



# KipwareXC®

CNC - G Code Conversion Software

*KipwareXC® from Kentech Inc. is an innovative and powerful application that can help your shop floor eliminate the “control incompatibility” issues that plague so many shops. Whether your “control incompatibility” issues stem from re-posting or re-programming G code to move it around the shop ... to forcing the purchasing of new equipment that only fits into your programmers wheelhouse ... or having to purchase expensive post processors for every machine control on the floor ... KipwareXC® can solve them all.*

***Below is a feature list for KipwareXC® ... that when explored in-depth will have you thinking about converting and editing G code in a whole new light.***

**STANDARD** KipwareXC® Features include :

1. Users can define and create an **unlimited number of PROFILES** which basically define the KipwareXC® what-to-convert-into-what. That means a single purchase of KipwareXC® can solve ALL your shop floor control incompatibility issues.
2. **Multiple machine types are supported** including diameter X lathes, radius X lathes, any type of equipment with X/Y/Z movement like vertical and horizontal mills and more.
3. KipwareXC® can convert complete G code programs from **INCH to METRIC** ... and reverse ... with a single click. In addition, users have complete control over letter addresses that are converted ... for example, don't convert S commands or do convert U and W commands.
4. Use user defined **START and STOP commands** to define and convert only sections of a G code program rather than the complete program.
5. In milling programs, **auto convert INCREMENTAL X/Y/Z commands to ABSOLUTE X/Y/Z commands** ... with a single click.
6. The **powerful KipwareXC® MACRO language** allows users to move beyond simple search and replace to powerful search and replace functions using user defined parameters and trigger codes.

7. The number of KipwareXC® **MACRO definitions** in each conversion PROFILE **is unlimited**. That means the amount and type of conversions that will be auto-executed in one conversion sequence is limitless. Creating an even more powerful scenario ... macros are executed in the order in which they are entered. They can be arranged in such a way to add even more power and flexibility to the conversion process.
8. Define an **unlimited Complete Line Replacements** using user defined trigger codes. For example ... use G28 as a trigger and remove the line completely or replace it with a user defined line.
9. Delete **BLANK LINES** during the conversion
10. Delete **SPACES** during the conversion
11. Auto **RE-SEQUENCE** programs after the conversion

#### **12. Circular Conversion Options :**

- a. Convert R to INCREMENTAL I/J/K
- b. Convert R to ABSOLUTE I/J/K
- c. Convert INCREMENTAL or ABSOLUTE I/J/K to any user defined LETTER ADDRESS
- d. Convert INCREMENTAL I/J/K to ABSOLUTE
- e. Convert ABSOLUTE I/J/K to INCREMENTAL
- f. Auto break up complete circle commands into ARC QUADRANTS

#### **13. Turning Options :**

- a. Convert X from RADIUS to DIAMETER
- b. Convert X from DIAMETER to RADIUS
- c. Convert INCREMENTAL U/W to ABSOLUTE X/Z
- d. INVERT X / U commands
- e. REVERSE G02/G03 commands
- f. INVERT I / K commands in G02/G03

#### **14. Formatting Options :**

- a. SEPARATE user defined G codes ... place user defined G codes on a separate line
- b. SEPARATE user defined M codes ... place user defined M codes on a separate line
- c. SEPARATE X/Y/Z commands into X/Y then Z

15. **REVERSE CALCULATE TAPPING FEEDRATES** using a user define G trigger code from IPM to PITCH or PITCH to IPM.
16. **RE-ORDER LETTER ADDRESSES** per user defined sequence ... for example convert T03M06 to M06T03
17. **AUTO ADD USER DEFINED HEIGHT OFFSET COMMAND** to Z axis move after tool change. Convert machine code that doesn't use a height offset command like G43 and have it auto-inserted on the first Z move after a tool change.

**18. Decimal Conversion Options :**

- a. REMOVE DECIMAL POINT from user defined letter addresses
- b. ADD DECIMAL POINTS to user defined letter addresses
- c. Tons of decimal point formatting options including 3 place decimal, 4 place decimals, add decimals to user defined places from the right or left, remove decimal and convert value to user defined place value.

**19. AUTO-OFFSET Feature :**

- a. ADD or SUBTRACT user defined value to all X/Y/Z/R current values.
- b. Perform for entire program or use START and STOP commands to define up to (10) sections of the program. For example, add 3.00" to all Z commands between T01 and M01.
- c. Convert X/Y/Z commands and matching I/J/K values if desired.

**20. INSERTION OPTION Feature :**

- a. User can define an UNLIMITED number of LINE insertion options where a user defined code triggers the software to auto-insert a user defined line of information prior to or after the line where the trigger code is found.
- b. User can define an UNLIMITED number of FILE insertion options where a user defined code triggers the insertion of the complete contents of a user defined file prior to or after the line where the trigger code is found.
- c. Auto insert a user defined character at the end of every line.
- d. Auto insert a user defined character or line of information on the line where a user defined trigger code is found.

**21. T CODE LOOK AHEAD Feature :**

- a. KipwareXC will automatically scan the program for T codes and using a user defined placement will auto-insert the NEXT TOOL T COMMAND into a tool's sequence. User defined placements include :
  - i. Before the M06 command on the same line
  - ii. At the end of the line where the M06 command is found
  - iii. On the first X/Y movement line after the M06 command

22. KipwareXC® includes **TONS of PRE-DEFINED PROFILES** ... just turn them ON or OFF with a checkbox. PRE-DEFINED Profiles include :
- a. Turning - Convert FANUC single line canned cycle commands to double line commands
  - b. Turning - Convert FANUC double line canned cycle commands to single line commands
  - c. Turning - Convert FANUC single line canned cycle commands to OKUMA OSP canned cycle commands
  - d. Turning - Convert FANUC double line canned cycle commands to OKUMA OSP canned cycle commands
  - e. Milling - Convert FADAL formatted G code programs for FANUC format
  - f. Milling - Convert FANUC formatted G code to Bridgeport DX-32
  - g. Milling - Convert FANUC formatted G code to Dynapath format
  - h. Milling - Convert FANUC formatted G code to ACRAMATIC A950 format
  - i. Fanuc Macro Formatter to auto-check macro programs for the correct formatting including bracket check and other format checks.
  - j. Custom Specific Replacement where user defined letter addresses can be replaced.
  - k. Milling - Drilling Cycle Re-Formatting to move or add X/Y positioning to the canned cycle command line or directly after the canned cycle command line.

**23. KipwareXC® EDITOR Features :**

- i. KipwareXC® includes a full EDITOR with many CNC and G code editing features such as :
  1. Re-Sequence G code programs
  2. REMOVE line numbers
  3. REMOVE or ADD SPACES for readability
  4. COLOR HIGHLIGHTING using a user defined color pallet
  5. REMOVE blank lines
  6. OVERRIDE feature that allows users to add or subtract values for user defined letter addresses. Values can be a direct numeric value or a percentage value. For example ... reduce all F commands by 50% or add 1.00" to all Z commands.
  7. SEPARATE G and / or M codes using a user defined parameters.
  8. Convert G code from INCH to METRIC ... and reverse.

**EVEN MORE IMPORTANTLY** ... since Kentech Inc. is the creator and developer of KipwareXC® ... we can create any user required customization to KipwareXC® to handle any user specific requests. Being able to use the base KipwareXC® source code

and to build upon that as desired allows us the ability to customize at a lower development rate and keep all the powerful standard KipwareXC® features in place ... just ADD the new features you require. Contact [Sales@KentechInc.com](mailto:Sales@KentechInc.com) to start the discussion of your custom G code conversion requirements.

