Here is a quick listing of G-codes
Posted by ivanirons - 2008/01/21 21:38

Common Fanuc G Codes Code Description

G00 Rapid positioning
G01 Linear interpolation
G02 CW circular interpolation
G03 CCW circular interpolation
G04 Dwell
G05.1 Q1. Ai Nano contour control
G05 P10000 HPCC
G10/G11 Programmable Data input/Data write cancel
G17 X-Y plane selection
G18 X-Z plane selection
G19 Y-Z plane selection
G20 Programming in inches
G21 Programming in mm
G28 Return to home position
G30 2nd reference point return
G31 Skip function (used for probes and tool length measurement systems)
G33 Constant pitch threading
G34 Variable pitch threading
G40 Tool radius compensation off
G41 Tool radius compensation left
G42 Tool radius compensation right
G43 Tool offset compensation positive
G44 Tool offset compensation negative
G45 Axis offset single increase
G46 Axis offset single decrease
G47 Axis offset double increase
G48 Axis offset double decrease
G49 Tool offset compensation cancel
G53 Machine co-ordinate system
G54 to G59 Work co-ordinate systems
G54.1 P1 to P48 Extended work co-ordinate systems
G73 High speed drilling canned cycle
G74 Left hand tapping canned cycle
G76 Fine boring canned cycle
G80 Cancel canned cycle
G81 Simple drilling cycle
G82 Drilling cycle with dwell
G83 Peck drilling cycle
G84 Tapping cycle
G84.2 Direct right hand tapping canned cycle
G90 Absolute programming (type B and C systems)
G91 Incremental programming (type B and C systems)
G92 Programming of absolute zero point
G94/G95 Inch per minute/Inch per revolution feed (type A system)
G98/G99 Inch per minute/Inch per revolution feed (type B and C systems)
G96/G97 Constant cutting speed (Constant surface speed)/Constant rotation speed (constant RPM)

==========

Re:Here is a quick listing of G-codes
Posted by shogun - 2008/03/11 12:12

---

hey thanks its very useful, for those of us who are starting

==========

Re:Here is a quick listing of G-codes
Posted by involute - 2008/06/17 12:27

---

am looking for Sinumeric 802D G codes ,M codes and D codes

==========

Re:Here is a quick listing of G-codes
Posted by nomaihg - 2008/06/24 12:36

---

hai hai hai ,,,very nice topic and i saved this G-codes for the future ,,, thanx

==========

Re:Here is a quick listing of G-codes
Posted by FallenZERO - 2008/08/04 20:22

Thank you, this really helped.

==========

Re:Here is a quick listing of G-codes
Posted by franc33 - 2008/08/17 10:10

---

ciao,

very useful for the study of the code and relative simulations.

http://www.cncezpro.com/index.cfm

franc

==========

Re:Here is a quick listing of G-codes
Posted by cncbayer - 2008/09/10 15:21

Thanks and are youu fimilar with fadal's (1990)s style

==========

Re:Here is a quick listing of G-codes
Posted by jdc - 2008/09/11 09:08

What did you need to know about Fadal?

==========

Re:Here is a quick listing of G-codes
Posted by sdtasman - 2008/09/12 05:51

I reckon we could do a bit better by adding the G-Code for turning as well.
And would it be spoon feeding if we laid out the details of each code.
Eg. G71 U_ R_
G71 P_ Q_ U_ W_ R_ F_
G80

And explained how it works.

I have found quite often Using code manaually when your in front of the machine at that moment is a lot more efficient

than sitting down in front of a Cam Software program and spitting out childlike programs to the machine. Understanding and using canned cycles can be so much quicker and use much less memory to run, most cases even having to do away with part programming.

==========

Re:Here is a quick listing of G-codes
Posted by Machine Man - 2008/10/07 21:53

Awesome ivan! this will really help me in the future! I really want to learn more about programming!:woohoo:

==========

Re:Here is a quick listing of G-codes
Posted by ddoug - 2009/01/28 02:45

A good listing of G Code is always useful. Thanks.

Even if not all are applicable to our machine.

But any idea why the G Code Quick Book has trouble downloading? Can't download or view ebook. The CNC Info ebook downloaded just fine.

dick

==========

Re:Here is a quick listing of G-codes
Posted by highbird - 2010/07/23 06:00

Thanks for the information ! and this link is useful :cheer:

==========

Re:Here is a quick listing of G-codes
Posted by r2d2f16 - 2010/07/24 04:38

---

this is nice for learning thanks:cheer:

==========

Re:Here is a quick listing of G-codes
Posted by lksahu - 2010/08/21 05:39

THANKS FOR THE LIST

==========

Re:Here is a quick listing of G-codes
Posted by motel - 2010/09/15 17:30

I thought G70 and G71 were inch and millimeter codes. Anyway I heard once that you should master I J and K programming since some machines don't recognize the Radius words. Is this true? Thx.
Matt

==========

Re:Here is a quick listing of G-codes
Posted by govermentcheese - 2010/09/16 18:38

Hello starting to learn g codes and my first project is to spell my initials of my name which are CTR we are doing this on a 2inch by 3 inch graph paper i understand most of the concept but I am having a problem on how to start the g codding sequence in ncplot to see how it is turning out. Iam 50 years old in 2 weeks. I lost my job last year company sold and moved over seas. I had 16 years there I ran a one man brake press so I have a good understanding of machining equipment. They had plasma and other gcode machines next to me. While plotting this out in ncplot program will I see it as I put it in the program.I think I have the c allready done . My class is only 2 hours a day twice a week have to learn fast.. Can anyone help me on this..

==========

Re:Here is a quick listing of G-codes
Posted by BlackOnyx - 2010/09/18 07:11

My experience with G2/G3 XYZ R or G2/G3 XYZ IJK is that the R option only works if the arc either starts or ends exactly on one of the quadrants, if it is an arc which does not either start and/or end on a quadrant (XYZ axis), then the R option does not work. This is why when I set up a post processor for a CAM system, I always revert to the IJK option, because it covers all arc possibilities.
For anyone who doesn't yet understand the difference,
 http://www.cncinformation.com/images/fbfiles/images/Arc.jpg
G1 X+1 Y+2 (position at the start of the arc)
G2 X+0 Y+1 R1 (perform the arc)

G1 X+1 Y+2 (position at the start of the arc)
G2 X+0 Y+1 I-1 J+0 (perform the arc)

The IJK represent incremental values (- or + direction as for the axes) in XYZ axes, from the position at the start of the arc, to the center point of the arc, in this case, I is -1, because the center of the arc is in the - direction along X axis. J is 0, because the arc center is on the same Y position, as the arc start. (R on some controls may be CR)
Also, note that the zeros in codes are not necessary,
G00/G0 G01/G1 G02/G2 G03/G3 G04/G4 M00/M0 M01/M1 etc. are functionally identical to a control.

==========

Re:Here is a quick listing of G-codes
Posted by BlackOnyx - 2010/10/19 01:36

Some very good general CNC code info
http://en.wikipedia.org/wiki/G-code

==========