A DEVELOPMENT OF CNC SOFTWARE FOR A SUPER SMALL PRECISION LATHE

Daisuke Iijima*, Akira Hayashi*, and Masashi Yamanaka**
*Nano Corporation, Tokyo, Japan
**Tohoku University, Sendai, Japan
*URL: http://www.nanowave.co.jp/

Introduction
Micro Turning System 4 (MTS4) is a super small CNC (computer numerical control) precision lathe that has a machine base size of 300 x 200 mm (A4 paper size). The MTS4 was developed for exclusively machining micro-parts as a substitution of conventional precision lathes that were too big compared to the size of parts to be machined. When the MTS was developed, one of the technical objectives was to improve the operationality in software of the conventional CNC machine tools. The software was developed so that the operation can be processed visually. As a result, the users could machine micro-parts for a few hours even if they were beginners. Figure 1 shows the picture of MTS4.

Background
Though general CNC machine tools can correctly machine various forms such as circular arc or complex shapes, it is one of the problems that it takes a lot of time for beginners to master the use of the machine tools because the operating instructions are difficult than manual machine tools. And it is also other problem that it needs much human energy and time to design tool paths that are used for manipulation of the machine tools for users who cannot introduce CAM (computer-aided manufacturing) system that is usually high cost. The smaller the target machining forms are, the more frequently human errors such as input error of small decimal number will easy to occur. Additionally, the fact remains that users are restricted to arbitrarily
modifying tool paths because the software of the CNC machine tools inclusive of the CAM software is usually nondisclosure.

Policy of Software Design
It is one of the solutions to disclose the software so as to let the users contrive the operating instructions in their own. Therefore, in order to make a source code of CNC software for the MTS open, the software was developed based on a personal computer (PC) and the Microsoft Visual C++ was chosen as the development environment. For easy addressing micro order dimensions of the parts, the software was designed as a visual-based input method of machining forms mainly using a mouse device. Especially, a simple computer-aided design (CAD) and CAM system was developed. The CAD/CAM system can automatically generate a tool path from a machining form, drawn by mouse-click over the displayed figure of the work on the PC monitor was implemented. After drawing of the machining form, each mouse-clicked point can be confirmed as coordinate data on the scale of the work. If the coordinate data is undesirable, the user can retype with respect to each coordinate data. Almost in case of retyping the data, the user directly input numerical data. The CAD/CAM system can also generate commonly used G-code program from the drawn form. The specifications of MTS software are listed on the Table 1.

<table>
<thead>
<tr>
<th>Specifications of MTS Software.</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>NC type</td>
<td>Personal Computer based NC</td>
</tr>
<tr>
<td>Development Environment of Software</td>
<td>Microsoft Visual C++</td>
</tr>
<tr>
<td>Operation Panel</td>
<td>Display on a PC monitor</td>
</tr>
<tr>
<td>Data Input Station</td>
<td>Mouse Device (main use)</td>
</tr>
<tr>
<td>Tool Path Generation</td>
<td>Automatic (by the simple CAD/CAM system) / Hand Coding of NC program (G-code)</td>
</tr>
<tr>
<td>Other Functions</td>
<td>- External / Internal / End Face Machining</td>
</tr>
<tr>
<td></td>
<td>- Machining Simulation</td>
</tr>
<tr>
<td></td>
<td>- Loading / Saving G-code Program</td>
</tr>
<tr>
<td></td>
<td>- Screw Cut</td>
</tr>
<tr>
<td></td>
<td>- Direct NC (option)</td>
</tr>
</tbody>
</table>

Operating Procedure
1) Input of Tool Offset Data
If a user newly reset tools on tool holders, tool offset data of the reset tools are needed to input into the software by using manual control panel. Figure 2 shows the manual control panel.

- If the X / Z table is not on each origin point, make the table return to the origin (click the “Zero return” button).

- Make the spindle revolve toward processable direction by the tools (click the “CW/CCW” button).
- Make the X and Z tables feed till the tool touches the work (use the X/Z arrow button).
- When the tool touches the work, input the moving distance from origin to current point (click the “X/Z-axis offset” button only).
- Input the work size (click the “Parameter” button, then enter the work size into the text box of the parameter dialogue).

After this procedure, the software calculates the distance from the machine origin to the program origin as tool offset data. If necessary, the user can save the data as a file.

Figure 2: Manual control panel.

Figure 3: Input of the work size.

2) Displaying Work Outline

Figure 3 shows the window to input work size in the automatic control panel.
- Start up the automatic control panel by clicking the “Automatic” button.
- Choose machining surface from external / internal / end face.
- When the “Update” button is clicked, an outline of the work is displayed on the PC monitor.

3) Drawing Machining Form

Tool paths can be generated by way of drawing a machining form on the work outline by mouse.
- First, click the first point outside of the work area.
- If the user wants to draw line segment, click the end point of the line inside of the work area.
- If the user wants to draw arc segment, click the end point of the arc, then click the point which is in the arc orbit inside of the work area with the “Ctrl” key of keyboard down.
- Draw arbitrary machining form by connecting line and arc segments.
- Finally, click the last point outside of the work area again.

Figure 4 shows an example of drawing form.

![Figure 4: Drawing machining form consists of line and arc segments by mouse click.](image1)

![Figure 5: Tool path is automatically calculated and displayed.](image2)

4) Tool Path Generation
When the “Tool path” button is clicked, a tool path that is based on G71 and G70 is automatically calculated from the mouse-clicked points. The tool path is also displayed on the PC monitor as shown in Figure 5. Furthermore, a G-code program is also created in conjunction with the tool path generation.

5) Modification of Data
If it is necessary to modify the data inputted by mouse click, the user can retype the numerical data from keyboard every block. Similarly, machining parameters can be also changed. The machining parameters are followings, “Spindle revolution”, “Depth of cut at rough cut”, “Depth of cut at final cut”, “Feed at rough cut”, and “Feed at final cut”. The machining parameters are enough to use even if they are default.

6) Machining
If the NC program is forwarded to the control box and the “Start” button is clicked, a sequence of machining is done.

Result and Future Work
The users got to machine micro-parts easily through an instruction for a few hours by using the developed visual-based CNC software even if they were beginners. Though one tool is applied to one tool path at the present time, it is planed to allocate multiple tools appropriately to one tool path so that more complex forms can be machined more easily in future.