

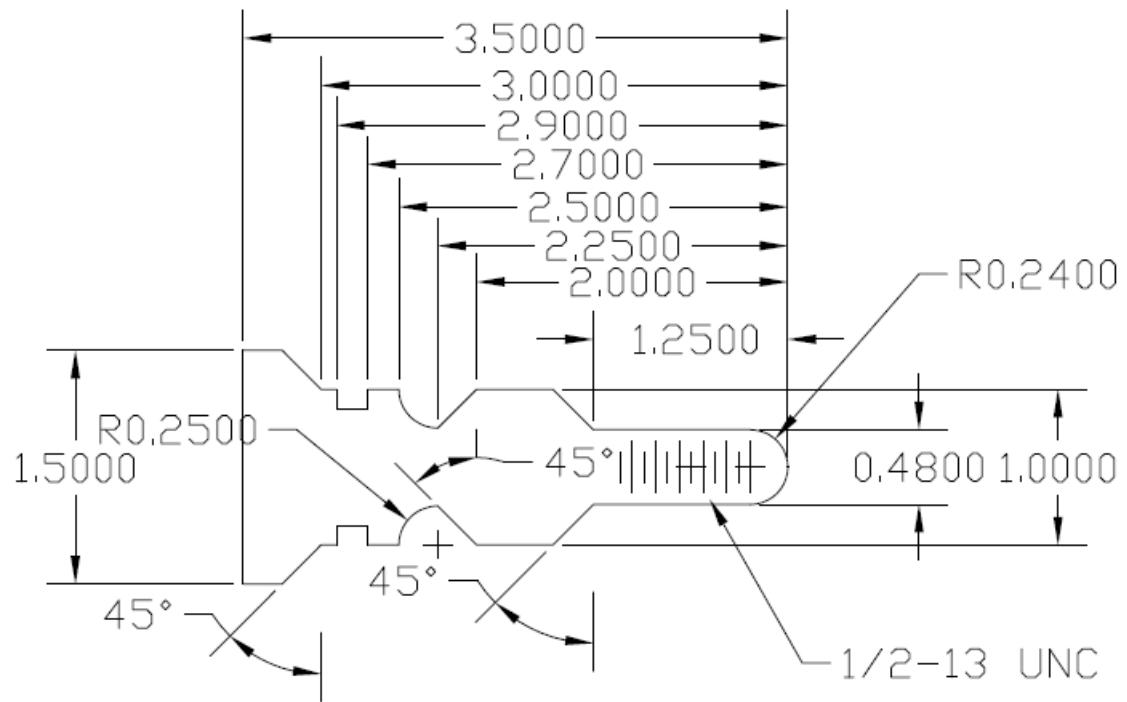
FAGOR AUTOMATION TC TRAINING MANUAL 8060/8065



The screenshot displays the FAGOR TC interface with the following data:

Parameter	Value	Unit
Tool ID	1 D 1	
Offs X	1.5080	
Offs Z	0.8468	
Tool change		
X	-1.800	
Z	0.500	
Feed Rate (F)	0.000	inch/min
Real Feed Rate (F real)	0.000	inch/min
%	100	
Dyn %	100	
Spindle Speed (S)	100.00	RPM
%	100	
SMAX	9000	
Gear	1	
Offset	0.0000	
Spindle Speed (S)	0.0	RPM
X-axis Position	1.5000	
Z-axis Position	0.0500	

The interface also includes a status bar at the top with 'READY', 'FC 1/5', and '16:55:42'. A bottom toolbar contains icons for tool inspection, graphics, offsets, and measurement.



INDEX

INTRODUCTION.....	4
CONVERSATIONAL SCREEN.....	5
REFERENCE SEARCH.....	6
TOOL CALIBRATION.....	7
MANUAL PRESET.....	9
FACING CYCLE.....	10
SIMULATION.....	11
STORING PART PROGRAMS.....	12
TAPER CYCLE.....	17
TURNING CYCLE.....	20
TAPER CYCLE.....	23
ROUNDING CYCLE.....	26
THREADING CYCLE.....	29
PROFILING CYCLE.....	32
GROOVING CYCLE.....	35
CUT OFF CYCLE.....	38
ZERO OFFSETS.....	40

INTRODUCTION

This manual has been generated to be a guideline for your assistance for the conversational programming of the Fagor control. In this manual we will be generating a part-program using the canned cycles that will be the most beneficial for your programming needs. In this manual we will be exercising all of the main canned cycles including:

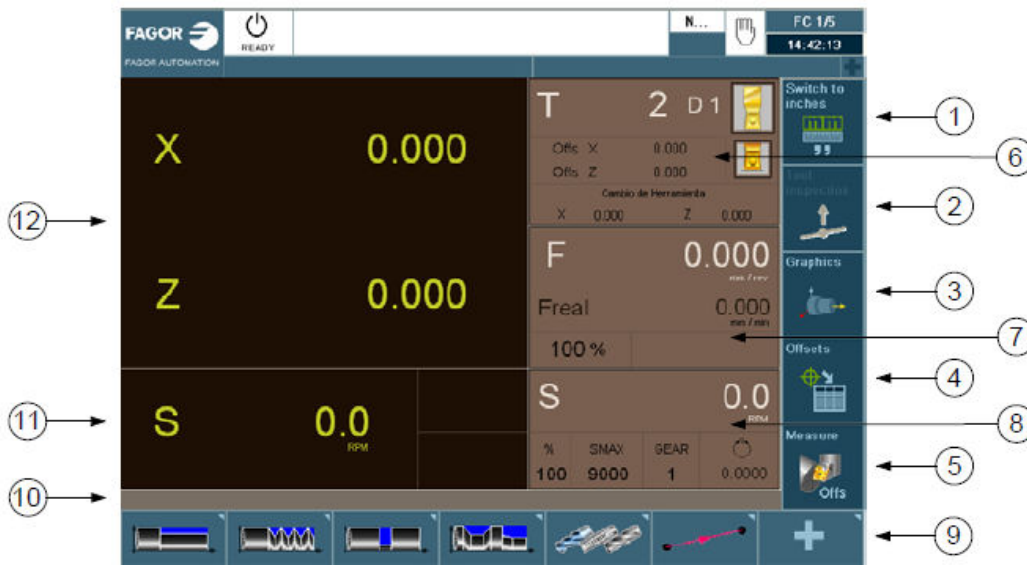
- Tool Calibration
- Turning Cycle
- Facing Cycle
- Taper Cycle
- Rounding Cycle
- Threading Cycle
- Grooving Cycle
- Profiling Cycle

Along with some of the other operations that will be important to no how to use:

- Homing of the Axes
- Presetting your Zero
- Simulation/Graphics
- File Maintenance

After the completion of these operations, you will begin to understand all the steps that are necessary in programming the Fagor CNC Control. Once you grasp each of these concepts you will be able generate any part in which you desire. Using this manual as your guideline will also help you generate part-programs in a timely manner. You will soon realize once you learn the programming of one canned cycle, you will understand the programming concept of each and every one of the other canned cycles. Whether it is your first time using a CNC or if you are a CNC programming expert, the conversational programming of the Fagor CNC Control will benefit all users.


CONVERSATIONAL SCREEN

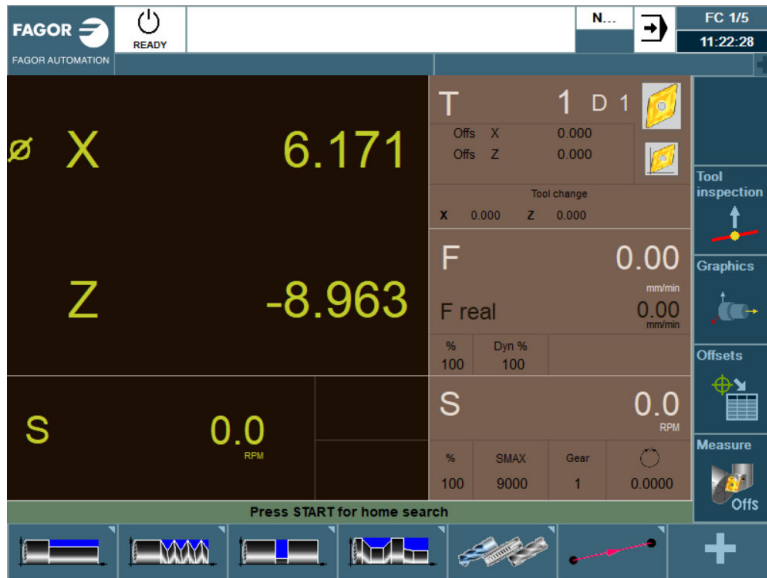


1. Softkey for selecting unit mm/inches.
2. Softkey to go into tool inspection.
3. Softkey to access the graphics in execution mode.
4. Softkey for selecting OFFSETS.
5. Softkey for tool calibration.
6. Window that shows:
 - The selected tool (T).
 - The (D) offset number associated with the selected tool.
 - The offset defined for the tool.
 - The position values (coordinates) of the tool change point referred to machine reference zero.
7. Window showing the axis feedrate F currently selected, the % of F being applied and the real F value.
8. Window showing spindle related information:
 - The selected theoretical turning speed. S value when working in rpm and CSS value when working at constant surface speed.
 - Spindle status. It is represented with an icon and may be turning clockwise, counterclockwise or stopped.
 - The % of spindle speed being applied.
 - Maximum spindle rpm (Smax).
 - Active spindle range (gear).
9. Softkeys for cycle editing.
10. Message bar.
11. Real spindle rpm.
12. Position (coordinates) of the axes. The f symbol indicates that the axis is working in diameter.


REFERENCE SEARCH

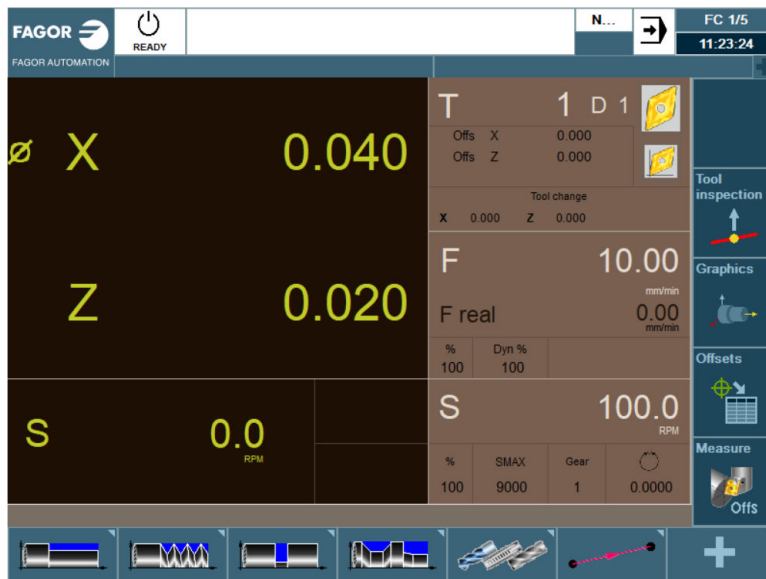
On power-up, it is always recommended to home your machine before performing any specific task. This is more of a safety precaution. When referencing your machine, the CNC will know exactly where it is located. To reference your machine, perform the following operation:

1. From the Jog Screen select the Zero Icon 
 - The following screen will appear



** Note: Pressing the ESC Key instead of Cycle Start will cancel the operation*


2. To execute the Home Search press cycle start 
 - At this time the CNC will execute the subroutine associated with the home search.
 - Both axes will now move and find their home positions.

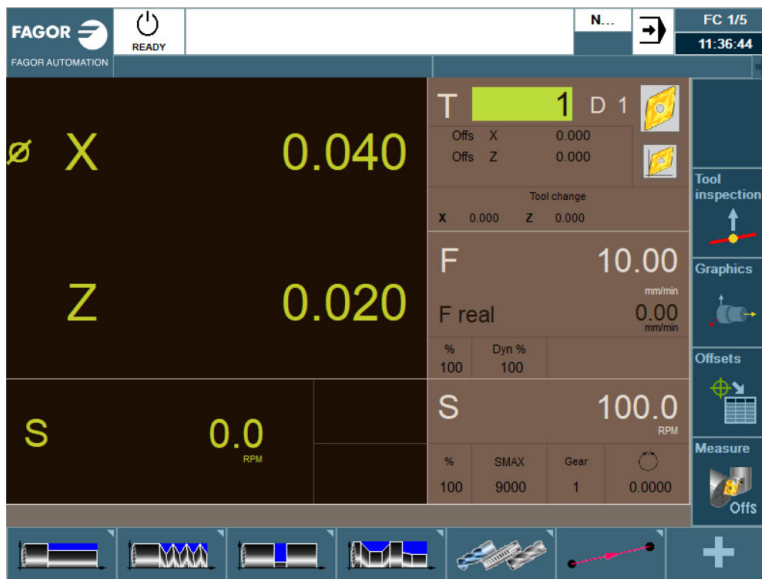


TOOL CALIBRATION

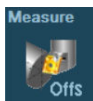
For this specific program we will be using four tools. Before we begin any type of programming, we will calibrate all four of our tools on both axes.

1. From the Jog Screen enter the following commands:

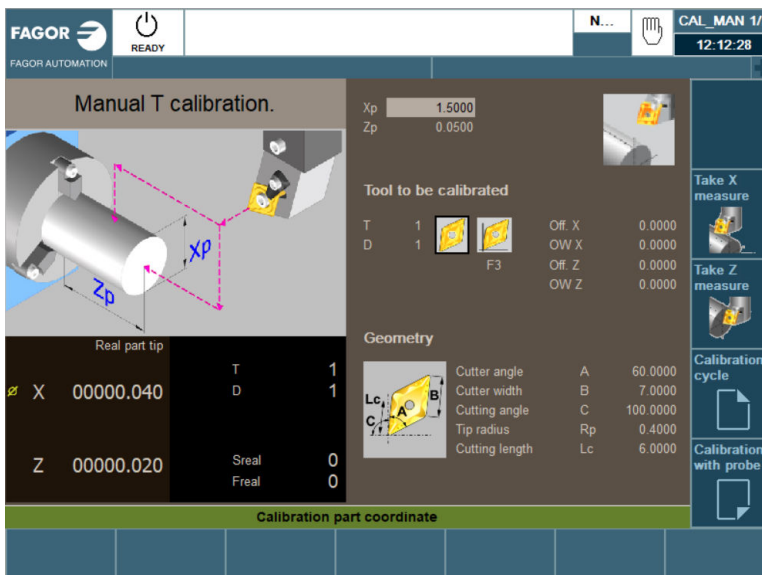
- T (the box will highlight in yellow)
- 1 (Tool 1)
- Cycle Start 
-



2. The axes will now move to its Tool Change Position.




3. Select the Icon.




- The first section that is highlighted grey is for the dimensions of your part. In this example we will use 1.5 for X (Xp) and 3.5 for Z (Zp).
- Select your tool family by using the space key to toggle through them. We will be using the diamond shape for tool 1.




- Select the tool shape by using the space. We will use  for tool 1.
- Jog your X axis accordingly until the tip of your tool is touching your part.
- Tool calibration:



- Press  then press ENTER to confirm the value. This will automatically put your tool offset in X.
- OW X - This is the tool offset wear for X axis. At this time, it is unnecessary to put any value in this field.

- Jog your Z axis accordingly until the tip of your tool is touching your part.



- Press  then press ENTER to confirm the value. This will automatically put your tool offset in Z.
- OW Z - This is the tool offset wear for Z axis. At this time, it is unnecessary to put any value in this field.

11. Geometry:


- Enter the appropriate information for defining your tool geometry.

A	60.0000
B	0.5000
C	95.0000
Rp	0.0020
Lc	0.2500

- For tool 1 we will use the following example:

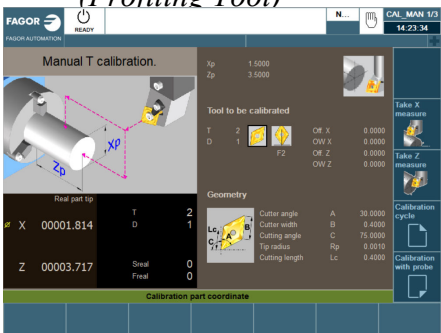
Note: It is very important that you define your tool geometry correctly.

TOOLS 2, 3, 4

- Select T (Tool)
- Enter 2, 3, 4 (Tool Number)
- Press Cycle Start 
- Repeat steps 5-10.

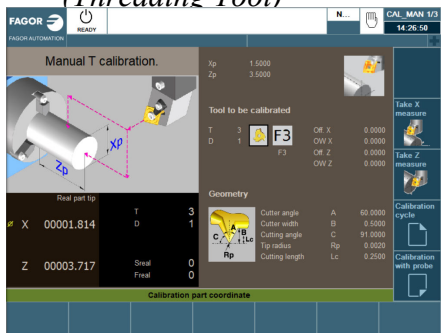
Tool 2

(Profiling Tool)



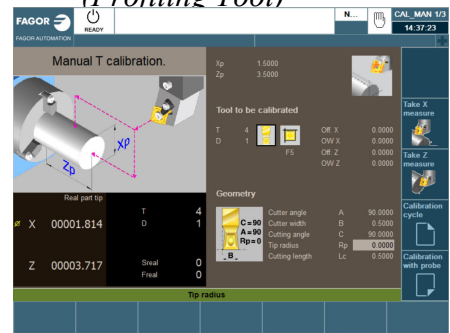
Tool 3

(Threading Tool)



Tool 4

(Profiling Tool)



MANUAL PRESET

Note: Not required if using the tool calibration.

1. From the Jog Mode, jog your X axis until you are at the desired position. Enter the following keys:
 - X (select your axis)
 - The number (your diameter preset value) 1.5”.
 - Enter (confirming your position)
 - Enter (executing your command)
- *Repeat these steps for your Z axis*

Figure 1



Figure 2

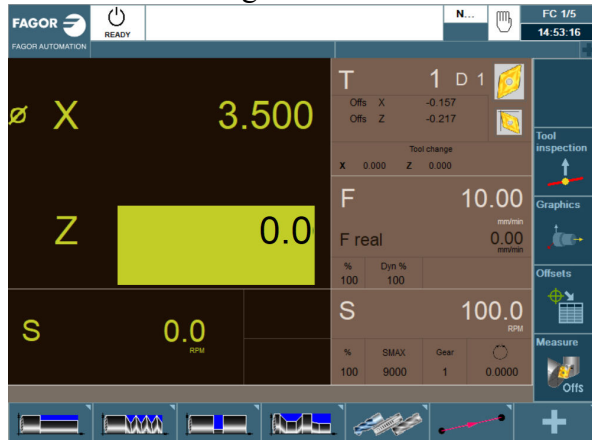


Figure 3

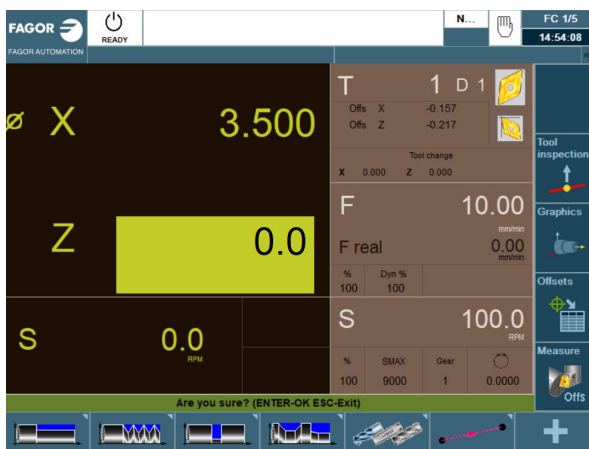
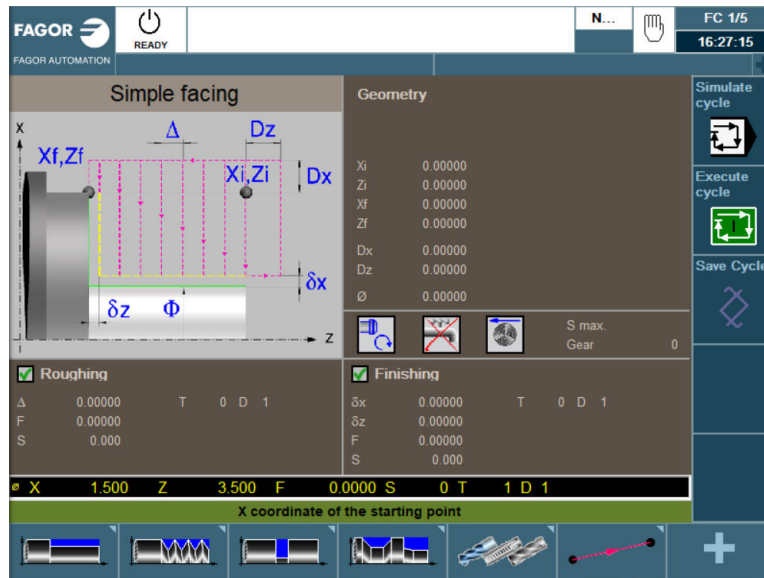


Figure 4



FACING CYCLE



1. Select , then 

2. Define your part:

- (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z0.05.
- (Xf, Zf)- This is your ending point of your X and Z coordinates. In this example we will be using X1.5 and Z0.
- Safety distances (Dx, Dz) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
- “Ø” This is the diameter that you want to turn the part down to. In this example we will use -0.25.

3. Define your machining conditions:

- Spindle direction. The space key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
- Coolant on or off. Use the space key to select.
- You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
- SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.


4. Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.01.

5. Define your Roughing Pass:

- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S- This is your spindle speed. In this example we are using a value of 300.
- T- Enter your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.

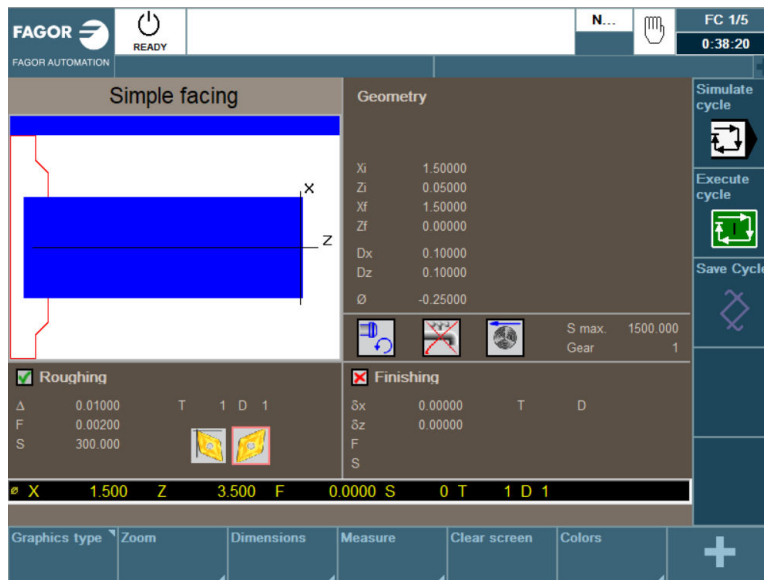
SIMULATION

Before the execution of any part program, running your program in simulation is strongly recommended. The following steps will guide you on how to set-up your graphics.

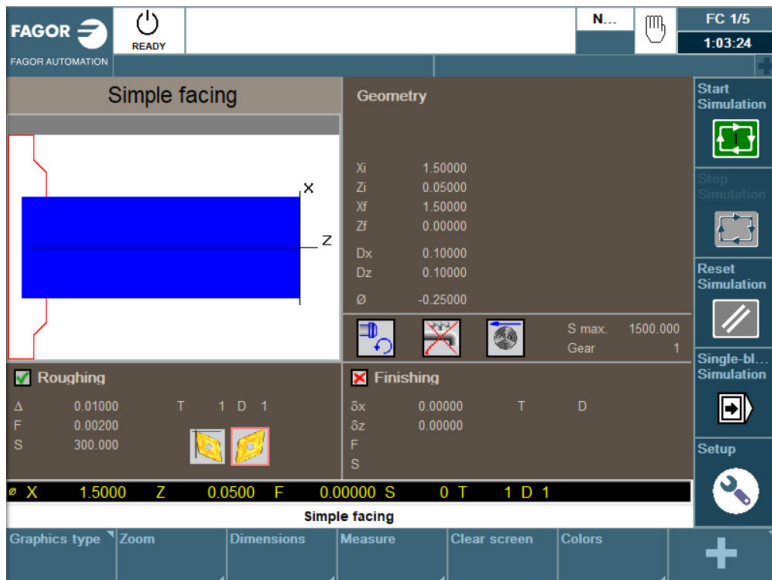
1. Press the Simulate  from your canned cycle page.
2. Press F1- Graphics type
3. Select Solids XZ
4. Next step is to define your part size.
5. Press F3 - Dimension.
6. Press F7 - Edit
7. For this example we will be using the following:

*Note: Press the down arrow to get to the next box.
Press ENTER when everything's filled out correctly.*

Z MIN	-5.000
Z MAX	0.05
INSIDE D	0.000
OUTSIDE D	1.500



8. Now to run your program in simulation press the Start Simulation button
9. After simulating the cycle the following screen should appear:

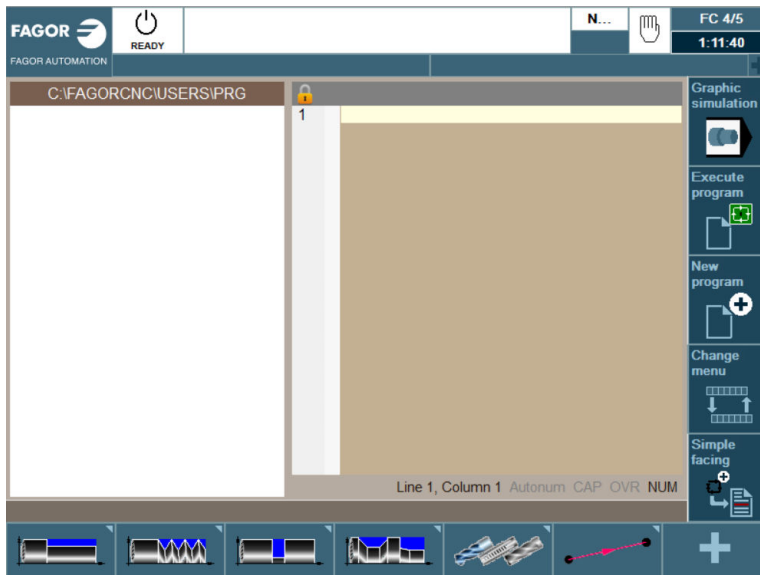



Note: If you notice no change in your graphics while running in simulation, go through step 6 again.

10. You are now ready to save this cycle to your Part-Program.

STORING PART PROGRAMS

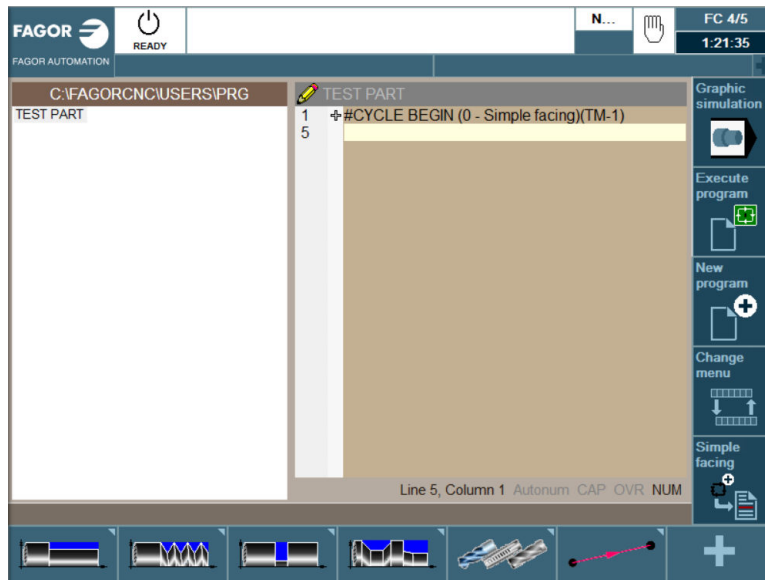
1. Select Save Cycle 
 - *The following screen will appear*



2. Select  and type the name Test Part. Press Enter.

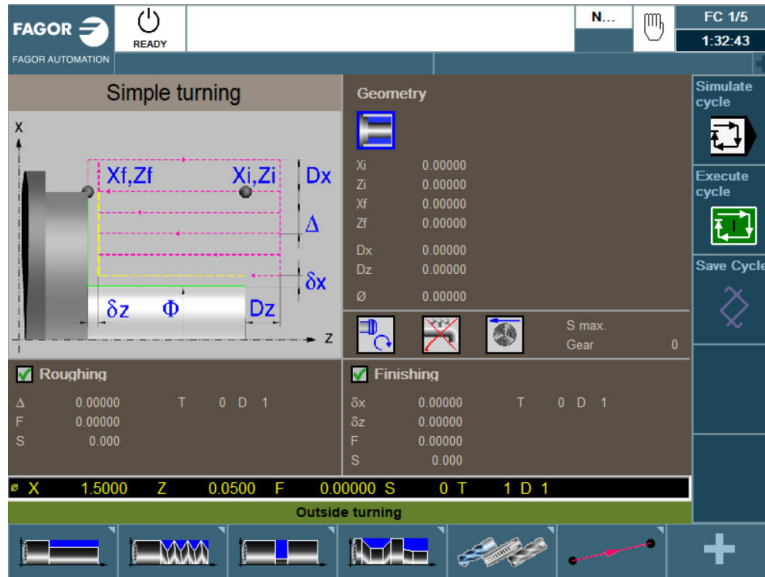
3. Select 



- *The CNC will add the cycle to the program*



4. The program is saved automatically. You can add M30 to the end of the program now or wait to add after the last cycle.
5. You have three options after saving the program, Graphic simulation, Execute Program, and selecting a new cycle.

TURNING CYCLE



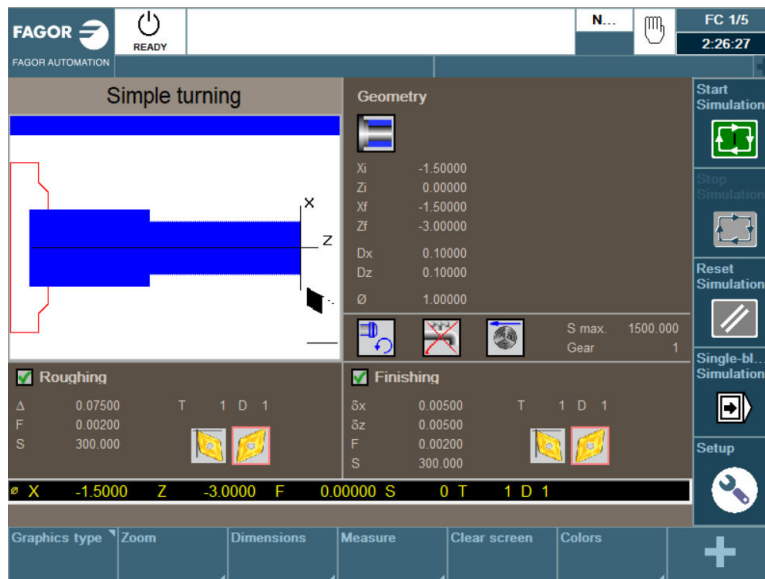
1. Select , then  Simple turning operation
2. For this operation we will be using Outside Turning. You can use the space key to toggle from inside to outside turning when the icon is highlighted in blue.
3. Define your part:
 - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z0.
 - (Xf, Zf)- This is your ending point of your X and Z coordinates. In this example we will be using X1.5 and Z-3.
 - Safety distances (Dx, Dz) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
 - “Ø” This is the diameter that you want to turn the part down to. In this example we will use 1.
4. Define your machining conditions:
 - Spindle direction. The space key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
 - Coolant on or off. Use the space key to select.
 - You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
 - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
5. Define your Roughing Pass:

- Δ - This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.
- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S- This is your spindle speed. In this example we are using a value of 300 rpm.
- T- Enter is your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.

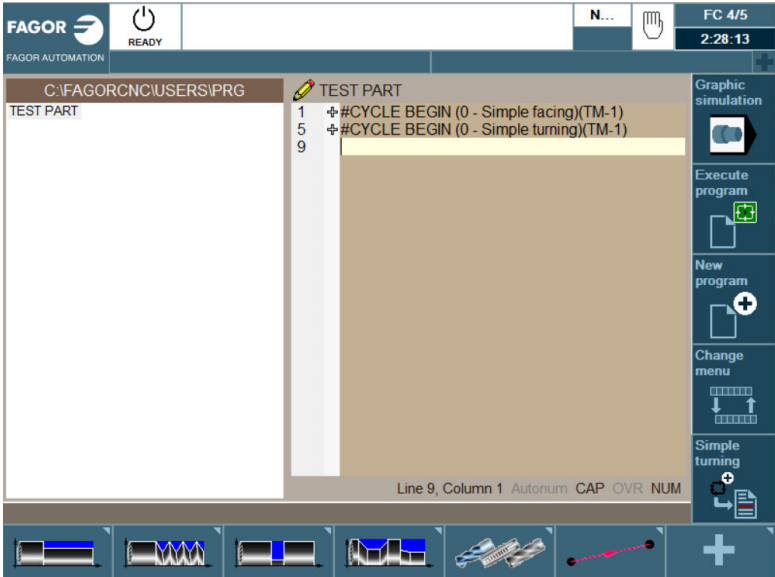
6. Define your Finishing Pass:

- δx - This is the amount of finishing stock that is going to be left on x for the finishing pass. For this example we will be using 0.005.
- δz - This is the amount of finishing stock that is going to be left on z for the finishing pass. For this example we will be using 0.005.
- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- VCC- This is your surface speed. In this example we are using a value of 300.
- T - Enter is your tool number. In this example we are using T1.
- D - Enter in your Tool Offset Number. In this example we are using D1.

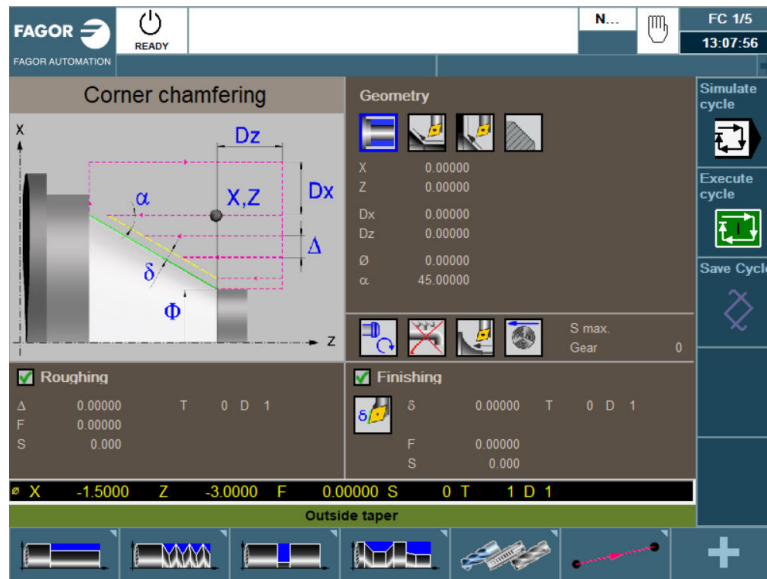
7. Simulate the program: (If necessary, Refer to page 11)


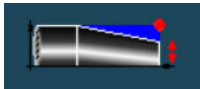





8. Save your part-program. (Refer to page 12)



TAPER CYCLE



1. Select , then  Cornering Chamfering operation.
2. For this operation we will be using Outside Turning. You can use the space key to toggle from inside to outside turning when the icon is highlighted blue.
3. Concave or convex entry of compensation. Use the space key to toggle. For this example, we will be using convex. 
4. Concave or convex entry of compensation. Use the space key to toggle. For this example, we will be using convex. 
5. The turning area that we will be working is Quadrant 1 
6. Define your part:
 - (X, Z)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z-3.
 - Safety distances (Dx, Dz) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
 - Ø - This is the diameter that you want to turn the part down to. In this example we will use 1.
 - α - This is the angle the taper will be on. In the example we will be using 45 degrees.

7. Define your machining conditions:

- Spindle direction. The space key will toggle from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
- Coolant on or off. Use the space key to select.
- You can select to either machine along the Z axis or X axis. For this example, we will use



along Z

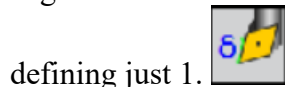
- You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
- SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 2000.

8. Define your Roughing Pass:

- Δ - This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.
- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S- This is your spindle speed. In this example we are using a value of 300 rpm.
- T- Enter is your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.

9. Define your Finishing Pass:

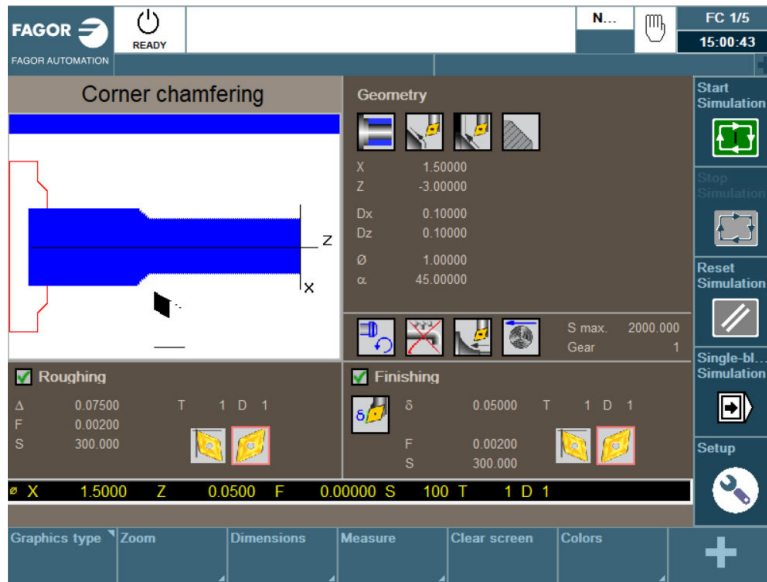
- Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). For this example, we will be using



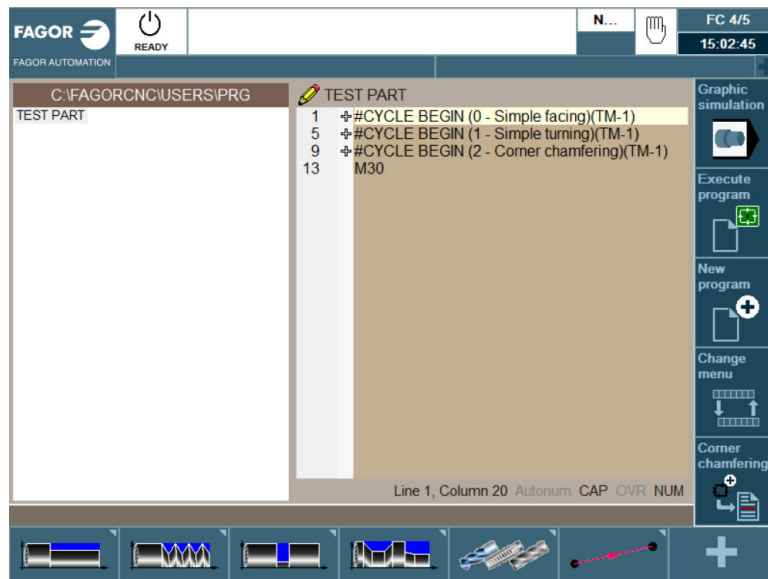
defining just 1.

- δ – This is the amount of finishing stock that is going to be left on for the finishing pass. For this example, we will be using 0.05.
- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S- This is your surface speed. In this example we are using a value of 300 rpm.
- T- Enter is your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.

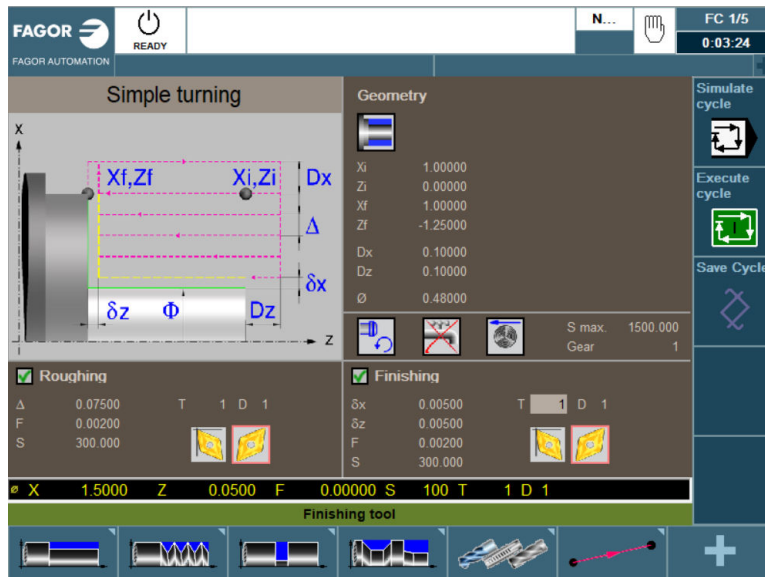
10. Simulate the program: (Refer to page 11)





11. Save your Cycle to your part-program: (Refer to page 12)



TURNING CYCLE



1. Select , then  Simple turning operation
2. For this operation we will be using Outside Turning. You can use the space key to toggle from inside to outside turning when the icon is highlighted blue.
3. Define your part:
 - (X_i, Z_i) - This is your starting point of your X and Z coordinates. In this example we will be using X_1 and Z_0 .
 - (X_f, Z_f) - This is your ending point of your X and Z coordinates. In this example we will be using X_1 and $Z_{-1.25}$.
 - Safety distances (D_x, D_z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
 - \emptyset - This is the diameter that you want to turn the part down to. In this example we will use 0.48 to undercut for the thread.
4. Define your machining conditions:
 - Spindle direction. The space key will toggle you from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
 - Coolant on or off. Use the space key to select.
 - You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
 - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.

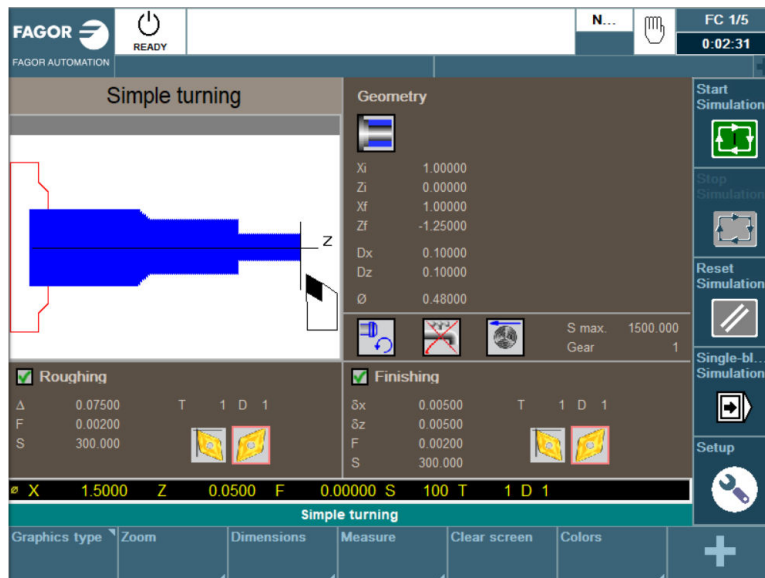
5. Define your Roughing Pass:

- Δ - This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.
- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S- This is your spindle speed. In this example we are using a value of 300.
- T- Enter is your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.

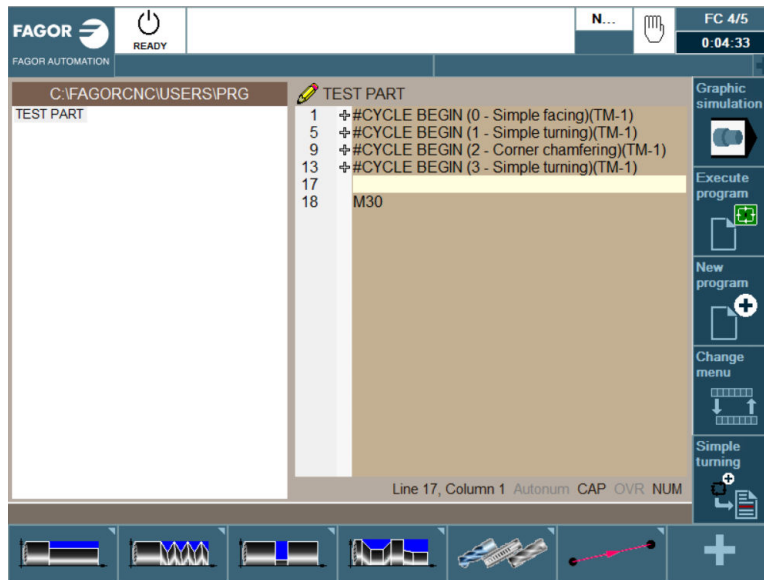
6. Define your Finishing Pass:

- δX - This is the amount of finishing stock that is going to be left on x for the finishing pass. For this example, we will be using 0.005.
- δZ - This is the amount of finishing stock that is going to be left on z for the finishing pass. For this example, we will be using 0.005.
- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S- This is your spindle speed. In this example we are using a value of 300.
- T- Enter is your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.

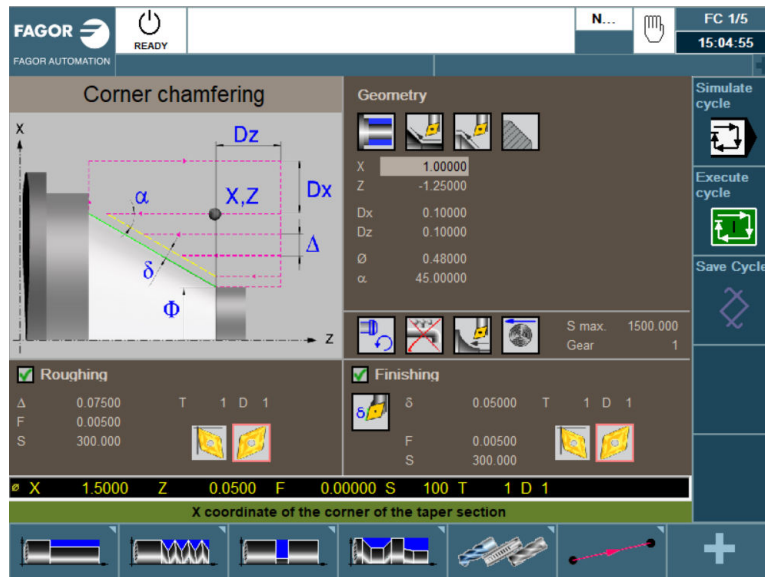
7. Simulate the program: (Refer to page 11)


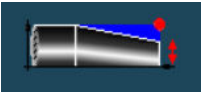





8. Save your Cycle to your part-program: (Refer to page 12)



TAPER CYCLE



1. Select , then  Cornering Chamfering operation.
2. For this operation we will be using Outside Threading. You can use the space key to toggle from inside to outside tapering when the icon is highlighted blue.
3. Concave or convex entry of compensation. Use the space key to toggle. For this example, we will be using concave. 
4. Concave or convex entry of compensation. Use the space key to toggle. For this example, we will be using convex. 
5. The turning area that we will be working is Quadrant 1 
6. Define your part:
 - (X, Z)- This is your starting point of your X and Z coordinates. In this example we will be using X1 and Z-1.25.
 - Safety distances (Dx, Dz) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
 - Ø - This is the diameter that you want to turn the part down to. In this example we will use 0.48.

- α - This This is the angle the taper will be on. In the example we will be using 45 degrees.

7. Define your machining conditions:

- Spindle direction. The space key will toggle from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
- Coolant on or off. Use the space key to select.
- You can select to either machine along the Z axis or X axis. For this example, we will use

along Z .


- You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
- SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.

8. Define your Roughing Pass:

- Δ - This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.
- F - This is your roughing feedrate for your axes. In this example we are using a value of 0.005 inches per revolution.
- S - This is your spindle speed. In this example we are using a value of 300 rpm.
- T - Enter is your tool number. In this example we are using T1.
- D - Enter in your Tool Offset Number. In this example we are using D1.

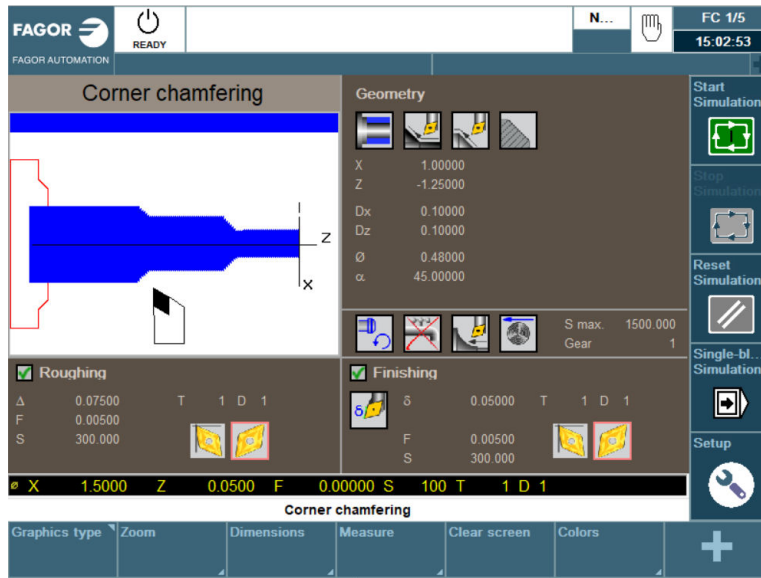
9. Define your Finishing Pass:

- Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). For this example, we will be using

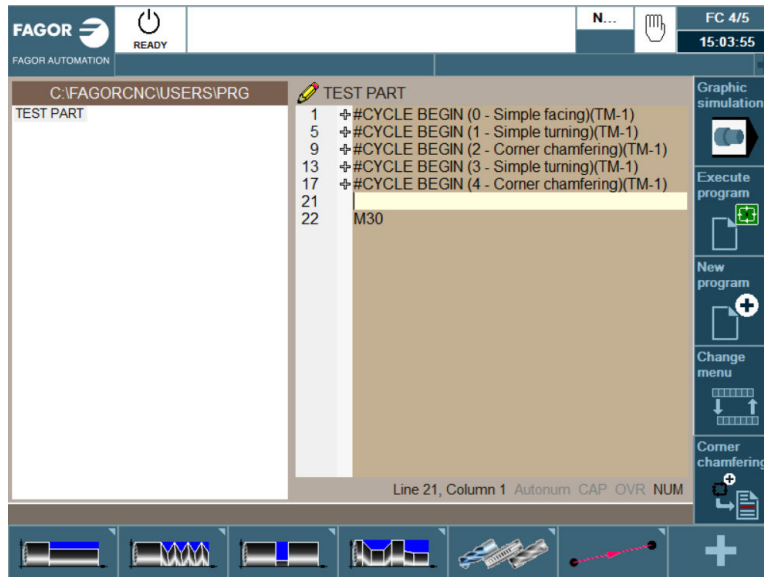
defining just 1. .

- δ - This is the amount of finishing stock that is going to be left on for the finishing pass. For this example, we will be using 0.05.
- F- This is your roughing feedrate for your axes. In this example we are using a value of 0.005 inches per revolution.
- S- This is your surface speed. In this example we are using a value of 300 rpm.
- T- Enter is your tool number. In this example we are using T1.
- D- Enter in your Tool Offset Number. In this example we are using D1.

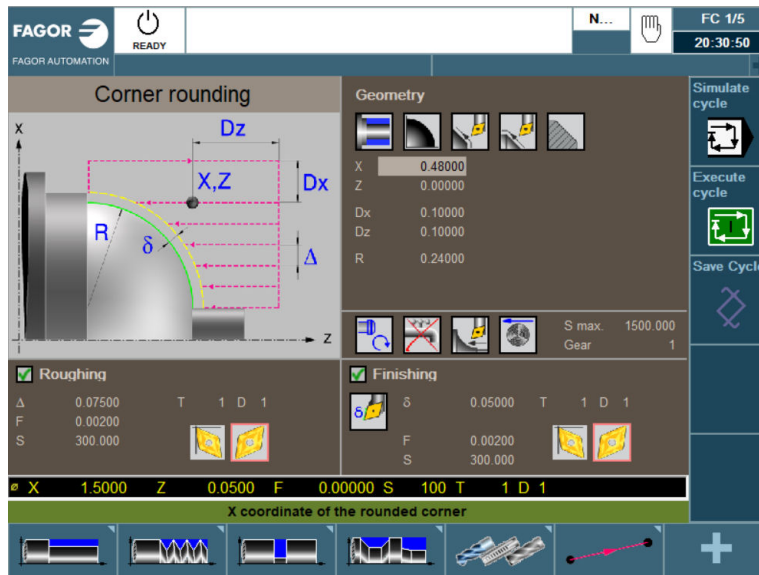
10. Simulate the program: (Refer to page 11)










11. Save your Cycle to your part-program: (Refer to page 11)



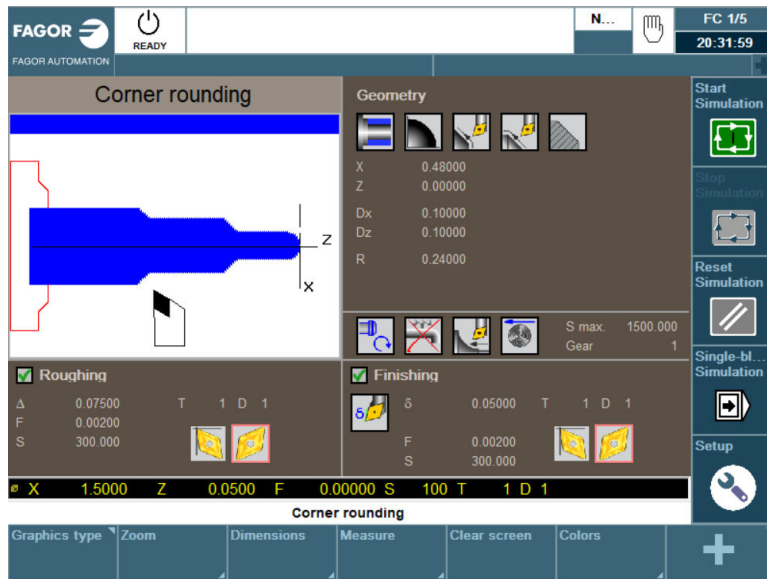
ROUNDING CYCLE



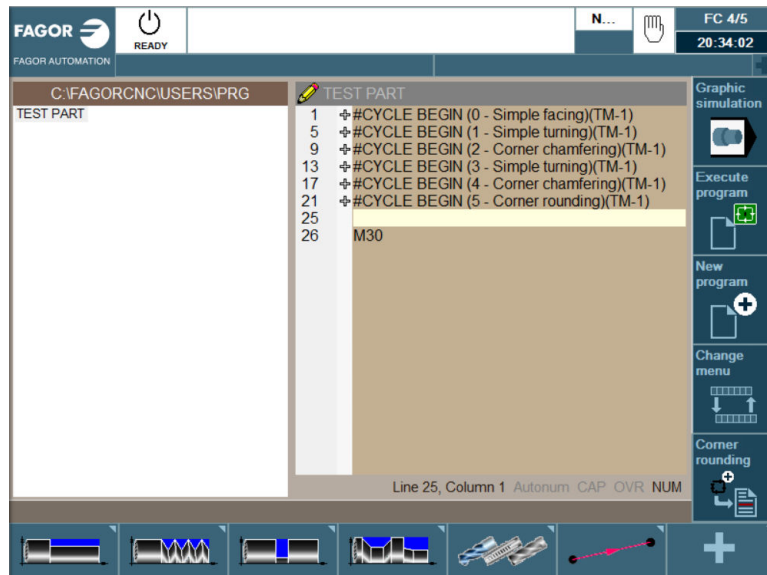
1. Select , then  Corner Rounding operation.
2. For this operation we will be using Outside Turning. You can use the space key to toggle from inside to outside turning when the icon is highlighted blue.
3. You can choose either convex or concave rounding. Use the space key to toggle. We will be using convex rounding for this example. 
4. Concave or convex entry of compensation. Use the space key to toggle. For this example, we will be using convex. 
5. Concave or convex entry of compensation. Use the space key to toggle. For this example, we will be using convex. 
6. The turning area that we will be working is Quadrant 1 
7. Define your part:
 - (X, Z)- This is your starting point of your X and Z coordinates. In this example we will be using X0.48 and Z0.

- Safety distances (Dx, Dz) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
 - R- This is where you put your radius value. In this example we will be using 0.24.
8. Define your machining conditions:
- Spindle direction. The space key will toggle from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
 - Coolant on or off. Use the space key to select.
 - You can select to either machine along the Z axis or X axis. For this example, we will use along Z.
 - You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
 - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
9. Define your Roughing Pass:
- Δ- This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.075.
 - F- This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
 - S- This is your spindle speed. In this example we are using a value of 1000 rpm.
 - T- Enter is your tool number. In this example we are using T1.
 - D- Enter in your Tool Offset Number. In this example we are using D1.
10. Define your Finishing Pass:
- Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). For this example, we will be using defining just 1. 
 - δ – This is the amount of finishing stock that is going to be left on for the finishing pass. For this example, we will be using 0.05.
 - F - This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
 - S - This is your surface speed. In this example we are using a value of 300 rpm.
 - T - Enter is your tool number. In this example we are using T1.
 - D - Enter in your Tool Offset Number. In this example we are using D1.

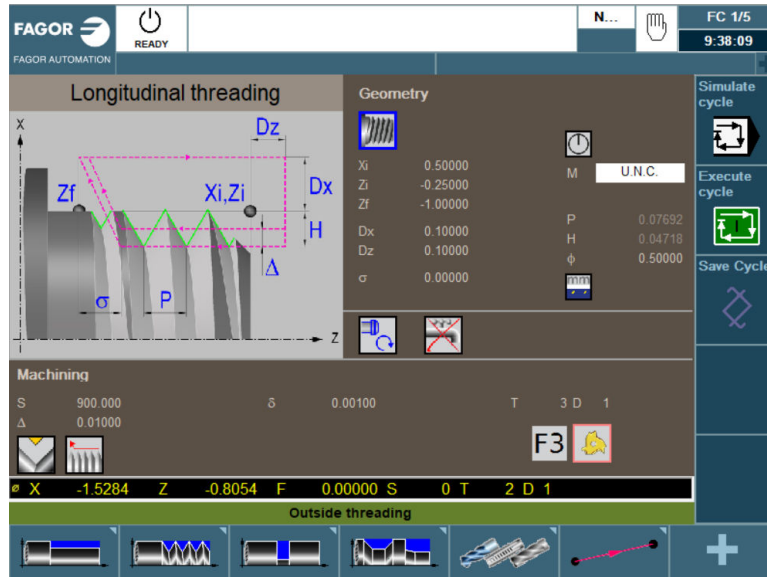
11. Simulate the program: (Refer to page 11)

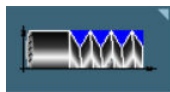
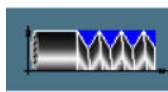



12. Save your Cycle to your part-program: (Refer to page 12)



THREADING CYCLE



1. Select  then  Longitudinal threading operation.
2. For this operation we will be using Outside Threading. You can use the space key to toggle from inside to outside threading when the icon is highlighted blue.
3. Define your part:
 - (X_i, Z_i) - This is your starting point of your X and Z coordinates. In this example we will be using X0.5 and Z-0.25
 - (Z_f) - This is your ending point of your Z coordinate. In this example we will use Z-1.
 - Safety distances (D_x, D_z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
 - σ - This indicates at what distance from the end of the thread the tool starts exiting. In this example we will be using 0.
 - IO angle- you can program with or without entry angle. For this example, we will program without.
 - Select the thread type that you would like to use. You can use the space key to toggle through the different types of thread. In this example we will be using U.N.C.
 - "Ø" This is the diameter of the material that is being threaded. We will be using 0.5 for our example.
 - Define the type of thread data, mm or inches. For this example, we will use inches. 
4. Define your machining conditions:
 - Coolant on or off. Use the space key to select.

- Spindle direction. The space key will toggle you from clockwise to counterclockwise. We will be using counter-clockwise in this example.

5. Define your passes:

- S - This is your spindle speed. In this example we are using a value of 900 RPM.
- Δ – This is the maximum penetration pass. In this example we will be using 0.01.
- Repeat last threading pass or not. We will not be repeating last pass for this example.

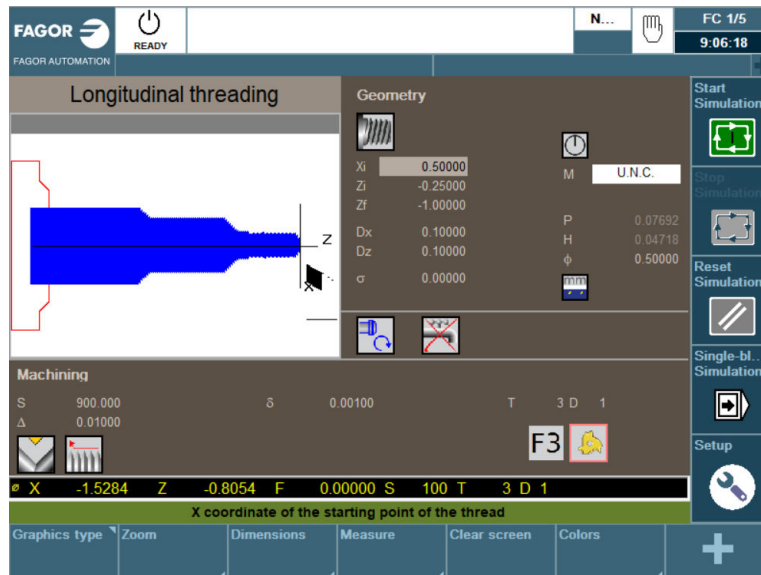


- This is to select the type of exit. Smooth or abrupt. For this example, we will use smooth.

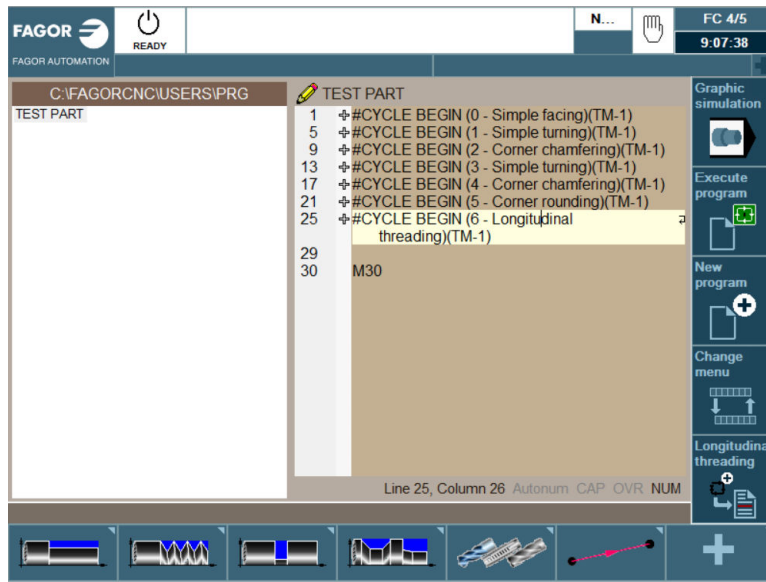


- δ – This is the minimum penetration pass. In this example we will be using 0.001.
- T- Enter is your tool number. In this example we are using T3.
- D- Enter in your Tool Offset Number. In this example we are using D1.

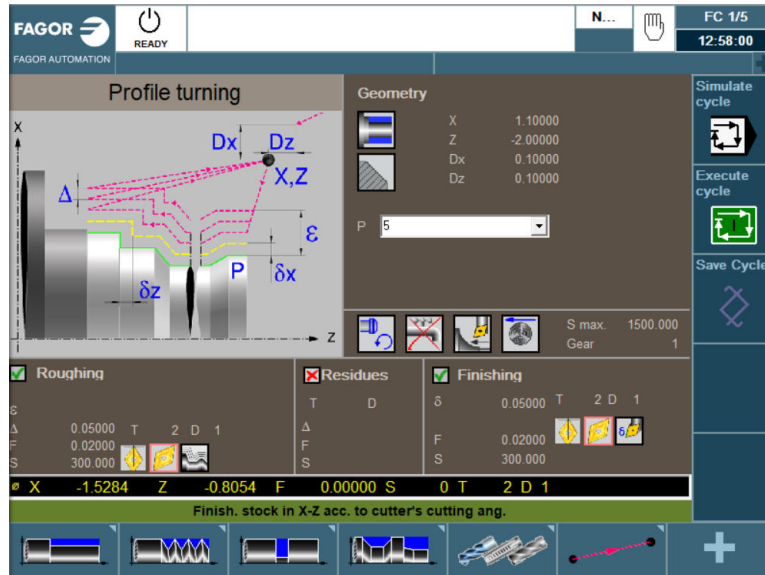
13. Simulate the program: (Refer to page 11)

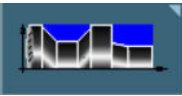
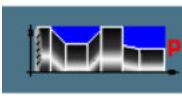



14. Save your Cycle to your part-program: (Refer to page 12)



PROFILING CYCLE




1. Select  then  Profile Turning operation.
2. For this operation we will be using Outside Profiling. You can use the space key to toggle from inside to outside profiling when the icon is highlighted blue.
3. The turning area that we will be working is Quadrant 1 .
4. (X, Z) – coordinates of the starting points. For this example, we will be using X: 1.1, Z: -2.
5. Safety distances (Dx, Dz) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
6. P – This is going to be our new profile number. In this example put in number 5 and then press RECALL. You will now be on the mini-cad page. Follow the following steps for our demo part:
 - Press Edit (F1)
 - Press Profile (F1)
 - Insert the initial points on the right-hand side. (Z1: -2, X1: 1)
 - Press Validate (F7)
 - Press Straight Line (F1)
 - Insert the following: (Z2: -2.25, X2: 0.5)
 - Angle: Leave blank
 - Press Validate (F7)

- Press Clockwise arc (F2)
- Insert the following: (Z2: -2.25, X2: 0.5, R: 0.25)
- Zc, Xc: Leave blank
- Press Validate (F7)
- Press Save & Continue (F7)
- Press ESC
- Press Finish (F5)

7. Define your machining conditions:


- Spindle direction. The space key will toggle from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
- Coolant on or off. Use the space key to select.
- You can select to either machine along the Z axis or X axis. For this example, we will use along Z.
- You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
- SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.

8. Define your Roughing Pass:

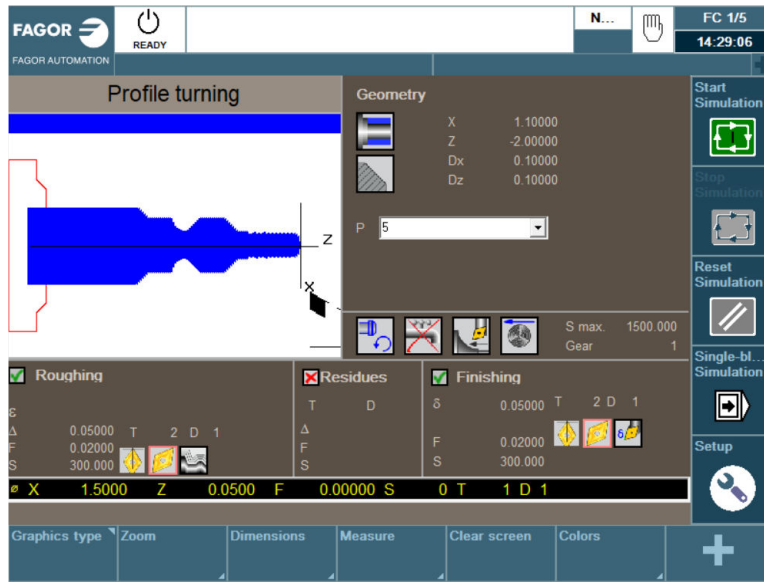
- Fp- This is your roughing feedrate for your axes. In this example we are using a value of 0.01 inches per revolution.
- Δ - This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.05.
- F - This is your roughing feedrate for your axes. In this example we are using a value of 0.02 inches per revolution.
- S - This is your spindle speed. In this example we are using a value of 300 RPM.
- T - Enter is your tool number. In this example we are using T2.
- D - Enter in your Tool Offset Number. In this example we are using D1.
- You can either machine your profile in 2 ways: paraxial machining (1 axis at a time) and machining following the profile. For this example, we will be following the profile. 

9. Define your Finishing Pass:

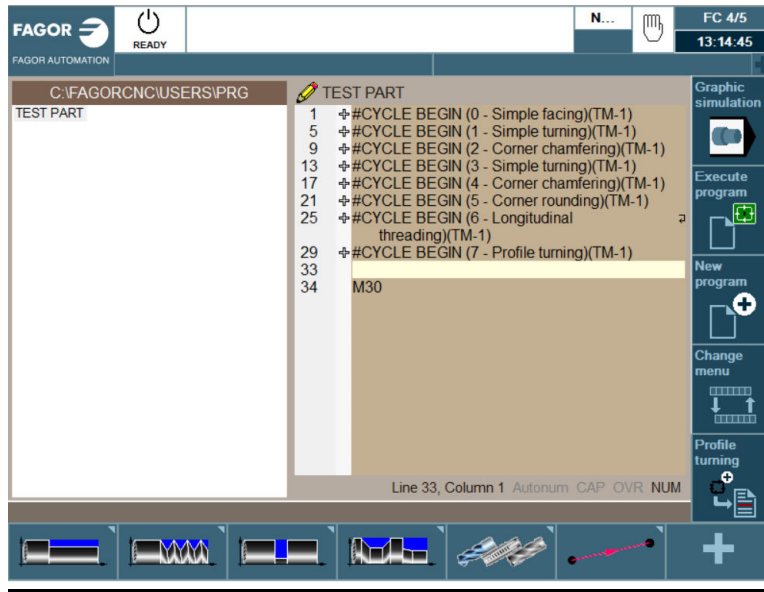
- δ - This is the amount of finishing stock that is going to be left on for the finishing pass. For this example we will be using 0.05.
- F - This is your roughing feedrate for your axes. In this example we are using a value of 0.02 inches per revolution.
- S - This is your spindle speed. In this example we are using a value of 300 RPM.
- T - Enter is your tool number. In this example we are using T2.
- D - Enter in your Tool Offset Number. In this example we are using D1.
- Either a single residual stock may be defined that is applied depending on the cutter's edge or 2 different ones, one for each axis (X, Z). For this example, we will be using

defining just 1. 

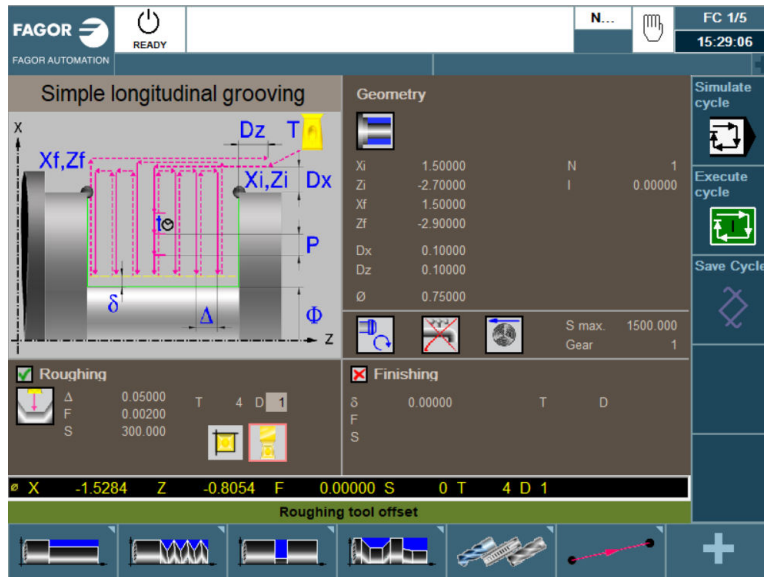
10. Simulate the program: (Refer to page 11)





11. Save your Cycle to your part-program: (Refer to page 12)



GROOVING CYCLE



1. Select  then  Simple Longitudinal Grooving operation.
2. For this operation we will be using Outside Grooving. You can use the space key to toggle from inside to outside grooving when the icon is highlighted blue.
3. Define your part:
 - (Xi, Zi)- This is your starting point of your X and Z coordinates. In this example we will be using X1.5 and Z-2.7
 - (Xf, Zf)- This is your ending point of your X and Z coordinates. In this example we will be using X1.5 and Z-2.9
 - Safety distances (Dx, Dz) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
 - “Ø” This is the diameter that you want to turn the groove down to. In this example we will use 0.75.
 - N- Number of grooves. We will only be making 1 groove in this example.
 - This is the offset in between each groove. We will put it at 0 since we are only making 1 groove.
4. Define your machining conditions:
 - Spindle direction. The space key will toggle from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
 - Coolant on or off. Use the space key to select.

- You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
- SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.

5. Define your machining conditions:

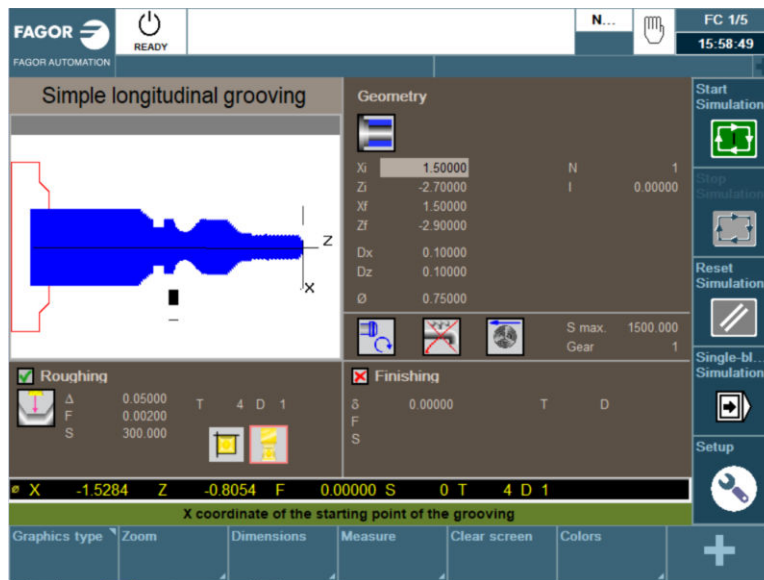
- This symbol means with or without penetration chip removal. We will use penetration



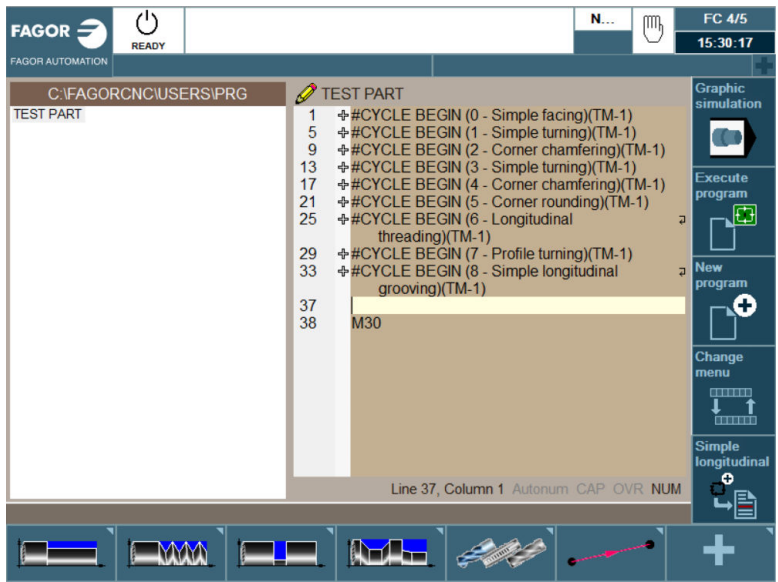
without chip removal for this example

- Δ - This is your turning step-in. How much material you are going to take off per pass. In this example we will be using 0.05.
- F - This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S - This is your spindle speed. In this example we are using a value of 300 RPM.
- T - Enter is your tool number. In this example we are using T4.
- D - Enter in your Tool Offset Number. In this example we are using D1.

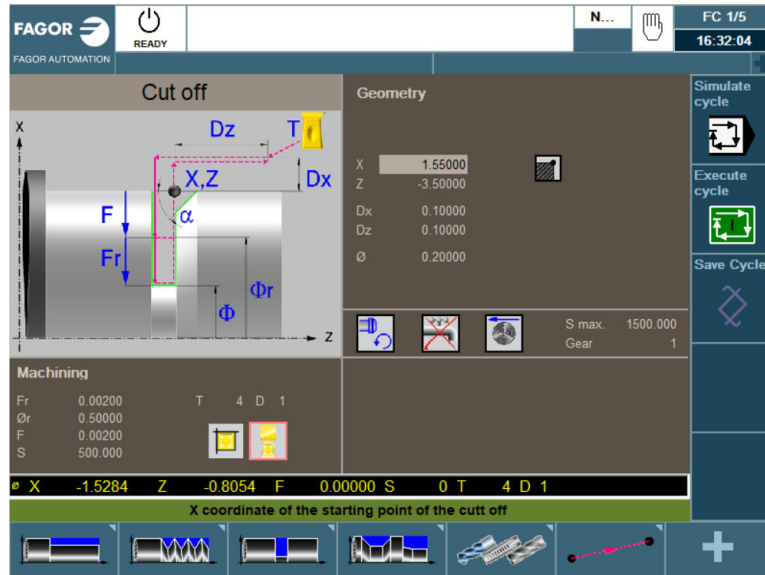
6. Simulate the program: (Refer to page 11)


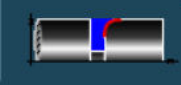


7. Save your Cycle to your part-program: (Refer to page 12)



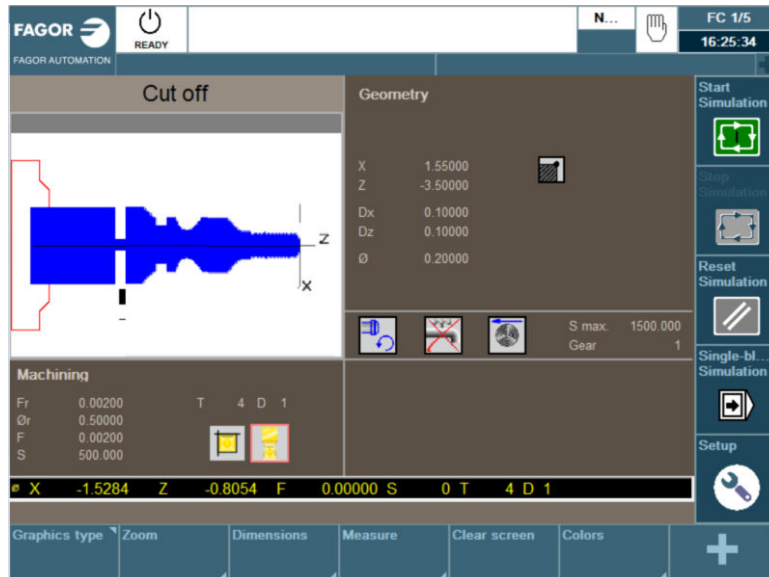
CUT OFF CYCLE



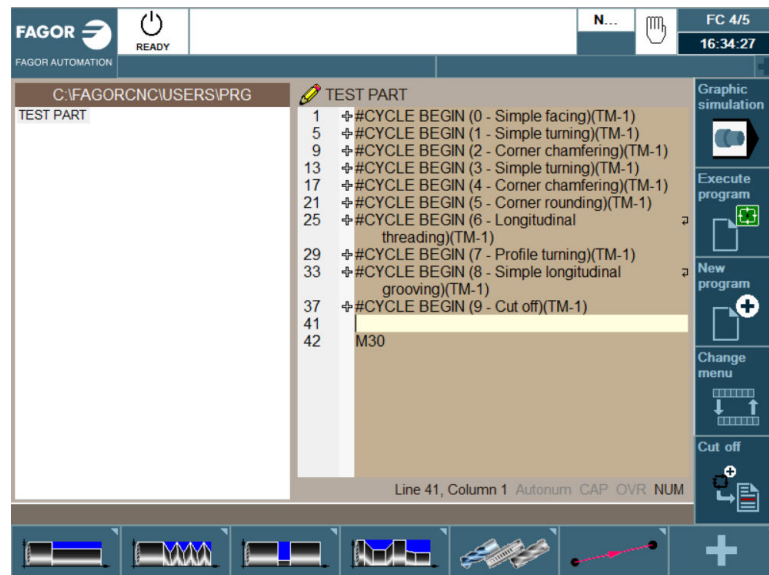
1. Select  then  Cut Off operation.
2. (X, Z)- This is your starting point of your X and Z coordinates. In this example we will be using X1.55 and Z-3.5
3. Safety distances (X, Z) This is the amount your tool will be away from your starting point before beginning its operation. In this example we will be using a value of 0.1 for both axes.
4. Ø - This is the diameter that you want to turn the groove down to. In this example we will use 0.2.
5. Define your machining conditions:
 - Spindle direction. The space key will toggle from clockwise to counter-clockwise. We will be using counter-clockwise in this example.
 - Coolant on or off. Use the space key to select.
 - You can use either RPM or CSS when machining. Use the space key to toggle from one to the other. In this example we will be using CSS.
 - SMAX- This is the maximum spindle speed that you can use while running this canned cycle. In this example we will be using 1500.
6. Define your Roughing Pass:
 - Fr - is your Final slow feedrate. How much material you are going to take off per pass. In this example we will be using 0.002.
 - Ør - This is your Gear ratio diameter for the feedrate. In this example we will be using 0.5.

- F - This is your roughing feedrate for your axes. In this example we are using a value of 0.002 inches per revolution.
- S- This is your Spindle speed. In this example we are using a value of 500.
- T- Enter is your tool number. In this example we are using T4.
- D- Enter in your Tool Offset Number. In this example we are using D1.

7. Simulate the program: (Refer to page 11)



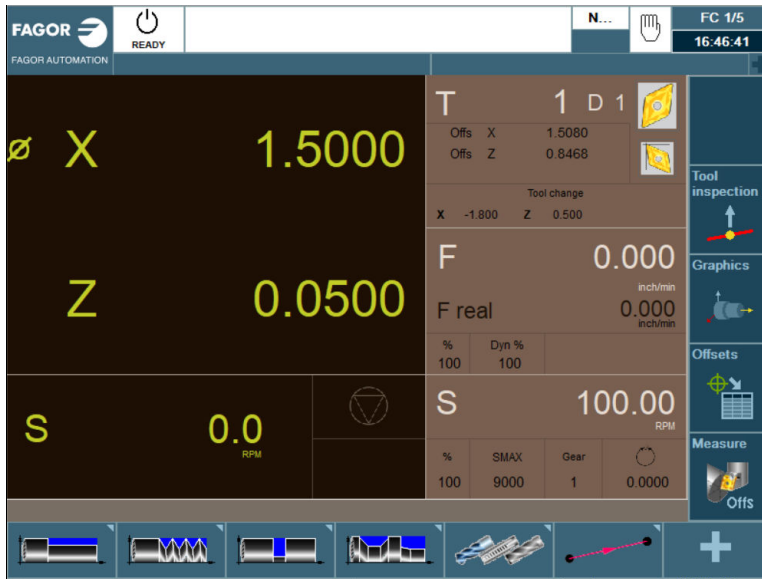
8. Save your Cycle to your part-program: (Refer to page 12)



ZERO OFFSETS

G54, G55, G56, G57, and up to 99 are our absolute zero offsets. This feature allows us to store a part zero position into a table. With this option you are able to recall your part zero position without having to touch off on your part again. This feature is useful when having parts that are commonly machined. To setup your Zero Offsets perform the following task:

1. From the Jog Screen setup your part zero position



2. Select the Offsets icon



3. Select the OFFSET you want to apply the new zero and press the Enter key to calculate de new position. X and Z on the bottom are now the exact value of their position away from home. Press cycle start key to confirm it.

Start to apply OFFSET	
Enter to calculate it	
OFFSETS	CLAMPS
G54 (G159=1)	1
G55 (G159=2)	2
G56 (G159=3)	3
G57 (G159=4)	4
G58 (G159=5)	5
G59 (G159=6)	6
G159=7	7
G159=8	8
G159=9	9
X	Z
0.0115	0.0000

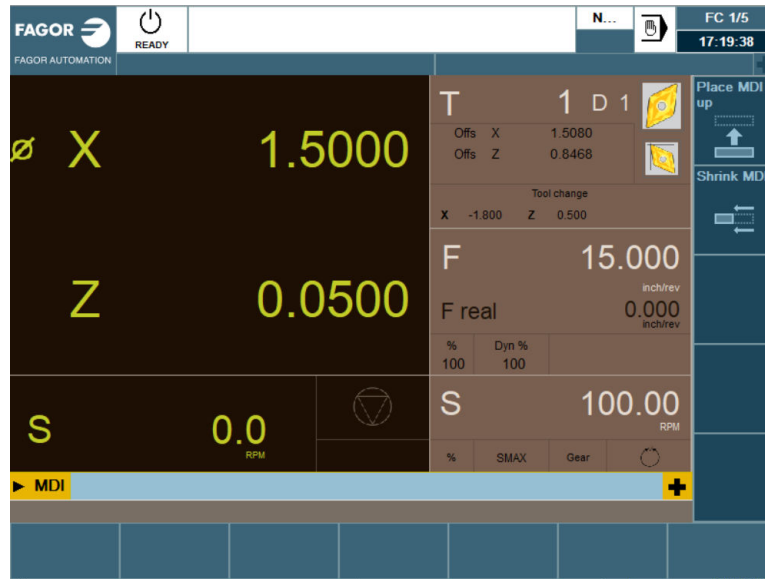
MDI MODE

MDI feature allows you to do one or more functions at a time. It allows you to move more than one axes at a time, start the spindle, set the feed, insert G-Codes, etc. To operate in MDI Mode, execute the following instructions:

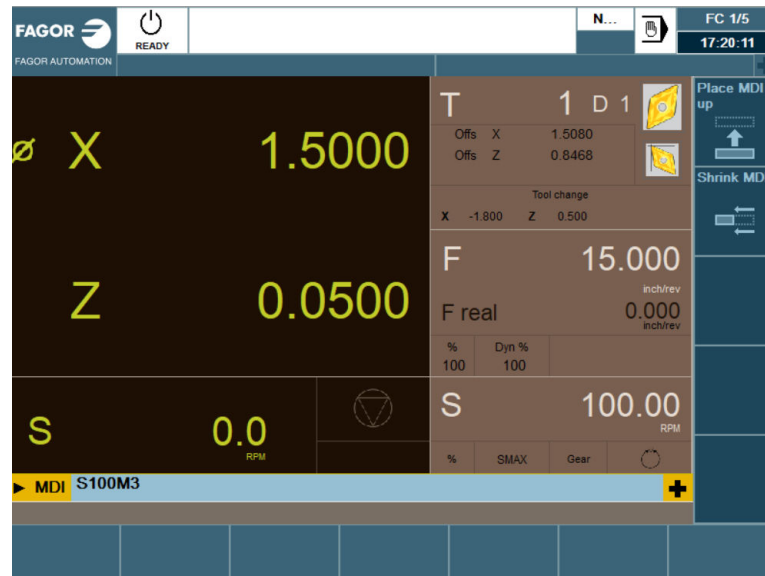


1. From the Jog Screen select the MDI key

*In this example we will be turning on the spindle at a clockwise rotation at 100 rpm
The CNC will display a light blue bar at the bottom of the screen*



2. Type in S100 M3



3. Press Cycle Start 