# Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
</table>
| OVERVIEW OF THE MANUAL GUIDE | 5
| i PROGRAMMING PROCESS                                                   | 5    |
| Structure of a MANUAL GUIDE i Program                                   | 5    |
| Structure of an Operation                                               | 5    |
| Fixed Form Sentences                                                    | 6    |
| DEFINING ARBITRARY PROFILES                                            | 8    |
| Start Point                                                             | 8    |
| Lines                                                                   | 9    |
| Two Lines and a Corner Radius                                           | 10   |
| Closing the Profile                                                    | 10   |
| Saving the Arbitrary Profile                                           | 11   |
| Example Profile                                                        | 11   |
| Modifying an Arbitrary Profile                                         | 26   |
| PLANNING                                                                | 28   |
| Workpiece Drawing                                                      | 28   |
| Part Blank Size                                                        | 28   |
| Setup                                                                  | 28   |
| Workpiece Program Zero                                                 | 28   |
| Safe Approach Position                                                 | 29   |
| Visualize Operations                                                   | 29   |
| Tooling and Feeds and Speeds                                           | 30   |
| Drilling Cycle                                                         | 30   |
| Tolerance Analysis                                                     | 30   |
| SETTING TOOL DATA                                                      | 32   |
| T0101 Roughing Tool                                                    | 33   |
| T0202 Finishing Tool                                                   | 37   |
| T0909 3/8” Drill                                                       | 42   |
| CREATING THE PROGRAM                                                   | 45   |
| START THE PROGRAM                                                      | 47   |
| CYLINDER BLANK FIGURE (G1900) PARAMETERS                               | 50   |
| DEFINE THE MATERIAL BLANK                                              | 51   |
| TURNING OUTER ROUGH (G1120) PARAMETERS                                 | 53   |
| OPERATION 1 - ROUGH TURNING                                            | 58   |
| Start Tool T0101                                                       | 58   |
| Turning (Outer Rough) Cycle                                            | 59   |
| End Tool T0101                                                         | 71   |
| VERIFY OPERATION 1                                                      | 73   |
DRILLING CYCLE PARAMETERS

OPERATION 2 - DRILLING

Start Tool T0909

Drilling Cycle

End Tool T0909

VERIFY OPERATION 2

TURNING OUTER FINISH (G1126) PARAMETERS

OPERATION 3 - FINISH TURNING

Start Tool T0202

TURNING (OUTER FINISH) Cycle

TURNING FACE FINISH (G1128) PARAMETERS

OPERATION 4 - FINISH FACING

TURNING (FACE FINISH)

End Tool T0202

END PROGRAM

VERIFY OPERATIONS 3 & 4

DETAILED VERIFICATION

APPENDIX

Fixed Form Sentences

Tool Data

Tool Geometry Offsets

Program Listing
Whether machining a part with a manual lathe or using a CNC turning center, much of the planning and process development is the same.

1. Study the workpiece drawing to visualize the part in 3-dimensions, noting material type, blank size, geometric features and accuracy and tolerance requirements.
2. Visualize the sequence of operations to machine the part.
3. Select the tools required for each machining operation.
4. Calculate the feeds and speeds for each tool.
5. Design the workholding to hold the part during machining.
6. Generate and verify the program (not required for manual machining).
7. Setup the machine (tooling and workpiece coordinate system zero).
8. Verify the program on the machine.
9. First part inspection (using trial machining for tight tolerances).
10. Run production.

Programming in G-code, a CAM programming system or conversational programming all require you to think in machining operations and only the physical coding of the part program is different. Manual machinist and Conversational Programmers often have the advantage over office bound G-code and CAM programmers in that they can collect the workpiece blanks and tooling right at the machine, which simplifies the visualization and decision making processes.

**Structure of a MANUAL GUIDE i Program**

MANUAL GUIDE i programs follow are structured and programmed using the sequence of operations. The overall structure is:

START PROGRAM (Program number, program comment, material blank definition)

OPERATION 1 (Start tool, machining cycle, end tool)

OPERATION 2 (Start tool, machining cycle, end tool)

OPERATION 3 (Start tool, machining cycle, end tool)

..  

OPERATION n (Start tool, machining cycle, end tool)

END PROGRAM

**Structure of an Operation**

An operation in MANUAL GUIDE i also follows a structure. A machining cycle can be further broken down into cutting conditions, cutting detail and geometry. The overall structure is:

START TOOL (Restart blocks, tool change and tool offset activation, spindle start, coolant etc)

CUTTING CONDITIONS (approach position, initial feed rate etc)

CUTTING DETAIL (optional cutting condition details)

GEOMETRY (geometry of the shape to be machined)

END TOOL (move to tool change position, coolant off, spindle stop, optional stop etc).
Fixed Form Sentences
Fixed Form Sentences are simply snippets of G-code used to control the CNC machine auxiliary functions. The cycles in MANUAL GUIDE i are responsible for generating CNC axes motion to machine the workpiece features. Fixed Form Sentences are responsible for tool changes, selecting tool offsets, activating tool offsets, starting and stopping the spindle, starting and stopping coolant and the other machine auxiliary functions.

Fix Form Sentences may be provided and installed by the machine tool builder, machine tool distributor or customized by a qualified person at the end-user to suit their particular programming style. The person writing the part program does not have to fully understand the G-code that is inside the Fixed Form Sentence, just how to insert them in the appropriate places in their programs. Samples for a simple horizontal lathe are included in the appendices of this document.

Fixed Form Sentences may have some items that must be modified after insertion. For example, the START TOOL CSS Fixed Form Sentence used in this example requires the Conversational Programmer to specify the tool/offset number (T), the initial spindle speed (S in the G97 block) in RPM at the approach diameter, the approach diameter (X) and the CSS cutting speed (S in the G96 block) in surface feet per minute (sfm).

When the START TOOL CSS Fixed Form Sentence is inserted, the T, S, X and letter addresses are displayed with red question marks (?). Each of these words must be selected in turn and values entered.

The T-value will typically be the turret station number followed by the same number for the tool offset. So for turret station number 1 the value is T0101.

The value of the S-word in the G96 CSS activate block must be calculated based on the tool manufacturer’s recommendations.

The approach position will be a little larger than the blank diameter, in the example we choose 4.2-inches which is 0.1-inches away from the part on either side. Since MANUAL GUIDE i also includes an approach move at rapid, the initial position in the Fixed Form Sentences could be further away from the part without any real cost of performance.

The S-word in the G97 block sets the initial speed of the spindle prior to activating CSS. It is included to ensure the spindle is up to speed when machining begins. This can be easily calculated from the cutting speed in feet-per-minute and the diameter position of the approach position.

For example, if the cutting speed is 1000 fpm and the approach position is 4.2-inches the RPM = sfm * 3.82 / diameter = 1000 * 3.82 / 4.2 = ~900 RPM.
Tool Data
Conventional CNC machines use tool geometry and wear offsets. Every tool has geometry and wear offset for tool length. Any contouring tool also has a geometry and wear offset for cutter radius compensation.

MANUAL GUIDE i requires a geometry offset for tool nose radius for every tool. This is used for visualizing the tool in the 3D solid model simulation and to determine residual machining geometry. This example does not use residual machining but it does use 3D solid model simulation.

Additional information is also required to fully describe the tool. The information required is different depending on the type of the tool. Sample tool data is provided with this example, but it is important to note that MANUAL GUIDE I requires that that tool data be installed prior to program generation. Without it, MANUAL GUIDE i cannot make appropriate decisions about the tool path and residual machining.

Secrets to Success
Before starting to program a part with MANUAL GUIDE i

- Install a set of Fixed Form Sentences setup that suit your needs
- Markup the workpiece drawing to identify each tool and operation
- Calculate all feeds and speeds for each tool and operation
- Setup tool geometry radius and the other tool data
- Follow the recommended structures outline above
DEFINING ARBITRARY PROFILES

Turning parts typically feature an arbitrary outer profile that must be roughed and finished machined. Many part also feature inner profiles that also must be drilled, rough turned and finished machined. Most of the other turning features have a more regular profiles that can be defined with simple geometry such as depth, length, diameter and pitch—for example, holes, tapped holes, grooves and threads.

Most arbitrary profiles can be defined by a series of lines, chamfers and arcs using the soft keys LINE, ARC (CW), ARC (CCW), CR (corner radius) and CC (corner chamfer).

The main challenge in defining arbitrary profiles is that not every feature is fully dimensioned. For example, parts often include two lines with a radius in between them. The start-point of one line is dimensioned as well as the end-point of the second line. The arc radius is also called out. However, the point that the lines meet the arc or are tangent to the arc is not typically dimensioned and must be calculated by manufacturing engineers. Consider the simple workpiece drawing opposite. The R.200 arc sits between two lines. The first line is a face starting at a diameter of 1.250-inches and 1.875 from the end of the part. The line is at a 90° angle to the centerline of the part. The second line ends at a diameter of 1.750-inches and 2.875-inches. The line is parallel to the centerline of the workpiece. The point that the R.200 arc blends we either line is not specified.

MANUAL GUIDE i includes a simple interface to describe arbitrary profiles using lines and arcs. The goal is to define the profile using only the information on the workpiece drawing. It includes several helper functions to handle the cases such as blended arcs and every data input field allows expression to be used to make calculations.

Profiles entered for the roughing pass can be reused for the finishing pass, eliminating the need to describe the profile twice.

Start Point

When defining an arbitrary profile, most of the data input is around determining the end-point of a line or arc. The initial start-point for one feature is assumed to be the end-point of the prior feature. However, the very first feature we program on an arbitrary profile does not have a prior end-point. Therefore, all arbitrary profiles begin with specifying an initial start-point.

Arbitrary profiles are typically defined from right to left, because the workpiece coordinate or part zero point is typically assigned on the end of the part. Turning parts are also often dimension from right to left, as in the example case opposite. The first feature on the profile of the example part is a .060-inch chamfer at 45°—a common turning part feature. We must specify the start-point with a X-axis diameter and a Z-axis position.

The Z-axis position is easy, it is the end of the part or the position Z0. However, the start-point of the chamfer is not dimensions.

The start diameter of the chamfer is easy to calculate. We know the end-point diameter is .750-inches. We also know the chamfer is .060-inches, but that is a radius value. We calculate the chamfer start diameters by subtracting 2 times the chamfer length (2 x .060) from the end-point diameter (.750). Before reaching for a calculate, understand that MANUAL GUIDE i allows you to enter an expression into any field. So you enter the expression “.75-(2*.06). The parentheses are not strictly required, since the rules of order of operations means that the multiplication “2*.06” will be calculated before the minus. When the input key is pressed, the expression is evaluated and the value of .63 is entered in the field.
**Lines**

Defining line segments in an arbitrary profile is quite simple.

**Step 1.**

Press the **LINE** soft key.

**Step 2.**

Press one of the eight soft key that best describes the line direction.

![Soft Key Options](image)

For example the chamfer on the end of the part is moving left and up from the start point, so you would press the **L-UP** soft key. The next line is moving left parallel to the centerline of the part, so you press the **LEFT** soft key for that segment. The next two segments are another chamfer moving left and up and a shoulder to the left.

**Step 3.**

Enter the information for the end-point of the line.
Two Lines and a Corner Radius
Two lines with a corner blend radius is a very common feature on turning parts. In the example part above, one line moves up from the 1.250 diameter and blends with a R.200 radius. The second line leaves the blends with the radius and moves left to at a diameter of 1.750 to the point 2.875. It would be easy in this case to calculate the blend points, but MANUAL GUIDE i provides and easier way.

Step 1.
Indicate the direction of the first line (we already know the start-point from the prior segment).

In the example we would press the \text{UP} soft key.

Step 2.
When the form requesting the end-point of the first line is displayed, press the \text{OK} soft key without entering any data.

Step 3.
Press the \text{CR} (corner radius) soft key and enter the radius value.

Step 4.
Enter a full definition of the end-point of the second line. In the example case the line is moving left, so the angle is known. Only the end diameter and the end Z-axis position is required. Enter the required information and press the \text{OK} soft key.

Closing the Profile
When an arbitrary profile has been specified, the final step is to indicate where the workpiece blank is to be machined. It may be clear in you mind as you define the profile, but MANUAL GUIDE i does not know if you are defining an outer profile or an inner profile. For parts machined from bar stock, this is relatively simple. The screen below shows the situation after defining the last line segment for the example part shown previously.

If you press the \text{CREATE} soft key at this stage, when you ran the program a message “FIGURE IS NOT CLOSED” would be displayed.

To closed a bar stock arbitrary profile, press the \text{BLCONT} (blank element connection) soft key until the \text{BLCONT} soft key is displayed. Then press the \text{BLCONT} soft key. Two candidates for closing the figure will be displayed. One of the candidate closes will be displayed in green (1). The other candidate is
Saving the Arbitrary Profile

When the soft key is pressed after an arbitrary profile is defined and the profile is closed, you are presented with two choices. Either you can save the profile in the current part program, or save the profile as a subprogram. Arbitrary profiles are typically used twice in a MANUAL GUIDE i program, once for the roughing and then again for the finishing pass. Therefore, it typically makes most sense to save the profile as a subprogram.

When you save the arbitrary profile as a subprogram, it makes sense to use the program numbers 8000-8499. When you reuse the profile created in the roughing cycle for the finishing cycle profile, MANUAL GUIDE i displays a list of available profiles based on a series of program numbers in the range specified by parameters 14720 and 14721 (14720=8000, 14721=8499).

It also helps to provide a comment that describes which part program uses the profile.

Example Profile

Creating and using arbitrary profile is very important for MANUAL GUIDE i applications, so we will first go through to process one time with the simple part shown above. We will start with a partially developed part program. Already programmed are the initial G-codes, the workpiece blank, the tool selection and spindle speed, and the cutting conditions for TURNING (OUTER ROUGH) cycles.

You can just read through the steps and hopefully understand the process. If you want to create the profile yourself, the initial program listing is provided in the appendix of this document.

Print out the workpiece drawing so that you can refer to it as we program.
The program has already been partially developed, and the cursor is highlighting the G1120 block that includes all the cutting conditions for a roughing cycle (see the description of the roughing cycle later in this document).

Normally, MANUAL GUIDE i automatically jumps to building a profile right after defining the cutting conditions, but since we are starting from a partially developed program we must initiate the arbitrary profile definition.

Press the \[ \text{soft key} \] until the \[ \text{soft key} \] is displayed.

Press the \[ \text{soft key} \].

With the TURNING FI (turning figure) tab selected and the ZX PLANE TURNING FIGURE highlighted, press the \[ \text{soft key} \].
This screen is normally display right after defining the **TURNING (OUTER ROUGH)** cutting conditions.

From our discussions above, you know that the first thing we must do is specify a start-point for the profile. In the example case it is the lower right of the chamfer.

Type the expression to calculate the **START POINT DX** (start diameter in X), 

```
.75 - 2 * .06
```

Press the **INPUT** key on the CNC MDI panel.

**Note**
If using MANUAL GUIDE i in NCGuide, you can press **ENTER** on the PC keyboard.

Notice that the expression 

```
.75-2*0.06
```

has been evaluated and replace with the result .063.

Also, after pressing the **INPUT** key, the cursor moved to the next field, in this case **START POINT Z**.

The start-point of the chamfer is on the end of the part at coordinate Z0.

Enter a value of 0. and press the **INPUT** key.
The value of 0. in entered into the START POINT field and the cursor has moved to the next field, in this case back to the first field START POINT DX.

Now that all the required information has been entered, we move to the next step.

Press the **OK** soft key.

MANUAL GUIDE \(i\) provides graphical feedback to confirm the information you entered was at least logical. We can not tell much from the start-point since is a single point. Also note the coordinates displayed at the bottom of the screen.

Now we will define the .060 x 45° chamfer. It is, of course a line.

Press the **LINE** soft key.

Next we specify the direction of the line from the prior position, in our case the start-point.

Notice that the current **LINE DIRECTION** at the top of the data input form is RIGHT, and that with a RIGHT line, the only value that can be entered is the END POINT Z.

Review the arrows on the soft keys for guidance. Relative to the start-point, the chamfer is a line moving left and up, which most resembles the

Press the **L-UP** soft key.
The **LINE DIRECTION** field at the top of the data input form has changed to LEFT-UP. Now there are three values that can be entered; **END POINT DX**, **END POINT Z** and **ANGLE**.

Notice that each of the fields has an asterisk “*” on their right-hand side. An asterisk means that a field is optional, and you do not have to enter a value. In fact you must enter any two of the three values to define the end point of the line.

From the drawing, the chamfer ends on the .75 diameter and the chamfer angle is 45°, so entering the **END POINT DX** and the **ANGLE** make sense.

Enter .75 in the **END POINT DX** and press the **INPUT** key.

The cursor moves the the **END POINT Z** field. To move to the **ANGLE** field, you can either press the **INPUT** without entering a value, or press the **cursor key** one time.

Press the **INPUT** key.

Enter the chamfer angle, 45. (degrees) and press the **INPUT** key.
Now that all the required information has been entered, we move to the next step.

Press the soft key.

MANUAL GUIDE provides graphical feedback to confirm the information you entered was at least logical. The chamfer appears to be logical. The coordinates displayed at the bottom of the screen includes the diameter we entered (0.075) and the Z coordinate (-0.060) that MANUAL GUIDE calculated.

The next segment in the arbitrary profile is another line moving to the left—a straight diameter of 0.75 moving to the dimension .625.

Press the soft key.

Press the soft key.
The LEFT line has only one value that can be specified, **END POINT Z**. Note how, by selecting the **LINE DIRECTION**, MANUAL GUIDE simplifies and error proofs the information you can enter. The most common error you will make in the beginning is entering Z coordinates. The drawing states the end of the diameter is at .625, so it is easy just to enter the value.

Enter .625 in the field, press the **key** and then press the **soft key**.

MANUAL GUIDE provides again provides graphical feedback to confirm the information you entered was at least logical. Hopefully you can see the problem. The coordinates displayed at the bottom of the screen includes the correct diameter (0.075) and the Z coordinate we entered (0.625) but the line is clearly not correct.

That is because our part zero is the end of the part. All Z-axis coordinates to the left of the workpiece zero point are negative. We should have entered minus .625. Let’s fix it.

Press the **ALTER** (figure alter) soft key.

Use the **key** to reselect the **END POINT Z** field, enter the new value “-.625 and press the **key**.

To complete the ALTER operation, press the **soft key**.
Now graphic looks like what we expected. Just remember, and Z-axis coordinates to the left of the workpiece zero will be negative or minus.

The next feature looks very much like a chamfer, again a line moving left and up.

Press the **LINE** soft key.

Again, with the **LINE-UP** line direction, we have the choice of entering two of the three possible values.

The angle of the line is not provided on the drawing, but with a bit of simple math we could calculate or infer the angle.

However, we do have the end-point diameter (1.250) and the Z-axis coordinate (.875), so it make sense to enter those values.

Enter each value below, pressing the **INPUT** each time:

- **DX**: .875
- **Z**: -.625
Double check that you entered a negative number for the **END POINT Z (Z)** value, and then press the **OK** soft key.

The graphic looks like what we expect. The next feature is another simple diameter moving to the left.

Press the **LINE** soft key.

Press the **LEFT** soft key.
Enter the values below (press \[\text{OK}\] after each):

Z: -1.875

Press the \[\text{OK}\] soft key.

The arbitrary profile is starting to look pretty good. However, the next segment is a line moving up, but the end-point diameter is not provided. We know there is a line there because the start diameter is 1.250, the diameter after the radius is 1.750. That is a change of .500 or .250 on each side. Since the 90° radius is only .0200 there must be a line .050 in length.

Notice that the blend point of the radius and the next line moving left is also not provided on the drawing. It is again easy to calculate with a 90° radius, but could be error-prone. So we will use the two lines and corner radius feature discussed above.

Press the \[\text{LINE}\] soft key.

Press the \[\text{UP}\] soft key.
Notice that there is only one coordinate value we can enter **END POINT DX**, and it has an asterisk because it is optional.

Press the **OK** soft key without entering a value.

MANUAL GUIDE i provides graphical feedback with a yellow line moving all the way from the bottom of the window to the top of the window. Also note the DX and Z end-point positions at the bottom of the screen have now values. MANUAL GUIDE i is indicating that it knows the end-point is somewhere on that line, but it needs more information to calculate the value.

Press the **CR** (corner radius) soft key.

Enter the radius value below (press afterwards):

\[ R: .2 \]

Press the **OK** soft key.
The vertical line has change from yellow to red to confirm that something has changed, but MANUAL GUIDE i still does not have sufficient information to draw the profile.

We have defined the line and the corner radius in the two lines and a corner radius sequence. Now lets define the second line end-point.

Press the **LINE** soft key.

Press the **LEFT** soft key.

You may remember that in the previous cases, the LINE LEFT direction only requires an END POINT Z value. This time, because we have the line-up and corner radius line hanging around in space, we must provide a complete definition of this lines end-point. Luckily, the drawing has that information.

Enter the following values:

- DX: 1.75
- Z: -2.875

Press the **OK** soft key.
Now MANUAL GUIDE i has all the information it needs to calculate the blend points of the corner radius with the original line up and the second diameter line.

The arbitrary has one last line to define, the face to the final diameter (2.250).

Press the **LINE** soft key.

Press the **UP** soft key.

Enter the following values:

**DX**: 2.25

Press the **OK** soft key.
The part definition of the arbitrary profile is complete, and we now have to tell MANUAL GUIDE i on which side of the line to remove material.

Press the continuous menu right soft until the _BLCONT_ soft key is displayed.

Press the _BLCONT_ soft key.

MANUAL GUIDE i provides two candidates for metal removal. The area enclosed by the green lines (1) and area enclosed by the magenta lines (2).

The green lines identify the material to be removed in this case.

Press the _FIG. 1_ soft key.
MANUAL GUIDE i provides two choices for saving arbitrary profiles. Either they can be saved in the main body of the program, or they can be saved as a subprogram.

Since most turning arbitrary profiles will be used at least twice, once for the roughing cycle and once for the finishing cycle, it makes sense to save them as subprograms.

Press the ↓ key to select the CREATE AS SUB PROGRAM choice.
Enter a subprogram number and figure name in the fields provided.

**SUBPRO NO.: 8010**

**FIGURE NAME: PROFILE FOR PROGRAM 1010**

It is recommended that the subprogram number be in the range 8000-8499. This range is specified in the CNC system parameters 14720 and 14721.

One way to organize programs and subprograms is to use the range of main program numbers 1000-1499 and add 7000 to those numbers for the associated subprogram number (1010+7000 = 8010).

Press the **OK** soft key.

The main program now includes a subprogram call (M98) to the subprogram 8010.

That completes the definition of the arbitrary profile.

**Modifying an Arbitrary Profile**

If the arbitrary profile must be modified, first highlight the subprogram call in the main program using the cursor keys.

Press the **ALTER** soft key.

Note the START POINT (lower right of the chamfer is highlighted in yellow. The rest of the arbitrary profile is in blue.

Press the **key** several time to see the yellow highlight move to the various line segments.
This screen shows the second shoulder diameter highlighted.

Press the \[ \text{cursor key} \] move the highlight back towards the beginning of the arbitrary profile.

To edit a arbitrary segment, highlight it and press the \( \text{ALTER} \) soft key.

Edit any of the data displayed.

To accept the changes, press the \( \text{OK} \) soft key.

To exit without making any changes, press the \( \text{CANCEL} \) soft key.

When all the editing of all the segments required are complete,

press the \( \text{CREATE} \) soft key to save the changes, or the \( \text{CANCEL} \) soft key to abandon all the changes.

\[ \text{Note} \]

When you press CREATE during an edit, you have a choice of altering the current program (most likely) or creating a completely new subprogram.
In this example we will create a program for a relatively simple turned part with a hole drilled on the centerline. The material is 1018 steel. The finished part is 4-inches in diameter and 3.125-inches long.

**Workpiece Drawing**

![Workpiece Drawing](image)

**Part Blank Size**

The finished part is 4-inches in diameter and 3.125-inches long. To provide safe workholding and to ensure clearance to machine the part a part blank of 4-inches in diameter and 5-inches long is selected. Blank size noted on the workpiece drawing.

**Setup**

The part will be chucked with 3.2-inches protruding from the chuck jaws. Setup information noted on the workpiece drawing.

**Workpiece Program Zero**

The workpiece dimensioning and common shop practice determine that the workpiece zero should be on the far right-hand and on the centerline of the workpiece. To provide a machine face, the workpiece program zero is set 50-thousandths inside of the workpiece when we define the material blank in MANUAL GUIDE i. The selected workpiece zero is sketched on the workpiece drawing.
**Safe Approach Position**

After a tool change, tools rapid to a position just off the part called the approach position. For parts that use constant surface speed programming, this allows the spindle to get up to speed at a known diameter. We choose to be 100-thousandths off the end of the part in Z (Z0.1), and 100-thousandths off the part in X (0.2 in diameter). The approach in Z is a standard programmed into the START TOOL Fixed Form Sentences. The X approach must be specified when the START TOOL Fixed Form Sentence is inserted into the program. Since the diameter of the part is 4-inches, the approach diameter is X4.2. This value is noted on the workpiece drawing.

**Visualize Operations**

The workpiece drawing has two main features:

1. Outer contour that must be rough and finished turned and finish faced.
2. A 3/8-inch diameter hole drilled along the centerline.

Standard machining practice determines that all roughing operations must be completed before finishing operations. There are four machining operations:

- **Op1**: Rough Turn (leaving 30-thousandths on diameters and faces for finishing cuts)
- **Op2**: Drill
- **Op3**: Finish Turn
- **Op4**: Finish Face

The operations are noted and sketched on the workpiece drawing.
### Tooling and Feeds and Speeds

Three tools are used.

The outer roughing operation is performed with an 80° standard turning tool, which is mounted in turret station number 1 (T0101). The tooling manufacturer recommends a cutting rate of 1000 sfm, a feedrate of 0.012 ipr and a depth of cut (DOC) per pass or 40-thousandths. The tool nose radius is 1/32-inches, 80° nose angle and 93° cutting angle.

The drilling operation is performed with a 3/8" drill placed in turret station 9 (T0909). The tool manufacturer recommends a spindle speed of 150 fpm and a feedrate of 0.007 ipr. The drill point angle is 118°.

The finish turn and finish facing operations are performed using another 80° standard turning tool that is reserved for finishing operations. The finishing tool is loaded in turret station number 2 (T0202). The tooling manufacturer recommends a cutting rate of 1000 sfm and 0.012 ipr. The tool nose radius is 1/32-inches, 80° nose angle and 93° cutting angle.

The roughing and finishing tools use constant surface speed (CSS). The spindle speed in surface feet per minute (SFM) is set in the START TOOL CSS Fixed Form Sentence in the G96 block. To ensure that the spindle is at the correct speed when reaching the approach position, it is useful to set the appropriate spindle speed in RPM at the approach diameter prior to switching to CSS mode. The approach spindle speed in RPM is specified in the START TOOL CSS Fixed Form Sentence in the G97 block.

The roughing and finishing tool RPM is calculated from standard formula:

\[
RPM = \frac{3.82 \times SFM}{DIA.} = \frac{3.82 \times 1000}{4.2} = 909 = \sim 900RPM
\]

The drill does not use CSS and must programmed in RPM:

\[
RPM = \frac{3.82 \times SFM}{DIA.} = \frac{3.82 \times 150}{.375} = 1528 = \sim 1500RPM
\]

The feeds, speeds and depth of cut for each operation and tool are noted on the workpiece drawing. The spindle speeds will be required when editing the TOOL START Fixed Form Sentences and the feedrates and depth of cuts will be set in the cutting conditions of the machining cycles.

Constant surface speed program is typical used in turning applications because provides a consistent surface finish and maximizes tool life since the tool is always cutting at the optimum rate. However, as the tool approached the workpiece centerline in the X-axis the spindle speed reaches the maximum spindle speed of the machine. This very high spindle speed can be disconcerting to new operators. Also, there maybe workholding issues some parts and many bar feeders have spindle speed limitations. Therefore, it is common practice to limit the maximum speed of the spindle when in CSS mode. There is no good way to predict the maximum spindle speed for a given application, and of course you must error on the side of caution - it must be determined by experience and testing.

The START PROGRAM Fixed Form Sentence includes a G50 command to limit the spindle speed in CSS. The arbitrary value of S3000 (3000 RPM) is used in this example.

#### Drilling Cycle

The hole depth is less that three times the drill diameter, so peck drilling is not required, unless there are problems found when machining the first part. Peck drilling retracts the tool periodically during machining to break stringy chips and to allow more coolant flow to the drill tip.

No peck required is noted on the workpiece drawing.

#### Tolerance Analysis

No specific tolerances are stated on the drawing so shop standard will be used. The nominal part geometry will be used for programming.
Op1: rough turn (.030 left on dia/face), T0101, 1/32tnr/80°na/93°aa, 1000sfm/900rpm, .012ipr, .040doc
Op2: Ø3/8" drill, T0909, 118°dpa, 1500rpm, 0.007ipm, no peck req.
Op3: finish turn, T0202, 1/32tnr/80°na/93°aa, 1000spm/900rpm, .012 ipr
Op4: finish face, T0202, 1/32tnr/80°na/93°aa, 1000spm/900rpm, .012 ipr

Blank 4.0 dia. x 5.0in long steel
Work Origin Z - 0.050"
Chuck with 3.2" from jaws
Safe Approach dia. X4.2
Two tuning tools and one drill are used to machine this part. Their geometry must be specified prior to creating the program, otherwise the paths generated by MANUAL GUIDE i may not be correct. Most of the information was gathered during the planning stage and marked on the workpiece drawing. The virtual tip and set of the tools depend on the turret and tool setup.

On the GEOMETRY OFFSET tab:

<table>
<thead>
<tr>
<th>Tool</th>
<th>Offset #</th>
<th>Radius</th>
<th>Virtual Tip</th>
</tr>
</thead>
<tbody>
<tr>
<td>T0101 Roughing Tool</td>
<td>01</td>
<td>0.0313 (1/32)</td>
<td>DIR. 3</td>
</tr>
<tr>
<td>T0202 Finishing Tool</td>
<td>02</td>
<td>0.0313 (1/32)</td>
<td>DIR. 3</td>
</tr>
<tr>
<td>T0909 3/8” Drill</td>
<td>09</td>
<td>0.375 (3/8)</td>
<td>-</td>
</tr>
</tbody>
</table>

On the TOOL DATA tab,

<table>
<thead>
<tr>
<th>Tool</th>
<th>Offset #</th>
<th>Tool Type</th>
<th>Tool Name</th>
<th>Set</th>
<th>Cut Angle</th>
<th>Nose Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>T0101 Roughing Tool</td>
<td>01</td>
<td>ROUGH</td>
<td>ROUGH (edited)</td>
<td>93.0°</td>
<td>80.0°</td>
<td></td>
</tr>
<tr>
<td>CNMG 432</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>T0202 Finishing Tool</td>
<td>02</td>
<td>FINISH</td>
<td>FINISH (edited)</td>
<td>93.0°</td>
<td>80.0°</td>
<td></td>
</tr>
<tr>
<td>CNMG 432</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>T0909 3/8” Drill</td>
<td>09</td>
<td>DRILL</td>
<td>DRILL</td>
<td>-</td>
<td>118.0°</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The tool geometry offsets and the tool data values can either be entered using the MDI keyboard, or by reading the data from a memory card. Suitable memory card files are provided in the appendix of this document.
T0101 Roughing Tool

Set the CNC mode to EDIT.

Press the continuous menu right soft key several times until soft key is displayed.

Note:
If using MANUAL GUIDE i in NCGuide, the keyboard short cut Cttl-3 selects EDIT mode.

Press the soft key.

Press the soft key.

Note:
In MANUAL GUIDE i, pressing the right-cursor key when on the top line of a table can either select the next tab (GEOMETRY OFFSET, WEAR OFFSET and TOOL DATA) or the next column of data (X-AXIS, Z-AXIS, RADIUS and VIRT. TIP) depending on whether the cursor mode is TAB mode or ITEM mode. The current mode is displayed on the right hand side of the title bar (TAB mode in the example). The change cursor soft key switches between TAB mode and ITEM or column mode.
With the cursor in **ITEM** mode, press the cursor right key until the RADIUS field of GEOMETRY OFFSET number 001 is selected.

**Note:**
The current cursor mode (TAB or ITEM) is displayed in the top right of the TOOL OFFSET window.

The tool inserts used in this example have a 1/32-inch tool nose radius (see marked up workpiece drawing). With MANUAL GUIDE *i*, you can enter the expression 1/32 (1 divided by 32), eliminating the need to first calculate the decimal value.

Using the MDI keyboard, enter the expression \[
\frac{1}{32}
\] and then press the key.

**Note:**
You can enter an expression in any input field in MANUAL GUIDE *i*.

MANUAL GUIDE *i* simplifies programming by allowing the operator to ignore the radius on the tool. The virtual tip field identifies the orientation of the tool so the CNC can adjust the tool path appropriately for the radius as the part is machined.

Press the cursor right key until the VIRT. TIP field of GEOMETRY OFFSET number 001 is selected and press the soft key.
There are no values that need to be entered for **WEAR OFFSETS** at this time. Those will be entered during the machine setup.

Press the cursor key until the first field or column in the **TOOL DATA** tab is selected.

Press the **GENERAL** soft key to select the general turning tool type.

Note: There are several tool types to select from including **GENERAL, THREAD, GROOVE, BUTTON, STRAIGHT, DRILL, CHAMFER, END MILL, BALL MILL, TAP, REAMER, BORING and FACE MILL**. Selecting the tool type and entering the tool data correctly is important. **GENERAL** is a standard turning tool.

The tool name displayed is the default **GENERAL**. Since we will have two standard turning tools, one for roughing and one for finishing, it is helpful to edit the default text to identify this one as the roughing tool.

Press the cursor right key one time to highlight the tool name **GENERAL**.
Enter the new tool type label “ROUGH” and press the \[ INPUT \] key on the MDI panel.

**Note:**
The text field identifying tools must be 8-characters long or less.

Press the \[ \rightarrow \] cursor key one time to select the SET field.

Refer to the tool sets in the graphic displayed. They represent the typical set or orientation of tools in a turret.

A set value of 1 describes a standard turning tool in a rear turret.

Enter a value of 1 and press the \[ INPUT \] key on the MDI panel.

**Note:**
*In NCGuide, the ENTER key on the PC keyboard is the shortcut key for INPUT on the MDI panel.*
Press the cursor key one time to select the **CUT ANGLE** field.

Refer to the graphic displayed, angle A, and enter the cut angle for the tooling being used. This is often referred to as the entering angle.

In the example the value is noted on the workpiece drawing, enter 93 for 93°, and press **INPUT** key on the MDI panel.

Press the cursor key one time to select the **NOSE ANGLE** field.

Refer to the graphic displayed and enter the nose angle for the tooling being used. In the example the value is noted on the workpiece drawing, enter 80 for 80° and press **INPUT** key on the MDI panel.

**T0202 Finishing Tool**

Press the cursor key one time and the cursor key one time to select the first field for tool number 2 (002).

Entering the information for tool number 2, the finishing tool, is similar to the information entered for tool number 1, the roughing tool.
Press the cursor key until the RADIUS field of GEOMETRY OFFSET for tool number 002 is selected.

Note: The current cursor mode (TAB or ITEM) is displayed in the top right of the TOOL OFFSET window.

The tool inserts used in this example have a 1/32-inch tool nose radius, as noted on the workpiece drawing.

Using the MDI keyboard, enter the expression \(1 \div 3 \downarrow 2\) and then press the key.

Press the cursor key until the VIRT. TIP field of GEOMETRY OFFSET for tool number 001 is selected, then press the soft key.
There are no values that need to be entered for **WEAR OFFSETS** at this time.

Press the cursor key until the first field or column in the **TOOL DATA** tab is selected.

Press the **GENERAL** soft key to select the general turning tool type.

Again, the tool name displayed is the default **GENERAL**. Since we will have two general tools, the roughing tool and the finishing tool, it is helpful to edit the default text in this case.

Press the cursor right key one time to highlight the tool name **GENERAL**.
Enter the new tool type label “FINISH” and press the \( \text{INPUT} \) key on the MDI panel.

Press the \( \text{cursor right} \) key one time to select the \( \text{SET} \) field.

Refer to the tool sets in the graphic displayed and enter a value of 1 - a turning tool in a rear turret, and press the \( \text{INPUT} \) key on the MDI panel.
Press the cursor key one time to select the **CUT ANGLE** field.
Refer to the graphic displayed and enter the cut angle (or entering angle) for the tooling being used. In this example, enter 93 for 93°, and press the **INPUT** key on the MDI panel.

Press the cursor right key one time to select the **NOSE ANGLE** field.
Refer to the graphic displayed and enter the nose angle for the tooling being used, in the example, enter 80 for 80°, and press the **INPUT** key on the MDI panel.

Press the cursor key one time and the **↓** key six times to select the first field for tool number 9.
T0909 3/8” Drill

Press the cursor right key until the RADIUS field of GEOMETRY OFFSET of tool number 009 is selected.

Note:
The current cursor mode (TAB or ITEM) is displayed in the top right of the TOOL OFFSET window.

In this example, tool number 9 (009) is a 3/8-inch diameter drill. With MANUAL GUIDE i, there is no need calculate a radius value, simply enter the expression 3/8/2 (3 divided by 8 divided by 2).

Using the MDI keyboard, enter the expression, \[ \frac{3}{8} / \frac{1}{2} \] and then press the key.

Note:
Though tool nose radius will not be used with a drill, MANUAL GUIDE i will use this field to display the diameter of the tool during 3D animation.

Press the cursor right key until the VIRT. TIP field of GEOMETRY OFFSET 001 is selected.

If necessary, press the soft key.

Note:
Centerline cutting tools such as drills do not use tool nose radius compensation, that is why we set the drill to a virtual tip of zero. However, MANUAL GUIDE i does use the RADIUS value to display the tool in 3D animation.
There are no values that need to be entered for **WEAR OFFSETS** at this time.

Press the \( \rightarrow \) cursor right key until the first field or column in the **TOOL DATA** tab is selected.

Initially, no soft key for a tool type **DRILL** is displayed.

Press the \( \uparrow \) one time to display the next set of tool types.

**DRILL** is now the first tool type on the second set of tool type soft keys.

Press the **DRILL** soft key to select the **DRILL** tool type.

**Note:**
There are several tool types to select from including **GENERAL**, **THREAD**, **GROOVE**, **BUTTON**, **STRAIGHT**, **DRILL**, **CHAMFER**, **END MILL**, **BALL MILL**, **TAP**, **REAMER**, **BORING** and **FACE MILL**. Selecting the tool type and entering the tool data correctly is important.

The tool name displayed is the default **DRILL**. Since we only have one drill in this example, we can leave it set to the default value.

Press the \( \rightarrow \) cursor key two times highlight the **SET FIELD**.

**Note:**
The tool type label is limited to 8 characters. We could label this drill ".375 DRL" or ".375DRIL" if we had more drills in the turret and needed to differentiate between them.
Press the cursor right key one time to select the NOSE ANGLE field.

Refer to the graphic displayed and enter the nose angle for the tooling being used, in the example, 118° for a standard drill (as noted on the workpiece drawing), and press the key on the MDI panel.

Note:
There is no field for the CUT ANGLE for a drill. The data to be entered in tool angle varies depending on the tool type selected.

All of the tool data necessary for programming the example part has been entered.

To close the tool data edit screens, press the soft key.

Note:
In this tutorial, we only set the tool geometry and tool data values required for programming. At runtime, to X and Z geometry offsets must be set before machining a part.
Set the CNC mode to EDIT.

Click the **NEWPRG** soft key.

Enter the program number (1020 used in this example) using the MDI keyboard.

Press the **CREATE** soft key.

To edit the program header comment and give the program a description, press the **D LIST** soft key.
With the program just created highlighted, press the 

soft key.

Enter a comment in the text box displayed, 

(EXAMPLE TURNING PART 2 in the example).

To complete the operation, press the soft key.

The comment is now displayed in the program directory.

To exit the program operations menus, press the soft key.
Note:
The comment entered in the previous step is also displayed on the first line of the program in parentheses.

Press the continuous menu right soft key once to display the soft key.

Notice that the soft key has a small turning workpiece and turning tool. There is another START soft key for milling operations, make sure you select the correct one.

Press the soft key.

The available START fixed form sentences are displayed.

If necessary, use the key on the MDI panel to highlight the START PROGRAM selection.

Press the soft key to insert the START PROGRAM Fixed Form Sentence.

Note:
The G-code in the example START PROGRAM Fixed Form Sentence and the selections available may be different on your machine. Fixed Form Sentences are customizable by the machine tool builder/distributor and the end-user.
With the example Fixed Form Sentence, the message “IMPERFECT WORD MUST BE CHANGED” is displayed.

The START PROGRAM Fixed Form Sentence has a word that must be modified by the operator. It is identified by the “?” characters in the program.

Use the ⬇️ cursor key to highlight and select just the S-word.

Enter the value of the maximum spindle speed allowed in CSS, 3000 for example, and press the ⏬️ key on the MDI keyboard to confirm the new value.

Note:
When modifying an individual word in a part program like the S-word in the Fixed Form Sentence inserted, it must be altered with the MDI panel ALTER key and NOT the ALTER soft key.

DISCUSSION: Constant Surface Speed

Constant surface speed program is typical used in CNC turning center applications because it provides a consistent surface finish and maximizes tool life—since the tool is always cutting at the optimum rate.

However, as the tool approached the workpiece centerline in the X-axis, the spindle speed reaches the maximum spindle speed of the machine. This very high spindle speed can be disconcerting to new operators. Also, there maybe workholding issues with some parts and many bar feeders have spindle speed limitations.

It is common practice to limit the maximum speed of the spindle when in CSS mode. There is no good way to predict the maximum spindle speed for a given application and you must error on the side of caution—it must be determined by experience and testing.

The START PROGRAM Fixed Form Sentence includes a G50 command to limit the spindle speed in CSS. The arbitrary value of S3000 (3000 RPM) is used in this example.
After manual editing data, press the key until the whole line is again highlighted.

Note:
After manual editing Fixed Form Sentence, you must always position the cursor back on the last line in the program (the line before the %) and all the way to the left so that the whole line is highlighted.

This completes the START PROGRAM requirements.
The material blank is defined mostly for the 3D animation of machining. Note that data item names with an asterisk (*) are optional and do not have to be specified. The graphic is the one displayed on the CNC screen when that field is selected. The data required is typically highlighted in yellow.

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>D</td>
<td><img src="image" alt="Diagram" /></td>
<td>The diameter of the blank (positive value).</td>
</tr>
<tr>
<td>L</td>
<td><img src="image" alt="Diagram" /></td>
<td>Length of the cylinder blank (positive value).</td>
</tr>
<tr>
<td>K</td>
<td><img src="image" alt="Diagram" /></td>
<td>Cutting allowance from the end of the workpiece blank to the workpiece origin (part program zero point). This value is typically positive (note sign display in graphic).</td>
</tr>
</tbody>
</table>
The material blank size is used to display the workpiece during 3D animation of the machining process. It does not determine tool paths in any way.

Press the **soft key**

Press the **key on the CNC MDI panel to select the **tab.**

Press the **key on the CNC MDI panel to highlight the **choice.
With the **CYLINDER BLANK FIGURE** choice highlighted, press the **SELECT** soft key.

Refer to the notes written on the workpiece drawing and enter the following values in the three fields using the MDI keyboard, pressing the key on the MDI panel to complete each value entered:

- **D**: 4. (DIAMETER of the workpiece blank)
- **L**: 5. (LENGTH of the workpiece blank)
- **K**: 0.05 (the position of the workpiece program zero from the end of the workpiece blank)

Complete defining the workpiece blank by pressing the **INSERT** soft key.

**Note:**

The blank part dimensions entered in the blank definition are only used for the graphic simulation of the workpiece and have no influence on the tool path generated by MANUAL GUIDE i.

The program now has a new G1900 block whose parameters are derived from the dialog.

The text below the animation window explains what the G-code means. If you move the cursor over each parameter, they also will be described.
The first operation is a outer rough turning cycle. This section explain the various parameters of the cycle. Note that data item names with an asterisk (*) are optional and do not have to be specified. The graphic is the one displayed on the CNC screen when that field is selected. The data required is typically highlighted in yellow.

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>P CUTTING DIRECTION</td>
<td><img src="image1.png" alt="Diagram" /></td>
<td>Direction of cut for the roughing cycle. Press the -Z soft key to machine towards the main spindle in the Z minus direction. Press the +Z soft key to machine away from the main spindle in the Z plus direction or for subspindles.</td>
</tr>
<tr>
<td>Q CUT DEPTH</td>
<td><img src="image2.png" alt="Diagram" /></td>
<td>Depth of cut for each roughing pass. This value is provided by the tooling manufacture recommendations for the tooling and material type. This value is always a positive radius value, so the workpiece diameter will be reduced twice this amount per roughing pass.</td>
</tr>
<tr>
<td>H RATE OF CUT DEPTH %</td>
<td><img src="image3.png" alt="Diagram" /></td>
<td>If this parameter is set to 100%, the depth of cut for all the passes will be the same. If the value is less that 100%, each cutting depth will be less than the prior pass. If the value is more than 100%, each pass will be deeper than the prior pass. The parameter has a range of 1 to 200 (%). The depth of cut of the next pass is the depth of cut of the current pass multiplied by this value / 100.</td>
</tr>
<tr>
<td>C X-AXIS FINISH AMT.*</td>
<td><img src="image4.png" alt="Diagram" /></td>
<td>The amount of material to be left on each side of the workpiece for a finishing cut. The general rule of thumb is to leave just slightly more than the radius of the finishing tool. If left blank, the finish amount is zero. The value is always a positive, radius amount.</td>
</tr>
<tr>
<td>D Z-AXIS FINISH AMT.</td>
<td><img src="image5.png" alt="Diagram" /></td>
<td>The amount of material to be left the faces of the workpiece for a finishing cut. The amount left on faces is more critical for small lead-angle tools, especially when finish machining 90° shoulders in the +X direction. Finish amounts on faces are typically small 0.003 - 0.006 inches.</td>
</tr>
</tbody>
</table>
### Cutting Conditions Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>F CUT DIRC.FEEDRATE</td>
<td><img src="image1.png" alt="Diagram" /></td>
<td>The cutting feedrate when the tool cuts in the parallel to the workpiece centerline. This value is provided by the tooling manufacture guidelines for a material type and application.</td>
</tr>
<tr>
<td>E CUT DEPTH FEEDRATE</td>
<td><img src="image2.png" alt="Diagram" /></td>
<td>The feedrate when the tool cuts into the workpiece in the radius (-X) direction. This value is provided by the tooling manufacture guidelines for a material type and application. If no particular application guidelines are provided, make it the same as (F).</td>
</tr>
<tr>
<td>V CUT RISE FEEDRATE</td>
<td><img src="image3.png" alt="Diagram" /></td>
<td>The feedrate when the tool cuts away from the centerline of the workpiece in the radius (+X) direction, up shoulders and faces. This value is provided by the tooling manufacture guidelines for a material type and application. If no particular application guidelines are provided, make it the same as (F).</td>
</tr>
</tbody>
</table>

### Detail Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>K 1ST FEED OVERRIDE</td>
<td>No specific graphic for this parameter</td>
<td>The feedrate override value for the first cutting pass. Set a positive value between 1 and 200% (1% increments). This allows the feedrates specified on the cutting conditions tab to be reduced or increased on the first cutting pass. For example, the outer surface of a casting may be harder than the material below, so the feedrate for the first pass may need to be reduced.</td>
</tr>
<tr>
<td>W CUT RISE METHOD</td>
<td>No specific graphic for this parameter</td>
<td>There are two methods of retract at the end of each roughing pass: (1) The SPEED method immediately retracts at rapid at a 45° angle at the end of each pass to the ESCAPE AMOUNT (U). The SPEED method is faster, but leaves 45° pips on faces that may need to be removed by a semi-finishing pass before the finishing pass; (2) The CUT method machines up the face profile at the programmed feedrate before retracting at the rapid rate at 45°. The CUT method is slower but produces a smoother surface for immediate finishing. Press the SPEED or CUT soft keys to select the preferred method.</td>
</tr>
</tbody>
</table>
### Detail Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
</table>
| **U** ESCAPE AMOUNT | ![Graphic](image1) | The distance that the tool retracts from the cutting surface after each roughing pass before moving at the rapid rate back to the Z-axis approach position. This is a positive, radius amount.  
A minimum ESCAPE AMOUNT value can be set in CNC system parameter 27128. If set, this becomes the initial value of the ESCAPE AMOUNT field.  
Note the graphic shows the HIGH SPEED cut method where the tool retracts at 45° at the end of each pass and does not machine up the face. With the CUT method, the tool has already moved away from diameters by the CUT DEPTH amount before the 45° retract, so a smaller value could be considered in this field. |
| **L** X-AXIS CLEARANCE | ![Graphic](image2) | The approach position in the X-Axis. The X-Axis moves to this position at rapid before the roughing passes begin, or before the optional face rough move. This value is a positive, radius value.  
A minimum X-AXIS CLEARANCE value can be set in CNC system parameter 27129. If set, this becomes the initial value of the X-AXIS CLEARANCE field. |
| **M** Z-AXIS CLEARANCE | ![Graphic](image3) | The approach position in the Z-Axis (positive value). The Z-Axis moves to this position at rapid before the roughing passes begin, and return to this position after each pass.  
A minimum Z-AXIS CLEARANCE value can be set in CNC system parameter 27130. If set, this becomes the initial value of the Z-AXIS CLEARANCE field. |
| **Z** APPROACH MOTION | ![Graphic](image4) | Soft keys describe choices  
Determines how the X and Z axes will approach the workpiece from their position prior to the cycle (the positions in the START TOOL Fixed Form Sentence in this case). Use the soft keys to specify the motion required.  
- **Z → X**: the tool moves in Z first, then X (initial value).  
- **X → Z**: the tool moves in X first, then Z.  
- **2 AXES**: the tool moves in X and Z simultaneously. |
<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>S  CUT DEPTH DIRECTION</td>
<td><img src="image" alt="Soft keys describe choices" /></td>
<td>Allows the selection of the depth of cut direction. This field is displayed when CNC system parameter 27100#0=1. Press - or + soft keys to select direction.</td>
</tr>
<tr>
<td>X  POCKET CUTTING</td>
<td><img src="image" alt="Diagram" /></td>
<td>Determines if MANUAL GUIDE attempts to cut into pockets in the defined profile. Note that depending on the tool geometry, some areas of the pocket may remain. This fields is displayed when CNC system parameter 27100#1=1. If this field is not displayed (parameter 27100#1=0), pockets are always machined. Press the CUT soft key to machine as much of the pocket that is feasible with the tool selected. Press the NOTHIN soft key to not machine the pocket.</td>
</tr>
<tr>
<td>Y  OVERHANG CUTTING</td>
<td><img src="image" alt="Diagram" /></td>
<td>Determines if MANUAL GUIDE attempts to cut into overhangs in the face of the defined profile. Note that depending on the tool geometry, some areas of the overhang may remain. This fields is displayed when CNC system parameter 27100#1=1. If this field is not displayed (parameter 27100#1=0), overhangs are always machined. Press the CUT soft key to machine as much of the pocket that is feasible with the tool selected. Press the NOTHIN soft key to not machine the pocket.</td>
</tr>
</tbody>
</table>
### End Facing Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>WT/WTOUT ED FACE MC</td>
<td><img src="image1" alt="Graphic" /></td>
<td>Determines if rough facing passes in X are performed prior top to the profile roughing in Z. This tab and its associated fields are displayed when CNC system parameter 27120#2=1. Press the <strong>soft key rough face the end of the workpiece blank.</strong> Press the <strong>soft key skip rough facing the end of the workpiece blank.</strong></td>
</tr>
<tr>
<td>END FACE REMOVAL*</td>
<td><img src="image2" alt="Graphic" /></td>
<td>End facing removal amount (positive value). This is the total amount of material to be removed from the face of the part. It determines the number of facing passes required to reach the Z-AXIS FINISH AMT. for the end of the workpiece. Rough facing the end of the workpiece before performing the outer roughing operation provides a know end surface and may allow you to optimize the Z-AXIS CLEARANCE amount. This tab and its associated fields are displayed when CNC system parameter 27120#2=1. If parameter 27120#3=1, end facing is performed without leaving a finishing amount. If this field is left blank, the end facing allowance is determined based on the defined profile and the material blank definition. If the end facing allowance cannot be determined, simultaneous facing is not performed.</td>
</tr>
</tbody>
</table>

*GE FANUC*
OPERATION 1 - ROUGH TURNING

Start Tool T0101

Press the **START** soft key

Use the cursor key to highlight the **START TOOL CSS** Fixed Form Sentence.

With the Fixed Form Sentence **START TOOL CSS** highlighted, press the **INSERT** soft key.

The message "**IMPERFECT WORD MUST BE CHANGED**" is displayed.

The **START TOOL CSS** Fixed Form Sentence used here has several words that must be specified. They are identified by the "?" characters in the program.

Use the CNC cursor keys to select each field in turn. Refer to the marked up drawing and enter the correct value and press the key on the MDI keyboard to confirm the new value,

- **T0101** (tool number),
- **S900** (spindle speed at approach diameter - G97 RPM mode),
- **X4.2** (approach diameter) and
- **S1000** (constant surface speed - G96 FPR).
Note:

Use the key on the MDI keyboard and NOT the ALTER soft key to change the Fixed Form Sentence values.

If you are using NCGuide, the PC keyboard shortcut Ctrl-ENTER can be used.

After manually editing the G-code, press the so the last line is completely highlighted again before continuing.

Turning (Outer Rough) Cycle

Press the soft key.

Press the cursor key to select the cycles tab.
With the **TURNING (OUTER ROUGH)** choice highlighted, press the **SELECT** soft key.

**Notes:**
Fields that are predetermined choices use soft keys to select a choice. For example, the first field **CUTTING DI-RECTION** has two soft keys, and to specify the choice.

Any field marked with an asterisk "*" is optional and you do not have to enter a value in that field.

As you move between the fields using the cursor keys, the graphic on the right changes to highlight the information that is required to be entered in the currently selected field.

The **CUT COND.** tab includes the base cutting conditions for the roughing cycle.

Refer to the values on the marked up drawing and enter the following values using the soft keys and MDI keyboard and press the **key:

- **P**: -Z (cut in Z direction – soft key selection)
- **Q**: 0.04 (depth of cut per pass)
- **H**: 100 (first pass feedrate override)
- **C**: 0.03 (finish amount on diameters)
- **D**: 0.005 (finish amount on faces)
- **F**: .012 (cutting feedrate FPR)
- **E**: .012 (approach feedrate FPR)
- **V**: .012 (exit feedrate FPR)

Press the **key to display the DETAIL tab.
The values on the DETAIL tab will default to the last value entered so they may already be set. Enter the following values using the soft keys and MDI keyboard and press the key:

- **K**: 100 (no override)
- **W**: CUT RIZE (press the soft key)
- **U**: 0.05 (escape 50-thousandths)
- **L**: 0.1 (approach clearance in X)
- **M**: 0.1 (approach clearance in Y)
- **Z**: Z->X MOV (press soft key)

**Notes:**
- The parameter 27100#0=0 so the CUT SHIFT DIRECTION field is not displayed and the direction is -X.
- The parameter 27100#1=0, so the POCKET CUTTING and OVERHANG CUTTING fields are not displayed. Pockets and overhangs will be machined where possible.
- Press the key to display the ED FACE MC tab.

**Notes**
- The ED FACE MC tab is only displayed when parameter 27120 bit 2 is set to a value of 1.
- This tab specifies whether you want a rough facing pass to be performed to establish a known surface before performing the axial roughing passes.

Enter the following values using the soft keys and MDI keyboard:

- **H**: AVAILABLE (press soft key)
- **I**: 0.05 (total stock on face)

**Note:**
- The END FACE REMOVAL FIELD is not displayed until after you press the AVAILABLE soft key.
- Parameter 27120#3 determines if the Z-AXIS FINISH AMT. is left on the workblank face.
Complete the data input by pressing the \textbf{INSERT} soft key.

We will now specify the geometry to be machined by the \textbf{TURNING (OUTER ROUGH)} cycle.

With turning, the part profile is always a custom shape so \textit{MANUAL GUIDE i} automatically guides you to define the profile of the part.

First define a starting point, beginning at the right-most point of the part.

Enter the following values using the MDI keyboard:

\textbf{DX: 1.13} \ (enter expression 1.25-2*.06)

\textbf{Z: 0.}

Complete the data input by pressing the \textbf{OK} soft key.

\textbf{Note:}

In \textit{MANUAL GUIDE i}, always specify the actual coordinates of the part profile ignoring approach positions. The profile is created with a series of lines and arcs.

The first feature is the chamfer on the end of the part, a straight line from the start point just defined to the diameter of 1.25, and at an angle of 45°.

From the soft key menu, select line by pressing the \textbf{LINE} soft key.
Next, specify the direction of the line from the choices displayed in the soft key menu.

Select a line moving left and up by pressing the soft key (LEFT and UP).

MANUAL GUIDE i provides several ways to enter position of the destination end point. In this case, the workpiece drawing provides the angle and the end point diameter.

Enter the following values using the MDI keyboard:
DX: 1.25 (end point diameter in X)
A: 45. (the line angle of 45°)

Complete the data input by pressing the soft key.

MANUAL GUIDE i displays the line so you can visually check that the data entered was logical.

Next we have a diameter that connects to a 0.125 radius and then to another line at a 60° angle.

Start by again selecting a line geometry by pressing the soft key.
This line is a diameter parallel to the workpiece centerline, press the LEFT soft key.

Note: The end point of the diameter or the blend point of the diameter to the radius is not given on the drawing. MANUAL GUIDE i allows us to specify the start point of one line (provided in the last step) and the end point of another line and a corner radius or chamfer between the two lines. MANUAL GUIDE i will then calculate all the intersecting blend points automatically.

Since we don’t know the end point in Z, just press the OK soft key.

The side-to-side yellow line signifies that the diameter is somewhere on that line, exactly where will be resolved later.

To specify the radius between the two lines, press the CR soft key.
Enter the corner radius value using the MDI keyboard:
R: .125 (corner radius)

Complete the data input by pressing the OK soft key.

We must now specify the end-point of the second line.

Press the LINE soft key.

The second line is moving to the left an up to a larger diameter.

Select a left-up line by pressing the L-UP soft key.
Since we did not specify the start-point of this line (tangent to the radius), or the end-point previous line (also tangent to the radius), we must provide a full specification of the end-point of this line and the angle of the line.

Enter the following values directly from the workpiece drawing using the MDI keyboard:
- **DX:** 2.375 (end point in X)
- **Z:** -1.125 (end point in Z – negative value)
- **A:** 60. (angle of the line)

Complete the data input by pressing the soft key.

MANUAL GUIDE \( i \) calculates the intersections of the radius blend points based on the start-point of the diameter, the corner radius between the lines, and the end-point and angle of the second line. It then draws a graphical representation to help you determine if logical information was entered.

The next set of part features are similar – two lines with a corner radius – so we repeat the procedure.

Press the \( \text{LINE} \) soft key.

This first line segment is a diameter parallel to the workpiece centerline, press the soft key.
The side-to-side yellow line signifies that the diameter is somewhere on that line, exactly where will be resolved later.

To specify the corner radius between the two lines, press the CR soft key.

Enter the corner radius value using the MDI keyboard:
R: .125 (corner radius)

Complete the data input by pressing the OK soft key.

Note: Again, the end point of the diameter or the blend point of the diameter to the radius is not given on the drawing. MANUAL GUIDE i allows us to specify the start-point of one line (provided in the last step) and the end-point of another line and a corner radius or chamfer between the two lines. MANUAL GUIDE i will then calculate all the intersecting blend points automatically.
We must now specify the end-point of the second line.

Press the **LINE** soft key.

The second line is moving to the left an up to a larger diameter.

Select a left-up line by pressing the **L-UP** soft key.

Since we did not specify the start-point of this line (tangent to the radius), or the end-point previous line that is tangent to the radius, we must provide a full specification of the end-point of this line and the angle of the line.

Enter the following values from the workpiece drawing using the MDI keyboard:

- DX: 4. (end point in X)
- Z: -2. (end point in Z – negative value)
- A: 80. (angle of the line)

Complete the data input by pressing the **OK** soft key.
MANUAL GUIDE calculates all the intersections based on the start-point of the diameter, the corner radius between the lines, and the end-point and angle of the second line. It then draws a graphical representation to help you determine if logical information was entered.

That complete the roughing profile of the part, we now just need to close the shape defining the blank material to be removed by the cycle.

Press the continuous menu right soft until the BLCONT soft key is displayed.

Press the BLCONT soft key.
MANUAL GUIDE i needs to know if the material to be removed is the green area (1) or the magenta area (2).

Press until the **FIG. 1** soft key to select the green area (1).

Confirm your selection of the green area, press the **YES** soft key.

Now that the profile is closed and complete, press the **CREATE** soft key.
You have two choices of where to keep the turning profile definition - 1. In the current program or 2. as a subprogram.

Since we nearly always use the same profile definition for the finish turning operation, it is more convenient to create a subprogram.

Use the cursor key to select the CREATE AS SUB PROGRAM choice.

Enter a subprogram number. It is recommended that the subprogram be in the range 8000-8499 with the last three digits the same as the main program. So if the main program is 1020, in this example, make the subprogram 8020.

Add a figure name to help with program management, for example “PROFILE FOR 1020”.

Complete the data input by pressing the OK soft key.

**End Tool T0101**

That completed entering the TURNING (OUTER ROUGH) cycle. All we need to do now to complete the operation is insert the **END TOOL** Fixed Form Sentence.

Press the **END** soft key.

**Note:**
The G-code G1120 is the cutting conditions for the TURNING (OUTER ROUGH) cycle. The M98 is a call to subprogram 8020, that contains the turning profile we just defined.
If necessary, press the \[ \text{ Arrow Down} \] key to highlight the END TOOL Fixed Form Sentence and press the \[ \text{INSERT} \] soft key.

That completes the outer roughing operation.
It is a best practice to verify each machining operation as you create them. It is much simpler than trying to test and debug a completed program at the end.

Press the continuous menu right key twice to display the simulation soft key.

Press the SIMLAT soft key.

Note: The blank displayed is the last part that was simulated. When we run the simulation for the first time it will read the blank size and display it correctly. Then that blank will become the last part that was simulated.

Press the REWIND soft key to rewind the program.
The soft keys for the 3D animation are like those used on a VCR or media player.

Press the **START** soft key to display the machining animation.

The animation shows the cutting rough the material at a rapid speed.

Note: The message 5010 END OF RECORD is displayed because we do not have inserted an END PROGRAM (M30) yet. We will fix this later.

If you want to step through the cycle step by step, press the **REWIND** soft key and then repeatedly press the **SINGLE** soft key until the program is back at the M01 in the program.

To exit the 3D solid model animation screen, press the **GRPOFF** soft key.
The next operation is a drilling cycle. This section explains the various parameters of the drilling cycle. Note that data item names with an asterisk (*) are optional and do not have to be specified. The graphic is the one displayed when that field is selected. The data required is highlighted in yellow.

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic Description</th>
</tr>
</thead>
</table>
| **Q** PECKING CUT DEPTH* | Depth of cut per peck. (positive value)  
Specify a peck cut depth if hole depth is 3 times the drill diameter or more. Leave blank to drill hole in one motion.  
The retract of the peck can be as small as 5-thousandths, so peck drilling does not significantly affect cycle time on a modern machine. |
| **K** GO PAST AMOUNT* | Drilling operations are dimensioned to the bottom of the full diameter, not to the point of the drill. However, tools are set by measuring to the end of the point of the tool.  
The go past amount parameter moves the tip of the tool deeper so that the CUT DEPTH specified will be achieved. For 118° use expression 3 * drill diameter.  
The go past amount is only displayed if parameter 27020#0=0. |
| **H** GO PAST AMOUNT SET | Determines if a go past amount is calculated on the DRILLING cycle DETAIL tab. Use the soft key to specify a go past amount should be calculated on the DETAIL tab.  
The GO PAST AMOUNT SET field is only displayed if parameter 27020#0=1. |
| **C** CLEARANCE | The distance between the hole BASE POSITION and the R-plane where the drill feed will change from the axis rapid rate to the feedrate specified. |
| **F** FEED RATE | Cutting feed rate. (positive value). |
### Cutting Conditions Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>P DWELL TIME</td>
<td><img src="image" alt="Graphic" /></td>
<td>Dwell time at the bottom of the hole before retract (in seconds, positive value).</td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td><img src="image" alt="Graphic" /></td>
<td>Determines how the X and Z axes will approach the hole from their position prior to the cycle (the positions in the START TOOL Fixed Form Sentence in this case). Use the soft keys to specify the motion required.</td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td><img src="image" alt="Graphic" /></td>
<td></td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td><img src="image" alt="Graphic" /></td>
<td></td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td><img src="image" alt="Graphic" /></td>
<td></td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td><img src="image" alt="Graphic" /></td>
<td></td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td><img src="image" alt="Graphic" /></td>
<td></td>
</tr>
</tbody>
</table>

### Position/Size Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>B BASE POSITION (Z)</td>
<td><img src="image" alt="Graphic" /></td>
<td>Absolute Z coordinate of the surface of the workpiece. This is typically used to reduce air cut time when the drilling is performed in a previously machined pocket. Specify a value of zero if machining on the end of the part.</td>
</tr>
<tr>
<td>L CUT DEPTH</td>
<td><img src="image" alt="Graphic" /></td>
<td>Incremental hole depth from the BASE POSITION. This is typically a negative value. A go past amount should be specified or calculated to allow for the tool point or to make sure you machine a through hole cleanly.</td>
</tr>
</tbody>
</table>
### Face Position Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y FACE POSITION</td>
<td><img src="image" alt="Diagram" /></td>
<td>Determine if the hole specification depth is for a Z positive face or a Z negative face (using subspindle). Press the (+FACE) for a positive face. Press the (-FACE) for a negative face. The Position/Size tab and the FACE POSITION field is only displayed if parameter 27100#4=1</td>
</tr>
</tbody>
</table>

### Detail Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A START CUT DEPTH*</td>
<td><img src="image" alt="Diagram" /></td>
<td>Tooling manufacturers may recommend a lower feedrate if the material surface is inclined, concaved or irregular. This field provides an optional feedrate to be used for until the drill is fully engaged with the material. This parameter specifies the distance to use the feedrate specified by the START FEED RATE field.</td>
</tr>
<tr>
<td>S START FEED RATE*</td>
<td><img src="image" alt="Diagram" /></td>
<td>Tooling manufacturers may recommend a lower feedrate if the material surface is inclined, concaved or irregular. This field provides an optional feedrate to be used for until the drill is fully engaged with the material. This parameter specifies the feedrate to use for the depth specified by the START CUT DEPTH field.</td>
</tr>
<tr>
<td>D END CUT DEPTH*</td>
<td><img src="image" alt="Diagram" /></td>
<td>Tooling manufacturers may recommend a lower feedrate when exiting through or cross holes. This field provides an optional feedrate to be used for the last portion of the drilling cycle. This parameter specifies the distance to use the feedrate specified by the END FEED RATE field.</td>
</tr>
</tbody>
</table>
### Detail Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>E</strong> END FEED RATE*</td>
<td>Tooling manufacturers may recommend a lower feedrate when exiting through or cross holes. This field provides an optional feedrate to be used for the last portion of the drilling cycle. This parameter specifies the feedrate to use for the depth specified by the END CUT DEPTH field.</td>
</tr>
<tr>
<td><strong>U</strong> TOOL DIAMETER</td>
<td>The diameter of the drill to calculate the go past amount for the drill point. This field is only displayed when parameter 27020#0=1 and the the GO PAST AMOUNT SET field is set to SETTING.</td>
</tr>
<tr>
<td><strong>V</strong> NOSE ANGLE</td>
<td>The point angle of the drill to calculate the go past amount for the drill point. This field is only displayed when parameter 27020#0=1 and the the GO PAST AMOUNT SET field is set to SETTING.</td>
</tr>
<tr>
<td><strong>K</strong> GO PAST AMOUNT</td>
<td>This field is calculated using the TOOL DIAMETER (U) and the NOSE ANGLE (V) fields when the soft key is pressed. This field is only displayed when parameter 27020#0=1 and the the GO PAST AMOUNT SET field is set to SETTING.</td>
</tr>
<tr>
<td><strong>R</strong> PECKING CLEARANCE*</td>
<td>Canned cycles for drilling on a turning CNC can either retract by a small amount (high-speed peck drilling - parameter 5101#2=0) or retract all the way to the R-plane between peck (peck drilling - parameter 5101#2=1). When advancing from the R-plane to make the next peck, this value specifies clearance to transition from the rapid rate to the feed rate. If this field is left blank when displayed, the clearance amount is specified by parameter 5115.</td>
</tr>
</tbody>
</table>
If necessary, press the continuous menu left key until the soft keys are displayed.

Start Tool T0909

Press the soft key.

Use the key to highlight the START TOOL RPM Fixed Form Sentence and press the soft key.

Note: Centerline tools like drills do not use Constant Surface Speed programming mode.
The message “IMPERFECT WORD MUST BE CHANGED” is displayed.

Use the CNC cursor keys to select each field in turn. Enter the correct value and press the key on the MDI keyboard to confirm the new value. T0909 (station for 3/8 drill), S1500 (spindle speed in RPM) and X4.2 (initial approach diameter).

Note:
Again, make sure you use the ALTER key on the MDI keyboard and not the ALTER soft key to change the value of the fields.

Note:
After manual editing data, use the cursor keys to position the cursor back on the last line in the program (the line before the %) and all the way to the left so that the whole line is highlighted.

Drilling Cycle

Press the CYCLE soft key

Use the cursor key to highlight the DRILLING cycle on the HOLE MACHINING cycles tab.

Press the SELECT soft key.
Enter the following values using the MDI keyboard:

Q: - (blank) (depth of each peck)

H: SETTING (press soft key)

C: 0.1 (approach clearance from end of part)

F: 0.007 (IPM feedrate for .375 drill)

P: (blank - dwell time)

Z: X->Z MOV (press soft key)

Note:
The GO PAST AMOUNT SET field is only displayed if parameter 27020#0=1.

Press the key to select the POS./SIZE tab.

Enter the following values using the MDI keyboard:

B: 0. (base position)

L: -0.75 (incremental depth of hole)

Press the key to select the FACE POSIT tab.

Note:
You may consider setting the base position to .030 to allow for the finishing allowance on the part face. If you do, you must also add 0.030 to the CUT DEPTH (L=-.78), because it is an incremental value from the base position. In practice, do not worry about the finishing allowance, because the first “peck” will be a feed move from the approach position Z0.1 by the PECKING CUT DEPTH.
Enter the following values using the MDI keyboard:
A: (leave blank)
S: (leave blank)
D: (leave blank)
E: (leave blank)
U: 0.375 (diameter of drill)
V: 118. (point angle of drill)

K: press the CALC soft key to calculate this field.

Complete the data input, press the INSERT soft key.

Note:
Most MANUAL GUIDE i cycles generate a least two blocks, one for the cutting conditions and one for the machined geometry. The drilling cycle contains all the information required in the single G1101 block.
End Tool T0909
That completes the DRILLING cycle. All we need to do now to complete the operation is insert the END TOOL Fixed Form Sentence.

Press the soft key.

Use the cursor key to highlight the END TOOL Fixed Form Sentence and press the soft key.
It is a best practice to verify each machining operation as you create them. It is much simpler than trying to test and debug a completed program.

Press the **continuous menu right key** twice to display the **SIMLAT** simulation soft key.

Press the **SIMLAT** soft key.

Press the **REWIN** soft key to rewind the program.
Press the **START** soft key to display the machining animation.

The animation shows the cutting rough the material at a rapid speed.

**Note:**
The message 5010 END OF RECORD is displayed because we do not have inserted an END PROGRAM (M30) yet. We will fix this later.

If you want to step through the program step by step, press the **REWIND** soft key and then repeatedly press the **SINGLE** soft key until the program is back at the M01 in the program.

To exit 3D solid model animation mode, press the **GRPOFF** soft key.
The next operation is the finish outer turning cycle. This section explains the various parameters of the cycle. Note that data item names with an asterisk (*) are optional and do not have to be specified. The graphic is the one displayed when that field is selected. The data required is highlighted in yellow.

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CUTTING DIRECTION</td>
<td>Press the soft key to finish in the Z minus direction. Press the soft key to finish in the Z plus direction. Press the soft key to optimize the finish in the Z minus direction. Press the soft key to optimize the finish in the Z plus direction. When either of the optimize soft keys are selected, and optimize tab is displayed. Those parameters are not discussed here.</td>
</tr>
<tr>
<td>FEED RATE</td>
<td>The cutting feedrate for finishing (positive value).</td>
</tr>
<tr>
<td>X-AXIS CLEARANCE</td>
<td>The approach position in the X-Axis. The X-Axis moves to this position at rapid before the finishing pass begins. This value is a positive, radius value.</td>
</tr>
<tr>
<td>Data Item</td>
<td>Graphic Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>--------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>M Z-AXIS CLEARANCE</td>
<td>The approach position in the Z-Axis. The Z-Axis moves to this position at rapid before the finishing pass begins. This value is a positive value.</td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td>Soft keys describe choices. The tool moves in Z first, then X (initial value). The tool moves in X first, then Z. The tool moves in X and Z simultaneously.</td>
</tr>
<tr>
<td>S CUT DEPTH DIRECTION</td>
<td>Soft keys describe choices. Press -X or +X soft keys to select direction.</td>
</tr>
<tr>
<td>X POCKET CUTTING</td>
<td></td>
</tr>
</tbody>
</table>
## Cutting Conditions Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y OVERHANG CUTTING</td>
<td><img src="image" alt="Diagram" /></td>
<td>Determines if MANUAL GUIDE attempts to cut into overhangs in the face of the defined profile. Note that depending on the tool geometry, some areas of the overhang may remain. This field is displayed when CNC system parameter 27100#1=1. If this field is not displayed (parameter 27100#1=0), overhangs are always machined. Press the <strong>CUT</strong> soft key to machine as much of the pocket that is feasible with the tool selected. Press the <strong>NOTHING</strong> soft key to not machine the pocket.</td>
</tr>
</tbody>
</table>
If necessary, press the continuous menu left key until the soft keys are displayed.

Start Tool T0202

Press the soft key.

Use the key to highlight the START TOOL CSS Fixed Form Sentence and press the soft key.
The message “IMPERFECT WORD MUST BE CHANGED” is displayed.

Use the CNC cursor keys to select each field in turn. Enter the correct value and press the key on the MDI keyboard to confirm the new value.

- T0202 (station for finishing tool),
- S900 (spindle speed in RPM at φ4.2),
- X4.2 (approach diameter), and
- S1000 (CSS spindle speed in FPM).

Note:
Again, make sure you use the ALTER key on the MDI keyboard and not the ALTER soft key to change the value of the fields.

Note:
After manual editing data, position the cursor back on the last line in the program (the line before the %) and all the way to the left so that the whole line is highlighted.

TURNING (OUTER FINISH) Cycle

Press the soft key

Press the cursor key to select the tab.
Enter the following values using the MDI keyboard and the soft keys:

P: -Z (press the \(-z\) soft key)
F: 0.12 (0.012 ipr - tool mfg. recommendation)
L: .05 (clearance in X)
M: .05 (clearance in Z)

Z: Z->X MOV (press the \(z\rightarrow x\) soft key)

To complete the finish turning cutting conditions data entry, press the \(\text{INSERT}\) soft key.
MANUAL GUIDE

i now needs the profile for the finishing pass. We could define a new profile, but since we already saved the roughing profile as a subprogram, we can re-use it.

Cancel the profile definition by pressing the **CANCEL** soft key.

Press the **→** key to select the **SUBPROGRAM** tab.

**Note:**

On the FANUC Series 0i-TD CNC, the parameters 14720 and 14721 specify the range of subprograms to be displayed. In this example, 14720=8000, 14721=8499.

If necessary, use the cursor keys to highlight **O8020**, the profile we created for program 1020 and press the **SELECT** soft key.
The next operation is the finish face turning cycle. This section explain the various parameters of the cycle. Note that data item names with an asterisk (*) are optional and do not have to be specified. The graphic is the one displayed when that field is selected. The data required is highlighted in yellow.

### Cutting Conditions Tab

<table>
<thead>
<tr>
<th>Data Item</th>
<th>Graphic</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>P</strong> CUTTING DIRECTION</td>
<td><img src="image" alt="Graphic" /></td>
<td>- Press the soft key to finish in the X minus direction.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Press the soft key to finish in the Z plus direction.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Press the soft key to optimize the finish in the minus direction.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- Press the soft key to optimize the finish in the plus direction.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Optimized tab and parameters not discussed here.</td>
</tr>
<tr>
<td><strong>F</strong> FEED RATE</td>
<td><img src="image" alt="Graphic" /></td>
<td>The cutting feedrate for finishing (positive value).</td>
</tr>
<tr>
<td><strong>L</strong> X-AXIS CLEARANCE</td>
<td><img src="image" alt="Graphic" /></td>
<td>The approach position in the X-Axis. The X-Axis moves to this position at rapid before the finishing pass begins. This value is a positive, radius value.</td>
</tr>
<tr>
<td><strong>M</strong> Z-AXIS CLEARANCE</td>
<td><img src="image" alt="Graphic" /></td>
<td>The approach position in the Z-Axis. The Z-Axis moves to this position at rapid before the finishing pass begins. This value is a positive value.</td>
</tr>
<tr>
<td>Data Item</td>
<td>Graphic</td>
<td>Description</td>
</tr>
<tr>
<td>-----------------</td>
<td>---------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Z APPROACH MOTION</td>
<td>Soft keys describe choices</td>
<td>Determines how the X and Z axes will approach the workpiece from their position prior to the cycle (the positions in the START TOOL Fixed Form Sentence in this case). Use the soft keys to specify the motion required.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- the tool moves in Z first, then X (initial value).</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- the tool moves in X first, then Z.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>- the tool moves in X and Z simultaneously.</td>
</tr>
<tr>
<td>X POCKET CUTTING</td>
<td></td>
<td>Determines if MANUAL GUIDE attempts to cut into pockets in the defined profile. Note that depending on the tool geometry, some areas of the pocket may remain.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This fields is displayed when CNC system parameter 27100#1=1.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If this field is not displayed (parameter 27100#1=0), pockets are always machined.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Press the <strong>CUT</strong> soft key to machine as much of the pocket that is feasible with the tool selected.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Press the <strong>NOTCH</strong> soft key to not machine the pocket.</td>
</tr>
<tr>
<td>Y OVERHANG CUTTING</td>
<td></td>
<td>Determines if MANUAL GUIDE attempts to cut into overhangs in the face of the defined profile. Note that depending on the tool geometry, some areas of the overhang may remain.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>This fields is displayed when CNC system parameter 27100#1=1.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>If this field is not displayed (parameter 27100#1=0), overhangs are always machined.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Press the <strong>CUT</strong> soft key to machine as much of the pocket that is feasible with the tool selected.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Press the <strong>NOTCH</strong> soft key to not machine the pocket.</td>
</tr>
</tbody>
</table>
The profile defined in the roughing operation is displayed.

**TURNING (FACE FINISH)**

Typically, we would now insert the **END TOOL** Fixed Form Sentence, but we will use the same tool to finish face the end of the part.

Press the \( \text{CYCLE} \) soft key.

Use the cursor keys to select the **TURNING** cycles tab and to highlight the **TURNING (FACE FINISH)** cycle.

Press the \( \text{SELECT} \) soft key.

The **TURNING (FACE FINISH)** cycle is capable of turning a complex profile, so the choice for **CUTTING DIRECTION** may look confusing. Our profile is a flat face in the \(-X\) direction.

If necessary, set the **CUTTING DIRECTION (P)**, press the \(-X\) soft key.
Enter the remaining values using the MDI keyboard and the soft keys:

- **F**: .012 (0.012 ipr - tool mfg. recommendation)
- **L**: .05 (clearance in X - preference)
- **M**: .05 (clearance in Z - preference)
- **Z**: Z→X MOV (press the **soft key**)

To complete the cutting conditions data entry, press the **soft key**.

The roughing contour did not include a definition of the finished face, because we started the profile at the first chamfer. Therefore, we must define the face of the part in a simple rectangle on the end of the part. The starting position is the edge of the machining blank in X. Since we have already roughed the part, we can start at the 1.25 inch diameter.

Enter the following values using the MDI keyboard:

- **DX**: 1.25 (outer diameter of machined blank)
- **Z**: 0. (end of part in Z)

To complete the cutting conditions data entry, press the **soft key**.

Press the **soft key**.
Press the \textit{DOWN} soft key.

Enter the following values using the MDI keyboard:

\texttt{DX: -.03125} (enter expression \texttt{-1/32}, one tool nose radius across the centerline of part)

To complete the data entry, press the \textit{OK} soft key.

Press the \textit{LINE} soft key.
Press the **RIGHT** soft key.

*Note:* Since the LINE DIRECTION is already set to RIGHT by default, you can just confirm and skip this step if you like.

Since we left 0.030-in on the faces during roughing, we must define out to that 0.03-inch surface. Because this position is to the right of the program zero point, it is a positive Z-value.

Enter the following values using the MDI keyboard:

Z: 0.03 (edge of blank)

To complete the data entry, press the **OK** soft key.

Press the **BLCONT** soft key until the **BLCONT** soft key is displayed.
Press the \textbf{BLCONT} soft key.

No specify which side of the defined profile to remove material, in this case the only logical one is shown as FIG. 1.

Press the FIG. 1 soft key.

Confirm the selection, press the \textbf{YES} soft key.
The profile is now closed and complete, press the CREATE soft key.

Since this exact facing profile is unlikely to be reused, just insert it in the current program.

Press the OK soft key.

Note that the G1128 is the finish facing cutting conditions and it is followed by several G-code defining the machining profile, rather than a macro call. The G1456 ends the profile definition.

End Tool T0202
Now we can end the finishing tool.

Press the END soft key.
With the **END TOOL** Fixed Form Sentence highlighted, press the **INSERT** soft key.
Since this is also the end of the program, we can also insert the **END PROGRAM** Fixed Form Sentence.

Press the **END** soft key.

With the **END PROGRAM** Fixed Form Sentence highlighted, press the **INSERT** soft key.
Press the continuous menu right key twice to display the simulation soft key.

Press the SIMLAT soft key.

Press the REWIND soft key to rewind the program.
Press the **START** soft key to display the machining animation.

The animation shows the machining at a rapid speed.

**Note:** The message 5010 END OF RECORD no longer appears since we have inserted an END PROGRAM (M30).

If you want to step through the cycle step by step, repeatedly press the **SINGLE** soft key until the program is back at the M30 in the program. Note that the program automatically rewound after the initial run through because of the M30 in the part program.
MANUAL GUIDE i has several features to help you verify a program more effectively. Let start by zooming the display to get a closer look.

Press the **soft key one time.**

Press the **soft key twice to enlarge the part viewed.**

Press the **soft key, twice. Note that the window moves down, not the part.**
Press the \( \text{MOVE} \) soft key, twice.

The finished part should now be reasonably centered in the SIMULATE-ANIIMATE window.

You can return to the original view easily by pressing the \( \text{AUTO} \) soft key, but do not do it now.

Press the \( \text{soft key} \) one time to return to the play soft keys.

Press the \( \text{SINGLE} \) soft key multiple times until the rough facing is completed (see image left).

Note the X-axis and Z-axis positions after the facing move. The X-axis has moved across the workpiece centerline to eliminate any small artifact being left on the part. The Z-axis is at +0.03, leaving a 30-thousands finishing amount on the face as instructed.
Press the **SINGLE** soft key multiple times until the roughing tool is at X3.14 and Z0.1 (see image left). You will notice that the roughing passes move in negative Z as programmed. The Z.01 position represents the 100-thousands approach distance from the finished profile as programmed. That is 100-thousands from the finished profile but only 70-thousandths from the material surface, since we left 30-thousandths on the face.

Press the **SINGLE** soft key one time until the roughing tool is at X2.88 and Z0.1 (see image left). This is the position prior to making the next roughing cut in negative Z.

Press the **SINGLE** soft key four times, noting the X and Z positions. First the tool cut in negative Z. Next the tool machine cuts up the face of the part to X3.36. That is 80-thousandths on diameter or the 40-thousandths depth of cut on radius that we specified. Next the tool retracts at 45° 100-thousandths on diameter (50-thousandths on radius) to X3.06 and plus 50-thousandths in Z. Finally the Z-axis retracts to the clearance position.

So when the CUT RIZE method is used, the actual escape distance for the Z retract will be the escape distance + depth of cut, or in this example, (.050 + .040) * 2 = 0.180 = 3.06 - 2.88.
Press the **START** soft key and let the animation complete.

Note the smoother nature of the surface of the workpiece after the finishing passes.

Press the **CUTDSP** soft key.

Press the **START** soft key and let the animation complete.

The part is processed with one quarter cut away, so you can see the internal details.

Press the **** soft key.

*Note: You can also set parameter 27195 bit 4 to a 1 to allow you to specify the retract distance each time in the drilling cycle cutting conditions.*
Press the **ROTATE** soft key.

Press the **Z** soft key, then press the **OK** soft key.

Press the **AUTO** soft key to resize the blank and then press the **OK** soft key to return to the animation play screen.
Press the **START** soft key and let the animation complete.

Now you can see the part profile more clearly. (image shown during the finish turning operation.)
Fixed Form Sentences

<Mode>=1,<Tab>=1,<Title>=START PROGRAM,<Code>=(START PROGRAM);G20 G18 G40 G80 G97 G99;G50 S????;
<Mode>=1,<Tab>=1,<Title>=START TOOL CSS,<Code>=(START TOOL);G40G80G97G99;T?M41;G97S?M03;G00X?Z0.1M08;G96S?;
<Mode>=1,<Tab>=1,<Title>=START TOOL RPM,<Code>=(START TOOL);G40G80G97G99;T?M41;G97S?M03;G00X?Z0.1M08;
<Mode>=1,<Tab>=5,<Title>=END PROGRAM,<Code>=(END PROGRAM);G30U0.M08;G30W0.M05;M30;
<Mode>=1,<Tab>=5,<Title>=END TOOL,<Code>=(END TOOL);G97;G30U0.M05;G30W0.M05;M01;

Tool Data

MGISYS : 0021 14/10/21
%
G1981 P001 K10 T  ‘ROUGH’ S 1 A 93.0 B  80.0
G1981 P002 K10 T  ‘FINISH’ S 1 A 93.0 B  80.0
G1981 P009 K20 T  ‘DRILL’ S 2 A 118.0
%

Tool Geometry Offsets

G10P10001X0.0000Z0.0000R0.0313
G10P10002X0.0000Z0.0000R0.0313
G10P10009X0.0000Z0.0000R0.1875
Program Listing

%  
O1020 (EXAMPLE TURNING PART 2)  
(START PROGRAM)  
G20G18G40  
G54G80G97G99  
G50S3000  
G1900D4.L5.K0.05  
(START TOOL CSS)  
G40G80G97G99  
T0101M41  
G97S900M03  
G00X4.2Z0.1M08  
G96S1000  
G1120P1.Q0.04H100.C0.03D0.03F0.012E0.012V0.012W2.U0.05L0.1M0.1Z10.  
N2.I0.03K100.  
M98P8020 (PROFILE FOR 1020)  
(END TOOL)  
G97  
G30U0.M05  
G30W0.M05  
M01  
(START TOOL RPM)  
G40G80G97G99  
T0909M41  
G97S1500M03  
G00X4.2Z0.1M08  
G1101H2.C0.1F0.007Z11.B0.L0.75Y1.U0.375V118.K0.1127  
(END TOOL)  
G97  
G30U0.M05  
G30W0.M05  
M01  
(START TOOL CSS)  
G40G80G97G99  
T0202M41  
G97S900M03  
G00X4.2Z0.1M08
G96S1000
G1126P1.F0.012L0.05M0.05Z10.
M98P8020(PROFILE FOR 1020)
G1128P3.F0.012L0.05M0.05Z10.S1.
G1450H0.V0.625A0.
G1451H0.V-0.015625K7.D-0.015625L0.M0.T1.
G1451H0.03V-0.015625K1.C0.03L0.M0.T1.
G1451H0.03V0.625K3.C0.03D0.625L0.M0.T2.
G1451H0.V0.625K5.C0.D0.625L0.M0.T2.
G1456
(END TOOL)
G97
G30U0.M05
G30W0.M05
M01
(END PROGRAM)
G30U0.M08
G30W0.M05
M30
O8020(PROFILE FOR 1020)
G1450H0.V0.565A0.
G1451H-0.06V0.625K4.D0.625A45.L0.M0.T1.
G1451H-0.853072V0.625K5.L0.M0.T1.
G1455H-0.961325V0.6875R0.125I-0.853072J0.75K2.T1.
G1451H-1.25V1.1875K4.C-1.25D1.1875A60.L0.M0.T1.
G1451H-1.751847V1.1875K5.L0.M0.T1.
G1455H-1.874948V1.290794R0.125I-1.751847J1.3125K2.T1.
G1451H0.V2.K1.C0.D2.L0.M0.T2.
G1451H0.V0.565K7.C0.D0.565L0.M0.T2.
G1456
M99
%
% 
O1010(PARTIAL EXAMPLE TURNING PART 1) 
(START PROGRAM) 
G20G18G40 
G54G80G97G99 
G50S3000 
G1900D2.25L4.75K0.025 
(START TOOL CSS) 
G40G80G97G99 
T0101M41 
G97S1560M03 
G00X2.450Z0.1M08 
G96S1000 
G1120P1.Q0.04H100.C0.03D0.005F0.012E0.012V0.012K100.W1.U0.05L0.05M0.1Z10. 
%