OmniTurn Training

Jeff Richlin
631 694 9400
jrichlin@gmail.com
# Codes Honored by the OmniTurn control

<table>
<thead>
<tr>
<th>Code</th>
<th>Usage</th>
<th>Description</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>G00</td>
<td>Rapid move</td>
<td>11,12</td>
</tr>
<tr>
<td>G01</td>
<td>G01Fn</td>
<td>Feed move</td>
<td>12,13</td>
</tr>
<tr>
<td>G02</td>
<td>G02XnZnInKnFn</td>
<td>Arc -Clockwise</td>
<td>6,15,17-24</td>
</tr>
<tr>
<td>G02</td>
<td>G02XnZnRn</td>
<td>Arc -Clockwise</td>
<td>6,17-24</td>
</tr>
<tr>
<td>G03</td>
<td>G03XnZnInKnFn</td>
<td>Arc -Counter Clockwise</td>
<td>6,17-24</td>
</tr>
<tr>
<td>G03</td>
<td>G03XnZnRn</td>
<td>Arc -Counter Clockwise</td>
<td>6,17-24</td>
</tr>
<tr>
<td>G04</td>
<td>G04Fn</td>
<td>Dwell</td>
<td>6,25,62</td>
</tr>
<tr>
<td>G10</td>
<td>G10XnZn</td>
<td>Work Shift</td>
<td>6,26-28,73</td>
</tr>
<tr>
<td>G33</td>
<td>G33XnZnInKnCnPO</td>
<td>Threading cycle</td>
<td>6,29-36</td>
</tr>
<tr>
<td>G35</td>
<td>G35n</td>
<td>Extra Course feeds in IPR (G3 and G2 control only)</td>
<td>6,29,36,74</td>
</tr>
<tr>
<td>G36</td>
<td>G36</td>
<td>Cancels G35 (G3 and G2 controls only)</td>
<td>6,36,74</td>
</tr>
<tr>
<td>G40</td>
<td>G40</td>
<td>Cancels Tool Nose Radius Compensation</td>
<td>16,37-43</td>
</tr>
<tr>
<td>G41</td>
<td>G41</td>
<td>Left hand Tool Nose Radius Compensation</td>
<td>16,37-43</td>
</tr>
<tr>
<td>G42</td>
<td>G42</td>
<td>Right hand Tool Nose Radius Compensation</td>
<td>37-43</td>
</tr>
<tr>
<td>G70</td>
<td>G70</td>
<td>Inch mode</td>
<td>6,44</td>
</tr>
<tr>
<td>G71</td>
<td>G71</td>
<td>Metric mode</td>
<td>6,44</td>
</tr>
<tr>
<td>G72</td>
<td>G72</td>
<td>Diameter programming mode</td>
<td>6,10,14,16,21,22,29,38,44,46,49,59</td>
</tr>
<tr>
<td>G73</td>
<td>G73</td>
<td>Radius programming mode</td>
<td>6,10,14,16,21,20,22,29,38,44,46,49,59</td>
</tr>
<tr>
<td>G74</td>
<td>G74XnZnInUnFn</td>
<td>Box Roughing cycle</td>
<td>45-47</td>
</tr>
<tr>
<td>G75</td>
<td>G75UnFnPn</td>
<td>Box Contour Roughing cycle</td>
<td>48-52,54</td>
</tr>
<tr>
<td>G76</td>
<td>G76Sn</td>
<td>Minimum spindle speed for constant surface feet</td>
<td>6,60</td>
</tr>
<tr>
<td>G77</td>
<td>G77Sn</td>
<td>Maximum spindle speed for constant surface feet</td>
<td>6,60</td>
</tr>
<tr>
<td>G78</td>
<td>G78UnFnPn</td>
<td>Rough Contour Cycle</td>
<td>51-55</td>
</tr>
<tr>
<td>G81</td>
<td>G81ZnFn</td>
<td>Drill cycle</td>
<td>6,56</td>
</tr>
<tr>
<td>G83</td>
<td>G83ZnKnFn</td>
<td>Peck drill cycle</td>
<td>6,57,58</td>
</tr>
<tr>
<td>G88</td>
<td>G88</td>
<td>Precision C axis orientation</td>
<td></td>
</tr>
<tr>
<td>G89</td>
<td>G89</td>
<td>Stop spindle and lock (C-Axis only)</td>
<td></td>
</tr>
<tr>
<td>G90</td>
<td>G90</td>
<td>Absolute mode selection</td>
<td>5,6,10,12,17,19,21,20,56,57,59</td>
</tr>
<tr>
<td>G91</td>
<td>G91</td>
<td>Incremental mode selection</td>
<td>5,6,10,17,56,57,59</td>
</tr>
<tr>
<td>G92</td>
<td>G92XnZn</td>
<td>Preset axis position</td>
<td>36,59,74</td>
</tr>
<tr>
<td>G94</td>
<td>G94Fn</td>
<td>Inches per minute mode</td>
<td>6,7,11,12,20,45,49,56,59</td>
</tr>
<tr>
<td>G95</td>
<td>G95Fn</td>
<td>Inches per revolution mode</td>
<td>6,7,11,12,45,49,56,59</td>
</tr>
<tr>
<td>G96</td>
<td>G96Sn</td>
<td>Spindle speed set as surface feet</td>
<td>6,60,62,65</td>
</tr>
<tr>
<td>G97</td>
<td>G97</td>
<td>Spindle speed set as RPM</td>
<td>6,60</td>
</tr>
<tr>
<td>M00</td>
<td>M00</td>
<td>Program stop - does not cancel active &quot;M&quot; functions</td>
<td>61</td>
</tr>
<tr>
<td>M01</td>
<td>M01</td>
<td>Optional stop</td>
<td>61</td>
</tr>
<tr>
<td>M02</td>
<td>M02</td>
<td>End program - does not cancel active&quot;M&quot; functions</td>
<td>26,62,65</td>
</tr>
<tr>
<td>M03</td>
<td>M03Sn</td>
<td>Spindle on, CW (spindle top coming)</td>
<td>16,62,65,74</td>
</tr>
<tr>
<td>M04</td>
<td>M04Sn</td>
<td>Spindle on, CCW (spindle top going)</td>
<td>62,65,74</td>
</tr>
</tbody>
</table>
## Codes Honored by the OmniTurn control
(Sort by Code)

<table>
<thead>
<tr>
<th>Code</th>
<th>Usage</th>
<th>Description</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>M05</td>
<td>M05</td>
<td>Spindle off, stop</td>
<td>62,65,74</td>
</tr>
<tr>
<td>M08</td>
<td>M08</td>
<td>Coolant on</td>
<td>16,62,65</td>
</tr>
<tr>
<td>M09</td>
<td>M09</td>
<td>Coolant off</td>
<td>62</td>
</tr>
<tr>
<td>M12</td>
<td>M12</td>
<td>Collet clamp</td>
<td>62</td>
</tr>
<tr>
<td>M13</td>
<td>M13</td>
<td>Collet unclamp</td>
<td>62</td>
</tr>
<tr>
<td>M19</td>
<td>M19</td>
<td>Spindle Positioning (optional C-Axis only)</td>
<td>62,74</td>
</tr>
<tr>
<td>M25</td>
<td>M25</td>
<td>User assigned on</td>
<td>62</td>
</tr>
<tr>
<td>M26</td>
<td>M26</td>
<td>User assigned off</td>
<td>62</td>
</tr>
<tr>
<td>M30</td>
<td>M30</td>
<td>End of program - cancels all active &quot;M&quot; functions</td>
<td>26,62,65</td>
</tr>
<tr>
<td>M31</td>
<td>M31</td>
<td>Cancels Cycle Repeat mode</td>
<td>62</td>
</tr>
<tr>
<td>M88</td>
<td>M88</td>
<td>C axis spindle orientation - Precision</td>
<td></td>
</tr>
<tr>
<td>M89</td>
<td>M89</td>
<td>Stop the spindle and lock it (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M91</td>
<td>M91</td>
<td>Wait for TB2-5 to be open circuit (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M92</td>
<td>M92</td>
<td>Wait for TB2-5 to be short to 0VDC (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M93</td>
<td>M93</td>
<td>Wait for TB2-7 to be open circuit (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M94</td>
<td>M94</td>
<td>Wait for TB2-7+ to be short to 0VDC (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M95</td>
<td>M95</td>
<td>Jump to subroutine 1 if TB2-9 is short to 0VDC (opt: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M97</td>
<td>M97InCnPn</td>
<td>Jump to subroutine, conditional (optional: PLC option only)</td>
<td>63</td>
</tr>
<tr>
<td>M98</td>
<td>M98Pn</td>
<td>Jump to subroutine</td>
<td>63</td>
</tr>
<tr>
<td>M99</td>
<td>M99</td>
<td>End subroutine</td>
<td>63</td>
</tr>
<tr>
<td>CI</td>
<td>CInnn.nn</td>
<td>Incremental spindle angle (optional: C-Axis only)</td>
<td>74</td>
</tr>
<tr>
<td>CA</td>
<td>CAnnn.nn</td>
<td>Absolute spindle angle (optional: C-Axis only)</td>
<td>74</td>
</tr>
<tr>
<td>C</td>
<td>XnZnCn</td>
<td>Automatic chamfer at intersection</td>
<td>15,16,67</td>
</tr>
<tr>
<td>D</td>
<td>Dn</td>
<td>Secondary offsets, axis correction or TNR comp value</td>
<td>68-71</td>
</tr>
<tr>
<td>F</td>
<td>Fn</td>
<td>Feedrates, dwell</td>
<td>48,56</td>
</tr>
<tr>
<td>LS</td>
<td>LSn</td>
<td>Loop start</td>
<td>72</td>
</tr>
<tr>
<td>LF</td>
<td>LF</td>
<td>Loop finish</td>
<td>73</td>
</tr>
<tr>
<td>R</td>
<td>XnZnRn</td>
<td>Automatic radius at intersection</td>
<td>15,16</td>
</tr>
<tr>
<td>S</td>
<td>Sn</td>
<td>Spindle speed selection, SFM or RPM</td>
<td>60,65,66,74</td>
</tr>
<tr>
<td>T</td>
<td>Tn</td>
<td>Tool offset call command</td>
<td>9</td>
</tr>
<tr>
<td>/</td>
<td>/</td>
<td>Block delete</td>
<td>Section 5.3</td>
</tr>
<tr>
<td>}</td>
<td>}n</td>
<td>Begin subroutine</td>
<td>63</td>
</tr>
</tbody>
</table>
## Codes Honored by the OmniTurn control  
(Sort by Description)

<table>
<thead>
<tr>
<th>Code</th>
<th>Usage</th>
<th>Description</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>G90</td>
<td>G90</td>
<td>Absolute mode selection</td>
<td>5,6,10,12,17,19-21,56,59</td>
</tr>
<tr>
<td>CA</td>
<td>CA&lt;nn.nn</td>
<td>Absolute spindle angle (optional: C—Axis only)</td>
<td>74</td>
</tr>
<tr>
<td>G02</td>
<td>G02XnZnKnKnFn</td>
<td>Arc -Clockwise</td>
<td>6,15,17-24</td>
</tr>
<tr>
<td>G02</td>
<td>G02XnZnRn</td>
<td>Arc -Clockwise</td>
<td>6,17-24</td>
</tr>
<tr>
<td>G03</td>
<td>G03XnZnKnKnF</td>
<td>Arc -Counter Clockwise</td>
<td>6,17-24</td>
</tr>
<tr>
<td>G03</td>
<td>G03XnZnRn</td>
<td>Arc -Counter Clockwise</td>
<td>6,17-24</td>
</tr>
<tr>
<td>C</td>
<td>XnZnCn</td>
<td>Automatic chamfer at intersection</td>
<td>15,16,67</td>
</tr>
<tr>
<td>R</td>
<td>XnZnRn</td>
<td>Automatic radius at intersection</td>
<td>15,16</td>
</tr>
<tr>
<td>}</td>
<td>}n</td>
<td>Begin subroutine</td>
<td>63</td>
</tr>
<tr>
<td>/</td>
<td>/</td>
<td>Block delete</td>
<td>Section 5.3</td>
</tr>
<tr>
<td>G75</td>
<td>G75InUnFnPn</td>
<td>Box Contour Roughing cycle</td>
<td>48-52,54</td>
</tr>
<tr>
<td>G74</td>
<td>G74XnZnUnInFnPn</td>
<td>Box Roughing cycle</td>
<td>45-47</td>
</tr>
<tr>
<td>M31</td>
<td>M31</td>
<td>Cancels Cycle Repeat mode</td>
<td>62</td>
</tr>
<tr>
<td>G36</td>
<td>G36</td>
<td>Cancels G35</td>
<td>6,36,74</td>
</tr>
<tr>
<td>G40</td>
<td>G40</td>
<td>Cancels Tool Nose Radius Compensation</td>
<td>16,37-43</td>
</tr>
<tr>
<td>M12</td>
<td>M12</td>
<td>Collet clamp</td>
<td>62</td>
</tr>
<tr>
<td>M13</td>
<td>M13</td>
<td>Collet unclamp</td>
<td>62</td>
</tr>
<tr>
<td>M09</td>
<td>M09</td>
<td>Coolant off</td>
<td>62</td>
</tr>
<tr>
<td>M08</td>
<td>M08</td>
<td>Coolant on</td>
<td>16,62,65</td>
</tr>
<tr>
<td>G72</td>
<td>G72</td>
<td>Diameter programming mode</td>
<td>6,10,14,16,21,22,29,38,44,46,49,59</td>
</tr>
<tr>
<td>G81</td>
<td>G81ZnF</td>
<td>Drill cycle</td>
<td>6,56</td>
</tr>
<tr>
<td>G04</td>
<td>G04F</td>
<td>Dwell</td>
<td>6,25,62</td>
</tr>
<tr>
<td>M30</td>
<td>M30</td>
<td>End of program - cancels all active M functions</td>
<td>26,62,65</td>
</tr>
<tr>
<td>M02</td>
<td>M02</td>
<td>End program - does not cancel active M functions</td>
<td>26,62,65</td>
</tr>
<tr>
<td>M99</td>
<td>M99</td>
<td>End subroutine</td>
<td>63</td>
</tr>
<tr>
<td>G35</td>
<td>G35n</td>
<td>Extra Course feeds in IPR</td>
<td>6,29,36,74</td>
</tr>
<tr>
<td>G01</td>
<td>G01F</td>
<td>Feed move</td>
<td>12,13</td>
</tr>
<tr>
<td>F</td>
<td>Fn</td>
<td>Feedrates, dwell</td>
<td>48,56</td>
</tr>
<tr>
<td>G70</td>
<td>G70</td>
<td>Inch mode</td>
<td>6,44</td>
</tr>
<tr>
<td>G94</td>
<td>G94F</td>
<td>Inches per minute mode</td>
<td>6,7,11,12,20,45,49,56,57,59</td>
</tr>
<tr>
<td>G95</td>
<td>G95F</td>
<td>Inches per revolution mode</td>
<td>6,7,11,12,45,49,56,57,59</td>
</tr>
<tr>
<td>G91</td>
<td>G91</td>
<td>Incremental mode selection</td>
<td>5,6,10,17,56,57,59,70</td>
</tr>
<tr>
<td>CI</td>
<td>CI&lt;nn.nn</td>
<td>Incremental spindle angle (optional: C-Axis only)</td>
<td>74</td>
</tr>
<tr>
<td>M98</td>
<td>M98Pn</td>
<td>Jump to subroutine</td>
<td>63</td>
</tr>
<tr>
<td>M95</td>
<td>M95</td>
<td>Jump to subroutine 1 if TB2-9 is short to 0VDC (opt: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M97</td>
<td>M97InCnPn</td>
<td>Jump to subroutine, conditional (optional: PLC option only)</td>
<td>63</td>
</tr>
</tbody>
</table>
# Codes Honored by the OmniTurn control

## (Sort by Description)

<table>
<thead>
<tr>
<th>Code</th>
<th>Usage</th>
<th>Description</th>
<th>Pages</th>
</tr>
</thead>
<tbody>
<tr>
<td>G41</td>
<td>G41</td>
<td>Left hand Tool Nose Radius Compensation</td>
<td>16,37-43</td>
</tr>
<tr>
<td>LF</td>
<td>LF</td>
<td>Loop finish</td>
<td>73</td>
</tr>
<tr>
<td>LS</td>
<td>LSn</td>
<td>Loop start</td>
<td>72</td>
</tr>
<tr>
<td>G77</td>
<td>G77Sn</td>
<td>Maximum spindle speed for constant surface feet</td>
<td>6,60</td>
</tr>
<tr>
<td>G71</td>
<td>G71</td>
<td>Metric mode</td>
<td>6,44</td>
</tr>
<tr>
<td>G76</td>
<td>G76Sn</td>
<td>Minimum spindle speed for constant surface feet</td>
<td>6,60</td>
</tr>
<tr>
<td>M01</td>
<td>M01</td>
<td>Optional stop</td>
<td>61,62</td>
</tr>
<tr>
<td>G83</td>
<td>G83ZnKnFnRnLnCn</td>
<td>Peck drill cycle</td>
<td>6,57,58</td>
</tr>
<tr>
<td>G92</td>
<td>G92XnZn</td>
<td>Preset axis position</td>
<td>36,59,74</td>
</tr>
<tr>
<td>M00</td>
<td>M00</td>
<td>Program stop - does not cancel active M functions</td>
<td>61,62</td>
</tr>
<tr>
<td>G73</td>
<td>G73</td>
<td>Radius programming mode</td>
<td>6,10,17,18,20,29,31,32,38,44,46,49,59</td>
</tr>
<tr>
<td>G00</td>
<td>G00</td>
<td>Rapid move</td>
<td>11,12</td>
</tr>
<tr>
<td>G42</td>
<td>G42</td>
<td>Right hand Tool Nose Radius Compensation</td>
<td>37-43</td>
</tr>
<tr>
<td>G78</td>
<td>G78UnFnPn</td>
<td>Rough Contour Cycle</td>
<td>51-55</td>
</tr>
<tr>
<td>D</td>
<td>Dn</td>
<td>Secondary offsets, axis correction or TNR comp value</td>
<td>68-55</td>
</tr>
<tr>
<td>M05</td>
<td>M05</td>
<td>Spindle off, stop</td>
<td>62,65,74</td>
</tr>
<tr>
<td>M04</td>
<td>M04Sn</td>
<td>Spindle on, CCW</td>
<td>62,65,74</td>
</tr>
<tr>
<td>M03</td>
<td>M03Sn</td>
<td>Spindle on, CW</td>
<td>16,62,65,74</td>
</tr>
<tr>
<td>M19</td>
<td>M19</td>
<td>Spindle Positioning (optional C-Axis only)</td>
<td>62,74</td>
</tr>
<tr>
<td>S</td>
<td>Sn</td>
<td>Spindle speed selection, SFM or RPM</td>
<td>60,65,66,74</td>
</tr>
<tr>
<td>G97</td>
<td>G97</td>
<td>Spindle speed set as RPM</td>
<td>6,60</td>
</tr>
<tr>
<td>G96</td>
<td>G96Sn</td>
<td>Spindle speed set as surface feet</td>
<td>6,60,62,65</td>
</tr>
<tr>
<td>G89</td>
<td>G89</td>
<td>Stop spindle and lock (C-Axis only)</td>
<td></td>
</tr>
<tr>
<td>M89</td>
<td>M89</td>
<td>Stop the spindle and lock it (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>G33</td>
<td>G33XnZnKnKnCnPn</td>
<td>Threading cycle</td>
<td>6,29-36</td>
</tr>
<tr>
<td>T</td>
<td>Tn</td>
<td>Tool offset call command</td>
<td>9</td>
</tr>
<tr>
<td>M26</td>
<td>M26</td>
<td>User assigned off</td>
<td>62</td>
</tr>
<tr>
<td>M25</td>
<td>M25</td>
<td>User assigned on</td>
<td>62</td>
</tr>
<tr>
<td>M91</td>
<td>M91</td>
<td>Wait for TB2-5 to be open circuit (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M92</td>
<td>M92</td>
<td>Wait for TB2-5 to be short to 0VDC (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M93</td>
<td>M93</td>
<td>Wait for TB2-7 to be open circuit (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>M94</td>
<td>M94</td>
<td>Wait for TB2-7+ to be short to 0VDC (optional: C-Axis only)</td>
<td>63</td>
</tr>
<tr>
<td>G10</td>
<td>G10XnZn</td>
<td>Work Shift</td>
<td>6,26-28,73</td>
</tr>
</tbody>
</table>
The slide has two axes of travel. 

**X**: Up and down (on GT) or towards and away from you (on Attachments). Travel up or away from you is (-) minus; down or towards you is plus (+).

**Z**: Towards or away from the spindle. Going towards the spindle is (-) minus. Away from the spindle is (+) plus.

**ABSOLUTE - G90**

In absolute (G90) the following move brings you to the same point no matter where you start:

- X: -3
- Z: -4

**INCREMENTAL - G91**

In incremental (G91) the following moves you the same amount from every start:

- X: -3
- Z: -4
• **The default** mode for X moves is G73 - radius moves, to program in diameters you must use G72 in the beginning of the program.

• **The first command** of a program must be G90 or G91 to define if the program is in absolute or incremental.

• **No blank** lines are allowed in a program, blank spaces are OK.

• **Comments** are any text or data enclosed in parentheses“( )”. Their purpose is to convey to the operator any information that the programmer might think is useful. Comments are displayed in the lower left corner of the screen. They stay on the screen till the comment is changed. As an example, you may want to use the comment to tell the operator what action to take when the spindle stops. For an example, the slide is told to go "HOME” and then the comment is displayed on the screen. Then the slide stops with the message on the screen.

- Do not put text on lines by itself. Comments must be on a line with a command!
- Keep the amount of text to a minimum, to much text can cause problems.
- A good place to put comments is on a line with a tool call ie: T1(LH turn tool)
- Use only text, do not use periods or commas or any other symbol such as i.e.:

  ! @ # $ % ^ & * " ' ? > < / \| = -.

• **Do not put** any text in a loop.

• **Commas are not allowed anywhere in the program**

• **Dimensional** data is interpreted with a resolution of .00005”. The fifth digit to the right of a decimal point must be a 0 or a 5. NOTE: when programming in diameter mode the X axis resolution is .0001”, not .00005”.

• **Decimal** point programming is used. Leading and trailing zeros need not be entered. For example “X1” is interpreted as 1 inch. X1 = X1.00000

• **G and M codes** must be programmed as two digit codes. ”G2” is not a legal code and it will be ignored. Also be sure to use the zero and not the letter O as part of the G and M codes.

• **Model commands**: These are commands that remain active until canceled:
  G90, G91 -G94, G95 -G70,G71 -G76, G77, G96,G97 -G72, G73
  All "M" codes, G35, G36 (GT-75 only)-G10

• **One shot commands**: These act only on the statement they are programmed in:
  G02, G03 -G04 -G33, G34 -G81, G83 -G92

• **Conflicting commands**:
  There can be only one "M" command per line of code
  There can be only one “one shot” G code per line of code
  There can be more than one nonconflicting modal G code per line
  The S and F commands can be with any other command

• **N sequence** are not allowed. They can cause intermittent problems.
**G10 Work shift**

Work shift is used to offset a program from the original starting point. Typical applications are:
- Machining multiple parts off a single shootout of a bar.
- Shifting a program away from the spindle the first time it is run

### G10XnZn

**G10** will shift the reference of the slide incrementally. If G10 is put into a loop the program will shift each time the command is used. The shift will take place on the next tool call. If you put the shift after a tool call the effect will take place the next time through the loop.

**Note:** *The shift is executed on tool calls! Use the G10 command before tool calls otherwise there will be no effect.*

The shift will be canceled with these commands: T0 - M30 - M02

The command must have a value for both X and Z.

#### Example of shifting a program for test running

In the following example we show using the G10 work shift for running a program the first time way from the work to make sure that the program looks like it will run OK. In this example you would set the tools to make the part. After the program is run a few inches away with the work shift the G10 command would be removed from the program. Then the program would be run to make a part.

```
G90G72G94F300
M03S2000
G04F2
G10X0Z3
T1(LH TURN TOOL)
X0Z.2
Z0G95F.01
X.1Z-.05F.002
Z-.15
X.2
G94F300Z1
T2(PART OFF TOOL)
```

**Shifts program 3” to the right**
**Tool 1** (left hand turn tool)  
**Tool 8** (work stop)

**Process:**
Use the front of the tool as a work stop
- T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown
G90G72G94F300 (PART-1)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T1(LH TURN TOOL)
X.6Z2
Z0
G95F.003X-.015
X.2F.01
Z-.25F.003
X.375
Z-.5
X.6
G94F300Z2
M30
It is possible to automatically generate a chamfer or radius between two connecting linear moves. Just program the lines to the theoretical intersection point of the two move and put a C or R with the absolute amount of the radius or chamfer needed.

**Format**

\[XnZnRn - ZnRn - XnRn\]

\[XnZnCn - ZnCn - XnCn\]

- **XnZn**: The linear move leading to the intersection point of two lines
- **Rn**: The n is the absolute value of the radius used to blend the two lines
- **Cn**: The n is the absolute value of the chamfer used to blend the two lines

**RULES**

The moves that are connected by the auto chamfer or radius **must be linear moves**. The C or R command will not work with blending arcs or arcs and lines. If you want to blend these use G02 and G03.

The moves do not have to be at right angles

A chamfer created is set back equally from the intersection point of the two lines.

A radius created is made tangential to the two intersecting lines. The direction (CW or CCW) of the radius is determined automatically by the OmniTurn. It looks ahead to the next move.

The n value must be the absolute (+) value

**Running programs using C or R**

When you use the automatic corner radius or chamfer commands the OmniTurn creates a number of moves to generate what you want. If you look at the command line while you run a program you will notice lines of code that you did write. In the single block mode you can see arc (G02 or G03) commands. This is normal. When you leave the editor the OmniTurn automatically recreates the new moves. The program is also recreated whenever you change the
**Topic:** Corner chamfer and corner radius

**Tool 1** (left hand turn tool)  
**Tool 8** (work stop)  

**Process:**
- Use the front of the tool as a work stop  
  T8, stop motion, pull part to stop, close collet  
- Use T1 (LH turn tool) to rough the part as shown adding the chamfer and radius moves

**Sample 2**
G90 G72 G94 F300 (PART-2)
G10 X0 Z2
T8 (WORK STOP)
X0 Z2
Z.2
F50 Z.01
M00 (PULL PART TO STOP)
M12
Z2 F300
M03 S1500
M08
T1 (LH TURN TOOL)
X.6 Z2
Z0
G95 F.003 X-.015
X.2 C.03 F.01
Z-.25 F.003
X.375 R.04
Z-.5
X.6
G94 F300 Z2
M30


G33 Threading

The format is: G33XnZnInKnAnCnP0

X The X axis location (as a radius) of the final pass of the cycle in G72 mode this is the final pass as a diameter.
Z The Z axis location of the end of the thread
I The starting incremental amount of material to be removed after the first pass.

This is to be defined as the diameter removal in diameter mode

K The lead of the thread, amount per revolution,.2” max. For larger see G35
A Used for tapered threading, it specifies the amount the X axis will move over the length of a tapered thread
C Causes the infeed to be at an angle, the default is 29°
P Used when you want the tool to keep traveling forward while it pulls out of the work. This will leave no undercut
O Including the letter O makes a single pass at the finished depth

Notes:

Diameter or radius mode
The use of the threading cycles is the same for either diameter (G72) or radius (G73) mode. Only the the values of X will be different. The values will correspond to the mode.

Starting position in Z
The tool in most cases will be started at least 1” away from the start of the thread to allow the slide to get up to speed before it makes contact with the material. This number will vary depending on the spindle speed and the pitch of the thread. The courser the thread and faster the spindle speed, the farther away you will need to start. Under worst case conditions the slide can get up to full threading speed in about 1/2 revolution of the ball screw. In most cases this does not matter, however if you are threading from an undercut and the tool has very little room to ramp up to speed, this is very important. You will have to slow the spindle down until the thread gauge goes on.

Starting position in X
The tool should be positioned to take the first pass. The farther away you start the tool, the more passes will be needed. In production runs it pays to experiment a little for the best results and speed.

Depth of each pass: I
The control will start with removing the amount given as I. Then the control will automatically reduce the depth of the cut as the tool gets deeper. This is a fixed procedure that cannot be changed, it keeps the amount of material removed constant. Start the tool so that it takes a full cut on the first pass.
G33 Threading

Retraction position between passes:
The tool will back away from the starting position plus 3 times the amount of I. Even as the tool gets deeper into the material it will always retract to the same point.

Pullout position in Z when using P option:
The tool will start to pull out at the location given in Z. It will travel beyond Z the same amount as it has to travel in X to reach the retraction position.

Angle infeed C option
If C is included in the G33 command the tool will feed in at an angle. This defaults to 29°. The maximum angle is 30° (based on standard 60° tool geometry) the min is 0°. If you wanted the tool to angle in at 27°, add C27 to the threading cycle command.

The single pass option O can be used for a cleanup pass:
When a single pass is needed write the same threading pass as used for cutting the thread. Just add a O to the command. Be sure to start the thread at the same point and at the same spindle speed. This option can be used with all variations of the threading command.

End of cycle position:
At the end of the threading cycle the tool will return to the starting point.
<table>
<thead>
<tr>
<th>Machine Screw Size</th>
<th>Threads Per Inch</th>
<th>Minor Dia</th>
<th>Tap Drills</th>
<th>Clearance Hole Drills</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>75% Thread</td>
<td>50% Thread</td>
</tr>
<tr>
<td>0</td>
<td>80</td>
<td>.0600</td>
<td>.0447</td>
<td>3/64 .0469</td>
</tr>
<tr>
<td>1</td>
<td>72</td>
<td>.0538</td>
<td>53 .0595</td>
<td>53 .0595</td>
</tr>
<tr>
<td>2</td>
<td>56</td>
<td>.0560</td>
<td>55 .0700</td>
<td>49 .0730</td>
</tr>
<tr>
<td>3</td>
<td>48</td>
<td>.0568</td>
<td>50 .0700</td>
<td>44 .0860</td>
</tr>
<tr>
<td>4</td>
<td>40</td>
<td>.0571</td>
<td>45 .0820</td>
<td>43 .0890</td>
</tr>
<tr>
<td>5</td>
<td>32</td>
<td>.0577</td>
<td>38 .1015</td>
<td>7/64 .1094</td>
</tr>
<tr>
<td>6</td>
<td>24</td>
<td>.0597</td>
<td>36 .1065</td>
<td>32 .1160</td>
</tr>
<tr>
<td>10</td>
<td>12</td>
<td>.0722</td>
<td>11 .3200</td>
<td>8 .3480</td>
</tr>
<tr>
<td>5/16</td>
<td>10</td>
<td>.0727</td>
<td>9/32 .2812</td>
<td>19/32 .2812</td>
</tr>
<tr>
<td>3/8</td>
<td>8</td>
<td>.0722</td>
<td>9/32 .2812</td>
<td>19/32 .2812</td>
</tr>
<tr>
<td>7/16</td>
<td>1/4</td>
<td>.0727</td>
<td>7/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>21/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>13/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>7/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>19/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>31/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>43/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>55/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>67/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td>1/2</td>
<td></td>
<td></td>
<td>67/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>59/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>41/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>23/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>11/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>7/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1/32 .2188</td>
<td>5/32 .2188</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1/32 .2188</td>
<td>5/32 .2188</td>
</tr>
</tbody>
</table>
The pitch \( P \) is the distance between thread peaks. For UTS threads, which are single-start threads, it is equal to the lead, the axial distance that the screw advances during a 360° rotation. UTS threads do not usually use the pitch parameter; instead a parameter known as threads per inch (TPI) is used, which is the reciprocal of the pitch.

\[
P = \frac{1}{\text{TPI}} \quad 0.05 = \frac{1}{20} \quad 0.03125 = \frac{1}{32}
\]

The outermost 0.125 and the innermost 0.25 of the height \( H \) of the V-shape are cut off from the profile.

\[
H = 0.866 \times P
\]

**Major Diameter = Screw # \times 0.013'' + 0.060''.**

For example, a number 10 calculates as: \#10 \times 0.013'' + 0.060'' = 0.190'' Major Diameter.

The formula for number sizes smaller than size \#0 is given by Major Diameter = 0.060'' - Zero size \times 0.013'', with the zero size being the number of zeroes after the first. So a \#00 screw is .047'' dia, \#000 is .034'' dia, etc.

**Classes 1A, 2A, 3A apply to external threads; Classes 1B, 2B, 3B apply to internal threads.**

**Class 1 threads are loosely fitting threads intended for ease of assembly or use in a dirty environment.**

**Class 2 threads are the most common. They are designed to maximize strength considering typical machine shop capability and machine practice.**

**Class 3 threads are used for closer tolerances.**
Sample 3 Process:
Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown
Leave .01 to clean up on a finish pass
Use T2 (LH turn tool - finish)
perform a cleanup pass with the chamfers and radius

Sample 4 Process:
Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T1 (LH turn tool) to rough the part as shown
Leave .01 to clean up on a finish pass
Use T2 (LH turn tool - finish)
perform a cleanup pass with the chamfers and radius

Use T3 (LH thread tool) to thread OD
Major diameter .375, minor diameter .298

Topic: Repeat basic programming example
Topic: Single point threading
Topic: Tools on the (-) side of the spindle

Part #3 and Part #4

Richlin Machinery Inc
OmniTurn Training 631 694-9400
G90 G72 G94 F300 (PART-3)  
G10 X0 Z2  
T8 (WORK STOP)  
X0 Z2  
Z.2  
F50 Z.01  
M00 (PULL PART TO STOP)  
M12  
Z2 F300  
M03 S1500  
M08  
T1 (LH TURN TOOL - rough)  
X.6 Z2  
Z0  
G95 F.003 X-.015  
X.22 F.01  
Z-.24 F.003  
X.395  
Z-.74  
X.6  
G94 F300 Z2  
T2 (LH TURN TOOL - finish)  
X.2 Z2  
Z.2  
G95 F.003  
Z-.25 F.003  
X.375 C.05  
Z-.75  
X.6 C.075  
Z-.95  
G94 F300 Z2  
M30  

G90 G72 G94 F300 (PART-4)  
G10 X0 Z2  
T8 (WORK STOP)  
X0 Z2  
Z.2  
F50 Z.01  
M00 (PULL PART TO STOP)  
M12  
Z2 F300  
M03 S1500  
M08  
T1 (LH TURN TOOL - rough)  
X.6 Z2  
Z0  
G95 F.003 X-.015  
X.22 F.01  
Z-.24 F.003  
X.395  
Z-.74  
X.6  
G94 F300 Z2  
T2 (LH TURN TOOL - finish)  
X.2 Z2  
Z.2  
G95 F.003  
Z-.25 F.003  
X.375 C.05  
Z-.75  
X.6 C.075  
Z-.95  
G94 F300 Z2  
T3 (THREADING TOOL)  
X-.365 Z2  
Z.2  
G95  
G33 X-.298 Z-.65 K.0625 I.03 C  
M30
Secondary Offsets

What are secondary offsets?

Secondary offsets are corrections that you can put into your program that the operator can adjust when running the program without having to go into the program to edit it. Once the program has been written with the secondary offsets incorporated, these corrections are made by pressing F9 while in the Automatic mode and inputting the amounts. This procedure is very similar to adjusting tool offsets. The big difference with secondary offsets is that there can be more than one correction to a tool.

There are a number of ways that they can be used. Below are a few examples of typical applications.

In all three cases it would be very advantageous to be able to have the operator make corrections to the parts that entail more than just moving the tool by changing the tool offset (T). If you made a change to the tool offset, the overall size of the part would change in each of the above examples.

If, however, you had a taper in the long thin part (lower left sample) and had to correct it to get the part straight, offset changes would not help. The secondary offset allows you to add or subtract a little, to any move, at any point in the program. So the correction of the taper can easily be taken care of.

NOTE: Clear secondary offsets before using them!

Before you run a program that uses secondary offsets be sure that you have reset the secondary offsets that you are using to zero! This can be done by pressing C when asked to make a correction to the offset table (See F9 in the Automatic section)
**Topic:** Repeat Single point threading

**Topic:** Grooving with secondary offsets

---

**Process:**

- Use the front of the tool as a work stop
  - T8, stop motion, pull part to stop, close collet
- Use T1 (LH turn tool) to rough the part as shown
  - Leave .01 to clean up on a finish pass
- Use T2 (LH turn tool - finish)
  - perform a cleanup pass with the chamfers
- Use T4 (.078 wide groove tool)
  - Undercut behind the thread.
  - Use secondary offsets to adjust size.
- Use T3 (LH thread tool) to thread OD
  - Major diameter .190, minor diameter .151
- Use T5 to cut off

---

**Tool List:**

- T5 .093 Cutoff tool
- T4 .078 groove tool
- T2 (LH turn tool - Finish)
- T3 (LH thread tool)
- Tool 1 (left hand turn tool)
- Tool 8 (work stop)
G90G72G94F300 (PART-5) X.15
G10X0Z2 G04F.3
T8(WORK STOP) X-.25
X0Z2 G94F300Z2
Z.2 T3 (THREADING TOOL)
F50Z.01 X-.18Z2
M00 (PULL PART TO STOP) Z.2
M12 G95
Z2F300 G33X-.15Z-.38K.03125I.025C
M03S1500 G94F300Z2
M08 T5 (CUT OFF TOOL)
T1(LH TURN TOOL - rough) X-.6Z2
X.6Z2 Z-1.1
Z0 G95F.003X.1
G95f.003X-.015 G94F300X-.6
X.3F.01 M05
Z-.44F.003 M08
X.6F94F300 Z2
Z.01 M30
X.1
Z0G95F.003
X.21
Z-.44
X.395
Z-.49
X.52Z-.615
X.6
G94F300Z2
T2 (LH FINISH TOOL) X.3Z2
X.3Z2
Z0
G95F.003X.15
X.19C.025
X.19C.025
Z-.45
X.375C.02
Z-.5
X.5Z-.625
X.6
Z2G94F200
T4 (078 GROOVE TOOL) X-.5Z2
X-.5Z2
Z-.45
G95F.002X-.15
G04F.3
X-.25
Z-.625
Z-.625
G94F300X-.6
T5 (CUT OFF TOOL)
X-.6Z2
Z-.44
X.375C.02
Z-.5
X.5Z-.625
X.6
Z2G94F200
T4 (078 GROOVE TOOL) X-.5Z2
X-.5Z2
Z-.45
G95F.002X-.15
G04F.3
X-.25
Z-.625
Z-.625
G74 is a box roughing cycle where a rectangular area of material is removed in many passes.

G74XnZnInUnFn

X and Z is the corner of the box area to be cleared out
In is the maximum amount to be roughed per pass, defined as the depth of cut per side
Un amount of material to be left by the cycle for a finish pass in X only.
(depth of cut, as a radius)
Fn is the feedrate

The box cycle starts at the current position, then makes cutting passes parallel to the Z axis at a cutting depth no greater than the I ending at X, Z. At the end of the cycle the tool is returned to the start point.

If you want to leave material for a finish pass the X and Z values must be offset for this.

The feedrate is IPM (G94) or IPR (G95), depending on the mode when the cycle is started.

The X, Z coordinate may be absolute or incremental, based on the current mode of the control.

The return passes are at a fixed clearance distance (.02") from the last cutting pass.
G83 Peck Drill Cycle

G83 is a one shot command. It is used to peck drill to a specific distance in Z and then rapid back to the starting point. The format is:

G83 Zn Kn Fn Rn Ln Cn

In G90, absolute mode: Z specifies the end of the point of the hole from the part Zero.
In G91, incremental mode: Z specifies the distance the tool will travel from the starting point.

**Start location:** Position the drill where you want the first drill peck to start. After the first peck the drill will rapid out to the R location, and then back to where it started less the C value.

K specifies the depth of cut per peck.
F is the drilling feedrate in inches per rev or minute depending on whether you are in G94 or G95.
R is the retraction plane, the tool will rapid back to this location at the end of each peck. **Default is the starting point of the cycle**
L is the rapid travel feedrate for the retraction move, noted in IPM. **Default is 200ipm**
C is the clearance distance left when the drill returns to the cut. **Default is .02”**
**Process:**

Use the front of the tool as a work stop

- T8, stop motion, pull part to stop, close collet

Use T1 (left hand turn tool) turn the part as shown

- Finish to size with chamfers

Use T3 (LH thread tool) to thread OD

- Major diameter .4375, minor diameter .348

Use T1 to clean tops of threads

Use T3 (LH thread tool) to thread OD, one pass to clean up.

Use T6 (spot drill) to spot and chamfer face

Use T7 (#7 drill) use G83 peck drill to drill through part

Use T9 (1/4-20 tap) to tap at least 5/8" deep

Use T5 to cut off

**Variation**

Cut the 7/16 - 20 thread with three starts instead of 1 start
G90G72G94F300 (PART-6) T9 (1-4 20 TAP)
G10X0Z2 X0Z2
T8(WORK STOP) Z.2
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP) M04
M00
M12
Z2F300
M12
Z.2
M03S1500
M03S1500
T1(LH TURN TOOL - rough) G94F300Z2
X.6Z2 X-.6Z2
Z0 Z-1.25
G95f.003X-.015 G94F300X-.6
X.4375C.05 M05
Z-.500
X.6Z-.5825 M08
G94F300Z2
T3 (THREAD TOOL) Z2
T1
X.427Z2
X.4375Z2
G95F.004Z-.5
X.6
G94F300Z2
T3 (THREAD TOOL) M30
X.427Z2
X.4375Z2
G33X.336 Z-.4I.03K.05C G94F300Z2
G33X.336 Z-.4I.03K.05CO
G94F300Z2
T6 (SPOT DRILL) G94F300Z2
X0Z2 T7 (NUMBER 7 DRILL)
Z.2
X0Z2
Z.2
G95F.005Z-.2
G94F300Z2
T7 (NUMBER 7 DRILL)
X0Z2
Z.1
G95 G83Z-1.35K.3R.5L300
G94F300Z2S500
Process:

Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet
Use T2 (LH turn tool) to turn the part as shown
Finish to size all that can be reached
Use T6 (spot drill) to spot and chamfer face
Use T7 (1/8" drill) use G83 peck drill to drill through part
Use T4 to finish grooves behind barb (use secondary offsets) and in front of end wall.
Use T5 to cut off
G90G72G94F300 (PART-7)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2F300
M03S1500
M08
T2(LH TURN TOOL)
X.6Z2
Z0
G95F.003X-.015
X.312R.04
Z-.125
X.392Z-.275
Z-.4
X.312
G04F.3
X.392Z-.55
Z-.675
X.312
G04F.3
X.392Z-.825
Z-1
X.312
G04F.3
X.51C.025
Z-1.1
G94F300Z2
T6 (SPOT DRILL)
X0Z2
Z.2
G95F.005Z-.2
G94F300Z2
T7 (125 DRILL)
X0Z2
Z.1
G95
G83Z-1.35K.2R.5L300
G94F300Z2
T4 (078 WIDE GROOVE TOOL)
X.4Z2
Z-.275 D1 (SET TO -078)
Topic: Secondary offsets for correcting diameters and tapers

Process:
Use the front of the tool as a work stop
T8, stop motion, pull part to stop, close collet

Use T2 (LH turn tool) to turn the part as shown
Use secondary offsets to adjust for exact sizes and taper.
G90G72G94F300 (PART-8)
G10X0Z2
T8(WORK STOP)
X0Z2
Z.2
F50Z.01
M00 (PULL PART TO STOP)
M12
Z2G300
M03S1500
M08
T1(LH TURN TOOL)
X.6Z2
Z0
G95f.003X-.015
X.2
Z-.2D1
X.4Z-.6D2
Z-1
X.6
G94F300Z2
M30
**Topic:** Boring  
**Topic:** Thread ID  
**Topic:** Knurl

---

**Process:**
Load blank into collet. The left side of the part is done.

- The 1.46 diameter is turned to the start of the knurl.
- T1 (.5 DRILL) Drill to .54 deep
- T2 (RH bore tool) Rough and finish bore
- T3 (ID under cut) Clear out under cut
- T4 (ID thread) Thread minor diameter .938", Major 1.000"
- T5 (LH turn tool) Face and turn the OD
- T7 (knurl) Knurl OD

---

**Sample 9**
G90 G72 G94 F300 (PART-9)
G10 X0 Z2
M08
M03 S1200
T1 (500 DRILL)
X0 Z2
Z.2
G95 F.002 Z-.54
G94 F300 Z2
T2 (BORE TOOL)
X.6 Z2
Z.1
G95 F.003 Z-.54
X.5
G94 F300 Z.1
X.8
G95 F.003 Z-.54
X.6
G94 F300 Z.1
X1.19
G95 F.003 X.96 Z-.03
Z-.546
X-.01
G94 F300 Z2
T3 (GROOVE TOOL)
X0.9 Z2
Z-.3
G95 F.003 Z.546
X1.060 F.002
G04 F.5
X.9
Z-.444 F.01 D1
X1.060 F.002
G04 F.5
X.9
Z-.394
X1.060 Z-.444
G94 F300 X.8
Z2
T4 (ID THREAD TOOL)
X.97 Z2
Z.2
G95
G33 X1 Z-.444 I.025 K.05 C
G94 F300 Z2
T5 (LH TURN TOOL)
Spindle Positioning - G3

Spindle positioning system specifications

Spindle power: 5HP  
Voltage: 200 - 230V 3 phase or single phase (contact the factory for wiring)  
Resolution: .02 °  
Max Speed: 4000 rpm  
Min Speed: .004 rpm

**M19**  
Programmed by itself causes the spindle to position via the shortest route to 0°. After the command is executed the spindle is locked in position. To release the spindle use **M05**. This is a one shot command, it’s modal.

**CI(-)nnn.nn**  
This makes the spindle move an incremental amount of degrees.

**CA(-)nnn.nn**  
This makes the spindle move to an absolute location of degrees.

**Snnn.nn**  
The "S" number if programmed along with a M19 indicates the spindle speed in RPM. With no sign the spindle will rotate in the M03 direction. The "-" sign will cause the spindle to rotate in the M04 direction.

**G35/G36 -**  
*(see notes in G33 section on use and formats)*

**Extra course long-lead ipr feeds.** The G35 allows long lead ipr feeds. G35 sets max feedrates to 1 ipr. G36 cancels G35. When G35 is active the system resolution drops to .00025”. G35 may be activated any time. There is also a G35F2 mode for 2”/rev feeds. Please refer to the threading section for details on format and use.

**After G35 and G36 there must be a G92 command**

**NOTE:** Both axis’s must be returned to the position they were in when the G35 was invoked before G36 is programmed. G35 must be canceled before a tool change!

**Notes on use:**

• Before a spindle positioning in absolute command can be executed there must be a M19 command to orient the spindle.

• Be sure that you calculate the amount of C needed for a coordinated C and Z move. In the following example there is not enough C given to complete the Z move, the slide will then hang up. A solution would be to increase C to 432° to complete the Z move.

  Formula to find number of degrees needed = the distance travel \[ \text{IPR} \times 360 \]

Z0  
G35  
G92X020  
G95F25  
C360Z-.3S5  
G94F5020  
G36  
G92X020

• Currently there is no feedback from the spindle drive that a move to a location has been completed. When you rotate the spindle into position you will have to put a dwell after a rotation command to allow it time to complete the move.
Spindle positioning system specifications

Spindle power: 5HP
Voltage: 200 -230V 3 phase
Resolution: .02 °
Max Speed: 4000 rpm
Min Speed: .004 rpm

M19 Programmed by itself causes the spindle to position via the shortest route to 0°. After the command is executed the spindle is locked in position. To release the spindle use M05. This is a one shot command, it's modal. M88 is for high precision orientation.

CI(-)nnn.nn This makes the spindle move an incremental amount of degrees.

CA(-)nnn.nn This makes the spindle move to an absolute location of degrees.

Snnn.nn The "S" number if programmed along with a M19 indicates the spindle speed in SFP at 1" diameter. With no sign the spindle will rotate in the M03 direction. The "-" sign will cause the spindle to rotate in the M04 direction.

As an example, if you want 5" per minute feed rate for a milling cut at 1" diameter

\[ S = \frac{1.57 \times \text{feedrate}}{\text{diameter of cut}} \quad S = \frac{1.57 \times 5}{1} \quad S = 7.9 \]

As an example, if you want 5" per minute feed rate for a milling cut at .5" diameter

\[ S = \frac{1.57 \times \text{feedrate}}{\text{diameter of cut}} \quad S = \frac{1.57 \times 5}{.5} \quad S = 15.8 \]

Notes on use:

• Before a spindle positioning in absolute command can be executed there must be a M19 command to orient the spindle.
**Process:**
Load blank into collet.
- T1 .475 DRILL
- T2 .61 DRILL
- T5 TURN OD
- T3 .125 LIVE TOOL
- T4 BORE TOOL

**Part #10**

**Topic:** Live tool cross drilling
**Topic:** C axis, M19, CA, CI
**Topic:** Looping
G90G72G94300 (PART-10) Z2
G10X0Z2 M30
M03S1500
T1 (DRILL 475) X0Z2
Z.2
G95F.003
G83Z-1.65K.5R.5L300 T2 (DRILL 61)
G94F300Z2 X0Z2
Z.2
G95F.003
G94F300Z2
T5 (GROOVE TOOL) T3 (125 LIVE TOOL)
X1.1Z2 Z-687
Z-.875 M19
G95F.003Z-.376 M17
G94F300Z2 LS4
T5 (GROOVE TOOL)
X1.1F.01 X.3F3
Z-1.016 CI60
X.818Z-.875F.003 X1.1F300
Z-.775 M18
X.818F.003 Z2
X1.1F.01 M30
Z-.684D1
X.818Z-.5
G04F.3
G94F300X1.2
Z2
T4 (BORE TOOL) X.625Z2
Z.1
G95F.003Z-.376 LS4
X.5Z-.438
Z-1.6
X.48
G94F300Z2
T3 (125 LIVE TOOL) CI60
X1.1Z2 LF
Z-.687 XI60
M19
M17
M30
X.3F3
X1.1F300
M18
Topic: Arc Statement
Topic: Live tool slotting

Process:
Load blank into collet.
T1 Turn face radius, chamfer and major diameter
T9 Live tool - create slot
Using Tool Nose Radius Compensation
Solving partial arcs with G02 and G03

t1 (lh turn tool 015 tnr)
x.8 z1
z0
g95f.003x1.2
z.1

\textbf{g42}

z0 x1.17 \textbf{D1}
x.98375
g03 x.93953 z-.00972 r.03
g02 x.79746 z-.0762 r.638
g03 x.775 z-.0996 r.03
z-2
x.7

\textbf{g40}
z1g00

\[
D1 = x \cdot 0.015 \ z \cdot -0.015 \ r \cdot 0.015
\]
Tool nose radius compensation

Notes on use:
When radii or angles are programmed and you need a very accurate reproduction you have to take into account the size of the tool nose radius. Otherwise there will not enough material removed in the area of the radius or angle. The tool nose radius compensation is very helpful when programming any moves that are not parallel to the axes. With the G41 and G42 codes you can compensate for the size of the tool nose radius without any complicated computations. The amount of compensation can be changed by correcting a radius value stored with the secondary tool offset table. The direction of the offset correction is also done with the secondary tool offset values of X and Z.

Format
Right Compensation G42
Left Compensation G41
Cancel Compensation G40

Compensation Value Location XnnnZnnnDn

G41 or 42 specifies the type of compensation to be turned on
G40 turns the compensation off
Dn is the secondary offset that stores the value of the tool nose radius value to be used. This value is taken from the R register in that offset table. This also can be used to shift the tool path to fit a previously completed path.

Sequence
G42 Turn compensation on
XnnnZnnnDn Move with secondary offset radius value used to turn on comp
......
......
Xnnn Move used to turn off compensation
G40 Turn compensation off

Rules
• The compensation must be turned on before a linear move, the command must be on a line by itself.
• The secondary offset (Dn) must be with a linear move on the line after either the G41 or G42
• The compensation must be turned off after a linear move, the command must be on a line by itself. To turn the compensation off put the G40 on the line after you make the move to clear the work. The turning off of the compensation will be done on this move. Be sure the move off the work is larger than the size of the tool nose radius being compensated.
• Compensation must be turned off before it can be turned on again. If you have to go from right to left compensation you must have a move off the part to turn one off before the other is turned on.

• The compensation can be used on all types of moves except:
  - Drilling
  - Threading

• The value of the R in the secondary offsets must be (+). It is the incremental value of the tool nose radius. ie: a .007” radius tool has a compensation value of .007

• Tool changes automatically turn off compensation

• Tool nose radius compensation can be used in either Radius (G73) or Diameter (G72) modes

• When the compensation is turned on or off the tool must be off the part by no less than the size of the radius being compensated. The clearance move off the part must be to a distance off the part by atleast twice the TNR value.

• The compensation looks ahead at the next move to help eliminate over travel into corners

• When using the TNR compensation the tool path gets shifted off the finished size. This does not matter if the tool being used to take a finish pass is different than the roughing tool. The tool is shifted in the setup to give a correct finished size. If the same tool is used to do the rough and finish pass then the tool path must be shifted to correct for the error created with the TNR comp. Next is a sample of what would happen without correction for size.

**Right or left?:**
The right and left compensations are based on the type of move you are performing, not the type of cutter. The type of compensation is described by looking at what side of the cut the center of the tool nose radius is. Imagine that you are sitting at the center of the tool nose radius, looking in the direction of the cut. The type of compensation that you have to apply is determined by whether the center of the tool is on the right or left of the material. In the following example you would want to apply G42 - right handed compensation:
TOOL NOSE RADIUS COMPENSATION  G41, G42, G40

In the following examples we use the same cutters, and the part geometry is the same. The only difference is the direction of the tool path:

![Diagrams showing right and left handed compensation](image)

**Shifting the TNR compensation**

The direction of the correction will depend on the direction of the tool path and desired TNR compensation.

![Diagram showing direction of corrections](image)

Notice the following table for the direction of the corrections to be added to the same secondary offset as the tool nose radius.
If you enter the incorrect sign for the secondary offset value the result will be the part will not be
the right size.

Setting the TNR value:
The value used for the compensation of the tool nose radius is stored in the secondary offset table. To enter a value in the table press F9 - SECCMP from the automatic page. This will bring up the secondary offset table:

First: Select a secondary offset number
Next: Enter the tool path correction. Enter the value with the correct sign. Refer to the previous table. If the value should be - use the sign. If the value is + just enter the value.

X value: Enter twice the value of the tool tip radius. i.e. if TNR=.007 enter .014
Z value: Enter the value of the tool tip radius.
Then: Enter the value of tool nose compensation, IE .007 and then press ESC

Changing a compensation value:
When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in the offset and you want to change it to .008, just enter the new value. You do not have to clear the register first. If you want to correct a value slightly, you must enter in the final value needed. ie: if you have .007 and want to increase it by .001 you must enter .008. \textit{Do not enter -.001}
**Worked examples**

In the first example a turning tool is used in one direction.

G90G94F300  
M03S2000  
T1  (LH turn tool with .015 tnr)  
X0Z1  
Z.05  
G95F.003  
G41  
X0Z0D1  
X.22  
X.25Z-.03  
Z-.3  
X.27  
G94F300Z2  
G40  
M30

**Turn on left hand tool nose radius compensation**

**Use the radius value found in secondary offset #1**

**Turn off the TNR compensation on the Z2 move**

Before running the program the setup person must make sure that there are the correct values in the secondary offset D1.

\[ X = -.01500 \quad Z = -.01500 \quad R = .01500 \]

If the values are not correct, then clear them and enter new ones. Remember when entering the X value you must enter twice the TNR value, i.e. -.03 for the above example.
The next example shows a tool being used with both right and left compensation. First the tool will be used to face onto the part. In this move the compensation is G42 (right). After the face move is done, the tool has to come off the material so it can come back on with G41 (left) compensation.

G90G94F300
M03S2000
T1
X0Z1
X.28
G42
X.28Z-.595D1
G95F.003X.225Z.005
X0
G94F200Z.1
G40
G95F.003
G41
X0Z0D2
X.22
X.25Z-.03
Z-.3
X.3
G94F300Z2
G40
M30

Turn on right hand tool nose radius compensation
Use the radius value found in secondary offset #1

Turn off the TNR compensation on the Z.1 move

Turn on left hand tool nose radius compensation
Use the radius value found in secondary offset #2

Turn off the TNR compensation on the Z2 move

The values for the secondary offsets for TNR compensation should be:
D1 X = -.015 Z = -.015 R = .015
D2 X = -.015 Z = -.015 R = .015
Notice in this example that the values of D1 and D2 are the same. You could use the same secondary offset (ie D1) for both the G41 and G42 offsets.

Running a program that used Tool Nose Radius Compensation
When you write a program with TNR compensation there is another program that is created automatically that has all of the moves that make up the compensated program. When you run the program you will see extra moves in your program that you did not write. This is normal. If you run a program in single block mode you will see the newly created moves. You will not see the G40, G41, or G42 code in the executed program. There will be moves that get the tool ready and in place for the rest of the compensated moves. The values of the X and Z’s will be changed to compensate for the TNR.

When you go to the editor you will be working on your original program. After you leave the editor the program will automatically be rewritten and stored so it is ready to run. Also every time you leave the secondary offset table the program will be rewritten to compensate for the new tool nose radius values given.
**Topic: Live tool - milling a rotary slot**

**Process:**
Load blank into collet.
T19 Live tool - create slot
Constant Surface feet spindle speeds - G96, G97, G77, G76

To use the following codes the OmniTurn must be equipped with a spindle control package. There are two types of spindle speed control modes that the OmniTurn control can use:

**Spindle speed in RPM -(G97).** In this mode the S value will set the spindle speed in turns per minute, "RPM". The speed will stay at this value until it is changed. If the spindle is turned off and then back on in the program the speed will still be the previously set value.

This mode is good for drilling and fixed spindle speed operations.

**Constant Surface Feet -G96.** In this mode the S value will set the amount of surface feet the tool will see. The speed of the material passing the tool will stay constant, no matter what the tool's distance from center is. As the tool gets closer to center the speed of the spindle will increase. Many tool and material suppliers give suggested feeds and speeds in terms of surface feet. This mode is good for turning and facing operations. *(See notes on use below)*

**Minimum spindle speed -G76:** Sets the minimum spindle speed, G76Sn.

**Maximum spindle speed -G77:** Sets the Maximum spindle speed, G77Sn

**Notes:** The default spindle speed mode is G97, RPM mode.

• M03, M04, and M05 operate the same for both modes of spindle control

<table>
<thead>
<tr>
<th>Important Note</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Notes on use:</strong></td>
</tr>
<tr>
<td>The constant surface speed control is not intended to be turned on at the beginning of the program and then left on. If you do this the spindle speeds will vary greatly every time the machine moves! This will create excessive ware on the spindle motor and drive. Turn the constant surface feet mode on just after the tool has been positioned for the cut. Estimate the spindle speed that the CSF mode will start at and have the spindle turned on before you make the positioning moves. After the cut has been finished turn the constant surface feet mode off. Then use RPM commands. DO NOT LEAVE THE G96 ACTIVE FOR TOOL CHANGES.</td>
</tr>
</tbody>
</table>

Simple formulas to convert these values are:

\[
\text{SFM} = \frac{(\text{RPM}) (2) (3.14) (\text{distance from center})}{12} \\
\text{RPM} = \frac{\text{SFM} x 12}{2 (3.14) (\text{distance from center})}
\]

Sample program showing constant surface feet:

```
G90G94F300
M03S1500
T1(LH TURN TOOL .008 RADIUS)
X.25Z.2
G96S250
G76S500
G77S2500
20
G95F.002X0
G94F.300Z2
G97
S2000
T2(DRILL)
X0Z.2
G95F.003Z-.5
G94F.300Z2
M30
```

OmniTurn Training Manual
Richlin Machinery - (631) 694 9400
Program commands -”M-codes” continued

M89 - Stop the spindle and lock it (optional: C-Axis only)
This code is used to quickly stop the spindle to put a hole or a slot in an arbitrary C-Axis location. It is quicker because the spindle does not go through its “homing” routine before locking, as it does with M19.

M91, M92, M93, M94 - Wait for input (optional: C-Axis only)
These M-codes stop the program until an input is “on” or “off”. This is useful for coordinating activity for an auto-loader primarily. The OmniTurn ‘waits’ (the program stops, like M00 or M01) until the input is in the correct state.

Relay closure to 0VDC (COM) sets the input “on”.
The input is “off” when the relay is open.
The inputs are located on TB2 in the spindle cabinet. (see page 6-22 for spindle panel layout).
The commands are as follows:

- **M91**  Wait for TB2-5 to be open circuit
- **M92**  Wait for TB2-5 to be short to 0VDC
- **M93**  Wait for TB2-7 to be open circuit
- **M94**  Wait for TB2-7+ to be short to 0VDC

M95 - Conditional jump to subroutine (optional: C-Axis only)
This command will cause the program to jump to subroutine 1 if input 7 is “on” (shorted to 0VDC). Input 7 is located at TB2-9 in spindle cabinet (see page 6-22 for spindle panel layout). The condition must exist before the command is executed. Use dwell (G04) if necessary to insure that the state of the input is stable before the program executes the M97 command.

M97 - Conditional jump to subroutine (optional: PLC only)
This command will cause the program to jump to any subroutine if any available PLC input is either “on” or “off”. The syntax is M97InCnPn. In is the input which is being tested Cn is the condition; either 1 (“ON”) or 0 = (“OFF”) Pn is the subroutine which will be executed The condition must exist before the command is executed. Use dwell (G04) if necessary to insure that the state of the input is stable before the program executes the M97 command.

M98 - Jump to subroutine (unconditional)
When this command is executed, the program will jump to the specified subroutine. The syntax is M98Pn, where n is the subroutine number.

}n - Begin subroutine n
The first line in any subroutine must be the brace } followed by the subroutine number. No other text on that line.

M99 - End subroutine
The last line in any subroutine. The next line which will execute will be the line immediately after the line that called the subroutine.
Looping is used to perform repetitive moves without having to write long programs. The start of a loop is defined by LS and then the number of times you want to execute the loop. IE: LS35 will start a loop with 35 repetitions. This command has to be on a line by itself. As the end of the loop put a LF on a line by itself.

**NOTE: Text statements can not be used inside the loop!!!!!**

An example of this is having to make lots of notches on a part that are evenly spaced:

![Diagram of notches](image)

For this example the could could be:

G90G94F300
T1 (notch tool 045 wide)
X.25Z.2
Z-.035
G91 -------------------------- NOTE THIS LOOP IS DONE IN INCRIMENTAL

**LS16**
G95F.001X-.1
G04F.05
G00X.1
Z-.085

**LF**
G90 -------------------------- BACK TO ABSOLUTE MODE
Z1
M30
There is no order that the tools must be set up on the slide. It is not important in what order you do the offsets. It is possible to only do one offset and call it #3. The control would not care that there were no values in unused offsets. If however you call a tool that has not been set, there may be a collision.

For the following examples we will assume:

- The material is approx. .5” Ø.
- The part will be programmed so that all of the tools will start at the center of the part in X, and .1” away from the face in Z. Follow this format for the first few programs you write.

Later you can be more efficient with time and movements after you have more experience!

**To Set Left Hand Turning Tools (Tool #3)**

- Be sure that the slide has been HOMED.
- From the main screen go to the Jog mode by typing “J”
- Using the jog keys and joystick move the cutting tool until it is just off the material and slightly smaller than the major diameter.

• Turn the spindle on, select the jog speed for slow, and take a skim pass of the material as shown next.
• Then, move the slide back in Z. **Do not move the slide in X.** This cut will be used to establish the offset.

**To Set Left Hand Turning Tools** continued

<table>
<thead>
<tr>
<th>Jog</th>
<th>Automatic</th>
<th>Single Block</th>
<th>Manual Data Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slow</td>
<td>7.1000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Medium</td>
<td>8.1.0000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fast</td>
<td>9. Est. Home</td>
<td></td>
<td></td>
</tr>
<tr>
<td>.00005</td>
<td>S.Set Zero</td>
<td></td>
<td></td>
</tr>
<tr>
<td>.0010</td>
<td>H.Go Home</td>
<td></td>
<td></td>
</tr>
<tr>
<td>.0100</td>
<td>T.Set Tool</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Select "T" to start entering the tool offset

At this point, you are still in the jog mode. Instead of selecting a new jog speed now, select “T”. The control will now begin the sequence for entering a tool offset. See the next screen.
To Set Left Hand Turning Tools - continued

After you have selected T, the control will ask what tool it is that you are about to enter. Type the number tool, in this case 3. Then hit the “RETURN” key.

After you have selected the #3 tool offset, the control will ask you whether you want to enter the Z or X offset. In this case we have set the tool on the diameter of the material and we are ready to enter the X offset, so hit X.
Enter the diameter of the material just cut

Then measure the diameter of the material you just cut accurately with a micrometer. Enter this diameter when the screen asks for it. Remember that this will be a diameter measurement.

NOTE: If the tool was touched off on the back side of the part (-X), then enter the diameter as a negative.
To Set Left Hand Turning Tools - continued

Take some care but do not be overly careful since any error made here can be easily corrected with the tool offset correction later when you are making the first piece. After typing .4923 hit “RETURN”.

Now establish the Z offset, for tool #3

The setting of the Z offset is a little different.
- Touch the tool off in the Z axis and then press T like you did with the X axis.
- The control will now ask for a tool number. In this example you would press 3 and then enter.
- Then press Z when asked which axis you are setting.
- Then the control asks what the location of the tool is from absolute zero in Z.

<table>
<thead>
<tr>
<th>Jog</th>
<th>Automatic</th>
<th>Single Block</th>
<th>Manual Data Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.Slow</td>
<td>7..1000</td>
<td>-X</td>
<td>-Z</td>
</tr>
<tr>
<td>3.Fast</td>
<td>9..00005</td>
<td>S.Set Zero</td>
<td>+X</td>
</tr>
<tr>
<td>4..00005</td>
<td>H.Go Home</td>
<td>T.Set Tool</td>
<td></td>
</tr>
<tr>
<td>5..0010</td>
<td>O IS NOT A VALID OFFSET NUMBER</td>
<td>PRESS ESCAPE TO RETURN TO JOG MENU</td>
<td></td>
</tr>
<tr>
<td>6..0100</td>
<td>PRESS 'ESC' TO RETURN TO MENU</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Using a finished part to touch off in Z:
If you put a finished part in the collet against a stop this would give you the absolute face of the part. The parts you machine should have the same location in Z when they are done. So you can jog the tool over the face of the part and touch off and when asked what the location is in Z you could enter 0.
To Set Left Hand Turning Tools continued

At this point, you are still in the jog mode. Instead of selecting a new jog speed now, select “T”. The control will now begin the sequence for entering a tool offset. See the next screen.

After you have selected T, the control will ask what tool it is that you are about to enter. Type the number tool, in this case 3. Then hit the “RETURN” key.
After you have selected the #3 tool offset, the control will ask you whether you want to enter the Z or X offset. In this case we have set the tool on the diameter of the material and we are ready to enter the X offset, so hit X.

Then measure the diameter of the material you just cut accurately with a micrometer. Enter this diameter when the screen asks for it. Remember that this will be a diameter measurement.

**NOTE:** If the tool was touched off on the back side of the part (-X), then enter the diameter as a negative.
Take some care but do not be overly careful since any error made here can be easily corrected with the tool offset correction later when you are making the first piece. After typing .4923 hit "RETURN”.

Now establish the Z offset, for tool #3

The setting of the Z offset is a little different.

- Touch the tool off in the Z axis and then press T like you did with the X axis.
- The control will now ask for a tool number. In this example you would press 3 and then enter.
- Then press Z when asked which axis you are setting.
- Then the control asks what the location of the tool is from absolute zero in Z.

Using a finished part to touch off in Z:
If you put a finished part in the collet against a stop this would give you the absolute face of the part. The parts you machine should have the same location in Z when they are done. So you can jog the tool over the face of the part and touch off and when asked what the location is in Z you could enter 0.
Setting ID Tools, ie Boring tools & Threading tools

The procedure for setting ID tools is similar to the two previous tools. The only difference is how you will touch off to determine the turned diameter.

Setting Threading tools

The threading tool is set similar to the other tools.

Setting X: The turned diameter is set like the other tools you have done. The offset can be set on the OD or ID.
OmniTurn Training

Finding X 0 for a Drill

- Setting a drill to be exactly on center is easier than it looks.
- First get a 5/8” collet and a 5/8” shaft. A common shaft to use is a Multibar tool holder or a OTC drill shank holder.
- Put the drill holder on the tooling plate. The one shown here is an Omni-305 with three holes. Do not tighten it down yet, just hold it enough so it does not fall down. Put the holder where you want it for your setup

- Now put the 5/8” 5C collet in the spindle.

- Slide the 5/8” shank into the holder.
- Jog the tooling plate so that the 5/8” shaft lines up with the collet. Loosen the Omni-305 so you can slide the shaft into the collet.
- Then lock the collet on the shaft.
- Now tighten the Omni-305 to the tooling plate.
- The tool offset for this first location can now be set for X = 0.
- Do not move the slide yet in the X axis. You can move in Z needed.
- Open the collet, take the 5/8” shaft out.

- It is now easy to set X centerline for the other two holes. The center distance between all OmniTurn holders is 1.100”. So we can use the jog commands to bring the second hole on center to set it as well.
- Jog the X using the #7 (.1”) and #8 (1.0”) commands.
- With the second hole on center set the X offset value for this tool to 0.

- Repeat the same steps as the previous step to set the 3rd hole.
- Now go back and set the Z values with the drilling tools, work piece and correct work holding.

Richlin Machinery (631) 694-9400 www.omni-turn.com
### Automatic Mode, Functions & Switches

**The "F" keys have the following functions:**

**NOTE:** These keys are not effective while program is running

<table>
<thead>
<tr>
<th>F 2</th>
<th>Tool Offsets</th>
<th>Adjust tool offsets.</th>
</tr>
</thead>
<tbody>
<tr>
<td>F 3</td>
<td>Edit</td>
<td>Edit existing programs or create new programs.</td>
</tr>
<tr>
<td>F 4</td>
<td>Verify</td>
<td>Verify the program in memory and display the tool paths.</td>
</tr>
<tr>
<td>F 5</td>
<td>New File</td>
<td>This will remove the program from active memory and open the file handler screen.</td>
</tr>
<tr>
<td>F 6</td>
<td>Prog Search</td>
<td>This enables the program to be started some place other than the start</td>
</tr>
</tbody>
</table>
| F 8   | File Ops     | File Operations:  
|        |              | File handler screen  
|        |              | Erase programs  
|        |              | Create new program  
|        |              | Up- or Download programs from another computer over RS232 |
| F 9   | SecOffsets   | Adjust values of secondary tool offsets and TNR compensation values |
| F10   | Spec Functions | Special functions:  
|        |              | Load tool offset values for program (previously saved)  
|        |              | Save tool offset values with program in memory  
|        |              | Set/reset parts counter  
|        |              | Set value of countdown counter |
| J     | Jog Mode     | Switch to Jog page |
| M     | MDI          | Switch to Manual Data Input page |

**HOT keys on the keyboard while in the automatic mode**  
**NOTE:** These keys are effective whether program is running or not

<table>
<thead>
<tr>
<th>C</th>
<th>Cycle Repeat</th>
<th>The program will run continuously. To cancel, Preass “C” again, or press “A”.</th>
</tr>
</thead>
<tbody>
<tr>
<td>/</td>
<td>Block delete</td>
<td>With this active the control will skip over program lines starting with &quot;/&quot;. In your program, put the &quot;/&quot; symbol as the first character in any block you might want to skip.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Press &quot;/&quot; again to cancel.)</td>
</tr>
<tr>
<td>O</td>
<td>Optional stop</td>
<td>This makes the M01 act as a stop program command (M00). In your program, put M01 wherever you might want to stop. (Press “O” again to cancel.)</td>
</tr>
<tr>
<td>S</td>
<td>Single block</td>
<td>The program will run one line at a time with each press of the cycle start button.</td>
</tr>
<tr>
<td>Pg Up</td>
<td>Coolant on/off (M08/09). Press to turn coolant on, press again to turn coolant off.</td>
<td></td>
</tr>
<tr>
<td>Pg Dn</td>
<td>Parts catcher Out/In (M25/26). Press to turn M25 on, press again to turn M26 off. Note that M25 is an auxiliary output on all OmniTurn machines; if a parts catcher is not installed, this M-function is available for any auxiliary function you need. See page 5.19 for keyboard.</td>
<td></td>
</tr>
</tbody>
</table>
### Automatic Mode, Switches

#### Functions and switches available from Automatic Mode

<table>
<thead>
<tr>
<th>To Go</th>
<th>Position</th>
<th>Functions</th>
<th>Switches</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>X</strong>: +0.00000</td>
<td><strong>X</strong>: +0.00000</td>
<td>F2-Tool Offsets</td>
<td>C.Cycle Repeat</td>
</tr>
<tr>
<td><strong>Z</strong>: +0.00000</td>
<td><strong>Z</strong>: +0.00000</td>
<td>F3-Edit</td>
<td>/Block Skip</td>
</tr>
<tr>
<td></td>
<td></td>
<td>F4-Verify</td>
<td>O.Optional Stop</td>
</tr>
<tr>
<td></td>
<td></td>
<td>F5-New File</td>
<td>S.Single Block</td>
</tr>
<tr>
<td></td>
<td></td>
<td>F6-Program Search</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>F8-File Ops</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>F9-Sec. Offsets</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>F10-Spec. Functions</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>J-Jog Mode</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>M-Manual Data Input</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Position</td>
<td>Press Cycle Start to Begin</td>
<td></td>
</tr>
<tr>
<td></td>
<td>X: +0.00000</td>
<td>Feed 0.000 IPM</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Z: +0.00000</td>
<td>100% Speed</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Tool 0</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Press key to enable function</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Press key to enable switch; press again to disable. Effective at any time.</td>
<td></td>
</tr>
</tbody>
</table>

#### Cycle Repeat Mode

To run the program over and over without pressing Cycle Start, select Cycle Repeat by pressing the C key. This is the preferred mode for bar jobs and loaders, which automatically supply material for machining. To cancel the Cycle Repeat mode, press C again, or press A.
Block Skip Mode
If there are lines in your program that need to be skipped over sometimes, add the forward slash character “/” which is under the question mark to the beginning of each line to skip. The entire program will run normally unless you press the “/” key; then only lines not marked with “/” will run. The marked lines will be skipped over. Press “/” again to return to normal automatic mode.

Optional Stop Mode
If there are specific lines in your program where you need to stop sometimes, put the M01 (Optional Stop) code on a line by itself. The program will run without stopping at these places unless you press O. The Optional Stop switch will highlight and the program will stop at each line with M01. Press Cycle Start to continue. Press O again to return to normal automatic mode. See Section Two, page 2.61 for more about Optional Stop.

Single Block Mode
When running a program for the first time it is very useful to step through the program one line at a time. The tool positions can be checked and offsets can be verified. Press S to initiate Single Block mode. Cycle Start must be pressed to execute each line. Press S again to return to normal automatic mode.
On the top of the keyboard is a group of “F” keys. These are used differently throughout the control software. Notations are made on the screen to help the operator remember how the keys are being used with the different sections of software. Care should be taken to remember that these keys change depending on the “Mode” the control is in. Following will be the description of how the Function keys are used in the Automatic mode.

---

**Automatic Mode, Function Keys**

---

**Automatic Mode, F2: Tool Offsets**

**F2 Tool offset screen, used to modify tool offsets**

This function key brings up the screen to adjust the tool offsets. Tool offsets are used to correct the starting location of the tools, and they will effect the finished part dimensions. These values are created when the tools are setup in the jog mode. When the F2 key is pressed the screen will then ask what tool number you want to adjust. The distances shown are the amount needed to travel from the Home position to the offset location. See below:

In the example above you see the offset screen with three tools being used. When it is necessary to correct a tool offset, enter the amount of change that is required. As an example, we will assume that tool 2 in the above example is a turning tool and is cutting apart .001” too large. So enter the offset change of -.001” for the X Diameter offset. When this value is entered you will notice that the total value of X has changed. This addition does not have to be done by the operator.

<table>
<thead>
<tr>
<th>OFFSET NUMBER</th>
<th>X Offset</th>
<th>Z Offset</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+0.86480</td>
<td>-1.25340</td>
</tr>
<tr>
<td>2</td>
<td>+1.65025</td>
<td>-1.99200</td>
</tr>
<tr>
<td>3</td>
<td>+2.91130</td>
<td>-0.93885</td>
</tr>
<tr>
<td>4</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>5</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>6</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>7</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>8</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>9</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>10</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>11</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>12</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>13</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>14</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>15</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>16</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>17</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>18</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>19</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>20</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>21</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>22</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>23</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>24</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>25</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>26</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>27</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>28</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>29</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>30</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>31</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>32</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
</tbody>
</table>

OFFSET NUMBER:
Press Esc to exit offset adjustment screen
After selecting a number and pressing Return the screen will ask:

<table>
<thead>
<tr>
<th>Tool</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+0.86480</td>
<td>-1.25340</td>
</tr>
<tr>
<td>2</td>
<td>+1.65025</td>
<td>-1.99200</td>
</tr>
<tr>
<td>3</td>
<td>+2.91130</td>
<td>-0.93885</td>
</tr>
<tr>
<td>4</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>5</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>6</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>7</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>8</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>9</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>10</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>11</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>12</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>13</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>14</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>15</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>16</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
</tbody>
</table>

**X DIAMETER ADJUSTMENT:**
Press Esc to exit offset adjustment screen

Now enter the value of change-(ie: -.001) and press Return. The value of X will update and then ask you about Z.

<table>
<thead>
<tr>
<th>Tool</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+0.86480</td>
<td>-1.25340</td>
</tr>
<tr>
<td>2</td>
<td>+1.64975</td>
<td>-1.99200</td>
</tr>
<tr>
<td>3</td>
<td>+2.91130</td>
<td>-0.93885</td>
</tr>
<tr>
<td>4</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>5</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>6</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>7</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>8</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>9</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>10</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>11</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>12</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>13</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>14</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>15</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
<tr>
<td>16</td>
<td>+0.00000</td>
<td>+0.00000</td>
</tr>
</tbody>
</table>

**Z ADJUSTMENT:**
Press Esc to exit offset adjustment screen

Enter the amount of change in Z and then press Return.

To correct another tool, enter the tool number now and press Return. **To exit the tool offset** correction screen press ESC and press return. This will tell the control that you are done and bring you back to the Automatic mode.
Notes on Tool Offsets:

1. The control will allow you to clear all the offsets by pressing C (for clear). Please only do this when you have had experience with the control and understand what you are doing. Clearing offsets can cause you to crash tools if it is done incorrectly!

2. The smallest offset changes are:
   - 0.00005” in Z
   - 0.0001” in X, (this is equal to .00005” on the radius)

3. Tool Offset changes greater than .05” will invoke “Do you really want to move [distance] (y/n)?” prompt to alert the you of possible mistake. This is a safety feature to ensure that you do not put in a large correction in error, ie 0.1” instead of .01”

4. If you have no change to a offset value just press Return without inputting a value. The control assumes that you want zero change.

5. Tool offsets can be changed at any program stop: m00, m01 or Motion Stop.

Automatic Mode, F3, Program Editing

F3 Program Editing

This function starts the text editor, used to change existing programs, or to create new ones.

The editor is described starting on page 5.19.
Automatic Mode, F5: New File, F6: Program Search

F5  **New Program**: used to change the running program to a new one
This function is used to change the program that is in the current file and allow the operator to enter in a new program name to be run. When F5 is pressed the file handler screen will appear. You will see the list of programs available. Use the arrow keys to highlight the desired program.
See page 1.17 - 1.20, Jog Mode, File Handler for details.

F6  **Program Search**: used to start the program at a certain tool instead of the beginning.
The program “DEMO” is used in the following example.
This function will allow you to start the program at a certain tool instead of the beginning.
**NOTE**: If there are less than two tools in your program, pressing F6 will generate an error as follows:

```
Program contains no Search Points
Press any key to return to Auto mode
```

If there is more than one tool in your program they will be listed, with highlight on first tool. Use arrow keys to select desired tool, then press Enter.

```
T 1
T 2
T 0

Use arrow keys to highlight Search Point
Enter to select, or ESC to cancel search
```

When Enter is pressed, another screen will appear defining the tool and the line number, and any M-functions that are active at that point:

```
Searched to tool 2, line 46
The following conditions will be established:
M03 S2500
M08
Press cycle start to proceed or Esc to cancel
```

Pressing Cycle Start will return you to Auto screen, in Single Block mode, with highlight on selected tool:

```
To Go
X: +0.00000
Z: +0.00000

Switches
C.Cycle Repeat
/.Block Skip
O.Optional Stop
S.Single Block

Functions
F2-Tool Offsets
F3-Edit
F4-Verify
F5-New File

Position
X: +7.86285
Z: -0.62225

Feed
0.000.000 IPM
100%

Speed
0 RPM
100%
Tool 2
```

The Program Search function reads every block of the program up to the tool selected, so the machine will start running as if it has just executed all the previous blocks. When Cycle Start is pressed, the current feed rate will be displayed and the tool will move to the part. **Note that no collision avoidance calculation is done**, so be sure that the tool can move safely to the part before pressing Cycle Start.
Automatic Mode, F8: File Operations

**F8 File Ops**: File Operations. This Function allows you to select, copy, delete and create new programs, and to send or receive programs via RS232 interface.

See page 1.17 - 1.20, Jog Mode, File Handler for details.

The screen is described below:

Directories and disks to search
(a: = floppy; d: = USB stick)

Program files in alphabetical order. Use arrow keys to move highlight, or select initial letter or number to highlight first file in series. Press Enter to load selected program into memory.

Options selectable with "Alt" key

- **Alt-C = Copy file**: You will be prompted for a location to copy to. This option is most useful for copying programs written at your desktop. You must first use Alt-S to select directory. See next page for details.

- **Alt-D = Delete file**: You will be prompted “Delete filename Y/N” Press N if you change your mind.

- **Alt-N = Create New file**: You will be prompted “Name of file to create?” Type the name then press Enter. The OmniTurn editor will start. When you leave the editor, you will have opportunity to load this program into memory (make it the current machine file). The editor is described in Section Five.

- **Alt-S = Select directory**: Use the up/dn arrow keys to highlight desired directory. Press Enter to display program files in that directory. Pressing the “D” key will remove the highlighted directory from the list. Highlight “OTHER” to choose a directory not on the list.

- **Alt-R = RS232 operations**: This option allows you to send/recieve programs via the serial port.

Options Selectable with “Alt” key

Alt-C = Copy file: You will be prompted for a location to copy to. This option is most useful for copying programs written at your desktop. You must first use Alt-S to select directory. See next page for details.

Alt-D = Delete file: You will be prompted “Delete filename Y/N” Press N if you change your mind.

Alt-N = Create New file: You will be prompted “Name of file to create?” Type the name then press Enter. The OmniTurn editor will start. When you leave the editor, you will have opportunity to load this program into memory (make it the current machine file). The editor is described in Section Five.

Alt-S = Select directory: Use the up/dn arrow keys to highlight desired directory. Press Enter to display program files in that directory. Pressing the “D” key will remove the highlighted directory from the list. Highlight “OTHER” to choose a directory not on the list.

Alt-R = RS232 operations: This option allows you to send/recieve programs via the serial port.
Automatic Mode, F9: Secondary Offsets

F9 Secondary tool offset screen; used to modify secondary tool offsets

This function will call up the secondary offset table. There are 32 offsets available and 32 tool nose radius compensation offsets. Please refer to the section on secondary offsets for their use. Notice that this differs from the offset table screen in that almost all of the offset values are set to 0.00000. Secondary offsets are corrected like offsets.

First: Select a secondary offset number
Next: Use the return key to enter past the X and Z inputs.
Then: Enter the value of tool nose compensation, IE .007 and then press ESC

Changing a TNR compensation value:
When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in the offset and you want to change it to .008, just enter the new value. You do not have to clear the register first. If you want to correct a value slightly, you must enter in the final value needed. IE: if you have .007 and want to increase it by .001 you must enter .008. Do not enter -.001

Clearing secondary offsets to Zero.
It is possible to clear all of the secondary offsets by pressing C when asked for a secondary offset number. This will set the entire table to

Individual offsets can be set to zero by pressing C when asked to enter a correction amount.
F10  Special Functions, used to call up a list of special functions.

- **Save and recall tool offset tables.**
  This function enables you to load or save the tool offsets with the program. Press L at this screen to load the offsets for the current program from the disk; press S to save the offsets for the current program to the disk. Use the OmniTurn 805 eight-position tool-holder (p/n 805PFB), which allows you to remove and replace tooling exactly as it was, and this function cuts setup time for periodic jobs to just a few moments. Also works well for re-loading offsets for different programs which use same set of tooling.

- **Set the number of cycles to repeat:**
  Press C at this screen, then enter the number of cycles to repeat, then press Enter to return. Press C at Auto screen to enable Cycle Repeat. This is good for use with a barfeed. As an example you could set the machine up and tell it you need 20 pieces. The OmniTurn will count down to zero and then stop.

- **Turn on the Parts Counter:** Press P at this screen. You will return to the Auto mode screen, and there will be a counter on the screen. The counter will count up each time an M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again.
M function keyboard controls

Toggle M functions on and off with keyboard controls.
- Press the key once to turn the function on, press again to turn it off.
- Works only in Jog or Automatic mode.

To review the functions of all the front panel controls, refer to pages 1.4, 1.5 and 1.6 in Section 1.
Automatic Mode

The "F" keys have the following functions:

F1  Quit          Go back to the Main menus
F2  Offset        Adjust tool offsets, correct part size
F3  Edit          Input and correct programs
F4  DIR           When no file is in memory this will list all the programs on the user disk
                 With a program in memory it will verify and plot the program
F5  Newprog       This will remove the program from active memory and allow a new one to
                  be entered
F6  Searchto      This enables the program to be started somewhere other than the start
F7  Prog          Runs Calcaid programming system
F8  Diskop        Disk Operations
                 Erase programs
                 Make a new System disk
                 Make a copy of the user program disk
                 Down and Up load programs from another computer over RS232
                 Set communication parameters
F9  Seccmp        Adjust values of secondary tool offsets and TNR compensation values
F10 Sp.fun        Special functions
                 Parts Counter
                 Set value of countdown counter for Continuous cycle
                 Preset feed rate override before starting a program
                 Store tool offset values with program in memory
                 Load tool offset values from memory
                 Set max spindle speed of machine

HOT keys on the keyboard while in the automatic mode

C  Continuous     The program will run continuously - This toggles on / off
O  Optional stop  This makes the M01 act as a stop program command - This toggles on / off
/  Block delete   With this active the control will skip over program lines starting with "/"
A  Automatic      The program will run from start to finish with one cycle start
S  Single block   The program will run one line at a time with each press of the cycle start
F1 - F10         Feed rate overrides. The function keys will adjust feed rates
                 (only while program is in motion)
Pg Up            Coolant on/off (M08/09)
Pg Dn            Parts catcher Out/In (M25/26)
M function keyboard controls

Toggle M functions on and off with keyboard controls.
- Press the key once to turn the function on, Press again to turn it off
- Works only in Jog or Automatic mode
Running programs

In the Automatic mode the control displays the program that it is currently running. When the control is turned on there is no program selected to run and this space is blank.

Be sure that the tool offsets are correct for the program to be run. If this program is the same as when the control was last shut down, the offsets should still be the same and the program will run without resetting the tools. For example, if you are running a program and shut the control down for the night. When you start up the control the next morning all you have to do is enter the program name in the file to be run once you enter the Automatic mode the next morning. See below. To recall tool offsets from memory refer to F10 in this section.

1. Running an existing program

If you have a program saved on the program disk it’s name can be entered now and the screen will show that this is the file now in memory. When the cycle start button is pressed the program will be executed.

2. Running a different existing program (F5)

Once a program has been selected it stays in memory until it is changed. In the Automatic mode, F5 will delete the currently running program and ask for a new program name. You will notice that after F5 has been pressed the file in memory is blank. If you forget the exact name of the program that you want to run you can press F4 (directory) after F5 has been pressed. This F4 command will list all of your programs on the A: drive, 5-1/4”, this is where your programs are stored.
Automatic Mode

There is a program in memory ready to run

- POSITION : X +0.00000 Z +0.00000
- COMMAND : X +0.00000 Z +0.00000
- FEED 10.0 IPM
- PERCENT FEED: 100

- PRESS CYCLE START

<table>
<thead>
<tr>
<th>Jog</th>
<th>Automatic</th>
<th>Single Block</th>
<th>Manual Data Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1-F10 FEED 10-100%</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FILE IN MEMORY: DEMOPR</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>'O' FOR OPTIONAL STOP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>'^' FOR BLOCK DELETE</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>'C' FOR CYCLE REPEAT</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PRESS 'S' FOR SINGLE BLOCK</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- IQUIT 2 OFFSET 3 EDIT 4 DIR 5 NEWPROG 6 SEARCHTO 7 PROG 8 DISKOP 9 SECCMP OSPEFUN

Press F5 to select new program

Program "DEMOPR" is being run

Program Run - Single step - "S"

It is possible to run the program one line at a time. This is useful when running a new program for the first time. The control displays the next command to be executed before it is run. You can look to see what is going to happen before a mistake is made. To accomplish this get the control into the Automatic mode & input the file to be run. At this point do not push the cycle start yet. Press "S" to activate the single block mode.

- POSITION : X +0.00000 Z +0.00000
- COMMAND : X +0.00000 Z +0.00000
- FEED 10.0 IPM
- PERCENT FEED: 100

- PRESS CYCLE START

<table>
<thead>
<tr>
<th>Jog</th>
<th>Automatic</th>
<th>Single Block</th>
<th>Manual Data Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1-F10 FEED 10-100%</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FILE IN MEMORY: DEMOPR</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>'O' FOR OPTIONAL STOP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>'^' FOR BLOCK DELETE</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>'C' FOR CYCLE REPEAT</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PRESS 'S' FOR SINGLE BLOCK</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- IQUIT 2 OFFSET 3 EDIT 4 DIR 5 NEWPROG 6 SEARCHTO 7 PROG 8 DISKOP 9 SECCMP OSPEFUN

Options used in Automatic mode

To toggle back to the regular automatic cycle, press “A” to turn off the single block mode.
**Automatic Mode**

**Parts counter - P**

<table>
<thead>
<tr>
<th>POSITION</th>
<th>Z</th>
<th>FEED</th>
<th>IPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>X +0.00000</td>
<td>Z +0.00000</td>
<td>10.0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>COMMAND</th>
<th>Z</th>
<th>PERCENT</th>
<th>FEED</th>
</tr>
</thead>
<tbody>
<tr>
<td>X +0.00000</td>
<td>Z +0.00000</td>
<td>100</td>
<td></td>
</tr>
</tbody>
</table>

**PRESS CYCLE START**

<table>
<thead>
<tr>
<th>Jog</th>
<th>Automatic</th>
<th>Single Block</th>
<th>Manual Data Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1-F10 FEED 10-100%</td>
<td></td>
<td></td>
<td>Parts counter 3</td>
</tr>
<tr>
<td>FILE IN MEMORY: DEMOPR</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>O FOR OPTIONAL STOP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>/ FOR BLOCK DELETE</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>C FOR CYCLE REPEAT</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PRESS 'S' FOR SINGLE BLOCK</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

1QUIT 2OFFSET 3EDIT 4DIR 5NEWFROG 6SEARCHTO 7PROG 8DISKOP 9SECCMP 0SPEFUN

The parts counter is turned on in F10

- **Turn on a PARTS COUNTER** first press F10, then Press P while you have this screen open. There will be no effect here. When you go to the Automatic mode main screen there will be a counter on the screen. The counter will count up each time a M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again. See notes at the end of this chapter on Function keys, F10.

**Program run - Optional stop activation - M01 - “O”**

Optional stops can be put into the program, M01. This stop command is one that can be skipped over. To turn the optional stops on go to Automatic mode, once the program is selected and before the program is run, press “O”. This will cause the program to stop like a M00. To get past the stop, press “cycle start”. To turn the optional stop off, press “O” again.

Uses for the optional stop:
- insert an M01 after a G92 statement for a new tool. This will help when running a new program to be sure that the tool offsets have been entered correctly. Once the program is tested you can turn off the stop and let the program run automatically.
- Have an M01 at the beginning of a program that is going to use an automatic bar feed or parts loader. This way, you can have the optional stop activated when you are setting up the machine. Once the cycle and program are proven correct, you can turn off the stop and let the machine run automatically.

**Program run - Cycle repeat - “C”**

This is useful for automatic bar feeder or automatic loader operations. When this is activated the program will automatically go back to the beginning of the program after a M30 is encountered and run the program again. The program will continue to run until it is stopped or the continuous counter is set. (see F10 in automatic mode). To turn it off, press “C” again.
Automatic Mode

/ - Block Delete

| POSITION | X +0.00000 | Z +0.00000 | FEED 10.0 IPM |
| COMMAND  | X +0.00000 | Z +0.00000 | PERCENT FEED: 100 |

PRESS CYCLE START

<table>
<thead>
<tr>
<th>Jog</th>
<th>Automatic</th>
<th>Single Block</th>
<th>Manual Data Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1-F10 FEED 10-100%</td>
<td>FILE IN MEMORY: DEMOPR</td>
<td>'O' FOR OPTIONAL STOP</td>
<td>'C' FOR CYCLE REPEAT</td>
</tr>
<tr>
<td>'/ ' FOR BLOCK DELETE</td>
<td>PRESS 'S' FOR SINGLE BLOCK</td>
<td>BLOCK DELETE ACTIVE</td>
<td></td>
</tr>
</tbody>
</table>

The block delete is used to bypass lines of the program. Put the forward slash “/” at the beginning of a line. When you want to skip the line just press the “/” key in the automatic mode. Then the Block Delete Active text will show up on the screen. This is commonly put on the line with the coolant on command. This way you can turn the coolant off (skip over the coolant on line) by activating the Block Delete.

Creating a new program

There are a number of ways to create a new program. Here are a few:

- **Use the text editor in OmniTurn.** First a new program name has to be created. This is done by going into the Automatic mode and typing in the new name when the control asks “FILE TO BE PROCESSED”. After the RETURN key is hit the control will answer “FILE NOT FOUND, PRESS ANY KEY TO CONTINUE”. By doing this you have accomplished two things.
  1. If there was already a program with the name you just entered, the control would now be ready to run it. If this is the case, then you would have to select a new name or change the program of the existing one already there.
  2. If there was no other program that had the new name then there was one created and loaded into the text editor.

Once the new name is entered into the text editor, press F3 to enter the editor. The text editor will ask “PRESS F1 TO CREATE A NEW FILE, ESC TO ABORT”. After pushing F1 the editor will provide a new blank screen to enter your program.

- **CAM system off line.** Transfer a file via floppy or RS-232. Once they are on the OmniTurn program disk they can be run like any other existing program. Please refer to the section in DOS notes on the format.

- **RS-232 or Disk transfer.** Manually enter a program in a text editor on another computer, transfer as above. Once they are on the OmniTurn program disk, they can be run like any other existing program. Please refer to the section in DOS notes on the format.

- **Use Calcaid in OmniTurn.** See the section on using Calcaid.
Automatic Mode

Function Keys

On the left side of the keyboard is a group of “F” keys. These are used differently throughout the control software. Notations are made on the screen to help the operator remember how the keys are being used with the different sections of software. Care should be taken to remember that these keys change depending on the “Mode” the control is in. Following will be the description of how the Function keys are used in the Automatic mode.

Function Keys - Automatic Mode - Program not in process

Following are the definitions of the function keys when the control is in the automatic mode and the program is not in motion.

F1 Exit Automatic mode, go to main screen

Pressing the F1 key will exit you from the Automatic mode. This is necessary to get to any of the other modes, ie. Jog or MDI.

F2 Tool offset screen, used to modify tool offsets

This function key brings up the screen to adjust the tool offsets. Tool offsets are used to correct the starting location of the tools, and they will effect the finished part dimensions. These values are created when the tools are setup in the jog mode. When the F2 key is pressed the screen will then ask what tool number you want to adjust. The distances shown are the amount needed to travel from the Home position to the offset location. See below:

| OFFSET NUMBER: |
| Press Esc to exit offset adjustment screen |

In the example above you see the offset screen with three tools being used. When it is necessary to correct a tool offset, enter the amount of change that is required. As an example, we will assume that tool 2 in the above example is a turning tool and is cutting a part .001” too large. So enter the offset change of - .001” for the X Diameter offset. When this value is entered you will notice that the total value of X has changed. This addition does not have to be done by the
Automatic Mode

operator.

After selecting a number and pressing Return the screen will ask

<table>
<thead>
<tr>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+0.86480</td>
</tr>
<tr>
<td>2</td>
<td>+2.91130</td>
</tr>
<tr>
<td>3</td>
<td>+0.00000</td>
</tr>
<tr>
<td>4</td>
<td>+0.00000</td>
</tr>
<tr>
<td>5</td>
<td>+0.00000</td>
</tr>
<tr>
<td>6</td>
<td>+0.00000</td>
</tr>
<tr>
<td>7</td>
<td>+0.00000</td>
</tr>
<tr>
<td>8</td>
<td>+0.00000</td>
</tr>
<tr>
<td>9</td>
<td>+0.00000</td>
</tr>
<tr>
<td>10</td>
<td>+0.00000</td>
</tr>
<tr>
<td>11</td>
<td>+0.00000</td>
</tr>
<tr>
<td>12</td>
<td>+0.00000</td>
</tr>
<tr>
<td>13</td>
<td>+0.00000</td>
</tr>
<tr>
<td>14</td>
<td>+0.00000</td>
</tr>
<tr>
<td>15</td>
<td>+0.00000</td>
</tr>
<tr>
<td>16</td>
<td>+0.00000</td>
</tr>
</tbody>
</table>

X DIAMETER ADJUSTMENT:
Press Esc to exit offset adjustment screen

Now enter the value of change (i.e., -0.001) and press Return. The value of X will update and then ask you about Z.

<table>
<thead>
<tr>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+0.86480</td>
</tr>
<tr>
<td>2</td>
<td>+1.64975</td>
</tr>
<tr>
<td>3</td>
<td>+2.91130</td>
</tr>
<tr>
<td>4</td>
<td>+0.00000</td>
</tr>
<tr>
<td>5</td>
<td>+0.00000</td>
</tr>
<tr>
<td>6</td>
<td>+0.00000</td>
</tr>
<tr>
<td>7</td>
<td>+0.00000</td>
</tr>
<tr>
<td>8</td>
<td>+0.00000</td>
</tr>
<tr>
<td>9</td>
<td>+0.00000</td>
</tr>
<tr>
<td>10</td>
<td>+0.00000</td>
</tr>
<tr>
<td>11</td>
<td>+0.00000</td>
</tr>
<tr>
<td>12</td>
<td>+0.00000</td>
</tr>
<tr>
<td>13</td>
<td>+0.00000</td>
</tr>
<tr>
<td>14</td>
<td>+0.00000</td>
</tr>
<tr>
<td>15</td>
<td>+0.00000</td>
</tr>
<tr>
<td>16</td>
<td>+0.00000</td>
</tr>
</tbody>
</table>

Z ADJUSTMENT:
Press Esc to exit offset adjustment screen

Enter the amount of change in Z and then press Return.

To correct another tool, enter the tool number now and press Return. **To exit the tool offset correction screen press ESC and press return.** This will tell the control that you are done and bring you back to the Automatic mode.
**Automatic Mode**

**Notes:**

1. The control will allow you to clear the offsets by pressing C (for clear). Please only do this when you have had experience with the control and understand what you are doing. Clearing offsets can cause you to crash tools if it is done incorrectly!

2. The smallest offset changes are:
   .00005” in Z
   .0001” in X, (this is equal to .00005” on the radius)

3. For Tool Offset changes of more than .02” the control will ask the operator if this is correct. If not it will ask you to re-enter the correction. This is a safety feature to ensure that you do not put in a large correction in error, ie 1” instead of .001”

4. If you have no change to a offset value just press Return without inputting a value. The control assumes that you want Zero change.
Automatic Mode

Program Editing - F3

F3  *Edit, On screen text editor, used to change existing programs, or enter new ones*

The editor is a full function text editor. In the OmniTurn you will be using only a small part of the capability of the editor. In the following description the most basic functions. If you want to learn more follow the instructions given in the HELP screens. (F1 while in the editor is active)

Starting the editor:
The editor is accessed from the Automatic mode by pressing F3 at any time. The program listed as “FILE IN MEMORY” will be activated. If you want to work on a program you have to make it the active program. When you enter the Automatic mode it asks “FILE TO BE PROCESSED”, type the file name that you want to edit and press RETURN. When the file name appears as the file in memory press F3. If you have to correct the program that is currently running just press F3.

- It will ask if you want to make a Backup copy. Making these backup copies is not required. If you are new to PC’s and DOS it is suggested that you make the backup copy so that if you loose the work that you have created it can be brought back.

- Either press “ESC” for no backup or F1 to create the backup file

The editor can also be used to enter new programs.

- Get to the Automatic mode
- If there is a program in file memory press F5
- Enter the new program name when prompted: “FILE TO BE PROCESSED”
- If the control does not tell the program is not found then you are using a name that already exists. Either pick a new name or plan on erasing the program that already exists with that name.

- Press any key to continue
- Press F3 to enter the editor
- The editor will ask if you really intend to create a new program, press F1 if you do, if not press “ESC”

Exiting the editor and saving corrections made

- Press F1 - this is for HELP
- Press F2 - this will exit the editor and save the corrections that have been made

Exiting the editor and NOT saving the corrections made

- Press F1 - this is for HELP
- Press “ESC” - this will halt the automatic saving function of the exit routine
- Press F2 - this will exit the editor without saving the corrections
Automatic Mode

F4  Directory, list all of the programs on the disk - in use when a program is not active

This function key is not always active. Once you have selected a program to run and there is an active file in memory this function is deactivated. If you have yet to select a current program this key will bring up a list of programs available on the program disk. This will list all of the programs you have stored on the A: disk, the 5-1/4". Once you have reviewed the programs available press “Esc” to return to the Automatic mode screen.

F5  Different file, used to change the running program to a new one

This function key is used to change the program that is in the current file and allow the operator to enter in a new program name to be run. After F5 is pressed the control will ask “FILE TO BE PROCESSED”. Type the new name in and Return. Then press the cycle start and the new program will run. Be sure that the tool offsets are set before running the new program. This key is not always active. If there is no active file in memory F5 does not appear.

F6  Search to, used to start the program at another location other than the beginning

This function key will allow you to start the program at a point other than the beginning. This is very useful for running new programs and skipping over sections of program that you do not have to check. It is intended for skipping to a tool change, this is an easily noted beginning section of code. F6 can also be used with programs that have line numbers.

After the F6 command is pushed (while in the Automatic mode) the screen will ask “SEARCH TO?”

| POSITION | X +0.00000 Z +0.00000 | FEED 10.0 IPM |
| COMMAND  | X +0.00000 Z +0.00000 | PERCENT FEED: 100 |

SEARCH TO?

Jog  Automatic  Single Block  Manual Input

F1-F10 FEED 10-100%
FILE IN MEMORY: DEMO
'O' FOR OPTIONAL STOP
'C' FOR CYCLE REPEAT
'/' FOR BLOCK DELETE
PRESS 'S' FOR SINGLE BLOCK

QUIT 2OFFSET 3EDIT 4DIR 5NEWPROG 6SEARCHTO 7PROG 8DISKOP 9SECCMP 0SPECFUN

After typing in the text to search to press the Return key. Then press the cycle start button to start the slide.

As an example, if you are want to skip to Tool #2 type in T2. The control will skip the code before this line and start the program with the T2 command. If you are using an OmniTurn with spindle control be sure that after your tool changes you have a spindle on (M03 or M04) and an S command. The F6 - Search to does not read the previous commands and you will have to be sure that the spindle is running.

If you use line numbers in your program, it is possible to skip to these instead of the tool changes.
Automatic Mode

8. This will exit the file handling screen and bring you back to the Automatic mode screen.

F9 Secondary tool offset screen, used to modify secondary tool offsets

This function will call up the secondary offset table. There are 32 offsets available and 32 tool nose radius compensation offsets. Please refer to the section on secondary offsets for their use. Notice that this differs from the offset table screen in that almost all of the offset values are set to 0.00000. Secondary offsets are corrected like offsets.

<p>| | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>17</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>18</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>19</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>20</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>21</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>22</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>23</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>24</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>25</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>26</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>27</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>28</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>29</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>30</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>31</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td>32</td>
<td>X: +0.00000 Z: +0.00000 R:0.00000</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Secondary offset number:
Press C to clear all offsets:
Press Esc to exit offset adjustment screen

First: Select a secondary offset number
Next: Use the return key to enter past the X and Z inputs.
Then: Enter the value of tool nose compensation, IE .007 and then press ESC

Changing a TNR compensation value:
When you put a value in the R offset table it writes over the old value. So if you have a number already in the register that you want to use and it is not the value needed, all you have to do is enter the correct value. As an example if you have a value of .032 in the offset and you want to change it to .008, just enter the new value. You do not have to clear the register first. If you want to correct a value slightly, you must enter in the final value needed. IE: if you have .007 and want to increase it by .001 you must enter .008. Do not enter -.001

Clearing secondary offsets to Zero.
It is possible to clear all of the secondary offsets by pressing C when asked for a secondary offset number. This will set the entire table to zeros.

Individual offsets can be set to zero by pressing C when asked to enter a correction amount.
Automatic Mode

F10 Special Function, used to call up a list of special functions.

This screen will allow you to:

- **Save and recall tool offset tables.** If you are using a tooling system that allows you to remove and replace tooling exactly this function is useful to you. Also if you use the same tool set for a number of different programs. When you save your program it is possible to save the tool offsets as well. The saved offsets are put on the A: disk with the programs.

  The screen above is shown if you do not have a part program active. Offsets can not be saved or recalled. If you have a part program active the control will allow you to save and load the offsets from memory:

  - L - press L to load the offsets from memory
  - S - press S to save the offsets to the disk

- **Modify the feedrate** to be used in a program before you start running it. This eliminates the need to race to the function keys after you press the cycle start. When you are running a program for the first time you might want to lower the feedrate to only 20% by pressing F2 so that you can watch the motion of the tool before you cut material.

- **Set the number of cycles** the Automatic mode will run before stopping when you set the cycle repeat to "C". This is good for use with a barfeed. As an example you could set the machine up and tell it you need 20 pieces. The OmniTurn will make the required amount and then stop.

- If you have the infinitely variable spindle speed control this will let you tell the control what spindle speed the machine is set at. That way the control will output the speed requested in your program without having to figure any ratios out. This is covered in greater detail with the documentation on the option.


**Automatic Mode**

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X +0.00000</th>
<th>Z +0.00000</th>
<th>FEED 10.0 IPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>COMMAND</td>
<td>X +0.00000</td>
<td>Z +0.00000</td>
<td>PERCENT FEED: 100</td>
</tr>
</tbody>
</table>

---

**FILE TO BE PROCESSED:**

- **Jog**
- **Automatic**
- **F1-F10 FEED**
- **FILE IN MEMORY**
- **'O' FOR OPTION**
- **'C' FOR CYCLE**
- **PRESS 'S' FOR DATA**

---

- **Press P to activate parts counter**
  - Press R to set max. spindle speed
  - Press C to set Cycle Repeat counter
  - Press Esc to return normal operation

---

- **Press P now, it will turn the counter on in the Automatic mode**

---

- **Turn on a PARTS COUNTER**: Press P while you have this screen open. There will be no effect here. When you go to the Automatic mode main screen there will be a counter on the screen. The counter will count up each time a M30 or M02 is executed. To clear the counter come back into the F10 screen and press P again.
Automatic Mode

Feedrate override
Function Keys - Automatic Mode - Program in process

When the program is running it is possible to change the feedrates. The function keys will select a percentage of the original feedrate. $F1 = 10\%, F2 = 20\%, \ldots F10 = 100\%$. IE if you push $F1$ while the program is running the feedrate will drop to 10% of what ever you have set in the program. The feedrate and what percentage of that feedrate is being run is displayed on the automatic mode screen on the upper right corner of the screen.

<table>
<thead>
<tr>
<th>POSITION</th>
<th>X +0.00000</th>
<th>Z +0.00000</th>
<th>FEED</th>
<th>10.0 IPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>COMMAND</td>
<td>X +0.00000</td>
<td>Z +0.00000</td>
<td>PERCENT FEED:</td>
<td>100</td>
</tr>
</tbody>
</table>

PRESS CYCLE START

<table>
<thead>
<tr>
<th>Jog</th>
<th>Automatic</th>
<th>Single Block</th>
<th>Manual Data Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>F1-F10 FEED 10-100%</td>
<td>FILE IN MEMORY: DEMOPR</td>
<td>'O' FOR OPTIONAL STOP</td>
<td>'C' FOR CYCLE REPEAT</td>
</tr>
<tr>
<td></td>
<td>PRESS 'S' FOR SINGLE BLOCK</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

QUIT 2OFFSET 3EDIT 4DIR 5NEWPROG 6SEARCHTO 7PROG 8DISKOP 9SECCMP 0

If you want to change the feedrate before you press cycle start select F10. This will allow you to preload a percentage before the program is started.