ABSTRACT

This workbook is intended for students taking a course in basic computer numerical control (CNC) operation that was developed during a project to retrain defense industry workers at risk of job loss or dislocation because of conversion of the defense industry. The workbook contains daily training guides for each of the course's 13 sessions. Among the topics covered in the course sessions are the following: CNC machine terms, machine specifications, the CNC coordinate grid system, and absolute/incremental programming; toolholder manufacturing processes, toolholding systems, and toolholder styles; functions and keys on CNC operator panels and writing/editing CNC programs; alarm codes and messages; program codes and preset tool methodology; operating CNC machines; using programming codes and locating errors in programs; operating a horizontal machine center; defining and determining machinability factors (speed, feed, and depth of cut); operating a CNC lathe; and troubleshooting CNC machine tools. Each daily training guide includes some/all of the following: session objectives, diagrams and specifications of the machine(s) introduced during the session, information sheets, and learning activities. An assessment and training guide and hands-on assessment for students in the course are also provided. (MN)
Basic CNC Operation

Training Workbook
Section One

Your Notes:
Session One

During this two hour session, participants will:

1. Locate, identify and explain the purpose of the basic components of CNC machine terms.

2. Explain machine specifications.

3. Explain machine home and the coordinate grid system and how it relates to the CNC machine.

4. Explain the four most common axes: X, Y, Z, and B.

5. Define absolute/incremental programming and the use of coordinates and signed numbers.
Axes Travel

Axes Travel

Table (X axis)

Saddle (Y axis)

Head (Z axis)

Additional 'Z' stroke for toolchange

Positioning Speed

Auto (X & Y)

AUTO (Z)

Manual (X, Y, & Z)

Feedrate Range

Minimum increment
<table>
<thead>
<tr>
<th><strong>Table</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Table</td>
<td></td>
</tr>
<tr>
<td>Working Surface</td>
<td></td>
</tr>
<tr>
<td>T Slots</td>
<td></td>
</tr>
<tr>
<td>T Slots Size</td>
<td></td>
</tr>
<tr>
<td>Height above Floor</td>
<td></td>
</tr>
<tr>
<td>Maximum Table Load</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Spindle</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Spindle Drive</td>
<td></td>
</tr>
<tr>
<td>Spindle Speed Range</td>
<td></td>
</tr>
<tr>
<td>Spindle Speed Control</td>
<td></td>
</tr>
<tr>
<td>Spindle Diameter</td>
<td></td>
</tr>
<tr>
<td>Spindle Taper</td>
<td></td>
</tr>
<tr>
<td>Machine Specification</td>
<td></td>
</tr>
<tr>
<td>------------------------</td>
<td></td>
</tr>
<tr>
<td><strong>Spindle</strong></td>
<td></td>
</tr>
<tr>
<td>Tool Holder Clamping</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool Holder</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle Speed Range</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Minimum Distance Spindle Nose to Table Top</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum Distance Spindle Nose to Table Top</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Automatic Tool Changer</strong></td>
<td></td>
</tr>
<tr>
<td>Magazine Capacity</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool Selection</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum Tool Diameter</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum Tool Weight</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum Total Weight of all Tools</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>Maximum Tool Length</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

ERI C
# Machine Specification

**Automatic Tool Changer**

Tool Change Time

Tool Change Time Chip to Chip

**Standard Equipment**

Spindle Orientation

Flood Coolant

Automatic Centralized Lubrication System

Full Machine Guarding

Slideway Protection

Swarf Collection Trays

Low Voltage Worklight
## Machine Specification

### Accuracy

<table>
<thead>
<tr>
<th>Positioning Accuracy</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Repeatability</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

### Mains Services Requirements

<table>
<thead>
<tr>
<th>Electrical Supply</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Standard Electrical Supply</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Low Volt Option Electrical Supply</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Compressed Air</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
STANDARD SPECIFICATION

a) System Features

- 4 axis control with automatic acceleration and deceleration.
- 3 axis simultaneous linear interpolation (XYZ)
- Multi Quadrant circular interpolation (XY, YZ, XZ)

Input range

- 0.001 to 9999.99mm (XYZ)
- or 0.0001 to 999.9999 inch (XYZ)

Part program storage length 120m

Registered programs 63

T-nal offset memory 99

ISO/EIA Automatic recognition

9" CRT Mono chrome screen with MDI keyboard

Feedrate command Direct programming of mm/min or inch/min

Feedrate override 0 to 150%

Rapid traverse override 0.25, 50, 100%

Spindle speed override 50% to 120% in 10% steps

Override cancel

Automatic coordinate system setting

Absolute/incremental command

Program number display/search 4 digits

Sequence number display/search 4 digits

Main program/subprogram (Subprogram: 2 levels)

Optional block skip

Stored pitch error compensation

Backlash compensation

Servo off

Cycle start/feed hold

Buffer register

Reset

Manual pulse generator multiplier X1, X10, X100

Machine lock (all axes)

Dry run

Single block

Part program storage and editing

Data protect switch

Self diagnostic function

Emergency stop

Stored stroke check 1

Interlock error display

Status output: CNC ready, servo ready, alarm, distribution end, automatic operation

Automatic operation start-lamp, feed hold

Digital Servo Control

b) Programming Features

- Pocket calculator type decimal point input
- Background edit
- Inch/metric operation (G20/G21)
- Exact stop/dwell G04/G09/G61
- Reference point return Manual automatic (G27, G28, G29)
- Second reference point return G30
- Coordinate system setting G92
- Feedrate F5 digit direct command
- Spindle speed S4/S5 digit command
Tool selection T4 digits
Miscellaneous function M3 digits
Skip function G31
Tool length compensation G43, G44, G49
Programme input/program end M00/M01/M02/M30
Canned cycles for drilling boring G73, G74, G76, G80 to G89
Cutter compensation C (G40-G59)
Mirror image (M21 to M24)
Work co-ordinate system (G54-G59)
Programme input of offset data (G10)
Absolute/incremental (G90/G91)
<table>
<thead>
<tr>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RNCS</td>
<td>NC Rotary Tables</td>
</tr>
<tr>
<td>1</td>
<td>Multi-Spindle</td>
</tr>
<tr>
<td>2</td>
<td>TRNC.B</td>
</tr>
<tr>
<td>3</td>
<td>RNC</td>
</tr>
<tr>
<td>4</td>
<td>RNC</td>
</tr>
<tr>
<td>5</td>
<td>RNC</td>
</tr>
<tr>
<td>6</td>
<td>RNC</td>
</tr>
<tr>
<td>7</td>
<td>RNC</td>
</tr>
<tr>
<td>8</td>
<td>RNC</td>
</tr>
<tr>
<td>9</td>
<td>RNC</td>
</tr>
<tr>
<td>10</td>
<td>RNC</td>
</tr>
<tr>
<td>11</td>
<td>RNC</td>
</tr>
<tr>
<td>12</td>
<td>RNC</td>
</tr>
<tr>
<td>13</td>
<td>RNC</td>
</tr>
<tr>
<td>14</td>
<td>RNC</td>
</tr>
<tr>
<td>15</td>
<td>RNC</td>
</tr>
</tbody>
</table>

**Others**
- Built-in and Special Rotary Tables
- Auto connector system
- Automatic dividing heads
- Air driven with controller
- Multi-spindle
- NC Dividing Heads
- NC Tilting Rotary Tables
- NC Rotary Tables

**Numerical Control**
- Auto Connector System (for NC Rotary Tables)
- Built-in and Special Rotary Tables
- TPC2
- TPC-Jr
- DNC-N 135-200
- RNCV-N 160-400
- RNCK 250-1250
- RNCB 250-1250
- RNCM 250-650
- RNCV 160-1500
- RNC 800-2000
- RNCX 125-320

**Numerical Control**
- Model 225

*(Many deliveries have been made. Consult a Tsudakoma sales person.)*
<table>
<thead>
<tr>
<th>Model</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RNCS</td>
<td>Super High Speed, 500r.p.m. NC Rotary Table with High Speed continuous rotation and various indexing function</td>
</tr>
<tr>
<td>RNCK</td>
<td>For horizontal machining centers</td>
</tr>
<tr>
<td>RNCX</td>
<td>Standard type only, Low price, Rear motor mounted type (251-301)</td>
</tr>
<tr>
<td>RNCBK</td>
<td>For horizontal machining centers, Table center has a large thru-hole</td>
</tr>
<tr>
<td>RNC</td>
<td>Horizontal positioning only, Machines large and heavy workpieces</td>
</tr>
<tr>
<td>TRNC</td>
<td>Unmovable motor location, Wide inclination</td>
</tr>
<tr>
<td>TTNC</td>
<td>Rotation and tilting are both numerically controlled, Complicated polyhedron machinings are possible</td>
</tr>
<tr>
<td>RNCV</td>
<td>Standard type, Easy application for multi-rotary tables</td>
</tr>
<tr>
<td>THNC</td>
<td>NC tilting rotary table with manual tilt, Low price / high quality</td>
</tr>
<tr>
<td>RNCM</td>
<td>Compact design without splash guard interference</td>
</tr>
<tr>
<td>RDH</td>
<td>Compact, Heavy-duty clamping, Large allowable workpiece inertia, High precision, Easy cable connection</td>
</tr>
<tr>
<td>RNCB</td>
<td>Table center has a large thru-hole, Machines long workpieces Applications: Perfect for insisting of an automatic power chuck, pull stud device, rotary joint device and others</td>
</tr>
<tr>
<td>RD</td>
<td>Low price, Quick delivery, High speed, High accuracy cross roller bearing and coupling attached</td>
</tr>
</tbody>
</table>

**Workpiece Examples**

<table>
<thead>
<tr>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turret</td>
<td>Cylinder</td>
<td>Serration</td>
<td>Pallet</td>
<td>Index plate</td>
</tr>
<tr>
<td>6</td>
<td>7</td>
<td>8</td>
<td>9</td>
<td>10</td>
</tr>
<tr>
<td>Index plate</td>
<td>Nozzle</td>
<td>Bevel gear</td>
<td>Barrel cam</td>
<td>Gear</td>
</tr>
<tr>
<td>11</td>
<td>12</td>
<td>13</td>
<td>14</td>
<td>15</td>
</tr>
<tr>
<td>Lift cam</td>
<td>Barrel cam</td>
<td>Cam</td>
<td>Cam</td>
<td>Lead cam</td>
</tr>
<tr>
<td>16</td>
<td>17</td>
<td>18</td>
<td>19</td>
<td>20</td>
</tr>
</tbody>
</table>
The Bridgeport machine has three axes that are under numerical control. Each axis can move independently, the direction being designated as a plus or minus movement.

The nomenclature shown assumes viewing from the Operator's Control Panel.
When we programme a numerical controlled machine, we always consider that the cutting tool is moving.
In Absolute (G90) all dimensioning is taken from a fixed point which is specified by the programmer (see Figure 5).

Figure 5 Absolute Co-ordinate Programming
Incremental (G91) Co-ordinate Programming

Diagram showing coordinates and movements: 15.00, 25.00, 35.00, 50.00, 25.00, 15.00, and 30.00.
Absolute and Incremental Programming

EXAMPLE 1

<p>| | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>8</td>
<td>7</td>
<td>6</td>
<td>5</td>
<td>4</td>
<td>3</td>
<td>2</td>
<td>1</td>
</tr>
</tbody>
</table>

**ABSOLUTE**

- 6 to 1 X
- 5 to 3 X
- 8 to 4 X
- 2 to 5 X
- 4 to 6 X
- 12 to 7 X
- 10 to 8 X
- 4 to 9 X

**INCREMENTAL**

- 6 to 1 X
- 5 to 2 X
- 8 to 3 X
- 2 to 4 X
- 4 to 5 X
- 5 to 6 X
- 12 to 7 X
- 10 to 8 X
- 4 to 9 X

**EXERCISE:** Find the absolute and incremental co-ordinates of the points listed above.

**ABSOLUTE SYSTEM:** A numerical control system in which all co-ordinate locations are programmed from a fixed or absolute zero point (origin).

**INCREMENTAL:** A numerical control system in which each co-ordinate location is given in terms of distance and direction along rectangular axes from the previous point.
Linear Interpolation (G00/G01)

The axes of the machine will move in linear at either Rapid or Feedrate traverse rates. Any movement preceded by:

a) G00 will occur at RAPID TRAVERSE
b) G01 will occur at FEEDRATE

NOTE: These commands are MODAL and will stay in effect until changed.

Maximum of 3 axes can be programmed in one BLOCK.

Figure 9 Linear Interpolation
Circular Interpolation (G02/G03)

Circular interpolation can be performed in any of three planes (G17, G18, G19).

There are two directions in which arcs are produced - G02 Clockwise and G03 Counter Clockwise.

The information required to move in an arc involves four 'word' addresses. Assuming G17 plane is used, the program would be N14 G02 X100.0 Y0.0 10.0 J-100.0

i.e.,
- G02 - Clockwise movement
- X100.0 - Tool finishing position in X
- Y0.0 - Tool finishing position in Y
- 10.0 - Arc offset in X axis
- J-100.0 - Arc offset in Y axis

Figure 10 Circular Interpolation

Clockwise Direction

Again using G17, the program for counter clockwise movement would be:
N15 G03 X0.0 Y100.0 J-100.0 10.0

Counter Clockwise Direction
The Interact can perform circular interpolation in 3 planes. Either:

a) X and Y (G17) see figure 11
b) X and Z (G18) see figure 12
c) Y and Z (G19) see figure 13

NOTE: Plane selection is MODAL and once selected will stay in effect until another plane is selected.

Figure 11 G17 XY Plane (Plan view above spindle)

Figure 12 G18 ZX Plane (View from behind spindle)

Figure 13 G19 YZ Plane (View from table end)
Arc Offsets

Arc offsets are addressed using the 'Words'.

I - is the sign distance parallel along the X axis from the starting position of the arc to the arc centre.

J - is the sign distance parallel along the Y axis from the starting position of the arc to the arc centre.

K - is the sign distance parallel along the Z axis from the starting position of the arc to the arc centre.

The 'word' values describe the direction in which the centre of arc lies in relation to the starting position. Values are automatically assumed to be +(plus). If a -(minus) value is required it MUST be designated. See Figure 14.

Figure 14 Arc Offsets

![Diagram of arc offsets]

Explanation:
In the X axis, the circle is in the X negative direction.
In the Y axis, the circle is in the Y negative direction.

Example:
Block of program required to move from A to B:
N15 G02 XU.0 Y-100.0 I-57.357 J-81.915
This command is used as an alternative to replace the I and J words used in circular interpolation.

Example:  
N15 G02 X0.0 Y-100.0 R100.0

NOTE:  
The +R word can only be used for arcs up to and include 180°.  
The -R word can only be used for arcs greater than 180°, and less than 360°.

Figure 15 Radius Command

To move from Position 1 to Position 2 (clockwise):  
N100 G02 X57.357 Y81.915 R100.0

To move from Position 1 to Position 2 (counter clockwise):  
N105 G03 X57.357 Y81.915 R-100.0
The machine has its own "Machine Reference co-ordinate System", from which the maximum travels of the X, Y and Z axes are measured.

Figure 16. Spindle positions at Machine Reference in X, Y and Z axes.
Z Axis G28 & G30 Reference Planes

There are 2 reference planes in the Z axis, to which the spindle will return automatically:

1. G28 plane which is at the Machine reference location.
2. G30 plane is where all toolchanges occur.

Program Format for G28 and G30 reference return planes.

The G28 command constitutes 2 movements:

a) Move to dimension
b) Move to reference
   example N50 G28 G91 Z0

The first movement to take place is an incremental move of Z0.

The second movement will return directly to the reference plane.

Care should be taken when programming this move in the absolute mode, as the Z dimension must be clear of the workpiece.

   N50 G28 G90 Z150.0

The G30 command works in the same way as the G28, but is only to be used for toolchange purposes.

   i.e. N60 G30 G91 Z0 T4 M6

After movement to the G30 reference plane, the machine will initiate a toolchange cycle, selecting tool number 4.
Work Co-ordinate System Programming (G54-G59)

The work co-ordinate system allows for the setting of datums relative to the machine reference coordinate system.

X and Y axis values for use with G54-G59 co-ordinate system.

When the position of the component datum has been determined, it can be entered into the relative work offset register.

This can be done in 2 different ways:

a) Manually, whereby the dimensions can be entered directly through the keyboard.

b) Programmed, whereby the dimensions can be entered into the program in the following format:

```
N10 G10 G90 L2 PO1 X-275.0 Y-300.0
```

Explanation:

- G10 L2 => Work co-ordinate offset input
- PO1 => Specifies offset register G54
- PO2 => G55 - G59
To determine the Z axis value, it is necessary to use a tool of known overall length (Gauge line - to tip). The distance travelled in the positional data display added onto the tool length, becomes the Z axis value.

This can also be entered manually into the work offset registers or within the program, e.g.

```
N10 G10 G90 L2 P01 X-275.0 Y-300.0 Z-250.0
```

Once the datum positions have been recorded, it is necessary to measure the overall tool lengths which are to be used as offset values.
The tool to be measured is placed in the measuring system, and the tool height can be measured using the micrometer.

NOTE: This is a mechanical measuring system.
With the Z axis positioned at the G28 Z axis reference position, we can determine the tool offset by using the formula:

\[ H = A - B \]

Where:
- \( H \) is the offset value
- \( A \) is the distance from spindle gauge line to component datum
- \( B \) is the distance between the tool tip and component datum

Example:

\[ H = 250 - 161.0 \]
\[ H = 89.0 \]

NOTE: Each offset value found must be positive and entered into the appropriate offset number.
Your Notes:
Session Two

During this two hour session, participants will:

1. Discuss toolholder manufacturing practices.
2. Examine NC toolholding systems.
3. Overview styles of toolholders.
4. Discuss styles of collets, tool adapters, and extensions.
5. Identify toolholders and various items.
A. MATERIAL - 8620 ALLOY STEEL

B. HEAT TREAT - CARBURIZE AND HARDENED (Rc 58 EXTERIOR )

(Rc 35 INNER CORE )

C. FINISH - BLACK OXIDE (NO BUILDUP ON OUTER SURFACE )

D. BLACK OXIDE - HELPS RELEASE TOOLHOLDER FROM THE SPINDLE
NC Toolholding Systems

1. TOOL HOLDER TYPES

A. V - FLANGE TOOLING (ALSO KNOWN AS CAT)

1. V - 30
2. V - 40 (POPULAR)
3. V - 45
4. V - 50 (POPULAR)

![Diagram of V-Flange Tooling]

**ENGINEERING DATA**

<table>
<thead>
<tr>
<th>Taper</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
<th>T</th>
</tr>
</thead>
<tbody>
<tr>
<td>V-30</td>
<td>1.87</td>
<td>1.25</td>
<td>1.25</td>
<td>1.81</td>
<td>(\frac{3}{8}) 13 x 100</td>
</tr>
<tr>
<td>V-40</td>
<td>2.69</td>
<td>1.75</td>
<td>1.75</td>
<td>2.50</td>
<td>(\frac{3}{8}) 11 x 12</td>
</tr>
<tr>
<td>V-45</td>
<td>3.25</td>
<td>2.25</td>
<td>2.25</td>
<td>3.25</td>
<td>(\frac{3}{8}) 10 x 150</td>
</tr>
<tr>
<td>V-50</td>
<td>4.00</td>
<td>2.75</td>
<td>2.75</td>
<td>3.88</td>
<td>1&quot; 8 x 175</td>
</tr>
</tbody>
</table>

B. BT TOOLING (JAPAN ORIGIN)

1. BT - 35
2. BT - 40 (POPULAR)
3. BT - 45
4. BT - 50 (POPULAR)

![Diagram of BT Tooling]

**ENGINEERING DATA**

<table>
<thead>
<tr>
<th>Taper</th>
<th>D</th>
<th>E</th>
<th>M</th>
<th>T</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>BT35</td>
<td>2.15</td>
<td>1.50</td>
<td>2.08</td>
<td>M12 x 1.75mm</td>
<td>67</td>
</tr>
<tr>
<td>BT40</td>
<td>2.57</td>
<td>1.75</td>
<td>2.48</td>
<td>M16 x 2.00mm</td>
<td>106</td>
</tr>
<tr>
<td>BT45</td>
<td>3.25</td>
<td>2.25</td>
<td>3.34</td>
<td>M20 x 2.50mm</td>
<td>130</td>
</tr>
<tr>
<td>BT50</td>
<td>4.00</td>
<td>2.75</td>
<td>3.94</td>
<td>M24 x 3.00mm</td>
<td>150</td>
</tr>
</tbody>
</table>
Styles of Toolholders

A. SHELL MILL ADAPTER - SHELL MILL

B. END MILL ADAPTER - END MILL, BORING CHUCK - BAR
Styles of Toolholders

C. MORSE TAPER AND JACOBS TAPER

D. E - COLLET ADAPTER - E - COLLETS
Styles of Toolholders

E. FLEX COLLET CHUCKS - GRIP & FLEX COLLETS

F. BORING RING ADAPTER
Styles of Toolholders

G. LENGTH COMPENSATING TAP CHUCK - TAP NUT / COLLET REQUIRED

[ .88 TENSION (EXPAND) & .38 COMPRESSION ]

H. TAP COMPENSATING TAP ADAPTER - USE UNIVERSAL TAP ADAPTER

[ .88 TENSION (EXPAND) & .38 COMPRESSION ]

[ SIZE 1  0 - 9/19 TAP & 1/8 PIPE ]
[ SIZE 2  5/16 - 7/8 TAP & 1/4 , 1/2 PIPE ]
[ SIZE 3  13/16 - 1-3/8 TAP & 3/4 , 1" PIPE ]
### TOOLHOLDER IDENTIFICATION SYSTEM

**Shank Style**
- **C**: V-Flange
- **B**: BT Flange
- **M**: MILACRON

**Taper**
- **2**: 30 Taper
- **3**: 35 Taper
- **4**: 40 Taper
- **5**: 45 Taper
- **6**: 50 Taper

**Holder**
- **A**: AUTOMATIVE
- **C**: COLLET
- **E**: END MILL
- **F**: STUB ARBOR
- **H**: BORING HEAD
- **J**: JACOBS TAPER
- **M**: MORSE TAPER
- **P**: RIGID TAP HOLDER
- **S**: SHELL MILL
- **T**: TAP
- **W**: TEST BAR
- **X**: BLANK BAR

**Length**
- **1**: 100

**Size (Dec)**
- **0**: 0
A. "E" COLLET (SINGLE ANGLE)

A. FITS ERICKSON STYLE COLLET CHUCKS

B. EACH SIZE HAS A GRIPPING RANGE OF 1/64"

C. SERIES AVAILABLE 075, 100, AND 150
Four different series to fit many needs
Different nose diameters to solve clearance problems
Collets collapse 1/64"
Black oxide finish to protect precision ground surfaces
The last four digits of our collet part numbers are the ID size

NOTE: For best grip and precision we recommend the Micro Precision™ Collet series as found on preceding pages

**DOUBLE ANGLE COLLETS**

<table>
<thead>
<tr>
<th>PART NO.</th>
<th>RANGE</th>
<th>COLLET</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>E</th>
</tr>
</thead>
<tbody>
<tr>
<td>DA30-0046</td>
<td>DA20-0062</td>
<td>DA10-0125</td>
<td>DA10-0159</td>
<td>DA10-0375</td>
<td>DA18-0250</td>
<td>DA18-0531</td>
</tr>
<tr>
<td>DA30-0062</td>
<td>DA20-0078</td>
<td>DA10-0140</td>
<td>DA10-0230</td>
<td>DA10-0430</td>
<td>DA18-0255</td>
<td>DA18-0546</td>
</tr>
<tr>
<td>DA30-0078</td>
<td>DA20-0093</td>
<td>DA10-0156</td>
<td>DA10-0406</td>
<td>DA18-0281</td>
<td>DA18-0565</td>
<td></td>
</tr>
<tr>
<td>DA30-0093</td>
<td>DA20-0109</td>
<td>DA10-0171</td>
<td>DA10-0421</td>
<td>DA18-0296</td>
<td>DA18-0576</td>
<td></td>
</tr>
<tr>
<td>DA30-0109</td>
<td>DA20-0125</td>
<td>DA10-0187</td>
<td>DA10-0437</td>
<td>DA18-0312</td>
<td>DA18-0593</td>
<td></td>
</tr>
<tr>
<td>DA30-0125</td>
<td>DA20-0140</td>
<td>DA10-0203</td>
<td>DA10-0452</td>
<td>DA18-0326</td>
<td>DA18-0599</td>
<td></td>
</tr>
<tr>
<td>DA30-0140</td>
<td>DA20-0156</td>
<td>DA10-0218</td>
<td>DA10-0468</td>
<td>DA18-0343</td>
<td>DA18-0625</td>
<td></td>
</tr>
<tr>
<td>DA30-0156</td>
<td>DA20-0171</td>
<td>DA10-0234</td>
<td>DA10-0484</td>
<td>DA18-0359</td>
<td>DA18-0640</td>
<td></td>
</tr>
<tr>
<td>DA30-0171</td>
<td>DA20-0187</td>
<td>DA10-0250</td>
<td>DA10-0500</td>
<td>DA18-0375</td>
<td>DA18-0655</td>
<td></td>
</tr>
<tr>
<td>DA30-0187</td>
<td>DA20-0203</td>
<td>DA10-0265</td>
<td>DA10-0515</td>
<td>DA18-0390</td>
<td>DA18-0671</td>
<td></td>
</tr>
<tr>
<td>DA30-0203</td>
<td>DA20-0218</td>
<td>DA10-0281</td>
<td>DA10-0531</td>
<td>DA18-0406</td>
<td>DA18-0687</td>
<td></td>
</tr>
<tr>
<td>DA30-0218</td>
<td>DA20-0234</td>
<td>DA10-0296</td>
<td>DA10-0546</td>
<td>DA18-0421</td>
<td>DA18-0703</td>
<td></td>
</tr>
<tr>
<td>DA30-0234</td>
<td>DA20-0250</td>
<td>DA10-0312</td>
<td>DA10-0562</td>
<td>DA18-0437</td>
<td>DA18-0718</td>
<td></td>
</tr>
</tbody>
</table>

**REPLACEMENT NUT**

<table>
<thead>
<tr>
<th>PART NO.</th>
<th>D</th>
<th>E</th>
</tr>
</thead>
<tbody>
<tr>
<td>DACN-0030</td>
<td>5/6</td>
<td>.56</td>
</tr>
<tr>
<td>DACN-0020</td>
<td>3/4</td>
<td>.83</td>
</tr>
<tr>
<td>DACN-0010</td>
<td>7/16</td>
<td>1.06</td>
</tr>
<tr>
<td>DACN-0018</td>
<td>7/4</td>
<td>1.44</td>
</tr>
</tbody>
</table>
A. Fits Erickson style collet chucks
B. Each size has a gripping range of \(1/32\)"
C. Designed for holding small dia. tools
D. Not as good as single angle – seating problem

**Series Available**

- **300 Series**: .013 – .25
- **200 Series**: .125 – .375
- **100 Series**: .25 – .5625
A. UNIVERSAL TAP ADAPTER
   A. POSITIVE DRIVE STYLE
   B. TORQUE CONTROL STYLE
      1. BEST USED FOR BLIND HOLES
      2. IF TORQUE SET TOO LIGHT, NOT ENOUGH THREADS
      3. TORQUE SETTINGS TRIAL AND ERROR

B. COLLET / NUT POSITIVE TAP DRIVE
   A. USE FOR RIGID TAPPING

B. TAP EXTENSION
   1. POSITIVE TAP DRIVE
   2. SEVERAL SIZES AVAILABLE
Floating Reamer Holders

A. ALLOWS REAMER TO ALIGN TO PREVIOUSLY DRILLED HOLE
B. MOST ALLOW .06 TOTAL RADIAL FLOAT
C. TOOLHOLDER AND COLLET TO MATCH IN SIZE CAPACITY
6. TOOLHOLDER RETENTION STUD (RETENTION KNOB)

A. TOOLHOLDER RETENTION STUDS ARE NOT NECESSARILY INTERCHANGEABLE ACROSS VARIOUS TYPES OF TOOLHOLDERS OR MACHINE TOOL MODELS.

**Japanese Style Retention Knobs**

<table>
<thead>
<tr>
<th>Taper Part No.</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 RB2M-0001</td>
<td>.43</td>
<td>.27</td>
<td>.90</td>
<td>45°</td>
</tr>
<tr>
<td>RB2M-0002</td>
<td></td>
<td></td>
<td></td>
<td>60°</td>
</tr>
<tr>
<td>35 RB3M-0001</td>
<td>.51</td>
<td>.33</td>
<td>1.10</td>
<td>45°</td>
</tr>
<tr>
<td>RB3M-0100</td>
<td>.54</td>
<td>.35</td>
<td>1.10</td>
<td>60°</td>
</tr>
<tr>
<td>RB3M-0101</td>
<td>.55</td>
<td>.31</td>
<td>.91</td>
<td>90°</td>
</tr>
<tr>
<td>40 RB4M-0001</td>
<td>.59</td>
<td>.39</td>
<td>1.38</td>
<td>45°</td>
</tr>
<tr>
<td>RB4M-0002</td>
<td></td>
<td></td>
<td></td>
<td>60°</td>
</tr>
<tr>
<td>RB4M-0003</td>
<td></td>
<td></td>
<td></td>
<td>90°</td>
</tr>
<tr>
<td>45 RB5M-0001</td>
<td>.75</td>
<td>.55</td>
<td>1.57</td>
<td>45°</td>
</tr>
<tr>
<td>RB5M-0002</td>
<td></td>
<td></td>
<td></td>
<td>60°</td>
</tr>
<tr>
<td>RB5M-0003</td>
<td></td>
<td></td>
<td></td>
<td>90°</td>
</tr>
<tr>
<td>50 RB6M-0001</td>
<td>.91</td>
<td>.67</td>
<td>1.78</td>
<td>45°</td>
</tr>
<tr>
<td>RB6M-0002</td>
<td></td>
<td></td>
<td></td>
<td>60°</td>
</tr>
<tr>
<td>RB6M-0003</td>
<td></td>
<td></td>
<td></td>
<td>90°</td>
</tr>
</tbody>
</table>

**Other Retention Knobs for Domestic and Imported Machines**

<table>
<thead>
<tr>
<th>Taper Part No.</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 RB2M-0100</td>
<td>.47</td>
<td>.31</td>
<td>.94</td>
<td>R°</td>
</tr>
<tr>
<td>50 RS6M-0114-C</td>
<td>1.10</td>
<td>.83</td>
<td>1.34</td>
<td>75°</td>
</tr>
<tr>
<td>40 RB4E-0100</td>
<td>.59</td>
<td>.39</td>
<td>1.05</td>
<td>90°</td>
</tr>
<tr>
<td>RB4E-0101</td>
<td>.59</td>
<td>.39</td>
<td>.98</td>
<td>90°</td>
</tr>
<tr>
<td>RS4E-0104</td>
<td>.59</td>
<td>.41</td>
<td>1.44</td>
<td>45°</td>
</tr>
<tr>
<td>RS4E-0106</td>
<td>.59</td>
<td>.39</td>
<td>1.38</td>
<td>45°</td>
</tr>
<tr>
<td>RS4E-0109</td>
<td>.52</td>
<td>.39</td>
<td>.47</td>
<td>50°</td>
</tr>
<tr>
<td>RS4E-0110</td>
<td>.93</td>
<td>.83</td>
<td>.39</td>
<td>75°</td>
</tr>
<tr>
<td>RS4E-0111</td>
<td>.50</td>
<td>.31</td>
<td>.75</td>
<td>75°</td>
</tr>
<tr>
<td>RS4E-0114-C</td>
<td>.75</td>
<td>.55</td>
<td>1.02</td>
<td>75°</td>
</tr>
<tr>
<td>50 RB6E-0100**</td>
<td>.91</td>
<td>.67</td>
<td>1.78</td>
<td>90°</td>
</tr>
<tr>
<td>RS6E-0101</td>
<td>.94</td>
<td>.71</td>
<td>1.23</td>
<td>90°</td>
</tr>
<tr>
<td>RS6E-0101-C</td>
<td>.94</td>
<td>.71</td>
<td>1.23</td>
<td>90°</td>
</tr>
<tr>
<td>RS6E-0114-C</td>
<td>1.10</td>
<td>.83</td>
<td>1.34</td>
<td>75°</td>
</tr>
</tbody>
</table>

* R designates radius. ** Wrench flats are on knob end. C at end of Part No. designates coolant hole.
# Flanges

## V-Flange

### U.S. ANSI for machines WITH Thru-The-Spindle Coolant

<table>
<thead>
<tr>
<th>TAPER</th>
<th>PART NO.</th>
<th>A</th>
<th>B</th>
<th>Thread Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>30</td>
<td>RC2E-0001</td>
<td>.52</td>
<td>.76</td>
<td>1/2-13</td>
</tr>
<tr>
<td>40</td>
<td>RC4E-0001</td>
<td>.74</td>
<td>.64</td>
<td>5/8-11</td>
</tr>
<tr>
<td>45</td>
<td>RC5E-0001</td>
<td>.94</td>
<td>.82</td>
<td>3/4-10</td>
</tr>
<tr>
<td>50</td>
<td>RC6E-0001</td>
<td>1.14</td>
<td>1.00</td>
<td>1-8</td>
</tr>
</tbody>
</table>

### U.S. ANSI for machines WITHOUT Thru-The-Spindle Coolant

<table>
<thead>
<tr>
<th>TAPER</th>
<th>PART NO.</th>
<th>A</th>
<th>B</th>
<th>Thread Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>RC4E-0002</td>
<td>.74</td>
<td>.64</td>
<td>5/8-11</td>
</tr>
<tr>
<td>45</td>
<td>RC5E-0002</td>
<td>.94</td>
<td>.82</td>
<td>3/4-10</td>
</tr>
<tr>
<td>50</td>
<td>RC6E-0002</td>
<td>1.14</td>
<td>1.00</td>
<td>1-8</td>
</tr>
</tbody>
</table>

## BT-Flange

### MODIFIED U.S. ANSI with metric thread and coolant hole

<table>
<thead>
<tr>
<th>TAPER</th>
<th>PART NO.</th>
<th>A</th>
<th>B</th>
<th>Thread Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>RS4M-0105</td>
<td>.74</td>
<td>.64</td>
<td>M16-2</td>
</tr>
<tr>
<td></td>
<td>RS4M-0115</td>
<td>.74</td>
<td>.75</td>
<td>M16-2</td>
</tr>
<tr>
<td>45</td>
<td>RS5M-0106</td>
<td>.94</td>
<td>.82</td>
<td>M20-2.5</td>
</tr>
<tr>
<td>50</td>
<td>RS6M-0105</td>
<td>1.14</td>
<td>1.00</td>
<td>M24-3.0</td>
</tr>
</tbody>
</table>
**Adapter Locking Fixture**

A. Holds and protects the adapter shank when changing tools in collet or shell mill adapters.

B. V - 30 and BT - 35 2.5 inch hex shank on fixture
C. V - 40 and BT - 40 3.0 inch hex shank on fixture
D. V - 45 and BT - 45 3.5 inch hex shank on fixture
E. V - 50 and BT - 50 4.0 inch hex shank on fixture

8. Pre-setting height gage

A. Allows for setting of tool lengths off of machine.
Identify the Toolholders & Items

A

B

C

D

E

F

G

H

I
DESCRIPTION: CT-50, .750 I.D. ADAPTOR SHANK
ASSEMBLED WITH XRTH-0001

<table>
<thead>
<tr>
<th>DATE</th>
<th>SCALE</th>
<th>DRAWN</th>
<th>SHEET</th>
<th>PRT NO: C6P2-0001</th>
</tr>
</thead>
<tbody>
<tr>
<td>5-27-93</td>
<td>.75</td>
<td>B.H.</td>
<td>1 OF 1</td>
<td>DWG NO: 122700-1</td>
</tr>
</tbody>
</table>

**Diagram:**
- 7/24 TAPER
- CT-50 Y-FLANGE
- 1/2-20 THD
- GAGE LINE
- 2.75 DIA
- 3.58
- 1.38
- 1.50 DIA
DESCRIPTION: BT-40 RETENTION KNOB
WITH 3/16 COOLANT HOLE

<table>
<thead>
<tr>
<th>DATE</th>
<th>SCALE</th>
<th>DRAWN</th>
<th>SHEET</th>
<th>PRT NO: RB4E-0001-C</th>
</tr>
</thead>
<tbody>
<tr>
<td>7-1-93</td>
<td>1.25</td>
<td>B.H.</td>
<td>1 OF 1</td>
<td>DWG NO: 6152-A</td>
</tr>
</tbody>
</table>

- **Φ .903**
- **.392**
- **5/8-11 THD**
- **3/16 THRU**
- **.124**
- **.157**
- **.589**
- **.988**
- **1.264**
- **2.124**
DESCRIPTION: BT-40, EXTENDED LENGTH
1.500 SHELL MILL HOLDER

DATE: 8-29-92
SCALE: .75
DRAWN: B.H.
SHEET: 1 OF 1
PRT NO: B4S5-1500
DWG NO: 242140-A

COMPONENT PARTS: XSDK-1500 - DRIVE KEY, 2 REQUIRED
XSAS-1500 - ARBOR SCREW, 3/4-16 THD
DESCRIPTION: CT-40 WITH BT-40 BACK HOLE & THD DR32 COLLET CHUCK
NOSE LENGTH = 3.50 ASSEMBLED

ASSEMBLY DRAWING
REFERENCE ONLY

DATE: 12-15-93
SCALE: .75
DRAWN: B.H.
SHEET: 1 OF 1
PRT NO: SPECIAL
DWG NO: 148022-A

PART NO: RSRA-0114-C
RETENTION KNOB, METRIC THREADS

PART NO: 148022
7/24 TAPER
CT-40 V-FLANGE
BALANCE WITHIN 1 gmm

M22-1.5 L.H. THD
3.38 REFERENCE TO STOP SCREW

DR32 SERIES
MICRO PRECISION COLLETS
RANGE .078 - .787

STOP SCREWS
PART NO: XCCS-2215
XCCS-2215-C (COOLANT, .12 THRU HOLE AND URETHANE WASHER INCLUDED)

TOLERANCES U.O.S.

<table>
<thead>
<tr>
<th>TOLERANCE</th>
<th>U.O.S.</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>± .030</td>
</tr>
<tr>
<td>XX</td>
<td>± .020</td>
</tr>
<tr>
<td>XXX</td>
<td>± .010</td>
</tr>
<tr>
<td>XXXX</td>
<td>± .005</td>
</tr>
<tr>
<td>ANGLES</td>
<td>± 1.0°</td>
</tr>
</tbody>
</table>

56
DESCRIPTION: BT-40, STANDARD LENGTH
DR16 COLLET CHUCK
NOSE LENGTH 2.88" ASSEMBLED

DATE SCALE DRAWN SHEET PRT NO: B4C4-0016
6-29-92 1-1 B.H. 1 OF 1 DWG NO: 241110-A

7/24 TAPER BT-40 FLANGE
GAGE LINE

.41 DRILL THRU
R.12

.83 DIA
1.08 DIA

30°

.69

DR16 SERIES
MICRO PRECISION COLLETS
RANGE .019 - .393
DESCRIPTION: BT-40, EXTENDED LENGTH

DR20 COLLET CHUCK
NOSE LENGTH = 6.0" ASSEMBLED

DATE: 6-29-92
SCALE: .75
DRAWN: B.H.
SHEET: 1 OF 1
PRT NO: B4C5-0020

DRAWN SHEET

7/24 TAPER BT-40 FLANGE

6.00

R.38

2.58

GAGE LINE

1.06

3.33 REFERENCE TO STOP SCREW

1.24 DIA

M14-1.5 L.H. THREAD THRU

STOP SCREWS
PART NO: XCCS-1415

XCCS-1415-C (COOLANT .12 THRU HOLE AND URETHANE WASHER INCLUDED)
<table>
<thead>
<tr>
<th>DESCRIPTION</th>
<th>MAT'L:</th>
</tr>
</thead>
<tbody>
<tr>
<td>BT-50, STANDARD LENGTH</td>
<td></td>
</tr>
<tr>
<td>1.500 END MILL HOLDER</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DATE</th>
<th>SCALE</th>
<th>DRAWN</th>
<th>SHEET</th>
<th>PRT NO:</th>
<th>Dwg No:</th>
</tr>
</thead>
<tbody>
<tr>
<td>3/7/95</td>
<td>.75</td>
<td>R.J.</td>
<td>1 OF 1</td>
<td>B6E4-1500</td>
<td>221760-A</td>
</tr>
</tbody>
</table>

**Dimensions:**
- **DIA**: 1.535
- **4.000**
- **2.63**
- **1.504**
- **1.504**
- **1.000**
- **1.125**
- **2X 3/4-16 THD**
- **2.500 DIA**
- **1.75 DIA**
- **1.500 DIA**

**Notes:**
- SMS-650, NC4-3.0 THD, 3.28 DP
- 5/64 DR, 4.39 DP
- 7/24 TAPER
- BT-50 FLANGE

**Drafting Information:**
- C

**Technical Details:**
- ERIC
- DRAWN: R.J.
- DATE: 3/7/95
- SCALE: .75
DESCRIPTION: CT-50, STANDARD LENGTH
DF10 COLLET CHUCK
NOSE LENGTH = 3.25" ASSEMBLED

DATE: 9-18-91
SCALE: .80
DRAWN: B.H.
SHEET: 1 OF 1
PRT NO: C6C4-1000
DWG NO: 121320-A

1 1/8-16 L.H. THREAD
4.50 REFERENCE TO STOP SCREW
2.75 DIA
2.25 DIA
1.08
2.53 DIA

7/24 TAPER CT-50 V-FLANGE

4.00
3.25
1.38

CT-50 SERIES
COMPLEX COLLETS
RANGE: .094 - 1.000

STOP SCREWS
PART NO: XCCS-1816
XCCS-1816-C (COOLANT .25 THRU HOLE AND URETHANE WASHER INCLUDED)
DESCRIPTION: CT-50, STANDARD LENGTH
DR32 COLLET CHUCK
NOSE LENGTH: 4.0" ASSEMBLED

DATE       SCALE   DRAWN   SHEET   PRT NO: C6C4-0032
6-4-93      .75     B.H.    1 OF 1   DWG NO: 121210-A

DRAWN

3.88 REFERENCE TO STOP SCREW

DR32 SERIES
MICRO PRECISION COLLETS
RANGE: .078 - .787

STOP SCREWS
PART NO: XCCS-2215
XCCS-2215-C (COOLANT, .12 THRU HOLE AND URETHANE WASHER INCLUDED)
Sections Three & Four

Your Notes:
During this four hour segment, participants will:

1. Discuss functions and keys as found on a CNC Operator Panel.
2. Execute a short MDi command using appropriate keys.
3. "Write" a short CNC drill program using data input keys.
4. "Write" a short CNC milling program using data input keys.
5. "Edit" an existing program using alter, delete, insert, cursor and page keys.
CNC Operator Panel

- CRT Display
- Cursor Keys
- Reset Key
- Data Input Keys
- Program/Edit Keys
- Software Panel
- Input Key
- Start/Output Key
- Function Keys
- Page Change Keys
- Percentage Spindle Load Meter
- Spindle Override
- Cycle Start
- Feed Hold
- Fault Reset
- Manual Pulse Generator
- Emergency Stop
- Feed Rate Override
- Power On
Identify the location of the following elements on the control panel:

1. Mode Select - OPR/Alarm Function Key
   A] Edit  
   B] Auto (Memory)  
   C] M.D.i.  
   D] Handle  
   E] Jog  
   F] Zero Return (ZRN)

2. Reset
   A] Clears Alarm  
   B] Back to Start-Up Condition

3. Override Knobs
   A] Spindle Override  
   B] Feed Rate Override  
   *C] Rapid Override  
   *May have to set dry run to "ON" if no button exists.  
   Will control rapid and feed.

4. Feed Hold
   A] Stops axes motion  
   B] Coolant and spindle not stopped
CNC Operator Panel Usage

5. Manual Pulse Generator (Job Knob)
   A] Increments Switchable
      \[X1 = \]
      \[X10 = \]
      \[X100 = \]

6. Program/Edit Keys
   A] Edit Mode
   B] INS = \[of block\]
   C] DEL = \[of word\]
   D] ALT = \[of word\]
   E] EOB = \[command\]
   F] CAN = \[of last entry\]

7. Input Key Use
   A] Tool Length Offsets
      1. D Word \[\]
      2. H Word \[\]

   B] Work Coordinate Offsets
      1. G54-G59
         a] 01 = \[\]
         b] 02 = \[\]
         c] 03 = \[\]
CNC Operator Panel Usage

8. Power On
   A] Enables drive motors

9. Emergency Stop
   A] Panic button

10. Percentage Spindle Load Meter
    A] Working percent of spindle

11. Fault Reset
    A] Machine reset after error

12. Soft Keys
    A] Left arrow (function menu)
    B] Right arrow (operation menu)
13. X Y Z B Keys
   A] Use in zero return

14. Data input keys
   A] Program entry or edit

15. Start/Output Key
   A] Upload program to computer

16. Cursor Keys (Up and Down)
   A] Search use

17. Page (Up and Down)
   A] Different screens
   B] Program editing
CNC Operator Panel Usage

18. Z AT
   ATC
   A] Z axis at auto tool change position

19. Tool No.
   A] Tells; when pushed, which tool no. is in spindle
CNC Operator Panel

1. M.D.i. the machine to start spindle in a clockwise direction at 400 RPM.

2. Turn coolant on

3. Use MPG to mill edge of part

Steps:

1. 

2. 

3. 

4. 

5. 

6. 

7. 
Correctly input a CNC program to drill one hole using keyboard.

**Steps:**

1. 
2. 
3. 
4. 
5. 
6. 
7. 
8. 
9. 
10. 
1. Correctly input a CNC program to mill a slot using the keyboard.

Steps:

1. 
2. 
3. 
4. 
5. 
6. 
7. 
8. 
9. 
10. 
1. Correctly edit a CNC program using EDIT keys found on the keyboard.

Program # ____________________________

Steps:

1. ____________________________________________________________________________

2. ____________________________________________________________________________

3. ____________________________________________________________________________

4. ____________________________________________________________________________

5. ____________________________________________________________________________

6. ____________________________________________________________________________

7. ____________________________________________________________________________

8. ____________________________________________________________________________

9. ____________________________________________________________________________

10. ____________________________________________________________________________
Session Five

During this two hour segment, participants will:

1. Discuss the importance of Alarm Code Numbers as found on FANUC and similar controls.

2. Discuss Alarm contents.

3. Define and explain important letters and numbers pertaining to Alarm Codes.

4. Explain what to do when clear Alarm message are on the CRT screen.
Alarm List

Trouble shooting by Alarm Display...

Alarm List
When an alarm occurs, the alarm message is automatically displayed on the CRT.

<table>
<thead>
<tr>
<th>Classification</th>
<th>Alarm Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program errors, operation errors</td>
<td>000-222</td>
</tr>
</tbody>
</table>

The next few pages give the alarm number and the contents of that alarm.
# Alarm Codes

<table>
<thead>
<tr>
<th>Number</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>000</td>
<td>A parameter which requires the power off was input, turn off power.</td>
</tr>
<tr>
<td>001</td>
<td>TH alarm (A character with incorrect parity was input). Correct the tape.</td>
</tr>
<tr>
<td>002</td>
<td>TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective. Correct the tape.</td>
</tr>
<tr>
<td>003</td>
<td>Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)</td>
</tr>
<tr>
<td>004</td>
<td>A numeral or the sign &quot;-&quot; was input without an address at the beginning of a block.</td>
</tr>
<tr>
<td>005</td>
<td>The address was not followed by the appropriate data but was followed by another address or EOB code.</td>
</tr>
<tr>
<td>006</td>
<td>Sign &quot;-&quot; input error (Sign &quot;-&quot; was input after an address with which it cannot be used. Or two or more &quot;-&quot; signs were input.)</td>
</tr>
<tr>
<td>007</td>
<td>Decimal point &quot;-&quot; input error (A decimal point was input after an address with which it cannot be used. Or two decimal points were input.)</td>
</tr>
<tr>
<td>009</td>
<td>Unusable character was input in significant area.</td>
</tr>
<tr>
<td>010</td>
<td>An unusable G code was commanded.</td>
</tr>
<tr>
<td>011</td>
<td>Feedrate was not commanded to a cutting feed or the feedrate was inadequate.</td>
</tr>
</tbody>
</table>

(Note)  "T" is a general term for 0-TC and 00-TC. "M" is a general term for 0-MC and 00-MC.

<table>
<thead>
<tr>
<th>Number</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>014</td>
<td>In variable lead threading, the lead incremental and decremental outputted by address K exceed the maximum command value or a command such that the lead becomes a negative value is given. M only</td>
</tr>
<tr>
<td>015</td>
<td>The number of the commanded axes exceeded that of simultaneously controlled axes. M only</td>
</tr>
<tr>
<td>021</td>
<td>An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. M only</td>
</tr>
<tr>
<td>023</td>
<td>In circular interpolation by radius designation, negative value was commanded for address R. M only</td>
</tr>
<tr>
<td>027</td>
<td>No axis is specified in G43 and G44 blocks for the tool length compensation type C. Offset is not canceled and another axis is offset for the tool length compensation type C.</td>
</tr>
<tr>
<td>028</td>
<td>In the plane selection command, two or more axes in the same direction are commanded.</td>
</tr>
<tr>
<td>029</td>
<td>The offset value specified by H code is too large. M only</td>
</tr>
<tr>
<td>030</td>
<td>The offset number specified by H code for tool length offset or cutter compensation is too large. M only</td>
</tr>
<tr>
<td>031</td>
<td>In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. T only</td>
</tr>
<tr>
<td>032</td>
<td>In setting an offset amount by G10, the offset amount was excessive.</td>
</tr>
</tbody>
</table>
## Alarm Codes

<table>
<thead>
<tr>
<th>Number</th>
<th>Contents</th>
</tr>
</thead>
</table>
| 033    | A point of intersection cannot be determined for cutter compensation C.  
        M only  
        A point of intersection cannot be determined for tool nose radius compensation.  
        T only |
| 034    | The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C.  
        M only  
        The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation.  
        T only |
| 035    | G39 is commanded in cutter compensation B cancel mode or on the plane other than offset plane.  
        M only  
        Skip cutting (G31) was specified in tool nose radius compensation mode.  
        T only |
| 036    | Skip cutting (G31) was specified in cutter compensation mode.  
        M only |
| 037    | G40 is commanded at the plane other than offset plane in cutter compensation B. The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode.  
        M only  
        The offset plane is switched in tool nose radius compensation.  
        T only |
| 038    | Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center.  
        M only  
        Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center.  
        T only |
| 039    | Chamfering or corner R was specified with a start-up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R.  
        T only |
| 040    | Overcutting will occur in tool nose radius compensation in a canned cycle G90 or G94. T only |
| 041    | Overcutting will occur in cutter compensation C.  
        M only  
        Overcutting will occur in tool radius compensation.  
        T only |
| 042    | Tool position compensation is commanded in tool radius compensation.  
        M only |
| 044    | One of G27 to G30 is commanded in canned cycle mode.  
        M only |
<p>| 046    | Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference point return command. |
| 050    | Chamfering and corner R are commanded in the thread cutting block. |
| 051    | Improper movement or the move distance of the block next to that for which chamfering and corner R are commanded. |
| 052    | The block next to the block for which chamfering and corner R are commanded is not G01. |
| 053    | In the chamfering and corner R commands, two or more of I, K and R are specified. Otherwise, the character after a comma (&quot;,&quot; ) is not C or R in direct drawing dimensions programming. |
| 054    | A block in which the chamfering or the corner R was specified includes a taper command. |
| 055    | In the block for which chamfering and corner R are commanded, the move distance is commanded less than the corner R amount. |
| 056    | Neither the end point nor angle is specified in the command for the block next to that for which only the angle is specified (A). In the chamfering command, I (K) is commanded for the X (Z) axis. |
| 057    | Block end point is not calculated correctly in direct dimension drawing programming. |
| 058    | Block end point is not found in direct dimension drawing programming. |
| 059    | The program with the selected number cannot be searched, in external program number search. |</p>
<table>
<thead>
<tr>
<th>Number</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>060</td>
<td>Commanded sequence number was not found in the sequence number search.</td>
</tr>
<tr>
<td>061</td>
<td>Address P or G is not specified in G70, G71, G72, or G73 command. T only</td>
</tr>
</tbody>
</table>
| 062    | - The depth of cut in G71 or G72 is zero or negative value.  
  - The repetitive count in G73 is zero or negative value.  
  - The negative value is specified to \( \Delta i \) or \( \Delta k \) in G74 or G75.  
  - A value other than zero is specified to address \( U \) or \( W \), though \( \Delta i \) or \( \Delta k \) is zero in G74 or G75.  
  - A negative value is specified to \( \Delta d \), though the relief direction in G74 or G75 is determined.  
  - Zero or a negative value is specified to the height of thread or depth of cut of 1st time in G76.  
  - The specified minimum depth of cut in G76 is greater than the height of thread.  
  - An unusable angle of tool tip is specified in G76. |
| 063    | The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. T only |
| 065    | - G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. T only  
  - Address Z (W) or X (U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. |
| 066    | An unallowable G code was commanded between two blocks specified by address P and G in G71, G72 or G73. T only |
| 067    | G70, G71, G72, or G73 command with address P and G was specified in MDI mode. T only |
| 069    | The final move command in the blocks specified by P and G of G70, G71, G72 and G73 ended with chamfering or corner R. T only |

<table>
<thead>
<tr>
<th>Number</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>070</td>
<td>The memory area is insufficient.</td>
</tr>
<tr>
<td>071</td>
<td>The address to be searched was not found. Or the program with specified program number was not found in program number search.</td>
</tr>
<tr>
<td>072</td>
<td>The number of programs to be stored exceeded 63 or 125 (option).</td>
</tr>
<tr>
<td>073</td>
<td>The commanded program number has already been used.</td>
</tr>
<tr>
<td>074</td>
<td>The program number is other than 1 to 9999.</td>
</tr>
<tr>
<td>076</td>
<td>Address P was not commanded in the block which includes an M98 command or a G66 command.</td>
</tr>
<tr>
<td>077</td>
<td>The subprogram was called in three or five folds.</td>
</tr>
<tr>
<td>078</td>
<td>A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found.</td>
</tr>
<tr>
<td>079</td>
<td>The contents of the program stored in the memory did not agree with that in tape in collation.</td>
</tr>
<tr>
<td>080</td>
<td>In the area specified by parameter c, the measuring position reach signal does not come on. (Automatic tool compensation function) T only</td>
</tr>
<tr>
<td>081</td>
<td>Automatic tool compensation was specified without a T code. (Automatic tool compensation function) T only</td>
</tr>
<tr>
<td>082</td>
<td>T code and automatic tool compensation were specified in the same block. (Automatic tool compensation function) T only</td>
</tr>
<tr>
<td>083</td>
<td>In automatic tool compensation, an invalid axis was specified or the command is incremental. (Automatic tool compensation function) T only</td>
</tr>
<tr>
<td>085</td>
<td>When entering data in the memory by using ASR or Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate is incorrect.</td>
</tr>
</tbody>
</table>
Alarm Codes

<table>
<thead>
<tr>
<th>Number</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>086</td>
<td>When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off.</td>
</tr>
<tr>
<td>087</td>
<td>When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read.</td>
</tr>
<tr>
<td>090</td>
<td>The reference point return cannot be performed normally because the reference point return start point is too close to the reference point or the speed is too slow.</td>
</tr>
<tr>
<td>092</td>
<td>The commanded axis by G27 (Reference point return check) did not return to the reference point.</td>
</tr>
<tr>
<td>094</td>
<td>P type cannot be specified when the program is restarted. (After program interruption, the coordinate system setting operation was performed.)</td>
</tr>
<tr>
<td>095</td>
<td>P type cannot be specified when the program is restarted. (After program interruption, the external work offset amount changed.)</td>
</tr>
<tr>
<td>096</td>
<td>P type cannot be specified when the program is restarted. (After program interruption, the work offset amount changed.)</td>
</tr>
<tr>
<td>097</td>
<td>P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P / S 94 to 97 reset, no automatic operation is performed.)</td>
</tr>
<tr>
<td>098</td>
<td>A command of the program restart was specified without the reference point return operation after power ON and emergency stop, and G28 was found during search.</td>
</tr>
<tr>
<td>099</td>
<td>After completion of search in program restart, a move command is given with MDI.</td>
</tr>
<tr>
<td>100</td>
<td>Setting data PWE is set to 1. Turn it to 0 and reset the system.</td>
</tr>
<tr>
<td>101</td>
<td>The power was turned off while rewriting the contents of the memory in the part program storage &amp; editing operation. When this alarm is generated, set the setting data PWE to 1 and turn on the power while pushing the DELET to clear the memory.</td>
</tr>
<tr>
<td>110</td>
<td>The absolute value of fixed decimal point display data exceeds the allowable range.</td>
</tr>
<tr>
<td>111</td>
<td>The calculation result of macro instruction exceeds the allowable range (-2^{32} to 2^{32} - 1).</td>
</tr>
<tr>
<td>112</td>
<td>Division by zero was specified. (Including tan 90*)</td>
</tr>
<tr>
<td>113</td>
<td>A function which cannot be used in custom macro is commanded.</td>
</tr>
<tr>
<td>114</td>
<td>An undefined H code is designated in G65 block. For custom macro A</td>
</tr>
<tr>
<td></td>
<td>There is an error in other formats than &lt;Formula&gt;. For custom macro B</td>
</tr>
<tr>
<td>115</td>
<td>A value not defined as a variable number is designated.</td>
</tr>
<tr>
<td></td>
<td>The header contents are improper. This alarm is given in the following cases: High speed cycle machining</td>
</tr>
<tr>
<td></td>
<td>1. The header corresponding to the specified call machining cycle number is not found.</td>
</tr>
<tr>
<td></td>
<td>2. The cycle connection data value is out of the allowable range (0 - 999).</td>
</tr>
<tr>
<td></td>
<td>3. The number of data in the header is out of the allowable range (1 - 32767).</td>
</tr>
<tr>
<td></td>
<td>4. The storing start data variable number of executable format data is out of the allowable range (#20000 - #85535).</td>
</tr>
<tr>
<td></td>
<td>5. The last storing data variable number of executable format data is out of the allowable range (#85535).</td>
</tr>
<tr>
<td></td>
<td>6. The storing start data variable number of executable format data is overlapped with the variable number used in the header.</td>
</tr>
<tr>
<td>116</td>
<td>The variable number designated with P is forbidden for assignment.</td>
</tr>
<tr>
<td></td>
<td>The left side of substitution statement is a variable whose substitution is inhibited.</td>
</tr>
<tr>
<td>118</td>
<td>The nesting of bracket exceeds the upper limit (quintuple).</td>
</tr>
<tr>
<td>119</td>
<td>The argument of SQRT or BCD is negative.</td>
</tr>
<tr>
<td></td>
<td>The SQRT argument is negative. Or BCD argument is negative, and other values than 0 to 9 are present on each line of BIN argument. For custom macro B</td>
</tr>
<tr>
<td>122</td>
<td>The macro modal call is specified in double. M only</td>
</tr>
<tr>
<td>Number</td>
<td>Contents</td>
</tr>
<tr>
<td>--------</td>
<td>----------</td>
</tr>
<tr>
<td>123</td>
<td>Macro control command is used during DNC operation.</td>
</tr>
<tr>
<td>124</td>
<td>DO-END does not correspond to 1:1.</td>
</tr>
<tr>
<td>125</td>
<td>The unusable address is used in G65 block. For custom macro A, &lt;Formula&gt; format is erroneous. For custom macro B.</td>
</tr>
<tr>
<td>126</td>
<td>In DOn, 1 ≤ n ≤ 3 is not established.</td>
</tr>
<tr>
<td>127</td>
<td>NC and macro commands are confused.</td>
</tr>
<tr>
<td>128</td>
<td>The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched.</td>
</tr>
<tr>
<td>129</td>
<td>An address which is not allowed in &lt;Argument Designation&gt; is used. T only</td>
</tr>
<tr>
<td>130</td>
<td>In axis control, a 3rd axis control command was given by PMC during CI control. On the contrary, an attempt was made for CI control from PMC during axis control.</td>
</tr>
<tr>
<td>131</td>
<td>Five or more alarms have generated in external alarm message.</td>
</tr>
<tr>
<td>132</td>
<td>No alarm No. concerned exists in external alarm message clear.</td>
</tr>
<tr>
<td>133</td>
<td>Small section data is erroneous in external alarm message or external operator message.</td>
</tr>
<tr>
<td>135</td>
<td>Without any spindle orientation, an attempt was made for spindle indexing. T only</td>
</tr>
<tr>
<td>136</td>
<td>A move command of other axes was specified to the same block as spindle indexing addresses C, H. T only</td>
</tr>
<tr>
<td>137</td>
<td>A move command of other axes was specified to the same block as M-code related to spindle indexing. T only</td>
</tr>
<tr>
<td>139</td>
<td>An axis is selected in commanding by PMC axis control.</td>
</tr>
<tr>
<td>141</td>
<td>G51 (Scaling ON) is commanded in the tool offset mode. M only</td>
</tr>
<tr>
<td>142</td>
<td>Scaling magnification is commanded in other than 1 - 999999. M only</td>
</tr>
<tr>
<td>143</td>
<td>The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. M only</td>
</tr>
<tr>
<td>144</td>
<td>The coordinate rotation plane and arc or tool offset C plane differ from each other. M only</td>
</tr>
<tr>
<td>145</td>
<td>The condition at the polar coordinate interpolation start or cancel is not correct. G112/G113 are commanded by other mode than G40. There are error at the plane selection. (Error of parameter setting) T only</td>
</tr>
<tr>
<td>146</td>
<td>G code which cannot be commanded are specified during the polar coordinate interpolation mode. T only</td>
</tr>
<tr>
<td>148</td>
<td>Automatic corner override deceleration rate is out of the settable range of judgement angle. Check parameter No. 0213, 0214, 0215. M only</td>
</tr>
<tr>
<td>150</td>
<td>Tool Group No. exceeds the maximum allowable value. M only</td>
</tr>
<tr>
<td>151</td>
<td>The tool group commanded in the machining program is not set. M only</td>
</tr>
<tr>
<td>152</td>
<td>The number of tools within one group exceeds the maximum value registerable. M only</td>
</tr>
<tr>
<td>153</td>
<td>A T-code is not stored in the due block. M only</td>
</tr>
<tr>
<td>154</td>
<td>When the group is not commanded, H99 or D99 was commanded. M only</td>
</tr>
<tr>
<td>155</td>
<td>In the machining program, M06 and T code in the same block do not correspond to the group in use. M only</td>
</tr>
<tr>
<td>156</td>
<td>P and L commands are missing at the head of program in which the tool group is set. M only</td>
</tr>
<tr>
<td>157</td>
<td>The number of tool groups to be set exceeds the maximum allowable value.</td>
</tr>
</tbody>
</table>
### Alarm Codes

<table>
<thead>
<tr>
<th>Number</th>
<th>Contents</th>
</tr>
</thead>
<tbody>
<tr>
<td>158</td>
<td>The tool life to be set is too excessive. M only</td>
</tr>
<tr>
<td>159</td>
<td>During setting program execution, power was OFF. M only</td>
</tr>
<tr>
<td>160</td>
<td>Different M code is commanded in Heads 1 and 2 as waiting M code. OTT only</td>
</tr>
<tr>
<td>165</td>
<td>An attempt was made to execute a program of an even number in Head 1 or an odd program number in Head 2. OTT only</td>
</tr>
<tr>
<td>175</td>
<td>Conditions when performing circular interpolation start or cancel not correct. 1. G107 is not commanded simultaneously with rotation axis radius. 2. G107 is commanded simultaneously with two axes. 3. G107 is commanded during cutter radius compensation. T only.</td>
</tr>
<tr>
<td>176</td>
<td>G code is commanded during circular interpolation mode when commands cannot be performed. T only</td>
</tr>
<tr>
<td>178</td>
<td>Commanded in the G41/G42 mode.</td>
</tr>
<tr>
<td>179</td>
<td>The number of controlled axes set by the parameter 597 exceeds the maximum number.</td>
</tr>
<tr>
<td>190</td>
<td>In the constant surface speed control, the axis specification is wrong. (Misprogram) M only</td>
</tr>
<tr>
<td>197</td>
<td>The program commanded the CI axis to move when the COFF signal was ON.</td>
</tr>
<tr>
<td>200</td>
<td>In the rigid tap, an S value is out of the range or is not specified. (programming error)</td>
</tr>
<tr>
<td>201</td>
<td>In the rigid tap, no F value is specified. (programming error)</td>
</tr>
<tr>
<td>202</td>
<td>In the rigid tap, spindle distribution value is too large (system error)</td>
</tr>
<tr>
<td>203</td>
<td>In the rigid tap, position for M29 or an S command is incorrect.</td>
</tr>
</tbody>
</table>
Alarm Code Worksheet

I. Define

A] M only ____________________________
   ____________________________
   ____________________________

B] T only ____________________________
   ____________________________
   ____________________________

II. Explain

A] Code 065 ____________________________
   ____________________________
   ____________________________

B] Code 074 ____________________________
   ____________________________
   ____________________________

C] Code 010 ____________________________
   ____________________________
   ____________________________
<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>030</td>
<td></td>
</tr>
<tr>
<td>023</td>
<td></td>
</tr>
<tr>
<td>072</td>
<td></td>
</tr>
</tbody>
</table>
III. Apply your knowledge

A code number comes on your CRT screen that is larger in value than #222.

Does the alarm have anything to do with program or operation errors?

_____ Yes
_____ No

Explain your answer:

________________________________________________________________________
________________________________________________________________________
________________________________________________________________________
________________________________________________________________________
________________________________________________________________________
Your Notes:

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________

__________________________________________________________________________________
Sections Six & Seven

Your Notes:
Sessions Six and Seven

During this four hour segment, participants will:

1. Define "Tool Length Offset" Referring to this manual.

2. Explain "G" Codes: G43, G44 and G49 which deal with tool length compensation.

3. Understand command format using G43 or G44 in the CNC program.

4. Apply "H" and "D" words used for tool offsets.

5. Understand preset tool methodology.

6. Review tool presetter information pages.

7. Complete a tool length setting using a tool presetter on the shop floor.

One tool length offset is known, a "G" Code will cap up the offset length for that tool in the program.

Command format:

\[ N \text{_________ G43 Z_________ H_________} \]

Where:
- \( N \) = Sequence number
- \( G43 \) = Tool length offset positive direction
- \( Z \) = Z coordinate of tool
- \( H \) = Length amount
- Ex. = 5.250 inches

\[ N \text{_________ G44 Z_________ H_________} \]

Where:
- \( N \) = Sequence number
- \( G43 \) = Tool length offset negative direction
- \( Z \) = Z coordinate of tool
- \( H \) = Length amount
- Ex. = -5.250 inches

Tool cancel
- \( G49 \) = Cancel offset amount
Two words which apply to tool offsets.

"H" Word  Used generally for tool length values

"D" Word  Used generally for tool radius values

Tool offset numbers in control

01 - 21  [Use for "H" words]
22 - 64  [Use for "D" words]
**Tool Offset**

*Example: Where found in program*

```
:0 2021
N5  G20  G40  G49  G80  G90
N10  T2  M6  (.T Dia End Mill)
N15  51600  M3
N20  60  G54  X0  Y6.
N25  G43  Z.1  H2  M8
N30  G1  Z-.1  H2  M8
```

```plaintext
G43  
Z.1  
H2  
M8  
```
Preset Tool Method

Your Notes

_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________
_____________________________________________________________________


Overview of machine
OVERVIEW

The Parset MASTER control is based on the principle that there are OPERATIONS that are done to COMPONENTS.

The four basic operations available are:

STORE / ADD
MEASURE / PRESET
MODIFY / DELETE / CANCEL
VIEW / DISPLAY

The three basic components that the operations are applied to are:

ZERO GAGE
TOOL / TOOL ASSEMBLY
TOOL PROGRAM

By combining an operation with a component, you store information into the control and recall that information when necessary to measure or preset a particular tool.

COMPONENTS

ZERO GAGE: This is the calibration point which is used in the control to calculate the measurements. On all controls, you must the crosshairs of the projector on the Zero Gage tip, before you STORE / ADD the zero gage. This is necessary because the glass scales are read to get the current position of the machine when you STORE / ADD a zero gage.

The data that is stored with a Zero Gage is:

ZERO GAGE # - ID number
X value - X value of the calibration point
Z value - Z value of the calibration point

TOOL / TOOL ASSEMBLY. A TOOL is a holder, cutting tool, retention knob, etc. that is ready to be placed in the machine tool. The only difference between a TOOL and TOOL ASSEMBLY is that a TOOL ASSEMBLY is stored in the memory of the control, and a TOOL is not.
PARSET MASTER

The Parset Master is a DRO type control designed exclusively for use on tool measuring machines.

STARTUP

When the control is first turned on, it will go through a self diagnostic procedure that takes about ten seconds. When it is finished, the control will beep, and the prompt display will show "Reference scales". To reference the scales, move the joystick in the X axis until the control beeps twice in rapid succession. Then move the Z axis until the control beeps twice in rapid succession. If you want, you can move both axes at the same time. Once the control has beeped for both axes, the control is referenced.
5) Type in the Z value for this tool from the number pad. This is normally the nominal length of the tool being stored.

6) Press ENTER - Prompt display shows "X:?"

7) Type in the X value for this tool from the number pad. This is normally the nominal diameter of the tool being stored.

8) Press ENTER - prompt display shows "ZG #:?"

9) Type in the zero gage number for this tool from the number pad. This must have been already stored with Store Zero Gage.

10) Press ENTER - Prompt display shows "TA stored" for a couple of seconds.

STORING A TOOL PROGRAM

1) Press STORE - prompt display shows "Store what?"

2) Press TOOL PROGRAM - prompt display shows "TP #:?"

3) Type in the tool program ID from the number pad. The ID can be up to 10 digits long, and hypens (-) and decimal points (.) are allowed.

4) Press ENTER - prompt displays shows "Step 1" for a couple of seconds.

5) Prompt displays "TA or ZG".

6) Press either TOOL ASSEMBLY or ZERO GAGE depending on what the first step of the program is going to be.

IF YOU CHOOSE TA

7) Press TOOL ASSEMBLY - prompt displays shows "TA #:?"

8) Type in the tool assembly ID using the number pad. The tool assembly must have already been stored using Store Tool Assembly.

9) Press ENTER - Prompt display shows "Pkt #:?"

10) Type in the pocket number for this step of the program using the number pad.

11) Press ENTER - prompt display shows "TA step stored" for a couple of seconds.
length and diameter of a given tool. The only information you need to be able to Measure/Preset is the ID of the component.

MEASURING A TOOL

1) Put the tool in the Parsetter spindle
2) Press MEASURE/PRESET - prompt display shows "Measure what?"
3) Press ZERO GAGE - prompt display shows "ZG #:?"
4) Type in the Zero Gage number from the number pad.
5) Press ENTER
6) You can now measure the tool.

MEASURING A TOOL ASSEMBLY

1) Put the tool in the spindle of the Parsetter.
2) Press MEASURE/PRESET - Prompt display shows "Measure what?"
3) Press TOOL ASSEMBLY - prompt display shows "TA #:?"
4) Type in the tool assembly ID number from the number pad
5) Press ENTER
6) You can now measure the Tool Assembly
VIEW / DISPLAY: This allows you to look at what components are already stored in the memory of the control. You would use this if you needed to see if the component you were looking for was in the control or if someone had deleted it.

VIEWING A ZERO GAGE

1) Press VIEW - prompt display shows "View what?"
2) Press ZERO GAGE
3) Use the UP ARROW and DOWN ARROW keys on the number pad to scroll the list of ZERO GAGES. The numbers that you see in the Z and X displays are the values of the Zero gage.
4) To finish the view operation, either scroll off the end of the list, or press the CANCEL / NO key.

VIEWING A TOOL ASSEMBLY

1) Press VIEW - prompt display shows "View what?"
2) Press TOOL ASSEMBLY
3) Use the UP ARROW and DOWN ARROW keys on the number pad to scroll the list of tool assemblies. The numbers that you see in the Z and X displays are the nominal values of that tool. To see which Zero Gage the tool assembly is using, press the MORE key once. This will change the prompt display so it will show the Zero gage. To change back to viewing the Tool Assemblies, press the MORE key once again.
4) To finish the view operation, either scroll off the end of the list, or press the CANCEL / NO key.
MODIFY / DELETE / CANCEL: This function allows you to permanently remove a given component from the memory of the control. This would be done either because that component was no longer used, or the memory of the control is full and you need to free some of it.

DELETING A ZERO GAGE

WARNING - DO NOT DELETE ZERO GAGE # 0. This Zero Gage in required for the control to function correctly.

1) Press CANCEL / NO - prompt display shows "Clear what?"

2) Press ZERO GAGE - prompt display shows "ZG #:?"

3) Using the number pad, type in the number of the Zero Gage that you wish to delete.

4) Press ENTER

5) If the prompt display shows "ZG Cleared", then that zero gage was removed from the memory of the control.

6) If the prompt display shows "ZG not cleared" - "Used in TA or TP", it means that either a Tool assembly or a Tool program is using that Zero gage, and it cannot be cleared.

DELETING A TOOL ASSEMBLY

1) Press CANCEL / NO - prompt display shows "Clear what?"

2) Press TOOL ASSEMBLY - prompt display shows "TA #:?"

3) Using the number pad, type in the number of the Tool Assembly that you wish to delete.

4) Press ENTER

5) If the prompt display shows "TA Cleared", then that Tool Assembly was removed from the memory of the control.

6) If the prompt display shows "TA not cleared" - "Used in TP", it means that a Tool program is using that Tool Assembly, and it cannot be cleared.
Projector Operation

Thin band of light

Position the tool tip in the projector's field of vision. Focus the tool by rotating until the image is sharp. Position the tool tip as shown in Figure, leaving a thin band of light between the tool tip and the crosshair. The magnification of the projector is 20 times so that the error caused by this thin band is virtually insignificant.

Cross hair rotation wheel

Turn the cross hair rotation wheel until the desired angle is reached when viewed in the projector's internal protractor. This feature is used when measuring tools with leading edges.

The Parsetter 240 is equipped with a lamp dimmer circuit to extend bulb life. If the machine is idle for 20 minutes the power to the projector bulb is interrupted which dims the bulb. Touching the joy-stick turns this back on.
Rough positioning is accomplished by moving the joy-stick in the direction desired. Two speeds are available for this jog feature, fast and slow. These are selected by depressing or releasing the slow fast switch on the control panel. Fine adjustment is made utilizing the thumb wheels.

The slow fast switch is a dead man switch which defaults to slow speed. Pressing and holding in the switch engages high speed.

Pushing the joy-stick in the direction of an arrow will move the projector in that direction. Pushing the joy stick at 45° between two arrows will move the projector in both directions.

Fine adjustment range is unlimited. Two thumb wheels are provided for the X axis. Arrows on the machine indicate which direction to turn the thumb wheels to get the desired movement.
What is presetting?
Presetting is the act of positioning a tool tip at a specific set of coordinates. These locations are called the nominal tool dimensions.

Why should you preset?
There are four basic reasons to preset. These are as follows:
1. Certain diametrically adjustable tools (boring tools, reamers) require a diameter setting to produce a good part.
2. Multiple spindle machines with the spindles at different heights require that tools be set for each spindle to produce identical parts.
3. Special requirements of the program for clearance or machining conditions.
4. Tools used on older controls that have no tool offset capabilities.

How to preset an adjustable tool.

1. Diameter adjustable tools.
Position the machine so that the diameter (or radius) desired is showing on the x axis display field of the control. Adjust the cutting bringing the tool tip to the cross hairs of the projector.

2. Length adjustable tools.
2.1. Positive lock end mill holders.
Position the machine so that the length desired is showing on the Z axis display field of the control. Adjust the tool length using the back up screw. Tighten the side lock screw. Final tighten the side lock screw and snug the back up screw in a tightening fixture, not the presetter spindle.

2.2. Collet chucks.
Collets draw down .005 - .015 as they are tightened. Position the machine so that the length desired + .015 is showing on the Z axis display field of the control. Tighten the collet nut. Reposition the machine to measure the actual tool length. Snug up the back up screw.

NOTE: Never tighten a collet chuck with the tool snugged up against a back up screw. This prevents the proper pull down action of the collet and can reduce the gripping power by as much as 50%.
Tool Measuring Basics

What is tool measuring?

Tool measuring is the act of establishing a tool's actual length and diameter. This is different from presetting in that no adjustment to the tool length and diameter to achieve specific dimensions is made.

Why should you measure your tools?

You can't make a good part on a CNC machine until you at least know the tool length. At some point in the machine set up process you must establish this dimension either by a measuring operation or by touching off.

How to measure a tool.

1. Tools with small radii

Position the machine until the tool tip is in the field of vision of the projector. Focus the tool by rotating until the image is sharp. Clamp the tool. Position the machine until the tool tip is aligned with the cross hairs of the projector.

2. Tools with large radii

Position the machine and focus the tool as above. Position the machine and measure either the length or diameter. Press the hold button for the axis measured. Position machine and measure the other axis. Print a label if desired. Release the axis hold by depressing the button.

3. Drills

Drilled holes are generally dimensioned as shown in figure one. It is very easy to measure a drill to produce this feature by aligning the drills outside diameter and point with the projector cross hairs.

![Figure 1](image1.png)  
**Figure 1.** Dimension of drill depth.

![Figure 2](image2.png)  
**Figure 2.** Measuring set length to full drill depth.

4. Runout

The runout of drills and other tools can be quickly and easily checked. Locate one edge of the drill on the cross hair of the projector. Press the absolute/incremental switch on the control so the each axis reads zero. Rotate the tool 180°. Any run will be easily seen and can be measured by moving projector so that the other edge is located on the cross hair. Point runout of drills can be measured in the same way.
Replacing and centering the light bulb:

Turn off the power switch. Loosen the set screw and pull the lamp socket out. Pull the old bulb out. Using a soft cloth pick up the new bulb and insert into the socket. Never grip the bulb with bare fingers as this may adversely affect bulb life. Push the socket back in the housing making sure that the filament is in the vertical position. Adjust the socket in and out of the lens housing to achieve the greatest projector illumination. Tighten the set screw.

Preventive maintenance:

Standard spindles
Periodically oil spindle oiler with 30 weight oil.

Optional air vacuum spindles
Periodically check the water trap. If an excessive amount of water has accumulated, drain the reservoir.
Periodically check the oiler. If the oil level is low refill the reservoir.

Trouble shooting guide:

We turn the machine on and nothing happens! Make sure machine is plugged in.

Machine is plugged in and nothing happens! Check fuse and replace as required.

Projector light works but the control is not on! Turn control on with switch located in front panel.

When the tool pressure switch is engaged nothing happens! Make sure air line is properly connected, then increase pressure by turning the pressure control valve clockwise.

Projector light is not on! Replace projector bulb. Before replacing the bulb make sure that the machine is not in a dimmer cycle. If the machine is not used for 20 minutes the light bulb will automatically dim to improve bulb life. Moving the joy-stick turns the light back on.

Axes will not move. Remove shipping bolts.

Thank you again for purchasing a Parsetter 240. If you need additional information on accessories or service please call 1-800-PARSET.
Considerations for the Preset Area

Organizational ideas
Do I have to reorganize my shop?
No you don't have to reorganize your shop to benefit from your PARSETTER. What is required is some discipline and planning.

Considerations for the preset area
1. Locate an assembly/breakdown bench adjacent to the machine. Equip this with tightening fixtures and a vise. Use this bench to assemble tools, install retention knobs and to do any final tightening.
2. Establish some method of communicating the tooling requirements for a given program to the preset operator.
3. DO equip your PARSETTER with a label printer.
4. Have enough toolholders to set up the next job while the current job is running.
5. Provide some means of transporting set up tool assemblies to the machine without banging the tapers together.
6. Send any required perishable tools (inserts, drills, taps) to the machine along with the tool assemblies.

Set up sheets
One method of communicating tooling requirements is through the use of a set up sheet. A sample of one type of set up sheet is shown on the next page.
Practical Operation of Presetter

Steps:

1. 

2. 

3. 

4. 

5. 

6. 

7. 

8. 

9. 

10. 

Reading = Z

---

79 112
Tool Offset

Complete setup sheet for Tool #1, after performing practical tool preset.
## CNC MILL TOOL SETUP SHEET

<table>
<thead>
<tr>
<th>OP NO.</th>
<th>PART NAME</th>
<th>PART #</th>
<th>FIXTURE NO.</th>
<th>PROGRAMMER</th>
<th>L/O #</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TOOL NO.</th>
<th>TOOL DESCRIPTION</th>
<th>SPINDLE ADAPTER</th>
<th>TOOL ADAPTER</th>
<th>SETUP (TLO) HEIGHT</th>
<th>COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

NOTES

---

DATE

C

81 114
Section Eight

Your Notes:

------------------------------------
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
|                                  |
Session Eight

*During this two hour segment, participants will:*

1. Become familiar with the Matsuura CNC control panel.
2. Power up a CNC machine in the proper sequence of events.
3. Set the switches on the main operator’s control panel to recommend settings.
4. Understand procedures regarding automatic operation of CNC machines.
5. Power down CNC machines in proper sequence of events.
6. Understand how to access the program directory (library) on the Matsuura CNC machine.
Matsuura Control Panel

Name the control features

6. __________________________
7. __________________________
8. __________________________
9. __________________________
10. __________________________
11. __________________________
12. __________________________
13. __________________________
14. __________________________
15. __________________________
16. __________________________
17. __________________________
18. __________________________
19. __________________________
20. __________________________
21. __________________________
22. __________________________
23. __________________________
24. __________________________
25. __________________________
26. __________________________
27. __________________________
28. __________________________
29. __________________________
30. __________________________
31. __________________________
32. __________________________
33. __________________________
34. __________________________
35. __________________________
36. __________________________
37. __________________________
38. __________________________
39. __________________________
40. __________________________
41. __________________________
42. __________________________
43. __________________________
44. __________________________
45. __________________________
46. __________________________
47. __________________________
48. __________________________
Powering Up CNC Machines

Lubrication
1] Slide lube filled sufficiently
2] Coolant oil tank
3] Air lubricator

Spindle Taper
1] Make sure it is clean
2] Tapers of tool holders clean
3] Pull stud is securely tightened

Doors of NC Unit
1] Make sure door is securely closed

Turning Power On
1] Turn on main power switch and check if main power lamp is lit
2] Press power of button on operation panel
   Note: some control require this to be pressed twice; first time for NC unit, second time for machine side

Air Pressure
1] Air lubricator normally requires 70 psi
Manual Zero Return Axes*

1] Set mode switch (10) to zero return

2] Depress (+) button of the Manual Feed
Push button switch (48)

3] Continue pushing (+) push button unit all axes are
at zero return position

4] The first zero return speed after power ON is set at
50% of the rapid feedrate.

5] Axes traverse if first performed at rapid feedrate,
then when the axis runs over the deceleration
DOG, it decelerates and stops at zero point. Zero
position lamp is then lit.

*Note: some controls allow automatic
zero return of all axes at power up.
Automatic operation (memory operation)

Automatic operation is performed by the following procedures.

a) Make sure that the tools are arranged in the magazine according to the program data sheet.

b) Make sure that each axis is returned to each zero point manually after power ON.

c) Make sure that [PS ALARM] lamp and lamp(40) on the main control panel are extinguished.

d) Make sure that no alarm message is displayed on the CRT screen. Alarm message, if any, is displayed when the ALARM key on the CRT unit is depressed.
**Automatic Operation (Memory Operation)**

e) Make sure that a tool is not in the spindle. In the presence of a tool, check the CRT screen to see whether the tool number is registered on the control. If it is not registered, set the tool number according to the parameters.

f) Check and make sure the tool length offset and cutter radius compensation values are input correctly.

g) Depress [FORWARD] switch for the chip conveyer (option).

h) Registration of program

Work programs made according to program FORMAT are registered into NC memory. Each program must have its own program number at the head of the program.

i) Make sure that each axis is positioned at the specific position shown by program data sheet and check the position display.
Automatic Operation (Memory Operation)

<table>
<thead>
<tr>
<th>Switch</th>
<th>Setting</th>
<th>Purpose</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODE switch (10)</td>
<td>On or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>RAPID TRAVERSE OVERRIDE</td>
<td>On or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>FEEDRATE OVERRIDE</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>SPINDLE OVERRIDE</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>SINGLE BLOCK switch (18)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>DRX RUN switch (19)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>BLOCK SKIP switch (20)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>OPTIONAL STOP switch (22)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>Z-FEED NEGLECT switch (23)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>M,S,T LOCK switch (24)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>MACHINE LOCK switch (25)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>MIRROR IMAGE switch (26)</td>
<td>ON or OFF</td>
<td>Normally at ON or OFF.</td>
</tr>
<tr>
<td>EDIT switch (29)</td>
<td>LOCK</td>
<td>Normally at LOCK.</td>
</tr>
</tbody>
</table>

**NOTE:** Underlined settings are normally used for machining operations.

j) Set the switches on the main operation control panel.

k) The program is started by depressing the CYCLE START push-button (16) on the main operation control panel. The CYCLE START lamp is lit when the automatic operation is started.
Stopping the Machine During Operation

Stopping the machine during an automatic operation

The machine can be stopped during an automatic operation in various ways, as occasion requires.

1) Programming of M00 or M01 beforehand.

When M00 or M01 is commanded, spindle rotation, coolant and oil mist are stopped and the axis feed is temporarily stopped after completion of its block. (M01 is effective only when OPTIONAL STOP switch is ON.) Restart is effected by depressing CYCLE START pushbutton (16).

This function is used:
- to check the work dimensions.
- to exchange workpieces.
- to remove the cutting chips.
- to perform manual operation.
- to change the tools manually.
2) Temporary stop by FEED HOLD pushbutton (17)

Axis feed is decelerated and stopped by depressing FEED HOLD pushbutton switch during an auto. operation. When M.S.T. functions are already commanded, they are performed. The CYCLE START lamp (16) is extinguished and FEED HOLD lamp is lit.

FEED HOLD is ignored when the Z axis cutting feed is performed by a tapping cycle (G74 or G84).

FEED HOLD is effective on DWELL, the DWELL is stopped.

During a temporary stop by FEED HOLD, the following manual operations can be performed.

   a) After a FEED HOLD stop, note down the coordinate values of the stopped points of each axis by means of the position display.
   b) Set MANUAL ABSOLUTE switch to ON or OFF.
   c) Perform the manual operation.
   d) Bring each axis back to the position of the noted coordinate value by using the position display.
   e) Change MODE select switch (10) back to TAPE or MEMORY.
   f) Depress the cycle start button (16) to re-start the auto. operation.
2. Manual operation of the machine
   - ON/OFF operation of the spindle rotation.
   - Rotation of magazine after spindle stop
   - Removing/inserting the tool holder to/from the spindle after spindle stop.
   etc. can be performed.

Note: MDI operation cannot be performed during a temporary stop by FEED HOLD. To perform MDI operation, the system must be reset once. If it is desired to change the operation mode, wait until the operation is stopped with a single block.
5) Emergency Stop

When the machine must be stopped immediately during an operation, depress EMERGENCY STOP (9) button. Then, the power of the machine is shut off, spindle rotation, axis feed are stopped immediately. Emergency stop is also applied to NC and servo system is turned off. All the motions of the machine are stopped.

To resume the operation, turn on POWER again, and return each axis to its zero return manually again.
Shutting off of the Power

a. Make sure that CYCLE START lamp (6) on the main operation panel is extinguished.

b. Make sure that each motion (axis feed or ATC motion etc.) of the machine is not performed. If the spindle is rotated, stop it.

c. Remove the tool if the tool is inserted into the spindle or in the ATC arm grip.

d. Clean the inside of spindle taper carefully, apply rust-preventive oil.

e. Return each axis (especially X axis) to the center to maintain the balance of the machine.

f. Depress [STOP] switch of the chip conveyer to stop the chip conveyer.

g. Depress EMERGENCY STOP pushbutton (9) on the main operation panel. Then the power of the machine is shut off and emergency stop is allied to the NC system, causing the servo system to be turned off.

h. Depress POWER OFF switch (7) on the CRT unit. Then the power of NC unit is shut off.

i. Turn the main power switch to OFF.
Procedure to bring up directory on CRT screen.

1] Place mode switch in **EDIT** mode.

2] Turn editor switch to **ON**.

3] Press **CANCEL** button.

4] Press **ORIGIN** button.

5] Press **CURSOR DOWN** button.

*Note: wait a few seconds and the programs residing in memory will be displayed on the machine.*
Questions.

1. Reference switch No. 10. Which mode is selected to manually references the axes?

2. Which axis is reverenced first? Why?

3. What tells you on the control that the axes are at their home or zero return position?

4. What does light No. 39 indicate?
5. For automatic operation, what should the following switches be set to?

<table>
<thead>
<tr>
<th>Switch</th>
<th>On or Off</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z Feed Neglect</td>
<td></td>
</tr>
<tr>
<td>M, S, T, Lock</td>
<td></td>
</tr>
<tr>
<td>Machine Lock</td>
<td></td>
</tr>
<tr>
<td>Mirror Image</td>
<td></td>
</tr>
</tbody>
</table>

6. Where should the axes X, Y, and Z position prior to shutting off the power? Why?

Why? ____________________________________

7. When the "Key Switch" is turned "ON", what does this allow you to do?

________________________________________

________________________________________

________________________________________
Section Nine

Your Notes:

____________________________________________________________________________________

____________________________________________________________________________________

____________________________________________________________________________________

____________________________________________________________________________________

____________________________________________________________________________________
Session Nine

During this segment, participants will:

1. Understand the work address system.
2. Understand preparatory and miscellaneous functions used in CNC programming.
3. Understand how G Codes groups work in program format.
4. Explain most common G and M codes in CNC language.
5. Locate common errors in CNC program.
7. Explain the function of G99/G98 initial/R planes.
8. Review questions on G and M codes.
Definitions:

Work Address Format
Addressing each work in a block by one or more characters which identify the meaning of the word.

Word
An ordered set of characters composed of a letter and some numeric value which may be used to cause a specific action of a CNC machine.

Variable Block Format
A format which allows the number of works in successive blocks to vary. A word address system is also a variable block format.

Block
A group of "words" considered as a unit and separated from other such units by an End of Block (EOB) character.
CNC Code Language

**Words and Numbers:**

**N-Word**
A sequence number used to identify a block of information in your program.

**Sequence Number**
A number addressed by the letter "N" identifying the location of blocks in a program.

Examples:

N5......;
N10......;
N15......;
Preparatory Codes (G Function)
A number following address "G" determines the meaning of the command for the concerned block.

G codes are divided into the following types:

One Shot G Code
G Code which is only effective in the block which it is specified.

Modal G Code
G Code which is effective until another G code of the same group is specified.

Example:
G01 and G00 are modal G codes in Group 01.

```
G01 X_______
    2_______
G00 X_______
    2_______
```
<table>
<thead>
<tr>
<th>G00</th>
<th>01</th>
<th>RAPID POSITIONING</th>
</tr>
</thead>
<tbody>
<tr>
<td>G01</td>
<td>01</td>
<td>LINEAR INTERPOLATION - USES FEEDRATE</td>
</tr>
<tr>
<td>G02</td>
<td>01</td>
<td>CW CIRCULAR INTERPOLATION</td>
</tr>
<tr>
<td>G03</td>
<td>01</td>
<td>CCW CIRCULAR INTERPOLATION</td>
</tr>
<tr>
<td>G04</td>
<td>*</td>
<td>DWELL - USES P FOR TIME, 1 SEC = P1000</td>
</tr>
<tr>
<td>G06</td>
<td>*</td>
<td>POSITIONING ERROR DETECT OFF ... YASNAC ONLY</td>
</tr>
<tr>
<td>G09</td>
<td>*</td>
<td>EXACT STOP CHECK -- FOR SHARP EDGES</td>
</tr>
<tr>
<td>G12</td>
<td>*</td>
<td>CIRCLE CUTTING CW ID ONLY ... YASNAC ONLY</td>
</tr>
<tr>
<td>G13</td>
<td>*</td>
<td>CIRCLE CUTTING CCW ID ONLY ... YASNAC ONLY</td>
</tr>
<tr>
<td>G17</td>
<td>02</td>
<td>XY PLANE DESIGNATION</td>
</tr>
<tr>
<td>G18</td>
<td>02</td>
<td>XZ PLANE DESIGNATION</td>
</tr>
<tr>
<td>G19</td>
<td>02</td>
<td>YZ PLANE DESIGNATION</td>
</tr>
<tr>
<td>G20</td>
<td>06</td>
<td>INCH INPUT DESIGNATION</td>
</tr>
<tr>
<td>G22</td>
<td>06</td>
<td>METRIC INPUT DESIGNATION</td>
</tr>
<tr>
<td>G27</td>
<td>*</td>
<td>REFERENCE POINT RETURN CHECK</td>
</tr>
<tr>
<td>G28</td>
<td>*</td>
<td>RETURN TO REFERENCE POINT (MACHINE &quot;0&quot;)</td>
</tr>
<tr>
<td>G29</td>
<td>*</td>
<td>RETURN FROM REFERENCE POINT</td>
</tr>
<tr>
<td>G40</td>
<td>07</td>
<td>TOOL RADIUS COMPENSATION CANCEL</td>
</tr>
<tr>
<td>G41</td>
<td>07</td>
<td>TOOL RADIUS COMPENSATION LEFT</td>
</tr>
<tr>
<td>G42</td>
<td>07</td>
<td>TOOL RADIUS COMPENSATION RIGHT</td>
</tr>
<tr>
<td>G43</td>
<td>08</td>
<td>TOOL LENGTH COMPENSATION PLUS DIRECTION</td>
</tr>
<tr>
<td>G44</td>
<td>08</td>
<td>TOOL LENGTH COMPENSATION MINUS DIRECTION</td>
</tr>
<tr>
<td>G49</td>
<td>08</td>
<td>TOOL LENGTH COMPENSATION CANCEL</td>
</tr>
<tr>
<td>G45</td>
<td>*</td>
<td>TOOL OFFSET INCREASE</td>
</tr>
<tr>
<td>G46</td>
<td>*</td>
<td>TOOL OFFSET DECREASE</td>
</tr>
</tbody>
</table>

* - ONE SHOT CODE
1. ONLY G-CODES FROM THE SAME GROUP CAN CANCEL EACH OTHER
2. ONLY 1 G-CODE FROM THE SAME GROUP IN A LINE
<table>
<thead>
<tr>
<th>GROUP</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>G81</td>
<td>CANNED CYCLE # 1 FEED IN RAPID OUT SPOT DRILLING</td>
</tr>
<tr>
<td>G82</td>
<td>CANNED CYCLE # 2 FEED IN, DWELL, RAPID OUT CHAMFERING</td>
</tr>
<tr>
<td>G83</td>
<td>CANNED CYCLE # 3 DEEPHOLE DRILLING</td>
</tr>
<tr>
<td>G84</td>
<td>CANNED CYCLE # 4 TAPPING ( PITCH * RPM * .98 = FEEDRATE )</td>
</tr>
<tr>
<td>G85</td>
<td>CANNED CYCLE # 5 FEED IN, FEED OUT BORING</td>
</tr>
<tr>
<td>G86</td>
<td>CANNED CYCLE # 6 FEED IN, SPINDLE STOP, RAPID OUT - BORING</td>
</tr>
<tr>
<td>G87</td>
<td>CANNED CYCLE # 7 FEED IN, SPINDLE STOP, MANUAL OUT</td>
</tr>
<tr>
<td>G88</td>
<td>CANNED CYCLE # 8 FEED IN, DWELL, SPINDLE STOP, MANUAL OUT</td>
</tr>
<tr>
<td>G89</td>
<td>CANNED CYCLE # 9 FEED IN, DWELL, FEED OUT, - BORING</td>
</tr>
<tr>
<td>G73</td>
<td>CANNED CYCLE # 10 HIGH SPEED PECK CYCLE</td>
</tr>
<tr>
<td>G74</td>
<td>CANNED CYCLE # 11 COUNTER TAPPING ( PITCH * RPM * .98 = FEEDRATE )</td>
</tr>
<tr>
<td>G76</td>
<td>CANNED CYCLE # 12 FINE BORING CYCLE</td>
</tr>
<tr>
<td>G80</td>
<td>CANNED CYCLE CANCEL</td>
</tr>
<tr>
<td>G90</td>
<td>ABSOLUTE PROGRAMMING ( USES ABSOLUTE ZERO )</td>
</tr>
<tr>
<td>G91</td>
<td>INCREMENTAL PROGRAMMING ( POINT TO POINT )</td>
</tr>
<tr>
<td>G92</td>
<td>ABSOLUTE ZERO POINT PRESET ( PART ZERO )</td>
</tr>
<tr>
<td>G94</td>
<td>FEEDRATE PER MINUTE</td>
</tr>
<tr>
<td>G95</td>
<td>FEEDRATE PER REVOLUTION</td>
</tr>
<tr>
<td>G98</td>
<td>RETURN TO INITIAL LEVEL IN CANNED CYCLE</td>
</tr>
<tr>
<td>G99</td>
<td>RETURN TO R-LEVEL IN CANNED CYCLE</td>
</tr>
</tbody>
</table>

* - ONE SHOT CODE
1. ONLY G-CODES FROM THE SAME GROUP CAN CANCEL EACH OTHER
2. ONLY 1 G-CODE FROM THE SAME GROUP IN A LINE
### M Codes

<table>
<thead>
<tr>
<th>GROUP</th>
<th>DESCRIPTION</th>
<th>USES</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>PROGRAM STOP, STOPS SPINDLE, COOLANT</td>
<td>SHIFT OR FLIP PART</td>
</tr>
<tr>
<td>M01</td>
<td>OPTIONAL STOP, STOPS SPINDLE AND COOLANT</td>
<td>BEFORE TOOL CHANGES</td>
</tr>
<tr>
<td>M02</td>
<td>END OF PROGRAM</td>
<td>LOOP TAPE ONLY</td>
</tr>
<tr>
<td>M03</td>
<td>SPINDLE CLOCKWISE</td>
<td>TURNS ON SPINDLE</td>
</tr>
<tr>
<td>M04</td>
<td>SPINDLE CCW</td>
<td>TURNS ON SPINDLE</td>
</tr>
<tr>
<td>M05</td>
<td>SPINDLE STOP</td>
<td>SEE M19 NOTE</td>
</tr>
<tr>
<td>M06</td>
<td>TOOL CHANGE</td>
<td>ONLY AT ATC POSITION</td>
</tr>
<tr>
<td>M07</td>
<td>OIL MIST</td>
<td>OPTIONAL FUNCTION</td>
</tr>
<tr>
<td>M08</td>
<td>COOLANT ON</td>
<td>FLOOD COOLANT</td>
</tr>
<tr>
<td>M09</td>
<td>COOLANT OFF</td>
<td>ALSO TURNS OFF M07</td>
</tr>
<tr>
<td>M18</td>
<td>NEUTRAL</td>
<td>FOR GEAR DRIVEN HEADS</td>
</tr>
<tr>
<td>M19</td>
<td>SPINDLE ORIENT</td>
<td>WILL SHUT OFF SPINDLE USED W/ZERO RET Z AXIS</td>
</tr>
<tr>
<td>M20</td>
<td>SPINDLE ORIENT OFF</td>
<td>USUALLY FOR OPERATOR USE ONLY</td>
</tr>
<tr>
<td>M21-28</td>
<td>OPTIONAL M-FUNCTIONS</td>
<td>STANDARD</td>
</tr>
<tr>
<td>M29</td>
<td>TOOL POCKET VERTICAL POSITION</td>
<td>DOUBLE ARM MACHINES ONLY</td>
</tr>
<tr>
<td>M30</td>
<td>END OF PROGRAM</td>
<td>USED AT END OF MEMORY</td>
</tr>
<tr>
<td>M42</td>
<td>LOW RANGE SPEED RUN IN HIGH GEAR</td>
<td>TAPPING, YASNAC ONLY</td>
</tr>
<tr>
<td>M46</td>
<td>CLEARS TOOL POCKET AT END OF PROGRAM</td>
<td></td>
</tr>
</tbody>
</table>
## M Codes

### Table of M Codes

<table>
<thead>
<tr>
<th>GROUP</th>
<th>DESCRIPTION</th>
<th>USES</th>
</tr>
</thead>
<tbody>
<tr>
<td>M48</td>
<td>FEEDRATE OVERRIDE EFFECTIVE</td>
<td></td>
</tr>
</tbody>
</table>
| M49   | FEEDRATE OVERRIDE CANCEL  
LOCK FEEDRATE AT 100 % |               |
| M73   | X - AXIS MIRROR IMAGE ON   | FANUC ONLY    |
| M74   | Y - AXIS MIRROR IMAGE ON   | FANUC ONLY    |
| M76   | 4TH AXIS MIRROR IMAGE ON   | FANUC ONLY    |
| M77   | MIRROR IMAGE CANCEL        | FANUC ONLY    |
| M94   | MIRROR IMAGE OFF           | YASNAC ONLY   |
| M95   | MIRROR IMAGE ON            | YASNAC ONLY   |
| M98   | JUMP COMMAND INTO SUB-     | MULTIPLE OPERATIONS |
|       | PROGRAM                   |               |
| M99   | JUMP BACK INTO MAIN PROGRAM |               |
056432; (NINE ERROR PROGRAM)
N1 G0 G20 G90 G40 G80
N5 T1 M6 (.5 DIA RH CUT DRILL)
N10 S1000 M4 M8
N20 G43 Z.1 H0
N25 G82 X1.0 R.1 Z-.3 P100 F50
N30 X3.
N35 X6.
N40 X9
N45 G0 G28 G91 Z0 M
N50 G28 X0 Y0 M0
N55 M10

LIST THE ERRORS FOUND BELOW AND CORRECTION.

<table>
<thead>
<tr>
<th>ERROR</th>
<th>CORRECTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td></td>
</tr>
<tr>
<td>2.</td>
<td></td>
</tr>
<tr>
<td>3.</td>
<td></td>
</tr>
<tr>
<td>4.</td>
<td></td>
</tr>
<tr>
<td>5.</td>
<td></td>
</tr>
<tr>
<td>6.</td>
<td></td>
</tr>
<tr>
<td>7.</td>
<td></td>
</tr>
<tr>
<td>8.</td>
<td></td>
</tr>
<tr>
<td>9.</td>
<td></td>
</tr>
</tbody>
</table>

# Canned Z Axis Cycles

<table>
<thead>
<tr>
<th>G #</th>
<th>PLUNGING</th>
<th>AT HOLE BOT</th>
<th>RETRACTION</th>
<th>APPLICATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>G73</td>
<td>WOODPECKER FEED</td>
<td>_______</td>
<td>RAPID TRAVERSE</td>
<td>HIGH SPEED DEEP HOLE DRILLING</td>
</tr>
<tr>
<td>G74</td>
<td>FEED</td>
<td>SPINDLE FWD AFTER Dwell</td>
<td>SPINDLE REVERSE AFTER FEED</td>
<td>REVERSE TAPPING</td>
</tr>
<tr>
<td>G76</td>
<td>FEED</td>
<td>SPINDLE INDEX-SHIFT</td>
<td>RAPID TRAVERSE SHIFT SP. START</td>
<td>BORING</td>
</tr>
<tr>
<td>G77</td>
<td>SPINDLE INDEX RAPID</td>
<td>Dwell</td>
<td>RAPID TRAVERSE SPINDLE INDEX</td>
<td>BACK BORING</td>
</tr>
<tr>
<td>G80</td>
<td>_______</td>
<td>_______</td>
<td>_______</td>
<td>CANCEL</td>
</tr>
<tr>
<td>G81</td>
<td>FEED</td>
<td>_______</td>
<td>RAPID TRAVERSE</td>
<td>DRILLING</td>
</tr>
<tr>
<td>G82</td>
<td>FEED</td>
<td>Dwell</td>
<td>RAPID TRAVERSE</td>
<td>SPOT, FACING</td>
</tr>
<tr>
<td>G83</td>
<td>WOODPECKER</td>
<td>_______</td>
<td>RAPID TRAVERSE</td>
<td>DEEP HOLE DRILLING</td>
</tr>
<tr>
<td>G84</td>
<td>FEED</td>
<td>SPINDLE REV AFTER Dwell</td>
<td>SPINDLE FWD RUN AFTER FEED</td>
<td>TAPPING</td>
</tr>
<tr>
<td>G85</td>
<td>FEED</td>
<td>_______</td>
<td>FEED</td>
<td>BORING</td>
</tr>
<tr>
<td>G86</td>
<td>FEED</td>
<td>SPINDLE STOP</td>
<td>RAPID TRAVERSE SPINDLE START</td>
<td>BORING</td>
</tr>
<tr>
<td>G87</td>
<td>FEED</td>
<td>SPINDLE STOP</td>
<td>MANUAL RETRACT SPINDLE START</td>
<td>BORING</td>
</tr>
<tr>
<td>G88</td>
<td>FEED</td>
<td>SPINDLE STOP AFTER Dwell</td>
<td>MANUAL RETRACT SPINDLE START</td>
<td>BORING</td>
</tr>
<tr>
<td>G89</td>
<td>FEED</td>
<td>Dwell</td>
<td>FEED</td>
<td>BORING</td>
</tr>
</tbody>
</table>
NOTE: DRILL MUST CLEAR PROJECTIONS BETWEEN HOLES TWO AND THREE AND FOUR AND FIVE.

O2255
N1 G90 G40 G80 G20
N5 T1 M6
N10 S3000 M3
N15 G54 X1.0 Y0
N20 G43 Z 2. H01

N30 G98 X2.
N35 G99 X3.
N40 G98 X4.
N45 G99 X5.
N50 X6.
N55 X7.
N45 G80 M9
N50 G91 G28 Z0 G0
N55 G28 X0 Y0
N60 M30
G & M Code Questions

1. Which three G Codes have to do with tool radius compensation?
   A] ________________
   B] ________________
   C] ________________

2. Which G Code cancels any canned Z axis cycle?
   A] ________________

3. Which reference plane is set in the program by a rapid (G00) command followed by a Z axis position?
   A] ________________

4. Which three G Codes have to do with tool length compensation?
   A] ________________
   B] ________________
   C] ________________
G & Mode Code Questions

5. Which G code is the standard start-up condition for feed rates on CNC milling machines?
   A] ________________

6. Which G codes could cancel a G01 command?
   A] ________________
   B] ________________
   C] ________________

7. What is a one-shot G code?
   A] ________________

8. How many G codes are in Group #9?
   A] ________________
G & Mode Code Questions

9. How many different group of G codes are there?
   A] ________________

10. A M09 cancels which M codes?
    A] ________________
    B] ________________

11. Explain what happens when a M01 is commanded.
    A] ________________
    ________________
    ________________

12. Which M code would be used for left-hand cut cutting tools?
    A] ________________
Section Ten

Your Notes:
During this two hour segment, participants will:

1. Understand components of the Niigata CNC horizontal machine center
2. Understand main and sub operating panels
3. Know how to input tool length offsets
4. Know how to zero return the axes to machine home
5. Understand MDI operations
6. Understand program entry and editing
7. Know how to use soft keys for program manipulation
8. Understand operation of APC and ATC units on machines
9. Know how to recover from a feed hold (example: tool breakage)
10. Understand procedure to mount and dismount tools in spindle and tool magazine

more on next page
11. Understand use of screen displays

12. Know where to look for typical alarm codes and how to explain the reasons for alarms

13. Recognize a sub-program which is used to make the machine go to a safe home position prior to starting the main program.
## Machine Specifications

### Niigata CNC Horizontal M.C. 

### MACHINE SPECIFICATIONS

<table>
<thead>
<tr>
<th>Item</th>
<th>Metric</th>
<th>Inch</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>TRAVEL &amp; WORK CAPACITY</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X axis column travel</td>
<td>610 mm (330+280)</td>
<td>24.0” (13.0”+11.0”)</td>
</tr>
<tr>
<td>Y axis vertical head travel</td>
<td>500 mm</td>
<td>19.7”</td>
</tr>
<tr>
<td>Z axis table travel</td>
<td>560 mm</td>
<td>22.0”</td>
</tr>
<tr>
<td>Spindle center line to pallet surface</td>
<td>50-550 mm [150-650 mm]</td>
<td>1.97-21.6” [5.9”-25.6”]</td>
</tr>
<tr>
<td>Spindle nose to table center line</td>
<td>150-710 mm</td>
<td>5.9-27.9”</td>
</tr>
<tr>
<td><strong>TABLE</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Table working surface</td>
<td>400 x 400 mm</td>
<td>15.7” x 15.7”</td>
</tr>
<tr>
<td>Table increments</td>
<td>1” [0.001”]</td>
<td>1” [0.001”]</td>
</tr>
<tr>
<td>Maximum load on pallet</td>
<td>400kg</td>
<td>880 lbs.</td>
</tr>
<tr>
<td><strong>SPINDLE</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spindle drive motor</td>
<td>AC11/7.5kw</td>
<td>AC15/10 HP</td>
</tr>
<tr>
<td>Spindle speeds</td>
<td>12,000min-”1” (rpm)</td>
<td>12,000 rpm</td>
</tr>
<tr>
<td>Spindle max. torque</td>
<td>292 N-m</td>
<td>215 ft-lbs.</td>
</tr>
<tr>
<td>Spindle taper</td>
<td>No. 40</td>
<td>No. 40</td>
</tr>
<tr>
<td><strong>FEEDRATE</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rapid traverse X-Y-Z</td>
<td>24 m/min.</td>
<td>945 ipm</td>
</tr>
<tr>
<td>Cutting X-Y-Z</td>
<td>1-15,000 mm/min/min.</td>
<td>0.04-590 ipm</td>
</tr>
<tr>
<td>Table Index speed/90°</td>
<td>2.5 sec.</td>
<td>2.5 sec.</td>
</tr>
<tr>
<td><strong>AUTOMATIC TOOL CHANGER (ATC)</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool magazine capacity</td>
<td>30 [40/60/90/120/185]</td>
<td>30 [40/60/90/120/185]</td>
</tr>
<tr>
<td>Tool selection</td>
<td>Short cut random</td>
<td>Short cut random</td>
</tr>
<tr>
<td>Tool shank</td>
<td>BT40</td>
<td>CT40</td>
</tr>
<tr>
<td>Maximum tool length</td>
<td>350 mm</td>
<td>13.8”</td>
</tr>
<tr>
<td>Maximum milling cutter dia.</td>
<td>95 mm</td>
<td>3.7”</td>
</tr>
<tr>
<td><em>(adjacent pockets empty)</em></td>
<td>200 mm</td>
<td>7.9”</td>
</tr>
<tr>
<td>Maximum tool mass</td>
<td>10kg</td>
<td>22 lbs.</td>
</tr>
<tr>
<td>Tool change time (tool to tool)</td>
<td>1.4 sec.</td>
<td>1.4 sec.</td>
</tr>
<tr>
<td><strong>AUTOMATIC PALLET CHANGER (APC)</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Type</td>
<td>Direct rotary</td>
<td>Direct rotary</td>
</tr>
<tr>
<td>Pallet change time</td>
<td>5.5 sec.</td>
<td>5.5 sec.</td>
</tr>
<tr>
<td>Number of pallet</td>
<td>2 [6/8/10/12]</td>
<td>2 [6/8/10/12]</td>
</tr>
<tr>
<td><strong>ACCURACY</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Positioning/full stroke X-Y-Z</td>
<td>±0.003 mm</td>
<td>±0.00012”</td>
</tr>
<tr>
<td>Positioning with scales</td>
<td>±0.002 mm</td>
<td>±0.00008”</td>
</tr>
<tr>
<td>Repeatability X-Y-Z</td>
<td>±0.0015 mm</td>
<td>±0.00006”</td>
</tr>
<tr>
<td>Repeatability with scales</td>
<td>±0.001 mm</td>
<td>±0.00004”</td>
</tr>
<tr>
<td>Table index</td>
<td>±2.5 sec.</td>
<td>±2.5 sec.</td>
</tr>
<tr>
<td><strong>GENERAL</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Base machine mass approx.</td>
<td>9,000 kg</td>
<td>20,000 lbs.</td>
</tr>
<tr>
<td>Machine space W/D/H approx.</td>
<td>2,460/3,700/2,467 mm</td>
<td>97”/146”/97”</td>
</tr>
<tr>
<td>Floor to table surface</td>
<td>1,016 mm</td>
<td>40”</td>
</tr>
<tr>
<td>Power</td>
<td>45 KVA</td>
<td>45 KVA</td>
</tr>
</tbody>
</table>

* General specifications
* Product specifications, accessories and machine appearance are subject to change without notice.
Niigata CNC Horizontal M.C.
### STANDARD

- Simultaneous controllable axes: Three axes (X, Y, Z)
- Least input increment: 0.001 mm (0.0001")
- Least command increment: 0.001 mm
- Linear interpolation G01
- Multi-quadrant circular interpolation
- Feed rate command: Direct designation of mm/min or inch/min feed.
- Feed rate override: 0 - 200% (10% increments)
- Tangential speed constant control
- Cutting feedrate clamp
- Override cancel
- Automatic acceleration/deceleration
- Exact stop/cutting mode G00, G01, G03, G04
- Dwell
- Reference point return: Manual, automatic (G27, G28, G29)
- Machine coordinate system selection: G53
- Work coordinate system selection: 6 types (G54-G59)
- Local coordinate system setting: G52
- Work coordinate system change: G92
- Absolute/incremental programming
- Decimal point input
- Program number search
- Program number: 4 digits/Program name: 16 characters
- Sequence number search
- Main program/subprogram Subprogram/4 folds nested
- Tape code: EIA, RS232A, ISO-840 automatic recognition
- Label skip
- Control input
- Optional blocks skip
- ISO code input
- Tool offset selection by T code 99
- Radius designation on arc
- 2nd auxiliary function
- Tool length compensation G43, G44
- Tool offset G43-G48
- Cutter compensation C G40-G42
- Tool offset amount memory A:
- ±6 digits, common to all tools, 32 pairs
- Backlash compensation
- Editing during automatic operation
- Registerable programs:
- TC0 (Program name display is possible)
- Part program storage length 80m (2520)
- Reader/Puncher interface:
- RS232 x 1, RS422 x 1, RS232C or 20mA current loop x 1
- Keyboard type MD/CRT character display:
- 9 monochrome CRT
- Data protection key:
- 3 types
- Self-diagnosis function
- Follow-up:
- At emergency stop
- Servo off
- Buffer register
- Manual continuous feed
- Incremental feed
- Manual absolute on/off
- All axis Machine lock
- Machine lock on each axis
- Auxiliary function lock
- Dry run
- Single block
- Over travel
- Stored stroke check 1
- Interrupt
- Stored pitch error compensation
- Clock function
- 14" color CRT
- Part program storage length 320, 640, 1280, 2560, 5120m
- Tool offset selection by T code 200, 499, 99
- Tool offset amount memory C
- Registered program:
- 200, 400, 1000
- External data input/output
- Humid interpolation
- Single direction positioning
- Conversational automatic programming function incl. 14 inch CRT and Graphic display
- Automatic corner override
- Handy line
- Sequence No. comparison and stop
- Tool life management
- Program restart
- High speed skip signal input
- Stored stroke check 2
- Tool length measurement
- Programmable data input G10L1
- Scaling
- Linear acceleration before cutting feed interpolation
- Polar coordinate command
- Optional angle chamfering corner R
- Programmable mirror image
- Interruption type cutout macro
- Coordinate system rotation
- Additional axes:
- Max. 5 axes
- Simultaneously controllable axes expansion:
- Max. 5 axes
- RUN hour display
- Background editing:
- Editing during automatic operation
TURN ON PROCEDURE OF CONTROL

2. Turning ON Procedure of Control Power

1. Turn ON primary power to the machine.

2. Turn ON Circuit Breaker.

3. Turn ON Control Power of NC Unit.

Ten seconds later after power ON, Position Screen is displayed.

4. Turn ON Control Power of Machine Operation Panel.

Hydraulic pump is started, and Power Indicating Lamp is lit.

Note-1: If Hydraulic pump is not started despite Control Power button is depressed, check whether ENG Button of either ATC, APC Operation Panel is kept ON. If kept ON, click ENG button ON direction light for reset.

Note-2: Hydraulic pump is actuated during Control Power button is depressed, and deactuated and stopped immediately after depressing:

Either of X, Y and Z-axis has possibility of Overrun (2nd Overrun).

Recovery procedure:

Keep depressing Control Power Button ON, and return overrun axis its inside of stroke range by MPG Handle.

Then depress NC RESET button.
NIIGATA CNC HMC

TURN OFF PROCEDURE OF CONTROL

3. Turning OFF Procedure of Control Power

①. Considering about machine balance and easier zero return operation, move each axis to its stroke center area as long as each axis has no interference with workpiece on the pallet.

Mode selection switch “RAPID” is selected. — Axis moving by +1 or –1 button (Refer to P. Rapid Traverse in details.)

②. Depress Mushroom type EMG button.

Lamp for R CONTROL ON J indicating is turned OFF, and hydraulic pump is stopped.

Turn EMG button CW direction little light for button reset.

③. Depress OFF button of NC-unit.

Screen display is turned OFF.

④. Turn OFF Circuit Breaker of Main Control Cabinet.

⑤. Turn OFF the power to the machine.
NIIGATA CNC HMC

MAIN OPERATING PANEL

PN 40 Operating Panel

Main Operating Panel

[Diagram of the main operating panel with various controls labeled]
MAIN OPERATING PANEL

NAME THE CONTROL FEATURES

1. ____________________________  15. ____________________________
2. ____________________________  16. ____________________________
3. ____________________________  17. ____________________________
4. ____________________________  18. ____________________________
5. ____________________________  19. ____________________________
6. ____________________________  20. ____________________________
7. ____________________________  21. ____________________________
8. ____________________________  22. ____________________________
9. ____________________________  23. ____________________________
10. ____________________________  24. ____________________________
11. ____________________________  25. ____________________________
12. ____________________________  26. ____________________________
13. ____________________________  27. ____________________________
14. ____________________________
NIIGATA CNC HMC

ATC MANUAL OPERATING PANEL
APC OPERATING PANEL
MANUAL HANDLE OPERATING PANEL

ATC Manual Operating Panel

APC Operating Panel

Manual HANDLE Operating Panel
NIIGATA CNC HMC

EXPLAIN THE FUNCTION OF EACH FEATURE

MANUAL HANDLE

MPG (1)

MULTIPLY (2)

SPINDLE COLLET (3)

WORKLAMP (4)
EXPLAIN THE FUNCTION OF EACH FEATURE

AUTOMATIC PALLET CHANGER

EMG (1)

CYCLE START (2)

FEED HOLD (3)

PALLET READY (4)

LIFTER UP (5)
EXPLAIN THE FUNCTION OF EACH FEATURE

AUTOMATIC TOOL CHANGER

EMG (1) ________________

MAN OPER (2) ________________

MAGAZINE CW (3) ________________

MAGAZINE CCW (4) ________________
MOVING THE AXES USING RAPID

5. Rapid Traverse. JOG Feed & Manual Pulse Generator (MPG)

5-1. Rapid Traverse

Each axis is moved to any desired position by rapid traverse.

1. Depress "RAPID" button of mode section switch.

2. Select desired axis to be moved on AXIS selector.

3. Depress "+" or "-" button of MAN FEED.

Axis moves during button depressing.
Moving speed is set by Rapid Override.

Note:
If an axis moves out of stroke limit (overrun) accidentally, move the axis to be in the stroke range.

If "CONTROL ON" indicating lamp is turned OFF due to overrun, recover with the following procedures.

i). Depress "CONTROL ON" button and keep depressing it.

ii). Move overrun axis to the opposite direction by MPG handle.
(Refer to Item 5-3. Manual Pulse Generator / P.8)

iii). Keep OFF from "CONTROL ON" button.

iv). Depress RESET button of NC unit. (OVERRUN indication signal is turned OFF on the screen.)
NIIGATA CNC HMC

MOVING THE AXES USING JOG OR MPG

5-2. JOG Feed

Each axis can be moved by set feed for JOG Feed manually.

JOG FEED RATE mm/min

100
27.2
1.4
1

Set jog feed rate to desired speed.

Select desired axis to be moved on AXIS selector.

Depress the JOG button of mode selector.

5-3. Manual Pulse Generator (MPG) / Handle feed

Each axis can be moved to desired position by Handle (Manual Pulse Generator: MPG).

Select MPG position of axis selector.

Depress the MPG button of mode selector.

Select desired axis on AXIS selector of MPG operation box right.

Set Desired position of MULTIPLY switch of MPG operation box right to either X1, X10, X100 or X1000.

X1 = 0.001 mm per 1DIV.
X10 = 0.1 mm per 1DIV.
X100 = 0.1 mm per 1DIV.
X1000 = Same as X100
INPUT OF TOOL OFFSETS

3. Tool Offset

Tool Length and Tool Dia. can be input to display on CRT screen for Tool Length Offset and Tool Dia. Offset compensation.

① Display OFFSET screen to CRT as follows.

Depress function MENU key located at bottom left to display 'OFFSET'.

② Depress OFFSET soft key. Then the following screen is displayed.
If not displayed, depress the same OFFSET key few more times.

<table>
<thead>
<tr>
<th>TOOL OFFSET</th>
<th>TOOL OFFSET</th>
</tr>
</thead>
<tbody>
<tr>
<td>NO. DATA NO. DATA</td>
<td>NO. GEOMETRY WEAR GEOMETRY WEAR</td>
</tr>
<tr>
<td>001 12.500 011 0.000</td>
<td>001 12.354 3.624 12.000 -3.623</td>
</tr>
<tr>
<td>002 23.400 012 0.000</td>
<td>002 0.000 0.000 0.000 0.000</td>
</tr>
<tr>
<td>003 15.300 013 1.234</td>
<td>003 0.000 0.000 0.000 0.000</td>
</tr>
<tr>
<td>004 0.000 014 3.456</td>
<td>004 <strong>MEM</strong> 0.000 0.000 0.000</td>
</tr>
<tr>
<td>005 <strong>MEM</strong> 015 0.000</td>
<td>005 0.000 0.000 0.000 0.000</td>
</tr>
<tr>
<td>006 60.000 016 0.000</td>
<td>006 0.000 0.000 0.000 0.000</td>
</tr>
<tr>
<td>007 4.250 017 0.000</td>
<td>007 0.000 0.000 0.000 0.000</td>
</tr>
<tr>
<td>008 0.000 018 -1.200</td>
<td>008 0.000 0.000 0.000 0.000</td>
</tr>
<tr>
<td>009 0.000 019 -4.300</td>
<td>009 0.000 0.000 0.000 0.000</td>
</tr>
<tr>
<td>010 0.000 020 0.000</td>
<td>010 0.000 0.000 0.000 0.000</td>
</tr>
</tbody>
</table>

Standard screen

Screen with Offset Memory 'C'( OP.)

③ Move Cursor to desired position using PAGE keys (↑ ↓ ) and CURSOR move keys (← → ).

④ Key in desired numeric value as shown below.

⑤ Depress 'INPUT' soft key. Offset value is input or altered.
**NIIGATA CNC HMC**

**ADDING TO OFFSET VALUE**

### REFERENCE

1. When Offset value is additionally added to the present offset value:

   1. Move cursor to desired position and key in additional value.

   ![Screen 1](image1.png)

   - **TOOL OFFSET**
     - NO. DATA
     - 001  0.000
     - 002  150.000
     - 003  0.000
   
   2. Depress '+ INPUT' soft key, and above additional value is added.

   ![Screen 2](image2.png)

   - **TOOL OFFSET**
     - NO. DATA
     - 001  0.000
     - 002  150.000
     - 003  0.000
   
   When '+INPUT' as -0.2, offset value is reduced to 149.800 from 150.000.
NIIGATA CNC HMC

TOOL OFFSET MEMORY " C "

2). Tool Offset 'Memory 'C' ( Option ) :

Both of Tool Length Offset ( H-code ) and Tool Radius Offset ( D-code ) using the same Offset No. can be used. and also GEOMETRY and WEAR Offset can be set together as shown left.

<table>
<thead>
<tr>
<th>NO.</th>
<th>GEOMETRY WEAR</th>
<th>GEOMETRY WEAR</th>
</tr>
</thead>
<tbody>
<tr>
<td>001</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>002</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>003</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>004</td>
<td>100.000</td>
<td>0.000</td>
</tr>
<tr>
<td>005</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>006</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>007</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>008</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>009</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>010</td>
<td>0.000</td>
<td>0.000</td>
</tr>
</tbody>
</table>

| (MM) |

GEOMETRY : Tool geometry offset 
WEAR : Tool wear offset

H1 works as offset value : 150.0 + 0.2 = 150.2
D1 works as offset value : 20.0 + (-0.15) = 19.85
4. Zero Return Method (Reference Point Return)

After turning ON the power, Zero Return (Reference Point Return) must be required.

Zero Point (Reference Point) of each axis is PLUS (+) stroke end.

4-1. Zero Return of X, Y, Z and B-axis (NC-table: OP.)

○ When an axis is on Zero position or near by Zero position:

1. Depress "RAPID" button on Mode selector switches.

   Green lamp is lit.

2. Select desired axis (X, Y, Z or B) to move on axis selection switch.

3. Depress "-" button of axis movement.

   During keeping to depress, the axis keeps to move and stops by releasing.

   Above 2 & 3 operations are repeated about axes near by or on zero position.

   Moving distance: Approximately 100 mm

4. Depress "ZERO" button on mode selection switch.
NIIGATA CNC HMC

ZERO RETURN METHOD

- When an axis is 100 mm or more distance from its zero point:
  Execute from above item ① for zero return on required axis. Repeat the same procedure for another axis if any.

① Select desired axis to be returned.
  Z-AXIS MUST BE SELECTED 1ST to avoid tool and work or fixture interference. If either X, Y, or B-axis is selected, tool may interfere with work or fixture.

② Depress 「+」 button of axis movement.
  Selected axis moves to zero point in plus (+) direction, and decelerated when it reaches close to the zero point.
  Keep depressing the button until reaching this deceleration range.
  Zero position indicating lamp is lit for the axis in this operation.
  Repeat ② & ③ operation for each axis.
  Make sure zero return lamps are lit for X, Y, and Z-axis (B-axis : NC-table)

ZERO RETURN

④-2. Zero return of B-axis (Either 72 or 360 position index table)
  ① Depress 「ZERO」button on mode selection switch.
  ② Select 「B」on axis selector.
  ③ Depress 「+」button of axis movement.

Motion:
  Pallet UNCLAMP → 0° indexing → Pallet CLAMP
  (Lifted.) (Down)
NIIGATA CNC HMC

MDI OPERATION

6. MDI (Manual Data Input) Operation

When either motion of Spindle rotation, Tool change, Pallet change or simple axis movement is desired, motion program is input manually to NC unit and executed automatically.

1. Depress "MDI" button as shown right.

2. Check display of top left on the CRT screen.
   To display PROGRAM (MDI) on the screen:

   I). Depress left-function MENU key, and display PROGRAM as shown.

   II). Depress PROGRAM soft key. PROGRAM (MDI) is shown at left top corner of the screen. If not displayed, repeat to depress it again until displaying.

3. Key in desired command using address keys and numeric keys.

---

For more detailed information, please refer to the manual.
MDI OPERATION

Keyed in command is displayed on the corner left below of the screen as shown.

Ex. > T01; M06; _

2. Depress right function MENU key, and keep depressing until displaying 'INSERT'.

3. Depress INSERT soft key, and then the program is shifted to top side of the screen.

4. Depress CYCLE START button.

Commanded program is executed.
1. Program Editing

1) New program registration

①. Depress EDIT button on mode selection switches.

②. Display PROGRAM (MEMORY) screen.

③. Key in Program No.

④. Depress INSERT soft key as shown.

By depressing PROGRAM soft key, PROGRAM (MEMORY) is displayed on upper left of the screen. If not displayed, repeat to depress the soft key again.

⑤. New program No. is displayed on the screen and registered as new one.

⑥. Key in the following program and depress INSERT soft key.

Max. letters in one time are 78 letters. Keeping 78 letters or within, no block number limit is given. After INSERT, each block is separated automatically.
2). ALTER. INSERT and DELETE of registered program ( Mode : EDIT )

①. Alteration ( ALTER )

i ). Move cursor to the desired position to be altered.

ii ). Key in the word to be altered.

iii ). Depress ' ALTER ' soft key.

②. INSERT

i ). Move cursor to one word before inserted.

ii ). Key in the word to be inserted.

iii ). Depress ' INSERT ' soft key.
NIIGATA CNC HMC

PROGRAM EDITING - EXISTING PROGRAM

①. DELETE

①-1. DELETE of Word (DLT-WRD)

i). Move cursor to be deleted.

ii). Depress DLT-WRD (Delete Word) soft key.

①-2. DELETE of designated range (DELETE)

i). Move cursor to the top word to be deleted.

②. Key Program No. to be called and depress FR_SRCH or BR_SRCH soft key.

Keyed in Program is called.
12. Program Call

1. Depress 'EDIT' or 'MEMO' button on mode selection switch.

2. Display PROGRAM (MEMORY) screen.
   i) Depress Function MENU key left below to display 'PROGRAM' (Program).

   ii). Depressing 'PROGRAM' soft key, PROGRAM (MEMORY) is displayed on top left of the screen. If not displayed, repeat depressing.

3. Keep depressing Right MENU key until displaying of FW_SRC H or BW_SRC H.
13. Operation

1. Depress 'MENO' button on mode selection switch.

2. Check whether machining program is called or not.

3. Depress 'CYCLE START'.

Program is started automatically.
NIIGATA CNC HMC

PROGRAM CHECKING METHOD

★ Program checking method:

In case of new program, program mistakes might be existed somewhere in the program. Therefore, operate the cycle with 'SIN BLOCK' (Single Block) selection and reduce 'RAPID OVERRIDE' to 25% without work mounting.

In case no load operation, use 'DRY RUN' operation is recommended to shorter checking time, because all programmed feed rates in the program are switched to the designated feed rate by 'JOG FEED RATE' button.

CAUTION:

If program is not started despite 'CYCLE START' button is depressed, check the following items.

1. Check 'SPINDLE' selector switch. If 'MAN. ROT' is selected, CYCLE is not started. 'AUTO' position must be selected in advance.
9-2. In case of PN 40

1). Manual ATC operation

①. Execute ZERO return (Reference Point Return) by MAN or MDI operation.

②. Execute spindle orientation stop and ATC position return for X-axis (2nd Reference Point Return) by MDI operation.

\[
\begin{align*}
M19 & : \\
G91 & G30 X0 \\
\end{align*}
\]

③. Select 'MAN' side (either one of RAPID JOG, MPG or ZERO) in mode selection switch.

④. Depress 'MAG' button and select magazine No. of new tool by depressing either FOR or REV button.

FOR : Tool magazine forward (CW)  
REV : Tool magazine reverse (CCW)
NIIGATA CNC HMC

PN - 40 MANUAL ATC OPERATION

③. Depress 'PICK UP' button and keep depressing 'FOR' button.

New tool is picked up from the magazine and moves to stand by position.

In this case, stand-by pot must be at the position as Fig.-A shown below.
When stand-by pot is positioned as Fig.-B below, 'PICK UP' motion does not work, and DGN No. '7 5 / ATC OTHERS' is displayed.
To recover it, select empty pot desired (if not designated, select any empty pot) in above Item-② operation by keeping to depress 'REV' button.

![Fig.-A](image1)
![Fig.-B](image2)

④. Depress 'ARM' button and keep depressing 'FOR' button.

Swing arm is moved to spindle side, and ATC shutter is opened at the same time.

⑤. Depress 'CHANGE' button and keep depressing 'FOR' button.

Tool-change is executed (Old Tool is picked up from the spindle and New Tool is inserted to the spindle).
By depressing 'REV' button and keep depressing 'FOR' button, reverse motion is possible (New Tool is picked up from the spindle and Old Tool is inserted to the spindle).

⑥. Return X-axis to zero position (reference point) by MDI operation.

G 91 G 28 X 0 ;

⑦. Select 'MAN' mode.

⑧. Select the Magazine No. for old tool to be stored in Item-⑧ operation.

⑨. Depress 'ARM' button and keep depressing 'REV' button.

Swing arm is returned to magazine and ATC shutter is closed.

⑩. Depress 'PICK UP' button and keep depressing 'REV' button.

Old Tool is stored into the magazine pot.

CAUTION: When FOR or REV button is released and turned OFF during in motion, the motion is stopped immediately.
Motion executing can be checked on MACHINE screen whether each motion is completed or not.
15-2. In case of PN40

1). ATC (Automatic Tool Changer):

If EMG situation is taken place in mid-motion, basically recovery is possible by MAN.(Manual) button operation.
However, Swing Arm can not be followed to the correct position sequence (Magazine side or Spindle side) depending on mid-motion, for example:

- Tool pickin up or storing motion at Tool Magazine side;
- ATC motion at spindle side;

In this case, check EMG stop button is depressed and kept 1st. and move the swing arm manually (by hands) to either magazine side or spindle side.
Then turn ON the control power again.

1)-1. In case of EMG stop in tool changing motion:

1. NC POWER button ON —— In case of power black out.
2. Control Power button ON.

After above operation, ALARM Lamp on Main Operation Panel is lit, and ‘OT 007 X+ OVERTRAVEL (HARD)’ is displayed on DGN of MACHINE screen. This alarm is appeared due to spindle orientation stop OFF by EMG stop. Therefore, ATC recovery motion can be executed with this alarm.

3. Select MAN side (either one of RAPID, JOG, MPG or ZERO) in mode selecting switch.
4. Depress ‘CHANGE’ button and keep depressing ‘FOR’ button until motion end.
   ‘REV’ button does not work in this case.

**WARNING**: NEVER TRY TO ROTATE SPINDLE WITHOUT A TOOL ON SPINDLE.

When machine is stopped in tool retracted position (no tool on spindle), NEVER TRY TO ROTATE SPINDLE, because spindle orientation is OFF.
If accidentally rotated, orient the spindle to the correct position (Drive key of the spindle is parallel with X-axis: Horizontal)

If spindle orientation operation recovery is not possible or not sure, take off drive keys to avoid tool interference with the keys.

5. Move X-axis to minus direction away from ZERO position enough and depress RESET button ON for ALARM OFF.
6. Depress ‘MAG’ (Magazine) button and select desired magazine (tool pot) No. by depressing either ‘FOR’ or ‘REV’ button.
7. Depress ‘ARM’ button and keep depressing ‘REV’ button until motion end.

Swing arm moves to magazine side and ATC shutter is closed at the same time.
NIIGATA CNC HMC

RECOVERY OF ATC

1. Depress 'PICK UP' button and keep depressing 'REV' button until motion end.
   Tool is stored to tool magazine selected.

2. 2 APC

Recovery motion by MAN operation is possible for all mid-motions for 2 APC.
2). Manual APC operation

①. Manual APC operation can be executed by selecting either 'MAN' or MDI operation.

Condition:
- Z-axis zero position (Reference Point): G91 G28 Z0 ;
- B-axis zero degree

②. Select 'MAN' side (either one of RAPID JOG, MPG or ZERO) in mode selection switch.

③. Depress 'APC' button and keep depressing either 'FOR' or 'REV' button. The following motion is executed.

Pallet UNCLAMP → Pallet index (APC shutter is indexed at the same time) → Pallet CLAMP

FOR button: For A-PALLET indexing
REV button: For B-PALLET indexing

When FOR or REV button is released and turned OFF during in motion, the motion is stopped immediately.

3). Manual NC-table CLAMP / UNCLAMP operation (In case of B-axis: NC table / OP.)

①. Select 'MAN' side (either one of RAPID JOG, MPG or ZERO) in mode selection switch.

②. Depress 'TABLE' button and depress either 'FOR' for clamp or 'REV' for unclamp.
NIIGATA CNC HMC

RECOVERY OF 2APC (2 PALLET APC)

2). 2 APC (2 Pallet type Automatic Pallet Changer)

Basically, APC motion is composed of 3 motions, i.e. LOADING, INDEXING and UNLOADING.

- Unloading motion: LOADER ADVANCE → PALLET UNCLAMP → LOADER RETRACTION
- Indexing motion: A/B-PALLET INDEXING
- Loading motion: LOADER ADVANCE → PALLET CLAMP → LOADER RETRACTION

In EMG stop situation, Automatic Pallet Changing condition is reset. Therefore, APC Loader takes NO advance motion and Retracting motion only is possible. To resume APC motion possible, the following recovery procedure must be taken first in any case.

1. NC POWER button ON. → In case of power black out.
2. CONTROL POWER button ON.
14. Recovery Procedure after FEED HOLD in Mid-operation

1). Program correction in mid-operation

When program mistake is found and its correction is required in memory operation, the following procedure must be taken.

①. Depress 'SIN BLOCK' or 'FEED HOLD' button to stop axis feed.
②. Depress NC RESET button in MEMO (Memory) mode.
③. Return Z-axis to Zero (reference point) position in 'ZERO' return mode. If required, return X and/or Y-axis also to Zero position.
④. Switch to 'EDIT' mode and correct the program.
⑤. In case of restart of the cycle, move cursor to the 1st program where the tool on spindle now, and depress 'CYCLE START' button.

**WARNING:**

WHEN THE CYCLE IS RESTARTED, EXECUTE WITH SINGLE BLOCK 1ST FOR SAFETY. RESTARTING POSITION IS DEPENDED ON STOPPED SITUATION IN MID-OPERATION.

**CAUTION:**

WHEN 'NC-RESET' BUTTON IS DEPRESSED, INFORMATION OF CANNED CYCLE & TOOL OFFSET ARE CANCELLED. THEREFORE, TO RESTART THE CYCLE, THESE COMMANDS MUST BE EXECUTED ONE MORE TIME IN ADVANCE.

2). Tool breakage in mid-operation

If a tool is broken during machining, operation is stopped by depressing 'FEED HOLD' or 'EMG' (Emergency) button.

2)-1. **FEED HOLD button is depressed:**

①. Stop spindle and check tool breakage and machined section.
   Eliminate broken tool on the work surface out of the machine.
②. Depress NC RESET button.
③. Execute ZERO return for each axis.
④. Change tool to new one.
⑤. Tool length setting for new tool.
⑥. Move cursor to the restarting position, and depress 'CYCLE START' button.
7-2. Tool mounting and dismounting on the Spindle

Tool mounting and dismounting on the spindle is executed.

①. Tool dismounting from spindle

ⅰ). Select 'UNCLAMP' side of 'SPINDLE COLLET CLAMP / UNCLAMP' switch.

ⅱ). Hold the tool and holder manually to prevent tool dropping.

ⅲ). Depress SPINDLE COLLET button at center side.

CAUTION: THIS OPERATION IS NOT POSSIBLE IN HN-8 SERIES IF M19 (SPINDLE ORIENTATION STOP) IS NOT EXECUTED YET.

②. Tool mounting to Spindle

ⅰ). Select 'CLAMP' side of 'SPINDLE COLLET CLAMP / UNCLAMP' switch.

ⅱ). Insert the tool and holder together to the spindle manually.

ⅲ). Depress SPINDLE COLLET button at center side.
7. Tool mounting and dismounting

7-1. Tool mounting and dismounting at Tool Magazine

Tool mounting & dismounting is executed at Tool Magazine located at machine left side.

1. Depress 'MAN. OPER.' switch of ATC Operation Box.

   - MAN. OPER. lamp is lit in Green.

2. Index a desired tool pot either CW or CCW direction.

3. When tool is dismounted, use Tool Dismounting kit shown in previous page (P.10). When mounting, just insert each tool and holder set the tool Pot.

4. After above operation finished, depress MAN. OPER. switch again.

   Green lamp is turned OFF.

CAUTION: IF THIS OPERATION IS NOT EXECUTED, ATC MOTION BY MEMORY OR MDI OPERATION CAN NOT BE EXECUTED.
NIIGATA CNC HMC

MACHINE SCREEN DISPLAY

16. MACHINE Screen
16-1. Display operation of MACHINE screen

1. Depress 'MC DISPLAY' button (MC DISPLAY button is lit with green).

MACHINE screen is displayed on CRT screen.

SCREEN 1.1 (MACHINE SCREEN, MAIN)

<table>
<thead>
<tr>
<th>TOOL #</th>
<th>TABLE</th>
<th>CONDITION</th>
</tr>
</thead>
<tbody>
<tr>
<td>SP</td>
<td>9999</td>
<td>B360DEG</td>
</tr>
<tr>
<td>GP</td>
<td>9999</td>
<td>APC POSITION</td>
</tr>
<tr>
<td>MG</td>
<td>9999</td>
<td>B CLAMP</td>
</tr>
<tr>
<td></td>
<td></td>
<td>P CLAMP</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SP ORIENT</td>
</tr>
<tr>
<td></td>
<td></td>
<td>C CLAMP</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>SPINDLE</th>
<th>LOADING</th>
<th>SPEED</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD</td>
<td>100 %</td>
<td>10000 RPM</td>
</tr>
</tbody>
</table>

GEAR: HIGH OVERRIDE: 100 %

DGN FNC NM24 CM4 MAPC +

( TOOL # ) ( TABLE )
SP : Tool No. on Spindle
GP : Tool No. on Gripper
MG : Tool No. of Magazine indexed

( SPINDLE )
LOAD : Spindle load meter (OP.)
SPEED : Spindle speed meter (OP.)

GEAR: HIGH -- Spindle gear OVERRIDE: 100 %
HIGH: High range
LOW: Low range

( CONDITION ) : Machine condition is displayed.
ATC POSITION: ATC tool change position
APC POSITION: APC pallet change position
B CLAMP: B-axis clamped
P CLAMP: Pallet clamped
SP ORIENT: Spindle orientation completed
C CLAMP: C-axis clamped (OP.)

2. Depress 'MC DISPLAY' button again to return to NC screen.
NIIGATA CNC HMC

MACHINE SCREEN DISPLAY

16-2. Spindle Override Setting (OP.)

<table>
<thead>
<tr>
<th>(SPINDLE)</th>
<th>SPEED</th>
</tr>
</thead>
<tbody>
<tr>
<td>LOAD 100 %</td>
<td>10000 RPM</td>
</tr>
<tr>
<td>GEAR: HIGH</td>
<td>OVERRIDE: 100 %</td>
</tr>
</tbody>
</table>

- Depress MENU key.
- Set override by +S or -S.
- Setting finished.

16-3. DGN ('Diagnosis') DISPLAY screen

1. Depress soft key 'DGN'.

In alarm situation, DGN No. and alarm unit are displayed on the screen.

SCREEN 2 (DGN DISPLAY)

<table>
<thead>
<tr>
<th>(DGN DISPLAY)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DGN#</td>
</tr>
<tr>
<td>25</td>
</tr>
<tr>
<td>30</td>
</tr>
<tr>
<td>100</td>
</tr>
<tr>
<td>125</td>
</tr>
</tbody>
</table>

2. Depressing MENU key left, screen display is returned to SCREEN 1.1 (P.32)
NIIGATA CNC HMC

MACHINE SCREEN DISPLAY

16-4. FUNC (Function) switch setting

1. Depress 'FUNC' soft key.

<table>
<thead>
<tr>
<th>RT</th>
<th>FUNC</th>
</tr>
</thead>
</table>

SCREEN 3 (FUNCTION SETTING)

<table>
<thead>
<tr>
<th>(FUNCTION SETTING)</th>
<th>ON-BLK SKIP 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>OVR CANCEL</td>
<td></td>
</tr>
<tr>
<td>ON-OP STOP</td>
<td>-BLCK SKIP 2</td>
</tr>
<tr>
<td>WC.LOCK</td>
<td>-BLCK SKIP 3</td>
</tr>
<tr>
<td>Z NEGLECT</td>
<td>-BLCK SKIP 4</td>
</tr>
<tr>
<td>AF.LOCK</td>
<td>-BLCK SKIP 5</td>
</tr>
<tr>
<td>DISP LOCK</td>
<td>-BLCK SKIP 6</td>
</tr>
<tr>
<td>MEMO LOCK</td>
<td>-BLCK SKIP 7</td>
</tr>
<tr>
<td>ON-ABS</td>
<td>-BLCK SKIP 8</td>
</tr>
<tr>
<td>AUTO &amp; MANU</td>
<td>-BLCK SKIP 9</td>
</tr>
<tr>
<td>HANDLE IR</td>
<td>ON-POWER OFF</td>
</tr>
<tr>
<td>PROG RESTART</td>
<td>-BLK LS</td>
</tr>
<tr>
<td>BLK RESTART</td>
<td>-AIR CUT 0</td>
</tr>
</tbody>
</table>

All functions including optional are displayed in this screen.

Refer to FANUC OPERATOR'S MANUAL and NIIGATA INSTRUCTION MANUAL in details for each function.

CAUTION: ABS function must be kept ON all the time.

<table>
<thead>
<tr>
<th>RT</th>
<th>ON</th>
<th>OFF</th>
</tr>
</thead>
</table>

Page 148 of 262

2. Move cursor to the desired item to be set using [↑][↓] keys.

3. Depress 'ON' or 'OFF' soft key for ON/OFF setting.

4. Depress 'RT' key to return to original MACHINE screen.
NIIGATA CNC HMC

MACHINE SCREEN DISPLAY

16-5. NM 24 Function screen (OP.): AOU, AMU, ATM etc.

1. Depress 'NM24' soft key.

<table>
<thead>
<tr>
<th>NM24</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
</tbody>
</table>

SCREEN 4 ( NM24 FUNCTION )

<table>
<thead>
<tr>
<th>MODEL</th>
<th>FUNCTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>AOU</td>
<td></td>
</tr>
</tbody>
</table>

CONTACT

<table>
<thead>
<tr>
<th>AIR ALARM</th>
<th>ALARM</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALARM</td>
<td>ALARM</td>
</tr>
</tbody>
</table>

ALARM NO: 22

RT

RESET

FUNCTION ALARM: In alarm situation of a function. ALARM signal is displayed.

ALARM NO: Alarm No. is displayed.

2. Depress either AIR ALARM or FUNCTION ALARM 'RESET' button.

3. Depress 'RT' key to return to original MACHINE screen.
16-6. CM 4 Function setting screen

1. Depress 'NM24' soft key.

2. Move cursor to the desired item and input set value (%).

3. Depress 'RT' key to return to original MACHINE screen.

16-7. Program No. setting screen of APC (OP.)

1. Depress 'M APC' soft key.

PRG #: Program No.
SEQ #: Pallet change sequence

[In case of 6 APC]
NIIGATA CNC HMC

MACHINE SCREEN DISPLAY

( APC PROGRAM 4 )

NO  PRG 
A : 1001  
B : 1002

[ IN CASE OF 2 APC ]

RT

① Move cursor to the desired position and input set value by numeric keys.
③ Depress 'RT' key to return to original MACHINE screen.

Note-1 : Refer to 'FANUC Series 15-MA OPERATOR'S MANUAL ( B-61224E )' for more details on MACHINE screen.

Note-2 : Items with ( OP.) means optional specifications. Refer to each machine specification.
### Alarm Message Display

#### 9 Trouble Diagnosis List (DGN Table)

<table>
<thead>
<tr>
<th>DGN No</th>
<th>DGN</th>
<th>Contents</th>
<th>Normal</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>DGN</td>
<td>Spindle stop despite of spindle rotation command ON. (Reset: Select MAN mode.)</td>
<td>R44.2 OFF</td>
</tr>
<tr>
<td>2</td>
<td>DGN</td>
<td>Spindle drive unit signal confirmation error</td>
<td>X6.0 ON</td>
</tr>
<tr>
<td>3</td>
<td>DGN</td>
<td>Spindle drive unit signal confirmation error</td>
<td>X6.1 ON</td>
</tr>
<tr>
<td>4</td>
<td>DGN</td>
<td>Spindle drive unit signal return error</td>
<td>X6.0 OFF</td>
</tr>
<tr>
<td>5</td>
<td>DGN</td>
<td>Spindle drive unit signal return error</td>
<td>X6.1 OFF</td>
</tr>
<tr>
<td>6</td>
<td>DGN</td>
<td>Spindle drive unit signal return error</td>
<td>X6.3 ON</td>
</tr>
<tr>
<td>9</td>
<td>DGN</td>
<td>Stroke end LS confirmation error</td>
<td>LS-16 X3.0 ON</td>
</tr>
<tr>
<td>10</td>
<td>DGN</td>
<td>Stroke end LS confirmation error</td>
<td>LS-15 X3.1 ON</td>
</tr>
<tr>
<td>11</td>
<td>DGN</td>
<td>Stroke end LS confirmation error</td>
<td>LS-13 X3.3 ON</td>
</tr>
<tr>
<td>12</td>
<td>DGN</td>
<td>Stroke end LS confirmation error</td>
<td>LS-14 X3.2 ON</td>
</tr>
<tr>
<td>13</td>
<td>DGN</td>
<td>Limit switch return error</td>
<td>LS-16 X3.0 OFF</td>
</tr>
<tr>
<td>14</td>
<td>DGN</td>
<td>Limit switch return error</td>
<td>LS-15 X3.1 OFF</td>
</tr>
<tr>
<td>15</td>
<td>DGN</td>
<td>Limit switch return error</td>
<td>LS-13 X3.3 OFF</td>
</tr>
<tr>
<td>16</td>
<td>DGN</td>
<td>Limit switch return error</td>
<td>LS-14 X3.2 OFF</td>
</tr>
<tr>
<td>19</td>
<td>DGN</td>
<td>Solenoid motion error</td>
<td>Y1.4 SOL-3098 ON</td>
</tr>
</tbody>
</table>

(Continues to next.)
# Trouble Diagnosis List (DGN Table)

Revised on Aug. 10, 1990

<table>
<thead>
<tr>
<th>DGN No</th>
<th>DGN</th>
<th>Contents</th>
<th>Normal</th>
</tr>
</thead>
<tbody>
<tr>
<td>74</td>
<td>ATC pos. confirmation OFF</td>
<td>X-axis 2nd reference point (G30) [ X-axis reference point (G28) ] Either one of above is OFF.</td>
<td>R1.6 ON</td>
</tr>
<tr>
<td>75</td>
<td>Tool pick up refused</td>
<td>Tool pick up is prohibited. (Reset: Execute tool storing motion manually once.)</td>
<td>K2.7 ON</td>
</tr>
<tr>
<td>76</td>
<td>Arm swing refused to go to magazine side</td>
<td>Tool in magazine &amp; tool in gripper</td>
<td>R56.7 ON</td>
</tr>
<tr>
<td>78</td>
<td>Magazine index command 0</td>
<td>Magazine indexes with 0 (zero) command. (Recommend by T-code or reset spindle tool number.)</td>
<td>R426. R427 = 0</td>
</tr>
<tr>
<td>79</td>
<td>Larger command for magazine index</td>
<td>Larger No. by T-command is given than magazine tool number</td>
<td>R426. R427 ≤D46. D47</td>
</tr>
<tr>
<td>80</td>
<td>Magazine stop position discrepancy</td>
<td>Stop position is not coincided with commanded position</td>
<td>R62.5 OFF</td>
</tr>
<tr>
<td>81</td>
<td>8-axis drive unit abnormal (T2T. 360T)</td>
<td>8-axis drive unit abnormal (Misposition, vibration, command data ≥ 360° &amp; etc.) FEED HOLD comes. (Reset : After depressing RESET SW (SW-1), turn ON control power.)</td>
<td>X4.4 OFF</td>
</tr>
<tr>
<td>82</td>
<td>8-axis drive unit servo alarm (T2T. 360T)</td>
<td>8-axis drive unit servo alarm (TG alarm, low voltage, overload, overcurrent, overspeed &amp; etc. : Refer to MANUAL.) EMERGENCY STOP comes (Reset: After eliminating the cause of alarm, depress RESET SW (SW-1) on 8-axis drive unit. then turn ON control power.)</td>
<td>X4.5 ON</td>
</tr>
<tr>
<td>84</td>
<td>8-axis clamped LS OFF (T2T. 360T. LS-17)</td>
<td>Limit switch confirmation error</td>
<td>(NC-T) LS-17 ON</td>
</tr>
<tr>
<td></td>
<td>NC-T: PS-2</td>
<td>X3.6 ON</td>
<td>(T2T. 360T)</td>
</tr>
<tr>
<td>85</td>
<td>8-axis unclamped LS OFF (T2T. 360T) (LS-18)</td>
<td>Limit switch confirmation error</td>
<td>LS-18 ON</td>
</tr>
<tr>
<td></td>
<td></td>
<td>X3.5 ON</td>
<td>(To be continued.)</td>
</tr>
</tbody>
</table>

---

NIIGATA ENGINEERING CO., LTD. 
T-906
Niigata CNC Horizontal M.C.

Restart Program (Safe Home)

PN40 Machine

07998
G91 G28 Z0 M19;
G91 G28 X0 Y0 M9;
G90 G17 G20 G40 G49 G80;
#3003 = 0;
#3004 = 0;
#508 = 0;
M99;
90

07998 is a sub-program and will normally be imbedded at or near the top of your main program.

Above variables explained.

#3003 = 0     Means turn single block off
#3004 = 0     Means turn feed hold to off
#508 = 0      Means this common variable is holding
Questions: Machine & Control

1. What is the maximum tool mass allowable on the automatic tool changer?
   A] ________________

2. How accurately can the machine position in X, Y, and Z?
   A] ________________

3. A tool is calculated by the operator to run at 11,000 RPM on this machine. Will the machine reach this high RPM?
   A] ________________
   Because max. RPM is________________

4. Which two "G" codes are acceptable for tool length compensation?
   A] ________________
   B] ________________
Questions: Machine & Control

1. Can this machine do circulator interpolation using multi-quadrant circular interpolation?
   A] _________________

2. Explain the advantage of multi-quadrant circular interpolation.
   ____________________________________________________
   ____________________________________________________
   ____________________________________________________

3. Which work coordinate offsets ("G" codes) are allowable on this machine?
   A] _________________
   B] _________________
   C] _________________
   D] _________________
   E] _________________
   F] _________________

4. How many sub-programs can be contained in a main program?
   A] _________________
   B] _________________
Section Eleven

Your Notes:

________________________
________________________
________________________
________________________
________________________
________________________
________________________
________________________
Session Eleven

During this two hour segment, participants will:

1. Define machinability factors of: Speed, feed and depth of cut

2. Understand the use of speeds and feed conversion charts

3. Determine correct cutting speeds and feeds for common cutting tools used in milling applications

4. Understand terms: Cutting speed and surface feet per minute

5. Understand basic formulas used for determining RPM and feed rate

6. Determine correct feed rates for rigid and non-rigid tapping applications
The cutting conditions that determine the rate of metal removal are:

- The cutting speed (CS),
- The feed rate, and
- The depth of cut.

The cutting speed is a value used to determine RPM (Speed).

The feed rate variables are the number of teeth, feed per tool, and RPM.

The depth of cut will be limited by the amount of metal that is to be machined, by the power available on the machine tool, as well as the setup.
Tool life is influenced most by:

1. Cutting speed (RPM)
2. Feed rate
3. Depth of cut

The first step in selecting cutting conditions is to select: ________________________________

The second step in is to select: ____________________________________________

The third step in is to select: ____________________________________________
# Speeds and Feeds

**UNIT: SPEEDS AND FEEDS - RPM VERSUS CUTTING SPEED CONVERSION CHART**

<table>
<thead>
<tr>
<th>DRAMETER</th>
<th>18</th>
<th>20</th>
<th>30</th>
<th>40</th>
<th>50</th>
<th>60</th>
<th>70</th>
<th>80</th>
<th>90</th>
<th>100</th>
<th>120</th>
<th>140</th>
<th>160</th>
<th>180</th>
<th>200</th>
<th>225</th>
<th>250</th>
<th>275</th>
<th>300</th>
</tr>
</thead>
<tbody>
<tr>
<td>in</td>
<td>31</td>
<td>32</td>
<td>33</td>
<td>34</td>
<td>35</td>
<td>36</td>
<td>37</td>
<td>38</td>
<td>39</td>
<td>40</td>
<td>41</td>
<td>42</td>
<td>43</td>
<td>44</td>
<td>45</td>
<td>46</td>
<td>47</td>
<td>48</td>
<td>49</td>
</tr>
<tr>
<td>1/8 44</td>
<td>45</td>
<td>46</td>
<td>47</td>
<td>48</td>
<td>49</td>
<td>50</td>
<td>51</td>
<td>52</td>
<td>53</td>
<td>54</td>
<td>55</td>
<td>56</td>
<td>57</td>
<td>58</td>
<td>59</td>
<td>60</td>
<td>61</td>
<td>62</td>
<td>63</td>
</tr>
<tr>
<td>1/4 31</td>
<td>32</td>
<td>33</td>
<td>34</td>
<td>35</td>
<td>36</td>
<td>37</td>
<td>38</td>
<td>39</td>
<td>40</td>
<td>41</td>
<td>42</td>
<td>43</td>
<td>44</td>
<td>45</td>
<td>46</td>
<td>47</td>
<td>48</td>
<td>49</td>
<td>50</td>
</tr>
</tbody>
</table>

**PERIPHERAL SPEED IN FEET PER MINUTE**

**REVOLUTIONS PER MINUTE**

**ERI**
MIL

UNIT: SPEEDS AND FEEDS - RPM VERSUS CUTTING SPEED
CONVERSION CHART

DIAMETER

350 1

PERIPHERALPEEDINFEETPERMINITTE
Not 15000001 10501 11001 1150112001
4001 4501 Sat 3601 $00 1 1501 2001 2501 1001 $50

1250

REI/OLUTIONSPER MINUTE

Inches

53613 35141 34669 3$197
12224 13752 15279 16807 18335 19862 21390 22918 24448 25974 27502 29030 10560 32013
16044
16807 17571 11335 19099
15280
6112 6876 7639 8403 9168. 9931 10696 11460 12224 12987 13751 14515

1/2

10696

1/4

5348

3/8

3566

4076

4585

5095

5622

6112

6421

7130

7639

1149

8658

9167

9677 10186 10695

2674

3057

3439

3821

4202

4584

4966

5348

5730

6112

6493

6875

1260

7639

8021

8403

8785

9167

9549

1/2

4278

4584

4890

5194

5500

5806

6112

6418

6723

702$

7334

7639

3820

4075

432,

4584

4838

5093

5348

5602

5857

6112

6366

3929

4147

4365

4514

4802

5020

.i238

1457

3438

3629

3820

4012

4202

4393

4584

4773

3056

3201

3361

3514

3647

3$20

2931

3036

3114

5/8
3/4
7/8
1

14/4

2139
1783

2445

2038

2751

2292

3057

2547

3362
2101

3667

3056

3972

3310

3565

1528

1746

1965

2183

2401

2620

2837

3036

3274

3492

3711

1337

1528

1719

1910

2101

2292

2483

2674

2865

3055

3247

1069

1222

1375

1528

1681

1834

1986

2139

2292

2445

2597

2750

2903

11204 13714

12223 12732

1012

1146

1273

1400

1528

1655

1783

1909

2037

2165

2292

2419

2546

2674

1-1/2

891

2101

764

873

982

1091

1200

1310

1419

1528

1637

1746

1855

1964

2074

2183

2292

2401

2510

2619

2728

1-3/4

668

764

159

955

1050

1146

1241

1337

1432

1528

1622

1719

1814

1910

2005

2101

2196

2292

2387

2

849

934

1011

1103

118$

1273

1358

1443

1528

1613

1698

1782

1867

1956

2037

2122

764

to

916

993

1069

1146

1222

1299

1375

1451

1528

1604

1681

1758

1833

1910

1250

1320

1389

1458

1521

1598

1667

1736

1146

1210

1273

1336

1401

1464

1528

1592

1352

1411

1468

2-1/4

24/2

594
534

679

611

764

687

2-3/4

486

553

625

694

764

133

903

972

1041

1110

1181

3

443

509

572

637

700

764

12t

891

954

1018

1012

14/4

411

470

528

517

646

704

764

123

812

940

999

1051

1117

1175

1234

1293

311

436

490

546

600

654

709

764

819

873

928

982

1037

1091

1144

1201

1236

1110

1364

14/2

662

713

764

815

$66

917

968

1019

1070

1121

1172

1223

1274

621

668

715

764

812

859

907

955

1002

1050

1098

1145

1194

674

718

764

809

854

899

642

987

1034

1079

1123

191

932

976

1017

1061

11/4

356

407

458

509

560

610

4

334

382

429

478

525

572

44/4

314

359

404

449

494

539

584

629

4-1/2

297

339

312

424

467

509

552

594

636

679

721

764

806

$49

4-3/4

221

321

361

402

442

482

323

563

603

643

682

724

764

$04

844

885

924

965

1006

267

305

143

312

420

458

496

535

572

611

649

688

726

764

802

$40

878

916

954

5

473

SIO

544

582

619

656

691

728

764

800

837

873

909

764

799

133

:68

5-1/4

235

290

327

364

402

436

3-1/2

242

277

312

347

382

416

451

486

520

556

590

625

660

694

730

51/4

212

266

298

332

364

398

431

464

491

532

564

596

630.

664

696

731

762

796

830

222

254

286

318

150

380

414

446

476

:109

540

573

605

637

661

701

732

764

796

6

161

BEST COPY AVAILABLE

218


## Speeds and Feeds

**UNIT: SPEEDS AND FEEDS - CONVERSION CHART FOR FEED**

<table>
<thead>
<tr>
<th>NUMBER of Teeth</th>
<th>.0002</th>
<th>.0004</th>
<th>.0006</th>
<th>.0008</th>
<th>.0010</th>
<th>.0012</th>
<th>.0014</th>
<th>.0016</th>
<th>.0018</th>
<th>.0020</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>.0004</td>
<td>.0008</td>
<td>.0012</td>
<td>.0016</td>
<td>.0020</td>
<td>.0024</td>
<td>.0028</td>
<td>.0032</td>
<td>.0036</td>
<td>.0040</td>
</tr>
<tr>
<td>3</td>
<td>.0006</td>
<td>.0012</td>
<td>.0018</td>
<td>.0024</td>
<td>.0032</td>
<td>.0040</td>
<td>.0048</td>
<td>.0056</td>
<td>.0064</td>
<td>.0080</td>
</tr>
<tr>
<td>4</td>
<td>.0008</td>
<td>.0016</td>
<td>.0024</td>
<td>.0032</td>
<td>.0040</td>
<td>.0056</td>
<td>.0064</td>
<td>.0080</td>
<td>.1048</td>
<td>.1280</td>
</tr>
<tr>
<td>6</td>
<td>.0012</td>
<td>.0024</td>
<td>.0036</td>
<td>.0048</td>
<td>.0064</td>
<td>.0096</td>
<td>.0128</td>
<td>.0192</td>
<td>.0256</td>
<td>.0384</td>
</tr>
<tr>
<td>8</td>
<td>.0016</td>
<td>.0032</td>
<td>.0048</td>
<td>.0064</td>
<td>.0104</td>
<td>.0144</td>
<td>.0224</td>
<td>.0320</td>
<td>.0432</td>
<td>.0640</td>
</tr>
<tr>
<td>10</td>
<td>.0020</td>
<td>.0040</td>
<td>.0060</td>
<td>.0080</td>
<td>.0120</td>
<td>.0160</td>
<td>.0240</td>
<td>.0360</td>
<td>.0480</td>
<td>.0720</td>
</tr>
<tr>
<td>12</td>
<td>.0024</td>
<td>.0048</td>
<td>.0072</td>
<td>.0096</td>
<td>.0144</td>
<td>.0192</td>
<td>.0280</td>
<td>.0424</td>
<td>.0576</td>
<td>.0864</td>
</tr>
<tr>
<td>14</td>
<td>.0028</td>
<td>.0056</td>
<td>.0084</td>
<td>.0112</td>
<td>.0160</td>
<td>.0224</td>
<td>.0336</td>
<td>.0480</td>
<td>.0640</td>
<td>.0960</td>
</tr>
<tr>
<td>16</td>
<td>.0032</td>
<td>.0064</td>
<td>.0096</td>
<td>.0128</td>
<td>.0192</td>
<td>.0256</td>
<td>.0384</td>
<td>.0512</td>
<td>.0672</td>
<td>.0992</td>
</tr>
<tr>
<td>18</td>
<td>.0036</td>
<td>.0072</td>
<td>.0108</td>
<td>.0144</td>
<td>.0208</td>
<td>.0272</td>
<td>.0400</td>
<td>.0560</td>
<td>.0768</td>
<td>.1152</td>
</tr>
<tr>
<td>20</td>
<td>.0040</td>
<td>.0080</td>
<td>.0120</td>
<td>.0160</td>
<td>.0224</td>
<td>.0288</td>
<td>.0432</td>
<td>.0600</td>
<td>.0800</td>
<td>.1152</td>
</tr>
<tr>
<td>22</td>
<td>.0044</td>
<td>.0088</td>
<td>.0132</td>
<td>.0176</td>
<td>.0240</td>
<td>.0304</td>
<td>.0460</td>
<td>.0640</td>
<td>.0840</td>
<td>.1200</td>
</tr>
<tr>
<td>24</td>
<td>.0048</td>
<td>.0096</td>
<td>.0144</td>
<td>.0192</td>
<td>.0256</td>
<td>.0324</td>
<td>.0480</td>
<td>.0680</td>
<td>.0880</td>
<td>.1280</td>
</tr>
<tr>
<td>26</td>
<td>.0052</td>
<td>.0104</td>
<td>.0156</td>
<td>.0208</td>
<td>.0272</td>
<td>.0344</td>
<td>.0512</td>
<td>.0720</td>
<td>.0936</td>
<td>.1344</td>
</tr>
<tr>
<td>28</td>
<td>.0056</td>
<td>.0112</td>
<td>.0168</td>
<td>.0224</td>
<td>.0296</td>
<td>.0372</td>
<td>.0540</td>
<td>.0768</td>
<td>.1008</td>
<td>.1408</td>
</tr>
<tr>
<td>30</td>
<td>.0060</td>
<td>.0120</td>
<td>.0180</td>
<td>.0240</td>
<td>.0320</td>
<td>.0400</td>
<td>.0584</td>
<td>.0816</td>
<td>.1080</td>
<td>.1512</td>
</tr>
<tr>
<td>32</td>
<td>.0064</td>
<td>.0128</td>
<td>.0192</td>
<td>.0256</td>
<td>.0340</td>
<td>.0432</td>
<td>.0624</td>
<td>.0876</td>
<td>.1152</td>
<td>.1632</td>
</tr>
<tr>
<td>34</td>
<td>.0068</td>
<td>.0136</td>
<td>.0208</td>
<td>.0272</td>
<td>.0360</td>
<td>.0460</td>
<td>.0656</td>
<td>.0920</td>
<td>.1200</td>
<td>.1712</td>
</tr>
<tr>
<td>36</td>
<td>.0072</td>
<td>.0144</td>
<td>.0216</td>
<td>.0288</td>
<td>.0384</td>
<td>.0480</td>
<td>.0680</td>
<td>.0960</td>
<td>.1256</td>
<td>.1816</td>
</tr>
<tr>
<td>40</td>
<td>.0080</td>
<td>.0160</td>
<td>.0240</td>
<td>.0320</td>
<td>.0416</td>
<td>.0512</td>
<td>.0704</td>
<td>.0992</td>
<td>.1344</td>
<td>.1880</td>
</tr>
<tr>
<td>44</td>
<td>.0088</td>
<td>.0176</td>
<td>.0264</td>
<td>.0352</td>
<td>.0448</td>
<td>.0552</td>
<td>.0744</td>
<td>.1032</td>
<td>.1400</td>
<td>.1984</td>
</tr>
<tr>
<td>48</td>
<td>.0096</td>
<td>.0192</td>
<td>.0288</td>
<td>.0384</td>
<td>.0480</td>
<td>.0584</td>
<td>.0776</td>
<td>.1064</td>
<td>.1440</td>
<td>.2048</td>
</tr>
<tr>
<td>52</td>
<td>.0104</td>
<td>.0208</td>
<td>.0312</td>
<td>.0416</td>
<td>.0520</td>
<td>.0624</td>
<td>.0816</td>
<td>.1104</td>
<td>.1512</td>
<td>.2112</td>
</tr>
<tr>
<td>56</td>
<td>.0112</td>
<td>.0224</td>
<td>.0336</td>
<td>.0448</td>
<td>.0560</td>
<td>.0672</td>
<td>.0864</td>
<td>.1152</td>
<td>.1560</td>
<td>.2168</td>
</tr>
<tr>
<td>60</td>
<td>.0120</td>
<td>.0240</td>
<td>.0360</td>
<td>.0480</td>
<td>.0592</td>
<td>.0704</td>
<td>.0900</td>
<td>.1192</td>
<td>.1600</td>
<td>.2224</td>
</tr>
<tr>
<td>72</td>
<td>.0144</td>
<td>.0288</td>
<td>.0432</td>
<td>.0576</td>
<td>.0672</td>
<td>.0872</td>
<td>.1076</td>
<td>.1384</td>
<td>.1800</td>
<td>.2424</td>
</tr>
<tr>
<td>90</td>
<td>.0180</td>
<td>.0360</td>
<td>.0540</td>
<td>.0720</td>
<td>.0900</td>
<td>.1100</td>
<td>.1300</td>
<td>.1600</td>
<td>.2000</td>
<td>.2600</td>
</tr>
</tbody>
</table>

**FEED PER MINUTE FOR ONE RPM**

**EXAMPLE:** For .006 inch feed per tooth, using a 20 tooth cutter, turning at 78 RPM.

**Feeds for one RPM (from table) = .120**

**Feed for 78 RPM = .120 * 78 = 9.39 inches per minute**
UNIT: SPEEDS AND FEEDS - CENTER DRILLING

SUGGESTED CENTER DRILLING SPEEDS IN R.P.M.

<table>
<thead>
<tr>
<th>SIZE</th>
<th>SUGGESTED RPM</th>
<th>SUGGESTED FEED (I.P.R.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>2,000</td>
<td>.001 - .003</td>
</tr>
<tr>
<td>1</td>
<td>1,750</td>
<td>.001 - .003</td>
</tr>
<tr>
<td>2</td>
<td>1,500</td>
<td>.002 - .006</td>
</tr>
<tr>
<td>3</td>
<td>1,300</td>
<td>.002 - .006</td>
</tr>
<tr>
<td>4</td>
<td>1,100</td>
<td>.002 - .006</td>
</tr>
<tr>
<td>5</td>
<td>900</td>
<td>.002 - .006</td>
</tr>
<tr>
<td>6</td>
<td>700</td>
<td>.002 - .006</td>
</tr>
<tr>
<td>7</td>
<td>500</td>
<td>.003 - .008</td>
</tr>
</tbody>
</table>

I.P.M. = I.P.R. * RPM
# Speeds and Feeds

**UNIT: SPEEDS AND FEEDS - SPOT DRILLING**

**SUGGESTED SPOT DRILLING SPEEDS IN R.P.M.**

*FORMULA*  
\[ \text{RPM} = \frac{\text{S.F.P.M.} \times 4}{\text{DIA.}} \times 2 \]

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>SPOT DRILL SPEED S.F.P.M.</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALUMINUM</td>
<td>200</td>
</tr>
<tr>
<td>BRASS</td>
<td>200</td>
</tr>
<tr>
<td>MAGNESIUM</td>
<td>150</td>
</tr>
<tr>
<td>CAST IRON</td>
<td>70</td>
</tr>
<tr>
<td>LOW CARBON STEEL</td>
<td>80</td>
</tr>
<tr>
<td>STAINLESS STEEL</td>
<td>30</td>
</tr>
<tr>
<td>THERMOPLASTICS</td>
<td>100</td>
</tr>
<tr>
<td>TITANIUM</td>
<td>20</td>
</tr>
</tbody>
</table>

**SUGGESTED FEEDS FOR HIGH SPEED STEEL SPOT DRILLS**

<table>
<thead>
<tr>
<th>SPOT DRILL DIA.</th>
<th>FEED / REV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/8</td>
<td>.001</td>
</tr>
<tr>
<td>1/4</td>
<td>.002</td>
</tr>
<tr>
<td>3/8</td>
<td>.003</td>
</tr>
<tr>
<td>1/2</td>
<td>.0035</td>
</tr>
<tr>
<td>3/4</td>
<td>.005</td>
</tr>
<tr>
<td>1.0</td>
<td>.007</td>
</tr>
</tbody>
</table>
# Speeds and Feeds

**UNIT: SPEEDS AND FEEDS - DRILLING**

**SUGGESTED SPEEDS FOR HIGH SPEED STEEL DRILLS**

*FORMULA: \( \text{S.F.P.M.} \times 4 \)*

\[
\text{RPM} = \frac{\text{S.F.P.M.}}{\text{DIA.}}
\]

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>REGULAR DRILLS SPEED</th>
<th>REGULAR DRILLS SFPM</th>
<th>TURBOFLUTE DRILLS SPEED</th>
<th>TURBOFLUTE DRILLS SFPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALUMINUM</td>
<td>200</td>
<td>200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BRASS</td>
<td>200</td>
<td>200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MAGNESIUM</td>
<td>150</td>
<td>200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CAST IRON</td>
<td>70</td>
<td>70</td>
<td></td>
<td></td>
</tr>
<tr>
<td>LOW CARBON STEEL</td>
<td>80</td>
<td>80</td>
<td></td>
<td></td>
</tr>
<tr>
<td>STAINLESS STEEL</td>
<td>30</td>
<td>30</td>
<td></td>
<td></td>
</tr>
<tr>
<td>THERMOPLASTICS</td>
<td>100</td>
<td>100</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TITANIUM</td>
<td>20</td>
<td>20</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**SUGGESTED FEEDS FOR HIGH SPEED STEEL DRILLS**

<table>
<thead>
<tr>
<th>DRILL DIA.</th>
<th>REGULAR DRILLS FEED / REV</th>
<th>TURBO-FLUTE DRILLS FEED / REV</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNDER 1/8</td>
<td>.001 - .002</td>
<td>.001 - .002</td>
</tr>
<tr>
<td>1/8 - 1/4</td>
<td>.002 - .003</td>
<td>.003 - .008</td>
</tr>
<tr>
<td>1/4 - 1/2</td>
<td>.004 - .007</td>
<td>.006 - .014</td>
</tr>
<tr>
<td>1/2 - 1.0</td>
<td>.007 - .015</td>
<td>.010 - .030</td>
</tr>
<tr>
<td>1/0 +</td>
<td>.015 - .025</td>
<td>.022 - .050</td>
</tr>
</tbody>
</table>

\[
\text{I.P.M.} = \text{I.P.R.} \times \text{RPM}
\]

**QUICK FORMULA**

\[
\text{RECOMMENDED I.P.R}
\]

**I.P.M. = INCH PER MINUTE**

\[
165 \quad \text{DRILL DIA.}
\]
UNIT: SPEEDS AND FEEDS - REAMING

SUGGESTED SPEEDS FOR HIGH SPEED STEEL REAMERS

FORMULA: \[ S.F.P.M. \times 4 = \text{CUTTING SPEED} \]

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>CUTTING SPEED S.F.P.M.</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALUMINUM</td>
<td>100</td>
</tr>
<tr>
<td>BRASS</td>
<td>100</td>
</tr>
<tr>
<td>MAGNESIUM</td>
<td>75</td>
</tr>
<tr>
<td>CAST IRON</td>
<td>50</td>
</tr>
<tr>
<td>LOW CARBON STEEL</td>
<td>60</td>
</tr>
<tr>
<td>STAINLESS STEEL</td>
<td>20</td>
</tr>
<tr>
<td>THERMOPLASTICS</td>
<td>50</td>
</tr>
<tr>
<td>TITANIUM</td>
<td>20</td>
</tr>
</tbody>
</table>

SUGGESTED FEEDS FOR HIGH SPEED STEEL REAMERS

<table>
<thead>
<tr>
<th>REAMER DIAMETER</th>
<th>FEED PER REVOLUTION I.P.R.</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNDER 1/8 DIA.</td>
<td>.002 - .004</td>
</tr>
<tr>
<td>1/8 - 1/4</td>
<td>.004 - .006</td>
</tr>
<tr>
<td>1/4 - 1/2</td>
<td>.006 - .008</td>
</tr>
<tr>
<td>1/2 - 1.0</td>
<td>.008 - .015</td>
</tr>
<tr>
<td>1.0 +</td>
<td>.015 - .025</td>
</tr>
</tbody>
</table>

FEED RATE FORMULA

\[ \text{I.P.M.} = \text{I.P.R.} \times \text{RPM} \]

I.P.M. = INCH PER MINUTE
UNIT: SPEEDS AND FEEDS - COUNTERSINKING

SUGGESTED COUNTERSINKING SPEEDS IN R.P.M.

<table>
<thead>
<tr>
<th>C’SINK SIZE</th>
<th>ALUMINUM</th>
<th>C. IRON</th>
<th>STEEL</th>
</tr>
</thead>
<tbody>
<tr>
<td>.25 DIA</td>
<td>500</td>
<td>200</td>
<td>175</td>
</tr>
<tr>
<td>.50 DIA</td>
<td>250</td>
<td>150</td>
<td>150</td>
</tr>
<tr>
<td>.75 DIA</td>
<td>200</td>
<td>100</td>
<td>125</td>
</tr>
<tr>
<td>1.0 DIA</td>
<td>150</td>
<td>75</td>
<td>80</td>
</tr>
<tr>
<td>2.0 DIA</td>
<td>100</td>
<td>75</td>
<td>75</td>
</tr>
</tbody>
</table>

FEED RATE RANGE - .003 TO .010 I.P.R.

NOTE: FEED RATE VARIES FROM MACHINE TO MACHINE.

I.P.M. = I.P.R. * RPM
## Speeds and Feeds

**UNIT: SPEEDS AND FEEDS - COUNTERBORING**

**Suggested Speeds**

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>SPEED ( SFPM )</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALLOY STEEL</td>
<td>40 - 80</td>
</tr>
<tr>
<td>BRASS</td>
<td>150 - 300</td>
</tr>
<tr>
<td>CAST IRON SOFT</td>
<td>120 - 140</td>
</tr>
<tr>
<td>MILD STEEL</td>
<td>75 - 85</td>
</tr>
</tbody>
</table>

**Suggested Feeds**

<table>
<thead>
<tr>
<th>COUNTERBORE DIAMETER</th>
<th>FEED ( IPR )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4 - 3/8</td>
<td>.003 - .005</td>
</tr>
<tr>
<td>7/16 - 5/8</td>
<td>.004 - .006</td>
</tr>
<tr>
<td>11/16 - 7/8</td>
<td>.005 - .007</td>
</tr>
<tr>
<td>15/16 - 1 3/16</td>
<td>.006 - .008</td>
</tr>
<tr>
<td>1 1/4 - 1 1/2</td>
<td>.007 - .009</td>
</tr>
<tr>
<td>1 9/16 - 2.0</td>
<td>.008 - .010</td>
</tr>
</tbody>
</table>
# Speeds and Feeds

**UNIT: SPEEDS AND FEEDS - TAPPING**

**SUGGESTED TAPPING SPEEDS IN R.P.M.**

<table>
<thead>
<tr>
<th>TAP SIZE</th>
<th>CAST IRON</th>
<th>ALUMINUM</th>
<th>MILD STEEL</th>
<th>ST. STEEL</th>
<th>BRASS</th>
</tr>
</thead>
<tbody>
<tr>
<td>10-24</td>
<td>1300</td>
<td>1400</td>
<td>1100</td>
<td>500</td>
<td>1500</td>
</tr>
<tr>
<td>1/4-20</td>
<td>1000</td>
<td>1200</td>
<td>750</td>
<td>400</td>
<td>1200</td>
</tr>
<tr>
<td>5/16-18</td>
<td>850</td>
<td>1100</td>
<td>650</td>
<td>300</td>
<td>1200</td>
</tr>
<tr>
<td>7/16-14</td>
<td>600</td>
<td>800</td>
<td>450</td>
<td>200</td>
<td>950</td>
</tr>
<tr>
<td>1/2-13</td>
<td>500</td>
<td>650</td>
<td>400</td>
<td>200</td>
<td>850</td>
</tr>
<tr>
<td>5/8-11</td>
<td>375</td>
<td>500</td>
<td>300</td>
<td>190</td>
<td>700</td>
</tr>
<tr>
<td>3/4-10</td>
<td>325</td>
<td>400</td>
<td>250</td>
<td>125</td>
<td>575</td>
</tr>
<tr>
<td>1 - 8</td>
<td>250</td>
<td>300</td>
<td>175</td>
<td>75</td>
<td>425</td>
</tr>
</tbody>
</table>

**SUGGESTED TAPPING FEED RATES**

**FEED RATE = PITCH OF THREAD * PRM * .95**

**PITCH = 1 / NUMBER OF THREADS PER INCH**

.95 IS 95% OF VALUE TO ALLOW SOME TAP "FLOATING" DURING CUT.

**EX. TAP CAST IRON USING A 1/2 - 13 TAP**

\[ P = \frac{1}{N} = \frac{1}{13} = 0.0769 \]

**FEED RATE = 0.0769 \times 500 \times 0.95 = 36.52 I.P.M.**

**THUS IN YOUR PROGRAM** \[ F36.52 \]
UNIT: SPEEDS AND FEEDS - BORING

SUGGESTED BORING SPEEDS IN S.F.P.M.

\[
\text{S.F.P.M.} \times 4 \\
\text{RPM} = \text{---------} \\
\text{DIA.}
\]

<table>
<thead>
<tr>
<th>CAST IRON</th>
<th>BRASS</th>
<th>ALUM.</th>
<th>MILD STEEL</th>
<th>TOOL STEEL</th>
<th>STAIN STEEL</th>
</tr>
</thead>
<tbody>
<tr>
<td>H.S.S</td>
<td>70</td>
<td>125</td>
<td>150</td>
<td>100</td>
<td>60</td>
</tr>
<tr>
<td>CARBIDE</td>
<td>225</td>
<td>300</td>
<td>500</td>
<td>200</td>
<td>100</td>
</tr>
</tbody>
</table>

FEEDS FOR ROUGHING

FEEDS FOR ROUGH BORING \( \cdot 007 - \cdot 015 \text{ INCH PER REV (I.P.R.)} \)

FEEDS FOR FINISH BORING \( \cdot 001 - \cdot 005 \text{ INCH PER REV. (I.P.R.)} \)
UNIT: SPEEDS AND FEEDS - END MILLING

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>FEED / TOOTH .25 D</th>
<th>FEED / TOOTH .50 D</th>
<th>FEED / TOOTH 1.0 D</th>
<th>S.F.P.M.</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALUMINUM</td>
<td>.003</td>
<td>.006</td>
<td>.009</td>
<td>150</td>
</tr>
<tr>
<td>MAGNESIUM</td>
<td>.004</td>
<td>.006</td>
<td>.010</td>
<td>150</td>
</tr>
<tr>
<td>BRONZE</td>
<td>.003</td>
<td>.005</td>
<td>.007</td>
<td>250</td>
</tr>
<tr>
<td>CAST IRON</td>
<td>.003</td>
<td>.005</td>
<td>.008</td>
<td>60</td>
</tr>
<tr>
<td>STEEL LC</td>
<td>.001</td>
<td>.002</td>
<td>.004</td>
<td>80</td>
</tr>
<tr>
<td>STEEL ALY</td>
<td>.0005</td>
<td>.001</td>
<td>.003</td>
<td>50</td>
</tr>
<tr>
<td>STEEL SS</td>
<td>.001</td>
<td>.002</td>
<td>.004</td>
<td>55</td>
</tr>
<tr>
<td>INCONEL</td>
<td>.0002</td>
<td>.001</td>
<td>.003</td>
<td>30</td>
</tr>
<tr>
<td>TITANIUM</td>
<td>.001</td>
<td>.002</td>
<td>.004</td>
<td>25</td>
</tr>
</tbody>
</table>

S.F.P.M. * 4
RPM = ------------
DIA.

FEED RATE = R * T * RPM

R = FEED / TOOTH
T = # OF TEETH ON END MILL

RPM = RPM CALCULATED THROUGH FORMULA
## Speeds and Feeds

**UNIT: FACE MILLING**

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>DEPTH OF CUT</th>
<th>HSS TOOL</th>
<th>CARBIDE INSERT</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>SPEED</td>
<td>FEED</td>
</tr>
<tr>
<td>ALUMINUM</td>
<td>0.250</td>
<td>800</td>
<td>0.022</td>
</tr>
<tr>
<td></td>
<td>0.025</td>
<td>1200</td>
<td>0.010</td>
</tr>
<tr>
<td>MAGNESIUM</td>
<td>0.250</td>
<td>900</td>
<td>0.022</td>
</tr>
<tr>
<td></td>
<td>0.025</td>
<td>1500</td>
<td>0.010</td>
</tr>
<tr>
<td>MILD STEEL</td>
<td>0.150</td>
<td>175</td>
<td>0.012</td>
</tr>
<tr>
<td></td>
<td>0.025</td>
<td>230</td>
<td>0.008</td>
</tr>
<tr>
<td>SS STEEL</td>
<td>0.150</td>
<td>130</td>
<td>0.010</td>
</tr>
<tr>
<td>TYPE 303</td>
<td>0.025</td>
<td>160</td>
<td>0.008</td>
</tr>
</tbody>
</table>

**SPEED = SFPM**

**FEED = IN PER TOOTH**

\[
\text{SFPM} \times 4
\]

\[
\text{RPM} = \frac{\text{DIA}}{
\]

**FEED RATE = R * T * RPM**

\[
\text{R} = \text{FEED IN INCH PER TOOTH}
\]

\[
\text{T} = \text{NUMBER OF TEETH}
\]
PROBLEM # 1

OPERATION - END MILLING       MATERIAL - CHILLED CAST IRON
TOOL SIZE - .375 DIA. ( HSS ) 2 FLUTE
FEED / TOOTH = 
RPM =            FEED RATE = 

PROBLEM # 2

OPERATION - FACE MILLING       MATERIAL - INCONEL X
TOOL SIZE - 3.5 DIA. ( CARBIDE ) 4 TEETH
FEED / TOOTH = 
RPM =            FEED RATE = 

PROBLEM # 3

OPERATION - FACE MILLING       MATERIAL - COPPER
TOOL SIZE - 2.5 DIA. ( HSS ) 6 TEETH
FEED / TOOTH = 
RPM =            FEED RATE = 

PROBLEM # 4

OPERATION - END MILLING       MATERIAL - MAGNESIUM ALLOY
TOOL SIZE - .75 DIA. ( CARBIDE TOOL ) 3 FLUTE
FEED / TOOTH = 
RPM =            FEED RATE = 

230
# Speeds and Feeds Questions

Calculate the correct RPM and feed rate for each machining situation below.

1. Operation - center drilling  
   Material - cast iron  
   #3 C-Drill (H.S.S.)  
   [RPM]  
   [Feed Rate]

   For this problem, use RPM vs. cutting speed conversion chart.

2. Cutter diameter - 1.25  
   Cutting speed - 450  
   [RPM]
Use conversion chart for feed.

3. Feed per tooth - .002
   Number of teeth - 8
   RPM - 125
   Find Per Minute] ________

4. Operation - spot drilling
   Material - cast iron
   1/2 diameter (H.S.S.) spot drill
   RPM] ________________
   Feed Rate] __________

5. Operation - drilling
   Material - cast iron
   1/4 diameter drill (regular)
   RPM] ________________
   Feed Rate] __________
### Speeds and Feeds Questions

4. **Operation - reaming**  
   **Material** - stainless steel  
   .500 dia reamer (H.S.S.)  
   **RPM**]  
   **Feed Rate**]  

5. **Operation - countersinking**  
   **Material** - cast iron  
   .75 diameter C-sink  
   **RPM**]  
   **Feed Rate**]  

5. **Operation - tapping (non-rigid)**  
   **Material** - cast iron  
   5/8 - 11 tap  
   **RPM**]  
   **Feed Rate**]
9. Operation - tapping (rigid)
   Material - brass
   1-8 tap
   RPM] ____________
   Feed Rate] ____________

10. Operation - boring
    Material - cast iron
    Carbide tool
    Fine boring - .950 dia
    RPM] ____________
    Feed Rate] ____________
11. Operation - end milling
   Material - titanium
   .500 dia H.S.S. end mill
   RPM] ________________
   Feed Rate] __________

12. Operation - face milling
   Material - mild steel
   5.0 dia carbide
   Face mill with 8 inserts
   RPM] ________________
   Feed Rate] __________
Session Twelve

During this four hour segment, participants will:

1. Understand the features found on a Yasnac control for CNC lathes
2. Understand the features found on a machine control station for CNC lathes
3. Understand axes movement found on a 2 axis CNC lathe
4. Understand common "G" and "M" codes used in programming a CNC lathe
5. Understand special codes of G96 and G99; and how they relate to spindle speed
6. Understand special codes of G96 and G99; and how they relate to feed rate
7. Understand the reasons behind selecting right hand or left hand cut turning tools for CNC lathe operation
8. Understand tool call and offset commands

more on next page
9. Apply the correct programming does for a finishing pass on a turned part

10. Understand how to apply a sub-program within a main program

11. Understand basic lathe formulas of conversion factors and production time

12. Understand specific fixed cycles used in CNC lathes

13. Apply knowledge of CNC training center by answering question on worksheets
Nakamora CNC Lathe

Yasnac CNC Control

Yasnac Controller
Nakamora CNC Lathe

Yasnac LX3 Control

YASNAC LX3 Operator Panel
Nakamora CNC Lathe

Panel Functions.

1. Power On
   Push once to ________________.
   Push again to ________________.

   Power Off
   Push once to de-energize both the ________________ and the ________________ power system of the NC unit.

2. CRT graphic display

3. Function keys
   (1) ALM  Alarm Key  Displays ______ Codes.
   (2) DGN  Diagnostic Key  Displays I/O Signal Info.
   (3) PRM  Parameter Key  Displays parameters
   (4) SET  Setting Key  For setting and writing data.
   (5) COM  Command Key  For displaying automatic operation command values.
   (6) PROG  Program Key  For part program display and writing
   (7) POS  Position Key  For displaying various current values.
   (8) OFS  Offset Key  For displaying and writing tool offset values.
4. Software Function Keys
Depressing Keys F1 through F5 calls up the corresponding functions.

5. Address Keys
The keys for keying the address characters when wiring in various data.
   * **Special Characters**
     / Key: For ________________ skip.
   
   EOB Key: For commanding end of one block. On the CRT, "j" is displayed instead of EOB.

3. Data Keys
The 15 data keys are used for writing:
   (1) MDI command values
   (2) Tool offset values
   (3) Setting values
   (4) Parameters
   (5) Other numerical values
   CAN (Cancel) Key: for deleting working values or address data
   WR (Write) Key: for storing address and data key in values
   Shift Key: for inputting special codes found on upper left corner of keys. Depressing the respective key.
7. **Page Keys**
   - Key: For displaying the next page
   - Key: For displaying the preceding page

8. **Cursor Keys**
   - Key: Moves cursor backward
   - Key: Moves cursor forward

9. **Next Key**
   Used for special functions
   1. Selects search type in edit mode
   2. Selects between line cursor and word cursor in edit mode

10. **Origin Key**
    For setting the current tool position as the new origin of the new coordinate system
11. Memory data keys
   OUT, VER, IN keys are effective only in edit mode.

   OUT key: For sending a program out from control to a computer, etc.

   IN key: For sending a program into control from a computer.

   VER key: (Verify) checks data in memory against data sent in by computer or tape reader

12. Edit keys:
   ERASE, INSRT, ALTER keys are for editing part programs stored in one memory.

   ERASE key: For erasing data in memory

   INSRT key: For inserting data

   ALTER key: For changing data

13. Reset key
   Resets internal condition of the NC
   1. Clears buffer
   2. Clears alarm
Machine control station
Cycle Start - Depress to start system in the automatic mode (tape, MDI, and mem.)

Feed Hold - Temporary stop of automatic operation
1. Not active during thread cutting
2. Depress cycle start to restart operation

Emergency stop push button - stops power
1. Machine stopped immediately by dynamic brake
2. Will display alarm code 330
3. To restart operation
   a] turn E-stop knob clockwise
   b] depress reset
   c] press Power On pushbutton
4. Use for turning off system
Nakamora CNC Lathe

Handle Dial (Manual pulse generator)
1. Set mode to handle
2. Select axis
3. Set mode amount per graduation
4. Rotate dial to move select axis
   CW = + direction
   CCW = - direction

Handle axis select
1. Selects X or Z axis

Manual pulse multiply switch
1. Selects value for handle mode
2. X1 to X100
   X1 = .0001 inch/step
   X100 = 10 inches/step

Jog and rapid pushbuttons
1. Used to feed tool manually
2. +X, -X, +Z, -Z
3. Jog speed set by jog feed rate switch
4. Each time the pushbutton is depressed in the STEP mode, the tool is moved by the value per step by Manual Pulse Multiply select switch.
Nakamora CNC Lathe

Jog feedrate switch and feedrate override switch.
1. In auto mode, feedrate adjustable from 0 - 200% of programmed feedrate.
2. Jog feedrate - used to select feedrate in the jog mode.

Rapid traverse rate override switch
1. Settable to F0, 25, 50 and 100%

Feedrate override cancel switch
1. Turning this switch on, feedrate is fixed at 100%
Nakamora CNC Lathe

Spindle speed override switch
1. Spindle speed adjustable from 50 - 120% of current spindle speed.

Manual reference point return switch
1. Switch used to bring the tool back to reference point manually.

Reference pint lamps
1. Lamps indicate that the tool is positioned on the reference pint
2. G28 auto return to reference point

Single block switch
1. Block-by-block can be obtained when this switch is turned on

Optional block skip switch
1. When On all commands in a block following / are neglected
2. When Off, all blocks including those preceded by / are executed.
Dry run switch
1. When On, tool moves at speed selected by the JOG FEEDRATE switch

Display lock/machine lock switch
1. When OFF, normal position display
2. When Display Lock excludes tool movement from display
3. When Machine Lock, axis movement including zero return is inhibited

M-Function Lock Switch
1. When ON, it ignores M, S, and T commands
2. To check program, it is used in combination with Machine Lock function
Manual Absolute Switch

1. When On, the tool moves as per diagram below.

   Tool Movement with MANUAL ABSOLUTE Switch On

2. When Off, the tool moves as per diagram below.

   Tool Movement with MANUAL ABSOLUTE Switch Off
Edit Lock Switch
1. When On, the following operations will not work.
   a] Erase, Insert, and Alter keys
   b] Storing of NC tape

G50 point return switch
1. Returns tool to the coordinate system setup point manually (where G50 has been programmed)

Manual interruption point return switch
1. When On, returns the tool to where the NC was switched over from the auto mode to the manual mode.

Program Restart Switch
1. Used for restarting the part program from any desired sequence number
Nakamora CNC Lathe

X Axis mirror image
   1. When On, all X axis coordinate are reversed.

Auto mode handle offset switch
   1. Enables tool motion through the use of MPG knob.

Cutting depth override switch
   G71 is a rough turning cycle
   G72 is a rough facing cycle
   1. Can override depths of cut from 10 - 200%

Positing store pushbutton
   1. Used for inputting measured workpiece values direction into control.
All major applications for turning can be accomplished with this form of turning center including; shaft work (with tailstock), chucking work, and bar work (if a bar feeder is used).
X axis is a diameter controlling axis.

Z axis is a length (depth) controlling axis.
# List of G Codes

<table>
<thead>
<tr>
<th>G Code</th>
<th>Special G Code I</th>
<th>Special G Code II</th>
<th>Group</th>
<th>Function</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>G00</td>
<td>G00</td>
<td>B</td>
<td>Positioning (rapid traverse feed)</td>
<td>B</td>
</tr>
<tr>
<td>G01</td>
<td>G01</td>
<td>G01</td>
<td>01</td>
<td>Linear interpolation, angle programming for linear interpolation</td>
<td>B, O</td>
</tr>
<tr>
<td>G02</td>
<td>G02</td>
<td>G02</td>
<td>02</td>
<td>Circular interpolation CW, (radius R designation)</td>
<td>B, O</td>
</tr>
<tr>
<td>G03</td>
<td>G03</td>
<td>G03</td>
<td>03</td>
<td>Circular interpolation CCW, (radius R designation)</td>
<td>B, O</td>
</tr>
<tr>
<td>G04</td>
<td>G04</td>
<td>G04</td>
<td></td>
<td>Dwell</td>
<td>B</td>
</tr>
<tr>
<td>G06</td>
<td>G06</td>
<td>G06</td>
<td></td>
<td>ERROR DETECT OFF positioning</td>
<td>B</td>
</tr>
<tr>
<td>G10</td>
<td>G10</td>
<td>G10</td>
<td></td>
<td>Tool offset value setup</td>
<td>O</td>
</tr>
<tr>
<td>G11</td>
<td>G11</td>
<td>G11</td>
<td>01</td>
<td>Beveling</td>
<td>Cornering</td>
</tr>
<tr>
<td>G12</td>
<td>G12</td>
<td>G12</td>
<td>02</td>
<td>Rounding</td>
<td>O</td>
</tr>
<tr>
<td>G20</td>
<td>G20</td>
<td>G20</td>
<td>05</td>
<td>Inch input specification</td>
<td>O</td>
</tr>
<tr>
<td>G21</td>
<td>G21</td>
<td>G21</td>
<td></td>
<td>Metric input specification</td>
<td>O</td>
</tr>
<tr>
<td>G22</td>
<td>G22</td>
<td>G22</td>
<td>01</td>
<td>Radius programming for circular interpolation CW</td>
<td>O</td>
</tr>
<tr>
<td>G23</td>
<td>G23</td>
<td>G23</td>
<td>02</td>
<td>Radius programming for circular interpolation CCW</td>
<td>O</td>
</tr>
<tr>
<td>G27</td>
<td>G27</td>
<td>G27</td>
<td></td>
<td>Reference point return check</td>
<td>B</td>
</tr>
<tr>
<td>G28</td>
<td>G28</td>
<td>G28</td>
<td></td>
<td>Automatic return to reference point</td>
<td>B</td>
</tr>
<tr>
<td>G29</td>
<td>G29</td>
<td>G29</td>
<td></td>
<td>Return from reference point</td>
<td>B</td>
</tr>
<tr>
<td>G30</td>
<td>G30</td>
<td>G30</td>
<td></td>
<td>Return to 2nd reference point</td>
<td>O</td>
</tr>
<tr>
<td>G31</td>
<td>G31</td>
<td>G31</td>
<td></td>
<td>Skip function</td>
<td>O</td>
</tr>
<tr>
<td>G32</td>
<td>G32</td>
<td>G32</td>
<td>01</td>
<td>Threadcutting, continuous threadcutting, multi-start threadcutting</td>
<td>B, O</td>
</tr>
<tr>
<td>G34</td>
<td>G34</td>
<td>G34</td>
<td></td>
<td>Variable lead threadcutting</td>
<td>O</td>
</tr>
<tr>
<td>G35</td>
<td>G35</td>
<td>G35</td>
<td></td>
<td>Tool set error compensation</td>
<td>O</td>
</tr>
<tr>
<td>G36</td>
<td>G36</td>
<td>G36</td>
<td>07</td>
<td>Stored stroke limit 2nd area ON</td>
<td>O</td>
</tr>
<tr>
<td>G37</td>
<td>G37</td>
<td>G37</td>
<td></td>
<td>Stored stroke limit 2nd area OFF</td>
<td>O</td>
</tr>
<tr>
<td>G38</td>
<td>G38</td>
<td>G38</td>
<td>08</td>
<td>Stored stroke limit 3rd area ON</td>
<td>O</td>
</tr>
<tr>
<td>G39</td>
<td>G39</td>
<td>G39</td>
<td></td>
<td>Stored stroke limit 3rd area OFF</td>
<td>O</td>
</tr>
<tr>
<td>G40</td>
<td>G40</td>
<td>G40</td>
<td>06</td>
<td>Tool radius compensation cancel</td>
<td>O</td>
</tr>
<tr>
<td>G41</td>
<td>G41</td>
<td>G41</td>
<td></td>
<td>Tool radius compensation No. 1</td>
<td>O</td>
</tr>
<tr>
<td>G42</td>
<td>G42</td>
<td>G42</td>
<td></td>
<td>Tool radius compensation No. 2</td>
<td>O</td>
</tr>
<tr>
<td>G43</td>
<td>G43</td>
<td>G43</td>
<td></td>
<td>Tool radius compensation No. 3</td>
<td>O</td>
</tr>
<tr>
<td>G44</td>
<td>G44</td>
<td>G44</td>
<td></td>
<td>Tool radius compensation No. 4</td>
<td>O</td>
</tr>
</tbody>
</table>

**Note:**
1. G codes in groups from 01 through 11 are modal. When the control is energized with the power switch or reset, the G codes marked with * are automatically selected.
   For G00/G01, G98/G99, and G90/G91, either one is selected as initial state by setting parameters.
2. G codes of * group are non-modal. They should not be commanded together with the other G codes in one block.
3. The modal G codes can be commanded mixedly in a block.
4. G codes in section B are basic.
5. Standard G codes can be converted to special G codes I by parameters. (basic feature)
6. Special G code II can be selected as optional function. When selected, the standard G codes and special G code II cannot be used.
7. The initial states of G codes of 05, 07, 08 group when the control is powered correspond to their respective setting data.
# List of G Codes

<table>
<thead>
<tr>
<th>G Code</th>
<th>Special G Code I</th>
<th>Special G Code II</th>
<th>Group</th>
<th>Function</th>
<th>Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>G 50</td>
<td>G 92</td>
<td>G 92</td>
<td></td>
<td>Coordinate system setup</td>
<td>B</td>
</tr>
<tr>
<td>G 51</td>
<td>G 51</td>
<td>G 51</td>
<td></td>
<td>Maximum spindle revolution setup</td>
<td>O</td>
</tr>
<tr>
<td>G 55</td>
<td>G 55</td>
<td>G 55</td>
<td></td>
<td>Return of current display value to origin</td>
<td>O</td>
</tr>
<tr>
<td>G 66</td>
<td>G 66</td>
<td>G 66</td>
<td></td>
<td>User macro simple call</td>
<td>O</td>
</tr>
<tr>
<td>G 67</td>
<td>G 67</td>
<td>G 67</td>
<td></td>
<td>User macro modal call</td>
<td>O</td>
</tr>
<tr>
<td>G 68</td>
<td>G 68</td>
<td>G 68</td>
<td></td>
<td>User macro modal call cancel</td>
<td>O</td>
</tr>
<tr>
<td>G 69</td>
<td>G 69</td>
<td>G 69</td>
<td></td>
<td>Mirror image by programming ON</td>
<td>O</td>
</tr>
<tr>
<td>G 70</td>
<td>G 70</td>
<td>G 72</td>
<td></td>
<td>Mirror image by programming OFF</td>
<td>O</td>
</tr>
<tr>
<td>G 71</td>
<td>G 71</td>
<td>G 73</td>
<td></td>
<td>Finishing cycle</td>
<td>O</td>
</tr>
<tr>
<td>G 72</td>
<td>G 72</td>
<td>G 74</td>
<td></td>
<td>Stock removal in turning</td>
<td>O</td>
</tr>
<tr>
<td>G 73</td>
<td>G 73</td>
<td>G 75</td>
<td></td>
<td>Stock removal in facing</td>
<td>O</td>
</tr>
<tr>
<td>G 74</td>
<td>G 74</td>
<td>G 76</td>
<td></td>
<td>Pattern repeating</td>
<td>O</td>
</tr>
<tr>
<td>G 75</td>
<td>G 75</td>
<td>G 77</td>
<td></td>
<td>Multiple repetitive cycles</td>
<td>O</td>
</tr>
<tr>
<td>G 76</td>
<td>G 76</td>
<td>G 78</td>
<td></td>
<td>Autotmatic threadcutting cycle</td>
<td>O</td>
</tr>
<tr>
<td>G 90</td>
<td>G 77</td>
<td>G 20</td>
<td></td>
<td>Turning cycle A</td>
<td>B</td>
</tr>
<tr>
<td>G 92</td>
<td>G 78</td>
<td>G 21</td>
<td></td>
<td>Threading cycle</td>
<td>B</td>
</tr>
<tr>
<td>G 94</td>
<td>G 79</td>
<td>G 24</td>
<td></td>
<td>Facing cycle B</td>
<td>B</td>
</tr>
<tr>
<td>G 96</td>
<td>G 96</td>
<td>G 96</td>
<td></td>
<td>Constant surface speed control</td>
<td>O</td>
</tr>
<tr>
<td>G 97</td>
<td>G 97</td>
<td>G 97</td>
<td></td>
<td>Constant surface speed control cancel</td>
<td>O</td>
</tr>
<tr>
<td>G 98</td>
<td>G 94</td>
<td>G 94</td>
<td></td>
<td>Feed per minute (mm/min)</td>
<td>B</td>
</tr>
<tr>
<td>G 99</td>
<td>G 95</td>
<td>G 95</td>
<td></td>
<td>Feed per revolution (mm/rev)</td>
<td>B</td>
</tr>
<tr>
<td>G 90</td>
<td>G 90</td>
<td>G 90</td>
<td></td>
<td>Absolute command</td>
<td>B</td>
</tr>
<tr>
<td>G 91</td>
<td>G 91</td>
<td>G 91</td>
<td></td>
<td>Incremental command</td>
<td>B</td>
</tr>
<tr>
<td>G 122</td>
<td>G 122</td>
<td>G 122</td>
<td></td>
<td>Tool registration start</td>
<td>B</td>
</tr>
<tr>
<td>G 123</td>
<td>G 123</td>
<td>G 123</td>
<td></td>
<td>Tool registration end</td>
<td>B</td>
</tr>
<tr>
<td>G 111</td>
<td>G 111</td>
<td>G 111</td>
<td></td>
<td>Taper multiple beveling/rounding</td>
<td>B</td>
</tr>
<tr>
<td>G 112</td>
<td>G 112</td>
<td>G 112</td>
<td></td>
<td>Arc multiple beveling/rounding</td>
<td>B</td>
</tr>
<tr>
<td>G 150</td>
<td>G 150</td>
<td>G 150</td>
<td></td>
<td>Cancel groove width compensation</td>
<td>O</td>
</tr>
<tr>
<td>G 151</td>
<td>G 151</td>
<td>G 151</td>
<td></td>
<td>Groove width compensation</td>
<td>O</td>
</tr>
</tbody>
</table>

Note:
1. G codes in groups from 01 through 11 are modal. When the control is energized with the power switch or reset, the G codes marked with * are automatically selected. For G00/G01, G 96/G 99, and G 90/G 91, either one is selected as initial state by setting parameters.
2. G codes of * group are non-modal. They should not be commanded together with the other G codes in one block.
3. The modal G codes can be commanded mixedly in a block.
4. G codes in section B are basic.
5. Standard G codes can be converted to special G codes I by parameters. (basic feature)
6. Special G code II can be selected as optional function. When selected, the standard G codes and special G code II cannot be used.
7. The initial states of G codes of 05, 07, 08 group when the control is powered correspond to their respective setting data.
## Typical M Codes

<table>
<thead>
<tr>
<th>M code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Program stop</td>
</tr>
<tr>
<td>M01</td>
<td>Optional stop</td>
</tr>
<tr>
<td>M02</td>
<td>End of program (does not rewind program)</td>
</tr>
<tr>
<td>M03</td>
<td>Spindle on in a clockwise direction</td>
</tr>
<tr>
<td>M04</td>
<td>Spindle on in a counter clockwise direction</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle stop</td>
</tr>
<tr>
<td>M08</td>
<td>Flood coolant on</td>
</tr>
<tr>
<td>M09</td>
<td>Coolant off</td>
</tr>
<tr>
<td>M10</td>
<td>Step part counter</td>
</tr>
<tr>
<td>M13</td>
<td>Bar feeder on</td>
</tr>
<tr>
<td>M14</td>
<td>Chuck jaws close</td>
</tr>
<tr>
<td>M15</td>
<td>Chuck jaws open</td>
</tr>
<tr>
<td>M16</td>
<td>Tailstock quill forward</td>
</tr>
<tr>
<td>M17</td>
<td>Tailstock quill back</td>
</tr>
<tr>
<td>M28</td>
<td>Tailstock body forward</td>
</tr>
<tr>
<td>M29</td>
<td>Tailstock body back</td>
</tr>
<tr>
<td>M41</td>
<td>Low spindle range</td>
</tr>
<tr>
<td>M42</td>
<td>High spindle range</td>
</tr>
<tr>
<td>M30</td>
<td>End of program (rewinds memory)</td>
</tr>
<tr>
<td>M98</td>
<td>Subprogram call</td>
</tr>
<tr>
<td>M99</td>
<td>End of subprogram</td>
</tr>
</tbody>
</table>


Spindle Speed Choices

I. Constant surface speed [G96]
   A. Use for operations:
      1] Contouring operations (turning)
      2] Boring operations
      3] Grooving tools
   B. Feedrate in I.P.R. remains constant
   C. Improved tool life

example: N005 G96 X300
         (Selects speed of 300 S.F.P.M.)

II. RPM [G97]
    A. Use for operations
       1] Machining on spindle centerline
          a. drilling
          b. reaming
          c. tapping
       2] Threading
          a. RPM and Feedrate must be synchronized

example: N005 G97 S300
         (Selects speed of 300 RPM)
Nakamora CNC Lathe

Feedrate Choices

I. Inches per revolution (i.p.r.)
   A. Maintains even witness marks on workpiece
   B. Use for almost all turning center operations

   example:    N010 G99 F.015
               (Selects .015 inches per revolution)

II. Inches per minute
   A. Some preset for knurling
   B. Use for stopping spindle
      (i.e. bar feed move)

   example:    N010 G98 F 5- 0
               (Selects 5 inches per minute)
Turning Tools

Spindle Direction
M03  CW  For R.H. cut tools
M04  CCW For L.H. cut tools

Example of right hand tool holder.
T Codes

Example: T0505

T05 = Turret station #5
05 = Offset number 5 (wear offset)

Note: Tool coordinate offset is derived by adding 50 to the value of index position. Thus, for position of the offset coordinate is 51.

Geometry Offsets = Used to designate program zero location

Wear Offsets = Workpiece sizing and tool nose radius compensation
Most CNC Controls have at least 16 offsets. Some have 32, 99 and more.

The offset table may look like this.

<table>
<thead>
<tr>
<th>#</th>
<th>X</th>
<th>Z</th>
<th>R</th>
<th>T</th>
</tr>
</thead>
</table>

Where:

- \#
- X
- Z
- R
- T
Tool offset (T) in tool offset register direction of imaginary tool nose

Numbers 1 - 4: quadrant numbers
Numbers 5 - 8: point to +Z, +X, -Z, and -X
Programming a turned part

For the part sketch shown below, write a NC program to finish turn this part.
<table>
<thead>
<tr>
<th>Program Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>00002</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N005</th>
</tr>
</thead>
<tbody>
<tr>
<td>N010</td>
</tr>
<tr>
<td>N015</td>
</tr>
<tr>
<td>N020</td>
</tr>
<tr>
<td>N025</td>
</tr>
<tr>
<td>N030</td>
</tr>
<tr>
<td>N035</td>
</tr>
<tr>
<td>N040</td>
</tr>
<tr>
<td>N045</td>
</tr>
<tr>
<td>N050</td>
</tr>
<tr>
<td>N055</td>
</tr>
<tr>
<td>N060</td>
</tr>
<tr>
<td>N065</td>
</tr>
<tr>
<td>N070</td>
</tr>
<tr>
<td>N075</td>
</tr>
<tr>
<td>N080</td>
</tr>
<tr>
<td>N085</td>
</tr>
<tr>
<td>N090</td>
</tr>
<tr>
<td>N095</td>
</tr>
<tr>
<td>N100</td>
</tr>
</tbody>
</table>
Sub-program example used for warn up of turning center.

Main program

00001
(overnight dwell program for spindle warm up)

N005 M98 P1000 L6
L work specified the number of hours of dwell

N010 G97 S1500 M03
Turn spindle on at 1500 RPM

N020 M30
End of program

Sub Program

01000
Actual dwelling program

N1G04 X3600
Dwell for 3600 seconds, one hour

N10 M99
Return to main program
Nakamora CNC Lathe

Basic lathe formulas

Conversion factors

To convert from "inches per revolution" (ipr) to "Inches per minute" (ipm), or vice versa, use the following:

\[ \text{ipr}^* = \text{RPM} = \text{ipm} \]
\[ \text{ipr} = \frac{\text{ipm}}{\text{RPM}} \]

To convert from "surface feet per minute" (SFM) to "RPM" or vice versa, use the following:

\[ \text{SFM} = \frac{3.1416 \times \text{RPM} \times \text{dia.}}{12} \]

\[ \text{RPM} = \frac{12 \times \text{SFM}}{3.1416 \times \text{dia.}} \]
Determining time required to perform an operation:

A. Find RPM

\[ \text{RPM} = \frac{12 \times \text{SFM}}{3.1416 \times \text{dia.}} \]

B. Time in seconds

\[ \text{Time (secs)} = \frac{\text{Distance to go} \times 60 \text{ secs.}}{\text{i pr} \times \text{RPM}} \]

Solve the following problem:
How long will it take to turn a part in one pass to a one inch diameter at 350 SFM and .004 ipr over 3 inches?

Answer:  
- RPM 
- Time in seconds
1. Special fixed cycle

G70: Finishing cycle (effective after G71, G72, G73)

G71: Outer diameter rough cutting cycle

G72: Face rough cutting cycle

G73: Closed loop cutting cycle

...... Cutting feed is done while repeating a certain cutting pattern.

G74: Face cutting-off cycle

...... Effective during drilling

G75: Outer diameter cutting-off cycle

G76: Threading cutting cycle

---

(1) Finishing cycle

After rough cutting by G71, G72 and G73, finishing cycle can be performed by the following command.

Command system

\[ G70 \ P_{(ns)} \ Q_{(nf)} \]

P: Sequence number at the beginning of cycle (ns)
Q: Sequence number at the end of cycle (nf)

Application example is given together with G71, G72 and G73.

(Note 1) With F and S code, codes commanded between sequence number ns and nf become effective.

(Note 2) Tip nose radius compensation is possible with this cycle.

(Note 3) Sequence number and block designated by P and Q should be placed before the block of G70.

(Note 4) Program sequence number at the beginning of the block of G70.
(2) G71 Outer diameter rough cutting cycle

This cycle is achieved by designating cutting conditions and finishing dimensions.

Command system

\[
G71 \quad P_{(ns)} \quad Q_{(nf)} \quad U_{(\Delta u)} \quad W_{(\Delta w)} \quad D_{(\Delta d)} \quad F_{(f)} \quad S_{(s)}
\]

\[P:\] Sequence number at the beginning of cycle (ns)
\[Q:\] Sequence number at the end of cycle (nf)
\[U:\] Distance and direction of cutting tool relief in X axis direction (\(\Delta u\)) —— Diameter designation
\[W:\] Distance and direction of cutting tool relief in Z axis direction (\(\Delta w\)) —— Finishing allowance
\[D:\] Cutting feed amount/each time (\(\Delta d\)) —— radius designation
\[F,S:\] F or S function is ignored when it is placed in P or Q block. F or S function which is placed in the block before G71 is effective.

(Note 1) F or S function cannot be changed while G71 cycle is in operation.

(Note 2) Tip nose radius compensation is not effective for this cycle.

(Note 3) Multi-quadrant cutting is not possible with this cycle.

(Note 4) Program sequence number at the beginning of the block of G71.
(4) G73 Closed loop cutting cycle

This cycle is achieved by designating cutting conditions and finishing dimensions.

Command system

\[
\text{G73 } \text{P} (\text{ns}) \text{ Q} (\text{nf}) \text{ I} (i) \text{ K} (k) \text{ U} (\Delta u) \text{ W} (\Delta w) \text{ D} (\Delta d) \text{ S} (s)
\]

P: Sequence number at the beginning of cycle (ns)
Q: Sequence number at the end of cycle (nf)
I: Distance and direction of cutting tool relief in X axis direction (i) ——— Radius designation
K: Distance and direction of cutting tool relief in Z axis direction (k)
U: Finishing allowance in X axis direction (\Delta u) ——— Diameter designation
W: Finishing allowance in Z axis direction (\Delta w)
D: Number of cutting

(Note 1) S function cannot be changed while G73 cycle is in operation.
(Note 2) Tool tip "R" compensation is not effective for this cycle.
(Note 3) Multi-quadrant cutting is not possible with this cycle.
(Note 4) Four different cutting shapes are produced according to the mark used. Therefore, care should be taken in programming.
(Note 5) Program sequence number at the beginning of the block of G73.
G70 G71 Example

n001 g50 x-20000 z1000
n002 g00 t0101 (M41 or M42)
 n003 g96 s400 m03
n004 g00 x-7800 z500 m08
n005 g71 p006 q011 u200 w100 d700 f30
n006 g00 x-4000
n007 g01 z-2000 f20
n008 g03 x-5000 z-3000 i-10 j0
n009 g01 x-6000 z-5000
n10 z-6000
n011 x-9000
n012 g00 x-20000 z1000
n013 t0100
n014 g50 x-20000 z1000
n015 t0202
n016 g96 s500 m03
n017 g70 p006 q011
n018 g00 x-20000 z1000 m09
n019 t0200
n020 m02 (or M30)
G70 G73 Example

n001 g50 x-20000 z10000
n002 g00 t0101 (M41 or M42)
n003 g96 s300 m03
n004 g00 x-12000 z10000 m08
n005 g73 p006 q011 i1000 k500 u200 w200 d3 f30
n006 g00 x-4000 z200
n007 g01 z-1500 f20
n008 x-6400 z-2500
n009 z-3700
n010 g03 x-8000 z-4500 i-8000
n011 g01 x-9000 z-5000
n012 g00 x-20000 z10000
n013 t0100
n014 g50 x-20000 z10000
n015 t0202
n016 g96 s350 m03
n017 g70 p006 q007
n018 g00 x-20000 z10000 m09
n019 t0200
n020 m02 (or M30)
<table>
<thead>
<tr>
<th></th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>G73 closed loop waiting cycle is also referred to as &quot;contour repetitive cycle&quot;</td>
</tr>
<tr>
<td>2.</td>
<td>G70 can be regarded as a single-pass contouring cycle. It cannot be used alone; it must be used with one of the three stock removal cycles.</td>
</tr>
<tr>
<td>3.</td>
<td>G73 is best suited for some for GED and casted workpieces that have been made to their rough shapes in which the dimensions are only slightly larger than the finished part size. Use of G71 on this part type would not be efficient because the diameter of the workpieces at various section is not the same.</td>
</tr>
<tr>
<td>4.</td>
<td></td>
</tr>
<tr>
<td>5.</td>
<td></td>
</tr>
</tbody>
</table>
1) Name the two most basic operation panels on a CNC turning center.

________________________________________________________________________

For the following descriptions, name the control panel button which activates the feature.

2) Turns on power to the control panel.

________________________________________________________________________

3) Turns on power to the machine unit.

________________________________________________________________________

4) Turns off power to the machine unit.

________________________________________________________________________

5) Shows display screen position display.

________________________________________________________________________

6) Shows the active program.

________________________________________________________________________

7) Shows the offset table.

________________________________________________________________________

8) Enters data except when editing programs.

________________________________________________________________________

9) Allows program data to be changed.

________________________________________________________________________

10) Allows new program data to be entered.

________________________________________________________________________

11) In the edit mode, this button rewinds the program back to the beginning.

________________________________________________________________________

12) This series of buttons is on some Fanuc controls below the display screen.

________________________________________________________________________
For the following descriptions, name the machine panel button which activates the feature.

1) This multi position switch is the heart of the CNC control.
   __________________________

2) This button stops axis motion (only).
   __________________________

3) This button activates the program.
   __________________________

4) This multi position switch controls the motion rate in the cutting mode.
   __________________________

5) This switch controls the motion rate in the rapid mode.
   __________________________

6) This switch gives the operator total control of all movements the machine makes.
   __________________________

7) This switch makes the machine stop after every command.
   __________________________

8) When this switch is on, absolutely no axis motion can occur.
   __________________________

9) This switch works in conjunction with an M01.
   __________________________

10) This device allows the operator to manually move the axes with a dial.
    __________________________

11) This device allows the operator to jog the axes.
    __________________________

12) This meter allows the operator to monitor how much stress the spindle is currently under.
    __________________________
1) Name the three most basic modes of operation.

________________________
________________________
________________________

2) Which mode switch positions are included in each mode?

________________________
________________________
________________________

3) In which mode does the machine behave the most like an engine lathe?

________________________

4) In which mode does the operator enter commands through the display screen and keyboard?

________________________

5) In which mode is the operator actually running workpieces?

________________________

6) In which mode is the operator editing programs?

________________________
1) What is the G code used to specify a dwell?

2) What happens during a dwell command?

3) What are the three words that can be used to specify the length of time in a dwell command?

4) What word is used to specify an optional block skip command?

5) When can the optional block skip command be used?

6) What word is used to specify the radius required when automatic corner rounding and chamfering is used?

7) What are the two words used to specify chamfers when automatic corner rounding and chamfering is used? What are their functions?
1) What are the four words involved with subprogramming and what are their meanings?

__________________________________________________________________________

__________________________________________________________________________

__________________________________________________________________________

__________________________________________________________________________

2) What is the most basic purpose for using subprogramming techniques.

__________________________________________________________________________

__________________________________________________________________________

3) Name two possible applications for subprograms.

__________________________________________________________________________

__________________________________________________________________________

4) What command can be used to end a main program that will not cause the cycle to stop (commonly used for bar feeding programs)?

__________________________________________________________________________
During this segment, participants will:

1. Review troubleshooting items pertaining to CNC machining.
CNC Troubleshooting

Problems Caused by tooling:

1. Dimensions
2. Finish of workpiece
3. Number of parts

What is tooling:

1. Workholding device
2. Tool holder
3. Clamps
4. Other
CNC Troubleshooting

Cutting Tool Problems:

What is tooling:

1. Edge (Blue or Burned)
2. Nicked/chipped
3. Rake - material buildup
4. Out of round or bent

Chatter - indicator of trouble.
Causes of Chatter

1. Dull tool - most common problem
2. Newly sharpened tool - overly sharp tool
3. Feed rate - increase or decrease
4. Spindle speed - change speed

*Example*: Feed of F4.0 i.p.m. Works best at 70% override
4.0 * .70 = 2.8
Thus 2.8 (F2.8) is feed rate to be using.

5. Loose fixture or workpiece
CNC Troubleshooting

Causes of Chatter

6. Lack of coolant
7. Wrong tool used (HSS)
8. Spindle rotation wrong
9. Wrong canned cycle (G81)
CNC Troubleshooting

Operator Input
Welcome to the Dana C.N.C. Operator Assessment Exam.

You are taking this examination to demonstrate your skills in the areas of:

- Shop math
- Blueprint reading and specification sheet reading
- Reasoning skills including:
  - Logical start-up and shut-down procedures
  - Measuring instrument identification
  - Tool identification

This assessment was developed by a team representing Dana C.N.C. operators, supervisors, training staff and management with Anoka-Hennepin Technical College.
Your Goal

Your goal is to correctly answer 75% of more of the questions to receive a passing score.

<table>
<thead>
<tr>
<th>Area of Study</th>
<th>Questions:</th>
<th>Your Goal:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Math</td>
<td>Questions 1 - 30</td>
<td>22 problems correct</td>
</tr>
<tr>
<td>Blueprint reading</td>
<td>Questions 31 - 76</td>
<td>34 problems correct</td>
</tr>
<tr>
<td>Reasoning skills</td>
<td>Questions 77 - 97</td>
<td>15 problems correct</td>
</tr>
</tbody>
</table>

Passing each section...
After you take the three sections of this exam, you will be asked to demonstrate your hands-on machining skills in a 30 to 45 minute assessment on the shop floor during your regular shift.

An instructor, selected by Anoka-Hennepin Technical College's (AHTC) Custom Training Division, will assess your skills in a consistent and fair manner.
Passing Each Section

If you do not pass the math section:
• You will be asked to take a mathematics refresher course at Dana, presented by AHTC

If you do not pass the blueprint reading section:
• You will be asked to take a blueprint reading course at Dana, presented by AHTC

If you do not pass the reasoning skills section:
• You will be asked to take a C.N.C. theory and hands-on course at Dana, presented by AHTC

If you do not pass the shop floor assessment:
• You will be asked to take a C.N.C. theory and hands-on course at Dana, presented by AHTC
Section 1:
Math Skills
<p>| | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Add</td>
<td>13</td>
<td>33</td>
<td>125</td>
<td>109</td>
</tr>
<tr>
<td>2. Add</td>
<td>47111</td>
<td>3134</td>
<td>256</td>
<td>9989</td>
</tr>
</tbody>
</table>
3. Subtract

\[
\begin{array}{c}
156 \\
- 34 \\
\hline
\end{array}
\]

A

10"

B

5

2"

C

1"

D

1"

5. Find the lengths of A, B, C, & D

\[
\begin{array}{c}
1,564 \\
- 198 \\
\hline
\end{array}
\]

1"
6. Multiply

\[
93 \\times 12
\]

7. Solve through Multiplication

A machine screw has 8 threads to the inch. How many threads are there in a threaded piece 7 inches long?

8. Solve through Division

A machine shop owner spends $2,368 to buy tools. Each tool costs an average of $16. How many tools can be bought?
Math Skills

10. \[
\begin{array}{c}
\phantom{0} .01 \\
- .015 \\
\hline
\phantom{0}
\end{array}
\]

11. \[
\begin{array}{c}
\phantom{0} .200 \\
+ .210 \\
\hline
\phantom{0}
\end{array}
\]
12. Circle your answer.

Which is greater?

0.015 or -0.026

13. Circle your answer.

Which is greater?

-0.024 or -0.026
The pitch of a thread is the distance from a given point on one thread to the corresponding point on the next thread. Pitch is usually expressed as a fraction. Machinists sometimes make the mistake of calling the number of threads in one inch the pitch. For example, a one-inch screw has 8 threads to the inch. The pitch of this thread is 1/8 inch. It is wrong to say that it is an 8 pitch thread.

14. A screw 4 inches long has 52 threads.
   a. How many threads per inch are there? _________________________
   b. What is the pitch? _________________________

15. A screw is 3 1/2 inches long and has 56 threads.
   a. Find the number of threads per inch. _________________________
   b. Find the pitch of the screw. _________________________
16. What is the total length of the template (dimension L)?
17. What is the total height H?
18. Find dimension A.
19. What is dimension B?
20. Find dimension C.
21. What is dimension D?
<table>
<thead>
<tr>
<th></th>
<th>Math Skills</th>
</tr>
</thead>
<tbody>
<tr>
<td>22.</td>
<td>Subtract</td>
</tr>
<tr>
<td></td>
<td>56.9</td>
</tr>
<tr>
<td></td>
<td>3.6</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>23.</td>
<td>Subtract</td>
</tr>
<tr>
<td></td>
<td>9883.456</td>
</tr>
<tr>
<td></td>
<td>298.179</td>
</tr>
<tr>
<td></td>
<td>298.179</td>
</tr>
<tr>
<td>24.</td>
<td>Multiply</td>
</tr>
<tr>
<td></td>
<td>3.4567</td>
</tr>
<tr>
<td></td>
<td>3.9876</td>
</tr>
<tr>
<td></td>
<td>13.450</td>
</tr>
<tr>
<td>25.</td>
<td>Divide</td>
</tr>
<tr>
<td></td>
<td>5.35</td>
</tr>
<tr>
<td></td>
<td>29.425</td>
</tr>
<tr>
<td></td>
<td>5.425</td>
</tr>
</tbody>
</table>
26. Express these subdivisions of an inch in machinists' terms.

Two problems have been done for you.

27. Convert the fraction 7/8 to a decimal number ________

28. Convert .75 to a fraction

.000001 Millionth
.00001
.0001
.001
.01 Hundredth
29. Circle the correct answer.

You have a 1 5/16 - 12 tap and you need to go one turn deeper, what numerical value would you assign?

a) .083  
b) .083  
c) .016

30. Circle the correct answer.

You have a 1 5/16 - 12 tap and you're two turns shallow, what numerical value would you assign?

a) .317  
b) .166  
c) -.166
Section 2: Print Reading Skills
Use the folded print found in your assessment package.

31. Identify the 4 views on the print:
   Write: top, bottom, end, back.
   The front view is the second one down on the left side of the print.

32. What is the name of the casting?

33. What reference would you use to determine what the casting is made out of?

34. What is the decimal tolerance of this print?

35. What is the latest revision?

36. What is the scale of the print?

37. Draw a finish symbol in the space provided.

38. When a finish is not specified on a view, what finish is required?
Throughout this assessment, symbols, like the one above, will be shown as: —A—

39. What features determine —A—?
   Hint: draw the symbols below

40. What is the starting depth from —A—?
   __________________________________________

41. How would you establish —B—?
   Hint: draw the symbols below

42. What is the overall thickness of the part?
   __________________________________________

43. What is the inlet/port SAE #16 location?
   From —E— ____________________________
   From —H— ____________________________

44. What number would you use to determine the correct Gresen spec sheet to reference this inlet/port?
   __________________________________________
Use the correct Gresen spec sheet, provided in your packet, to help you with the following questions.

45. What is the size of thread SAE #16?

______________________________

46. What does the 12 in your answer above represent?

______________________________

47. Thread SAE #16 is .750 deep. How many turns are required?

______________________________

48. What is the minor thread diameter of SAE #16? Use your spec sheet

______________________________

49. What is the U diameter of SAE #16?

______________________________

50. What is the tolerance zone of the U diameter?

______________________________
51. **Look at the relief Eng Spec 2.003.049 on the print**
What is the location of the center of the relief.

   From –E– __________________________
   From –A– __________________________

52. On the back view, how many counterbores [C'BORE] are shown?

53. What is the diameter of the counterbores?
   *(Show minimum and maximum)*

54. What is the minimum and maximum depth of the counterbores?

55. How many 3/8-16 UNC - 2B (bolt holes) are shown on the front view?
56. What is the minimum and maximum drill depth of 3/8-16 UNC - 2B (bolt holes) on the front view?

57. What is the minimum and maximum thread depth of 3/8-16 UNC - 2B (bolt holes) on the front view?

58. How many 7/16-14 UNC - 2B (bolt holes) are shown on the back view?

59. What is the minimum and maximum drill depth of 7/16-14 UNC - 2B (bolt holes) on the back view?

60. What is the minimum and maximum thread depth of 7/16-14 UNC - 2B (bolt holes) on the back view?

61. How many Trepan Grooves are located on the:
   - Front view
   - Back view

62. Are all the Trepans the same on the front and back views?
List the spec numbers and the quantity of Trepan Grooves located on the print.

<table>
<thead>
<tr>
<th>Spec</th>
<th>Quantity</th>
<th>Spec</th>
<th>Quantity</th>
<th>Spec</th>
<th>Quantity</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

67. Refer to the Gresen Spec sheet for the following question

What are the depths of the Trepans listed above?
Refer to Gresen spec sheet number 2.003.070 - 2.003.077 for Trepan Grooves.

What are the minimum and maximum tolerances for diameter $\theta_A$?

<table>
<thead>
<tr>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>.071</td>
<td>.071</td>
</tr>
</tbody>
</table>
69. Refer to Gresen spec sheet number 2.003.070 - 2.003.077 for Trepan Grooves. What are the minimum and maximum tolerances for diameter **Ø B**?

<table>
<thead>
<tr>
<th>For</th>
<th>Min</th>
<th>Max</th>
</tr>
</thead>
<tbody>
<tr>
<td>.075</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
70. What is the maximum and minimum size of the tooling holes shown on the bottom view?

71. How many tooling holes are shown on the bottom view?

72. What is the maximum and minimum depth of the tooling holes shown on the bottom view?

73. What is the distance between the tooling holes shown on the bottom view from surface E?

74. The two tooling holes are dimensioned from surface H. What are the dimensions?

75. Look at the back view. What is the vertical dimension from the lower counterbore hole to the centerline of the top Trepan?

76. What is the finish required for the relief spec 2.003.049 from surface A?

Hint: Check the spec sheet.
Section 3: Reasoning Skills
77. **Start-Up Procedures for C.N.C. Machines**

Number the following start up procedure steps in the order that you think is most reasonable. A total of seven steps.

*Hint: There are several reasonable ways to number these.*

- Reset E stop
- Assign program
- Check oil, grease and coolant
- Turn main power on
- Check pallet assignment for job
- Home out or zero out axes
- Verify tool and fixture offsets
78. **End of Shift Shut Down Procedures for C.N.C. Machines**

Number the following end-of-shift shut down procedure steps in the order that you think is most reasonable. A total of four steps.

- Depress E stop
- Jog X, Y, Z & B axes off home positions
- Switch main power off (if required)
- Depress master stop
Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.

79. a) Depth micrometer  
    b) Protractor  
    c) Height gage

80. a) Blade micrometer  
    b) Thickness gage  
    c) One inch micrometer
Circle the letter that corresponds with the correct instrument term.

81.
- a) Protractor
- b) Vernier
- c) Scale

82.
- a) Feeler or thickness gage
- b) Dial indicator
- c) Thread gage
Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.

83.
- a) Disc micrometer
- b) One inch micrometer
- c) Blade micrometer

84.
- a) Depth micrometer
- b) Vernier
- c) Protractor
Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.

85. a) Height gage  
    b) Thread gage  
    c) Depth micrometer

86. a) Disc micrometer  
    b) Dial indicator  
    c) Gage blocks
Identify the Measuring Instruments

Circle the letter that corresponds with the correct instrument term.

87.  
a) Vernier  
b) Dial indicator  
c) Thread gage

88.  
a) Telescopic gages  
b) Depth micrometers  
c) Protractors
Circle the letter that corresponds with the correct instrument term.

89. a) Height gages  
     b) Gage blocks  
     c) Thread gages
Circle the letter that corresponds with the correct tool term.

91.  a) Straight thread tap  
     b) Pipe tap  
     c) Drill/counterbore combo.

90.  a) Drill  
     b) Pipe tap  
     c) Reamer
Identify the Tools

Circle the letter that corresponds with the correct tool term.

92.
- a) Pipe tap
- b) Straight shank reamer
- c) Drill

93.
- a) Reamer
- b) Drill
- c) Straight thread tap
Identify the Tools

Circle the letter that corresponds with the correct tool term.

94.  
   a) Straight thread tap  
   b) Straight shank reamer  
   c) Drill
You bore and ream a hole and find the hole is tapered. What may have caused the problem?
96. Fill in the blanks with your answer.

You're running a job and notice a feature has chatter.
What would you consider before you change the feeds and speeds?
Fill in the blanks with your answer.

You're at a machine and you hear a noise that could lead to tool failure.

How should you react?
Basic CNC Operation

Hands-On Assessment
Assessment Tasks 1 - 3

Scoring Key: Your goal is to score a 4 or 5 in each area.
5 = Outstanding  4 = Above Average  
3 = Average    2 = Below Average   1 = Poor

The operator...

1. Start up and shut down
A. Demonstrates start up procedure 1 2 3 4 5
B. Demonstrates shut down procedure 1 2 3 4 5

2. Routine maintenance
A. Determines oil levels 1 2 3 4 5
B. Determines coolant levels 1 2 3 4 5

3. Controls
A. Identifies manual data input 1 2 3 4 5
B. Identifies auto run 1 2 3 4 5
C. Identifies set up modes 1 2 3 4 5
D. Identifies dry run feature 1 2 3 4 5
E. Identifies single block feature 1 2 3 4 5
4. Basic editing

A. Calls up program from memory
B. Identifies differences between sub programs and main programs
C. Edits main or sub program
D. Inserts tool radius and length offsets
E. Inserts work coordinate offsets
F. Identifies custom macro blocks of information

5. Machine set-up

A. Mill: Sets X-Ø, Y-Ø and Z-Ø
   Using edge finder or dial indicator or probe
   Lathe: Sets X-Ø and Z-Ø
   Using probe or tools
B. Pre-sets tools with tool presetter
   [optional: use touch off method]
C. Utilizes one or more work coordinate offset G54 – G59
D. Identifies tool holder styles and numbers
   Mill: BT, CAT, Milacron, etc.
   Lathe: right or left hand tool holder
E. Explains proper part loading and holding procedures
### Controls Descriptions

The following descriptions depict similar controls found in your work area. Use the descriptions to match the illustrations found on pages 1 through 14.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Nakamuura, Fanuc, TM System 6</td>
</tr>
<tr>
<td>2.</td>
<td>Matsuura, Fanuc, System 6</td>
</tr>
<tr>
<td>3.</td>
<td>Methods, Lathe/Slant 50, Yasnac</td>
</tr>
<tr>
<td>4.</td>
<td>Matsuura, RATTF, Yasnac, with pallet shuttle</td>
</tr>
<tr>
<td>5.</td>
<td>Matsuura, MC600 VF, without shuttle, w/4th Axes</td>
</tr>
<tr>
<td>6.</td>
<td>K.T. 200</td>
</tr>
<tr>
<td></td>
<td>KT D Control</td>
</tr>
<tr>
<td></td>
<td>KT D Control APL</td>
</tr>
<tr>
<td></td>
<td>KT B Control</td>
</tr>
<tr>
<td>7.</td>
<td>Mori Seiki, Partner M300, Fanuc MFM4</td>
</tr>
<tr>
<td>8.</td>
<td>Niigata, Fanuc 15M, HCNiigata PN40</td>
</tr>
<tr>
<td>9.</td>
<td>Niigata, Fanuc 11M V20, with mounting system</td>
</tr>
<tr>
<td>10.</td>
<td>Niigata, Fanuc V50 - V70, 6 Pallet</td>
</tr>
<tr>
<td>11.</td>
<td>Allen-Bradley, Hardinge, Relief Cell</td>
</tr>
<tr>
<td>12.</td>
<td>Cincinnati, Mylacron, Gear Lathe</td>
</tr>
<tr>
<td>13.</td>
<td>Parlec Tool Presetter</td>
</tr>
<tr>
<td>14.</td>
<td>Cincinnati, Acramatic</td>
</tr>
</tbody>
</table>
Not shown in available illustrations.