OCCASION

This publication has been made available to the public on the occasion of the 50th anniversary of the United Nations Industrial Development Organisation.

DISCLAIMER

This document has been produced without formal United Nations editing. The designations employed and the presentation of the material in this document do not imply the expression of any opinion whatsoever on the part of the Secretariat of the United Nations Industrial Development Organization (UNIDO) concerning the legal status of any country, territory, city or area or of its authorities, or concerning the delimitation of its frontiers or boundaries, or its economic system or degree of development. Designations such as "developed", "industrialized" and "developing" are intended for statistical convenience and do not necessarily express a judgment about the stage reached by a particular country or area in the development process. Mention of firm names or commercial products does not constitute an endorsement by UNIDO.

FAIR USE POLICY

Any part of this publication may be quoted and referenced for educational and research purposes without additional permission from UNIDO. However, those who make use of quoting and referencing this publication are requested to follow the Fair Use Policy of giving due credit to UNIDO.

CONTACT

Please contact publications@unido.org for further information concerning UNIDO publications.

For more information about UNIDO, please visit us at www.unido.org

COMPUTER AIDED DESIGN AND COMPUTER AIDED MANUFACTURING CENTRE

DP/SRL/86/014

SRI LANKA

Technical report: NC Part Programming[*]

Prepared for the Government of Sri Lanka
by the United Nations Industrial Development Organization
acting as executing agency for the United Nations Development Programme

Based on the work of Tamus Markus
Expert in CAD/CAM-NCPP

Backstopping officer: P. Prijapratama, Engineering Industries Branch

United Nations Industrial Development Organization
Vienna

3a/26

-----

[*] This document has not been edited.

ABSTRACT

The mission has been undertaken under the project 'The Establishment of a Computer Aided Design and Computer Aided Manufacturing Centre' DP/SRL/86/014

The immediate objective of the mission was to assist the University of Moratuwa, Department of Mechanical Engineering in the development of training capabilities in NC Part Programming.

The mission lasted one month from June 12th to July 11th 1989.

TABLE OF CONTENTS

FINDINGS AND RECOMMENDATIONS

1. Having discussions with the faculty members and considering the industrial activities of the country the basic fields can be identified for the economical and successful utilisation of CAM. Some of these fields are:

   - agricultural machinery and equipment
   - plastic industry
   - rubber industry
   - and basicallly at all sectors where spare parts are manufactured (maintenance depots of the transportation system, of paper industry machinery, of machinery for oil industry etc).

2. Based on experience and recent discussions and mee tings the following organisations can be especially interested in the near future work of the CAD/CAM Centre.

   Central Engineering Consultancy Bureau
   Ceylon Institute of Scientific and Industrial Research
   Ceylon Petroleum Corporation
   Ceylon State Hardware Corporation
   Ceylon Steel Corporation
   Colombo Commercial Co. Ltd.
   Colombo Dockyard Ltd.
   Engineering Consultants Ltd.
   Hemas Marketing Ltd.
   National Engineering Research and Development Centre
   Sri Lanka Central Transport Board
   Sri Lanka Tyre Corporation
   State Engineering Corporation of Sri Lanka

3. The utilisation of new technologies highly depends on the knowledge and attitude of the engineers and executives of the industry. The present capacity of the CAD/CAM Centre already can serve as a starting basis for a series training courses and demonstrations for the graduated engineers actively engaged in the field. This activity should be a nation-wide mission, therefore the support and the participation of the Institution of Engineers Sri Lanka is desirable.

4. The level of cooperation and the number of contracts between the University and the industrial sector can be raised as soon as the CAD/CAM facilities of the University be widely publicised with the training courses. Even the most potential users are likely to require a series of production tests under contract before deciding the installation of the CNC or CAM technology.

5. During the period of the mission the Computer Centre of the University was installing the latest shipment of computers including 16 AT compatibles, micro VAX 3600-s etc. The hardware capacity of the University makes possible the receiving of both more sophisticated CAE/CAD/CAM software packages and large student groups. The CAD/CAM Centre has to have on its premises the full processing power of the systems used therefore the developing of the local area network is strongly recommended.

The lack of a computer aided part programing system was severly felt. The purchase of a CAD/CAM system in the nearest future is desirable. The more sophisticated systems (eg EUCLID [F] or EUKLID [CH] ANVILL5000 [US]etc) are modular structured. The consequence is that the purchase of one of these systems needs careful analysis of the present needs (just basic CAD/CAM, MOULDFLOW module or finite element module is also needed etc.) and the future needs of system integration (standardised interfaces exist or not etc.) and the budget limitations also should be considered.

These circumstances and the variable need of industry to be fulfilled by the CAD/CAM Centre, and the fact that the majority of parts are not designed but only remanufactured suggest that besides the purchase of a sophisticated CAD/CAM system the acquiring of low cost but very effective software packages for CAPP (Computer Aided Process Planning) and CAM is also highly recommended (MICLASS, GTIPROG, TUSY etc).

6. The present undergraduate courses in Production Engineering have the very basics of CAM. Considerable increase of the topics in these courses is beyond the realities. Slight changes should be done in favour of more CAM topics in the curriculum. Programming lab work and demonstrations on the TAKISAWA/FANUC CNC Manufacturing Centre for the students is recommended.

Dramatic change in the CAM educational work will be resulted if students for Master Degree will enroll in the Department. Considering the recent capacity and the UNIDO plans for the year 1990 the basic facilities are available to open M.Sc Courses in Mechanical Engineering (Production Engineering). Based on a properly prepared curriculam the materials and syllabi needed could be compiled by the Fall of 1990 to start M.Sc. studies. Some expert support for thye first acedemic year is needed to launch such a new programme.

7. Some text books are available at the University Library on CAM (see Annexe 1). More books are needed as soon as possible because very important parts of CAD/CAPP/CAM haven't been covered recently. See in Annexe 2 the list of recommended books and journals.

8. A list of basic international standards on NC programming was compiled (see Annexe 3). To get in contact with the Sri Lanka Standards Institution is recommended to obtain the ISO standards.

9. As part of the program a plan of study tours for the persons concerned with the project has been prepared and is due to be undertaken this year. It is suggested that the Department of Manufacture Engineering of Budapest Technical University be included in the program partly because of its long experience in the training of CAD/CAM specialists partly because of its standing as a national CAD/CAM demonstartion centre.

OBJECTIVE OF THE ACTIVITIES AND THE DUTIES

a) Objective

The objective of the mission was to assist the University of Moratuwa, Department of Mechanical Engineering in the development of training capabilities in NC Part Programmming.

b) Duties

1. To provide lectures on numerically controlled machine tools and programming and train local staff in NC Part Programming.

2. To elaborate a proper training programme for local staff and prepare in co-operation with the counterpart a detailed training programme for fellowsnip holders.

3. To assist in elaboration of training methodologies, syllabi and lecture notes for courses for students and industrial engineers.

DESCRIPTION OF ACTIVITIES

1. The topics of the lectures were discussed in detail.
   Since University has installed at the CAD/CAM Centre a
   TAKISAWA-FANUC CNC Manufacturing Centre the staff members
   have shown good progress in manual NC part programing
   techniques. As a consequence the conclusion was drawn
   that only the more sophisticated part of manual part
   programming should be dealt with and also emphasize com-
   puter aided part programming methods.

   The topics covered in the lectures are given in Annexe 4,
   the schedule is listed in Annexe 5.

2. On the afternoon sessions the staff members were engaged
   in CNC part programming on the CNC manufacturing centre,
   to get practice in the topics that were of immediate in-
   terest in the lectures. A part programme was elaborated
   for the workshop to demonstrate to the participants the
   flexibility of the CNC techniques.

3. A workshop was organised (see Annexe 6) for industry
   engineers and invited participants in NC part
   programming. Demonstration CNC programs, computer
   programs and lecture notes were prepared. As a con-
   sequence of the prevailing transport and communications
   problems the workshop had to be postponed to a future
   date.

4. To assist the educational work and to give deep under-
   standing and knowledge to the staff members in computer
   aided part programming an APT like programming systems
   with limited capabilities was specified both in input
   language set and in CLDATA records to be used.

   The staff members with great enthusiasm were involved in
   developing the processor, which produces CLDATA files
   for 2.5 D contouring works. After the basic principles
   of the post processor structure was discussed the staff
   members individually were able to continue and to finish
   the post processor for the TAKISAWA-FANUC CNC machining
   centre. More test runs were produced and this little sys-
   tem now can be used both in demonstration and education.

5. The extensive lecture notes elaborated (see Annexe 7) can
   be used both for education of undergraduates and field
   engineers.

ANNEX 1

Books now available at the University Library on CAPP/CAM technology:

CHAMBERS : Drafting and manual programming for numerical control (1980)

GIBBS : An introduction to NCN machining (1984)

HITOMI : Manufacturing systems engineering (1979)

KUSIAK : Flexible Manufacturing systems: methods and studies (1986)

MARTIN : Numerical Control of Machine Tools (1970)

ANNEX   2

Suggested books and journals

ENCARNACAO: CAD/CAM INTERFACES (Springer, 1986)

GRROVER-ZIMMMERS : CAD/CAM   (Prentice Hall 1985)

PUSTAY-SAVA: COMPUTER NUMERICAL CONTROL (Prentice Hall 1983)

REMBOLD-BLUME-DILLMAN: COMPUTER INTEGRATED MANUFACTURING
                       TECHNOLOGY AND SYSTEMS
                       (Springer 1988)

TEN HAGEN- CHAUDRY: DESIGN RULES FOR CIM SYSTEMS
                     (North Holland 1985)

WARNECKE   : TOWARDS THE FACTORY OF THE FUTURE
                       (Springer 1985)

WARNECKE   : FLEXIBLE MANUFACTURING SYSTEMS
                       (Springer 1986)

                    : 1988 /NC GUIDE BOOK
                       (Modern Machine Shop, Cincinatti, Ohio
                       1988)

COMPUTER AIDED DESIGN (Pergamon)
COMPUTERS IN INDUSTRY (North Holland)
COMPUTER INTEGRATED MANUFACTURING AND ROBOTICS (Pergamon)
MANUFACTURING SYSTEMS   (CIRP)

ANNEX 3

(ISO standards to be obtained)

ISO/DP 841	Industrial Automation Systems-Numerical Control of Machine Tools- Axis and motion nemenclature
ISO 3592	Numerical Control of Machines-NC processor output-Logical structure (and major words)
ISO 4342	Numerical Control of Machines-NC processor input-Part programming reference language
ISO 4343	Numerical Control of Machines-NC processor output-Minor El;ements of Zero-Type records (post-processor commands)
ISO 6983	On NC program codes
ISO/DIS 6132	Extended Format and Data Structures

CAD/CAM PROJECT   ANNNEX 4

Lecture programme on NC PART PROGRAMMING

Lecturer   :Dr.T.Markus (UNIDO Expert)

Venue   :Seminar Room, Department of Mechanical Engineering

Lecture No	Topics
Introductory	Basics of NC,CNC,DNC, applications, manual and computer aided part programming.
1.	Computer aided part programming. Processor/post-processor concept, sample part-program, level of automation.
2.	Computer aided part programming. Sample part-Program for contouring. The CLDATA structure, role and structure of post- processors.
3.	Computer aided part programming. Principles of 3D-5D manufacturing. Programming the manufacturing of complex surfaces. System structure of a 5D CAPP/CAM system, principles of system design philosophy.
4.	Advanced manual part programming. Address words, motion axis,special use of address codes. International standards.
5.	Advanced manual part programming. Tool coordinate system, tool compensation techniques for turning, milling.
6.	Advanced manual part programming. Subroutines, parameters and macros-1.
7.	Advanced manual part programming. Subroutines parameters and macros-2, extended format and data structure.
8.	Production planning for NC machines, documents used.
9.	More on computer aided part programming, future trends. Sculptured surface programming,the effects of control unit capability on post-processors, systems of high level of automation. Towards Computer Intergrated Manufacturing (CIM) systems.

ANNEX 5

CAD/CAM PROJECT (DP/SRL/86/014)
WORK SCHEDULE FOR NC PART PROGRAMMING EXPERT
Dr.T.MARKUS (UNIDO)

		Morning 9 am - 12 noon	Afternoon 1.30 - 3.45 pm
JUNE			
Monday	19th	Discussion with UNDP/UNIDO officials	
Tuesday	20th	Arrival at the Department. Preliminary discussion, finalisation of programme.	
Wednesday	21st	Introductory Lecture	Lecture 1
Thursday	22nd	Lecture 2	Demonstration
Friday	23rd	Lecture 3	Demonstration

		WEEKEND	
Saturday	24th	Preparation of lecture notes	
Sunday	25th		

Monday	26th	Lecture 4	Demonstration
Tuesday	27th	Lecture 5	Demonstration
Wednesday	28th	Lecture 6	Demonstration
Thursday	29th	Lecture 7	Demonstration
Friday	30th	Lecture 8	Demonstration

JULY

		WEEKEND	
Saturday	01st		
Sunday	02nd	Preparation of report to UNDP.	

Monday	03rd	Lecture 9	Demonstration
Tuesday	04th	Preparations for the Workshop for Industrialists.	
Wednesday	05th	Workshop for Industrialists - Day 1.	
Thursday	06th	Workshop for Industrialists - Day 2.	
Friday	07th	Drawing up training/fellowship programme for local staff. Finalisation of report to UNDP.	

		WEEKEND	
Saturday	08th		
Sunday	09th	Departure.	

Department of Mechanical Engineering
University of Moratuwa.

Workshop on NC Part Programming in CAD/CAM

Date: 05   06 July 1989

Objectives:

The series of workshops on CAD/CAM topics are intended to create an awareness and generate interest among practising engineers in this technology. It is envisaged that these workshops would enable industry personnel to identify key areas of CAD/CAM technology and judge the potential of its application in their own working environment.

In due course, the CAD/CAM Centre will be in a position to conduct comprehensive training programmes for industry personnel in specific areas of CAD/CAM application.   The present series of workshops would thus serve as a familiarisation programme.

Programme:

05th Wednesday

9000 - 1215 hrs	Session I Lecture: The role of NC, CNC techniques Programming NC, CNC machines.
1215 - 1330 hrs	Lunch
1330 - 1600 hrs	Session II Demonstration on computers/ CNC machining centre.

06th Thursday

9000 - 1215 hrs	Session III Lecture: The use of computers in NC programming. Part programming systems.
1215 - 1330 hrs	Lunch
1330 - 1600 hrs	Session IV Demonstration on computers/ CNC machining centre.

Venue: Seminar Room
Department of Mechanical Engineering, University of Moratuwa.

The workshop will be conducted by UNIDO Expert Dr. T. Markus (Associate Professor, Department of Manufacturing Engineering, Budapest Technical University. Hungary)

CAD/CAM Centre
Department of Mechanical Engineering,
University of Moratuwa.

ANNEX 7

CONTENTS

0. INTRODUCTION TO NC/CNC/DNC TECHNIQUES AND APPLICATIONS

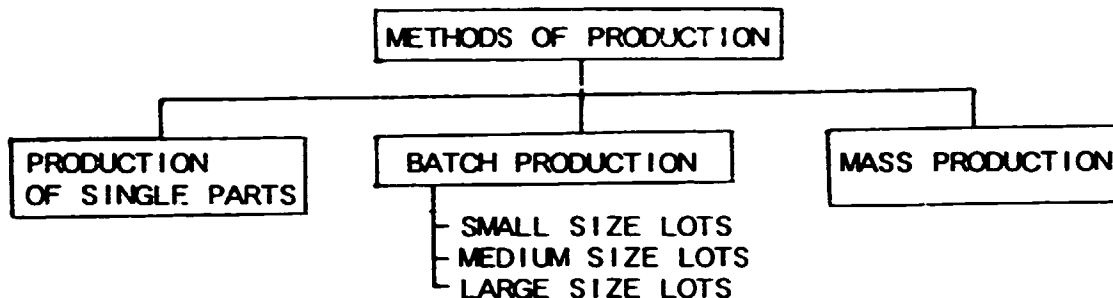The methods of production is shown on Fig.0.1.

Fig. 0.1

The automation of large size lots and mass production has been solved by using different types of automation (e.g. cam operated automatic lathes, production lines etc.)

The methods used for the automation of the production of large size lots and of mass production are not economical for single parts (and small/medium lot sizes).

However one of the basic charateristic features of machine industry is the production of small and medium size lots. On this field of application NC/CNC technique is the excellent solution.

0.1 BASICS OF NC/CNC/DNC TECHNIQUES

These techniques are one type of programmable automation methods where the whole process is controlled by commands which are constructed of letters, numbers and symbols. The control commands are prepared either manually or by using computers.

The technology has a wide variety of applications including:
- drafting,
- sheet metal press working,
- cutting technologies,
- assembly,
- spot welding,
- inspection etc.

The general reasons for NC/CNC techniques are as follow:

- high efficiency of NC/CNC machine-tools
  (an average factor of 4 is used for estimations)
- the workpieces need higher precision and
  uniformity.
- overall trend for higher flexibility because
  of smaller production lots and shorter product
  life cycles.
- optimization of the production flow

- decreasing costs of computer hardware
- need to increase the quality of work.
- obvious trend to integrate CAD with CAM.

Disadvantages are: higher investment cost, higher maintenance cost, trained personnel is needed.

The more concrete examination of NC/DNC techniques suggests the application if the following characteristics exist:

- frequent production of parts in small/medium lot sizes
- the part geometry is complicated
- many operations are executed on the part
- frequent changes of engineering design
- tight tolerances on the workpiece
- expensive material/workpiece, where production failure results high costs.
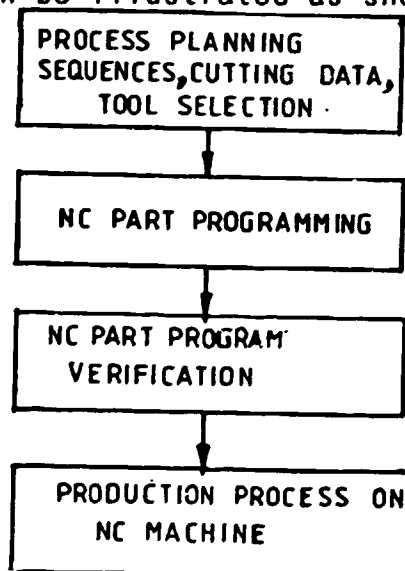
0.1.1 NUMERICAL CONTROL

The basic components of NC systems are:
- part programme (control commands)
- NC controller unit (or machine control unit = MCU)

The numerical controller generates those movements which carry out the machining of workpieces with the relative motions between the tool and the workpiece. The information source is the NC part program which contains the necessary geometrical and technological data for the machining.

The NC process can be illustrated as shown on Fig 0.2.

Fig 0.2

PROCESS PLANNING
SEQUENCES, CUTTING DATA, TOOL SELECTION

NC PART PROGRAMMING

NC PART PROGRAM VERIFICATION

PRODUCTION PROCESS ON NC MACHINE

The basic elements of the conventional control units:
- tape reader
- data buffer
- signal output channel to machine tool

- feedback channel from machine tool
- sequence controls for coordinating the operations.

The machine-tools are very flexible, minimising idle time. Automatic tool changers are equipped, one of the basic functions in the workpiece positioning. On the newer machining centers two tables or pallets are used, the workpiece loading or unloading can be made during the production of an other part.

0.1.2 COMPUTER NUMERICAL CONTROL (CNC)

The evolution of computer technology and control techniques resulted the appearance of CNC units. This means, that the conventional hard wired NC controller unit is replaced by a minicomputer (microcomputer).

The CNC technique brought solutions for some problems which existed in NC. These difficulties were as follows:

- The NC part program always needs a debugging verification procedure, the necessary changes meant an uncomfortable procedure at NC machines
- during the manufacturing process there was not possibility to change certain parameters (eg. speeds and feeds)
- the punched tape/tape reader coplex was always the last reliable unit of the system.

The CNC system is extremely flexible, where a Control Processing Unit (CPU) supervises the logical control, geometric data processing, program execution. CNC is a typical software control system.

The basic tasks of the microprocessor are:

-system management
-data input/output
-data corrections
-control of NC program execution
-processing of operator commands
-output of NC process variables to the display

The advantages of CNC can be summerised in the following statements:

- NC tape and tape reader have to be used just once (if the NC program media is punched tape), program editing and verification is done at machine-tool.

- increase of flexibility, wide variety in NC

program use (parameter programming, use of
macros or subroutines).

- ability to serve information for a com
puterised factory-wide manufacturing system.

0.1.3   Direct Numerical Control (DNC)

DNC is a manufacturing system where the machine
tool controls are interconnected to an external
process computer.
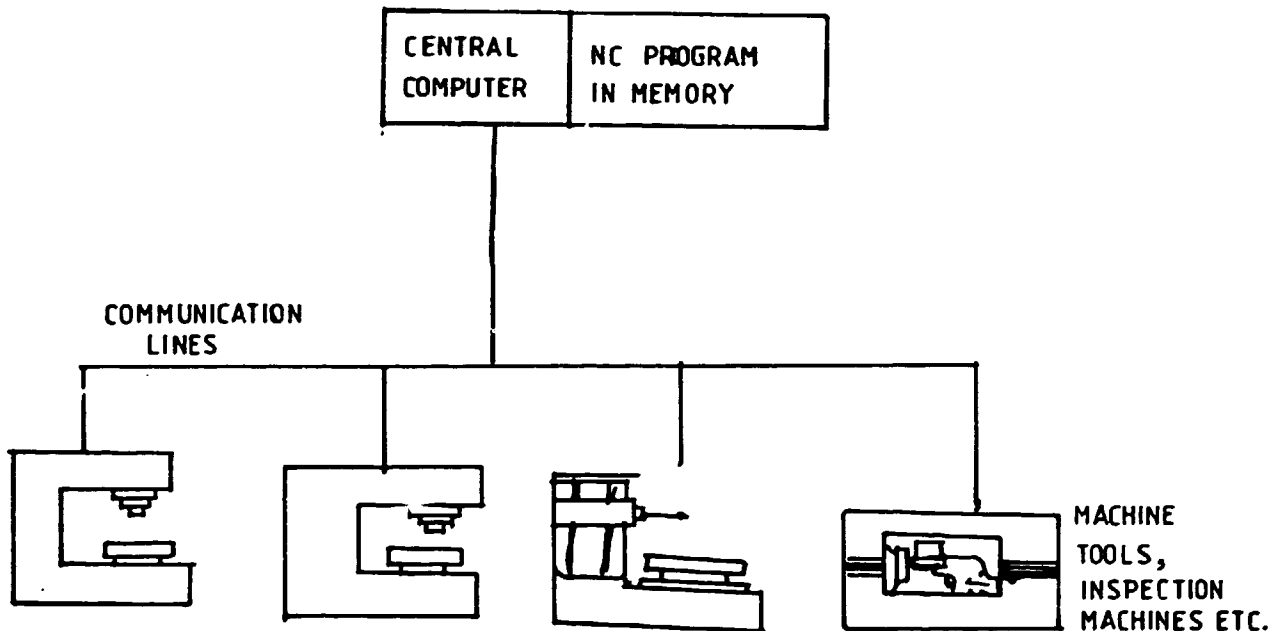The basic components are shown on Fig.0.3

Fig 0.3

The DNC systems are generally 'tapeless' systems, the
main error source this way is eliminated. Some of the
basic functions of DNC system are:
  - Administration of part program for the machine
    tool
      . input, storage, output, protocoling,
        editing, copying, erasing

. administration of NC part program and NC
  data

. library functions, storing, updating,
  retrieving, controlling memory access

- Distribution of NC part program and NC data

. activation of NC programs, acknowledgement
  of program calls, controlling start
  conditions, status reporting

. activation of communication interface

. transfer of program as blocks to the machine
  tool control

. control of the data transfer, status, and
  completion of transfer

The real flexible capabilities are experienced at com-
bined DNC/CNC systems. Beyond those another advantage
is resulting from the redundancy of the combined sys-
tem ie. if the central DNC computer fails the CNC
units of the system can be operated as stand-alone
machines.

0.2 NC PROGRAMMING

The basic methods of NC programming are of two types:
- manual programming
- computer-aided part programming

The manual part programming basically means a machine
level coding of the control instructions. Its ef-
ficiency and usability is strictly depending on the
complexity of the workpiece. Generally speaking upto
2.5 D production manual programming can be applied,
but in many cases even at 2D contouring the necessary
work of the programmer goes far beyond the acceptable
economic and time factors. The manual programming of
complex surfaces for 3D-5D manufacturing is practi-
cally imposible.

0.2.1 MANUAL NC PART PROGRAMMING

The basic document to start with is the part drawing.
For a successful program coding the usual production
engineering tasks also have to be completed, ie:

- operations sequence planning
- operations planning
- operation elements planning

The actual program coding can be started only if these
preceeding steps have been completed. This is a very

important factor because the optional utilisation of
the NC/CNC machines greatly depends on the quality of
part program - and the latter depends on the quality
of technological planning.

The control commands (blocks) are using word addresses
for the execution of specific functions. Some of the
word addresses are as follows (detailed list will be
given in Chapter 4):

    N - sequence number (block identification)
    G - preparatory functions (movement conditions,
        type of dimensional data - ie. absolute or
        incremental, fix cycles etc.)

    X ⎫
    Y ⎬ - primary linear motion coordinates
    Z ⎭

    F - feedrate function
    S - spindle speed function
    T - Tool function

    M - miscellaneous functions (spindle on/off,
        coolant on/off, end of program etc.)

For example a typical NC block is (linear movement to
X = 215.5   Y = 103.0, coolant on and spindle speed is
1600 min with counter-clockwise rotation on. The
sequence number of the control-block is 143).
N143 G01 X215500 Y103000 S1600 M04 M08(End of Block)

The coordinate values of the movements are calculated
by the NC part programmer. The basic thing what is
needed for this task is to create a coordinate system,
the 'part-coordinate system'. The origin of the part-
coordinate system can be selected freely but practical
reasons should be considered depending on part sym-
metry etc.

0.2.2 COMPUTER AIDED PART PROGRAMMING

The part programming work needs tedious labour in many
cases,including large amount of calculations for
complicated contours, surfaces etc. Besides these
factors the production engineering phase of the work
(operation sequences, cutting parameters) can result
serious errors. The reliable automation of this
process was of high importance. The solution came up
with the spreading use of the computers. Since the
late 50s many computer aided programming systems have
been developed with a wide variety in capacity, in the
level of automation and in the field of application.

The end-product of the computer aided part programming
systems are the same as of the manual NC programming
i.e. the NC control blocks, but the way leading to the
creation of NC part program is absolute different.

Instead of direct programming of the commands a 'lanugage' is used to describe the production task. This 'input part program' is processed by the computer and the NC part program codes are produced as output.

The details will be discussed in Chapter 2 - Chapter 4.

1. COMPUTER AIDED PART PROGRAMMING I. PROCESSOR/POST-PROCESSOR CONCEPT, LEVEL OF AUTOMATION

The early efforts of computer aided part programming didn't aim to be general systems, basically the machine-tool and control unit manufacturers developed some sort of programming aid which worked for the specific machine and control unit.

The decisive step was made with the development of the APT system, which has its marks on the majority of the systems even in our times. The language, the processing concept serve as basis for international standards and numerous 'APT-like' systems are in use all over the world.

The APT as a problem oriented system came up with the following philosophy (see Fig.1.1):

- use a high level quasi-English language to describe the production task

- this 'part programme' will be processed by the system ('processor')giving a solution which is independent of the specific machine-tool and specific control unit parameters.

- using the CLDATA as an input, a second processing phase follows,the machine-tool and control unit features will be considered in this phase by the programme ('post-processor') resulting the appropriate control blocks as output for the NC complex.

APT-like input (problem oriented, symbolic language to describe the production task

PROCESSOR ────▶ Processing of input part program

CLDATA   CLDATA (the machine and control unit independent output of the processing)

POST-PROCESSOR ──── transforming CLDATA into NC commands according to machine-tool and control unit features.
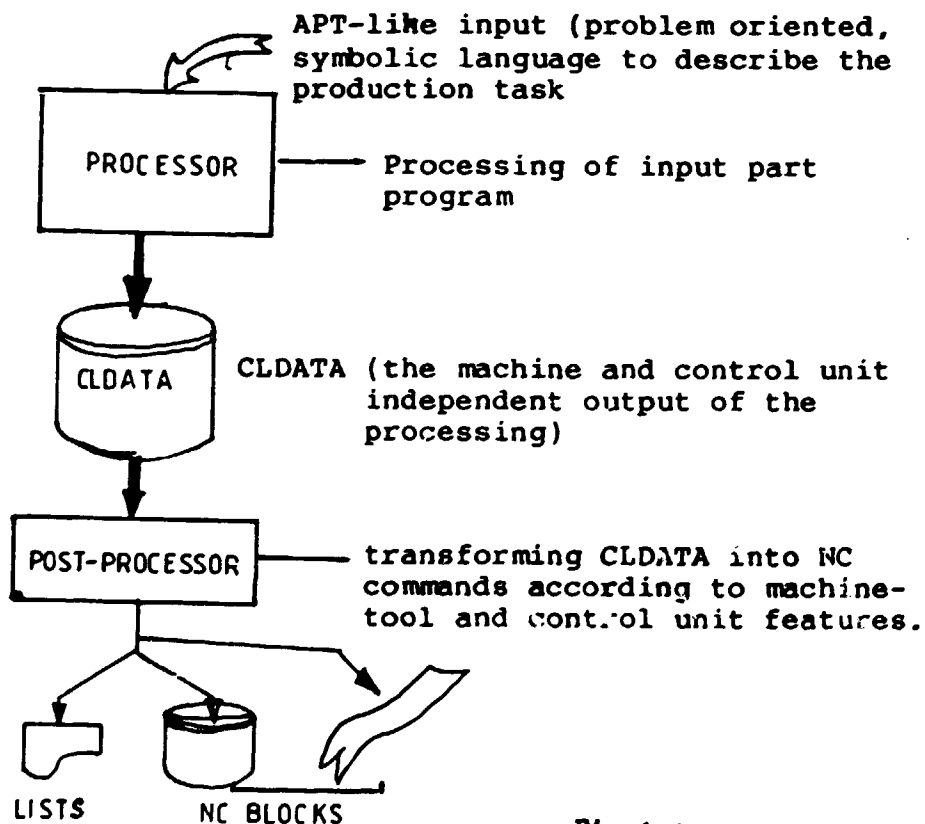
LISTS   NC BLOCKS

Fig 1.1

Many APT-like systems are available, but there are some differences between the systems. The first and crucial selection can be made on the basic potentials:

- primarily used for complex geometrical problems, no technological data calculation is done automatically

- medium or small complexity for the gemotrical possibility but some or the production engineering functions are automatically executed.

Between the two extremes many systems exist with different level of automation. As an example see the EXAPT family which was developed in GFR (West-Germany) as shown on Fig.1.2.
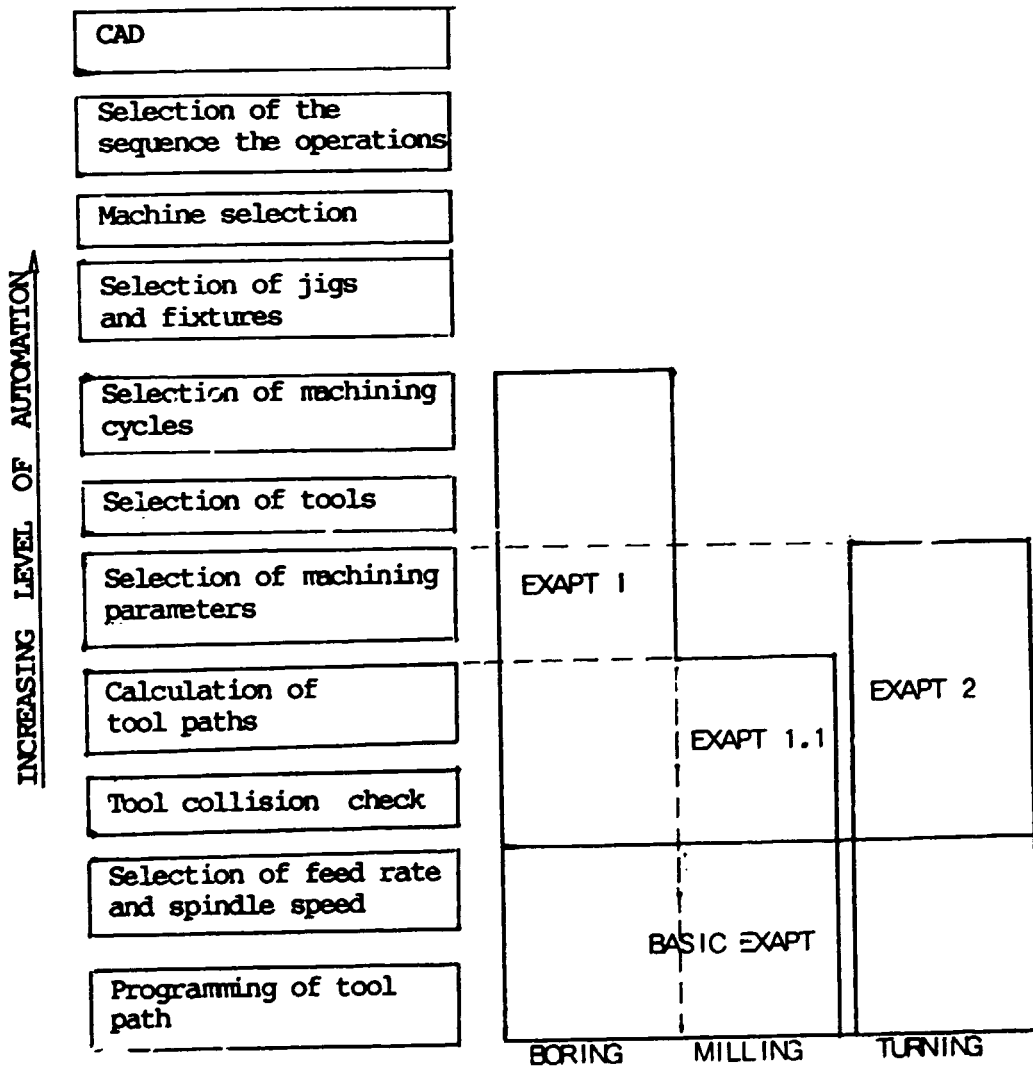

Fig. 1.2

The EXAPT systems have technological processor (with different
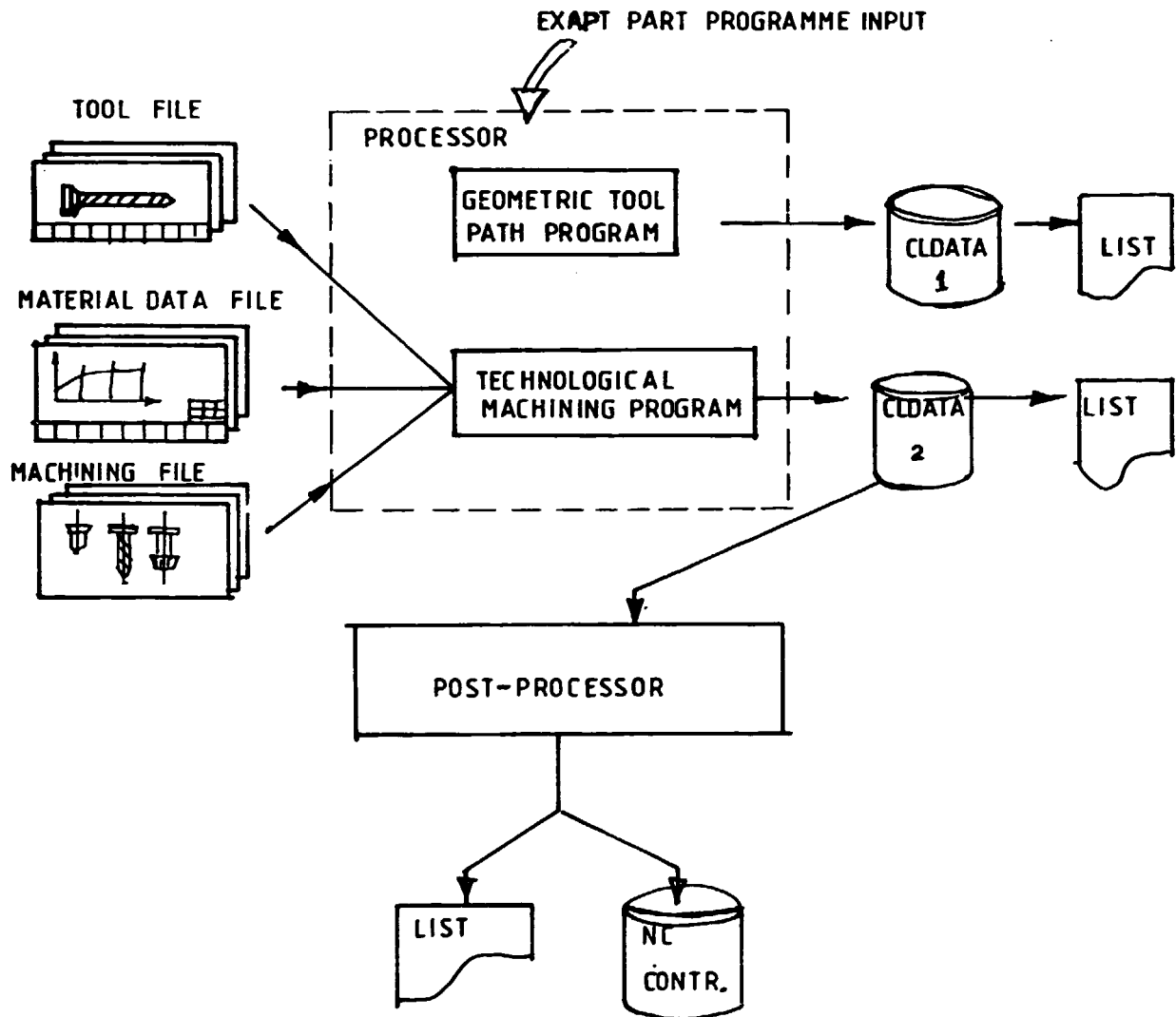level of capacity). The rough sketch of system structure
is shown on Fig.1.3

Fig 1.3

The work of the technological processor is supported by
the tool file, material data file and the machining file.
The first one has all data of the tools available for
the system, the material file among other data gives the
feed rate/diameter function etc. The calculation algorithms
for cycles and other machining related calculations are
stored in the machining file.

To illustrate the APT-like input languages and the part programming a simple manufacturing task is programmed with EXAPT. (See workpiece on Fig.1.4).
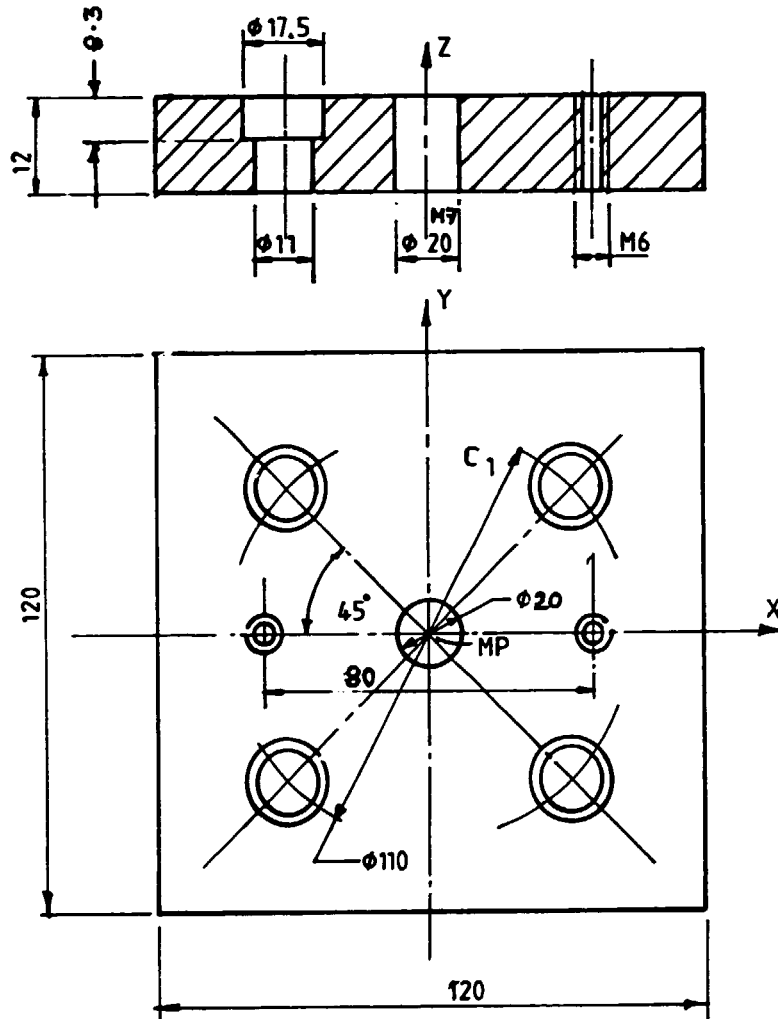

Fig 1.4

The part program is demonstrated on Fig.1.5. This little sample program is written for a point-to-point work, i.e. no contour manufacturing is needed. Looking at Fig.1.5 the typical program structure is obvious (introductory [header] statements, geometry, technology, execution), some explanations on the program follows:

```
1.    PARTNO/SAMPLE PLATE, MORATUWA
2.    MACHIN/TAKISAWA                                          HEADER
3.    TRANS/200,100,50                                         STATEMENTS
```

```
4.    ZSURF/12
5.    MP = POINT/0,0,12
6.    C1 = CIRCLE/CENTER.MP,RADIUS,55                          GEOMETRICAL
7.    PAT = PATERN/ARC,C1,45,CCLW,4                            DEFINITION
```

```
8.    PART/MATERL,12
9.    CLDIST/.8
10.   BORE1 = DRILL/SO,DIAMET,5,DEPTH,12,12,TOOL,315
11.   THREAD = TAP/SO,DIAMET,6,DEPTH,12,TOOL,416              TECHNOLOGICAL
12.   BORE2 = DRILL/SO,DIAMET,11,DEPTH,12                     DEFINITION
13.   SINK = SINK/SO,DIAMET,17.5,DEPTH,8.3
14.   REM = REAM/DIAMET,20,DEPTH,12
```

```
15.   COOLNT/ON
16.   FROM/-100,0,100
17.   WORK/BORE 1.,THREAD
18.   GOTO/-40,0,12
19.   GOTO/40,0,12                                            EXECUTION
20.   WORK/REM                                                DEFINITION
21.   GOTO/MP
22.   WORK/BORE2,SINK
23.   GOTO/PAT
24.   FINI
```

The header statements give the workpiece ID, the type
of machine-tool (post-processor) and the zero point shift
of the workpiece coordinate system. The geometry follows,
the definitions are simple, just one thing must be mentioned:
the identifiers of those elements can be cited only in
the statements which elements explicitely had been defined
in one of the preceeding definitions.

The technological statements with numbers 10-13 have an
SO modifier but statement No. 14 has not. This modifier
stands for single operation i.e. just the programmed operation
will be executed. If the SO modifier is omitted (as in
No. 14) the statement is a so called 'Ream Cycle' definition
and all the necessary operations needed prior to the reaming
operation will be calculated automatically by the system.

The execution of cutting operation are activated from statement No. 15. As mentioned before the reaming cycle (execution in statement No. 20) will active all the necessary preceeding operations.

When programming contouring cutting operations, the geometry and the execution statements are somewhat more complex, this is discussed in Chapter 2.

2 COMPUTER AIDED PART PROGRAMMING II
CONTOURING, THE CLDATA STRUCTURE, POST-PROCESSORS

2.1 CONTOURING

To start the discussion on the programming of contouring a sample part program for the workpiece of Fig.2.I is listed on Fig.2.2. No second view of the part is given, let us work with Z = 0 coordinate when executing contouring.
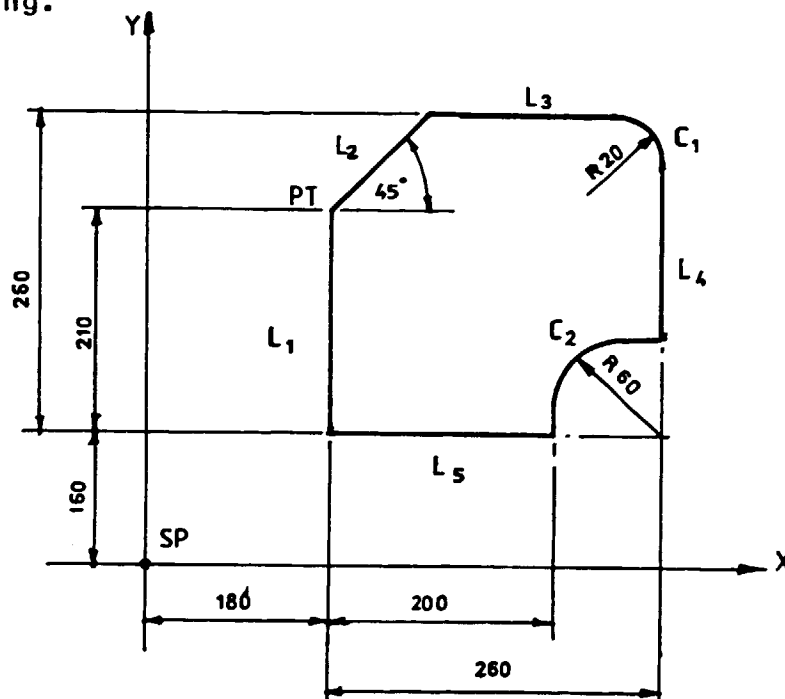

Fig. 2.1

If reading the part-program on Fig. 2.2 the special features of this concept of geometry are obvious, i.e. the system uses infinite elements (independently of the defined points of a line the lines have no directions and they are infinite, only complete circles can be described etc.)

```
 1 PARTNO/SAMPLE2,MORATUWA
 2 MACHIN/TAKISAWA,FANUC
 3 TRANS/300,250,150
 4 SP=POINT/0,0,0
 5 L1=LINE/180,160,180,210
 6 PT=POINT/180,210,0
 7 L2=LINE/PT,ATANGL,45
 8 L5=LINE/180,160,380,160
 9 L3=LINE/PARLEL,L5,YLARGE,260
10L4=LINE/PARLEL,L1,XLARGE,260
11C1=CIRCLE/YSMALL,L3,XSMALL,L4,RADIUS,20
12C2=CIRCLE/260,160,0,60
13CUTTER=16
14SPINDL/1200,CLW
15COOLNT/ON
16FEDRAT/180
17FROM/SP
18GO/TO,L1
19TLLFT,GOLFT/L1,PAST,L2
20GORGT/L2,PAST,L3
21GORGT/L3,TANTO,C1
22GOFWD/C1,TANTO,L4
23GOFWD/L4,PAST,C2
24GORGT/C2,PAST,L5
25GORGT/L5,PAST,L1
26SPINDL/OFF
27COOLNT/OFF
28RAPID
29GOTO/SP
30FINI
```

Fig. 2.2

The actual part-geometry are cut-out of the set of in-finite elements by the programmed tool movements (Seq no. 19-25).
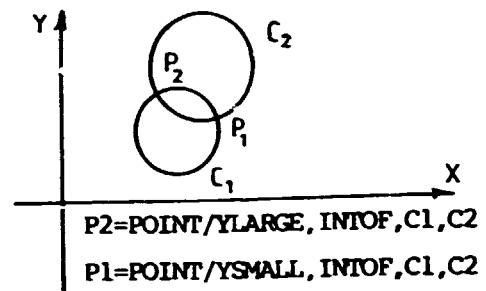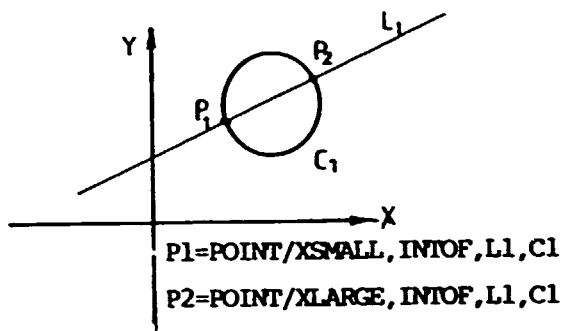
The geometrical definitions use modifiers to select the desired specific element. On Fig 2.3 a limited number of geometrical definitions are given.

The statement No. 19 specifies the relative position of the tool and workpiece. TLLFT means, that the tool is on left of the workpiece when executing the contouring machining (left or right should be judged by looking into the direction of the tool movement).
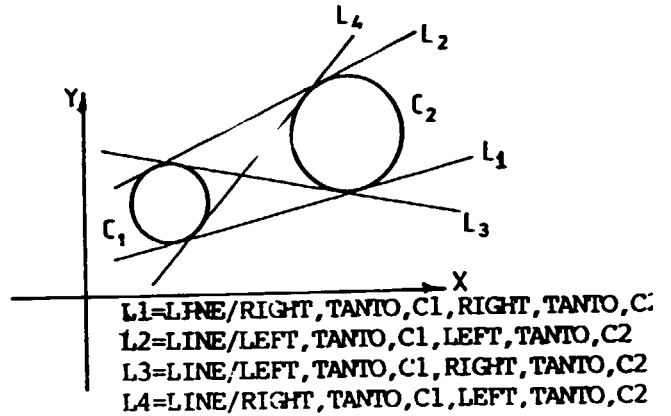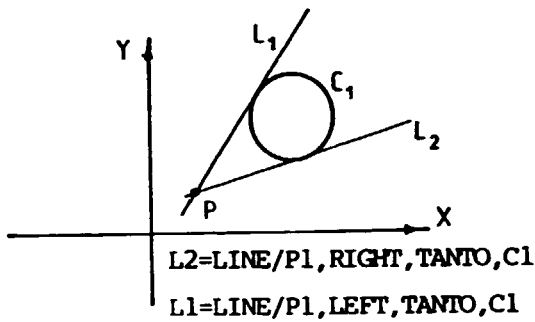
Comparing the definitions to those of the EXAPT sample program the difference is obvious between a processor with technological modules and a geometrical processor. In the latter case no separate definitions exist to declare machining operations but the motion statements themselves are the definition and the same time the ex-ecution statements. (Exceptions exist especially for the machining of sculptured surfaces where no series of in-dividual tool movements are programmed; examples shown in Chapter 3 and Chapter 9).

The position of tool is controlled by the TO,ON,PAST, TANTO modifiers. The concept of tool control is based on the 'three surface' philosophy, see. Fig. 2.4.
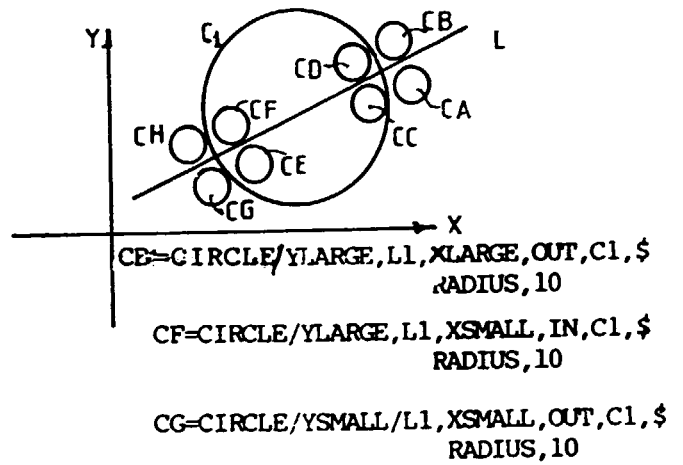
POINT

P1=POINT/XSMALL,INTOF,L1,C1

P2=POINT/XLARGE,INTOF,L1,C1

P2=POINT/YLARGE,INTOF,C1,C2

P1=POINT/YSMALL,INTOF,C1,C2

LINE

L2=LINE/P1,RIGHT,TANTO,C1

L1=LINE/P1,LEFT,TANTO,C1

L1=LINE/RIGHT,TANTO,C1,RIGHT,TANTO,C2

L2=LINE/LEFT,TANTO,C1,LEFT,TANTO,C2

L3=LINE/LEFT,TANTO,C1,RIGHT,TANTO,C2

L4=LINE/RIGHT,TANTO,C1,LEFT,TANTO,C2

CIRCLE

C1=CIRCLE/P1,P2,P3

CB=CIRCLE/YLARGE,L1,XLARGE,OUT,C1,$
 RADIUS,10

CF=CIRCLE/YLARGE,L1,XSMALL,IN,C1,$
 RADIUS,10

CG=CIRCLE/YSMALL/L1,XSMALL,OUT,C1,$
 RADIUS,10

LOFT CONIC

LC=LCONIC/5PT,PT1,PT2,PT3,PT4,PT5

CONEA=CONE/PTA,VECB,22

Fig.2.3

Fig. 2.4

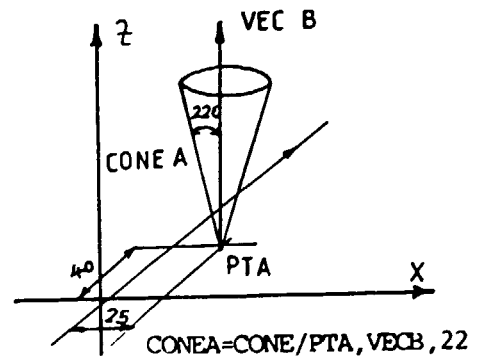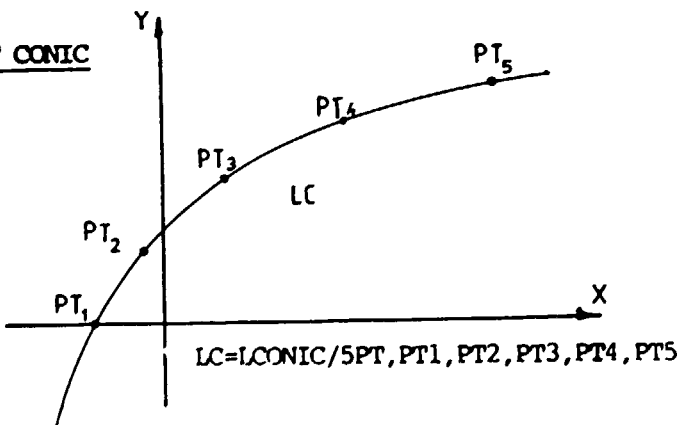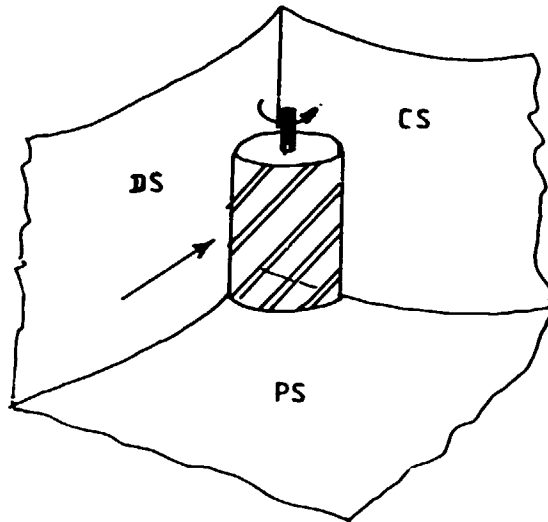The bottom of the tool (tool tip) is in connection with the PS (Part Surface) while the tool is executing the movement. The direction of movement is controlled by the DS (Drive Surface), the limit of movement is controlled by the CS (Check Surface) according to specific modifier programmed (the Figure 2.4 shows a TO position). If the sample program is processed the tool centre coordinate for the contouring will be written on CLDATA file with the appropriate codes of other statements (no basic geometrical definition will be stored on CLDATA).
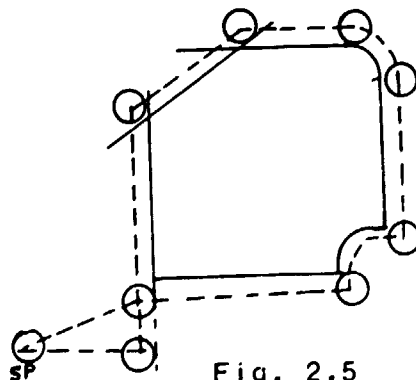
The contouring will result the tool path of Fig 2.5

Fig. 2.5

This 'APT-like' language structure what has been dis-
cussed serves as a basis for international
standardization. Many extensions and refinements had
been added to the basic definitions and ISO 4342 was pub-
lished as a reference input language set. Continuous
work has been done on the development of this reference
language (including sculptured surface machining) even in
our times.

2.2 CLDATA STRUCTURE

The end-product of the NC processor is the CLDATA which
serves as input to the post-processor.

The CLDATA is a data file with records of variable
lengths. Each processor output statement represent one
(or more) CLDATA record(s). The type of information can
be identified by major codes (and in some cases addi-
tional minor codes are also used).

The standardised form (major codes) and the role of the
record are listed on Table 2.1

The vertical structure of the CLDATA records shows the
following rules:

- the rocords are preceeded by a record of 1000 type
  which carries the original sequence number of the
  statement in question.

Type	Explanation
1000	INPUT SEQUENCE NO. OF DEFINITION
2000	POST-PROCESSOR INSTRUCTION
3000	CANONICAL FORM OF SURFACE DATA
4000*	TOOL POSITION RELATIVE TO DRIVE/PART SURFACES
5000	TOOL POSITION (MOTION VECTOR INFORMATION)
6000	TOLERANCE OR CUTTER INFORMATION
7000*	TOOL POSITION WITH RESPECT TO THE START-UP SURFACE
8000*	MOTION INFO (TOOL DIRECTION WITH RESPECT TO LAST MOVE)
9000	MULTI-AXIS OR UNITS INFORMATION
12000*	CARRIES DATA FOR CALLING WITH STRING ARGUMENT SEPC. PROGRAMS
14000	FINI (TERMINATION RECORD)
15000	INFORMATION FOR UNSEGMENTED NON-LINEAR TOOL PATH
16000	WORKPIECE CONTOUR DESCRIPTION
17000	TOOL DESCRIPTION
18000	MATERIAL DESCRIPTION
19000	MACHINE DESCRIPTION
20000	LITERAL TYPE POST-PROCESSOR COMMAND
28000 to 32000	RESERVED FOR PRIVATE USE

Note: RECORDS MARKED * CAN BE USED BUT DO NOT BELONG TO THE STANDARDIZED BASIC TYPES

Table 2.1

- for unsegmented tool paths (eg. circle) a record of type 3000 is used and followed immediately by a record of type 15000.

- for linear (seg·mented) movements frequently more than one record of type 5000 is needed (eg. for a sculptured surface). In this case the sequence number record (1000) is preceeding only the first record of type 5000 and is not repeated for all continuation records.

For example consider the sample program of Fig. 2.2.

The definitions from seq. no. 20 to 23 will generate the following records:

  .

  .

1000 carrying seq.no 20
5000 carrying the terminating coordinates axes of
     linear motion
1000 carrying seq.no 21
5000 carrying the terminatiny coordinates of
     linear motion .
1000 carrying the seq. no 22
3000 canonical data of circle CI
15000terminating coordinates of movement along CI
     circle
1000 carrying seq.no 23
5000 carrying the terminating coordinate of linear
     motion

  .

  .

  .

  .

The structure of records type 1000

W(ord)	1	-	integer	= record sequence number
W	2	-	integer	= 1000
W	3	-	integer	= seq. no of statement in part program
W	4	-	character	= ID of original statement
W	5	-	character	= ID of original statement

The record of type 5000 (linear motions)

W1 - integer = record sequence number

W2 - integer = 5000

W3 - integer = 3 for motion start (FROM)
4 for incremental coordinates (GODLTA)
5 for absolute coordinates (GO...)
6 if this is a continuation record

W4 - character= ID of geometrical element

W5 - integer = subscript or point index (eg. at patterns)

W6 - real = X coordinate of point

W7 - real = Y coordinate of point

W8 - real = Z coordinate of point

----

if the record is carrying more than one point information (movement on curved surface, segmented circle etc.) then the record continues up to W 245 and if necessary continuation records follow:

W9 - real = X coordinate of 2nd point

W10- real   = Y coordinate of 2nd point

W11- real   = Z coordinate of 2nd point

----

if multiaxis machining is used (eg. 5D) the record is carrying Dextets and not triplets:

W9 - real   = I component of first tool axis
                vector

W10- real   = J component of first tool axis
                vector

W11- real   = Z component of first tool axis
                vector

.

.

.

The post-processor records carry informations on spindle speeds, feedrates, coolants, tool changes, optional block skips etc..

Statements of seq. no 14 to 16 load as follows (without seq. records)

W1 (integer)   = record seq.number

W2 (integer)   = 2000

W3 (integer)   = 1031 - minor code for SPINDL major
                 word

W4 (real)   = 1200 - spindle speed

W5 (integer)   = 60 -code for minor word CLW

W1 (integer)   = record seq. number

W2 (integer)   = 2000

W3 (integer)   = 1030 - minor code for COOLNT major

```
                           word
        W4 (integer)   = 71 - code for minor word ON
-------------------------------------------------------
        W1 (integer)   = record sequence number
        W2 (integer)   = 2000
        W3 (integer)   = 1009 - minor code for FEDRAT major
                           word
        W4 (real)      = 120 - feedrate in MM/MIN
-------------------------------------------------------
```

The processor outputs the CLDATA records, the movements
are calculated into movement elements in case of com-
plex shapes or of group of movements (exceptions exist
eg. hole machining cycles etc.).

The processing of CLDATA is executed by the post-
processor.

(The CLDATA records are also compiled in international
standards: ISO 3592 and ISO 4343)

2.3   POST-PROCESSOR

The post-processor serves as a link between the proces-
sor and the machine tool/control unit. Its basic task
is to convert the information carried in CLDATA to NC
control blocks. All significant data of the machine
tools and of the control unit are accessible for the
post-processor.

To fulfill its task the post-processor reads the CLDATA
records and processes them (depending on the number of
reading of the CLDATA file 'one-pass' and 'two-pass'
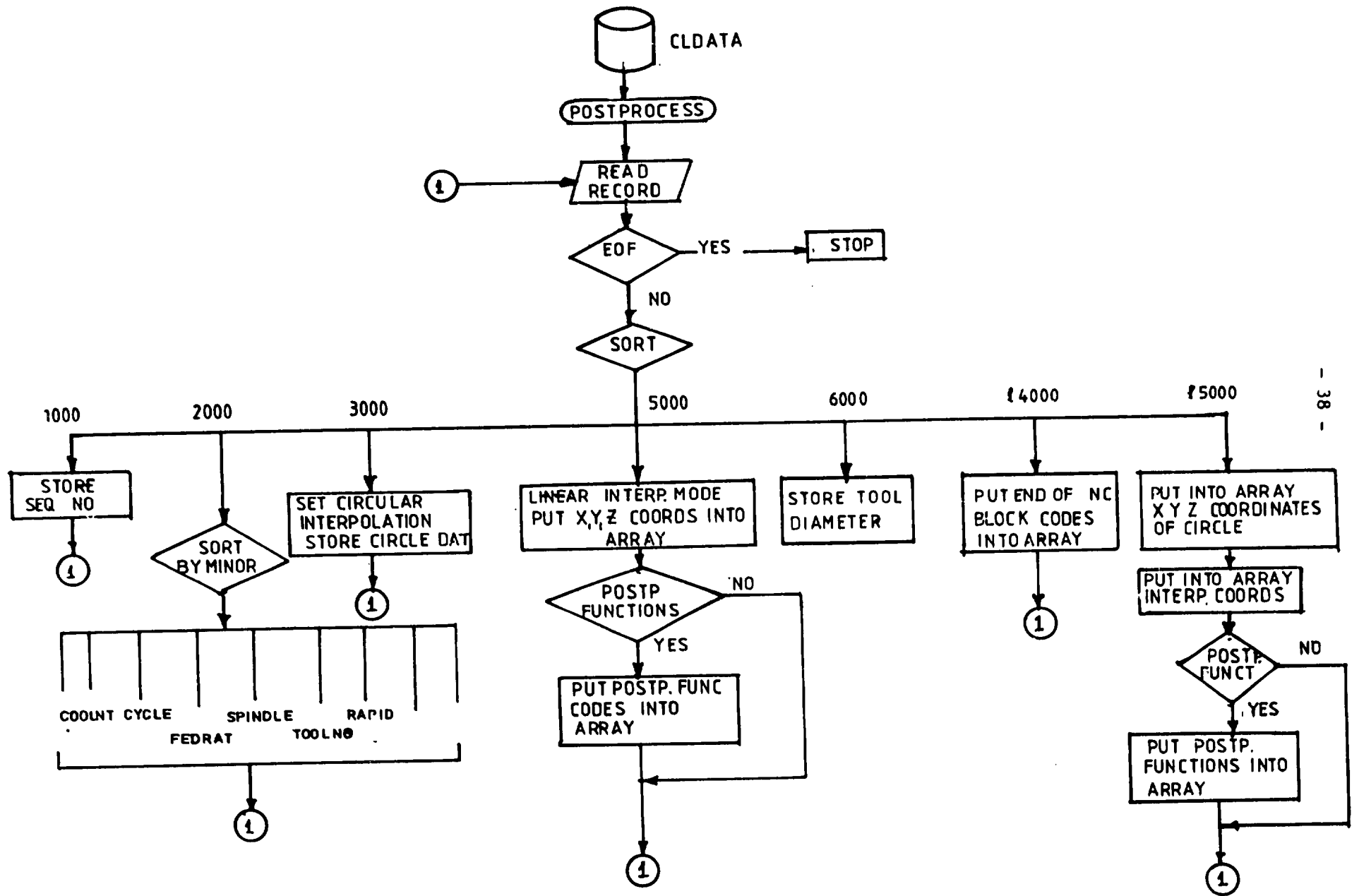post-processors exist). A very rough sketch of the

Fig 2.6

post-processor structure is given on Fig 2.6.

The first step is a sorting procedure according to record types. The processing of the records differ in many ways.

- seq. no records are stored up to the next record reading to serve as a flag in case of error messages (1000 type records)

- most of the post-processor functions are not written into separate control blocks, but they are stored and will be put into those control blocks which carry tool movement information (2000 type records for spindle speeds, feedrates, coolant etc.)

- when processing movement records, the necessary interpolation code (if circle is used also the interpolation parameters) should be written into the control block, together with the 'waiting' miscellaneous functions (feed, spindle, speed etc.)

The end product of post-processing is the file of NC control blocks and lists for the machine-tool operator (list of control commands, instructions for tooling, instructions for the setting of correction data etc.)

3. COMPUTER AIDED PART PROGRAMMING III
PROGRAMMING 3D-5D MANUFACTURING

Workpieces with surfaces of curved shapes need full 3D or 5D manufacturing capabilities. 5D machining is especially powerful and serves as the most economical manufacturing technique for special dies/moulds, aircraft parts, turbine impellers, compressors etc..

Those machine tools are 5D machines where besides the linear movements (x,y and z) the tool and/or workpiece can rotate about two rotational axes. One of the typical solution is the application of two rotary tables. Different possibilities also exist, eg. the tool can rotate about two rotational axes etc..

The rotary tables generally have guide limited capacity concerning workpiece dimensions, but five sides of a workpiece can be manufactured, in one set-up. This machine construction is also the best suited for the machining of impeller blades.

3.1 MANUFACTURING OF 5D SURFACES

In this and in the next paragraph the discussion is based on the capabilities and structure of the FAUN 5D programming system. Other possibilities will be shown in Chapter 9.

Basically three special surfaces can be programmed.

a. Ruled surfaces defined by two curves (general planes contain these curves) and line rulings which connect the curves (Fig. 3.1)

CURVE (Major directrix)

Line ruling (generator)

CURVE 2 (Slave directrix)
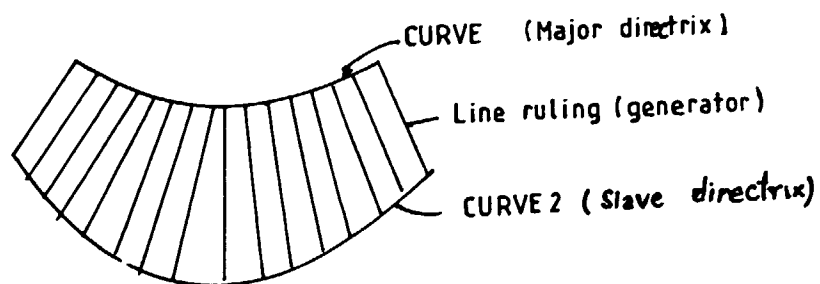
Fig. 3.1

b. Translational surfaces generated with the sweep motion of a space curve (generatrix) along an other space curve (directrix). See Fig. 3.2.
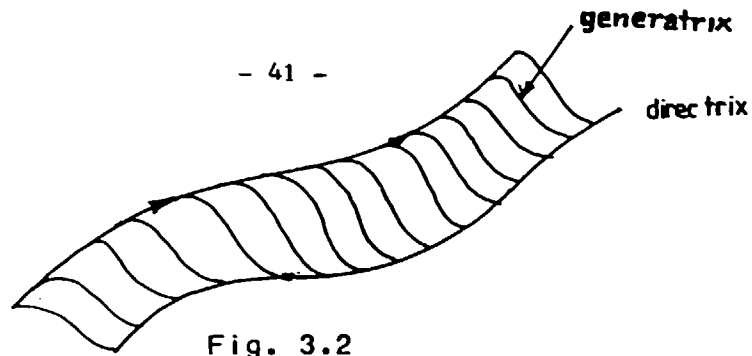
generatrix

directrix

Fig. 3.2

While moving along the directrix the generatrix is able to tilt or to execute zooming, resulting a very complex surface. Besides these operations the generatrix can behave as a cone ruling or as a cylinder ruling.

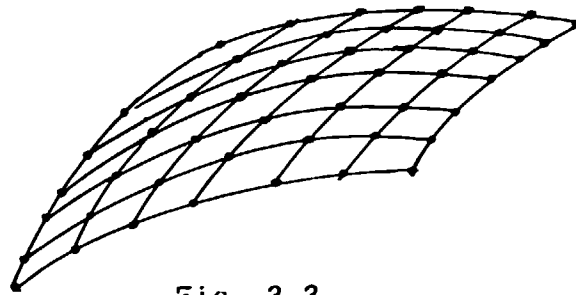c. Sculptured surface as built up by rectangular grid of M*N points (Fig 3.3)

Fig. 3.3

The points represent surface points (as on Fig 3.3) or Bezier polygon points.

The machining statements for the special surfaces have definitions for the strategy of movements (one-way, zig-zag, spiral), for tool tilt, for the strategy of ap-proaching movements and departing movements (in direction of surface normal, in direction of tool axis etc.) for possible lifts when changing the tool paths etc. Eg. the machining of ruled surface R1 from P1 starting point with zig-zag strategy, with a ball-tool which is tilted by 5[0] and approaching of surface is needed in surface normal direction (length of movement is 100 mm), at tool path changes 10 mm of tool-lift is needed and the depart from the finished surface is programmed in tool-axis direction (length of movement is 200 mm) looks like follows:

MKRSF/R1,START,P1,ZIGZAG,TOOL,FACE,TILT,5,3,IN,SNORM,100,
LIFT,10,OUT,TAXIS,200
The zig-zag and one-way strategy is used for the machin-ing of ruled surfaces and translative surfaces.

Sculptured surfaces allow zig-zag and spiral (meander movements (see Fig 3.4)
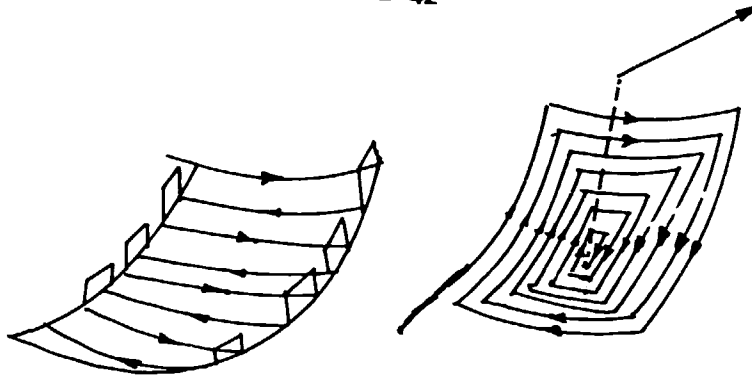
Fig. 3.4

At the processing and manufacturing of special surfaces numerous aspects need careful consideration.

- Eg. an interesting and not too attractive speciality of the ruled surfaces that in many cases the surface cannot be manufactured to the exact geometrical shape even theoretically if tool side is used. Explanation is given on Fig. 3.5.
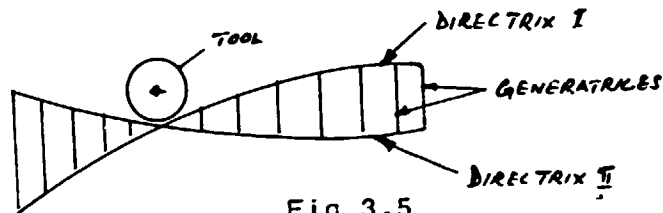

Fig 3.5

The tool is located with its side on one of the generatries (tool is shown from axis direction). The surface normal along the generatrix is changing, which means that tool diameter should be 0 mm diameter to avoid immediate geometrical errors (fortunately in practice the tolerance many times makes possible the use of tool-side for machining).

- The surface definitions (and consequently the processing algorithms) are different for the surfaces. Because of the tedious calculations needed for sculptured surfaces and because of the existence of some sort of analytical rules for the other two surface types the processing time shows significant differences.

Processing about the same surfaces with identical tolerances the computer time needed is the following.

Ruled surfaces   5 - 10 min
Translative surfaces   12 - 14 min
Sculptured surfaces   90 -120 min

- Practice is needed for the effective work, because programming and processing the same surface with the

only difference that major and cross movement direc-
tions are changed results great differences in time
(depending on the curvature of surface).

- Processing - and the number of CNC blocks - highly
depends on tolerances. The effect of changing the
tool size is not so significant (see Fig. 3.6) but
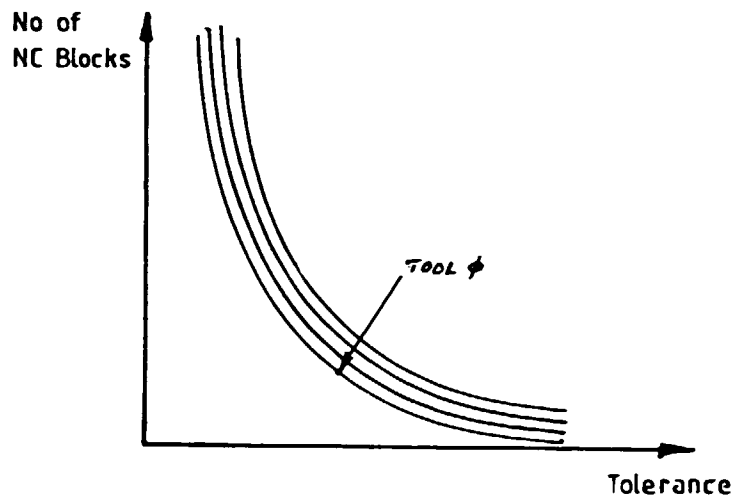naturally the increasing the tool size also helps.

Fig 3.6

- Typical feature of the 5D movements the existence of
'non-linear'. This means that between two points
when the 5D movement is executed the linear and
rotational movements are accomplished simultaneously
(start and termination in the same movement for all
five motions) with possible interference occurance
on the theoretical surface (see Fig 3.7)

Fig 3.7
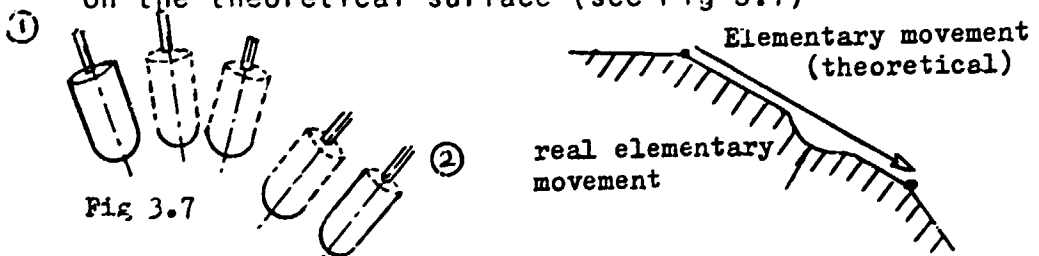
The control of this error within the acceptable
tolerance limits needs many calculations from the
processor, i.e. this problem should be solved by the
system designer and the user (part programmer) need
not take care of it.

The part programming is made by a menue based dialog sys-
tem which applies APT-like input language structure (in
semantical meaning). The basic version of the FAUI system

does not calculate cutting parameters, the required data of spindle speeds, feedrates etc. should be defined by the user the same way as it was introduced in the 2nd sample part-program.

The execution statements are individual movement statements (eg. GOTO type) as complex WK..(work ...) Statements as it was illustrated in this paragraph for a ruled surface.

3.2   SYSTEM STRUCTURE OF A 5D CAPP/CAM SYSTEM (FAUN)

The modular structure of the system makes possible to compile different versions with varying capabilities.

The system structure is shown on Fig. 3.8.

The system control, handling of the several entry points selection, file archivation, error listing are accomplished by the monitor program. The input module handles the menues of the input dialogue procedure, offers full editing capability for the user and gives list of the input data in APT-like statement format. The decoding module calculates and converts all geometrical elements into canonical formats (i.e. the basic elements eg. circles will be stored in the same format independently of the original definition version). This canonisation procedure is not valid for the special surfaces.

The processing module has double function:

- processing up to 2.5 D the input part program (plus 3D machining of planes of arbitrary positions).

- serving as a sub-monitor program to activate the special surface and roughing modules.

The surface modules calculates the tool paths according to tolerances and tool sizes. The result is a series of coordinates (surface points, surface normal components tool axis components) written into an intermediate file.

These data are converted into CLDATA records by the processing module.

The postprocessor is not a regular processor module but it also can be activated by the moniter program.

The latest version of the system has a technological processor for calculating and optimisation of cutting
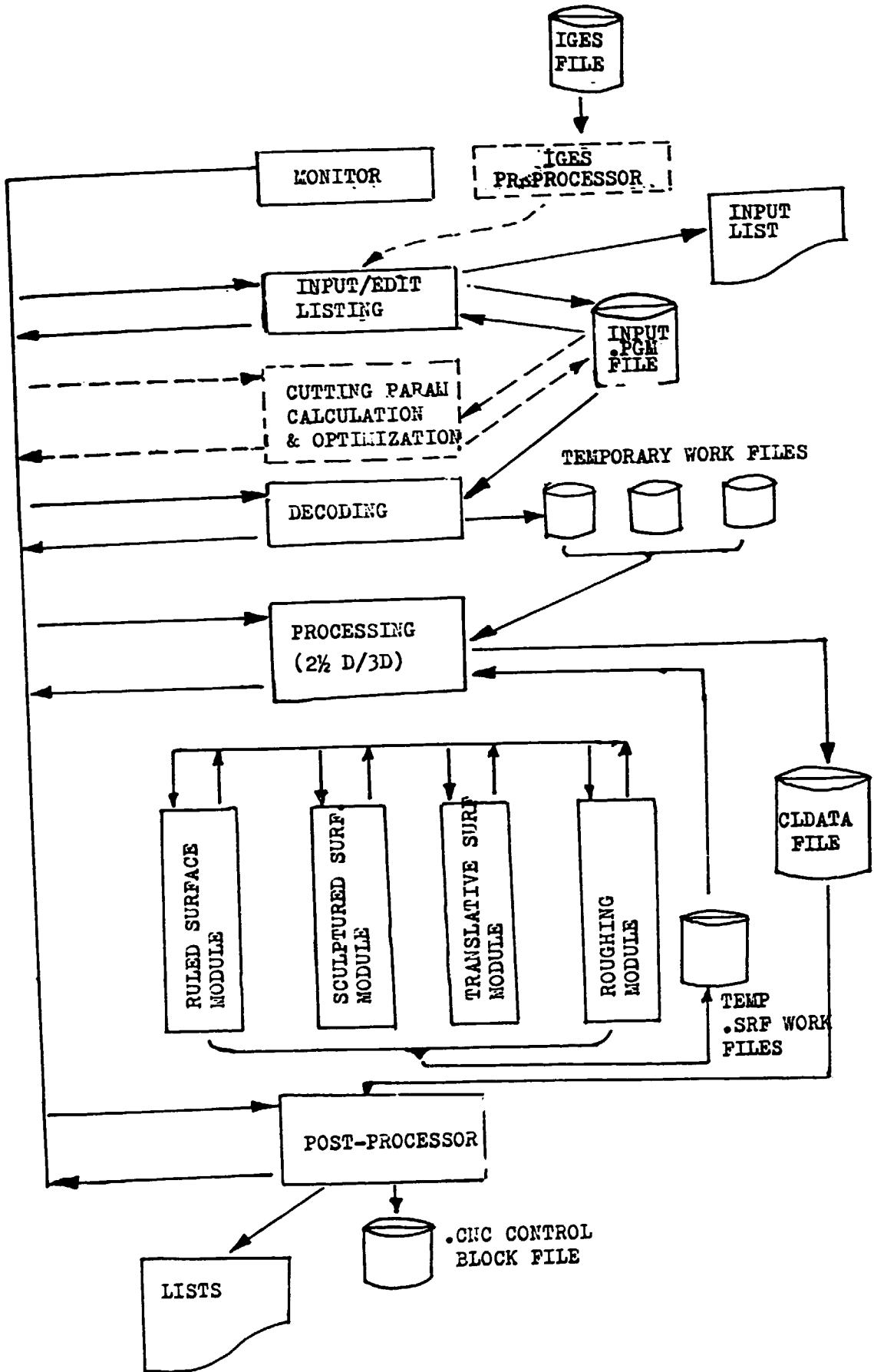
Fig 3.8

parameters. This capability can be utilised up to 2.5 D.

The programming of feedrates, spindle speeds are done by the usual way (SPINDL, FEDRAT etc. definitions). If the program does not have cutting parameters for the machining statements, then

TPROC/ON

Machining definitions

TPROC/OFF

should be programmed and the automatic calculation will be carried out.

The FAUN system has also an IGES preprocessor module. The IGES (Initial Graphical Exchange Specification) file structure serves as a CAD/CAM data interface and input geometry for the CAPP/CAM system can be directly written onto the input part program file.

4. ADDRESS CODES, AXIS AND MOTION NOMENCLATURE

The characters used for NC, CNC control functions are listed in Table 4.1. The unassigned codes have different uses, just for illustration some possible applications are given in parentheses. The multiple use is valid for the majority of the codes too, explanations come later.

4.1 MOTIONS AND AXIS

The coordinate system and machine movements are defined in such a way, that the part programmer can describe the machining operations without knowing whether the tool approches the workpiece or the workpiece approaches the tool. There are international standards (ISO) for this topic.

General Considerations

- The standard coordinate system is a rectangular right-handed system with the three principal axes labelled X,Y,Z with rotary motions about these axis labelled A,B and C respectively. (See Fig.4.1)
- The standard coordinate system is used to provide the coordinates of a moving tool with respect to a stationary workpiece.
- The positive direction of movement of a component of the machine is that which shall cause an increasing positive dimension of the workpiece.
- Unprimed letter is used to designate tool movement. Primed letter denotes the movement of the workpiece. Movement in the primed direction is opposite to the corresponding unprimed movement. (see Fig. 4.2)
- The location of the origin (X=0, Y=0, Z=0) or the angular origins (A=0, B=0, C=0) shall be located at the machine origin which is defined by the manufacturer. Workpiece zero is arbitrary and may be adjustable.

The Z axis

- Z axis is parallel to the principal spindle of the machine
- If several spindles exist one of them shall be selected as principle spindle , preferably perpendicular to the workholding surface.
- In case of swivelling principal spindle the Z axis shall be parallel to spindle axis when the spindle is in the zero position. The preferred zero position is with the spindle perpendicular to the work-holding surface.

- If the swivelling spindle can lie in only one position
  parallel to one of the axis of the standard three axis
  system, this standard axis is the Z axis.
- When the machine has no spindle, the Z axis shall be
  perpendicular to the work-holding surface.
- The +Z direction is defined from the workpiece to the
  tool-holder.
  (The tailstock is considered for the lathes to be the
  toolholder)
- The +Z axis of the coordinate measuring machines is
  collinear but in the opposite direction to the
  acceleration of gravity vector.

The X-axis of motion

- The X-axis is horizontal and parallel to the
  work-holding surface (where it is possible).

Machine with rotating tools

- With horizontal Z axis: positive X shall be to the
  right when viewed in the negative Z-direction.
- Vertical Z axis, single column: +X axis shall be to
  the right when viewed from the front of the machine
  into the column.
- Vertical Z axis, Gantry type machine: +X axis shall be
  to the right when viewed from the left-hand gantry
  support.

Machine with rotating workpieces

- The X axis of motion shall be radial and parallel to
  the cross slide. The positive axis shall be away from
  the axis of rotation.

The Y axis motion

- Positive Y shall be in the direction to make a
  right-hand set of coordinates.

A,B and C rotary axis

- These are rotary axes about X-, Y- and Z- axes or
  parallel to X, Y and Z.
- Positive A,B and C are in the direction to advance
  right-hand screws in the positive X, Y and Z
  directions respectively.

Additional Axes
Linear Axes

- When in addition to primary linear axes X, Y and Z there exist secondary linear axes parallel to the primary axes, these shall be noted U,V and W respectively. If tertiary axes exist they shall be designated P, Q and R respectively. When the existing linear motion is not parallel to X, Y and Z, they can be noted as U,V,W,P,Q and R as is most convenient.
- The primary linear axes should be those nearest the principal spindle, the secondary the next nearest, etc.

Rotary axes

When there exist secondary rotary axes, either parallel to A, B and C or compounded or swivelled, they shall be designated D or E.

Direction of Spindle rotation

Clockwise spindle rotation shall be in the direction to advance a right-hand screw into the workpiece.

Direction of Motion

Motion in positive direction of linear or rotary axes increases the positive position values and decreases negative position values.

- If the tool is moving the direction of movement and axes directions are equal. (Positive movements are designated with +X, +Y, +Z, +A, +B,...)
- If the workpiece is moving, the direction of movement and axes direction are opposite. (Positive movements are designated with +X', +Y', +Z', +A', +B',...)

Characteristic examples of axis nomenclature and motions are given on Fig. 4.3 - Fig. 4.10.

4.2 CONTROL COMMANDS OF NC/CNC UNITS

To give a wider overview on programming this paragraph deals with a 6D control unit. Specific FANUC programming will be discussed in Chapters 6 and 7.

4.2.1 Control codes of a 5D machining centre/5D CNC unit (UNIMERIC 700 CNC)

This control unit is able to control six axes simultaneously; the control codes are listed in Table 4.2. Because of machine tool limitations in fact only 5 axes are used; see Fig. 4.11.

Some of the address codes need more explanations because of multiple use (like E, F, H, L, U, V, W) or special use (as P, Q and R).

The E code is basically used for register operations. The CNC equipment offers special functions which make the NC programs more versatile and same time the length of the programs are considerably reducing.

The possibilities are:

- use of memory registers (variables)
- arithmetic calculations with memory registers
- conditional and unconditional branches in NC part program.
- using program cycles

The memory registers of the control unit can be addressed from the part program and the data stored in registers are used or altered by the part program. (E00 - E99 --> one hundred memory register is offerred by this control unit)
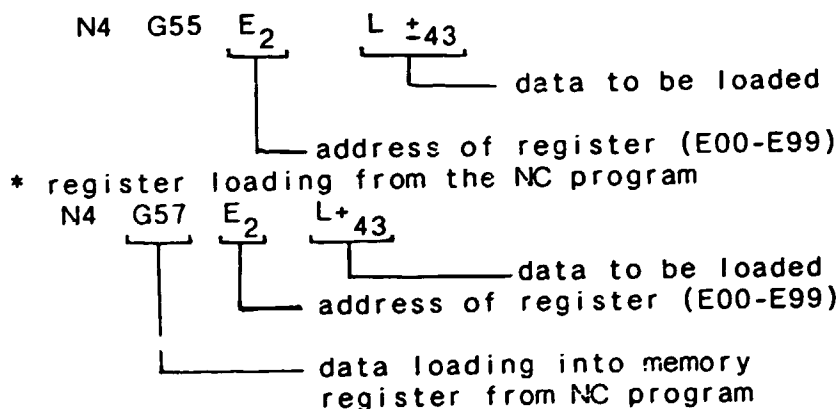Eg. Loading data into memory registers by three ways:

* if [OFF SET] used, with activating [E] address on the key board.

* using data loading punched tape

    N4   G55   E$_2$ L $\pm$43
                                 └─── data to be loaded

                └─── address of register (E00-E99)

* register loading from the NC program
    N4   G57   E$_2$ L+43
                          └─── data to be loaded
                └─── address of register (E00-E99)

        └─── data loading into memory
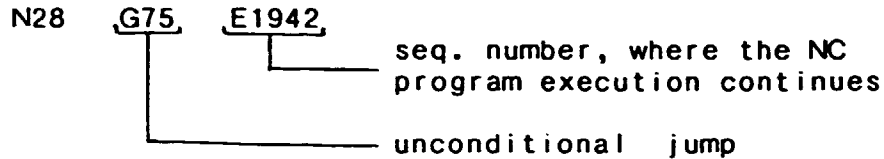   register from NC program

Not only the value of L can be loaded into a memory register, but the data stored in another register too.
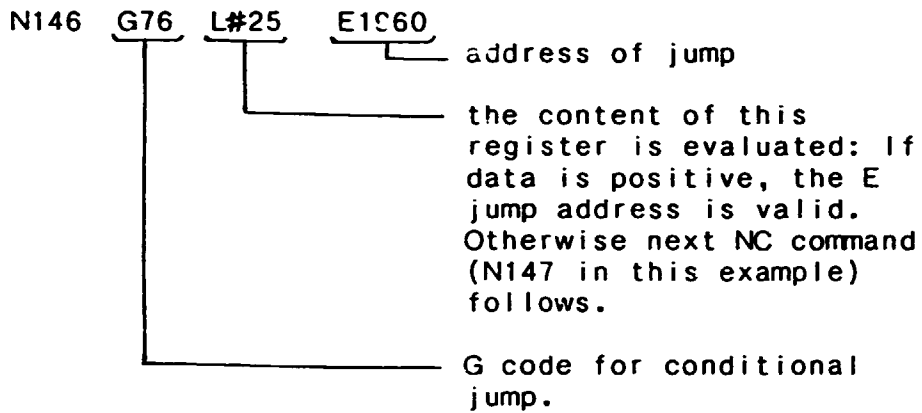
N10 G57 E14 L#10 means, that the data of register 10 will be loaded into register 14. This is the parametric use of E memory registers.

<u>Programming of jumps</u> is also based on the E address words.

Unconditional jump is eg.:

N28 G75, E1942,

     └──── seq. number, where the NC program execution continues

└──── unconditional jump

Conditional jump is also allowed in the NC part program.

N146 G76 L#25 E1060

     └──── address of jump

└──── the content of this register is evaluated: If data is positive, the E jump address is valid. Otherwise next NC command (N147 in this example) follows.

└──── G code for conditional jump.

Subroutine calls and subrouting technique is discussed in Chapter 6.

Next a simple demonstration follows on the excellent flexibility of NC program cycles.

<u>Sample NC Program</u>

NC program is needed for drilling of a pattern m*n holes (Fig 4.12). The algorithm of program logic is shown on Fig 4.13. The numbers in ☐ mean the explanations on Table 4.3 which lists the complete NC part program.
Memory registers used:

        E10 : A
        E11 : B
        E16 : m-1
        E20 : n-1
        E21 : z coordinate at the end of machining.

Conserving <u>F codes</u> the special use is the <u>G37</u> control code which gives in seconds the time of simultaneous movements along the six axes. The speed of the individual movements is determined by the CNC control unit. The other cases are G38 and G39.

The <u>G38</u> code activates a path velocity

$$\cdot \; F_{path} \; = \; \sqrt{F_x^2 + F_y^2 + F_z^2}$$

for the linear movements. If A, B and C are also programmed,, first the linear movements will be executed, then the rotational movements follow.

The G39 code defines velocity for the rotational axes.

$$F = \sqrt{F_A^2 + F_B^2 + F_C^2}$$

The rotational movements will be executed first, then follow the linear movements.

For rotational and linear movements the following criteria is valid:

$$F_{rot} \ (0/min) = \frac{\sqrt{A^2 + B^2 + C^2}_{[0]}}{\sqrt{X^2 + Y^2 + Z^2}_{[mm]}} * F_{linear} \ (mm/min)$$

The P, Q, R addresses are used for the compensation of tool radius. The U, V, W coordinates are the tool length compensation components in case of tilting spindle. The use of compensation (correction) codes are discussed in Chapter 5.

Finally concerning the G and M codes the existence of the 'three digit' codes should be mentioned. Efforts have been made to workout a suitable international standard in which the whole wide spectrum of NC/CNC machines have the appropriate codes. Eg the M codes are grouped the following way (1 st digit of the codes):

Group	Field of Application
0	General NC/CNC use
1	Drilling, Milling machines and machining centres
2	Turnin g
3	Punchinng, nibbling, flame-cutting, laser and plasma cutting, water-beam machining etc.
4	Unassigned
5	Optimisation for adaptive control
6	Special multi-spindle machines and machines with several working units
7-8	Unassigned
9	Reserved for code extension

The G codes are planned to be the same as before, but as a first digit the group code will precede the 2nd and 3rd numericals.
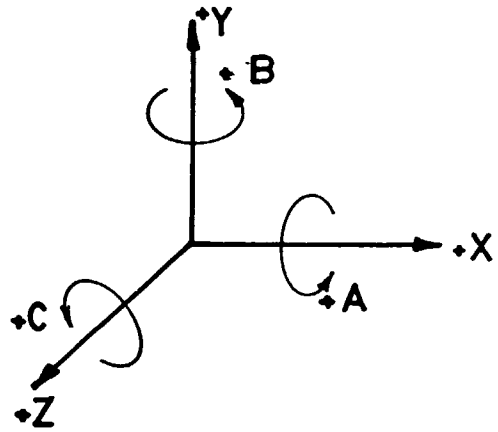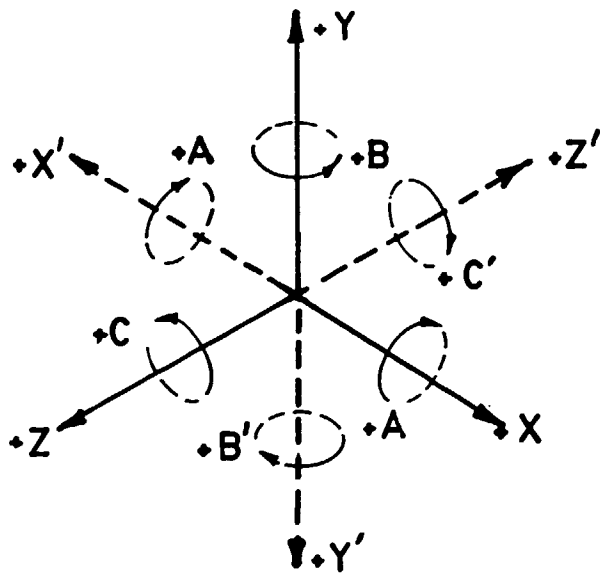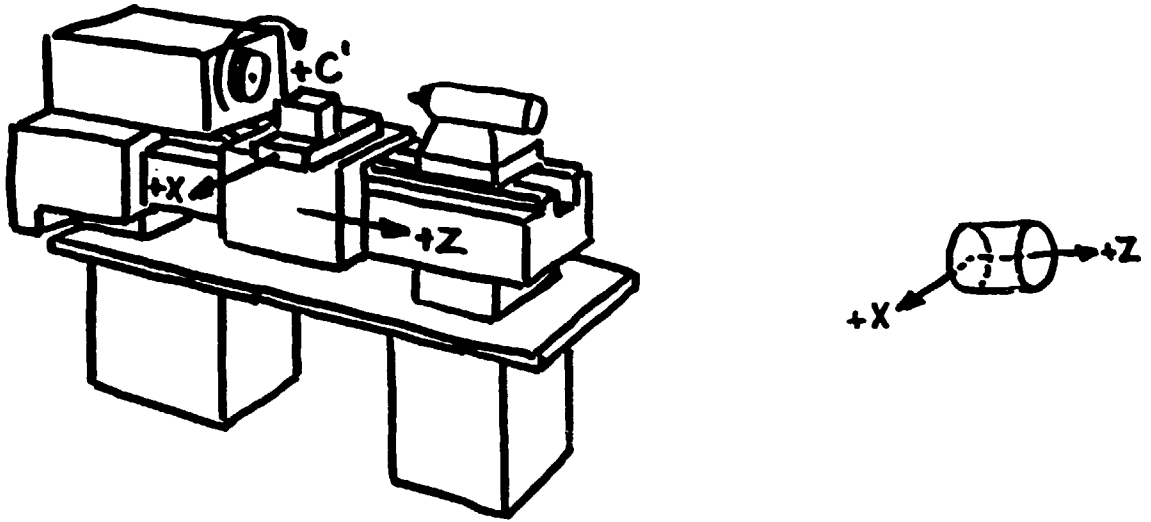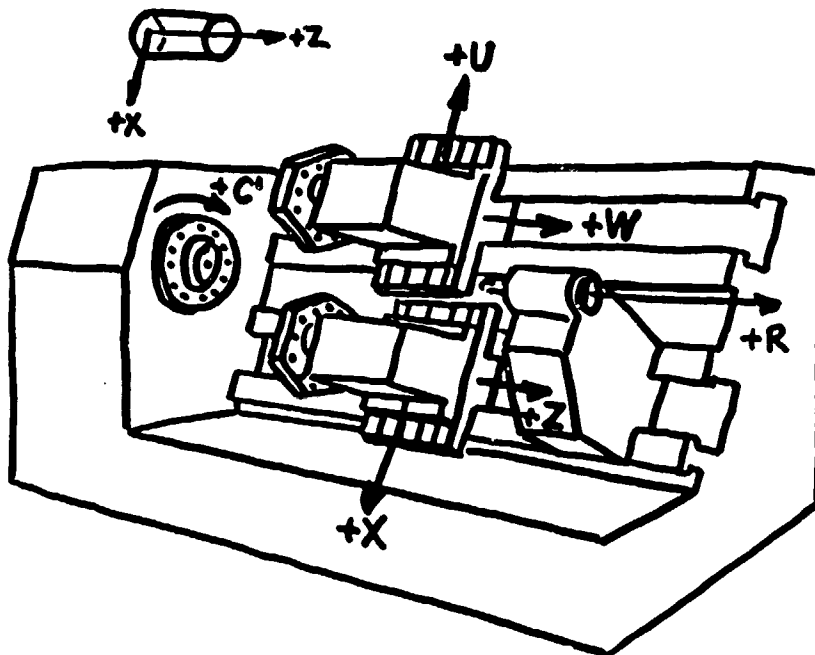
Fig 4.1

Fig 4.2

Fig. 4.3 PARALLEL LATHE

Fig. 4.4 TWIN TURRET LATHE WITH PROGRAMMABLE TAILSTOCK
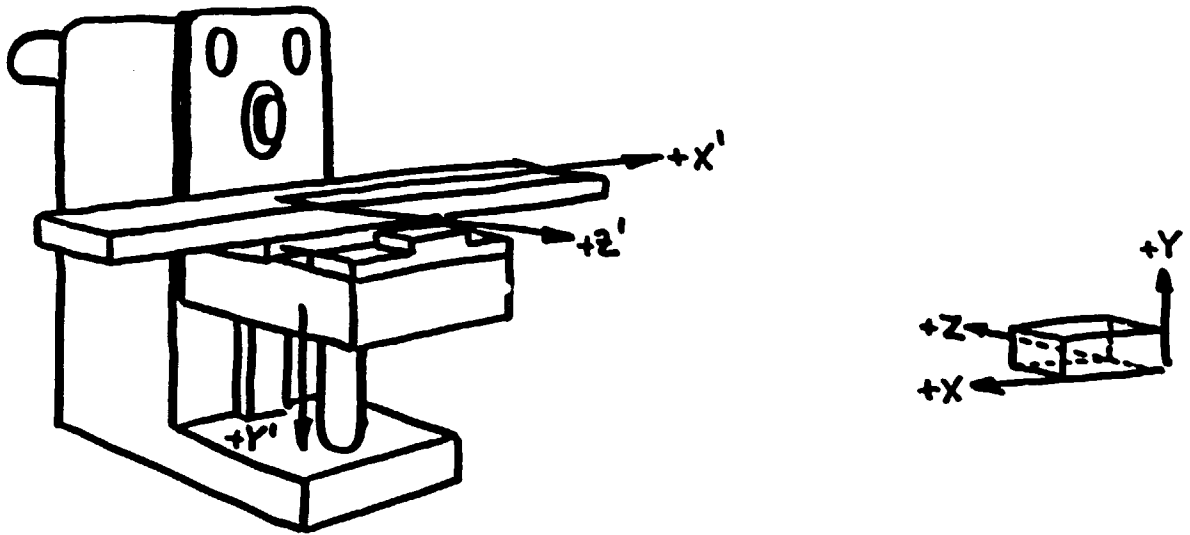
Fig. 4.5   MILLING MACHINE (HORIZONTAL SPINDLE)
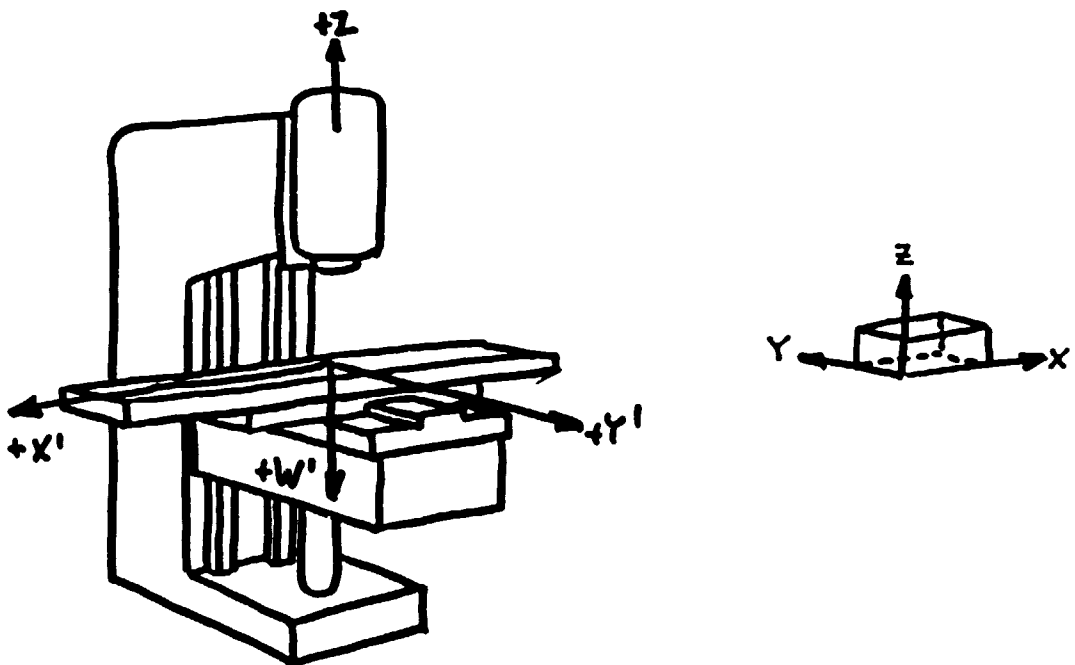
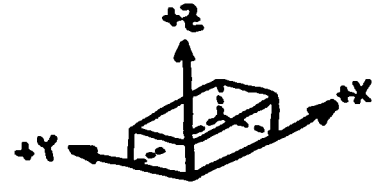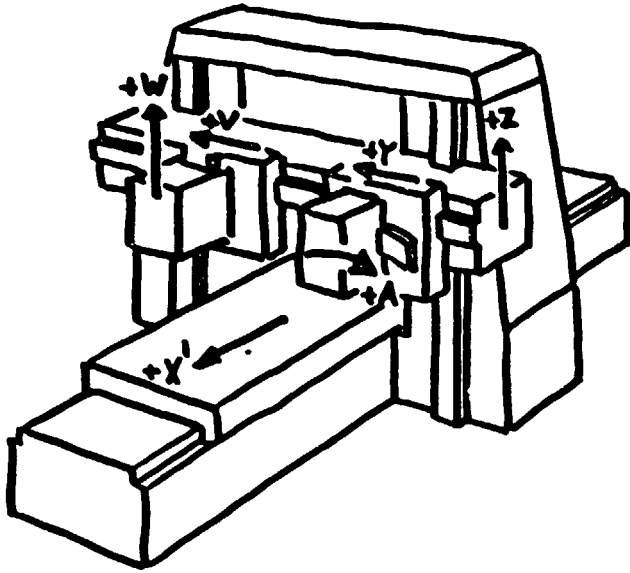Fig. 4.6   MILLING MACHINE (VERTICAL SPINDLE)

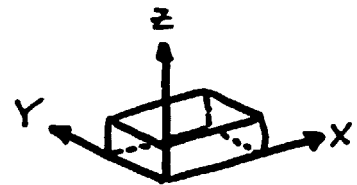Fig 4.7   PLANO-MILLING MACHINE (BRIDGE PROFILER)
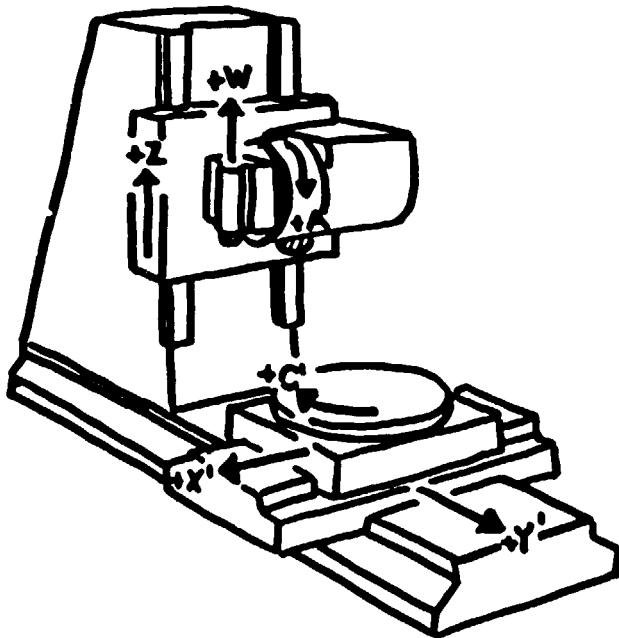
Fig. 4.8   PROFILE AND CONTOUR MILLING MACHINE (WITH TILTING HEAD)
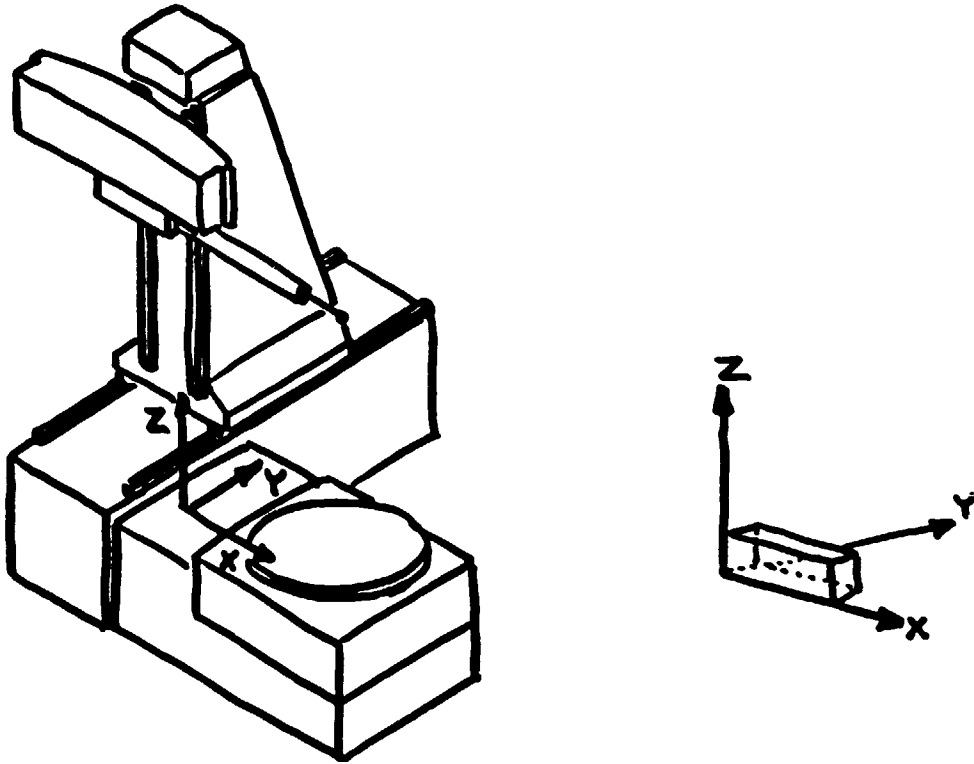
Fig 4.9   COORDINATE MEASURING MACHINE
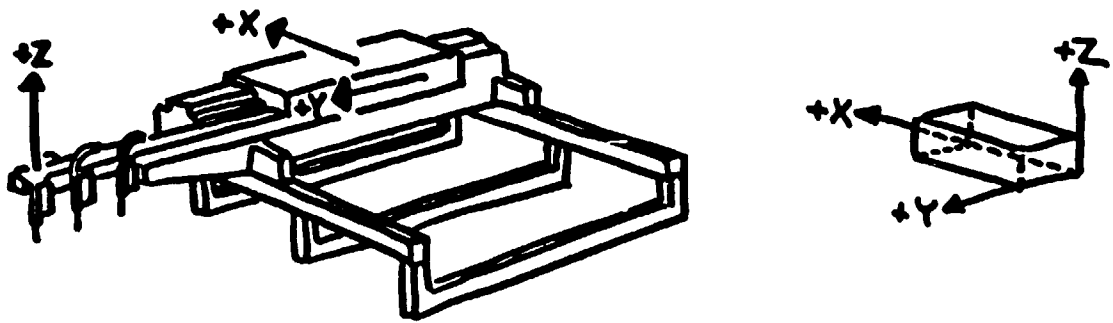
Fig. 4.10   FLAME-CUTTING MACHINE
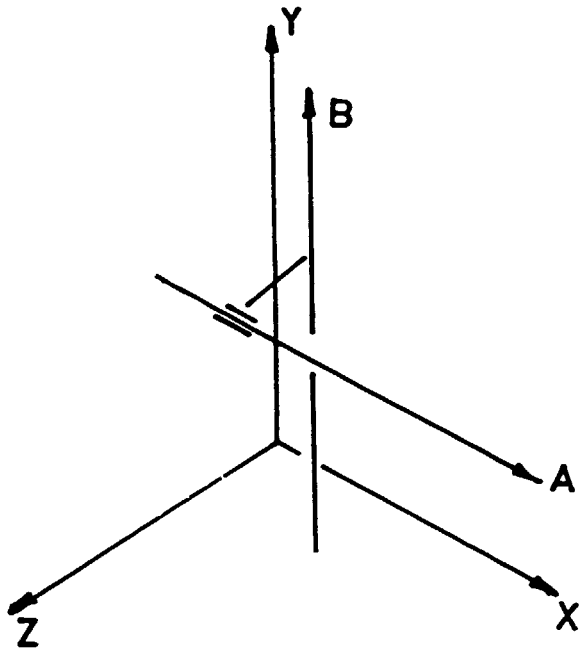
Fig 4.11 3D Machine (3 Linear Axes and 2 Rotary Tables
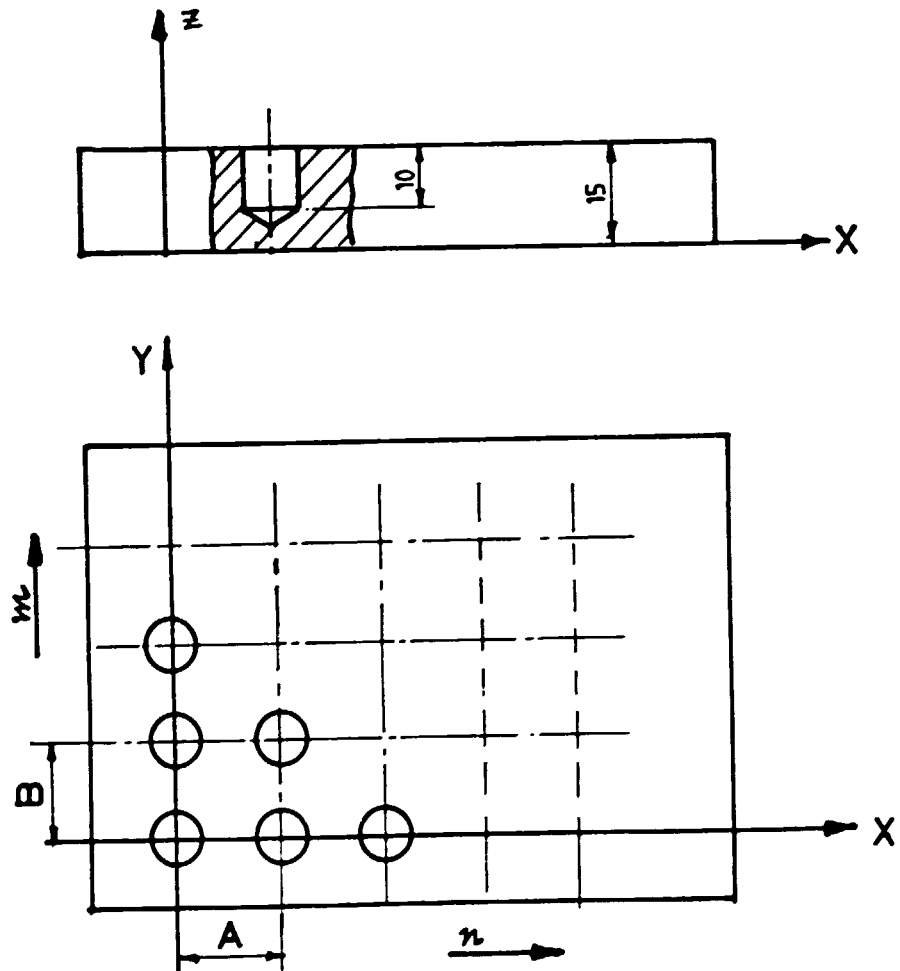
Fig 4.12 Example on NC program cycles
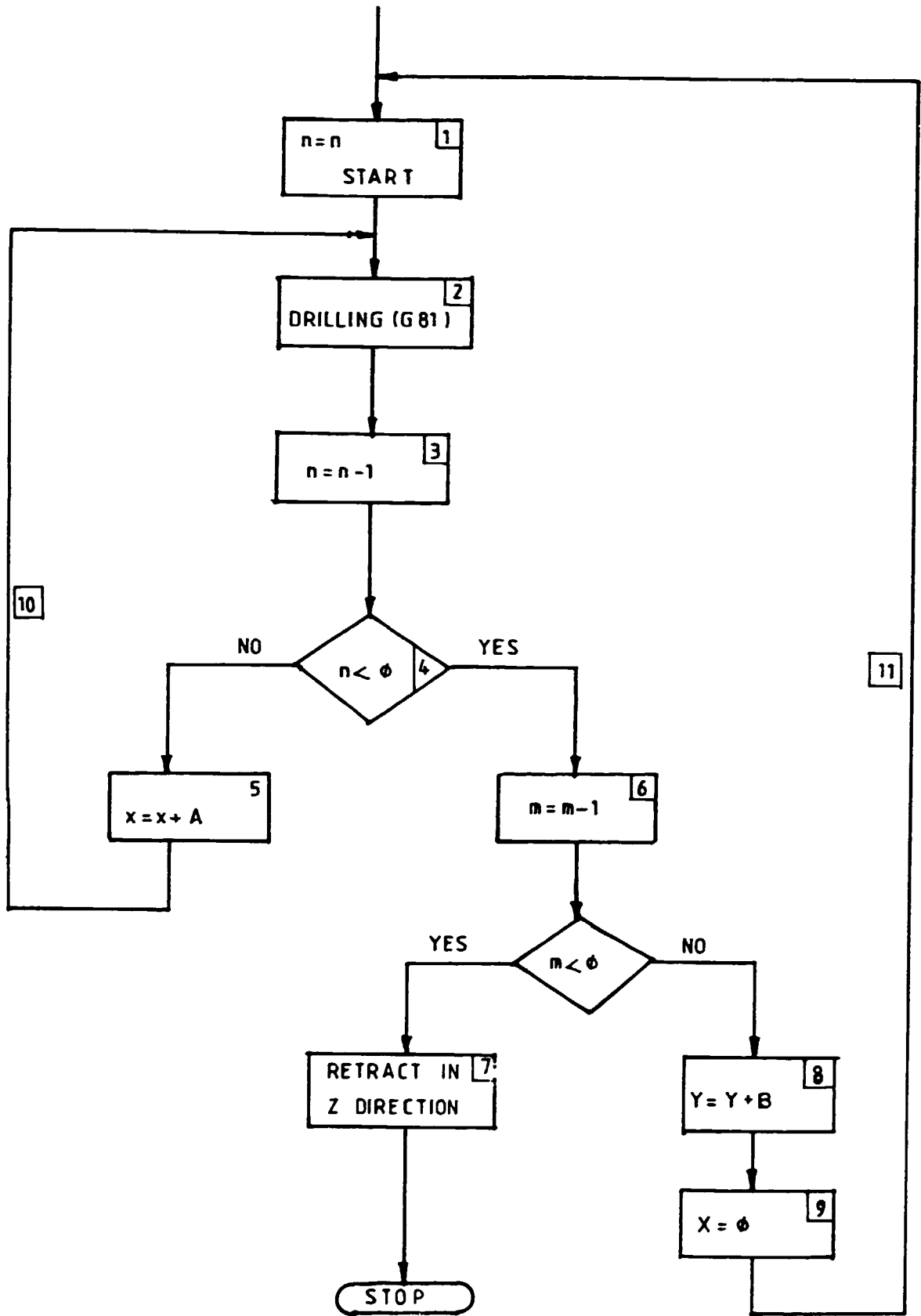
Fig 4.13 Flowchart of NC Program

TABLE 4.1

A - ANGULAR DIMENSION AROUND X AXIS

B - ANGULAR DIMENSION AROUND Y AXIS

C - ANGULAR DIMENSION AROUND Z AXIS

D - ANG. DIM. AROUND SPECIAL AXIS;THIRD FEED FUNCTION

E - ANG. DIM. AROUND SPECIAL AXIS; SECOND FEED FUNCTION

F - FEED FUNCTION

G - PREPARATORY FUNCTION

H - UNASSIGNED (NUMBER OF REPETITIONS OF CANNED CYCLES)

I - INTERPOLATION PARAMETER TO THE X AXIS

J - INTERPOLATION PARAMETER TO THE Y AXIX

K - INTERPOLATION PARAMETER TO THE Z AXIX

L - UNASSIGNED (NUMBER OF REPITITIONS OF SUBPGMS)

M - MISCELLANEOUS FUNCTION

N - SEQUENCE NUMBER

O - NOT RECOMMENDED (COMMENTS - GO)

P - 3RD RAPID TRAVERSE PARALLEL TO X-AXIS

Q - 2ND RAPID TRAVERSE PARALLEL TO Y-AXIS

R - 1ST RAPID TRAVERSE PARALLEL TO Z-AXIS

S - SPINDLE SPEED

T - TOOL FUNCTION

U - SECONDARY MOTION PARALLEL TO X-AXIS

V - SECONDARY MOTION PARALLEL TO Y-AXIS

W - SECONDARY MOTION PARALLEL TO Z-AXIS

X - PRIMARY X-MOTION

Y - PRIMARY Y-MOTION

Z - PRIMARY Z-MOTION

TABLE   4.2

ADDRESS	FORMAT	FUNCTION
A	$\pm\,43$	CONTROLLED ROTARY AXIS
B	$\pm\,43$	CONTROLLED ROTARY AXIS
C	$\pm\,43$	CONTROLLED ROTARY Axis
E	23	LENGTH OF DEEP DRILLING STEPS (G83)
	4	START ADDRESS OF SUBROUTINE FOR SUBR. CALL (G72)
	4	ADDRESS FOR UNCONDITIONAL OR CONDITIONAL JUMP (G75-G77)
	2	PARAMETER ADDRESS (G55,G57,G58)
	1	SELECTION OF ZERO POINT SHIFT (G92)
F	4	FEEDRATE (MM/MIN-G94)⌉FOR LINEAR
	13	FEEDRATE (MM/REV-G95)⌋ AXES
	22	FEEDRATE (SEC-G37) FOR 6 AXES
	3	REGISTER ADDRESS OF MACHINE DATA
G	2	PREPARATORY FUNCTION
H	43	DELAY (SEC)
	3	MEMORY ADDRESS FOR THE COMPENSATION OF MECHANICAL ERROR
I	$\pm\,43$	CIRCULAR INTERPOLATION - CENTER X COORD
J	$\pm\,43$	CIRCULAR INTERPOLATION - CENTER Y COORD
K	$\pm\,43$	CIRCULAR INTERPOLATION -. CENTER Z COORD

ADDRESS	FORMAT	

(TABLE 4.2 CONTD.)

L	±43	SAFETY DISTANCE FOR RAPID TRAVERSE APPROACH AT CANNED CYCLES
	±43	DATA INPUT FOR TOOL CORRECTION (G55,G57)
	±43	PARAMETER DATA INPUT (G55, G57, G58)
	±5	CONTENT OF MACHINE DATA REGISTER
	±5	DATA FOR MECHANICAL ERROR COMPENSATION
	4	NUMBER OF REPETITIONS OF SUBROUTINE EXECUTION
M	2	MISCELLANEOUS FUNCTIONS
N	4	SEQUENCE NUMBER
O	4	SEQ. NUMBER OF MAJOR STATEMENT (IN EIA CODE ONLY)
P	±16	COEFFICIENT OF TOOL RADIUS COMP.-X
Q	±16	COEFFICIENT OF TOOL RADIUS COMP.-Y
R	±16	COEFFICIENT OF TOOL RADIUS Comp.-Z
S	2	SPINDLE SPEED CODE
	4	SPINDLE SPEED (REV/MIN)
T	6	TOOL FUNCTION/MAGAZINE NO.AND CORRECTION
U	±43	SECONDARY LINEAR MOTION PARALLEL TO X
	±16	COEFFICIENT OF TOOL LENGTH-X
V	±43	SECONDARY LINEAR MOTION-P.TO Y
	±16	COEFFICIENT OF TOOL LENGTH-Y
W	±43	SECONDARY LINEAR MOTION-P.TO-Z
	±16	COEFFICIENT OF TOOL LENGTH-Z
X	±43	LINEAR MOTION
Y	±43	LINEAR MOTION
Z	±43	LINEAR MOTION

TABLE 4.3

```
%

N1 T010102                          TOOL ID
N2 S800 M40 M03                     SPINDLE SPEED,RANGE,CLW
N3 G90 G00 X0 Y0 Z20000 M08 M36     POSITIONING
-----------------------------------------------------------------
N4 G57 E15 L#20 G73                 1   NUMBER OF HOLES
                                        IN X DIRECTION
-----------------------------------------------------------------
N5 G73 G81 G90 Z5000 L20000         2   CANNED CYCLE
-----------------------------------------------------------------
N6 G58 E15 L-1                      3   MODIFICATION OF CYCLE
                                        'VARIABLE' :
                                        DECREASING NOMBER OF
                                        HOLES IN X DIR.
-----------------------------------------------------------------
N7 G77 E10 L#15                     4   CONDITIONAL JUMP
-----------------------------------------------------------------
N8 G91 G00 X#10                     5   STEP IN X DIR. WITH
                                        'A' SIZE
-----------------------------------------------------------------
N9 G75 E5                           10  UNCONDITIONAL JUMP
-----------------------------------------------------------------
N10 G73 G58 E16 L-1                 6   MODIFICATION OF CYCLE
                                        VARIABLE. DECREASING
                                        NUMB.OF HOLES IN Y DIR
-----------------------------------------------------------------
N11 G77 E15 L#16                    7   CONDITIONAL JUMP
-----------------------------------------------------------------
N12 G91 G00 Y#11                    8   Y COORDINATE OF THE
                                        NEW LINE OF HOLES
-----------------------------------------------------------------
N13 G90 X0                          9   X COORDINATE OF THE 1ST
                                        HOLE
-----------------------------------------------------------------
N14 G75 E4                          11  UNCONDITIONAL JUMP
-----------------------------------------------------------------
N15 G90 G00 Z#21                    12  RETRACT

N16 M30
```

5   COORDINATE SYSTEMS, TOOL CORRECTION TECHNIQUES

5.1   Coordinate Systems

The machine-tool coordinate system is a fixed coordinate system of the machine. The origin usually is beyond the range of the movements. The origin at the workpiece coordinate system is defined in the machine-tool coordinate system by a zero-shift.

The reference point is located in the range of the possible movements. Reaching the reference point the measuring system of the machine will have the coordinates of this point (the coordinate are those of the machine coordinate system).
(There are machines/control units where the design of the machine-tool coordinate system and the reference point are the same points).

In some cases more than one reference points exist, eg;
-   1st reference point for setting automatic reference point of the machine-tool coordinate system,
-   2nd reference point for the position of the automatic tool change,
-   3rd reference point for the position of palet changing etc.

The origin of the workpiece - coordinate system can be fixed to any of the points of the workpiece depending on the shape of the workpiece, symmetry, properties etc. The directions of axes are the same as of the machine system if the tool is executing the movement in the specific direction in question.

The CNC control units offer more than one possible workpiece coordinate system, which can be selected by commands of the NC part program.

(eg. UNIMERIC 700 CNC - 8 workpiece coordinate systems
      MITSUBISH MELDAS m0 - 6 workpiece coordinate
                              systems
      FANUC 0 - MB   - 6 workpiece coordinate systems)

The very economical use of the automatically programmable workpiece coordinate-systems are eg:

-   the production of more identical workpieces in the same time with the same NC part program - each of the workpiece to be produced will get different zero point shifts.
-   manufacturing a workpiece with rotary table shifts - each side of the workpiece will get a different zero point shift.

The coordinate systems and the connections between them are illustrated on Fig. 5.1.

The control unit calculates the coordinates of the con-
trolled point of the tool as follows.

$$X_{TO} = X_O + X_P + \Delta X \qquad\qquad \Delta X = P * r_T$$

$$Y_{TO} = Y_O + Y_P + \Delta Y \qquad\qquad \Delta Y = Q * r_t$$

$$Y_{TO} = Z_O + Z_P + L + (\Delta Z) \qquad (\Delta Z = R * r_t)$$

In the part program the coordinates of P or T are
programmed (in case of T coordinates no Δx and Δy are
needed).

The tool-coordinate system has its orgin at the con-
trolled point of the tool, directions are of those, that
increasing tool dimensions show into positve axes
directions.
Examples for lathes are given on Fig 5.2, the total
coordinate-systems complex is shown in Fig. 5.3.

5.2 Programming tool movements on milling machines

The connection between the controlled tool point (To)
and the programmed tool point (P) was shown on Fig 5.1.
In the NC part program the path of the To point is not
used because a number of inconveniences and difficulties
exist.

- The coordinates of NC programs are not of those can
  be read on the part drawings.
- the calculation of the path needs the tool data.
- dificulties of program checking and modification etc.

The most convenient method is the programming of the P
point, the path of To will be calculated by the control
unit.

There are three ways for considering the tool diameter:

a. the path of T point is programmed (programming of
   tool 'center points')
b. programming of tool with 1 mm radius (see Fig 5.4).
   The P,Q and R correction vectors give the position of
   T point relative to point P in case of R_t = 1 mm;
   some cases shown on Fig 5.5.
.. path (contour) generating, where the part contour is
   programmed and the path of point T is calculated by
   the control unit. In this case the X_p, Y_p, Z_p coor-
   dinates and also the tool position relative to the
   contour (Right/Left) should be programmed.

There are three basic types of corrections at contour
generating.

a. G41 or 42 for the offset depending on the side, no
   circles are generated automatically at the corners,

these should be programmed separately with G39 codes.

b. G41 or G42 for the offset, circles at the corners are generated automatically.

c. G41 or G42 for the offset with no circles (G39's are ignored), but where it is necessary 3 linear movements will be generated automatically.

When programming contour generation, care should be taken:
- on the execution of the first (start-up) movement (examples for the MITSUBISHI MELDAS CNC control unit is shown on Fig 5.6)
- on the movement strategy used at the corner points. Especially when the contouring is executed on the inner side many possibilities exist to violate the contour (using correction method b, see examples for the MITSUBISHI unit on Fig 5.7)

Tool length compensation is a more simple case at milling machines. Basically three methods are used for the elimination of the different length of the tools (Fig 5.8).

a. selecting a tool ('reference tool') and the relative length differences are used for the other tools.

b. all of the tools are compensated with their total lengths (this should be a preffered alternative to a.)
and again:

c. compensating of tool length of 1 mm (this case means that all the tools are considered with length of 1mm and U,V,W components are calculated as are the P,Q,R,'s at radius correction. The actual tool lengths are corrected when executing the NC program).

The relative simplicity of tool length correction should be understood up to 3D cases. In 5D manufacturing where the tool is able to tilt and the momentary tool axis is not parallel with the Z -axis, the situation is more complicated. Tool axis compenents should be calculated (as V,V,W's in method C.) in order to get the actual momentary tool length (in Z direction).

5.3 Tool compensation on lathes

As it was shown on Fig 5.2-5.3 the coordinate systems on lathes are a little more complex. The basic idea of compesation technique is the same, but some special featu es exist for turning.
The most characteristic difference is in the profile-distorting effect of the tool-tip radius (see Fig 5.9).
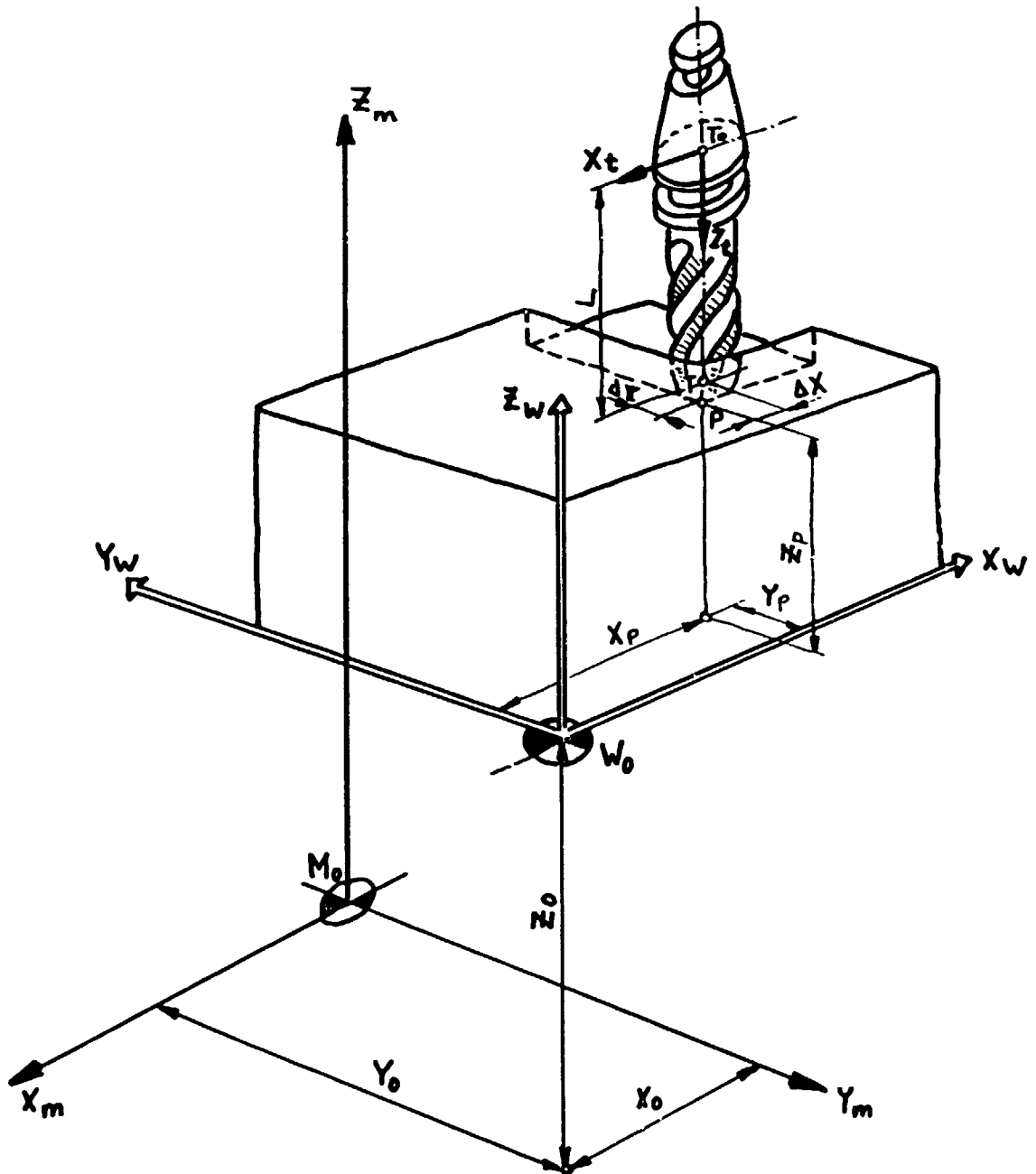
P = programmed point (contour point)

T = programmed point of the tool

T_o = controlled point of the tool (origin of tool coordinate system)

W_o = origin of workpiece coordinate system

M_o = origin of machine tool coordinate system
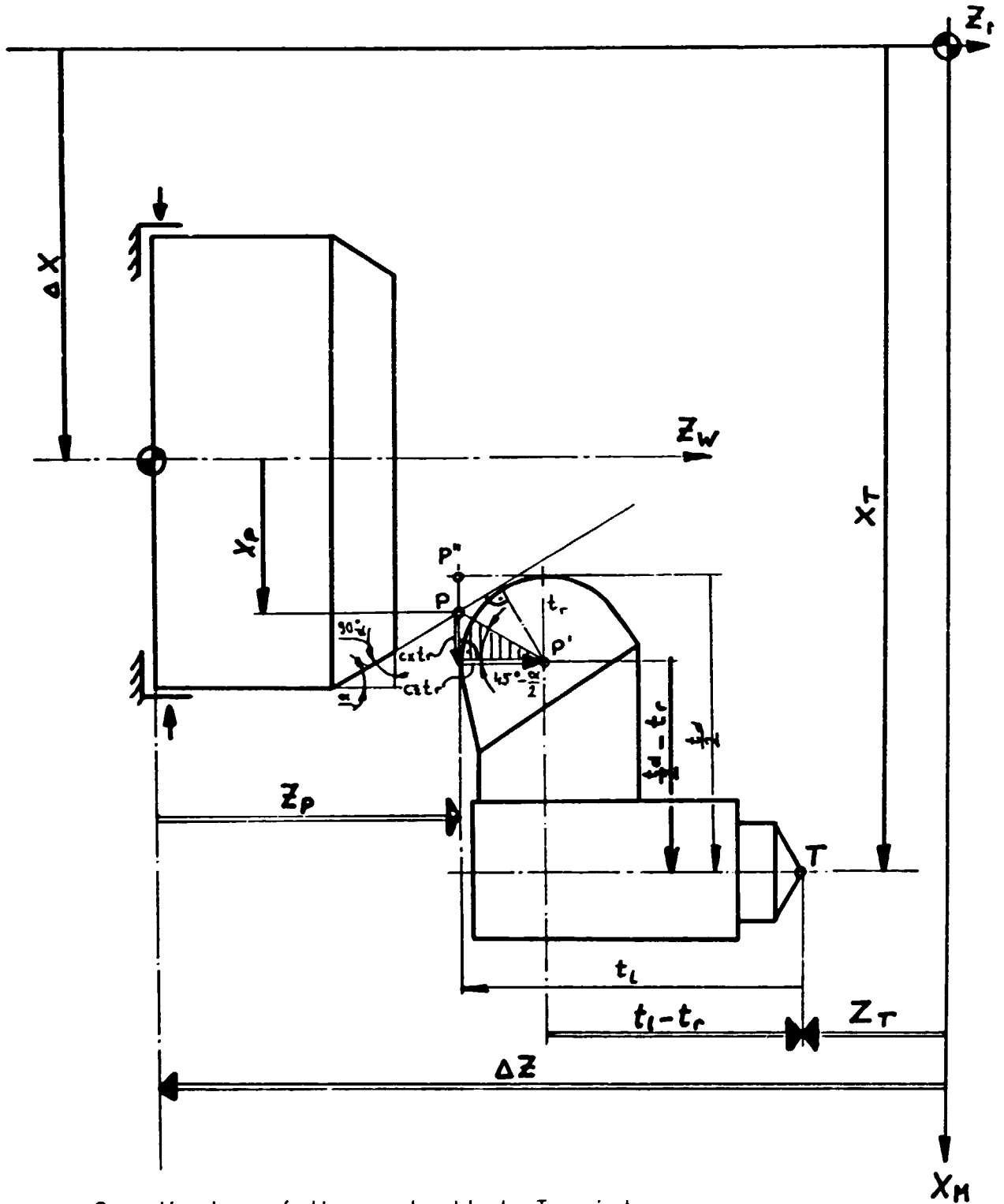
Fig.5.1   COORDINATE SYSTEMS
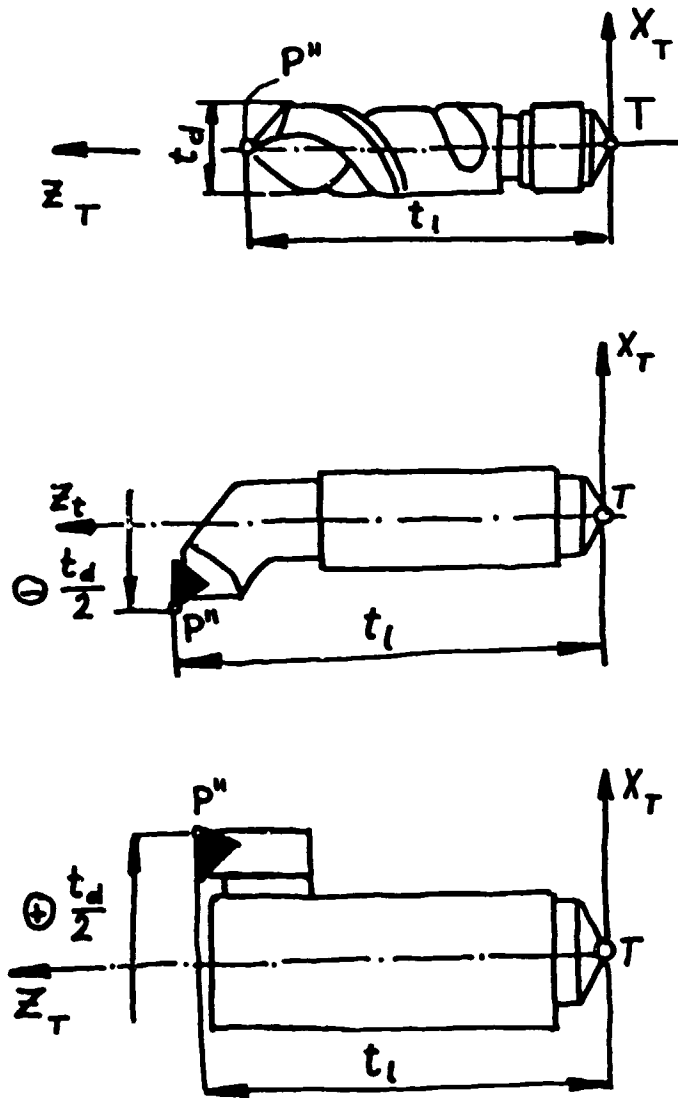
T = controlled tool point, origin of tool
    coordinate system

t_d = diameter of tool (drills) or tool extension
    multiplied by two

P" = point of tool tip

t_l = tool lengtn

Fig.5.2 TOOL COORDINATES FOR CNC LATHE TOOLS

Coordinates of the controlled T point:

$$X_T = \Delta X + X_p + \left[\frac{t_d}{2} - t_r \right] + c_x \times t_r \quad ;$$

$$Z_s = -\Delta Z + Z_p + (t_l - t_r) + c_z \times t_r$$

Fig.5.3 CONTROLLED COORDINATES FOR LATHES

$$R_t = \frac{D}{2}$$

$$\Delta x = P \cdot R_t$$

$$\Delta y = Q \cdot R_r$$

$$x_{T_o} = X_p \cdot \Delta x$$

$$y_{T_o} = Y_p \cdot \Delta y$$

$$Z_{T_o} = Z_o \cdot L$$
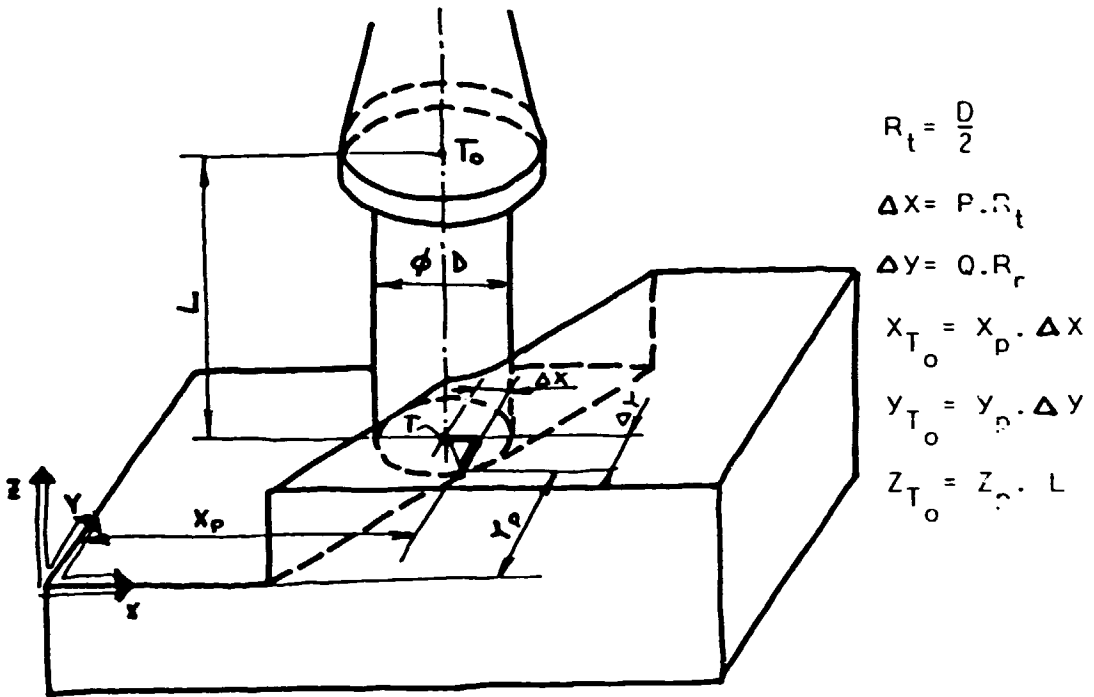
Fig.5.4   RADIUS CORRECTION OF 1 MM
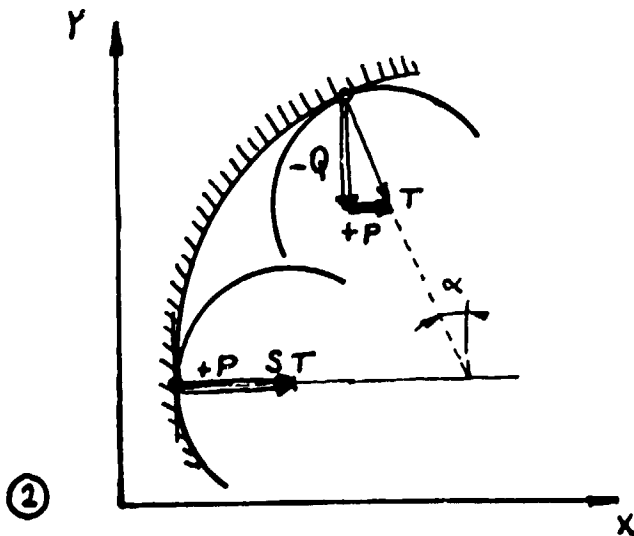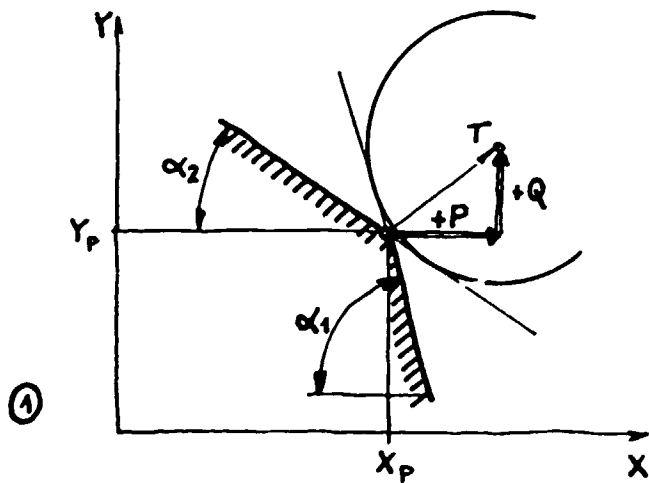

Fig.5.5   P-Q   CORRECTION-VECTORS (EXAMPLES)

Fig. 5.6

CONTOUR GENERATING (CUTTING OUTSIDE)

CONTOUR GENERATING (CUTTING INSIDE)

line-line

line-arc

arc-line

arc-arc

$0° \leq \varphi \leq 0,6°$

$179° \leq \varphi \leq 180°$
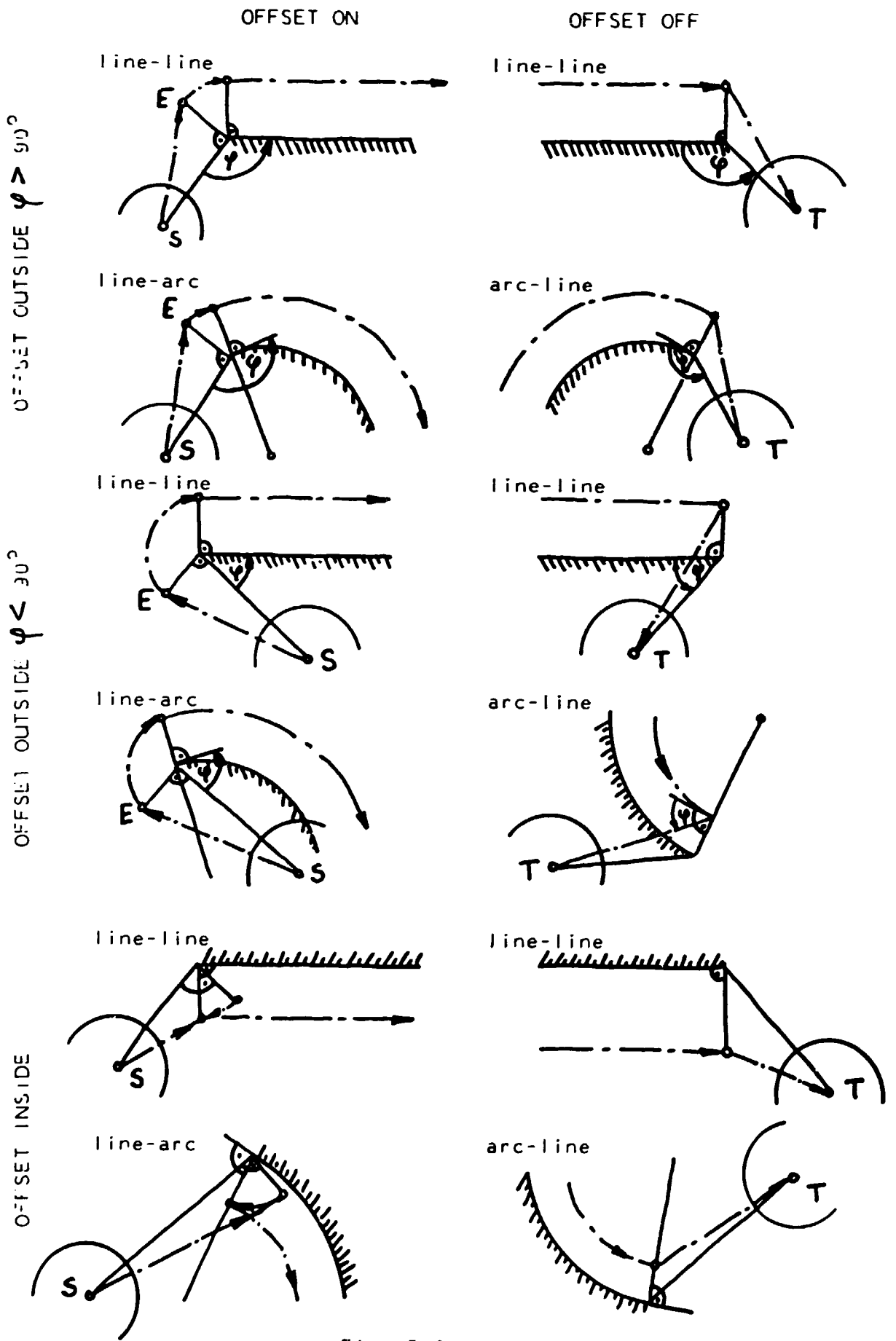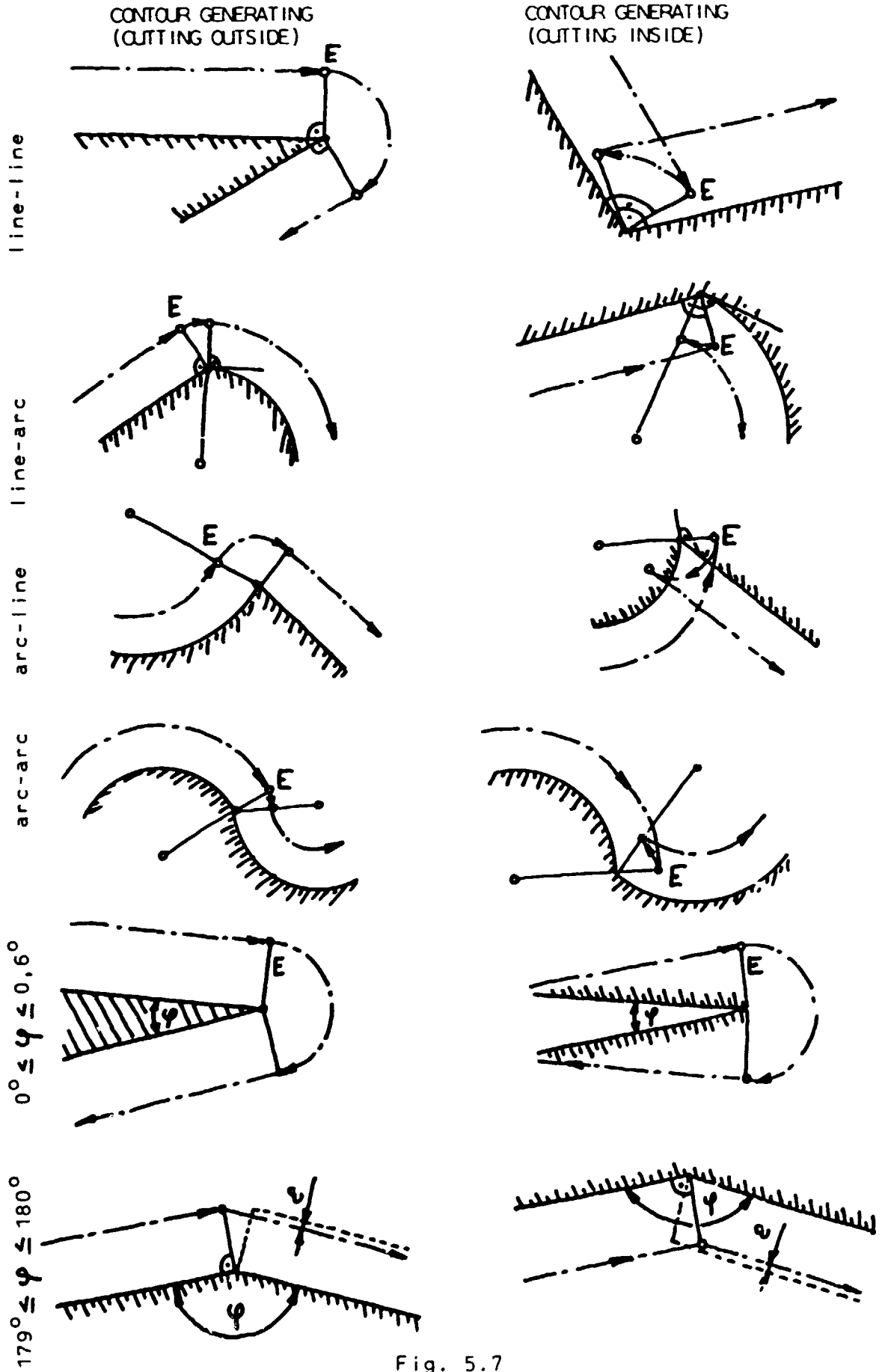
Fig. 5.7

Fig.5.8   TOOL LENGTH COMPENSATION

R_t = radius of tool tip

P_1, P_2 = corner points of profile

P'' = programmed tool edge

P' = center point of tip radius

Fig.5.9   PROFILE ERROR OF TURNING
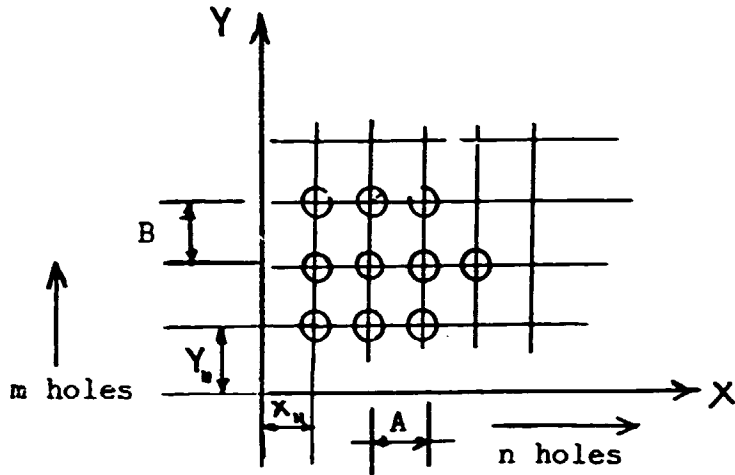
Fig 6.2

Variables

```
#500    A
#501    B
#502    m        .....    #102
#503    n        .....    #103
#504    X11      .....    #104
#505    Y11      .....    #105
```
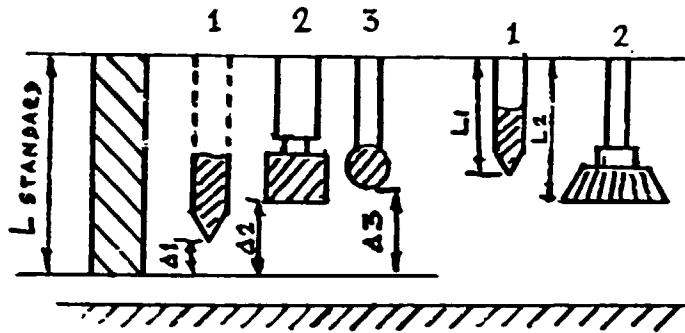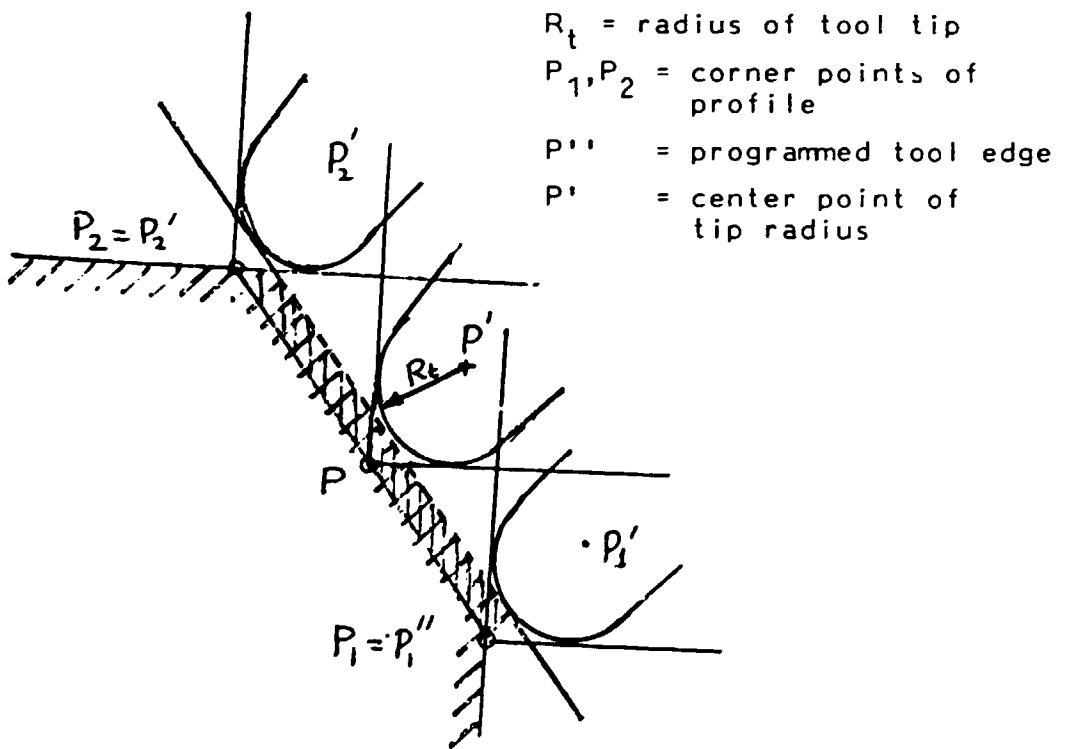
```
G65   H01   P#102   Q#502 ⎤
G65   H01   P#103   Q#503 ⎬   input data definition
G65   H01   P#104   Q#504 ⎮
G65   H01   P#105   Q#505 ⎦
```

```
N100 G65 H81 P*300 Q=103 R0          IF N.EQ=C
                                     GOTO 300
```

control statements (blocks) for drilling cycle.

```
G65 H03 P#103 Q#103 R1               (N=N-1)
G65 H02 P#104 Q#104 R#504            (X=X+A)
G65 H80 P#100                        GOTO 100
```

```
N300    G65 H81 P#500 Q#102 R0       (IF M.EQ=0 GOTO 500)
        G65 H01 P#103 Q#503          ( N = Nstart)
        G65 H01 P#104 Q#504          ( X = X11)
        G65 H02 P#105 Q#105 R#501    ( Y = Y+B)
        G65 H80 P100                 (GOTO 100)
```

N500   end of cycle   in macro body.

6. ADVANCED MANUAL PROGRAMMING. SUBROUTINES AND MACROS - 1. (FANUC-0-MB)

6.1 SUBPROGRAMS

NC Programs ⟵——— Main program

⟍Subroutines (subprogramme)

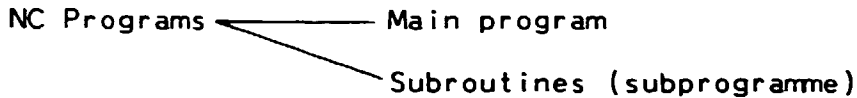The use of subroutines or subprograms is especially useful when the same pattern or a series of fixed NC command blocks frequently occurs in the NC part program.

The main program calls a subprogram and the subprogram is also able to activate other subprograms (see Fig 6.1)

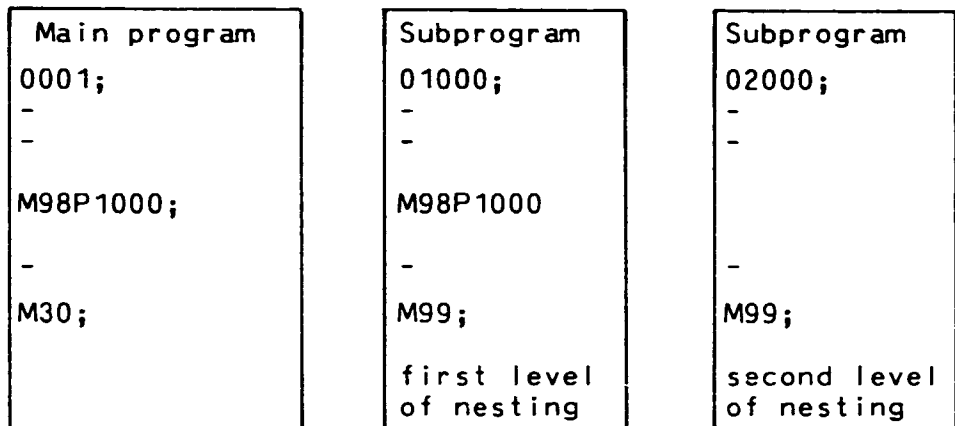| Main program	Subprogram	Subprogram
0001;	01000;	02000;
-	-	-
-	-	-
M98P1000;	M98P1000	
-	-	-
M30;	M99;	M99;
	first level of nesting	second level of nesting

Fig 6.1

The control unit can store up to 125 programs and subprograms in the memory.

Subprogram calls are activated with the M99 control word.

The call looks like as follows.

M98 P x x x x x x x x
       └——ID number of subprogram
       └————number of repetitions
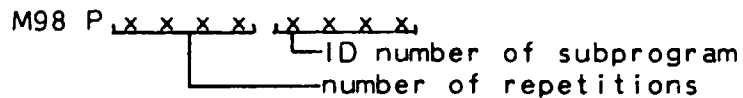
Subprogram return is activated with the M99 code. In this case the NC program execution in the calling main program (or subprogram) will be continued at the next control block.

Special cases:

- if the last block of the subprogram has an adress with P code, the execution in the calling program (subprogram) will be continued at the address which is specified with the P code.

- if M99 is performed in the main program the consequences are:
  x control returns to start of main program (usually this feature is used with /M99 ie

> > optional block skip – to ensure the termination of the loop on request.
> > * if /M99 Pn is programmed, the return address is not the main program start but the n adress.
> - subprogram execution from the start is possible as it were a main program by specifying a search from MDI keyboard.
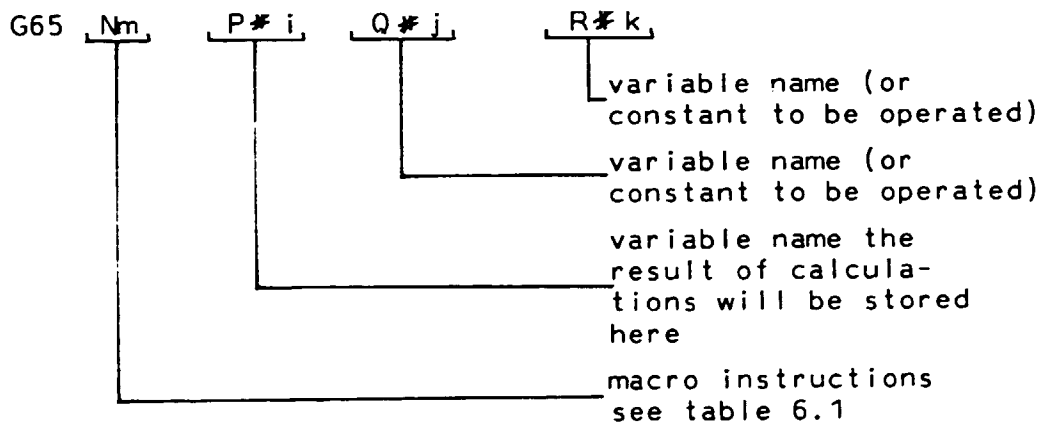
The level of acceptable subprogram nesting is 4.

6.2 CUSTOM MACRO A.

Macros behave like subprograms but more flexible application is possible.'Macro instruction'means the macro call,'Macro body' means the macro itself. The special features of macros are as follows.

- variables can be used
- operations can be performed on the variable
- in the macro instructions (calls) actual values can be assigned to the variables.

Macro instructions

G65 Nm, P#i, Q#j, R#k,

- variable name (or constant to be operated)
- variable name (or constant to be operated)
- variable name the result of calculations will be stored here
- macro instructions see table 6.1

G65	H01	Definitions, substitution	$\#i = \#j$
G65	H02	Addition	$\#i = \#j + \#k$
G65	H03	Subtraction	$\#i = \#j - \#k$
G65	H04	Product	$\#i = \#j \times \#k$
G65	H05	Division	$\#i = \#j \div \#k$
G65	H11	Logical sum	$\#i = \#j \ .OR. \ \#k$
G65	H12	Logical product	$\#i = \#j \ .AND. \ \#k$
G65	H13	Exclusive OR	$\#i = \#j \ .XOR. \ \#k$
G65	H21	Square Root	$\#i = \sqrt{\#j}$
G65	H22	Absolute Value	$\#i = |\#j|$

G65	H34	Arctangent	i = ATAN (j/k)
G65	H80	Unconditional divergence	GOTO n
G65	H81	Conditional divergence 1	IFj = k, GOTO n
G65	H82	Conditional divergence 2	IFi $\neq$ k, GOTO n
G65	H83	Conditional divergence 3	IFj > k, GOTO n
G65	H84	Conditional divergence 4	IFj < k, GOTO n
G65	H85	Conditional divergence 5	IFj >= k, GOTO n
G65	H86	Conditional divergence 6	IFj <= k, GOTO n
G65	H99	P/S alarm condition	P/S alarm number 500 + n

Table   6.1

There are two types of variables which occur at the FANUC unit. These are

- Common variables   #100 – #149
and #500 – #531

These are common variables because #i   is the same in the main program and in the macros. The difference between the #1xx and #5xx type variables is, that the #1xx variables are cleared when the power is turned off and reset whit 0 (zero) with power on. The #5xx variables are independent of power turn on/off and they are unchanged.

- System variables – meaning, that their application is fixed.
(eg tool offset amount #1   – #99
#2000   – #2200)

Macro calls

Single call with
M98 P⌐⌐ ; program number of called macro

Modal macro call can be activated with
G66 P⌐⌐ ;
program number of called macro.

This modal call means, that the macro will be called in each subsequent block, as long as a cancel command will not be encountered.

The cancel of modal call is programmed with G67 code.

For the illustration of the practical use of the macros the sample part with variable nxm holes (see chapter 4) will be programmed with FANUC codes. The basic data used is illustrated on Fig 6.2.

7 ADVANCED MANUAL PROGRAMMING II.
MACROS - II (FANUC - 0-MB). FUNCTIONS AND EXTENDED FORMAT.

7.1 CUSTOM MACRO - B

Macro call in Chapter 6 was activated with an M98 P_
block what was used also for subroutine call.
If actual parameter values are needed for the macro ex-
ecutions the call has to be activated with:
G65 P(prog.no)L(iteration times) ⟨argument assignment⟩ ;
Argument assignment can be handed by two ways:

- Method I.
  With addresses: A_B_C_D_..Z_ . All of the addresses
  can be used (except G,L,N,O and P)
  Alphabatical order is necessary only for I,J and K.
  For illustration the macro variable #2 has to be
  10 and variable #5 has to get 6 as actual value.
  In this case the macro call is:
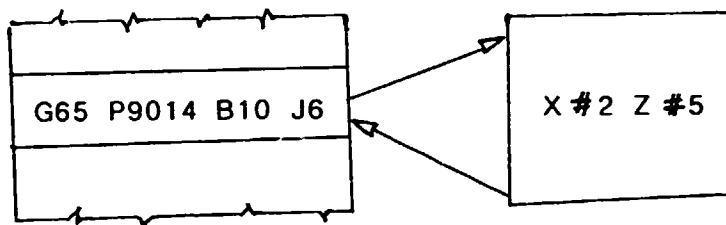
The reason of using B__ and J__ addresses is the
connection between argument assignment and the
variables of the macro body as it is shown in Table
7.1.

| Address	Variable in custom macro body
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19
.	.
.	.
Z	#26

Table 7.1

- Method II.

In this case arguments can be assigned to A_,B_ and C_ and maximum ten arguments can be assigned to variables I,J and K as it is shown on Table 7.2.

ADDRESS	VARIABLE
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
.	.
.	.
I_{10}	#31
J_{10}	#32
K_{10}	#33

Table 7.2

If Method I and Method II are mixed, in case of the same variable that argument is effective which was specified later, eg:

G65 A1.0 B2.0 I-3.0 I4.0 D5.0 P1000

〈 Variable〉
#1: 1.0
#2: 2.0
#3: -
#4:-3.0
　　　　　4.0　　5.0
#7: 5.0
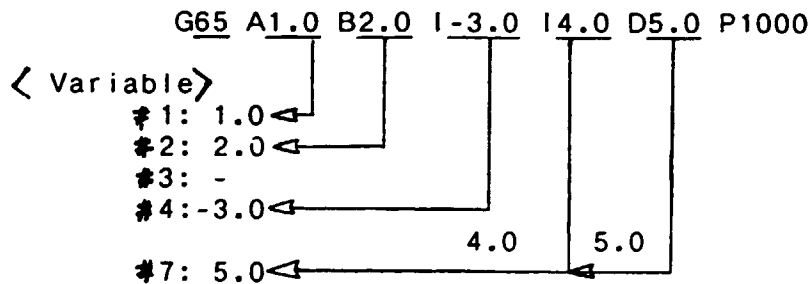
Modal call of macros is executed with

G66 P program number L repetitive count 〈argument assignment〉 ;
The cancel of modal call is with G67 code

Multiplex calls are allowed up to level four (ie. this is the macro call nesting). The multiplex modal effects that the specified macro is called each time a motion command is executed. If more than one modal macros are specified, the next macro is called each time a motion command in the first macro is executed.

The M98 and G65 calls

- Only G65 call can have arguments

- The level of local variables is changed if G65 is used, no change is when using M98 (ie. variable #5 specified before M65 call is dirferent that of after the M65 call)

- Up to four G65 and G66 calls can be activated

- The block with M98 has an execution stop if other addresses are included than O,N,P and L.
  G65 is not affected this way.

Variables are of three types

- Local variables (#1- #33) are locally used in the macro, basically for argument transfer. Locality means that if a variable is used in macro B what is called from macro A the value in macro A will not be lost.
- Common variables #100- #199, #500- #531   These behave as real common variables, ie. their value is the same for all macros.
- System variables with fixed applications (tool-compensation, macro-alarm, cycle operation control, clock .nformation etc.)
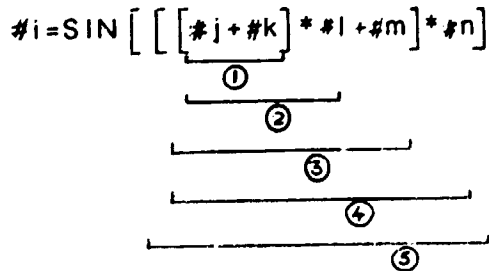
Arithmetic commands

Numerous arithmetic operations can be performed on variables.

$$\#i = \langle formula \rangle$$

ec.

#i = #j	definition
#i = #j + #k	sum
#i = #j * #k	product
#i = SIN [#j]	sine(in degree)
#i = COS [#j]	cosine (in degree)
#i = ROUND [#j]	rounding off
etc.	

The arithmetic operations can be combined and nested up to level five. Example on a three fold nesting:

$$\#i = SIN \left[\left[\left[\#j + \#k \right] * \#l + \#m \right] * \#n \right]$$

① ② ③ ④ ⑤

Control commands

The NC program execution can be influenced by
- divergence (GOTO)
- iteration

commands.

Divergence has the format:

IF [⟨conditional expression⟩] GOTO u

The n sequence number can be replaced by a variable or by a ⟨Formula⟩

The iteration has the following form:

WHILE [⟨conditional expression⟩] DO m (m=1,2,3,...)
...
...
...
END m

If the conditional expression is not satisfied, the loop will not be executed.

7.2 EXTENDED FORMAT AND DATA STRUCTURE

The trend in numerical control is going towards full automation. Therefore an increasingly flexible programming is desired both in individual machines as well as in machine cells or flexible manufacturing cells.

The extended format has the necessary functions for machine monitory, measuring cycles, subroutines and parameter programming.

The basic conception for the extended data format is shown on Fig. 7.1

According to ISO 6983 integration of the extended data format in the NC format is initiated by the symbol '[' and is terminated by the symbol ']'.

Program example:

```
     %
[DATA
   LOCAL REAL:XPOS:YPOS
  END DATA;]
N100 G00 X100 F1000
N110 G81 Z100 R400 F500
[     XPOS:=100; YPOS:=110
      WHILE XPOS =1000 DO
             XPOS:=XPOS+100;]
             N120X[XPOS] Y [YPOS]
[     ENDWHILE;]
N200 M30
```
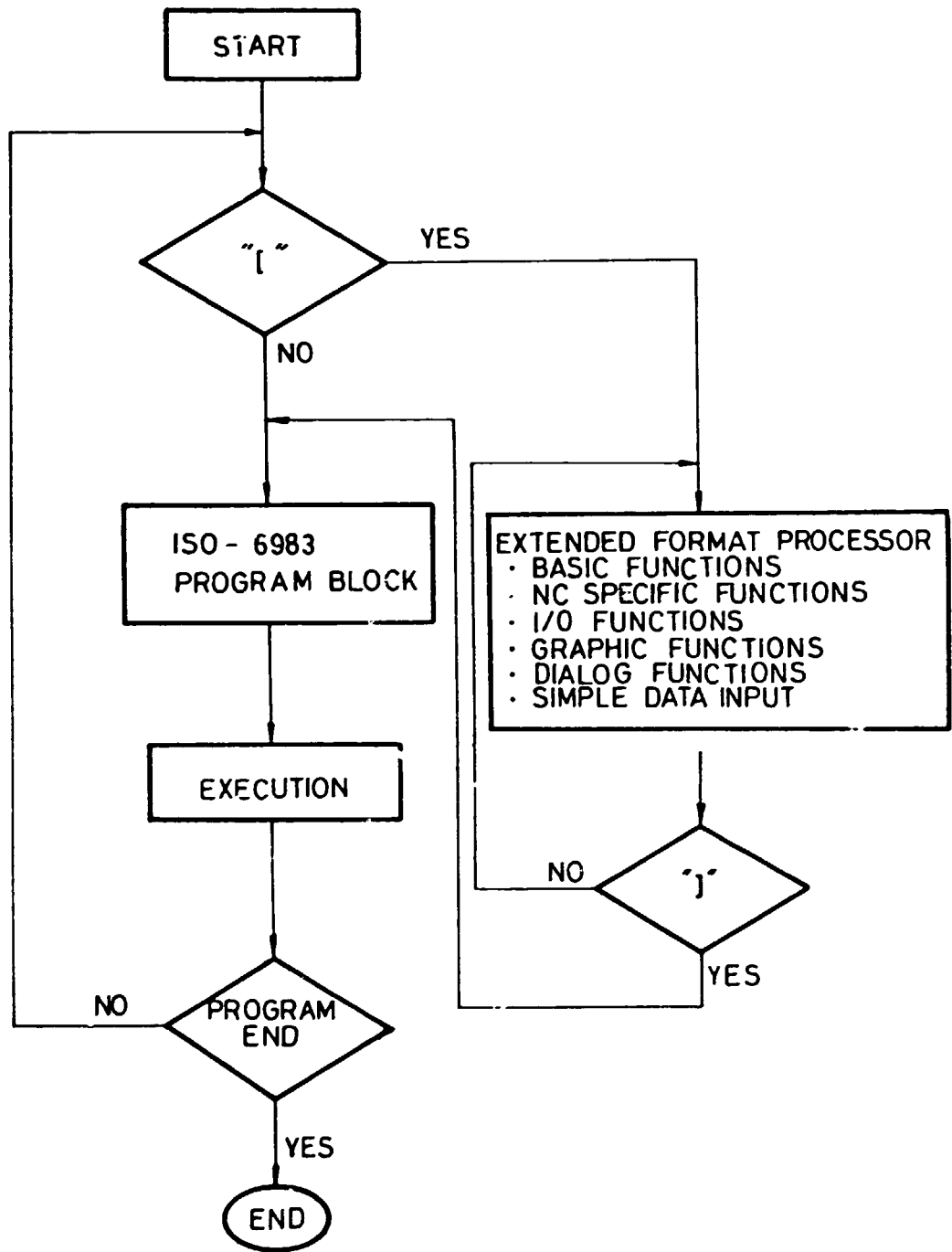
Fig 7.1

The program structure very briefly is as follows:

- declaration part (external for procedures, programs and functions which were declared in other external programs, data declaration with COMMON, GLOBAL and LOCAL variables)

- procedure and function part

The standard functions, procedures and operations are about the same as with FANUC control unit.

This field of the NC programming technology is under international standardization procedure, details are discussed in ISO/DIS 6132.

8. PRODUCTION PLANNING AND DOCUMENTATION FOR NC/CNC MANUFACTURING

This problem is one of the very basic questions: uncompleted or false documents result faulty product and many times huge financial losses. Just coding the NC blocks, editing and modifying the programs with no rigid procedural rules promotes the total chaos in manufacturing.

Just briefly discussing the topics focusing primarily on manual NC programming the following conclusions can be drawn:

a) - all the usual steps of the production process planning should precede the NC coding procedure;

b) - seperate files should be opened for the NC/CNC documentation for each workpiece;

c) - list of NC program is needed for each set-up with:

- machine-tool and control unit name
- ID of work piece
- operation number (set-up number) of machining
- data of program preparation
- programmer's name

d) - the plan of tooling has to be prepared:

- number of utilised magazine sequences;
- type of tool;
- type and quality of the cutting edge or insert;
- data on chip breaking;
- characteristic data of the tool
- correction switches (registers) used for the tool

(The desirable goal is: one plan of tooling could be applied for more set-ups)

e) - list of manual operations for the machine operator:

- all those operations are listed which should be executed before starting the machining;
- the equipments to be used for these operations also should be listed;

(e.g. - coordinates of zero point shift

- type of chuck to be used

- data of corrections etc.)

f) - list of <u>operation instructions</u> (in the same form as it is
used in conventional production. See Fig. 8.1)

Seq. No.	Operations	ID of Surface	Tool/meas. equipment	Cutting speed m/min	n 1/min	feeds mm/rev	depth of cut	No.of cuts
4	Roughing of hole ⌀ 77	4	STB25 boring tool	120	710	0.30	5	4

g) - change of technology:

Separate sheet is needed with

- Serial number of change

- reasons of change

- date

- name of the person who asked for change

9. **MORE ON COMPUTERAIDED PART PROGRAMMING. FUTURE TRENDS**

9.1 SCULPTURED SURFACE PROGRAMMING

Originating from the APT system and using many other
suggestions the sculptured surface language is now comes to
be at the end of an international standardization procedure.

There are only six major words used but a very wide variety
of minor words and modifiers make the input language flexible.
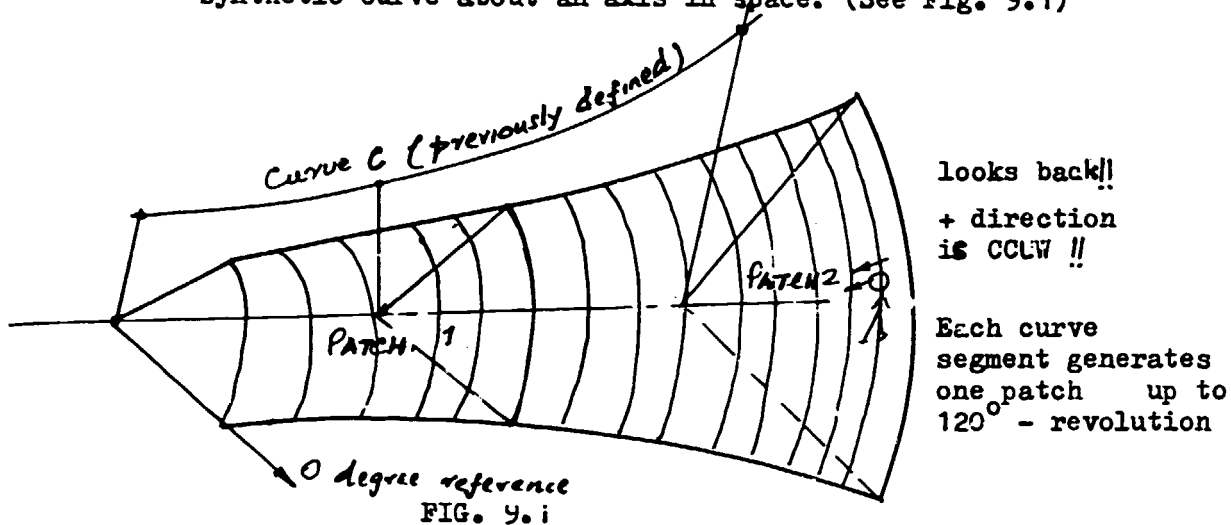
The major words are:

POINT   - point definitions for a point on a synthetic
                curve or on a sculptured surface

SCON   - specification of regional milling conditions
- part surface parameters
- drive control parameters
- tool axis orientation
- feedrate selection
- stopover control (between passes or'lifts')
- clearance plane specification

SCURV   - synthetic curve defintion

SMIL   - used for regional milling tool control
- tool positioning
- generating a single tool path
- area clearance

SSURF   - sculptured surface definition

VECTOR   - for the definition of vectors with respect to
                existing synthetic curves and sculptured surfaces

9.1.1. SURFACE DEFINITIONS

Just for example let us see in details the possible
sculptured surface definition

a) $S1 = SSURF/REVOLV, C, AXIS \begin{bmatrix} P1,V \\ P1,P2 \end{bmatrix} \begin{bmatrix} CCLW \\ CLW \end{bmatrix}, \quad s1,s2$

This forms a surface of revolution by rotating a
synthetic curve about an axis in space. (See Fig. 9.1)

FIG. 9.1

Curve C (previously defined)

looks back‼

+ direction
is CCLW ‼

Each curve
segment generates
one patch   up to
120° - revolution

PATCH 1

PATCH 2

O degree reference

the <u>axis</u> is a directed space line from the point P1 to P2

<u>s1, s2</u> the starting and terminating angle of revolution
(0, 360 is accepted for a full rotational surface)

b) S = SSURF/RULED,C,AXIS $\begin{bmatrix} V \\ P1,P2 \end{bmatrix}$ is a simple cylindrical
ruled surface

(See Fig. 9.2)

C - previously defined synthetic curve

FIG. 9.2

C - the patches are
centered on this
curve!

V is the cross
spline direction
(or P1 to P2 vector)

V is the cross
spline direction
(or P1 to P2 vector)
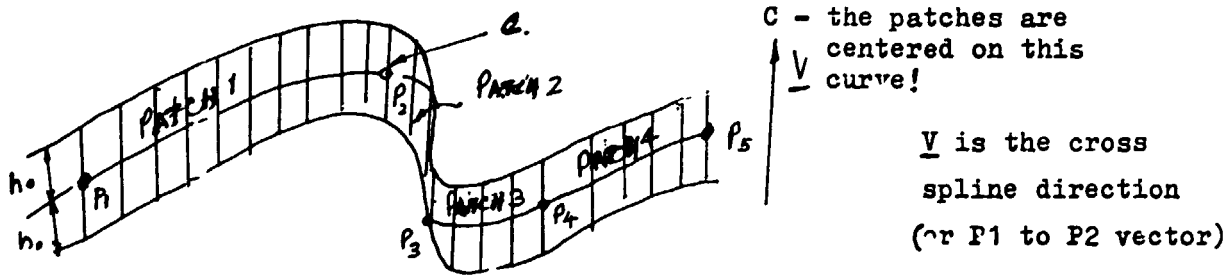
c) S1 = SSURF/SMESM, XYZ, SPLINE,P1,P2,P3,P4,P5,TANSPL,V5,$
SPLINE,Q1,Q2,Q3,Q4,Q5,$
SPLINE,R1,R2,R3,R4,R5,$
SPLINE,S1,CRSSPL,VC,S2,S3,S4,$
NORMAL,VN,S5

Rectangularly ordered set of N by M points with
coordinates (i.e. identifiers of points which were
defined by their coordinate values), see Fig. 9.3

Parametric bicubic patches are calculated to fill the
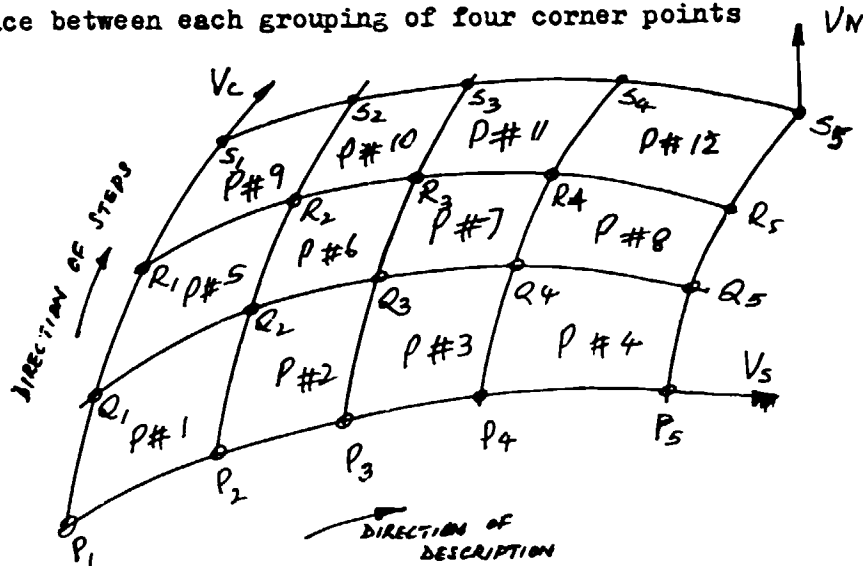space between each grouping of four corner points

FIG. 9.3

d) PATCH modifier (minor word) indicates, that a single
bicubic parametric patch is defined with specified order
of 16 points and/or vectors.

Three modifiers exist:

PNTSON - the points specified will be included in
a ticubic surface interpolated to fit those
sixteen points

PNTVEC - the points specify patch curves, the vectors
specify tangent and cross-spline direction
vectors and corner twist vectors

POLYGN - the points represent corners and Bezier control
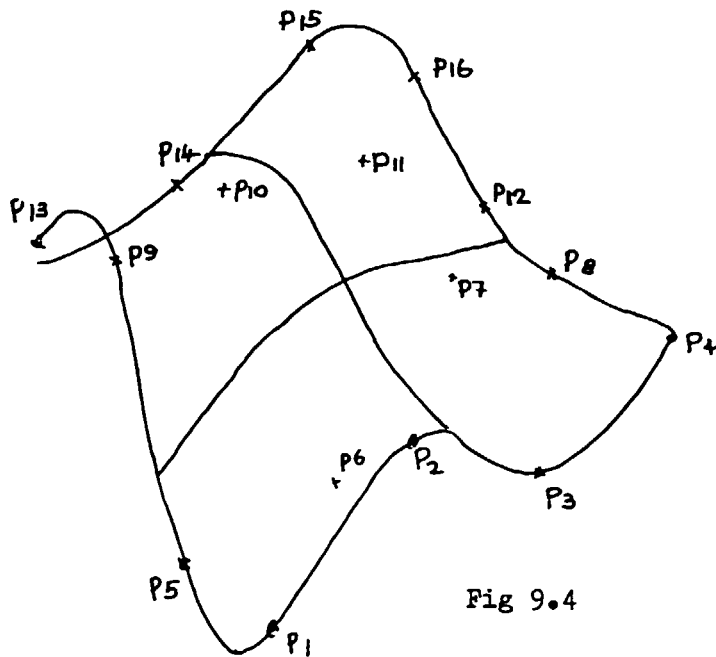points of a polygn defining a bicubic Bezier
surface

PNTSON eg:

Fig 9.4

Patch boundary curves are cubic curves which interpolate the

four sets of points (see Fig 9.4).

$$P_1, \ P_2, \ P_3, \ P_4$$
$$P_1, \ P_5, \ P_9, \ P_{13}$$
$$P_4, \ P_8, \ P_{12}, P_{16}$$
$$P_{13}, P_{14}, P_{15}, P_{16}$$

internal input points (P6, P7, P10, P11) control the twist

vectors at the patch corners and hence the internal shape of

the patch.

$$\text{SSURF/PATCH,PNTSON} \begin{bmatrix} \text{PLUS} \\ \text{,MINUS} \end{bmatrix}, \ P1, \ P2, \ P3, \ P4, \ \ldots, \ P15, \ P16$$

9.1.2 TECHNOLOGICAL DEFINITIONS

The regional milling conditions are controlled with the

parameters of the SCON major word.

The tool axis can be either fixed or variable. If the tool

axis vectors are needed, then MULTAX must be programmed (see

CLDATA structure 5000 type records with sextet coordinates -

Chapter 2).

Fixed tool axis:

SCON/AXIS, vector

meaning, that the tool axis will be parallel to a defined
vector.

Variable tool axis:

$$\text{SCON/AXIS, NORMDS } \begin{bmatrix} \text{PLUS} \\ \text{,MINUS} \end{bmatrix}.$$

which case results a tool axis with maintained position (i.e.
parallel to the normal vector).

The drive control for a sculptured surface:

e.g.

$$\text{SCON/DS, S, PARAM, u1, u2, v1, v2} \begin{bmatrix} \text{AT} \\ \text{,ON} \end{bmatrix} \begin{bmatrix} \text{NORMAL} \\ \text{,vector} \end{bmatrix}$$

Here S is the ID of a previously defined sculptured surface.
The v1 and v2 are the parametric values in the major direction
on the surface (representing lower and upper extents); v1 and
v2 are the parametric values in minor direction. The vector
represents a fixed tool projection vector.

The part surface information for regional milling is defined:

$$\text{SCON/PS } \begin{matrix} \text{TO} \\ \text{,ON} \end{matrix} \text{ , S} \begin{bmatrix} \text{PLUS} \\ \text{,MINUS} \end{bmatrix} \text{, thick}$$

S is an ID for the sculptured surface; PLUS means that the
tool side of the surface is determined by the cross product
of a TANSPL and a CRSSPL vector (tangent vector for both
incoming and outgoing curve areas; cross-spline vector respec-
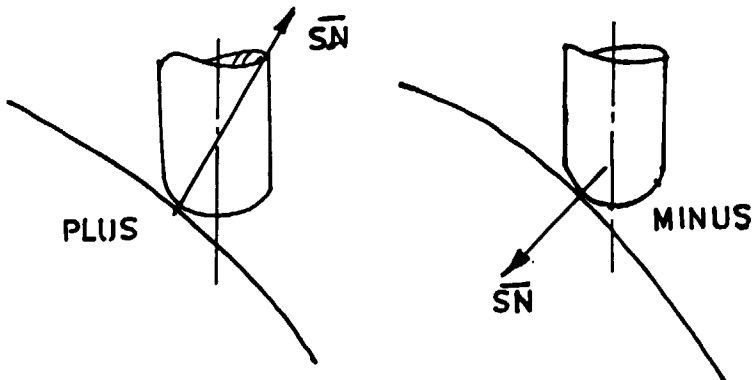tively) and MINUS is the opposite direction. (see Fig 9.5)

Fig 9.5

The thick scalar represents the thickness of material to be
left or to be removed on the surface (positive if to be left
and negative the other case).

The regional milling tool control is programmed with SMIL
major word.

Single tool path can be programmed

SMIL/PATH, DS, PARAM, u, v $\begin{bmatrix} \text{CRSSPL} \\ \text{,TANSPL} \end{bmatrix} \begin{bmatrix} \text{PLUS} \\ \text{,MINUS} \end{bmatrix}$

where u and v are the major and minor direction parameters
respectively of the initial point on the drive control surface,
the other parameters are known.

The tool path generated will start at the selected initial point
and extend to the boundary of the drive geometry (specified in
a previous SCON/DS definition).

Single tool position is also programmable with

SMIL/POSN, DS, PARAM, u, v, INCR $\begin{bmatrix} \text{scalar} \\ \text{vector} \\ \text{plane} \end{bmatrix}$

Typical use of this command to bring the tool safely into contact
with the surface to be machined.

The most frequent milling strategy is the ZIGZAG movement.

SMIL/ZIGZAG, DS, PARAM, u,v $\begin{bmatrix} \text{CRSSPL} \\ \text{,TANSPL} \end{bmatrix} \begin{bmatrix} \text{PLUS} \\ \text{,MINUS} \end{bmatrix}$, STEPOV, \$ $\begin{bmatrix} \text{PLUS} \\ \text{,MINUS} \end{bmatrix}$

Here u and v parameters define the initial point where the
zigzag movements begin. STEPOV indicates, that the stepover
is to be in the direction of increasing parameter values (PLUS)
or decreasing parameter values (MINUS).

The parameters for the stepover movements can be programmed in,

SCON/STEPOV,<parameters>

definition.

List of materials used to compile the lecture notes:

ACZEL - DR. TAKACS: PROGRAMMING THE MK-500/ MITSUBISHI MELDAS MO (BUDAPEST 1987, GTE - in Hungarian.

FARKAS ed: INTRODUCTION TO MANUAL PROGRAMMING OF NC MACHINES (BUDAPEST 1982, TECHNICAL PUBLISH CO.- in Hungarian)

GROOVER - ZIMMERS: CAD/CAM (PRENTICE - HALL 1985)

MATYASI - KRANCZLER: PROGRAMMING THE UNIMERIC CNC IN 3D-5D (BUDAPEST TECHNICAL UNIVERSITY 1987 - in Hungarian)

REMBOLD - BLUME - DILLMANN: COMPUTER INTEGRATED MANUFACTURING TECHNOLOGY AND SYSTEMS (SPRINGER VERLAG 1988)

- :FANUC O-MO OPERATOR'S MANUAL (FANUC 1987)
- :SCULPTURED SURFACE LANGUAGE (CAM - I 1985)
- :ISO DP -:and working papers of ISO/TC 97/SC8 and ISO/TC 184.