FANUC

*i*HMI Cycle Programming for Turning Exercises

Version 1.0

0. Table of Contents

1. Preface

The following document is dedicated to programmers who has no experience with iHMI Cycle Programming. Using Step-by-step Hands-on method, it introduces operation of iHMI and simplifies training of iHMI Cycle Programming for Turning machines. Gradually increasing complexity it gives the possibility to intuitively review operation and programming methods and creatively increase the knowledge, using 3 Examples of turning programs, made on iHMI NC Operation programming (CNCOpera). The generation of programs is shown step-by-step. All details regarding operation and programming are explained extensively. The Exercises can be done beside on real CNC also on available CNC Guide project "30iTB iHMI Programming Training". The simulator has 2 Paths and 4 Spindles. It gives you opportunity to check other standard iHMI operations and even to deepen knowledge preparing the below exercises on the second (right) main spindle.

To get the simulator Ready, use key  to set Servo Ready. This key emulates the key "Ready" on standard machines. To start programming go to "EDIT" mode and select "NC Operation" on iHMI Home Screen.

Options used in CNC Guide project

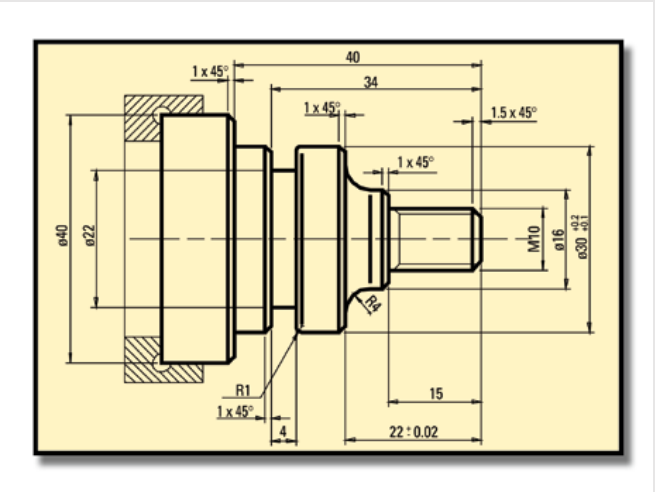
Designation of Machine control type (Turning center system)	S838#T
BACKGROUND EDITING	J956
CANNED CYCLES	J890
CHAMFERING/CORNER R	J875
CONTROLLABLE AXES EXPANSION	J801
CONSTANT SURFACE SPEED CONTROL	J855
CUSTOM MACRO	J873
CUSTOMER SOFTWARE SIZE 512KB	J738#512K
CYLINDRICAL INTERPOLATION	J816
GRAPHIC DISPLAY	J972
iHMI BASIC FUNCTION	**R901**
iHMI Machining Cycle (for 2 path system)	**R912**
MACRO EXECUTOR	J888
PART PROGRAM STORAGE 8MB	J959
SIMULTANEOUSLY CONTROLLED AXES EXPENSION	J803
SPINDLE SERIAL OUTPUT	J850
TOOL GEOMETRY/WEAR COMPENSATION	J931
TOOL RADIUS – TOOL NOSE RADIUS COMPENSATION	J930
MULTIPLE REPETITIVE CYCLES	J877
MULTIPLE REPETITIVE CYCLES II	J889

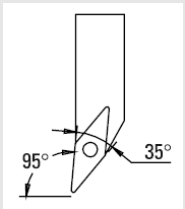
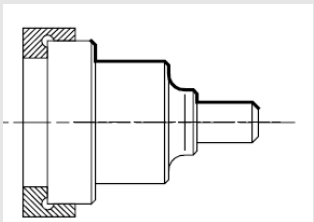
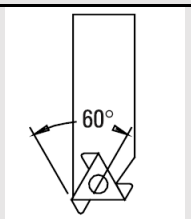
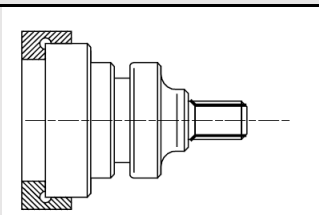
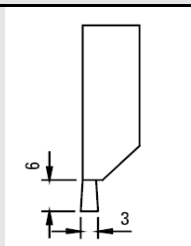
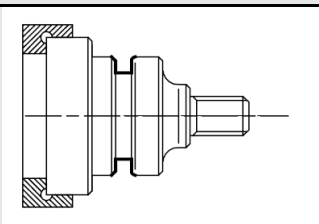
Please find related documents under Chapter 5 – Related documents.

2. Example 1

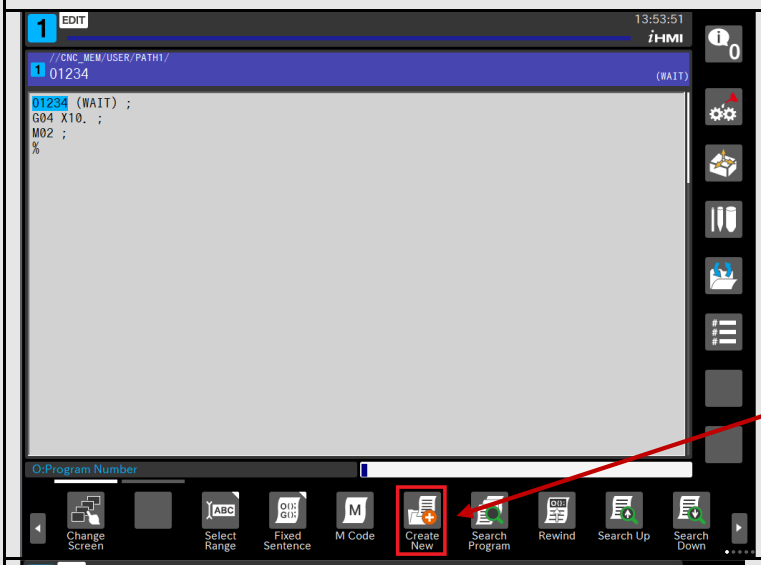
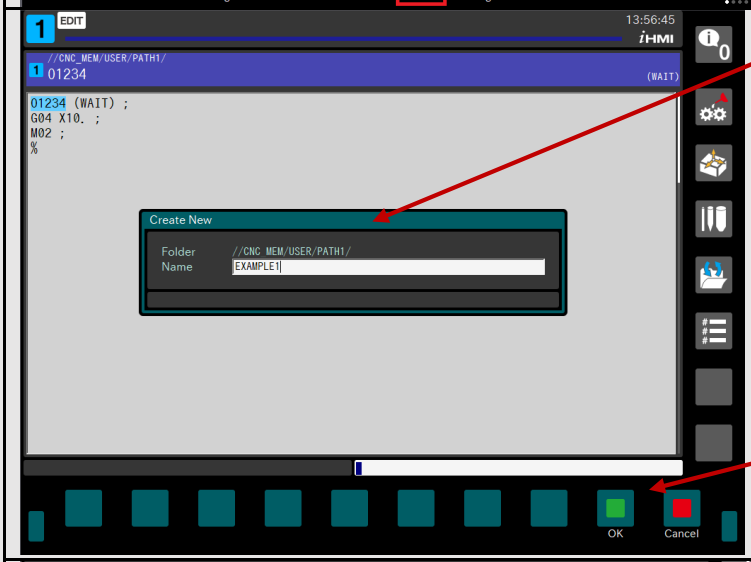
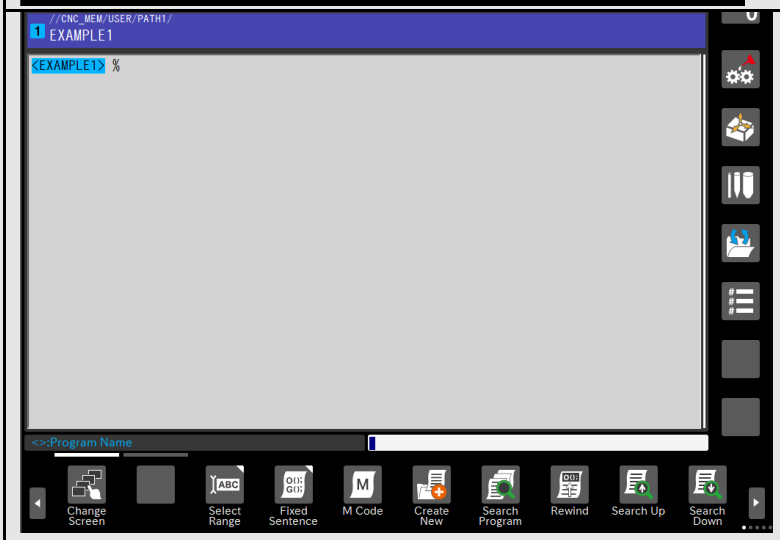
In this example a workpiece, which has an overall rising turning contour and for which all geometric elements are known or have been determined, is programmed with the iHMI Cycle Programming. The following machining cycles are created:

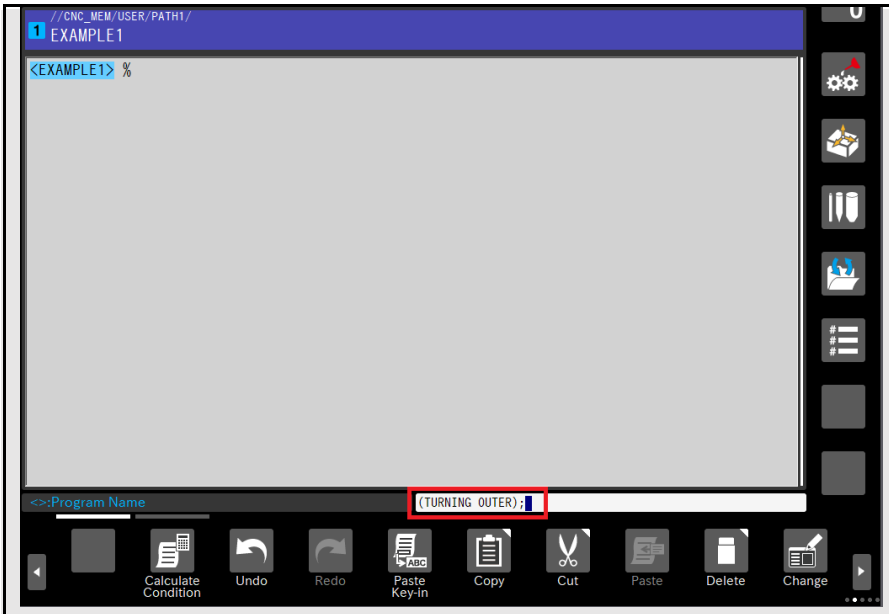
- External turning: roughing and finishing (contour as a subprogram)
- Threading
- Groove



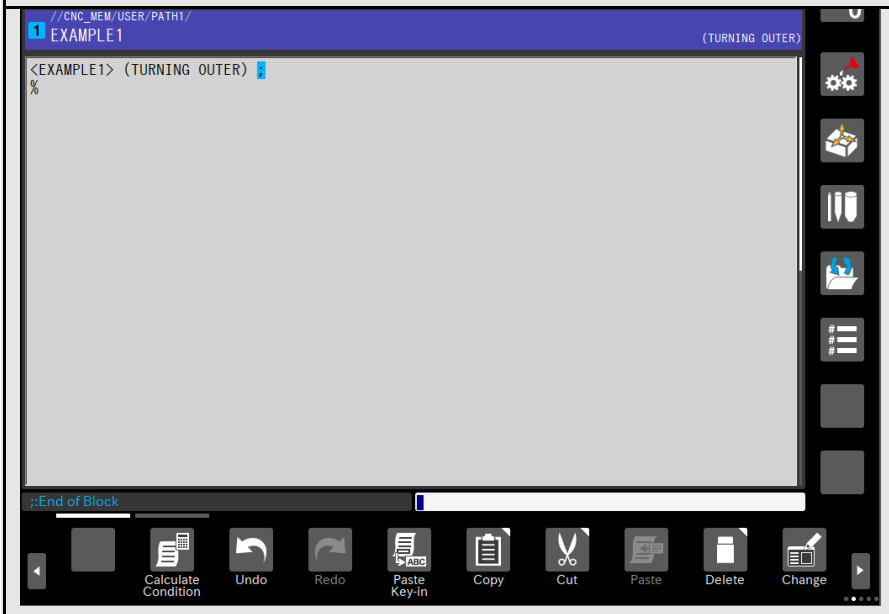
Tool data		Process steps	
T0101		• External turning-roughing • External turning-finishing	
T0202		• Thread cutting M10	
T0303		• Grooving 4 x 4	

2.1 New Program

	Select „EDIT“Mode. In „Home Screen“ select “NC OPERATION”.
	*Click on “Create New”*  *New program* *The new program will be created in the current folder.*
	*Enter the program number (Oxxxx) or program name “EXAMPLE” using the keyboard.* *Confirm the program input by pressing the soft key “OK”.*
	*The new program opens automatically .*

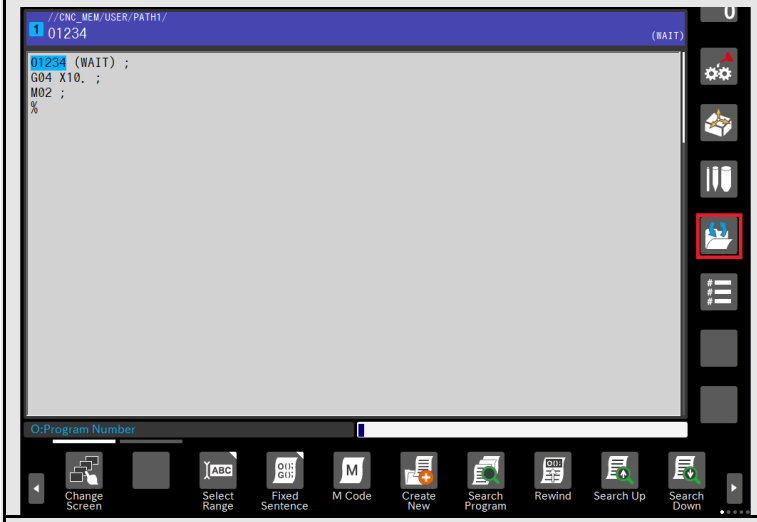


To add comment to the program type "(xxxxxx)" and ";" as End Of Block. Push "Insert" on MDI.

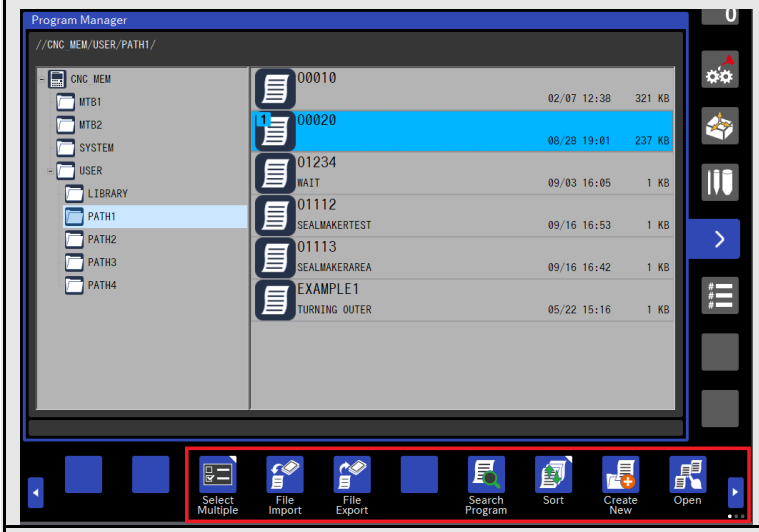


Comment is added.

2.2. Select Program to be edited



Click on Folder icon  *- side bar. You open the Program Manager.*

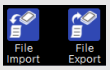


Here you can change folder or select a program just by clicking with the mouse.

With "Open"  *you can open the selected program in Fore Ground for editing.*

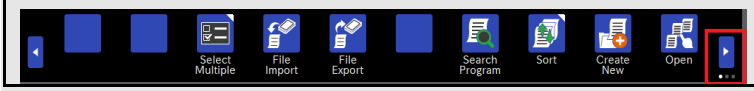
You can select multiple programs for output by "Select..."  .

You can Export or Import files by



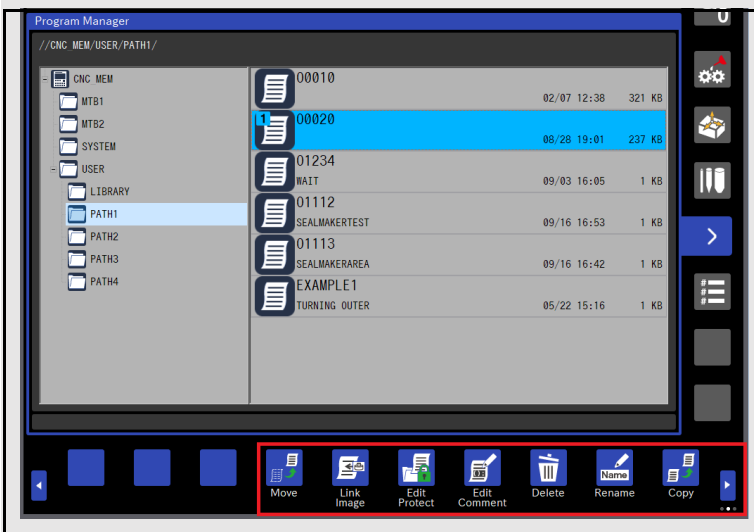
You also can create a new program or folder here.

Click "Create New" 



The horizontal softkey line consists of 3 bars indicated by the points below the arrow.

2.3. Various functions of Program Manager



On the second horizontal bar you can:

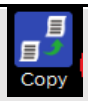
Link an image from a list to each program.

Lock and protect from editing.

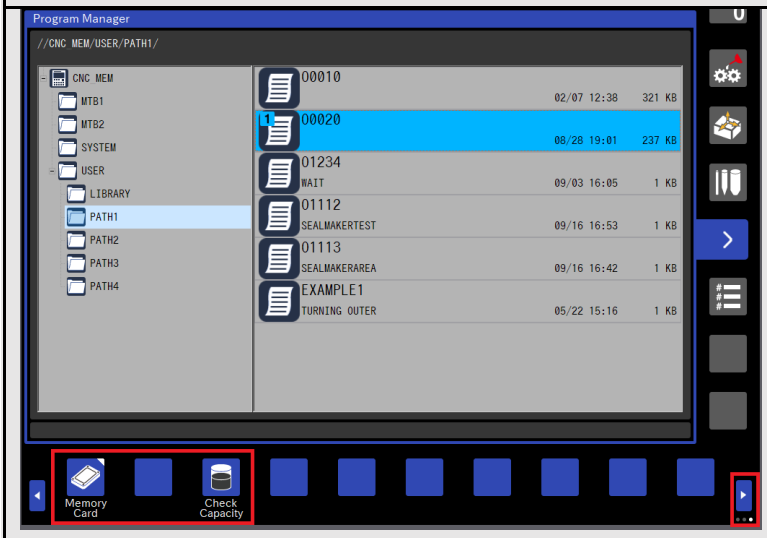
Edit comments.

You can delete one program.

You can rename or copy .

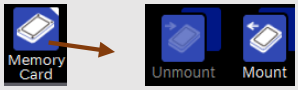


Copy to other folders or peripherals

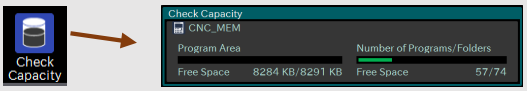


On the 3rd horizontal bar you can:

Mount or Unmount Memory Card with installed Mass Storage.

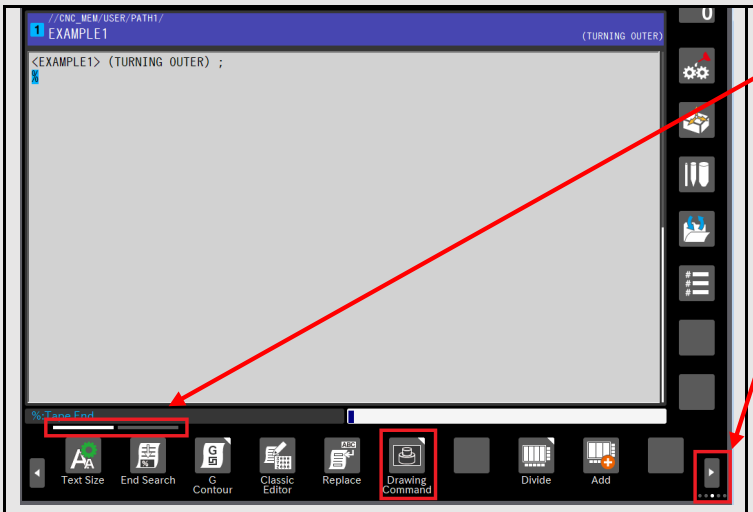


or check memory capacity



If you want to return to Program Edit screen, push the blue arrow.

2.4. Generating Blank Definition

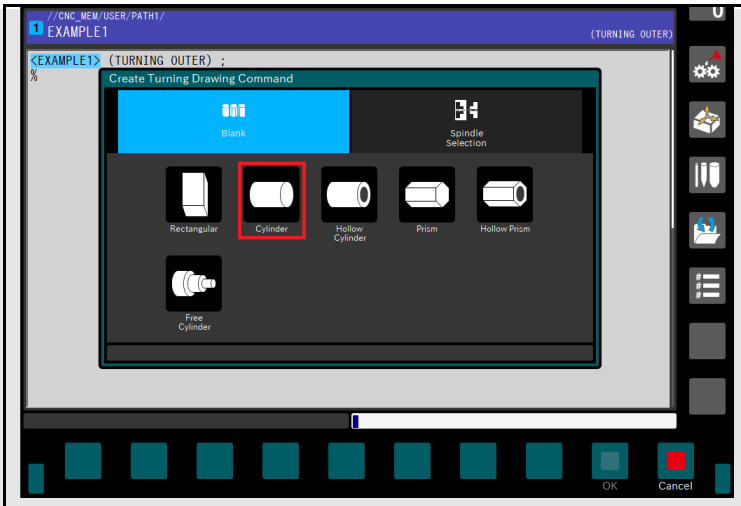


On the first screen – the 2 lines show that there are 2 screens related to this display.

Select a Blank Workpiece form using "Drawing Command".

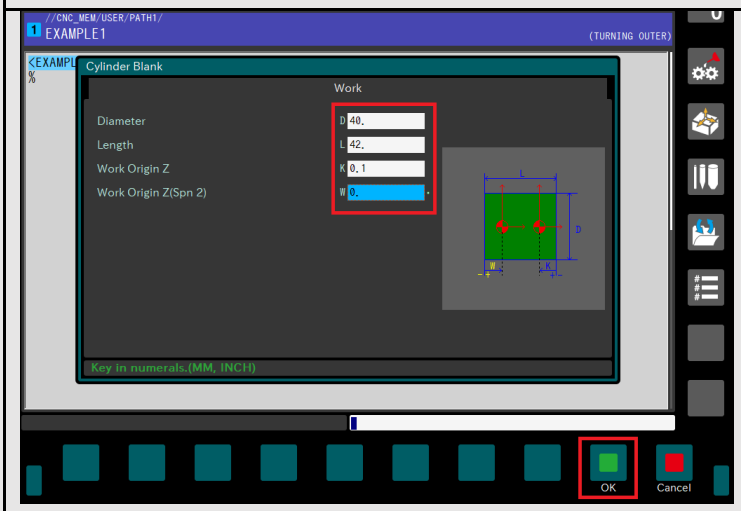
This is the third horizontal bar of soft keys.

If there is coice, select Turning blank figure. If the machine doesn't have milling operations, "Milling" key may not appear.



Choose "Cylinder".

Blank form is mainly necessary for Graphic, but data entered here are used also for determining Coordinate System when describing geometry and for interference in simulation.



Enter the following values on the keyboard (see drawing):

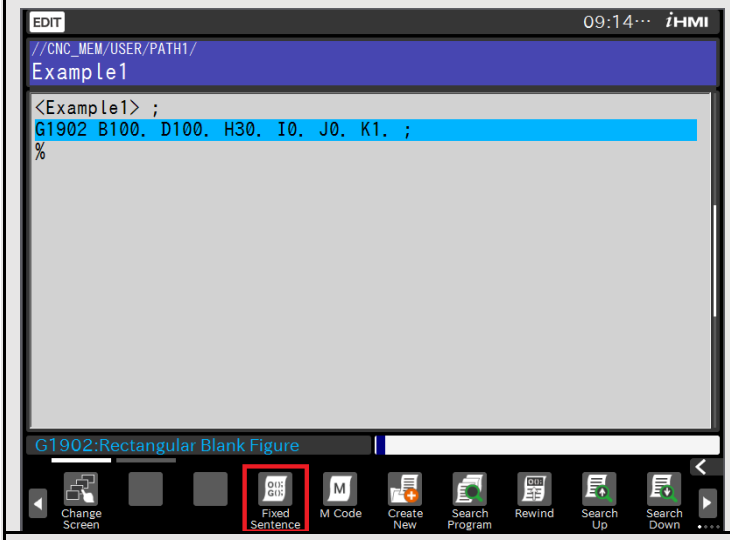
D: 40
L: 42
K: 0.1
W: 0

NOTE:

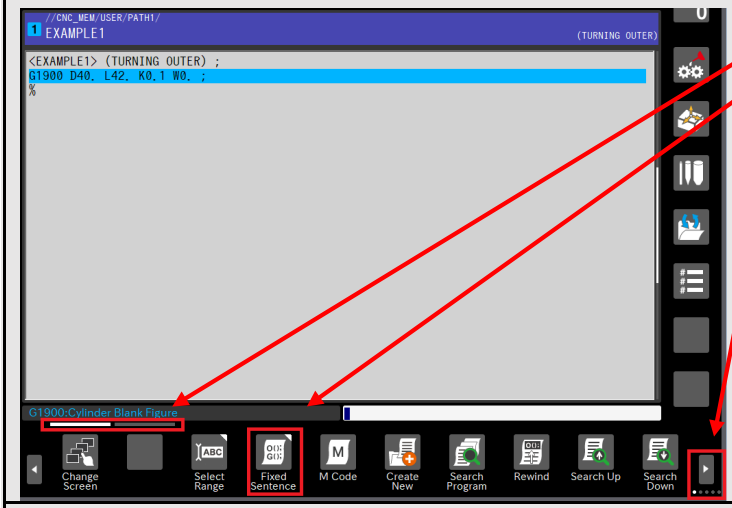
The machine configuration here is for 2 spindles. In the Exercises we use only the main, left one.

Confirm data input by pressing the soft key "OK".

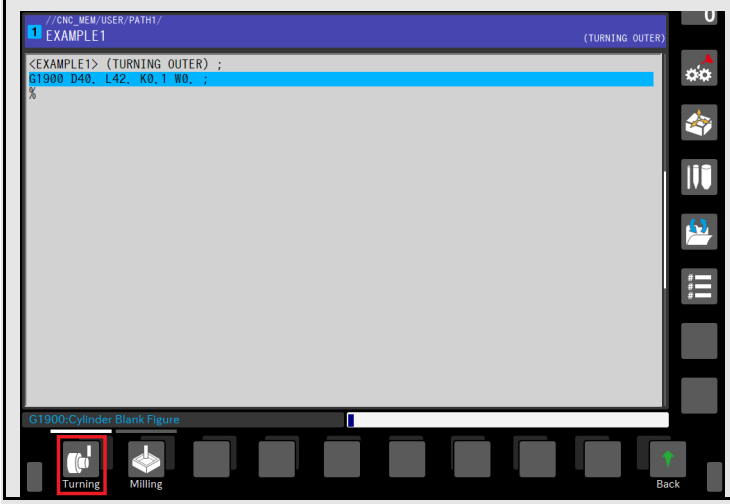
2.5. Fixed Sentences



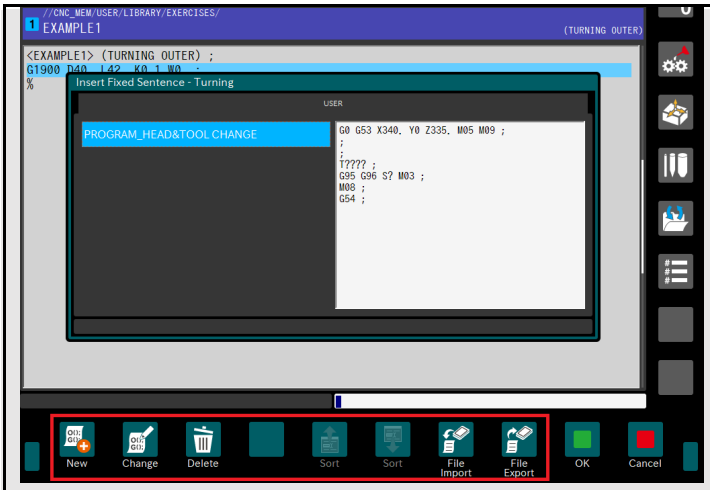
Using "Fixed Sentence", you can insert by 1 click several ISO lines (Group of Commands), prepared by programmer according to machine specific features, into the program.



On the first screen and first horizontal soft key bar, you have the "Fixed Sentences" key.



If there is a choice, select "Fixed Sentences" for turning. Otherwise you have "Basic" where Turning and Milling Fixed Sentences are specified together.



By "New" is possible to prepare new group, defined by name, like: 1. Start Tool, 2. Head of program, 3. End of program, etc.

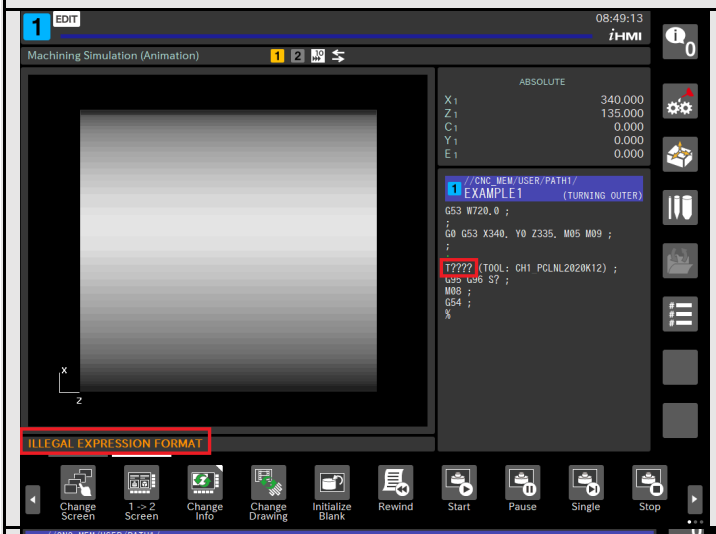
By "Change", you can edit existing group of commands.

You can "Delete" a group.

Question marks (?) are space holder in the program for the real data.

- *Program cannot run with (?) – syntax error*
- *After program is completed, following (?) can be searched/jumped by 1 click.*

Confirm by „OK".

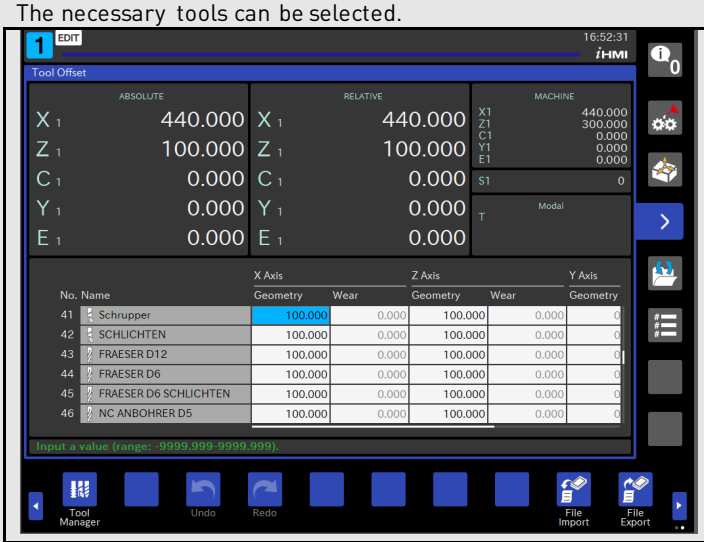


If operator forgets to change question mark to respective data, simulation and execution in MEM mode will stop after Syntax check.

Using Tool Manager screen already prepared tools can be selected.

2.6. Necessary Tools

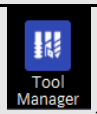
The necessary tools can be selected.



In Tool Offset screen, tools which are already defined (set up), can be used/called in the program by Tool Number (T0101; selects Tool Number 1, Offset Number 1). Tool Offset screen shows Type and Offset. If the necessary tool is missing, generate it.

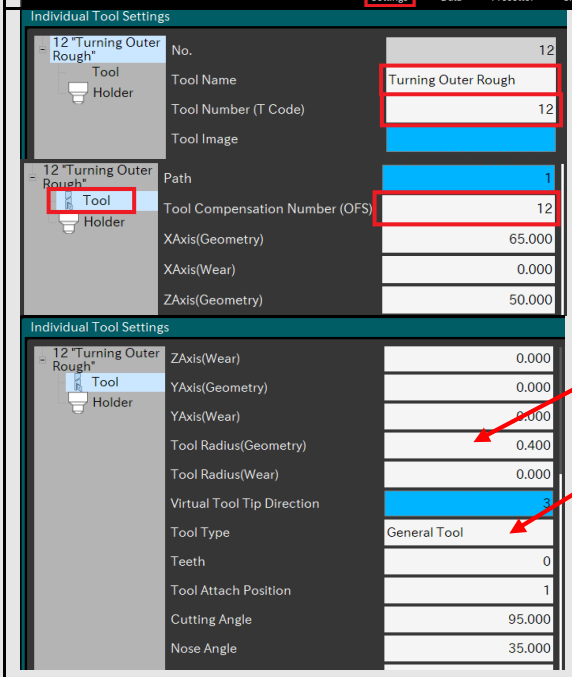
T0101, T0202, T0303 could have been configured to be used with other programs. In this Example1 we set up the necessary tools in free positions – T0101 from drawing to T1212, T0202 to T1313, T0303 to T1414.

Generation of New tools is done in Tool Manager.

Tool Manager - 

gives the possibility to set up New Tools.

To generate a New Tool, change to "Edit" mode, go by cursor to free number and select "Individual Setting".

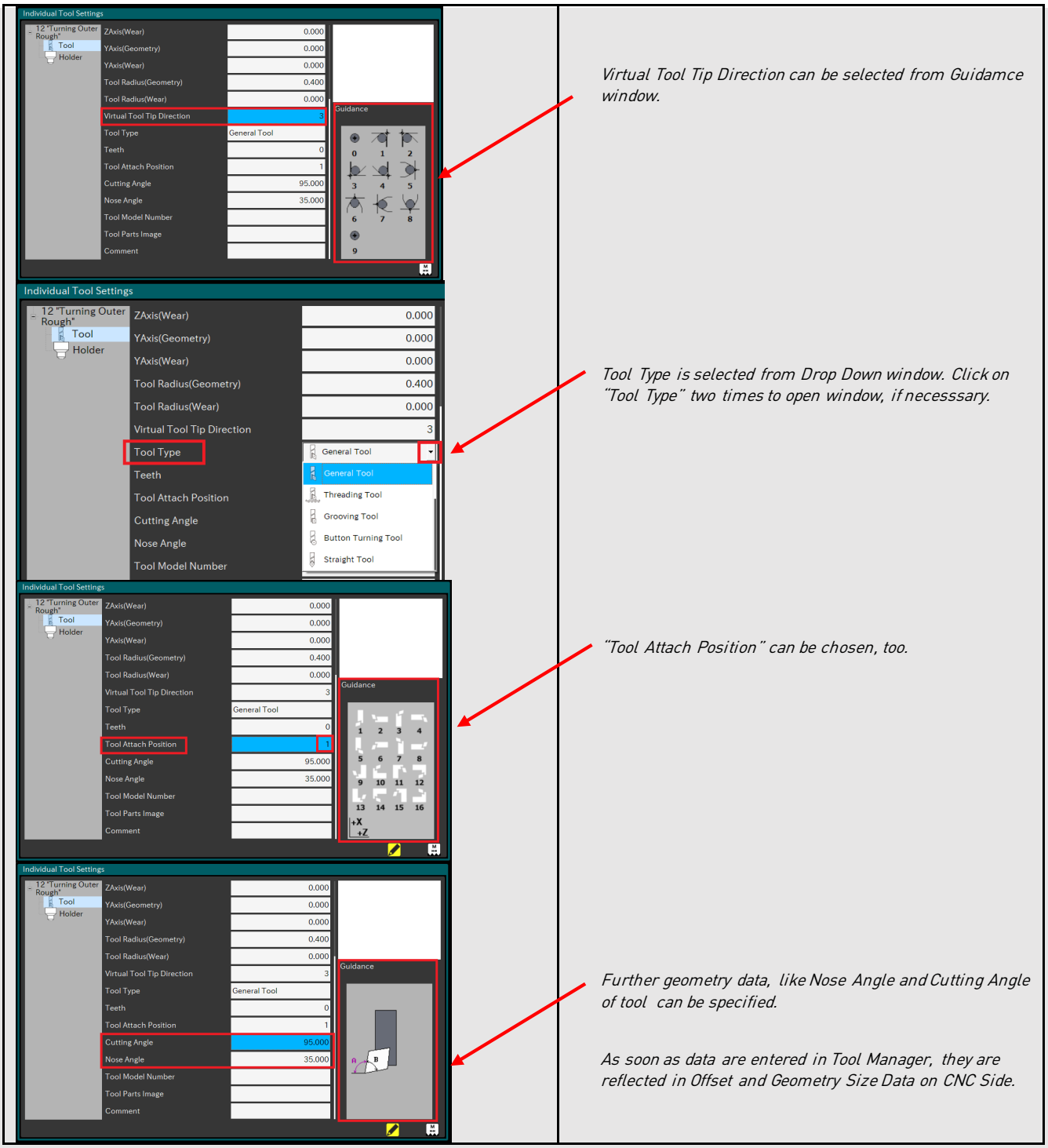


With cursor on Tool Manager list select number "12" you can select the "T-Code" number. Input data for name in "Tool Name" it is reflected on the left side of screen.

With cursor on "Tool" and setting the "Tool Compensation Number", you can define the tool geometry and offsets.

Set "Z/X/Y Axis (Geometry and Wear)" and "Tool Radius (Geometry and Wear)"

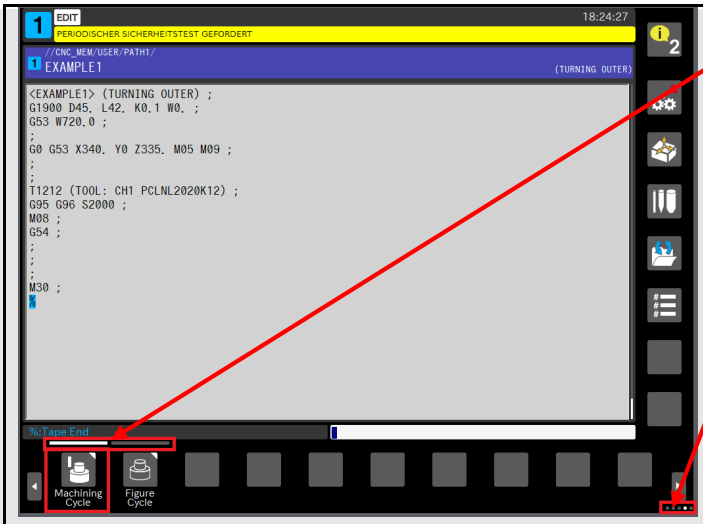
Select "Tool Type" from Drop Down list. It takes a moment the list to appear.



Exercise: This was an example for setting up T1 from the drawing above as T12 in the program. Please complete setting up all other necessary tools (T2 and T3, as described on drawing Page 4 above) respectively as T1313 and T1414.

2.7. Cycle selection

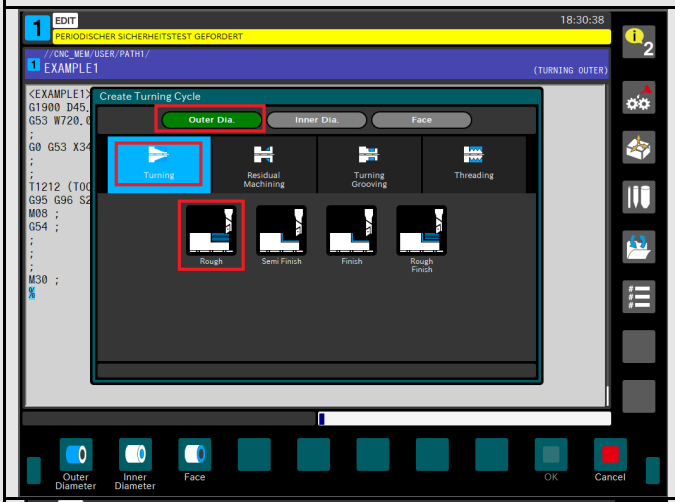
Using „NC Operation Programming "the process is generally divided in 2 steps – first choose and set up machining cycle, then – respective geometry. The geometry, belonging to the cycle appears automatically after inputting the cycle into the program.



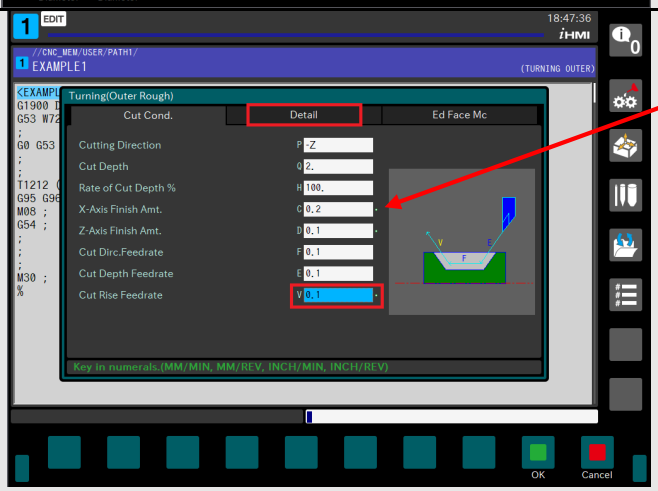
"Machining Cycle" soft key appears under "EDIT" mode:

1. On the Edit screen (1st screen group)
2. And the 4th horizontal bar.

Select "Turning".



Select "Outer Dia.", "Turning" and "Rough". Confirm by "OK"



Note1:

*The values with * are not obligatory*

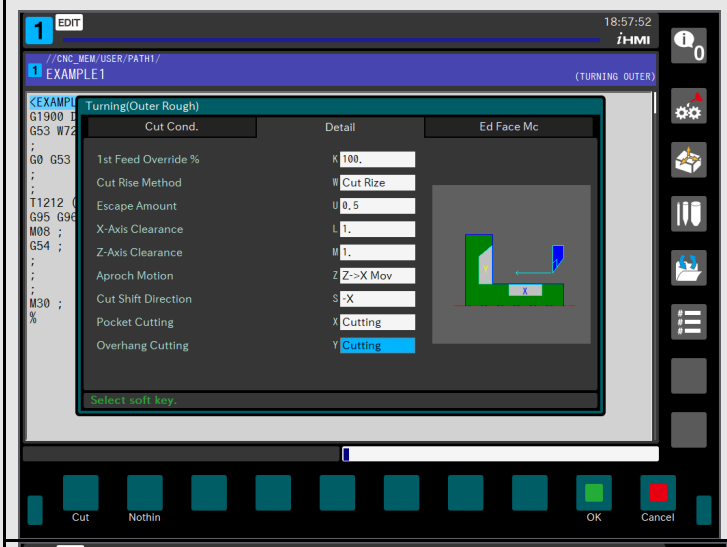
Complete parameter

Pushing "Input" on the keyboard in the last line, the screen automatically changes to "Detail" tab.

You can also, click on "Detail" or use keyboard arrow – right.

Note2:

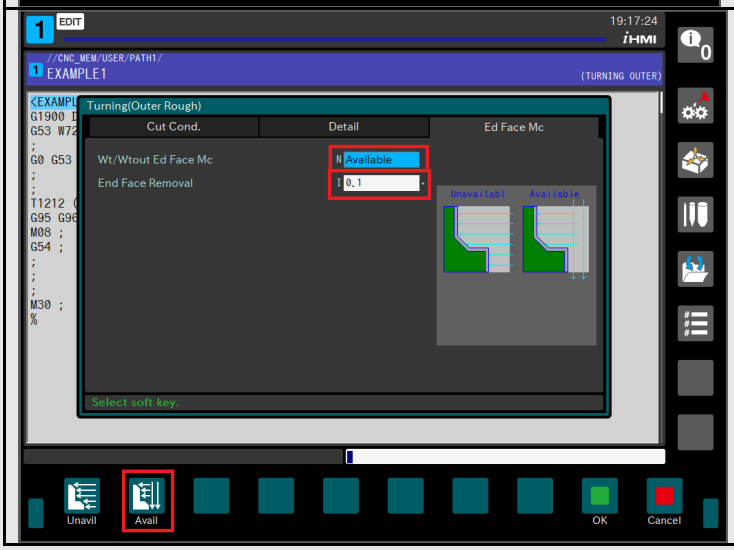
The feed rate for turning is mm/rev per default. For mm/min, enter in the ISO the respective comand in advance.



Complete data

To use pocket cutting "multiple repetitive canned cycle II" – J889 optional function is required.

To come to tab "Ed Face Mc" push "Input" in the last line or select tab by arrow from keyboard.



Make End Face Machining with the same roughing tool available.

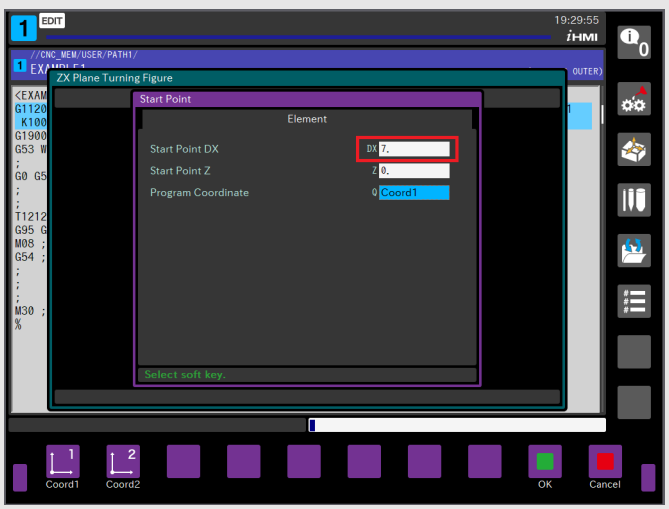
"End Face Removal" should be the data "Work Origin Z – K" allowance from page 9 – distance from rough part front to program zero point of workpiece. In this case K=0.1 mm.

Data entry is not obligatory (star) the removal value can be calculated automatically.

If completed, push "OK" to confirm.

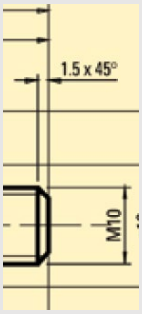
2.8. Geometry

Describe workpiece contour
Confirming the Cycle by "OK", it is inserted into the program and respective contouring definition screen opens automatically.

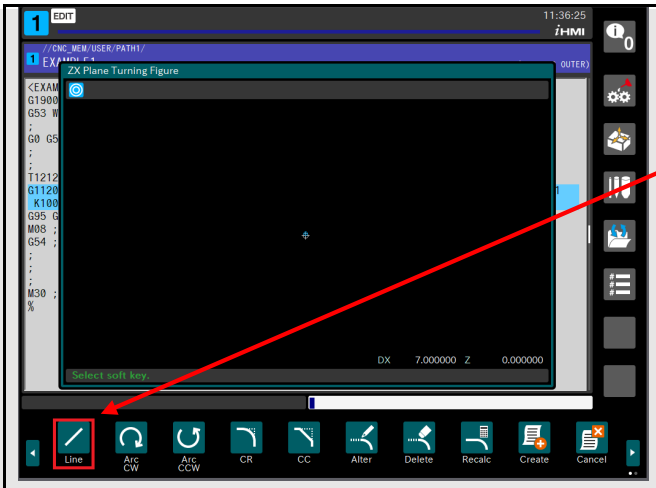


For turning outer roughin process "ZX" plane is selected.

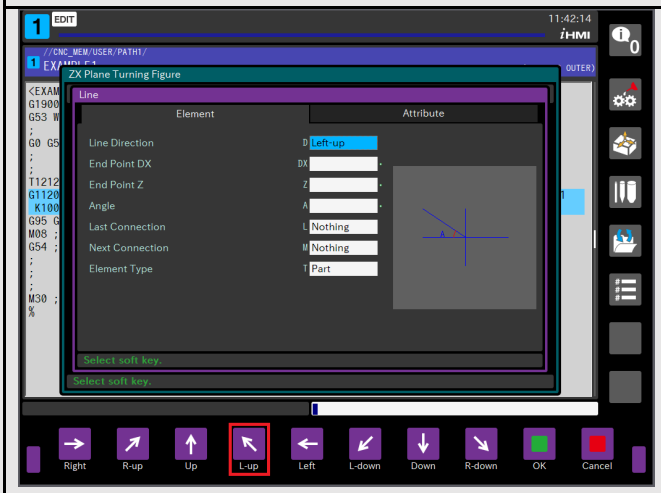
Start point DX7. considers chamfer of 1.5 mm on M10 size. (10mm – 1.5 mm – 1.5 mm = 7 mm. for the starting point)



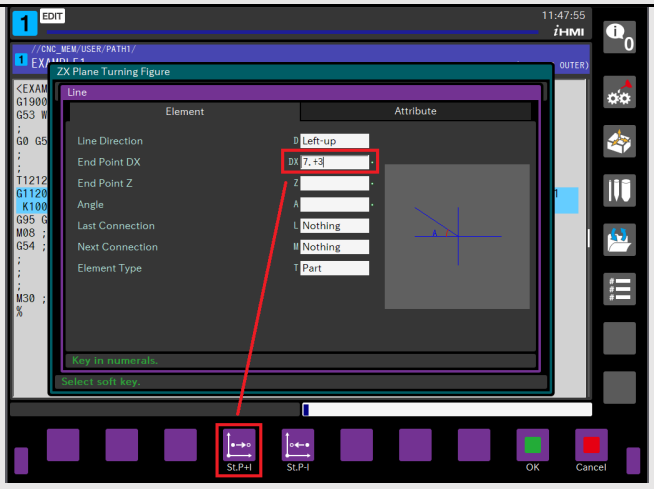
Confirm by "OK".



Select "Line" element.

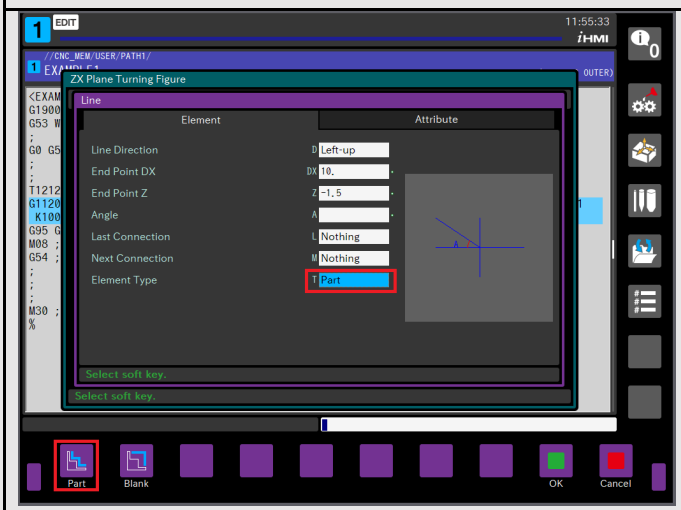


Select direction "L-up". Click "Input".



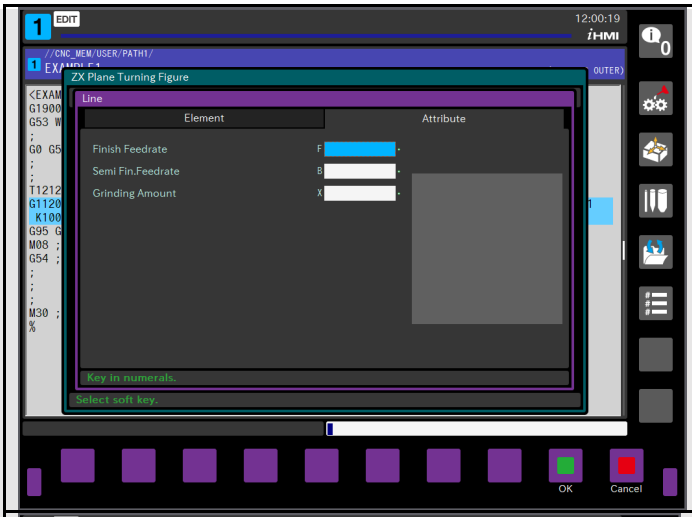
Diameter of End Point X and End Point Z can be entered in absolute or incremental, using the current position and incremental increase/decrease. Push "Input".

In CNC Guide this function is not yet applied.

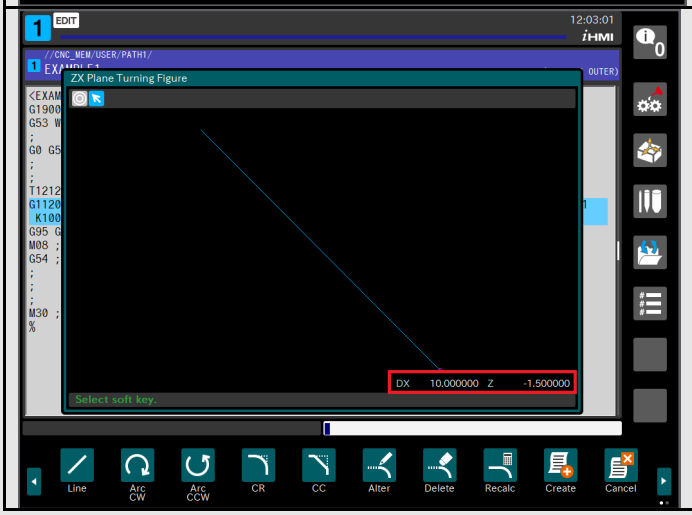


Selecting "Part" means that the element belongs to finished workpiece contour and not to blank.

Pushing "Input" the tab switches to "Attribute".

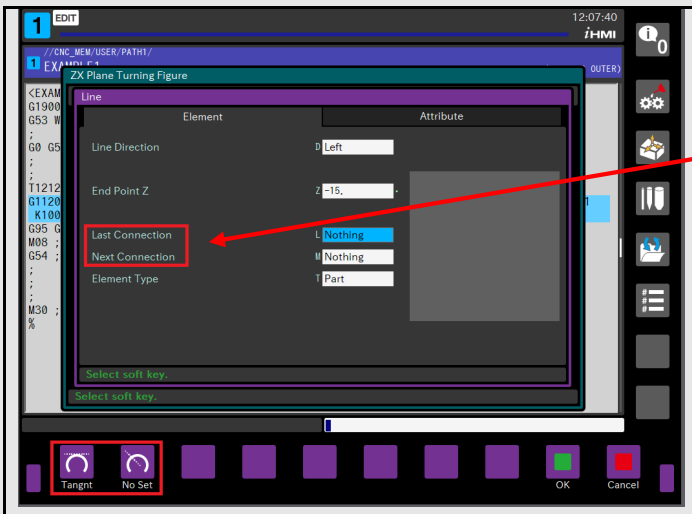


Here you can enter optional data (not obligatory – (*)).
By pushing "OK", the element appears on the screen.



In the marked field you can see the current position of the end point of element.

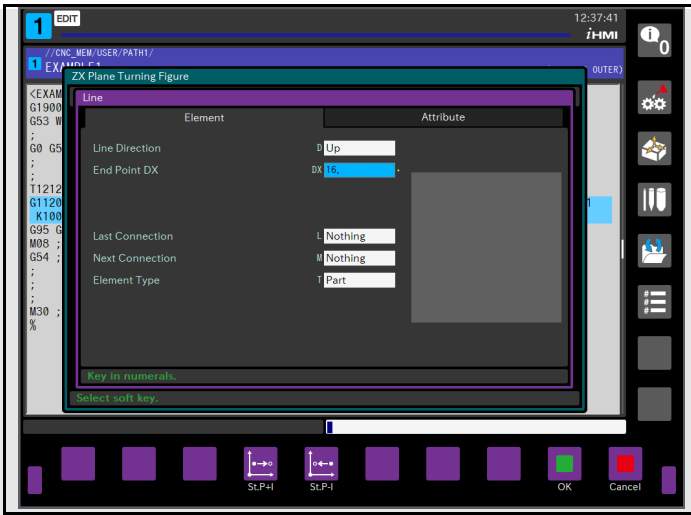
Next element is horizontal line in direction –Z (left).



Complete parameter.

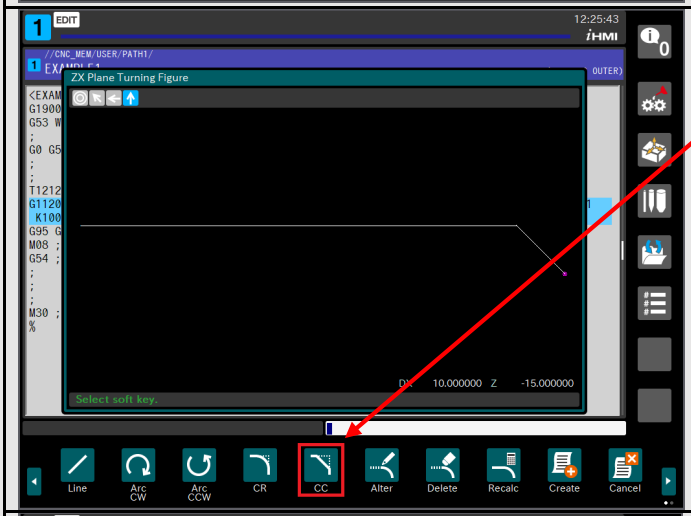
Using "Last Connection" "Next Connection" you can define Tangent or Cutting point to Last/Next element. The point would be calculated automatically, if the control has enough data.

Here "Attribute" tab consists of the same data as above. Confirm by "OK".

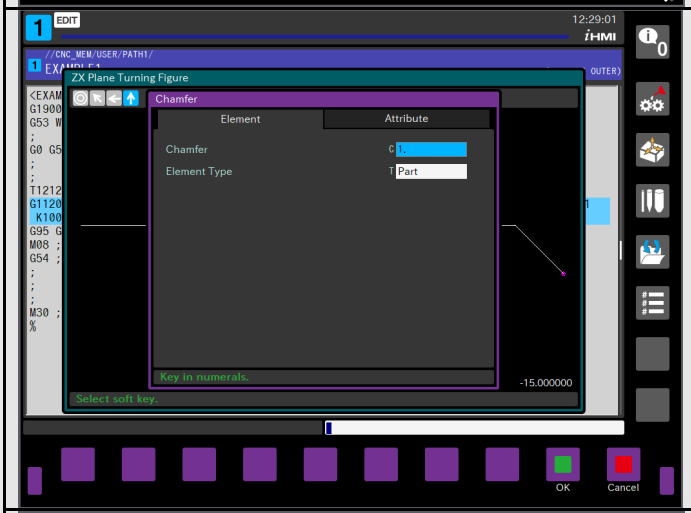


Next element is "Line Up" to diameter 16 mm.

Confirm by "OK".

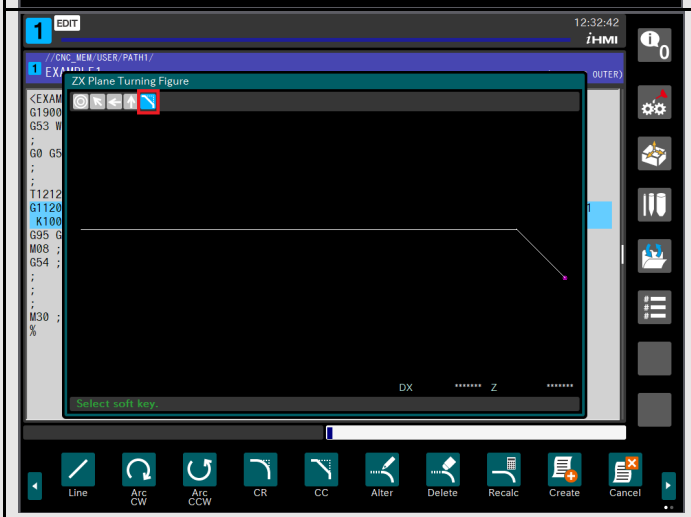


Select "CC" – corner chamfer.

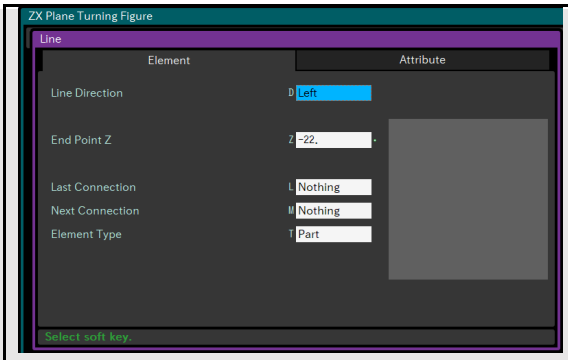


Chamfer is 1 mm.

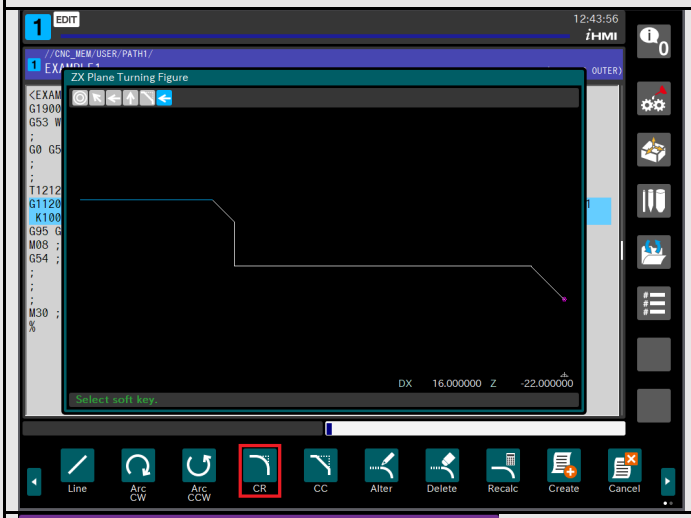
Confirm by "OK"



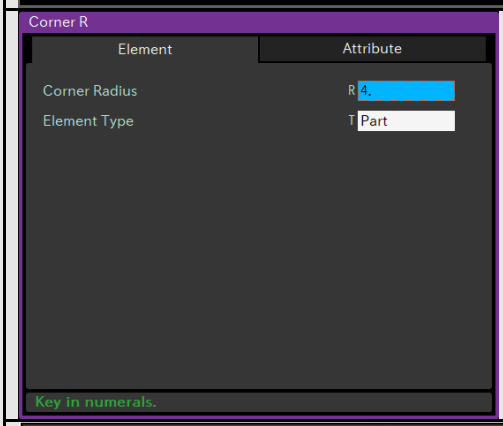
Control will wait for the next element to fit the chamfer between Last and Next.



Next element is horizontal line - "Left" to Z-22 mm.



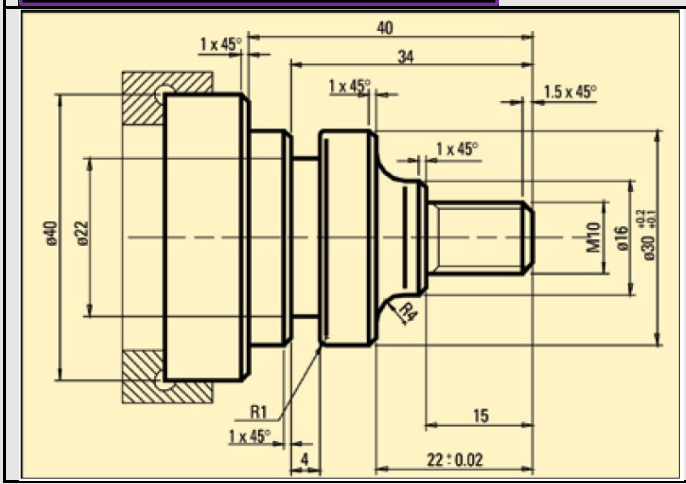
Select "CR" – Corner Radius.



Corner radius is 4. mm.

"Atributes" are not obligatory.

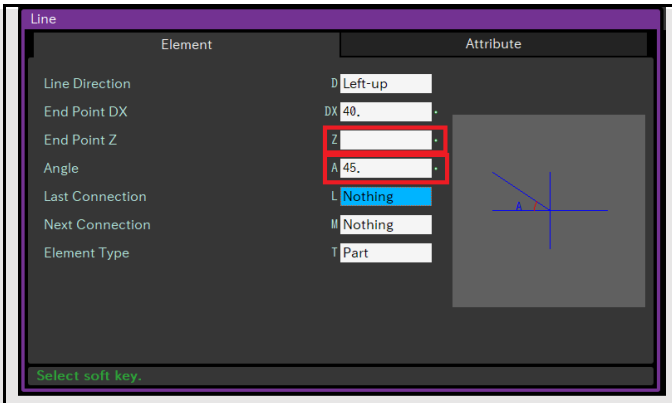
Confirm by "OK".



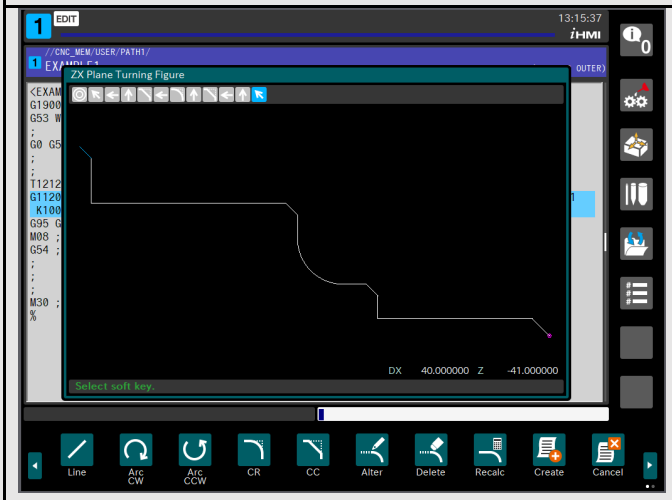
Exercise:
Please complete the contour according to the drawing on the left side.
Endpoint after chamfer – DX40, Z-41.

Leave Outer Grooving without geometrical description.

The end Chamfer should be defined as a line "Left Up" by 45 deg. without Z dimension.
Caution: do not over define elements. This may confuse control calculations.

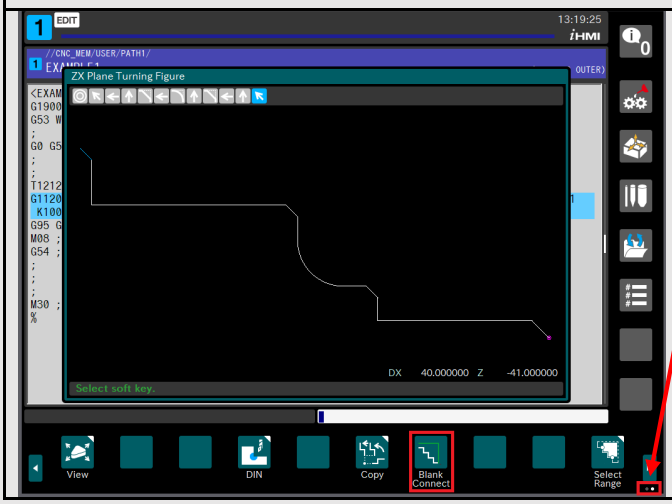


Complete the parameter for the last element.

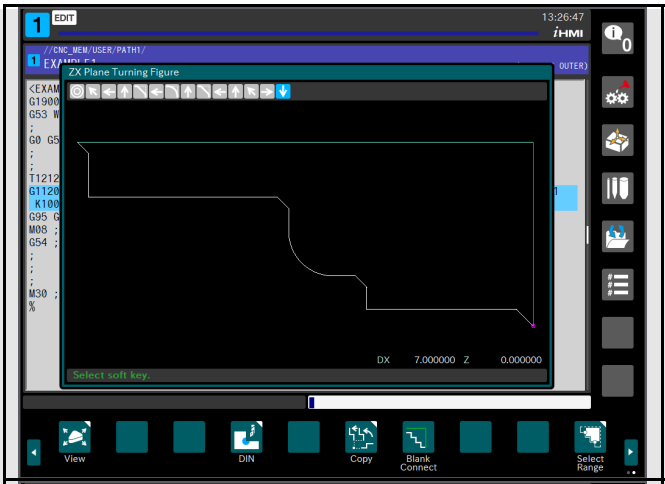


This is how the finished Workpiece contour should look like.

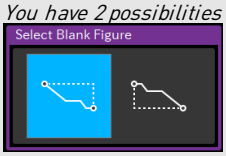
Now the description should be completeted by closing with Blank definition.



Go to the second horizontal key bar and select "Blank Connect" soft key.



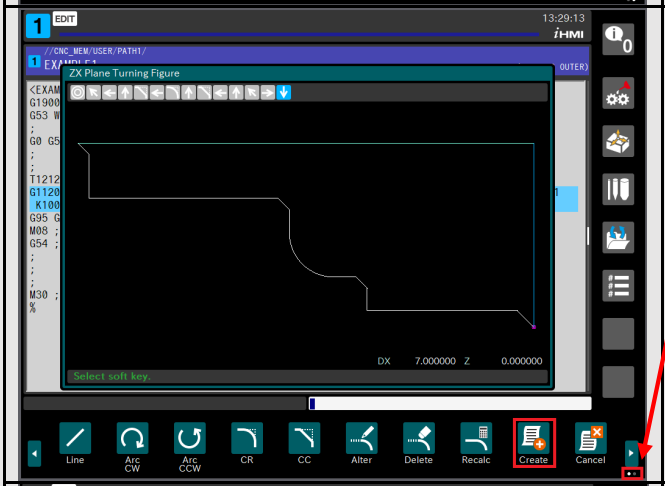
You have 2 possibilities to define the blank contour:



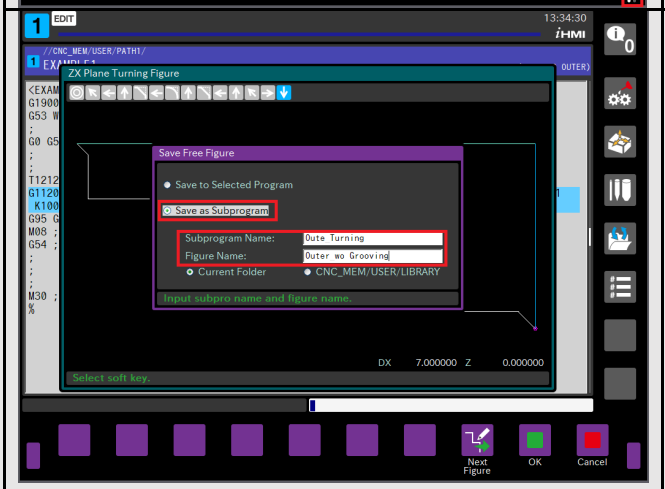
Select the marked.

Confirm by "OK".

The contour is automatically closed. The "Blank" contour elements are shown in different colours (Green).



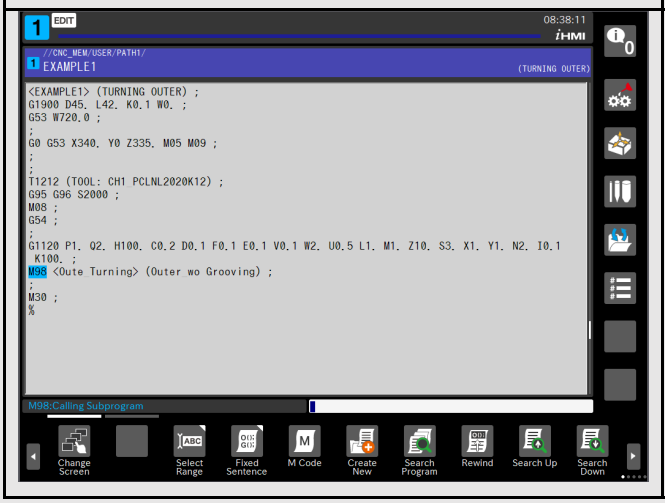
Go back ti the first horizontal key line and select "Create".



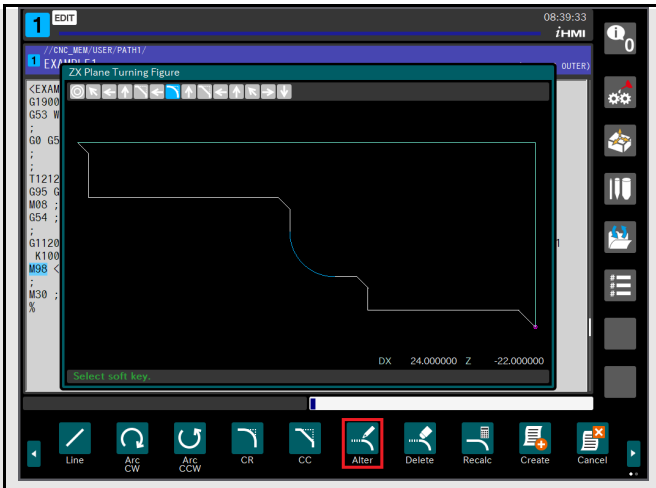
You can save the contour program as a sparate Subprogram, which can be used later.

Under "Subprogram Name" and "Figure Name" you can enter identifying comments for the prepared Part Contour.

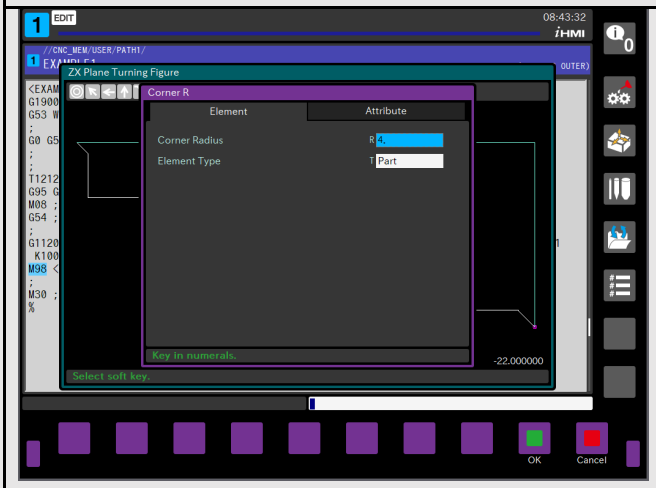
Confirm by "OK".



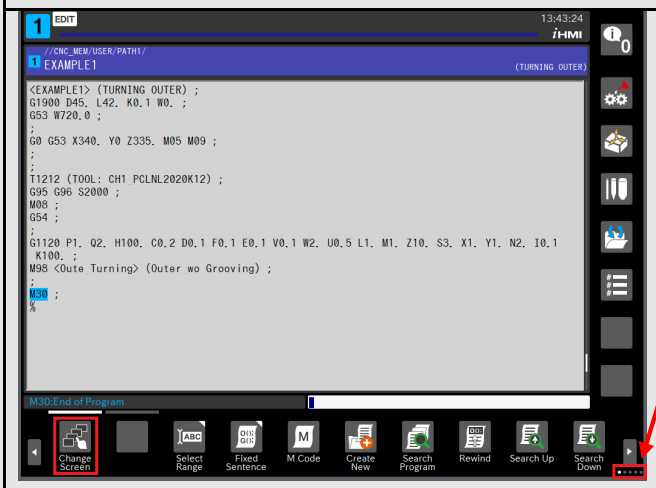
You can edit the contour going by cursor to M98 and pushing "Input".



Go by Left/Right arrow key to the element you want to modify and push "Alter" soft key.

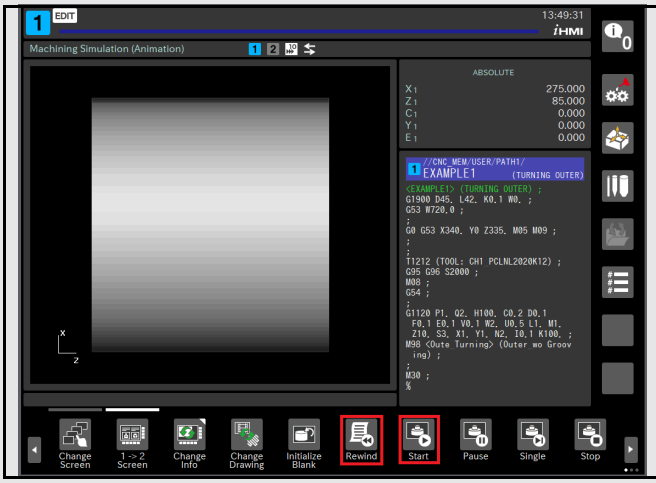


The input screen for the element opens and you can modify the data for the element.



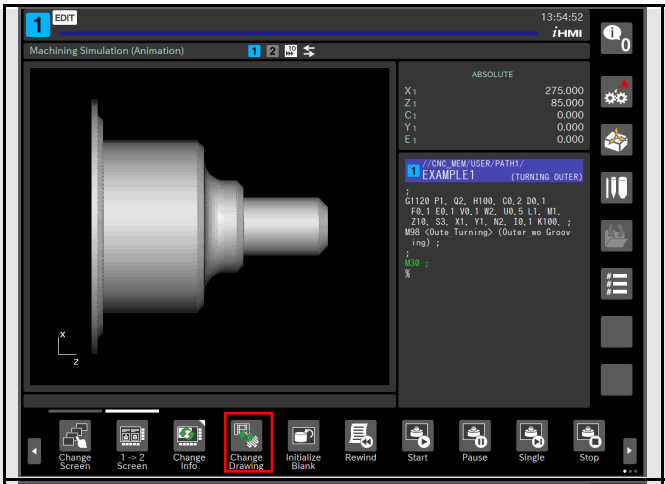
Setting M30 after the first created process you can check/simulate and, if necessary, edit the program on this stage.

On the first horizontal softkey bar push "Change Screen".



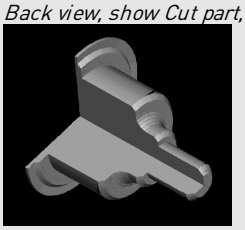
Push "Rewind" to get the cursor to the first line of program and then – "Start".

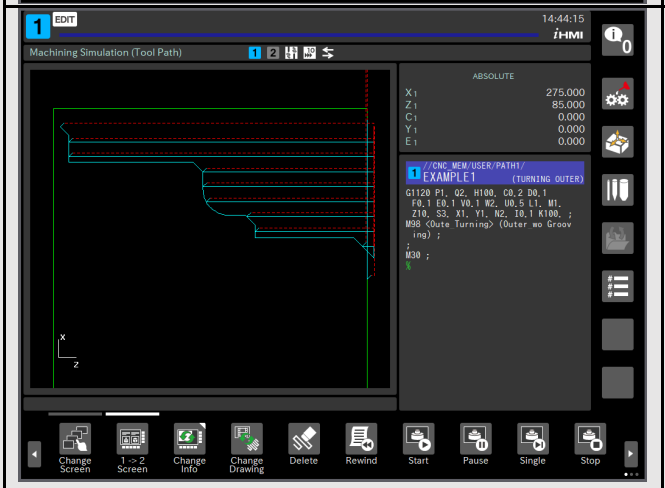
The program is executed, 3D simulation is shown.



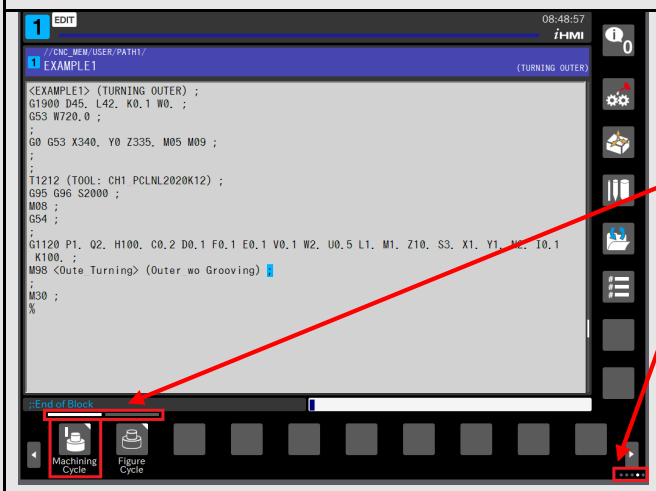
You can change the 3D view on the second horizontal key bar.

You can Zoom In/Out, Move, Rotate with fixed coord. system and freely rotating around coord. axes, change by one key from Front to Back view, show Cut part, etc.





Using the soft key "Change Drawing" you can simulate the tool path, checking every movement of the tool.



Next process – Finishing Outer contour.

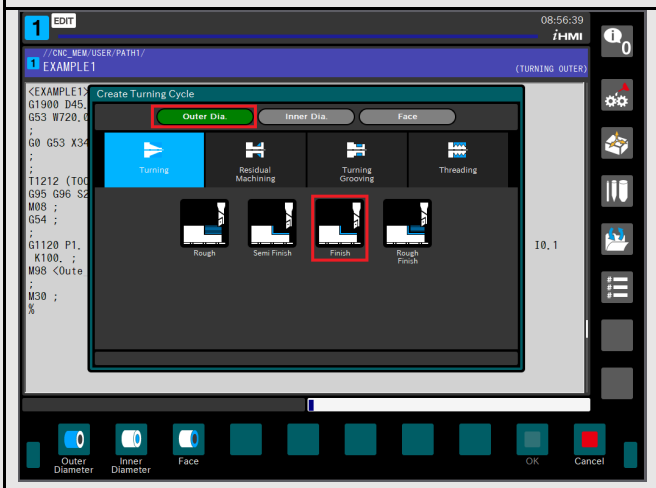
Go by cursor to the end of geometry definingtion for Outer Roughing proces.

Go to the first Screen (out of simulation screen).

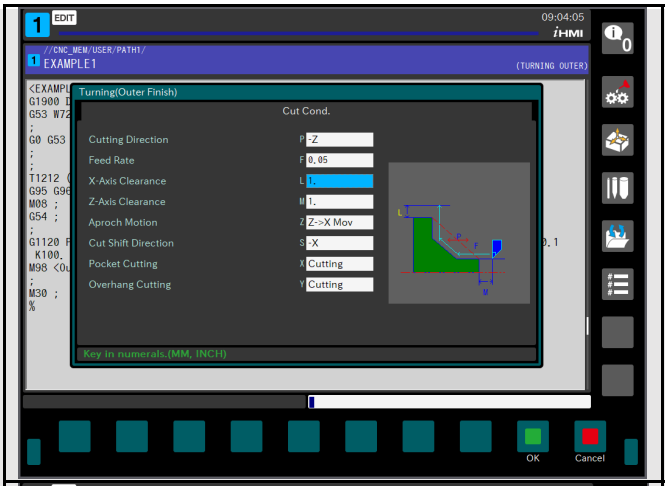
Click softkey arrow to come to the 4th horizontal bar.

Select "Machining Cycle".

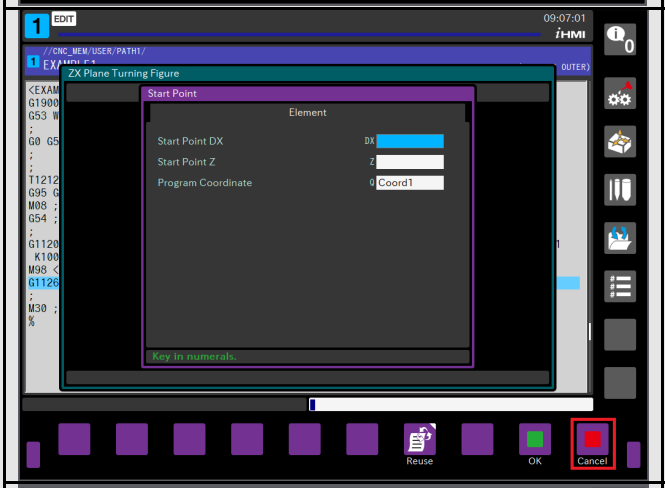
We will use the same tool, so no toolchange is necessary.



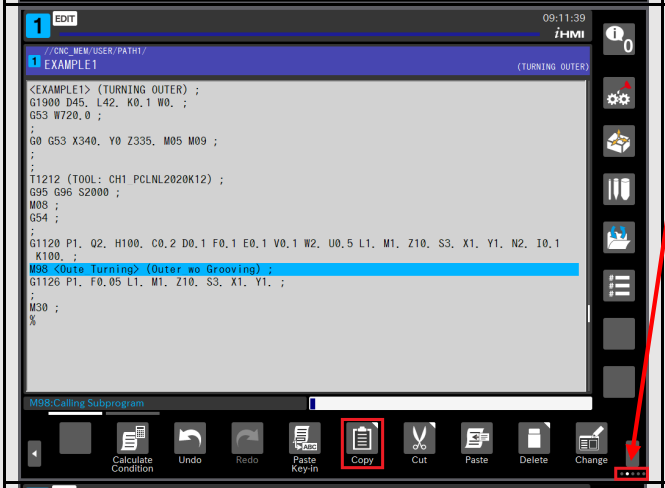
Select on "Outer Dia." - "Finish".



Set up as shown at the left. Confirm by "OK".



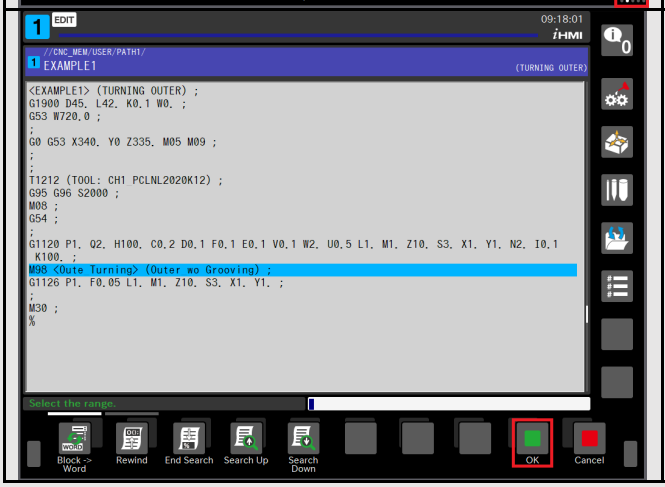
After process is defined, the geometry (part contour) description opens.
We will use the already prepared contour. Select "Cancel".



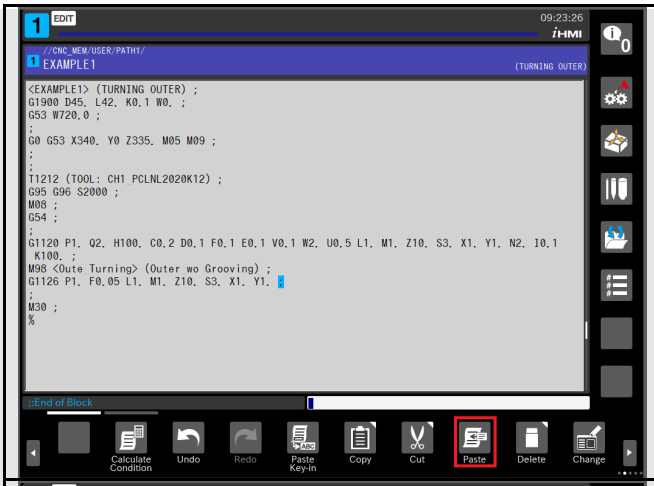
Go by cursor to the subrogram for the contour, using left/right arrow mark the line as shown.

On the second horizontal soft key bar push "Copy".

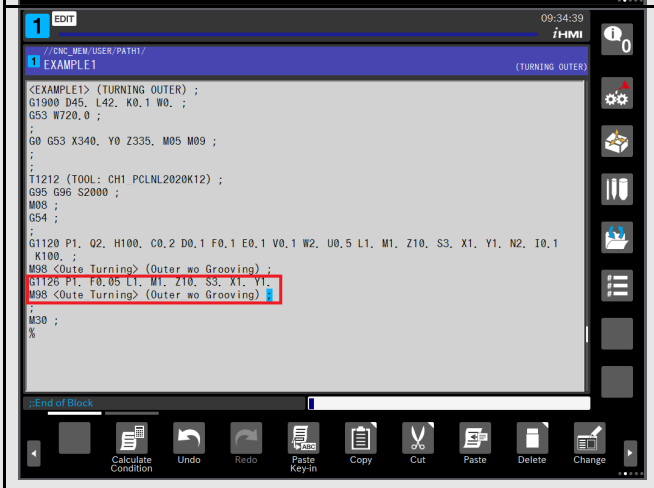
If you move the cursor up/down, you can select a range of lines to be copied.



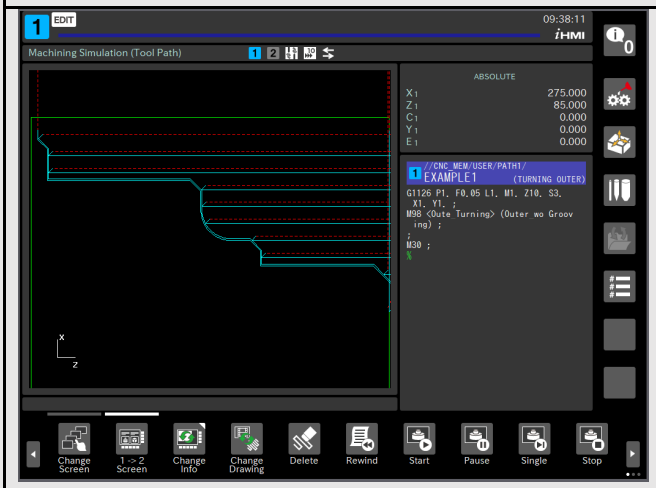
A new screen appears. Confirm your choice by "OK".



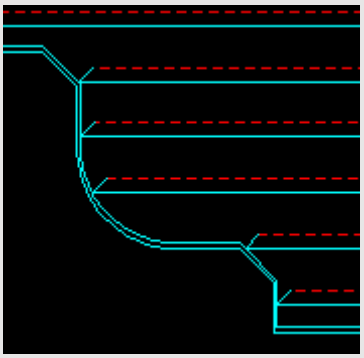
Go by cursor to the end of Finishing Cycle and push "Paste".

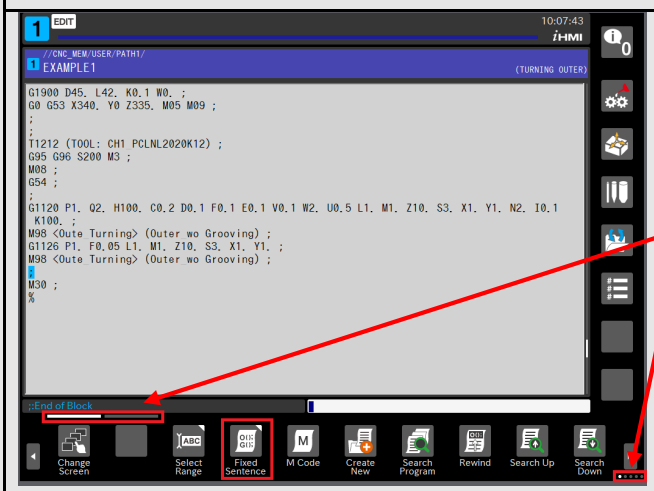


Now Finishing Cycle is completed and you can test - simulate.



Finishing tool path appears.





Next proces is Grooving.

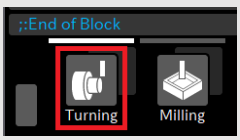
We start with moving to Index position and Tool cange using "Fixed Sentences"

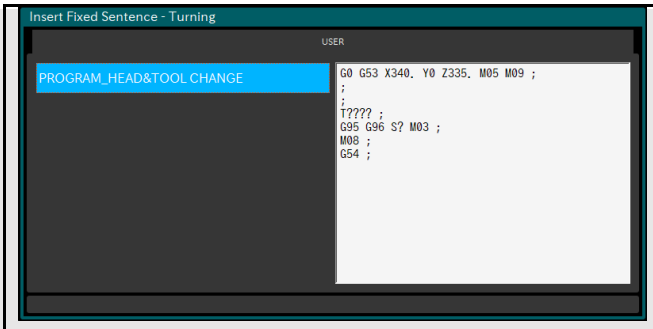
Put the cursor to the right place.

Go to first screen

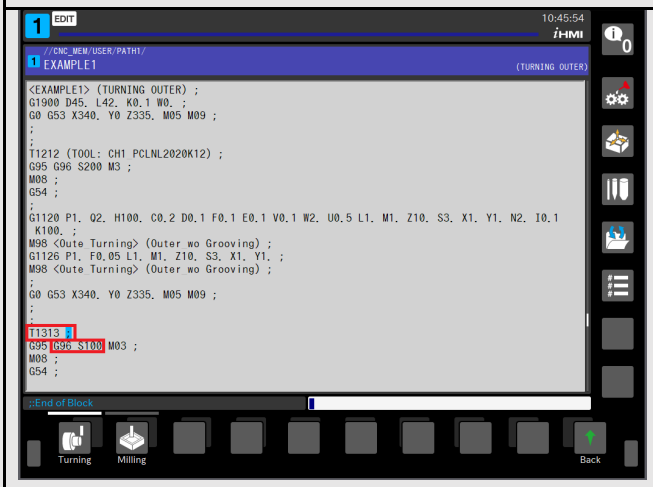
and first horizontal softkey bar.

Select "Fixed Sentence". Select "Turning".



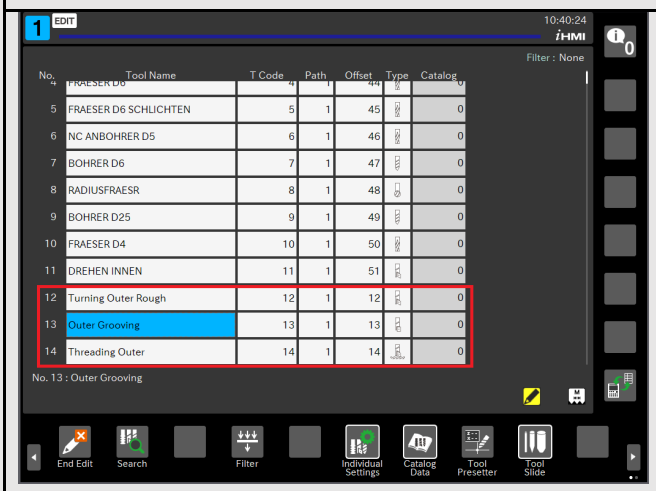


Push "OK" to transfer blocks to program.

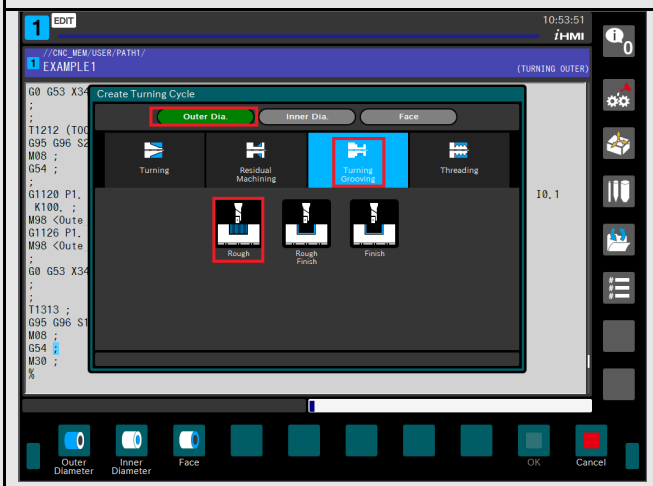


Modify the program for Tool number and Spindle speed in Constant Surface speed. Amend the question marks.

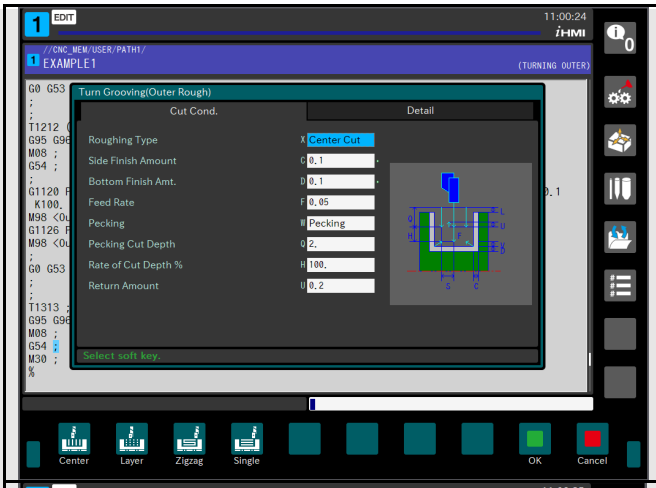
G96S100 means 100 m/min tangential cutting speed between tool tip and workpiece. It G96 is used mainly for Finishing of front face and deeper side grovings. It is not recmmendable to be used in rough cutting (it coul cause overload of spindle if the time for a cut is short) and for thread cutting (the inertia of spindle could cause thread damage).



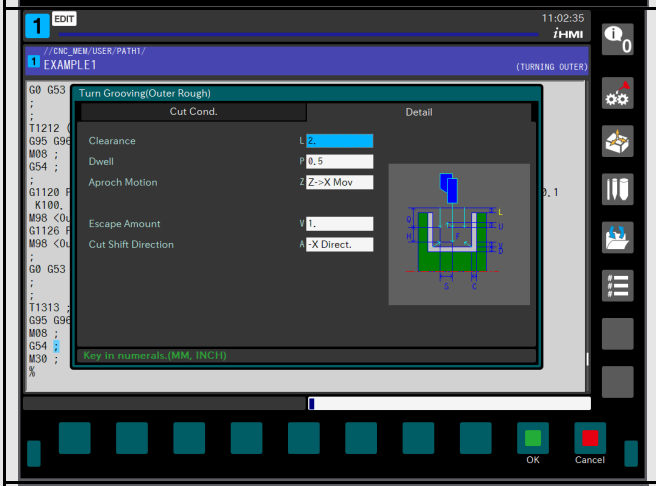
Select the respective Grooving tool, which you set up under the exercise above on page 13.



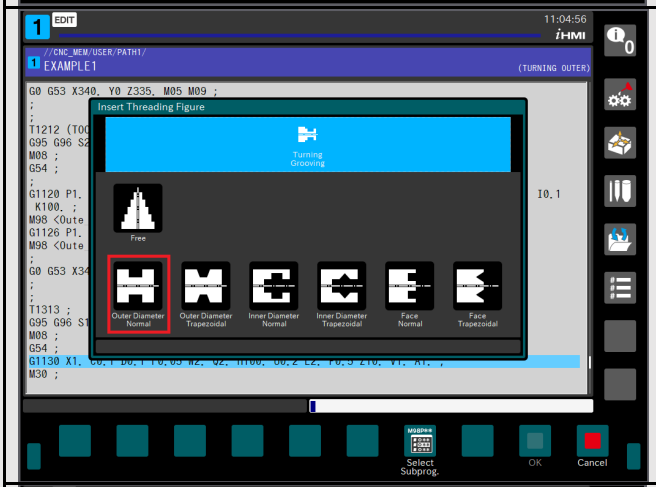
Select Turning cycle "Outer Dia.", "Turning Grooving", "Rough".



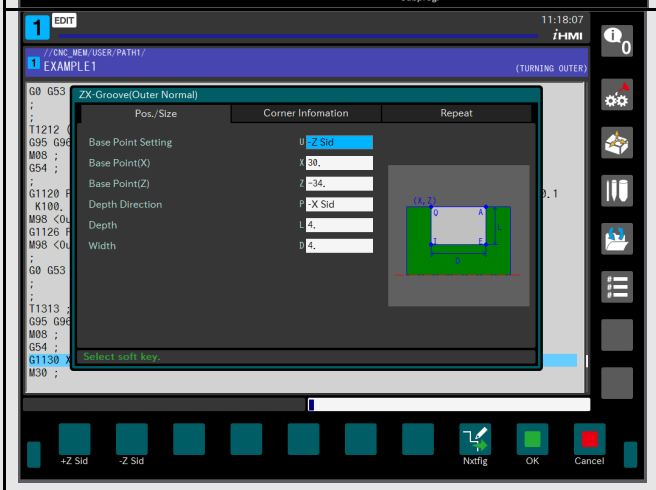
Complete parameters as shown. Do not forget "Detail" tab.



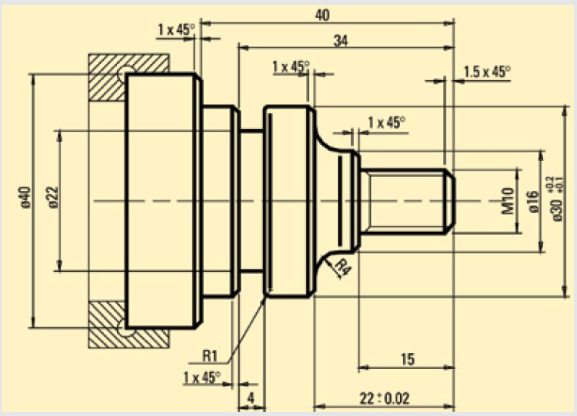
All data are obligatory. Confirm by "OK".

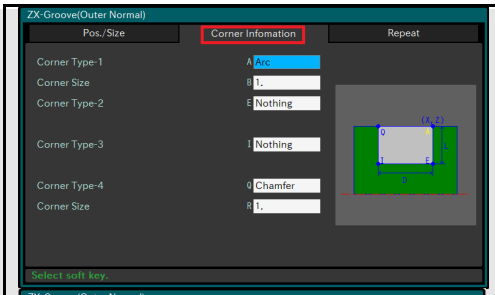


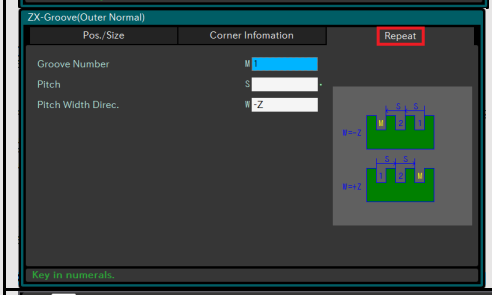
Geometry screen appears. Select "Outer Diameter Normal".



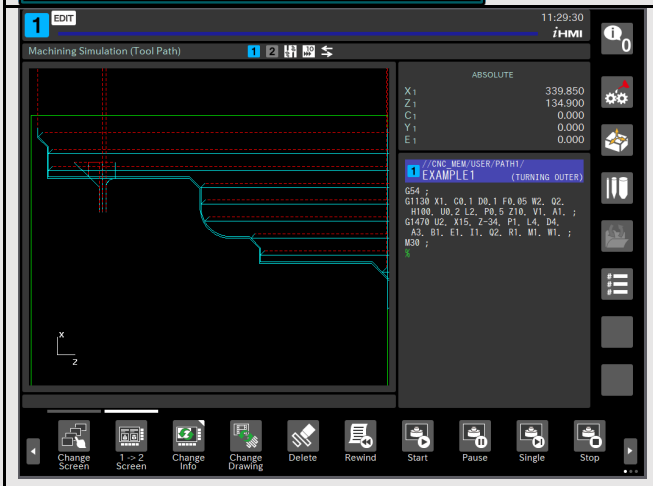
Set the data according to drawing.



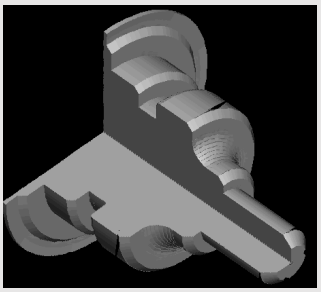


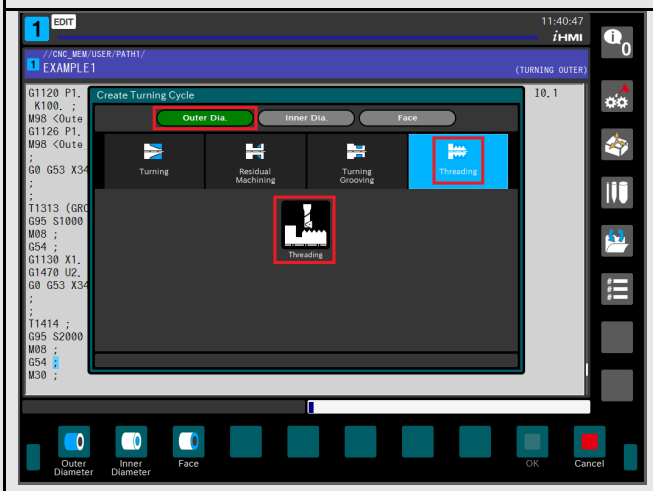


Set up "Corner Information" and "Repeat".



Grooving cycle is completed and can be tested/simulated.



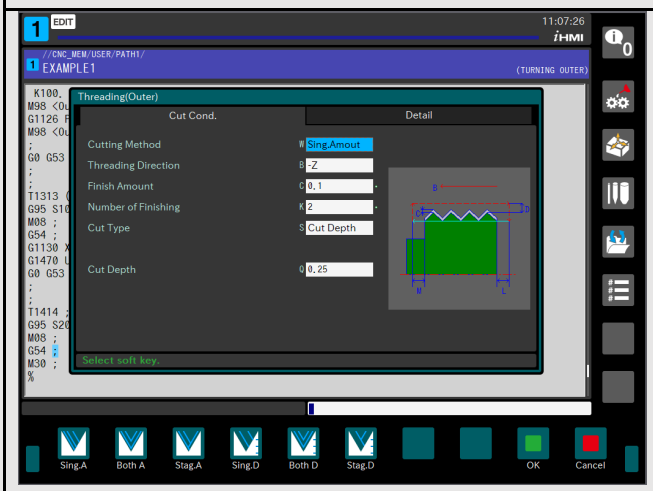


Next process is Threading.

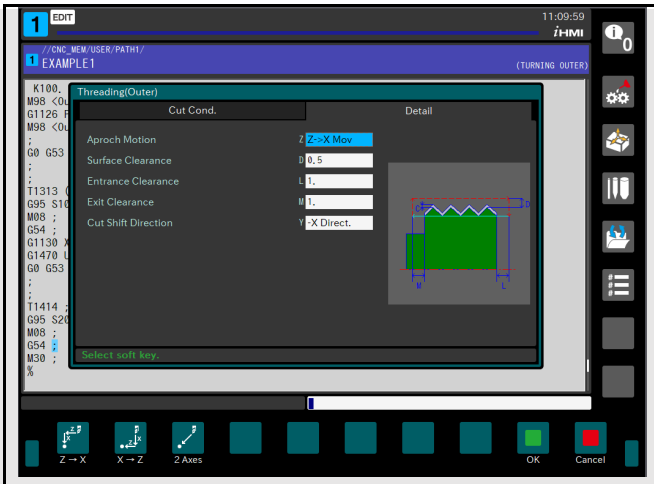
Input "Fixed Sentence", modify Tool and Spindle speed. Select G95 S2000 M3;

Go to "Machining Cycle" and "Turning".

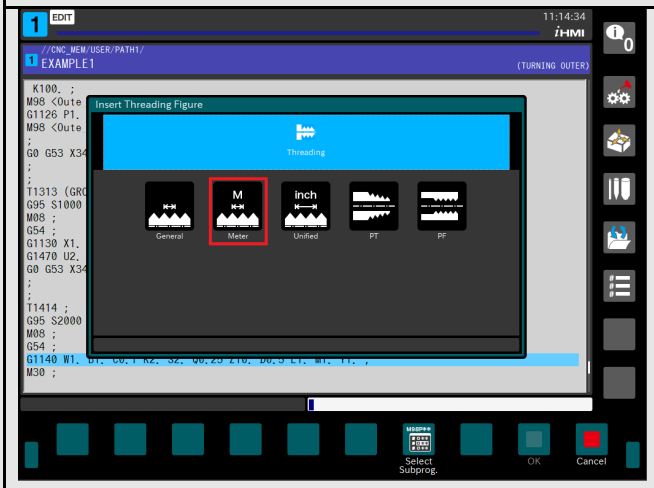
Select "Outer Dia.", "Threading" and "Threading".



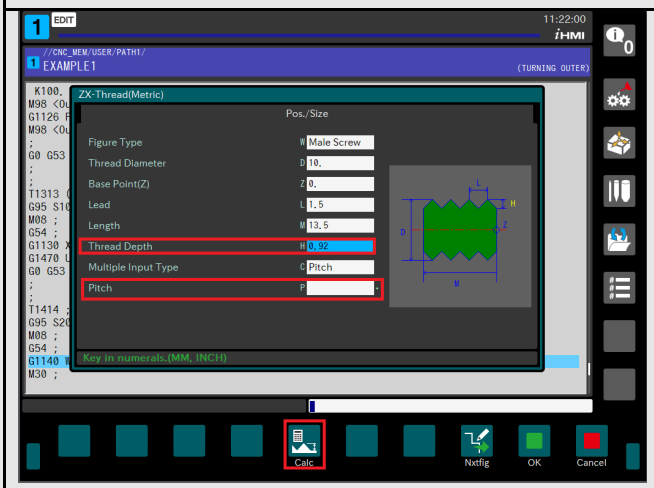
Set parameters as shown at left.
Do not forget the "Detail" tab. All data there are obligatory.



Set parameters as shown at left. Confirm by "OK".



The geometry screen – "Insert Threading Figure" appears. Regarding drawing we will machine metric thread M10.

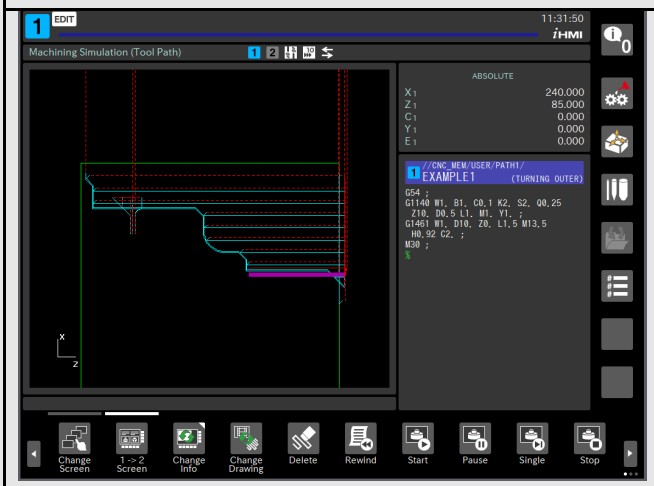


Set up parameter as left.

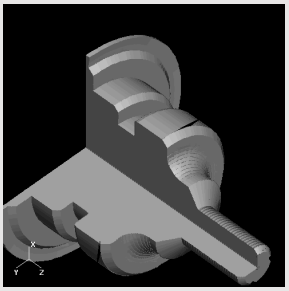
"Thread Depth" can be calculated automatically, based on standard formula for Metric Thread.

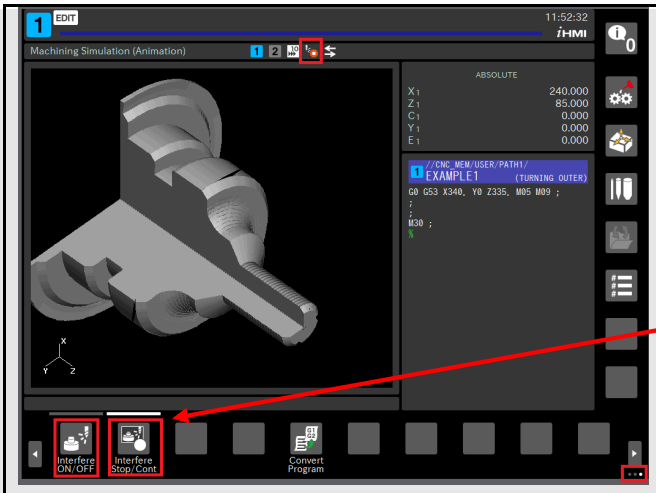
Once you have set up "Lead", choosing "Multiple Input Type" – "Pitch", for a single thread it is not necessary to input data.

Confirm by "OK".



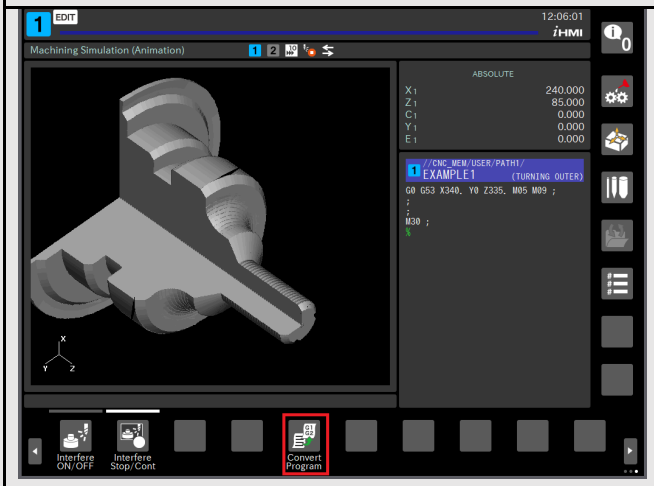
Program can be tested.





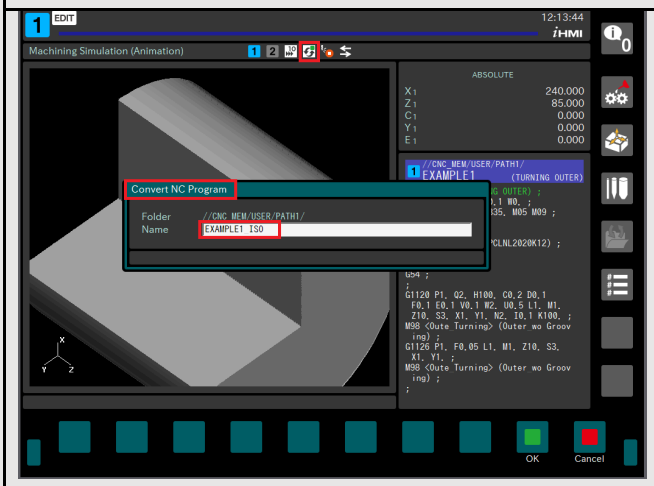
On the third horizontal key bar is possible to activate interference function – "Interference ON/OFF". The icon on the top indicates whether the function is active and whether the program will continue after encountering an interference (the touching point would be marked) or stop in the program line of interference. The selection is done by the key shown here.

Please keep in mind that this interference check is made based on graphic data. If there are too small path elements, this function may not work properly.

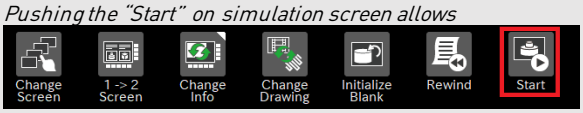


If you push "Change Screen" you can check and edit, if necessary, the program. If the "Fixed Sentence" consists of End of Program blocks, they can be input from there. Otherwise go to Tool Change/Index position using Copy/Paste function.

The iHMI Cycle Program consists of Gxxxx codes.
The program can be converted to ISO codes to see every step of the machining program.
The function is available in "Edit" mode on the simulation (second screen) and third horizontal key bar.

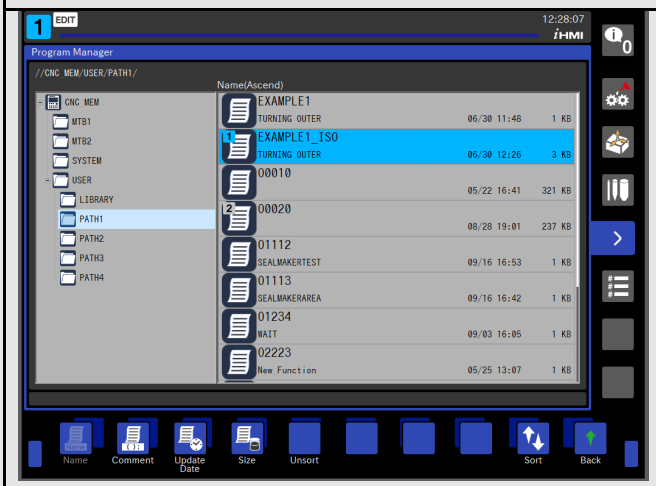


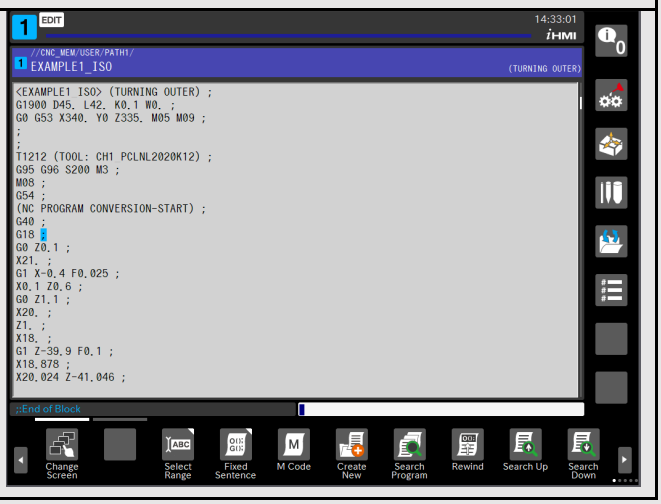
Pushing soft key "Convert Program" confirmation icon appears on the top of screen.
Pushing the "Start" on simulation screen allows



to register a name for the new ISO program.
Confirm by "OK".

Conversion takes several minutes, depending on the complexity of program and is times bigger than the Cycle Program.

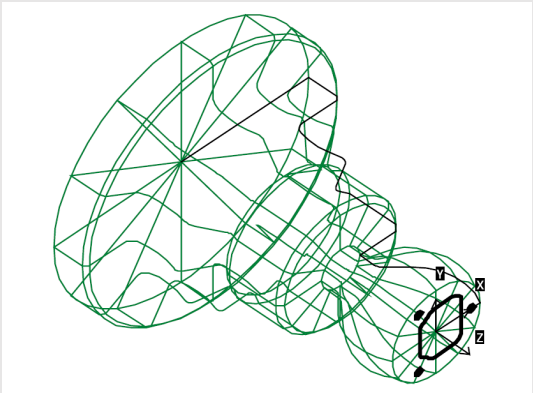


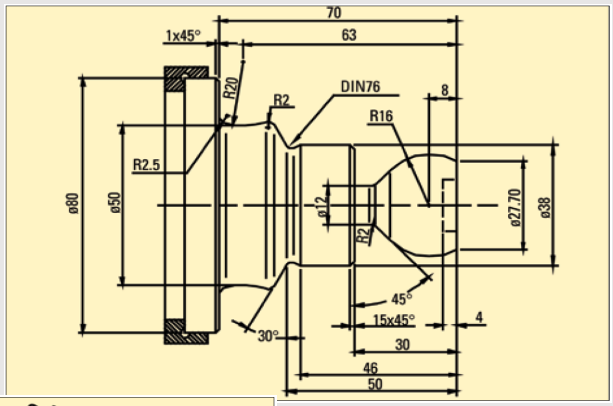


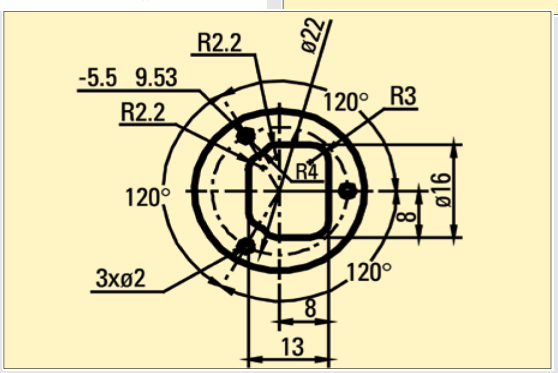
3. Example 2

NOTE: The following explanations will focus on new topics. The already considered procedures will be indicated as hints.
This example contains programming of a workpiece with iHMI Cycle Programming Tool, which has a falling contour and two unknown elements of geometric data.
The following machining cycles are introduced:

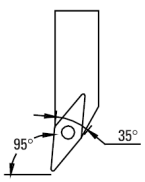
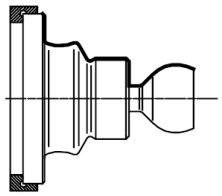
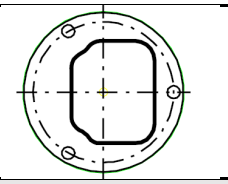
- External turning (roughing and finishing)
- Milling
- Drilling.

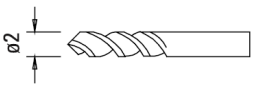
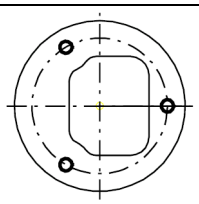


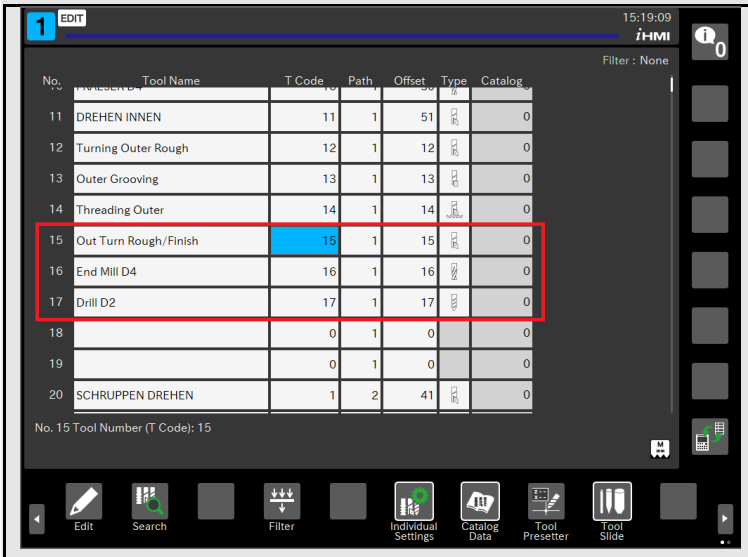




3.1 Necessary Tools

Tool Data		Machining processes	Machining Area
T1515		○ External turning-roughing ○ External turning-finishing	
T1616		○ Pocketing	

T1717		○ Drilling	

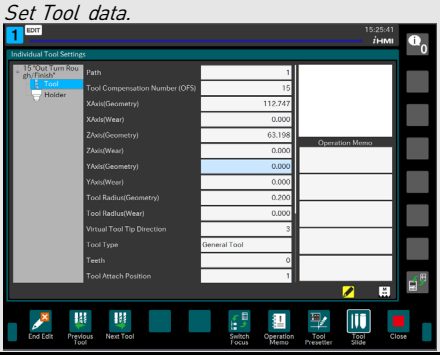


Go to "Tool Manager" 

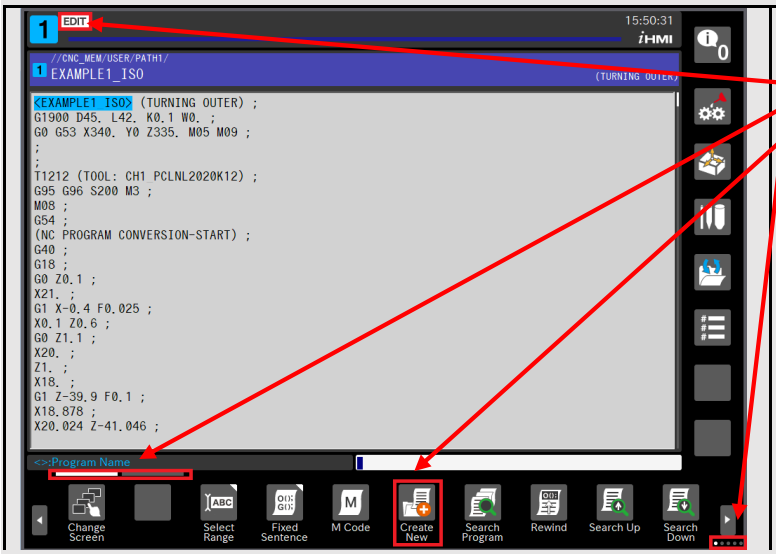
on tool manager screen

Change to "Edit".

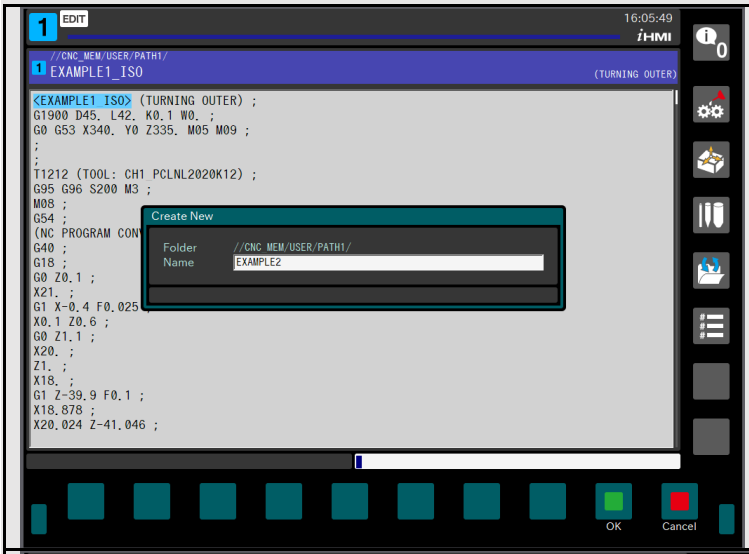
Set Tool data.



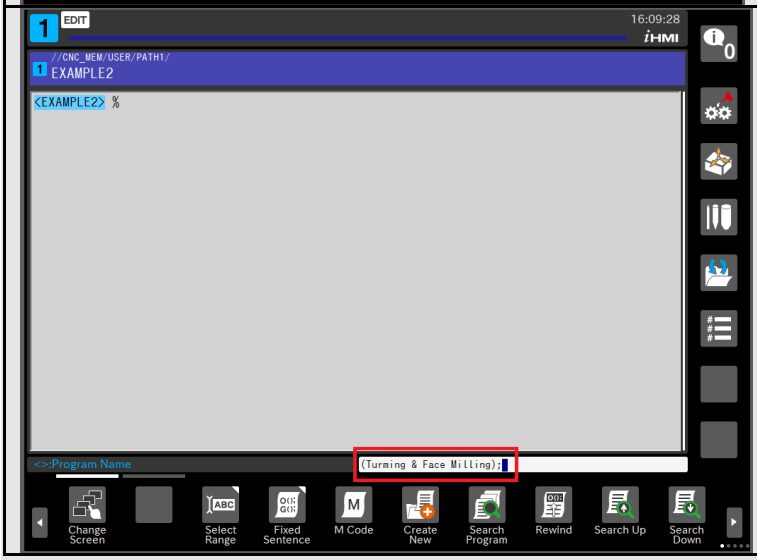
3.2 New Program



"Edit" mode
First screen
"Create New"
First horizontal soft key bar

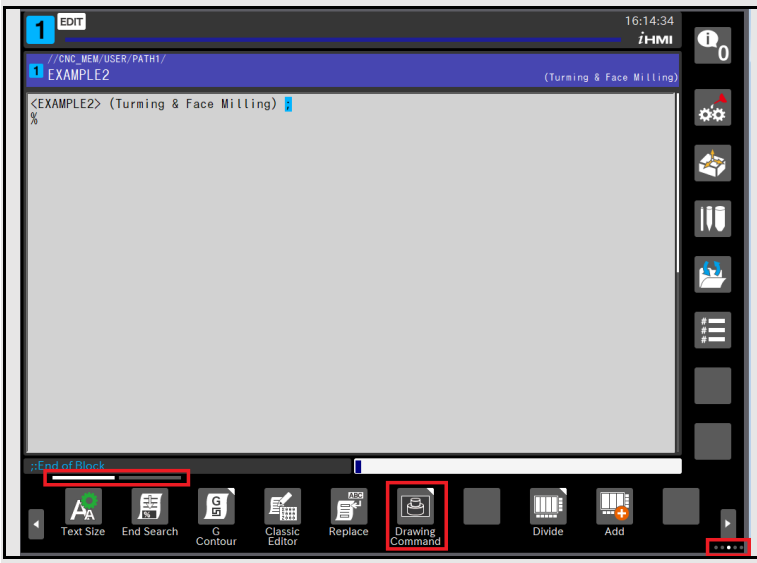


Type in name and confirm by "OK".

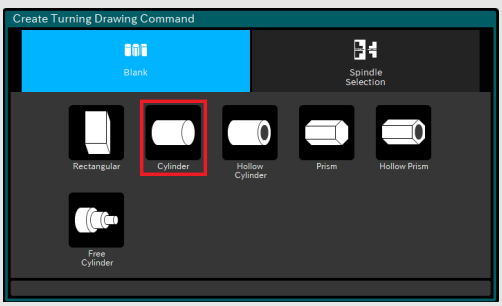


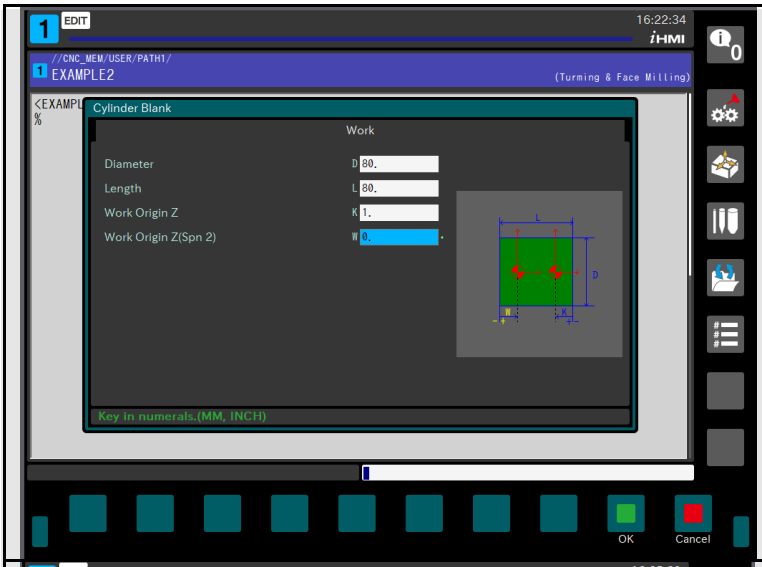
Enter comment with EOB (;) and push "Insert" on MDI keyboard.

3.3 Blank Definition

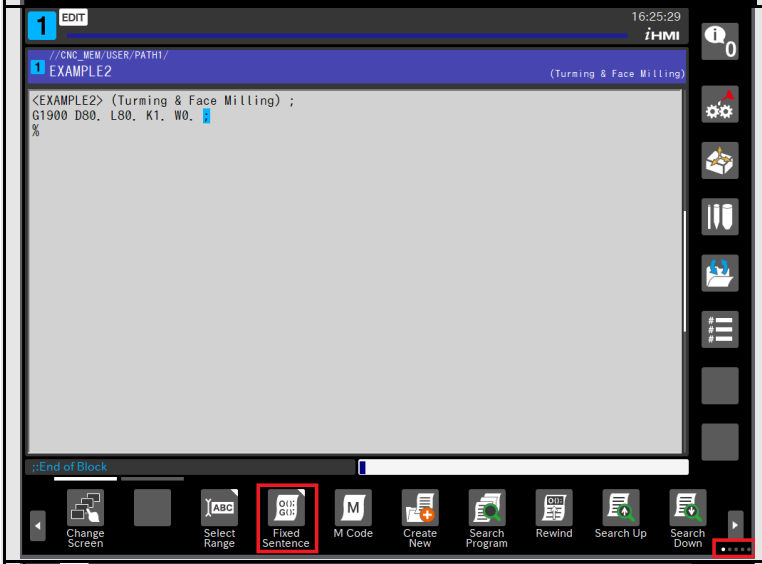


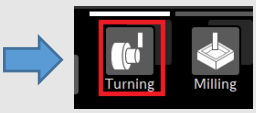
Blank definition



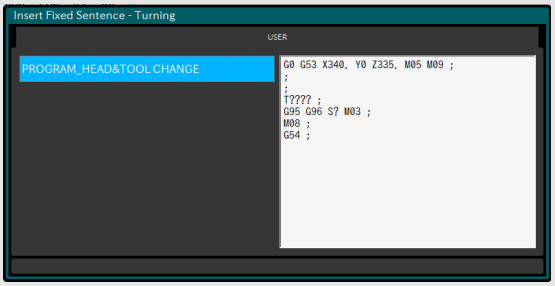


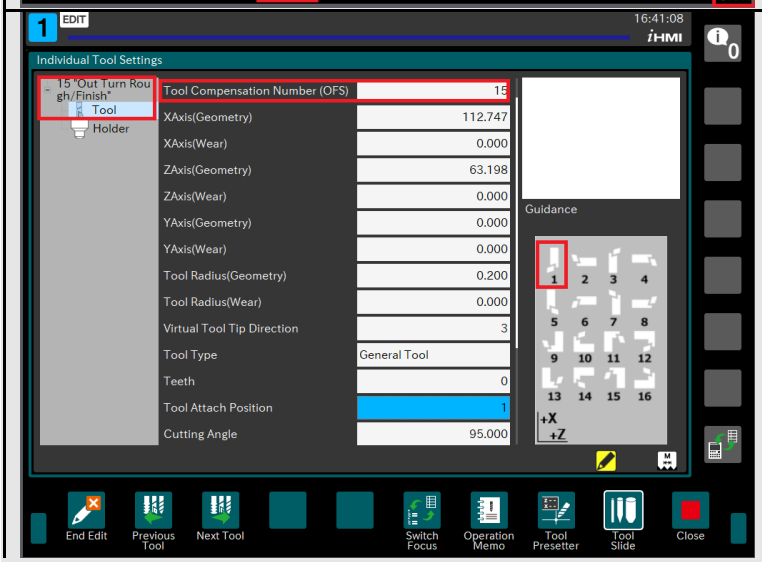
Complete parameter like shown left and confirm by "OK".





Insert "Fixed Sentence" for program head.
See the horizontal key bar number.



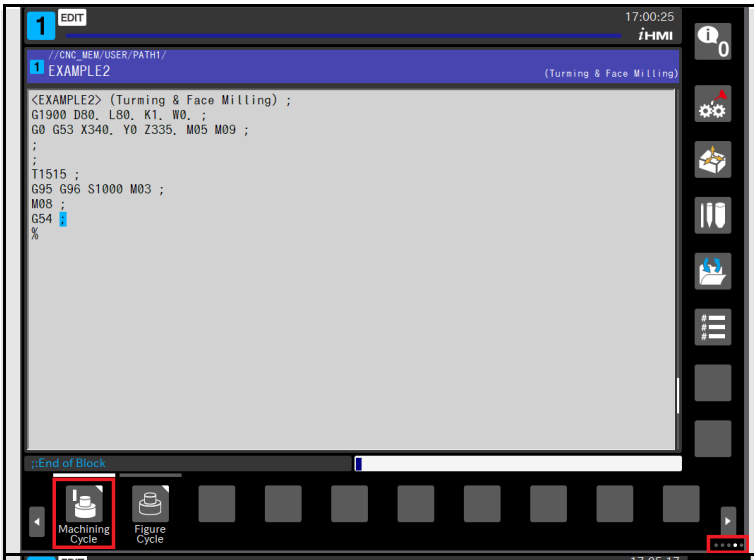


Turning tool "Out Turn Rough/Finish" is registered under number 15 and offset number 15.

Set the correct data in the program instead of questionmarks.

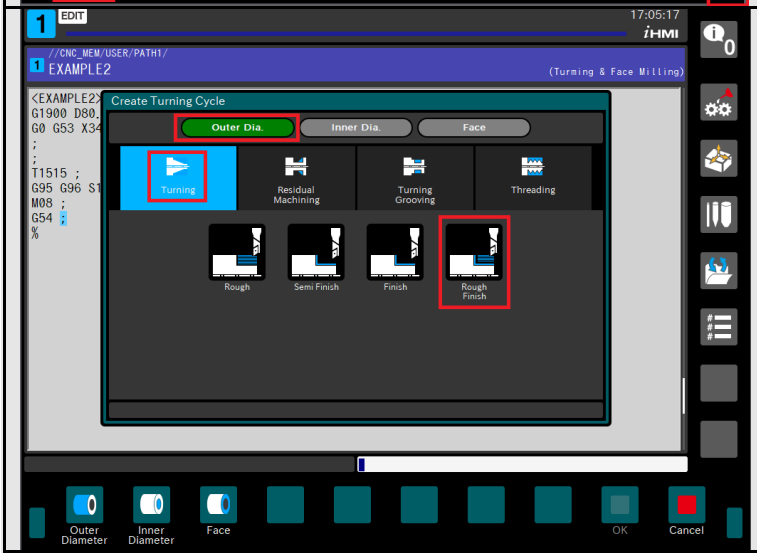
```
<EXAMPLE2> (Turming & Face Milling) ;
G1900 D80. L80. K1. W0. ;
G0 G53 X340. Y0 Z335. M05 M09 ;
;
;
T1515 ;
G95 S1000 M03 ;
M08 ;
```

3.4 Process: Outer Turning Rough & Finish and Contour

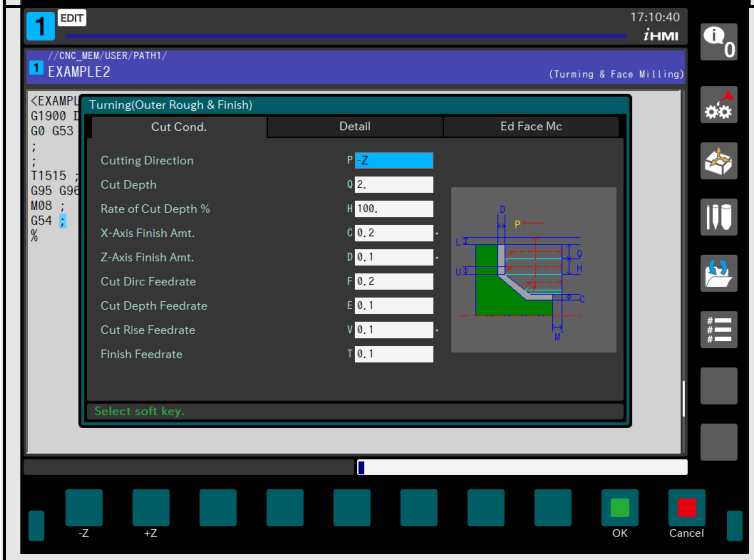


Push "Machining Cycle" on the 4th horizontal soft key bar.

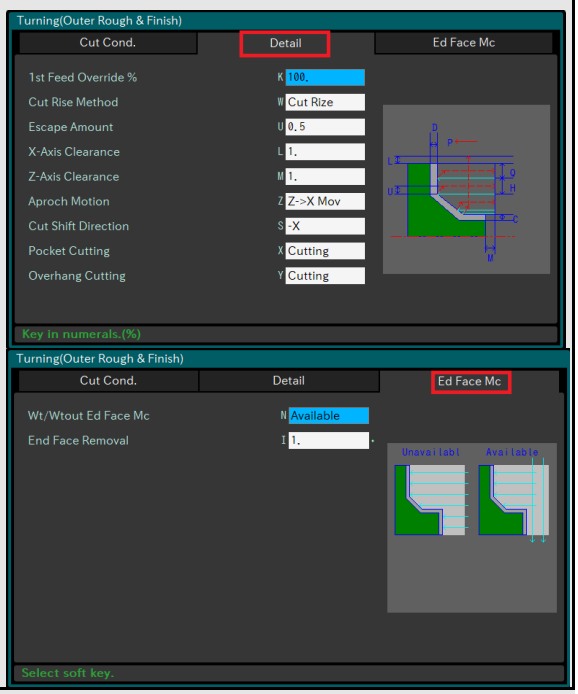
Select "Turning".



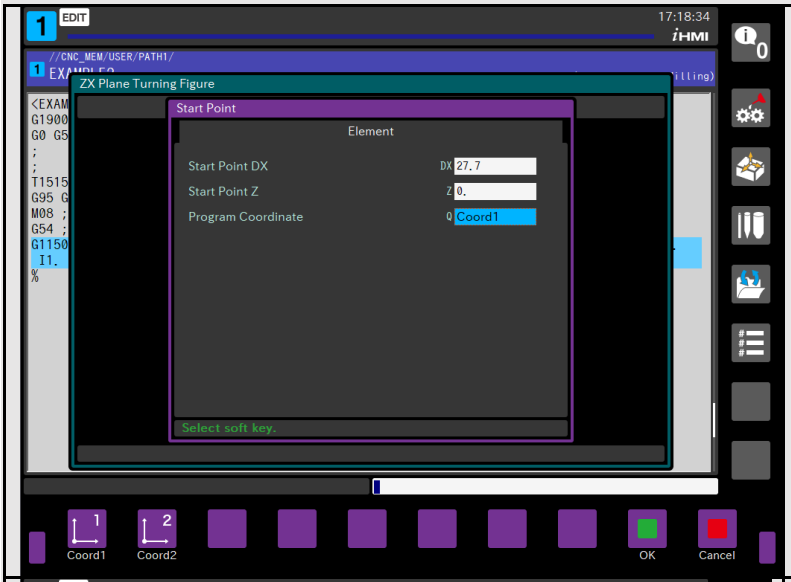
Select "Outer Dia.", "Turning" and "Rough Finish".



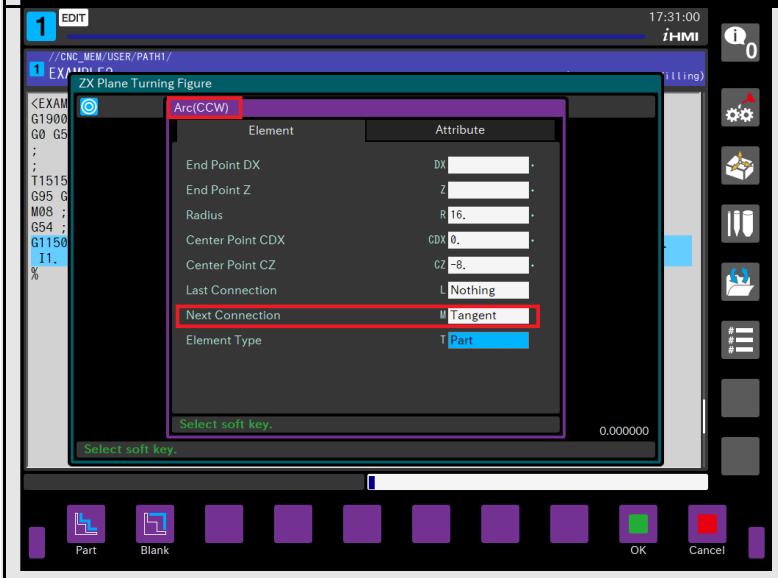
Set up data as shown and confirm by "OK"



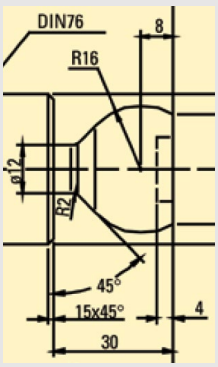
Contour: Outer Turning



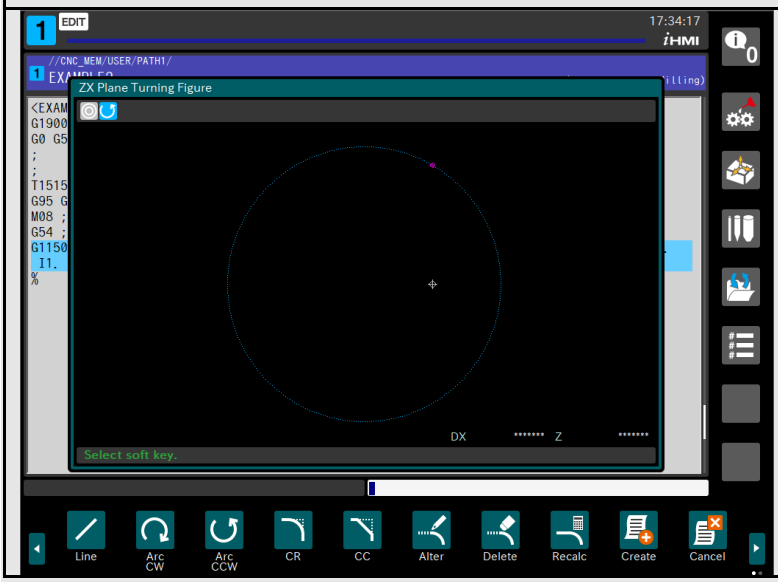
Start describing the work piece contour by "Start Point". Confirm by "OK".



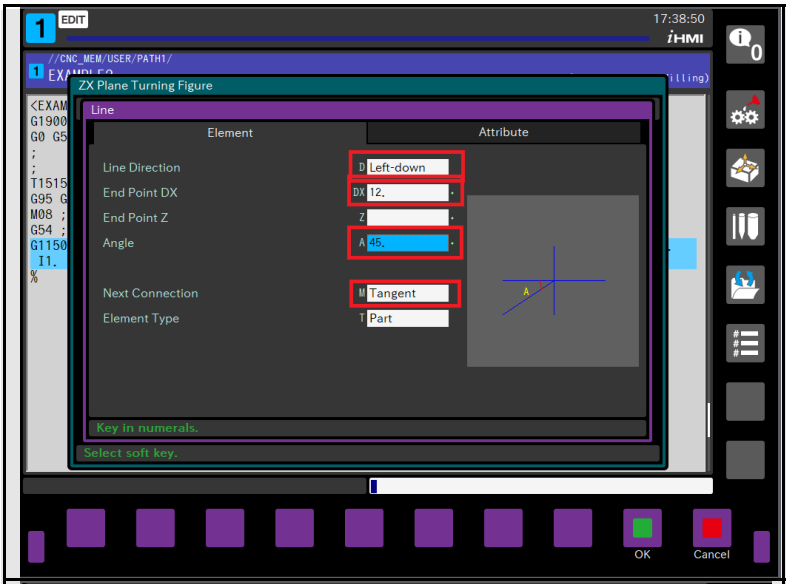
Next Arc (CCW) is defined sufficiently – Center Point and Radius. The Arc has no endpoint. It is tangent to the next line – 45 deg.



Set as shown to the left. Consider "Next Connection" – "Tangent".
"Atribute" here are not obligatory.



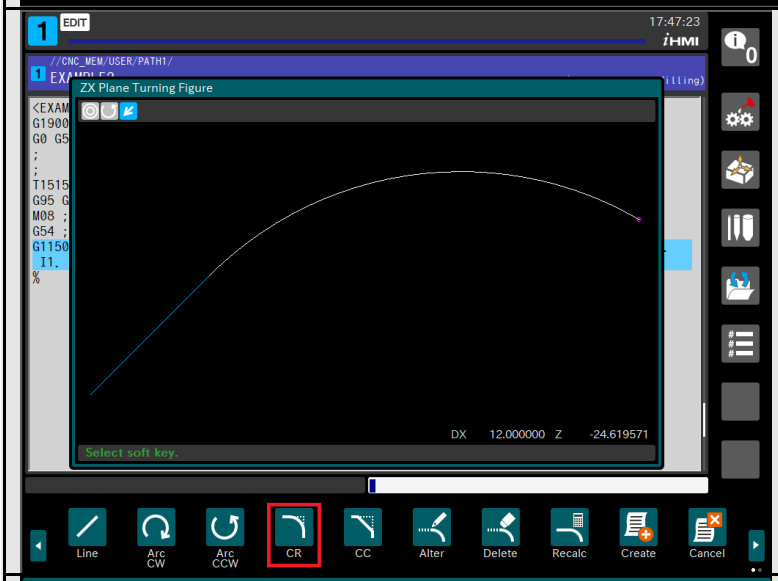
Element is shown, waiting for further data to calculate the endpoint.



Next line – "Left-down", ends on X12. mm and touches the previous element by 45 deg. from the horizontal line. Therefore, the line is defined sufficiently.

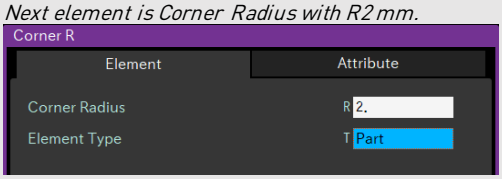
The next element (corner radius) will touch the line, too.

"Atribute" are not obligatory.

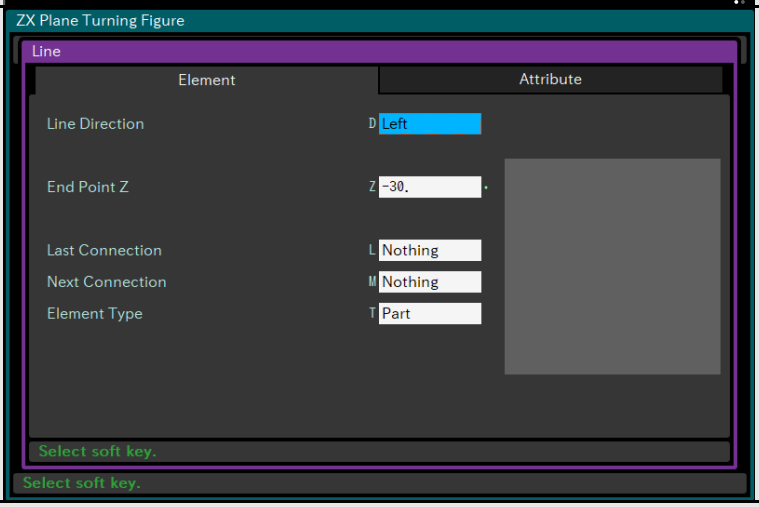


The end points of bot elements – CCW Ark and Tangent line can be calculated ad they are shown on the graph.

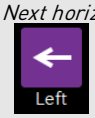
Next element is Corner Radius with R2 mm.

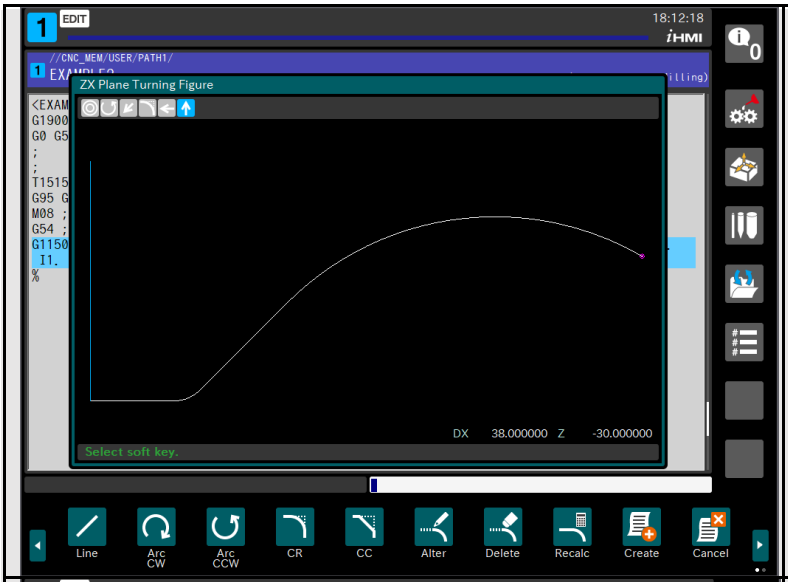


"Atribute" are not obligatory.

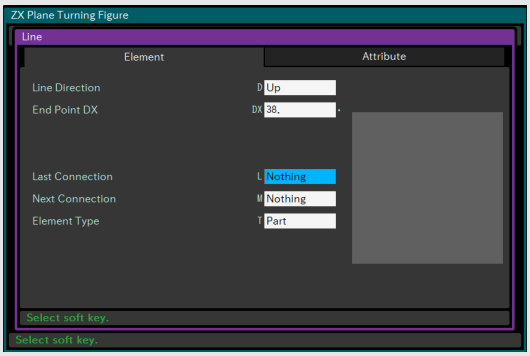


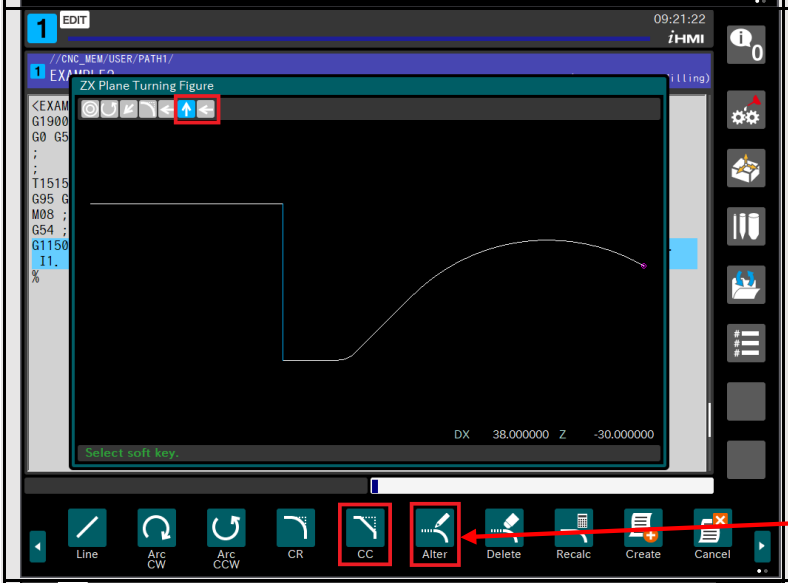
Next horizontal line – left to Z-30. mm





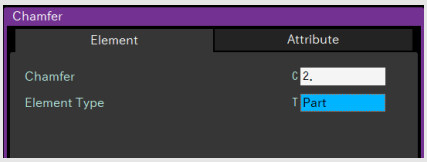
Next vertical line – up ends in D38., (without chamfer). Chamfer and Corner Radius can be embedded between two elements after their definition.



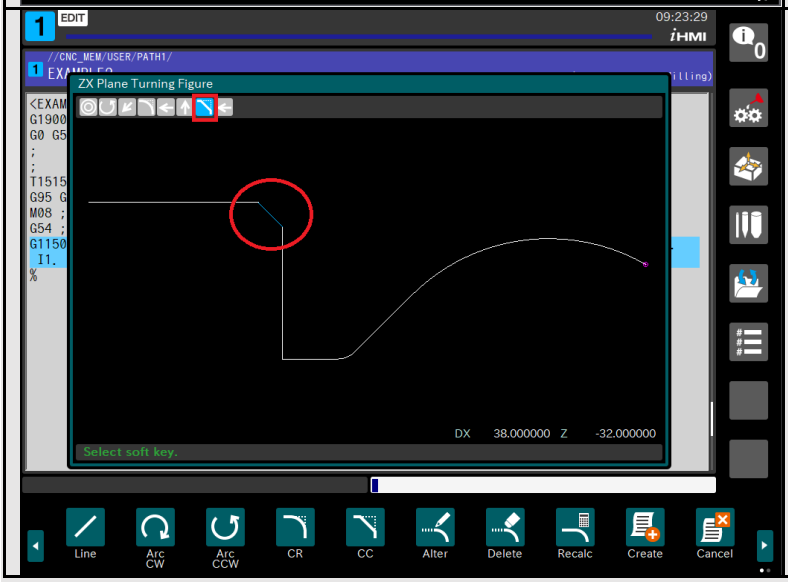


Insert "Line", "Left" to Z-46. mm. The contour looks like shown at the left.

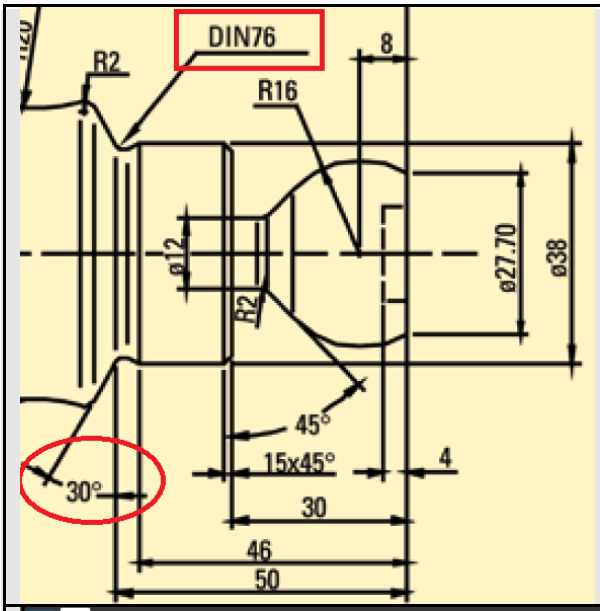
You can edit the prepared contour going by cursor (left) to the element after which you want to enter additional element (chamfer - CC). Push CC and set value.



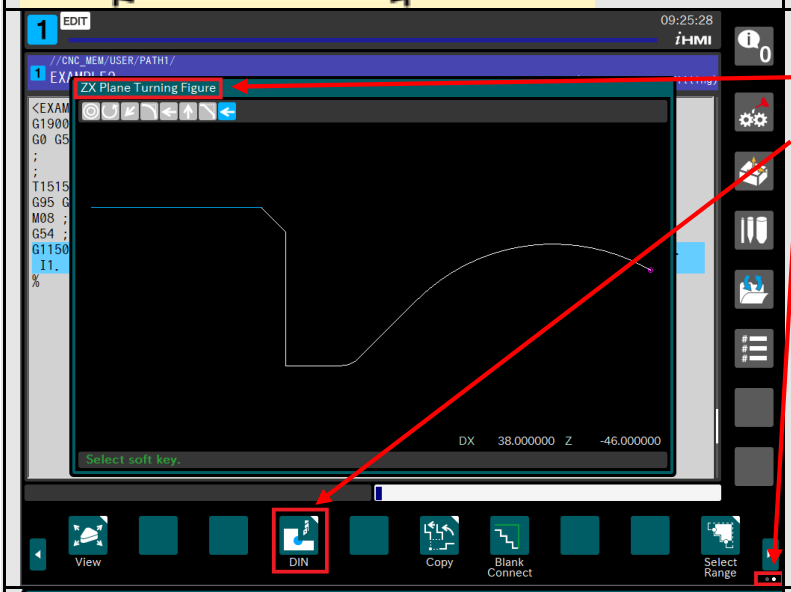
If you want to change data of the selected contour element, push "Alter".



Chamfer was inserted.



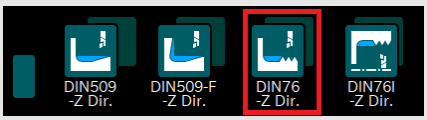
Next element is groove, standardized under DIN76, running over to a line under 30 deg.



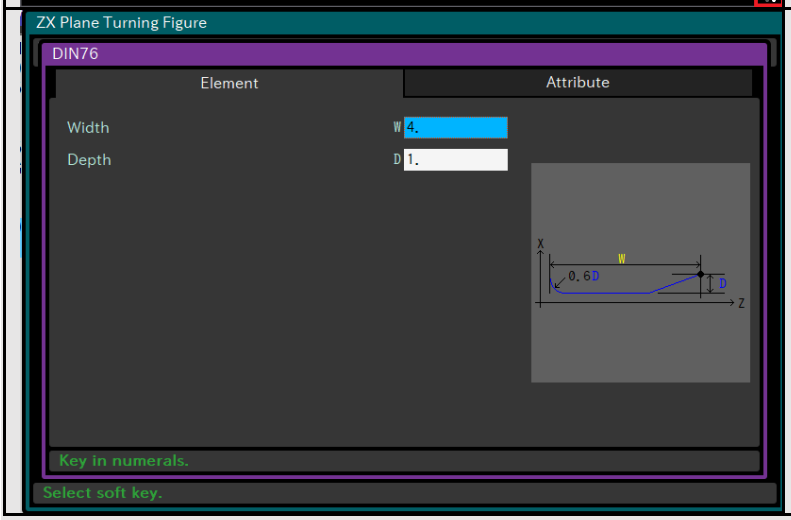
On the „ZX Plane Turning Figure“,

you can find “DIN” normed grooving elements under DIN.

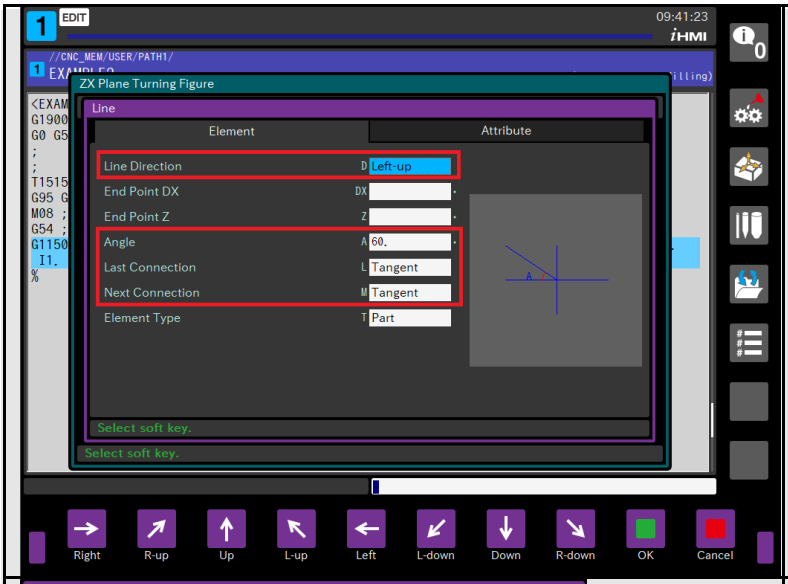
Second horizontal soft key bar.



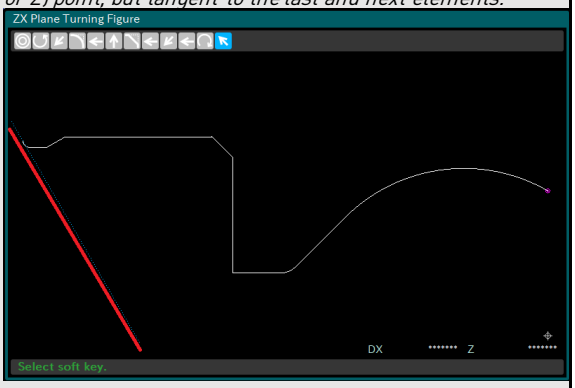
Select „DIN76 –Z Dir.“



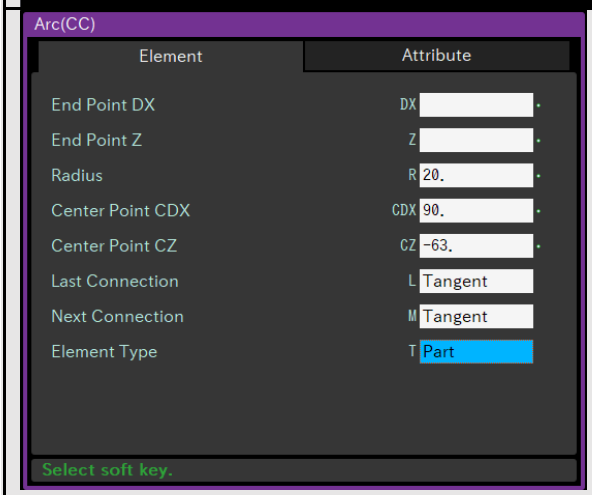
Set „Width“ and „Depth“ for the complete groove.
“Atribute” data are not obligatory.
Confirm by “OK”.



Next element is a line with angle 30 deg. to the vertical line (90-30=60 deg. to the horizontal line), without end (X or Z) point, but tangent to the last and next elements.



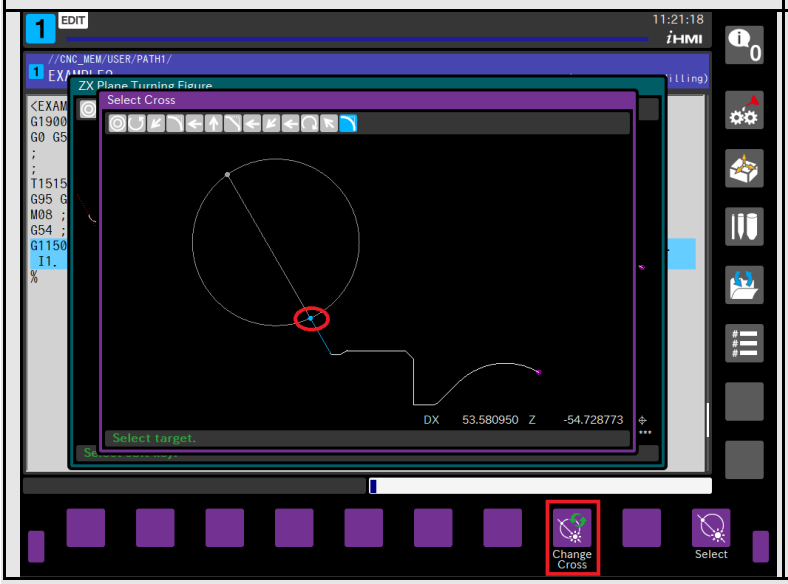
Line is shown, waiting for data to calculate end point.



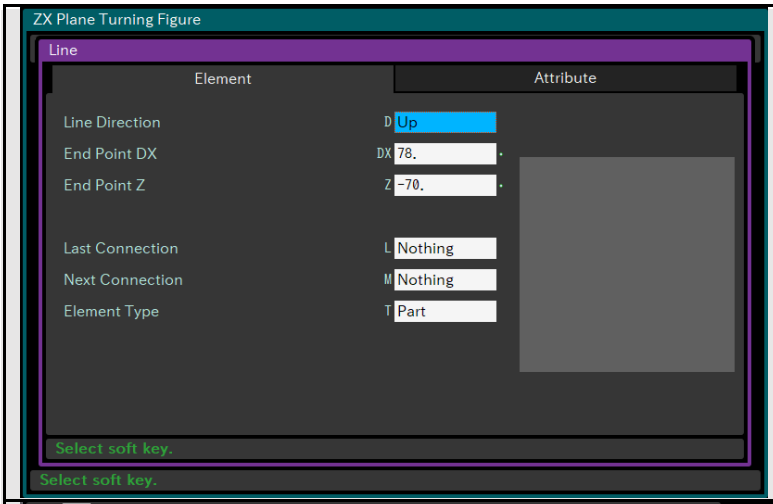
Enter corner radius of 2 mm.
*Next element is Ark Clock Wise (CW) with R20., Center Point Z-63. and X 90. (50+20*2 = 90). End point is unknown. Element is tangent to Last and Next element.*

(Last connection is omitted in new iHMI SW version 40)

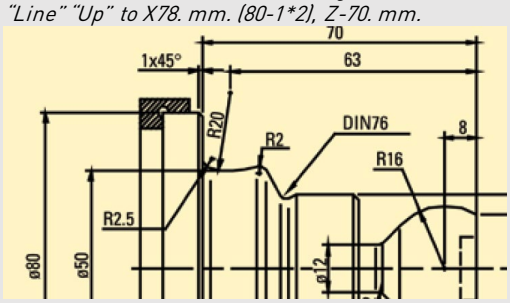
"Atribute" data are not obligatory.

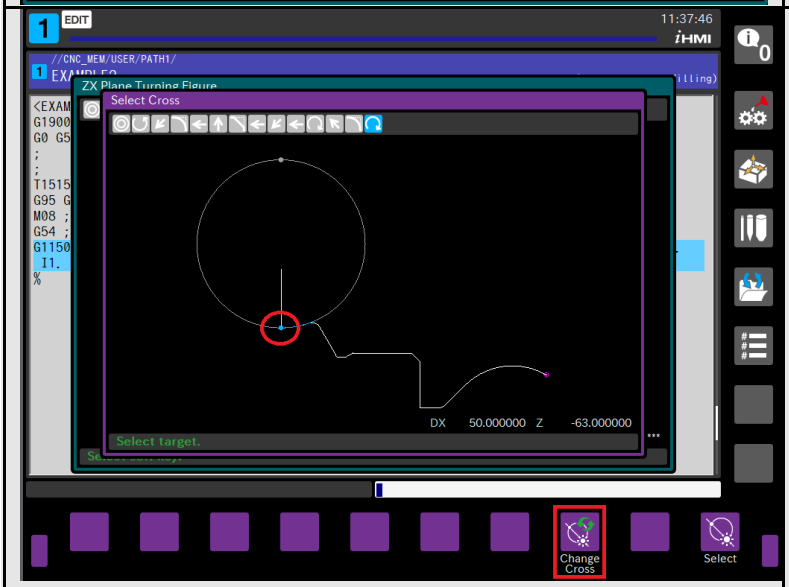


Select the below intersection point using "Change Cross" key. Push "Select".

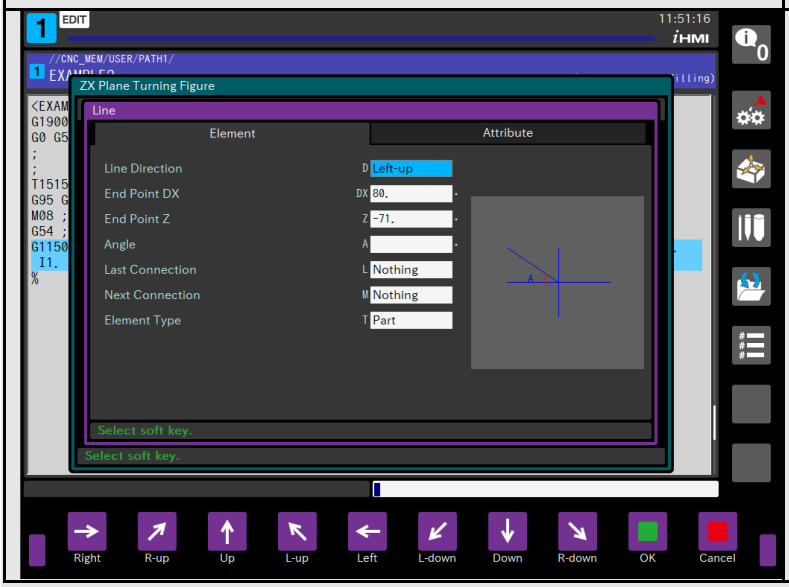


*Next elements are Corner Rounding (CR) R2.5 mm., and "Line" "Up" to X78. mm. (80-1*2), Z-70. mm.*





Select the lower crossing point. Push "Select".

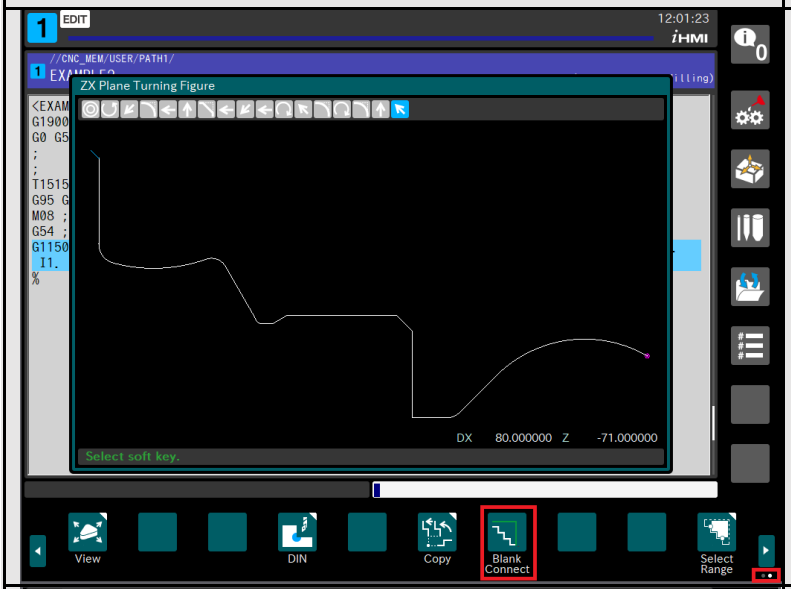


Complete parameter setting like shown at left.
The worpiece contour is completed.

Last line of the contour should be defined as line, not chamfer (CC cannot be the last element of a contour.) .

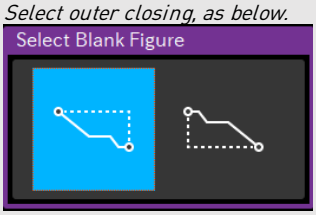
Now it has to be closed with Blank definition.

To edit an element, go back by arrow on the MDI keyboard to select the element and push "Input".

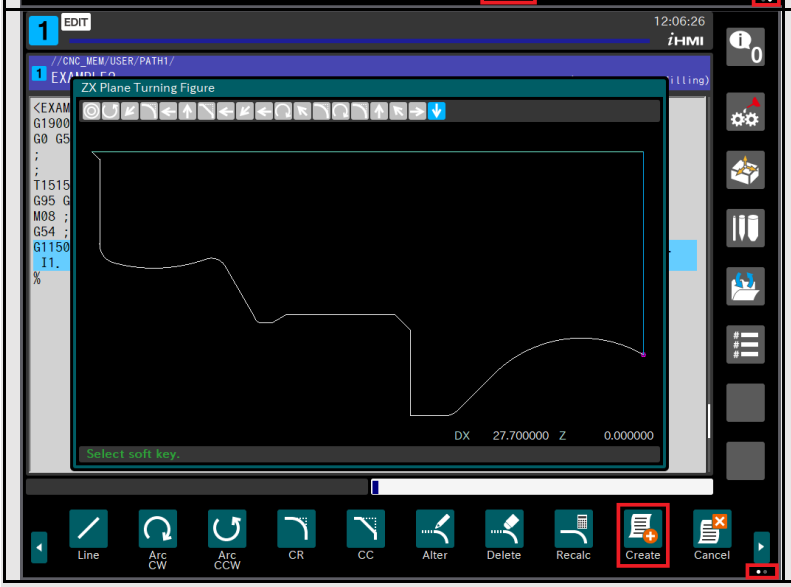


To close the contour automatically, go to the second horizontal soft key bar and push "Blank Connect".

Select outer closing, as below.

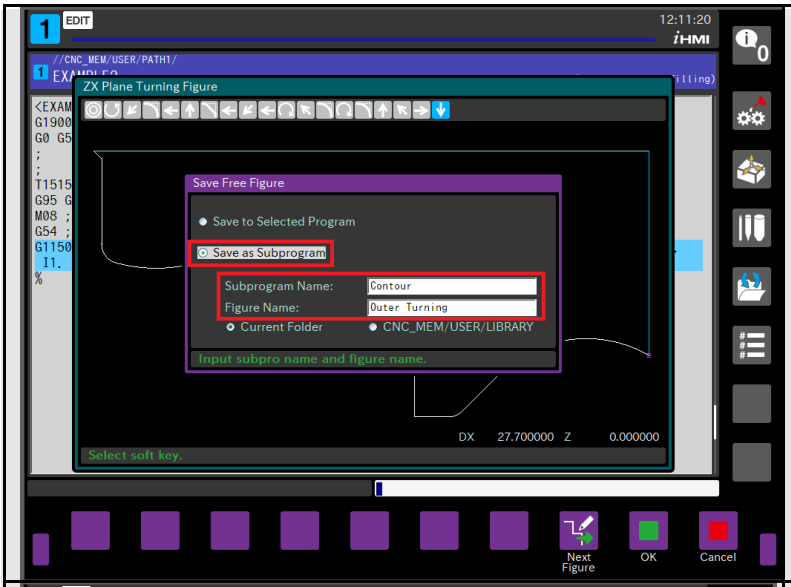


Confirm by „OK"



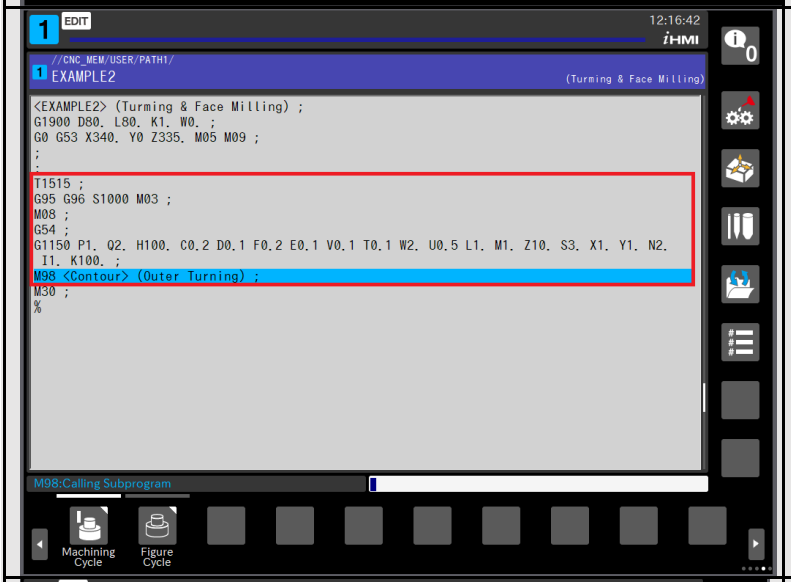
Contour for Outer Roughing & Finishing cycle is completed.

Push "Create" to save the geometry data.



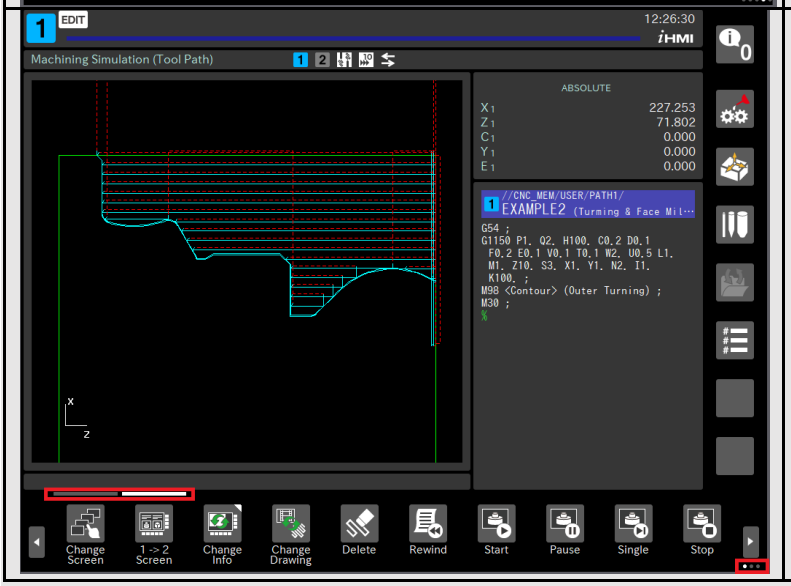
The geometry data can be saved as a "Subprogram". It can be edited or/and used at any time again for Outer Roughing / Finishing cycles.

In the main program the geometry subprogram is called by M98 (only 1 line).

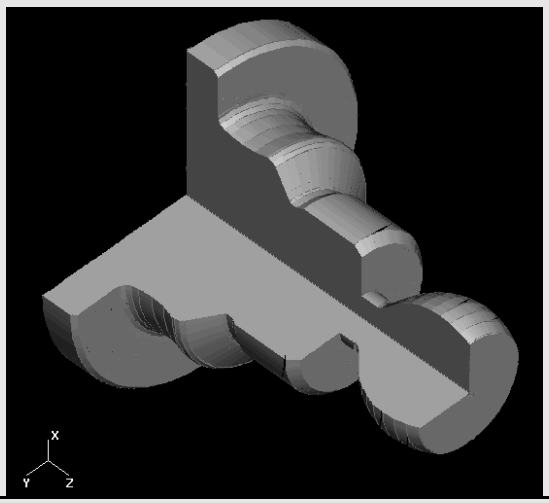


The complete main program for the Outer Turning Rough&Finish consists of 6 lines.

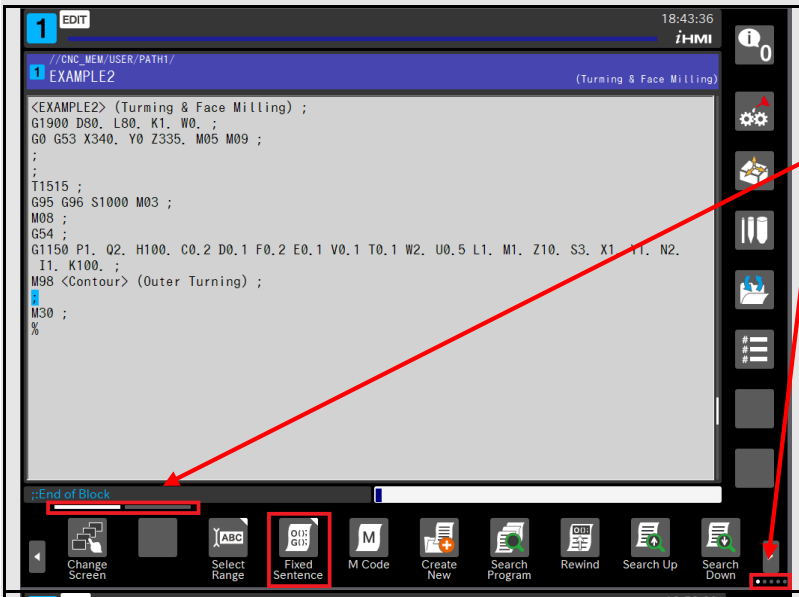
Entering M30 at the end you can test / simulate the prepared process withou alarms.



By "Change Drawing" the 3D simulation is enabled.

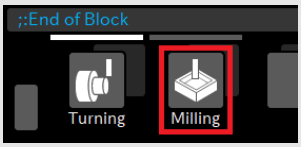


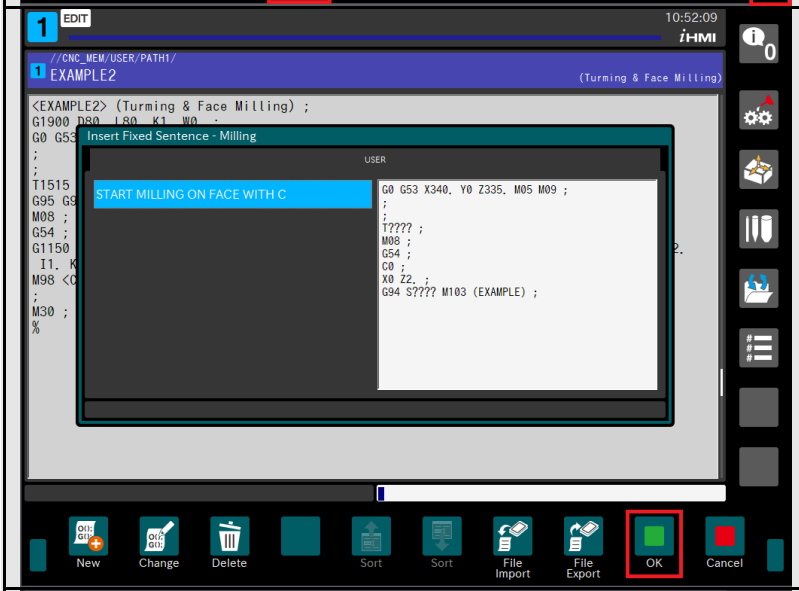
3.5 Process: Face Pocketing with C-axis



Select the suitable template from "Fixed Sentence", "Milling"

First Screen
First horizontal soft key bar.

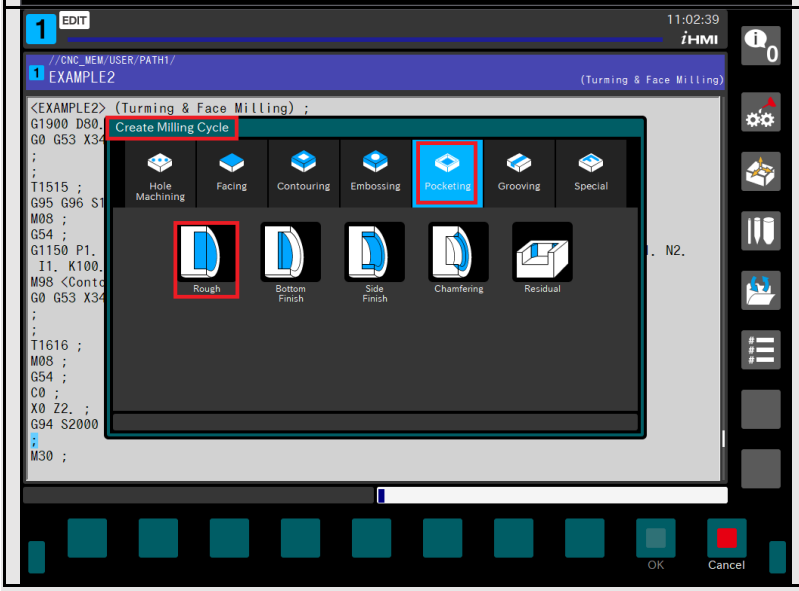




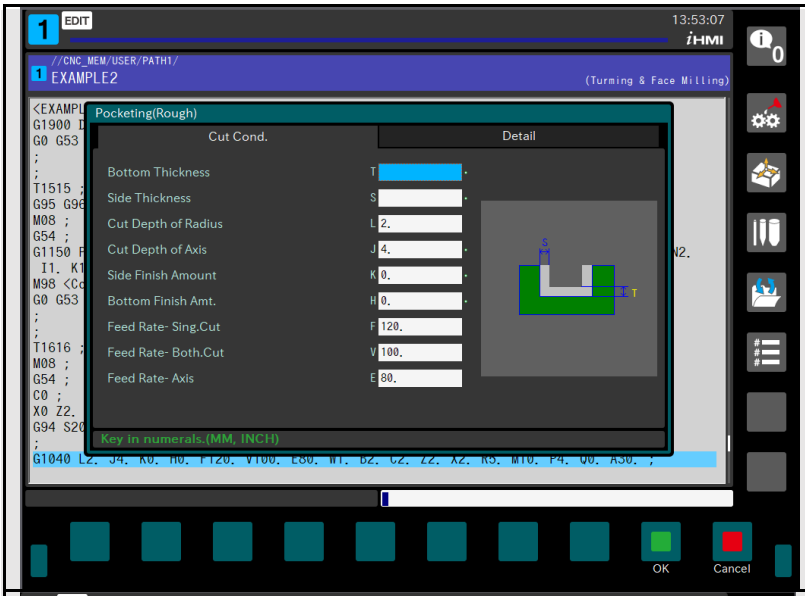
Push "OK" to insert data into the program.

M-Code for changing Spindle command from Main to Lifetool depends on Machine Tool Builder. M103 here is an example only.
MTB may have additional M codes to activate brake, for example.

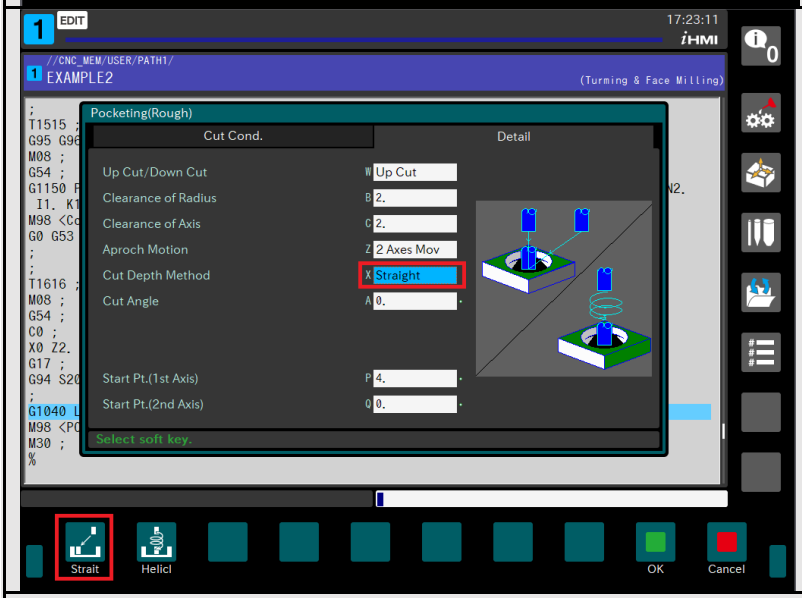
Tool number is 16, offset 16. Spindle speed is 2000 rpm.



Select "Milling Cycle", "Pocketing" and push "Rough".



Complete parameters as shown.
Check "Details".

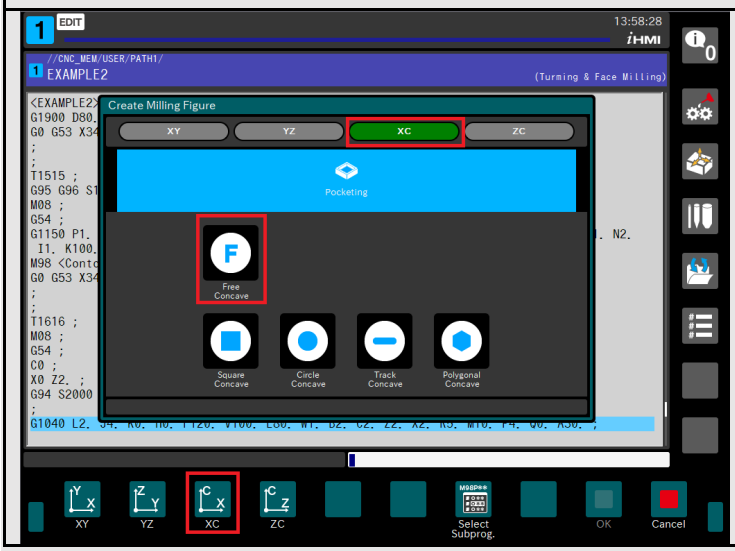


Select "Straight" cut-in method.

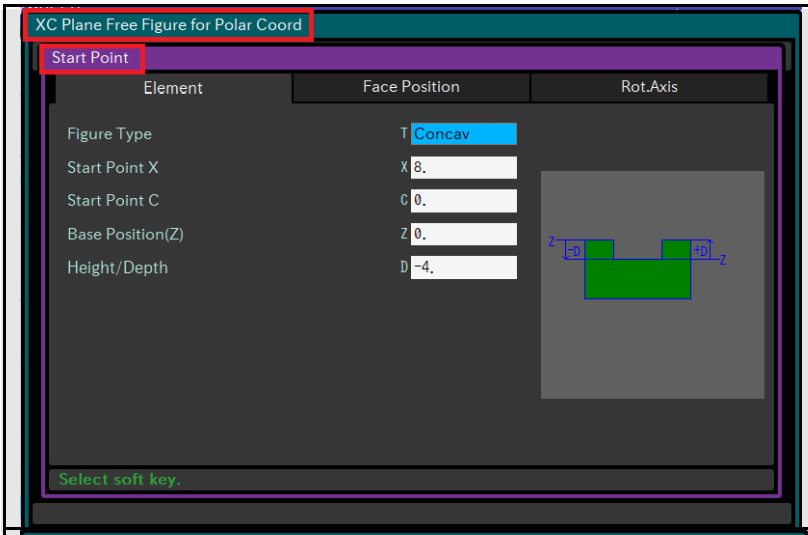
"Helical" - Spiral plunge of pocket machining cannot be used on ZC, XC or XA plane. Alarm message (MGi 3529) - WRONG SELECTING THE PLANE is the result, if it is selected.

Confirm by "OK".

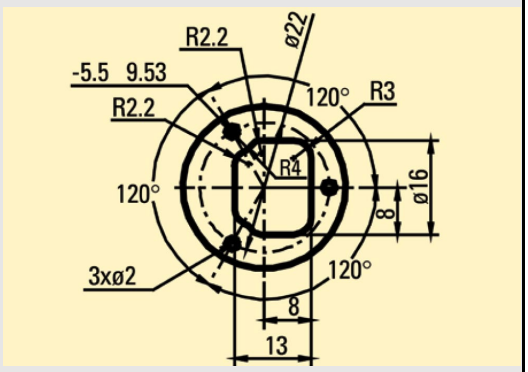
3.6 Geometry: Face Pocketing with C-axis

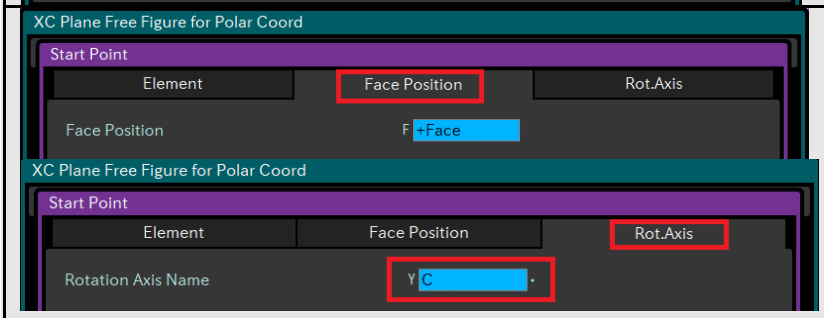


Screen switches automatically to geometry definition. Select "XC" – front face of part. Select "Free Concave" and confirm by "OK".



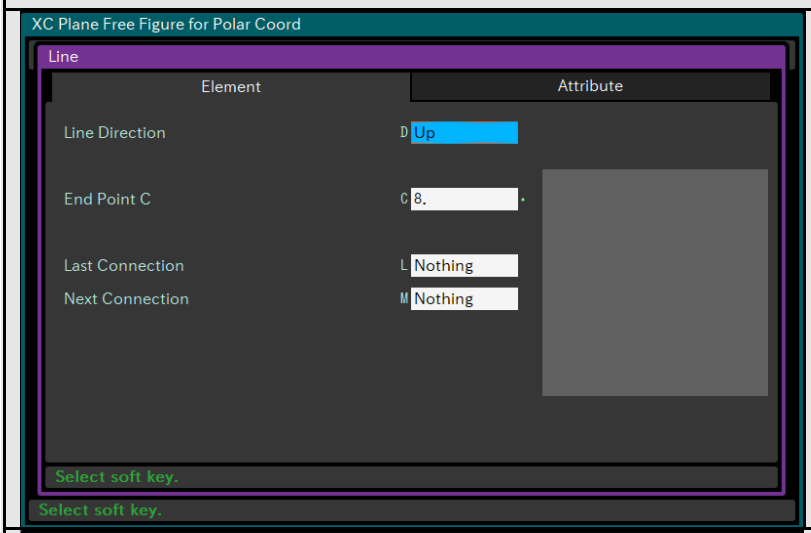
Set data according to drawing.





Check also "Face Position"

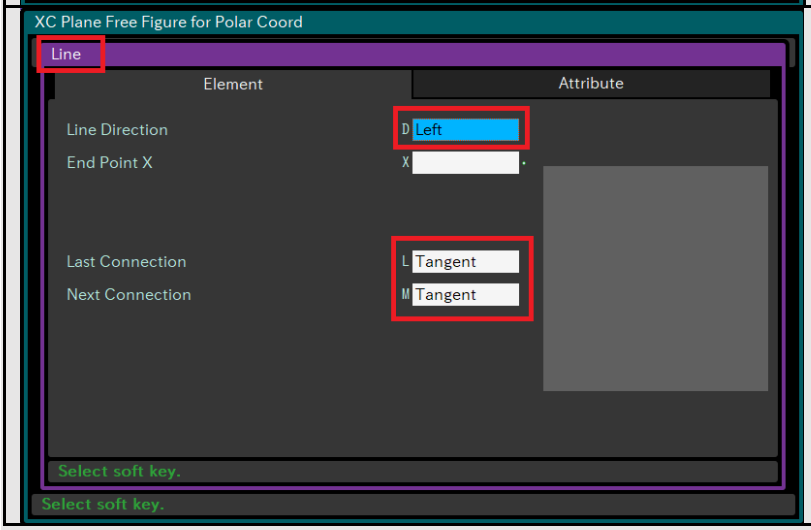
and "Rot.Axis"



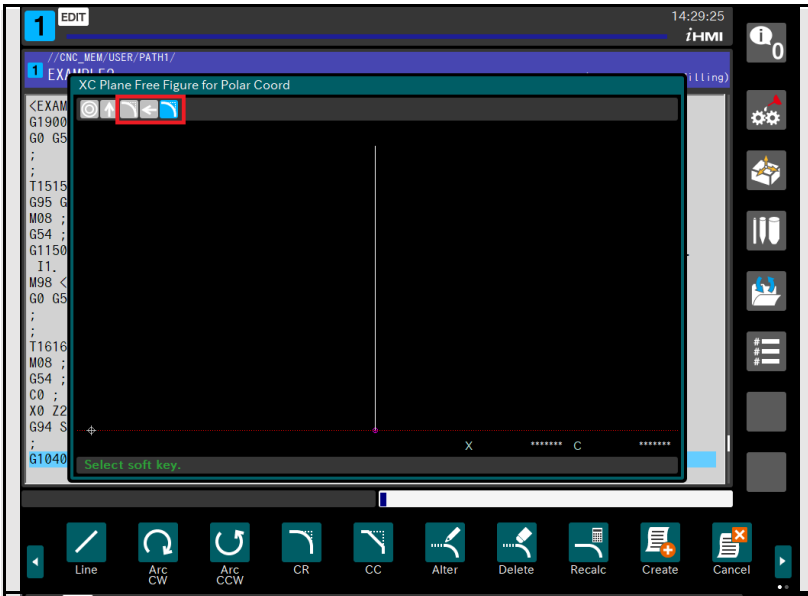
Select "Line", "Up" – 8 mm.
Please remember, although the line is in C coordinate, the value corresponds with the drawing data in mm. (the control calculates the deg. motion of C-axis automatically) and input is in Radius.

"Attribute" is not necessary to be set.

Next elememt is "CR" (Corner Radius) with 3 mm.

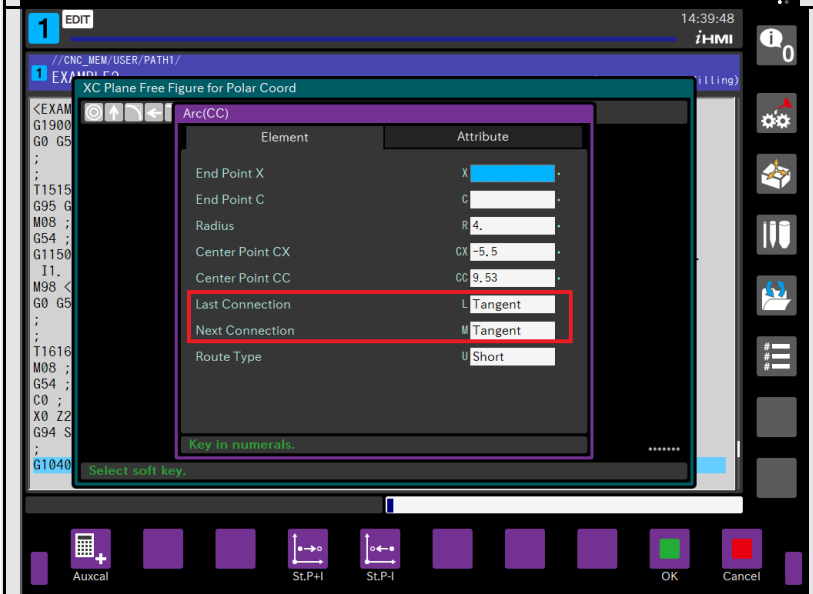


Next element is "Line" to "Left", which is tangent to the last "CR" and to the next element – a new "CR".



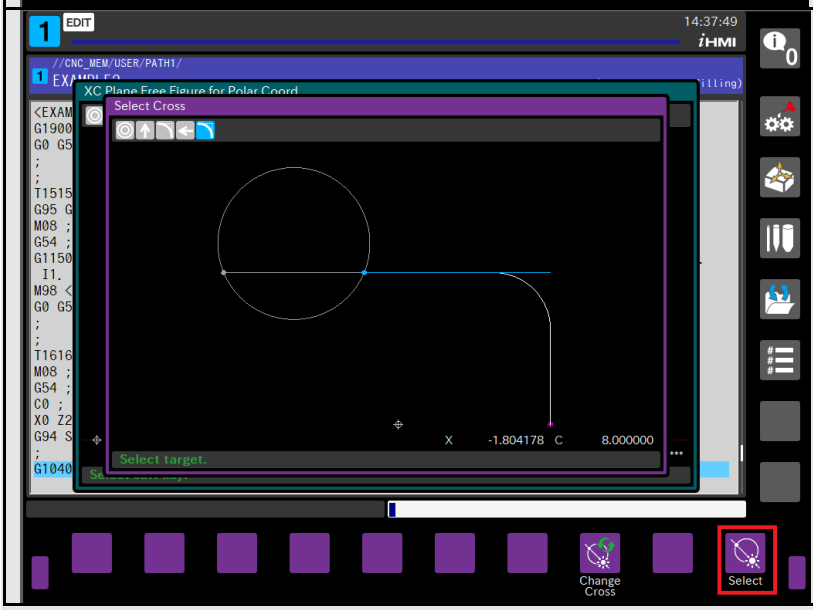
The last 3 elements are not enough defined to be drawn. Control is waiting for final definition to connect them to each other and show on display.

The next element Arc, will submit the necessary data.



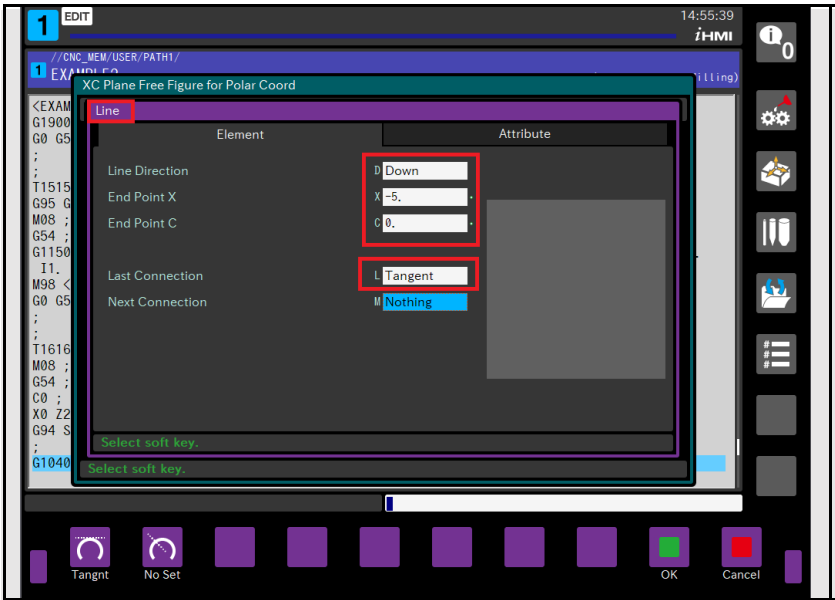
According to the drawing, the next element is "Arc CCW" with Radius 4. mm and centre point X-5.5 and C9.53 mm. This element should be tangent to "Last..." and "Next Connection".

"Atribute" are not obligatory.

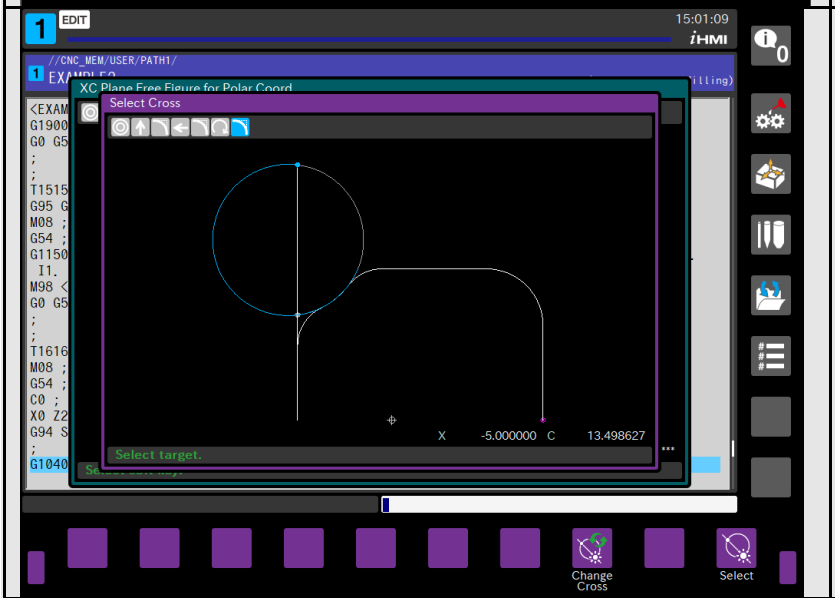


The defined arc has 2 crossing points with the line. Change selection by "Change Cross" and confirm by "Select".

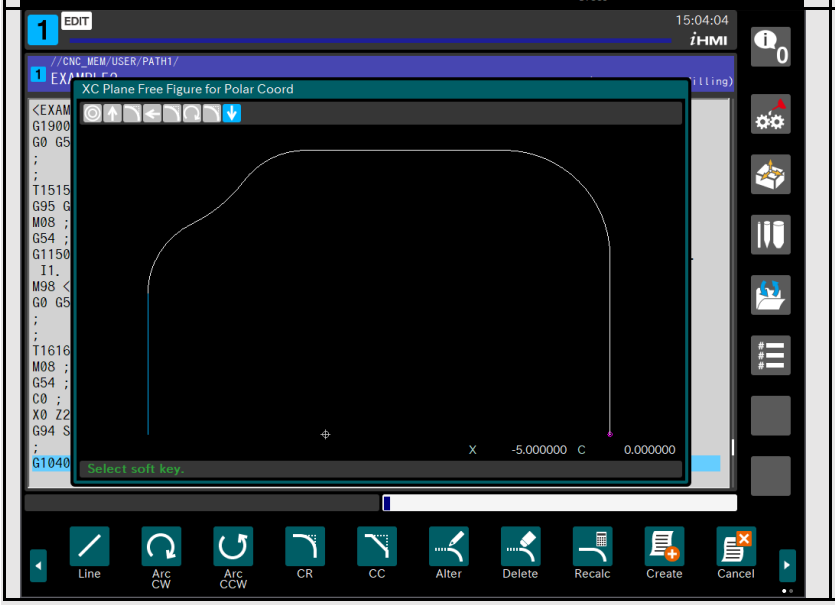
Next element will be "CR" – Corner Radius of 2.2 mm. again, which is tangent to the arc, but will not have enoug information to define finaly the last elemet in the chain.



The next element "Line", "Down", with end point X-5.mm and C0. mm., and tangent to the last connection, gives enough data to determine the last elements.

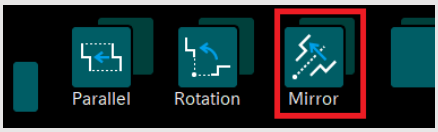


Intersection point has to be defined.

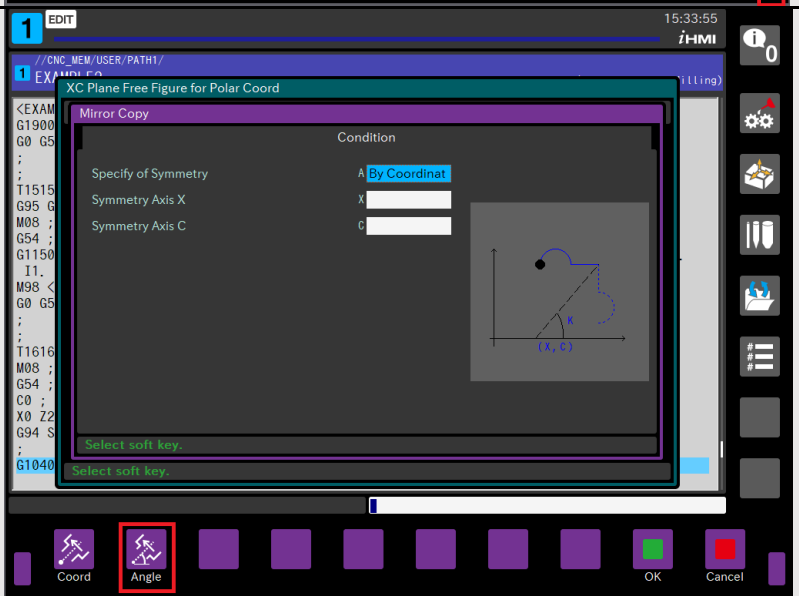


Now half of the contour (over the horizontal line) is prepared.
If we can Mirror the Half over the horizontal line, we can close and finalize it.

Go to second horizontal soft key bar and push "Copy".

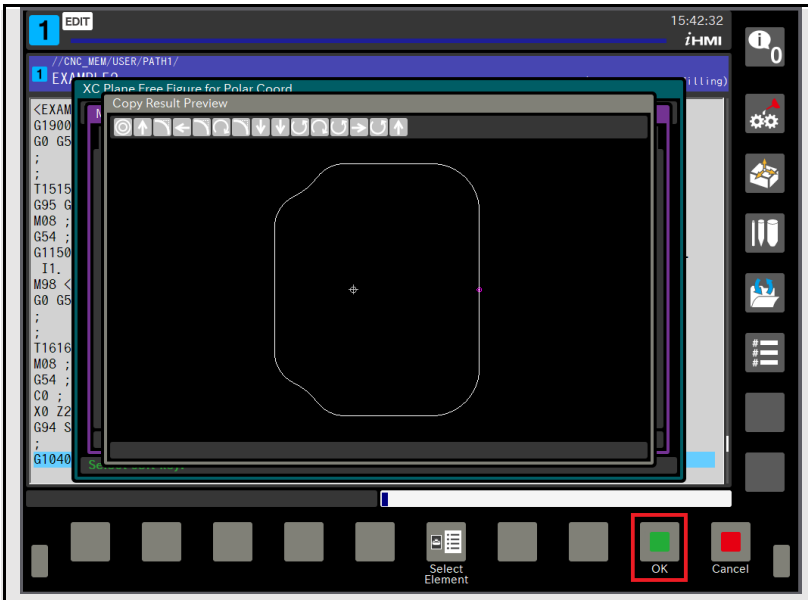


Select „Mirror".

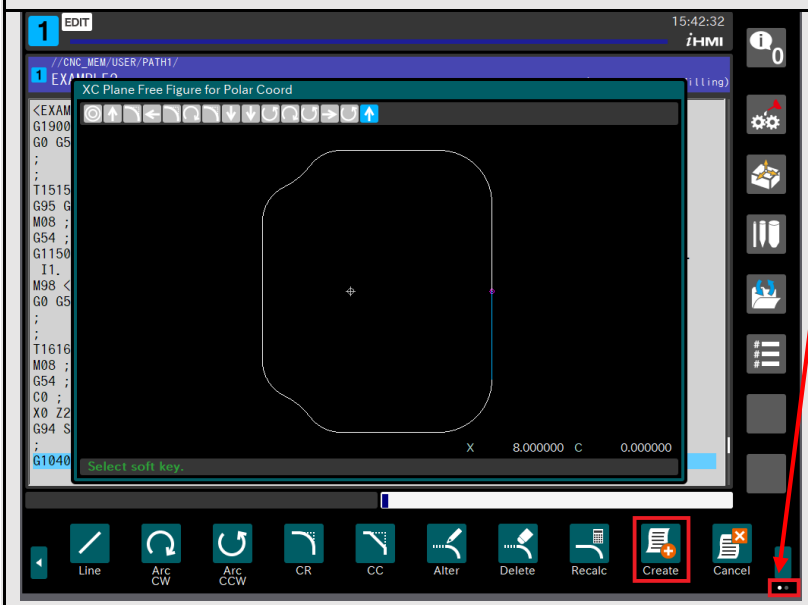


Select "Angle" and insert angle K = 180 deg.

Select, using Left/Right arrow on key board the elements to be mirrored. By "Preview" key is possible to see whether element selection was correct.

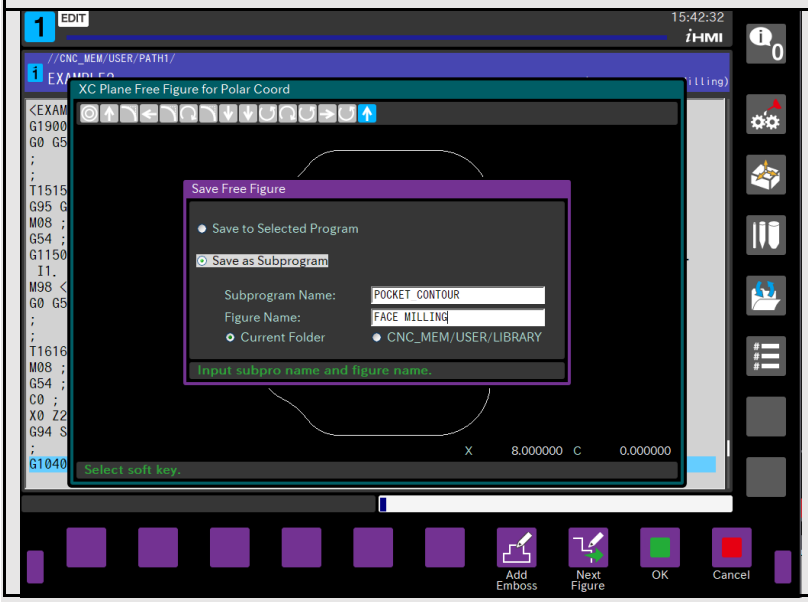


If YES, confirm by "OK".

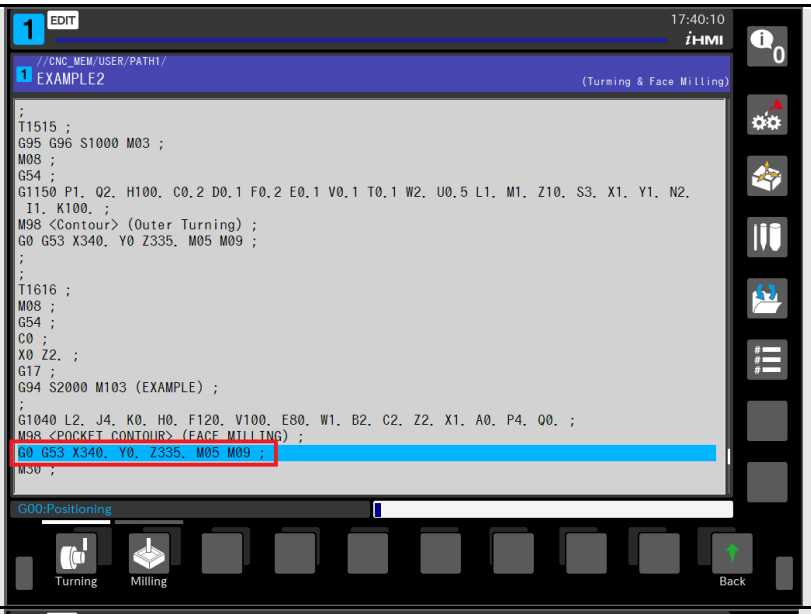


Now selected elements are mirrored and the contour is closed. Here you can edit every element as desired.

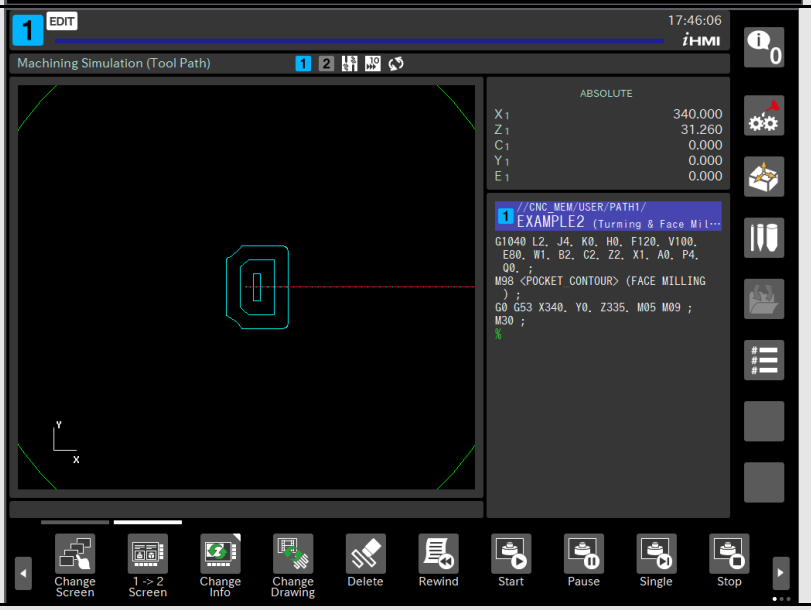
Go to first Soft Key Bar and create subprogram for the pocket contour.



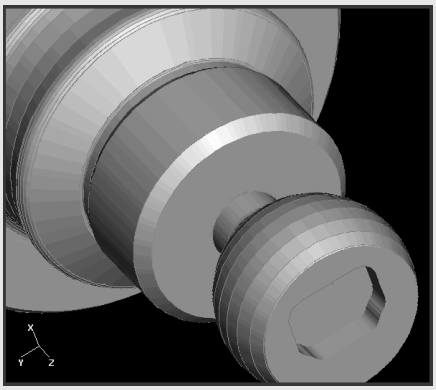
Set es shown on the left and confirm by "OK".



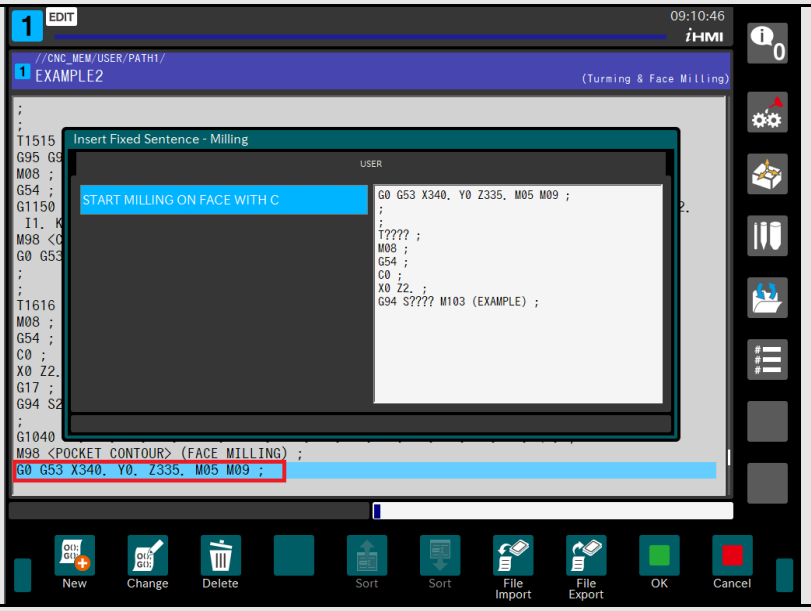
Command movement to Index / Tool change position and test / simulate the program including the pocket cycle on the XC face plane.



3D immage:

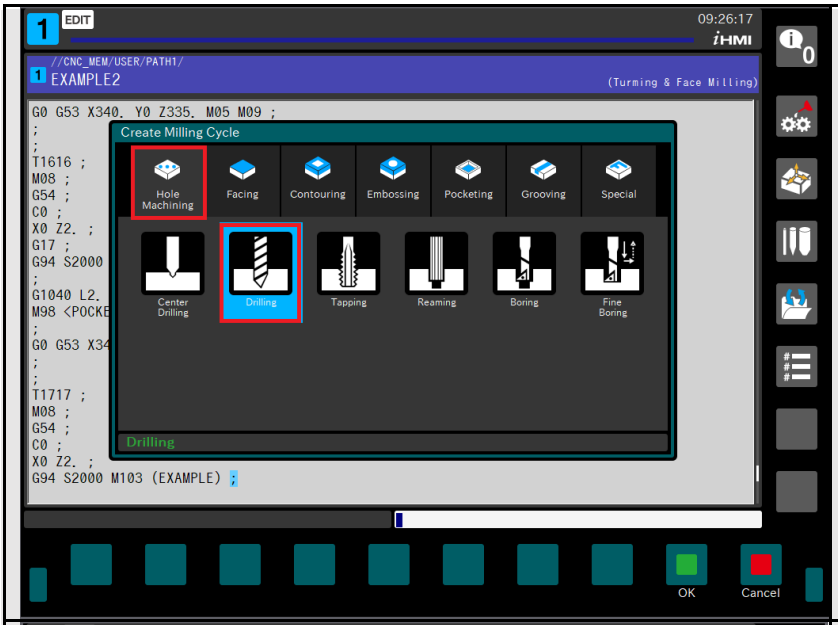


3.7 Process: Drilling on XC face

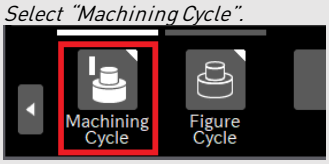


Insert the "Fixed Sentence" for "START MILLING ON FACE C" and edit the program:

1. *Delete the double line G0G53X340.Y0Z335.M5M9;*
2. *Set Tool number T1717*
3. *Spindle speed – S2000*



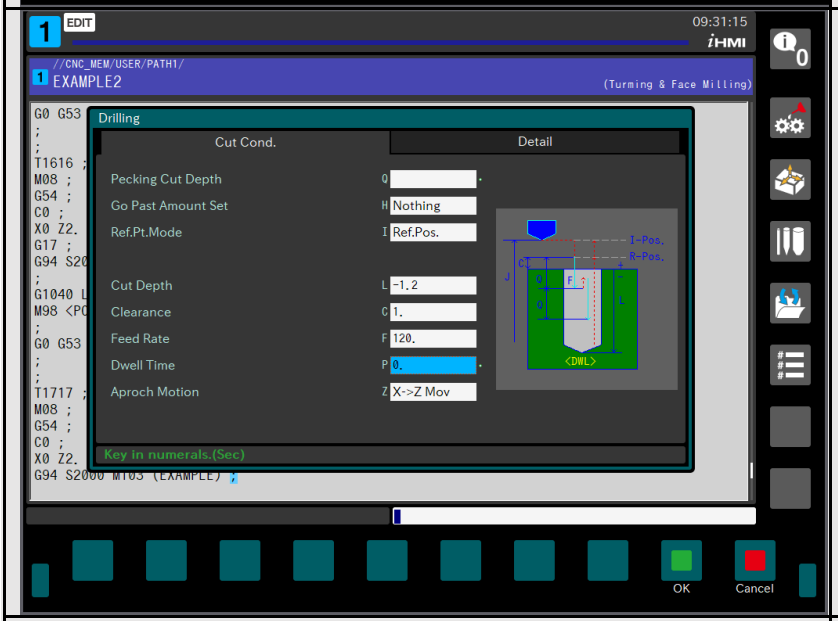
Select "Machining Cycle".



"Milling"

"Hole Machinning" and "Drilling"

Confirm by "OK".

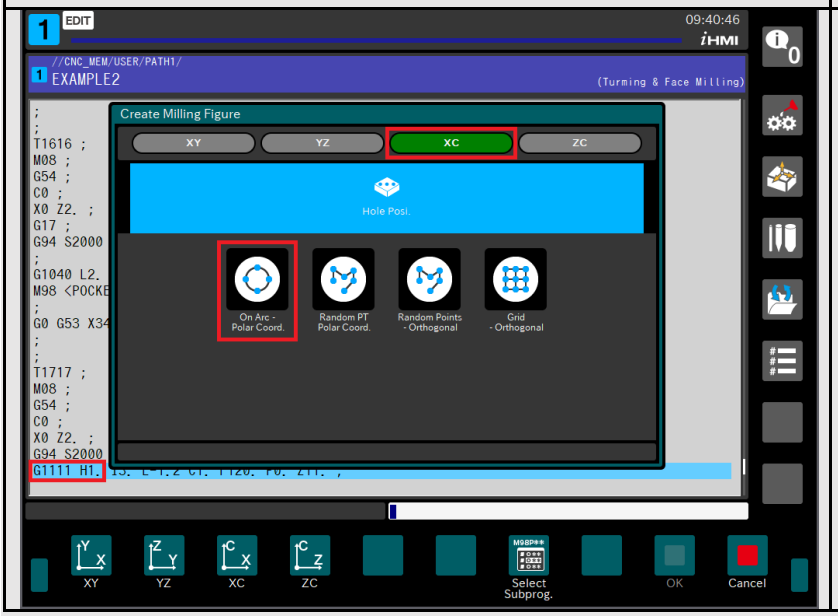


Complete data as shown at the left.

Data in "Detail" are not obligatory.

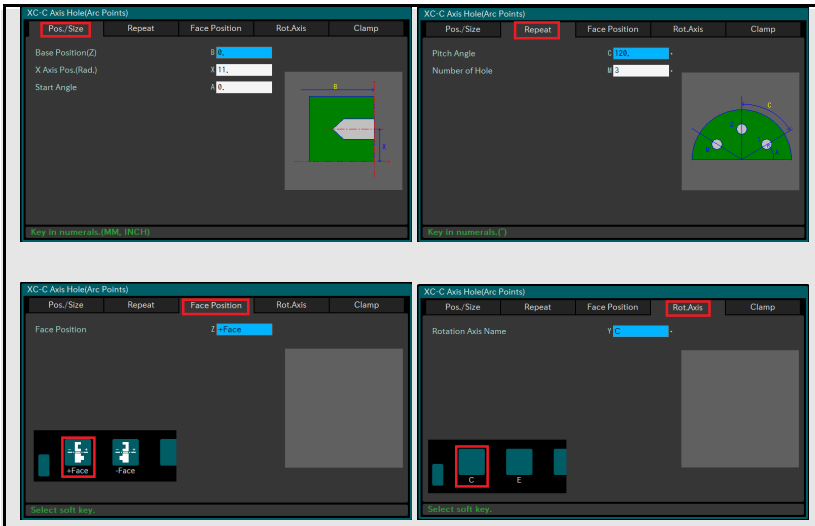
Confirm by "OK".

3.8 Geometry: Drilling on XC plane

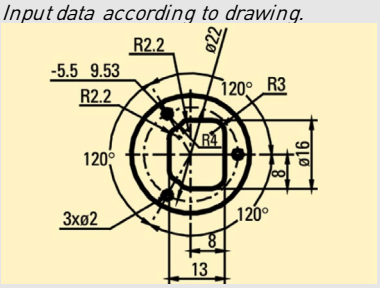


Cycle is inserted in the program, Geometry screen opens automatically.

Select "XC" plane and "On Arc - Polar Coord.".

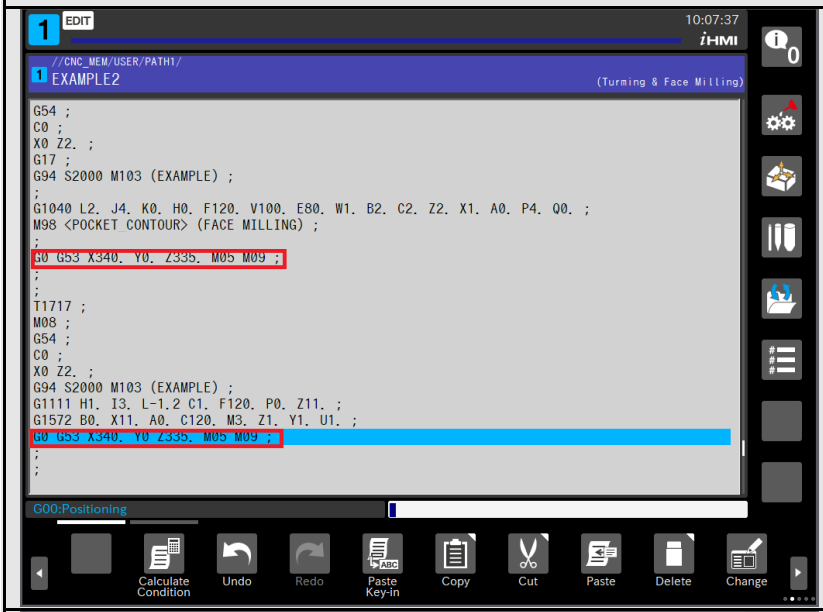


Input data according to drawing.



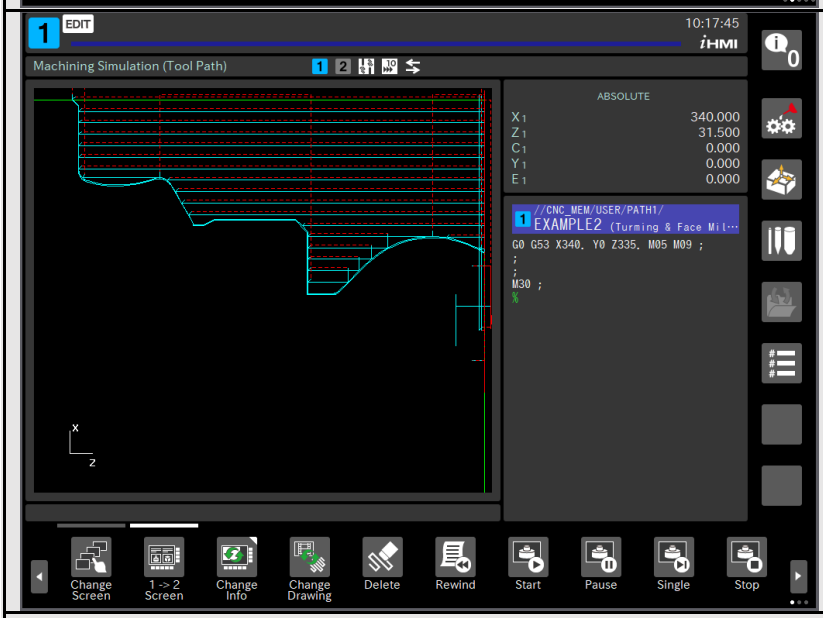
The macine has 2 main spindles (left and right) and 2 spindle axes (C and E). We are working on the left main spindle – C Axis.
"Clamp" means – in Drilling operations can the spindle axis be clamped. The M-command is set in CNC Parameter. In our case this setting is not obligatory.

Confirm by "OK".

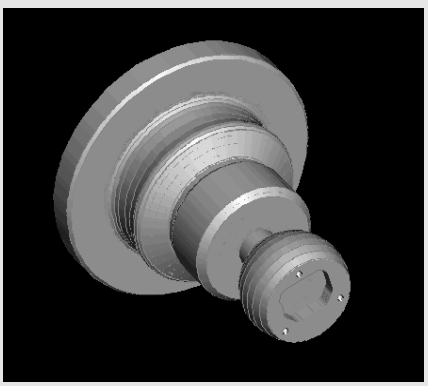


"Copy" the command line to return to Index/Tool Change position and "Paste" before end of program (M30).

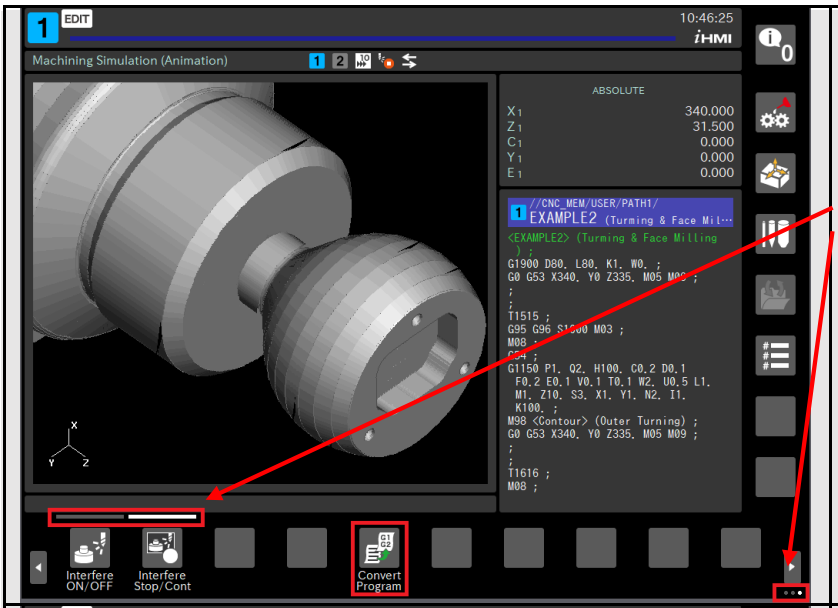
So you can test / simulate the Drilling Cycle.



At the end of program you can enter dedicated Fixed Sentence blocks or modify the "START MILLING ON FACE C" to stop spindle rotation and coolant.
The program is completed.

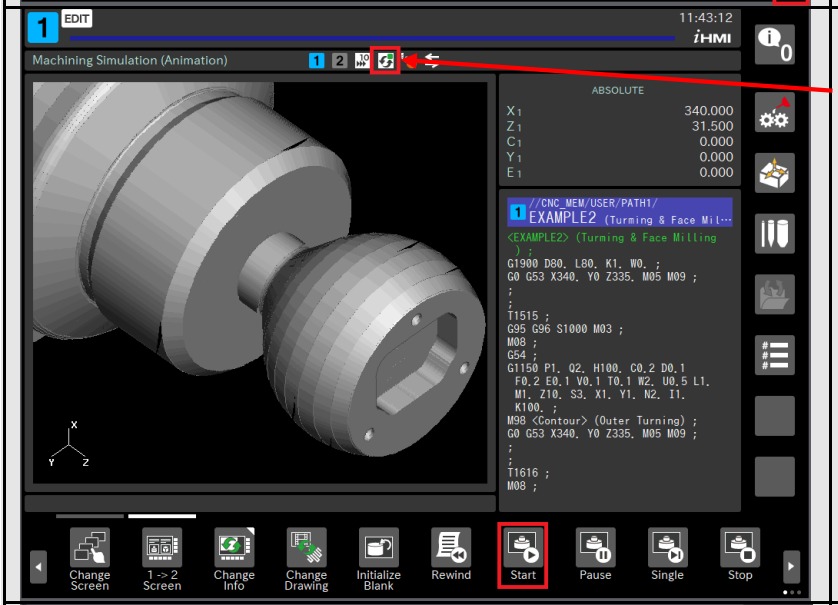


3.9 Conversion of Cycle Program into ISO Program



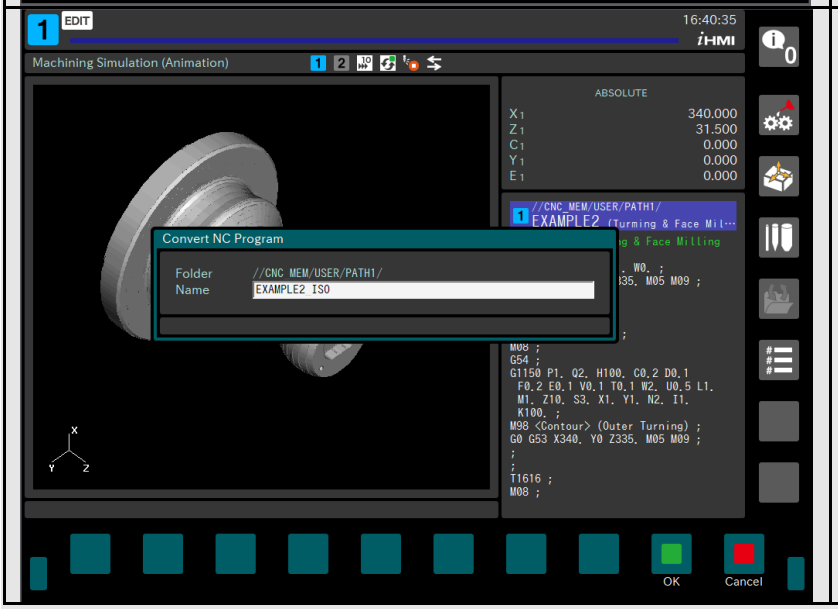
Advantages: In ISO program every step / movement of the machine can be checked. ISO program can be used on different similar machines, having the same configuration (without iHMI and Manual Guide i) and machine commands.

"Convert Program" key is the Second screen (simulation screen) and 3rd horizontal soft key bar.

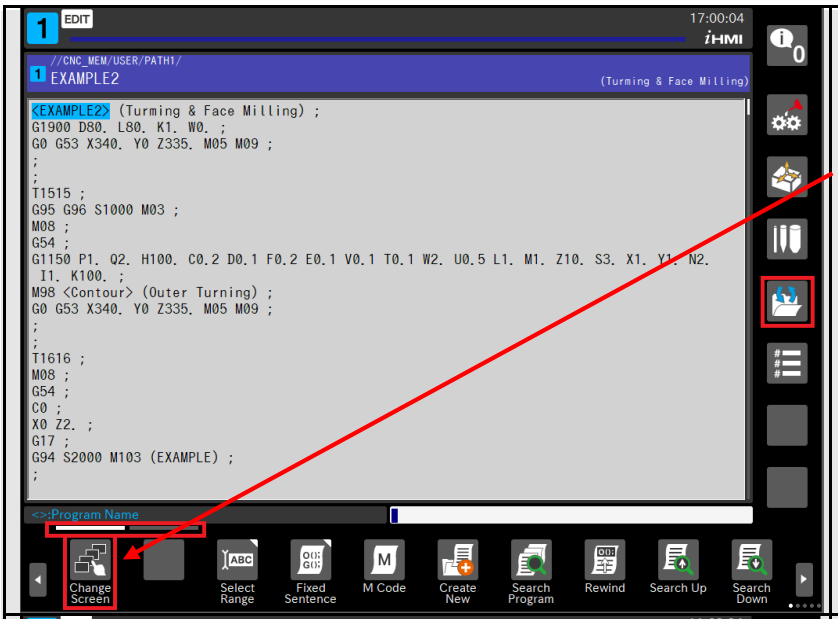


Pushing Convert Program" key the icon on the top indicates – active function.

After selecting "Start" on the simulation screen, the name of the ISO program can be input.



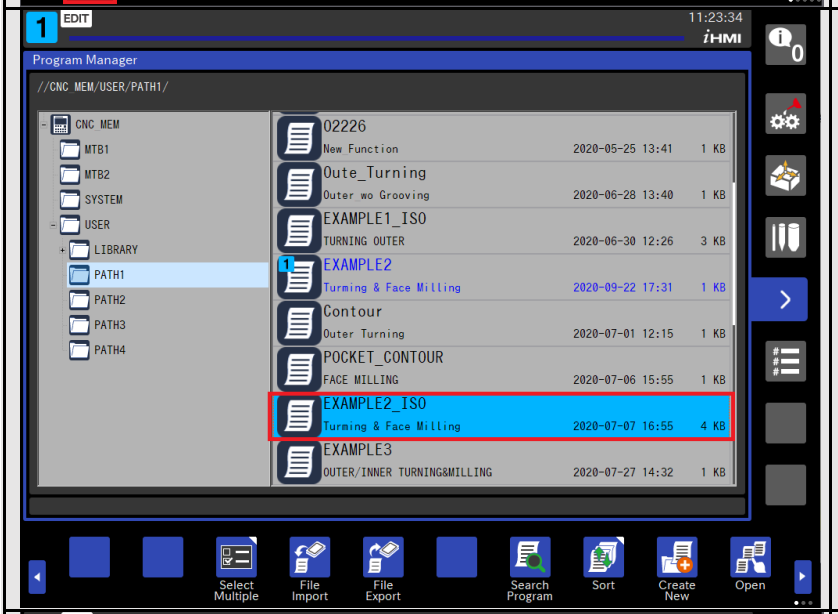
Conversion strts by "OK".



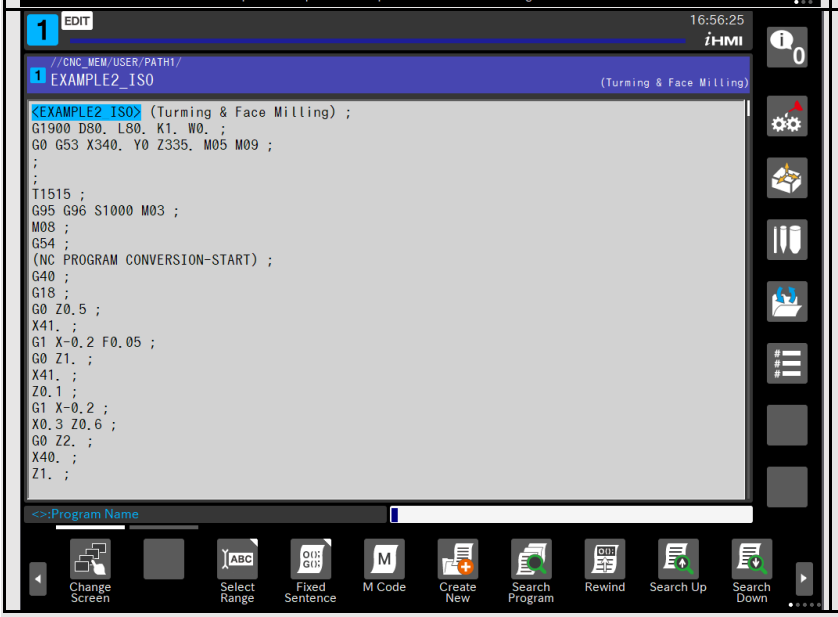
During conversion the simulation is activated, showing the movement of tools.

After finishing, the screen remains on Simulation screen. Go to "Change Screen" to the first screen.

Go to Program memory and open EXAMPLE2_ISO.



Go by cursor to EXAMPLE2_ISO and push "Open".

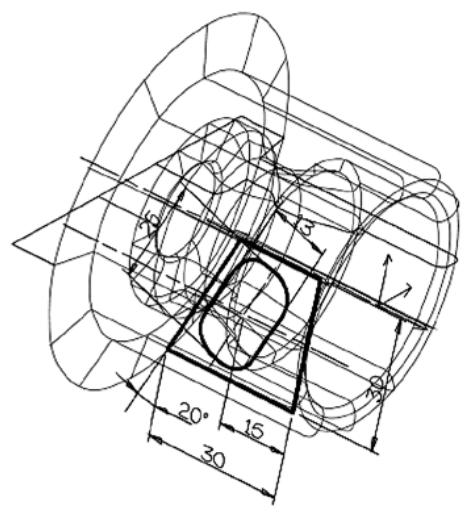


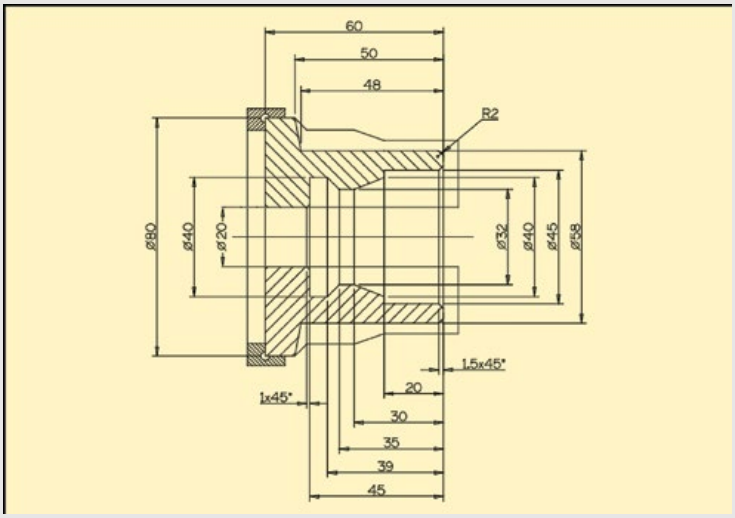
The converted ISO program is using simplest comands, so it can be used on other machines working with ISO code.

4. Example 3

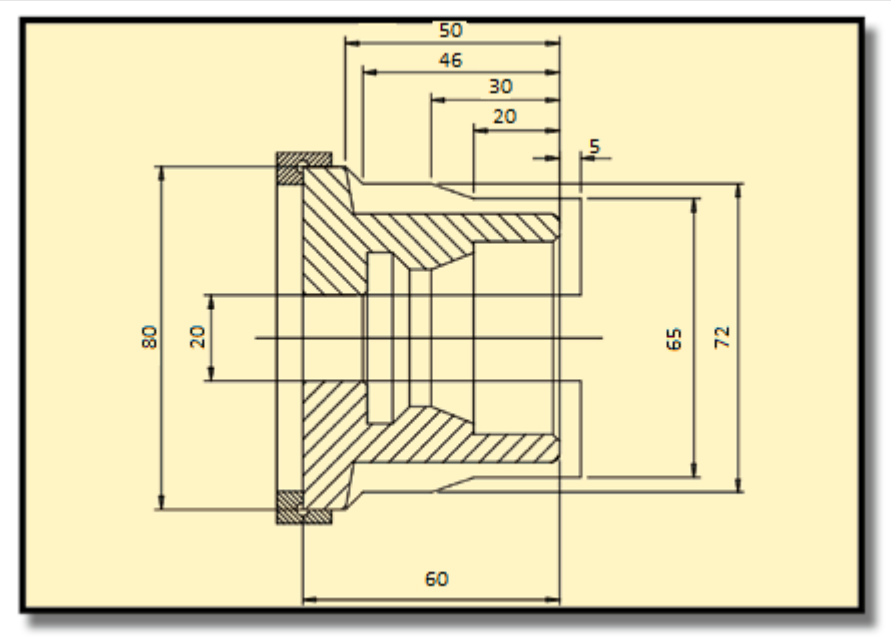
In this example, the blank part is a pre-machined hollow cylinder. A pocket is to be milled out on the lateral surface. The following machining cycles are programmed:

- Facing
- External turning (roughing)
- Internal turning (roughing)
- Contour milling
- Pocket milling

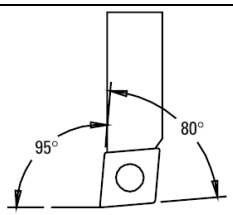
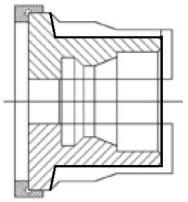
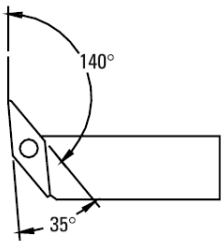
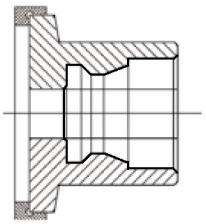
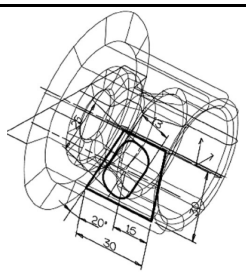


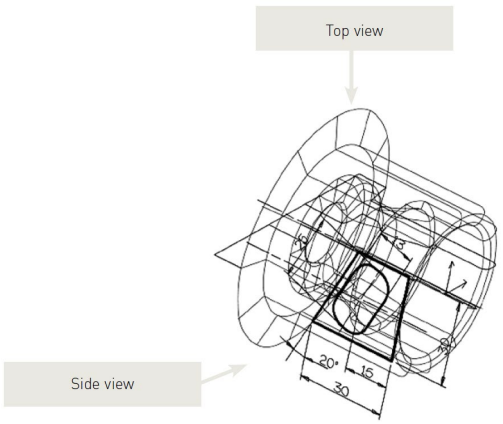


Blank Part Dimensions

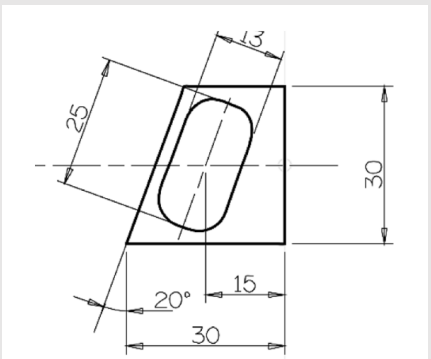


Necessary Tools

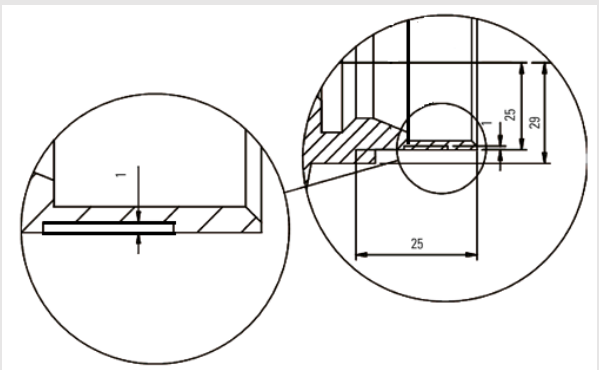
Tool Data		Machining processes	Machining Area
T1818		o Facing and External turning-roughing; Roughing and Finishing	
T1919		o Inner Turning	
T2020		o Milling	



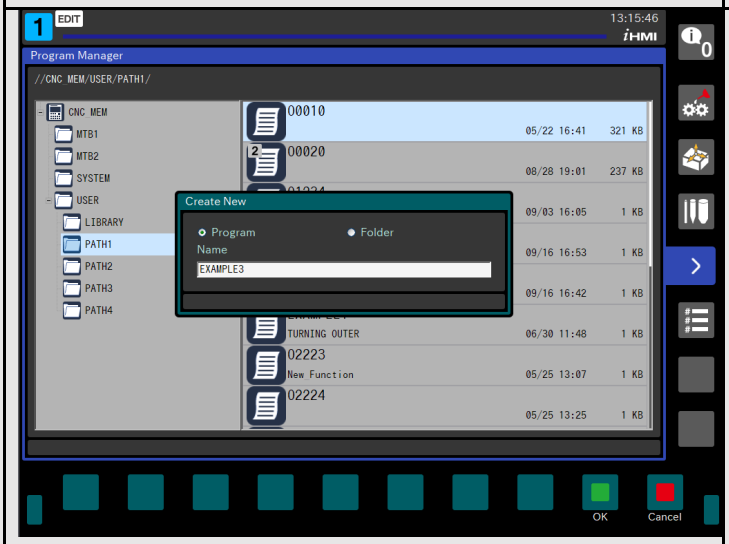
Side View:



Top View:

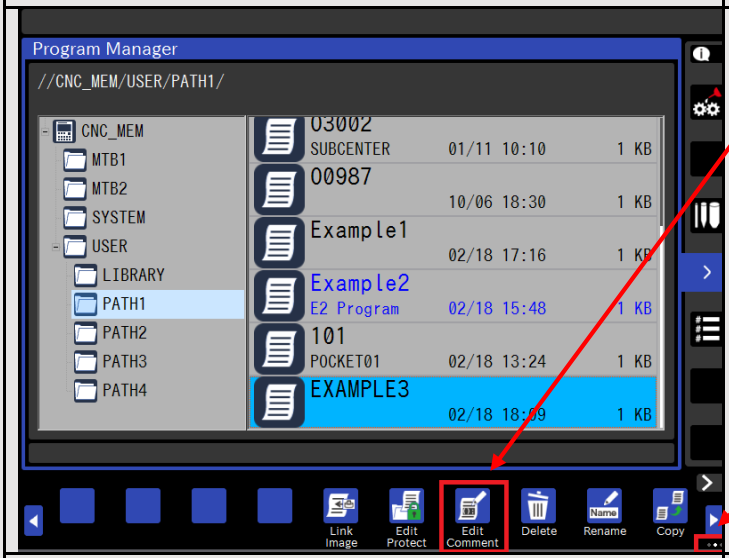


New Program

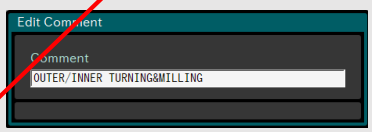


Select EDIT mode and Edit screen. Open Vertical bar and select Program Management.

Create "New Program" named Example3.

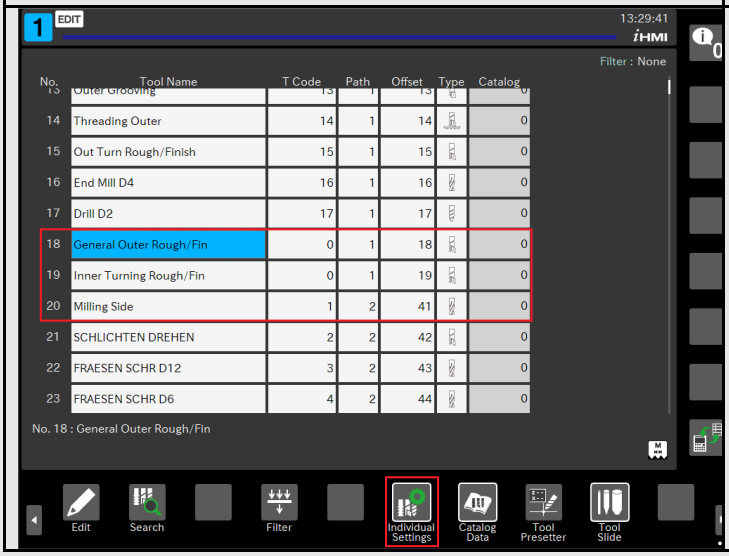


Select "Edit Comment" on the second horizontal bar.

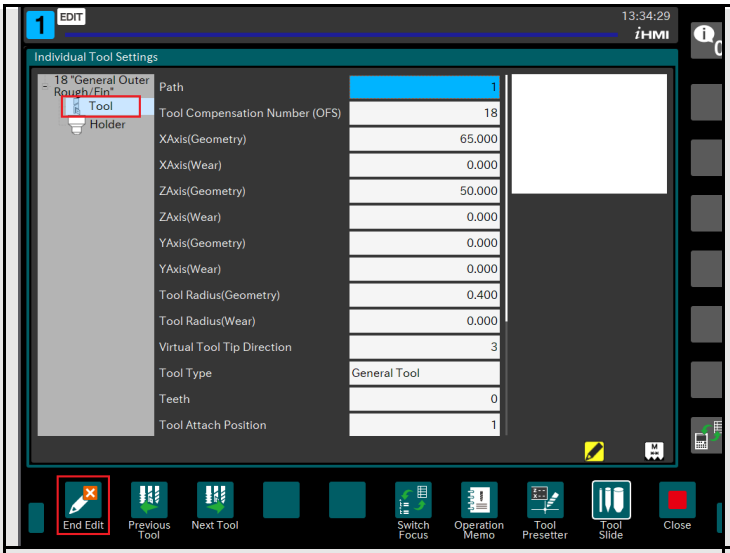


Confirm by „OK".

Tools

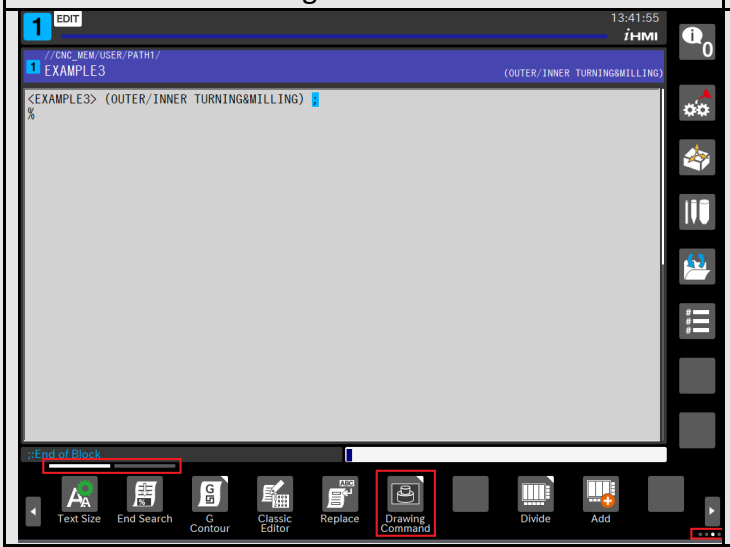


Prepare required tools (T1818, T1919, T2020). Use "Individual Settings".



Activate "Edit" and complete data entry. Here Example for Tool number 18.

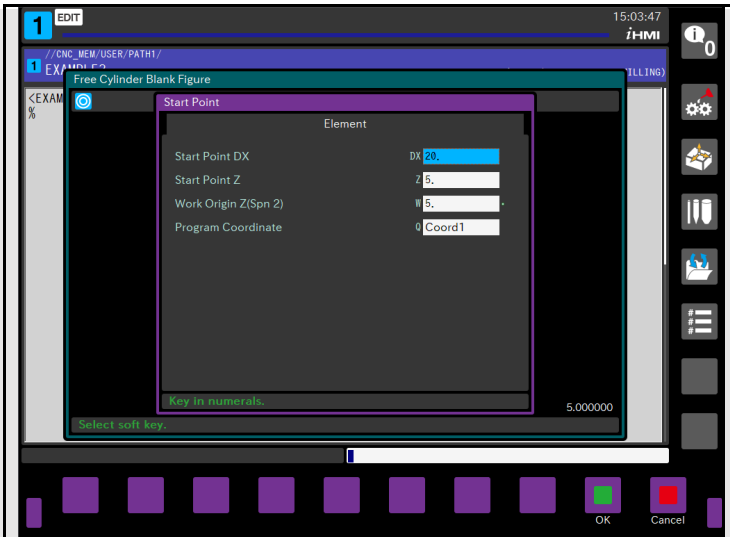
Process: Outer Turning



Open program <Example3> and push "Drawing Command" to prepare entering Blank data.

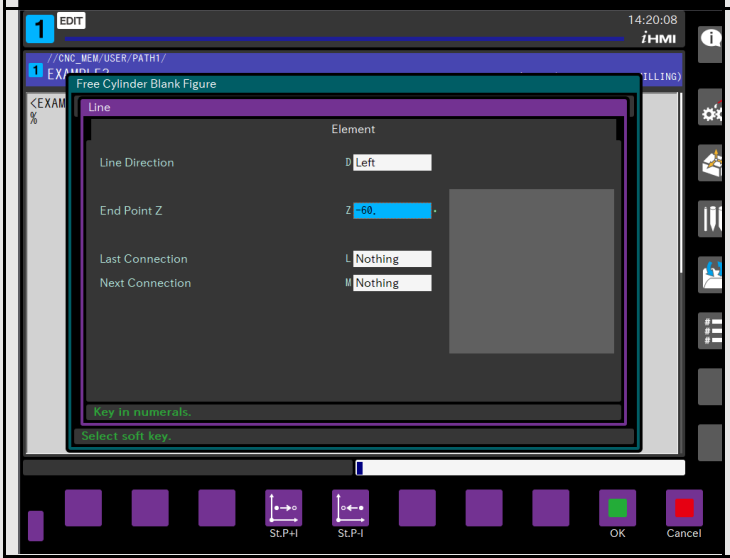
Select "Turning".

This time the form of Blank is specific and has to be described. Select "Free Cylinder". As soon as you select "Free Cylinder" the screen will be changed.



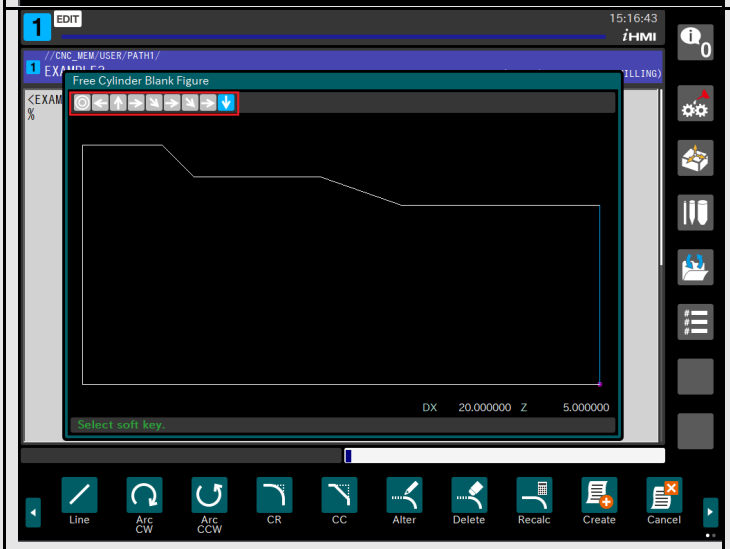
Startpoint for the Blank description will be as shown at the left.

Using 2 spindles, the "Coord 2" of the right spindle is different than "Coord 1". We are working with left spindle only.



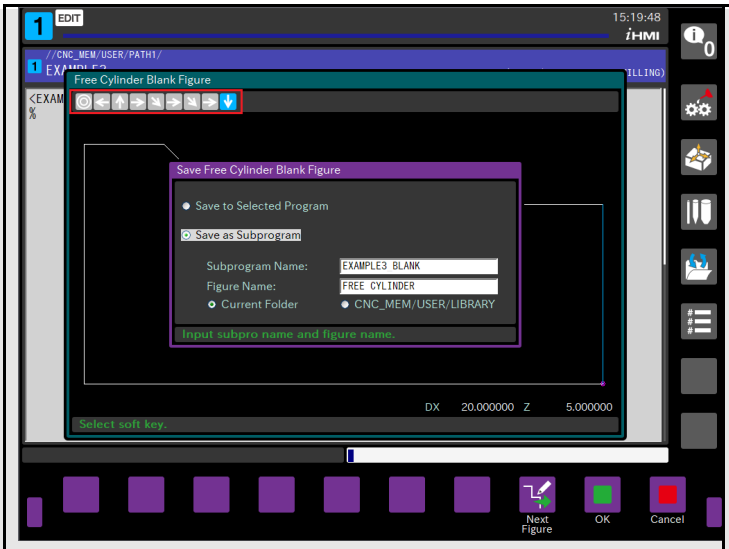
Select "Line" and "Left" to Z-60.0 mm. Confirm by „OK".

The blank part definition entered here is only used for the graphic simulation and has no influence on the tool path.

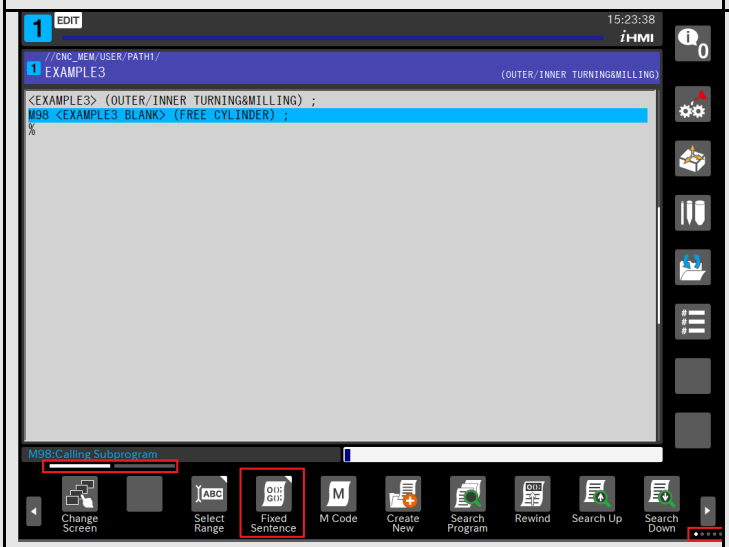


Next element – "Line", "Up" to X80.0 mm., "Right" to Z-50.0 mm., "R-down" to X72.0 mm. Z-46.0 mm., "Right" to Z-30.0 mm., "R-down" to X65.0 mm. and Z-20.0 mm., "Right" to Z5.0 mm., "Down" to X20.0 mm.

Blank workpiece is described. It can be saved in separate sub program. It can be reused at any time.

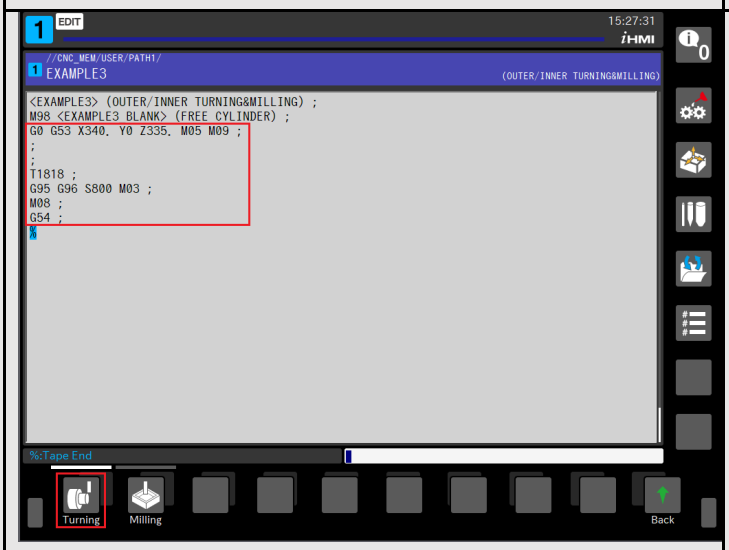


Confirm by "OK".

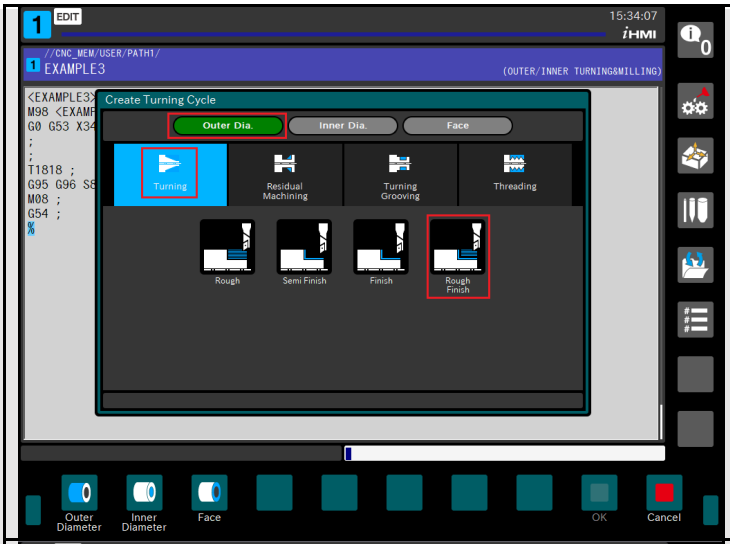


Select on first (Edit) screen and first soft key bar the "Fixed Sentence" and "Turning".

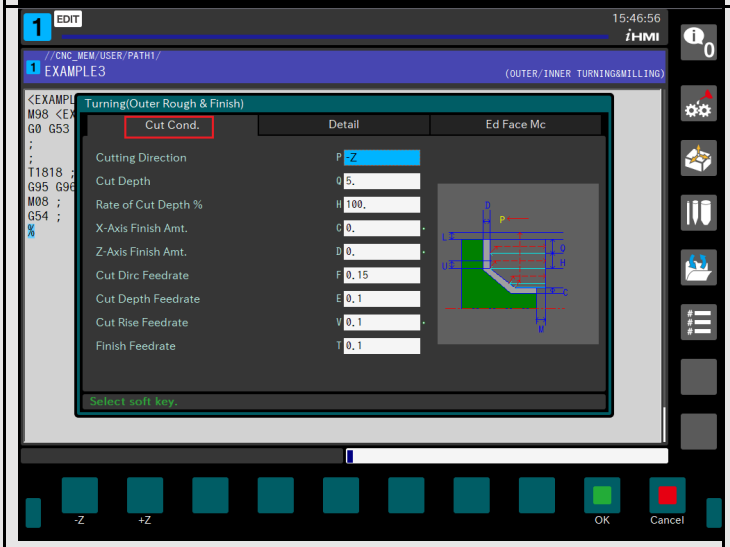
Enter data and edit the program.



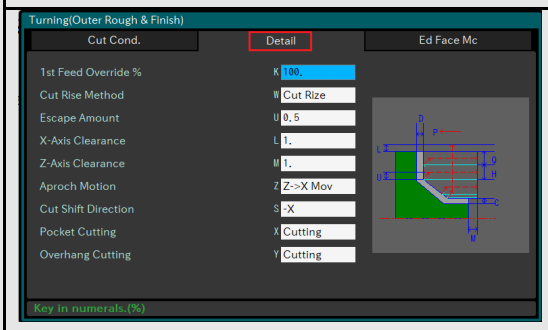
The correct data are shown at the left side.

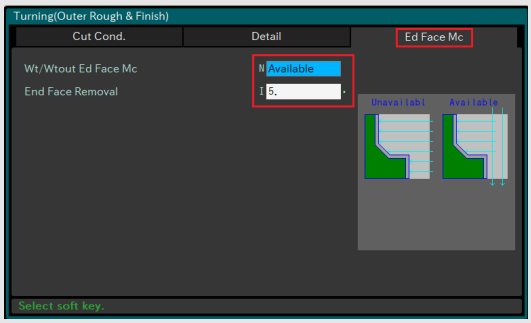


Select "Machining Cycle" on the 3rd horizontal soft key bar, and "Turning", "Rough Finish".



Set "Cut Cond." Data as shown on the left side.

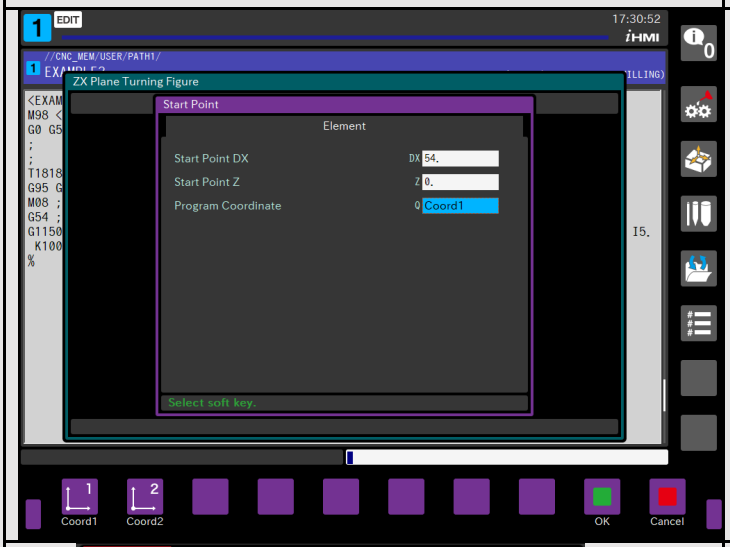




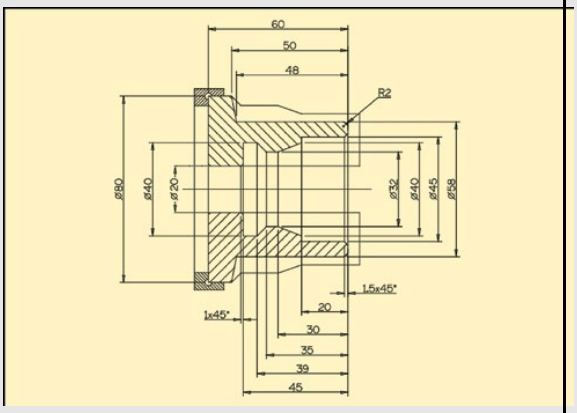
Consider also "Detail" and "Ed Face Mc."

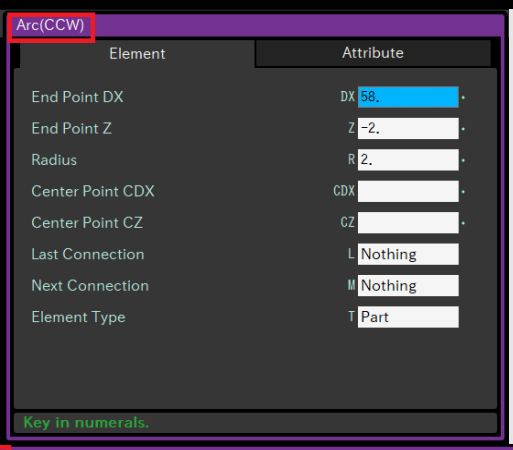
Confirming by "OK" screen changes to start point of contour description for turning.

Geometry: Outer Turning



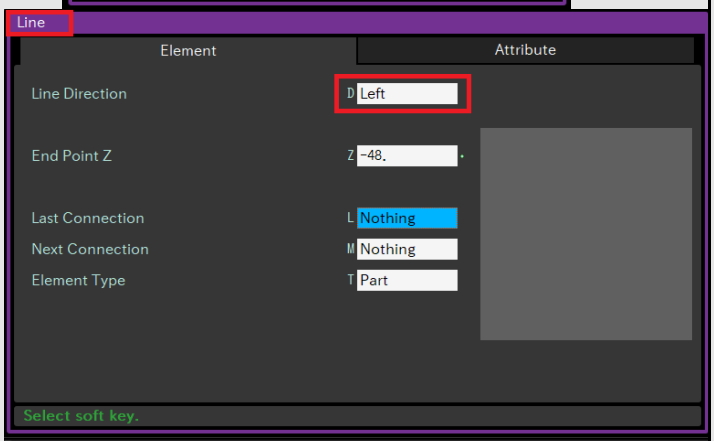
Complete the data as shown left.

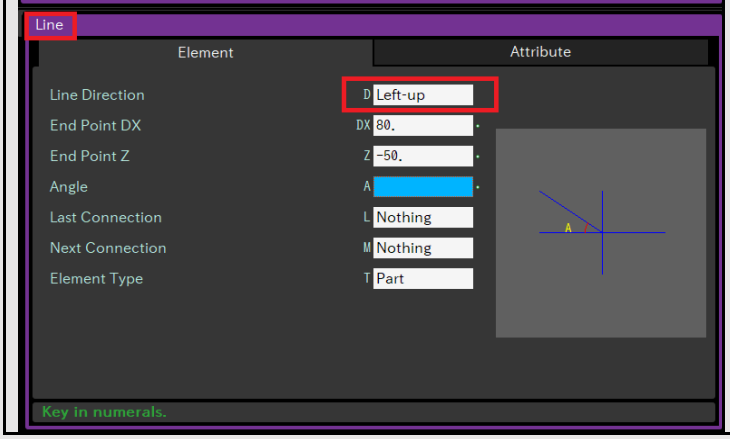


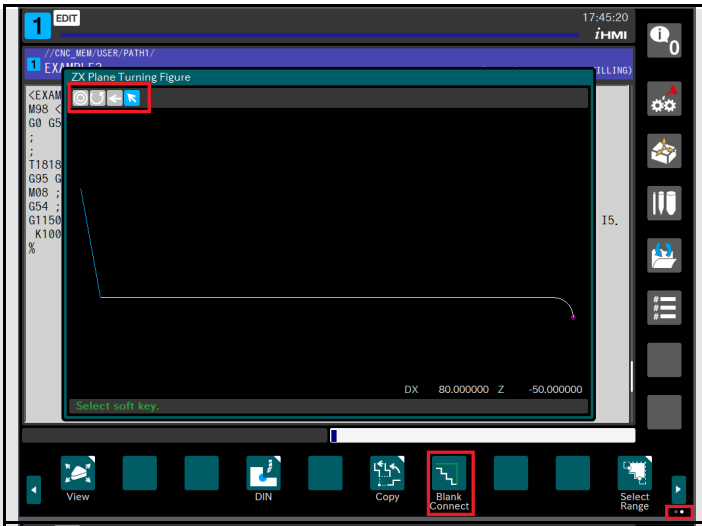


First element is Arc CCW. Contour cannot start with "CR" – Corner Rounding.
Next element is "Line", "Left".
Next – "Line", "Left-up".

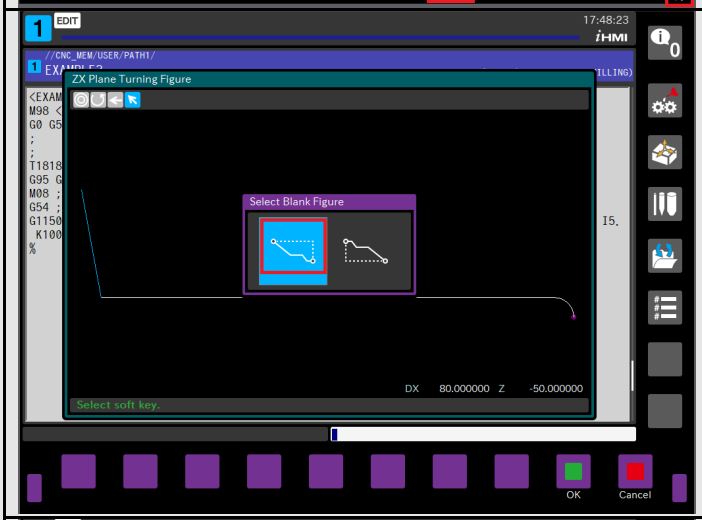
"Attribute" are not obligatory.



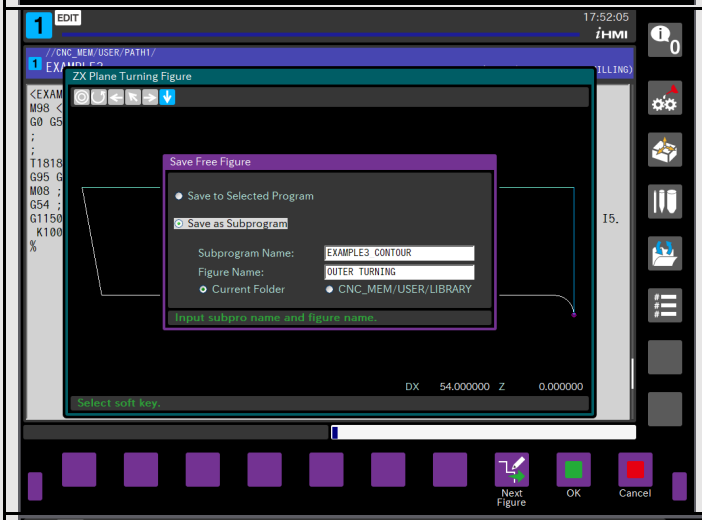




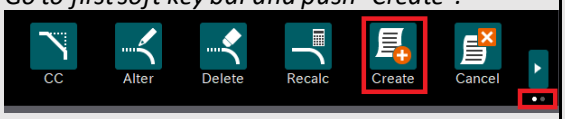
Contour is ready to be closed. On the second horizontal soft key bar is the key "Blank Connect".



Select closing outer contour and push "OK".

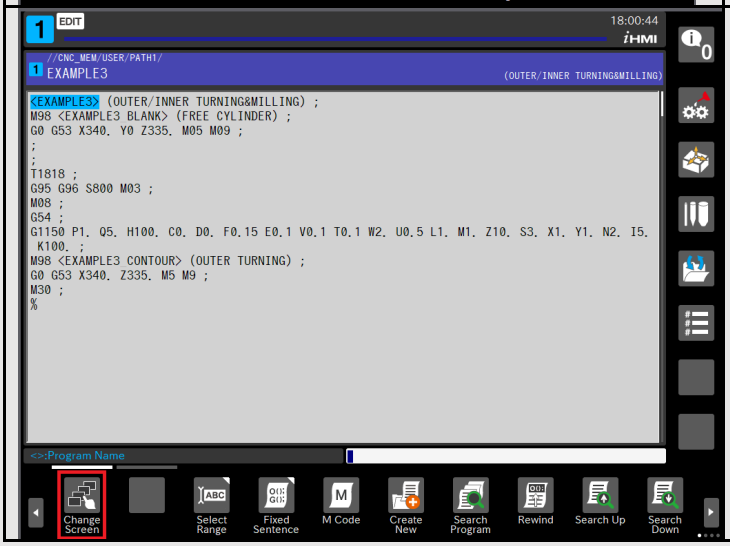


Go to first soft key bar and push "Create".

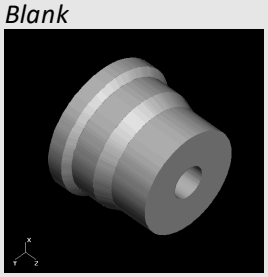


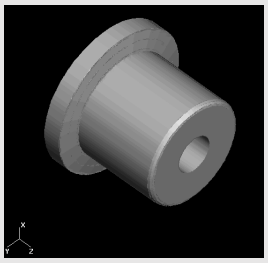
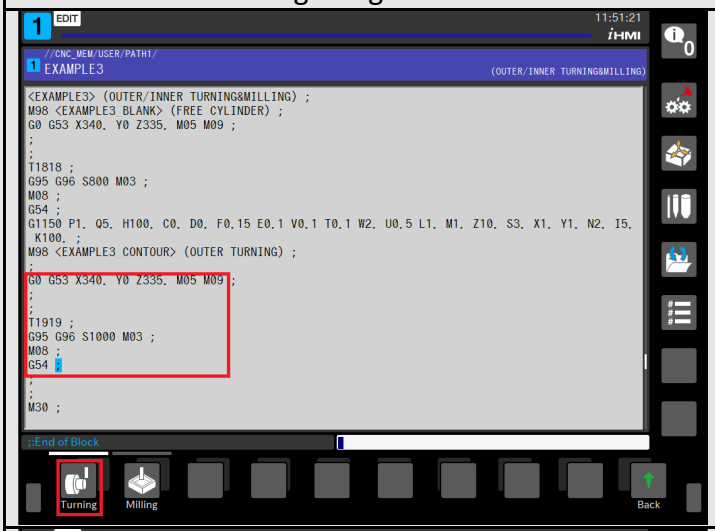
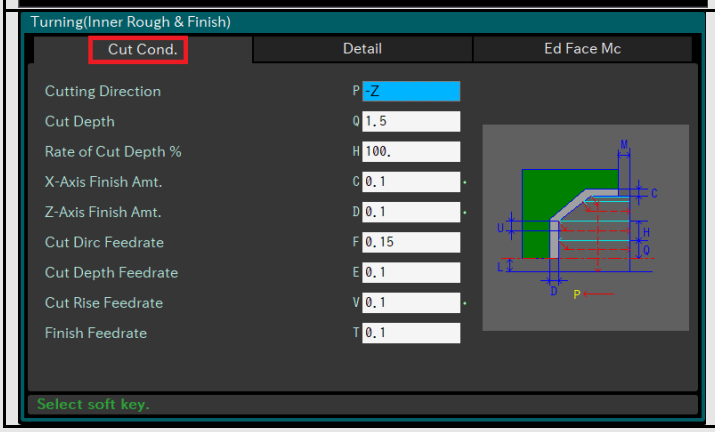
„Save as Subprogram", „EXAMPLE3_CONTOUR", comment „OUTER TURNING".

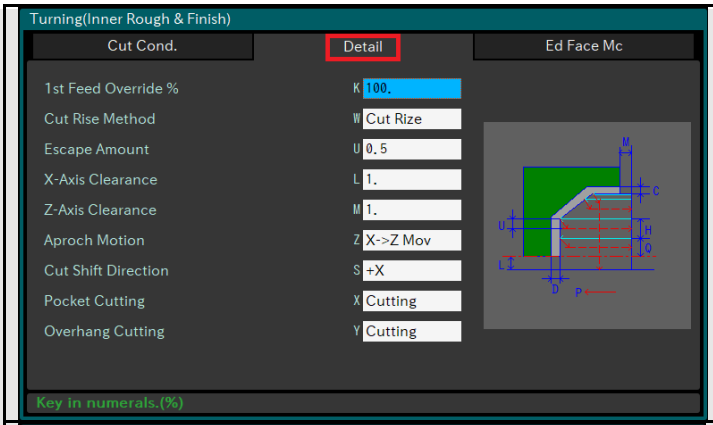
Escape to Index/Tool Change point and enter M30.
Test/simulate the program.



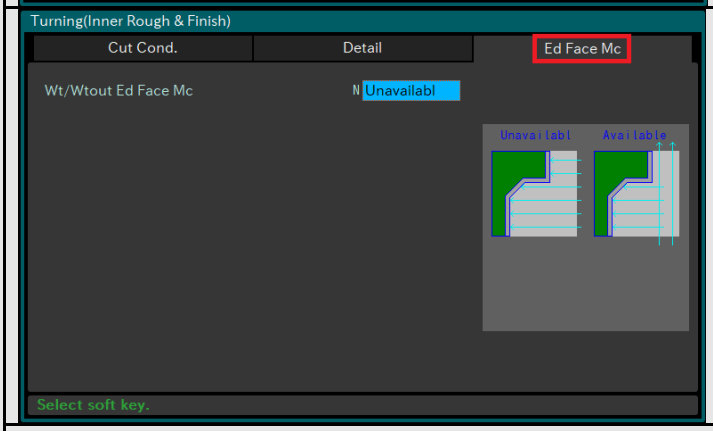
Blank



	Finished 
Process: Inner Turning Rough and Finish	
	*Insert the "Fixed Sentence", "Turning" and modify the program.*
	*Select "Machining Cycle", "Turning", than "Inner Dia." and "Rough Finish".* *Screen is automatically switching.*
	*Complete data entry as shown at the left.*



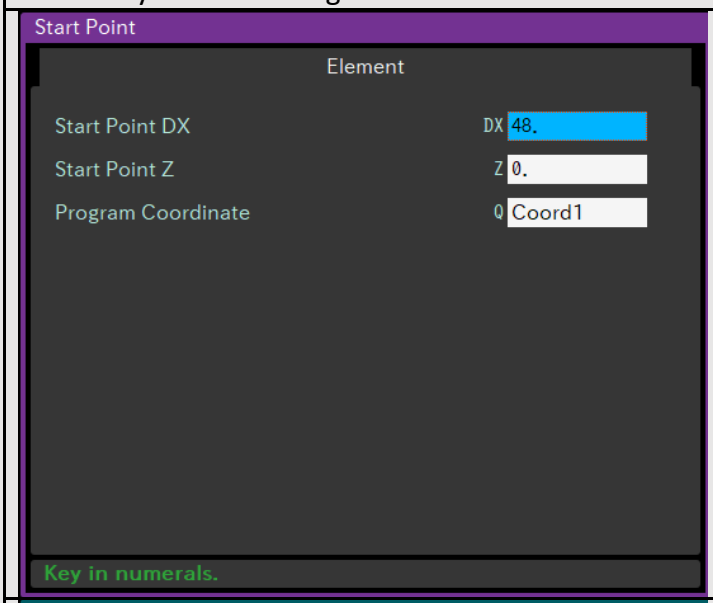
"Detail" should be completed, too.



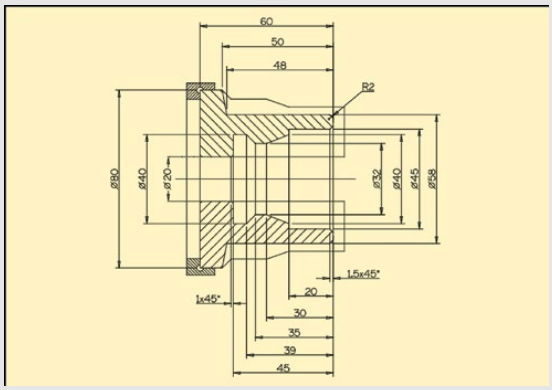
"Ed Face Mc" – End Face Machining was done at the beginning of Outer Turning cycle. For the Inner Turning it is not necessary.

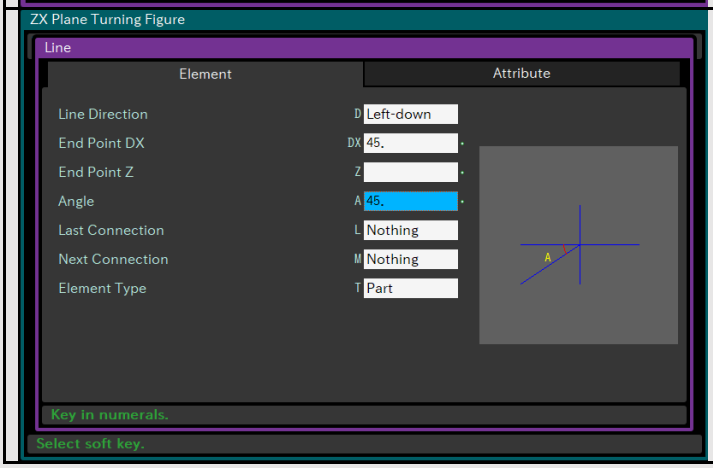
Confirm by "OK".

Geometry: Inner Turning Contour



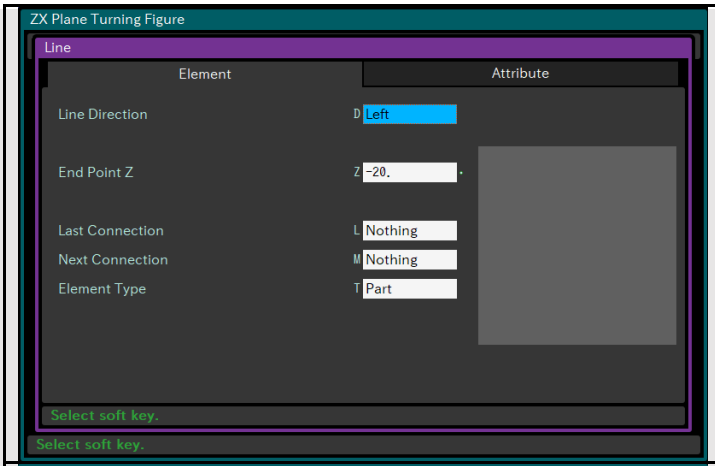
*Start point DX=45.+1.5*2=48. mm*



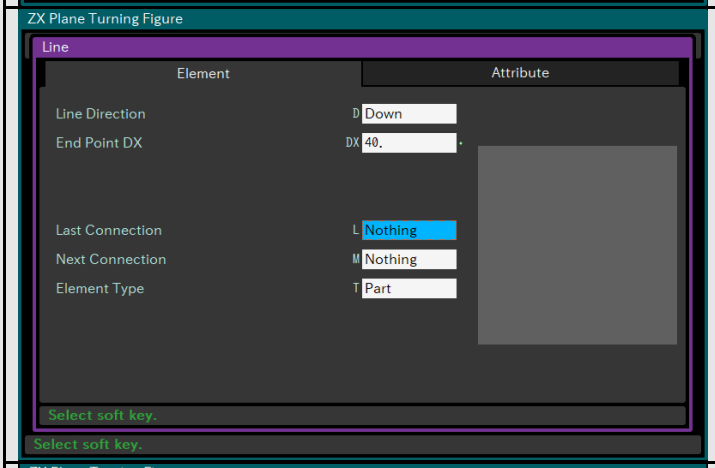


First element is "Line", "Left-down", defined by End Point DX and Angle 45 deg. "Attribute" are not obligatory for the complete Turning Figure.

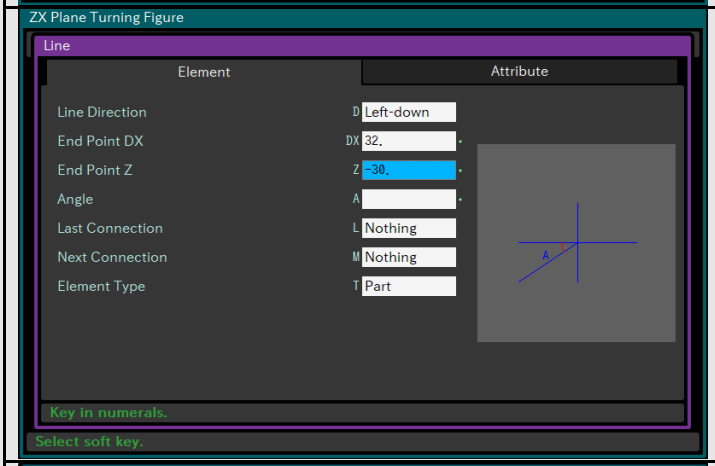
Corner Chamfer "CR" cannot be selected as the first element. Alternatively you can select a "Line", "Down" to DX 45 and fit "CR" 1.5 between this and next elements.



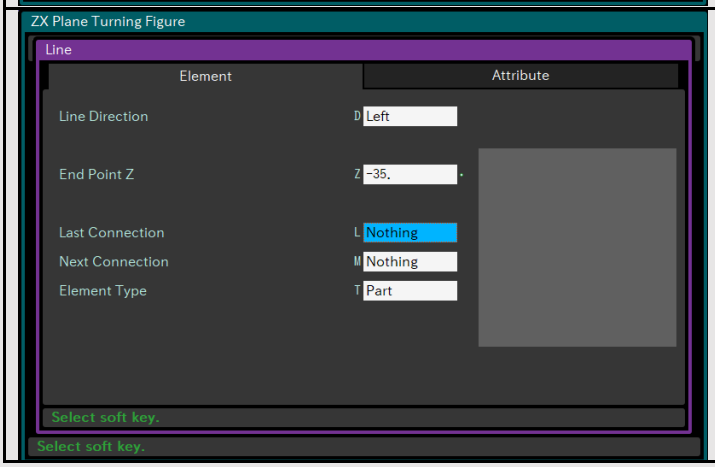
Next element is “Line”, “Left”.



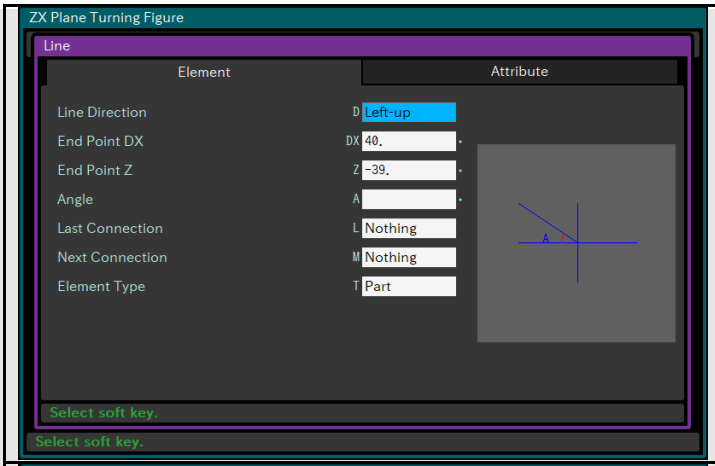
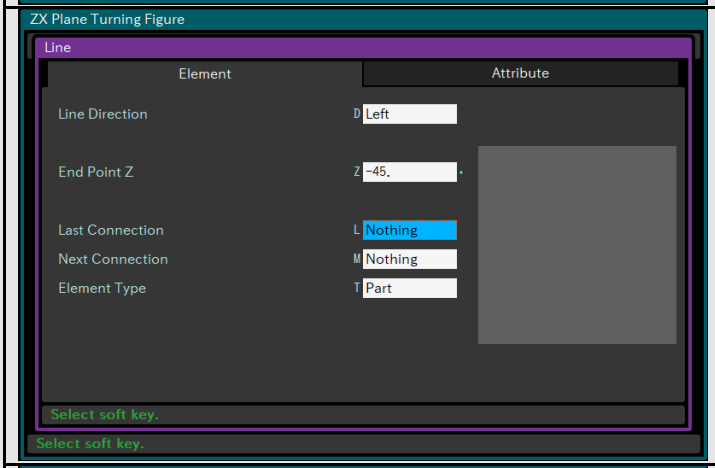
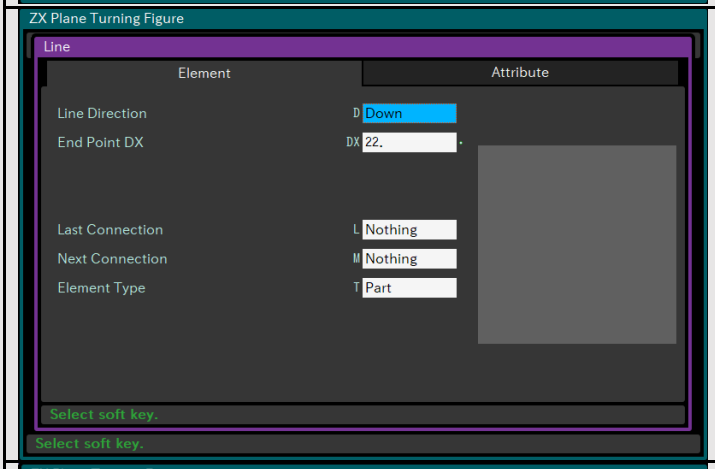
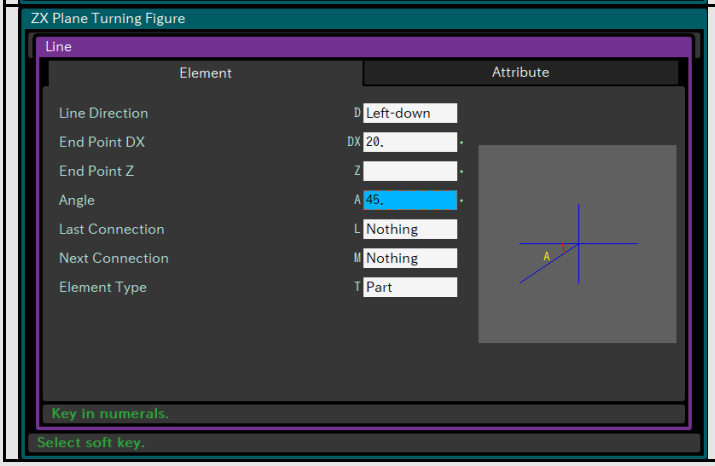
Next – “Line”, “Down”.

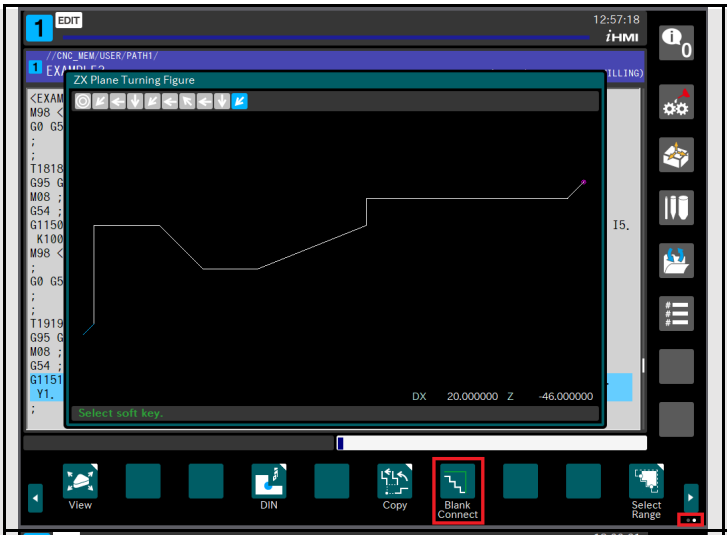


Next ”Left-down” element can be defined by the endpoint according to the drawing.



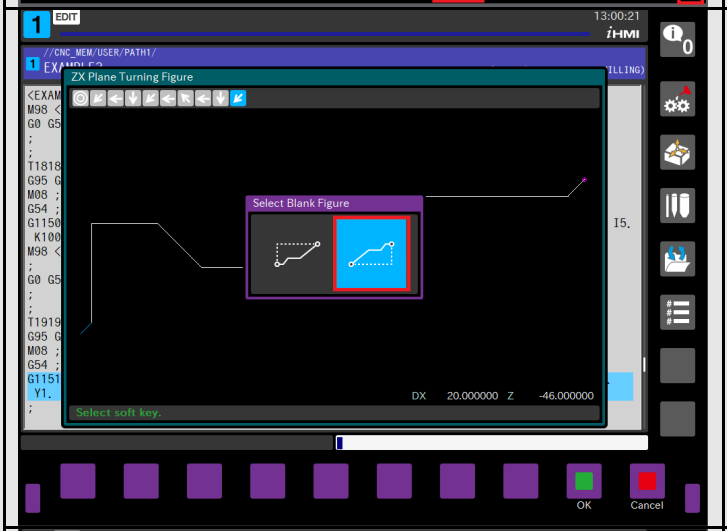
Next line “Left” element goes to Z-35 mm.

	*Next line "Left-up" is determined by endpoint according to the drawing, too.*
	*Next line "Left" goes to Z-45. mm.*
	*Next "Down" line goes to DX22. mm.*
	*Next chamfer has to be defined as a "Line", "Left-down" by 45 deg.*

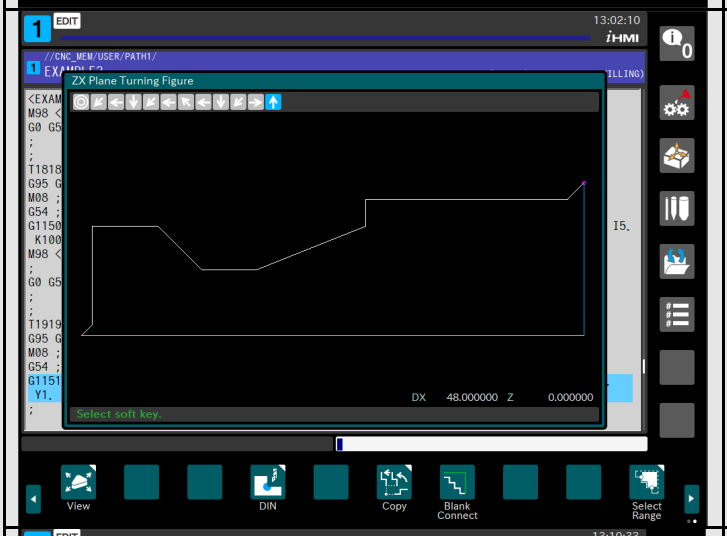


Workpiece contour is completed. Now Blank lines have to be determined.

Go to the second horizontal soft key bar and select "Blank Connect".

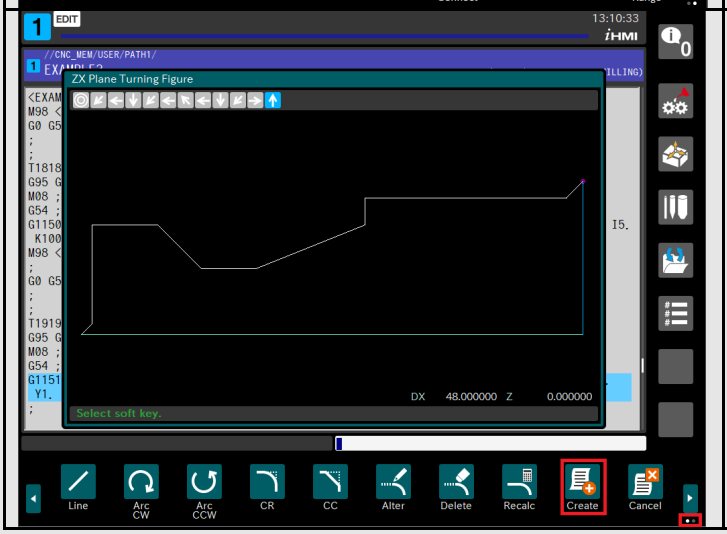


Select as shown and confirm by "OK".

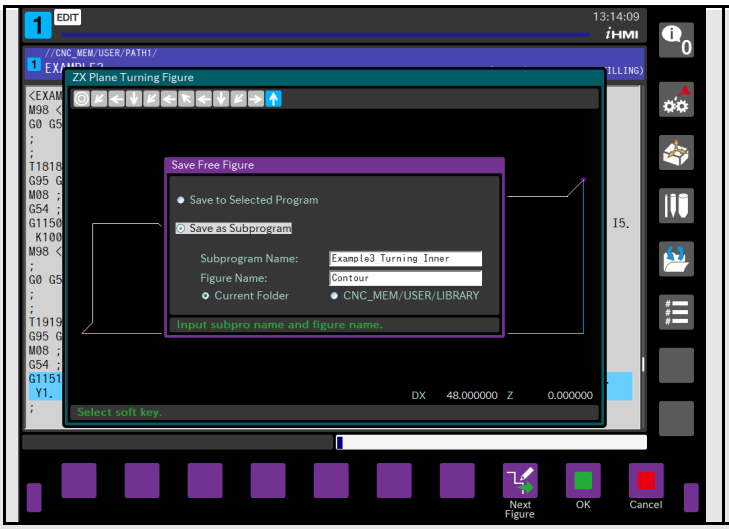


Cutting area is completed by Blank lines.

Register a subprogram for the inner turning contour.



Go to first horizontal soft key bar and select "Create".



Complete as shown at the left. Confirm by "OK".

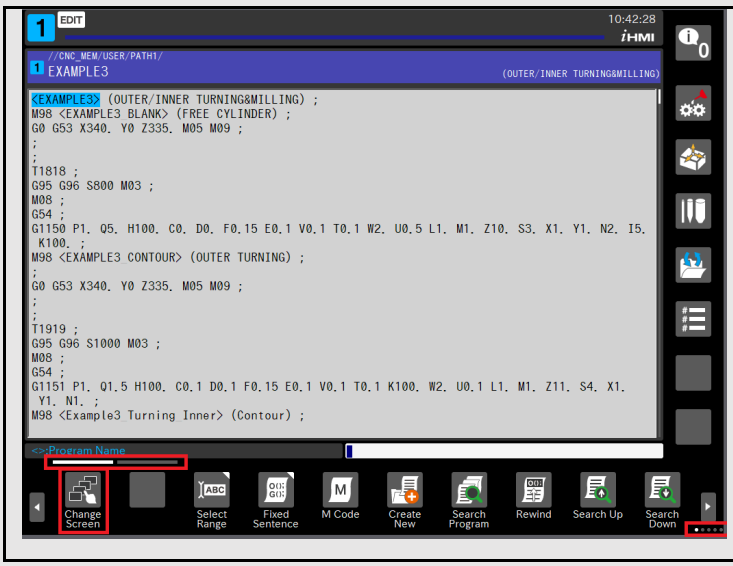
After inserting the geometry of inner contour, insert "Fixed Sentence" to go to index/tool change point and stop coolant and spindle, edit the program and test the program.

4.1 Graphical Interference Check

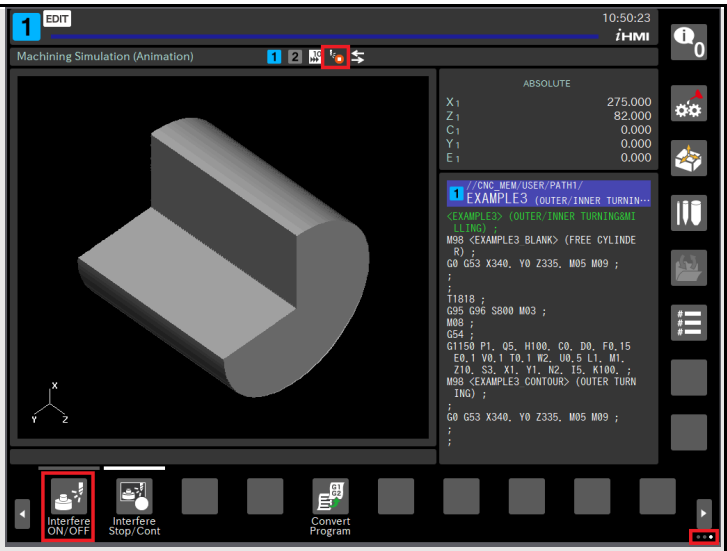
iHMI Cycle Programming has the possibility to check Interference between Work Piece and Tool in Rapid Traverse rate. The Interference Check is based on Graphic 3D simulation. The Execution can be stopped as soon as Collision is encountered or just mark the collision spot and continue simulation. As the Interference Check is made on Graphical basis in 3D simulation only, it depends on the resolution of the simulation display. If a touching between Tool and Workpiece in extend of 0.1 mm, for example, happens on a turning bar of 1000 mm, the Collision may not be indicated. Zooming In the process can show the interference area.

The reasons for Interference could be incorrect programming (edit the program) or tool description.

Please keep in mind that the Tool geometry per default depends on data fixed in CNC Parameter. The random description of Tool Geometry will be considered in the next chapter.

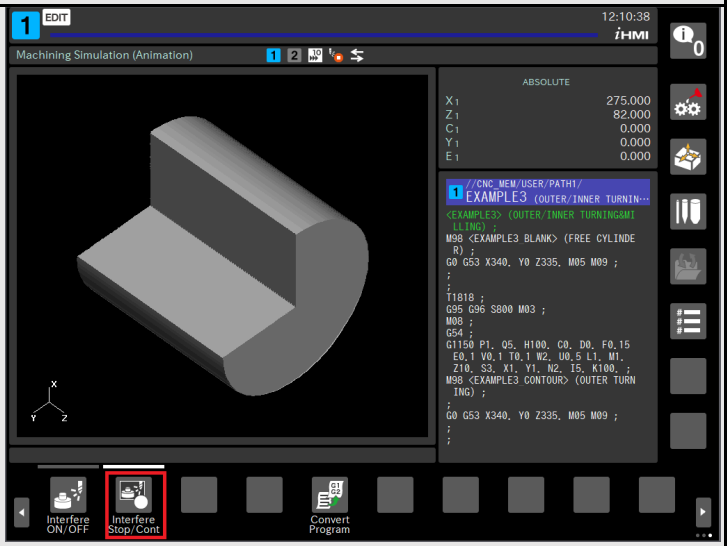


Stay on first screen – Editing. On first horizontal soft key bar push the "Change Screen" to go to second – Simulation screen.



On the third horizontal soft key bar you can activate the collision check by "Interference ON/OFF".

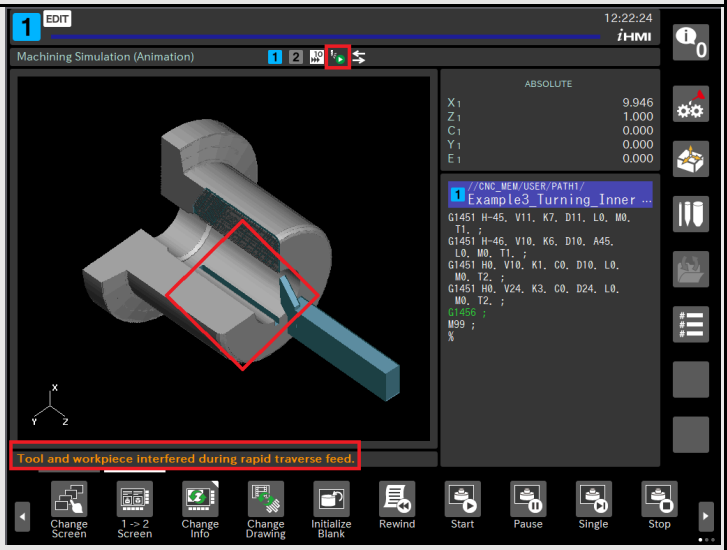
On the top you can find the indication of the activated function.



Using the key "Interfere Stop/Cont" you can toggle between Interference check with stopping the simulation and check with marking the interference area, but continuing the simulation.

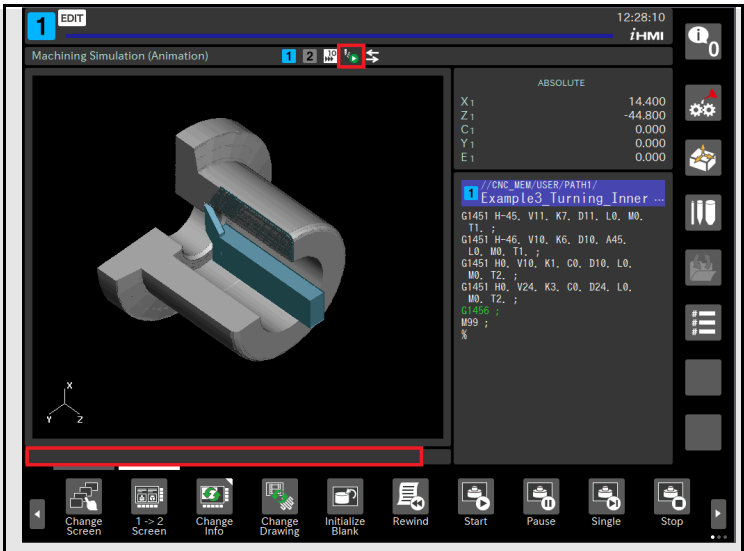
- Stop simulation when Interference

- Mark interference area but continue simulation



On the left you can see how the interference is indicated when the function is selected to continue after marking the interference.

Please keep in mind, that area of interference is well marked, but as soon as it is overcut, the indication of interference is deleted.

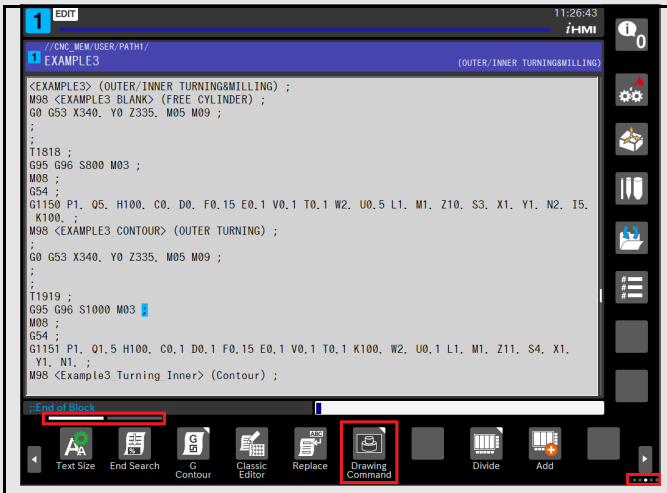


Interference indication disappears as soon as interference area is overcut.

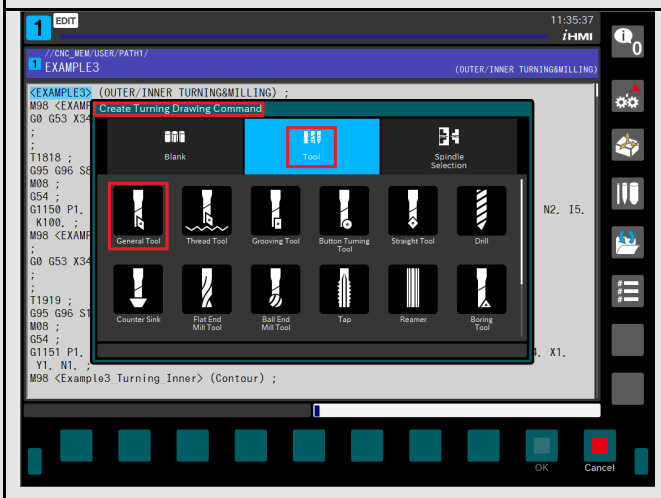
4.2 Geometrical Tool Definition

Tool animation on 3D graph is based on default tool dimensions fixed in CNC Parameter 27350 ~ 27386. If the workpiece sizes vary is recommendable to describe the tool geometry accordingly. Setting CNC Parameter 27310, bit 6 (TLD) to 1, the Tool Geometry definition for all tool types appears as a Tab on the "Drawing Command" screen 

.



On the firs, edit screen and the third horizontal soft key bar, the "Drawing Command" soft key is situated.

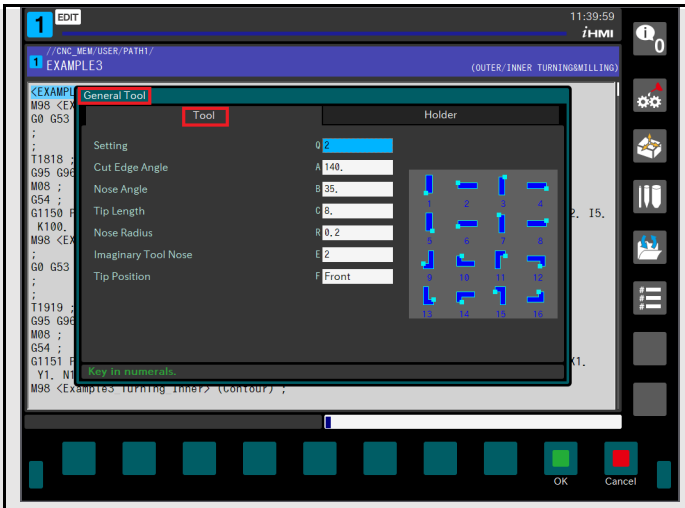


Selecting Turning 

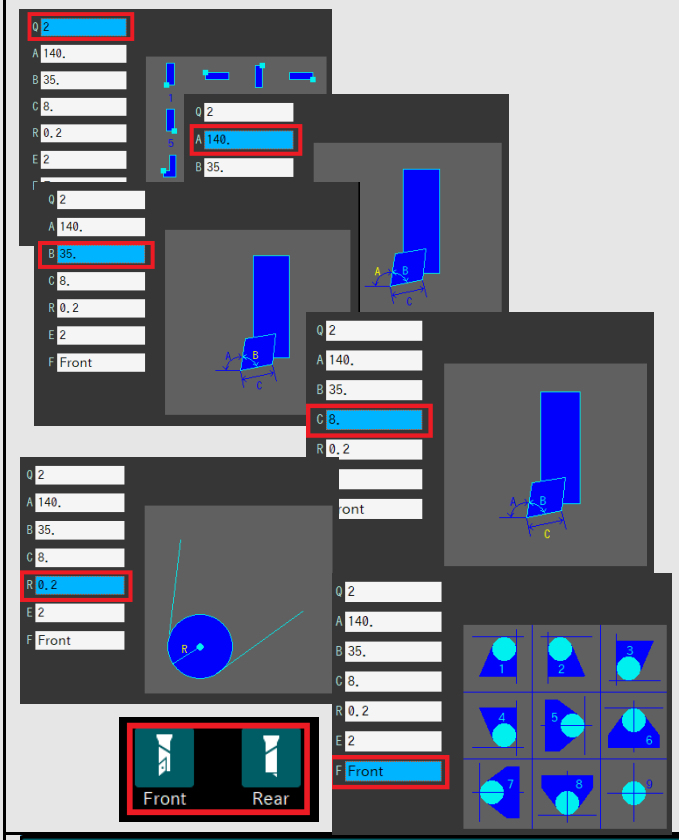
or Milling 

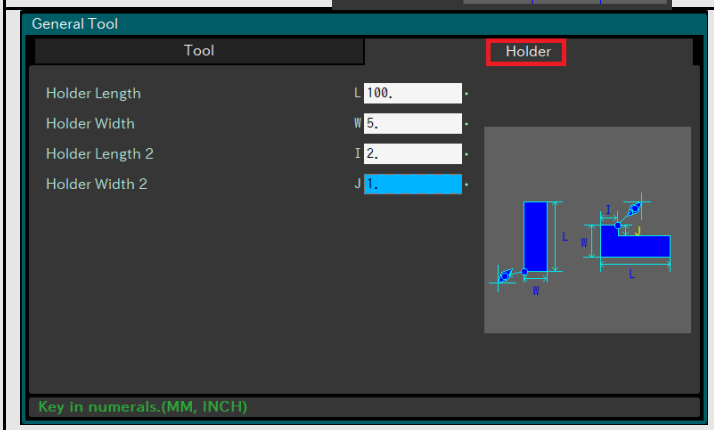
the correspondent tools can be described.

Go to "Tool" and "General Tool". As soon as "General Tool" is selected the screen changes.



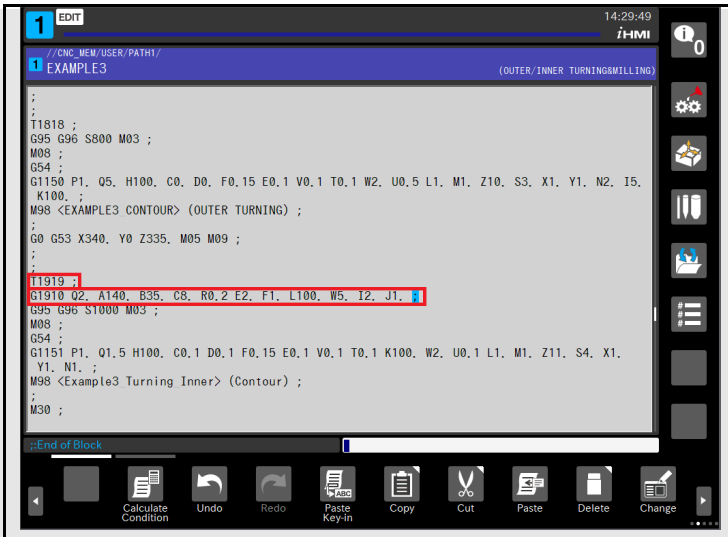
A number of tool dimensions can be determined in those tabs.



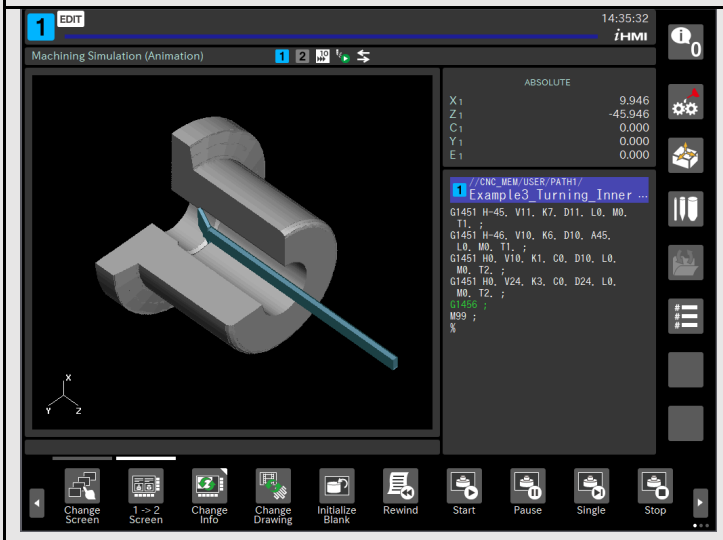


In a similar way, the geometry of the tool holder can be described.

Depending on the cursor position in the program the specific Tool geometry description for the tool animation can be inserted for every process.

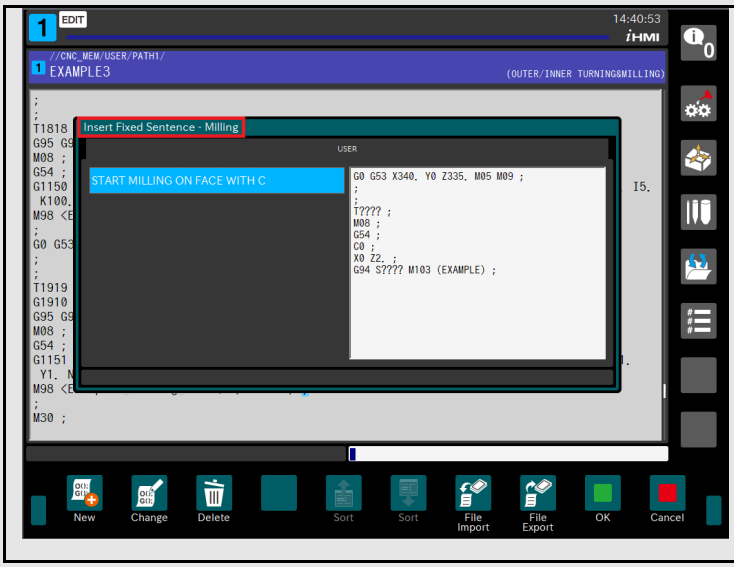


Go by cursor to T1919 in the program and insert by "OK" the geometry description of Inner cutting tool.

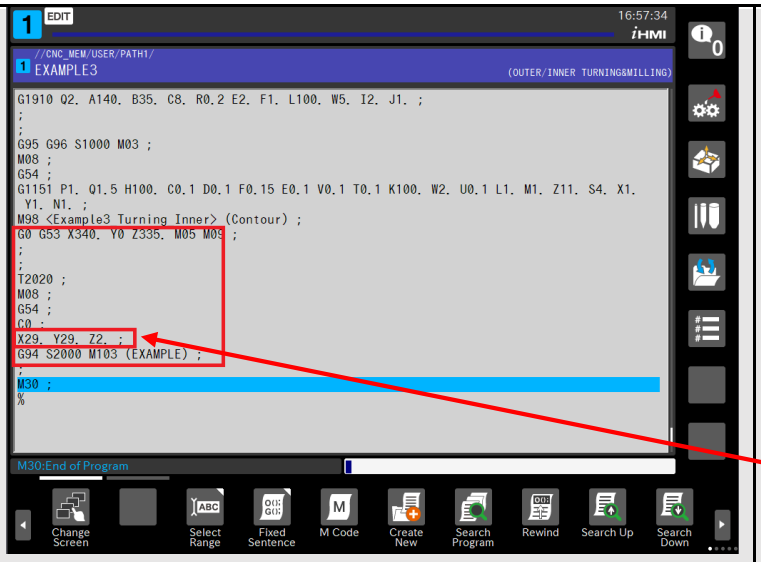


Now there is no interference between tool and work piece any more.

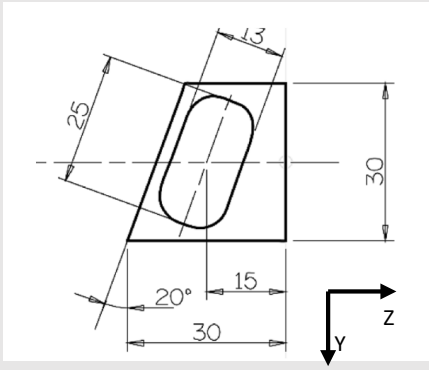
4.3 Milling process



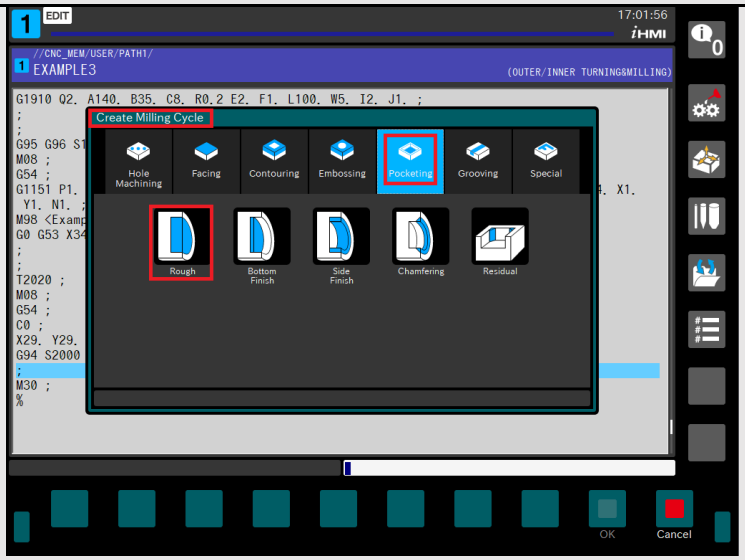
Insert "Fixed Sentence" for "Milling" and edit the program respectively.



Complete as shown at the left.

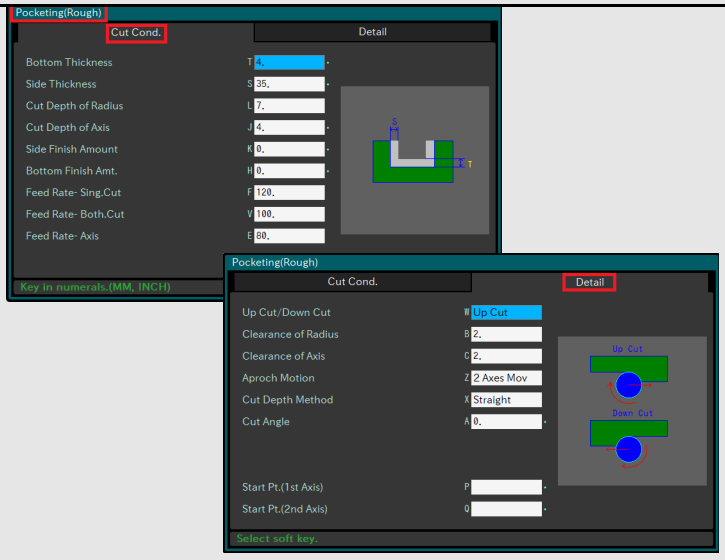


Prepare free position for start of process.

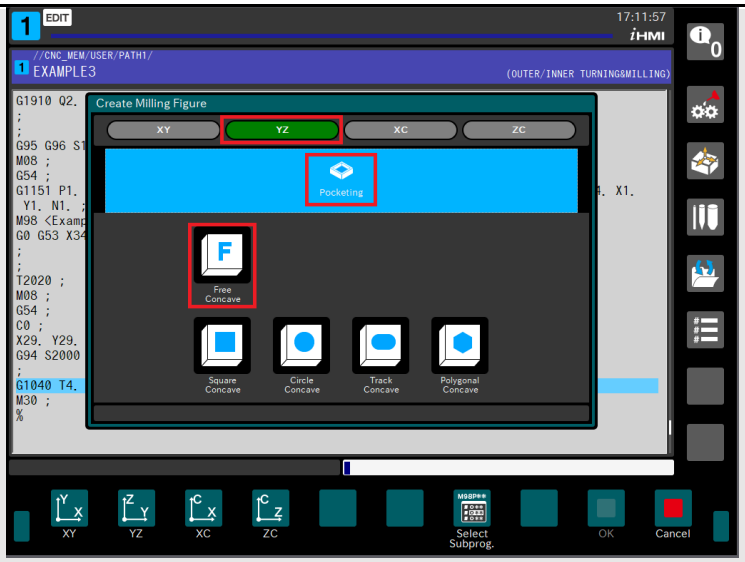


Select "Machining Cycle" and "Pocketing".

Pushing "Roughing" the screen changes automatically.

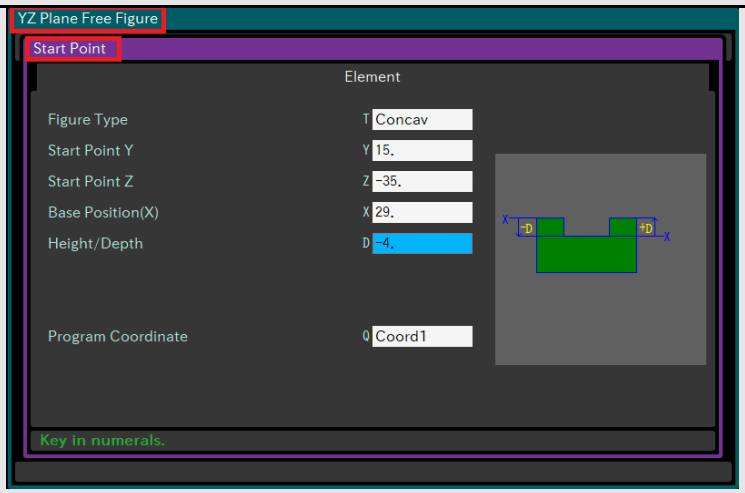


Complete data as shown. Do not forget "Detail".

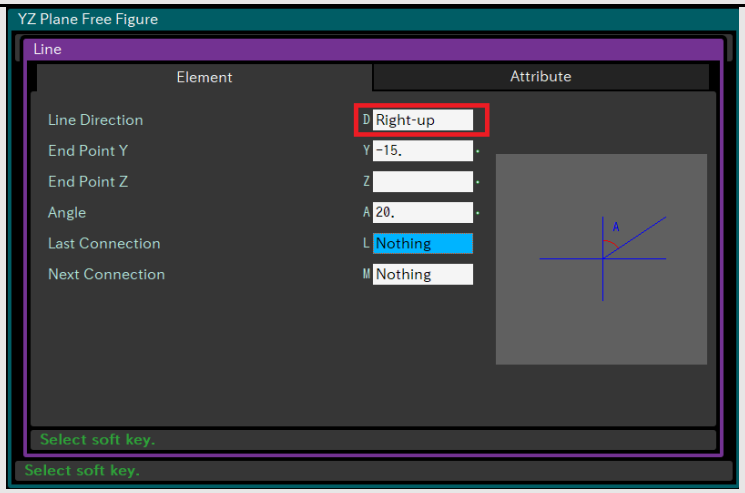


Geometry screen is opened automatically as soon as you confirm the process by "OK".

Select "YZ" plane, and "Free Concave".

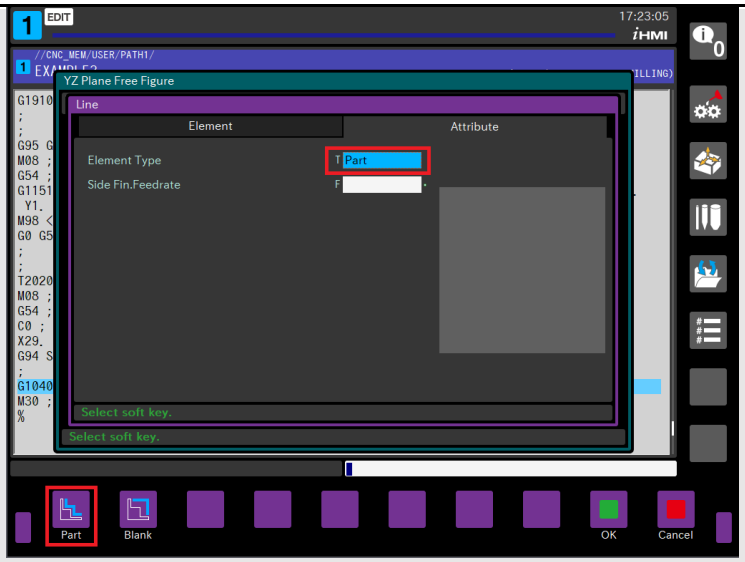


Complete the data for the Start Point according to the drawing, as shown on the left.



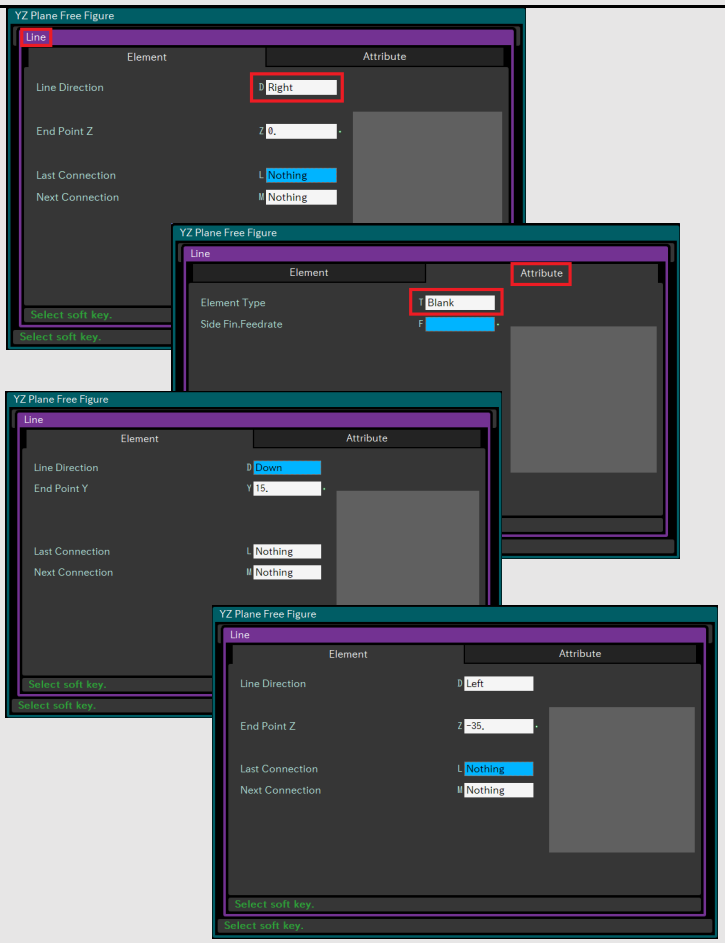
Next elements are Lines:

"Right-up" by angle 20 deg. and End Point in Y.



Under "Attribute" is possible to select "Part" or "Blank". This gives you the possibility to prepare a contour where some elements determine the workpiece, other – the blank.

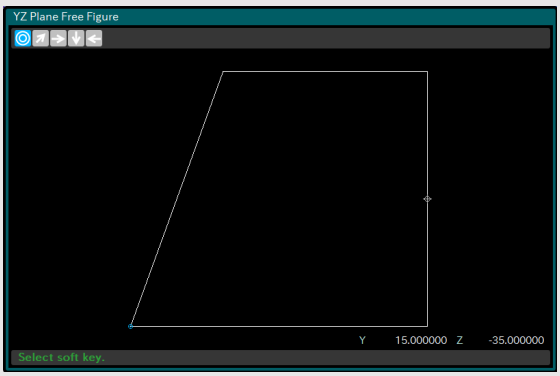
"Part" is a workpiece element – process will cut along the element. "Blank" element is not a border for the cutting process. The process will not restrict overcutting of those contour elements.

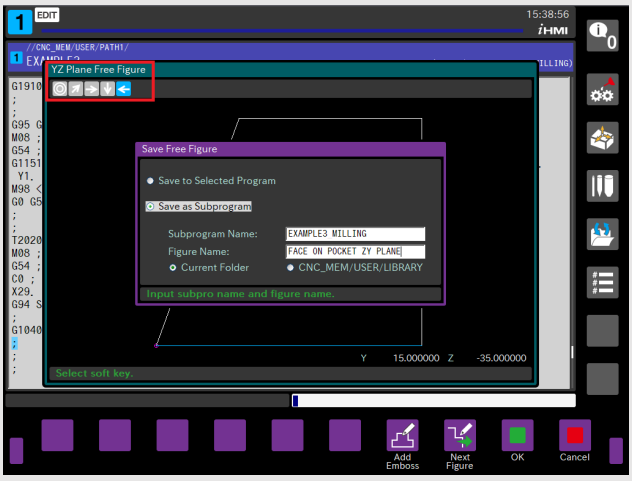


Next line goes "Right" to Z0.

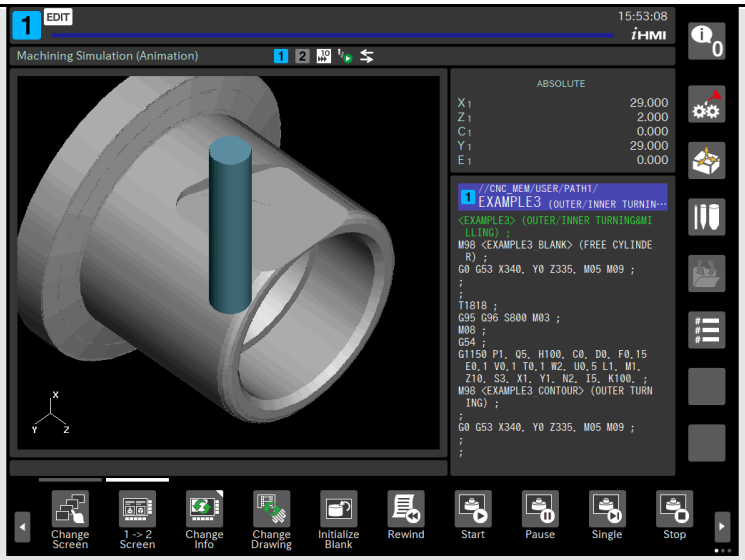
"Attribute" is "Blank".

Subsequent elements are: "Down" to Y15. – "Blank"; "Left" to Z-35. – "Blank".



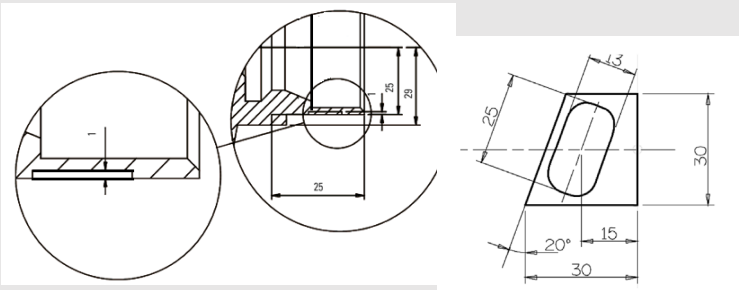


Save contour as subprogram "EXAMPLE_MILLING".

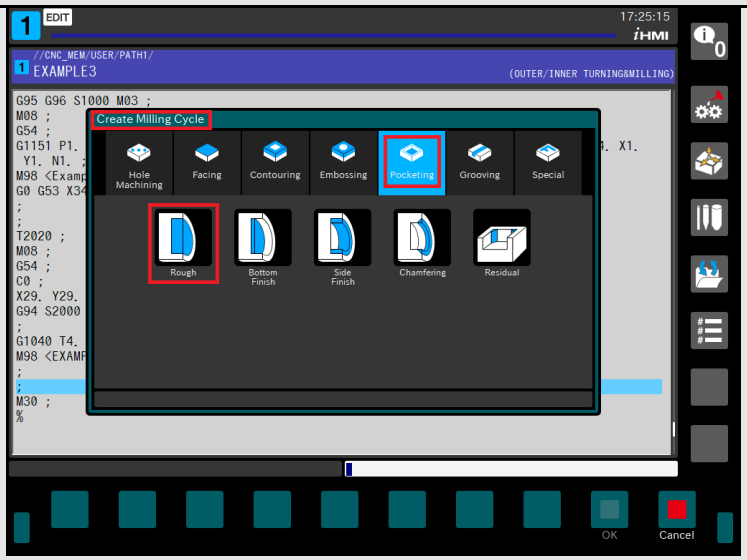


Test/simulate the program. The end position of the cycle is the tool position before the cycle starts.

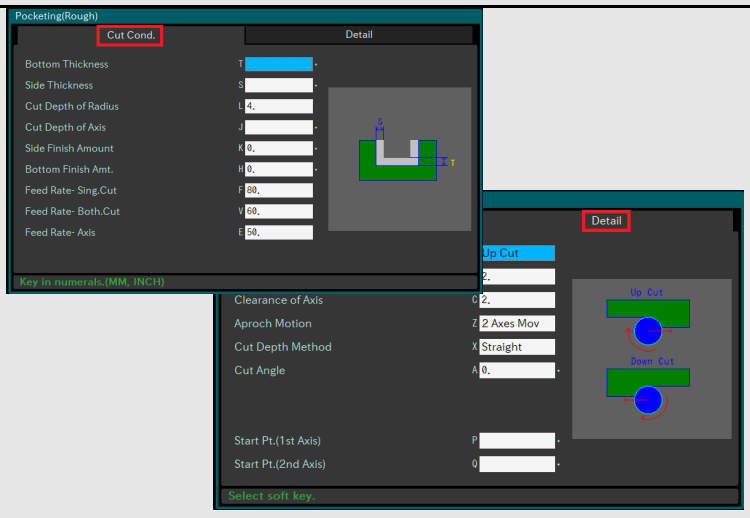
The next process – milling pocket, will use the same tool as previous process, escape to index/tool change position is not necessary.



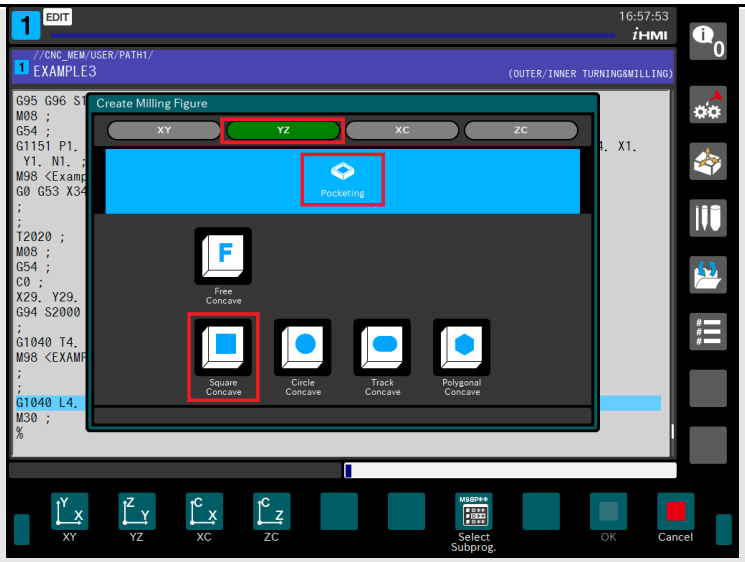
Next milling process will machine the pocket on the prepared face ZY.



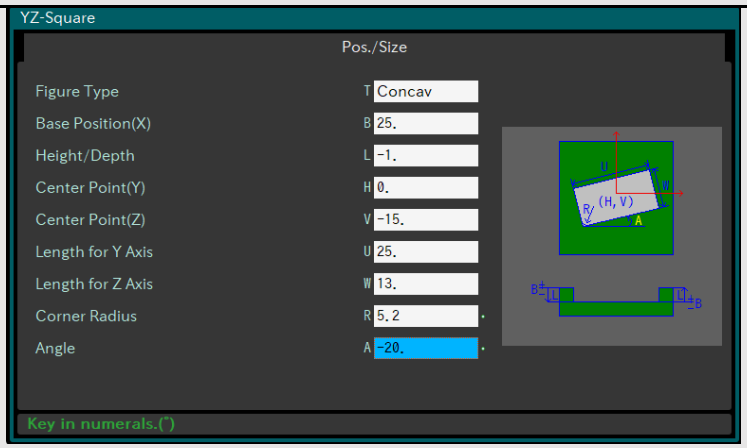
Select "Milling" cycle, "Pocketing" and "Rough".



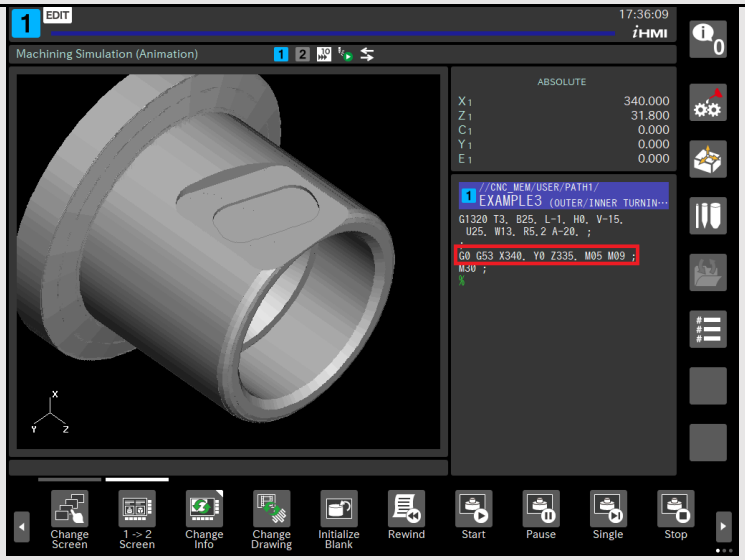
Complete the data, do not forget "Detail".



Select "YZ" plane, "Pocketing" and "Square Concave".



Set up data shown on the left.



Insert movement to Index position.

Test/simulate the program.

Example 3 is completed.

5. Related documents

Description	Document name	Document specification
FANUC iHMI Home Screen	OPERATOR'S MANUAL	B-64644EN
FANUC iHMI CNC Operation Screen	OPERATOR'S MANUAL	B-64644EN-1
FANUC iHMI Machining Cycle	OPERATOR'S MANUAL	B-64644EN-2
FANUC iHMI Set-up Guidance	OPERATOR'S MANUAL	B-64644EN-3
FANUC iHMI Setup Manual	Setting Up iHMI	B-64647EN
CNC Guide (A08B-9010-J770 – J773), V18.0	CNC Guide Operator's manual included in DVD	SPEC19/018_GFXE-19060-EN/01
CNC Guide Update Notice V18.0		A-42147-00431EN

One common servo and control platform –
Infinite opportunities
THAT's FANUC!

WWW.FANUC.EU

FA
CNCs,
Servo Motors
and Lasers

ROBOTS
Industrial Robots,
Accessories
and Software

ROBOCUT
CNC Wire-Cut
Electric Discharge
Machines

ROBODRILL
Compact
CNC Machining
Centres

ROBOSHOT
Electric CNC
Injection Moulding
Machines

IoT
Industry 4.0
solutions

MBR-04262-EN V1.0-10/2020