension keywords and values. The default keywords are:

D = Diameter: Integer, decimal or fraction.

CR = Corner Radius: Integer, decimal or fraction.

A = Taper Angle: Integer, decimal or fraction.

FL = Flute Length: Integer, decimal or fraction.

SL = Shaft Length: Integer, decimal or fraction.

BL = Body Length: Integer, decimal or fraction.

AD = Arbor Diameter: Integer, decimal or fraction.

> You can define tools in the NC programs with and without '=' between the dimension keyword and value.

You can change between metric and imperial tools by adding UM for Metric (mm) and UI for Imperial (inch).

Examples of tool lines (with and without '='):

ISO comments

(TOOL1 "FACE MILL" UM D=42 CR=0.8 FL=6.25 SL=30 BL=50 AD=12.5)

(TOOL3 "DRILL" FL36.1234 SL=0 BL = 36 AD6 A120 UM D6 CR0)

Heidenhain comment

; TOOL12 "TAPER MILL" A=12.345 D=8 AD=21.13 CR=0 FL=20 SL=30 BL=50 UM

Imperial tool

(TOOL136 "END MILL FLAT" UI D=1/8 CR=0 FL=1 SL=2 BL=3 AD=1/8)

> If a dimension keyword is missing, the value of that keyword is taken from the default tool.
>
> Remember to insert space between parameter definitions.

6.2.8 Multi Channel

The Multi Channel option enables you to view NC programs for 2 or 3 channel machines correctly in the editor, at the same time. The NC codes for each channel are automatically displayed in their own window when the corresponding *Wait* code is specified.

To configure Multi Channel settings, select **Multi Channel** from the **File types** menu in the left window of the Editor Setup dialog, or click the **Setup Multi Channel** icon in the *Multi Channel* menu within the NC Functions tab. The Multi Channel configuration dialog is shown below.

The windows can be synchronized with the wait codes and editing can be done in all windows. Wait codes that match are shown in yellow and missing wait codes are shown in purple.
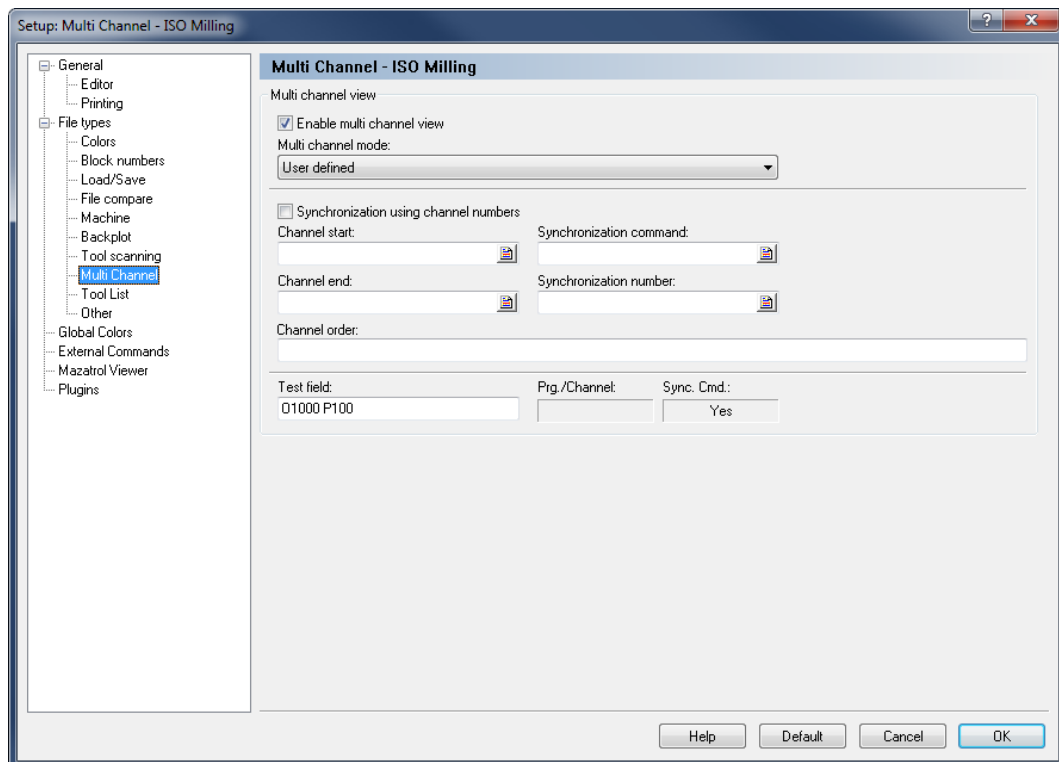
Multi Channel configuration dialog.

Multi Channel View

Enable multi channel view

Check this field to enable the multi channel view.

Multi channel mode

Use this drop-down list to specify how to read the multi channel information. You can either select one of the predefined types, or the custom type to manually define the multi channel commands.

Synchronization using channel numbers

Check this field if the synchronization commands specify which channel to synchronize with.

Example: !2 means synchronize with channel 2.

If this field is not checked, then the synchronization is done using synchronization numbers.

Example: M303 means synchronize with all channels that contain M303.

Channel start

Use this field to specify the start of a channel information, when multiple channels are stored in the same file.

Example: Specify O if each channel starts with an O number. You can specify multiple strings separated by comma. For example, if the first channel starts with G13 and the second channel starts with G14, then specify G13,G14 in this field. You can also specify the start of channel information using an advanced trigger.

Channel end

Use this field to specify the end of a channel information, when multiple channels are stored in the same file.

Example: Specify M30 if each channel ends with M30. You can also specify the end of channel information using an advanced trigger.

Synchronization command

Use this field to specify the synchronization command.

Example: If synchronization points are indicated with !1, !2, ..., specify ! in this field. If you specify the command using an advanced trigger, then you must place <> around the synchronization information.

Example: To synchronize on M300-M399, specify M3 in this field. To synchronize on all Pxxx, specify P in this field.

Synchronization number

Use this field to specify the synchronization number.

Example: If synchronization points have an ID number like WAIT(1001, channel 1, channel 2) where the ID number is 1001, that must also match in addition to the synchronization command.

abc **Channel order**

Use this field to specify the channel order, for example 1,3,2.

abc **Test field**

Use this field to test the multi channel setup.

6.2.9 Tool List

This section describes how to set up tool list triggers and generate tool lists.

To configure the tool list settings, select **Tool List** from the **File types** menu in the left window of the Editor Setup dialog. The Tool List configuration dialog is shown below.
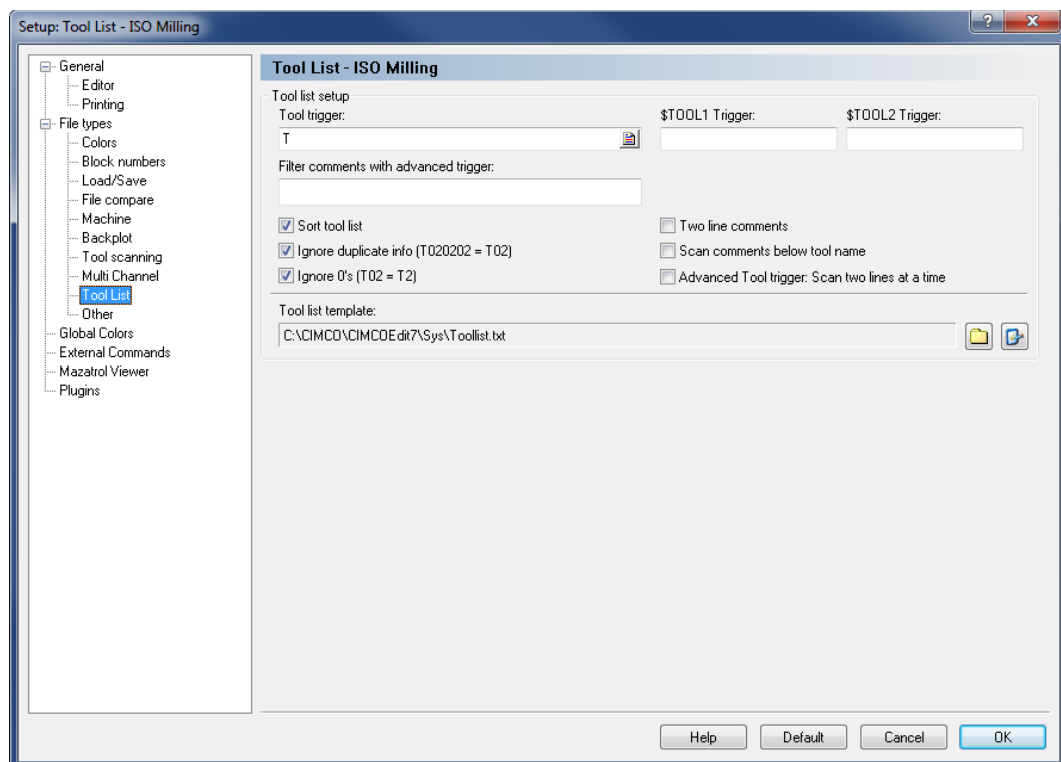
Tool List dialog.

Tool List Setup

abc **Tool trigger**

Specify the tool trigger in this field.

Example: If the machine uses T01 for tool 01, specify T in this field.

$TOOL1 / $TOOL2 Trigger

Use these fields to specify the advanced trigger that should be used to generate the $TOOL1/2 variable.

Example: To include the value specified after R into the following line: (T1 5MM DRILL R33), specify the trigger R. This will assign the value R33 to $TOOL1.

Filter comments with advanced trigger

Use this field to specify the advanced trigger (regular expression) used to filter comments. The comment lines are only used if they match the expression.

The comment filter is useful when each tool is surrounded by multiple comments as shown below:

        (*********************)
        (* BALL MILL D6   *)
        (*********************)
        T3

With the standard settings the tool list will be:

        T3 *********************

To get the correct tool description, you can add the following regular expression: [A-Z0-9]+

This ensures that the tool comment includes letters, numbers and spaces. The tool list will then be:

        T3 * BALL MILL D6 *

Sort tool list

Check this field to sort the tool list.

> When you check this field, each tool is only listed once.

Ignore duplicate info (T020202 = T02)

Ignores duplicate/magazine info in the tool number, i.e. T020202 should be listed as T02.

Ignore 0's (T02 = T2)

Ignores leading zeros in tool numbers, i.e. T02 should be listed as T2.

☑ **Two line comments**

Check this field if the tool comment consists of two lines.

☑ **Scan comments below tool name**

Check this field if the tool comment is always located below the tool change line.

☑ **Advanced Tool trigger: Scan two lines at a time**

Check this field if the Advanced Tool trigger should scan two lines at a time.

abc **Tool list template**

Use this field to specify the directory of the tool list template. Click the folder icon to the right to select a tool list template.

> ⓘ Click the icon next to the folder icon if you wish to edit the specified tool list template.

6.2.10 Other

This section describes how to configure File type specific settings, i.e. the settings made here will only be applied to the selected file type, ex. ISO Milling.

To configure Other settings, select **Other** from the **File types** menu in the left window of the Editor Setup dialog, or click the **Other Settings** icon ▣ in the *File Type* menu within the Editor tab. The following dialog appears:
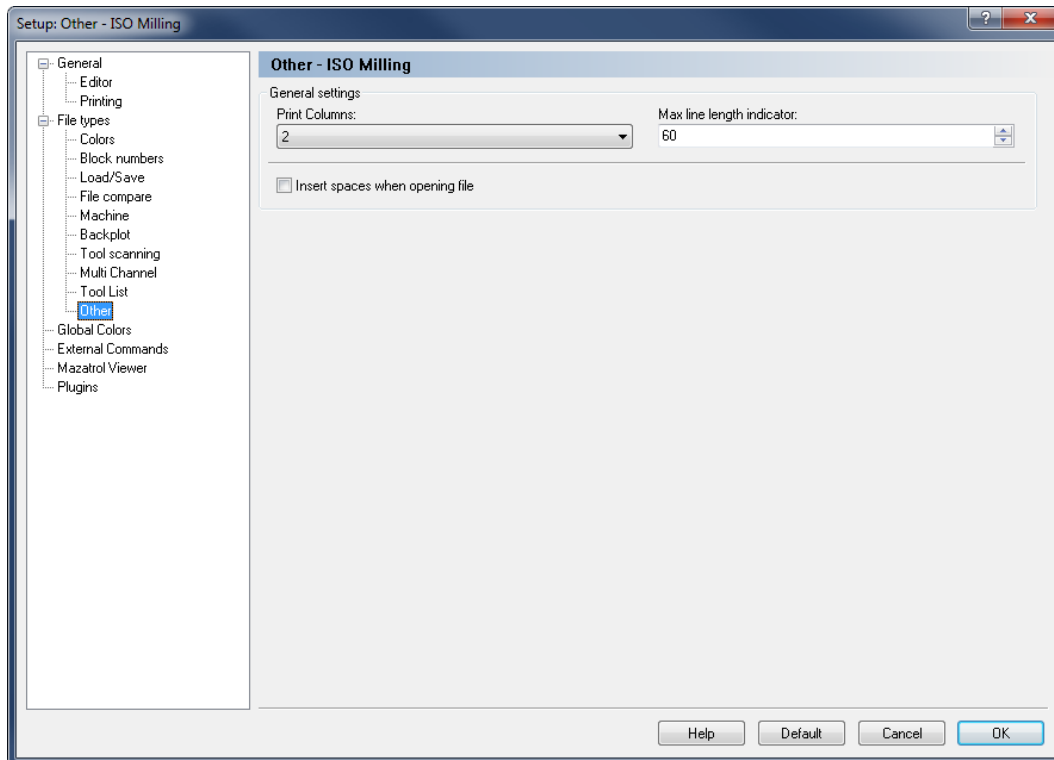
Other configuration dialog.

General Settings

Print Columns

Use this field to specify the number of columns to be used when CNC programs of the selected file type should be printed.

Max line length indicator

Use this field to set the position of the Max line length indicator.

> ⓘ   If you are using a proportional font (i.e. all letters are not the same width), the indicator does not show the exact line length.

Insert spaces when opening file

Check this field to automatically insert spaces when opening a file.

6.3 Global Colors

This section describes how to configure the general colors settings. Here, you can create your own colors for program simulations etc., and the CNC-Calc window (background, axes, contour, etc.).

To configure Global Colors, click on **Global Colors** in the left window of the Setup dialog. The following dialog appears:
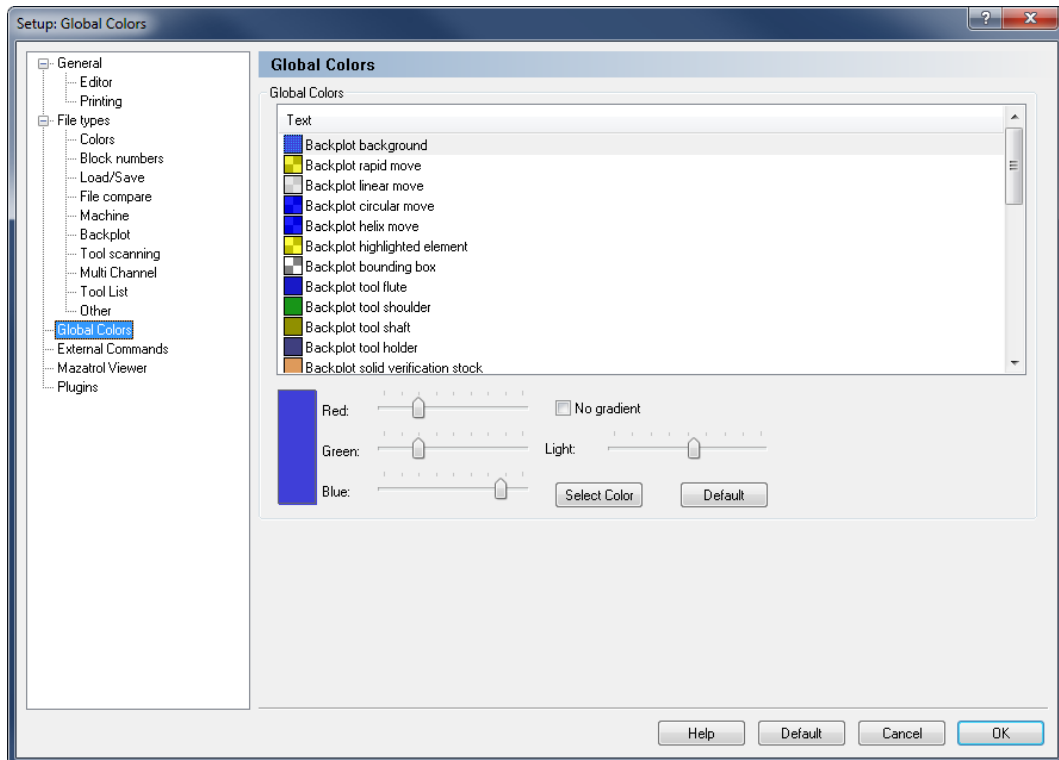
Global Colors dialog.

Global Colors

To quickly modify a specific color, select it from the list and move the sliders to get the desired color.

`Button`   **Select Color**

Click this button to select a predefined color from the list or create a custom color.
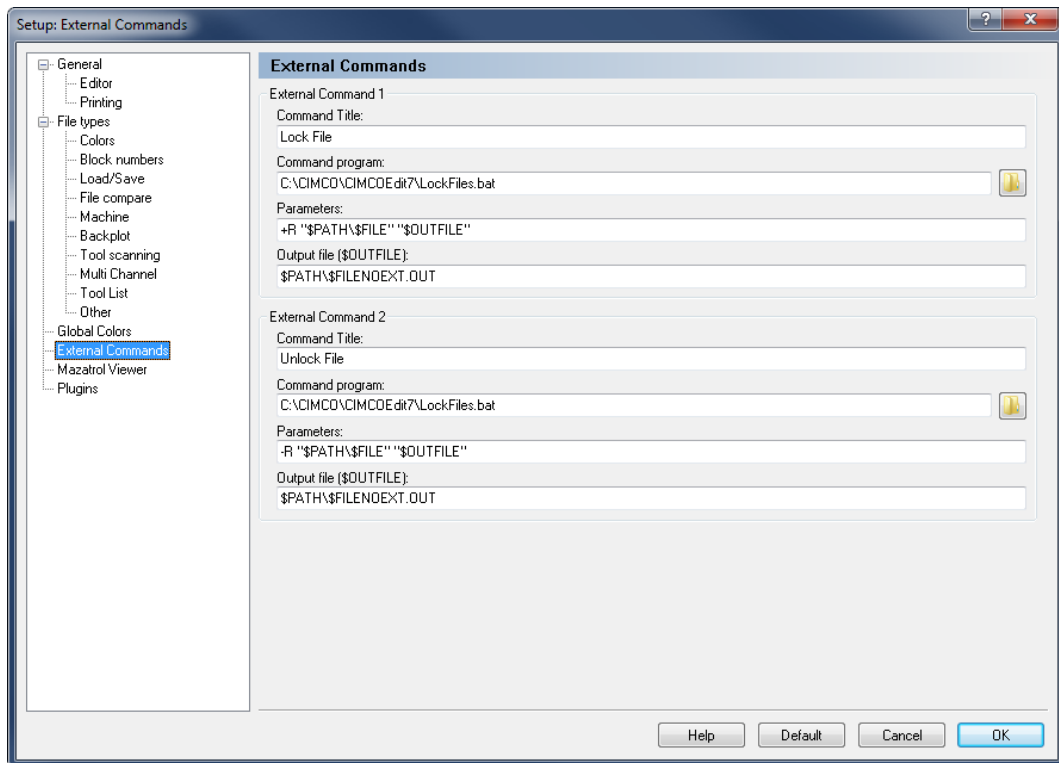
`Button`   **Default**

Click this button to apply the color by default to the selected element.

6.4 External Commands

This section describes the setup and use of external commands.

To configure External Commands, click on **External Commands** in the left window of the Setup dialog. The following dialog appears:

The External Commands dialog.

External Command 1 - 2

abc   **Command Title**

Use this field to specify the menu text shown in the *NC Functions* menu.

abc   **Command program**

Use this field to specify the application to execute. You can click the folder icon 📁 to browse for the executable file.

abc   **Parameters**

Use this field to specify the parameters to the command. The following variables are available:

- $FILE: Defines the file name without path.
- $PATH: Specifies the path of the file.

- $FILEPATH: Specifies the file name with path.
- $FILENOEXT: Defines the file name without extension.

Example: To execute the command with the parameters *-g C:\NCPRG\MOULD.NC C:\NCPRG\MOULD.NEW*, specify `-g $FILE $OUTFILE`, and then `$PATH\$FILENOEXT.NEW` in the **Output file** field below.

Output file ($OUTFILE)

Use this field to specify the name of the output file. The following variables are available:

- $FILE: Defines the file name without path.
- $PATH: Specifies the path of the file.
- $FILEPATH: Specifies the file name with path.
- $FILENOEXT: Defines the file name without extension.

Example: To execute the command with the parameters *-g C:\NCPRG\MOULD.NC C:\NCPRG\MOULD.NEW*, specify `$PATH\$FILENOEXT.NEW`, and `-g $FILE $OUTFILE` in the **Parameters** field.

> If no output filename is specified, it is assumed that the input file is modified.
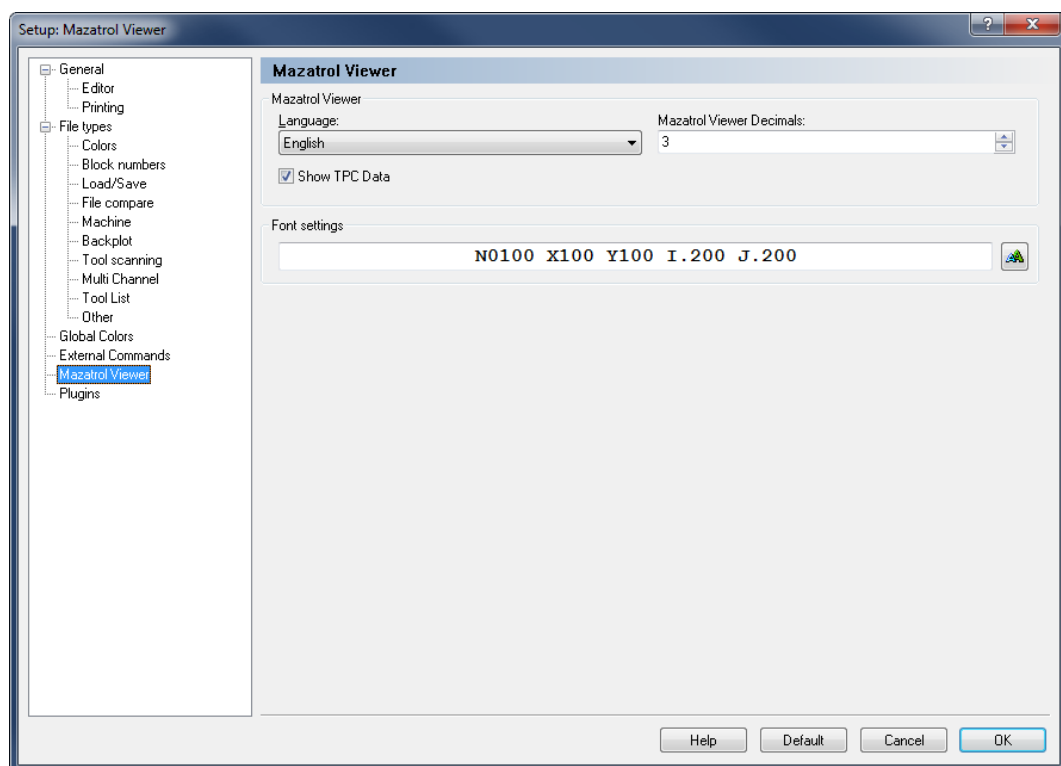
6.5 Mazatrol Viewer

This section describes how to set up the Mazatrol Viewer. The Mazatrol Viewer enables you to open and view Mazatrol files directly from your PC.

Mazatrol viewer can show binary files without header, as it determines the type from the file extension.

> Mazatrol Viewer is an optional module, and requires therefore a Mazatrol Viewer enabled license key. Mazatrol Viewer is activated under Plugins.

Mazatrol Viewer configuration dialog.

Mazatrol Viewer

Language

Select the language used for showing the Mazatrol data.

Mazatrol Viewer Decimals

Use this field to specify how many decimals should be used to display values in the Mazatrol Viewer.

☑ **Show TPC Data**

Check this field to include TPC data when viewing Mazatrol files.

Font settings

abc This field shows an example of the selected font. Click on the font button ⛰ to the right to select another font.
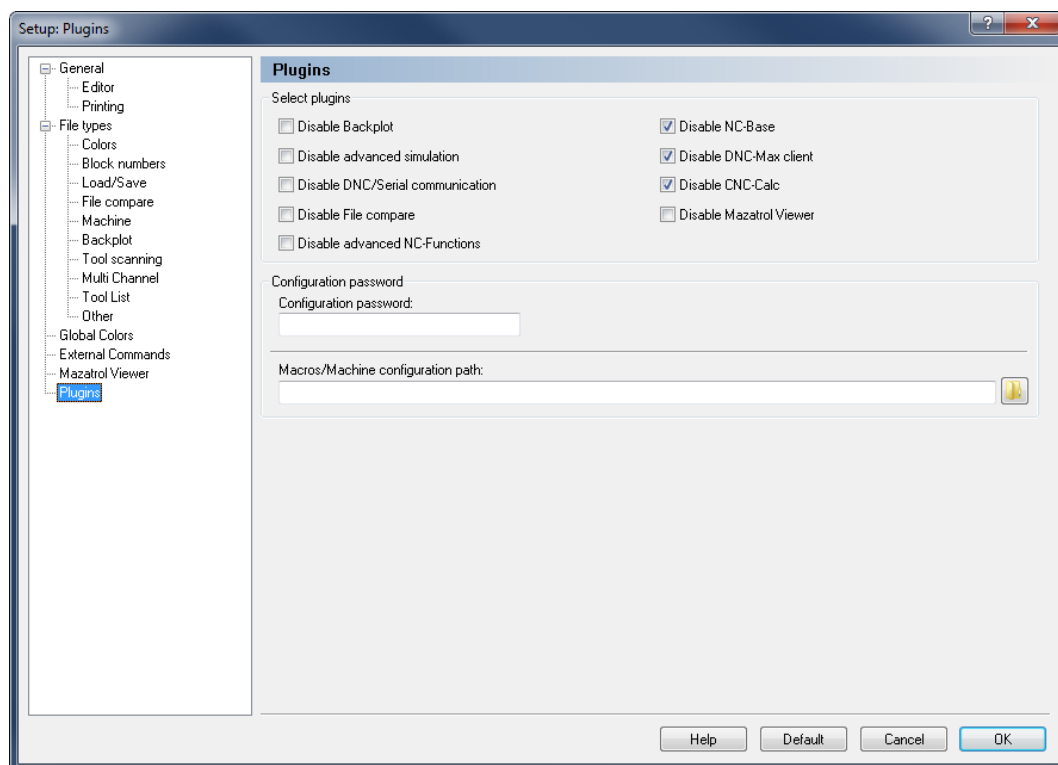
6.6 Plugins

In this section, you will find general information about how to manage Plugins.

ⓘ   The Editor must be licensed according to the plugins you wish to run in the Editor.

When you select **Plugins** in the Editor Setup window, the following dialog appears:

The Plugins dialog.

Select Plugins

☑ **Disable Backplot**

Check this field to disable the Backplot module.

ⓘ   Backplot requires a CIMCO Edit Professional license.

☑ **Disable advanced simulation**

Check this field to disable the advanced simulation module.

☑ **Disable DNC/Serial communication**

Check this field to disable the DNC/Serial communication module.

ⓘ   Requires DNC-Option extended license for CIMCO Edit.

☑ **Disable File compare**

Check this field to disable the File compare function.

☑ **Disable advanced NC-Functions**

Check this field to disable the advanced NC functions (Macros, Mirror, Rotate, ...).

ⓘ   Advanced NC-Functions requires a CIMCO Edit Professional license.

☑ **Disable NC-Base**

Check this field to disable the NC-Base Client.

ⓘ   If installed along with NC-Base Server, requires extended license for CIMCO Edit.

☑ **Disable DNC-Max Client**

Check this field to disable the DNC-Max Client.

ⓘ   If installed along with NC-Base Server and DNC-Max Server, requires extended license for CIMCO Edit.

☑ **Disable CNC-Calc**

Check this field to disable the CNC-Calc module.

> ⓘ   CNC-Calc requires an extended license for CIMCO Edit Professional.

☑ **Disable Mazatrol Viewer**

Check this field to disable the Mazak Mazatrol viewer module.

> ⓘ   Mazatrol Viewer requires an extended license for CIMCO Edit.

Configuration Password

abc **Configuration password**

Use this field to specify the configuration password.

> ⊘   If you forget the password, hold down *Ctrl+Shift* when you start the editor.

abc **Macros/Machine configuration path**

Use this field to specify where the file types (*.mac) should be stored. Click the folder icon 📁 to the right to select a path.

> ⓘ   You can use *$USERNAME* and *$COMPUTERNAME* as directory variables.

7. DNC Settings

If you wish to configure the settings for the communication between the PC and a selected machine, you can do it in two ways:

- You can select 📑**DNC Setup** from the *Transmission* menu, select the desired machine, and click the **Setup** button.

- Or you can select the desired machine from the **Transmission** toolbar and click on the DNC Setup icon 🔽 next to **Machine**.

If you select **DNC Setup** from the *Transmission* menu, this window will appear:
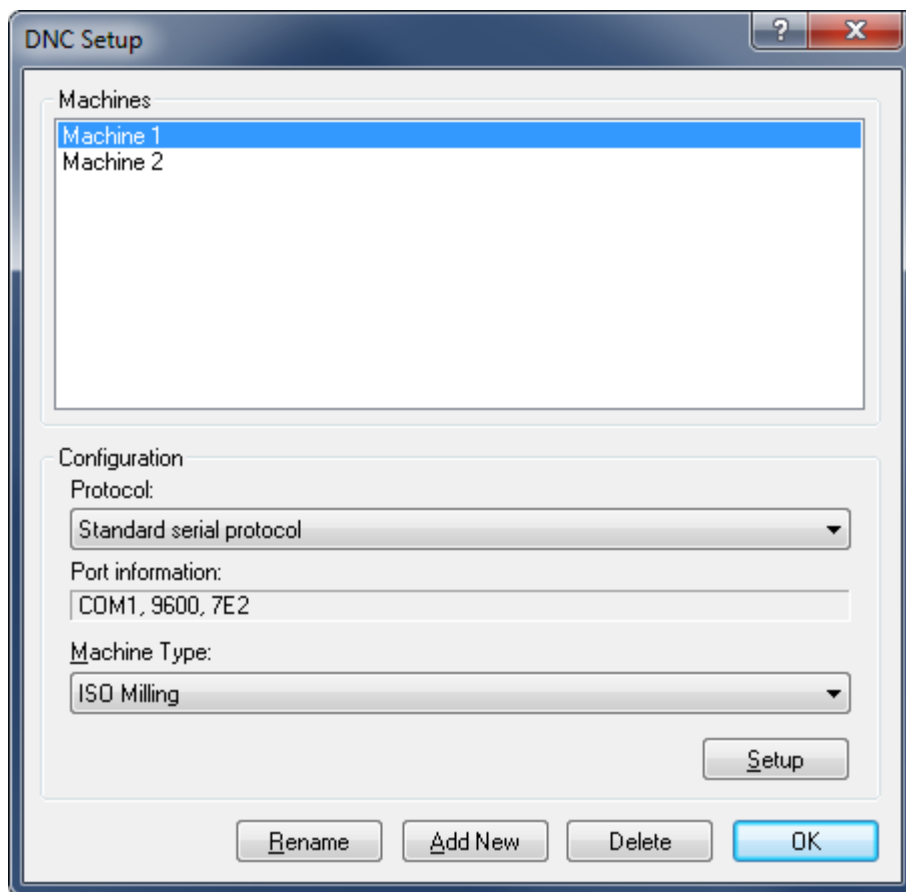
DNC Setup dialog.

Machines

If there are several machines connected to the computer, you can select which machine to communicate with or configure.

Configuration

Protocol

Here you select the communication protocol for the selected machine. The Standard serial protocol is default in CIMCO Edit v7. Optional protocols are available for CNC controls like Heidenhain, Mazak, etc.

> You cannot select a protocol which you have no license for. If tried, the computer will automatically select the earlier used protocol.

Port information

Shows in shorthand the most important of the machine settings.

Example: COM1, 9600, 7E2 (Port: COM1, Baud rate: 9600, Data bits: 7, Parity: E (even), Stop bits: 2).

Machine Type

Shows a list of different machines, which can be used as templates (e.g. Heidenhain, ISO Milling, ISO Turning, etc.)
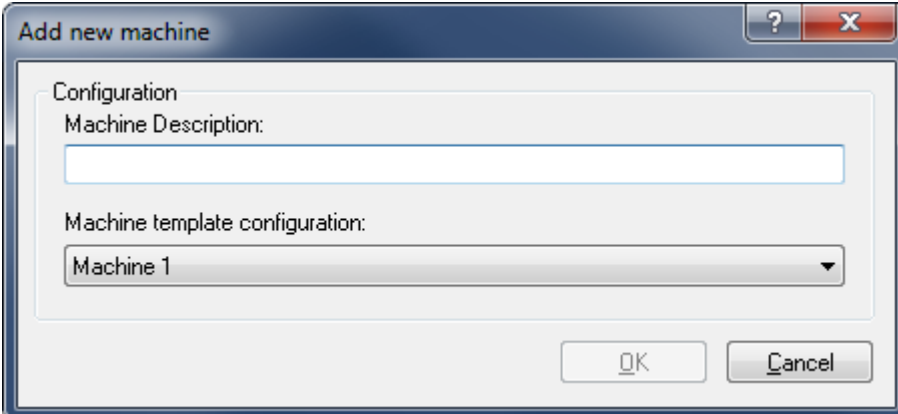
Setup

Click this button to enter the configuration dialog for the selected machine.

Rename

Click this button to rename a machine configuration.

Add New

Click this button to add a new configuration. The following window appears:

Add a new DNC machine.

Enter the name and machine template for the new machine, and then click **OK**. Click **Cancel** to cancel the operation.

[Button] **Delete**

Click this button to delete a configuration. You will be prompted if you really want to delete the machine configuration.

[Button] **OK**

Click this button to execute the new settings and close the configuration dialog.

7.1 Port Setup

To change the DNC settings, select **DNC Setup** from the **Transmission** tab, or click the DNC setup icon on the transmission menu.

The basic port configuration dialog is shown below:
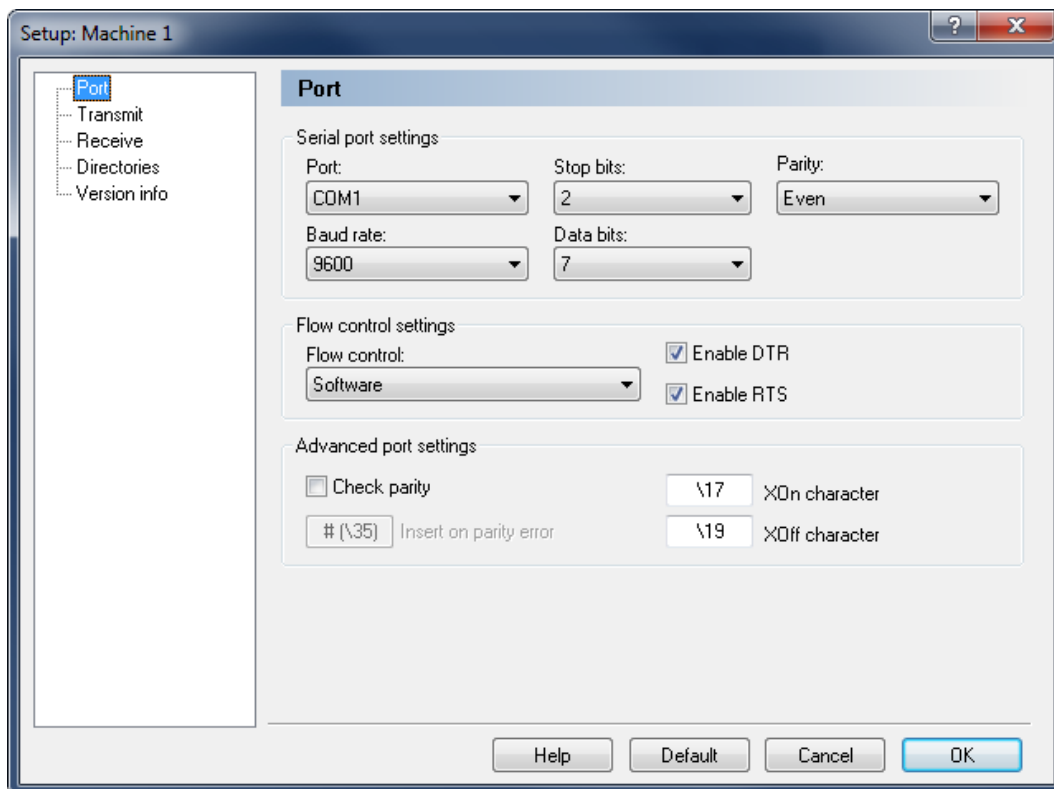
Basic port settings.

Serial Port Settings

[abc ▾] **Port**

List of the serial ports on your PC. Select the one you wish to use with this machine configuration.

Baud rate

The transmission speed in bits per second. Select the desired Baud rate from the drop-down list.

Stop bits

Select the number of stop bits appended after each character in the communication to separate the data bits.

> The number of stop bits required depends on the CNC control. Please refer to the reference manual for more details.

Data bits

Select the number of bits in each character. By default, the 7-bit ASCII code is used for NC data transmission.

Parity

The parity is an error checking method which indicates if a transmission was successful or not, by adding a 'parity bit' (8th bit when using 7-bit ASCII code for data transmission) to ensure the number of bits with the value '1' in a set of bits is *even* or *odd*.

The method can operate in different modes:

- **None**: Is selected when no parity error checking is going to be used.
- **Even** parity: An even number of 1's is used in each character.
- **Odd** parity: An odd number of 1's is used in each character.

Only for special cases (test purposes):

- **Mark** parity: The parity bit is always '1' and the line is 'low'.
- **Space** parity: The parity bit is always '0' and the line is 'high'.

Flow Control Settings

Flow control

The handshake procedure for synchronizing the communication between the computer and the CNC control. There are two types of flow control:

Software handshake uses the XOn/XOff ASCII characters.
Hardware handshake uses the RTS/CTS control lines in the RS-232 communication specifications.

You can select one of the following flow control methods from the dropdown list:

- **None**: No flow control is performed.
- **Software**: Software handshake is used for the flow control.
- **Hardware**: Hardware handshake is used for the flow control.
- **Hardware and Software**: Both Hardware and Software handshakes are used.

☑ **Enable DTR**

Check this box to set DTR high.

☑ **Enable RTS**

Check this box to set RTS high.

Advanced Port Settings

☑ **Check parity**

If you want CIMCO Edit v7 to report parity errors, then mark the *Check parity* box. Every time an error occurs, the specified character is inserted into the file at the receiving point.

Non standard XOn and XOff characters can be specified.

abc **Insert on parity error**

Use this field to specify a character to insert in the received file, if a parity error occurs while receiving data from the remote machine. If the field is left blank, no character is inserted.

abc **XOn character**

Use this field to specify the XOn character. If the field is left blank, the standard XOn character is used. If your machine needs a non-standard XOn character, you can specify it here.

abc **XOff character**

Use this field to specify the XOff character. If the field is left blank, the standard XOff character is used. If your machine needs a non-standard XOff character, you can specify it here.

ⓘ   You can specify the *XOn*, *XOff*, and *Insert on parity error* characters by entering the ASCII character, i.e. enter *$* to get ASCII character 36, or you can enter the ASCII value like this *\36*.

7.2 Transmit Settings

To configure transmit settings, click on **Transmit** in the left hand list of the Machine Setup window. The transmit configuration dialog is shown below:
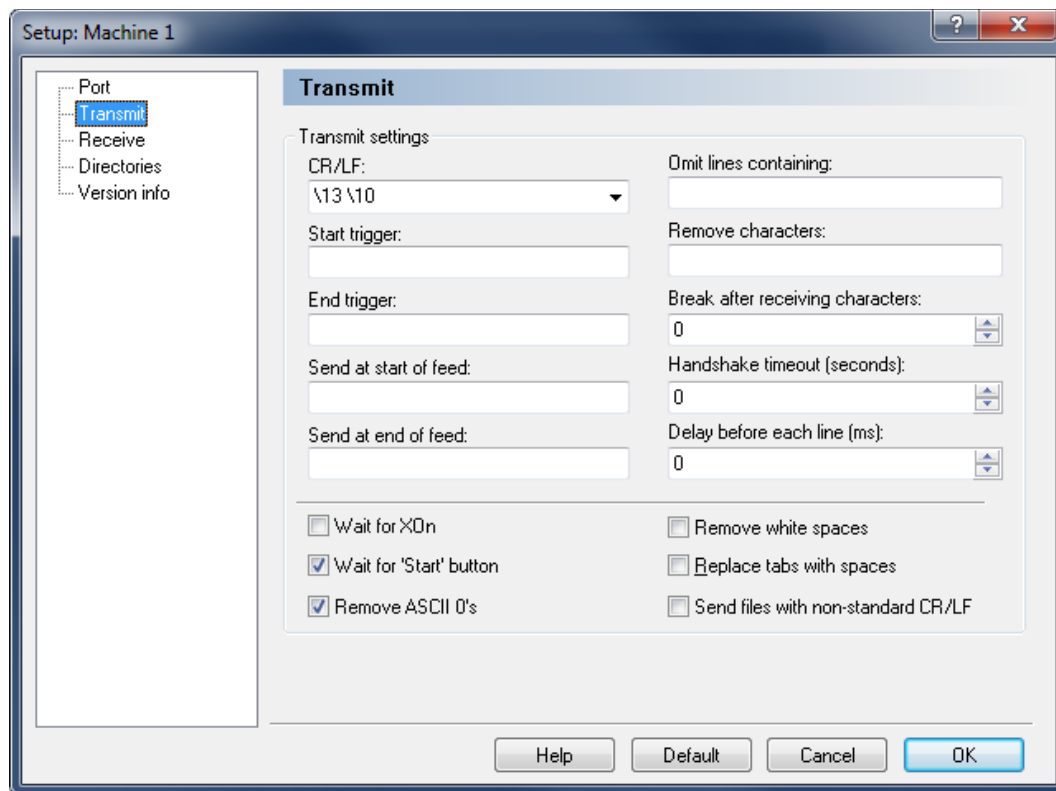
Transmission setup.

Transmit Settings

CR/LF

In this field, you can select the carriage-return or line-feed character(s) from the drop-down list, or you can enter them manually.

> ASCII 13 and ASCII 10 have no character representation, so they must be entered as \13 and \10 respectively.

Start trigger

Use this field to specify the start trigger. The DNC will start transmitting data from the first line in the file that contains the specified start trigger. If no start trigger is specified, the transmission will start at the beginning of the file.

End trigger

Use this field to specify the end trigger.

> The line containing the end trigger is not transmitted.

Send at start of feed

Here, you can specify a character or a string that should be sent at the start of the transmission.

Send at end of feed

Specify a character or a string that should be sent at the end of the transmission.

Omit lines containing

Excludes lines containing one or more selected characters. The consequence of entering anything here is that any line containing one of these characters or strings will not be transmitted.

Remove characters

Use this field to specify characters that should be removed from the transmitted data.

Break after receiving characters

Ends the transmission to the control after the specified number of characters have been sent, even if the program is not finished. If this field is left empty, any incoming characters are ignored.

Handshake timeout (seconds)

The chosen time that CIMCO Edit v7 should wait after receiving a stop flow signal (XOff and/or CTS Low) from a remote machine before it terminates the connection. If the timeout value is not specified, it will wait until a start flow is received.

Delay before each line (ms)

Specify the time in milliseconds CIMCO Edit v7 should wait before it begins transmitting a new line.

Wait for XOn

This box is checked if you wish CIMCO Edit v7 to wait for an Xon before transmitting data. This option is only available if Software handshake is enabled in the port settings.

☑ **Wait for 'Start' button**

Check this field if the transmission should not start until you click the Start button on the transmit status dialog.

☑ **Remove ASCII 0's**

Check this field if ASCII 0's should be removed from the transmitted data.

☑ **Remove white spaces**

White spaces (ASCII 32) and tabs (ASCII 9) are removed automatically from the file transmitted to the remote machine.

☑ **Replace tabs with spaces**

Converts tabs (ASCII 9) to spaces (ASCII 32) before sending the file to the remote machine.

☑ **Send files with non-standard CR/LF**

Check this field if you wish to send files that contain non-standard linefeed characters like \CR\LF\LF or \CR\CR.

ⓘ When entering characters in one of the character entry fields, the characters can be entered in two ways: You can enter the characters themselves, like *$** to get ASCII character 36 followed by ASCII 42, or you can enter the ASCII values of the characters like *\36\42*.

7.3 Receive Settings

To configure receive settings, click on **Receive** in the left hand list of the Machine Setup window. The receive configuration dialog is shown below:
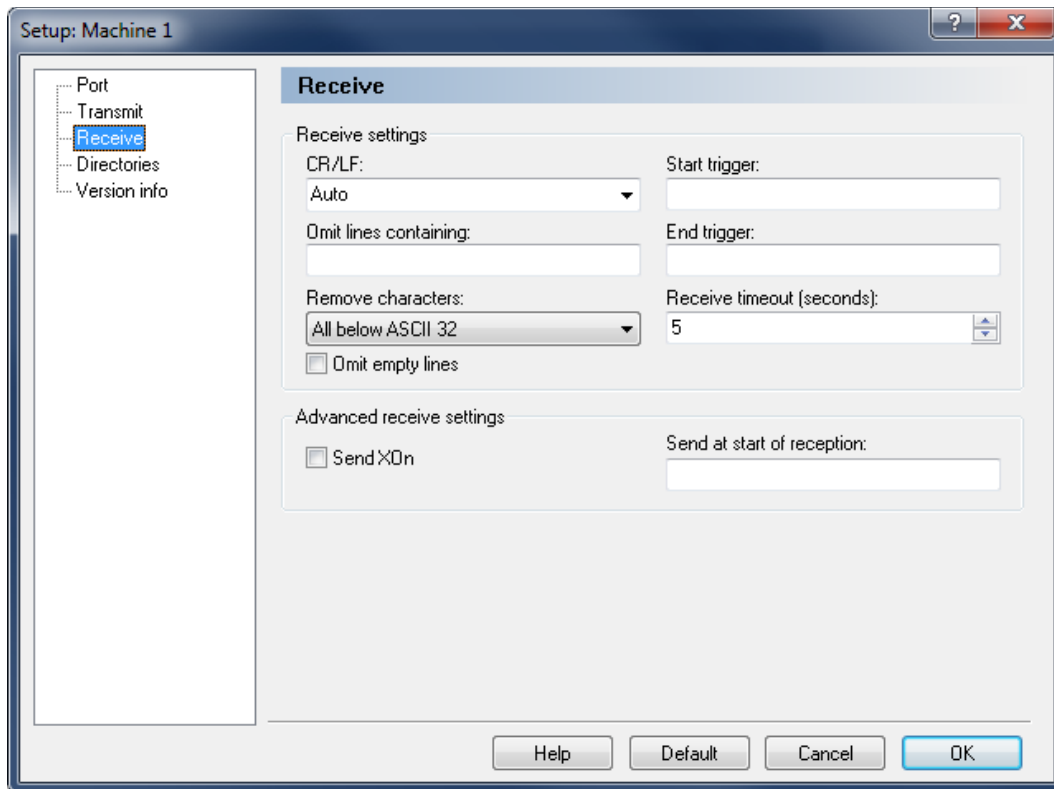
Receive settings.

Receive Settings

abc ▾ **CR/LF**

Use this field to specify the line feed character(s).

If you select *Auto*, the DNC will try to auto-detect the CR/LF combination used. If the remote machine does not use any of the standard combinations of ASCII 13 and ASCII 10, you must enter the appropriate CR/LF combination manually.

> ⓘ The normal combination is ASCII 13 followed by ASCII 10. ASCII 13 and ASCII 10 have no character representation, so they must be entered as \13 and \10 respectively.

abc **Omit lines containing**

Excludes lines containing one or more selected characters. The consequence of entering anything here is that any line containing one of these characters or strings will be discarded when received.

Remove characters

Selected characters will automatically be removed from the received stream. You can select None, ASCII 0 or All below ASCII 32 from the drop-down list if one of those is appropriate.

Omit empty lines

Check this field if empty lines should not be saved.

Start trigger

Start triggers are characters indicating that CIMCO Edit v7 should start saving incoming data. If no start trigger is used and the field therefore is left empty, CIMCO Edit will start saving at the beginning of the file.

> The start trigger is the first sequence of characters received.

End trigger

End triggers are characters indicating that CIMCO Edit v7 should stop saving incoming data. If no end trigger is used and the field therefore is left empty, CIMCO Edit v7 will continue to save until the end of the file, where a timeout occurs.

> The end trigger is the last sequence of characters received.

Receive timeout in seconds

Indicates the time delay after the last character has been received from the remote machine, until the computer concludes that the receiving operation is finished. If no end trigger is specified, you should specify a value here, or you will have to stop the receive process manually from the receive status dialog.

Advanced Receive Settings

Send XOn

When this field is checked, the DNC will send an XOn character when it is ready to receive data.

Send at start of reception

Specify a string that should be sent to the CNC when a receive operation is started.

> When entering characters in one of the character entry fields, the characters can be entered in two ways: You can enter the characters themselves, like *$** to get ASCII character 36 followed by ASCII 42, or you can enter the ASCII values of the characters like *\36\42*.

7.4 Directory Settings

In the following dialog, you can define the Transmit and Receive directories. To specify the directories, click on **Directories** in the left hand list of the Machine Setup window. The following dialog appears:
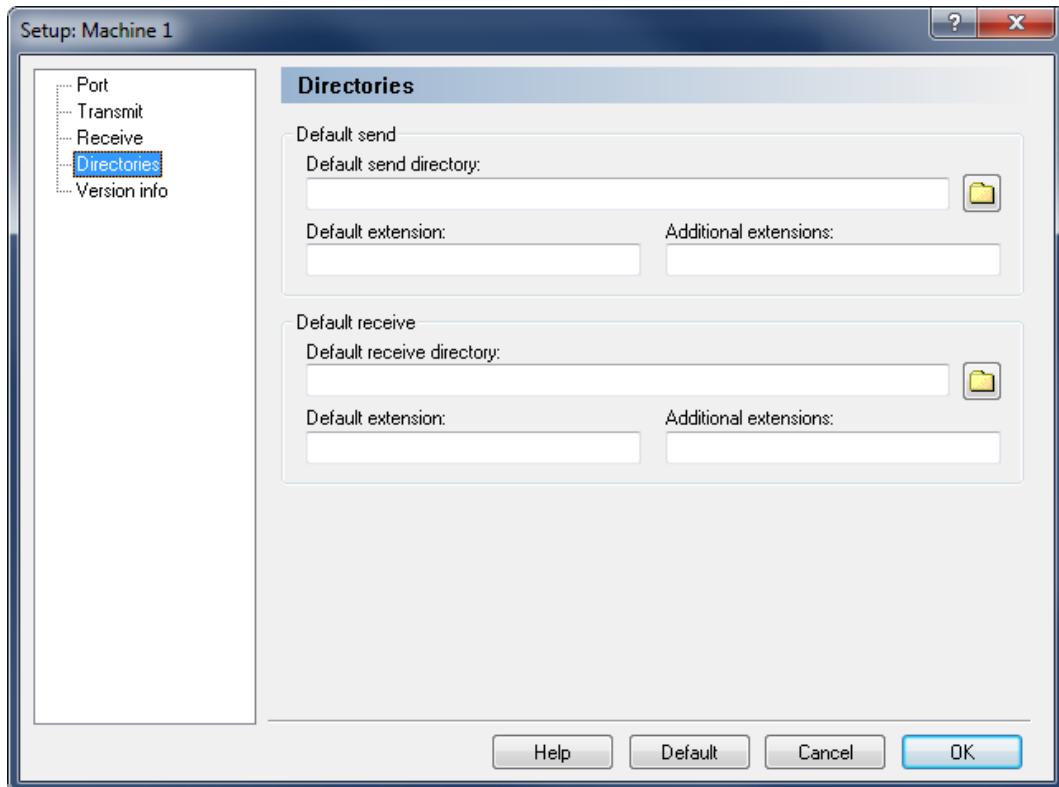
Transmit and Receive directories setup.

Default Send

abc   **Default send directory**

Specify the default send path in this field, or click on the folder icon 🗀 to the right to select a directory.

abc   **Default extension**

Use this field to specify the default extension of files to be sent.

abc   **Additional extensions**

Use this field to specify additional extensions for files to be sent.

Default Receive

abc   **Default receive directory**

Specify the default receive directory, or click on the folder icon 📁 to the right to select a directory.

abc   **Default extension**

Use this field to specify the default extension for received files.

abc   **Additional extensions**

Use this field to specify additional extensions for received files.

7.5 Version Info

If you want to find out which version of CIMCO Edit v7 you are running, just click on **Version Info** to the left. You can also get this information if from the main menu, you select **Help** and click **About**.
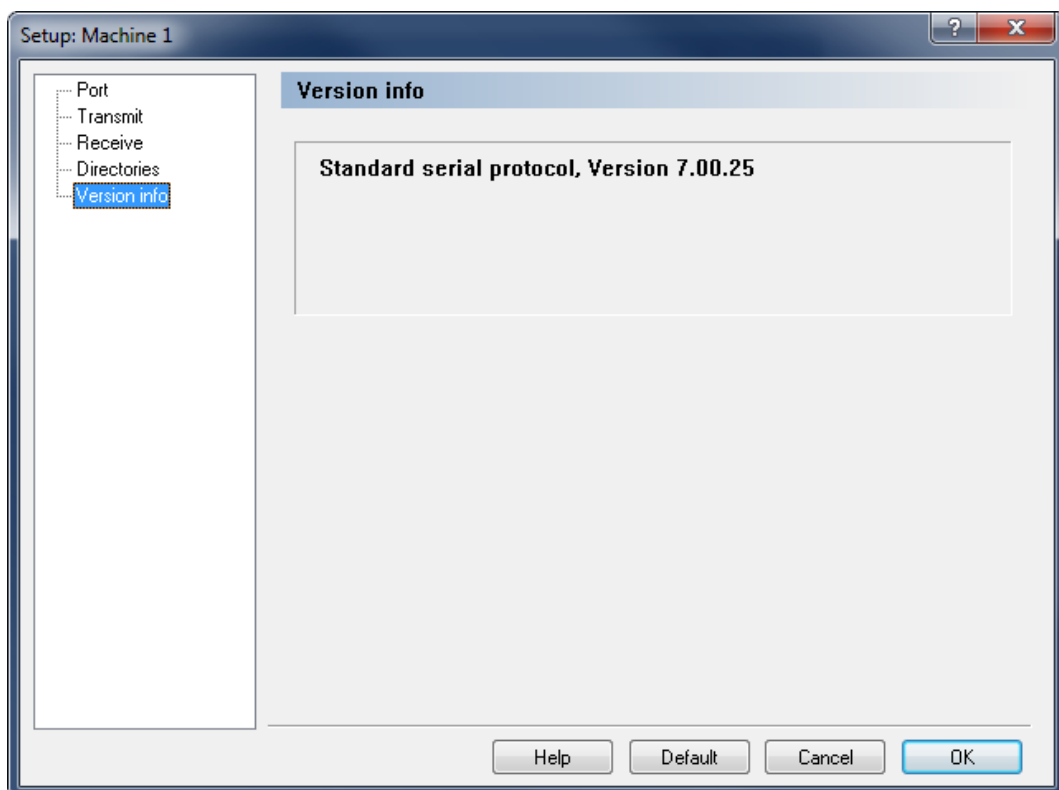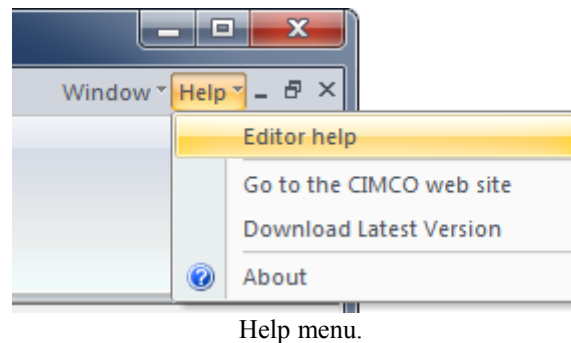

Version info.

8. Using Editor Help

This section describes how to use the help system in CIMCO products.

A standard Windows Help file is available through the drop-down **Help** menu by clicking **Editor help**.

Help menu.

8.1 Using Help in Dialogs

CIMCO Edit v7 contains multiple dialogs that provide check boxes, text fields and buttons for specific configurations.

To get help for a particular dialog item, click the **?** at the top of a dialog box, as shown below.


Click the question mark.

This will change the cursor to an arrow and a question mark indicating that you are in *Help Mode*. The new pointer is shown below.

The question cursor.

Now click on the item that you want further information about. If help is available for the selected item, a small window will pop up displaying the help text. If no help is available the general help file is launched. An example of a pop-up help is shown below.
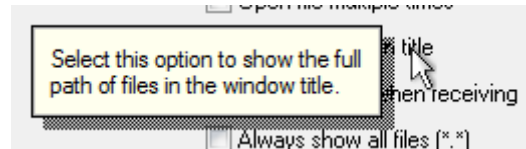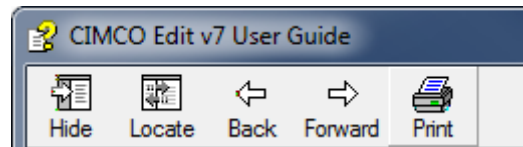
The pop-up information.

You can also display the pop-up help for a dialog item by clicking in the field and then clicking **F1**.

8.2 Printing Help Information

If you wish to print just a single section of this help file, click the **Print** button on the toolbar (shown below).

Help buttons.

If you wish to print more than a few sections, you should open the PDF version of the on-line documentation. The PDF document produces a much nicer printout.

You can download the PDF version of the user guide from the CIMCO website.

> ⓘ   You will need Adobe Reader to open the PDF file.