# Practical No.4

## Objective:

Manual Part Programming on CNC Milling,  
i. Introduction to G-Codes & M-Codes  
ii. Formulation of part programming in cimco. Use ISO milling operations, and write your roll no. as 1333

## Tools & Equipment:

Computer with CIMCO Installed.

## Theory: Introducton to G Codes & M Codes

**G-code** is the common name for the most widely used [numerical control](http://en.wikipedia.org/wiki/Numerical_control) (NC) [programming language](http://en.wikipedia.org/wiki/Programming_language), which has many [implementations](http://en.wikipedia.org/wiki/Implementation). Used mainly in [automation](http://en.wikipedia.org/wiki/Automation), it is part of [computer-aided engineering](http://en.wikipedia.org/wiki/Computer-aided_engineering). G-code is sometimes called **G programming language**.

In fundamental terms, G-code is a language in which people tell computerized [machine tools](http://en.wikipedia.org/wiki/Machine_tool) what to make and how to make it. The "how" is defined by instructions on where to move to, how fast to move, and through what path to move. The most common situation is that, within a [machine tool](http://en.wikipedia.org/wiki/Machine_tool), a [cutting tool](http://en.wikipedia.org/wiki/Cutting_tool_(machining)) is moved according to these instructions through a tool path, cutting away excess material to leave only the finished work piece. But the same concept also extends to noncutting tools, such as forming or burnishing tools; to [additive](http://en.wikipedia.org/wiki/3D_printing) methods; and to measuring probes that validate the results.

Codes that begin with M are called miscellaneous words. They control machine auxiliary options like coolant and spindle direction. Only one M-code can appear in each block of code.

**LIST of Codes and their function:**

G00 Rapid move G0 X# Y# Z# up to eight axes or G0 Z# X#

G01 Feed Rate move G1 X# Y# Z# up to eight axes or G1 Z# X#

G02 Clockwise move

G03 Counter Clockwise move

G04 Dwell time G04 L#

G08 Spline Smoothing On

G09 Exact stop check, Spline Smoothing Off

G10 A linear feedrate controlled move with a decelerated stop

G11 Controlled Decel stop

G17 XY PLANE

G18 XZ PLANE

G19 YZ PLANE

G28 Return to clearance plane

G33 Threading (Lathe)

G35 Bypass error checking on next line

G40 Tool compensation off

G41 Tool compensation to the left

G42 Tool compensation to the right

G43 Tool length compensation - negative direction

G44 Tool length compensation - positive direction

G49 Tool length compensation cancelled

G53 Cancel work coordinate offsets

G54-G59 Work coordinate offsets 1 through 6

G61 Spline contouring with buffering mode off

G64 Spline contouring with buffering mode on

G65 Mill out rectangular pocket

G66 Mill out circular pocket

G67 Flycut

G68 Mill out rectangular pocket with radius corners

G70 Inch mode

G71 Millimeter mode

G74 Peck drilling (Lathe) G83 Z# X# R#

G81 Drill cycle G81 X# Y# Z# R#

G82 Dwell cycle G82 X# Y# Z# R#

G83 Peck cycle G83 X# Y# Z# R#

G84 Tapping cycle G84 X# Y# Z# R# C#

G85 Boring cycle 1 G85 X# Y# Z# R#

G86 Boring cycle 2 G86 X# Y# Z# R#

G88 Boring cycle 3 G88 X# Y# Z# R#

G89 Boring cycle 4 G89 X# Y# Z# R#

G90 Absolute mode

G91 Incremental mode

G92 Home coordinate reset G92 X# Y# Z#

G94 IPM mode (Lathe) default

G95 IPR mode (Lathe)

G96 Constant Surface Feed On (Lathe)

G97 Constant Surface Feed Off (Lathe)

G110 Lathe Groove Face

G111 Lathe Groove OD

G112 Lathe Groove ID

G113 Lathe Thread OD

G114 Lathe Thread ID

G115 Lathe Face Rough

G116 Lathe Turn Rough

G120 Mill Outside Square

G121 Mill Outside Circle or Island

G122 Mill Out Counter Bore

G123 Mill Outside Ellipse pocket

G124 Mill Inside Ellipse pocket

G125 Mill Outside Slot

G126 Mill Inside Slot pocket

G130 3D tool compensation with gouge protection

G131 3D offset parallel to 3D profile

G132 3D tool compensation with gouge protection in the Z axis only

G135 5 axis tool compensation with gouge protection

G136 Included angle limit for gouge protection. G136 L#

G140 3D part rotation and plane tilting G140 U# V# W# R#

G141 Scale factor for the X axis only. G141 L#

G142 Scale factor for the Y axis only. G142 L#

G143 Scale factor for the Z axis only. G143 L#

G160 Mill 3D Cylinder

G162 Mill 3D Sphere

G163 Mill 3D Ramped Plane

G170 Set soft limits and crash fixture/chuck barriers to defaults

G171 Set backward crash fixture/chuck barriers G171 U# V# W#

G172 Set forward crash fixture/chuck barriers G172 U# V# W#

G181 Bolt Hole Drill

G182 Bolt Hole Dwell

G183 Bolt Hole Peck

G184 Bolt Hole TapG185 Bolt Hole Bore

**Common M Codes**

Default M codes used on most machines types. User customizable M codes will change based on application and user definition.

M02 End of Program

M03 Spindle On Clockwise, Laser, Flame, Power ON

M04 Spindle On Counter Clockwise

M05 Spindle Stop, Laser, Flame, Power OFF

M06 Tool Change

M08 Coolant On

M09 Coolant Off

M10 Reserved for tool height offset

M13 Spindle On, Coolant On

M30 End of Program when macros are used

M91 Readout Display Incremental

M92 Readout Display Absolute

M97 Go to or jump to line number

M98 Jump to macro or subroutine

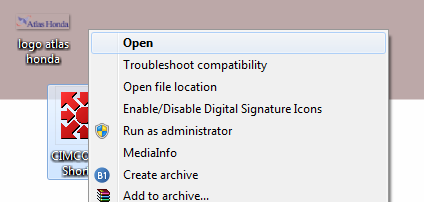
M99 Return from macro or subroutine

M100 Machine Zero Reset

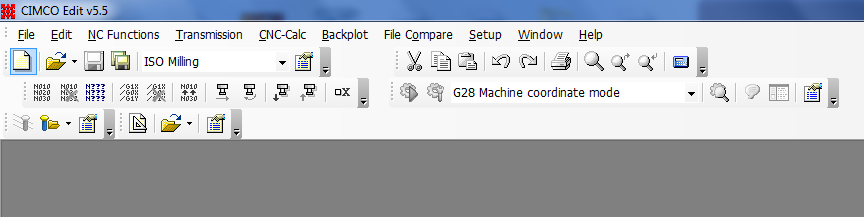
M199 Mid program start

**B. Using ISO Milling, simulation of tool path like my roll number “1333”**

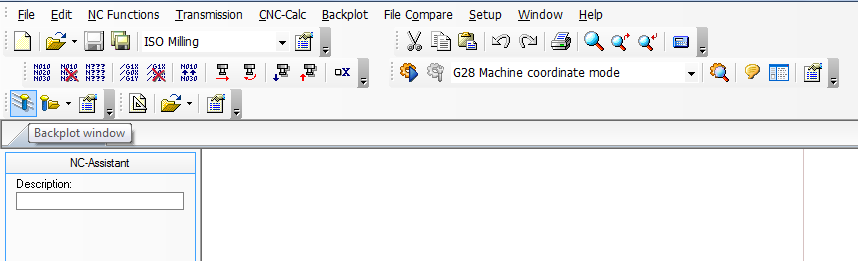
**Step 1. Start CIMCO EDIT Software**



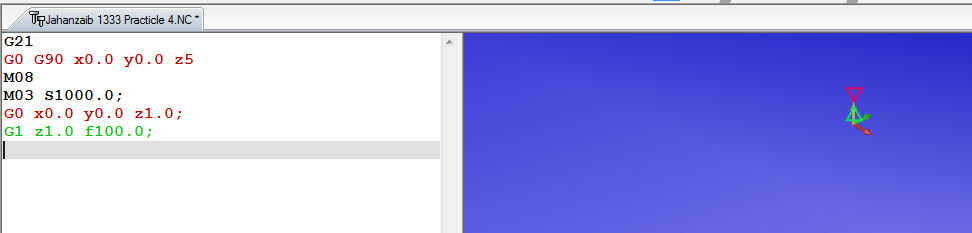
**Step 2. Click New in the top end corner below file.**



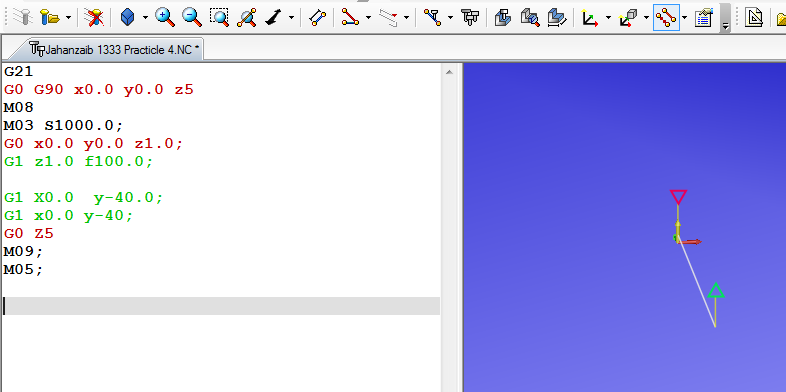
**Step 3. Click Back plot window to see a real time effect of code we have entered.**



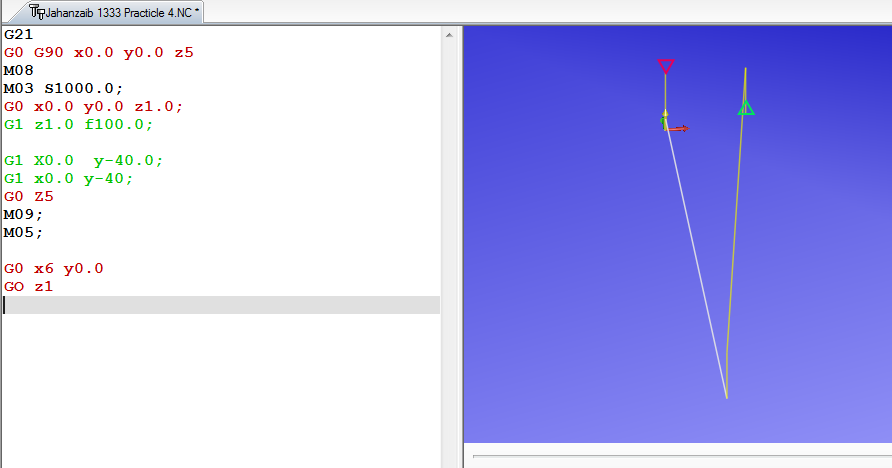
**Step 4. Enter the basic machine codes and start defining the geometry to be created by milling operation. The code shows that the tool is ready.**



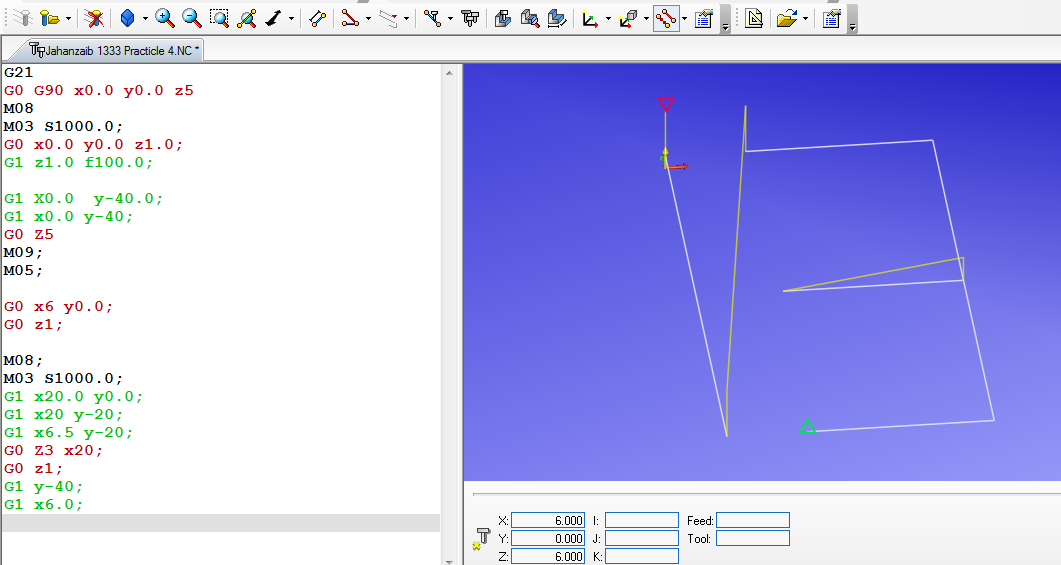
**Step 5. Enter the code for 1. After entering the code we see that the spindle & coolant are turned off while the spindle is ready to move for next digit ‘3’.**



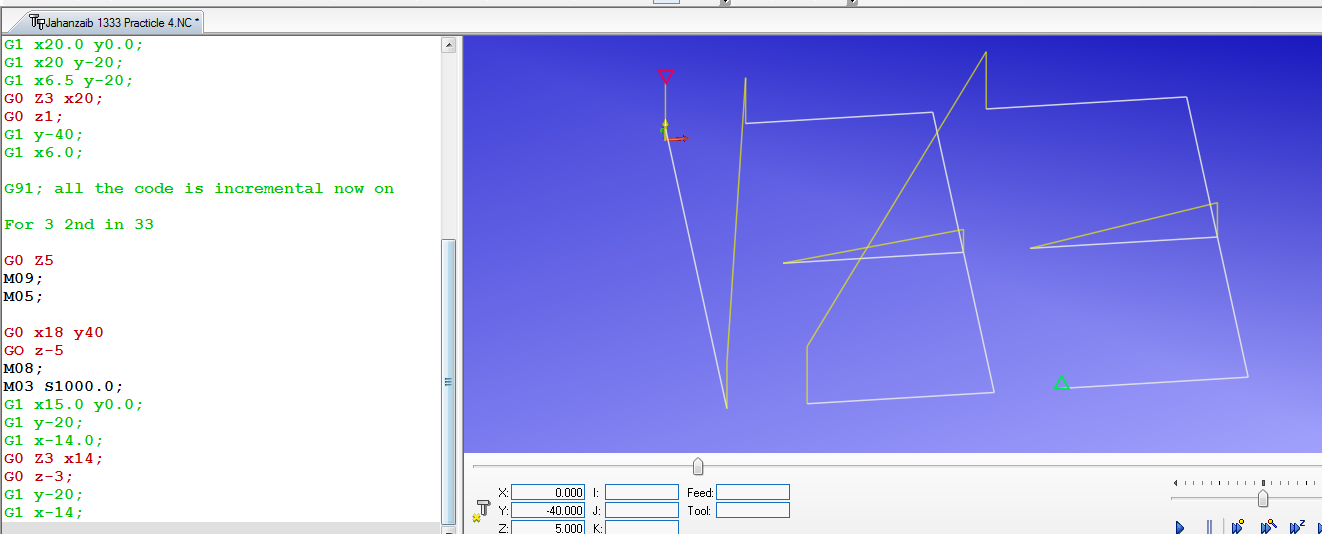
**Step 6. Move the spindle at height Z=1 to next point as follows:**



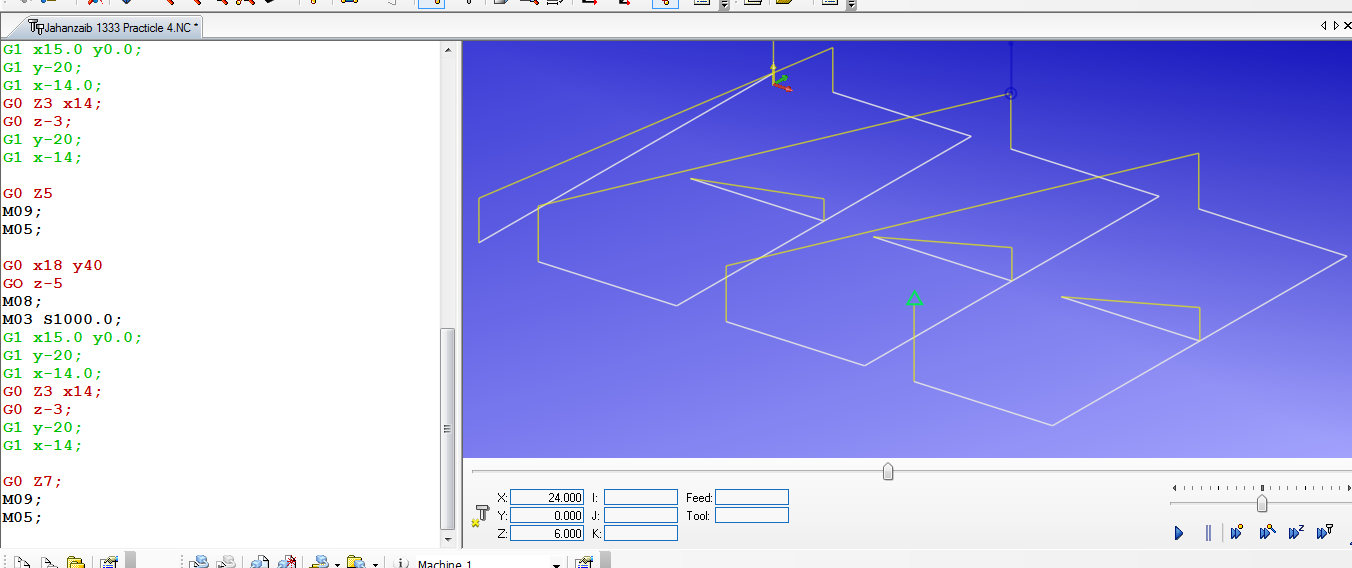
**Step 7. Define geometry for digit 3 in absolute positioning.**



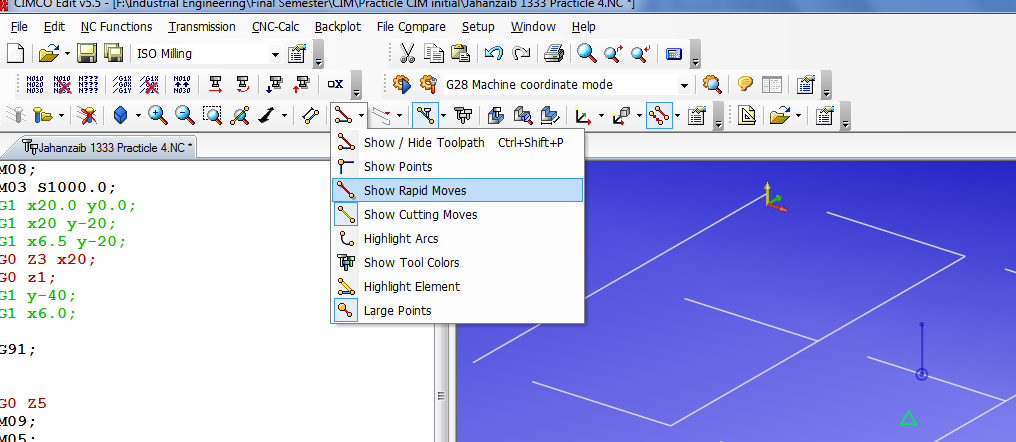
**Step 8. Enter code G91 (All the codes are incremental now) and define geometry for next digit i.e 3.**



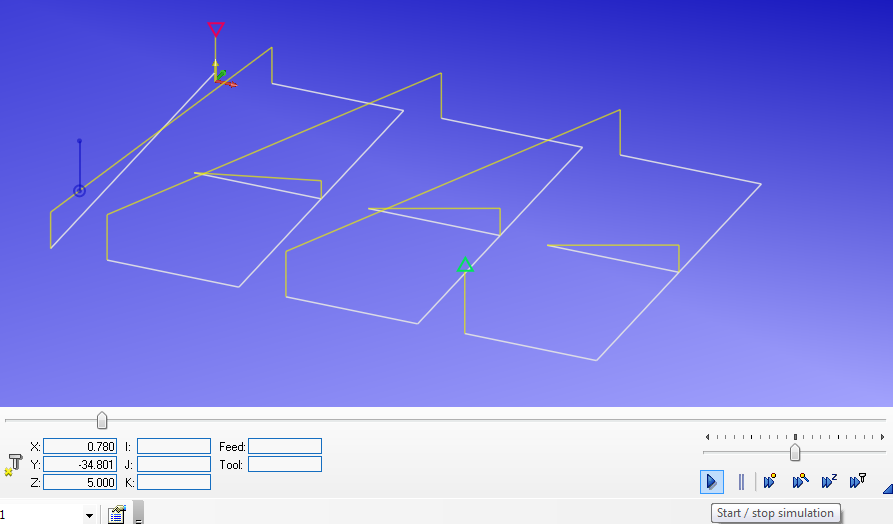
**Step 9. Copy the same incremental code as in step 8 along with machine codes to generate the similar tool path as we have the same digit, i.e 3.**



**Step 10. Hide rapid moves of spindle.**

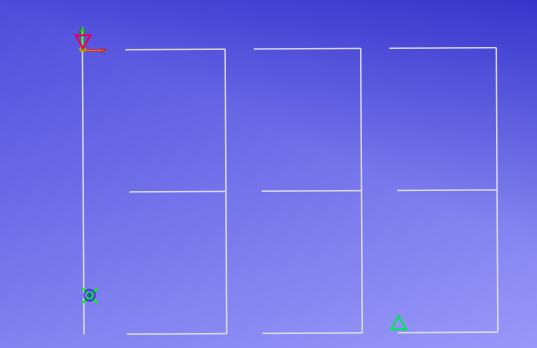


**11. The formulation of Code is complete. Run the Simulation**

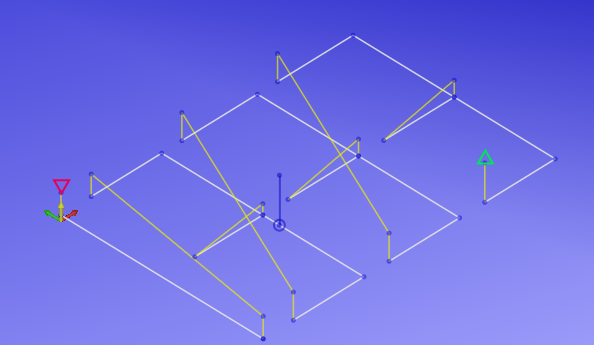


**Results:**

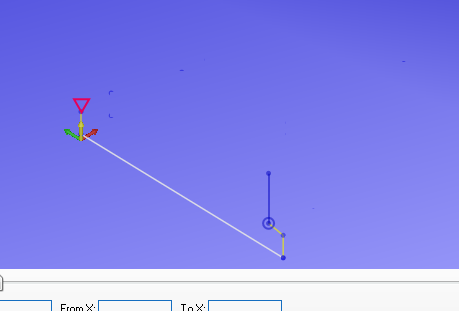
1. **Following Geometry is generated with the code we made:**

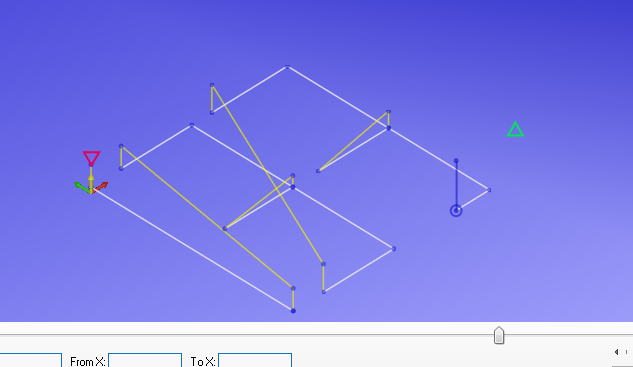


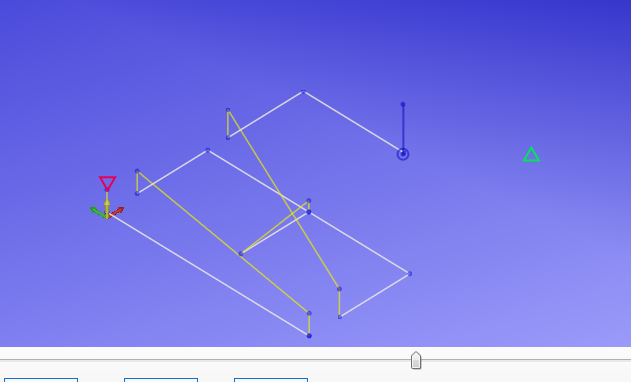
## 

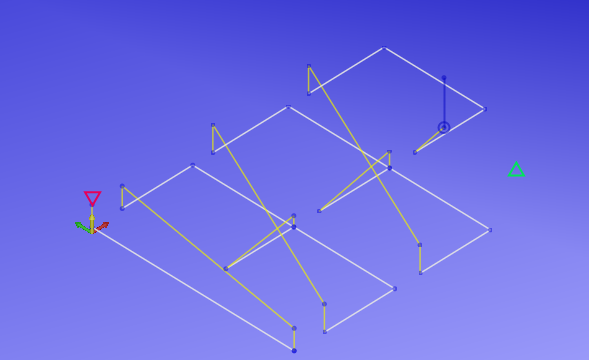


**Result 2. The simulation was carried out successfully.**









**Result 3.**

**Following Code was successfully simulated for defining the geometry of the Roll Number 1333.**

Codes for writing 1333

**Start Here**

G21

G0 G90 x0.0 y0.0 z5

M08

M03 S1000.0;

G0 x0.0 y0.0 z1.0;

G1 z1.0 f100.0;

G1 X0.0 y-40.0;

G1 x0.0 y-40; G0;

G0 Z5

M09;

M05;

**For 3**

G0 x6 y0.0

GO z1

M08;

M03 S1000.0;

G1 x20.0 y0.0;

G1 x20 y-20;

G1 x6.5 y-20;

G0 Z3 x20;

G0 z1;

G1 y-40;

G1 x6.0;

G91; **all the code is incremental now on**

**For 3 2nd in 33**

G0 Z5

M09;

M05;

G0 x18 y40

GO z-5

M08;

M03 S1000.0;

G1 x15.0 y0.0;

G1 y-20;

G1 x-14.0;

G0 Z3 x14;

G0 z-3;

G1 y-20;

G1 x-14;

**For 3rd 3 in 1333**

G0 Z5

M09;

M05;

G0 x18 y40

GO z-5

M08;

M03 S1000.0;

G1 x15.0 y0.0;

G1 y-20;

G1 x-14.0;

G0 Z3 x14;

G0 z-3;

G1 y-20;

G1 x-14;

G0 Z7;

M09;

M05;