DENFORD

# TRIAC FANUC - 0 M

# INSTRUCTION

# MANUAL

*Version 11 . 92*

*WARNING*

*WARRANTY ON THIS MACHINE WILL BE INVALIDATED IF ANY MODIFICATIONS, ADDITIONAL ANCILLIARY EQUIPMENT IS FITTED OR ANY ADJUSTMENTS MADE TO THE CONTROLLING DEVICES WITHOUT PRIOR NOTIFICATION FROM DENFORD MACHINE TOOLS LTD.*

# INTRODUCTION

THANK YOU FOR YOUR SELECTION AND PURCHASE OF OUR TRIAC FANUC MILLING MACHINE.

THIS MANUAL DESCRIBES THE INSTRUCTIONS AS TO THE INSTALLATION, OPERATION AND PROGRAMMING OF THE TRIAC FANUC MILLING MACHINE, IN ORDER TO USE THE MACHINE TO ITS FULL CAPACITY.

THIS MANUAL IS COMPLIMENTED BY A SERVICE MANUAL CONTAINING:

ROUTINE MAINTENANCE, PARAMETER LISTINGS, DIAGNOSTICS, PLC LISTINGS, ELECTRICAL WIRING DIAGRAMS, SPARE PARTS AND ASSEMBLY DRAWINGS AND TROUBLE SHOOTING.

IN ADDITION TO THESE MANUALS, PLEASE REFER TO THE INSTRUCTION MANUALS OF OUR CONTROL SUPPLIERS FOR DETAILED OPERATING TECHNIQUES.

This page intentionally left blank.

## INSTRUCTION MANUAL

- SECTION 1 STANDARD EQUIPMENT SUPPLIED
- SECTION 2 SPECIFICATIONS
- SECTION 3 INSTALLATIONS
- SECTION 4 OPERATIONS
- SECTION 5 CNC APPLICATIONS
- SECTION 6 CUSTOMERS OBSERVATIONS

## SECTION INDEX

## SECTION INDEX

This page intentionally left blank.

## STANDARD EQUIPMENT SUPPLIED:

| | |
|---|---|
| • TRIAC FANUC CNC MILLING MACHINE | 1 |
| • TRIAC FANUC INSTRUCTION MANUAL | 1 |
| • TRIAC FANUC SERVICE MANUAL | 1 |
| • FANUC SERIES OMC MANUALS | 1SET |

## STANDARD EQUIPMENT:

| | |
|---|---|
| • SET OF KEYS | 1 |
| • SET OF METRIC ALLEN KEYS | 1 |
| • SET OF MACHINE FUSES | 1 |

## COOLANT OPTION ONLY:

| | |
|---|---|
| • SET OF ALTERNATIVE COOLANT PIPE NOZZLES | 1 |

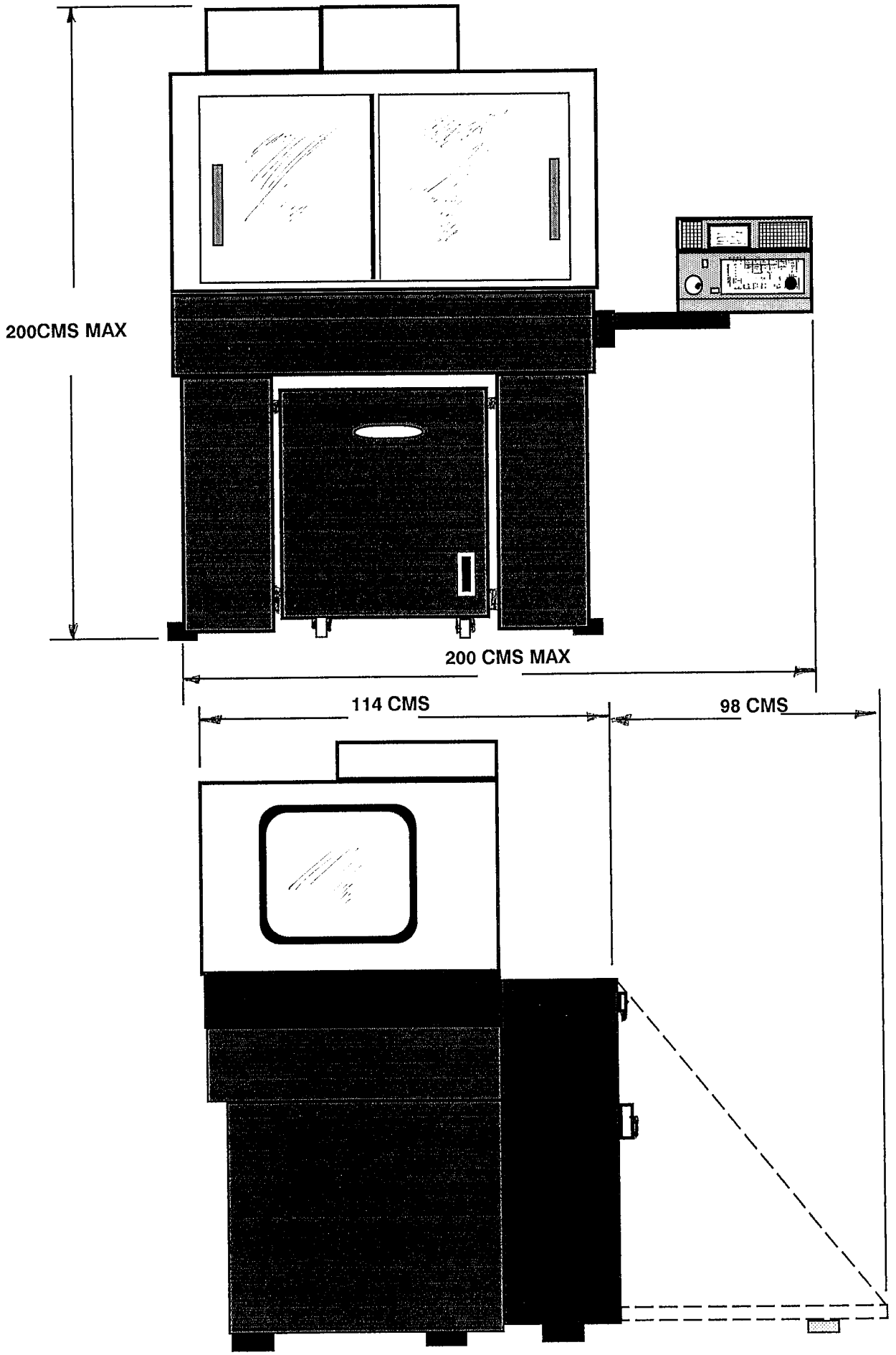
This page intentionally left blank.

## SECTION 2

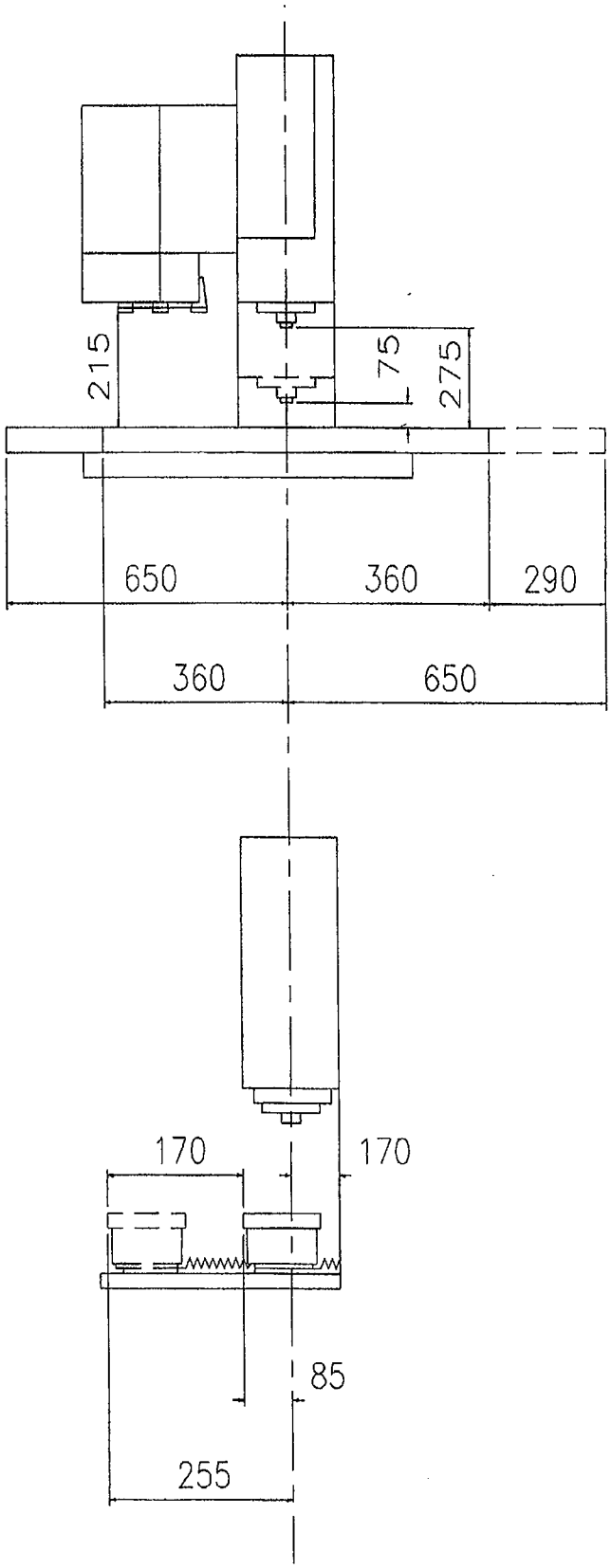
### SPECIFICATION

This page intentionally left blank.

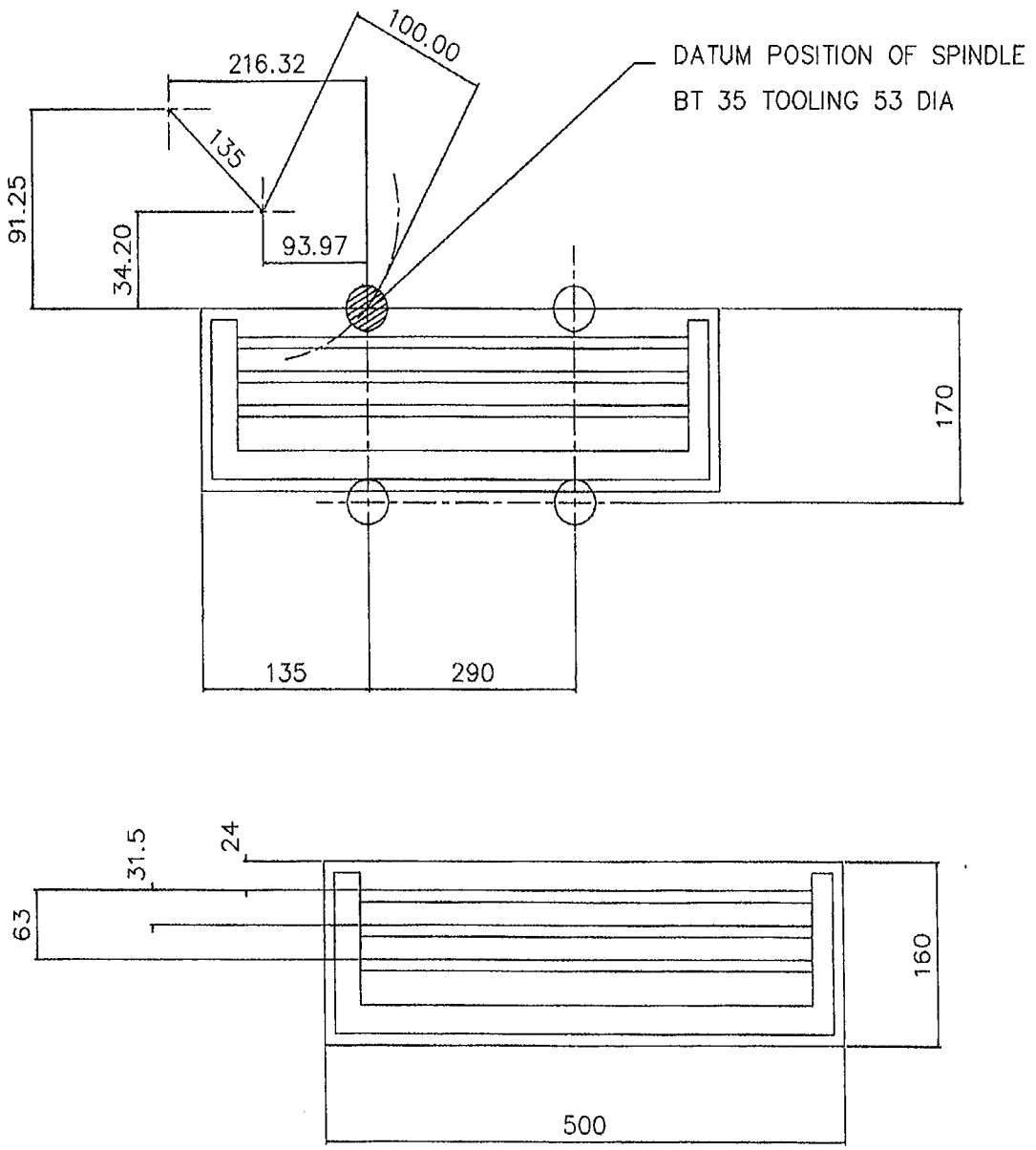
## 2.1 TRIAC FANUC MACHINE EXTENT



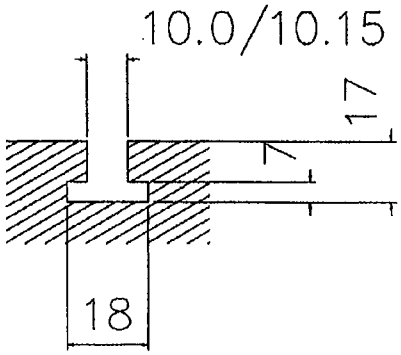
## 2.2 CAPACITY DIAGRAMS



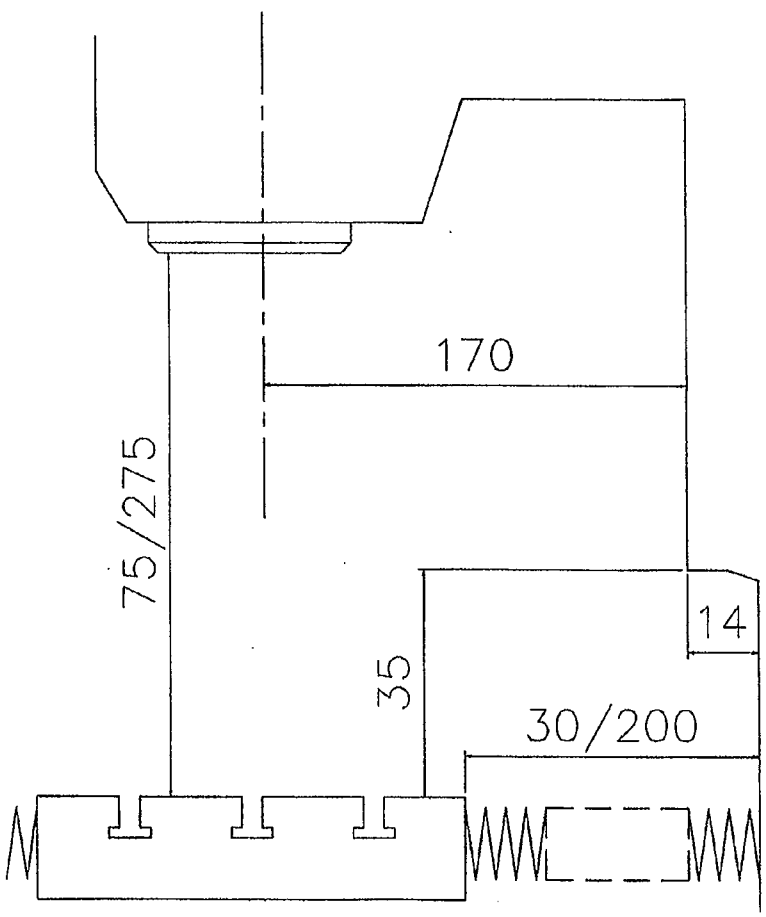
## MOVEMENT OF A.T.C.OVER TABLE & SPINDLE TRAVEL OVER TABLE



**TEE SLOT SECTION**




**HEAD POSITION**



## 2.3 SPECIFICATIONS

### 2.3.1 MACHINE SPECIFICATION

1 STANDARD FEATURES - TRIAC FANUC (ATC)

A)General:

1) Machine dimension list:

Cabinet:

| | |
|---|---|
| Length: | 1000mm (39 1/2") |
| Width: | 1140mm (45") |
| Height: | 2000mm (78 3/4") |

2) Weight:

| | |
|---|---|
| Machine Weight: | 340 kg (750 lbs) |

B)Capacities:

Table Area: 500 x 160mm (19 1/2" x 6 1/4")

| | |
|---|---|
| T Slots: | 10mm slots |
| Traverses: | X: 290mm |
| | Y: 175mm |
| | Z: 200mm |
| Maximum feed Rate: | 5 m/min |
| Spindle nose to table top: | Max: 275 mm |
| | Min: 75mm |
| Spindle nose to column: | 177mm |
| Maximum feeding force: | 17.6 KN |

C)Main Spindle

| | |
|---|---|
| Drive Motor: | Fanuc DC 1 HP |
| Speed range: | 0 - 4000 RPM |

| | |
|---|---|
| Spindle Type: | ATC - No 35 INT, |
| | STD R8 |
| Spindle Orientation: | Programmable |

D) Axis Motors

FANUC AC Digital Servo (Brushless)

| | |
|---|---|
| Maximum Torque: | 7 - 8 NM |

E) Tooling

| | |
|---|---|
| Maximum Tool Diameter: | 80mm |
| Maximum Tool Length: | 90mm |

F) ATC (Optional)

| | |
|---|---|
| Number of Stations: | 6 |
| Time for Tool Change: | 7 Secs |

G) Coolant (Cutting Fluid)

| | |
|---|---|
| Pump Motor: | 20 l/min (4.4 G/min) |
| Tank Capacity: | 40. Litres |

H) Machine Working Lights

12v AC Halogen Lo-Vo Light

I) Enviromental Conditions

| | |
|---|---|
| Power Supply: | 50/60 HZ 3PH |
| | 380/415V 12 Amps |
| Total Power Consumption: | 5 KW |

## 2-3-1 NC CONTROL SPECIFICATION

1 STANDARD FEATURES - FANUC OM MODEL C

A) Controlled Axes

| | |
|---|---|
| Controlled Axes 3 Axes (X Y & Z) | |
| | Simultaneous Contouring Control |
| Least input increment | 0.001mm (0.00004") |
| Least command increment | 0.001mm |

B) Interpolation Functions

| | |
|---|---|
| Positioning | G00 |
| Linear Interpolation | G01 |
| Multi-quadrant circular Interpolation | G02, G03 |

C) Feed Functions

| | |
|---|---|
| Rapid Traverse | 5m/min |
| Rapid Traverse Override | 100%, 50%, 25%, 0% |
| Manual Pulse Generator | Increments, 0.001/0.01/0.1 |
| Manual Continuous Feed | |
| Cutting Feed Rate | G94 (mm/min), G95 (mm/rev) |
| Cutting Feed Rate Clamp | |
| Feed Rate Override | 0 to 150% at 10% increment |

| | |
|---|---|
| Tangential Speed Constant Control | |
| Automatic Acceleration/ Deceleration | Linear for rapid traverse exponential for cutting feed |
| Dwell G04 | 0 to 99999.999 sec |
| Dry Run | |
| Free Hold | |
| Reference Point Return | G27, G28 and G29 |
| Exact Stop | G09 |

D) Spindle Functions

| | |
|---|---|
| Spindle Speed Command | S-5 Digit Direct RPM |
| Constant Surface Speed Control | G96 and G97 |
| Spindle Speed Override | 50% to 120% at 10% increment |

E) Tool Functions

| | |
|---|---|
| T-Function | 2 Digit Tool Number + 2 Digit Offset Numbers |
| Tool Offset Amount | ±6 Digits separate settings of geometry offsets. 16 Pairs in Memory |

| | |
|---|---|
| Tool Radius Compensation | G40, G41 and G42 |
| Programmable Data Input | G10 Option |
| Direct input of offset value measured | |
| Incremental Offset | |
| Counter Input of Offset Amount | |

F) Miscellaneous Funtions

| | |
|---|---|
| M-Functions 2 Digits | |

G) Programming Functions

| | |
|---|---|
| Plane Switching | XY, XZ, YZ |
| Co-ordinate System Setting | G54 - G59 |
| Inch/Metric conversion | G20 and G21 |
| Decimal Point Programming | |
| Radius Programming on arc | |
| Canned Cycles | G73, G74 G76 and G80-89 |
| Program Number | O (EIA code) or : (ISO) 4 digits |
| Program Number Search | |
| Program Name | |
| Main Program and Sub Programs | |
| Sequence Number | N 4 Digits |

| | |
|---|---|
| Tape Code | EIA (RS-244A)/ ISO(R-804) |
| Input/Output Interface | |
| Optional Block Skip | |
| Buffer Register | |
| Program Stop | M00 |
| Optional Stop | M01 |
| Program End | M02 or M30 |
| Single Block | |
| Part Program Storage & Editing | |
| Tape Storage Length | 10M (4000 char) |
| Registerable Programs | 63 Programs |

H) Safety Functions

| | |
|---|---|
| Emergency Stop | |
| Software Limits | X,Y and Z |
| Machine Lock | |
| Cutting Block Start | |
| Interlock | |

I) Others

| | |
|---|---|
| Manual Data Input (MDI) | Keyboard Type |
| Software Keys | 5 Keys |

9" Monochrome CRT Character Display

| | |
|---|---|
| Self Diagnosis Functions | |
| External Data Input | |
| External Tool Compensation | |
| *Alarm and OperatorMessage Only | |
| Programmable Controller | PC-model L |

**NOTES:**

| | |
|---|---|
| **Power Supply** | 415 Volts 3PH + Neutral |
| **Temperature** | 0 degrees C to |
| | 45 degrees C |
| **Relative Humidity** | Less than 75% |

## 2.3.2. OPTIONAL EXTRAS

Additional Programmable Memory: Total 320m

Conversation Program with Graphic Functions

Program Input of Offset Data

Cutter Compensation Addition A : TTL 64

Custom Macro A

Pattern Data Input

Graphic Display

Clock Function

Handwheel Intervention

MDI Operation B

Background Editing

Work Co-ordinate System G54 - G59

Program Restart

2nd Auxiliary Function

Tool Life Management

Helical Cutting

Additional Axis Control

Co-ordinate System Rotation

Scaling

Automatic Corner Override

Simultaneous 4 Axis Control

Macro Executer

Reader/Puncher Interface

This page is intentionally blank.

## SECTION 3

### INSTALLATION

This page is intentionally blank.

## 3.1 INTRODUCTION

This section describes the correct procedures for installing your TRIAC FANUC Milling machine.

These procedures should be followed precisely to ensure your TRIAC FANUC is not damaged in any way during the installation period

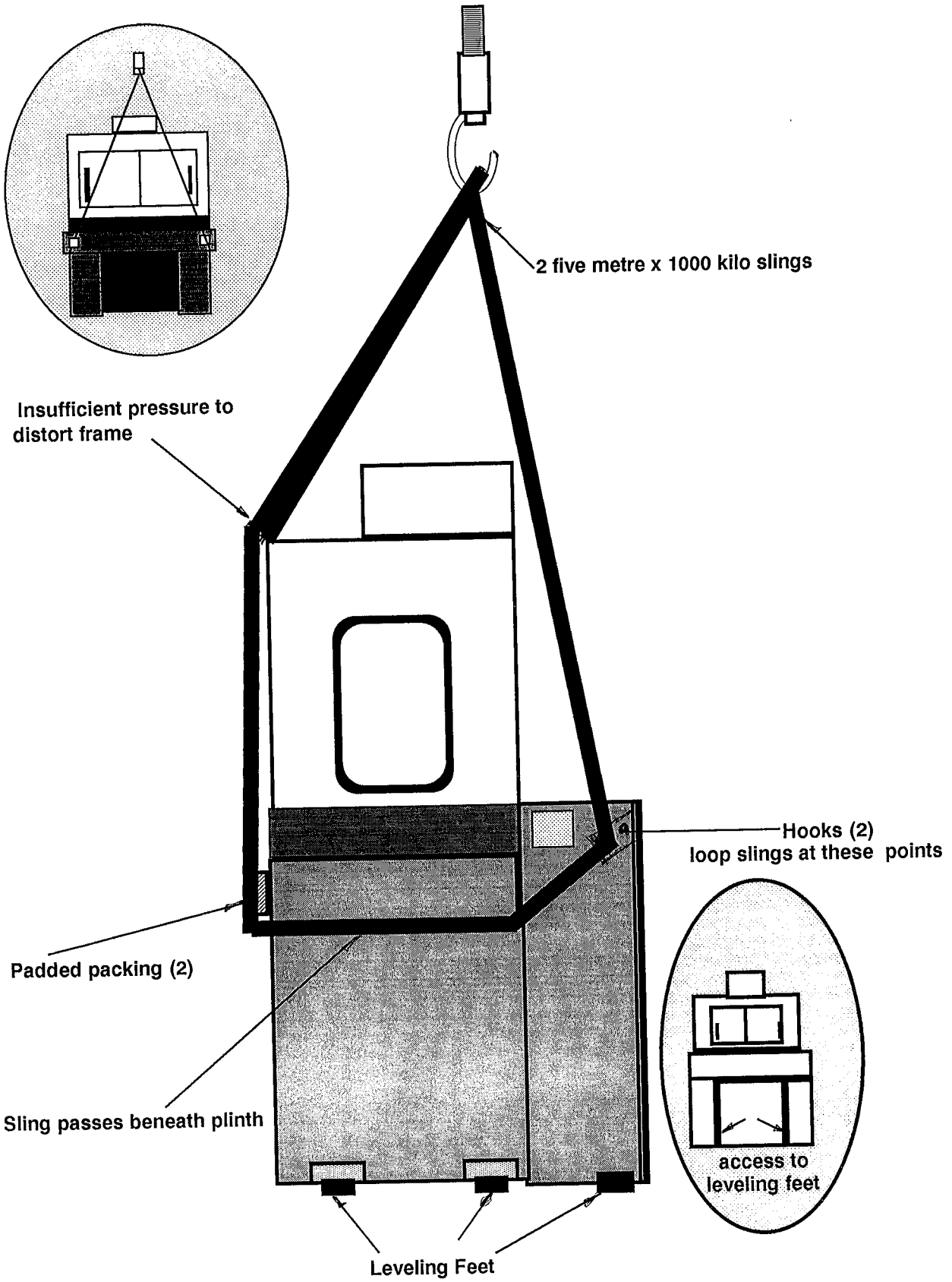
During the installation period the protective coats and coverings applied prior to despatch should NOT be removed.

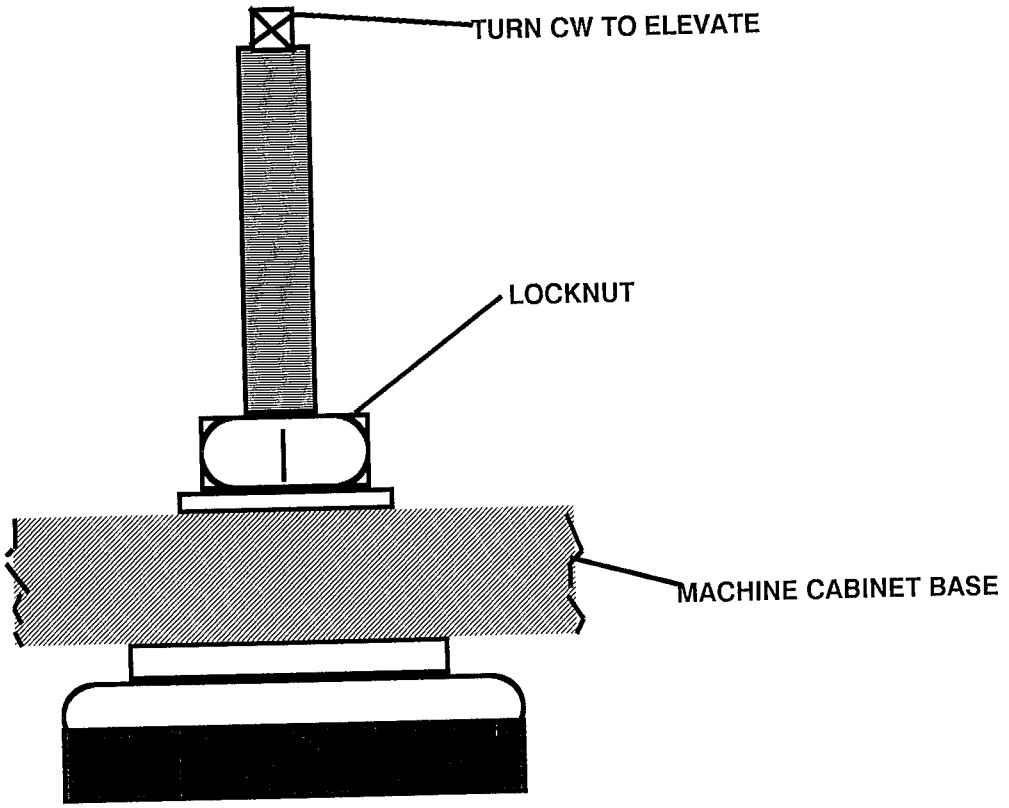
All installation work should be carried our by qualified personnel.

TRIAC FANUC is a floor mounted milling machine, hence it is important to ensure a level site is prepared.

If in doubt contact DENFORD's Servicing Department for further details.

## 3.2 LIFTING A TRIAC FANUC






The TRIAC FANUC stands on six anti-vibration leveling feet situated at each corner of the machine and also in the electrical cabinet at the rear of the machine. The machine bed is leveled to the cabinet during manufacture hence it is only necessary to ensure that the cabinet is leveled during installation.

## FEET LOCATION

Once the coolant tank has been pulled clear of the machine cabinet base, the four feet are accessible through the inside of the cabinet side panels - see diagram on previous page. The two feet on the electrical cabinet are accessible through the main electrical cabinet door at the rear of the machine.

## LEVELING

Tools required:- Spirit level, 10mm & 24mm A/F Spanners.

Place the spirit level on the centre of the table parallel to the "T" slots. If the bubble is not aligned adjust asfollows:-

Release the locknut on the leveling screw whilst holding the screw stationary with the 10mm spanner. By turning the screw CW the machine can be raised, conversely

turning the screw CCW the machine can be lowered. When the bubble is aligned, turn the spirit level through 90 degrees and again adjust to align the bubble. Re- check for level at the extreme ends of the table. Once the machine is level, tighten the locknuts taking care to hold the screw stationary.

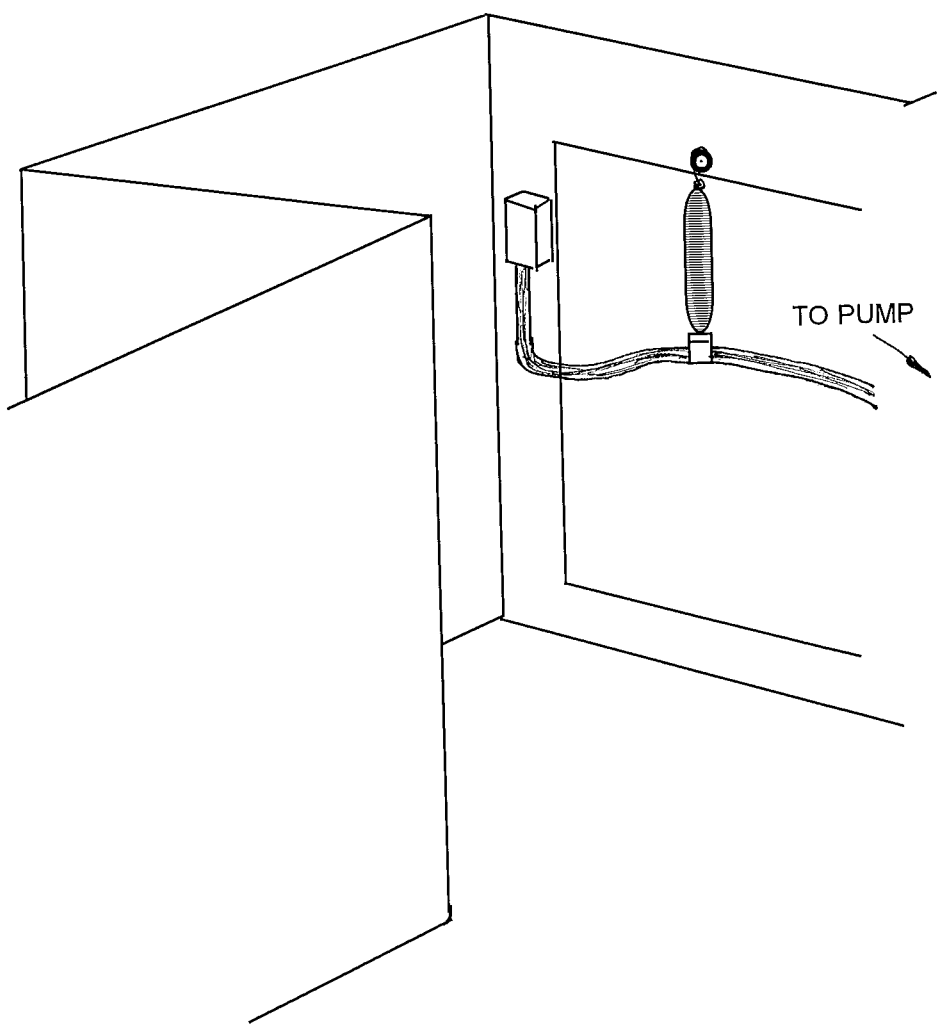
NOTE:- The two feet inside the electrical cabinet should not be used to level the machine and should be adjusted LAST and only to support the electrical cabinet.

ELECTRICAL CABINET

NB. THE ELECTRICAL CONTROL BOX IS INSPECTED THEN SEALED WITH A YELLOW SEAL, IF THIS SEAL IS BROKEN ON DELIVERY INFORM THE SUPPLIERS IMMEDIATELY. THE SEAL SHOULD ONLY BE BROKEN FOR THE INITIAL MAINS POWER CONNECTION.

PUMP CABLE

The coolant pump cable beneath the machine is attached to the machine cabinet by a spring and hook to prevent entrapment when the mobile coolant tank is pulled out to the front of the machine for filling and emptying.



## 3.4 ELECTRICAL CONNECTION

**Cable Needed:-**

3 Core + Neutral & Earth, 2.5mm per phase

**For U.K Market:-**

3 Phase + Neutral 415v 50Hz Current Taken 8 Amps/Phase

**For Export Market:-**

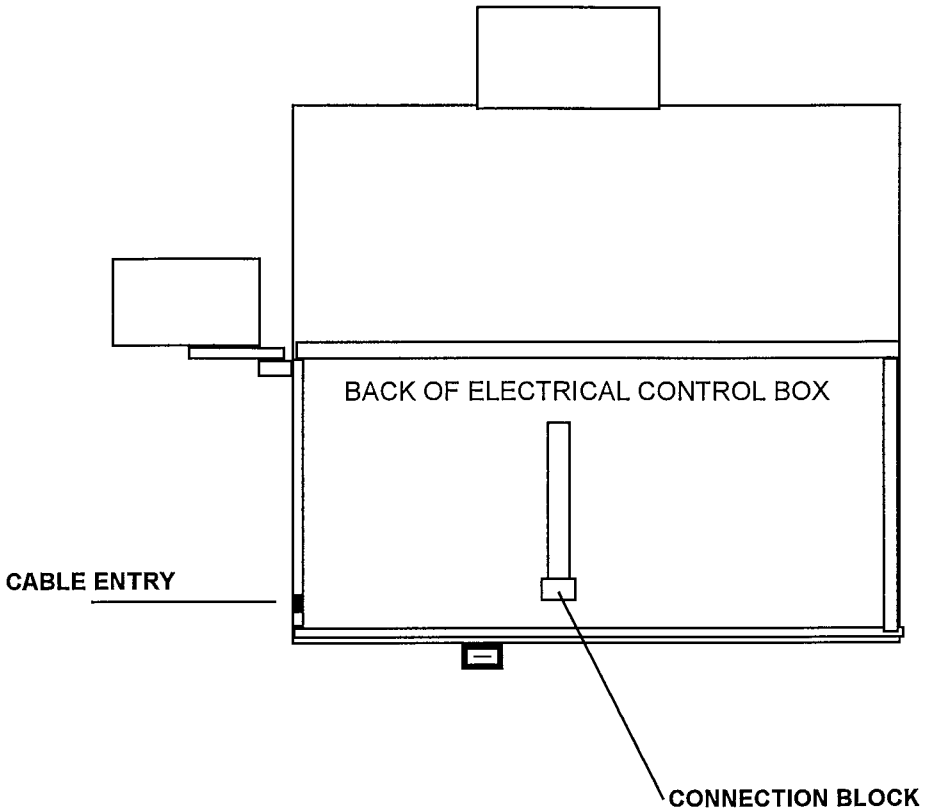
Contact Denford Servicing Department

Tools Needed:- Wire Strippers, Small Flat Blade Screwdriver

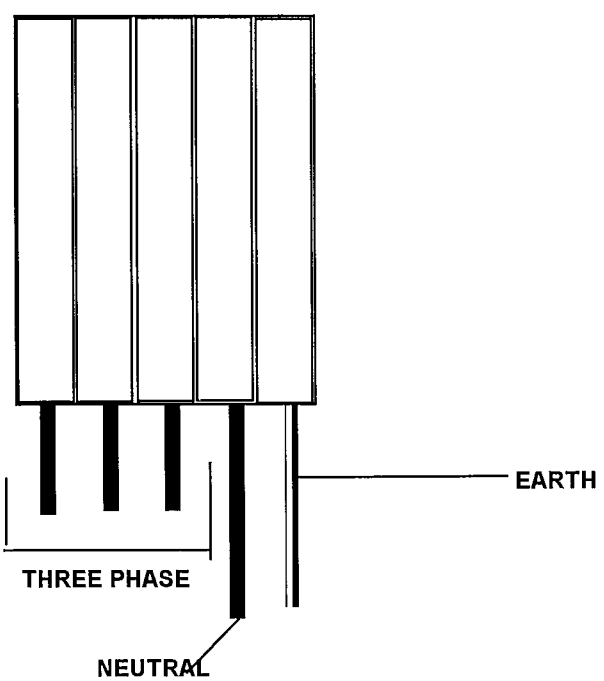
**CONNECTION PROCEEDURE:-**

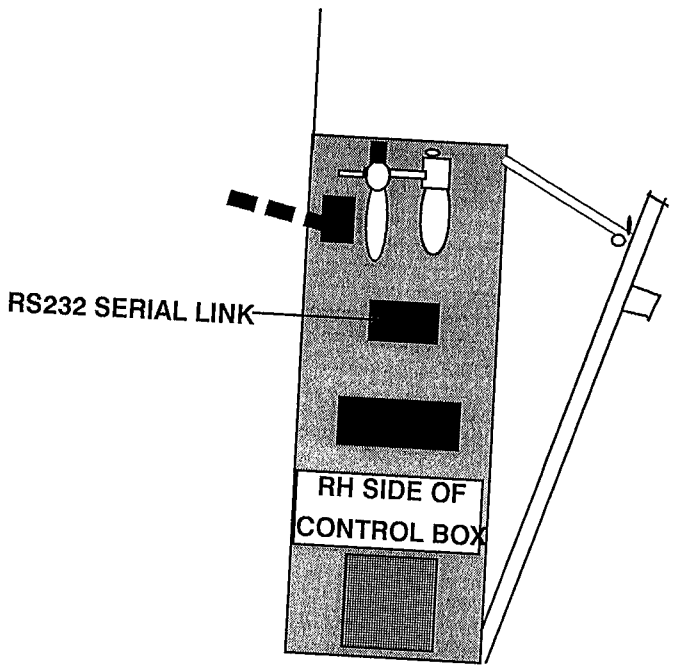
- Unlock and open cabinet at rear of machine.
- Mains connection block is located on the panel in the bottom centre
- Feed cable through cable entry as shown on next page.
- Connect seperate wires into connection block as shown on next page.

## 3.4 TRIAC FANUC ELECTRICAL CONNECTIONS

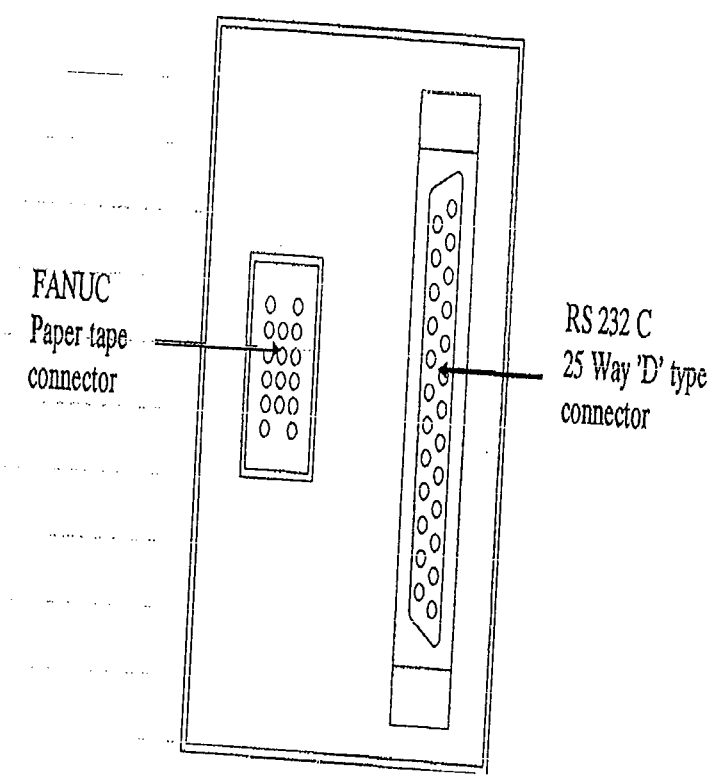


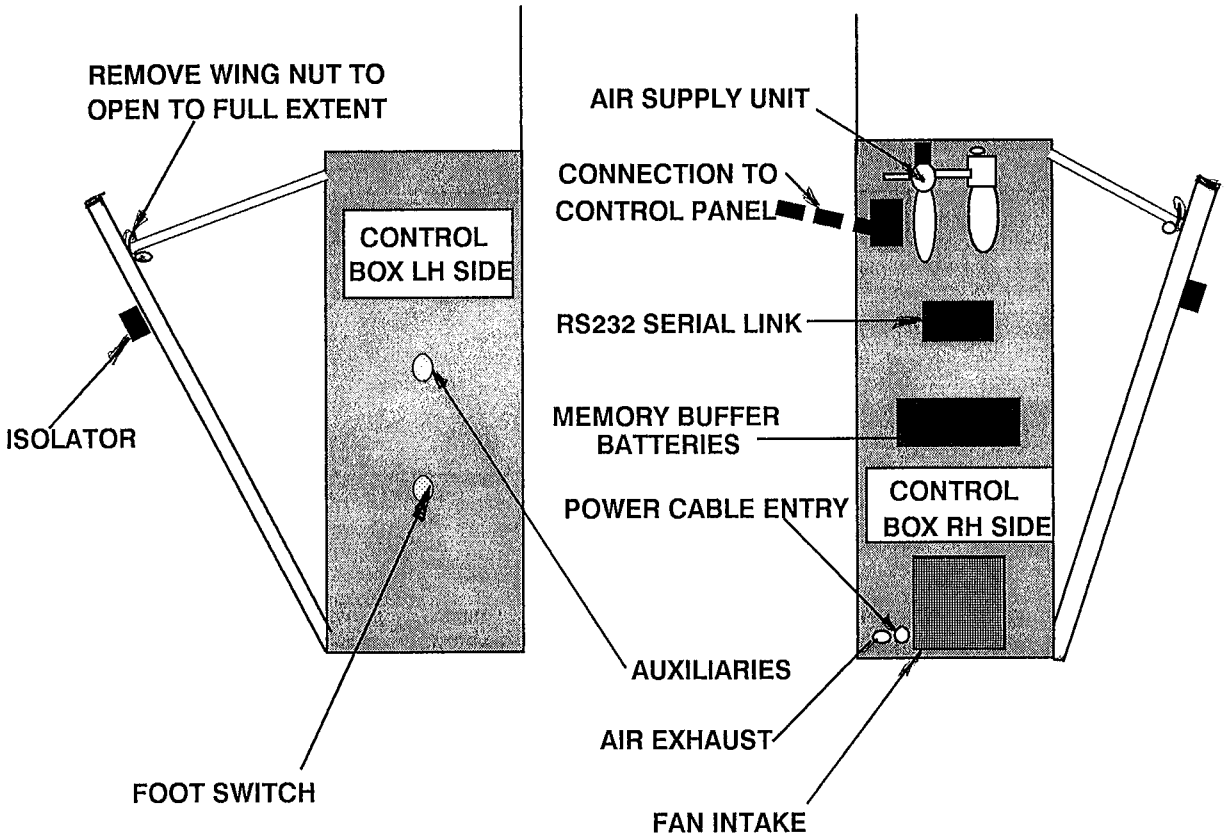

### CONNECTION BLOCK

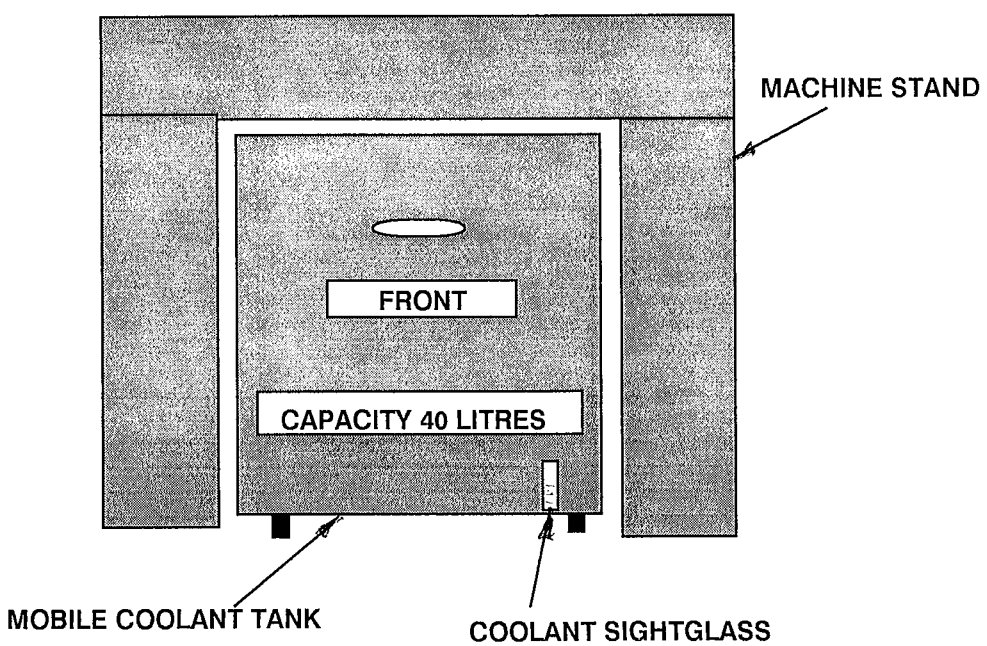





## RS 23 2 SERIAL LINK



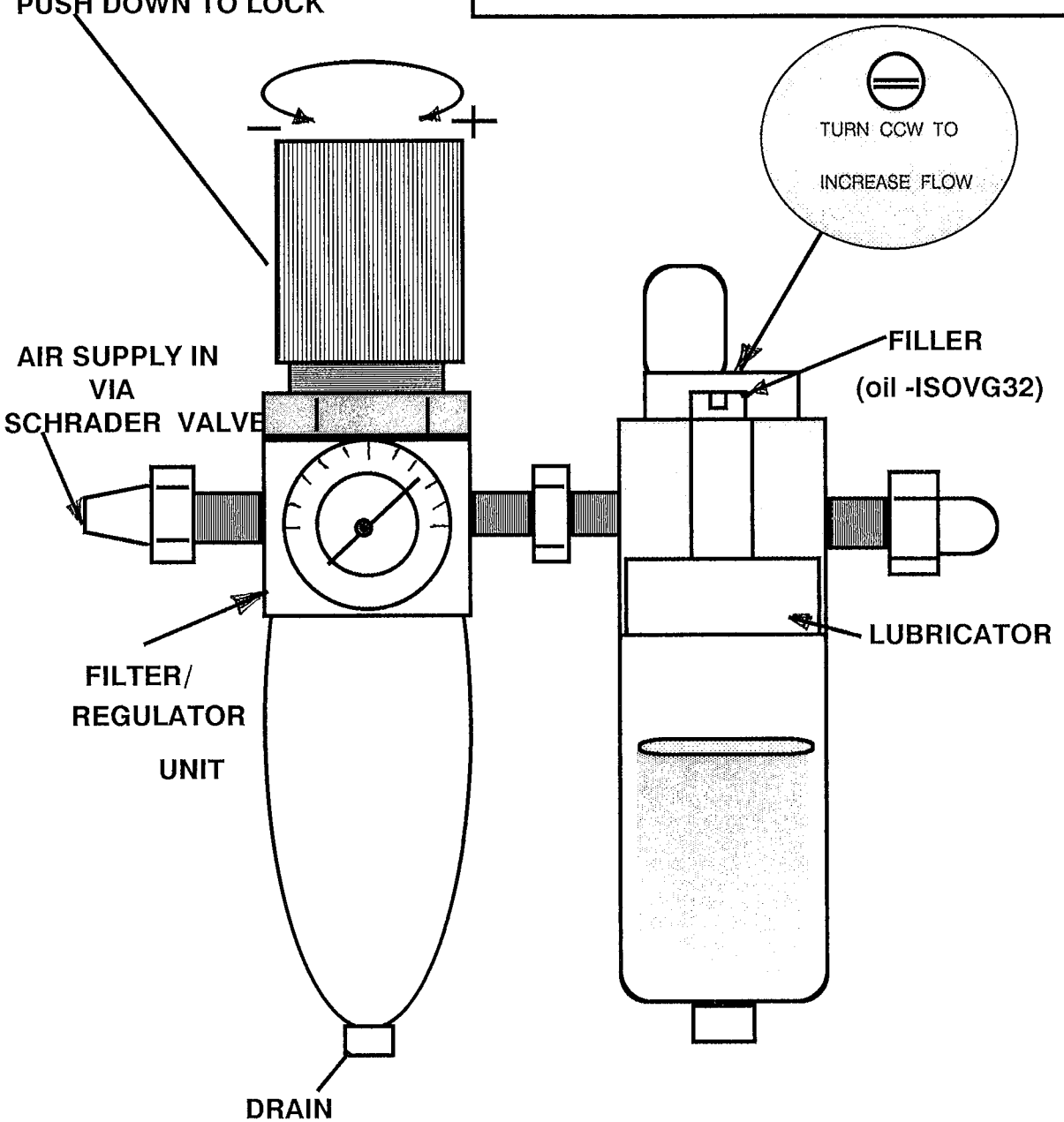

REMOVE WING NUT TO
OPEN TO FULL EXTENT
CONTROL
BOX LH SIDE
ISOLATOR
FOOT SWITCH
AIR SUPPLY UNIT
CONNECTION TO
CONTROL PANEL
RS232 SERIAL LINK
MEMORY BUFFER
BATTERIES
POWER CABLE ENTRY
CONTROL
BOX RH SIDE
AUXILIARIES
AIR EXHAUST
FAN INTAKE


MACHINE STAND
FRONT
CAPACITY 40 LITRES
MOBILE COOLANT TANK
COOLANT SIGHTGLASS

## AIR FILTER REGULATOR AND LUBRICATOR

Air consumed = ATC : 0.87 $dm^3$/cycle

Power Vice : +0.092 $dm^3$/cycle




Maximum pressure for air regulator 9.9Kg F/$cm^2$(150 lbs sq")

Operating pressure 100lbs sq."

Check main supply pressure before adjusting pressure.

## 3.7 MACHINE PREPARATION

Equipment Required:- Machine slide way oil (See Service Manual Section 1.5), Glass and Perspex Cleaner

On delivery, the machine will have all untreated surfaces within the work area covered with a protective solution.

This is removed using a kerosene base solvent and should be done before any attempt is made to move the slides or operate the machine.

Once the protective coating is removed all untreated surfaces should be covered with a light smear of machine oil (i.e B.P: CS 68).

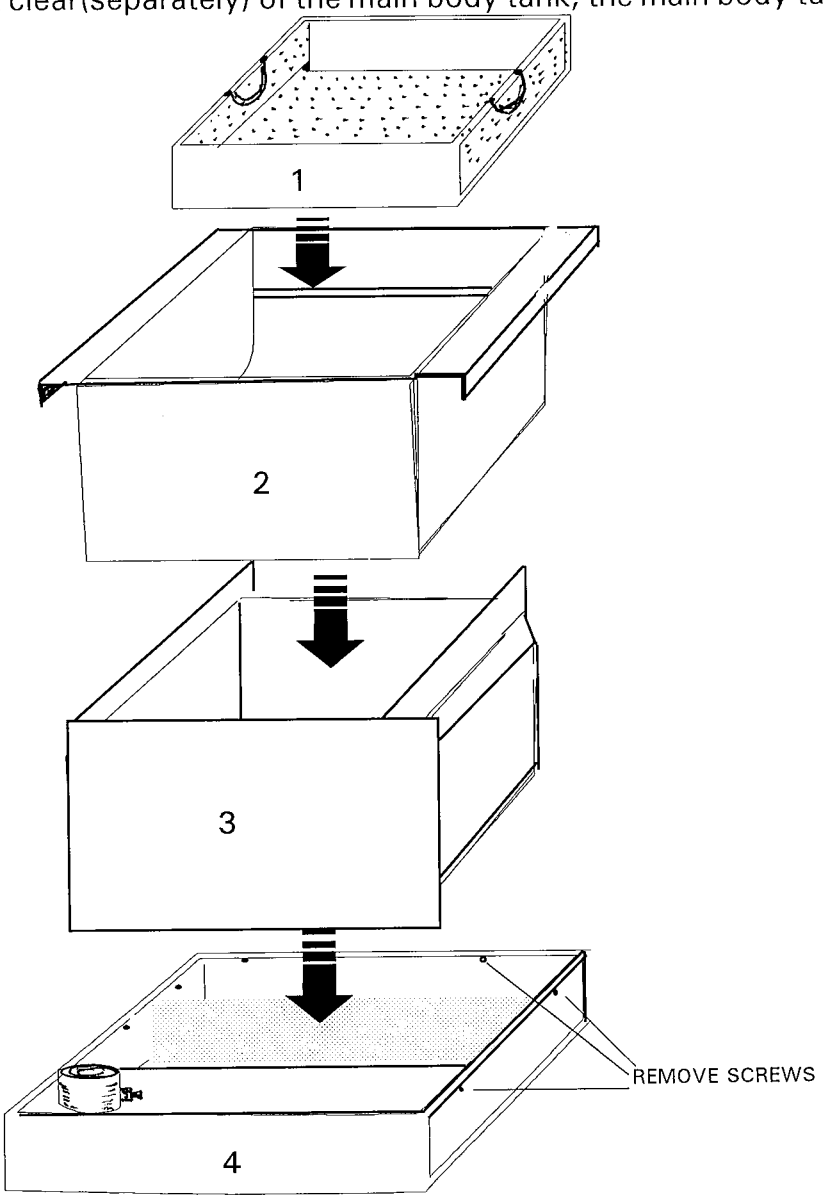
The protective plastic sheet on the windows is pulled off and the glass and perspex should be cleaned with an anti-static cleaner.

# *COOLANT TANK*

The coolant tank breaks down into four component parts :-

1. The Ferrous filter tray
2. The Non - Ferrous filter tray
3. The Main Body tank
4. The Pump Filter Tank Base.

Trays 1 and 2 lift clear(separately) of the main body tank, the main body tank can then be

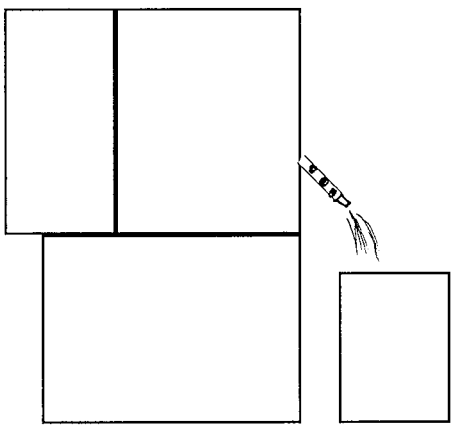



separated from the base by removing the six screws.

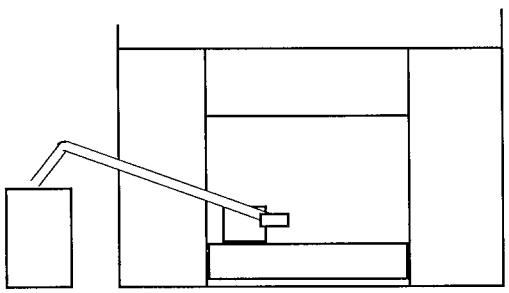
# *COOLANT DRAINING*

There are basically two methods of draining the coolant tank :-

1. With the coolant tank still in place beneath the cabinet base, place the drum or container which is to hold the drained off coolant as near to the machine cabinet as possible. Angle the coolant nozzel(s) into the drum and switch on the coolant flow (see diagram below). When the coolant ceases to flow, switch off the coolant and pull out the tank from beneath the cabinet (front or rear depending on the cabinet model). The tank can then be separated as in the diagram on the previous page and the remainder of the coolant removed from the base.



2. With the coolant tank still in place beneath the cabinet and with the coolant switched off, remove the coolant delivery pipe from the pump by removing the clip. Place a pipe of the same size over the pump nozzel which you have just removed the delivery pipe from, clamp with the same clip and place the other end in a drum or container, then switch on the coolant pump to drain the coolant into the drum. When the coolant ceases to flow, switch off the coolant pump, pull out the coolant tank from beneath the cabinet. The tank can then be separated as in (1) above. Ensure delivery pipe is reconnected to pump after the draining operation is completed.



## SECTION 4

### OPERATION

## 4.1 INTRODUCTION

This operation section of the manual is intended to allow a competent machinist to get started on TRIAC FANUC.

In 4.3 the control description part, each button is identified in turn with a brief description of what it is intended to do. For further details, the FANUC Manuals supplied and Section 5 of this Manual should be consulted.

This section also covers the type of tooling used on TRIAC FANUC and describes how to mount different tools.

The final part of this section goes through the correct procedure for starting up TRIAC FANUC. The operator should ensure he/she reads the start up procedures thoroughly and adhere to them whenever turning on the machine.

## 4.2 SAFETY PRECAUTIONS

This machine has safety devices in order to protect the operator and machine from any unexpected accident.

### 1. Tidyness

- Do not place objects on top or around the machine so that it interferes with guards or the operation of the machine.
- Operate the machine in enviroment conditions which are suitable for CNC Control.

### 2. Power Source

- Ensure correct cable for power source is used. (See Section 3.4)
- When power fails turn off the isolator (found on the left hand side of the machine cabinet ) immediately.
- When leaving TRIAC FANUC at the end of a shift ensure power is turned off.
- Ensure that TRIAC FANUC is isolated before any maintenance of the machine is carried out.

### 3. Lubricating Oil and Inspection

- For lubrication, please refer to maintenance Section, (See Service Manual Section 1.5)
- Check oil reservoir daily,
- Check state of slideway lubrication each day. The machine is fitted with an auto-lubricating system. Ensure the reservoir is topped up regularly when indicated by the low-lube indicator situated at base of LO VO lamp.

### 4. Hints on Safety Operation

- When operating the machine, wear clothing suitable for safe operation.
- Secure workpiece firmly on table
- Only operate the machine after raising all safety screens.
- DO NOT lower safety screens during operation.
- Before starting daily operation, run machine for a few minutes to warm up.

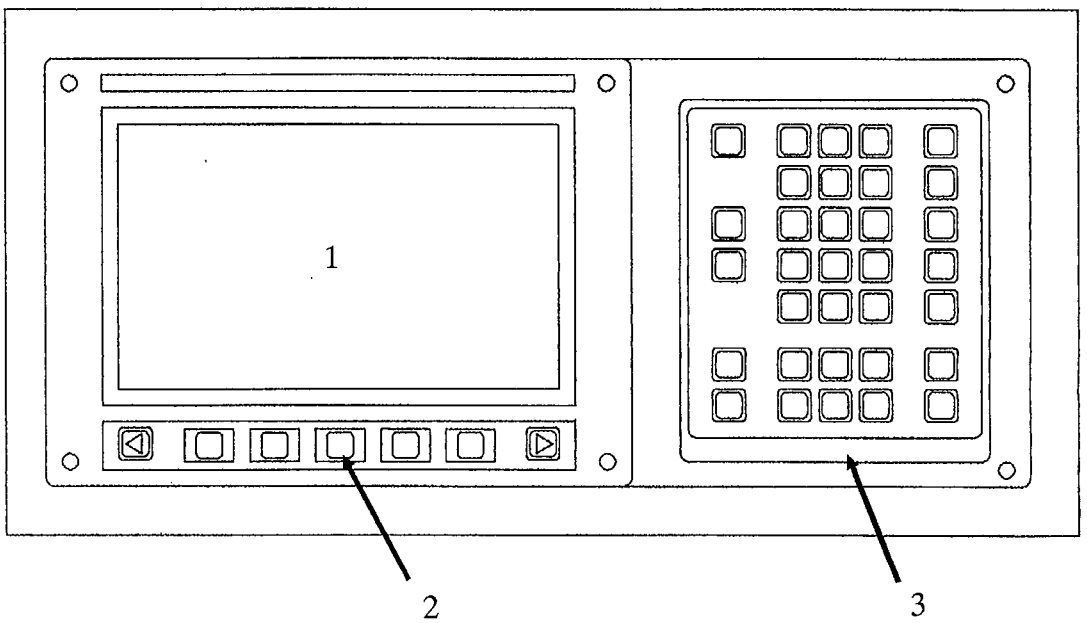
- It is very dangerous to handle swarf, the tool nose and any rotating part of the machine. Never, try to remove the swarf during operation.
- After setting tools, perform trial cutting by manual operation.
- Use a tool offset function effectively in order to avoid a machining failure.
- When cleaning the machine and its attachments, be sure to stop machine operation.
- Do not forget to inspect and clean the machine and NC unit.
- Do not operate the machine with the power control box open.
- When opening the doors of the electrical cabinet, turn OFF the isolator in advance.
- When an emergency stop is required, operate the EMERGENCY STOP switch on the operation panel.

## 4.2 SAFETY DEVICES

This machine is equipped with the following safety devices for the safety of the operator and protection of the machine.

| NAME | LOCATION |
|---|---|
| • MOTOR OVERLOAD | INSIDE ELECTRICAL CABINET |
| • AUTOMATIC SERVO | INSIDE ELECTRICAL CABINET |
| • EMERGENCY STOP SWITCH | OPERATION PANEL |
| • ISOLATOR | OUTSIDE ELECTRICAL CABINET |
| • LIMIT SWITCES | ALL AXIS TRAVELS |

## 4.3 FANUC DISPLAY SCREEN AND DATA INPUT PANEL




- 1) Display Screen
- 2) Software controlled keys used to select option shown above each key.
- 3) Input Panel

## 4.3 FANUC DATA INPUT PANEL ( DIAGRAM ON NEXT PAGE )

- RESET Resets any alarm messages. Resets program to start in edit mode.
- ALPHA/NUMERIC PAD Inputs character expected by controller when inputting program. Multi character keys toggle between characters shown
- CURSOR Moves cursor through program element by element in defined direction
- PAGE Moves cursor through program page by page in defined direction.

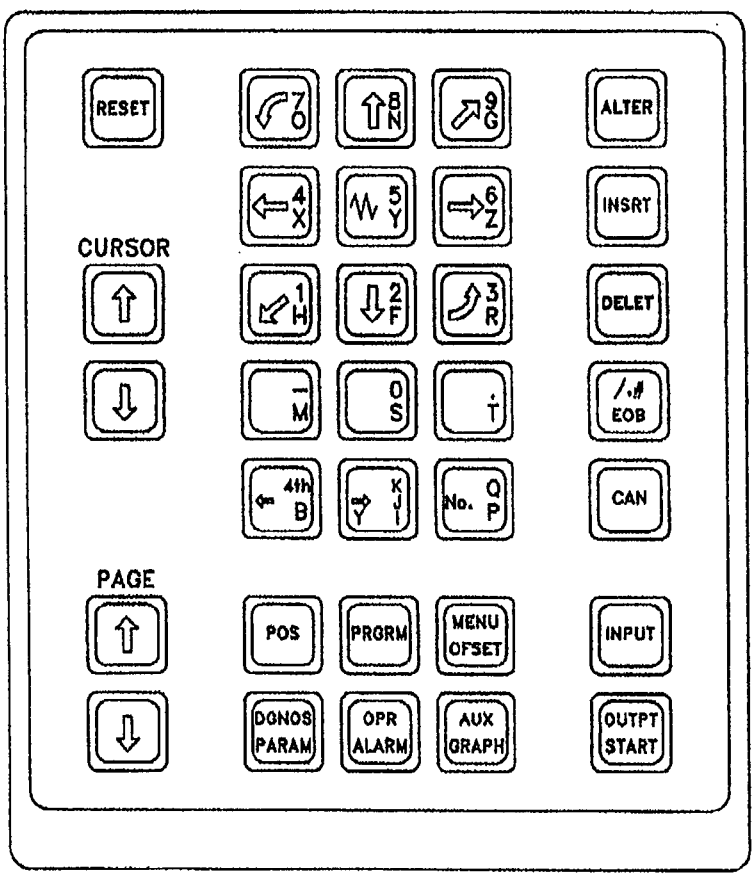
**SELECTION KEYS EACH HAVING SEVERAL PAGES**

- POS Axis positions
- PRGRM Program selection screen
- MENU Tool offset and macro variables OFFSET (Macros optional)
- DGNOS Machine diagnostics and parameters PARAM
- OPR Displays operator and machine alarm messages. ALARM (May also display operators panel on OM Model A Controllers)
- AUX Graphics (Optional) GRAPH

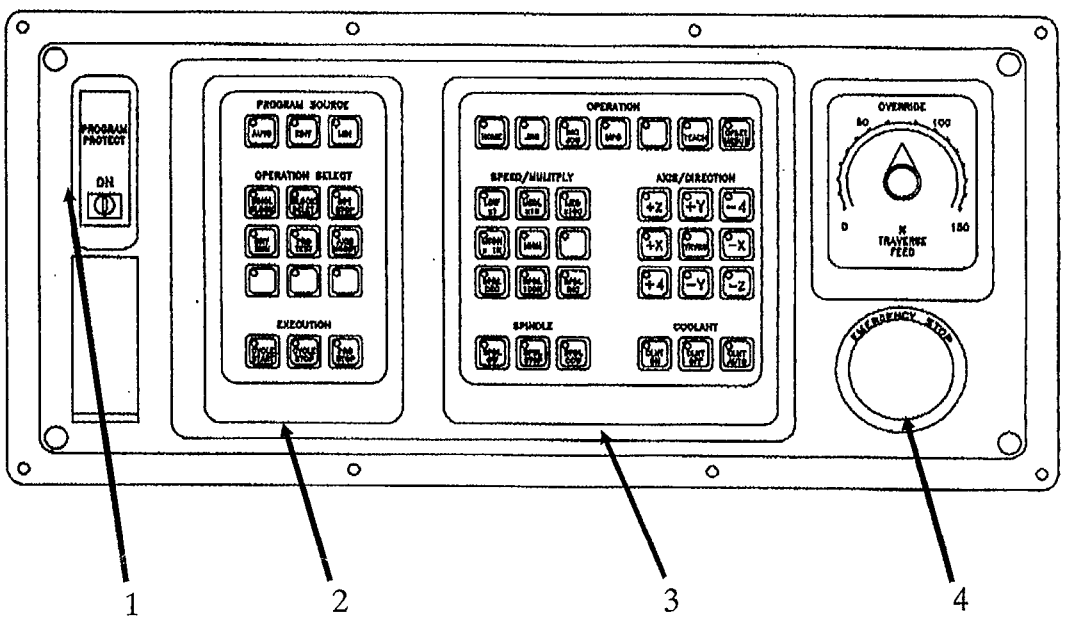
**EDITING KEYS**

- ALTER Alters addresses
- INSRT Inserts addresses. (Also used to initialise new programs)
- DELET Deletes addresses.
- / ; # E.O.B. - charactures shown are toggle EOB operated when editing.
- CAN Cancels an address. (Before EOB is executed)
- INPUT Allows input of data, tool offsets etc. Allows input of a program from an external source when in edit mode.
- OUTPUT/START Allows output of a program to an external unit. START (May also start cycle on OM Model A)

## 4.3 FANUC DATA INPUT PANEL

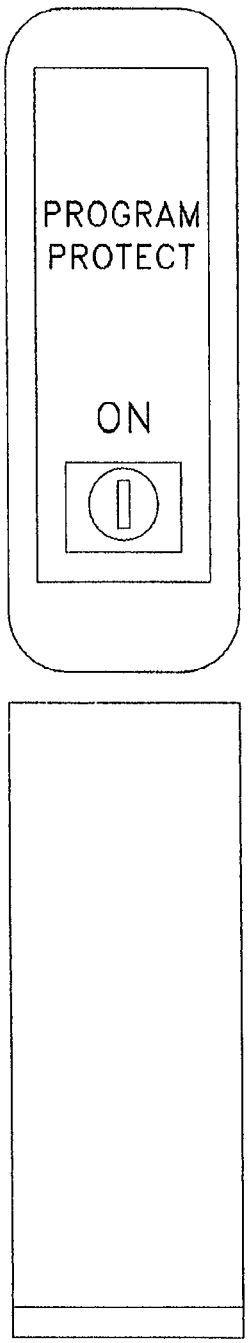


## 4.3 FANUC OPERATORS PANEL



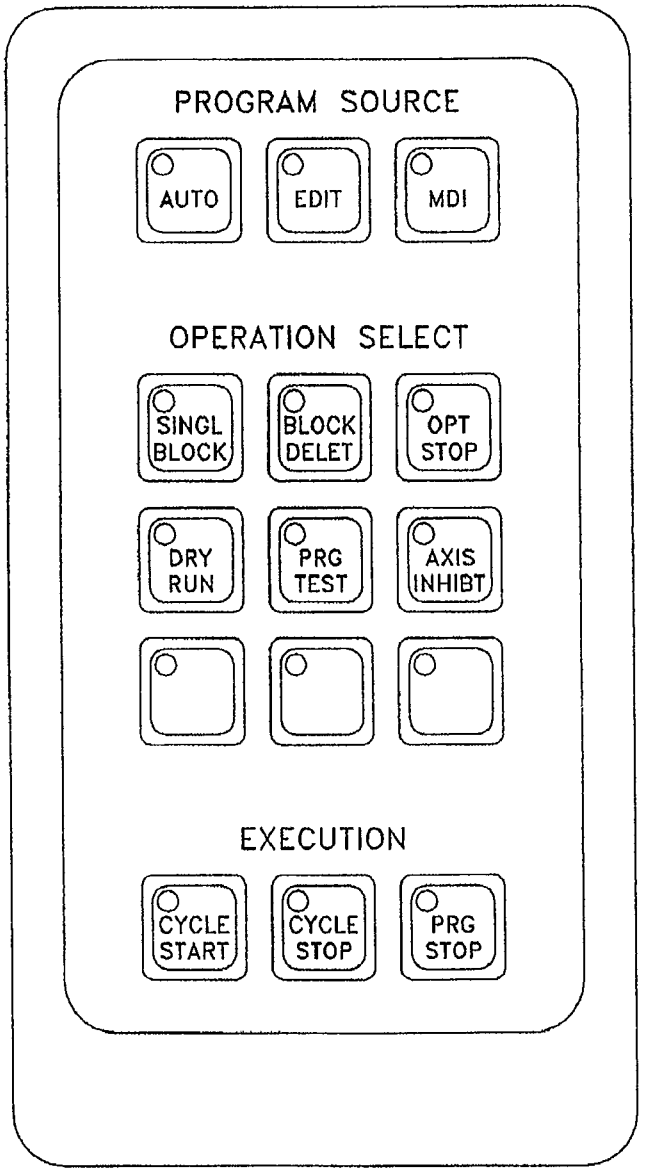

- 1) Program Protection key Switch.
- 2) Automatic Operation Panel.
- 3) Manual Operation Panel.
- 4) Emergency Stop and Manual Feed Override

## 4.3 PROGRAM PROTECTION UNIT




- ON - Protects program from tampering or accidental erasure.
- OFF - Allows full program editing

## 4.3 AUTOMATIC OPERATION PANEL



**PROGRAM SOURCE**

- AUTO - Select to run program.
- EDIT - Select to edit program.
- MDI - Select to manually key in G & M Codes out of program mode.

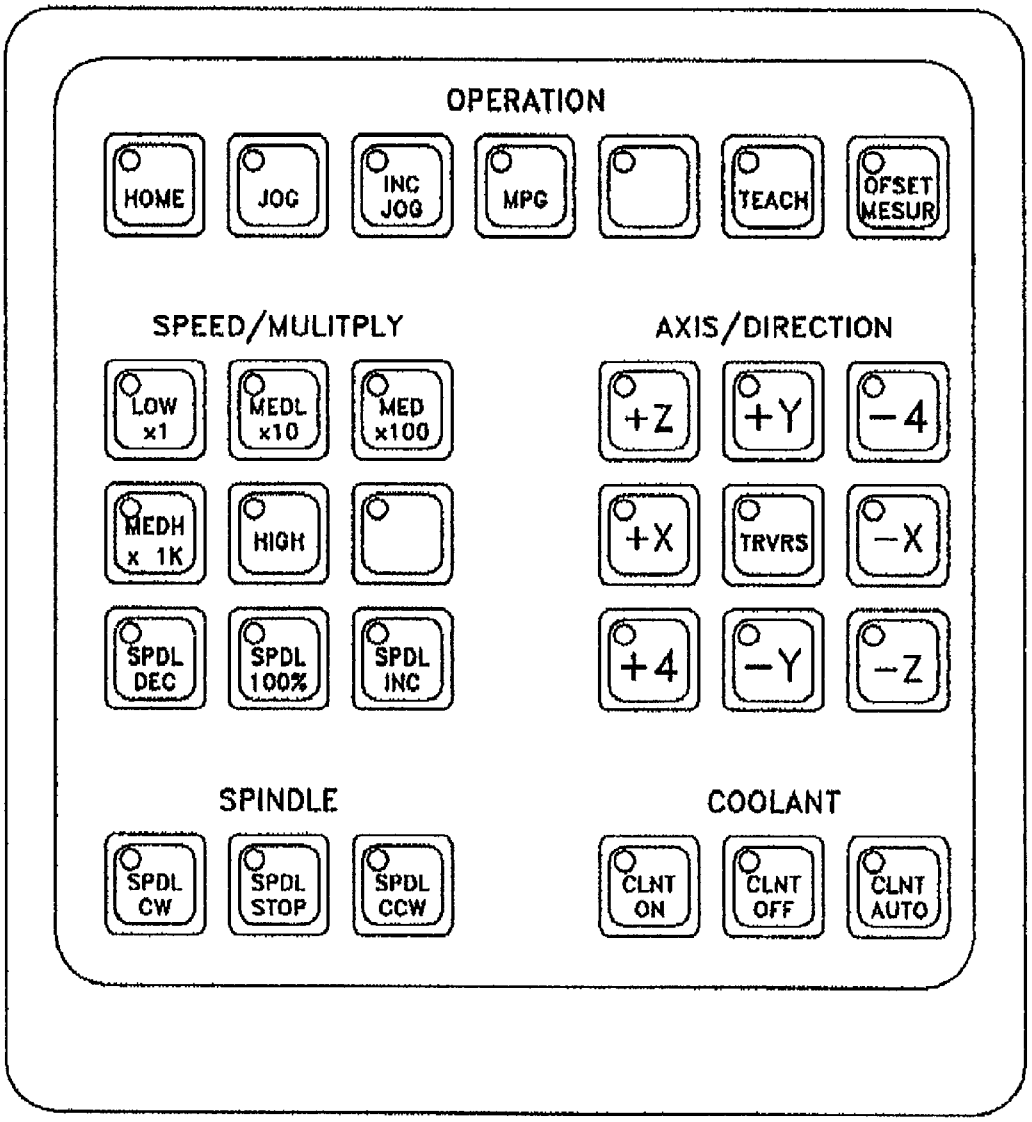
**OPERATION SELECT**

- SINGL BLOCK - Allows single step execution of program.
- BLOCK DELETE - Select in edit mode to ignore block when running program (Activates \ in front of block)
- OPT STOP - Used in conjunction with M01 to optionally stop program.
- DRY RUN - Runs program through at jog feed rate.
- PRG TEST - Runs program through ignoring all M codes.
- AXIS INHIBT - Runs program through with axes locked.

**EXECUTION**

- CYCLE START - Starts program.
- CYCLE STOP - Stops program.
- PRG STOP - Stops program at end of current block.

## 4.3 FANUC MANUAL OPERATION PANEL




- HOME - Zeros machine around its own reference points.
- JOG - Moves axes around at feeds as set on override.
- INC JOG - Moves axes at 0.001, 0.01, 0.1, 1 Increments.
- MPG - Manual Pulse Generator (Electronic Handwheel Control)

- TEACH - Inputs actual machine position into program.
- X 1 ) - Multiplier selection for handwheel control MPG or
- X 10 ) INC JOG mode
- X 100)
- X 1K - Multiplier for INC JOG mode
- SPDL DEC - Spindle Decrease (Override of SPDL 100%), -10 %.
- SPDL 100% - Reverts spindle speed back to programmed value.
- SPDL INC - Spindle increase (Override of SPDL 100%), +10%.

**AXIS/DIRECTION - Press either JOG or MPG to operate.**

- -X - Movement in -X direction
- +X - Movement in +X direction
- -Y - Movement in -Y direction
- +Y - Movement in +Y direction
- -Z - Movement in -Z direction
- +Z - Movement in +Z direction
- -4th - Movement in -4th direction
- +4th - Movement in +4th direction
- TRVRS - Rapid Traverse (toggle switch)

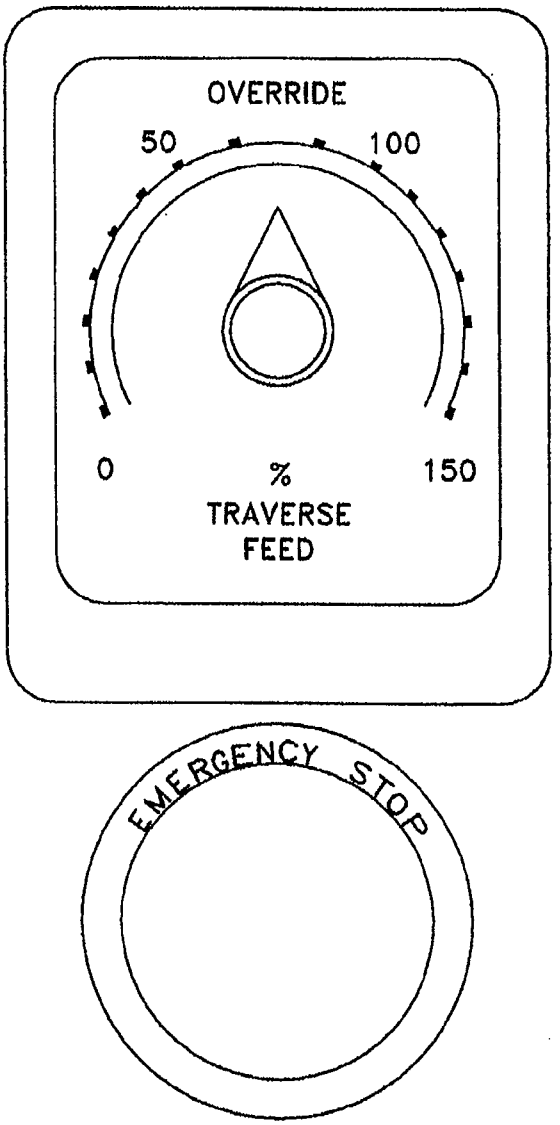
**SPINDLE**

- CW - Spindle movement clockwise
- STOP - Spindle Stop.
- CCW - Spindle movement counter clockwise

**COOLANT**

- CLNT ON - Coolant ON
- CLNT OFF - Coolant OFF
- CLNT AUTO - Coolant operated by program

## 4.3 EMERGENCY STOP AND MANUAL FEED OVERRIDE

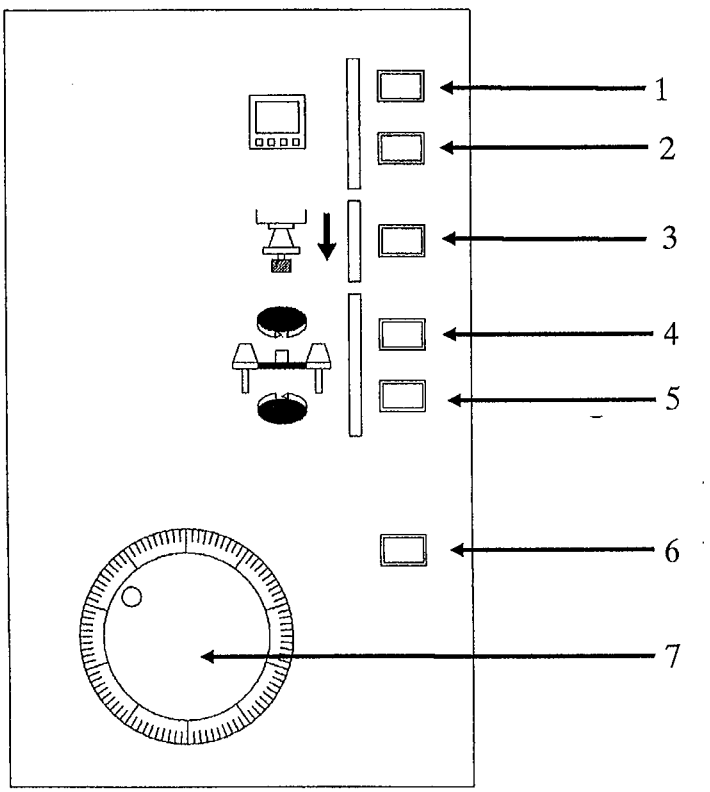



- Manual override of program feed and rapid traverse.
- Override of axis direction feed rates.

**EMERGENCY STOP**

- Cuts all power to drives.

## 4.3 OPERATORS PANEL




**Control**

- 1) Green button to start control system.
- 2) Red button to close down control system.

**A.T.C.**

- 3) Spindle clamp operator, push to open, release to close.
- 4) Carousel index clockwise by one position.
- 5) Carousel index counter clockwise by one position.

**Limit Switch**

- 6) Manual limit switch override.

**Handwheel**

- 7) Manual pulse generator. Manual movements of X,Y and Z axes when in handwheel mode.

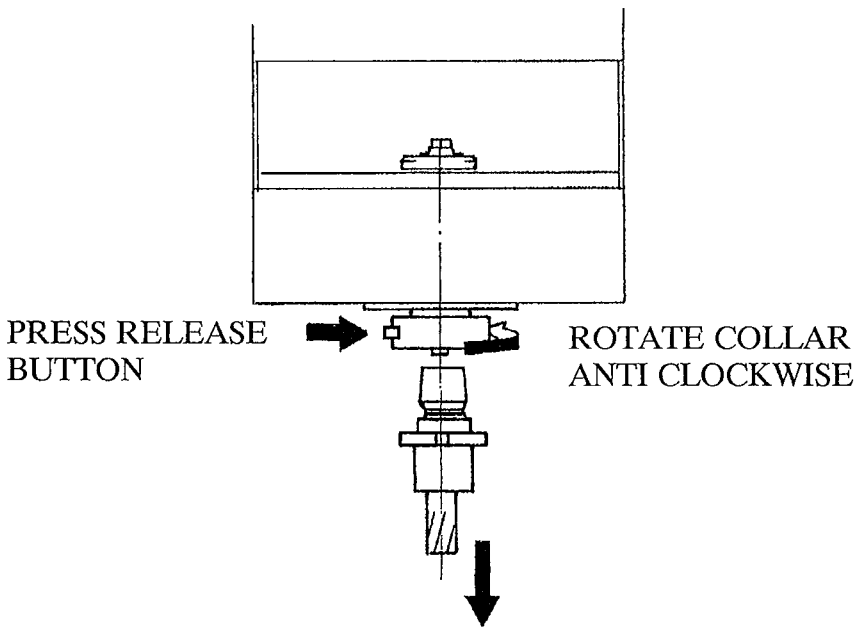
  Clockwise : Positive direction

  Counter clockwise : Negative direction

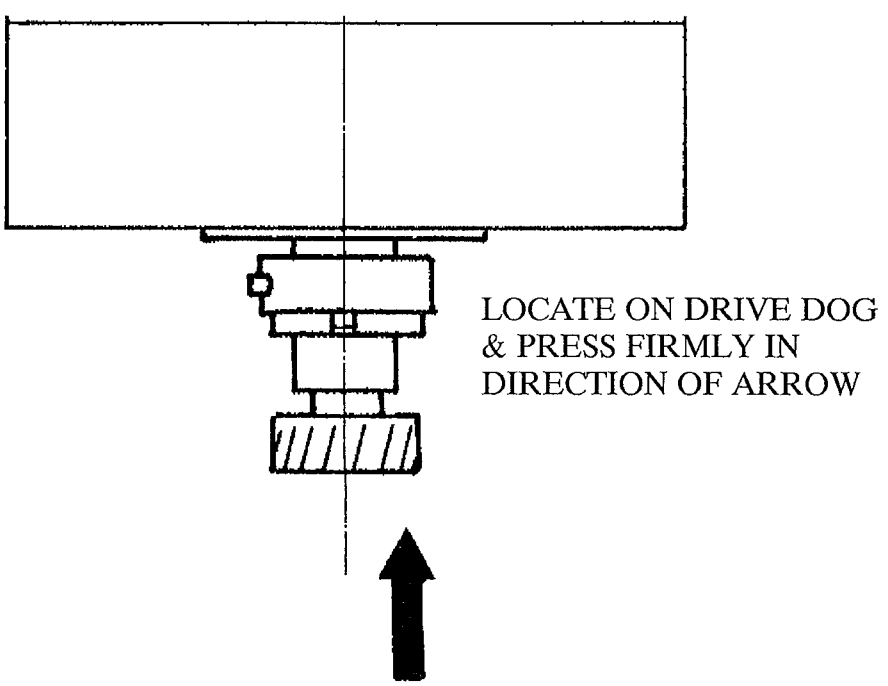
## 4.4 TRIAC FANUC TOOL CHANGING ( MANUAL )

### UNLOADING TOOL

DRAWBAR ( 7/16" UNF )
TO RELEASE R8 ARBOR
ONLY




### LOADING TOOL



## 4.4 TRIAC FANUC TOOL CHANGING ( PNEUMATIC DRAWBAR )

To change the tool when using a pneumatic drawbar use the flowing procedure:

- 1) Stop spindle.
- 2) Use spindle clamp operator button on operators panel ( See diagram section 4.3 ) to release tool. Press to release.
- 3) Replace old tool with new and release spindle clamp operator button to hold tool in spindle.

## 4.4 TRIAC FANUC TOOL CHANGING ( A.T.C. )

If the machine is fitted with an automatic tool changer then use the following procedure to change tools:

- 1) Change to M.D.I. mode.
- 2) Use M06 command followed by the tool number corresponding to the desired tool in the carousel.

## 4.5 TRIAC FANUC START UP PROCEDURE

The start up procedure for TRIAC FANUC is as follows:-

- Switch ON isolator found at rear of cabinet.
- Switch ON Control, Green button on Operators Panel, No.5 (See diagram Section 4.3)

**Wait for screen to display:-**

OPERATORS MESSAGE

DENFORD MACHINE TOOLS LIMITED

TRIAC FANUC

Serial _____

Date_____

P.L.C._____

## 4.6 DATUM OF AXES

**DATUM AXES AS FOLLOWS:-**

- Start up machine with P and CAN depressed when switching on power.(Special case , use only on delivery of machine.).
- Jog off limits.
- Press Home button on FANUC Manual Operations Panel.
- Press "+ X" button on FANUC Manual Operations Panel and wait for table to come to a stand still.
- Press "+ Y" button on FANUC Manual Operations Panel and wait for table to come to a stand still.
- Press "+ Z" button on FANUC Manual Operations Panel and wait for spindle head to come to a stand still
- Press "POS" button on FANUC Input Panel.
- Press Page arrow pointing downwards until page showing absolute, relative and machine positions is displayed.
- Ensure the X,Y and Z values are ZERO on "Machine" read out.

**The machine is now ready for use.**

**NOTE:-**

**If X,Y and Z values are not ZERO seek expert advice.**

## 4.7 RUN IN PROCEDURES

Before working on TRIAC FANUC for the first time it is important that the spindle is "RUN-IN" for a time.

This should be done to the following procedure:-

| SPEED | TIME |
|---|---|
| • 500 RPM | 15 MINUTES |
| • 1000 RPM | 10 MINUTES |
| • 2000 RPM | 10 MINUTES |

**Hence TRIAC FANUC is run in for a total of 35 Minutes**

To run in TRIAC FANUC follow the procedure on the next page.

## STARTING SPINDLE ON TRIAC FANUC

**To " RUN IN " TRIAC FANUC follow the following procedure:**

- 1. Select M.D.I. MODE on the automatic operations panel.(See section 4.3)
- 2. Press the PRGRM Key on the data input panel. (See section 4.3)
- 3. Press PAGE ↓ UNTIL the M.D.I. Screen is displayed.
- 4. Press M 03 INPUT
- 5. Press S 500 INPUT
- 6. Press CYCLE START

**NOTE:**

**ENSURE SPDL 100% IS ACTIVATED ON OPERATION PANEL.**

**To change speed:**

- 1. Press S 1000 INPUT ( the value after the S code should be changed to the numbers given on the previous page after the correct time interval ).
- 2. Press CYCLE START

**To stop spindle:**

- 1. Press M 05 INPUT
- 2. Press CYCLE START

## SECTION 5

### CNC APPLICATIONS

This page is intentionally blank.

## MANUAL PROGRAMMING

The first step in manual programming is to decide upon a machining method to achieve the specified shape, size and finish. This takes the form of a planning sheet which would be very similar to that for the same operation on a conventional miller.

Number of passes, tools, depths of cut, feeds and speeds are all included, and additionally, setting distanceS for tools, from a datum in each plane, are required.

The programmer then uses all this information to complete the program sheet, which is writen in rows of information, each row containing a block of tape data.

In writing the program, the programmer must perform calculations to allow for tool point radius when machining chamfers, tapers or arcs, when work suface programming is not active.

It is also necessary to calculate values for 3 further 'address' letters 'I', 'J' and 'K' when machining parts of a circle. These values represent the arc centre, and enable the interpolator to issue movement instructions to each axis.

When complete, the program sheet is typed into the machine memory and then recorded on a cassette tape.

This page is intentionally blank.

## SUMMARY OF TYPICAL ADDRESS CHARACTERS

| | |
|---|---|
| N | SEQUENCE NUMBER |
| X | PRIMARY MOTION 'X' AXIS |
| Y | PRIMARY MOTION 'Y' AXIS |
| Z | PRIMARY MOTION 'Z' AXIS |
| G | PREPARATORY FUNCTIONS |
| I | INCREMENTAL DISTANCE PARALLEL TO 'X' AXIS |
| J | INCREMENTAL DISTANCE PARALLEL TO 'Y' AXIS |
| K | INCREMENTAL DISTANCE PARALLEL TO 'Z' AXIS |
| R | RADIUS |
| M | MISCELLANEOUS FUNCTIONS |
| T | TOOL NUMBERS |
| S | SPINDLE SPEEDS |
| F | FEED RATES |

This page is intentionally left blank

## SECTION 5.2

### WORD DESCRIPTIONS

## 5.2.1 PART PROGRAMMING WORDS

In numerical control, a word by definition is considered to be a specific command or piece of data in a part program block. A word begins with an alphabetic character(the word address) and is followed by a value of one or more numerical digits.

The words on the following pages are used in part programming for the FANUC OM.

### GENERAL RULES

For words which are programmed in a decimal point format,the OM assumes leading zero suppression, when the decimal point is not programmed.

For words which are not programmed with a decimal point,

Leading Zero suppression is active.

Example:

- N10 is N0010
- G1 is G01

The number of numerical digits in a word is established. See Tape Format Specification.

The position of the decimal point for display is also established, See Tape Format Specification.

Leading or trailing zeros may be programmed in all words to clarify intention and simplify readability.

Program all minus (-) signs.

## 5.2.2 TAPE FORMAT SPECIFICATION

This specification is to describe the digit field which may follow each letter address given here. The meaning of the two digits following a letter address shows the implied position of the decimal point.

Example : X ± 5.3

This implies that X can be plus or minus with up to 5 digits before the decimal point and up to 3 digits after the point.

a) The standard Tape Format Specification with the feedrate in mm/rev (G95) is:-

| | | | |
|---|---|---|---|
| /1 :4 | N4 | G2 | X/U ± 4.3 |
| Y/V ± 4.3 | Z/W ± 4.3 | | |
| I ± 4.3 | J ± 4.3 | K ± 4.3 | |
| R4.3 | F3.4 | P7 | Q7 |
| S4 | T4 | M2 | |

b) For Programming Feedrate in mm/min (G94) the Format is:-

F5.0

### IMPERIAL AND METRIC

When commanding a dwell (G04) in seconds the Format for the X, U or P words in Imperial or Metric Mode is:-

| | | |
|---|---|---|
| X4.3 | U4.3 | P4300 |

## 5.2.3 WORD DETAILS

Although the Control will, in general, accept part programming words in any sequence, it is recommended that the word order for each block is used:-

/ N.G. X or U.Y or V.Z or W.I.K.F.S.T.M.

**/ Block**

A slash as the first character in a block allows that block to be ignored when the Block Delete pushbutton is activated.

Example: /N1020 M00

**O: Program Number**

The : followed by a 4 digit numerical value is used to assign a program number.

**N : Sequence Number**

The N word may be omitted. When programmed, the sequence number following the N address is a four digit numerical value and is used to identify a complete block of information. Although ascending, decending, or duplicate numbering is allowed, it is best to program in ascending order in increments of 10. This allows for future editing and simplified sequence number search.

G : Preparatory Command

The two digit G command is programmed to set up the control to perform an automatic machine operation. A fulllist of G codes are given. One G word from each modal group and one non modal G word can be programmed on the same block.

EXAMPLE : Valid N100 G00 G40 G90 G95

Non Valid N100 G00 G40 G41 G90 G95

- G40 & G41 are from the same group.

A retained G word (Modal) from one group remains active

until another G word from the same group is programmed.

One-shot G words (Non-Modal) must be programmed in every block when required.

A power down/up reactivates the G words which are indicated by an asterisk (*). See following pages.

## G CODES LISTING FOR DENFORD MILLERS

| | | | |
|---|---|---|---|
| Group | 1 | G00 | Positioning (Rapid Traverse) |
| | 1 | G01 | Linear Interpolation (Cutting Feed) |
| | 1 | G02 | Circular Interpolation/Helical CW |
| | 1 | G03 | Circular Interpolation/Helical CW |
| | 0 | G04 | Dwell, Exact Stop |
| | 0 | G09 | Exact Stop |
| | 0 | G10 | Data Setting |
| | 0 | G11 | Data Setting Mode Cancel |
| * | 2 | G17 | XY Plane Selection |
| | 2 | G18 | ZX Plane Selection |
| | 2 | G19 | YZ Plane Selection |
| | 6 | G20 | Input in inch |
| | 6 | G21 | Input in mm |
| | 0 | G27 | Reference Point Return Check |
| | 0 | G28 | Return to Reference Point |
| | 0 | G29 | Return from Reference Point |
| | 0 | G30 | 2nd Reference Point Return |
| | 0 | G31 | Skip Function |
| | 1 | G33 | Thread Cutting |
| | 0 | G39 | Corner Offset Circular Interpolation |
| | 7 | G40 | Cutter Compensation Cancel |
| | 7 | G41 | Cutter Compensation Left |
| | 7 | G42 | Cutter Compensation Right |

| | | | |
|---|---|---|---|
| | 8 | G43 | Tool Length Compensation + Direction |
| | 8 | G44 | Tool Length Compensation + Direction |
| * | 8 | G49 | Tool Length Compansation Cancel |
| * | 11 | G50 | Scaling Cancel |
| | 11 | G51 | Scaling |
| * | 14 | G54 | Work Co-ordinate System 1 Selection |
| | 14 | G55 | Work Co-ordinate System 2 Selection |
| | 14 | G56 | Work Co-ordinate System 3 Selection |
| | 14 | G57 | Work Co-ordinate System 4 Selection |
| | 14 | G58 | Work Co-ordinate System 5 Selection |
| | 14 | G59 | Work Co-ordinate System 6 Selection |
| | 0 | G60 | Single Direction Positioning |
| | 15 | G61 | Exact Stop Mode |
| | 15 | G62 | Automatic Corner Override |
| | 15 | G63 | Tapping Mode |
| * | 15 | G64 | Cutting Mode |
| | 0 | G65 | Macro Call, Macro Command |
| | 12 | G66 | Macro Modal Call |
| * | 12 | G67 | Macro Modal Call Cancel |
| | 16 | G68 | Co-ordinate Rotation |
| * | 16 | G69 | Co-ordinate Rotation Cancel |
| | 9 | G73 | Peck Drilling Cycle |
| | 9 | G74 | Counter Tapping Cycle |

| | | | |
|---|---|---|---|
| | 9 | G76 | Fine Boring |
| * | 9 | G80 | Canned Cycle Cancel |
| | 9 | G81 | Drilling Cycle, Spot Boring |
| | 9 | G82 | Drilling Cycle, Counter Boring |
| | 9 | G83 | Peck Drilling Cycle |
| | 9 | G84 | Tapping Cycle |
| | 9 | G85 | Boring Cycle |
| | 9 | G86 | Boring Cycle |
| | 9 | G87 | Back Boring Cycle |
| | 9 | G88 | Boring Cycle |
| | 9 | G89 | Boring Cycle |
| * | 3 | G90 | Absolute Command |
| | 3 | G91 | Incremental Command |
| | 0 | G92 | Programming of Absolute Zero Point |
| * | 5 | G94 | Feed per Minute |
| | 5 | G95 | Feed per Rotation |
| * | 10 | G98 | Return to Initial Point in Canned Cycle |
| | 10 | G99 | Return to R in Canned Cycle |

## NOTES FOR G CODE LISTING

**NOTE 1:**

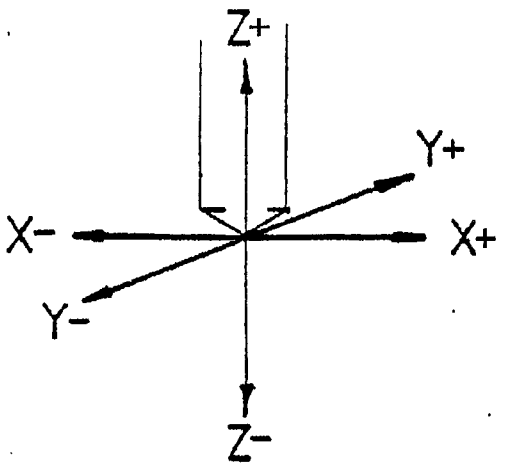
G Codes of 00 group represent those not modal and are effective to the designated block

**NOTE 2:**

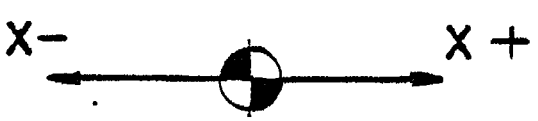
G Codes of different groups can be commanded to the same block infinely. If more than one G code of the same group are commanded the one commanded latter becomes effective.

## Axis Definitions

TRIAC FANUC is a three axis mill. The illustrations show the positive and negative axes, indicating the direction of the tool movement. This is an important concept to understand. Because all programming is done with this in mind, that is all motion programmed is the movement of the tool , not the movement of the table.

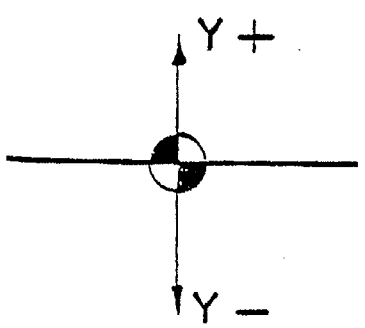



**X Direction from datum:**



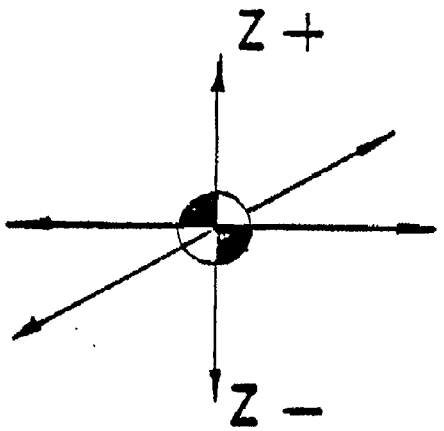

- X- Left forward movement of the tool.
- X+ Right forward movement of the tool.

**Y Direction from datum:**




- Y+ Tool moves away from the operator.
- Y- Tool moves towards the operator.

**Z Axis directions:**




**Incremental Axes ( G91 SET )**

**X Axis Command**

The X word is an incremental distance which is used to command a change in position perpendicular to the spindle centre line.

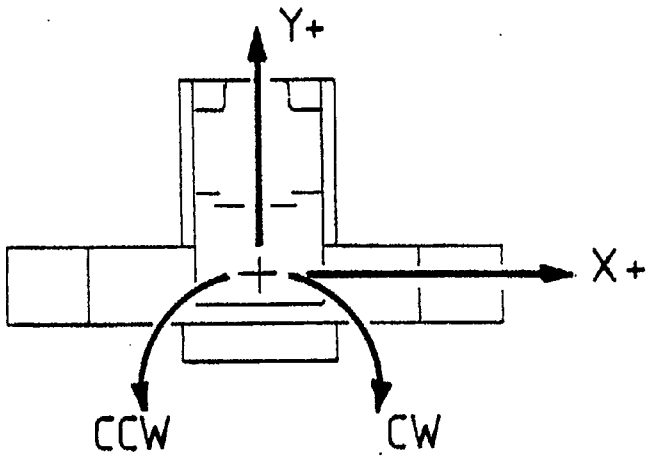
**Y Axis Command**

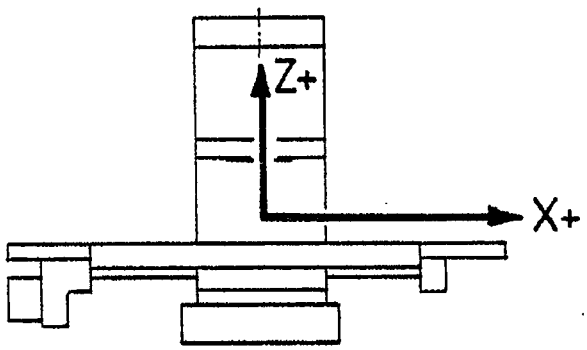
The Y Word is an incremental distance which is used to command a change of position away and towards the operator.

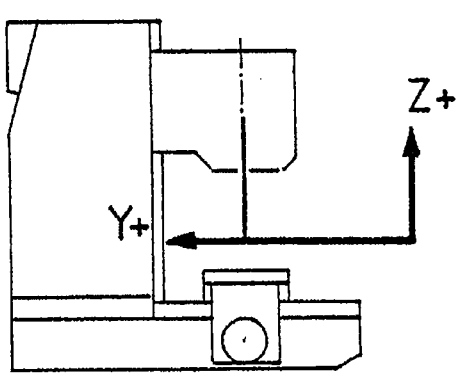
**Z Axis Command**

The Z word is an incremental distance which is used to command a change of position parallel to the spindle centre line.

Do not program X , Y or Z in the same block.


Y+
X+
CCW
CW


Z+
X+


Z+
Y+

**X : Dwell**

The X word is used with G04 to command a dwell in seconds.

**I WORD**

**a) G40 Active**

For arc programming (G02 or G03) the I value (with sign) is programmed to define the incremental distance parallel with the X axis between the cutter centre and the component arc centre at the start of arc.

**b) G41/G42 Active**

For arc programming (G02 or G03) the I value (with sign) is programmed to define the incremental distance parallel to the X axis between the start of the arc and the arccentre.

**J WORD**

**a) G40 Active**

For arc programming (G02 and G03) the J value (with sign) is programmed to define the incremental distance parallel with the Y axis between the cutter centre and the component arc centre at the start of arc.

**b) G41/G42 Active**

For arc programming (G02 and G03) the J value (with sign) is programmed to define the incremental distance parallel to the Y axis between the start of the arc and the arc centre.

**K WORD**

**a) G40 Active**

For arc programming (G02 or G03) the K value (with sign) is programmed to define the incremental distance parallel with the Z axis between the cutter centre and the component arc centre at the start of arc.

**b) G41/G42 Active**

For arc programming (G02 or G03) the K value (with sign) is programmed to define the incremental distance parallel to the Z axis between the start of the arc and the arc centre.

**F WORD**

a) In G95 mode the F word is used to command feed/rev.(optional)

b) In G94 mode the F word is used to command feed/min.

c) In G33 mode the F word specifies the lead of the

thread. (optional)

**P WORD**

Used with G10 to define the offset number in the tool geometry file.

**Q WORD**

Used with G10 the Q word defines the tool location code.

**R WORD**

a) For circular interpolation (G02 or G03) the arc radius is to the centre of the tool nose radius (G40 active) - or the actual radius required (G41/G42 active).

b) The R word is programmed with G10 to indicate the tool radius in the tool geometry/wear file for use with work surface programming (G41/G42).

**S WORD**

The four digit S word is used to command the required surface speed in either feet or metres per minute.

**T WORD**

Tool words are used to cause a tool change, and to activate tool geometry offsets to obtain the desired machining accuracy.

**M Word**

An M Word is used to initialise auxiliary functions particular to the machine. One M code can be programmed within one program block together with any other part program information.

## M CODE LIST FOR DENFORD MILLERS

* ALL M CODES MARKED WITH AN ASTERISK WILL BE EXECUTED AT THE END OF A BLOCK (I. E. AFTER THE AXIS MOVEMENT)

| | | |
|---|---|---|
| * | M00 | PROGRAM STOP |
| * | M01 | OPTIONAL STOP |
| * | M02 | PROGRAM RESET |
| | M03 | SPINDLE FORWARD |
| | M04 | SPINDLE REVERSE |
| * | M05 | SPINDLE STOP |
| | M06 | AUTO TOOL CHANGE |
| | M07 | COOLANT 'B' ON |
| | M08 | COOLANT 'A' ON |
| * | M09 | COOLANT OFF |
| | M10 | WORK CLAMP OPEN |
| | M11 | WORK CLAMP CLOSE |
| | M12 | |
| | M13 | SPINDLE FORWARD & COOLANT ON |
| | M14 | SPINDLE REVERSE & COOLANT ON |
| | M15 | PROGRAM INPUT USING, 'MIN P' (SPECIAL FUNCTION) |
| | M16 | |
| | M17 | |
| | M18 | |
| | M19 | SPINDLE ORIENTATE |

M20 ATC ARM IN

M21 ATC ARM BACK

M22 ATC ARM DOWN

M23 ATC ARM UP

M24 ATC DRAWBAR UNCLAMP

M25 ATC DRAWBAR CLAMP

M26

M27 RESET CAROUSEL TO POCKET ONE

M28 RESET CAROUSEL TO POCKET POSITION

M29 SELECT 'DNC' MODE

M30 PROGRAM RESET & REWIND

M31 INCREMENT PARTS COUNTER

M32

M33

M34

M35

M36

M37 DOOR OPEN TO STOP

M38 DOOR OPEN

M39 DOOR CLOSE

M40 PARTS CATCHER EXTEND

M41 PARTS CATCHER RETRACT

M42

| | | |
|---|---|---|
| | M43 | SWARF CONVEYOR FORWARD |
| | M44 | SWARF CONVEYOR REVERSE |
| * | M45 | SWARF CONVEYOR STOP |
| | M46 | |
| * | M47 | |
| | M48 | LOCK % FEED AND % SPEED AT 100% |
| | M49 | CANCEL M48 (DEFAULT) |
| | M50 | |
| | M51 | |
| | M52 | |
| | M53 | |
| | M54 | |
| | M55 | |
| | M56 | |
| | M57 | |
| | M58 | |
| | M59 | |
| | M60 | |
| | M61 | |
| | M62 | AUX. 1 ON |
| | M63 | AUX. 2 ON |
| | M64 | AUX. 1 OFF |
| | M65 | AUX. 2 OFF |

M66 WAIT FOR INPUT 1

M67 WAIT FOR INPUT 2

M68 ONLY INDEX WITH ALL AXIS AT HOME POSITION

M69 INDEX TURRET ANYWHERE

M70 MIRROR IN X ON

M71 MIRROR IN Y ON

M72

M73 MIRROR IN IV ON

M74

M75

M76 WAIT FOR INPUT 1 TO GO LOW

M77 WAIT FOR INPUT 2 TO GO LOW

M78

M79

M80 MIRROR IN X OFF

M81 MIRROR IN Y OFF

M82

M83 MIRROR IN IV OFF

M84

M85

M86

M87

M88

M89

M90

M91

M92

M93

M94

M95

M96

M97

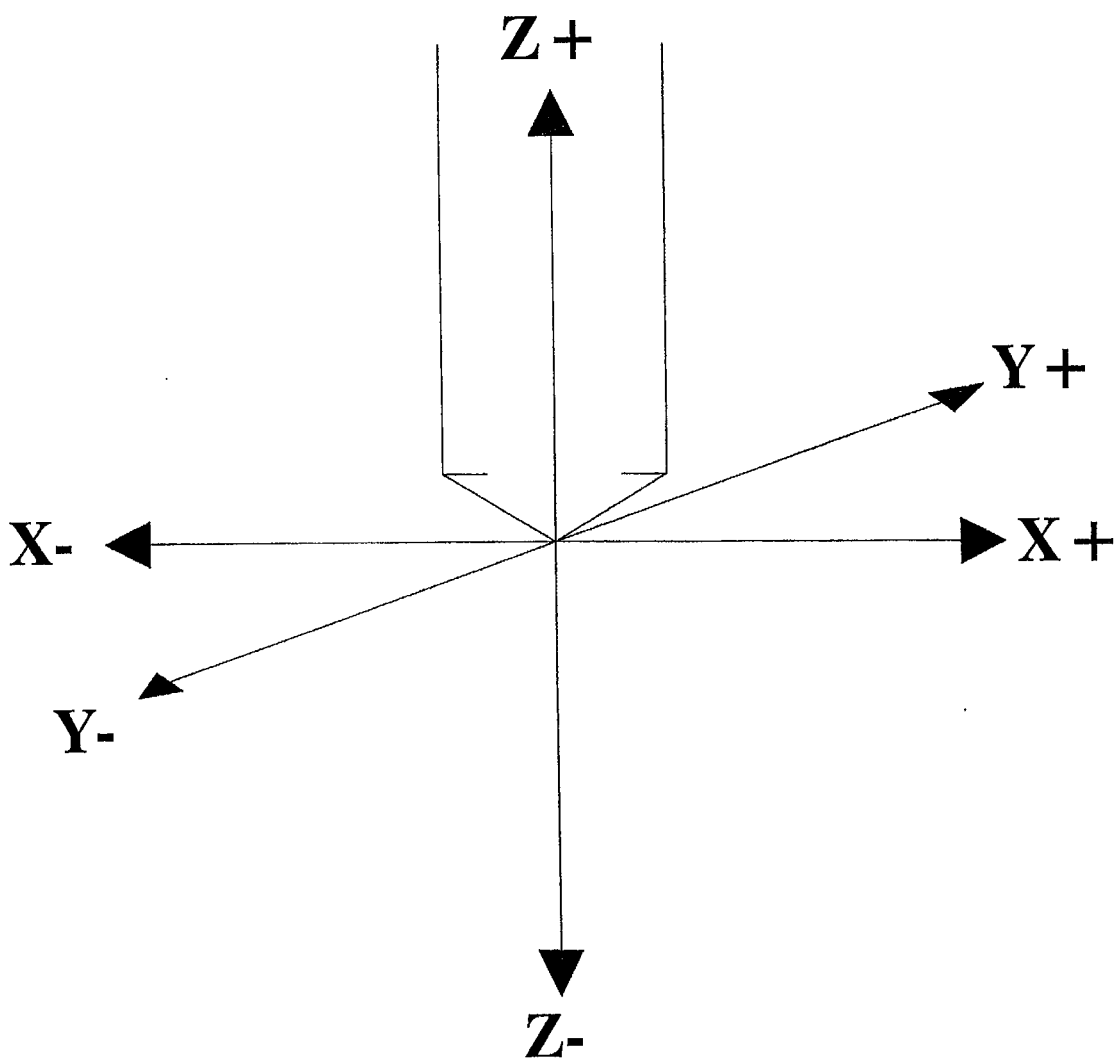
M98 SUB PROGRAM CALL

M99 SUB PROGRAM END

## SECTION 5.3

### BASIC MOVEMENTS

- BASIC MOVES



## 5.3.1 BASIC MOVEMENTS

i) Straight Line Motion (G00, G01) (Descriptions of the appropriate 'G' codes are given in Section 5.2.3 ).

Almost all motion commands will be programmed as straight line motion for moving to or cutting along a face. During cutting statements, it is only when an arc is to be generated that it will be necessary to change this mode of operation.

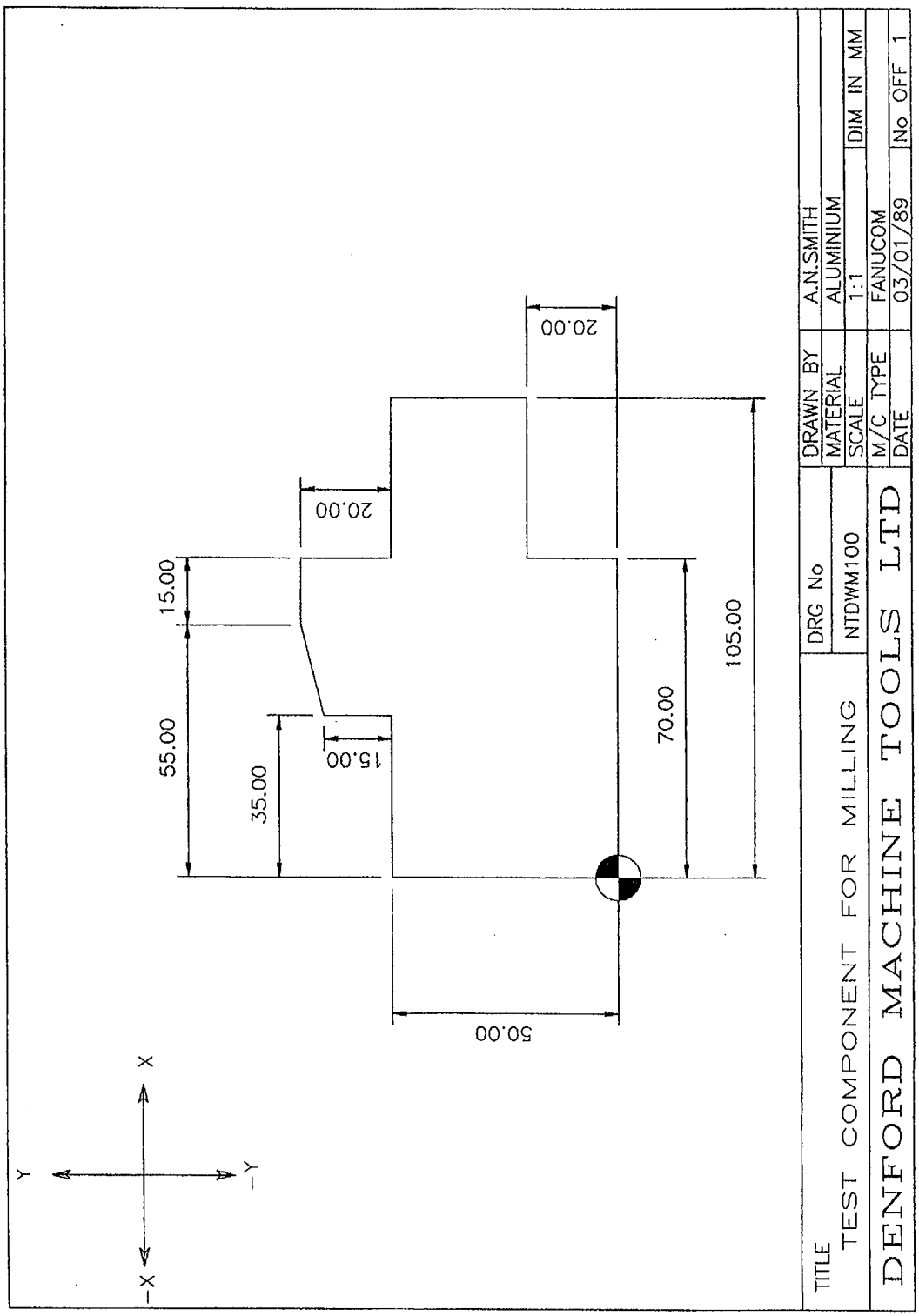
Any point on a component can be defined by X,Y & Z dimensions.

In absolute programming the programmer commands the tool to move to a position which is relevant to a component zero. The direction of motion is determined by the system and is dependent on the previous position of the tool.

The programmed word which instructs the control to accept 'X', 'Y' and 'Z' values as absolute is G90.

In incremental programming ( G91 ) the programmer commands the tool point to move to a new position which is a specific distance and direction from its previous position. The direction of motion is determined by the sign of the value for the command.

## EXAMPLE 1


20.00
20.00
15.00
55.00
35.00
15.00
70.00
105.00
50.00
X
Y
-Y
-X
DENFORD MACHINE TOOLS LTD
TITLE
TEST COMPONENT FOR MILLING
DRG No
NTDWM100
DRAWN BY
A.N.SMITH
MATERIAL
ALUMINIUM
SCALE
1:1
DIM IN MM
M/C TYPE
FANUCOM
DATE
03/01/89
No OFF 1

This page is intentionally left blank

## SECTION 5.4

### PROGRAMMING INSTRUCTIONS

## 5.4.1 PLANNING AND PROGRAMMING PROCEDURE

The following procedure may be used as a guide to assist the programmer, by describing each step required in preparing the method of production. Before writing the part program a detailed planning procedure is required.

### Planning Procedure

- 1. Receive part drawing. From part drawing information, check suitability of part to be machined against the machine capacity (clearances).
- 2. Determine a method for holding the component.
- 3. Determine the tooling required to suit the method of machining and utilise as much as possible the tools which are permanently in the carousel on the machine.
- 4. Determine the order of machining and the carousel stations.
- 5. Determine planned stops (cycle interrupt procedure incorporating block delete codes) for checking dimensional sizes where required by the operator.
- 6. Determine cutting speeds bearing in mind the following:
- a) Component material, tooling, rigidity of component.
- b) The tooling selected: toolholders with carbide inserts, the grade of the carbide insert selected, carbide drills, high speed steel drills, ceramics; for roughing and finishing operations.
- 7. Determine depths of cut and feeds for roughing operations bearing in mind the horsepower/kilowatts available for cutting and the rigidity of the part.
- 8. Determine from surface finish requirements the tooling most suited for the finishing operations and determine feedrates.
- 9. Allocate tool offsets as required.
- 10. Complete planning sheet.

## PROGRAMMING PROCEDURE

- 1. After completing the planning sheet draw the component to scale showing the cutter paths (a scale drawing is not mandatory where the component shape and the cutting paths are simple. A sketch may prove sufficient).
- 2. Select a component datum and dimension in scale drawing (or sketch) with the length of cuts relative to the component datum. Carry out necessary calculations at slopes and arcs.
- 3. Draw tooling layout sheet showing tools to be used in the program and indicate the carousel numbers for each tool.
- 4. Complete the tooling layout sheet by indicating the ordering code for each tool and the grade and type of tool to be used.
- 5. Commence writing starting procedure onto program sheets.
- 6. Input instruction codes into machine.

## 5.4.2 SETTING OF WORKING DATUM

A working datum is essential in all work done on a CNC Milling Machine so that the machine knows the position of the work piece in relation to its own machine datum.

When entering a program the offset values are inputted via the G92 code. The offsets are usually incorporated at the beginning of a program.

e.g

N30 G92 X0 Y0 Z0

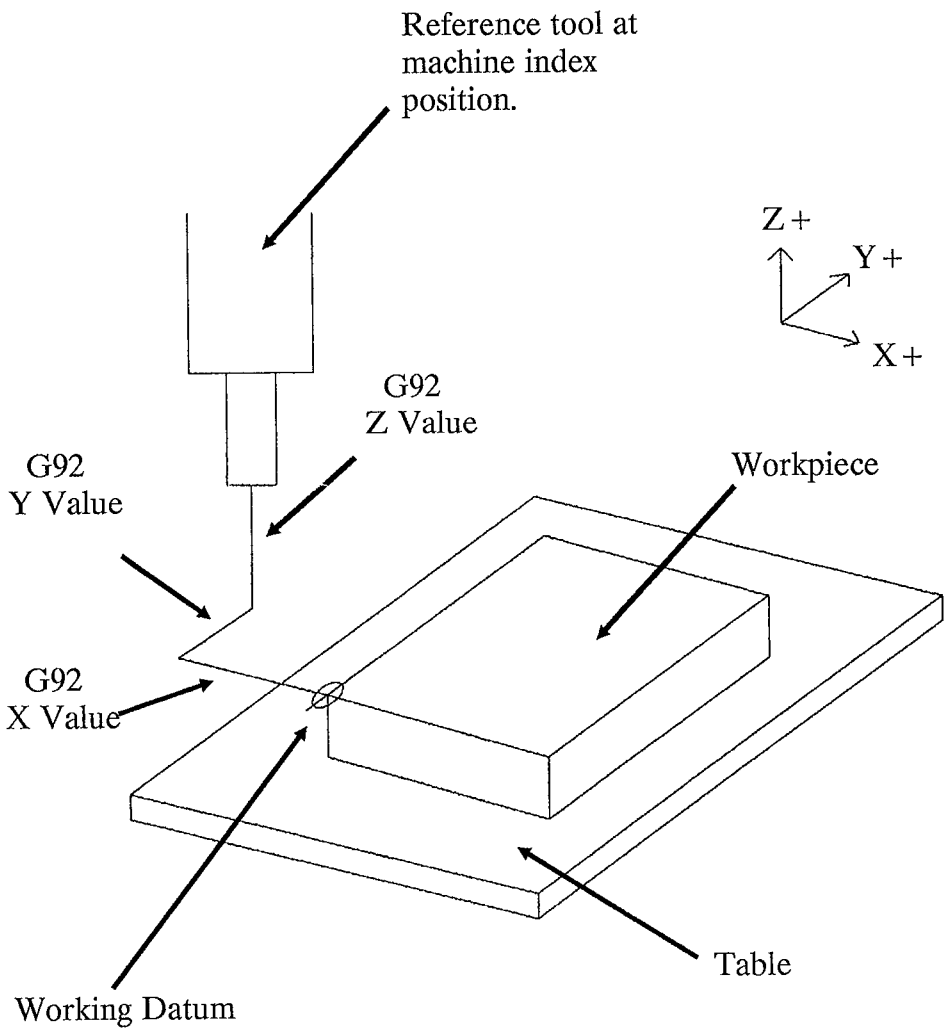
When programming these values are usually set to zero and then determined during setting up.

**Note:**

If details concerning the dimensions of the workpiece, the precise location of its fixtures and the precise lengths of tools are available the actual axis offset values maybe put in at this stage.

Once the program is in the offset values are usually inputted by the operator driving the reference tool (usually either the longest or the most frequently used) to a known position relative to the workpiece. Comparison of the position readout at machine reference point and at known position relative to the workpiece will reveal the appropriate G92 values, See diagram on next page.

## 5.4.2 SETTING WORKING DATUM



## 5.4.2 G92 - TOOLSETTING - TRIAC FANUC

1. DATUM MACHINE AS IN SECTION 4.6.

2. INDEX CAROUSEL TO TOOL NO. 1

3. IN JOG OR HANDWHEEL MODE, MOVE TOOL TIP TO COMPONENT ZERO.

4. PRESS - POS, TO SELECT POSITION RELATIVE SCREEN

5 PRESS X, Y AND Z and CAN, THIS WILL SET ALL THE AXES TO ZERO.

6. PRESS - HOME

7. PRESS - +Z, +X, -X . THE 3 VALUES FOR THESE AXES ARE ENTERED IN G92 - THIS HAS SET REFERENCE TOOL ie. TOOL NO1, SO OFFSET FOR TOOL1 WILL READ 0 IN THE OFFSET FILE.

8. JOG TO POSITION IN Z FOR ALL REMAINING TOOLS AS FOR TOOL No. 1.

9. PRESS - OFFSET MENU, AND SELECT OFFSET TO MATCH TOOL NUMBER USING CURSOR

10. PRESS - EOB AND WHILST HOLDING THIS KEY PRESS - 6Z KEY, THIS WILL DISPLAY TOOL 2 AND RELATIVE POSITION AT THE BOTTOM OF THE SCREEN

11. PRESS - INPUT AND THIS WILL ENTER THE VALUE IN THE OFFSET FILE.

12. REPEAT 1. TO 11. FOR REMAINING TOOLS.

13. TO ENTER THE TOOL RADIUS COMPENSATION IN THE OFFSET FILE , KEY IN THE VALUES AND PRESS - INPUT OPPOSITE THE RELATIVE OFFSET NUMBER.

## 5.4.3 TOOL LENGTH COMPENSATION (OFFSET)

After setting up the G92 command the reference tool will drive the X0 Y0 Z0 (i.e component datum) if given the command:

G00 X0 Y0 Z0

If a second tool is given the same command it will drive to

X0 Y0 Z ?.

The value ? is equal to the difference in length between the reference tool and the second tool. This discrepancy can be compensated for by having a tool length offset associated with the second tool. In this case a value of ? would be stored in the tool offset file. The tool offset file may be addressed and modified by the machine operator.

Any movement called for by a tool is modified by the tool offset value currently held on file for that tool, i.e 0 for the reference tool and, if the second tool is shorter than the reference tool, + ? for the second tool.

## 5.4.3 SETTING TOOL LENGTH COMPENSATION ON TRIAC FANUC

The following procedure should be followed to set the tool length compensation for each tool.

- 1) Pick tool to be set.
- 2) In JOG mode move tool tip to component zero.
- 3) Set display to tool offset page.
- 4) Move cursor to an unused offset number.
- 5) Enter Z value using numeric key pad.
- 6) Enter INPUT

This insets the offset value into the address. The value can be called up in the program using the command H.....

Example:

H01

Selects tool offset number one.

## 5.4.4 CUTTER RADIUS COMPENSATION

Each tool has a cutter radius compensation associated with it. This is of particular value for contouring operations, when the control is capable of modifying the programmed "cutter centre" path to take into account the actual point of cutter contact.

In most cases the "nominal" diameter of the cutter will provide an appropriate level of accuracy. In some cases however, e.g the contouring of a pocket with a tight tolerance on width, the cutter radius offset may have to be adjusted after measuring or probing in a similar fashion to the length offset.

Cutter radius compensations are held in H values and called into the program in the same way as G92 but using the radius compensation commands G41 and G42

e.g N150 G41 G00 X5-0 Y10.0 H05

**Note:**

G41 calls for compensation on left hand side of work looking along direction of travel.

G42 call for compensation on right hand side of work looking along direction of travel.

This page is intentionally left blank

## EXAMPLE PROGRAM FOR TRIAC FANUC

- **TEST PROGRAM FOR TRIAC FANUC**

  :0100

  N10 G21 G40 G49 G80

  N20 G91 G94 G00 G28 X0 Y0 Z0

- **THE FIRST TWO LINES SET THE MACHINE UP IN THE UNITS AND FORMAT REQUIRED. MACHINE SENT TO REF. POSITION.**

  N30 M06 T0101

- **TOOL 1 IS INSERTED INTO THE SPINDLE (6MM SLOT DRILL)**

  N40 G92 X Y Z

- **DATUM POSITION** - X, Y AND Z VALUES TAKEN FROM THE MACHINE POSITION READ-OUT.

  N50 G90 G00 X0 Y20 Z10 G43 H01 M03 S3000

  N60 G01 Z-2 F120

  N70 Y50

  N80 X35

  N90 Y65

  N100 X55 Y70

  N110 X70

  N120 Y50

  N130 X105

  N140 Y20

  N150 X70

  N160 Y0

  N170 X0

  N180 Y22

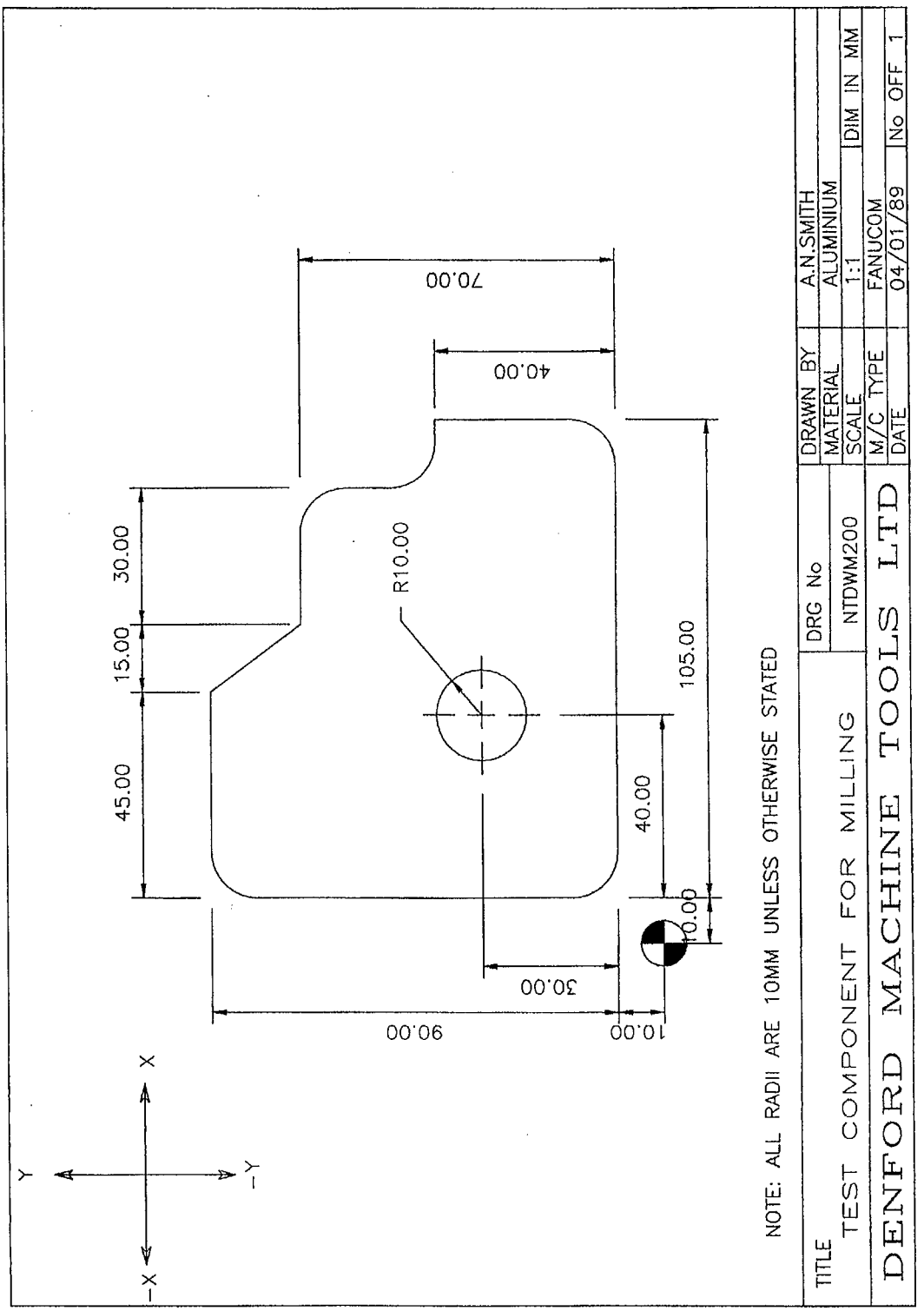
  N190 G00 Z10

N200 M05

N210 G91 G28 X0 Y0 Z0

N220 M30

## EXAMPLE 2


70.00
40.00
30.00
15.00
45.00
R10.00
105.00
40.00
10.00
30.00
90.00
10.00
X
-X
Y
-Y
NOTE: ALL RADII ARE 10MM UNLESS OTHERWISE STATED
DRG No
NTDWM200
DRAWN BY
A.N.SMITH
MATERIAL
ALUMINIUM
SCALE
1:1
DIM IN MM
M/C TYPE
FANUCOM
DATE
04/01/89
No OFF 1
TITLE
TEST COMPONENT FOR MILLING
DENFORD MACHINE TOOLS LTD

**TEST PROGRAM FOR TRIAC FANUC**

:0200

N10 G21 G40 G49 G80

N20 G91 G94 G00 G28 X0 Y0 Z0

- **THE FIRST TWO LINES SET THE MACHINE UP IN THE UNITS AND FORMAT REQUIRED. MACHINE SENT TO REF. POSITION.**

N30 M06 T0101

- **TOOL 1 IS INSERTED INTO THE SPINDLE.**

N40 G92 X Y Z

- **DATUM POSITION** - X, Y AND Z VALUES TAKEN FROM THE "MACHINE" READ-OUT.

N50 G90 G41 G00 X0 Y-10 Z10 G43 H01

- **CUTTER COMPENSATION USED H17 = OFFSET No.17**

N60 M03 S3000

N80 Z-2 F120

N85 G00 Y0 X0 G14 H17 (OFFSET 17 IS RADIUS OF TOOL 1.).

N86 G01 Y20

N90 Y80 F150

N100 G02 X10 Y90 R10

N110 G01 X45

N120 X60 Y70

N130 X80

N140 G02 X90 Y60 R10

N150 G01 Y50

N160 G03 X100 Y40 R10

N170 G01 X105

N180 Y10

N190 G02 X95 Y0 I-10 J0

N200 G01 X10

N210 G02 X0 Y10 I0 J10

N220 G01 Y22

N230 G00 Z2 G40

N240 Y40 G42 H17

N250 X40

N260 G01 Z-2

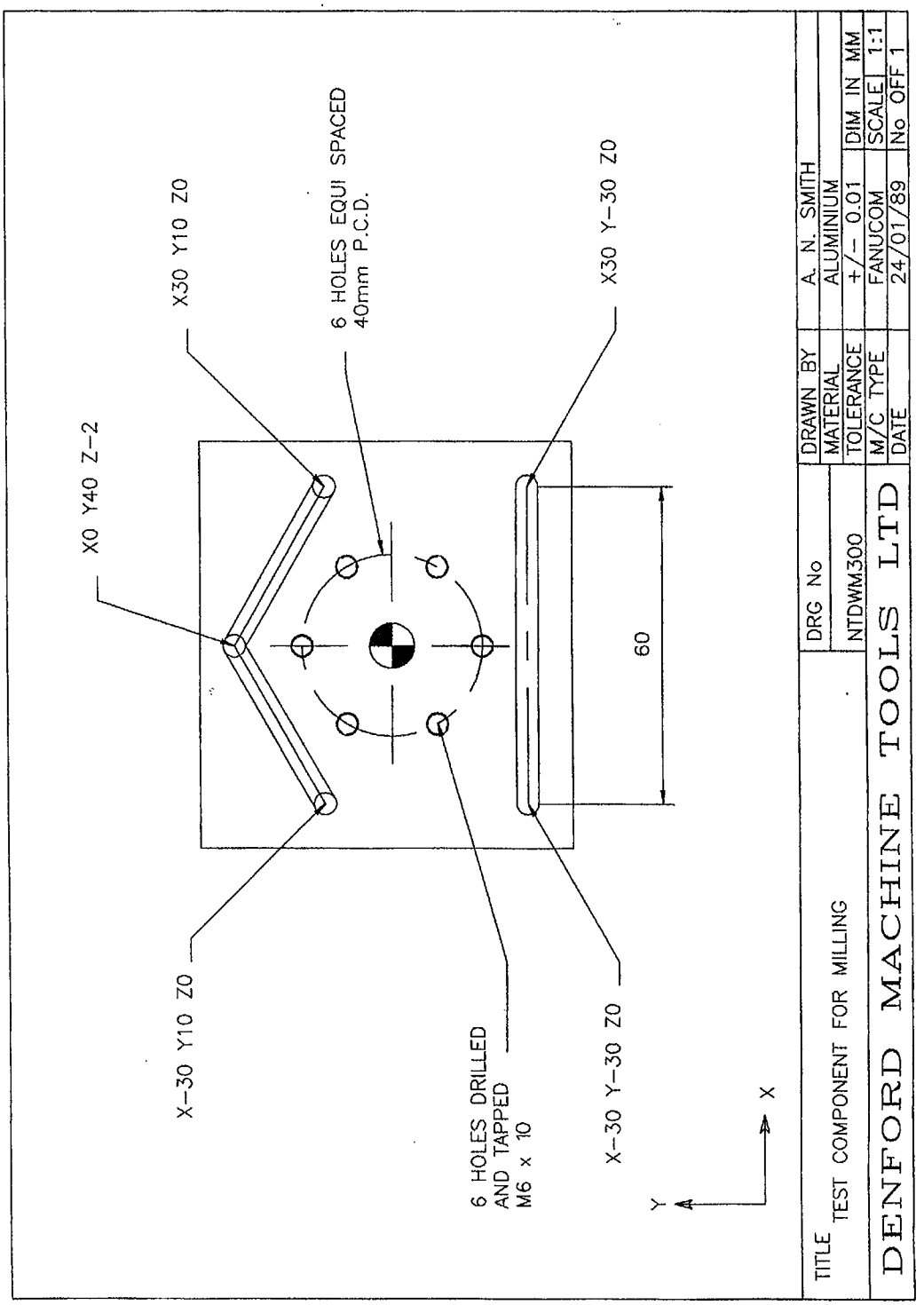
N270 G02 X40 Y40 I0 J-10

N280 G01 Z10 G40

N290 M05

N300 G91 G00 G28 X0 Y0 Z0

## EXAMPLE 3


X0 Y40 Z-2
X30 Y10 Z0
6 HOLES EQUI SPACED
40mm P.C.D.
X30 Y-30 Z0
60
X-30 Y10 Z0
6 HOLES DRILLED
AND TAPPED
M6 x 10
X-30 Y-30 Z0
Y
X
DENFORD MACHINE TOOLS LTD
TITLE
TEST COMPONENT FOR MILLING
DRG No
NTDWM300
DRAWN BY
A. N. SMITH
MATERIAL
ALUMINIUM
TOLERANCE
+/- 0.01
DIM IN MM
M/C TYPE
FANUCOM
SCALE
1:1
DATE
24/01/89
No OFF 1

:3000

N10 G21 G40 G49 H00 G80

N20 G91 G94 G00 G28 X0 Y0 Z0

N30 G92 XYZ ( **DATUM POSITION** )

**(No. 4 CENTRE DRILL)**

N40 M06 T0101

N50 M03 S2000

N60 G90 G43 H01 G00 X0 Y20 Z10

N70 G81 G99 X0 Y20 Z-7 R2 F80

N80 G99 X17.32 Y10

N90 G99 Y-10

N100 G99 X0 Y-20

N110 G99 X-17.32 Y-10

N120 G98 Y10

N130 G80 M05

N140 G91 G28 X0 Y0 Z0

**(5mm TWIST DRILL)**

N150 G49 M06 T02 H00

N160 M03 S3000

N170 G90 G43 H02 G00 X0 Y20 Z10

N180 G83 G99 X0 Y20 Z-12 R2 Q7 F100

N190 G99 X17.32 Y10

N200 G99 Y-10

N210 G99 X0 Y-20

N220 G99 X-17.32 Y-10

N230 G98 Y10

N240 G80 M05

N250 G91 G28 X0 Y0 Z0

**(M6 x 1 TAP)**

N260 G49 M06 T03 H00

N270 M03 S200

N280 G90 G43 H03 G00 X0 Y20 Z10

N290 G84 G99 X0 Y20 Z-8 R5 F200

N300 G99 X17.32 Y10

N310 G99 X0 Y-20

N320 G99 X-17.32 Y-10

N330 G98 Y10

N340 G80 M05

N350 G91 G28 X0 Y0 Z0

**(6mm SLOT DRILL)**

N360 G49 M06 T04 H00

N370 M03 S2600

N380 G90 G43 H04 G00 X-30 Y10 Z10

N390 Z2

**(THREE AXES LINEAR MOVE)**

N400 G01 Z0 F180

N410 X0 Y40 Z-2

N420 X30 Y10 Z0

N430 G00 Z2

N440 X-30 Y-30

**(PLANE SWITCHING)**

N450 G18 G01 Z0

N460 G02 X30 Z0 R300

N470 G17 G00 Z10

N480 M05

N490 G91 G28 X0 Y0 Z0

N500 M30

This page is intentionally left blank

## SECTION 5.5

### SUBROUTINES AND EXAMPLES OF CYCLES

## PROGRAM AND SUBROUTINE IDENTIFICATION

The first block of a program/subroutine must contain an identification number.

The program would be as follows:-

| | |
|---|---|
| % | Wind/Rewind Stop |
| :1234 | Program Number |
| N10 | |
| .... ) | |
| ..... ) | Part Program Blocks ) |
| ..... ) | |
| N ..... M30 | Last Block |
| % | Wind/Rewind Stop |

The Subroutine would be as follows:-

| | |
|---|---|
| % | |
| :5678 | Subroutine Number |
| N10 ..... | |
| ..... | |
| ..... | |
| N..... M99 | Jump Back to Main Program. |

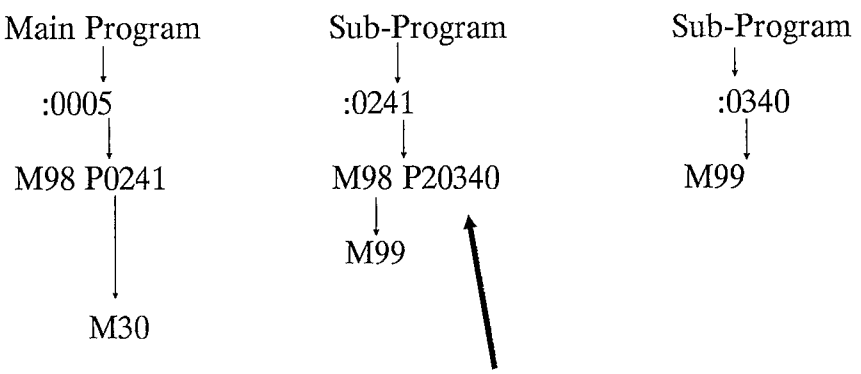
## 5.5.1 SUBPROGRAMS (SUBROUTINES)

Using the program jump functions, a machining program with repeated machining or function sequences can be simplified.

The machining sequences, which are repeated and can be used several times, are stored as subroutines and called up using the program jump functions.

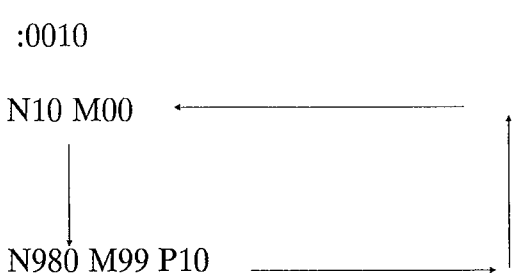
M98 - Jump command to another program.

M99 - Return command.




This repeats program 0340 twice.

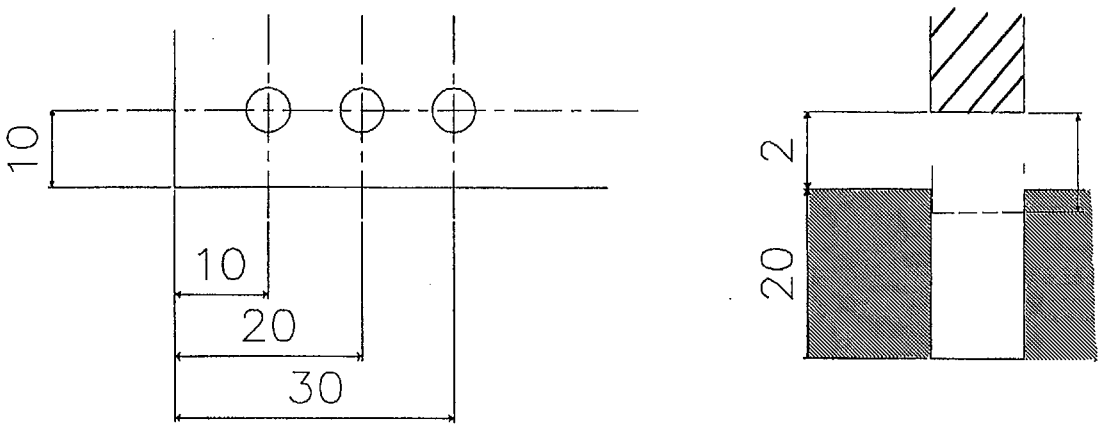
M99 can be used to return to the start of the program.




M99 generally indicates the end of a subroutine and allows the jump back to the main program. If it is used with 'P' address this indicates the 'Jump To' block number. The program will read the M99 P10 (GOTO N10), i.e. automatic return to the start.

Line N10 must read M00 to stop the cycle to enable a component to be loaded. All information prior to N10 (i.e. standard tool geometry/wear) would not be read after the first cycle. M30 would not be programmed in this case.

## 5.5.2 G73 HIGH SPEED PECK DRILLING




Used for efficient machining, permits very low retraction value.

N100 M06 T03

N110 M03 S2000

N120 G90 G00 X10 Y10 Z10 P43 H03

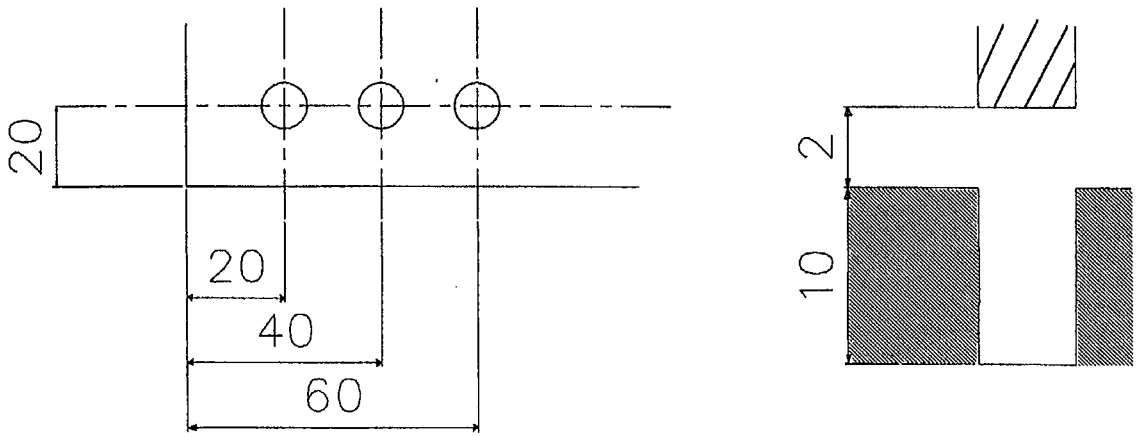
N130 G73 G99 X10 Y10 Z-20 Q3 R2 F2000

N140 X20

N150 X30

N160 G80

## 5.5.3 G74 LEFT HAND TAPPING




N100 M06 T04

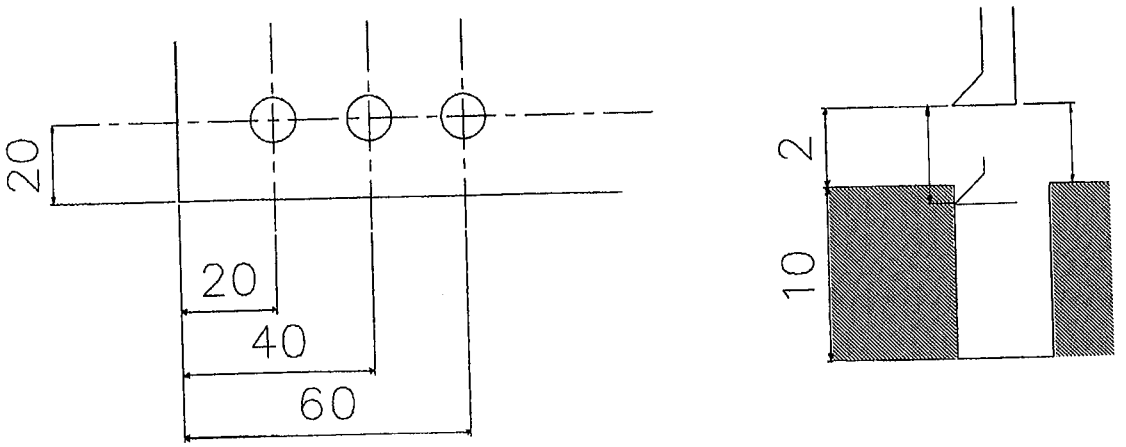
N110 M03 S2000

N120 G90 G00 X00 Y20 Z10

N130 G91 G74 G99 X20 Z-12 R-8 K3 F100

N140 G80

## 5.5.4 G76 FINE BORING CYCLE




N100 M06 T05

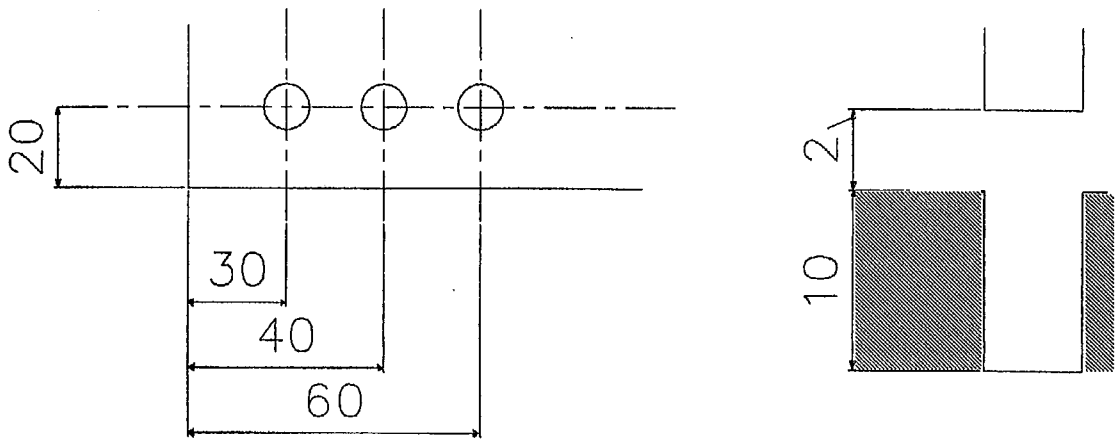
N110 M03 S2000

N120 G90 G00 X00 Y20 Z10

N130 G91 G76 G99 X20 Z-12 Q0.5 R-8 K3 F200

N140 G80

## 5.5.5 G81/G82 DRILLING CYCLE, SPOT BORING CYCLE




N100 M06 T05

N110 M03 S2000

N120 G90 G00 X30 Y20 Z10

N130 G81 G99 X30 Y20 Z-10 R2 F200

N140 X40

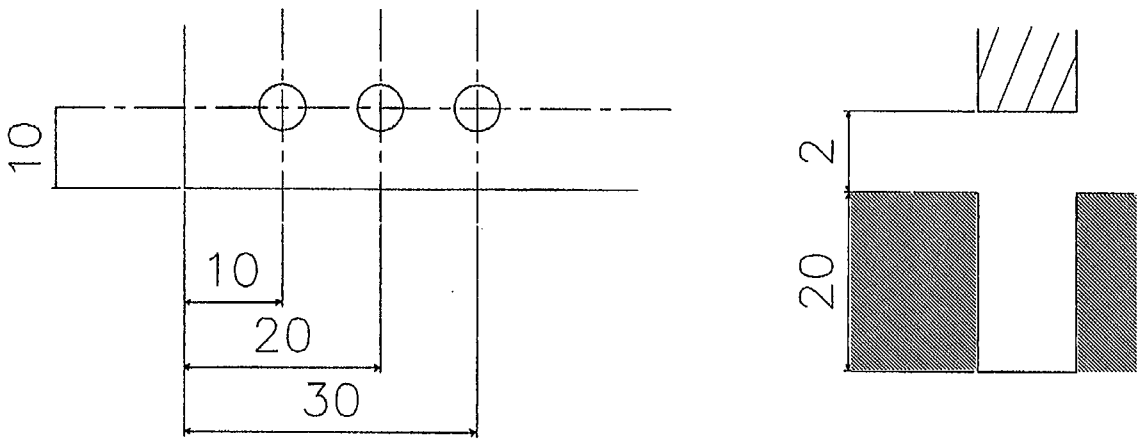
N150 G98 X60

N160 G80

**Note:**

G82 is the same as G81 but has a dwell time at the bottom of the hole.

## 5.5.6 G83 PECK DRILLING CYCLE




N100 M06 T03

N110 M03 S2000

N120 G90 G00 X10 Y10 Z10

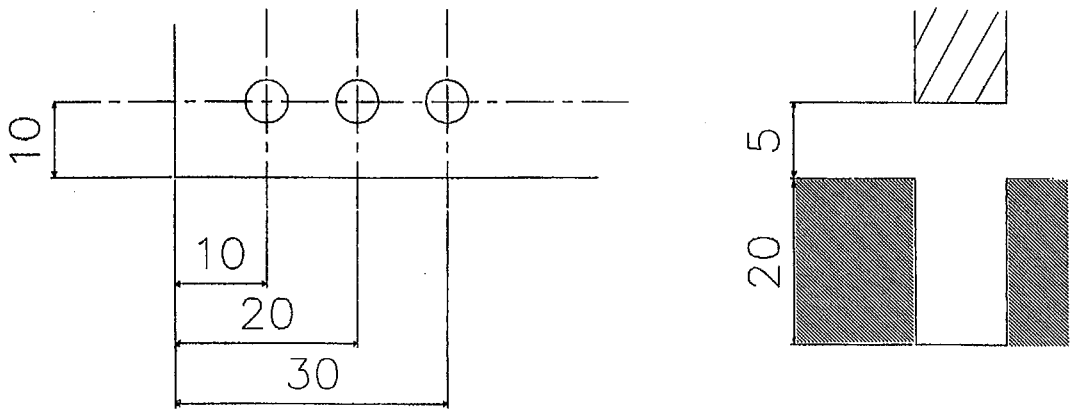
N130 G83 G99 X10 Y10 Z-20 Q3 R2 F200

N140 X20

N150 G98 X30

N160 G80

## 5.5.7 G84/G85 TAPPING CYCLE




N100 M06 T03

N110 M03 S250

N120 G90 G00 X10 Y10 Z10

N130 G84 G99 X10 Y10 Z-20 R5 F250

N140 X20

N150 G98 X30

N160 G80

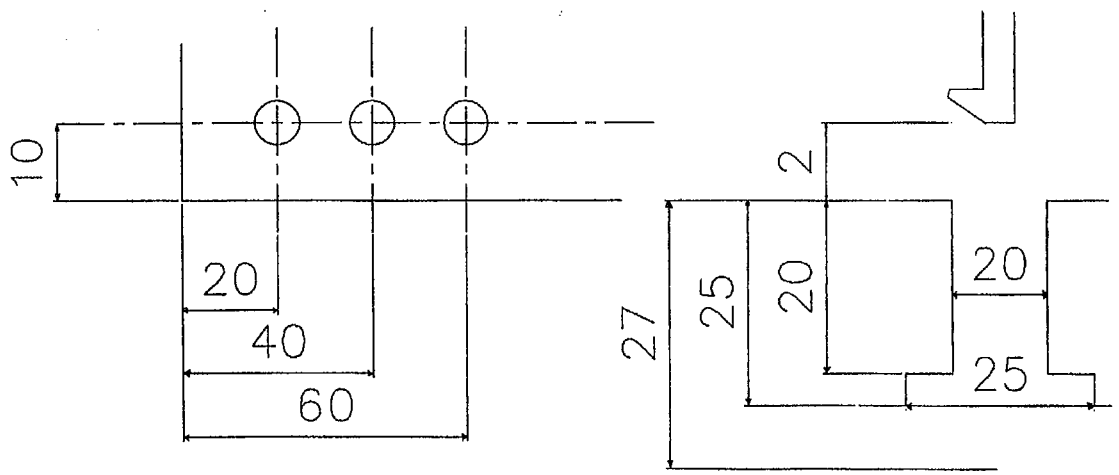
**Note:**

Spindle direction is reversed at the bottom of the hole for G84, but for G85 the spindle direction is constant.

## 5.5.8 G87 BORING CYCLE, BACK BORING CYCLE

**NOTE:**

**MOUNT WORK CLEAR OF TABLE.**




N100 M06 T04

N110 M03 S2000

N120 G00 X20 Y10 Z20

N130 G87 G98 X20 Y10 Z20 Q4 R-27 F100

N140 X40

N150 X60

N160 G80

**Note:**

After tool is in position spindle stops. The spindle then shifts in direction opposite to the tool and the tool is placed at the bottom of the hole. The spindle spindle rotates clockwise and machining is perfomed upwards.

## SECTION 6

### CUSTOMER'S OBSERVATIONS OF THIS MANUAL

This page is intentionally blank.

**PLEASE RETURN TO DENFORDS IF YOU FEEL ANY AMENDMENTS ARE REQUIRED TO THIS MANUAL.**

**FROM:**

**TO:**

THE TECHNICAL AUTHOR,
DENFORD MACHINE TOOLS LIMITED,
BIRDS ROYD,
BRIGHOUSE, TEL.: 0484 712264
WEST YORKSHIRE. TELEX : 517478
HD6 1NB FAX.: 0484 722160

ALL AMENDMENTS WILL BE CHECKED AT DENFORDS THEN INCORPORATED INTO THE NEXT REPRINT OF THE TRIAC FANUC INSTRUCTION MANUAL.

**PART OF MANUAL REQUIRING AMENDMENT:**

**SECTION:.....PAGE:.....PARA:.....FIG:.....**

**AMENDMENT TO BE MADE / ANY OTHER COMMENT:**

**.....SIGNATURE.....POSITION IN COMPANY.**

This page is intentionally blank.

# APPENDIX

This page is intentionally blank.

# APPENDIX 1 GLOSSARY OF TERMS

**ARC**

A portion of a circle

**ARC CENTRE OFFSET**

The incremental distance between the programs cutter path and the arc centre in the axes directions. The X,Y and Z values are written as the I,J and K amounts without a directional sign.

**ASCII**

American Standard for Informatioin Interchange.

**AUTOMATIC CYCLE**

Is a mode of control operation that continously runs the tape or stored program until a program stop or end of program word is read.

**AUXILIARY FUNCTION**

The function of a machine other than the co-ordinate commands. F00, S100, T0100, M08.

**AXIS**

X axis: Line perpendicular to the spindle centre line (absolute)
U axis: Line perpendicular to the spindle centre line (incremental)
Z axis: Line parallel with the spindle centre line (absolute).
W axis: Line parallel with the spindle centre line (incremental).
Y axis: Line perpendicular to both the X and Y axes (absolute).
V axis: Line perpendicular to both the U and W axes (incremental).

**BINARY**

A system for describing numbers using only two digits.

**BINARY CODED DECIMAL**

A system of represented numbers comprised of a combination of four binary bits running accross the tape; letters are also described in this way.

**BIT**

The smallest programmable unit (i.e 1 or 0) in machine code.
8 Bits = 1 Byte

**BLOCK**

A word or words that collectively provide sufficient information for a complete operation. The block is separated from other blocks by an end of block character.

this page is intentionally left blank

**BLOCK DELETE**

Is a feature which provides means for skipping blocks (which have been preceded by a slash/code) at the discretion of the operator.

**BUFFER STORAGE**

Is an information storing area which is utilised to transfer the stored data to active storage almost instantaneously.

**CANNED CYCLE**

Is an automatic motion cycle which is held in buffer storage for the duration of the amount of cycle repeats programmed.

**CHARACTER**

A number, letter or symbol as punched accross the tape and constituting part of the row.

**CIRCULAR INTERPOLATION**

A means of programming an arc by providing a few basic statements.

**CLOSED LOOP**

A system in which the result of the output is measured and fed back for comparison with the input.

**CNC**

Computer numerical control See SOFTWIRED N.C

**COMMAND**

A signal or group of signals initiating one step in the execution of a program.

**CONTOURING (CONTINUOUS PATH)**

Co-ordinated simultaneous motion of two or more axes.

**CORNERING**

The effect at the machine due to the difference of electrical commands and slide positioning whilst commanding an abrupt cahnge of direction.

**C.S.S.**

Constant Surface Speed. A feature in a control system which tracks the point of a cutting tool so that the spindle can be constantly monitored to give the correct peripheral speed condition.

**C.P.U.**

Central Processing Unit.

**DECODE**

The traslation from tape coded language to control recognisable language.

**DRY RUN**

The use of this function enables the operator to run through the program replacing the programmed feed rates with the maximum jog feed, to test run the program and check tool clearance.

**DWELL**

A programmed time delay.

**EDIT**

Having put the program into memory (via the tape reader), the machine can then be operated from memory and suspect block of data can then be displayed for examination.

The faulty block can then be erased or correct information put in its place by means of MDI. The batch can then be run off entirely from the memory.

The addition of a tape punch enables correct tape to be produced automatically from edited and stored data.

**EIA**

Electrical Industries association has been responsible for setting many N.C standards, one being tape coding infoamtion.

**ENCODE**

The translation from control recognisable language to tape coded alnguage.

**ENCODER**

Device for comparing the position with the control input instruction.

**END OF BLOCK**

A character punched on a N.C. tape which denotes the end of a block of data.

**EXECUTIVE**

In CNC systems the executive software determines the manner in which the program data is processed.

**FEED HOLD**

At any stage in the cycle, the operator can stop the machine movements using the cycle stop key.

**FEEDRATE**

Is commanded in the N.C program by the F word and used by the control to drive the axes. The rate of feed may be programmed as FEED/MIN or FEED/REV.

**FEEDRATE**

Is an interrupt used to hold axes motion

**FEEDRATE OVERRIDE**

A manual function can be used by the operator to override the programmed feed rate within certain limits.

**FOLLOWING ERROR**

The lag distance between the actual machine position and the command position.

**FORMAT**

The arrangement and overall pattern in which the input data is organised (formatted).

**G CODE**

A preparatory code in the program which determines the control mode.

**HARDWARE**

The physical components of a control system or computer.

**HARDWIRED**

A numerical control sstem which is dedicated to driving a machine through committed circuit connections, and requiring the input data to be in a fixed format.

**I REGISTER**

A value which when programmed is used by the control as an X axis arc centre offset.

**INCREMENTAL PROGRAMMING**

A method of programming in which the motion statements relate from the previous programmed positon. the signs which accompany the departure commands in this system are directional in meaning.

**INPUT**

The transfer of external information into the control system.

**INTERGRATED CIRCUIT**

A complete circuit constructed within or on the surface of a silicon chip.

**INTERFACE**

The medium through which the control or computer directs the machine tool.

**INTERPOLATION**

The method used by the control system to achieve a series of approximations via. straight lines to enable an acceptable execution for tapers or arcs.

**ISO**

International Standards Organisation.

**K REGISTER**

A value which when programmed is used by the control as a Z axis arc centre offset or as a velocity rate for feed and threading.

**LEADER**

A portion of tape which precedes and follows the coded section to allow for handling or loading onto reels.

**LOOP**

A tape which is spliced together and not loaded to the reels.

**M CODE (Miscellaneous Function)**

The M words are used by the machine tool builder to indicate certain auxiliary functions such as coolant on, turret index, speed range etc.

**MANUAL DATA INPUT (MDI)**

A means of inserting data into the control system manually. The data entered is identical to that entered from a control tape or stored program.

**MANUSCRIPT**

A form used by the programmer for listing the detailed instructions which constitute a part program. These can be transcribed directly by the tape preparation device.

**MEMORY**

The storage capacity of a system to retain a part program or Programs.

**MODAL**

Information that is entered into the control and retained until changed.

**N WORD**

A word comprising a unique sequence number which is used to identify a complete block of information.

**OCTAL NUMBERS**

A numbering system which operates to a base of 8. Octal is more readily convertable to the decimal system than is the binary system.

**OPEN LOOP**

A system in which the output cannot be measured and fedback for comarison with the input.

**OPTIONAL STOP**

A miscellaneous command which is given by the programmer to stop the spindle and feed at a specific point in the program at the discretion of the operator.

**PARITY CHECK**

An automatic method of checking tape rows to establish that an odd or even parity system has been consistently punched. Should this check fail, the reader will stop to indicate an error in the preparation equipment or the reader itself.

**PROGRAM STOP**

A miscellaneous command which is given by the programmer to stop the spindle and feed at a specific point in the program.

**RESOLVER**

A device for comparing the position with the control input instruction.

**RECTANGULAR CO-ORDINATES**

A component graphically shown as three perpendicular axes (X,Y and Z) along which any point can be described in terms of distance and direction from any other point. The part program is written from this source.

**RESOLUTION**

The smallest increment of distance that will be developed by the control system in order to command machine motion.

**ROW**

A row of holes perpendicular to the tape edge which contain the character.

**S ADDRESS**

A word used in commanding the spindle speed.

**SEQUENCE NUMBER**

See N Word.

**SOFTWIRED N.C.**

A control system that has been pre-programmed with software to enable it to drive a machine tool.

**SUB-ROUTINE**

A stored sub program which is called from the main program.

**STEP-BY-STEP OPERATION**

An extension of MDI permits the machine to operate block-by-block, to permit the checking of each stage of the job, if required.

**TAB**

Tab characters may be used to space out the words in a program manuscript in columns if required.

**TOOL OFFSET**

A feature that allows the operator to make tool adjustments to compensate for the difference between the actual and the programmed setting dimensions.

**TRACK**

A path of holes parallel with the tape along which information may be stored by the presence or absence of holes.

**U WORD**

The U Word is used to command motion perpendicular to the spindle centre line (incremental).

**W WORD**

The W Word is used to command motion parallel to the spindle centre line (incremental).

**V WORD**

The V Word is used to command motion perpendicular to both U and W axes.

**WORD**

A combination of the letter address and digits.

| ADDRESS | DIGITS | WORD |
|---|---|---|
| X | +002.0000 | X+002.0000 |
| F | 1.9990 | F1.9990 |

**X WORD**

On 2 and 4 axis machines X Word is used to command motion perpendicular to the spindle centre line (absolute).

**Z WORD**

On 2 and 4 axis machines Z Word is used to command motion parrallel to the spindle centre line (absolute).

**Y WORD**

On 2 and 4 axis machines Y Word is used to command motion perpendicular to both X and Z (absolute)

**ZERO**

In absolute programming, zero is the point from which all other dimensions are referenced.

## APPENDIX 2 LIST OF ABBREVIATIONS

| | |
|---|---|
| G.B.T. NO. | = Group Block Terminal |
| EXECUTE OR MDI | = Manual Data Input |
| MSD | = Machine Set Up Data (or Parameters) |
| RAM | = Random Access Memory |
| ROM | = Read Only Memory |
| PROM | = Programmable Read Only Memory |
| EPROM | = Erasable Programmable Read Only Memory |
| PCI | = Program Controlled Interface (i.e. Ladder Diagram in Software). |
| PWM | = Pulse Width Modulated G.E Servo Drive or Hi-Ak Drives. |
| SCR | = Silicon Controlled Rectifier (i.e Thyristor Drives used on Spindle) |
| TRIAC | = Solid State Relay |
| I.P. | = Initial Position or Zero Ref. Point |
| BCD | = Binary Coded Decimal |
| LED | = Light Emitting Diodes |
| IC | = Input Conditioner or Input Filter |
| OD | = Output Driver |
| +VE | = Positive |
| -VE | = Negative |

| | |
|---|---|
| DVM | = Digital Voltmeter |
| POT | = Potentiometer or Variable Resistor |
| LSI | = Large Scale Integration |
| PCL | = Programmable Controller Language |
| MCL | = Machine Control Logic |

This page is intentionally blank.

# APPENDIX 3

## TRIGONOMETRY AND GEOMETRY

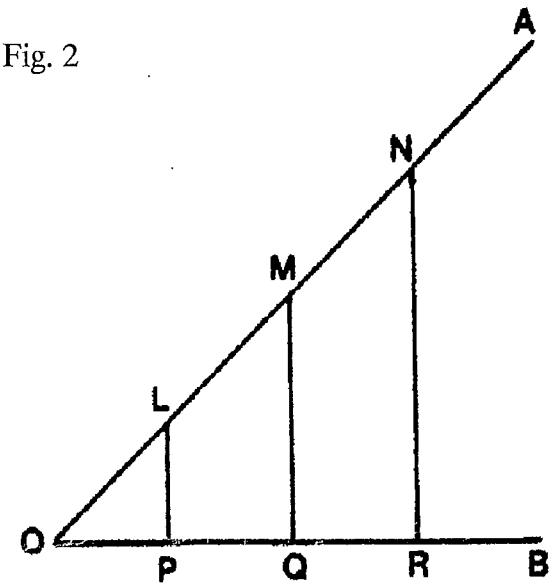
- TRIGONOMETRY
- GEOMETRY

### a) Tangent

In Fig. 2 AOB is an angle of say 30 Degrees. The triangles LPO, MQO, NRO are similar because they all share the common angle of 30 Degrees. Therefore a common ratio for the three triangles can be shown as:

$$\frac{LP}{OP} = \frac{MQ}{OQ} = \frac{NR}{OR}$$

Fig. 2




If these sides were known, then dividing LP by OP and MQ by OQ and so on will produce the same common ratio.

If side LP is 5 mm and OP is 8.66 mm then LP/OP produces a ratio of 0.5774 or tangent value for 30 Degrees, a set of tangent tables are compiled in this way.

In the triangle LOP, side LP is opposite to the angle AOB, side OP is adjacent to the angle LOP, so to calculate a tangent ratio for the triangle LOP may be shown as:

$$\text{Tangent value} = \frac{\text{Opposite Side}}{\text{Adjacent side}}$$

### b) Sine

In Fig. 2 the triangles LPO, MQO, NRO are similar because they all share the same angle (90 Degrees) at P, Q and R. Therefore a common ratio for three triangles can be shown as:

$$\frac{PL}{OL} = \frac{QM}{OM} = \frac{RN}{ON}$$

Therefore the ratio of PL/OL depends on the angle value of AOB only and dividing these two values will always produce the same common ratio.

If side PL is 5 mm and OL is 10 mm then PL/OL produces a ratio of .5000 or sine value for 30 Degrees, a set of sine tables are compiled in this way. In the triangle LOP, side PL is opposite to the angle AOB, side OL is the hypotenuse of the triangle LOP, so to calculate a sine ratio for the triangle LOP may be shown as:

$$\text{Sine Value} = \frac{\text{Opposite Side}}{\text{Hypotenuse}}$$

### C) Cosine

It is also true that in Fig. 2 the ratios:

$$\frac{OP}{OL} = \frac{OQ}{OM} = \frac{OR}{ON}$$

Therefore the ratio of OP/OL depends on the angle value of AOB only and dividing these two values will always produce the same common ratio.

If side OP is 8.66 mm and OL is lO mm then OP/OL produces a ratio of .866 or cosine value for 30 Degrees, a set of cosine tables are compiled in this way. In the triangle LOP, side OP is adjacent to the angle AOB, side OL is the hypotenuse of the triangle LOP, so to calculate a cosine ratio for the triangle LOP may be shown as:

$$\text{Cosine Value} = \frac{\text{Adjacent Side}}{\text{Hypotenuse}}$$

## SUMMARY

### Tangent

TAN = Opp. Side/Adj. Side

also Opp. Side = Tan x Adj. Side

Adj. Side = Opp. Side/Tan

### Sine

SIN = Opp. Side/Hypot.

also Opp. Side = Sin x Hypot.

Hypot. = Opp. Side/Sin

### Cosine

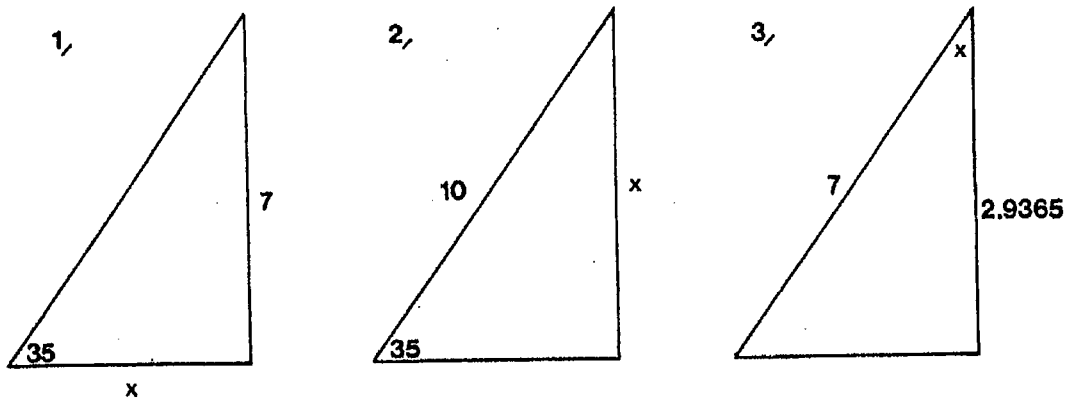
COS = Adj. Side/Hypot.

also Adj. Side = Cos x Hypot.

Hypot. = Adj. Side/Cos

## EXAMPLES:

Determine X values




### 1) Solution

Find adjacent side knowing opposite.

$$\text{Adj} = \frac{\text{Opp}}{\text{Tan 35 Degrees}}$$

$$A = \frac{7}{0.7002}$$

$$A = 9.9971 = X$$

### 2) Solution

$0 = \text{Sin 35 Degree x Hypot}$

$0 = .5736 \times 10$

$0 = 5.7360 = X$

### 3) Solution

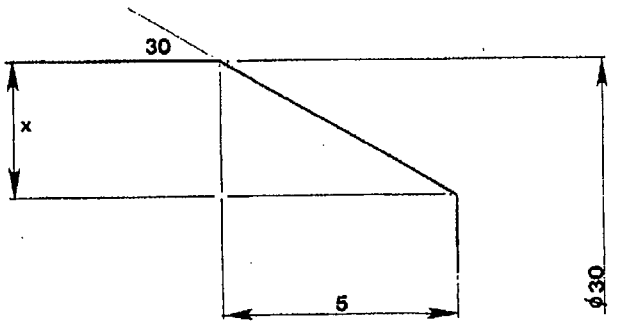
Find angle knowing adjacent side and hypotenuse.

$$\text{Cos} = \frac{\text{Adj.}}{\text{Hypot.}}$$

$$\text{Cos} = \frac{2.9365}{7} = 0.4195 \text{ ( as cosine ratio )}$$

0.4195 from cosine tables = 65 deg 12' = X

4/



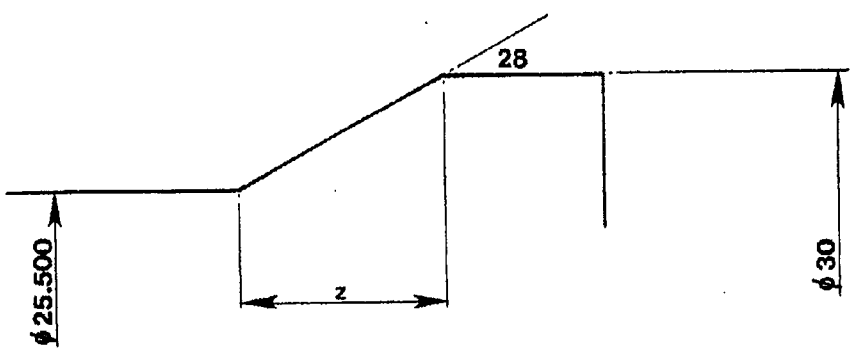

**4) Solution**

Find X distance on component chamfer

X = Tan 30 x 5

X = 2.887 mm

5/




**5) Solution**

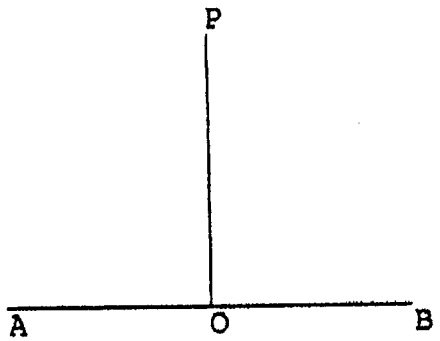
Determine Z distance on thread relief.

$$Z = \frac{2.250}{\text{Tan } 28}$$

Z = 4.232 mm

**DEFINITION Right Angle**

If a straight line OP meets another straight line AOB as shown in Fig. 1, so as to make the adjacent angles POA, POB equal, each angle is called a right angle.




**Acute Angle.**

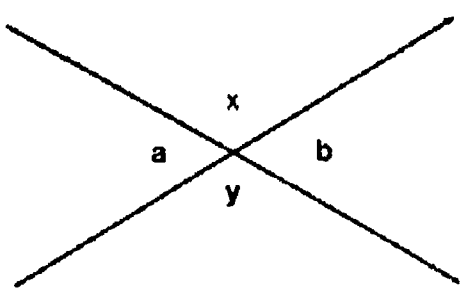
An acute angle is an angle less than a right angle.

**Obtuse Angle.**

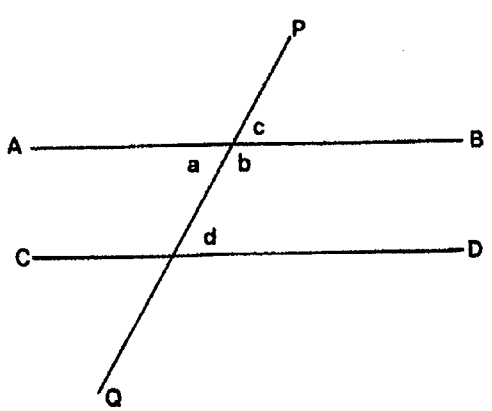
An obtuse angle is an angle greater than a right angle but less than two right angles.

**Reflex Angle.**

A reflex angle is an angle between two and four right angles. Any two angles whose sum is two right angles are called supplementary. Two angles whose sum is one right angle are called complementary.




If two straight lines intersect, the vertically opposite angles are equal. In fig. 2, where two lines intersect, a is equal to b and x is equal to y.



If two straight lines in the same plane do not meet they are called parallel lines. In fig. 3 the line is called a transversal line, angles a and d are called alternative angles; angles c and d are called corresponding angles; angles b and d are called interior angles.

Therefore if a transversal line PQ cuts two parallel lines, angles a and d are equal, angles c and d are equal.

Angle properties of the circle.

**Circle 1**

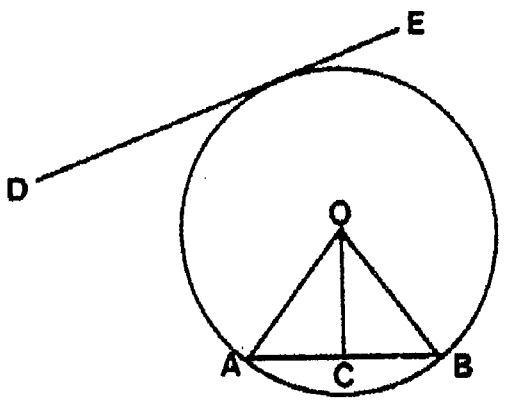



**fig 4**

In figure 4, line DE is tangent to circle 1, because it only touches the circle at one point, line AB is a chord of circle 1, if OC is perpendicular to the chord AB from the centre of circle 1 then AC is equal to CB.

**Circle 2**

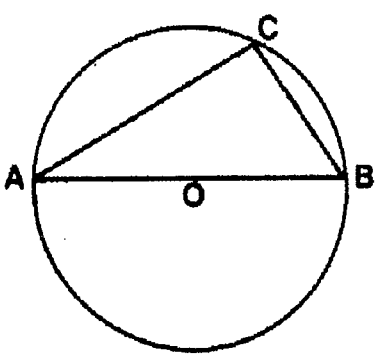



**fig 5**

The angle in a semi circle is a right angle. In fig 5 line AB passes through the centre of circle 2 at O. Any point chosen within the semi circle at C will produce a right angle ACB.

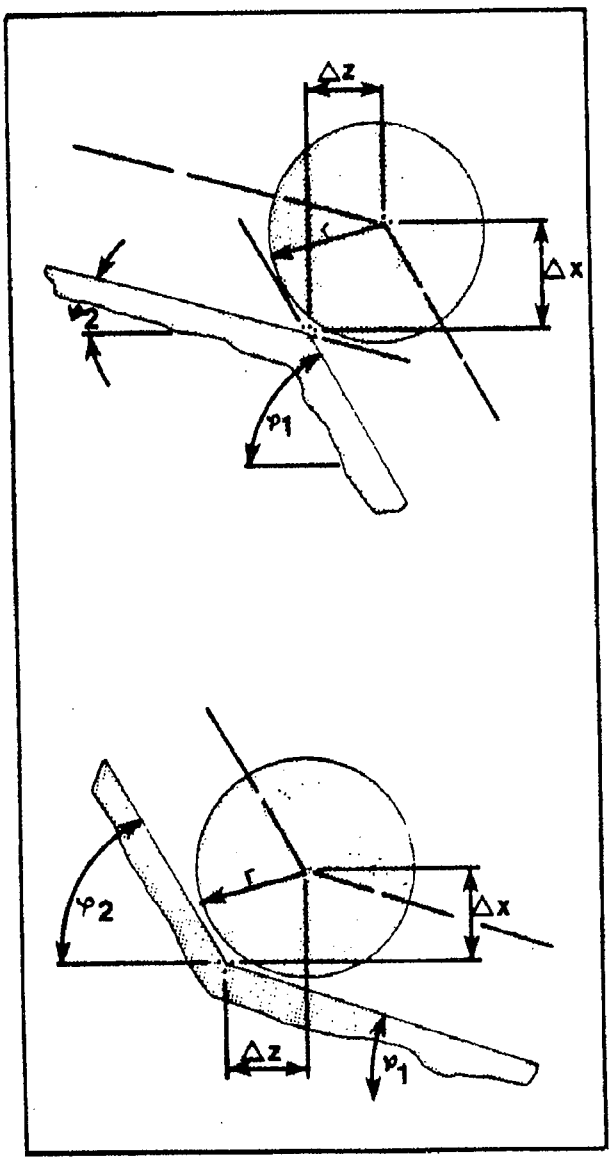
# APPENDIX 4

## TABLES OF INFORMATION

- USEFUL FORMULAE

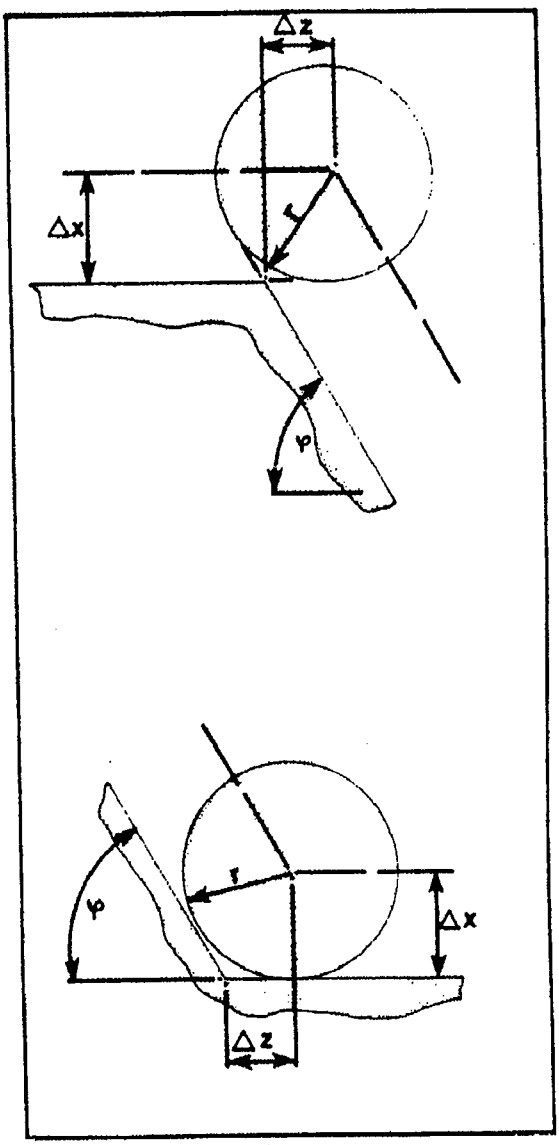
This page is intentionally blank.

## 5.8.1 USEFUL FORMULAE FOR DETERMINING CONTOUR CHANGE POINTS



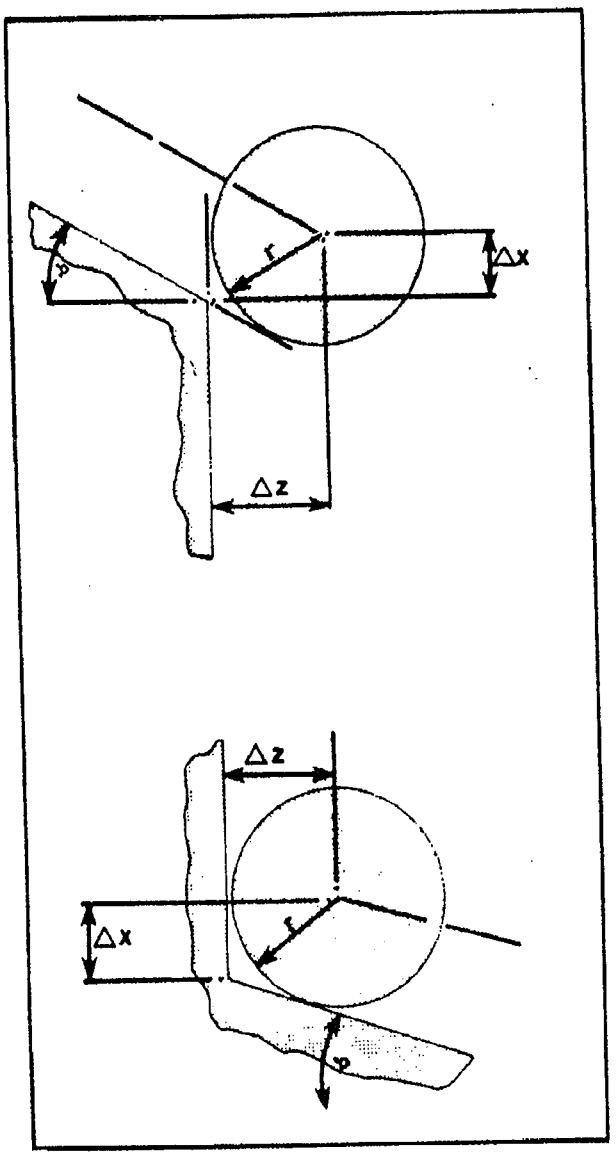

$$\Delta x = r.\ \frac{\cos\frac{\alpha_2 + \alpha_1}{2}}{\cos\frac{\alpha_2 - \alpha_1}{2}}$$

$$\Delta z = r.\ \frac{\sin\frac{\alpha_2 + \alpha_1}{2}}{\cos\frac{\alpha_2 - \alpha_1}{2}}$$



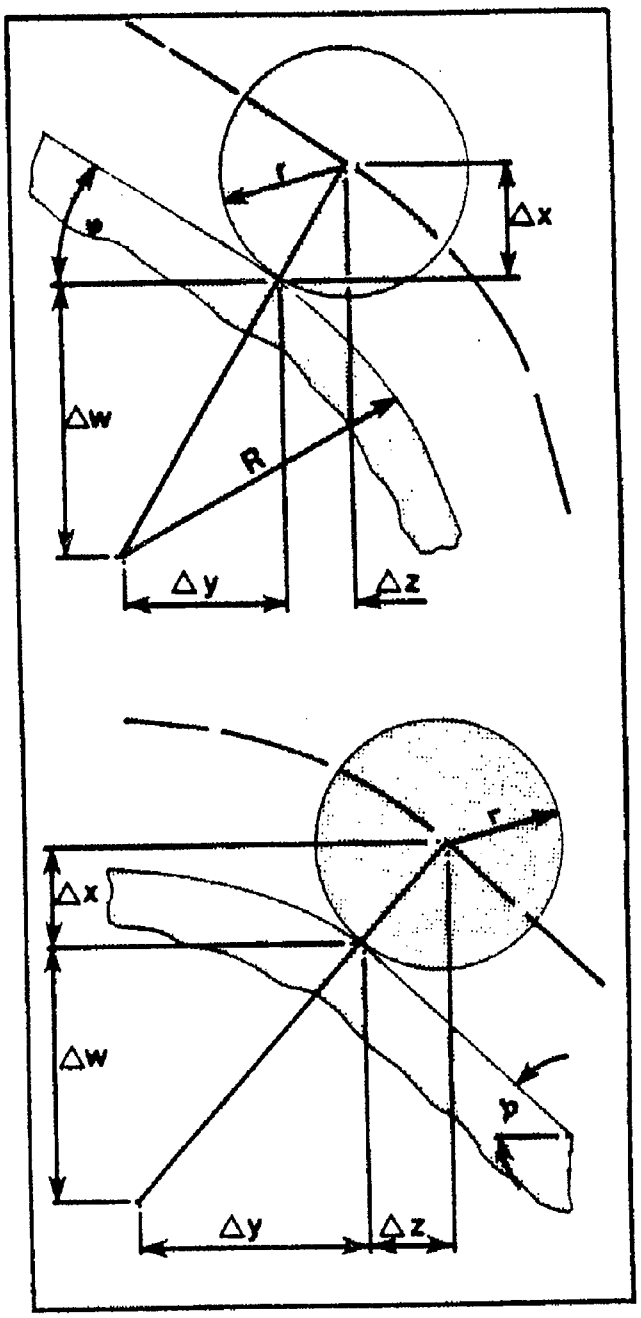

$$\Delta x = r$$

$$\Delta z = r.\tan\frac{\alpha}{2}$$




$$\Delta x = r. \tan\left(45 - \frac{\alpha}{2}\right)$$

$$\Delta z = r$$



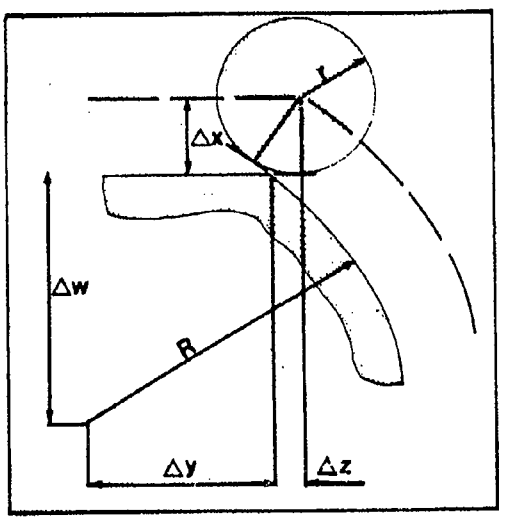

$\Delta w = R \cos \alpha$

$\Delta x = r \cos \alpha$

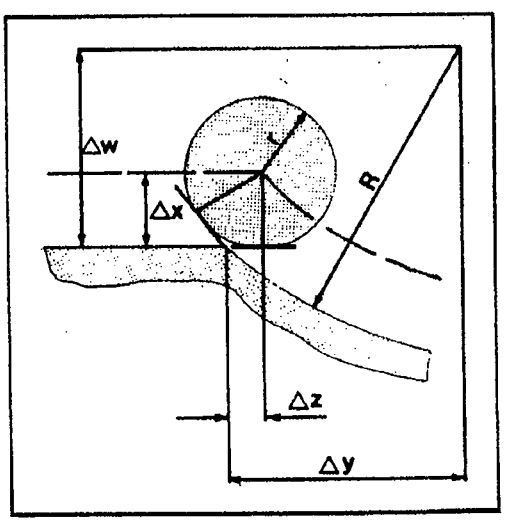
$\Delta y = R \sin \alpha$

$\Delta z = r \sin \alpha$

$\Delta x = r$




$\Delta z = \sqrt{[(R+r)^2-(\Delta w+r)^2]}-\Delta y$

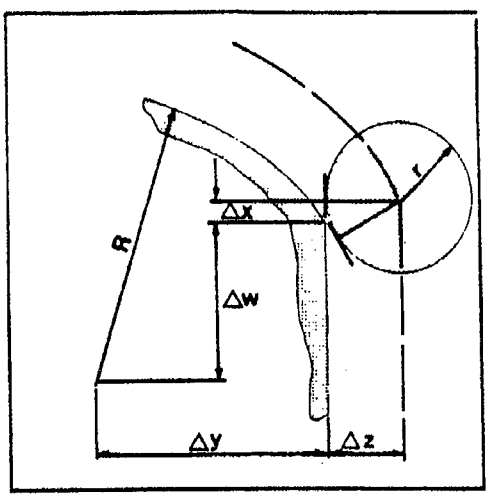


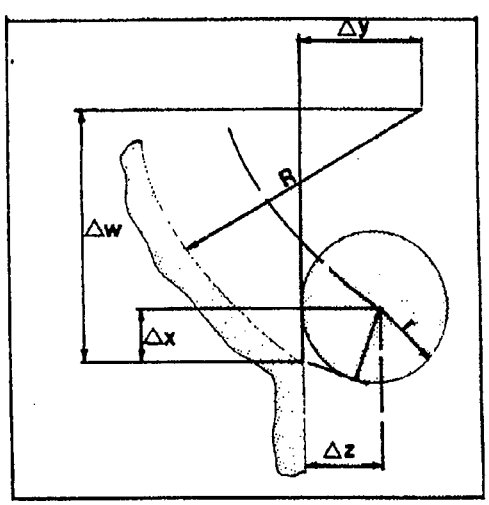

$\Delta x = r$

$\Delta z = \Delta y - \sqrt{[(R-r)^2-(\Delta w-r)^2]}$

$$\Delta y = r$$

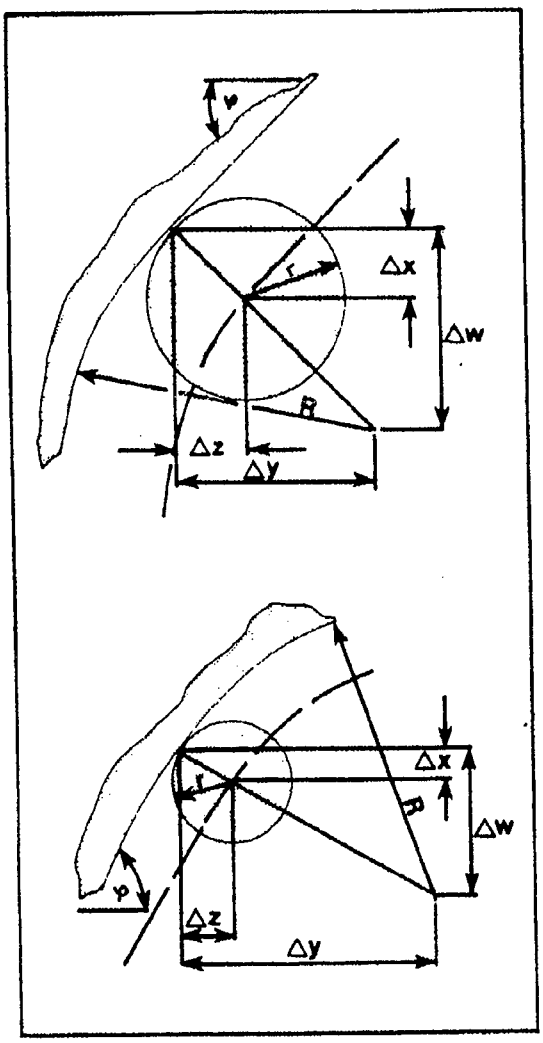
$$\Delta x = \sqrt{[(R+r)^2-(\Delta y+r)^2]}-\Delta w$$




$$\Delta z = r$$

$$\Delta x = \Delta w - \sqrt{[(R-r)^2-(\Delta y-r)^2]}$$




$\Delta w = R \cos \alpha$

$\Delta x = r \cos \alpha$

$\Delta y = R \sin \alpha$

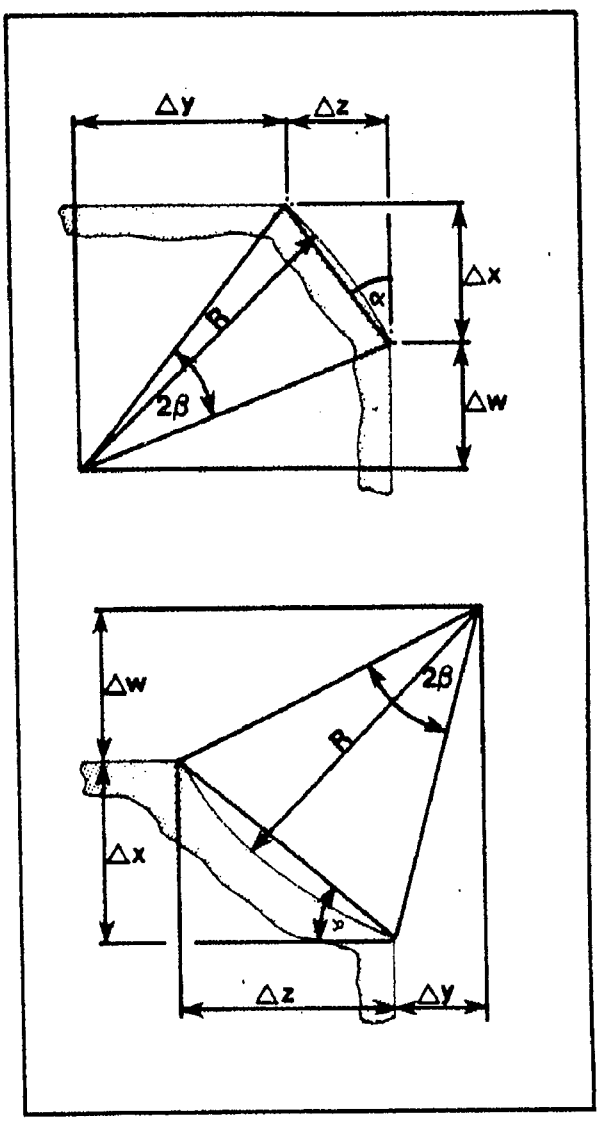
$\Delta z = r \sin \alpha$

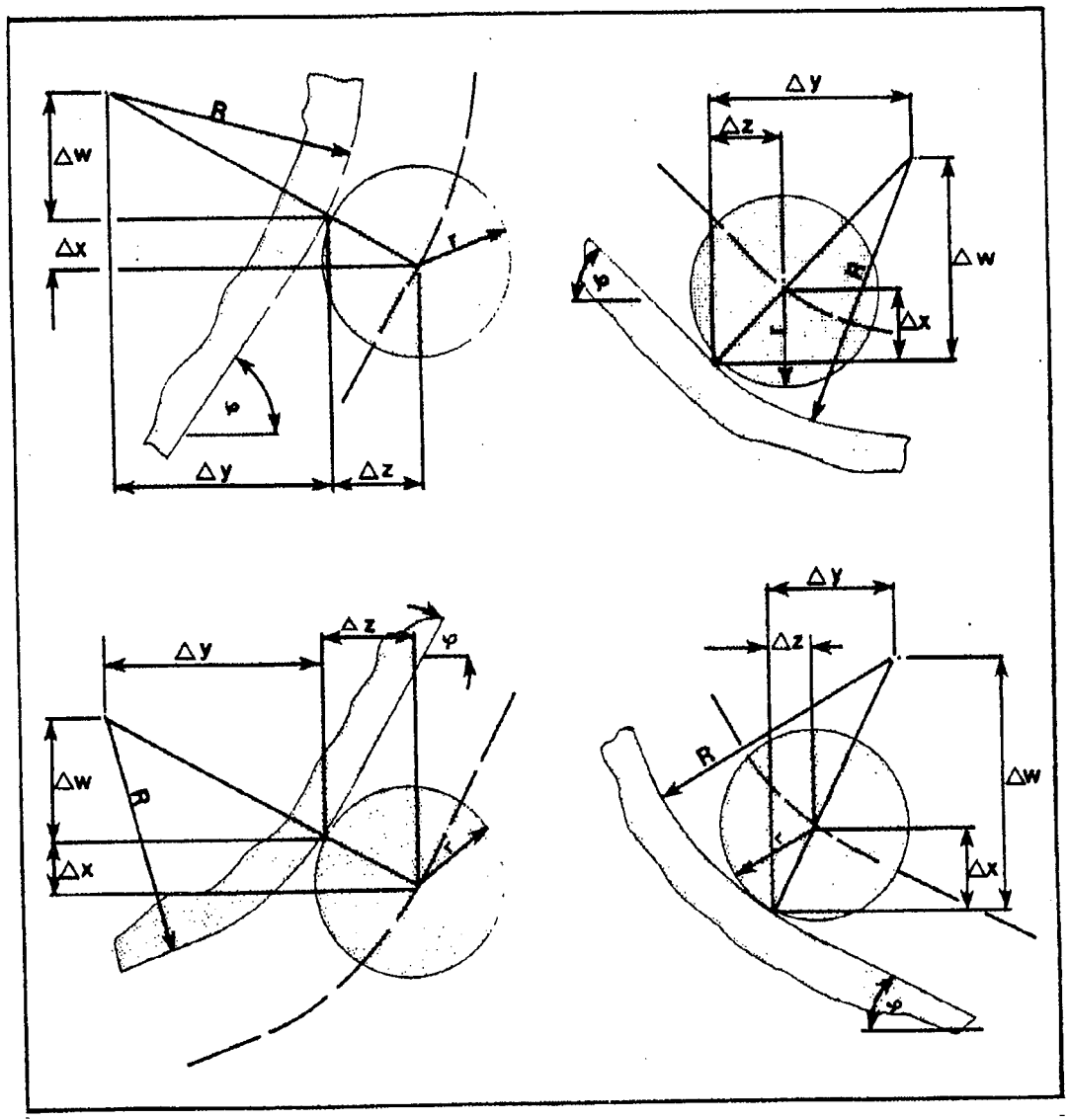
$$\Delta y = R \sin ( 90 - \alpha - \beta )$$

$$\Delta w = R \sin ( \alpha - \beta )$$

$$\tan \alpha = \frac{\Delta z}{\Delta x}$$

$$\sin \beta = \frac{\Delta z}{2R \sin \alpha}$$



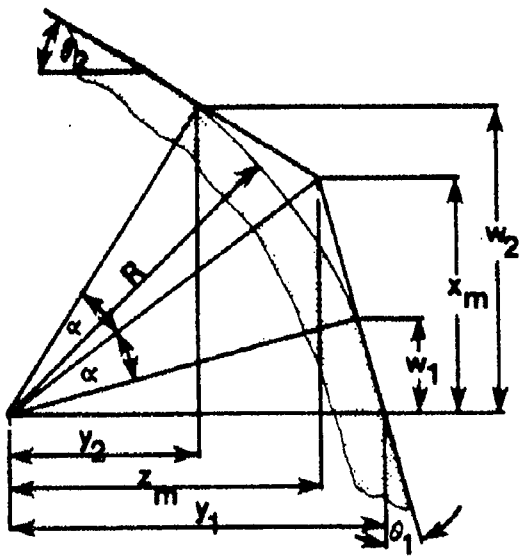



$$\Delta w = R \cos \alpha$$

$$\Delta x = r \cos \alpha$$

$$\Delta y = R \sin \alpha$$

$$\Delta z = r \sin \alpha$$




$$x_m = R\frac{\sin(\alpha + \theta_1)}{\cos\alpha}$$

$$z_m = R\frac{\cos(\alpha + \theta_1)}{\cos\alpha}$$

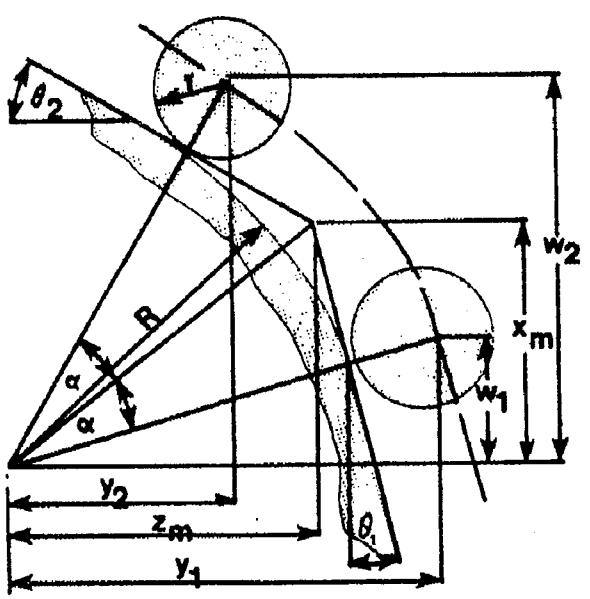
$$\alpha = \frac{90 - \theta_1 - \theta_2}{2}$$

$$w_1 = R \sin\theta_1$$

$$w_2 = R \cos\theta_2$$

$$y_1 = R \cos\theta_1$$

$$y_2 = R \sin\theta_2$$




$$x_m = R\frac{\sin(\alpha + \theta_1)}{\cos\alpha}$$

$$z_m = R\frac{\cos(\alpha + \theta_1)}{\cos\alpha}$$

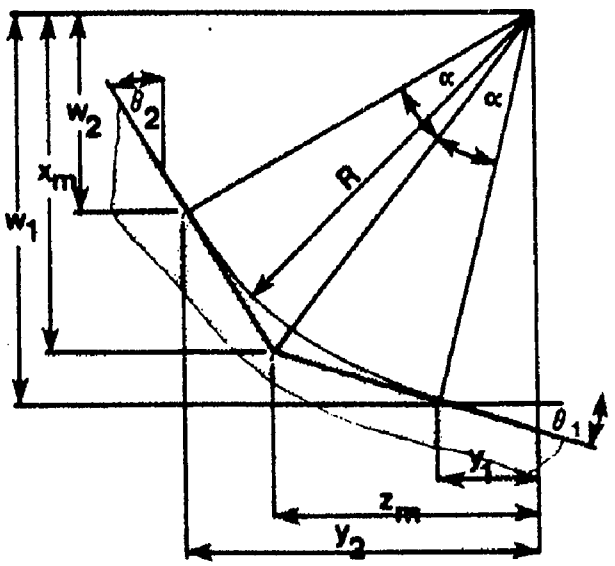
$$\alpha = \frac{90 - \theta_1 - \theta_2}{2}$$

$$w_1 = (R + r)\sin\theta_1$$

$$w_2 = (R + r)\cos\theta_2$$

$$y_1 = (R + r)\cos\theta_1$$

$$y_2 = (R + r)\sin\theta_2$$




$$x_m = R \frac{\cos(\alpha + \theta_1)}{\cos \alpha}$$

$$z_m = R \frac{\sin(\alpha + \theta_1)}{\cos \alpha}$$

$$\alpha = \frac{90 - \theta_1 - \theta_2}{2}$$

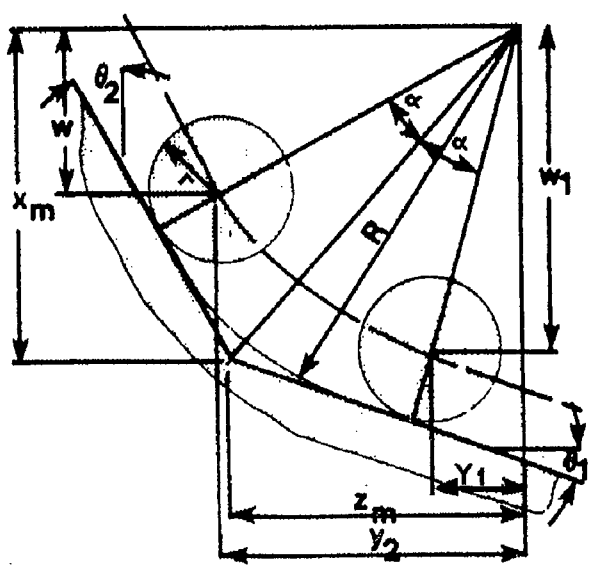
$$w_1 = R \cos \theta_1$$

$$w_2 = R \sin \theta_2$$

$$y_1 = R \sin \theta_1$$

$$y_2 = R \cos \theta_2$$

$$x_m = R \frac{\cos(\alpha + \theta_1)}{\cos \alpha}$$




$$z_m = R \frac{\sin(\alpha + \theta_1)}{\cos \alpha}$$

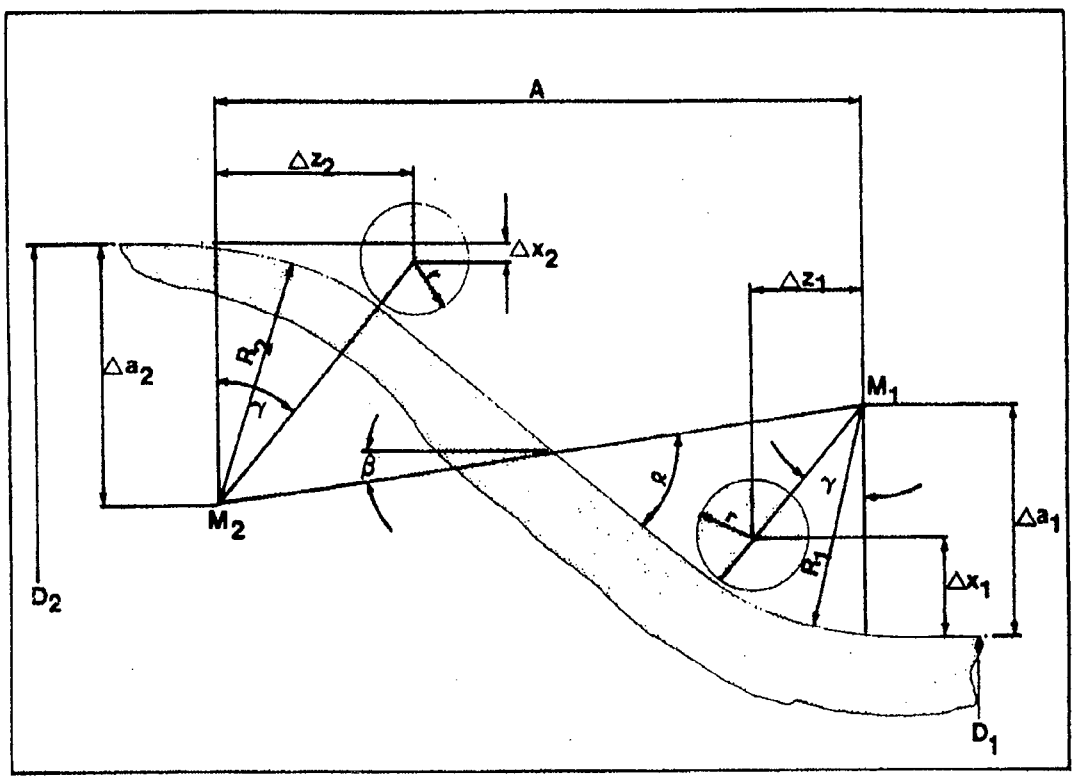
$$\alpha = \frac{90 - \theta_1 - \theta_2}{2}$$

$$w_1 = (R - r) \cos \theta_1$$

$$w_2 = (R - r) \sin \theta_2$$

$$y_1 = (R - r) \sin \theta_1$$

$$y_2 = (R - r) \cos \theta_2$$




$$\Delta z_1 = \sin\gamma\,(R_1 - r) \qquad \Delta x_1 = R_1 - \cos\gamma\,(R_1 - r)$$

$$\Delta z_2 = \sin\gamma\,(R_2 + r) \qquad \Delta x_2 = R_2 - \cos\gamma\,(R_2 + r)$$

$$\tan\beta = \frac{R_1 + R_2 - \frac{D_2 - D_1}{2}}{A}$$

$$\gamma = \alpha - \beta$$

$$\sin\alpha = \frac{R_1 + R_2}{\sqrt{\left[A^2 + \left(R_1 + R_2 - \frac{D_2 - D_1}{2}\right)^2\right]}}$$

# INDEX

This page is left intentionally blank

# INDEX

## I



## L



## M



## N



## O



## P



## R

## S



## T



## W