Computer Aided Manufacturing

Dr.S.RAMABALAN,
PRINCIPAL,
E.G.S. PILLAY ENGINEERING COLLEGE,
NAGAPATTINAM.

Cleansing through Breathing

1 min

Recap and review of previous class

5 mins

Unit III

PROGRAMMING OF CNC LATHE

Coordinate system - structure of a part program -G & M Codes - Programming for FANUC and SIEMENS controller -Single pass and canned cycle -Turning, facing and threading -Multi-pass canned cycle -Rough and Finish turning, facing, pattern repeating, grooving, threading, drilling, boring, peck drilling, high speed drilling cycle - **Subprogram and Macro programming -Tool length and nose radius compensation - offset -Tool, work and coordinate -Insert - Materials, Classification, Nomenclature and Selection** -Tool and Work holding devices -Automatic tool changer -Turret and drum type -Tool holder nomenclature and selection -CNC part programming using CAD/CAM software and interfacing with CNC machines

Prerequisite Knowledge

- CNC Programming

3.EVOCATION

4. General Objective (GO)

- Students will be able to apply the appropriate programming method for fabrication of components with necessary settings.

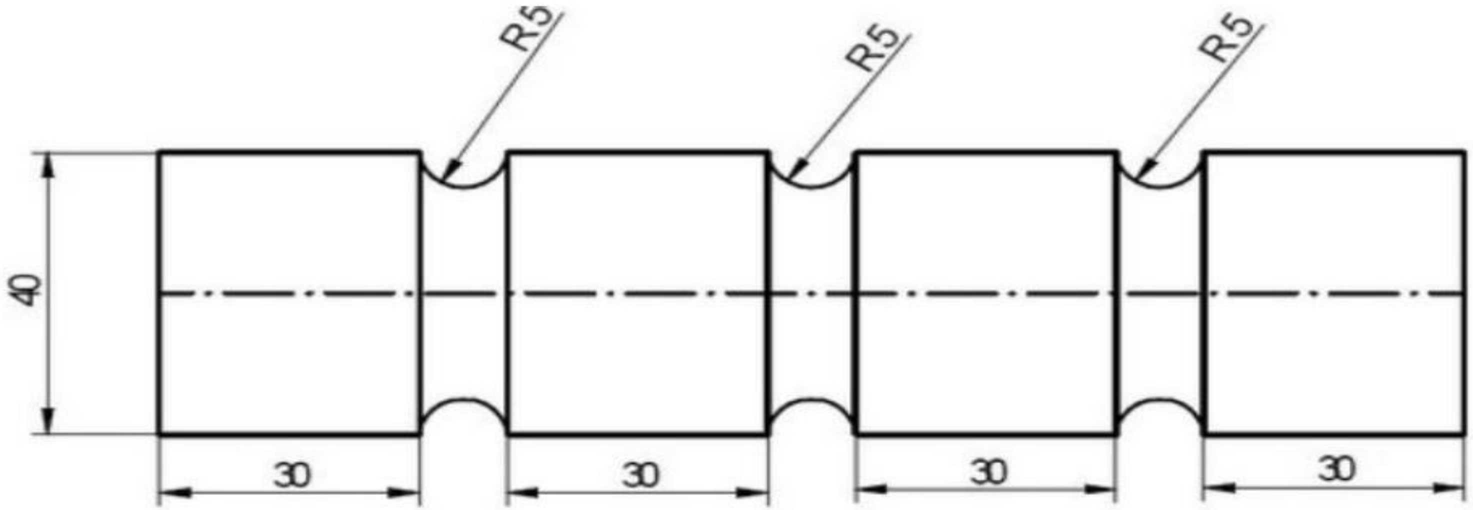
5. SPECIFIC OBJECTIVE (SO) MAPPED WITH STEM

The students will be able to

1. Exemplify the needs of the subprogram and macro program concept in CNC part programming. (U-C) (E)
2. Exemplify the tool nose radius compensation for different lathe operations. (U-C) (E)
3. Compare tool offset and work offset in CNC lathe. (U-C) (E)
4. Explain the four types of insert materials, nomenclature and selection of insert for given work material. (U-C) (E)

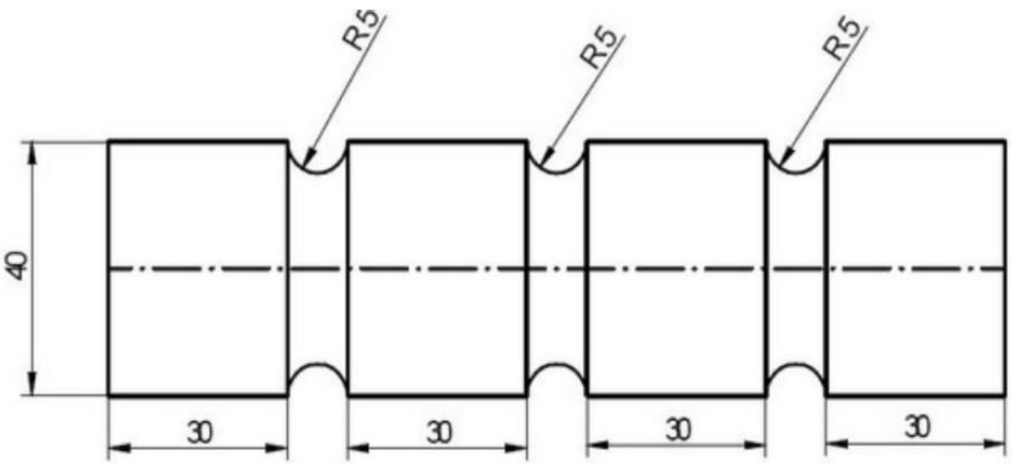
CONCEPT OF SUBPROGRAM

- CNC lathe program which calls a subprogram multiple times to cut the same pattern.
- Subprogram uses UW instead of XZ to make program easier to understand and debug.



SUBPROGRAM

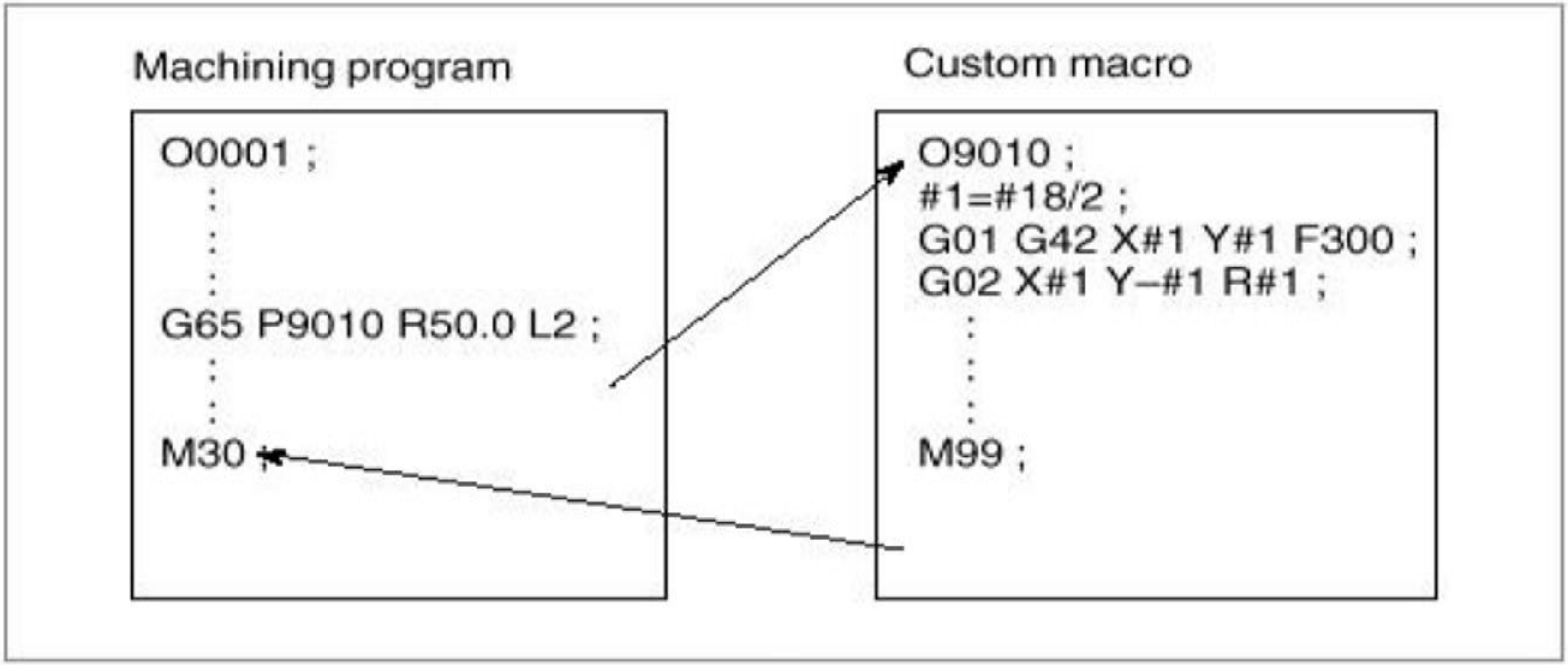
- Main Program:
- O0005
- N5 G90 F0.2 S1200 T0101 M04
- N10 G00 X40 Z0
- **N20 M98 P37000 (call subprogram O7000 three times)**
- N30 G01 W-30
- N40 G28 U0 W0
- N50 M05 M30



- Subprogram:
- O7000;
- G01 U0 W-30
- G02 U0 W-10 R5
- N60 M99

MACRO PROGRAMMING

- Macros can be used to
  - ✓ automate repetitive tasks
  - ✓ document what you did
  - ✓ share common procedures
  - ✓ add tools to the toolbar
  - ✓ add keyboard shortcuts



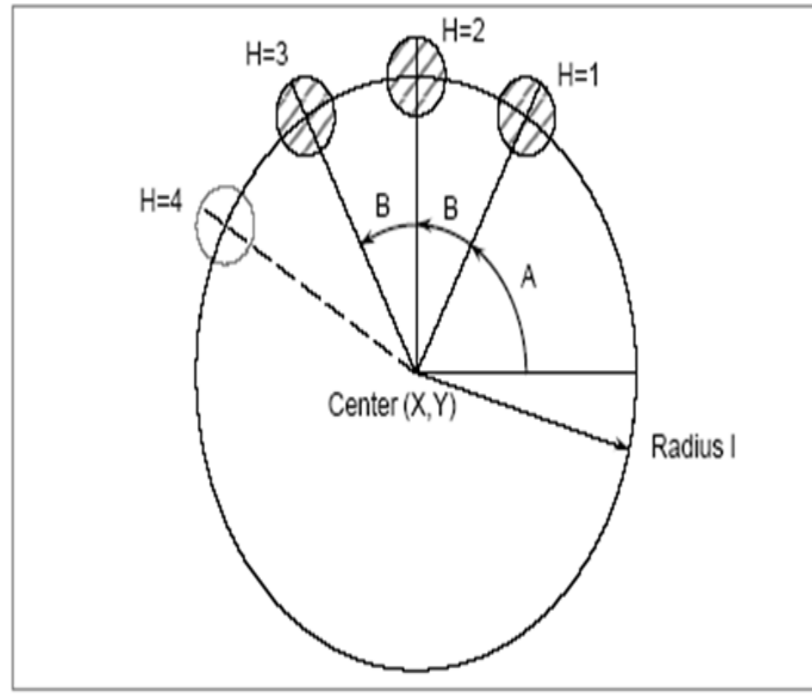
- Part Program prepared using variables
- Arithmetic operations possible
- Conditional statements like GOTO , DO WHILE are used
- For developing user defined cycles
- Can be easily called from user program

Range of Custom Macro Variables

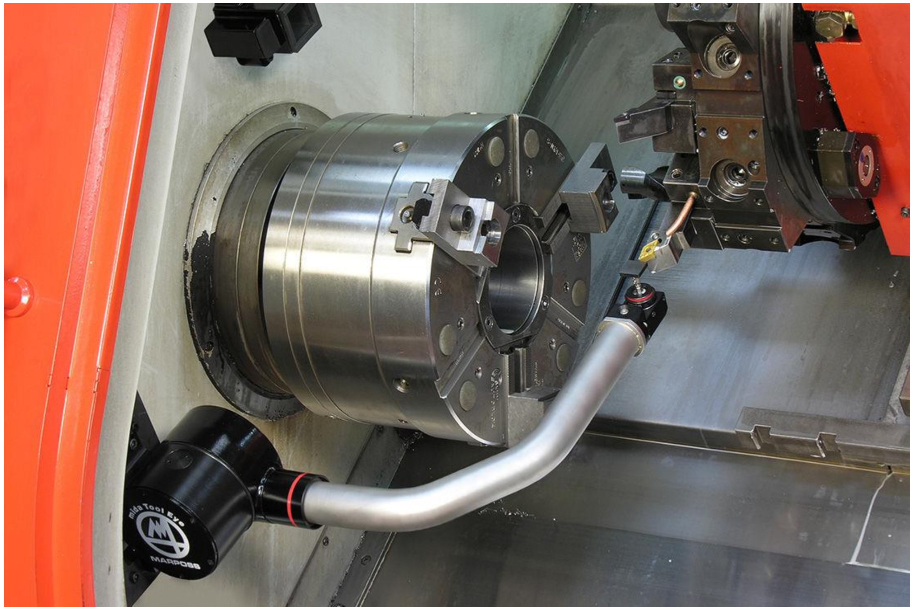
Variable number	Type of variable	Function
#0	Always null	This variable is always null. No value can be assigned to this variable.
#1 – #33	Local variables	Local variables can only be used within a macro to hold data such as the results of operations. When the power is turned off, local variables are initialized to null. When a macro is called, arguments are assigned to local variables.
#100 – #199 #500 – #999	Common variables	Common variables can be shared among different macro programs. When the power is turned off, variables #100 to #199 are initialized to null. Variables #500 to #999 hold data even when the power is turned off.
#1000 –	System variables	System variables are used to read and write a variety of NC data items such as the current position and tool compensation values.

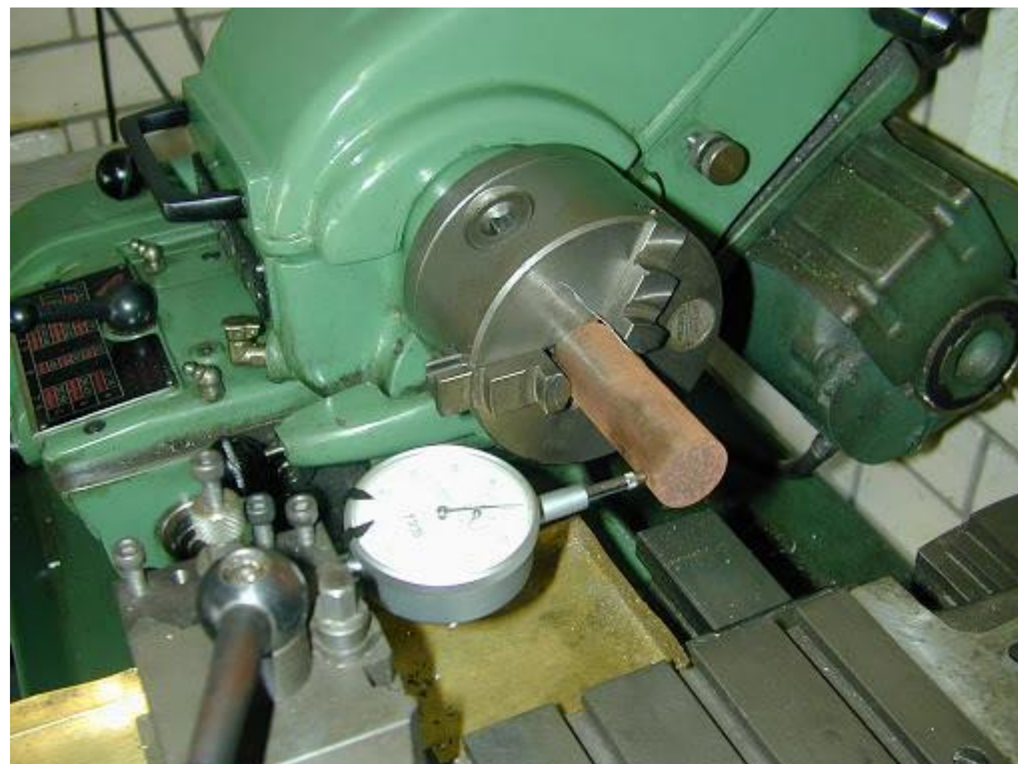
Example – 1

- **O0002;**
- N10 G21 G94;
- N20 G91 G28 Z0;
- N30 T01 N06 D01;
- N40 G90 G54 X0 Y0 Z100;
- N50 G65 P5000 X100 Y50 R30 Z-50 F500 I100 A0 V45 H5; N60 M30;
-
- **O5000;**
- N10 #3=#4003;
- N20 G81 2#26 R#18 F#9 K0;
- N30 N1 WHILE[#11GTO]D01;
- N40 #5=#24+#4*COS[#1];
- N50 #6=#25+#4*SIN[#1];
- N60 G90 X#5 Y#6;
- N70 #1=#1+#2;
- N80 #11=#11-1;
- N90 END1;
- N100 G#3 G80;
- N110 M99;



Tool length and nose radius compensation


mida Tool Eye
MARPOSS

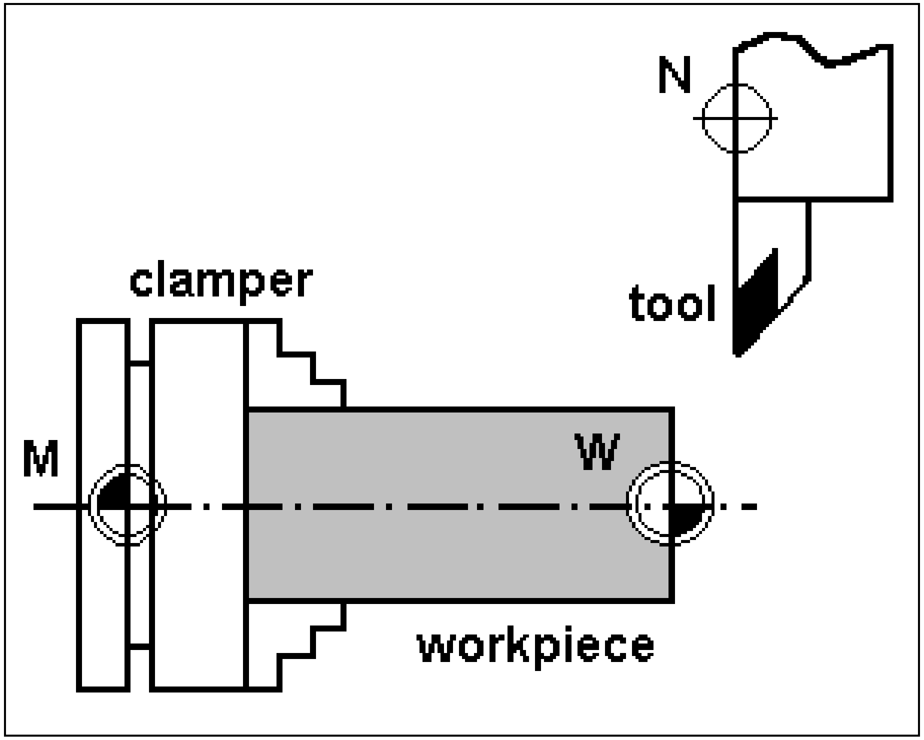


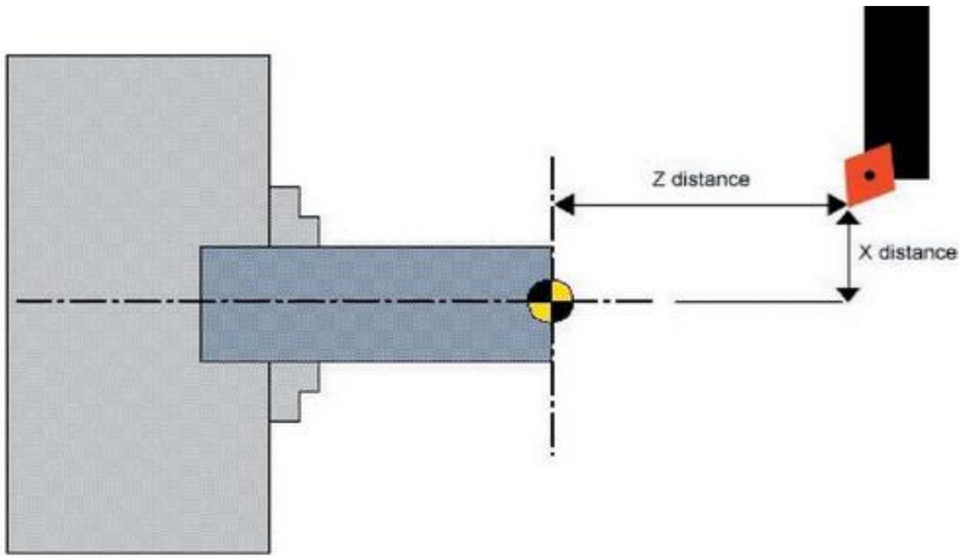
WORK COORDINATE SYSTEM OR WORK ZERO OR WORK OFFSETS (G54-G59)

- **Machine Zero** – Machine Builder
- **Work Zero** – Point Assumed in the work-piece for giving its dimensional value called as work zero.
- **Offset** – the distance of W/s zero from M/c zero is called offset.
- Basic work offsets are very simple to specify: simply enter one of G54, G55, G56, G57, G58, or G59. Most machine startup with G54 selected. It's a good practice to put a G54 into the safety line at the top of all of
- your g-code programs to make sure you know what work offset is being used unless you have reason to want to leave that aside.

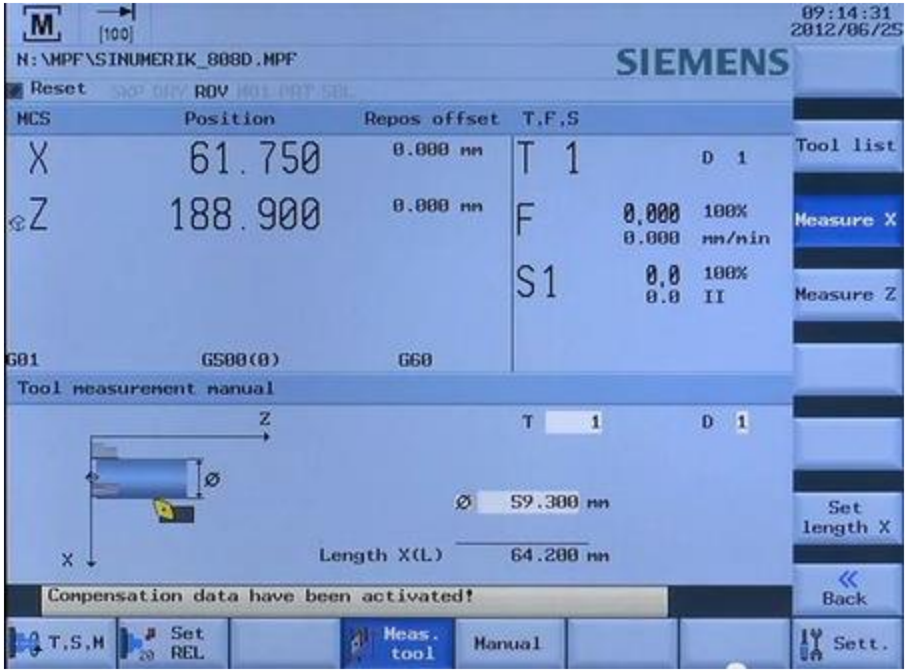
Reference Points

M = Machine zero point. (unchangeable ref. Point determined by machine manufacturer)
N = Tool mounting reference point
W = Work piece zero point. (can be freely determined by the programmer and can be moved within program

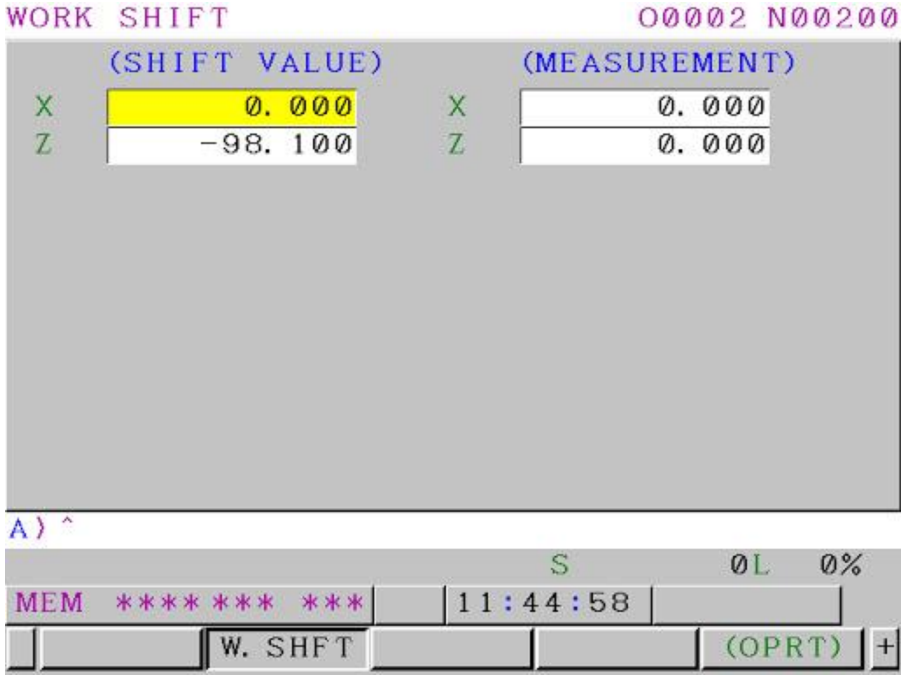




Z distance
X distance

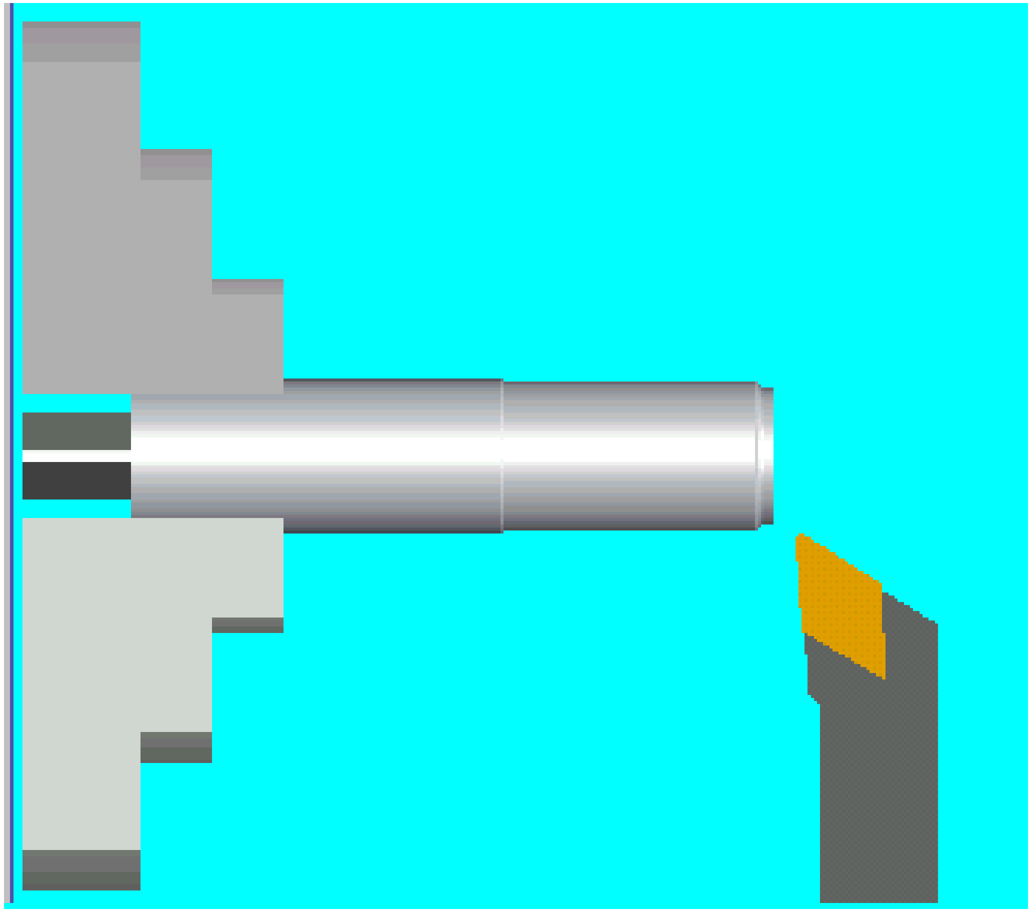


09:14:31
2012/06/25
N:\MPF\SINUMERIK_808D.MPF
SIEMENS
Reset
ROV
MCS
Position
Repos offset
T,F,S
X
61.750
0.000 mm
Z
188.900
0.000 mm
T 1
D 1
F
0.000
100%
0.000
mm/min
S1
0.0
100%
0.0
II
G01
G500(0)
G60
Tool list
Measure X
Measure Z
Tool measurement manual
T 1
D 1
Ø 59.300 mm
Length X(L) 64.200 mm
Set length X
Back
Compensation data have been activated!
T,S,M
Set REL
Meas. tool
Manual
Sett.

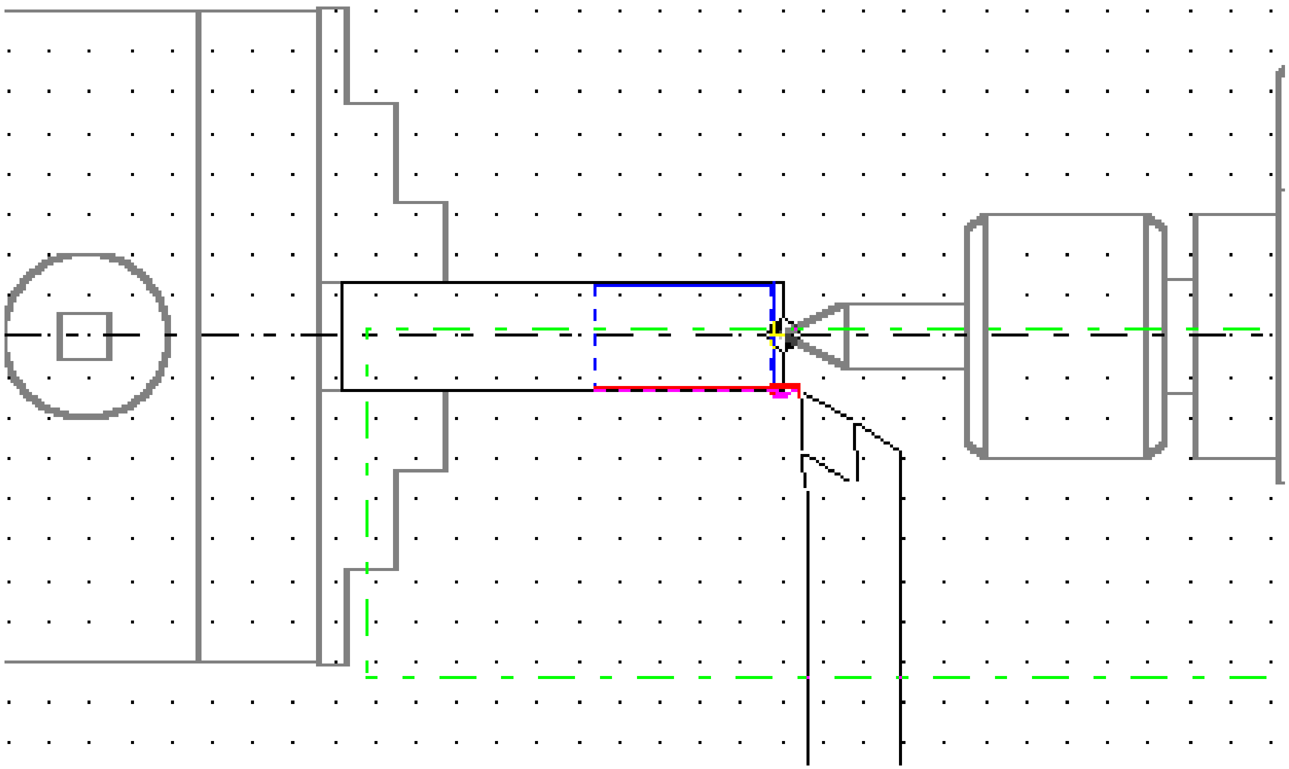


WORK SHIFT
O0002 N00200
(SHIFT VALUE)
(MEASUREMENT)
X 0.000
Z -98.100
X 0.000
Z 0.000
A)^
S 0L 0%
MEM **** *** ***
11:44:58
W. SHFT
(OPRT)
+

Work piece (1)



Work piece (2)



TOOL NOSE RADIUS COMPENSATION

- **Function and purpose**
- The tool nose is generally rounded and so a hypothetical tool nose point is treated as the tool nose for programming .with such a programming, an error caused by the tool rounding arises during taper cutting or arc interpolation between the actually programmed shape and the cutting shape. Nose R or Tool radius compensation is a function for automatically calculating and offsetting this error by setting the nose radius or tool radius value.

TOOL NOSE RADIUS COMPENSATION

- **Programming Format:-**
- G40 = Nose R/Tool radius compensation mode cancel
- G41 = Nose R/Tool radius compensation left mode ON
- G42 = Nose R/Tool radius compensation right mode On

1. G40 serves to cannel the tool nose radius compensation mode.

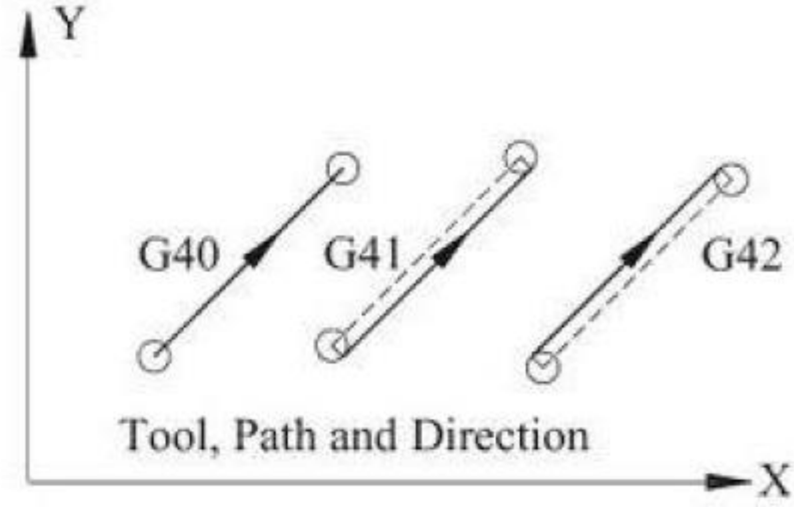
2. Tool nose radius compensation function pre reads the data in the following two move command blocks and controls the tool nose radius center path by the intersection point calculation method so that it is offset from the programmed path by an amount equivalent to the nose radius .

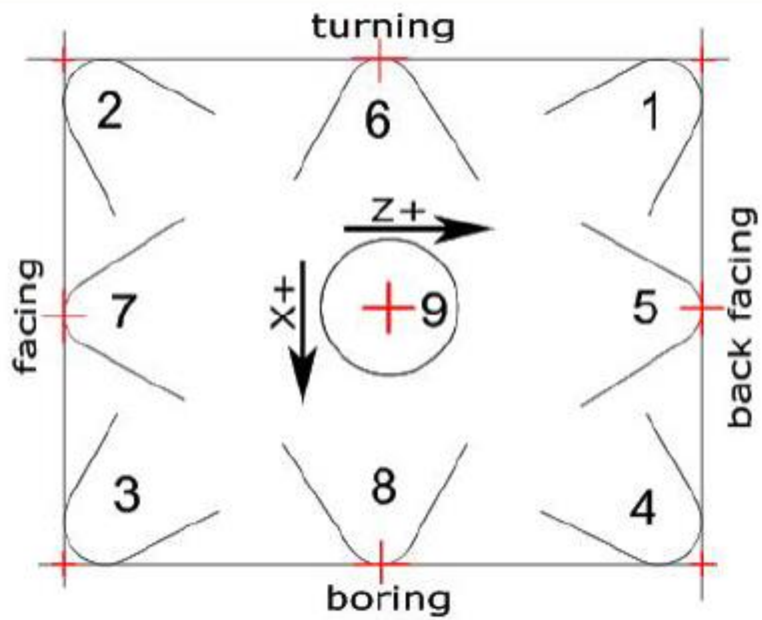
3. The tool nose radius compensation amount corresponds to the tool length number and it should be preset with the tool nose point.

4. Tool nose radius compensation function is also valid for fixed cycles (G77 to G79 or *G90 to G92) and for roughing cycles (G70, G71, G72 and G73). However, in the roughing cycles, the tool nose radius compensation function

5. If four or more blocks without move commands exist in five continuous blocks, overcutting or undercutting will result. However, blocks in which optional block skip is valid are ignored. Applied for finish shape is cancelled and upon completion of the roughing, NC unit will re-center the compensation mode.

6. With threading commands, compensation is temporarily cancelled in one block before.


Y
G40
G41
G42
Tool, Path and Direction
X


turning
2
6
1
Z+
facing
7
X+
9
5
back facing
3
8
4
boring

INSERT

- Another name for a tipped tool, a cutting tool used in metalworking.
- General turning inserts come in variety of shapes and sizes. One thing is important to remember and understand is that every turning insert has a nomenclature associated with it.

Turning Inserts

- Turning inserts are specified by their ANSI insert designation that provides a shorthand method for identifying the shape and size of an insert.

MATERIALS

- Carbide
- Ceramic
- Cermet
- Diamond

CLASSIFICATION

- Turning Inserts
- Grooving Inserts
- Threading Inserts
- Drilling Tools
- Boring Tools


SCMT-HM
SD
SEET-DR
SEET-DM
SEHT
SEKN
SNKN
SNMA
SNMG-DF
SNMG-DF
SNMG-DM
SNMG-DM
SNMG-EF
SNMG-PM
SPG
SPKN
SPU
TCMT-HF
TCMT-HM
TCMT-HR
TNMA
TNMG
TNMG-DF
TNMG-DF
TNMG-DM
TNMG-DM
TNMG-EF
TNMG-PM
TNMM
TNMM
TPGH
TPG
TPGN
TPKN
TT/TPMM
TRMR
TPU
TPUN
VBMT-HM
VBMT-HR
VNMG-DF
VNMG-DF
VNMG-DM
VNMG-DM
VNMG-EF
VNMG-PM
WCMX
WNMA
WNMG-DF
WNMG-DM
WNMG-EF
WNMG-PM

Nomenclature- Insert

T N M G — 4 3 2 — (..)

SHAPE

A – Parallelogram 85°
B – Parallelogram 82°
C – Diamond (Rhombic) 80°
D – Diamond (Rhombic) 55°
E – Diamond (Rhombic) 75°
H – Hexagon
K – Parallelogram 55°
L – Rectangle
M – Diamond (Rhombic) 86°
O – Octagon
P – Pentagon
R – Round
S – Square
T – Triangle
V – Diamond (Rhombic) 35°
W – Trigon 80°

RELIEF ANGLE

N – 0°
A – 3°
B – 5°
C – 7°
P – 11°
D – 15°
E – 20°
F – 25°
G – 30°
NP – 0° 10°

TOLERANCE CLASS

Insert Inscribed Circle (I.C.)	Thickness
A = ±.0002	±.001
B = ±.0002	±.005
C = ±.0005	±.001
D = ±.0005	±.005
E = ±.001	±.001
[1,2]**F** = ±.002 to ±.004	±.002
G = ±.001	±.005
[1]**M** = ±.002 to ±.010	±.005
[1]**U** = ±.005 to ±.012	±.005

SIZE (I.C.)

Number of $\frac{1}{8}$ths on inserts $\frac{1}{4}$″ Inscribed Circle (I.C.) and over.

Number of $\frac{1}{32}$nds on inserts less than $\frac{1}{4}$″ I.C.

Rectangle and parallelogram inserts require two digits.

1st digit – No. of $\frac{1}{8}$ths in widths

2nd digit – No. of $\frac{1}{4}$ths in length

CUTTING POINT CONFIGURATION

0 – Sharp corner (.005R)
1 – $\frac{1}{64}$ radius
2 – $\frac{1}{32}$ radius
3 – $\frac{3}{64}$ radius
4 – $\frac{1}{16}$ radius
6 – $\frac{3}{32}$ radius
8 – $\frac{1}{8}$ radius
12 – $\frac{3}{16}$ radius
A – Square insert w/45° chamfer
D – Square insert w/30° chamfer – R
E – Square insert w/15° chamfer – R
[2]G – Square insert w/30° chamfer – L
[2]H – Square insert w/15° chamfer – L
K – Square insert w/15° dbl. chamfer
L – Square insert w/15° dbl. chamfer
N – Truncated triangle insert
P – Flattened corner triangle – R
[2]R – Flattened corner triangle – L

INSERT TYPE

A – With hole
B – With hole and one countersink
C – With hole and two countersinks
D – Smaller than $\frac{1}{4}$″ I.C. with hole
E – Smaller than $\frac{1}{4}$″ I.C. clamp type
F – Clamp type with chip groove on both top surfaces
G – With hole and chip groove on both top surfaces
H – With hole, one countersink and chip groove on one top surface
J – With hole, two countersinks and chip groove on both top surfaces
K – Smaller than $\frac{1}{4}$″ I.C. with hole, and chip groove on both top surfaces
L – Smaller than $\frac{1}{4}$″ I.C. clamp type, and chip groove on both top surfaces
M – With hole and chip groove on one top surfaces (except Pos./Neg. insert)
[2]**N** – Clamp type with chip groove except ceramic–no chip groove
P – With hole and 10° chip groove on both top surfaces (except Pos./Neg. insert)
R – Without hole, with chip groove on one top surface
[2]**U** – Smaller than $\frac{1}{4}$″ I.C. clamp type and with chip groove on one top surface
X – Special design

THICKNESS

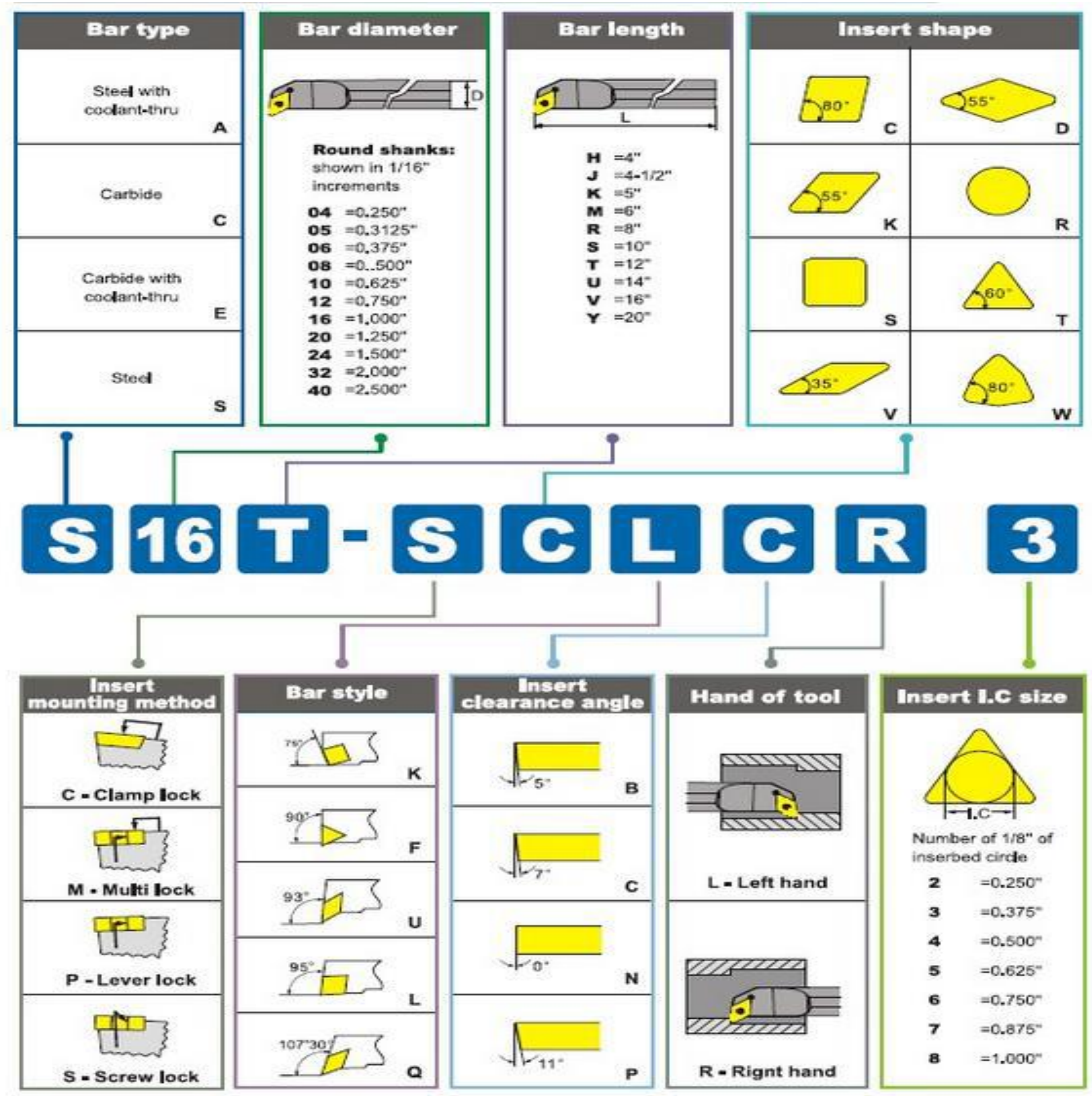
Number of $\frac{1}{32}$nds on inserts less than $\frac{1}{4}$″ I.C.

Number of $\frac{1}{16}$ths on inserts $\frac{1}{4}$″ I.C. and over

CHIPBREAKER

[1]*Exact tolerance is determined by the size of the insert.*
[2]*Valenite standard only.*

Nomenclature- Tool Bar



SELECTION

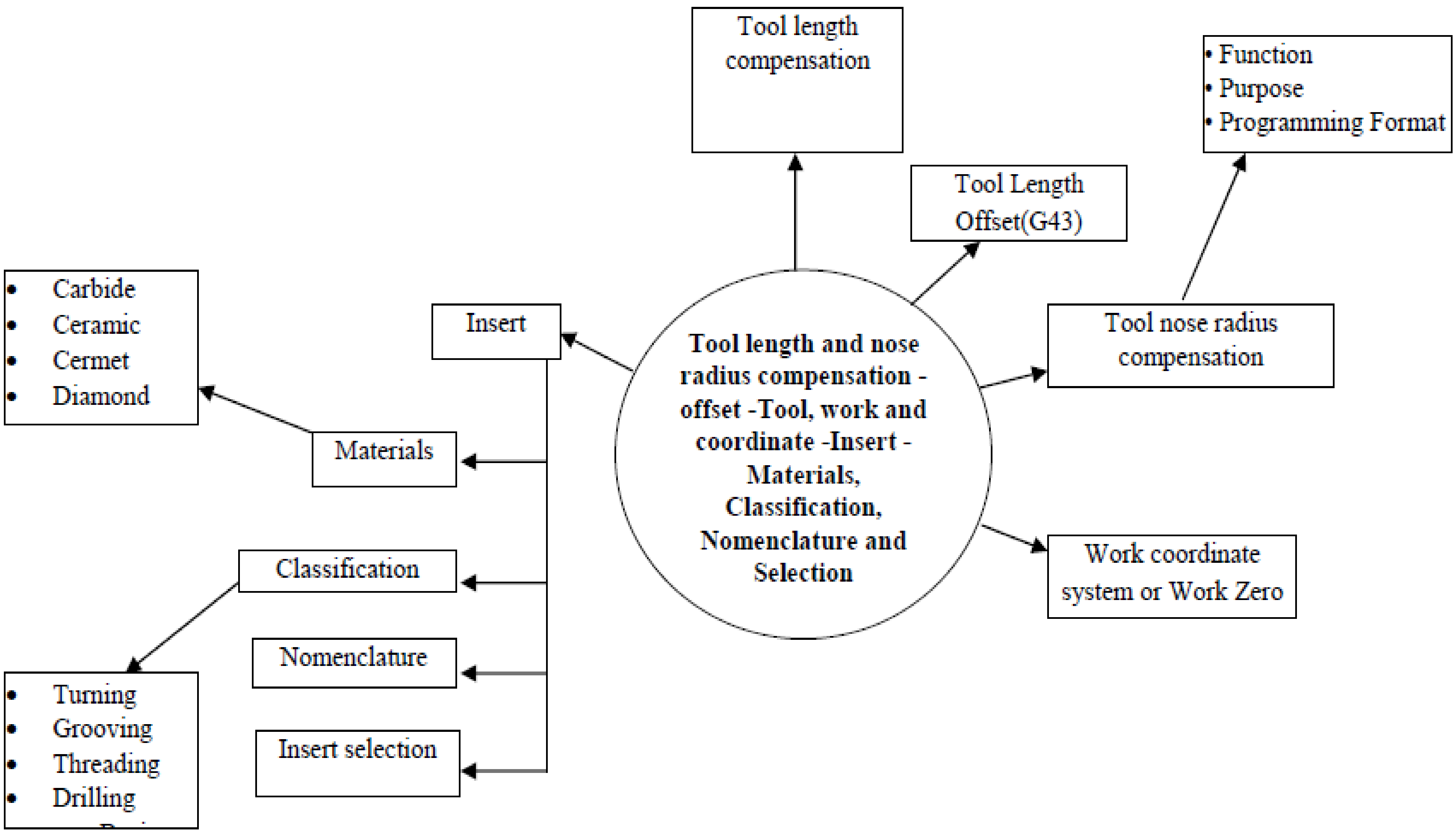
1. What would be your primary material for machining? **Stainless Steel, carbon steel, cast iron , titanium** etc. Choose the appropriate grade from manufacturer's **catalog for your application material**.

2. Choose the manufacturer. For ex **Sandvik, Walter and ISCAR** have excellent life but are expensive. Kyocera and TAEGUTEC-Korea offer best value for money.

3. For general turning, go for **WNMG 060408 inserts for roughing. For profiling and finishing, VNMG12T304 inserts** .

4. You would also require tools for boring, drilling, grooving, parting, face grooving, etc.

Mind map


Tool length compensation
• Function
• Purpose
• Programming Format
Tool Length Offset(G43)
Carbide
Ceramic
Cermet
Diamond
Insert
Tool length and nose radius compensation - offset -Tool, work and coordinate -Insert - Materials, Classification, Nomenclature and Selection
Tool nose radius compensation
Materials
Classification
Work coordinate system or Work Zero
Nomenclature
Turning
Grooving
Threading
Drilling
Insert selection

Discussion

10 mins