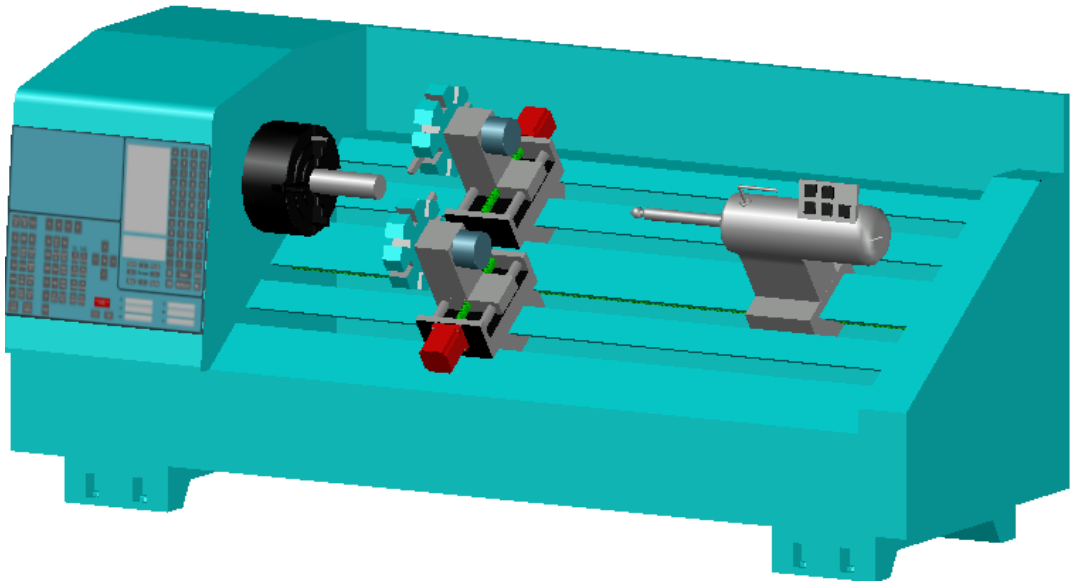
Virtual CNC Lathe

with Twin Turrets



Technical Manual

2020

ABASYN TECHNO LTD
City Plaza, Main Topi Bazar, Distt, Swabi, KP, Pakistan

Ph: Email:

Preface

In the modern high-tech manufacturing field, the computer numeric control (CNC) machines due to its excellent features of accuracy and repeatability completely replaced the manual machines.

Learning CNC has not been a very easy task in the modern industrial world. The training institutions are constrained to provide sufficient details/information to the learners about the processes involved in the running of machines in the industrial sector. That is why, there are very few institutions which can provide such type of trainings but they charge heavily. Moreover, learning CNC and its application has never been easy where chance of damage exists at various levels.

The intent of developing Virtual Factory is to introduce CNC with programmers / professionals in virtual environment in order to not only provide them detailed know-how in affordable charges but reduce the possible damages to a maximum length.

By the time readers go through this book, they will have a solid understanding of CNC machines before their deployment into the real manufacturing setup..

Every effort has been made to ensure that the material given in the manual is error free and work properly. However, any positive suggestions would be gratefully appreciated. Feedback can be sent to xyz@abasyntechno.com

Table of Contents

Level -6
CNC Lathe – Machining

Level – 7
Miscellaneous

Level - 1

Introduction & Equipment Overview

1.1 Overiveiw

The goal of developing virutal CNC factory is to bridge the space between the industrial professionals and the expensive CNC machines in an affordable budget with minimal physical damage. This shall provide them an opportunity to program and manufacture a part in such an environment where they can learn all machining operations as real. Lathe machines work by spinning the part and moving the tools.

1.2 Prerequisites and Target Audience

The programmers are expected to have the following:

- Engineering / Technical degree or diploma or equivalent experience
- Basic know how and use of conventrional tools and metrology
- Computer Skills
- **Target Audience**:
    - Manufacturers, Engineers, designers, scientists, inovators, technicians and students.

1.3 Required Tools and Equipment's

Audience will need:

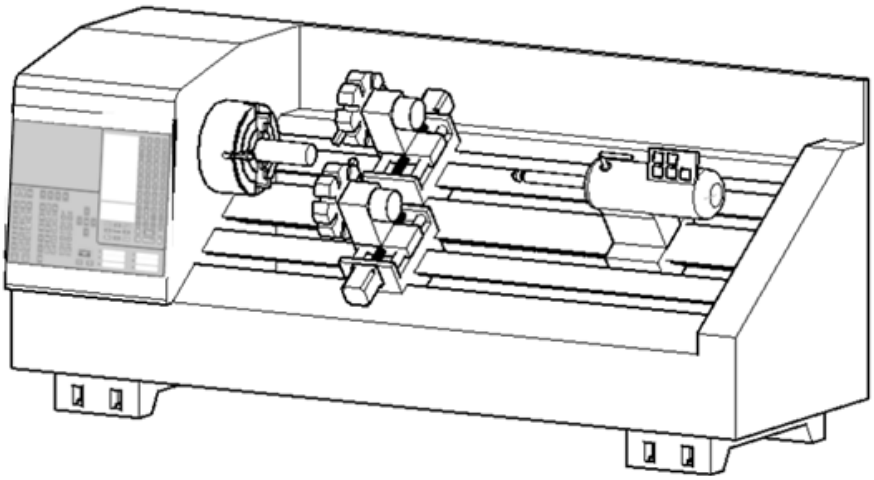
- Computer (Computer specs)
- Virtual Factory
- Basic computer know-how

1.4 Topics

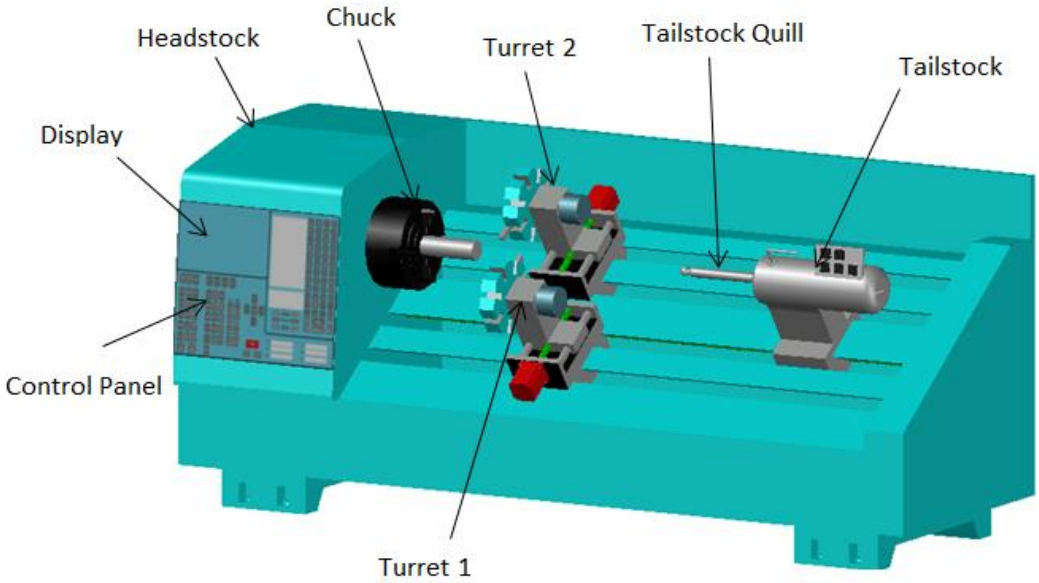
- Safety
- Coordinate Systems
- CNC Programming Language
- CNC Tools and its Operation
- CNC Turning

1.5 Machine Orientation

CNC system simulation and functionalities are widely welcomed these days. Demanding complex machining requires increasingly virtual machines. Virtual machining supports the user to reduce time and cost of manufacturing. Major components of the Virtual CNC Lathe with Twin Turret, is shown below in Figure 1.1.



(a)



(b)

Figure 1.1: Virtual CNC Lathe with Twin Turret

Level – 2

Safety

Though CNC machines are safer than conventional machines, this is still important topic to be there in this manual. Ignorance or incompetency can seriously injure you or damage to the equipment. The purpose of this level is to provide specific safety rules and develop an overall attitude of safety awareness. Hence this is totally different environment compared with the physical one which has its own rules of safety. Normally, you learn an attitude of safety and care from experience shown in Figure 2.1. Following strict rules in the shop promotes a progressive and professional environment.

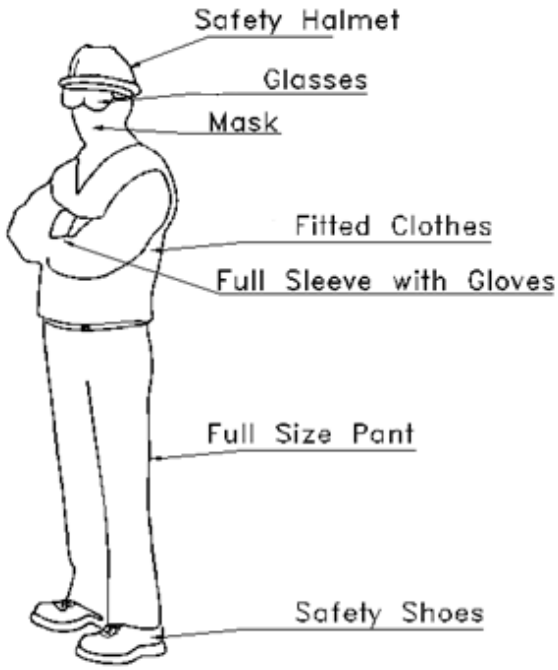


Figure 2.1: A Person with Typical Safety

The following are some additional instructions which can give you a healthy working environment when strictly applied.

1. No jokes are allowed in the shop nor should be engaged in unnecessarily talk.
2. Never interrupt someone while working at the machine. This can cause them making an irreparable damage.
3. Borrowing any tool without prior permission from the owner is strictly prohibited.
4. Respect your seniors and there you learn from.
5. Leave your workplace and surrounding area at least as clean as you were given.
6. Place the tools and equipment to its origin / location.
7. Never use Virtual Factory if you have not been trained to operate by a qualified person.
8. Follow process sequence for the maximum output
9. Read the operating manual thoroughly and follow instructions.
10. Do not eat /drink on the desk
11. Keep on saving your work periodically

Coordinate System

3.1 Overview

All motions in a CNC machine are based on a system namely Cartesian coordinate system. Any CNC machine cannot be operated without understanding its coordinate systems which are defined within. It describes you how the machine coordinate system is set on the machine so the CNC machine knows where the stock is located. Tools length and diameter compensation are two important parameters where know-how of coordinate system is required.

3.2 Coordinate System in Twin Turret CNC Lathe

Though Cartesian coordinate system consists of three number lines namely X, Y, Z, yet Y axis is absent in this machine. Moreover, it has Primary Axis (X, Z), Secondary Axes (U, W) and Tertiary Axes (P, R). These axes are set perpendicularly at 90 degree angles to each other as shown in Figure 3.1.

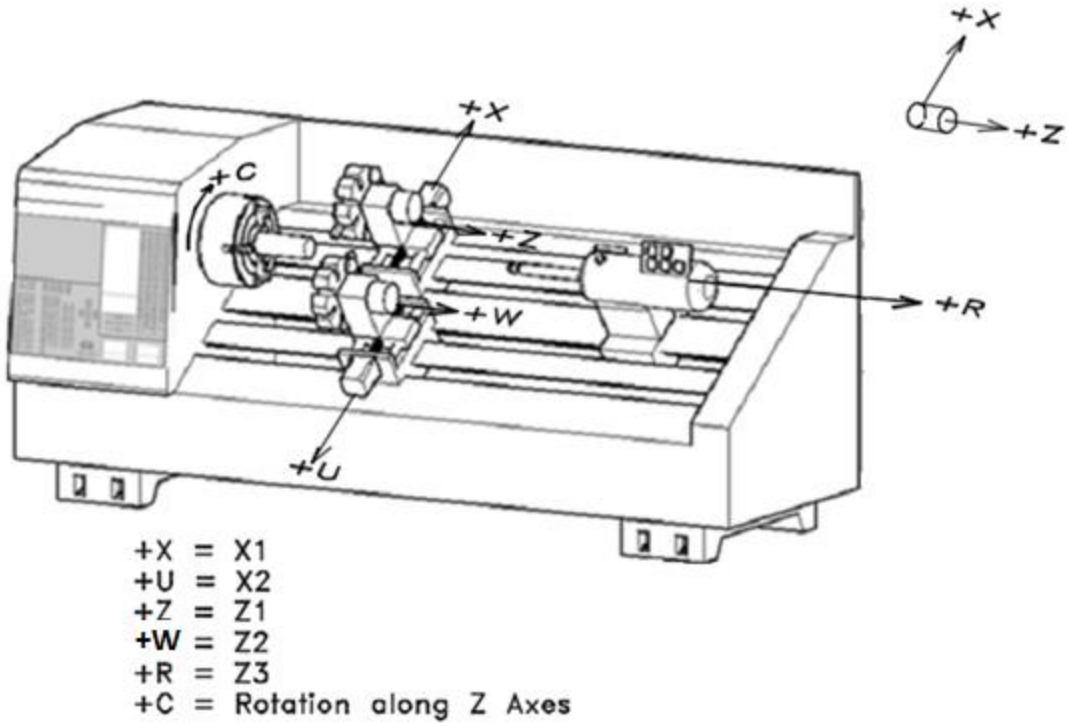


Figure 3.1 : Coordinate System in Twin Turret CNC Lathe Machine

3.3 Units

CNC programs are written in either Inch or metric units. A single code is required to switch the machine from one unit to the other. Old programmers had been using inch units because most of the tools were available in inches and they were more familiar with the inch measurement system. However, metric tools are easily available these days and new programmers are tending to use metric units.

3.4 Machine Home Position

When you turn on a CNC machine, it doesn't know where its axes are currently positioned in the workspace. After turning on the control power a button (Home) on the machine panel requires a press which sends all its axes to their extreme limits and a micro switch (limit switch) is activated. This signals to the control that the home for the respective axis is reached. Once all the axes are stopped, the machine is labled as "homed". Machine coordinates are manipulated in relation to this home position.

3.5 Tool Length Offset and Radius Compensation

Every tool has a different length. When a tool is replaced, the length of its replacement will likely change because it is almost not possible to set a new tool in the holder in exactly the same place as the old one. The CNC machine needs to know how far each tool extends from the turret / spindle (in case of milling) to the tip. These values are noted and called Tool Length Offset (TLO). The TLO is found by jogging the turret with tool from the machine *home* Z- position to the *part* Z- zero position, as shown in Figure 3.2.

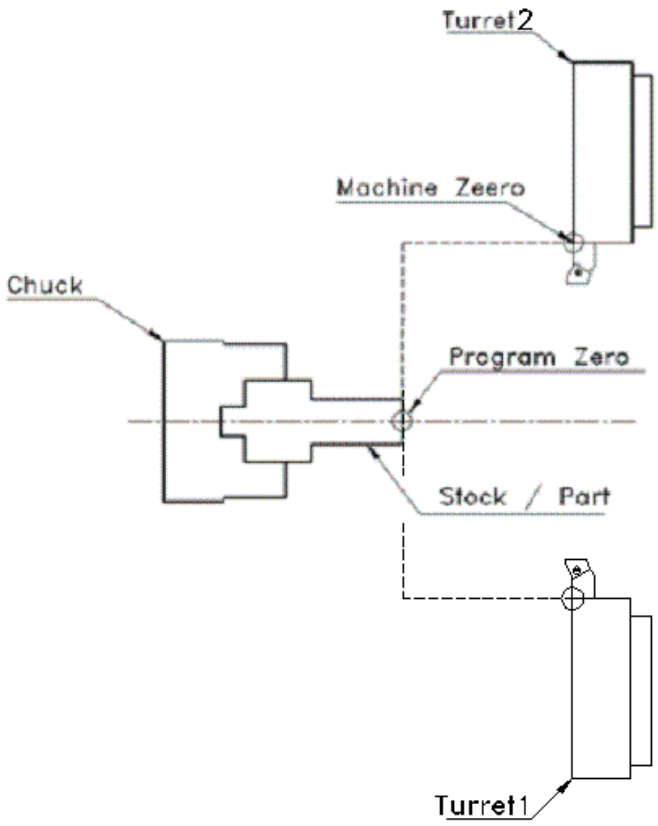


Figure 3.2 : Machine and program zero from the spindle along both axis.

Sometime a programmer ignores **cutter** size when calculating coordinates while writing a program and **Radius compensation setup** allows the programmer to choose from a range of **cutter** sizes . When the program is run, the machine will keep the cutter away from the programmed path (Figure 3.3) which is based upon the values in the offset table. This is the most important parameter table, which a programmer should consider while starting a program or machine.

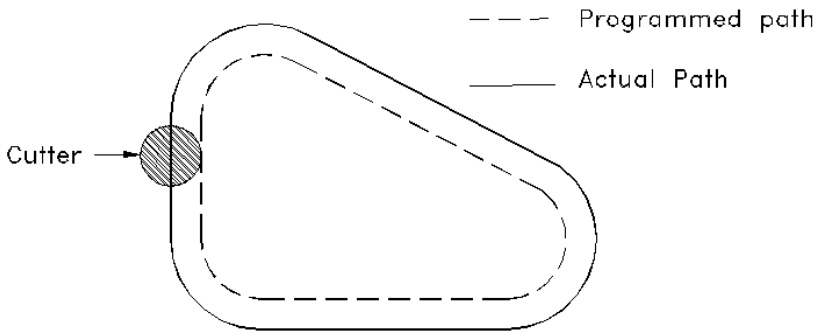


Figure 3.3 : Radius Compensation for a Programmed Tool Path

Level - 4

CNC Programming Language

4.1 Overview

CNC machines were developed by MIT in 1950. Most CNC machines used a language set by Electronics Industry Association (EIA) in 1960. Initially, this language was named as “RS-274D” but it got its fame as “G-code” etc.

Most programmer use about one third of codes available in the controller vocabulary; the reason is they are common among every CNC machine. These codes, along with other parameters and coordinates, comprise to CNC program.

4.2 CNC Program Editor

CNC program is a simple ASCII character text files that can be viewed or edited in any text editor like Notepad etc.

4.3 CNC Language Format and a Sample Program

CNC instruction set is written in a line with an order from left to right. Each line of instruction set is called “block”. They are written in a particular sequence that covers safety and feature operation. So it is important to follow a specific program format. These blocks are arranged in the following sequence:

1) Start Program
2) Select Units
3) Load Tool
4) Spindle On
5) Rapid Traverse to Position
6) Coolant On
7) Machining operation
8) Coolant Off
9) Spindle Off
10) Move to safe position
11) End program

The above program has simply one tool for one operation. Program with multiple tools repeat, steps from three to nine for each operation. Table 4.1 shows a sample program with codes explained. This sample program is designed to teach you basic concept of programming.

Sample Program for Turret1

Block	Description	Operation
%	Start of Program	Program Start
N10 G90 G70	Block Number, Absolute mode, Set units to inch	Coordinate System Selection Unit selection
N20 T1 M06	Select Tool No 1	Tool Selection
N30 S500 M03	Speed 500, Clockwise (CW) Spindle on	Spindle On
N40 G00 X20 Z5	Rapid Move Near Part	Move Turret to Position
N50 M08		Coolant On
N60 G01 G95 Z0 F1	Move to Job Zero Position	Linear interpolation with feed
N70 G01 Z-4.5 F0.2	Cutting Surface	Linear Interpolation with a Depth of cut
N80 G01 X50 Z-30	Movement in X Direction with Cut	Linear Interpolation with new depth of Cut
N90 G01 X20 Z5		Turret back to safe position
N100 M09		Coolant Off
N110 G00 X0 Z0	Rapid Traverse to Home the Turret	Home Position
N120 M05		Spindle off
N130 M30		Reset Program to beginning
%	End of Program	Program End

4.4 List of Codes Used in CNC Programming

Almost all letters are used in CNC machines. Table 4.1 shows some codes which are used frequently, but their functions are totally different based on which G-code is used in combination with.

Code	Description
A	Rotary motion along X–Axes
B	Rotary motion along Y-Axes
C	Rotary motion along Z-Axes
D	Cutter diameter compensation (CDC)
F	Feed rate
G	G-codes
H	Tool Length Offset (TLO)
I	Arc Center in X-vector, also used in drill cycles
J	Arc Center in Y-vector, also used in drill cycles
K	Arc Center in Z-vector, also used in drill cycles
M	M-Codes
N	Block Index
O	Program number
P	Dwell time
Q	Used in drill cycles
R	Arc Radius, also used in drill cycles
S	Spindle Speed (RPM)
T	Tool number
X	X-coordinate
Y	Y-Coordinate
Z	Z-coordinate

Table 4.1: List of all Codes Used in CNC Programming

4.5 G and M Codes

4.5.1 G-Codes

Codes that start with letter G are called preparatory codes (some experts call them Geometric codes) because they prepare a machine for a certain type of motion. The most common G-codes with their meanings are shown in Table 4.2

Code	Function	Description	Standard
G00	Rapid positioning	A mode of control in which movement to the programmed point occurs With maximum, e.g. rapid, feed rate, a feed rate previously programmed is Ignored but not cancelled, and the movements in different axes may be uncoordinated	Iso6983
G01	Linear Interpolation	A mode of control. used for a uniform slope or straight line motion, that uses the information contained in a block to produce velocities proportional to the distances to be moved in two or more axes simultaneously	Iso6983
G02	Circular interpolation arc clockwise Remark to circular interpolation	Circular interpolation in which the curvature of the path of the tool with respect to the work piece is clockwise when the plane of motion is viewed in the negative direction of the axis perpendicular to lt. A mode of contouring control that uses the information contained in one block to produce an arc or a circle, the velocities of the axes used to generate the arc being varied by the control	Iso6983
G03	Circular interpolation arc counter-clockwise	Circular Interpolation in which the curvature of the path of the tool With respect to the work piece counter-clockwise when the plane of motion is viewed in the negative direction of the axis perpendicular to it.	Iso6983
G04	Dwell	A timed delay of programmed or established duration. not cyclic or sequential; i.e. not an interlock or hold.	Iso6983
G18	ZX plane selection	Used to identify the plane for such functions as circular interpolation. cutter compensation; and others as required.	Iso6983
G20	Turning Operation	It Takes X value which is the Diameter to be cut and Z value that is End point in Z-Axis	assigned
G28	Return to home position (machine zero, aka machine reference point)	Takes X Y Z addresses which define the intermediate point that the tool tip will pass through on its way home to machine zero. They are in terms of part zero (aka program zero), NOT machine zero	Iso6983 & Fanuc
G33	Thread cutting, constant lead	Mode selection for machines equipped for thread cutting	Iso6983
G34	Thread cutting, Increasing lead	Constantly increasing lead.	Iso6983
G40	Cutter compensation cancel	Command which cancels any cutter compensation (diameter or radius') or tool offset.	Iso6983
G41	Cutter compensation left	Direction of cutter compensation of the tool path looking from the cutter in the direction of the relative cutter motion.	Iso6983
G42	Cutter compensation right	Direction of cutter compensation of the tool path looking from the cutter in the direction of the relative cutter motion.	Iso6983
G43	Tool offset positive	Indicates that the value of the tool offset shall be added to the coordinate dimension of the relevant	Iso6983

		block, or blocks.	
G44	Tool offset negative	Indicates that the value of the tool offset shall be added to the coordinate dimension of the relevant block, or blocks.	Iso6983
G63	Tapping	The selection for the particular case shall be defined in the format specification	Iso6983
	Continuous-path mode	Used to move the axis with programmed feed rate across two or more blocks (no exact at each end of block)	Iso6983
G68	Rotate coordinate system	Rotates coordinate system in the current plane given with G17, G18, or G19. Center of rotation is given with two parameters, which vary with each vendor's implementation. Rotate with angle given with argument R. This can be used, for instance, to align the coordinate system with a misaligned part. It can also be used to repeat movement sequences around a center. Not all vendors support coordinate system rotation.	Used for Milling in Fanuc
G69	Turn off coordinate system rotation	Cancel G68	
G70	Dimension input inch	Mode selection for dimension input.	Iso6983
G71	Dimension input metric	Mode selection for dimension input.	Iso6983
G75	Peck grooving cycle for turning		Fanuc
G76	Threading cycle for turning, multiple repetitive cycle		Fanuc
G80	Fixed cycle cancel	Fixed cycles Will discontinue.	Iso6983
G81 to G89	Fixed cycle	A preset series of operations which direct machine axis movements and/or cause spindle operation to complete such actions as boring, drilling, tapping or combinations thereof	Iso6983
G90	Absolute dimension	Mode of control for interpretation of dimensions as relative to a specified origin	Iso6983
G91	Incremental dimension	Positioning defined with reference to previous position	Iso6983
G92	Preload registers	Used to modify or set registers by the programmed data words. NO motion occurs	Iso6983
G93	Inverse time feed rate	Feed input is reciprocal to the time to execute the block	Iso6983
G94	Feed per minute	Feed rate units are millimeters/inches per minute	Iso6983
G95	Feed per revolution	Feed rate units are millimeters/inches per revolution.	Iso6983
G96	Constant surface	The spindle speed codes specify the constant surface	Iso6983

	speed	speed in meters/feet per minute. The spindle speed is automatically controlled to maintain the programmed value	
G97	Revolution Per Minute	The spindle speed code specifies the spindle speed in revolutions per minute	Iso6983

Table 4.2: List of G-Codes Used in CNC Programming

4.5.2 M-Codes

Codes that start with letter M are called miscellaneous codes (some experts call them Mechanical Codes) shown in Figure 4.3. They control auxiliary options of machine like coolant on/off and spindle direction. Mostly one M-code appear in a block.

Code	Function	Description	Standard
M00	Program stop	A miscellaneous function command to cancel the spindle or other functions (e.g. coolant function) and terminate further processing after the completion of commands in the block	Iso6983
M01	Optional (planned) stop	A miscellaneous function command similar to a program stop except that the control ignores the' command unless the operator has previously pushed a button to validate the command	Iso6983
M02	End of program	A miscellaneous function indicating completion of work piece. Cancels spindle or other function (e.g. coolant function) after completion of all commands in the block. Used to reset control and/or machine	Iso6983
M03	Spindle on (clockwise rotation)	A miscellaneous function command to start the spindle rotation in the clockwise (CW) direction. The spindle (rotation) speed is specified with the S-Word	Iso6983
M04	Spindle on (counterclockwise rotation)	A miscellaneous function command to start the spindle rotation in the counter-clockwise (CCW) direction. The spindle (rotation) speed is specified with the S-Word.	Iso6983
M05	Spindle OFF	A miscellaneous function command to cancel the spindle rotation.	Iso6983
M06	Tool change	Command to execute the change of toot(s) manually or automatically, not to Include tool selection May or may not automatically shut-off coolant and spindle.	Iso6983
M07	Coolant on (mist)		Fanuc
M08	Coolant on (flood)		Fanuc
M09	Coolant off		Fanuc
M21	Tailstock forward		Fanuc
M22	Tailstock Backward		Fanuc
M23	Thread gradual pullout ON		Fanuc

M24	Thread gradual pullout OFF		Fanuc
M30	End of Data	A miscellaneous function which cancels spindle or other function (e g. coolant function) after completion of all commands in the block Used to reset control and/or machine. Resetting control Will include return to the program start character.	Iso6983
M60	Work piece change	A miscellaneous function indication that the work piece needs to be removed or reoriented. Cancels spindle and coolant functions after completion of all commands in the block	Iso6983
M94	Mirror image cancel		Fanuc list 2
M95	Mirror image of X axis		Fanuc list 2
M98	Subprogram call		Fanuc
M99	Subprogram End		Fanuc

Table 4.3: List of M-Codes Used in CNC programming

4.5 Canned Cycles

A canned cycle is a way to perform repetitive CNC machine operations. In other words this allows one compact block of code to command many moves. Canned cycles automate certain machining functions such as boring, drilling, threading etc.

Level - 5

CNC Lathe Operation

5.1 Overview

Though you will be using virtual CNC lathe machine, learning the following setup process is important for actual machine operation.

1- Before Start
Ensure oil and coolant levels are full in the machine. Check if machine requires any maintenance service. Clean the working area and remove if any loose tool in the working space. Ensure the air supply which is required to the machine.

2- Start/Home
Switch on the main breaker, usually at the back side of machine followed by switching to power on button located in on the control panel.

3- Loading Tools
Load tools into the carousel listed in the CNC program or mentioned in the process plan if any.

4- Tool Length Offsets
For each tool find the TLO and update the TLO table.

5- Fixture Offset (have or have not)
When fixture is properly installed and aligned on the machine, set the fixture offset to find the part XZ datum.

6- Loading CNC Program
Send / Copy CNC program from your computer to the machine control using RS-232 communications cable, USB flash memory, or FDD / HD disk.

7- Run Program
Dry run or run the program, using extra caution until the program is proven to be error-free. Single block execution is a safer way to verify your codes.

8- Shut Down
Remove job from the spindle, clean the work area, and properly shut down the machine. Be sure to clean the work area and return tools to its proper origin.

5.2 Virtual CNC Lathe with Twin Turret - GUI

Overview of the GUI control panel, shown in Figure 5.1, where you know the location of buttons and their functions

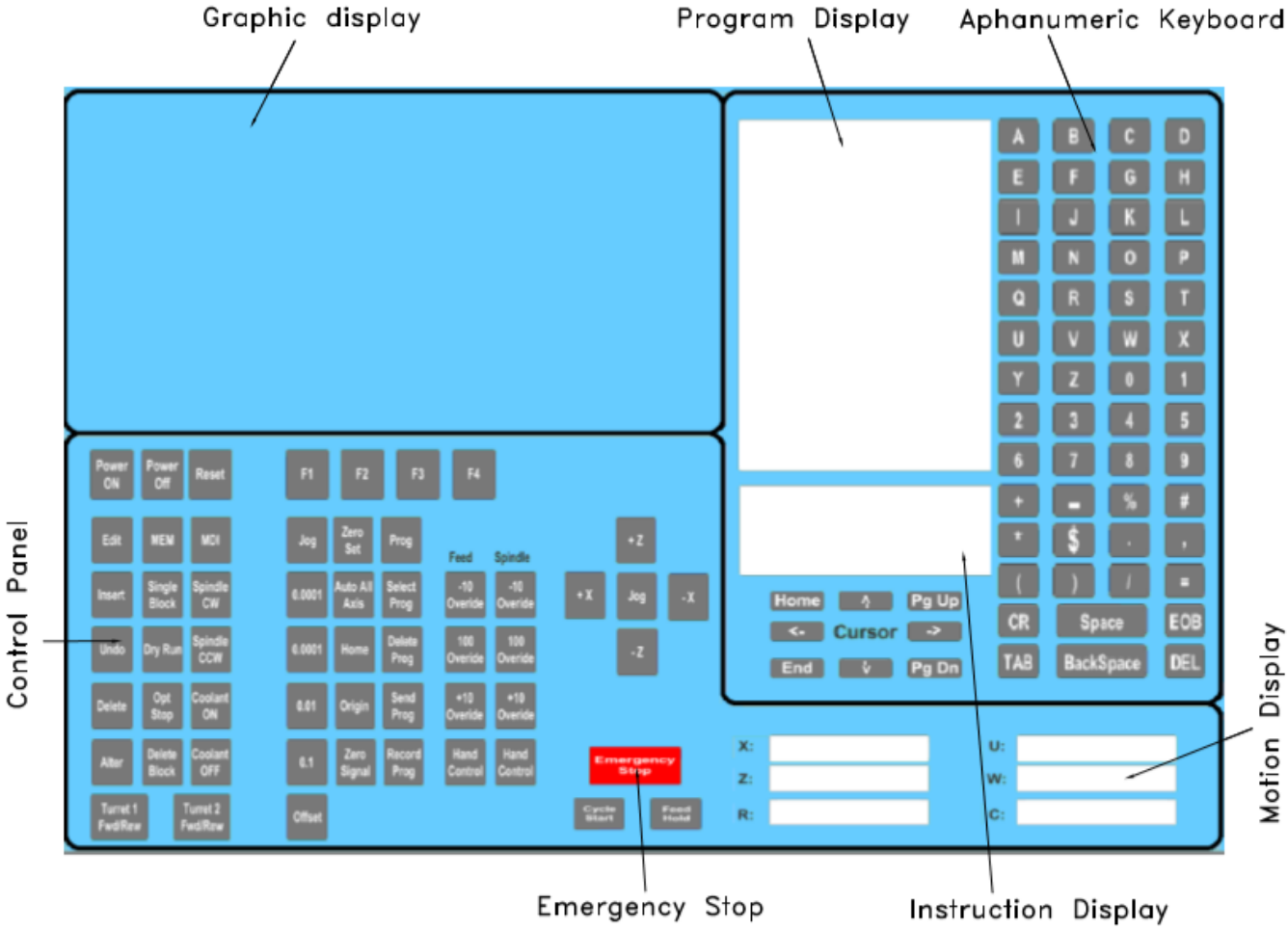


Figure 5.1 : The Virtual CNC Lathe with Twin Turret GUI Control Panel

5.3 Start and Home Machine

In the virtual environment you may not be required the following procedure however the actual machine would need a similar procedure as given in Figure 5.2

No	Machine Starting Order	Button Name and Location
1	**Machine working Space** : Well Cleaned	
2	**Air Supply:** On (to check if we need air or not)	
3	**Main Supply Breaker:** on	
4	**AC supply:** Press POWER ON Button (Make sure emergency Button is on)	Power ON

5	Reset Button : Press	
6	**Home Machine** : Press Bring machine axes to home	

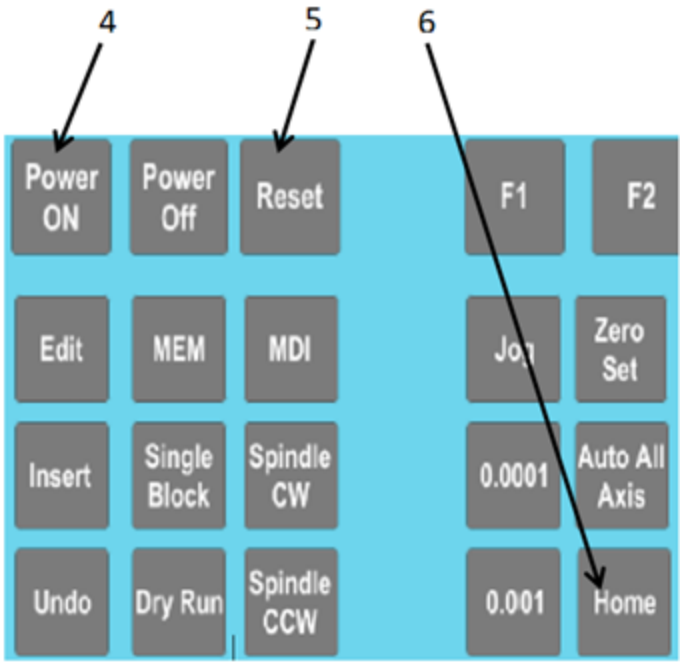


Figure : 5.2 Start /Home Machine

5.4 Loading Tools in the Turrets

No	Procedure	Button Name and Location
1	**MDI**: Press	
2	**Select Turret 1 Fwd / Rev:** Press (Turret selection between 1 and 2)	
3	**Tool Number Selection:** Press	

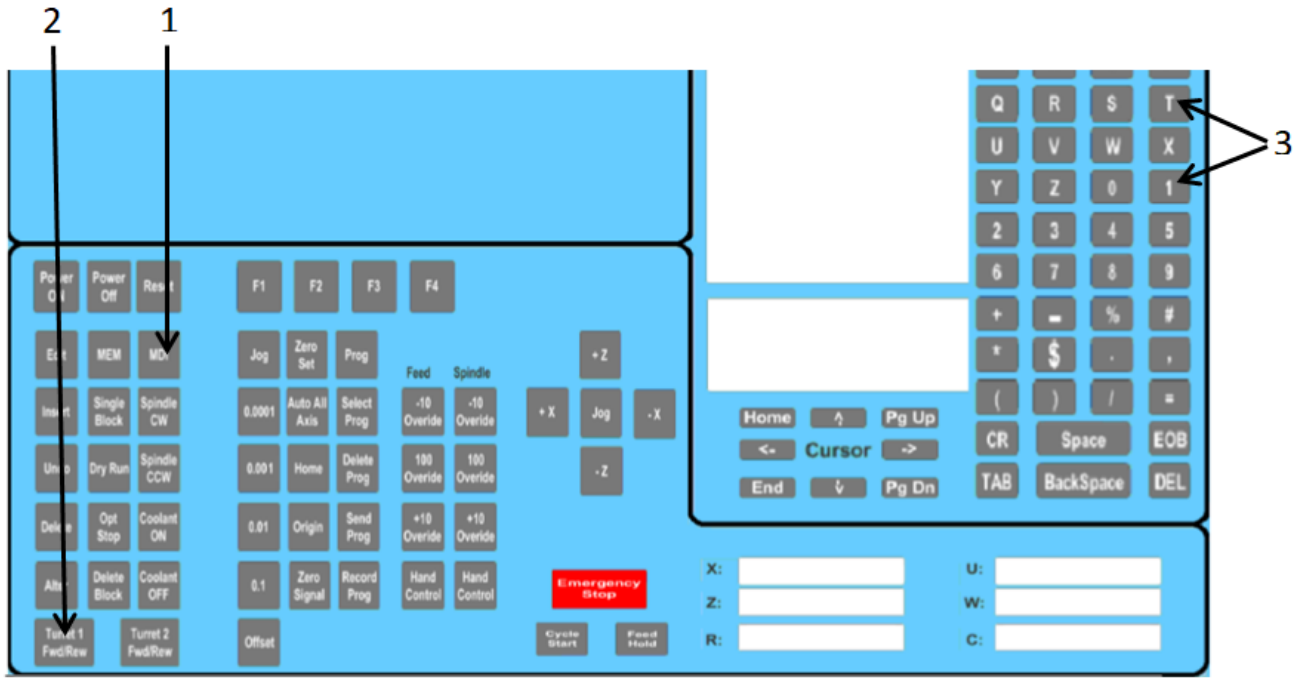
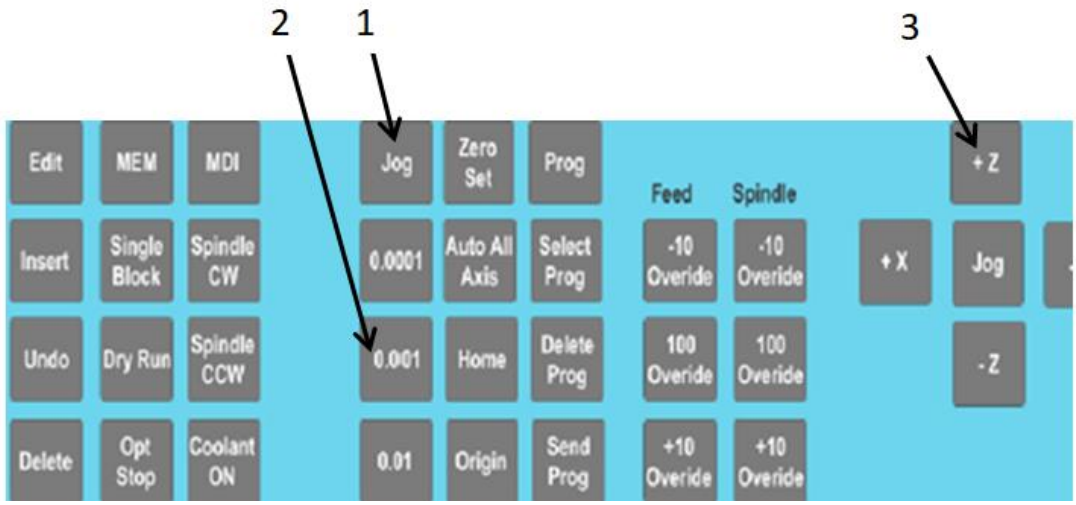


Figure : 5.3 Loading Tools in the Turrets

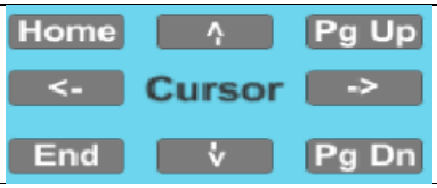
The same procedure can be followed for the Turret 2 provided if Turret 2 button is selected in Figure 5.3.

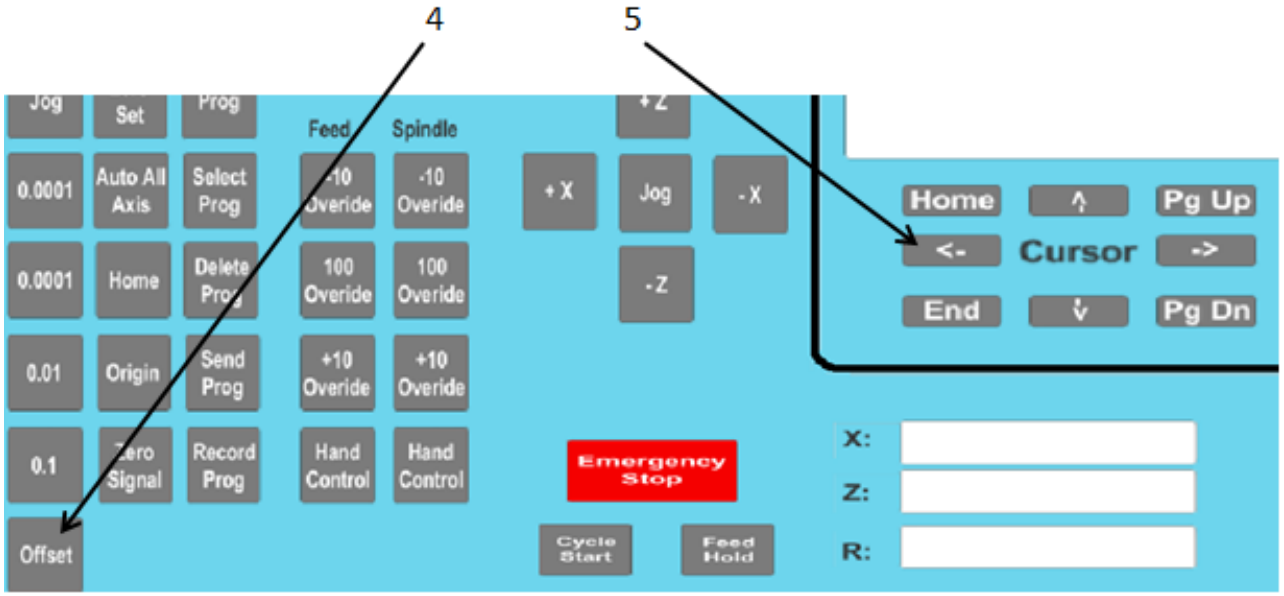
5.5 Setting Tool Length Offset - TLO

No	Procedure	Button Name and Location
1	**Jog**: Press Sets the machine to control by manual (use cursor keys to adjust the tool)	
2	**Jog Increment:** Press Set the jog increment to find the exact measurement.	
3	**Jog Direction :** Press Sets the tool to move in Z direction	



(a)

No	Procedure	Button Name and Location
4	**Offset**: Press Press until the too offset table displays	
5	**Cursor Arrows:** Use cursor keys for active tool to align	

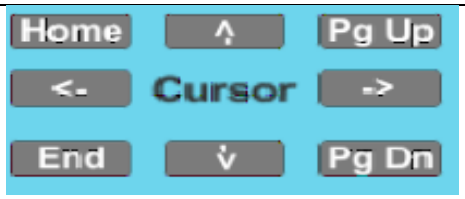
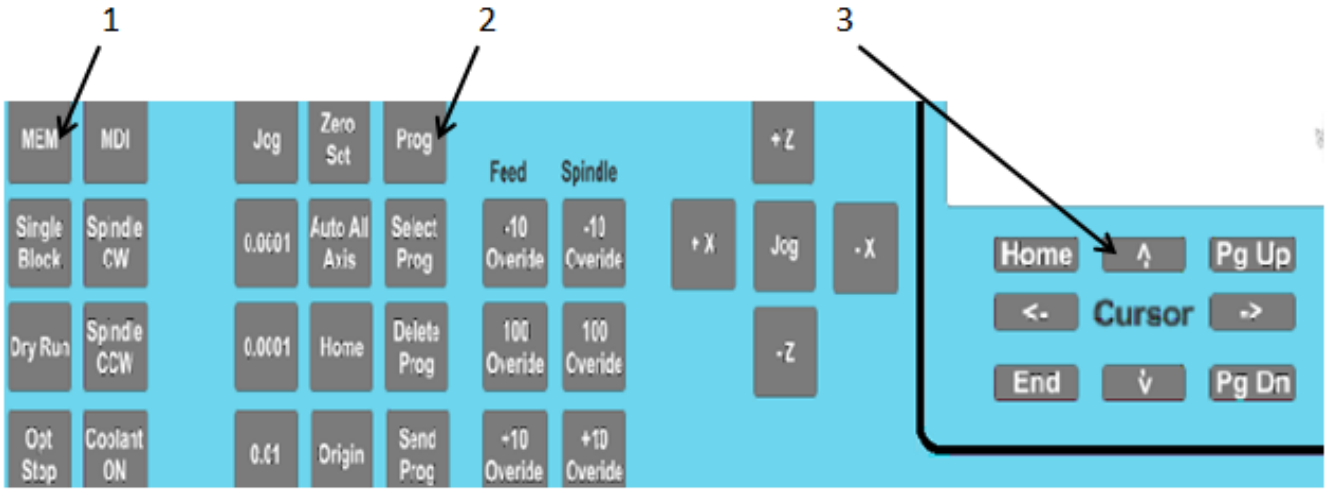
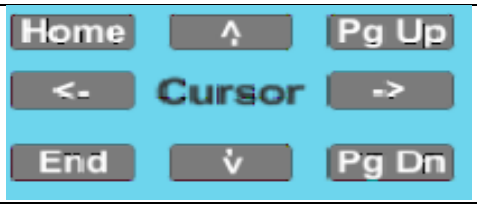


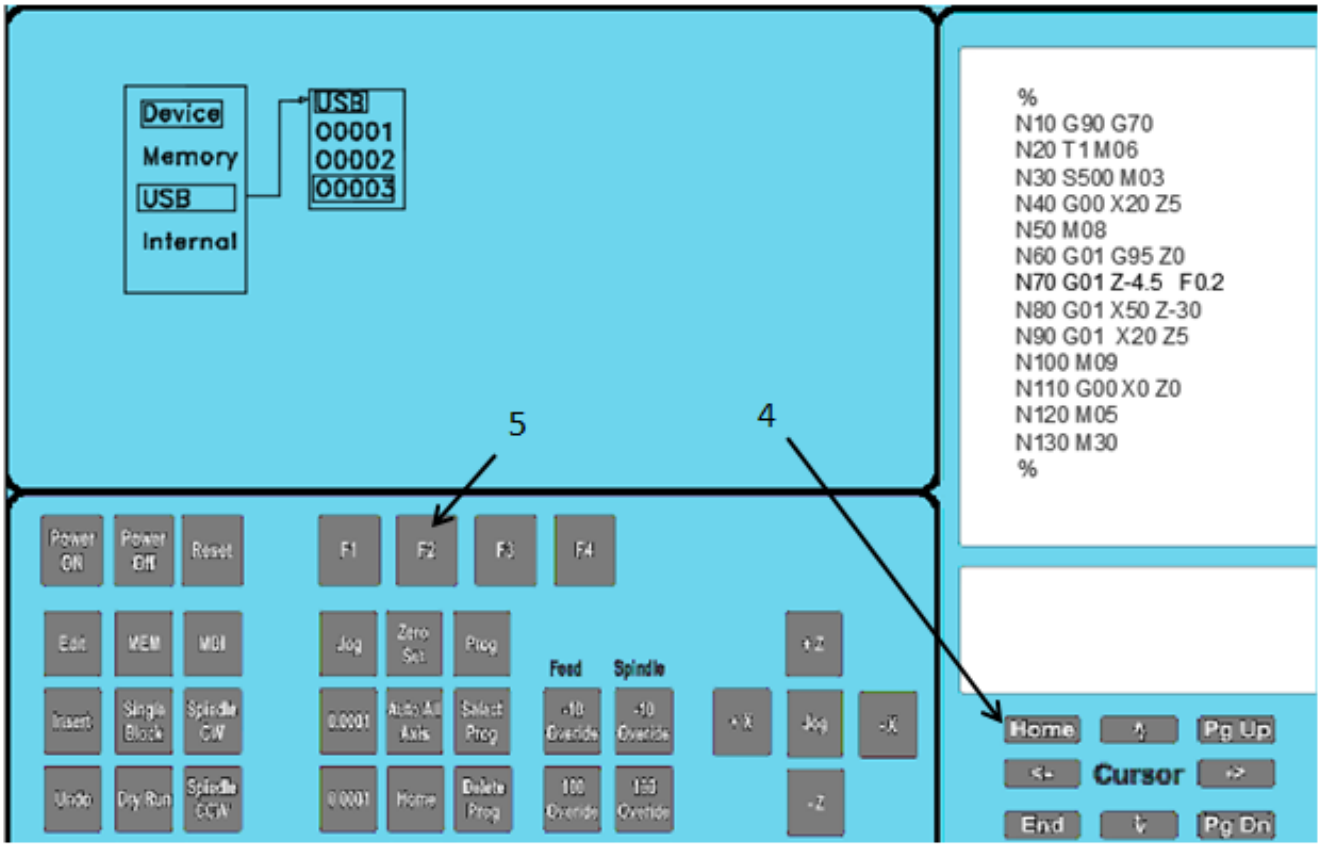
(b)

Figure : 5.4 Setting Tool Length Offset - TLO

Make sure the Tool Length Offset Table is updated before run.

5.6 Load CNC Program

No	Procedure	Button Name and Location
1	**Memory**: Press	
2	**Prog:** Press	
3	**Device :** Select **USB Device** Select USB Device from the table appeared on the screen. A list of files appeared on the screen. Select F3 to select the file.	
 (a)		
4	**Program Select**: Use Right Cursor Key to Highlight and select the desired file from USB	
5	**F2:** Press F2 to copy the program to internal memory of the machine. If a same name file exists, select “y” when prompts to overwrite.	

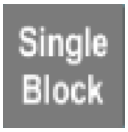


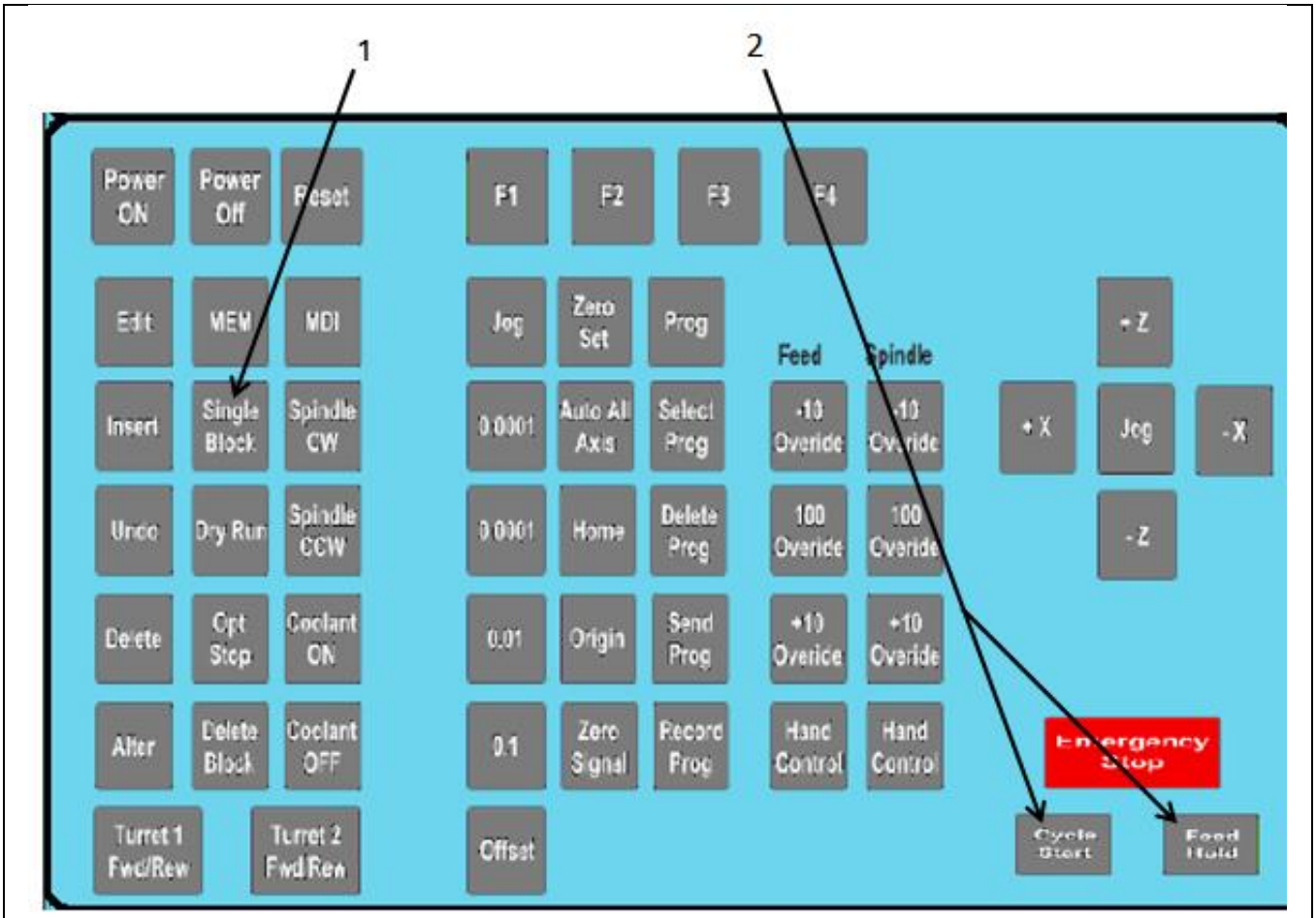
(b)

Figure : 5.6 Load CNC Program

5.7 Run CNC Program

Dry run is an appropriate method to verify a program before auto run, if program proven, feed rates can be set to 100% while single block mode to off, Figure 5.7

No	Procedure	Button Name and Location
1	**Pre-Start**: • Ensure that you have taken all the safety measures. • Material and tools have been properly installed • Ensure the area is cleaned and the door is closed. • Turn "**Single Block**" button on • Set Rapid Feed Rate to -10	 
2	**Start:** • Hold on to **Feed Hold** button and be ready to press it in case feed was not appropriate. • Press **Cycle Start** button	



5.7 Run CNC Program

Tool Length Offset can be accessed if change is required in any parameter in the table.

5.8 Machine Shut Down

No	Procedure	Button Name and Location
1	**Jog machine to safe area**: Select Jog	
2	**Shut Down Button: Press**	

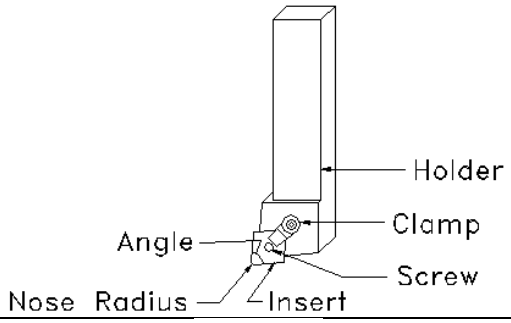
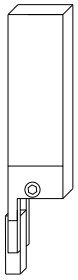
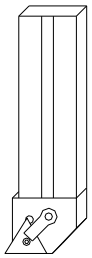
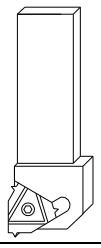
CNC Lathe – Machining

6.1 Overview

CNC lathes can be found with different specs even under the one roof. Some have two turrets with one spindle, some have two spindles with one turret, some have one spindle with one turret and some have milling or grinding options as well.

6.2 Type of Tools

It is important to have a glance on the tools you will be using with CNC lathe. CNC Lathes have left-handed and right-handed tools found in different size, angle, shape, tolerance, clearance, nose radius, insert and tool holder. Based on operations, a typical lathe can have five types of different tools shown in Table 6.1

No	Tool Name	Tool
1	**CNC Turning - Facing** For rough turning, use a rigid tool such as a round, square or 80 degree diamond. For finishing use more versatile tool, such as 55 or 35 degree diamond. Ensure each tool selected is right-sided for the respective side.	
2	**Groove** Width and corner radii justify the Groove tools. These tools are used mostly for making groves features such as O and Snap ring cuts. New version of the tools can also be used for rough, finish and contouring operations.	
3	**Bore** Holes are often finished and précised with a boring tool. Such tool requires a hole large enough to allow the bar to safely enter and exit.	
4	**Thread** Thread is an other important feature among the lathe operations. Thread tool is used to perform it.	

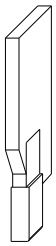
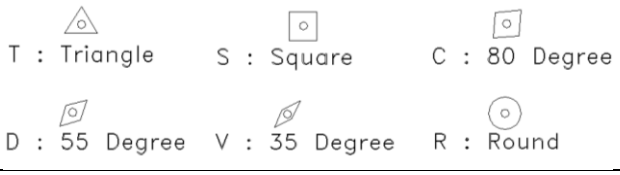
5	**Cutoff** A cut off tool is same as groove tool but designed for deeper cuts to remove the finished part/portion from the stock.	
Common Insert Shapes and Codes		

Table - 6.1 : Types of Tools

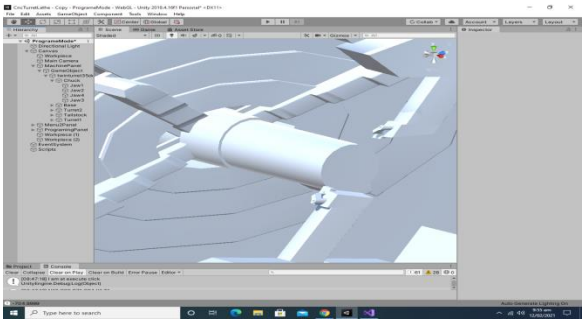
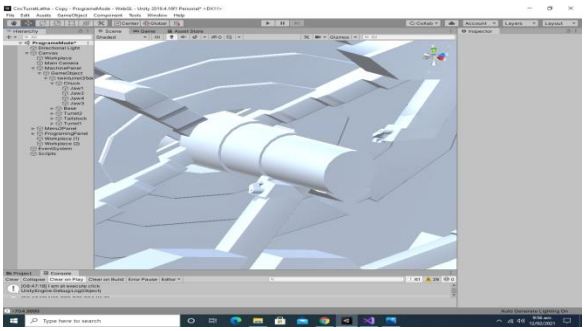
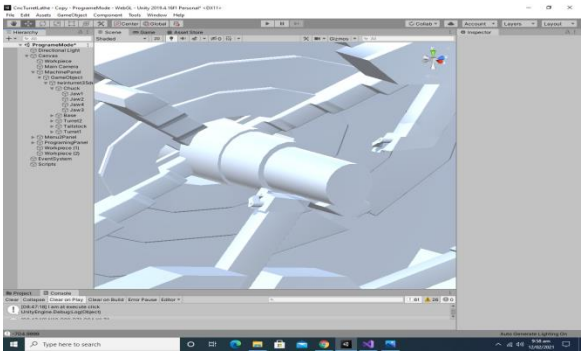
6.3.1 Facing

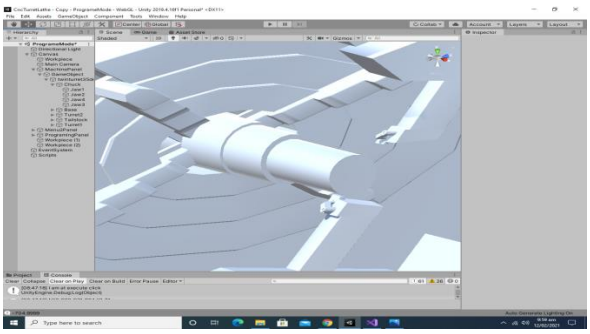
The part is gripped in hard jaws of the chuck and faced to provide a known location to set the Z-offset of other tools and a flat surface for face drilling. The zero for CNC lathe is usually the center-face of the finished part therefore, it is recommended to take at least one roughing and facing pass to ensure the precision.

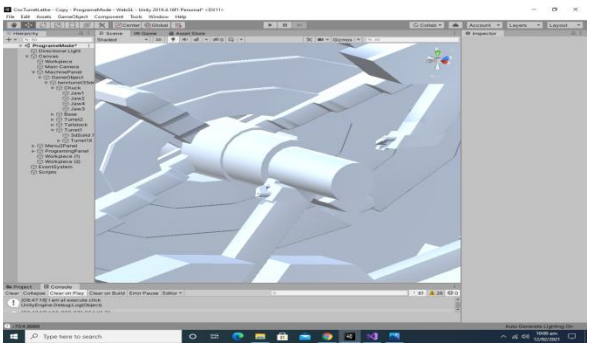
Important Rules

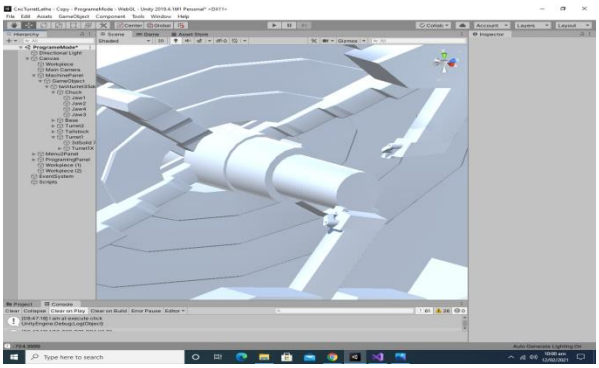
- Start from Outer Diameter (OD) of the stock as the rough stock diameter varies usually.
- Face straight up/down until the contact point of the tools reach at X0 and then pull away.
- Possibly taking one rough and finish pass.
- Using an 80 degree diamond rigid tool is highly recommended for the operation.
- Remove any extra tool or material lying on the machine bed.
- Check air pressure and coolant required if any.

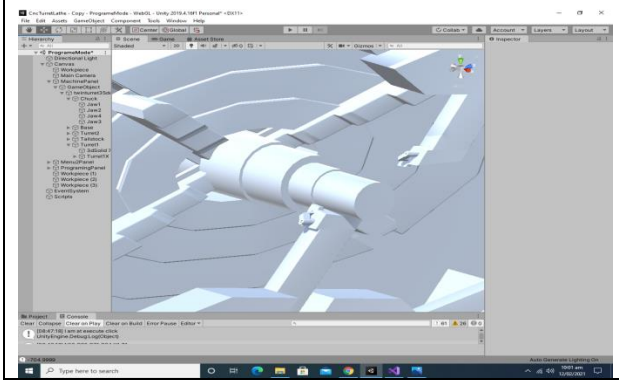
Sample Program for Facing with Turret1 & 2

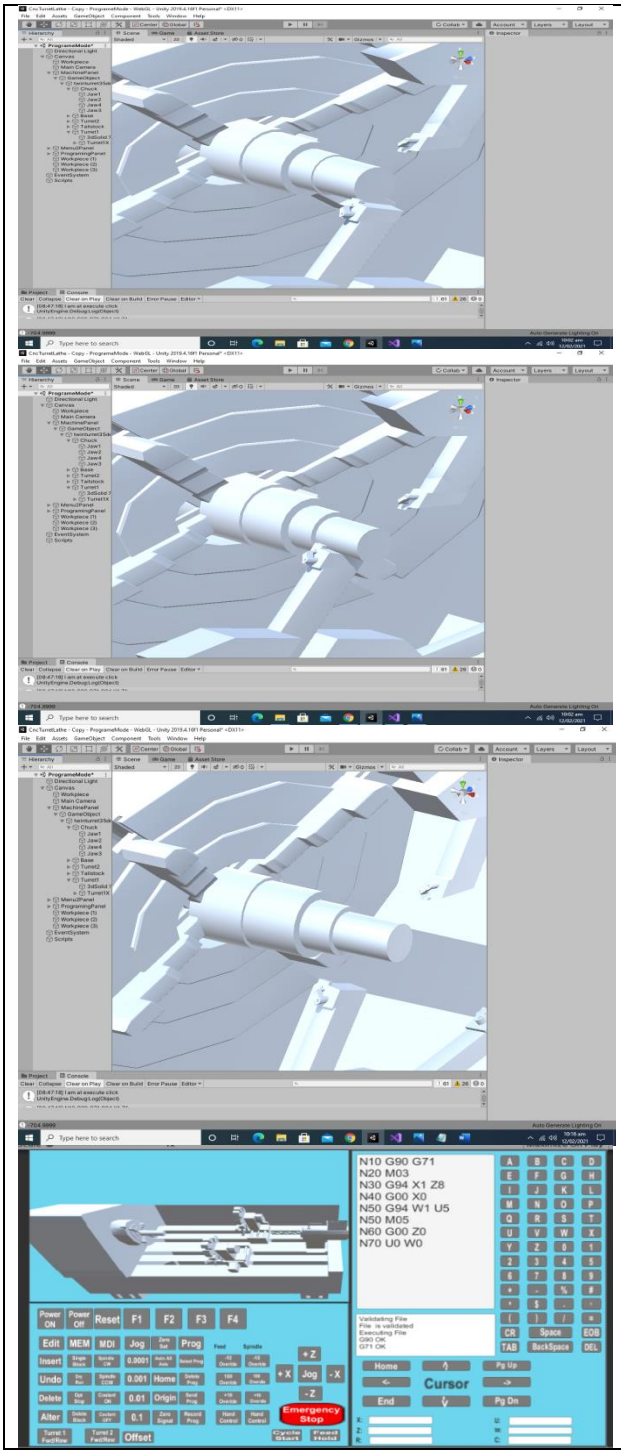
Status	Block
  	N10 G90 G71 N20 M03 N30 G20 X1 Z8 N40 G00 X-1 Z-8 X1 N50 G20 X3 Z6 N60 G00 X-1 Z-6 X1 N70 G20 X2 Z4.5 N80 M05 N80 G00 X0 Z0









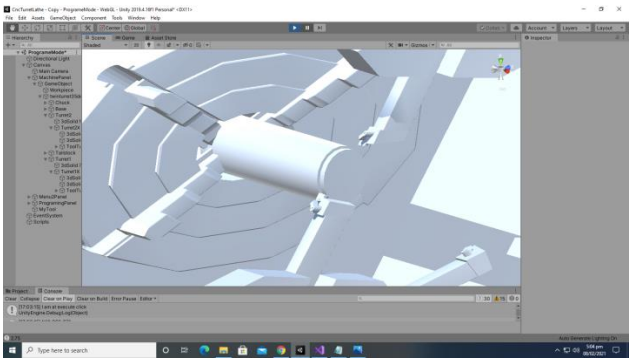
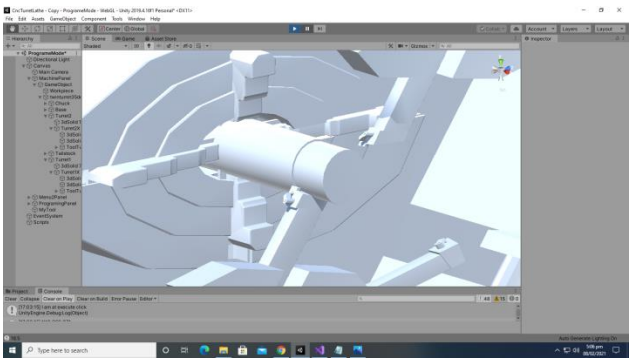
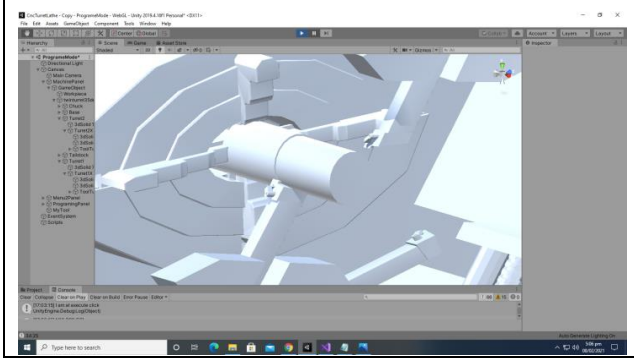

N10 G90 G71
N20 M03
N30 G94 X1 Z8
N40 G00 X0
N50 G94 W1 U5
N50 M05
N60 G00 Z0
N70 U0 W0
Validating File
File is validated
Executing File
G91 OK
G71 OK
Power ON
Power Off
Reset
F1
F2
F3
F4
Edit
MEM
MDI
Jog
Prog
Insert
Undo
Delete
Alter
Offset
+Z
+X
Jog
-X
-Z
Emergency Stop
Home
Cursor
End
Pg Up
Pg Dn
Space
BackSpace
TAB
CR
EOB
DEL

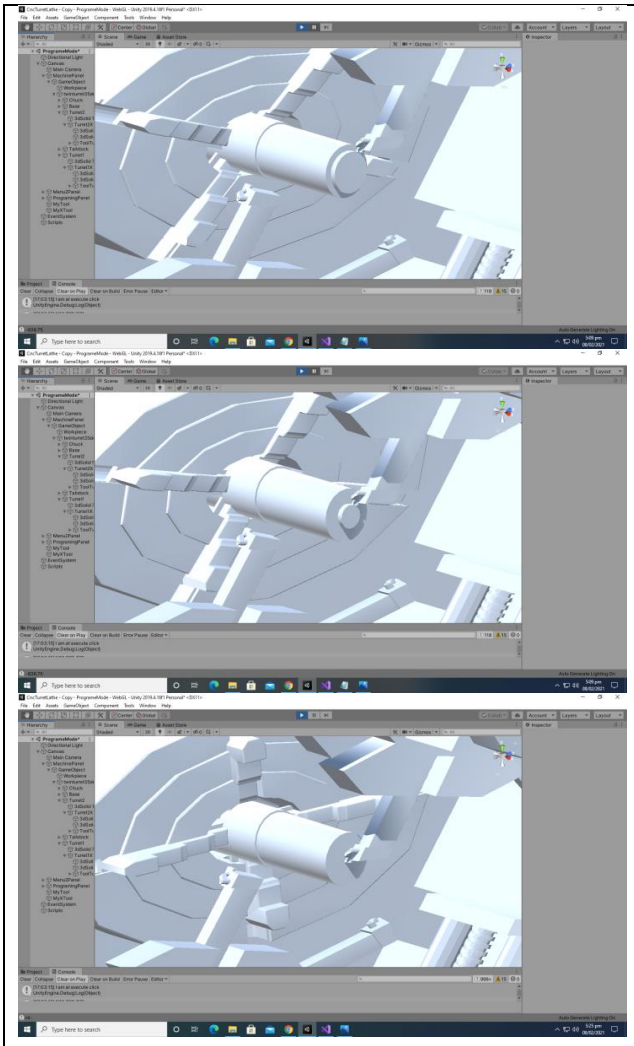
6.3.2 Rough and Finish

Rough the part leaving stock in Z and X for finish paths.

Rules for Roughing and Finishing

- Start slightly away from the face of the part.
- It is recommended to use the same tool for both facing and OD roughing when possible.
- A Constant thickness of material over the part for the finishing operation is required.
- Avoid to grooves and few other features that will be finished with some other tools.
- Moving the tool at the end of each path to remove scallops is highly recommended.
- Start slightly away from the part face.
- Use diamond tool, with sufficient side and end cutting angles to provide tool clearance.
- Use a nose radius tool equal or smaller, than the smaller ID radius of the profile.
- Skip the grooves as they will be machined using other tools and operations.
- Finishing at the major diameter of the thread is highly advised.

Status	Block
  	N10 G90 G71 N20 M03 N30 G94 X1 Z8 N40 G00 X0 N50 G94 W1 U5 N50 M05 N60 G00 Z0 N70 U0 W0

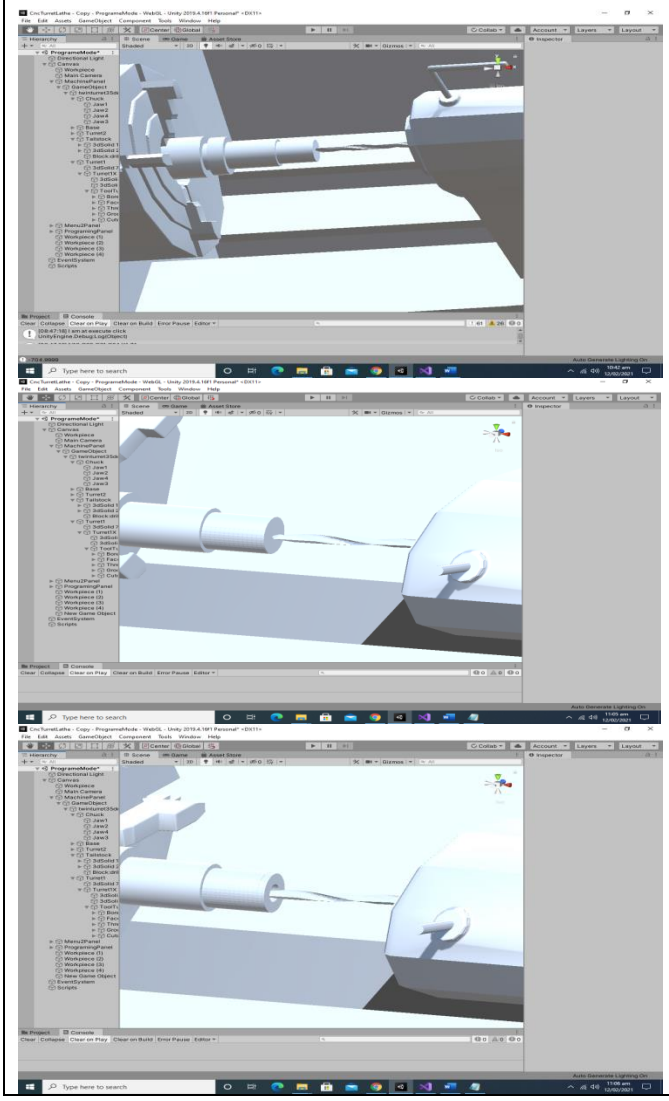


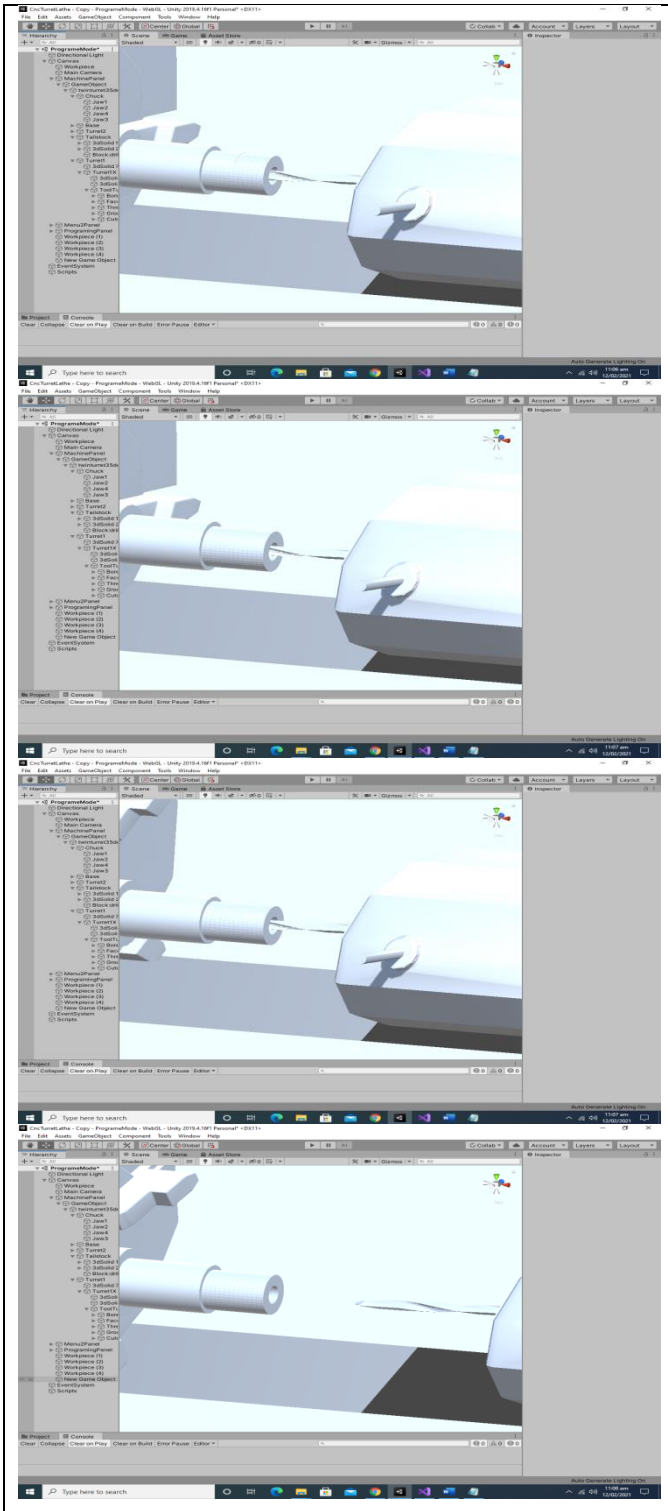
6.3.3 Drill

CNC Lathes support common drill G-code cycles including G81.

Rules for Drilling

• Spot drilling.
• Considering progressively drills to make a large hole.
• Peck drill holes.
• Use a canned cycle if possible..
• Use the shortest drill possible to produce more accurate hole.

Status	Block
	N10 G90 G71 N20 M03 N30 G81 R-3 N40 G01 R0 N50 M05 N60 G00 R50



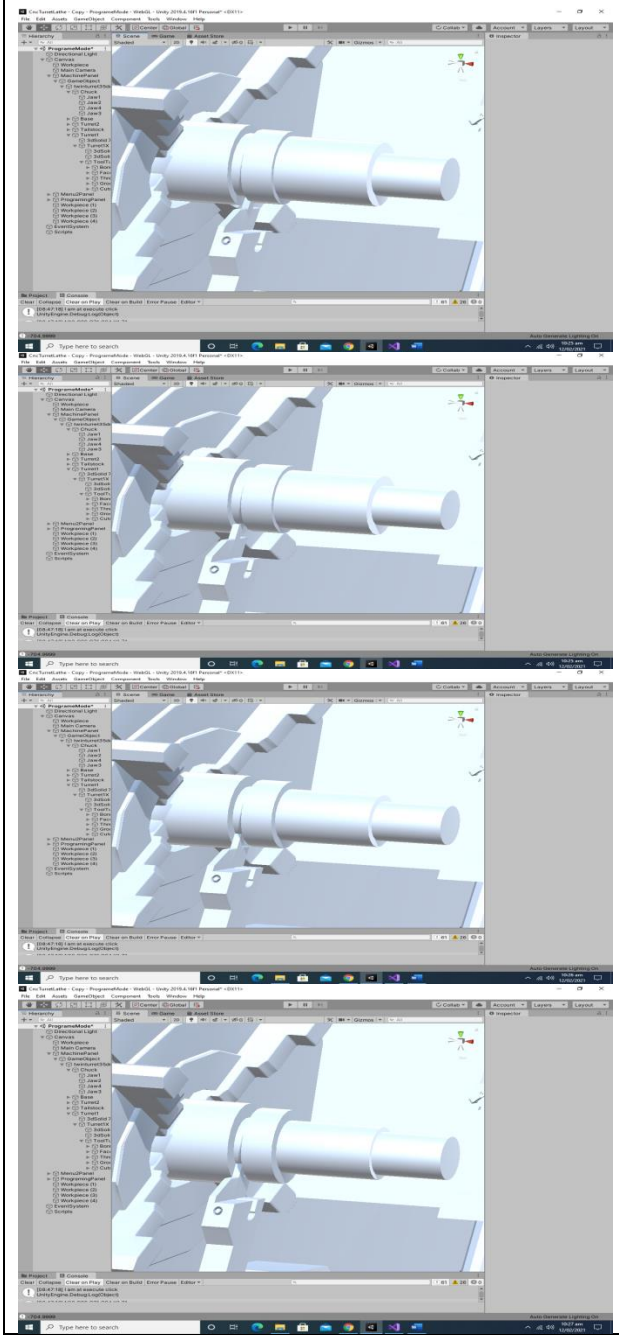
6.3.4 Groove

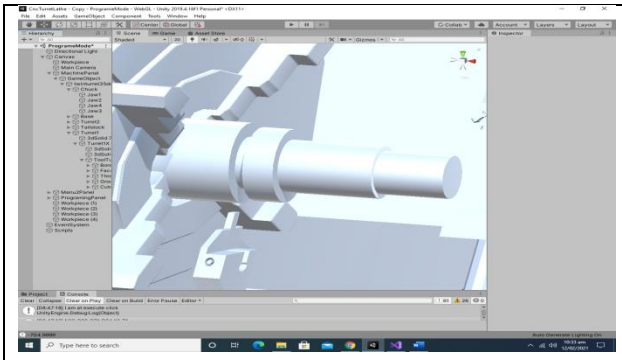
The following sequence is often used to create a groove.

A) The tool is usually plunged near the center of the groove. .
B) Other plunge is required to remove excess material.
C) A contouring path, starting outside and moving inward to create radii and contour.

Rules for Grooving

• Use a groove tool narrower than the groove.
• Use lathe control groove cycles to program the grooves.

Status	Block
	N10 G90 G71 N20 M03 N30 G81 R-3 N40 G01 R0 N50 M05 N60 G00 R50

6.3.5 Threads

Lathes can create any type of thread, including tapered pipe threads, OD, ID, and multi-start. They are done mostly with an insert. Many roughing passes remove a small amount of material to form a thread that is accurate and smooth. A CNC Program is written in a way that each time the tool begins the cut pass at the same rotational position.

Rules for Threading

- The manufacturer's instructions strictly be followed for thread parameters if any.
- Use lathe control thread canned cycles. They look same as mill drill cycle to program all the motion required to form the thread.
- Spindle timing is important to reach full speed before engaging the thread.

Status	Block
Awaiting for input	

6.3.6 Cutoff

This can be the last machining operation on CNC lathes. It separates the part from the stock.

Rules for Cutoff

- Cutoff is same as grooving operations.
- The tool usually retracts, moves back, and then plunges to cut away the part.
- Part catching facility keeps the part from falling into the bottom of the machine and protects from being scratched by chips.

Status	Block
Awaiting for input	