CNC-Calc v7 User Guide

February 2014 | Copyright © 1991-2014 by CIMCO A/S | Web: www.cimco.com | Email: info@cimco.com

Table of Contents

CN	C-C	alc v7 User Guide	1
Lice	ense	Information	6
1.	Ov	erview	8
1	.1.	Designed for usability and productivity	8
1	.2.	User Interface	9
2.	Ins	tallation	10
2	.1.	Enabling CNC-Calc	10
3.	Mo	ouse Functions	11
4.	CN	C-Calc Tab	12
4	.1.	File	12
4	.2.	View	13
4	.3.	Modify	14
4	.4.	Snap	16
4	.5.	Points and Lines	17
4	.6.	Arcs and Circles	19
4	.7.	Pattern	21
4	.8.	Text	22
4	.9.	Milling Operations	23
4	.10.	Turning Operations	25
5.	Pro	ogram Menu	27
5	.1.	File Menu	27
6.	CN	C-Calc Settings	
6	.1.	Main Configuration	
6	.2.	Color Configuration	
7.	Usi	ing CNC-Calc Help	
7	.1.	Using Help In Dialogs	31
7	.2.	Printing Help Information	32
CN	C-C	alc Tutorials	
8.	Bef	fore You Start	
9.	Tu	torial Mill 1 - 2D Construction of a Part for Milling	35
9	.1.	Open a New Drawing	
9	.2.	Draw the Geometry	
	9.2	.1. Draw a rectangle with round corners	

9.2.2.	Draw a circle defined by its center and radius	
9.2.3.	Draw vertical and horizontal lines	
9.2.4.	Offset a circle	40
9.2.5.	Create a fillet between elements	41
9.2.6.	Mirror elements	42
9.2.7.	Connect the inner elements	43
9.3. S	ave the File	44
10. Tut	orial Mill 2 - Machining the Part	45
10.1.	Open an Existing Drawing	46
10.2.	Face Milling	47
10.2.1	Creation of Facing Toolpaths	47
10.2.2	2. Inserting a Tool with Feed and Speed Calculator	51
10.3.	Contour Milling	53
10.3.1	Creation of Contour Toolpaths	53
10.3.2	2. Inserting a Tool with Feed and Speed Calculator	61
10.4.	Pocket Milling	62
10.4.1	Creation of Pocket Toolpaths	62
10.5.	Drilling	70
10.5.1	. Generate a Drill Cycle	70
11. Tut	orial Mill 3 - Backplot in the Editor	75
11.1.	Before You Start	75
11.2.	Open an Existing NC Program	76
11.3.	The First Backplot	77
11.3.1	. Backplot Tool Setup	78
11.3.2	2. Backplot Stock Setup	80
12. Tut	orial Mill 4 - Milling of Letters	83
12.1.	Open a New Drawing	83
12.2.	Simple Letters Milling	84
12.2.1	. Drawing the Text	
12.2.2	2. Milling of Letters	85
12.3.	TrueType Letters Milling	89
12.3.1	. Drawing the Text	89
12.3.2	2. Contour Milling of Letters	91
12.3.3	B. Pocket Milling of Letters	94
13. Tut	orial Lathe 1 - 2D Construction of a Part for Turning	97

13.1. Op	en a New Drawing	
13.2. Dra	aw the Geometry	99
13.2.1.	Draw vertical lines defined by start point and length	99
13.2.2.	Offset an element	101
13.2.3.	Draw a circle defined by its center and radius	
13.2.4.	Draw a line defined by its endpoints	
13.2.5.	Draw a horizontal line defined by its length	104
13.2.6.	Draw a polar line	105
13.2.7.	Chamfer the foremost corner	106
13.2.8.	Draw a line defined by its endpoints	107
13.2.9.	Trim between points of intersection	108
13.2.10.	Fillet intersections	109
13.3. Sav	e the File	112
14. Tutoria	l Lathe 2 - Machining the Part	113
14.1. Op	en an Existing Drawing	114
14.2. Fac	ing the Front of the Stock	115
14.2.1.	Creation of Facing Toolpaths	115
14.2.2.	Exporting the Toolpath and Backplot in the Editor	118
14.3. Ro	ughing the Part	119
14.3.1.	Creation of Roughing Toolpaths	119
14.3.2.	Exporting the Toolpath and Backplot in the Editor	
14.4. Gro	poving the Part	126
14.4.1.	Creation of Grooving Toolpaths	126
14.4.2.	Exporting the Toolpath and Backplot in the Editor	131
14.5. Fin	ishing the Part	132
14.5.1.	Creation of Finishing Toolpaths	132
14.5.2.	Exporting the Toolpath and Backplot in the Editor	137
14.6. Th	reading the Part	139
14.6.1.	Creation of Threading Toolpaths	139
14.6.2.	Thread Form Tab	140
14.6.3.	Exporting the Toolpath and Backplot in the Editor	143
14.6.4.	Select Thread	145
14.7. Dri	lling the Part	146
14.7.1.	Generate a Drill Cycle	146
14.7.2.	Exporting the Toolpath and Backplot in the Editor	149

14.8. Ta	apping the Part	150
14.8.1.	Generate a Tap Cycle	150
14.8.2.	Exporting the Toolpath and Backplot in the Editor	153
14.9. Pa	arting-off the Part	154
14.9.1.	Create a Cutoff Toolpath	154
14.9.2.	Exporting the Toolpath and Backplot in the Editor	

License Information

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

TERMS OF USE FOR:

Software: CNC-Calc v7 Version: 7.x.x Date: February 2014 Copyright © 1991-2014 by CIMCO A/S

Notice

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

Software License

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

Disclaimer of all Warranties and Liability

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

Notice:

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

Copyright © 1991-2014 by CIMCO A/S. All rights reserved.

This software contains confidential information and trade secrets of CIMCO A/S. Use, disclosure, or reproduction is prohibited without the prior express written permission of CIMCO A/S.

CIMCO CNC-Calc, CIMCO Edit and the CIMCO Logo are trademarks of CIMCO A/S.

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

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

1. Overview

CIMCO CNC-Calc v7 is an add-on for CIMCO Edit that enables novice programmers to draw 2D geometrical contours, lay out toolpaths for mill and lathe, and simulate the resulting NC program.

CNC-Calc v7 is a great tool for the operators and toolmakers who are untrained in the use of advanced CAD/CAM systems. For them, CNC-Calc v7 can help increase productivity and assist in the day-to-day NC programming. For a small company it can be the first step into the CAD/CAM world.

1.1. Designed for usability and productivity

CIMCO CNC-Calc v7 is designed for ease-of-use that enables the user to draw contours fast and easily. It features common functions for drawing lines and circles in relation to the coordinate system and/or existing geometry. Functionality ranges from the plain "horizontal line" to the complex "circle tangent to three elements". It includes advanced trimming capabilities and an easy point and click approach for toolpaths layout.

CIMCO CNC-Calc v7 imports DXF files. From DXF files it is possible to generate toolpaths for lathe and mills, such as ISO, Fanuc and Heidenhain controllers. Other features include generation of user-defined compensation types like computer, controller, wear, and reverse wear.

Since CIMCO CNC-Calc v7 is an integrated part of CIMCO Edit it is an easy task to view, edit, and simulate generated toolpaths. This enables the user to validate programs and thereby optimize the use of machine resources.

1.2. User Interface

CNC-Calc v7 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:

							(СІМСС) Edit v	7.0 - (l	Intitle	d Drawi	ng]													-		×
Editor NC Functions	Backp	lot	File Co	mpare		nsmissio		CNC-C	alc															W	indow	r≛ Helj	p ~ _ d	۶×
📄 🚔 Open Drawing 👻 💽	•	XI	Y	24		/ AU		- I.		(.)(27		F	1	(3		A	A a	ISO Mi	lling				•	
New Drawing P Setup CNC-Calc Ir						×**				Cent	er Tv	vo ints	X	Rec	tangu	lar Boll	t Circu	lar Bol		ext ntry	a a	-	0			A	a	3
	/iew		Modi	-		Snap	Drav	/ Point				arcs / C			101e Pi	attern Patte		Patterr		Text	*		M	illing	Opera	ations		
Untitled Drawing																											4 ۵	×
CNC-Calc	1								-		-				-												-	-
			*						*		•			•	*			•		*			-			•	*	,
								Ż			1						÷				÷							
			,																	,				,				,
																							-					
																										• •		
											+			4														
	· •									• •									•							• •		
		• •		• •											-			-									-	
														-		•									-	•		
Element Info	· ·	• •	*						*		*	•		•	*	• •		•		*		•		*		• •	*	*
	1	• •		• •				•	*	•	4			4	•			4				•	*			• •	*	
			,					,			4			•	*			*	, .								•	
								÷			÷.						÷				÷			Ì	÷			
														4						i.				"4 S	. po		T	
																				Ļ							ļ	
	<u> </u>										-																	
		(achine)	Zero X:		0		ine Zero	Y:	0.0	Machi	ne Zer	οZ:	0.0						[Use	Y-Axi	s Subst	itution		t Diam		5	0.0
														renser	tto D	MO - N		D DFC/						¥: 50 I	000	INC	15:54:46	
														icensed		.wi0 - P		IN NESH	a.c C	wie07	473	N: 0	5.145	1. 50.		1143 .	5.54:40	

CNC-Calc main window.

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. Below an example of the Element Info pane display is shown.

	Rectangle
	First Corner X:
1	Î 🔭
	First Corner Y:
1	l Tr
_	Second Corner X:
	
	Second Corner Y:
	<u>ц</u>
	Corner Radius:
	\mathbf{X}

CNC-Calc pane with entry fields.

Element Info - Arc						
Center X:	-57.500					
Center Y:	20.729					
Radius:	5.000					
Length:	6.405					
Start Angle:	106.602					
End Angle:	180.000					
Sweep:	73.398					

Element Info pane with information.

2. Installation

CNC-Calc v7 is installed as part of CIMCO Edit v7. Please see the CIMCO Edit v7 documentation for installation instructions.

If you are upgrading from an existing installation without CNC-Calc, reinstallation of CIMCO Edit v7 is not necessary. Just copy the new key file (named "license.key") to the appropriate directory.

2.1. Enabling CNC-Calc

Start CIMCO Edit v7 and verify that the CNC-Calc tab is available in the Ribbon.

If not, open the editor configuration by clicking the **Global Setup** icon in the *Editor* tab. Go to the Plugins section in the left tree of the configuration window. Now ensure that the option **Disable CNC-Calc** is unchecked. You must restart the program to activate the CNC-Calc tab.

Setup: Plugins		? ×
Setup: Plugins General Gitor File types Goodsave Slock numbers Cod/Save File compare Machine Backplot Tool scanning CNC-Calc Multi Channel Tool List Other Global Colors External Commands Mazatrol Viewer Plugins	Plugins Select plugins Disable Backplot Disable advanced simulation Disable DNC/Serial communication Disable File compare Disable advanced NC-Functions Configuration password Configuration password: Macros/Machine configuration path:	
	1	Help Default Cancel OK

Enabling the CNC-Calc tab.

3. 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 (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 \mathbb{N} 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.

4. CNC-Calc Tab

In this section, the functions under the CNC-Calc tab are described.

The functions in CNC-Calc v7 are activated using the appropriate icons on the Ribbon. The commands are organized into different menus under the *CNC-Calc* tab.

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



The CNC-Calc add-on is optional and will be present in the Editor only if you also purchased it.

If the *CNC-Calc* tab is not available, please refer to Installation to read about how to start CNC-Calc.

4.1. File

This section describes the functions in the **File** menu. These functions are used to handle file operations like loading and saving files. You can also access the CNC-Calc configuration through this menu.





New Drawing

Clears the graphics area and opens a new document.



Open Drawing

Opens existing CNC-Calc drawing files or DXF files.

The downwards arrow next to the *Open Drawing* 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* function can also be accessed with Ctrl+S.



Setup CNC-Calc

Lets you access the configuration for CNC-Calc.

Please refer to the section CNC-Calc Settings for more information on how to configure CNC-Calc.

4.2. View

The functions in the **View** menu control the way the drawing is displayed in the graphics area. Here it is possible to perform various zoom functions.





Zoom In

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



Zoom out

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



Zoom All

Use this function to fit 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*.

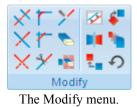


Zoom Window

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.

4.3. Modify

This section describes the functions in the **Modify** menu. These functions are used to modify the drawing geometry in different ways.





Trim To Intersection

Use this option to 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 to 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

Use this option to 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

Use this option to 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

Use this option to create 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

Use this option to create a chamfer between two elements, with a defined angle and/or distance.

It is optional whether the two elements should also be trimmed to the chamfer.



Break Element

Break element 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

Use this option to join two selected elements into one.



Delete

Use this option to delete the elements you select. They can be restored with the Undo function (the \mathcal{O} Undo icon in the Modify toolbar).



Remove Duplicates

Select this option to delete elements that are duplicated in the drawing.



Offset Elements

Use this option to offset the elements you select by a specified distance.

It is optional whether the original element should be kept.



Mirror Elements

Use this option to perform a mirror operation of the elements you select along a line selected as the mirror axis.

It is optional whether the original element should be kept.



Translate Elements

Use this option to 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

Use this option to rotate elements. This 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 element should be kept.



Scale Elements

Use this option to 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 element should be kept.



Undo

Select this option to 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*.

4.4. Snap

This section describes the different kinds of snap options that are available in the **Snap** menu.

The snap option is only available when some point position has to be selected or indicated.



The Snap menu.

+ + + + • + + + +

Snap to Grid

Toggle snap to the grid points.

Snap to Points

Toggle snap to points drawn by the user.



Snap to Center Points

Toggle snap to the centers of circles and arcs.



Snap to Mid Points

Toggle snap to the midpoints of elements.



Snap to End Points

Toggle snap to the endpoints of elements.



Snap to Intersections

Toggle snap to the intersections between elements.



ALL Enable All Snap Types

Activates all snap types.

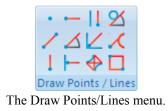


Disable All Snap Types

Deactivates all snap types.

Points and Lines 4.5.

This section describes the functions in the Draw Points/Lines menu. This menu contains functions for drawing points and lines defined in different ways.



Point

Use this option to draw a point at the selected position.



Between 2 Points

Use this option to draw a line between two selected points.

Vertical

Use this option to 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

Use this option to 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

Use this option to draw a line defined by its polar coordinates.

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

Perpendicular

Use this option to draw 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

Use this option to draw 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

This option allows you to draw 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

Use this option to 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

Use this option to draw 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

This option lets you draw a line tangent to an arc or circle that goes through 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 through, and last you select which of the two solutions you want to keep.



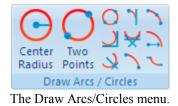
Rectangle

Use this option to draw 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).

4.6. Arcs and Circles

This section describes the functions in the **Draw Arcs/Circles** menu. These functions let you draw full circles (360 degree arcs) and arcs defined in different ways.



Center Radius

This option lets you define the center of the circle, followed by the radius.

Two Points

Use this option to define the circle by selecting two (diametrically opposite) points.



Three Points

Use this option to define the circle by selecting three points on the periphery of the circle.



Tangent Two Elements

This option 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

This option 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

This option 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.



Use this option to define 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

This option 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

This option lets you draw an arc by selecting three points.

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

Use this option to define an arc by its center point, radius, starting angle, and end angle.

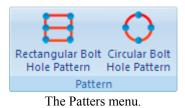


Tangent Dynamic

Use this option to construct dynamic tangent from one element through a point.

4.7. Pattern

This section describes the functions in the **Pattern** menu. These functions let you easily create rectangular and circular bolt hole patterns.





Rectangular Bolt Hole Pattern

Use this option to define a rectangular bolt pattern.

You first select the start point of the pattern (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

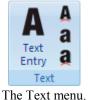
Use this option to define a circular bolt pattern.

You first 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.

4.8. Text

This section describes the functions in the Text menu.

These functions let 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.







Text Entry

Use this option to define 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.



Text Circular Entry

Use this option to define 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.



TrueType Text Entry

Use this option to define TrueType 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.



TrueType Text Circular Entry

Use this option to define TrueType 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.

4.9. Milling Operations

The functions in the **Milling Operations** menu allow 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.

ISO Milling 🔹					
🛨 🖻 🔍 🖿 🌒 🗊 🗛 ai	3				
Milling Operations					
The Milling Operations many					

The Milling Operations menu.



Face Milling

Use this option to create a facing operation based on a selected outline contour.



Pocket Milling

Use this option to create 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

Use this option to create operations for contour milling.

A contour operation can machine multiple contours with roughing and finishing cuts, but only with one tool.



Drill Holes

Use this option to create 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 Drilling

Use this option to 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

Use this option to create threading milling operations. The threads can be inside or outside, and can be created for tools with one or multiple teeth.

Multiple threads can be selected with the use of filters, or by selecting the individual circles from the drawing.



Mill Letters

Use this option to generate operations for milling simple letters on the drawing. These letters will have to be drawn using the simple letter function.

All letters can be selected with the window function, and machined in operations based on their start and end depths.

A Mill TrueType Letters

Use this option to create operations to mill the outline and/or the interior of the individual letters. These letters will have to be drawn using the TrueType letter function.

The letters can be selected with the window function, and all letters with the same parameters can be machined in one operation.



Export Contour

This option 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.



Calculator

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.

4.10. Turning Operations

The functions in the Turning Operations menu allow 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.

ISO Turning 🔹	.				
Turning Operations					

The Turning Operations menu.



Roughing Turning

Use this option to 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.



Face Turning

Use this option to create a facing operation.

This operation can contain both roughing and finishing cuts, but only with one tool.



Finish Turning

Use this option to create a finish operation based on a selected contour. This operation will make only one toolpath that fits the selected contour.



Grooving

Use this option to create a grooving operation based on a selected contour.

This will enable machining of the areas that could not be machined with either the Roughing or Finishing operation.



End Drilling

This option 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 Horizontal

Use this option to create 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

This option 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

This option 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.



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.

5. Program Menu

Most of the CNC-Calc v7 functions are accessed through the menus in the CNC-Calc tab. However, a few are logically located under the global *** File** menu.

5.1. File Menu



Close

Closes the active file. If the active file has been modified, you will be prompted to save it. Close can also be accessed with Ctrl+F4.

	A	
C	9	
		1

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. **Save** can also be accessed with *Ctrl+S*.



Save As

Saves the drawing under a different name.

Exit

Closes CIMCO Edit v7, and thus also CNC-Calc v7. If any files have been modified, you will be prompted to save them. **Exit** can also be accessed with Alt+F4 (Alt+F4 is a Windows standard).



The functions *New* and *Open* under the sicon respectively create an empty NC document and open an NC file from disk. To create or open CNC-Calc drawings use the functions *New Drawing* and *Open Drawing* located in the File toolbar.

6. CNC-Calc Settings

This section describes in details the configuration of CNC-Calc.

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

6.1. Main Configuration

The main configuration is entered most easily by selecting the icon **Setup CNC-Calc** from the *File* menu.

	音 Open Drawing 🔹
New	Reference Save
New Drawing	Setup CNC-Calc
	File

Opening the CNC-Calc setup window.

It is important to select first the correct machine type under the *File types* setup dialog, since all settings will apply for the selected machine. The window below shows the main configuration dialog.

Setup: CNC-Calc - ISO Milling	-?	x
General Editor Printing Colors Block numbers Load/Save File compare Machine Backplot Tool scanning	CNC-Calc - ISO Milling CNC-Calc Settings Turning Ø Diameter programming Selected Post Processor: C:\CIMC07BETA\CIMC0Edit7\Posts\Mill\fanuc.js Grid	
Other Other Other Global Colors External Commands Mazartol Viewer Plugins	Show grid Grid size: 10.00 Show origin	
	Help Default Cancel	ок

Main CNC-Calc configuration window.

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.

CNC-Calc Settings

Turning

Check this option to select turning (lathe) output.

Diameter programming

Selects whether X axis output is in diameter measurement or in radius measurement. This option is only available if turning (lathe) output is selected.

Selected Post Processor

Use this field to specify the post processor that should be used to format the output of the CNC-Calc operations. Click the folder icon \square at the right of the field to browse for the desired post processor.

If changes need to be made to a post processor to suit your personal needs, feel free to contact CIMCO A/S at info@cimco.com or at telephone +45 4585 6050.

Grid

Show grid

Use this check box to specify if the grid should be shown.

💴 🗄 Grid size

Use this field to set the spacing between main grid points.

Show sub-grid

Select this option to have a sub-grid visible when zoomed in to a degree that shows few main grid points.

Show origin

Select this option to have lines along X and Y zero visible.

6.2. Color Configuration

The Color configuration is entered by selecting *Global Colors* in the configuration tree, and scrolling down to the CNC-Calc colors in the list.

Setup: Global Colors		? ×
General - Editor - Printing File types - Colors - Block numbers - Load/Save - File compare - Machine - Backplot - Tool scanning - CNC-Calc - Multi Channel - Tool List - Other - Eternal Commands - Mazatrol Viewer - Plugins	Global Colors Fext CNC Calc Background CNC Calc Grid CNC Calc Sub-grid CNC Calc Stelectable CNC Calc Selectable CNC Calc Selectable CNC Calc Selectable CNC Calc Element Hit CNC Calc Element Hit CNC Calc Freed Move Red: Blue: Select Color Default	
	Help Default Cancel	OK

CNC-Calc color configuration window.

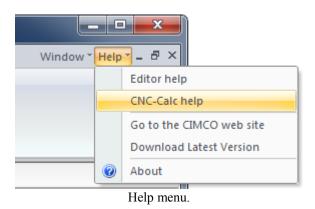
To change the color of a CNC-Calc element, either left click the element in the list to select it and then click the **Select Color** button, or you can double-click the element in the list. The color can then be picked from a standard palette, or a custom color can be defined.

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

7. Using CNC-Calc Help

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

A standard Windows Help file is available through the drop-down **Help** menu by clicking **CNC-Calc help**.



7.1. Using Help In Dialogs

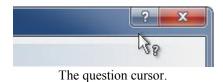
CIMCO CNC-Calc v7 contains multiple dialogs that provide drop-down lists, check boxes, text fields and buttons for specific configurations.

To get help for a particular dialog item, click the ? at the top of a dialog box, as shown below.

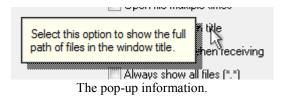


Click the question mark.

This will change the cursor to an arrow and a question mark indicating that you are in *Help Mode*. The new pointer is shown below.



Now click on the item that you want further information about. If help is available for the selected item, a small window will pop up displaying the help text. If no help is available the general help file is launched. An example of a pop-up help is shown below.



You can also display the pop-up help for a dialog item by clicking in the field and then pressing **F1**.

7.2. Printing Help Information

If you wish to print just a single section of this help file, click the **Print** button on the toolbar (shown below).



If you wish to print more than a few sections, you should open the PDF version of the help file. The PDF document produces a much nicer printout.

You can download the PDF version of the user guide from the CIMCO website.



You will need Adobe Reader to open the PDF file.

8

CNC-Calc Tutorials

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

Important

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.

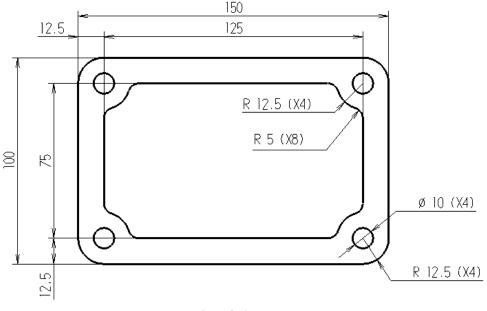
The same is the case in terms of program start and program finish codes. These codes could for instance be inserted with the macro functionality provided in the CIMCO Edit editor.

8. Before You Start

The first thing to do before proceeding with any of the tutorials is to set the toolbar parameters. Start CIMCO Edit v7 and verify that the CNC-Calc tab is available in the Ribbon. If not, open the editor configuration by clicking the **Global Setup** icon in the *Editor* tab. Go to the Plugins section in the left tree of the configuration window, and ensure that the option **Disable CNC-Calc** is unchecked. You must restart the program to activate the CNC-Calc tab.

Enabling the CNC-Calc tab.

9. Tutorial Mill 1 - 2D Construction of a Part for Milling



Sample 2D part.

This tutorial demonstrates one of many ways in which the 2-dimensional part above can be drawn in CNC-Calc v7. 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:

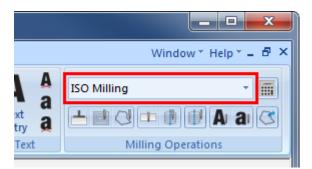
- Draw a rectangle with a corner radius.
- Draw a circle with known center and radius.
- Draw vertical and horizontal lines from known points.
- Offset a circle.
- Make curves between elements.
- Delete elements.
- Mirror elements about lines.
- Join end points with straight lines.

9.1. Open a New Drawing

To make a new drawing, click on the icon **New Drawing** in the *File* toolbar under the CNC-Calc tab. When the new drawing is opened the following window is displayed:

🔊 🗄 9 🚄) =									CIMC	O Edit	v7.0 -	Untitle	ed Drav	/ing]														l	-		×
Editor NC Functions	Back	plot	File	Comp	are	Tran	smissio	n	CNC-C	alc																	Wi	ndow	r* Hel	p* -	. 8
📄 💣 Open Drawing 🔹 🧃	<\X⊤Y ፼₽			/ AU-			· - \(\texts)										()		A	A a	ISO	Milli	ing				+			
New Setup CNC-Calc	om 🔍 n 🗔	X				0	<u>/ **</u>			∠`X ₽ □		ter T	wo pints	٦¥.		Rectan	gular	Bolt	Circul	ar Bol		Text intry	a	L.	ek.	0			A	a	C
	'n 🔍 View		Mo	odify	•)		nap	Drav		7 🖵 ts / Li	Rac		Arcs / (Hole		rn l 'atteri	Hole F	atter	ľ	ntry Tex							tions		
Lintitled Drawing																														4	۱Þ>
CNC-Calc		-				-																								_	-
																									ĵ.	ĵ.					
																		Ċ					÷		Ì	ĺ.					
																		,									,				
							,															,									
						. ,	,			,					ļ.,		,				. ,					,					
	1.1						,																								
															-											÷					
										-					+		-							-						-	
Element Info	•				•		,	• •	*	*	, ,			, <i>,</i>				*		•	• •	*					*		• •		
	· ·	•	•		•	• •	,	• •	•	*	•	٠	•			•	*	•		•	• •			*	•	*	·		•		
	1.1	*			•						•	*								4				4	*	•			• •		
	1	•	•		•	•		• •	•		•				1	•	*	•		*	• •	•		*	*	*	•		• •		
	1								-	-		-				-	-			-				-						-	
		÷.							÷			•					ĵ.	ć				,			÷.	÷.	ţ,	1			
				÷					Ì			ż					Ĺ		÷				÷	Ĵ	Ì	ĺ	.45	. DO			
							,					į						,				ŀ			,					_	
										-																					
		Machin	e Zero	X:	0.0		Mach	ine Zero	Y:	0.0	Mac	nine Ze	ro Z:	0.0									se Y-A	vis Si	ihstitu	tion					50.
					0.0					0.0				0.0								шо							INS		

Make sure that the file type **ISO Milling** is selected in the **File Type** field of the *Milling Operations* toolbar.



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 \mathbf{I} Setup CNC-Calc and then Global Colors from the tree menu. For this tutorial we have chosen to use blue as our drawing color and white for the background.

9.2. Draw the Geometry

9.2.1. Draw a rectangle with round corners

- Click on the **Rectangle** icon \Box in the *Draw Points / Lines* toolbar.
- Enter the values shown in the figure below to draw a rectangle with sides = 150, height = 100 and corner radius = 12.5, and centered in the drawing area:

Rectangle		
First Corner X:		
	-75	۲x
First Corner Y:		
	-50	۲r
Second Corner X:		
	75	×.
Second Corner Y:		
	50	Ц.
Corner Radius:		
	12.5	r.
	$\mathbf{\times}$	✓

• Click on \checkmark to approve the command and draw the rectangle.

4	4	1.								4				\cdot		
	4					•			e	*						
*							•		*					·		
											-		-		-	
			-					 	 							
-	-	1	-	-	-	-			-	-	-	-	-	1	-	
														.		
4		1.	*	*	*	P	•			4	4	÷	*	•]		
														1		
4	4															

9.2.2. Draw a circle defined by its center and radius

- Click on the Center Radius icon 🕞 in the Draw Arcs / Circles toolbar.
- Enter the value **Circle Radius** = 5 in the CNC-Calc pane.

Circle Center And Radius	
Center Point X:	
	\odot
Center Point Y:	
	0
Circle Radius:	
5.0	C
$\mathbf{\mathbf{x}}$	
	\checkmark

- Activate the function **Snap to Center Points** in the *Snap* toolbar.
- Snap to the center of the left topmost corner arc. The circle will automatically move to the center point when the cursor is close to it. The pointer also changes indicating snap to arc/circle center.
- Left-click to add the circle.

4																
		\square														
4	4	((3			P.			a	0	e	-		•)		
														Ť		
								-	e							
						_										
						~		 -								
				-	-	-						-		- I	-	
	4		*	*		÷	4	h i i			-	-	-			
	1	1	÷		-							-	-	<u> </u>		
4	4			-					a	4	•	-	÷	•]		
		$\overline{\ }$												1		
4	4		L .	-								-				

9.2.3. Draw vertical and horizontal lines

9.2.3.1. Vertical Lines

- Activate the functions Snap to Center Points 🖸 and Snap to Mid Points 🖊.
- Click on **Vertical** in the *Draw Points / Lines* toolbar to draw a vertical line.
- Enter the value Line Length = -20.0 into the CNC-Calc pane. The negative value indicates the vertical line is in the -Y direction from the start point of line.

	Vertical Line		
Start	Point Of Line X:		
			Îx
Start	Point Of Line Y:		
			Îr
Line l	Length:		
		-20.0	ÎΙ
		\sim	\checkmark

- Snap to the center of the circle drawn in the previous step.
- Click to add the vertical line.
- Snap to the midpoint of the topmost horizontal line (match up with the Y-axis).
- Click to add another vertical line with the same values. This one will serve as a mirror line for the mirroring of our part about the Y-axis.

9.2.3.2 Horizontal Lines

- Click on **Horizontal** in the *Draw Points / Lines* toolbar to draw a horizontal line.
- Similar to the above, draw a line with Line Length = 20 from the same circle center. This time Line Length is set to a positive value to indicate the horizontal line is in the +X direction.

• Add then a horizontal mirror line from the center of the left vertical line (match up with the X-axis).

4													-				
		$\left(\right)$															
4	4	$ \cdot $	2		*							*	-		·)		
			2														
*	*												-	-			
		-	-		-	-							-		·	-	
4	÷			*	*			4		• · · ·		÷		-			
							2						-		·		
	4											4			.)		
4	4		-	-	-	-	-	-	-	-	-	-	-				

9.2.4. Offset a circle

- Click on the **Offset Element** icon in the *Modify* toolbar.
- Enter the value **Offset Distance** = **7.5** (12.5 5 = 7.5) into the CNC-Calc pane. Select the option *Copy (uncheck to Move)*.

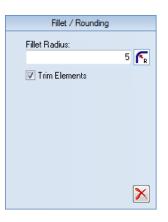
Offset Elemen	t	
Offset Distance:		
	7.5	\$
Steps:		
	1	.
🔽 Copy (uncheck to f	Move)	
		\times

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

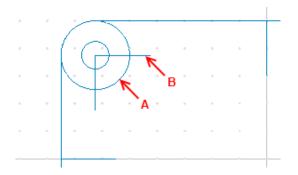
			~														
4		(\mathcal{A}		
			ケ														
*	Ŧ												~		× .		
		1	-		-	-							-	-	÷.,	-	
									 						-		
				_													
4	4	1		*	*	*	*	4	h	6 · ·	6	÷	÷	*	*	*	*
4	4				÷					6 · · ·	6		÷				
4															.		
															Ϊ		
4	4					-							-				

9.2.5. Create a fillet between elements

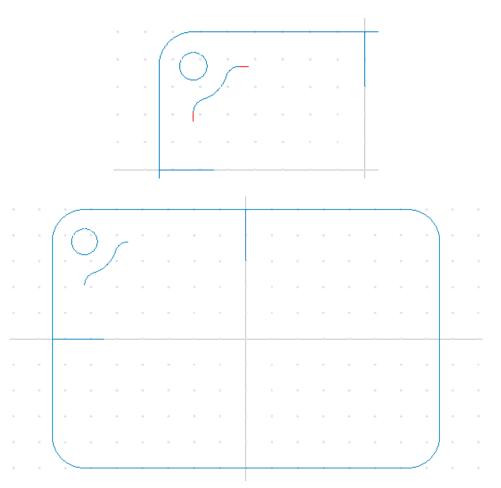
- Click on the **Fillet Elements** icon *T* in the *Modify* toolbar.
- Enter the value Fillet Radius = 5, and leave the option *Trim Elements* checked.



• To select the elements for the fillet operation, click on the circle by A and on the line by 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.
- Click on the **Delete** icon s and delete the two residual lines (the ones pointing away from the center of the circle).



9.2.6. Mirror elements

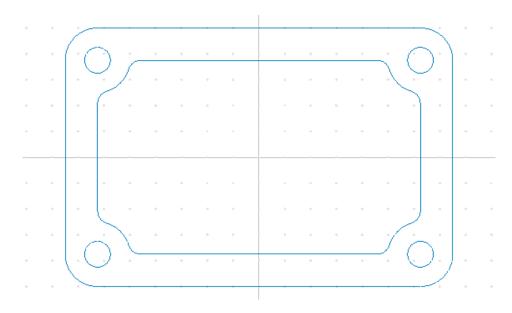
- Click on the **Mirror Elements** icon **II** in the *Modify* toolbar.
- First, click on the vertical mirror line.
- Then click on all the elements which should be mirrored (the circle and the inner corner). You can hold down the left mouse button while dragging out a window around the elements.
- Now do the same and mirror along the horizontal mirror line. Continue mirroring until your drawing looks similar to the one below.

· · · [·C	Section and the section of the secti	1 4 4 4	$\langle (\bigcirc) \rangle = 0$
			· · · · · · ·
· · · ·	$\sum_{i=1}^{n} (i - i) = \sum_{i=1}^{n} (i - i) $		
	.)		$2 \bigcirc 1$
			· · · ·

• Click on the **Delete** icon **S** and delete the mirror lines.

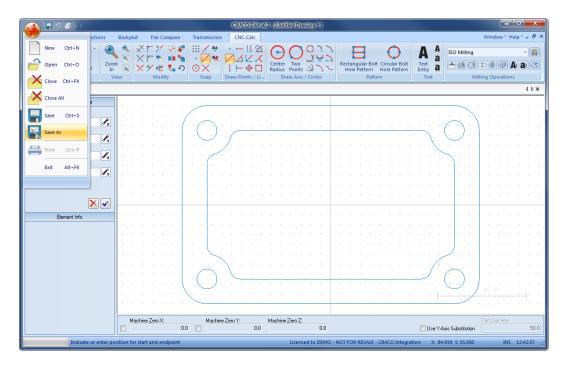
9.2.7. Connect the inner elements

- Activate the snap function **Snap to End Points** */*.
- Click on **Between 2 Points** icon / 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.



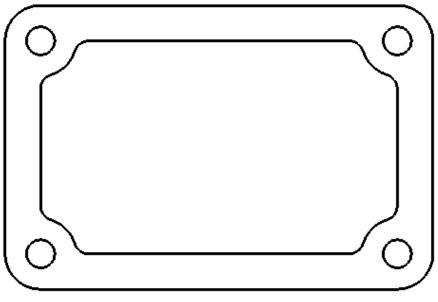
9.3. Save the File

Click on the **Main Menu** Sicon and then select **Save As** from the drop-down menu. Give the file the name *CNC-Calc Milling Tutorial 1* and save it (the file name extension is added automatically).



10. Tutorial Mill 2 - Machining the Part

With CNC-Calc v7 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. There are several machine types in CNC-Calc, but the most commonly used are ISO G-code programming and Heidenhain plain text.



Sample 2D part.

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

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

- Face Milling
- Contour Milling
- Pocket Milling
- Drilling



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

10.1. Open an Existing Drawing

To open a drawing, click on the icon $\stackrel{\frown}{=}$ **Open Drawing** in the *File* toolbar under the CNC-Calc tab. An Open file dialog will appear.

💽 Open		
Look in: 🔋	Drawings	- G Ø ▷ □.
Name	*	Date modified 🔺
CNC-Ca	alc Milling Tutorial 1.cdd	02-01-2012 13:45
№ inch-g7	1.cdd	01-07-2012 21:19 =
🛛 🖹 lathe1.c	dd	01-07-2012 21:19
🖹 metric-	g71.cdd	01-07-2012 21:19
🖹 tutorial	1.cdd	01-07-2012 21:19
🖹 tutorial	4.cdd	01-07-2012 21:19
N ⊤	L	15 00 0010 10.01
File name:	CNC-Calc Milling Tutorial 1.cdd	Open
Files of type:	CNC-Calc Drawing files (*.cdd, *.dxf) Cancel

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

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

				CI	MCO Edi	t v6.2 -	[C:\CIM	ICO/CIN	ACOE	dit6\Dra	wings\	INC-Calc	Tuto	rial Mill	1.cdd *										×
Editor NC Functions	Back	olot	File Co	ompare	Trans	mission	n Cl	NC-Calc														Wind	low * H	elp * -	- 8 ×
Drawing Y Setup CNC-Calc	om 🔍 n 🔍 View		Modif	図 4 第一章 1 fy		<u>/ ak</u>	/ 2 - Draw	- ! ∠ ∠ - ⊕ Points /		Center Radius Dr	Points	O イ ン シン / Circles	インノ	Rectano Hole	gular Bo Pattern Patt		ular Bo Patter	lt Tr n Er	A ext ntry Text	1	D Milling	illing Op	eration	l a	
CNC-Calc Milling Tutorial 1.cdd *																									٩Þ×
CNC-Calc	-	· · · · · · · · · · · · · · · · · · ·	• •					· · ·	•	· · · · · · · · · · · · · · · · · · ·	· · ·	· · ·			· · ·	-	· · ·		-				· · · · · · · · · · · · · · · · · · ·		
Bement Info	-	· ·	· ·		· ·	-	 	· ·	•	• •	• •	· · ·		· ·	· ·	•		• •	•	· ·	· ·		· ·	•	· ·
		• • •	• • • •			L D		 	•	• •	• •	+ + + +		• • • • •	• • • •	•	, , , , ,		J				 	•	
		Machine	Zero X:			Machin	ne Zero Y		0.0	Machine	Zero Z:	0.0				-					ubstitution		iameter	•	50.0

 $\mathbf{\hat{0}}$

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 \mathbf{I} Setup CNC-Calc and then Global Colors from the tree menu.

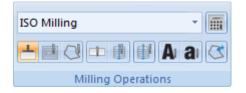
10.2. Face Milling

CNC-Calc v7 can generate CNC toolpaths for face milling, with or without finishing passes and using different cutting strategies.

10.2.1. Creation of Facing Toolpaths

First, in the *Operations* toolbar, select the programming format of the NC-program in the *File Type* drop-down menu. Select **ISO Milling**.

Then click on the icon **here Face Milling** to generate a CNC-Toolpath for face milling.



Write the text FACING in the *Comment* field of the CNC-Calc pane Face Milling. 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 of the drawing. This will select the bounding contour that the facing operation will operate on.

🔬 🖬 9 🗉 🔹	CIMO	O Edit v6.2 - [C:\CIMCO\CIMCOE	dit6\Drawings\CNC-Calc Mill	ing Tutorial 1.cdd *]	
Editor NC Functions	Backplot File Compare	Transmission CNC-Calc			Window * Help * = 🗗 ×
Per Drawing * Save Drawing ₩ Setup CNC-Calc File	● × × × 2 € ● × × * ● ■ ● /iew Modify		Center Two Radius Points 2	Rectangular Bolt Circular Bolt Hole Pattern Hole Pattern Pattern Tev	
CNC-Calc Miling Tutorial 1.cdd *					4 b x
Face Miling	1	N			
Comment: FACING					
Single Step Show Toolpath Parameters			• • • •		
Back New Export Editor					
	· · · · · · ·	· · · · · · · · ·		· · · · · · · · · · · · · · · · · · ·	
Bement Info - Line					
Start X: 62.500					
Start Y: 50.000 End X: -62.500 End Y: 50.000					
Angle: 180.000 Length: 125.000			· · · · · · ·	<u> </u>	/ 1 → 1 ↓ 1 1 1 1 1 1
			e » e » 1 » .		
	Machine Zero X:	Machine Zero Y:	Machine Zero Z: 0.0	m u	Part Diameter Se Y-Axis Substitution
Select the next eler	ment CNC contour		Licensed 1	to DEMO - NOT FOR RESALE - CIMCO Integr	ation X: -38.589 Y: 52.560 INS 12:49:20 ,;;

Click on the button **Parameters** in the CNC-Calc pane Face Milling. This will open the configuration dialog for setting the face milling parameters.

Enter the values into the dialogs as shown in the pictures below.

Depths Tab

Face Milling Parameters	×
Depths Strategy	
Cutter Diameter: 30.0 Depth Settings Start Depth: 2.0 End Depth: 0.0 Retract Height: 10.0	Roughing Roughing Stepdown: 2.0 Finish Finish Stepdown: 0.1 Finish Cuts: 1 Stock to Leave: 0.0
	OK Cancel

- 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.
- **Stock to Leave**: The amount of stock that is left at the end of the operation (after both roughing and finishing cuts).

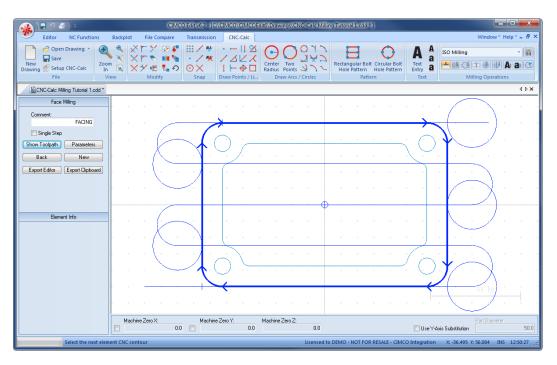
Strategy Tab

Strategy	
Cutting Method	
Zigzag	•
Move between Cuts	
High Speed Loops	•
Overlap Across:	Facing Angle:
15.0	0.0
Overlap Along:	Stepover:
30.0	30.0
Entry Distance:	
20.0	
Exit Distance:	
10.0	

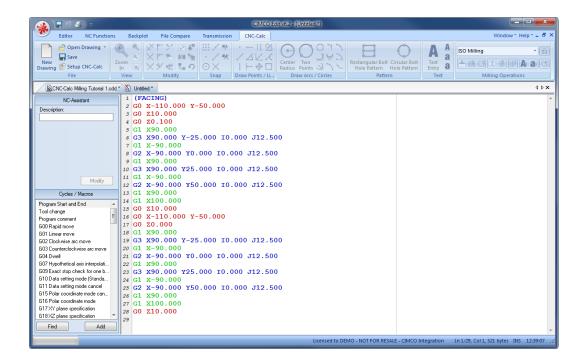
• **Cutting Method**: The method used to perform the face operation. It is possible to select *Zigzag*, *Climb*, or *Conventional*.

- 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.
- 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.
- **Stepover**: The distance between each of the parallel cuts of the facing operation.

After entering the values, close the parameters dialog with **OK**. To show the generated toolpath click on **Show Toolpath** button in the Face Milling pane.



Now click on the button **Export Editor** to export the generated NC code to a new window in the Editor. The following screen should now be displayed.



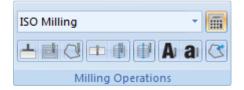
See section "Inserting a Tool with Feed and Speed Calculator" for information on how to insert a tool using the Feed and Speed Calculator.

10.2.2. 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 for milling operations, select the icon **Calculator** in the *Milling Operations* toolbar.



Fill in the following values into the Feed and Speed Calculator pane:

- **Tool** #: Number assigned to the tool, let's say that the face mill have a tool number of 1.
- Diameter (D): Diameter is 30 mm.
- **# Flutes (Z)**: The number of flutes is 5.
- Feed per tooth (Sz): In this example it is set to 0.08 mm.
- Cutting Speed (V): Is set to 190.

The fields are linked together, so as entries are made in the *Cutting Speed* field, the fields *RPM* and *Feedrate* will be automatically updated.

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

Change the RPM to 2000 rpm and the feedrate to 800 mm/min.

Click on the button **Export Clipboard** on the Feed and Speed Calculator pane. 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 the icon **Paste** from the *Edit* toolbar in the Editor tab.

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:	1
190	
BPM (n): 2015	
2010	
Feedrate (F) in mm/min: 806.385	1
000.003	
💿 Metric 🛛 🔘 Inches	
Clear	
Export Editor Export Clipbo	bard

(A)	CIMCO Edit v6.2 - [Untit	ied *]	
Editor NC Functions	Backplot File Compare Transmission CNC-Calc		Window - Help - 🗗 🗙
		Redo 🛞 Insert File 😪 Replace	Linc/Block Find
CNC-Calc Milling Tutorial 1.cdd	Vitiled *		4 Þ ×
NC-Assistant Description Cycles / Macris Cycles / Macris Progen Start and End Tool change Progen comment G00 Regid move G01 Linear move G02 Dockwise ac move G03 Counterclockwise ar move G03 Counterclockwise ar move G04 Durel G05 Phytophetical asis interpolati G08 Exact stop-check for one b G10 Data setting mode cancel G15 Polar coordinate mode G17XY plane specification G18XZ plane specification G18XZ plane specification G18XZ plane specification	1 T1 M06 \$2000 F8000.000 2 (FACING) 3 G0 X-110.000 Y-50.000 4 G0 Z10.000 6 G1 X90.000 7 G3 X90.000 Y-25.000 I0.000 J12.500 8 G1 X-90.000 9 G2 X-90.000 Y0.000 I0.000 J12.500 11 G3 X90.000 Y25.000 I0.000 J12.500 12 G1 X-90.000 13 G2 X-90.000 Y50.000 I0.000 J12.500 14 G1 X90.000 15 G1 X100.000 16 G0 Z10.000 17 G0 X-110.000 Y-50.000 18 G0 Z0.000 19 G1 X90.000 20 G3 X90.000 Y-25.000 I0.000 J12.500 21 G1 X-90.000 22 G2 X-90.000 Y0.000 I0.000 J12.500 23 G1 X90.000 24 G3 X90.000 Y25.000 I0.000 J12.500 25 G1 X-90.000 26 G2 X-90.000 Y0.000 I0.000 J12.500 27 G1 X90.000 26 G2 X-90.000 Y0.000 I0.000 J12.500 27 G1 X90.000 26 G2 X-90.000 Y0.000 I0.000 J12.500 27 G1 X90.000 26 G2 X-90.000 Y0.000 I0.000 J12.500 27 G1 X90.000		
Find Add	29 G0 Z10.000 30		
Calculate Feed a	nd Speed	Licensed to DEMO - NOT FOR RESALE - CIMCO Integration	Ln 2/30, Col 1, 544 bytes INS 13:10:00 ,;

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

10.3. Contour Milling

CNC-Calc v7 can generate contour milling toolpaths - with and without radius compensation.

10.3.1. Creation of Contour Toolpaths

To begin the creation of an NC program for the contour operation, select **Contour Milling** to generate a CNC-toolpath for contour milling (ensure that **ISO Milling** is selected in the field *File Type*).

ISO Milling 🔹	
🛨 📑 🔁 🖿 🌒 🖶 A) a)	C
Milling Operations	

Write the text CONTOUR in the *Comment* field of the CNC-Calc pane Contour Milling. 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.

Move the pointer over the outlining contour of the drawing. This highlights the contour element; the arrows indicate the direction the tool will travel. Click on the part of the element that makes the contour direction clockwise like in the picture below.

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

🔬 🗐 🤊 🖉) 🕫 👘	CIM	CO Edit v6.2 - [C:\CIMCO\CIMCOE	dit6\Drawings\CNC-Calc Millin	ng Tutorial 1.cdd *]	
Editor NC Functions	Backplot File Compare	Transmission CNC-Calc			Window * Help * 🗕 🗗 🗙
P Open Drawing ★ Save Drawing ★ Save Setup CNC-Calc File		$\begin{array}{c c} \vdots & \swarrow & \psi \\ & & & & & \\ & & & \\ & & & & & \\ & & & & \\ & & & & & \\ & & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\ & & & & \\$	Center Two Radius Points 2	Rectangular Bolt Circular Bolt Hole Pattern Pattern Text	ISO Milling - III
CNC-Calc Miling Tutorial 1.cdd *					4 Þ x
Contour Milling					<u> </u>
Comment: CONTOUR		\mathcal{C}			
Single Step		\mathbf{V}	· · · · · · · · · · · · · · · · · · ·	$ \cdot \cdot$	
Select Contour					
Select Entry Point	• • • • • • •	· · · · · · · ·			· · · · · · · · · ·
Show Toolpath Parameters Back New	· · · · · ·		• • • • • • •		· · · · · · · · · · ·
Export Editor Export Clipboard					
Bement Info					
				· · · · · · · · · · · · · · · · · · ·	
		\sim			↓ 45.00
	Machine Zero X:	Machine Zero Y:	Machine Zero Z: 0.0	🗖 Use Y-4	Axis Substitution Part Diameter 50.0
Select the next elem	ment in the current contour, or	the start element in a new contour	Licensed to	DEMO - NOT FOR RESALE - CIMCO Integration	X: -70.302 Y: 52.339 INS 12:51:59 ,;;

Click on the button **Parameters** in the CNC-Calc pane Contour Milling. This will open the configuration dialog for setting the contour milling parameters.

Enter the values into the Parameters dialogs as shown in the pictures below.

General Tab

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

Contour Milling Parameters	×
General Side Cuts Depth Cuts Lead	d In/Out
Cutter Diameter: 10.0	
Depth Settings	
Retract Height:	Start Depth:
10.0	0.0
Safe Distance:	End Depth:
2.0	-5.0
Stock to Leave Stock to Leave XY: 0.0	Stock to Leave Z: 0.0
Compensation Apply on Roughing Side Cuts Compensation Type Computer Work Side Left	•
	OK Cancel

- 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.
- **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 option 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

Configure the cuts taken in the XY direction.

ontour Milling Parameters	X
General Side Cuts Depth Cuts Lead	d In/Out
✓ Use Side Cuts	
Roughing Passes Number of Passes:	Finish Passes Number of Passes:
2	1
Spacing:	Spacing:
2.0	0.1
Machine finish passes at	All Depths
Finish Overlap Overlap Distance:	
	10.0
	OK Cancel

- Use Side Cuts: If this option 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 Passes): The number of roughing side cuts in the operation.

- **Spacing** (Roughing Passes): If more than one roughing pass is taken, this is the distance between them.
- Number of Passes (Finish Passes): The number of finishing side cuts in the operation.
- Spacing (Finish Passes): The distance of each finishing pass.
- **Final Depth**: If this radio button is checked, the finishing passes will only be taken at the final depth.
- All Depths: If this radio button 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.

Depth Cuts Tab

Configure the cuts taken in the Z direction.

Contour Milling Parameters	×
General Side Cuts Depth Cuts Lead	In/Out
Use Depth Cuts	
Roughing Passes Max Roughing Steps:	Finish Passes Number of Cuts:
2.0	
Use Even Depth Cuts	Steps:
	0.1
Helix Linearization	
Linearize Helix Movements	
Linearization Tolerance	0.05
Toolpath Linking	
By Depth	By Contour
	OK Cancel

- Use Depth Cuts: If this option 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 option 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: The number of finishing depth cuts in the operation.
- Steps: The distance of each finishing pass.
- Linearize Helix Movements: Some machines cannot make helix movements, and if this option 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.

Contour Milling Parameters		×
General Side Cuts Depth Cuts	Lead In/Ou	t
Vise Lead In/Out Parameters		
Lead In	n r	.ead Out
✓ Use Line Line Length: 5.0	->	✓ Use Line Line Length: 5.0
Perpendicular		Perpendicular
Tangent	<-	Tangent
IV Use Arc Radius:		Vuse Arc
5.0		5.0
Sweep:		Sweep:
90.0		90.0
Feedrates Use custom feedrates	Feedrates	
		OK Cancel

- Use Lead In/Out Parameters: Enables or disables the lead in and out.
- Use Line: Enables or disables the lead in/out lines.
- Line Length: The length of the lead in/out line.
- **Perpendicular**: If this option 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.
- Use custom feedrates: Check this option to enable using custom feedrates for the milling operation.
- Feedrates: Click this button to open a new window to enter custom values for *Cutting (XY), Helix/Ramp*, and *Plunging (Z)* feedrates.

Now, close the parameters dialog with **OK**. To show the generated toolpath click on **Show Toolpath** button in the Contour Milling pane.

🔬 🖬 🕫 🛋) 🕫		CIMCO E	dit v6.2 - [C:\CIMCO\CIMCOE	dit6\Drawings\CNC-Calo	: Milling Tutorial 1.cdd *]			x
Editor NC Functions	Backplot File		ransmission CNC-Calc				Window* Help	×
New Save Drawing Setup CNC-Calc File	🔍 X 🗡 '	s 💵 🔓		Center Two Radius Points Draw Arcs / Circles	Rectangular Bolt Circu Hole Pattern Hole Pattern	Daten Pattern Text	-	• 🔳
CNC-Calc Miling Tutorial 1.cdd *								4 Þ 🗙
Contour Milling								
Comment								
CONTOUR Single Step			$\ $					
Select Contour			∥≜⊖ ∕─			$\neg \bigcirc \mathbf{I}$		
Reverse Contour								
Select Entry Point			· · · ·					
Show Toolpath Parameters Back New					•			
Export Editor Export Clipboard								
Bement Info								
							• • •	
			((₹) ∪)			$= \cup J_{/}$		
	· · ·		6 p p d			1 D 4 4		1
	Machine Zero		Machine Zero Y:	Machine Zero Z: (.0	📃 Use Y-Axis Subs	Part Diameter	50.0
Select the next elem	ent in the current (contour, or the st	art element in a new contour	Licer	sed to DEMO - NOT FOR RESAL	E - CIMCO Integration X: -7	9.395 Y: 55.000 INS 1	2:52:12 ,;

Click on the button **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 of the file. Insert the text from the clipboard, either by *pressing* Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

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. See section "Inserting a Tool with Feed and Speed Calculator" for information on how to insert a tool using the Feed and Speed Calculator.

10.3.2. Inserting a Tool with Feed and Speed Calculator

Follow the steps from the previous operation for inserting a tool with the built-in calculator.

Instead of the values used there, use the following values into the Feed and Speed Calculator pane:

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

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

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

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

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

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

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 the **Find** \triangleleft icon from the *Find* toolbar under the Editor Tab.

Go to the start of the comment line and insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar.

F	eed and Speed Calculator	
Too	ol #:	
	2	
Dia	meter (D) in mm:	
	10	
# F	lutes (Z):	
	4	
Fee	ed per tooth (Sz) in mm:	
	0.06	
Cut	tting Speed (V) in mm/min:	
	172.7876	
RP	M (n):	
	5500	
Fee	edrate (F) in mm/min:	
	1320	
Metro	ric 💿 Inches	
	Clear	
Expo	ort Editor	ard

Editor NC Functions Badzplot File Compare Transmission CNC-Calc Window" Hdp" = 6 New Open Close Size Size Size File Iso Mailing Iso Mailing<	🕢 🗖 🧉 🗧 🔹	CIMCO Edit v6.2 - IC:\CIMCO\CIMCOEdit6\Sa	mples\CNC-Calc Milling Tutorial 2.NC]	
New Open Cites Save Are Print Global Description Description Other Are Print Global Description Other Are Print Global Description Previous Tool change Image: State State </th <th>100</th> <th>Backplot File Compare Transmission CNC-Calc</th> <th></th> <th>Window * Help * 🗕 🗗 🗙</th>	100	Backplot File Compare Transmission CNC-Calc		Window * Help * 🗕 🗗 🗙
NC.4sistart 22 G1 X-90.000 Y50.000 Y50	A A	e Print Global v Setup	Undo ≩ Append File C Redo € Insert File G Redo	Go to Line/Block Number
Normalasian 26 (c2 x -90.000 Y50.000 J12.500 Pecereption 27 (G1 X90.000 27 (G1 X90.000 26 (J1 X100.000 28 (G1 X100.000 29 (G1 X100.000 29 (G1 Z10.000 32 (G1 Z10.000 31 (CONTOUR) 32 (G0 Z10.000 32 (G1 Z2.000 33 (G0 Z1.000 33 (G0 Z1.000 75 (J1 Z-1.667 36 (J1 X-5.000 75 (J1 X-5.000 7 (G1 X-5.000 75 (J1 X-5.000 9 (G1 X-5.000 75 (J1 X-5.000 9 (G1 X-5.000 75 (J1 X-5.000 9 (G1 X-5.000 75 (J1 X-5.000 10 clange 76 (J1 X-37.500 10 clange 76 (J1 X-37.500 10 clange 76 (J1 X-37.500 10 clange 76 (J1 X-57.100 10 clange 76 (J1 X-57.100 10 clange 76 (J1 X-73.500 10 clange 76 (J1 X-73.500 10 clange 76 (J1 X-62.500 Y-57.100 10 clange 76 (J1 X-62.500 Y-57.100 10 clange 16 (J1 X-37.500 10 clange 16 (J1 X-37.500 10 clange 177 (J1 J1 J1 J1 J1 J1 J	CNC-Calc Milling Tutorial 1.cdd*	CNC-Calc Miling Tutorial 2.NC *		4 Þ ×
Mody 32 Go X0.000 Y62.100 33 GO Z10.000 33 GO Z2.000 34 GO Z2.000 35 G1 Z-1.667 35 G1 Z-1.667 36 G1 X-5.000 36 G1 Z-5.00 39 G2 X82.100 Y57.100 I5.000 J0.000 36 G1 X-62.500 39 G2 X82.100 Y37.500 I0.000 J-19.600 37 G3 X-0.000 Y57.100 I-19.600 J0.000 39 G2 X82.100 Y37.500 I0.000 J-19.600 36 G1 X-62.500 Y-57.100 I-19.600 J0.000 40 G1 Y-37.500 I0.000 J19.600 37 G2 X-62.500 Y-57.100 I-19.600 J0.000 42 G1 X-62.500 Y-57.100 I-19.600 J0.000 38 G2 X62.100 Y37.500 I0.000 J19.600 46 G1 Y37.500 39 G2 X62.100 Y37.500 I0.000 J19.600 46 G1 X0.000 Y57.100 G30 Caboring a cnowe 46 G1 X37.500 G30 Experiments 46 G1 X37.500 G30 Hapdimode Standa 47 G2 X1.500 Y55.100 I.500 J0.625 G11 Data setting mode caread 46 G1 X-62.500 Y-55.100 I.500 J0.602 G15 Polar condinate mode caread 50 G2 X60.100 Y37.500 I0.000 J-17.600 G16 Polar condinate mode caread 50 G2 X60.100 Y37.500 I0.000 J-17.600 G17X/ phare pacelication 36 G1 X-62.500 Y-55.100 36 G2 X-62.500 Y-55.100 I.500 J0.000 36 G1 X-62.500 Y-55.100 37 G2 X-62.500 Y-55.100 37 G2 X-62.500 Y-55.100		26 G2 X-90.000 Y50.000 I0.000 J12.500 27 G1 X90.000 28 G1 X100.000 29 G0 Z10.000 30 T2 M06 S5500 F1320.000		ŕ
Program Start and End 36 G1 X62.500 Tool charge 39 G2 X82.100 Y37.500 I0.000 J-19.600 Program Start and End 39 G2 X82.100 Y37.500 I0.000 J-19.600 G00 Rapid move G2 X62.500 Y-57.100 I-19.600 J0.000 G1 X62.500 G00 Linest move G3 G2 X82.100 Y-37.500 I0.000 J19.600 G2 X82.100 Y-37.500 I0.000 J19.600 G01 Linest move G3 C2 X-62.100 Y-37.500 I0.000 J19.600 G3 C2 X-62.500 Y-57.100 G02 Declowite a cmove G3 C2 X-62.500 Y-57.100 G2 X-62.500 Y-57.100 G02 Contractockwise ac move G3 C2 X-62.500 Y-57.100 G3 C2 X-62.500 Y-57.100 G02 Declowite ac move G3 C3 X-62.500 Y-57.100 G3 C3 X-000 J5 C4 G02 Declowite ac move G3 C3 X-000 Y57.100 G3 C3 X-000 J0.000 J-1.625 G3 Dia setting mode Carrele G3 X3.000 Y55.100 I1.500 J0.625 G3 C3 X0.100 Y37.500 I0.000 J-17.600 G15 Paid condrate mode G3 C2 X-60.100 V-37.500 I0.000 J-17.600 G3 G1 X-62.500 Y-55.100 G17 Mpohenication G3 C2 X-60.100 Y-37.500 I0.000 J-17.600 G3 G1 X-62.500 Y-55.100 G18 Zobarcostinate mode G3 C2 X-60.100 Y-37.500 I0.000 J17.600 G3 C2 X-60.100 Y-37.500 I0.000 J17.600		32 GO X0.000 Y62.100 33 GO Z10.000 34 GO Z2.000 35 GI Z-1.667 36 GI X-5.000		E
GBD Counterclockwise air cmove GBd Dwell 44 (1 Y37,500 45 G2 X-62.500 Y57.100 I19.600 J0.000 45 G2 X-62.500 Y57.100 I19.600 J0.000 47 G2 X1.500 Y56.100 I0.000 J-1.625 46 G1 X03.000 Y55.100 I1.500 J0.625 49 G1 X62.500 615 Polar condinate mode G15 Polar condinate mode G16 Polar condinate mode G16 Polar condinate mode G17XV plane specification G16 Polar condinate mode G1 X-737.500 G2 X60.100 Y37.500 I0.000 J-17.600 51 G1 X-62.500 Y-55.100 G2 G2 X62.500 Y-55.100 G2 G2 X-62.500 Y-55.100 G2 G2 X-62.500 Y-55.100	Tool change Program comment G00 Rapid move G01 Linear move	39 G2 X82.100 Y37.500 I0.000 J-19.600 40 G1 Y-37.500 I Intervention Intervent		
G11 Data setting mode cancel 49 G1 X62.500 G15 Folar condinate mode cancel 50 G2 X80.100 Y37.500 I0.000 J-17.600 G16 Folar condinate mode 50 G2 X80.100 Y37.500 I0.000 J-17.600 G17 X/r plane specification 50 G2 X80.100 Y-55.100 I-17.600 J0.000 G18/22 plane specification 50 G2 X62.500 Y-55.100 I-17.600 J0.000 G18/22 plane specification 50 G2 X-80.100 Y-37.500 I0.000 J17.600	G03 Counterclockwise arc move G04 Dwell G07 Hypothetical axis interpolati G09 Exact stop check for one b	44 G1 Y37.500 45 G2 X-62.500 Y57.100 I19.600 J0.000 46 G1 X0.000 Y57.100 47 G2 X1.500 Y56.100 I0.000 J-1.625		
	G11 Data setting mode cancel G15 Polar coordinate mode can G16 Polar coordinate mode G17 XY plane specification	49 G1 X62.500 50 G2 X80.100 Y37.500 I0.000 J-17.600 51 G1 Y-37.500 52 G2 X62.500 Y-55.100 I-17.600 J0.000		
	Find Add	54 G2 X-80.100 Y-37.500 I0.000 J17.600	Licensed to DEMO - NOT FOR RESALE - CIMCO Integration	Ln 30/137, Col 1, 2.847 bytes INS 13:31:07 .:

The NC program should now look like the following:

10.4. Pocket Milling

CNC-Calc v7 can generate pocket milling toolpaths.

10.4.1. Creation of Pocket Toolpaths

To start creating an NC program for the pocket operation, select the function **Pocket Milling** to generate a CNC-toolpath for pocket milling (ensure that **ISO Milling** is selected in the field *File Type*).

ISO Milling	
- 🛃 🖓 🖿 🌒 🖪 a	2
Milling Operations	

Write the text POCKET in the *Comment* field of the CNC-Calc pane Pocket Milling. 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 of the drawing. This will highlight the inner contour that will be used as boundary for the pocket operation.

🔬 🖬 9 🗉 🔍	CIMCO	Edit v6.2 - [C:\CIMCO\CIMCOE	dit6\Drawings\CNC-Calc Milling	Tutorial 1.cdd *]	
Editor NC Functions	Backplot File Compare	Transmission CNC-Calc			Window * Help * = 🗗 🗧
Open Drawing * Save Drawing * File			Center Two Radius Points 2 2 2 Draw Arcs / Circles	Rectangular Bolt Hole Pattern Pattern Te	
CNC-Calc Milling Tutorial 1.cdd *	CNC-Calc Milling Tutorial 2.NC *				4 Þ ×
Pocket Milling					
Comment: POCKET					
Single Step		\bigcup			$\mathbf{V} \subseteq \mathbf{V}$
Show Toolpath Parameters Back New Export Editor Export Clipboard					
Bement Info		· · · · · · · · · · · · · · · · · · ·			
		$\sum_{i=1}^{n}$		· · · · · · · · · · · · · · · · · · ·	$\sum_{i=1}^{n}$
					45.00
	Machine Zero X:	Machine Zero Y:	Machine Zero Z: 0.0	— L	Part Diameter Use Y-Axis Substitution 50.0
Select the starting	point of the CNC contour (use end	point snap)	Licensed to DEMO - N	IOT FOR RESALE - CIMCO Integration	X: 88.710 Y: 55.000 INS 14:49:34 ,

Click on the button **Parameters** in the CNC-Calc pane Pocket Milling. This will open the configuration dialog for setting the pocket milling parameters.

Enter the values into the Parameters dialogs as shown in the pictures below.

General Tab

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

Pocket Milling Parameters	— X—		
General 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	Stock to Leave Z:		
0.0	0.0		
Compensation Type			
Computer	•		
Milling Type			
Conventional	Olimb		
Feedrates			
Use custom feedrates	Feedrates		
	OK Cancel		

- Cutter 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.
- 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.
- Use custom feedrates: Check this option to enable using custom feedrates for the milling operation.
- Feedrates: Click this button to open a new window to enter custom values for *Cutting (XY), Helix/Ramp*, and *Plunging (Z)* feedrates.

Side Cuts Tab

Configure the cuts taken in the XY direction.

General Side Cuts Depth Cuts E	intry Strategy	Lead In/Out	
Roughing Passes:	Finish F	asses:	
Max Roughing Spacing:	Cuts:		
3.			1
	Spacing	j :	
			0.1
Finish Passes at			
At Final Depth	⊚ Ata	I Depths	
Finish Overlap Overlap Distance:			1.0
Roughing Smoothing			
Disabled			Max
		1 1	i I

- **Max Roughing Spacing**: The maximum side stepover used in the roughing of the part.
- Cuts: The number of finishing side cuts in the operation.
- **Spacing**: The distance of each finishing pass.
- At Final Depth: If this radio button is checked, the finishing passes will only be taken at the final depth.
- At All Depths: If this radio button 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

Configure the cuts taken in the Z direction.

General Side Cuts Depth Cuts En	try Strategy Lead In/Out
☑ Use Depth Cuts	
Roughing Passes:	Finish Passes:
Max Roughing Steps: 2.0	Number of Cuts:
Use Even Depth Cuts	Steps: 0.1
Toolpath Linking	
By Depth	By Pocket
	OK Cancel

- Use Depth Cuts: If this option 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 option 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: The number of finishing depth cuts in the operation.
- Steps: The distance of each finishing pass.

- **By Depth**: This is only used if multiple pockets are milled in the same operation. If selected, the cut on 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 Strategy Tab

This tab is used to configure how the tool cuts from one Z level to the next.

Helix Diameter: Minimum Helix Diameter:		Side Cuts Depth Cuts	Entry St	ategy	Lead In/O	ut	
Ramp Ramp Angle: Ramp Clearance: 5.0 3.0 Ramp Length: 20.0 Image: Ramp Clearance: Pelix Entry Pelix Parameters Helix Parameters Ramp Clearance: 1minum Helix Diameter: 20.0 Helix Diameter: Minimum Helix Diameter: 3.0 0.2 Helix Linearization Imearizet Helix Movements Linearization Tolerance Image:							
Ramp Parameters Ramp Clearance: Ramp Angle: S.0 Ramp Length: 20.0 Image: Ramp Clearance: Helix Entry Helix Parameters Helix Angle: Ramp Clearance: Image: S.0 Helix Diameter: Minimum Helix Diameter: Image: 3.0 Helix Linearization 0.2 Linearizet Helix Movements Linearization Tolerance		-					
Ramp Angle: Ramp Clearance: 5.0 3.0 Ramp Length: 20.0 Image: Ramp Clearance: Helix Parameters Helix Angle: Helix Angle: Ramp Clearance: 1mage: 5.0 1mage: Ramp Clearance: 1mage: 1mage: 1mag	\odot						
S.0 S				Ramp	Clearance:		
Ramp Length: 20.0 Helix Entry Helix Parameters Helix Angle: 5.0 2.0 Helix Diameter: 3.0 0.2 Helix Linearization Linearize Helix Movements Linearization Tolerance		[5.0				2.0
20.0 Helix Entry Helix Parameters Helix Angle: 5.0 2.0 Helix Diameter: 3.0 Clearance: 0.2 Helix Linearization Linearize Helix Movements Linearization Tolerance		Dere Levelle	5.0				5.0
Helix Entry Helix Parameters Helix Angle: 5.0 2.0 Helix Diameter: 3.0 0.2 Helix Linearization Linearize Helix Movements Linearization Tolerance		Ramp Length:	20.0				
Helix Parameters Ramp Clearance: Helix Angle: 5.0 5.0 2.0 Helix Diameter: Minimum Helix Diameter: 3.0 0.2 Helix Linearization 0.2 Linearize Helix Movements Linearization Tolerance			20.0				
3.0 0.2 Helix Linearization Linearize Helix Movements Linearization Tolerance			5.0	Ramp	Clearance:		2.0
Helix Linearization Interview Contents Linearization Tolerance		Helix Diameter:		Minimu	um Helix Dia	meter:	
Linearize Helix Movements Linearization Tolerance			3.0				0.2
0.01		Linearization					

- 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.

- Linearize Helix Movements: Some machines cannot make helix movements, and if this option 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.

Lead In/Out Tab

Configures the way the tool will approach the pocket at the start/end of the roughing, and for each finishing pass.

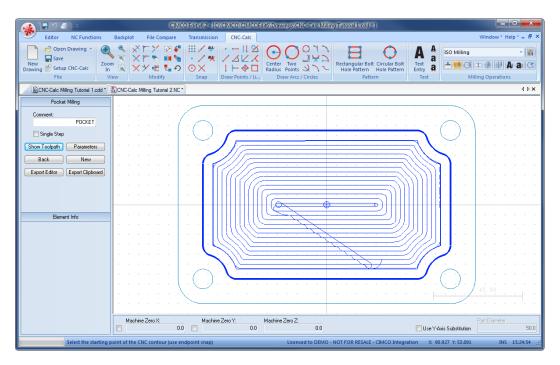
The use of lead in/out is optional, when the compensation is set to computer or none. It is however mandatory, when any compensation is performed by the controller.

Pocket Milling Parameters			×
General Side Cuts Depth Cuts	Entry Strate	egy Lead In/Out	
Use Lead In/Out Parameters			
Lead In		Lead Out	
Ivse Line Line Length:		☑ Use Line Line Length:	
5.0	->		5.0
Perpendicular		Perpendicular	
Tangent	<-	Tangent	
Use Arc		Use Arc	-
Radius:		Radius:	5.0
5.0 Sweep:		Sweep:	D.U
90.0		Sweep.	90.0
		ОК	Cancel

- Use Lead In/Out Parameters: Enables or disables the lead in and out.
- Use Line: Enables or disables the lead in/out lines.
- Line Length: The length of the lead in/out line.
- **Perpendicular**: If this option 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**. To show the generated toolpath click on **Show Toolpath** button in the Pocket Milling pane.



Click on the button **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 of the file. Insert the text from the clipboard, either by *pressing* Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

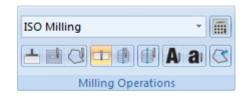
The NC program in the Editor now consists of three operations, and since we use the same tool for the contour and pocket operations we will not insert a tool before the pocket operation.

10.5. Drilling

CNC-Calc v7 can generate codes for drilling in either canned cycles or as longhand.

10.5.1. Generate a Drill Cycle

To start creating a NC-program for the drilling operation, select the function **Drill Holes** in the *Milling Operations* toolbar to generate a drill cycle (ensure that **ISO Milling** is selected in the field *File Type*).



Write the comment DRILLING in the *Comment* field of the CNC-Calc pane Drilling. 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.

Drilling				
Comment:	DRILLING			
📃 Use Selecti	ion Filter			
Filter	Drill Parameters			
Reorder Circ	Reorder Rect			
Back	New			
Export Editor	Export Clipboard			

Click on the button **Drill Parameters** to open the parameter dialog window shown below. For this drilling operation, please enter the parameters shown.

Drilling Parameters

Controller Type TNC 430	Output Type
	Conghand
	Threading
·/.U	Pitch:
	50.0
Use Retract	 Tapping Speed:
500.0	100.0
	Retract Speed:
	200.0
Pecking Retract:	200.0
3.0	Dwelling
Minimum Depth:	Use Upper Dwelling
2.5	1.0
First Feedrate Factor:	Use Lower Dwelling
0.75	2.0
	TNC 430 Reference Plane: 0.0 Depth: Incremental Absolute 7.0 Use Retract 500.0 Pecking Retract: 25 First Feedrate Factor: Comparison Com

- **Drilling Type**: This drop-down box is used to select the operation type. The posible parameters depend on the type selected.
- **Canned** (Output Type): Select this radio button to use a canned cycle. The canned cycle depends on the selected machine, and the possible parameters reflect this canned cycle.
- **Retract Plane**: The retract plane is the height that the tool is moved to before it traverses between holes.
- **Reference Plane**: This is the height of the material. For some machines like Maho, this is also the height that the operation is calculated around.
- **Safe Distance**: The safe distance is the distance above the reference plane where all moves toggle between feed and rapid.
- **Depth**: This field is used to enter the final depth of the operation.
- Use Plunging: This radio button is used to indicate if plunging moves should be performed with the entered plunging feedrate.
- **First Depth**: This field is used to enter the first depth for a pecking operation. The following pecks will be calculated based on degression and minimum depth.

Notice that in this example it makes no difference if **Incremental** or **Absolute** is selected for *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:

- 1. Select each hole location with the cursor. In order to get the correct hole center for circles and arcs, the **Snap to Center Points** function should be used.
- 2. Select the actual circle or arc. This will create a new hole location at the center 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.

Click on the button **Filter** in the CNC-Calc pane Drilling. 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. Click **OK** after entering the values shown.

Selection Filter	
🔲 All Points	
Circles All Circles	Arcs All Arcs
🔘 No Circles	No Arcs
Oircles in Range	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 option **Use Selection Filter** in the left hand pane, and then make a window selection that includes the entire drawing.

🔬 🖬 🤊 🐔) =	CIMO	O Edit v6.2 - [C:\CIMCO\CIMCOE	lit6\Drawings\CNC-Calc Milli	ng Tutorial 1.cdd *]	
Editor NC Functions	Backplot File Compare	Transmission CNC-Calc			Window * Help * = 🗗
Open Drawing +			$\Theta O O O O O O O O O O O O O O O O O O O$		A A ISO Milling
New Drawing Setup CNC-Calc In		O X I ⊢ ↓ □ Snap Draw Points / Li	Center Two Radius Points 2 2 2 Draw Arcs / Circles	Rectangular Bolt Circular Bolt Hole Pattern Hole Pattern I Pattern	Text Entry a Text Milling Operations
CNC-Calc Miling Tutorial 1.cdd *	CNC-Calc Milling Tutorial 2.NC	•]			4 Þ
Drilling					
Comment					
DRILLING					· · · · · · · · · · · · · · ·
Use Selection Filter					
Filter Drill Parameters		a a a a a a a a			
Reorder Circ Reorder Rect					
Back New					
Export Editor Export Clipboard					
Element Info					
					45.00
		× · · · · · · · ·			
		. ~		·····	
	Machine Zero X:		Machine Zero Z: 0.0		Part Diameter.
Calast as in director					
Select or indicate ne	ext hole's center point. Selection	is possible from circles and points	Licensed to DEMC) - NOT FOR RESALE - CIMCO Integrati	ion X: 94.919 Y: 53.004 INS 14:13:52

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

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

Click on the button **Export Clipboard**. The drilling 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 of the file. Insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

📣 🖬 🤊 🖨 🔹	CIMCO Edit v6.2 - [Ci\CIMCO\CIMCOEdit6\Samples\CNC-Calc Milling Tutorial 2.NC]	- • ×
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window	v≚ Help≚ = 🗗 ×
New Open Close Save Sav File		-
CNC-Calc Miling Tutorial 1.cd	CNC-Calc Miling Tutorial 2.NC *	4 Þ ×
NC-Assistant Description:	975 G2 X57.400 Y-20.655 Z-4.000 II6.845 J-5.100 976 G1 X57.400 Y20.655 Z-4.000 J5.100 J16.845 977 G2 X45.655 X32.400 Z-4.000 IS.100 J16.845 978 G1 X-45.655 Y32.400 Z-4.000 IS.100 J16.845 978 G2 X-57.400 Y20.655 Z-4.000 I-16.845 J5.100 979 G2 X-57.400 Y-20.655 Z-4.000 I-5.100 J-6.845 980 G1 X-57.400 Y-20.655 Z-4.000 I-5.100 J-16.845 982 G2 X-16.13 Y-32.400 Z-4.000 I-5.100 J-16.845 982 G2 X21.613 Y-32.400 Z-4.000 I J16.845 984 G3 X21.763 Y-32.400 Z-4.000 I J16.845 984 G3 X21.763 Y-32.500 Z-4.000 I J16.845 984 G3 X21.763 Y-32.500 Z-4.000 I J16.845	•
Cycles / Macros Program Start and End Tool change Program comment G00 Rapid move	966 G2 X57.500 Y-20.729 Z-4.000 I16.771 J-5.000 967 G1 X57.500 Y20.729 Z-4.000 988 G2 X45.729 Y32.500 Z-4.000 J5.000 J16.771 989 G1 X-45.729 Y32.500 Z-4.000 990 G2 X-57.500 Y20.729 Z-4.000 I-16.771 J5.000 991 G1 X-57.500 Y-20.729 Z-4.000	
G01 Linear move G02 Clockwise arc move G03 Counterclockwise arc move G04 Dwell G07 Hwoothetical axis interpolati	992 G2 X-45.729 Y-32.500 Z-4.000 I-5.000 J-16.771 993 G1 X22.763 Y-32.500 Z-4.000 94 G3 X27.763 Y-27.500 Z-4.000 I0.000 J5.000 955 G0 X27.763 Y-27.500 Z10.000	
GU/ Hypothetical assi interpolati G09 Exact top check for one b G10 Data setting mode (Standa G11 Data setting mode cancel G15 Polar coordinate mode G17X/ plane specification G18XZ plane specification T18XZ plane specification	996 [DRILLING] 997 GO X62.500 Y37.500 998 GO Z10.000 999 G83 X62.500 Y37.500 Z-7.000 R2.000 Q1.000 F200.000 1000 X-62.500 Y37.500 1011 X-62.500 Y-37.500 1022 X62.500 Y-37.500	
Find Add	1003 G80 1004	
Select elements to	delete (For multiple elements use window selection) Licensed to DEMO - NOT FOR RESALE - CIMCO Integration Ln 1.004/1.004, Col 1, 36.247 bytes	INS 15:35:31 ,

The NC program should look like the following.

Since the feedrate for the operation is defined in the canned cycle, we will enter manually the tool change. Write the following line just before the DRILLING comment:

T3 M06 S1200

This will assign the tool no. 3 with a spindle speed of 1200 rpm to the drilling operation.

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

11. Tutorial Mill 3 - Backplot in the Editor

One of the advantages of running CNC-Calc inside CIMCO Edit v7 is that the editor can be used to manipulate and backplot the NC programs generated in CNC-Calc.

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.

In the following, we will setup the backplot and verify the program generated by the previous tutorial.

11.1. Before You Start

The first thing to do before proceeding with the simulation is to set the menu parameters. Start CIMCO Edit v7 and verify that the Backplot tab is available in the Ribbon. If not, open the editor configuration by clicking the **Global Setup** ficon in the *Editor* tab. Go to the Plugins section in the left tree of the configuration window. Now ensure that the option **Disable Backplot** is unchecked. You must restart the program to activate the Backplot tab.

Setup: Plugins		? ×
General Editor Printing File types Colors Block numbers Load/Save File compare Machine Backplot CNC-Calc Multi Channel Tool List Other Global Colors External Commands Mazatrol Viewer Plugins	Plugins Select plugins Disable Backplot Isable NC-Base Disable advanced simulation Isable DNC-Max client Disable DNC/Serial communication Disable CNC-Calc Disable File compare Disable Mazatrol Viewer Disable advanced NC-Functions Configuration password Configuration password: Macros/Machine configuration path:	
	Help Default Cancel	ОК

Enabling the Bacplot tab.

11.2. Open an Existing NC Program

To open a NC Program, click on the icon rightarrow **Open** in the *File* toolbar under the Editor tab. This will open an Open File dialog where you can select the file you want to open.

🎋 Open			×
Look in: 🌗 Samples		- 🕝 🤌 📂 🛙	•
Name		Date modified	*
MillPlus		25-07-2012 10:	17
📗 Multi Channel		25-07-2012 10:	17
🌗 Siemens 840D		25-07-2012 10:	17 _
🛯 🛯 Tests		26-07-2012 10:	46 ≡
CNC-Calc Milling Tutorial	2.NC	25-07-2012 16:	24 👻
·			P.
File name: CNC-Calc Milling 1	utorial 2.NC		Open
Files of type: ISO Milling Files (*	NC;*.NCL;*.ISO;*.N	NCF) 🔻 C	ancel
🔲 Open as read-o	nly		

Please select the NC file from the last tutorial (CNC-Calc Milling Tutorial 2.NC) and click **Open**.

You should now see the NC program from the CNC-Calc Tutorial Mill 2 displayed.

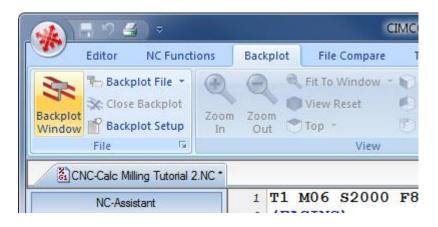
🔬 🗄 🤊 🖆 👘	CIMCO Edit v6.2 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc Milling Tutorial 2.NC]	
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window* Hel	p* = 6 ×
New Open Close Save Save File	Iso Milling Iso	ige
CNC-Calc Milling Tutorial 2.NC		∢≬×
NC-Assistant Description:	1 T1 M06 S2000 F8000.000 2 (FACING) 3 G0 X-110.000 Y-50.000 4 G0 Z10.000 5 G0 Z0.100	
Modify	6 (c1 X90.000 7 (c3 X90.000 8 (c1 X=90.000 9 (c2 X=90.000 Y0.000 I0.000 J12.500 10 (c1 X90.000 11 (c3 X90.000 Y25.000 I0.000 J12.500 12 (c1 X=90.000	
Cycles / Macros	13 G2 X-90.000 Y50.000 I0.000 J12.500	
Program Statt and End > Tool change Program comment G00 Rapid move E G01 Linear move E G02 Cocketwise act move E G03 Counterclockwise act move E G04 Cockwise act atops check for one b E G10 Data setting mode cancel E G15 Polar coordinate mode E G15 Polar coordinate mode E G15 Polar coordinate mode T G15 Polar coordinate mode T	14 (31 x30.000 15 (31 x100.000 16 (30 210.000 17 (30 x-110.000 y-50.000 18 (31 x30.000 20 (33 x30.000 y-25.000 10.000 J12.500 21 (31 x-90.000 22 (32 x-90.000 y0.000 10.000 J12.500 23 (31 x30.000 24 (33 x30.000 y25.000 10.000 J12.500 25 (31 x-90.000 y50.000 10.000 J12.500 26 (32 x-90.000 y50.000 10.000 J12.500 27 (31 x30.000 28 (31 x100.000 29 (30 210.000 20 (31 x100.000 20 (31 x100.000 20 (31 x100.000 21 (31 x30.000 23 (31 x100.000 24 (33 x30.000 25 (31 x100.000 26 (31 x100.000 27 (31 x30.000 28 (31 x100.000 29 (30 210.000 30 72 M06 s5500 F1320.000	·
	Licensed to DEMO - NOT FOR RESALE - CIMCO Integration Ln 1/1.005, Col 1, 36.260 bytes INS	10:42:42 ,

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

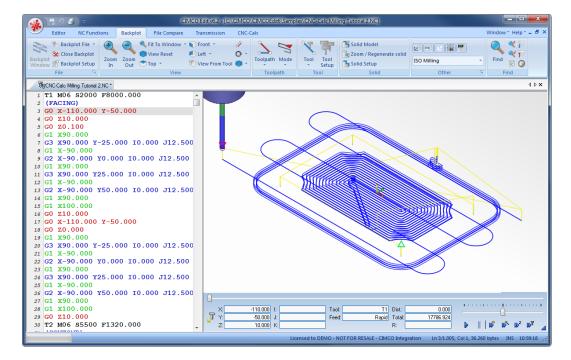
11.3. The First Backplot

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

Click on the **Backplot** tab at the top of the Ribbon bar. Then click on the icon \gg **Backplot Window** to backplot the NC program opened in the current window.



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.

11.3.1. Backplot Tool Setup

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

Click on the icon **Tool Setup** in the Backplot *Tool* toolbar. This will open the Tool Setup window.

То	ol Setu	р							?	x
		T1 - (Default) End Mill Bull 4MM R1		Type: End Mill Bull	•	Diameter:	4	Corner rac		Ty
		T2 - (Default) End Mill Bull 4MM R1								
		T3 - (Default) End Mill Bull 4MM R1								
			Description		Туре			D		L
			End Mill Flat 2MM		End N	fill Flat		2	6	0
			End Mill Ball 2MM		End N	fill Ball		2	6	0
			End Mill Bull 2MM	R0.5	End N	fill Bull		2	6	0
			End Mill Flat 3MM		End N	fill Flat		3	6	0
			End Mill Ball 3MM		End N	fill Ball		3	6	0
			End Mill Bull 3MM	R0.8	End N	fill Bull		3	6	0
			End Mill Flat 4MM		End N	fill Flat		4	6	0 .
		H	Delete	Modify	Add			Assign to) 'T1'	
	🗸 Save	tool information					Canc	el 🗌	OK	

When the Backplot parsed the NC program, it detected that three tools were being used.

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

In order to backplot the program correctly, we need to define these three tools. By changing the type and the diameter of the tools, we can create the correct setup. If you want to change tool color or define tool lengths, click on the **Tool Setup** \boxed{Im} icon at the right of the selected tool.

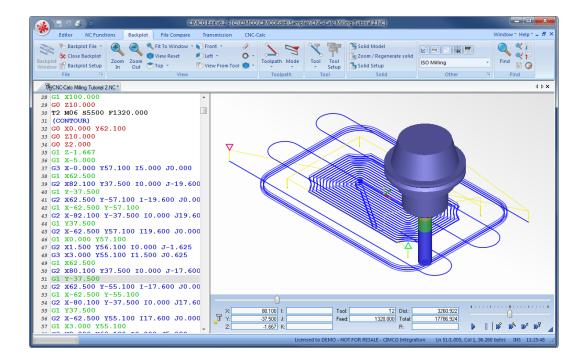
Tool Setup				? ×
Face mill 30MM				
T2 End Mill Flat 10MM				
T3 Drill 10MM	Type: Drill	Diameter:	0	Tr
	Description	Туре	D	L A
	End Mill Flat 2MM	End Mill Flat	2	60
	End Mill Ball 2MM	End Mill Ball	2	60
	End Mill Bull 2MM R0.5	End Mill Bull	2	60
	End Mill Flat 3MM	End Mill Flat	3	60
	End Mill Ball 3MM	End Mill Ball	3	60
	End Mill Bull 3MM R0.8	End Mill Bull	3	60
	End Mill Flat 4MM	End Mill Flat	4	60 🚽
	Delete Modify	Add	Assign to	'T3'
Save tool information		Ca	ancel	ОК

Please select and enter the values for the three tools shown on the picture below.

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

Now exit the tool configuration dialog with **OK**.

After the tools have been configured, the screen should look like the one below. Try to find the tool changes (M06) and verify that the tool does indeed change when a tool change is encountered in the NC program.



11.3.2. Backplot Stock Setup

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

Click on the icon **Solid Setup** in the Backplot *Solid* toolbar. This will open the Solid Setup window.

Solid Setup				Į	? ×							
- Stock Dimensi	ions											
	Min X: -110 🚔	Min Y: -57.1 🚔	Min Z: -7.1	09 🚖 📃	+5 %							
	Max X: 102.5 🜩	Max Y: 62.1 🚔	Max Z:	2	Round							
	+5% .5%	+5% .5%	+5 %		Reset							
Dimensions:	212.5	119.2		9.09								
🔲 Use cylind	Use cylindrical stock											
Options	Options											
📃 Use a diff	erent color for each	tool										
📃 Do not sh	iow tool setup when	all tools are defined										
📃 Smooth s	urface											
Stop if too	ol collision detected (rapid through stock)										
	tting with tool should											
Stop if cu	- tting with tool shaft											
🔲 Stop if too	ol holder collision det	ected										
Low Quality (Fast)	Medium Quality		High Qua	ality (Slow)							
	· · ·	<u> </u>	1	1	1							
Tool Library			Ca	ancel	ОК							

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.

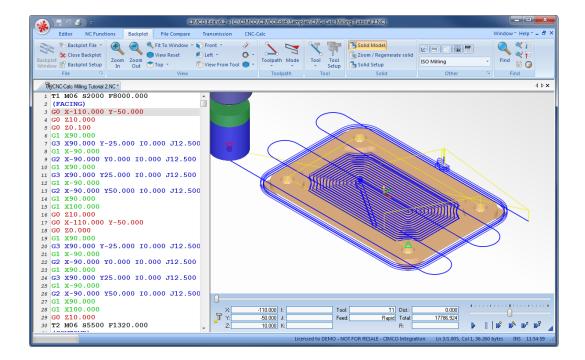
From the facing operation, we know that the top of the stock should be Z = 2.00 mm. From the drawing, we know that the values of the corners are (-75.00, -50.00) and (75.00, 50.00).

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 below. You can also adjust the quality of the solid by moving the slider at the bottom of the dialog.

Solid Setup							? <mark>x</mark>					
Stock Dimensi	ions											
	Min X:	-77 🚔	Min Y:	-52 ≑	Min Z:	-5 ≑	+5 %					
	Max X:	77 🔷	Max Y:	52 ≑	Max Z:	2 🌲	Round					
	+5 %	.5%	+5 %	-5%	+5 %	-5%	Reset					
Dimensions:		154		104		7						
🔲 Use cyline	Use cylindrical stock											
Options	Options											
📃 Use a diff	erent colo	for each I	tool									
📃 Do not sh	iow tool se	tup when	all tools are	e defined								
📃 Smooth s	urface											
🔲 Stop if too	ol collision	detected (rapid throu	gh stock)								
📃 Stop if cu	tting with t	ool should	er									
📃 Stop if cu	tting with t	ool shaft										
🔲 Stop if too	ol holder co	ollision det	ected									
Low Quality (Fast)		Mediu	m Quality		Hig	gh Quality (Slow)					
· · · · · · · · · · · · · · · · · · ·	1	1		1	1		<u> </u>					
Tool Library						Cancel	ОК					

Please enter these values and exit the dialog with **OK**.

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.



12. Tutorial Mill 4 - Milling of Letters

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

12.1. Open a New Drawing

To make a new drawing you must click on the icon **New Drawing** in the *File* toolbar under the CNC-Calc tab. When the new drawing is opened the following window should now be displayed:

🔬 🗄 🤊 🖉) 🕫								. 1	CIMC	O Edit	v7.0 -	(Untit	led Dra	wing)													l			×
Editor NC Functions	Back	plot	File	Comp	are	Tran	smissio	n	CNC-C	alc																	Wi	indow	/* He	lp * -	. 8
📄 💣 Open Drawing 👻 🤮	•	×	r>	1 0	F	,				12				0.	17				- (3		A	A	IS	D Mill	ing				+	
New Setup CNC-Calc		X					<u>/ **</u>	14				nter	Two	2	¥ -)	Recta	inguli	ar Bolt	Circu	ilar Bo	olt		a	L P	100	Q			A	a	C
	n 🔍 View	X		l⊑ ⁼∎ odify	+)	() Si	nap	Drav		🏞 🖵 ts / Li			oints		<u>ک</u> کر 5	Ho	le Pat	tern Patte		Patte	rn	Entry Te							ations		
Untitled Drawing							-																	<u> </u>		_	_	-			4 Þ:
CNC-Calc						-							-		- T		-					-	-			_				-	_
	- ·	*			• •		*		*	*	, ,	•	*	*	-	•		*		•	*			4	*	*	*		•		
	1	•			• •	•		• •	•	*	•	•	•	•	•	*	•			•	•		•	4	•	*	•		•		
	1	*	• •		• •	, ,	,		*			•	*	•	• •	*					*	•		4	*	•			•	, ,	
	1		•		• •	•		• •	•		• ·			•	•	*	•	•			*	*		*	*	*	•				
	1.1								-	-		-	-	-	•	-				-	-			-	-	-					
	1 ·	•			• •	•		• •	•	*		•	*	•	·			,		•	*		•		•	^	*		•		
	1	•										*				•				*				4	•	*					
	1	*	•••		• •						• •	•	*	•	.	*		•		•	*	•	• •	4	*	•	•		•	•	
D																															
Element Info																															
															.																
	I								-	-					.																
	I																														
	1.															4													4		
	1.														.							•	, .				,45	.00		· - r	
															.															, ,	
									-													-									_
		Machir	ne Zero	X:	0.0		Mach	ne Zero	Y:	0.0		hine Z	ero Z:		0.0							m	Jse Y-	Axis S	ubstitu	ution					50
					5.0					5.0						ensed 1													INS		

Make sure that file type **ISO Milling** is selected in the **File Type** field in the *Operations* toolbar. Text machining is not suported for turning operations.



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 CNC-Calc** and then **Global Colors** from the tree menu. For this tutorial we have chosen to use blue as our drawing color and white for the background.

12.2. Simple Letters Milling

In this part of the tutorial we will 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 (no accents, dieresis, tildes, etc.), numbers, and some special characters can be entered.

12.2.1. Drawing the Text

To begin the creation of the text that will be used for machining, click on the icon **A** Text Entry in the CNC-Calc *Text* toolbar.

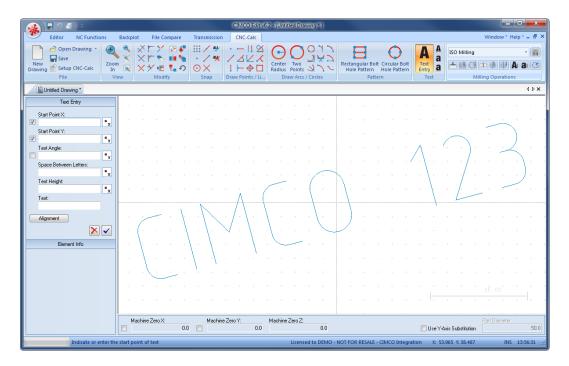
The Text Entry pane appears to the left to define the text that will be drawn.

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 (relative to the horizontal axis), distance between letters, and letter height. Fill in the fields with the values shown below.

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	
$\mathbf{\times}$	✓

When you are done entering text and values, click on the blue check mark button \checkmark at the bottom right of the dialog to accept the insertion. This is important, since otherwise the text will disappear once you start doing other things.

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.

12.2.2. Milling of Letters

To begin the creation of an NC program for the letters milling operation, ensure that **ISO Milling** is selected as programming format for the NC program in the *File Type* drop-down list in the *Operations* toolbar.

Then select the feature Letter Milling by clicking on the icon **A**Mill Letters in the *Milling Operations* toolbar to generate a CNC-toolpath for milling letters.

ISO Milling	
占 🖻 🖓 🖿 🌒 🕑 🗛 ai	3
Milling Operations	

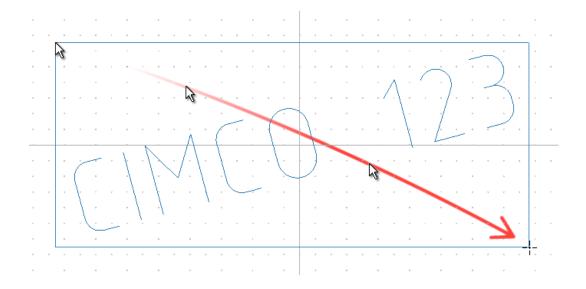
The CNC-Calc Letter Milling pane will appear on the left side of the screen as shown. Write the comment LETTER MILLING 'CIMCO 123' in the *Comment* field. It is a good practice to include the text to machine in the comment for the letter milling operations.

Letter	Milling
Comment: TTER MILLIN	G 'CIMCO 123'
	Parameters
Back	New
Export Editor	Export Clipboard

Now click on the **Parameters** button and enter the values shown below.

Letter Milling Parameters	
Depth Settings Retract Height:	
10.0	
Safe Distance:	
2.0	
Start Depth:	
0.0	
End Depth:	
-1.0	
Stepdown Z:	
1.0	
Cancel OK	

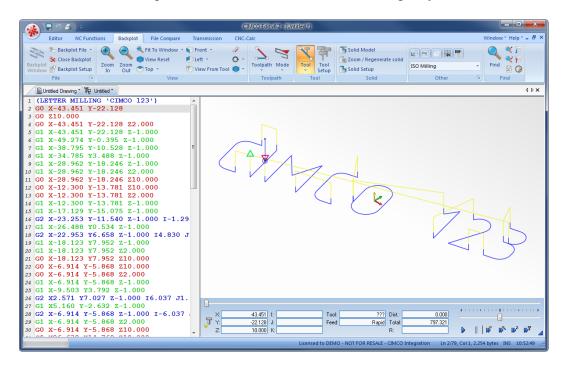
Now, use window selection to select the text you want to mill. On the drawing, left-click one corner, hold down the mouse button, and drag diagonally. When the desired text is framed, release the button. The frame disappears and the text will have turned yellow to indicate that it has been selected.



Next click on the **Export Editor** button, and the NC codes for machining the text will be shown in a new window 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 at the top of the Ribbon and then on the icon \gg **Backplot Window**.

🔬 🔒 9 🗉 🕫		CIMCO Edit v6.2 - [Un	titled *]				×
Editor NC Functions	Backplot File Compare Transmission CN	NC-Calc				Window* Help* = 6	8 ×
Window Backplot Setup In		* Toolpath Mode	Tool Tool Setup	Solid Model Com / Regenerate solid Solid Setup	ISO Milling	Find D	
File	View	Toolpath	Tool	Solid	Other	Find	
Untitled Drawing * 🗿 Untitled						4 ۵	×
NC-Assistant Description:	i (LETTER MILLING 'CIMCO 123' 2 G0 X-43.451 Y-22.128 3 G0 Z10.000 4 G0 X-43.451 Y-22.128 Z2.000 5 G1 X-43.451 Y-22.128 Z-1.00 6 G1 X-49.274 Y-0.395 Z-1.000 7 G1 X-38.795 Y-10.528 Z-1.00 9 G1 X-28.962 Y-18.246 Z-1.00 10 G1 X-28.962 Y-18.246 Z2.000 11 G0 X-28.962 Y-18.246 Z2.000))0)0)0)0					
Cycles / Macros Program Stat and End Tool change Program comment 600 Rapid move 602 Cookwise arc move 603 Cookwise arc move 603 Courtectockwise arc move 604 Dwell 604 Physothetical asis interpolati 609 Exact top check for one b))0)0 I-1.294 J4					
G10 Data setting mode (Standa G11 Data setting mode carnel G15 Polar coordinate mode carn G16 Polar coordinate mode G17XY plane specification G18×Z plane specification Find Add	24 G1 X-6.914 Y-5.868 Z-1.000 25 G1 X-9.503 Y3.792 Z-1.000 26 G2 X2.571 Y7.027 Z-1.000 T4 27 G1 X5.160 Y-2.632 Z-1.000 28 G2 X-6.914 Y-5.868 Z-1.000 29 G1 X-6.914 Y-5.868 Z2.000 30 G0 X-6.914 Y-5.868 Z0.000			10 - NOTFOR RESALE - CIMCO		2.254 bytes INS 10:40:33	



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.

At the bottom right of the backplot window, start the simulation by clicking the **Start** / **stop simulation** icon.

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

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*.

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** functions in CIMCO Edit v7.

12.3. TrueType Letters Milling

In this part of the 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 special characters can be entered.

12.3.1. Drawing the Text

To begin the creation of the TrueType text that will be used for machining, click on the icon **a TrueType Text Entry** in the CNC-Calc *Text* toolbar.

The Text Entry pane appears to the left to define the text that will be drawn.

In the input field **Text** at the bottom, write the text to be milled with TrueType letters; in this example "CIMCO 456". 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.

Text	Entry
Start Point X:	
	0 °x
Start Point Y:	
	0 °,
Text Angle:	
	30 °r
Text Height:	
	25 📍
Text:	
	CIMCO 456
Alignment	Select Font
	\mathbf{X}

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.

Font					×
Font Font: Times New Roman Tempus Sans ITC Times New Roman Trebuchet MS Verdana Viner Hand ITC	•	Font style: Bold Regular Italic Bold Bold Italic Sample AaBby	- - Yy	Size: 10 11 12 14 16 18 20 7 Zz	OK Cancel
		Script: Western		, ,	•

Using the **Alignment** button, you can select the text horizontal and vertical alignment relative to the starting point entered.

Text Alignment
Horizontal Alignment
 Center
─ Right
Vertical Alignment Top
🔘 Center
e Bottom
Cancel OK

When done, click on the blue check mark button \checkmark at the bottom right of the dialog to insert the text.

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

🔬 🖬 9 🗉 🔹		CIMCO Edit v	5.2 - [Untitled Drawing *]			
Editor NC Functions	Backplot File Compare	Transmission CNC-Calc				Window* Help* = 🗗 🗙
Open Drawing * Save Drawing * Setup CNC-Calc File			Center Two Radius Points 2 2 2 Draw Arcs / Circles	Rectangular Bolt Hole Pattern Pattern	A a Text Entry a Text	• 🖩 🕑 A a 🔇
Untitled Drawing *						4 Þ 🗙
Text Entry						
Start Point X:						
Start Point Y:					1/10	
V Punk I.					$\left \left \right\rangle\right $)
Text Angle:					1-1-2	*******
Text Height:				1	$S \rightarrow $	
•x Text				AR	50	
Text			· · · · · · · · · · · · · · · · · · ·	$(\langle \cdot \rangle)$	*	
Alignment Select Font						
\mathbf{X}			1			
Element Info			$\mathbb{A} \setminus \mathbb{A}$	9		
		a a a a fin				
	· · · · · · · ·					
		a a a g 2 a a				40.00
	`					
	Machine Zero X:	Machine Zero Y: 0	Machine Zero Z: 0.0		Use Y-Axis Substitution	Part Diameter 50.0
Indicate or enter th	e start point of text		Licensed to DEMO -	NOT FOR RESALE - CIMCO Integratio	on X: -20.092 Y: 52.816	INS 14:25:13 .;;

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

12.3.2. Contour Milling of Letters

To begin the creation of an NC program for the TrueType letters milling operation, ensure that **ISO Milling** is selected as programming format for the NC program in the *File Type* drop-down list in the *Operations* toolbar.

Then select the feature TruType Letter Milling by clicking on the icon **a** Mill TrueType Letters in the *Milling Operations* toolbar to generate a CNC-toolpath for milling letters.

ISO Milling 🔹	
	3
Milling Operations	

Now the CNC-Calc Letter Milling pane is opened to the left of the drawing area. Write LETTER MILLING 'CIMCO 456' in the *Comment* field.

Letter	Milling
Comment: TTER MILLIN	G 'CIMCO 456'
Show Toolpath	Parameters
Back	New
Export Editor	Export Clipboard

Next, click on the **Parameters** button. The TrueType Text Milling Parameters dialog appears. Check the **Mill Outline** option at the top and fill in the rest of the fields as shown below. Click **OK** when done.

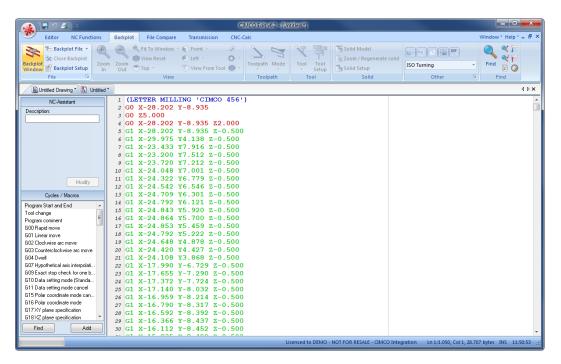
True Type Text Milling Parameter	s 🗾
Milling Operation Image: Mill Outline	Mill Interior
Cutter Diameter: 0.7	Stepover Stepdown Z:
Depth Settings	1.0
Retract Height:	Stepover XY:
5.0	0.4
Safe Distance:	
2.0	Stock to Leave Stock to Leave Z:
Start Depth:	0.0
0.0	Stock to Leave XY:
End Depth: -0.5	0.0
	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.

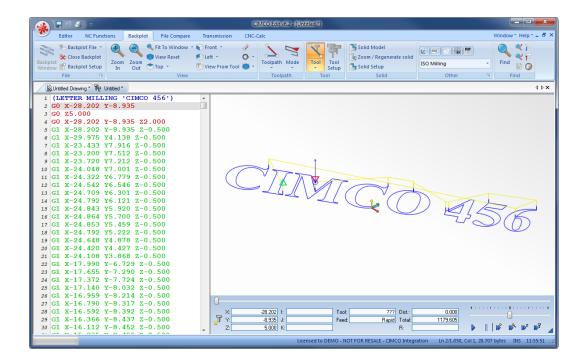
Click on the **Show Toolpath** button. Now the generated toolpath will be shown on the drawing. Since the toolpath coincides with the contour of the letters it can be difficult to see.

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** tab and then clicking on the **Backplot Window** icon.



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.



12.3.3. Pocket Milling of 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 *Untitled Drawing* (or the name of your drawing if you have saved the drawing).

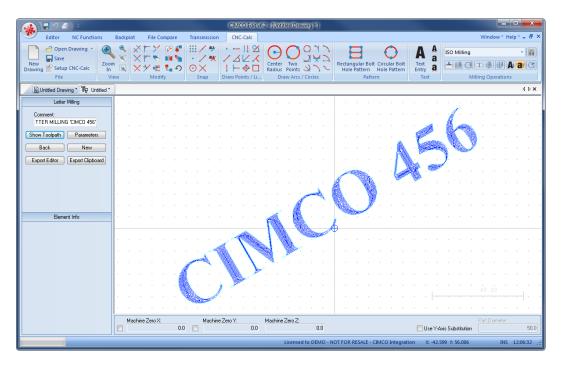
2. Click on the button **New** in the Letter Milling pane, and enter a comment for the operation.

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

4. Insert the new parameters as shown below (remember to check the option Mill Interior and disable Mill Outline).

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

- 5. Window-select the text.
- 6. Click on Show Toolpath 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.

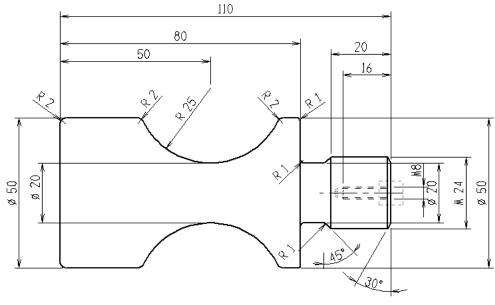
	🖬 १ 🗉) र				CIMCO Edit v6.2 - [U	ntitled *]			
	Editor NC Functions	Backplo	t File Compare	Transmission	CNC-Calc				Window * Help * = d
Backplo Window		pom Zoom In Out	 Fit To Window View Reset Top * View 	 Front Left View From Tool 	Toolpath Mode	Tool Setup Tool	Solid Model Com / Regenerate solid Solid Setup Solid	ISO Milling Other	Find C C
	Intitled Drawing * The Untit	ed 🔨 🕅 🛱 Un	titled *						4 Þ
2 G 3 G 4 G 5 G 6 G 7 G 8 G 9 G 10 G 11 G 11 G 13 G 14 G 13 G 14 G 13 G 14 G 13 G 14 G 13 G 14 G 12 G 13 G 14 G 12 G 12 G 14 G 12	ILETER MILLIN 30 x - 29.093 y - 3 50 Z5.000 y - 3 50 Z5.000 y - 3 50 Z - 29.093 y - 3 11 x - 29.093 y - 3 31 x - 29.173 y - 3 31 x - 29.273 y - 3 31 x - 38.936 y - 3 31 x - 38.936 y - 3 33 x - 29.073 y - 3 33 x - 29.073 y - 3 32 x - 29.037 y - 3 31 x - 38.820 y - 3 31 x - 38.820 y - 3 31 x - 39.519 y - 3 <	10.870 2 10.870 2 10.867 2 10.867 2 10.828 2 10.828 2 3.501 2 3.501 2 3.505 2 10.931 2 10.884 2 10.690 2 10.690 2 10.690 3 10.409 2 3.198 2 3.482 2 3.183 2 3.508 2	22.000 -0.500 -0.500 -0.500 I2. -0.500 I0. -0.500 I-0. -0.500 I-0. -0.500 I0. -0.500 I0. -0.5	56 60 03 226 03 22 00 90 52 13 441			C C	4	56
27 G	31 X-28.789 Y- 33 X-28.951 Y-3	10.783 2 10.564 2	1-0.500 1-0.500 I-0	0.2 ×	-29.093 l:	Tool		0.000	0
29 G	2 X-29.073 Y- 2 X-29.224 Y-	10.202 2			5.000 K:		B: _) ₩° ₩ ⁶ ₩ ² ₩ ⁷

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

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 v7.

13. Tutorial Lathe 1 - 2D Construction of a Part for Turning



Sample 2D part.

This tutorial demonstrates one of many ways in which the 2-dimensional part above can be drawn in CNC-Calc v7. Since the part is symmetric about the horizontal axis, only a subsection of the part needs to be drawn. The complete part emerges from mirroring. The part will be used to generate turning toolpaths, so we only need to draw the outer contour of the upper half.

This tutorial demonstrates the use of the following functions:

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

13.1. Open a New Drawing

To make a new drawing you must click on the icon **New Drawing** in the *File* toolbar under the CNC-Calc tab. When the new drawing is opened the following window should now be displayed:

🔬 🗄 9 🗲 🔍 🔹								C	IMCC) Edit	v6.1 -	(Untit	led Di	awing	9)															- 6	
Editor NC Functions	Backp	lot	File C	ompar		Fransm			NC-Ca																			Win	dow	Help	~ = é
New Save Setup CNC-Calc		×1 ×1 ×1	Mod		•) /) /) / Sna	₩.		∠ ∕		C.	hter dius F	Two Points w Arcs		インシン	Re	ctang Hole F	attern	olt Ci H	ircular ole Pat	Bolt tern		xt try	a	ISO T	þ.		i.			
	view		Mod	пу		Sna	p	Draw	Point	S / LI		Drav	N Arcs	/ Circ	ies			Pa	ttern				Text				Tum	ing C	perat)	tions	4 ۵
Untitled Drawing																															4 P
CNC-Calc	- I .		,													•															
	1.1	• •								*								,							4	•					
			,																						4						
		• •																	•				,		*		÷				
	1 × 1									-		-	-	-	.	-	-	-			-	-			-		-			-	
	- A -	• •					, ,	• •				•				•															
	1.1	• •	,			,				,		*						,							4	•				•	
	1.	• •						• •				٠								•					4		•				•
		• •																							*	÷	*				*
	<u> </u>									-						-		-			-						-				
Element Info	1 ·	• •	*			*				*	, ,		*	*		•	*	*	*			*	*			*	*	*			
	1.4	• •	·					• •	•		•	٠				4	•	,		•	•			·	4	•	*				
	1.4								*			4	*			4	*				*				4	*					
	1.1	• •						• •			•			•		*	•		•	• •					*	*	*	•			*
	1.1								-			-	-	-		-	-	-			-	-			-	-	-			-	-
		• •				•		• •	•	*		٠	*	•		•	•	*	*	•	*	*	*	•	*	*	*	•			*
		• •				*				*	•	4	•	*	•	4	•	*				*			4	•	*				*
	1.1	• •	·			•		• •			•	٠	*	•		*			•	•	*	•	Ŀ		4		•	,45.	ρο.	-	1
	1.1	• •					•				•	*			•	*			•			*	ļ		*	*	*				ţ
	<u> </u>	lachine						e Zero>					-								-				-						
		lachine	Zero Z:		0.0		lachini	e ∠ero >	s	0.0												E	Use	Y-As	is Sub	stituti					50
															ensed 1	- 051		IOTE			cm 10							co.co			0.07.40

Make sure that file type **ISO Turning** is selected in the **File Type** field in the *Operations* toolbar.



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 \mathbf{I} Setup CNC-Calc and then Global Colors from the tree menu. For this tutorial we have chosen to use blue as our drawing color and white for the background.

13.2. Draw the Geometry

13.2.1. Draw vertical lines defined by start point and length

- Click on the Vertical icon in the *Draw Points / Lines* toolbar.
- Enter the values shown in the figure below to draw a vertical line of 12 mm:

	Vertical Line	
:	Start Point Of Line Z:	
		0 îz
	Start Point Of Line X:	
		0 îx
ļ	Line Length:	
		12 👖
		\times

• Click on \checkmark to approve the command and draw the line.

Depending on the axes' color it can be difficult to see the line, since it is situated on the X-axis.

• Draw another vertical line by entering the following values in the dialog that is already open:

Vertical Line	
Start Point Of Line Z:	
	-110 🔓
Start Point Of Line X:	
	0 îx
Line Length:	
	25 °]
	\times

• Click on \checkmark to approve the command and draw the line.

-	-											-		-							ł	
	4			4			÷	4	÷			4								4	ł	
	4					-	-	4				4					4				ł	
													-								ł	
	1			4	÷	4		4	÷	4		4		+		r.	4			4	ł	4
-					-	-	-		v	-	-	-		-			-				ł	-
		1		4													*			4	ł	
				4		4	*	4				4		-			4			4	ł	4
-			-		-	-	-			-	-	-		-			-					-
							 				 										-	
*	-			5	*	-	-	4	*		-	-	*	*			*			5	ł	-
		4		7								4			2			1			ł	
×.	4			4		4	-	4	×.			4		4		*	4	×.	4		ł	4
-	-		-		-	-	-		-	-	-	-		-		-	-				ł	-

0

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

13.2.2. Offset an element

- Click on the **Offset Element** icon in the *Modify* toolbar.
- Enter the value **Offset Distance = 80** in the CNC-Calc pane.

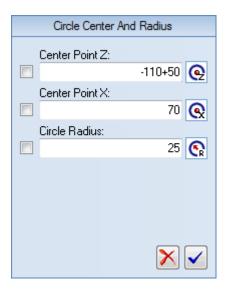
Offset Element		
Offset Distance:		
	80	\$
Steps:		
	1	
Copy (uncheck to Mo	ve)	
		X

- Now click on the leftmost line (the 25 mm line drawn in the last step). After doing this, 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 (it turns blue after clicking on it).

	-				-	-		-		-		-		-					-
-	4		4	•			4				4				4			4	
÷		-		*	÷	-	4			-	4	-			4			h	
÷					÷	÷													
	ī.		4		4		4		4		4	+	r i						
-				-		-	-			-		-							
			4																
													,			1			
-	4		4		4	-	4			*	4	4			4				
	-	-		-	-	-			-	-	-	-		-	-		-		-

13.2.3. Draw a circle defined by its center and radius

- Click on the Center Radius icon 🕞 in the Draw Arcs / Circles toolbar.
- Enter the values shown in the figure below:

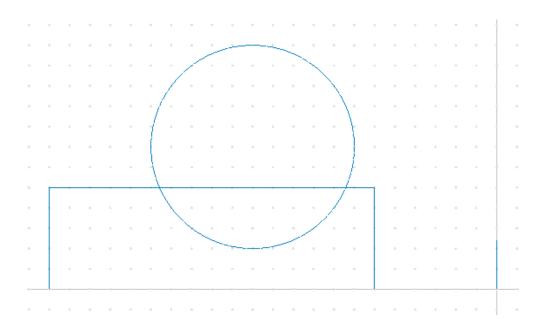


• Click on \checkmark to approve the command and draw the circle.

			4																4	ł	
	4		4		4			1	-4-		 -	4							4	ł	4
-	-			-		-	/	-		-	-	$\overline{\ }$		-		-	-			ł	
	4		4		4	/		4				4					4		4	ł	4
	4	-			-/			4				4	.)	1		-		4		ł	
			,		1									£.				4			
	4		4		- É			4	÷					-			4		4	ł	4
-					1					-	-			F		-	-			ł	
	T		4		-\			4				4	- ,	/	I.				4	ł	
								4				4								ļ	
			,									/	1						,	ļ	
					4			-			 						4				
		-			-					-		-		-		-	-				
	4		4																4	ļ	

13.2.4. Draw a line defined by its endpoints

- Click on the **Snap to End Points** icon / in the *Snap* toolbar.
- Click on the Between 2 Points icon / in the Draw Points / Lines toolbar to draw a line between two points.
- Select the top point of the two long lines of 25 mm to draw a line between those elements. 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.

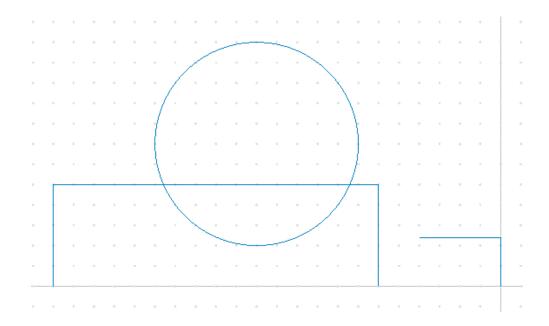


13.2.5. Draw a horizontal line defined by its length

- Click on the **Horizontal** icon in the *Draw Points / Lines* toolbar.
- Enter the value Line Length = -20 to draw a horizontal line of 20 mm in the -Z direction.

Horizontal Line
Start Point Of Line Z:
Start Point Of Line X:
V
Line Length:
-20 罩
\mathbf{X}

• With the **/** Snap to End Points function enabled, select the top end of the short vertical line furthest to the right. This will draw the horizontal line.

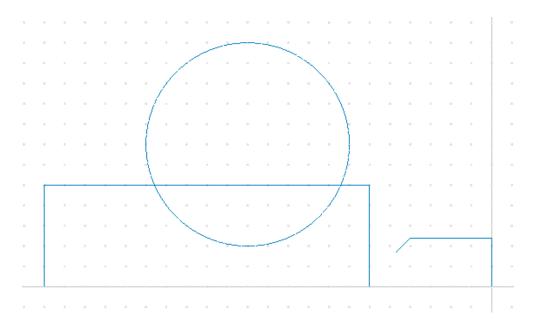


13.2.6. Draw a polar line

- Click on the **Polar** icon \angle in the *Draw Points / Lines* toolbar.
- Enter the following values into the CNC-Calc pane:

	Polar Line		
	Start Point Of Line Z:		
V	Start Point Of Line X:		٧z
			∕x
	Line Angle:		
		180+45	⊿
	Line Length:	5	1
			•⁄
		×	✓

• With the **/** Snap to End Points function enabled, select the left end of the short horizontal line drawn in the previous step. The drawing should look like the one below.



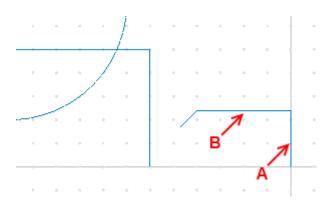
13.2.7. Chamfer the foremost corner

- Click on the **Chamfer Elements** icon *^{*}* in the *Modify* toolbar.
- Define the chamfer using *Distance and Angle* and enter the values shown in the picture below into the CNC-Calc pane:

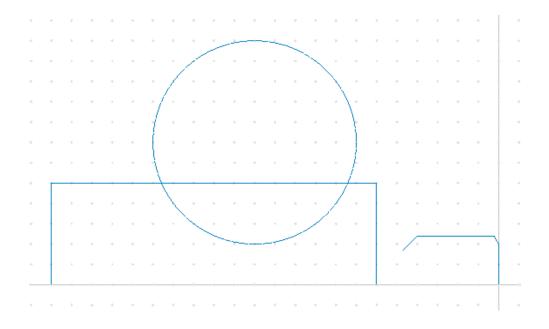
Chamfer Lines	
🔘 One Distance	
🔘 Two Distances	
Oistance and Angle	
First Length	
	2.0 🌾
Second Length	
	0.0 🏲
Chamfer Angle	
	30 🏲
🔽 Trim Elements	
	\times

• 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.



• From the four possible solutions select the one which makes the right chamfer. In the picture below you can see how it should look.

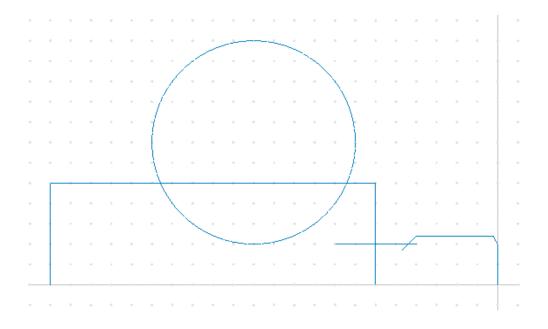


13.2.8. Draw a line defined by its endpoints

- Click on the **Between 2 Points** icon / in the *Draw Points / Lines* toolbar.
- Enter the following values into the CNC-Calc pane:

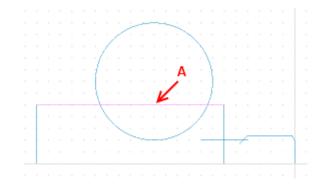
Line Through 2 Points	
First Point On Line Z:	
-20 🛃	
First Point On Line X:	
20 🖍	
Second Point On Line Z:	
-40 🖌	
Second Point On Line X:	
20 🖍	
Polyline	
\mathbf{X}	

• Click on \checkmark to approve the command and draw the line.

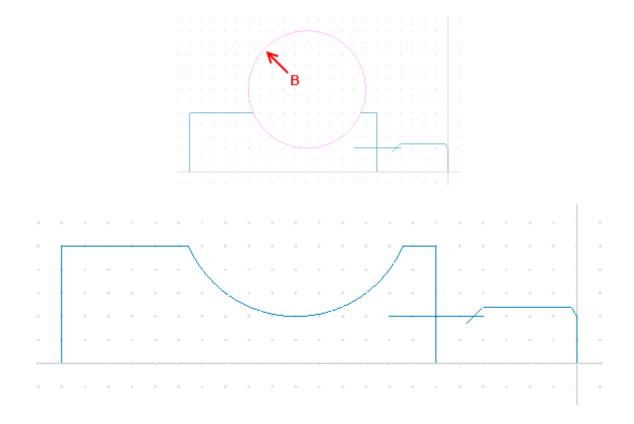


13.2.9. Trim between points of intersection

- Click on the **Trim to Intersection** icon \times in the *Modify* toolbar.
- Now trim the long horizontal line. To do this, click on the part of the line you want to remove indicated by A on the picture below.

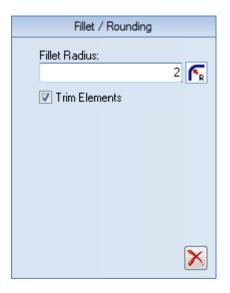


• Now trim the large circle. To do this, click on the part of the circle you want to remove - indicated by B on the picture below.



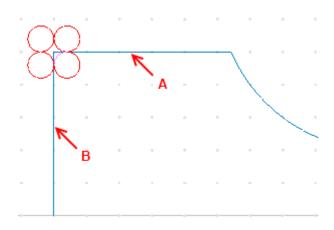
13.2.10. Fillet intersections

- Click on the **Fillet Elements** icon T in the *Modify* toolbar.
- Enter the value **Fillet Radius** = 2 in the CNC-Calc pane.



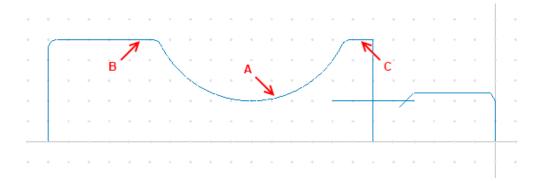
• Select the elements on which the fillet operation should be performed. This is done by left-clicking on the part of the elements that you want to keep.

To fillet the leftmost corner, select the elements as indicated on the picture below by A and B. Then, you must select and left-click precisely on the arc element you want to keep. This is the pink 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 picture below.

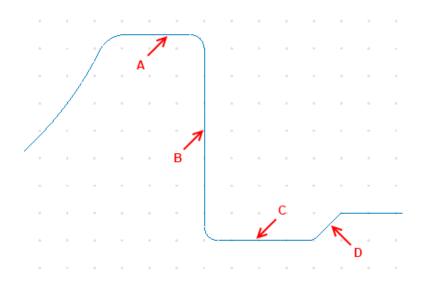


Repeat fillet operation with radius 1 corners

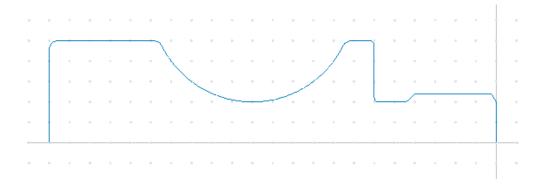
• First change the **Fillet Radius** value in the CNC-Calc pane from 2.00 to 1.00.

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

• 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 following picture.

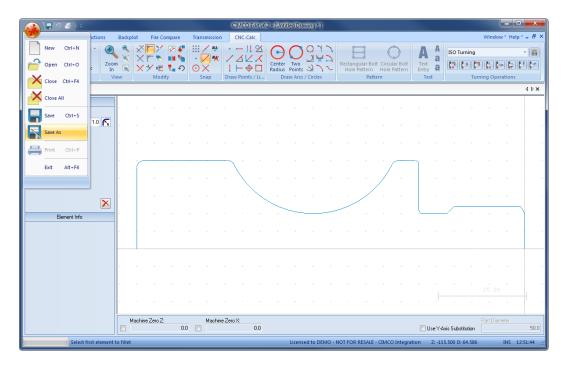


Now the drawing is finished, and it should look like the one below:



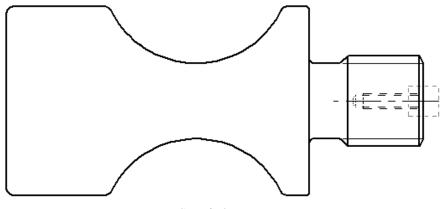
13.3. Save the File

Click on the **Main Menu** icon and then select **Save As** from the drop-down menu. Give the file the name *CNC-Calc Turning Tutorial 1* and save it (the file name extension is added automatically).



14. Tutorial Lathe 2 - Machining the Part

With CNC-Calc v7 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.



Sample 2D part.

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

In the following we assume that the stock used is Ø60 mm, and that it projects sufficiently from the chuck jaws.

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

- Facing the Front of the Stock
- Roughing the Part
- Grooving the Part
- Finishing the Part
- Threading the Part
- Drilling the Part
- Tapping the Part
- Parting-off the Part

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

14.1. Open an Existing Drawing

To open a drawing, click on the icon rightarrow **Open Drawing** in the *File* toolbar under the CNC-Calc tab. An Open file dialog will appear.

🔜 Open	×
Look in: 🕕 Drawings	• 🎯 🤌 📂 🛄 •
Name	Date modified
CNC-Calc Milling Tutorial 1.cdd	02-01-2012 13:45
CNC-Calc Turning Tutorial 1.cdd	16-04-2012 12:23 =
🗟 inch-g71.cdd	20-07-2012 12:35
🗟 lathe1.cdd	20-07-2012 12:35
🗟 metric-g71.cdd	20-07-2012 12:35
🗟 tutorial 1.cdd	20-07-2012 12:35
	00 07 0010 10 0F
File name: CNC-Calc Turning Tutorial 1.cdd	Open
Files of type: CNC-Calc Drawing files (*.cdd, *.dxf)	

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

🔈 🖬 १ 🛫 🔍 न				CO Edit v6.2 -	[C:\CIMCO\C	MCOEdit	6\Drawings\(INC-Calc Turn	ing Tutorial	1.cdd *]						×
Editor NC Functions	Backp	lot File	Compare	Transmiss	ion CNC-C	alc								Windo	w* Help*	- 8
New rawing * Setup CNC-Calc Ir File	om 💌	× × × × * Mo	- 🖉 🗳				Center Two Radius Point Draw Arr	O い い S / Circles	Rectangu Hole Pa	llar Bolt Circulz attern Hole P Pattern	ar Bolt attern	A A a a a a a a a a a a a a a a a a a a	ISO Turning	🔊 🚺 🖡		•
CNC-Calc Turning Tutorial 1.cdd	•															4 Þ 🗙
CNC-Calc																
										/		ĵ '				
					X					· /						
		-							- /					-		
Element Info	-											<u> </u>				
	1															
		fachine Zero		Mac	hine Zero X:	0.0						🗖 Use Y 4	uxis Substitution	Part Diar	neter	

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

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 \mathbf{I} Setup CNC-Calc and then Global Colors from the tree menu.

14.2. Facing the Front of the Stock

In the following section of the 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.

14.2.1. Creation of Facing Toolpaths

Ensure the file type (NC Format) for our facing example program is ISO Turning.

Then select the feature Face Turning by clicking on the **Face Turning** icon in the *Turning Operations* toolbar.



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

	Face T	uming	
	Comment:		
		FACING	
	Start Point Z:		
		5	۲z
	Start Point X:		
		60	۲x
	End Point Z:		
		0	z,
	End Point X:		
		0	×
		Parameters	
S	how Toolpath	New	
	Export Editor	Export Clipbo	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.
- **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 button **Parameters** in the CNC-Calc pane to define how the operation will be performed. Enter the following values into the parameter dialogs shown below.

Cuts Tab

Use this tab to configure the tool setup and other parameters for the operation.

Face Turning Parameters	
Cuts Entry/Exit	
Tool Definition	Roughing
Tool Orientation	Roughing Stepover
	3.0
	Finish ▼ Finish Stepover
	0.2
	Finish Cuts:
Tool Radius 0.8	1
0.0	Stock to Leave:
Overcut Amount:	0.0
1.0	
Finish Compensation	
Compensation Type Computer	
	OK Cancel

- Tool Orientation: The nine icons represent the possible nine 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 End Point 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/Exit Tab

This tab is used to configure how the tool approaches and leave the part.

Face Turning Parameters	
Cuts Entry/Exit	
Entry/Retract Entry Amount: 2.0 Retract Amount:	Entry Vector Use Entry Vector Entry Angle 0.0
2.0	Entry Length
	Exit Vector Vector Exit Angle
	0.0
	1.0
	OK Cancel

- 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.

After entering the values, close the parameters dialog with **OK**. To show the generated toolpath on the drawing click on **Show Toolpath** button in the Face Turning pane.

🔬 🖬 🤊 🐔 🔹			CI	MCO Edit	v6.2 - [C:	(CIMCO)	CIMCOEd	dit6\Draw	ings\CN	C-Calc Ti	urning	g Tutorial 1	.cdd *]								×
Editor NC Functions	Backplo	ot File	e Compare		smission		-Calc												Windo	w * Help	- # C
📄 🔐 Open Drawing 👻 💽	2	XE	Y 🛛	• •••	AU-		112	\odot	\bigcirc	0.7	2	E		\bigcirc			ISO T	urning			- 📰
New Save Drawing Setup CNC-Calc In	m 📉	XY		ຸ ຄຸ	× ^			Center	Two Points	37		Rectangula Hole Pat			olt Te	ext a	17	1		-	2
	iew	м	lodify			Draw Po			aw Arcs ,				Pattern			Text		Tur	ning Op	erations	
CNC-Calc Turning Tutorial 1.cdd *																					4 Þ 🗙
Face Turning	1																				
Comment: FACING																					
Start Point Z:																					
Start Point X:																				m	
End Point Z:																				\sim	
0 Z) (() (\square							
End Point X:		1					\mathbf{X}														
Parameters								~				1									
Show Toolpath New									-											_ ∖ ‡	
Export Editor Export Clipboard																					
																				X	+
Bement Info																					
	м	achine Zerr	07		Machine	Zero X:					1					1			Part Dia	meter	
		ucrimite 2,010		0.0	Machine	2010 / .	0.0								E	Use Y	Axis Sub:	titution	i al Dia	motol	
None										Licensed	i to D	EMO - NOT	FOR RE	SALE - CIN	MCO Inte	gration	Z: -4	1.170 D	: 88.537	INS 1	.0:13:35

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

14.2.2. Exporting the Toolpath and Backplot in the Editor

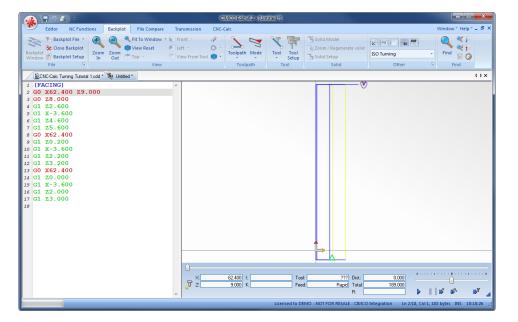
Click on **Export Editor** in order to generate the actual program. Now, a window like the one shown below will appear. Notice that the comment from the dialog is inserted at the top of the program as a comment.

Ester NC Functions Backpict File Transmission CNC-Calc Window* Heip* - d New New New New New New New New New New	🔬) 🖶 🤈 🚄) 🔹			CIMCO Ed	it v6.2 - [Untitled *]					
Image: Save more Coll Curcles Ing and the set of the set	Editor NC Functions	Backplot File Comp	oare Transmission	CNC-Calc					Windov	v * Help * = 1
NC-Assistant 1 (PACTNG) Description 2 GO X52,400 Z9,000 00 28,000 40,000 00 26,000 29,000 00 26,000 29,000 00 26,000 29,000 4 G1 25,600 4 G1 25,600 6 G1 24,600 7 G1 25,600 8 G2 26,00 9 G1 25,600 10 G1 25,600 10 G1 25,600 11 25,200 12,200 12 G1 2,3,600 12 G1 2,3,600 12 G1 2,3,600 12 G1 2,200 12 G1 2,3,600 12 G1 2,000 13 G1 2,000 14 G1 2,000 15 G1 2,3,600 16 G1 2,2,000 16 G1 2,2,000 </th <th>New rawing Setup CNC-Calc</th> <th>Zoom III XY</th> <th>・/株 のOX</th> <th></th> <th>Radius Points 💥 🤉 🛰</th> <th>Hole Pattern Ho</th> <th>cular Bolt Text le Pattern Entry</th> <th>a</th> <th></th> <th></th>	New rawing Setup CNC-Calc	Zoom III XY	・/株 のOX		Radius Points 💥 🤉 🛰	Hole Pattern Ho	cular Bolt Text le Pattern Entry	a		
NC-Assistant 1 (PACTNG) Description 2 GO X52,400 Z9,000 00 28,000 40,000 00 26,000 29,000 00 26,000 29,000 00 26,000 29,000 4 G1 25,600 4 G1 25,600 6 G1 24,600 7 G1 25,600 8 G2 26,00 9 G1 25,600 10 G1 25,600 10 G1 25,600 11 25,200 12,200 12 G1 2,3,600 12 G1 2,3,600 12 G1 2,3,600 12 G1 2,200 12 G1 2,3,600 12 G1 2,000 13 G1 2,000 14 G1 2,000 15 G1 2,3,600 16 G1 2,2,000 16 G1 2,2,000 </td <td>CNC-Calc Turning Tutorial 1.cd</td> <td>dd * 📓 Untitled *</td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td></td> <td>4 Þ</td>	CNC-Calc Turning Tutorial 1.cd	dd * 📓 Untitled *								4 Þ
	NC-Assistant Description: Cycles / Macros Program State of End Order Ange with Contart Stud. Tool change with Contart Stud. Tool change with Contart Stud. Stud Change with Contart Stud.	1 (PACLNG) 2 GO X62.400 3 GO Z8.000 4 GI Z2.600 5 GI X-3.600 6 GI X-3.600 7 GI Z5.600 9 GI Z0.200 9 GI Z0.200 10 GI X-3.600 12 GI X-3.600 13 GO X62.400 13 GO X62.400 14 GI X-3.600 15 GI X-3.600 16 I Z0.000	29.000							

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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.

Now a window like the one below will appear.





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

14.3. Roughing the Part

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

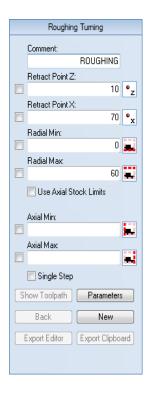
14.3.1. Creation of Roughing Toolpaths

Ensure the file type (NC Format) for our roughing example program is ISO Turning.

Then select the feature Roughing Turning by clicking on the **Roughing Turning** icon in the *Turning Operations* toolbar.



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

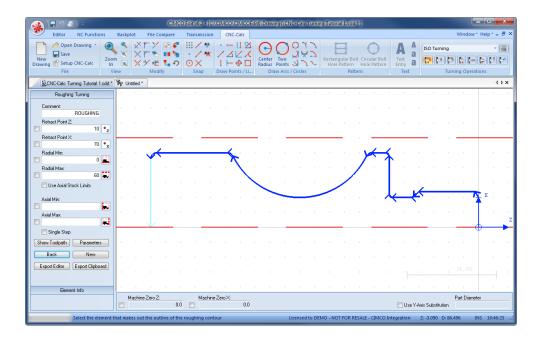


- **Comment**: This comment will be shown in the final NC-Program. It is always good to include a comment in order to distinguish the various operations in the final program.
- **Retract Point Z**: This is the Z value to where the operation will retract the tool after completion.
- **Retract Point X**: This is the X value to where the operation will retract the tool after completion.
- **Radial Min**: This is the lower machining limit for the operation in the X direction.
- **Radial Max**: This is the upper machining limit for the operation in the X direction.

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 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 below.



Click on the button **Parameters** in the Roughing Turning pane to open the parameters dialog. Enter the following values into the parameter dialogs shown below.

Tool Tab

This tab is used to configure the settings for tool, work orientation and compensation type used for the operation.

Lathe Roughing Parameters	
Tool Cuts Entry/Exit	
Tool Definition	Work Orientation
Tool Orientation	F
$\mathbf{E} \mathbf{O} \mathbf{F}$	
Tool Radius	
0.5	
Plunge Settings	
Use Axial Plunge	Use Radial Plunge
Plunge Angle:	Plunge Angle: 0.0
Compensation	
Compensation Type Computer	
	OK Cancel

- **Tool Orientation**: The nine icons represent the possible nine 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.
- Use Axial Plunge: If the tool permits it, check this option to allow horizontal plunge.
- Use Radial Plunge: If the tool permits it, check this option to allow vertical 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 operation. The two most commonly used are *Controller* or *Computer*.

Cuts Tab

Use this tab to configure cutting parameters for the operation.

Lathe Roughing Parameters	×
Tool Cuts Entry/Exit	
Cuts Overlap: 2.0 Depth of Cut: 5.0 V Use Even Steps Retract Retract Distance: 1.0	Finish Passes Vuse Finish Passes Passes: 1 Spacing: 0.2 Stock to Leave Stock to Leave X: 0.1 Stock to Leave Z: 0.1
	OK Cancel

- **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: Check this option if any finish passes should be taken.
- **Passes**: The number of finish 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.

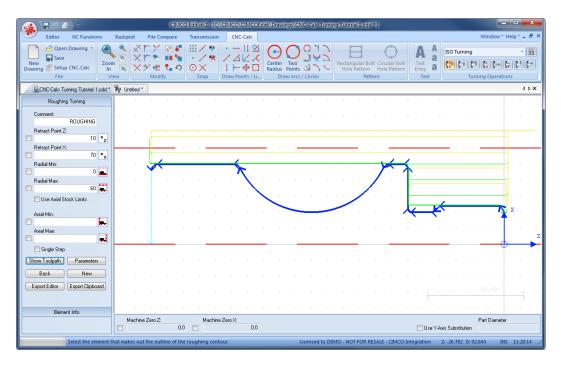
Entry/Exit Tab

Configure how the tool approaches and leaves the part.

Lathe Roughing Parameters	
Tool Cuts Entry/Exit	
Entry/Retract Intry Amount: 1.0 Extension: 1.0	Entry Vector I Use Entry Vector Entry Angle 45.0 Entry Length 1.0 Exit Vector Exit Vector Exit Angle 45.0 Exit Length 1.0
	OK Cancel

- Entry Amount: This value is used to extend the toolpath before it starts the actual cut.
- Extension: This value 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.

After entering the values, close the parameters dialog with **OK**. To show the generated toolpath on the drawing click on **Show Toolpath** button in the Roughing Turning pane.



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

14.3.2. Exporting the Toolpath and Backplot in the Editor

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 the icon **Paste** from the *Edit* toolbar in the Editor tab.

🔀 📑 💽 🚔 💣 Iso Turning Cut Paste (* Redo) Insert File 🔍 Find Pre 😰 Next Tool change Go to Line/Block Number 🎝 Undo 🛛 🛱 Append File V Find Next Save Print Global Find Ky Replace Close Save File Type CNC-Calc Turning Tutorial 1.cdd * 🗿 Untitled * 4 Þ 🗙
 It GI Z2.200

 12 GI Z3.200

 13 GO X62.400

 14 GI Z0.000

 15 GI X-3.600

 16 GI Z2.000

 17 GI Z3.000

 18 (BOUCHING)
 NC-Assistant 10 GA DEVENTION 10 (GOUGHING) 19 GO X71.000 Z10.000 20 GO Z1.207 21 GO X51.531 22 GI X50.117 Z0.500 23 GI Z-30.599 24 G3 X50.600 Z-31.500 I-1.558 K-0.901 25 GI Z-110.500 27 GI X66.000 27 GI X67.414 Z-109.793 28 GO Z1.207 29 GO X41.648 30 GI X40.234 Z0.500 31 GI Z-29.700 32 GI X47.000 Z-29.700 33 GI Z-110.500 34 GI Z-110.500 18 (ROUGHING) Modify Cycles / Macros Cycles / Macros Program Start and End Tool change with Constant Surf... Tool change with Constant RPM Program comment 600 Rapid move 601 Linear move 602 Clockwise acr move 602 Clockwise acr move 603 Courterclockwise acr move 604 Devel 601 Data setting mode 601 Betwise data input 621 Metric data input 627 Reference point return che... 628 Return to reference point 33 (G3 X50.600 Z-31.500 1 34 (G1 Z-110.500 35 (G1 X56.117 36 (G1 X57.531 Z-109.793 37 (G0 X11.765 38 (G0 X31.765 39 (G1 X30.351 Z0.500 40 (G1 Z-29.700 40 (G1 Z-29.700

Now the NC program should look like the following screen.

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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.

Now a window like the one below will appear.

🕢 🖬 🤊 🖆 🔻	(CIMCO Edit v6.2 - [Ur	ntitled *]				×
Editor NC Functions Backplot File Compare	Transmission CNC	-Calc				Window * Help *	- 8 ×
The Backplot File → Sackplot Setup Window Backplot Setup File → View Reset Jon 200m Out → Top → View View	Front Pront Pront	Toolpath Mode	Tool Tool Setup	Solid Model	ISO Turning Other	Find O Find	
		Toopath	1001	Solid	Other		4 Þ X
CNC-Calc Turning Tutorial 1.cdd * Tr Untitled * (FACING)							A b X
1 (FACING) 2 G0 X62.400 Z9.000 3 G0 Z8.000 4 G1 Z2.600 5 G1 X-3.600 6 G1 Z4.600 7 G1 Z5.600 8 G0 X62.400 9 G1 Z0.200 10 G1 X-3.600 11 G1 Z2.200 12 G1 Z3.200 13 G0 X62.400 14 G1 Z0.000 15 G1 X-3.600 16 G1 Z2.000 16 G1 Z2.000 17 G1 Z3.000 18 (ROUGHING) 19 G0 X71.000 Z10.000 20 G2 Z1.207 21 G0 X51.531 22 G1 X50.117 Z0.500 23 G1 Z-30.599 24 G3 X50.600 Z-31.500 I-1.558 K-0.9 25 G1 X67.414 Z-109.793 26 G0 Z1.207 27 G0 Z41.648 30 G1 X40.234 Z0.500 11 X40.234 Z0.500 21 X40.234 Z0.500		62.400 E 9.000 K	too		0.000		
	,		Licensed to DEN	IO - NOT FOR RESALE - CIMCO Ir	tegration Ln 2/64, Col :	1, 958 bytes INS 11:	29:26 ,;;

14.4. Grooving the Part

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

14.4.1. Creation of Grooving Toolpaths

Ensure the file type (NC Format) for our grooving example program is ISO Turning.

Then select the feature Grooving Turning by clicking on the **Grooving** icon in the *Turning Operations* toolbar.

ISO Turning 👻	
	1
Turning Operations	

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



- **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.

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 Turning pane.
- 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 two elements, so the selection looks like the one on the picture below.

Now your drawing should look something like the one below.

(▲) = ? =) =		CIM	CO Edit v6.2 - [0	:\CIMCO\CIMCOE	dit6\Drawings\	CNC-Calc Turni	ng Tutorial 1.cdd *]		- D X
C. N.	Backplot	File Compare	Transmissio						Window * Help * = 🗗 ×
New Drawing Save Sstup CNC-Calc File			• / 44 • / 44 • / 44 • / 44	· ⊗ / ⊿ ∠ ⋌ 	Center Two Radius Poir Draw Ar	ts Circles	Rectangular Bolt Circular Bolt Hole Pattern Hole Pattern Pattern		y v 📰
CNC-Calc Turning Tutorial 1.cdd *	To Untitled *								4 Þ ×
Grooving Turning	· ·			1 A	· ·		A		
Comment GR00VING									
Retract Point Z:									
Retract Point X:									
Single Step Show Toolpath Parameters				X					
Back New									
Export Editor Export Clipboard							a server		x
Element Info									
	Machine		Machir 1.0 🔳	ne Zero X: 0.0				📃 Use Y-Axis Substitution	Part Diameter
						Licensed to DEM	O - NOT FOR RESALE - CIMCO Int	egration Z: -60.614 D: 8.	2.512 INS 11:57:49 ,;

Click on the button **Parameters** in the Grooving Turning pane to open the parameters dialog. Enter the following values into the parameter dialogs shown below.

Cuts Tab

Use this tab to configure cutting parameters for the operation.

Lathe Grooving Parameters	×
Cuts Finish	
Tool Definition Tool Width: 3.0	Groove Angle Angle: 90.0
Corner Radius:	Cuts
Tool Orientation	Direction Bi-Directional Stepover:
Depth Settings	1.5
Stock Clearance:	Use Pecking
0.5	Pecking Depth:
Stock Amount 2.0	2.0 Pecking Retract:
Wall Backoff: 0.1	0.5
Stock To Leave Stock To Leave Z: 0.0	Stock To Leave X:
	OK Cancel

- **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 Z: Indicates how much stock should be left in the Z-direction after the whole operation is performed.
- Stock to Leave X: Indicates how much stock should be left in the X-direction after the whole operation is performed.

Finish Tab

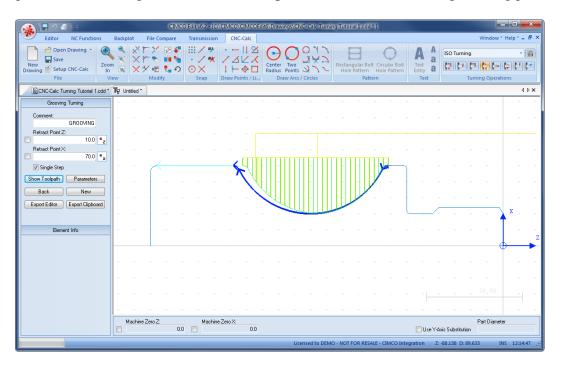
Use this tab to configure how the finish passes for the operation should be performed.

Lathe Grooving Parameters	×
Cuts Finish	
Cuts Number of cuts: Cut Depth: 0.1	First Cut Direction
Finish Pass Distances First Distance: 5.0	Overlap: 1.0
Finish Compensation Compensation Type	
Computer	•
L	OK Cancel

- Use Finish: Indicates whether or not finish cuts should be performed.
- Number of 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*.

After entering the values, close the parameters dialog with **OK**. To show the generated toolpath on the drawing click on **Show Toolpath** button in the Grooving Turning pane.



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

14.4.2. Exporting the Toolpath and Backplot in the Editor

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 the cursor to the very end by pressing Ctrl+End. Insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

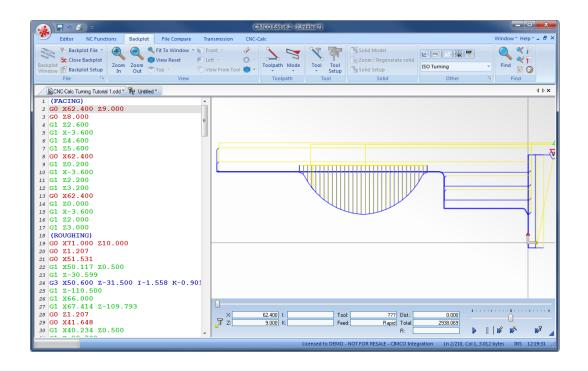
Now the NC program should look like the following screen.

🔬 🖬 🤊 🗉 🔍	CIMCO Edit v6.2 - [Unitiled *]	x
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window * Help * -	8×
New Open Close Save Art		
CNC-Calc Turning Tutorial 1.cdd	1 ⁷ 👸 United • 4 1	¢×
NC-Assistant Description:	58 G1 X47.000 39 G3 X50.200 Z-31.500 10.000 60 G1 Z-110.500 61 G1 61 G1 X51.614 Z-109.793 62 G0 X71.000 62 G0 X10.000 63 G0 Z10.000 64 (GROOVING) 65 G0 X70.000 210.000 64 (GROOVING) 65 G0 X70.000 210.000	*
Modify Cycles / Macros	68 60 X70.000 X10.000 66 60 Z-57.768 67 C0 X55.000 68 G1 X20.367 69 G0 X55.000 70 G0 Z-59.232	ш
Program Start and End Tool change with Constant Surf Tool change with Constant RPM Program comment G00 Rapid move	71 G1 X20.367 72 G1 X20.567 Z-59.132 73 G0 X55.000 74 G0 Z-56.305 74 G0 Z-56.305 75 G1 X20.697	
G01 Linear move G02 Clockwise arc move G03 Counterclockwise arc move G04 Dwell G10 Data setting mode	76 G1 X20.897 Z-56.405 77 G0 X55.000 78 G0 Z-60.695 79 G1 X20.697 90 G1 X20.697 60 G1 X20.897 Z-60.595	
G11 Data setting mode cancel G18 ZX plane designation G20 Inch data input G21 Metric data input G27 Reference point return che	81 G0 X55.000 82 G0 Z-54.841 83 G1 X21.206 84 G1 X21.406 Z-54.941	
G27 Hetelence point return che G28 Return to reference point	85 G0 X55.000 86 G0 Z-62.159 87 G1 X21.206 101 M21 M21 M21 M21 M21 M21 M21 M21 M21 M2	Ţ
	Licensed to DEMO - NOT FOR RESALE - CIMCO Integration Ln 210/210, Col 1, 3.012 bytes INS 12:17:5	j4 .;

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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.

Now a window like the one below will appear.



1

You can perform another grooving operation in order to machine the groove at the end of the outer thread.

14.5. Finishing the Part

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

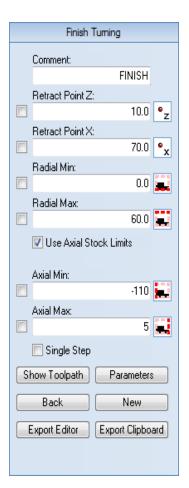
14.5.1. Creation of Finishing Toolpaths

Ensure the file type (NC Format) for our finishing example program is ISO Turning.

Then select the feature Finish Turning by clicking on the **Finish Turning** icon in the *Turning Operations* toolbar.



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

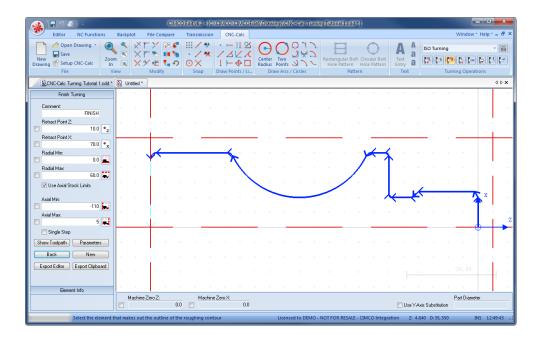


- **Comment**: This comment will be shown in the final NC-Program. It is always good to include a comment in order to distinguish the various operations in the final program.
- **Retract Point Z**: This is the Z value to where the operation will retract the tool after completion.
- **Retract Point X**: This is the X value to where the operation will retract the tool after completion.
- **Radial Min**: This is the lower machining limit for the operation in the X direction.
- **Radial Max**: This is the upper machining limit for the operation in the X direction.
- Use Axial Stock Limits: This option indicates if axial limits should be used.
- Axial Min: This is the leftmost machining limit for the operation in the Z direction.
- Axial Max: This is the rightmost machining limit for the operation in the Z direction.

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 Turning pane.
- 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.

Now your drawing should look something like the one below.



Click on the button **Parameters** in the Finish Turning pane to open the parameters dialog. Enter the following values into the parameter dialogs shown below.

Tool Tab

Configure settings for tool, work orientation and compensation type used for the operation.

Lathe Finish Parameters	X
Tool Cuts Entry/Exit	
Tool Definition	Work Orientation
Tool Orientation	🔄 📑
Tool Radius 0.4	
Plunge Settings	
🔲 Use Axial Plunge	✓ Use Radial Plunge
Plunge Angle:	Plunge Angle: 90.0
Compensation Compensation Type Computer]
	OK Cancel

- **Tool Orientation**: The nine icons represent the possible nine 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.
- Use Axial Plunge: If the tool permits it, check this option to allow horizontal plunge.
- Use Radial Plunge: If the tool permits it, check this option to allow vertical 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 operation. The two most commonly used are *Controller* or *Computer*.

Cuts Tab

Configure cutting parameters for the operation.

Lathe Finish Parameters	×
Tool Cuts Entry/Exit	
Finish Passes Passes:	Stock to Leave Stock to Leave X:
1	0.0
Spacing:	Stock to Leave Z:
0.1	0.0
Retract Retract Distance: 1.0	OK Cancel

- **Passes**: The number of finish passes to take in the operation.
- **Spacing**: The depth of each of the finish passes.

- **Retract Distance**: The distance that the tool retracts from the stock before a return move is made.
- 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/Exit Tab

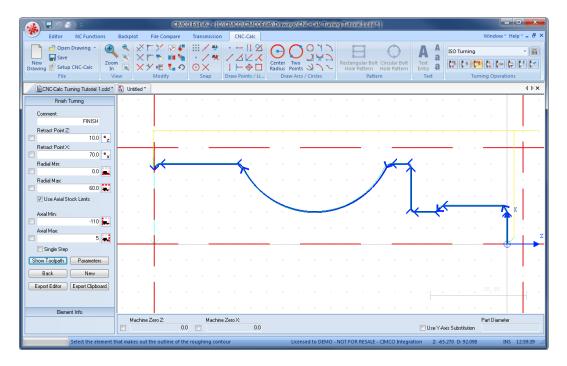
Configure how the tool approaches and leaves the part.

Lathe Finish Parameters	×
Tool Cuts Entry/Exit	
Entry/Retract Entry Amount: 1.0 Extension: 1.0	Entry Vector Use Entry Vector Entry Angle Entry Length 3.0
	Exit Vector Use Exit Vector Exit Angle 90.0
	Exit Length 3.0
	OK Cancel

- Entry Amount: This value is used to extend the toolpath before it starts the actual cut.
- Extension: This value 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 at which the tool will approach the part.
- Entry Length: The length of the approach.

- Use Exit Vector: Enable/Disable the use of exit vector.
- Exit Angle: The angle at which the tool will retract from the part.
- Exit Length: The length of the retract.

After entering the values, close the parameters dialog with **OK**. To show the generated toolpath on the drawing click on **Show Toolpath** button in the Finish Turning pane.



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

14.5.2. Exporting the Toolpath and Backplot in the Editor

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 the cursor to the very end by pressing Ctrl+End. Insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

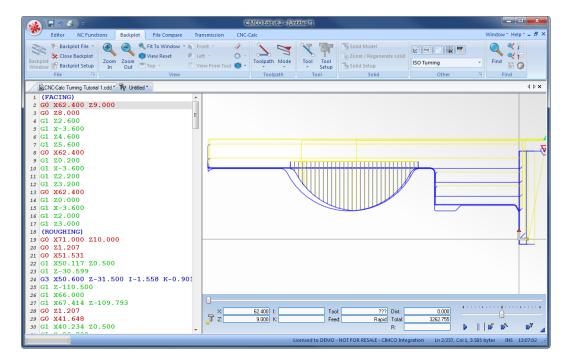
Now the NC program should look like the following screen.

🔬 🖬 🕈 🗉 🕫	CIMCO Edit v6.2	[Untitled *]	
Editor NC Functions	Backplot File Compare Transmission CNC-Calc		Window * Help * = 🗗 ×
New Open Close Save Sa	RESO Turning Print: Global Setup File Type	b Select All X Del → y Undo Append File te W Redo Insert File Edit	Find
CNC-Calc Turning Tutorial 1.cd	V 🛐 Untitled *		4 Þ x
NC-Assitant Description Modify	200 GO X70.000 209 GO Z10.000 210 (FINISH) 211 GO X70.800 Z10.000 212 GO Z2.221 213 GO X4.243 214 GI X0.000 Z0.100 215 GI Z0.000 216 GI X19.200 217 G3 X19.600 Z-0.054 I-0.000 K-0.400 218 GI X23.600 Z-1.208 219 G3 X24.000 Z-1.555 I-0.200 K-0.346		
Cycles / Macros Program Start and End Tool change with Constant Staff Tool change with Constant Staff Tool change with Constant RPM E00 Rapid move G01 Linear move G02 Clocivities arc move G03 Counterclockwise arc move G03 Counterclockwise arc move G04 Owel G10 Data setting mode G11 Data setting mode G	220 G1 Z-20.400 222 G3 Z23.766 Z-20.683 I-0.400 K-0.000 223 G2 X20.351 Z-22.390 223 G2 X20.000 Z-22.814 I0.424 K-0.424 224 G1 X20.000 Z-29.400 225 G2 X21.200 Z-30.000 I0.600 K0.000 227 G3 X50.000 Z-31.400 I-0.000 K-1.400 228 G3 X47.333 Z-38.363 I-2.400 K0.000 230 G2 Z-82.437 I10.933 K-22.037 231 G3 X50.000 Z-84.587 I-1.067 K-2.150 232 G3 X47.853 Z-110.400 I-2.400 K0.000 233 G3 X47.853 Z-110.400 I-2.400 K0.000 234 G1 X-3.853		
G27 Reference point return che G28 Return to reference point * Find Add	235 G0 X70.800 236 G0 Z10.000 237	Licensed to DEMO - NOT FOR RESALE - CIMCO Integration	E In 237/237, Col 1, 3.585 bytes INS 13:06:35

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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.

Now a window like the one below will appear.



14.6. Threading the Part

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

14.6.1. Creation of Threading Toolpaths

Ensure the file type (NC Format) for our threading example program is ISO Turning.

Then select the feature Threading Horizontal by clicking on the **Threading Horizontal** icon in the *Turning Operations* toolbar.

ISO Turning 🔹	
	5
Turning Operations	

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

	Threading	, Z Tuming	
	Comment:		
		THREADING	
	Start Z:		
		0	z
	End Z:		
		-20	۳z
S	how Toolpath	Parameter	8
	Back	New	
	Export Editor	Export Clipbo	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.
- Start Z: This is the Z value where the operation starts.
- End Z: This is the Z value where the operation ends.

Click on the button **Parameters** in the Threading Z Turning pane to open the parameters dialog. Enter the following values into the parameter dialogs shown below.

14.6.2. Thread Form Tab

e Threading Parameters	
read Form Cutting	
Taper Type	Taper Angle: 0.0
Thread Lead: Pitch 3.0	Number of Starts:
0	
Operation Type External	Internal
Left to Right	Right to Left
Thread Form	
Select From Table	
Included Angle: 60.0	Thread Angle: 30.0
Major Diameter:	Minor Diameter:
24.0	20.752
Cutting Parameters	
Stock to Leave: 0.0	Stock Clearance: 5.0
	OK Annulle

Configure the form of the thread and the taper in 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.
- **Pitch**: Defines the distance it takes for the thread to revolve one round around the part.
- **Number of Starts**: This defines how many starts the thread will have. The normal number is one.
- **Operation Type**: Determines whether the thread is at the outside or inside of the part, and is going from the left to the right or from right to left.
- 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 section Select Thread 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 diameter.
- Minor Diameter: The smallest measure of the thread diameter.
- Stock to Leave: Defines how much stock should be left at the end of the operation.
- **Stock Clearance**: Defines how far away from the stock the tool should move before it moves back to the start.

Cutting Tab

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

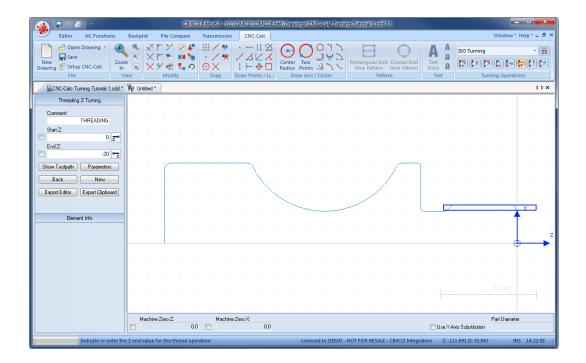
Lathe Threading Parameters	
Thread Form Cutting	
Operation Type Canned G76 🔹	Angle: 30.0
Acceleration Distance	
 Absolute 	Revolutions
	2
Pulloff Distance	
 Absolute 	Revolutions
	6.0
Type of Cut	
Constant Area	Constant Depth
Depth calculation	
First Cut Depth	1.0
Number Of Cuts:	3
Number Of Spring Cuts:	1
Overcut	
 Absolute 	Revolutions
	3.0
	OK Annuller

- **Operation Type**: Use this field to select a Canned G76 or Box G92 cycle for the threading operation.
- **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.

- Acceleration Distance: The Distance that the tool will start before reaching the thread in order to accelerate to achieve a more uniform thread.
- **Absolute** (Acceleration Distance): With this option the user 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.
- **Pulloff Distance**: Defines the distance (*Absolute* or in *Revolutions*) that should be covered at the end of each cut (only used with canned cycles).
- **Constant Area** (Type of Cut): Using the constant area option, the tool will remove equal amounts of the area per cut.
- **Constant Depth** (Type of Cut): 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 selected, the operation will be performed with this number of cuts (plus 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.
- 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.

Click on **OK** to use the values and close the dialog.

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.



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

14.6.3. Exporting the Toolpath and Backplot in the Editor

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 the cursor to the very end by pressing Ctrl+End. Insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

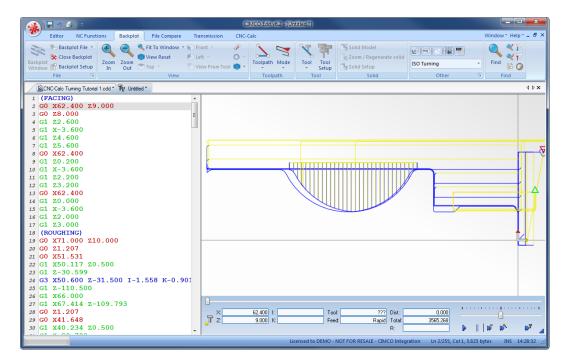
Now the NC program should look like the following screen.

🔬 🖬 🤊 🗉 🔍	CIMCO Edit v6.2	[Untitled *]	
Editor NC Functions	Backplot File Compare Transmission CNC-Calc		Window≚ Help≚ = ×
New Open Close Save Sa File		Select All X Del → Oldo Append File W Dudo Append File W Redo Edit	o to Line/Biock Number Find
BCNCCalc Turning Tutoral 1.cdt ⁻⁷ 3 Untitled 4 D ×			
NC-Assistant Description Cycles / Macros Program Start and End Tool charge with Constant Surt. Tool charge with Constant Surt. Tool charge with Constant Surt. Coll Rayde with Constant Surt. Edit Deate setting mode Coll Deate setting mode Soft Duet a setting mode Soft Due	226 G1 X47.200 227 G3 X50.000 Z-31.400 I-0.000 K-1.400 228 G1 Z-36.213 229 G3 X47.333 Z-38.363 I-2.400 K0.000 230 G2 Z-82.437 I10.933 K-22.037 231 G3 X50.000 Z-84.587 I-1.067 K-2.150 232 G1 Z-108.400 233 G3 X47.853 Z-110.400 I-2.400 K0.000 243 G1 X53.853 250 X70.800 246 G0 Z10.000 247 (THEADING) 248 G3 X4.000 Z6.396 249 G0 X21.25 240 G32 Z-23.000 F3.000 244 G3 Z-23.000 F3.000 245 G0 X34.000 246 G3 Z-23.000 F3.000 247 G3 X40.00 248 G3 Z-23.000 F3.000 249 G0 X34.000 246 G3 Z-23.000 F3.000 247 G3 X4.000 246 G3 Z-23.000 F3.000 247 G3 X4.000 248 G3 Z-23.000 F3.000 249 G3 X4.000 249 G3 Z-23.000 F3.000 240 C34.000 250 G X34.000 250 G Z6.000 250 G2 Z-23.000 F3.000 250 G3 X4.000 250 G3 Z-23.000 F3.000 250 G3 X4.000 250 G3 Z-23.000 F3.000 250 G3 X4.000 250 G3 Z-23.000 F3.000 250 G3 Z		•
Find Add	254 G0 Z6.000 255		
		Licensed to DEMO - NOT FOR RESALE - CIMCO Integration	Ln 255/255, Col 1, 3.823 bytes INS 14:25:22 ,;

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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.

Now a window like the one below will appear.



14.6.4. Select Thread

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	lect Thread					×
	Export Units Metric		C	Inches		
	Metric					-
	Major Dia.	Lead	Minor Dia.	Description		•
	20.0000	1.0000	18.9170	Fine		
	22.0000	2.5000	19.2940	Coarse		
	22.0000	1.5000	20.3760	Fine		
	24.0000	3.0000	20.7520	Coarse Fine		
	25.0000	2.0000	23.3760	Fine		
	27.0000	3.0000	23.7520	Coarse		-
				OK	Canc	el

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.

14.7. Drilling the Part

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

14.7.1. Generate a Drill Cycle

Ensure the file type (NC Format) for our drilling example program is **ISO Turning**.

Then select the feature *Drilling* by clicking on the **End Drilling** icon in the *Turning Operations* toolbar.

ISO Turning 🔹	
	5
Turning Operations	

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

	End Drilling
	Comment:
	DRILLING
	Retract Distance (Abs.):
	10 🔭
	Safe Distance (Abs.)
	3 🔭
	Start Depth:
	2 🗾
_	End Depth:
	-20 🏹
	Parameters
	New
E	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 Distance (Abs.)**: The distance to which the tool is retracted after the operation has been completed.
- Safe Distance (Abs.): 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.

🔬 🔒 9 🛃 🕫				CIMCO) Edit vé	5.2 - [C	:\CIMC	0\CIMC	OEdit6\	Drawin	gs\CNC	Calc Tu	urning	futorial	1.cdd *]									x
Editor NC Functions	Backplo	t Fi	ile Comp	are	Trans	mission	C	NC-Calc														Wind	low * H	elp * 🗕	8
New Drawing Save Drawing Setup CNC-Calc File		×r ×r ×Ý	Y R	2 4 • •		/ ** <	/ .] Draw	- 9 - 1 9 - 1 9 Points /	Ra		Two oints				alar Bolt attern Patte	Hole P		A Tex Entr	- a	ISO T	Furning	🔊 🚺 ming Oj	reation	- s	
CNC-Calc Turning Tutorial 1.cdd *	ି ଅଳ Unti	tled *					^		-															٥	Þ×
End Drilling					4																		-		•
Comment: DRILLING																									
Clearance Height:																									
Retract Height:																									
Start Depth:															./		Ì								
End Depth:								\mathbb{N})	/_										
Parameters													-						/,				x		
Export Editor Export Clipboard																							T		7.
Element Info																			+				фн	-	ŕ
	Ma	ichine Ze	ro Z:	0.0		Machin	e Zero >	e 0	.0						-				Use Y+	Axis Sub	stitution	Part D	iameter		
										L	censed 1	to DEMO	D - NOT	FOR RE	ESALE - (Integral	tion	Z: -31	.173 D:	66.354		INS	15:03:	29

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

Click on the button **Parameters** in the End Drilling pane to open the parameters dialog. Enter the following values into the parameter dialogs shown below.

Drilling Parameters

Drilling Parameters		×
Operation Type Peck Drilling	Canned O Canned O Longhand	
Dwell Use Upper Dwelling 500.0	Vise Lower Dwelling	Tip Compensation Use Tip Compensation Tip Angle
		114.0
Feedrates V Use Plunging Feedrate:	Vuse Retract Feedrate:	Drill Diameter
100.0		10.0
Pitch: 4.5	200.0	Tip Compensation 3.247
Pecking		
Ver Verking Peck Clearance: 3.0	Peck Retract: 5.0	
First Peck: 10.0	Subsequent Pecks: 6.0	
Pecking Use Pecking Peck Clearance: 3.0 First Peck: 10.0		Cancel OK

- **Operation Type**: The operation type can be Drilling G83 or Tapping CW G84 for Canned cycles, and Drilling or Peck drilling for Longhand cycles.
- Use Upper Dwelling: Toggles the use of upper dwelling. Upper Dwelling is the time that the drill will rest at the top of a peck.
- Use Lower Dwelling: Toggles the use of lower dwelling. Lower Dwelling is the time that the tool will rest at the bottom of each cut in order to break the chip.
- Use Plunging Feedrate: Toggles the use of Plunging Feedrate. This is the feedrate used during the drilling part of the operation.
- Use Retract Feedrate: Toggles the use of Retract Feedrate. This is the feedrate used during the retract part of the operation.
- 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 length of the tip of the drill. This is the length from the tip to where the drill reaches the full diameter.
- Use Pecking: By selecting this option the operation will be performed with pecking movements.
- **Peck Clearance**: The distance away from the current bottom of the hole where the feedrate will switch from rapid to feed.
- **Peck Retract**: The distance that the tool will retract between pecks (must be larger than Peck Clearance).
- **First Peck**: The distance that the tool moves down in the first peck. This is the depth of the first peck.
- **Subsequent Pecks:** After the first peck is performed, the entered distance will be used for the remaining pecks.



The selected operation type determines which drill parameters are available.

Click on **OK** to use the values and close the dialog.

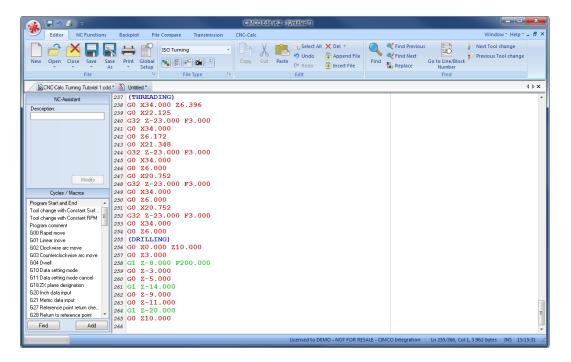
Try experimenting with the various parameters and see how they change the generated toolpath.

14.7.2. Exporting the Toolpath and Backplot in the Editor

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.

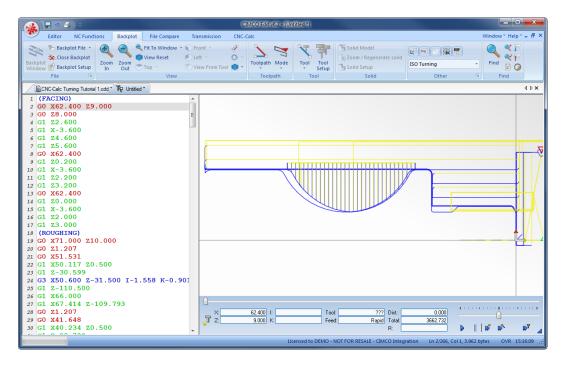
Change the window to the NC program and move the cursor to the very end by pressing Ctrl+End. Insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

Now the NC program should look like the following screen.



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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.



Now a window like the one below will appear.

14.8. Tapping the Part

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

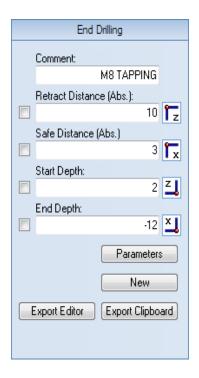
14.8.1. Generate a Tap Cycle

Ensure the file type (NC Format) for our taping example program is ISO Turning.

Then select the feature *Drilling* by clicking on the **End Drilling** icon in the *Turning Operations* toolbar.

ISO Turning 👻	
	1
Turning Operations	

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



- **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 Distance (Abs.)**: The distance to which the tool is retracted after the operation has been completed.
- Safe Distance (Abs.): 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. This represents the top of the part.
- 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 distances entered are shown as crosses on the drawing.

Editor NC Functions	Backp	lot	File Com	CIMCO	Trans	smissio		CNC-Ca		notorat	vings(t	INC-C	aic Turr	ing Tu	itorial 1	cud -	1					W	6	- C	8 X
New Drawing Save Drawing Setup CNC-Calc File		×1 ×1 ×2	Modify	2 \$ • •	· · ·	/ 44/ / 44/ / A4/ / A4/ / A4/	· / Drav			Center Radius D	Two Point	ls 🖌		Rec H	tanguli ole Pat	ar Bolt tern Patter	Hole Pa) ar Bolt attern	A a a ext	ISO T	Furning	urning) 📴 Opera		- 🗐
CNC-Calc Turning Tutorial 1.cdd *	Tę Ur	titled *																							4 Þ 🗙
End Drilling																									
Comment: M8 TAPPING																									
Clearance Height:																									
Retract Height 3	1																								
Start Depth:	1	ľ						Ń								./		Ì							
End Depth:															/										
Parameters														-					 /				\neg	X.	
Export Editor Export Clipboard																							Ī		
Bement Info																					-		-0		Z N
Lienent Into																									
																							0.00		
		fachine i	7007			Machir	, ne Zero	V.		v				*								Pari	t Diam	ator	
		rocaline.	2010 2.	0.0		machi	10 2 610		0.0										Use Y-A	xis Sub	stitution		- Dialin	2001	

Click on the button **Parameters** in the End Drilling pane to open the parameters dialog. Enter the following values into the parameter dialogs shown below. Notice that only a few parameters are available for configuring. It is no longer possible to use pecking, feedrate, and dwelling options because the *Operation Type* is selected as **Tapping CW G84**.

Drilling Parameters

Operation Type Tapping CW G84	 Canned Longhand 	
Dwell Use Upper Dwelling 500.0	Use Lower Dwelling	Tip Compensation Use Tip Compensation Tip Angle
Feedrates		114.0
Use Plunging Feedrate:	Use Retract Feedrate:	Drill Diameter 10.0
100.0	200.0	Tip Compensation
Pitch:		3.247
4.5		
Pecking ✓ Use Pecking		
Peck Clearance: 3.0	Peck Retract: 5.0	
First Peck:	Subsequent Pecks: 6.0	

- **Operation Type**: The operation type can be Drilling G83 or Tapping CW G84 for Canned cycles, and Drilling or Peck drilling for Longhand cycles.
- 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 tip.
- **Drill Diameter**: The diameter of the drill.
- **Tip Compensation**: The length of the tip of the drill. This is the length from the tip to where the drill reaches the full diameter.
- **Pitch**: The amount that the tool should move down for each revolution.

Click on **OK** to use the values and close the dialog.

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

14.8.2. Exporting the Toolpath and Backplot in the Editor

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.

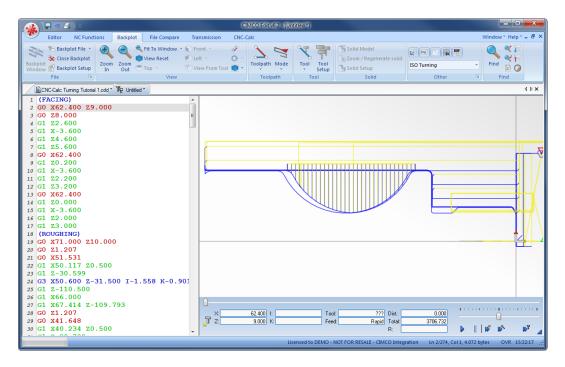
Change the window to the NC program and move the cursor to the very end by pressing Ctrl+End. Insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Paste** from the *Edit* toolbar in the Editor tab.

Now the NC program should look like the following screen.

🔬 🖬 🤊 🗉 🔹	CIMCO Edit v6.2 - [Untitled *]	×
Editor NC Functions	Backplot File Compare Transmission CNC-Calc Window* Help* = 5	8 ×
New Open Close Save Sa File	Iso Turning Iso Turning Select All X Del ~ ver Pint Global Stup Our Paste Our Redo iso Find Previous Find Previous iso Find Previous Find Previous	
CNC-Calc Turning Tutorial 1.cdd	d* 👸 Untiled *	×
NC-Assistant Description:	245 GO X34.000 246 GO X60.00 247 GO X20.752 248 G32 Z-23.000 F3.000 259 GO Z6.000 251 GO X20.752 252 G32 Z-23.000 F3.000 253 GO X34.000 253 GO X34.000 254 GO Z6.000	-
Modify Cycles / Macros	235 (DRILLING) 255 (DRILLING) 257 (50 X3.000 257 (50 X3.000 258 (51 Z=8.000 F200.000	
Program Start and End Tool change with Constant Surf Tool change with Constant RPM Program comment	26 G0 Z-5.000 26 G0 Z-5.000 26 G1 Z-14.000 26 G2 G2 -9.000	
G00 Rapid move G01 Linear move G02 Clockwise arc move G03 Counterclockwise arc move G04 Dwell	262 G0 Z-9,000 263 G0 Z-11.000 264 G1 Z-20.000 265 G0 Z10.000 265 G0 Z10.000	
G10 Data setting mode G11 Data setting mode cancel G18 Z≺ plane designation G20 Inch data input	267 GO X0.000 Z10.000 268 GO Z3.000 269 MO3 270 G32 Z-12.000 F125.000 M05	
G21 Metric data input G27 Reference point return che G28 Return to reference point	271 G32 Z3.000 F125.000 M04 272 G0 Z10.000 273 M03 274	
	Licensed to DEMO - NOT FOR RESALE - CIMCO Integration In 266/274, Col 1, 4.072 bytes OVR 15:31:23	3

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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.



Now a window like the one below will appear.

14.9. Parting-off the Part

In the following section of the tutorial we will generate a NC program for parting-off the part. In order to select the Cutoff operation, the below described steps must be performed.

14.9.1. Create a Cutoff Toolpath

Ensure the file type (NC Format) for our Cutoff example program is ISO Turning.

Then select the feature Cutoff Turning by clicking on the **Cutoff** Ficon in the *Turning Operations* toolbar.

ISO Turning 👻	
	1
Turning Operations	

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

	Cutoff	Tuming	
	Comment:		
		CUTOFF	
	Distance Z:		
		-110	1 2
	Start Depth X:		
		50	^{X1}
	End Depth X:		
		0	Ϊ Χ _ε
S	how Toolpath	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.
- **Distance Z**: The position at which the part will be cut perpendicularly to the Z-axis.
- Start Depth X: The depth at which the actual operation is started.
- End Depth X: The final depth of the operation.

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

The three values entered are shown as a vertical line on the drawing.

🔬 🖬 🤊 🗉) 🔹				CIMCO	O Edit v6.	2 - [C:\	CIMCO	CIMCOE	dit6\Drav	vings\CI	NC-Calc	Turning	g Tutoria	1.cdd *	1								□ X
Editor NC Functions	Backplo	t f	ile Com	pare	Transm			C-Calc													Winde	ow≚ Hel	lp*= 8 ×
Prevent Save New Drawing Save Setup CNC-Calc File V		×r ×r ×*	Modify	Ø ≰ • •		**	∕⊿ ⊥⊢	II ⊗ ∠ ⋌ ◆ □ pints / Li		Points	O シ 、 / Circles	インノ	Rectange Hole P	ular Bolt attern Patte	Hole Pa		A Text Entry Te	<u>d</u>	ISO TU			erations	• 🔳
CNC-Calc Turning Tutorial 1.cdd *	Ty Untit	tled *																					4 Þ 🗙
Cutoff Turning																							
Comment: CUTOFF																							
Distance Z:	· · ·																						
-110 -110 Jack Start Depth X:																							
End Depth X:	· _	+					-								. 1		-Y						
							\sim																
Show Toolpath Parameters								$\overline{\}$															
Export Editor Export Clipboard													/						_				_
Experteduce Expertemptione																	~	-					
Element Info	• • • •																						
		-																					_
																							4
		-	-	-										-	-	-	-			-			
	Ma	ichine Z	ero Z:	0.0		fachine	Zero X:	0.0									 u	Jse Y-A	xis Subsl	itution	Part Dia	ameter	
None										Licens	ed to DE!	40 - N	DT FOR R	ESALE - I	CIMCO I	Integrati	ion	Z: -58.	361 D:5	4.379		OVR	15:44:21 ;;

Click on the button **Parameters** in the Cutoff Turning pane to open the parameters dialog. Enter the following values into the parameter dialogs shown below.

Cutting Tab

Lathe Cutoff Parameters	— X
Cutting Comer Geometry	
Tool Definition Tool Side	Tool Comer Radius
H H	Tool Width 3.0
Entry/Exit Entry Amount	Retract Amount
5.0	5.0
Finish Compensation Compensation Type Computer	T
Compensation Side Right	
Pecking Use Pecking	Pecking
	OK Cancel

Use this tab to configure cutting parameters for the operation.

- **Tool Side**: Use these two icons to define the right side of the tool as the cutting side.
- Tool Corner Radius: The corner radius of the tool.
- Tool Width: The width of the tool used for the operation.
- Entry Amount: The length that the tool will start the cut above Start Depth X.
- **Retract Amount**: The length that the tool will pull free along the radial direction at the end of the cut.
- **Compensation Type**: The compensation type that is used for the operation. The two most commonly used are *Controller* or *Computer*.
- Compensation Side: The compensation side that should be used in the operation.
- Use Pecking: By selecting this option the operation will be performed with pecking movements.

• **Pecking**: When *Use Pecking* is selected, use this button to open a new window to define the cut-off pecking parameters.

Corner Geometry Tab

This tab is used to configure the geometry of the corner - sharp, round, chamfer.

	er Geometry None			
۲	Comer Radius Comer Radius			
	Corner Radius	2.0		
0	Chamfer			
	- Chamfer Angle		Outher Radius	
		2578.3101		1.0
	Width		Inner Radius	
		2.0		1.0
	Height	2.0		
	rance Cut			
	Use Clearance Cut pover:		Stock To Leave Z:	
		1.0		0.0
	Use Pecking		Pecking	

- None: Select this option to perform a straight down cut with no corner machining (square corner).
- **Corner Radius**: Select this option if a rounded corner should be machined with the specified *Corner Radius*.
- **Chamfer**: Select this option if a chamfered corner should be machined with the specified *Angle* and *Width/Height*. Round corners for the chamfer can be defined with the *Inner/Outer Radius* values.
- Use Clearance Cut: Check this option if a clearance cut should be done to clear the corner fillet/chamfer before the cutoff operation is performed.
- Stepover: This is the amount of material removed in each clearance cut.
- Stock to Leave Z: Is the amount of material that will be left in the Z-direction after the clearance operation is performed.

- Use Pecking: By selecting this option the clearance cut will be performed with pecking movements.
- **Pecking**: When *Use Pecking* is selected, use this button to open a new window to define the clearance pecking parameters.

Click on **OK** to use the values and close the dialog.

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

14.9.2. Exporting the Toolpath and Backplot in the Editor

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 the cursor to the very end by pressing Ctrl+End. Insert the text from the clipboard, either by pressing Ctrl+V, or selecting the icon **Raste** from the *Edit* toolbar in the Editor tab.

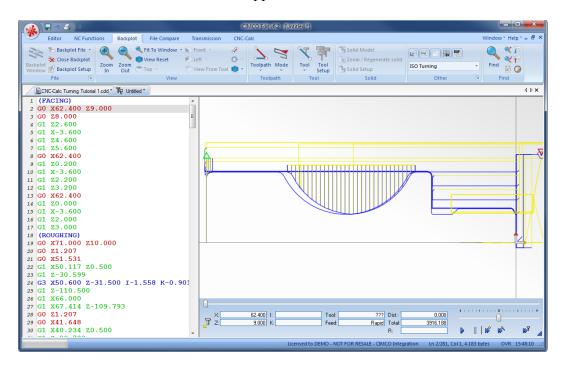
Now the NC program should look like the following screen.

🔬 🖥 a 🐔 🕫	CIMCO Edit v6.2 -	[Untitled *]	
Editor NC Functions	Backplot File Compare Transmission CNC-Calc		Window* Help* = 🗗 🗙
New Open Close Save St		Select All × Del → Other Select All	Go to Line/Block Number Find
CNC-Calc Turning Tutorial 1.cd	T the Obs	LUIL	4 b x
	260 G0 Z-5.000		11/2
NC-Asistant Description	261 G1 Z-14.000 262 G0 Z-9.000 263 G0 Z-11.000 264 G1 Z-20.000 265 G0 Z10.000 266 (MB TAPPING) 267 G0 X0.000 Z10.000 268 G0 Z3.000 269 M03		
Modify Cycles / Macros	270 G32 Z-12.000 F125.000 M05 271 G32 Z3.000 F125.000 M04 272 G0 Z10.000 273 M03		
Program Start and End A Tool change with Constant Surf	275 G0 X60.000 Z-110.000 276 G1 X45.800		
G00 Rapid move G01 Linear move G02 Clockwise arc move	277 GO X60.000 278 GO Z-109.267 279 GI X49.080		
G03 Counterclockwise arc move G04 Dwell G10 Data setting mode	280 GO X60.000 281 GO Z-108.533 282 G1 X49.948		
G11 Data setting mode cancel G18 ZX plane designation G20 Inch data input G21 Metric data input	283 G0 X60.000 284 G0 Z-107.800 285 G1 X50.000		
G27 Reference point return che G28 Return to reference point * Find Add	286 G3 X45.600 Z-110.000 I-2.200 K0.000 287 G1 X-0.400 288 G0 X60.000 289		E
		Licensed to DEMO - NOT FOR RESALE - CIMCO Integration	Ln 274/280, Col 1, 4.176 bytes OVR 15:47:23 ,;

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

To open the backplot window, click on the **Backplot** tab at the top of the Ribbon and then on the **Backplot Window** icon in the *File* toolbar.

Now a window like the one below will appear.



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