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

CNC-Calc v7 User Guide 1
License Information 6
1. Overview 8
1.1. Designed for usability and productivity 8
1.2. User Interface 9
2. Installation 10
2.1. Enabling CNC-Calc 10
3. Mouse Functions 11
4. CNC-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. Program Menu 27
5.1. File Menu 27
6. CNC-Calc Settings 28
6.1. Main Configuration 28
6.2. Color Configuration 30
7. Using CNC-Calc Help 31
7.1. Using Help In Dialogs 31
7.2. Printing Help Information 32
CNC-Calc Tutorials 33
8. Before You Start 34
9. Tutorial Mill 1 - 2D Construction of a Part for Milling 35
9.1. Open a New Drawing 36
9.2. Draw the Geometry 37
9.2.1. Draw a rectangle with round corners 37

9.2.2. Draw a circle defined by its center and radius 38
9.2.3. Draw vertical and horizontal lines 39
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. Save the File 44
10. Tutorial 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. 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. 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. Tutorial 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. Backplot Stock Setup 80
12. Tutorial 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 84
12.2.2. Milling of Letters 85
12.3. TrueType Letters Milling 89
12.3.1. Drawing the Text 89
12.3.2. Contour Milling of Letters 91
12.3.3. Pocket Milling of Letters 94
13. Tutorial Lathe 1 - 2D Construction of a Part for Turning 97

13.1. Open a New Drawing 98
13.2. Draw 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 102
13.2.4. Draw a line defined by its endpoints 103
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. Save the File 112
14. Tutorial Lathe 2 - Machining the Part 113
14.1. Open an Existing Drawing 114
14.2. Facing 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. Roughing the Part 119
14.3.1. Creation of Roughing Toolpaths 119
14.3.2. Exporting the Toolpath and Backplot in the Editor 125
14.4. Grooving 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. Finishing 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. Threading 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. Drilling 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. Tapping 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. Parting-off the Part 154
14.9.1. Create a Cutoff Toolpath 154
14.9.2. Exporting the Toolpath and Backplot in the Editor 158

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

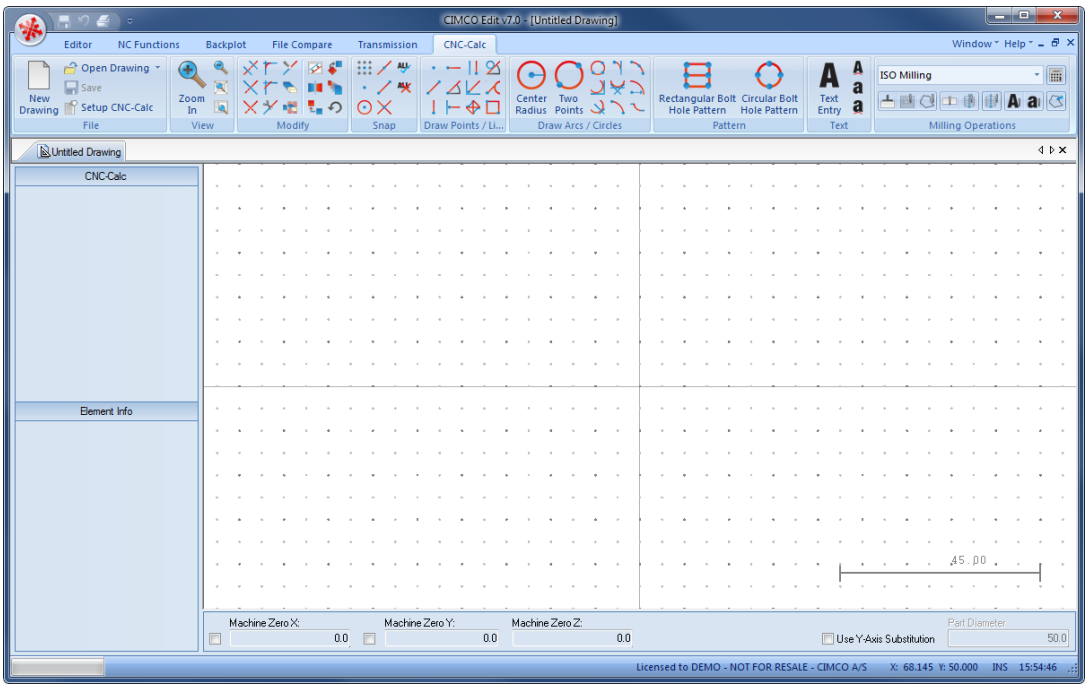
CIMCO CNC-Calc v7 is designed for ease-of-use that enables the user to draw contours fast and easily. It features common functions for drawing lines and circles in relation to the coordinate system and/or existing geometry. Functionality ranges from the plain "horizontal line" to the complex "circle 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.

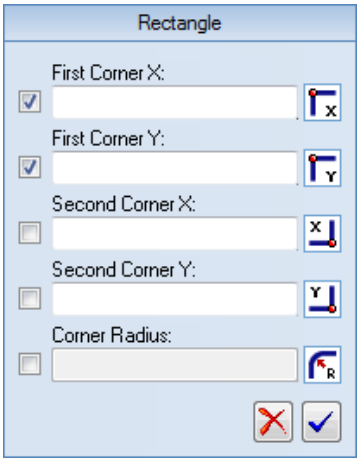
1.2. User Interface

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:

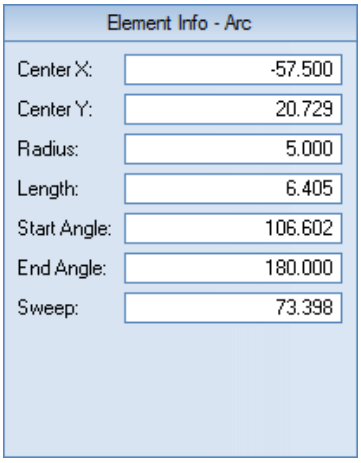


CNC-Calc main window.

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.



CNC-Calc pane with entry fields.



Element Info pane with information.

2. Installation

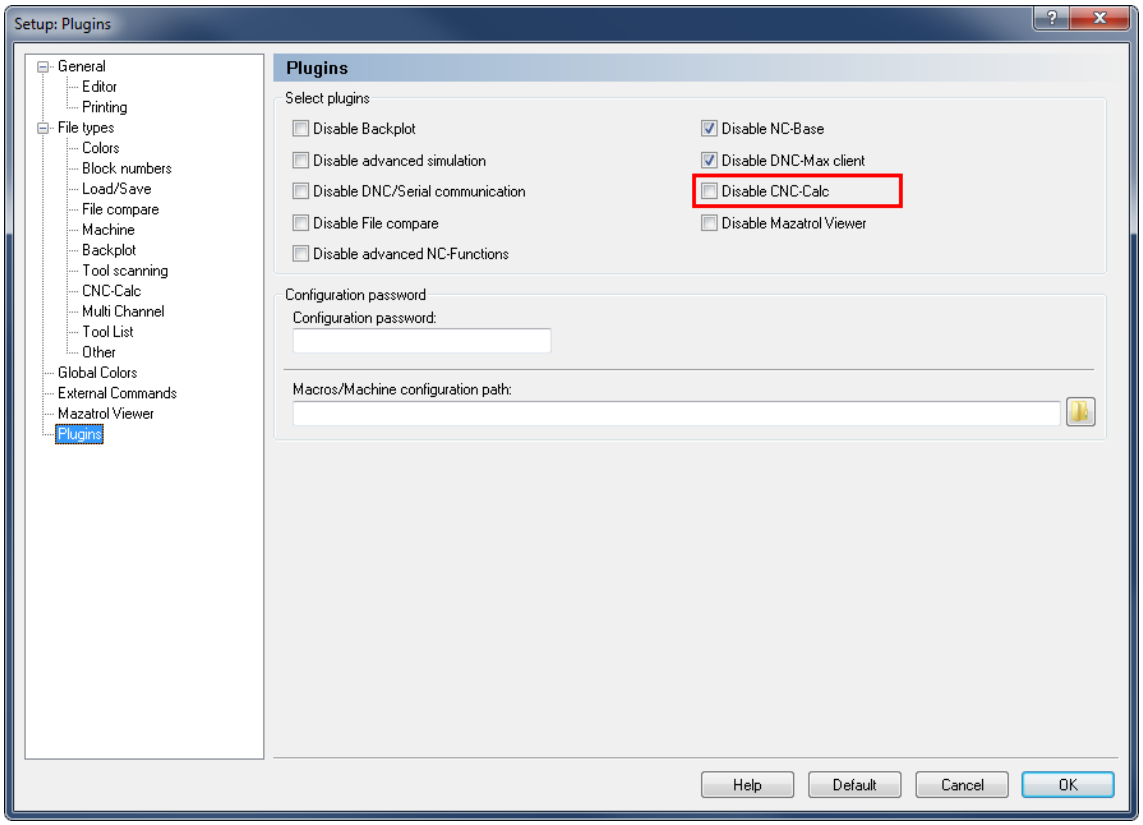
CNC-Calc v7 is installed as part of CIMCO Edit v7. Please see the CIMCO Edit v7 documentation for installation instructions.

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.



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 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.

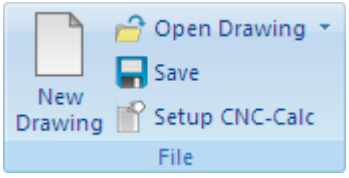
If the function can be activated through the Ribbon bar or a keyboard shortcut, the icon or the shortcut is displayed next to the command name.

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.



The File 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*.

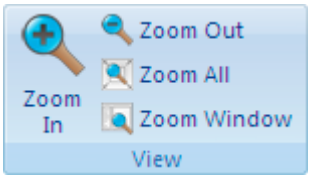
Setup CNC-Calc

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.



The View menu.

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

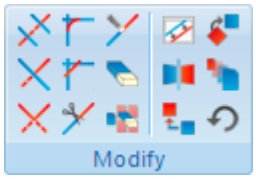
Use this function to fit the drawing to the graphics area. This can also be done by clicking the middle button on the mouse (on most mice, pressing the scroll wheel), or with *Ctrl+End*.

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.



The Modify menu.

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 **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.

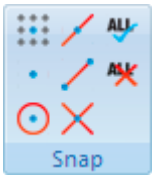
Undo

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.

Enable All Snap Types

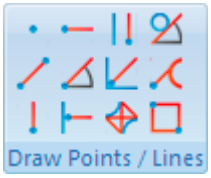
Activates all snap types.

Disable All Snap Types

Deactivates all snap types.

4.5. Points and Lines

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



The Draw Points/Lines menu.

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.

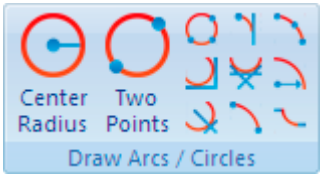
Rectangle

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.



The Draw Arcs/Circles menu.

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.

Tangent Three Elements

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

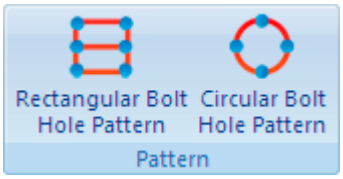
Use this option to define an arc by its center point, radius, starting angle, and end angle.

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.



The Patters menu.

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

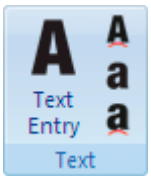
Use this option to define a circular bolt pattern.

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.



The Text menu.

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.

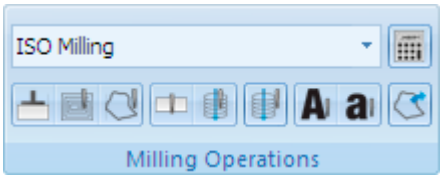
TrueType Text Circular Entry

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.



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.

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.

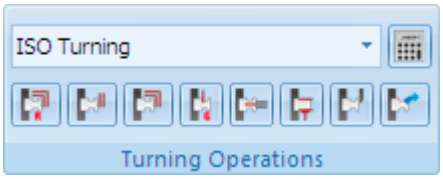
If a controller has smart canned cycles, for example a specific pocket operation, the user can create a macro in the editor to support this, and then export the actual contour for insertion in the canned cycle.

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.



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*.

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 icon 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.

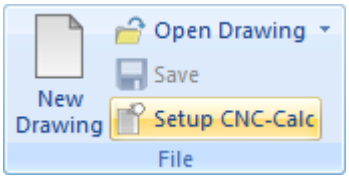
6. CNC-Calc Settings

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.

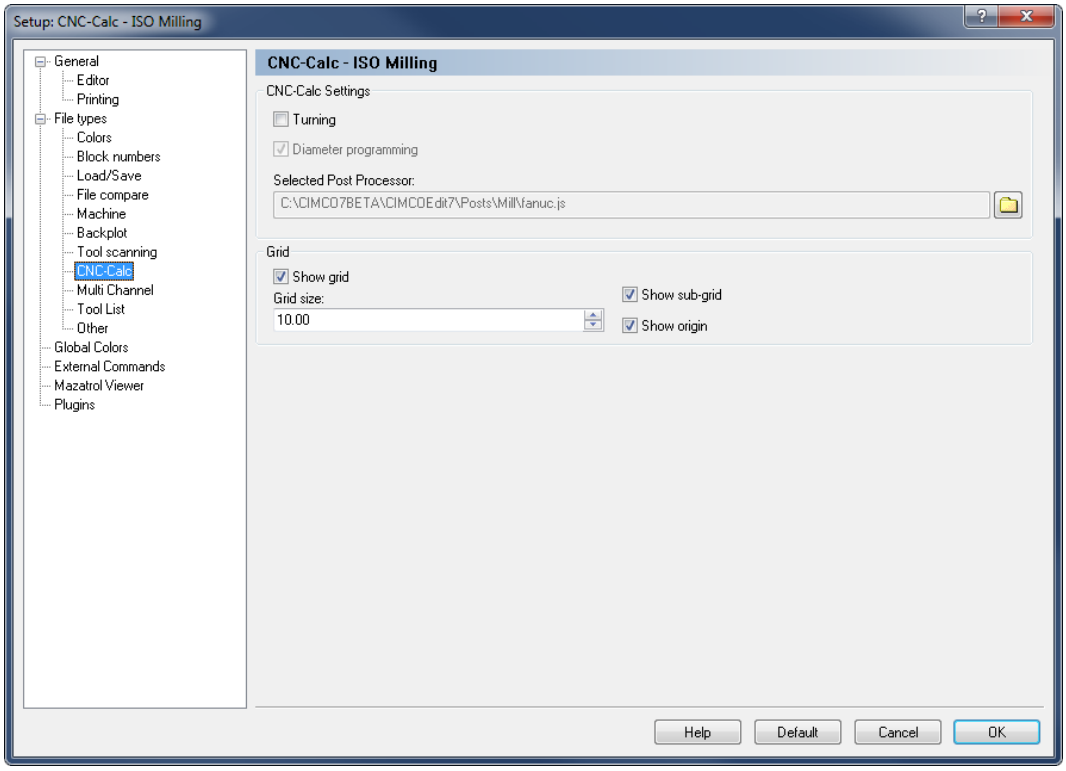
6.1. Main Configuration

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



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.



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 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

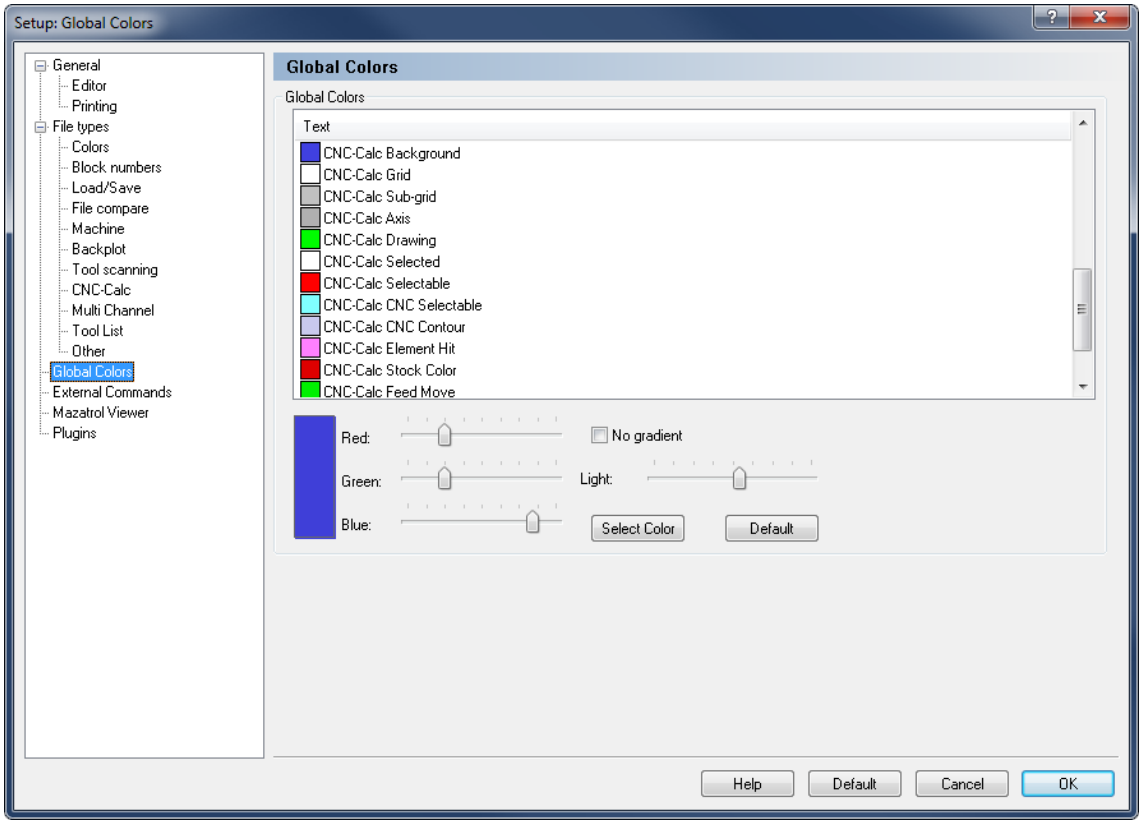
Select this option to have a sub-grid visible when zoomed in to a degree that shows few main grid points.

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.



CNC-Calc color configuration window.

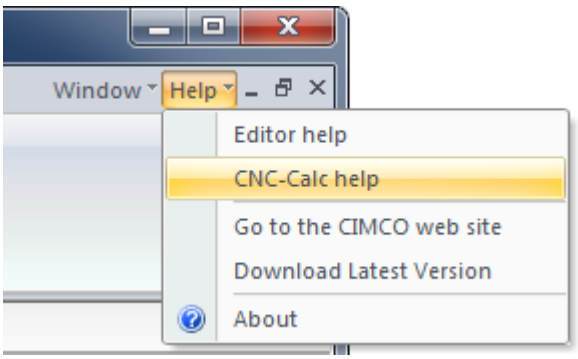
To change the color of a CNC-Calc element, either left click the element in the list to select it and then click the **Select Color** button, or you can double-click the element in the list. The color can then be picked from a standard palette, or a custom color can be defined.

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**.



Help menu.

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.

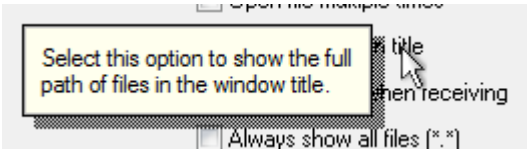
To get help for a particular dialog item, click the **?** at the top of a dialog box, as shown below.

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.

The question cursor.

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

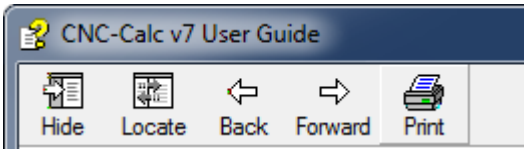


The pop-up information.

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



Help buttons.

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.

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

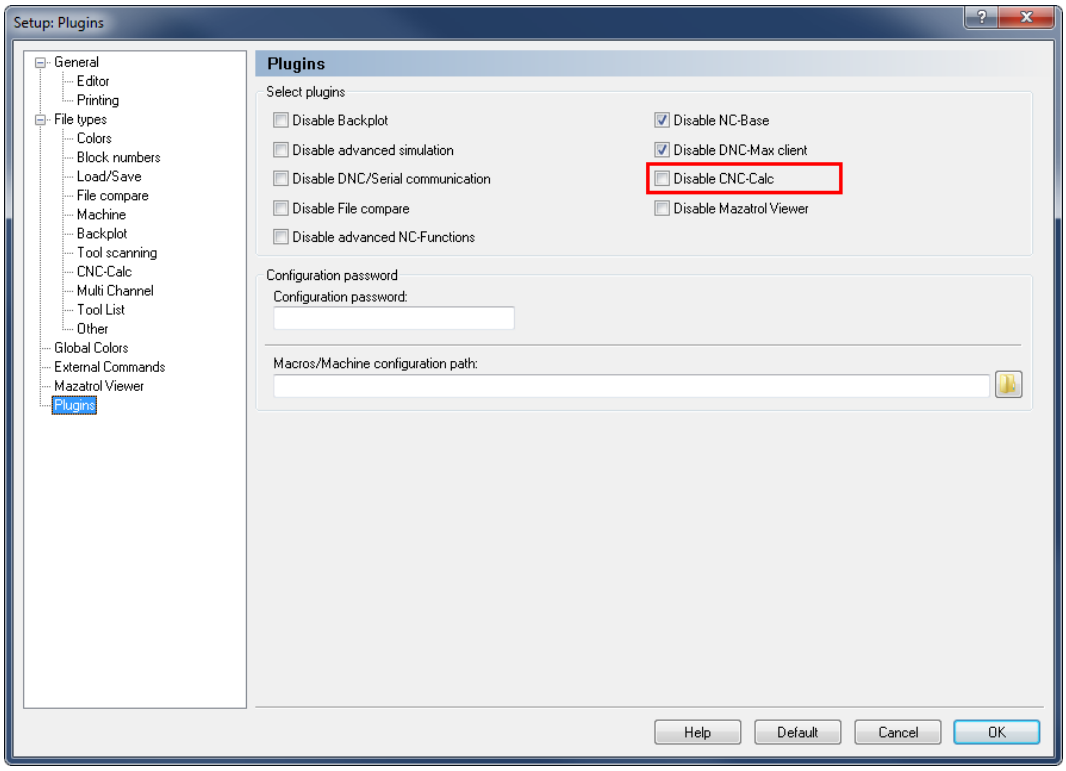
Do not run the generated NC programs as is!

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.

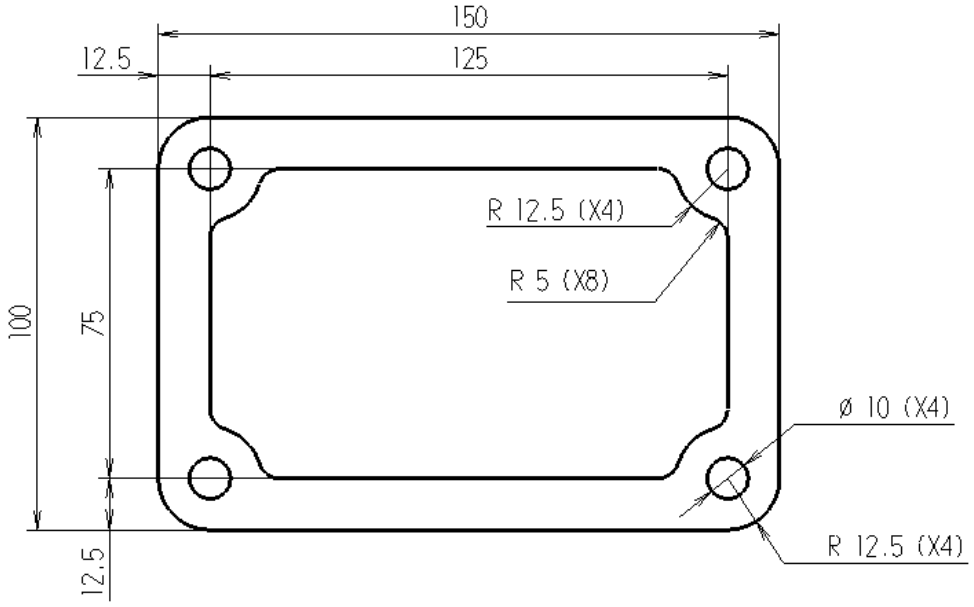
8. Before You Start

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.

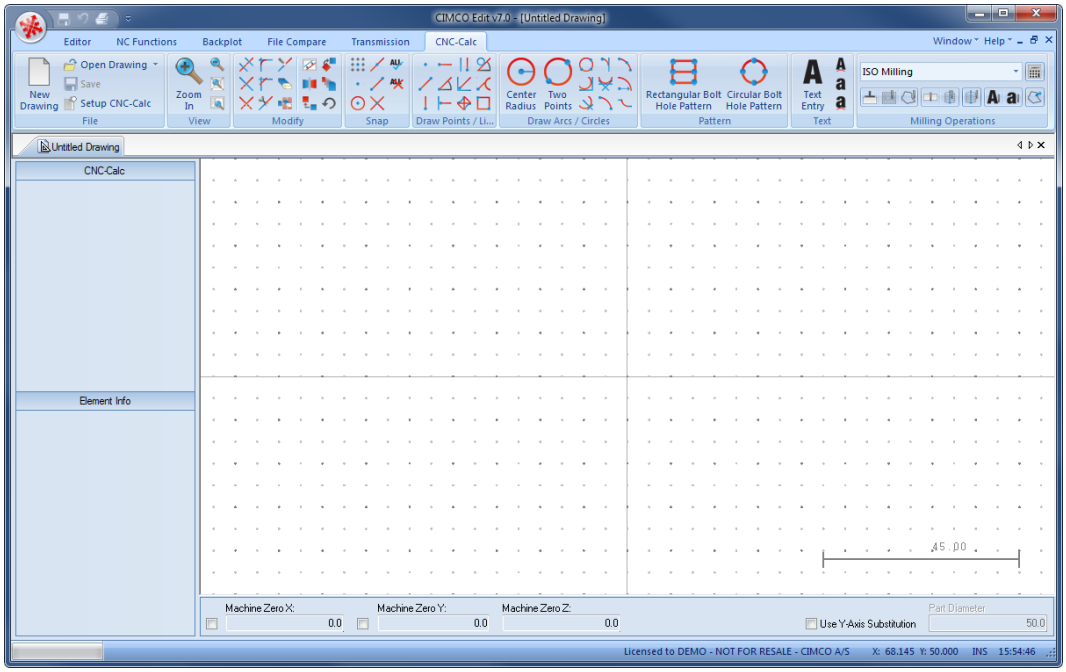
This tutorial demonstrates one of many ways in which the 2-dimensional part above can be drawn in CNC-Calc v7. Since the part consists of a number of similar elements and since its part-elements are symmetrical, only a subsection of the part needs to be drawn. The rest emerges from mirroring. Finally, joining the mirrored elements with straight lines completes the part.

This tutorial demonstrates the use of the following functions:

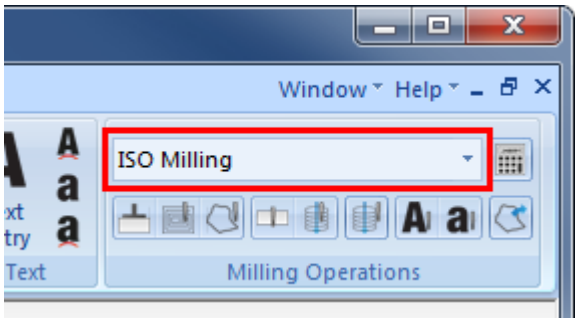
- Draw a rectangle with a corner radius.
- Draw a circle with known center and radius.
- Draw vertical and horizontal lines from known points.
- Offset a circle.
- Make curves between elements.
- Delete elements.
- Mirror elements about lines.
- Join end points with straight lines.

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:



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



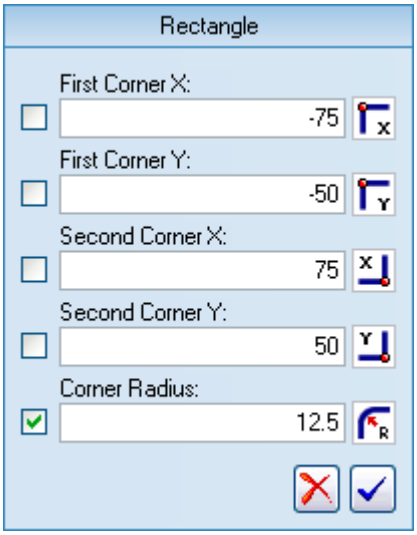
If you hold the cursor over an icon, a short description of the icon's functionality will appear.

You can change the colors of the drawing area by selecting **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.

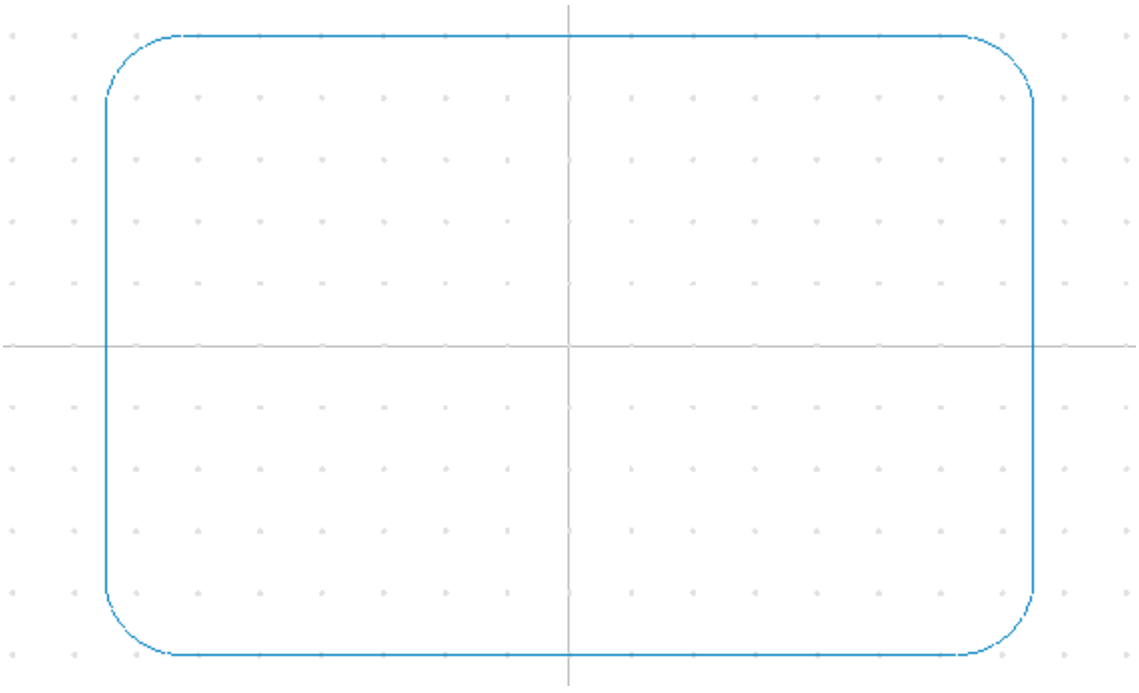
9.2. Draw the Geometry

9.2.1. Draw a rectangle with round corners

- Click on the **Rectangle** icon 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:

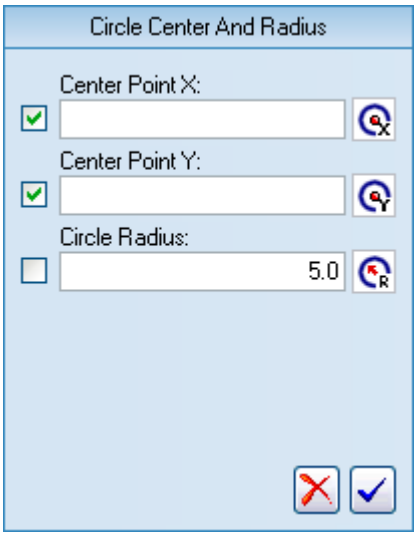


- Click on to approve the command and draw the rectangle.

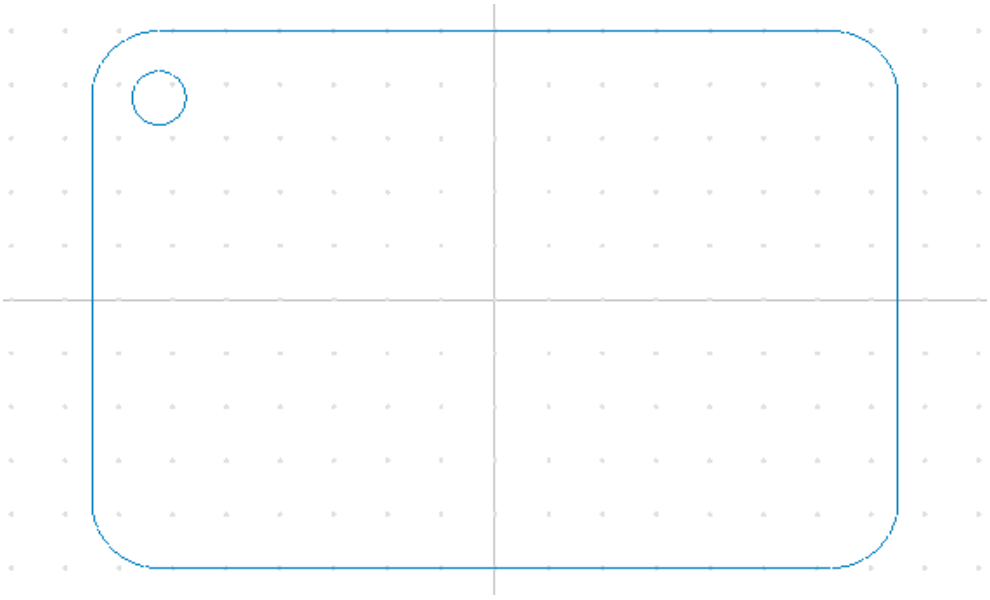


9.2.2. Draw a circle defined by its center and radius

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



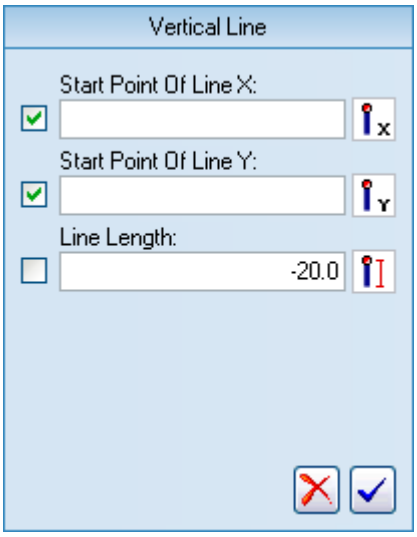
- Activate the function **Snap to Center Points** in the *Snap* toolbar.
- Snap to the center of the left topmost corner arc. The circle will automatically move to the center point when the cursor is close to it. The pointer also changes indicating snap to arc/circle center.
- Left-click to add the circle.



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.

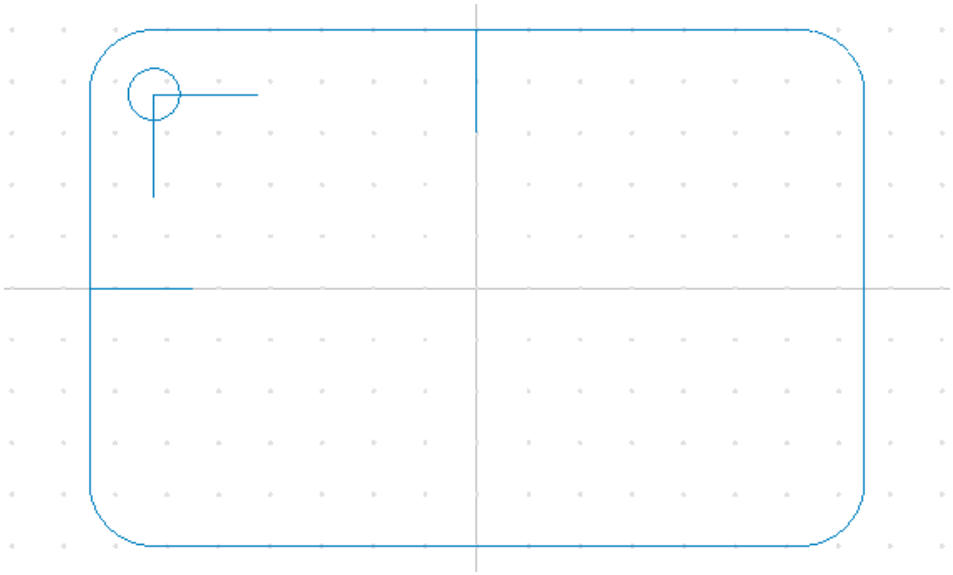


- Snap to the center of the circle drawn in the previous step.
- Click to add the vertical line.
- Snap to the midpoint of the topmost horizontal line (match up with the Y-axis).
- Click to add another vertical line with the same values. This one will serve as a mirror line for the mirroring of our part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

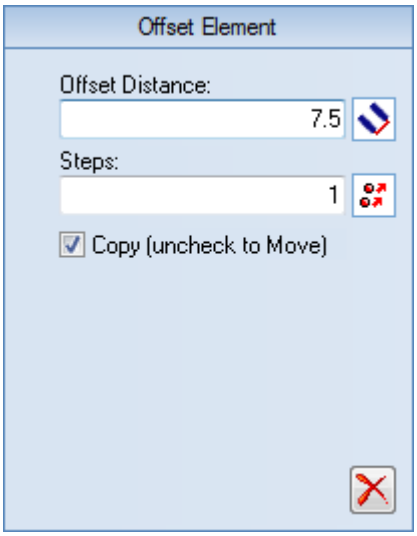
- Add then a horizontal mirror line from the center of the left vertical line (match up with the X-axis).



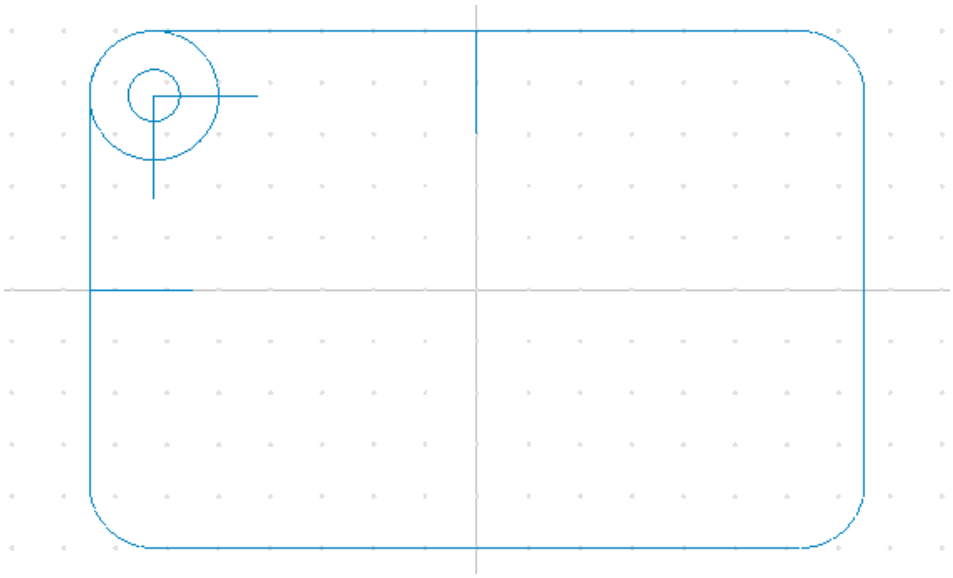
9.2.4. Offset a circle

- 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)*.

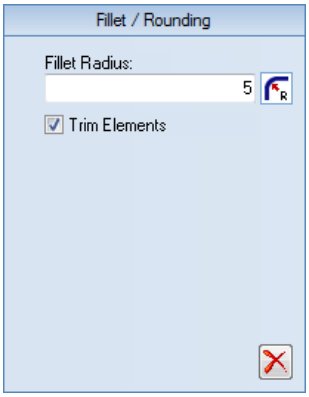


- Click on the circle and select the outermost of the appearing circles.

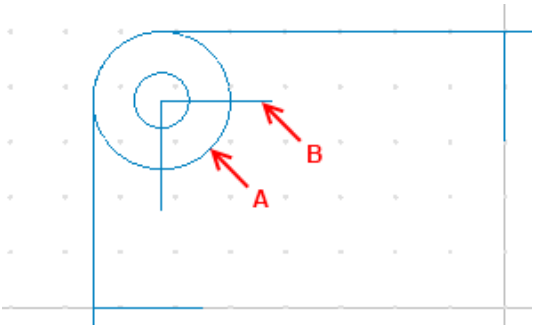


9.2.5. Create a fillet between elements

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



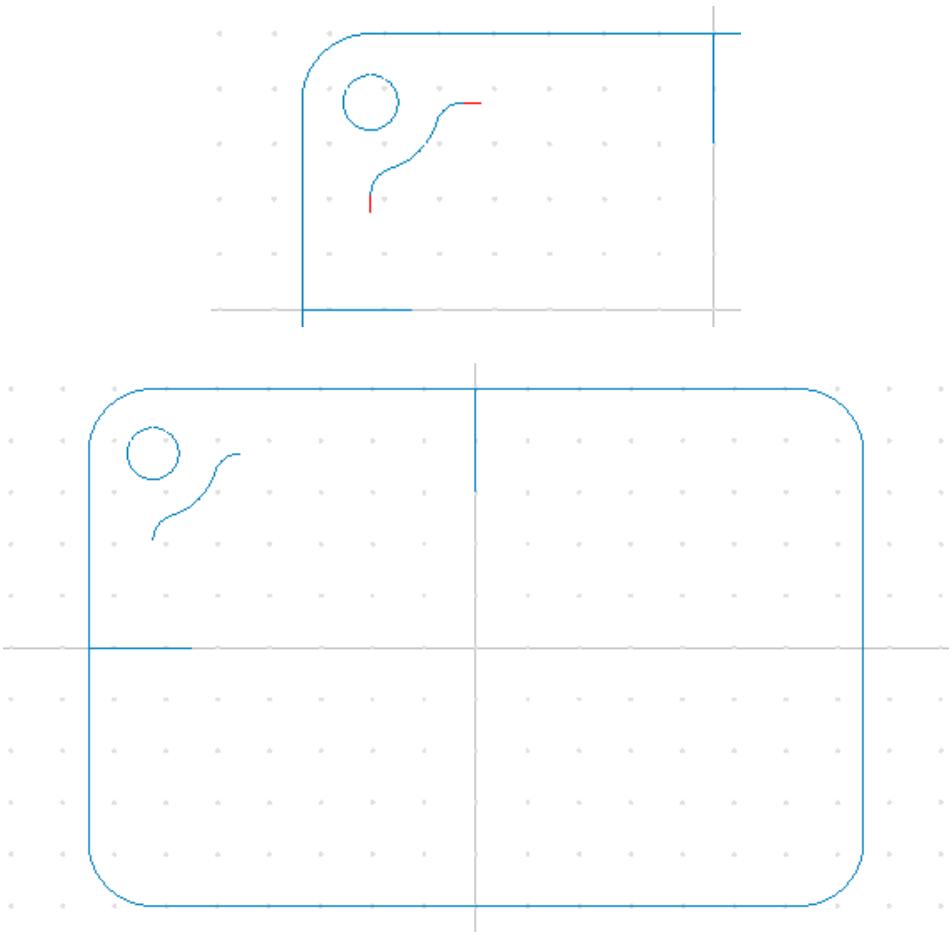
- To select the elements for the fillet operation, click on the circle by A and on the line by B.



- 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 and delete the two residual lines (the ones pointing away from the center of the circle).



9.2.6. Mirror elements

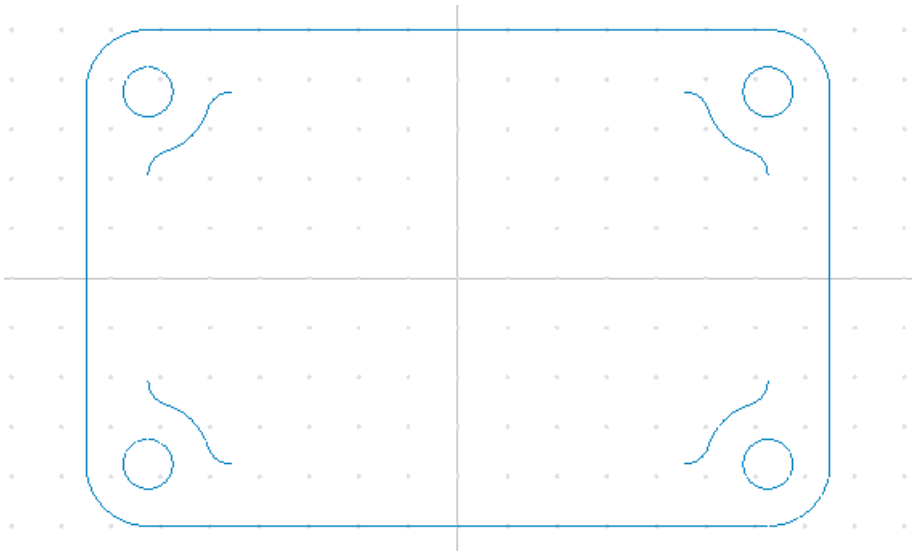
- Click on the **Mirror Elements** icon in the *Modify* toolbar.

- 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.

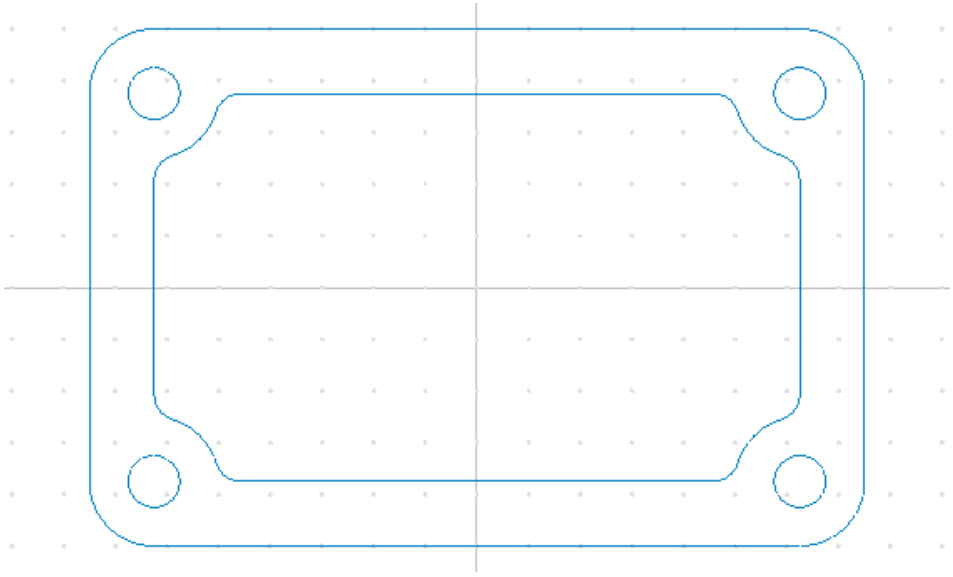
- Now do the same and mirror along the horizontal mirror line. Continue mirroring until your drawing looks similar to the one below.

- Click on the **Delete** icon and delete the mirror lines.



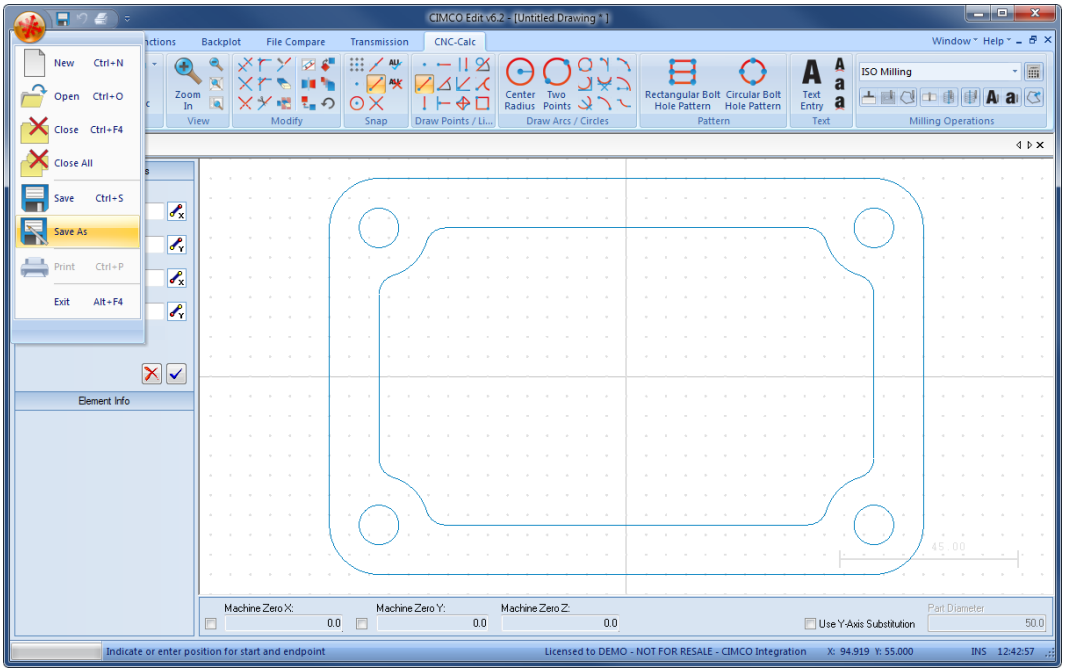
9.2.7. Connect the inner elements

- 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.



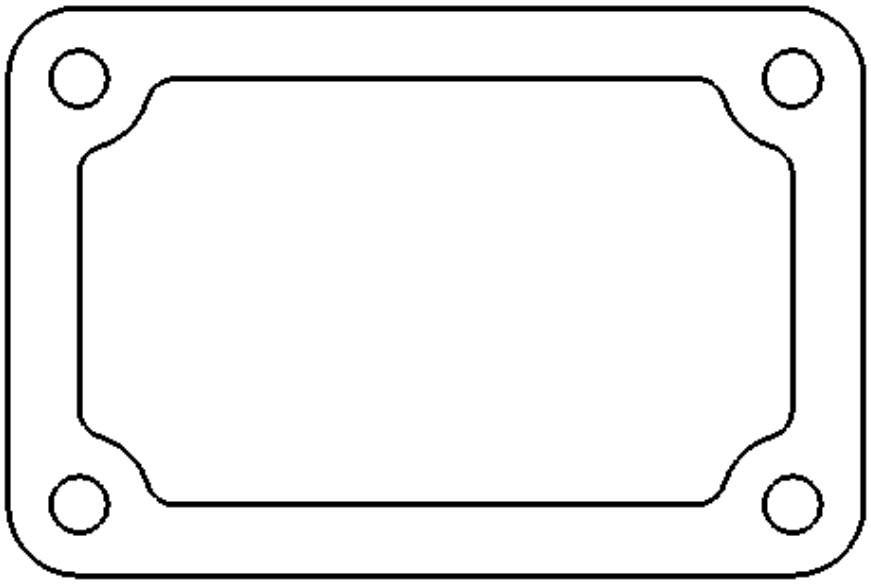
9.3. Save the File

Click on the **Main Menu** icon 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 NC-codes for various types of machining.

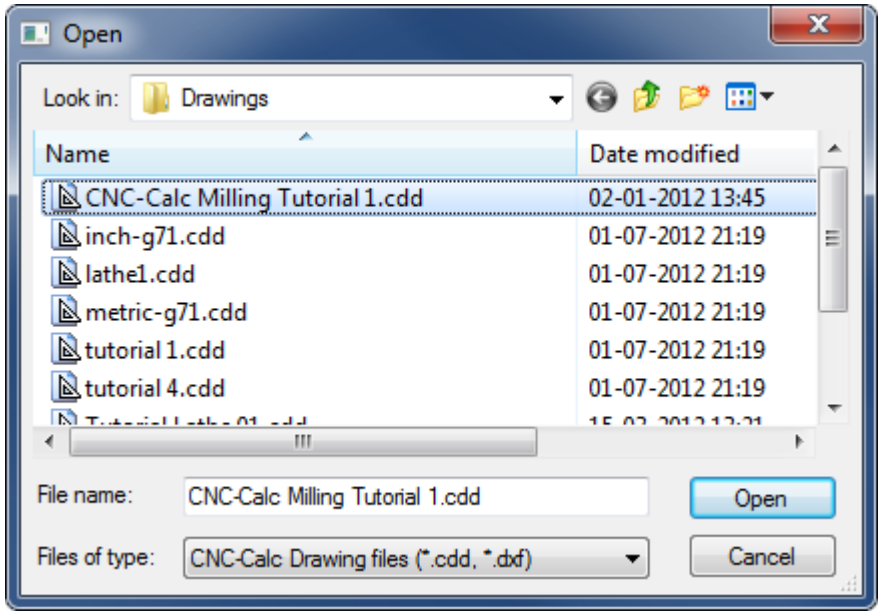
In order to produce the final part we will use the following operations:

- Face Milling
- Contour Milling
- Pocket Milling
- Drilling

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

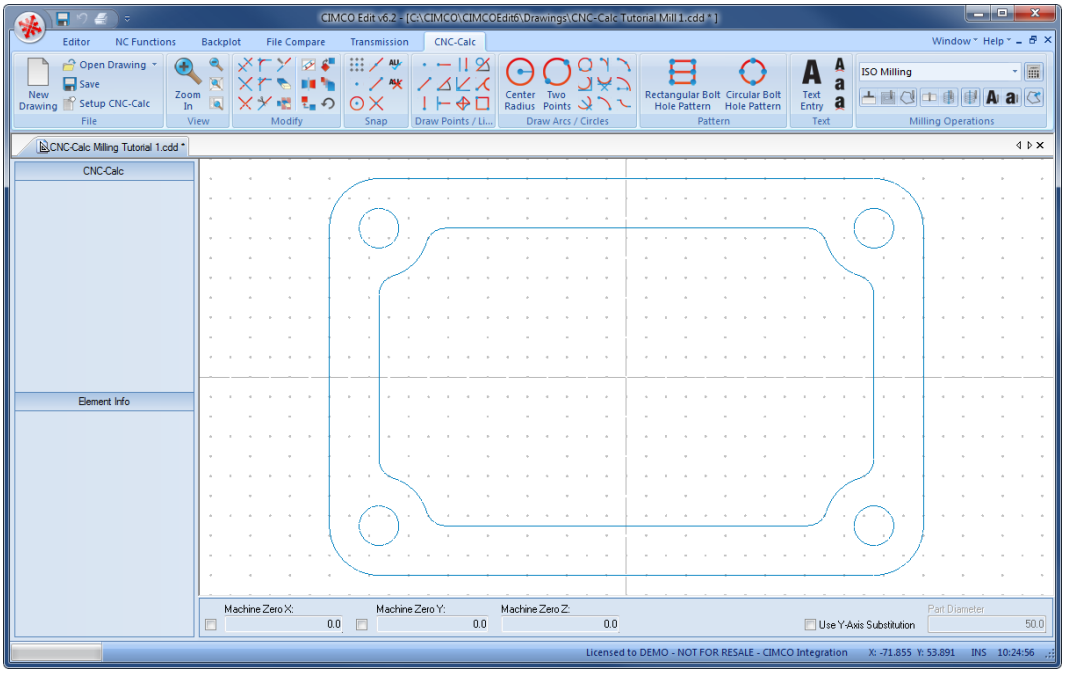
10.1. Open an Existing Drawing

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



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.



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.

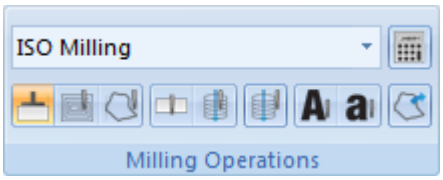
10.2. Face Milling

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

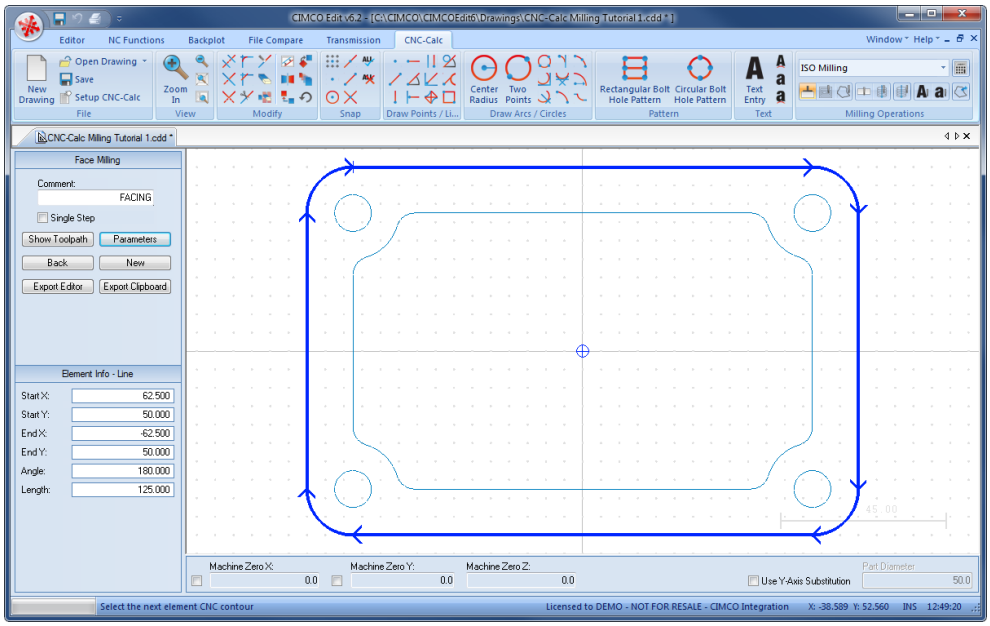
First, in the *Operations* toolbar, select the programming format of the NC-program in the *File Type* drop-down menu. Select **ISO Milling**.

Then click on the icon **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.

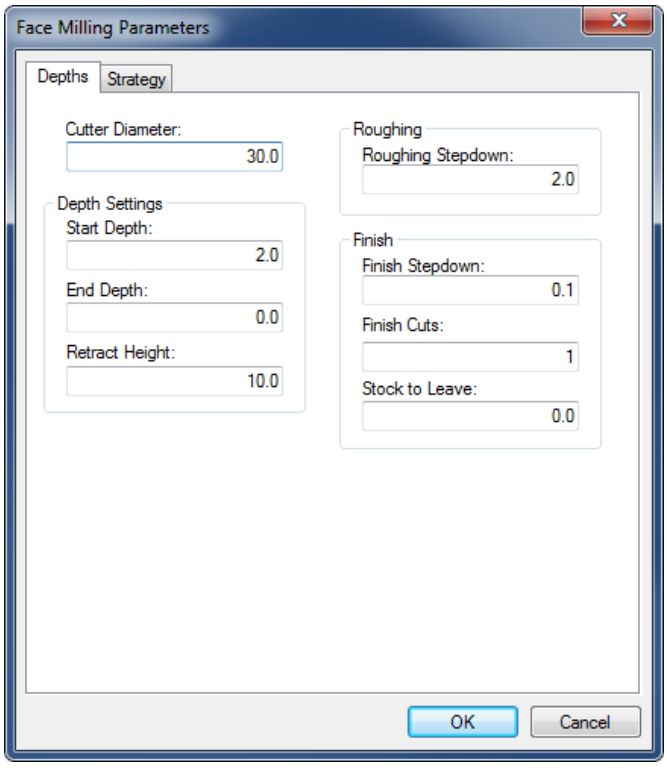
Click on the outlining contour of the drawing. This will select the bounding contour that the facing operation will operate on.



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

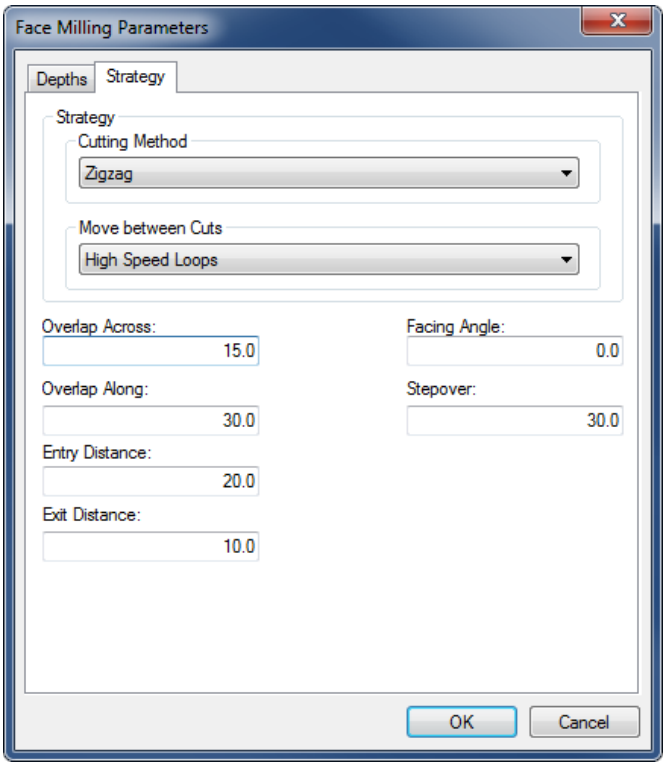
Enter the values into the dialogs as shown in the pictures below.

Depths Tab



- **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).

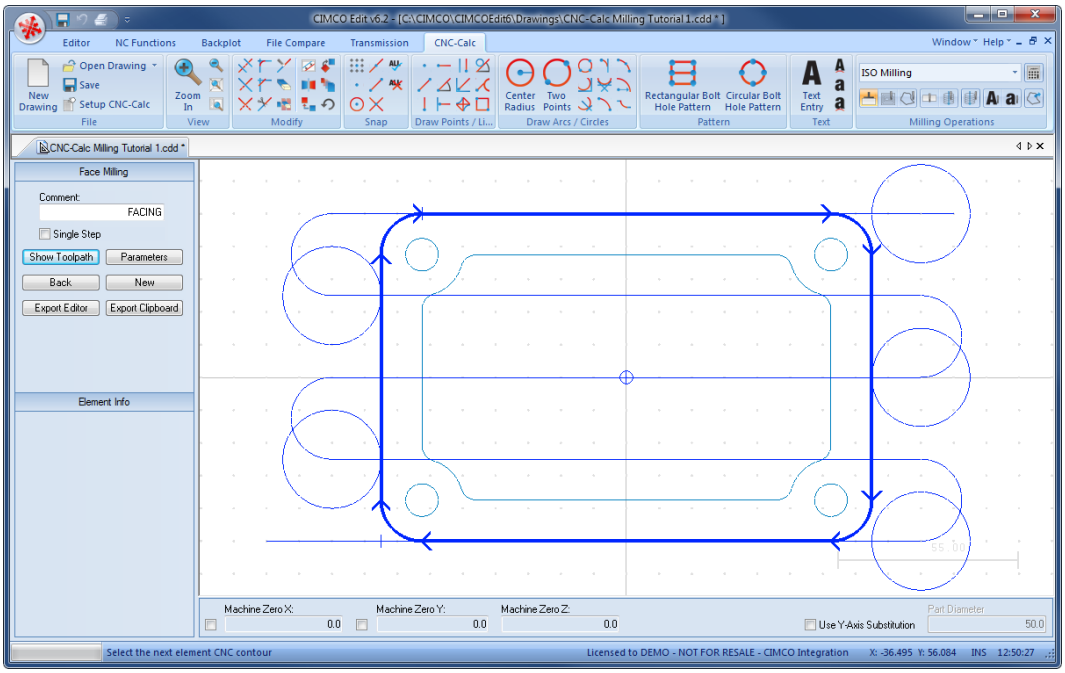
Strategy Tab



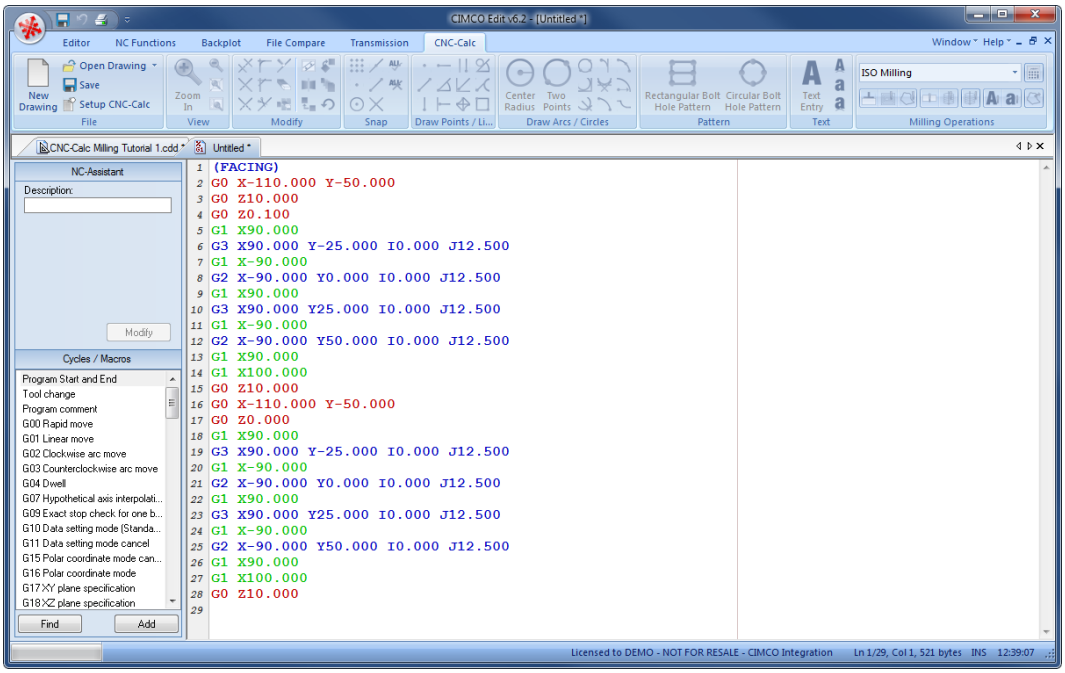
- **Cutting Method**: The method used to perform the face operation. It is possible to select *Zigzag*, *Climb*, or *Conventional*.

- **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.



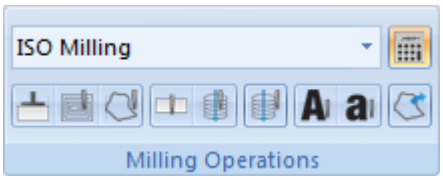
See section "Inserting a Tool with Feed and Speed Calculator" for information on how to insert a tool using the Feed and Speed Calculator.

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.

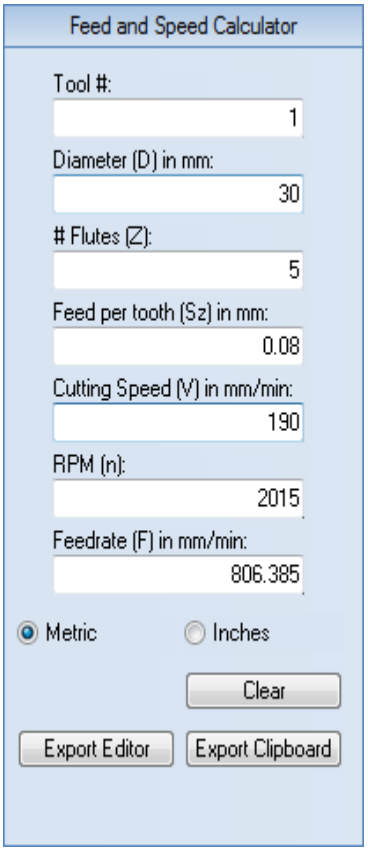
In the facing example, we use a face mill that we give the following characteristics: diameter is 30 mm, it has 5 flutes, a cutting feed of 0.08 mm per tooth, and a cutting speed of 190 mm/min.

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.

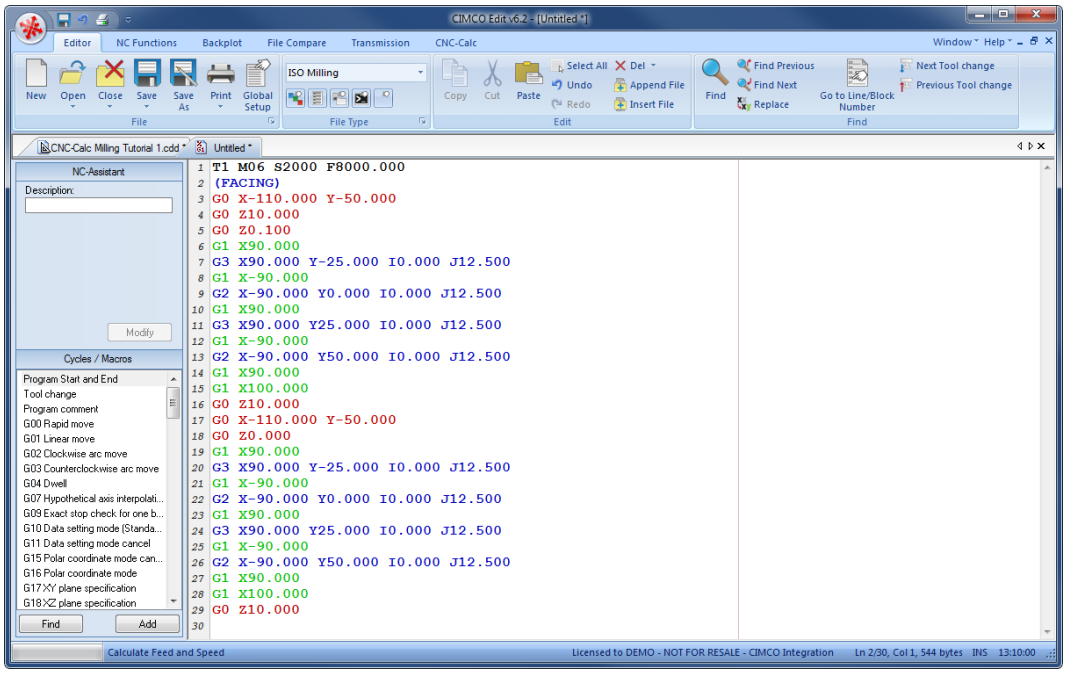
If we then want to have 2000 rpm and a feedrate of 800 mm/min instead of the calculated 2015 and 836.385, the value for the cutting speed will be updated to 188.5 mm/min.

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.

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

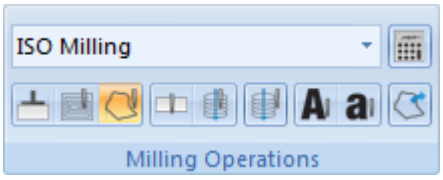


10.3. Contour Milling

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

10.3.1. Creation of Contour Toolpaths

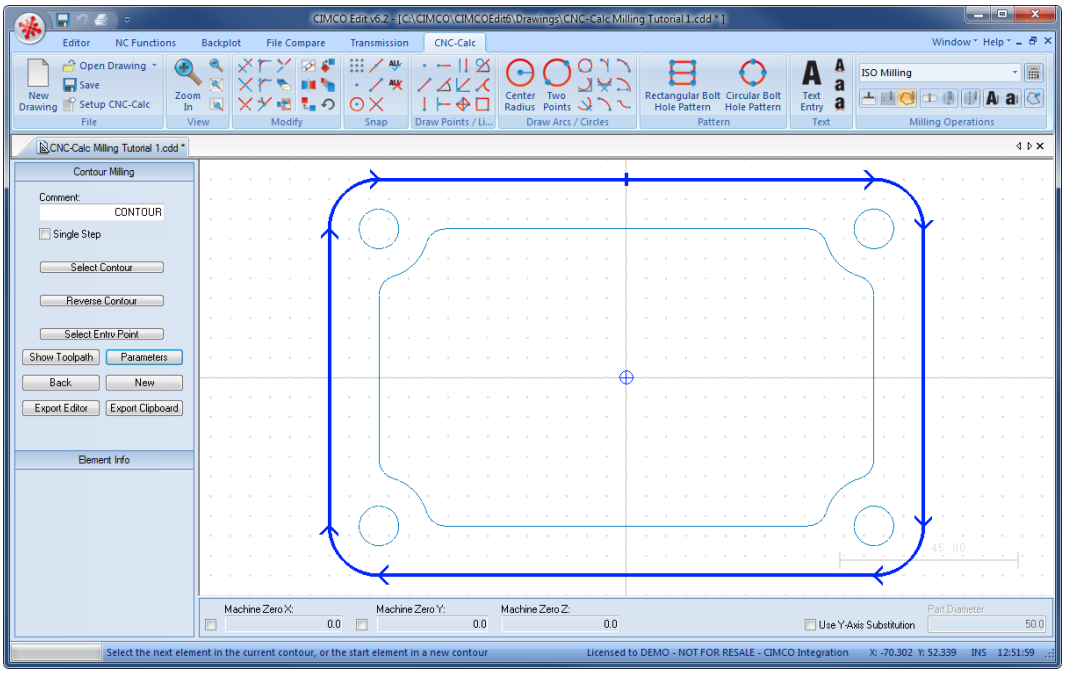
To begin the creation of an NC program for the contour operation, select **Contour Milling** to generate a CNC-toolpath for contour milling (ensure that **ISO Milling** is selected in the field *File Type*).



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.

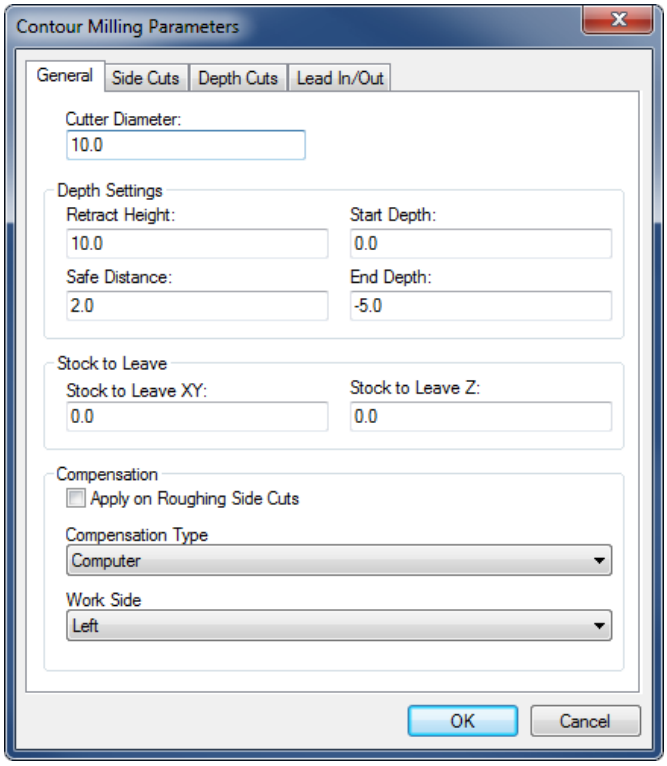


Click on the button **Parameters** in the CNC-Calc pane Contour Milling. This will open the configuration dialog for setting the contour milling parameters.

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.



- **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).

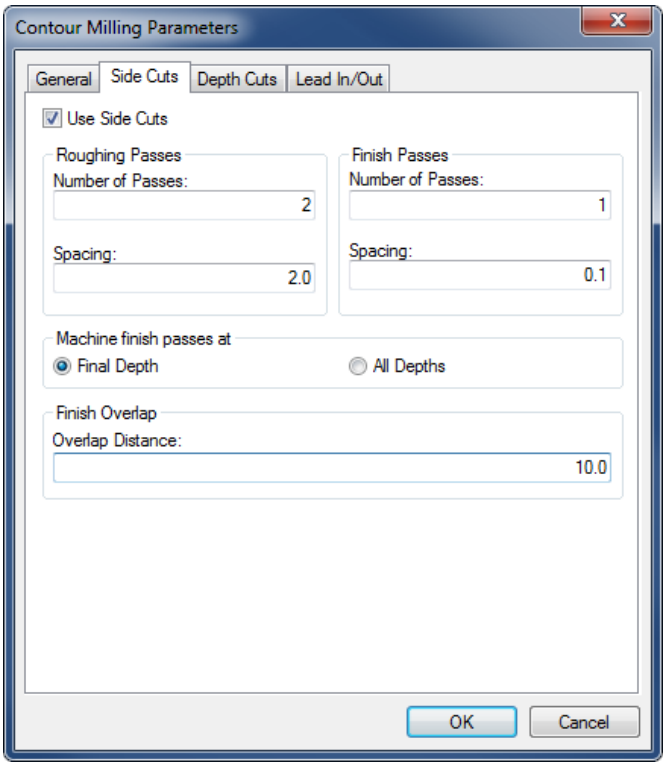
- **Apply on Roughing Sidecuts**: If this option is checked, the compensation type will be applied to both roughing and finishing side cuts. Otherwise computer compensation is used for roughing cuts, and the selected compensation type for finishing cuts.

- **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.



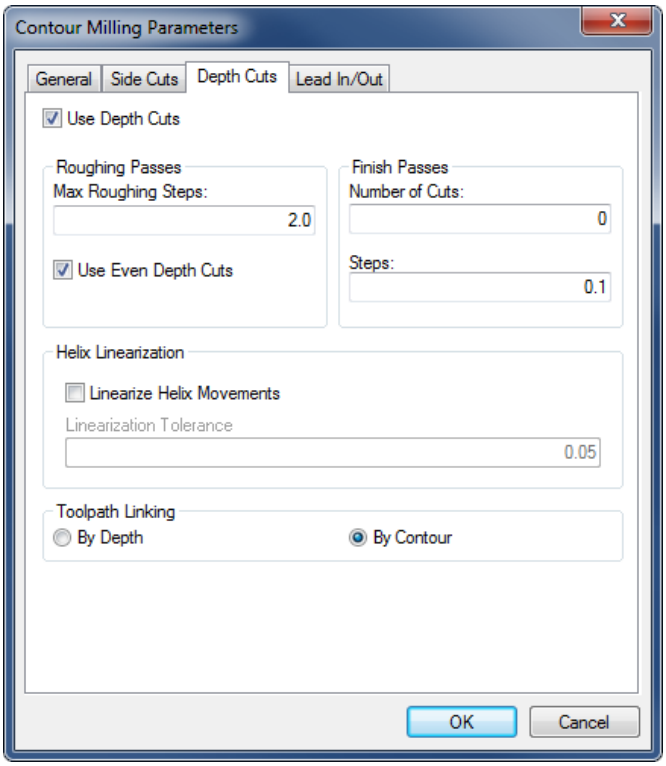
- **Use Side Cuts**: If this option is checked, the operation will perform the cuts defined by the parameters. Otherwise, only one cut at the final contour will be performed.

- **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.



- **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.

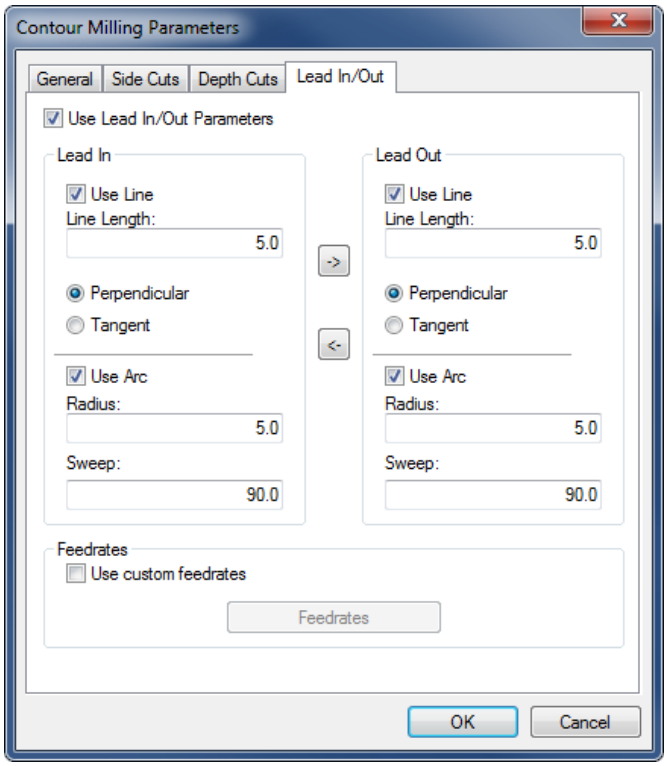
- **By Depth**: This is only used if multiple contours are milled in the same operation. If selected, the cut on each depth will be performed on all contours, before any cuts are made at a new depth.

- **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.

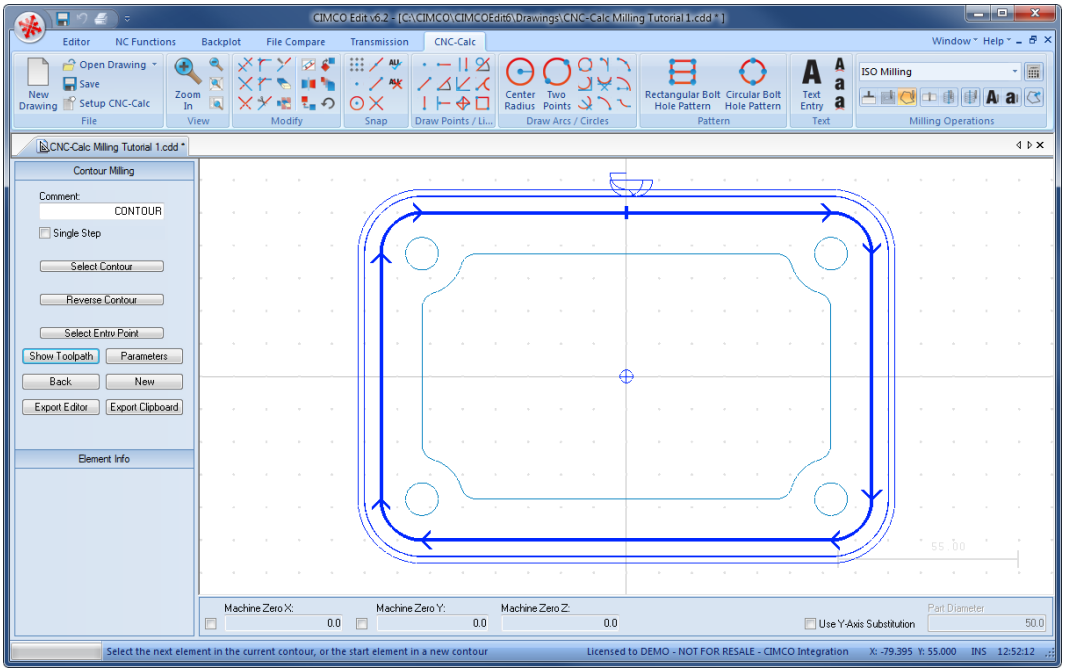
The use of lead in/out is optional, when the compensation is set to computer or none. It is however mandatory, when any compensation is performed by the controller.



- **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.



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

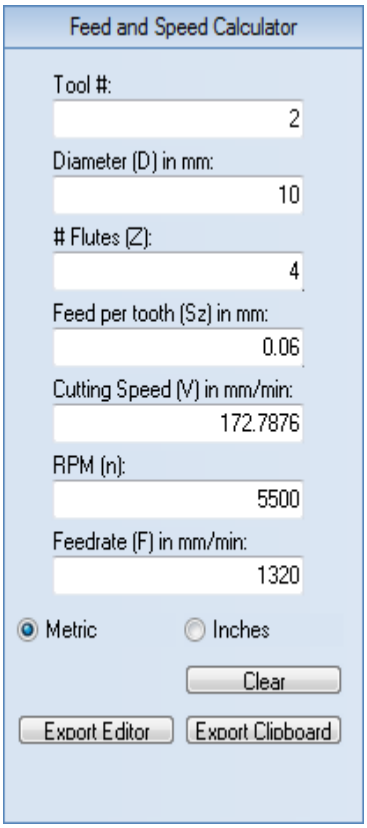
The NC program in the Editor now consists of two operations, and currently they are both made with the same tool. Now we need to insert a new tool for the contour operation. See section "Inserting a Tool with Feed and Speed Calculator" for information on how to insert a tool using the Feed and Speed 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.

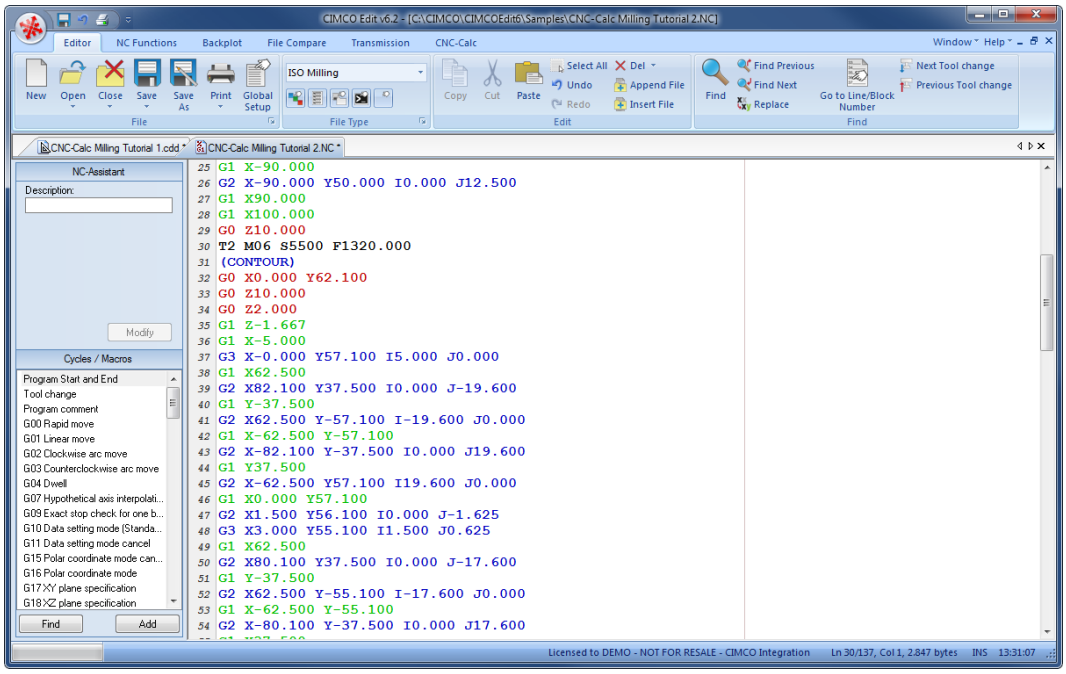
Click on the button **Export Clipboard** to copy the generated line to the Windows clipboard.

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** 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 **Paste** from the *Edit* toolbar.

The NC program should now look like the following:

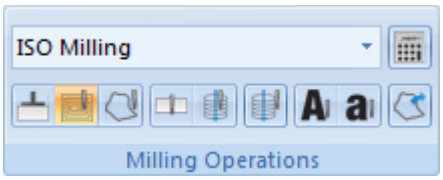


10.4. Pocket Milling

CNC-Calc v7 can generate pocket milling toolpaths.

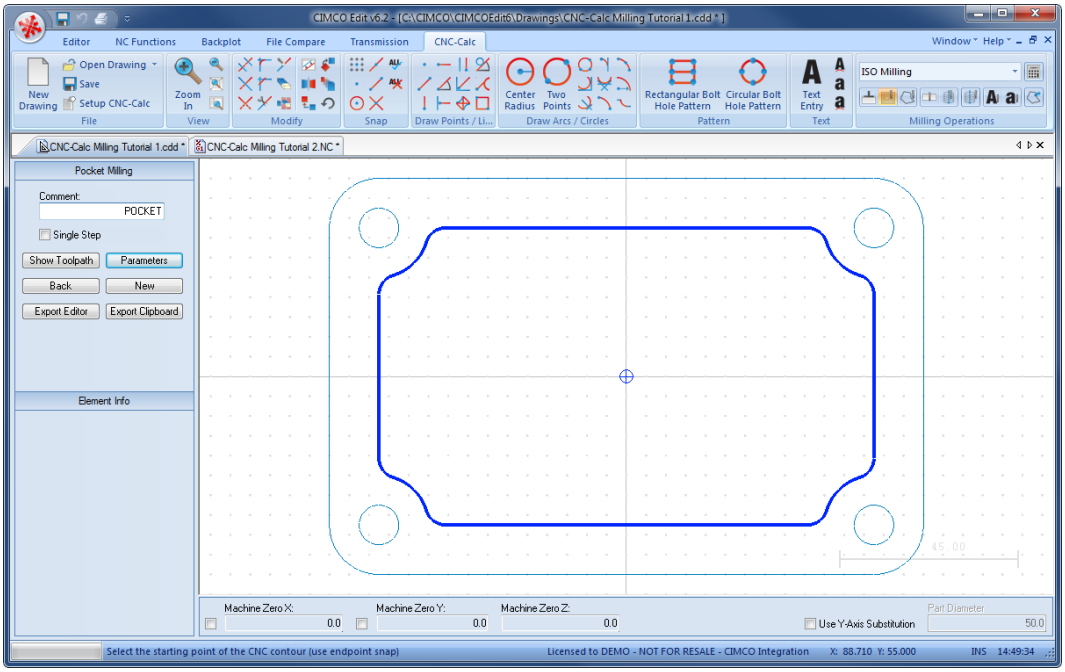
10.4.1. Creation of Pocket Toolpaths

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*).



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.

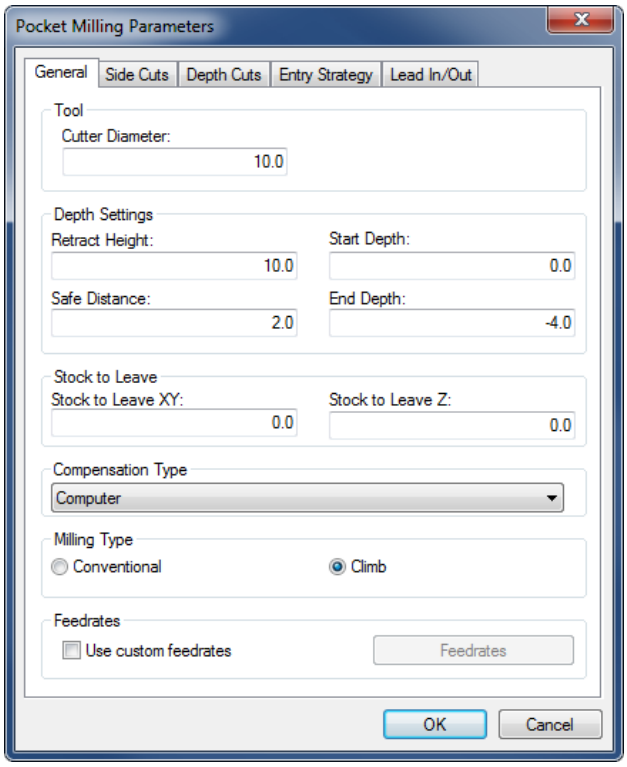


Click on the button **Parameters** in the CNC-Calc pane Pocket Milling. This will open the configuration dialog for setting the pocket milling parameters.

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.

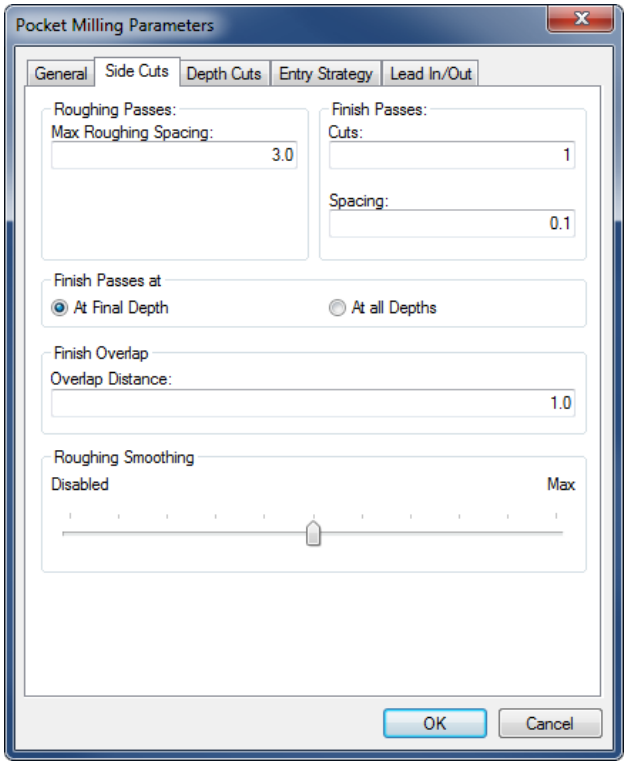


- **Cutter Diameter**: The diameter of the used tool.
- **Retract Height**: The height to which the tool will move between contours, and where it will stop at the end of the operation.
- **Safe Distance**: The distance above the part, where the feedrate will change from rapid to cutting speed.
- **Start 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.



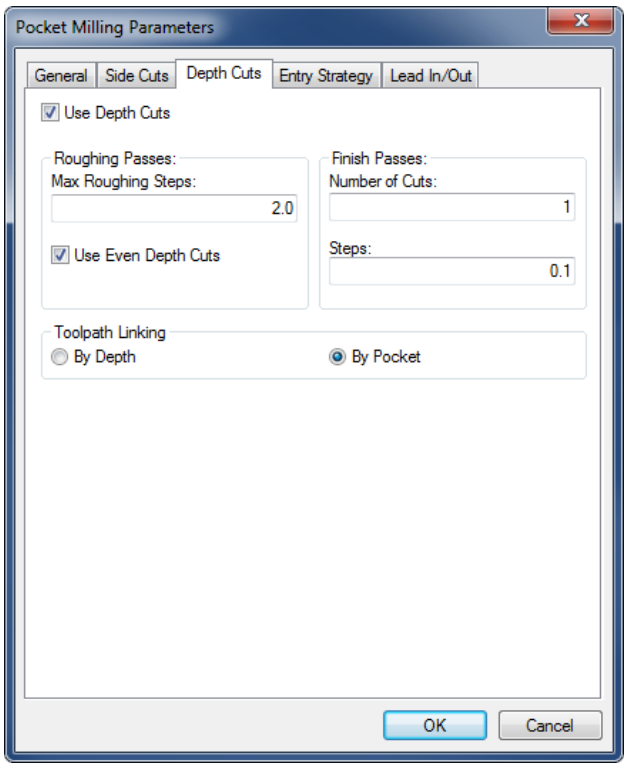
- **Max Roughing Spacing**: The maximum side stepover used in the roughing of the part.
- **Cuts**: The number of finishing side cuts in the operation.
- **Spacing**: The distance of each finishing pass.
- **At Final Depth**: If this radio button is checked, the finishing passes will only be taken at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



- **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.

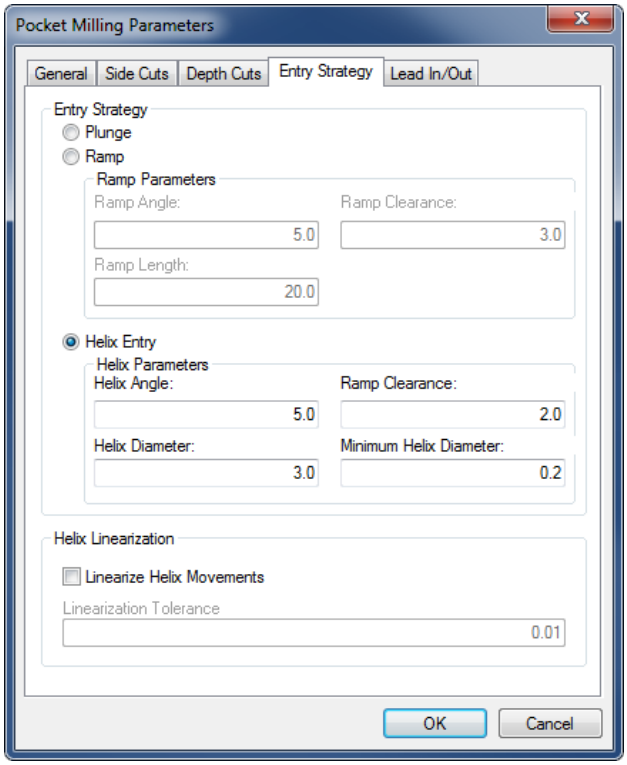
- **Steps**: The distance of each finishing pass.

- **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.



- **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.

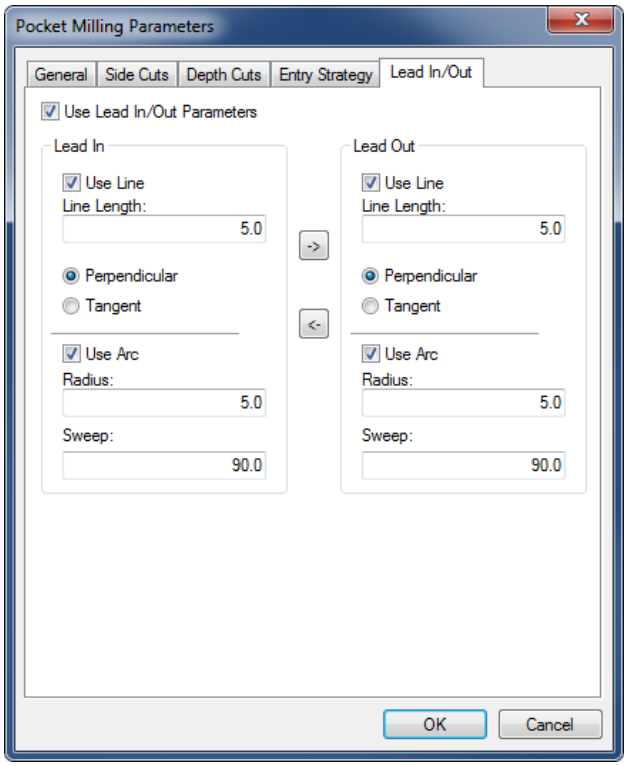
- **Linearize Helix Movements**: Some machines cannot make helix movements, and if this option is checked, all helix movements will be converted to lines in the NC operation.

- **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.



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

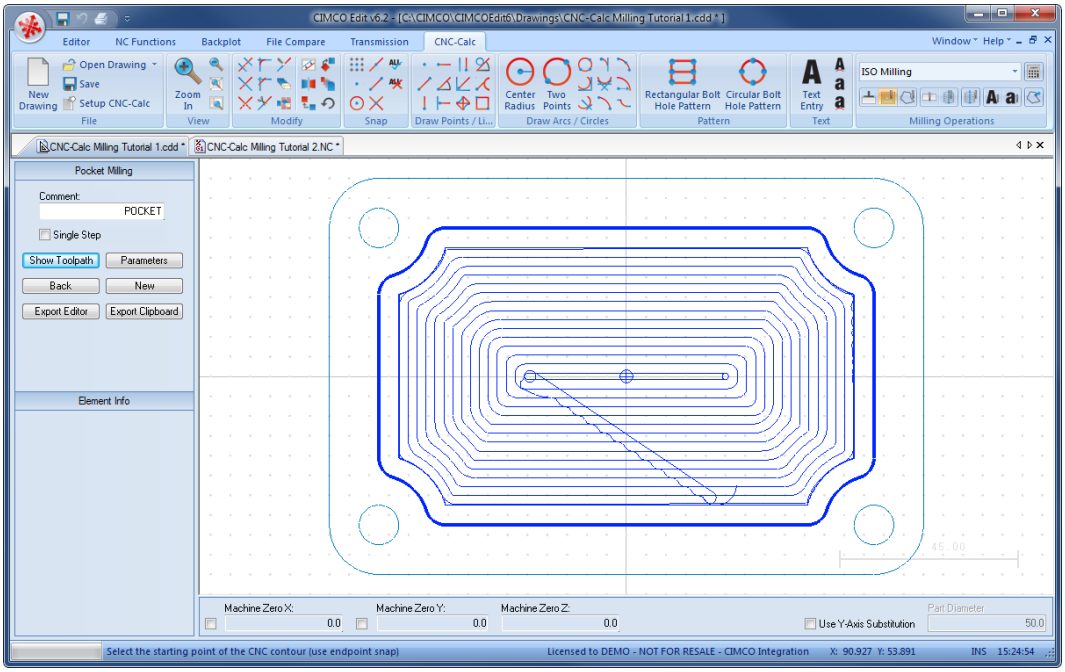
- **Use Line**: Enables or disables the lead in/out lines.

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

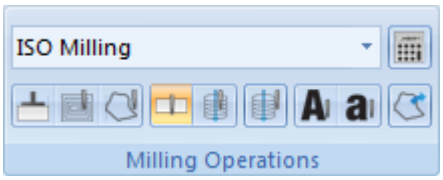
The NC program in the Editor now consists of three operations, and since we use the same tool for the contour and pocket operations we will not insert a tool before the pocket operation.

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

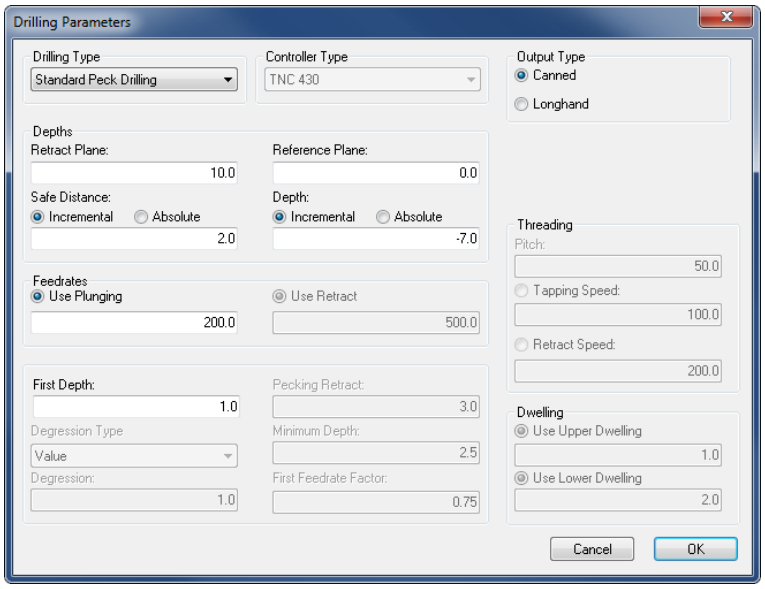
To start creating a NC-program for the drilling operation, select the function **Drill Holes** in the *Milling Operations* toolbar to generate a drill cycle (ensure that **ISO Milling** is selected in the field *File Type*).



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.

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



- **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.

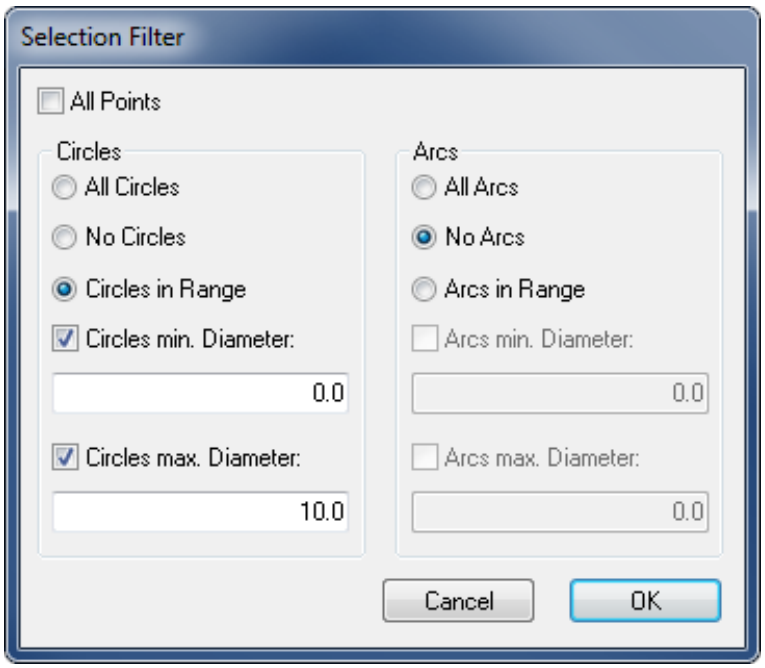
Notice that in this example it makes no difference if **Incremental** or **Absolute** is selected for *Safe Distance* and *Depth*, since these incremental values refer to the *Reference Plane*, which is 0.

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.

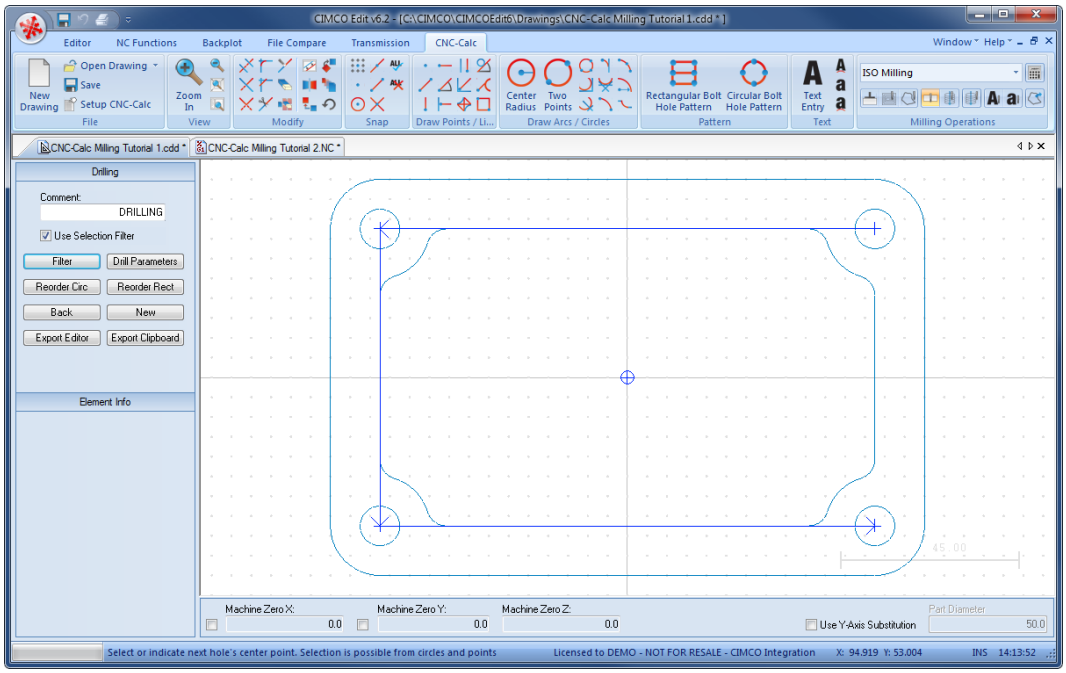
In the following we will use the filter to select the corner holes, but not any of the arcs.

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.



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

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

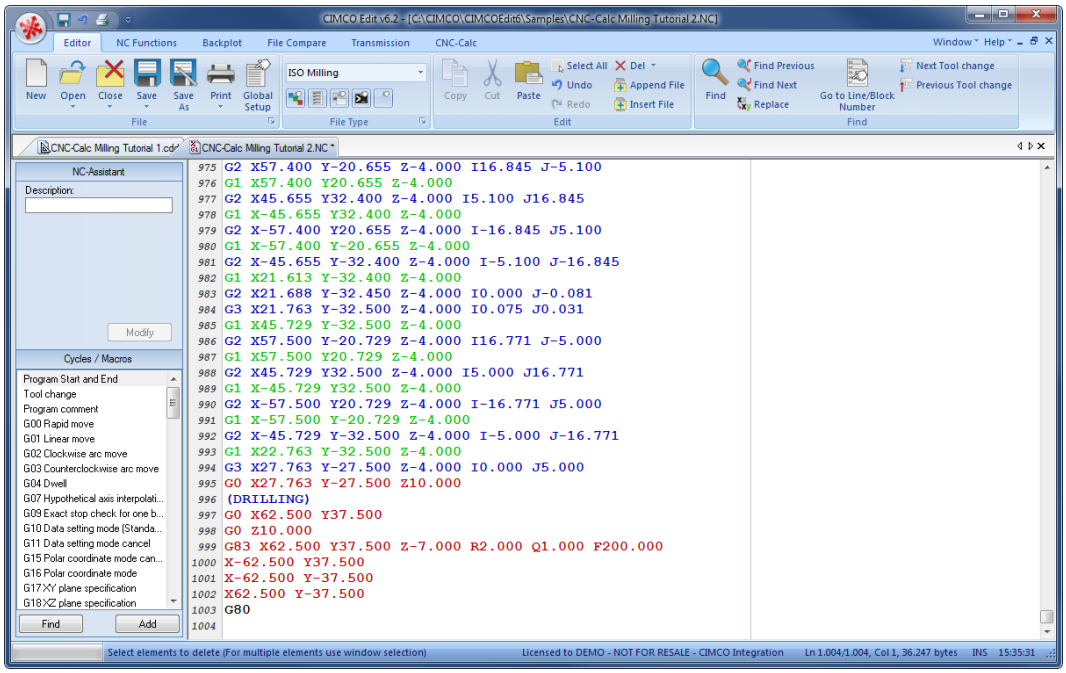


The order of operation can then be changed by clicking on **Reorder Circ** and **Reorder Rect** in the Drilling pane.

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

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

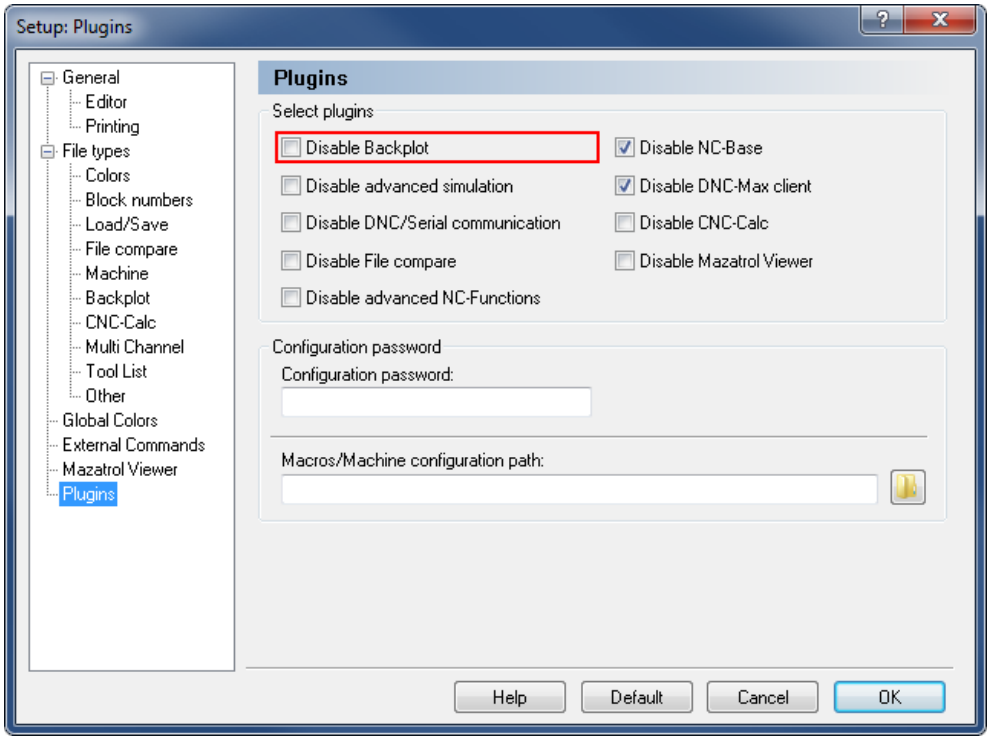
One of the advantages of running CNC-Calc inside CIMCO Edit v7 is that the editor can be used to manipulate and backplot the NC programs generated in CNC-Calc.

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

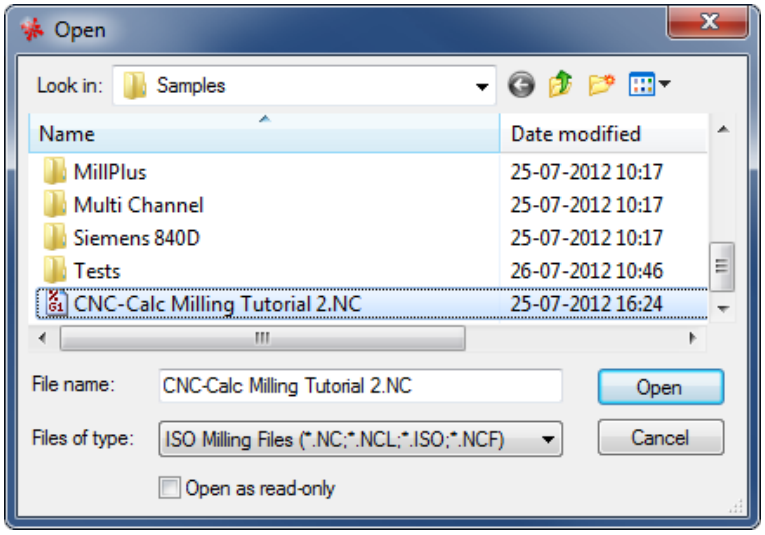
The first thing to do before proceeding with the simulation is to set the menu parameters. Start CIMCO Edit v7 and verify that the Backplot tab is available in the Ribbon. If not, open the editor configuration by clicking the **Global Setup** icon in the *Editor* tab. Go to the Plugins section in the 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.



Enabling the Bacplot tab.

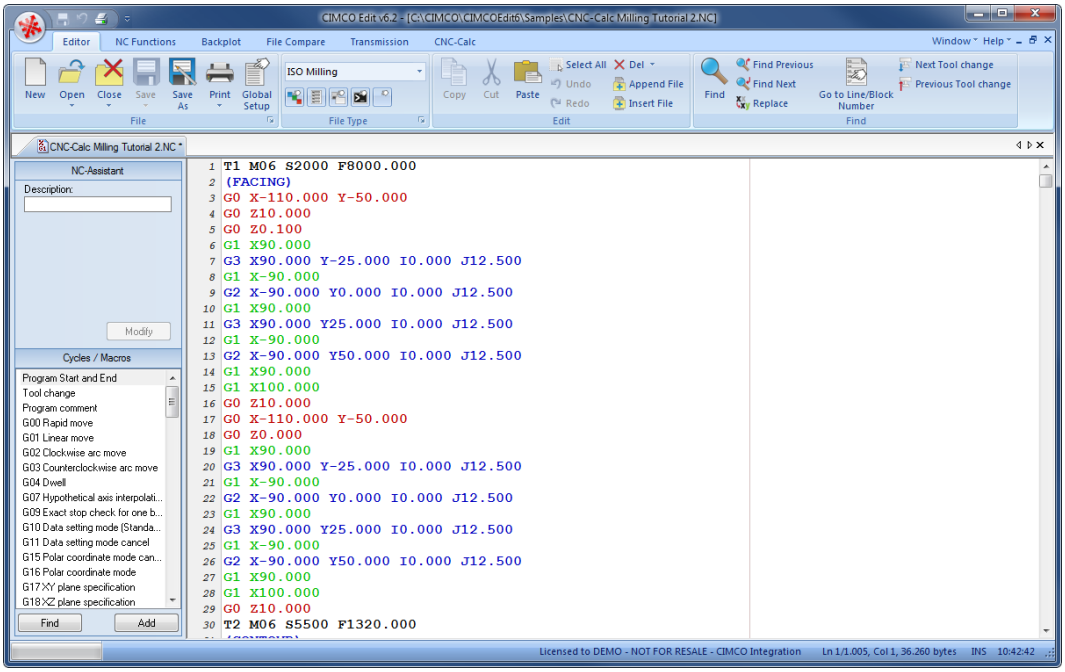
11.2. Open an Existing NC Program

To open a NC Program, click on the icon **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.



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.

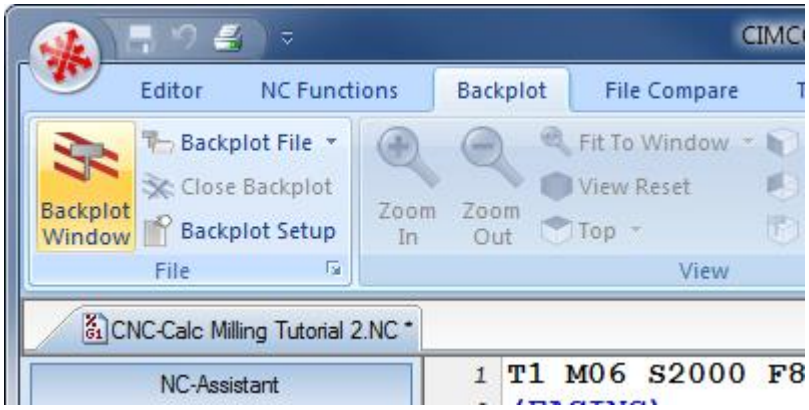


If you hold the cursor over an icon a short description of the icon's functionality will appear.

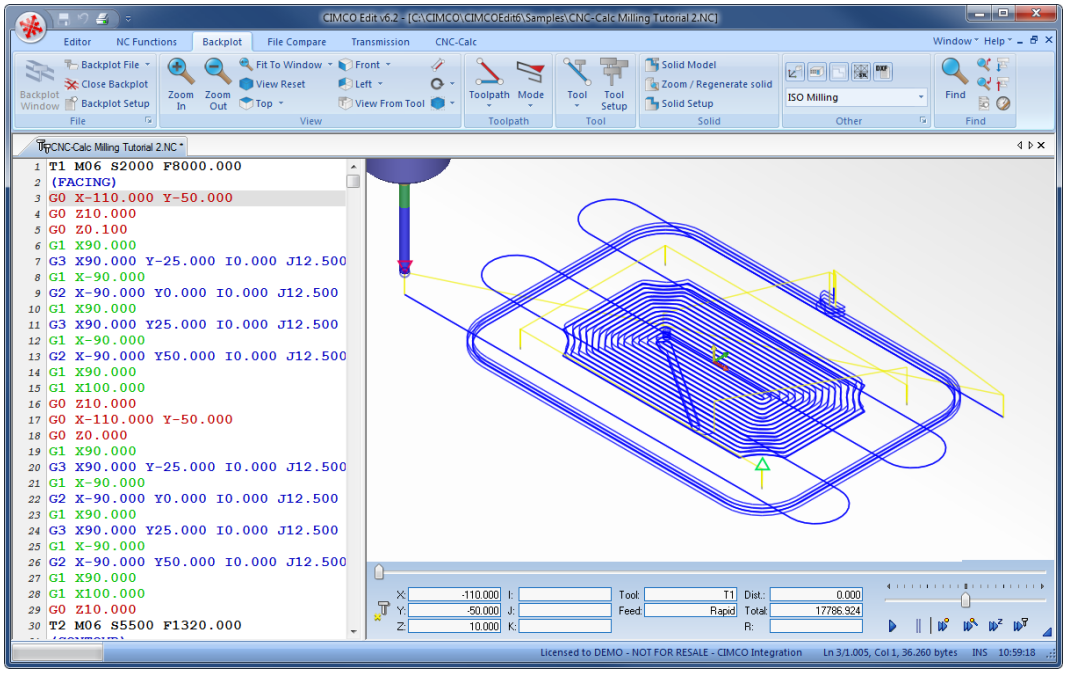
11.3. The First Backplot

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 **Backplot Window** to backplot the NC program opened in the current window.



Now the screen should look like the one below.

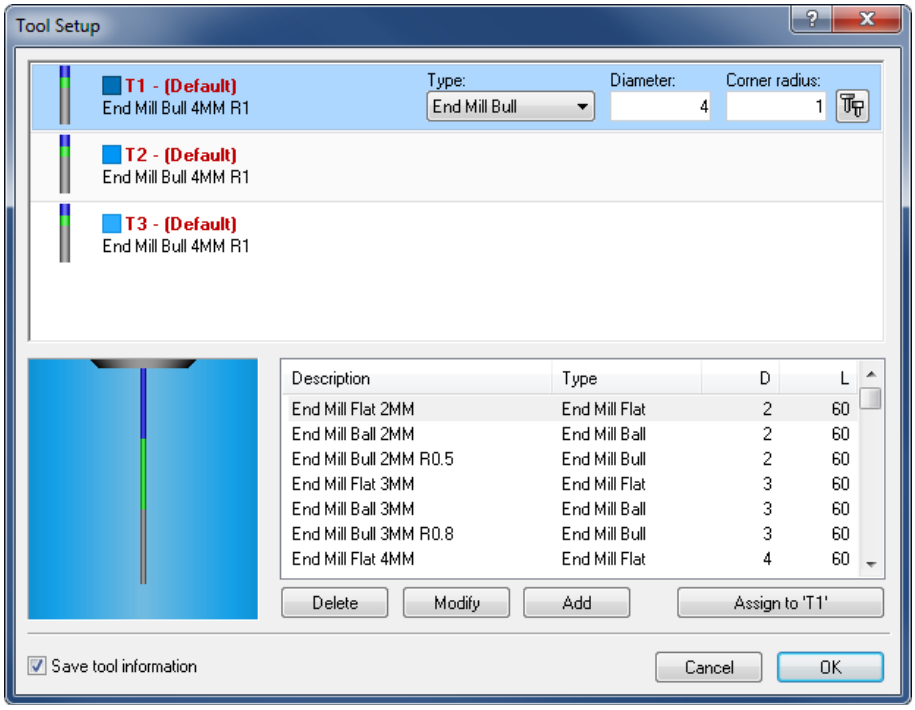


Here, the toolpath is shown. It looks OK, but it can be configured to look a lot more like the final part. To do this, we need to define the individual tools and the stock.

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.

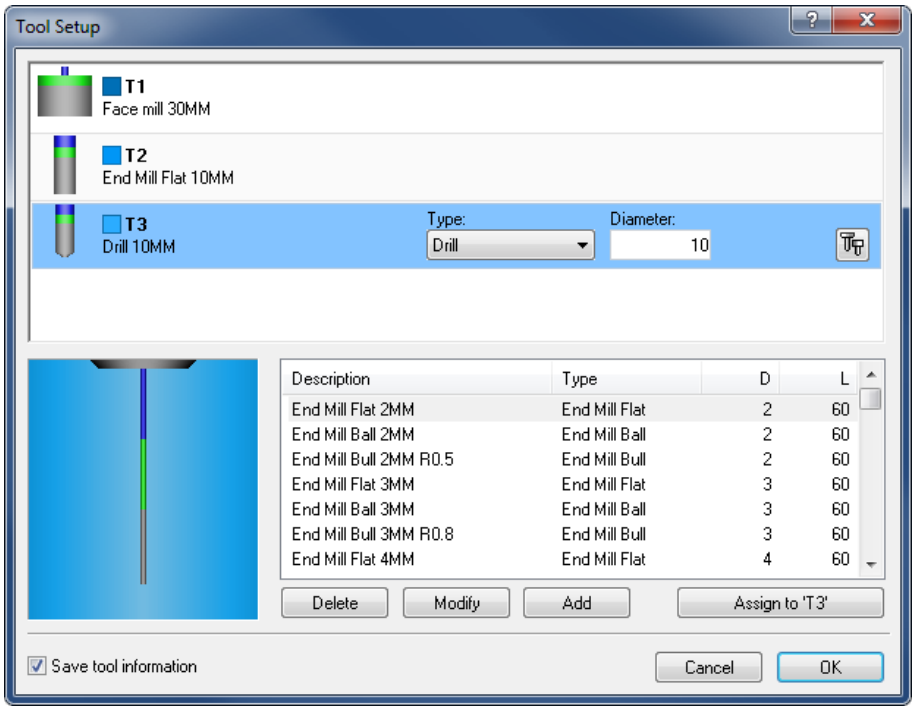


When the Backplot parsed the NC program, it detected that three tools were being used.

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** icon at the right of the selected tool.

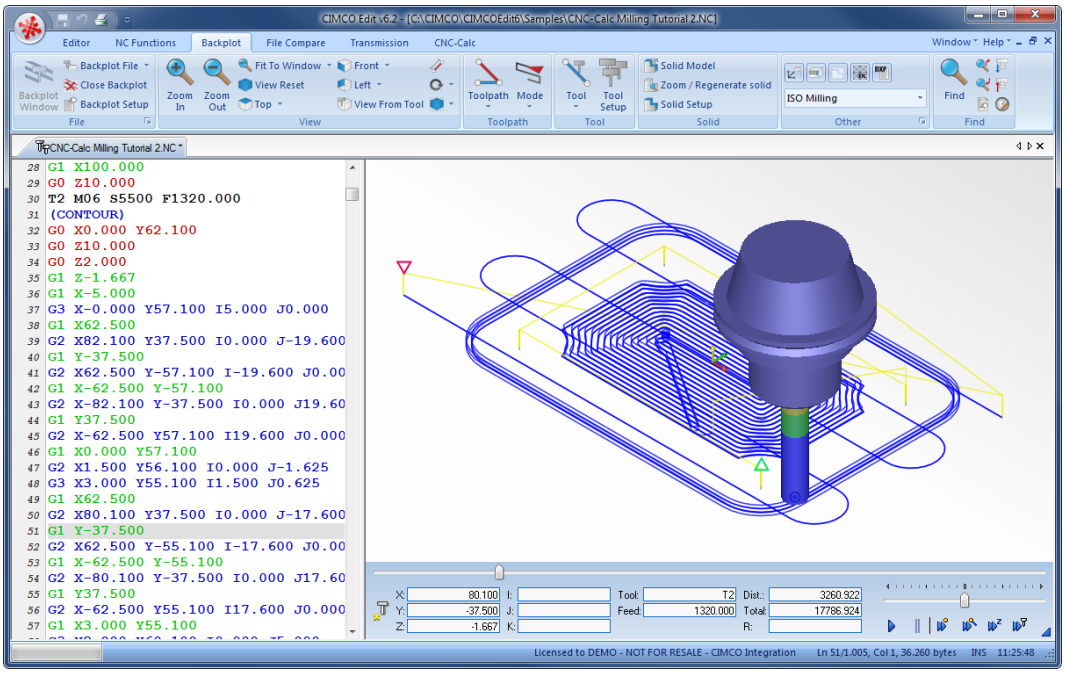
Please select and enter the values for the three tools shown on the picture below.



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**.

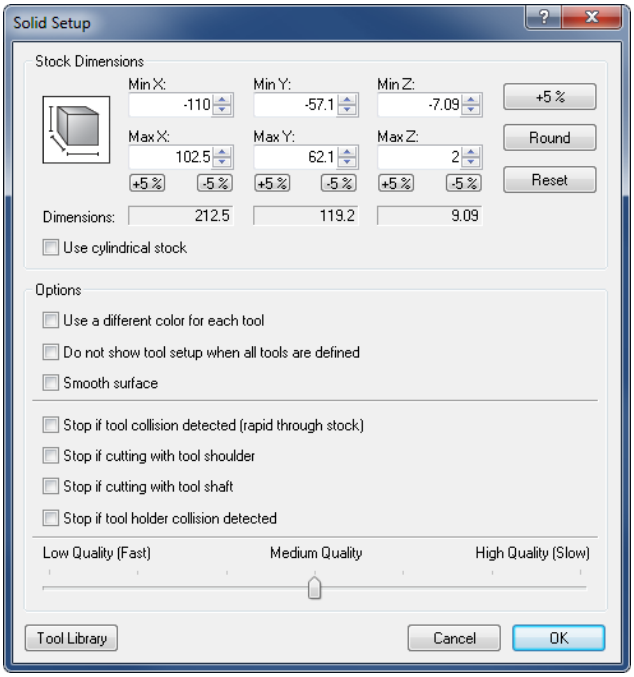
After the tools have been configured, the screen should look like the one below. Try to find the tool changes (M06) and verify that the tool does indeed change when a tool change is encountered in the NC program.



11.3.2. Backplot Stock Setup

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

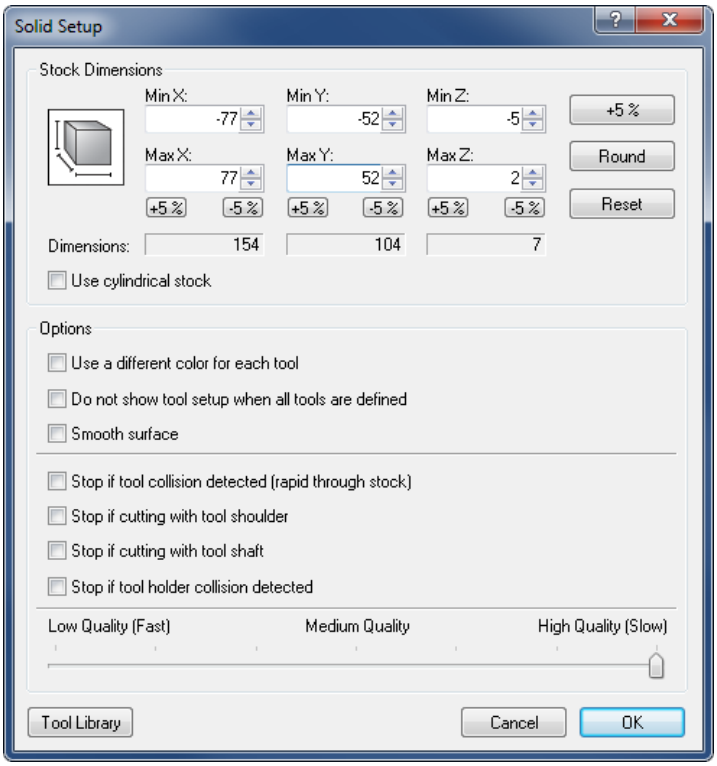
Click on the icon **Solid Setup** in the Backplot *Solid* toolbar. This will open the Solid Setup window.



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.

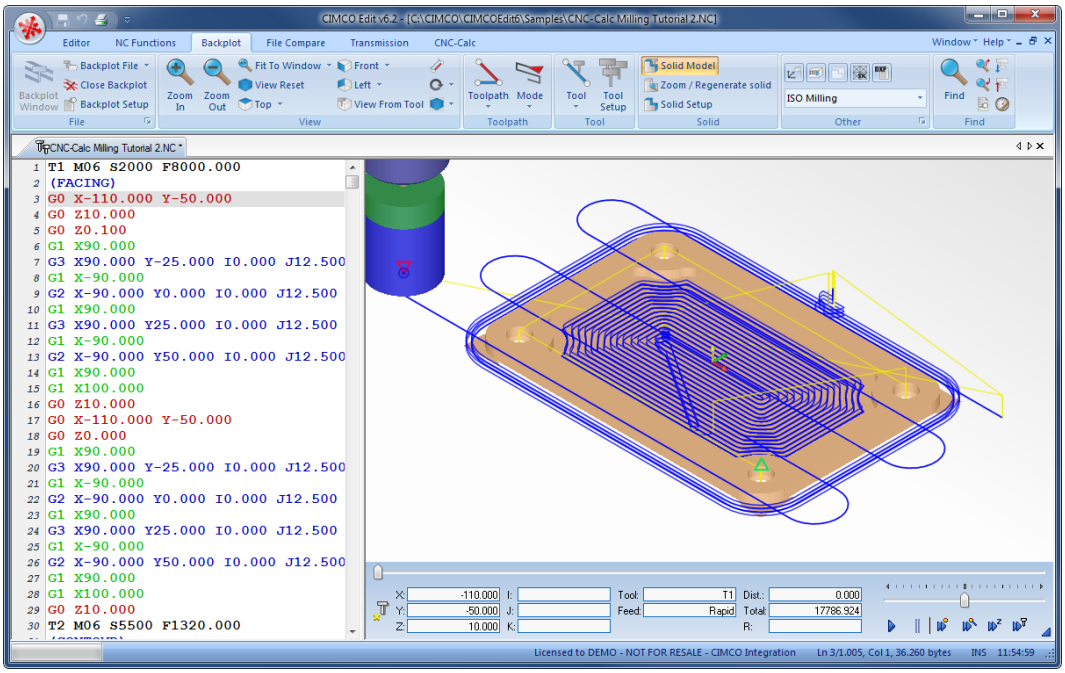
From the facing operation, we know that the top of the stock should be Z = 2.00 mm. From the drawing, we know that the values of the corners are (-75.00, -50.00) and (75.00, 50.00).

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.



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.

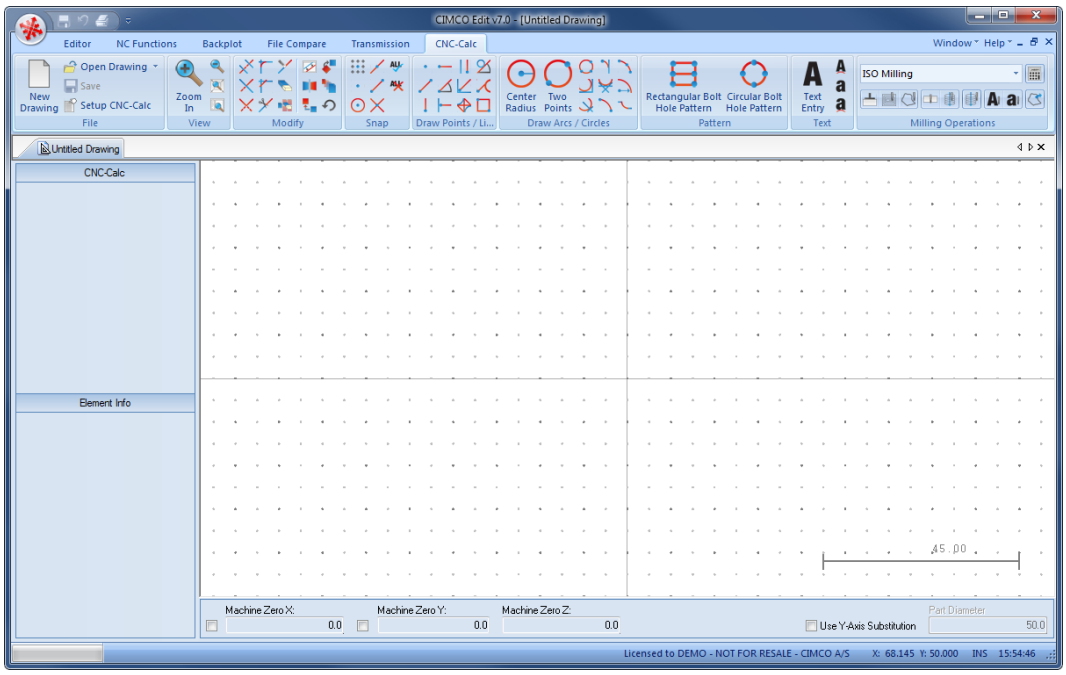

CIMCO Edit v6.2 - [C:\CIMCO\CIMCOEdit6\Samples\CNC-Calc Milling Tutorial 2.NC]
Editor
NC Functions
Backplot
File Compare
Transmission
CNC-Calc
Backplot File
Close Backplot
Backplot Setup
Zoom In
Zoom Out
Fit To Window
View Reset
Top
Front
Left
View From Tool
Toolpath
Mode
Tool
Tool Setup
Solid Model
Zoom / Regenerate solid
Solid Setup
ISO Milling
Find
CNC-Calc Milling Tutorial 2.NC *
T1 M06 S2000 F8000.000
(FACING)
G0 X-110.000 Y-50.000
G0 Z10.000
G0 Z0.100
G1 X90.000
G3 X90.000 Y-25.000 I0.000 J12.500
G1 X-90.000
G2 X-90.000 Y0.000 I0.000 J12.500
G1 X90.000
G3 X90.000 Y25.000 I0.000 J12.500
G1 X-90.000
G2 X-90.000 Y50.000 I0.000 J12.500
G1 X90.000
G1 X100.000
G0 Z10.000
G0 X-110.000 Y-50.000
G0 Z0.000
G1 X90.000
G3 X90.000 Y-25.000 I0.000 J12.500
G1 X-90.000
G2 X-90.000 Y0.000 I0.000 J12.500
G1 X90.000
G3 X90.000 Y25.000 I0.000 J12.500
G1 X-90.000
G2 X-90.000 Y50.000 I0.000 J12.500
G1 X90.000
G1 X100.000
G0 Z10.000
T2 M06 S5500 F1320.000
X: -110.000
Y: -50.000
Z: 10.000
I:
J:
K:
Tool: T1
Feed: Rapid
Dist.: 0.000
Total: 17786.924
R:
Licensed to DEMO - NOT FOR RESALE - CIMCO Integration
Ln 3/1,005, Col 1, 36,260 bytes
INS
11:54:59

12. Tutorial Mill 4 - Milling of Letters

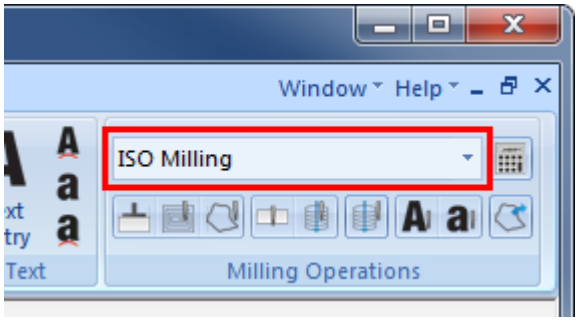
This tutorial demonstrates how a 2-dimensional text can be used as the basis for an NC program milling letters and numbers.

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:



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

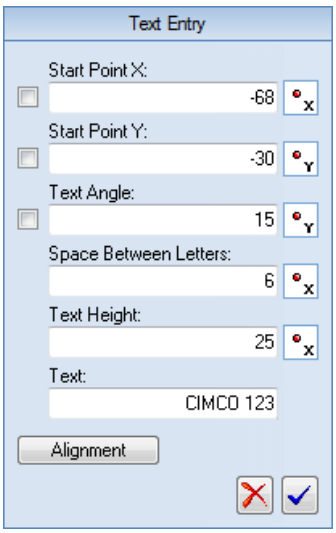
Normal upper and lower case letters (no accents, dieresis, tildes, etc.), numbers, and some special characters can be entered.

12.2.1. Drawing the Text

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

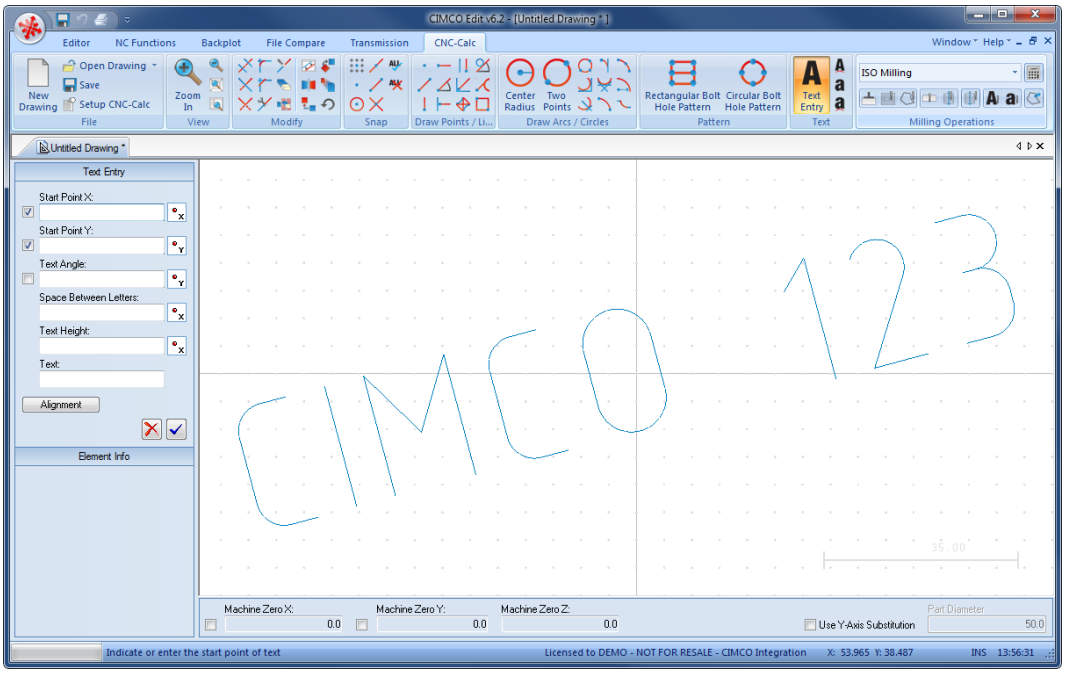
The Text Entry pane appears to the left to define the text that will be drawn.

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.



When you are done entering text and values, click on the blue check mark button 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.

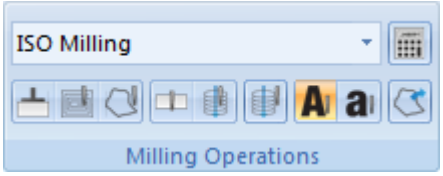


The drawing now shows the text, and its geometry can now be used for generating the toolpaths.

12.2.2. Milling of Letters

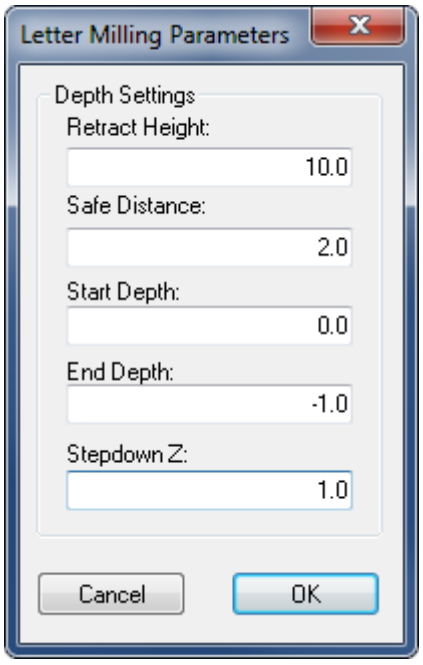
To begin the creation of an NC program for the letters milling operation, ensure that **ISO Milling** is selected as programming format for the NC program in the *File Type* drop-down list in the *Operations* toolbar.

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

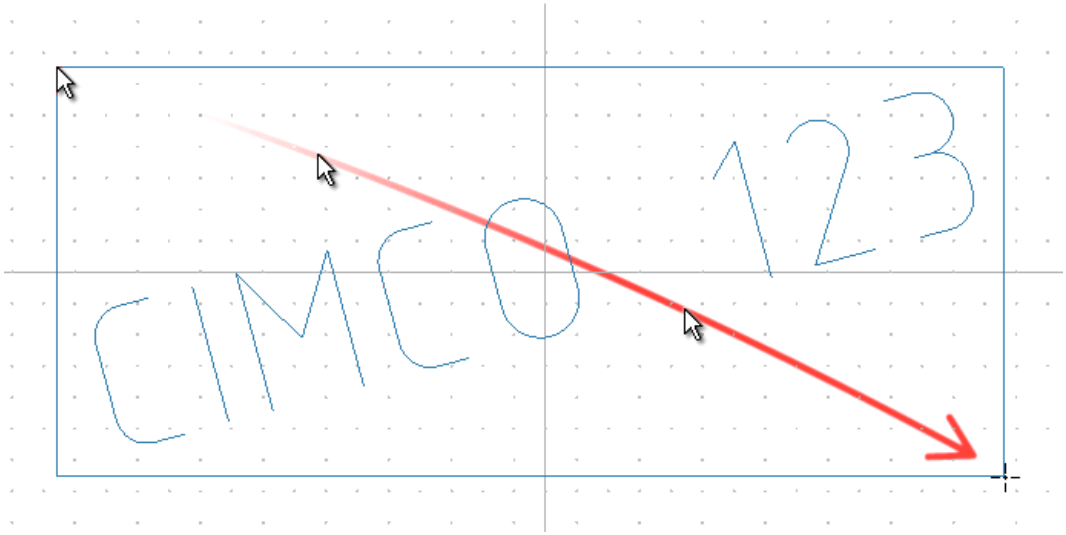


The CNC-Calc Letter Milling pane will appear on the left side of the screen as shown. Write the comment LETTER MILLING 'CIMCO 123' in the *Comment* field. It is a good practice to include the text to machine in the comment for the letter milling operations.

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

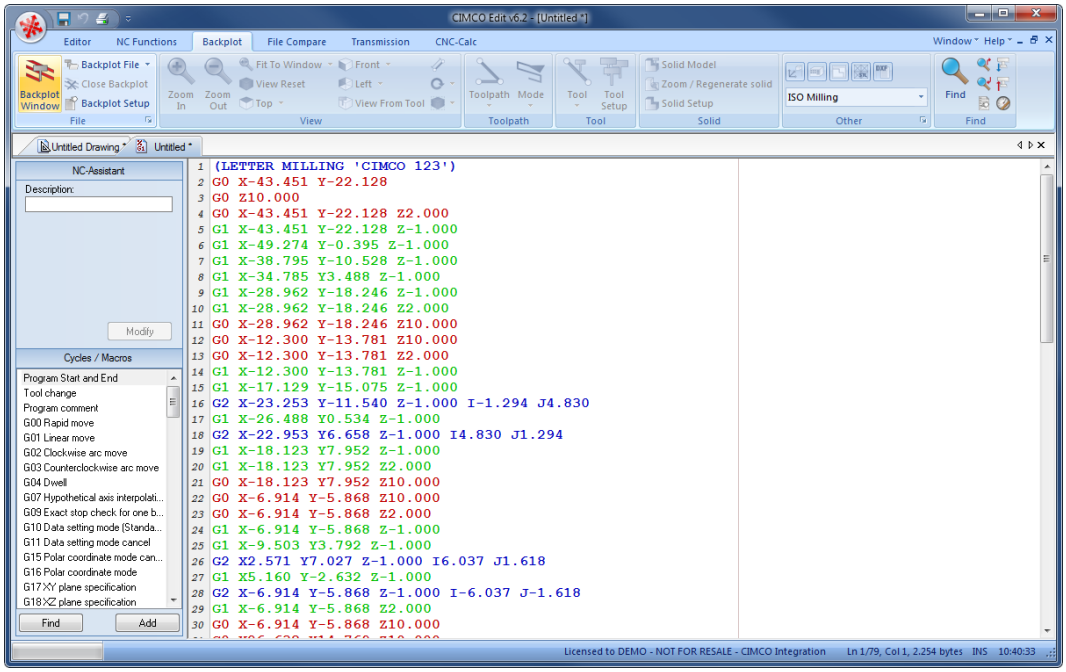


Now, use window selection to select the text you want to mill. On the drawing, left-click one corner, hold down the mouse button, and drag diagonally. When the desired text is framed, release the button. The frame disappears and the text will have turned yellow to indicate that it has been selected.

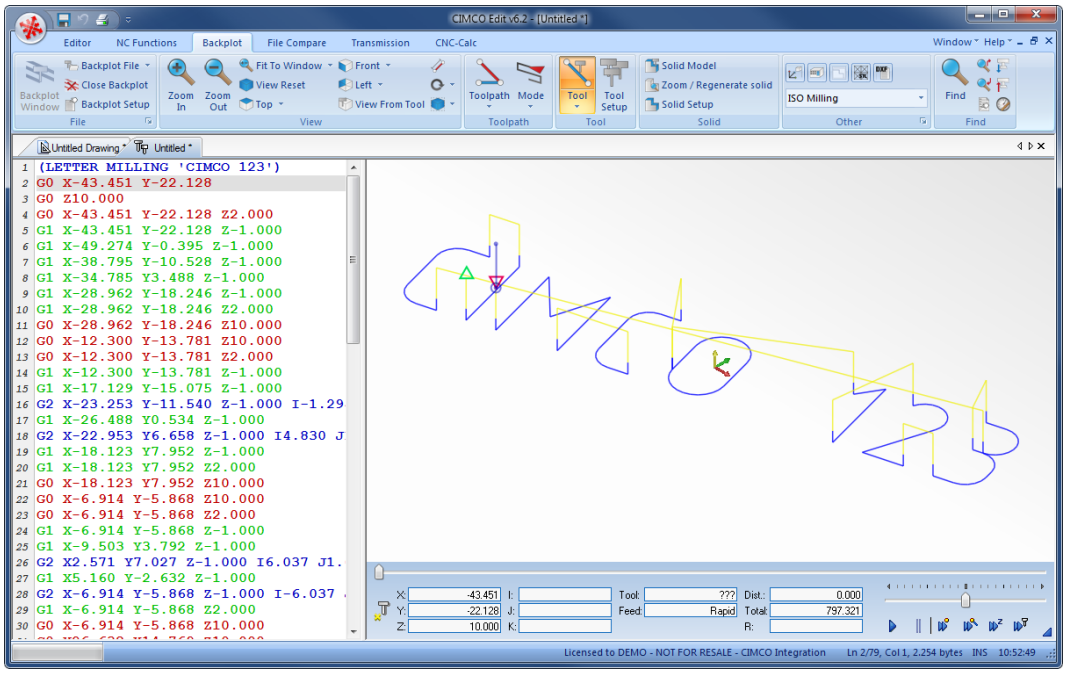


Next click on the **Export Editor** button, and the NC codes for machining the text will be shown in a new window in the Editor.

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 **Backplot Window**.



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".

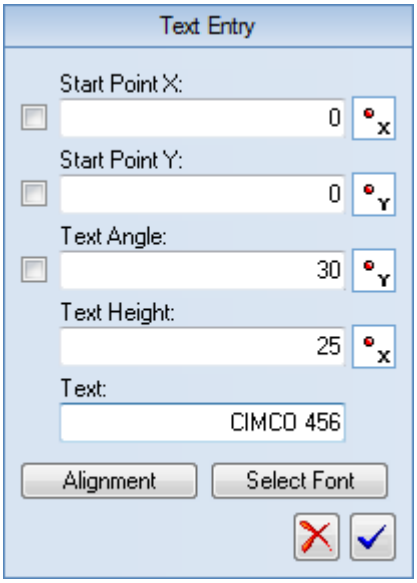
Normal upper and lower case letters, numbers, and special characters can be entered.

12.3.1. Drawing the Text

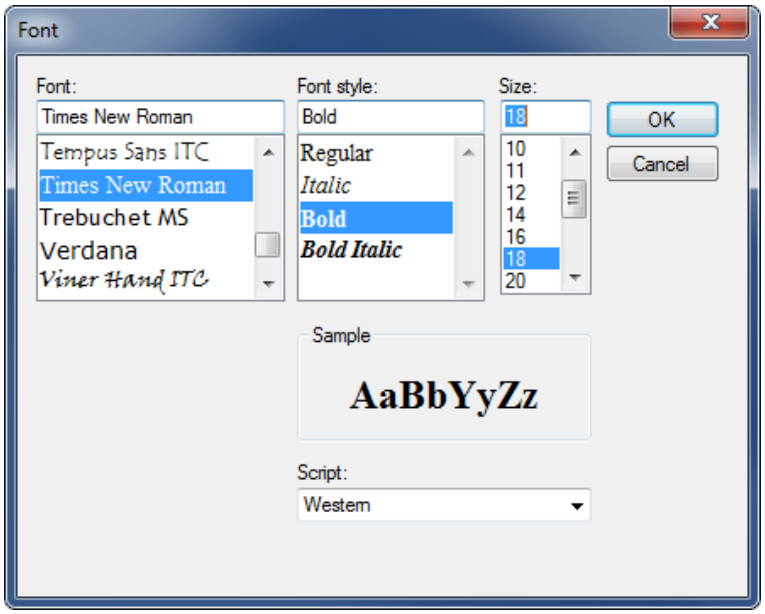
To begin the creation of the TrueType text that will be used for machining, click on the icon **TrueType Text Entry** in the CNC-Calc *Text* toolbar.

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

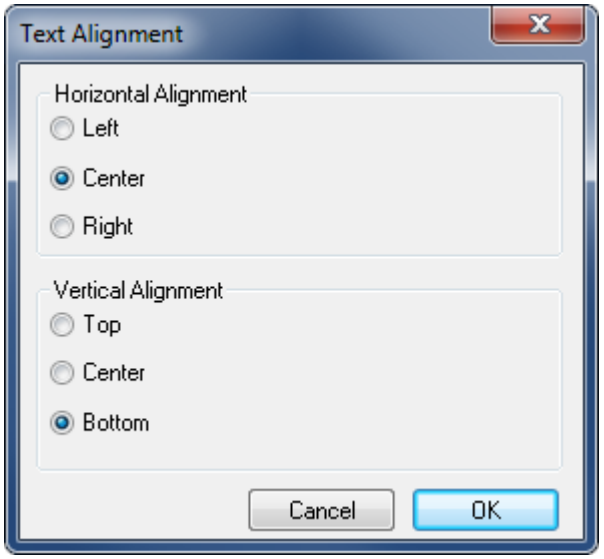
In the input field **Text** at the bottom, write the text to be milled with TrueType letters; in this example "CIMCO 456". Enter the coordinates for the starting point of the text, the angle (relative to the horizontal axis) of the text baseline, and the height of the letters.



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.

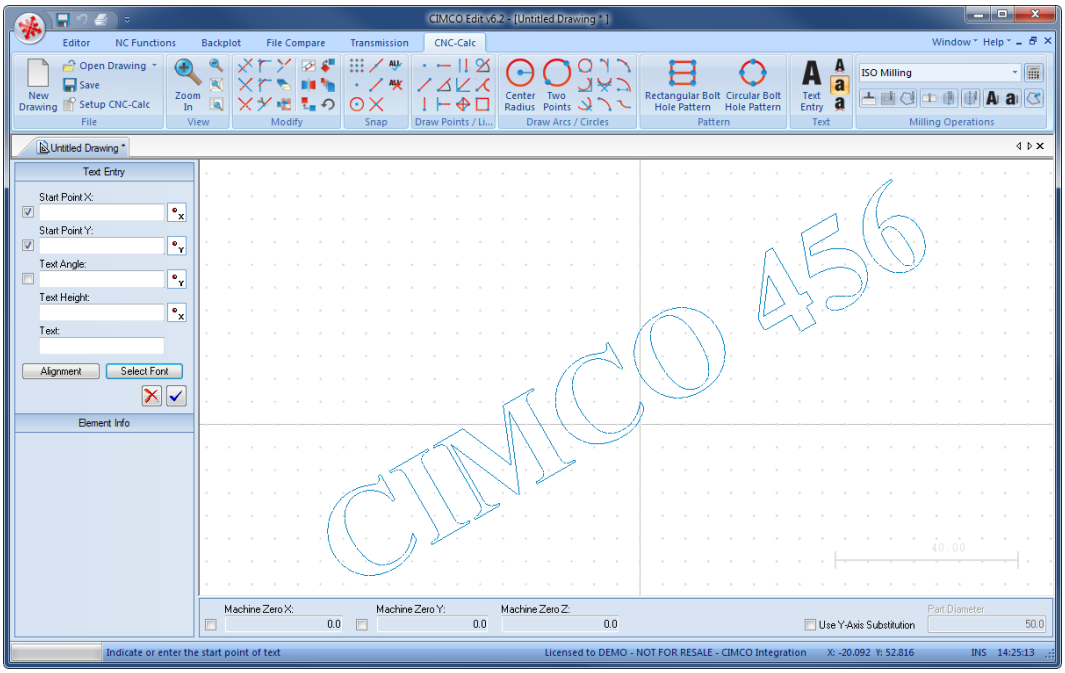


Using the **Alignment** button, you can select the text horizontal and vertical alignment relative to the starting point entered.



When done, click on the blue check mark button ☑ 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.

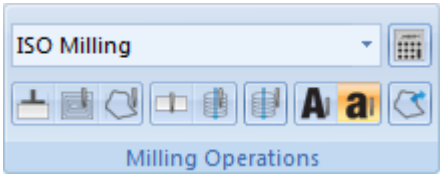


The geometry is now finished and can be used for the generation of toolpaths.

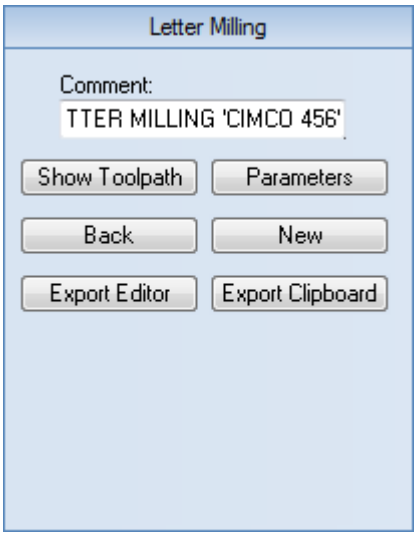
12.3.2. Contour Milling of Letters

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.

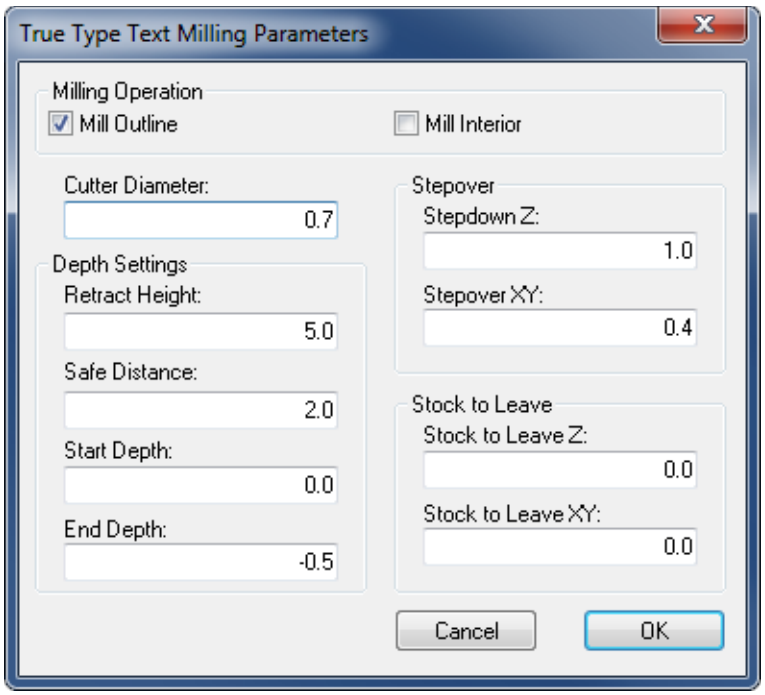
Then select the feature TruType Letter Milling by clicking on the icon **Mill TrueType Letters** in the *Milling Operations* toolbar to generate a CNC-toolpath for milling letters.



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.



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.

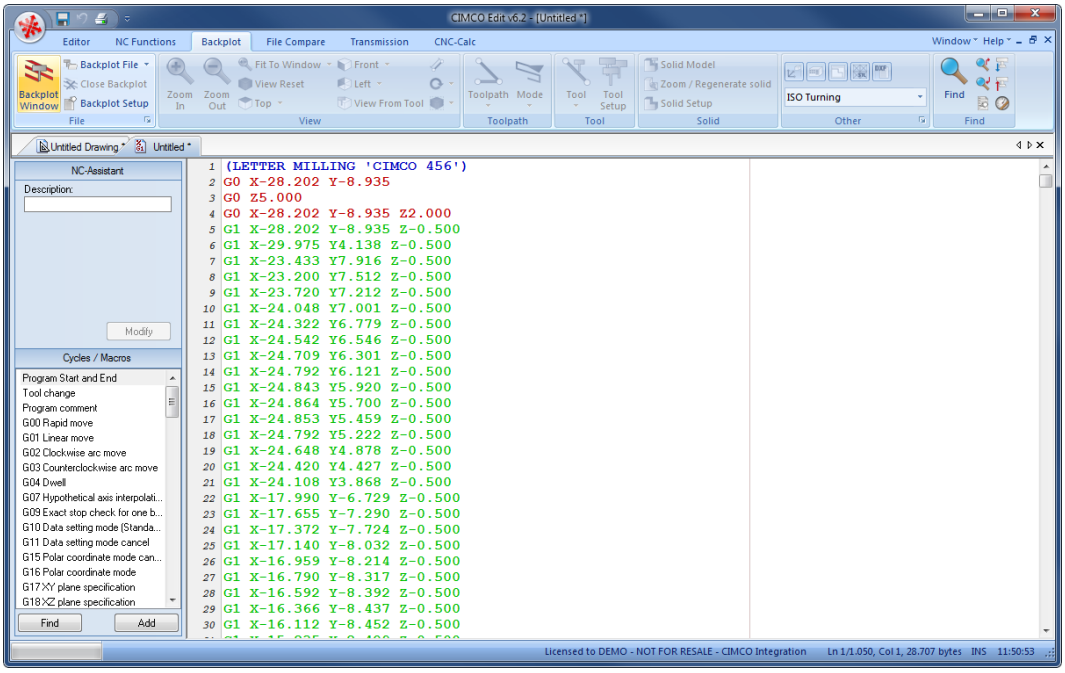


Use window selection to select the letters to mill. This is done by left-clicking in the upper-left corner of the drawing. Now hold the mouse button down and drag the cursor to the lower right corner and release the mouse button.

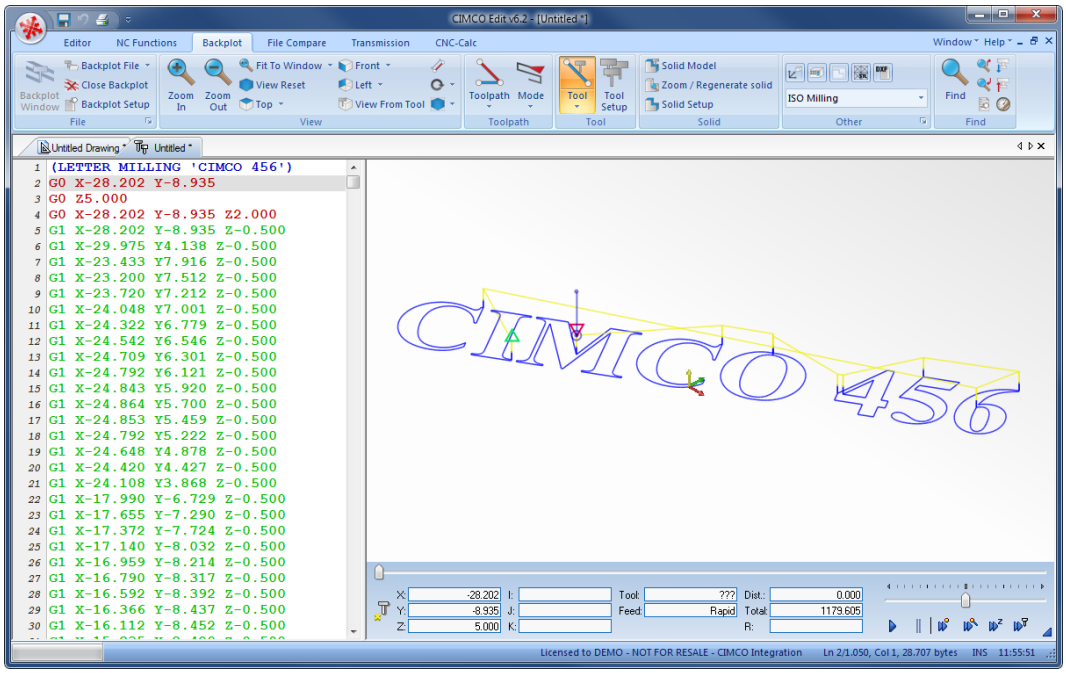
Click on the **Show Toolpath** button. Now the generated toolpath will be shown on the drawing. Since the toolpath coincides with the contour of the letters it can be difficult to see.

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

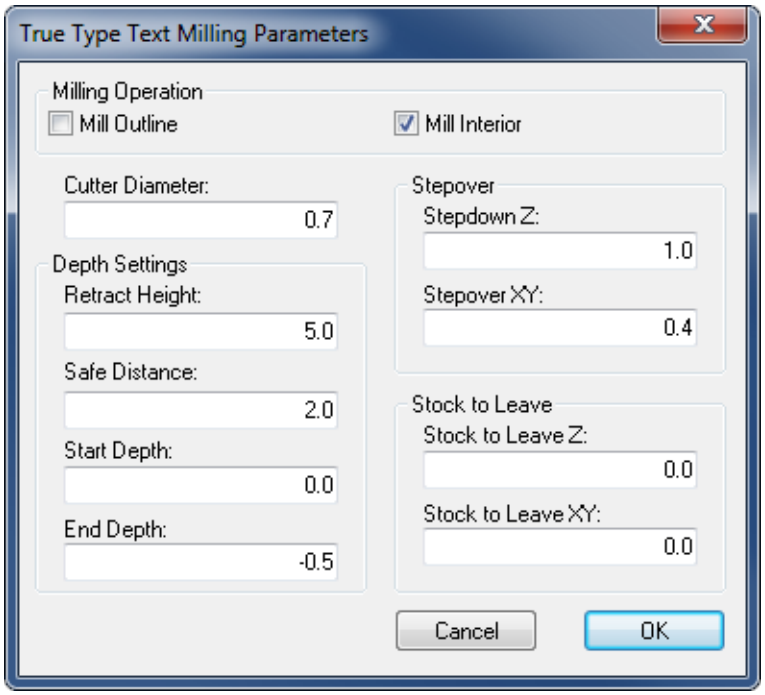
The toolpath generated from the text looks like a contour operation formed by the outlines of the letters and numbers. The area inside these contours - inside the letters - can be milled as a special pocket milling operation. The rest of this tutorial will show you how to do this.

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.

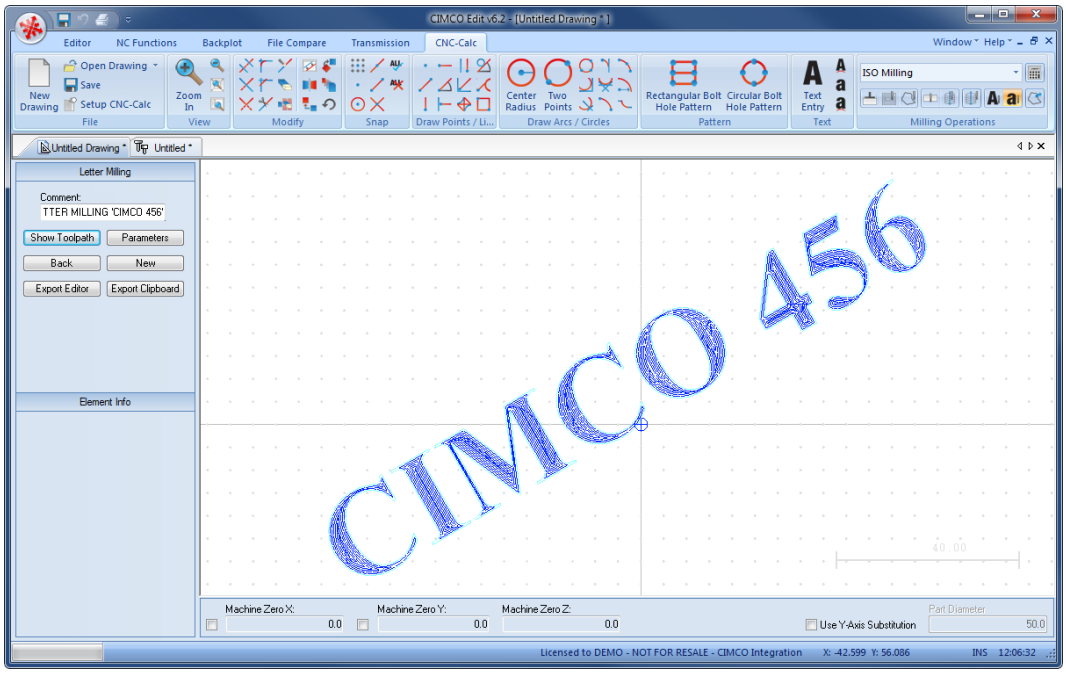
3. Click on the button **Parameters**.

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



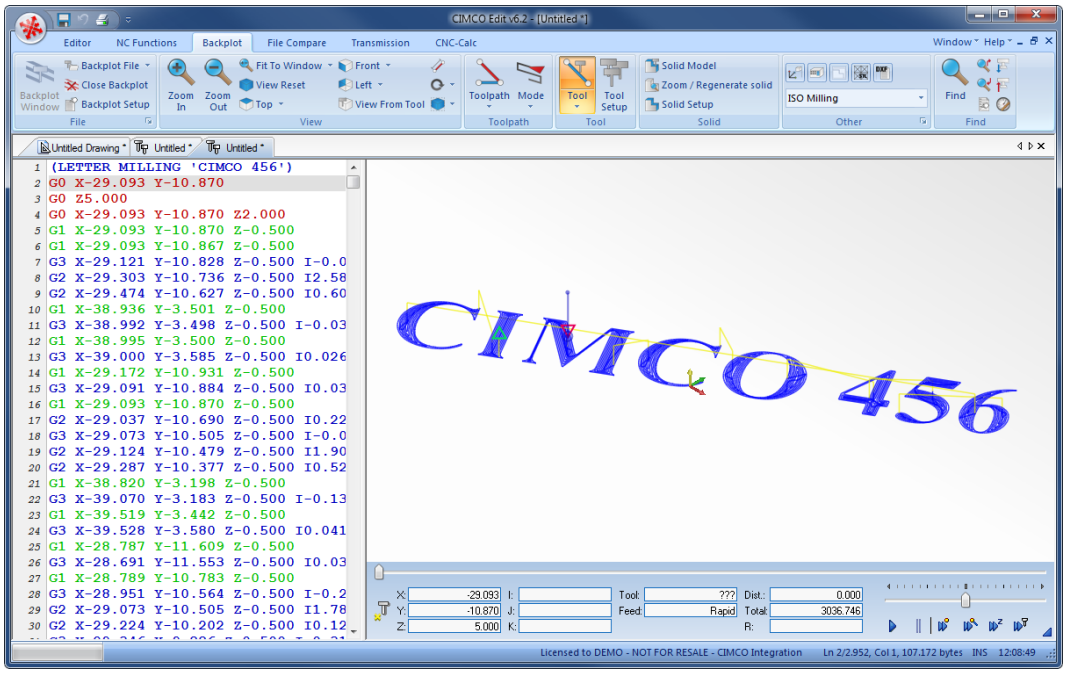
5. Window-select the text.

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.

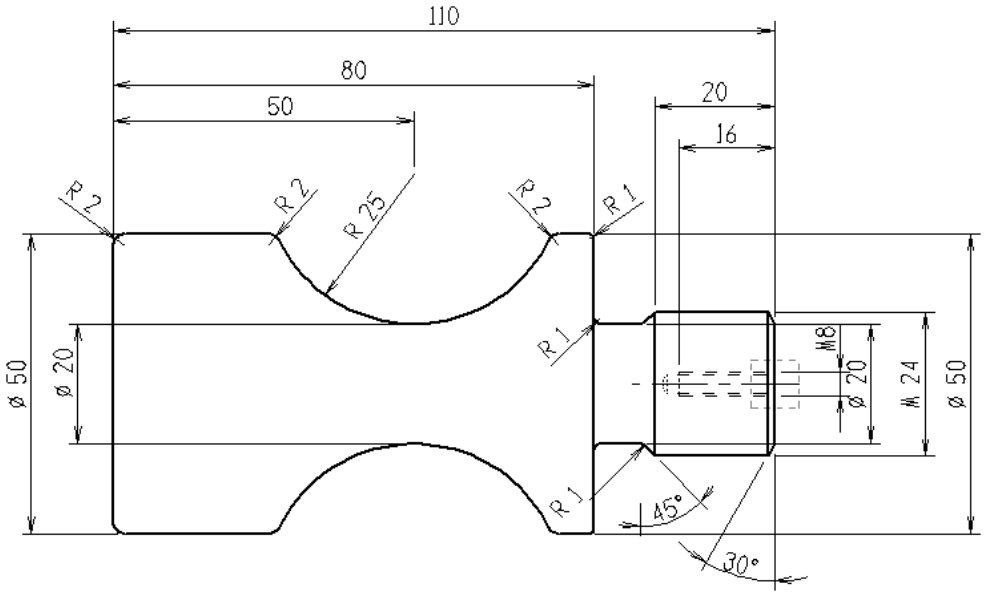
8. Simulate the program to verify the toolpath the same way we did earlier in this tutorial.



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.

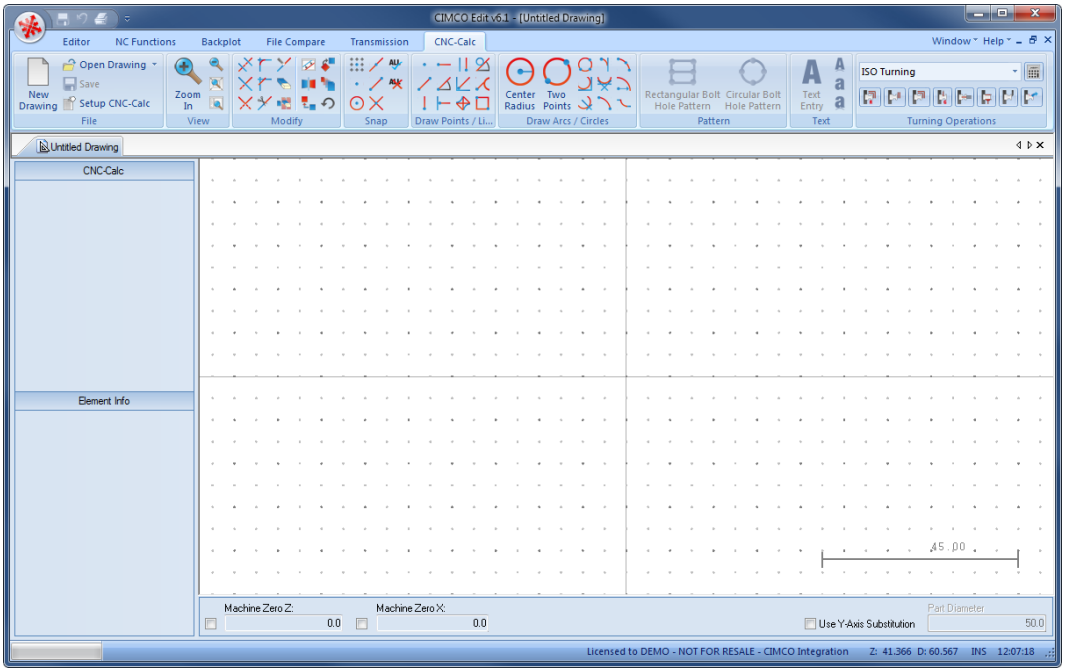
This tutorial demonstrates one of many ways in which the 2-dimensional part above can be drawn in CNC-Calc v7. Since the part is symmetric about the horizontal axis, only a subsection of the part needs to be drawn. The complete part emerges from mirroring. The part will be used to generate turning toolpaths,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

This tutorial demonstrates the use of the following functions:

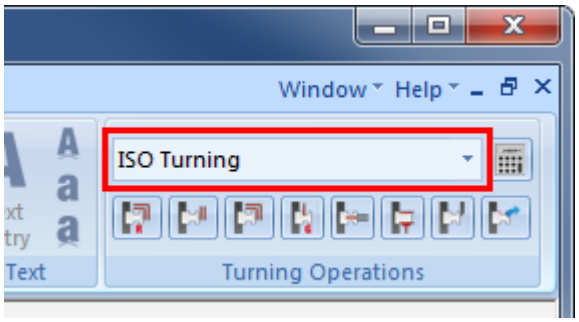
- Draw a vertical line defined by its starting point and length.
- Draw a horizontal line defined by its length.
- Draw a circle with a given radius defined by its center.
- Draw a line defined by its end points.
- Draw a polar line.
- Offset a geometric element.
- Trim element between points of intersection 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:



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



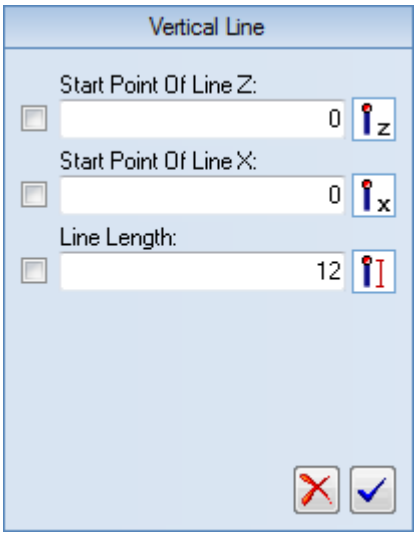
If you hold the cursor over an icon a short description of the icon's functionality will appear.

You can change the colors of the drawing area by selecting **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

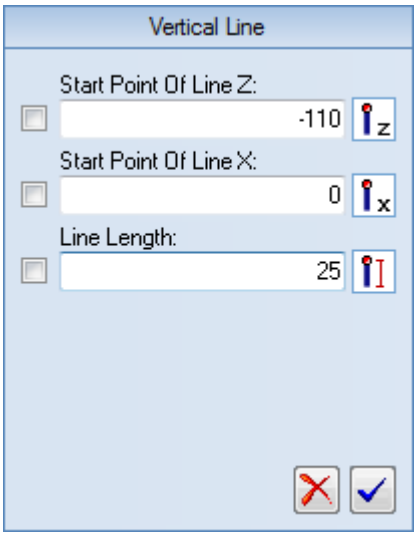
- Click on the **Vertical** icon in the *Draw Points / Lines* toolbar.
- Enter the values shown in the figure below to draw a vertical line of 12 mm:



- Click on 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:

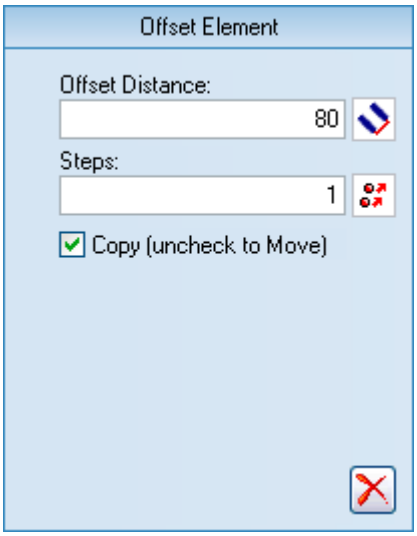


- Click on ☑ to approve the command and draw the line.

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

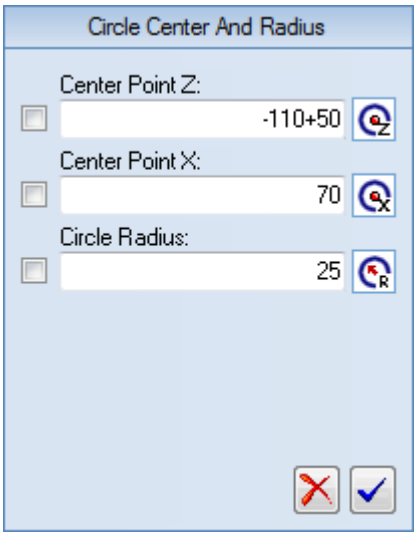
- Click on the **Offset Element** icon in the *Modify* toolbar.
- Enter the value **Offset Distance = 80** in the CNC-Calc pane.



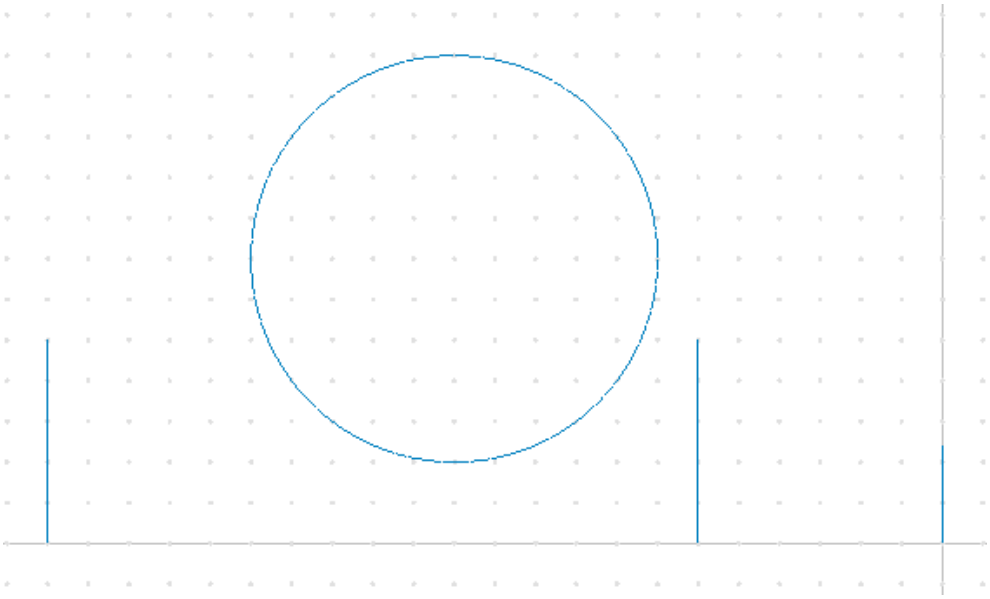
- Now click on the leftmost line (the 25 mm line drawn in the last step). After doing this, two lines will appear (only the rightmost can be seen on the drawing).
- Now click to the right of the selected line to keep the line shown in red (it turns blue after clicking on it).

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:

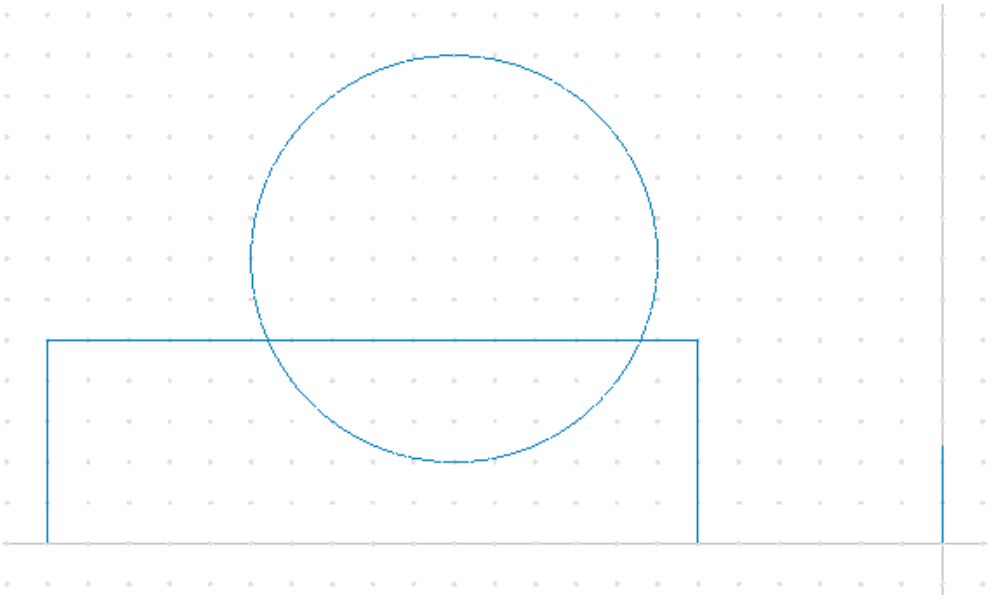


- Click on to approve the command and draw the circle.



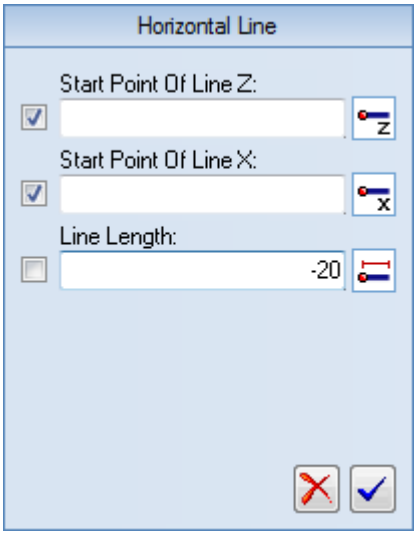
13.2.4. Draw a line defined by its endpoints

- 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.

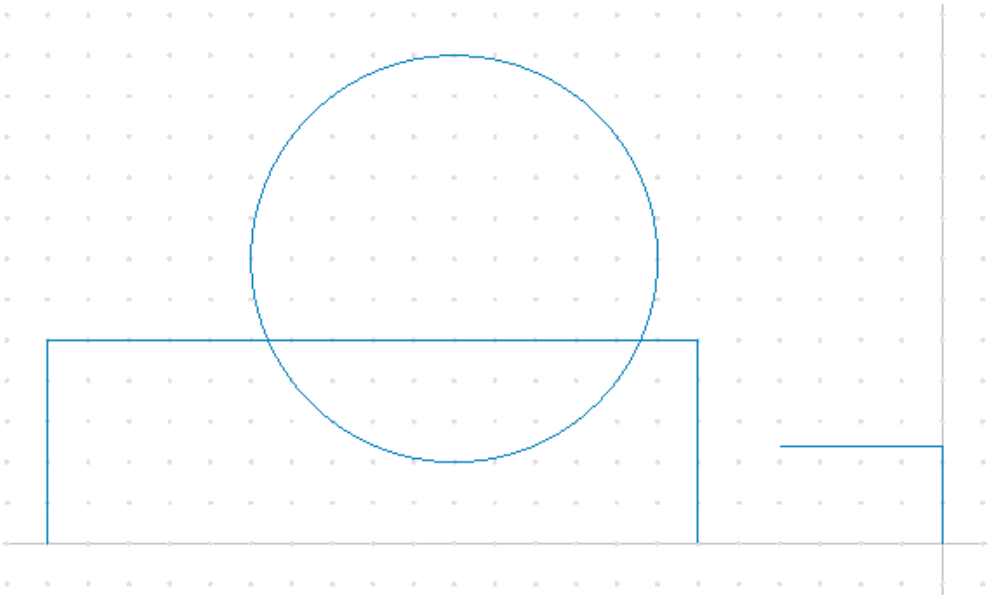


13.2.5. Draw a horizontal line defined by its length

- 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.

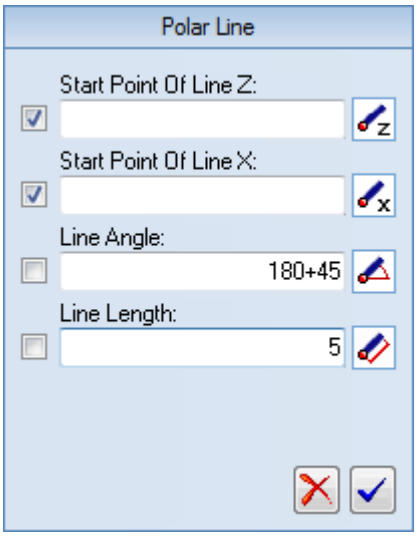


- With the **Snap to End Points** function enabled, select the top end of the short vertical line furthest to the right. This will draw the horizontal line.

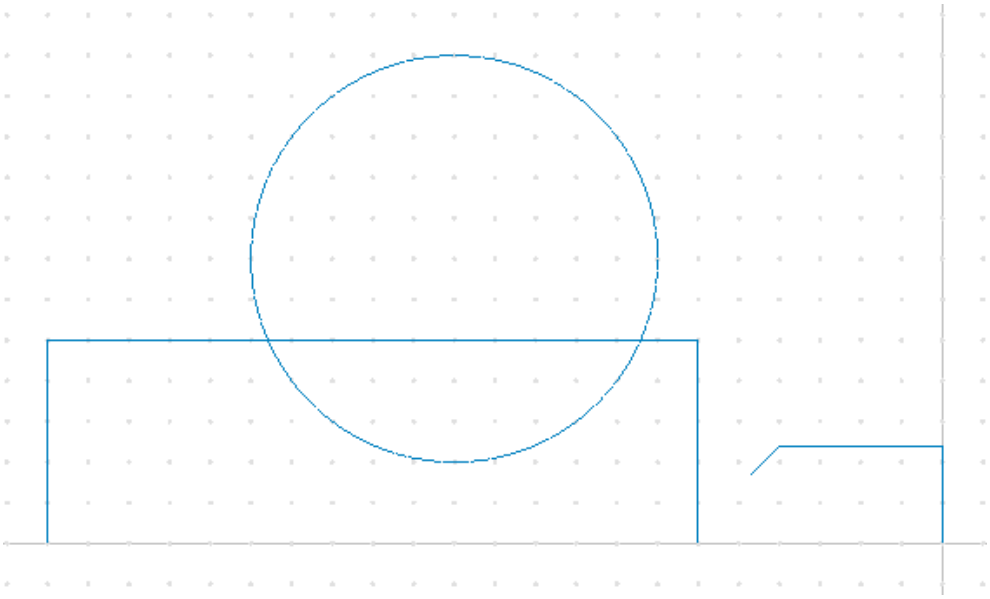


13.2.6. Draw a polar line

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



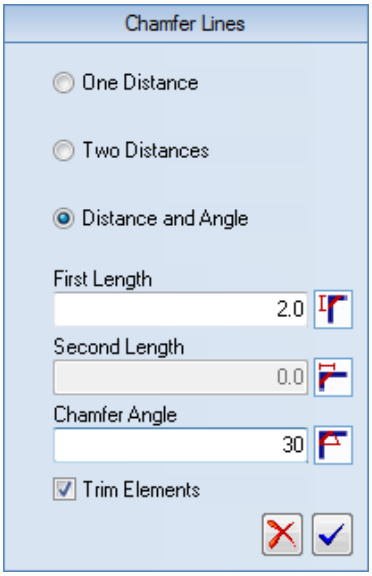
- With the **Snap to End Points** function enabled, select the left end of the short horizontal line drawn in the previous step. The drawing should look like the one below.



13.2.7. Chamfer the foremost corner

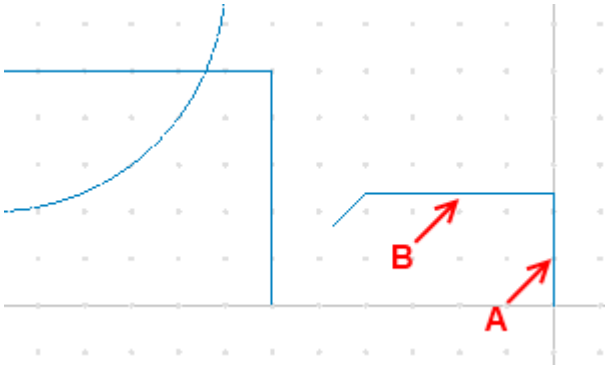
- Click on the **Chamfer Elements** icon in the *Modify* toolbar.

- Define the chamfer using *Distance and Angle* and enter the values shown in the picture below into the CNC-Calc pane:

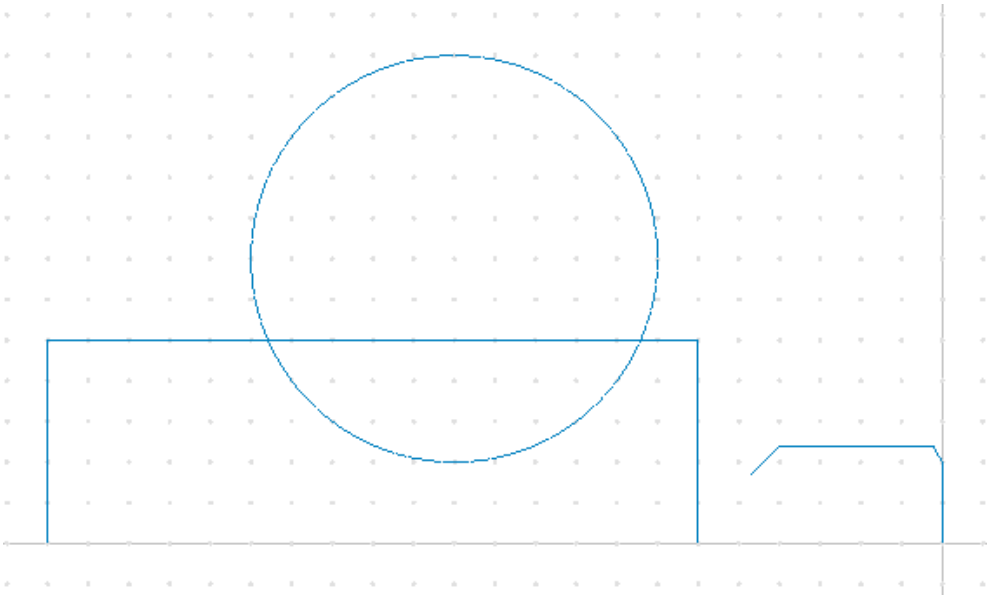


- Since the chamfer angle is different from 45 degrees, it is important to select the lines in the right order. The angle will always be measured from the first element selected.

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

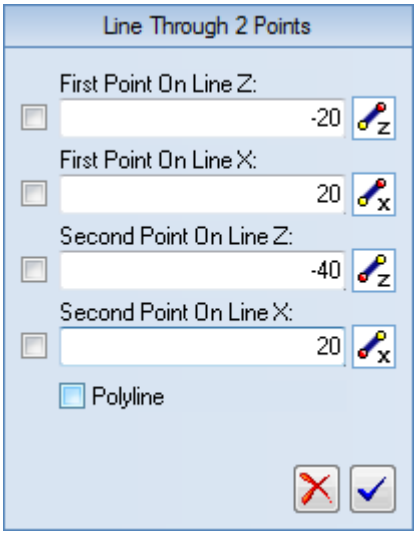


- From the four possible solutions select the one which makes the right chamfer. In the picture below you can see how it should look.

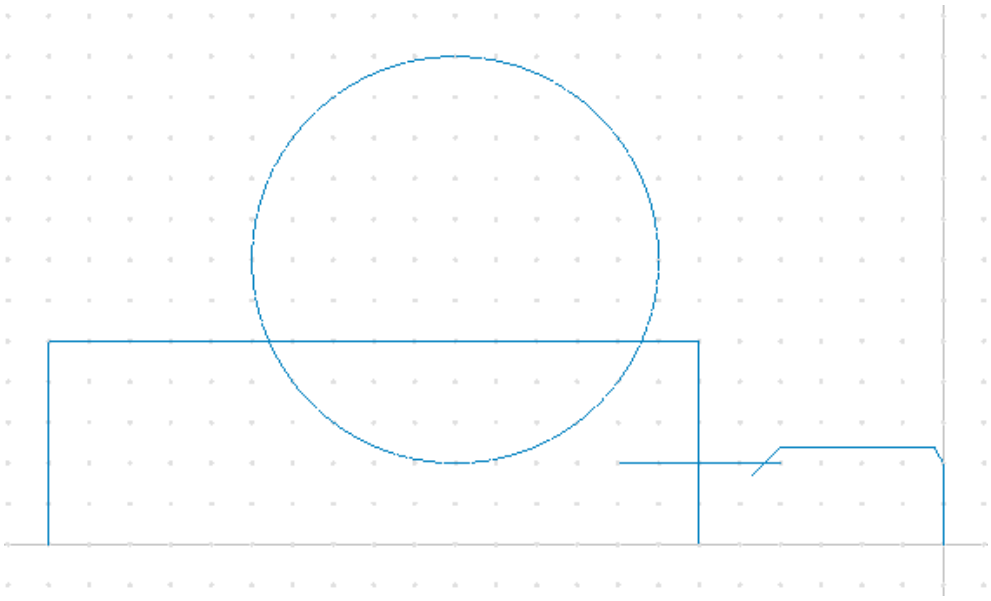


13.2.8. Draw a line defined by its endpoints

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

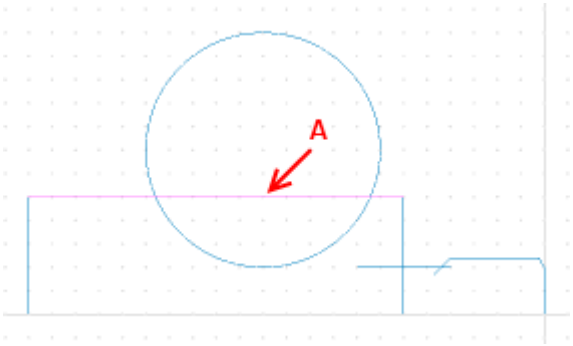


- Click on to approve the command and draw the line.

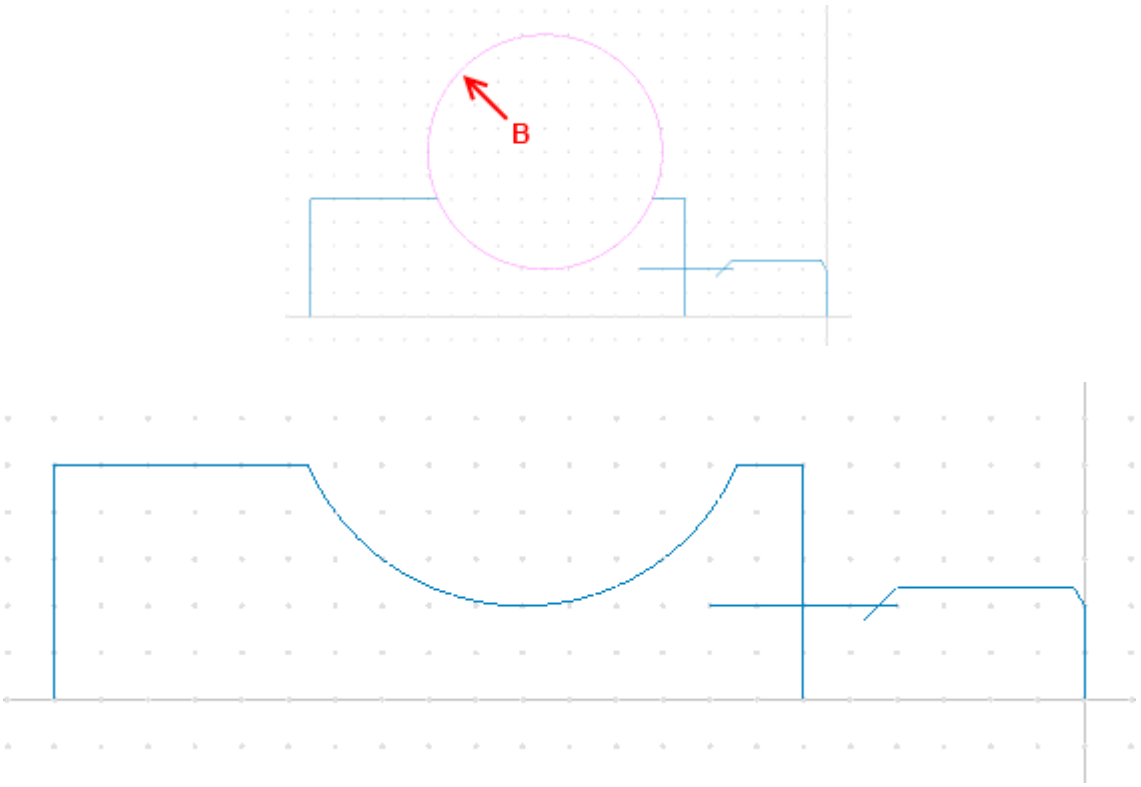


13.2.9. Trim between points of intersection

- Click on the **Trim to Intersection** icon 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.

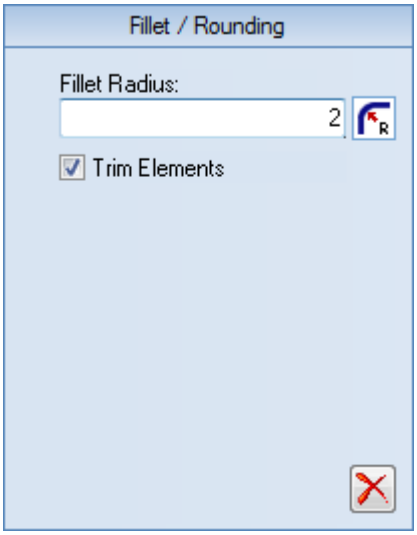


- Now trim the large circle. To do this, click on the part of the circle you want to remove - indicated by B on the picture below.



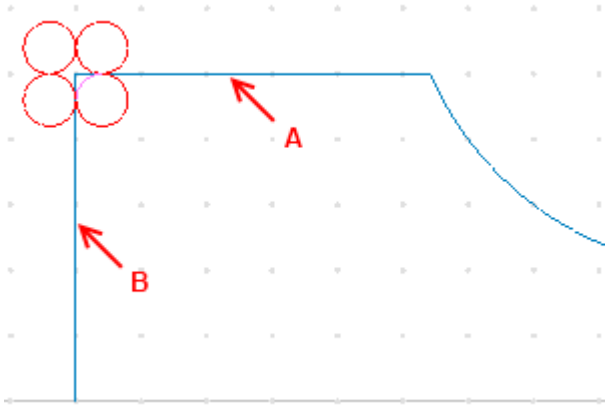
13.2.10. Fillet intersections

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



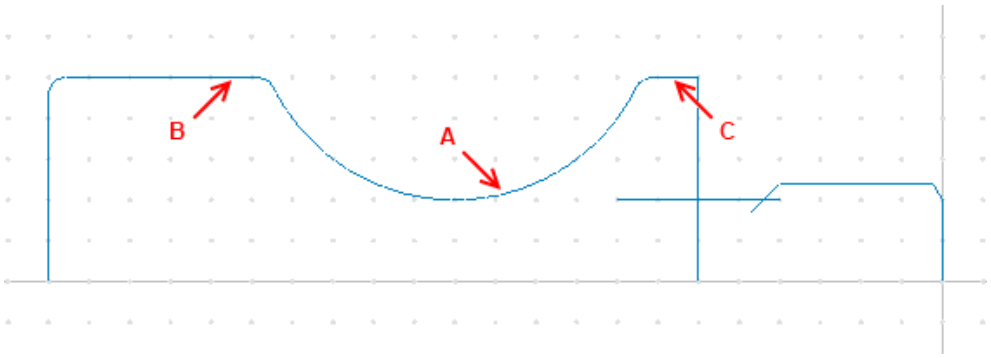
- Select the elements on which the fillet operation should be performed. This is done by left-clicking on the part of the elements that you want to keep.

  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.

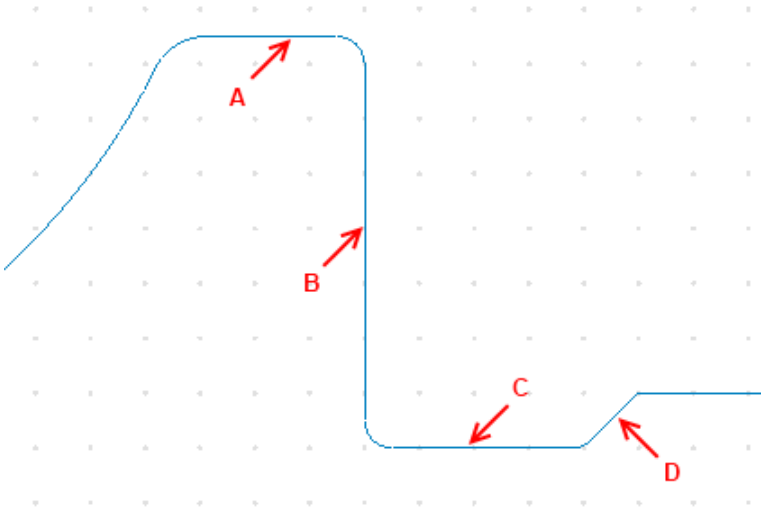


Repeat fillet operation with radius 1 corners

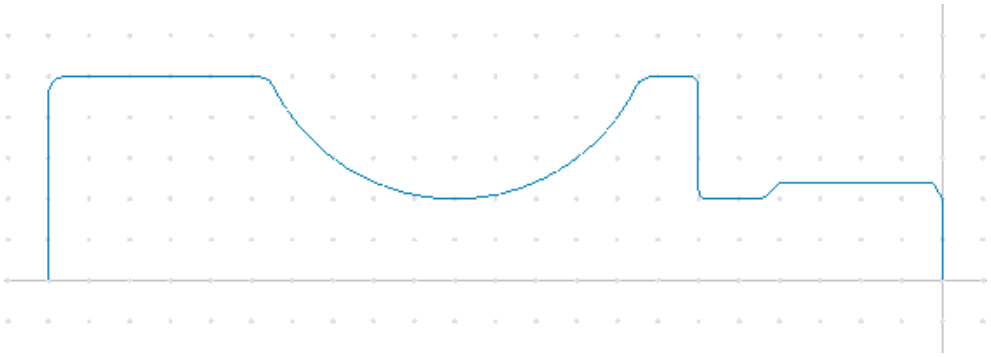
- First change the **Fillet Radius** value in the CNC-Calc pane from 2.00 to 1.00.

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.

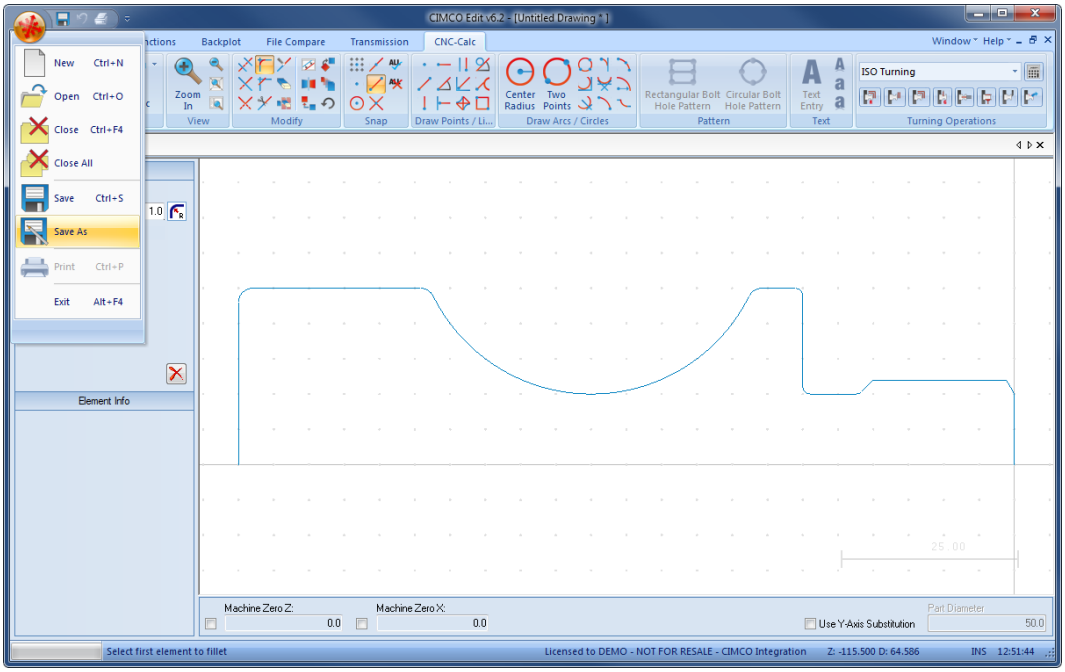


Now the drawing is finished, and it should look like the one below:



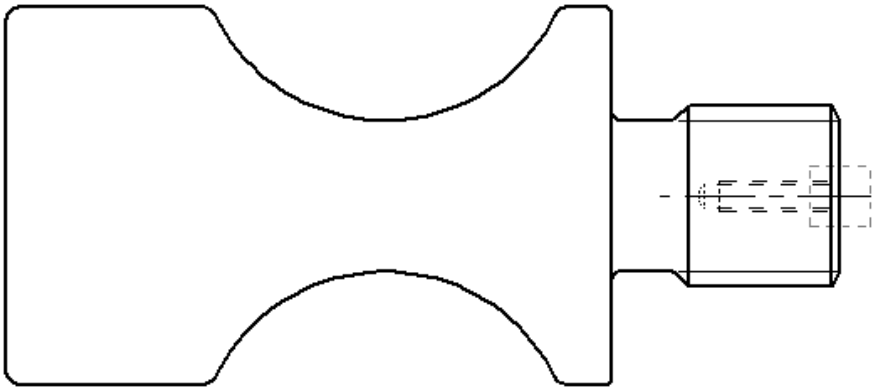
13.3. Save the File

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.

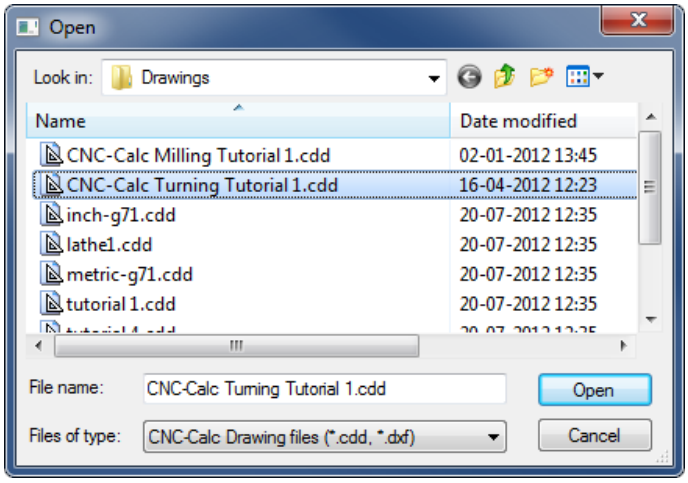
In order to produce the final part we will use the following operations:

- 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.

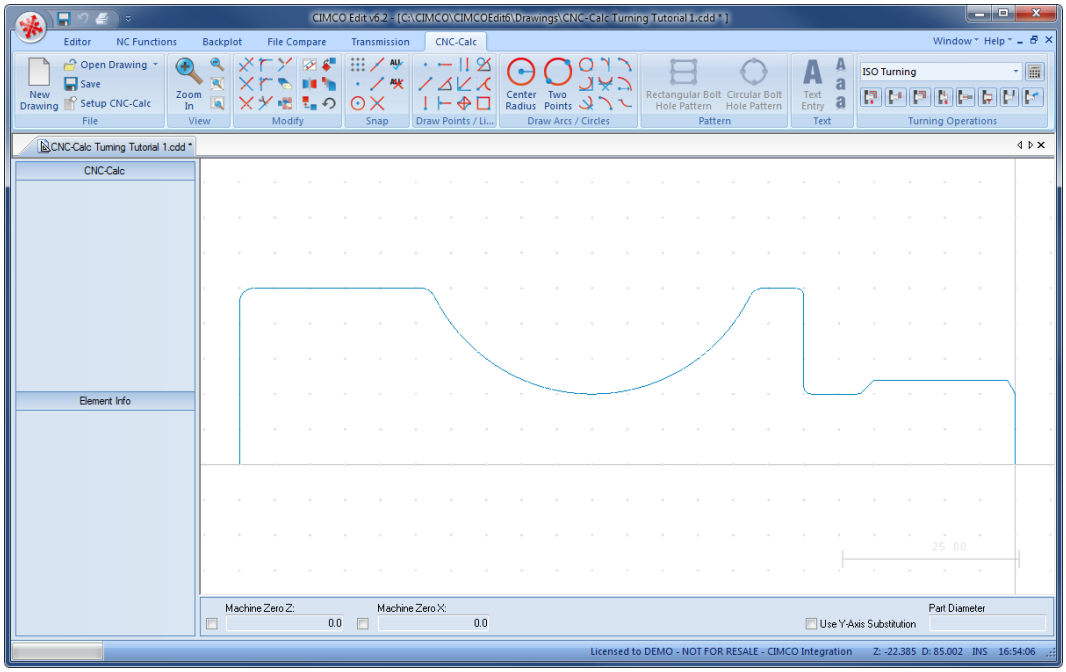
14.1. Open an Existing Drawing

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



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

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 **Setup CNC-Calc** and then **Global Colors** from the tree menu.

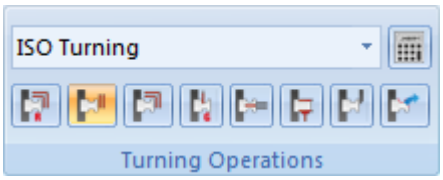
14.2. Facing the Front of the Stock

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.

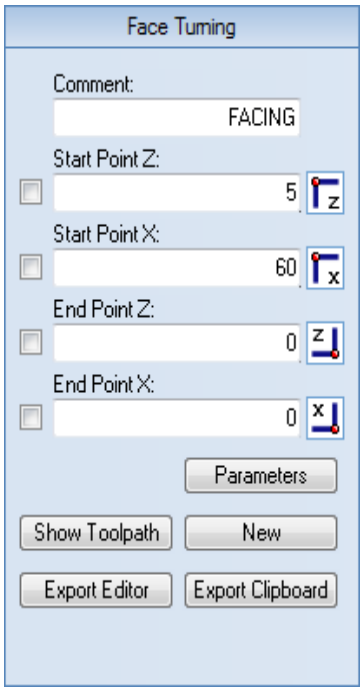
14.2.1. Creation of Facing Toolpaths

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.



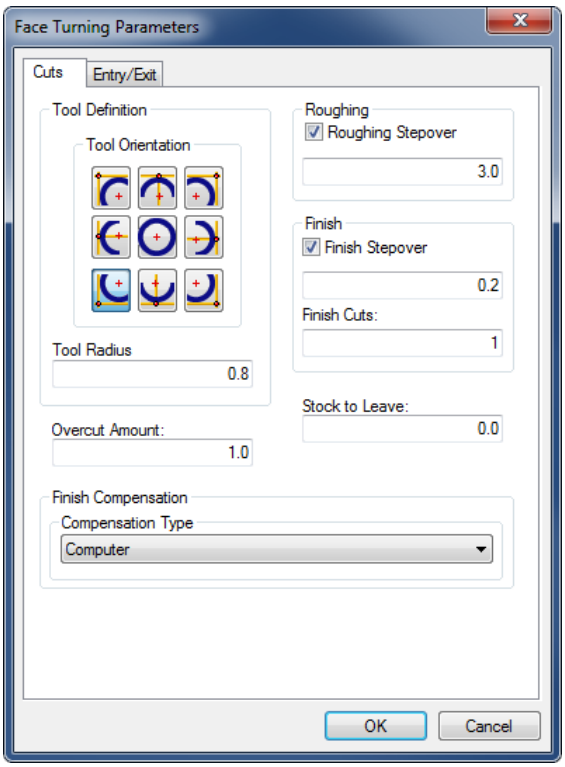
- **Comment**: This comment will be shown in the final NC-Program. It is always good to include a comment, in order to distinguish the various operations in the final program.
- **Start Point Z**: This is where the facing operation will start along the Z-axis.
- **Start Point X**: This is the start diameter 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.



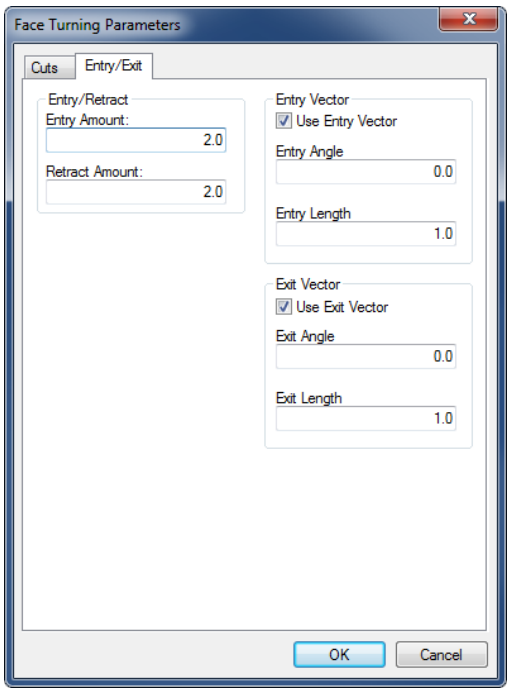
- **Tool Orientation**: The nine icons represent the possible nine orientations of the tool.
- **Tool Radius**: This is the nose radius of the tool.
- **Overcut Amount**: The distance the tool will cut longer than the End Point X value.
- **Roughing**: Here, the check-box can be used to enable or disable 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.



- **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.

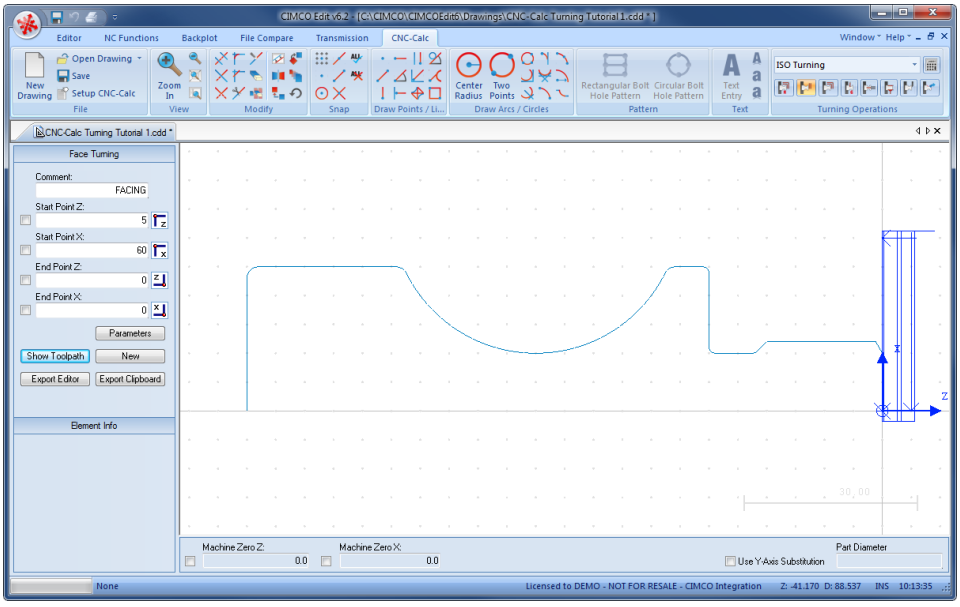
- **Entry Length**: The length of the entry vector.

- **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.

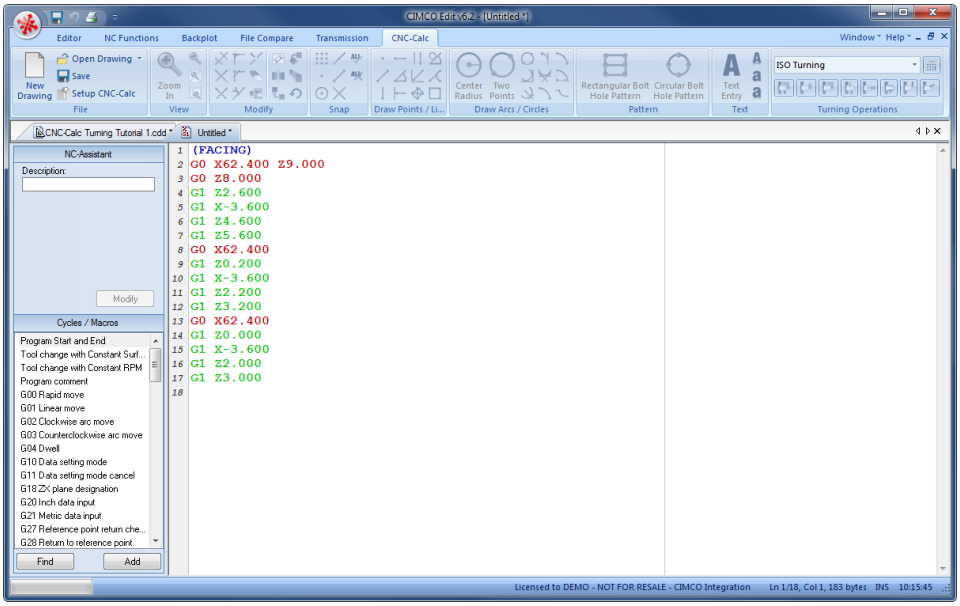
After entering the values, close the parameters dialog with **OK**. To show the generated toolpath on the drawing click on **Show Toolpath** button in the Face Turning pane.



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

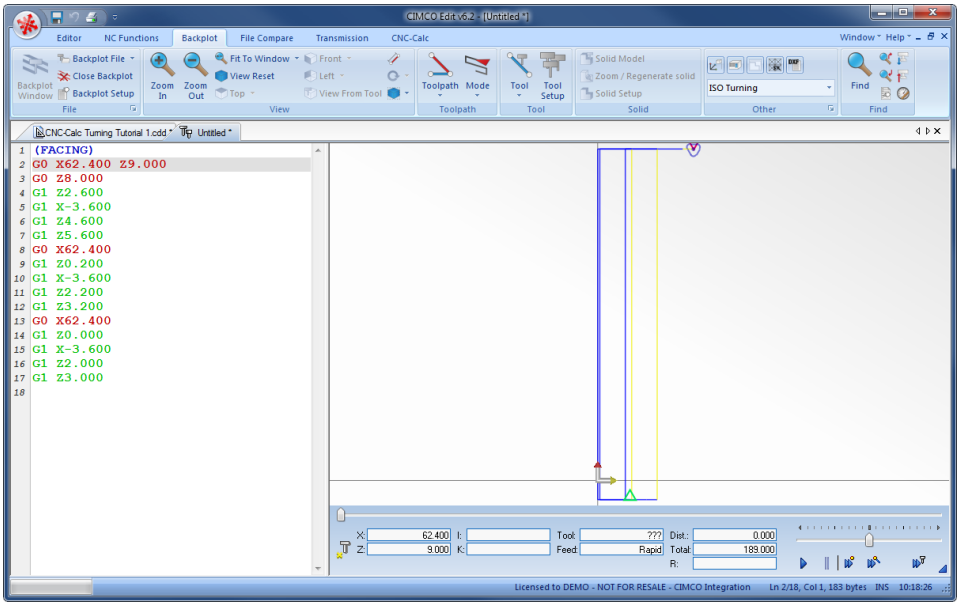
Click on **Export Editor** in order to generate the actual program. Now, a window like the one shown below will appear. Notice that the comment from the dialog is inserted at the top of the program as a comment.



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.

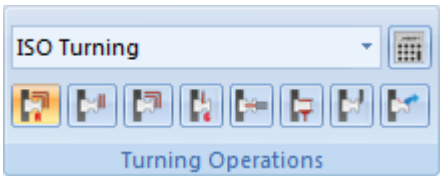
14.3. Roughing the Part

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.

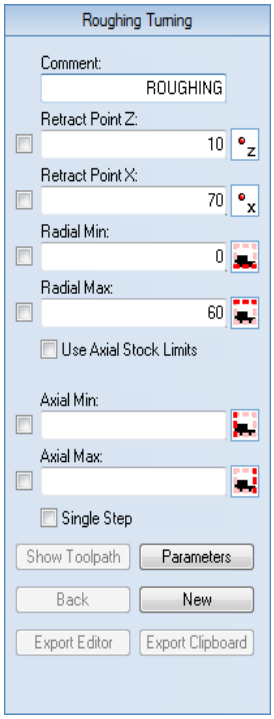
14.3.1. Creation of Roughing Toolpaths

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.

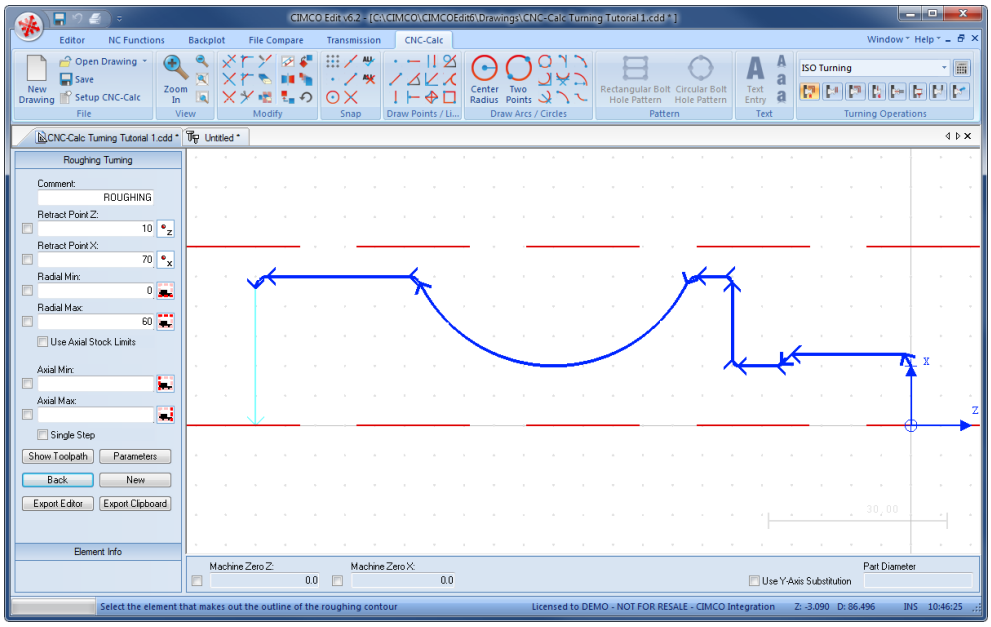


- **Comment**: This comment will be shown in the final NC-Program. It is always good to include a comment in order to distinguish the various operations in the final program.
- **Retract Point Z**: This is the Z value to where the operation will retract the tool after completion.
- **Retract Point X**: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

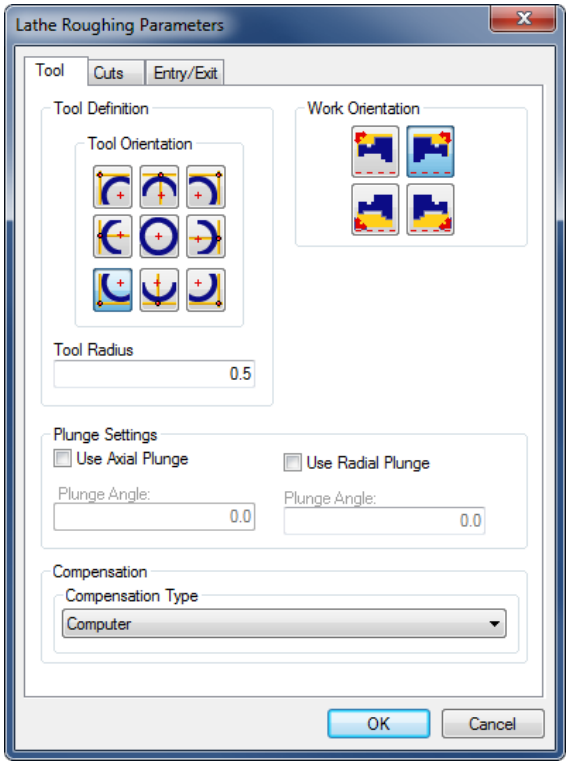
Now your drawing should look something like the one below.



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

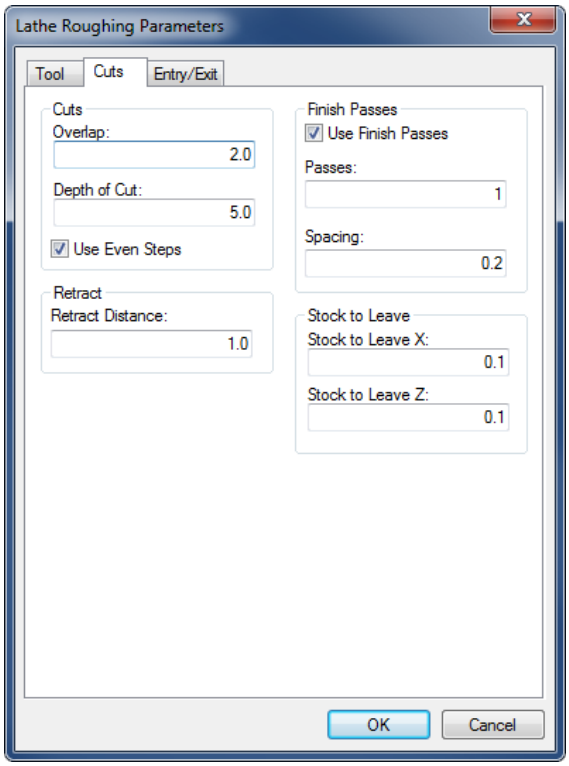
This tab is used to configure the settings for tool, work orientation and compensation type used for the operation.



- **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.

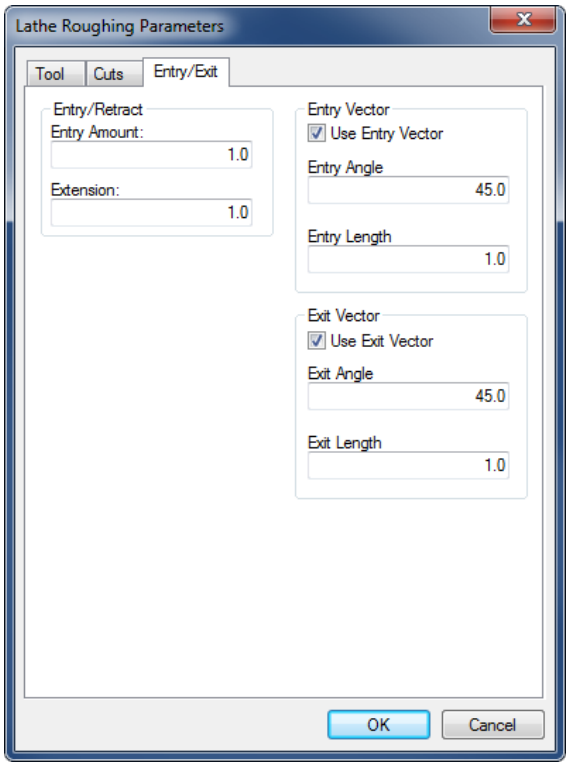


- **Overlap**: The distance that a cut will overlap the previous cut.
- **Depth of Cut**: The amount of material that is taken in each cut.

- **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.

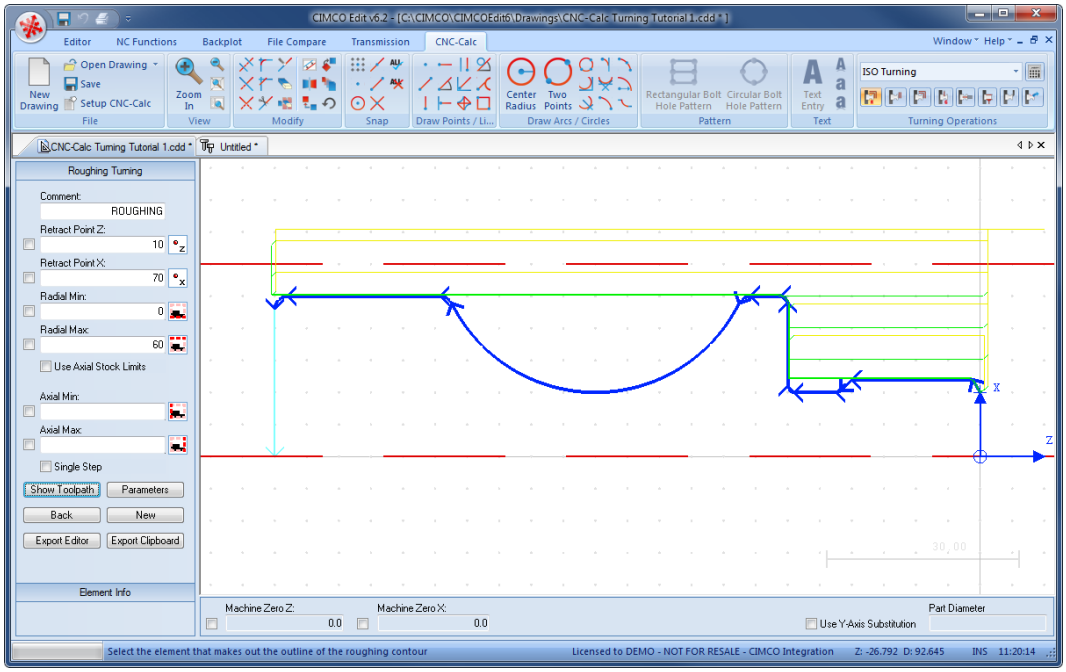
Entry/Exit Tab

Configure how the tool approaches and leaves the part.



- **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.



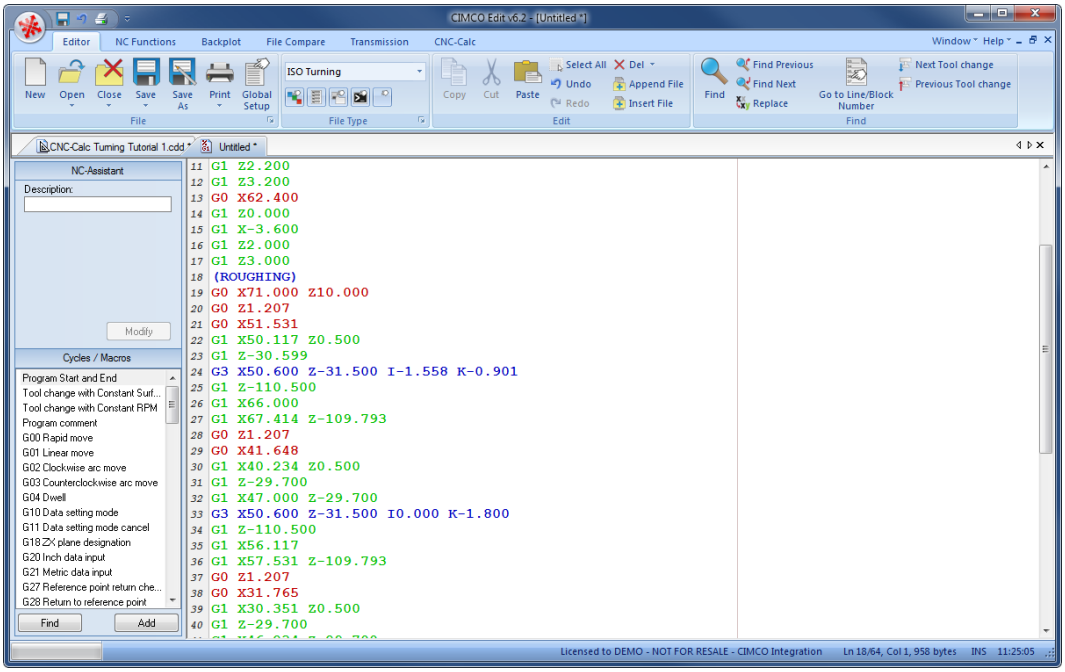
Try experimenting with the various parameters and see how they change the generated toolpath.

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.

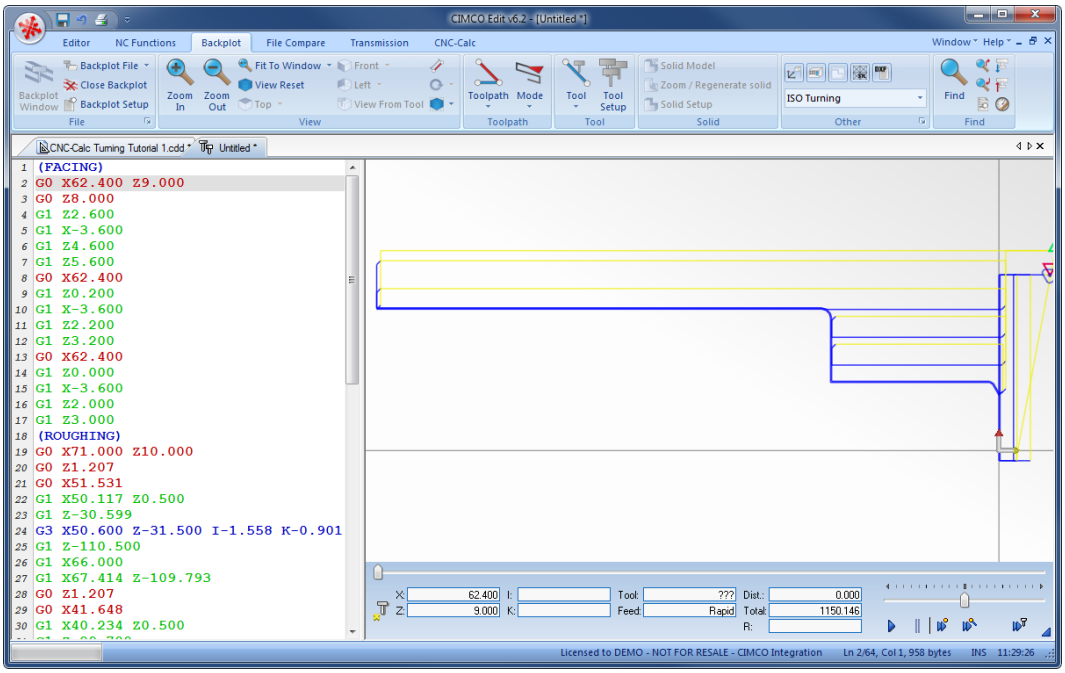
Now the NC program should look like the following screen.



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.



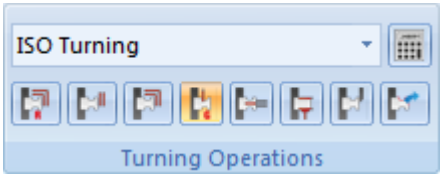
14.4. Grooving the Part

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.

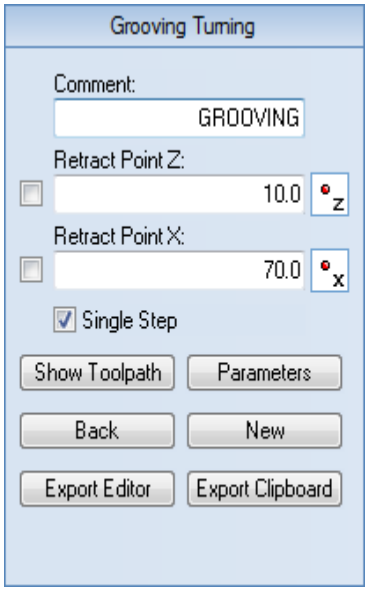
14.4.1. Creation of Grooving Toolpaths

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.



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

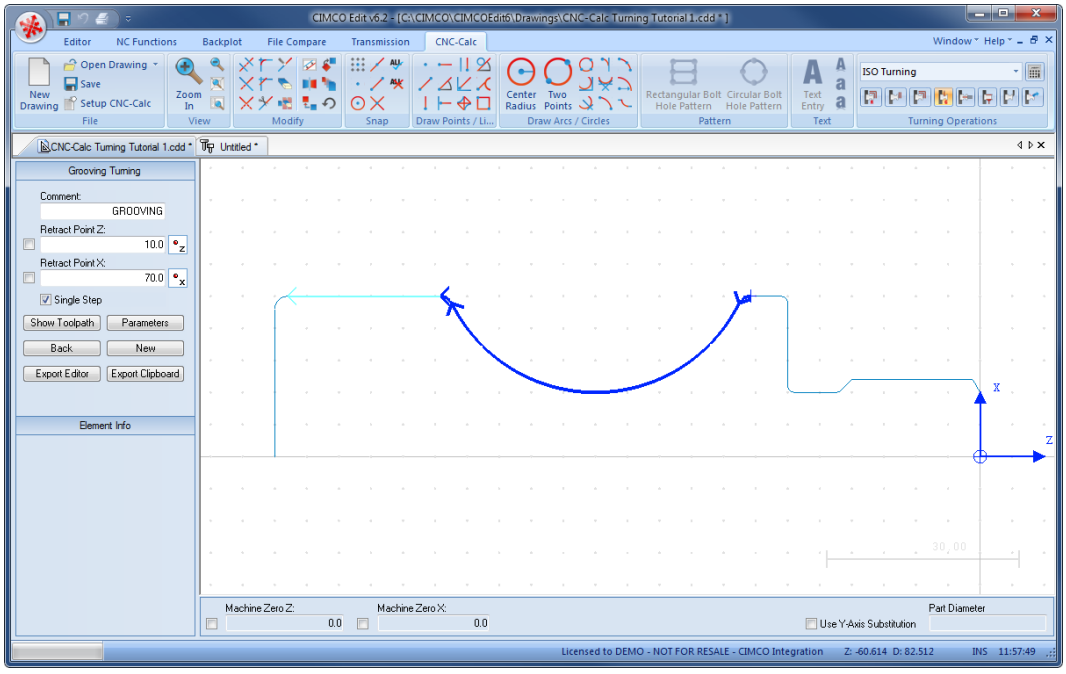


- **Comment**: This comment will be shown in the final NC program. It is always good to include a comment in order to distinguish the various operations in the final program.
- **Retract Point Z**: This is the Z value, to where the operation will retract the tool after completion.
- **Retract Point X**: 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.

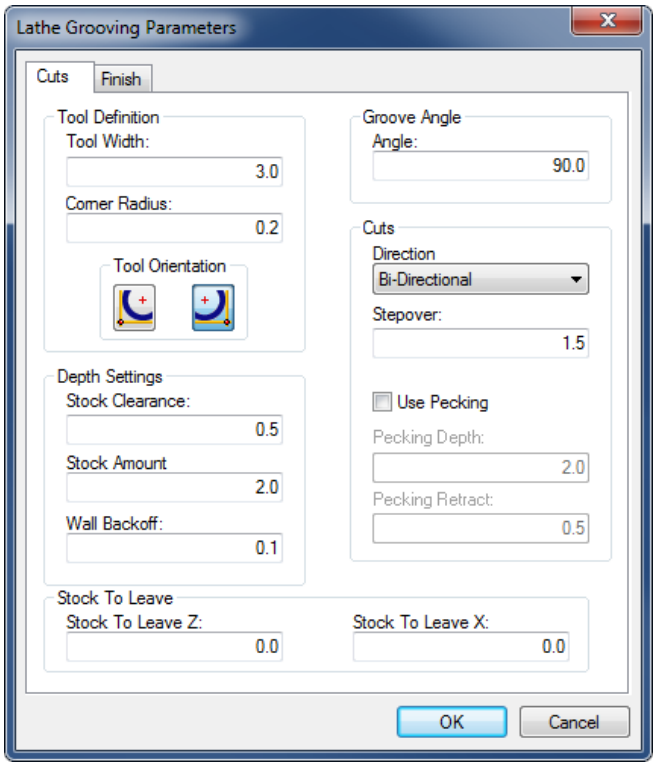
Now your drawing should look something like the one below.



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.

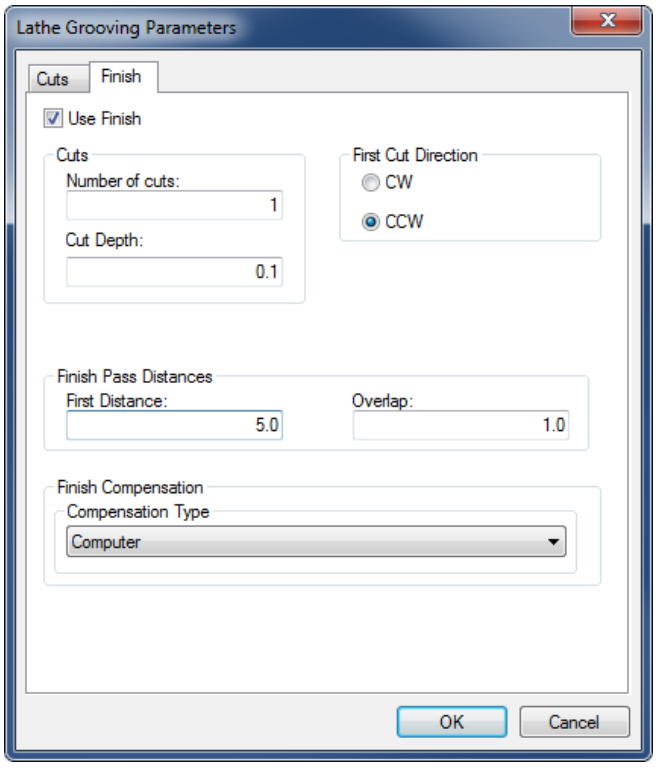


- **Tool Width**: The width of the tool.
- **Corner Radius**: The corner radius of the tool.
- **Tool Orientation**: The two icons indicate how the tool is zeroed.
- **Stock Clearance**: Indicates how far off the part the tool should move before making sideways moves.
- **Stock Amount**: Indicates how 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

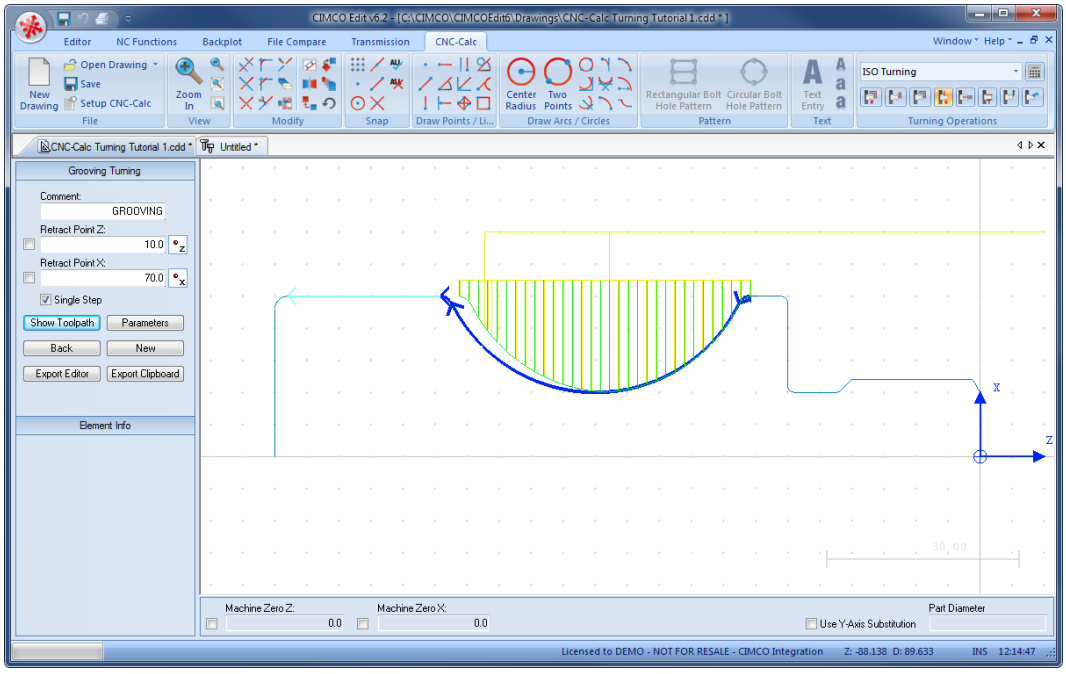
Use this tab to configure how the finish passes for the operation should be performed.



- **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.



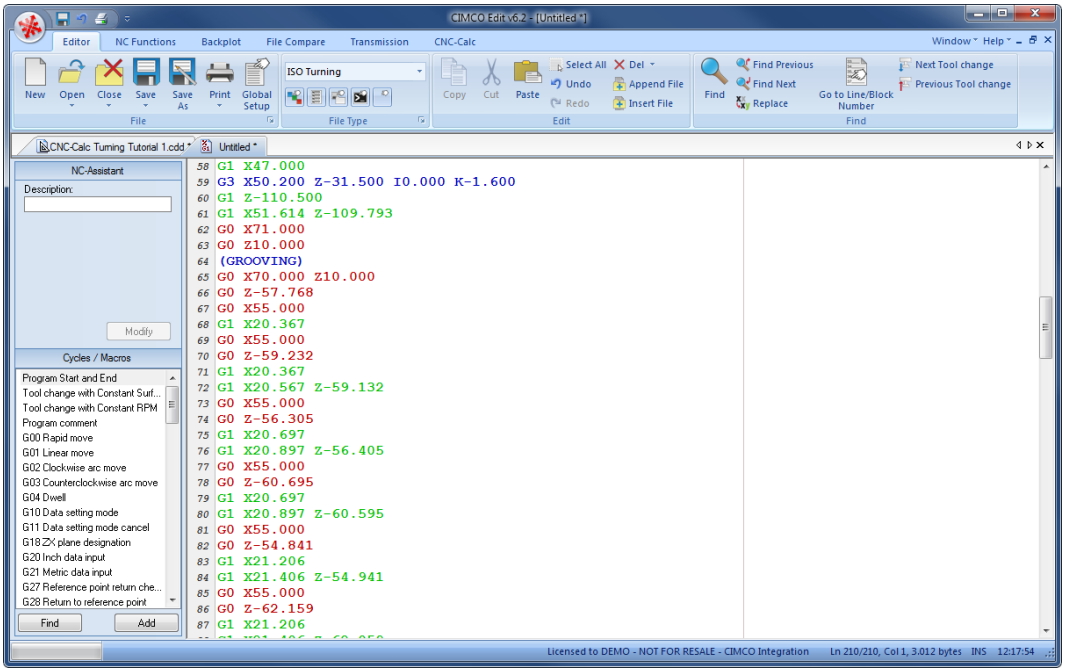
Try experimenting with the various parameters and see how they change the generated toolpath.

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

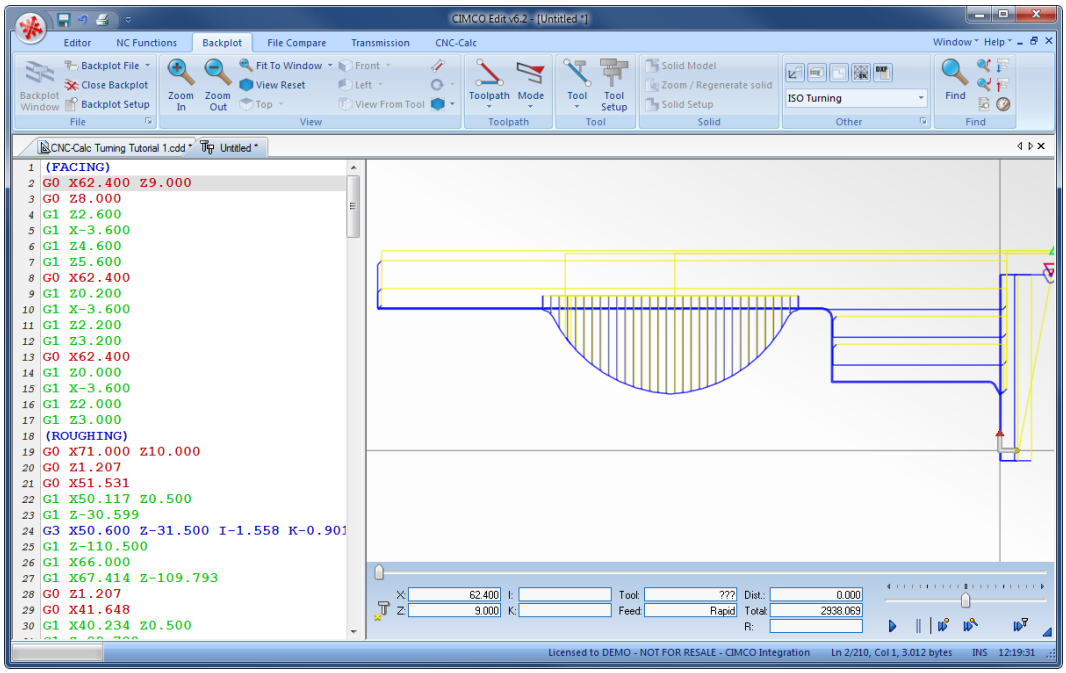
Now the NC program should look like the following screen.



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.



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

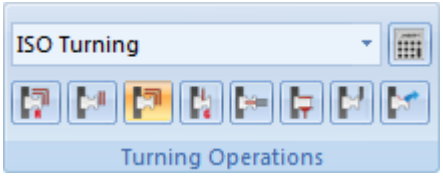
14.5. Finishing the Part

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.

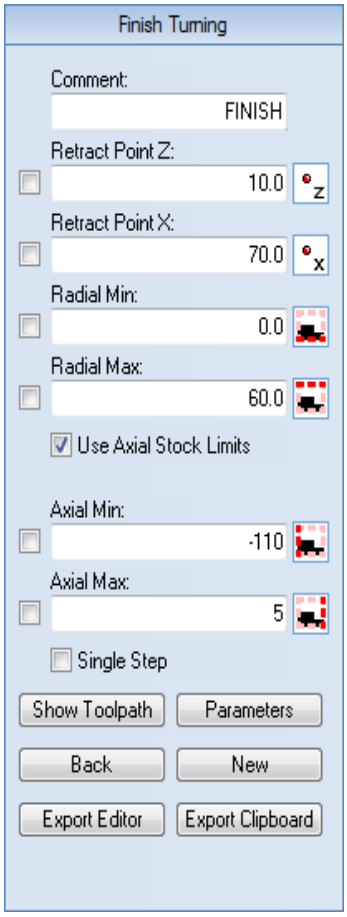
14.5.1. Creation of Finishing Toolpaths

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.

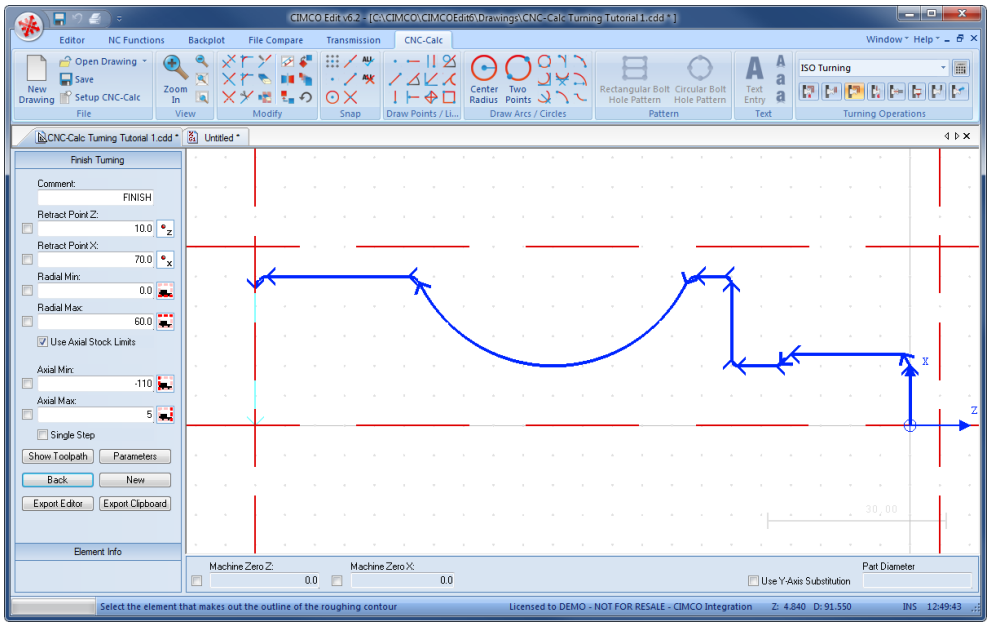


- **Comment**: This comment will be shown in the final NC-Program. It is always good to include a comment in order to distinguish the various operations in the final program.
- **Retract Point Z**: This is the Z value to where the operation will retract the tool after completion.
- **Retract Point X**: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

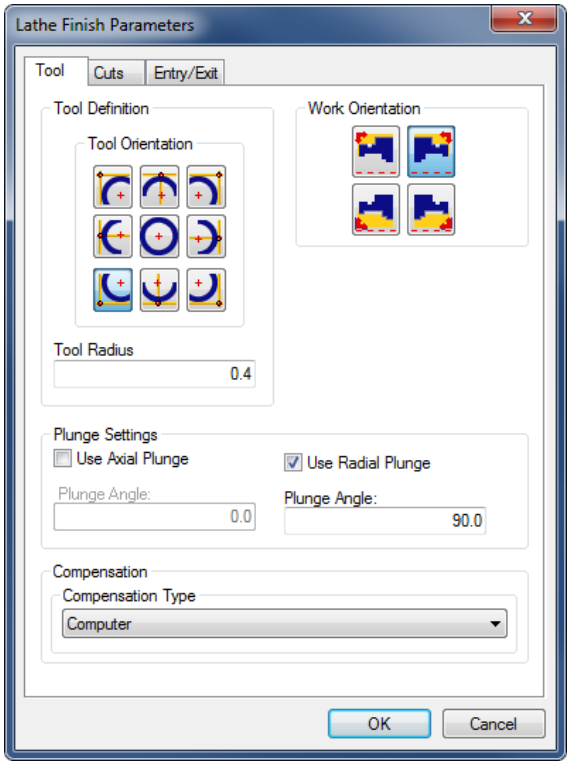
Now your drawing should look something like the one below.



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

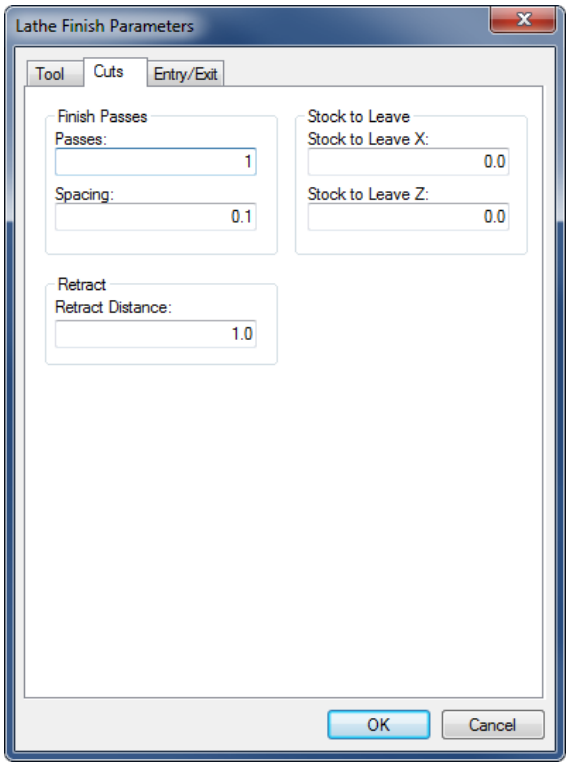
Configure settings for tool, work orientation and compensation type used for the operation.



- **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.



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

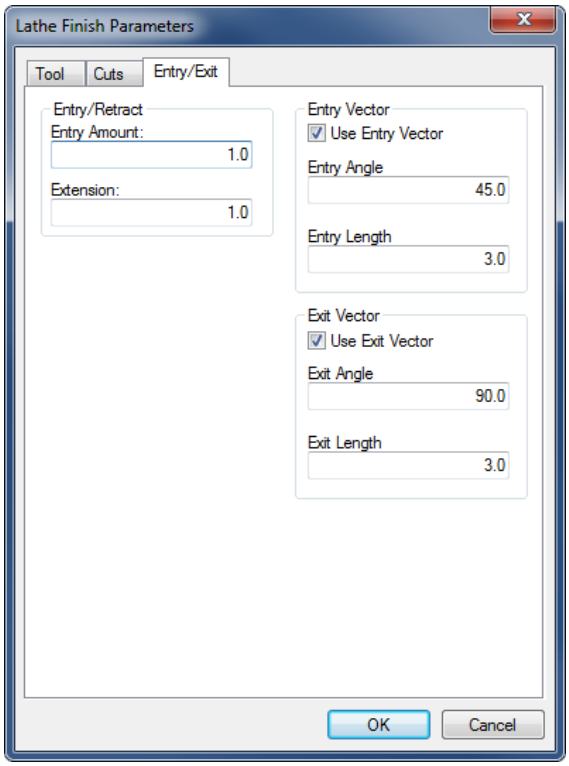
- **Retract Distance**: The distance that the tool retracts from the stock before a return move is made.

- **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.



- **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.

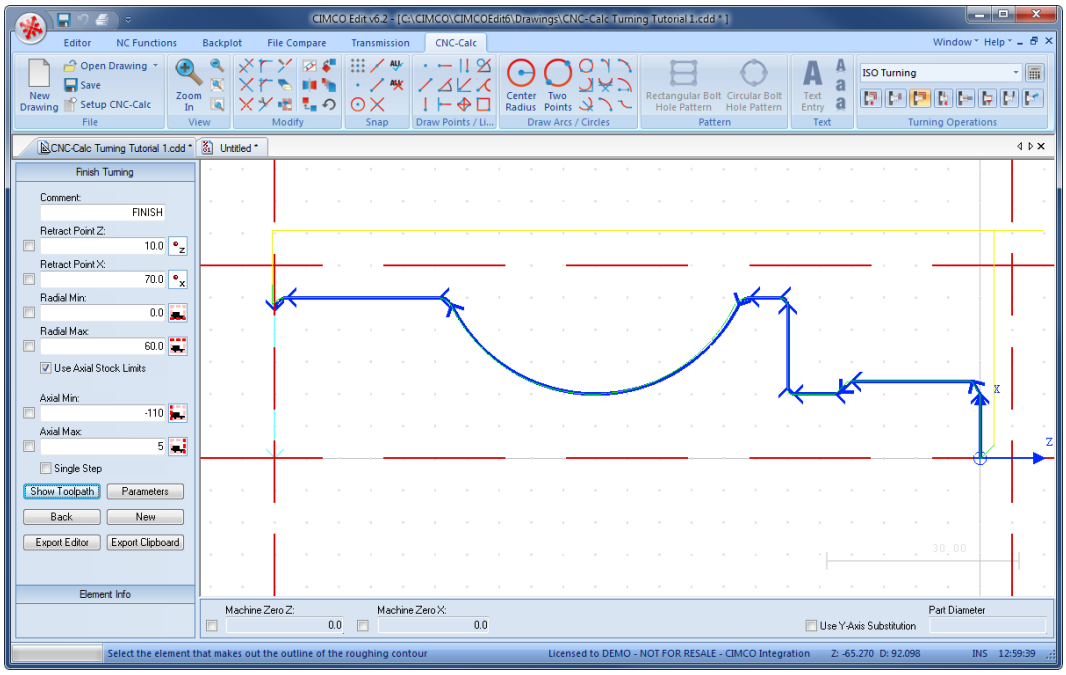
- **Use Entry Vector**: Enable/Disable the use of entry vector.

- **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.



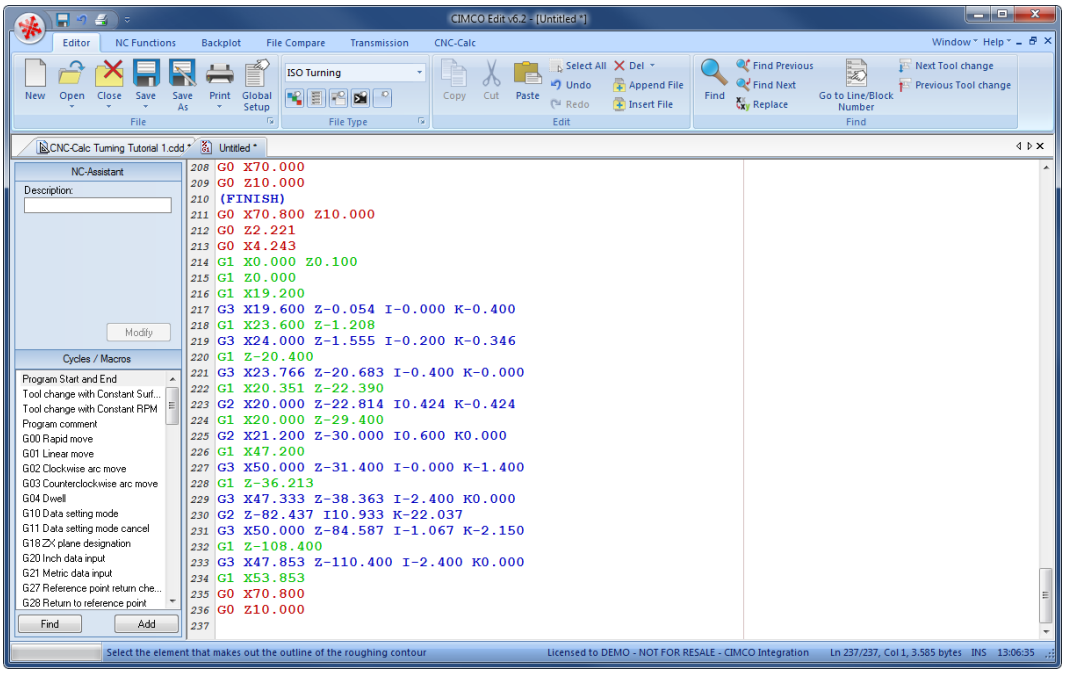
Try experimenting with the various parameters and see how they change the generated toolpath.

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

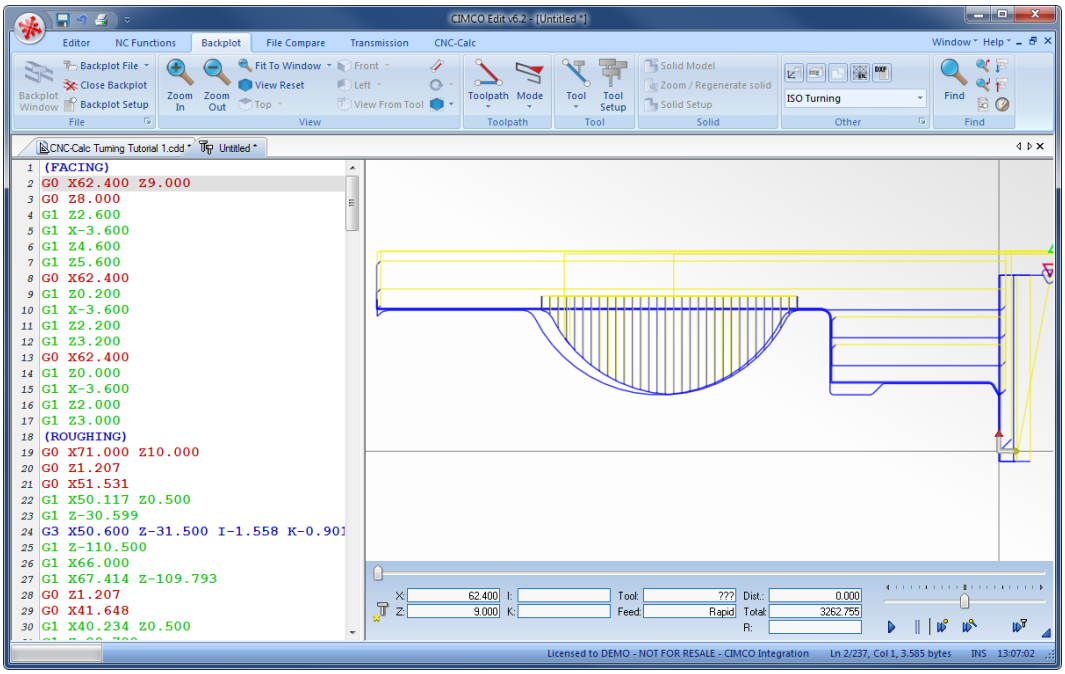
Now the NC program should look like the following screen.



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.



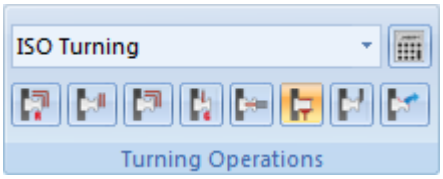
14.6. Threading the Part

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.

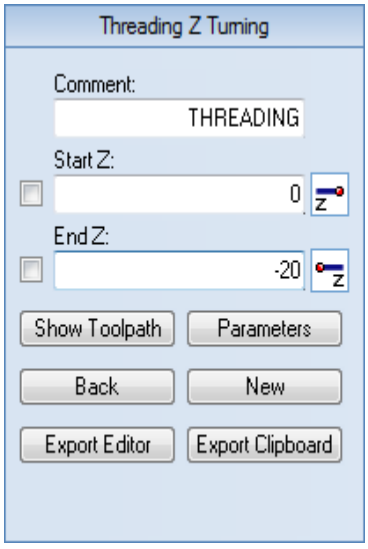
14.6.1. Creation of Threading Toolpaths

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.



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

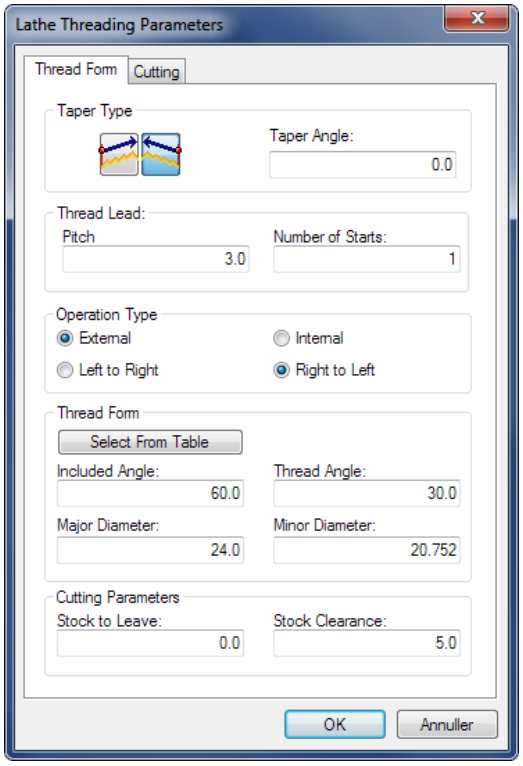


- **Comment**: This comment will be shown in the final NC-Program. It is always good to include a comment in order to distinguish the various operations in the final program.
- **Start Z**: This is the Z value where the operation starts.
- **End Z**: This is the Z value where the operation ends.

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

Configure the form of the thread and the taper in the operation.

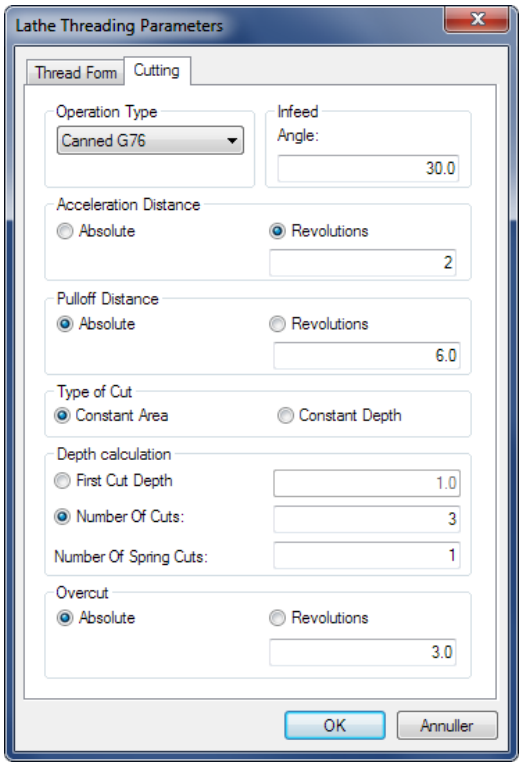


- **Taper Type**: If the taper angle is not zero, a conical thread will be produced. The two icons represent the two ways the cone can go.
- **Taper Angle**: Is the angle of the conical thread.
- **Pitch**: Defines the distance it takes for the thread to revolve one round around the part.
- **Number 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.



- **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).

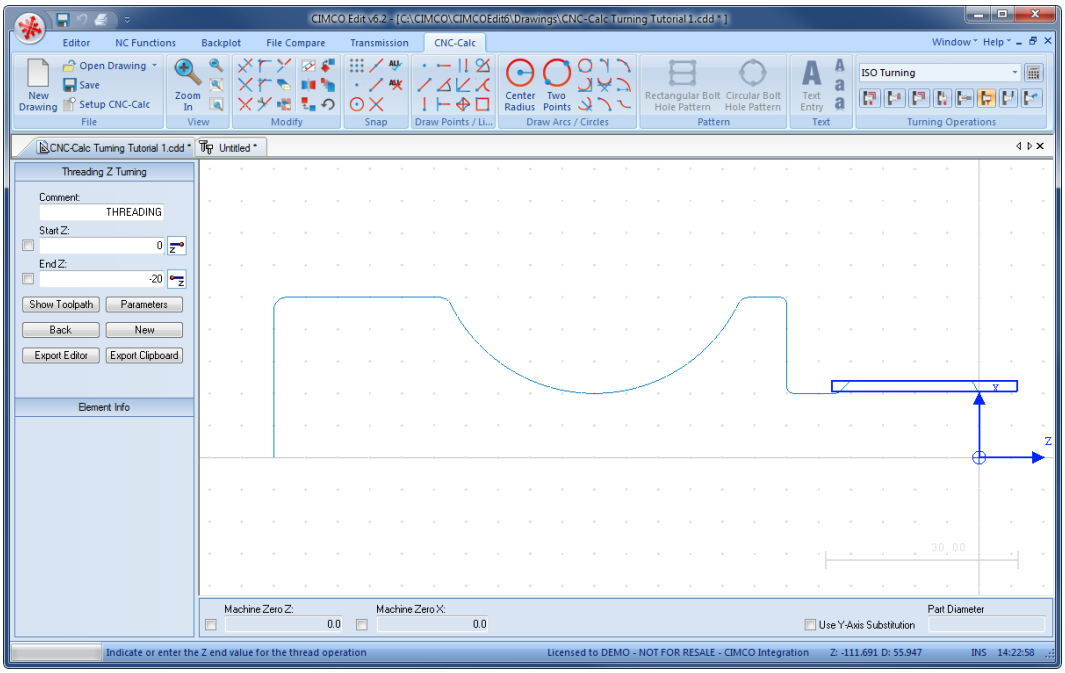
- **Number of Spring Cuts**: If spring cuts are used, this many cuts will be made at the final depth.

- **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.



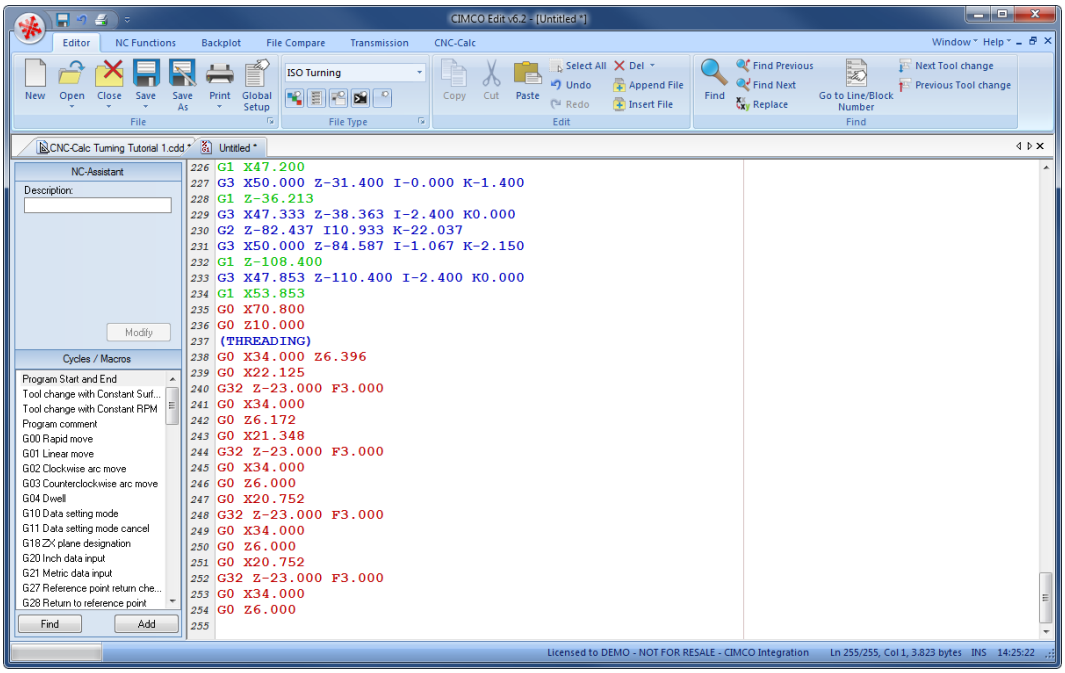
Try experimenting with the various parameters and see how they change the generated toolpath.

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

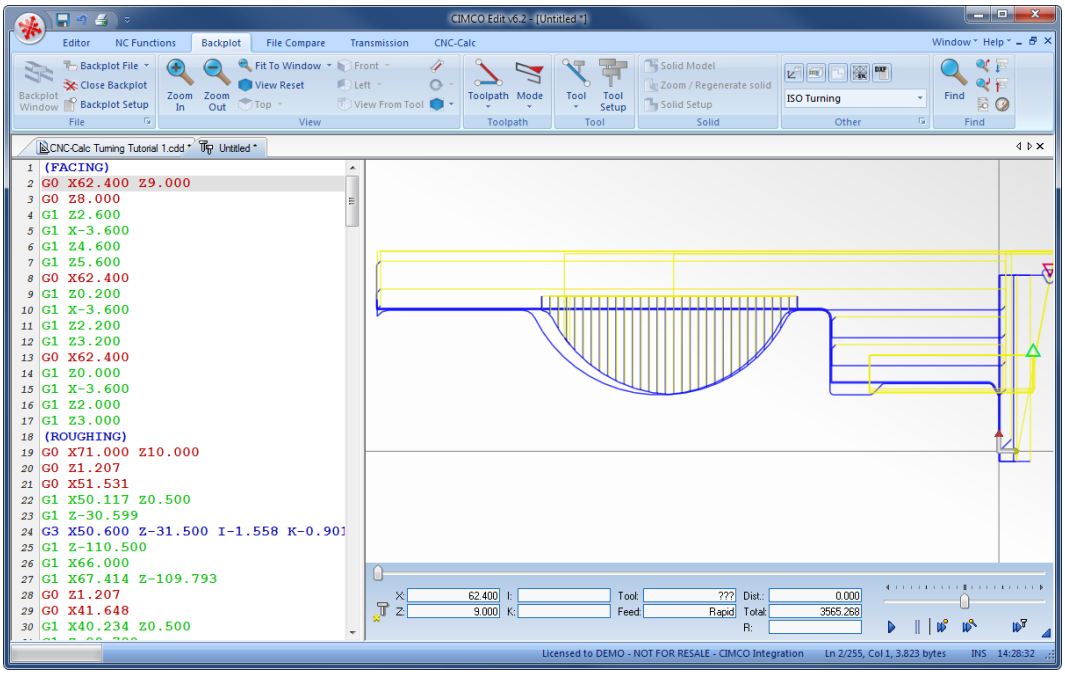
Now the NC program should look like the following screen.



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.

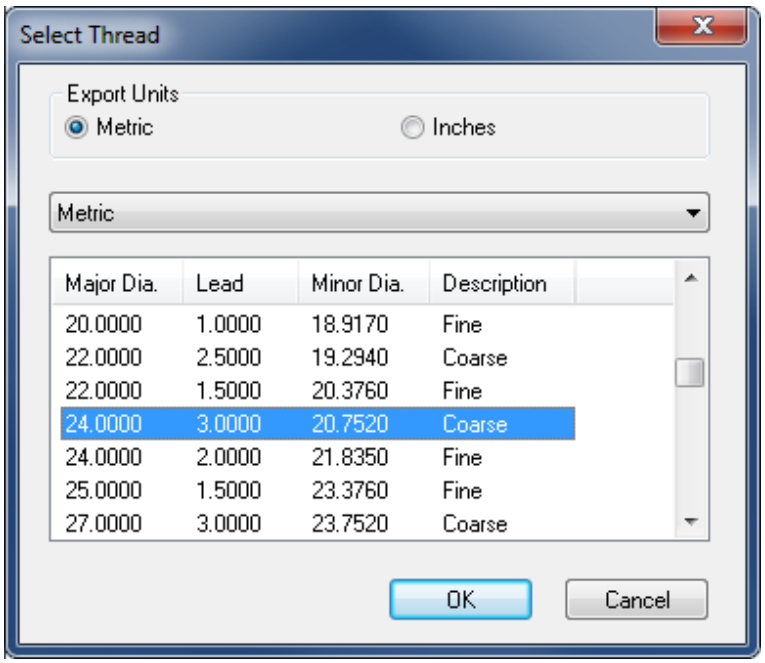
Now a window like the one below will appear.



14.6.4. 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.



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.

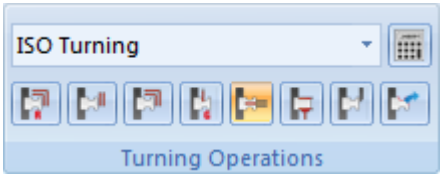
14.7. Drilling the Part

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.

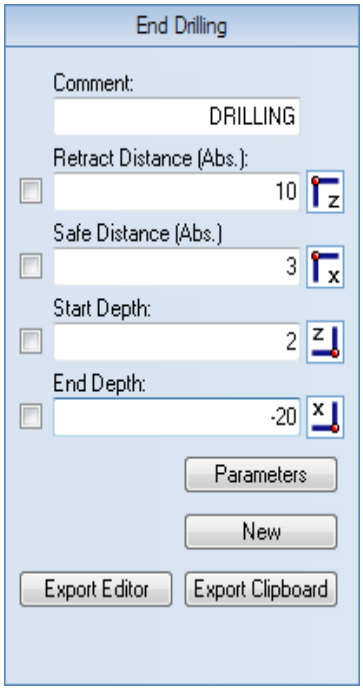
14.7.1. Generate a Drill Cycle

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.



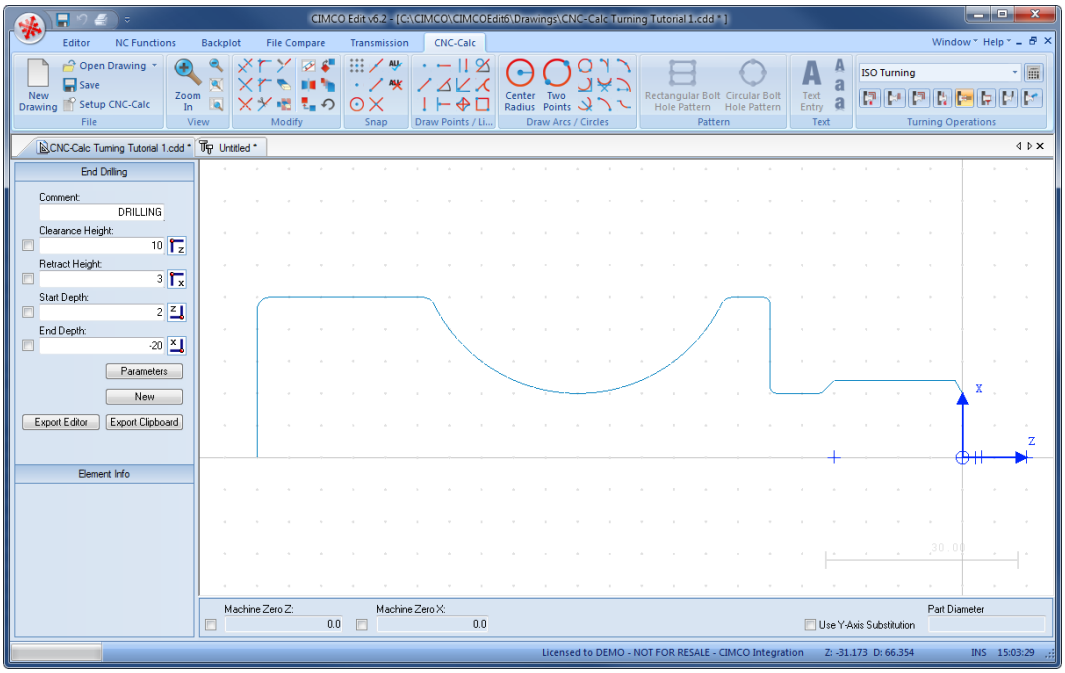
This will open the End Drilling pane to the left of the drawing area. Now insert the values shown in the dialog below.



- **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.
- **End Depth**: The final depth of the operation.

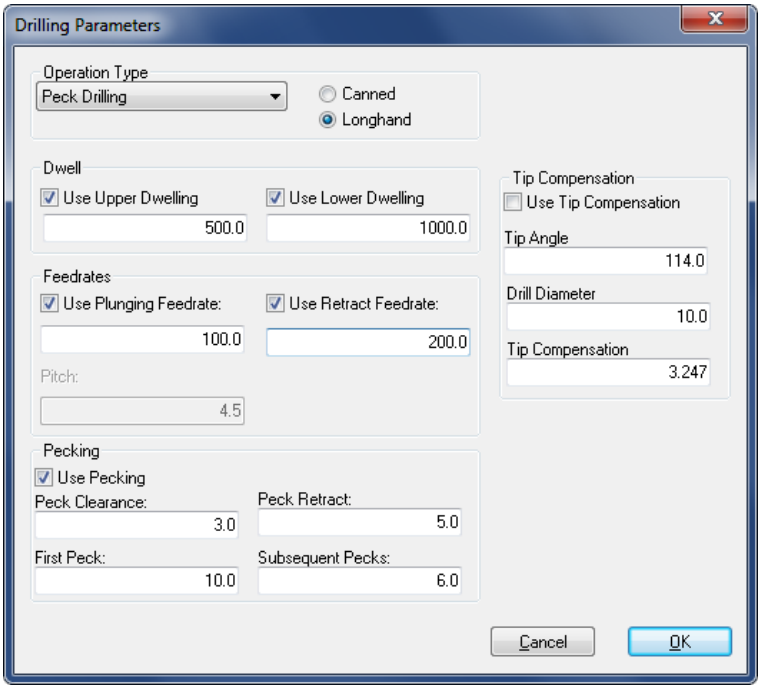
The Drilling operation is defined by the above parameters, after the entry of which the screen will look something like the one shown below.

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



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



- **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.

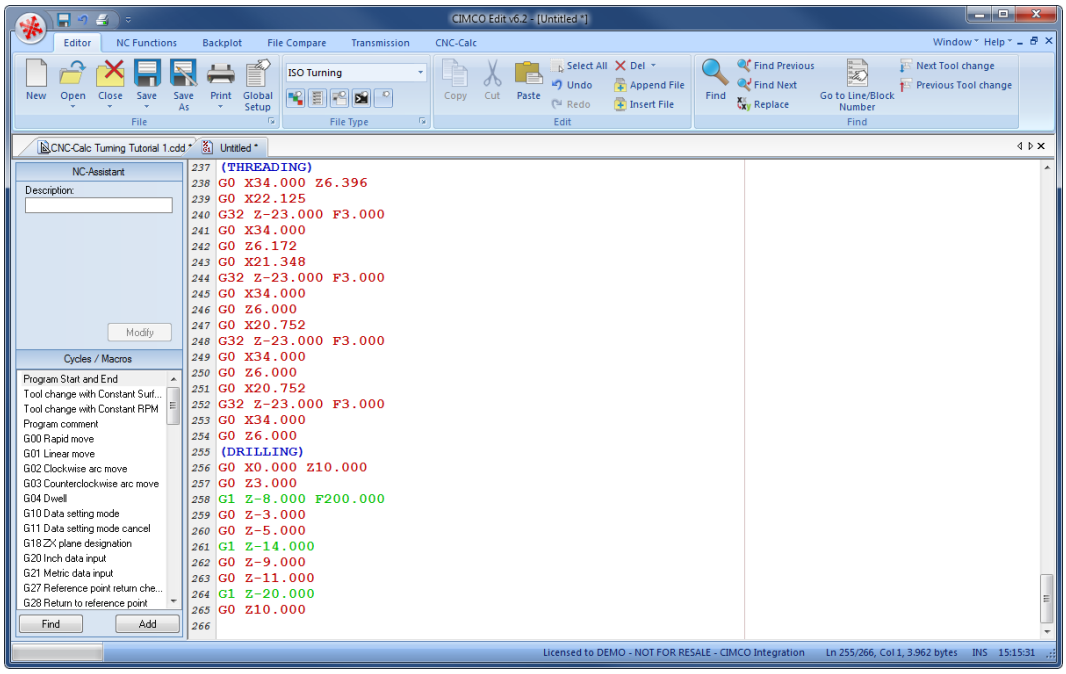
Try experimenting with the various parameters and see how they change the generated toolpath.

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

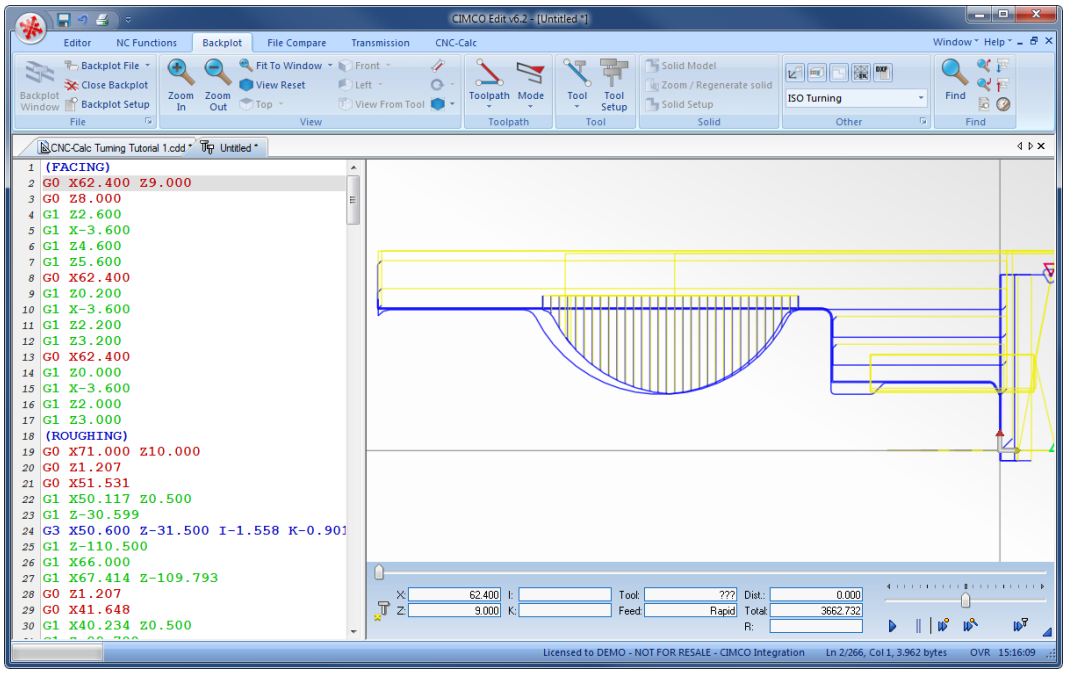
Now the NC program should look like the following screen.



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.



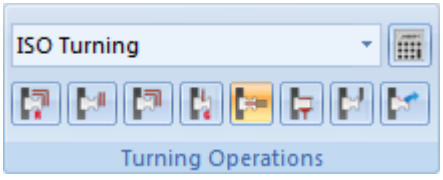
14.8. Tapping the Part

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.

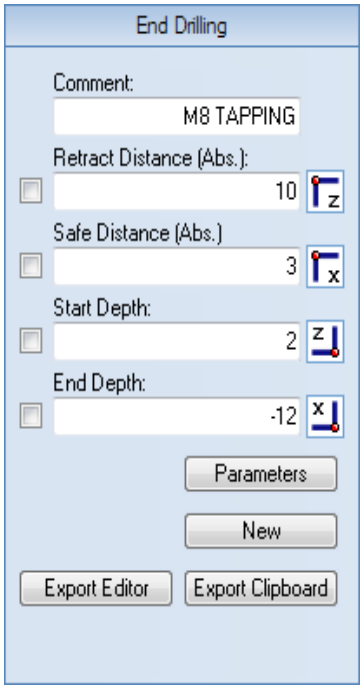
14.8.1. Generate a Tap Cycle

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.



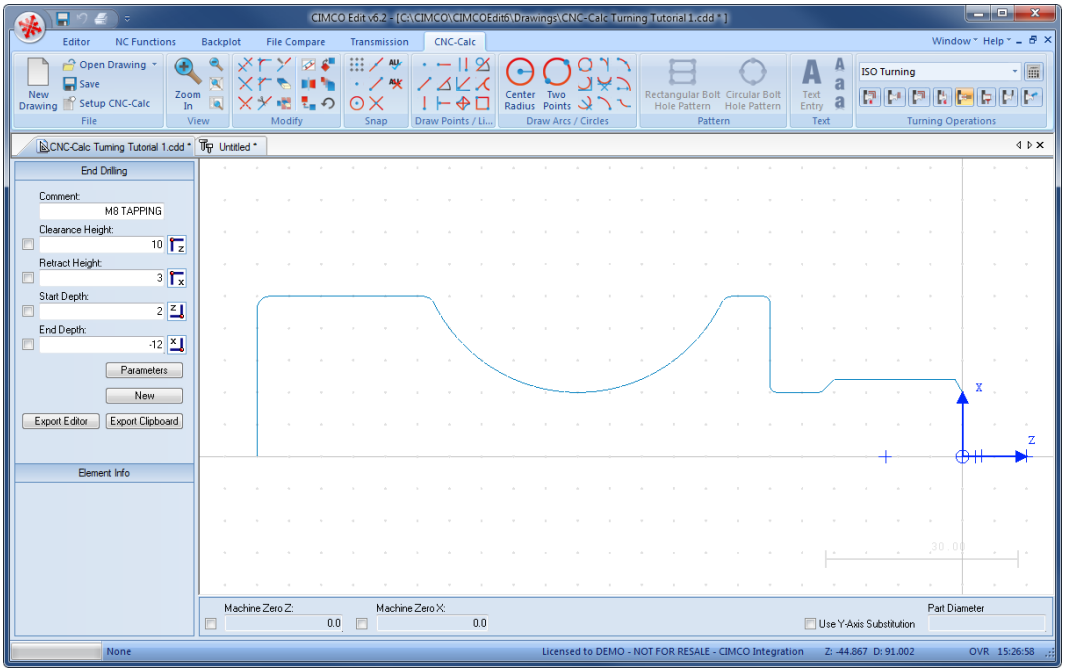
This will open the End Drilling pane to the left of the drawing area. Now insert the values shown in the dialog below.



- **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.

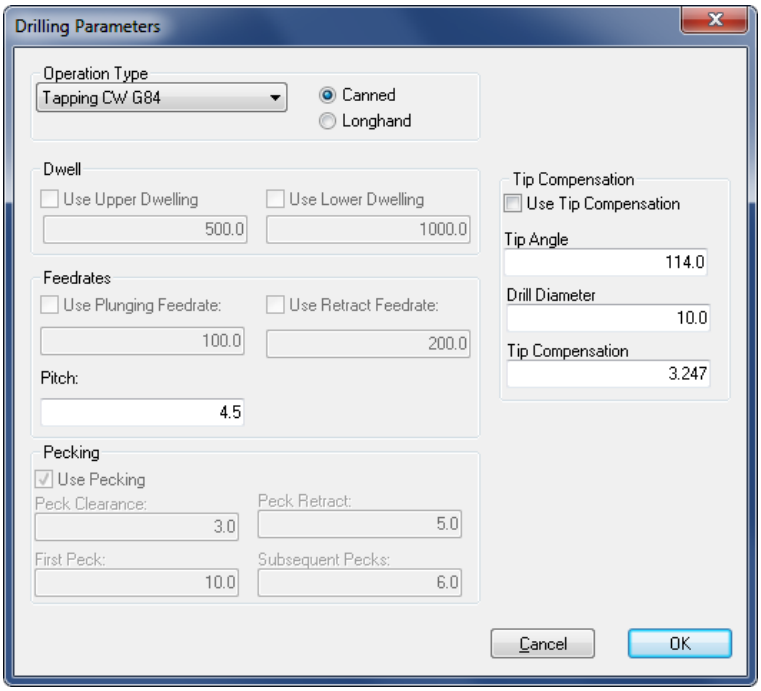
The four distances entered are shown as crosses on the drawing.



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**: 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.

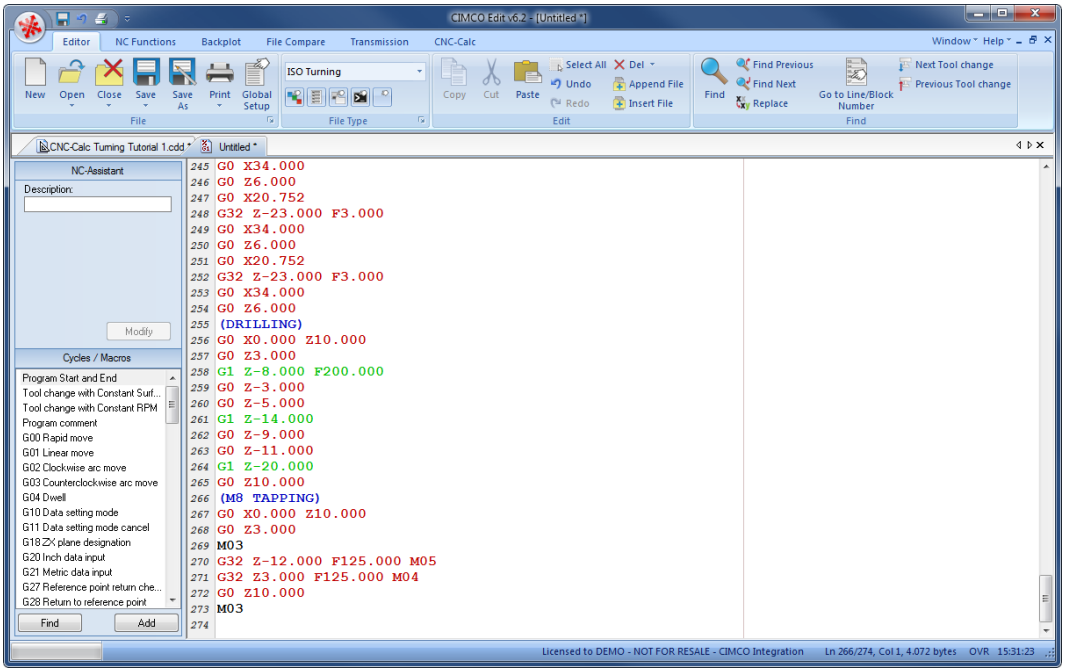
Try experimenting with the various parameters and see how they change the generated toolpath.

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.

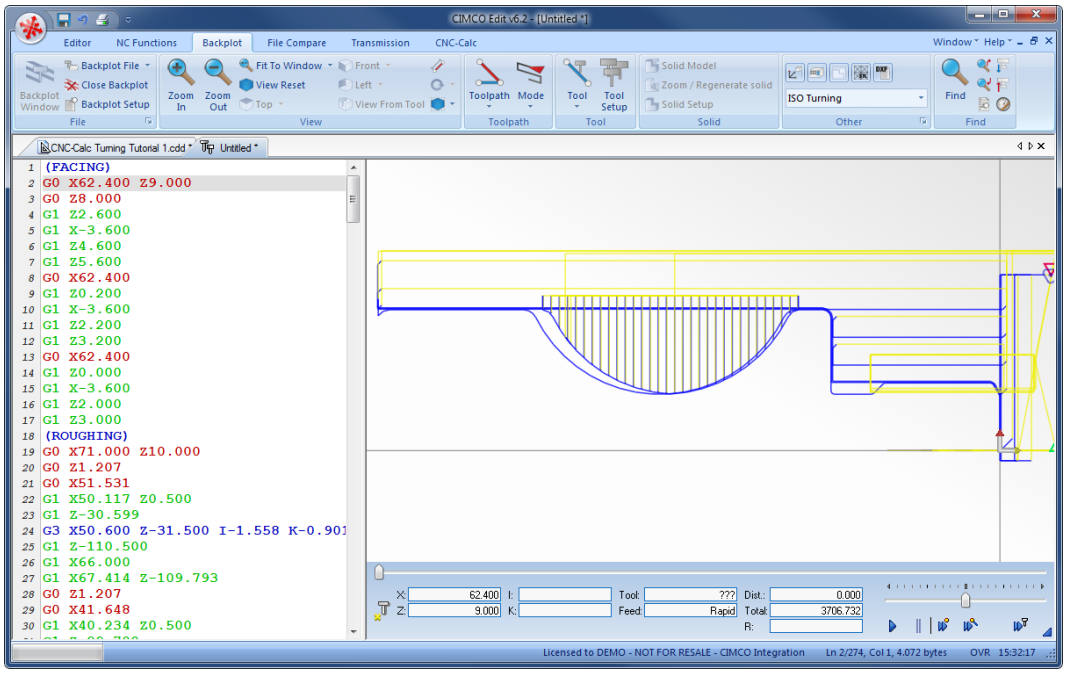
Now the NC program should look like the following screen.



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.



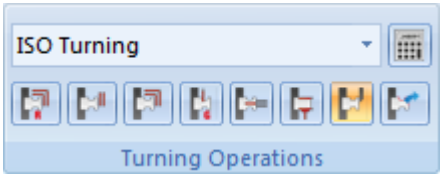
14.9. Parting-off the Part

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.

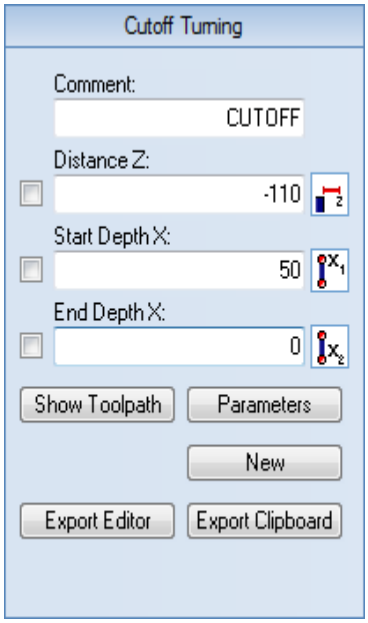
14.9.1. Create a Cutoff Toolpath

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** icon in the *Turning Operations* toolbar.



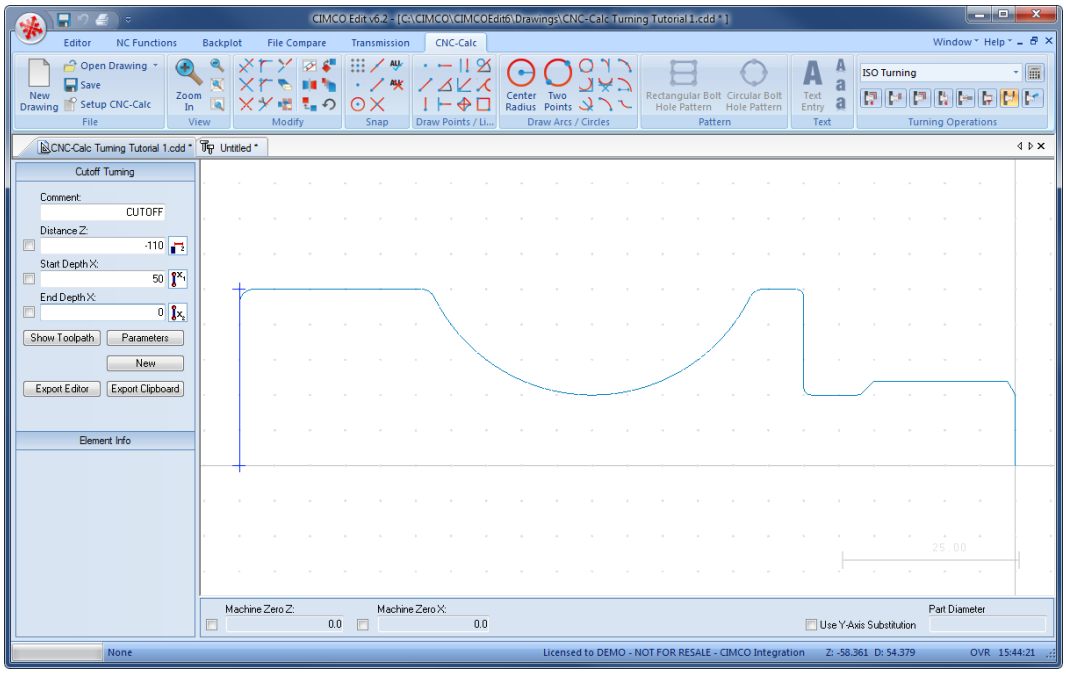
This will open the Cutoff Turning pane to the left of the drawing area. Now insert the values shown in the dialog below.



- **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.
- **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.

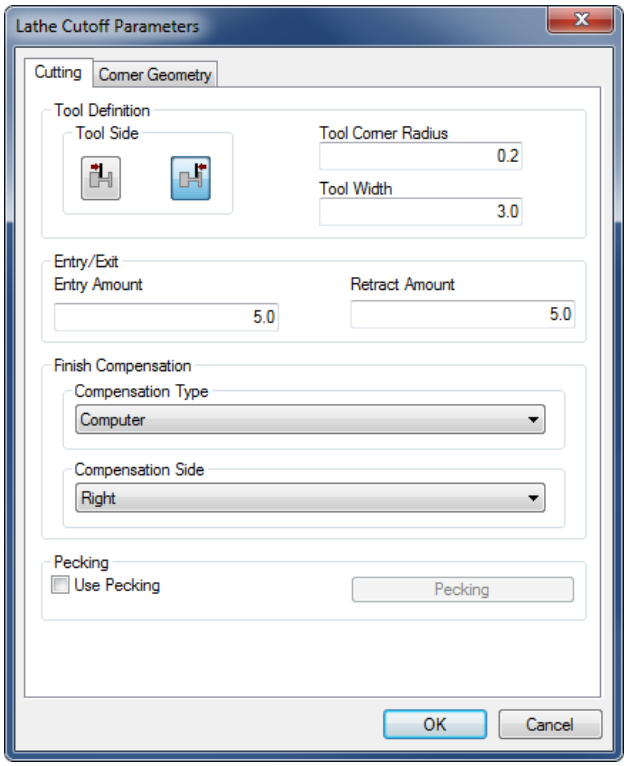
The three values entered are shown as a vertical line on the drawing.



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

Use this tab to configure cutting parameters for the operation.

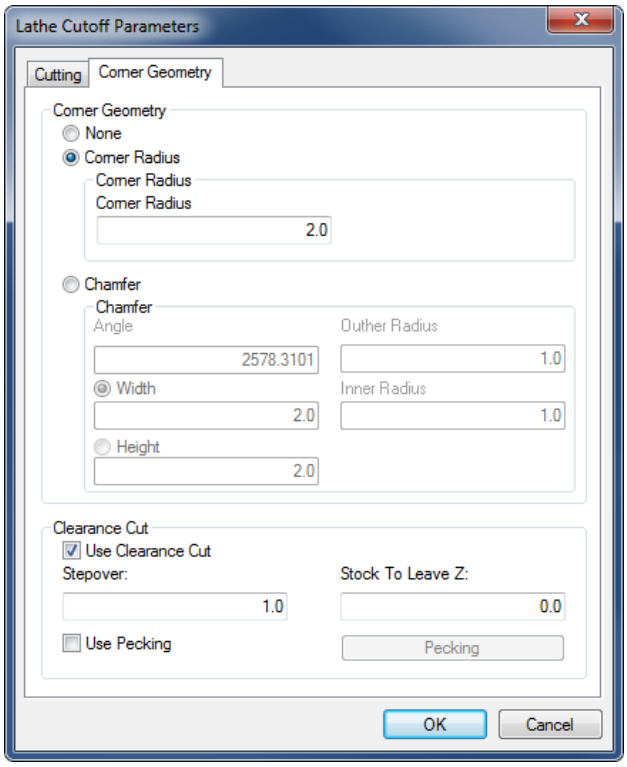


- **Tool Side**: Use these two icons to define the right side of the tool as the cutting side.
- **Tool Corner Radius**: The corner radius of the tool.
- **Tool Width**: The width of the tool used for the operation.
- **Entry Amount**: The length that the tool will start the cut above Start Depth X.
- **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.



- **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.

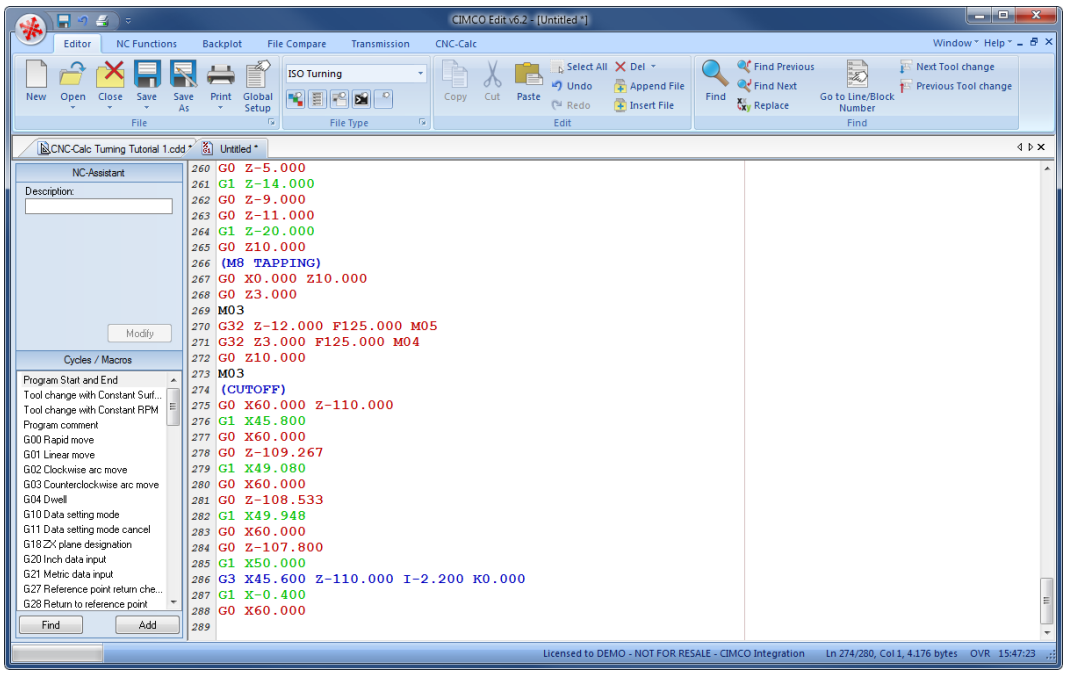
Try experimenting with the various parameters and see how they change the generated toolpath.

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

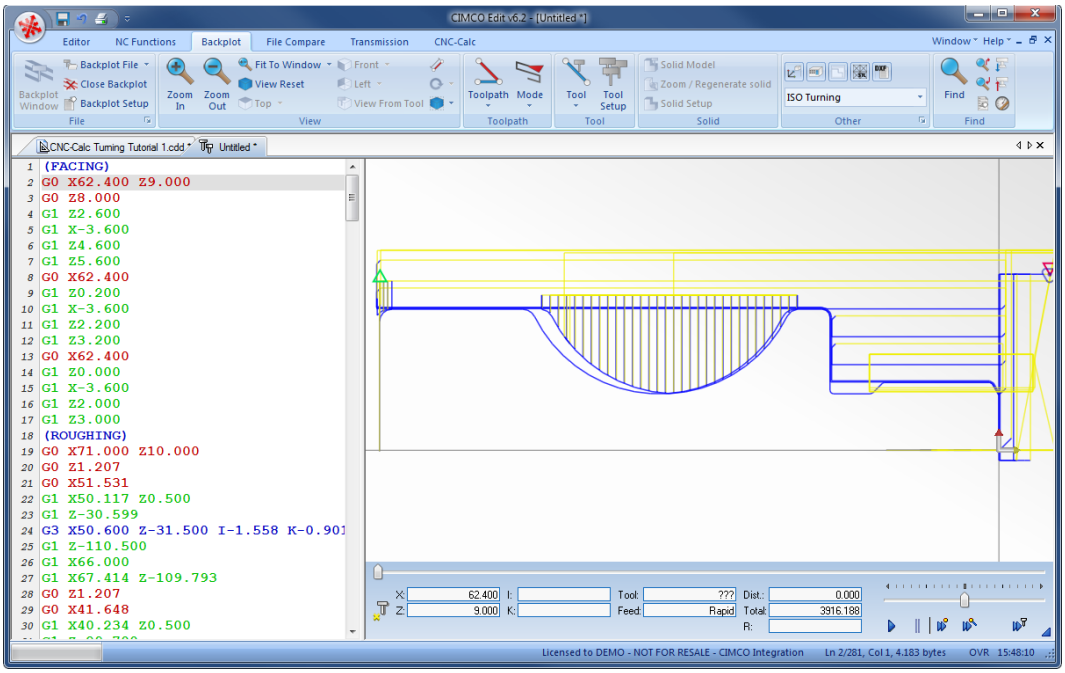
Now the NC program should look like the following screen.



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.



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