

Skill Building Exercise



Durham College
School of Technology

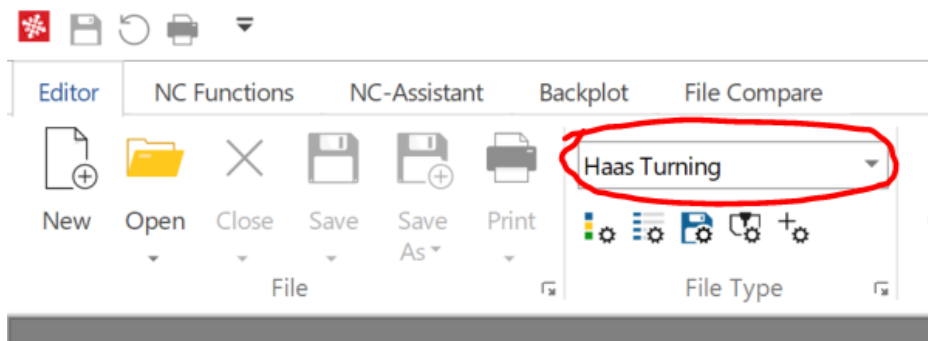
Preparation

Create a folder on H-drive (or on your USB stick/drive if you prefer) called **CAM4132**

In that folder create a folder whose name is the same as **your student number**. (for example, 100001234)

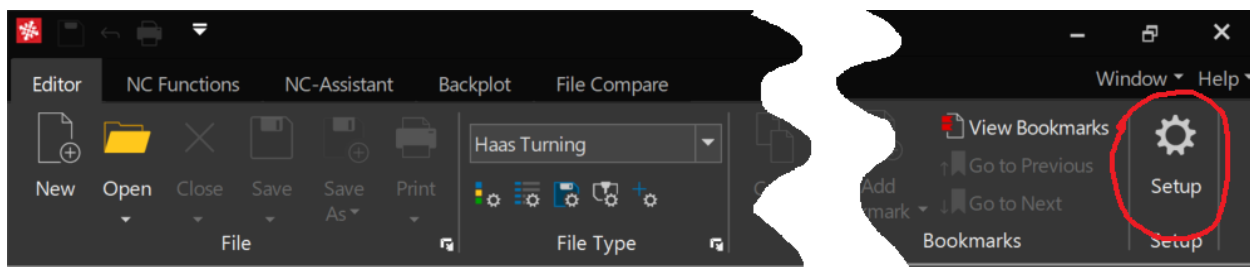
In that folder create another folder called **exercise_4**

Open Cimco edit.

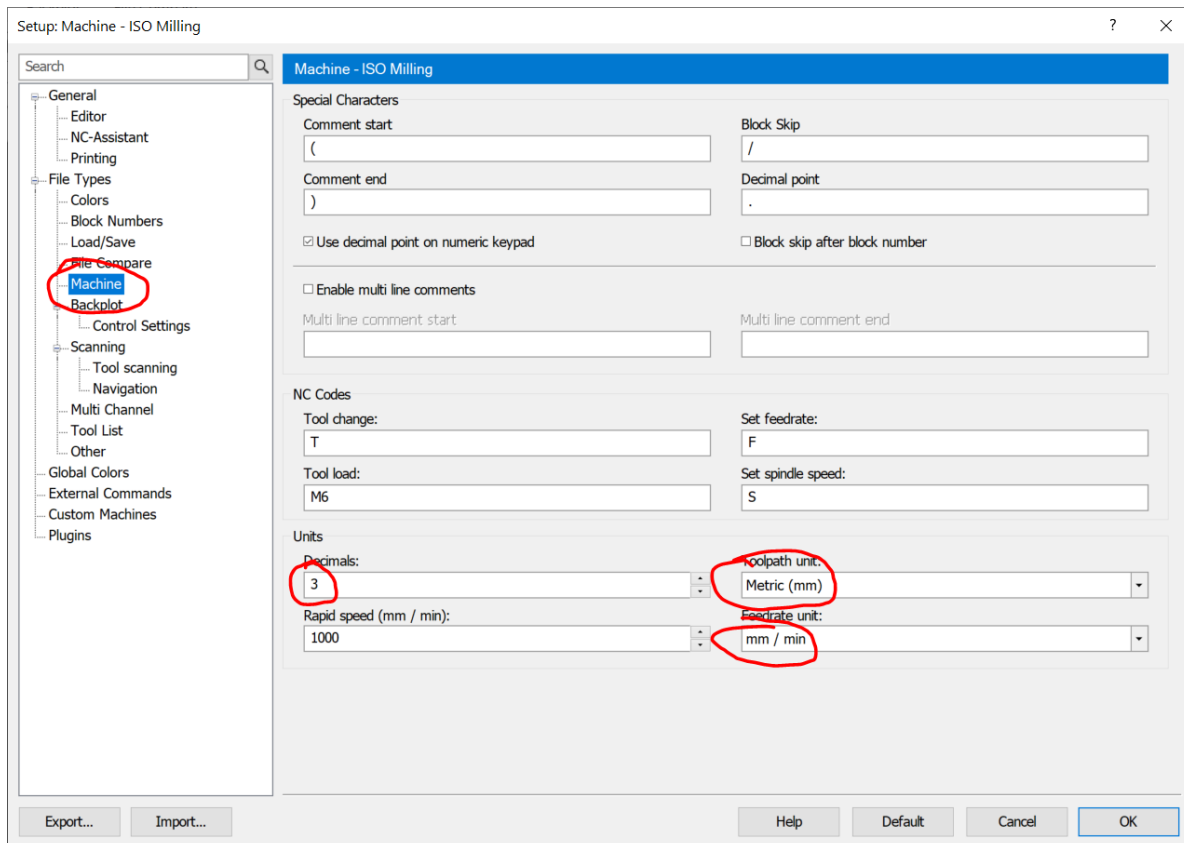


Make sure you set the program type to a turning program.

Set the units to match the program, in this case inches.

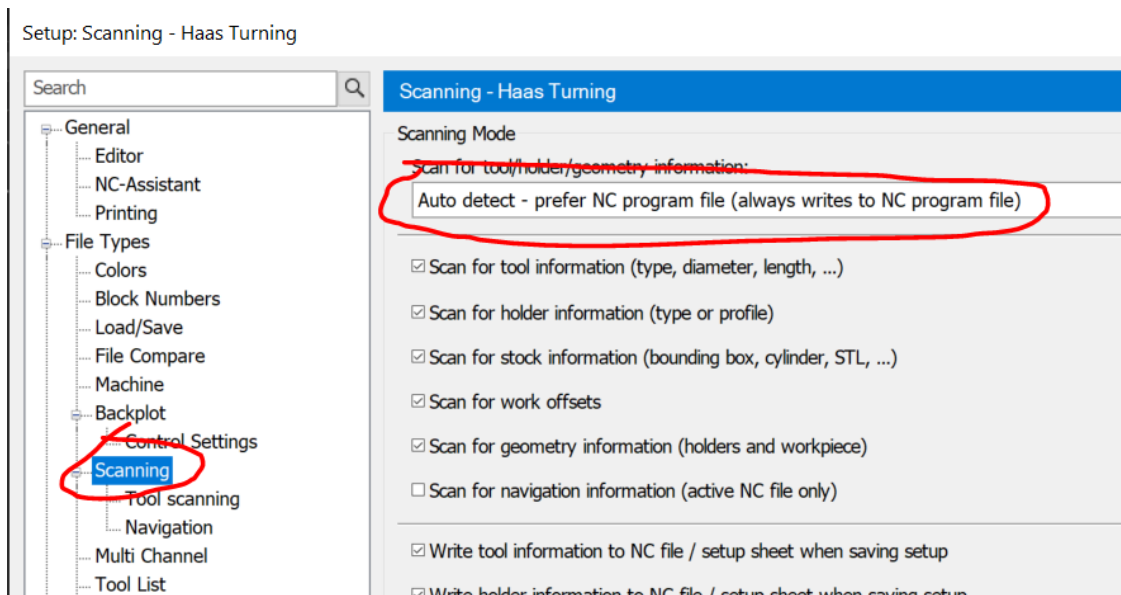


Click on the Setup panel at the far right end of the ribbon menu.



Select Machine and then check that Toolpath Units are mm and the Feedrate Unit is Unit/rev.

Click on the Scanning setting.



Ensure that Auto detect scans from and writes setting to the NC file.

Activity

Attached to this exercise you will find the following documents:

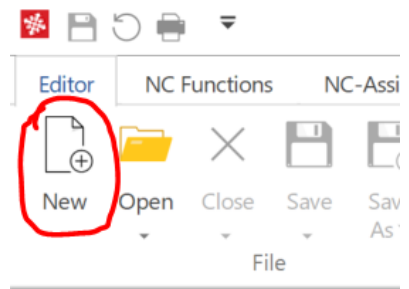
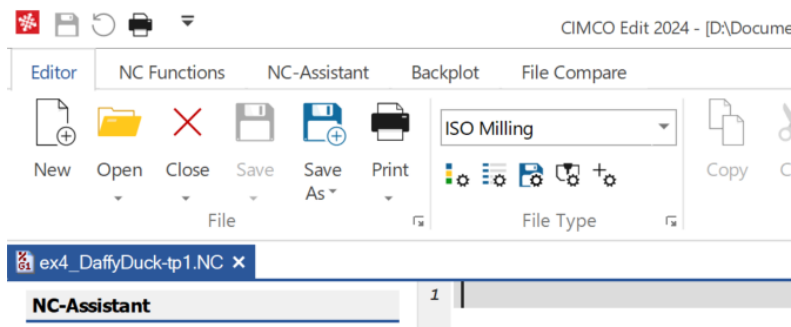
- exercise_4.pdf
- EX4-L1-DG1.pdf
- EX4-L1-SE1.pdf
- EX4-L1-TL1.pdf
- chuck_assy3IN.stl
- chuck_assy2.75IN.stl

Save these files into the **exercise_4** folder. (NOT INTO DOWNLOADS!!)

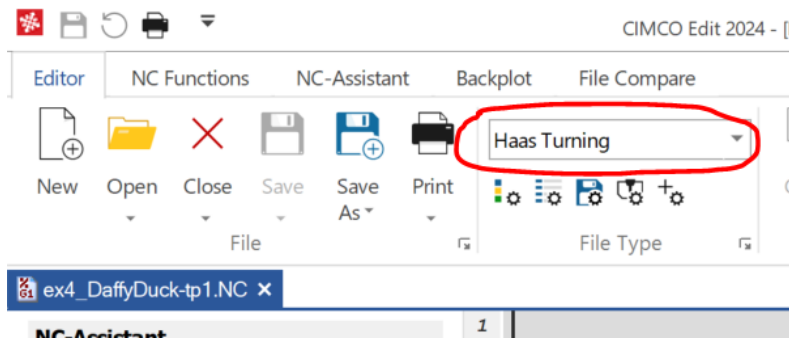
With these documents it is possible to use CIMCO Edit to create a program for a CNC lathe.

Program 1

Using Cimco Edit, open a new file and save it as **ex4_<YourLastName>-tp1.nc** in the **exercise_4** folder just created.



The first task will be to change the **File Type** to reflect the fact that this is a turning program.



Develop the Program

Begin by adding program start/end code using the NC Assistant.

Insert: Program Start and End

Parameters for 'Program Start and End'

1032

Program number [1000 - 6000]

EX4

Program description

2500

Max spindle speed

* = Optional parameter

Default

Cancel

OK

Then add a tool change (with constant surface speed):

```

%
O1032
(-- EX4 --)
G21 G40
G54
G50 S2500
M05
M30
%
```

← Add tool change here

Insert: Tool change with Constant Surface Speed

Parameters for 'Tool change with Constant Surface Speed'

OD80L Tool description

01 Tool number

01 Tool offset number

30 Constant surface speed (units/min)

* 03 Spindle direction [03 - 04]

* = Optional parameter

Default Cancel OK

```

%
O1032
(-- EX4 --)
G21 G40
G54
G50 S2500
(-- OD80L --)
(-- CONSTANT SURFACE SPEED --)
N1 T0101
G18 G99
G96 S30 M03
(-- START MOTION --)
|
} Add code in here
(-- END MOTION --)
M09
G28 U0. W0.
M01
M05
M30
%
```

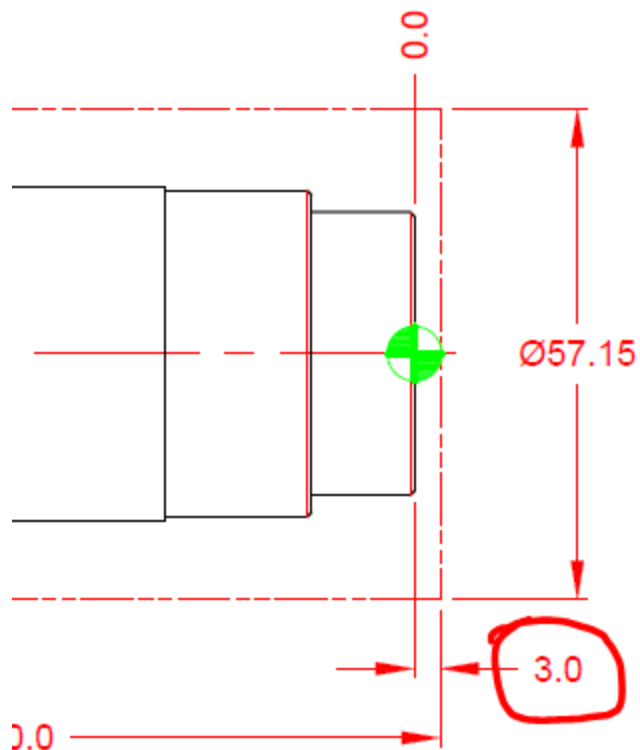
Move to the corner of the stock with the tool nose near the corner of the stock:

X60. Z3

According to the setup sheet, the first operation will end face the bar:

Machining Operations	Tool Description	Tool /Off	R	T
1: Rough Face $\varnothing 33$ +.5 on length	80° L/H turning tool	01 01	.8	3
2: Rough Turn $\varnothing 39$ +.5 on diameter & +.5 on length				
Rough Turn $\varnothing 38$ +.5 on diameter & +.5 on length				
Rough Turn $\varnothing 33$ +.5 on diameter & +.5 on length				
3: Finish Face $\varnothing 33$	55° L/H turning tool	03 03	.4	3
Finish Turn $\varnothing 33$ to diameter and length				
Finish Turn $\varnothing 38$ to diameter and length				
Finish Turn $\varnothing 39$ to diameter and length				
Finish Turn $\varnothing 53.98$ to diameter and length				
4: Part Off	0.125 parting tool	05 05	.2	3

If we look at the setup drawing we can see there is 3mm of material to end face.



If we consider cutting that in 3 passes then the first operation will cut 1mm per pass.

There are three ways to program this cut:

1. Manually
2. Using basic facing cycle, G94
3. Using advanced cycle, G72

As there are not that many cuts to make, option three is a bit overly complex.

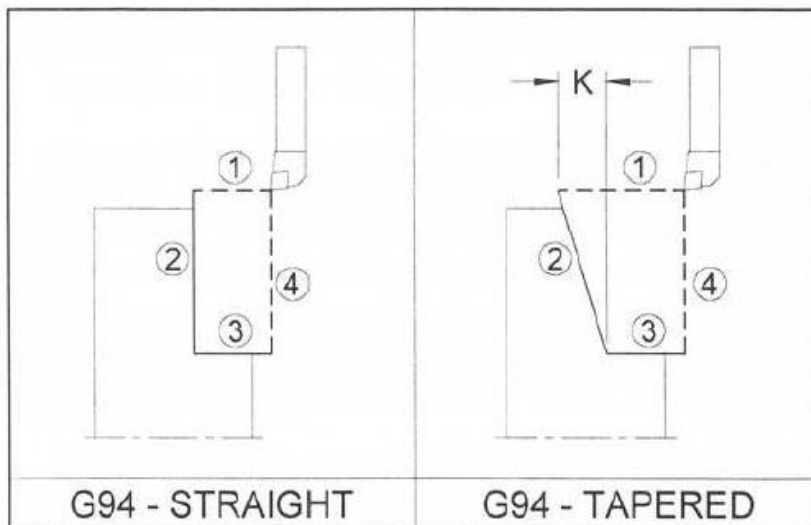
That leaves the G94 cycle, or manual coding.

The G94 cycle requires less information, so we shall use it.

G94 X(U).. Z(W).. F..

where ...

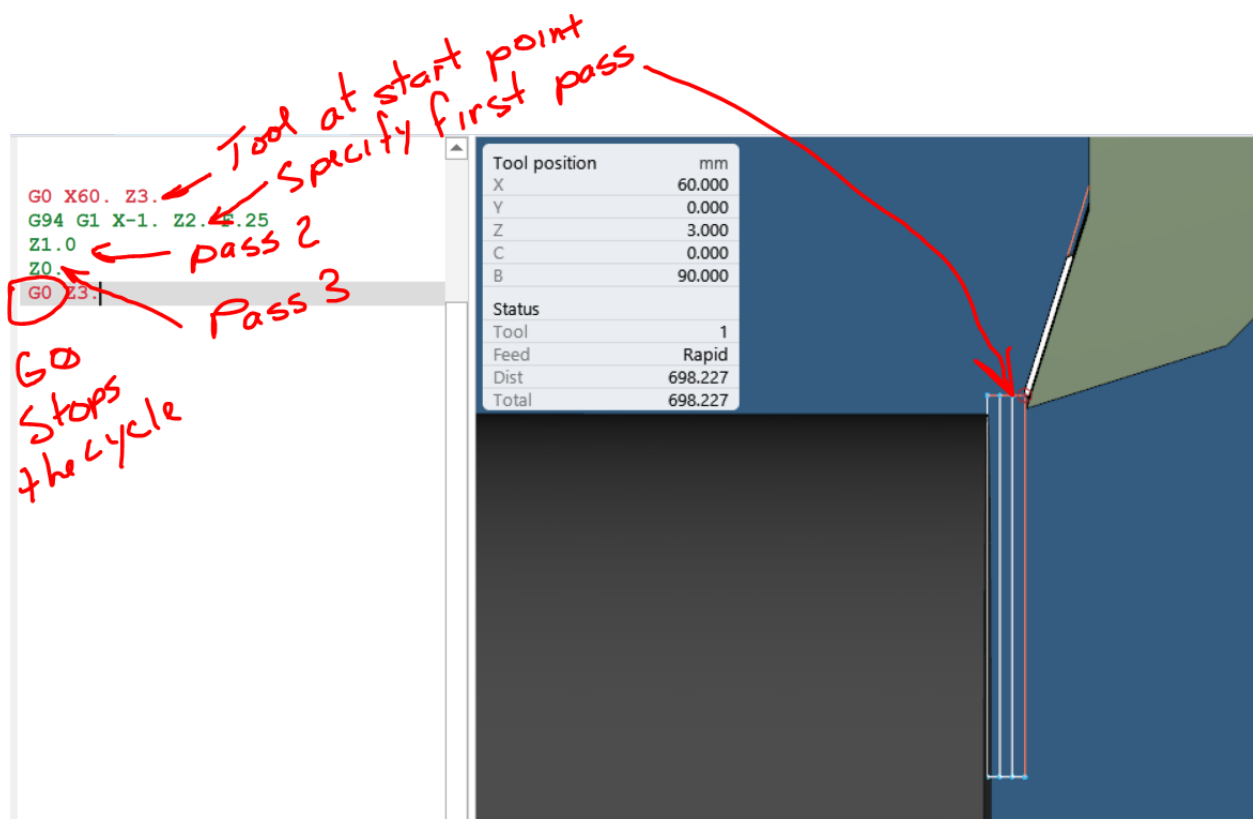
X = Diameter to be cut
Z = End of cut in Z position
F = Cutting feedrate (usually *in/rev* or *mm/rev*)



G94 cuts in a box shape so we only need to instruct it how far in Z we need to travel and it fills in the rest.

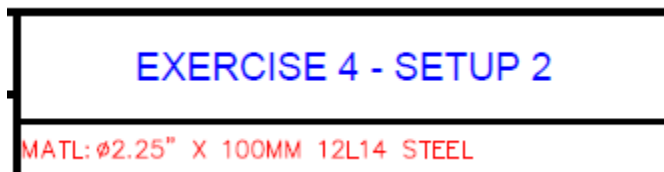
So add the following lines to the code:

```
G0 X60. Z3.  
G94 G1 X-1. Z2. F.25  
Z1.0  
Z0.5  
G0 Z3.
```



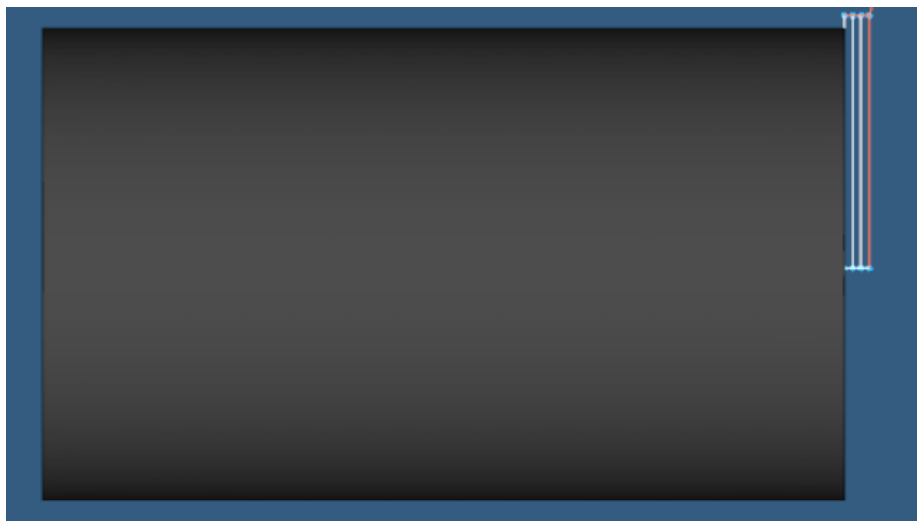
Now we can go into Backplot and see that it looks like.

Add a Stock as a cylinder per the drawing.



Properties	
Name	Stock Cylinder
Quality	Very High
Translation Z:	3.0000
Outer Diameter	57.1500
Inner Diameter	0.0000
Length	100.0000

After playing the Backplot it should appear similar to the following.



The next operation, according to the setup sheet, is to rough the various diameters.

Machining Operations	Tool Description	Tool /Off	R	T
1: Rough Face $\varnothing 33$ +.5 on length	80° L/H turning tool	01 01	.8	3
2: Rough Turn $\varnothing 39$ +.5 on diameter & +.5 on length				
Rough Turn $\varnothing 38$ +.5 on diameter & +.5 on length				
Rough Turn $\varnothing 33$ +.5 on diameter & +.5 on length				
3: Finish Face $\varnothing 33$	55° L/H turning tool	03 03	.4	3
Finish Turn $\varnothing 33$ to diameter and length				
Finish Turn $\varnothing 38$ to diameter and length				
Finish Turn $\varnothing 39$ to diameter and length				
Finish Turn $\varnothing 53.98$ to diameter and length				
4: Part Off	0.125 parting tool	05 05	.2	3

Again, as was demonstrated in the tutorial, we could manually compute all the roughing passes, or, use a roughing cycle.

There are two roughing cycles that could be used:

1. G90 straight cutting cycle
2. G71 repetitive roughing cycle

Of these, the G90 cycle is limited to just straight line cutting.

Since we have fillets in this profile, the G71 will be necessary.

After the facing cycle completes, we might code a move back to the start point:

G0 X100 Z100

Then add the G71 code.

The G71 cycle can be input in either of two formats:

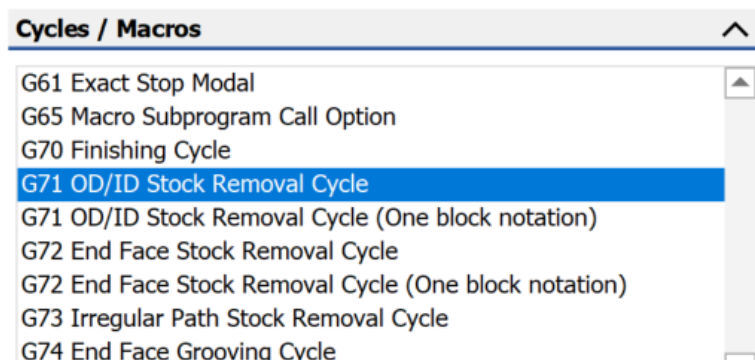
- Double line format
- Single line format

G71 U.. R..
G71 P.. Q.. U.. W.. F.. S..

G71 P.. Q.. I.. K.. U.. W.. D.. F.. S..

The style you can use depends on the type of controller that the machine has.

The Leadwell lathe uses the two line format so that is what we must code.
Cimco Edit has an NC assistant macro for this:



Double click it and fill in the table as seen below.

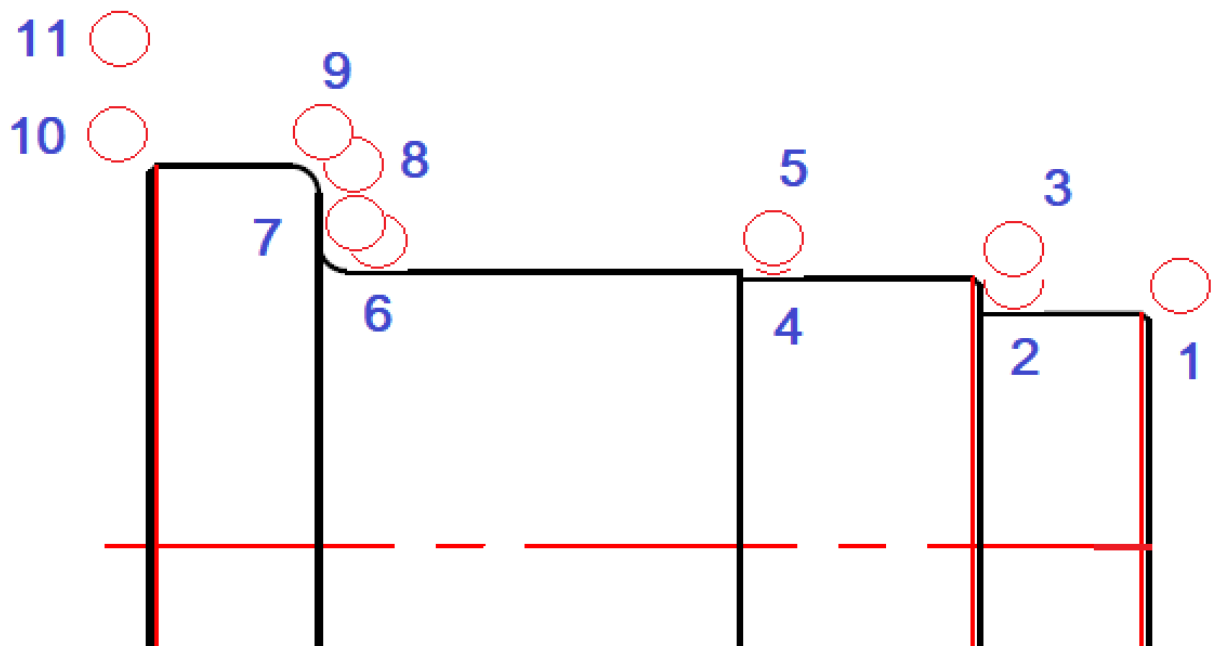
A screenshot of the 'Insert: G71 OD/ID Stock Removal Cycle' dialog box. The dialog box has a title bar with a close button (X). Inside, there is a section titled 'Parameters for 'G71 OD/ID Stock Removal Cycle''. Below this title is a table of parameters. Each row consists of a parameter value in a text box, an asterisk (*) indicating it is optional, and a description. The parameters are: Depth of cut for each pass of stock removal, positive radius (2.0); Retract height for each pass of stock removal (0.5); Feedrate [>= 0.0001] (0.25); X-axis size and direction of G71 rough pass allowance (empty); Z-axis size and direction of G71 rough pass allowance (empty); Starting block number (10); Ending block number (60); Spindle speed [>= 1] (30); Tool and offset (0101); X-axis size and direction of G71 finish allowance (0.5); and Z-axis size and direction of G71 finish allowance (0.5). At the bottom left, there is a legend: '* = Optional parameter'. At the bottom right, there are three buttons: 'Default', 'Cancel', and 'OK'.

Now these lines have been added:

```
G0 Z3.  
X100.  
Z100.  
X60. Z3.  
N2 G71 U2.0 R0.5  
N3 G71 F0.25 P10 Q60 S30 T0101 U0.5 W0.5
```

The parameter P and Q refer to a series of locations that define the part profile.

For this part:



We can make a chart of these values:

	X	Z	R
1	33	3	
2	33	-12	
3	38	-12	
4	38	-29	

5	39	-29	
6	39	-57	
7	43	-59	R2
8	49.98	-59	
9	53.98	-61	R2
10	53.98	-74	
11	60	-74	

Then they can be added to the program:

```

X60. Z3.
G71 U2.0 R0.5
G71 F0.25 P10 Q60 S30 T0101 U0.5 W0.5
N10 G0 X33.
G1 Z-12.
X38.
Z-29.
X39.
Z-57.
G02 Z-59. X43. R2.
G01 X49.98
G03 Z-61. X53.98 R2.
G01 Z-74
N60 X60.
G0 X100
Z100

```

After the rough cut, a finish cut can be defined.

Use the G70 finish cycle.

This is simple,

G70 P.. Q.. F.. S..

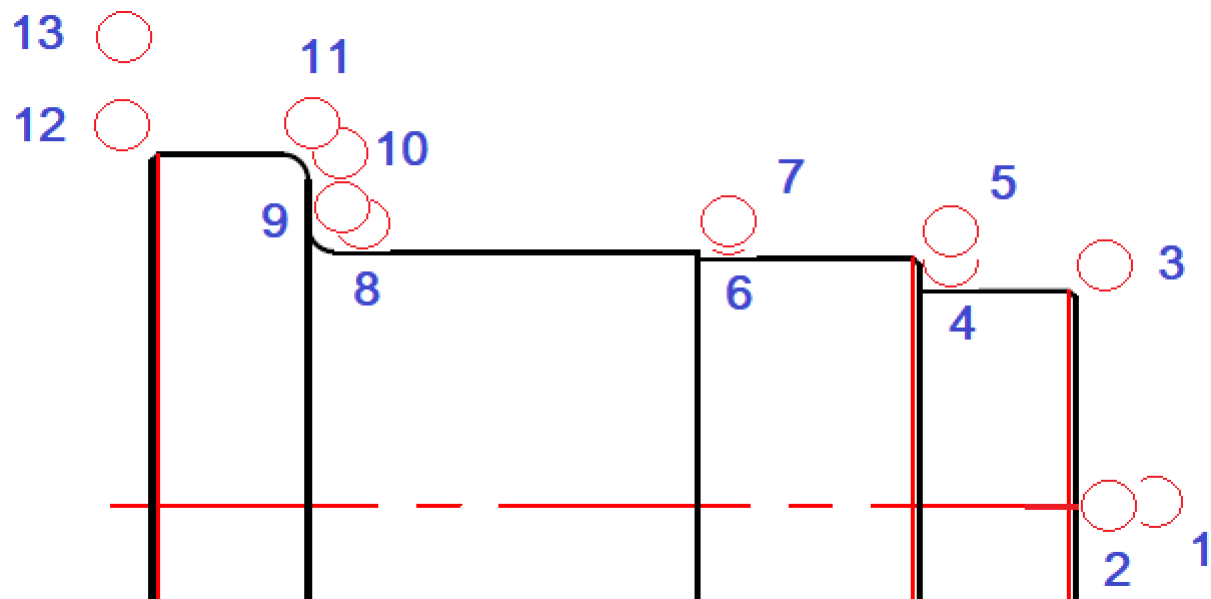
Once again, the P and Q define a boundary.

Most often, the same boundary is used for finishing and roughing.

In this case, the two are slightly different.

The finish cut needs a pass up the end face as well as the diameters.

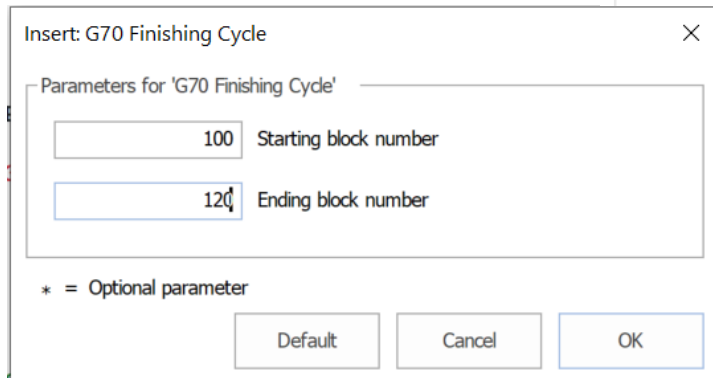
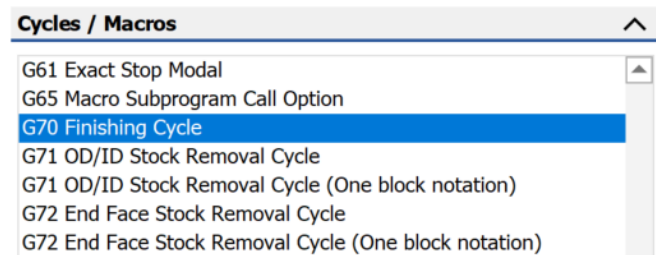
So, a new table can be created.



	X	Z	R
1	-.1	3	
2	-.1	0	
3	33	0	
4	33	-12	
5	38	-12	
6	38	-29	
7	39	-29	
8	39	-57	
9	43	-59	R2
10	49.98	-59	

11	53.98	-61	R2
12	53.98	-74	
13	60	-74	

Add the G70 using the Cimco edit macro:



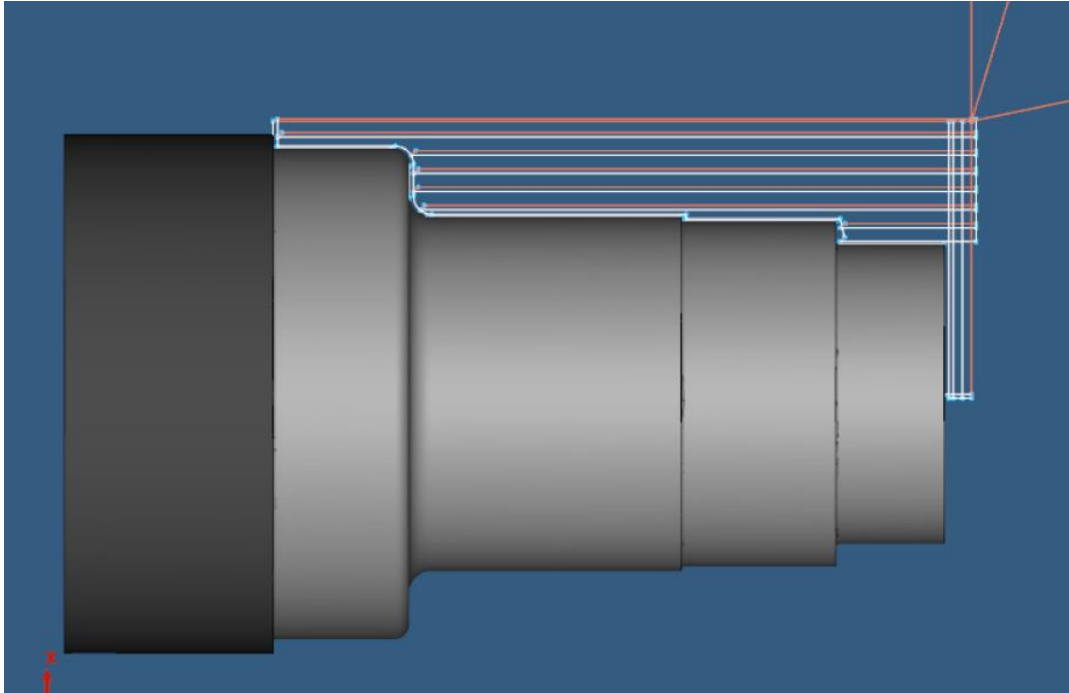
Then add the profile.

The part should appear somewhat like that seen below.

```

Z100
X60. Z3.
N61 G70 P100 Q120
N100 G0 X-.1
G1 Z0.
X33.
Z-12.
X38.
Z-29.
X39.
Z-57.
G02 Z-59. X43. R2.
G01 X49.98
G03 Z-61. X53.98 R2.
G01 Z-74
N120 X60.
G0 X100
Z100
(-- END MOTION --)

```



The last task is to part it off.

The part off tool is 0.125" wide, and the tracking point is on the chuck side of the tool.

So, make sure the blade goes to the correct Z-value to cut the part to the correct size.

```
(-- START MOTION --)
X60. Z-74.175
G1 X-.1 F.2
G0 X60.
M09
```

Add this after the finish cut.

Now the tools should be added as more than one are requested on the setup sheet.

Use the Cimco edit tool change (with constant surface speed) macro.

Cycles / Macros ^

Program Start and End

Tool change with Constant Surface Speed

Tool change with Constant RPM

Program comment

G00 Rapid Motion Positioning (XZ)

G00 Rapid Motion Positioning (XYZCB)

G01 Linear Interpolation Motion (XZ)

Insert: Tool change with Constant Surface Speed X

Parameters for 'Tool change with Constant Surface Speed'

OD55L

Tool description

03

Tool number

03

Tool offset number

30

Constant surface speed (units/min)

* 03

Spindle direction [03 - 04]

* = Optional parameter

Default

Cancel

OK

Insert: Tool change with Constant Surface Speed X

Parameters for 'Tool change with Constant Surface Speed'

PART OFF

Tool description

05

Tool number

05

Tool offset number

30

Constant surface speed (units/min)

* 03

Spindle direction [03 - 04]

* = Optional parameter

Default

Cancel

OK

The program should resemble the following:

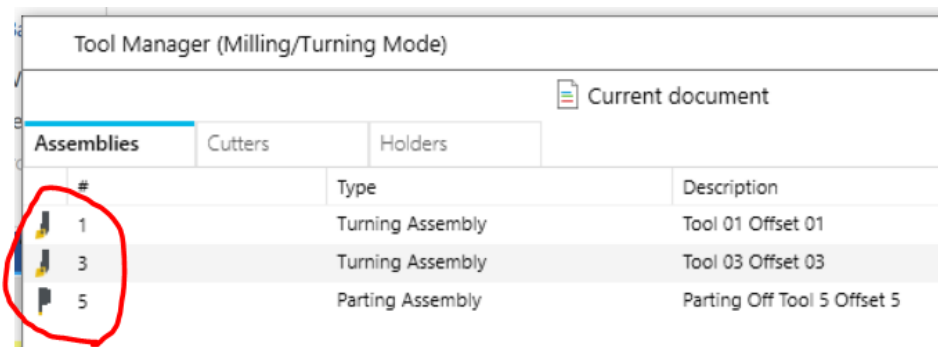
```
%  
O1032  
(-- EX4 --)  
G21 G40  
G54  
G50 S2500  
(-- OD80L --)  
(-- CONSTANT SURFACE SPEED --)  
T0101  
G18 G99  
G96 S30 M03  
(-- START MOTION --)  
G0 X60. Z3.  
G94 G1 X-1. Z2. F.25)  
Z1.0  
Z0.5  
G0 Z3.  
X100.  
Z100.  
X60. Z3.  
G71 U2.0 R0.5  
G71 F0.25 P10 Q60 S30 T0101 U0.5 W0.5  
N10 G42 G0 X33.  
G1 Z-12.  
X38.  
Z-29.  
X39.  
Z-57.
```

G02 Z-59. X43. R2.
G01 X49.98
G03 Z-61. X53.98 R2.
G01 Z-74
N60 X60.
G40 G0 X100
Z100
(-- OD55L --)
(-- CONSTANT SURFACE SPEED --)
T0303
G18 G99
G96 S30 M03
(-- START MOTION --)
X60. Z3.
N61 G70 P100 Q120
N100 G42 G0 X-.1
G1 Z0.
X33. K.5
Z-12.
X38. K.5
Z-29.
X39.
Z-57.
G02 Z-59. X43. R2.
G01 X49.98
G03 Z-61. X53.98 R2.
G01 Z-74
N120 X60.
G40 G0 X100

```

(-- PART OFF --)
(-- CONSTANT SURFACE SPEED --)
T0505
G18 G99
G96 S30 M03
(-- START MOTION --)
X60. Z-74.175
G1 X-.1 F.2
G0 X60.
(-- END MOTION --)
M05 M09
G28 U0. W0.
M30
%
```

Set up the tool definitions in Cimco Edit.



Now add the lathe chuck.

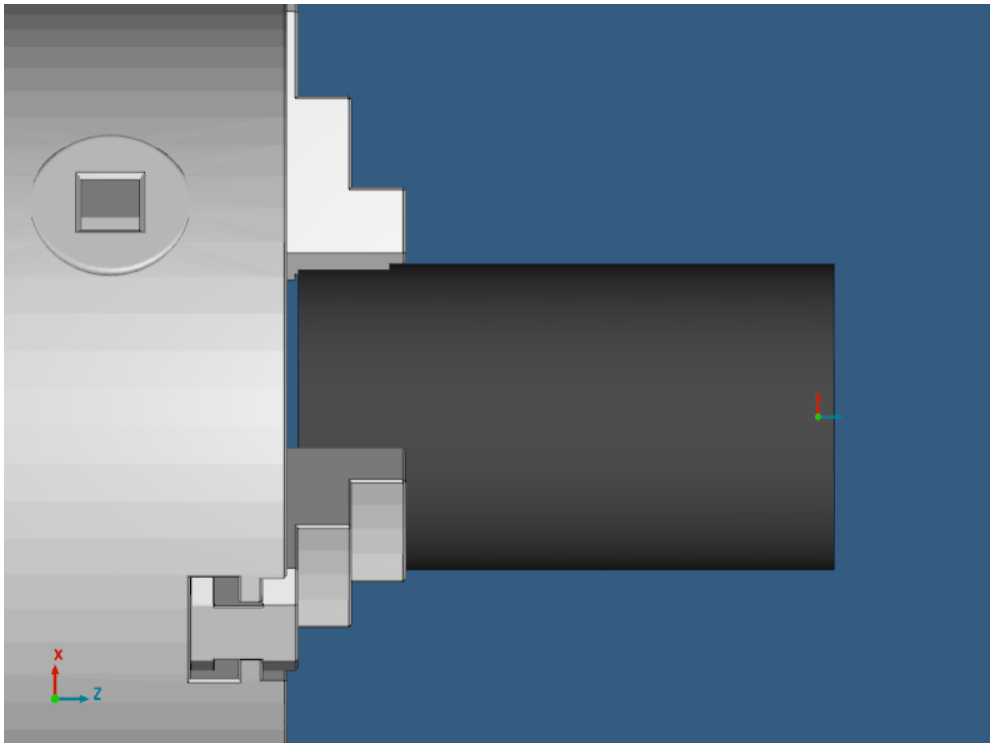
Use the file:

J:\unfs2\system\caddslib\cam\tool_lib\Cimco_Edit\chuck_assy-2.25in.stl

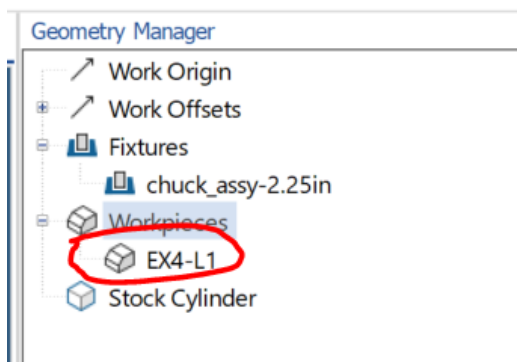
It will locate the face of the jaw pocket at Z0.

Move it Z-80.0 to locate it correctly.

It should appear somewhat like the following:

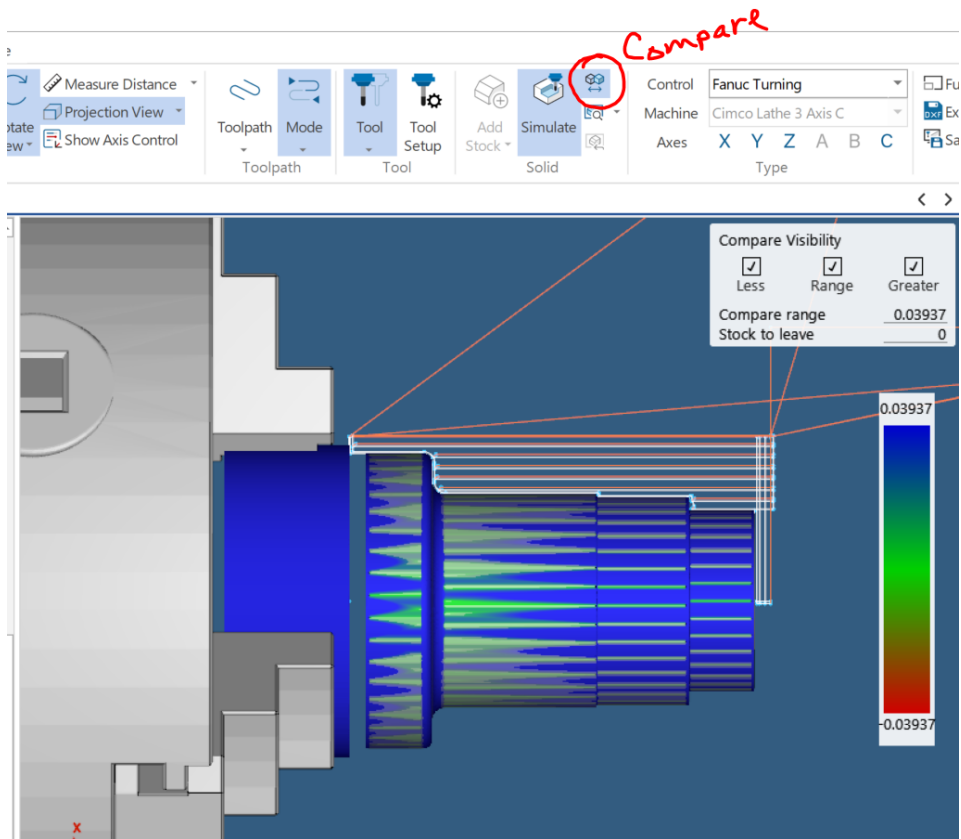


You can also add the supplied model of the part itself as a workpiece.



It will locate directly at the Z0 of the setup, so no further input is required.

With these two items in place, you can run the simulation and then check to see if there is any uncut material, or, if it has been undercut.



Submission

Create a compressed copy of the **exercise_4** folder. (Zip file)
Submit the zip file to the DC Connect dropbox for evaluation.