© 1991 - 2011 CIMCO Integration - March 22, 2011

1. License information

Information in this document is subject to change without notice and does not represent a commitment on the part of CIMCO Integration. 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 Integration.

TERMS OF USE FOR:

Software: CNC-Calc v6 Version: 6.x.x Date: March 2011 Copyright © 2011 by CIMCO Integration

Notice:

CIMCO Integration reserves the right to make changes to the CNC-Calc v6 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 Integration. 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 Integration.

Disclaimer of all Warranties and Liability

CIMCO Integration 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 CNC-Calc v6 software prove defective following its purchase, the buyer (and not CIMCO Integration, 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 Integration be liable for direct, indirect, or consequential damages resulting from any defect in the software, even if CIMCO Integration 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 Integration. No use or disclosure is permitted other than as expressly set forth by written license with CIMCO Integration.

Copyright © 2011 CIMCO Integration. All rights reserved.

THIS SOFTWARE CONTAINS CONFIDENTIAL INFORMATION AND TRADE SECRETS OF CIMCO INTEGRATION. USE, DISCLOSURE, OR REPRODUCTION IS PROHIBITED WITHOUT THE PRIOR EXPRESS WRITTEN PERMISSION OF CIMCO INTEGRATION.

The CIMCO Logo is a trademark of CIMCO Integration. Microsoft, Windows, and Windows NT are registered trademarks of Microsoft Corporation. Other brand and product names are trademarks or registered trademarks of their respective holders.

Contacting CIMCO Integration

Phone: +45 4585 6050 Fax: +45 4585 6053 E-mail: info@cimco.com Web: http://www.cimco.com

2. Table of Contents

License information		2
Table of Contents		.4
Important Notice		6
Installation		7
Overview		. 8
Mouse functions		9
Toolbars		10
7.1. File		10
7.2. View		10
7.3. Snap		11
7.4. Points and Lines		12
7.5. Arcs and Circles		13
7.6. Pattern		15
7.7. Letters		15
7.8. Modify		16
7.9. Milling Operations		18
7.10. Turning Operations		19
Menus		21
B.1. File menu		21
Configuring CNC-Calc		22
0.1. Main configuration		22
0.2. Color configuration		25
0.3. Other configurations that	affect CNC-Calc	26
Tutorial 1 2D Construction (Mi	1)	27
0.1. Before you start		28
0.2. Draw the geometry		30
Tutorial 2 CNC toolpaths and F		35
	ace milling (Mill)	55
	ace milling (Mill)	
1.1. Before you start		36
 Before you start Creation of Facing toolpa 	,	36 38
 Before you start Creation of Facing toolpation Inserting a Tool with Feed 	hs	36 38 41
 Before you start Creation of Facing toolpation Inserting a Tool with Feed Tutorial 3 Contour milling (Milling) 	hs I and Speed Calculator	36 38 41 43
 Before you start Creation of Facing toolpation Inserting a Tool with Feed Tutorial 3 Contour milling (Mil Before you start 	hs I and Speed Calculator	36 38 41 43 44
7 7 7 7 7 7	Important NoticeInstallationOverviewMouse functionsToolbars1.File2.View3.Snap4.Points and Lines5.Arcs and Circles6.Pattern7.Letters8.Modify9.Milling Operations10.Turning OperationsMenus1.File menuConfiguring CNC-Calc1.Main configuration2.Color configuration3.Other configurations that a Tutorial 1 2D Construction (Mil0.1.Before you start	 View

13. Tut	torial 4 Pocket milling (Mill)	
13.1.	Before you start	
13.2.	Creation of Pocket toolpaths	
14. Tut	torial 5 Backplot in the Editor (Mill)	
14.1.	Before you start	
14.2.	The first backplot	61
14.3.	Backplot tool setup	
14.4.	Backplot Stock setup	64
15. Tut	torial 6 Drilling (Mill)	66
15.1.	Before you start	66
15.2.	Generate a drill cycle	67
16. Tut	torial 7 Milling of Letters (Mill)	70
16.1.	Before you start	70
17. Tut	torial 8 Milling of TrueType Letters (Mill)	76
17.1.	Before you start	76
17.2.	Pocket milling letters	
18. Tut	torial L1 - Drawing the part (Lathe)	
18.1.	2D construction of a part for turning	
18.2.	Before you start	
18.3.	Draw the geometry	
19. Tut	torial L2 - Machining the part (Lathe)	
19.1.	CNC toolpaths	
19.2.	Before you start	96
19.3.	Facing the front of the stock	
19.4.	Roughing the part	
19.5.	Grooving the part	
19.6.	Finishing the part	
19.7.	Threading the part	
19.8.	Drilling the part	
19.9.	Tapping the part	

3. Important Notice

Do not run the generated NC programs as is!

This is due to the fact that no tool changes or other codes for spindle speed etc. are generated.

The same is the case in terms of program start and program finish codes. These codes could for instance be inserted with the macro functionality provided in the Cimco Edit editor.

4. Installation

CNC-Calc v6 is installed as part of CIMCO Edit v6. Please see the CIMCO Edit v6 documentation for installation instructions.

If you are upgrading from an existing installation without CNC-Calc v6, reinstallation of CIMCO Edit v6 is not necessary. Just copy the new keyfile (named "license.key") to the appropriate directory.

5. Overview

CNC-Calc v6 can draw 2D geometry and generate NC code in ISO and Heidenhain conversational format for contours and drilling cycles. The main program window (with an empty drawing pane) looks like this:

k)1							~					<i>c</i> 11 <i>c</i>	.		1	CIMC	O Ed	it v6.0	0																A firm of a	ow≁ I	-	
	Editor NC Fun Open Drawing Save Setup CNC-Calo File	*		kplot Zoo Zoo Zoo View	om Oi om Al om W			\times 1 \times 1 \times 2		iission S	•	2	**	· / Draw				Radiu		nts 🔍	ン ン ン にrcles	500	Rec	tangu lole Pa	lar Bol ttern Patt	Hol	cular B e Patte	olt	A Text Entr	a		so tu		7	H		2	1
Ur	ntitled Drawing																																					41
	CNC-Calc																																					
			1																																			
			1	•		•									•				·				•									•		•		·		
																			÷				•	•														
	Element Info																																					
	Element Into																																					
			1																																			
			1																				•															,
			1																																			
			1	•				,													÷		•	•										'				,
			1																																			
			1.					,											·				•	•												·		
																								, .										· 4	10:0	ο.		
																														H								۲.
				Mac	:hine :	Zero2	Z:	30	.0	1achin	e Zero	oX:	-18.0	D																Use'	Y.āvie	Sube	titutioe		art Dia	ameter		

To the left of the drawing pane, you see the CNC-Calc and Element Info panes. The CNC-Calc pane shows coordinate entry fields and other information about the activity you are performing at any given time, while the Element Info pane shows the statistics of any element hovered over by the mouse. To the right an example of the Element Info pane display is shown.

EI	ement Info - Arc
Center Z:	0.000
Center D:	0.000
Radius:	20.000
Length:	62.832
Start Angle:	0.000
End Angle:	180.000
Sweep:	180.000

The following sections will describe first the mouse functions, then the toolbars and menus.

6. Mouse functions

The mouse buttons are used to perform the following functions:

Left button	Selects whatever is described in the lower left corner of the program window.
Middle button (o	on most mice, pressing the scroll wheel)
	Fits/zooms the geometry drawing to the entire graphics area. This
	can also be achieved by clicking the icon.
Right button	Drag the geometry drawing across the graphics area by holding down the right mouse button as you drag the mouse.
Scroll wheel	Zoom in and out, centered on the cursor position.

7. Toolbars

7.1. File



The File toolbar handles file operations like loading and saving of files.



New Drawing clears the graphics area and opens a new document. This can also be accessed with Ctrl-N.



Open Drawing opens existing CNC-Calc v6 or DXF files. This can also be accessed with Ctrl-O. The downwards arrow next to the Open icon gives access to a recent files list, making it easy to reopen a file that you have been editing recently.



Save saves the drawing to your hard disk. If it is the first time you save the drawing, you will be prompted for a file location and name. *Save* can also be accessed with Ctrl-S.

Setup CNC-Calc lets you access the configuration for CNC-Calc.

7.2. View



The *View* toolbar controls the way we look at the drawing. Here it is possible to perform all the various zoom functions.



Zoom in centered on the middle of the graphics area. *Zoom in* can also be accessed with Page Down.



Zoom out centered on the middle of the graphics area. *Zoom out* can also be accessed with Page Up.

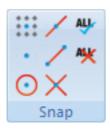


Zoom All fits the drawing to the graphics area. This can also be done by clicking the middle button on the mouse (on most mice, pressing the scroll wheel), or with Ctrl-End.

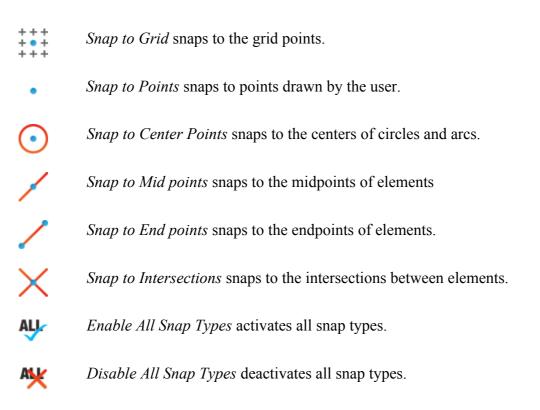


Zoom Window lets you zoom in on an area, which you select by first clicking at one corner, then dragging the rectangle and clicking at the opposite corner.

7.3. Snap



The *Snap* toolbar handles the different kinds of snap options that are available. The snap option is only available when some point position has to be selected or indicated.



7.4. Points and Lines



The *Draw Points/Lines* toolbar contains functions for drawing lines and points defined in different ways.

Point will draw a point at the selected position.

Between 2 Points will draw a line between two selected points.

Vertical will draw a vertical line. The first point selected defines the starting point (and the X coordinate), the second point selected defines the length (and need not be directly above or below the first point).

Horizontal will draw a horizontal line. The first point selected defines the starting point (and the Y coordinate), the second point selected defines the length (and need not be directly to the left or right of the first point).



Polar lets you select the starting point of a line, and you then select (or write) the angle and length of the line.

Perpendicular draws a line perpendicular to another line. You first select the line your new line is to be perpendicular to, then the starting point of your new line. You then select the length of your new line, and last you select in which direction from the starting point your new line is to go.

Parallel draws a line parallel to another line. You first select the line your new line is to be parallel to, then the starting point of your new line. You then select the length of your new line, and last you select in which direction from the starting point your new line is to go.

/

Bisector draws a line bisecting two other lines, i.e. a line that halves the angle between two lines. You first select the two lines you want to bisect, then you select the length of your new line (from the intersection of the two lines you are bisecting), and last you select which of the four possible solutions you want to keep.



Tangent Two Elements lets you draw a line tangent to two circles or arcs. You select the two circles or arcs your new line is to be tangent to, and then you select which of the solutions you want to keep.

Tangent Angle draws a line tangent to an arc or circle, at a selected angle. You first select the arc or circle your new line is to be tangent to, then the angle and length, and last you select which of the two solutions you want to keep.

Tangent Through Point draws a line tangent to an arc or circle, to a selected point. You first select the arc or circle your new line is to be tangent to, then the point it is to go though, and last you select which of the two solutions you want to keep.

Rectangle draws a rectangle where you select the two opposite corners. It is possible to define a corner radius for the rectangle (the corner radius is ignored if there is not room for it).

7.5. Arcs and Circles



The Draw Arcs/Circles toolbar lets you draw full circles (360 degree arcs) and arcs.



Center Radius lets you define the center of the circle, followed by the radius.

Two Points lets you define the circle by selecting two (diametrically opposite) points.

Three Points lets you define the circle by selecting three points on the periphery of the circle.

Tangent Two Elements lets you define a circle tangent to two elements, of a defined radius. You first write the radius, then you select the two elements the circle is to be tangent to. Last you select which of the solutions you want to keep.

Tangent Center on Line lets you define a circle tangent to one element, with its center on a line, of a defined radius. You first write the radius, then you select the line the center is to be on, and the element the circle is to be tangent to. Last you select which of the solutions you want to keep.

Tangent through Point lets you define a circle tangent to one element, through a point, of a defined radius. You first select the point the circle is to go through then write the radius, and select the element the circle is to be tangent to. Last you select which of the solutions you want to keep.

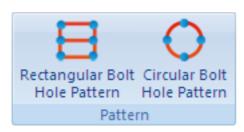
Tangent Three Elements defines a circle tangent to three elements. You select the three elements the circle is to be tangent to, and then you select which of the solutions you want to keep.

Two Points lets you draw an arc by selecting the endpoints of the arc, entering the radius, and selecting which solution you want to keep.

Three Points lets you draw an arc by selecting three points. Note that the arc created will not cross the zero degree point (3 o'clock). The selection order of the three points is unimportant.

Start and End Angles lets you define an arc by its center point, radius, starting angle, and end angle.

7.6. Pattern





Rectangular Bolt Hole Pattern defines a rectangular bolt pattern. You select the start point (one of the corners), and then enter the step in X, step in Y, number of holes in X, number of holes in Y, and the hole diameter.

Circular Bolt Hole Pattern defines a circular bolt pattern. You select the center of the bolt pattern, select the radius of the bolt pattern, and then enter the start angle, step angle, number of holes, and the hole diameter.

7.7. Letters



The *Letters* toolbar lets you draw two kinds of letters: simple letters and TrueType letters. The simple letters are like the letters used on drawings. These letters can be used to mill, for instance, a part number on a part. The TrueType letter is more artistic, and any TrueType font installed in the Windows operating system can be used.



Simple Text Linear Alignment defines simple text written on a line. You enter the starting point, angle of the line, then the distance between and height of the letters. It is also possible to select the horizontal and vertical alignment of the text.



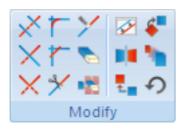
Simple Text Circular Alignment defines simple text that is written on a circle. You select or enter the center and radius of the alignment circle, then the start angle, space between and height of the letters. It is also possible to select the horizontal and vertical alignment of the text on the circle.

True Type Text Linear Alignment defines True Type text written on a line. You enter the starting point and angle of the line, and then the height of the letters. It is also possible to select the horizontal and vertical alignment of the text.



True Type Text Circular Alignment defines True Type text written on a circle. You select or enter the center and radius of the alignment circle, then the start angle and height of the letters. It is also possible to select the horizontal and vertical alignment of the text, on the circle.

7.8. Modify



The Modify toolbar modifies the geometry in different ways.



Trim To Intersection will trim the selected element to the nearest intersection(s). Select the element to be trimmed on that part to be removed. It is then trimmed to the intersection(s) nearest the point where it was selected. The trimmed element is also broken in two if there are intersections on both sides of the selected point.



Trim One Element will trim one element to another. Select the element to be trimmed first, on the section to be kept, and then select the element to trim to. This kind of trimming may extend the trimmed element to the intersection with the element it is trimmed to.



Trim Two Elements will trim two elements to each other. Select the two elements to be trimmed, on the sections to be kept. This kind of trimming may extend the trimmed elements to their intersection.

Fillet Elements creates a fillet between two elements, with a fillet radius you select. It is optional whether the two elements should also be trimmed to the fillet.

Chamfer Elements creates a chamfer between two elements, with a define angle and/or distance. It is optional whether the two elements should also be trimmed to the chamfer.



Break Elements will divide an element into two pieces. First select the element to be broken into two, and then select the point at which it should be divided.



Join Elements will join two selected elements into one.

Delete will delete the elements you select. They can be restored with the *Undo* function (the \mathfrak{O} icon in the *Modify* toolbar).



Delete Duplicates will delete elements that are duplicated in the drawing.



Offset Elements will offset the elements you select by a specified distance. It is optional whether the original element should be kept. *Mirror Elements* will mirror the elements you select along a line selected as the mirror axis.

Translate Elements will translate the elements you select along a vector defined by selecting two points. It is optional, whether the original should be kept, and it is possible to create multiple copies, where each copy is translated one step further along the selected vector.



Rotate Elements will create one or more copies of the selected elements, rotated around a selected point, at a specified angle per copy. It is optional whether the original should be kept.

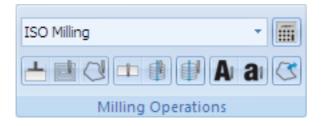


Scale Elements will create one or more copies of the selected elements, scaled about a selected point by a specified scale factor. It is optional whether the original should be kept.



Undo will undo one or more operations. This can mean deleting elements created, restoring deleted elements, and/or undoing modifications to elements. Undo can also be accessed with Ctrl-Backspace.

7.9. Milling Operations



The *Milling Operations* toolbar allows the user to perform various operations used in the manufacturing of parts. All the operations can be exported directly to CIMCO Edit or to the clipboard, for insertion in a user-defined location.

Face Milling creates a facing operation based on a selected outline contour.

Pocket Milling creates pocket operations for one or multiple pockets. These pockets can contain multiple islands, or none. In a single operation, it is possible to make both roughing and finishing cuts, but only with one tool.

Contour Milling creates operations for contour milling. A contour operation can machine multiple contours with roughing and finishing cuts, but only with one tool.

Drilling creates drilling operations for hole drilling. From a drawing, the hole positions can be selected with the use of a filter, or by simply indicating the hole position. If multiple holes are drilled, they can be arranged in both rectangular and circular patterns.

Helix Milling can generate operations for helix drilling. Like normal drilling, multiple holes can be selected with the use of filters, or by selecting the individual circles from the drawing.

Thread Milling can create threading milling operations. The threads can be inside or outside, and can be created for tools with one or multiple teeth. Again, multiple threads can easily be selected with the use of the filter function.

Ø



Simple letter milling can generate operations for milling simple letters on the drawing. These letters will have to be drawn using the simple letter function, but then all letters can be selected with the window functions, and machined in operations based on their start and end depths.



True Type letter milling can create operations to mill the outline and/or the interior of the individual letters. The letters can be selected with the window function, and all letters with the same parameters can be machined in one operation.

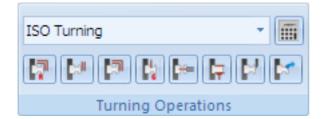


Export Contour can export a contour that the operator has selected on the drawing. If a controller has smart canned cycles, for example a specific pocket operation, the user can create a macro in the editor to support this, and then export the actual contour for insertion in the canned cycle.



Feed and Speed Calculator is used to generate tool changes, or to simply calculate the feed and speed, based on the data of a specific tool.

7.10. Turning Operations



The *Turning Operations* toolbar allows the user to perform various operations used in the manufacturing of parts. All the operations can be exported directly to CIMCO Edit or to the clipboard, for insertion in a user-defined location.



Roughing will create a roughing operation based on a selected outline contour. In this operation it is possible to create both roughing and semi-finishing cuts, but only with one tool.



Facing will create a facing operation. This operation can contain both roughing and finishing cuts, but only with one tool.

Finishing will create a finish operation based on a selected contour. This operation will make only one toolpath that fits the selected contour.

Grooving creates a grooving operation, based on a selected contour. This will enable machining of the areas that could not be machine with either the Roughing or Finishing operation.

H=

Þ

Drilling generates operations for drilling a center hole in the part. The drilling operation will be performed as one continuous motion or as pecking with or without dwelling at the end of each plunge.

Threading creates threading operations for inside and outside threads. The geometry of the thread can be entered by hand or selected from tables. The thread geometry can be normal or conical.

Cutoff generates an operation for cutting off the part from the stock. In the operation it is possible to define the corner geometry as sharp, round or chamfered.

Export Contour can export a contour that the operator has selected on the drawing. If a controller has smart canned cycles for example a specific roughing operation, the user can create a macro in the editor to support this, and then export the actual contour for insertion in the canned cycle.

246091

Calculator The feed and speed calculator is used to generate tool changes, or to simply calculate the feed and speed, based on the data for a specific tool.

J

h

8. Menus

Most of the CNC-Calc v6 functions are accessed through the CNC-Calc menu, however, a few are logically located under the *File* menu.

8.1. File menu

Close	Closes the active file. If the active file has been modified, you will be prompted to save it. <i>Close</i> can also be accessed with Ctrl-F4.
Close All	Closes all open files. If any files have been modified, you will be prompted to save them.
Save	Saves the drawing to your hard disk. If it is the first time you save the drawing, you will be prompted for file location and name. <i>Save</i> can also be accessed with Ctrl-S.
Save As	Saves the drawing under a different name.
Exit	Closes CIMCO Edit v6, and thus also CNC-Calc v6. If any files have been modified, you will be prompted to save them. <i>Exit</i> can also be accessed with Alt-F4 (Alt-F4 is a Windows standard).

9. Configuring CNC-Calc

Perhaps the most important thing to remember when configuring CNC-Calc is that the configuration is specific to each machine type.

9.1. Main configuration

The main configuration is entered most easily by selecting the icon Setup CNC-Calc

from the CNC-Calc File toolbar.



It is important to select the correct machine type, and it should be noted that the selection between ISO and Heidenhain conversational NC code output is made by the template used when creating the machine type. The window below shows the main configuration dialog.

Setup: CNC-Calc - Heid	lenhain	? 🔀
General Editor Printing	CNC-Calc - Heidenhain	Machine type:
 File types Colors Block numbers Load/Save 	 Turning Diameter programming Arc center is specified as diameter 	Heidenhain Arc Type: Absolute arc center
File compare Machine Backplot Multi Channel Tool List Other Global Colors External Commands Mazatrol Viewer Plugins	 Always add sign Modal X/Y values Show grid Output G0x Output FMAX as F9999 	Number of decimals 3 Image: Constraint of the constraint
	Grid Grid size: 10.00	0.0005 Show sub-grid Show origin Default Cancel OK

The top part of the CNC-Calc configuration dialog contains the settings for toolpath output, with the settings for the drawing grid at the bottom section. The correct settings for toolpath output depend on the machine and the control that is to run the NC code. If in doubt, you should consult the programming manual for your specific machine.

The settings for toolpath output are:

Turning	Select this for turning (lathe) output. This option is unavailable for Heidenhain conversational NC code output.
Diameter program	Selects whether X axis output is in diameter measurement or in radius measurement. This option is only available if turning (lathe) output is selected.
Arc center is speci	fied as diameter (lathe) Selects whether the I value for arcs is specified in diameter measurement or in radius measurement. This option is only available if turning (lathe) output is selected.
Always add sign	Selects whether to always output the sign of the coordinate (giving a + sign on positive and zero coordinates), or whether the sign is only output on negative coordinates.
Modal X/Y values	Selects whether the coordinates are modal or not. Modal coordinates means the coordinates are only output when changed, while non-modal coordinates means both X and Y are output on every line, regardless of whether they are changed or not.
Show grid	Select this to make the grid visible.
Output G0X	Is used to format the ISO G-Code output format. With <i>Output</i> $G0X$ disabled G01 will be formatted as G1.
Output FMAX as	F9999
	This only affects Heidenhain machines. Here the Rapid moves are normally formatted as FMAX but with the field enabled the format will be F9999 that the older Heidenhain controllers use.
Machine Type	Select the machine that the output should be formatted for. For milling this could be Heidenhain or ISO milling.

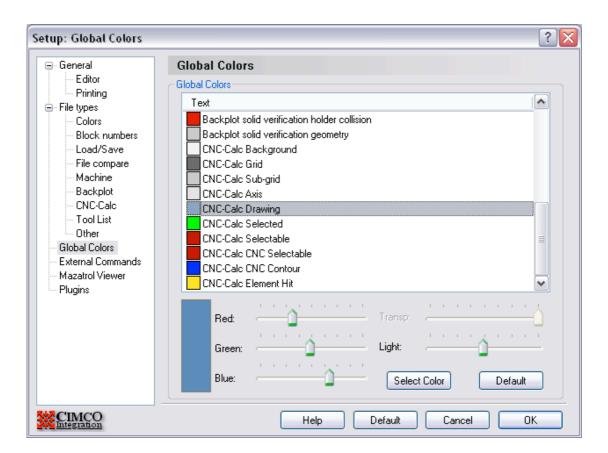
Absolute arc o	enter	
		The arc center is given in absolute I and J coordinates.
Relative to sta	rt	The arc center is given in I and J coordinates, relative to the start point of the arc.
Radius (R) va	lues	The arc radius is given (with the address R), rather than the arc center.
Number of de	cimals	All coordinates are rounded to this number of decimals, and if padding with trailing zeroes is selected below, coordinates are padded with trailing zeroes to this number of decimals.
Trailing 0's		
X123.000		inates are padded with trailing zeroes to the number of als selected above, if rounding results in fewer non-zero als.
X123.0	coordi	e numbers are output with one trailing zero. Other inates are output rounded to the number of decimals ed above, without trailing zeroes.
X123.	zeroes	e numbers are output with a decimal point, with no trailing b. Other coordinates are output rounded to the number of als selected above, without trailing zeroes.
Maximum arc out	put ang	gle
	contro handle	s primarily used in milling, and it allows the operator to I the maximum sweep of the arcs. Some controllers can not e arcs with a sweep larger than 180 degrees, and here it is I to ensure that these arcs will not be generated.
Rotary axis		xis substitution is used, this field contains the address letter axis used in this substitution.
Rotary axis linear	ization	tolerance
	transfo	axis substitution is used, all Y-axis movements are ormed to an axis rotation. To control the precision of this ormation, the entered linearization tolerance is used.

The grid settings are:

Grid size	This sets the spacing between main grid points.
Show sub-grid	Select this to have a sub-grid visible when zoomed in to a degree that shows few main grid points.
Show origin	Select this to have lines along X and Y zero visible.

9.2. Color configuration

The *Color configuration* is entered by selecting *Global Colors* in the configuration tree (after entering the configuration and selecting the correct machine type), and scrolling down to the CNC-Calc colors in the list.



To change the color of a CNC-Calc element, either left click the element in the list to select it and click the select color button , or you can double-click the element in the list. The color can then be picked from a standard palette, or a custom color can be defined.

9.3. Other configurations that affect CNC-Calc

There are a few other configurations in CIMCO Edit v6 that affect CNC-Calc.

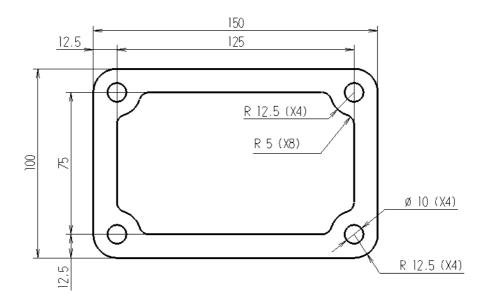
Machine type template

As has already been mentioned, the template used when creating a machine type determines whether the NC code output from CNC-Calc is in ISO or Heidenhain conversational NC code.

Machine under file types

In the *Machine* dialog under *File types* in the configuration, the *Comment start*, *Comment end* and *Decimal point* settings are used by CNC-Calc, when creating NC code.

10. Tutorial 12D Construction (Mill)



This tutorial demonstrates one of many ways in which the 2-dimensional part above can be drawn in CNC-Calc v6. Since the part consists of a number of similar elements and since its part-elements are symmetrical, only a subsection of the part needs to be drawn. The rest emerges from mirroring. Finally joining the mirrored elements with straight lines completes the part.

This tutorial demonstrates the use of the following functions:

- Draw a rectangle with a corner radius
- Draw a circle with known center and radius
- Draw vertical and horizontal lines from known points
- Offset a circle
- Make curves between elements
- Delete elements
- Mirror elements about lines
- Join end points with straight lines
- Save file with a given name

10.1. Before you start

The first thing to do before drawing a new part is to set the toolbar parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

•		CIMCO Edit v6.0			_ = ×
Editor NC Functions Backplot File Co	ompare Transmission CN	VC-Calc			Window * Help *
		λ 🗈 🗈 +) (+	X Del → Append File Tinsert File X Find Previous X Find Next Find Kay Replace	Kext Tool change Previous Tool change Number Find	
	Setup: Plugins		?	3	
A	General Editor Editor Editor Printing File types Elock numbers Load/Save File Colors Glock numbers Colors Glock numbers Colors Colors Glock numbers Colors Colors	Plugins Select plugins Disable Backplot Disable DNC/Selial communication Disable DNC/Selial communication Disable DNC/Selial communication Configuration password Configuration password Type file path:	Disable NC Base Disable DNC-Max client Disable DNC-Calo Disable Mazatrol Viewer		С
B		Help	Default Cancel OK		
					it 6.0
				Licensed to Frank Carlsen DEMO Key	INS 2:48:23 PM

To make a new drawing you must click on **CNC-Calc** menu indicated by **A** and then select **New Drawing** indicated by **B**.

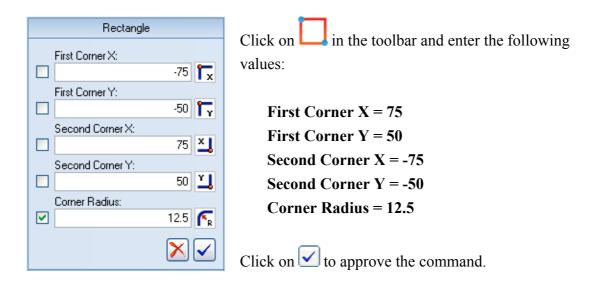
When the new drawing is opened the following window should now be displayed:

🚯 🖬 " 🗃 🔹			CIMCO Edit v6.	D		_ = ;
Editor NC Functions	Backplot File	Compare Transmission	CNC-Calc			Window * Help * 🗕 🗗
- Save	Zoom All	×		O SY H	n a	ISO Turning
New Drawing Setup CNC-Calc In File	Tana Zoom Windo View	w X Y 🐮 🛃 🔊 Modify	😳 🗶 🚺 🛏 🛠 🛄 Radiu	r Two s Points A A A Wraw Arcs / Circles Pattern	nole Pattern Entry 👾	Turning Operations
LUntitled Drawing	view	Modily	Shap Diaw Points / Eres E	naw Arcs / Circles Patt	em lext	4 b>
CNC-Calc	1		$ \longrightarrow $			
chocat				`		
			· · · · · · · ·	· · A · · · · ·		
	<i>.</i> .	 П				
		Винин				
	1					
				• • • • • • • •		
	1					
Element Info	· · · · ·					
						40.00 •
					 	40.00
	Machine Zero	Z: Machine 30.0	Zero X: -18.0			Part Diameter
	sition for start and end		10.0			xis o ubstitution

Note If you hold the cursor over an icon a short description of the icon's functionality will appear.

10.2. Draw the geometry

Draw a rectangle with sides = 150, height = 100 and corner radius = 12.5



Draw a circle with radius = 5 defined by its center

Circle Center And Radius	
Center Point X:	
Center Point Y:	
	0
Circle Radius:	
5.0	0 💽
×	

Click on in the toolbar.

Enter Circle Radius = 5

Activate the snap function (circles center points).

Snap to the center of the left topmost corner arc.

Left-click to add the circle.

Draw a vertical and horizontal line defined by its end point and length

Vertical Line	٨
Start Point Of Line X:	А
✓	p
Start Point Of Line Y:	г
VÎ₁	
Line Length:	С
-20.0	
	L
	S
$\mathbf{\times}$	

	Activate the snap function (circles center
:	points) and (midpoint of lines).
•	Click on and enter the following into the dialog
	Line Length = -20.0
	Snap to the center of the left topmost corner arc.
	Click to add the vertical line.
	Snap to the midpoint of the topmost horizontal line.

Click to add a vertical line. This will serve as a mirror line for the mirroring of our part about the Y-axis.

Similar to the above draw a horizontal line with length = 20 from the same center. This time Line Length is set to 20 and next the mirror line is added from the center of the left horizontal line (the X-axis, see picture below).

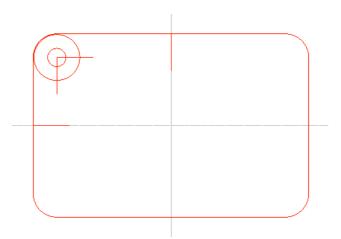
Offset a circle

Click on *in the toolbar and enter:*

Offset Distance = 7.5 (12.5 - 5 = 7.5).

Click on the circle and select the outermost of the appearing circles.

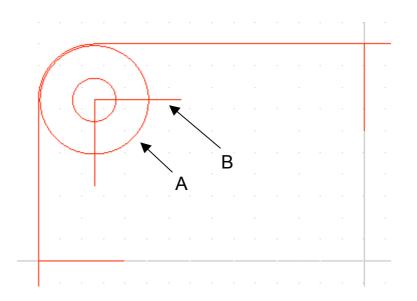
Your drawing should now look like the one below.



Create a fillet between elements

Click on T in the toolbar.

Enter Fillet Radius = 5.



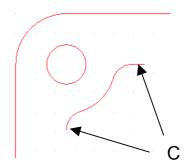
Click on the circle by \mathbf{A} and on the line by \mathbf{B} .

From the possible solutions select the part of the circle, which makes the right fillet. In the picture below you can see how it should look.

Do the same by the vertical line.

The topmost left part of your drawing should now look similar to the picture on the right.

Click on and delete the two lines by **C** (the ones pointing away from the center of the circle).



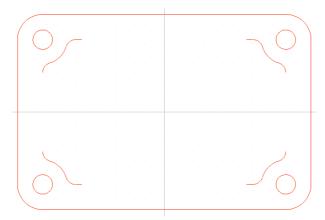
Mirror elements about mirror lines

Click on in the toolbar.

First, click on the vertical mirror line.

Click on all the elements, which should be mirrored (the circle and the inner corner). You can hold down the left mouse button while dragging out a window around the elements.

Now do the same and mirror along the horizontal mirror line. Continue mirroring until your drawing looks similar to the one below.



Click on and delete the mirror lines.

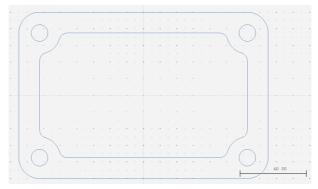
Connect the inner elements

Activate this snap function / (*snap to end points*).

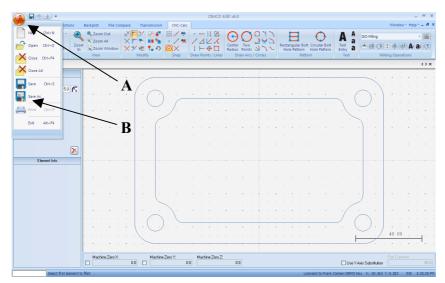
Click on / in the *Draw Points/Lines* toolbar.

Snap to the two arcs' end points and add the remaining horizontal and vertical lines to finish the part.

Your part should now look as the one below.



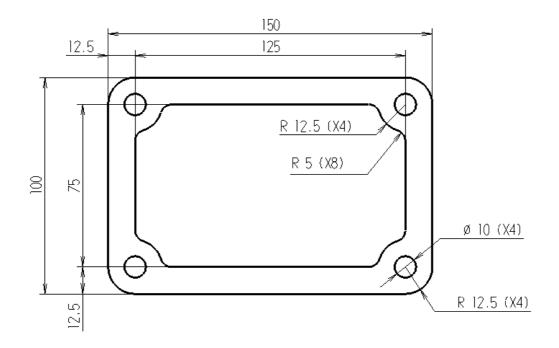
Name the file and save it



Click on *Main Menu* indicated by **A** and then select *Save as* indicated by **B** from the dropdown menu. Give the file the name **CNC-Calc Milling Tutorial 1** and save it (the file name extension is added automatically).

11. Tutorial 2 CNC toolpaths and Face milling (Mill)

With CNC-Calc v6 it is possible to create toolpaths directly from the program's geometrical drawings. Thereby, calculations become more secure and programming becomes much faster compared to doing it manually. At the same time you get a big advantage, since it is possible to move, copy, rotate, scale, and mirror elements with the result of instant NC-code generation.

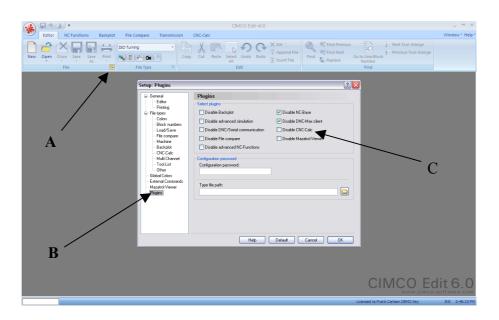


This tutorial demonstrates how the 2-dimensional part above can form the basis for NC-codes for various types of machining.

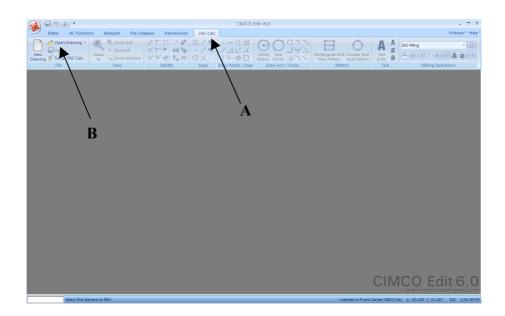
Note This tutorial builds upon the result from CNC-Calc v6 Tutorial 1.

11.1. Before you start

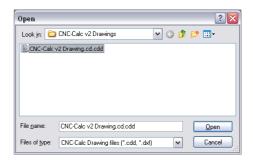
The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.



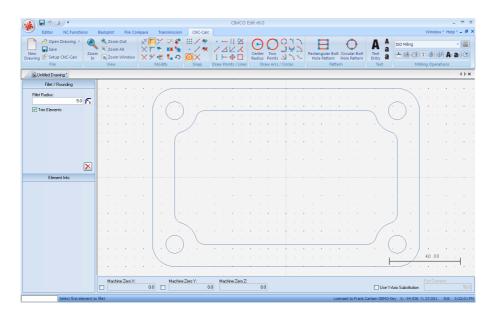
To open a drawing, click on **CNC-Calc** indicated by **A** and then on **Open Drawing** indicated by **B**.



Select the file **CNC-Calc Milling Tutorial 1** and click Open.



You should now see the part from CNC-Calc v6 Tutorial 1 displayed.



Note If you hold the cursor over an icon for a moment, a short description of the icon's functionality will appear.

11.2. Creation of Facing toolpaths

First, in the drop down menu indicated by arrow **A**, select the programming format of the NC-program.

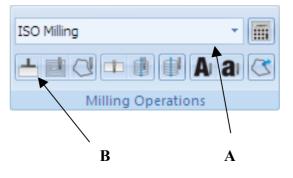
Select: ISO Milling.

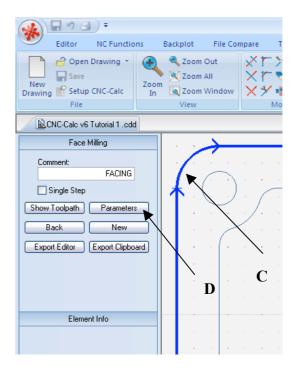
Then select **Generate a CNC-Toolpath for Face Milling** by clicking on the icon by arrow **B**.

Write the text FACING in the **Comment** field. This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.

Click on the outlining contour at the place indicated by arrow **C**. This will select the bounding contour that the facing operation will operate on.

Click on **Parameters** by arrow **D**.





Enter the values into the dialog as shown in the table below.

Depths Tab: Cutter Diameter:

This is the diameter of the cutter. Here it is a 30 mm Face Mill.

Start Depth: This is the top of the part.

End Depth: The final depth (will be corrected by *Stock to Leave*).

Retract Height: When the operation is finished, this is the height that the tool will retract to.

Roughing Stepdown: The maximum roughing cuts that the operation will take.

Finish Stepdown: If *Finish Cuts* is larger than zero, this is the cut that will be taken in each finishing cut.

Finish Cuts: The number of finishing cuts that the operation will perform. If the value is left at zero, only roughing cuts will be made.

Strategy Tab:

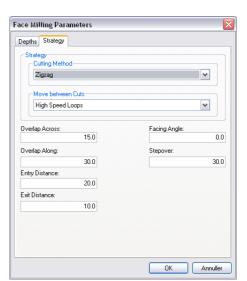
Cutting Method: The method used to perform the face operation. It is possible to select Zigzag, Climb, or Conventional.

Move Between Cuts: Is only used for the Zigzag Cutting Method, since the other methods will move free between cuts.

Overlap Across: The amount that the mill will hang out over the side diagonal to the cutting direction.

Overlap Along: The distance that the tool will move out over the end before the high speed loops are taken.

epths Strategy	
Cutter Diameter:	- Roughing
30.0	Roughing Stepdown:
	2.0
Depth Settings	
Start Depth:	 Finish
2.0	Finish Stepdown:
End Depth:	0.1
0.0	Finish Cuts:
Retract Height:	1
10.0	Stock to Leave:
·	0.0



Entry Distance: The distance that the tool will start out at before the actual cut is taken.

Exit Distance: The distance the tool moves out after the final cut is taken.

Facing Angle: The angle at which the operation is performed. An angle of zero is along the X-axis, and an angle of 90 is along the Y-axis.

Now close the parameters dialog with OK and click on **Export Editor.** The following screen should now be displayed.

ء (🖌 🤊 🖓 🖓	CIMCO Edit v6.0 - [Untitled *]	_ = >
Editor NC Functions	s Backplot File Compare Transmission CNC-Calc	Window * Help * = 🖉
Drawing Setup CNC-Calc File	Access Out Image: Constraint of the second sec	attern Entry A Milling Operations
CNC-Calc v6 Tutorial 1 .cd	3 Unitied *	4 b x
NCAsistant Description:	2 c00 X-110.000 Y-50.000 3 c00 Z10.000 4 c00 Z0.100 5 c01 X50.000 6 c03 Y-25.000 J0.000 J12.500 7 c01 X-50.000 8 c02 Y0.000 J10.000 J12.500 9 c01 X50.000 2 c03 Y25.000 J10.000 J12.500 11 c01 X-90.000 2 c02 Y50.000 J10.000 J12.500	
Cpcles / Masce Pagana Stat and End A Tool change Pagana comment 600 Facid move 601 Linear move 602 Dockwise arc move 603 Conterciclocivies and move 604 fund 607 Hippotentical axis interpolisi. 601 Data setting mode cancel 615 Polar accidante mode cancel	JJ GOI X90.000 JJ GOI X90.000 JS GOO X10.000 JS GOO X10.000 JJ GOO X10.000 JJ GOO X0.000 JJ GOO X2.000 JJ GOO X2.000 JJ GOO X2.000 JJ GOO X2.000 JS GOI X9.000 JS GOI X2.000	
G16 Polar coordinate mode G17XY plane specification G19XZ plane specification G19XZ plane specification G20 English units G22 Stored stroke check ON G23 Stored stroke check ONF G23 Stored stroke check ONF G23 Stored stroke check ONF G25 Scindle speed fluctuation d	ac GOI X90.000 27 GOI X100.000 28 GOO Z10.000 29	wk Carlsen DBH0 Xey In 123; Col 1: 400 bytes 186 10:1515 X

11.3. Inserting a Tool with Feed and Speed Calculator

The Feed and Speed calculator built into CNC-Calc is used to insert feed and speed data into the NC program. All the data used in the calculations can normally be found in the reference material supplied by the manufacturer.

In the facing example, we use a face mill that we give the following characteristics: diameter is 30 mm; it has 5 flutes, a cutting feed of 0.08 mm per tooth, and a cutting speed of 190 mm/min.

To use the feed and speed calculator, select Feed and Speed Calculator for Milling Operations by clicking on the icon indicated by arrow A. Fill in the following values: Tool #: Lets say that the face mill have a tool number of 1. Diameter is 30 mm. # Flutes: The number of flutes is 5. Feed per tooth: In this example it is set to 0.08 mm. Cutting Speed: Is set to 190. ISO Miling Feed ar Tool #: Diameter Event are the face mill have a tool Diameter is 5. Feed per tooth: In this example it is set to 0.08 mm. Cutting Speed: Is set to 190.

The fields are linked together, so as entries are made in the cutting speed field, the other fields will be automatically updated.

If we then want to have RPM 2000 and a feedrate of 800 instead of the calculated 2015 and 836.385, the value for the cutting speed will be updated to 188.5.

Change the RPM to 2000 and the feedrate to 800. Click on **Export Clipboard** indicated by arrow **B**.

ISO Milling 👻
Milling Operations
A /
Feed and Speed Calculator
Tool #:
1
Diameter (D) in mm:
30
Flutes (Z):
5
Feed per tooth (Sz) in mm:
0.08
Cutting Speed (V) in mm/min:
190
BPM (n):
2015
Feedrate (F) in mm/min:
806.385
Metric OInches
B Clear
Export Editor Export Clipboard

The line for the NC program is now in the clipboard, and it is ready for insertion. Change the window to that of the NC program, and press Ctrl-Home to move to the very start.

Insert the text from the clipboard, either by pressing Ctrl-v, or selecting

Editor in the menu and the *Paste* icon from the toolbar.

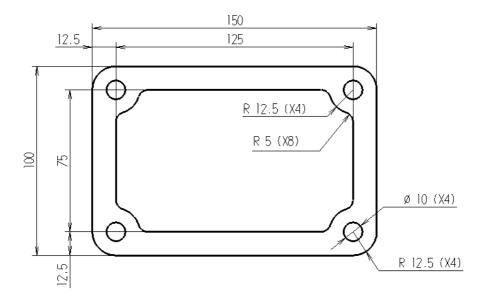
The NC program should now look similar to the one below.

📣 🖬 🤊 🖨 🔻	CIMCO Edit v6.0 - [Untitled *] _ 🖉 🛪
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window * Help * - 🕫 ×
Open Drawing 🔹 🙆	
New Save Z	
	a la Zoom Window X Y 🖷 🖬 🕈 ⊙ X I I ⊢ ↔ 🗆 Radius Points 🎝 ∖ ∨ Hole Pattern Hole Pattern Entry a
File	View Modify Snap Draw Points / Lines Draw Arcs / Circles Pattern Text Milling Operations
CNC-Calc v6 Tutorial 1 .cd	() United *
NC-Assistant	1 TI M06 S2000 F800.000
	2 (FACING)
Description:	3 G00 X-110.000 Y-50.000
	4 GOO Z10.000
	5 GOO 20.100
	G G01 X90,000
	7 G03 Y-25.000 I0.000 J12.500
	s G01 X-90.000
	9 G02 Y0.000 I0.000 J12.500
	20 G01 X90.000
	11 G03 Y25.000 I0.000 J12.500
	12 G01 X-90.000
Modify	13 G02 Y50.000 I0.000 J12.500
Cycles / Macros	24 G01 X90.000
	25 G01 X100,000
Program Start and End	26 G00 Z10.000
Program comment	17 G00 X-110,000 Y-50,000
G00 Rapid move	1 500 Z 0.000
G01 Linear move	29 GOI 20.000
G02 Clockwise arc move	
G03 Counterclockwise arc move G04 Dwell	20 G03 Y-25.000 I0.000 J12.500
G07 Hypothetical axis interpolati	22 G01 X-90.000
G09 Exact stop check for one b	22 G02 Y0.000 I0.000 J12.500
G10 Data setting mode (Standa	23 G01 X90.000
G11 Data setting mode cancel	24 G03 Y25.000 I0.000 J12.500
G15 Polar coordinate mode can	25 G01 X-90.000
G16 Polar coordinate mode G17 XY plane specification	26 G02 Y50.000 I0.000 J12.500
G18 XZ plane specification	27 G01 X90.000
G19 YZ plane specification	28 G01 X100.000
G20 English units	29 GOO 210.000
G21 Metric units	30
G22 Stored stroke check ON G23 Stored stroke check OFF	
G25 Spindle speed fluctuation d	
Find Add	· · · · · · · · · · · · · · · · · · ·
Calculate Feed an	d Speed Licensed to Frank Carlsen DEMO Key Ln 1/30, Col 22, 502 bytes INS 10:36:47 AM

Now save the NC program as CNC-Calc v6 Tutorial 2.NC

12. Tutorial 3 Contour milling (Mill)

CNC-Calc v6 can generate contour milling - with and without radius compensation. There are several machine types in CNC-Calc, but the most commonly used are ISO G-code programming and Heidenhain plain text.



This tutorial demonstrates how the above 2-dimensional part can form the basis of NC-codes for various types of machining.

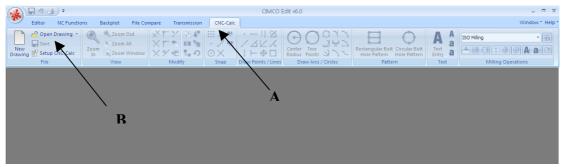
Note This tutorial builds upon the result from CNC-Calc v6 Tutorial 1.

12.1. Before you start

The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

•	CIMCO Edit v6.0	_ = ×
Editor NC Functions Backplot File C	iompare Transmission CNC-Calc	Window * Help *
• • • As •	ng X Ind Previous Ned Tool change Copy Cut Paste Select Undo Redo Hile All Lot Find Redo Redo Redo File All Select Select Undo Redo File All Select Select Undo Redo File Find Redo File Find Redo File Find Redo File Find Redo File	
_	Setup: Plugins	
A	E Bior Printing Printing Printing Printing Printing Printing Printing Printing Printing Disable Backplot Disable BAC Base Disable BAC BASE Disa	С
B	Hep Default Carcel OK	

To open a drawing, click on **CNC-Calc** indicated by **A** and then on **Open Drawing** indicated by **B**.



Select the file **CNC-Calc Tutorial 1** and click Open.

Open		? 🗙
Look jn: 隘) CNC-Calc v2 Drawings 🛛 🔽 🌀 💋) 📂 🛄 -
CNC-Calc	v2 Drawing.cd.cdd	
File <u>n</u> ame:	CNC-Calc v2 Drawing.cd.cdd	<u>Open</u>
Files of type:	CNC-Calc Drawing files (*.cdd, *.dxf)	Cancel

Note If you hold the cursor over an icon for a moment, a short description of the icon's functionality will appear.

12.2. Creation of Contour toolpaths

To begin the creation of an NC program for the contour operation, select the **Generate CNC-Toolpath for Contour Milling** by clicking on the icon indicated by arrow **A**.

Write CONTOUR in the **Comment** field.

This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.

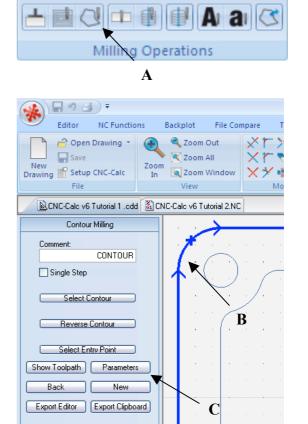
Click on the outlining contour at the place indicated by arrow **B**.

This highlights the outline contour; the contour arrows indicate the direction the tool will travel.

What side the tool will machine is controlled by the *Work Side* drop down box on the *General* tab in the parameters dialog.

Click on Parameters by arrow C.

Enter the values in the Parameter dialog as shown below.



ISO Milling

General Tab contains all the general parameters that are used for roughing and finishing in both depth and side cuts.

Cutter Diameter: The diameter of the tool in use.

Retract Height: The height to which the tool will move between contours, and where it will stop at the end of the operation.

eneral	Side Cuts	Depth Cu	ts Lead Ir	n/Out	
Cutte	r Diameter:				
10.0					
10.0					
	Settings				
	act Height:			Start Depth:	
10				0.0	
Safe	Distance:			End Depth:	
2.0				-5.0	
Stock	to Leave-				
Stoc	k to Leave >	<u>۲:</u>		Stock to Leave Z:	
0.0				0.0	
	ensation				
	pply on Rou	ghing Side	outs		
Comp	ensation T	ре			
Com	iputer				
Work	Side				
Left					

Safe Distance: The distance above the part, where the feedrate will change from rapid to cutting speed.

Start Depth: This is the top of the stock.

End Depth The depth at which the last cut will be taken. This value is corrected by the Stock to leave Z value.

Stock to Leave XY: The amount of stock that is left in the XY/side direction at the end of the operation (after both Roughing and Finishing).

Stock to leave Z: The amount of stock that is left in the Z/depth direction at the end of the operation (after both Roughing and Finishing).

Apply on Roughing Sidecuts: If this check box is checked, the compensation type will be applied to both roughing and finishing side cuts. Otherwise computer compensation is used for roughing cuts, and the selected compensation type for finishing cuts.

Compensation Type: This is the compensation type used for the operation.

Work Side: This field determines on which side of the contour the tool will pass. Together with the selected direction of the contour it determines if the milling type will be climb or conventional. *Side cuts Tab* Configures the cuts taken in the XY direction.

Use Side Cuts: If this check box is checked the operation will perform the cuts defined by the parameters. Otherwise, only one cut at the final contour will be performed.

Number of Passes (Roughing): The number of roughing side cuts in the operation.

Spacing (Roughing): If more than one roughing pass is taken, this is the distance between them.

Number of Passes (Finish): The number of finishing side cuts in the operation.

Spacing (Finish): The distance of each finishing pass.

Final Depth: If this check box is checked, the finishing passes will only be taken at the final depth.

All Depths: If this check box is checked, the finishing passes will be taken at every depth.

Overlap Distance: The distance that all the finishing laps will overlap, in order to smooth the surface.

eneral Side Cuts Depth Cuts Le	ead In/Uut
Use Side Cuts	
Roughing Passes	Finish Passes
Number of Passes:	Number of Passes:
2	1
Spacing:	Spacing:
2.0	
Machine finish passes at	~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~
 Final Depth 	All Depths
~ · · · · · · · · · · · · · · · · · · ·	
Finish Overlap	
Overlap Distance:	
	10.0

Depth cuts Tabs Configures the cuts taken in the Z direction.

Use Depth Cuts: If this check box is checked, the operation will perform the cuts defined by the parameters. Otherwise, only one cut at the final depth will be performed.

Max Roughing Steps: The maximum cut that will be taken in a roughing cut.

ontour Milling Parameters	×
General Side Cuts Depth Cuts Lead	d In/Out
Use Depth Cuts	
Roughing Passes Max Roughing Steps: 2.0	Finish Passes Number of Cuts:
Use Even Depth Cuts	Steps: 0.1
Helix Linearization	0.05
Toolpath Linking	O By Contour
	OK Annuller

Use Even Depth Cuts: If this check box is

checked, all the roughing passes will have the same distance. If it is left unchecked, cuts will be taken at the Max Roughing Steps distance, and any rest material will be taken with the last cut.

Number of Cuts (Finish): The number of finishing depth cuts in the operation.

Steps (Finish): The distance of each finishing pass.

Linearize Helix Movements: Some machines cannot make helix movements, and if this check box is checked, all helix movements will be converted to lines in the NC operation.

Linearization Tolerance: When the helix is converted to lines, this will be the maximum error for the final lines.

By Depth: This is only used if multiple contours are milled in the same operation. If selected, the cut on each depth will be performed on all contours, before any cuts are made at a new depth.

By Contour: If selected, one contour will be milled from start to finish, before the next contour is worked upon.

Lead in/out tab Configures the way the tool will approach the contour at the start/end of the roughing, and for each finishing pass.

The use of lead in/out is optional, when the compensation is set to computer or none. It is however mandatory, when any compensation is performed by the controller.

Use Lead In/Out Parameters: Enables or disables the lead in and out.

neral Side Cuts Depth Cuts Lead	In/Out
Use Lead In/Out Parameters	
.ead In	- Lead Out
✓ Use Line Line Length:	Use Line Line Length:
5.0	5.0
Perpendicular Tangent	Perpendicular Tangent
Use Arc	Use Arc
Radius:	Radius:
5.0	5.0
Sweep:	Sweep:
90.0	90.0
Feedrates	rates

Use Line: Enables or disables the lead in/out lines.

Line Length: The length of the lead in/out line.

Perpendicular: If this is selected, the line will be perpendicular to the following element for lead in, and the previous element for lead out.

Tangent: If this is selected, the line will be tangent to the following element for lead in, and the previous element for lead out.

Use Arc: Enables or disables the lead in/out arcs.

Radius: The radius of the lead in/out arc.

Sweep: The sweep angle of the lead in/out arc.

The two arrows in the middle of the dialog are used to copy all values from lead in to lead out, and vice versa.

Now, close the parameters dialog with OK and click on **Export Clipboard**. The NC operation is now in the clipboard, and it is ready for insertion.

Change the window to that of the NC program and press Ctrl-End to move to the very end. Insert the text from the clipboard, either by pressing Ctrl-v, or selecting *Editor* in

the menu and then selecting the *Paste* icon in the toolbar.

The NC program in the Editor now consists of two operations, and currently they are both made with the same tool. Now we need to insert a new tool for the contour operation.

12.3. Inserting a Tool with Feed and Speed Calculator

Follow the steps from the previous tutorial for the Feed and Speed on page 41. Instead of the values used there, use the following values:

```
Tool #: 2
Diameter (D) in mm: 10
# Flutes (Z): 4
Feed per tooth (Sz) in mm: 0.06
Cutting Speed (V) in mm/min: 175
```

Now, the last two values have been calculated and inserted in the dialog. They should be:

RPM: 5570 **Feedrate (F) in mm/min:** 1336.9015

Correct the **RPM** to 5500 and then **Feedrate (F) in mm/min:** to 1320.

Now the Feed and Speed dialog should look like the one to the right:

Click on **Export Clipboard** to copy the generated line to the clipboard.

10 # Flutes [Z]: 4 Feed per tooth (Sz) in mm: 0.06 Cutting Speed (V) in mm/min: 172.7876 RPM (n): 5500 Feedrate (S) in mm/min: 1320.0 Metric Inches Clear Export Editor Export Clipboard

Feed and Speed Calculator

2

Tool #

Diameter (D) in mm

Change to the NC program in the editor. After the contour

operation was copied to the editor, the cursor is at the very end of the program. In order to insert the tool line from the clipboard, we must locate the start of the contour operation. Since the comment *CONTOUR* was inserted, it is easy to locate the start of the operation.

Find the text *CONTOUR*, either by pressing Ctrl-f, or selecting *Editor* in the menu and then the *Find* icon in the *Find* toolbar.

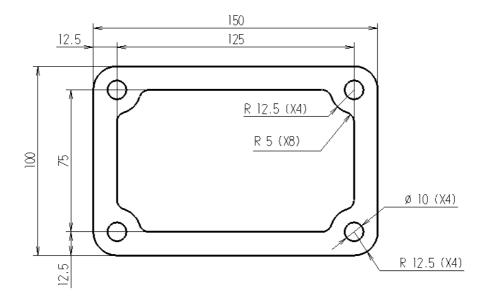
Go to the start of the comment line and insert the text from the clipboard. The NC program should now look like the following.

🔬 🔒 🤊 🖨 🔻	CIMCO Edit v6.0 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc v6 Tutorial 2.NC *]	- = x
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window* H	elp = @ ×
New Open Close Save Save File		
CNC-Calc v6 Tutorial 1 .cdr		4 Þ ×
	24 G03 Y25.000 I0.000 J12.500	N P A
NC-Assistant Description Cycles / Mactos Program Start and End Cycles / Mactos Program Start and End Tool change Program comment 601 Bajding are move 601 Linear move 602 Linear move 603 Counterclockwise are move 604 Counterclockwise are move 603 Counterclockwise are move 603 Counterclockwise are move 604 Counterclockwise are move 604 Counterclockwise are move 605 Counterclockwise are move 605 Counterclockwise are move 606 Counterclockwise are move 607 Counterclockwise are move 607 Counter move 607 Counterclockwise are move 608 Counterclockwise are move 608 Counterclockwise are move 609 Counterc	25 G01 x-90.000 26 G02 x50.000 I0.000 J12.500 27 G01 x00.000 28 G01 x100.000 29 G00 210.000 30 T2 H06 85500 F1320.000 31 (CONTOUR) 32 G00 x-79.895 y54.895 33 G00 21.0 000 34 G00 22.000 35 G01 z-1.667 36 G01 x-83.430 y51.359 37 G03 x-76.359 I3.536 J3.536 38 G02 x-62.500 y57.100 I13.859 J-13.859 39 G01 x62.500 y57.100 40 G02 x82.100 y37.500 I0.000 J-19.600 41 G01 x-37.500 42 G02 x-82.500 y-57.100 I-19.600 J0.000 43 G01 x-62.500 y57.100 44 G02 x-82.100 y-37.500 I0.000 J19.600 45 G01 x-7.6359 y51.359 y19.600 J0.000 47 G02 x-74.517 y51.697 J1.165 J-1.165 47 G02 x-72.654 y51.876 I0.775 J1.721 48 G02 x-62.500 y55.100 40 G02 x-62.500 y55.100 40 G02 x-74.517 y51.607 J1.721 40 G02 x-74.517 y51.607 J1.721 40 G02 x-62.500 y55.100 41 G01 x-2.650 y55.100 42 G02 x-62.500 y55.100 43 G01 x-62.500 y55.100 44 G02 x-74.517 y51.607 J1.721 45 G01 x-72.654 y51.876 I0.775 J1.721 45 G01 x-72.654 y51.876 I0.775 J1.721 45 G02 x-62.500 y55.100 45 G02 x-62	z
G20 English units G21 Metric units G22 Stored stroke check ON G23 Stored stroke check OFF G25 Spindle speed fluctuation dM Find Add	32 CO1 X80.100 Y-37.500 33 CO2 X62.500 Y-55.100 I-17.600 J0.000 54 GO1 X-62.500 Y-55.100 55 GO2 X-80.100 Y-37.500 I0.000 J17.600 56 CO1 X7.500	
Calculate Feed and	000 07 010 174 017 017 010 000 000 000 000 000 000 000	► 11:15:41 AM

Save the NC program as CNC-Calc v6 Tutorial 3.NC

13. Tutorial 4 Pocket milling (Mill)

CNC-Calc v6 can generate pocket milling. There are several machine types in CNC-Calc, but the most commonly used are ISO G-code programming and Heidenhain plain text.



This tutorial demonstrates how the above 2-dimensional part can form the basis of NC-codes for various types of machining.

Note This tutorial builds upon the result from CNC-Calc v6 Tutorial 1.

13.1. Before you start

The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

🔬 🖬 🔊 🚍 🔻	CIMCO Edit v6.0	_ = x
Editor NC Functions Backplot File C	ompare Transmission CNC-Calc	Window * Help *
• • • As • OO	Append File V Find Next Previous Tool change	
A	Setup: Plugins Image: Setup Plugins Extended of the setup of the set	С
В	Help Default Cancel OK	

To open a drawing, click on **CNC-Calc** indicated by **A** and then on **Open Drawing** indicated by **B**.

CIMCO Edit v6.0	- ¤ X
Editor NC Functions Backplot File Compare Transmission CNC-Calc	Window* Help*
New B Save Save Zoom Out X T Y M Image: Comparing the same same same same same same same sam	
B	
Select the file CNC-Calc Tutorial 1 and click	Open ?X
	Look jn: 🗀 CNC-Calc v2 Drawings 🛛 🕥 🎲 📂 🖽 •
Open.	File pame: CNC-Calc v2 Drawing cd.cdd
	Files of type: CNC-Calc Drawing files (*.cdd, *.dxf) Cancel

Note If you hold the cursor over an icon for a moment, a short description of the icon's functionality will appear.

13.2. Creation of Pocket toolpaths

To start creating an NC program for the pocket operation, select the function **Generate a CNC-Toolpath for Pocket Milling** by clicking on the icon indicated by arrow **A**.

Write the text POCKET in the **Comment** field.

This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.

Click on the inner contour where indicated by arrow **B**. This will highlight the inner contour.

Click on Parameters by arrow C.

Enter the values in the Parameter dialog as shown.

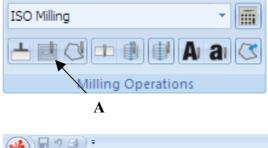
General Tab Configures the parameters used for roughing and finishing in both depth and side cuts.

Cutting Diameter: The diameter of the used tool.

Retract Height: The height to which the tool will move between contours, and where it will stop at the end of the operation.

Safe Distance: The distance above the part

where the feedrate will change from rapid to cutting speed.



	•			
Editor	NC Functions	Backplot	File Compare	т
New Drawing Setup File	Drawing ~ CNC-Calc		AII	
CNC-Calc v6	Tutorial 1 .cdd 📓	CNC-Calc v6 Tu	utorial 3.NC	
Pocket Comment: Single Step Show Toolpath Back Export Editor	Milling POCKET Parameters New Export Clipboard			
Eleme	nt Info			
		· · ·	(1) (1) (2) (2) (2) (2) (2) (2) (2) (2) (2) (2	

eneral Side Cuts Depth Cuts Entry	y Strategy Lead In/Out
Tool	
Cutter Diameter:	
10.0	
Depth Settings	
Retract Height:	Start Depth:
10.0	0.0
Safe Distance:	End Depth:
2.0	-4.0
Stock to Leave Stock to Leave XY:	Stock to Leave Z:
Stock to Leave XY: 0.0	Stock to Leave Z:
0.0	0.0
Compensation	
Compensation Type	
Computer	×
Milling Type	
Conventional	 Climb
· · · · · · · · · · · · · · · · · · ·	·

Start Depth: This is the top of the stock.

End Depth The depth at which the last cut will be taken. This value is corrected by the Stock to leave Z value.

Stock to Leave XY: The amount of stock that is left in the XY/side direction at the end of the operation (after both Roughing and Finishing).

Stock to leave Z: The amount of stock that is left in the Z/depth direction at the end of the operation (after both Roughing and Finishing).

Compensation Type: This is the compensation type used for the operation.

Conventional: When checked, the operation will be generated using conventional milling.

Climb: When checked, the operation will be generated using climb milling.

Side Cuts Tab Configures the cuts taken in the XY direction.

Max Roughing Spacing: The maximum side stepover used in the roughing of the part.

Number of Passes (Finish): The number of finishing side cuts in the operation.

Spacing (Finish): The distance of each finishing pass.

eneral Side Cuts Depth Cuts Entry	Strategy Lead In/Out
Roughing Passe: Max Roughing Spacing: 3.0	Finish Passes: Cuts: 1
	Spacing: 0.1
- Finish Passes at	
 At Final Depth 	🔿 At all Depths
Finish Overlap Overlap Distance:	1.0
Roughing Smoothing Disabled	Max
(

At Final Depth: If this check box is checked, the finishing passes will only be taken at the final depth.

At All Depths: If this check box is checked, the finishing passes will be taken at every depth.

Overlap Distance: The distance that all the finishing laps will overlap in order to smooth the surface.

Roughing Smoothing: This slider controls the amount of smoothing used. The higher the value (rightmost), the smoother the resulting toolpath will be.

Depth Cuts Tab Configures the cuts taken in the Z direction.

Use Depth Cuts: If this check box is checked, the operation will perform the cuts defined by the parameters. Otherwise, only one cut at the final depth will be performed.

Max Roughing Steps: The maximum cut that will be taken in a roughing cut.

Use Even Depth Cuts: If this check box is checked, all the roughing passes will have the same distance. If it is left unchecked, cuts will be

Roughing Passe: Max Roughing Steps:	Finish Passes: Number of Cuts:
2.0	1
Use Even Depth Cuts	Steps: 0.1
Linearization Tolerance	0.01
Toolpath Linking	
~ - - ·	By Pocket
🔵 By Depth	

Pocket Milling Parameters

taken at the Max Roughing Steps distance, and any rest material will be taken with the last cut.

Number of Cuts (Finish): The number of finishing depth cuts in the operation.

Steps (Finish): The distance of each finishing pass.

Linearize Helix Movements: Some machines cannot make helix movements, and if this check box is checked, all helix movements will be converted to lines in the NC operation.

Linearization Tolerance: When the helix is converted to lines, this will be the maximum error for the final lines.

By Depth: This is only used if multiple pockets are milled in the same operation. If selected, the cut at each depth will be performed on all pockets before any cuts are made at a new depth.

By Pocket: If selected, one pocket will be milled from start to finish before the next pocket is worked upon.

Entry strategies Tab Configures how the tool cuts from one Z level to the next.

Plunge: When this is selected, the tool will move straight down.

Ramp: With the ramp entry, the tool moves down to the **Ramp Clearance** above the part. Then it makes a ramp movement with the length **Ramp Length** and the angle **Ramp Angle**.

Helix Entry: Moves down to Helix Clearance above the part. Then it will spiral down with the angle Helix Angle in a circular movement with a diameter between Helix Diameter and Minimum Helix Diameter. How big the actual diameter will be depends on the geometry.

Lead in/out Tab Configures the way that the tool will approach the contour at the start/end of the roughing, and for each finishing pass.

The use of lead in/out is optional when the compensation is set to computer or none. It is however mandatory when any compensation is performed by the controller.

Use Lead In/Out Parameters: Enables or disables the lead in and lead out. Use Line: Enables or disables the lead in/out lines.

Line Length: The length of the lead in/out line.

eneral	Side Cuts Depth Cuts	Entry Strateg	y Lead In/Out
Entry	Strategy		
~	lunge		
OH	lamp Ramp Parameters		
	Ramp Angle:	Ran	np Clearance:
		5.0	3.0
	Ramp Length:		
		20.0	
L			
٥H	lelix Entry		
ſ	Helix Parameters Helix Angle:	Heli	x Clearance:
		5.0	2.0
	Helix Diameter:	Mini	mum Helix Diameter:
		3.0	0.2

neral Side Cuts Depth Cuts	Entry Strat	egy Lead In/Out
Use Lead In/Out Parameters		
Lead In		Lead Out
Use Line		Vse Line
Line Length:		Line Length:
5.0		5.0
	 ▶ 	<u></u>
 Perpendicular 		 Perpendicular
○ Tangent	<-	 Tangent
Use Arc		Use Arc
Radius:		Radius:
5.0		5.0
Sweep:		Sweep:
90.0		90.0
00.0		00.0
Feedrates		
Use custom feedrates		
	Feedrates	
	T COULTUCS	

Perpendicular: If this is selected, the line will be perpendicular to the following element for lead in, and the previous element for lead out.
Tangent: If this is selected, the line will be tangent to the following element for lead in, and the previous element for lead out.
Use Arc: Enables or disables the lead in/out arcs.
Radius: The radius of the lead in/out arc.
Sweep: The sweep angle of the lead in/out arc.

The two arrows in the middle of the dialog are used to copy all values from lead in to out, and vice versa.

Now close the parameters dialog with OK and click on **Export Clipboard**. The NC operation is now in the clipboard, and is ready for insertion.

Change the window to that of the NC program and press Ctrl-End to move to the very end. Insert the text from the clipboard, either by pressing Ctrl-v, or selecting *Editor* in

the menu and then selecting the *Paste* icon in the toolbar.

The NC program in the Editor now consists of three operations, and since we use the same tool for the contour and pocket operations we will not insert a tool before the pocket operation.

Now save the NC program as CNC-Calc v6 Tutorial 4.NC

14. Tutorial 5 Backplot in the Editor (Mill)

One of the advantages of running CNC-Calc inside CIMCO Edit v6 is that the editor can be used to manipulate and backplot the NC programs generated in CNC-Calc. In the following, we will setup the backplot and verify the program.

14.1. Before you start

The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable Backplot* indicated by **C** is unchecked.

🔬 🗐 🗐 🗐 =	CIMCO Edit v6.0	_ = ×
Editor NC Functions Backplot File C	iompare Transmission CNC-Calc	Window * Help *
New Open Close Save Print File	Append File Find Next Previous Tool change	
7	Setup: Plugins	
A	General Plugins File type Disable Backplot Disable BCBackplot File type Disable Backplot Disable Backplot Bickok numbers Disable Bolt/Pointal communication Disable Macadial Viewer Bickok numbers Disable Bolt/Pointal communication Disable Macadial Viewer Bickok numbers Disable Sockplot Disable Macadial Viewer Disable Sockplot Disable Macadial Viewer Disable Macadial Viewer Bickol Colors Configuration password Configuration password Configuration password Global Colors Type file path: Disable Macadial Viewer Disable Macadial Viewer	
В	Help Default Cancel OK	
	CIMCO Ec	li <u>t</u> 6.0
	W W W . CLIMIC O- Licensed to Frank Carlsen DEMO Key	software.com INS 2:48:23 PM

To open an **NC Program**, click the open program icon indicated by arrow **A**.

This will open the Open File dialog, where the file you want to open can be selected.

Please select the file from the last tutorial and click Open.

New Open Save Save Print ISO Miling File Save Save Print File File Sagi Samples File File File Sagi Samples ISO Milling ISO Milling ISO Milling Mazarol ISO CNC-Calc v6 Tutorial 3.NC ISO Milling ISO CNC-Calc v6 Tutorial 4.NC MulPlus ISO CNC-Calc v6 Tutorial 4.NC ISO Milling Files (".NC.".NCL.".ISO.".NCF) Annuller Filown: CNC-Calc v6 Tutorial 4.NC Åbn Filown: CNC-Calc v6 Tutorial 4.NC Åbn	Edit	or NC Fun	ctions Backplot	File Compare	Transmission
As File As File File File Segit Samples Signers File Mazatrol File Mult Channel File Signers Stotorial 4.NC File File File File		XE		ISO Milling	-
File File Type G Åbn Image: Im	New Oper	n Close Sav		📲 🖻 📽 🖬	Co
Sag t Samples Samples Source-Calc v6 Tutorial 3.NC Mazatrol SCNC-Calc v6 Tutorial 4.NC Miliplus Multi Channel Siemens 840D Siemens 840D Filnavn: CNC-Calc v6 Tutorial 4.NC K		File		File Type	G
Seg i Samples Si Samples Si Si CNC-Calc v6 Tutorial 3.NC Mazatrol Si CNC-Calc v6 Tutorial 4.NC MilPlus MilPlus Siemens 8400 Sicro-Calc v6 Tutorial 2.NC Simens CNC-Calc v6 Tutorial 4.NC Siemens 8400 CNC-Calc v6 Tutorial 4.NC Simens Size Size Size Size Size Size Size Size					
ISO MILL Interface ISO MILL Interface Mazatrol Interface Multi Channel Interface Siemens 840D CNC-Calc v6 Tutorial 2.NC Image: State Interface Filnavn: CNC-Calc v6 Tutorial 4.NC	Åbn				2 🔀
Mazatrol Millius Multi Channel Siemens 840D CNC-Calc v6 Tutorial 2.NC Filnavn: CNC-Calc v6 Tutorial 4.NC Abn	Søgi: 📔	🔁 Samples		💌 🕝 🧊 📂	-
MilPlus MilDus Milc Channel Stemens 9400 CNC-Calc v6 Tutorial 2.NC Filnavn: CNC-Calc v6 Tutorial 4.NC Åbn	🛅 ISO MILL		👔 CNC-Calc v6 T	utorial 3.NC	
Multi Channel Siemens 840D CNC-Calc v6 Tutorial 2.NC CNC-Calc v6 Tutorial 4.NC Filnavn: CNC-Calc v6 Tutorial 4.NC	Mazatrol		💹 CNC Colo US T	utorial 4 MC	
Siemens 840D CINC-Calc v6 Tutorial 2.NC Finavn: CINC-Calc v6 Tutorial 4.NC Abn			01 CNC*Calc VO I	dional Hind	
Signal CNC-Calc v6 Tutorial 2.NC Image: Signal CNC-Calc v6 Tutorial 4.NC	🚞 MillPlus			donal 4.NC	
Filnavn: CNC-Calc v6 Tutorial 4.NC Abn	MillPlus	annel		utonal Hind	
Filnavn: CNC Cale v6 Tutorial 4.NC	MillPlus	annel 840D			
	MillPlus Multi Cha Siemens	annel 840D			
Filtype: ISO Milling Files (".NC/.*.NCL/*.ISO,*.NCF) 🗸 Annuller	MillPlus Multi Cha Siemens	annel 840D			>
Pikype. TSU Milling Files (LINC; LINC; LISU; LINCF)	MillPlus Multi Cha Siemens CNC-Cale	annel 840D c v6 Tutorial 2.N	c		
	MillPlus Multi Cha Siemens CNC-Cale Filnavn:	annel 840D c v6 Tutorial 2.N CNC-Calc v6	C Tutorial 4.NC		Åbn
Abn som skrivebeskyttet	MillPlus Multi Cha Siemens CNC-Cale Filnavn:	annel 840D c v6 Tutorial 2.N CNC-Calc v6	C Tutorial 4.NC		Åbn

You should now see the NC program from the last tutorial.

🔬 🗐 " 🗃 🔹	CIMCO Edit v6.0 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc v6 Tutorial 4.NC] 🗕 🖻
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window - Help - 🗗
New Open Close Save Save File	
CNC-Calc v6 Tutorial 4.NC	(b)
NC-Assistant	1 F1 M06 S2000 F800.000
Description:	2 (FACING) 3 G00 X-110.000 Y-50.000 4 G00 Z10.000 5 G00 Z0.100 6 G01 X90.000 5 G01 X90.000 I0.000 J12.500 2 G01 X90.000 10 G01 X90.000 12 G01 X90.000 13 G02 Y0.000 J12.500 24 G01 X90.000 15 G01 X100.000 16 G00 Z10.000 J12.500 25 G01 X90.000 17 G00 X-110.000 Y-50.000 18 G00 Z0.000 19 G01 X90.000 26 G03 Y-50.000 I0.000 J12.500 22 G01 X-90.000 23 G01 X90.000 24 G03 Y25.000 I0.000 J12.500 25 G01 X90.000 26 G03 Y-50.000 27 G01 X90.000 27 G01 X90.000 28 G01 X90.000 29 G01 X90.000 20 G03 Y-50.000 I0.000 J12.500 22 G01 X-90.000 24 G03 Y25.000 I0.000 J12.500 25 G01 X-90.000 26 G01 Y-50.000 27 G01 X90.000 26 G01 Y-50.000 27 G01 X90.000 26 G01 Y-50.000 26 G01 Y-50.000 26 G01 Y-50.000 26 G01 Y-50.000 26 G01 Y-50.000 26 G01 Y-50.000 27 G01 X90.000 26 G01 Y-50.000 26 G01 Y-50.000 27 G01 X90.000 27 G01 X90.000 28 G01 X100.000 29 G01 X100.000 20 G01 Y-10.000 20
G21 Metric units G22 Stored stroke check ON G23 Stored stroke check OFF	29 G00 210.000 30 T2 M06 55500 F1320.000 32 (CONTOUR)
G25 Spindle speed fluctuation d M Find Add	32 G00 X-79.895 Y54.895 33 G00 210.000
	Licensed to Frank Carlson DEMO Key Ln 1/851, Col 1, 23,452 bytes INS 11:47:06 /

14.2. The first backplot

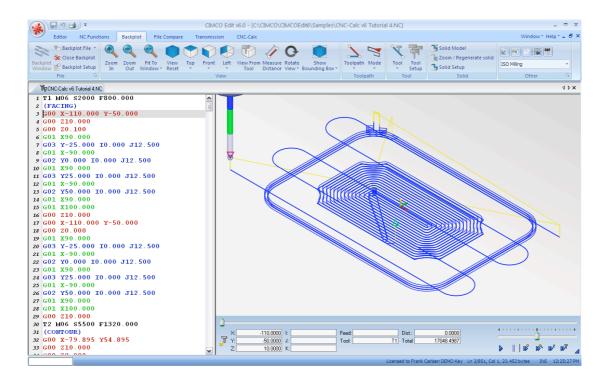
When the NC program has been loaded into the editor, it is possible to backplot it.

Click on the Backplot tab indicated by

arrow **A**. Then click on the $\widehat{}$ icon indicated by **B** to backplot the NC program.

	- 17 🛃	;					C
	Editor	NC Functi	ions	Backplo	ot Fi	le Compare	Transi
Backplot Window		Backplot Backplot Blot Setu	Zoom In	Zoom Out	Fit To Window	View Reset	Top Fr
	File	5					
	NC-Calc v6	Tutorial 4.N0					
	NC-Assi:	stant				s2000	F800.
Descript	ion:				ACING 0 X-1) 10.000	0 Y-50

Now the screen should look like the one below.



Here, the toolpath is shown. It looks OK, but it can be configured to look a lot more like the final part. To do this, we need to define the individual tools and the stock.

14.3. Backplot tool setup.

The following steps will show you how to set up the tools used in the backplot.

Click on the *Tool Setup* icon in the Backplot *Tool* toolbar indicated by arrow **A**. This will open the window Tool Setup.

Tool Tool A Setup Tool

When backplot parsed the NC program, it detected that two tools were being used.

Since these tools have not been defined, the screen will look like the one to the right.

In order to backplot the program correctly, we need to define these two tools.

 Tool Setup
 Type:
 Diameter:

 End Mill Ball 4MM
 End Mill Ball
 4

 T2 - (Dorbauh)
 End Mill Ball
 4

 End Mill Ball 4MM
 End Mill Ball
 4

 T2 - (Dorbauh)
 End Mill Ball
 4

 End Mill Ball 4MM
 End Mill Flat:
 5

 End Mill Flat:
 5
 20.0

 End Mill Ball 5MM
 End Mill Flat:
 5

 End Mill Ball 5MM
 End Mill Ball 5
 50

 End Mill Ball 5MM Flat:
 End Mill Ball 5
 10

 End Mill Ball 5MM Flat:
 End Mill Ball 5
 10

 End Mill Ball 5MM Flat:
 End Mill Ball 5
 10

 End Mill Ball 5MM Flat:
 End Mill Ball 5
 10

 End Mill Ball 5MM Flat:
 End Mill Ball 5
 10

 End Mill Ball 5MM Flat:
 End Mill Ball 5
 10

By changing the type and the diameter of the tools, we can create the correct setup.

Please select and enter the values for both tools shown on the picture to the right.

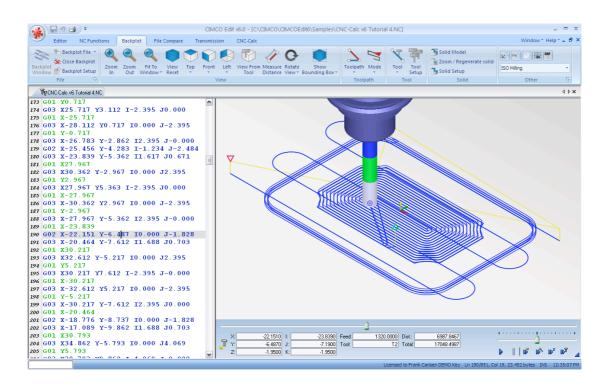
These are the same tool values that were used when the NC programs was generated in CNC-Calc.

Now exit the configuration dialog with OK.





After the tools have been configured, the screen should look like the one below. Try to find the tool changes and verify that the tool does indeed change when a tool change is encountered in the NC program.



14.4. Backplot Stock setup.

The following steps will show you how to set up the stock used in the backplot.

Click on the *Solid Setup* icon in the Backplot *Solid* toolbar, indicated by arrow **A**. This will open the backplot Solid Setup window.



The default values shown in the dialog are based on the cutting moves in the NC program.

Since the tool moves down in cutting speed, the Z-max will nearly always be too big.

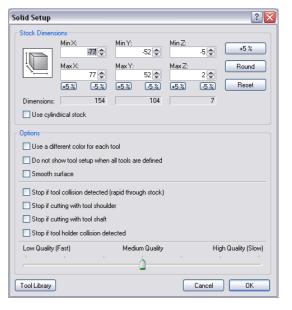
The same is the case for the facing operation. That will give a too large stock along both the X- and Y- axes.

From the facing operation, we know that the top of the stock should be Z:2.00. From the drawing, we know that the values of the corners are (-75.00, -50.00) and (75.00, 50.00).

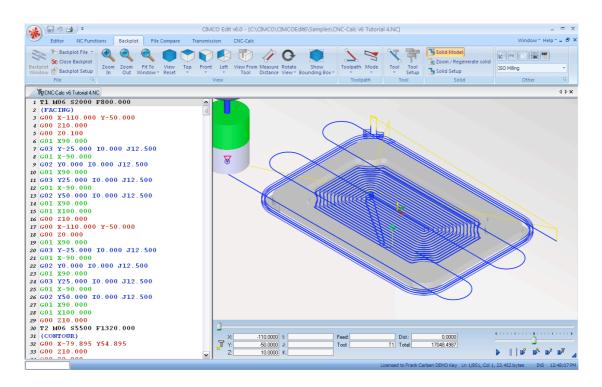
We now make the stock 2 mm larger along the X- and Y-axes, so the values will be the ones shown in the dialog to the right.

Please enter these values and exit the dialog with OK.

lid Setup							? 🔀
Stock Dimens	ions						
	Min X:	-35 🗢	Min Y:	7.098 🗢	Min Z:	-5.07 📚	+5 %
	MaxX:	77.5 🜩	Max Y: 11().549 🤤	Max Z:	2 🜩	Round
	+5 %	-5%	+5%	-5%	+5 %	-5%	Reset
Dimensions:		212.5		117.647		7.07	
Stop if cu	ol collision Itting with t Itting with t	detected (i ool should ool shaft ollision dete	er	ıgh stock)			
Low Quality	(Fast)		Mediu	ım Quality		Hi	gh Quality (Slow)
Tool Library					_	Cancel	ОК



Now, everything has been configured and the backplot can be used to verify the operations. The screen should now look like the one shown below.



15.Tutorial 6 Drilling (Mill)

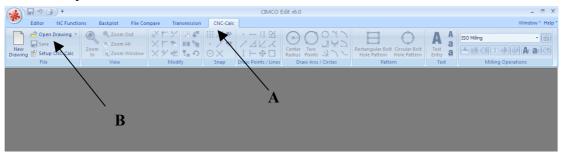
CNC-Calc v6 can generate codes for drilling in either canned cycles or as longhand. There are several machine types in CNC-Calc, but the most commonly used are ISO G-code programming and Heidenhain plain text.

15.1. Before you start

The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

Editor NC Functions Backplot File Compare Transmission CNC-Cale Wind Image: Compare So Turning Image: Compare	_ = X	CIMCO Edit v6.0
New Open Close Save Save Print So Utiling File File File File File File File File	Window * Help *	Clitor NC Functions Backplot File Compare Transmission CNC-Calc
General Plugins		New Open Close Save Print Il Copy Cut Paste Select Undo Redo All Il Copy Cut Paste Select Undo Redo All Il Copy Cut Paste Select Undo Redo Insert File Replace Go to Une/Block New Open Close Save Save Print Select Copy Cut Paste Select Undo Redo All Select Paste Select Copy Cut Paste Select Copy C
A Generative advanced bit/Seriel compare Disable File compare Disable Mazard Viewer Disable Mazard Viewer	С	A Plugins Plug

To open a drawing, click on **CNC-Calc** indicated by **A** and then on **Open Drawing** indicated by **B**.



Select the file **CNC-Calc Tutorial 1** and click Open.



Note If you hold the cursor over an icon for a moment, a short description of the icon's functionality will appear.

15.2. Generate a drill cycle

Start with selecting the programming format of the NC-program, the field indicated by arrow **A**.

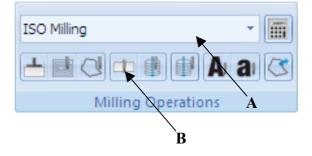
Select: ISO - Milling.

Then select **Generate a drill Cycle** by clicking on the icon indicated by arrow **B**.

Write the comment DRILLING in the *Comment* field.

This text will be included at the start of the final NC code for this operation. When multiple operations exist in the same NC program, the comments will help to locate and identify the start of each operation.

Click on *Drill Parameters*, to open the parameter dialog. window shown below.



Dri	illing
Comment:	
	DRILLING
📃 Use Selecti	on Filter
Filter	Drill Parameters
Reorder Circ	Reorder Rect
Back	New
Export Editor	Export Clipboard

Drilling Type Standard Peck Drilling	Controller Type TNC 430	Output Type Canned Longhand
Depths Retract Plane:	Reference Plane:	
10.0	0.0	
Safe Distance: Incremental O Absolute	Depth: O Incremental	Threading
2.0	-7.0	Pitch:
Feedrates		50.0
Use Plunging	Use Retract	O Tapping Speed:
200.0		100.0
		Retract Speed:
First Depth:	Pecking Retract:	
1.0		Dwelling
)egression Type	Minimum Depth:	Use Upper Dwelling
Value 😽	2.5	1.0
) egression:	First Feedrate Factor:	Use Lower Dwelling
1.0		

For this drilling operation, please enter the parameters shown. Notice that in this example it makes no difference if **Incremental** or **Absolute** are selected as **Safe Distance** and **Depth**, since these incremental values refer to the **Reference Plane**, which is 0.

For the selection of the location of the holes several options are available:

- Select each hole location with the cursor. In order to get the correct hole center for circles and arcs, the Snap to Center function should be used
- 2. Select the actual circle or arc. This will create a new hole location, at the centre of the circle/arc.
- 3. Use window selection with or without filter. If the filter is used, it is possible to limit the selection to circles or arcs in different ranges.

In the following we will use the filter to select the corner holes, but not any of the arcs.

By setting up the filter as shown, we will limit the window selection to include only circles in the range from 0 to 10 in diameter.

Selection Filter				
All Points				
Circles O All Circles	Arcs O All Arcs			
🔿 No Circles	 No Arcs 			
 Circles in Range 	O Arcs in Range			
Circles min. Diameter:	Arcs min. Diameter:			
0.0	0.0			
Circles max. Diameter:	Arcs max. Diameter:			
10.0	0.0			
	Cancel OK			

Now enable the **Use Selection Filter** in the left hand pane, and then make a window selection that includes the entire drawing.

When this selection is made, only the four corner holes should be selected.

Click on Export Editor. The following program is displayed in the editor.

(DRILL) G00 X12.5 Y87.5 G00 Z10.0 G83 X12.5 Y87.5 Z-7.0 R2.0 Q1.0 F200.0 X137.5 Y12.5 X12.5 G80

The order of operation can then be changed by clicking on **Reorder Circ** and **Reorder Rect** in the dialog.

16. Tutorial 7 Milling of Letters (Mill)

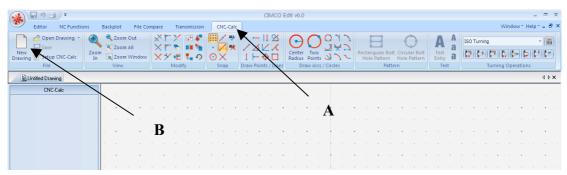
This tutorial demonstrates how a 2-dimensional text can be used as the basis for an NC program milling letters and numbers.

16.1. Before you start

The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

•		CIMCO Edit v6.0			_ = ×
Editor NC Functions Backplot File C	ompare Transmission	CNC-Calc			Window * Help *
• • • As •	ng Copy File Type	Cut Paste Select Undo Redo All Edit	X Del → Append File Dinsert File X Del → Find Previous X Find Previous X Find Next Find Kay K Replace	So to Line/Block Number Find	
A	Setup: Plugins Graneal Edicr Printing File types Colos Block numbers Load/Save File compare Machine Backplot CNC Call Mat Channel Tool Litt Other Global Coloss External Commands Mazartol Viewer Pluginis	Plugins Select plugins Diable Backplot Diable advanced simulation Diable ND/Sraid communication Diable File compare Diable Model Advanced NC-Functions Configuration password Configuration password Type file path: Help.			С
D		нер			

To make a new drawing you must click on **CNC-Calc** menu indicated by **A** and then select **New Drawing** indicated by **B**.



Note If you hold the cursor over an icon a short description of the icon's functionality will appear.

In this tutorial we will try to machine a single line of text composed of letters and numbers. For this example we have chosen the text "CIMCO 123".

Normal upper and lower case letters, numbers, and characters can be entered when the icon **A** is clicked in the CNC-Calc *Text* toolbar indicated with A.

The dialog **Text Entry** appears to the left. In the input field at the bottom named **text**, enter the text that will be machined. In this example "CIMCO 123".

In addition to the text, five additional parameters are needed to specify start point, baseline angle, letter distance, and letter height. Fill in the fields with the values shown to the right.

A	A A Text Entry A Text	
	Text Entry	
	Start Point X:	
	-68	°x
	Start Point Y:	
	-30	°۲
	Text Angle:	
	15	°۲
	Space Between Letters:	
	6	°x
	Text Height:	
	25	°x
	Text:	
	CIMCO 123	
	Alignment	
	\mathbf{X}	

When you are done entering text and values click the blue check mark button \checkmark at the bottom right of the dialog to accept.

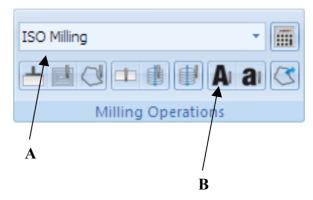
				CIMCO Edit v6.	D			_ = ×
10	Backplot File Con	npare Transmission	CNC-Calc					Window * Help * 🗕 🗗 🕻
New Drawing * Save Save File Com In	Coom Out Coom All Coom Window View	×たく ×たい ×たい ・ ・ Modify	· / · · · · · · · · · · · · · · · · · ·		r Two s Points & Circles	Rectangular Bolt Hole Pattern Pattern	Text A a ISO Miling	• IIIIng Operations
Untitled Drawing								4 ⊳ ×
Text Entry								
Start Point X:								
-68 • x							· · · ·	· · · · ·
-30 ° _Y								
Text Angle: 15 • _Y						A		<u>)</u>
Space Between Letters:								
6 •x							\setminus /)
25 ° _x	· · ·				· / · / ·		$\langle \cdot \cdot \cdot \rangle + \cdot \cdot$	· · · · · · ·
Text: CIMCD 123			Δ	C				
Alignment		1 1 1					1.5	
Element Inro		· · · / · /·	- × - \.	- <u>(</u> - , - , - , - , - , - , - , - , - , -				
		<u>.</u>						
	- i - i 🔪 i -	ang a Arras						
	· · · + ·							
							1	30.00
	Machine Zero X:	0.0 Machine	Zero Y: Ma 0.0	achine Zero Z: C	.0		Use Y-Axis Substitution	Part Diameter 50.0
Enter the text to draw						Licensed to Frank Carlsen DEMO	Key X: 9.502 Y: 10.879	INS 3:34:36 PM

Your screen should now look like the following.

The drawing now shows the text, and its geometry can now be used for generating the toolpaths.

Select the file type (NC Format) for our example letter milling program (e.g. ISO Milling) as indicated with the letter **A**.

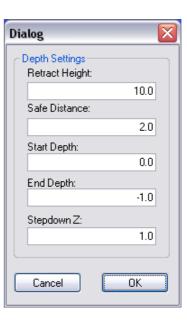
Then Select the feature **Letter Milling** by clicking on the *Mill Letters* icon indicated by the letter **B** in the Milling Operations toolbar.

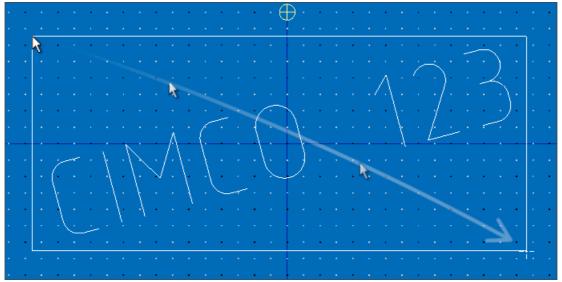


The dialog **Letter Milling** will appear on the left side of the screen as shown. Write the comment LETTER MILLING in the *Comment* field. Now click on the *Parameter* button and enter the values shown below.

Letter Milling
Comment: LETTER MILLING
Parameters
Back New
Export Editor Export Clipboard

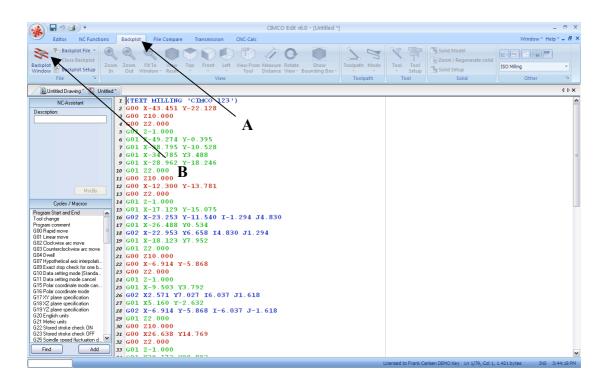
Now, use window selection to select the text you want to mill. On the drawing, left-click one corner, hold down the mouse button, and drag diagonally. When the desired text is framed, release the button. The frame disappears and the text will have turned yellow to indicate that it has been selected.





Next click on the **Export Editor** button, and the NC codes for machining the text will be shown in the Editor.

To verify the generated toolpath we must simulate it using the integrated Graphical Backplot. To open the backplot window, click on the **Backplot** tab (indicated with letter **A**) and then on the **Backplot Window** icon (indicated with letter **B**) as shown below.

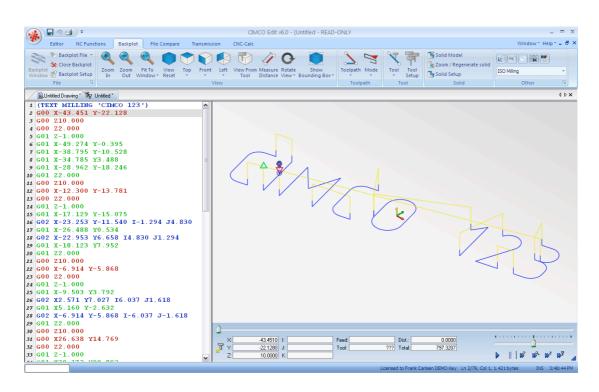


At the bottom right of the backplot window, start the simulation by clicking **Start/Stop Simulation**, indicated by the icon **D**.

Simulation speed and direction is infinitely variable both forwards and backwards. This is controlled by dragging the slider either to the left or to the right, where right is forward.



If you want to verify a certain operation in the NC program, simply click on a line of the NC code to the left. The simulation tool will immediately position itself on the corresponding place in the simulation. You can move the tool one line at a time using the up and down arrow keys on your keyboard, or skip through the code a page at a time using PageUp and PageDown.



Your screen should now present "CIMCO 123" in the following way.

Notice the rapid moves, indicated by the yellow lines, retracting to the level we defined using the Parameters dialog.

The example text milling program "CIMCO 123" can be used as a subprogram to another program by simply cut-and-paste, but can also be completed as an independent program, providing it is supplied with the code lines for **Program Start/Program Stop**, **Tool Change**, and **Feed/Speed**, which you can quickly add either manually or using the **Macro function** in CIMCO Edit 6.

Important notice

The final execution of the program depends to a high degree on the applied macro programs. It is also important that the correct set-up of CNC-Calc is used for each machine/control.

It is very important to verify/simulate the programs before they are executed on a machine. Please pay special attention to the movements in the Z axis, and make sure that they run with the required feed and rapid move speed.

17. Tutorial 8 Milling of TrueType Letters (Mill)

This tutorial demonstrates how a 2-dimensional TrueType text can provide the basis for an NC program milling letters and numbers.

17.1. Before you start

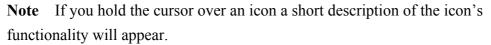
The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

		CIMCO Edit v6.0	_ = x
Editor	NC Functions Backplot File	Compare Transmission CNC-Calc	Window * Help *
New Open	Close Save Save Print	Append File V Find Next Previous Tool cha	
	_	Setup: Plugins	
В	A	General Plugins Printing Disable BACkplot Disable BACkplot Disable DACMasc client Disable BACkplot Disable DACMasc client Disable DACMasc client Disable DACMasc client Disable DACMasc client Disable DACMasc client Disable DACMasc client Disable DACMasc client Disable CACCate Disable CACCate Disable DACMasc and NCFunctions Disable Mascator Viewer Disable DACMasc and Disable Mascator Viewer Disable DAccate Compare Disable DACCate Disable DAccate Viewer Disable CACCate Disable DAccate Viewer Disable DACCate Disable Daccate Viewer Disable Daccate DACCate Disable Daccate Viewer Disable Daccate DACCate Disable Daccate Viewer Disable Daccate DACCate Disable Daccate DACCate Disable Daccate DACCate Disable Daccate DACCate	- C
		CIMCO	Edit 6.0
		Licensed to Frank Carlsen DEMO Key	INS 2:48:23 PM

To make a new drawing you must click on the **CNC-Calc** menu indicated by **A** and then select **New Drawing** indicated by **B**.

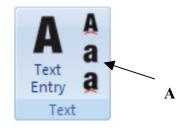
When the new drawing is opened the following window should now be displayed:

A			CIMCO Edit v6.0		- 0
Editor NC Functions		ompare Transmission CNC-	N N		Window* Help* 🗕 🗗
New Save Zoc Drawing Vetup CNC-Calc	Zoom Out Zoom All	Xrs 🖬 🔽		Rectangular Bolt Circular Bolt Hole Pattern Hole Pattern	ISO Turning • 📰
File	View	Modify Snap	Draw Points / Does Draw Arcs / Circles	Pattern Text	Turning Operations
Untitled Drawing					4 ⊳3
CNC-Calc					
			Α		
		B			
		D			
Element Info					
LIGHTER HIE					
					· · · · 40:00 · · ·
			<u>.</u>		
	Machine Zero Z:	Machine Zero X: 30.0	-18.0	Use Y	Part Diameter
]	sition for start and endpo			Licensed to Frank Carlsen DEMO K	



In this tutorial we will machine a single line of TrueType text composed of letters and numbers. For this example we have chosen the text "CIMCO 456".

Normal upper and lower case letters, numbers, and characters can be entered when the icon **a** is clicked in the CNC-Calc *Text* toolbar indicated with A.



The dialog **Text Entry** appears to the left. Enter the coordinates for the starting point of the text, the angle (relative to the horizontal axis) of the text baseline, and the height of the letters.

In the bottom field *text* you write the text (here: "CIMCO 456") to be milled with TrueType letters.

Text Entry	
Start Point X:	
-	68 °x
Start Point Y:	
	30 °r
Text Angle:	
	30 ° _Y
Text Height:	
25	i.0 °x
Text:	
CIMCO 4	56
Alignment Select F	ont
2	<

Next, by clicking on the button **Select Font** and opening the font dialog, select the font type and font size. End this dialog by clicking 'OK'. As a result of the changes you make in the font dialog the look of the text changes. You can enter the font dialog again until you are satisfied with the layout.

When done, click on the blue check mark button \checkmark to insert the text. This is important, since otherwise the text will disappear once you start doing other things.

Font					×
Eont: Times New Roman TRAJAN PRO Trebuchet MS Tw Cen MT	•	Font style: Bold Regular <i>Italic</i> Bold Bold Italic	*	Size: 18 10 11 12 14 16 18	OK Cancel
Verdana	Ŧ	Sample AaBb	÷ Yy	¹⁸ 20 ∓	
		Script: Western		•	

With parameters, text, and font defined your screen should look like the following. Depending on your choice of font this might vary.

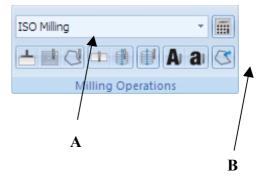
			O Edit v6.0		_ = X
(nission CNC-Calc			Window * Help * 🗕 🗗 🗙
Copen Drawing *	Zoom Out	ଃ≰" Ⅲ/₩ · ー ⊻ ■ ►	$\Theta O S S$		A ISO Miling -
New Drawing Setup CNC-Calc In	Zoom Window		Center Two Radius Points 🕺 🦒 🔪		a 🗕 🖻 🖓 🖿 🌒 🕼 A a 🔇
File	View Modify	Snap Draw Points / Lir		Pattern Text	Milling Operations
Untitled Drawing					4 Þ 🗙
Text Entry					
Start Point X:					1
-68 • _x					
Start Point Y: -30 • Y					$\{\mathcal{N}_{i}\}_{i=1}^{n}$, \mathcal{N}_{i} , \mathcal{N}_{i}
Text Angle:				and the second second	$\langle \langle \rangle \rangle$ \rightarrow \rightarrow \rightarrow \rightarrow \rightarrow
30 • Y	• • • • • • •				
Text Height: 25.0 • x					Y
Text					
CIMCD 456				a fa fa la comencia de la	
Alignment Select Font				·)	
) a a a a a a	
Element Info			$\langle \cdot \cdot \rangle = - h \sum_{i=1}^{n} h_{i}$	· · · · · · · ·	
		STA V			
			19		
		$\simeq 1 / 1 / 1 / 1$	de la construcción de la constru		
		· · · · · · · · · · · · · · · · · · ·			
	· · · · · · · · · · · (· · ·				
		Se e e e e e		· · · · · · · · ·	40.00
	· · · · · · · +				
		Machine Zero Y: Machine Zero			Part Diameter
	0.0	0.0	0.0	🛄 Us	e Y-Axis Substitution 50.0
Enter the text to draw				Licensed to Frank Carlsen DEM	IO Key X: 31.402 Y: 60.029 INS 12:15:51 PM

The geometry is now finished and can be used for the generation of toolpaths.

Select the file type (NC Format) for our example letter milling program (e.g. ISO Milling) as indicated with the letter A.

Then Select the feature **True Type Milling** by clicking on the *Mill True Type Letters* icon indicated by the letter B in the Milling Operations toolbar.

Now the dialog Letter Milling dialog is opened to the left of the drawing area. Write "LETTER MILLING" in the *Comment* field. Click on the *Parameter* button.



Letter Milling
Comment: LETTER MILLING
Show Toolpath Parameters
Back New
Export Editor Export Clipboard

The **TrueType Text Milling Parameters** dialog appears. Check the **Mill Outline** checkbox at the top and fill in the rest of the fields as shown below. Click 'OK' when done.

True Type Text Milling Paran	neters 🛛 🔀
Milling Operation	Mill Interior
Cutter Diameter: 0.7	Stepover Stepdown Z:
Depth Settings Retract Height: 5.0	1.0 Stepover XY: 0.4
Safe Distance:	Stock to Leave
Start Depth: 0.0	Stock to Leave Z:
End Depth: -0.5	Stock to Leave XY:
	Cancel OK

Use window selection to select the letters to mill. This is done by left-clicking in the upper-left corner of the drawing. Now hold the mouse button down and drag the cursor to the lower right corner and release the mouse button.

Click on the **Show Toolpath** button. Now the generated toolpath will be shown on the drawing. Since the toolpath coincides with the contour of the letters it can be difficult to see.

Now click on the **Export Editor** button in order to transfer the generated program to the editor.

With the generated program in the editor it is now possible to backplot it. This is done by first selecting the **Backplot** Menu indicated by **A** below and then clicking on the

			CIMCO Edit v6.0	- Il Intitled *1				x
Editor NC Functions	Backplot File Compare	Transmission	CNC-Calc	founded 1			Window * He	elp×_ 8 ×
	om Zoom Fit To Out Window - Reset	op Front Left View	View From Measure Rotate Tool Distance View * Box	Show unding Box	Tool Tool Setup Tool	Solid Model	ISO Miling Other	•
Untitled Drawing	d*							4 Þ ×
NC-Assistant Description:	1 (TRUE TYPE TE 2 G00 X-41.822 3 G00 25.000 6 G0 21.500 6 G0 X-41.589 7 G01 X-49.681 10 G01 X-49.681 10 G01 X-49.681 11 G01 X-49.087 12 G01 X-49.681 12 G01 X-49.687 12 G01 X-49.687 12 G01 X-48.689	Y-14.348 Y-14.752 Y-19.596 Y-19.193 Y-18.885 Y-18.678 Y-18.459	A					1
Cycles / Macis Cycles / Macis Program Star And End Tod change Program comment G00 Appid move G01 Lines move G02 Clockwise arc move G03 CounterCockwise arc move G04 Ovel G03 CounterCockwise arc G05 Plata coordinate mode can G15 Plata coordinate mode G15 Plata coordinate mode	13 G01 X-48 703 14 G01 X-48 761 15 G01 X-48 562 16 G01 X-48 562 17 G01 X-48 562 18 G01 X-48 562 19 G01 X-48 766 20 G01 X-48 94 21 G01 X-48 94 22 G01 X-54 754 22 G01 X-55 758 23 G01 X-55 758 24 G01 X-56 714	Y-17.985 Y-17.802 Y-17.600 Y-17.378 Y-17.136 Y-16.900 Y-16.557 Y-16.107 Y-15.549 Y-4.952 Y-4.951 Y-3.957 Y-3.649						
G17 ½ plane specification G18 ½ plane specification G18 ½ plane specification G20 English units G21 Metic units G22 Stored stoke check DN G23 Stored stoke check DF G25 Spindle speed fluctuation d	26 G01 X-56.454 27 G01 X-56.622 28 G01 X-56.819 29 G01 X-57.043 30 G01 X-57.295 32 G01 X-57.570 32 G01 X-57.862 33 G01 X-58.172	Y-3.363 Y-3.288 Y-3.242 Y-3.225 Y-3.247 Y-3.318 Y-3.439						V

Backplot Window icon indicated by the letter B.

Use the buttons in the lower right corner of the window to control the simulation speed and direction. By clicking on a line in the NC code to the left, the tool will jump to that position in the simulation. The up and down keys move the line selection to the previous or next line, and the tool will be moved accordingly.

📣 🖬 🔊 🔒) :	Ŧ				CIMCO E	dit v6.0 - [Untitled	[*]			-	
Editor NO	C Functions Back	plot File Co	mpare Tra	ansmission	CNC-Calc					Window * Help	8
Backplot F Backplot Window	kplot		Tiew Top eset *	Front Left	View From Measure Ro Tool Distance Vi	otate Show ew * Bounding Box	Toolpath Mode	Tool Tool Setup	Solid Model Com / Regenerate solid Solid Setup Solid	ISO Miling	*
Untitled Drawing*	* belftell)(4 Þ ×
2 500 X-41. 3 600 Z5.00 4 600 Z1.50 5 601 Z-0.5 6 601 X-41. 7 601 X-49. 8 601 X-50.	0 0 589 Y-14.75 980 Y-19.59 980 Y-19.59 681 Y-18.67 087 Y-18.45 869 Y-18.45 869 Y-18.22 703 Y-17.98 615 Y-17.80 562 Y-17.30 562 Y-17.13 662 Y-17.43 662 Y-16.50 94 Y-16.10 305 Y-15.54 423 Y-4.952 758 Y-4.952 758 Y-4.952 758 Y-3.9649 454 Y-3.467 273 Y-3.649 454 Y-3.467	12 12 16 13 15 15 15 15 15 15 15 15 15 15				U	C (56	
29 G01 X-57. 30 G01 X-57.				-41.822	20 1	Feed	Dist.:	0.0000		4	

17.2. Pocket milling letters

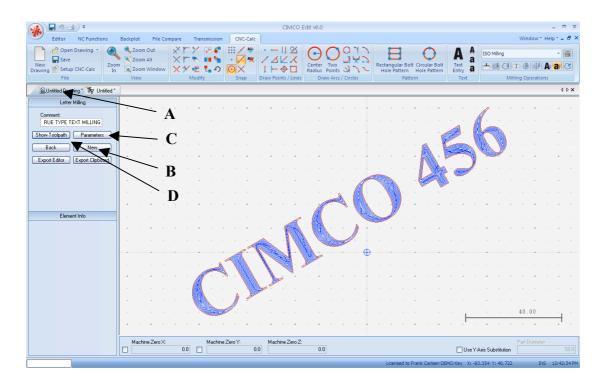
The toolpath generated from the text looks like a contour operation formed by the outlines of the letters and numbers. The area inside these contours - inside the letters - can be milled as a special pocket milling operation. The rest of this tutorial will show you how to do this.

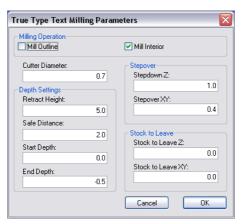
1. Return to the drawing by clicking the tab marked A in the Figure below.

2. Click on the button New marked B, and select the letters for the operation.

3. Click on the button **Parameters** marked **C**.

4. Insert the new parameters as shown below (remember to check the checkbox **Mill Interior**).





- 5. Window-select the text.
- 6. Click on **Show Toolpath** marked **D** the generated toolpaths will now be shown on the screen.
- 7. Click the button **Export Editor.** Now the NC codes for the machining are shown in the editor.

Simulate the program to verify the toolpath the same way we did earlier in this tutorial.

CIMCO Edit v6.0 - [Untitled "]	_ = ×
Editor NC Functions Backplot File Compare Transmission CNC-Calc	Window * Help * 🗕 🗗 🗙
Resulptor File - Received and Acadel Received and Acadel Received and	2 🗆 🗅 🗰 🖷
Window 们 Backplot Setup In Out Window* Reset * * * Tool Distance View* Bounding Box* * * Setup In Solid Setup	SO Miling *
	Other 🕫
Burniled Drawing * Top United * Top United *	4 b ×
1 (TRUE TYPE TEXT MILLING)	
3 GOO 25,000	
4 600 22.000	
5 G01 Z-0.500	
6 G03 X-47.002 Y-15.277 I1.620 J0.804	
7 G03 X-46.871 Y-15.316 I0.084 J0.042	
8 G01 X-46.851 Y-15.305	
9 G03 X-46.819 Y-15.171 I-0.047 J0.082	
20 G01 X-47.078 Y-14.777	
11 G02 X-47.311 Y-14.398 16.599 J4.324	
10 601 x-47, 078 y-14, 777 100 502 11 602 x-47, 078 y-14, 777 100 502 11 602 x-73 311 y-14, 398 16.5 593 J.324 12 601 x-53, 747 Y-3 210 100 100 100 12 602 x-53, 747 Y-3 910 108 70.5 11 14 603 x-53, 861 Y-309 10.687 100 100 100 16 53, 538 Y-2988 1-1.666 J-0.042 100	
B 502 X-53.501 Y-5.099 I-1.66 J-0.704	
15 GO3 X-53,984 Y-2,984 J-0,084 J-0,042	
16 GO1 X-54, 003 Y-2, 960	
27 G03 X-54.035 Y-3.092 10.047 J-0.082	
18 G01 X-53.761 Y-3.513	
29 G02 X-53.544 Y-3.867 I-6.408 J-4.179	
20 G01 X-47.107 Y-15.055	
22 G03 X-47.007 Y-15.266 I1.666 J0.704	
23 G02 X-47.006 Y-15.525 I-0.263 J-0.131	
24 G03 X-46.816 Y-15.700 I0.122 J-0.059 25 G01 X-46.362 Y-15.438	
26 601 A - 40, 302 I - 10, 430 26 603 A - 46, 323 Y - 15, 272 I - 0, 059 J0, 102	
27 G03 X-46.669 Y-14.750 I-57,771 J-37,8	
28 G02 X-46, 999 Y-14, 218 17, 945 J5, 310	
29 G01 X-53, 475 Y-2, 974	
30 G02 X-53.584 Y-2.740 I3.675 J1.858	
32 G03 X-53.693 Y-2.506 I-3.785 J-1.624 X 47.0070 : Feed Dist: 0.0000	
32 G03 X-53.854 Y-2.458 I-0.103 J-0.052 TY. 152860 J. Tool ???? Total 3041.4204	
33 G01 X-54, 513 Y-2, 838 Z 50000 K	🕷 📽 🖓 🖓 🙍
Licensed to Frank Certain DEMO Key Ln 2/2.957, Col	1. 86.504 hytes INS 12:46:45 PM

It is possible to generate toolpaths with both **Mill Interior** and **Mill Outline** selected under **Parameters**. This will create both the pocket operation on the inside and the milling of the contours.

The example program "CIMCO 456" can be used as a subprogram to another program by simply cut-and-paste, but can also be completed as an independent program, providing it is supplied with the code lines for **Program Start/Program Stop**, **Tool Change**, and **Feed/Speed**, which you can quickly add either manually or using the Macro function in CIMCO Edit 6.

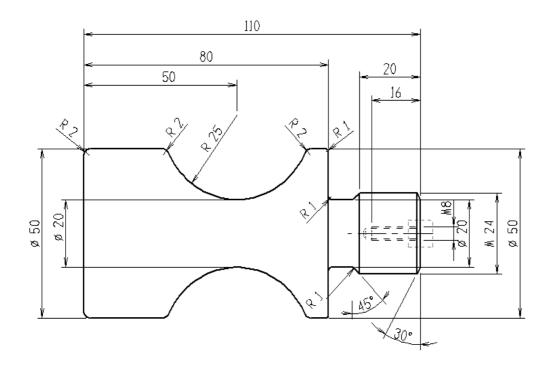
Important notice

The final execution of the program depends to a high degree of the applied macro programs. It is also important that the correct set-up of CNC-Calc is used for each machine/control.

It is very important to verify/simulate the programs before they are executed on a machine. Please pay special attention to the movements in the Z axis, and make sure that they run with the required feed and rapid move speed.

18. Tutorial L1 - Drawing the part (Lathe)

18.1. 2D construction of a part for turning.



This example demonstrates one of many ways in which the 2-dimensional part above can be drawn in CNC-Calc v6. Since the part consists of a number of similar elements and since its part-elements are symmetrical only a subsection of the part needs to be drawn. The rest emerges from mirroring. Finally joining the mirrored elements with straight lines completes the part.

This tutorial demonstrates the use of the following functions

- Draw a vertical line defined by its starting point and length.
- Offset a geometric element.
- Draw a circle with a given radius defined by its centre.
- Draw a line defined by its end points.
- Draw a horizontal line defined by its start point and length.
- Draw a line defined by its start point, angle, and length.
- Trim element between points of intersection with other elements.
- Connect and round with a given radius between two elements.
- Connect and bevel two elements by a given angle and distance.

18.2. Before you start

The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

(A)	CIMCO Edit v6.0	_ = ×
Editor NC Functions Backplot File Co	ompare Transmission CNC-Calc	Window * Help *
AS T	Image: Select Undo <	
A	Setup: Plugins Image: Setup setu	С
B	Hep Default Cancel DK	

To make a new drawing you must click on the **CNC-Calc** menu indicated by **A** and then select **New Drawing** indicated by **B**.

When the new drawing is opened the following window should now be displayed:

🔬 🗐 🕫 🖶 🕫	CIMCO	Edit v6.0	_ = X
Editor NC Functions Backplot File Co	ompare Transmission CNC-Calc		Window * Help * 🗕 🗗 🗙
Preving Save Drawing Stup CINC-Calc File	X T Y Z C X X T C X Z X X X X X X X X X X X X X X X X X	Center Two Radius Points 2010 Draw Arcs / Circles s	A A A ISO Turning ISO Turning Text Text Turning Operations
Luntitled Drawing			4 ⊳ ×
CNCCak	B		

Note If you hold the cursor over an icon a short description of the icon's functionality will appear.

You can change the colors of the drawing area by selecting **Setup** and then **Colors** from the dropdown menu. For this tutorial we have chosen to use red as our drawing color and white for the background.

18.3. Draw the geometry

Draw vertical lines defined by start point and length

Vertical Line	Click on	in the Draw Points / Lines toolbar and				
Start Point Of Line X:	enter the fo	enter the following values:				
0 î _x						
Start Point Of Line Y:	Start]	Point of Line Z = 0				
Line Length:	Start]	Point of Line X = 0				
12 1 2	Line I	Length $= 12$				
	Click on	to approve the command.				
$\mathbf{\times}$		are shown it can be difficult to see the it is situated on the X-axis				

Draw another vertical line defined by start point and length

Vertical Line	Enter the followi
Start Point Of Line X:	already open:
-110 î _x	
Start Point Of Line Y:	Start Point
Line Length:	Start Point
25 1	Line Length
	Click on 🔽 to a
$\mathbf{\times}$	

Enter the following values in the dialog that is already open:

```
Start Point of Line Z = -110
Start Point of Line X = 0
Line Length = 25
```

Click on \checkmark to approve the command.

Click on in the *View* toolbar. This will make the drawing fill the whole drawing area on the screen.

Offset an element

	Offset E	lement		
Offset D	istance:		80	>
Steps:				
			1	87 87
🔽 Сору	(unche	ck to Mo	ve)	
				$\mathbf{\times}$

Click on *in the Modify* toolbar and enter the following value:

Offset Distance = 80

Now click on the leftmost line indicated by **A** on the picture below.

After this is done two lines will appear (only the rightmost can be seen on the drawing). Now click to the right of the selected line to keep the line shown in red.

(MA)	96	•									(CIMCO	Edit v6.0													. = x
	Editor	NC Functio	ons	Backplot	File Cor	mpare	Transm	ission	CNC-	Calc														Wind	low* Help	- 8 ×
New Drawin	🔚 Save	p CNC-Calc	Zoom	Zoom Zoom Zoom View		×r ×r ×*	Nodify	• •	· / • / • X Snap	**				Points	O 1 シマン シン ノ ノ ノ ノ	CD7	Rectangul Hole Par	ar Bolt ttern I Patterr		Bolt ern	A A a a a a a a a a a a a a a a a a a a		Turning		erations	
	Untitled Dra	wing *	<u> </u>																			^				4 Þ 🗙
		Element																					_			
	ifset Distance eps:	80	>																							
	Copy (unch	eck to Move)		л <u>к</u>										,												
	Elem	ent Info	\mathbf{X}					•						•		•								•		
																,										
					•		- A																			
																					<i>•</i>			25.0		
				Machi	ine Zero Z:	0.0		1achine i	Zero X:	0.0											Use Y	Axis Su	ibstitution			
		Select the off	set elem	ent to keep														Licer	nsed to Fr	rank Ca	arisen DEMO	Key Z:	-52.607	D: 27.67	7 INS	2:24:08 PM

Draw a circ	le with	radius =	= 25 d	lefined	by	its	center
-------------	---------	----------	--------	---------	----	-----	--------

Circle Center And Radius	Click on \bigcirc in the toolbar.
Center Point Z: Center Point X: Circle Radius: 25 S	Center Point Z = -110+50 (-60) Center Point X = 70 Circle Radius = 25
	Click on 🗹 to approve the command.

Draw a line defined by its endpoints.

First click on \checkmark to activate the snap function '*Snap to Endpoints*' labeled A on the drawing below.

Now click on \checkmark to enter '*Draw line Between 2 Points*' labeled **B** on the drawing below.

Select the top point of the two long lines indicated by C and D. Please notice that the cursor changes when it snaps to the endpoint of the lines. After the line is drawn the drawing should look like the one below.

•			CIMCO	Edit v6.0		_ = ×
Editor NC Functions	Backplot File Co		CNC-Calc			Window* Help* = 🗗 🗙
📄 🔗 Open Drawing 👻 💮	Coom Out	XTYZ		$\Theta O O O O O O O O O O O O O O O O O O O$		ISO Turning 🔹 🧰
New Drawing Setup CNC-Calc In	n Zoom All	XXX		Center Two Radius Points 2 7 C Rectangula Hole Pat	ar Bolt Circular Bolt tern Hole Pattern Entry	N
File	View	Modify	Snap Draw Points / Lines		Pattern Text	Turning Operations
Untitled Drawing *						4 ⊳ ×
Line Through 2 Points			$\sim 1 \sim 10^{-1}$	D		
First Point On Line Z:			A	В		
First Point On Line X:	· · ·					
Second Point On Line Z:				· · · · · · · · · · · · · · · · · · ·	<pre></pre>	
Second Point Un Line 2:					$\langle \rangle$	
Second Point On Line X:					- <u>_</u>	
Polyline			· · · ·	• • •		
Element Info	· ·				· /· · · ·	
					/ _ · _ · ·	
				/		T
		, i C i A	a sa ka 🔪 ka			$\mathbf{D}_{\mathbf{r}}$, \mathbf{r} , \mathbf{r}
						25.00
				· · · ·	I . F.	· · · · · · · · · · · · · · · · · · ·
	Machine Zero Z:	Machine	Zara V.			Part Diameter
	Machine Zero Z:	0.0	0.0		Use Y-	Axis Substitution 50.0
Indicate or enter posi	ition for start and endpo	int			Licensed to Frank Carlsen DEMO H	Key Z: -21.701 D: 65.869 INS 2:27:51 PM

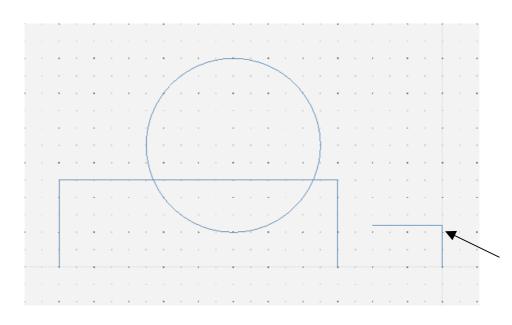
Draw horizontal line defined by length and using snap

Horizontal Line	
Start Point Of Line X:	
	x
Start Point Of Line Y:	
	T
Line Length:	
-20	=

Click on in the toolbar and enter the following values:

Line Length = -20

with the end point snap enabled. Select the end of the short vertical line furthest to the right. This is indicated on the picture below.



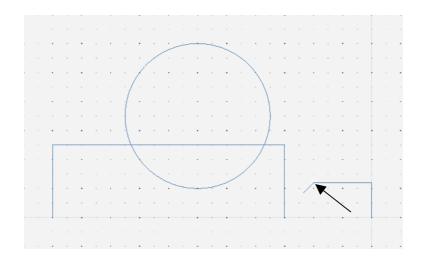
Draw Polar line defined by angle, length, and using snap

Polar Line		
Start Point Of Line X:		
		√ x
Start Point Of Line Y:		
		ИY
Line Angle:		_
	180+45	
Line Length:		
	5.0	1
	$\mathbf{\times}$	\checkmark

Click on \checkmark in the toolbar and enter the following values:

Line Angle = 180+45 Line Length = 5

With the end point snap enabled. Select the end of the short horizontal line. This is indicated on the picture below.



Chamfer the foremost corner

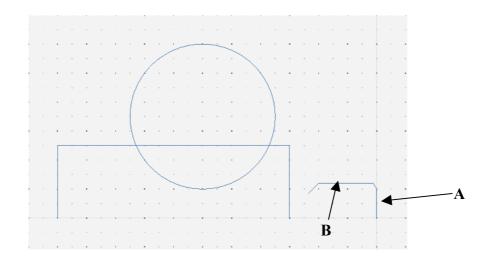
Chamfer Lines
O One Distance
◯ Two Distances
 Distance and Angle
First Length
2 4
Second Length
0.0 🎽
Chamfer Angle
30 🏲
✓ Trim Elements
\mathbf{X}

Click on T in the toolbar and enter the following values:

First Length = 2 Chamfer Angle = 30

Since the chamfer angle is different from 45 degrees it is important to select the lines in the right order. The angle will always be measured from the first element selected.

So first select the vertical line marked **A** and then the horizontal line marked **B** on the drawing below.



Draw a line defined by its endpoints

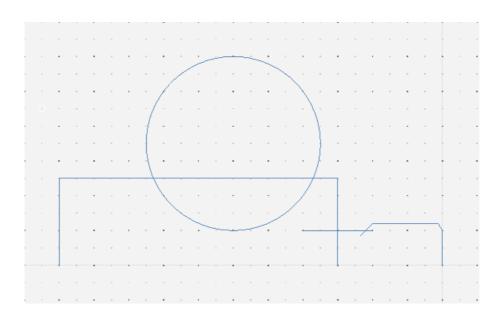
Line Through 2 Points
First Point On Line X:
-20 🖍
First Point On Line Y:
🗖 20 🖧
Second Point On Line X:
-40 🖍
Second Point On Line Y:
🗖 20 🖍
Polyline
$\mathbf{\times}$

Click on *Line through 2 points* / in the toolbar and enter the following values:

First Point On Line Z = -20 First Point On Line X = 20 Second Point On Line Z = -40 Second Point On Line X = 20

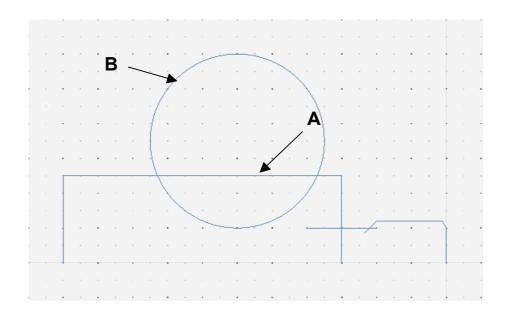
Click on \checkmark to approve the command.

Now the drawing should look like the picture below.



Trim Between Points of intersection

- Click on \times to select *Trim To Intersection*.
- Now trim the long horizontal line. To do this, select it as indicated by **A** on the picture below.
- Now trim the large circle. To do this, select it as indicated by **B** on the drawing below.



Fillet intersections with radius 2.0

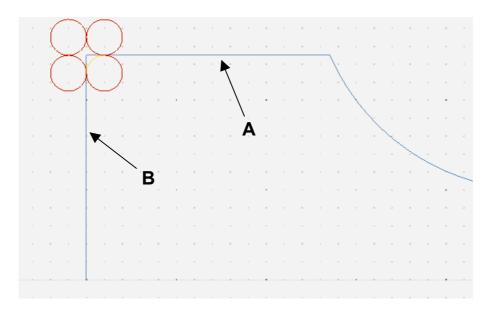
Fillet / Rounding	
Fillet Radius:	
	2 💦
🗹 Trim Elements	
	\mathbf{X}
	<u> </u>

Click on T in the toolbar and enter the following values:

Fillet Radius = 2

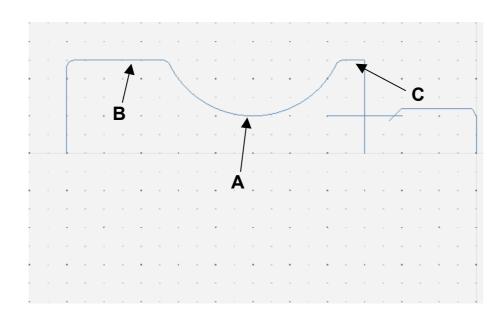
Select the elements on which the fillet operation should be performed. This is done by left-clicking on the part of the elements that you want to keep. Select the elements as indicated on the picture below by **A** and **B**.

Now you must select and left-click precisely on the arc element you want to keep. This is the yellow arc on the picture below.



Now repeat the operation to fillet the additional radius 2 corners

Select the other two corners A+B and A+C as shown at the picture below. Then select the correct arcs to keep. At the end of the operation the drawing should look something like the drawing below.

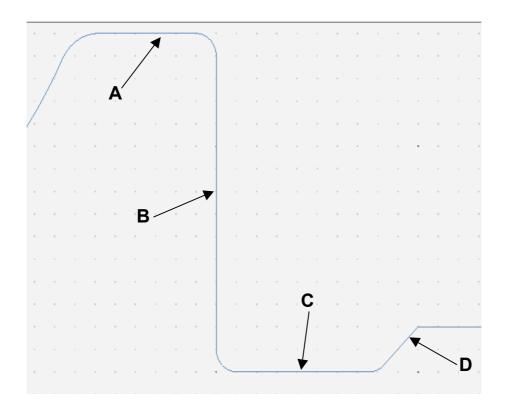


Now repeat the operation for the fillet operations with radius 1.00 corners

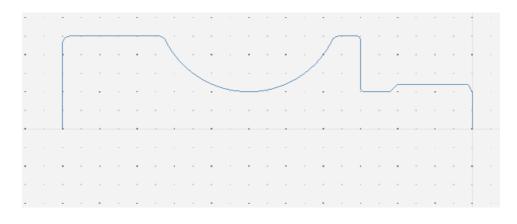
First change the Fillet Radius in the dialog from 2.00 to 1.00.

You might want to zoom in on the area we will be working on, as on the following picture.

Then select the 3 corners A+B, B+C and C+D as shown on the picture below. Then select the correct arcs to keep. At the end of the operation the drawing should look something like the drawing below.



Now the drawing is finished, and it should look like the one below

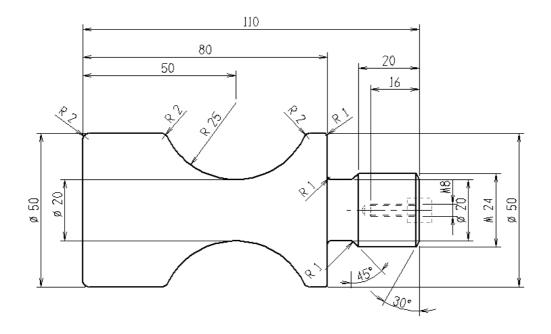


Name the file and save it

Click on Files and then select 'Save as' from the dropdown menu. Save the file with the name CNC-Calc V6 Lathe Tutorial 1 (the file extension is added automatically).

19. Tutorial L2 - Machining the part (Lathe)

19.1. CNC toolpaths



With CNC-Calc v6 it is possible to create toolpaths directly from the program's geometrical drawings. Thereby, calculations become more secure and programming becomes much faster compared to doing it manually. At the same time you get a big advantage since it is possible to move, copy, rotate, scale, and mirror elements with the result of instant NC-code generation.

In the following we assume that the stock used is Ø60, and that it projects sufficiently from the Chuck Jaws.

In order to produce the final part we will use the following operations:

- 4. Facing the front of the stock.
- 5. Roughing the part.
- 6. Grooving the areas that could not be handled by the roughing tool.
- 7. Finishing the part.
- 8. Threading the front of the part.
- 9. Drilling the center hole in the part.

This tutorial demonstrates how the 2-dimensional part shown can form the basis of NC-codes for various types of machining.

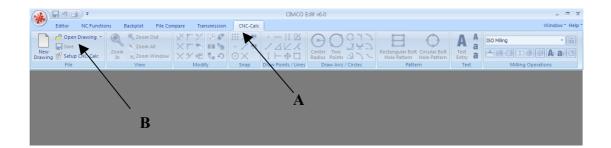
Note This tutorial builds upon the result from CNC-Calc v6 Tutorial 1.

19.2. Before you start

The first thing to do before drawing a new part is to set the menu parameters. Start **CIMCO Edit v6** and open the editor configuration by clicking on the icon indicated by **A**. Go to the Plugins section indicated by **B**. Now ensure that the checkbox *Disable CNC-Calc* indicated by **C** is unchecked.

Editor NC Functions	Backplot File Con	npare Transmission	CIMCO Edit v6.0			_ ♂ X Window™ Help™
New Open Close Save Save File		Copy e Type	Cut Paste Select Undo Redo All Edit	X Del ~ Append File Insert File	IS IN Next Tool change Go to Line/Block Number Find	
A		Setup: Plugins General Edicr Pining File types Colors Colors Bock numbers Load/Save File compate Machine Backplot OHCCale Other Olobel Colors External Commands Macatrol Viewer Plugins	Plugins Select plugins Ditable dokptot Ditable advanced simulation Ditable File compare Ditable edvanced NC-Foreid communication Ditable File compare Ditable File compare </th <th>Disable NCBase Disable NCCBase Disable ONCMax clent Disable ONCCas Disable Mazotal Viewer</th> <th></th> <th>С</th>	Disable NCBase Disable NCCBase Disable ONCMax clent Disable ONCCas Disable Mazotal Viewer		С
В			Help	Default Cancel OK		

To open a drawing, click on **CNC-Calc** indicated by **A** and then on **Open Drawing** indicated by **B**.



Select the file CNC-Calc V6 Lathe Tutorial 1.cdd and click Open.

Åbn	? 🔀
<u>S</u> øgi: 🗀	Drawings 💽 🕑 😥 🖽 🗸
CNC-Calc \	/6 Lathe Tutorial 1.cdd
Fil <u>n</u> avn:	CNC-Calc V6 Lathe Tutorial 1.cdd
Filtype:	CNC-Calc Drawing files (*.cdd, *.dxf)

A • a •								_	CIMCO	Edit v6.0)											
Editor NC Function	ns E Zoom In	Backplot Coor Coor Coor Coor View	n All n Window	×r ×r ×*	Transm	€ 4 " • *	CNC-Calc	· - / \ -	II ⊗ ∠ ⋌ ◆ □ ints / Lines		Two Points raw Arcs			ectangul Hole Pat	ar Bolt (tern F Pattern	Circular Bol tole Pattern	A A a iext ntry Text	ISO T		Windo	e 🕞	,
CNC-Calc V6 Lathe Tutorial	1.cdd																					4 Þ
CNC-Calc	_																					
									•		•							•		·		į
		- (. \															
Element Info	_								_				2						-		-	
																		/.				
																				• 25.0		•
					•	•					-	-			-							
		Mach	ine Zero Z:			lachine.	Zero X: O	.0									🗌 Use Y	'Axis Sub	ostitution	Part Dia	meter	

You should now see the part from CNC-Calc v6 Tutorial 1 displayed.

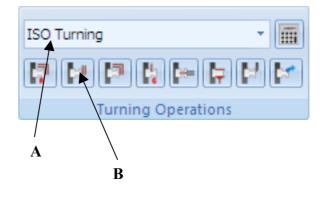
Note If you hold the cursor over an icon a short description of the icon's functionality will appear.

19.3. Facing the front of the stock

In the following tutorial we will generate an NC program to face the front of the part. In order to select the facing operation the below described steps must be performed.

Select the file type (NC Format) for our facing example program (e.g. ISO Turning) as indicated with the letter **A**.

Then Select the feature **Face Turning** by clicking on the *Face Turning* icon indicated by the letter **B** in the *Turning Operations* toolbar.



This will open the Face Turning dialog to the left of the drawing area. Now insert the values shown in the dialog below.

Face Turning	
Comment:	
FACING	
Start Point Z:	
5 🔽	
Start Point X:	
60 🔽	
End Point Z:	
0 Z	
End Point X:	
O 🛀	
Parameters	
T drameters	
Show Toolpath New	
Export Editor Export Clipboard	

Comment:

This comment will be shown in the final NC-Program. It is always good to include a comment, in order to distinguish the various operations in the final program.

Start Point Z:

This is where the facing operation will start along the Z-axis.

Start Point X:

This is the start diameter of the facing operation. **End Point Z:**

This is where the facing operation will end along the Z-Axis.

End Point X:

This is the end diameter of the facing operation.

Now we have defined where the facing operation will work on the stock. This will be shown on the drawing as a rectangle with arrows that indicate the direction of the operation. Click the parameters button in the dialog to define how the operation will be performed. Enter the following values shown below into the parameter dialogs.

Face Turning Parameters	X
Cuts Entry/Exit	
Tool Definition	Roughing Roughing Stepover
	3
$\mathbf{\overline{C}} \mathbf{\overline{O}} \mathbf{\overline{)}}$	Finish Finish Stepover
	0.2
	Finish Cuts:
Tool Radius	1
0vercut Amount:	Stock to Leave: 0.0
Finish Compensation	
Computer	~
	OK Annuller

Cuts Tab Configures the tool setup and other parameters for the operation

Tool Orientation: The 9 icons represent the possible 9 orientations of the tool. **Tool Radius:** This is the nose radius of the tool.

Overcut Amount: The distance the tool will cut longer than the Endpoint X value. **Roughing:** Here, the check-box can be used to enable or disable the use of roughing passes. If roughing is used, each cut will be the size of the *Roughing Stepover*.

Finish: The check box can be used to enable or disable the use of finish passes. If finish passes are used, *Finish Cuts* passes will be made with a depth of the *Finish Stepover*.

Stock To Leave: The stock to leave is the amount of material that will be left after the whole operation is performed.

Compensation Type: The compensation type that is used for the operation. The two most commonly used are *Controller* or *Computer*.

Entry/Retract	
ntry Amount:	Entry Vector
2.0	Use Entry Vector
	Entry Angle
etract Amount:	0.0
2.0	Futur Lawath
	Entry Length
	1.0
	- Exit Vector
	Use Exit Vector
	Exit Angle 0.0
	0.0
	Exit Length
	1.0

Entry/Exit Tab Configure how the tool approaches and leave the part.

Entry Amount: The length that the tool will start each cut above *Start Point X*. Retract Amount: The length that the tool will pull free along the Z-Axis before it makes moves for the next cut. Use Entry Vector: Enable/Disable the use of entry vector.

Entry Angle: The angle of the entry vector.

Entry Length: The length of the entry vector.

Use Exit Vector: Enable/Disable the use of exit vector.

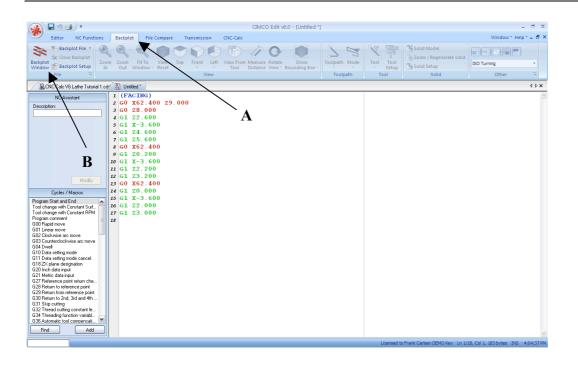
Exit Angle: The angle of the exit vector. **Exit Length:** The length of the exit vector.

Click on **OK** to use the values,

Click on Show Toolpath, and the toolpath will be shown on the drawing.

Try experimenting with the various parameters and see how they change the generated toolpath.

Click on **Export Editor** in order to generate the actual program. Now, a window like the one shown below will appear. Notice that the comment from the dialog is inserted at the top of the program as a comment.



To verify the generated toolpath, we must simulate it using the integrated Graphical Backplot.

To open the backplot window, click on the Backplot menu (indicated with letter A

above) and then on the **Backplot Window** icon (indicated with letter **B**) as shown above.

Now a window like the one below will appear.

Entry NC fundion Bodgiol The Compary Translation CAC/dit Window Window Sold Model	📣 🖶 47 🎿 🗧	CIMCO Edit v6.0 - [Untitled *]			_ = x
Backatel Window Rest 2000 FRETo View Tool Distance View Bounding Son Tool Distan	Editor NC Functions Backplot File	Compare Transmission CNC-Calc			Window * Help * _ 🗗 🗙
1 (PACTHG) ∧ 2 60 289.000 ∧ 4 61 22.600 ∧ 5 1 X3.500 ∧ 6 1 X-3.600 ∧ 7 1 Z2.600 ∧ 9 0 X 22.400 > 10 1 X - 3.600 > 12 0 X 22.400 > 13 0 X 22.400 > 16 1 X 23.600 > 16 1 X 23.600 > 17 2 3.000 18 > > 19 × 5 10 × > 10 × > 10 × > 10 × > 11 × > 12 3.00	Backplot Window P Backplot Setup	Reset * * Tool Distance View* Bounding Box*	* * Setup	Solid Setup	Turning *
1 (PACTHG) ∧ 2 60 289.000 ∧ 4 61 22.600 ∧ 5 1 X3.500 ∧ 6 1 X-3.600 ∧ 7 1 Z2.600 ∧ 9 0 X 22.400 > 10 1 X - 3.600 > 12 0 X 22.400 > 13 0 X 22.400 > 16 1 X 23.600 > 16 1 X 23.600 > 17 2 3.000 18 > > 19 × 5 10 × > 10 × > 10 × > 10 × > 11 × > 12 3.00	CNC-Calc V6 Lathe Tutorial 1.cd* The Untitled *				4 Þ ×
	2 50 x62.400 29.000 3 60 28.000 4 61 22.600 5 61 X-3.600 5 61 25.600 7 61 25.600 9 61 20.200 9 61 20.200 10 61 X-3.600 11 61 22.200 12 61 32.200 13 60 x62.400 14 61 20.000 15 61 X-3.600 16 61 22.000 17 61 23.000	Image: Second	Dist: 00000 1950000	Þ	I 10° 10° 10 ⁷

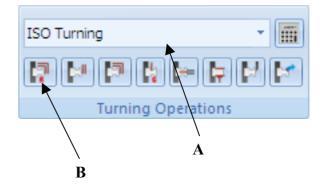
• The Backplot animation is controlled using the slider and command buttons bottom right.

19.4. Roughing the part

In the following tutorial we will generate a Roughing NC program. In order to select the roughing operation the below described steps must be performed.

Select the file type (NC Format) for our facing example program (e.g. ISO Turning) as indicated with the letter **A**.

Then Select the feature **Roughing Turning** by clicking on the *Roughing Turning* icon indicated by the letter **B** in the *Turning Operations* toolbar.



This will open the Roughing Turning dialog to the left of the drawing area. Now insert the values shown in the dialog below.

Roughing Turning
Comment:
ROUGHING
Retract Point Z:
□ 10.0 ° _z
Retract Point X:
70.0 ° _x
Start Diameter:
60.0
Single Step
Show Toolpath Parameters
Back New
Export Editor Export Clipboard

Comment:

This comment will be shown in the final NC-Program. It is always good to include a comment in order to distinguish the various operations in the final program.

Retract Point Z:

This is the Z value to where the operation will retract the tool after completion.

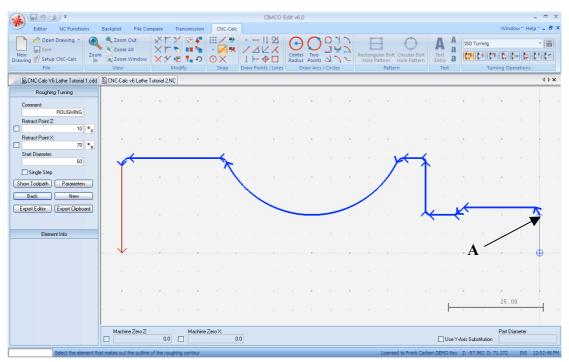
Retract Point X:

This is the X value to where the operation will retract the tool after completion.

Start Diameter:

This is the start diameter of the used stock.

The roughing operation works on a contour, and in order to generate a toolpath we must select that contour. This is done by clicking on the contour as indicated by A in the picture below.



When the selection is made the contour is selected until the end. This is OK for this operation but we really do not want any work done on the leftmost face, so in order to exclude this click the **Back** button once. This will deselect the leftmost face.

Now your drawing should look something like the one above.

Click on the **Parameters** Button to open the parameters dialog. Enter the following values shown below into the parameter dialogs.

Lathe Roughing Parameters	×
Tool Cuts Entry/Exit	
Tool Definition	Work Orientation
Compensation Compensation Type Computer	

Lathe Roughing Parameters \mathbf{X} Tool Cuts Entry/Exit Cuts **Finish Passes** Overlap: 🔽 Use Finish Passes 2.0 Passes: Depth of Cut: 1 5.0 Spacing: 🔽 Use Even Steps 0.2 Retract Retract Distance: Stock to Leave 1.0 Stock to Leave X: 0.1 Stock to Leave Z: 0.1 Annuller ΟK

Tool Tab Configures settings for tool, work orientation and compensation type used for the operation

Tool Orientation: The 9 icons represent the possible 9 orientations of the tool.Tool Radius: The nose radius of the tool.Work Orientation: The four icons control the way we machine the part. In the following we are machining outside from right to left.

Horizontal Plunge: If the tool permits it we could allow horizontal plunge.Plunge Angle: Is the maximum angle we will allow the tool plunge.Compensation Type: This is the compensation type that is used for the

Cuts Tab Configures cutting parameters for the operation.

operation. The two most commonly used

are Controller or Computer.

Overlap: The distance that a cut will overlap the previous cut.

Depth of Cut: The amount of material that is taken in each cut.

Use Even Steps: Indicate what should happen if the total depth is not dividable by the cut depth. You can select whether even steps or the entered amount should be used.

Retract Distance: The distance that the tool retracts from the stock before a return move is made.

Use Finish Passes: Should any finish passes be taken.

Passes: The number of passes to take in the operation.

Spacing: The depth of each of the finish

passes.

Stock to Leave X: Is the amount of material that will be left in the X-direction after the whole operation is performed.

Stock to Leave Z: Is the amount of material that will be left in the Z-direction after the whole operation is performed.

Entry ID strengt	Enter Martin
Entry/Retract Entry Amount:	Entry Vector
1.0	
	Entry Angle
Extension:	45.0
1.0	Entry Length
	1.0
	- Exit Vector
	Use Exit Vector
	Exit Angle
	45.0
	Exit Length
	1.0

Entry/Exit Tab Configure how the tool approaches and leaves the part.

Entry Amount: Is used to extend the toolpath before it starts the actual cut. Extension: Is used to extend the toolpath at the end of the cut.

Use Entry Vector: Enable/Disable the use of entry vector.

Entry Angle: The angle of the entry vector.

Entry Length: The length of the entry vector.

Use Exit Vector: Enable/Disable the use of exit vector.

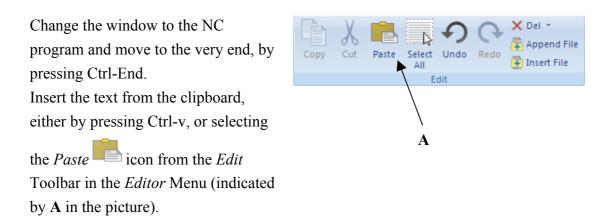
Exit Angle: The angle of the exit vector. **Exit Length:** The length of the exit vector.

Click on **OK** to use the values,

Click on Show Toolpath, and the toolpath will be shown on the drawing.

Try experimenting with the various parameters and see how they change the generated toolpath.

Click on **Export Clipboard** in order to generate the actual program. The program is now in the computer's clipboard and is ready to be inserted into the CNC program.



Now the NC program should look like the following screen.

CIMCO Edit v6.0 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc v6 Lathe Tutor	rial 2.NC *] _ 🖻 🗶
Editor NC Functions Backplot File Compare Transmission CNC-Calc	Window* Help* 🗕 🗗 🗙
Image: Second Line	Find Previous Find Previous Find Previous Find Previous Find Find Find
CNCCalc V6 Lathe Tutorial 1.cd* 3 CNC-Calc v6 Lathe Tutorial 2.NC*	4 Þ x
NCAssistant 5 61 X-3.600 Descriptor. 6 G1 24.600 7 G1 25.600 7 G1 25.600 9 G1 20.200 10 G1 X-3.600 10 G1 X-3.600 12 G1 Z2.200 12 G1 Z2.200 12 G1 Z3.200 12 G1 Z2.200 12 G1 Z3.200 12 G1 Z3.200 12 G1 Z3.200	-
Mody 15 G1 X=3 600 Cycles / Marcos 16 G1 22 000 Tod charge with Constart RM 19 G0 X10.000 210.000 Tod charge with Constart RM 22 G0 Z0.900 22 G0 Z0.900 20 G0 20.900 210.000 210.000 20.000	
111 Data sensity mode concert 27 G0 20.900 121 Zybard dispation 28 G0 X44.734 122 Xybard dispation 29 G1 X43.320 20.193 123 Kybard dispation 29 G1 X43.320 20.193 124 Kybard dispation 29 G1 X43.320 20.193 127 Return form reference point 12 G1 X47.000 128 Return form reference point 12 G1 X47.000 129 Return form reference point 13 G1 Z-10.600 130 Return form reference point 13 G1 Z-110.600 133 G1 Z-110.600 13 G1 Z-51.160 134 G1 X55.160 16 G3 X1rsed cutting contrast Ha	
Select the element that makes out the outline of the roughing contour	Licensed to Frank Carlsen DEMO Key Ln 75/75, Col 1, 1.086 bytes INS 12:47:29 PM

To verify the generated toolpath, we must simulate it using the integrated Graphical Backplot.

To open the backplot window, click on the Backplot menu (indicated with letter A

below) and then on the **Backplot Window** icon (indicated with letter **B**) as shown above.

Now a window like the one below will appear.

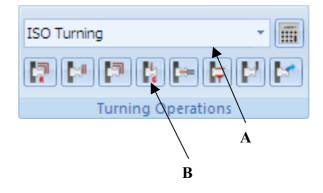
at	📕 🔊 🗃) =	CIMCO Edit ve	5.0 - [C:\CIMCO\CIMCOEdit6\Samples\CN	C-Calc v6 Lathe Tuto	rial 2.NC *]			_ = x
-	Editor NC Functions Backplot File Compare	Transmission	CNC-Calc				Window* H	elp * 🗕 🗗 🗙
Backp Wind	Backplot Setup In Out Window * Rest	Front Lef	View From Measure Rotate Tool Distance View Bounding Box	Toolpath Mode	Tool Tool	Solid Model Com / Regenerate solid Solid Setup	La Carlon	•
	File 😨	View		Toolpath	Tool	Solid	Other	G.
	CNDCalc V6 Lathe Tutorial 1.cd TPCNC-Calc v6 Lathe Tutorial 2.NC							4 ▷ ×
2 G 3 G 5 G 6 G 7 G 8 G 9 G 11 G 12 G 13 G 14 G 13 G 14 G 13 G 14 G 13 G 14 G 20 G 21 G 22 G 24 G 25 G 27 G 28 G 20 G 20 G 20 G 20 G 20 G 20 G 20 G 21 G 20 G 21 G 22 G 22 G 24 G 22 G 22 G 24 G 22 G 24 G 25 G 26 G 27 G 28 G 20 G 20 G 20 G 20 G 20 G 20 G 20 G 21 G 22 G 23 G 24 G 22 G 24 G 22 G 24 G 25 G 27 G 28 G 28 G 28 G 28 G 28 G 28 G 28 G 29 G 28 G 20 G 20 G 20 G 20 G 20 G 20 G 20 G 20 G 21 G 22 G 24 G 25 G 26 G 27 G 28 G 29 G 29 G 20 G 2	FACTNG) 0 XEA 400 29.000 0 XEA 400 29.000 1 22.000 1 22.000 1 24.00 1 24.00 1 25.000 0 XE2.400 1 20.200 1 22.200 1 22.200 1 23.200 0 XE2.400 1 23.000 0 XE2.400 1 23.000 0 XE2.400 1 23.000 0 XE2.400 1 23.000 0 XE2.514 1 X51.160 20.193 1 2-110.600 1 X65.000 1 X65.000 1 X65.000 1 X65.000 1 X65.000 0 X20.933 0 20.900 0 X20.933 1 X43.320 20.193		`A					
30 G 31 G 32 G 33 G	1 x-29.700 1 x47.000 3 x50.600 z-31.500 I-0.000 K-1.800 1 z-110.600	v V	X 62 4000 I: Z 9,0000 K:	Feed:	Dist: ??? Total:	0.0000 1241.2675	 ^⁄u °u∣∥ ∢	₩ ⁷

19.5. Grooving the part

In the following tutorial we will generate a Grooving NC program. In order to select the Grooving operation the below described steps must be performed.

Select the file type (NC Format) for our facing example program (e.g. ISO Turning) as indicated with e letter **A**.

Then Select the feature **Grooving Turning** by clicking on the *Grooving Turning* icon indicated by letter **B** in the *Turning Operations* toolbar.



This will open the Grooving Turning dialog to the left of the drawing area. Now insert the values shown in the dialog below.

Grooving Turning							
Comment:							
GROOVING							
Retract Point Z:							
10.0 °z							
Retract Point X:							
70.0 ° x							
🖌 Single Step							
Show Toolpath Parameters							
Back New							
Export Editor Export Clipboard							

Comment:

This comment will be shown in the final NC program. It is always good to include a comment in order to distinguish the various operations in the final program.

Retract Point Z:

This is the Z value, to where the operation will retract the tool after completion.

Retract Point X:

This is the Z value, to where the operation will retract the tool after completion.

The Grooving operation works on a contour, and in order to generate a toolpath we must select that contour. To select the contour for the operation perform the following steps:

- Ensure that *single step* is checked in the Grooving dialog.
- Select the contour shown on the picture below. To do this, start at the far right by selecting the R2 rounded corner with the indication arrow pointing to the left. Now select the next 2 elements, so the selection looks like the one on the picture below.

			(CIMCO Edit v6.0		_	
1	Backplot File Co	mpare Transmission	CNC-Calc			Window* Help	* - Ø
New Setup CNC-Calc File	Coom Out Zoom All Zoom Window View	× × * * * * * * * * * * * * *	· ✓ ↓ · ✓ ↓	Center Two Radius Points 2	Rectangular Bolt Circular Bolt Hole Pattern Hole Pattern Pattern	Text Entry Text Text	- III 2 [**
CNC-Calc V6 Lathe Tutorial 1.cdd	TGCNC-Calc v6 Lathe	Tutorial 3.NC					4 Þ 3
Grooving Turning Comment: GR00VING		· · ·					
Retract Point Z:							
Retract Point X: 70.0 • x	· · ·	• • •		• • • •	· · · · ·		į
Show Toolpath Parameters Back New							
Export Editor Export Clipboard							
Element Info					L		
							÷
	· · ·						
	Machine Zero Z:	Machine	Zero X: 0.0			Part Diameter	

Now your drawing should look something like the one below.

Click on the **Parameters** Button to open the parameters dialog. Enter the following values shown below into the parameter dialogs.

ts Finish	
Tool Definition	Groove Angle
Tool Width:	Angle:
3.0	90.0
Corner Radius:	
0.2	Cuts
~ Tool Orientation ->	Direction
	Bi-Directional 👻
	Stepover:
	1.5
Depth Settings	
Stock Clearance:	Use Pecking
0.5	Pecking Depth:
Stock Amount	2.0
2.0	
Wall Backoff:	Pecking Retract:
0.1	5.0
0.1	
Stock To Leave	
Stock To Leave Z:	Stock To Leave X:
0.0	0.0

Cuts Tab Configures cutting parameters for the operation.

Tool Width: The Width of the tool. **Corner Radius:** The corner radius of the tool.

Tool Orientation: The two icons indicate how the tool is zeroed.

Stock Clearance: Indicates how far off the part the tool should move before making sideways moves.

Stock Amount: Indicates how much stock there is above the actual groove.Wall Backoff: If possible, the tool will move this far away from the wall before it retracts.

Angle: This is the angle of the groove's center line. An angle of 90 degrees is a vertical angle on the outside, while an

angle of 0 is a horizontal groove from the right.

Direction: This is the direction in which the groove is machined. It can be

Positive, Negative, or Bi-Directional.

Stepover: This is the amount of material removed in each cut.

Use Pecking: Indicates weather pecking is used or not.

Pecking Depth: Defines how deep each peck should be.

Pecking Retract: Determines how far the tool should retract between pecks. **Stock to Leave:** Indicates how much stock should be left after the whole operation is performed.

Lathe Grooving Parameters	
Cuts Finish	
🔽 Use Finish	
Cuts Number of cuts: 1 Cut Depth: 0.1	First Cut Direction CW © CCW
Finish Pass Distances First Distance:	Overlap: 1.0
 Finish Compensation Compensation Type 	
Computer	
	OK Annuller

Finish Tab Configures how the finish passé for the operation should be performed.

Use Finish: Indicates whether or not finish cuts should be performed. Number of Finish Cuts: Describes how many finish cuts should be taken. Cut Depth: This is the amount of material that will be removed with each cut.

First Cut Direction: The finish cut is made from both sides. The First Cut Direction is the direction of the first of the finish cuts.

First Distance: This is how far the first cut will be taken along the contour. **Overlap:** The second finish cut will overlap the first finish cut by this distance.

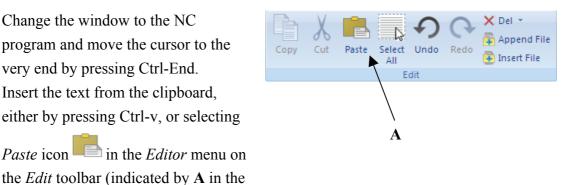
Compensation Type: This is the compensation type that is used for the

operation. The two most commonly used are Controller or Computer.

Click on **OK** to use the values, Click on **Show Toolpath**, and the toolpath will be shown on the drawing.

Try experimenting with the various parameters and see how they change the generated toolpath.

Click on **Export Clipboard** in order to generate the actual program. The program is now in the computer's clipboard and is ready to be inserted into the CNC program.



Now the NC program should look like the following screen.

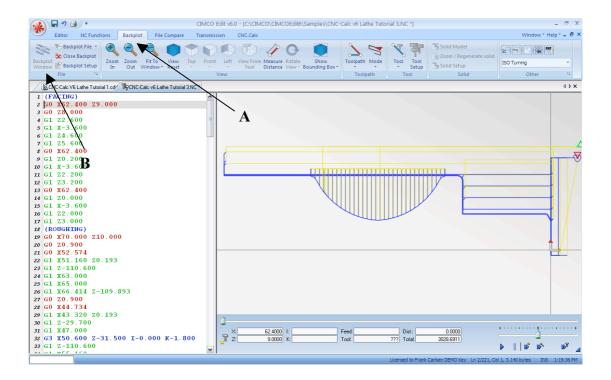
picture).

👧 🗐 📲 🖣 🗐 🔻	CIMCO Edit v6.0 - IC\CIMCO\CIMCOEdit6\Samples\CNC-Calc v6 Lathe Tutorial 3.NC *1	_ = ×
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window * H	Help 🛀 🗗 🗄
	Save Print So Turning Image: Copy Cut Paste Select Undo Redo Print Image: Copy Cut Paste Select Image: Copy Cut Select Image: Copy Cut <td< th=""><th></th></td<>	
CNC-Calc V6 Lathe Tutorial 1.c	od / (%) CNC-Cale v6 Lathe Tutorial 3.NC *	4 Þ 🗙
NCAsistant Description:	73 G0 X70.000 74 G0 Z10.000 75 (GRCOVING) 76 (GRCOVING) 77 G0 X50.000 78 G1 X20.367 80 G2 X55.000 81 G0 X55.000 82 G1 X20.367 83 G1 X20.57 84 G0 X55.000	
Coles / Macros Coles / Macros Pognan Stat and Evid Tod charge with Constant RPM Program comment G01 Lines move G01 April move G01 April move G01 Constant RPM G02 Construints are move G03 ConstructorConstant RPM G03 ConstructorConstant RPM G03 ConstructorConstant G03 Constant G03 ConstructorCon	85 G0 Z-56.305 86 G1 X20.697 87 G1 X20.897 Z-56 405	3
G31 Skip cutting G32 Thread cutting constant le G34 Threading function variabl G36 Automatic tool compensati M Find Add	101 103 21 823 11	

To verify the generated toolpath, we must simulate it using the integrated Graphical Backplot. To open the backplot window, click on the **Backplot** tab (indicated with

letter **A**) and then on the **Backplot Window** icon (indicated with letter **B**) as shown below.

Now a window like the one below will appear.

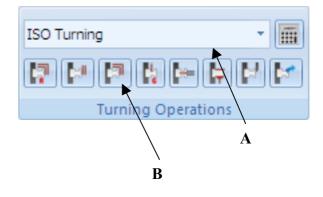


19.6. Finishing the part

In the following tutorial we will generate a Finish NC program. In order to select the Finish operation the below described steps must be performed.

Select the file type (NC Format) for our facing example program (e.g. ISO Turning) as indicated with letter **A**.

Then Select the feature **Finish Turning** by clicking on the *Grooving Turning* icon indicated by letter **B** in the *Turning Operations* toolbar.



This will open the Finish Turning dialog to the left of the drawing area. Now insert the values shown in the dialog below.

Finish Turning
Comment:
FINISH
Single Step
Show Toolpath Parameters
Back New
Export Editor Export Clipboard

Comment:

This comment will be shown in the final NC Program. It is always good to include a comment, in order to distinguish the various operations in the final program.

The Finish operation works on a contour, and in order to generate a toolpath we must select that contour. To select the contour for the operation perform the following steps:

- Ensure that *single step* is unchecked in the Finish dialog
- Select the contour shown on the picture below. To do this, start at the far right by selecting the vertical line with the indication arrow pointing up. Now the whole contour is selected, so unselect the last vertical line by clicking the Back button.

🚯 🖬 🕫 🕈				CIMCO Edit v6.0			_ 5
Editor NC Functions	Backplot File Co	mpare Transmiss					Window * Help * 🗕 🗗
Save	Zoom Out			$- \ \mathfrak{A} \mathfrak{O}$			ISO Turning *
New Drawing Setup CNC-Calc In	Zoom Window	×× 🗉 🖬		- 🔶 🗋 Center T Radius Po	ints 🔍 🤉 🔍 Rectang	jular Bolt Circular Bolt Text Pattern Hole Pattern Entry	
File	View	Modify	Snap Draw	Points / Lines Draw	Arcs / Circles	Pattern Text	Turning Operations
CNC-Calc V6 Lathe Tutorial 1.cdd	TCNC-Calc v6 Lathe	Tutorial 3.NC *					4 6 3
Finish Turning							
Comment: FINISH	•••		• • •				
Single Step							
Show Toolpath Parameters					. .		
Back New							
Export Editor Export Clipboard	~ ~ ~		<u> </u>				
Element Info							K
							` <u>↑</u>
	· · ·						and the second second
					· · · ·		
							Ψ
							25.00
	Machine Zero Z:		hine Zero X:				Part Diameter
		0.0	0.0				r⁄Axis Substitution
Select the next element	t CNC contour or push	Forward or Fast Forwa	rd		Lic	ensed to Frank Carlsen DEMO Key Z:	-40.530 D: 59.978 INS 1:57:56 P

Now your drawing should look something like the one below.

Click on the **Parameters** Button to open the parameters dialog. Enter the following values shown below into the parameter dialogs.

Lathe Finish Parameters		
Tool Definition	Entry Vector Entry Angle Entry Length Exit Vector Exit Angle Exit Length	45.0 3.0 90.0 3.0
Compensation Compensation Type Computer Compensation Side Right		
	Cancel	ОК

Tool Orientation: The 9 icons represent the possible 9 orientations of the tool.
Tool Radius: The nose radius of the tool.
Entry Angle: The angle at which the tool will approach the part.
Entry Length: The length of the approach.
Exit Angle: The angle at which the tool will retract from the part.
Exit Length: The length of the retract.
Compensation Type: This is the approach for the part is used for the

compensation type that is used for the operation. The two most commonly used are *Controller* or *Computer*.

Compensation Side: The side of the contour that the tool will move on. This determines if it is an inside or outside toolpath, since the tool will always move in the direction of the selection.

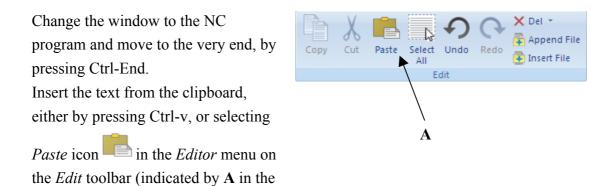
Click on **OK** to use the values.

picture).

Click on Show Toolpath, and the toolpath will be shown on the drawing.

Try experimenting with the various parameters and see how they change the generated toolpath.

Click on **Export Clipboard** in order to generate the actual program. The program is now in the computer's clipboard and is ready to be inserted into the CNC program.



Now the NC program should look like the following screen.

🔬 🖥 🤊 🖨 🔹	CIMCO Edit v6.0 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc v6 Lathe Tutorial 3.NC *] – C	5.5
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window * Help * _	Ð
	So Turning Image: Copy Cut Pasts Set Undo Image: Copy Cut Pasts Image: Copy Cut Pasts </td <td></td>	
CNC-Calc V6 Lathe Tutorial 1.c	d ²¹ 3]CRCGale v6 Lable Tutorial 3NC ¹ 4	Þ×
NC-Assistant	211 G0 Z-35,813	
	212 G1 X49, 982	
Description:	213 G3 X47, 556 Z-37, 584 I-2, 191 K0, 200	
	214 G1 2-37, 584	
	215 G2 X20.068 Z-58.500 I11.022 K-22.216	
	226 G1 Z-58.500	
	217 G2 X40.235 Z-77.187 I24.766 K1.300	
	228 G0 X55.000	
	229 G0 X70.000	
	220 G0 Z10,000	
	221 (FINISH)	
	222 G0 X3. 443 Z2. 121	
Modify		
	223 G1 X-0.800 Z0.000	
Cycles / Macros	224 G1 X19.200	
Program Start and End	225 G3 X19.600 Z-0.054 I0.000 K-0.400	
Tool change with Constant Surf	226 G1 X23.600 Z-1.208	
Tool change with Constant RPM	227 G3 X24.000 Z-1,555 I-0,200 K-0,346	
Program comment _	228 G1 Z-20,400	
G00 Rapid move	229 GJ X23.766 Z-20.683 I-0.400 K0.000	
G01 Linear move		
G02 Clockwise arc move G03 Counterclockwise arc move	230 G1 X20.351 Z-22.390	
G04 Dwell	231 G2 X20.000 Z-22.814 I0.424 K-0.424	
G10 Data setting mode	232 G1 Z-29.400	
G11 Data setting mode cancel	233 G2 X21.200 Z-30.000 I0.600 K0.000	
G18 ZX plane designation	234 G1 X47.200	
G20 Inch data input	235 G3 X50,000 Z-31,400 I0,000 K-1,400	
G21 Metric data input	236 G1 Z-36.213	
G27 Reference point return che G28 Return to reference point	237 G3 X47, 333 Z-38, 363 I-2, 400 K0,000	
G29 Return to reference point	23 62 7-82, 437 110, 933 K-22, 037	
G30 Return to 2nd, 3rd and 4th		
G31 Skip cutting	239 G3 X50.000 Z-84.587 I-1.067 K-2.150	
G32 Thread cutting constant le	240 G1 Z-108.400	
G34 Threading function variabl	241 G3 X45.200 Z-110.800 I-2.400 K0.000	
G36 Automatic tool compensati M	242 G0 X51.200	
Find Add	243	ų.
	1	
	Licensed to Frank Carlsen DEMO Key Ln 243/243, Col 1, 3.649 bytes INS 2:26:	25 PM

To verify the generated toolpath, we must simulate it using the integrated Graphical Backplot. To open the backplot window, click on the **Backplot** tab (indicated with

letter **A**) and then on the **Backplot Window** icon (indicated with letter **B**) as shown below.

Now a window like the one below will appear.

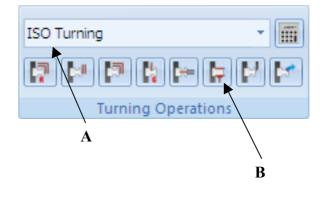
(A) ↓ ? (B) ↓	IMCO Edit v6.0 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-	Calc v6 Lathe Tutorial 3.NC *]		_ 0	= x
Editor NC Functions Backplot File Compare T	ransmission CNC-Calc			Window * Help * -	. 8 ×
Backplot File Close Backplot Window File File File	Front Left View From Measure Rotate Show Tool Distance View Bounding Box *	Toolpath Mode Toolpath Mode Toolpath Tool	Solid Model Coom / Regenerate solid Solid Setup Solid	ISO Turning Other	•
CNCCalc V6 Lathe Tutorial 1.cd+ TrpCNC Calc v6 Lathe Tutorial 3.NC	۷			4	1 Þ ×
1 (FA THG) 2 50 X52.400 Z9.000 3 60 Z8 000 4 61 Z2 000 5 0 X52.400 Z9.000 6 1 Z4.500 Z8 5 0 X52.400 Z8 9 61 Z0.200 Z8 10 1 X-3.60 Z8 12 12.2.00 Z9 Z9 12 61 Z-3.60 Z9 12 12.2.200 Z9 Z9 Z9 12 12.3.600 Z9 Z9 Z9 12 12.3.600 Z9 Z9 Z9 12 12.3.000 Z9 Z9 Z9 Z9 12 12.3.000 Z9 Z9	A				*
30 G1 2-29.700 31 G1 X47.000 32 G3 X50.600 Z-31.500 I-0.000 K-1.800 33 G1 Z-110.600		Feed: Dist: Tool 2??? Total	0.0000 3218.9497 Carlsen DEMO Key Ln 2/243, Co	⁰ ⁰ ¹	7 4

19.7. Threading the part

In the following tutorial we will generate a Threading NC program. In order to select the Threading Horizontal operation the below described steps must be performed.

Select the file type (NC Format) for our facing example program (e.g. ISO Turning) as indicated with letter **A**.

Then Select the feature **Threading Horizontal** by clicking on the *Grooving Turning* icon indicated by letter **B** in the *Turning Operations* toolbar.



This will open the Threading Z Turning dialog to the left of the drawing area. Now insert the values shown in the dialog below.

Threading Z Turning
Comment:
THREADING
Start Z:
0 z •
End Z:
-20 😋
Show Toolpath Parameters
Back New
Export Editor Export Clipboard

Comment:

This comment will be shown in the final NC-Program. It is always good to include a comment in order to distinguish the various operations in the final program.

Start Z:

This is the Z value where the operation starts.

End X:

This is the Z value where the operation ends.

Click on the Parameters button to open the parameters dialog.

Enter the following values shown below into the parameter dialogs.

the Threading Paran	neters
hread Form Cutting Ta	aper
Threading Type	ead Lead: 3.0 nber of Starts:
	1
C Thread Form Select From Tabl	e
Included Angle:	60.0
Thread Angle:	
	30.0
Major Diameter:	24.0
Minor Diameter:	
	20.752
	OK Annuller

Thread Form Tab Configures the form of the thread in the operation.

Threading Type: The 4 icons represent threading outside or inside made from left to right, or right to left. Thread Lead: Defines the starting position of the thread. Number of Starts: This defines how many starts the thread will have; the normal number is one. Select From Table: Instead of typing all the various values for the thread, these can be inserted from a table. This table contains all of the most common threads for both Imperial and Metric. Please see the end of this section for further explanations on the use of the table. Included angle: The total angle of the

thread profile.

Thread Angle: The forward angle of the thread profile measured from vertical. **Major Diameter:** The largest measure of the thread.

Minor Diameter: The smallest measure of the thread diameter.

nread Form Cutting Taper	
Type of Cut	Cutting Parameters
💿 Constant Area	Stock Clearance:
🔿 Constant Depth	5.0
·	Stock to Leave:
Depth calculation	0.0
🔘 First Cut Depth	
1.0	Pulloff Distance
Number Of Cuts:	Absolute
3	
	6.0
umber Of Spring Cuts:	_ Infeed
1	Angle:
NC Export Type	30.0
Longhand (G32)	

Cutting Tab Configure the number of and the type of cuts that should be used in the operation

Constant Area: With constant area the tool will remove equal amounts of the area per cut.

Constant Depth: Using the constant depth option each cut will have the same depth. Since the removed area is triangular an increasing amount will be removed the deeper the tool cuts. **First Cut Depth:** If this option is selected the first cut defines how the following cuts will be made based on what method (Constant Area/Depth) is used.

Number of Cuts: If this option is used the operation will be performed with this number of cuts (+ the selected number of spring cuts).

Number of Spring Cuts: If spring cuts are used this many cuts will be made at the final depth.

Stock Clearance: Defines how far away from the stock the tool should move before it moves back to the start.Stock to Leave: Defines how much stock

should be left at the end of the operation. **Pulloff Distance:**

Infeed Angle: The angle at which the tool will move down. The reason for this is to minimize the chip pressure at the front of the tool and thereby obtain a more even thread.

Taper Type	Acceleration Distance
0.0 Overcut Absolute Revolutions	Revolutions
3.0	

Taper Tab Configures extension of the cuts and a possible taper for the operation.

Taper Type: If the taper angle is not zero, a conical thread will be produced. The two icons represent the two ways the cone can go.

Taper Angle: Is the angle of the conical thread.

Absolute Overcut: With this option the tool will continue the defined distance at the end of the thread.

Revolutions Overcut: With this option the thread will be extended by the number of revolutions entered.

Acceleration Distance: The Distance that the tool will start before reaching the thread in order to accelerate to achieve a more uniform thread.

Calculation Acceleration Distance: With this option the acceleration distance will be calculated by the operation.

Absolute Acceleration Distance: Here, the operator can enter how far away the tool should start before reaching the thread.

Revolutions Acceleration Distance: With this option the tool will use the given number of revolutions to accelerate. It will therefore start the number of revolutions multiplied by the thread pitch before reaching the thread.

Click on **OK** to use the values.

After the values have been entered in the dialog the screen will look something like the picture below. Notice that the area of operation is shown with a blue rectangle.

🔬 🗐 " 🗇 🕘 🔹										CI	MCO E	dit v6.0													-	
Editor NC Functions	Back	plot	File Co	ompare	Tran	nsmissio	n C	NC-Calc																Windov	r* Help	* = <i>0</i>
📄 🚔 Open Drawing 👻 💽	< · · ·	Zoom C		XI	Y	24		<u> </u>	٠.		X	\bigcirc	\bigcirc	01	2	F	=	(3	Δ	A	ISO T	urning			•
New Setup CNC-Calc In	om 📒	Zoom A Zoom V						<mark>/</mark> **	1			Center Radius	Two			Rectan	gular Bi	olt Circu Hole	Iar Bolt	Tex	a					
File		View			Modif	ten v J	U .	nap	Draw	Points				/ Circles		Hole		ttern	Pattern	T	y 🦗 ext		Tur	ning Opera	tions	
CNC-Calc V6 Lathe Tutorial 1.cdd	1 Դրգ	NC-Calc	v6 Lathe	Tutorial 3	NC *																					4 Þ 3
Threading Z Turning																									-	
Comment																										
THREADING Start Z:																										
5tatt 2.																										
End Z:																										
Show Toolpath Parameters																										
Back New	1																÷.,		- j							
Export Editor Export Clipboard									$\langle \cdot \rangle$																	
Export Editor Export Clipboard																/										
	1										<				1						1					
Element Info	· ·											-							Ļ		4.					-
											,														€	
	· ·												-		-						-					
	1																							30.00		
																				H						-1
			-							_										_						_
		Machine	e∠ero Z:		0 🗖		ne Zero >		1.0												Use Y	Axis Sub	stitution	Part Diam	eter	

Try experimenting with the various parameters and see how they change the generated toolpath.

Click on Export Clipboard in order to generate the actual program.

The program is now in the computer's clipboard and is ready to be inserted into the CNC program.

Change the window to the NC program and move to the very end by pressing Ctrl-End. Insert the text from the clipboard, either by pressing Ctrl-v, or selecting Paste icon in the Editor menu on

the *Edit* toolbar (indicated by **A** in the picture).

A in the

Now the NC program should look like the following screen.

Editor NC Functions Solution Solution <td< th=""><th>🕕 🗐 🗐 🗧 🕫</th><th></th><th>_ = x</th></td<>	🕕 🗐 🗐 🗧 🕫		_ = x
New Open City Is So Turney Itel Itel Syste So Turney Itel Syste Itel Tool (hange Itel Syste <thit< th=""><th>1</th><th></th><th></th></thit<>	1		
New Open Cose Save Save Pint Copy Cut Parte Save Undo Redo Pinterfile Find Cate Cose Control Cost Cost Cost Cost Cost Cost Cost Cost			
New Open Cose Save Save Pint Copy Cut Parte Save Undo Redo Pinterfile Find Cate Cose Control Cost Cost Cost Cost Cost Cost Cost Cost	- C) 🗁 🧡 🛄 🖻	🔜 🔜 ISO Turning 🔹 📄 🚶 🛃 🛒 🏠 🖓 🖓 X Del * 🔍 🔍 Rind Previous 🔛 👔 Prexit Tool change	
All Org Project Number Edd First C First		and an	
File C File Type East Find INCCack V6 Labe Tuoted 1 cold CNCCack v6 Labe Tuoted 31C 4 Px INCCacing Cold C2 0.351 7-0.400 K0.000 220 C1 C2 0.351 7-2.2.390 221 C2 2.2.390 221 C1 C2 2.2.391 221 C1 C2 0.2.0 C2			
NC-Assistent 229 G3 X23.766 2-20.683 I-0.400 K0.000 Decipitor 20 G1 X20.351 Z-22.390 Z <td></td> <td></td> <td></td>			
Nuckasida 230 G1 2:20:3 2:30:00 7:22:3:30 Decription 233 G2 X:20:00 X:22:3:30 X:20:3:30 X:20:3:3:30 X:20:3:3:30 X:20:3:3:3:30 X:20:3:3:3:3:3:3:3:3:3:3:3:3:3:3:3:3:3:3:	CNC-Calc V6 Lathe Tutorial 1.cr	ve/ 3) CNC-Calc v6 Lathe Tutorial 3.NC *	4 ⊳ ×
Decorption: 221 (22 X20.000 7-22.814 IO.424 K-0.424 222 (21 Z-29.400 223 (21 Z-29.400 223 (21 Z-29.400 223 (21 Z-29.400 224 (21 X47.200 233 (21 X-30.000 K-0.000 226 (21 Z-36.213 237 (23 X17.200 237 (23 Z2.23.33 (2-33.363 I-2.400 K0.000 238 (2 Z-82.437 IIO.933 K-22.037 238 (23 Z-82.437 IIO.933 K-22.037 239 (3 X50.000 Z-81.587 I-1.067 K-2.150 246 (3 Z-108.400 246 (3 Z-108.400 248 (3 X45.200 Z-110.800 I-2.400 K0.000 248 (3 X45.200 Z-110.800 I-2.400 K0.000 248 (3 X45.200 Z-110.800 I-2.400 K0.000 248 (3 X45.200 Z-110.800 I-2.400 K0.000 248 (3 Z-2.3.000 F3.000 IF3.000 248 (3 Z-2.3.000 F3.000 249 (3 X24.000 Z6.396 246 (3 Z 2-23.000 F3.000 241 (3 X45.200 Z2.125 290 (3 X21.348 246 (3 Z 2-23.000 F3.000 224 (3 Z 2-23.000 F3.000 247 (3 X34.000 240 (2 Z 3.1.348 240 (0 X21.348 240 (3 Z 2.2.3.000 F3.000 251 hohd angrad 250 (3 Z 2 -23.000 F3.000 252 (3 C Z 2.2.3.000 F3.000 252 (3 Z 2-23.000 F3.000 251 hohd angrad 250 (3 Z 2 -23.000 F3.000 252 (0 X X4.000 250 (0 Z 2.000 F3.000 253 hohd angrad 250 (0 Z 2.0.000 <td>NC-Assistant</td> <td>229 G3 X23.766 Z-20.683 I-0.400 K0.000</td> <td>^</td>	NC-Assistant	229 G3 X23.766 Z-20.683 I-0.400 K0.000	^
Image: Constraint State and End Tool St	Description	230 G1 X20.351 Z-22.390	
Logical field 23 62 x17.200 23 62 x17.200 23 62 x17.200 23 63 x17.200 23 63 x17.200 236 61 x2-36.213 12 -36.313 1-2.400 K0.000 236 62 x2-36.213 10.933 K-22.037 23 63 x50.000 2-84.587 1-1.067 K-2.150 240 61 x10.00 26.3 x51.200 2-110.800 1-2.400 K0.000 242 60 x51.200 2-110.800 1-2.400 K0.000 24 60 x1.200 24 60 x1.200 24 60 x1.200 24 60 x1.200 24 60 x2.125 60 26 63 x2.200 10.00 24 60 x2.125 63 25 63 24 60 x2.1.205 24 60 x2.1.205 25 63 x2.2000 73.000 24 60 x2.1.205 25 60 x2.1.205 25 60 x2.1.205 25 60	Description	231 G2 X20.000 Z-22.814 IO.424 K-0.424	
Mody 239 61 X50.000 7-31.400 10.000 K-1.400 226 63 X50.000 7-31.400 10.000 K-1.400 226 63 X50.000 7-31.400 10.000 K-1.400 226 61 2-36.213 27 63 X50.000 2-31.601 K-1.400 226 63 X50.000 2-36.213 22 C3 X50.000 2-36.213 227 63 X50.000 2-84.587 I-1.067 K-2.150 24 G1 C-106.400 226 G3 X40.000 Z6.396 240 G1 Z-10.800 I-2.400 K0.000 226 G3 X40.000 Z6.396 240 G3 Z2.2125 Z4 G0 X21.128 Z4 G0 X34.000 Z4 G0 X34.000 Z2 Z2 Z2.3000 F3.000 Z3.000 Z3.000 F3.000 Z3 G0 X21.348 Z4 G2 Z2 Z2.20.000 Z3.000 F3.000 Z3 G0 X21.000 Z3 G0 X21.000		232 G1 Z-29.400	
Logical Constant Start, Start, Start Start, Start Start, Start Start, Start Start, Start Start, Start Start, St		233 G2 X21.200 Z-30.000 I0.600 K0.000	
226 61 2-36.213 227 63 <x47.333< td=""> 2-38.363 1-2.400 229 62 2-82.437 110.933 K-22.037 229 63 X50.000 2-84.587 1-1.067 K-2.150 240 61 2-10.800 1-2.400 K0.000 242 60 X51.200 2-110.800 1-2.400 K0.000 243 60 X51.200 2-110.800 1-2.400 K0.000 244 60 X51.200 2-110.800 1-2.400 K0.000 245 60 X21.125 60 2-2.125 60 2-2.23.000 63 2-2.23.000 53.000 2-2.23.000 2-2.23.000 2-2.23.000 2-2.23.000 2-2.20.000 2-2.20.000 2-2.20.000 2-2.20.000 2-2.20.000 2-2.20.000 2-2.20.000 2-2.20.000 2-2.20</x47.333<>		234 G1 X47.200	
Logics / Marcias 227 63 X47 333 X-22.037 Logics / Marcias 228 62 Z-82.437 110.933 K-22.037 Logics / Marcias 229 63 X50.000 Z-84.587 L1.067 K-2.150 Logics / Marcias 240 61 Z-108.400 241 63 X50.000 Logic Angewing Contrart Sut., Tod Charge with Contrat Sut., Tod Charge with Contra		235 G3 X50.000 Z-31.400 I0.000 K-1.400	
Mody 229 62 2-2 2.2 63 X50.000 2-84.587 1-1.067 K-22.037 240 61 2-108.400 240 61 2-108.400 240 61 2-108.400 240 63 X50.000 2-84.587 1-1.067 K-2.150 240 61 2-108.400 241 63 X51.200 240 K51.200 700 charge with Contart Stut, To dcharge with Contart Stut, Stut and Education as a converse Stut and Stut and Education as a converse Stut and Stut and Stut, 2000 F3.000 F3.000 247 60 X21.348 240 632 2-23.000 F3.000 250 F3.000 F3.000 F3.000 221 632 2-23.000 F3.000 251 F3.20 C.752 225 60 X20.00 F3.000 252 F3.000 F3.000 256 632 2-23.000 F3.000 252 F3.22 C32 C32		236 G1 Z-36.213	
Mody 229 63 X50.000 2-84.587 I-1.067 K-2.150 Velocity 61 2-108.400 24 61 Z-108.400 Velocity 43 X45.200 Z-110.800 I-2.400 K0.000 Velocity 63 X45.200 Z-110.800 I-2.400 K0.000 Velocity Contract Status Velocity K1.000 K1.000 K1.000 Velocity Contract Status Velocity K1.000		237 G3 X47,333 Z-38,363 I-2,400 K0.000	
Mody 240 61 2-108:400 241 63 X45.200 2-10.800 1-2.400 K0.000 Cycles / Macco 242 60 X45.200 2-110.800 1-2.400 K0.000 Progen Star adEnd 70 60 X25.1200 244 60 X3.400 26:396 To charge wth Contart Stuff. 744 60 X3.4000 26:396 246 632 2:23.000 F3.000 G01 Fage move 60 X24 60 X3.4000 26:396 247 60 X3.4000 26:396 G10 Lassetting mode G10 Lassetting mode G10 Lassetting mode G10 Z3.1.348 246 G2 Z : 1.25 20:32 G0 22:60 G3 Z : 2-23.000 F3.000 22:60		238 G2 Z-82 437 I10.933 K-22.037	
Mody 240 61 2-108:400 241 63 X45.200 2-10.800 1-2.400 K0.000 Cycles / Macco 242 60 X45.200 2-110.800 1-2.400 K0.000 Progen Star adEnd 70 60 X25.1200 244 60 X3.400 26:396 To charge wth Contart Stuff. 744 60 X3.4000 26:396 246 632 2:23.000 F3.000 G01 Fage move 60 X24 60 X3.4000 26:396 247 60 X3.4000 26:396 G10 Lassetting mode G10 Lassetting mode G10 Lassetting mode G10 Z3.1.348 246 G2 Z : 1.25 20:32 G0 22:60 G3 Z : 2-23.000 F3.000 22:60		239 G3 X50.000 Z-84.587 T-1.067 K-2.150	
Modify 242 63 X15.200 2-110.800 T-2.400 K0.000 Cycles / Macros 242 63 X15.200 2-110.800 T-2.400 K0.000 Tod charge wh/ Contart StM 242 60 X51.200 244 60 X51.200 Tod charge wh/ Contart StM 245 60 X22.125 246 63 22.125 Flogtan corment G00 Fad move 246 60 X21.125 246 60 X21.125 G02 Clockwise acr more 246 60 X21.125 246 60 X21.125 G02 Clockwise acr more 246 60 X21.125 246 60 X21.125 G02 Clockwise acr more 246 60 X21.125 246 60 X21.125 G02 Clockwise acr more 246 60 X21.125 246 60 X21.125 G02 Clockwise acr more 246 60 X21.000 F3.000 251 60 X21.000 252 60 X21.000 252			
Opder / Macros 242 60 X51.200 Progen Stat and End Tool change with Contart Stat Tool change with Contart Stat Tool change with Contart Stat Tool change with Contart Stat Tool change with Contart Stat Stat. and End COL black with mode G10 Lab setting mode G10 Lab setting mode G10 Lab setting mode G10 Lab setting mode G11 Dab setting mode G11 Dab setting mode G12 Black with a profession G23 Return to reference point G23 Return to reference point G24 Return to reference point G25	Modify		
Update / Hadual 243 (FHEE AD DING) Program Sitt and Rod To d charge with Contart SEM, Program comment 244 (G) X34.000 25.396 Cold Anage with Contart SEM, Program comment 246 G3 22.2.23.000 F3.000 G0 Anad move G0 Contart for the setting mode G10 Data setting mode G20 Inch data input G22 Helici data input G22 Helici data input G23 Helici data input G24 Helici data input G25 Helici data input	Contra di Manaza		
Program Stati and Lind 24 GO X334_000_Z6.336 Tod Charge with Contart Stat., 245 GO X22.125 GO Randmow Rove 632 GO X22.125 GOT Randmow Rove 246 GO X22.125 GOT Randmow Rove 246 GO X22.125 GOT Randmow Rove 247 GO X34.000 GOT Load setting mode 247 GO X34.000 GIT Data setting mode 251 GO X21.1348 GOT Data setting mode 251 GO X20.752 GIT Data setting mode 251 GO X20.752 GZP Return tom reference point return Char 256 GO X34.000 GZP Return tom reference point return Char 256 GO X20.752 GZP Return tom reference point return Char 256 GO X20.752 GZP Return tom reference point return Char 256 GO X21.400 GZP Return tom reference point return Char 256 GO X21.000 GZP Return tom reference point return Char 256 GO X21.000 GZP Return tom reference point return Char 256 GO X21.000 GZP Return tom reference point return Char 256 GO X21.000 GZP Return tom reference point C		242 (THEFADINC)	
Ted charge with Constant RFM 245 GO X22.125 Program commed 246 GO X22.2.23.000 F3.000 GOI Andrew Constant RFM 246 GO X22.125 GOI Andrew Constant RFM 246 GO X22.125 GOI Linear move 246 GO Z2.2.23.000 F3.000 GOI Constructed Constant RFM 246 GO Z2.125 GOI Linear move 247 GO X34.000 GOI Constructed Constant RFM 250 GO Z2.1.72 GOI Constructed Constant RFM 250 GO Z2.1.348 GOI Data setting mode cancel 251 GO X34.000 GID Lata setting mode cancel 252 GO Z6.000 GZD Inch data riput 253 GO X30.752 GZD Heiric data riput 253 GO Z6.000 GZB Heain Intoin Eleisence point 256 GO Z6.000 GZB Heain Intoin Eleisence point 256 GO Z6.000 GZB Heain Intoin Constant Hum Chair 256 GO Z6.000 GZB Heain Intoin Eleisence point 256 GO Z6.000 GZB Heain Intoin Eleisence point 257 GO X20.752 GZB Heain Intoin Eleisence point 258 GO Z6.000 GZB Heain Intoin Eleisence point 259 GO X34.000 GZB Heain Intoin Eleisence point 259 GO X34.000 GZB Heain Intoin Eleisence point 259 GO X34.000			
Program comment G00 Rapid move G01 Lines move G01 Lines move G01 Clockwise acr move G02 Lines move G03 Clockwise acr move G03 Clockwise a			
G00 Rapid move 247 G0 X34,000 G01 Lines move 246 G0 Z6.172 G02 Colume ac move 246 G0 Z6.172 G04 Dead 240 G32 Z-23.000 F3.000 G11 Data setting mode 221 G0 X34,000 G12 Data setting mode 222 G0 Z5.000 G12 Data setting mode 223 G0 Z2.0.00 G12 Data setting mode 225 G0 Z3.000 F3.000 G22 Ment data input 225 G0 X3.4.000 G23 Return to indeference point 226 G0 Z2.0.752 G23 Return to indeference point 226 G0 X2.0.752 G23 Return to indeference point 226 G0 Z2.0.000 G23 Return to indeference point 226 G0 Z2.0.000 F3.000 G23 Return to indeference point 226 G0 Z2.0.000 F3.000 G23 Return to indeference point 226 G0 Z2.0.000 F3.000 G24 Tread cuting constant L	Program comment		
102 Charlen monore 246 60 76.172 603 Charlen control is a monore 246 60 72.172 603 Charlen control is a monore 249 60 72.172 604 Duelation monore 249 60 72.172 605 Charlen monore 249 60 72.172 610 Data setting mode 632 632 632 22.22.3000 F3.000 611 Data setting mode 252 60 72.6 60 72.1 632 60 72.1 632 60 72.1 60 72.1 60 72.1 60 72.1 7			
G03 Counterclockview acrower 249 G0 X21 348 G04 Oved 250 G32 Z-23.000 F3.000 G10 Dastering mode 251 G0 X24 G0 X20 G10 Zastering mode 252 G0 X20.752 S2 G0 X20.752 G27 Reterone point Future Ches 256 G0 Z6.000 F3.000 S2 G0 X20.752 G27 Reterone point Future Ches 256 G0 Z6.000 S2 G0 G0 Z6.000 S2			
Gid Queil Gid Queil Gid Queil Gid Queil Gid dairpud Zi Gid Xi 4.000 Eis Gid Xi 4.000 Eis Gid Xi 4.000 Gid Alerence point Zi Gid Xi 4.000 Eis Gid Xi 4.000 Eis Gid Xi 4.000 Gid Pheuni Condition Queil Zi Gid Xi 4.000 Eis Gid Xi 4.000 Eis Gid Xi 4.000 Gid Pheuni Condition Queil Zi Gid Xi 4.000 Eis Gid Xi 4.000 Eis Gid Xi 4.000 Gid Pheuni Condition Condition Zi Gid Zi Zi Zi Zi 2.23.000 Fill 0.000 Eis Gid Zi Zi Zi Zi 2.23.000 Fill 0.000 Eis Gid Zi Zi Zi Zi 2.23.000 Fill 0.000 Gid Hindright function vanide Zi Gid Cid Ci Condot Eis Gid Cid Cid Cid Cid Cid Cid Cid Cid Cid C			
a 10 Jas setting mode cancel a 11 Jas setting mode cancel 11 Jas setting mode cancel 12 Jas Jas Jas mode cancel 12 Jas			
01827/glane deignation 292 60 26.000 0230 her deignation 293 60 226.000 0231 Gel ante deignation 294 632 7.52 0231 Gel ante deignation 294 632 7.52 0231 Gel ante briefence point 295 60 254.000 0231 Gel ante briefence point 295 60 256.000 0231 Head outing constant 295 60 27.60 0.000 237 Head outing constant 295 60 27.60 0.000 237 Head outing constant 295 60 27.60 0.000 236 1000 F3.000 297 60 22.0.752 296 60 22.72 2.30.000 F3.000 22.75 297 60 32.000 F3.000 22.97 60 22.000 22.000 286 60 7.6.000 260 60 7.6.000 260 26.000 260 26.000 260 26.000 260 26.000 260 26.000 260 27.6.000 260 26.000 260 27.6.000 260			
G20 Inch data input G27 Relatence point return Cas 253 GO X20.752 G28 Return to reference point G28 Return to reference point G29 Return to reference point G20 Return to reference point G2			
G21 Metic data input 254 G32 2-23.000 F3.000 G27 Metic data input 254 G32 2-23.000 F3.000 G28 Metun to inference point 255 G0 X34.000 G37 Heterne point 255 G0 X34.000 G37 Heterne point 255 G0 X34.000 G31 Skp culing 257 G0 X20.752 G32 Thread culing constant. 258 G0 X34.000 G34 Heterne point 259 G0 X34.000 G37 Heterne point 257 G0 X20.752 G37 Heterne point 259 G0 X34.000 G37 Heterne point 259 G0 X34.000 G37 Heterne point 257 G0 X20.752 G38 Heterne point 260 G0 Z6.000 G37 Heterne point 260 G0 Z6.000 G38 Heterne point 260 G0 Z6.000 G38 Heterne point 260 G0 Z6.000 G38 Heterne point 260 G0 Z6.000 G40 G0 Z6.000 261			
235 Got X34.000 236 Got X34.000 237 Got X26.000 238 Got X20.752 231 Station one delay 234 Tread culting contrant to G35 Automatic tool concentration Find Add Concentration 261			
G32 Return ton references point 256 G0 26.000 G30 Return to 30 and 4th 256 G0 26.000 G31 Skip cutting constant e 258 G3 2 2 -23.000 F3.000 G34 Return do in commental 259 G0 X34.000 G34 Return do in commental 259 G0 X34.000 Find Add	G27 Reference point return che		
630 Flexu to 2nd, 3d and th 227 G0 X20.752 631 Ship culing 227 G0 X20.752 632 Flexad culing constant 228 G32 Z - 2.3.000 F3.000 F3.000 636 Automatic tool comentati 260 G0 Z6.000 F3.000 Find Add 261 C1 C1 C1			
G31 Skip culting E207 Gio X.20.152 G32 Thread culting constant (a E207 Gio X.20.152 G34 Threading Innotion variable E36 Gio X.20.232 G35 Automatic tool comentation E36 Gio X.20.232 Find Add			
G32 Thready Luting constant. 22% G32 Z -23.000 F3.000 G32 Thready Luting Constant. 22% G0 X24.000 G38 Automatic tool concensati. 260 G0 Z 6.000 Find Add 262 262			
G36 Automatic tool concensal	G32 Thread cutting constant le		
Find Add 262			Ξ
	G 36 Automatic tool compensati	260 G0 26.000	
Licensed to Frank Carlsen DEMO Key Ln 261/261, Col 1, 3.887 bytes INS 2:3815FM	Find Add		~
		Licensed to Frank Carlsen DEMO Key Ln 261/261, Col 1, 3.387 bytes INS	2:38:15 PM

To verify the generated toolpath, we must simulate it using the integrated Graphical Backplot. To open the backplot window, click on the **Backplot** tab (indicated with

letter **A**) and then on the **Backplot Window** icon (indicated with letter **B**) as shown below.

Now a window like the one below will appear.

🔬 🖬 🤊 🗃 🔻	CIMCO Edit v6.0 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc v6 Lathe Tutorial 3.NC *]
Editor NC Functions Backplot File Compare	Transmission CNC-Calc Window* Help* = 🗗
Backplot File Zoom Zoom Fit To View To	P Front Left View From Measure Rotate Show Toolpath Mode Tool Tool Tool
Window Backplot Setup In Out Window Neset	Tool Distance View* Bounding Box* * Setup Solid Setup
CCCalc V6 Lathe Tutorial 1.cd+ TeCNC-Calc v6 Lathe Tutorial 3.N	40×
229 G3 223.766 2-20.683 T-0.400 K0.000 230 G1 X0.51 7-22 390 231 G2 X0.000 Z-22.814 I0.424 K-0.424 232 G1 Z-20.400 234 G2 X1.000 Z-30.000 I0.600 K0.000 234 G2 X21.000 Z-30.000 I0.600 K0.000 234 G1 X47.400 235 G3 X50.000 Z-31.400 I0.000 K-1.400 236 G1 Z-36.213 237 G3 X47.333 Z-38.363 T-2.400 K0.000 238 G2 Z-82 J10.933 K-22.037 239 G3 X50.000 Z-84.587 T-1.067 K-2.150 240 G1 Z-108.400 242 G0 X51.200 243 (THREADING) 244 G0 X34.000 Z-110.800 T-2.400 K0.000 245 G3 Z-23.000 F3.000 247 G0 X34.000 248 G2 Z-82 J2 Z-30 249 G0 X21.348 250 G32 Z-23.000 F3.000 251 G0 X34.000 252 G0 Z52 C-23.000 F3.000 255 G0 X34.000 255	
227 G0 X20.752 238 G32 - 23.000 F3.000 229 G0 X34.000 260 G0 26.000 261	Image: Signal for the second secon
	Licensed to Frank Carlsen DEMO Key Ln 244/261, Col 16, 3.887 bytes INS 2:40:52 PM

When entering the form parameters for the thread it is possible to use *Select From Table* on the Thread Form tab.

When this button is clicked the dialog below is shown.

Se	elect Threa	d			
	Export Units Metric			◯ Inches	
	Metric				~
	Major Dia.	Lead	Minor Dia.	Description	~
	24.0000	3.00	20.7520	Coarse	
	24.0000	2.00		Fine	
	25.0000	1.50		Fine	
	27.0000	3.00	23.7520	Coarse	
	27.0000 30.0000	2.00 3.50	24.8350 26.2110	Fine Coarse	
	30.0000	2.00		Fine	
	30.0000		28.3760	Fine	
	33,0000	2.00	30 8350	Fine	×
				ОК	Cancel

The selection is then performed in the following way:

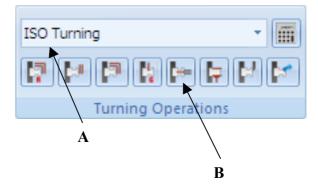
- First select if the thread is a metric or imperial thread.
- Based on the above selection the different types of threads can be selected in the drop down box.
- It is now possible to select the specific thread in the list box, and when OK is clicked the corresponding data will be copied to the form fields in the parameter window.

19.8. Drilling the part

In the following tutorial we will generate a Drilling NC program. In order to select the Drilling operation the below described steps must be performed.

Select the file type (NC Format) for our facing example program (e.g. ISO Turning) as indicated with letter **A**.

Then Select the feature **Drilling** by clicking on the *Drilling* icon indicated by letter **B** in the Turning Operations toolbar.



This will open the Drilling dialog to the left of the drawing area. Now insert the values shown in the dialog below.

End Drilling	
Comment:	
DRILLING	
Clearance Height:	
10	Γz
Retract Height:	
3	۲x
Start Depth:	
2	z,
End Depth:	
-20	×,
Parameters	
New	
Export Editor Export Clipboa	ard

Comment:

This comment will be shown in the final NC Program. It is always good to include a comment in order to distinguish the various operations in the final program.

Clearance Height:

The position to where the tool is moved before the actual operation, and where it will end after the drilling is finished.

Retract Height: The distance from the start of the operation where the feedrate will switch from rapid to feed.

Start Depth: The depth at which the actual operation is started.

End Depth: The final depth of the operation.

The Drilling operation is defined by the above parameters, after the entry of which the screen will look something like the one shown below.

The four distances entered are shown as crosses on the drawing.

■ 9 3) =						
(*)	Backplot File Co	mpare Transmission		CIMCO Edit v6.0		Window * Help * - 🗗
New Setup CNC-Calc File	Coom Out	メ 大 大 大 で い は い の い し		Center Two Radius Points 2	Rectangular Bolt Circular Bolt Hole Pattern Pattern	ISO Turning *
CNC-Calc V6 Lathe Tutorial 1.cdd	TeCNC-Calc v6 Lathe	Tutorial 3.NC *				4 Þ
End Drilling			1		and the second	
Comment DRILLING		· · · ·				
Clearance Height 10 Tz						
Start Depth:						• • • •
End Depth: .20						
Parameters				/		
Export Editor Export Clipboard						
Element Info						
						25.00
	Machine Zero Z:		e Zero X:			Part Diameter
		0.0	0.0		Use Y	Axis Substitution
None					Licensed to Frank Carlsen DEMO Key Z: -3	14.346 D: -20.750 INS 3:29:06

Click on the **Parameters** button to open the parameter dialog. Enter the following values shown below into the parameter dialog.

rilling Parameters			
Operation Type			
Drilling			*
Feedrate:		Dwell:	
	200.0		0.0
Pecking			
✓ Use Pecking Peck Clearance:		Peck Betract:	
reck Lieafance:	3.0	Teck fieldct.	5.0
First Peck:		Subsequent Pecks:	
	10.0		6.0
Tip Compensation	ion		
Tip Angle		Drill Diameter	
	30.0		10.0
		Tip Compensation	
			18.6603
		Cancel	ΟΚ
			UN

Operation Type: The operation type can be drilling or threading that is either Clock or Counter Clock Wise. **Feedrate:** The feedrate used for all feed moves.

Dwell: The time that the drill will dwell at the bottom of each cut in order to break the chip.

Use Pecking: By selecting this option the operation will be performed with pecking movements.

Peck Clearance: The distance above the previous cut to which the drill will move in rapid after the retraction is performed. **Peck Retract:** The distance the drill will retract at each peck.

First Peck: The depth of the first peck. **Subsequent Pecks:** After the first peck is performed the entered distance will be

used for the remaining pecks.

Use Tip Compensation: Toggles the use of tip compensation. This option is used for drilling through a part. It will extend the hole based on the geometry of the drill.

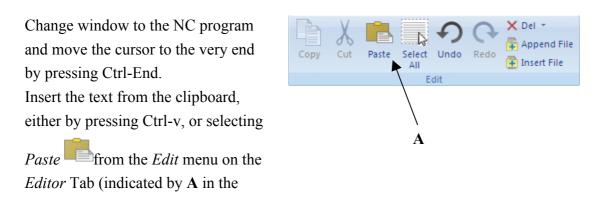
Tip Angle: The angle of the drill. **Drill Diameter:** The diameter of the drill. **Tip Compensation:** The calculated amount, by which the hole will be extended.

Click on **OK** to use the values,

picture).

Try experimenting with the various parameters and see how they change the generated toolpath.

Click on **Export Clipboard** in order to generate the actual program. The program is now in the computer's clipboard and is ready to be inserted into the CNC program.

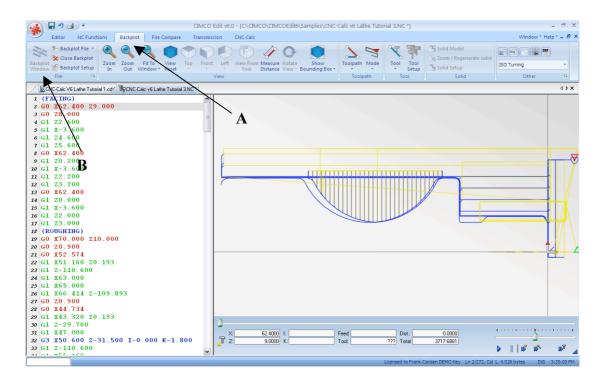


Now the NC program should look like the following screen.

🔬 🖬 🤊 🖨 🔻		CIMCO Edit v6.0 - [C:\CIMCO	\CIMCOEdit6\Samples\CNC-	Calc v6 Lathe Tuto	rial 3.NC *]			- = x
Editor NC Functions	Backplot File Compare	Transmission CNC-Calc					Window* H	elp * 💷 🗗 🤇
Backplot File * Backplot Window Backplot Setup File ©	m Zoom Fit To Out Window Reset		easure Rotate Show stance View * Bounding Box *	Toolpath Mode	Tool Tool Setup	Solid Model Com / Regenerate solid Solid Setup Solid	ISO Turning Other	*
NCCalc V6 Lathe Tutorial 1 or	CNC-Calc v6 Lathe Tutorial 3.1	IC *						4 Þ X
NC-Assistant	240 G1 Z-108.400							
Description: Modify Cycles / Macros Program Stat and End Tool change with Constant Staff. Tool change with Constant Staff. E00 Bapdirows 601 Lones rove 603 Curverclockvise arc move 603 Curverclockvise arc move 603 Curverclockvise arc move 603 Lones rove 603 Lones rove 604 Devel 611 Data setting mode 611 Data setting mode 611 Data setting mode 611 Data setting mode 611 Data setting mode	242 60 X51.200 243 (THREADING) 244 60 X34.000 Z6. 245 60 X22.125 246 632.2-23.000 F 247 60 X34.000 249 60 X21.348 250 632 2-23.000 F 251 60 X34.000 252 60 Z6.000 256 60 X20.752 256 60 X34.000 256 60 X34.000 257 60 X20.752 259 632 X-23.000 F 259 60 X34.000 260 (DRILLING) 261 (DRILLING) 262 60 X0.000 Z10.	3.000 3.000 3.000	000					
G18 ZX plane designation G20 Inch data input G21 Metric data input G27 Reference point return che G28 Return to reference point	263 G0 Z3.000 264 G1 Z-8.000 F20 265 G0 Z-3.000 266 G0 Z-5.000	0.000						
G29 Return from reference point G30 Return to 2nd, 3rd and 4th G31 Skip cutting G32 Thread cutting constant le G34 Threading function variabl G36 Automatic tool comeensati Find Add	267 G1 Z-14.000 268 G0 Z-9.000 269 G0 Z-11.000 270 G1 Z-20.000 271 G0 Z10.000 272 C1 C10.000							
					Licensed to Frank O	arlsen DEMO Key Ln 272/272, C	ol 1, 4.026 bytes INS	3:31:56 PM

To verify the generated toolpath, we must simulate it using the integrated Graphical Backplot. To open the backplot window, click on **Backplot** tab (indicated with letter

A) and then on **Backplot Window** icon (indicated with letter **B**) as shown below. Now a window like the one below will appear.

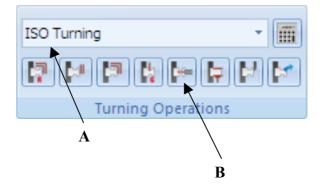


19.9. Tapping the part

In the following tutorial we will generate a Tapping NC program. In order to select the Drilling operation the below described steps must be performed.

Select the file type (NC Format) for our facing example program (e.g. ISO Turning) as indicated with letter **A**.

Then Select the feature **Drilling** by clicking on the *Drilling* icon indicated by letter **B** in the *Turning Operations* toolbar.



This will open the End Drilling dialog to the left of the drawing area. Now insert the values shown in the dialog below.

End Drilling	
Comment:	
M8 TAPPING	
Clearance Height:	_
10 📔	z
Retract Height:	
3 👔	x
Start Depth:	
2 Z	Į
End Depth:	
-12 ×	Į
Parameters)
New)
Export Editor Export Clipboard)

Comment:

This comment will be shown in the final NC Program. It is always good to include a comment, in order to distinguish the various operations in the final program.

Clearance Height:

The position, to where the tool is moved before the actual operation, and where it will end after the drilling finished.

Retract Height: The distance from the start of the operation where the feedrate will switch from rapid to feed.

Start Depth: The depth at which the actual operation is started.

End Depth: The final depth of the operation.

The Tapping operation is defined by the above parameters, after the entry of which the screen will look something like the one shown below.

The four entered distances are shown as crosses on the drawing.

				CIMCO Ed	4.45.0				
	Backplot File Co	mpare Transmission	CNC-Calc	CIVICO EU	11 10.0				Window * Help * = 8
Copen Drawing * A	Zoom Out				$\sim \sim \sim$				
Save	Zoom All	Xrs is		XVX	$\Theta O $		\bigcirc A \ge	ISO Turning	- 🔳
lew wing TSetup CNC-Calc In	n 📃 Zoom Window	XYELO			lenter Two Ladius Points 💐 🕇	Rectangular Bo	It Circular Bolt Text Hole Pattern Entry		집 🔚 🖬 🖬
File	View	Modify	Snap Dra	w Points / Lines	Draw Arcs / Circle		tern Text	Turnii	ng Operations
CNC-Calc V6 Lathe Tutorial 1.cdd	Techo Calc v6 Lathe	Tutorial 3.NC *							41
End Drilling	· · · ·	· · · ·							· · · · ·
Comment: M8 TAPPING									
Clearance Height 10									
Retract Height:									
Start Depth: 2									
End Depth:			$\langle \cdot \rangle$						
Parameters						1			
Export Editor Export Clipboard									
Element Info									
								T • • •	wit t
								• •	
									25.00
									· · ·
	Machine Zero Z:	0.0 Machir	e Zero X: 0.0				Use	Y-Axis Substitution	Part Diameter

Click on the **Parameters** button to open the parameter dialog.

Enter the following values shown below into the parameter dialog. Notice that the dialog is different from the one in the previous drilling operation. It is no longer possible to use pecking because the Operation Type is selected as **CW Tapping**.

rilling Parameters	\$		
Operation Type			
CW Tapping			*
Feedrate:		Dwell:	
	125.0		0.0
- <mark>Pecking</mark> Vise Pecking			
Peck Clearance:		Peck Retract:	
			5.0
First Peck:		Subsequent Pecks:	
	10.0		6.0
Tip Compensation			
Use Tip Compensa	ation		
Tip Angle		Drill Diameter	
	30.0		10.0
		Tip Compensation	0.0000
			8.6603
		Cancel	ОК

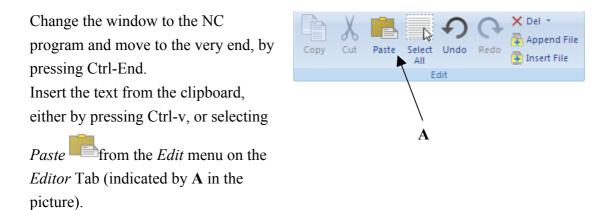
Operation Type: The operation type can be drilling or threading that is either Clock or Counter Clock Wise. **Feedrate:** The feedrate used for all feed moves. Depending on the machine this may be in a different format. For this machine's G32 code a feedrate of 125 will actually result in a pitch of 1.25. **Dwell:** Is the time that the drill will dwell at the bottom of each cut in order to break the chip.

Use Tip Compensation: Toggles the use of tip compensation. This option is used for drilling through a part. It will extend the hole based on the geometry of the drill.

Tip Angle: The angle of the drill. **Drill Diameter:** The diameter of the drill. **Tip Compensation:** The calculated amount by which the hole will be extended. Try experimenting with the various parameters and see how they change the generated toolpath.

Click on Export Clipboard in order to generate the actual program.

The program is now in the computer's clipboard and is ready to be inserted into the CNC program.



Now the NC program should look like the following screen.

CIMCO Edit v6.0 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc v6 Lathe Tutoria	al 3.NC *] _ 🖛 🗙
Editor NC Functions Backplot File Compare Transmission CNC-Calc	Window - Help - 🗗 🗙
New Open Close Save Print File File File File File File File File	Ind Previous Image: The Next Tool change Image: The Next Tool Change Image: Tool Change Image: Tool Change Image: Too
CNC Calc V6 Lathe Tutorial 1.cd ² 👸 CNC-Calc v6 Lathe Tutorial 3.NC *	4 Þ ×
NCAssistant 248 60 26.172 249 60 x21.348 250 632 x23.000 F3.000 250 632 x23.000 F3.000 252 60 x20.752 254 632 x20.752 254 632 x20.752 254 60 x23.000 F3.000 255 60 x34.000 255 60 x20.752 254 32 50 22.27 300 F3.000 255 60 x20.752 254 50 22.7 300 F3.000 255 60 x20.752 25 52 52 52 52 52 52 50 x20.752 25 52 25 53 300 53 30	
Modily 259 G0 X34.000 260 C2.000 260 G0 26.000 260 C2.000 260 G0 26.000 261 DETLLING) 262 G0 20.000 Tod charge with Contart Start, Tod charge with Contart NBM 262 G0 23.000 100 dharge with Contart NBM 263 G0 23.000 266 G0 2.5.000 260 Dil Lines move 266 G0 2.5.000 266 G0 2.5.000 261 Detartsmove 266 G0 2.5.000 267 G1 2.14.000 263 Currant NBM 266 G2 2.5.000 267 G1 2.10.000 261 Data setting mode careling mode 267 G1 2.10.000 267 G1 2.72.0.000 276 G1 2.72.0.000 276	
G20 Inch dda roud 272 (M6 TAPP TING) G27 Aleeince point return from reference point 273 G0 X0.000 Z10.000 G28 Return for reference point 274 G0 Z3.000 G37 Biturn for reference point 274 G0 Z3.000 G38 Return for reference point 275 G0 Z 2.10.000 G31 Stip cutting 277 G32 Z -12.000 F125.000 M05 G32 Thread cutting constant le 276 G32 Z -12.000 F125.000 M05 G34 Lumack to commerail 279 M03 279 M03 G34 Lumack to commerail 279 M03 280 279 Find Add 279 M03 280 279	=
نا ا	icensed to Frank Carlsen DEMO Key Ln 280/280, Col 1, 4.136 bytes INS 3:39:42 PM

To verify the generated toolpath, we must simulate it using the integrated Graphical Backplot. To open the backplot window, click on **Backplot** tab (indicated with letter

A) and then on **Backplot Window** icon (indicated with letter **B**) as shown below.

Now a window like the one below will appear.

