Program construction

Program:

The most controller are using as a job control language the symbols from the DIN66025.
After this the partprogram are contains a sequence of lines.
A line contains several words.
A word contains a letter and a number.

Part program:

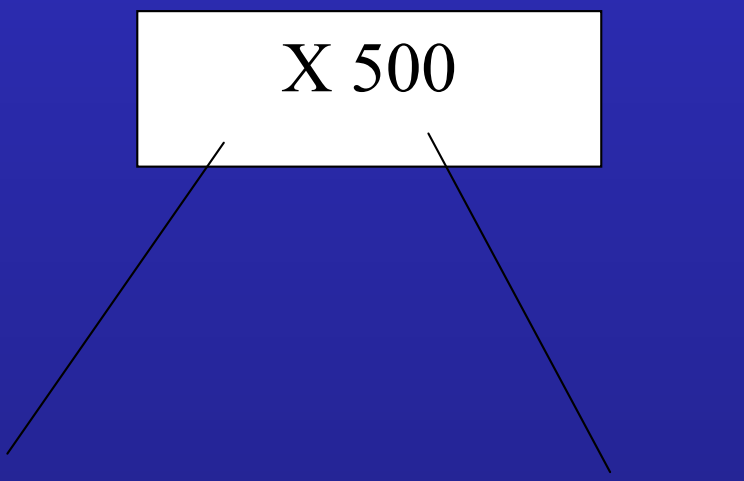
```
N10 G50 S2500
N20 G0 X500 Z500
N30 G0 X50 Z2 T0101 G97 S2500 M3 M8
```

OKUMA

Line:

N20 G0 X500 Z500

Word:



The separate line contains:

- Program technical information.
- Geometrical information.
- Technical information.

OKUMA

Program technical information:
For the execution of the program are necessary.
For example : + Plus
— Minus
. decimal point

Geometrical information.
Means motion of some axis in the machine, the word for the motion is from that address **G** (Engl. Word for **Go**) and some numbers behind.

The most important G commands are:
G00 positioning
G01 linear interpolation

OKUMA

Technical information.

F = Feedrate

T = Tool

S = Spindlespeed

M = additional function

For example:

F – command = F0.25 (mm/rpm)

T – command = T0101 (Tool no. 1)

S – command = S1000 (1000 rpm)

M – command = M03 (spindle direction CW)

Main Address Characters

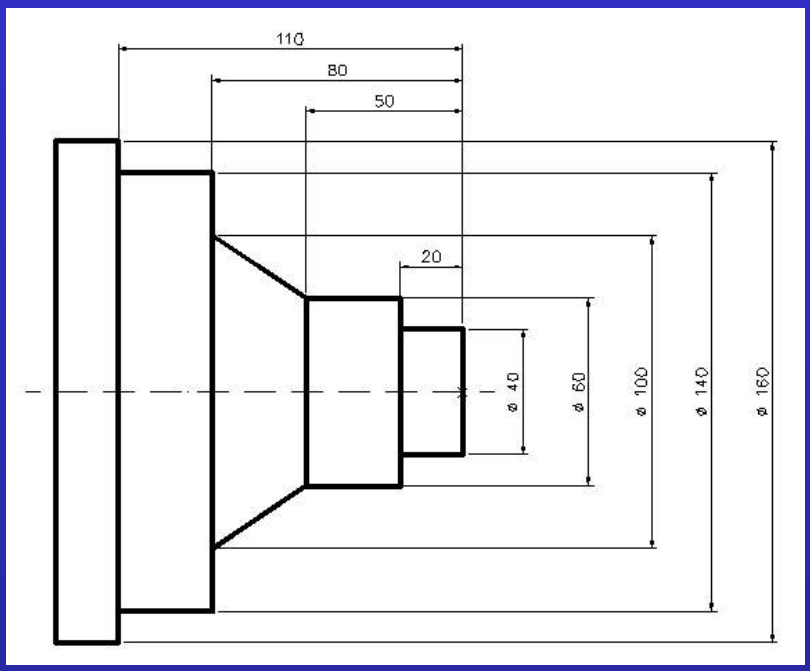
- N Block Number
- G Preparatory Function (See List)
- X Diameter Value
- Z Length Value
- F Feedrate (mm/min or mm/rev)
- (or Dwell time in seconds)
- S Spindle Speed (m/min or rev/min)
- T Turret Station/Offset Number
- M Miscellaneous Function (See List)

M – Function

M – function are help function, just to switch on some additional function, e.g. coolant on or off, C –axis on or off.

OKUMA

Lesson with G01



Program construction for 2 axis lathe machine

G50 S4500
G00 X500 Z500
G00 X0 Z2 T0101 G96 S250 M03 M08
G01 Z0 F0.15
G01 X40
G01 Z-20
G01 X60
G01 Z-50
G01 X100 Z-80
G01 X140
G01 Z-110
G01 X160
G01 Z-130
G00 X500 Z500 M09
M02

Tool command

G Code for constant cutting speed

M- code coolant on

M-code for spindle direction

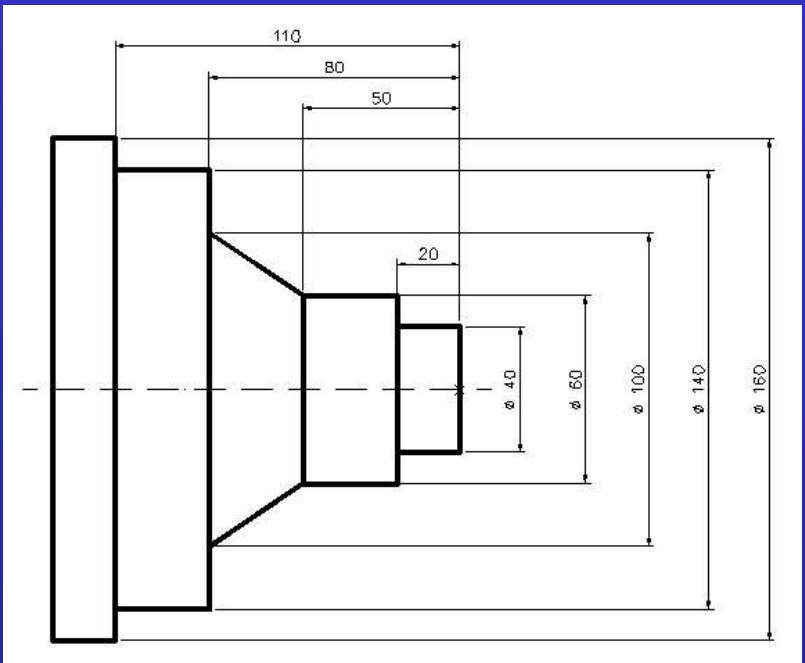
Cutting speed in m/min

Note

The G00 command means that the machine will move with rapid feedrate , the rapid feedrate is dependent on the machine. The unit for rapid feedrate is m/min.
The G01 command means that the machine will move with feedrate, therefore it is necessary to program in the first line with G01 a feedrate command.
The address for feedrate is ”F ” for example F0.25 = 0.25mm by one rotation of the spindle

OKUMA

Lesson with G85 Lap cycle



Program construction for G85 Lap cycle

```
G50 S4500
G00 X500 Z500
G00 X160 Z2 T0101 G96 S250 M03 M08
G85  NAP1  D5  U0.5  W0.1  F0.35
```

F = Feedrate

W = Stock removal in Z

U = Stock removal in X

D = Cuttingdepth in diameter

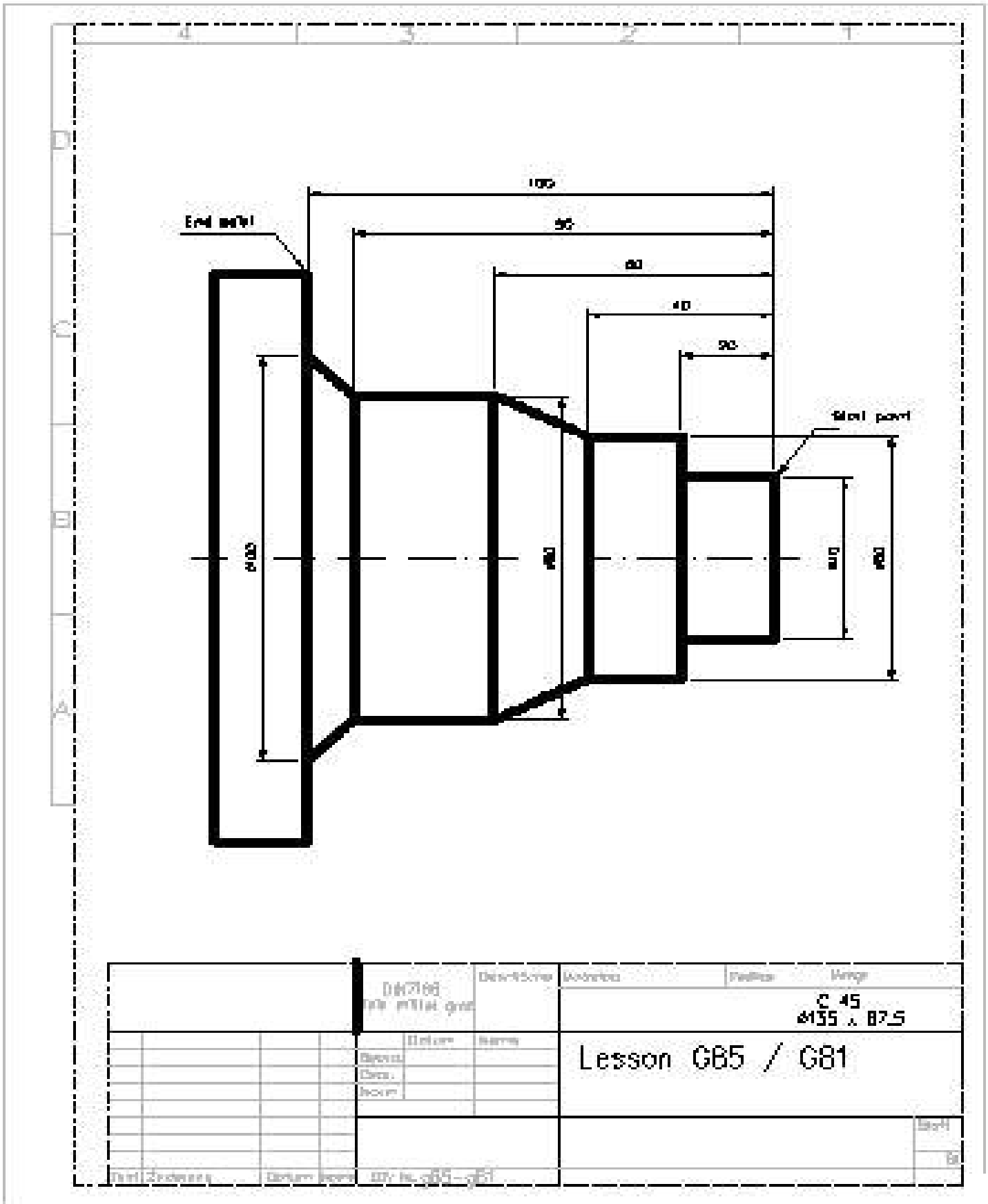
```
NAP1 G81
```

G81 = Cutting direction in Z - axis

```
G01  X0   Z0
     X40
     Z-20
     X60
     Z-50
     X100 Z-80
     X140
     Z-110
     X160
G80
G00 X500 Z500 M09
M02
```

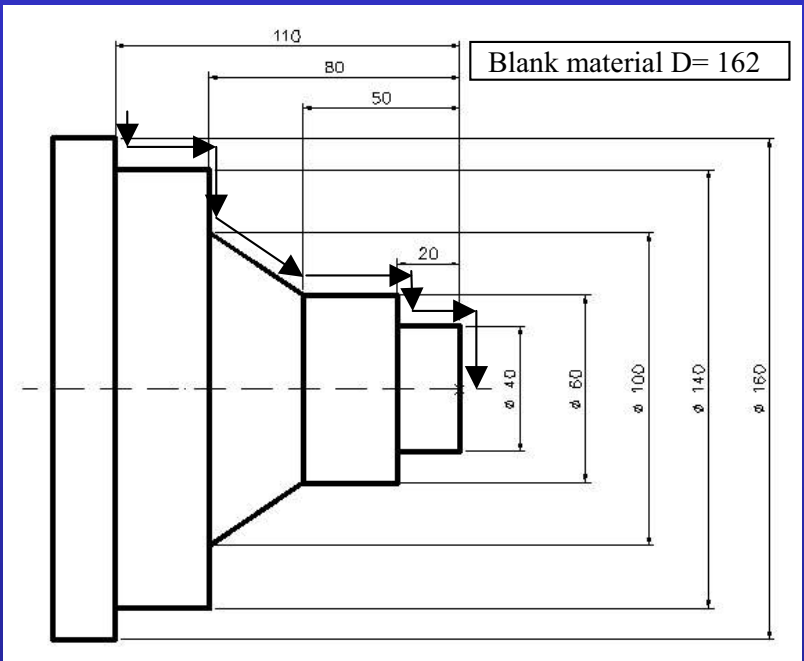

OKUMA

Exercise G85 /G81



OKUMA

Lesson G85 Lap Cycle in X - Direction



Program construction for G85 Lap cycle in X -Direction

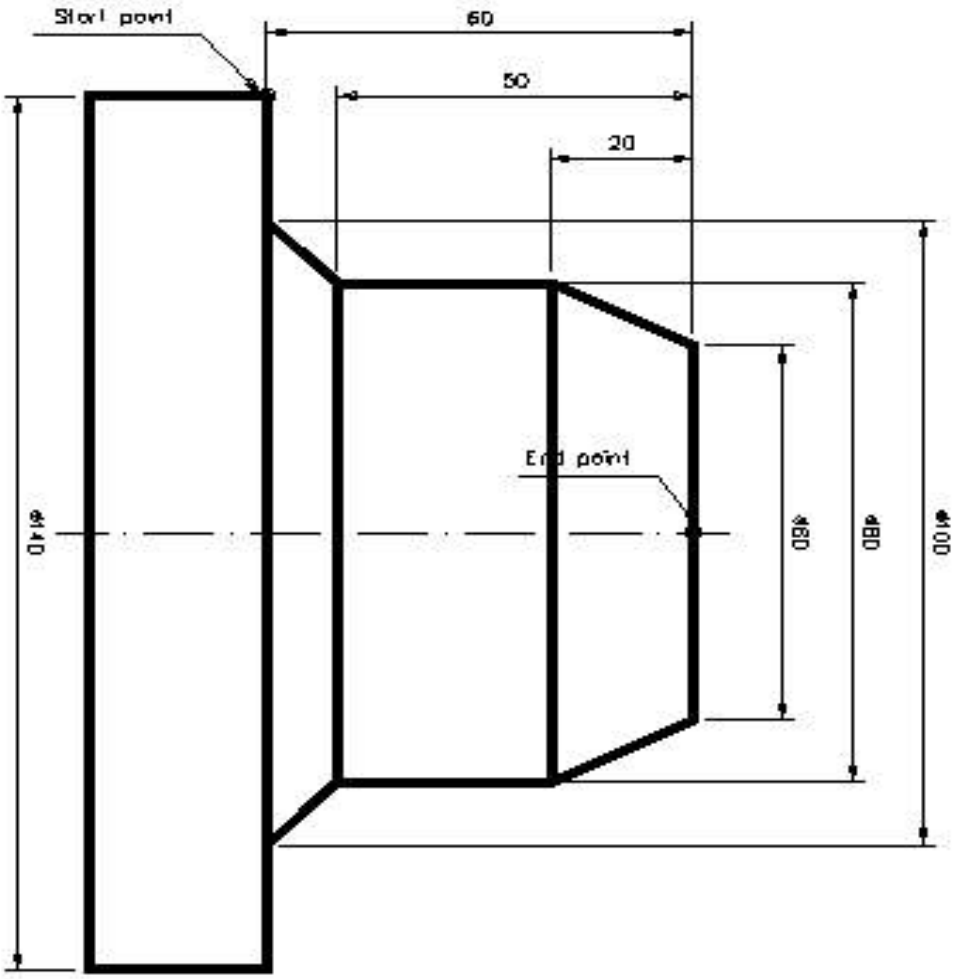
```
G50 S4500
G00 X500 Z500
G00 X162 Z2  T0101 G96  S250  M03  M08
G85 NAP1 D4 U0.5 W0.1 F0.35
NAP1 G82
G0 Z-110
G1 X140
G1 Z-80
G1 X100
G1 X60 Z-50
G1 Z-20
G1 X40
G1 Z0
G80
G00 X500 Z500
M02
```

Note

Please consider, the G82 contour description start from the spindle and will go in direction of the tailstock.

OKUMA

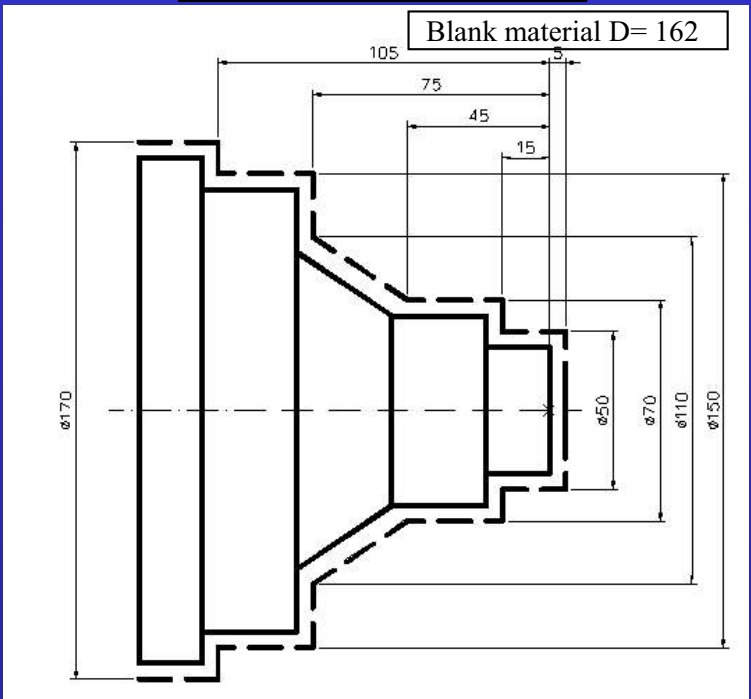
Exercise G85 / G82



	DIN7168 fein mittel grob	Oberfläche	Material	Position	Menge
			C 45 ø135 x 87.5		
		Datum	Name	Lesson G85 / G82	
		Bearb.			
		Gepr.			
		Norm			
					Blatt
Zust. Änderung	Datum	Name	IDV Nr. G85-G82		

OKUMA

Lesson G85 Lap Cycle in Z – direction with blank contour definition




```
G50 S4500
G00 X500 Z500
G00 X165 Z2  T0101  G96  S250  M03  M08
G85 NAP1 D4 U0.5 W0.1 F0.35
NAP1 G83
```

G – Code for blank material

```
G0 X0 Z5
G1 X45
G1 Z-15
G1 X65 Z-45
G1 X105 Z-75
G1 X145
G1 Z-105
G1 X170
```

Blank material definition

```
G81
```

G – Code for longitudinal shape designation

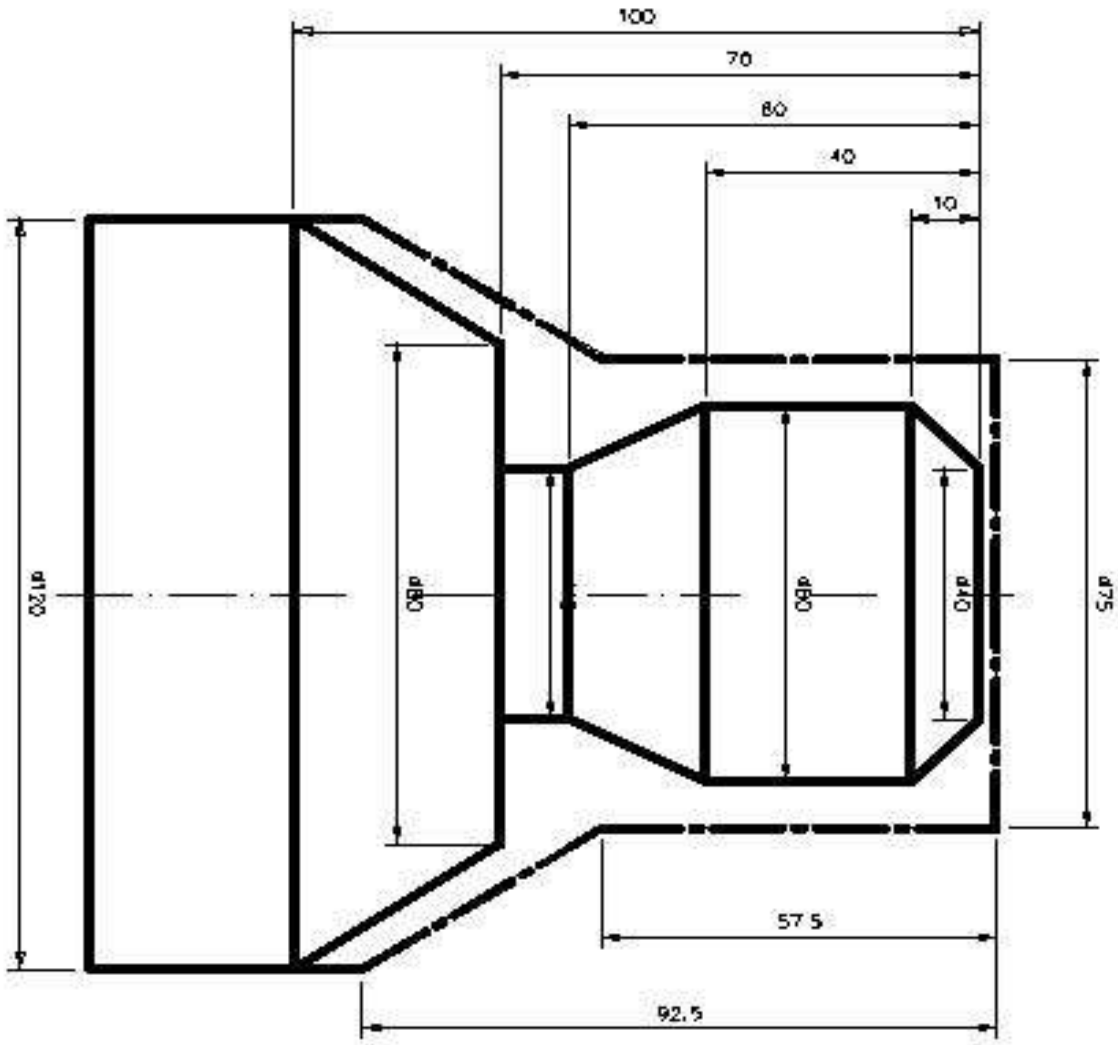
```
G0 X0 Z2
G1 Z0
G1 X40
G1 Z-20
G1 X60 Z-50
G1 X100 Z-80
G1 X140
G1 Z-110
G1 X160
G1 Z-130
G1 X162
```

Finish shape

```
G80
G00 X500 Z500
M02
```


OKUMA

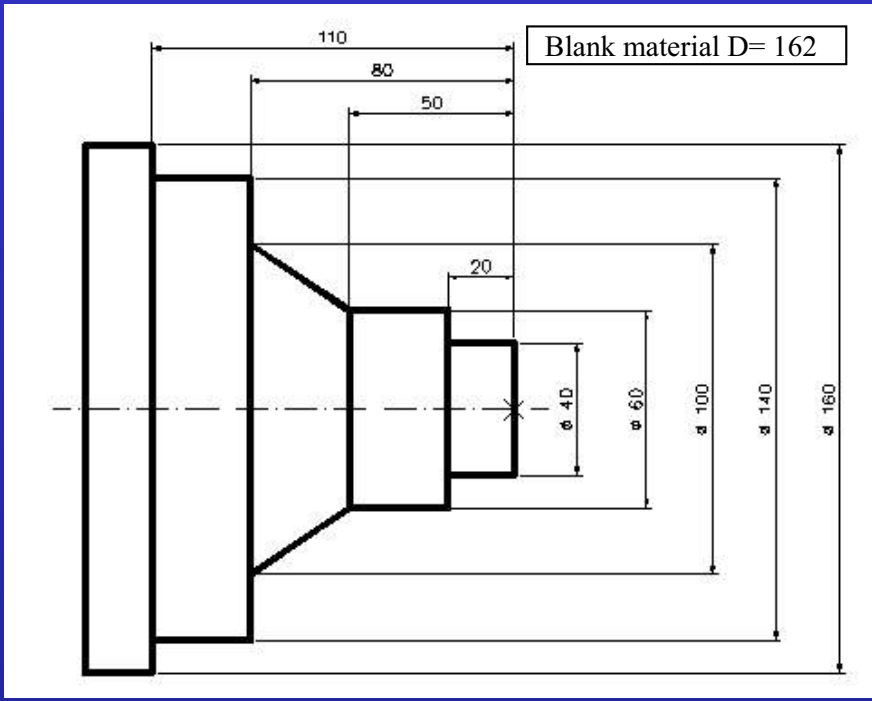
Exercise G85 / G83



	DIN7168 fein mittel grob	Oberfläche	Maßstab	Position Menge
				C 45 ø 145 x 142,5
		Datum	Name	Lesson G85/G81/G83
		Bearb.		
		Gepr.		
		Norm		
Zust. Änderung	Datum	Name	NC-Nr. G85-G81-G83	Blatt

OKUMA

Lesson G85 Lap Cycle with G84 changing cutting condition



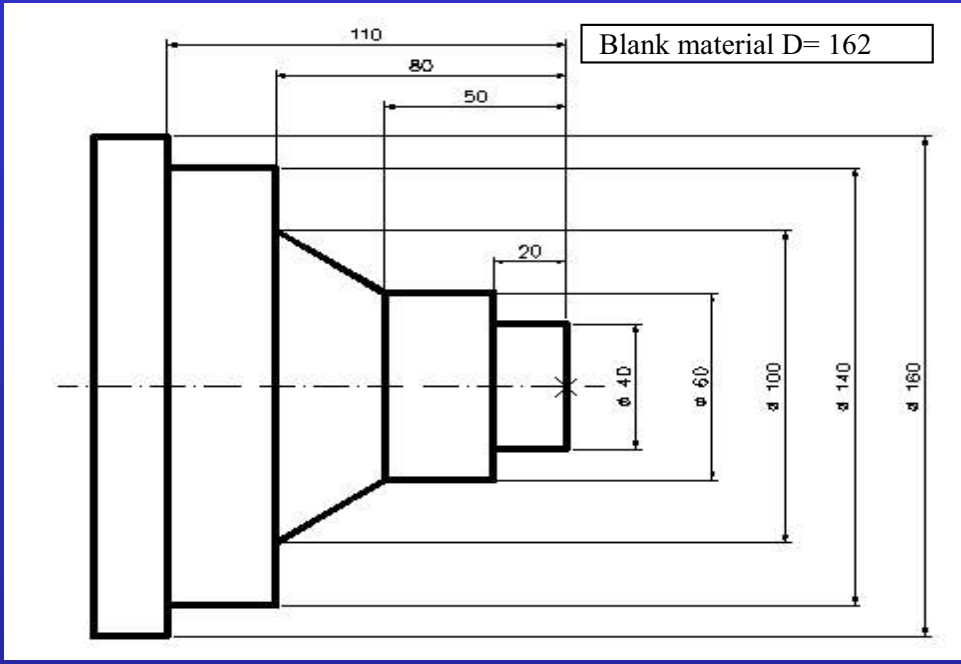
Program construction for G85 Lap cycle with changing cutting condition

```
G50 S4500
G00 X500 Z500
G00 X165 Z2  T0101  G96  S250  M03  M08
G85 NAP1 D4 U0.5 W0.1 F0.35
$ G84 XA=100 ZA=2 DA=1 FA=0.1
NAP1 G81
G00 X0 Z2
G01 Z0 F0.1
G01 X40
G01 Z-20
G01 X60
G01 Z-50
G01 X100 Z-80
G01 X140
G01 Z-110
G01 X160
G80
G00 X500 Z500
M02
```

Feedrate
Cutting depth
Start point in Z for reduction of the cutting condition
Start point in X for reduction of the cutting condition

OKUMA

Lesson G87 Lap Cycle finish cutting cycle

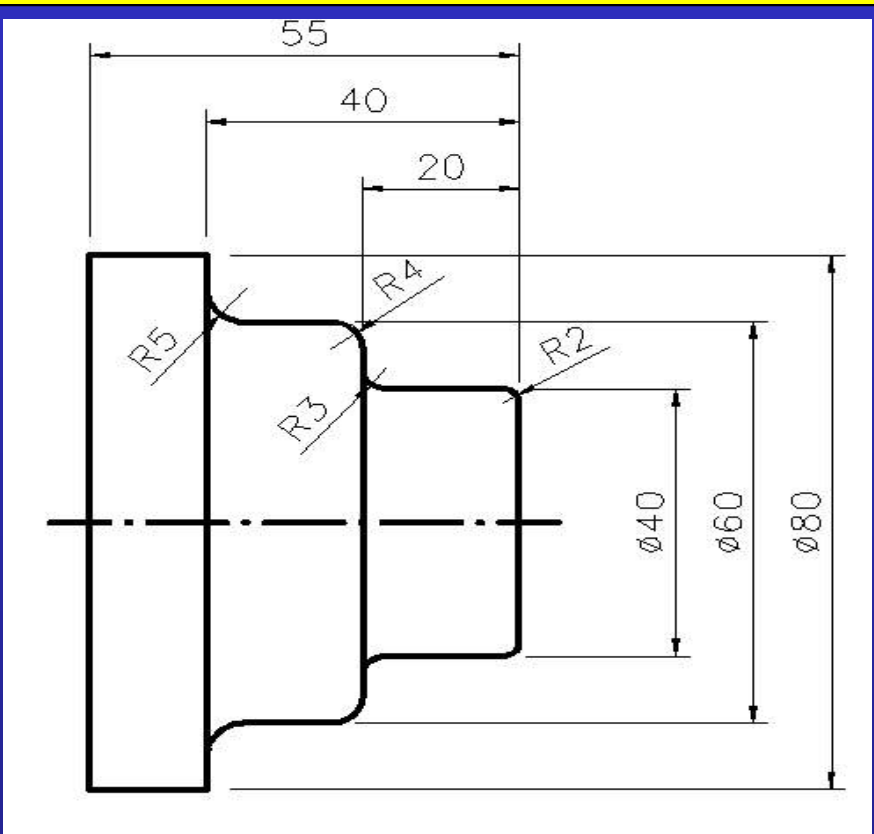


Program construction for G87 finish cutting cycle

```
G50 S4500
G00 X500 Z500
( OD Rough )
G00 X162 Z2  T0101  G96  S250  M03  M08
G85 NAP1 D4 U0.5 W0.1 F0.35  G84 XA=100 ZA=2 DA=1 FA=0.1
NAP1 G81
G00 X0 Z2
G01 Z0 F0.1
G01 X40
G01 Z-20
G01 X60 Z-50
G01 X100 Z-80
G01 X140
G01 Z-110
G01 X160
G01 Z-130
G80
G00 X500 Z500
( OD Finish )
G00 X165 Z2  T0202  G96  S250  M03  M08
G87 NAP1
G00 X500 Z500
M02
```


OKUMA

Lesson G76 automatic rounding

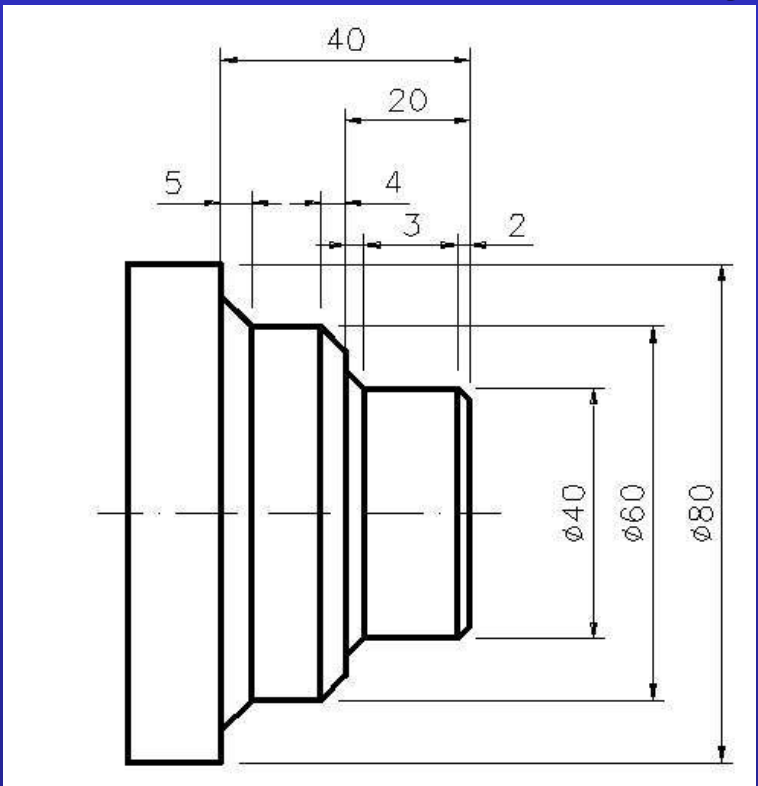


Program construction for G76 automatic rounding

G50 S4500
G00 X500 Z500
G00 X165 Z2 T0101 G96 S250 M03 M08
G00 X0
G01 Z0 F0.1
G01 G76 X40 L2
G01 G76 Z-20 L3
G01 G76 X60 L4
G01 G76 Z-40 L5
G01 X85
G00 X500 Z500 M09
M02

OKUMA

Lesson G75 automatic chamfering

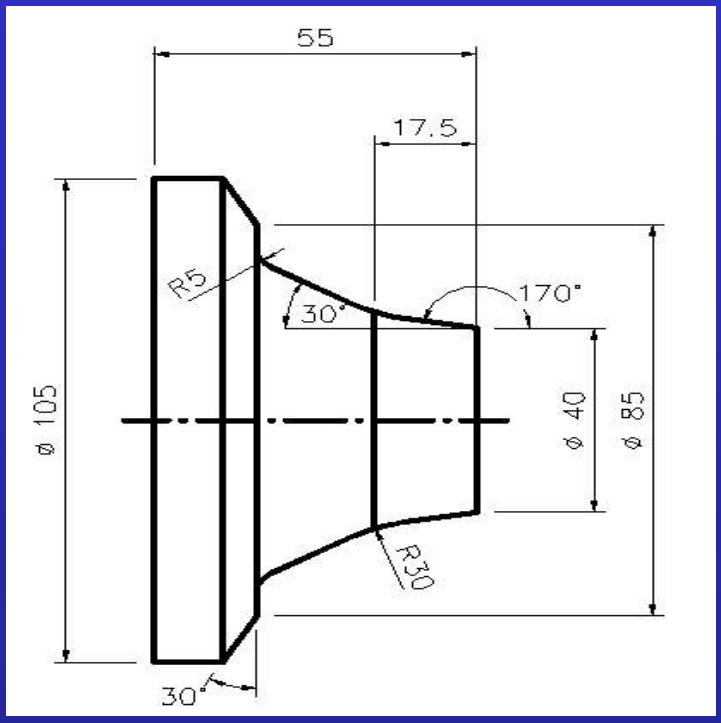


Program construction for G75 automatic chamfering

G50 S4500
G00 X500 Z500
G00 X165 Z2 T0101 G96 S250 M03 M08
G00 X0
G01 Z0 F0.1
G01 G75 X40 L2
G01 G75 Z-20 L3
G01 G75 X60 L4
G01 G75 Z-40 L5
G01 X85
G00 X500 Z500 M09
M02

OKUMA

Lesson Taper cutting by angle designation and G76 function

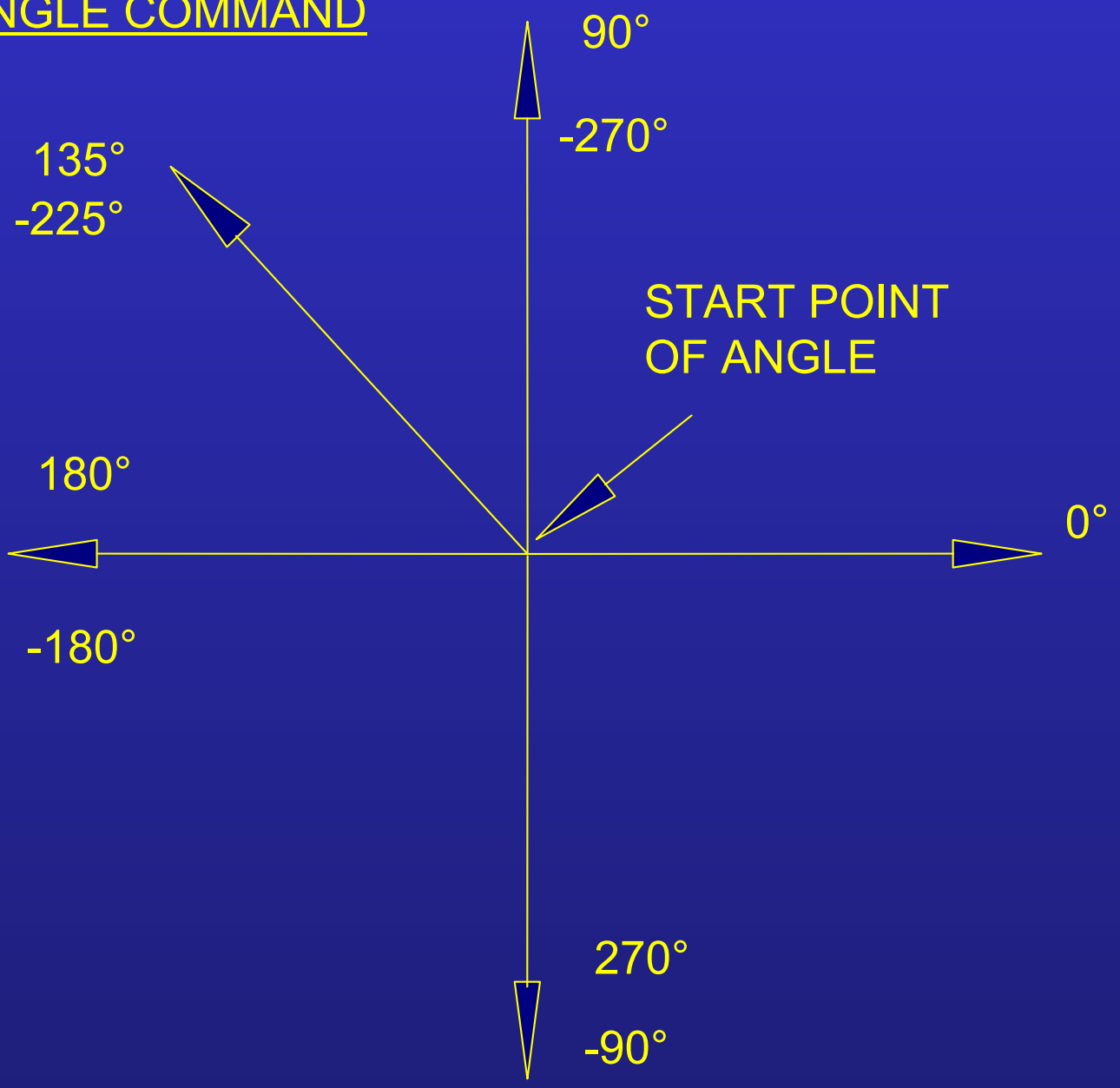


Program construction for Taper cutting by angle designation and G76 function

```
G50 S4500
 G00 X500 Z500
G00 X165 Z2  T0101 G96  S250  M03  M08
G00 X0
G01 Z0 F0.1
G01 X40
G01 G76 Z-17.5 A170 L30
G01 G76 Z-37.5 A150 L5
G01 X85
G01 X105 A120
G01 Z-55
G01 X110
G00 X500 Z500 M09
M02
```

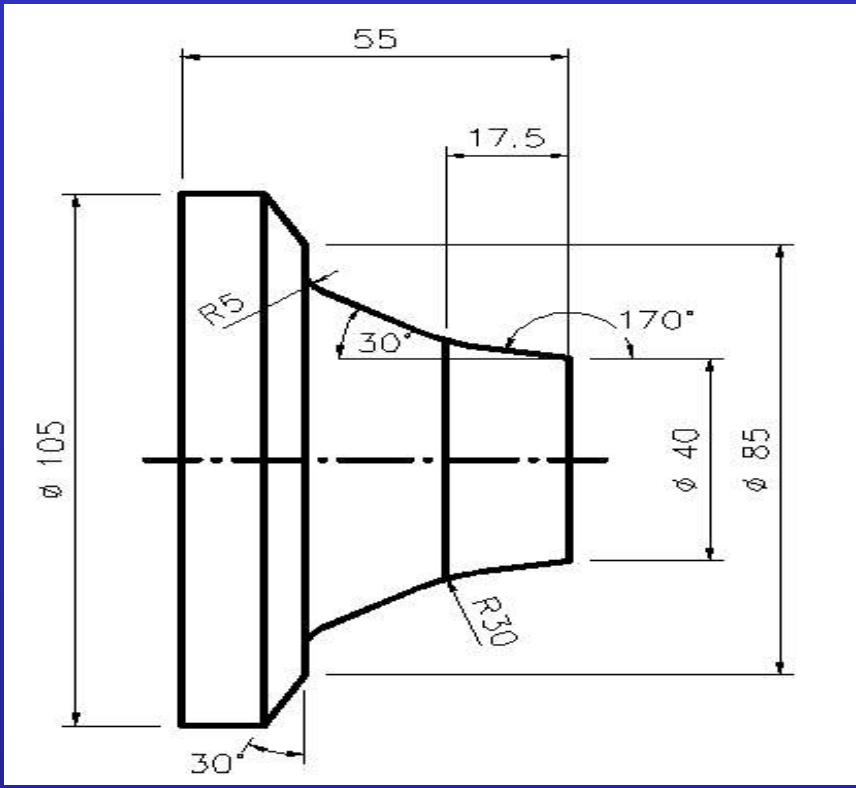

OKUMA

A VALUE
DIRECT ANGLE COMMAND



OKUMA

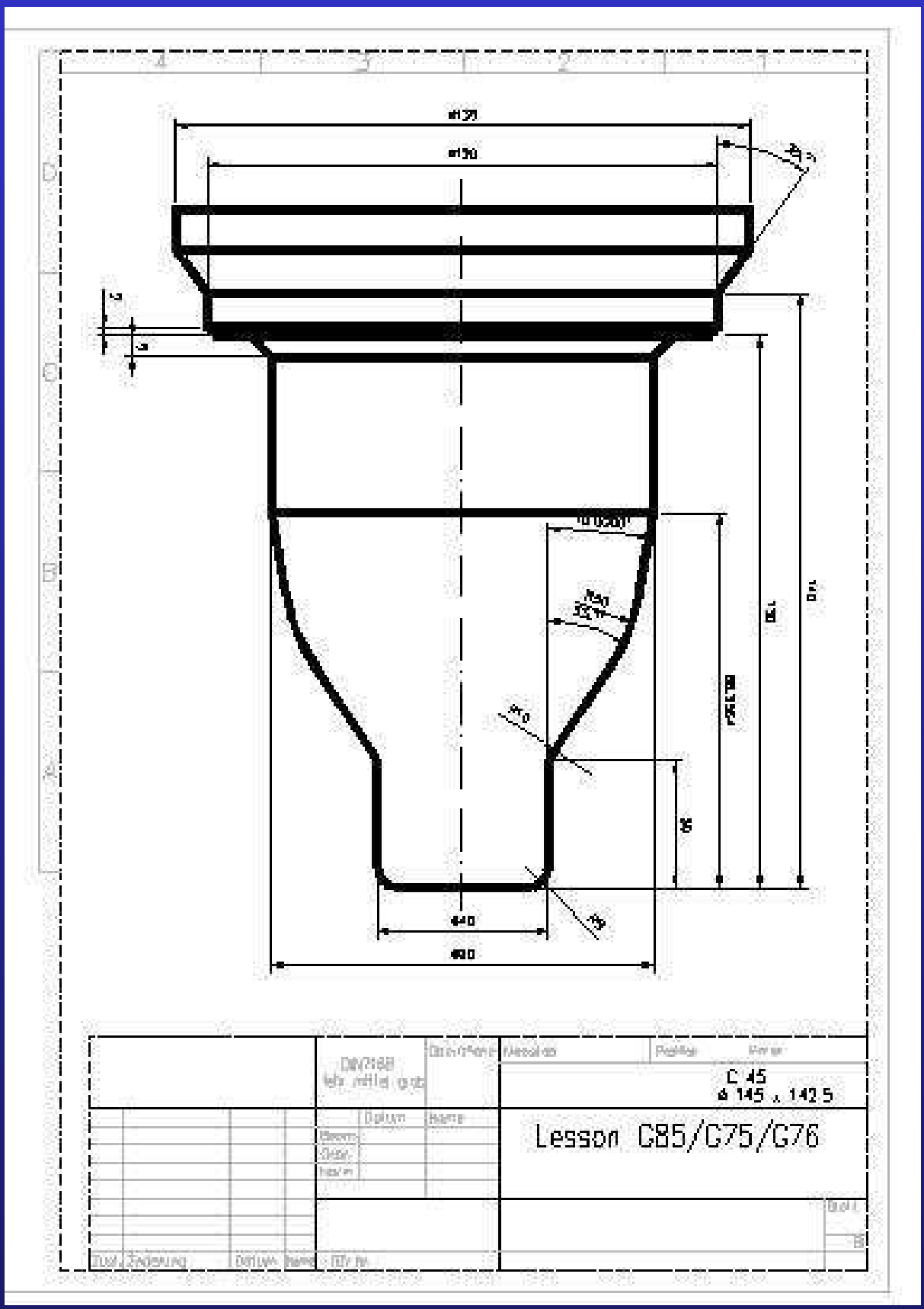
Lesson Taper cutting by angle designation and G76 function



Program construction for Taper cutting by angle designation and G76 function

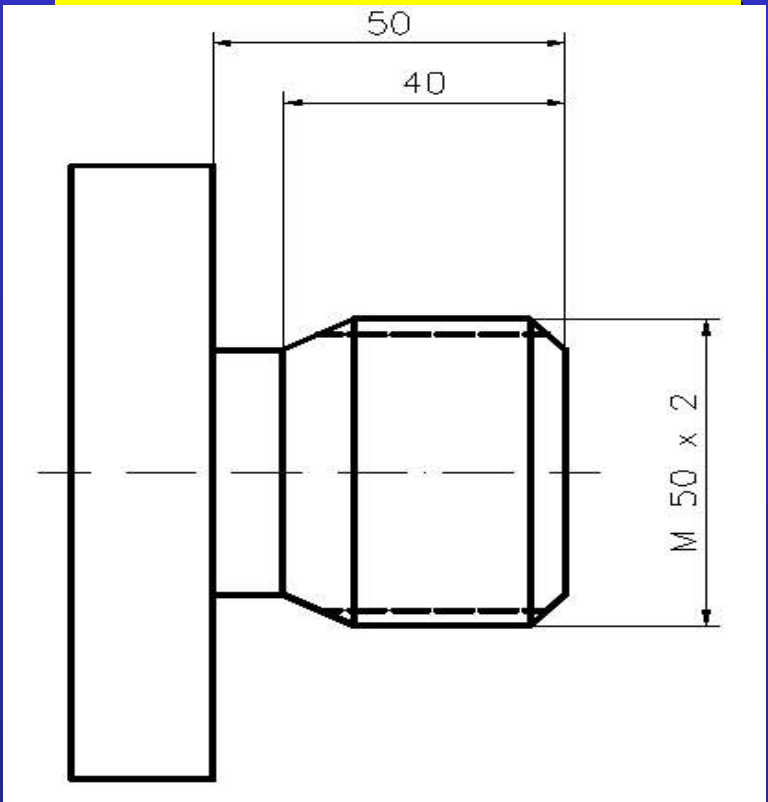
```
G50 S4500
G00 X500 Z500
G00 X165 Z2  T0101 G96  S250  M03  M08
G00 X0
G01 Z0 F0.1
G01 X40
G01 G76 Z-17.5 A170 L30
G01 G76 Z-37.5 A150 L5
G01 X85
G01 X105 A120
G01 Z-55
G01 X110
G00 X500 Z500 M09
M02
```


OKUMA



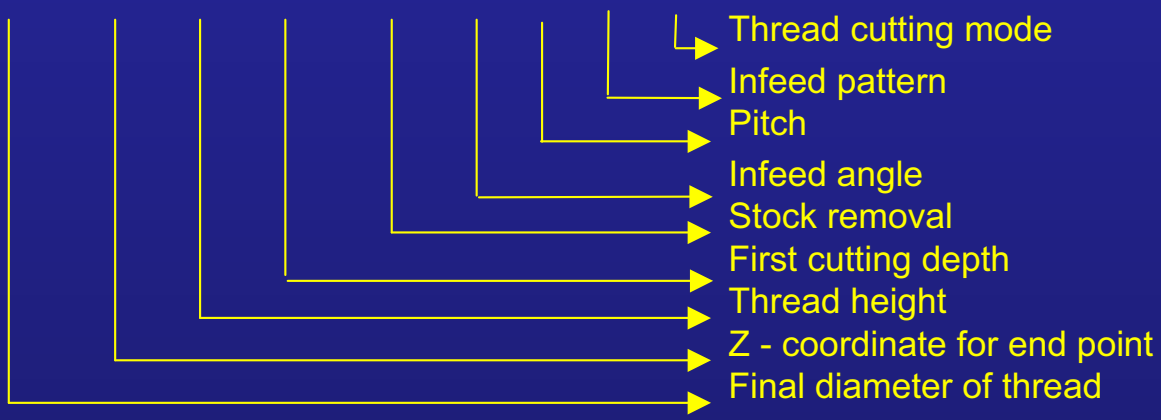
OKUMA

Lesson G71 thread cutting



Program construction for G71 thread cutting cycle

G71 X 47.4 Z-40 H2.6 D0.25 U0.04 B60 F2 M73 M33

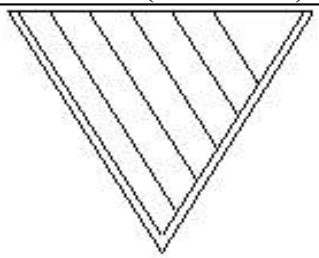
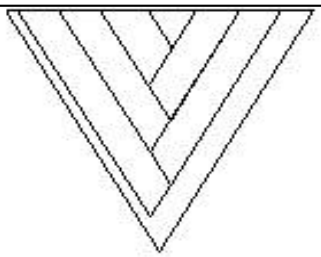
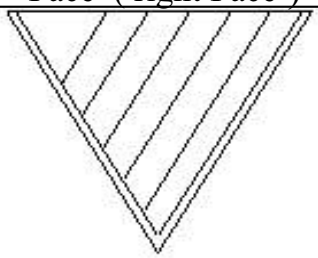


G50 S2500
G00 X500 Z500
G0 X54 Z4 T0101 G97 S510 M3 M8
G71 X47.4 Z-40 H2.6 D0.25 U0.04 B60 M73 M33
G00 X500 Z500
M2

Note:
In case of G71 cycle it is not possible to use G96 command.

OKUMA

Lesson G71 thread cutting cycle

B60 M32 Straight Infeed along thread Face (left Face)	B60 M33 Zig zag Infeed	B60 M34 Straight Infeed along thread Face (right Face)
		

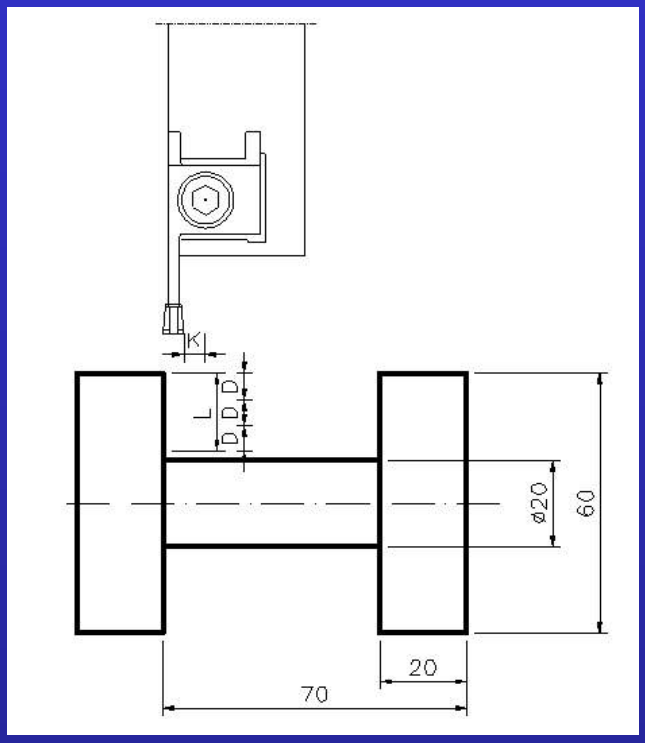
Cuttingdepth calculation:

M73 Infeed is made by D (in diameter) in each thread cutting cycle up to the point D mm away from " H –U (W) position. After that point is reached, Infeed amount change to D/2 D/4 D/8, leaving stock removal U (W) if specified. And in the finishing cycle, Infeed is made as much as the specified amount U (W). (until 800 Kg / mm²)

M74 Infeed is made by D (in diameter) until the point is away from " H –U (W) position . (Aluminium, Brass, plastic)

M75 Infeed is made always by the same chip section . (from 800 Kg / mm²)

Lesson G73 Grooving cycle



Program construction for G73 grooving cycle

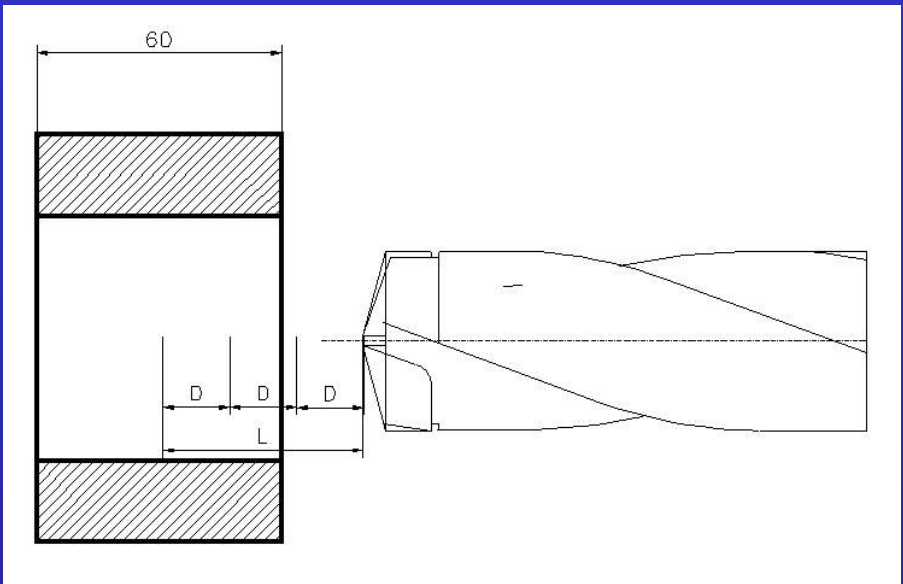
G73 X20 Z-70 K4 D2 L10 E0.2

- E0.2 → Dwell time
- L10 → Total Infeed amount to the cutting start point
- D2 → Depth of cut per peck feed
- K4 → Shift amount to the startpoint
- Z-70 → Endpoint in Z
- X20 → Final diameter in X

```
G50 S2500
G00 X500 Z500
G00 X62 Z-24 T0404 G96 S150 M3 M42 M08
G73  X20  Z-70  K4  D2  L10  E0.2 F0.15
G00 X500 Z500
M02
```


OKUMA

Lesson G74 Drill cycle



Program construction for G74 drill cycle

G74 X0 Z-70 D20 L40 E0.2

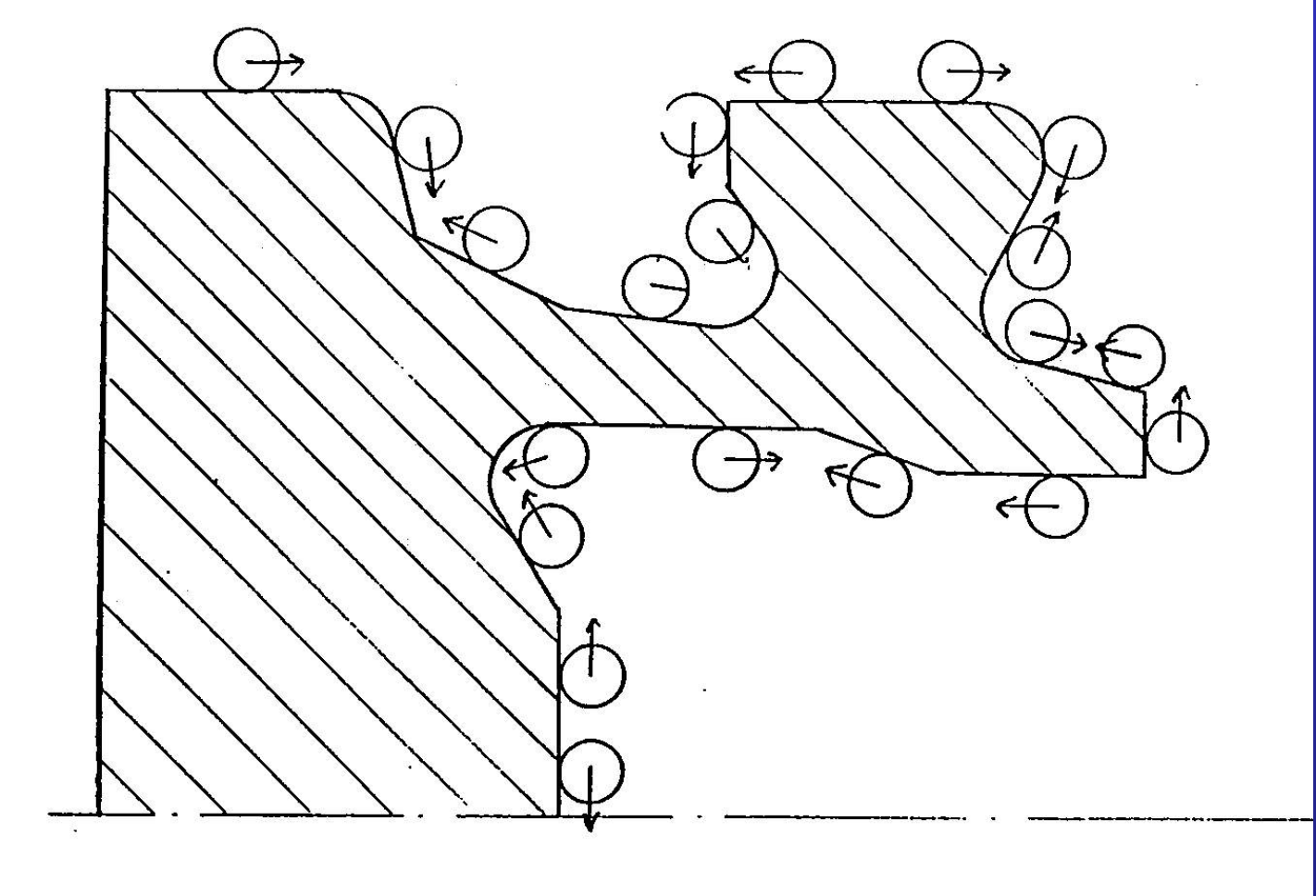
- E0.2 → Dwell time
- L40 → Total Infeed amount to the cutting start point
- D20 → Depth of cut per peck feed
- Z-70 → Final point in Z
- X0 → End point in X

```
G50 S2500
G00 X500 Z500
G00 X0 Z4 T0404 G97 S1500 M3 M42 M08
G74  X0  Z-70  D20  L40  E0.2 F0.15
G00 X500 Z500
M02
```


OKUMA

Lesson Cutterradius compensation

Recognition aid for different cutting direction during works with automatic cutting radius compensation.

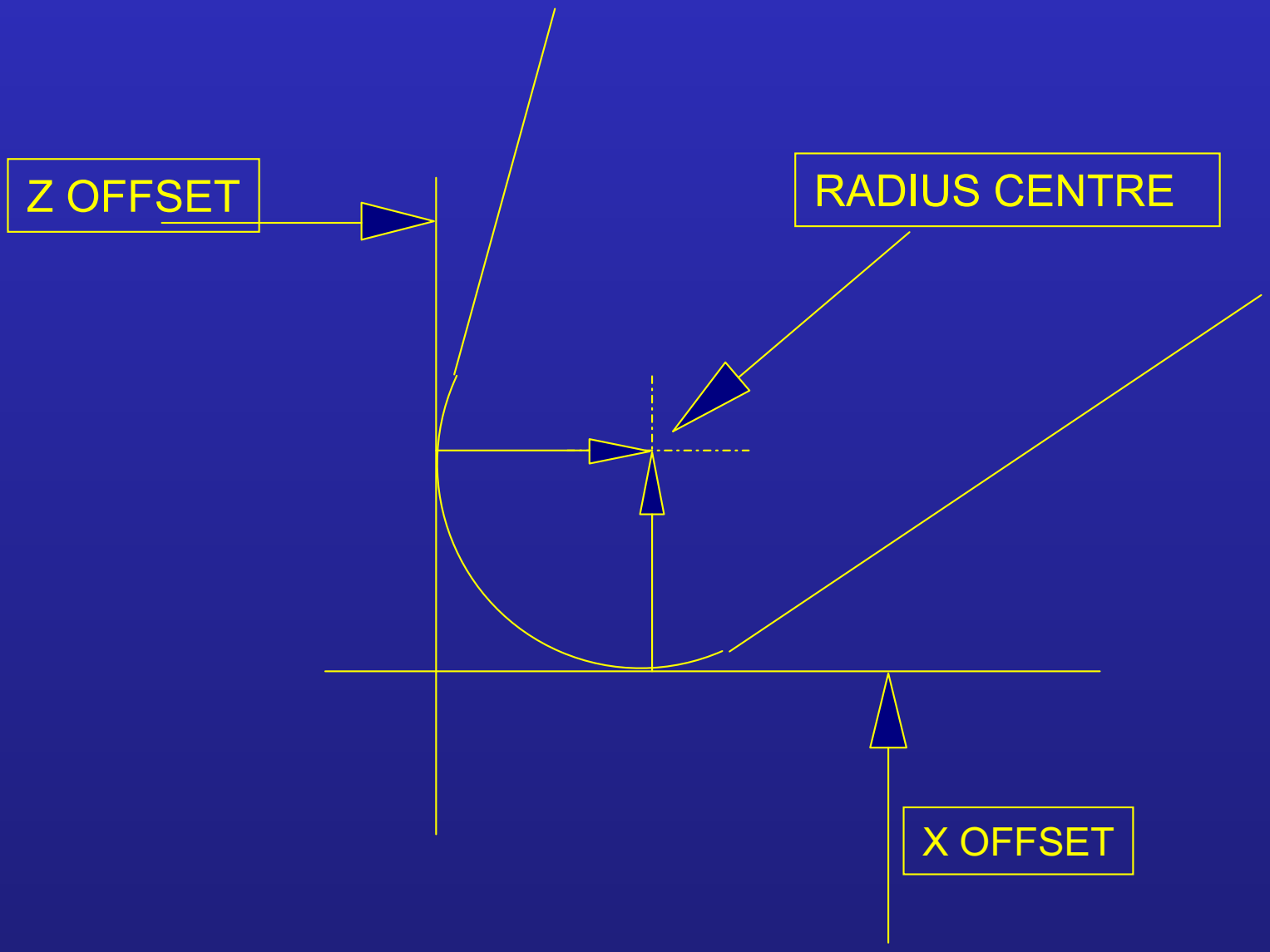


One sees in the direction of feedrate (arrows) and the tool is to the right of the outline, than it is necessary to program G42.

If is to the left there of the cutting direction then, the command must be G41.

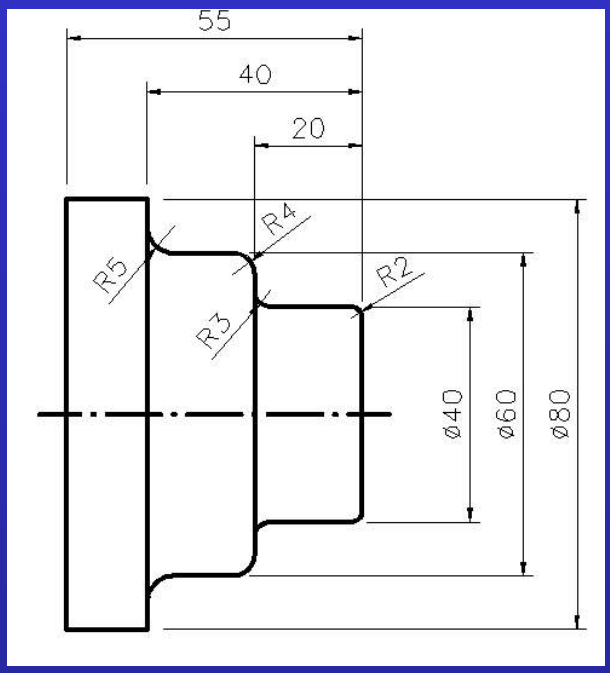
OKUMA

Lesson Cutterradius compensation



OKUMA

Lesson G41/G42 Cutter radius compensation

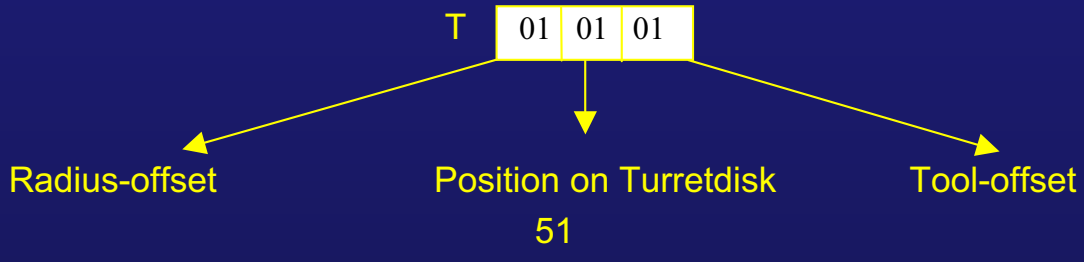


Program construction for cutter radius compensation

G50 S4500
G00 X500 Z500
G00 X165 Z2 T010101 G96 S250 M03 M08
G00 X0
G01 G42 Z0 F0.1
G01 G76 X40 L2
G01 G76 Z-20 L3
G01 G76 X60 L4
G01 G76 Z-40 L5
G01 X85
G40
G00 X500 Z500 M09
M02

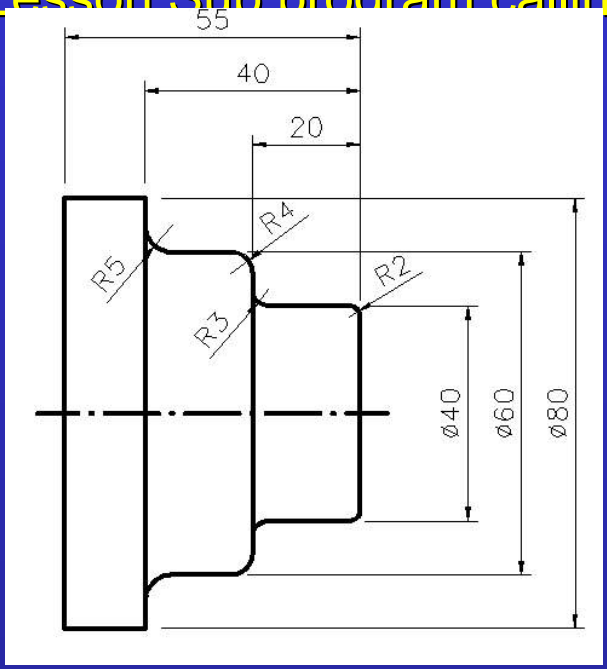
Note:

If it is necessary to use G41 / G42 than the tool command must change to T010101.



OKUMA

Lesson Sub program calling



Program construction

```
G50 S4500
G00 X500 Z500
G00 X165 Z2  T010101 G96  S250  M03  M08
CALL OSUB
G00 X500 Z500 M09
M02

OSUB
G00 X0
G01 G42 Z0 F0.1
G01 G76 X40 L2
G01 G76 Z-20 L3
G01 G76 X60 L4
G01 G76 Z-40 L5
G01 X85
G40
RTS
```

Note:

To call an Subprogram in a main program it is necessary the use the command ” CALL ”
The subprogram name must begin with an " O " and may have not more than 4 signs.
The subprogram must end with command ” RTS ” .

OKUMA

Macro's

What is a Macro?

A group of instructions, which are possible to store and called as an unit, this make it possible the reduce the time of programming for repeatable jobs or family parts.

Variables Function:

In OSP controller it is possible to use 5 kind of Variable.
1.)Common variables
2.)Local variables

Common Variables

The term ”common” in ”common variables” can be literally understood as common; they can be used in common for main and subprograms. When the same variable is used in two or more programs, the variable number used in those programs must be identical. Therefore, a common variable, the result of calculation in one program, can be referred to in other programs.

[Format]

V	numerals = numerical data or expression

Common variable designations consist of up to three digits following ”V”. The usable common variables are V1 through 200.

Examples:
N101 V5 = 10
N101 V5 = V5 + 1

[Details]
- Common variables are effective both in main programs and subprograms.
- Common variables are not affected by resetting the control or turning power off. That is, the data are retained unless they are re-set or a control software is installed.
- Besides setting or changing them in a program, common variables can be set or changed by setting a parameter. For detailed information on parameter setting, refer to SECTION PARAMETER SETTING, DATA OPERATION in OPERATION MANUAL.

Local Variables

As is apparent from the term ”local”, local variables are the variables that a user can set as desired with meaningful names assigned to them. Up to 127 local variables each can be used for the A and B saddles.

[Format]

Letter	Letter	two alphanumeric = Numerical data or expression

O, N and V cannot be used.

Example: ‘DlA1’ ‘ITH5’

[Details]

A local variable cannot be assigned the same name as already used for a function name, comparison operator, Boolean operator, or extended address character.

Extended address characters are provided to realise LAP, pattern processing, and user-specific fixed cycles. They are necessary because there are not enough letters in the alphabet to cover the required number of extension names. The following extended address characters are currently used.

<AA> <AB> <DA> <DB> <FA> <FB> <IA> <IB> <KA> <KB>
<LA> <LB> <RA> <RB> <SA> <SB> <TA> <TB> <UA> <UB>
<WA> <WB> <XA> <XB> <ZA> <ZB> <BC>

Characteristics of Local Variables

- Local variables are cleared when the control is reset.
- When a new local variable is set in a main program, that is, when data is assigned to a new local variable name, that local variable name and corresponding data are registered in the memory.

NOTICE

If a local variable name is used without setting any data for it, an alarm results.

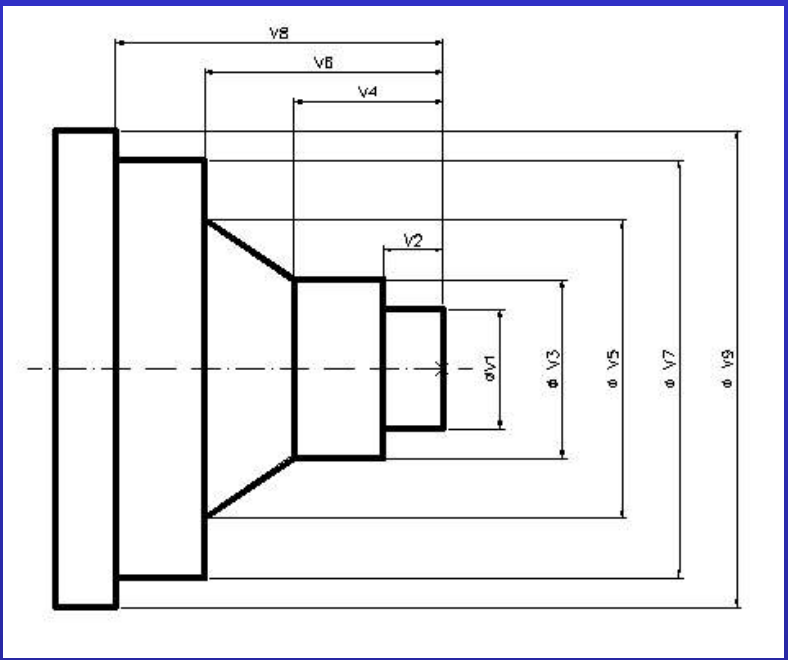
- When new data is assigned to a local variable already registered with other data, that old data is updated.

```
N0010      DIA1 = 160
:
:
:
N0049
N0050      DIA1 = 20
:
:
:
```

In N0010, numerical data "160" is assigned to local variable name "DIA1", and this data remains effective up to sequence N0049. In N0050, the new numerical data "200" is assigned to the same local variable name "DlA1". This clears the old data "160" and replaces it with the new data "200"

- Up to 127 local variables can be used.

Lesson with common variable

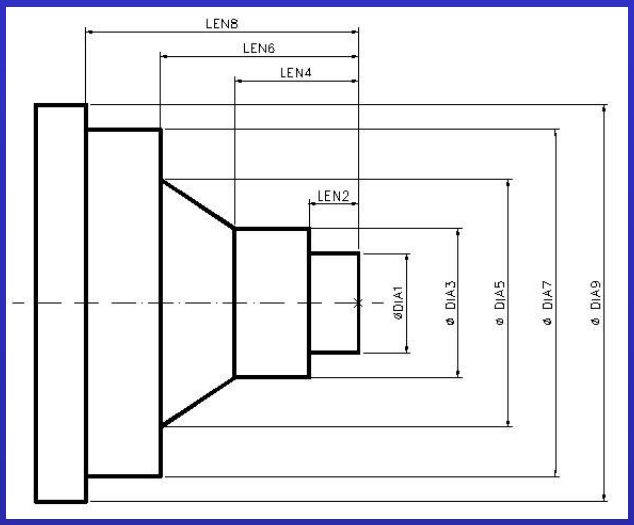


Program construction with common variable

```
V1=40
V2=20
V3=60
V4=50
V5=100
V6=80
V7=140
V8=110
V9=160

G50 S4500
G00 X500 Z500
G00 X=V9 Z2 T0101 G96 S250 M03 M08
G01 X0 Z0
      X=V1
      Z=-V2
      X=V3
      Z=-V4
      X=V5 Z=-V6
      X=V7
      Z=-V8
      X=V9
G00 X500 Z500 M09
M02
```


OKUMA

Lesson with local variable



Program construction with local variable

```
DIA1=40
LEN2=20
DIA3=60
LEN4=50
DIA5=100
LEN6=80
DIA7=140
LEN8=110
DIA9=160

G50 S4500
G00 X500 Z500
G00 X=DIA9 Z2 T0101 G96 S250 M03 M08
G01 X0 Z0
      X=DIA1
      Z=-LEN2
      X=DIA3
      Z=-LEN4
      X=DIA5 Z=-LEN6
      X=DIA7
      Z=-LEN8
      X=DIA9
G00 X500 Z500 M09
M02
```


OKUMA

Arithmetic Operation Function

This function allows arithmetic operation using variables. The programming can be done in the same way as for general arithmetic expressions.
Address character, Extended address character, Variable = Expression
The expression on the right-hand side, requesting an arithmetic operation, is made up of constants, variables, comparison expressions, and operators.
The arithmetic and comparison expressions are described below.

1.)Arithmetic Expression

Operator	Meaning	Example
+ –	Positive sign Negative sign	+1234 –1234
+ –	Sum (addition) Difference (subtraction)	X = 12.3 + V1 X = 12.3 – V1
* /	Product (multiplication) Quotient (division)	X = V10 * 10 X = V11/10

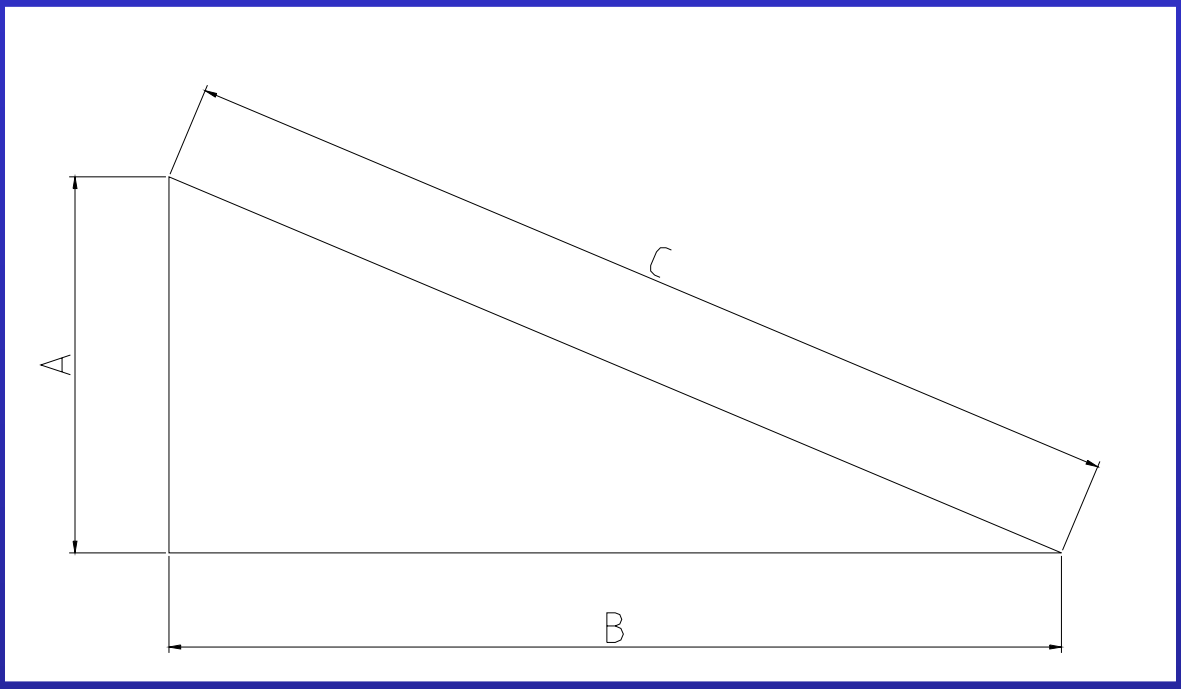
2.) Comparison Expression

Operator	Meaning	Example	Contents	Rule
LT	(Less Than, <)	IF [V1 LT 5] N100	Jump to N100 when V1 is less than 5.	Provide a space on either side of the operator.
LE	(Less than or Equal to, ð)	IF [V1 LE 5] N100	Jump to N100 when V1 is less than or equal to 5.	
EQ	(EQual to, =)	IF [V1 EQ 5] N100	Jump to N100 when V1 is equal to 5.	
NE	(Not Equal to, ¦)	IF [V1 NE 5] N100	Jump to N100 when V1 is not equal to 5.	
GT	(Greater Than, >)	IF [V1 GT 5] N100	Jump to N100 when V1 is greater than 5.	
GE	(Greater than or Equal, Š)	IF [V1 GE 5] N100	Jump to N100 when V1 is greater than or equal to 5.	

3.) Function

Function	Meaning	Example	Rule and Remark
SIN COS TAN	Sine Cosine Tangent	V1= V1 * SIN [V3] V1= V1 * COS [V3] V1= V1 * TAN [V3]	Numbers after function operation symbols must be enclosed in square brackets.
ATAN	Arctangent (1) Value range: -90° to 90°	V1 = ATAN [V2]	
ATAN2	Arctangent (2) Angle of point defined by coordinate value (a, b). Value range -180° to 180°	V1 = ATAN2 [2]	When two elements are specified within square brackets, place a comma between them.
SQRT	Square root	V1 = SQRT [V2]	
ABS	Absolute value	V1 = ABS [V2]	
BIN	Decimal to binary conversion	V1 = BIN [V2]	The position of the decimal point is determined in accordance with the unit system selected. The unit systems for angle commands are: 1 deg. for 1mm and 1 inch unit system 0.001 deg. for 1µm unit system 0.0001 deg. for 0.0001 inch system
BCD	Binary to decimal conversion	V1 = BCD [V2]	
ROUND	Rounding off fractions	V1 = ROUND [V2]	
FIX	Cutting off fractions	V1 = FIX [V2]	
FUP	Counting fractions as a whole number	V1 = FUP [V2]	
DROUND	Rounding off fractions to three decimal places (metric system) or to four decimal places (inch system)	V1 = DROUND [V2]	
DFIX	Cutting off fractions below the third decimal place (metric system) or below the fourth decimal place (inch system)	V1 = DFIX [V2]	
DFUP	Count the figures below the third decimal place (metric syst em) or below the fourth decimal place (inch system) as a whole number	V1 = DFUP [V2]	
MOD	Remainder (a - fix[a/b]:b)	V1 = MOD [V2/V3]	

Lesson for triangle calculation



Formula for calculate the sides of a triangle: $A^2 + B^2 = C^2$ (Pythagorean)

A = 25
B = 55
C = ?

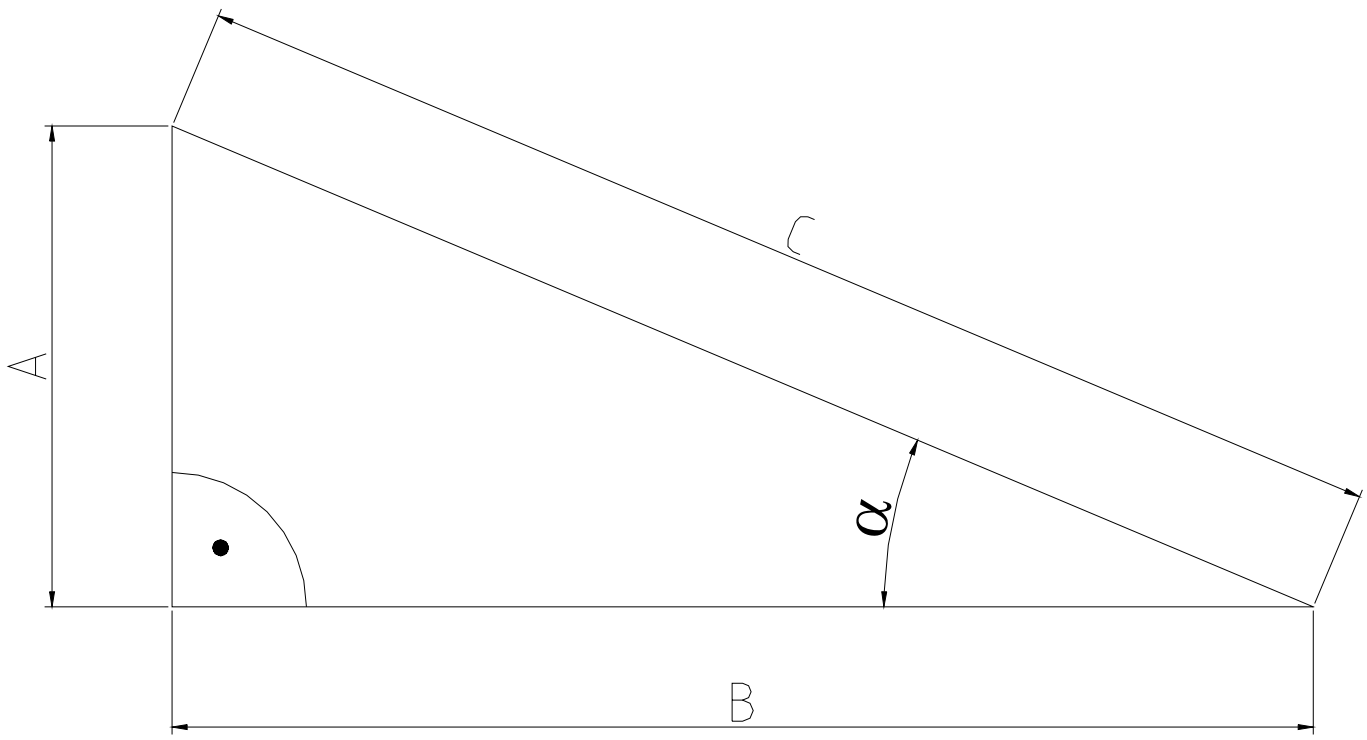
One possibility for calculation.
V1=25
V2=55
V10=V1*V1 (625)
V11=V2*V2 (3025)
V12=V10+V11 (3650)
V13=SQRT[V12] (60.415)
M2

Another possibility for calculation
V1=25
V2=55
V3=SQRT[[V1*V1]+[V2*V2]]
M2

A maximum of seven pairs of [ ] can used.

OKUMA

Lesson for triangle calculation



Formula for angle calculation:

$$\text{Tan } \alpha = \frac{A}{B} \qquad \text{Sin } \alpha = \frac{A}{C} \qquad \text{Cos } \alpha = \frac{B}{C}$$

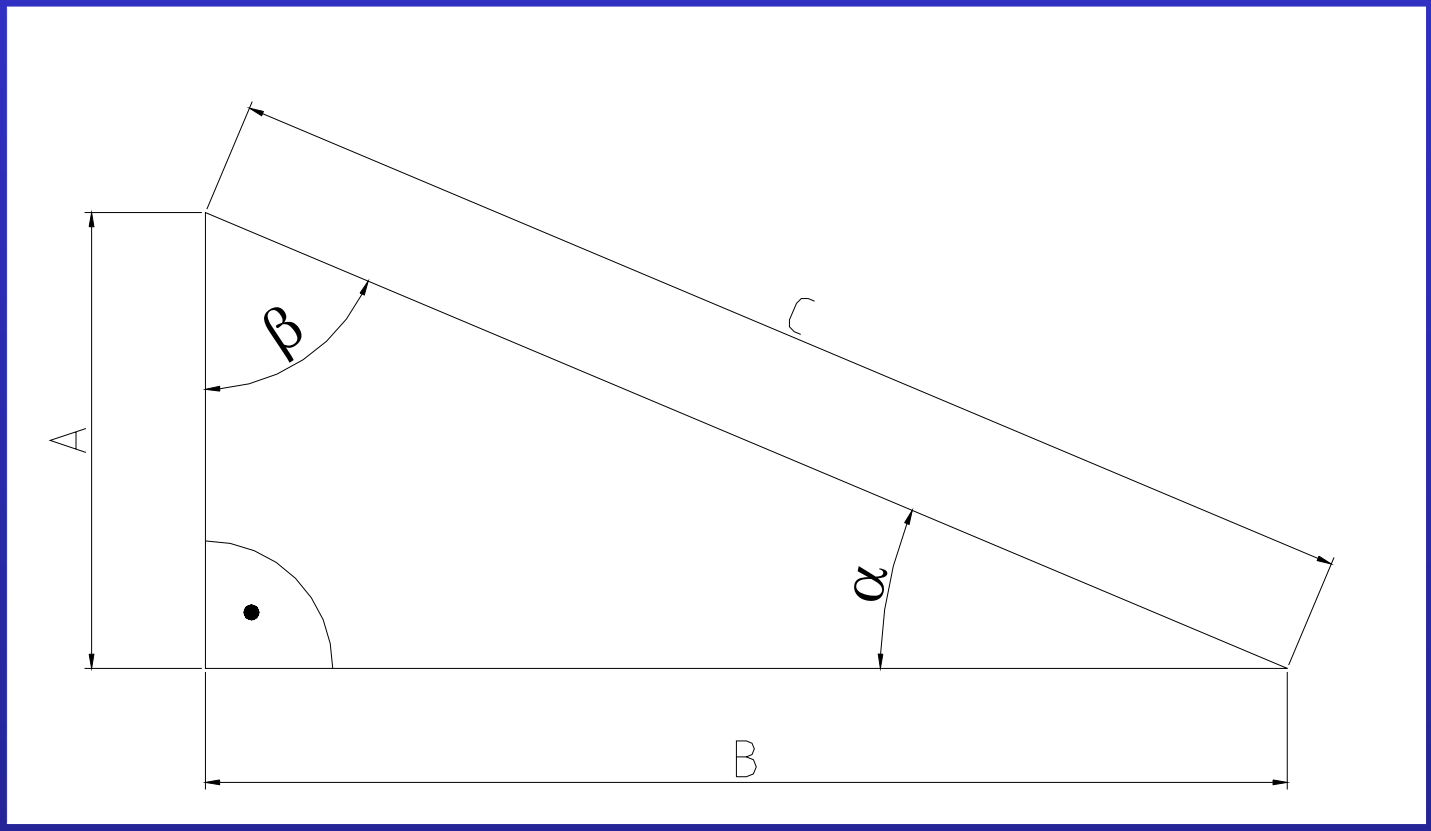
A = 25
B = 55
C = 60.415
α = ?

V1=25
V2=55
V3=60.415

V10=V1/V2 (0.454545)
V11=ATAN[V10] (24.444°)
M2

OKUMA

Exercise for triangle calculation



Exercise:

Please calculate

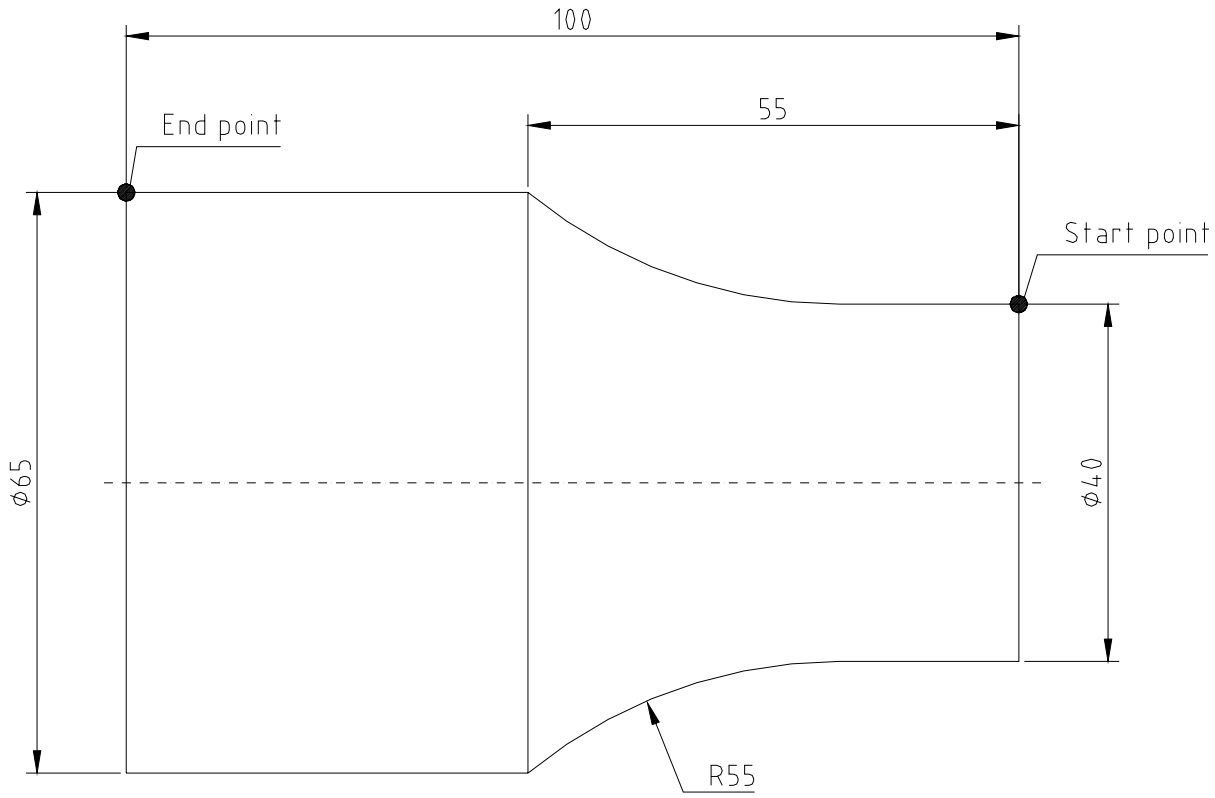
Side A and B and angle β

We have:
C = 75.716
α = 32.335

Solution:

OKUMA

Practical Exercise



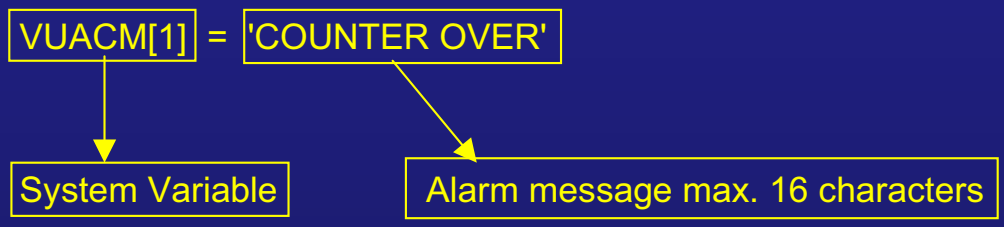
Please make a macro for the workpiece shape above.

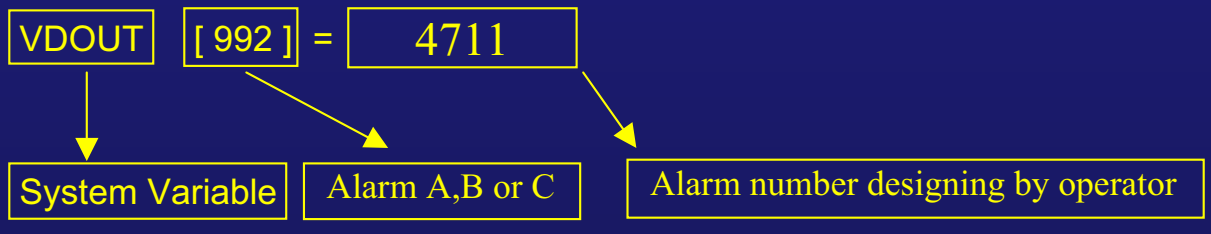
OKUMA

Example for make an counter program with alarm message

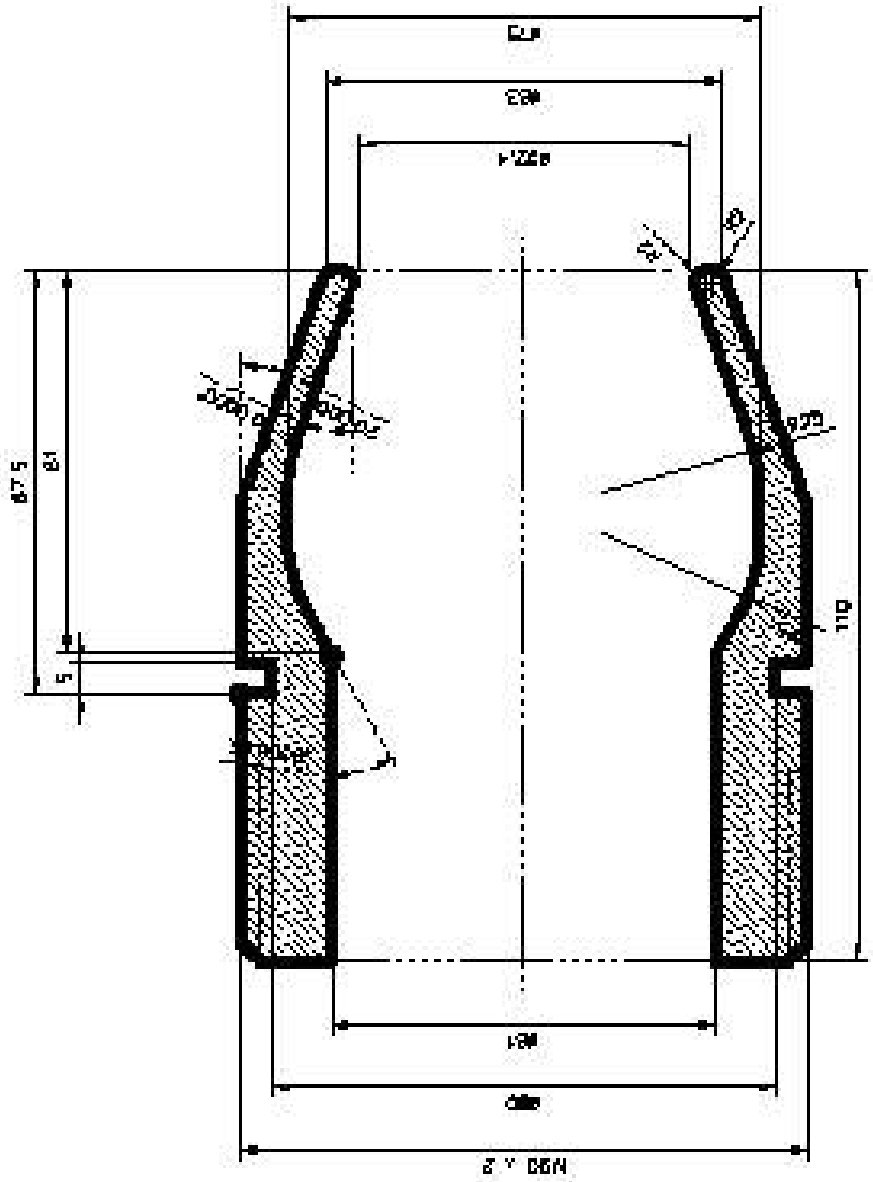
```
V1=0
V2=20
N10
N20
.
.
.
.
.
.

N90
N100
V1=V1+1
IF[V1 GE V2] NALM
GOTO N10
NALM VUACM[1]='COUNTER OVER'
VDOUT[992]=4711
M2
```

Alarm message programming



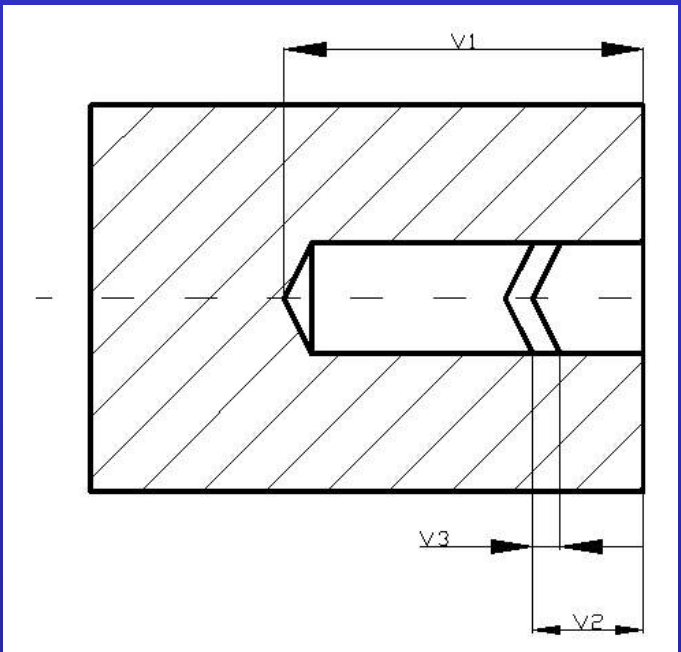


OKUMA



	DIN7168 fein mittel grob	Oberflächen	Material	Position	Menge
			C45 ∅ 95 x 112		
		Datum	Name	Buchse	
		Bearb.			
		Gepr.			
		Norm			
				4711	Blatt
Zust.	Änderung	Datum	Name	DIN-Nr. Buchse	

Test



Please make a macro for the deep hole drilling, after every step, drill should retract at the Z - position where the macro starts.

V1=50 (Z – endpoint of hole)
V2=5 (Depth of cut per peck feed)
V3=0.5 (Approaching distance)

OKUMA