5-24-2019


Daniel M. Small
Portland State University

Let us know how access to this document benefits you.
Follow this and additional works at: https://pdxscholar.library.pdx.edu/honorstheses

Recommended Citation

10.15760/honors.734

by

Daniel Small

An undergraduate honors thesis submitted in partial fulfillment of the requirements for the degree of Bachelor of Science in University Honors and Mechanical Engineering

Thesis Adviser

Dr. Faryar Etesami

Portland State University

2019
Abstract:
This paper presents a method and analysis for determining the geometric accuracy of CNC milling machines. The method measures the accuracy of five basic geometric values: straightness, circularity, size, angularity, and position. Tolerance prediction models are found using statistical analysis. The tolerance prediction models are used to find more complex tolerance values. The results of the paper will allow manufacturers to measure the actual tolerance capabilities of their machines and no longer rely on guess work. The method and analysis can also be applied to other machining processes.

Introduction:

Until the twentieth century, the relationship between the designer and manufacturer has been very close. The designer would explain their design and how parts should fit together. The manufacturer, usually a skilled craftsman, would make the parts and refine them until they fit. Today however, designers and manufacturers are spread out all over the world. Designers need to be able to clearly and effectively convey design intent to a manufacturer even if they do not speak the same verbal language. On the other hand, manufacturers also need to tell the designer the capabilities of their manufacturing processes.

GDT is great for designers to communicate with manufacturers. GDT communicates how parts will be used by defining simple, geometric volumes [1, 3].

Figure 1: This is an example of traditional coordinate tolerancing.
However, GDT is currently a one-way conversation. Most manufacturers cannot tell the designers what tolerance levels they can produce for each GDT feature. A manufacturer might provide a tolerance standard based on traditional coordinate dimensioning system for some manufacturing processes. After receiving the designs, the manufacturer then quotes a price based on how much time and resources it will take to make the part. If the manufacturer thinks that they will need advance processes to produce the requested tolerance, they will charge more money per part.

One of the most common manufacturing processes is CNC milling. CNC milling is a subtractive manufacturing process. The manufacturer starts with a block of material and removes material with the mill to form the part. CNC milling has a good balance between cost and accuracy. Unfortunately, manufactures usually determine their machining tolerances based on experience rather than testing.

The goal of this research is to determine the GDT values that a CNC mill can produce. If the GDT values for individual machines can be determined, then manufacturers can be confident in producing a higher level of accuracy. A manufacturer will charge much less if a part can be manufactured perfectly every time on a single machine. This will remove the need for large price safety factors by reducing the uncertainty of manufacturing. A designer could also avoid extra manufacturing processes by designing within the tolerances of a CNC mill.

### Methods:

1. **Part Design**

   A machined part was designed to have multiple straight edges, circles, features of size, and angles. The stock part is a 6061-T6 extruded aluminum block. The dimensions are 0.75x4x9 inches. The part was also designed to be made with only one end mill size. Multiple tools would increase time and cost to produce the part. To save more time, the part only requires one clamping position.

2. **Machining**

   The part model was then imported into Mastercam to generate the G-codes. Dynamic 2D milling tool path was used for all features in the part. The part was manufactured in the Haas TM-1P CNC milling machine in the PSU machine shop. No finishing cuts were used for the tool path. A 3/8-inch end mill was used to cut the
features. Spindle speed was set to 5000 RPM and very light cuts were used for each layer. Each depth of cut was a max of 0.075 inches. This made the total machining time over 2 hours. However, the amount of tool wear was extremely minimal. No post processing was performed on the machined part.

3. Measuring

The part dimensions were measured on a Tesa Micro-hite coordinate measurement machine (CMM). Eight features of straightness, circularity, size, angularity and position were measured. For location measurements, sides 10 and 6 were chosen as the datums.

Results:

Table 1: This table shows the tolerance statistics for each measured tolerance.

<table>
<thead>
<tr>
<th>Tolerance (mm)</th>
<th>Average</th>
<th>Max</th>
<th>Min</th>
<th>Standard Deviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straightness</td>
<td>0.005</td>
<td>0.008</td>
<td>0.003</td>
<td>0.002</td>
</tr>
<tr>
<td>Circularity</td>
<td>0.006</td>
<td>0.008</td>
<td>0.003</td>
<td>0.002</td>
</tr>
<tr>
<td>Size</td>
<td>-0.012</td>
<td>-0.007</td>
<td>-0.021</td>
<td>0.005</td>
</tr>
<tr>
<td>Orientation</td>
<td>0.004</td>
<td>0.013</td>
<td>-0.003</td>
<td>0.005</td>
</tr>
<tr>
<td>Position</td>
<td>0.030</td>
<td>0.056</td>
<td>0.017</td>
<td>0.012</td>
</tr>
</tbody>
</table>

Straightness:

![Straightness graph](image)

Figure 5: This graph shows the measured straightness tolerance and best fit line.

The CNC mill produced very straight lines with precise tolerance. The average straightness tolerance was only 5 microns. The tolerance increased as the size of the feature increased. A longer measured line had a larger tolerance. The R-squared value for the best fit line was 0.4242.

Circularity:

![Circularity graph](image)

Figure 6: This graph shows the measured circularity tolerance and best fit line.

The circles had very similar tolerance characteristics as the lines. They were very precise and the tolerance increased as the size of the circles increased. The correlation
between the size and tolerance was much stronger as the R-squared value was 0.8367.

Size:

The FOS that were measured in this research were circles, rails, and slots. The tolerance for FOS were about twice as large as the straightness and circularity tolerance. The average tolerance for FOS was -12 microns. No correlation was found between the size of a feature and the tolerance. The R-squared value for the best fit line was 0.0132.

Angularity:

Angularity was very similar to FOS in tolerance. While the average was closer to zero than the average of FOS, the standard deviation was the same. Once again, there did not seem to be any correlation between the angle and the tolerance value. The R-squared value for the best fit line was 0.0361.

Position:

Position had the largest tolerance value out of all the measured tolerances. It had both the highest average and standard deviation. The amount of tolerance did correlate to the distance from the datums. The R-squared value was 0.5765 for the best fit line.
**Discussion:**

**Toolpath Distance:**

The straightness and circularity seemed to be very similar in tolerance average, range, and standard deviation. However, the slope of the models for straightness and circularity were different. When the circularity characteristic length was converted from the diameter of the circle to the circumference of the circle, the slopes lined up very well. The common factor between the circumference and the line length is the toolpath distance traveled by the end mill.

Because the toolpath distance seems to equally affect straightness and circularity, the data was combined to form a single model for straightness and circularity. This increases the sample size of the model and reduces uncertainty. It also simplifies the equations for finding the advanced GDT values.

**Tolerance Prediction Model Statistics:**

To find the upper and lower tolerance limit predictions for each measured tolerance, the following equations were used [1]:

\[ T = \pm t_{n-1} \sqrt{\frac{1}{n}} \]

\[ T = \pm t_{n-2} \sqrt{1 + h_i} \]

\[ h_i = \frac{1}{n} \sum (X_i - \overline{X})^2 \]

Eq. 1 finds the margin of error for the tolerance confidence interval of an independent variable. The FOS and angularity tolerances are both independent. Eq. 2 finds the margin of error for the tolerance confidence interval of a dependent variable. Straightness and circularity are both dependent on the toolpath length, while the position is dependent on the distance from the datums.

Eq. 3 accounts for potential uncertainty in the model if the predictor value is outside of the measured value range. The margin of error is then added to the model for the upper tolerance prediction and subtracted from the model for the lower tolerance prediction. For straightness, circularity and position, the lower tolerance predation is not useful for GDT and therefore is not calculated.
Tolerance Predictions:

All predictions were calculated at a 99.9% confidence level. This means that 99.9% of all the features produced on the Haas TM-1P CNC milling machine in the PSU machine shop will be more precise than the tolerance prediction lines.

For example, using the graph, the straightness tolerance of a line that is 40 mm long would be under 9.5 microns. For a circle, one would multiply the diameter of the circle by \( \pi \) to find the circumference of the circle. A circle with a diameter of 20 mm would have a circularity tolerance under 10.5 microns.

Using all of the tolerance prediction models, a GDT callout calculator was created based on previous models used for finding the GDT values for 3D printers [1]. The models were simplified slightly because straightness and circularity share the same tolerance prediction model. The calculator allows a designer to know the tolerance value that can be produced for a given GDT callout and feature dimensions on a specific endmill. A machine shop could send this GDT calculator, that is calibrated to their machines to designers to indicate the tolerance level that they can producing.
Table 2: This table is an example of the GDT calculator. Dimensions are in millimeters and angles are in degrees.

<table>
<thead>
<tr>
<th>GDT Callouts Calculator</th>
<th>User Input 1</th>
<th>User Input 2</th>
<th>User Input 3</th>
<th>Tolerance Upper</th>
<th>Tolerance Lower</th>
</tr>
</thead>
<tbody>
<tr>
<td>Straightness (line length)</td>
<td>40</td>
<td></td>
<td></td>
<td>0.009503243</td>
<td></td>
</tr>
<tr>
<td>Circularity (diameter)</td>
<td>20</td>
<td></td>
<td></td>
<td>0.010588001</td>
<td></td>
</tr>
<tr>
<td>Flatness (diagonal length)</td>
<td>80</td>
<td></td>
<td></td>
<td>0.011482635</td>
<td></td>
</tr>
<tr>
<td>Cylindricity (diameter, axis depth)</td>
<td>10</td>
<td>30</td>
<td></td>
<td>0.018191009</td>
<td>-0.112559091</td>
</tr>
<tr>
<td>Feature of Size (feature size)</td>
<td>20</td>
<td></td>
<td></td>
<td>0.088359091</td>
<td>-0.112559091</td>
</tr>
<tr>
<td>Perpendicularity-Axes (axis length)</td>
<td>50</td>
<td></td>
<td></td>
<td>0.012127172</td>
<td></td>
</tr>
<tr>
<td>Perpendicularity-Surface (projection length)</td>
<td>75</td>
<td></td>
<td></td>
<td>0.013650347</td>
<td></td>
</tr>
<tr>
<td>Parallelism-Axes (axis length)</td>
<td>30</td>
<td></td>
<td></td>
<td>0.011082503</td>
<td></td>
</tr>
<tr>
<td>Parallelism-Surface (diagonal length)</td>
<td>60</td>
<td></td>
<td></td>
<td>0.012771805</td>
<td></td>
</tr>
<tr>
<td>Angularity-Surface (diagonal length, projected length, angle)</td>
<td>20</td>
<td>25</td>
<td>45</td>
<td>0.010595159</td>
<td>-0.112059091</td>
</tr>
<tr>
<td>Profile of a Line (largest profile length)</td>
<td>25</td>
<td></td>
<td></td>
<td>0.088859091</td>
<td>-0.112059091</td>
</tr>
<tr>
<td>Profile of a Line with Datum (largest profile length, distance from datum)</td>
<td>10</td>
<td>20</td>
<td></td>
<td>0.172316103</td>
<td>-0.028602078</td>
</tr>
<tr>
<td>Profile of a Surface (largest profile length, profile depth)</td>
<td>25</td>
<td>30</td>
<td></td>
<td>0.097924246</td>
<td>-0.102993935</td>
</tr>
<tr>
<td>Profile of a Surface with Datum (largest profile length, distance from datum, profile depth)</td>
<td>10</td>
<td>20</td>
<td>30</td>
<td>0.181381259</td>
<td>-0.019536923</td>
</tr>
<tr>
<td>Runout (diameter, distance from datum)</td>
<td>40</td>
<td>20</td>
<td></td>
<td>0.099132121</td>
<td></td>
</tr>
<tr>
<td>Total Runout (diameter, length from end to end, distance to datum)</td>
<td>35</td>
<td>60</td>
<td>100</td>
<td>0.120909357</td>
<td></td>
</tr>
<tr>
<td>True Position (distance to datum)</td>
<td>42</td>
<td></td>
<td></td>
<td>0.0871312</td>
<td></td>
</tr>
</tbody>
</table>

Figure 14: This is a graph that shows the upper and lower tolerance prediction model for features of size.
Conclusion:

In this paper, the GDT tolerance capabilities, that a specific end mill machine can produce, have been determined. Using the methods and analysis presented in this paper, anyone can find the GDT values for their own CNC endmill. The process has already been applied to other manufacturing processes such as 3D printing [1, 2]. Future research could focus on verifying the models used to calculate the more complex GDT callouts. This method should also be applied to lathe machining, laser cutting, broaching, grinding boring, and honing.

Citations:

Citation 1
F. Etesami and T. Griffin, 2013
“Characterizing the Accuracy of FDM Rapid Prototyping Machines for Machine Design Applications”, *Syst. Des.*, vol. 12, p. V012T13A060

Citation 2

Citation 3
P. Witherell, G. Herron, and G. Ameta 2016