ثانويـة التكنولوجيـا التطبيقيـة
Applied Technology High School

CNC Turning

Module2: Introduction to MTS-TopTurn and G & M codes

PREPARED BY

Academic Services

January 2013

Module2: Introduction to MTS-TopTurn and G & M codes

Module Objectives:

Upon the successful completion of this module, the student will be able to:

1. Demonstrate familiarity with the MTS Simulator.
2. Create a setup sheet.
3. Identify the different parts that compose the NC program.
4. Identify the difference between rapid positioning and feed rate positioning.
4. Utilize both G-codes and M-codes.
5. Demonstrate the ability to write a simple NC part program.

Module Contents

1. CNC Machine Simulator:

A simulator is used to create NC programs, simulating them and checking their quality. These programs can be transferred and executed on the real CNC machine to produce the work part. The simulator that we are going to deal with throughout the term is called the **MTS Simulator**.

2. MTS simulator software:

2.1 How to start the software?

To start the MTS simulator:

- Click on windows **"start"** button in the task bar.
- Point on **"all programs".**
- Point on **"MTS TopCAM 7.4"**.
- Click on **"TOPCAM".**
- Or click on the "**MTS TopCAM**" shortcut on the desktop.

The start-up menu shown in Fig. 2.1 will offer you the following software modules for selection: TOPCAM, TOPTURN and TOPMILL.

TOPTURN is used for turning programming. However, TOPMILL is used for milling programming. As Turning is introduced in this course so you will choose TOPTURN to start turning.

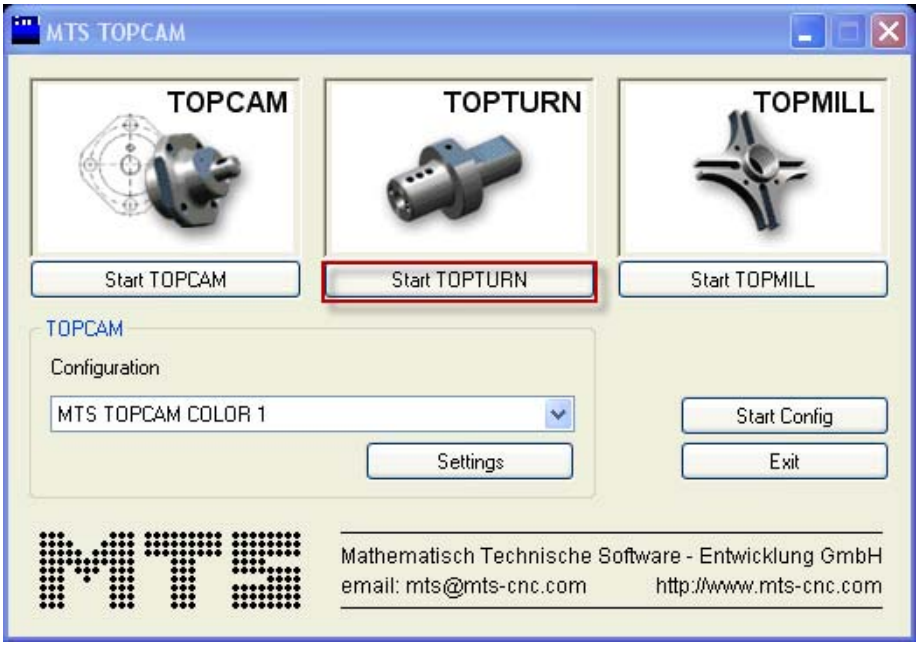


Fig. 2.1: The start-up menu

The next window that appears is to select the controller. The controller is the link between the software (NC program) and the hardware (CNC machine). From the controllers list, choose the **"FANUC 21 TB Code-C EMCO@EMCO TURN55 R1 T8VDI10"** since it is the one used by the EMCO CNC Turning Machine available in the lab, then click on "start turning" as shown in Fig.2.2.

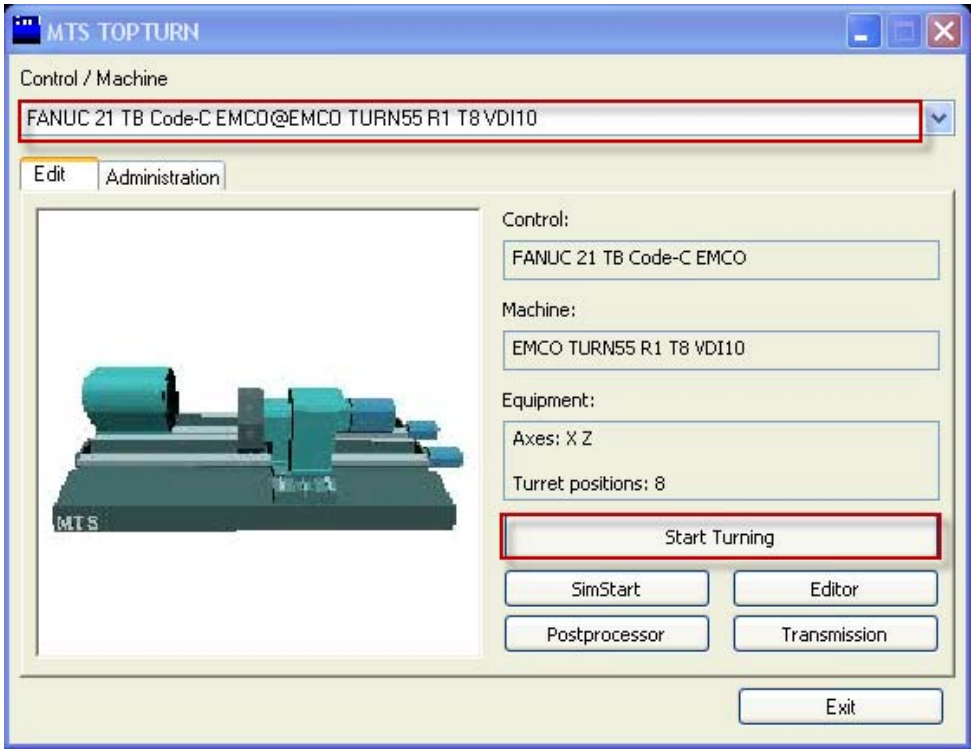


Fig. 2.2: Selecting the controller

MTS TOPTURN simulator will start. The main areas of the screen are shown on Fig. 2.3.

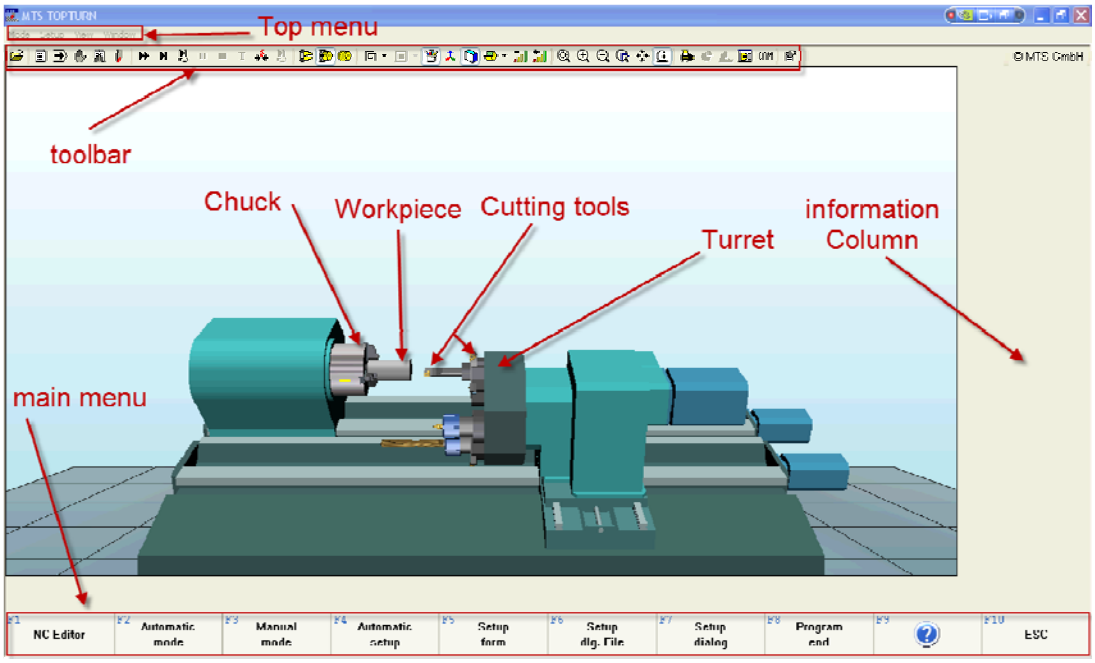


Fig. 2.3: MTS TOPTURN simulator interface

2.2 How to change the display?

1. Graphic display modes:

Click on **"Window"** on the top menu bar as shown in Fig.2.4, You will find seven different views. Try each view to find the difference between them.

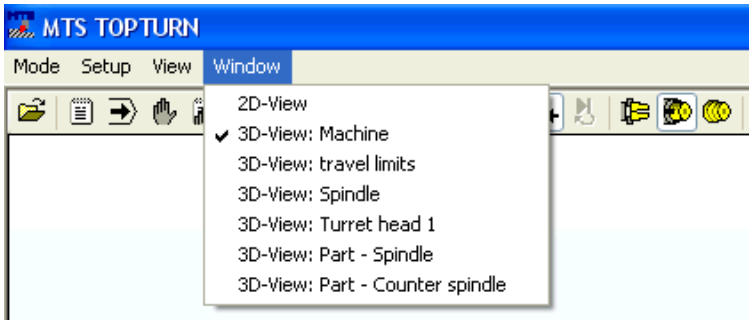


Fig. 2.4 Display views

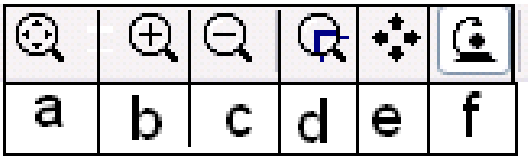
The following icons are also available on the toolbar for quick access for the same views offered under window:

a. 2D Graphic : shows the simulation in 2D.

b. 3D graphic : shows the simulation in 3D.

c. 3D graphic (part) : shows the simulation in 3D (workpiece only).

2. Display modes icons:



a. Reset: Reset the view after using the icons below.
b. Scale up: to zoom in
c. Scale down: to zoom out
d. Zoom rectangle: to zoom in by using a selected area.
e. Move 3D graphic: to move the 3D graphic up and down, right and left. (Drag)
f. Study 3D graphic: to rotate the 3D graphic around 3 axes. (Drag on the machine)

3. Setup Sheet:

3.1 Create a setup sheet

In order to set the machine, a setup sheet is created. The setup means to enter the information of the tools, workpiece dimensions, material used, holding devices and the position of zero point.

Example 1:

Prepare a "Setup sheet" with the following information:

- Workpiece length: *60 mm*
- Workpiece diameter: *40 mm*
- Workpiece material: *Aluminium/Almg1*
- The chucking depth: *14 mm (Chucking depth is the length of the workpiece that inserted into the chuck)*
- Workpart zero location: *"Right side – center of the workpart surface*
- Tool position*: position 4 (The turret of the machine can hold many tools, so this is the position of the tool on the turret.)*
- Tool to be used:

  Tool type: *Corner Tool Left*,

  Tool Name: *CCMT 060202_SCACR 0808 B06_B 3.10 1213.*

Solution:

1. Click **"Setup dlg. File"** or "F6". On the main menu. Fig. 2.5

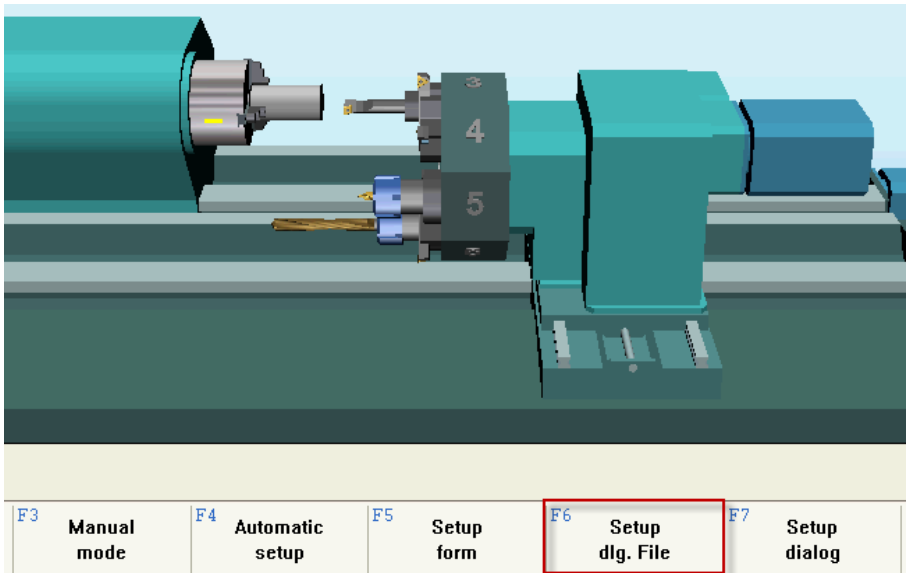


Fig. 2.5: MTS TOPTURN

2. Write the File name: The file name is "Setup1", then click "Open" as shown in Fig. 2.6

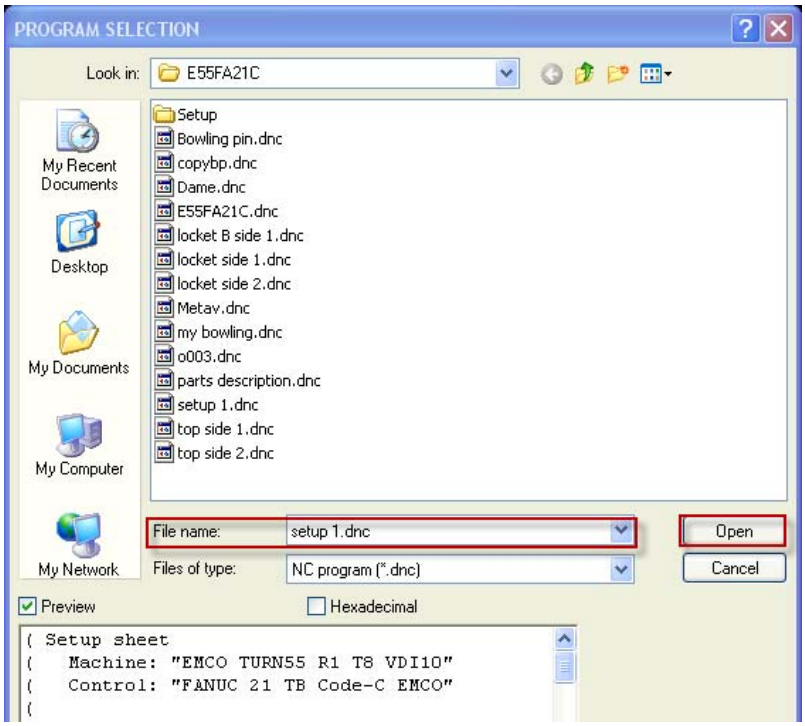


Fig. 2.6: Inserting the File name

3. Click on tab 1: **"General information"** and then enter the **"program number"** starting with "O" letter. In this example you can write "O1".

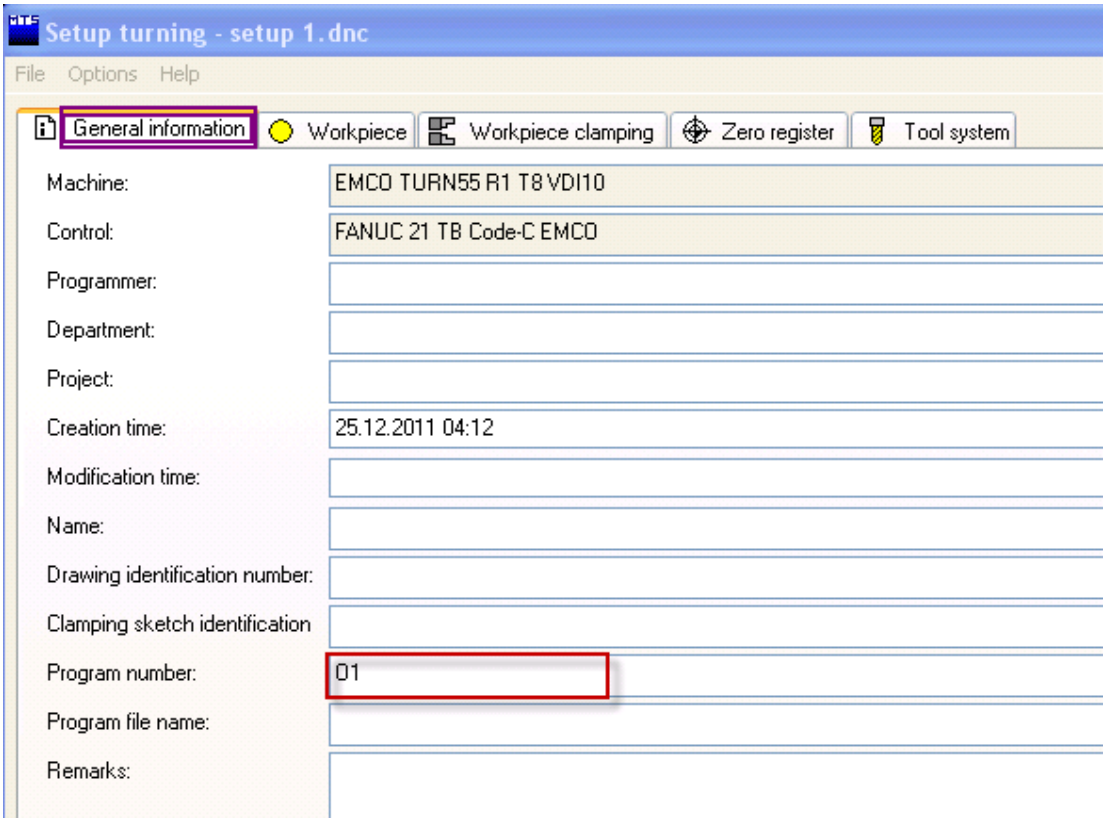


Fig. 2.7: General information tab

4. Click on tab 2: **"Workpiece"**, the **"Workpiece type"** list box, the cylinder is the default. Click the list box arrow to see the different shapes available in the software then select.**"Cylinder"** for the workpiece type. The next step is to fill in the information for the length, Diameter, and material type.
   a. **Length L**: "60"
   b. **Diameter D**: "40"
   c. For material selection click on the folder in the field of **"Material"**

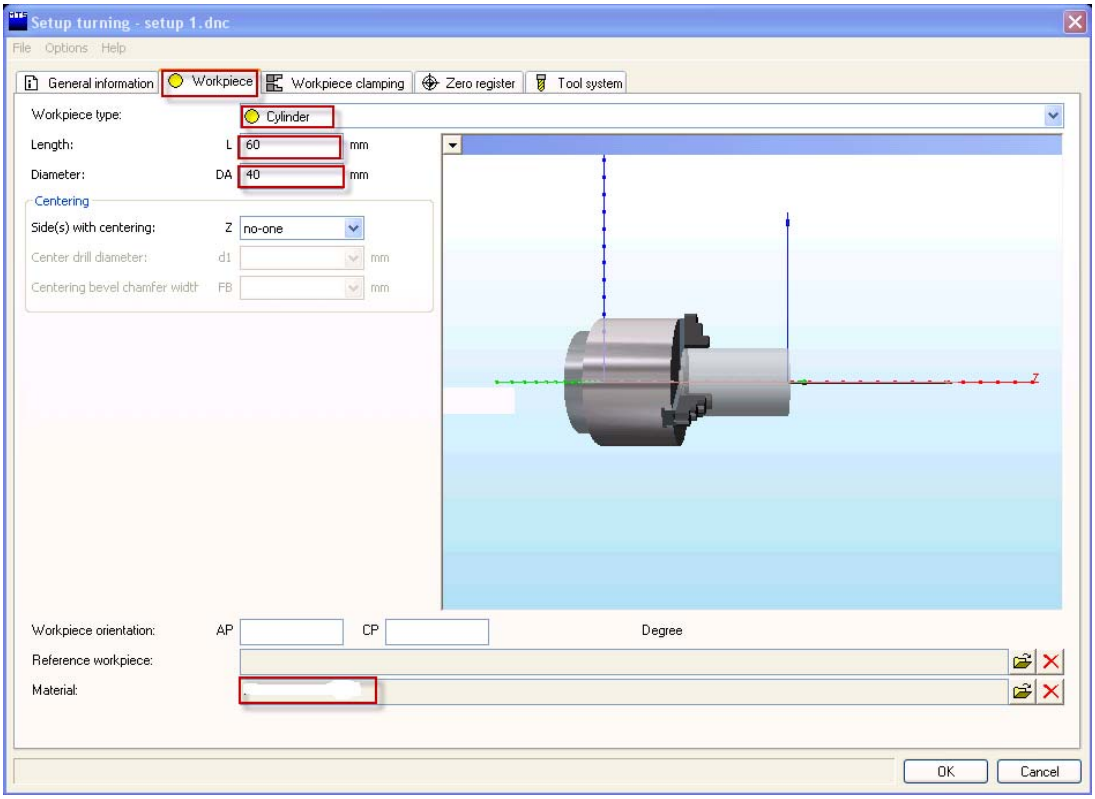


Fig. 2.8: Workpiece tab

d. Select **"Non ferrous folder"** or click on the green **"N"** to select the required material as shown in Fig. 2.9

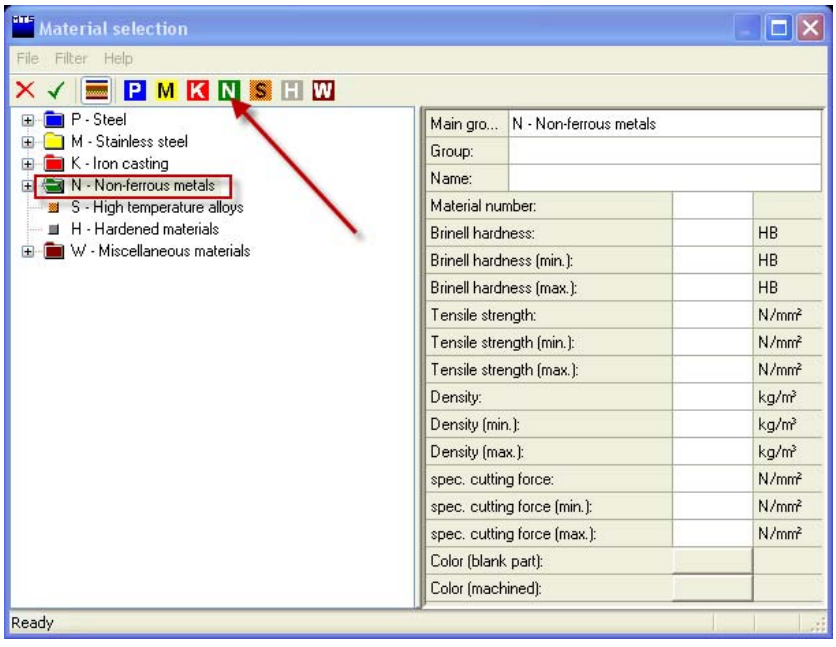


Fig. 2.9: Material class selection

e. Click on **"Aluminum folder".** Fig. 2.10

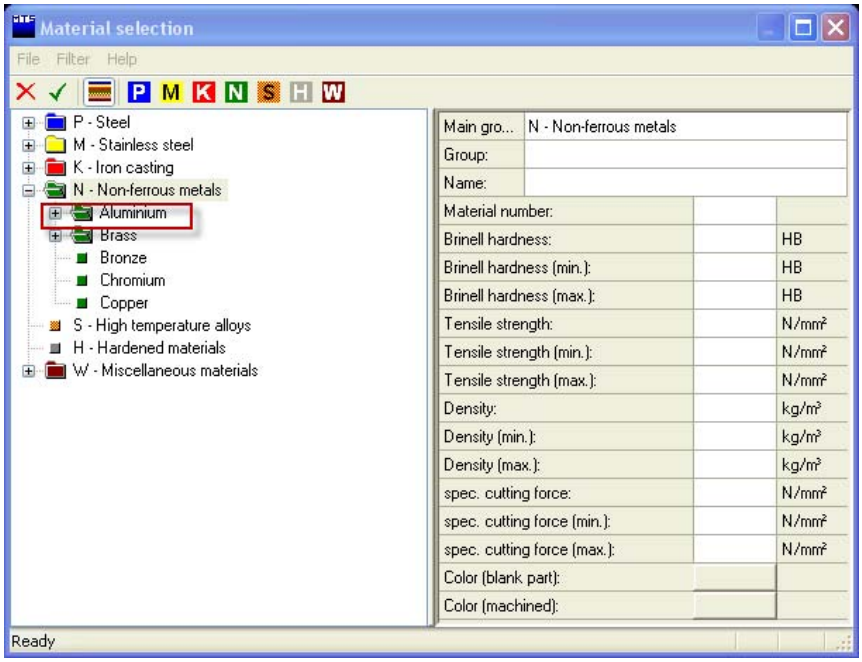


Fig. 2.10: Material selection

f. Click on **"Almg1",** then Click select and quit  . Fig. 2.11

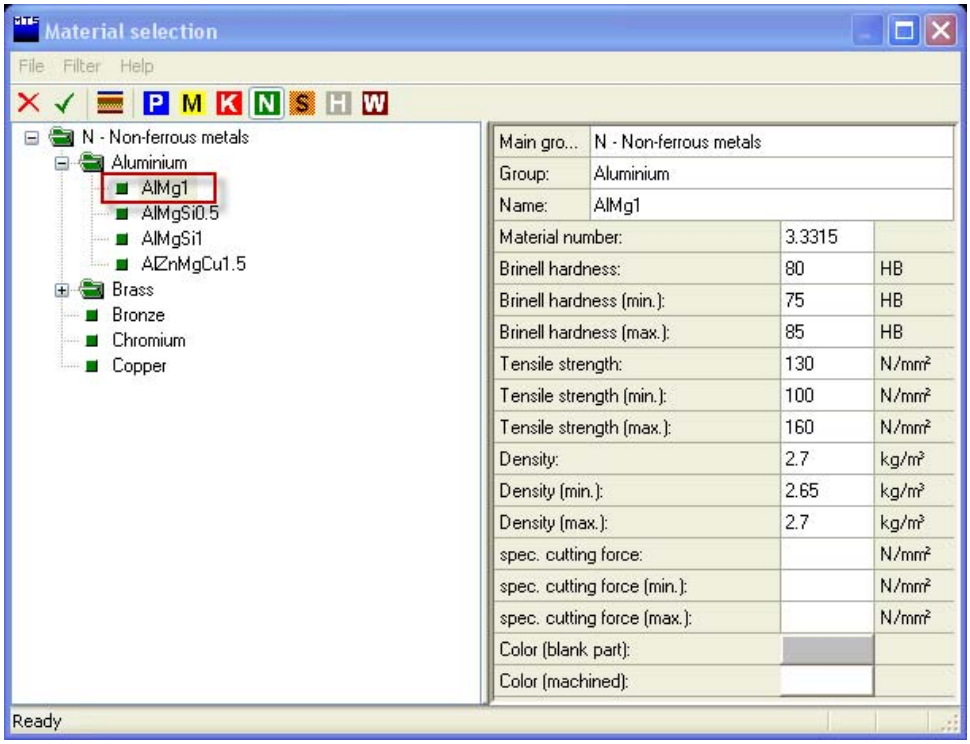


Fig. 2.11: selecting the Almg1 material

5. Click on tab 3: **"Workpiece clamping"** to fill in the clamping device information as shown in Fig. 2.12.
   - **Clamping device type**: "Lathe chuck"
   - **Lathe chuck:** "KSF 74 – 3AsB" (depend on the type and size of the workpiece)
   - **Chuck jaws:** Step jaw\HB 3–B09 x L14_7_4xH20_14_7 (depend on the type and size of the workpiece)
   - **Chucking depth:** "14"

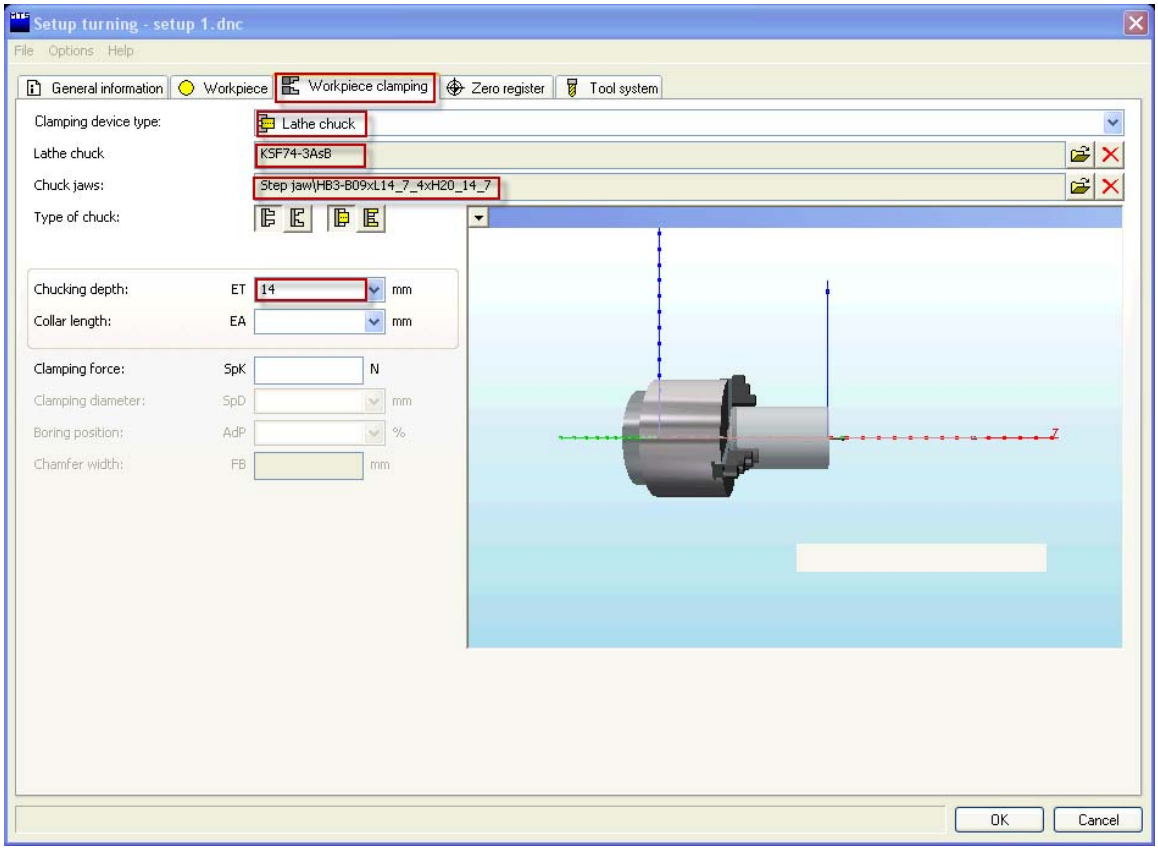


Fig. 2.12 Workpiece clamping tab

6. Click on tab 4: **"Zero register"** to fill in the information related to the workpiece Zero point as shown in Fig. 2.13.

   Specification of zero offset value relative to:
   - **"Right side – center of the workpart surface"**
   - G54 Z: **"0"** X:**"0"**

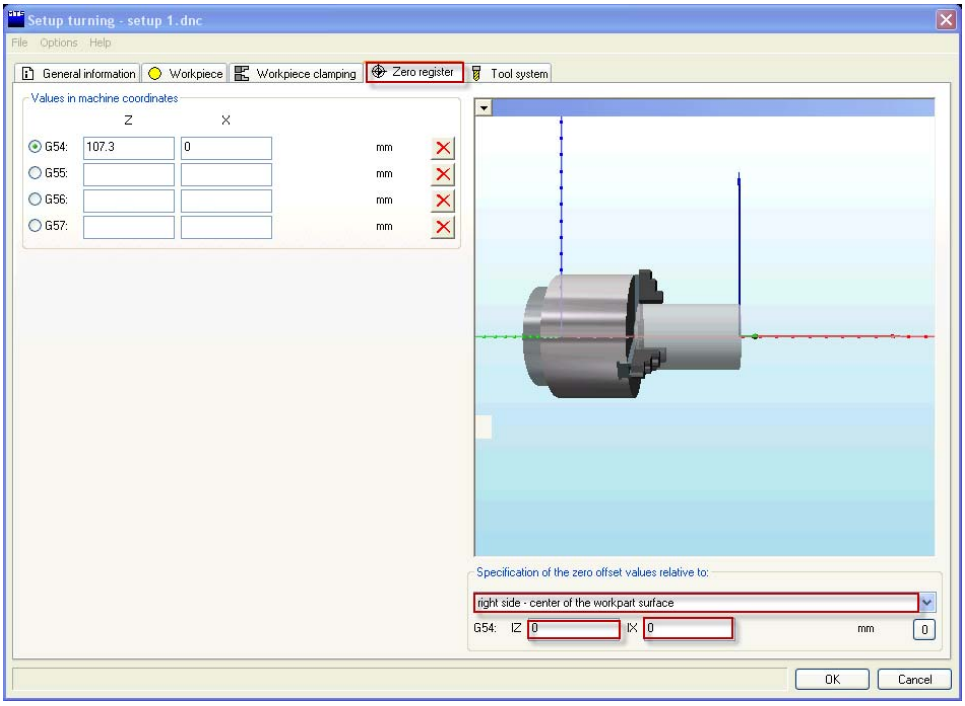


Fig. 2.13: Zero register tab

7. Click on tab 5: **"Tool system"** in order to select the tools that will be used in the program.
8. Click **"Edit"** as shown in Fig. 2.14 for tool selection.

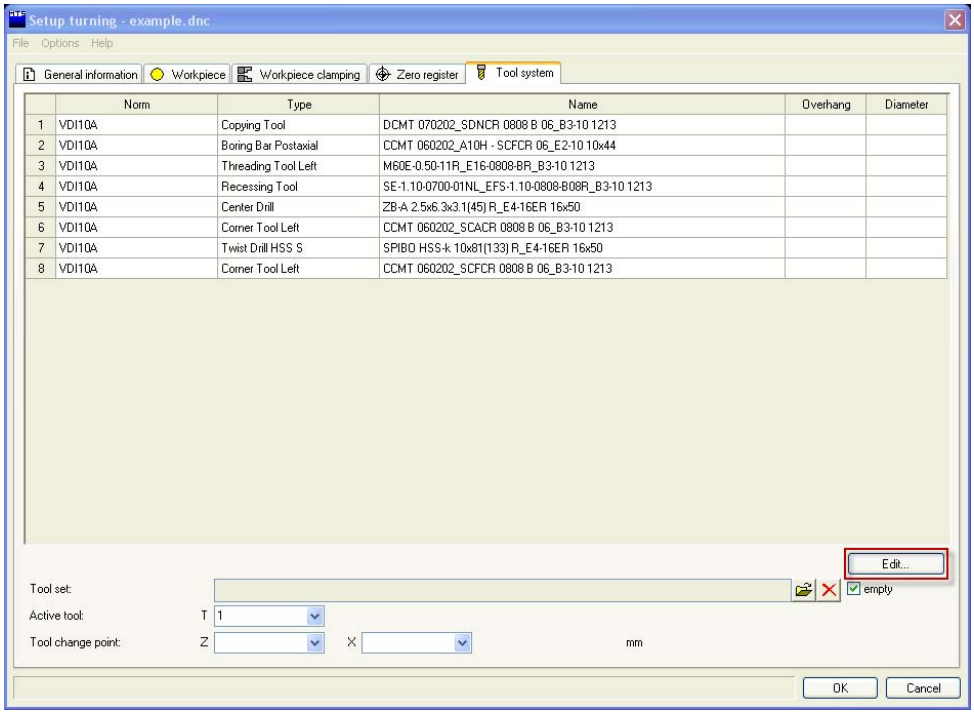


Fig. 2.14 Tool system tab

a. From the drop down menu "**File**" to open a new file as shown in Fig. 2.15.

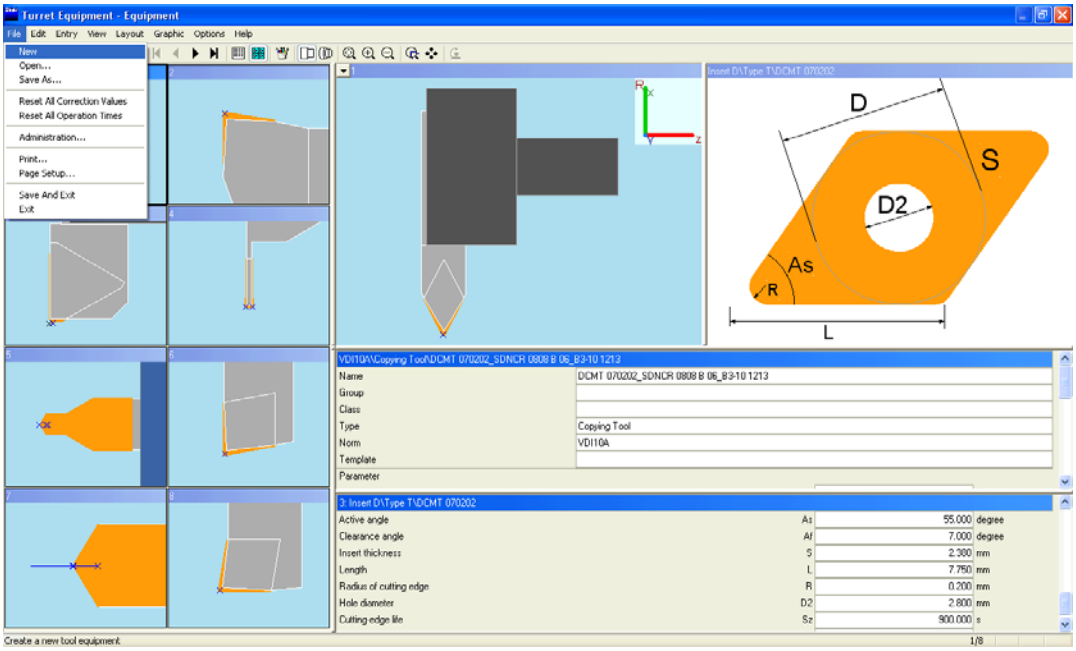


Fig. 2.15

b. A message appears asking the following questio "***Remove all tools from the tool equipment?***" , Select **"yes"** if you want to select a new tool set as shown in Fig. 2.16.

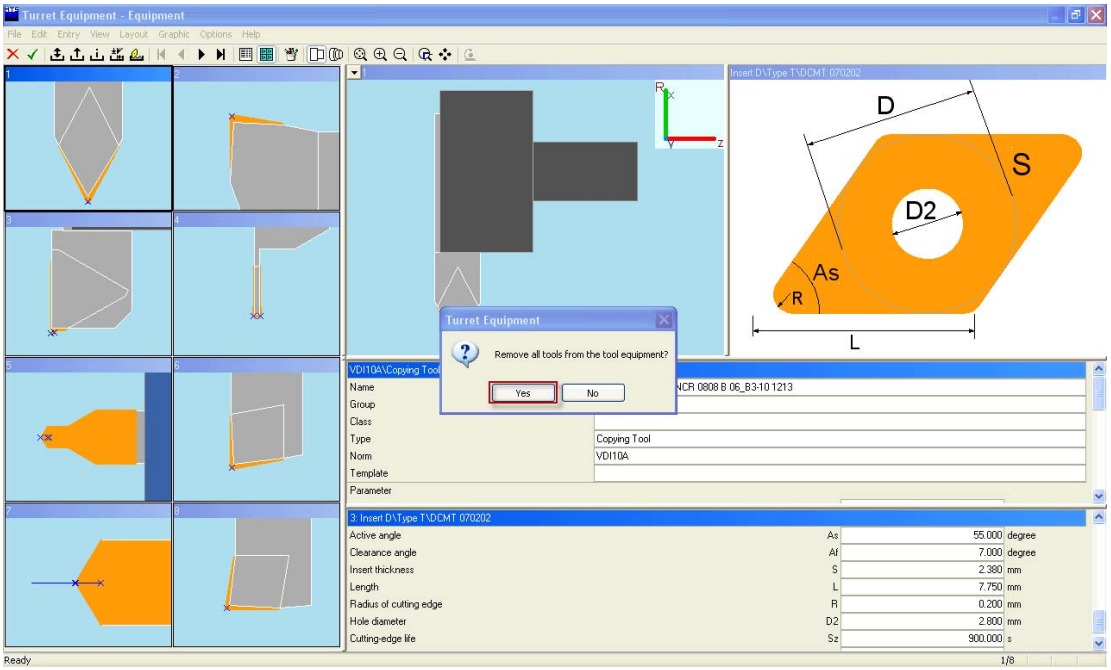


Fig. 2.16: Selecting new tool set

c. Double click on position 4 as the selected tool should be placed in position No. 4 as shown in Fig 2.17.

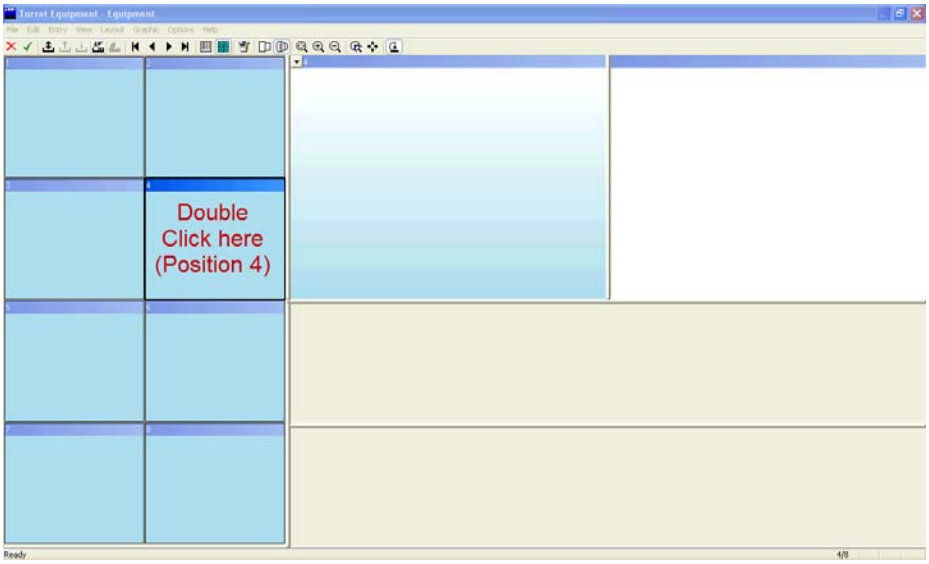


Fig. 2.17: Select tool position

d. Double click on"**Corner tool left"** to select the type of the tool mentioned in the setup sheet as shown in Fig. 2.18.

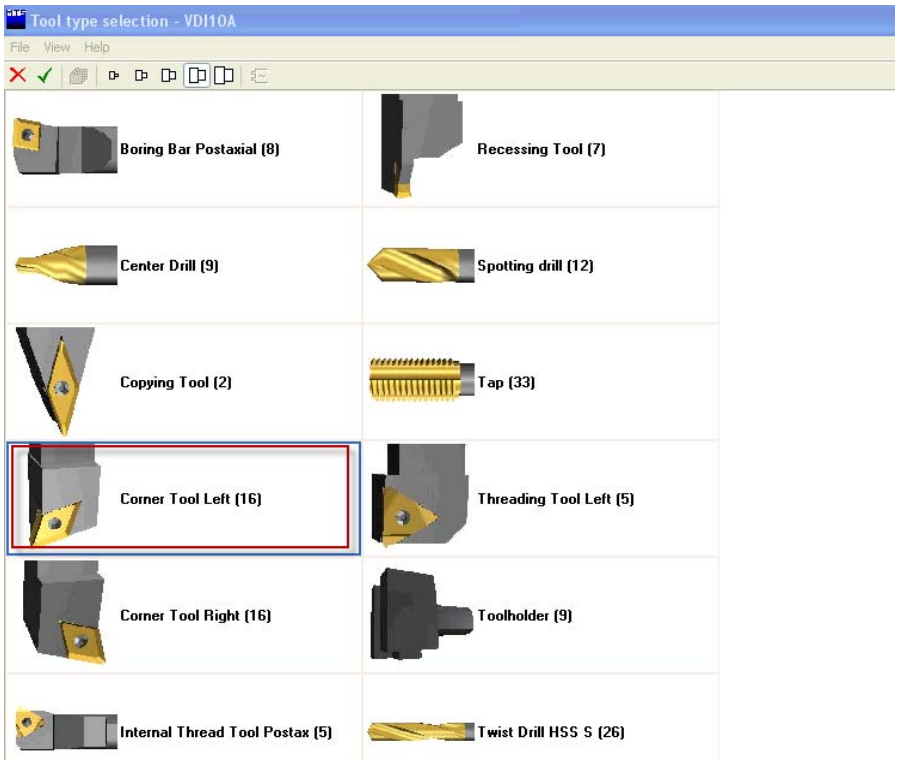


Fig. 2.18 Tool type selection

e. Click on the correct tool name **"CCMT 060202_SCACR 0808 B 06_B 3.10 1213",** then, save it by clicking  . The right column under the tool name shows the properties and dimensions of the tool as shown in Fig.2.19.

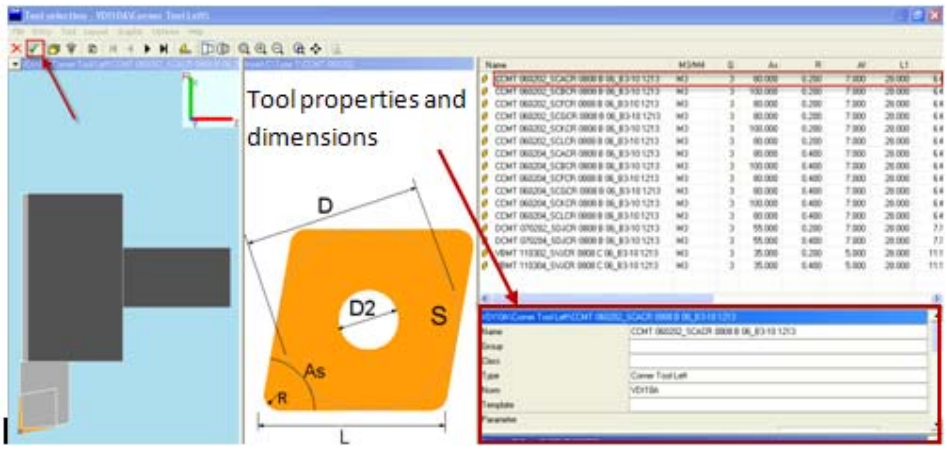


Fig. 2.19 Tool name selection

On the toolbar of the tool equipment Fig.2.19, there are some important icons that will help you in your **future settings**:

- Click on  to give a color to the tool, so when it operates, the machined area will have the same color as the tool.
- If you want to remove a tool, click on "**remove**" 
- Click on  for tool type selection.

f. Click **"Save and quit"**  . Fig. 2.20

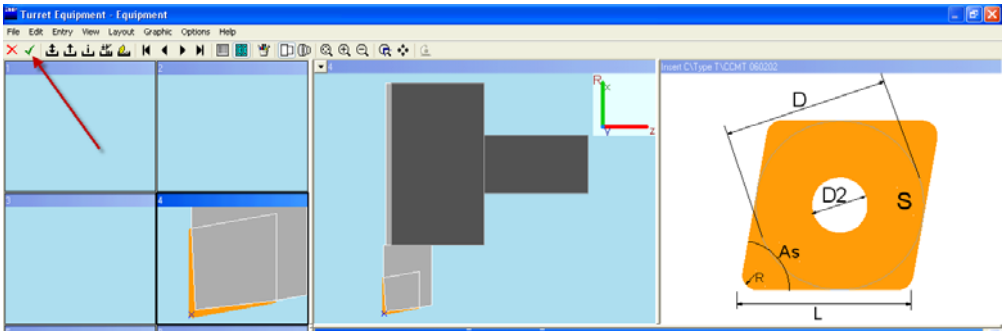


Fig. 2.20

9. Click **"OK"** to end the setup as shown in Fig. 2.21.

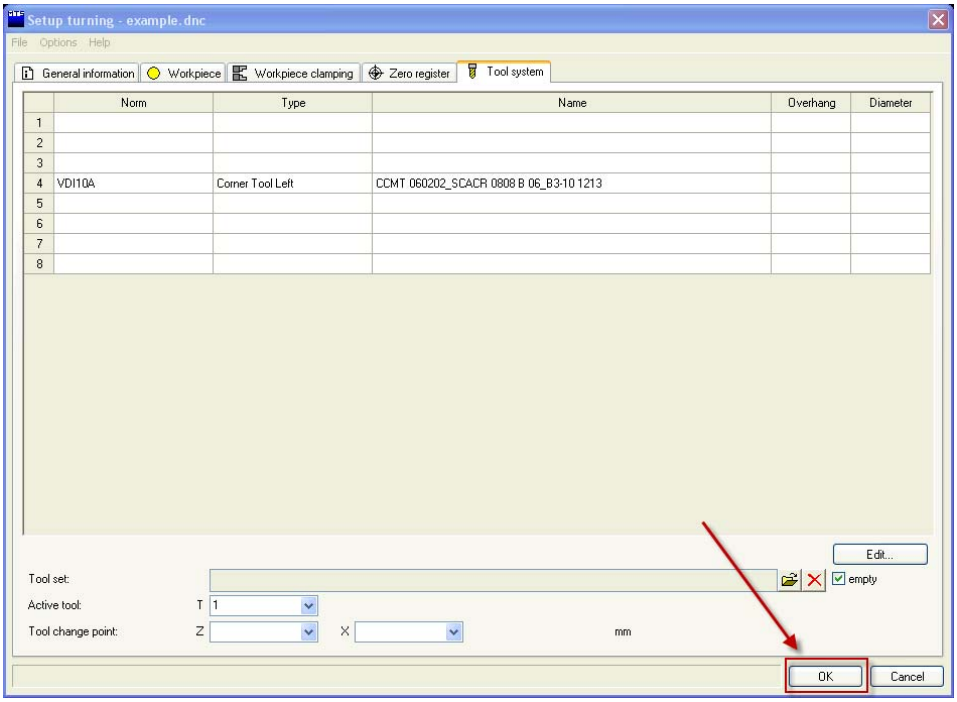


Fig. 2.21

3.2. Check and edit the setup sheet

- To check the prepared setup sheet, click on **"editor"** Or press **"F4"** or select **"NC editor from the main menu"**

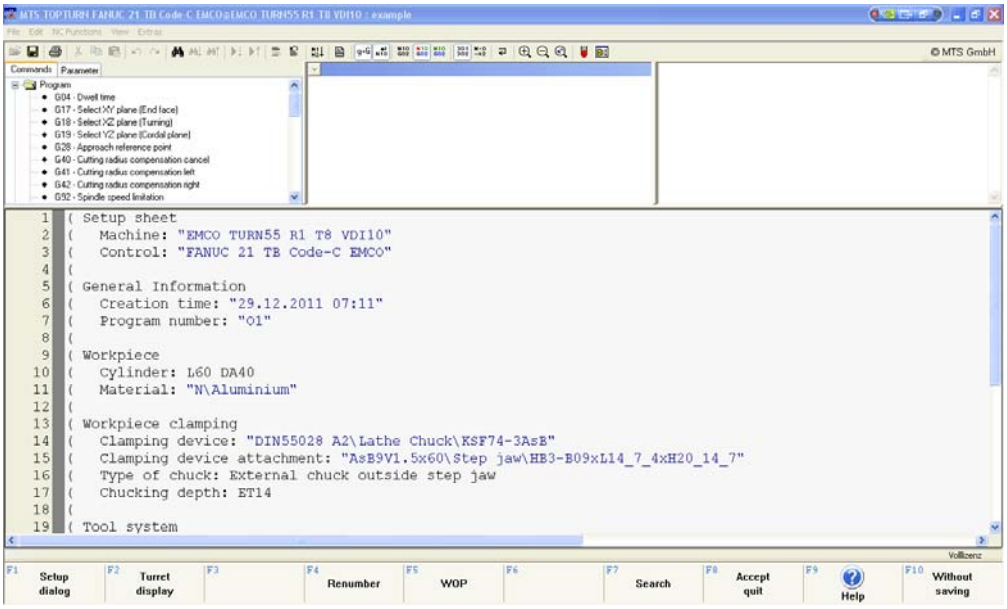


Fig. 2.22: Setup Sheet

- Click **"Accept Quit"** from the editor menu at the bottom.
- On the main screen try the following icons which allow you to edit the tool equipment:
  - Click on **"setup mode"**  to activate the tool equipment icon. Setup mode is used to make the machine setup (clamping devices, tools … etc).
  - Click on **"tool equipment"**  to see the tools mounted on the turret.
  - Click on **"quit"**  to exit from the tool equipment.
  - On the bottom menu press **"Esc"** or **"F10"** to return to the main menu.
  - Click **"Esc"** or **"F10"**
  - Click **"Yes"**
  - Open the MTS again as explained earlier, (Click Start Turning)
  - To open your setup again click on **"Open NC Program"**  (Open NC Program is to open new or existing NC program)
  - Select "setup1" then click **"open"**
  - Click on **"editor"**  twice to see your setup again.
  - Press **"Accept Quit"** from the editor menu at the bottom.
  - On the bottom menu press **"Esc"** or "F10" to return to the main menu.
  - Click **"Esc"** or "**F10**".

Practical Task 1 :

Use MTS software to do the following setup. Give your setup the name "Example 2"

Setup sheet" information (Parameters):

- Workpiece length: *67 mm*
- Workpiece diameter:*40 mm*
- Workpiece material: *Brass\CuZn 30*
- The chucking depth: *14 mm*
- Workpiece zero location: *"Right side – center of the workpart surface"*
- Tool position*: position 6*
- Tool to be used:

  Tool type: *Corner Tool Left*,
- Tool Name: *CCMT 060204_SCGCR 0808 B 06_B3-10 1213*

4. Basics of CNC programming

A programmer writes a set of instructions describing how he or she would like the machine to move and then feeds the program into the machine's computer which is called the controller.

The computer reads the instructions and sends electrical signals to a motor, which then turns a screw to move the machine. A sensor mounted on the machine sends positioning information back to the computer.

CNC machine tools produce only the movements that are described in a part program (a program that describes the machining operations and dimensions). Therefore, a programmer must write a part program in a language that the controller can understand. The most popular language for NC programming is ***G & M*** programming codes. One of the popular controller manufacturers using standard G & M codes is ***Fanuc***.

5. Structure of a NC Program:

The NC program consists of a series of commands with which the CNC machine tool is instructed to manufacture a certain work part.

The first line in the NC Program contains the program name. The NC-program names can contain alphanumerical (containing letters and numbers) or numerical characters.

Program core: Consists of a sequence of blocks (Lines). Each block contains the technical and geometric information that the controller requires for the correspondent machining step. It starts always with the block Number (N) e.g. N10, then the instructions or commands follow. Example of one block of a program: *N10 G0 X30 Z20* Where G0 is a G-code command and (X30 Z20) are the co-ordinates of the target point.

Program end: Consists of one block having one command.

The following table contains few basic G-codes, plus some additional and Miscellaneous functions (M-codes).

6. Basic G and M codes:

Command	Description
G54	(Workpiece co-ordinate setting) Inform the controller to use the workpiece zero point as an origin for the co-ordinate system instead of machine zero point.
G90	Absolute dimensioning system
G91	Incremental dimensioning system
G00 X…. Z…..	Rapid positioning. The tool will move to the target point at rapid speed. This will save time when the tool is moving in open space. Example: G00 X30 Z20. (X30 Z20 is target point)

G01 X…. Z…..F	Feed rate positioning. The cutting tool moves at a certain feed rate (F). **This is used to move the tool to cut the workpiece in straight line at certain feed rate.** Example: G01 X30 Z5 F0.8
G97 S ….	Set the spindle speed at certain value in RPM Example: G97 S2000 (spindle speed is 2000 RPM)
T …..	Select the tool in a certain position. T0404 (The active tool is tool number 4)
F	F: is the feed rate usually in mm/rev
M3	Spindle rotation clockwise
M4	Spindle rotation counterclockwise
M30	End the program

The following examples illustrate the use of the above codes.

Example 2:

Open the prepared setup sheet "**Example 2**". In the "**editor**" write the given *NC part program* using absolute dimensioning mode, to move the cutting tool in rapid positioning to the points shown below from (A) to (F) as shown in Fig. 2.23.

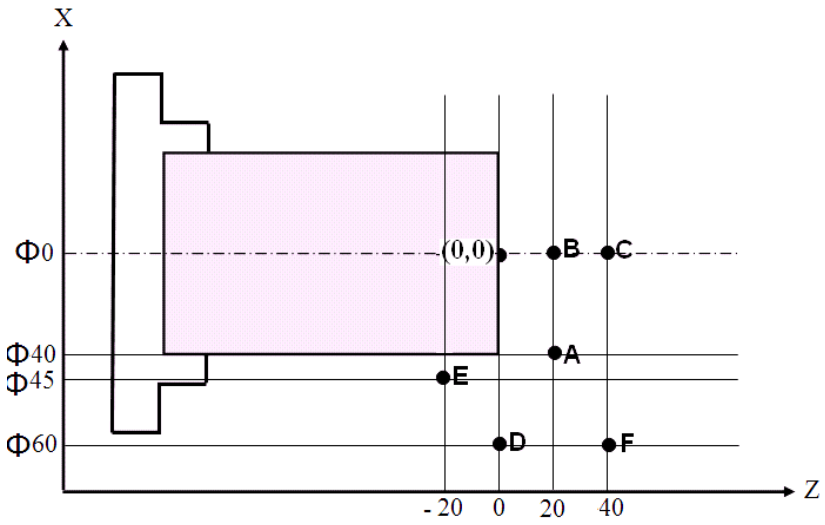


Fig. 2.23: Rapid positioning – G0

The part program:

Block No.	Program blocks		Description
N10	**G54**		*The controller will consider the workpiece point as zero point instead of machine zero point.*
N20	**G90**		*Absolute dimensioning system*
N30	**T0606**		*Select tool in position No.6.*
N40	**G0 X40 Z20**		*Move rapid (fast with maximum feed rate) to point (A) X40 Z20*
N50	**G0 X0 Z20**		*Move rapid to point (B) X0 Z20*
N60	**G0 X0 Z40**		*Move rapid to point (C) X0 Z40*
N70	**G0 X60 Z0**		*Move rapid to point (D) X60 Z0*
N80	**G0 X45 Z-20**		*Move rapid to point (E) X45 Z-20*
N90	**G0 X60 Z40**		*Move rapid to point (F) X60 Z40*
N100	**M30**		*End the program*

Solution:

Steps using MTS to write the above program:

1. Click: Open NC Program 
2. Select “example 2”
3. From the above menu Click **“Interactive mode”** 
   (Interactive mode is the simulating mode in which you can write, edit and simulate you program.)
4. Write the program as follows:

G54 ↵ (Enter ↵)

{Every time click (Enter ↵) again or (Y) to accept what you entered or if there is a mistake click (N) to be able to edit line}.

G90 ↵

T0606 ↵

G0 X40 Z20 ↵ (Select 2D-Graphic display mode .)

G0 X0 Z20 ↵

G0 X0 Z40 ↵

G0 X60 Z0 ↵

G0 X45 Z-20 ↵

G0 X60 Z40 ↵

M30 ↵

7. MTS modes of simulation

The following icons allow you to simulate the above program in different modes: Fig. 2.24

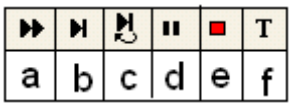


Fig. 2.24: Simulation modes and their controls

a. **Automatic mode:** to run an NC program automatically.
b. **Single block mode:** to run an NC program block by block where the operator need to confirm the execution of each block by click enter.
c. **Interactive mode:** to run an NC program block by block with the possibility to edit the executed block if not confirmed with the needed operation.

The above run mode could be controlled by the following:

d. **Pause the simulation**
e. **Stop:** To stop the simulation at any step while running the program.
f. **Test run:** The test run option is used to simulate the program very fast to reach a certain step in the program. However, the

simulation speed while using the test run option has nothing to do with the real machining of the workpiece.

Practical Task 2

Run the program "**Example 2**" in the different modes of simulation available in the MTS software.

- Run the program using "**automatic mode**" (click on "**pause**" then resume simulation while running).
- Run the program using "**single block mode**" (click on "**test run**" while running)
- Click on **"editor"** : On the editor you can write and edit your program and from the toolbar of the editor you can copy, paste, cut and do many tasks.
- From the menu bar click on **"NC Functions".** Fig. 2.25

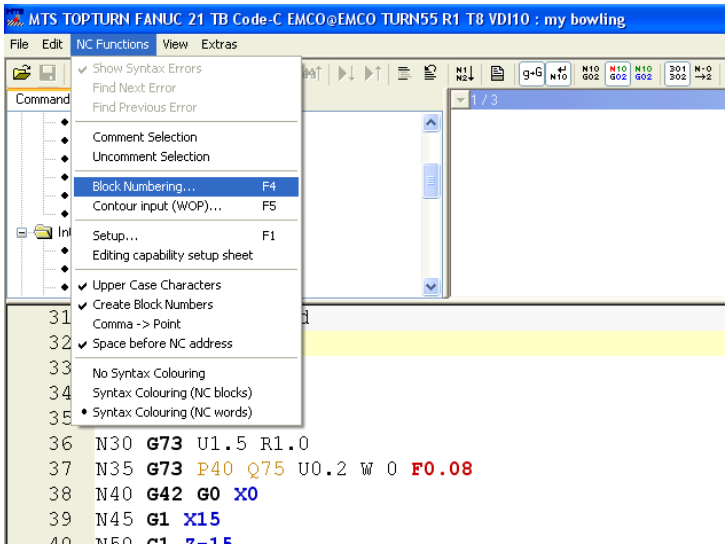


Fig. 2.25: NC functions

- Click on **"Block Numbering"** from the drop menu
- Change the start block number and the step size, then click **"Ok".** This will change your blocks' numbers. Fig. 2.26

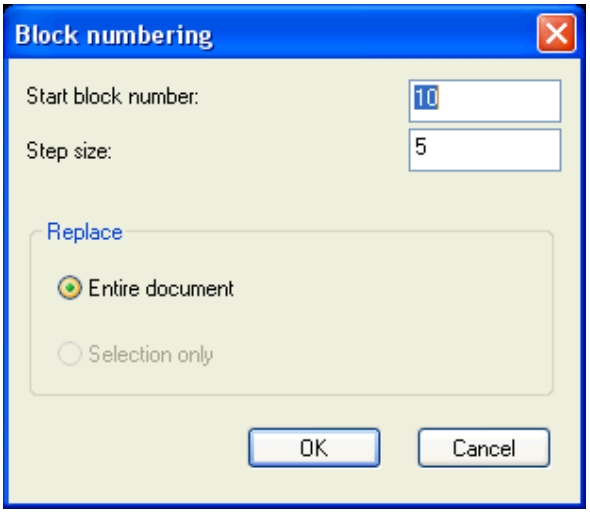


Fig. 2.26: Block Numbering

Practical Task 3 :

Prepare a setup sheet with the following information and write NC program using absolute dimensioning, to move the tool in rapid positioning to the points (From A to F), shown on the drawing Fig. 2.27:

"Setup sheet" information (parameters):

- Workpiece length: *64 mm*
- Workpiece diameter: *35 mm*
- Workpiece material: *Aluminium/Almg1.*
- Workpiece zero location: *"Right side – center of the workpart surface"*
- The chucking depth: *14 mm*
- Tool position*: position 8*
- Tool to be used:

  Tool type: *Corner Tool Left*,

  Tool Name: *CCMT 060202_SCACR 0808 B06_B 3.10 1213*

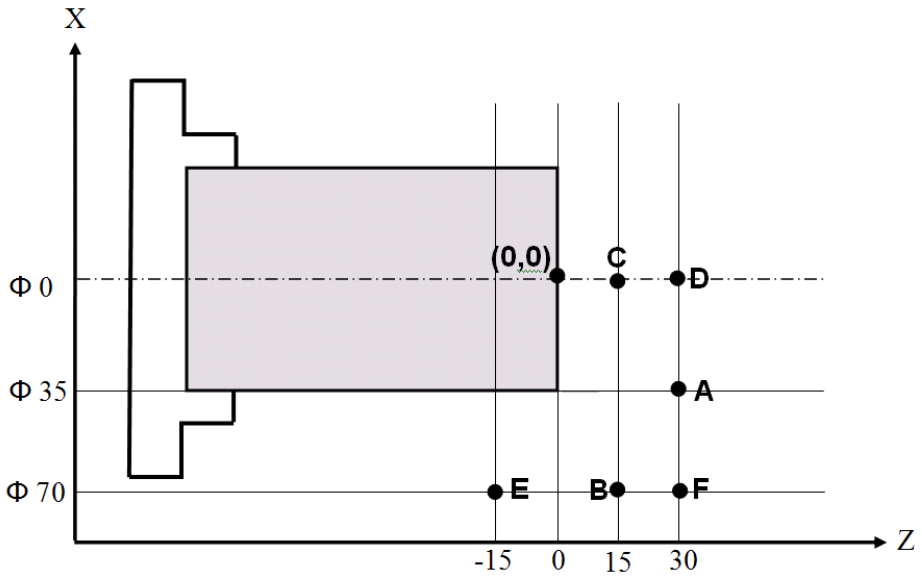


Fig. 2.27

Solution:

Block No.	**Part program blocks**
N5	
N10	
N15	
N20	
N25	
N30	
N35	
N40	
N45	
N50	

Example 3:

Prepare a setup sheet with the following information then write the NC program given below using absolute dimensioning, to move the tool in feed rate positioning to the points shown on the drawing Fig. 2.28:

"Setup sheet" information:

- Workpiece length: *70 mm*
- Workpiece diameter: *40 mm*
- Workpiece material: *Aluminium/Almg1*
- The chucking depth: *14 mm*
- Workpiece zero location: *"Right side – center of the workpart surface"*
- Tool position*: position 2*
- Tool to be used:

  Tool type: *Corner Tool Left*,

  Tool Name: *VBMT 110302_SVJCR 0808 C 06_B3-10 1213*

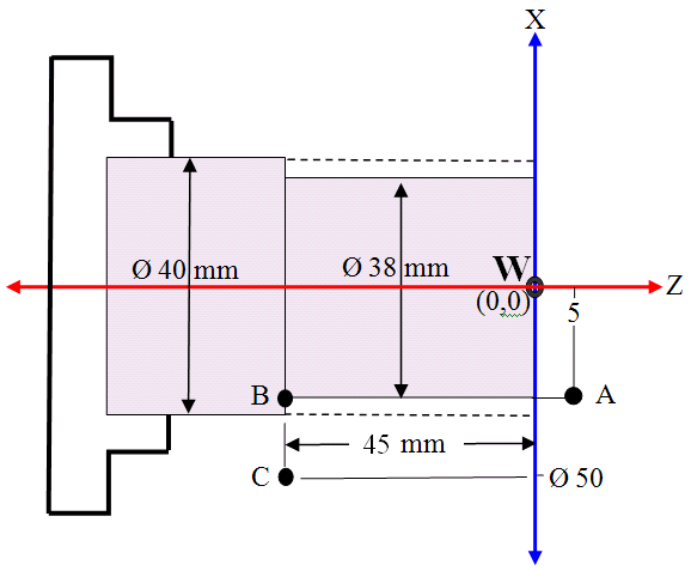


Fig. 2.28: Feed rate Positioning – G1

The given program:

Block No.	Part program blocks		Description
N10	**G54**		*The controller will consider the workpiece point as zero point instead of machine zero point.*
N20	**G90**		*Absolute dimensioning system*
N30	**G97 S1000 M3**		*Set the spindle speed to 1000 RPM, The direction of spindle rotation is clockwise (M3),*
N40	**T0202**		*Select the tool in position No.2*
N50	**G0 X38 Z5**		*Move the cutting tool rapid to point (A) X38 Z5*
N60	**G1 X38 Z-45 F0.8**		*Move the cutting tool to point (B) X38 Z-45 at feed rate=0.8 mm/rev. to cut the workpiece.*
N70	**G1 X50 Z-45**		*Move the cutting tool at feed rate=0.8 to the point (C) X38 Z-45 to remove any material behind the cutting the tip of the cutting tool.*
N80	**G0 X50 Z100**		*Move rapidly to X50 Z100 (Any point away from the workpiece)*
N90	**M30**		*End the program*

Solution:

Follow the same steps to prepare the setup sheet, to write the program and to simulate.

Explore MTS icons (3):

TOPTURN icons exploration. The program “Example 3” already open

- **Open NC program**
- Choose “Example 3”
- Click “**open**”
- To see the program in the editor mode, Click “**editor**” .twice.
- Click on “**save and exit**” at the top to exit from the program.
- Run the program using “**automatic mode**” (click on “**pause**” then resume simulation while running).
- Run the program using “**single block mode**” (click on “**test run**” while running)
- Run the program using “**interactive mode**” (switch between “**2D graphic**”, “**3D graphic**” or “**3D graphic (part)**” while running).
- You can click “**stop**” at any step while running the program.
- Use the zoom icons to view specific areas.

- On the toolbar, click on “**Dimensioning**”.

- Explore icons to check the dimensions of the work part. Fig.2.29

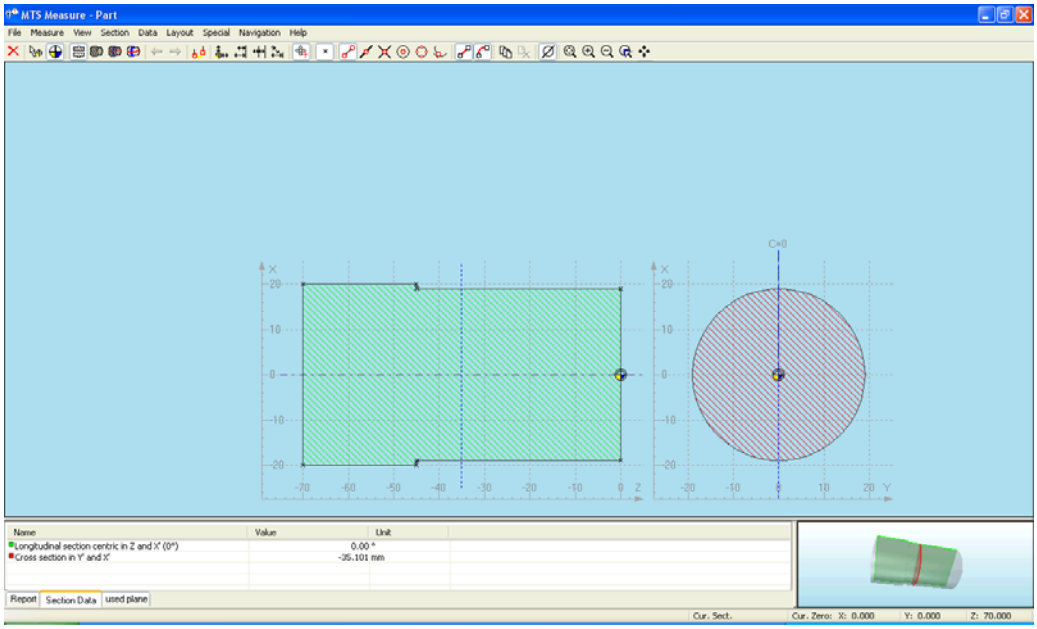


Fig. 2.29: Dimensioning

The dimensioning screen allows you to check the dimensions of your workpart. The default view gives you two section views from the workpiece. The following are some important tools:

Move current section: By picking this icon and drag the selection line. As shown below Fig. 2.30

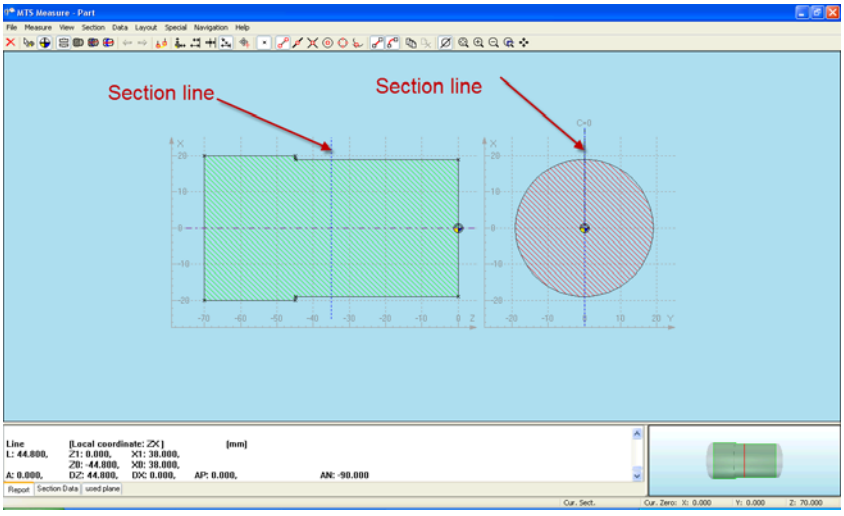
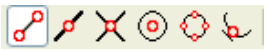
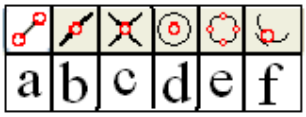


Fig. 2.30: The two section lines could be moved

 **measure points**: activate and deactivate the measure points view (ON/OFF)

 **Display points:** each icon display a certain points on the screen (this is compared to Object snap setting in Autocad).



a. Contour points

b. Middle line point

c. Intersection point

d. Circle center

e. Quadrant points

f. Middle arc

to deactivate any icon, just double click on it.

 **Point to point measurement**: Allow you to take measurements from one point to another.

Practical Task 4 :

Prepare a setup sheet with the following information then write NC program using absolute dimensioning, to move the tool in feed rate positioning as shown on the drawing. Fig. 2.31:

"Setup sheet" information (parameters):

- Workpiece length: *60 mm*
- Workpiece diameter: *32 mm*
- Workpiece material: *Aluminium/Almg1*
- The chucking depth: *14 mm*
- Workpiece zero location: *"Right side – center of the workpart surface"*
- Tool position*: position 8*
- Tool to be used:

  Tool type: *Corner Tool Left*,

  Tool Name: *VBMT 110302_SVJCR 0808 C 06_B3-10 1213*

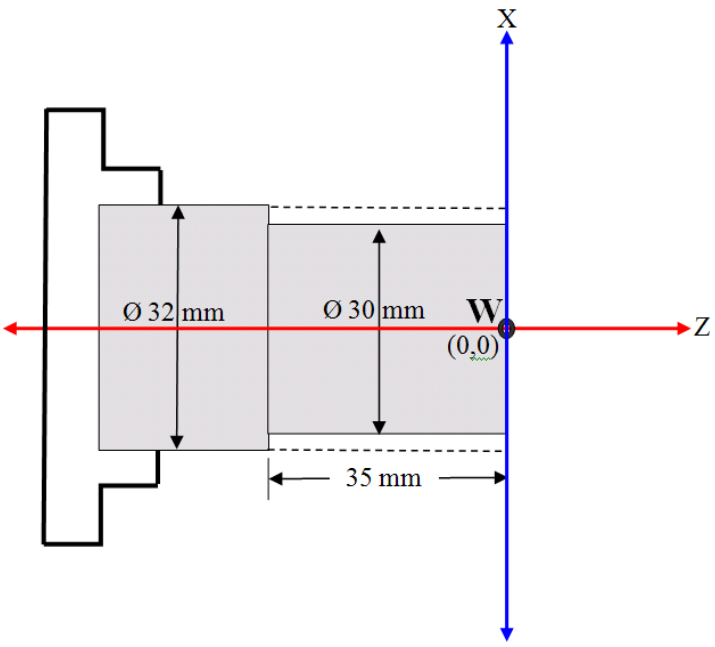


Fig. 2.31

Block No.	Part program blocks
N5	
N10	
N15	
N20	
N25	
N30	
N35	
N40	
N45	
N50	

Example 4:

Prepare a setup sheet with the following information then write the NC program given below using incremental mode to move the tool in rapid positioning from point A to B as shown on the drawing Fig. 2.32:

"Setup sheet" information (parameters):

- Workpiece length: *40 mm*
- Workpiece diameter: *30 mm*
- Workpiece material: *Brass\CuZn 30*
- The chucking depth: *14 mm*
- Workpiece zero location: *"Right side – center of the workpart surface"*
- Tool position*: position 6*

- Tool to be used:

  Tool type: *Corner Tool Left*,

  Tool Name: *CCMT 060202_SCACR 0808 B06_B 3.10 1213*

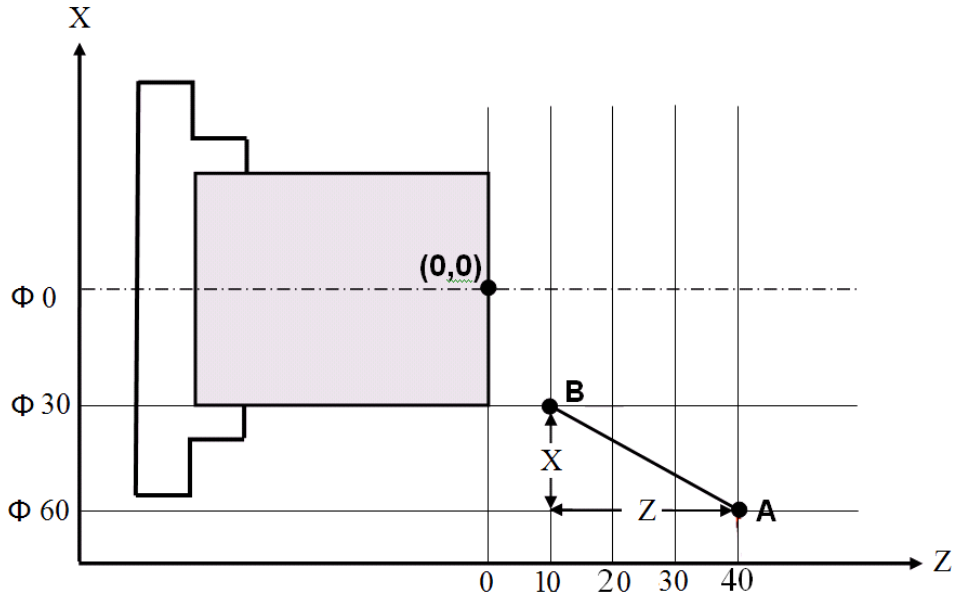


Fig. 2.32

Block No.	Part program blocks		Description
N10	**G54**		*The controller will consider the workpiece point as zero point instead of machine zero point.*
N20	**G90**		*Absolute dimensioning system.*
N30	**T0606**		*Select tool in position No.6.*
N40	**G0 X60 Z40**		*Move rapid (fast with maximum feed rate) to point X60 Z40*
N50	**G91**		*Incremental dimensioning system*
N60	**G0 X-30 Z-30**		*Move rapid to point X-30 Z-30 (measured incrementally) X = 60 – 30 = 30 (diameter values), the cutting tool moves toward the center of the workpiece (negative X) Z = 40 – 10 = 30 (the cutting tool moves in negative direction)*
N70	**M30**		*End the program*

Student's Notes

Reference:

1. **CNC-Simulator Turning *with Driven Tools and Counter Spindle* Programmer's Guide**

2. **EMCO WinNC GE Series Fanuc 21 TB Software description/ Software version**