WINMAX LATHE
NC PROGRAMMING

Dual-screen and Max Consoles for Hurco Turning Centers
# Table of Contents

WinMax Lathe NC Programming

Documentation Conventions .................................................. xix

Programming and Operation Information ................................. xxii

Overview .................................................................................. 1 -  1
  NC Part Program Components .................................................. 1 -  2
  Program Start ......................................................................... 1 -  2
  Sequence Number .................................................................... 1 -  3
  Address Characters .................................................................. 1 -  3
  Special Characters .................................................................... 1 -  5
  Words ....................................................................................... 1 -  5
  Block ....................................................................................... 1 -  5
  Block Skip Code ........................................................................ 1 -  6

Off-line Part Program Formats .................................................. 1 -  7
  Block Code Processing ............................................................... 1 -  7
  Block Data Formats .................................................................. 1 -  9

Default M and G Codes ............................................................. 1 - 10

Navigation .................................................................................. 1 - 10

NC Editor .................................................................................... 1 - 11

Starting a New NC Program ...................................................... 1 - 13
  NC Programming Rules .............................................................. 1 - 13

Program Editing Features .......................................................... 1 - 15
  Program Review ........................................................................ 1 - 16

NC Editor Menus ......................................................................... 1 - 17
  Basic Programming Menu .......................................................... 1 - 17
  Jump and Search Functions Menu .............................................. 1 - 19
    Search Submenu ...................................................................... 1 - 20
  Edit Functions Menu ................................................................. 1 - 21
  Renumbering and Tagging Menu ................................................. 1 - 23
    Block Renumbering Submenu .................................................. 1 - 24
    NC Tag List ............................................................................ 1 - 25
  Program Execution Menu .......................................................... 1 - 26
  NC Editor Settings Menu ........................................................... 1 - 27
    NC Editor Settings .................................................................. 1 - 28

Importing and Exporting M Code Data ........................................ 1 - 29

Basic NC G Codes ........................................................................ 2 - 1
  Overview .................................................................................. 2 - 3
  G Code Groups ......................................................................... 2 - 3
  G Code Table ............................................................................ 2 - 4

G00 - Linear Motion at Rapid (default) ........................................ 2 -  6
G01 - Linear Motion at Feed ....................................................... 2 -  6
G02 - Clockwise Circular Motion at Feed ...................................... 2 -  7
G03 - Counterclockwise Circular Motion at Feed ......................... 2 -  8
  Example—Program Arc with Endpoint and Center Coordinates .... 2 -  8
  Example—Program Arc with Endpoint and Radius .................... 2 -  9
G04 - Dwell .................................................................................. 2 - 10
G06 - Probe/Block Skip ............................................................... 2 - 10
G07 - Radius Programming ................................................. 2 - 11
   Programming Display Mode .................................... 2 - 12
G08 - Diameter (default) Programming ................................. 2 - 13
   Programming Display Mode .................................... 2 - 14
G09 - Exact Stop ....................................................... 2 - 15
G20 - Inch Mode (default) ........................................... 2 - 15
   Unit Display Mode .............................................. 2 - 15
G21 - Millimeter Mode ................................................ 2 - 16
   Unit Display Mode .............................................. 2 - 16
G33 - Threading ......................................................... 2 - 17
G40 - Cutter Radius Compensation Off (default) ................. 2 - 18
G41 - Cutter Radius Compensation Left ............................. 2 - 19
G42 - Cutter Radius Compensation Right ............................ 2 - 19
   Cutter Compensation Turned Off (G40) ...................... 2 - 21
   Cutter Compensation Right Turned On (G42) ............... 2 - 21
G53 - Program Machine Coordinates ................................ 2 - 22
G59 - Cancel Work Coordinate Offsets (default) .................. 2 - 22
G73 - Peck Drill with Chip Break Drill Cycle ..................... 2 - 23
G74 - Left Hand Tapping ............................................. 2 - 24
G78 - Threading Cycle ................................................ 2 - 25
   Turn and Face Threading ....................................... 2 - 28
   Turn Threading - Inner Diameter and Outer Diameter .... 2 - 29
   Clearance, Lead In and Lead Out Parameters ............... 2 - 29
   Multiple Start Parameters ..................................... 2 - 30
   Tapered Thread .................................................. 2 - 31
   Cutting Parameters .............................................. 2 - 31
   Sample Thread Part Programs .................................. 2 - 33
      Straight OD Thread ......................................... 2 - 33
      Straight OD Thread with Lead In/Lead Out Angles .... 2 - 34
      Tapered OD Thread ........................................ 2 - 35
      Straight ID Thread ....................................... 2 - 36
G80 - Cancel Drill Cycle (default) ................................ 2 - 37
G81 - Drill Cycle ..................................................... 2 - 37
G82 - Drill Cycle with Dwell ........................................ 2 - 38
G83 - Peck Drill Cycle ............................................... 2 - 39
G84 - Right Hand Tapping ............................................ 2 - 40
G90 - Absolute Programming (default) ............................. 2 - 41
G91 - Incremental Programming ..................................... 2 - 43
G92 - Work Coordinate Offsets or Spindle Max Speed ............ 2 - 45
G94 - Feed per Minute (default) .................................. 2 - 45
G95 - Feed per Revolution .......................................... 2 - 45
G96 - Constant Surface Speed (CSS) ................................. 2 - 46
G97 - Direct Spindle Speed (default) ............................... 2 - 46
G98 - Drill Cycle Initial Level Return (default) ................. 2 - 47
G99 - Drill Cycle R Plane Return .................................. 2 - 48
G101 - Axial Surface of the Part -
   Main Spindle without Linear Y ............................... 2 - 49
G102 - Radial Surface of the Part -
   Main Spindle without Linear Y ................................ 2 - 50
G103 - Axial Surface of the Part -
   Main Spindle with Linear Y ................................... 2 - 52
G104 - Radial Surface of the Part -
   Main Spindle with Linear Y ................................... 2 - 53
G105 - Axial Surface of the Part -
   Sub-spindle without Linear Y ............................... 2 - 54
<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>G106</td>
<td>Radial Surface of the Part – Sub-spindle without Linear Y</td>
<td>2 - 55</td>
</tr>
<tr>
<td>G107</td>
<td>Axial Surface of the Part – Sub-spindle with Linear Y</td>
<td>2 - 57</td>
</tr>
<tr>
<td>G108</td>
<td>Radial Surface of the Part – Sub-spindle with Linear Y</td>
<td>2 - 58</td>
</tr>
<tr>
<td>G109</td>
<td>Turning / Lathe Mode</td>
<td>2 - 59</td>
</tr>
</tbody>
</table>

**Basic NC M Codes**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>Program Stop</td>
<td>3 - 1</td>
</tr>
<tr>
<td>M01</td>
<td>Optional Stop</td>
<td>3 - 4</td>
</tr>
<tr>
<td>M02</td>
<td>End of Program (no rewind)</td>
<td>3 - 4</td>
</tr>
<tr>
<td>M03</td>
<td>Spindle Clockwise</td>
<td>3 - 4</td>
</tr>
<tr>
<td>M04</td>
<td>Spindle Counterclockwise</td>
<td>3 - 4</td>
</tr>
<tr>
<td>M05</td>
<td>Spindle Off</td>
<td>3 - 4</td>
</tr>
<tr>
<td>M07</td>
<td>Secondary Coolant On</td>
<td>3 - 4</td>
</tr>
<tr>
<td>M08</td>
<td>Primary Coolant On</td>
<td>3 - 4</td>
</tr>
<tr>
<td>M09</td>
<td>Coolant Off</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M10</td>
<td>Set Chucking Pressure to High</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M11</td>
<td>Set Chucking Pressure to Low</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M12</td>
<td>Turret Index Reverse</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M13</td>
<td>Turret Index Forward</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M14</td>
<td>Chuck Open</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M15</td>
<td>Chuck Close</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M16</td>
<td>Chuck Air Blow On</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M17</td>
<td>Chuck Air Blow Off</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M18</td>
<td>Main Chuck Coolant On</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M19</td>
<td>Spindle Orient</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M20</td>
<td>Chuck Open for Bar Feeder Start</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M21</td>
<td>Bar Feeder Finished Close Chuck</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M22</td>
<td>Start Bar Feeder (Bar Load)</td>
<td>3 - 5</td>
</tr>
<tr>
<td>M23</td>
<td>Start Bar Feeder for Z Guided Feed</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M24</td>
<td>Part Conveyor On</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M25</td>
<td>Part Conveyor Off</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M28</td>
<td>Tailstock Quill Advance</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M29</td>
<td>Tailstock Quill Retract</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M30</td>
<td>End of Program (rewind)</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M31</td>
<td>Increment Cycle Counter</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M32</td>
<td>Increment Setup Cycle Counter</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M38</td>
<td>Steady Rest Clamp</td>
<td>3 - 6</td>
</tr>
<tr>
<td>M39</td>
<td>Steady Rest Unclamp</td>
<td>3 - 7</td>
</tr>
<tr>
<td>M41</td>
<td>Spindle Gear 1 / Low Gear Range</td>
<td>3 - 7</td>
</tr>
<tr>
<td>M42</td>
<td>Spindle Gear 2 / High Gear Range</td>
<td>3 - 7</td>
</tr>
<tr>
<td>M48</td>
<td>Use Feedrate Override</td>
<td>3 - 7</td>
</tr>
<tr>
<td>M49</td>
<td>Ignore Feedrate Override</td>
<td>3 - 7</td>
</tr>
<tr>
<td>M50</td>
<td>Conveyor On</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M51</td>
<td>Conveyor Off</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M52</td>
<td>Auxiliary Output 1 On</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M53</td>
<td>Auxiliary Output 2 On</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M54</td>
<td>Auxiliary Output 3 On</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M55</td>
<td>Auxiliary Output 4 On</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M57</td>
<td>Use Part Catcher with Bar End Eject</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M58</td>
<td>Part Catcher Advance</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M59</td>
<td>Part Catcher Retract</td>
<td>3 - 8</td>
</tr>
<tr>
<td>M60</td>
<td>Select External Chucking</td>
<td>3 - 8</td>
</tr>
</tbody>
</table>
M61 - Select Internal Chucking ........................................ 3 - 8
M62 - Auxiliary Output 1 Off ........................................ 3 - 8
M63 - Auxiliary Output 2 Off ........................................ 3 - 8
M64 - Auxiliary Output 3 Off ........................................ 3 - 8
M65 - Auxiliary Output 4 Off ........................................ 3 - 9
M71 - Tool Setter Retract ............................................ 3 - 9
M72 - Tool Setter Advance ........................................... 3 - 9
M74 - Activate Rigid Tapping ........................................ 3 - 9
M85 - Auto Door Open ................................................ 3 - 9
M86 - Auto Door Close ................................................ 3 - 9
M91 - Single Block Inactive ......................................... 3 - 9
M92 - Single Block Active ............................................ 3 - 9
M94 - Set Program Maximum Rapid Rate ............................ 3 - 9
M114 - Chuck Open - Sub-spindle .................................. 3 - 9
M115 - Chuck Close - Sub-spindle .................................. 3 - 9
M118 - Sub Chuck Coolant On ....................................... 3 - 10
M128 - Z-axis Move to and Hitch to Tailstock ...................... 3 - 10
M129 - Z-axis Unhitch from Tailstock .............................. 3 - 10
M130 - Turning Mode .................................................. 3 - 10
M131 - Milling Mode ................................................... 3 - 10
M132 - Enable “C” Axis Clamp ...................................... 3 - 10
M133 - Enable “C” Axis Hold Assist ................................. 3 - 10
M134 - Release Both “C” Axis Clamps ............................... 3 - 10
M160 - Select External Chucking - Sub-spindle .................... 3 - 10
M161 - Select Internal Chucking - Sub-spindle .................... 3 - 10
M186 - Activate Torque Monitoring for W .......................... 3 - 11
M187 - De-activate Torque Monitoring for W ....................... 3 - 11
M200 - Block Skip Synchronization .................................. 3 - 11
M203 - Sync Spindles Forward Mode ................................. 3 - 11
M204 - Sync Spindles Reverse Mode ................................ 3 - 11
M205 - Clear Spindle Sync ............................................ 3 - 11
M231 - Bypass Chuck Open Interlock - Main Spindle ............ 3 - 11
M232 - Enable “C3” Axis Clamp .................................... 3 - 12
M234 - Release “C3” Axis Clamp .................................... 3 - 12
M241 - Allow Sub-spindle Chuck to Open while Spindle is Running for Part Transfer .......................... 3 - 12

ISNC G Codes .......................................................... 4 - 1
Overview ................................................................. 4 - 3
G Code Groups .......................................................... 4 - 3
G Code Table ............................................................ 4 - 3
G00 - Rapid Traverse (default) ....................................... 4 - 6
G01 - Linear Interpolation ............................................. 4 - 7
G02/G03 - Clockwise/Counterclockwise Arc ....................... 4 - 8
  Program Arc with Endpoint and Center Coordinates .......... 4 - 9
  Program Arc with Endpoint and Radius ........................ 4 - 11
G04 - Dwell ............................................................. 4 - 12
G06 - Probe/Block Skip .............................................. 4 - 12
G07 - Radius Programming ........................................... 4 - 13
  Programming Display Mode ....................................... 4 - 14
G08 - Diameter Programming (default) ............................. 4 - 15
  Programming Display Mode ....................................... 4 - 16
G09 - Exact Stop ...................................................... 4 - 17
<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>G84</td>
<td>Face CW Tapping</td>
<td>75</td>
</tr>
<tr>
<td>G83.1</td>
<td>Chip Breaker Peck Drilling</td>
<td>74</td>
</tr>
<tr>
<td>G83</td>
<td>Face Drilling with Pecks</td>
<td>72</td>
</tr>
<tr>
<td>G82</td>
<td>Drill Cycle with Dwell</td>
<td>70</td>
</tr>
<tr>
<td>G81</td>
<td>Drill Cycle</td>
<td>69</td>
</tr>
<tr>
<td>G80</td>
<td>Canned Cycle Cancel (default)</td>
<td>68</td>
</tr>
<tr>
<td>G79</td>
<td>Stock Removal in Facing Cycle</td>
<td>67</td>
</tr>
<tr>
<td>G78</td>
<td>Multiple Threading Cycle</td>
<td>65</td>
</tr>
<tr>
<td>G77</td>
<td>Outer Diameter/Inner Diameter Cutting Cycle</td>
<td>64</td>
</tr>
<tr>
<td>G76</td>
<td>Multiple Threading Cycle</td>
<td>62</td>
</tr>
<tr>
<td>G75</td>
<td>Outer Diameter/Inner Diameter Drilling</td>
<td>60</td>
</tr>
<tr>
<td>G74</td>
<td>End Face Peck Drilling</td>
<td>58</td>
</tr>
<tr>
<td>G73</td>
<td>Pattern Repeating</td>
<td>56</td>
</tr>
<tr>
<td>G72</td>
<td>Stock Removal in Facing</td>
<td>54</td>
</tr>
<tr>
<td>G71</td>
<td>Stock Removal in Turning</td>
<td>51</td>
</tr>
<tr>
<td>G70</td>
<td>Finishing Cycle</td>
<td>49</td>
</tr>
<tr>
<td>G69</td>
<td>Constant Lead Thread Cutting</td>
<td>47</td>
</tr>
<tr>
<td>G68</td>
<td>Canned Cycle Cancel (default)</td>
<td>46</td>
</tr>
<tr>
<td>G67</td>
<td>Modal Subprogram Call Cancel (default)</td>
<td>44</td>
</tr>
<tr>
<td>G66</td>
<td>Modal Subprogram Call</td>
<td>42</td>
</tr>
<tr>
<td>G65</td>
<td>Subprogram Call</td>
<td>40</td>
</tr>
<tr>
<td>G63</td>
<td>Macro Instruction (G65)</td>
<td>41</td>
</tr>
<tr>
<td>G62</td>
<td>Specifying Subprogram Iterations</td>
<td>40</td>
</tr>
<tr>
<td>G61</td>
<td>Precision Cornering</td>
<td>40</td>
</tr>
<tr>
<td>G60</td>
<td>Tool Offset or Tool Geometry</td>
<td>39</td>
</tr>
<tr>
<td>G59</td>
<td>Work Coordinates (G54.93 - Aux Work Coordinates)</td>
<td>39</td>
</tr>
<tr>
<td>G58</td>
<td>Work Coordinates (G54 default)</td>
<td>37</td>
</tr>
<tr>
<td>G57</td>
<td>Machine Coordinates</td>
<td>36</td>
</tr>
<tr>
<td>G56</td>
<td>Data Setting Cancel (default)</td>
<td>35</td>
</tr>
<tr>
<td>G55</td>
<td>Data Setting (Work Offsets, Tool Offsets, or Tool Geometry)</td>
<td>17</td>
</tr>
<tr>
<td>G54-G59</td>
<td>Work Coordinates (G54 default)</td>
<td>37</td>
</tr>
<tr>
<td>G53</td>
<td>Machine Coordinates</td>
<td>36</td>
</tr>
<tr>
<td>G52</td>
<td>Aux Work Coordinates</td>
<td>35</td>
</tr>
<tr>
<td>G51</td>
<td>Inch Mode (default)</td>
<td>34</td>
</tr>
<tr>
<td>G50</td>
<td>Unit Display Mode</td>
<td>33</td>
</tr>
<tr>
<td>G49</td>
<td>YZ Plane</td>
<td>32</td>
</tr>
<tr>
<td>G48</td>
<td>XZ Plane</td>
<td>31</td>
</tr>
<tr>
<td>G47</td>
<td>XY Plane</td>
<td>30</td>
</tr>
<tr>
<td>G46</td>
<td>Unit Display Mode</td>
<td>32</td>
</tr>
<tr>
<td>G45</td>
<td>Millimeter Mode</td>
<td>31</td>
</tr>
<tr>
<td>G44</td>
<td>G28 - Automatic Return to Reference Point</td>
<td>30</td>
</tr>
<tr>
<td>G43</td>
<td>Constant Lead Thread Cutting</td>
<td>29</td>
</tr>
<tr>
<td>G42</td>
<td>Threading</td>
<td>29</td>
</tr>
<tr>
<td>G41</td>
<td>Cutter Radius Compensation Left</td>
<td>28</td>
</tr>
<tr>
<td>G40</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>27</td>
</tr>
<tr>
<td>G39</td>
<td>Cutter Compensation (G40–G42)</td>
<td>26</td>
</tr>
<tr>
<td>G38</td>
<td>Cutter Compensation Off (default)</td>
<td>25</td>
</tr>
<tr>
<td>G37</td>
<td>Cutter Compensation Left</td>
<td>24</td>
</tr>
<tr>
<td>G36</td>
<td>Cutter Compensation Right</td>
<td>23</td>
</tr>
<tr>
<td>G35</td>
<td>Cutter Compensation Off (default)</td>
<td>22</td>
</tr>
<tr>
<td>G34</td>
<td>Cutter Compensation Off (default)</td>
<td>21</td>
</tr>
<tr>
<td>G33</td>
<td>Cutter Compensation Off (default)</td>
<td>20</td>
</tr>
<tr>
<td>G32</td>
<td>Cutter Radius Compensation Right</td>
<td>19</td>
</tr>
<tr>
<td>G31</td>
<td>Cutter Radius Compensation Right</td>
<td>18</td>
</tr>
<tr>
<td>G30</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>17</td>
</tr>
<tr>
<td>G29</td>
<td>Cutter Radius Compensation Left</td>
<td>16</td>
</tr>
<tr>
<td>G28</td>
<td>Cutter Radius Compensation Right</td>
<td>15</td>
</tr>
<tr>
<td>G27</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>14</td>
</tr>
<tr>
<td>G26</td>
<td>Cutter Radius Compensation Right</td>
<td>13</td>
</tr>
<tr>
<td>G25</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>12</td>
</tr>
<tr>
<td>G24</td>
<td>Cutter Radius Compensation Right</td>
<td>11</td>
</tr>
<tr>
<td>G23</td>
<td>Cutter Radius Compensation Right</td>
<td>10</td>
</tr>
<tr>
<td>G22</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>9</td>
</tr>
<tr>
<td>G21</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>8</td>
</tr>
<tr>
<td>G20</td>
<td>Cutter Radius Compensation Right</td>
<td>7</td>
</tr>
<tr>
<td>G19</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>6</td>
</tr>
<tr>
<td>G18</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>5</td>
</tr>
<tr>
<td>G17</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>4</td>
</tr>
<tr>
<td>G16</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>3</td>
</tr>
<tr>
<td>G15</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>2</td>
</tr>
<tr>
<td>G14</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>1</td>
</tr>
<tr>
<td>G13</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>0</td>
</tr>
<tr>
<td>G12</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-1</td>
</tr>
<tr>
<td>G11</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-2</td>
</tr>
<tr>
<td>G10</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-3</td>
</tr>
<tr>
<td>G9</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-4</td>
</tr>
<tr>
<td>G8</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-5</td>
</tr>
<tr>
<td>G7</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-6</td>
</tr>
<tr>
<td>G6</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-7</td>
</tr>
<tr>
<td>G5</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-8</td>
</tr>
<tr>
<td>G4</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-9</td>
</tr>
<tr>
<td>G3</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-10</td>
</tr>
<tr>
<td>G2</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-11</td>
</tr>
<tr>
<td>G1</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-12</td>
</tr>
<tr>
<td>G0</td>
<td>Cutter Radius Compensation Off (default)</td>
<td>-13</td>
</tr>
</tbody>
</table>

**Table of Contents**

- G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry)  17
- Work Offsets  17
- Tool Offset or Tool Geometry  18
- G11 - Data Setting Cancel (default)  18
- G17 - XY Plane  19
- G18 - XZ Plane (default)  21
- G19 - YZ Plane  22
- G20 - Inch Mode (default)  23
- Unit Display Mode  23
- G21 - Millimeter Mode  24
- Unit Display Mode  24
- G28 - Automatic Return to Reference Point  25
- G32 - Constant Lead Thread Cutting  27
- G33 - Threading  29
- Cutter Compensation (G40–G42)  31
- G40 - Cutter Radius Compensation Off (default)  32
- G41 - Cutter Radius Compensation Left  32
- G42 - Cutter Radius Compensation Right  33
- G50 - Set Part Setup or Set Max RPM  35
- Set Part Setup  35
- Set Max RPM  35
- G53 - Machine Coordinates  36
- G54-G59 - Work Coordinates (G54 default)  37
- G54.1 - G54.93 - Aux Work Coordinates  39
- G61 - Precision Cornering  40
- G64 - Precision Cornering Cancel (default)  40
- G65 - Subprogram Call  40
- Arguments  40
- Multiple Arguments  41
- Passing Argument Lists to Subprograms in Macro Mode B  41
- Layering of Local Variables within Subprogram Calls  42
- Specifying Subprogram Iterations  42
- Using G65 and G66  42
- Using M98  42
- Macro Instruction (G65)  44
- G66 - Modal Subprogram Call  47
- G67 - Modal Subprogram Call Cancel (default)  49
- G70 - Finishing Cycle  49
- G71 - Stock Removal in Turning  51
- Sign of U and W  51
- G72 - Stock Removal in Facing  54
- G73 - Pattern Repeating  56
- G74 - End Face Peck Drilling  58
- G75 - Outer Diameter/Inner Diameter Drilling  60
- G76 - Multiple Threading Cycle  62
- G77 - Outer Diameter/Inner Diameter Cutting Cycle  64
- Sign of U and R  64
- G78 - Threading Cycle  66
- G79 - Stock Removal in Facing Cycle  67
- G80 - Canned Cycle Cancel (default)  68
- G81 - Drill Cycle  69
- G82 - Drill Cycle with Dwell  70
- G83 - Face Drilling with Pecks  72
- G83.1 - Chip Breaker Peck Drilling  74
- G84 - Face CW Tapping  75
| G84.2 (or G84 M29) - Rigid CW Tapping | 4 - 77 |
| G85 - Boring Cycle | 4 - 78 |
| G86 - Bore Rapid Out Cycle | 4 - 80 |
| G87 - Side Drilling with Pecks/Dwell | 4 - 82 |
| G88 - Side CW Tapping | 4 - 83 |
| G89 - Side Boring Cycle | 4 - 84 |
| G90 - Absolute Programming (default) | 4 - 85 |
| G91 - Incremental Programming | 4 - 87 |
| G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle | 4 - 89 |
| Work Coordinate Offsets | 4 - 89 |
| Spindle Max Speed | 4 - 89 |
| Multiple Thread Cutting Cycle | 4 - 90 |
| G93 - Inverse Time Feedrate | 4 - 91 |
| G94 - Feed per Minute (default) | 4 - 92 |
| G95 - Feed per Revolution | 4 - 92 |
| G96 - Constant Surface Speed (CSS) | 4 - 93 |
| G97 - Direct Spindle Speed (default) | 4 - 93 |
| G98 - Return to Initial Level (default) | 4 - 94 |
| G99 - Return to R Point Level | 4 - 95 |
| G101 - Axial Surface of the Part - Main Spindle without Linear Y | 4 - 96 |
| G102 - Radial Surface of the Part - Main Spindle without Linear Y | 4 - 97 |
| G103 - Axial Surface of the Part - Main Spindle with Linear Y | 4 - 99 |
| G104 - Radial Surface of the Part - Main Spindle with Linear Y | 4 - 100 |
| G105 - Axial Surface of the Part - Sub-spindle without Linear Y | 4 - 101 |
| G106 - Radial Surface of the Part - Sub-spindle without Linear Y | 4 - 102 |
| G107 - Axial Surface of the Part - Sub-spindle with Linear Y | 4 - 104 |
| G108 - Radial Surface of the Part - Sub-spindle with Linear Y | 4 - 105 |
| G109 - Turning / Lathe Mode | 4 - 106 |

**ISNC M Codes**

5 - 1

**ISNC M Code Overview**

5 - 2

**M Code Translation**

5 - 2

**Setting up M Codes**

5 - 2

**ISNC M Code Definitions**

5 - 5

- ISNC M Code Translation List - Page 1
  5 - 5

- ISNC M Code Translation List - Page 2
  5 - 6

- ISNC M Code Translation List - Page 3
  5 - 7

- ISNC M Code Translation List - Page 4
  5 - 8

**ISNC M Code Translation List - Page 1**

5 - 9

- Program Stop (ISNC M0)
  5 - 9

- Optional Stop (ISNC M1)
  5 - 9

- Main Spindle CW (ISNC M3)
  5 - 9

- Main Spindle CCW (ISNC M4)
  5 - 9

- Main Spindle Stop (ISNC M5)
  5 - 9

- Live Tool CW (ISNC M33)
  5 - 10

- Live Tool CCW (ISNC M34)
  5 - 10
Live Tool Stop (ISNC M35) .................................................. 5 - 10
Sub-spindle CW (ISNC M103) ............................................. 5 - 11
Sub-spindle CCW (ISNC M104) ............................................ 5 - 11
Sub-spindle Stop (ISNC M105) ............................................ 5 - 11
Secondary Coolant (ISNC M7) ........................... 5 - 11
Primary Coolant On (ISNC M8) ............................... 5 - 11
Coolant Off (ISNC M9) ..................................................... 5 - 11
Chuck Pressure High (ISNC M67) ....................... 5 - 11
Chuck Pressure Low (ISNC M66) .............................. 5 - 12
Turret Index Reverse (ISNC M12) .......................... 5 - 12
Turret Index Forward (ISNC M13) ......................... 5 - 12
Chuck Open (ISNC M69) ................................................. 5 - 12
Chuck Close (ISNC M68) ............................................... 5 - 12
Main Spindle Orient (ISNC M19) .............................. 5 - 12
Chuck Open for Bar (ISNC M20) ......................... 5 - 12
Chuck Close for Bar (ISNC M21) ............................ 5 - 12
Start Bar Feeder (ISNC M22) ........................................... 5 - 12
Start Bar with Z (ISNC M23) ........................................... 5 - 13
Part Conveyor On (ISNC M110) ............................ 5 - 13
Part Conveyor Off (ISNC M111) ............................ 5 - 13
Tailstock Quill Adv (ISNC M78) ......................... 5 - 13
Tailstock Quill Retr (ISNC M79) ............................. 5 - 13
End of Program (ISNC M30) ............................................. 5 - 13
Incr Cycle Count (ISNC M31) ................................. 5 - 13
Incr Setup Cycle (ISNC M32) ................................. 5 - 13
Feedrate Override (ISNC M48) .............................. 5 - 13
Ignore Feed Override (ISNC M49) ...................... 5 - 13
Conveyor On (ISNC M24) .............................................. 5 - 13
Conveyor Off (ISNC M25) .............................................. 5 - 14
Part Catcher w/Eject (ISNC M57) ......................... 5 - 14
Part Catcher Advance (ISNC M58) ....................... 5 - 14
Part Catcher Retract (ISNC M59) ........................... 5 - 14
Sel External Chuck (ISNC M60) .......................... 5 - 14
Sel Internal Chuck (ISNC M61) ............................ 5 - 14
Tool Setter Retract (ISNC M71) ............................. 5 - 15
Tool Setter Adv (ISNC M72) ...................................... 5 - 15
Auto Door Open (ISNC M85) ..................................... 5 - 15
Auto Door Close (ISNC M86) ..................................... 5 - 15
Single Block Off (ISNC M91) ..................................... 5 - 15
Single Block On (ISNC M92) ..................................... 5 - 15
Sub-spindle Chuck Open (ISNC M168) ............... 5 - 15
Sub-spindle Chuck Close (ISNC M169) ............... 5 - 15
Turning Mode (ISNC M130) ......................................... 5 - 15
Milling Mode (ISNC M131) .......................................... 5 - 15
C Axis Clamp (ISNC M41) ............................................. 5 - 15
C Axis Hold (ISNC M133) ............................................. 5 - 15
C Axis Unclamp (ISNC M40) ..................................... 5 - 15
Sub-spndl Ext Chuck (ISNC M160) ..................... 5 - 16
Sub-spndl Int Chuck (ISNC M161) ....................... 5 - 16
W Torque Mon On (ISNC M186) ......................... 5 - 16
W Torque Mon Off (ISNC M187) ............................ 5 - 16
Block Skip Sync (ISNC M200) .............................. 5 - 16
Sync Spindles Forward (ISNC M203) ................. 5 - 16
Sync Spindles Reverse (ISNC M204) ........................................ 5 - 16
Clear Spindle Sync (ISNC M205) ........................................ 5 - 17
Bypass Chuck Intlk (ISNC M231) ........................................ 5 - 17
C3 Axis Clamp (ISNC M232) ........................................ 5 - 17
C3 Axis Unclamp (ISNC M234) ........................................ 5 - 17
Spindle CW/Coolant On (ISNC M63) .................................... 5 - 17
Spindle CCW/Coolant Off (ISNC M64) .................................. 5 - 17
Spindle Off/Coolant Off (ISNC M65) ..................................... 5 - 18
Sub-Spdl CW/CLNT On (ISNC M163) ................................. 5 - 18
Sub-Spdl CCW/CLNT Off (ISNC M164) ............................... 5 - 18
Sub-Spdl Off/CLNT Off (ISNC M165) ................................. 5 - 18
Rigid Tap (ISNC M29) .................................................. 5 - 19
ISNC M Code Translation List - Page 4 ............................. 5 - 19
  Auxiliary Output 1 On (ISNC M52) .................................. 5 - 19
  Auxiliary Output 2 On (ISNC M53) .................................. 5 - 19
  Auxiliary Output 3 On (ISNC M54) .................................. 5 - 19
  Auxiliary Output 4 On (ISNC M55) .................................. 5 - 19
  Auxiliary Output 1 Off (ISNC M152) ............................... 5 - 19
  Auxiliary Output 2 Off (ISNC M153) ............................... 5 - 19
  Auxiliary Output 3 Off (ISNC M154) ............................... 5 - 19
  Auxiliary Output 4 Off (ISNC M155) ............................... 5 - 19
  Chuck Air On (ISNC M16) ........................................... 5 - 19
  Chuck Air Off (ISNC M17) .......................................... 5 - 19
  Set Max Rapid (ISNC M94) ........................................... 5 - 20
  Part Transfer Sub-spindle (ISNC 241) ............................ 5 - 20
  End of Program (no rewind) (ISNC M2) ............................ 5 - 20
  Steady Rest Clamp (ISNC M38) ..................................... 5 - 20
  Steady Rest Unclamp (ISNC M39) ................................... 5 - 21
  Spindle Gear 1 (ISNC M41) ......................................... 5 - 21
  Spindle Gear 2 (ISNC M42) ......................................... 5 - 21
  Z-axis Tailstock Hitch (ISNC M128) ............................... 5 - 21
  Z-axis Tailstock Unhitch (ISNC M129) ......................... 5 - 21

E Codes ................................................................. 6 - 1

T Codes ................................................................. 7 - 1
  Tool Change Sequence ................................................ 7 - 1
  Tool Offsets ......................................................... 7 - 2

Record of Changes .................................................. 1

Index ................................................................. IX - 1
LIST OF FIGURES

Figure 1–1. NC Editor Screen .................................................. 1 - 2
Figure 1–2. Typical NC Block .................................................. 1 - 6
Figure 1–3. NC Editor Status Bar .......................................... 1 - 12
Figure 1–4. NC Program Review screen ................................. 1 - 16
Figure 1–5. ISNC Basic Programming Menu ......................... 1 - 17
Figure 1–6. ISNC Jump and Search Functions Menu .............. 1 - 19
Figure 1–7. ISNC Search Submenu ....................................... 1 - 20
Figure 1–8. ISNC Edit Functions Menu ................................. 1 - 21
Figure 1–9. ISNC Renumbering and Tagging Menu ................. 1 - 23
Figure 1–10.ISNC Block Renumbering Submenu ..................... 1 - 24
Figure 1–11.ISNC NC Tag List Screen .................................. 1 - 25
Figure 1–12.ISNC Program Execution Menu ......................... 1 - 26
Figure 1–13.NC Editor Settings Menu .................................. 1 - 27
Figure 1–14.ISNC NC Editor Settings screen ......................... 1 - 28
Figure 1–15.Import and Export Functions screen .................... 1 - 29
Figure 1–16.Save NC State to File screen .............................. 1 - 30
Figure 1–17.Save NC State to File screen with new folder and NC State file ... 1 - 31
Figure 1–18.Import NC State screen ..................................... 1 - 31
Figure 2–1. G02 (Clockwise) circular motion at feed ................ 2 - 7
Figure 2–2. Arc with Endpoint and Center Coordinates ............ 2 - 8
Figure 2–3. Arc with Endpoint and Radius .............................. 2 - 9
Figure 2–4. G07 Radius Programming .................................... 2 - 11
Figure 2–5. WinMax Status bar ............................................. 2 - 12
Figure 2–6. G08 Diameter Programming .................................. 2 - 13
Figure 2–7. WinMax Status bar ............................................. 2 - 14
Figure 2–8. Cutter Compensation Right Turned Off (G40) ......... 2 - 21
Figure 2–9. Cutter Compensation Turned On (G42) ................. 2 - 21
Figure 2–10. Tool Motion for G73 Chip Break Drill Cycle ......... 2 - 23
Figure 2–11. G74 Left Hand Tapping ...................................... 2 - 24
Figure 2–12. Thread Parameters ............................................ 2 - 28
Figure 2–13. OD Thread with Offsets U and W Example ......... 2 - 29
Figure 2–14. OD Thread with Chase in (A) and Chase out (C) Example .... 2 - 29
Figure 2–15. OD with multiple starts at 120°, 170°, and 180° Example ........ 2 - 30
Figure 2–16. OD with 3 Evenly Spaced Threads Example ........ 2 - 30
Figure 2–17. Tapered Thread Example ................................... 2 - 31
Figure 2–18. No Programmed Finish Pass Example ................ 2 - 31
Figure 2–19. Three Programmed Finished Passes Example ....... 2 - 32
Figure 2–20. Straight OD Thread Example ............................. 2 - 33
Figure 2–21. Straight OD Thread with Lead In/Lead Out Angles ... 2 - 34
Figure 2–22. Tapered OD Thread Example ............................. 2 - 35
Figure 2–23. Straight ID Thread Example ............................... 2 - 36
Figure 2–24. G81 Drill Cycle .................................................. 2 - 37
Figure 2–25. G82 Drill Cycle with Dwell ................................. 2 - 38
Figure 2–26. G83 Peck Drill Cycle ......................................... 2 - 39
Figure 2–27. G84 Right Hand Tapping .................................... 2 - 40
Figure 2–28. Absolute Dimensions ....................................... 2 - 41
Figure 2–29. Incremental Dimensions .................................... 2 - 43
Figure 2–30. G98 Drill Cycle Initial Level Return (default) ....... 2 - 47
Figure 2–31. General Drill Cycle Tool Motion ......................... 2 - 48
Figure 2–32. G101 TMX Axis Coordinates .................................................. 2-49
Figure 2–33. G102 TMX Axis Coordinates .................................................. 2-50
Figure 2–34. Y Axis Coordinate System Detail for G102 ............................... 2-51
Figure 2–35. G103 TMX Axis Coordinates .................................................. 2-52
Figure 2–36. G104 TMX Axis Coordinates .................................................. 2-53
Figure 2–37. G105 TMX Axis Coordinates .................................................. 2-54
Figure 2–38. G106 TMX Axis Coordinates .................................................. 2-55
Figure 2–39. Y Axis Coordinate System Detail for G106 ............................... 2-56
Figure 2–40. G107 TMX Axis Coordinates .................................................. 2-57
Figure 2–41. G108 TMX Axis Coordinates .................................................. 2-58
Figure 2–42. G109 Turning/Lathe Mode (Main Spindle Axis Map) ................. 2-59
Figure 2–43. G109 Turning/Live-tool Mode (Main Spindle Axis Map) .......... 2-60
Figure 2–44. G109 Turning/Lathe Mode (Sub-spindle Axis Map) ................. 2-60
Figure 2–45. G109 Turning/Live-tool Mode (Sub-spindle Axis Map) .......... 2-60
Figure 4–1. G02 (Clockwise) circular motion at feed .................................. 4 - 9
Figure 4–2. Arc with Endpoint and Center Coordinates ............................... 4 - 9
Figure 4–3. Arc with Endpoint and Center Coordinates ............................... 4-10
Figure 4–4. Arc with Endpoint and Radius ................................................. 4-11
Figure 4–5. G07 Radius Programming ....................................................... 4-13
Figure 4–6. WinMax Status bar ................................................................. 4-14
Figure 4–7. G08 Diameter Programming .................................................... 4-15
Figure 4–8. WinMax Status bar ................................................................. 4-16
Figure 4–9. G17, G18, and G19 Planes ....................................................... 4-19
Figure 4–10. XY Plane Selection .............................................................. 4-20
Figure 4–11. G18 XZ Plane Selection ........................................................ 4-21
Figure 4–12. G19 YZ Plane Selection ........................................................ 4-22
Figure 4–13. G32 Constant Lead Thread Cutting ....................................... 4-28
Figure 4–14. G40 Cutter Compensation Off ............................................... 4-31
Figure 4–15. G41 Cutter Compensation Left .............................................. 4-31
Figure 4–16. G42 Cutter Compensation Right ........................................... 4-31
Figure 4–17. Cutter Compensation Off (G40) and Right (G42) .................... 4-33
Figure 4–18. G54-G59 Work Offset G Codes for Multiple Parts ................. 4-38
Figure 4–19. G70 Finish Cycle ................................................................. 4-50
Figure 4–20. Sign of U and W determines stock removal ......................... 4-51
Figure 4–21. G71 Stock Removal in Turning ............................................. 4-53
Figure 4–22. G72 Stock Removal in Facing ............................................... 4-55
Figure 4–23. G73 Pattern Repeating ........................................................ 4-57
Figure 4–24. G74 End Face Peck Drilling .................................................. 4-59
Figure 4–25. G75 Outer Diameter/Inner Diameter Drilling ......................... 4-61
Figure 4–26. G76 - Multiple Threading Cycle .......................................... 4-63
Figure 4–27. Sign of U and R determines the angle of the taper for canned cycles 4-64
Figure 4–28. G77 External Diameter/Internal Diameter Cutting Cycle ......... 4-65
Figure 4–29. G81 Drill Cycle ................................................................. 4-70
Figure 4–30. G82 Drill Cycle with Dwell .................................................... 4-71
Figure 4–31. G83 Peck Drill Cycle ............................................................ 4-73
Figure 4–32. G84 Face CW Tapping ........................................................ 4-76
Figure 4–33. G85 Tool Movement for the Boring Cycle ............................... 4-79
Figure 4–34. G86 Tool Movement for the Bore Rapid Out Cycle ................. 4-81
Figure 4–35. G90 Absolute Dimensions (Mode B) ..................................... 4-85
Figure 4–36. G91 Incremental Dimensions (Mode B) ................................. 4-87
Figure 4–37. G92 Multiple Thread Cutting—Tapered Thread ..................... 4-90
Figure 4–38. G98 Drill Cycle Initial Level Return (default) ......................... 4-94
Figure 4–39. G99 General Drill Cycle Tool Motion ................................... 4-95
Figure 4–40. G101 TMX Axis Coordinates ............................................... 4-96
Figure 4–41. G102 TMX Axis Coordinates ............................................. 4- 97
Figure 4–42. Y Axis Coordinate System Detail for G102 .......................... 4 - 98
Figure 4–43. G103 TMX Axis Coordinates ............................................. 4- 99
Figure 4–44. G104 TMX Axis Coordinates ............................................. 4- 100
Figure 4–45. G105 TMX Axis Coordinates ............................................. 4- 101
Figure 4–46. G106 TMX Axis Coordinates ............................................. 4- 102
Figure 4–47. Y Axis Coordinate System Detail for G106 ......................... 4 - 103
Figure 4–48. G107 TMX Axis Coordinates ............................................. 4- 104
Figure 4–49. G108 TMX Axis Coordinates ............................................. 4- 105
Figure 4–50. G109 Turning/Lathe Mode (Main Spindle Axis Map) ............. 4- 106
Figure 4–51. G109 Turning/Live-tool Mode (Main Spindle Axis Map) ......... 4- 107
Figure 4–52. G109 Turning/Lathe Mode (Sub-Spindle Axis Map) .............. 4- 107
Figure 4–53. G109 Turning/Live-tool Mode (Sub-spindle Axis Map) ......... 4- 107

Figure 5–1. ISNC NC Settings screen .................................................... 5 - 2
Figure 5–2. ISNC M Code Translation List Page 1 screen ........................ 5 - 3
Figure 5–3. ISNC M Code Translation List Page 2 screen ........................ 5 - 3
Figure 5–4. ISNC M Code Translation List Page 3 screen ........................ 5 - 4
Figure 5–5. ISNC M Code Translation List Page 4 screen ........................ 5 - 4

Figure 6–1. Work Offset: Immediate Activation ..................................... 6 - 1

Figure 7–1. Tool Offset Behaviors ......................................................... 7 - 2
LIST OF TABLES

Table 1–1. Address Characters ................................................. 1 - 4
Table 1–2. Order of Block Code Processing .............................. 1 - 8
Table 1–3. Block Data Formats ............................................... 1 - 9

Table 2–1. G Code Groups ...................................................... 2 - 3
Table 2–2. G Codes ............................................................. 2 - 5
Table 2–3. G04 Dwell Examples ............................................ 2 - 10
Table 2–4. Effects of Using G07 ............................................ 2 - 11
Table 2–5. Effects of Using G08 ............................................ 2 - 13
Table 2–6. Inch Mode formats ............................................. 2 - 15
Table 2–7. Millimeter Mode formats ..................................... 2 - 16
Table 2–8. Thread Parameters .............................................. 2 - 27
Table 2–9. Threading Equivalents between Turn and Face Thread Macros ............................................. 2 - 28
Table 2–10. R Values and G96 Programming .......................... 2 - 46

Table 4–1. G Code Groups ...................................................... 4 - 3
Table 4–2. G Codes ............................................................. 4 - 5
Table 4–3. G04 Dwell Examples ............................................ 4 - 12
Table 4–4. Effects of Using G07 ............................................ 4 - 13
Table 4–5. Effects of Using G08 ............................................ 4 - 15
Table 4–6. Inch Mode formats ............................................. 4 - 23
Table 4–7. Millimeter Mode formats ..................................... 4 - 24
Table 4–8. Macro Mode B Local Variables and Subprogram Arguments ............................................. 4 - 41
Table 4–9. H Code Descriptions and Instruction Functions for G65 Macro Instructions ............................................. 4 - 45
Table 4–10. R Values and G96 Programming .......................... 4 - 93
DOCUMENTATION CONVENTIONS

This documentation uses several conventions to explain the safety features and emphasize key concepts. These conventions are described in this section.

Additional information is available on the machine’s Documentation CD.

Sample Screens

Sample screens in this documentation were taken from a WinMax single-screen control. All screens are subject to change. The screens on your system may vary slightly. The sample screen here illustrates softkeys and includes a software version.

Softkeys

Softkeys are located on the side of the screen. You can set the softkeys to appear on either the right or left side of the screen. Refer to the Getting Started with WinMax for information about making this selection. Softkeys may change upon field entries or other softkey selection. References to softkeys in the documentation appear with the softkey’s corresponding F-key. For example, the Part Setup softkey from the Input screen above is referenced as the PART SETUP F1 softkey.

Screen Areas

The screens are divided into the following areas, in addition to the row of softkeys:
Data Entry

The data entry area is located on the opposite side of the screen from the softkeys. Available softkeys may change even when the text and data entry area does not.

Fields in the data entry area display or receive information. Refer to Using the Touchscreen for information on entering information in fields.

Prompts and Error/Status Area

The bottom portion of the screen is reserved for prompts, program status and error messages.

Prompts provide help on data entry selections based on the field with the blinking cursor.

Errors and status messages occur anytime the status or error occurs. They are not based on the field with the blinking cursor. These messages provide machine information to the operator.

Error messages may also stop and/or prevent machine operation until the cause of the error is corrected.

Status Bar

The status bar contains

- The name of the open, selected program.
- A calculator icon—select the icon to display a working, on-screen calculator.
- Units of measure (Inch or Millimeters)—select the units of measure in the status bar to toggle between Inch and Metric.
- Programming mode (R for Radius; D for Diameter)—select the programming mode in the status bar to toggle between Radius and Diameter.
- A yellow icon—indicates the feed hold is on when visible.
- A red icon—indicates the Emergency Stop button has been pressed when visible.
- A keyboard icon—select the icon to display a working on-screen keyboard.
- The current time.

When viewed on a single-screen console, all icons appear in the same status bar; when viewed on a dual-screen console, the program name and calculator icon appear on the left screen status bar, and the unit of measure, keyboard icon and time appear on the right screen status bar.

Console Buttons and Keys

References to console buttons and keys appear in bold text throughout the documentation. For example, the Start Cycle button appears as the Start Cycle button and the Manual key appears as the Manual console key in text.
Refer to the *Getting Started with WinMax* manual for information about console buttons and keys, in addition to other information about using softkeys and the pop-up text entry window.

## Using the Touchscreen

The console has a touchscreen for entering programming data. To make a selection, tap the screen on a softkey, field, or drop-down list using the stylus attached to the side of the console or another suitable pointing device.

## Icons

This manual may contain the following icons:

### Caution/Warning

⚠️ The operator may be injured and the machine severely damaged if the described procedure is not followed.

### Important

➡️ Ensures proper operation of the machine and control.

### Troubleshooting

❓ Steps that can be taken to solve potential problems.

### Hints and Tricks

💡 Useful suggestions that show creative uses of the WinMax features.

### Where can we go from here?

🌐 Lists several possible options the operator can take.

### Table of Contents

To assist with onscreen PDF viewing, this icon is located on the cover page. Click the icon to access the Table of Contents (TOC).

You can also access many of the same TOC entries from the Adobe Reader bookmarks located on the left side of the PDF page.
Hurco provides documentation for using WinMax software on a control or desktop in two formats: on-screen Help and PDF. The information contained in both formats is identical.

On-screen Help contains information about the current screen. If Help is not available for a screen, a Welcome screen appears with access to the Table of Contents, Index, or Search functions.

- To view the on-screen Help directly on a Hurco control, select the Help console key.
- To view the on-screen Help on the desktop software, select the Help icon in the menu bar.

PDF files are available on the hard drive. These files can be copied from the hard drive to a USB memory device and transferred to a PC for viewing and printing.

Using the On-screen Help

On-screen Help provides information about using WinMax. The Help is context-sensitive to the screen level. Press the console Help button to display the Help topic for the current screen. The following list describes Help functions:

- Buttons in the upper left-hand corner of the Help screen are used to move through Help topics and print screens.
  - Use the Hide button to hide the navigation pane.
  - Use the Back button to return to the previous Help screen.
  - Use the Print button to print the current displayed Help topic, if a printer is attached and configured. See Printing the Programming Manuals, on page - xxiii for more information about printing.

- Use the arrow buttons to move between pages within a Help topic and to move through topics.

- Use the Contents tab for a list of information sorted by subject:
  1. Select the “+” to expand the topic and view sub-topics.
  2. Select the topic to display it.

- Use the Index tab to show the Help index:
  1. Quickly scroll to an index topic by typing the topic in the box at the top of the index.
  2. Select a topic and the Display button to view the topic.
• Use the **Search** tab to search the Help for a word or phrase:
  1. Type the search word(s) into the text box at the top of the pane.
  2. Select the List Topics button. A list of topics that contain the search word(s) is displayed.
  3. Select a topic and the Display button to view that topic.

• Use the **Favorites** tab to save Help topics for quick access:
  1. Select the Add button at the bottom of the pane to add the current topic.
  2. Select a topic from the Favorites list, and select the Display button to view it.
    • Select a topic from the Favorites list, and select the Remove button to remove it from the list.

---

**Printing the Programming Manuals**

The WinMax On-screen Help is also provided in PDF format for easy printing. The information contained in the PDF files is identical to the on-screen Help. The PDF files may be copied to a floppy disk or USB memory device to be transferred to a PC for printing. Here are the steps to access the PDF files:

1. From the Input screen, select the **PROJECT MANAGER F8** softkey.
2. Select the **FILE MANAGER F7** softkey.
3. In the left-hand pane, navigate through the folders:
   • For WinMax Lathe on a machine, the path is D:\Hurco\Hurco Lathe\hlp.
   • For WinMax Desktop on a PC, the path is C:\Program Files\Hurco\Hurco Lathe\hlp.

   The PDF files will appear in the right-hand pane.

   - The SHOW ALL FILE TYPES field in User Interface Settings must be set to YES (default is NO) in order to see the PDF files in the directory. Access the SHOW ALL FILE TYPES field in Auxiliary Mode, Utilities/ User Preferences/ User Interface Settings.

4. Highlight the PDF file(s) in the right-hand pane, and select the COPY **F2** softkey.
5. Ensure that your media is loaded (either a floppy disk in the disk drive or a USB memory device in the USB port), and navigate to the proper location in the left-hand pane of the DISK OPERATIONS screen (either the floppy drive A: or the USB port E:). Highlight the desired location.
6. Place the cursor in the right-hand pane and select the PASTE **F3** softkey to paste the PDF file(s) to the desired location.

You may now remove your media and load the PDF file(s) onto a PC for printing.
OVERVIEW

This documentation describes the BNC (Basic Numerical Control) and ISNC (Industry Standard Numerical Control) Part Programming Editor portion of the CNC software as it is used on the machine tool console.

NC programming is designed to use as much of the Conversational system as possible. As a result, the screen layout is the same in both the NC and the Conversational systems. This allows a smooth transition between the two editors.

The primary difference between Conversational and NC programming is the program editors. The NC programming uses standard G and M codes; whereas, Conversational programming uses a supported programming language, such as plain English.

The CNC software can read NC files from the serial port directly into dynamic memory. NC files can be serially loaded to the hard disk.

NC part programs can be created using the CNC on the machine tool or off-line CNC programming software running on a personal computer. NC programs cannot be converted to Conversational programs, nor can NC programs be converted automatically to any other NC format.

Please refer to Getting Started with WinMax Lathe, NC Settings, on page 2 - 17 for information about default NC Settings.

NC Part Program Components .......................................................... 1 - 2
Off-line Part Program Formats ....................................................... 1 - 7
Default M and G Codes ................................................................. 1 - 10
Navigation ....................................................................................... 1 - 10
NC Editor ......................................................................................... 1 - 11
Starting a New NC Program ............................................................ 1 - 13
Program Editing Features ............................................................... 1 - 15
NC Editor Menus ........................................................................... 1 - 17
Importing and Exporting M Code Data .......................................... 1 - 29
NC Part Program Components

NC programs are a series of characters and words that form program blocks. These program blocks tell the machine tool how and where to move. The operator needs to understand the basic program structure and the types of codes in order to create, edit, and run a program successfully.

Program Start

All NC programs begin with a “%” (percent) character. When a percent character is received, the control starts to accept, check, and load blocks into its memory. If you are creating a new part program at the control, the percent character is automatically inserted at the beginning of the program Address Characters

In the following sample screen, the S [% indicates the beginning of the program and the E ] % line indicates the end of the program. Both characters appear automatically when beginning a new NC program.

![Figure 1–1. NC Editor Screen](image-url)
Sequence Number

A sequence number serves as a block label; it has no other significance within the part program except being required with GOTOs in the NCPP option and the M99 jump command. Sequence numbers are often used to mark the beginning of milling sequences so you can restart at a given sequence number or recall specific operations within the program.

When programming on an off-line system, sequence numbers should be used sparingly. Sequence numbers (N words) are optional in the NC Editor, and they are useful in programs sent over the RS-232 link. However, the absence of sequence numbers permits faster processing (loading, syntax checking, and parsing) of the part program and can result in improved part program execution. In addition, omission of these numbers increases the amount of the program that can fit into memory.

If you request renumbering of part program sequence numbers, any sequence numbers in GOTO statements will not be updated. In general, you will not want to renumber part programs that use GOTO statements. Refer to Renumbering and Tagging Menu, on page 1-23 for renumbering information.

Address Characters

An address character is the first character of a word in a program block. The following is a list of the address characters recognized by this system:

<table>
<thead>
<tr>
<th>Character</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>/</td>
<td>Block Skip Command. Specify blocks of code that are skipped when this control feature is enabled. The specified block, or portion of a block, begins with a “/” (forward slash). Refer to Block Skip Code, on page 1-6 for more information about Block Skip.</td>
</tr>
<tr>
<td>( )</td>
<td>Comment Command. Comment statements provide information about the part program. You can insert comment statements at the end of any part program block by enclosing the comment within parentheses. You may make an entire block of code a comment statement by enclosing it within parentheses. The Comment Command characters are used to delimit comments.</td>
</tr>
<tr>
<td>F</td>
<td>Feedrate. Sets the modal feedrate for cutting moves. Pitch. Sets the pitch for threading.</td>
</tr>
<tr>
<td>G</td>
<td>Preparatory Functions. G codes have two basic functions: • specify a modal condition. For example, G20 establishes Inch mode, G21 establishes Millimeter mode, G90 establishes Absolute mode and G91 establishes Incremental mode. Programming G20 Inch mode and G90 Absolute mode in the first block of a part program tells the control to remain in inch mode until G21 Millimeter mode is programmed, and to remain in Absolute mode until G91 Incremental mode is programmed. • specify the type of tool motion (example: G00 programs a rapid move, G01 programs a linear feed move, G02 and G03 program circular moves).</td>
</tr>
<tr>
<td>I</td>
<td>X-axis Center/Offset coordinate for programming geometric information needed to determine the endpoint of a motion command.</td>
</tr>
<tr>
<td>K</td>
<td>Z-axis Center/Offset coordinate for programming geometric information needed to determine the endpoint of a motion command.</td>
</tr>
</tbody>
</table>
### Address Characters (continued)

<table>
<thead>
<tr>
<th>M</th>
<th>Miscellaneous Functions. M Codes perform miscellaneous functions such as turning on the spindle, turning on coolant, specifying a program stop, or specifying an end of program.</th>
</tr>
</thead>
<tbody>
<tr>
<td>N</td>
<td>Sequence Number. Each block may contain a nine-digit integer N Number. N Numbers are not required; however, programmers can use them as sequence numbers to order the part program. It is not required for the numbers to appear in order within the part program.</td>
</tr>
<tr>
<td>P</td>
<td>Thread Angle. Starting thread angle for threading, G33 with P Value.</td>
</tr>
<tr>
<td>R</td>
<td>Radius Programming. Generally used for programming the Radius. Can also be used with G73, Pattern Repeating for programming the number of repetitions. Can also be used with G74 G75 for programming the retract distance. See G Code descriptions for parameter definitions.</td>
</tr>
</tbody>
</table>
| S   | Spindle Speed Function. Sets the spindle speed for two-axis machines (TM and TMX). If using the S command on machines with multiple spindles (including live-tooling), the S command refers to ALL spindles.  
  • S:M19 orients all spindles on a multiple spindle machine.  
  • S1:M19 orients only the main spindle.  
  • S:S2000 ramps multiple spindles to 2000 RPM.  
  • S1:S2000 commands only the main spindle. |
| S1  | Spindle Speed Function. Sets the spindle speed for main spindle for live-tooling machines. Please refer to the S address character definition above for examples. |
| S2  | Spindle Speed Function. Sets the spindle speed for live-tooling spindle 2. Please refer to the S address character definition above for examples. |
| S3  | Spindle Speed Function. Sets the spindle speed for sub-spindle. Please refer to the S address character definition above for examples. |
| T   | Tool Select. Identifies the active tool and activates the offsets for the tool. |
| X   | Primary X Motion Dimension, Dwell Time coordinate for programming geometric information needed to determine the endpoint of a motion command. |
| Z   | Primary Z Motion Dimension coordinate for programming geometric information needed to determine the endpoint of a motion command. |
| C   | Primary C Motion Dimension coordinate for programming geometric information needed to determine the endpoint of a motion command. This code is used for Live Tooling only. |
| W   | Primary W Motion Dimension coordinate for programming geometric information needed to determine the endpoint of a motion command. |
| Y   | Primary Y Motion Dimension coordinate for programming geometric information needed to determine the endpoint of a motion command. |
Special Characters

Special characters are ASCII characters within a file which have special meaning to the system and cannot be edited. The following special characters are recognized by the NC software:

- **% (Beginning/End of tape)**—signals the system that all of the following characters are part of the program. The system automatically adds this character to the beginning of a new program. You can also include the % character to signal the End of Tape.

- **E. of tape (EOT)** (optional)—signals the NC system that no more legal program characters follow. This character is optional to provide compatibility with existing programs that include EOT characters at the end.

- **[CR]** (Carriage Return)—signals the End of a Program Block.

- **[CRLF]** (Carriage Return/Line Feed Pair)—signals the End of a Program Block (identical to [CR]).

  ![CRLF is not shown when the program is viewed in the NC Editor.](image)

Words

A word is a group of alphanumeric characters. The first character of a word is an address character—a letter such as M or G. The address character is followed by a signed or unsigned numeric value. Some sample NC words are “X-.03” and “G00.” One word or group of words form a program block.

Block

A block is a group of words terminated by the end-of-block character. Each block within a part program must be terminated with either a carriage return [CR] or a carriage return line feed [CRLF].

When changing the cutting type (for example, hole drilling or profile turning) for a data block, a different data block must be created to describe the new cutting type. When a different tool is needed in a program, a new data block must be created to describe the actions of that tool.
The following illustration shows a typical NC block and its components:

![Figure 1–2. Typical NC Block](image)

<p>| | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Word</td>
<td>2</td>
<td>Address character</td>
<td>3</td>
<td>Numeric indicator</td>
<td>4</td>
<td>Feedrate word</td>
<td>5</td>
</tr>
<tr>
<td>6</td>
<td>Modal preparatory functions</td>
<td>7</td>
<td>One-shot preparatory functions</td>
<td>8</td>
<td>Dimension words</td>
<td>9</td>
<td>Miscellaneous function</td>
<td></td>
</tr>
</tbody>
</table>

**Block Skip Code**

Block Skip will skip specified blocks of code when turned on. To skip a block of code, place a slash (/) at the front of the block.

N120 G00 Z4 (rapid to Z)
/N125 G42 (cutter compensation right, skip if block skip on)

Please refer to *NC Editor Settings Menu, on page 1 - 27* for information about enabling the Block Skip state.

The program processes blocks before they appear on the screen. If you change block skip mode while a part program is running, the control may have already processed blocks marked to be skipped. The control will not go back and reprocess these blocks.

When programming with **M200**, program BNC M200 - Block Skip Synchronization or ISNC Block Skip Sync (ISNC M200) so it can be activated when the tool is off the part. Using this code when in a cutting mode may cause the tool to leave dwell marks.
Off-line Part Program Formats

This section describes how an NC post processor part program code should be configured on an off-line CAM system. The order in which the control processes a block of code (block code processing) and the data format for each code are important to a post processor.

Block Code Processing

A block of code can contain 0 to 10 G codes and 0 to 10 M codes. The table below shows the order in which the control will process a block of part program code. The order in which each entry appears within a block of code does not affect the order of processing, except for block delete codes, which are processed at the point they appear in a block.

Order for Processing Blocks of Code

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
</table>
| Block Delete Codes            | Place a slash at the front of a block to skip it during processing. Block Delete Codes are processed at the point they appear in a block. Example—
                                 | /N125 G42 (cutter compensation right, skip if block del on)                  |
| 1. N Numbers                  | N Numbers can contain up to 9 digits. Example—N123456789                      |
| 2. Comment Statements         | Enclose comments in parentheses. Example—(this is an example of a Comment Statement) |
| 3. Specific G Codes           | Refer to *Basic NC G Codes, on page 2 - 1* or *ISNC G Codes, on page 4 - 1* for a list of G Codes and their descriptions. |
| 4. Feedrate Override          | Choose between M48 or M49.                                                   |
| 5. Beginning of Block M Codes | Choose between M07 and M08.                                                   |
| 6. Tool (T Code) Commands     | Tool motion does not occur unless axis motion is programmed in the block containing the T Code, or until motion is programmed in a following block. |
                                 | T0303                                                                        |
                                 | M03 S1200                                                                    |
                                 | T00 cancels tool offsets.                                                    |
### Order for Processing Blocks of Code (continued)

<table>
<thead>
<tr>
<th>Block</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>7. After Tool Change, Before Motion M Codes</td>
<td>Choose between M3 and M4.</td>
</tr>
<tr>
<td>8. Spindle Speed (S Code) Command</td>
<td>Set the spindle speed (rpm and css) using an S code.</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Example—G92 S5000 (prevent the spindle from exceeding 5000 rpm)</td>
</tr>
<tr>
<td>9. Dwell Command</td>
<td>Enter <strong>G04</strong>. Refer to BNC G04 - Dwell or ISNC G04 - Dwell for information about this code.</td>
</tr>
<tr>
<td>10. Feedrate (F code) or Dwell</td>
<td>Choose between G94 and G95 to set up the feedrate values.</td>
</tr>
<tr>
<td></td>
<td>Dwell commands programmed with G94 or G95 are interpreted as the number of seconds or spindle revolutions to dwell.</td>
</tr>
<tr>
<td>11. Motion Commands</td>
<td>Choose among G00, G01, G02, G03, and G33.</td>
</tr>
<tr>
<td>12. Exact Stop</td>
<td>Enter G09.</td>
</tr>
<tr>
<td>13. Spindle Stop</td>
<td>Enter M5 to stop spindle motion.</td>
</tr>
<tr>
<td>14. End of Block System M Codes</td>
<td>Choose among M0, M1, M2, and M30.</td>
</tr>
</tbody>
</table>

*Table 1–2. Order of Block Code Processing*
## Block Data Formats

The table below shows the format (ranges) for the different types of blocks of part program code.

<table>
<thead>
<tr>
<th>Entry</th>
<th>Description</th>
<th>Format (range)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G</td>
<td>G Code</td>
<td>5 (0-65535)</td>
</tr>
<tr>
<td>M</td>
<td>M (Miscellaneous) Code</td>
<td>5 (0-65535)</td>
</tr>
<tr>
<td>S</td>
<td>Spindle Speed (rpm) and (css)</td>
<td>9 (0 - 2 x 10^9) (no practical limit)</td>
</tr>
<tr>
<td>T</td>
<td>Tool Code + Offset</td>
<td></td>
</tr>
<tr>
<td></td>
<td>T00 cancels tool offsets</td>
<td>0 (0-999999999)</td>
</tr>
<tr>
<td>N</td>
<td>N Numbers</td>
<td>9 (0-999999999)</td>
</tr>
<tr>
<td>F</td>
<td>G94 FPM Mode</td>
<td>Refer to BNC G20 - Inch Mode (default) and G21 - Millimeter Mode or ISNC G20 - Inch Mode (default) and G21 - Millimeter Mode for these formats.</td>
</tr>
<tr>
<td></td>
<td>G95 FPR Mode</td>
<td></td>
</tr>
<tr>
<td>F</td>
<td>Dwell in G94 fpm mode</td>
<td>3.4 seconds (0.0001 - 999.9999)</td>
</tr>
<tr>
<td></td>
<td>Dwell in G95 fpr mode</td>
<td>3.4 revs (0.0001 - 999.9999)</td>
</tr>
</tbody>
</table>

### Table 1–3. Block Data Formats

Where decimals are included in the format, the number on the left of the decimal (3) represents the number of places to the left and the number on the right (4) of the decimal represents the number of places to the right of the decimal.

3.4 = nnn.nnnn
Default M and G Codes

Upon power up, control reset, initial entry into the NC Editor, or after erasing a program, the system presets these M codes as defaults:

- M05  Spindle Off
- M09  Both Coolant Systems Off

The system also presets certain G codes as the default active codes. The default G codes are highlighted in the BNC G Code Table, on page 2 - 4 and the ISNC G Code Table, on page 4 - 3.

The system uses the units specified when the NC Editor is selected, not the G codes, for graphics display and running the part program.

Navigation

To move the cursor from a block to the beginning of the next block, press the down arrow (\(\downarrow\)). Use the right/advance arrow (\(\rightarrow\)) and the left/back arrow (\(\leftarrow\)) to move the cursor within a block. Use the Enter key to move the cursor between words and blocks.

To move to the beginning of the current block, press the Home key or the up arrow (\(\uparrow\)). If the cursor is already at the beginning of the block, pressing the up arrow moves the cursor to the beginning of the last word in the previous block.

To move from a word to the beginning of the next word, press the Enter key. If the cursor is at the end of the current block when the Enter key is pressed, the editor automatically presents the next legal address character. To move from one character to the next, press the right arrow. If the cursor is at the end of the current block, the cursor wraps around to the beginning of the block.

To move from one character to the preceding character, press the left arrow. If the cursor is at the start of the current block, it wraps around to the end of the current block.

Delete characters or words from a block using these methods:

- To delete numeric data, position the cursor on the number and press the Delete key.
- To erase the entire word, position the cursor on the address character and then press the left arrow or the Delete key. The entire word is removed since numeric data is not allowed in an NC program without an address character to introduce it.
NC Editor

The NC Editor contains information about the program. You can view the NC Editor by either opening an existing NC program or creating a new NC program and accessing the Part Programming screen.

NC Editor provides a wide range of tools to review, create, and modify NC part programs. The editor has the following features:

- Several ways of selecting the code, such as by dragging the pointing device across the screen, and by using keyboard keys (if available).
- Copying/Pasting/Cutting of a block of selected code, including to applications outside WinMax.
- Undo/Redo functionality.
- Unlimited number of tags.
- Real-time syntax check. Incorrect syntax is indicated by showing the incorrect text in red. Comments are shown in a user-defined color (green in this example):

```
N9 Z-44.45
N10 X101.6
G1 G40 X104.14 Z2.54
M9
N65 [M1]
(FINISH WITH A DIFFERENT TOOL)
G91 G28 U0. W0.
G90
N75 [GE]
T1212
G97 S2000 M3
M8
```

See NC Editor Settings, on page 1 - 28 for information about the comment color.

- Real-time indicators of the meanings of the G codes and M codes. Place the cursor on the G code or M code and the definition is displayed in the prompt area of the screen.
- Simplified and friendlier access to common editing tasks, such as jumping and searching operations.
- Keyboard shortcuts:
  - Ctrl + Home—jump to the beginning of program.
  - Ctrl + End—jump to the end of program.
  - Ctrl + C—copy selected text to the clipboard.
  - Ctrl + F—opens a Find window. Enter text to find in the current program.
  - Ctrl + G—opens a Goto window. Enter a block number to navigate to that block.
  - Ctrl + H—opens a Find and Replace window.
  - Ctrl + V—paste the copied or cut text
• Ctrl + X—cut the selected text from screen and copy to clipboard
• Ctrl + Y—redo the last undo. This command can also be disabled using the Disable Undo and Redo Yes/No field selections on the NC Editor Settings screen.
• Ctrl + Z—undo the last change. This command can also be disabled using the Disable Undo and Redo Yes/No field selections on the NC Editor Settings screen.

The status bar provides real-time updates on the status on the program being edited. The information shown in the status bar is as follows:

![Status Bar](image)

1. Part Program Name
2. Units
3. Programming Mode
4. Coding Standard
5. Current Line Number
6. Insert/Overstrike Mode
7. Block Skip On/Off

**Figure 1–3. NC Editor Status Bar**

The Part Program name is shown first in the status bar. Programming Mode can either be radius (R) or diameter (D). Coding standard can either be Industry Standard (I) or Basic NC (B). Current Line number indicates the current location of the cursor in the part program. Insert mode is indicated as INS, while overstrike mode is indicated as OVR. SKIP On indicates Block Skip is turned on; SKIP OFF indicates Block Skip is turned off.

Refer to *NC Editor Menus, on page 1 - 17* for information about the NC Editor softkey menus.
Starting a New NC Program

Procedures for entering the program name, the part setup information, the tool
descriptions, and the program parameters are described in Getting Started with WinMax
Lathe, Project Manager, on page 3 - 1 and Programming Basics, on page 4 - 1, in addition
to information about saving files.

The parameters, part setup (except work offsets), and tool setup (except tool offsets)
used for NC programs are not stored as part of the NC program. This information can be
loaded from a Conversational program before going into the NC mode.

Part and tool setup information is not automatically deleted or replaced when loading a
new NC program. If part and tool setup information appears on the setup screens after
loading in a new NC program, it is the setup information from the previously displayed
part program.

When there is no NC program loaded in memory, the system automatically assigns the
filename NONAME#, where the # represents a sequential number from 0 to 99. The file
extension is set in User Preferences. Refer to Getting Started with WinMax Lathe, Project
Manager, on page 3 - 1.

These steps for creating an NC part program help determine the most efficient tool
movement and basic program structure to save time during programming:

1. Determine the tool path on the print and label the points where the path
direction changes.
2. Make a chart showing the coordinates of each point identified in the previous
step.
3. Identify the turret movements that will be necessary during cutting.

NC Programming Rules

Here are some basic rules to follow when creating NC part programs:

- The axis letter always precedes the numeric information.
- In most cases an integer works the same as a decimal or real number. In the
following cases an integer is scaled by the appropriate scaling factor to
maintain compatibility with existing NC programs:

  Feedrate: F (BNC only)
  Rotation: R (ISNC only)
  Dwell: P, X (Both BNC and ISNC)
  Scaling: P (ISNC only)

  If an integer is below the acceptable range after scaling, a “Below
  Minimum Value” error message occurs.
• All axis dimensions are considered to be positive unless a minus sign is entered. When describing axis motion, the codes for the program block must contain the following information in order to move properly:
  • Axis identification (e.g., X, Y, Z).
  • Direction the axis will move (+ or -).
  • Distance the axis will move (e.g., 4.0).
  • Enter the speed preceded by the F address character to program a feedrate in a block.
  • Include a Z parameter to permit the system to draw the part on the graphics screen. An absolute Z command must occur after a tool change before making another move command.
Program Editing Features

Several editing features are available with the WinMax Lathe Max control. Keyboard and software features provide you with the editing capabilities to create and update part programs.

Please refer to Getting Started with WinMax Lathe, Consoles, on page 1-5 for additional information about the console.

The console keyboard contains these keys:

- **Programming Mode keys**—Input, Review, and Menu keys.
- **Standard AT-keyboard keys**—Insert, Delete, Home, End, Page Up, Page Down.

  Please refer to Getting Started with Your WinMax Lathe, Standard Keys, on page 1-28 for details about using these keys for specific functions for Tool Setup or Tool Review.

  Refer to Program Review, on page 1-16 and for details about using these keys with the Program Review screen.

- **Cursor Control keys**—Arrows, Enter, alt, C, and F function keys. Refer to Getting Started with WinMax Lathe, Cursor Control Keys, on page 1-29.

- **Draw key**—displays the graphic in wireframe view. Refer to Getting Started with WinMax Lathe, Wireframe Graphics, on page 4-86 for more information about Wireframe Graphics.

- **Verify key**—displays the graphic in 2D or 3D view. Refer to Getting Started with WinMax Lathe, Verification Graphics, on page 4-92 for more information about Verification Graphics.

- **Optional AT-Keyboard**—allows you to enter data using a keyboard. Refer to Getting Started with WinMax Lathe, AT-Keyboard, on page 1-30.

The software provides these features:

- **Pop-up Text Entry window**—allows you to enter text, such as naming a part program, if the console does not have an optional AT-keyboard. Access this feature by selecting the keyboard icon on the bottom of the screen.

- **NC Subprogram Review screen**—lists the data blocks in a program and the tools programmed for each block. Refer to Program Review, on page 1-16 for information about using this screen.

- **Tool Review screen**—lists the tool number, type, orientation, size, programming mode (diameter or radius), speed, and feed for each tool programmed in the data block. Please refer to Getting Started with WinMax Lathe, Tool Review, on page 4-77 for details about using Tool Review.
Program Review

Press the Review console key to display the NC Program Review screen. A summary of the NC subroutines appears by number.

![NC Program Review screen](image)

**Figure 1–4. NC Program Review screen**

To go directly to a subroutine, highlight the subroutine and press the Enter key or the JUMP TO SUBPROGRAM F3 softkey. Refer to *NC Editor Menus, on page 1 - 17* for information about the NC Editor’s screen and softkeys.

These Standard Console keys perform these functions with the Program Review screen:

- The **Home** console key jumps to the first subroutine listed in the Program Review screen.
- The **End** console key jumps to the last subroutine listed in the Program Review screen.
- The **Page Up** console key jumps to the first subroutine visible on the screen.
- The **Page Down** console key jumps to the last subroutine visible on the screen.

When the entire part program contains more subroutines than can be displayed on one screen, use the **Page Down** and **Page Up** console keys or the scroll bar located on the right-hand side of the screen to scroll through the listed subroutines.

You can clear, delete, copy, and move data blocks to another location in a part program or even another program using the Edit Functions described in the *Edit Functions Menu, on page 1 - 21*.

- To select multiple blocks, select the first block and move the pointing device down to the last block without lifting the stylus.
- With the optional AT-keyboard, you can use **Ctrl + select** (using the pointing device for selecting) to select non-consecutive data blocks.
NC Editor Menus

The Hurco NC system provides many levels of program editing, as well as editing tools, to simplify the task. Press the MORE ➔ F7 softkey on each menu to move to the next menu selection. The menus appear in a circular loop; every time you press the MORE ➔ F7 softkey you access the next menu.

The NC Editor contains these top level menus:

- Basic Programming Menu .................................................. 1 - 17
- Jump and Search Functions Menu ....................................... 1 - 19
- Edit Functions Menu .......................................................... 1 - 21
- Renumbering and Tagging Menu .......................................... 1 - 23
- Program Execution Menu .................................................... 1 - 26
- NC Editor Settings Menu ..................................................... 1 - 27

Basic Programming Menu

While entering NC codes to create blocks, you may wish to insert new blocks, delete blocks, or display different sections of the part program on the screen.

The Basic Programming softkeys provide these functions:

- **Insert Block Before**—inserts a blank line before the block where the cursor is located. This permits addition of a new block of data. This softkey will be disabled if text can't be inserted at the current cursor location.
• **Delete Block**—removes the block where the cursor is positioned. This softkey will be disabled if the block can't be deleted.

• **Jump to Beginning**—moves the cursor to the beginning of the first program block in memory. If a keyboard is available, Ctrl + Home combination will result in the same action.

• **Jump to End**—moves the cursor to the beginning of the last program block in memory. If the keyboard is available, Ctrl + End combination will result in the same action.

• **Jump and Search Functions**—invokes Jump and Search Functions menu.

• **Edit Functions**—invokes Edit Functions menu

• **More ➔**—displays the next menu.

• **Exit Editor**—exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.
Jump and Search Functions Menu

The Jump and Search provides the flexibility to locate hard-to-find items in program memory using the block or sequence number or searching for specific address characters, numeric parameters, or words.

When the Jump & Search Functions softkey is selected from the Basic Programming menu, these softkeys appear:

- **Jump to Block Number**—enter a block number in the popup box. The cursor is positioned on the specified block. The number entered refers to the position that block has in NC memory.

  ![Jump to Block Number](image1.png)

  You can also use the touchscreen to select the Line number in the Editor status bar to open the Jump to Block popup box.

- **Jump to Sequence Number**—enter a sequence number (N code) in the popup box. The cursor is positioned on that block.

- **Search for Text**—activates the Search submenu. Refer to Search Submenu, on page 1 - 20 for more information.

- **Jump to Beginning**—moves the cursor to the beginning of the first program block in memory. If a keyboard is available, Ctrl + Home combination will result in the same action.

- **Jump Page Forward**—moves the cursor to show the screen one page forward.

- **Jump Page Backward**—moves the cursor to show the screen one page backward.

- **Jump To End**—moves the cursor to the beginning of the last program block in memory. If the keyboard is available, Ctrl + End combination will result in the same action.

- **Exit**—returns to the previous menu.
**Search Submenu**

The Search submenu allows you to search for specific text in the part program.

![ISNC Search Submenu](image)

**Figure 1–7. ISNC Search Submenu**

The menu contains the following softkeys:

- **Enter Text To Search**—opens a popup box where search text is entered. Type search term and select the Find Next button. The NC editor finds the next occurrence of the entered text following the current cursor position. If found, the text is highlighted. Selecting the Search Up checkbox will search for the search term prior to the current cursor position.

- **Search Again**—repeats the last search operation without prompting the user.

- **Find and Replace**—opens a popup box where search text is entered and replacement term is specified.
  - The **Replace Next** screen button finds the next instance of the search term and replaces it with the new term. To find and replace the next instance you must select the Replace Next button again.
  - The **Replace All** screen button finds and replaces all instances of the search term.

- **Exit**—invokes the Jump and Search menu.
Edit Functions Menu

The Edit menu provides advanced editing functionality to the user.

When the Edit Functions softkey is selected from the Basic Programming menu, these softkeys appear:

- **Insert/Overstrike Mode Toggle**—switches the data entry style between insert and overwrite. Currently active mode is shown on the status bar of the window - “INS” indicates Insert and “OVR” indicates Overwrite.

  You can also use the touchscreen to toggle the Insert/Overstrike Mode in the Editor status bar.

  Or, you can press the Insert console key to toggle between the two modes.

- **Copy Selection**—copies the selected text to the Windows’ clipboard memory. If the keyboard is available, this action can be achieved by pressing Ctrl + C combination. Text can be selected by dragging the pointing device (up/down/right/left) across the editing area of the screen, or by holding down the keyboard Shift key and using the arrow keys to select text.

- **Cut Selection**—copies the selected text to the Windows’ clipboard. In addition, the selected text is deleted from the part program. If the keyboard is available, Ctrl + X combination will cut the selected text and place it in the Windows’ clipboard.

- **Paste Selection**—text previously copied to the Windows’ clipboard is inserted at the current cursor position. If the keyboard is available, this action can be achieved by pressing Ctrl + V keys.
• **Redo**—redo the editing operation(s) previously undone by the Undo softkey. If the keyboard is available, Ctrl + Y combination will result in the same action. The feature can be disabled using the NC Editor Settings screen and the Disable Undo and Redo field.

• **Undo**—undo the previous editing operations starting from the latest to the earliest. The depth of the undo buffer is 100. If more than one text modification was performed in a single step (e.g. "Replace All"), all those modifications will be undone in one step as well. If the keyboard is available, Ctrl + Z combination will result in the same action. The feature can be disabled using the NC Editor Settings screen and the Disable Undo and Redo field.

• **Find and Replace**—activates a popup box where search text and replacement text are entered. NC editor finds occurrences of the search text and replaces it with the replacement text.

• **Exit**—loads the Basic Programming menu.
Renumbering and Tagging Menu

Renumbering and tagging menu offers access to renumbering and tagging functionality. This menu can be accessed by selecting More from the Basic programming menu.

![ISNC Renumbering and Tagging Menu](image)

**Figure 1–9. ISNC Renumbering and Tagging Menu**

The menu contains the following softkeys:

- **Block Renumbering Mode**—invokes Block Renumbering Mode submenu. See *Block Renumbering Submenu, on page 1 - 24*.

- **Tag Block**—tags a block by underlining it when selected. Select the Tag Block softkey a second time to remove the tag from the block. There is no limit on the amount of tagged blocks you can have.

- **Jump to Previous Tagged Block**—NC editor tries to find the closest tagged block before the current cursor's position. If such block is found, the cursor is placed in the beginning of the block.

- **Jump to Next Tagged Block**—NC editor tries to find the closest tagged block after the current cursor's position. If such block is found, the cursor is placed in the beginning of the block.

- **Tagged Block List**—invokes Tagged Blocks List screen, which allows the user to review the tagged block and manage the tags. See *NC Tag List, on page 1 - 25*.

- **More**—invokes Program Execution menu.

- **Exit Editor**—exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.
**Block Renumbering Submenu**

The Block Renumbering mode submenu allows the user to search for specific text in the part program.

![ISNC Block Renumbering Submenu](image)

*Figure 1–10. ISNC Block Renumbering Submenu*

The menu contains the following softkeys:

- **Jump & Search Functions**—opens the Jump and Search Functions menu. Refer to *Jump and Search Functions Menu, on page 1 - 19*.
- **Enable Optional Numbering**—enables user-assigned block numbering mode.
- **Enable Auto Numbering**—enables automatic block numbering mode.
- **Renumber Numbered Blocks**—enables block renumbering. The renumbering interval is entered into the popup box and the blocks are automatically renumbered. All numbered blocks will be renumbered in the program.
- **Renumber Selected Blocks**—enables renumbering of selected blocks. First select the blocks to be renumbered by dragging the pointing device (up/down/right/left) across the editing area of the screen, or by holding down the keyboard Shift key and using the up/down/left/right arrow keys to select text. Then select the softkey and specify the renumbering interval. The selected blocks are renumbered.
- **Exit**—invokes the Renumbering and Tagging menu.
NC Tag List

The NC Tag List screen is accessed with the Tagged Block List softkey on the Renumbering and Tagging Menu. This softkey is not active (grayed out) if there are no tagged blocks. The tagged blocks in a program are displayed:

Figure 1–11. ISNC NC Tag List Screen

Softkeys are:

- **Jump to Selected Tag**—goes to the selected tag in the program.
- **Clear Tag**—deletes the selected tag.
- **Clear All Tags**—deletes all tags in the list.
- **Exit**—invokes the Renumbering and Tagging menu.
Program Execution Menu

Program Execution menu allows selection of the part program to be processed for graphics and for execution. This menu can be accessed by twice pressing More ➔ (first from the Basic Programming menu and then from the Renumbering and Tagging menu).

The menu contains the following softkeys:

- **Set Start Marker**—indicates the block that the system should use to start program execution when running and verifying the program. A letter S is inserted to the left of the block. All blocks previous to the Start block are grayed out.

- **Set End Marker**—indicates the block that system should use to end program verification and execution. A letter E is inserted to the left of the block. All blocks following the End block are grayed out.

  => If the Start and End markers overlap, the dollar ($) sign is displayed to the left of the line.

- **Reset Start/End Markers**—restores start and end markers to their defaults. The defaults are at the beginning and end of the program.

- **More ➔**—invokes NC Editor Settings menu.

- **Exit Editor**—exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.
NC Editor Settings Menu

NC Editor Settings allows you to modify the editor’s behavior.

Softkeys are

- **NC Editor Settings**—invokes NC Editor Settings screen. See *NC Editor Settings*, on page 1 - 28.
- **More ➔**—invokes Basic Programming menu.
- **Exit Editor**—exits the NC Editor and moves to the Input screen. Select Part Programming to return to the NC Editor screen.

![NC Editor Settings Menu](image-url)
**NC Editor Settings**

The NC Editor Settings screen allows you to set the Block Skip Enable state, adjust the color of commented text, and disable the Undo and Redo feature.

![NC Editor Settings Screen](image)

*Figure 1–14. ISNC NC Editor Settings screen*

Fields and softkeys are

- **Block Skip Enable**—indicates whether blocks with the Ignore character (/) are interpreted and executed or ignored. Yes (Skip Off) and No (Skip On) softkeys appear when the cursor is in this field. To change the Block Skip state,
  - Select the **Yes** softkey to ignore the blocks with the Ignore character (/). **Skip On** appears in the status bar.
  - Select the **No** softkey to execute the blocks with the Ignore character (/). **Skip Off** appears in the status bar.

- **Comment Color**—displays the current comment color. The color can be changed by selecting the Change... screen button or the Change Comments Color softkey. A window opens with Red, Green, and Blue slide adjustments for changing the color of comment text.
  - Select **OK** to save changes and return to the NC Editor Settings screen.
  - Select **Cancel** to close the window without saving changes.

- **Current Font**—displays the current NC Editor font. To change the font, select the **Change Current Font** softkey. A Choose Font window opens with Font and Size selections. A Sample of the selected font appears in the lower portion of the window.
  - Select **OK** to save changes and return to the NC Editor Settings screen.
  - Select **Cancel** to close the window without saving changes.

  The **OK** screen button is inactive if a font size other than a pre-defined size selected from the list is entered.
• **Disable Undo and Redo**—indicates whether the NC Editor’s Undo and Redo functions are disabled.
  • When **Yes** is selected, the Undo and Redo softkeys are not available (grayed out) on the Edit Functions softkey menu.
  • When **No** is selected, the Undo and Redo softkeys are available on the Edit Functions softkey menu for use following an editing operation.

**Importing and Exporting M Code Data**

The import and export functions feature allows you to import NC files written for machine types for use with controls other than WinMax Lathe Max controls. In addition, this feature makes it possible for you to export NC files written for WinMax Lathe Max controls to machine types for use with other controls.

From the Input screen, select the Import/Export Functions F7 softkey. The Import and Export Functions screen appears.

![Figure 1–15. Import and Export Functions screen](image)

Softkeys are:

• **Import NC State From File** F1—imports M Codes from a file.
• **Save NC State To File** F3—saves NC setup data to a file.
Select Save NC State To File F3 softkey and this screen appears:

![Save NC State to File screen](image)

*Figure 1–16. Save NC State to File screen*

This screen shows folders on the left pane and file names on the right pane. The File Name field defaults to the name “ncstate.ncl.”

You can select the Create Folder F4 softkey to set up a folder to hold the NC State File. A pop-up window appears for you to enter the folder name. Select OK and the folder appears in the left pane. When Save NC State to File is selected, the screen opens to the last folder used.

To name the file and save it to the folder,

1. Open the folder.
2. Type the file name in the File Name field.
3. Select the Save F1 softkey. A message appears confirming the file was saved successfully.
4. Select OK to clear the message. The Input screen appears.
To view the file in the directory listing,

1. Select the Import/Export Functions F7 softkey on the Input screen
2. Select the Save NC State to File F3 softkey. The screen shows directory information, including the folder and file.

![Figure 1–17. Save NC State to File screen with new folder and NC State file](image)

To load this file from the Import and Export Functions screen,

1. Select the Import NC State From File F1 softkey. The screen opens to the last folder used.
2. Select the file name from the right-hand pane.
3. Select the Load Selected File(s) F1 softkey. The Import NC State to Conversational screen appears.

   The Import NC State to Conversational screen appears if a conversational program (.LNC) is currently loaded. Load an NC file or create a new NC file if a conversational program is loaded.

![Figure 1–18. Import NC State screen](image)
4. Select the program components to be restored.
   
   To import only the M Codes from an NC file, select only ISNC M Codes. This selection retains existing tool offsets, work offsets, and parameters.

5. Select the Begin Operation F2 softkey. A pop-up window appears confirming the NC State being successfully restored. Select OK to close the window.
BASIC NC G CODES

The following G Codes are defined in this chapter:

G00 - Linear Motion at Rapid (default) ........................................ 2 - 6
G01 - Linear Motion at Feed ....................................................... 2 - 6
G02 - Clockwise Circular Motion at Feed ..................................... 2 - 7
G03 - Counterclockwise Circular Motion at Feed ......................... 2 - 8
G04 - Dwell ................................................................................... 2 - 10
G06 - Probe/Block Skip .............................................................. 2 - 10
G07 - Radius Programming .......................................................... 2 - 11
G08 - Diameter (default) Programming ......................................... 2 - 13
G09 - Exact Stop ........................................................................... 2 - 15
G20 - Inch Mode (default) ............................................................. 2 - 15
G21 - Millimeter Mode ................................................................. 2 - 16
G33 - Threading ............................................................................ 2 - 17
G40 - Cutter Radius Compensation Off (default) ......................... 2 - 18
G41 - Cutter Radius Compensation Left ....................................... 2 - 19
G42 - Cutter Radius Compensation Right .................................... 2 - 19
G53 - Program Machine Coordinates ......................................... 2 - 22
G59 - Cancel Work Coordinate Offsets (default) ....................... 2 - 22
G73 - Peck Drill with Chip Break Drill Cycle ............................... 2 - 23
G74 - Left Hand Tapping ............................................................... 2 - 24
G78 - Threading Cycle ................................................................. 2 - 25
G80 - Cancel Drill Cycle (default) ............................................... 2 - 37
G81 - Drill Cycle .......................................................................... 2 - 37
G82 - Drill Cycle with Dwell ........................................................ 2 - 38
G83 - Peck Drill Cycle ................................................................ 2 - 39
G84 - Right Hand Tapping ............................................................. 2 - 40
G90 - Absolute Programming (default) ....................................... 2 - 41
G91 - Incremental Programming .................................................. 2 - 43
G92 - Work Coordinate Offsets or Spindle Max Speed ............... 2 - 45
G94 - Feed per Minute (default) .................................................... 2 - 45
G95 - Feed per Revolution ............................................................ 2 - 45
G96 - Constant Surface Speed (CSS) .......................................... 2 - 46
G97 - Direct Spindle Speed (default) ............................................. 2 - 46
G98 - Drill Cycle Initial Level Return (default) ......................... 2 - 47
G99 - Drill Cycle R Plane Return .................................................. 2 - 48
<table>
<thead>
<tr>
<th>G Code</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>G101</td>
<td>Axial Surface of the Part - Main Spindle without Linear Y</td>
<td>2 - 49</td>
</tr>
<tr>
<td>G102</td>
<td>Radial Surface of the Part – Main Spindle without Linear Y</td>
<td>2 - 50</td>
</tr>
<tr>
<td>G103</td>
<td>Axial Surface of the Part – Main Spindle with Linear Y</td>
<td>2 - 52</td>
</tr>
<tr>
<td>G104</td>
<td>Radial Surface of the Part – Main Spindle with Linear Y</td>
<td>2 - 53</td>
</tr>
<tr>
<td>G105</td>
<td>Axial Surface of the Part – Sub-spindle without Linear Y</td>
<td>2 - 54</td>
</tr>
<tr>
<td>G106</td>
<td>Radial Surface of the Part – Sub-spindle without Linear Y</td>
<td>2 - 55</td>
</tr>
<tr>
<td>G107</td>
<td>Axial Surface of the Part – Sub-spindle with Linear Y</td>
<td>2 - 57</td>
</tr>
<tr>
<td>G108</td>
<td>Radial Surface of the Part – Sub-spindle with Linear Y</td>
<td>2 - 58</td>
</tr>
<tr>
<td>G109</td>
<td>Turning / Lathe Mode</td>
<td>2 - 59</td>
</tr>
</tbody>
</table>
Overview

G Codes initiate axis motion, plane changes, and feedrate changes. This information is often needed when using an off-line CAM or CAD/CAM system to create NC part programs.

Some G Codes are on by default at the start of a program until another G Code cancels it. Defaults are identified below.

Any unrecognized code will stop execution and display an error message. More than one G code can be specified in the same block. If more than one is from the same group, the last G code entered is active.

G Code Groups

The G codes are grouped by function. These groups are listed in the following table:

<table>
<thead>
<tr>
<th>Group</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>One-Shot</td>
</tr>
<tr>
<td>01</td>
<td>Interpolation</td>
</tr>
<tr>
<td>03</td>
<td>Dimension</td>
</tr>
<tr>
<td>05</td>
<td>Feed</td>
</tr>
<tr>
<td>06</td>
<td>Measurement</td>
</tr>
<tr>
<td>07</td>
<td>Cutter Compensation</td>
</tr>
<tr>
<td>09</td>
<td>Canned Cycles</td>
</tr>
<tr>
<td>10</td>
<td>Return from Canned Cycles</td>
</tr>
<tr>
<td>14</td>
<td>Coordinate System Selection</td>
</tr>
<tr>
<td>20</td>
<td>Speed</td>
</tr>
<tr>
<td>21</td>
<td>Programming Mode</td>
</tr>
<tr>
<td>30</td>
<td>Lathe Axis Map Coordinates</td>
</tr>
</tbody>
</table>

Table 2–1. G Code Groups

Specifying a Group 01 (Interpolation) G code in a canned cycle automatically enters the G80 condition (Cancel Drill Cycle). Conversely, a group 01 G code is not affected by the canned cycle G codes.
## G Code Table

This table lists the G codes, identifies the defaults in bold text, identifies groups, lists Modal or Non-modal types, and describes the G codes’ functions.

<table>
<thead>
<tr>
<th>G Code</th>
<th>Group</th>
<th>Type</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>01</td>
<td>Modal</td>
<td>Linear Motion at Rapid (default)</td>
</tr>
<tr>
<td>G01</td>
<td></td>
<td></td>
<td>Linear Motion at Feed</td>
</tr>
<tr>
<td>G02</td>
<td></td>
<td></td>
<td>Clockwise Circular Motion at Feed</td>
</tr>
<tr>
<td>G03</td>
<td></td>
<td></td>
<td>Counterclockwise Circular Motion at Feed</td>
</tr>
<tr>
<td>G04</td>
<td>00</td>
<td>Non-modal</td>
<td>Dwell</td>
</tr>
<tr>
<td>G07</td>
<td>21</td>
<td>Modal</td>
<td>Radius Programming</td>
</tr>
<tr>
<td>G08</td>
<td></td>
<td></td>
<td>Diameter Programming (default)</td>
</tr>
<tr>
<td>G09</td>
<td>00</td>
<td>Non-modal</td>
<td>Exact Stop</td>
</tr>
<tr>
<td>G20</td>
<td>06</td>
<td>Modal</td>
<td>Inch Mode (default)</td>
</tr>
<tr>
<td>G21</td>
<td></td>
<td></td>
<td>Millimeter Mode</td>
</tr>
<tr>
<td>G33</td>
<td>00</td>
<td>Non-modal</td>
<td>Threading</td>
</tr>
<tr>
<td>G40</td>
<td>07</td>
<td>Modal</td>
<td>Cutter Radius Compensation Off (default)</td>
</tr>
<tr>
<td>G41</td>
<td></td>
<td></td>
<td>Cutter Radius Compensation Left</td>
</tr>
<tr>
<td>G42</td>
<td></td>
<td></td>
<td>Cutter Radius Compensation Right</td>
</tr>
<tr>
<td>G53</td>
<td>14</td>
<td>Non-modal</td>
<td>Program Machine Coordinates</td>
</tr>
<tr>
<td>G59</td>
<td></td>
<td></td>
<td>Cancel Work Coordinate Offsets (default)</td>
</tr>
<tr>
<td>G73</td>
<td>09</td>
<td>Modal</td>
<td>Peck Drill with Chip Break Drill Cycle</td>
</tr>
<tr>
<td>G74</td>
<td></td>
<td></td>
<td>Left Hand Tapping</td>
</tr>
<tr>
<td>G78</td>
<td>00</td>
<td>Non-modal</td>
<td>Threading Cycle</td>
</tr>
<tr>
<td>G80</td>
<td>09</td>
<td>Modal</td>
<td>Cancel Drill Cycle (default)</td>
</tr>
<tr>
<td>G81</td>
<td></td>
<td></td>
<td>Drill Cycle</td>
</tr>
<tr>
<td>G82</td>
<td></td>
<td></td>
<td>Drill Cycle with Dwell</td>
</tr>
<tr>
<td>G83</td>
<td></td>
<td></td>
<td>Peck Drill Cycle</td>
</tr>
<tr>
<td>G84</td>
<td></td>
<td></td>
<td>Right Hand Tapping</td>
</tr>
<tr>
<td>G Code (Continued)</td>
<td>Group</td>
<td>Type</td>
<td>Function</td>
</tr>
<tr>
<td>-------------------</td>
<td>-------</td>
<td>------</td>
<td>----------</td>
</tr>
<tr>
<td>G90</td>
<td>03</td>
<td>Modal</td>
<td>Absolute Programming (default)</td>
</tr>
<tr>
<td>G91</td>
<td></td>
<td></td>
<td>Incremental Programming</td>
</tr>
<tr>
<td>G92</td>
<td>14</td>
<td>Modal</td>
<td>Work Coordinate Offsets or Spindle Max Speed</td>
</tr>
<tr>
<td>G94</td>
<td>05</td>
<td>Modal</td>
<td>Feed per Minute (default)</td>
</tr>
<tr>
<td>G95</td>
<td></td>
<td></td>
<td>Feed per Revolution</td>
</tr>
<tr>
<td>G96</td>
<td>20</td>
<td>Modal</td>
<td>Constant Surface Speed (CSS)</td>
</tr>
<tr>
<td>G97</td>
<td></td>
<td></td>
<td>Direct Spindle Speed (default)</td>
</tr>
<tr>
<td>G98</td>
<td>10</td>
<td>Modal</td>
<td>Drill Cycle Initial Level Return (default)</td>
</tr>
<tr>
<td>G99</td>
<td></td>
<td></td>
<td>Drill Cycle R Plane Return</td>
</tr>
<tr>
<td>G101</td>
<td>30</td>
<td>Modal</td>
<td>Axial Surface of the Part - Main Spindle without Linear Y</td>
</tr>
<tr>
<td>G102</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Main Spindle without Linear Y</td>
</tr>
<tr>
<td>G103</td>
<td></td>
<td>Modal</td>
<td>Axial Surface of the Part - Main Spindle with Linear Y</td>
</tr>
<tr>
<td>G104</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Main Spindle with Linear Y</td>
</tr>
<tr>
<td>G105</td>
<td></td>
<td>Modal</td>
<td>Axial Surface of the Part - Sub-spindle without Linear Y</td>
</tr>
<tr>
<td>G106</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Sub-spindle without Linear Y</td>
</tr>
<tr>
<td>G107</td>
<td></td>
<td>Modal</td>
<td>Axial Surface of the Part - Sub-spindle with Linear Y</td>
</tr>
<tr>
<td>G108</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Sub-spindle with Linear Y</td>
</tr>
<tr>
<td>G109</td>
<td></td>
<td>Modal</td>
<td>Turning / Lathe Mode (default)</td>
</tr>
</tbody>
</table>

Table 2–2. G Codes
G00 - Linear Motion at Rapid (default)

G00 sets rapid linear motion as the machine's modal condition. Any block that executes while G00 is modal moves the axes at rapid feedrate to the programmed endpoint. The feedrate of each axis is controlled to ensure that all programmed axes reach their respective endpoints simultaneously.

Default—Yes
Modal—Yes
Cancels G01, G02, or G03

G01 - Linear Motion at Feed

G01 sets feed linear motion as the machine's modal condition. Any block that executes while G01 is modal moves the programmed axes directly to their programmed endpoint at a specified feedrate using the F command. G01 requires programming X or Z or both and you may include F if not specified previously. The vector feedrate of each axis is controlled to ensure that all axes reach their endpoints simultaneously.

Default—No
Modal—Yes
Cancels G00, G02, or G03

Examples—G01

```
N5  G90  G07  G01  X0.25  Z-0.5  F ___   (G90 = Absolute;  G07 = Radius)
N7  G90  G08  G01  X1.00  Z-0.5  F ___   (G90 = Absolute;  G08 = Diameter)
N5  G91  G07  G01  X0.25  Z-0.5  F ___   (G91 = Incremental; G07 = Radius)
N7  G91  G08  G01  X0.50  Z-0.5  F ___   (G91 = Incremental; G08 = Diameter)
```
G02 - Clockwise Circular Motion at Feed

G02 sets clockwise circular interpolation as the modal condition. Either program the arc's endpoint and center coordinates, or program the arc's endpoint and radius. Refer to Example—Program Arc with Endpoint and Center Coordinates, on page 2 - 8 and Example—Program Arc with Endpoint and Radius, on page 2 - 9 for programming examples.

Default—No
Modal—Yes
Cancels G00, G01, or G03

The following figure illustrates G02 (Clockwise) circular motion at feed:

![Figure 2–1. G02 (Clockwise) circular motion at feed](image)
G03 - Counterclockwise Circular Motion at Feed

G03 sets counterclockwise circular interpolation as the modal condition. Either program the arc's endpoint and center coordinates, or program the arc's endpoint and radius. Refer to Example—Program Arc with Endpoint and Center Coordinates, on page 2 - 8 and Example—Program Arc with Endpoint and Radius, on page 2 - 9 for programming examples.

Default—No

Modal—Yes

Cancels G00, G01, or G02

Example—Program Arc with Endpoint and Center Coordinates

This method requires you to define the arc's endpoint and center coordinates. Dimensions for the arc's endpoint (X, Z) may be defined with absolute or incremental dimensions, depending on the modal G90/G91 condition. All arc center dimensions (I, K) must be defined using signed incremental dimensions (the distance from the arc's start point to the arc's center).

If G07 Radius Programming is active, the arc's endpoint (X) and center (I) coordinates must be programmed as nominal values.

If G08 Diameter Programming is active the arc endpoint (X) must be programmed as a diameter value, but the arc center (I) must be programmed as the nominal value.

Example—G03 Arc

The following example part program shows arcs programmed with X, Z, I, and K.

N2 G90 G07 G03 X0.8 Z-0.6 I0 K-0.6 (G90 = Absolute; G07 = Radius)
N7 G90 G08 G03 X1.6 Z-0.6 I0 K-0.6 (G90 = Absolute; G08 = Diameter)
N2 G91 G07 G02 X0.8 Z-0.6 I0 K-0.6 (G91 = Incremental; G07 = Radius)
N7 G91 G08 G02 X1.6 Z-0.6 I0 K-0.6 (G91 = Incremental; G08 = Diameter)
Example—Program Arc with Endpoint and Radius

The second method of programming a G02 or G03 block is to define the arc's endpoint and radius. There are two possible arcs when you program X, Z, and R. The sign of the R radius value determines whether you want the arc that spans less than 180° (by programming R as a positive value), or the arc that spans more than 180° (by programming R as a negative value).

N2 G90 G07 G02 X__ Z__ R+__ (G90 = Absolute; G07 = Radius; G02 = CW)
N2 G90 G07 G03 X__ Z__ R-__ (G90 = Absolute; G07 = Radius; G03 = CCW)
N7 G90 G08 G02 X__ Z__ R+__ (G90 = Absolute; G08 = Diameter; G02 = CW)
N7 G90 G08 G03 X__ Z__ R-__ (G90 = Absolute; G08 = Diameter; G03 = CCW)
N2 G91 G07 G02 X__ Z__ R+__ (G91 = Incremental; G07 = Radius G02 = CW)
N2 G91 G07 G03 X__ Z__ R-__ (G91 = Incremental; G07 = Radius; G03 = CCW)
N7 G91 G08 G02 X__ Z__ R+__ (G91 = Incremental; G08 = Diameter G02 = CW)
N7 G91 G08 G03 X__ Z__ R-__ (G91 = Incremental; G08 = Diameter; G03 = CCW)
G04 - Dwell

G04 programs a pause in part program execution. F is the only entry allowed in a **G04** block. F Codes set the modal feedrate for cutting moves.

Default—No

Modal—No

Cancels—None

<table>
<thead>
<tr>
<th>Active G Code</th>
<th>F in the G04 Block Programs</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>G94</strong> Feed Per Minute</td>
<td>The number of seconds to dwell. For example, the block <strong>G04</strong> F2.1 pauses for 2.1 seconds before it executes the next block of code.</td>
</tr>
<tr>
<td>Refer to <strong>G94 - Feed per Minute (default)</strong>, on page 2 - 45 for information about G93 and G94.</td>
<td></td>
</tr>
<tr>
<td><strong>G95</strong> Feed Per Revolution</td>
<td>The number of spindle revolutions to dwell. For example, the block <strong>G04</strong> F10 pauses for 10 spindle revolutions before it executes the next block of code.</td>
</tr>
<tr>
<td>Refer to <strong>G95 - Feed per Revolution</strong>, on page 2 - 45 for information about G95.</td>
<td></td>
</tr>
</tbody>
</table>

*Table 2–3. G04 Dwell Examples*

G06 - Probe/Block Skip

G06 arms the Probe mechanism and provides a means for stopping the axis, for instance, when the torque is detected to be too high. Remaining Motion programmed after a high torque detection is deleted from memory. Please refer to **M186 - Activate Torque Monitoring for W**, on page 3 - 11 for details.
G07 - Radius Programming

You can program X axis dimensions using the radius or diameter of the part. Program G07 when you wish to program radius dimensions.

Default—No
Modal—Yes
Cancels—G08

Format

G07 X ____ (X is interpreted as a radius value.)

This mode will affect live-tool (milling) when running G codes. When G07 is active, live-tool programming is the same as programming a 3-axis milling machine.

The table below provides information about how the modal setting affects part programs.

<table>
<thead>
<tr>
<th>Programmed Value</th>
<th>G07 Modal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Endpoint of linear move</td>
<td>X use radius</td>
</tr>
<tr>
<td>Endpoint of circular move</td>
<td>X use radius</td>
</tr>
<tr>
<td>Center and radius of circular move</td>
<td>IKR use radius</td>
</tr>
<tr>
<td>X axis infeed distances</td>
<td>infeed distance use radius</td>
</tr>
</tbody>
</table>

Table 2–4. Effects of Using G07

G07 Radius Programming Example

The following example part program shows arcs programmed with X, Z, I, and K.

G90 G07 X0.8 Z-0.6 I0 K-0.6 (G90 = Absolute; G07 = Radius)
G91 G07 X0.8 Z-0.6 I0 K-0.6 (G91 = Incremental; G07 = Radius)

G07, Radius programming, establishes a modal setting that interprets X-axis dimensions in radius values. The following figure illustrates radius programming:

Figure 2–4. G07 Radius Programming
Programming Display Mode

G07 does not affect the Programming Display Mode. The correct units are interpreted from the part program.

⇒ The Programming Display Mode appears on the screen’s Status bar. Select the D (diameter) on the screen to toggle the mode to R (radius). The display toggles between choices.

Figure 2–5. WinMax Status bar
G08 - Diameter (default) Programming

You can program X axis dimensions using the radius or diameter of the part. Program G08 when you wish to program diameter dimensions.

Default—Yes
Modal—Yes
Cancels—G07

This mode will affect live-tool (milling) when running G codes. When G07 is active, live-tool programming is the same as programming a 3-axis milling machine.

The table below provides information about how the modal setting affects part programs.

<table>
<thead>
<tr>
<th>Programmed Value</th>
<th>G08 Modal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Endpoint of linear move</td>
<td>X use diameter</td>
</tr>
<tr>
<td>Endpoint of circular move</td>
<td>X use diameter</td>
</tr>
<tr>
<td>Center and radius of circular move</td>
<td>IKR use radius</td>
</tr>
<tr>
<td>X axis infeed distances</td>
<td>infeed distance use radius</td>
</tr>
</tbody>
</table>

*Table 2–5. Effects of Using G08*

G08 Diameter Programming Example

The following example part program shows arcs programmed with X, Z, I, and K.

G90 G08 X1.6 Z-0.6 I0 K-0.6 (G90 = Absolute; G08 = Diameter)
G91 G08 X1.6 Z-0.6 I0 K-0.6 (G91 = Incremental; G08 = Diameter)

G08, Diameter programming, specifies that all X-axis positions are interpreted in diameter values. The following figure illustrates diameter programming:

*Figure 2–6. G08 Diameter Programming*

The arc center dimension (K) stays the same for both commands, but the X-axis dimension (X) changes. The program may switch between G07 Radius and G08 Diameter at any time.
Programming Display Mode

G08 does not affect the Programming Display Mode. The correct units are interpreted from the part program.

Figure 2–7. WinMax Status bar
G09 - Exact Stop

G09 causes the axes to decelerate to zero velocity at the end of a block. Since G09 is non-modal, you must program it in every block that requires the tool to come to a complete stop.

G09 is not used in conjunction with G00, G01, G02, or G03 commands.

Default—No
Modal—No
Cancels—None
Cancelled by—None

G20 - Inch Mode (default)

This code sets Inch mode. Program this command in the first block of each part program that uses inches for the dimension.

Default—Yes
Modal—Yes
Cancels—G21

The default format for Inch mode for different types of blocks is shown in the table below:

<table>
<thead>
<tr>
<th>Blocks</th>
<th>Default</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometric Entries X, Z, I, K, R</td>
<td>3.4</td>
<td>-845.0000 to 845.0000</td>
</tr>
<tr>
<td>Feedrate (Feed/min) and (Feed/rev) F</td>
<td>3.4</td>
<td>000.0001 to 845.0000</td>
</tr>
<tr>
<td>Thread Lead (distance/rev) K</td>
<td>2.6</td>
<td>00.000001 to 10.000000</td>
</tr>
</tbody>
</table>

1. The default format 3.4 represents 3 places to the left of the decimal point and 4 places to the right. The default format 2.6 represents 2 places to the left of the decimal point and 6 places to the right.

Table 2–6. Inch Mode formats

Unit Display Mode

G20 does not affect the Unit Display Mode as viewed in the position or offset screens. The correct units are interpreted from the part program.
G21 - Millimeter Mode

This code sets Millimeter mode. Program this command in the first block of each part program that uses millimeters for the dimension.

Default—No

Modal—Yes

Cancels—G20

The default format for Millimeter mode for different types of blocks is shown in the table below:

<table>
<thead>
<tr>
<th>Blocks</th>
<th>Default¹</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometric Entries X, Z, I, K, R</td>
<td>5.3</td>
<td>-21463.000 to 21463.000</td>
</tr>
<tr>
<td>Feedrate (Feed/min) and (Feed/rev) F</td>
<td>5.3</td>
<td>0000.001 to 21463.000</td>
</tr>
<tr>
<td>Thread Lead (distance/rev) K</td>
<td>3.5</td>
<td>000.00001 to 10.000000</td>
</tr>
</tbody>
</table>

¹ The default format 5.3 represents 5 places to the left of the decimal point and 3 places to the right. The default format 3.5 represents 3 places to the left of the decimal point and 5 places to the right.

Table 2–7. Millimeter Mode formats

Unit Display Mode

G21 does not affect the Unit Display Mode as viewed in the position or offset screens. The correct units are interpreted from the part program.
G33 - Threading

This code allows you to program a simple threading cycle. Refer to G78 - Threading Cycle, on page 2 - 25 for information about programming more the complex, multi-pass Threading Cycle for OD and ID threads,

Default—No

Modal—No

Cancels—None

Example—G33

To program a threading move, X and Z and either I or K are required. P (the thread start angle) and F (feedrate) may be specified.

1. Program the thread's endpoint (using Z only, X only, or a combination of X and Z for a taper thread).
2. Program the pitch of the thread using either of these codes:
   - K (the pitch along the Z axis)
   - I (the pitch along the X axis)

If you are programming a tapered thread (both X and Z axes move) you must program the lead (I or K) for the axis that has the longest move.

G0 X3.5 Z4 (Position for thread)
G33 Z1 K0.2 (Cut thread)
G1 X3.6 (Pullout of thread)

Example—G33 with P Value

You can also start a thread at a given angle. The angle value may range from -360.0000 to 360.0000. A zero angle starts the thread at the index mark of the spindle encoder, and is the default. The P value is not modal and must appear in the same block as the G33.

The thread start angle is specified by a P value in a G33 block, such as:

G0 X3.5 Z4 (Position for thread)
G33 Z1 K0.2 P180 (Cut thread using 180° thread start angle)
G1 X3.6 (Pullout of thread)
G40 - Cutter Radius Compensation Off (default)

Cutter compensation may be used for two purposes. First, it may be used when the dimensions in the program and the part surface are the same. The system calculates the proper tool path by using the part surface and the tool diameter information.

Second, cutter compensation corrects the difference between the diameters of the tool specified and the tool actually used to cut the part. This situation often occurs when the program originates from an off-line device. Note that the coordinates of those programs are usually tool center line data.

Cutter compensation is based on the direction of travel of the tool. To determine which type of cutter compensation to use, look at the part as if you are moving around the part always keeping the tool ahead of you. Then it becomes obvious whether the tool needs to be on the right or the left of the programmed line or the boundary of the part.

G40 turns off cutter compensation. When G40 is active, the tool center will move along the path defined in the part program. Refer to Figure 2–8. Cutter Compensation Right Turned Off (G40), on page 2 - 21 for an illustration of the tool path with G40. Refer to Examples—Cutter Compensation, on page 2 - 20 for programming examples.

Default—Yes
Modal—Yes
Cancels G41, G42

G00 or G01 must be selected in order for this command to cancel the offset compensation. Each axis moves straight (G01) or at rapid traverse (G00) from the point of the old vector at the start point toward the end point. The machine should be in G40 mode before the end of a program. Otherwise, when the program ends in the offset mode, positioning cannot be made to the terminal point of the program, and the tool position will be separated from the terminal position by the vector value.
G41 - Cutter Radius Compensation Left

This code turns cutter compensation on to the left of the programmed part profile. When G41 is active, the tool path is offset by the tool radius value stored in the tool table. The offset will be to the left side of the part profile. Refer to Examples—Cutter Compensation, on page 2 - 20 for programming examples.

Default—No

Modal—Yes

Cancels G40, G42

G00, G01, G02, or G03 must be specified.

G42 - Cutter Radius Compensation Right

This code turns cutter compensation on to the right of the programmed part profile. When G42 is active, the tool path is offset by the tool radius value stored in the tool table. The offset will be to the right side of the part profile. Refer to Examples—Cutter Compensation, on page 2 - 20 for programming examples.

Default—No

Modal—Yes

Cancels G40, G41
Examples—Cutter Compensation

The following examples use the part program below. One example turns cutter compensation on, and the other example turns cutter compensation off. Figure 2–8. Cutter Compensation Right Turned Off (G40), on page 2 - 21 shows the tool path with cutter compensation turned off (G40). Figure 2–9. Cutter Compensation Turned On (G42, on page 2 - 21 shows the tool path with G42 (Cutter Compensation Right) turned on.

%  
(MSG, LATHE DOC SAMPLE, ABS G42, TOOL RAD 0.1)  
G90 G40 G94 T0000 (ABSOLUTE MODE,CUTTER COMPENSATION OFF,OFFSET 0)  
G07 G00 X2.5 T0101 (RADIUS PROGRAMMING,)  
E1 (CALL WORK OFFSET)  
(Set to 10 for example)  
(RAPID TO X, TOOL 1 OFFSET 1)  
Z0 (RAPID TO Z)  
X3.05 (RAPID TO X)  
G96 R.5 S350 M03 (CSS, AXIS AT 0.5IN.,)  
(350 SFM AND CLOCKWISE)  
G01 G95 F.01 X0 (FACE, IPR 0.01)  
G00 Z.5 (RAPID TO Z)  
G00 X.5 Z.1 (RAPID TO XZ)  
G42 (CUTTER COMPENSATION RIGHT)  
Z.05  
G01 Z0  
X1 Z-1 (TAPER)  
X1.7 (FACE)  
Z-2.7 (TURN)  
X1.9 (FACE)  
Z-3 (TURN)  
G00 X2.5 (CUTTER COMPENSATION OFF, RAPID TO X)  
G40 (CUTTER COMPENSATION OFF)  
Z.5 (RAPID TO Z)  
END OF PROGRAM, REWIND)  
M30
**Cutter Compensation Turned Off (G40)**

The following figure illustrates the tool path with G40. Notice that the tool travels very close to the part. The nose radius of the tool programmed in Tool Setup determines the space between the part and the tool.

**Figure 2–8. Cutter Compensation Right Turned Off (G40)**

**Cutter Compensation Right Turned On (G42)**

The following figure illustrates the tool path with G42 programmed.

**Figure 2–9. Cutter Compensation Turned On (G42)**
G53 - Program Machine Coordinates

G53 assumes absolute positioning in BNC. G91 has no effect. This code causes the part program to ignore the following offsets for one block so you can program the axes to a position that you define using machine coordinates:

- active work coordinate offsets (G92)
- active tool offset
- active fixture offset

Default—No
Modal—No
Cancels G40, G42

Before you program G53 in a block, Cutter radius compensation (G40) must be cancelled.

G59 - Cancel Work Coordinate Offsets (default)

This code cancels any active offset set by G92 Work Coordinate Offsets.

Default—Yes
Modal—Yes
Cancels G92
G73 - Peck Drill with Chip Break Drill Cycle

G73 sets chip break drilling as the modal drill cycle.

Default—Yes

Modal—Yes

Cancels G74, G80, G81, G82, G83, G84, G98, G99

The tool motion for the chip break drill cycle is illustrated in Figure 2–10. Tool Motion for G73 Chip Break Drill Cycle, on page 2 - 23. This cycle requires programming X or Z, R (Retract Clearance Plane), and Q. G73 causes the tool to feed by the amount programmed in Q, then rapid back an incremental amount, A (2.5 mm). Next the tool will rapid to a distance slightly above (2.5 mm) the last infeed depth, then feed another Q distance into the part. This sequence repeats until the tool reaches the programmed Z depth. Program G99 (Drill Cycle R Plane Return) to move the tool back to the R plane.

The following figure illustrates tool motion for G73 chip break drill cycle. The short broken lines represent rapid moves; the long broken lines represent feed moves.

Example—G73

G90

G73 X-0.1 Z-1.2 R0.1 Q.5 G99
G74 - Left Hand Tapping

G74 executes the same as G84 Right Hand Tapping except the spindle directions are reversed.

Default—No
Modal—No
M74 is required for the optional Rigid Tapping.
Canceled—None

The tool motion for this cycle is described below. The feedrate and spindle override controls are forced to 100% are disabled while this cycle is active.

To stop tool motion when the tap is in the bore, either press the Emergency Stop button or use Retract.

The control will not respond to a motion-stop command until the tool returns to the reference plane.

The drawing below shows left-hand tapping tool motion:

1. Tool rapids to R plane.
2. Tool feeds to the programmed depth.
3. Spindle reverses direction (CCW).
4. If G94 is active, the axis will dwell for P seconds; if G95 is active, the axis will dwell for P revolutions.
5. Tool feeds to R plane.

Figure 2–11. G74 Left Hand Tapping
G78 - Threading Cycle

This feature provides a multi-pass Threading Cycle for OD (outside diameter) and ID (inside diameter) threads.

Default—No
Modal—No
Cancels—None

Prior to executing this cycle, the spindle must be running at speed in G97 mode and the tool must be positioned so it can rapid directly to the thread's X and Z clearance position.

This cycle supports taper threads, face threads, lead in and lead out angles, multiple spring passes, and various methods of specifying depth of cut. Refer to G33 - Threading, on page 2 - 17 for simple Threading programming information.

Threading Cycle Parameters ........................................ 2 - 26
Turn and Face Threading ........................................... 2 - 28
Turn Threading - Inner Diameter and Outer Diameter .......... 2 - 29
Clearance, Lead In and Lead Out Parameters .................. 2 - 29
Example—OD Thread with Offsets U and W .................... 2 - 29
Example—OD Thread with Chase in (A) and Chase out (C) .... 2 - 29
Multiple Start Parameters ......................................... 2 - 30
Tapered Thread ...................................................... 2 - 31
Cutting Parameters .................................................. 2 - 31
Sample Thread Part Programs ................................. 2 - 33
The following table contains definitions for the types of threading cycle parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Parameter Type</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Lead In</td>
<td>Lead in Angle. This is the infeed (chase in) angle with respect to the X axis. Default 0°</td>
</tr>
<tr>
<td>B</td>
<td>Tapered Thread</td>
<td>Taper angle relative to the Z axis. -45° &lt;= B &lt;= 45°</td>
</tr>
<tr>
<td>C</td>
<td>Lead Out</td>
<td>Lead out Angle. Pullout (chase out) angle with respect to X axis. Default 0°.</td>
</tr>
<tr>
<td>D</td>
<td>Cutting</td>
<td>Start rough depth. <strong>Required.</strong></td>
</tr>
<tr>
<td>E</td>
<td>Cutting</td>
<td>Final rough depth. If E is not programmed, the value defaults to D unless V is programmed.</td>
</tr>
<tr>
<td>F</td>
<td>Turn</td>
<td>Pitch. Inches/millimeters per revolution.</td>
</tr>
<tr>
<td>F-</td>
<td>Turn</td>
<td>Lead. Threads per inch.</td>
</tr>
<tr>
<td>H</td>
<td>Length</td>
<td>The Z end position at the major radius.</td>
</tr>
<tr>
<td>I</td>
<td>Face</td>
<td>X position where thread starts.</td>
</tr>
<tr>
<td>J</td>
<td>Turn</td>
<td>Total depth of cut from the major radius to the minor radius. End Depth = Z, Start Depth - Z+J.</td>
</tr>
<tr>
<td>K</td>
<td>Turn</td>
<td>Z position where thread starts.</td>
</tr>
<tr>
<td>L</td>
<td>Cutting</td>
<td>Depth of cut per finish pass. If L is not programmed, then there are no finish passes. If P is nonzero and L equals zero, then the number of spring passes equals P + R.</td>
</tr>
<tr>
<td>M=0</td>
<td>Face</td>
<td>Turn Thread (default).</td>
</tr>
<tr>
<td>M=1</td>
<td>Face</td>
<td>Face Thread. No M value or M=0 activates the turn thread.</td>
</tr>
<tr>
<td>P</td>
<td>Cutting</td>
<td>Number of finish passes implemented before starting spring passes. If P is not programmed and L is nonzero, set L to 1. Otherwise, no finish passes will occur.</td>
</tr>
<tr>
<td>Q</td>
<td>Multiple Start</td>
<td>Thread cut start angle. The default is 0°.</td>
</tr>
<tr>
<td>+Q</td>
<td>Multiple Start</td>
<td>Multiple Start number. The angle per thread start equals 360/Q.</td>
</tr>
<tr>
<td>R</td>
<td>Cutting</td>
<td>Number of spring passes at 0 depth per pass. A spring pass is a repetition made by the finish tool at the final finish depth.</td>
</tr>
</tbody>
</table>
### Threading Cycle Parameters (continued)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Parameter Type</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>S</td>
<td>Face</td>
<td>The X end position at Z Start Depth.</td>
</tr>
<tr>
<td>T</td>
<td>Length</td>
<td>Length of thread between the infeed move and pullout move.</td>
</tr>
<tr>
<td>U</td>
<td>Clearance</td>
<td>The X return clearance. Measured from the major diameter. Default 0.1&quot; or 2.54 mm.</td>
</tr>
<tr>
<td>V</td>
<td>Cutting</td>
<td>When set to 1, the depth per pass results in constant volume removal per pass. When programming constant volume, do not program a final depth parameter (E) in the same block.</td>
</tr>
<tr>
<td>W</td>
<td>Clearance</td>
<td>Z clearance. Measured from the start of thread. Default 0.1&quot; or 2.54 mm.</td>
</tr>
</tbody>
</table>
| X         | Turn/Face      | **Turn**: Radius at the start of the final thread cut at the end of the infeed motion. Effective final pass radius = X or minR.  
**Face**: The X end position at Z Clearance (Z + J + W). The interpretation of this parameter is not affected by the Radius/Diameter Mode state (G07 or G08). |
| Z         | Length         | The Z end position at the start of the final thread pass at the end of the infeed motion, or X Clearance (T + Z = K). |

*Table 2–8. Thread Parameters*
Turn and Face Threading

Turn threading cuts the thread horizontally on the stock. Face threading cuts the thread vertically on the stock. The type of threading is determined by the M (Face) parameter. If no M is specified, M0 is the default.

The following entries are required to define a turn thread.

![Thread Parameters Diagram](image)

*Figure 2–12. Thread Parameters*

The following equivalents exist between the turn and face thread macro arguments.

<table>
<thead>
<tr>
<th>Thread Axis</th>
<th>Turn Thread</th>
<th>Face Thread</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thread Start</td>
<td>K</td>
<td>I</td>
</tr>
<tr>
<td>Thread Normal End</td>
<td>Z</td>
<td>X</td>
</tr>
<tr>
<td>Optional Thread End</td>
<td>H</td>
<td>S</td>
</tr>
<tr>
<td>Optional Thread End #2</td>
<td>Thread Start + T</td>
<td>Thread Start + Td</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Depth Axis</th>
<th>Turn Thread</th>
<th>Face Thread</th>
</tr>
</thead>
<tbody>
<tr>
<td>Last Pass Depth</td>
<td>X</td>
<td>Z</td>
</tr>
<tr>
<td>Initial Depth</td>
<td>Last Depth + J</td>
<td>Last Depth + J</td>
</tr>
</tbody>
</table>

*Table 2–9. Threading Equivalents between Turn and Face Thread Macros*

- The thread start, last pass depth, and total depth arguments (K, X, and J for turn thread; I, Z, and J for face thread) must be programmed for each thread cycle.
- Exactly one of the three thread arguments (Z, H, or T for turn thread, X, S, or T for face thread) must be programmed for the given type of thread cycle (turn or face).
Turn Threading - Inner Diameter and Outer Diameter

The type of thread cut (Inner or Outer Diameter) is defined by the signs of the \( X \) and \( J \) parameters. The \( X \) parameter specifies the radius or diameter at the start of the final thread pass. The \( J \) parameter specifies the total depth of the thread.

- If \( X \) and \( J \) are of the same sign (both positive or both negative), an OD (outer diameter or external) thread cut will be implemented.
- If \( X \) and \( J \) are of the opposite sign (one positive and one negative), an ID (inner diameter or internal) thread cut will be implemented.

If the sign of \( X+J \) is not the same as the sign of \( X \), the macro is aborted. The \( X \)-axis is not allowed to cross the spindle centerline when cycling a threading macro.

Clearance, Lead In and Lead Out Parameters

These parameters have default values and may not need to be programmed.

Example—OD Thread with Offsets U and W

The following figure illustrates the use of \( U \) (X return clearance) and \( W \) (Z clearance) parameters:

![Figure 2–13. OD Thread with Offsets U and W Example](image)

Example—OD Thread with Chase in (A) and Chase out (C)

The following figure illustrates the use of \( A \) (lead in angle) and \( C \) (lead out angle) parameters:

![Figure 2–14. OD Thread with Chase in (A) and Chase out (C) Example](image)
Multiple Start Parameters

Multiple Start parameters allow you to define multiple starting points at different angles from 0 to 359.999° around the thread. This parameter is optional.

Example—Multiple Starting Points at Different Angles

The following figure illustrates programming multiple starting points at different angles: 120°, 170°, and 190°:

G78 X.9 J.2 Z1.2 K2 F.15 D.04 E.03 P2 L.02 R3 Q-120
G78 X.9 J.2 Z1.2 K2 F.15 D.04 E.03 P2 L.02 R3 Q-170
G78 X.9 J.2 Z1.2 K2 F.15 D.04 E.03 P2 L.02 R3 Q-180

Example—Multiple Starting Points at Evenly Spaced Angles

The following figure illustrates programming multiple starting points at 3 evenly spaced angles: 0°, 120°, and 240°. Q3 represents 360° divided by 3:

G78 X.9 J.2 Z1.2 K2 F.15 D.04 E.03 P2 L.02 R3 Q3

Figure 2–15. OD with multiple starts at 120°, 170°, and 180° Example

Figure 2–16. OD with 3 Evenly Spaced Threads Example
Tapered Thread

For a tapered thread, you must program B to specify the taper angle with respect to the Z axis for turn thread, or X axis for face thread.

G78 X0.9 J0.6 B14.7 Z1.2 K4.0 F.375 D0.1 E0.08 P2.0 L0.02 R3.0 U0.1 W0.1

Refer to Tapered OD Thread, on page 2-35 for more information about the parameters shown in this example code.

Cutting Parameters

The following cutting parameters affect how the thread will be cut. You can specify the start (D) and final rough depth (E), number of finish passes (L), or the number of spring passes. A spring pass is a repetition made by the finish tool at the final finish depth.

Example—No Finish Pass Programmed

Only the start rough depth (D) and final rough depth (E) are programmed. The tool moves progressively deeper from D to E through a number of control-calculated passes.

G78 X1.05 J.45 D.1 E.05

Figure 2–18. No Programmed Finish Pass Example
Example—Three Finished Passes Programmed

Three finish passes (L) are programmed along with the D and E values. The tool moves progressively deeper from D to E through a number of control-calculated passes, then moves in for three consecutive finishing passes.

\[
G78 \ X1.05 \ J.45 \ D.1 \ E.05 \ L.01 \ P3
\]

*Figure 2–19. Three Programmed Finished Passes Example*
Sample Thread Part Programs

The following sample code shows four types of threads:

Straight OD Thread ................................................................. 2 - 33
Straight OD Thread with Lead In/Lead Out Angles ................. 2 - 34
Tapered OD Thread ................................................................. 2 - 35
Straight ID Thread ................................................................. 2 - 36

Straight OD Thread

This example shows a straight thread cut with:

- Variable depth per pass.
- Two finish passes of 0.02" per finish pass (P, L).
- Three spring passes (R).

Thread cut example - straight thread

G90 S1:M03 S1:120
G0 Z4.1
G78 X0.9 J0.2 Z1.2 K2.0 F0.15 D0.04 E0.03 P2.0 L0.02 R3.0 U0.1 W0.1
M30

Figure 2–20. Straight OD Thread Example
Straight OD Thread with Lead In/Lead Out Angles

This example shows a straight thread cut with:

- Lead In angle (A) of 30°.
- Lead Out angle (C) of 5°.
- Two finish passes of 0.02" per finish pass (P, L).
- Three spring passes (R).

Thread cut example—straight thread with lead in/lead out angles

```
G90 M03 S120
G00 X4.1 Z2.5
G78 X0.9 J0.6 Z1.2 K2.0 A30 C5.0 F0.15 D0.04 E0.03 P2.0 L0.02 R3.0 U0.1 W0.1
M30
```

Figure 2–21. Straight OD Thread with Lead In/Lead Out Angles
**Tapered OD Thread**

This example shows

- A tapered thread cut with a Taper angle (B) of 14.7°.
- Two finish passes of 0.02" per finish pass (P, L).
- Three spring passes (R).

**Thread cut example—tapered thread**

G90 S1:M03 S120  
G00 Z4.1  
G78 X0.9 J0.6 B14.7 Z1.2 K4.0 F.375 D0.1 E0.08 P2.0 L0.02 R3.0 U0.1 W0.1  
M30

*Figure 2–22. Tapered OD Thread Example*
**Straight ID Thread**

This example shows a straight thread ID cut with:

- Variable depth per pass
- Turn thread (M=0)
- Pitch and Lead (F, F-)
- Two finish passes of 0.02" per finish pass (P, L)
- Three spring passes (R)

The figure below illustrates the bottom half of the stock.

**Thread cut example—straight ID thread**

```
G90 S1:M03 S120
G0 Z4.1
G78 X0.9 J-0.2 Z1.2 K2.0 F0.15 D0.04 E0.03 P2.0 L0.02 U0.1 W0.1
M30
```

*Figure 2–23. Straight ID Thread Example*
G80 - Cancel Drill Cycle (default)

G80 cancels any active drilling cycle. When G80 is active, X and Z motion commands move the tool directly to the programmed endpoint without performing a drilling cycle. In addition to turning off the drilling cycle, G80 activates Linear Motion at Rapid and clears the modal drilling parameters such as R, Z, Q, and P.

If the drill cycle depth is not programmed after G80, then the drill cycle depth defaults to the R plane.

Default—Yes
Modal—Yes
G00 cannot be programmed in the same block.

G81 - Drill Cycle

G81 sets drilling as the modal drill cycle.

Default—No
Modal—Yes
Cancels G74, G80, G82, G83, G84, G98, G99

The tool motion for the drill cycle is shown below.

1. Position to XZ
2. Rapid to reference plane
3. Feed to depth
4. Rapid to reference plane

Example—G81 Drill Cycle

G90 (Absolute)
G81 X2 Z-1.2 R.1 F10

Figure 2–24. G81 Drill Cycle
G82 - Drill Cycle with Dwell

G82 sets drilling with dwell as the modal drill cycle.

Default—No

Modal—Yes

Cancels G74, G80, G81, G83, G84, G98, G99

The tool motion for this cycle is the same as the drill cycle except a dwell for $P$ seconds occurs at the bottom of the hole.

1. Position to XZ.
2. Rapid to reference plane.
3. Feed to depth.
4. Dwell for $P$ seconds (2.2 in example) or revolutions (depending on FPM or FPR mode). Refer to Threading Cycle Parameters, on page 2 - 26 for information about P format.
5. Rapid to reference plane.

Example—G82 Drill Cycle with Dwell

G90 \( \text{(Absolute)} \)
G82 X2 Z-1.2 R.1 F10 P2.2 G99

Figure 2–25. G82 Drill Cycle with Dwell
G83 - Peck Drill Cycle

G83 sets peck drilling as the modal drill cycle.

Default—No

Modal—Yes

Cancels G74, G80, G81, G82, G84, G98, G99

The tool motion for the chip break drill cycle is illustrated in Figure 2–26. G83 Peck Drill Cycle, on page 2 - 39. This cycle requires programming X or Z, R (Retract Clearance Plane), and Q. This cycle causes the tool to feed by the amount programmed in Q, then rapid back to the R plane. Next the tool rapid to a distance slightly above (2.5mm) the last infeed depth, then feeds another Q distance into the part. This sequence repeats until the tool reaches the programmed Z depth.

The following figure illustrates tool motion for G83 peck drill cycle. The short broken lines represent rapid moves; the long broken lines represent feed moves.

**Example—G83 Peck Drill Cycle**

G90 (Absolute)

G83 X2.0 Z-1.2 R0.1 Q0.5
G84 - Right Hand Tapping

G84 sets right-hand tapping as the modal drill cycle.

Default—No

Modal—Yes

M74 is required for the optional Rigid Tapping.

Cancels G74, G80, G81, G82, G83, G98, G99

The tool motion for this cycle is described below. The feedrate and spindle override controls are forced to 100% are disabled while this cycle is active.

⚠️ To stop tool motion when the tap is in the bore, either press the Emergency Stop button or use Retract.

The control will not respond to a motion-stop command until the tool returns to the reference plane.

The drawing below shows right-hand tapping tool motion:

1. Tool rapids to R plane.
2. Tool feeds CW to the programmed depth.
4. If **G94** is active, the axis will dwell for P seconds; if **G95** is active, the axis will dwell for P revolutions.
5. Tool feeds to R plane.

![Figure 2–27. G84 Right Hand Tapping](image)
G90 - Absolute Programming (default)

You can use **absolute (G90)** or **incremental (G91)** dimensions directly from the blueprint. Most part programs should begin with a **G90** Absolute command that moves the tool to a fixed position on the machine. Once the start point is defined, you can study the part blueprint and determine whether to use **G90** Absolute or **G91** Incremental dimensions.

G90 establishes a modal setting that tells the control to interpret all XZ endpoint dimensions in absolute coordinates. G90 Absolute Programming specifies that all tool endpoint positions are measured from the current part zero position.

Default—Yes

Modal—Yes

Cancels G91

Cancelled by G91

Most part programs should begin with a **G90** command to establish part zero. This setting remains active until you program **G91** to select incremental dimension programming. A program may switch between **G90** Absolute mode and **G91** Incremental mode at any time.

Absolute dimensions are measured from part zero. The figure shows the path that the tool center will follow.

![Figure 2–28. Absolute Dimensions](image)

Figure 2–28. Absolute Dimensions
Example—G90 Absolute Mode

The following sample NC program is in Absolute mode:

N10  (msg, lathe doc sample, abs G42, tool radius 0.1)
N20  G90 G40 G94 T0000  (absolute mode, cutter compensation off, offset 0)
N30  G07 G00 X2.5 T0101  (radius programming, rapid to X, tool 1 offset 1)
N40  Z3  (rapid to Z)
N60  X.5  (rapid to X)
N70  G96 S350 M03  (CSS, X axis at 0.5in, 350 SFM and clockwise)
N80  G01 G95 F.01 X0  (face, IPR 0.01)
N120  G00 Z4  (rapid to Z)
N125  G42  (cutter compensation right, skip if block del on)
N130  X.5 Z3  (rapid to XZ)
N160  G01 X1 Z2  (taper)
N170  X1.7  (face)
N180  Z0.3  (turn)
N190  X1.9  (face)
N200  Z0  (turn)
N230  G00 G40 X2.5  (cutter compensation off, rapid to X, skip if block del on)
N240  Z3  (rapid to Z)
N900  M30  (end of program, rewind)
G91 - Incremental Programming

G91 establishes a modal setting that tells the control to interpret all XZ endpoint dimensions in incremental coordinates. G91 Incremental Programming specifies that all tool endpoint positions are measured from the tool's position at the start of the motion. This setting remains active until you program G90 to select absolute dimension programming. A program may switch between G90 Absolute and G91 Incremental at any time within a part program.

Default—No

Modal—Yes

Cancels G90

Incremental dimensions specify the distance that the tool must move during each block (i.e. the distance from the start of the move to the end of the move).

Figure 2–29. Incremental Dimensions
Example—G91 Incremental

The following sample NC program changes from G90 to G91:

N10  (msg, lathe doc sample inc G42 tool rad 0.1)
N20  G90  G40  G94  T0000  (absolute mode, cutter compensation off, offset 0)
N30  G07  G00  X2.5  T0101  (radius programming, rapid to X, tool 1 offset 1)
N40  Z3  (rapid to Z)
N60  G91  X-2  (incremental mode, rapid to X)
N70  G96  S350  M03  (CSS, X axis at 0.5in, spindle speed 350 and clockwise)
N80  G01  G95  F.01  X-.5  (face, RPM 0.01)
N100 G00  Z1  (rapid to Z)
N125 G42  (cutter compensation right, skip if block del on)
N130  X.5  Z-1  (rapid to XZ)
N160  G01  X.5  Z-1  (taper)
N170  X.7  (face)
N180  Z-1.7  (turn)
N190  X.2  (face)
N200  Z-.3  (turn)
/N230  G00  G40  X.6  (cutter comp. off, rapid to X, skip if block del on)
N240  Z3  (rapid to Z)
N900  M30  (end of program, rewind)
G92 - Work Coordinate Offsets or Spindle Max Speed

G92 lets you establish the part program coordinates at the current position without generating any tool motion.

Default—No
Modal—Yes
Cancels G59

G92 can also set a maximum spindle speed for constant surface speed mode.
G92 cannot include X and Z coordinates.

Example—G92
G92 S5000 (prevent the spindle from exceeding 5000 rpm)

G94 - Feed per Minute (default)

G94 puts the control into feed per minute mode. When G94 is active, all F feedrate values are interpreted in inch/minute (G20) or millimeter/minute (G21) units. Any dwell command that is programmed while G94 is active will be interpreted as number of seconds to dwell.

Default—Yes
Modal—Yes
Cancels G95

G95 - Feed per Revolution

G95 puts the control into feed per revolution mode. When G95 is active, all F feedrate values are interpreted in inch/revolution (G20) or millimeter/revolution (G21) units. Any dwell command that is programmed while G95 is active will be interpreted as number of spindle revolutions to dwell.

Default—No
Modal—Yes
Cancels G94
G96 - Constant Surface Speed (CSS)

G96 sets feet per minute or meters per minute as the modal spindle command mode. When G96 is programmed, the spindle speed will increase as the tool tip moves toward the spindle centerline and decrease as the tool moves away from the spindle centerline. When G96 is active, the programmed S value is read as a feet per minute or meters per minute command.

Default—No
Modal—Yes
Cancels G97

To specify a radius of the tool tip that corresponds to the surface speed given in S, program the R value in the same block as G96.

- If you do not program an R value or the R value is 0, the control uses the X-axis position as the radius.
- If an R value has been previously programmed, and you program a G96 without an R value, the previously programmed R value will be used.

<table>
<thead>
<tr>
<th>Active Programming Mode</th>
<th>R Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>G07 Radius Programming</td>
<td>distance from the tool tip to the spindle centerline (radius)</td>
</tr>
<tr>
<td>G08 Diameter Programming</td>
<td>twice the distance from the tool tip to the spindle centerline (diameter)</td>
</tr>
</tbody>
</table>

Table 2–10. R Values and G96 Programming

G97 - Direct Spindle Speed (default)

G97 sets direct RPM as the modal spindle command mode. When G97 is active, the programmed S value is revolutions per minute.

Default—Yes
Modal—Yes
Cancels G96
G98 - Drill Cycle Initial Level Return (default)

When G98 is active, all G81-G89 drill cycles command the tool to return to the Z axis coordinate where the tool was when the cycle was initiated.

Default—Yes

Modal—Yes

Cancels G99

If the initial Z coordinate is changed for subsequent cycles, the Z coordinate for the first cycle after G80 is used as the reference plane.

![Diagram](image)

*Figure 2–30. G98 Drill Cycle Initial Level Return (default)*

**Example 1—G98**

In the following two blocks, the tool will rapid to Z10 in the second block even though Z20 (init plane) is specified:

```
N005 G98
N010 G00 Z10
N020 G81 Z5 F60 (feed to Z5, rapid to Z10)
N030 G00 Z20
N040 G81 Z15 (rapid to Z10, feed up to Z15, rapid to Z10)
N050 G80
```

**Example 2—G98**

To return to the init plane, G98 requires a G80 Cancel Drill Cycle to change to the init plane.

```
N005 G98
N010 G0 Z10
N020 G81 Z5 F60 (feed to Z5, rapid to Z10)
N030 G80 Z20
N040 G81 Z15 (rapid to Z10, feed up to Z15, rapid to Z20)
N050 G80
```
G99 - Drill Cycle R Plane Return

When G99 is active, all G81-G89 drill cycles command the tool to return to the R Reference plane at the end of the drill cycle.

Default—No

Modal—Yes

Cancels G98

Example—G99

Tool returns to the R Reference Plane.

N005 G99
N020 G81 Z5 (feed to Z5, rapid to R)

Figure 2–31. General Drill Cycle Tool Motion
G101 - Axial Surface of the Part - Main Spindle without Linear Y

G101 sets up the machine for Axial Milling without Linear Y Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G102, G103, G104, G105, G106, G107, G108, G109

Example

G101 (AXIS MAP)
M131 (MILLING MODE)
G00 Z.25

This part-relative, tool motion map is used when the Y Axis travel is inadequate for the geometry of the feature. The Y-axis positions are accomplished by coordinating the C axis and X axis motion. On TMM machines this map is always used for Axial Milling.

⚠️ This map is relative to the TMX MY and TMX MYS Main Spindle Only.
The Axial Milling map for the TMX MYS Sub-spindle uses G105.

Figure 2–32. G101 TMX Axis Coordinates
G102 - Radial Surface of the Part – Main Spindle without Linear Y

G102 sets up the machine for Radial Milling without Linear Y Motion and activates the coordinate map shown below.

Default—No

Modal—Yes


Example

G102 \textbf{(AXIS MAP)}

M131 \textbf{(MILLING MODE)}

G25 Q4 \textbf{(DIAMETER OF PART)}

G00 X.25

G25 Q must be used to establish the Z part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for wrapping the Y Axis coordinate system around the circumference of the part. The bottom surface of the feature will be parallel to the circumference of the part. On the TMM machine this map is always used for Radial Milling.

This map is relative to the TMX MY and TMX MYS Main Spindle only. The Radial Milling map for the TMX MYS Sub-spindle uses G106.

This map swaps the Z and X axes from G101.

The Y-axis travel limits are limited to +/- 800 inches (20320 mm) when using this map.

\textbf{Figure 2–33. G102 TMX Axis Coordinates}
In the following figure, the Cylinder Diameter is 4 inches (102 mm), and the Cylinder Circumference is 12.544 inches (319 mm).

Figure 2–34. Y Axis Coordinate System Detail for G102
G103 - Axial Surface of the Part – Main Spindle with Linear Y

G103 sets up the machine for Axial Milling with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No
Modal—Yes
Cancels—G101, G102, G104, G105, G106, G107, G108, G109

Example

G103  (AXIS MAP)
M131  (MILLING MODE)
Z.25
Y.1

When using tool holders with multiple turn stations that require a Y move on a tool change, the G103 axis map must be active.

Tool Change Example

G103  (AXIS MAP)
T0101
Y0.
G109

The linear Y-axis motion for this part-relative, tool motion map is created by using the combined motion of the X and X’ physical axes. The coordinate system for this axis map is the same as the G101 coordinate system.

⚠️ This map is relative to the TMX MY and TMX MYS Main Spindle only. The Axial Milling map for the TMX MYS Sub-spindle uses G107.

Figure 2–35. G103 TMX Axis Coordinates
G104 - Radial Surface of the Part – Main Spindle with Linear Y

G104 sets up the machine for Radial Milling with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G105, G106, G107, G108, G109

Example

G104  (AXIS MAP)
M131  (MILLING MODE)
G25  Q4.  (DIA. OF PART)

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for Radial Milling with true linear Y-Axis motion. The bottom surface of the feature will be perpendicular to the X physical axis. On TMM machines this map is always used for Radial Milling.

⚠️ This map is relative to the TMX MY and TMX MYS Main Spindle only. The Radial Milling map for the TMX MYS Sub-spindle uses G109.

This map swaps the Z and X axes from G101.

The Y+ and Y- axis has reversed direction from the G101 and G103 axis maps.

![Figure 2–36. G104 TMX Axis Coordinates](image-url)
G105 - Axial Surface of the Part – Sub-spindle without Linear Y

G105 sets up the machine for Axial Milling on the Sub-spindle and activates the coordinate map shown below.

Default—No
Modal—Yes
Cancels—G101, G102, G103, G104, G106, G107, G108, G109

Example

G105  **(AXIS MAP)**
M131  **(MILLING MODE)**

This part-relative, tool motion map is used when the Y-axis travel is inadequate for the geometry of the feature. The Y-axis positions are accomplished by coordinating the C and X axis motion.

⚠️ This map is relative to the TMX MYS Sub-Spindle only. The Axial Milling map for the TMX MYS Main spindle uses G101.

*Figure 2–37. G105 TMX Axis Coordinates*
G106 - Radial Surface of the Part – Sub-spindle without Linear Y

G106 sets up the machine for Radial Milling on the sub-spindle and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G107, G108, G109

Example

G106  (AXIS MAP)
M131  (MILLING MODE)
G25  Q4.  (DIA. OF PART)

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used to wrap the Y-axis coordinate system around the circumference of the part and the part is in the sub-spindle. The bottom surface of the feature will be parallel to the circumference of the part.

⚠️ This map is relative to the TMX MYS Sub-Spindle only. The Radial Milling map for the TMX MYS Main spindle uses G102.

This map swaps the Z and X axes from G101.

The Y-axis travel limits are limited to +/- 800 inches (20320 mm) when using this map.

Figure 2–38. G106 TMX Axis Coordinates
In the following figure, the Cylinder Diameter is 4 inches (102 mm), and the Cylinder Circumference is 12.544 inches (319 mm).

Figure 2–39. Y Axis Coordinate System Detail for G106
G107 - Axial Surface of the Part – Sub-spindle with Linear Y

G107 sets up the machine for axial milling on the sub-spindle with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes


Example

G107 (AXIS MAP)
M131 (MILLING MODE)

The linear Y-axis motion for this part-relative, tool motion map is created by using the combined motion of the X and X’ physical axes.

⚠️ This map is relative to the TMX MYS Sub-Spindle only. The Axial Milling map for the TMX MYS Main spindle uses G103.

The G107 coordinate system is the same as the G105 axis map.

Figure 2–40. G107 TMX Axis Coordinates
G108 - Radial Surface of the Part – Sub-spindle with Linear Y

G108 sets up the machine for Radial Milling on the sub-spindle with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G106, G107, G109

Example (Insert Example Here)

G108 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIA. OF PART)

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for Radial Milling on the sub-spindle with true linear Y-Axis motion. The bottom surface of the feature will be perpendicular to the X physical axis.

This map is relative to the TMX MYS Sub-Spindle only. The Radial Milling map for the TMX MYS Main spindle uses G104.

This map swaps the Z and X axes from G101.

This map reverses the direction of the Y-axis from the direction G101.

Figure 2–41. G108 TMX Axis Coordinates
G109 - Turning / Lathe Mode

G109 sets up the machine in Lathe Mode and activates the coordinate map shown below.

Default—Yes

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G106, G107, G108

Example (Turning Operations)

M130
G109

Example (Drilling operations with the C axis mapped instead of Y)

M131
G109

When G109 is used with M131 (Live Tool Mode) it allows for performing live tooling operations using polar coordinates. The G109 must be preceded by either the M130 (Lathe Mode) or M131 (Live-tool Mode – Main Spindle/C Axis).

⚠️ This axis map applies to all machine machines.

Polar Coordinate programming is not available for the sub spindle.

Figure 2–42. G109 Turning/Lathe Mode (Main Spindle Axis Map)
Figure 2–43. G109 Turning/Live-tool Mode (Main Spindle Axis Map)

Figure 2–44. G109 Turning/Lathe Mode (Sub-Spindle Axis Map)

Figure 2–45. G109 Turning/Live-tool Mode (Sub-spindle Axis Map)
BASIC NC M CODES

M codes belong to one of the following four execution categories:

- Execute before tool change or motion.
- Execute after T and S codes, but before motion.
- Execute after tool change or motion.
- Execute after tool change or motion (system M codes).

Understanding the execution category for each M code is important when the M code is programmed in a block that also contains a T code, S code, E code, and XZ axes motion.

The following M codes are available on most systems.

M00 - Program Stop .................................................. 3 - 4
M01 - Optional Stop ................................................... 3 - 4
M02 - End of Program (no rewind) ................................. 3 - 4
M03 - Spindle Clockwise .............................................. 3 - 4
M04 - Spindle Counterclockwise ................................. 3 - 4
M05 - Spindle Off ................................................. 3 - 4
M07 - Secondary Coolant On ........................................ 3 - 4
M08 - Primary Coolant On ........................................ 3 - 4
M09 - Coolant Off .................................................. 3 - 5
M10 - Set Chucking Pressure to High .......................... 3 - 5
M11 - Set Chucking Pressure to Low .......................... 3 - 5
M12 - Turret Index Reverse ........................................ 3 - 5
M13 - Turret Index Forward ...................................... 3 - 5
M14 - Chuck Open .................................................. 3 - 5
M15 - Chuck Close .................................................. 3 - 5
M16 - Chuck Air Blow On ......................................... 3 - 5
M17 - Chuck Air Blow Off ......................................... 3 - 5
M18 - Main Chuck Coolant On ................................... 3 - 5
M19 - Spindle Orient .................................................. 3 - 5
M20 - Chuck Open for Bar Feeder Start ....................... 3 - 5
M21 - Bar Feeder Finished Close Chuck ....................... 3 - 5
M22 - Start Bar Feeder (Bar Load) ............................... 3 - 5
M23 - Start Bar Feeder for Z Guided Feed .................... 3 - 6
M24 - Part Conveyor On ............................................. 3 - 6
M25 - Part Conveyor Off ........................................... 3 - 6
M28 - Tailstock Quill Advance ..................................... 3 - 6
<table>
<thead>
<tr>
<th>M Code</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>M29</td>
<td>Tailstock Quill Retract</td>
<td>6</td>
</tr>
<tr>
<td>M30</td>
<td>End of Program (rewind)</td>
<td>6</td>
</tr>
<tr>
<td>M31</td>
<td>Increment Cycle Counter</td>
<td>6</td>
</tr>
<tr>
<td>M32</td>
<td>Increment Setup Cycle Counter</td>
<td>6</td>
</tr>
<tr>
<td>M38</td>
<td>Steady Rest Clamp</td>
<td>6</td>
</tr>
<tr>
<td>M39</td>
<td>Steady Rest Unclamp</td>
<td>7</td>
</tr>
<tr>
<td>M41</td>
<td>Spindle Gear 1 / Low Gear Range</td>
<td>7</td>
</tr>
<tr>
<td>M42</td>
<td>Spindle Gear 2 / High Gear Range</td>
<td>7</td>
</tr>
<tr>
<td>M48</td>
<td>Use Feedrate Override</td>
<td>7</td>
</tr>
<tr>
<td>M49</td>
<td>Ignore Feedrate Override</td>
<td>7</td>
</tr>
<tr>
<td>M50</td>
<td>Conveyor On.</td>
<td>8</td>
</tr>
<tr>
<td>M51</td>
<td>Conveyor Off</td>
<td>8</td>
</tr>
<tr>
<td>M52</td>
<td>Auxiliary Output 1 On</td>
<td>8</td>
</tr>
<tr>
<td>M53</td>
<td>Auxiliary Output 2 On</td>
<td>8</td>
</tr>
<tr>
<td>M54</td>
<td>Auxiliary Output 3 On</td>
<td>8</td>
</tr>
<tr>
<td>M55</td>
<td>Auxiliary Output 4 On</td>
<td>8</td>
</tr>
<tr>
<td>M57</td>
<td>Use Part Catcher with Bar End Eject</td>
<td>8</td>
</tr>
<tr>
<td>M58</td>
<td>Part Catcher Advance</td>
<td>8</td>
</tr>
<tr>
<td>M59</td>
<td>Part Catcher Retract</td>
<td>8</td>
</tr>
<tr>
<td>M60</td>
<td>Select External Chucking</td>
<td>8</td>
</tr>
<tr>
<td>M61</td>
<td>Select Internal Chucking</td>
<td>8</td>
</tr>
<tr>
<td>M62</td>
<td>Auxiliary Output 1 Off</td>
<td>8</td>
</tr>
<tr>
<td>M63</td>
<td>Auxiliary Output 2 Off</td>
<td>8</td>
</tr>
<tr>
<td>M64</td>
<td>Auxiliary Output 3 Off</td>
<td>8</td>
</tr>
<tr>
<td>M65</td>
<td>Auxiliary Output 4 Off</td>
<td>9</td>
</tr>
<tr>
<td>M71</td>
<td>Tool Setter Retract</td>
<td>9</td>
</tr>
<tr>
<td>M72</td>
<td>Tool Setter Advance</td>
<td>9</td>
</tr>
<tr>
<td>M74</td>
<td>Activate Rigid Tapping</td>
<td>9</td>
</tr>
<tr>
<td>M85</td>
<td>Auto Door Open</td>
<td>9</td>
</tr>
<tr>
<td>M86</td>
<td>Auto Door Close</td>
<td>9</td>
</tr>
<tr>
<td>M91</td>
<td>Single Block Inactive</td>
<td>9</td>
</tr>
<tr>
<td>M94</td>
<td>Set Program Maximum Rapid Rate</td>
<td>9</td>
</tr>
<tr>
<td>M114</td>
<td>Chuck Open - Sub-spindle</td>
<td>9</td>
</tr>
<tr>
<td>M115</td>
<td>Chuck Close - Sub-spindle</td>
<td>9</td>
</tr>
<tr>
<td>M118</td>
<td>Sub Chuck Coolant On</td>
<td>10</td>
</tr>
<tr>
<td>M128</td>
<td>Z-axis Move to and Hitch to Tailstock</td>
<td>10</td>
</tr>
<tr>
<td>M129</td>
<td>Z-axis Unhitch from Tailstock</td>
<td>10</td>
</tr>
<tr>
<td>M130</td>
<td>Turning Mode</td>
<td>10</td>
</tr>
<tr>
<td>M131</td>
<td>Milling Mode</td>
<td>10</td>
</tr>
</tbody>
</table>
M132 - Enable “C” Axis Clamp ................................................................. 3 - 10
M133 - Enable “C” Axis Hold Assist ......................................................... 3 - 10
M134 - Release Both “C” Axis Clamps ....................................................... 3 - 10
M160 - Select External Chucking - Sub-spindle ....................................... 3 - 10
M161 - Select Internal Chucking - Sub-spindle ........................................ 3 - 10
M186 - Activate Torque Monitoring for W .............................................. 3 - 11
M187 - De-activate Torque Monitoring for W .......................................... 3 - 11
M200 - Block Skip Synchronization ......................................................... 3 - 11
M203 - Sync Spindles Forward Mode ..................................................... 3 - 11
M204 - Sync Spindles Reverse Mode ....................................................... 3 - 11
M205 - Clear Spindle Sync ................................................................. 3 - 11
M231 - Bypass Chuck Open Interlock - Main Spindle ......................... 3 - 11
M232 - Enable “C3” Axis Clamp .......................................................... 3 - 12
M234 - Release “C3” Axis Clamp .......................................................... 3 - 12
M241 - Allow Sub-spindle Chuck to Open while Spindle is Running for Part Transfer 3 - 12
**M00 - Program Stop**

M00 causes part program execution to stop and the spindle to turn off. Press the **START CYCLE** button to resume program execution and restart the look-ahead program. Execute after tool change or motion (system M codes).

**M01 - Optional Stop**

M01 performs the same functions as M00, but can be toggled on or off through the **Opt Stop** key on the control.

- When the **Opt Stop key** is **Off**, the program ignores M01.
- When the **Opt Stop key** is **On**, the program stops on M01 so you can inspect the part. Press the **START CYCLE** button to resume running the program.

Execute after tool change or motion.

**M02 - End of Program (no rewind)**

M02 stops program execution and leaves the part program pointer on the block that follows the M02 command. M02 is required at the end of all programs. It commands the spindle to turn off. Execute at the end of the block after all motion occurs.

**M03 - Spindle Clockwise**

M03 turns the spindle on in the clockwise direction. M03 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.

An S code specifying the spindle speed must be programmed before the spindle will turn on. The advanced RPM Look Ahead feature that reduces part cycle time by ramping the spindle up to speed before the feed move begins improves the cycle time because the control does not dwell while the spindle ramps up to the programmed speed. When this feature is active, it is possible to create a part program containing an M03 and spindle speed command that does not turn on the spindle.

Execute after T and S codes, but before motion.

**M04 - Spindle Counterclockwise**

M04 is the same as M03 except it commands the spindle to rotate in the counterclockwise direction. Execute after T and S codes, but before motion.

**M05 - Spindle Off**

M05 turns the spindle off. M05 executes at the end of the block after all motion occurs. Execute after tool change or motion (system M codes).

**M07 - Secondary Coolant On**

M07 commands the secondary coolant pump to turn on, if so equipped. This command is cancelled by an M09 command. Execute before tool change or motion.

**M08 - Primary Coolant On**

M08 commands the primary coolant pump to turn on, typically flood coolant. This command is cancelled by an M09 command. Execute before tool change or motion.
M09 - Coolant Off
M09 command tells the control to turn the primary and secondary coolant pumps off. Execute after tool change or motion.

M10 - Set Chucking Pressure to High
M10 command sets the chucking pressure to high.

M11 - Set Chucking Pressure to Low
M11 command sets the chucking pressure to low.

M12 - Turret Index Reverse
M12 is used to reverse the turret.

M13 - Turret Index Forward
M13 is used to move the turret forward.

M14 - Chuck Open
M14 is used to open the chuck for use with the optional Bar Feeder.

M15 - Chuck Close
M15 is used to close the chuck for use with the optional Bar Feeder.

M16 - Chuck Air Blow On
M16 turns on the chuck air blow.

M17 - Chuck Air Blow Off
M17 turns off the chuck air blow.

M18 - Main Chuck Coolant On
M18 turns on the main chuck coolant.

M19 - Spindle Orient
M19 orients all spindles to 0°.

M20 - Chuck Open for Bar Feeder Start
M20 is used with the optional Bar Feeder.

M21 - Bar Feeder Finished Close Chuck
M21 is used with the optional Bar Feeder.

M22 - Start Bar Feeder (Bar Load)
M22 is used with the optional Bar Feeder.
**M23 - Start Bar Feeder for Z Guided Feed**

M23 is used with the optional Bar Feeder to open the chuck then cause the bar feeder with Z to guide the stock.

**M24 - Part Conveyor On**

M24 turns on the optional Part Conveyor.

**M25 - Part Conveyor Off**

M25 turns off the optional Part Conveyor.

**M28 - Tailstock Quill Advance**

M28 is used to advance the optional Tailstock Quill. If either the Interrupt or Stop Cycle console key is pressed before the quill is fully advanced, the quill will automatically retract. If in Interrupt mode and you return to Auto Mode and press Start Cycle, the Tailstock will advance before the program continues.

**M29 - Tailstock Quill Retract**

M29 is used to retract the optional Tailstock Quill.

**M30 - End of Program (rewind)**

M30 stops program execution and moves the part program pointer to the top of the part program. M30 commands the spindle to turn off. Execute after tool change or motion (system M codes).

**M31 - Increment Cycle Counter**

M31 increments the cycle counter.

**M32 - Increment Setup Cycle Counter**

M32 increments the setup cycle counter.

**M38 - Steady Rest Clamp**

The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

⚠️ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

This optional device is used with turning centers to hold long pieces of stock in the center while cutting.
**M39 - Steady Rest Unclamp**

After the spindle is stopped, M39 unclamps the optional Steady Rest device during automatic cycle operation. The spindle MUST BE stopped when this M-Code executes or an error will appear and the cycle will be halted.

⚠️ The spindle must be stopped prior to executing the M39 Steady Rest Unclamp.

The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

⚠️ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

**M41 - Spindle Gear 1 / Low Gear Range**

M41 sets the spindle to Gear 1, or the low gear range.

Low Gear Speed Range for TM18:
- Minimum RPM = 5
- Maximum RPM = 600

**M42 - Spindle Gear 2 / High Gear Range**

M42 sets the spindle to Gear 2, or the high gear range.

High Gear Speed Range for TM18:
- Minimum RPM = 15
- Maximum RPM = 1600

**M48 - Use Feedrate Override**

M48 uses the value specified by the Feedrate Override dial (FPM %). Execute before tool change or motion.

**M49 - Ignore Feedrate Override**

M49 ignores the value specified by the Feedrate Override (FPR %) and uses the programmed feedrate.
**M50 - Conveyor On**
M50 is used to turn on the optional Conveyor.

**M51 - Conveyor Off**
M51 is used to turn off the optional Conveyor.

**M52 - Auxiliary Output 1 On**
M52 is used to turn on auxiliary equipment or a unique machine function.

**M53 - Auxiliary Output 2 On**
M53 is used to turn on auxiliary equipment or a unique machine function.

**M54 - Auxiliary Output 3 On**
M54 is used to turn on auxiliary equipment or a unique machine function.

**M55 - Auxiliary Output 4 On**
M55 is used to turn on auxiliary equipment or a unique machine function.

**M57 - Use Part Catcher with Bar End Eject**
M57 is used to advance the optional Part Catcher with bar end eject.

**M58 - Part Catcher Advance**
M58 is used to advance the optional Part Catcher.

**M59 - Part Catcher Retract**
M59 is used to retract the optional Part Catcher.

**M60 - Select External Chucking**
M60 is used to select external chucking.

**M61 - Select Internal Chucking**
M61 is used to select internal chucking.

**M62 - Auxiliary Output 1 Off**
M62 is used to turn off auxiliary equipment or a unique machine function.

**M63 - Auxiliary Output 2 Off**
M63 is used to turn off auxiliary equipment or a unique machine function.

**M64 - Auxiliary Output 3 Off**
M64 is used to turn off auxiliary equipment or a unique machine function.
**M65 - Auxiliary Output 4 Off**

M65 is used to turn off auxiliary equipment or a unique machine function.

**M71 - Tool Setter Retract**

M71 is used to retract the optional Tool Setter.

**M72 - Tool Setter Advance**

M72 is used to advance the optional Tool Setter.

**M74 - Activate Rigid Tapping**

M74 is used to activate rigid tapping. M74 must proceed the tapping G code and must be on a block by itself.

**M74 Example:**

```
M74
G84 Z-0.5 F0.05
```

**M85 - Auto Door Open**

M85 is used to open the optional Auto Door.

**M86 - Auto Door Close**

M86 is used to close the optional Auto Door.

**M91 - Single Block Inactive**

M91 is used for inactivating a single block.

**M92 - Single Block Active**

M92 is used for activating a single block.

**M94 - Set Program Maximum Rapid Rate**

M94 is used to set the maximum rapid rate for the program. M94 requires an argument to specify the maximum rapid rate and must be expressed as MMPM.

**M94 Example:**

```
M94:5080
```

This block sets the Maximum Rapid Rate to 5080 MMPM (200 IPM).

**M114 - Chuck Open - Sub-spindle**

M114 is used for opening the sub-spindle chuck.

**M115 - Chuck Close - Sub-spindle**

M115 is used for closing the sub-spindle chuck.
**M118 - Sub Chuck Coolant On**
M118 is used for turning on coolant for the sub-spindle chuck.

**M128 - Z-axis Move to and Hitch to Tailstock**
M128 is used to move the Z-axis to the Tailstock and subsequently to hitch to the Tailstock. The Tailstock can then move with the Z-axis to the specified Z location.

**M129 - Z-axis Unhitch from Tailstock**
M129 is used to unhitch the Z-axis from the Tailstock. The Z-axis can then move freely from the tailstock.

**M130 - Turning Mode**
M130 is used to set the control to turning mode.

**M131 - Milling Mode**
M131 is used to set the control to milling mode.

**M132 - Enable “C” Axis Clamp**
M132 enables the C axis clamp.

**M133 - Enable “C” Axis Hold Assist**
M133 enables the C axis hold assist.

**M134 - Release Both “C” Axis Clamps**
M134 releases both C axis clamps.

**M160 - Select External Chucking - Sub-spindle**
M160 selects external chucking for sub-spindle.

**M161 - Select Internal Chucking - Sub-spindle**
M161 selects internal chucking for sub-spindle.
**M186 - Activate Torque Monitoring for W**

M186 activates torque monitoring for the W-axis. Torque monitoring is available when picking up or transferring parts from either the Main spindle to the Sub-spindle or from the Sub-spindle to the Main spindle. The force, or load, of the material on the W-axis is monitored when M186 is used.

If the motor load is greater than 50% of the rated Full load for the W-axis, the program stops and an error message appears. The 50% value cannot and does not change. Use M186 in conjunction with G06 (Probe/Block Skip Function). With G06 active, the system stops the motion and deletes the remaining distance to go. G06 is a one-shot command and stays in effect until the end of the block.

M186 functions when Feed motions are used (G01, G02, G03). M186 does not function with Rapid motions (G00).

**M186 Example**

M241 (Allow sub-spindle chuck to open when spindle running)  
M114 (Open sub-spindle chuck)  
G0 W200 (Rapid W to approach position)  
M186 (Activate W-axis torque monitoring)  
G1 G6 W180 F20 (Feed W-axis forward to grab part and activate Probe/Block skip)  
M187 (Stop W-axis torque monitoring)  
G0 W200 (Rapid W back to approach position)

**M187 - De-activate Torque Monitoring for W**

M187 deactivates torque monitoring for the W-axis.

**M200 - Block Skip Synchronization**

M200 allows you to turn on block skip while the part program is executing. Using this code ensures that blocks marked with block delete codes are skipped after block skip has been turned on, without requiring you to stop and restart the program. Execute before tool change or motion. Refer to *Block Skip Code, on page 1 - 6* for information about Block Skip.

**M203 - Sync Spindles Forward Mode**

M203 synchronizes the main spindle and sub-spindle forward.

**M204 - Sync Spindles Reverse Mode**

M204 synchronizes the main spindle and sub-spindle in reverse.

**M205 - Clear Spindle Sync**

M205 clears the main spindle and sub-spindle synchronization.

**M231 - Bypass Chuck Open Interlock - Main Spindle**

M231 bypasses the main spindle chuck open interlock.
**M232 - Enable “C3” Axis Clamp**

M232 enables the C3 axis clamp.

**M234 - Release “C3” Axis Clamp**

M234 releases the C3 axis clamp.

**M241 - Allow Sub-spindle Chuck to Open while Spindle is Running for Part Transfer**

M241 allows the sub-spindle chuck to open while the sub-spindle is running. This interlock bypass is cancelled automatically when M115 (Close Sub-spindle Chuck) is executed.
ISNC G CODES

The following ISNC G Codes are defined in this section:

Overview .......................................................... 4 - 3
G00 - Rapid Traverse (default) .............................. 4 - 6
G01 - Linear Interpolation .................................. 4 - 7
G02/G03 - Clockwise/Counterclockwise Arc .......... 4 - 8
G04 - Dwell ....................................................... 4 - 12
G06 - Probe/Block Skip .................................... 4 - 12
G07 - Radius Programming ................................ 4 - 13
G08 - Diameter Programming (default) ................. 4 - 15
G09 - Exact Stop ................................................ 4 - 17
G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry) 4 - 17
G11 - Data Setting Cancel (default) ..................... 4 - 18
G17 - XY Plane .................................................. 4 - 19
G18 - XZ Plane (default) ................................... 4 - 21
G19 - YZ Plane .................................................. 4 - 22
G20 - Inch Mode (default) .................................. 4 - 23
G21 - Millimeter Mode ....................................... 4 - 24
G28 - Automatic Return to Reference Point .......... 4 - 25
G32 - Constant Lead Thread Cutting ....................... 4 - 27
G33 - Threading .................................................. 4 - 29
G40 - Cutter Radius Compensation Off (default) .... 4 - 32
G41 - Cutter Radius Compensation Left ................. 4 - 32
G42 - Cutter Radius Compensation Right ............... 4 - 33
G50 - Set Part Setup or Set Max RPM ................. 4 - 35
G53 - Machine Coordinates ................................ 4 - 36
G54-G59 - Work Coordinates (G54 default) ............ 4 - 37
G54.1 - G54.93 - Aux Work Coordinates ................. 4 - 39
G61 - Precision Cornering .................................. 4 - 40
G64 - Precision Cornering Cancel (default) .......... 4 - 40
G65 - Subprogram Call ....................................... 4 - 40
G66 - Modal Subprogram Call ............................... 4 - 47
G67 - Modal Subprogram Call Cancel (default) ...... 4 - 49
G70 - Finishing Cycle ......................................... 4 - 49
G71 - Stock Removal in Turning ......................... 4 - 51
G72 - Stock Removal in Facing ............................ 4 - 54
G73 - Pattern Repeating .............................................. 4 - 56
G74 - End Face Peck Drilling ........................................ 4 - 58
G75 - Outer Diameter/Inner Diameter Drilling ......... 4 - 60
G76 - Multiple Threading Cycle. ................................ 4 - 62
G77 - Outer Diameter/Inner Diameter Cutting Cycle .... 4 - 64
G78 - Threading Cycle. .............................................. 4 - 66
G79 - Stock Removal in Facing Cycle ......................... 4 - 67
G80 - Canned Cycle Cancel (default). ................. 4 - 68
G81 - Drill Cycle ....................................................... 4 - 69
G82 - Drill Cycle with Dwell ....................................... 4 - 70
G83 - Face Drilling with Pecks ................................. 4 - 72
G83.1 - Chip Breaker Peck Drilling ............................... 4 - 74
G84 - Face CW Tapping .............................................. 4 - 75
G84.2 (or G84 M29) - Rigid CW Tapping .................. 4 - 77
G85 - Boring Cycle ..................................................... 4 - 78
G86 - Bore Rapid Out Cycle ........................................ 4 - 80
G87 - Side Drilling with Pecks/Dwell ......................... 4 - 82
G88 - Side CW Tapping .............................................. 4 - 83
G89 - Side Boring Cycle ............................................. 4 - 84
G90 - Absolute Programming (default) ..................... 4 - 85
G91 - Incremental Programming ................................. 4 - 87
G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle 4 - 89
G93 - Inverse Time Feedrate ........................................ 4 - 91
G94 - Feed per Minute (default) ................................. 4 - 92
G95 - Feed per Revolution ......................................... 4 - 92
G96 - Constant Surface Speed (CSS) ......................... 4 - 93
G97 - Direct Spindle Speed (default) ......................... 4 - 93
G98 - Return to Initial Level (default) ....................... 4 - 94
G99 - Return to R Point Level .................................... 4 - 95
G101 - Axial Surface of the Part - Main Spindle without Linear Y .......... 4 - 96
G102 - Radial Surface of the Part – Main Spindle without Linear Y ........ 4 - 97
G103 - Axial Surface of the Part – Main Spindle with Linear Y ............ 4 - 99
G104 - Radial Surface of the Part – Main Spindle with Linear Y ............ 4 - 100
G105 - Axial Surface of the Part – Sub-spindle without Linear Y .......... 4 - 101
G106 - Radial Surface of the Part – Sub-spindle without Linear Y .......... 4 - 102
G107 - Axial Surface of the Part – Sub-spindle with Linear Y ............. 4 - 104
G108 - Radial Surface of the Part – Sub-spindle with Linear Y ............. 4 - 105
G109 - Turning / Lathe Mode ....................................... 4 - 106
Overview

G Codes initiate axis motion, plane changes, and feedrate changes. This information is often needed when using an off-line CAM or CAD/CAM system to create NC part programs.

⇒ Some G Codes are on by default at the start of a program until another G Code cancels it. Defaults are identified below.

Any unrecognized code will stop execution and display an error message. More than one G code can be specified in the same block. If more than one is from the same group, the last G code entered is active.

G Code Groups

The G codes are grouped by function. These groups are listed in the following table:

<table>
<thead>
<tr>
<th>Group</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>One-Shot</td>
</tr>
<tr>
<td>01</td>
<td>Interpolation</td>
</tr>
<tr>
<td>03</td>
<td>Dimension</td>
</tr>
<tr>
<td>02</td>
<td>Spindle Speed</td>
</tr>
<tr>
<td>05</td>
<td>Feed</td>
</tr>
<tr>
<td>06</td>
<td>Units of Measure</td>
</tr>
<tr>
<td>07</td>
<td>Cutter Compensation</td>
</tr>
<tr>
<td>10</td>
<td>Return from Canned Cycles</td>
</tr>
<tr>
<td>14</td>
<td>Coordinate System Selection</td>
</tr>
<tr>
<td>16</td>
<td>Plane Selection</td>
</tr>
<tr>
<td>21</td>
<td>Programming Mode</td>
</tr>
<tr>
<td>30</td>
<td>Lathe Axis Map Coordinates</td>
</tr>
</tbody>
</table>

Table 4–1.  G Code Groups

⇒ Specifying a Group 01 (Interpolation) G code in a canned cycle automatically enters the G80 - Canned Cycle Cancel (default) condition. Conversely, a Group 01 G code is not affected by the canned cycle G codes.

G Code Table

The following table lists the G codes, identifies the defaults in bold text, identifies groups, lists Modal or Non-modal types, and describes the G code functions.
<table>
<thead>
<tr>
<th>G Code</th>
<th>Group</th>
<th>Type</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>01</td>
<td>Modal</td>
<td>Rapid Traverse (default)</td>
</tr>
<tr>
<td>G01</td>
<td></td>
<td></td>
<td>Linear Interpolation</td>
</tr>
<tr>
<td>G02</td>
<td></td>
<td></td>
<td>Clockwise Circular Motion at Feed</td>
</tr>
<tr>
<td>G03</td>
<td></td>
<td></td>
<td>Counterclockwise Circular Motion at Feed</td>
</tr>
<tr>
<td>G04</td>
<td>00</td>
<td>Non-modal</td>
<td>Dwell</td>
</tr>
<tr>
<td>G07</td>
<td>21</td>
<td>Modal</td>
<td>Radius Programming</td>
</tr>
<tr>
<td>G08</td>
<td></td>
<td>Modal</td>
<td>Diameter Programming (default)</td>
</tr>
<tr>
<td>G09</td>
<td>00</td>
<td>Non-modal</td>
<td>Exact Stop</td>
</tr>
<tr>
<td>G10</td>
<td>00</td>
<td>Modal</td>
<td>Data Setting—Work Offsets or Tool Offsets or Tool Geometry Offsets</td>
</tr>
<tr>
<td>G11</td>
<td></td>
<td></td>
<td>Data Setting Cancel (default)</td>
</tr>
<tr>
<td>G17</td>
<td>16</td>
<td>Modal</td>
<td>XY Plane</td>
</tr>
<tr>
<td>G18</td>
<td></td>
<td>Modal</td>
<td>XZ Plane (default)</td>
</tr>
<tr>
<td>G19</td>
<td></td>
<td></td>
<td>YZ Plane</td>
</tr>
<tr>
<td>G20</td>
<td>06</td>
<td>Modal</td>
<td>Inch Mode (default)</td>
</tr>
<tr>
<td>G21</td>
<td></td>
<td></td>
<td>Millimeter Mode</td>
</tr>
<tr>
<td>G28</td>
<td></td>
<td>Non-modal</td>
<td>Automatic Return to reference Point</td>
</tr>
<tr>
<td>G32</td>
<td>01</td>
<td>Modal</td>
<td>Constant Lead Thread Cutting</td>
</tr>
<tr>
<td>G33</td>
<td></td>
<td></td>
<td>Threading</td>
</tr>
<tr>
<td>G40</td>
<td>07</td>
<td>Modal</td>
<td>Cutter Radius Compensation Off (default)</td>
</tr>
<tr>
<td>G41</td>
<td></td>
<td></td>
<td>Cutter Radius Compensation Left</td>
</tr>
<tr>
<td>G42</td>
<td></td>
<td></td>
<td>Cutter Radius Compensation Right</td>
</tr>
<tr>
<td>G50 or G92</td>
<td>00</td>
<td>Non-Modal</td>
<td>Set Part Setup or Set Max RPM</td>
</tr>
<tr>
<td>G53</td>
<td>14</td>
<td>Non-modal</td>
<td>Machine Coordinates</td>
</tr>
<tr>
<td>G54 - G59</td>
<td></td>
<td></td>
<td>Work Coordinates (default)</td>
</tr>
<tr>
<td>G54.1 - G54.93</td>
<td></td>
<td></td>
<td>Aux Work Coordinates</td>
</tr>
<tr>
<td>G61</td>
<td></td>
<td>Modal</td>
<td>Precision Cornering</td>
</tr>
<tr>
<td>G64</td>
<td></td>
<td></td>
<td>Precision Cornering Cancel (default)</td>
</tr>
<tr>
<td>G65</td>
<td>00</td>
<td></td>
<td>Subprogram Call</td>
</tr>
<tr>
<td>G66</td>
<td>12</td>
<td>Modal</td>
<td>Modal Subprogram Call</td>
</tr>
<tr>
<td>G67</td>
<td></td>
<td>Modal Subprogram Call Cancel (default)</td>
<td></td>
</tr>
<tr>
<td>G70</td>
<td>00</td>
<td>Non-Modal</td>
<td>Finish Cycle</td>
</tr>
<tr>
<td>G71</td>
<td></td>
<td></td>
<td>Stock Removal in Turning</td>
</tr>
<tr>
<td>G72</td>
<td></td>
<td></td>
<td>Stock Removal in Facing</td>
</tr>
<tr>
<td>G73</td>
<td></td>
<td></td>
<td>Stock Removal with Repeat Pattern</td>
</tr>
<tr>
<td>G74</td>
<td></td>
<td></td>
<td>Z Direction Grooving</td>
</tr>
<tr>
<td>G75</td>
<td></td>
<td></td>
<td>X Direction Grooving</td>
</tr>
<tr>
<td>G76</td>
<td>01</td>
<td>Modal</td>
<td>Multiple Thread Cutting</td>
</tr>
<tr>
<td>G77</td>
<td></td>
<td></td>
<td>Stock Removal in Turning Cycle</td>
</tr>
<tr>
<td>G78 or G92</td>
<td></td>
<td></td>
<td>Threading Cycle</td>
</tr>
<tr>
<td>G79</td>
<td></td>
<td></td>
<td>Stock Removal in Facing Cycle</td>
</tr>
<tr>
<td>G Code</td>
<td>Group</td>
<td>Type</td>
<td>Function</td>
</tr>
<tr>
<td>--------</td>
<td>-------</td>
<td>------</td>
<td>----------</td>
</tr>
<tr>
<td>G80</td>
<td>10</td>
<td>Modal</td>
<td>Canned Cycle Cancel <em>(default)</em></td>
</tr>
<tr>
<td>G81</td>
<td></td>
<td></td>
<td>Drill Cycle</td>
</tr>
<tr>
<td>G82</td>
<td></td>
<td></td>
<td>Drill Cycle with Dwell</td>
</tr>
<tr>
<td>G83</td>
<td></td>
<td></td>
<td>Face Drilling with Pecks</td>
</tr>
<tr>
<td>G83.1</td>
<td></td>
<td></td>
<td>Chip Breaker Peck Drilling</td>
</tr>
<tr>
<td>G84</td>
<td></td>
<td></td>
<td>Face CW Tapping</td>
</tr>
<tr>
<td>G84.2 (or G84 M29)</td>
<td></td>
<td></td>
<td>Rigid CW Tapping</td>
</tr>
<tr>
<td>G85</td>
<td></td>
<td></td>
<td>Face Boring Cycle</td>
</tr>
<tr>
<td>G86</td>
<td></td>
<td></td>
<td>Bore Rapid Out Cycle</td>
</tr>
<tr>
<td>G87</td>
<td></td>
<td></td>
<td>Side Drilling with Pecks/Dwell</td>
</tr>
<tr>
<td>G88</td>
<td></td>
<td></td>
<td>Side CW Tapping</td>
</tr>
<tr>
<td>G89</td>
<td></td>
<td></td>
<td>Side Boring Cycle</td>
</tr>
<tr>
<td>G90</td>
<td>03</td>
<td>Modal</td>
<td>Absolute Programming <em>(default)</em></td>
</tr>
<tr>
<td>G91</td>
<td></td>
<td></td>
<td>Incremental Programming</td>
</tr>
<tr>
<td>G92</td>
<td>14</td>
<td>Modal</td>
<td>Work Coordinate Offsets or Spindle Max Speed or Threading Cycle</td>
</tr>
<tr>
<td>G93</td>
<td>05</td>
<td>Modal</td>
<td>Inverse Time</td>
</tr>
<tr>
<td>G94</td>
<td></td>
<td></td>
<td>Feed per Minute <em>(default)</em></td>
</tr>
<tr>
<td>G95</td>
<td></td>
<td></td>
<td>Feed per Revolution</td>
</tr>
<tr>
<td>G96</td>
<td>02</td>
<td>Modal</td>
<td>Constant Surface Speed (CSS)</td>
</tr>
<tr>
<td>G97</td>
<td></td>
<td></td>
<td>Direct Spindle Speed <em>(default)</em></td>
</tr>
<tr>
<td>G98</td>
<td>11</td>
<td>Modal</td>
<td>Return to Initial Level <em>(default)</em></td>
</tr>
<tr>
<td>G99</td>
<td></td>
<td></td>
<td>Return to R Point Level</td>
</tr>
<tr>
<td>G101</td>
<td>30</td>
<td>Modal</td>
<td>Axial Surface of the Part - Main Spindle without Linear Y</td>
</tr>
<tr>
<td>G102</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Main Spindle without Linear Y</td>
</tr>
<tr>
<td>G103</td>
<td></td>
<td>Modal</td>
<td>Axial Surface of the Part - Main Spindle with Linear Y</td>
</tr>
<tr>
<td>G104</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Main Spindle with Linear Y</td>
</tr>
<tr>
<td>G105</td>
<td></td>
<td>Modal</td>
<td>Axial Surface of the Part - Sub-spindle without Linear Y</td>
</tr>
<tr>
<td>G106</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Sub-spindle without Linear Y</td>
</tr>
<tr>
<td>G107</td>
<td></td>
<td>Modal</td>
<td>Axial Surface of the Part - Sub-spindle with Linear Y</td>
</tr>
<tr>
<td>G108</td>
<td></td>
<td>Modal</td>
<td>Radial Surface of the Part - Sub-spindle with Linear Y</td>
</tr>
<tr>
<td>G109</td>
<td></td>
<td>Modal</td>
<td>Turning / Lathe Mode <em>(default)</em></td>
</tr>
</tbody>
</table>

*Table 4–2. G Codes*
G00 - Rapid Traverse (default)

G00 sets rapid traverse as the machine's modal condition. Any block that executes while G00 is modal moves the axes at rapid feedrate to the programmed endpoint. The feedrate of each axis is controlled to ensure that all programmed axes reach their respective endpoints simultaneously.

Default—Yes
Modal—Yes
Cancels G01 - Linear Interpolation, G02/G03 - Clockwise/Counterclockwise Arc

Format
G0 X(u) ____ Y(v) ____ Z (w) ____ C(h)____

- X indicates Absolute X and (u) indicates Incremental X.
- Y indicates Absolute Y and (v) indicates Incremental Y.
- Z indicates Absolute Z and (w) indicates Incremental Z.
- C indicates Absolute C and (h) indicates Incremental C.
G01 - Linear Interpolation

G01 sets feed linear interpolation as the machine's modal condition. Any block that executes while G01 is modal moves the programmed axes directly to their programmed endpoint at a specified feedrate using the F command. G01 requires programming X or Z or both and you may include F if not specified previously. The vector feedrate of each axis is controlled to ensure that all axes reach their endpoints simultaneously.

Default—No

Modal—Yes

Cancels G00 - Rapid Traverse (default), G02/G03 - Clockwise/Counterclockwise Arc

Format

G1 X(u) ____ Y(v) ____ Z(w) ____ C(h) ____

- X indicates Absolute X and (u) indicates Incremental X.
- Y indicates Absolute Y and (v) indicates Incremental Y.
- Z indicates Absolute Z and (w) indicates Incremental Z.
- C indicates Absolute C and (h) indicates Incremental C.

Examples

N5 G90 G07 G01 X0.25 Z-0.5 F ___ (G90 = Absolute; G07 = Radius)
N7 G90 G08 G01 X1.00 Z-0.5 F ___ (G90 = Absolute; G08 = Diameter)
N5 G91 G07 G01 X0.25 Z-0.5 F ___ (G91 = Incremental; G07 = Radius)
N7 G91 G08 G01 X0.50 Z-0.5 F ___ (G91 = Incremental; G08 = Diameter)
G02/G03 - Clockwise/Counterclockwise Arc

G02 sets clockwise arc as the modal condition. G03 sets counterclockwise arc as the modal condition.

Default—No

Modal—Yes

Cancels:
- G00 - Rapid Traverse (default)
- G01 - Linear Interpolation,
- G02/G03 - Clockwise/Counterclockwise Arc

Either program the arc endpoint and center coordinates, or program the arc endpoint and radius. Refer to Program Arc with Endpoint and Center Coordinates, on page 4 - 9 and Program Arc with Endpoint and Radius, on page 4 - 11 for programming examples.

Format

G02 X(u) ____ Z(w) ____ I ____ K ____ or R ____
G03 X(u) ____ Z(w) ____ I ____ K ____ or R ____

- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters
- I—X-axis Center/Offset coordinate for programming geometric information needed to determine the endpoint of a motion command.
- K—Z-axis Center/Offset coordinate for programming geometric information needed to determine the endpoint of a motion command.
- R—radius
Example

The following figure illustrates G02 (Clockwise) arc at feed:

![Figure 4-1. G02 (Clockwise) circular motion at feed](image)

**Program Arc with Endpoint and Center Coordinates**

This method requires you to define the arc endpoint and center coordinates. Dimensions for the arc endpoint (X, Z) may be defined with absolute or incremental dimensions, depending on the modal G90 - Absolute Programming (default) or G91 - Incremental Programming condition.

All arc center dimensions (I, K) must be defined using signed incremental dimensions (the distance from the arc start point to the arc center).

![Figure 4–2. Arc with Endpoint and Center Coordinates](image)

If G07 - Radius Programming is active, the arc endpoint (X) and center (I) coordinates must be programmed as radius values.

If G08 - Diameter Programming (default) is active the arc endpoint (X) must be programmed as a diameter value, but the arc center (I) must be programmed as a radius value.
Example G02/G03 Arcs

The following example part program shows arcs programmed with X, Z, I, and K.

```
%  
N1 G90 G20 G40 G54  (MAKE OFFSET 10 INCHES)  
N2 G00 T0101 G07 X.0 Z.1  (G07 = RADIUS PROGRAMMING)  (MAKE SURE OFFSET IS LARGER THAN 0.5")  
N3 G01 Z0 F10  
N4 X.2  
N5 G03 X0.8 Z-0.6 I0 K-0.6  
N6 G01 G91 Z-1.0  
N7 G90 G08 G02 X1.9512 Z-2.0243 R.6  (G08 = DIAMETER PROGRAMMING)  
N8 G01 X2.2 Z-2.1487  
N9 Z-2.5  
N1 G00 X2.5  
N11 Z.1  
M30
```

Figure 4–3. Arc with Endpoint and Center Coordinates
Program Arc with Endpoint and Radius

The second method of programming a G02 or G03 block is to define the arc endpoint and radius. There are two possible arcs when you program X, Z, and R (radius). The sign of the R radius value determines whether you want the arc that spans less than 180° (by programming R as a positive value), or the arc that spans more than 180° (by programming R as a negative value).

N2 G90 G07 G02 X__ Z__ R+__ (G90 = Absolute; G07 = Radius; G02 = CW)
N2 G90 G07 G03 X__ Z__ R-__ (G90 = Absolute; G07 = Radius; G03 = CCW)
N7 G90 G08 G02 X__ Z__ R+__ (G90 = Absolute; G08 = Diameter; G02 = CW)
N7 G90 G08 G03 X__ Z__ R-__ (G90 = Absolute; G08 = Diameter; G03 = CCW)
N2 G91 G07 G02 X__ Z__ R+__ (G91 = Incremental; G07 = Radius G02 = CW)
N2 G91 G07 G03 X__ Z__ R-__ (G91 = Incremental; G07 = Radius; G03 = CCW)
N7 G91 G08 G02 X__ Z__ R+__ (G91 = Incremental; G08 = Diameter G02 = CW)
N7 G91 G08 G03 X__ Z__ R-__ (G91 = Incremental; G08 = Diameter; G03 = CCW)

Figure 4–4. Arc with Endpoint and Radius
G04 - Dwell

G04 programs a pause in part program execution.

Modal—No

Format

G04 P ____ or X ____ or U ____

X and U can include a decimal point. For example, X10 is X10.0.

P cannot include a decimal point, and the value is divided by 1000 or 10000. For example, P3000 is 3.0 seconds.

<table>
<thead>
<tr>
<th>If Active G Code</th>
<th>Then P, X, or U in the G04 Block Programs</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>G93</strong> Inverse Time Feedrate</td>
<td>The actual feedrate is the distance divided by the time. The dwell is in seconds. Refer to <strong>G93 - Inverse Time Feedrate</strong>, on page 4 - 91 for information about G93.</td>
</tr>
<tr>
<td><strong>G94</strong> Feed Per Minute</td>
<td>The number of seconds to dwell. For example, the block G04 X2.1 pauses for 2.1 seconds before it executes the next block of code. Refer to <strong>G94 - Feed per Minute (default)</strong>, on page 4 - 92 for information about G94.</td>
</tr>
<tr>
<td><strong>G95</strong> Feed Per Revolution</td>
<td>The number of spindle revolutions to dwell. For example, the block G04 X10 pauses for 10 spindle revolutions before it executes the next block of code. Refer to <strong>G95 - Feed per Revolution</strong>, on page 4 - 92 for information about G95.</td>
</tr>
</tbody>
</table>

Table 4–3. G04 Dwell Examples

G06 - Probe/Block Skip

G06 arms the Probe mechanism and provides a means for stopping the axis, for instance, when the torque is detected to be too high. Remaining Motion programmed after a high torque detection is deleted from memory. Please refer to **W Torque Mon On (ISNC M186)**, on page 5 - 16 for details.
G07 - Radius Programming

You can program X axis dimensions using the radius or diameter of the part. Program G07 when you wish to program radius dimensions.

Default—No

Modal—Yes

Cancels—G08

Format

G07 X ____ (X is interpreted as a radius value.)

This mode will affect live-tool (milling) when running G codes. When G07 is active, live-tool programming is the same as programming a 3-axis milling machine.

The table below provides information about how the modal setting affects part programs.

<table>
<thead>
<tr>
<th>Programmed Value</th>
<th>G07 Modal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Endpoint of linear move</td>
<td>X use radius</td>
</tr>
<tr>
<td>Endpoint of circular move</td>
<td>X use radius</td>
</tr>
<tr>
<td>Center and radius of circular move</td>
<td>IKR use radius</td>
</tr>
<tr>
<td>X axis infeed distances</td>
<td>infeed distance use radius</td>
</tr>
</tbody>
</table>

Table 4-4. Effects of Using G07

G07 Radius Programming Example

The following example part program shows arcs programmed with X, Z, I, and K.

G90 G07 X0.8 Z-0.6 I0 K-0.6 (G90 = Absolute; G07 = Radius)
G91 G07 X0.8 Z-0.6 I0 K-0.6 (G91 = Incremental; G07 = Radius)

G07, Radius programming, establishes a modal setting that interprets X-axis dimensions in radius values. The following figure illustrates radius programming:

![Figure 4-5. G07 Radius Programming](image)
Programming Display Mode

G07 does not affect the Programming Display Mode. The correct units are interpreted from the part program.

⇒ The Programming Display Mode appears on the screen’s Status bar. Select the D (diameter) on the screen to toggle the mode to R (radius). The display toggles between choices.

*Figure 4–6. WinMax Status bar*
G08 - Diameter Programming (default)

You can program X axis dimensions using the radius or diameter of the part. Program **G08** when you wish to program diameter dimensions.

Default—Yes

Modal—Yes

Cancels—G07

This mode will affect live-tool (milling) when running G codes. When G07 is active, live-tool programming is the same as programming a 3-axis milling machine.

The table below provides information about how the modal setting affects part programs.

<table>
<thead>
<tr>
<th>Programmed Value</th>
<th>G08 Modal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Endpoint of linear move</td>
<td>X use diameter</td>
</tr>
<tr>
<td>Endpoint of circular move</td>
<td>X use diameter</td>
</tr>
<tr>
<td>Center and radius of circular move</td>
<td>IKR use radius</td>
</tr>
<tr>
<td>X axis infeed distances</td>
<td>infeed distance use radius</td>
</tr>
</tbody>
</table>

*Table 4–5. Effects of Using G08*

**G08 Diameter Programming Example**

The following example part program shows arcs programmed with X, Z, I, and K.

G90 G08 X1.6 Z-0.6 I0 K-0.6 *(G90 = Absolute; G08 = Diameter)*

G91 G08 X1.6 Z-0.6 I0 K-0.6 *(G91 = Incremental; G08 = Diameter)*

G08, Diameter programming, specifies that all X-axis positions are interpreted in diameter values. The following figure illustrates diameter programming:

![Figure 4–7. G08 Diameter Programming](image)

The arc center dimension (K) stays the same for both commands, but the X-axis dimension (X) changes. The program may switch between **G07** Radius and **G08** Diameter at any time.
Programming Display Mode

G08 does not affect the Programming Display Mode. The correct units are interpreted from the part program.

Figure 4–8. WinMax Status bar
G09 - Exact Stop

G09 causes the axes to decelerate to zero velocity at the end of a block. Since G09 is non-modal, you must program it in every block that requires the tool to come to a complete stop.

Modal—No

G09 is not used in conjunction with G00 - Rapid Traverse (default), G01 - Linear Interpolation or G02/G03 - Clockwise/Counterclockwise Arc commands.

Format

G09 X ____ Y ____ Z ____ (machine decelerates to stop at specified point.)

G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry)

G10 used in conjunction with specific parameters provides the following data setting functions:

Default—No

Modal—Yes

Cancels G11 - Data Setting Cancel (default)

G10 commands can not be performed by graphics at the same time the program is running.

Work Offsets

G10 L2 is used for setting work coordinate systems. One of six work coordinate systems can be changed as shown below where P is used to select the external work zero point offset value (P = 0), or one of the work coordinate systems (P = 1 to 6), and X, Y, Z is the work zero point offset value of each axis.

For Work Offsets, absolute values (X, Y, Z) replace work offset values. Incremental values (U, V, W) are added to the current work offset values.

Format—Work Offset

G10 L2 P0 or P1-6 X ____ Y ____ Z ____
G10 L2 P1-6 U ____ V ____ W ____
Tool Offset or Tool Geometry

G10 is used for setting tool length and radius offsets. G10 is used with the P and R parameters. P is the offset number 01 through 99, and R is the offset amount which may be absolute or incremental depending on G90 or G91.

For Tool Offsets, absolute values (X, Y, Z) replace tool offset values. Incremental values (U, V, W) are added to the current tool offset values.

Format—Tool Offset or Tool Geometry

\[
\text{G10 P01-99 X } \_\_\_ Y \_\_\_ Z \_\_\_ R \_\_\_ (\text{Tool Offset})
\]

\[
\text{G10 P10001-10099 U } \_\_\_ V \_\_\_ W \_\_\_ C \_\_\_ (\text{Tool Geometry})
\]

Parameters—Tool Offset or Tool Geometry

- **P**—program with a number 01 through 99 for the tool wear offset value.
- **P**—program with a number 10001 to 10099 for the tool geometry offset value.

G11 - Data Setting Cancel (default)

Default—Yes

Modal—Yes

Cancels—G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry)
**G17 - XY Plane**

G17, the XY Plane Selection code, sets the plane for circular interpolation modes G02/ G03 - Clockwise/Counterclockwise Arc.

Default—No

Modal—Yes

G17 cancels G18 - XZ Plane (default) or G19 - YZ Plane.

The three codes in the plane selection group and their relationships to each other are illustrated below:

![Figure 4–9. G17, G18, and G19 Planes](image)

<table>
<thead>
<tr>
<th>Planes</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G17</td>
<td>XY Plane</td>
</tr>
<tr>
<td>G18</td>
<td>XZ Plane (default)</td>
</tr>
<tr>
<td>G19</td>
<td>YZ Plane</td>
</tr>
</tbody>
</table>

**Format**

The format of the XY plane selection command is as follows:

```
G17 X____ Y____
```

⇒ G17 is only available for turning centers with Y-axis motion.

- The X, Y, Z, I, and J words are valid in circular interpolation blocks.
- K words are invalid.
- If a Z word is programmed in the circular interpolation block, a helix is generated in the XY plane.
• The direction of an arc or helix in the XY plane can be determined by looking at the XY plane with positive X to the right and positive Y going up.

• The XY plane is a right-handed coordinate system (thumb points to positive Z, and fingers wrap in counterclockwise direction).

• In G17, the arc end point is defined by the X and Y words in the block. The arc center point is defined by the I and J words in the block.

Example

The diagram below illustrates XY plane selection:

*Figure 4–10. XY Plane Selection*
G18 - XZ Plane (default)

G18, the XZ Plane Selection code, is the default and sets the plane for the circular interpolation modes G02/G03 - Clockwise/Counterclockwise Arc to the XZ plane.

Default—Yes

Modal—Yes

G18 cancels G17 - XY Plane or G19 - YZ Plane.

Format

The format of the XZ plane selection command is as follows:

G18 Z____ X____

- The X, Y, Z, I, and K words are valid in circular interpolation blocks.
- J words are invalid.
- If a Y word is programmed in the circular interpolation block, a helix is generated in the XZ plane.
- The direction of an arc or helix in the XZ plane can be determined by looking at the XZ plane with positive X to the right and positive Z going up.
- Basic NC and ISNC handle the XZ plane differently. For Basic NC, the XZ plane is a left-handed coordinate system (thumb points to positive Y, and fingers wrap in clockwise direction). For ISNC, the XZ plane is a right-handed coordinate system (thumb points to positive Y, and fingers wrap in a counterclockwise direction).
- In G18, the arc end point is defined by the X and Z words in the block. The arc center point is defined by the I and K words in the block.

Example

The diagram below illustrates XZ plane selection:

![Figure 4–11. G18 XZ Plane Selection](image)
G19 - YZ Plane

G19, the YZ Plane Selection code, sets the plane for circular interpolation modes G02/ G03 - Clockwise/Counterclockwise Arc to the YZ plane.

⇒ G19 is only available for turning centers with Y-axis motion.

Default—No

Modal—Yes

G19 cancels G17 - XY Plane or G18 - XZ Plane (default).

- The X, Y, Z, J, and K words are valid in circular interpolation blocks.
- I words are invalid.
- If an X word is programmed in the circular interpolation block, a helix is generated in the YZ plane.
- The direction of an arc or helix in the YZ plane can be determined by looking at the YZ plane with positive Y to the right and positive Z going up.
- The YZ plane is a right-handed coordinate system (thumb points to positive X, and fingers wrap in counterclockwise direction).
- In G19, the arc end point is defined by the Y and Z words in the block. The arc center point is defined by the J and K words in the block.

Format

The format of the YZ plane selection command is as follows:

G19 Y____ Z____

Example

The diagram below illustrates YZ plane selection:

Figure 4–12. G19 YZ Plane Selection
G20 - Inch Mode (default)

G20 sets Inch mode. Program this command in the first block of each part program that uses inches for the dimension.

Default—Yes

Modal—Yes

Cancels—G21 - Millimeter Mode

Format

The default format for Inch mode for different types of blocks is shown in the table below:

<table>
<thead>
<tr>
<th>Blocks</th>
<th>Default Format</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometric Entries X, Z, I, K, R</td>
<td>3.4</td>
<td>-845.0000 to 845.0000</td>
</tr>
<tr>
<td>Feedrate (Feed/min) and (Feed/rev) F</td>
<td>3.4</td>
<td>000.0001 to 845.0000</td>
</tr>
<tr>
<td>Thread Lead (distance/rev) K</td>
<td>2.6</td>
<td>00.000001 to 10.000000</td>
</tr>
</tbody>
</table>

Table 4–6. Inch Mode formats

The default formats are defined as follows:

- 3.4 represents 3 places to the left of the decimal point and 4 places to the right.
- 2.6 represents 2 places to the left of the decimal point and 6 places to the right.

Unit Display Mode

G20 does not affect the Unit Display Mode as viewed in the position or offset screens. The correct units are interpreted from the part program.
**G21 - Millimeter Mode**

G21 sets Millimeter mode. Program this command in the first block of each part program that uses millimeters for the dimension.

Default—No

Modal—Yes

Cancels—G20 - Inch Mode (default)

**Format**

The default format for Millimeter mode for different types of blocks is shown in the table below:

<table>
<thead>
<tr>
<th>Blocks</th>
<th>Default Format</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometric Entries X, Z, I, K, R</td>
<td>5.3</td>
<td>-21463.000 to 21463.000</td>
</tr>
<tr>
<td>Feedrate (Feed/min) and (Feed/rev) F</td>
<td>5.3</td>
<td>0000.001 to 21463.000</td>
</tr>
<tr>
<td>Thread Lead (distance/rev) K</td>
<td>3.5</td>
<td>000.00001 to 10.000000</td>
</tr>
</tbody>
</table>

*Table 4–7. Millimeter Mode formats*

The default formats are defined as follows:

- 5.3 represents 5 places to the left of the decimal point and 3 places to the right.
- 3.5 represents 3 places to the left of the decimal point and 5 places to the right.

**Unit Display Mode**

G21 does not affect the Unit Display Mode as viewed in the position or offset screens. The correct units are interpreted from the part program.
G28 - Automatic Return to Reference Point

Any point within the machine coordinate system can be selected as the reference point. The return reference point is often used to move the turret to the maximum machine position at the end of the program to get the turret out of the way to load and unload parts.

The G28 Automatic Return To Reference Point command specifies an automatic return to the reference point for the designated axes. An intermediate point can be specified with the X____Y____Z____ parameters. If no intermediate point coordinates are specified, the system uses the previous intermediate point coordinates. If no intermediate point coordinates are specified during the current program execution, the machine returns directly to the reference point.

Modal—No

Format

G28 U____ V____ W____

- X indicates Absolute X and (u) indicates Incremental X.
- Y indicates Absolute Y and (v) indicates Incremental Y.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters

- V—used with machines configured with Y-axis motion.

A milling map must be active in order for V parameter to be used, if it is not, the machine will be unable to complete the motion.

For information about milling maps, please refer to the following sections:

- G101 - Axial Surface of the Part - Main Spindle without Linear Y
- G102 - Radial Surface of the Part – Main Spindle without Linear Y
- G103 - Axial Surface of the Part – Main Spindle with Linear Y
- G104 - Radial Surface of the Part – Main Spindle with Linear Y
- G105 - Axial Surface of the Part – Sub-spindle without Linear Y
- G106 - Radial Surface of the Part – Sub-spindle without Linear Y
- G107 - Axial Surface of the Part – Sub-spindle with Linear Y
- G108 - Radial Surface of the Part – Sub-spindle with Linear Y
- G109 - Turning / Lathe Mode
Examples

These parameters specify the absolute or incremental location of the intermediate point in coordinates relative to the current coordinate system. The G28 command is only performed for the axis which follows the G28. For example, if an X value follows the G28, the machine moves to the X reference point, not the Y or Z reference point.

As another example, a typical method to send the turret home using incremental moves is shown below. This example will send the turret to the maximum machine position.

G28 U0. W0.
G32 - Constant Lead Thread Cutting

G32 allows you to perform single edge ID and OD thread cutting. The cutting amount per depth is held constant, and either straight or tapered threads can be cut using this cycle. Refer to G78 - Threading Cycle, on page 4 - 66 for information about programming the more complex, multi-pass Threading Cycle for OD and ID threads.

Default—No
Modal—Yes

Cancels:
- G33 - Threading
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

The starting angle is measured in 0.001 degree increments. If no decimal point is specified the scale factor is 0.001. For example, Q180000 is equivalent to 180.000 degrees.

This cycle can be interrupted.
- When the Feed Hold button is pressed, cutting will continue until the first non-thread cutting move. Then the tool will retract to Z start after completing the lead out move for the current depth of cut.
- When the Interrupt Cycle button is pressed, the tool will complete the motion for the G32 move as well as any thread cutting move following that line, and then stop the spindle and turn off the coolant.

The feedrate override is fixed at 100% during thread cutting.

When multiple threading commands are issued in sequence, the control will not wait for a spindle before continuing the next thread command.

Format

\[
\text{G32 X(u) ____ Z(w) ____ F or E ____ Q ____}
\]

- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters
- **X(u)**—X end point
- **Z(u)**—Z end point; only used for tapered threading.
- **F or E**—pitch
- **Q**—thread starting angle
Example

The following figure illustrates G32:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Straight Thread</td>
</tr>
<tr>
<td>2</td>
<td>Tapered Thread</td>
</tr>
</tbody>
</table>

*Figure 4–13. G32 Constant Lead Thread Cutting*
G33 - Threading

G33 allows you to program a simple threading cycle. Refer to G78 - Threading Cycle, on page 4 - 66 for information about programming the more complex, multi-pass Threading Cycle for OD and ID threads.

Modal—No

Cancels:

- G32 - Constant Lead Thread Cutting
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

Format

G33 X ____ Z ____ (F ____ or E ____)

Parameters

- Z—only used for tapered threading.
- F—pitch
- E—thread lead

Example—G33

To program a threading move, X and Z and either I or K are required. P (the thread start angle) and F (feedrate) may be specified.

1. Program the thread’s endpoint (using Z only, X only, or a combination of X and Z for a taper thread).
2. Program the pitch of the thread using either of these codes:
   - F (modal feedrate)
   - K (the pitch along the Z axis)
   - I (the pitch along the X axis)

If you are programming a tapered thread (both X and Z axes move), program the lead (I or K) for the axis with the longest move.

G00 X3.5 Z4 (Position for thread)
G33 Z1 K0.2 (Cut thread)
G01 X3.6 (Pullout of thread)
Example—G33 with P Value

You can also start a thread at a given angle. The angle value may range from -360.0000 to 360.0000. A zero angle starts the thread at the index mark of the spindle encoder, and is the default. The P value is not modal and must appear in the same block as the G33. The thread start angle is specified by a P value in a G33 block, such as:

G00 X3.5 Z4  \textit{(Position for thread)}
G33 Z1 K0.2 P180 \textit{(Cut thread using 180° thread start angle)}
G01 X3.6 \textit{(Pullout of thread)}
Cutter Compensation (G40–G42)

Cutter compensation may be used for two purposes. First, it may be used when the dimensions in the program and the part surface are the same. The system calculates the proper tool path by using the part surface and the tool diameter information.

Second, cutter compensation corrects the difference between the diameters of the tool specified and the tool actually used to cut the part. This situation often occurs when the program originates from an off-line device. Note that the coordinates of those programs are usually tool center line data.

Cutter compensation is based on the direction of travel of the tool. To determine which type of cutter compensation to use, look at the part as if you are moving around the part always keeping the tool ahead of you. Then it becomes obvious whether the tool needs to be on the right or the left of the programmed line or the boundary of the part.

![Figure 4–14. G40 Cutter Compensation Off](image1)

![Figure 4–15. G41 Cutter Compensation Left](image2)

![Figure 4–16. G42 Cutter Compensation Right](image3)

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>A</strong></td>
<td>Beginning point of programmed direction (clockwise tool movement from point A to point B)</td>
</tr>
<tr>
<td><strong>B</strong></td>
<td>Ending point of programmed direction (clockwise tool movement from point A to point B)</td>
</tr>
</tbody>
</table>
G40 - Cutter Radius Compensation Off (default)

G40 turns off cutter compensation. When G40 is active, the tool center will move along the path defined in the part program. Refer to Figure 4–14. G40 Cutter Compensation Off, on page 4 - 31 for an illustration of the tool path with G40.

Default—Yes

Modal—Yes

Cancels G41 - Cutter Radius Compensation Left or G42 - Cutter Radius Compensation Right

G40 must be active to perform G53 - Machine Coordinates.

G00 - Rapid Traverse (default) or G01 - Linear Interpolation must be selected in order for this command to cancel the offset compensation. Each axis moves straight (G01) or at rapid traverse (G00) from the point of the old vector at the start point toward the end point. The machine should be in G40 mode before the end of a program. Otherwise, when the program ends in the offset mode, positioning cannot be made to the terminal point of the program, and the tool position will be separated from the terminal position by the vector value.

Examples

Please refer to Examples—Cutter Compensation, on page 4 - 33.

G41 - Cutter Radius Compensation Left

G41 turns cutter compensation on to the left of the programmed part profile. When G41 is active, the tool path is offset by the tool radius value stored in the tool table. The offset will be to the left side of the part profile. Refer to Figure 4–15. G41 Cutter Compensation Left, on page 4 - 31 for an illustration of the tool path with G41.

Default—No

Modal—Yes

Cancels G40 - Cutter Radius Compensation Off (default), G42 - Cutter Radius Compensation Right

Either G00 - Rapid Traverse (default), G01 - Linear Interpolation, or G02/G03 - Clockwise/Counterclockwise Arc must be active.

Examples

Please refer to Examples—Cutter Compensation, on page 4 - 33.
G42 - Cutter Radius Compensation Right

G42 turns cutter compensation on to the right of the programmed part profile. When G42 is active, the tool path is offset by the tool radius value stored in the tool table. The offset will be to the right side of the part profile. Refer to Figure 4–16. G42 Cutter Compensation Right, on page 4 - 31 for an illustration of the tool path with G42.

Default—No

Modal—Yes

Cancels G40 - Cutter Radius Compensation Off (default), G41 - Cutter Radius Compensation Left

Either G00 - Rapid Traverse (default), G01 - Linear Interpolation, or G02/G03 - Clockwise/Counterclockwise Arc must be active.

Examples—Cutter Compensation

The following examples use the part program below. The figures illustrate the tool path with G40 and with G42. Notice that the tool travels very close to the part with G40. The nose radius of the tool programmed in Tool Setup determines the space between the part and the tool.

Figure 4–17. Cutter Compensation Off (G40) and Right (G42)

N10 (msg, lathe doc sample, abs G42, tool rad 0.1)
N20 G90 G40 G94 T0000 (absolute mode, cutter compensation off, offset 0)
N30 G07 G00 X2.5 T0101 (radius programming, rapid to X, tool 1 offset 1)
N40 Z3 (rapid to Z)
N60 X.5 (rapid to X)
N70 G96 R.5 S350 M03 (CSS, axis at R.5 in, 350 SFM and clockwise)
N80 G01 G95 F.01 X0 (face, IPR 0.01)
N120 G00 Z4 (rapid to Z)
N125 G42 (cutter compensation right)
N130 X.5 Z3  (rapid to XZ)
N160 G01 X1 Z2
N170 X1.7
N180 Z0.3
N190 X1.9
N200 Z0
N230 G00 G40 X2.5  (cutter compensation off, rapid to X)
N240 Z3  (rapid to Z)
N900 M30  (end of program, rewind)
G50 - Set Part Setup or Set Max RPM

G50 used in conjunction with specific parameters provides the following programming functions.

ISNC Mode A
Modal—No

Set Part Setup

G50 sets part setup RPM when an S Parameter is not specified. Please also refer to G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle, on page 4 - 89 for information about using G92 in setting work offsets.

Set Part Setup Format

G50 X ____ Y ____ Z ____

Set Part Setup Parameters

• No S parameter is used.

Set Part Setup Example

Set Max RPM

G50 in conjunction with an S Parameter sets the Maximum Spindle RPM. Please also refer to G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle, on page 4 - 89 for information about using G92 in setting maximum spindle RPM.

G97 - Direct Spindle Speed (default) must be active.

Set Max RPM Format

G50 S ____

Set Max RPM Parameters

• S—spindle speed (for Max RPM)

Set Max RPM Example

G97
G50 S1000
G94
G53 - Machine Coordinates

This one-shot usage code causes the part program to ignore the following offsets for one block so you can program the axes to a position that you define using machine coordinates:

- active work coordinate offsets (G50 - Set Part Setup or Set Max RPM, G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle)
- active tool offset
- active fixture offset

Default—No
Modal—No

Cancels:

- G54-G59 - Work Coordinates (G54 default)
- G54.1 - G54.93 - Aux Work Coordinates
- G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

Before you program G53 in a block, the program must meet the following conditions:

- G40 - Cutter Radius Compensation Off (default) must be active.
- G90 - Absolute Programming (default) must be active if using X instead of U.
- Either G00 - Rapid Traverse (default) or G01 - Linear Interpolation mode must be active.

Format

G53 X(u) ____ Y(v) ____ Z(w) ____

Example

N10 G90 G00 Z0 X0  (Absolute Mode)
N30  G53 22 X1 (Move to X1 Z2)

When the N30 block is finished, the machine will return to the G90 - Absolute Programming (default) mode.
G54-G59 - Work Coordinates (G54 default)

These modal commands select the work coordinate systems 1–6. The work coordinate systems are affected by the work offsets, the shift offset, and the G50 - Set Part Setup or Set Max RPM or G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle command. Coordinate system 1 is the same as the part setup and it is the default coordinate system. Coordinate systems 1–6 are established by manually entering work offset values for G55–G59 on the Work Offset screen or with the G10 command.

Default—Yes
Modal—Yes

Cancels:

- G53 - Machine Coordinates
- G54-G59 - Work Coordinates (G54 default)
- G54.1 - G54.93 - Aux Work Coordinates
- G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

Use the G10 - Data Setting (Work Offsets, Tool Offsets, or Tool Geometry) command to change work coordinate systems, and use the G50 - Set Part Setup or Set Max RPM or G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle command to set part zero. All six work coordinate systems can be moved an equal distance and direction by using the G50 or G92 command.

Format

The format of the multiple work coordinates command is as follows:

G54 (Select work coordinate system 1)
G55 (Select work coordinate system 2)
G56 (Select work coordinate system 3)
G57 (Select work coordinate system 4)
G58 (Select work coordinate system 5)
G59 (Select work coordinate system 6)
Example

When in NC mode, the Part Setup screen displays up to six work coordinates (G54–G59). As shown below, these codes are used to set multiple part zeroes for multiple parts using the same part program.

![Diagram showing G54-G59 work offsets for multiple parts]

<table>
<thead>
<tr>
<th></th>
<th>Machine Zero</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Part Zero</td>
</tr>
</tbody>
</table>

Figure 4–18. G54-G59 Work Offset G Codes for Multiple Parts

The coordinates defining G54 are the part zero coordinates for the original part defined on the Part Setup screen. Set the X, Z, C, W, and Y values for the G54 to G59 codes. These work offsets are stored in memory, but not with the part program.

The G54 work offsets are the same registers as those in the Part Setup screen for Part Zero X, Z, C, W, and Y. Editing G54 work offsets for multiple coordinate systems updates the part setup for X, Z, C, W, and Y on the Part Setup screen.
G54.1 - G54.93 - Aux Work Coordinates

There are 93 additional X, Z, C, W, and Y work offsets available in NC programming.

Default—No

Modal—Yes

Cancels:

- G53 - Machine Coordinates
- G54-G59 - Work Coordinates (G54 default)
- G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

Format

G54.1 P1- P93 X ____ Y ____ Z ____

Parameters

- P—auxiliary work offset

Examples

To access any of these offsets call G code G54.1 Pn, where n is 1 through 93. For example, to change to auxiliary work offset 46,

G54.1 P46

To update work offset values, use data setting G code G10 Pn to set the Auxiliary work offsets values. For example, to update work offset 46,

G10 P46 X12.5 Y3.0 Z-0.5
**G61 - Precision Cornering**

Precision cornering causes the subsequent line to come to an exact stop.

Default—No

Modal—Yes

Cancels G64 - Precision Cornering Cancel (default)

**G64 - Precision Cornering Cancel (default)**

Default—Yes

Modal—Yes

Cancels G61 - Precision Cornering.

**G65 - Subprogram Call**

The G65 subprogram command has the following form:

```
G65 P____ L_____ [followed by optional arguments]
```

The P represents the subprogram number and the L represents the number of iterations that the subprogram must perform. These two methods of argument passing can be used together.

**Arguments**

In a G65 subprogram call, the local variables in the calling program are not copied to the local variables in the called subprogram. Arguments which follow the G65 command are copied to the local variables in the subprogram as illustrated in the following command:

```
G65 P5080 A0.0 B8 C2.3 S6 T2 H81 I9 J3.5 K0 Z-1 R.1
```

The value which follows A is copied to the local variable #1 in the subprogram. *Table 4–8. Macro Mode B Local Variables and Subprogram Arguments, on page 4 - 41* shows the relationships between the subprogram arguments and the local variables in the subprograms.
Multiple Arguments

Multiple I, J, and K arguments can also be used as subprogram arguments. For example, if three I arguments are used in the subprogram call, the first I maps to the #4 variable, the second I maps to the #7 variable, and the third I maps to the #10 variable. The following subprogram call is legitimate:


Only numbers may be used as arguments in a G65 subprogram call; no variables or expressions can be used. If multiple iterations of the subprogram are to be performed, the local variables will be initialized to the same argument values.

Passing Argument Lists to Subprograms in Macro Mode B

There are several methods for passing arguments and parameters to subprograms. The G65 - Subprogram Call and G66 - Modal Subprogram Call subprogram calls allow an argument list to be provided after the G65 and G66, respectively. The user defined M Code and the user defined G Code allow an argument list to be provided after the user defined Code. The argument list consists of various letters followed by values. The values are then stored as local variables within the subprogram.

The table below lists the correspondence between the arguments and the local variables in Macro Mode B. The argument list is optional. Any arguments which are not included in the list are given vacant status.

<table>
<thead>
<tr>
<th>Local Variables</th>
<th>Subprogram Arguments</th>
<th>Local Variable</th>
<th>Subprogram Arguments</th>
</tr>
</thead>
<tbody>
<tr>
<td>#1</td>
<td>Argument A</td>
<td>#18</td>
<td>Argument R or K5</td>
</tr>
<tr>
<td>#2</td>
<td>Argument B</td>
<td>#19</td>
<td>Argument S or I6</td>
</tr>
<tr>
<td>#3</td>
<td>Argument C</td>
<td>#20</td>
<td>Argument T or J6</td>
</tr>
<tr>
<td>#4</td>
<td>Argument I or I1</td>
<td>#21</td>
<td>Argument U or K6</td>
</tr>
<tr>
<td>#5</td>
<td>Argument J or J1</td>
<td>#22</td>
<td>Argument V or I7</td>
</tr>
<tr>
<td>#6</td>
<td>Argument K or K1</td>
<td>#23</td>
<td>Argument W or J7</td>
</tr>
<tr>
<td>#7</td>
<td>Argument D or I2</td>
<td>#24</td>
<td>Argument X or K7</td>
</tr>
<tr>
<td>#8</td>
<td>Argument E or J2</td>
<td>#25</td>
<td>Argument Y or I8</td>
</tr>
<tr>
<td>#9</td>
<td>Argument F or K2</td>
<td>#26</td>
<td>Argument Z or J8</td>
</tr>
<tr>
<td>#10</td>
<td>Argument I3</td>
<td>#27</td>
<td>Argument K8</td>
</tr>
<tr>
<td>#11</td>
<td>Argument H or J3</td>
<td>#28</td>
<td>Argument I9</td>
</tr>
<tr>
<td>#12</td>
<td>Argument K3</td>
<td>#29</td>
<td>Argument J9</td>
</tr>
<tr>
<td>#13</td>
<td>Argument M or I4</td>
<td>#30</td>
<td>Argument K9</td>
</tr>
<tr>
<td>#14</td>
<td>Argument J4</td>
<td>#31</td>
<td>Argument I10</td>
</tr>
<tr>
<td>#15</td>
<td>Argument K4</td>
<td>#32</td>
<td>Argument J10</td>
</tr>
<tr>
<td>#16</td>
<td>Argument I5</td>
<td>#33</td>
<td>Argument K10</td>
</tr>
<tr>
<td>#17</td>
<td>Argument Q or J5</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 4–8. Macro Mode B Local Variables and Subprogram Arguments
Layering of Local Variables within Subprogram Calls

M98 subprogram calls use local variables differently from other subprogram calls since the called subprogram does not get a new set of local variables. Changes made to the local variables within the current subprogram will be retained when the calling program is re-instated.

Changes can be made to the local variables within the current subprogram, but when program execution returns to the calling program, the values of the local variables of the calling program are reinstated. The local variables in the subprogram can be changed, however, without affecting the local variables in the calling program. With other subprogram calls, unless an argument list is passed to the subprogram, the local variables are given vacant status.

Specifying Subprogram Iterations

The number of iterations for a subprogram to perform are specified with G65, G66, and M98 subprogram calls.

Using G65 and G66

When making G65 and G66 subprogram calls, the L parameter is used to specify iterations. The maximum number of iterations which can be specified with the G65 and G66 subprogram calls is 999.

Using M98

When making M98 subprogram calls, the P parameter is used to specify iterations as well as the subprogram number. Up to four digits can be used to specify iterations for a maximum of 9999 iterations. Leading zeros are not required when specifying iterations; however, leading zeros are required with a subprogram number that is less than 1000.
G65 Subprogram Example

In the following example a line of holes will be drilled along a line. The type of canned cycle can be determined along with the distance between the holes in both the X and Y axes. The type of canned cycle and various canned cycle parameters can also be set.

ISNC Part Program 1 Inch
BOLT_LN.LNC

O4000
T1 M06
M03 G00 G90 X0 Y0 Z0 S1800
/
(B REPRESENTS THE NUMBER OF BOLT HOLES)
(H REPRESENTS THE DESIRED CANNED CYCLE)
(X,Y REPRESENTS THE INCREMENTAL DISTANCE BETWEEN HOLES)
(Z REPRESENTS THE HOLE DEPTH)
(R REPRESENTS THE R PLANE LEVEL)
/
#500=99 (RETURN TO R LEVEL)
G65 P5070 X.5 Y.75 B10 H81 Z-1 Q0. R.1 F20.
M30
(**********************************************************************)
(BOLT HOLE LINE PATTERN - SUBPROGRAM 5070)
(**********************************************************************)
O5070
(#2 IS THE NUMBER OF HOLES)
(#11 IS THE CANNED CYCLE NUMBER)
(#26 IS THE HOLE DEPTH)
(#500 IS 99 FOR RETURN TO R LEVEL MODE OR 98 FOR RETURN TO INITIAL POINT)
(#5003 IS THE Z COORDINATE BEFORE THE CANNED CYCLE IS PERFORMED)
/
WHILE [#2GT0] DO250
G91 G#500 G#11 Z#26 Q#17 R[#5003-#18] F#9
G00 X#24 Y#25
#2 = #2-1
END250
M99

View the part using the Verify console key to verify that the part is programmed correctly.
Macro Instruction (G65)

G65 Macro instructions are G65 commands which are used to perform mathematical, trigonometric, or program control functions instead of subprogram calls. These commands are intended to support existing programs which use this program format.

The value in the H parameter defines the operation being performed. In all instructions except the GOTO commands H80 through H86, a variable number follows the P parameter. The operation’s result is stored in that variable number. In the following command the value stored in variable #100 is added to the number 1 and the resultant value is stored in variable #115.

G65 H02 P#115 Q#100 R1

For the GOTO commands, the value which follows the P is a positive or negative integer. If the number is negative, the software begins searching for the sequence number at the beginning of the file and continues to search for the sequence number until reaching the end of the file. If the number is positive, the search for the sequence numbers begins with the block after the GOTO command and continues until reaching the end of the file. The software then searches from the beginning of the file until reaching the GOTO command block.

The values which follow Q and R are general purpose parameters which are used in mathematical, logical, or GOTO operations. The specific operations are listed in the following table.

Format

The following is the G65 Macro Instruction format:

G65 H ____, P #a, Q #b, R #c.

For H80 through H86, if “a” has a negative value, the software performs a GOTO but begins looking for the sequence number at the beginning of the program. No variables can be used for the P parameter for H80 through H86.

The G65 Macro Instructions are intended to support existing Macro A programs. Use equations and regular GOTO statements in place of these instructions when developing new programs.

For example use #100 = 4.56 OR #110 instead of G65 #11 P#100 Q4.56 R#110.

And use IF [#150 EQ #160] GOTO 100 instead of G65 H81 P100 Q#150 R#160.

These commands can be used in either Macro A or B mode.
This table lists the Descriptions and Instruction Functions for the H codes used in the G65 macro instructions:

<table>
<thead>
<tr>
<th>H Code</th>
<th>Description</th>
<th>Instruction Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>H01</td>
<td>Definition, Substitution</td>
<td>#a = #b</td>
</tr>
<tr>
<td>H02</td>
<td>Addition</td>
<td>#a = #b + #c</td>
</tr>
<tr>
<td>H03</td>
<td>Subtraction</td>
<td>#a = #b - #c</td>
</tr>
<tr>
<td>H04</td>
<td>Product</td>
<td>#a = #b * #c</td>
</tr>
<tr>
<td>H05</td>
<td>Division</td>
<td>#a = #b / #c</td>
</tr>
<tr>
<td>H11</td>
<td>Logical Sum</td>
<td>#a = #b .OR. #c</td>
</tr>
<tr>
<td>H12</td>
<td>Logical Product</td>
<td>#a = #b .AND. #c</td>
</tr>
<tr>
<td>H13</td>
<td>Exclusive OR</td>
<td>#a = #b .XOR. #c</td>
</tr>
<tr>
<td>H21</td>
<td>Square Root</td>
<td>#a = Ö#b</td>
</tr>
<tr>
<td>H22</td>
<td>Absolute Value</td>
<td>#a =</td>
</tr>
<tr>
<td>H23</td>
<td>Remainder</td>
<td>#a = #b - trunc (#b/#c) * #c; trunc: discard fractions less than 1.</td>
</tr>
<tr>
<td>H24</td>
<td>Conversion from BCD to Binary</td>
<td>#a = BIN(#b)</td>
</tr>
<tr>
<td>H25</td>
<td>Conversion from binary to BCD</td>
<td>#a = BCD(#b)</td>
</tr>
<tr>
<td>H26</td>
<td>Combined Multiplication/Division</td>
<td>#a = (#a * #b) / #c</td>
</tr>
<tr>
<td>H27</td>
<td>Combined Square Root 1</td>
<td>#a = Ö(#b2 + #c2)</td>
</tr>
<tr>
<td>H28</td>
<td>Combined Square Root 2</td>
<td>#a = Ö(#b2 - #c2)</td>
</tr>
<tr>
<td>H31</td>
<td>Sine</td>
<td>#a = #b * SIN(#c)</td>
</tr>
<tr>
<td>H32</td>
<td>Cosine</td>
<td>#a = #b * COS (#c)</td>
</tr>
<tr>
<td>H33</td>
<td>Tangent</td>
<td>#a = #b * TAN(#c)</td>
</tr>
<tr>
<td>H34</td>
<td>Arc tangent</td>
<td>#a = TAN(#b/#c)</td>
</tr>
<tr>
<td>H80</td>
<td>Unconditional Divergence (GOTO)</td>
<td>GOTO a</td>
</tr>
<tr>
<td>H81</td>
<td>If Statement, Equals</td>
<td>IF #b = #c, GOTO a</td>
</tr>
<tr>
<td>H82</td>
<td>If Statement, Not Equal</td>
<td>IF #b ¹ #c, GOTO a</td>
</tr>
<tr>
<td>H83</td>
<td>If Statement, Greater Than</td>
<td>IF #b &gt; #c, GOTO a</td>
</tr>
<tr>
<td>H84</td>
<td>If Statement, Less Than</td>
<td>IF #b &lt; #c, GOTO a</td>
</tr>
<tr>
<td>H85</td>
<td>If Statement, Greater Than/Equals</td>
<td>IF #b &gt;= #c, GOTO a</td>
</tr>
<tr>
<td>H86</td>
<td>If Statement, Less Than/Equals</td>
<td>IF #b &lt;= #c, GO TO a</td>
</tr>
</tbody>
</table>

*Table 4–9. H Code Descriptions and Instruction Functions for G65 Macro Instructions*
Example

This example shows how to use G65 macro instructions in a Bolt Hole Circle subprogram:

**ISNC Part Program 1 Inch**

**G65INST.LNC**

```
%
(#600 IS BOLT HOLE CIRCLE X COORD)
(#601 IS BOLT HOLE CIRCLE Y COORD)
(#602 IS BOLT HOLE CIRCLE RADIUS)
(#603 IS STARTING ANGLE)
(#604 IS NUMBER OF BOLT HOLES)
#600 = 0
#601 = 0
#602 = 2.
#603 = 30.
#604 = 12.
T1 M6
G00 X0 Y0 Z0.05
M98 P3030
G00 X0 Y0 Z0.05
M02
O3030
(#110 IS BOLT HOLE COUNTER)
(#112 IS ANGLE OF CURRENT HOLE)
(#113 IS X COORD OF CURRENT HOLE)
(#114 IS Y COORD OF CURRENT HOLE)
N10 G65 H01 P#110 Q0
G65 H22 P#111 Q#604
N20 G65 H04 P#112 Q#110 R360
G65 H05 P#112 Q#112 R#604
G65 H02 P#112 Q#603 R#112
G65 H32 P#113 Q#602 R#112
G65 H02 P#113 Q#600 R#113
G65 H31 P#114 Q#602 R#112
G65 H02 X#114 Q#601 R#114
G90 H00 X#113 Y#114
G81 Z-1. F20.
G80
G65 H02 P#110 Q#110 R1
G65 H84 P-20 Q#110 R#111
M99
```
G66 - Modal Subprogram Call

Default—No

Modal—Yes

Cancels—G67 - Modal Subprogram Call Cancel (default).

Modal subprograms are executed every time a motion is performed (i.e., after a Move command). Use them for performing repetitive tasks at different locations. The repetitive tasks can be put inside a modal subprogram. A subprogram call can be made to a program which contains X, Y, and Z locations and will be executed at each of these locations.

A modal subprogram will not be modal within another modal subprogram. If the modal subprogram contains Move commands, the modal subprogram will not be performed after Move commands within the modal subprogram. This allows Move commands to be contained within modal subprograms.

These methods allow the subprogram call to be modal:

- A Modal Subprogram Call (G66 - Modal Subprogram Call) Command
- A Modal user defined G code

In a G66 Modal subprogram call, the subprogram is repeatedly executed after each Move command until the G67 - Modal Subprogram Call Cancel (default) command is performed.
Modal Subprogram Call (G66) Example

The following program draws a series of squares and rectangles:

**NC Part Program 1 Inch**
**G66.LNC**

%
(EXAMPLE OF MODAL SUBPROGRAM CALL G66)
(P6010 IS USED AS MODAL SUBPROGRAM)
(THE VALUES AFTER I AND J ARE PASSED TO)
(THE SUBPROGRAM. THE SUBPROGRAM IS ONLY)
(EXECUTED AFTER BLOCKS WITH MOVE COMMANDS.)

X0 Y0 Z0
X5 G66 P6010 I1. J1.5
Y-3
X-5
Y0

(MODAL SUBPROGRAM IS NOW CANCELED WITH G67)
G67
Y3

(THE MODAL SUBPROGRAM IS STARTED AGAIN WITH NEW PARAMETERS.)
Y0
Y-2
M02

:6010

(THE SUBPROGRAM CREATES A SIMPLE BOX SHAPE.)
G91
X#4
Y#5
X-#4
Y-#5
G90
M99

View the part using the Verify console key to verify that the part is programmed correctly.
G67 - Modal Subprogram Call Cancel (default)

The G67 command is used to cancel modal subprograms initiated with either the G66 or with a modal user defined G code.

Default—Yes
Modal—Yes
Cancels—G66 - Modal Subprogram Call

G70 - Finishing Cycle

G70 causes the tool to make a finish pass along the profile. Individual feeds and speeds are used in the profile. Remaining roughing stock is removed.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

Format

G70 P ____ Q ____ F ____ S ____ T ____

Parameters

- P—first sequence number of Profile
- Q—last sequence number of Profile
- F—cutting feedrate
- S—spindle speed
- T—tool
Example

The following sample program and figure illustrate G70:

```
% O1234
G21
G54
T303
G0 X52 Z2
G90
G94
G96 S190
G70 P10 Q40 F200
N10 G1 G42 X50 Z0
N20 W-10
N30 G2 U6 Z-13 I3 K0 F150
N40 G40 G1 X60 Z-11
M8
M5
M30
%
```

![Figure 4–19. G70 Finish Cycle](image)

*Figure 4–19. G70 Finish Cycle*
G71 - Stock Removal in Turning

G71 removes stock in a turning move.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

G71 allows Linear and Circular interpolation.

Sign of U and W

The sign of the U and W parameters determines the stock removed from point B to C, as shown below:

![Diagram of stock removal](image)

*Figure 4–20. Sign of U and W determines stock removal*

Format

\[
\begin{align*}
\text{G71 } & \ U \ \ \ R \\
\text{G71 } & \ P \ \ \ Q \ \ \ U \ \ \ W \ \ \ F \ \ \ S \ \ \ T
\end{align*}
\]

Parameters

- **U**—depth of cut (U/2 specified as diameter)
- **R**—retract distance (retracts at 45° angle)
- **P**—first sequence number of Profile
- **Q**—last sequence number of Profile
- **U**—second line with U is X stock finish (or finish tolerance)
- **W**—Z stock finish (or finish tolerance)
- **F**—cutting feedrate for roughing
- **S**—roughing speed
- **T**—tool with offset xxxx
Example

The following sample program and figure illustrate G71:

%  
O0000  
G0 G21  
G54  
G28 U0. W0.  
T0303  
G97 S1700 M3  
Z2.54  
X104.14  
G71 U4. R2  
G71 P5 Q10 U2 W3 F200  
N5 G1 G42 X57.15 Z 20.0  
N6 X57.15 Z-25.4  
N7 X69.85 S500  
N8 X82.55 Z-38.1  
N9 Z-44.45 F190  
N10 G40 X101.6 S300  
G0 X104.14 Z2.54  
M9  
(FINISH WITH A DIFFERENT TOOL)  
G28 U0. W0.  
M8  
G70 P5 Q10 T1212  
G0 X104.14 Z2.54  
G28 U0. W0.  
M30  
%
Figure 4–21. G71 Stock Removal in Turning

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Cutting depth</td>
</tr>
<tr>
<td>2</td>
<td>Retract distance</td>
</tr>
<tr>
<td>3</td>
<td>Finish tolerance in X</td>
</tr>
<tr>
<td>4</td>
<td>Finish tolerance in Z</td>
</tr>
</tbody>
</table>
G72 - Stock Removal in Facing

G72 removes stock in a facing move. The cutting passes are determined by the profile.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

G72 allows Linear and Circular interpolation.

Format

G72 U ____ R ____

G72 P ____ Q ____ U ____ W ____ F ____ S ____ T ____

Parameters

- **U**—depth of cut
- **R**—retract distance (retracts at 45° angle)
- **P**—first sequence number of Profile
- **Q**—last sequence number of Profile
- **U**—second line with U is X stock finish (or finish tolerance)
- **W**—Z stock finish (or finish tolerance)
- **F**—cutting feedrate for roughing
- **S**—roughing spindle speed
- **T**—tool with offset xxxx

Example

The following sample program and figure illustrate G72:

```
O1000
G21
T0303
G97 S1700 M3
X115
Z2
G72 U6 R1
G72 P5 Q10 U-2 W-2 F250
N5 G1 X115 Z-25
X105
X85
X75 Z-12
X25
Z0.
```

%
N10  G1  Z2
M5
M8
M09
M30

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Cutting depth</td>
</tr>
<tr>
<td>2</td>
<td>Retract distance</td>
</tr>
<tr>
<td>3</td>
<td>Finish tolerance in X</td>
</tr>
<tr>
<td>4</td>
<td>Finish tolerance in Z</td>
</tr>
</tbody>
</table>

Figure 4–22. G72 Stock Removal in Facing
G73 - Pattern Repeating

G73 cuts identical profiles in each pass, shifting by the current relief amount as each pass gets closer to the final profile.

Modal—No

F, S, and T functions specified in Profile blocks using G71 - Stock Removal in Turning, G72 - Stock Removal in Facing, and G73 - Pattern Repeating are ignored until a G70 is active.

G73 allows Linear and Circular interpolation.

Format

G73 U ____ W ____ R ____

G73 P ____ Q ____ U ____ W ____ F ____ S ____ T ____

Parameters

- **U**—depth of cut in X axis
- **W**—first line with W is depth of cut in Z axis
- **R**—number of repetitions for roughing passes
- **P**—first sequence number of Profile
- **Q**—last sequence number of Profile
- **U**—second line with U is finish allowance in X/2
- **W**—second line with W is finish allowance in Z
- **F**—cutting feedrate for roughing
- **S**—spindle speed
- **T**—tool with offset xxxx
Example

The following figure illustrates G73:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X Cut Depth</td>
</tr>
<tr>
<td>2</td>
<td>Z Cut Depth</td>
</tr>
<tr>
<td>3</td>
<td>X Cut Depth + X Finish Allowance/2</td>
</tr>
<tr>
<td>4</td>
<td>Z Cut Depth + Z Finish Allowance</td>
</tr>
<tr>
<td>5</td>
<td>X Finish Allowance/2</td>
</tr>
<tr>
<td>6</td>
<td>Z Finish Allowance</td>
</tr>
</tbody>
</table>

*Figure 4–23. G73 Pattern Repeating*
G74 - End Face Peck Drilling

G74 performs grooving in the Z Direction.

Modal—No

Format

G74 R ____

G74 X ____ Z ____ P ____ Q ____ R ____ F ____ S ____ T ____

Parameters

- **R**—retract distance between pecks
- **X**—total X distance X/2
- **Z**—total Z depth
- **P**—peck cutting depth in X
- **Q**—peck cutting depth in Z
- **R**—second line with R is relief distance for retract move
- **F**—cutting feedrate for roughing
- **S**—spindle speed
- **T**—tool with offset xxxx

 esposa

**Groove Width** is the starting Z value minus ending Z value plus tool width.

- The X and P parameters are not required when making single grooves.
- If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

Example

The following sample program and figure illustrate G74:

```
% G1000
G21 G54
T1313
M8
G97 S1700 M3
G0 X90.0 Z3.0
X60.0
G74 R1.0
G74 Z-6. X40.0 P3 Q4 F200
M9
```
M05
M30
%

Figure 4–24. G74 End Face Peck Drilling

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X Total X distance x/2</td>
</tr>
<tr>
<td>2</td>
<td>Z Total Z depth</td>
</tr>
<tr>
<td>3</td>
<td>R Retract distance</td>
</tr>
<tr>
<td>4</td>
<td>R Relief distance for retract move</td>
</tr>
</tbody>
</table>
G75 - Outer Diameter/Inner Diameter Drilling

G75 performs grooving in the X Direction.

Modal—No

Format

G75 R ____

G75 X ____ Z ____ P ____ Q ____ R ____ F ____ S ____ T ____

Parameters

- **R**—retract distance between pecks
- **P**—peck cutting depth in X
- **Q**—peck cutting depth in Z
- **X**—total distance x/2
- **Z**—total Z depth
- **R**—second line with R is relief distance for retract move
- **F**—cutting feedrate for roughing
- **S**—spindle speed
- **T**—tool with offset xxxx

**Groove Width** is the Z Start value minus Z End value plus tool width.

The Z and Q parameters are not required when making single grooves.

If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.

Example

The following sample program and figure illustrate G75:

```
% O1000
G21 G54 G90
T0202
M8
G97 S1700 M3
G0 X105 Z3.0
Z-20.0
G75 R1.0
G75 X97.75 Z-35.0 P2 Q3 F200
```
G28 U0
M9
M05
M30
%

Figure 4–25. G75 Outer Diameter/Inner Diameter Drilling

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X Total X Distance X/2</td>
</tr>
<tr>
<td>2</td>
<td>Z Total Z Depth</td>
</tr>
<tr>
<td>3</td>
<td>R Retract Distance</td>
</tr>
<tr>
<td>4</td>
<td>R Relief Distance for Retract Move</td>
</tr>
</tbody>
</table>
G76 - Multiple Threading Cycle

G76 performs single edge ID and OD thread cutting. The cutting amount per depth is held constant.

Default—No

Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

This cycle can be interrupted. If either the Feed Hold or Interrupt Cycle button is pressed, the tool will retract to Z start after completing the lead-out move for the current depth of cut.

Format

G76 Pxxxxxx Q ____ R ____

G76 X(u) ____ Z(w) ____ R0 P ____ Q ____ F ____

- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Please refer to Sign of U and R, on page 4 - 64 for information about determining the angle of the taper.

Parameters

- **P**—first line with P, P contains 6 digits. Moving from left to right, the digits represent the following values:
  - The first two digits represent the number of repetitive passes for the finish pass. The value ranges from 1 to 99.
  - The second two digits represent the chamfering distance along the Z axis for the lead-out move. This value ranges from 0.1 to 9.9 in increments of 0.1 expressed as a value from 00 to 99 and is a multiple of the thread lead specified by F.
  - The last two digits represent the angle of the tool tip.

  For example 2 finish passes, a chamfer of 1.5F using a tool tip angle of 55 would be expressed as: P021555

- **Q**—minimum depth of cut.
• **R**—finish allowance. This is the amount of material to leave for the finish pass.
• **X(u)**—minimum diameter for the thread depth.
• **Z(w)**—Z end position of the cycle.
• **R**—second line with R is start thread radius minus end thread radius.
• **P**—second line with P is thread height.
• **Q**—second line with Q is depth of cut for the first pass.
• **F**—thread lead.

⇒ The retract move is approximately 45° or less depending upon servo lag.

**Example**

The following figure illustrates G76:

<table>
<thead>
<tr>
<th></th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Chamfer distance along Z axis for lead-out move</td>
</tr>
<tr>
<td>2</td>
<td>Z end of thread</td>
</tr>
<tr>
<td>3</td>
<td>R start thread radius minus end thread radius</td>
</tr>
<tr>
<td>4</td>
<td>P thread height</td>
</tr>
<tr>
<td>5</td>
<td>Q depth of cut for first pass</td>
</tr>
</tbody>
</table>

*Figure 4–26. G76 - Multiple Threading Cycle*
G77 - Outer Diameter/Inner Diameter Cutting Cycle

G77 performs external or internal diameter cutting.

Default—No

Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G76 - Multiple Threading Cycle
- G78 - Threading Cycle
- G79 - Stock Removal in Facing Cycle

Sign of U and R

The sign of U and R determines the angle of the taper for canned cycles. In general, the value of R is added to the start point of the cycle.

Figure 4–27. Sign of U and R determines the angle of the taper for canned cycles
Format

\[ \text{G77 X(u) ___ Z(w) ___ R ___ F ___} \]

- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Parameters

- \( \text{X(u)} \) — X end position
- \( \text{Z(w)} \) — Z end position
- \( \text{R} \) — start radius minus end radius
- \( \text{F} \) — cutting feedrate

Example

The following figure illustrates G77:

\[ \text{Figure 4–28. G77 External Diameter/Internal Diameter Cutting Cycle} \]
G78 - Threading Cycle

G78 has a box shape. The cycle starts at the previous XZ point and goes to the XZ point specified in the current block. This block is modal and allows cutting threads with multiple passes. Please also refer to G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle for information about using G92 with Threading Cycles.

Default—No
Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G79 - Stock Removal in Facing Cycle

Format

G78 X ____ Z ____ R ____ F ____

Please refer to Sign of U and R, on page 4 - 64 for information about determining the angle of the taper.

Parameters

- **R**—taper distance.
- **R** is positive if the X Start position is larger than the X End position.
- **R** is negative if the X Start position is smaller than the X End position.
- **F**—pitch
G79 - Stock Removal in Facing Cycle

G79 removes stock in a facing move. Each cutting pass is programmed for this cycle.

Default—No

Modal—Yes

Cancels:

- G32 - Constant Lead Thread Cutting
- G33 - Threading
- G76 - Multiple Threading Cycle
- G77 - Outer Diameter/Inner Diameter Cutting Cycle
- G78 - Threading Cycle

Format

G79 X(u) ____ Z(w) ____ R ____

- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Please refer to Sign of U and R, on page 4 - 64 for information about determining the angle of the taper.

Parameters

- R—amount of taper, if any.

Example

G79 is modal so stock removal can be done as follows:

G79 X ____ Z ____ R ____
U ____
U ____
W ____
W ____
G80 - Canned Cycle Cancel (default)

G80 cancels any active drilling cycle. When G80 is active, X and Z motion commands move the tool directly to the programmed endpoint without performing a drilling cycle. In addition to turning off the drilling cycle, G80 activates Linear Motion at Rapid and clears the modal drilling parameters such as R, Z, Q, and P.

If the drill cycle depth is not programmed after G80, then the drill cycle depth defaults to the R plane.

Default—Yes
Modal—Yes

Cancels:

- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

G00 - Rapid Traverse (default) cannot be programmed in the same block.
G81 - Drill Cycle

G81 sets drilling as the modal drill cycle.

Default—No
Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G81 X ____ Z ____ F ____ S ____

Parameters

- F—Feedrate
- S—Spindle speed

The tool motion for the drill cycle is shown below.

1. Position to XZ
2. Rapid to reference plane
3. Feed to depth
4. Rapid to reference plane
Example—G81 Drill Cycle
G90 (Absolute)
G81 X2 Z-1.2 R.1 F10

Figure 4–29. G81 Drill Cycle

G82 - Drill Cycle with Dwell

G82 sets drilling with dwell as the modal drill cycle.

Default—No
Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G82 X ____ Z ____ P ____ F ____ S ____
Parameters

- **P**—Seconds to dwell at bottom of hole
- **F**—Feedrate
- **S**—Spindle speed

When P is an integer (no decimal point), the value will be divided by 10000. For example, P10000 becomes P1.0.

The tool motion for this cycle is the same as the drill cycle except a dwell for P seconds occurs at the bottom of the hole.

1. Position to XZ.
2. Rapid to reference plane.
3. Feed to depth.
4. Dwell for P seconds or revolutions (depending on FPM or FPR mode).
5. Rapid to reference plane.

**Example—G82 Drill Cycle with Dwell**

```
G90 (Absolute)
G82 X2 Z-1.2 R.1 F10 P2.2 G99
```

*Figure 4–30. G82 Drill Cycle with Dwell*
G83 - Face Drilling with Pecks

G83 sets face drilling with pecks as the modal drill cycle.

Default—No
Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83.1 - Chip Breaker Peck Drilling
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G83 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **Q**—Distance for feed
- **F**—Feedrate
- **S**—Spindle speed

⇒ If **P** or **Q** are integers (no decimal point), the values will be divided by 10000. For example, **P10000** becomes **P1.0**.
The tool motion for the chip break drill cycle is illustrated in Figure 4–31. G83 Peck Drill Cycle, on page 4 - 73This cycle requires programming X or Z, R (Retract Clearance Plane), and Q. This cycle causes the tool to feed by the amount programmed in Q, then rapid back to the R plane. Next the tool rapids to a distance slightly above (2.5mm) the last infeed depth, then feeds another Q distance into the part. This sequence repeats until the tool reaches the programmed Z depth.

The following figure illustrates tool motion for G83 peck drill cycle. The short broken lines represent rapid moves; the long broken lines represent feed moves.

![Figure 4–31. G83 Peck Drill Cycle](image)

**Example—G83 Peck Drill Cycle**

```
G90 (Absolute)
G83 X2.0 Z-1.2 R0.1 Q0.5
```
G83.1 - Chip Breaker Peck Drilling

G83.1 sets chip breaker peck drilling as the modal drill cycle.

Default—No
Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G83.1 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **Q**—Distance for feed
- **F**—Feedrate
- **S**—Spindle speed

⇒ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.
G84 - Face CW Tapping

G84 sets Face CW tapping as the modal drill cycle.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G83.1 - Chip Breaker Peck Drilling
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G84 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- P—Seconds to dwell at bottom of hole
- Q—Distance for feed
- F—Feedrate
- S—Spindle speed

If Q is an integer (no decimal point), the value will be divided by 10000. For example, Q10000 becomes Q1.0.
The tool motion for this cycle is described below. The feedrate and spindle override controls are forced to 100% are disabled while this cycle is active.

To stop tool motion when the tap is in the bore, either press the Emergency Stop button or use Retract.

The control will not respond to a motion-stop command until the tool returns to the reference plane.

This drawing shows right-hand tapping tool motion:

1. Tool rapids to R plane.
2. Tool feeds CW to the programmed depth.
4. If G94 is active, the axis will dwell for P seconds; if G95 is active, the axis will dwell for P revolutions.
5. Tool feeds to R plane.

![Figure 4–32. G84 Face CW Tapping](image-url)
G84.2 (or G84 M29) - Rigid CW Tapping

Rigid tapping allows the same hole to be tapped repeatedly with precision. The rigid tapping feature increases accuracy by synchronizing the rotation of the spindle with the feed of the Z axis. G84.2 is used for right-handed tapping. M29 is required for Rigid Tapping.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G84.2 (or G84 M29) X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- P—Seconds to dwell at bottom of hole
- Q—Distance for feed
- F—Feedrate
- S—Spindle speed

If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.
G85 - Boring Cycle

The Boring cycle provides a feed-in and feed-out sequence suitable for boring.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G85 X ____ Z ____ P ____ F ____ S ____

Parameters

- P—Seconds to dwell at bottom of hole
- F—Feedrate
- S—Spindle speed

The face boring cycle moves the axes in this manner:

1. The spindle should already be switched on using an M3 code.
2. The tool is positioned over the hole location.
3. At the G85, the spindle feeds to Z Bottom as specified.
4. At Z Bottom, the spindle feeds to the Z Start position.

⇒ It is possible to have an XY position move with the G85 code.
Example

The diagram below illustrates tool movement for the Boring cycle (G85):

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Stock</td>
</tr>
<tr>
<td>2</td>
<td>Z Start; Return Point</td>
</tr>
<tr>
<td>3</td>
<td>Z Bottom</td>
</tr>
</tbody>
</table>

*Figure 4–33. G85 Tool Movement for the Boring Cycle*
G86 - Bore Rapid Out Cycle

The Bore Rapid Out canned cycle is a feed-in, rapid-out sequence. The spindle stops at the bottom of the hole and is retracted at the rapid traverse rate.

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G87 - Side Drilling with Pecks/Dwell,
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G86 X ____ Z ____ P ____ F ____ S ____

Parameters

- P—Seconds to dwell at bottom of hole
- F—Feedrate
- S—Spindle speed

The Bore Rapid Out canned cycle moves the axes in this manner with the spindle switched on:

1. The tool is positioned at the Initial Z location and moves at the rapid speed to XY if it is in the block.
2. The spindle bores down to Z Bottom at the specified feedrate.
3. The spindle turns off.
4. The spindle moves up to Z Start at the rapid speed.
5. The spindle turns on.
Example

This diagram illustrates tool movement for the Bore Rapid Out cycle:

![Diagram showing tool movement for the Bore Rapid Out cycle]

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Stock</td>
</tr>
<tr>
<td>2</td>
<td>Z Start (Basic); Return Point (Industry standard)</td>
</tr>
</tbody>
</table>
| 3 | Z Bottom  
    | Spindle Stop |

*Figure 4–34. G86 Tool Movement for the Bore Rapid Out Cycle*
G87 - Side Drilling with Pecks/Dwell

Default—No

Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G88 - Side CW Tapping
- G89 - Side Boring Cycle

Format

G87 X ____ Z ____ P ____ Q ____ F ____ S ____

Parameters

- P—Seconds to dwell at bottom of hole
- Q—Distance for feed
- F—Feedrate
- S—Spindle speed

⇒ If P or Q are integers (no decimal point), the values will be divided by 10000. For example, P10000 becomes P1.0.
G88 - Side CW Tapping

Default—No
Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G89 - Side Boring Cycle

Format

G88 X ____ Z ____ P ____ F ____ S ____

Parameters

- **P**—Seconds to dwell at bottom of hole
- **F**—Feedrate
- **S**—Spindle speed

⇒ If **P** is an integer (no decimal point), the value will be divided by 10000. For example, P10000 becomes P1.0.
G89 - Side Boring Cycle

Default—No
Modal—Yes

Cancels:

- G80 - Canned Cycle Cancel (default)
- G81 - Drill Cycle
- G82 - Drill Cycle with Dwell
- G83 - Face Drilling with Pecks
- G84 - Face CW Tapping
- G84.2 (or G84 M29) - Rigid CW Tapping
- G85 - Boring Cycle
- G86 - Bore Rapid Out Cycle
- G87 - Side Drilling with Pecks/Dwell
- G88 - Side CW Tapping

Format

G89 X ____ Z ____ F ____ S ____

Parameters

- F—Feedrate
- S—Spindle speed
G90 - Absolute Programming (default)

G90 applies to Macro Mode B and establishes a modal setting that tells the control to interpret all XZ endpoint dimensions in absolute coordinates. G90 Absolute Programming specifies that all tool endpoint positions are measured from the current part zero position.

Default—Yes
Modal—Yes

Cancels G91 - Incremental Programming (Mode B)

You can use G90 absolute or G91 - Incremental Programming dimensions directly from the blueprint. Most part programs should begin with a G90 Absolute command that moves the tool to a fixed position on the machine. Once the start point is defined, you can study the part blueprint and determine whether to use G90 Absolute or G91 Incremental dimensions.

Most part programs should begin with a G90 command to establish part zero. This setting remains active until you program G91 to select incremental dimension programming. A program may switch between G90 Absolute mode and G91 Incremental mode at any time.

Absolute dimensions are measured from part zero. The figure shows the path that the tool center will follow.

*Figure 4–35. G90 Absolute Dimensions (Mode B)*
Example

The following sample NC program is in Absolute mode (Mode B):

N10 (msg, lathe doc sample, abs G42, tool radius 0.1)
N20 G90 G40 G94 T0000 (absolute mode, cutter compensation off, offset 0)
N30 G07 G00 X2.5 T0101 (radius programming, rapid to X, tool 1 offset 1)
N40 Z3 (rapid to Z)
N60 X.5 (rapid to X)
N70 G96 S350 M03 (CSS, X axis at 5 in, 350 SFM and clockwise)
N80 G01 G95 F.01 X0 (face, IPR 0.01)
N120 G00 Z4 (rapid to Z)
N125 G42 (cutter compensation right, skip if block del on)
N130 X.5 Z3 (rapid to XZ)
N160 G01 X1 Z2 (taper)
N170 X1.7 (face)
N180 Z0.3 (turn)
N190 X1.9 (face)
N200 Z0 (turn)
N230 G00 G40 X2.5 (cutter compensation off, rapid to X, skip if block del on)
N240 Z3 (rapid to Z)
N900 M30 (end of program, rewind)
G91 - Incremental Programming

G91 applies to Macro Mode B and establishes a modal setting that tells the control to interpret all XZ endpoint dimensions in incremental coordinates. G91 Incremental Programming specifies that all tool endpoint positions are measured from the tool's position at the start of the motion. This setting remains active until you program G90 - Absolute Programming (default) to select absolute dimension programming. A program may switch between G90 Absolute and G91 Incremental at any time within a part program.

Default—No

Modal—Yes

Cancels G90 - Absolute Programming (default) (Mode B)

Incremental dimensions specify the distance that the tool must move during each block (i.e. the distance from the start of the move to the end of the move).

![Figure 4–36. G91 Incremental Dimensions (Mode B)](image)
Example

The following sample NC program changes from G90 to G91 (Mode B):

N10 \texttt{(msg, lathe doc sample inc G42 tool rad 0.1)}
N20 \texttt{G90 G40 G94 T0000 (absolute mode, cutter compensation off, offset 0)}
N30 \texttt{G07 G00 X2.5 T0101 (radius programming, rapid to X, tool 1 offset 1)}
N40 \texttt{Z3 (rapid to Z)}
N60 \texttt{G91 X-2 (incremental mode, rapid to X)}
N70 \texttt{G96 S350 M03 (CSS, X axis at 5 in, spindle speed 350 and clockwise)}
N80 \texttt{G01 G95 F.01 X-.5 (face, RPM 0.01)}
N120 \texttt{G00 Z1 (rapid to Z)}
N125 \texttt{G42 (cutter compensation right, skip if block del on)}
N130 \texttt{X.5 Z-1 (rapid to XZ)}
N160 \texttt{G01 X.5 Z-1 (taper)}
N170 \texttt{X.7 (face)}
N180 \texttt{Z-1.7 (turn)}
N190 \texttt{X.2 (face)}
N200 \texttt{Z-.3 (turn)}

/ 
N230 \texttt{G00 G40 X.6 (cutter compensation off, rapid to X, skip if block del on)}
N240 \texttt{Z3 (rapid to Z)}
N900 \texttt{M30 (end of program, rewind)}
G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle

ISNC Mode B

Default—No

Modal—Yes

Cancels:

- G53 - Machine Coordinates
- G54-G59 - Work Coordinates (G54 default)
- G54.1 - G54.93 - Aux Work Coordinates

G92 used in conjunction with specific parameters provides the following programming functions:

Work Coordinate Offsets

G92 lets you establish the part program coordinates at the current position without generating any tool motion.

G92 Work Coordinates Offsets cannot include X and Z coordinates.

Work Coordinate Offsets Format

G92

Spindle Max Speed

G92 can also set a maximum spindle speed for direct spindle speed or constant surface speed mode when an S parameter is used.

G92 Spindle Max Speed cannot include X and Z coordinates.

Spindle Max Speed Format

G92 S____

Spindle Max Speed Parameters

- S—spindle speed

Spindle Max Speed Example

G92 S5000 (prevent the spindle from exceeding 5000 rpm)
Multiple Thread Cutting Cycle

G92 can perform a multiple thread cutting cycle when an F parameter is used.

Please refer to *Sign of U and R*, on page 4 - 64 for information about determining the angle of the taper.

Multiple Thread Cutting Cycle Format

\[
\text{G92 } X(u) \quad Z(w) \quad R \text{ or } I \quad F \quad
\]

- X indicates Absolute X and (u) indicates Incremental X.
- Z indicates Absolute Z and (w) indicates Incremental Z.

Multiple Thread Cutting Cycle Parameters

- **X(u)**—minimum thread depth diameter
- **Z(w)**—Z end cycle position
- **R** or **I**—thread radius start minus thread radius end
- **F**—cutting feedrate

Multiple Thread Cutting Cycle Example

The following figure illustrates G92 Multiple Thread Cutting Cycle:

*Figure 4–37. G92 Multiple Thread Cutting—Tapered Thread*

<table>
<thead>
<tr>
<th></th>
<th>Lead is based on commanded spindle RPM and F: cutting feedrate</th>
</tr>
</thead>
<tbody>
<tr>
<td>R or I</td>
<td>Thread radius start minus thread radius end</td>
</tr>
</tbody>
</table>

1. Lead is based on commanded spindle RPM and F: cutting feedrate

R or I | Thread radius start minus thread radius end
G93 - Inverse Time Feedrate

G93 can be specified to change the feedrate as a function of time and distance. If the time is unchanged but the distance changes, then the actual feedrate will change proportionally. The time is computed by dividing one by the Inverse Time programmed. The actual feedrate is the distance divided by the time.

Default—No

Modal—Yes

Cancels:

- G94 - Feed per Minute (default)
- G95 - Feed per Revolution

⚠️ The feedrate must be specified for every move.

⇒ If the inverse time value falls outside the contouring limits of the machine, then the control will instead use the minimum or maximum value.

Format

G93 X____ Z____ F____

The format for Inverse Time is F6.3 (maximum of six digits before the decimal point and maximum of three digits after the decimal point) and the units are minutes. Feedrates of up to 999999.999 can be programmed using G93.

Parameters

- **F**—feedrate as a function of time and distance

Example

Time is 1/10.0min = 0.1min

Actual Feedrate for first line is 5.0in/0.1 min = 50 ipm.

Actual Feedrate for second line is 7.0in/0.1 min = 70 ipm.

G93 G1 X5.0 F10.0
Y7.0 F10.0
G94 - Feed per Minute (default)

G94 puts the control into feed per minute mode. When G94 is active, all F feedrate values are interpreted in inch/minute (G20 - Inch Mode (default)) or millimeter/minute (G21 - Millimeter Mode) units. Any dwell command that is programmed while G94 is active will be interpreted as number of seconds to dwell.

Default—Yes
Modal—Yes

Cancels:

• G93 - Inverse Time Feedrate
• G95 - Feed per Revolution

Format

G94 X____ Z____ F____

Parameters

• F—feedrate in inch/minute or millimeter/minute

G95 - Feed per Revolution

G95 puts the control into feed per revolution mode. When G95 is active, all F feedrate values are interpreted in inch/revolution (G20 - Inch Mode (default)) or millimeter/revolution (G21 - Millimeter Mode) units. Any dwell command that is programmed while G95 is active will be interpreted as number of spindle revolutions to dwell.

Default—No
Modal—Yes

Cancels:

• G93 - Inverse Time Feedrate
• G94 - Feed per Minute (default)

Format

G95 X____ Z____ F____

Parameters

• F—feedrate in inch/revolution or millimeter/evolution
G96 - Constant Surface Speed (CSS)

G96 sets feet per minute or meters per minute as the modal spindle command mode. When G96 is programmed, the spindle speed will increase as the tool tip moves toward the spindle centerline and decrease as the tool moves away from the spindle centerline. When G96 is active, the programmed S value is read as a feet per minute or meters per minute command.

Default—No

Modal—Yes

Cancels G97 - Direct Spindle Speed (default)

To specify a radius of the tool tip that corresponds to the surface speed given in S, program the R value in the same block as G96.

- If you do not program an R value or the R value is 0, the control uses the X-axis position as the radius.
- If an R value has been previously programmed, and you program a G96 without an R value, the previously programmed R value will be used.

<table>
<thead>
<tr>
<th>Active Programming Mode</th>
<th>R Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>G07 - Radius Programming</td>
<td>distance from the tool tip to the spindle centerline (radius)</td>
</tr>
<tr>
<td>G08 - Diameter Programming (default)</td>
<td>twice the distance from the tool tip to the spindle centerline (diameter)</td>
</tr>
</tbody>
</table>

*Table 4–10. R Values and G96 Programming*

G97 - Direct Spindle Speed (default)

G97 sets direct RPM as the modal spindle command mode. When G97 is active, the programmed S value is revolutions per minute.

Default—Yes

Modal—Yes

Cancels G96 - Constant Surface Speed (CSS)
G98 - Return to Initial Level (default)

When G98 is active, all G81-G89 drill cycles command the tool to return to the Z axis coordinate where the tool was when the cycle was initiated. If the initial Z coordinate is changed for subsequent cycles, the Z coordinate for the first cycle after G80 - Canned Cycle Cancel (default) is used as the reference plane.

Default—Yes

Modal—Yes

Cancels G99 - Return to R Point Level

Examples

In the following example, the tool will rapid to Z10 in the second block even though Z20 (init plane) is specified:

```
N005  G98
N010  G00  Z10
N020  G81  Z5  F60  (feed to Z5, rapid to Z10)
N030  G00  Z20
N040  G81  Z15  (rapid to Z10, feed up to Z15, rapid to Z10)
N050  G80
```

In the following example, to return to the init plane, G98 requires a G80 Cancel Drill Cycle to change to the init plane.

```
N005  G98
N010  G0  Z10
N020  G81  Z5  F60  (feed to Z5, rapid to Z10)
N030  G80  Z20
N040  G81  Z15  (rapid to Z10, feed up to Z15, rapid to Z20)
N050  G80
```

Figure 4–38. G98 Drill Cycle Initial Level Return (default)
G99 - Return to R Point Level

When G99 is active, all G81-G89 drill cycles command the tool to return to the R Reference plane at the end of the drill cycle.

Default—No

Modal—Yes

Cancels G98 - Return to Initial Level (default)

Example

The tool returns to the R Reference Plane.

N005  G99
N020  G81  Z5  (feed to Z5, rapid to R)

Figure 4–39. G99 General Drill Cycle Tool Motion
G101 - Axial Surface of the Part - Main Spindle without Linear Y

G101 sets up the machine for Axial Milling without Linear Y Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G102, G103, G104, G105, G106, G107, G108, G109

Example
G101  (AXIS MAP)
M131  (MILLING MODE)
G00  Z.25

This part-relative, tool motion map is used when the Y Axis travel is inadequate for the geometry of the feature. The Y-axis positions are accomplished by coordinating the C axis and X axis motion. On TMM machines this map is always used for Axial Milling.

⚠️ This map is relative to the TMX MY and TMX MYS Main Spindle Only. The Axial Milling map for the TMX MYS Sub-spindle uses G105.

Figure 4–40. G101 TMX Axis Coordinates
G102 - Radial Surface of the Part – Main Spindle without Linear Y

G102 sets up the machine for Radial Milling without Linear Y Motion and activates the coordinate map shown below.

Default—No

Modal—Yes


Example

G102 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIAMETER OF PART)
G00 X.25

G25 Q must be used to establish the Z part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for wrapping the Y Axis coordinate system around the circumference of the part. The bottom surface of the feature will be parallel to the circumference of the part. On the TMM machine this map is always used for Radial Milling.

This map is relative to the TMX MY and TMX MYS Main Spindle only. The Radial Milling map for the TMX MYS Sub-spindle uses G106.

This map swaps the Z and X axes from G101.

The Y-axis travel limits are limited to +/- 800 inches (20320 mm) when using this map.

![Figure 4–41. G102 TMX Axis Coordinates](image-url)
In the following figure, the Cylinder Diameter is 4 inches (102 mm), and the Cylinder Circumference is 12.544 inches (319 mm).

*Figure 4–42. Y Axis Coordinate System Detail for G102*
G103 - Axial Surface of the Part – Main Spindle with Linear Y

G103 sets up the machine for Axial Milling with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G104, G105, G106, G107, G108, G109

Example
G103 (AXIS MAP)
M131 (MILLING MODE)
Z.25
Y.1

When using tool holders with multiple turn stations that require a Y move on a tool change, the G103 axis map must be active.

Tool Change Example
G103 (AXIS MAP)
T0101
Y0.
G109

The linear Y-axis motion for this part-relative, tool motion map is created by using the combined motion of the X and X’ physical axes. The coordinate system for this axis map is the same as the G101 coordinate system.

This map is relative to the TMX MY and TMX MYS Main Spindle only. The Axial Milling map for the TMX MYS Sub-spindle uses G107.
G104 - Radial Surface of the Part – Main Spindle with Linear Y

G104 sets up the machine for Radial Milling with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No
Modal—Yes
Cancels—G101, G102, G103, G105, G106, G107, G108, G109

Example
G104 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIA. OF PART)

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for Radial Milling with true linear Y-Axis motion. The bottom surface of the feature will be perpendicular to the X physical axis. On TMM machines this map is always used for Radial Milling.

This map is relative to the TMX MY and TMX MYS Main Spindle only. The Radial Milling map for the TMX MYS Sub-spindle uses G109.

This map swaps the Z and X axes from G101.

The Y+ and Y- axis has reversed direction from the G101 and G103 axis maps.

Figure 4–44. G104 TMX Axis Coordinates
G105 - Axial Surface of the Part – Sub-spindle without Linear Y

G105 sets up the machine for Axial Milling on the Sub-spindle and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G106, G107, G108, G109

Example
G105 (AXIS MAP)
M131 (MILLING MODE)

This part-relative, tool motion map is used when the Y-axis travel is inadequate for the geometry of the feature. The Y-axis positions are accomplished by coordinating the C and X axis motion.

⚠️ This map is relative to the TMX MYS Sub-Spindle only. The Axial Milling map for the TMX MYS Main spindle uses G101.

Figure 4–45. G105 TMX Axis Coordinates
G106 - Radial Surface of the Part – Sub-spindle without Linear Y

G106 sets up the machine for Radial Milling on the sub-spindle and activates the coordinate map shown below.

Default—No
Modal—Yes
Cancels—G101, G102, G103, G104, G105, G107, G108, G109

Example

G106 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIA. OF PART)

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used to wrap the Y-axis coordinate system around the circumference of the part and the part is in the sub-spindle. The bottom surface of the feature will be parallel to the circumference of the part.

⚠️ This map is relative to the TMX MYS Sub-Spindle only. The Radial Milling map for the TMX MYS Main spindle uses G102.

This map swaps the Z and X axes from G101.

The Y-axis travel limits are limited to +/- 800 inches (20320 mm) when using this map.

![Figure 4–46. G106 TMX Axis Coordinates](image-url)
In the following figure, the Cylinder Diameter is 4 inches (102 mm), and the Cylinder Circumference is 12.544 inches (319 mm).

*Figure 4–47. Y Axis Coordinate System Detail for G106*
G107 - Axial Surface of the Part – Sub-spindle with Linear Y

G107 sets up the machine for axial milling on the sub-spindle with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No
Modal—Yes

Example
G107 (AXIS MAP)
M131 (MILLING MODE)

The linear Y-axis motion for this part-relative, tool motion map is created by using the combined motion of the X and X’ physical axes.

This map is relative to the TMX MYS Sub-Spindle only. The Axial Milling map for the TMX MYS Main spindle uses G103.

The G107 coordinate system is the same as the G105 axis map.

Figure 4–48. G107 TMX Axis Coordinates
G108 - Radial Surface of the Part – Sub-spindle with Linear Y

G108 sets up the machine for Radial Milling on the sub-spindle with true linear Y-Axis Motion and activates the coordinate map shown below.

Default—No

Modal—Yes

Cancels—G101, G102, G103, G104, G105, G106, G107, G109

Example

G108 (AXIS MAP)
M131 (MILLING MODE)
G25 Q4. (DIA. OF PART)

G25 Q must be used to establish part-relative location. The surface of the part is Z.0, and a positive Z value is above the part in the air; a negative Z value moves into the stock.

This part-relative, tool motion map is used for Radial Milling on the sub-spindle with true linear Y-Axis motion. The bottom surface of the feature will be perpendicular to the X physical axis.

⚠️ This map is relative to the TMX MYS Sub-Spindle only. The Radial Milling map for the TMX MYS Main spindle uses G104.

⚠️ This map swaps the Z and X axes from G101.

⚠️ This map reverses the direction of the Y-axis from the direction G101.

Figure 4–49. G108 TMX Axis Coordinates
G109 - Turning / Lathe Mode

G109 sets up the machine in Lathe Mode and activates the coordinate map shown below.

Default—Yes
Modal—Yes
Cancels—G101, G102, G103, G104, G105, G106, G107, G108

Example (Turning Operations)
M130
G109

Example (Drilling operations with the C axis mapped instead of Y)
M131
G109

When G109 is used with M131 (Live Tool Mode) it allows for performing live tooling operations using polar coordinates. The G109 must be preceded by either the M130 (Lathe Mode) or M131 (Live-tool Mode – Main Spindle/C Axis).

This axis map applies to all machine machines.

Polar Coordinate programming is not available for the sub spindle.

Figure 4–50. G109 Turning/Lathe Mode (Main Spindle Axis Map)
Figure 4–51. G109 Turning/Live-tool Mode (Main Spindle Axis Map)

Figure 4–52. G109 Turning/Lathe Mode (Sub-Spindle Axis Map)

Figure 4–53. G109 Turning/Live-tool Mode (Sub-spindle Axis Map)
ISNC M Codes

This section contains information about the following topics:

- ISNC M Code Overview .......................................................... 5 - 2
- M Code Translation ............................................................... 5 - 2
- ISNC M Code Definitions ....................................................... 5 - 5
- ISNC M Code Translation List - Page 1 ................................. 5 - 9
- ISNC M Code Translation List - Page 2 ................................. 5 - 13
- ISNC M Code Translation List - Page 3 ................................. 5 - 15
- ISNC M Code Translation List - Page 4 ................................. 5 - 19
ISNC M Code Overview

M codes belong to one of the following four execution categories:

- Execute before tool change or motion.
- Execute after T and S codes, but before motion.
- Execute after tool change or motion.
- Execute after tool change or motion (system M codes).

Understanding each M code's execution category is important when the M code is programmed in a block that also contains a T code, S code, E code, and XZ axes motion.

M Code Translation

The ISNC option provides M Code translation, allowing you to customize M Codes from existing programs and controls for use on the WinMax Lathe Max Control. Many M Codes are named by the machine builder and vary from one machine to another. These M Codes can be converted to a default set of codes, or the codes can be customized and translated to the M Code that the WinMax Lathe Max control uses.

In addition, if an existing program contains M Codes for machine functions that are not supported, you can specify which M Codes to ignore.

Setting up M Codes

From the Utilities, User Preferences, NC Settings screen with an ISNC program open, select the ISNC M CODES F5 softkey. This softkey appears when the cursor is in the NC Dialect field.
The ISNC M Code Translation List - Page 1 screen opens.

![Figure 5–2. ISNC M Code Translation List Page 1 screen](image)

This screen lists M Code names and a column listing the corresponding default ISNC code. Please refer to *ISNC M Code Definitions, on page 5 - 5* for details about each M Code.

You can change the default ISNC codes, unless the field is inactive. Inactive fields contain commonly used code numbers. A change is saved when you select the Save ISNC M CODES F7 softkey. Select the RESET M CODES TO DEFAULT VALUES F5 softkey to return all ISNC codes to the default settings.

There are 3 of these Translation screens, accessed from Page 1 by selecting the PREVIOUS M CODE SCREEN F1 or NEXT M CODE SCREEN F2 softkey.

![Figure 5–3. ISNC M Code Translation List Page 2 screen](image)
The last of the ISNC M Code Translation List screens, Page 4, contains a listing of codes which can be ignored because the program contains M Codes for functions that the WinMax Lathe Max Control does not support. Enter these codes on this screen and the M Codes will be ignored.
ISNC M Code Definitions

Following are definitions for Hurco M Codes. A corresponding ISNC M Code for M Code Translation or Importing and Exporting M Code Data is provided for each M Code.

The following M codes are available on most systems.

ISNC M Code Translation List - Page 1

Program Stop (ISNC M0) ........................................ 5 - 9
Optional Stop (ISNC M1) ........................................ 5 - 9
Main Spindle CW (ISNC M3) ............................... 5 - 9
Main Spindle CCW (ISNC M4) ............................... 5 - 9
Main Spindle Stop (ISNC M5) ............................... 5 - 9
Live Tool CW (ISNC M33) ................................. 5 - 10
Live Tool CCW (ISNC M34) ................................. 5 - 10
Live Tool Stop (ISNC M35) ................................. 5 - 10
Sub-spindle CW (ISNC M103) ......................... 5 - 11
Sub-spindle CCW (ISNC M104) ......................... 5 - 11
Sub-spindle Stop (ISNC M105) ......................... 5 - 11
Secondary Coolant (ISNC M7) ......................... 5 - 11
Primary Coolant On (ISNC M8) ......................... 5 - 11
Coolant Off (ISNC M9) ......................................... 5 - 11
Chuck Pressure High (ISNC M67) ..................... 5 - 11
Chuck Pressure Low (ISNC M66) ....................... 5 - 12
Turret Index Reverse (ISNC M12) .................... 5 - 12
Turret Index Forward (ISNC M13) .................... 5 - 12
Chuck Open (ISNC M69) ..................................... 5 - 12
Chuck Close (ISNC M68) ..................................... 5 - 12
Main Spindle Orient (ISNC M19) ..................... 5 - 12
Chuck Open for Bar (ISNC M20) ..................... 5 - 12
Chuck Close for Bar (ISNC M21) ..................... 5 - 12
Start Bar Feeder (ISNC M22) ......................... 5 - 12
<table>
<thead>
<tr>
<th>ISNC M Code Translation List - Page 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start Bar with Z (ISNC M23) ..........</td>
</tr>
<tr>
<td>Part Conveyor On (ISNC M110) .......</td>
</tr>
<tr>
<td>Part Conveyor Off (ISNC M111) .......</td>
</tr>
<tr>
<td>Tailstock Quill Adv (ISNC M78) ......</td>
</tr>
<tr>
<td>Tailstock Quill Retr (ISNC M79) .....</td>
</tr>
<tr>
<td>End of Program (ISNC M30) ..........</td>
</tr>
<tr>
<td>Incr Cycle Count (ISNC M31) .......</td>
</tr>
<tr>
<td>Incr Setup Cycle (ISNC M32) .......</td>
</tr>
<tr>
<td>Feedrate Override (ISNC M48) ......</td>
</tr>
<tr>
<td>Ignore Feed Override (ISNC M49) .....</td>
</tr>
<tr>
<td>Conveyor On (ISNC M24) ............</td>
</tr>
<tr>
<td>Conveyor Off (ISNC M25) ............</td>
</tr>
<tr>
<td>Part Catcher w/Eject (ISNC M57) ...</td>
</tr>
<tr>
<td>Part Catcher Advance (ISNC M58) ....</td>
</tr>
<tr>
<td>Part Catcher Retract (ISNC M59) ...</td>
</tr>
<tr>
<td>Sel External Chuck (ISNC M60) ......</td>
</tr>
<tr>
<td>Sel Internal Chuck (ISNC M61) .....</td>
</tr>
<tr>
<td>Tool Setter Retract (ISNC M71) ....</td>
</tr>
<tr>
<td>Tool Setter Adv (ISNC M72) .......</td>
</tr>
<tr>
<td>Auto Door Open (ISNC M85) ........</td>
</tr>
<tr>
<td>Auto Door Close (ISNC M86) .......</td>
</tr>
<tr>
<td>Single Block Off (ISNC M91) .......</td>
</tr>
<tr>
<td>Single Block On (ISNC M92) .......</td>
</tr>
<tr>
<td>Sub-spindle Chuck Open (ISNC M168)</td>
</tr>
<tr>
<td>ISNC M Code Translation List - Page 3</td>
</tr>
<tr>
<td>---------------------------------------</td>
</tr>
<tr>
<td>Sub-spindle Chuck Close (ISNC M169) ........................................... 5 - 15</td>
</tr>
<tr>
<td>Turning Mode (ISNC M130) .................. 5 - 15</td>
</tr>
<tr>
<td>Milling Mode (ISNC M131) .................. 5 - 15</td>
</tr>
<tr>
<td>C Axis Clamp (ISNC M41) ................... 5 - 15</td>
</tr>
<tr>
<td>C Axis Hold (ISNC M133) .................. 5 - 15</td>
</tr>
<tr>
<td>C Axis Unclamp (ISNC M40) ............... 5 - 15</td>
</tr>
<tr>
<td>Sub-spndl Ext Chuck (ISNC M160) .......... 5 - 16</td>
</tr>
<tr>
<td>Sub-spndl Int Chuck (ISNC M161) .......... 5 - 16</td>
</tr>
<tr>
<td>W Torque Mon On (ISNC M186) ............. 5 - 16</td>
</tr>
<tr>
<td>W Torque Mon Off (ISNC M187) ............ 5 - 16</td>
</tr>
<tr>
<td>Block Skip Sync (ISNC M200) ............. 5 - 16</td>
</tr>
<tr>
<td>Sync Spindles Forward (ISNC M203) ...... 5 - 16</td>
</tr>
<tr>
<td>Sync Spindles Reverse (ISNC M204) ...... 5 - 16</td>
</tr>
<tr>
<td>Clear Spindle Sync (ISNC M205) .......... 5 - 17</td>
</tr>
<tr>
<td>Bypass Chuck Intlk (ISNC M231) .......... 5 - 17</td>
</tr>
<tr>
<td>C3 Axis Clamp (ISNC M232) ............... 5 - 17</td>
</tr>
<tr>
<td>C3 Axis Unclamp (ISNC M234) ............. 5 - 17</td>
</tr>
<tr>
<td>Spindle CW/Coolant On (ISNC M63) ...... 5 - 17</td>
</tr>
<tr>
<td>Spindle CCW/Coolant Off (ISNC M64) ..... 5 - 17</td>
</tr>
<tr>
<td>Spindle Off/Coolant Off (ISNC M65) ..... 5 - 18</td>
</tr>
<tr>
<td>Sub-Spdl CW/CLNT On (ISNC M163) ...... 5 - 18</td>
</tr>
<tr>
<td>Sub-Spdl CCW/CLNT Off (ISNC M164) ..... 5 - 18</td>
</tr>
<tr>
<td>Sub-Spdl Off/CLNT Off (ISNC M165) ..... 5 - 18</td>
</tr>
<tr>
<td>Rigid Tap (ISNC M29) ...................... 5 - 19</td>
</tr>
<tr>
<td>ISNC M Code Translation List - Page 4</td>
</tr>
<tr>
<td>--------------------------------------</td>
</tr>
<tr>
<td>Auxiliary Output 1 On (ISNC M52)</td>
</tr>
<tr>
<td>Auxiliary Output 2 On (ISNC M53)</td>
</tr>
<tr>
<td>Auxiliary Output 3 On (ISNC M54)</td>
</tr>
<tr>
<td>Auxiliary Output 4 On (ISNC M55)</td>
</tr>
<tr>
<td>Auxiliary Output 1 Off (ISNC M152)</td>
</tr>
<tr>
<td>Auxiliary Output 2 Off (ISNC M153)</td>
</tr>
<tr>
<td>Auxiliary Output 3 Off (ISNC M154)</td>
</tr>
<tr>
<td>Auxiliary Output 4 Off (ISNC M155)</td>
</tr>
<tr>
<td>Chuck Air On (ISNC M16)</td>
</tr>
<tr>
<td>Chuck Air Off (ISNC M17)</td>
</tr>
<tr>
<td>Set Max Rapid (ISNC M94)</td>
</tr>
<tr>
<td>Part Transfer Sub-spindle (ISNC 241)</td>
</tr>
<tr>
<td>End of Program (no rewind) (ISNC M2)</td>
</tr>
<tr>
<td>Steady Rest Clamp (ISNC M38)</td>
</tr>
<tr>
<td>Steady Rest Unclamp (ISNC M39)</td>
</tr>
<tr>
<td>Spindle Gear 1 (ISNC M41)</td>
</tr>
<tr>
<td>Spindle Gear 2 (ISNC M42)</td>
</tr>
<tr>
<td>Z-axis Tailstock Hitch (ISNC M128)</td>
</tr>
<tr>
<td>Z-axis Tailstock Unhitch (ISNC M129)</td>
</tr>
</tbody>
</table>
Program Stop (ISNC M0)

M0 causes part program execution to stop and the spindle to turn off. Press the START CYCLE button to resume program execution and restart the look-ahead program. Execute after tool change or motion (system M codes).

Optional Stop (ISNC M1)

M1 performs the same functions as M0, but can be toggled on or off through the Opt Stop key on the control.

- When the Opt Stop key is Off, the program ignores M1.
- When the Opt Stop key is On, the program stops on M1 so you can inspect the part. Press the START CYCLE button to resume running the program.

Execute after tool change or motion.

Main Spindle CW (ISNC M3)

M3 turns the main spindle (S1) on in the clockwise direction. M3 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.

An S code specifying the spindle speed must be programmed before the spindle will turn on. The advanced RPM Look Ahead feature that reduces part cycle time by ramping the spindle up to speed before the feed move begins improves the cycle time because the control does not dwell while the spindle ramps up to the programmed speed. When this feature is active, it is possible to create a part program containing an M3 and spindle speed command that does not turn on the spindle.

Execute after T and S codes, but before motion.

Format

S1:M3

Main Spindle CCW (ISNC M4)

M4 is the same as M0 except it commands the main spindle (S1) to rotate in the counterclockwise direction. Execute after T and S codes, but before motion.

Format

S1:M4

Main Spindle Stop (ISNC M5)

M5 turns the main spindle (S1) off. M5 executes at the end of the block after all motion occurs. Execute after tool change or motion (system M codes).

Format

S1:M5
**Live Tool CW (ISNC M33)**

⚠️ M33 is currently not supported.

M33 turns the live tooling spindle (S2) on in the clockwise direction. M3 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.

An S code specifying the spindle speed must be programmed before the spindle will turn on. The advanced RPM Look Ahead feature that reduces part cycle time by ramping the spindle up to speed before the feed move begins improves the cycle time because the control does not dwell while the spindle ramps up to the programmed speed. When this feature is active, it is possible to create a part program containing an M33 and spindle speed command that does not turn on the spindle.

Execute after T and S codes, but before motion.

**Format**

```
S2:M33
```

**Live Tool CCW (ISNC M34)**

⚠️ M34 is currently not supported.

M34 is the same as M0 except it commands the live tooling spindle (S2) to rotate in the counterclockwise direction. Execute after T and S codes, but before motion.

**Format**

```
S2:M34
```

**Live Tool Stop (ISNC M35)**

⚠️ M35 is currently not supported.

M35 turns the live tooling spindle (S2) off. M5 executes at the end of the block after all motion occurs. Execute after tool change or motion (system M codes).

**Format**

```
S2:M35
```
**Sub-spindle CW (ISNC M103)**

M103 turns the sub-spindle (S3) on in the clockwise direction. M103 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.

An S code specifying the spindle speed must be programmed before the spindle will turn on. The advanced RPM Look Ahead feature that reduces part cycle time by ramping the spindle up to speed before the feed move begins improves the cycle time because the control does not dwell while the spindle ramps up to the programmed speed. When this feature is active, it is possible to create a part program containing an M103 and spindle speed command that does not turn on the spindle.

Execute after T and S codes, but before motion.

**Format**

S3:M103

**Sub-spindle CCW (ISNC M104)**

M104 is the same as M0 except it commands the sub-spindle (S3) to rotate in the counterclockwise direction. Execute after T and S codes, but before motion.

**Format**

S3:M104

**Sub-spindle Stop (ISNC M105)**

M105 turns the sub-spindle (S3) off. M105 executes at the end of the block after all motion occurs. Execute after tool change or motion (system M codes).

**Format**

S3:M105

**Secondary Coolant (ISNC M7)**

M7 commands the secondary coolant pump to turn on, if so equipped. This command is cancelled by an M9 command. Execute before tool change or motion.

**Primary Coolant On (ISNC M8)**

M8 commands the primary coolant pump to turn on, typically flood coolant. This command is cancelled by an M9 command. Execute before tool change or motion.

**Chuck Pressure High (ISNC M67)**

M67 command sets the chucking pressure to high.
**Chuck Pressure Low (ISNC M66)**

M66 command sets the chucking pressure to low.

**Turret Index Reverse (ISNC M12)**

M12 is used to reverse the turret.

**Turret Index Forward (ISNC M13)**

M13 is used to move the turret forward.

**Chuck Open (ISNC M69)**

M69 is used to open the chuck for use with the optional Bar Feeder.

**Chuck Close (ISNC M68)**

M68 is used to close the chuck for use with the optional Bar Feeder.

**Main Spindle Orient (ISNC M19)**

M19 orient the main spindle to 0°.

**Chuck Open for Bar (ISNC M20)**

M20 is used with the optional Bar Feeder.

**Chuck Close for Bar (ISNC M21)**

M21 is used with the optional Bar Feeder.

**Start Bar Feeder (ISNC M22)**

M22 is used with the optional Bar Feeder.
ISNC M Code Translation List - Page 2

**Start Bar with Z (ISNC M23)**

M23 is used with the optional Bar Feeder to open the chuck then cause the bar feeder with Z to guide the stock.

**Part Conveyor On (ISNC M110)**

M110 turns on the optional Part Conveyor.

**Part Conveyor Off (ISNC M111)**

M111 turns off the optional Part Conveyor.

**Tailstock Quill Adv (ISNC M78)**

M78 is used to advance the optional Tailstock Quill. If either the **Interrupt** or **Stop Cycle** console key is pressed before the quill is fully advanced, the quill will automatically retract. If in Interrupt mode and you return to Auto Mode and press Start cycle, the Tailstock will advance before the program continues.

**Tailstock Quill Retr (ISNC M79)**

M79 is used to retract the optional Tailstock Quill.

**End of Program (ISNC M30)**

M30 stops program execution and moves the part program pointer to the top of the part program. M30 commands the spindle to turn off. Execute after tool change or motion (system M codes).

**Incr Cycle Count (ISNC M31)**

M31 increments the cycle counter.

**Incr Setup Cycle (ISNC M32)**

M32 increments the setup cycle.

**Feedrate Override (ISNC M48)**

M48 uses the value specified by the Feedrate Override dial (FPM %). Execute before tool change or motion.

**Ignore Feed Override (ISNC M49)**

M49 ignores the value specified by the Feedrate Override (FPR %) and uses the programmed feedrate.

**Conveyor On (ISNC M24)**

M24 is used to turn on the optional Conveyor.
**Conveyor Off (ISNC M25)**
M25 is used to turn off the optional Conveyor.

**Part Catcher w/Eject (ISNC M57)**
M57 is used to advance the optional Part Catcher with bar end eject.

**Part Catcher Advance (ISNC M58)**
M58 is used to advance the optional Part Catcher.

**Part Catcher Retract (ISNC M59)**
M59 is used to retract the optional Part Catcher.

**Sel External Chuck (ISNC M60)**
M60 is used to select external chucking.

**Sel Internal Chuck (ISNC M61)**
M61 is used to select internal chucking.
Tool Setter Retract (ISNC M71)
M71 is used to retract the optional Tool Setter.

Tool Setter Adv (ISNC M72)
M72 is used to advance the optional Tool Setter.

Auto Door Open (ISNC M85)
M85 is used to open the optional Auto Door.

Auto Door Close (ISNC M86)
M86 is used to close the optional Auto Door.

Single Block Off (ISNC M91)
M91 is used for inactivating a single block.

Single Block On (ISNC M92)
M92 is used for activating a single block.

Sub-spindle Chuck Open (ISNC M168)
M168 is used for opening the sub-spindle chuck.

Sub-spindle Chuck Close (ISNC M169)
M169 is used for closing the sub-spindle chuck.

Turning Mode (ISNC M130)
M130 sets the control to turning mode.

Milling Mode (ISNC M131)
M131 sets the control to milling mode.

C Axis Clamp (ISNC M41)
M41 enables the C axis clamp.

C Axis Hold (ISNC M133)
M133 enables the C axis hold. This code is valid for TMM machines only.

C Axis Unclamp (ISNC M40)
M40 releases both C axis clamps.
**Sub-spndl Ext Chuck (ISNC M160)**

M160 selects external chucking for sub-spindle.

**Sub-spndl Int Chuck (ISNC M161)**

M161 selects internal chucking for sub-spindle.

**W Torque Mon On (ISNC M186)**

M186 activates torque monitoring for the W-axis. Torque monitoring is available when picking up or transferring parts from either the Main spindle to the Sub-spindle or from the Sub-spindle to the Main spindle. The force, or load, of the material on the W-axis is monitored when M186 is used.

If the motor load is greater than 50% of the rated Full load for the W-axis, the program stops and an error message appears. The 50% value cannot and does not change. Use M186 in conjunction with G06 (Probe/Block Skip Function). With G06 active, the system stops the motion and deletes the remaining distance to go. G06 is a one-shot command and stays in effect until the end of the block.

M186 functions when Feed motions are used (G01, G02, G03). M186 does not function with Rapid motions (G00).

**M186 Example**

M241 (Allow Sub-spindle Chuck to Open when Spindle Running)

M114 (Open Sub-spindle Chuck)

G0 W200 (Rapid W to Approach position)

M186 (Activate W Axis Torque monitoring)

G1 G6 W180 F20 (Feed W Axis forward to Grab Part and activate Probe/Block skip)

M187 (Stop W Axis Torque Monitoring)

G0 W200 (Rapid W back to Approach position)

**W Torque Mon Off (ISNC M187)**

M187 deactivates torque monitoring for the W-axis.

**Block Skip Sync (ISNC M200)**

M200 allows you to turn on block skip while the part program is executing. Using this code ensures that blocks marked with block delete codes are skipped after block skip has been turned on, without requiring you to stop and restart the program. Execute before tool change or motion. Refer to *Block Skip Code, on page 1 - 6* for information about Block Skip.

**Sync Spindles Forward (ISNC M203)**

M203 synchronizes the main spindle and sub-spindle forward.

**Sync Spindles Reverse (ISNC M204)**

M204 synchronizes the main spindle and sub-spindle in reverse.
Clear Spindle Sync (ISNC M205)
M205 clears the main spindle and sub-spindle synchronization.

Bypass Chuck Intlk (ISNC M231)
M231 bypasses the main spindle chuck open interlock.

C3 Axis Clamp (ISNC M232)
M232 enables the C3 axis clamp.

C3 Axis Unclamp (ISNC M234)
M234 releases the C3 axis clamp.

Spindle CW/Coolant On (ISNC M63)
M63 performs 2 functions at once:

- M63 turns the spindle on in the clockwise direction. M63 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M63 commands the primary coolant pump to turn on, typically flood coolant. Execute before tool change or motion.

Format
S1 or S2:M63

Spindle CCW/Coolant Off (ISNC M64)
M64 performs 2 functions at once:

- M64 turns the spindle on in the counterclockwise direction. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M64 commands the primary coolant pump to turn off. Execute before tool change or motion.

Format
S1 or S2:M64
Spindle Off/Coolant Off (ISNC M65)

M65 performs 2 functions at once:

- M65 turns the spindle off. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M65 commands the primary coolant pump to turn off. Execute after tool change or motion.

Format

S1 or S2:M65

Sub-Spdl CW/CLNT On (ISNC M163)

M163 performs 2 functions at once:

- M163 turns the sub-spindle on in the clockwise direction. M63 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M163 commands the primary coolant pump to turn on, typically flood coolant. Execute before tool change or motion.

Format

S3:M163

Sub-Spdl CCW/CLNT Off (ISNC M164)

M164 performs 2 functions at once:

- M164 turns the sub-spindle on in the counterclockwise direction. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M164 commands the primary coolant pump to turn off. Execute before tool change or motion.

Format

S3:M164

Sub-Spdl Off/CLNT Off (ISNC M165)

M165 performs 2 functions at once:

- M165 turns the sub-spindle off. M64 will execute after Tool, Spindle, and E (Work Offset) codes execute, but before motion programmed in the block occurs.
- M165 commands the primary coolant pump to turn off. Execute after tool change or motion.

Format

S3:M165
Rigid Tap (ISNC M29)

M29 is used to activate rigid tapping. M29 must precede the tapping G code and must be on a block by itself.

M29 Example

M29
G84 Z-0.5 F0.05

ISNC M Code Translation List - Page 4

Auxiliary Output 1 On (ISNC M52)

M52 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 2 On (ISNC M53)

M53 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 3 On (ISNC M54)

M54 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 4 On (ISNC M55)

M55 is used to turn on auxiliary equipment or a unique machine function.

Auxiliary Output 1 Off (ISNC M152)

M152 is used to turn off auxiliary equipment or a unique machine function.

Auxiliary Output 2 Off (ISNC M153)

M153 is used to turn off auxiliary equipment or a unique machine function.

Auxiliary Output 3 Off (ISNC M154)

M154 is used to turn off auxiliary equipment or a unique machine function.

Auxiliary Output 4 Off (ISNC M155)

M155 is used to turn off auxiliary equipment or a unique machine function.

Chuck Air On (ISNC M16)

M16 turns on the chuck air blow.

Chuck Air Off (ISNC M17)

M17 turns off the chuck air blow.
**Set Max Rapid (ISNC M94)**

M94 is used to set the maximum rapid rate for the program. M94 requires an argument to specify the maximum rapid rate and must be expressed as MMPM.

**M94 Example:**

```
M94:5080
```

This block sets the Maximum Rapid Rate to 5080 MMPM (200 IPM).

**Part Transfer Sub-spindle (ISNC 241)**

M241 allows the sub-spindle chuck to open while the sub-spindle is running. This interlock bypass is cancelled automatically when M115 (Close Sub-spindle Chuck) is executed.

**End of Program (no rewind) (ISNC M2)**

M2 stops program execution and leaves the part program pointer on the block that follows the M2 command. M2 is required at the end of all programs. It commands the spindle to turn off. Execute at the end of the block after all motion occurs.

**Steady Rest Clamp (ISNC M38)**

The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

⚠️ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

This optional device is used with turning centers to hold long pieces of stock in the center while cutting.
**Steady Rest Unclamp (ISNC M39)**

After the spindle is stopped, M39 unclamps the optional Steady Rest device during automatic cycle operation. The spindle MUST BE stopped when this M-Code executes or an error will appear and the cycle will be halted.

⚠️ The spindle must be stopped prior to executing the M39 Steady Rest Unclamp.

The Steady Rest device can be manually hitched to the Z-Axis to assist with movement of the Steady Rest to the proper location. The position of the Steady Rest device is tracked in order to avoid colliding with the tailstock.

⚠️ The position of the Steady Rest can only be tracked when moving the Steady Rest with the Z-Axis. If the Steady Rest is moved manually without the use of the Z-Axis, then the control will not be aware of this new position for the Steady Rest which will either allow the tailstock to possibly collide with it or improperly limit the motion of the Z-Axis when hitched to the Tailstock.

A safety switch located on a bracket under the way cover also protects against the Steady Rest and Tailstock colliding. If the safety switch is tripped, the machine will enter the Emergency Stop mode.

This optional device is used with turning centers to hold long pieces of stock in the center while cutting.

**Spindle Gear 1 (ISNC M41)**

M41 sets the spindle to Gear 1, or the low gear range.

Low Gear Speed Range for TM18:
- Minimum RPM = 5
- Maximum RPM = 600

**Spindle Gear 2 (ISNC M42)**

M42 sets the spindle to Gear 2, or the high gear range.

High Gear Speed Range for TM18:
- Minimum RPM = 15
- Maximum RPM = 1600

**Z-axis Tailstock Hitch (ISNC M128)**

M128 is used to move the Z-axis to the Tailstock and subsequently to hitch to the Tailstock. The Tailstock can then move with the Z-axis to the specified Z location.

**Z-axis Tailstock Unhitch (ISNC M129)**

M129 is used to unhitch the Z-axis from the Tailstock. The Z-axis can then move freely from the tailstock. Bypass is cancelled automatically when M115 (Close Sub-spindle Chuck) is executed.
E Codes

For programs using Industry Standard NC (ISNC), work offsets are programmed using G54-G59 codes. Please refer to G54-G59 - Work Coordinates (G54 default), on page 4 - 37 and G54.1 - G54.93 - Aux Work Coordinates, on page 4 - 39. ISNC programs do not use E Codes.

For programs using Basic NC (BNC), offsets from machine zero are programmed with work offset E codes. The coordinates for each work offset are stored in the Work Offsets. Refer to Getting Started with WinMax Lathe, Part Setup—Work Offsets, on page 4 - 11.

All dimensions in the Work Offsets represent dimensions measured from a machine coordinate to the desired work offset position. E00 cancels any work offsets. A work offset value is provided for each axis that is installed on your machine.

Execution of a Work Offset E code does not cause tool motion unless axis motion is programmed in the block that contains the E code, or until motion is programmed in a following block.

There are two options for positioning Z0 along the Z axis. The option used depends on your machine-tool configuration:

- All machines can use G92 to establish the part program coordinates at the current position without generating any tool motion.
- Work Offsets are functional, depending on how the variables have been set.

Both axes move to a position relative to the new fixture offset as soon as any axis is programmed.

---

![Image of work offset activation](image.png)

*Figure 6–1. Work Offset: Immediate Activation*
T CODES

This section describes how to program a tool change and activate the tool's offsets.

Tool Change Sequence

You must precede a tool change sequence with the commands required to move the axes to the machine's tool change position.

A typical tool change sequence is shown below.

T0303
M03 S1200

⚠️ The T code will cause the turret to index into position. It is the responsibility of the part programmer or machine operator to ensure that the turret is in a safe position to perform an index.

The format for a T code is:

T __ nn

where:

__ is the tool number—The first digits in the T code identify the tool number that must be indexed into position.

nn is the tool offset number—The last two digits identify the tool table offsets to use for the current tool.

A T00 cancels any tool offsets.

The coordinates for each tool offset are stored in the Tool Offset table. All dimensions in the tool offset table represent dimensions measured from machine coordinate zero to the desired tool offset position. A tool offset value is provided for both axes.

Execution of a tool offset T code does not cause tool motion unless axis motion is programmed in the block that contains the T code, or until motion is programmed in a following block. This applies to Work Offset E codes as well.
Tool Offsets

The Tool Offset implements the Work Offset E codes. The drawings that follow show the tool movement.

- **Immediate Action**—the axes will move to a position relative to the new tool offset as soon as any axis is programmed.

![Tool Offset Diagram](image)

*Figure 7–1. Tool Offset Behaviors*
 RECORD OF CHANGES

704-0115-309, April 2013
 Revised by: K. Gross
 Approved by: J. Small, D. Skrzypczak, G. Traicoff April 2013

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>704-0115-309 Updates based on changes through v9.02.34 software.</td>
</tr>
<tr>
<td>Added G101-G109 for BNC and ISNC.</td>
</tr>
<tr>
<td>Updated some examples.</td>
</tr>
</tbody>
</table>

704-0115-308, January 2013
 Revised by: K. Gross
 Approved by: D. Skrzypczak January 2013

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>704-0115-308 Updates based on changes through v9.02.11 software.</td>
</tr>
<tr>
<td>Added information for Disable NC Editor Undo and Redo function.</td>
</tr>
</tbody>
</table>

704-0115-307, October 2012
 Revised by: K. Gross
 Approved by: D. Skrzypczak, K. Szabo, K. Van Blaircum October 2012

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>704-0115-307 Updates based on changes through v9.00.43 software.</td>
</tr>
<tr>
<td>Re-organized ISNC M Code presentation to follow the M Code Translation screens.</td>
</tr>
</tbody>
</table>

704-0115-306, March 2012
 Revised by: K. Gross
 Approved by: D. Skrzypczak March 2012

<table>
<thead>
<tr>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>704-0115-306 Updates based on changes through v8.1.3.48 software,</td>
</tr>
</tbody>
</table>
### Changes

**704-0115-305**, December 2011

Revised by: K. Gross

Approved by: D. Skrzypczak, J. Mulkey, G. Traicoff, K. Van Blaircum

#### Updates
- Based on changes through v8.1.2.30 software,
- Set cross-references between books.
- Added ScreenPath condition and paragraph tag.

**704-0115-304**, April 2011

Revised by: K. Gross

Approved by: D. Skrzypczak, J. Mulkey, G. Traicoff, K. Van Blaircum

#### Updates
- Based on changes through v8.1.2 software, including updated NC Editor screen and the addition of optional ISNC.

**704-0115-303**, June 2010, ECN 16538

Revised by: K. Gross

Approved by: D. Skrzypczak, J. Mulkey, C. Thale, G. Traicoff, K. Van Blaircum

#### Updates
- Based on changes through v8.1.1 software.
- Updated graphics for G02, G73, G74, G81, G82, G84, G98, and G99.
- Removed descriptions for M36 and M37.
704-0115-302, January 2009, ECN 16508

Revised by: K. Gross

Approved by: D. Skrzypczak, January 2009

Changes

- 704-0115-302 rB Updates based on v2.02.02 software. Added information to front matter about On-screen Help and accessing the On-screen Help in PDF format, November 2008.
- 704-0115-302 rA Updates based on v1.01, v1.02, and v2.0 software and the introduction of the Live Tooling (TMM Series) machine. October 2007.

704-0115-301, 14 July 2005, ECN 15866

Revised by: K. Gross

Approved by: G. Traicoff, C. Thale, D. Skrzypczak, July 2005

Changes

New manual release.
INDEX

Symbols
NC part programming 1 - 1
( ) Comment Command 1 - 3
/ Block Skip Command 1 - 3
/ Ignore Command 1 - 3

A
Absolute Programming G90 (default), ISNC 4 - 85
Absolute Programming G90, BNC 2 - 41
Activate Rigid Tapping M74, BNC 3 - 9
Activate Torque Monitoring for W M186, BNC 3 - 11
active tool offsets 7 - 1
AT-keyboard option 1 - 15
Auto Door Close M86, BNC 3 - 9
Auto Door Close M86, ISNC 5 - 15
Auto Door Open M85, BNC 3 - 9
Auto Door Open M85, ISNC 5 - 15
automatic calculations 1 - 16
Aux Work Coordinates G54.1 - G54.93, ISNC 4 - 39
Auxiliary Output 1 Off, M152, ISNC 5 - 19
Auxiliary Output 1 Off, M62, BNC 3 - 8
Auxiliary Output 1 On, M52, BNC 3 - 8
Auxiliary Output 1 On, M52, ISNC 5 - 19
Auxiliary Output 2 Off, M153, ISNC 5 - 19
Auxiliary Output 2 Off, M63, BNC 3 - 8
Auxiliary Output 2 On, M53, BNC 3 - 8
Auxiliary Output 2 On, M53, ISNC 5 - 19
Auxiliary Output 3 Off, M154, ISNC 5 - 19
Auxiliary Output 3 Off, M64, BNC 3 - 8
Auxiliary Output 3 On, M54, BNC 3 - 8
Auxiliary Output 3 On, M54, ISNC 5 - 19
Auxiliary Output 4 Off, M155, ISNC 5 - 19
Auxiliary Output 4 Off, M65, BNC 3 - 9
Auxiliary Output 4 On, M55, BNC 3 - 8
Auxiliary Output 4 On, M55, ISNC 5 - 19
Axial Surface of the Part – Sub-spindle with Linear Y G103, BNC 2 - 52
Axial Surface of the Part – Sub-spindle with Linear Y G103, BNC 2 - 99
Axial Surface of the Part – Sub-spindle with Linear Y G107, BNC 2 - 57
Axial Surface of the Part – Sub-spindle with Linear Y G107, ISNC 4 - 104
Axial Surface of the Part – Sub-spindle without Linear Y G105, BNC 2 - 54

B
Bar Feeder Finished Close Chuck M21, BNC 3 - 5
block
code processing order 1 - 7
data formats 1 - 9
Block Delete
Synchronization M200, BNC 3 - 11
Block Renumbering Mode softkey, NC 1 - 23
Block Skip
activating 1 - 6
code 1 - 6
Block Skip Enable field, NC Editor Settings screen 1 - 28
Block Skip Synchronization M200, ISNC 5 - 16
Bore Rapid Out Cycle G86, ISNC 4 - 80
Boring Cycle G85, ISNC 4 - 78
Bypass Chuck Open Interlock - Main Spindle M231, BNC 3 - 11
Bypass Chuck Open Interlock M231, ISNC 5 - 17

C
C Axis Clamp M41, ISNC 5 - 15
C Axis Hold M133, ISNC 5 - 15
C Axis Unclamp M40, ISNC 5 - 15
C Code 1 - 4
C motion dimension 1 - 4
C3 Axis Clamp M232, ISNC 5 - 17
C3 Axis Unclamp M234, ISNC 5 - 17
calculator, on-screen - xx
Cancel Drill Cycle G80, BNC 2 - 37
Cancel Work Coordinate Offsets G59, BNC 2 - 22
Canned Cycle Cancel (default) G80, ISNC 4 - 68
Change Comments Color softkey, NC Editor Settings screen 1 - 28
Change Current Font softkey, NC Editor Settings screen 1 - 28
Change... screen softkey, Comment Color field, NC Editor Setting screen 1 - 28
color
Percent 1 - 2
Special 1 - 2
chase in 2 - 26
chase out 2 - 26
Chip Breaker Peck Drilling G83.1, ISNC 4 - 74
Choose Font window, NC Editor Setting screen 1 - 28
Chuck Air Blow Off M17, BNC 3 - 5
Chuck Air Blow On M16, BNC 3 - 5
Chuck Air Off M17, ISNC 5 - 19
Chuck Air On M16 (ISNC) 5 - 19
Chuck Close - Sub-spindle M115, BNC 3 - 9
Chuck Close for Bar M21, ISNC 5 - 12
Chuck Close M15, BNC 3 - 5
Chuck Close M68, ISNC 5 - 12
Chuck Open - Sub-spindle M114, BNC 3 - 9
Chuck Open for Bar Feeder M20, ISNC 5 - 12
Chuck Open for Bar Feeder Start M20, BNC 3 - 5
Chuck Open M14, BNC 3 - 5
Chuck Open M69, ISNC 5 - 12
Chuck Pressure High M67, ISNC 5 - 11
Chuck Pressure Low M66, ISNC 5 - 12
Clear All Tags softkey, NC 1 - 25
Clear Spindle Sync M205, BNC 3 - 11
Clear Spindle Sync M205, ISNC 5 - 17
Clear Tag softkey, NC 1 - 25
clearance parameters, Threading 2 - 29
Clockwise Arc, G02, ISNC 4 - 8
Clockwise Circular Motion at Feed, G02, BNC 2 - 7
Clockwise Circular Motion at Feed, G02, ISNC 4 - 8
Counterclockwise Arc, G03, ISNC 4 - 8
Counterclockwise Circular Motion at Feed, G03, BNC 2 - 8
Current Font field, NC Editor Settings screen 1 - 28
cursor control keys 1 - 15
Cut Selection softkey, NC 1 - 21
cut, copy, and paste data blocks 1 - 16
Cutter Compensation 2 - 18
Cutter Compensation, ISNC 4 - 31
Cutter Radius Compensation Left G41, BNC 2 - 19
Cutter Radius Compensation Left G41, ISNC 4 - 32
Cutter Radius Compensation Off G40, BNC 2 - 18
Cutter Radius Compensation Off G40, ISNC 4 - 32
Cutter Radius Compensation Right G42, BNC 2 - 19
Cutter Radius Compensation Right G42, ISNC 4 - 33

D
Data Setting (Work/Tool Offsets/Tool Geometry) G10, ISNC 4 - 17
Data Setting Cancel G11, ISNC 4 - 18
De-activate Torque Monitoring for W M187, BNC 3 - 11
default format
inch mode 2 - 15
millimeter mode 2 - 16
millimeter mode, ISNC 4 - 24
Diameter Programming G08, BNC 2 - 13
Diameter Programming G08, ISNC 4 - 15
Direct Spindle Speed (default)G97, ISNC 4 - 93
Direct Spindle Speed G97, BNC 2 - 46
Disable Undo and Redo field, NC Editor Settings screen 1 - 29
display mode
programming 2 - 12, 4 - 14
display mode, unit 2 - 15, 2 - 16
display mode, unit, ISNC 4 - 23, 4 - 24
Draw key 1 - 15
Drill Cycle G81, BNC 2 - 37
Drill Cycle G81, ISNC 4 - 69
Drill Cycle Initial Level Return G98, BNC 2 - 47
Drill Cycle R Plane Return G99, BNC 2 - 48
Drill Cycle with Dwell G82, BNC 2 - 38
Drill Cycle with Dwell G82, ISNC 4 - 70
Dwell G04, BNC 2 - 10
Dwell G04, ISNC 4 - 12
M30, end of program 1 - 2
E00 6 - 1
editing
features 1 - 15
Emergency Stop - xx
Enable “C” Axis Clamp M132, BNC 3 - 10
Enable “C” Axis Hold Assist M133, BNC 3 - 10
Enable “C3” Axis Clamp M232, BNC 3 - 12
Enable Optional Numbering softkey, NC 1 - 24
Enable Auto Numbering softkey, NC 1 - 24
End Face Peck Drilling G74, ISNC 4 - 58
End key
Program Review screen 1 - 16
End of Program (no rewind) M02, BNC 3 - 4
End of Program (no rewind) M2, ISNC 5 - 20
End of Program (rewind) M30, BNC 3 - 6
End of Program M30, ISNC 5 - 13
end-of-block character 1 - 5
Enter Text To Search softkey, NC 1 - 20
error messages - xx
Exact Stop G09, BNC 2 - 15
Exact Stop G09, ISNC 4 - 17
Exit Editor softkey, NC 1 - 23, 1 - 26, 1 - 27
External Chucking, Select M60, BNC 3 - 8
External Chucking, Select M60, ISNC 5 - 14

F codes 1 - 3
F, Feedrate parameter 1 - 3
Face CW Tapping G84, ISNC 4 - 75
Face Drilling with Pecks G83, ISNC 4 - 72
Face Threading 2 - 28
Feed per Minute (default) G94, ISNC 4 - 92
Feed per Minute G94, BNC 2 - 45
Feed per Revolution G95, BNC 2 - 45
Feed per Revolution G95, ISNC 4 - 92
Feedrate Override M48, ISNC 5 - 13
feedrate parameter 1 - 3
file extension, NC 1 - 13
Find and Replace softkey, NC 1 - 20, 1 - 22
Finishing Cycle G70, ISNC 4 - 49
formats 1 - 9

G Code 2 - 3
cancel canned cycle 2 - 3
definition 1 - 3
groups 2 - 4
groups, ISNC 4 - 3
modal 2 - 4
preparatory functions, ISNC 4 - 1
same block 2 - 3
same group 2 - 3
G code functions 2 - 4
G code functions, ISNC 4 - 3
G Code Groups 2 - 3
G Code Groups 2 - 3
G Code Groups, ISNC 4 - 3
G Code Table 2 - 4
G Code Table, ISNC 4 - 3
G Code
preparatory functions 2 - 1
G00 - Linear Motion at Rapid (default), BNC 2 - 6
G00 - Rapid Traverse (default), ISNC 4 - 6
G01 - Linear Interpolation, ISNC 4 - 7
G01 - Linear Motion at Feed), BNC 2 - 6
G02 Clockwise Circular Motion at Feed, BNC 2 - 7
G02/G03 - Clockwise/Counterclockwise Arc, ISNC 4 - 8
G03 Counterclockwise Circular Motion at Feed, BNC 2 - 8
G04 - Dwell, BNC 2 - 10
G04 - Dwell, ISNC 4 - 12
G06 Probe/Block Skip, BNC 2 - 10
G06 Probe/Block Skip, ISNC 4 - 12
G07 Radius Programming, BNC 2 - 11
G07 Radius Programming, ISNC 4 - 13
G08 Diameter Programming (default), ISNC 4 - 15
G08 Diameter Programming, BNC 2 - 13
G09 - Exact Stop, BNC 2 - 15
G09 - Exact Stop, ISNC 4 - 17
G10 - Data Setting (Work/Tool Offsets/Tool Geometry), ISNC 4 - 17
G101 - Axial Surface of the Part - Main Spindle without Linear Y, ISNC 4 - 96
G101 - Axial Surface of the Part, Main Spindle without Linear Y, BNC 2 - 49
G102 - Radial Surface of the Part – Main Spindle without Linear Y, BNC 2 - 50
G102 - Radial Surface of the Part – Main Spindle without Linear Y, ISNC 4 - 97
G103 - Axial Surface of the Part – Main Spindle with Linear Y, BNC 2 - 52
G103 - Axial Surface of the Part – Main Spindle with Linear Y, ISNC 4 - 99
G104 - Radial Surface of the Part – Main Spindle with Linear Y, BNC 2 - 53
G104 - Radial Surface of the Part – Main Spindle with Linear Y, ISNC 4 - 100
G105 - Axial Surface of the Part – Sub-spindle without Linear Y, BNC 2 - 54
G105 - Axial Surface of the Part – Sub-spindle without Linear Y, ISNC 4 - 101
G106 - Radial Surface of the Part – Sub-spindle without Linear Y, BNC 2 - 55
G106 - Radial Surface of the Part – Sub-spindle without Linear Y, ISNC 4 - 102
G107 - Axial Surface of the Part – Sub-spindle with Linear Y, BNC 2 - 57
G107 - Axial Surface of the Part – Sub-spindle with Linear Y, ISNC 4 - 104
G108 - Radial Surface of the Part – Sub-spindle with Linear Y, BNC 2 - 58
G108 - Radial Surface of the Part – Sub-spindle with Linear Y, ISNC 4 - 105
G109 - Turning / Lathe Mode, BNC 2 - 59
G109 - Turning / Lathe Mode, ISNC 4 - 106
G11 - Data Setting Cancel, ISNC 4 - 18
G17 - XY Plane, ISNC 4 - 19
G18 - XZ Plane, ISNC 4 - 21
G19 - YZ Plane, ISNC 4 - 22
G20 - Inch Mode, BNC 2 - 15
G20 - Inch Mode, ISNC 4 - 23
G21 - Millimeter Mode, BNC 2 - 16
G21 - Millimeter Mode, ISNC 4 - 24
G28 - Return to Reference Point, ISNC 4 - 25
G32 - Constant Lead Thread Cutting, ISNC 4 - 27
G33 - Threading, BNC 2 - 17
G33 - Threading, ISNC 4 - 29
G40 - Cutter Radius Compensation Off (default), BNC 2 - 18
G40 - Cutter Radius Compensation Off (default), ISNC 4 - 32
G41 - Cutter Radius Compensation Left, BNC 2 - 19
G41 - Cutter Radius Compensation Left, ISNC 4 - 32
G42 - Cutter Radius Compensation Right, BNC 2 - 19
G42 - Cutter Radius Compensation Right, ISNC 4 - 33
G50 - Set Max RPM, ISNC 4 - 35
G50 - Set Part Setup, ISNC 4 - 35
G53 - Machine Coordinates, ISNC 4 - 36
G53 - Program Machine Coordinates, BNC 2 - 22
G54.1 - G54.93 - Aux Work Coordinates, ISNC 4 - 39
G54-G59 - Work Coordinates, ISNC 4 - 37
G59 - Cancel Work Coordinate Offsets, BNC 2 - 22
G61 - Precision Cornering, ISNC 4 - 40
G64 - Precision Cornering Cancel, ISNC 4 - 40
G65 - Subprogram Call, ISNC 4 - 40
G66 - Modal Subprogram Call, ISNC 4 - 47
G67 - Modal Subprogram Call Cancel (default), ISNC 4 - 49
G70 - Finishing Cycle, ISNC 4 - 49
G71 - Stock Removal in Turning, ISNC 4 - 51
G72 - Stock Removal in Facing, ISNC 4 - 54
G73 - Pattern Repeating, ISNC 4 - 56
G73 - Peck Drill with Chip Break Drill Cycle, BNC 2 - 23
G74 - End Face Peck Drilling, ISNC 4 - 58
G74 - Left Hand Tapping, BNC 2 - 24
G75 - Outer Diameter/Inner Diameter Drilling, ISNC 4 - 60
G76 - Multiple Threading Cycle, ISNC 4 - 62
G77 - Outer Diameter/Inner Diameter Cutting Cycle, ISNC 4 - 64
G78 - Threading Cycle, BNC 2 - 25
G78 - Threading Cycle, ISNC 4 - 66
G79 - Stock Removal in Facing Cycle, ISNC 4 - 67
G80 - Cancel Drill Cycle, BNC 2 - 37
G80 - Canned Cycle Cancel (default), ISNC 4 - 68
G81 - Drill Cycle, BNC 2 - 37
G81 - Drill Cycle, ISNC 4 - 69
G82 - Drill Cycle with Dwell, BNC 2 - 33
G82 - Drill Cycle with Dwell, ISNC 4 - 70
G83 - Face Drilling with Pecks, ISNC 4 - 72
G83 - Peck Drill Cycle, BNC 2 - 39
G83.1 - Chip Breaker Peck Drilling, ISNC 4 - 74
G84 - Face CW Tapping, ISNC 4 - 75
G84 - Right Hand Tapping, BNC 2 - 40
G84.2 (or G84 M29) - Rigid CW Tapping, ISNC 4 - 77
G85 - Boring Cycle, ISNC 4 - 78
G86 - Bore Rapid Out Cycle, ISNC 4 - 80
G87 - Side Drilling with Pecks/Dwell, ISNC 4 - 82
G88 - Side CW Tapping, ISNC 4 - 83
G89 - Side Boring Cycle, ISNC 4 - 84
G90 - Absolute Programming (default), BNC 2 - 41
G90 Absolute Programming (default), ISNC 4 - 85
G91 - Incremental Programming, BNC 2 - 43
G91 - Incremental Programming, ISNC 4 - 87
G92 - Work Coordinate Offsets / Spindle Max Speed / Multiple Thread Cutting Cycle, ISNC 4 - 89
G92 - Work Coordinate Offsets or Spindle Max Speed, BNC 2 - 45
G92 Multiple Thread Cutting Cycle, ISNC 4 - 89
G92 Spindle Max Speed, ISNC 4 - 89
G94 - Feed per Minute (default), BNC 2 - 45
G94 - Feed per Minute (default), ISNC 4 - 92
G95 - Feed per Revolution, BNC 2 - 45
G95 - Feed per Revolution, ISNC 4 - 92
G96 - Constant Surface Speed (CSS), BNC 2 - 46
G96 - Constant Surface Speed (CSS), ISNC 4 - 93
G97 - Direct Spindle Speed (default), BNC 2 - 46
G97 - Direct Spindle Speed (default), ISNC 4 - 93
G98 - Drill Cycle Initial Level Return (default), BNC 2 - 47
G98 - Return to Initial Level (default), ISNC 4 - 94
G99 - Drill Cycle R Plane Return, BNC 2 - 48
G99 - Return to R Point Level, ISNC 4 - 95
Increment Cycle Counter M31, BNC 3 - 6
Increment Cycle Counter M31, ISNC 5 - 13
Increment Cycle Counter M32, ISNC 5 - 13
Increment Setup Cycle Counter (M32), BNC 3 - 6
Incremental Programming G91, BNC 2 - 43
Incremental Programming G91, ISNC 4 - 87
Inner Diameter
  Threading 2 - 29
Insert/Overstrike Mode Toggle softkey, NC 1 - 21
Internal Chucking, Select M61, BNC 3 - 8
Internal Chucking, Select M61, ISNC 5 - 14
ISNC M Code Translation List - Page 1 screen, ISNC 5 - 3
ISNC M CODES F5 softkey, NC Settings screen, ISNC 5 - 2

J
Jump & Search Functions softkey, NC 1 - 24
Jump Page Backward softkey, NC 1 - 19
Jump Page Forward softkey, NC 1 - 19
Jump to Beginning softkey, NC 1 - 19
Jump to Block Number softkey, NC 1 - 19
Jump To End softkey, NC 1 - 19
Jump to Next Tagged Block softkey, NC 1 - 23
Jump to Previous Tagged Block softkey, NC 1 - 23
Jump to Selected Tag softkey, NC 1 - 25
Jump to Sequence Number softkey, NC 1 - 19
JUMP TO SUBPROGRAM F3 softkey 1 - 16

K
K parameter 1 - 3
keyboard 1 - 15
keyboard shortcuts 1 - 11
lead in parameters
  Threading 2 - 29
lead out parameters
  Threading 2 - 29
Left Hand Tapping G74, BNC 2 - 24
Linear Interpolation G01, ISNC 4 - 7
Linear Motion at Feed G01, BNC 2 - 6
Linear Motion at Rapid G00, BNC 2 - 6
Live Tool Clockwise M33, ISNC 5 - 10
Live Tool Counterclockwise M34, ISNC 5 - 10
Live Tool Stop M35, ISNC 5 - 10

M
M Code
definition 1 - 4
ISNC 5 - 1
M Codes
Basic NC 3 - 1
M0 - Program Stop, ISNC 5 - 9
M00 - Program Stop, BNC 3 - 4
M01 - Optional Stop, BNC 3 - 4
M02 - End of Program (no rewind), BNC 3 - 4
M03 - Spindle Clockwise, BNC 3 - 4
M04 - Spindle Counterclockwise, BNC 3 - 4
M05 - Spindle Off, BNC 3 - 4
M07 - Secondary Coolant On, BNC 3 - 4
M08 - Primary Coolant On, BNC 3 - 4
M09 - Coolant Off, BNC 3 - 5
M1 - Optional Stop, ISNC 5 - 9
M10 - Set Chucking Pressure to High, BNC 3 - 5
M103 - Sub-spindle clockwis, ISNC 5 - 11
M104 - Sub-spindle CCW, ISNC 5 - 11
M105 - Sub-spindle Stop, ISNC 5 - 11
M11 - Set Chucking Pressure to Low, BNC 3 - 5
M110 - Part Conveyor On, ISNC 5 - 13
M111 - Part Conveyor Off, ISNC 5 - 13
M114 - Chuck Open - Sub-spindle, BNC 3 - 9
M115 - Chuck Close - Sub-spindle, BNC 3 - 9
M118 - Sub Chuck Coolant On, BNC 3 - 10
M12 - Turret Index Reverse, BNC 3 - 5
M12 - Turret Index Reverse, ISNC 5 - 12
M128 - Z-axis Move to and Hitch to Tailstock, BNC 3 - 10
M128 - Z-axis Tailstock Hitch, ISNC 5 - 21
M129 - Z-axis Tailstock Unhitch, ISNC 5 - 21
M129 - Z-axis Unhitch from Tailstock, BNC 3 - 10
M13 - Turret Index Forward, BNC 3 - 5
M13 - Turret Index Fwd, ISNC 5 - 12
M130 - Turning Mode, BNC 3 - 10
M130 - Turning Mode, ISNC 5 - 15
M131 - Milling Mode, BNC 3 - 10
M131 - Milling Mode, ISNC 5 - 15
M132 - Enable “C” Axis Clamp, BNC 3 - 10
M133 - C Axis Hold, ISNC 5 - 15
M133 - Enable “C” Axis Hold Assist, BNC 3 - 10
M134 - Release Both “C” Axis Clamps, BNC 3 - 10
M14 - Chuck Open, BNC 3 - 5
M15 - Chuck Close, BNC 3 - 5
M152 - Auxiliary Output 1 Off, ISNC 5 - 19
M153 - Auxiliary Output 2 Off, ISNC 5 - 19
M154 - Auxiliary Output 3 Off, ISNC 5 - 19
M155 - Auxiliary Output 4 Off, ISNC 5 - 19
M16 - Chuck Air Blow On, BNC 3 - 5
M16 - Chuck Air On, ISNC 5 - 19
M160 - Select External Chucking - Subspindle, BNC 3 - 10
M160 - Sub-spndl Ext Chuck, ISNC 5 - 16
M161 - Select Internal Chucking - Subspindle, BNC 3 - 10
M163 Sub-spindle CW/Coolant On, ISNC 5 - 18
M163 - Sub-spindle CCW/Coolant Off, ISNC 5 - 18
M165 Sub-spindle Off/Coolant Off, ISNC 5 - 18
M168 - Sub-spindle Chuck Open, ISNC 5 - 15
M169 - Sub-spindle Chuck Close, ISNC 5 - 15
M17 - Chuck Air Blow Off, BNC 3 - 5
M17 - Chuck Air Off, ISNC 5 - 19
M18 - Main Chuck Coolant On, BNC 3 - 5
M186 - Activate Torque Monitoring for W, BNC 3 - 11
M186 - W Torque Mon On, ISNC 5 - 16
M187 - De-activate Torque Monitoring for W, BNC 3 - 11
M187 - W Torque Mon Off, ISNC 5 - 16
M19 - Main Spindle Orient, ISNC 5 - 12
M19 - Spindle Orient, BNC 3 - 5
M2 - End of Program (no rewind), ISNC 5 - 20
M20 - Chuck Open for Bar Feeder Start, BNC 3 - 5
M20 - Chuck Open for Bar, ISNC 5 - 12
M200 - Block Skip Synch, ISNC 5 - 16
M200 - Block Skip Synchronization, BNC 3 - 11
M203 - Sync Spindles Forward Mode, BNC 3 - 11
M203 - Sync Spindles Forward, ISNC 5 - 16
M204 - Sync Spindles Reverse Mode, BNC 3 - 11
M204 - Sync Spindles Reverse, ISNC 5 - 16
M205 - Clear Spindle Sync, BNC 3 - 11
M205 - Clear Spindle Sync, ISNC 5 - 17
M21 - Bar Feeder Finished Close Chuck, BNC 3 - 5
M21 - Chuck Close for Bar, ISNC 5 - 12
M22 - Start Bar Feeder (Bar Load), BNC 3 - 5
M22 - Start Bar Feeder, ISNC 5 - 12
M23 - Start Bar Feeder for Z Guided Feed, BNC 3 - 6
M23 - Start Bar with Z, ISNC 5 - 13
M231 - Bypass Chuck Intlk, ISNC 5 - 17
M231 - Bypass Chuck Open Interlock - Main Spindle, BNC 3 - 11
M232 - C3 Axis Clamp, ISNC 5 - 17
M232 - Enable “C3” Axis Clamp, BNC 3 - 6
M234 - C3 Axis Unclamp (ISNC) 5 - 17
M234 - Release “C3” Axis Clamp, BNC 3 - 12
M24 - Conveyor On, ISNC 5 - 13
M24 - Part Conveyor On, BNC 3 - 6
M241 - Sub-chuck open while spindle running for part transfer, BNC 3 - 12
M241 Part Transfer Sub-spindle, ISNC 5 - 20
M25 - Conveyor Off, ISNC 5 - 14
M25 - Part Conveyor Off, BNC 3 - 6
M28 - Tailstock Quill Advance, BNC 3 - 6
M29 - Rigid Tap, ISNC 5 - 19
M29 - Tailstock Quill Retract, BNC 3 - 6
M3 - Main Spindle CW, ISNC 5 - 9
M30 - End of Program (rewind), BNC 3 - 6
M30 - End of Program, ISNC 5 - 13
M31 - Incr Cycle Count, ISNC 5 - 13
M31 - Increment Cycle Counter, BNC 3 - 6
M32 - Incr Cycle Count, ISNC 5 - 13
M32 - Increment Setup Cycle Counter, BNC 3 - 6
M33 - Live Tool CW, ISNC 5 - 10
M34 - Live Tool CCW, ISNC 5 - 10
M35 - Live Tool Stop, ISNC 5 - 10
M38 - Steady Rest Clamp, BNC 3 - 6
M38 - Steady Rest Clamp, ISNC 5 - 20
M39 - Steady Rest Unclamp, BNC 3 - 7
M39 - Steady Rest Unclamp, ISNC 5 - 21
M4 - Main Spindle CCW (ISNC) 5 - 9
M40 - C Axis Unclamp, ISNC 5 - 15
M41 - C Axis Clamp, ISNC 5 - 15
M41 - Spindle Gear 1 / Low Gear Range, BNC 3 - 7
M41 - Spindle Gear 1, ISNC 5 - 21
M42 - Spindle Gear 2 5 - 21
M42 - Spindle Gear 2 / High Gear Range, BNC 3 - 7
M48 - Feedrate Override, ISNC 5 - 13
M48 - Use Feedrate Override, BNC 3 - 7
M49 - Ignore Feedrate Override, BNC 3 - 7
M49 - Ignore Feedrate Override, BNC 3 - 7
M5 - Main Spindle Stop, ISNC 5 - 9
M50 - Conveyor On, BNC 3 - 8
M51 - Conveyor Off, BNC 3 - 8
M52 - Auxiliary Output 1 On, BNC 3 - 8
M52 - Auxiliary Output 1 On, ISNC 5 - 19
M53 - Auxiliary Output 2 On, BNC 3 - 8
M53 - Auxiliary Output 2 On, ISNC 5 - 19
M54 - Auxiliary Output 3 On, BNC 3 - 8
M54 - Auxiliary Output 3 On, ISNC 5 - 19
M55 - Auxiliary Output 4 On, BNC 3 - 8
M55 - Auxiliary Output 4 On, ISNC 5 - 19
M57 - Part Catcher w/Eject, ISNC 5 - 14
M57 - Use Part Catcher with Bar End Eject, BNC 3 - 8
M58 - Part Catcher Advance, BNC 3 - 8
M58 - Part Catcher Advance, ISNC 5 - 14
M59 - Part Catcher Retract, BNC 3 - 8
M59 - Part Catcher Retract, ISNC 5 - 14
M60 - Select External Chuck, ISNC 5 - 14
M60 - Select External Chucking, BNC 3 - 8
M61 - Select Internal Chuck, ISNC 5 - 14
M61 - Select Internal Chucking, BNC 3 - 8
M62 - Auxiliary Output 1 Off, BNC 3 - 8
M63 - Auxiliary Output 2 Off, BNC 3 - 8
M63 - Spindle CW/Coolant On, ISNC 5 - 17
M64 - Auxiliary Output 3 Off, BNC 3 - 8
M64 - Spindle CCW/Coolant Off, ISNC 5 - 17
M65 - Auxiliary Output 4 Off, BNC 3 - 9
M65 - Spindle Off/Coolant Off, ISNC 5 - 18
M66 - Chuck Pressure Low, ISNC 5 - 12
M67 - Chuck Pressure High, ISNC 5 - 11
M68 - Chuck Close, ISNC 5 - 12
M69 - Chuck Open, ISNC 5 - 12
M7 - Secondary Coolant, ISNC 5 - 11
M71 - Tool Setter Retract, BNC 3 - 9
M71 - Tool Setter Retract, ISNC 5 - 15
M72 - Tool Setter Adv, ISNC 5 - 15
M72 - Tool Setter Advance, BNC 3 - 9
M74 - Activate Rigid Tapping, BNC 3 - 9
M78 - Tailstock Quill Adv, ISNC 5 - 13
M79 - Tailstock Quill Retr, ISNC 5 - 13
M8 - Primary Coolant On, ISNC 5 - 11
M85 - Auto Door Open, BNC 3 - 9
M85 - Auto Door Open, ISNC 5 - 15
M86 - Auto Door Close, BNC 3 - 9
M86 - Auto Door Close, ISNC 5 - 15
M9 - Coolant Off, ISNC 5 - 11
M91 - Single Block Inactive, BNC 3 - 9
M91 - Single Block Off, ISNC 5 - 15
M92 - Single Block Active, BNC 3 - 9
M92 - Single Block On, ISNC 5 - 15
M94 - Set Program Maximum Rapid Rate,
  BNC 3 - 9
M94 - Set Program Maximum Rapid Rate,
  ISNC 5 - 20
M98 subprogram calls, ISNC 4 - 42
Machine Coordinates G53, ISNC 4 - 36
Main Chuck Coolant On M18, BNC 3 - 5
Main Spindle Clockwise M3, ISNC 5 - 9
Main Spindle Counterclockwise M4, ISNC 5 - 9
Main Spindle Orient M19, ISNC 5 - 12
Main Spindle Stop M5, ISNC 5 - 9
messages
eroof G21, BNC 2 - 16
Millimeter Mode G21, ISNC 4 - 24
millimeter mode, default format 2 - 16
Millling Mode M131, BNC 3 - 10
Milling Mode M131, ISNC 5 - 15
Modal Subprogram Call Cancel (default)
  G67, ISNC 4 - 49
Modal Subprogram Call G66, ISNC 4 - 47
mode
  programming - xx
multiple blocks, select for editing 1 - 16
multiple start parameters
  Threading 2 - 30
Multiple Thread Cutting Cycle G92, ISNC 4 - 89
Multiple Threading Cycle G76, ISNC 4 - 62
N
N, sequence number 1 - 4
NC Editor
  viewing the screen 1 - 11
NC Editor Settings softkey, NC 1 - 27
NC part program
  components 1 - 2
  overview 1 - 1
  starting a new 1 - 15
NC part program 1 - 1
NC post processor
  formats 1 - 7
NC Subprogram Review screen 1 - 15
new NC program, starting a 1 - 15
NEXT M CODE SCREEN F2 softkey, ISNC 5 - 3
number, sequence 1 - 4
O
off-line part program formats 1 - 7
on-screen calculator - xx
optional AT-Keyboard 1 - 15
Optional Stop M01 (ISNC) 5 - 9
Optional Stop M01, BNC 3 - 4
Outer Diameter
  Threading 2 - 29
Outer Diameter/Inner Diameter Cutting
  Cycle G77, ISNC 4 - 64
Outer Diameter/Inner Diameter Drilling
  G75, ISNC 4 - 60
overview, NC 1 - 1
P
parameter, starting thread angle for
  threading G33 1 - 4
Page Down key
  Program Review screen 1 - 16
Page Up key
  Program Review screen 1 - 16
parameters, program 1 - 13
Part Catcher Advance M58, BNC 3 - 8
Part Catcher Advance M58, ISNC 5 - 14
Part Catcher Retract M59, BNC 3 - 8
Part Catcher Retract M59, ISNC 5 - 14
Part Catcher with Eject M57, ISNC 5 - 14
Part Conveyor Off (M25), BNC 3 - 6
Part Conveyor Off M111, ISNC 5 - 13
Part Conveyor On (M24), BNC 3 - 6
Part Conveyor On M110, ISNC 5 - 13
part program
  axis motion 1 - 14
  components 1 - 2
  feedrates 1 - 14
  NC 1 - 1
  sample Thread 2 - 33
  setup information 1 - 13
  start 1 - 2, 1 - 15
part setup 1 - 13
part transfer M241, BNC 3 - 12
part transfer M241, ISNC 5 - 20
Part Transfer Sub-spindle M241, ISNC 5 - 20
paste 1 - 16
Paste Selection softkey, NC 1 - 21
Pattern Repeating G73, ISNC 4 - 56
Peck Drill Cycle G83, BNC 2 - 39
Peck Drill with Chip Break Drill Cycle G73,
  BNC 2 - 23
Percent character 1 - 2
pitch parameter 1 - 3
pop-up text entry window 1 - 15

8 - Index 704-0115-309 WinMax Lathe NC Programming
power-on control - xx
Precision Cornering Cancel G64, ISNC 4 - 40
Precision Cornering G61, ISNC 4 - 40
preparatory functions - G codes, Basic NC 2 - 1
preparatory functions - G codes, ISNC 4 - 1
PREVIOUS M CODE SCREEN F1 softkey, ISNC 5 - 3
Primary Coolant On M08, BNC 3 - 4
Primary Coolant On M8, ISNC 5 - 11
Probe/Block Skip G06, BNC 2 - 10
Probe/Block Skip G06, ISNC 4 - 12
program components, NC 1 - 2
Display Mode 2 - 12, 4 - 14
ingredients features 1 - 15
NC 1 - 1
new 1 - 15
parameters 1 - 13
Review 1 - 16
Program Machine Coordinates G53, BNC 2 - 22
program status - xx
Program Stop M0, ISNC 5 - 9
Program Stop M00, BNC 3 - 4
programming mode keys 1 - 15

R

R parameter 1 - 4
Radial Surface of the Part – Main Spindle with Linear Y G104, BNC 2 - 53
Radial Surface of the Part – Main Spindle with Linear Y G104, ISNC 4 - 100
Radial Surface of the Part – Main Spindle without Linear Y G102, BNC 2 - 50
Radial Surface of the Part – Main Spindle without Linear Y G102, ISNC 4 - 97
Radial Surface of the Part – Sub-spindle with Linear Y G108, BNC 2 - 58
Radial Surface of the Part – Sub-spindle with Linear Y G108, ISNC 4 - 105
Radial Surface of the Part – Sub-spindle without Linear Y G106, BNC 2 - 55
Radial Surface of the Part – Sub-spindle without Linear Y G106, ISNC 4 - 102
Radius Programming G07, BNC 2 - 11
Radius Programming G07, ISNC 4 - 13
ranges 1 - 9
rapid rate, set program maximum M94, ISNC 5 - 20
Rapid Traverse G00, ISNC 4 - 6
Redo softkey, NC 1 - 22
Release “C3” Axis Clamp M234, BNC 3 - 12
Release Both “C” Axis Clamps M134, BNC 3 - 10
Renumber Numbered Blocks softkey, NC 1 - 24
Renumber Selected Blocks softkey, NC 1 - 24
RESET M CODES TO DEFAULT VALUES F5 softkey, ISNC 5 - 3
Reset Start/End Markers softkey, NC 1 - 26
Return to Initial Level (default) G98, ISNC 4 - 94
Return to R Point Level G99, ISNC 4 - 95
Return to Reference Point G28, ISNC 4 - 25
review 1 - 16
Right Hand Tapping G84, BNC 2 - 40
Rigid CW Tapping G84.2 (or G84 M29), ISNC 4 - 77
Rigid Tap M29, ISNC 5 - 19

S

S Code 1 - 4
S, spindle speed parameter 1 - 4
S1 Code 1 - 4
S1, spindle speed parameter 1 - 4
S2 Code 1 - 4
S2, spindle speed parameter 1 - 4
S3, spindle speed parameter 1 - 4
sample part programs 2 - 33
sample screens - xix
sample Thread part programs 2 - 33
Save ISNC M CODES F7 softkey, ISNC 5 - 3
Save NC State To File F3 softkey, ISNC 1 - 29
Search Again softkey, NC 1 - 20
Search for Text softkey, NC 1 - 19
Secondary Coolant M7, ISNC 5 - 11
Secondary Coolant On M07, BNC 3 - 4
Select External Chuck M60, ISNC 5 - 14
Select External Chucking - Sub-spindle M160, BNC 3 - 10
Select External Chucking M60, BNC 3 - 8
Select Internal Chuck M61, ISNC 5 - 14
Select Internal Chucking - Sub-spindle M161, BNC 3 - 10
Select Internal Chucking M61, BNC 3 - 8
sequence number 1 - 4
Set Chucking Pressure to High M10, BNC 3 - 5
Set Chucking Pressure to Low M11, BNC 3 - 5
Set End Marker softkey, NC 1 - 26
Set Max RMP G50, ISNC 4 - 35
Set Part Setup G50, ISNC 4 - 35
Set Program Maximum Rapid Rate M94, BNC 3 - 9
Set Program Maximum Rapid Rate M94, ISNC 5 - 20
Set Start Marker softkey, NC 1 - 26
setup
Part 1 - 13
Tool 1 - 13
Side Boring Cycle G89, ISNC 4 - 84
Side CW Tapping G88, ISNC 4 - 83
Side Drilling with Pecks/Dwell G87, ISNC 4 - 82
Single Block Active M92, BNC 3 - 9
Single Block Inactive M91, BNC 3 - 9
Single Block Off M91, ISNC 5 - 15
Single Block On M92, ISNC 5 - 15
Spindle CCW/Coolant Off M64, ISNC 5 - 17
Spindle Clockwise M03, BNC 3 - 4
Spindle Counter-clockwise M04, BNC 3 - 4
Spindle CW/Coolant On M63, ISNC 5 - 17
Spindle Gear 1 (M41), ISNC 5 - 21
Spindle Gear 1 / Low Gear Range (M41), BNC 3 - 7
Spindle Gear 2 (M42), ISNC 5 - 21
Spindle Gear 2 / High Gear Range (M42), BNC 3 - 7
Spindle Max Speed G92, BNC 2 - 45
Spindle Max Speed G92, ISNC 4 - 89
Spindle Off M05, BNC 3 - 4
Spindle Off/Coolant Off M65, ISNC 5 - 18
Spindle Orient M19, BNC 3 - 5
Spindle Speed parameter
live-tooling machines (TMM Series)
spindle 1 1 - 4
live-tooling machines (TMM Series)
spindle 2 1 - 4
two-axis machines (TM Series) 1 - 4
spring pass 2 - 31
start
program 1 - 2
Start Bar Feeder (Bar Load) M22, BNC 3 - 5
Start Bar Feeder for Z Guided Feed (M23), BNC 3 - 6
Start Bar Feeder M22, ISNC 5 - 12
Start Bar with Z M23, ISNC 5 - 13
status, program - xx
Steady Rest Clamp (M38), BNC 3 - 6
Steady Rest Clamp (M38), ISNC 5 - 20
Steady Rest Unclamp (M39), BNC 3 - 7
Steady Rest Unclamp (M39), ISNC 5 - 21
Stock Removal in Facing Cycle G79, ISNC 4 - 67
Stock Removal in Facing G72, ISNC 4 - 54
Stock Removal in Turning G71, ISNC 4 - 51
Stop (Optional) M01, BNC 3 - 4
Stop (program) M00, BNC 3 - 4
Stop program M0, ISNC 5 - 9
Stop, Optional M1, ISNC 5 - 9
Straight ID Thread 2 - 36
Straight OD Thread 2 - 33
Straight OD Thread with Lead In/Lead Out Angles 2 - 34
Sub Chuck Coolant On M115, BNC 3 - 10
Sub-chuck open while spindle running for part transfer M241, BNC 3 - 12
Subprogram Call G65, ISNC 4 - 40
subprogram calls M98, ISNC 4 - 42
Sub-Spdl CCW/CLNT Off M164, ISNC 5 - 18
Sub-Spdl CW/CLNT On M163, ISNC 5 - 18
Sub-Spdl Off/CLNT Off M165, ISNC 5 - 18
Sub-spindle Chuck Close M169, ISNC 5 - 15
Sub-spindle Chuck Open M168, ISNC 5 - 15
Sub-spindle Clockwise M103, ISNC 5 - 11
Sub-spindle Counter-clockwise M104, ISNC 5 - 11
Sub-spindle External Chucking M160 (ISNC) 5 - 16
Sub-spindle Internal Chucking M161, ISNC 5 - 16
Sub-spindle Stop M105, ISNC 5 - 11
Sync Spindles Forward M203, ISNC 5 - 16
Sync Spindles Forward Mode M203, BNC 3 - 11
Sync Spindles Reverse M204, ISNC 5 - 16
Sync Spindles Reverse Mode M204, BNC 3 - 11

T
T Code 7 - 1
T, tool select parameter 1 - 4
T00 7 - 1
Tag Block softkey, NC 1 - 23
Tagged Block List softkey, NC 1 - 23
Tailstock Quill Advance M28, BNC 3 - 6
Tailstock Quill Advance M78, ISNC 5 - 13
Tailstock Quill Retract M29, BNC 3 - 6
Tailstock Quill Retract M79, ISNC 5 - 13
Tapered OD Thread 2 - 35
tapered thread 2 - 17
Threading 2 - 31
text entry window 1 - 15

Tapered OD Thread 2 - 33
Stock Removal in Facing G72, ISNC 4 - 54

10 - Index 704-0115-309 WinMax Lathe NC Programming
Threading 2 - 33
  face 2 - 28
  lead in parameters 2 - 29
  lead out parameters 2 - 29
  multiple start parameters 2 - 30
  straight ID thread 2 - 36
  straight OD 2 - 33
  straight OD Thread with Lead In/Lead Out Angles 2 - 34
  Tapered OD Thread 2 - 35
turn 2 - 28
Threading Cycle G78, BNC 2 - 25
Threading Cycle G78, ISNC 4 - 66
Threading G33, BNC 2 - 17
Threading G33, ISNC 4 - 29
tool change 7 - 1
tool sequence 7 - 1
tool offset
  behaviors 7 - 2
table 7 - 1
Tool Review screen 1 - 15
tool select 1 - 4
Tool Setter Advance M72, ISNC 5 - 15
Tool Setter Retract M71, BNC 3 - 9
Tool Setter Retract M71, ISNC 5 - 15
tool setup 1 - 13
torque monitoring, BNC 3 - 11
torque monitoring, ISNC 5 - 16
turn and face thread equivalents 2 - 28
Turn Threading 2 - 28
  Inner Diameter 2 - 29
  Outer Diameter 2 - 29
Turning / Lathe Mode G109, BNC 2 - 59
Turning / Lathe Mode G109, ISNC 4 - 106
Turning Mode M130, BNC 3 - 10
Turning Mode M130, ISNC 5 - 15
Turret Index Forward M13, BNC 3 - 5
Turret Index Forward M13, ISNC 5 - 12
Turret Index Reverse M12, BNC 3 - 5
Turret Index Reverse M12, ISNC 5 - 12

V
  Verify key 1 - 15
  view
    the NC Editor 1 - 11

W
  motion dimension 1 - 4
  W Torque Monitoring Off M187, ISNC 5 - 16
  W Torque Monitoring On M186, ISNC 5 - 16
Work Coordinate Offsets / Spindle Max
  Speed / Multiple Thread Cutting Cycle
  G92, ISNC 4 - 89
Work Coordinate Offsets G92, BNC 2 - 45
Work Coordinates G54-G59, ISNC 4 - 37
work offset 6 - 1
  E offset 6 - 1

X
  axis center/offset coordinate 1 - 3
    motion dimension 1 - 3
  XY Plane G17, ISNC 4 - 19
  XZ Plane G18, ISNC 4 - 21

Y
  motion dimension 1 - 4
  YZ Plane G19, ISNC 4 - 22

Z
  axis center/offset coordinate 1 - 3
    motion dimension 1 - 4
  Z-axis Move to and Hitch to Tailstock (M128), BNC 3 - 10
  Z-axis Tailstock Hitch M128, ISNC 5 - 21
  Z-axis Tailstock Unhitch (M129), ISNC 5 - 21
  Z-axis Unhitch from Tailstock (M129), BNC 3 - 10

U
  Undo softkey, NC 1 - 22
  Unit Display Mode 2 - 15, 2 - 16
  Unit Display Mode, ISNC 4 - 23, 4 - 24
  units of measure - xx
  Use Feedrate Override M48, BNC 3 - 7
  Use Part Catcher with Bar End Eject (M57), BNC 3 - 8