CNC TURNING
Foreword

MIC has produced this book for us in its Industrial Maintenance Journeyman Programme and it is specifically designed to introduce the basics of maintenance.

This book is intended for use as a reference text to be supplemented by notes and explanations and does not stand alone.

Compilation of this book was completed with standard published material, Tel-A-Train and resource personnel at MIC. No claim is made to the ownership of any material contained herein.

THIS BOOK IS NOT FOR SALE

REFERENCE TEXT USED
# TABLE OF CONTENTS

## CNC TURNING

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>General Description</td>
<td>3</td>
</tr>
<tr>
<td>Operating Controls</td>
<td>4</td>
</tr>
<tr>
<td>CRT and MDI Panel</td>
<td>5</td>
</tr>
<tr>
<td>MDI and CRT Panel Functions</td>
<td>6</td>
</tr>
<tr>
<td>Operators Panel</td>
<td>8</td>
</tr>
<tr>
<td>Operators Panel Functions</td>
<td>9</td>
</tr>
<tr>
<td>Part Programme Structure</td>
<td>13</td>
</tr>
<tr>
<td>“M” Codes</td>
<td>13</td>
</tr>
<tr>
<td>Spindle Direction and Axis Movement Conventions</td>
<td>15</td>
</tr>
<tr>
<td>Preparatory Functions or “G” Codes</td>
<td>16</td>
</tr>
<tr>
<td>G97 Direct RPM Programming</td>
<td>17</td>
</tr>
<tr>
<td>Programming Axis Movements</td>
<td>18</td>
</tr>
<tr>
<td>Incremental Programming</td>
<td>19</td>
</tr>
<tr>
<td>Constant Surface Speed Programming</td>
<td>19</td>
</tr>
<tr>
<td>G00 Positioning at Rapid Traverse</td>
<td>20</td>
</tr>
<tr>
<td>G01 Linear Interpolation at Feedrate</td>
<td>21</td>
</tr>
<tr>
<td>Circular Interpolation</td>
<td>22</td>
</tr>
<tr>
<td>Tool Nose Radius Compensation</td>
<td>23</td>
</tr>
<tr>
<td>Automatic Tool Nose Radius Compensation</td>
<td>26</td>
</tr>
<tr>
<td>Imaginary Tool Nose Location Code Numbers</td>
<td>27</td>
</tr>
<tr>
<td>G41 - G42 Compensation Summary</td>
<td>31</td>
</tr>
<tr>
<td>Multi Repetitive Cycles</td>
<td>31</td>
</tr>
<tr>
<td>G70 Finishing Cycle</td>
<td>34</td>
</tr>
<tr>
<td>G72 Transverse Area Clearance Cycle</td>
<td>34</td>
</tr>
<tr>
<td>G73 Pattern Repeat Cycle</td>
<td>36</td>
</tr>
<tr>
<td>G74 Peck Drilling and Face Grooving Cycle</td>
<td>37</td>
</tr>
<tr>
<td>G76 Complex Threadcutting Canned Cycle</td>
<td>39</td>
</tr>
<tr>
<td>Direct Drawing Dimension Programming</td>
<td>44</td>
</tr>
</tbody>
</table>
GENERAL DESCRIPTION

The machine is a numerically controlled centre lathe of horizontal configuration. Both axes are driven by A.C. Servo Motors. The main slideways are induction hardened and ground. Lubrication of all surface is automatic. Manual jogging of the slides is effected using push buttons or handwheel.

FEED DRIVES

Feed motion in both axes is provided by A.C. Servo Motors driven by a transistorised (PWM) system within the interface. Drive is transmitted via toothed belts on the X axis and Z axis, to a recirculating ballscrew and pre-loaded nut assembly. Positional feed-back is by Pulse coders which are integral with the drive motors. A similar device is associated with the spindle, to allow the synchronising of screw cutting. Absolute slideway position is detected and known at all times by absolute encoders. The position of the slideways is stored in CMOS RAM memory which is battery backed whilst the control system is powered down. The slideway position is monitored such that in any event of a mis-programmed dimension the slideway travel is limited by software which shuts down the CNC. Overrun buffers protect the slideways from mechanical damage in the event of software failure to detect overtravel.
OPERATING CONTROLS

The operator control station on the CNC Centre Lathe consists of separate upper and lower panels.

1. The MDI and CRT Panel (Upper)
This consists of a C.R.T. Display and keyboard for MANUAL DATA INPUT into the O.T. Control System.

2. Operators Panel (Lower)
This consists of a series of switches and push buttons used to control the functions of the machine.

Many of the functions of the control station are dependent on the machine STATUS which is continuously displayed at the bottom of the C.R.T. scree. Status message indicated are:-

- NOT READY: Indicates that the control unit or the servo system is not ready for operation.
- ALARM: Indicates that an alarm is occurring. The kind of alarm can be seen by pressing the ALARM button.
- BAT: Indicates that the power level of the battery is lower than the specified level. The battery is used to protect data stored in memory when the power is off. If this status is indicated change the battery.
- BUF: Indicates that the command data read into the buffer register has not been executed.
- JOG: Indicates that the manual continuous feed (JOG MODE) is selected.
- STEP: Indicates that the manual step feed (STEP MODE) is selected.
- AUTO: Indicates that automatic operation (AUTO MODE) is selected.
- M.D.I: Indicates that manual input data (M.D.I. MODE) is selected.
- EDIT: Indicates that memory edit (EDIT MODE) is selected. This status is indicated near the centre of the bottom line on the screen.
- EDIT: Indicates that editing is being executed. This status is indicated at the bottom right of the screen.
- SEARCH: Indicates that searching such as sequence number or word search is being executed.
OUTPUT: Indicates that the programme is being output by the use of the input/output interface.

INPUT: Indicates that the programme is being input by the use of the input/output interface.

COMPARE: Indicates that the programme is being compared with the content of the memory by use of the input/output interface.

LSK: Indicates label skip condition is active. i.e. The control does not read titles, part numbers, labels etc. on tape heading.
### FANUC OT MDI AND CRT PANEL FUNCTIONS

<table>
<thead>
<tr>
<th>DRG REF NO</th>
<th>DESCRIPTION</th>
<th>FUNCTION/OPERATION</th>
</tr>
</thead>
</table>
| 1          | C.R.T. Display with soft key buttons for menu selection | Used to display:  
(i) Positional data and values  
(ii) Command and programme data  
(iii) Tool compensations (Offsets and Data)  
(iv) Setting Data  
(v) Diagnostic Data  
(vi) Alarm Messages  
(vii) Status |
|            | Function Buttons (push buttons) | Function buttons are used to indicate via the CRT display major functions of the control rather like chapters in a book. |
| 2          | **POS** | Indicates current position. |
| 3          | **PRGRM** | Conducts the following:  
In edit mode - edit and display of the programme in memory.  
In M.D.I. mode - input and display of the M.D.I data.  
In auto mode - display of command value |
| 4          | **MENU**  
**OFFSET** | Setting and display of offset value |
| 5          | **DGNOS**  
**PARAM** | Setting and display of parameter and diagnostic data. |
<p>| 6          | <strong>OPR ALARM</strong> | Alarm number display and display of software operators panel. |
| 7          | <strong>AUX GRAPH</strong> | Graphic display. |</p>
<table>
<thead>
<tr>
<th>ORG REF NO</th>
<th>DESCRIPTION</th>
<th>FUNCTION/OPERATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>PAGE</td>
<td>(Push Buttons)</td>
<td>Several pages of information are included in the chapter selected with each function button.</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td>Advances to next page when pressed.</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td>Returns to previous page when pressed.</td>
</tr>
<tr>
<td>CURSOR</td>
<td>(Push Buttons)</td>
<td>The cursor is displayed as a line (the width of one character) below the relevant word.</td>
</tr>
<tr>
<td>10</td>
<td></td>
<td>Moves cursor forward word by word.</td>
</tr>
<tr>
<td>11</td>
<td></td>
<td>Moves cursor backward word by word.</td>
</tr>
<tr>
<td>NOTE:</td>
<td></td>
<td>Advancing cursor beyond the page changes the screen to the next page.</td>
</tr>
<tr>
<td>12</td>
<td>RESET</td>
<td>(Push Button) Used to reset input data also used to remove alarm.</td>
</tr>
<tr>
<td>13</td>
<td>DATA INPUT KEYS</td>
<td>(Push Buttons) Data input with the DATA INPUT KEYS is indicated on the penultimate line of the screen. The key functions as ADDRESS KEY when ADRS is indicated and as a NUMERIC VALUE KEY when NUM. or NO. is indicated. The address and the numeral are automatically switched by the N.C. according to various conditions (such as page currently selected, etc).</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>A</th>
<th>Q</th>
<th>K</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

P, C, H, and EOB are input when pressed once. Q, A, I and / are input when pressed twice. K is input when pressed three times.
<table>
<thead>
<tr>
<th>DRG REF NO</th>
<th>DESCRIPTION</th>
<th>FUNCTION/OPERATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>13</td>
<td>DATA INPUT KEYS</td>
<td>/E is input when pressed after the address</td>
</tr>
<tr>
<td></td>
<td></td>
<td>EOB Deletes data displayed on screen when pressed with a function key.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CAN</td>
</tr>
<tr>
<td></td>
<td>PROGRAMME EDIT</td>
<td></td>
</tr>
<tr>
<td></td>
<td>(Push Buttons)</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>ALTER</td>
<td>Used to alter one word of data in programmed editing.</td>
</tr>
<tr>
<td>15</td>
<td>INSERT</td>
<td>Used to insert one word of data in programme editing.</td>
</tr>
<tr>
<td>16</td>
<td>DELETE</td>
<td>Used to delete one word of data in programme editing.</td>
</tr>
<tr>
<td>17</td>
<td>INPUT</td>
<td>Used to input data as keyed in.</td>
</tr>
<tr>
<td>18</td>
<td>START</td>
<td>Used to execute M.D.I comments. Also to output data outside N.C. through I/O interface.</td>
</tr>
</tbody>
</table>

**Operators Panel**
OPERATORS PANEL FUNCTIONS

Program Protect (Key Switch)

Used to prevent part programme storage and editing operations being performed unless removable key
sets switch at ON position.

Program Source Keys
i. AUTO: The program stored in memory can be executed.
ii. EDIT: a) Registration of program to memory
     b) Modification, addition and deletion of program
     c) Punch-out for editing can be performed
iii. MDI: Operation by commands on the M.D.I. Panel can be executed.

Override

Used to modify traverse rate in selected more of either Jog, Rapid or programme feed.

Reset (Emergency Stop)

Used to stop ALL machine movements immediately in an emergency. Power is removed from all motors and control assumes a RESET state. Release button by turning clockwise.

NOTE: After button is released a manual HOME operation should be executed.

Operation Select Keys

i. Single Block: When this switch is on the control executes only one block of information every time the cycle start button is pressed.
   i. If switched on during Threading Cycle the feed is stopped after the threadcutting and the next non threading block is completed.
   ii. If switched on during canned cycle or special G Code, feed hold is applied, after one complete cycle is executed.

ii. Block Delete: When switched on this allows the control to ignore all blocks of information which have a slash “/” code as the first character in the block.

iii. Opt Stop: When switched on the cycle stops after a block containing M01 is executed. Cycle resumes by pressing “Cycle Start”.

iv. Dry Run: If this switch is ON in the cycle the programmed federate is ignored and the value shown on the Traverse Feed Switch is applied.

v. M/C Lock: When this switch is in the ON position all axis move commands are suppressed. (M, S and T functions are executed). Position registers are updated irrespective of no slide movements.
Owing to the use of the absolute encoding system the CNC will lose the true absolute position because the slideway movement is locked. It is very important to reset the absolute encoders after running the machine in “MACHINE LOCK”.

Adopt the following procedure:-

1. Press “Emergency Stop” push button
2. Press Control Off
3. Release Emergency Stop push button
4. Press Control On

This will reset the position register.

**Execution Keys**

i. Cycle Start: For starting automatic operation of programme.

ii. Feed Hold: When feed hold is pressed the button illuminates and:-
   i. The feed is stopped after deceleration if the slides are moving.
   ii. Dwell is not continued if the button is pressed while dwell is being executed.
   iii. The machine is stopped after the operation of the M.S. or T function.
   iv. If the feed hold is applied during threading mode the feed hold is not applied until the threading cycle is complete.

**Operation Keys**

i. Jog: In this mode the slides can be moved one axis at a time by use of the appropriate Axis Direction Key.

ii. Inc Jog: When selected allows axis movement by multiplying amount selected for each press of the axis direction key.


iv. Teach: Selects Teach Mode. Used in conjunction with the M.P.G. where the machine movements are stored in memory thus enabling programs to be created.
**Axis Direction Keys**

The X and Z keys are used to increment (Step), jog or zero return (Home) the desired axis in the direction selected. When used in conjunction with the RAPID key, the slides move at rapid traverse rates.

**Step Keys**

Used to select the magnifying power of the movement amount when used in conjunction with the handwheel.

**Handle Keys**

Used to select Axis direction for manual pulse generator.

**Other Keys**

1. **Brake Rel:** Releases magnetic brake to allow spindle to be rotated by hand.

2. **Turr Index:** For each press of the key the turret advances one station forward. Active in JOG mode only.

3. **Chuck Enable:** Activates chuck open/close footswitch. NOTE: To be operative control must be on. Spindle must be stopped and guard must be open. Cancelled when spindle starts.

4. **T’stk Enable:** Activates tailstock quill advance/retract footswitch.

5. **Jog off:** Illuminates – Used to get machine out of over travel:- Select jog. Hold jog off over travel button in whilst pressing required Axis direction button.

6. **Cont On:** Switches control on.

7. **Cont Off:** Switches control off.

8. **Lub Low:** Lights when low slideway lub oil level is detected.

9. **Pos Rec:** This button can be used to record the “Cut and Measure” procedure. The main advantage in utilising this button is that once it has been depressed the tool can be moved away from the workpiece to a safe position in any direction and then the Mx or Mz position input in the normal way because the tools original position when the button was pressed was recorded.
MINIMUM WORKING POSITION

BASIC SETTING DIMENSIONS FOR absolute encoders without buffers
PART PROGRAMME STRUCTURE

A CNC Lathe program is sequential and is ordered in blocks. Blocks are sequentially ordered in the CNC memory by using an ‘N’ word. A block is composed of several words such as dimension words and various coding words which switch on various functions on the machine. A fuller description of this structure is given in the FANUC OT Operators Manual. In the following text the structure of the program is built up and examples will be introduced with each feature explained.

BLOCK NUMBERS

As previously mentioned the blocks are numbered in memory using an ‘N’ word or sequence numbers can have up to four digits up to N9999. Blocks numbers in program are written as shown below:-

- N10 (Information) E0B
- N20 (Information) E0B
- N30 (Information) E0B
- N120 (Information) E0B

Notes:-

1. It is better to write block numbers in decades i.e. 10, 20, 30 etc., so that there are spaces available to edit in additional blocks when proving a program on the machine.

2. E0B is the end of block character which is punched on the tape preparation machine and allows the CNC to identify the end of the block. An explanation of program tape composition is given in the FANUC OT Operators Manual.

3. M30 is the code which is used to tell the CNC that this block is the end of the program. This code also resets the program to the beginning automatically.

MISCELLANEOUS FUNCTIONS OR “M” CODES

As described above an M30 word resets the program. M codes are used for things such as turning the spindle on or off in the programme as desired. A list of the available M codes and their use is given below:-
<table>
<thead>
<tr>
<th>M Code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Programme Stop</td>
</tr>
<tr>
<td>M01</td>
<td>Optional Stop</td>
</tr>
<tr>
<td>M02</td>
<td>End of Programme</td>
</tr>
<tr>
<td>M03</td>
<td>Spindle on Anti-clockwise</td>
</tr>
<tr>
<td>M04</td>
<td>Spindle on Clockwise</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle Stop</td>
</tr>
<tr>
<td>M08</td>
<td>Coolant on (All Machines)</td>
</tr>
<tr>
<td>M09</td>
<td>Coolant off</td>
</tr>
<tr>
<td>M18</td>
<td>Second Coolant pump on (Machines with Sauter Turret Option)</td>
</tr>
<tr>
<td>M30</td>
<td>End of programme and reset</td>
</tr>
<tr>
<td>M34</td>
<td>Work Catcher (In)</td>
</tr>
<tr>
<td>M35</td>
<td>Work Catcher (Out)</td>
</tr>
<tr>
<td>M40</td>
<td>Spindle Gear Range 1 (Low)</td>
</tr>
<tr>
<td>M41</td>
<td>Spindle Gear Range 2 (High)</td>
</tr>
<tr>
<td>M78</td>
<td>Chuck Close (Bar Feed Appl)</td>
</tr>
<tr>
<td>M79</td>
<td>Chuck Open</td>
</tr>
<tr>
<td>M98</td>
<td>Sub Programme Call</td>
</tr>
<tr>
<td>M99</td>
<td>Return to Main Programme</td>
</tr>
</tbody>
</table>
SPINDLE DIRECTION & AXIS MOVEMENT CONVENTIONS.
PREPARATORY FUNCTIONS OR “G” CODES

As can be seen from the example in the Programming Rules of ‘M’ Codes the short program for turning the spindle on or off does not give a spindle speed. In practice if this program was executed the spindle would not start because no spindle speed is commanded. The example shows how a fixed spindle speed can be programmed.

Example:-

N10  G97  S625M03  (Spindle on c/clockwise, spindle speed 625rpm)
N20  M05   (Spindle off)
N30  M30   (Programme rewind)

N.B. from now on E0B will be ignored for the purpose of clarity.

The function G97 is Direct RPM programming and with the associated “S” word gives a spindle speed of 625 rpm. This is a preparatory function or G code and is used to turn on control system functions such as threadcutting for example or rapid traverse. A list of the available preparatory or G codes is shown and a brief explanation of what they are. Explanations of their use will be gradually introduced as they are somewhat different from M codes.

Programming Rules:-

1. Only one G code must be programmed in a block.¹
2. G codes can be programmed with other words in the same block.

Further Considerations:-

M and G codes are generally nodal. This means the control and/or machine will stay on this mode until another code is programmed which changes this mode.

e.g. M08 coolant will stay on until programmed off by M09.

This may seem to be obvious in the case of simple functions like coolant but with G codes potentially dangerous situation can occur if this is not remembered. E.g. rapid traverse code G00 is nodal and the machine will stay in this mode until G01 (Feedrate straight line mode) is programmed or some other mode (Threadcutting). Always remember to switch back or to the required mode.

¹Upgraded controllers can handle more than one G code per block.
G97 Direct RPM Programming

The S word associated with this has a 4 digit formula (S4). This means the system will handle up to 9999 rpm. The CNC Centre Lathe will only run at its maximum rpm, therefore the S word value is the maximum spindle speed that can be programmed. The spindle speed is directly written e.g.

10 rpm = S10
200 rpm = S200
1500 rpm = S1500

Preparatory Codes

G00  Rapid Traverse
G01  Linear Interpolation
G02  Circular Interpolation (c/clockwise viewed from above)
G03  Circular Interpolation (clockwise viewed from above)
G04  Dwell
G10  Offset Value Setting
G20  Inch Data Input
G21  Metric Data Input
G27  Reference Point Return and Check
G28  Reference Zero Return
G33  Threading
G40  Tool Nose Radius Compensation Cancel
G41  Tool Nose Radius Compensation Left
G42  Tool Nose Radius Compensation Right
G70  Finishing Cycle
G71  Rough Turning Cycle
G72  Rough Facing Cycle
G73  Pattern Repeat Cycle
G74  Peck Drilling Cycle
G76  Thread Cutting Cycle
G77  Rough Turning Canned Cycle
G78  Thread Cutting Canned Cycle
G79  Rough Facing Canned Cycle
G92  Absolute Position Register Preset
G94  In/min or mm/min Feedrate
G95  In/rev or mm/rev Feedrate
G96  Constant Surface Speed Programming
G97  Direct RPM Programming / CSS off
PROGRAMMING AXIS MOVEMENTS

The CNC uses two words for commanding slide movements. The “X” word and the “Z” word. The X word controls cross slide movement. The Z word controls longitudinal carriage or saddle movement.

Colchester CNC Centre Lathe axis drive motors have inbuilt position detectors in them which together with the high precision ballscrews enables dimensions to be programmed to an accuracy of 0.0001 inch in the inch mode or 0.001mm in the metric mode.

CNC Centre Lathes can be programmed in the inch or metric more by programming G20 or G21 as a block on its own at the start of the program.

```
e.g. N10 G20 E0B (inch mode)
or N10 G21 E0B (metric mode)
```

The machine is set up to switch on in the metric mode. N.B. See Operating Section for detailed set up explanation.

There is a sign convention for the direction of X and Z slide movements this is shown in the diagram below:-

![Sign Convention Diagram](image)

There are two methods of programming axis movements (i) Incremental (U and W) (ii) Absolute (X and Z). Incremental will be dealt with first as it enables the way the control works to be understood easily in the initial instance.
Incremental Programming

When programming incremental moves two other words are used instead of X and Z. These are U and W. U and W represent incremental moves in the X and Z axes respectively. The reason for using a different letter address will become clearer later.

Let us consider the tooling layout at the beginning of the programming section and also the part shape shown below.

Assuming that the part shape has already been roughed out and that we require to drive a tool around this shape to take a finishing cut.

CONSTANT SURFACE SPEED PROGRAMMING

This is a standard feature on Colchester CNC Lathes which allows the machine to control the surface speed automatically.

The machine is driven through two gear ranges M40 and M41 (low and high ranges) which are automatically selected from program.

Range changing can be controlled with the spindle running. The machine automatically stops, selects the required range and the spindle restarts.

Colchester CNC Centre Lathes are fitted with a continuously variable speed spindle motor and the CNC allows this to be used so that a constant cutting speed is maintained as the cut-
ting diameter of the tool changes.

If the machine was continuously programmed for spindle speed changes in the G97 mode, then one would have to change the spindle speed each time the diameter changes by use of the following formula:-

\[ S = \frac{\pi DN}{1000} \text{ (Metric) where } D = \text{diameter in mm} \]
\[ \text{and } S = \text{cutting speed in metres/min} \]
\[ \text{and } N = \text{spindle rpm} \]

or

\[ S = \frac{\pi DN}{12} \text{ (Inch) where } D = \text{diameter in inches} \]
\[ \text{and } S = \text{cutting speed in feet/min} \]
\[ \text{and } N = \text{spindle rpm} \]

The CNC will do this automatically when the G96 code is used. To do this we have to start the program as shown below.

N10 G21

N20 G92 S 2000 (preset maximum spindle speed)

N30 G96 S 150 (constant surface speed 150 metres/min)

N40 G00 X 25.0 Z 102.0

Etc.

**G00 POSITIONING AT RAPID TRAVERSE**

As seen previously G00 specifies straight line positioning as Rapid Traverse Rate i.e. Metric 6m/min, Imperial 236 inch/min.

A tool moves to the X, Z position in the work co-ordinate system or from its current position to the position specified as the U, W distance at rapid traverse rate along each axis independently.

When both axes are commanded to move simultaneously the resulting departure approximates a 45 degree move.

Should both axes be programmed with dissimilar departures the axes will move at 45 degrees until the smaller departure is completed, the remaining departure will be completed along the normal axis path.
It is important to bear this in mind as linear interpolation does not take place so be very careful that a collision situation does not occur.

**COMMAND**

G00 X (U)_____ Z (W)________

G00 is modal and will be changed by G01, G02, G03, G33, G77, G78 AND G79.

**G01 LINEAR INTERPOLATION AT FEEDRATE**

G01 specification straight line moves at the programmed federate defined by an F word which may be in the same block or may prevail from some previous block.

Linear Interpolation moves a tool in a straight line to the X, Z position in the work coordinate system or from its current position to the position specified as the U, W distance.
When both axes are commanded to move simultaneously the tool will move to the programmed destination in a straight line regardless of the lengths.

**COMMAND**

G00  X (U)_____  Z (W)_________ F

G00 is modal and will be changed by G01, G02, G03, G33, G77, G78 AND G79.

**CIRCULAR INTERPOLATION**

The FANUC OT CNC has the ability to enable the machine to generate radii on the workpiece by driving both slideways in a circular arc simultaneously. This is achieved by using either of two MODAL G codes G02 or G03.

G02 – Circular Interpolation C.C.W.

The G02 code permits the slides to be traversed in circular path. The extent of the travel, together with the radius must be programmed in the same block as this code. The direction of motion is C.C.W. (viewed from operators position above the turret). This code is MODAL and will only be cancelled by G00, G01, G03, G04, G33 AND G92.
G03 – Circular Interpolation C.W.

The G03 code is identical in effect to the G02 code except that it permits circular arc traverse in a CLOCKWISE direction (viewed from operators position over turret). Again this code is MODAL.

The Block Format is as follows:-

N100 G02/G03 X…… Z…… R……

The X and Z co-ordinates are the END POINTS of the arc. The R word is the RADIUS of the arc to be generated (up to 180 degrees is possible but not practical on a lathe).

TOOL AND NOISE RADIUS COMPENSATION

All the examples examined to date have looked at straight line cutting moves and have not mentioned the effect of the tool nose radius. All tools have finite radii and in particular commercially available replaceable carbide insert tools have standard radii of 0.5mm, 0.8mm, 1.2mm and 1.5mm. Button tools can have radii as big as 25mm.

This has an effect on the profile of the part when machining. The reason for this is the way tools are generally set.
The diagram shows the tip of a typical right hand turning tool. The tool is set to tangential dimensions in the X and Z directions in the intersection of which gives the point “P”. Point “P” is the tool setting point because it is the only point that can be conveniently measured although it becomes immediately apparent that “P” does not machine anything.

This is irrelevant when turning diameters and faces because point “P” coincides with the programmed data taken from the part drawing.
When machining tapers and radii however, programming point “P” data will give dimensional errors as can be seen from the simple example of machining a 45 degree taper below.

![Diagram of taper machining with point P located within the profile.](image)

It can be seen that we have a program point “P” within the profile to obtain the correct dimensional form. Programming point “P” to drawing dimensions will produce an undersize chamfer.

Starting Point A

Let us consider the general case first.
Angle b a b = 45 degrees
ab = da = Tool Radius TR
ca is common to both triangles
therefore cb = cd.
It therefore follows that:

\[
\text{Angle } b\ a\ c = \text{Angle } c\ a\ d = \frac{\text{Angle } b\ a\ d}{2}
\]

\[
\tan 22.5\ \text{deg} = \frac{cb}{ba} \text{ therefore } cb = \tan 22.5\ \text{deg} \times ba = TR \tan 22.5\ \text{deg}
\]

Therefore Point "P" has to be compensated by TR - TR \tan 22.5\ \text{deg}

i.e. TR \ (1 - \tan 22.5\ \text{deg})

or in general TR \ (1 - \tan \alpha) \text{ where } \alpha = \text{angle of taper}

### AUTOMATIC TOOL NOISE RADIUS COMPENSATION

It is a pre-requisite to understanding the mechanics of tool nose radius compensation before using the facility of the control which enables compensation calculations to be made automatically. In effect this enables the programmer to be able to program the tool path as if the tool had a dead sharp point and thus point “P” can be programmed all the time.

In order to do this it is necessary to tell the CNC whereabout the tool point “P” is in relationship to the programmed data. E.g. for turning the tool on the outer surface of the workpiece and for using the tool on the inner surface of the workpiece. This is illustrated below:

Two “G” codes are used to instruct the CNC as to where the tool tip is.

G41 – Switches on tool nose radius compensation to left of part – generally external right hand turning.

G42 – Switches on tool nose radius compensation to right of part – generally right hand boring.

It is easy to decide which code by the following rule of thumb.
Programming Rules

1. Imagine yourself sitting on the tool tip at the centre of the tool nose radius. Look in the direction of the desired tool travel along the workpiece surface and decide whether you are sitting to the LEFT or the RIGHT of the work surface. Program G41 or G42 accordingly.

2. When the machine is first switched on the automatic tool nose radius compensation is OFF.

3. Program G40, G41 or G42 in a block on their own.

4. Tool nose radius compensation becomes active/inactive in the next move following it being switched on/off. This means that a slide movement will be generated vectorially with the move, therefore make sure that these codes are switched on with the tool well away from the job.

5. G40, G41 and G42 are MODAL.

6. Do not program G41 in the G41 mode or G42 in the G42 mode. It is acceptable to program G40 in the G40 mode.

Imaginary Tool Nose Location Codes:-

When using TNRC, the CNC also has to know the actual position of point “P” in relation to the tool radius centre point. The reason for this is that different tool geometry shapes will require the control system to generate different compensation moves for the same tool path data. This is illustrated below (following page):-
In case (a) the compensation generated is part of the point on the work surface whereas in case (b) the compensation generated is before the work surface change point.

There are 9 Imaginary Tool Nose Location Codes which describe the relevant tool shape. These are illustrated below and the relevant code number must be set against the particular tool on the Tool Offset Page under the “T” column, alternatively the value can be input via the part program (if the G10 option has been purchased). Numbers 0 and 9 are actually the tool nose radius centre and for most practical purposes these are rarely used because programming to the tool radius centre means that the tool path X and Z dimensions do not relate to the part drawing and are therefore difficult to follow for the average operator.

N.B. The G41 and G42 convention adopted in this manual differs slightly from the Fanuc manual which uses the opposite codes. The convention used follows what is generally recognised BUT requires that the R value on the Tool Geometry Page is a NEGATIVE value for the TNRC to work correctly.

If the FANUC convention is elected to be adopted then the G41/42 codes described MUST be reversed (See Fanuc OT Operators Manual).
IMAGINARY TOOL NOSE LOCATION CODE NUMBERS

Point to which tool nose offset is set
As used in the 'T' work of offset page
G41 – G42

The difference between each theoretical point and the actual point of contact on the tip radius is automatically compensated for by the control when either G41 or G42 codes are commanded.

The side to be cut, as viewed in the direction of travel, is stated by G41: left or G42: right.

When compensation is no longer required it can be cancelled with G40.

The start-up status of the control is G40 Tool Tip Compensation Cancel. G40, G41 and G42 are MODAL.

The amount of compensation calculated by the control will depend on the size of the insert radius, which is inserted in the tool offset file under the address ‘R’.

MULTI REPETITIVE CYCLES

There are a series of more complex canned cycles available for roughing more difficult component shapes and for simple programming of groove plunging and peck drilling. The salient details of the cycles related to CNC Centre Lathes with specific examples shown in this section but this is only in supplement to the detail provided in the FANUC OT Operators Manual which should be studied thoroughly as well as the following information.

G71 LONGITUDINAL AREA CLEARANCE CYCLE

This is a powerful canned cycle which will rough cut to any arbitrary finish part profile by merely defining the profile, the depth of cut and the finishing stock allowance in particular. For the roughing of bars and billets this enables very fast programming with a minimum of blocks which can be easily edited by changing the depth of cut and federate to cope with vir-
ually any changes in material specification:

The cycle is commanded using one or two G71 blocks and the format is as follows:-

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

Where in the first block designated G71:-

U is the depth of cut in feed in the X axis. This value is factory set in parameters at 5.0mm and this value is MODAL and does not need to be programmed unless a different value is required for the depth of cut. This is a RADIUS value.

R is the tool lift off when the end of the cut hits the profile and rapid traverses back for the next cut. This value is also factory set in parameters at a value of 1.0mm and is MODAL and does not need to be programmed unless a different value is required.

This is a RADIUS value.

In the second block designated G71:-

P and Q are the start and end block numbers which respectively define the beginning and end of the finish part profile being roughed out.

U is the stock allowance for a finishing cut in the X axis. For external roughing U is a PLUS value and for internal roughing U is a MINUS value. U is a DIAMETER value.

W is the stock allowance for a finishing cut in the Z axis. For roughing from the tailstock to the headstock direction (the usual case) W is a plus value. For roughing from the headstock to the tailstock direction W is a MINUS value.

F is the roughing cut federate and is programmed in the usual way.

T, and the T word can be programmed in the G71 calling block BUT ONLY as a tool offset value i.e. a change can be made to the two least significant digits in the T word, DEFINITELY NO CHANGE IN THE PART OF THE T WORD WHICH INDEXES THE TURRET.

If it is elected to use this BE ABSOLUTELY SURE that the machine parameters have not been changed to call up TOOL GEOMETRY OFFSET by the same part of the T word as TOOL WEAR OFFSETS. It is RECOMMENDED by the required T word contents are completely called up prior to calling the G71 block.

NOTES:-

1. W and R (first G71 block) and U and W (second G71 block) require exactly the same
dimension word rules related to Decimal Point Programming.

2. F and S (and T) and programmed as with ordinary programming formats.

An example of this cycle is given below for examination and then a review of other precautions to be observed will be made:-

**Cutting Conditions:**
- Depth of Cut = 3mm
- Surface Speed = 150m/min
- Feedrate = 0.3mm/rev

**Stock Allowances:**
- X axis = 0.5mm
- Z axis = 0.25mm

**Programme Zero**

```plaintext
N10 G21
N20 G92 S2000 T0100
N30 G96 S150.0 M08
N40 G00 X 135.0 Z 155.0 M03 T0101
N50 G71 U 3.0 R 1.0
N60 G71 P70 Q130 U 1.0 W 0.25 F 0.3
N70 G00 X 30.0 F 0.15 S 200.0
N80 G01 U 20.0 W -20.0
N90 Z 120.0
N100 X 80.0 Z 85.0
N110 Z 65.0
N120 G02 X 120.0 Z 45.0 R 20.0
N130 G01 X 130.0 Z 40.0
N140 G00 X 400.0 Z 800.0 T0100
N150 T0200
N160 G41
N170 G00 X 135.0 Z 155.0 T0202
N180 G70 P70 Q130
N190 G00 X 620.0 Z 1064.0 M09 T0200
N200 T0100
N210 M05
N220 M30
```
**G70 FINISHING CYCLE**

It can be seen from the previous example that a separate G70 code is used for calling up the finishing pass after the roughing out has taken place. This is initiated as follows:-

G70  P…. Q  F

Where P is the start block number of the finish part profile definition and Q is the finishing block number. F is the feed rate, if required.

Programming Rules:-

1. Position the finishing tool to the same starting point as for the G71 cycle. The tool returns to this position after the G70 cycle is complete.

2. Observe the following rules from the previous section regarding G71:- Rule 2, Rule 5, Rule 6, Rule 8 and Rule 10.

The G70 Finishing Cycle is also used in the same manner for the G72 and G73 cycles to be described in the following text.

**G72 TRANSVERSE AREA CLEARANCE CYCLE:-**

This cycle is similar to the G71 cycle except that the major cut axis is in the X axis.

The cycle is commanded using one or two G72 blocks and the format is as follows:-

G72  W, R

G72  P, Q, U, W, F, T.

W is the depth of the cut infeed in the Z axis. All other addresses have exactly the same meaning as with the G71 cycle and the same values which are set in parameters apply unless otherwise programmed or set as before.

The same rules apply as with the G71 cycle with the following converse exceptions:-

1. The X axis applies instead of the Z axis in the previous rule 4 relating to the G71 cycle.

For further detail and a programming example consult the FANUC OT Operators Manual.
G71 LONGITUDINAL AREA CLEARANCE ROUTINE (G70 FINISHING CYCLE)

Depth of cut = 3.0
Tool Lift off = 1.0
Finishing allowance = 1.0 on Dia's
                           = 0.2 on faces
Roughing feedrate = 0.25
Finishing feedrate = 0.15

G72 TRANSVERSE AREA CLEARANCE ROUTINE (G70 FINISHING CYCLE)

Depth of cut = 3.0
Tool Lift off = 1.0
Finishing allowance = 0.2 on Dia's
                           = 0.1 on faces
Roughing feedrate = 0.25
Finishing feedrate = 0.15
G73 PATTERN REPEAT CYCLE

This cycle is used for the rough and finish turning of preformed parts such as casting and forgings.
The cycle is commanded using one or two G73 blocks and the format is as follows:-

\[
\begin{align*}
\text{G73} & \quad U, \quad W, \quad R \\
\text{G73} & \quad P, \quad Q, \quad U, \quad W, \quad F, \quad T.
\end{align*}
\]

Where in the first block designated G73:- U is the tool stand off to take into account the material thickness to be removed. The value is a RADIUS value and the amount is the total stock to be removed less the finishing allowance. The U value is the material thickness in the X axis. Normal dimension rules apply.

W is the similar value to allow for the material to be removed in the Z axis. The value is again the total stock removed LESS the finishing allowance. Normal dimension word rules apply.

R is the number of cuts required to remove the material but does not include the finishing pass to remove the stock allowance which is removed using the G70 cycle. R is a whole number.

In the second block designated G73:-

P and Q are the start and end block numbers which respectively define the beginning and end of the finish part profile being roughed out including the rapid infeed command move.

U and W are the stock allowances for the finish cut as in the previous G71 and G72 cycles.

Programming Rules

The same rules generally apply to the G73 cycle as are laid out in the section dealing with the G71 cycle, there are however some noteworthy precautions to observe with this cycle as follows:

1. The G73 cycle is not particularly efficient because it cannot take into account variations in stock thickness which varies considerably on forgings and castings from diameter to diameter. As a consequence in practice it may be that the tool is either cutting air or taking a heavy cut on parts of the profile.

2. It is very advisable to face out all fillet corners and faces before calling up the cycle because the tool moves in the same amount in the Z axis as the X axis for each cut sub-division. In general the geometry of most roughing tools will not accommodate this.
G74 PECK DRILLING AND FACE GROOVING CYCLE:

The use of this cycle enables either peck drilling or face grooving depending upon which addresses are called up in the G74 block(s). The cycle is commanded using one or two G74 blocks and the format is as follows:-

G74 R
G74 X (U).......Z (W).......P, Q, R, F

Where the first block designated G74.

R is the peck lift off, in the case of both drilling and grooving.

This value can be specified as a standard value in parameter or otherwise specify in R value of the first block.

Decimal point and normal dimension word rules apply to the R word. The value is MODAL and the first block does not need to be programmed if the value specified in parameters is used. If no pecking is required programme R equal to zero.

In the second block designated G74:--

X (U) is the final depth in the X axis (absolute or incremental) of the required groove moving in the X negative direction. This only applies to grooving. In the case of drilling DO NOT programme X (U) values. Normal dimension word and decimal point rules apply. In the case of U the value is a DIAMETER so programme U/Z as the width of the groove.

Z (W) is the final depth in the Z axis (absolute or incremental) of the required groove or drilled hole moving in the Z minus direction (generally). Normal dimension word and decimal point rules apply.

P is the tool indeed increment in the X axis and can be used to give either a groove clearing cycle in the case of P being equal or less than the width of the tool or repetitive face grooves in the case of P being greater than the width of the tool.

Decimal point format cannot be used with the P word programme leading zero suppression e.g. for a P value for 5mm programme P5000. P is a radius value.

Q is the infeed depth for each peck in the Z axis. Programme Q the same as P.

N.B. P and Q should not have any signs and are only values as described.

R is the tool lift off at the end of the cut in the case of peck turning or grooving. The lift off is in the X plus direction and the sign of R is always plus. In the case of drilling omit the R
word. The following specific example illustrates a peck drilling cycle.

Cutting Condition:
Drilling Speed - 300 rpm (250 drill at 25m/min)
Cutting speed)
Drill Feed - 0.15mm/rev
Peck Increment - 10mm
Peck Lift Off - 1mm

Programme:
N10 G21
N20 G00 X400.0 Z200.0 T0100 M03
N30  X0.0  25.0  T0101 M08
N40 G74 Z-65.0 Q10000 Fo.15
N60 G00 X400.0 Z200.0 T0100 M09
N70 M30

G75 Grooving in the X Axis

The use of this cycle enables the grooves to be plunged in a diameter and is the basic equivalent to the G74 cycle, used for grooving in the Z axis. The cycle is commanded using one or two G75 blocks and the format is as follows:

\[
G75, R \\
G75 X (U)… Z(W) …P, Q, R, F
\]

Where in the first block designated G75:

R is the peck lift off in the X axis and is similar to the G75 cycle, except that X and Z are transposed. The rules are the same as with the first G74 R word.

In the second block designated G75:-

X (U) is the final depth in the X axis (absolute or incremental) of the required groove moving in the X negative direction as before. The same rules apply as in G74.

Z (W) is the final depth in the Z axis of the required groove or groove series moving in the Z minus direction. Rules apply as before.
P is the peck increment in the X axis. The format for P is the same as in the G74 cycle.

Q is the tool infeed increment in the Z axis and can be used to give either a groove clearing cycle, in the case of Q being equal or less than the width of the tool or repetitive grooves on the diameter in the case of Q being greater than width of the tool.

N.B. Programme P and Q as before in the G74 cycle.

R is the tool lift off at the end of the cut infeed. The lift off is in the Z plus direction and sign R is always plus.

N.B. there will be a small cut width in the last cut owing to the small remainder on the last Z infeed position because the tool width is not equally divisible into the groove width.

**G76 Complex Threadcutting Canned Cycle:**

It can be realised from previous section, regarding simple canned cycles that there are disadvantages with this canned cycle:

1. Compound infeed cannot be readily achieved so the canned cycle is limited to fine pitch threads.

2. A large number of blocks may be generated if a large number of passes are required. Where there is a requirement for any of the above then there is a compound threadcutting canned cycle available.
This is commanded in a G76 block. The format of the G76 block is as follows:

```
G76 P (m, r, a) Q (d min) R (d)
G76 X U) …Z (W) …R (i) P (k) Q (d) F ()
```

Where in the first calling block the word addresses are as follows:-

P (m) is the number of spring passes. This is a factory set at 02 – 2 passes but can be changed in the calling block up to 99 maximum. The format must be 2 digits i.e. 10 etc.

P (a) is the included angle of the tool tip. This is factory set at 60° but can be changed in this calling block. There are six angles available 80°, 60°, 55°, 30°, 20° and zero. The format must be two digits i.e.60 etc.

Q (min) is the minimum cutting depth. Obviously as the infeed depth decreases to a small value it could occur on a large lead thread that an inordinate number of passes are generated so the prevent this a minimum value can be specified in the “Q” address. This value is factory set at 0.05mm (0.002 ins). The format is three digits maximum i.e. 0.05mm is written as 50 and 0.1mm is written as 100.

R (d) is finishing allowance. The infeed progresses down one flank and then the last pass is a plunge infeed to clean up both flanks at the depth set in the R (d) address. This is factory set at 0.25mm (0.001 in) and can be changed in this calling block. THE DECIMAL POINT MUST BE PROGRAMMED IN THE R (d) REGISTER.

N.B. All the addresses in the first calling block are MODAL and do not need to be programmed unless other than the factory set values are required.

In the second calling block the word addresses are as follow:-

X (u) is absolute (or incremental) thread root diameter.

Z (w) is the end point of the threadcutting departure including the pullout distance.

R (i) is programmed when taper threads are cut. The value is the difference in thread radius from the start point of the threading pass to the end point. For example for a 60° included angle taper thread.

\[
R (i) = \tan 30° \times W \text{ (thread length departure), in general}
\]

\[
R (i) = \tan \frac{\theta}{2} \times W \text{ where } \theta = \text{ included angle.}
\]

P (k) is the crest to root height of the thread and is specified as a radius value without a decimal point.
Q (d) is the depth of cut for the 1st pass, stated without a decimal point.

F (c) is the lead of the thread (as previously).

**Programming Rules**

1. The rules for G33 apply to G76

2. Normal dimension format rules apply unless otherwise stated.

3. For internal and external threads the infeed direction is determined by the values of X (u), Z (w). If programming is incremental i.e. u and w be careful to program the correct sign i.e. for minus, external in feed and for u plus internal outfeed.
As an alternative to the Fanuc G73, the programming of I and K words to define the arc centre, these would be programmed instead of the R word as already described.

When using I and K words, the arc may be generated, although for a lathe this is not practical.

I and K words are incremental and are used for direction dependent.

I and K words specify the distance and direction from the start point to the arc centre.

The feedrate 'F' word is not shown in line 40; it is assumed to prevail from the previous block.
DIRECT DRAWING DIMENSION PROGRAMMING

The facility enables simplified programming or angles, 45 degree chamfer, corner rounding and dimensional values that normally appear on drawings to be programmed by directly inputting their values, even if they are inserted between straight lines which have an optional angle.

![Programme example diagram]

There are two main ways of using this facility.

1. The component is described using X (U) Z (W) intersection points together with chamfer (C) and/or radius (R) values only.

2. The component is described using X (U) Z (W) intersection points chamfer (C) radius (R) and optional angle (A) values.

<table>
<thead>
<tr>
<th>METHOD 1</th>
<th>METHOD 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>X (x2) Z (x2), C(C1);</td>
<td>A(a1) C(c1);</td>
</tr>
<tr>
<td>X (x3) Z(Z3), R(r2);</td>
<td>X (x3) Z(z3) A(a2) R(r2)</td>
</tr>
<tr>
<td>X (x4) Z(z4);</td>
<td>X (x4) Z(z4);</td>
</tr>
</tbody>
</table>

It can therefore be see that: Method One, can be used when no angular values are known but intersection co-ordinates are given instead.

Method Two, is ideal when angular values are known but only one co-ordinate is given. Both methods can be used in the same programme.
METHOD 1
G01 Z-30;
X57.66 Z-80. C5;
X120. Z-85.5 R10
X120. Z-100;

METHOD 2
G01 Z-30;
A170. C5;
OR X120. Z-85.5 A100. R10
X120. Z-100

Rules:-

1. When commanding a straight line, only programme one or two values, from “X”, “A” or “Z”.
   i.e. X, A or A, Z or Z

   However, if only one value from X or Z or A is programmed the next block MUST contain all three X, Z, A, values.

   e.g. A (a1) C (c1); only one from X, Z, A

   X (x3) Z9z3) A (a2) R (r2) (must contain all three from X, Z, and A)

   Therefore if more that one value from X, Z or A is known it is wise to include it in order to avoid the need for all three values to be included in the following block as this may necessitate unnecessary calculations.
2. The following G codes are NOT allowed in the same block as a direct input of drawing dimensions command, or between blocks which define sequential figures.
   1) G codes other than G04 in groups 00.
   2) G02, G03, G33, G92 and G94 in group 1.

3. Corner rounding or chamfering cannot be inserted into a threading block.
<table>
<thead>
<tr>
<th>COMMANDS</th>
<th>TOOL MOVEMENT (EXTERNAL)</th>
<th>TOOL MOVEMENT (INTERNAL)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( X_2 ) ( Z_2 ) ( C_1 ); ( X_3 ) ( Z_3 ) ( C_2 ); ( X_4 ) ( Z_4 );</td>
<td>(START POINT) ( (X_2, Z_2) )</td>
<td>(START POINT) ( (X_3, Z_3) )</td>
</tr>
<tr>
<td>or</td>
<td>( (X_4, Z_4) ) ( (X_5, Z_5) ) ( X_6 ) ( Z_6 );</td>
<td>( (X_2, Z_2) ) ( (X_3, Z_3) )</td>
</tr>
<tr>
<td>( A_1 ) ( C_1 ); ( X_3 ) ( Z_3 ) ( A_2 ) ( C_2 ); ( X_4 ) ( Z_4 );</td>
<td>(START POINT) ( (X_3, Z_3) )</td>
<td>(START POINT) ( (X_3, Z_3) )</td>
</tr>
<tr>
<td>or</td>
<td>( (X_4, Z_4) ) ( (X_5, Z_5) ) ( X_6 ) ( Z_6 );</td>
<td>( (X_2, Z_2) ) ( (X_3, Z_3) )</td>
</tr>
<tr>
<td>( A_1 ) ( R_1 ); ( X_3 ) ( Z_3 ) ( A_2 ) ( C_2 ); ( X_4 ) ( Z_4 );</td>
<td>(START POINT) ( (X_3, Z_3) )</td>
<td>(START POINT) ( (X_3, Z_3) )</td>
</tr>
<tr>
<td>or</td>
<td>( (X_4, Z_4) ) ( (X_5, Z_5) ) ( X_6 ) ( Z_6 );</td>
<td>( (X_2, Z_2) ) ( (X_3, Z_3) )</td>
</tr>
<tr>
<td>( X_2 ) ( Z_2 ) ( C_1 ); ( X_3 ) ( Z_3 ) ( R_2 ); ( X_4 ) ( Z_4 );</td>
<td>(START POINT) ( (X_3, Z_3) )</td>
<td>(START POINT) ( (X_3, Z_3) )</td>
</tr>
<tr>
<td>or</td>
<td>( (X_4, Z_4) ) ( (X_5, Z_5) ) ( X_6 ) ( Z_6 );</td>
<td>( (X_2, Z_2) ) ( (X_3, Z_3) )</td>
</tr>
</tbody>
</table>
N001 G01  Z0.0  F.2
N002 X60.0  A90.0  C1.0
N003 Z-30.0  A180.0  R6.0
N004 X100.0  A90.0
N005 A170.0  R20.0
N006 X300.0  Z-180.0  A112.0  R15.0
N007 Z-230.0  A180.0

Detailed Explanation Over Leaf
N001 G01 Z0.0 F.2 Feed from start point of X0. Z5. To X0. Z0

N002 X60.0 A90.0 C1.0 Feed out to 60 dia at 90° to centre line and insert a 1 x 45° chamfer – NOTE: by including the A90.0 value there is no need to state all 3 X, Y or A values in the next block.

N003 Z-30.0 A180.0 R6.0 Feed to Z30. parallel to centre line (A180.0) & insert at the end of that move a radius of 6.0.

N004 X100.0 A90.0 Feed out to 100 dia at 90° to centre line. By incl. A90.0 there is no need for all values in the next block.

N005 A170.0 R20.0 Feed at an angle of 170° to centre line and insert a 20 radius at the end of that move. As only one value from X, Y or A was used the next block MUST contain all 3 values: X, Z & A

N006 X300.0 Z-180.0 A112.0 R15 Feed at an angle of 112° to centre line to intersect with a position of X300 Z-180. and insert at the end of that move a 15. radius

N007 Z-230.0 Z180.0 Feed to Z230 parallel to centre line (A180.0)

NOTE: When a contour has been described by using only one value X, Z or A, the end point for that contour cannot be determined from that block. It is actually determined by using values from the following block. Therefore in single block mode the single block stop is not made in the first block but as the completion of the second block. However the feed hold stop is made at the end of the first block; i.e. the control reads two blocks but the tool only moves one.