TB029 (Rev1) - Thread Milling

Overview

This document describes the steps necessary to implement internal and external threading operations using a CNC control capable of true simultaneous three-axis interpolation, such as the Centroid M-Series CNC controls. In software versions v6.02 and later, a thread milling cycle was added in Intercon. This document is intended for systems that have software versions prior to v6.02.

Tools

There are two types of cutting tools to consider for threading operations. One is the single point cutter which is cheaper and more versatile than the second type, the full form cutter, which can save time when cutting standard thread sizes. When selecting a tool, be sure to consider the minimum and maximum pitch the tool can be used for as well as the diameter.

Techniques for internal threading

The basic internal threading (nuts) technique involves the following steps. It is assumed that a hole has been bored with a diameter equal to the thread minor diameter and that a right-hand thread will be milled.

- 1.A rapid move to the center point of the thread.
- 2. A feed to the depth of the thread if milling from bottom to top, or a feed to the surface height if milling from top to bottom.
- 3. Turn on cutter compensation. Use left (G41) compensation if milling from bottom to top. Use right (G42) compensation when milling from top to bottom.
- 4. Move to a starting point on the arc defined by the major diameter of the thread. The diameter of the arc is equal to the thread major diameter for internal threading. This value can be found in a reference table, such as that listed by Machinery?s Handbook. For example, an internal thread with a one-inch diameter has a major diameter of 1.0000". Any move to a point 0.5000" (half of the thread diameter) from the center point of the thread is acceptable. This move is typically a one-axis move, and should not be an arc move.
- 5. Perform a helical move. This involves a 360-degree arc in the XY plane with a Z move equal to the thread pitch and in the appropriate direction. Use G2 (CW arc) if milling from top to bottom. Use a G3 (CCW arc) if milling from bottom to top.
- 6. Repeat step (5) for the number of threads needed. This can be calculated as thread depth (in inches) time?s threads per inch.

Techniques for external threading

The basic external threading (bolts) technique involves the following steps. It is assumed that a cylinder of proper height with a diameter equal to the thread major diameter exists and that a right-hand thread will be milled.

- 1. A rapid move to a point outside the cylinder. To achieve proper cutter compensation results, make the move to a point that is at least one half of the thread tool diameter from the cylinder that will be cut.
- 2. A feed to the depth of the thread if milling from bottom to top, or a feed to the surface height if

milling from top to bottom.

- 3. Turn on cutter compensation. Use right (G42) compensation if milling from bottom to top. Use left (G41) compensation when milling from top to bottom.
- 4. Move to a starting point on the arc defined by the minor diameter of the thread. This value can be found in a reference table, such as that listed by Machinery's Handbook. For example, a one-inch diameter thread with eight threads per inch has a minor diameter of 0.8492". Any move to a point 0.4246" (half of the thread minor diameter) from the center point of the thread is acceptable. This move should not be an arc move.
- 5. Perform a helical move. This involves a 360-degree arc in the XY plane with a Z move equal to the thread pitch and appropriate direction. Use G2 (CW arc) if milling from top to bottom. Use a G3 (CCW arc) if milling from bottom to top.
- 6. Repeat step (5) for the number of threads needed. This can be calculated as thread depth (in inches) time?s threads per inch.

Additional Tips

- 1. For cases where the fractional part of thread pitch exceeds four digits, it may be necessary to calculate the absolute Z position for each thread and then round off as opposed to moving the Z axis incrementally the distance obtained from rounding the pitch value to four digits at the start. Rounding the pitch from the start can cause errors that increase as the thread depth and number of threads per inch increases.
- 2. When using Centroid Shop Floor Programming (SFP), 360-degree arcs cannot be entered as an EP&R (end point and radius) type. Use the center point and end point type of arc (CP&EP).
- 3. When milling threads from top to bottom, it may be possible to use Intercon's Repeat-to-Depth Sub Program after the first helical is performed.
- 4. For left-hand threads, reverse all the cutter compensation and arc directions.

Document History

Rev1 Created on ***1999-08-23***