

Defect Identification in CNC Manufactured Products and their Control Measure

Chirag Kulkarni¹, Dr.Girish Bagale²

U.G. Student, MBA Tech Programme, NMIMS's Mukesh Patel School of Technology Management and Engineering,
Mumbai, Maharashtra, India¹

Assistant Professor, NMIMS's Mukesh Patel School of Technology Management and Engineering, Mumbai,
Maharashtra, India²

ABSTRACT: The main objective of this project is to identify the sources of defects in final products and to understand ways to eliminate those defects. This requires understanding the processes involved in manufacturing of components and how parameters of those processes affect the final product.

To accomplish the goal of the project, a step by step understanding of the processes approach is used. It starts right from studying the diagram of the component to be manufactured to the final inspection stages.

While checking the processes defects have to be identified that are present in the component that being manufactured and implement the necessary steps for eliminating those defects.

KEYWORDS: Workpiece, Tool Nose Radius, Feed, Chip

I. INTRODUCTION

Problem statement: To reduce defects in components manufactured by identifying causes of the defects.

The objective of the project is to understand and study the sources of error in the manufacturing in CNC manufacturing and to find methods to eliminate those errors.

II. METHODOLOGY ADPOTED

After thoroughly observing and understanding the machining process of the components over the course of internship, several common error points were noted down and analysed. These error points were then scrutinized. In order to understand the root causes of these errors, research and practical experience was used as a basis for coming up with solutions.

III. MACHINING DEFECTS

The following are the machining defects which are checked during the final inspection:

1. All gauges such as Air Pressure Gauge, Snap Gauge etc should be OK
2. Receiver gauges should be ok
3. Thread should be OK by visual and thread gauge inspection
4. Tool mark/chattering not allowed
5. Chamfer out not allowed
6. Tap/tool broken not allowed
7. Rough lining/rough finishing not allowed
8. Inside Diameter Under Size, Over Size, oval not allowed
9. Outside Diameter Under Size, Over Size, oval not allowed
10. Dimension Under Size, Over Size not allowed
11. Operation miss not allowed

IV.SOURCES OF DEFECTS AND THEIR CONTROL MEASURES

1.ERROR IN TOOL ALIGNMENT

Severity:Can affect entire batch

IDENTIFICATION: Dimensions beyond tolerance limits; incorrect shape/geometry of the machined component
Several problems arise in lathes when the chuck/collet is not properly aligned with the tailstock or tool holder. These problems contribute to a large number of errors produced as a result in the workpiece geometry (dimensions, tolerances).

Proper alignment of spindle, workpiece and the tool increases efficiency of a machine.

There are two types of misalignment:

- A. Centreline misalignment
- B. Angular misalignment

A. Centreline Misalignment

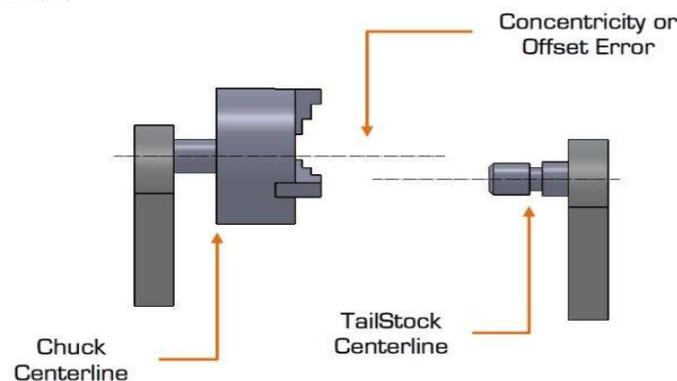


Figure 1: Centreline misalignment

Figure 1 shows centreline misalignment of chuck and tailstock. When the centre lines of spindle, tool holder or chuck are not aligned (displaced) from each other, concentricity misalignment occurs. Concentricity misalignment happens in the horizontal or vertical axes. The error due to this offset remains fixed and maintains a constant error value.

B. Angular Misalignment

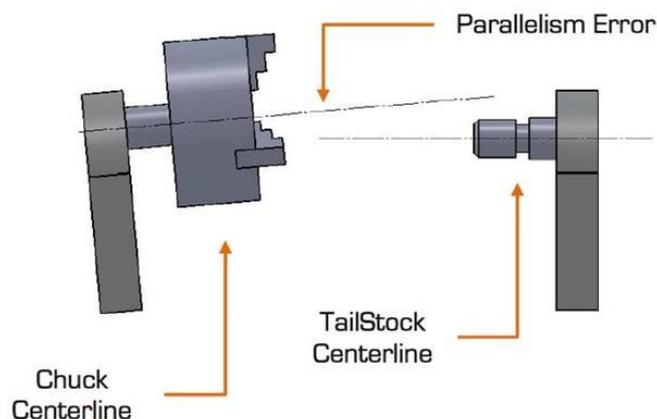


Figure 2: Angular misalignment

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 7, July 2017

Figure 2 shows when the axes are not parallel aligned. When the axes of rotation and axis of tool movement are not parallel to each other, parallelism misalignment occurs. The error is measured in angular value or displacement error. The error grows with distance, hence the further you move from the chuck, the greater the angular error becomes. Corrective measure: Properly mount the tool, and check the alignment of the chuck, tailstock and tool before starting production.

2.ERROR DUE TO IMPROPER CLAMPING

Severity:Affects the particular piece

IDENTIFICATION: Poor surface finish, damage to tool holder and insert, dimensions not in tolerance limit

Improper clamping of workpiece gives rise to increased vibrations between the workpiece and the cutting tool. As a result, the surface quality of the workpiece is decreased, dimensions may go beyond the indicated tolerances and the tool holder and insert might get damaged. Therefore, proper clamping of workpiece is important for high efficiency.

Corrective Measures:

- Ensure the hydraulic pump is running. If not, restart machine.
- Check whether the indicated/required hydraulic pressure is being imparted on the workpiece by the chuck.
- Check whether the foot switch is working and the workpiece is being clamped properly.
- Check whether the workpiece is aligned correctly with the rotating axis and the cutting tool axis.

3. ERROR DUE TO CHATTER

Severity:Chatter affects the entire batch of the workpieces produced. It happens because of two reasons:

A. Poor set up

B. Improper insert installation

IDENTIFICATION: Damage to tool, dimensions beyond tolerance limit, breakdown of machine

Chatter is a resonant phenomenon where the machine or workpiece vibrates. Chatter is responsible for the loud noise generated in machines. It also causes unwanted violent vibrations in machines. Chatter reduces tool life. It decreases the accuracy and life of machine. There are two types of chatter:

- Tool chatter
- Workpiece chatter

Tool ChatterIn tool chatter, the machine and tool vibrates and transfer the vibrations onto the workpiece. .

Workpiece Chatter In workpiece chatter the wall of the workpiece is vibrating. It is common in components with thin walls. It causes more severe problems as compared to tool chatter.

Increasing or decreasing how much the tool sticks out can reduce the vibrations by changing the frequency. Changing the deflection of tool reduces the probability of exciting chatter.

Corrective Measures:

- Reduce feed rate
- Select more positive chip breakers
- Increase feed rate
- Use smaller nose tool radius tool (TNR). A smaller TNR reduces cutting pressures. Always use a TNR smaller than the radius being cut.

4.ERROR DUE TO UNACCEPTABLE CHIP CONTROL

Severity:The long stringy chips produced when turning are extremely problematic in operation and production. Unbroken chips warp around the workpiece. Therefore, the operator has to spend more time in removing chips increasing the cycle time.

IDENTIFICATION: Chips attached to tool, unwanted marks of component's surface

Chip edges are generally very hot and sharp, hence pose a risk to the operator. Furthermore, chips can drag along the parts, causing damage to the workpiece. Moreover, the tool can also break if the chip is stuck between the cutting edge and the workpiece as the tool keeps on cutting the chip.

Solutions:

- Increase feed rate

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 7, July 2017

- Use more positive chip breaker
- Select smaller nose radius insert

4.1 Make Use of Insert Technology(Tips on Breaking Chips and Controlling Burrs when Machining Automotive Transmission Components, n.d.)

Most turning inserts today have a geometry pressed into the rake face to control chips. The edge line or primary land should be narrow. For one-pass machining where there are variable depths of cut, it is common to use geometries with lands that vary from 0.004 inch to 0.010 inch. The key here is to make sure your feed rate exceeds the width of the land. When your feed rate exceeds the width of the land it makes use of the rake angle and chip-forming ability of the insert geometry. The rake angle shears the material rather than the land to rub off the material. When machining with a feed rate that is smaller than the land width, heat is developed and has a negative effect on tool life. Optimizing feed rate and land width also makes use of the chip-forming ability of the geometry. The various angles and “bumps” that make up the geometry are designed to drive the chip into obstructions that can mechanically break the chip. For lower feed applications, these chip formers, or bumps, need to be close to the edge line, especially at the insert nose radius. The hardest chip breaking, which creates the shortest chips, occurs just above the nose radius for the cutting depth. So, adjusting the cutting depth and the nose radius is essential to optimize the cut. Using a smaller nose radius than usual, with a wiper, can be a successful, though less conventional, method.

5.ERROR DUE TO POOR SURFACE FINISH

Severity: Poor surface finish is highly undesirable factor, especially in production of work pieces with high grade surface finish requirement.

IDENTIFICATION: Poor surface finish

Causes:

- Feed rate too great for nose radius
- Low cutting speed
- Poor shearing action

Solution:

- Reduce feed rate
- Use larger nose radius insert
- Increase cutting speed
- Select more positive chipbreaker
- Flood cutting zone with coolant

6.ERROR DUE TO INCORRECT INSERT SELECTION(Schneider, 2010)

Insert plays a very important role in metal cutting operations. Selection of the correct geometry, nose radius and tolerances of the insert is important to get high efficiency in production.

There are two parameters to keep in mind while choosing an insert:

1. Insert geometry
2. Nose radius

IDENTIFICATION: Vibration in cutting operation

6.1 Insert geometry:

The geometry of the insert has a decisive influence on the cutting process. A positive insert has a positive rake angle. The insert's edge angle and clearance angle together will equal less than 90 degrees. A positive rake angle means a lower tangential cutting force. However, a positive rake angle is obtained at the cost of the clearance angle or the edge angle. If the clearance angle is small there is a risk of abrasion between the tool and workpiece, and friction can give rise to vibration. In those cases where the rake angle is large and the edge angle is small, a sharper cutting edge is obtained. The sharp cutting edge penetrates the material more easily, but it is also more easily changed or damaged by edge or other uneven wear.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 7, July 2017

Edge wear means that the geometry of the insert is changed, resulting in a reduction in the clearance angle. Therefore, with finish machining it is the required surface finish of the workpiece that determines when the insert must be changed. Generally, the edge wear should be between 0.004 and 0.012 in. for finishing and between 0.012 and 0.040 in. for rough machining.

6.2 Nose radius:

The nose radius of the insert also affects the distribution of cutting forces. The greater the nose radius, the greater the radial and tangential cutting force- and the emergence of vibration. However, this is not the case with radial cutting forces. The deflection of the tool in a radial direction is instead affected by the relationship between the cutting depth and the size of the nose radius. If the cutting depth is smaller than the nose radius, the radial cutting forces will increase with increased cutting depth. If the cutting depth is equal to or greater than the size of the nose radius, the radial deflection will be determined by the lead angle. Thus, it's a good idea to choose a nose radius that is somewhat smaller than the cutting depth. In this way the radial cutting forces can be kept to a minimum, while utilizing the advantages of the largest possible nose radius, leading to a stronger cutting edge, better surface finish and more even pressure on the cutting edge.

7 ERROR DUE TO BURNING OF MATERIAL

Severity: Burning of material of workpiece is detrimental to the efficiency of operation of the machine. Depending on how soon the defect is identified, this error can also affect the entire batch of production in a particular machine.

When a spinning tool is moving linearly through material, its flute allows material (chip) to be pushed inside of the tool which is then immediately sliced away. The chip carries the heat of the tool out with it. The thicker the chip that is sliced away the more heat the chip can be carried away from the bit. Therefore, it is important to control the chip flow. This can be done in a number of ways listed below.

IDENTIFICATION and Corrective measures:

Table 1: Problems, causes and solutions when material burns

PROBLEM	CAUSE	SOLUTION
The material burns on outside edge.	The tool is blunt.	Replace with sharp tool.
The material burns on outside edge.	Feed speed too slow.	Increase feed speed.
The material burns on internal corners of mould.	Feed speed too slow in corners.	Increase feed speed at corners.
The material burns on internal corners of mould.	The tool is blunt.	Replace with sharp tool.
The tool is burnt.	Feed speed too slow for tool being used.	Increase feed speed to tool specifications.
The tool is burnt.	The tool is blunt.	Replace with sharp tool.
Burnt edges of holes.	The tool is blunt.	Replace with sharp tool.
Burnt edges of holes.	Tool put in wrong spindle (Left hand drill bit in right hand spindle or vice versa).	Insert tool in correct spindle.
Burnt edges of holes.	Incorrect spindle speed (RPM).	Change spindle speed.
Raised mark along edge of moulding.	The tool has a gap on the cutting edge.	Replace the tool.
Rough edge, cutter marks visible.	Machine is travelling too fast (feed speed).	Reduce the feed speed.

Table 1 gives the probable causes of errors due to material burn and their solutions. These can be used as general corrective methods for the mentioned problems.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 7, July 2017

8.ERROR DUE TO WRONG TOOL OFFSET

Severity: Wrong tool offset often leads to workpieces having dimensions well beyond the tolerance limits. Entire batches of production can go wasted due to wrong tool offset.

IDENTIFICATION: Dimensions out of tolerance limits

Tool Offset

The machine must be informed the amount the machine zero and component zero point. The component of offset between zero points is the starting point of the machining operation on the work piece at X, Y and Z axis.

Tool Nose/Cutter Nose Radius Compensation(qhunt, n.d.)

Without compensation, the imaginary tool tip follows the programmed path resulting in undercutting (the shaded area), i.e., an over-sized job. For compensation, the centre of the tool nose (C) is made to follow the programmed path with tool nose radius compensation (right) command active.

In milling operation, the part program is developed for the cutter path with reference to the centre of the tool rather than the point on the periphery where the actual cutting takes place. At the time of writing a part program, a cutter of suitable diameter is selected and program is developed for centre line of the cutter. But when actual machining is done, if a cutter of smaller diameter is used, it will result in a larger work piece and if a cutter with larger diameter is used, it will result in a smaller work piece.

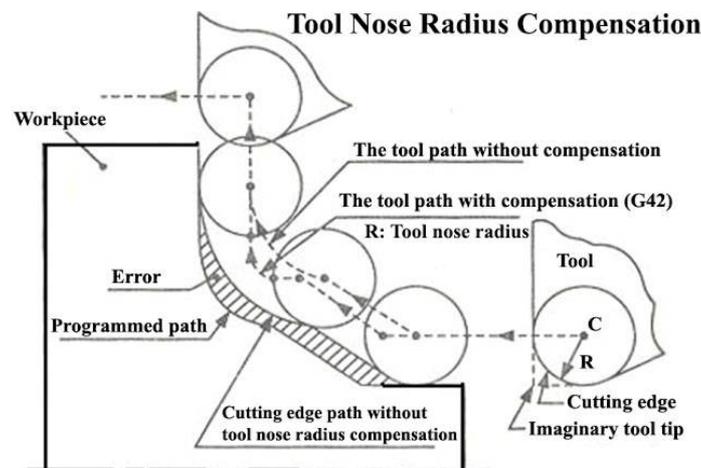


Figure 3: Nose Radius Compensation

Figure 3 shows how the cutting nose of the tool moves while cutting metal and how it can cause errors.

Corrective Measure:

- Tool offset compensation should be properly taken and with improved techniques.
- Nose radius compensation with the correct nose radius value should be input.

9.ERROR DUE TO INCORRECT PROGRAM(Lynch, n.d.)

Severity: A wrong program is the most catastrophic of errors as it is difficult for operators to identify. If not corrected soon, entire line of production can be wasted

Typical Mistakes

- **Syntax Mistakes:** A control will not even be able to execute the command with a syntax mistake. For example, the programmer may have meant to type a G code as G01, but by mistake typed G101. Most CNC controls do not even have a G101 command. They are easy to diagnose because it is likely that as soon as the programmer sees the command generating the alarm, the mistake will be very obvious.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 7, July 2017

- **Motion Mistakes:** They are caused by incorrect coordinate calculations. Other mistakes in this category include forgetting to instate or cancel tool offsets, reversing clockwise and counter clockwise commands, and improper mode selection for incremental versus absolute. In many cases, it means the program is telling the control to, at best, not machine properly, or at worst, crash the tool into the workpiece, workholding device, or machine.
- **Set-up Mistakes:** The operator has to make measurements such calculating the program zero, the tool offset depending on the tool length and entering these values in the machine. If these numbers are incorrectly measured or if mistakes are made during their entry, the control will not truly know where the program zero point is. All motions the program commands will be incorrect

VI. CONCLUSION

The nine sources of errors were identified and scrutinized and solutions were found to eliminate them. These identified sources of errors can be used as a precautionary measures by organizations. This will help organizations to reduce the number of defective components manufactured and tackle any problems arising due to these sources of errors easily.

REFERENCES

- [1] Author Unknown, RMIT Australia, Year of publication not mentioned
- [2] George Schneider, Cutting Tool Applications Chapter 10: Boring Operations and Machines, American Machinist, 2010
- [3] Turning Troubleshooting, Carbide Depot (<http://www.carbidedepot.com/formulas-turning-troubleshoot.htm#workpiecechatter>)
- [4] Dasarathi GV, 23. Chip Breaking in CNC Turning, CNC etc, Cadem Technologies, Year of publication not mentioned. (<http://cadem.com/cncetc/cnc-turning-chip-breaking/>)
- [5] SandvikCoromant, Tips on Breaking Chips and Controlling Burrs when Machining Automotive Transmission Components, Modern Machine Shop, 2015
(<http://www.mmsonline.com/articles/tips-on-breaking-chips-and-controlling-burrs-when-machining-automotive-transmission-components>)
- [6] Author not mentioned, Dealing With Chatter When Turning or Milling, CNC Cookbook (Dealing With Chatter When Turning or Milling)
- [7] Author not mentioned, Helpful Alignment Tips for Machine Shops, Modern Machine Shop, 2013
(<http://www.mmsonline.com/articles/helpful-alignment-tips-for-machine-shops>)
- [8] Qhunt Media, Zero Offset and Tool Offset on CNC Machine, Qhunt, 2015.
(<http://www.qhunt.com/2015/10/zero-offset-and-tool-offset-on-cnc.html>)
- [9] Qhunt Media, Zero Offset and Tool Offset on CNC Machine, Qhunt, 2015.
(<http://www.qhunt.com/2015/10/zero-offset-and-tool-offset-on-cnc.html>)