Five Axis Machining of Stamping Dies

By
Mark I. Zeni
B.A.Sc. Mechanical Engineering
University of Waterloo
Waterloo, Canada, 1990

Submitted to the Sloan School of Management and the Department of Mechanical Engineering in Partial Fulfillment of the Requirements for the Degrees of

Master of Science in Mechanical Engineering
and
Master of Science in Management

at the
Massachusetts Institute of Technology
May 1995

Signature of Author ______________________________
Department of Mechanical Engineering and Sloan School of Management

Certified by ______________________________
David E. Hardt
Professor of Mechanical Engineering and Leaders for Manufacturing Co-Director
Department of Mechanical Engineering
Thesis Advisor

Certified by ______________________________
Stephen C. Graves
Professor of Management Science and Leaders for Manufacturing Co-Director
Sloan School of Management
Thesis Advisor

Accepted by ______________________________
Ain Sonin
Chairman, Department Graduate Committee
Department of Mechanical Engineering

Five Axis Machining of Stamping Dies

By
Mark I. Zeni

Submitted to the Sloan School of Management and the Department of Mechanical Engineering on May 12, 1995 in partial fulfillment of the requirements for the Degrees of
Master of Science in Mechanical Engineering and
Master of Science in Management

Abstract

Die development is one of the most expensive and historically the longest lead time activity of a new car program. Auto makers incessantly look to reduce both the cost and time to design and build automotive stamping dies. Five axis machining is a technology that offers the potential of improving both these metrics.

This thesis analyzes and quantifies the benefits of five axis machining in the manufacture of automotive stamping dies. It will show how the process flexibility offered by five axis machining can be used to affect savings beyond the machining center. The process and equipment requirements to take full advantage of five axis machining at a major U.S. auto maker are explained.

For this major U.S. auto maker, there is significant financial benefit to pursuing five axis machining. The technology is only applicable to certain types of dies and certain die building processes, however. Although there are significant benefits, achieving those benefits require more than just 'plugging in' the technology. Organizational, process, and cultural factors must be taken into account and are considered.

Thesis Supervisors: Stephen C. Graves
Co-Director, Leaders for Manufacturing program
Professor of Management Science
Sloan School of Management

David E. Hardt
Co-Director, Leaders for Manufacturing program
Professor of Mechanical Engineering
Department of Mechanical Engineering

Acknowledgments

The author wishes to acknowledge the Leaders for Manufacturing Program for its support of this work.

I also wish to thank:

- The many people at the major U.S. auto maker who supported this study, particularly my managers and co-workers, the central engineering staff, and die plant managers, staff, and employees.
- The equipment manufacturer for its support of and assistance in the study; from the sales office to the equipment technicians.
- The benchmark companies for opening their doors and offering information on their die development operations.
- My advisors, Steven Graves and David Hardt, for their guidance and support that resulted in this thesis.

Dedication

I would like to dedicate this thesis to my fiancee, Roxann. Her ongoing patience, support, and encouragement has not only helped make this thesis possible, but helps me to realize my true potential.

Table of Contents

List of Figures

List of Tables

List of Variables

α = Angle between radial force vector and X axis (degrees)
c = Chip load (inches/tooth)
d = depth of cut (inches)
d_{min} = minimum distance between consecutive points (inches))
δ = Step over (inches)
D = Tool diameter (inches)
= 2*(outside radius-corner radius) for radius end mill
D_e = Cutter effective diameter (inches)
$D_{e,min}$ = Minimum cutter effective diameter (inches)
$D_{e,max}$ = Maximum cutter effective diameter (inches)
f = Feed rate (inches/min)
f_{min} = Minimum desired feed rate (inches/min)
F_a = Force in the axial direction (pounds)
F_r = Force in the radial direction (pounds)
F_t = Force in the tangential direction (pounds)
F_x = Sum of forces in the X direction or dimension (pounds)
F_y = Sum of forces in the Y direction or dimension (pounds)
F_z = Sum of forces in the Z direction or dimension (pounds)
γ_r, γ_a = Radial and axial rake angles, respectively (degrees)
h = cusp height (inches)
n = Number of effective flutes or teeth (number)
r = corner radius (inches)
t = throughput time (seconds)
V_c = Cutter Velocity (in/min)
V = Volume of material to remove (inch^3 of material per inch^2 of surface area)
V_{cusp} = Volume of cusp material to remove (inch^3 of material per inch^2 of surface area)
V_{wake} = Volume of wake material to remove (inch^3 of material per inch^2 of surface area)
RPM = Spindle RPM (rev/min)
ϕ = Tool inclination or heel angle (degrees)

1.0 Introduction to die construction and five axis machining

1.1 Introduction

A stamping die transforms a sheet of flat metal into a contoured part such as a hood or door panel for an automobile. To arrive at the final part shape, the sheet must be stamped (hit) by several consecutive dies. Depending on part complexity, it takes between three and five dies (hits) to go from flat metal to finished part. A die is an assembly of several die *components*. Each component consists of a casting and a set of parts (called *details*) that are assembled to the casting. A typical stamping die (exploded by component) is shown in Figure 1.1.1.

An explanation of stamping die operation follows. Refer to Figure 1.1.1 throughout the explanation. The upper shoe is fastened to the press ram (upper sliding section of a stamping press). The punch cap is fastened to the lower shoe which in turn attaches to the press bolster (lower stationary section of a stamping press). The lower binder is placed over the punch cap and sits in the lower shoe. The binder rests on a manifold of nitrogen cylinders that compress six to eight inches when pressure is applied to the binder. When the press is open, the surface of the lower binder sits six to eight inches above the surface of the punch cap. To stamp a part, a sheet metal blank is placed between the upper and lower sections, and the press closes. Forming begins when the blank is pinched between the upper shoe and lower binder. The press continues its stroke, compressing the nitrogen cylinders while stretching and forming the blank over the punch cap. At the end of press stroke, the press opens and a formed panel is removed. This is but one type of stamping die; there are many variations of stamping die composition and sequence of operations.

Most of the sheet metal parts found in an automobile are formed by stamping dies. In fact, automotive manufacturers and their suppliers form the greater part of the die building industry. One major U.S. auto maker requires approximately 350 new panels per year, which translates into approximately 1400 production dies per year. For this major U.S. auto maker alone, stamping dies are a $1 billion business.

Automotive stamping dies are generally built in a separate plant and placed into production at a stamping plant. The formed parts from the stamping plant are delivered to an assembly plant where they are welded together to form a vehicle body at the start of the assembly process.

Die development[1] is one of the most expensive and longest lead time activity of a new car program (Drees, 1991). For this reason auto makers and independent die plants are constantly concerned with shortening lead times, lowering cost, and improving quality in die development. One technology that can provide improvements in all three metrics is five axis machining.

This thesis will evaluate and quantify the benefits of five axis machining for a major U.S. auto maker's die development operations. The study is not limited to die building operations alone, but considers how five axis machining can affect other areas of die development. It will also define equipment and process requirements to take full advantage of five axis technology.

---

[1] Die Development involves planning, design, construction, and tryout.

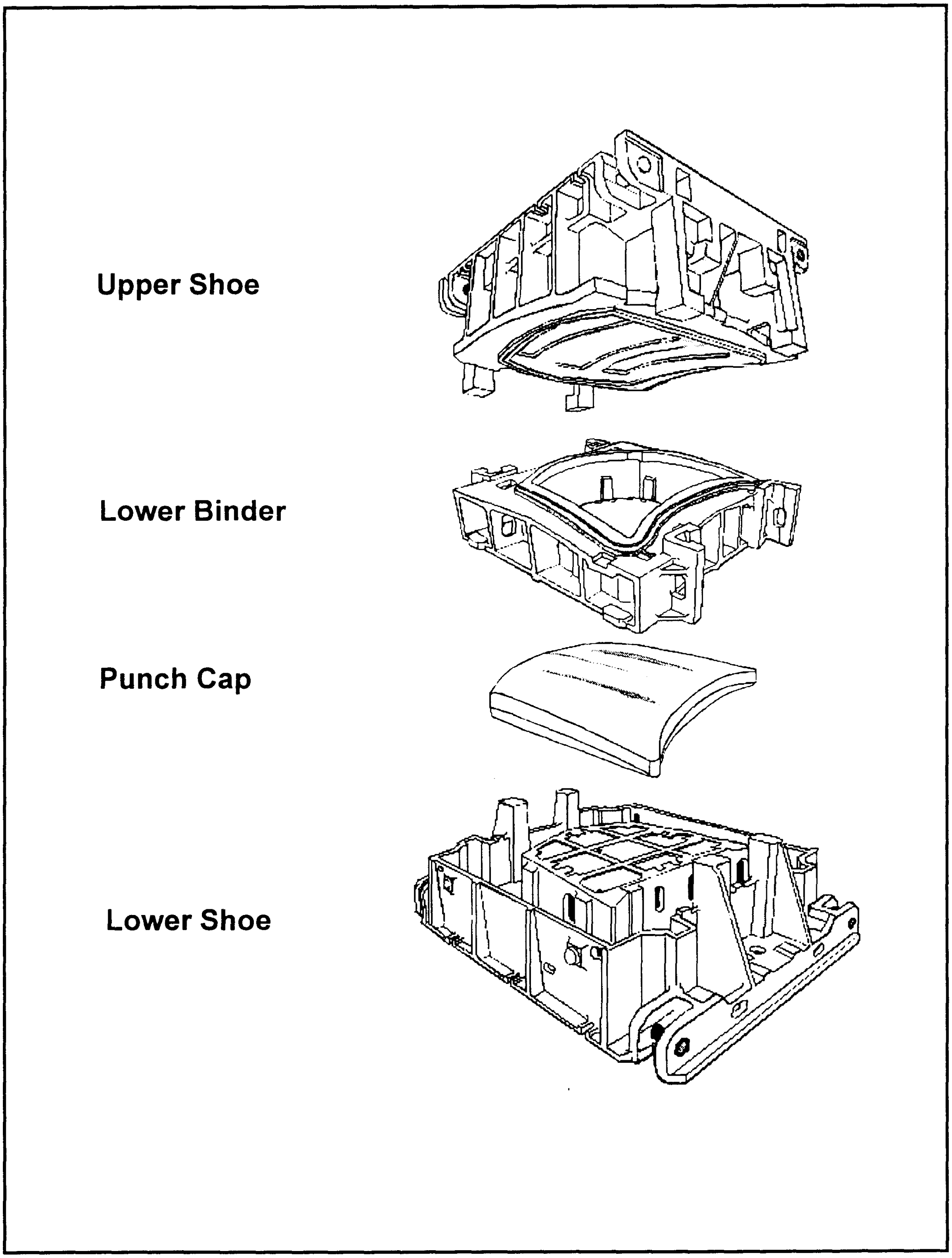


Figure 1.1.1 Exploded view of a stamping die.

1.2 Die construction process

Die development includes part design, die design, casting, construction, and tryout. Construction is the most expensive and longest lead time portion of die development for both U.S. and Japanese auto makers (Walker, 1993; Drees, 1991). Figure 1.2.1 shows a typical die development process. The hatched line below Die Design indicates the processes that are considered design. The hatched line below Die Construction indicate those processes that are considered construction.

Die design encompasses process design, layout, and detail design. Process designers determine the number of dies per part and the processes performed by each die. Die layout designers prepare the math data for individual die components. Detail designers complete the die support structure and prepare the design for a die plant.

When the die design is complete, a pattern shop builds a polystyrene model of each die component. The models become a pattern for casting. They are slightly larger than the actual die for machining allowance[2] and to accommodate metal shrinkage. The patterns are sent to a foundry for casting using a lost foam process. When the die components return from the foundry, the construction process begins.

In many cases CAM programming[3] begins long before castings arrive at the die plant loading docks. CAM programmers only need the die math data to perform their work.

---

[2] The top 1/4 inch of cast metal is porous and contaminated. It is removed to uncover clean metal.

[3] CAM is an abbreviation for Computer Aided Manufacturing and by definition refers to any use of computer technology in manufacturing. In the context of this thesis, CAM refers only to computer assisted part programming.

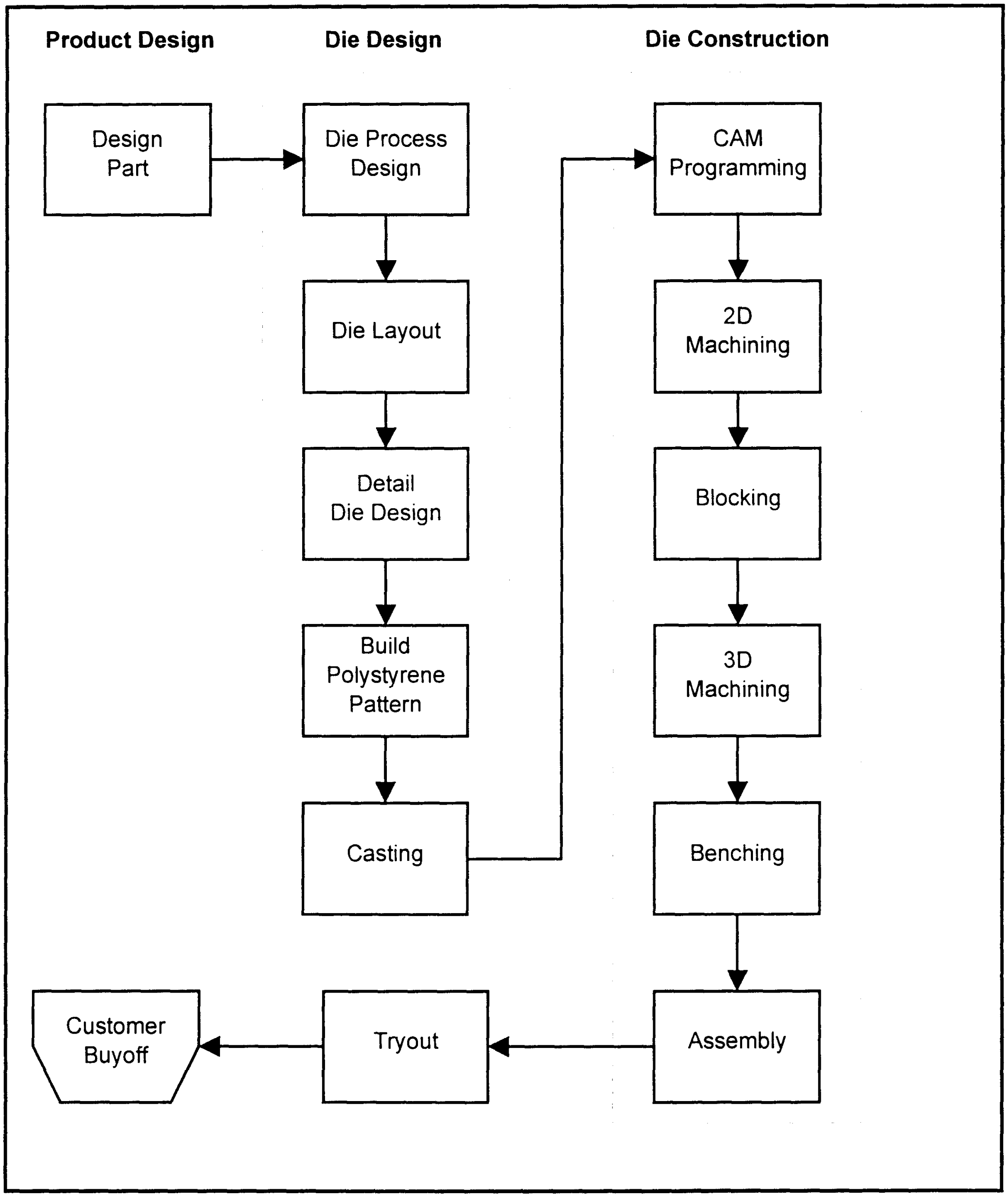


Figure 1.2.1 Die development process.

Die construction consists of six general processes. Once construction is complete, a die is ready for tryout. A brief explanation of each process follows.

CAM programming. When the castings arrive from the foundry, programmers perform 2-D and 3-D CAM programming on the die. Milling machine operators retrieve the completed programs from the CAM room when die components are ready for machining.

2-D machining. Refers to machining on a flat, horizontal or vertical plane. All mounting and mating surfaces are 2-D machined to ensure they are flat and level. Machining is performed in a two stage process; roughing and finishing.

Blocking. Intermediate stage between 2-D and 3-D machining. Die makers install keys, wear plates and slider plates at Blocking. They then assemble individual die components and details to prepare for 3-D machining.

3-D machining. This is where the part contour is defined. The milling machine operator executes the 3-D CAM program to cut away excess metal and define the part shape. Again, roughing and finishing passes are used.

Benching (or Bench). A die maker receives a die from 3-D machining and grinds away the cusp material not removed by the milling machine. After grinding, the die maker polishes the contoured surface with a series of boat stones. Die regions where metal is formed must be polished to allow for metal flow.

Assembly. In this stage a die maker will assemble any remaining die components, finish installing die details, heat treat the die surface, and prepare the die for the tryout presses.

After construction the die is tested in a tryout press where rework is performed. Once the die produces good parts, the customer (stamping plant) approves the die in a buy off procedure and it is shipped to the stamping plant for secondary tryout and production.

1.3 Five axis machining

Conventional milling is performed in three orthogonal axes. A CAM program commands a tool over a work piece in any combination of X, Y, and Z axes. Difficulty arises when trying to use three axis machining to define free formed surfaces such as the contour portion of a punch cap. When three axis machining a contoured surface, a ball nose cutter is programmed over the surface in parallel passes leaving excess material, or cusps (Figure 1.3.1, left side).

Five axis machining introduces two rotary axes to the three orthogonal axes. This allows a CAM programmer to select a flat bottom tool and orient that tool at any angle to match the contoured surface of the stamping die. Orienting the tool to the die significantly reduces the amount of cusp material a die maker must remove in the bench process (Figure 1.3.1, right side). To prevent gouging the part or smearing metal, the cutting tool must be slightly inclined to the part surface.

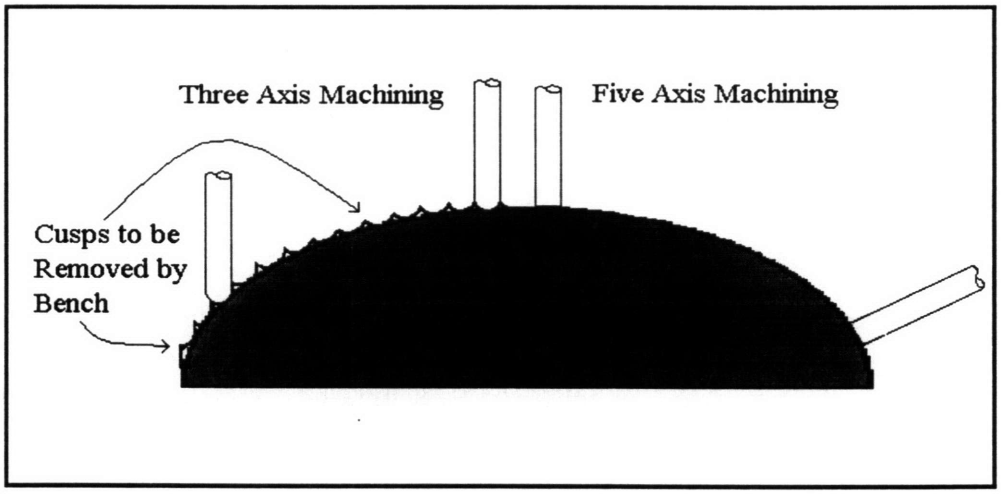


Figure 1.3.1 Three and five axis machining.

There are four stages of five axis machining. Each subsequent stage is more difficult than the former. The stages are:

1. *Five axis drilling.* Drilling in a plane other than those formed by the X, Y, or Z axes of a three axis machine tool. See Figure 1.3.2a. This is the easiest stage of five axis machining since the rotary axes are only used for positioning and there is no upsetting force on the rotary servomotors.
2. *Four axis machining.* Milling in three orthogonal axes (X, Y, Z) with an oblique tool vector. See Figure 1.3.2b. Again, the rotary axes are only used for positioning, but there is an upsetting force on the rotary servomotors.
3. *Five axis simultaneous contouring.* Traditionally considered 'five axis machining'. Refers to simultaneously moving all five axes while tracing out a contoured surface with the *bottom* of a cutting tool. See Figure 1.3.2c. This stage requires that the rotary servomotors maintain dynamic accuracy.
4. *Five axis simultaneous swarf cutting.* Simultaneously moving five axes while tracing out a contoured surface with the *side and bottom* of a cutting tool. See Figure 1.3.2d.

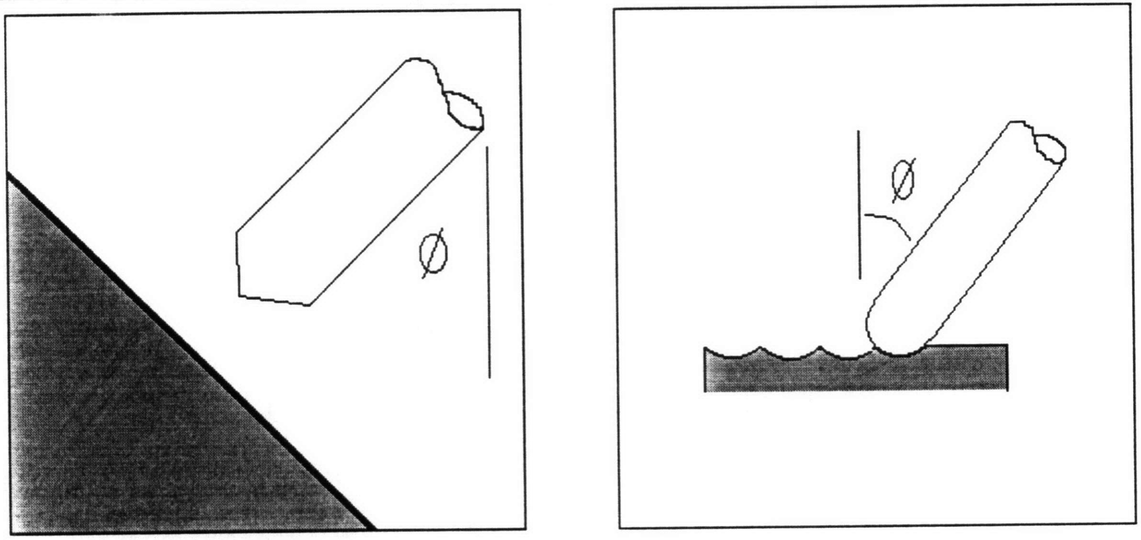


Figure 1.3.2a Five axis drilling.

Figure 1.3.2b Four axis machining.

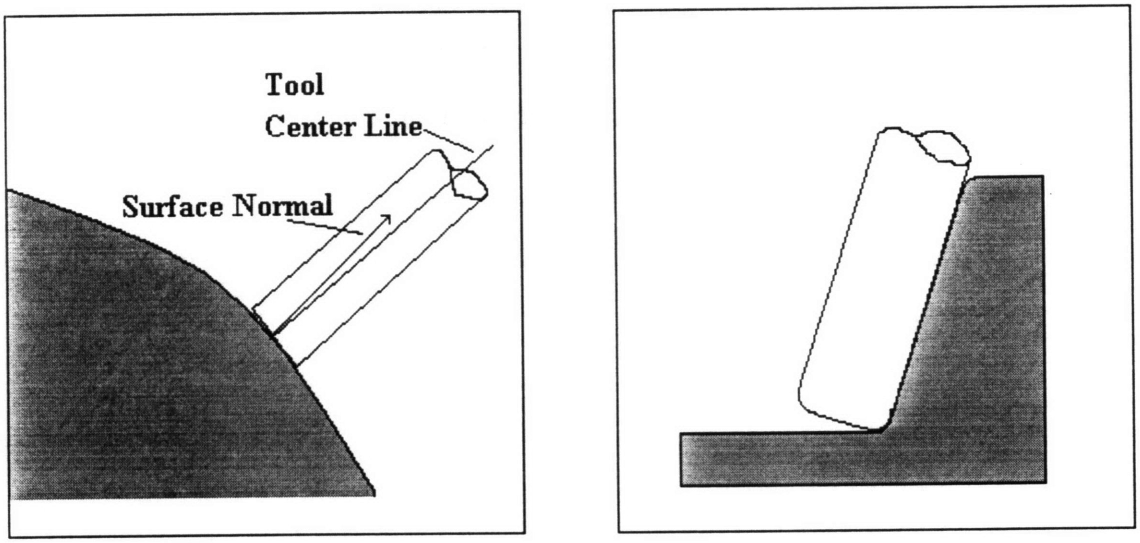


Figure 1.3.2c Five axis contour machining

Figure 1.3.2d Five axis swarf cutting.

Figure 1.3.2 Stages of five axis machining.

1.4 Die construction processes affected by five axis machining

The construction processes that are affected in a move from three to five axis machining are:

CAM programming. Few CAM software packages are able to accurately program a tool path in five axes. Visualizing and programming a cutting tool on a contoured surface is more difficult when the tool is following the surface normal.

Machining. The addition of two rotary axes to the milling machine will make the machining process more complex, reduce machining time, reduce machining accuracy (due to the addition of two axes), and increase CNC requirements.

Bench. Five axis machining reduces the amount of material to be removed at bench, and the material remaining is distributed in less defined cusps. This may require a change in the bench process.

Maintenance. Maintaining the accuracy and reliability of a five axis machine requires more effort compared to a three axis machine.

1.5 Why consider five axis machining

A recent internal benchmark study at the major U.S. auto maker showed that its die development costs are significantly higher than world class. The study analyzed die development costs and practices among several world class manufactures. One result of the benchmark study was the calculation of a performance gap that measured the difference in die development costs between the major U.S. auto maker and a world class manufacturer. The performance gap results are shown in Figure 1.5.1.

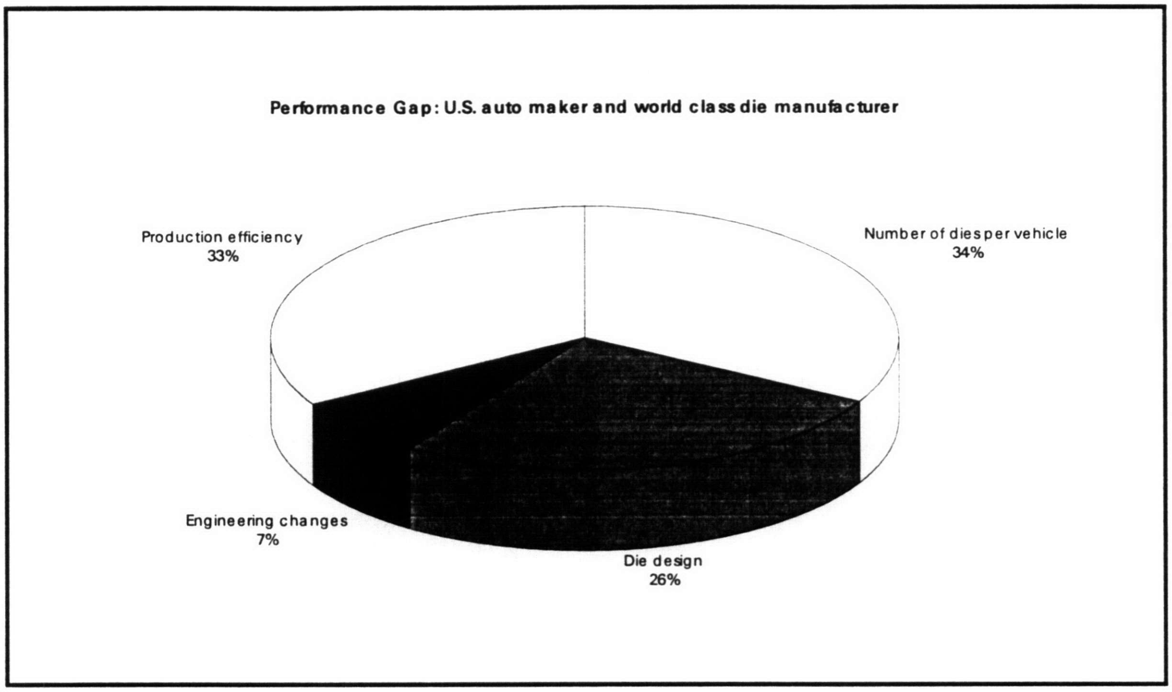


Figure 1.5.1 Result of competitiveness study.

The pie chart shows the amount this performance gap can be reduced by advancing each element to match world class practices. For example, by bringing engineering changes to world class levels (i.e. eliminating them) the major U.S. auto maker will be 7% closer to world class costs in die development. The results of the study are not surprising; one third of the performance gap is due to the number of dies per vehicle, one third due to design practices and engineering changes, and one third to production efficiencies. Taking a closer look at the production efficiencies element (Figure 1.5.2), machining efficiency is only 7% of the performance gap. That is, bringing machining to world class standards will only bring this auto maker 7% closer to world class die development costs. Further, five axis machining only affects the machine-cutting-time portion of the machining resource. Also included in the "machining" element are tool change time, program edit time, idle time, etc.

In summary, even if five axis machining were to significantly reduce machine cutting time, it would bring this auto maker only 1-2% closer to world class costs. This realization is quite sobering and a main premise for opponents of five axis machining.

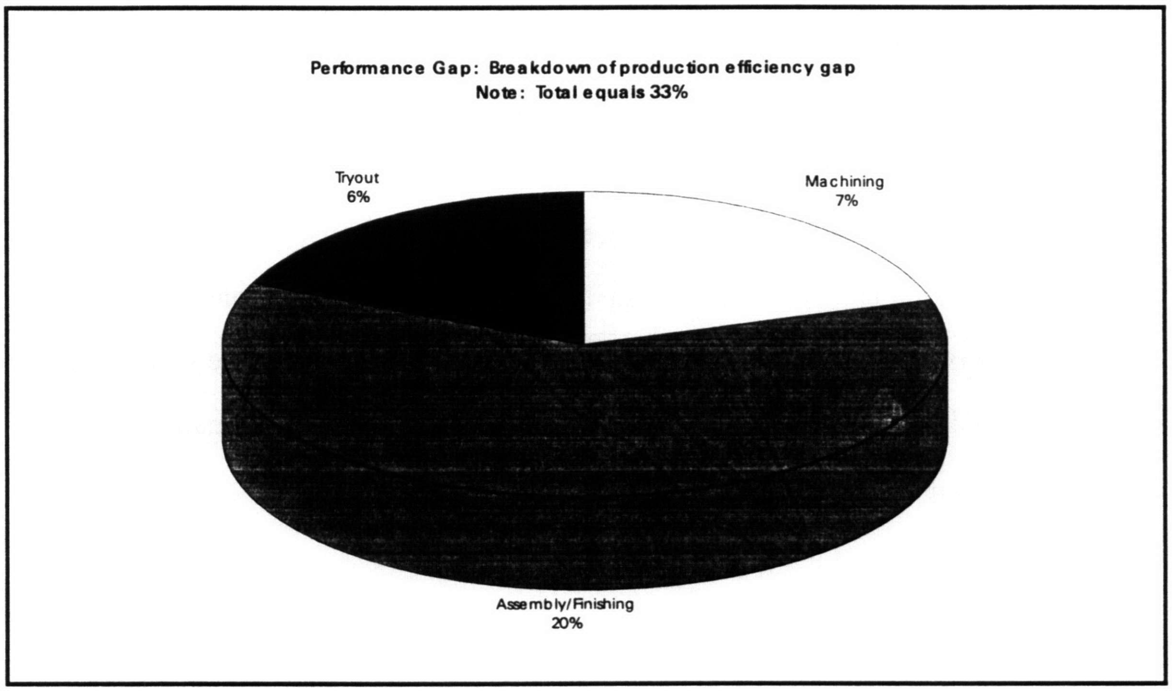


Figure 1.5.2 Production efficiency portion of performance gap.

The problem with this argument is that measurements against world class are made in units of 'machining cost' or 'time expended machining'. It fails to consider *how* the machining resource is used. Five axis machining will reduce machining time, but this is a small contribution in the drive to world class competitiveness. The major benefits of five axis machining are *not* its effects on machining time, but its potential effects on assembly and finishing (20% of performance gap) and design (26% of performance gap). By leveraging the process flexibility introduced with five axis machining, significant savings can be realized in assembly/finishing and indirectly in design. This thesis will present how this process flexibility can be used to affect greater costs and will determine the equipment and process requirements for this auto maker to take advantage of these greater savings.

1.6 Thesis outline

This thesis will look at five axis machining from the standpoint of a major U.S. auto maker. Its aim is to quantify and evaluate the benefits of five axis machining for that auto maker. It also aims to define and analyze equipment and process requirements to take full advantage of five axis technology. In the context of this thesis, five axis technology is not limited to equipment currently in use by the auto maker, it also considers alternate equipment and processes. The thesis contains eight chapters. Chapter one serves as an introduction to die making and five axis machining. An outline of the remaining seven chapters follows.

Chapter 2: Literature review of five axis machining.

The second chapter reviews current research and technology of five axis machining as it applies to die and mold making. The goal of the literature search is to determine and summarize the current state of the art for five axis machining.

Chapter 3: Benchmark of five axis mold and die shops.

This chapter presents the results of a *qualitative* benchmark study of several mold and die shops. The purpose of the study is to understand how other mold and die builders take advantage of five axis machining. The data collected includes type of equipment used, preventative maintenance practices, CAM programming tools and practices, machining and bench practices.

Chapter 4: History of five axis machining at a major U.S. auto maker.

The fourth chapter traces back to when this U.S. auto maker first considered five axis machining for its die plants. The chapter starts with an explanation of how dies were made before CNC machining. It describes a mid-eighties modernization program when five axis die making machinery was first purchased. The purpose of the chapter is to explain the current state of technology at the modernized die plants and how they arrived at that state. The chapter also explains why this five axis study was commissioned and why it is needed.

Chapter 5: Benefit of five axis machining

Each stage of five axis machining is analyzed to determine its potential to either reduce cost, reduce lead time, increase quality or increase throughput. The basis of evaluation is the current die building operations of a major U.S. auto maker. The chapter focuses on the potential benefit of five axis machining based on analytical and experimental studies. Current equipment and process limitations are not considered in this chapter.

Chapter 6: Equipment and process requirements for five axis machining

This chapter will focus on equipment and process requirements to take full advantage of five axis machining. The chapter begins with an Ishikawa diagram answering why five axis technology is not currently in use. The results of the Ishikawa diagram are combined with 'customer voices' from the die plants to come up with a list of die plant requirements for five axis machining. Current systems and customer requirements are combined to develop a set of equipment and process requirements for five axis machining.

Chapter 7: Implementation of technology

Any company can purchase new technology and 'plug it in'. Drawing the most from that technology not only requires a re-evaluation of current methods, but also a consideration of organizational and cultural issues within the company.

The previous chapter focused on equipment and process requirements. This chapter provides a means to achieve some of those requirements in light of the history, structure, and culture of the organization involved. Research on technology implementation in organizations and the history of technology implementation given in Chapter 4 will be the basis for analysis.

Chapter 8: Conclusions and Recommendations

The final chapter summarizes the findings of the research and outlines a list of recommendations and a plan for implementation for the major U.S. auto maker.

2.0 Literature review

There is considerable interest in five axis machining in both trade and research journals. The lead users of this technology are believed to be the aerospace, mold making, and die making industry. Mold and die makers must build large contoured steel parts containing many free flowing surfaces that seem well suited to five axis machining. The following review provides results and insights from several key articles on five axis machining as it applies to mold and die making. The chapter is divided into eight sections, the first seven dedicated to a particular article. The final section provides a summary of the research.

2.1 Vickers, Quan

Vickers (1989) studied the difference between three axis ball nose and five axis end mill cutting for machining low curvature surfaces. End mills were shown to better match the surface geometry and allow fewer passes to machine a part. Vickers indicated higher material removal rates and longer tool life with an end mill. The combination of fewer required passes and higher material removal rate resulted in a greater than 90% reduction in machining time with an end mill.

2.2 Takagaki, Okuno

Takagaki (1990) reported on the benefits of high speed five axis machining for stamping dies. The goal was to dramatically reduce bench (hand grinding) time. To achieve high speed machining, very light single contact point cuts were taken. Conventional machining calls for heavier cuts and lower feed rates that result in more heat, higher deflections, and ultimately less accuracy. The benefit of the former strategy is an accurate part, very low heat generation, and excellent surface finish without increasing machining time. By taking many light cuts instead of few heavy cuts, tool life increases substantially. The longer tool life allows a die shop to finish cut an entire contour with one tool. This avoids mismatched surfaces caused by replacing worn tools in the middle of a part.

Takagaki states that the CNC must be able to handle 'enormous' amounts of data to perform continuous simultaneous five axis cutting. His recommendations for a CNC to successfully perform high speed five axis machining are:

> A five axis block processing time of 5 milliseconds to handle the incoming data,
>
> A bell shaped acceleration/deceleration curve, feed forward compensation, and block look ahead to ensure path accuracy, and
>
> Accelerated backlash compensation to eliminate errors when the machine tool changes quadrants.

One important point raised by Takagaki that illustrates the importance of CAD and CAM systems is that a finished part is only as good as the CAD data defining the part and the CAM program that generates the cutter path.

2.3 Giessen

Giessen (1987) reported on the benefits of three and five axis CAD/CAM and CNC systems over traditional copy milling for mold and die building. Giessen estimates that approximately 1% of the surface of all dies for a car model are for the outer skin. The reported benefits of three axis machining over five is less programming effort, less expensive and less sophisticated machinery. Five axis milling time is only 20% of the original copy milling time and bench work is reduced 40%. By introducing CAM programming and more complex machines (capital cost and maintenance costs), some die shops report no payback with five axis machining. The author attributes these results to the (mid 80's) state of the art of CAD/CAM systems and die makers lack of experience with the systems.

Giessen believes that five axis CNC machining will provide few benefits beyond three axis. Slightly curved surfaces are much better suited to five axis machining than complex die surfaces. The author concludes that although three and five axis CNC machining shows marginal benefit to copy milling, the technology will improve and eventually return engineering and economic benefit.

2.4 Eman, You

Eman (1989) studied the removal of cusps in die finishing by superimposing a tertiary motion onto conventional cutting motions. Eman offset a second spindle on a machine tool and had the primary spindle provide a whirling motion. The second spindle held the cutting tool and was able to incline (Figure 2.4.1). Eman rotated the whirl spindle on the order of 30 rpm and maintained the whirl axis of rotation perpendicular to the work piece. As the cutting tool rotated about the primary spindle axis and advanced over the work piece, the machine Z axis was adjusted to ensure the cutting tool was in contact with the work piece. The results show a considerable reduction in cusp material to be removed using the tertiary cutter concept. Eman calculated a reduction in R.M.S. cusp height of approximately 87 percent.

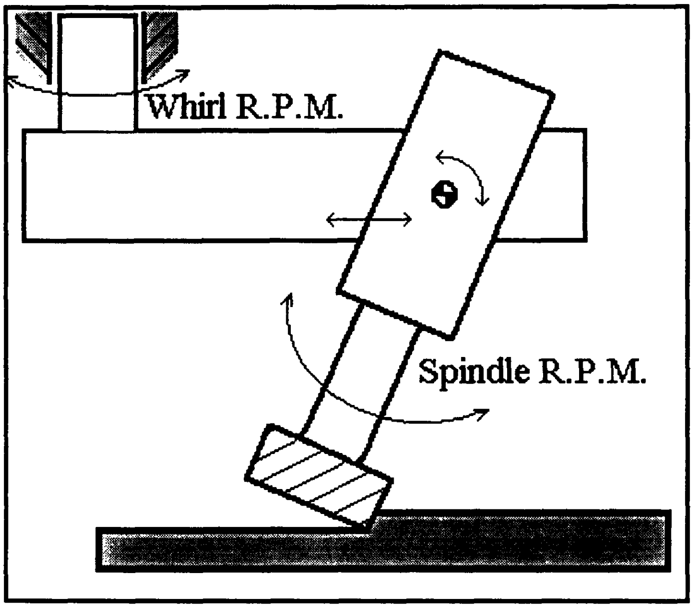


Figure 2.4.1 Eman's tertiary motion concept.

2.5 Hock, Janovsky

Hock and Janovsky (1991) summarize the use of five axis machining in die and mold making. The primary reported benefit of five axis machining is for slightly curved, convex surfaces. Savings were found principally in bench time and partially in machining time. In some cases bench time was nearly eliminated. These surfaces (typically automotive outer panel dies) only represent a small portion all dies, limiting the application of five axis machining.

Simultaneous five axis swarf cutting is another application of five axis machining for dies. It can be used to outline certain contours in one pass, eliminating the need for hand work. Two difficulties encountered by advocates of five axis machining are with the CAM software and bench process. Many users complain that CAM packages cannot generate a robust tool path that does not gouge or interfere with the part surface. When grinding a die surface, the die maker uses the cusps as a spotting aid. With five axis machining, the cusps are much lower and spread further apart, making it difficult for the die maker to spot the die (Figure 2.5.1).

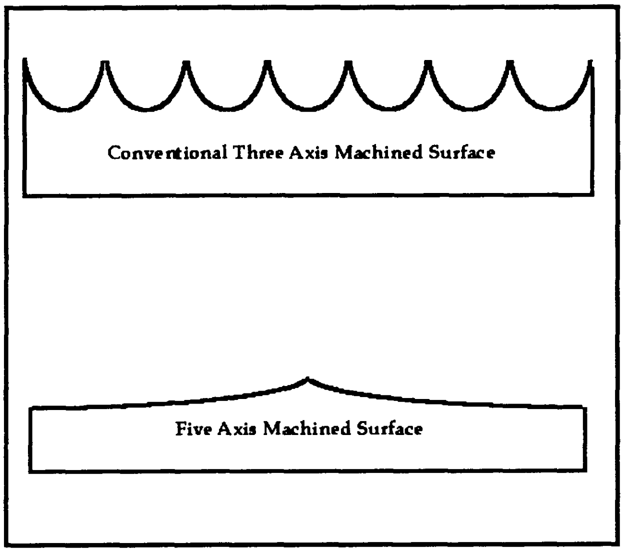


Figure 2.5.1 Difference between three and five axis machined surfaces in cross section.

High speed milling is a recent innovation where spindle speed and feed rate are increased and step over are decreased to give a much finer surface finish with equivalent machining times. Hock and Janovsky claim that the introduction of high speed milling has retarded the growth of five axis machining. Many die shops feel that the trend is to high speed three axis machining, not five axis machining.

2.6 Leverton, Scoones, Karima

The group (1993) studied alternative approaches to reduce die development lead time through the use of computer based simulation. The approaches studied were a parallel, risk management strategy and a serial, high performance process. Parallel processing of die development components requires that many processes which are traditionally performed in series are now performed in parallel. The method requires understanding and subsequently managing the risk of placing dependent processes in parallel. The high performance serial approach attempts to keep the process steps in series and drastically increase individual process performance to significantly reduce overall lead time. Two cases were studied for this latter (serial) method; computerized tryout and five axis machining.

Leverton et. al. found that five axis machining overcame several limitations associated with three axis machining but prevents the CAM programmer from having direct control over the tool path. Early attempts at five axis CAM programming required extensive checking of the tool path to assure gouge free and interference free cutting. Five axis required much larger engineering overhead compared to three axis and posed a barrier to adoption by die shops. A critical element to successful five axis machining is a CAM programming system that can accurately and repeatably provide gouge free and interference free tool paths. Computer simulation of a roof die contour showed a 91% reduction in machining time to achieve equivalent surface finish. Computer simulation of a door die contour showed a 95% reduction in machining time. Simultaneous five axis contour and swarf cutting was used for the door die. Five axis swarf cuts were used to define regions around door openings such as the window and door handle. The swarf cut regions required no subsequent hand work (the cutter exactly defined the part).

2.7 Bedi

Bedi studied methods of machining complex 3D surfaces on a four and five axis milling machine using a toroidal (radius end mill) cutter. He concentrated on maximizing surface definition and material removal rate during finish cut machining. Most advocates of five axis machining maintain an end mill or toroidal cutter at some arbitrary fixed angle to the part surface. Bedi suggests using the unique properties of a toroidal cutter to vary the inclination of the cutter and match the curvature of the tool to the part curvature. Varying the lead angle of a toroid or flat bottom end mill will both vary the curvature of the cutting surface of the tool. However, the sharp edge of an flat bottom end mill generates wake material that results in a poor surface finish. Toroidal cutters have a radius edge that nearly eliminates wake material.

Bedi found that by varying the angle between tool and part he could set the maximum curvature of the cutting tool equal to the minimum curvature of the part surface. The result is a close fit between the part surface and tool profile, minimizing cusp height. Bedi reports a 30% increase in amount of material removed using this technique on a five axis machine as compared to a three axis machine. The four axis machine showed a 5% increase in material removal over three axes.

2.8 Summary of research

From the above articles, several issues seem apparent in the research. First, there seems to be benefit in pursuing five axis machining in die and mold construction. Second, most of the research effort is designed around reducing either machining time, bench time, or both. Little emphasis is placed on how five axis machining can affect other areas of die development. Finally, the research states that CAD/CAM and CNC capabilities are critical to successfully implementing five axis machining.

These issues are explored further in this thesis; specifically as they apply to the major U.S. auto maker. The thesis will also provide insight on how five axis machining can reach beyond the boundaries of die construction to affect die and product design; an area that is generally not covered by the research.

3.0 Benchmark study

The seven benchmark companies were a mold builder, a heavy equipment manufacturer and five die shops. The following sections present benchmark results for each company.

3.1 Company A: Michigan die shop #1

Company. The company performs CAM programming, machining, bench and tryout of dies for the automotive industry. It will also perform some CAM programming and machining work for the aerospace industry. The location visited only does programming and machining.

Machinery. Company A has many three axis and two five axis machines. The five axis machine at this location is a Cincinnati Milacron™ with +/- 25 degree tilting head in the A and B axes. The second machine is a smaller Jobs™ machine at another location. The Jobs machine is too small for die making and is up for sale. The five axis Cincinnati Milacron is used for two, three, four, and five axis machining.

CAM system. Company A uses the Catia™ CAD/CAM system. No verification or simulation package is used. The programmers are confident that the tool paths they generate with Catia will not gouge the part surface, leave material full, or crash the machine head.

Transition. The transition from three to five axis was very easy for company A. Only CAM programmers required significantly more training.

Maintenance. The Cincinnati machine is quite robust. The general foreman claims the five axis machine requires no more maintenance than the three axis machines. It is checked every one to two weeks to ensure the spindle axis is perpendicular to the table. This is done by loading the spindle with a horizontal bar and instructing the machine to trace a circle in the spindle plane. The test will quickly indicate which axis is not zeroed. To correct the error the maintenance person increments B or C axis until the circle is perfectly parallel to the table and zeros the B and C axis encoders at that location. When a more serious error is detected, a local company is called in to align and calibrate the axes with a laser. Company A did find a loss of accuracy with the five axis machine for two reasons. First, an error in angular alignment is grossly exaggerated by the length of the spindle and tool. Second, the axes are not perfectly orthogonal, nor do they intersect exactly. The combination of these two errors makes it difficult to know exactly where the tip of the tool is at all times

Five axis applications. Very little five axis contouring is done with the Cincinnati machine. The B and C axes are mostly used to position a ball nose at an angle for four axis machining. The tool rotation angle is selected by the operator when he/she cuts the job. If the job is programmed using tool centers, the operator need not make any adjustments. If the job is programmed using tool tips, the operator must calculate the new tool tip. The B and C axes are also used for five axis drilling holes and to machining flat, inclined surfaces on dies. Most aerospace work is limited to five axis drilling.

The machine is rarely used for simultaneous five axis machining (swarf and contour) due to travel limits on the B and C axes. Most die surfaces require more than 25 degrees of rotation to be machined with five axes. The little five axis simultaneous swarf cutting done at company A is done for the aerospace industry. Swarf cutting causes excessive tool loads and subsequent deflections, degrading part accuracy. The aerospace industry allows tolerances on swarf cuts that are wider than the error caused by the deflection. Die industry tolerances are tighter than the deflection error.

3.2 Company B: Ohio die shop

Company. The Ohio die shop performs design, pattern making, CAM programming, machining, bench and tryout of dies for the automotive industry.

Machinery. Company B has ten three axis and one five axis machine. The five axis machine is a new Rambaudi™ high speed bridge mill with a trunion head capable of +/- 105 degrees B and 360 degrees C axis travel. The five axis Rambaudi has a maximum spindle speed of 20,000 rpm and rated accuracy of +/- 0.0006 inch within the entire workspace. The machine has interchangeable spindle heads for high torque five axis, high speed five axis, or high speed three axis machining.

CAM system. Company B supports various major CAD/CAM systems since it designs and manufactures dies for several major auto makers. Unigraphics™, Calma™, Cisigraph™, Catia™, CGS™, WorkNC™ and Autocad™ are some of the packages supported. There is no additional software verification or simulation performed beyond the capabilities of the CAM software used.

Transition. At the time of the benchmark study, the five axis machine had only been in operation for several months. To date there have been no difficulties in incorporating the machine into production.

Maintenance. There is very little maintenance history for the Rambaudi machine. The machine seems to be very robust to date and company B has not yet found any maintenance related problems.

Five axis applications. Company B did not purchase the Rambaudi machine with the express intent of five axis machining. Upon selecting Rambaudi as the equipment supplier it realized the incremental cost for five axis capabilities was very small. This led to a decision to purchase five axis attachments with the machine.

At the time of the benchmark study, the Rambaudi was only being used for three and four axis machining. Company B claims the biggest benefit of four axis machining is less machine time by being able to increase feed rates up to 300 inches per minute.

Over the next year or two company B intends to use the two axis spindle attachments to perform five axis simultaneous machining. Automotive outer panel dies are the intended candidates for simultaneous five axis machining.

3.3 Company C: Illinois mold shop

Company. The Illinois mold shop is the division of a company which builds and sells press-wood doors and other wood products. The mold shop is part of the company research and technical center that performs design, CAM programming, machining, finishing and tryout of all molds for its manufacturing plants.

Machinery. Company C has only one five axis milling machine at its technical center mold shop. The machine is an Ingersoll™ gantry type rail mill with an interchangeable spindle. The spindle options are a three axis spindle and a five axis nutating spindle. The nutating spindle is capable of +/- 100 degree rotation in B and 360 degree rotation in C axis. The Ingersoll machine is used for two, three, four and five axis cutting.

CAM system. Company C uses the NCL™ CAD/CAM system from NCCS™. It has Vericut™ N/C (numerical control) verification software available but does not use the software, claiming it takes too long to verify a cutter path. The company uses no other verification or simulation package. The programmer feels that the verification and simulation package within NCL is robust enough to warrant not using Vericut.

Transition. The transition from three to five axis was relatively simple for company C. The benefits gained from five axis machining helped to overcome any difficulties encountered. Only CAM programmers were required to pursue additional training.

Maintenance. Company C had relatively minor problems with the nutating spindle itself but did encounter several maintenance related problems with the Ingersoll machining center. The nutating spindle did have problems maintaining seal integrity. After multiple attempts trying to solve the problem, company C resorted to not using through-spindle coolant and living with oil dripping from the spindle unit when the nutating spindle is loaded.

Accuracy and repeatability of the machine and nutating spindle is rarely a problem. Calibration of the nutating spindle only occurs when the spindle crashes into a part, the program gouges the surface of a mold or the five axis cut "doesn't look right".

Five axis applications. There are no contoured or free formed surfaces on a door mold. The shapes of the mold lend themselves nicely to four and five axis swarf cutting. All molds at company C are machined to net shape. There is no hand grinding required after the mold comes off the machine. Hand polishing is done to remove machining marks and bring the mold surface to a mirror finish.

3.4 Company D: Michigan die shop #2

Company. The company builds dies for the automotive industry and engine parts for the aerospace industry. It performs CAM programming, machining, bench and tryout of dies for the automotive industry. Stamping dies represent approximately 90% of the work it sees. Company D has been programming N/C machines for 25 years and has been five axis machining since the late 1970's.

Machinery. Company D has many three axis and several five axis machines. Two of the five axis machines are Cincinnati Milacron™ with a two way tilting table, two are Pratt and Whitney™ bridge mills with a trunion type head and one small Boston Digital™ machine.

CAM system. Company D uses Unigraphics™ for its CAD/CAM system. It owns a copy of CGTech™'s Vericut™ software but no longer uses it. Unigraphics has very robust five axis software. The CAM programmers have been five axis programming for so long that they seldom have problems with five axis cutter paths gouging the part surface.

Transition. The transition from three to five axis in the late 1970's was a difficult one. The state of CAM software at the time made five axis machining very difficult. Once the software improved machining with five axes became much easier.

Maintenance. All five machines require some additional maintenance due to the complexity of the two axis tables and heads. Beyond this Company D does not see additional preventative maintenance requirements or loss in reliability with the five axis machines.

Five axis applications. Company D invested in five axis technology to build engine parts for the aerospace industry and decrease bench on outer panel dies. Aircraft engine parts have tighter tolerances than stamping dies but the surfaces on engine parts is much simpler. The simple surfaces make five axis contouring and swarf cutting ideal applications for these parts.

In addition to engine parts, the five axis machines are used for outer panel dies. The goal with five axis machining is to reduce bench time. All convex contours on outer panel dies are machined with a bull nose (toroidal) cutter in five axes. Only the finish pass is five axis machined. All rough work is performed with three axes. Company D views five axis drilling and four axis machining a waste of valuable five axis machine time. Five axis drilling is done with a portable indexable drill. Although four axis machining may show some machine time and tool life savings, it finds the big savings in simultaneous five axis machining.

3.5 Company E: Japanese transplant die shop

Company. The company is a major Japanese auto maker with a manufacturing plant in the United States. This plant not only assembles cars but also builds production dies for domestic stamping operations. It performs CAM programming, machining, bench and tryout of its dies. The benchmark location was an in-house research, development, and production die shop.

Machinery. Company E has several SNK™ three axis machines and one Ingersoll™ five axis machine with a nutating spindle. The five axis machine is very similar to that of company C. The Ingersoll machine is used for two, three, four, and some five axis machining.

CAM system. Company E uses its own proprietary CAD/CAM software and the Camax™ CAD/CAM system. Camax has a very robust verification and simulation package. The programmers have CGTech™'s Vericut™ verification software available but do not use it extensively. They are confident the tool paths generated by Camax will not gouge the part surface, leave material full, or crash the machine head. One problem with Vericut is it takes too long to gouge check a job with five axes.

Transition. The transition from three to five axis was easy for company E. Only CAM programmers required extensive training on five axis programming and the Camax system.

Maintenance. The nutating spindle requires some additional maintenance due to the complexity of the head. All machines are thoroughly checked annually for accuracy and repeatability. Additional maintenance is required for the nutating spindle when the head 'looks off' or gouges the part. At this point the nutating spindle is checked and re-calibrated. In most cases the Nutator™ maintains its accuracy.

Five axis applications. Most of the work done at this die shop is five axis drilling and tapping and four axis machining. Some simultaneous five axis machining is done but only for research and development purposes.

Most of the four axis work involves machining cam and die adapters, and cam driver faces that are cast directly into the die shoe. Four axis machining is used so long as the nutating spindle does not interfere with the die. The benefits of four axis machining for company E are quite large. It allows the company to cast cam components directly into the die shoe which results in fewer parts, less machining, less assembly time, lower part cost and much higher part accuracy.

At the time of the benchmark study company E was experimenting with simultaneous five axis contouring with the intention of machining contoured die surfaces in five axis.

3.6 Company F: Heavy equipment manufacturer

Company. The company is a large manufacture of off road equipment. The benchmark location is a mid volume production facility that fabricates power train parts for off road equipment.

Machinery. The benchmark location has many three axis and one five axis machine. The five axis machine is an Ingersoll™ with a nutating spindle and part of a six machine flexible manufacturing system (FMS) that machines transmission and transaxle cases and frames. The nutating spindle has the same specifications as that used by company C.

CAM system. Company F uses a Giddings and Lewis™ proprietary CAD/CAM system. No verification or simulation package is used.

Transition. There was no additional difficulty due to the nutating spindle. It was a small part of the FMS and is not required to perform five axis contouring. Extensive training was required to debug and run the FMS, but the nutating spindle did not add more time to the training.

Maintenance. The machine seems quite robust. No additional maintenance is required beyond the standard preventative maintenance on the FMS.

Five axis applications. Very little five axis contouring is done with the nutating spindle. It is part of a flexible manufacturing cell and only performs four axis machining. The spindle is set to a compound angle and machines flat regions at this angle. The process is a mid-volume production process that accepts parts from a three axis machining center and delivers parts to another three axis machining center. Tasks performed by the FMS are repetitive within batches. The only change to the nutating spindle between batches is to set a new compound angle.

3.7 Company G: Michigan die shop #3

Company. The company builds dies for the automotive industry. It performs CAM programming, machining, bench and tryout. The company was near bankruptcy and was taken over several years ago. It is now under new management.

Machinery. Company G has approximately 25 three axis and two five axis machines. The two five axis machines are Ingersoll™ machining centers with nutating spindles. The nutating spindles are similar to those of company C and are limited to 1100 rpm spindle speed.

CAM system. Company G builds dies for the automotive industry and as such uses several CAM packages. Gerber™, Cisigraph™, Catia™, PDGS™, Strim™, and WorkNC™ are all used by company G in programming dies. Only Gerber, Catia, and Strim have five axis capabilities.

Transition. The Ingersoll machines were purchased by the prior owner of the company over ten years ago. They were purchased with the intent that five axis machining would one day be pursued. That intention was lost when the company changed hands.

Maintenance. Company G only uses the nutating spindles occasionally. When they are used they tend to overheat. Maintenance requirements for the nutating spindles are unknown since they are used so rarely.

Five axis applications. Company G does not use their nutating spindles for either four or five axis machining. They are used occasionally to machine a die region that is difficult to reach with a straight or right angled head. The reason for not using the five axis capabilities of the Nutators™ was the limited spindle speed and the inability to program five axes. The 1100 rpm spindle speed limit requires a very slow feed rate which substantially limits the machine throughput. The company has no experience or training in five axis programming, nor do they feel it is required.

Company G has purchased a machine with a fixed 30 degree spindle with which they will perform high speed, four axis die machining. This is as far as they intend to go in the near term with four or five axis machining.

3.8 General conclusions from benchmark study

All of the benchmark plants had only a small portion (typically less than 10%) of its machines either dedicated to or capable of five axis machining.

There is little consensus among the benchmark plants as to the benefit of four and five axis machining.

Maintenance requirements increase by a negligible amount for five axis machining centers compared to three axis. A common theme was the need to confirm the accuracy of a five axis machine more regularly than three axis machines.

Verification packages are popular among the above plants, but interestingly enough are seldom ever used. It seems that once CAM programmers became familiar with five axis programming and develop confidence in the generated tool paths, detailed verification becomes more a burden than a benefit.

The transition from three to five axes for these plants went rather smoothly. The only significant impact was the additional training required for CAM programmers.

Company	4 Axis	5 Axis	Recent	Spindle type	Other uses
A	YES	NO	YES	Dedicated	Military
B	YES	YES	YES	Exchangeable	None
C	YES	YES	NO	Exchangeable	Molds
D	NO	YES	NO	Dedicated	Aerospace
E	YES	YES	NO	Exchangeable	None
F	YES	NO	NO	Exchangeable	Offroad equip.
G	NO	NO	NO	Exchangeable	None

Table 1. Summary of benchmark results.

Table 1. summarizes some key points from the benchmark study. The columns are defined as follows:

4 Axis: Company machines with four axes (three axis with a tilted tool vector).

5 Axis: Company presently does or intends to machine parts using five axis simultaneous machining in the near future.

Recent: Five axis machine is less than five years old.

Spindle type: Machines are dedicated five axis or have exchangeable spindles.

Other uses: Use of five axis machines for other than dies.

4.0 History of die plant technology for a major U.S. auto maker

4.1 Die building before CAM

Before the introduction of CAM programming systems and CNC machine tools, die plants used three axis copy milling and construction aids to build dies. Wooden models of each die contour were machined from CAD (math) data. Addendum surfaces were manually assembled together with the contour to complete the model. The wooden die model was used as a pattern to cast (female) plaster molds. Male plaster molds were cast from the female molds. The male and female plaster models were subsequently shipped to a plant for die construction.

Patterns were built in polystyrene from die prints and models, then shipped to a foundry for casting. Once the castings arrived at a die plant, the bottom of the die would be machined on a planer[4] to achieve a flat reference surface. The casting was then mounted on a copy mill together with the plaster pattern.

A copy mill had two heads: a tool head and a probe head. The machine operator would locate the die and pattern such that the tool (machining) head was located over the die and the probe was located over the pattern. Both tool and probe had to be positioned over the exact same spot on the die and pattern, respectively. To machine the die the operator would trace the probe over the pattern and the copy mill would 'copy' the movements with the machining head.

After machining, cusps (which were in the order of 0.040" high) were ground smooth by a die maker in a bench operation. After the cusps were ground, the die maker would use a spotting rack to ensure the die contour was correct. A spotting rack was another pattern

---

[4] An automatic 2-D milling machine that will lace cut a surface at a constant depth.

made of plaster composite or plastic that was to replicate the part exactly (plus metal thickness for male dies). With the aid of the spotting rack and templates, a die maker could grind and polish a die to the correct shape. Following bench came die assembly and finally die tryout, after which the die was ready to produce parts.

The copy milling process had many sources of variability that occasionally led to poor part fit. The main sources of variability were:

- Variation in the wood model,
- Shrinkage or deformation of the plaster,
- Accuracy of the copy mill,
- Accuracy of bench and bench aids.

Moving to CAM programming and CNC machining eliminated the wood model, plaster mold and all bench aids, greatly improving die and part quality.

4.2 Major body tooling program

The late '70's and early '80's saw the beginning of globalization for the auto industry. At that time, this major U.S. auto maker benchmarked several European and Japanese die builders to compare die cost and panel quality. The major U.S. auto maker was trailing in both part quality and manufacturing technology. A result of the benchmark study was a program to provide technologically advanced equipment to several die plants in order to meet world class cost and quality standards in die construction.

In 1982 a joint committee was formed to develop "... a plan for the implementation of a modern, efficient tool and die build facility...". Primary objectives of the committee were to:

1. Minimize lead time,
2. Improve quality (reduce variability, improve fit) of sheet metal stamped parts,
3. Reduce die costs.

Secondary objectives were:

1. Minimize bench work,
2. Maximize use of digital data.

The committee concluded that there were significant die construction cost savings in procuring a computer numerically controlled (CNC) system from the then current copy mill technology. The major technological shortfall at the time was the availability of CAM software to operate a CNC system. The key recommendations of the committee were:

> Assign dedicated personnel from central engineering and die plants to plan and implement a tool room modernization program.
>
> Modernize selected plants in parallel.
>
> Recognize that to ensure success, implementation must be a cooperative program among plants, general offices, and central engineering activities.
>
> Specify and procure major machining centers with tool changers, CNC, tracer controls, three axis and five axis capabilities.
>
> Generate software to design dies, generate N/C code, and distribute N/C data to machining centers.

Develop training and skill certification programs for employees.

Assign a champion at each plant responsible for implementing the new technology and report directly to the plant manager.

Ensure the ability to perform full five axis contouring is provided.

Recognize that work force involvement is important for the total success of the system.

Early experiments indicated that five axis machining would show great benefit to machining die contours once robust five axis CAM systems were developed. For this reason the committee recommended that all new machines have five axis capability.

4.3 Major body tooling program implementation

Actual implementation of the tooling program differed from the recommendations of the joint committee. Individual plants were responsible for installing the new machines and systems with the help of outside contractors. The equipment manufactures embarked on training programs for the CAM programmers, machinists, and maintenance people that would work with the new equipment.

The machining centers that were ultimately purchased contained multiple spindle units (also called attachment heads). The ability to switch spindle units gave the machines great flexibility. A typical machining center would have the following spindle units:

- Heavy duty straight (for heavy, rough cutting),
- Medium duty straight (for finishing cuts),
- Light duty, high speed straight (for high speed machining),
- Medium duty right angle for machining perpendicular to the Z plane, and

- Five axis attachment which added a second rotary axis to the machine; typically a nutating type spindle (See Figure 4.3.1).

To aid the transition to CNC machining, a group of engineers from central engineering were to be assigned to aid the die plants with any startup problems. Each engineer would have to be intimately involved with the program since its inception and have some specialization (software, control systems, machining, tooling, etc.). The trouble shooting group would operate as a S.W.A.T. team when any of the modernized die plants ran into problems with the new equipment. Once the problem was resolved and a solution implemented, team members would travel to the other modernized die plants and (proactively) implement the solution.

Shortly after equipment installation and prior to the formation of the trouble shooting team, the die making business of this major U.S. auto maker underwent a major reorganization. In the process of reorganization the central engineering group responsible for the modernization program was broken up and re-assigned to different divisions. No trouble shooting team of experts, as originally proposed, was ever formed.

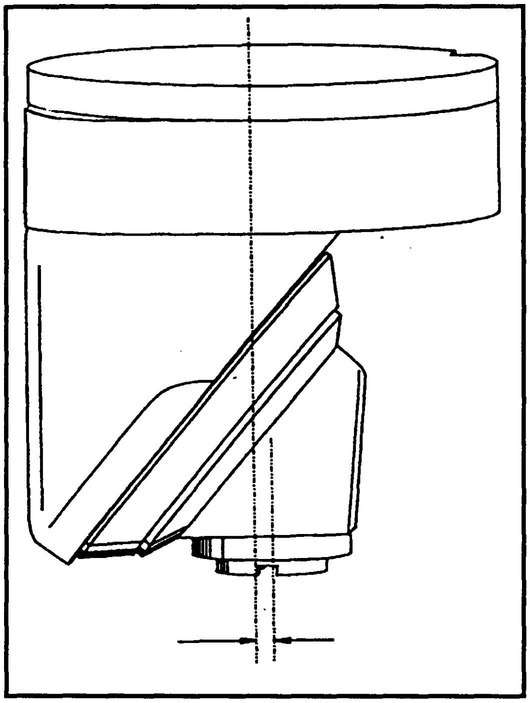


Figure 4.3.1 Nutating type spindle unit.

The plants were left with little outside support to solve problems with the new equipment when it failed. Resources it used were other modernized die plants, the equipment manufacturers, and (informally) members of the reassigned central engineering group.

4.4 Difficulties encountered by die plants

After installation of the new machinery, the modernized die plants went through a period of growing pains that lasted several years. Most of the modernized die plants had little experience with CNC machining and were left with little training. This combination resulted in a poor understanding of both the capabilities and requirements of the new equipment.

Much of the equipment relied on leading edge technologies that had limited prior exposure to a production environment. Once installed, the new machines experienced considerable down time in the die plant environment. Little attention was given to the integration of the various system components (product design, CAD/CAM, machining, bench, tryout, maintenance) that led to further delays and difficulties trying to get entire systems working.

Difficulties with the system led to difficulties and differences of opinion between plant management, central engineering staff and equipment suppliers. These difficulties resulted in animosity between the three groups.

The reorganization left the original implementation team scattered throughout the corporation with new job assignments. It is not clear if responsibility was assigned to the various plant or central engineering staff to meet performance targets with the new equipment. The burden of solving problems and getting the equipment up and running was principally the responsibility of each individual die plant.

4.5 Current state of die plants

This combination of events resulted in each die plant making the most of the new technology with the people it had. The basis of the change was a transition from copy milling to CNC machining in order to eliminate the use of patterns and construction aids.

Little emphasis was placed on the five axis spindles that were part of the modernization drive. Five axis machining was viewed as an option that could be incorporated at a later date once the new equipment was up and running. The move from three to five axis machining was considered a small one compared to the change from copy milling to CNC machining.

Each modernized plant progressed at its own rate. Some tried using the five axis heads (primarily nutating type heads) earlier than others. Several die plants viewed the nutating spindle as an all-purpose spindle; it could be used as a straight, right angle, or five axis head. These plants experienced accuracy, repeatability and reliability problems with the nutating spindle and realized it was not intended for all-purpose use. Other plants, after some experimenting realized that the nutating spindle performed best when subject to light, four and five axis cuts and limited its used to those applications. Finally, some plants had difficulty with the installation of their nutating spindles that led to problems whenever used.

The variety of experiences with the nutating spindle resulted in a lack of consensus among the plants and between the plants and equipment supplier. There was no agreement on the benefit (if any) of five axis machining or the capabilities of the nutating spindle unit. The purpose of this study is to settle the debate and determine:

1. The benefit of five axis machining,
2. The capability of current die plant equipment, and
3. Equipment and process requirements for five axis machining.

The goal is to quantify the benefit of five axis machining for the major U.S. auto maker and define equipment and process requirements to take full advantage of five axis machining.

5.0 Benefit of five axis machining

As mentioned Chapter 1, the benefits of five axis machining are not only to machining time but also to bench, assembly, and design. By taking advantage of the process flexibility introduced by five axis technology, greater savings can be realized. In this chapter these benefits are explained and where practical, quantified. The chapter is divided into three sections; five axis drilling, four axis machining, and five axis simultaneous machining. The third section on simultaneous machining combines contouring and swarf cutting. Swarf cutting is not explored in the same depth as the previous three stages (drilling, four axis, contouring) since process and equipment constraints at the auto maker prevent it from taking advantage of five axis swarf cutting. To take advantage of swarf cutting, significant organizational and process change is required. Equipment and processes required to take full advantage of the benefits will be considered in Chapter 6. Savings are based on current practices in place at a typical die plant of the major U.S. auto maker. The derivation of all equations used in this chapter are provided in Appendix A.

5.1 Five axis drilling

Five axis drilling (See Figure 1.3.2a) refers to drilling at an angle that is not parallel to any of the three (X, Y, Z) orthogonal axes of a conventional milling machine.

5.1.1 Use of five axis drilling in die construction

During construction, each die receives two location holes on the die face which must be normal to the die surface. These are referred to as coordinating holes, and are used during die tryout to nest panels between hits. Some dies require more than two coordinating holes and some are parallel to the machine Z axis (i.e. standard drilling procedure, not five axis drilling). On average, each die requires two five axis drilled holes.

5.1.2 Current process used to drill five axis coordinating holes

Each die plant has its own procedure for drilling coordinating holes. The estimated savings presented here are for one particular die plant which is an internal (captive) supplier for a major U.S. auto maker.

To drill a coordinating hole, a nutating spindle is loaded on a machining center and set to the angle of the first coordinating hole. Due to miscalibration of the nutating spindle, the tool tip is not located at the exact commanded position. In fact, the exact position of the tool tip at this point is not known. To move the tool to the correct location, a machinist will manually check the Nutator™ position to a tooling ball at a known coordinate. By comparing the Nutator's actual position to the known position of the tooling ball, the machinist is able to calculate a series of offsets which will bring the Nutator to the correct location. The machinist enters these offsets into the machine controller, then proceeds to drill the first coordinating hole. The entire procedure is repeated for subsequent coordinating holes. To drill two coordinating holes in a die using this procedure requires approximately 8 machine hours.

If the procedure is not closely followed, incorrect offsets are entered and the coordinating hole will be placed out of tolerance. When this happens, the die must be set up on the machining center a second time.[5] The machinist repeats the procedure, drilling a larger hole over top of the incorrect hole. A bushing is inserted into the larger hole to bring the hole diameter back to the correct size.

5.1.3 Savings with five axis drilling

With a calibrated and error free five axis spindle unit, the process of drilling coordinating holes should take less than one machine hour per die. Using an estimate of 7 hours saved per die and an annual die plant output of 400 dies[6]:

$$\frac{7\text{ machine hours}}{\text{die}} \times \frac{400\text{ dies}}{\text{year}} = 2800\text{ machine hours}$$

A savings of 2800 machine hours per year is possible. Using a plant rate of $100/machine hour, 500 machining hours per average die and 6900 hours available per machine per year[7], this savings equates to:

$$\frac{2800\text{ hours}}{\text{year}} \times \frac{\$100.00}{\text{hour}} = \frac{\$280{,}000}{\text{year}}$$

$$\frac{2800\text{ hours}}{\text{year}} \times \frac{1\text{ die}}{500\text{ machine hours}} = \frac{5.6\text{ dies}}{\text{year}}\text{ throughput improvement}$$

$$\frac{2800\text{ hours}}{\text{year}} \times \frac{1\text{ machining center}}{6900\text{ hours}} = 40\%\text{ of a machining center}$$

---

[5] The accuracy of coordinating hole locations are checked after the die is removed from the machining center.

[6] Actual values of output, cost, and productivity vary substantially from plant to plant.

[7] 345 days/year, 20 hours/day.

By simply maintaining a calibrated five axis spindle unit, a die plant could witness a 1.4% throughput improvement from coordinating holes alone. The change will not affect upstream or downstream processes. These savings are for one particular die plant of the major U.S. auto maker. Different plants use different procedures to drill coordinating holes. To determine savings for a specific die plant, the processing times for the various operations must be substituted for those given here.

5.1.4 Other uses of five axis drilling

The ability to perform five axis drilling is a process capability that can be used to generate revenue from other sources. Several of the benchmark companies use their five axis capabilities to perform complicated multi-axis drilling sequences on parts for the aerospace industry. The additional capacity yielded from five axis drilling may be used to pursue work in that industry. In fact, 40% of one machine's time can be allocated to generating other revenue sources given the savings from drilling coordinating holes.

5.2 Four axis machining

Four axis machining refers to machining with three orthogonal axes and a tilted tool vector (See Figure 1.3.2b).

5.2.1 Use of four axis machining in die construction

There are two types of four axis machining applicable to die construction. The first uses a tilted ball nose cutter to machine a contoured surface. This is the same type of machining that is currently used on die contours, with the exception of a tilted tool. The second type uses an end mill type cutter to machine flat regions such as angled plates that are not parallel to any of the three orthogonal (X, Y, Z) axes of a conventional milling machine.

5.2.2 Four axis ball nose machining

With a ball nose cutter, the velocity of the tool where it contacts the part surface (cutter velocity) depends on spindle RPM and cutter effective diameter (Appendix B):

$$V_c = \pi \times RPM \times D_e$$

The effective diameter (D_e) is actually a range of diameters ($D_{e\ min}$ to $D_{e\ max}$) where the tool contacts the part. The cutter velocity (V_c) varies over this range of diameters. Since feed rate is limited by maximum cutter velocity ($V_{c,max}$), V_c is used to denote $V_{c,max}$. Figure 5.2.1 shows the effective diameter range for a conventional three axis cut and a four axis cut.

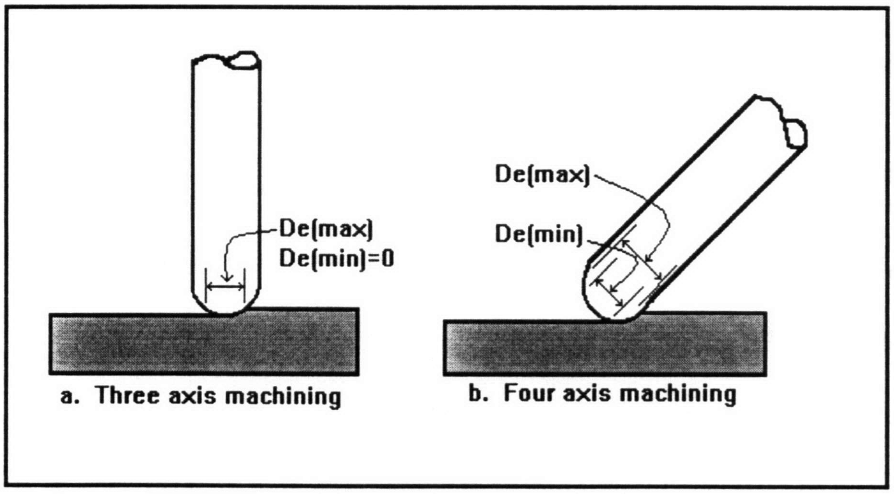


Figure 5.2.1 Effective cutter diameter.

With three axis ball nose cutting, the minimum cutter velocity is zero at the tool center. At this point the tool smears material instead of cutting, which results in:

- Low tool life,
- Poor surface finish, and
- Low feed rate.

<u>Low tool life</u>

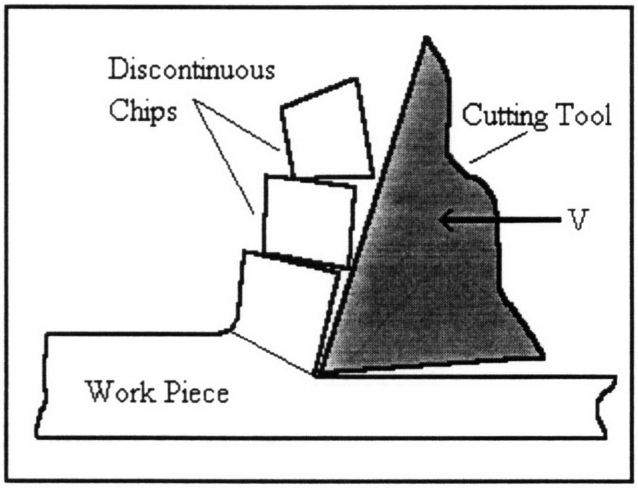


Figure 5.2.2 Discontinuous chips at low cutter speed.

Smearing material tends to 'burn' a tool and reduce tool life significantly. At very low cutting velocities (near the tool center) the tool literally plows through the work piece, forming discontinuous chips (Schey, 1987). See Figure 5.2.2. These discontinuous chips may not have adequate space to clear the cutting zone and may interfere with and damage the tool cutting edge. In fact, research by Tonshoff (1989) did show that tool chipping occurs in the region of the cutting edge near the tool center. Tonshoff also found that the combination of varying chip cross sections, various cutting speeds, and intermittent tool loading led to cyclical and variable mechanical (and thermal) loading on the tool. Variable loading occurs due to varying conditions (i.e. cutting speed) along the periphery of the tool. Cyclical loading occurs due to the intermittent nature of milling. In mill machining, the tool engages the work piece for only a portion of each revolution. The combination of these factors leads to a degradation of tool life. Tonshoff showed tool wear may be reduced by selecting a proper tilt angle. Figure 5.2.3 shows flank wear for various tilt angles when the tool is tilted parallel and perpendicular to the feed direction[8].

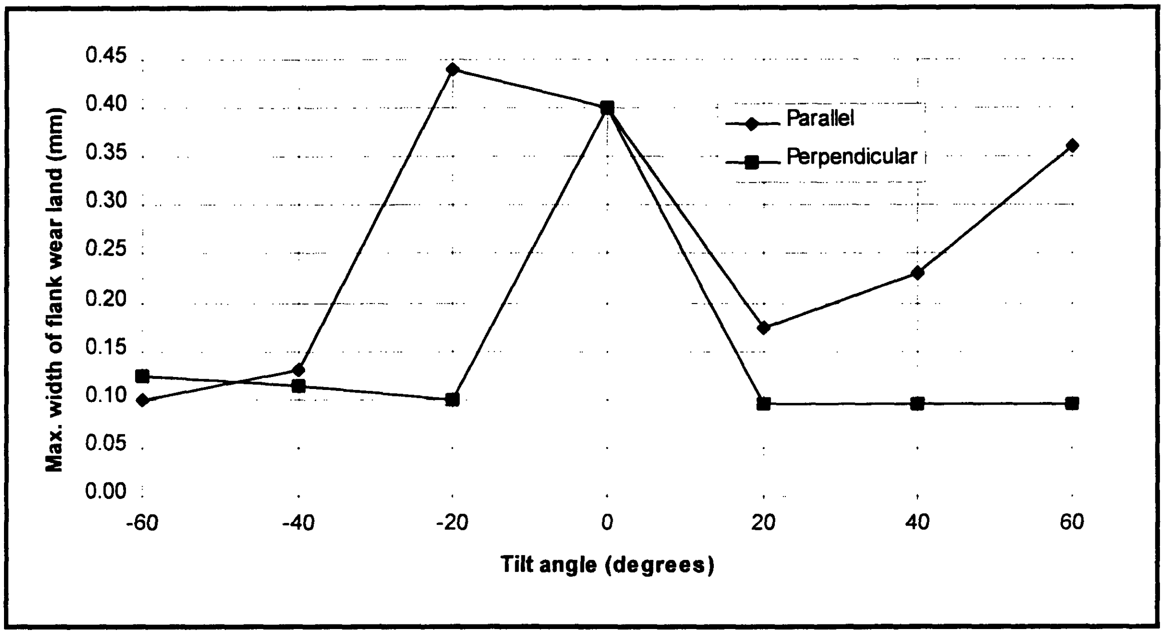


Figure 5.2.3 Flank wear for various tool angles[9].

When the tilt angle is perpendicular to the feed direction, the graph is approximately symmetrical about zero degrees. That is, tool wear is the same for both plunging (tool pointing in the direction of travel) and reverse cutting (tool pointing away from the direction of travel). However, for a tool angle parallel to the direction of travel, the graph is not symmetrical about zero degrees. This indicates that tool wear using an inclined ball nose and climb cutting is quite different than using an inclined ball nose and conventional cutting[10]. Figure 5.2.4 shows the difference between conventional and

---

[8] Flank wear is a standard measure of tool life. Generally, the lower the flank wear, the longer the tool life.

9 From Tonshoff (1989).

[10] Climb cutting is used for most CNC machining applications.

climb cutting. A positive tilt angle indicates a climb type cutting condition, and a negative tilt angle denotes a conventional cut.

From Tonshoff's data, a +/-20 degree tilt angle in the direction of travel (plunge and reverse cutting) will increase tool life four times. It is important to note that the ball nose diameter is much larger than the depth of cut in these experiments (on the order of ten to twenty times larger). Therefore when Tonshoff refers to plunge cutting, the tool center is still well above the part surface, and no smearing of material occurs. Considering these results stem from laboratory conditions and that in practice the tool will not remain at 20 degrees for the entire cut, a two time increase in tool life is assumed here.

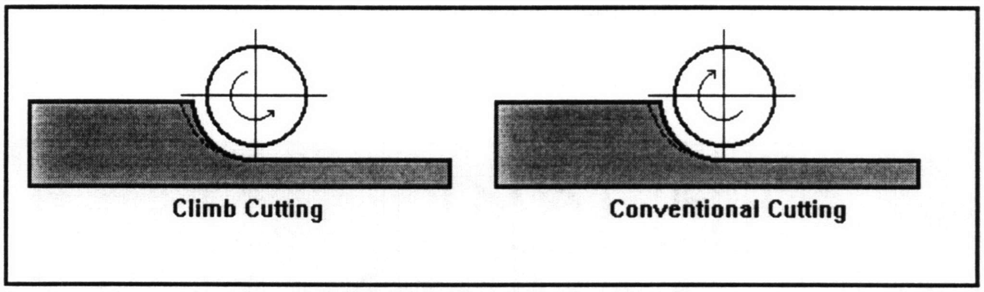


Figure 5.2.4 Climb and conventional cutting.

For three axis ball nose machining, the insert cost to finish machine a die is approximately \$42.00 per die. Each die requires two bottom inserts at \$17 each and one side insert at \$8 each. Tool life savings only applies to the bottom inserts. Doubling tool life for these bottom inserts will save \$17 per die in material cost. Not having to interrupt finish machining to replace a bottom insert will save an additional \$20 per die[11]. The total tooling savings per die is estimated at \$37. Considering die complexity, it is estimated that only 80% of all 3D finishing can accommodate a tilted tool:

$$\frac{\$37.00}{\text{die}} \times \frac{400 \text{ dies}}{\text{year}} \times 80\% = \frac{\$11{,}840}{\text{year}}$$

---

[11] Twelve minutes at \$100/hour.

Poor surface finish

Three axis ball nose machining yields a poorer surface finish than four axis ball nose machining. Two factors contribute to the degraded surface finish. First, the low cutter velocity prevents the tool from cutting near the center of rotation. The tool in effect "smears" the center material and pushes the material to the edges of the cut. Most of the smeared material is removed by the cutter at a larger effective diameter and the remainder piles up on top of the existing cusps much in the same way a snow plow piles snow on the side of a highway. Second, at low speeds, the work piece surface is strain hardened and sufficient strain is generated to shear the surface perpendicular to the tool direction. This sheering causes violent fluctuations in cutting forces that result in a rough, semi-strain hardened and scalloped surface (Schey, 1987). Figure 5.2.5 shows the surface of a three axis ball nose machined die. Figure 5.2.6 shows the surface of a four axis ball nose machined die. Although feed rate and step over were identical in the experiment, the four axis machined surface is smoother.

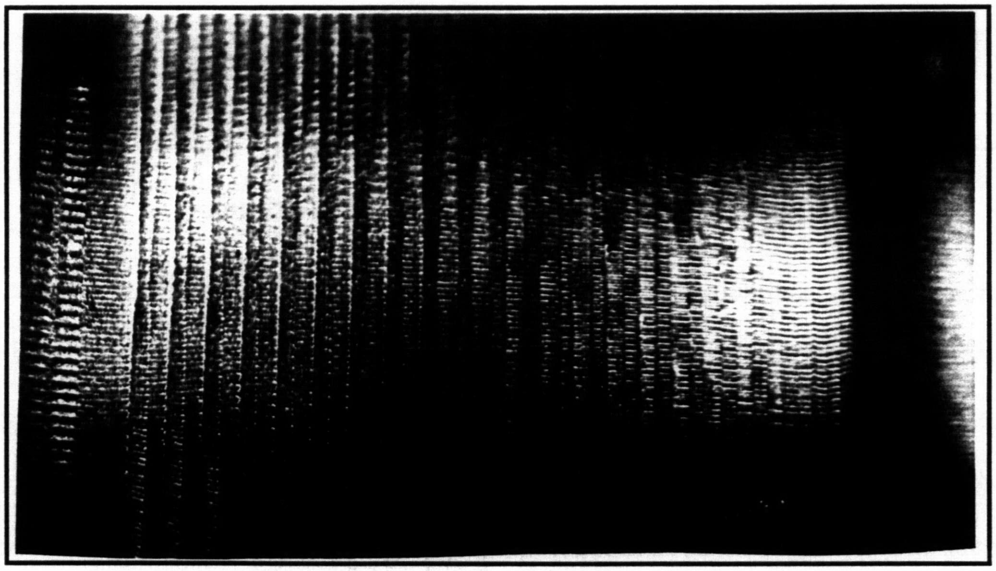


Figure 5.2.5 Three axis ball nose machined die.

Better surface finish and lower cusps translate to savings in bench time. Since no bench experiments were performed, an estimate of savings is not available.

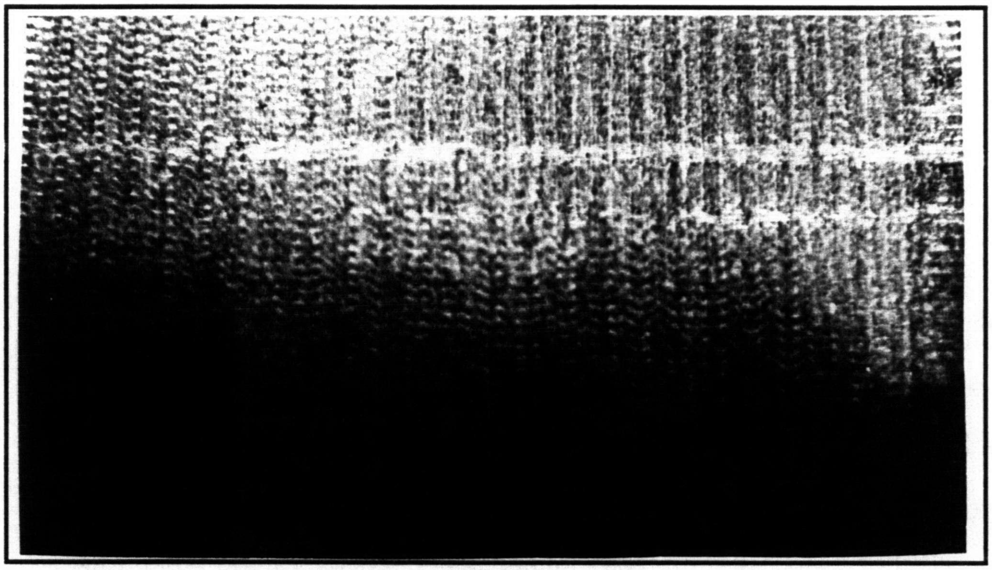


Figure 5.2.6 Four axis ball nose machined die.

Low feed rate

The feed rate for machining depends on cutter characteristics according to the following equation (Appendix A):

$$f = \frac{V_c \times c \times n}{\pi \times D_e}$$

The only variables that change in a move from three to four axis ball nose machining are effective diameter and number of effective flutes. Effective diameter for a three axis cut over a flat surface using a ball nose cutter (See Figure 5.2.1a) varies with tool diameter and depth of cut (*d*) and is given by (Appendix A):

$$D_{e,\,min} = 0$$

$$D_{e,\,max} = 2\sqrt{d(D_{tool} - d)}$$

With four axes, effective diameter varies with the inclination of the tool (ϕ) and depth of cut (*d*) (See Figure 5.2.1b) and varies according to (Appendix A):

$$D_{e,min} = D_{tool} \times \sin(\phi)$$

$$D_{e,max} = D_{tool} \times \sin(\phi + Cos^{-1}(\frac{D_{tool} - 2d}{D_{tool}}))$$

Feed rate is limited by the maximum effective diameter of the tool. Therefore, the maximum effective diameter is used to determine feed rate. The effective diameter is larger with four axis machining which would indicate a lower feed rate. However by tilting the tool, more teeth are brought into the cut. The number of effective teeth (or flutes) refers to the number of cutter teeth which engage the part every revolution. A three axis ball nose cut with an inserted cutting tool[12] generally has only one effective tooth. Rotating the same cutter brings a second tooth into the cut. For example, a 2 inch diameter inserted ball nose cutter with coated carbide inserts (V_C = 9000 inches/min), allowable chip load of 0.015 inches/tooth, and a 0.080 inch depth of cut:

$$\begin{aligned} f_{3axis} &= \frac{9000 \times 0.015 \times 1}{3.14159 \times 2\sqrt{0.08 \times (2 - 0.08)}} \\ &= 54.8 \text{ inches/min} \end{aligned}$$

$$\begin{aligned} f_{4axis} &= \frac{9000 \times 0.015 \times 2}{3.14159 \times 2 \sin(20^{o} + Cos^{-1}(\frac{(2 - 2 \times 0.080)}{2}))} \\ &= 62.9 \text{ inches/min} \end{aligned}$$

Note: Four axis feed rate is based on a 20 degree tool angle.

---

[12] Commonly used for die machining.

The four axis feed rate is 15% higher than the three axis feed rate. A typical machining center cuts for approximately 40% of available time. Of that time 40% is used for 3D machining, of which 60% is rough machining and 40% is finish machining[13]. Therefore a machining center is occupied 6% of the time with 3D finish machining work. The realizable machine time savings from four axis ball nose machining are:

$$\frac{500\text{ hours}}{\text{die}} \times 6\% = \frac{30\text{ hours}}{\text{die}} \text{ finish machining}$$

$$\frac{30\text{ hours}}{\text{die}} \times \frac{400\text{ dies}}{\text{year}} \times 15\% = \frac{1800\text{ hours}}{\text{year}}$$

$$\frac{1800\text{ hours}}{\text{year}} \times \frac{\$100}{\text{hour}} = \frac{\$180{,}000}{\text{year}}$$

The exact tilt angle must consider tool life, bench time, and machine time savings for each particular die plant. If the machining centers are a process bottleneck, than maximizing feed rate becomes the over-riding goal. In such a case, the tilt angle should be increased to the point where two teeth become effective, assuming that the time saved from the feed rate gain is larger than the time lost to insert replacement. If neither of the three are bottleneck processes, the tilt angle should be chosen to maximize total savings from tool life, bench, and machining time. Once the required information is collected, the maximum savings for each particular die plant can be determined using a fairly simple mathematical program.

The four axis ball nose cut will yield a longer tool life, better surface finish and allow a higher feed rate compared to a three axis ball nose cut.

---

[13] Again, this estimate must be modified to suit the particular die plant.

5.2.3 Four axis end mill machining

Additional benefits of four axis machining for die construction are possible with four axis end mill machining. Approximately one in every four die sets[14] has a cam mechanism for translating the vertical movement of a stamping press into a horizontal movement (See Figure 5.2.7). Cams are used to impart flanges or breaks into panels. A cam consists of a cam driver, cam & die adapters and cam slide. Current die development practices call for cam parts to be designed, cast, and manufactured as separate die details. Once completed, the details are assembled into a die shoe (See Figure 1.1.1) during the assembly process.

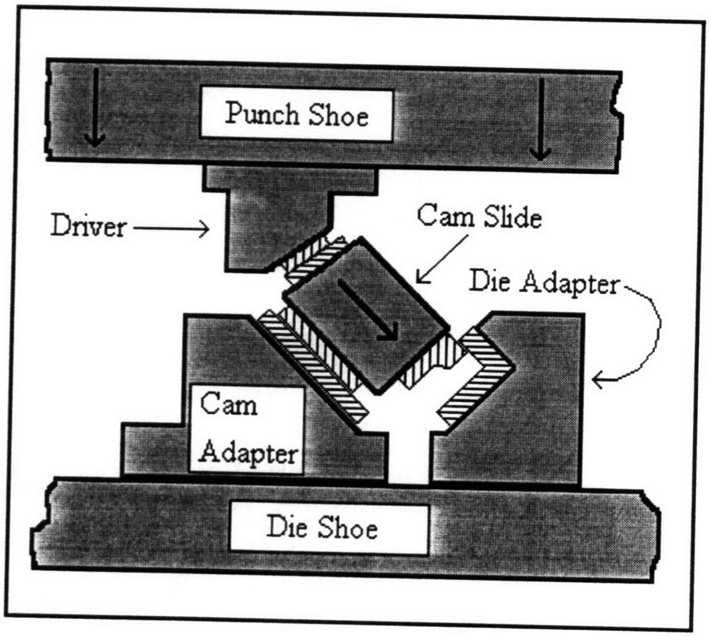


Figure 5.2.7 Cam assembly.

The process flexibility offered by four axis machining allow the driver and adapter details to be cast directly into the die shoe. During the machining process the five axis spindle is loaded with an end mill cutter, rotated to the specified cam angle, and the cam parts are machined in place. The savings that can be realized by using four axis end mill machining are:

- Reduced design time. Die designers need not detail individual cam parts or the mating surfaces for those parts.
- Fewer parts. Die makers have fewer parts to handle and assemble.

---

[14] A die set is a series of dies that form a part.

- Less machining time. Less time to setup and program parts, and less machining required since the mating surfaces and key ways need not be machined.
- More accurate location. Consolidating parts eliminates problems which may occur due to tolerance stack-up.

The dollar savings for casting cam parts directly into a die shoe varies from company to company. The major U.S. auto maker requires an estimated 350 new panels (or die sets) each year (See Chapter 1). Approximately 25% of these panels require a cam flange die and each cam flange die requires two adapters and a driver mechanism. The total number of parts that can be consolidated is approximately 260. The die construction[15] savings with four axis end mill machining are based on incorporating the 260 parts directly into die shoes.

Estimated setup and machining time saved per part: 20 hours[16]

Estimated assembly time saved per part: 35 hours

Total time saved: <u>55 hours per part.</u>

$$\frac{55\ \text{hours}}{\text{part}} \times \frac{\$75.00}{\text{hour}} \times \frac{260\ \text{parts}}{\text{year}} = \$1.07\ \text{Million}\big/_{\text{year}}\ ^{17}$$

In effect, four axis end mill machining offers a process flexibility in machining which can be exploited during die design to affect significant savings in machining and assembly.

---

[15] Design time savings were not available and therefore not included.

[16] Again, estimates vary plant to plant.

[17] The $75 per hour is a composite of machining, setup, and assembly costs.

5.3 Simultaneous five axis machining

Five axis contouring is traditionally considered 'true' five axis machining and refers to simultaneously moving five axes while cutting a part (See Figure 1.3.2c).

5.3.1 Five axis contour machining

Figure 1.3.1 illustrates the advantage of using five axis instead of three axis to machine contoured surfaces. The two rotary axes allows a flat bottom tool to be aligned to the surface normal. In practice, the tool is inclined several degrees from the surface normal in the direction of tool travel to avoid contact between the tool center and part surface. The incline is called a heel angle; it avoids the tool wear and metal smearing problems addressed in section 5.2.2. While a ball nose tool leaves a cusp profile defined by a series of circles, an inclined flat bottom tool leaves a cusp profile defined by a series of ellipses (See Figure 5.3.1). The elliptical surface matches the part surface more closely and results in:

- Lower cusp height
- Less material to remove by hand grinding
- Less machining time
- Smoother part surface

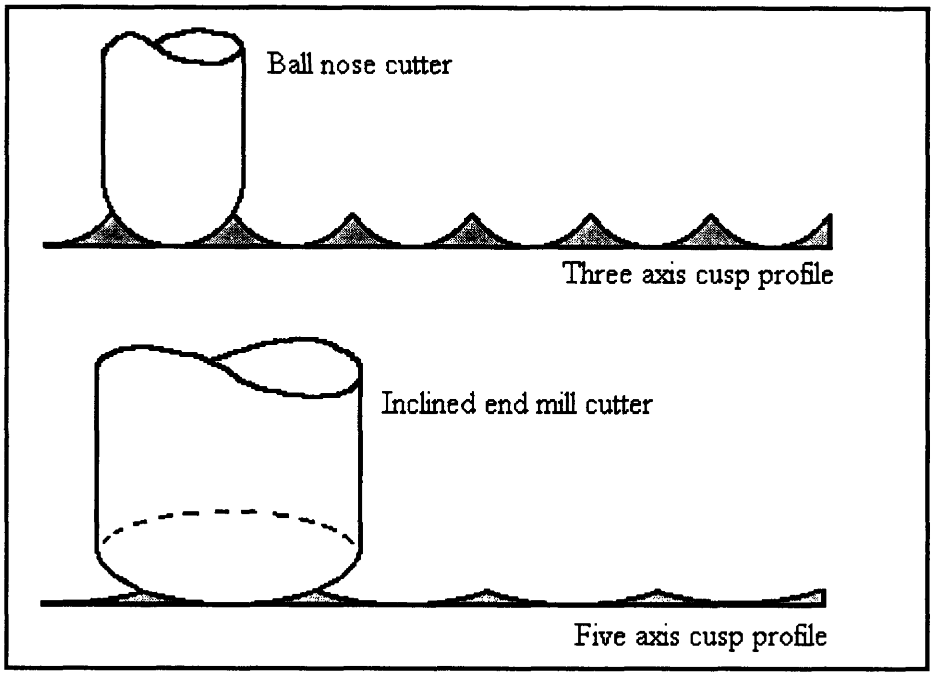


Figure 5.3.1 Three and five axis cusp profiles.

Cusp height

Cusp height (h) using a ball nose tool over a flat surface[18] (See Figure 5.3.2) varies according to the equation (Appendix A):

$$h = \tfrac{1}{2}(D - \sqrt{D^2 - \delta^2})$$

Variables are defined at the front of the paper.

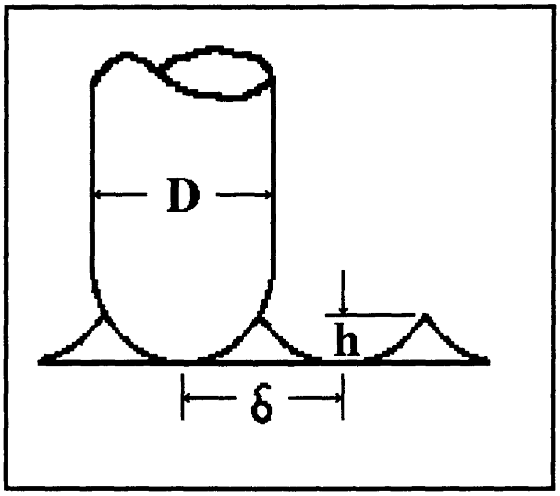


Figure 5.3.2 Three axis cusp height.

The cusp height (h) using an inclined end mill with a corner radius varies as (Appendix A):

$$h = (\tfrac{D}{2} + r\sin(\phi))\sin(\phi) \times$$

$$(1 - \sqrt{1 - \frac{\delta^2}{4 \times (\frac{D}{2} + r \times \sin(\phi))^2}})$$

An end mill with a corner radius is called a radius end mill or toroidal cutter. It is an end mill with a rounded bottom edge.

Note: Figure 5.3.3 shows an inclined end mill with no corner radius (r=0).

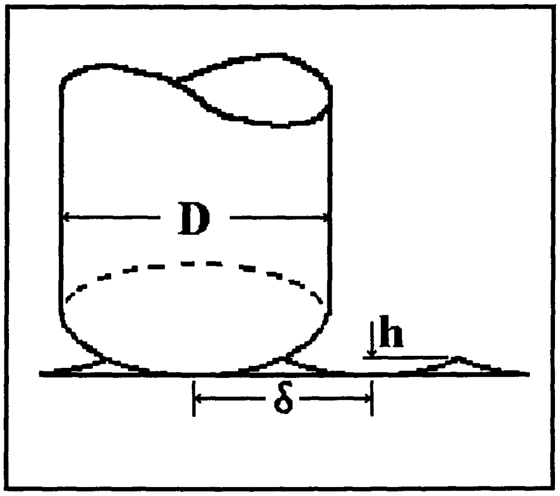


Figure 5.3.3 Five axis cusp height.

---

[18] Calculations are presented for flat part surfaces. Equations for curved surfaces are much more complicated and yield very similar results (See Appendix A).

For a given tool diameter and step over, the five axis (end mill) cusp height is much lower than the three axis (ball nose) cusp height. For example; a 2 inch diameter ball nose with a 0.1 inch step over yields a cusp height of 0.0013 inches. Using a 2 inch end mill with no corner radius, 0.1 inch step over and 5 degree heel angle yields a cusp height of 0.0001 inches, or one tenth of the three axis cusp height. Figure 5.3.4 shows the variation of cusp height with varying step over for 1 and 2 inch diameter tools. The five axis cusp height results in Figure 5.3.4 are generated with the above equations using a 5 degree heel angle and no end mill corner radius (r=0).

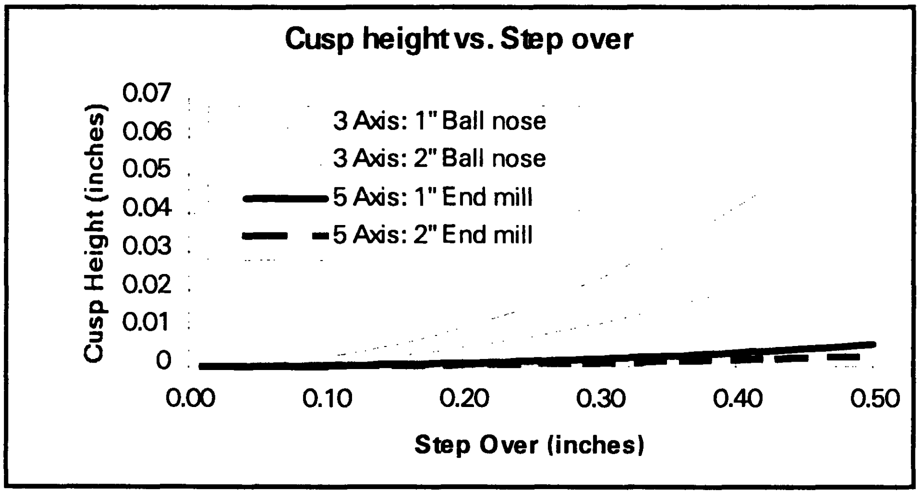


Figure 5.3.4 Three versus Five axis cusp height.

Material remaining

The material required to be removed by hand grinding and polishing equals the volume of cusp material remaining on the surface. For three axis ball nose machining over a flat or curved surface[19], the material to be removed is given by (Appendix A):

$$V = \frac{D}{2} - \frac{1}{4}\sqrt{D^2 - \delta^2} - \frac{D^2}{4\delta} Sin^{-1}(\frac{\delta}{D})$$

For five axis contour machining, the material remaining is the sum of cusp and wake material. Since the tool only cuts when a cutter is engaged, the machining process can be modeled as a series of evenly spaced discrete cuts. The result is a track of material (or wake) that is uncut. The space between these discrete cuts is determined by feed rate, spindle speed, and number of cutters on the tool periphery. Wake material refers to the material missed by the cutter as it advances over the part (See Figure 5.3.5).

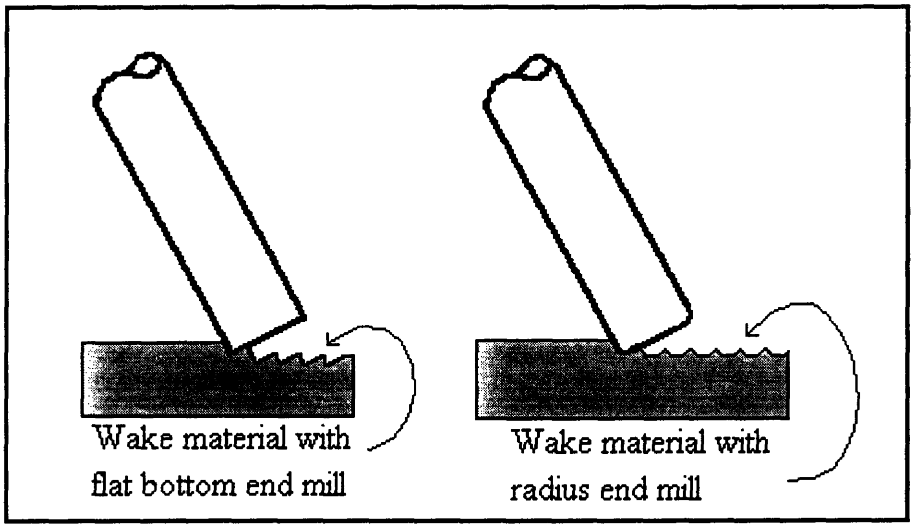


Figure 5.3.5 Wake material for flat bottom and radius end mills.

Observing Figure 5.3.5, it is obvious that a radius end mill cutter is much more effective than a flat bottom end mill in removing wake material. A radius edge is also more

---

[19] The difference is negligible for surfaces with curvature less than 10% of the ball nose curvature. See Appendix A.

resistant to breakage compared to a sharp edge. For these reasons, radius end mill cutters are generally used for five axis machining. Depending on cutter diameter, corner radius and tool heel angle, a radius end mill cutter will increase the cusp height between 5 and 25%. However, the reduction of wake material from using a radius cutter dwarfs the increase in cusp material (See Appendix A).

For five axis contour machining over a flat surface with a radius end mill cutter, the material to be removed is the sum of cusp and wake material remaining. Remaining cusp material is given by (Appendix A):

$$V_{cusp} = (\frac{D}{2} + r\sin\phi) \times r\sin\phi - \frac{\sin\phi}{2} \times \sqrt{(\frac{D}{2} + r\sin\phi)^2 - \frac{\delta^2}{4}} - \frac{(\frac{D}{2} + r\sin\phi)^2}{\delta} \times \sin\phi \times Sin^{-1}(\frac{\delta}{D + 2r\sin\phi})$$

Remaining wake material is given by (Appendix A):

$$V_{wake} = A \times A^{*} \times \frac{\omega \times n}{f \times d}$$

Where;

$$A = \frac{\delta\sin\phi}{2} \times \sqrt{(\frac{D}{2} + r\sin\phi)^2 - \frac{\delta^2}{4}} + (\frac{D}{2} + r\sin\phi)^2 \times \sin\phi \times Sin^{-1}(\frac{\delta}{D + 2r\sin\phi}) + \delta d - \delta\sin\phi \times (\frac{D}{2} + r\sin\phi)$$

and

$$A^{*} = \frac{r \times f}{n \times \omega} - \frac{f}{2n \times \omega}\sqrt{r^2 - \frac{f^2}{4n^2 \times \omega^2}} - r^2 Sin^{-1}\frac{f}{2n \times \omega \times r}$$

Total material remaining equals:

$$V = V_{cusp} + V_{wake}$$

Figure 5.3.6 compares material remaining for a three and five axis cut. The parameters from Figure 5.3.4 are used.

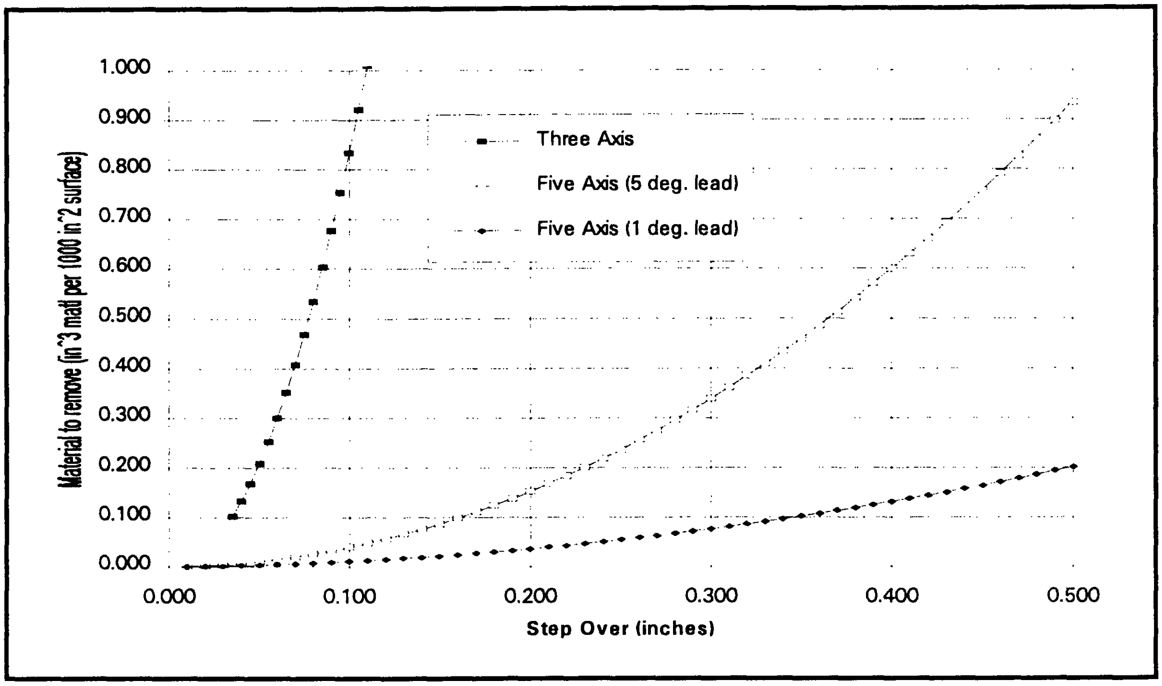


Figure 5.3.6 Material remaining: Three versus five axis.

Machining time

With all contour machining there is a trade-off between machine time and bench time[20]. By decreasing step over in any of the above equations, both cusp height and material to remove decrease, reducing bench time. However, a smaller step calls for more passes to cover a given area, increasing machine time. A typical trade-off curve for three axis contour machining is shown in Figure 5.3.7.

---

[20] Bench time is the time required to grind and stone polish the die surface by hand. Material to be removed is an indication of bench time.

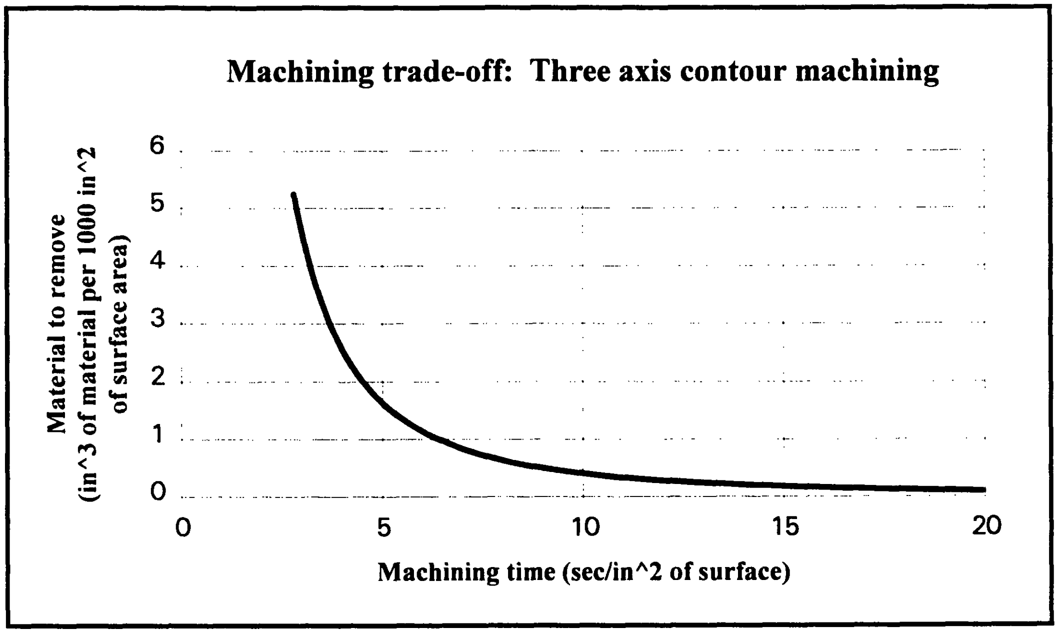


Figure 5.3.7 Trade-off between machine time and bench time.

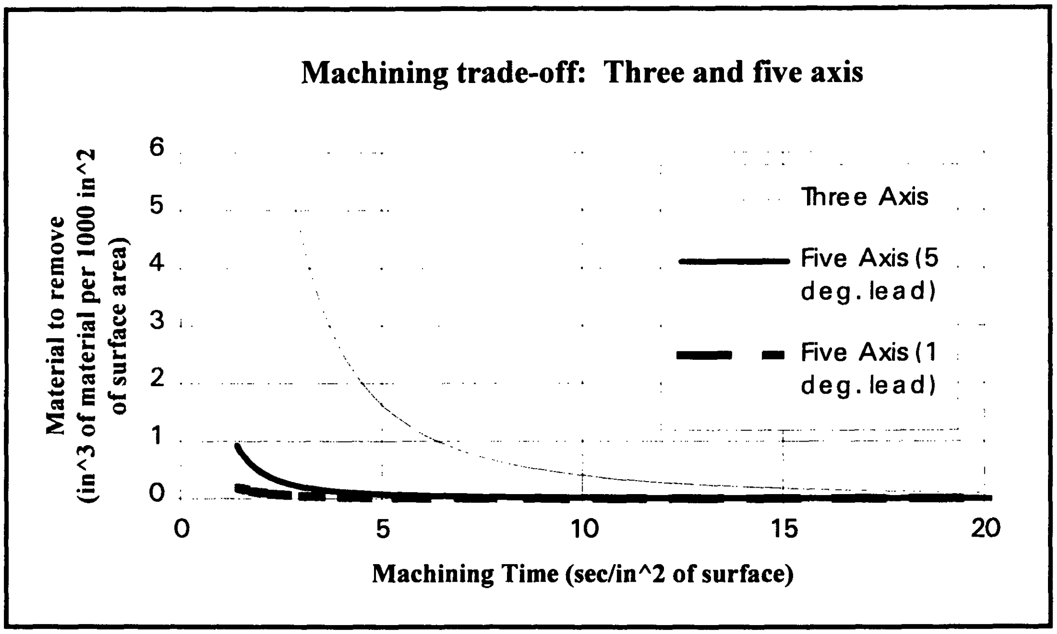


Figure 5.3.8 Three and five axis trade-off curves.

Five axis contour machining will shift this trade-off curve towards the origin. Figure 5.3.8 shows the traditional three axis trade-off curve and two five axis curves, the first with a 5 degree tool heel angle and the second with a 1 degree tool heel angle.

With less than 0.25 in^3 of material to remove per 1000 in^2 of surface area, hand grinding is no longer required. At this point the die need only be stone polished[21]. Therefore, once the material to remove reaches this level, there is no further benefit in increasing machine time; it will have no effect on polishing time and no further affect on bench time.

The results presented here compare three and five axis machining using the same tool diameter (i.e. 2 inch ball nose and 2 inch end mill). Machining a contour with five axis allows a CAM programmer to select a larger diameter tool. The benefit of a larger tool is it further reduces material remaining on the part surface and in effect pushes the machining trade-off curve further down Figure 5.3.9 shows five axis trade-off curves for a two and four inch diameter end mill with 1 degree heel angle and 1/8 inch corner radius.

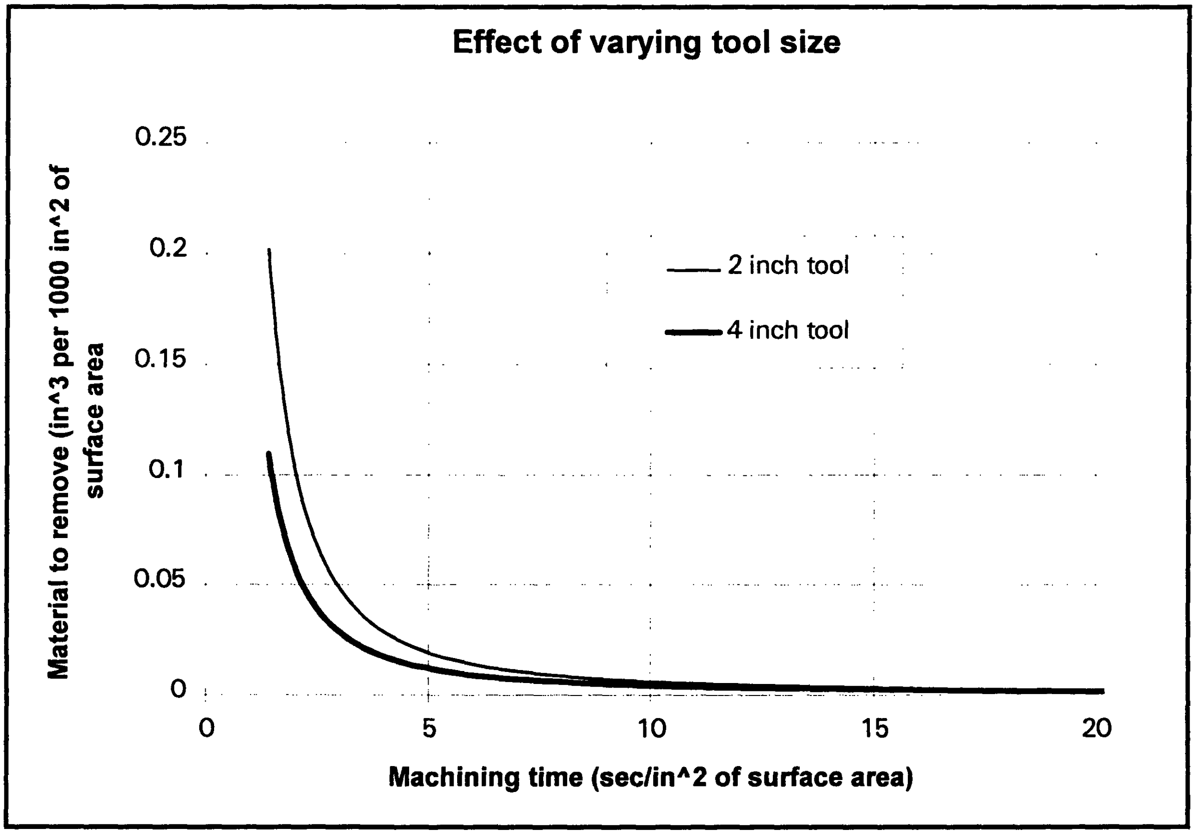


Figure 5.3.9 Five axis trade-off curves for different tool diameters.

---

[21] Schey (1987) and empirical data from die builders.

Part surface quality

The cutter velocity for five axis contour machining remains constant at (Appendix A):

$$V_c = \pi \times RPM \times D_e$$

The heel angle ensures the tool center does not contact the part surface, eliminating surface problems encountered with three axis ball nose machining (Explained in Section 5.2.2).

5.3.2 Use of five axis contouring in die construction

To evaluate simultaneous five axis machining (both contouring and swarf cutting), a CAM programming experiment was performed. The purpose of the experiment was to evaluate benefits of simultaneous five axis machining, determine what dies and die regions are ideal candidates for five axis machining, and determine process and equipment requirements to realize the benefits. The experiment involved CAM programming the draw contour die of two parts; a hood outer and center pillar male die. Each part was programmed twice: once using three axis and once using five axis machining. The CAM experiment is documented in Appendix C.

Results of the CAM experiment are shown in Figure 5.3.10. The hood outer is an ideal candidate for five axis contour machining. Machining time was reduced by 65% for the hood die. The center pillar die required a combination of three, four and five axis machining to complete. For that reason, the center pillar is not a good candidate for five axis machining. The CAM experiment is discussed in more detail in Chapter 6. A conclusion from the CAM experiment was that large, gently curving surfaces are ideal candidates for five axis contour machining. Major automotive outer panels such as hoods, roofs and doors are mainly composed of large gently sloping surfaces and are excellent candidates for five axis machining.

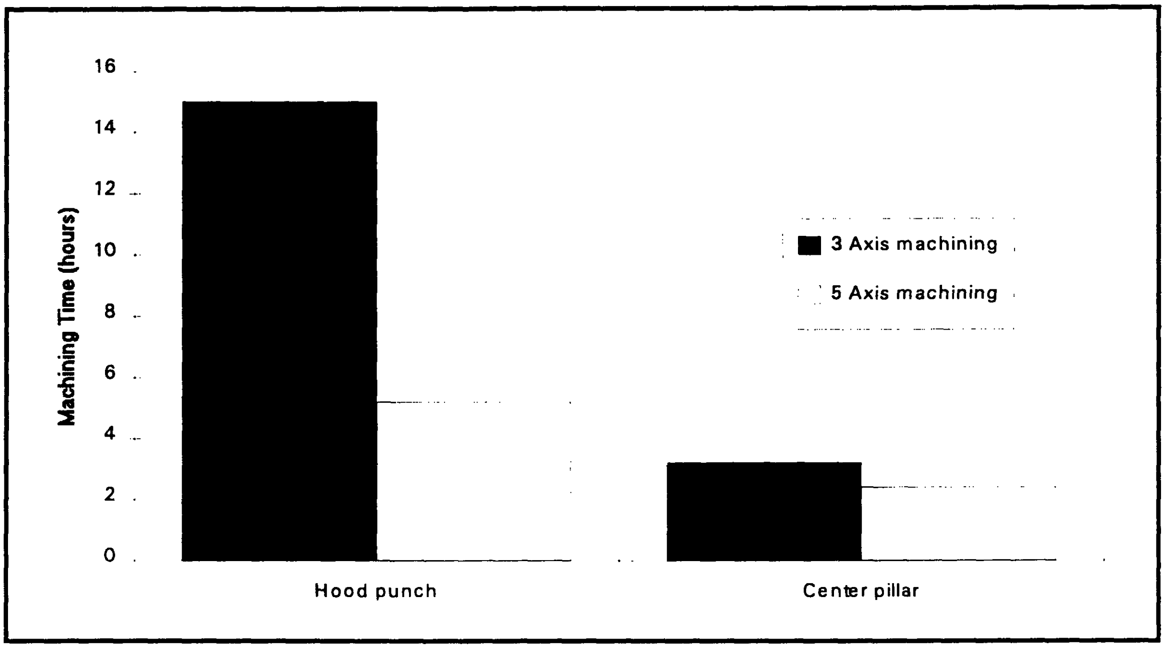


Figure 5.3.10 CAM experiment results.

5.3.3 Five axis contouring of outer panel draw dies

Five axis contouring is considered for 3D contour finishing only. Contour roughing is not considered. Five axis contour machining can eliminate hand grinding in the bench process (Section 5.3.1) and is ideally suited to large, gently sloping surfaces such as outer panel dies (Section 5.3.2). Approximately 6% of all machining and bench work done by a typical plant of the major U.S. auto maker is for outer panel draw dies; these are excellent candidates for five axis machining. A typical outer panel draw die requires 320 bench hours; 106 hours grinding and 214 hours polishing. Using the strategy recommended in Section 5.3.1, step over is decreased to the point where hand grinding is no longer required. At this point, machining time is reduced by 65% (See Appendix C) compared to traditional three axis machining with a ball nose cutter. Given that 30 hours/die are available for 3D contour finishing (Section 5.2.2), the savings for a die plant that produces 400 dies per year is:

Machine Time:

$$\frac{30\text{ hours}}{\text{die}} \times 65\% \times \frac{400\text{ dies}}{\text{year}} \times \frac{\$100}{\text{hour}} \times 6\%\text{ outer draw dies} = \frac{\$46{,}800}{\text{year}}$$

Bench Time:

$$\frac{106\text{ hours saved}}{\text{die}} \times \frac{400\text{ dies}}{\text{year}} \times 6\%\text{ outer draw} = \frac{2540\text{ hours}}{\text{year}}$$

$$\frac{2540\text{ hours}}{\text{year}} \times \frac{\$50}{\text{hour}} = \frac{\$127{,}000}{\text{year}}$$

$$\text{Total Savings: } \$46{,}800 + \$127{,}000 = \frac{\$173{,}800}{\text{year}}$$

Cusp height was selected to maximize the sum of machining and bench savings. If the machining centers are a process bottleneck, cusp height should be selected to minimize machining time. Note: Bench costs ($50/hour) are estimated and vary widely from plant to plant.

5.3.4 Five axis swarf cutting

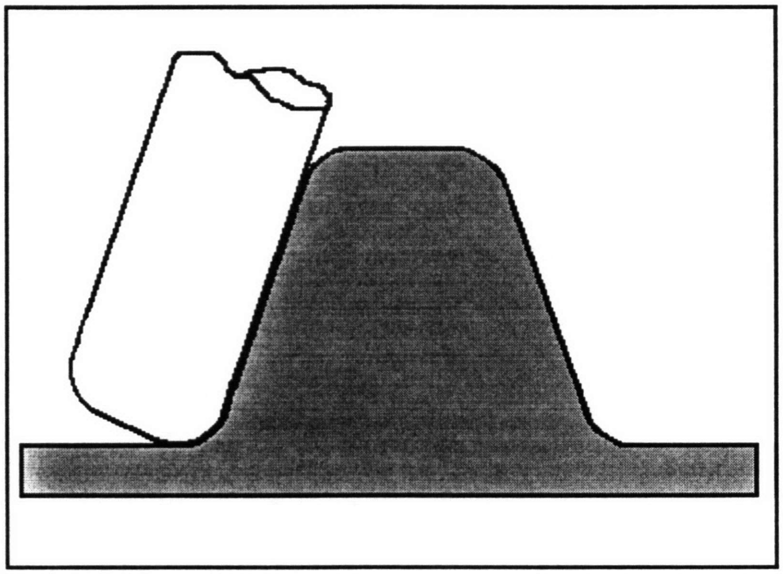


Figure 5.3.11 Five axis swarf cut.

Swarf cutting refers to simultaneously machining with the side and bottom of a cutting tool (See Figure 1.3.2d).

Automotive major body inner panels such as a hood, door, or deck lid inner contain reinforcing ribs that provide outer panel strength. In cross section these ribs contain straight line segments that are ideal candidates for five axis swarf cutting. A typical five axis swarf cut of a rib section is shown in Figure 5.3.11. Currently, a common practice used to machine inner panel ribs in three axis requires flow cutting the inner radii followed by bump cutting the rib. The flow cut clears out material at the lower radii to avoid plunging the tool during the bump cycle. In bump machining, the tool traces out the rib cross section (across left side base, up left side web, across flange, down right side web, across right side base), increments along the length of the rib (step over), then traces out the cross section again (across right side base, up right side web, across flange, down left side web, across left side base). This process is repeated for the entire rib.

Use of simultaneous five axis machining for inner panel ribs will eliminate both the flow and bump cuts to yield much lower machine and bench times. The process to five axis swarf cut a rib is as follows:

1. Five axis contour cut bases and flanges.
2. Five axis swarf cut web and lower radii.
3. Five axis contour cut upper radii with a radius form tool.

Although the five axis swarf cut requires an additional (non standard) radius form tool, the subsequent savings in machining and bench time should more than make up for the costs of carrying additional tools. Proliferation of additional tools can be avoided by standardizing the outer radii on ribs that are swarf cut.

Although the potential savings of five axis swarf cutting seem quite large, they have not been quantified here. A significant process, equipment, and organizational change is required to realize these savings at the major U.S. auto maker. For this reason, savings for five axis swarf cutting was not pursued. These process, equipment and organizational changes are discussed in Chapter 6.

5.4 Chapter summary

There are significant achievable benefits to five axis machining dies. The primary benefits are in drilling coordinating holes, four axis ball nose machining, four axis end mill machining, and five axis contour machining of outer panel draw dies. Four axis ball nose machining allows higher feed rates and results in longer tool life and better surface finish for 3D finish contour machining. Five axis contour machining offers savings in machine and bench time for outer panel dies by providing a nearer net shape part, allowing larger step over, and producing a smoother part surface.

Four axis end mill machining offers a process flexibility that can be exploited by reaching beyond the die plant boundaries and into die design and engineering. The added flexibility also provides a specialization that can be use to generate revenue from sources other than die building.

Although the benefits of five axis swarf cutting seem considerable, the organizational and process changes required to exploit these benefits are substantial and difficult to quantify. For these reasons costs and benefits of five axis swarf cutting were not estimated.

The benefits quantified in this chapter are compiled in Section 7.1: Net present value analysis. The next chapter will look at equipment and process requirements to exploit the benefits cited in this chapter.

6.0 Equipment and process requirements for five axis machining

To take advantage of the benefits of five axis machining in die development, certain equipment and process requirements must be met. These requirements assume the die developer wishes to take full advantage of its five axis technology, meaning it strives to realize all the benefits listed in Chapter 5 without increasing cost or reducing quality.

Figure 6.0.1 shows an Ishikawa diagram used to determine root causes of why five axis machining is not currently used by the major U.S. auto maker. The chart was generated with the aid of die plant personnel and central engineering staff of the major U.S. auto maker. Some common root causes from the diagram are:

- Inadequate training in five axis technology,
- Poor understanding of the technology,
- Lack of confidence in the five axis equipment,
- Animosity between auto maker and equipment supplier,
- Poor communication between die plants, central engineering, and designers,
- Deficiencies with the CAM software.

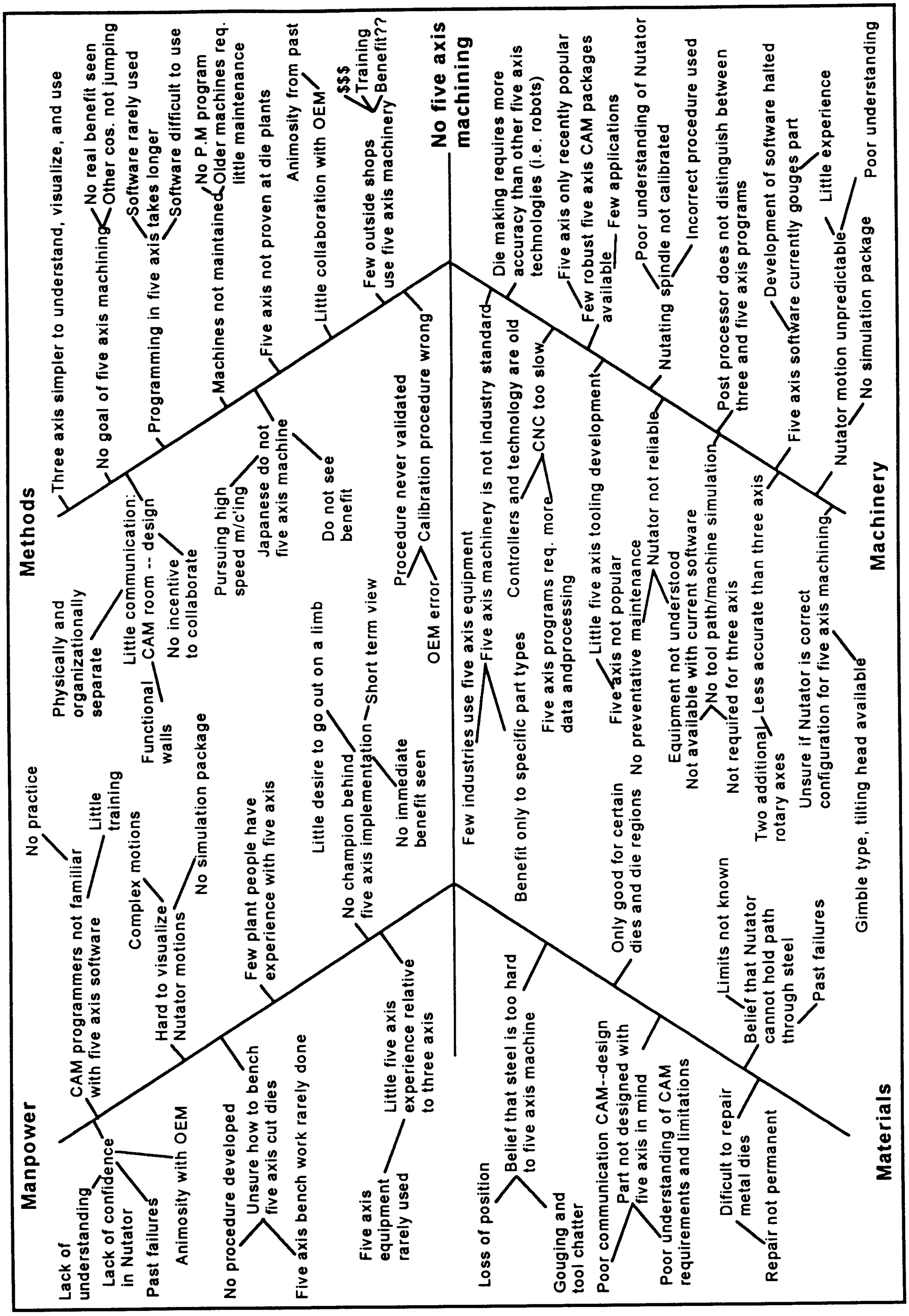


Figure 6.0.1 Ishikawa diagram for five axis machining.

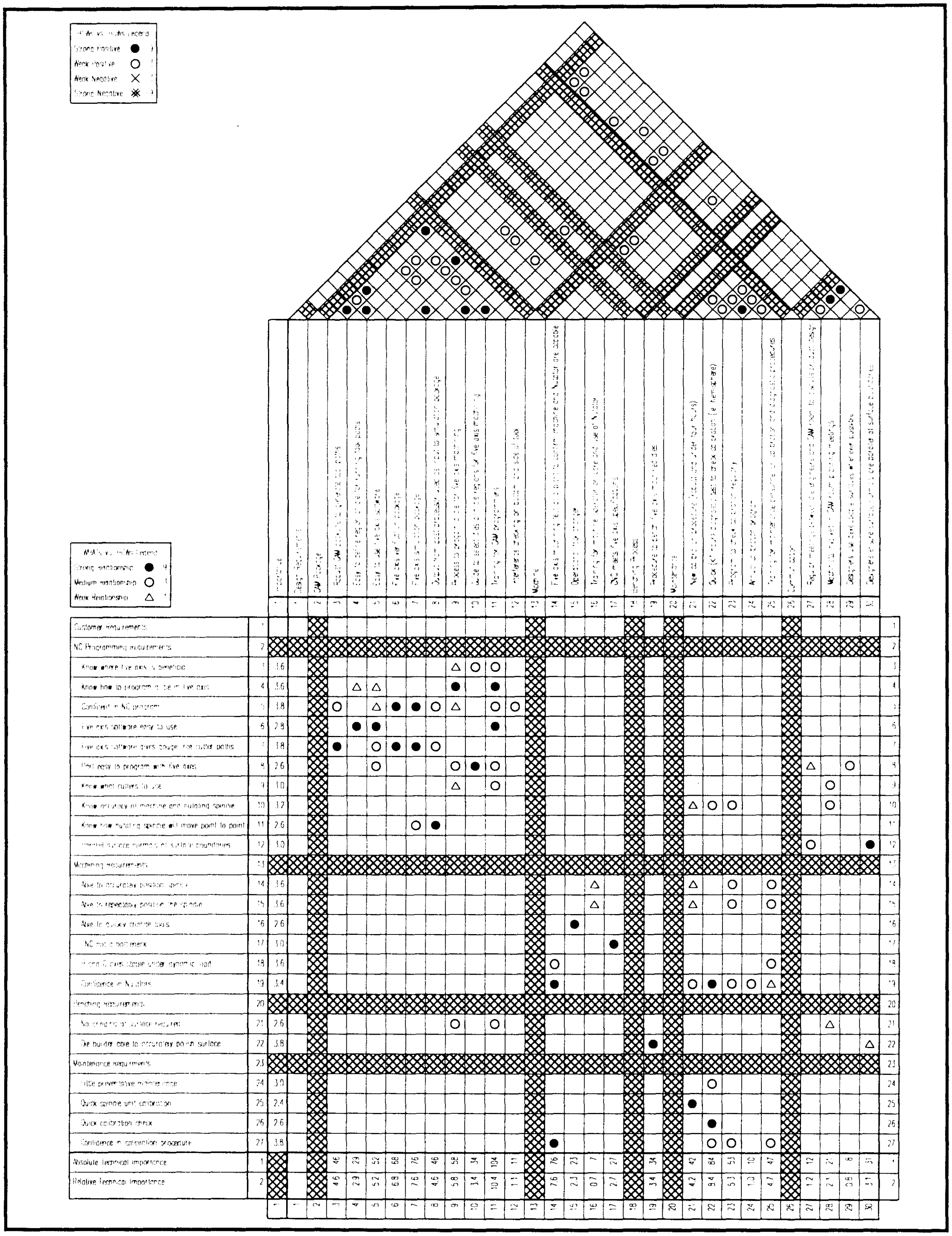


Figure 6.0.2 QFD matrix for five axis machining.

Design Requirement	Importance Ranking
Training for CAM programmers	10.4
Quick diagnostic test to check calibration	8.4
Five axis simulation package	7.6
Five axis machining test to confirm Nutator™ capabilities	7.6
Five axis verification package	6.8
Process to program a die for five axis machining	5.8
Program to check calibration regularly	5.3
Easy to use five axis software	5.2
Training for maintenance personnel	4.7
Robust CAM package to generate tool paths	4.6
Output from post processor is input into simulation package	4.6
New, robust calibration procedure	4.2
Guide to select dies and die regions for five axis machining	3.4
Procedure to bench five axis cut dies	3.4
Designers ensure surface normals are parallel at boundaries	3.1
Easy to define region of die for running tool paths	2.9
CNC meets five axis specifications	2.7
Operational tool changer	2.3
Machinists involved in CAM room planning meetings	2.1
Regular meetings between design and CAM room	1.2
Interference checking on bottom and side of tool	1.1
Annual calibration program	1.0
Designers use 'developable' surfaces whenever possible	0.8
Training for machine operators on care and use of Nutators™	0.7

Table 2. QFD design requirements ranked in order of importance.

The root causes were combined with customer[22] voices to generate a set of requirements for a QFD[23] matrix (See Figure 6.0.2). A survey of customers provided an importance ranking for the customer voices. Design requirements were generated to address the various customer requirements. The combination of how well the design requirements addressed customer voices and the ranking of customer voices provided an importance ranking of design requirements (See Table 2).

Many of the above requirements are not needed for the first stages of five axis machining (i.e. five axis drilling and four axis machining). Where this is the case, the early stage requirements will be specified. Although training requirements rank higher than both machining center and CAM software requirements, an understanding of the latter two is required to appreciate the former. For this reason, machining center and CAM software requirements will precede training requirements.

---

[22] The customers were CAM programmers, die makers, and machinists.

[23] Quality Functional Deployment; a Total Quality Management (TQM) tool.

6.1 Machining center

The exact requirements of a machining center to perform five axis work are dependent on the type of equipment and die manufacturing process employed. Although most of the requirements that follow are *generally* applicable to any five axis equipment or die construction process, they are specifically based on the equipment and processes employed by the major U.S. auto maker who is the focus of this study. The machining center requirements are broken into spindle unit, validation, and maintenance requirements.

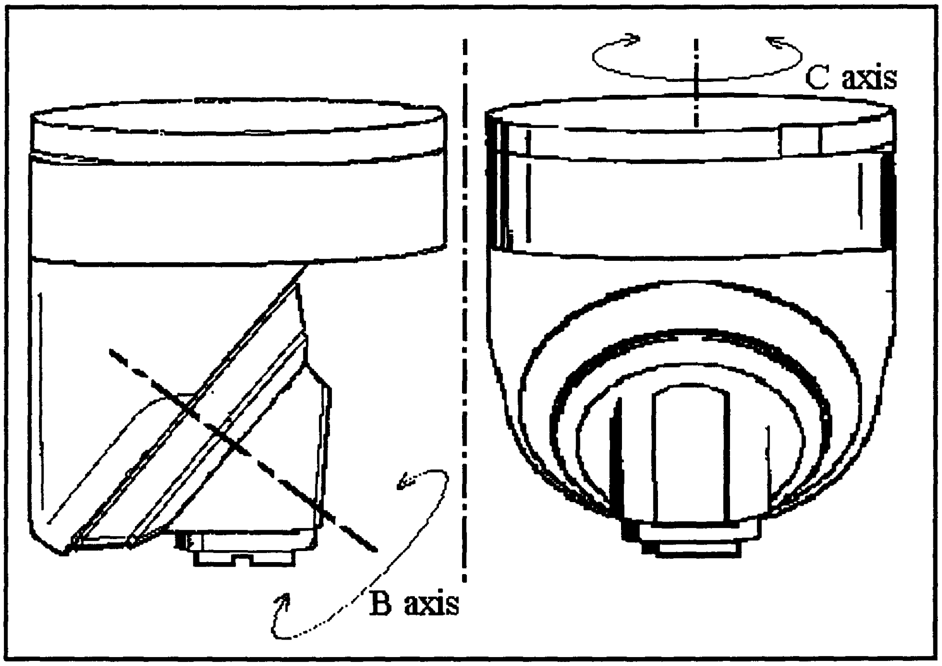


Figure 6.1.1 Nutating type spindle.

6.1.1 Five axis spindle unit

A five axis spindle unit is a detachable head that mounts on a machining center and provides the machine with two additional (rotary) axes[24]. A nutating type five axis spindle is shown in Figure 6.1.1. The five axis spindle must be able to accurately and repeatably position a tool tip to a known position in a given orientation. To do this while machining, the axis servo drives must be able to resist the machining forces imposed on it. Cutting forces in machining depend on cutter geometry, part geometry, and material to be cut according to:

$$F_x = \sum (F_r \cos(\alpha) + F_t \sin(\alpha))$$

$$F_y = \sum (F_r \sin(\alpha) - F_t \cos(\alpha))$$

$$F_z = \sum F_a \times (\alpha)$$

Where,

$$F_r = -F \sin(\gamma_r) \times \cos(\gamma_a)$$

$$F_t = F \cos(\gamma_r) \times \cos(\gamma_a)$$

$$F_a = F \sin(\gamma_a)$$

And,

$$F = A \times K_s$$

See Appendix B for an explanation of these equations. Using the above equations together with the cutter and part geometry, material to cut, and type of five axis spindle (nutating, trunion, tilting, etc.), it is possible to determine servo drive requirements. It is important to keep in mind that the above equations are empirical in nature and are only valid to within an order of magnitude estimate.

---

[24] In many cases the C axis is incorporated into the machine (rotation about the machine Z axis) and the five axis spindle only adds one additional axis.

To repeatably position a tool tip on a part, the error contributed by the rotary axes must not exceed the total allowable error budget for the machining center. The machining center error budget is that portion of final part tolerance which is allocated to the machining center. So long as the two rotary axes do not increase machine error beyond this allowable level, the five axis spindle will repeatably position the tool tip.

To accurately position a tool tip the CNC controller must know the *exact* spindle unit and tool dimensions. Any error in pivot or offset lengths are not only multiplied by a rotary axis but will also vary depending on the rotary axis position. For example, say a machining center with a nutating spindle contained a B axis error of 1 arc minute. With a 6 inch tool and 10 inch nutating pivot length (typical), a 1 arc minute error leads to a 0.005 inch error. When the tool points straight down, the error is in the Y dimension, when the tool is horizontal, the error is in the X and Z dimensions. A linear (X, Y, Z) axis error on the other hand, is not multiplied, and always affects the same dimension in the same direction.

To accurately determine the pivot lengths and offsets for a nutating spindle unit, a robust calibration procedure is required. This procedure is used by a maintenance mechanic to determine exact spindle unit dimensions. The order of the procedure is critical to avoid confounding offsets. For a nutating spindle, the required order to accurately determine offsets is :

- B axis offset,
- Nutating angle and C axis offset,
- X and Y axis offset,
- Pivot length and spindle unit overall length.

The B and C axes are shown in Figure 6.1.1. The nutating angle is defined as the acute angle between the B and C axes. The X and Y offsets are the horizontal displacement of the spindle center of rotation with respect to the C axis center of rotation. Pivot length is the vertical distance between the spindle face and the intersection of the B and C axes.

6.1.2 Five axis spindle validation

The result of an error in a rotary axis depends on the position of the machining center rotary axes (See example, above). This makes it difficult to pinpoint the source of error in a five axis cut. To ensure the five axis spindle unit is properly calibrated, a quick (relative to the calibration procedure) check is required to validate the five axis spindle. The test should take less than one hour of machine time, test the rotary axes under load, and be easy to interpret[25].

Work done by Bedi (1994) using a toroidal cutter and curvature matching can be extended to develop such a five axis diagnostic test. A toroidal tool sweeps out a donut shape profile and was used by Bedi to match the tool curvature to the part curvature (See Section 2.7).

By rotating any tool about some center point along the tool axis and a fixed distance away from the tool tip, the tool traces the surface of a sphere. This concept can be used to machine a perfect hemi-sphere with five axis machining. By keeping the center line of a toroidal tool perpendicular to the hemi-sphere surface, perfect circular contact

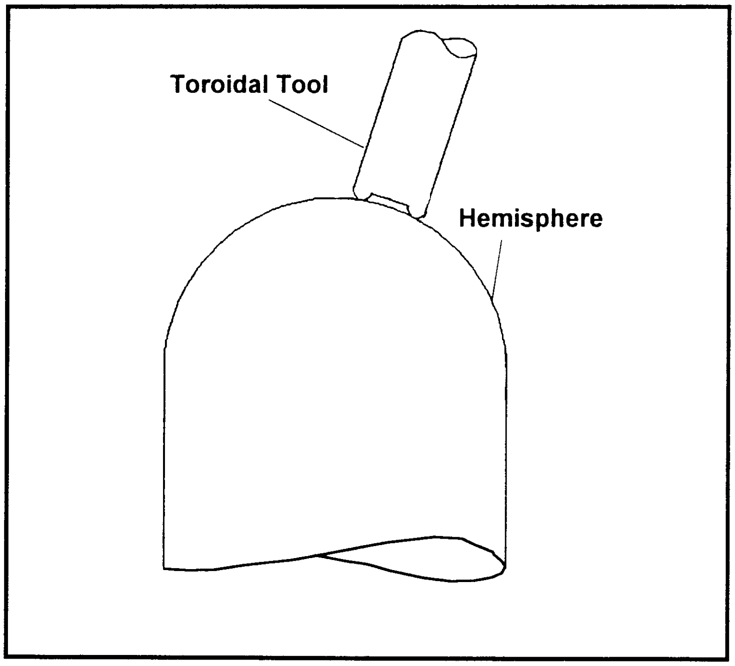


Figure 6.1.2 Diagnostic hemi-sphere.

---

[25] Many tests require cutting a part then checking the part under a CMM for accuracy.

between tool and hemi-sphere results (See Figure 6.1.2). Most contour machining utilizes point contact between the tool and part. With point contact there will always be a cusp height (See Figure 5.3.1). With circular (line) contact, it is possible to eliminate cusps and machine a net shape part.

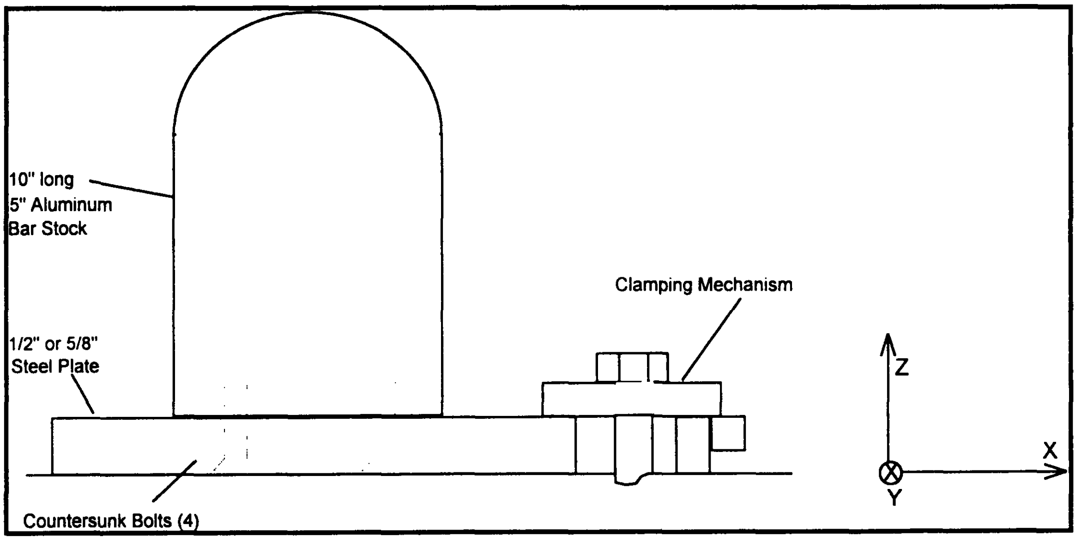


Figure 6.1.3 Hemi-sphere setup.

A test was developed using a 1 inch diameter toroidal tool and a section of five inch diameter aluminum bar stock. The test setup is shown in Figure 6.1.3. The aluminum bar stock is rough machined in three axes with a ball nose cutter. The final part is a five axis machined hemisphere consisting of 10 concentric circles machined in alternating directions. The procedure used in the finish pass is as follows:

- The initial tool vector is set on top of the hemi-sphere,
- The X, Y, and C axes trace a complete clockwise circle,
- The Z, X, and B axis move to position over the next concentric circle, below the previous circle,
- The X, Y, and C axes trace a complete counter clockwise circle, and
- The cycle is repeated for a total of 10 concentric circles.

At the present time point to point movements are used. A more accurate hemi-sphere is achievable using three dimensional circular interpolation, if the CNC supports this option.

The hemi-sphere test takes approximately 40 minutes of machine time to run at a programmed feed rate of 25 to 30 inches per minute. The real strength of the hemi-sphere test is in interpreting the result. If the machine is properly calibrated and tuned, the result is a perfect hemisphere. The entire contact circle of the tool on the hemi-sphere should be visible on each of the ten passes.

If a perfect hemi-sphere is not the result, there is an inaccuracy somewhere in the machine. The error is most likely in one of the two rotary axes. Visually inspecting the impression of the cutter on the hemi-sphere, it is possible to interpret where the machine inaccuracy lies. Appendix D contains a glossary of hemi-sphere patterns, their causes, and their cures. The glossary could be used by a maintenance mechanic to trouble shoot problems with the machining center or five axis spindle unit.

6.1.3 Maintenance requirements

To ensure it maintains accuracy and repeatability, five axis equipment must be maintained. One common theme among the benchmark companies was that the five axis equipment had to be regularly checked. Although it needed little additional maintenance effort, regular verification was crucial to maintaining accuracy, repeatability, and operator confidence in the machine. The calibration procedure should be performed at least annually together with an axis backlash check, even if a hemi-sphere test indicates the spindle unit is calibrated. There are two reasons to perform the calibration at least annually:

- The calibration procedure yields more accurate results than the hemi-sphere, and
- The maintenance mechanics are more likely to remain familiar with the calibration procedure if they perform it on a regular basis.

It is recommended that the hemi-sphere test be performed either monthly or quarterly depending on how often the five axis spindle is used.

6.2 CAM software requirements

Machining can only be as accurate as the tool paths generated by the CAM software. This is a well known fact in the die business and as a result the highest ranking customer equipment requirement is a CAM software requirement (See Figure 6.0.2). In fact, 5 of the top 11 ranking design requirements are CAM software requirements.

Several of the root causes that are addressed in the CAM software requirements are:

- Five axis software is difficult to use,
- No five axis simulation package,
- Very little development of five axis software.

To determine CAM software requirements, a CAM programming experiment was performed using the CAM software in use by die plants of the major U.S. auto maker. A report on the experiment is found in Appendix C. These CAM software requirements are for simultaneous five axis machining. Five axis drilling and four axis machining have the same CAM software requirements as three axis contour machining.

6.2.1 CAM programming experiment

The intended goal of the experiment was to compare three and five axis machining to determine what dies and die features are good candidates for five axis machining. Simultaneous five axis contouring and swarf cutting were evaluated. With the results a die plant will be able to select dies or die regions where five axis machining leads to shorter machining time, better part accuracy, and less hand grinding. In addition, the experiment provided data on the requirements of a five axis CAM package. Early in the experiment, it was clear that CAM software will play a critical role in determining the extent of five axis technology use at a die plant.

Two die contours were programmed using both three and five axis CAM software in use by die plants of the major U.S. auto maker. The two contours were the male draw stages of a hood outer panel and a center pillar.

6.2.2 Results of CAM experiment

The results of the CAM experiment are shown in Tables 3 and 4.

Hood Outer	Three Axis	Five Axis
Programming time	12.5 hours	24.5 hours
Verification time	negligible	3 hours
CPU processing time	1.2 hours	10.5 hours
File size	179,954 points	91,446 points
Machining time	14.9 hours	5.17 hours

Table 3. CAM experiment results: Hood outer die.

Center Pillar	Three Axis	Five Axis
Programming time	15 hours	34 hours
Verification time	negligible	9 hours
CPU processing time	1.3 hours	1.7 hours
File size	64,244 points	33,612 points
Machining time	3.2 hours	2.4 hours

Table 4. CAM experiment results: Center pillar die.

One interesting result of the experiment is the total programming and machining time. Figure 6.2.1 shows the sum of programming and machining times for both dies in three and five axis. Note that for both dies, any machining time savings were lost in the CAM room. This was especially the case for the center pillar die.

Although the results do not look promising, there is still benefit to five axis contouring an outer panel die such as the hood outer in this experiment for two reasons:

- Unloading a capacity constrained resource, and
- Savings at bench

Unloading a capacity constrained resource

Machining centers are very large and expensive pieces of equipment and as such are generally capacity constrained. Time savings on a capacity constrained resource should show as a throughput improvement. By moving process hours from a capacity constrained resource such as machining to a non-capacity constrained resource such as CAM programming, throughput should increase.

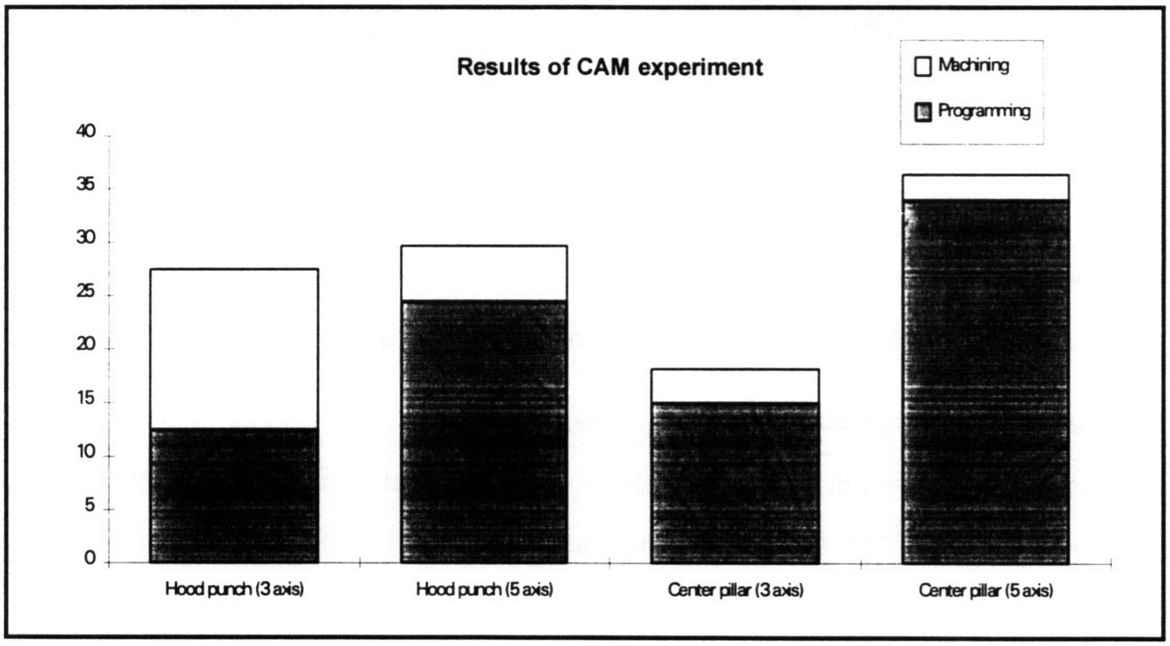


Figure 6.2.1 CAM experiment: Programming and machining times.

Savings at bench

Not only does five axis machining save machining time for outer panel dies, it eliminates hand grinding (See Section 5.3.1). A typical hood die outer requires approximately 300 hours of bench work. One hundred of these hours are used in hand grinding. Eliminating hand grinding on outer draw dies dwarfs the additional time spent in the CAM room (See Figure 6.2.2).

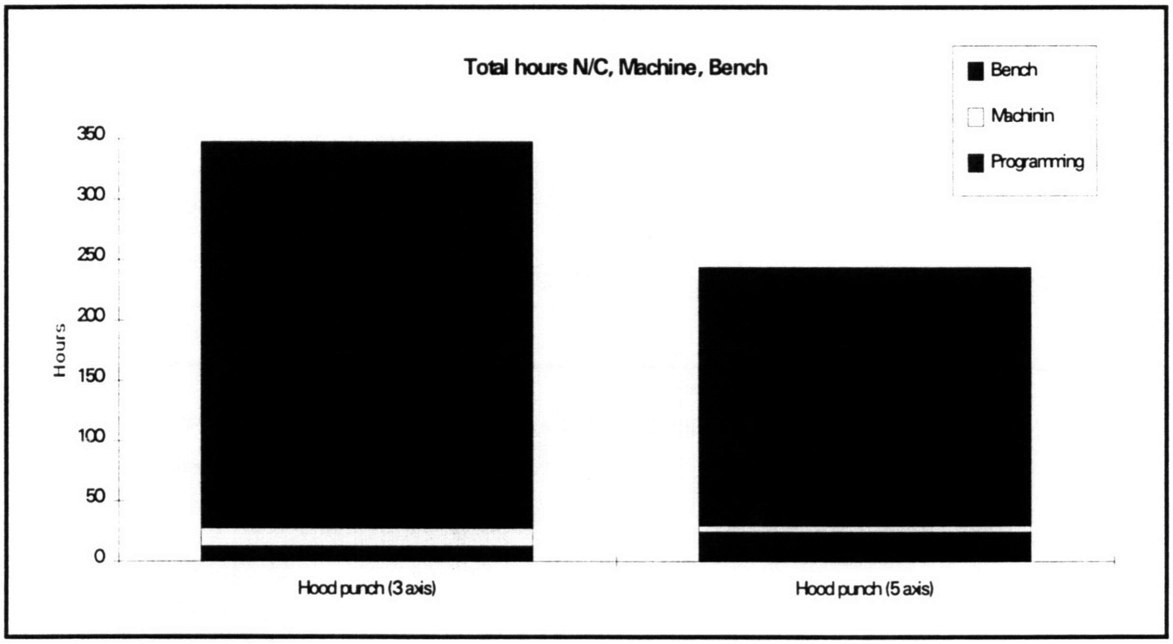


Figure 6.2.2 CAM experiment: Programming, machining, and bench times.

It seems that time saved at the machining center is lost in the CAM room. It was commonly believed that the reason for this lay in the inherent complexities of five axis CAM programming. This in fact is not the case. The reason for the additional time is not due to five axis per se, but due to the inadequacies of the CAM software used to program in five axis. A CAM programmer can work around these inadequacies, but the result is longer programming time.

Specific problems with the current CAM software in use by the die development operations of the major U.S. auto maker are in tool path definition, verification, and simulation.

Tool path definition

Defining a five axis tool path is very tedious using this software. Operators and options available in the three axis version of the software are not available in the five axis package.

Tool path verification

Although the five axis package has a verification operator, it is not robust and contains many 'glitches'. This makes the software very difficult for the CAM programmer to rely on without years of experience in using it.

Tool path simulation

Tool path simulation is limited to a wire frame drawing of the tool stepping over the surface. A good five axis simulation package will show a solid model of the tool spindle unit, work piece and machining center.

Other CAM software packages are commercially available today which provide much better five axis machining capabilities (See Appendix C).

The current software in use by the die development operations of the major U.S. auto maker is caught in a 'death spiral' (See Figure 6.2.3). A reinforcing loop can work to either strengthen or weaken a product. Generally, the more a product is used, the more additional development goes into that product. More development usually leads to better results when using the product (faster, cheaper, etc.). Better results in turn promote more use of the product, and so on. The five axis CAM software in question is caught in the negative version of this loop. For a variety of reasons (See Figure 6.0.2), five axis technology initially saw little use. Because so few CAM programmers used the five axis package, CAM software developers ranked five axis improvements very low on their development priorities list. The package in turn yielded poorer and poorer results (compared to the two and three axis versions of the software which were widely used and heavily developed). This led to less use of the software, less development, and ultimately a 'death spiral' effect. The phenomena in Figure 6.2.3 is explored further in Section 7.2.

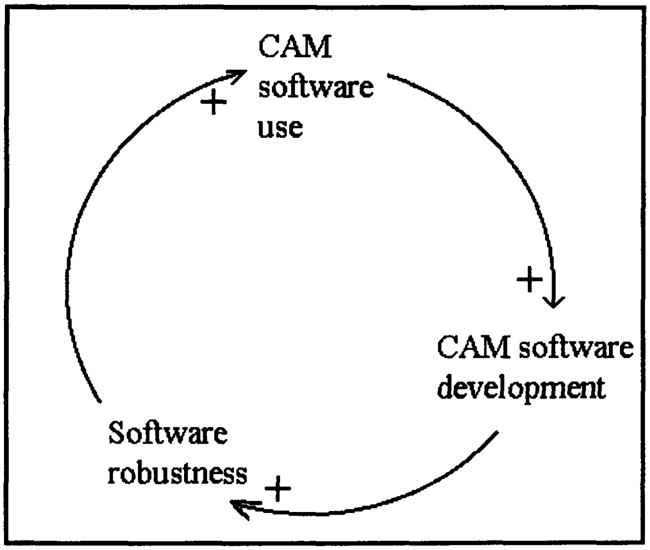


Figure 6.2.3 CAM software 'death spiral'.

6.2.3. Requirements of five axis software

The customer requirements for five axis CAM software are:

- Confidence in NC program,
- Five axis software easy to use,
- Five axis software gives gouge free cutter paths, and
- Part easy to program with five axis.

The CAM software design requirements to meet the customer requirements are:

- Robust CAM package to generate tool paths,
- Easy to use five axis software,
- Robust five axis verification and simulation packages.

All of these requirements can be met by benchmarking commercially available CAM software packages. The effects of the 'death spiral' has left the five axis CAM software used by the major U.S. auto maker far behind currently available commercial CAM software. The CAD and CAM packages of the major U.S. auto maker are currently under review. A long term goal of the auto maker is to bring this software up to 'world class' levels. The die development operations of the auto maker will become a large customer of the CAM package that results from this upgrade. It is vitally important that the die development group become involved early on in the review and subsequent decision process to include its requirements for such a package.

6.3 Training requirements

The Ishikawa diagram in Figure 6.0.1 shows many root causes that contain the phrase "lack of understanding/knowledge", or "no/little training". Lack of training is one of the prime root causes of why five axis machining is rarely used by this major U.S. auto maker. In fact, three of the top ten ranking design requirements are training related requirements. The training requirements considered in this section are for:

- CAM programmers,
- Maintenance personnel,
- Machine operators, and
- Die builders.

CAM programmers command by far the largest portion of the training requirement as evidenced by the benchmark study and CAM programming experiment.

6.3.1 CAM programmers

The process most heavily affected by five axis machining is CAM programming. Upstream processes such as design need only consider the additional flexibility offered by five axis machining. Downstream processes such as machining and bench will only be marginally affected by a shift from three to five axis. The physical change between three and five axis lies in machine motion which is ultimately determined in the CAM room by programmers. The training requirements that follow are for simultaneous five axis machining only (contouring and swarf cutting). Five axis drilling and four axis machining have near identical CAM requirements as three axis contour machining.

Moving from three to simultaneous five axis CAM programming requires visualization skills which go well beyond those needed to perform three axis contour machining. The combination of these factors make CAM programmer training not only the most critical training requirement but also the most important design requirement in the entire QFD matrix (See Figure 6.0.2).

This sentiment was echoed by many of the benchmark companies. When asked about the transition from three to five axis machining, the most common reply among benchmark companies was that:

- The transition went smoothly,
- CAM programmers required the most additional training, and
- CAM programmers had the longest five axis learning curve.

To successfully program dies in five axis, CAM programmers must be familiar with and trained to:

- Use five axis software,
- Know what types of dies are good five axis candidates,
- Process a five axis job,
- Visualize five axis motions,
- Understand how the CAM software will react to special geometries and conditions.

Five axis software

Obviously, CAM programmers need to know how to use the five axis software. Because five axis machining is only a small part of overall die machining, five axis CAM programmers must continually update their training to maintain their knowledge of the software and to stay abreast of developments from software updates. The major components of a CAM software package that a programmer must be familiar with are path definition, tool path generation, part verification, and machining simulation[26]. Learning curves vary person to person and between software packages. For a CAM programmer who is proficient in three axis contour machining to learn a robust five axis CAM software package would take approximately six months of training and practice.

Candidates for five axis machining

Not all dies are good five axis candidates. Dies or die regions with large, gently sloping surfaces are ideal candidates (See Appendix C). A CAM programmer must be able to look at a die and determine if that die (or any region on that die) is a candidate for five axis machining. To acquire this skill, a five axis process guide should be developed that walks a CAM programmer through a series of steps to determine where five axis machining would benefit the die plant for the specific die. The CAM programmer would refer to the guide until s/he became familiar with its contents. Once the knowledge in the guide is embedded in the programmer, s/he need only refer to the document when processing guidelines are changed or updated.

Process a five axis die

Not only must a programmer know which dies and die regions are good five axis candidates, s/he must know how to best define five axis tool paths over a certain die. The first step is to determine what type of tool is best suited to the die in question. The type of surface to be machined will guide tool selection. Depending on the surface geometry of the part, tool selected, and characteristics of the particular CAM software package, the programmer must decide whether and where s/he needs to section cut, flow cut, swarf cut, or bump cut the die. A die is generally cut using a combination of these methods.

---

[26] Some CAM packages do not contain verification or simulation software.

This knowledge can be contained in the same process guide mentioned earlier. The guide will walk the programmer through the five axis selection process, tool selection process, and finally the recommended cutting strategy (flow cut, section cut, etc.).

Visualization tools

To avoid possible machine head crashes or tool gouging, the CAM programmer must be able to visualize how a machine will move its axes to position a tool. An effective way to train programmers in visualization is with a simulation program. Simulation software will animate the machine tool, spindle unit and cutting tool as it moves over the part and teach the programmer how the machine moves in certain situations. After some experience with the simulation software, programmers will have a much better understanding of how the machine axes will react to a given five axis move.

Visualization tools benefit inexperienced five axis programmers. As was the case with many benchmark companies, the major U.S. auto maker can expect the use of simulation packages to decrease once CAM programmers become comfortable with five axis machining.

Special geometries and conditions

CAM software packages used to generate contoured three and five axis tool paths are very large and generally very complex. Like any piece of large machinery, these programs have peculiarities and 'glitches' that are not well understood or documented. To successfully and confidently program a die in five axes, a programmer needs to know what these peculiarities are and how they may affect his/her programs. Two ongoing training activities are recommended to keep abreast of these special conditions. First, a working document should be developed which details all the known special conditions, peculiarities, and die geometries that can lead to programming problems. The document would be updated and distributed to CAM programmers regularly. Ideally, the software company who developed the CAM package would be part of this process to 'close the loop' back to the supplier. Second, regular meetings between CAM programmers and machine operators should be held to address peculiarities found when running five axis jobs on the machining centers. Peculiarities or special conditions that are discovered would become part of the working document.

6.3.2. Maintenance personnel

Maintenance personnel need to be familiar with the workings of the five axis spindle, the five axis spindle calibration procedure, and the hemi-sphere diagnostic test.

Five axis spindle

It is not realistic to require the equipment manufacturer to be responsible for all repair work done on the five axis spindle. The maintenance personnel should be trained on the general mechanics of the spindle, seal and bearing replacement, backlash adjustment, and servo system debugging and tuning.

Five axis spindle calibration procedure

To properly use the calibration procedures, maintenance mechanics should be trained on the purpose of the procedure, how it works, and how to successfully calibrate and test the calibration of a five axis spindle. The required training consists of walking through several calibrations with an engineer or technician familiar with the procedure.

Hemi-sphere diagnostic test

The hemi-sphere test is used to periodically check the calibration of a five axis machine. Maintenance personnel who are responsible for performing the test should be trained on the hemi-sphere concept (See Section 6.1.2 above), how to set up and run the test, and how to interpret the results of the test. Appendix D contains a glossary of hemi-sphere results, what each pattern means, and how to correct the problem found. Again, a walk through of several tests with an engineer or technician familiar with the procedure is sufficient.

6.3.3 Machine operators

Operators need to be trained on spindle unit operating procedures, operating limits, and how to spot early indications of trouble with the unit. A list of 'early warning signs' needs to be developed and periodically updated. Several critical warning signs are spindle or tool overheating, fluid leaks, 'jerky' servo axis motions, backlash noise, and bearing wear noise. The training can be added to ongoing, continuous operator training on equipment and processes.

6.3.4 Die builders

Five axis contour machining not only reduces material remaining on the surface of a part (See Section 5.3.1), it also changes the way the remaining material is distributed over the part surface (See Figure 2.5.1). Die builders, when benching a five axis machined die, must understand the differences between three and five axis machined dies, and more specifically, know how to bench a five axis machined die. With five axis contour machining, die builders need only stone polish the die surface; hand grinding is not required. The training requirements here are rather simple and may be accomplished by developing a procedure to spot five axis machined surfaces and how to bench those surfaces. This training requirement is only applicable to simultaneous five axis machined parts; it does not apply to five axis drilling or four axis machining.

6.4 CNC requirements

The CNC requirements for five axis machining are not a high ranking design requirement. However, the requirements must be met for successful simultaneous five axis machining.

6.4.1 Introduction to CNC

Computer Numerical Controllers (CNC's) take a part program that was generated in the CAM room and turn that program into command voltages for the machine tool servo motors. Figure 6.4.1 shows the typical sequence of operations for a three axis CNC.

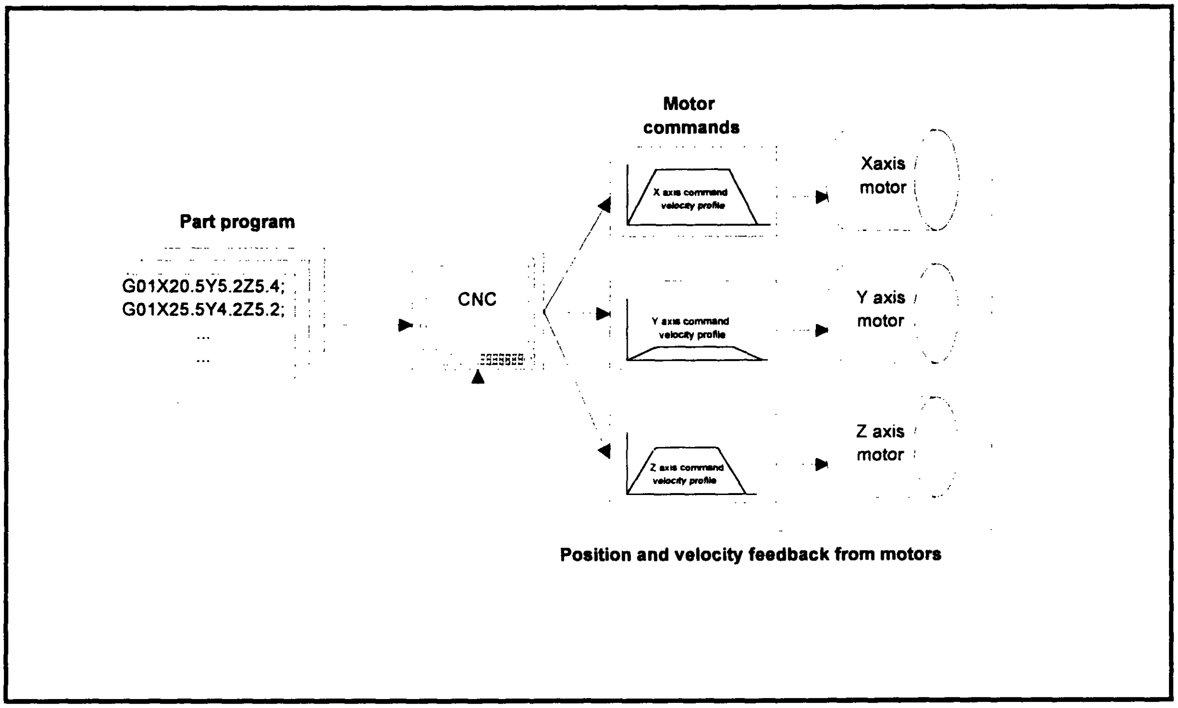


Figure 6.4.1 The CNC as an interface between part program and machine tool.

The CNC performs other functions in addition to controlling the machine tool axes. Most commercially available CNC's have features such as:

Communications interface (i.e. Ethernet) to directly transfer programs from a CAM station to the machine tool through a computer network,

Internal memory to store part programs and preprocessed instruction blocks,

MDI mode that allows an operator to program the machine tool at the CNC console,

Edit mode that allows an operator to change a part program at the CNC console,

Multitasking capabilities that allow an operator to edit a part program while the machine tool is executing some other part program,

Block look ahead feature that permits the CNC to preprocess blocks to anticipate and prepare for large directional changes in any axis,

Graphical simulation of the tool path on the CNC console to show the operator programmed machine motions before they occur.

6.4.2 CNC requirements for five axis machining

The CNC requirements for five axis drilling and four axis machining are the same as those for three axis contour machining. The additional requirements explained here are for simultaneous five axis machining (contouring and swarf cutting). A CNC must process data quickly enough to keep ahead of machine tool movements. With five axis simultaneous machining, there is a risk of data starvation or the CNC becoming a process bottleneck. Data starvation occurs when part program commands (called blocks) are communicated to the CNC at a slower rate than the machine executes the blocks. The CNC becomes a processing bottleneck when the machine travels faster than the CNC can supply it with instructions (the servo system executes commands faster than the CNC is able to process blocks). When this occurs, the machine pauses until it receives further instruction. The effect of a pause is an impression on the part surface which must be manually ground to blend with the remainder of the surface.

The goal of these requirements is to provide a set of specifications a CNC must meet or exceed to ensure it does not become a processing bottleneck for five axis machining.

The sequence of operations performed by a CNC is shown in Figure 6.4.2. Elements within the dashed box are internal CNC components. The part program is the input and the servo amplifier is the output. Each element must process a block within the amount of time it takes the machine to execute that block. The throughput time at which a CNC must provide instructions to a servo motor depends on the feed rate of the machine and spacing of the points, according to the following equation:

$$t = \frac{d_{min}}{f_{min}} \times 60 \text{ seconds}$$

For example, with a minimum required feed rate of 200 inch/min. and a point density of 0.040 inch, each CNC element must process a block in 12 msec or less. The following sections step through the requirements of each element to maintain a maximum throughput time.

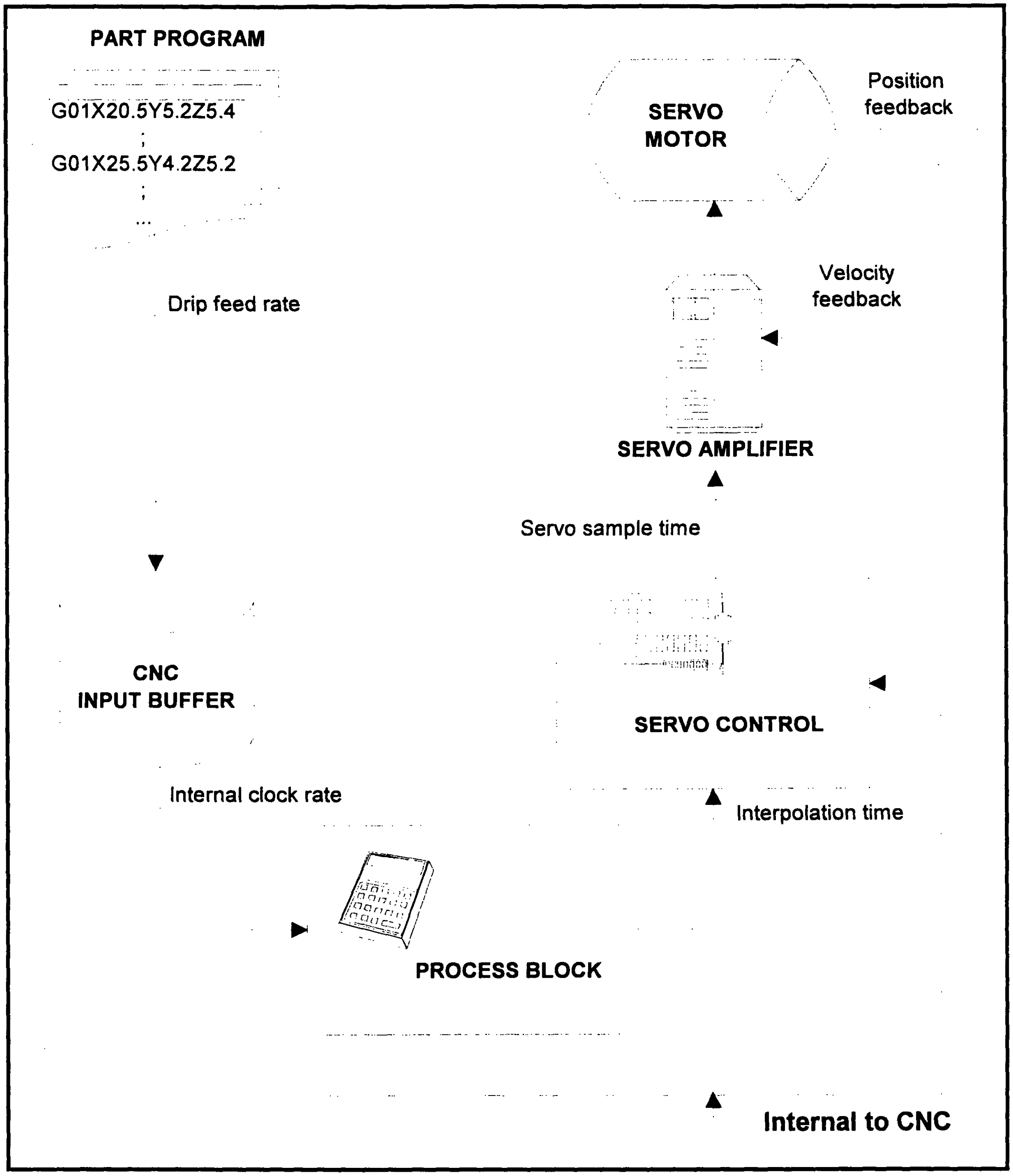


Figure 6.4.2 Sequence of operations performed by a CNC[27].

Drip feed rate. Part programs that are too large to fit into CNC memory must be drip fed from an input device such as a tape drive or network connection. To avoid data

---

[27] For a servo system with digital drives, both velocity and position feed back to the servo control.

starvation, each program block must be fed in a period that is less than or equal to the maximum throughput time (*t*) specified above. A typical worse case five axis program block looks like:

N#####G01X±####.###Y±####.###Z±####.###B±###.###C±###.###F#####;

Individual components of the block are explained in Table 5.

Segment	Purpose	Segment Length (# characters)
N#####	Program line number	6
G01	Point-to-point move command	3
X±####.###	X axis commanded position to 3 significant digits (mm)	10
Y±####.###	Y axis commanded position to 3 significant digits (mm)	10
Z±####.###	Z axis commanded position to 3 significant digits (mm)	10
B±###.###	B axis commanded position to 3 significant digits (deg)	9
C±###.###	C axis commanded position to 3 significant digits (deg)	9
F#####	Programmed feed rate (mm/min.)	6
;	End-of-block character	1
	Total length of typical five axis program block:	64

Table 5. Components of a five axis move command.

Each block requires 10 bits for transmission, therefore the drip feed rate must equal:

$$\frac{64}{t} \times 10 \text{ bits per second (baud)}$$

Using our previous example with a 200 inch/min feed rate and 0.040 inch point spacing, the drip feed rate must be greater than or equal to 53,300 baud, or 53.3 kbaud.

Internal clock rate between system components. This clock rate refers to the rate at which individual components within the CNC transfer information. It is dependent on the microprocessor clock rate and generally is not a bottleneck. A clock rate of 10 MHz (most CNC clock rates are above 10 MHz) will be more than sufficient here.

CNC input buffer size. The size of the input buffer must be such that it does not starve the controller for data. Theoretically, if the drip feed rate meets or exceeds requirements, a one block buffer will be sufficient. The CNC was simulated using a model very similar to that shown in Figure 6.4.2. The simulation estimates three axis CNC performance for a user-specified set of parameters and indicates relationships between those parameters[28]. To determine buffer size requirements, the simulation was run while varying the input buffer size. Figure 6.4.3 shows the results of that simulation. Due to variability in block size and CNC throughput time, a 1 to 4 block input buffer will occasionally starve the CNC and reduce machine feed rate. The feed rate drop is eliminated for an input buffer size of 5 blocks or more. The minimum required input buffer size is 5 five axis blocks. However, the simulation is based on three axis machining. An equivalent three axis block is only 46 characters in length, and a five axis block is 64 characters in length.

---

[28] The model used to determine input buffer size and block look ahead is a proprietary simulation package for simulating a specific CNC in use at several die plants of the major U.S. auto maker. Output from the simulation is CNC specific and only approximate.

Therefore the input buffer must be able to hold five 64 character blocks. The requirement can also be expressed as 320 characters (64*5).

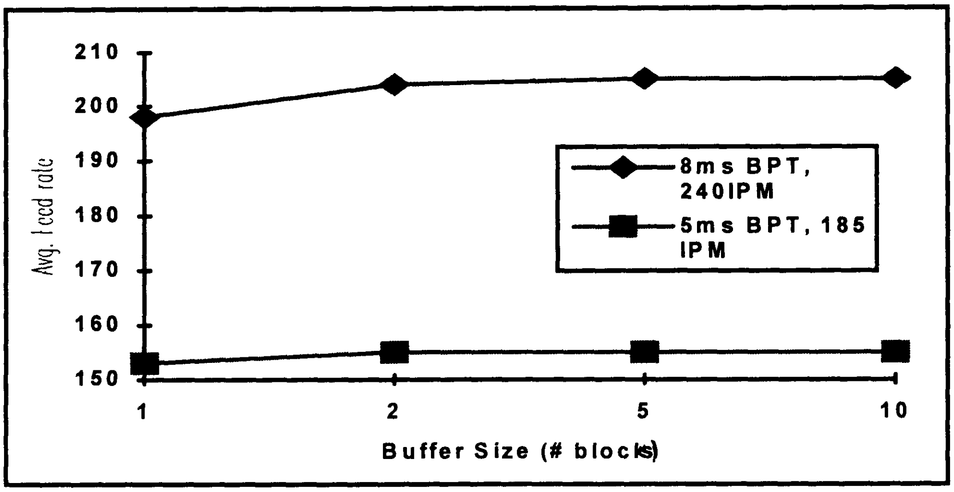


Figure 6.4.3 Effect of input buffer size on feed rate.

Block processing time. This is a very widely advertised CNC specification. The block processing time (BPT) is the time taken to translate a block of G-code into individual axis movements. Most CNC suppliers list BPT for three axis machining. Assuming a linear relationship[29] between BPT and number of axes, the effective BPT for a five axis move is 3/5, or 60% of the advertised, three axis BPT. The required BPT is based on the minimum distance between points and the desired feed rate, according to the following equation:

$$\text{BPT} = \frac{\text{Minimum Distance (inch)}}{\text{Feedrate (inch / min)}} \times 60000 \ [\text{msec}]$$

Using the example of a 200 inch/min feed rate and minimum 0.040 inch point spacing on a five axis move, the five axis BPT must be less than or equal to 12 msec, or an equivalent three axis BPT of 7.2 msec.

---

[29] Valid assumption so long as the CNC is not additionally burdened by calculations such as cutter radius compensation.

Block look-ahead The ability of the CNC to process blocks ahead of time is critical to avoid unnecessary machining errors such as gouging and overshoot. To prevent machining errors, the CNC must not only process blocks ahead of time but 'look ahead' at upcoming moves. This look-ahead feature allows the CNC to change feed rates based on the upcoming geometry. A CNC with block look ahead will reduce feed rate when it 'sees' upcoming rapid changes in direction (such as a tight radius) to avoid overshoot. Figure 6.4.4 shows the effect of block look-ahead on contour error from a three axis simulation. A 10 block look-ahead is sufficient to avoid contour errors caused by the control system.

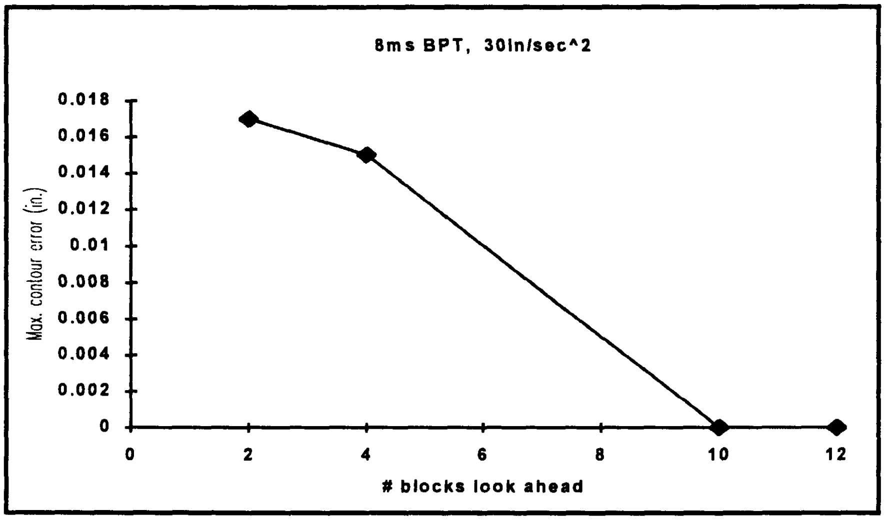


Figure 6.4.4 Effect of block look-ahead on contour error.

Interpolation time. The interval at which the CNC passes commands to the servo system. The CNC outputs a distance each axis should move based on the interpolation time. For example, if a CNC has an interpolation time of 8 msec, the CNC will break the motion into 8 msec size moves. Given a 60 inch per minute programmed feed rate and a 0.112 inch move, the CNC will command a 0.008 inch move every interpolation cycle (8 msec) for 14 interpolation cycles (0.112/0.008). To avoid being a bottleneck, the interpolation time must be less than or equal to the block processing time.

An issue related to interpolation rate is blending. If a CNC with an interpolation time of 8 msec is commanded to move 0.106 inches at 60 inches per minute, the CNC will command a 0.008 inch move every 8 msec for 13 interpolation cycles. This leaves a 0.002 inch distance remaining distance. To prevent a drop in feed rate, the CNC must support a blending algorithm which accounts for the remaining 0.002 inch move in the next block. CNC's that do not support blending must deal with the remaining distance independently, resulting in a drop in feed rate.

Interpolation time is sometimes stated as a rate (i.e. Interpolation rate). The interpolation rate is simply the inverse of interpolation time.

Servo sample and update time. This is the time taken for the system to read servo motor position and velocity feedback, compare actual to commanded position, and issues a command voltage to drive the axes. The servo sample and update time (SST) requirement is similar to the block processing time (BPT) requirement. To avoid being a processing bottleneck and meet the Nyquist sampling criteria, the SST must be less than or equal to half of the BPT. Given the example of a five axis move with a desired feed rate of 200 inch/min and minimum point spacing of 0.040 inch, the SST must be less than or equal to:

$$
\begin{aligned}
\mathrm{SST} &= \frac{0.040\ \mathrm{in}}{200\ \mathrm{in/min}} \times \frac{60000}{2} \\
&= 6\ \mathrm{msec}\ (5\ \mathrm{axis\ SST}) \\
&= 3.6\ \mathrm{msec}\ (3\ \mathrm{axis\ SST})
\end{aligned}
$$

Servo sampling time is sometimes stated as a rate (i.e. Servo sample and update rate). The servo sample rate is simply the inverse of servo sampling time.

Note on CNC requirements

The advent of faster microprocessors, better control systems, and faster data communications have made CNCs much quicker in recent years. The CNC requirements listed here are easily met my many current model CNCs and communication systems. The requirements can be added to a set of specifications when purchasing a CNC to ensure it will be capable of five axis machining. The greater benefit of these requirements however, is in checking if older model CNCs currently in use can support (or can be retrofitted to support) five axis machining.

6.5 Communication requirements

The three communications requirements ranked 15th, 20th, and 23rd in importance out of 24 design requirements. It is the author's contention that communication requirements were ranked so low because it was generally perceived that the requirement was considered too far fetched to be realistic. In fact, meeting the communication requirement is a necessary step in achieving the larger savings possible with four axis machining (See Section 5.2.3) and is a critical requirement for five axis swarf cutting (See Section 5.3.4). The communication requirements explained here are for achieving four axis end mill machining and five axis simultaneous (contour and swarf cut) machining. Communications already in place at the die plants are sufficient to take advantage of five axis drilling and four axis ball nose machining. The two communication requirements discussed here are:

- Increased communication and collaboration between die engineering and die plants, and
- Increase communication and collaboration between product designers and die plants.

Some of the root causes addressed by these communication requirements are:

- Poor communication between design and CAM room,
- Organizational walls between design and die plants, and
- No incentive for design staff and die plants to collaborate.

The importance of these requirements are revisited in Chapter 7.

6.5.1 Die engineering and die plants

To take advantage of the benefits offered by four and five axis machining, die engineering must understand how the new die plant process flexibility can favorably affect design. This requirement can be met by the engineering staff collaborating with die plant CAM programmers. In effect, the die plants 'offer' this new process flexibility to the engineering staff and the team decides how these new capabilities can best be used to reduce die development time and increase part quality. One such benefit (mentioned in Section 5.2.3) is incorporating cam adapters and drivers into die shoes. One benchmark company is already doing this with their five axis equipment. In general, die engineers can design dies that may seem more complicated to build (i.e. complex flanging and under-cutting) but in fact are easily machined using the process flexibility offered by five axis machining. With ongoing collaboration, similar benefits can be uncovered and exploited.

6.5.2 Product designers and die plants

Two problems prevent the use of five axis swarf cutting in die construction for the major U.S. auto maker (See Section 5.3.4). The first is CAM software that was addressed in Section 6.2. The second is the product math data. To swarf cut a die section, the product math data must contain surfaces that are 'developable'. This problem was encountered during the CAM experiment, specifically when trying to program the center pillar die (See Appendix C).

The main rib of the center pillar die was considered an ideal candidate for five axis swarf cutting. However, the rib web contains surfaces whose linear segments are not in the plane formed by the flange surface and flange surface normal (Appendix C). That is, to successfully swarf cut requires that the rib surfaces have straight elements in the plane perpendicular to the flange surface (a 'developable' surface). This not being the case with the center pillar caused the tool to rock back and forth over the flange surface during the swarf cut, which in turn caused the leading or trailing edge to gouge the flange surface.

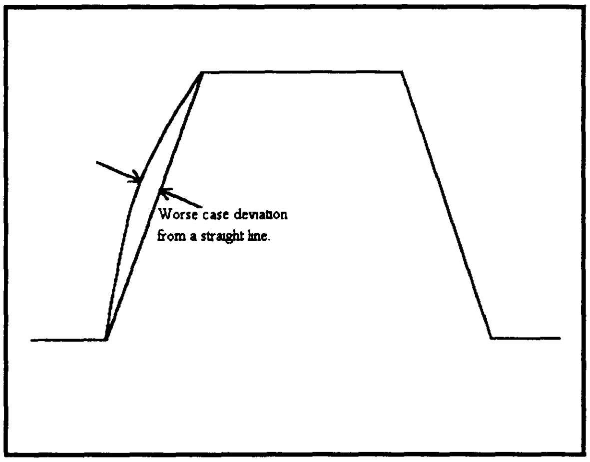


Figure 6.5.1 Rib section deviation.

A test was performed to ensure the surface could in fact be made 'developable' without losing product definition. Twenty five cross sections were made through the center pillar. Figure 6.5.1 shows a typical cross section. The web surface (that should be straight in cross section to be swarf cut) is slightly curved. A straight line was drawn from the top flange to the bottom flange over top of the web surface (straight line, Figure 6.5.1). The maximum deviation was calculated between the actual cross section and the ideal 'developable' (straight line) cross section. Figure 6.5.2 shows a histogram of results. For 19 of the 25 cross sections, the deviation from straight was less than 0.002 inches, which is less than the machining error allowance. This test proved that the curvature in the surface is not a product characteristic. Rather, it is the result of a process used by product designers to generate the mathematical representation of an otherwise flat, 'developable' surface.

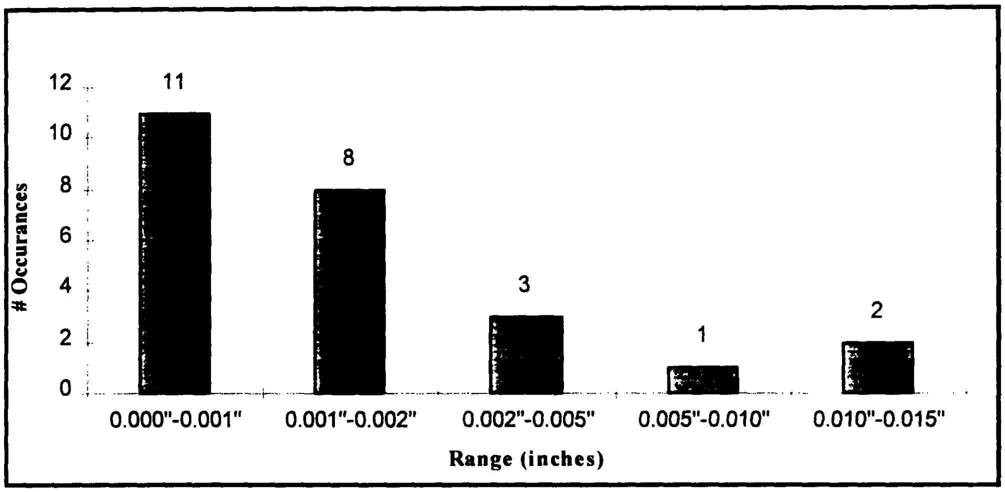


Figure 6.5.2 Deviations from a flat surface.

This difficulty can be overcome by increasing collaboration between product designers and die plants. Working together, product designers could spend more time on certain die regions to build 'developable' surfaces. The realizable savings in CAM programming time, machining time, and bench time will far outweigh additional design time.

In addition to 'developable' surfaces, product designers and die plants can work to eliminate 'glitches' found by CAM programmers, such as non-parallel surface normals which tend to cause tool rotation and surface gouging (See Appendix C). The purpose of increasing communication and collaboration is to strengthen the links between processes and to optimize the entire system, rather than have each individual die development function optimize its own activity.

6.6 Chapter summary

Though the benefits of five axis machining are substantial, the technology requirements to exploit those benefits are not. The most critical requirements are training and maintenance. Of these, CAM programmer training is crucial to successful five axis implementation.

It may seem that the biggest affect of five axis machining is on the machining center, when in fact it is in CAM programming. As such, the die development operators of this major U.S. auto maker must play an active role in defining the requirements of a five axis CAM software package. They must arm their die plants with a robust five axis CAM package and ensure CAM programmers are sufficiently trained on the use of the software.

The process that prevents realization of the largest single item savings (four axis ball nose machining) is communication. By increasing communication between different organizational groups, several barriers to successful five axis implementation will fall. This organizational change requires a shift from a functional focus to a process focus and is explored further in the following chapter.

7.0 Implementation of technology

When a company considers a technology for it's operations, it generally limits the scope of the analysis to benefits of the technology (i.e. savings) compared to cost of equipment and process changes. An equally important consideration, and one that is often overlooked, is how the technology will affect the organizational structure and how the culture will react to its implementation. To take full advantage of any technology, one must consider the scope[30] of the technology in the organization and implications of implementation. That is, a technology that affects many organizational areas has a very wide scope.

This chapter addresses some of these issues. Section 7.1 provides a traditional cost-benefit analysis for five axis machining at the major U.S. auto maker. Section 7.2 provides a causal loop analysis on the implementation and use of five axis technology at the die plants of the major U.S. auto maker. Finally, Section 7.3 views the implementation of five axis machining in the auto makers die development operations as an architectural innovation and considers the implications of that perspective.

7.1 Justification of five axis machining

This section consolidates the benefits of five axis machining quantified in Chapter 5. The benefits are compared to the cost of equipment, installation, and maintenance. The analysis covers two levels of implementation. The first level considers only low hanging fruit, the second looks at full five axis machining. It is very possible that a die development organization strives for and achieves only the first level benefits. For that

---

[30] Scope here refers to the areas of an organization the technology impacts.

reason, the analysis is given at both levels of implementation. Level one net present value is repeated for the major U.S. auto maker's die development operations, since it already owns the nutating spindles and its machining centers are capable of five axis machining.

The first level of implementation; low hanging fruit, considers five axis drilling, four axis ball nose machining, and five axis contour machining. The second level includes all of the savings of the first level and adds four axis end mill machining. Five axis swarf cutting should be included in second level implementation but is not included in the justification. To successfully swarf cut in five axis requires a significant overhaul of current equipment and processes (See Section 5.3.4) and is beyond the scope of this paper. The levels simulate the actual expected implementation of five axis machining in a die development organization. That is, first level savings are easy to achieve compared to those for four axis end mill machining and swarf cutting.

All first level elements can be realized by a die plant independent of other functions in the die development organization. Four axis end mill machining and swarf cutting require collaboration with the design and engineering functions of the die development organization.

7.1.1 Die plant assumptions used for cost justification

The analysis provides a framework to evaluate five axis machining. For confidentiality reasons, the numbers used to define a hypothetical die plant in the analysis are not representative of the die making operations of the major U.S. auto maker. A die making operation that wishes to evaluate five axis machining need only substitute the assumptions and costs for that particular organization.

The analysis will be performed on a per plant basis. The die plant considered here produces 400 dies per year and uses an average of 500 hours of machining center time per die. Assuming that each machining center has available 6900 hours per year[31], the die plant requires 29 machining centers. Generally, machining centers at a die plant are dedicated as either roughing or finishing machines. A roughing machine performs brute material removal and is not required to cut as accurately as a finishing machine. A finish machine is used for light, finish cuts, and is able to maintain its accuracy almost indefinitely. The hypothetical die plant considered here has 12 dedicated roughing machines, 12 dedicated finishing machines, and 5 machines allocated to both roughing and finishing.

7.1.2 Compilation of savings: First level implementation

Table 6 provides estimated savings for level one implementation. Bench savings from improved surface finish with four axis ball nose machining were not estimated.

Stage of five axis machining	**Estimated savings**
Five axis drilling; machine time savings, from Section 5.1.3:	$\frac{2800\text{ hours}}{\text{year}} \times \frac{\$100.00}{\text{hour}} = \frac{\$280{,}000}{\text{year}}$
Four axis ball nose machining; tool cost savings, from Section 5.2.2:	$\frac{\$37.00}{\text{die}} \times \frac{400\text{ dies}}{\text{year}} \times 80\% = \frac{\$11{,}840}{\text{year}}$
Four axis ball nose; machine time savings, from Section 5.2.2:	$\frac{1800\text{ hours}}{\text{year}} \times \frac{\$100}{\text{hour}} = \frac{\$180{,}000}{\text{year}}$
Five axis machining; outer panel draw dies, from Section 5.3.3: Machine time savings:	$\frac{19.5\text{ hours}}{\text{die}} \times \frac{24\text{ dies}}{\text{year}} \times \frac{\$100}{\text{hour}} = \frac{\$46{,}800}{\text{year}}$
Five axis machining; outer panel draw dies, from Section 5.3.3: Bench time savings:	$\frac{2544\text{ hours}}{\text{year}} \times \frac{\$50}{\text{hour}} = \frac{\$127{,}000}{\text{year}}$
Total:	**$645,640 per year**

Table 6. Level one estimated savings.

---

[31] 345 work days per year, 20 hours per day.

7.1.3 Compilation of savings: Second level implementation

Table 7 provides estimated savings for level two implementation. The savings combine the level one savings with those for four axis end mill machining.

Stage of five axis machining	**Estimated savings**
Level one savings (Section 7.1.2):	$\frac{\$645{,}640}{\text{year}}$
Four axis end mill machining, from Section 5.2.3:	$\frac{55\text{ hrs}}{\text{part}} \times \frac{\$75.00}{\text{hour}} \times \frac{260\text{ parts}}{\text{year}} = \frac{\$1{,}070{,}000}{\text{year}}$
Total:	**\$1,715,640 per year**

Table 7. Level two estimated savings.

7.1.4 Cost of equipment and installation

To machine in five axis, a die plant requires a five axis spindle unit and a machining center that is capable of accepting the five axis spindle unit. This justification is based on the assumption that all machining centers are capable of accepting a five axis spindle unit. This is the case for the die plants of the major U.S. auto maker. If this were not the case, a die plant must justify the purchase of new machining centers.

The cost estimate is for the purchase and installation of nutating type spindle units. Most machining centers in the die plants consulted for this study were nutating type spindle units. Table 8 provides an estimated installation cost summary per machining center. The numbers were obtained from historical data and equipment manufacturer estimates.

Item	**Cost estimate**
Nutating type spindle unit:	$250,000
Equipment manufacturer's technician:	$1\text{ week} \times \frac{40\text{ hours}}{\text{week}} \times \frac{\$100.00}{\text{hour}} = \$4{,}000$
Die plant maintenance mechanic:	$1\text{ week} \times \frac{40\text{ hours}}{\text{week}} \times \frac{\$50.00}{\text{hour}} = \$2{,}000$
Die plant machinist:	$1\text{ week} \times \frac{40\text{ hours}}{\text{week}} \times \frac{\$50.00}{\text{hour}} = \$2{,}000$
Total:	**$258,000**

Table 8. Cost estimate summary per machining center.

The five axis spindle units will require approximately one additional day per year of maintenance attention. An equipment manufacturer's technician, a maintenance mechanic, and a machinist are required to perform the maintenance. The additional cost per machining center is:

$$\frac{1\text{ day}}{\text{year}} \times \frac{8\text{ hours}}{\text{day}} \times \frac{\$100 + \$50 + \$50}{\text{hour}} = \frac{\$1600}{\text{year}}$$

Each die plant will require at least two five axis CAM software licenses. The estimated cost per license (including CAM programmer training) is $60,000[32].

---

[32] Average for several commercially available five axis CAM software packages.

7.1.5 Net present value analysis

Assumptions used in the net present value (NPV) analyses are:

1. Nutating spindles are purchased for 17 machines per die plant. Each plant has 12 finishing and 5 roughing/finishing machines.
2. A ten year equipment life. After ten years the equipment will be out-dated and near the end of its service life.
3. Two robust five axis CAM software package licenses purchased at a total price of $120,000, including training.
4. Corporate tax rate of 34%. Average tax rate from Brealey (1991).
5. Required rate of return of 14.4%. The capital asset pricing model (Brealey, 1991) was used to determine risk. Model details are provided below.
6. Average labor cost inflation of 2% per year (Wall Street Journal; 03/22/95).

Capital asset pricing model

According to Brealey (1991), investment risk can be determined using the capital asset pricing model. The model states that a discount rate (r) can be quantified by considering the risk free (treasury bill) interest rate (r_f), expected rate of return on the market portfolio (r_m), and the company's overall risk compared to the market (β) according to:

$$r - r_f = \beta(r_m - r_f)$$

The market risk premium (r_m-r_f) over a 63 year period (1926-1989) has averaged 8.4% per year (Brealey, 1991). The risk free rate of interest (r_f), is approximately 5.7% (Wall Street Journal, 03/22/95). The beta risk (β) used for this NPV is an average of the quoted beta risks of the three major U.S. auto makers: Chrysler; 1.1, Ford; 1.2, General Motors; 0.8 (A.G. Edwards, 03/23/95).

Using the capital asset pricing model, the discount rate for the NPV is:

$$r - r_f = \beta(r_m - r_f)$$

$$r - 5.7 = \frac{1.1 + 1.2 + 0.8}{3}(8.4)$$

$$r = 14.4\%.$$

Tables 9, 10, and 11 provide NPV summaries for level one, level two, and level one implementation for the major U.S. auto maker, respectively.

NPV for level one implementation

Discount Rate:		14.40%		Equip. cost per installation:		$258,000
Inflation Rate:		2.00%		Estimated annual savings:		$645,640
Corp. Tax Rate:		34.00%		Est. maintenance / machine:		$1,600
Level One Implementation				Cost of CAM Software		$120,000
		Additional	Estimated	Taxes:		
Year	Investment	Expenses	Savings	Savings(Cost)	Net Savings	Present Value
0	($4,506,000)			$1,532,040	($2,973,960)	($2,973,960)
1		($27,744)	$658,553	($214,475)	$416,334	$363,928
2		($28,299)	$671,724	($218,764)	$424,660	$324,481
3		($28,865)	$685,158	($223,140)	$433,154	$289,310
4		($29,442)	$698,861	($227,603)	$441,817	$257,952
5		($30,031)	$712,839	($232,155)	$450,653	$229,992
6		($30,632)	$727,096	($236,798)	$459,666	$205,063
7		($31,244)	$741,637	($241,534)	$468,859	$182,836
8		($31,869)	$756,470	($246,364)	$478,237	$163,018
9		($32,507)	$771,600	($251,292)	$487,801	$145,348
10		($33,157)	$787,032	($256,317)	$497,557	$129,593
				Total Net Present Value:		**($682,440)**

Table 9. Net present value: First level implementation.

NPV for second level implementation

Discount Rate:		14.40%		Equip. cost per installation:		$258,000
Inflation Rate:		2.00%		Estimated annual savings:		$1,743,800
Corp. Tax Rate:		34.00%		Est. maintenance / machine:		$1,600
Level Two Implementation				Cost of CAM Software		$120,000
Year	Investment	Additional Expenses	Estimated Savings	Taxes: Savings(Cost)	Net Savings	Present Value
0	($4,506,000)			$1,532,040	($2,973,960)	($2,973,960)
1		($27,744)	$1,778,676	($595,317)	$1,155,615	$1,010,153
2		($28,299)	$1,814,250	($607,223)	$1,178,727	$900,661
3		($28,865)	$1,850,535	($619,368)	$1,202,302	$803,037
4		($29,442)	$1,887,545	($631,755)	$1,226,348	$715,994
5		($30,031)	$1,925,296	($644,390)	$1,250,875	$638,387
6		($30,632)	$1,963,802	($657,278)	$1,275,892	$569,191
7		($31,244)	$2,003,078	($670,423)	$1,301,410	$507,495
8		($31,869)	$2,043,140	($683,832)	$1,327,439	$452,487
9		($32,507)	$2,084,002	($697,509)	$1,353,987	$403,441
10		($33,157)	$2,125,682	($711,459)	$1,381,067	$359,712
				Total Net Present Value:		**$3,386,598**

Table 10. Net present value: Second level implementation.

The results of the NPV analysis shows that level one implementation would not pay for five axis equipment purchase, installation, and maintenance (Table 9). For this hypothetical die plant, there is benefit in pursuing five axis technology only if level two implementation is carried out (Table 10). That is, there must be coordination between die design, die engineering, and die plants to cast cam adapters and die adapters directly into the die shoe casting, and to cast cam drivers directly into the punch shoe casting (See Figure 5.2.7).

The NPV is quite sensitive to annual cost savings and original equipment costs. For example, an 25% drop in equipment cost and installation results in a NPV of positive $41,000 (approximately) for level one implementation. Similarly, a 29% increase in annual savings for level one implementation results in a slightly positive NPV. Either of these changes (or a combination of the two) justifies the purchase and installation of five axis equipment for level one installation. Figure 7.1.1 shows the relationship between NPV, equipment cost, and annual savings for level one implementation.

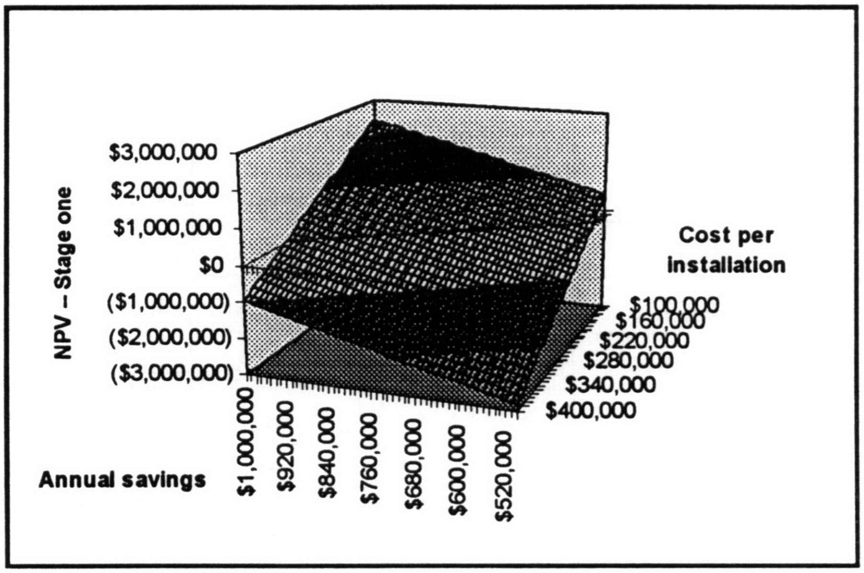


Figure 7.1.1 NPV with varying equipment cost and annual savings.

First level NPV for major U.S. auto maker

Discount Rate:		14.40%		Cost per installation:		$8,000
Inflation Rate:		2.00%		Estimated annual savings:		$673,800
Corp. Tax Rate:		34.00%		Est. maintenance / machine:		$1,600
Level One Implem. for major U.S. auto maker				Cost of CAM Software		$120,000
		Additional	Estimated	Taxes:		
Year	Investment	Expenses	Savings	Savings(Cost)	Net Savings	Present Value
0	($256,000)			$87,040	($168,960)	($168,960)
1		($27,744)	$687,276	($224,241)	$435,291	$380,499
2		($28,299)	$701,022	($228,726)	$443,997	$339,256
3		($28,865)	$715,042	($233,300)	$452,877	$302,484
4		($29,442)	$729,343	($237,966)	$461,934	$269,697
5		($30,031)	$743,930	($242,726)	$471,173	$240,464
6		($30,632)	$758,808	($247,580)	$480,597	$214,400
7		($31,244)	$773,984	($252,532)	$490,209	$191,161
8		($31,869)	$789,464	($257,582)	$500,013	$170,441
9		($32,507)	$805,253	($262,734)	$510,013	$151,966
10		($33,157)	$821,358	($267,989)	$520,213	$135,494
				Total Net Present Value:		**$2,226,902**

Table 11. Net present value: Level one implementation for auto maker.

Given that the major U.S. auto maker in this paper has purchased the five axis equipment[33] there is benefit in pursuing level one implementation.

This section provides a framework from which to financially evaluate five axis machining. Equipment cost and estimated savings must be calculated on a company (or in some cases die plant) specific basis. The NPV analysis for the hypothetical die plant

---

[33] Cost per installation reflects labor charge for setup and installation.

above shows that there is a definite financial benefit to level two implementation. For level one implementation, the NPV is negative. This may or may not be the case for other die development operations. A company that must purchase entirely new machining centers to exploit the benefits of five axis machining will probably calculate negative NPVs for both levels of implementation. On the other hand, the introduction of lower cost five axis spindles (less than $100,000) will lead to positive NPVs for both levels of implementation. This was in fact the case with one of the benchmark companies (See Section 3.2). The company purchased five axis spindle units as an option on a new machining center, since the estimated NPV of the incremental investment (in this case less than $100,000) was positive for even the first level of implementation. Finally, for a company that has already made the investment in capital equipment there is a definite financial benefit (i.e. large positive NPV) in exploiting five axis machining for die construction.

7.2 Causal loop analysis of technology implementation

7.2.1 Purpose of causal loop analysis

This section views the use of five axis technology at the die plants of the major U.S. auto maker from a systems dynamics perspective. With the use of causal loops, we can develop an explanation of how the current state came to be. The causal loop analysis aims to not only explain the auto maker's history with five axis machining, but provide insight into some key levers. The levers can be used to ensure successful utilization of the equipment in the future.

7.2.2 Causal loops and analysis

Figure 7.2.1 shows several causal loops that attempt to portray what has happened with the die plants. A (-) arrow between actions indicates an inverse relationship. A (+) arrow between actions indicates a positive relationship. For example, in loop B1, the arrow from 'Difficulty with five axis spindle' to 'Plant solves problem itself' is positive. This means that the more difficulty encountered with the spindle, the more the plant attempts to solve the problem. The arrow from 'Plant solves problem itself' to 'Difficulty with five axis spindle' is negative. This means that the more a plant tries to solve the problem, generally the fewer problems with the five axis spindle. A loop with an even number of negative arrows is called a reinforcing loop since the action is reinforced each cycle around the loop.

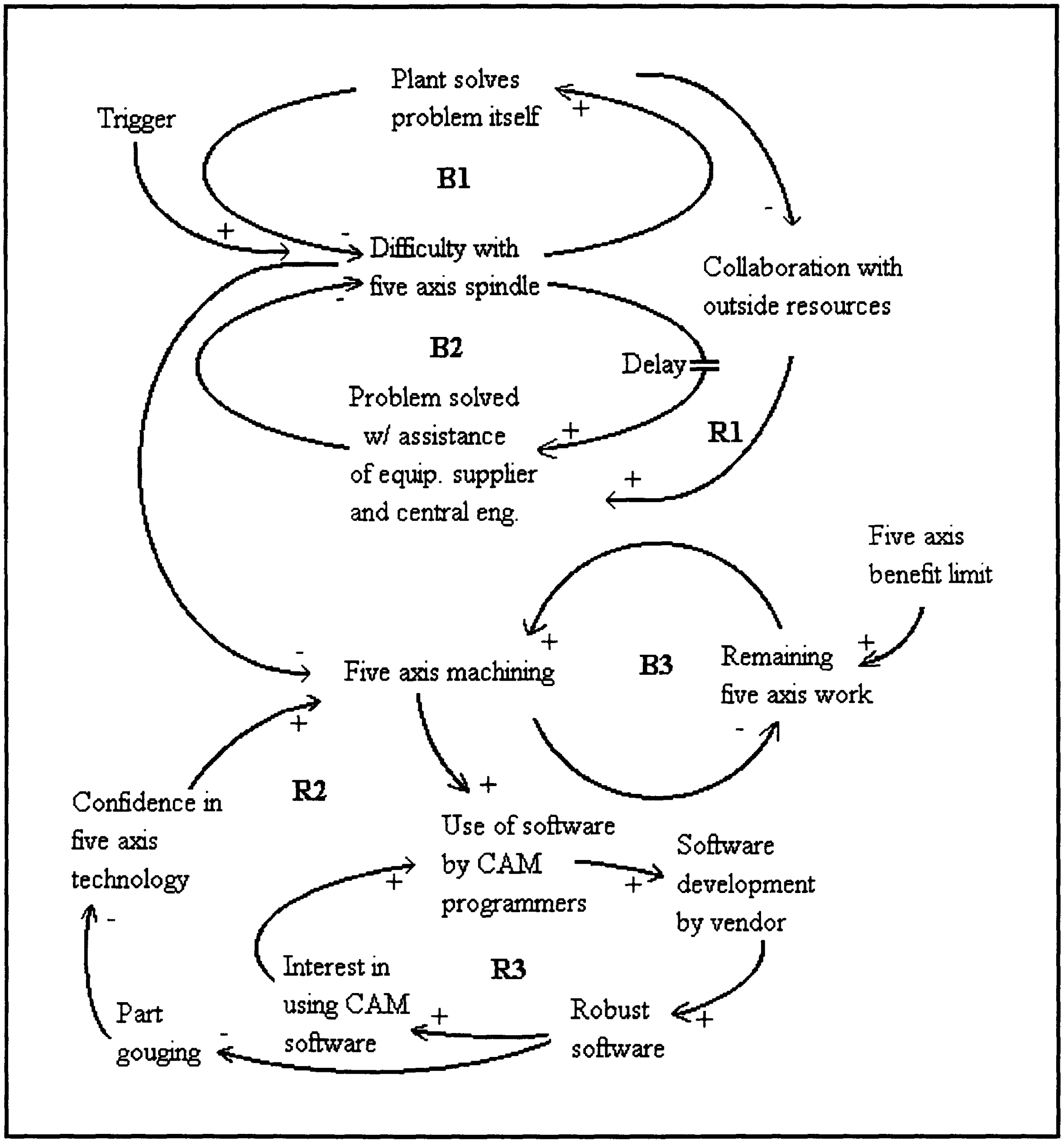


Figure 7.2.1 Causal loops for major U.S. auto maker.

A loop with an odd number of negative arrows is called a balancing loop since it has an equilibrium effect on the system. Two system archetypes (Senge, 1990) are evident in Figure 7.2.1. They are a 'shifting the burden' archetype and a 'limits to growth' archetype. The 'shifting the burden' archetype consists of balancing loops B1, B2, and reinforcing loop R1. When a die plant encounters problems with a five axis spindle, it has the option to solve the problem itself or to work with the equipment supplier and central engineering to solve the problem. The latter approach will bring the correct resources to bear on the problem, but using that option involves a delay. The delay consists of a approval delay (funding to pay for equipment manufacturer or central engineering) and a response time delay (unlike plant maintenance personnel, the equipment manufacturer and central engineering cannot respond instantly to a request). Due to budget and time constraints, the former option is generally taken. That is, the plant tries to solve the problem itself. Taking either route will usually result in a resolution of the difficulty. However, due to little training and experience with the five axis equipment, the plant solution tends to be a 'quick fix'.

The problem with using the 'quick fix' solution over time are two fold. First, the fundamental problem is never solved: 'Quick fixes' ultimately lead to more difficulties. Second, little communication and collaboration take place between the plants, equipment suppliers, and central engineering. This will lead to animosity between the groups, as was cited in Chapter 4. The reduced collaboration results in more difficulties with the five axis equipment since, as mentioned previously, the fundamental problem is not solved by the 'quick fix'. Initial difficulties with the five axis equipment and poor support on the part of both equipment suppliers and central engineering at the time of installation most likely provided the ‘trigger’ that led to die plants deciding to solve the problems in-house.

The second archetype, 'limits to growth', occupy balancing loop B3 and reinforcing loops R2 and R3. Balancing loop B3 simply says that the more five axis machining done, the less new work available for five axis. As more five axis machining is performed, there is less potential for new five axis work as all the benefits of five axis machining cited in Chapter 5 are exploited. This in turn results in less growth of five axis machining. At most die plants of the major U.S. auto maker, this balancing loop is dormant. So little five axis work is presently performed that for all intent, the 'five axis benefit limit' is equivalent to the 'remaining five axis work'.

Reinforcing loops R2 and R3 are very dominant in many of the die plants. Difficulties with the five axis spindle from the shifting the burden archetype triggers loops R2 and R3, but in a negative sense. That is, increased difficulties with five axis spindles leads to less five axis machining being done (equipment is down, frustrations are high). Less use of five axis machining leads to less use of the five axis CAM software. This in effect becomes a trigger for reinforcing loops R2 ad R3. As the software vendor sees the use of its five axis software fall, development priority for that software also falls (the 'death spiral' explained in Chapter 6). The drop in priority means that bugs are fixed slowly and the addition of new features is slow compared to the high use (three axis) software. This in turn makes the five axis software less robust relative to the three axis package and results in more part gouging when the five axis software is used (compared to when the three axis software is used). When the CAM programmers and machinists see more gouging after using the five axis machining, confidence in the five axis technology drops, which again leads to a drop in use of the technology.

Another reinforcing loop that applies to only the CAM programmers and software vendors is loop R3. As the use of five axis software drops, so does development for the software, resulting in a less robust package. With the package less robust than the high use (three axis) software, there is less interest in using the five axis software and CAM programmers try to do all their work with the more robust three axis package. This 'death spiral' has effectively eliminated use of the five axis package at the major U.S. auto maker. The software vender in turn, has suspended all development work on the software.

These two archetypes show how the die plants arrived at the state they are in today with their five axis technology. It is interesting to note that initial installation problems with the equipment supplier and central engineering triggered the 'shifting the burden' archetype, which in turn triggered the 'limits to growth' archetype. The next section will identify levers and how they can be used to realize the benefits offered in Chapter 5.

7.2.3 Levers for successful utilization

To reverse the effects of a reinforcing loop, one must select a lever (an element in that loop), that is controllable and will reverse the loop. A lever in the 'shifting the burden' archetype is to work with the equipment suppliers and central engineering. A lever in the 'limits to growth' archetype is the development of the software by the vendor. Each is briefly discussed.

Shifting the burden archetype

When a plant encounters difficulties with a five axis spindle, the reactive response is to solve the problem instantly. The proactive solution is to call in the equipment supplier and central engineering staff who have expertise on the equipment, are able to determine the root cause of the problem and repair it. This will not only reduce future difficulties with the five axis equipment, but properly managed will reverse the direction of the reinforcing loop and lead to increased collaboration with outside resources. This in turn will incite the plants to involve the equipment supplier and central engineering more often, reducing animosity and ultimately leading to a more collaborative approach.

However, a trigger is required to instigate the proactive response. This trigger may be in the form of central engineering assistance for the five axis spindles. If the central engineering organization, together with the equipment supplier were to approach each plant with an offer to repair and continually support the five axis equipment, the result could be a trigger to reverse reinforcing loop R1. The sustainability of this reversal depends on the commitment of the central engineering organization and the equipment supplier to continue supporting the equipment and work collaboratively with the plants. A sustained effort in this regard will lead to:

- Fewer equipment problems,
- Less animosity between the three groups,
- More communication and collaboration between the three groups.

Limits to growth archetype

To use more five axis machining, the users of the equipment must be confident in the technology. This means that the CAM software be robust and not gouge the part. To trigger both reinforcing loops R2 and R3 in the positive direction, the software vendors must provide a package that CAM programmers can use to confidently generate a gouge free tool path. The software vendors may select one of two routes here:

- Develop a world class, five axis software package, or
- Allow the die plants to purchase a world class five axis software package.

Either of these two routes will provide the die plants with a robust software package that will provide gouge free five axis tool paths. This will increase the interest in five axis programming and in turn result in more use of the five axis software by the CAM programmers. It will also build confidence in the five axis technology and lead to more five axis machining.

It is important to note that reversal of the 'shifting the burden' archetype is necessary to reverse the 'limits to growth' archetype, and both archetypes must be reversed to take advantage of the benefits cited in Chapter 5. Ongoing problems with the five axis spindle will ultimately eliminate gains made by an improved software package. Similarly, solving all the problems with the five axis equipment is of limited use if the software continues to gouge the part surface.

Once these archetypes are reversed and the die plants begin to approach the limit to five axis benefits, the requirement of increased communication and collaboration between functions (Section 6.5) becomes more important. The following section addresses the issues involved with that requirement and offers a framework from which to understand the difficulty such a requirement imposes.

7.3 Five axis machining as an architectural innovation

7.3.1 Introduction to architectural innovation

Research by Henderson (1990) on product innovation reveals that many innovations do not properly fit into a category of being either 'incremental' or 'radical'. She identifies a category of innovation that is considered 'architectural' because it does not affect the individual product components as much as it affects the relationship between the components: the system architecture, in other words[34].

		Core Concepts	
		Reinforced	Overturned
Linkages between Core Concepts and Components	Unchanged	Incremental Innovation	Modular Innovation
	Changed	Architectural Innovation	Radical Innovation

Figure 7.3.1 Henderson's innovation matrix.

Henderson developed an innovation matrix (Figure 7.3.1) that defines the various categories of innovation. It is important to note that most innovations occupy more than one cell in the matrix. The distinction here is to determine the dominant category. Core concepts refer to the components of the product. For example, a desktop personal computer's components are the motherboard, power supply, keyboard, monitor, etc. Each of these components embodies a core concept. The power supply, for example, may be 120 Volt AC, or battery powered DC. An incremental innovation for a desktop PC could be a faster motherboard, or a lighter keyboard. A modular innovation would involve overturning a

---

[34] Webster defines architecture as "A method or style of building". This same definition is used to denote a method of assembling a product.

core concept, such as moving from a CRT screen to an LCD screen. A radical PC innovation would be the currently popular hand-held personal data communicators which are hand-held, battery powered, pen based, and use an LCD screen. A PC innovation which may be considered an architectural innovation is the notebook computer. Although it contains several different core concepts (LCD screen and battery power), these core concepts are not new and the greater innovation is the new product architecture.

The architecture of a product is embedded in both the structure and information processing of the firm. A company that builds desktop PC's and wishes to enter the notebook market must face several structural changes. The co-location of the screen and chassis will require closer interaction by the groups responsible for those components. The problems of power management and the marketing constraints imposed on size and weight will require much closer interaction of all component groups. If this structural change conflicts with the company culture, the problem is magnified. Take the example of the computer company that wishes to build notebook computers. If the company is organized by components, the transition may be a difficult one. Being organized by components means that individual component divisions (representing monitors, keyboards, motherboards, etc.) operate separately in the organization and over time each has its own sub-culture. Before component groups can be expected to successfully work together, the problem of conflicting sub-cultures must be addressed.

It is important to note that product or process innovations may occupy several cells of the innovation matrix. In fact, most innovations have elements of several cells. The point is to recognize what cells are relevant and understand the subsequent effects these types of innovation can have on an organization.

7.3.2 Implications for five axis machining in die development

To take full advantage of five axis technology and achieve the maximum savings indicated in Section 7.1, a company must address the implications of the technology on the structure of the organization. A notebook computer can be viewed as a product architectural innovation, and five axis machining can be viewed as a process architectural innovation. Where the notebook computer affects linkages between product components, five axis machining similarly affects process components. Both of these innovations require a change in the structure of the organization.

		Core Concepts	
		Reinforced	Overturned
Linkages between Core Concepts and Components	Unchanged	Five axis drilling Four axis machining	EDM Five axis contouring
	Changed	Parts consolidation Swarf cutting	Rapid prototype

Figure 7.3.2 Innovation matrix for five axis machining.

Figure 7.3.2 shows Henderson's innovation matrix as applied to die construction and five axis machining, where applicable. Five axis drilling and four axis ball nose machining are incremental innovations as they do not change the core concepts employed by the individual processes or the linkages between those processes. Parts consolidation (as in cam components cast directly into die shoes) and using developable surfaces for areas to be swarf cut are more of an architectural innovation since they do not affect process components as much as they affect the linkage between those components. Electric Discharge Machining (EDM) and five axis contouring to finish machine a die is a modular innovation since it introduces a new core concept (EDM or simultaneous five axis) to replace traditional finish machining while not affecting the linkage between process elements. Finally, a radical innovation in die development could be some rapid prototyping tool where the prototype became the production die after a hardening process. This would change both core concepts and the linkage between process elements.

Henderson in her research cites many examples of companies having difficulties with architectural innovation. Both product and process architectural knowledge becomes embedded in the structure and procedures of a company. Therefore, to change the architecture requires changing the procedures and may even require modifying the structure to reflect the new architecture. For this specific auto maker's die development operations, this means changing the way different process elements interact. The 'increased communication' discussed in Section 6.5 addresses that very issue. It may seem at first a minor issue, to improve communication across functions, but history shows (Henderson, 1990) that time and again, companies who unknowingly embark on an innovation that is architectural in nature encounter many difficulties.

To achieve the 'increased communication' requires a conscious understanding that the change is architectural in nature and the way in which the individual process elements interact must be open for change. In other words, to achieve the benefit of parts consolidation, the die engineers must not only be willing but have an incentive to work with the die plants to consolidate parts into castings. With this incentive in place, the increased communication is much more likely to be fruitful. To have developable surfaces, the product designers and die plants must have a similar arrangement, that again will lead to more fruitful communications between the two. As noted in the computer company example, if the process changes are not compatible with the group culture, the culture problem must be addressed first.

8.0 Conclusions and recommendations

8.1 Conclusions

Based on the experimentation and analysis presented in this paper, there is significant benefit in pursuing five axis machining for the manufacture of stamping dies. Other die manufacturers are currently pursuing five axis machining as the cost of equipment drops and CAM software becomes more robust.

Past attempts at five axis machining at the major U.S. auto maker have failed. This is in large part due to three factors:

1. A 'shifting the burden' archetype,
2. Lack of support for and training on five axis CAM software, and
3. A view that five axis machining is solely an incremental process improvement.

The benefits cited in this paper are achievable in two steps or levels. The first level exploits savings from:

- Five axis drilling of coordinating holes,
- Four axis ball nose finish machining of 3D contours, and
- Five axis contour machining of outer panel draw dies.

Depending on actual equipment costs and die plant specific savings, the NPV for level one implementation may be negative. If this is the case, there is no benefit in pursuing five axis machining if the intent were only level one implementation. For the major U.S. auto maker, however, the only additional equipment needed is a robust CAM software package. Therefore, for this auto maker, even the level one NPV is positive.

The second level implementation involves:

- Four axis end mill machining, and
- Five axis swarf cutting.

Level two implementation looks beyond the die plant and exploits five axis machining to optimize the entire die development process. It represents a shift from a functional to a process focus and can be viewed as an architectural process innovation.

This paper only considered four axis end mill machining. Five axis swarf cutting will require significant process changes for the major U.S. auto maker. The NPV for level two implementation are positive by a large margin. It seems worthwhile for a die plant to invest in five axis equipment given it pursues and achieves level two implementation.

The current five axis machining center equipment (primarily nutating spindle units) at the die plants of the major U.S. auto maker are sufficient to exploit these benefits. The five axis spindle units need only to be properly calibrated, maintained, and periodically validated.

Robust five axis CAM software is commercially available and is only marginally more expensive than three axis software.

A critical requirement for success in five axis machining is the training of CAM programmers. They must understand how to select and process a die in five axis and fully understand the five axis CAM software.

8.2 Recommendations

The die development operations of the major U.S. auto maker should pursue level one implementation of five axis machining.

Central engineering should trigger the repair, maintenance, and use of the five axis equipment at the die plants. By committing resources to the die plants, central engineering can reverse the 'shifting the burden' archetype that led to animosity between die plants, equipment manufacturer, and central engineering.

Once the five axis equipment is used for drilling coordinating holes and four axis ball nose machining, the die plants should purchase robust five axis CAM software with a verification and simulation package. Given that the CAM software is such a critical element to the success of five axis machining, the die development operations should become involved in the definition and specification of the five axis CAM software used by its die plants.

Upon installing the software and training the CAM programmers, a process guide for five axis machining should be developed. Regular communications between the CAM room and machinists should be established to work out any 'glitches' in the software and to accelerate the die plant five axis learning curve.

Level two implementation should be pursued once the level one benefits are exploited, keeping in mind that level two benefits are an architectural process innovation, rather than an incremental or modular innovation.

The five axis die building project should begin with one machining center and one die plant. From this small group a sub-culture can be cultivated to avoid the difficulties encountered with a large scale improvement project. The initiative would grow from this machining center to the entire die plant and ultimately throughout the die development organization.

References

Albert, M., "Bypassing The Bench", Modern Machine Shop, Volume 66, Number 9, pp. 48-57, February 1994.

Bedi, Gravelle, Chen, "Principal Curvature Alignment Technique for Machining Complex Surfaces", Working paper, Department of Mechanical Engineering, University of Waterloo, Waterloo, Ontario, Canada, 1994.

Brealey, Richard A., Myers, Stewart C., “Principles of Corporate Finance”, Fourth Edition, McGraw Hill, 1991.

Drees, F.R., "A Comparative Analysis of Best Practice in the Manufacture of Hard Dies for Automotive Stamping Operations", M.S. thesis in Mechanical Engineering and S.M. thesis, M.I.T., 1991.

Eastman, M. "The CAM Path to Productivity", Cutting Tool Engineering, CTE Publications, Volume 46, Number 4, pp. 24-35, June 1994.

Esterling, D., "Reduce Machining Cost With NC Verification", Modern Machine Shop, Volume 66, Number 2, pp. 76-85, July 1993.

Giessen, Jurgen Held, "Milling machines for production of large presstools: CAD/CAM technology sets new standards", VDI-Z Development, Design, Production, VDI-Verlag GmbH, Issue 129 No.4 April 1987, pp. 53-63.

Griffith, W.H., "High Speed, High-Precision Machining -- A CNC Checklist", Modern Machine Shop, Volume 66, Number 2, pp. 64-73, July 1993.

Henderson, Rebecca M., “Managing Innovation in the Information Age”, Harvard Business Review, January-February 1994, pp. 100-105.

Henderson, Rebecca M., “Keeping Close to Your Customers: Architectural Innovation in the Market Place and the Failure of Established Firms”, Working Paper, MIT Sloan School of Management, February 1991.

Henderson, Rebecca M., Clark, Kim B., “Architectural Innovation: The Reconfiguration of Existing Product Technologies and the Failure of Established Firms”, Administrative Science Quarterly, Volume 35 (1990), pp. 9-30.

Hock, Stefan, Janovsky, Dirk, "Five Axis Milling Advances Die and Mould Making", European Production Engineering, Volume 16, Number 4, 1992, pp. 10-16.

Jones, T., "Horizontal Machining of Molds", Modern Machine Shop, Volume 66, Number 9, pp. 62-71, February 1994.

Leverton, T., Scoones, A., Karima, M., "On the use of Computer Based Simulation to Reduce Die Manufacturing Lead Times", International Body and Engineering Conference; Body Assembly and Manufacturing, pp. 99-105, 1993.

Lilly, B.W., Bailey, R.W., Altan, T., "Automated Finishing of Dies and Molds: A State of the Art Review", Computer Aided Design and Manufacturing of Dies and Molds, ASME, PED-Volume 32. 1988, pp. 75-90.

Manufacturing Solutions, "Five-axis adds a new machining dimension", Tooling and Production, March 1994, pp. 57-58.

Metcut Research Associates Inc., Machinability Data Center, "Machining Data Handbook, 3r Edition", Cincinnati, Ohio, 1980.

Modern Machine Shop, 1983 NC/CAM Guidebook, Gardner Publications, Volume 55, Number 8, January 1983.

Modern Machine Shop, 1993 Guide to CNC Technology and Manufacturing Software, Volume 65, Number 11A, April 1993.

Noaker, Paula M., "CAM at the Outer Limits", Manufacturing Engineering, ASME, November 1991, pp. 45-49.

Oetjens, Thomas J., "Automotive CAD/CAM Die: The Sculptured Surface", SME Autofact Conference Proceedings, SME.

Schein, Edgar H., “Organizational Culture and Leadership; Second Edition”, Jossey Bass, San Francisco, 1992.

Schey, John A., “Introduction to Manufacturing Processes; Second Edition”, McGraw Hill, 1987.
Senge, Peter M., “The Fifth Discipline”, Doubleday Currency, New York, 1990.

Shultz, D., "Why Five-Axis?", Modern Machine Shop, Volume 66, Number 2, pp. 50-60, July 1993.

Smith, G.T., "Advanced Machining: The Handbook of Cutting Technology", IFS Publications, UK, 1989.

Sprow, E.E., "Should we? Could we? Step up to five-axis programming?", Manufacturing Engineering, November 1993.

Takagaki, Toshikazu and Okuno, Toshiyuki, "5-Axis High-Precision High-Speed Milling Machine opens up A New Horizon", SNK Technical Report, Number 9, 1990.

Thomas, Robert J., "What Machines Can't Do: Politics and Technology in the Industrial Enterprise", University of California Press, Berkeley, 1994.

Tonshoff, H.K., Hernandez-Camacho, J., "Die Manufacturing by Five and Three Axis Milling", Journal of Mechanical Working Technology, Volume 20 (1989), pp. 109-119.

Trent, E.M., "Metal Cutting; Second Edition", Butterworths, London, 1984.

Vickers, G. W., Bedi, S., Haw, R., "The Definition and Manufacture of Compound Curvature Surfaces Using G-Surf", Computers in Industry, Volume 6, Number 3, pp. 173-183.

Vickers, G.W. and Quan, K.W., "Ball-Mills Versus End-Mills for Curved Surface Machining", Journal of Engineering for Industry; Transactions of the ASME, Volume 111, Number 1, pp. 22-26, February 1989.

Walker, Derrick Clarence, "Design of a Process Improvement Methodology for Die Construction", MIT, Master's Thesis, Cambridge, MA.

Whitney D.E., Tung, E.D., "Robot Grinding and Finishing of Cast Iron Stamping Dies", Journal of Dynamic Systems, Measurement, and Control, Volume 114, March 1992, pp. 132-140.

You, S.J. and Eman, K.F., "Scallop Removal in Die Milling by Tertiary Cutter Motion", Journal of Engineering for Industry; Transactions of the ASME, Volume 111 Number 7, pp. 213-9, August 1989.

Appendix A. Derivation of equations

MRR3.XLS Documentation

M. Zeni, October 1994.

This manual documents the Excel 4.0 spreadsheet named MRR3.XLS.

MRR3 calculates the material removal rate, cusp height, spindle horsepower, and material remaining for a milling operation. The spreadsheet can be used for both three and five axis machining over flat, inclined, and curved surfaces.

The eight parts of this document cover the different regions of the spreadsheet:

1. Cutter types
2. Data entry area
3. Data manipulation
4. Output area
5. Calculation area
6. Error checking
7. Cutting data
8. Equation derivation

Cusp Height, MRR, Horsepower, and Material Remaining										
For Ball Nose, Flat, and Radiused end mills										
M.Zeni 10/08/94										
			Flat	Radius	Tilted	BallNose	Flat EMill	Rad. EMill		
		Ball Nose	End Mill	End Mill	Ballnose	on incline	Contour	Contour		
Lead angle:		**XXXXXX**	**5**	**5**	**15**	**40**	**5**	**5**	**degrees**	
Cutter Diameter:		**1.00**	**2.00**	**2.00**	**1.00**	**1.00**	**2.00**	**2.00**	**inches**	
Corner Radius:		**XXXXXX**	**XXXXXX**	**0.250**	**XXXXXX**	**XXXXXX**	**XXXXXX**	**0.250**	**inches**	
Number of flutes:		**2**	**4**	**4**	**2**	**2**	**4**	**4**	**number**	
Chip Load:		**0.0080**	**0.0150**	**0.0150**	**0.0080**	**0.0080**	**0.0150**	**0.0150**	**inches**	
Depth of cut:		**0.0440**	**0.0440**	**0.0440**	**0.0440**	**0.0440**	**0.0440**	**0.0440**	**inches**	< = = (0.04+0.008/2)
Step Over:		**0.0600**	**0.2000**	**0.2000**	**0.0600**	**0.0600**	**0.2000**	**0.2000**	**inches**	
Cutting Speed:		**9000**	**9000**	**9000**	**9000**	**9000**	**9000**	**9000**	**in./min.**	< = = Coated Carbide
Surface Curvature (R)			**XXXXXX**	**XXXXXX**	**XXXXXX**	**XXXXXX**	**60.00**	**60.00**	**inches**	
						Valid			< = = Cut on ball, not shank	
Major Radius:		0.50	1.00	0.750	0.50	0.50	1.00	0.750	inches	
Effective Step Over:						0.0783			inches	
Lead angle (rads):			0.0873	0.0873	0.2618	0.6981	0.0873	0.0873	radians	
Effective Radius:		0.5000	1.0000	1.0000	0.5000	0.5000	1.0000	1.0000	inches	< = = +/- 45 deg. assumption
							0.00	0.00	radians	
Spindle RPM:		2865	1432	1432	1950	2865	1432	1432	rev/min.	< = = Max. 3800 Strght/1900
Feedrate:		46	86	86	31	46	86	86	in./min.	
Feed per cutter rev.:		0.0160	0.0600	0.0600	0.0160	0.0160	0.0600	0.0600	in./rev.	
Maximum Step Over:		0.4102	1.7376	1.7376	0.4102	0.4102	1.7376	1.7376	inches	
Cusp Height:		0.00090	0.00044	0.00044	0.00097	0.00262	0.00044	0.00052	inches	
Matl Removal Rate:		0.12	0.74	0.75	0.08	0.16	0.74	0.75	in^3/min	
MRR relative to Ball Nose:			618%	627%	68%	130%	618%	626%		
Alloy Spec.Power:		1.41	1.41	1.41	1.41	1.41	1.41	1.41		
Horsepower:		0.09	0.53	0.53	0.06	0.11	0.53	0.53	Hp	
Machining time:		**21.82**	**3.49**	**3.49**	**32.05**	**21.82**	**3.49**	**3.49**	**sec/in^2**	
Relative to 3 axis BN:			**16%**	**16%**	**147%**	**77%**	**16%**	**16%**		
Matl Remaining (in^3/in^2 surface)*1000:										
	Cusps:	0.3002	0.1455	0.1455	0.3002	0.5117	0.1510	0.1510		
	Wake:	0.0000	0.1298	0.0075	0.0000	0.0000	0.1297	0.0075		
	Total:	**0.3002**	**0.2753**	**0.1530**	**0.3002**	**0.5117**	**0.2808**	**0.1585**	in^3 per in^2 of	
									surface area (X1000)	
Calculation of MRR & wake area:			1	1			1	1	Cutting condition = = >	
Area Removed (A):			0.00877	0.00877			0.00877	0.00877	in.^2	1: depth of cut > = wake height
Ideal Cross Section:			0.00066	0.00066			0.00066	0.00066	in.^2	2: depth of cut < wake height
Area of Wake (A*):			0.00001	0.00000			0.00001	0.00000	in.^2	
Percentage lost:		0	1.48%	0.09%	0	0	1.48%	0.09%		
Difference due to curvature:							0.00001	0.00001	in.^2	
Error checking:										
	Total area:	0.00264	0.00880	0.00880	0.00264	0.00345	0.00880	0.00880	in.^2	
	Area cut+missed:	0.00264	0.00870	0.00879	0.00264	0.00345	0.00869	0.00879	in.^2	
	Difference:	0.00000	-0.00010	-0.00001	0.00000	0.00000	-0.00011	-0.00001	in.^2	
	Error:	0.00%	-1.18%	-0.07%	0.00%	0.00%	-1.23%	-0.12%	Percent	

Table A1. MRR3.XLS Spreadsheet.

1. Cutter types

The first row below the date is the title row. This row lists the seven types of cutters and cutting geometry that are calculated by the spreadsheet. Each of the seven columns of numbers references the cutter in the title row. The cutters are:

'Ball Nose': Traditional ball nose end mill used in conventional three axis machining. The surface this ball nose travels on is assumed planar and flat, and the axis of the ball nose is vertical and normal to the surface at all times.

'Flat End Mill': Flat bottom end mill cutting a planar surface; either flat or inclined. The end mill is inclined by a lead (or heel) angle and is cutting on its leading edge only. This column represents five axis machining with a flat end mill.

'Radius End Mill': Flat bottom end mill with a corner radius cutting a planar surface; either flat or inclined. Identical to the 'Flat End Mill' but with a corner radius. This tool is also called a toroidal cutter.

'Tilted Ball Nose': Traditional ball nose end mill cutting a planar flat surface. Unlike the first column, the tilted ball nose has its tool axis tilted some angle from the surface normal. The difference between tilted ball nose and traditional ball nose is that tilted ball nose may have more effective flutes (number of flutes) and the tilted ball nose is limited to the maximum Nutator spindle speed of 1900 rpm.

'Ball Nose on incline': Traditional ball nose end mill cutting an planar inclined surface. The ball nose tool vector is vertical at all times. The only difference between ball nose and ball nose on incline is the step over. This column can be used to approximate a ball nose on a curved surface.

'Flat EMill Contour': Flat bottom end mill cutting a curved surface. The end mill is inclined to the tangent of the surface at the point of contact by a specified amount.

'Rad. EMill Contour': Radius end mill cutting a curved surface. The radius end mill is inclined to the tangent of the surface at the point of contact by a specified amount.

The **'Ball Nose'** and **'Ball Nose on incline'** columns represent conventional three axis machining. The **'Tilted Ball nose'** column represents four axis machining, and the flat and radius end mill columns represent five axis machining.

2. Data entry area

The first region below the title row is the data entry area. Tool dimensions and cutting conditions are input in the data entry area. The area is boxed and highlighted in bold text. Data to be entered is:

> **Heel angle:** For the flat and radius end mills, this row specifies the tilt angle of the tool relative to the surface tangent. For the **'Tilted Ball nose column'**, the tilt angle specifies the rotation of the tool off the surface normal for four axis machining. For the **'Ball Nose on incline'** column the tilt angle represents the angle of a planar surface normal to the ball nose tool axis. All angles are specified in degrees.
>
> **Cutter Diameter:** Specifies the outside diameter of the tool in inches.
>
> **Corner Radius:** Specifies the corner radius on a radius end mill in inches. This field is applicable to the **'Radius End Mill'** and **'Rad. EMill Contour'** columns only.
>
> **Number of flutes:** The number of effective cutters per revolution of the tool. An indexable ball nose end mill may have two flutes but only one effective flute (or cutter).
>
> **Chip Load:** Specifies the desired metal chip thickness in inches. This is typically a tool material constraint.
>
> **Depth of cut:** Specifies the average depth the tool will cut when machining the surface (inches).
>
> **Step Over:** The distance in inches between subsequent passes over a surface. For all planar machining (flat and inclined), the distance is measured as viewed

from the machine XY plane. For all curved surface machining ('**Flat EMill Contour**' and '**Rad. EMill Contour**') the distance is measured as a straight line normal to the surface.

Cutting Speed: The maximum allowable cutting speed for the given tool material and cutting surface combination in inches per minute. This is a tabulated value.

Surface Curvature (R): The curvature (radius) of the surface for the '**Flat EMill Contour**' and '**Rad. EMill Contour**' columns in inches.

Tools for which the input does not apply are X'd out. For example there is no corner radius for a ball nose end mill; hence the corner radius cell for '**Ball Nose**' is X'd out.

		Flat	Radius	Tilted	Ballnose	FlatEMill	R.EMill	
	Bnose	E.Mill	E.Mill	Ballnose	on incl.	Contour	Contour	
Heel angle:	**XXXX**	**5**	**5**	**15**	**40**	**5**	**5**	**deg**
Cutter Dia.:	**1.00**	**2.00**	**2.00**	**1.00**	**1.00**	**2.00**	**2.00**	**in.**
Corner Radius:	**XXXX**	**XXXX**	**0.250**	**XXXXX**	**XXXXX**	**XXXXX**	**0.250**	**in.**
No. flutes:	**2**	**4**	**4**	**2**	**2**	**4**	**4**	**No.**
Chip Load:	**0.0080**	**0.0150**	**0.0150**	**0.0080**	**0.0080**	**0.0150**	**0.0150**	**in.**
Dep. of cut:	**0.0440**	**0.0440**	**0.0440**	**0.0440**	**0.0440**	**0.0440**	**0.0440**	**in.**
Step Over:	**0.0600**	**0.2000**	**0.2000**	**0.0600**	**0.0600**	**0.2000**	**0.2000**	**in.**
Cut. Speed:	**9000**	**9000**	**9000**	**9000**	**9000**	**9000**	**9000**	**ipm**
Surf. Curvature (R)		**XXXX**	**XXXX**	**XXXXX**	**XXXXX**	**60.00**	**60.00**	**in.**

Table A1-1: Data entry area.

3. Data manipulation

The data manipulation area translates the numbers input into the data entry area into values used by the equations to generate the output section. The data manipulation area has four rows:

Valid/Invalid cut: This row is only applicable for the **'Ball Nose on incline'** column. When the step over is so large that the ball nose begins to cut on the shank of the tool, the cell outputs an 'Invalid' and the results for that column are no longer valid. So long as the tool cuts on its radius, a 'Valid' flag is output which means the calculations for that column are valid.

Major Radius: For all cutters but the radius end mill (**'Radius End Mill'** and **'Rad. EMill Contour'** columns), the major radius equals the cutter radius. For the radius end mill cutters, the major radius equals the cutter radius minus the corner radius.

Effective Step Over: This row is only applicable for the **'Ball Nose on incline'** column. The effective step over equals the input step over divided by the cosine of the incline angle. As the incline increases and the step over normal to the tool vector remains constant, the step over normal to the part surface increases.

Heel angle (rads): Equals the angle from the data input area in radians.

Effective radius: Typically set to half the cutter diameter. Represents the largest radius where the tool touches the part.

					Valid	< Cut on ball, not shank		
Major Radius:	0.50	1.00	0.750	0.50	0.50	1.00	0.750	inches
Eff. StepOver:					0.0783			inches
Heel angle (rads):		0.0873	0.0873	0.2618	0.6981	0.0873	0.0873	rads
Effective Radius:	0.5000	1.0000	1.0000	0.5000	0.5000	1.0000	1.0000	inches

Table A1-2: Data manipulation area

4. Output area

The output area presents the spreadsheet results. Refer to the end of this Appendix for the math used to generate the output. The outputs are:

Spindle RPM: Maximum recommended spindle speed in revolutions per minute. The spindle speed is limited by the spindle motor. The maximum is set to 3800 rpm for a straight spindle and 1900 rpm for the Nutator.

Feed rate: Maximum recommended feed rate in inches per minute.

Feed per flute: Feed in inches for every revolution of the cutter (Feed rate/Spindle RPM).

Maximum Step Over: Increasing the step over beyond this limit will result in significant uncut material and make the calculations invalid (inches).

Cusp Height: The expected cusp height above the part surface in inches.

Material Removal Rate: The amount of material being removed given the feed rate and area of cut in cubic inches per minute.

MRR relative to Ball Nose: The material removal rate as a percentage of the **'Ball Nose'** material removal rate. Greater than 100% represents a higher material removal rate than **'Ball Nose'**.

Spec.Power: Specific horsepower for the given material and material removal rate. Number entered manually from the table at the bottom of the spreadsheet (in^3/min/hp).

Horsepower: The theoretical spindle horsepower. Equals **Material Removal Rate** divided by **Spec.Power**.

The next three rows calculate the material that was not removed by the cutter. The results are multiplied by 1000 to easily compare numbers.

Material Remaining -- Cusps: Material left between subsequent tool path step overs (in^3 remaining per in^2 of surface area times 1000) which will be removed by benching.

Material Remaining -- Wake: Material left between subsequent cutter rotations (in^3 remaining per in^2 of surface area times 1000).

Material Remaining -- Total: Total of above two rows. Total material required to remove via benching to arrive at part surface (in^3 remaining per in^2 of surface area times 1000). Material remaining is an indication of bench time.

Machining Time: Time taken to machine the surface in seconds per in^2 of surface.

Relative to 3 axis BN: Machining time as a percentage of '**Ball Nose**' machining time. Less than 100% represents a lower machining time than '**Ball Nose**'.

Spindle RPM:	2865	1432	1432	1950	2865	1432	1432	rev/min.
Feed rate:	46	86	86	31	46	86	86	in./min.
Feed per flute:	0.0160	0.0600	0.0600	0.0160	0.0160	0.0600	0.0600	in./rev.
Max. Step Over:	0.4102	1.7376	1.7376	0.4102	0.4102	1.7376	1.7376	inches
Cusp Height:	0.0009	0.0004	0.0004	0.0009	0.0026	0.0004	0.0005	inches
Matl Removal Rate:	0.12	0.74	0.75	0.08	0.16	0.74	0.75	in^3/min
MRR relative to Ball Nose:		618%	627%	68%	130%	618%	626%	
Matl. Spec. HP:	1.41	1.41	1.41	1.41	1.41	1.41	1.41	
Horsepower:	0.09	0.53	0.53	0.06	0.11	0.53	0.53	Hp
Machining time:	**21.82**	**3.49**	**3.49**	**32.05**	**21.82**	**3.49**	**3.49**	**sec/in^2**
Relative to 3 axis BN:		**16%**	**16%**	**147%**	**77%**	**16%**	**16%**	

Table A1-3: Output area.

5. Calculation area

This region is used to calculate material removal rate and material remaining. See Appendix for mathematical development. Variables such as A and A* are referenced at the end of this Appendix. All quantities (other than percentages) are in square inches.

Calculation of MRR & wake area: This row is used as a title for the calculation area and an indicator of the cutting conditions for flat and radius end mills. A cutting condition of 1 indicates that the specified depth of cut is higher than the wake left by the cutter. A cutting condition of 2 indicates that the wake left by the cutter is higher that the depth of cut. The calculations are valid for either cutting condition. A cutting condition of 2 only occurs with high heel angle, high feed rate cuts.

Area Removed (A): The frontal area removed by the cutter. The frontal area is the area projected in a plane perpendicular to the cutter path. In the case of a curved cutter path, it is the area projected in a plane perpendicular to a line tangent to the cutter path at the point the cutter intersects the cutter path.

Ideal Cross Section: The area removed as viewed from the side of the cutter (normal to the cutter path), not including wake material.

Area of Wake (A*): Material left between flutes of the cutter, as the cutter travels along the cutter path.

Percentage lost: Percentage of '**Ideal Cross Section**' lost to wake. Equals '**Area of Wake**' divided by '**Ideal Cross Section**'.

Difference due to curvature: Additional area missed due to the curved surface in **'Flat EMill Contour'** and **'Rad. EMill Contour'** columns. Additional area missed is approximately zero for all ball nose columns.

Calc.of MRR, wake area:		1	1			1	1	Cutting Cond.	
Area Remd. (A):		.00877	.00877			.00877	.00877	in.^2	
Ideal X Section:		.00066	.00066			.00066	.00066	in.^2	
Wake Area (A*):		.00001	.00000			.00001	.00000	in.^2	
Percentage lost:	0	1.48%	0.09%	0	0	1.48%	0.09%		
Diff. due to curvature:						.00001	.00001	in.^2	

Table A1-4: Calculation area.

6. Error checking

This region sums the material removed and material remaining per step over to ensure it sums up to the depth of cut per step over, in square inches.

Total area: Equals the total area to be removed per step over. Equals **'Depth of cut'** times **'Step Over'**.

Area cut + missed: Calculated area cut by cutter plus calculated material remaining, per step over.

Difference: Difference between previous two rows (should be zero).

Error: Error in calculations as a percent of **'Total area'**.

Error checking:								
Total area:	0.00264	0.00880	0.00880	0.00264	0.00345	0.00880	0.00880	in.^2
Area cut+misd:	0.00264	0.00870	0.00879	0.00264	0.00345	0.00869	0.00879	in.^2
Difference:	0.00000	-0.00010	-0.00001	0.00000	0.00000	-0.00011	-0.00001	in.^2
Error:	0.00%	-1.18%	-0.07%	0.00%	0.00%	-1.23%	-0.12%	Percent

Table A1-5: Error checking area.

7. Cutting data

The cutting data region at the bottom of the spreadsheet contains two tables. The first table lists the maximum cutting speed for a given cutter material machining a specific material. Once the tool material is selected, the corresponding **'Cutting Speed'** is entered in the data entry area. The second table lists specific force and horsepower requirements for various cast irons and steels. For a given material, the corresponding **'Spec.Power'** is entered into the output region.

For Alloy 4	**Vc**
Material	**(ipm)**
HSS	780
Coated HSS	1200
Sintered Carbide	6000
Coated Carbide	9000
Cermet	12000
CBN	18000
Ceramic Alum. Oxide	24000
Ceramic Silicone Nitrate	24000
Diamond Tipped	30000
Diamond Solid	36000

Table A1-6: Cutting speed data.

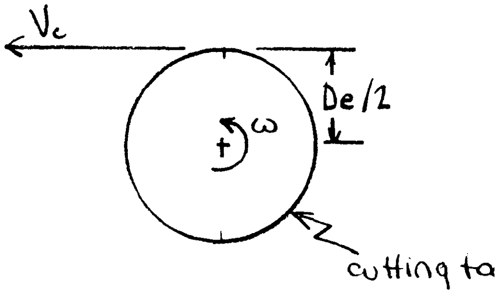
	Ks	**Vp**
Material	#/in^2	in3/mn/hp
Alloy 1	1227.26	2.23
Alloy2	1763.84	1.55
Alloy 3	1322.88	2.05
Alloy 4	1956.76	1.41
Alloy 5	2452.84	1.09
AISI-1020	2893.80	0.96
AISI-W2	3431.22	0.77
AISI-S7	3086.72	0.87
AISI-D2	3575.91	0.77
AISI-1	3431.22	0.77
AISI-6	3431.22	0.77
AISI-A2	3575.91	0.77

Table A1-7: Force and hp data.

8. Derivation of equations

Cutter Velocity

V_c = cutter velocity [in/min] or [ft/min]
De = cutter effective diameter [in]
RPM = spindle rev/min
ω = spindle rotation [rad/sec]



$$V_c = \omega \frac{D_e}{2} \quad [\text{in/sec}]$$

$$\omega = \frac{2\pi}{60} \cdot \text{RPM}$$

$$\therefore V_c = \pi \cdot \frac{\text{RPM}}{60} \cdot De \quad [\text{in/sec}]$$

$$V_c = \pi \cdot \text{RPM} \cdot De \quad [\text{in/min}] \longleftarrow$$

$$= \frac{\pi}{12} \text{RPM} \cdot De \quad [\text{ft/min}] \longleftarrow$$

Feedrate

f = feed rate [in/min]
V_c = cutter velocity [in/min]
c = chip load [in/tooth]
n = # effective teeth [teeth/rev]
D_e = cutter effective diameter

$$\frac{V_c}{\pi D_e} = \text{RPM} \quad [\text{rev/min}]$$

every revolution, the tool may advance $c \cdot n$ inches

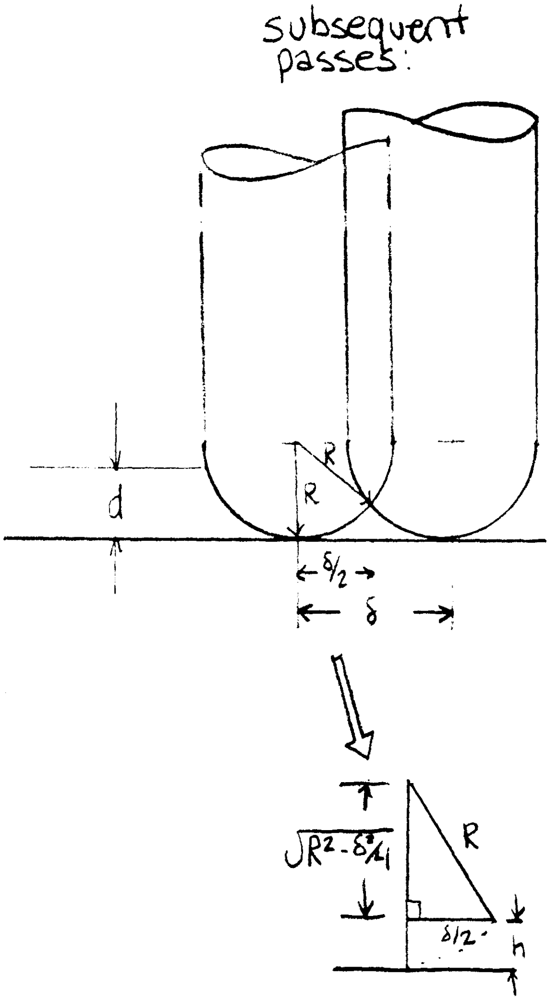
$$\therefore \ f = c \cdot n \cdot \text{RPM}$$

$$\boxed{f = \frac{V_c \cdot c \cdot n}{\pi D_e}} \longleftarrow$$

Ball Nose End Mill Calculations

(i) cusp height

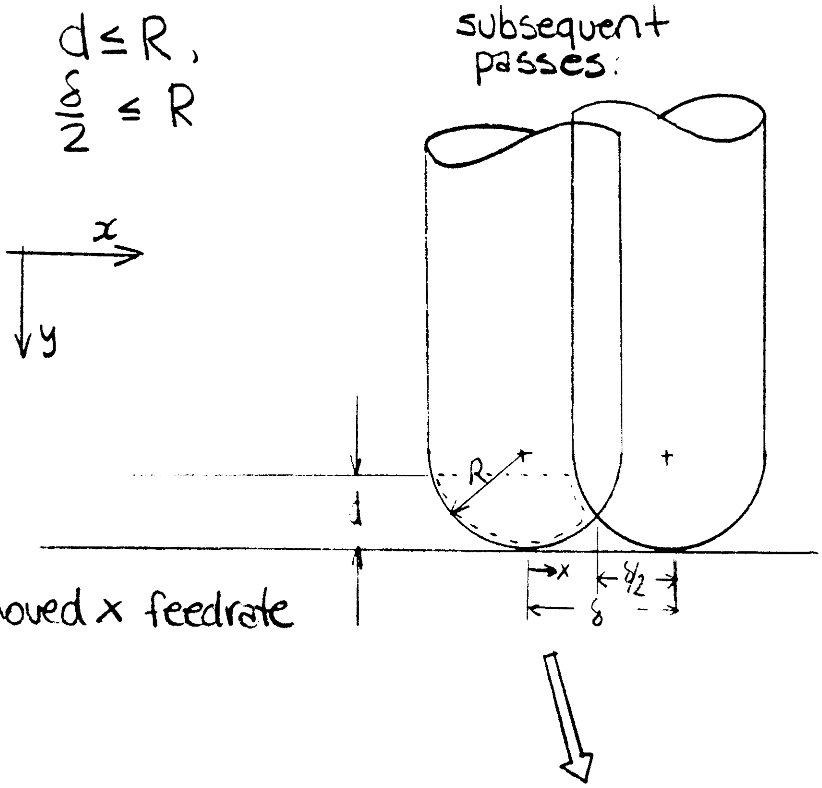
h= cusp height [in.]
R= ball nose radius [in.]
δ= step over [in.]
d= depth of cut [in.]



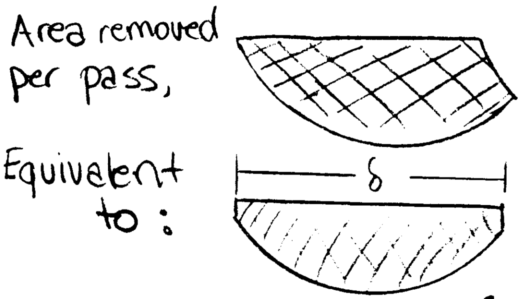
$$\boxed{h = R - \sqrt{R^2 - \delta^2/4}}$$

(ii) Material removal rate (MRR)

constraints: $d \le R$, $\frac{\delta}{2} \le R$



MRR = Area removed × feedrate



$$\text{Area removed} = 2\int_0^{\delta/2}\left(\sqrt{R^2-x^2}-R+d\right)dx = d\delta - R\delta + 2\int_0^{\delta/2}\sqrt{R^2-x^2}\,dx$$

$$= \delta(d-R) + 2\left[\frac{\delta}{4}\sqrt{R^2-\frac{\delta^2}{4}} + \frac{1}{2}R^2\sin^{-1}\frac{\delta}{2R}\right]$$

$$= \delta(d-R) + \frac{\delta}{2}\sqrt{R^2-\frac{\delta^2}{4}} + R^2\sin^{-1}\delta/2R$$

$$\boxed{MRR = f\delta(d-R) + \frac{f\delta}{2}\sqrt{R^2-\delta^2/4} + fR^2\sin^{-1}\frac{\delta}{2R} \quad \left[\frac{in^3}{sec}\right]}$$

Max step over (δ_{max}) @ $x = \delta/2$; $y = R-d$

$$\frac{\delta_{max}^2}{4} + (R-d)^2 = R^2 \quad , \text{or } \delta_{max} = 2\sqrt{d(2R-d)}$$

(iii) <u>Material Remaining</u>. (V) [in³/in² surface area]

$V = V_{cusp} + V_{WAKE}$ ($V_{WAKE} = 0$ for all ballnose)

$\therefore\ V = V_{cusp}$

$$= \int_0^{\delta/2} (R-y)\,dx = \int_0^{\delta/2} R - \sqrt{R^2-x^2}\,dx$$

$$= \frac{R\delta}{2} - \int_0^{\delta/2} \sqrt{R^2-x^2}\,dx$$

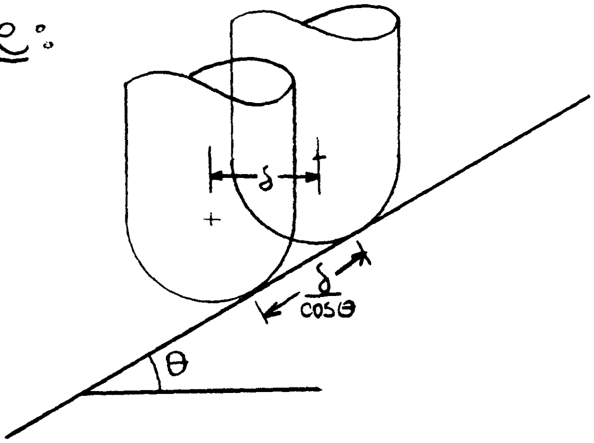
$$= \frac{R\delta}{2}\left[\frac{\delta}{4}\sqrt{R^2-\frac{\delta^2}{4}} + \frac{R^2}{2}\sin^{-1}\frac{\delta}{2R}\right]$$

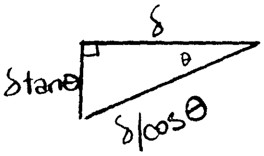
$$= \frac{R\delta}{2} - \frac{\delta}{4}\sqrt{R^2-\frac{\delta^2}{4}} - \frac{R^2}{2}\sin^{-1}\frac{\delta}{2R}$$ [per $\delta/2$ in² of surface area]

$$\boxed{\div\ \delta/2 ;\quad \underline{V = R - \frac{1}{2}\sqrt{R^2-\delta^2/4} - \frac{R^2}{\delta}\sin^{-1}\frac{\delta}{2R}}\ \left[\frac{\text{in}^3\ \text{mat'l.}}{\text{in}^2\ \text{surface}}\right]}$$

Ball nose on an incline:

θ = inclination angle [deg.]





cusp height (h),
material removal rate (MRR), and
material to remove (V)
same as flat (no incline) and $\delta = \delta/\cos\theta$

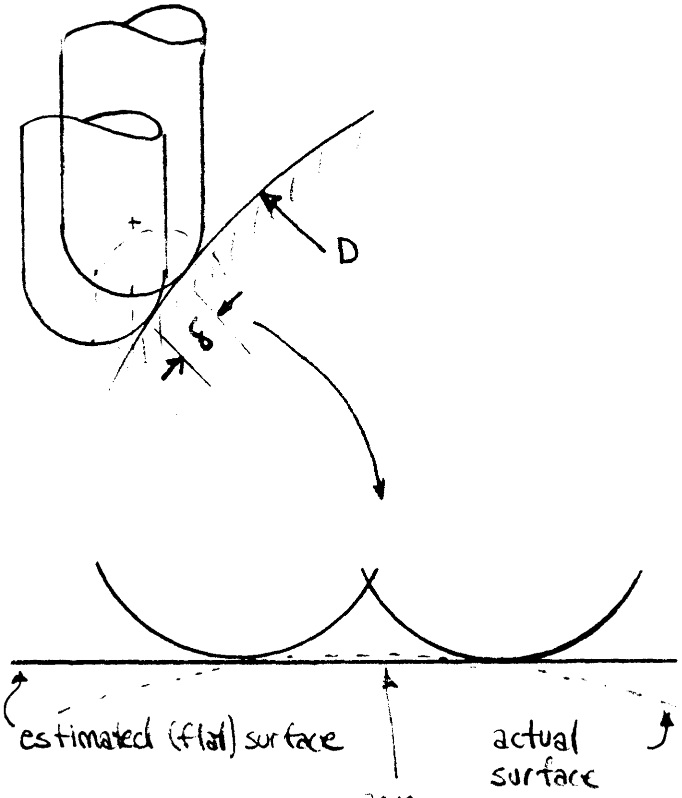
given max. δ, R, θ such that:

$$\delta(1+\tan^2\theta)-2R=0$$

Ball nose on a curved surface (contour)

Assume this case is the same as ball nose on an incline. Error due to this assumption:

D = contour diameter [in.]
θ = step over angle [rad]



∇ = wedge area [in²]
$\triangledown$ = triangle area [in²]
error = $\nabla - \triangledown$

$$\nabla = \frac{\theta D^2}{8} \text{ [in}^2\text{]}$$

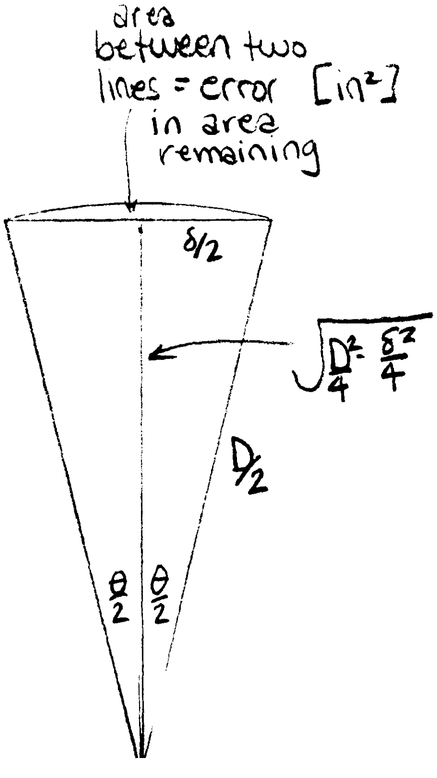
$$\triangledown = \frac{\delta}{4}\sqrt{D^2 - \delta^2}$$

$$\boxed{\text{error} = \frac{1}{4}\left(\frac{D^2\theta}{2} - \delta\sqrt{D^2-\delta^2}\right) \text{ [in}^2\text{]}}$$

w/ δ = 0.100" & 2" ϕ ball nose;

error in area remaining due to flatness assumption:

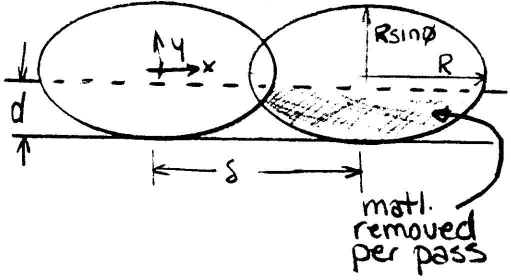
$\cong$ 1% with D = 40 inches
$\cong$ 0.5% with D = 80 inches



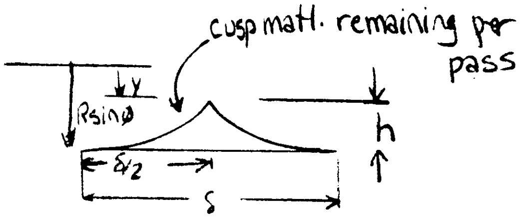
Flat bottom end mill calculations

(i) cusp height

subsequent passes:



R = end mill radius [in.]
ø = heel angle [rads]
δ = step over [in.]
d = depth of cut [in.]
h = cusp height [in.]



equation of ellipse:

$$\frac{x^2}{R^2} + \frac{y^2}{R^2\sin^2\phi} = 1$$

$$@\ x = \delta/2;\quad y_{\delta/2} = R\sin\phi\sqrt{1-\delta^2/4R^2} \qquad y_{\delta/2} + h = R\sin\phi$$

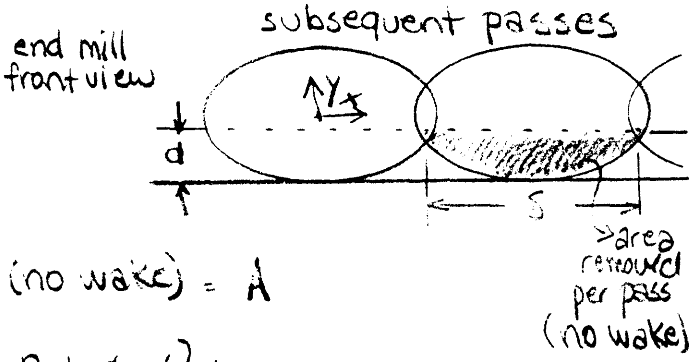
$$\therefore \boxed{h = R\sin\phi\left(1-\sqrt{1-\delta^2/4R^2}\right)} \quad [\text{in.}] \quad \{d \geq h\}$$

If $d < h$; cusp height = d [in.]

for MRR:

f = feed rate [in./min]
s = spindle speed [RPM]
f/s = advance per rev. [in./rev.]
n = # effective teeth [#]
$\frac{f}{s \cdot n}$ = advance per tooth [in]

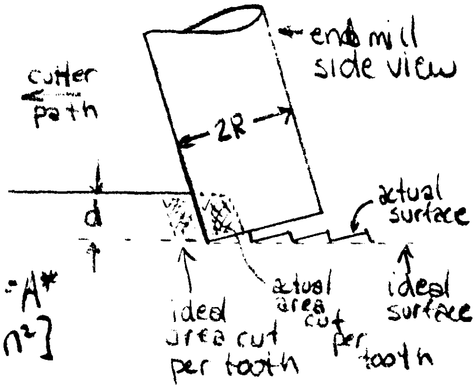
(ii) Material removal rate (MRR)



Area removed per pass (no wake) = A

$$A = 2\int_0^{\delta/2}\left\{\sin\phi\sqrt{R^2-x^2} - R\sin\phi + d\right\}dx$$

$$A = \frac{\delta\sin\phi}{2}\sqrt{R^2-\delta^2/4} + R^2\sin\phi\,\sin^{-1}\frac{\delta}{2R} - R\delta\sin\phi + d\delta \;[in^2]$$



material missed in wake = A^*
(in cross section)

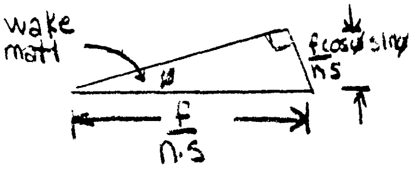
ideal area cut per tooth = $\frac{fd}{sn}$ $[in^2]$

area missed = $\frac{1}{2}\left(\frac{f}{n\cdot s}\right)^2\cos\phi\sin\phi = A^*$ $[in^2]$

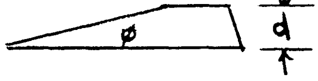
fraction of ideal area missed.

$$= \frac{f}{2\cdot n\cdot s\cdot d}\cos\phi\sin\phi$$

$$\left[d \geq \frac{f}{n\cdot s}\cos\phi\sin\phi\right]$$



if $d < \frac{f}{n\cdot s}\cos\phi\sin\phi$:



fraction of area missed
if $d < \frac{f}{n\cdot s}\cos\phi\sin\phi$:

$$A^* = \frac{1}{2}\left(\frac{f}{n s}\right)^2\cos\phi\sin\phi - \frac{\left(\frac{f}{ns}\cos\phi\sin\phi - d\right)^2}{2\sin\phi\cos\phi} \quad [in^2]$$

Assume $d \geq \frac{f}{n \cdot s} \cos\phi \sin\phi$ [in] (typical)

Actual area removed $= A\left(1 - \frac{A^*}{f \cdot d/(s \cdot n)}\right)$

MRR = feed rate * actual area removed

$$\boxed{MRR = fA\left(1 - \frac{s \cdot n}{f \cdot d} A^*\right) \; [in^3/min]}$$

(iii) Material remaining : = cusp + wake material

cusp material $= \frac{R\delta}{2} \sin\phi - \int_0^{\delta/2} \sin\phi \sqrt{R^2 - x^2}\, dx \quad [in^3/_{\delta/2} in^2 \text{ surface area}]$

$$= R\sin\phi - \frac{\sin\phi}{2}\sqrt{R^2 - \delta^2/4} - \frac{R^2}{\delta}\sin\phi \, \mathrm{Sin}^{-1}\frac{\delta}{2R} \quad \left[in^3/in^2 \text{ surface area}\right]$$

wake material $= A \cdot A^* \cdot \frac{s \cdot n}{f \cdot d} \quad \left[in^3/in^2 \text{ surface area}\right]$

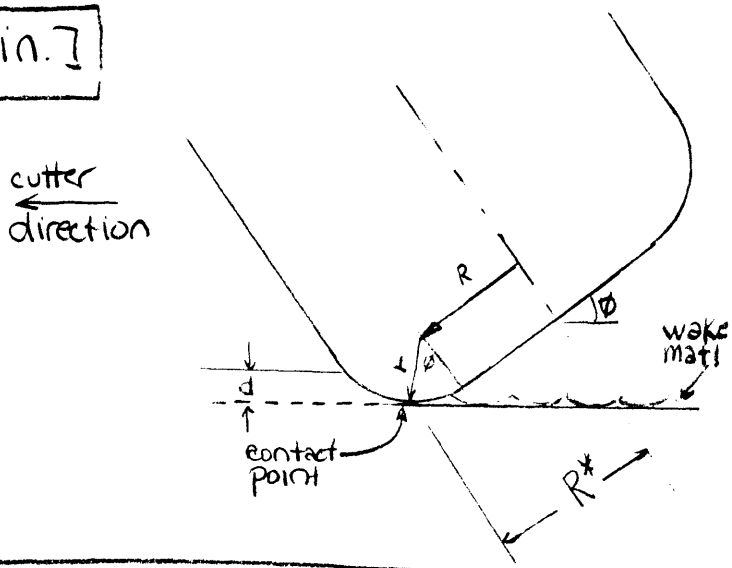
total material remaining

$$\boxed{R\sin\phi - \frac{\sin\phi}{2}\sqrt{R^2 - \delta^2/4} - \frac{R^2}{\delta}\sin\phi \, \mathrm{Sin}^{-1}\frac{\delta}{2R} + A \cdot A^* \cdot \frac{s \cdot n}{f \cdot d} \quad \left[\frac{in^3}{in^2 \text{ surface}}\right]}$$

Radius end mill calculations

Model a radius end mill as a flat bottom end mill with an "effective" cutter radius:

$$\boxed{R^* = R + r\sin\phi \quad [\text{in.}]}$$



R= major radius [in.]

r= minor radius [in.]
(= corner radius)

∴ from end mill calculations:

(i) cusp height

$$\boxed{h = (R + r\sin\phi)\left(1 - \sqrt{1 - \frac{\delta^2}{4(R + r\sin\phi)^2}}\right)(\sin\phi) \quad [\text{in.}]}$$

(ii) Material removal rate

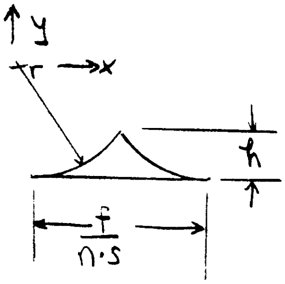
$$\boxed{MRR = fA\left(1 - \frac{sn}{fd}A^*\right) \quad [\text{in}^3/\text{min}]}$$

where:

$$\boxed{A = \frac{\delta\sin\phi}{2}\sqrt{R^{*2} - \frac{\delta^2}{4}} + R^{*2}\sin\phi\,\sin^{-1}\frac{\delta}{2R^*} - R^*\delta\sin\phi + \delta d \quad [\text{in}^2]}$$

A^* = matl. not removed in wake
= wake material

for $h \le d$:



h= height of cusp on wake [in.]

$$h = r - \sqrt{r^2 - \frac{f^2}{4n^2s^2}} \quad [\text{in.}]$$

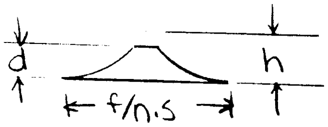
$$A^* = \text{wake area} = 2\int_0^{\frac{f}{2ns}} \left(r - \sqrt{r^2 - x^2}\right)dx$$

$$\boxed{A^* = \frac{rf}{ns} - \frac{f}{2ns}\sqrt{r^2 - \frac{f^2}{4n^2s^2}} - r^2 \sin^{-1}\frac{f}{2n\cdot s\cdot r} \quad [\text{in}^2]}$$

for $h > d$:

$$A^* = 2\left[\int_0^{\sqrt{r^2-(r-d)^2}} \left(r - \sqrt{r^2 - x^2}\right)dx + d\cdot\left(\frac{f}{2ns} - \sqrt{r^2-(r-d)^2}\right)\right]$$

$$\boxed{A^* = \sqrt{d(2r-d)}\,(r-d) - r^2\sin^{-1}\frac{\sqrt{d(2r-d)}}{r} + \frac{fd}{ns}}$$



$$MRR = f\left[A\left(1 - \frac{n\cdot s}{f\cdot d}\cdot A^*\right)\right]$$

$$\boxed{MRR = fA\left(1 - \frac{n\cdot s}{fd}A^*\right) \quad \frac{\text{in}^3}{\text{min}}}$$

(iii) Material remaining = cusp + wake material

cusp material (same a flat bottom end mill with $R^{*} = R + r\sin\phi$

$$V_c = (R + r\sin\phi)\sin\phi - \frac{\sin\phi}{2}\sqrt{(R + r\sin\phi)^2 - \frac{\delta^2}{4}} - \frac{(R + r\sin\phi)^2}{\delta}\sin\phi \, \mathrm{Sin}^{-1}\frac{\delta}{2(R + r\sin}$$

wake material $= A \cdot A^{*} \cdot \frac{s \cdot n}{f \cdot d}$ [in^3 / in^2 surface area]

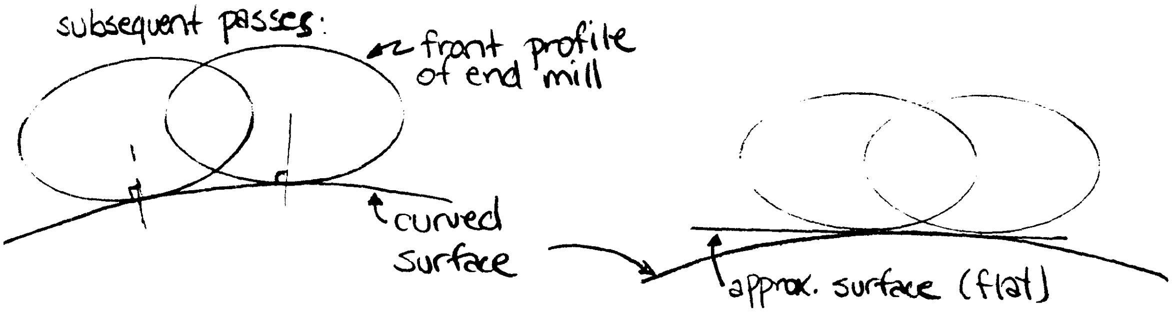
total material remaining:

$$\boxed{(R + r\sin\phi)\sin\phi - \frac{\sin\phi}{2}\sqrt{(R + r\sin\phi)^2 - \delta^2/4} - \frac{(R + r\sin\phi)^2}{\delta}\sin\phi \, \mathrm{Sin}^{-1}\frac{\delta}{2(R + r\sin\phi} + A \cdot A^{*}\frac{s \cdot n}{f \cdot d}}$$

[in^3 matl. / in^2 of surface area]

End mill (flat or radius) on a curved surface

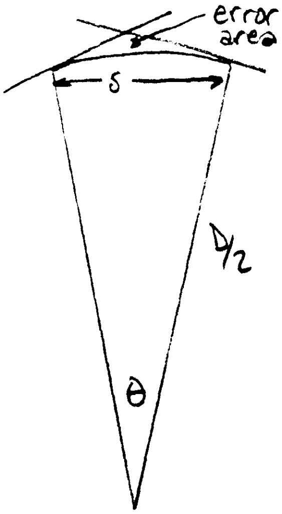
Assume surface is flat:



θ = step over angle [rad]
D: surface diameter [in.]

additional area missed = error area

$$\boxed{\begin{aligned} &= \frac{D^2}{4}\tan\frac{\theta}{2} - \frac{D^2\theta}{8}\left[\frac{\text{in}^2}{\text{stepover}}\right] \\ &\theta = \cos^{-1}\left[1-\frac{2\delta^2}{D^2}\right] = 2\sin^{-1}\delta/D \end{aligned}}$$



Appendix B. Machining forces, feeds, and speeds

In metal machining, cutting forces are perpendicular to the contact surface between the cutting tooth and the part surface according to:

$$F = A \times K_s$$

Where A is the area of contact between the tool and work piece, and K_s is an empirical value determined by the work piece material. The cutting force is defined in three dimensions by the tool radial rake angle (γ_r) and tool axial rake angle (γ_a). The three dimensions in the tool (local) coordinate system are radial, tangential, and axial, according to:

$$\begin{aligned}
F_r &= -F \times \sin(\gamma_r)\cos(\gamma_a) \\
F_t &= F \times \cos(\gamma_r)\cos(\gamma_a) \\
F_a &= F \times \sin(\gamma_a)
\end{aligned}$$

This only expresses the force applied to each tooth. Summing over all (n) teeth and representing the forces in the machine X, Y, and Z coordinates:

$$\begin{aligned}
F_x &= \sum_n (F_r \times \cos(\alpha) + F_t \times \sin(\alpha)) \\
F_y &= \sum_n (F_r \times \sin(\alpha) - F_t \times \cos(\alpha)) \\
F_z &= \sum_n F_a(\alpha)
\end{aligned}$$

Where (α) is the angle between the radial axis and X axis.

Appendix C. CAM experiment

Purpose

Compare three and five axis machining to determine which dies and what die features are good candidates for five axis machining. Five axis contouring and simultaneous five axis swarf cutting were evaluated. With the results a die plant will be able to select dies or regions of a die where five axis machining will lead to shorter machining time, better part accuracy, and less hand grinding.

Conclusions

The following conclusions are based on analysis of the data and informal interviews with CAM programmers, machinists, and die makers.

Due to the limitations imposed by THE CAM SOFTWARE (abbreviated TCS, the name of the software is not revealed here for confidentiality reasons) and the product design, only several types of dies and die features are good candidates for five axis machining. Dies or die regions with large, gently sloping surfaces such as any male outer panel die and many female outer panel dies are excellent candidates.

For outer panel dies, machining time can be reduced up to 67% and bench time reduced significantly by using five axis machining.

More CAM programming time is required when programming in five axis versus three axis due to the inflexibility of the TCS five axis software. Given that the machining resource is capacity constrained and CAM programming is not, the additional programming effort is well worth the savings at the milling machine.

Several existing CAM packages such as CAMAX™ and NUFORM™ contain excellent five axis capabilities and will minimize the additional CAM programming effort.

A robust NC verification or simulation package is required to ensure the tool does not gouge the part. The existing verification operator in TCS does not detect all gouging generated by the TCS cutter path software.

Improvements required for the TCS five axis swarf software are in path definition, tool path generation, and path verification.

For five axis swarf cutting to be possible, the CAM programmers must communicate with the product designers. Surfaces must be designed with five axis swarf cutting in mind. This does not necessarily mean that the product shape must change.

Concave form cutters for machining outer radii could substantially reduce both machining time and bench time. These form cutters could trace a radius while maintaining a tool vector normal to the part surface. eliminating the need to flow cut outer radii.

Recommendations

Five axis simultaneous machining of dies be used for and limited to outer panels; both male and female forms.

The largest acceptable cutter with the smallest acceptable heel angle be selected to minimize both machine and bench times for five axis simultaneous machining.

Develop a TCS function to allow CAM programmers the flexibility to generate flow lines across multiple surfaces. This will aid both three and five axis programming.

Improve TCS function that permits a CAM programmer to maintain a constant cusp height.

A robust verification package be purchased to verify all five axis CAM programs.

CAM programmers communicate with product designers to build surfaces that are designed for programmability and manufacturability.

Experimental procedure

The procedure used in this experiment was as follows:

Select the dies to program.

Work with central engineering and several die plants, and a prototype shop to select an ensemble of dies for programming.

Review die prints and files with CAM programmer.

Develop a strategy for programming each die. Determine regions to cut five axis, tools to use, speeds and feeds, and desired cusp heights.

Program each die with three and five axis TCS.

Record programming time, time spent correcting and working around problems, machine processing time, and verification time. Also record file size (indication of machining time) and explanation of problems encountered.

Post process cutter paths on machining center.

The post processor will output estimated machining time.

Check each cutter path with VERICUT™ verification software.

Run cutter paths through VERICUT package to determine material removal, machining time, obvious gouging or material missed. Use design data to compare cut part to design part; check for gouging or material missed.

Evaluate results and recommend dies and die regions for five axis machining.

Generate a set of recommendations for die plants.

Dies selected for programming

The contour portion of a stamping die is represented by several thousand mathematically defined surfaces. When we researched dies to program, we strove to select an ensemble that contained all the different types of surface one would find on a stamping die. With this in mind, we selected two dies that contain many of the surfaces encountered in die making. Both dies were draw dies. Contour surfaces on draw dies are generally more complicated and more critical than other dies, since most of the forming occurs at the draw stage. We programmed the hood die in three axis first, then in five axis. We reversed the order for the second die to balance learning effects. The two dies selected were:

The male punch from a draw die for a hood outer panel.

> This type of die is traditionally considered an ideal candidate for simultaneous five axis machining. Any die that is principally composed of large gently sloping surfaces (hood, roof, deck lid, and door outers) falls into this category. Interference problems are generally avoided with male surfaces. See Figure C1.

The male punch from a draw die for a center pillar

> The center pillar contour contains a wide variety of surfaces to test. There are flat, steeply inclined surfaces (ideal for swarf cutting), slightly inclined flat surfaces (ideal for four axis machining), and gently sweeping surfaces (ideal for simultaneous five axis). Programming the male punch avoided problems with spindle to die interference. Advocates of five axis machining are divided on whether this is a good candidate for five axis

machining. The experimental results will provide quantitative data to settle this debate. See Figure C2.

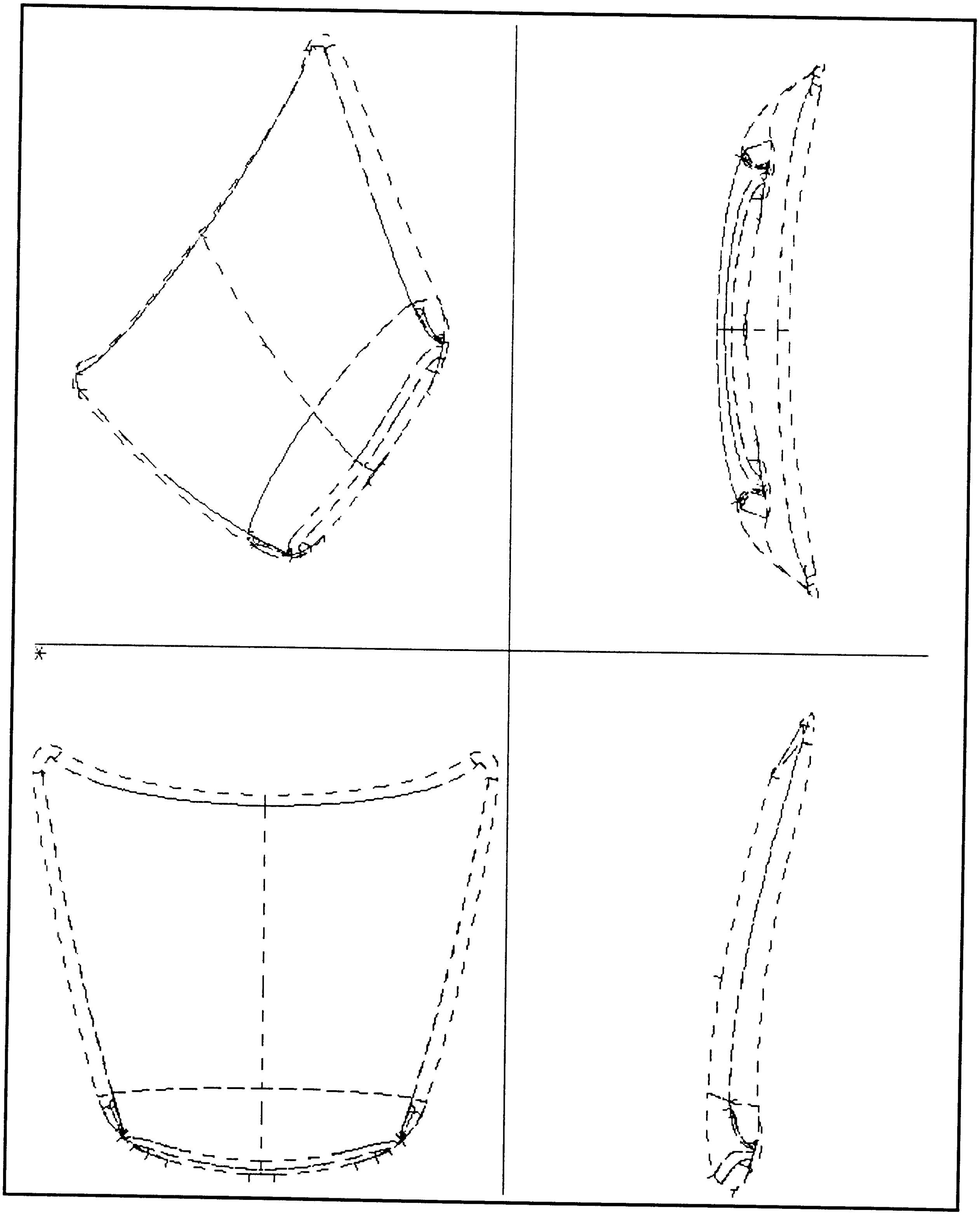


Figure C1. Hood die

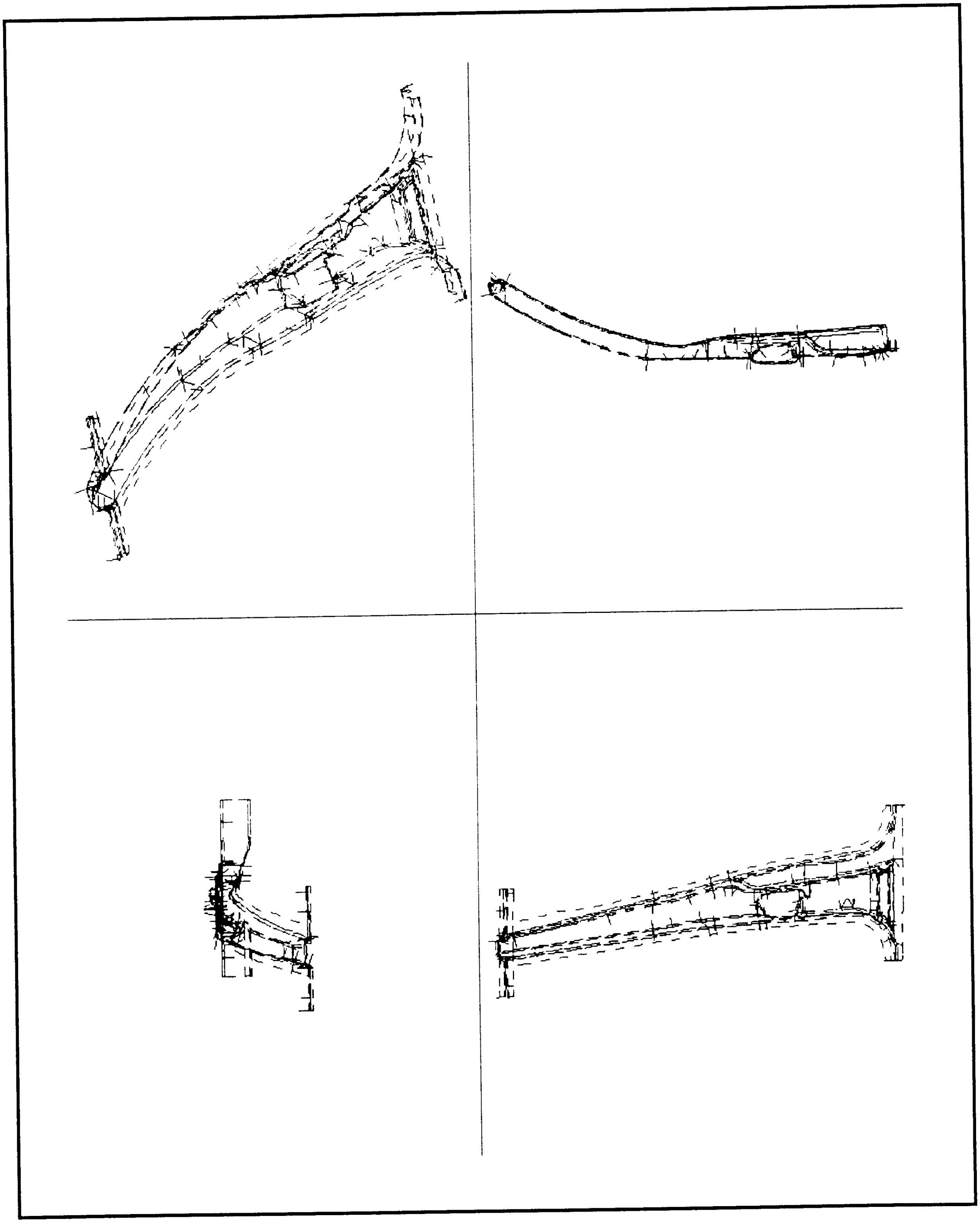


Figure C2. Center pillar die

Summary of Results

Hood die	**Three Axis**	**Five Axis**
Programming time	12.5 hours	24.5 hours
Verification time	negligible	3 hours
CPU processing time	1.2 hours	10.5 hours
File size	179,954 points	91,446 points
Machining time	14.9 hours	5.17 hours

Center pillar die	**Three Axis**	**Five Axis**
Programming time	15 hours	34 hours
Verification time	negligible	9 hours
CPU processing time	1.3 hours	1.7 hours
File size	64,244 points	33,612 points
Machining time	3.2 hours	2.4 hours

Hood die results and findings

The step over for both ball nose and radius end mill machining was based on a 0.001 inch cusp height and equivalent bench times. Feeds and speed were estimated from allowable cutting speed, effective tool diameter, and tool chip load. To reduce bench time and maintain the same machine time savings, a larger diameter cutter could be used.

The profile program traced the outside profile of the punch. This program was only generated once since it is a vertical cut.

Total programming time was 12.5 hours for three axis and 27.5 hours for five axis. The 12 hour difference is attributable to difficulties encountered with five axis TCS.

Programming flow cuts took 3.3 hours longer in three axis than five axis. For both three and five axes programs, offset lines had to be generated for each flow cut to define the tool path. These lines generally require editing to trim overlapping sections, join gaps, etc. The three axis program required several passes and hence several offset lines to define the horizontal radius; the five axis program required only one pass.

By far the largest discrepancy between three and five axis programming was the contour lace cut. It took 4.5 hours to program the lace cut in three axis and 18 hours in five axis. Four programs defined the five axis contour. The largest program defined the surface of the hood up to the horizontal radii. The remaining three programs cut the front, rear, and side horizontal radii. This strategy was selected to keep the cusp height low, avoid gouging the part surface, and avoid software problems caused with vertical cuts (see Discussion).

Seven hours of the difference between three and five axis lace programming is attributed to surface development which included generating tool lines. Tool lines are lines on the surface of the part that the tool is programmed to follow. Generating these lines was time consuming because the TCS does not contain an option to generate tool lines across multiple surfaces. With three axis programming, the entire part was section cut which will define tool lines over all surfaces. The TCS package is able to do this quite easily (see Figure C3).

Verification time for all three axis programs was negligible. Verification of the five axis cuts took 3 hours because TCS "gouge detect and update" function proved inadequate (see Discussion). TCS path verification software allows a programmer to develop a tool and run the tool over a generated tool path. All verification and correction is done manually. Given that there are thousands of points in each file, this process can be very time consuming.

The cutter path and original design file were run through CGTECH™'s VERICUT software to check for gouging. VERICUT did detect some gouging in the surface but was determined to be within a tolerance band of +/-0.005".

Estimated machining time for five axes was much less than for three axes. Total estimated machining time for three axis was 15 hours. Total estimated machining time for the five axis programs was 5.2 hours. A die plant can save approximately 10 hours of machining time when using five axis machining to build this die. The savings in machining time are a result of higher material removal rates and a larger step over with five axis.

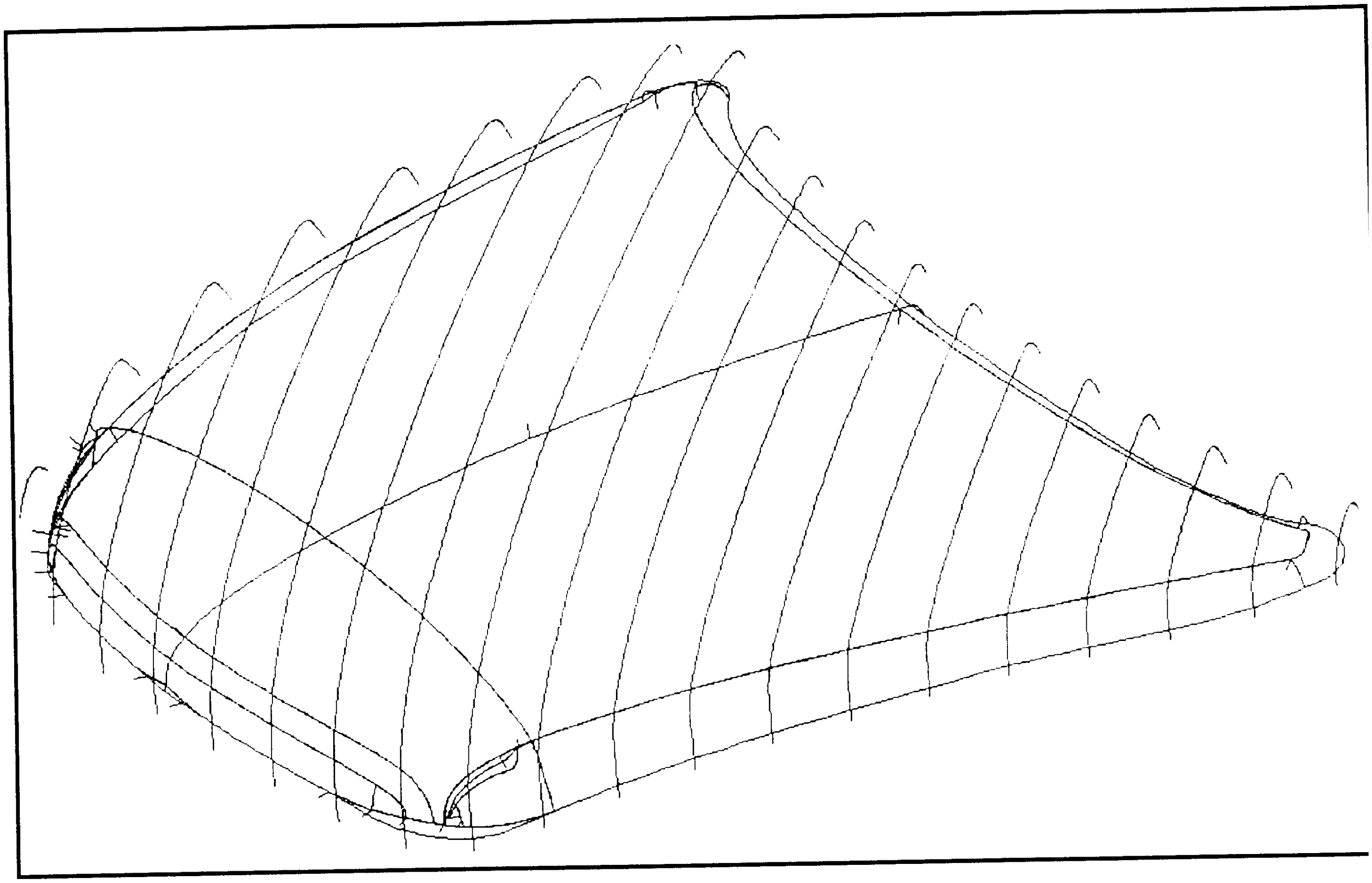


Figure C3. Definition of a section cut.

Center pillar results and findings

The step over for both ball nose and radius end mill machining was based on a 0.001 inch cusp height and equivalent bench times for contour cuts. Feeds and speed were estimated from allowable cutting speed, effective tool diameter, and tool chip load. Swarf cuts in five axis were intended to be to the exact part surface and would need no bench work.

The profile program traced the outside profile of the center pillar. This program was only generated once in three axis since it is a vertical cut.

Total programming time was 15 hours for three axis and 43 hours for five axis. The 28 hour difference is attributable to difficulties encountered with five axis TCS and the product design data.

All flow cuts were done in three axis. A flow cut will follow along a radius and machine away excess stock (see Figure C4). Programming flow cuts took approximately 8 hours for the entire die. As in the hood program, offset lines were generated to define the tool paths for flow cutting. In the three axis program, only inner radii (fillets) were flow cut; there was no need to flow cut outer radii. In the five axis program, both fillets and outer radii were flow cut since the contour was not machined around the horizontal radii. However, the five axis program left less material to remove at the fillets which meant fewer tool lines. The result was roughly the same programming time, machining time, and file size for both three and five axis flows.

Contouring (lace cutting and pocketing) took 3.5 hours to program in three axis; the equivalent work took 21 hours to program in five axis. The five axis programs involved swarf cutting, contour machining, and some three axis work. The vertical ribs were swarf cut, the flanges and top were contour machined, and the ends were machined in three axis using a ball nose. Several regions on the top section were cut in three axis since it proved too difficult to orient the tool in these regions without gouging the surface (see Figure C5).

The same strategy was employed to contour the center pillar top section and the hood die. That is, the top section was contour machined in five axis up to the horizontal radii. As with the hood die, this strategy was selected to avoid both gouging and software problems with vertical cuts (see Discussion). Unlike the hood die, the center pillar horizontal radii were flow cut in three axis. The center pillar radii are much smaller than the horizontal radii on the hood die and for that reason are much easier to program and cut in three axis. A 1/2" end mill was used to cut the flanges. A larger cutter could not be used as it would interfere with the ribs.

Most of the time difference between three and five axis contour programming is attributed to the number of programs required. To contour the center pillar required 8 programs using five axis and 3 programs using three axis (see Discussion).

Verification time for all three axis programs was negligible. Verification of the five axis cuts took 9 hours due to the problems encountered with the product data, inadequacy of TCS "gouge detect and update" function, and the absence of any gouge prevention or detection function in TCS five axis swarf cutting software. VERICUT was not run on the center pillar; we did not think the extra effort would result in any new information.

Total estimated machining time for three axis was 3.2 hours. Total estimated machining time for the five axis programs was 2.4 hours. Approximately 50 minutes of machining time could be saved when using five axis machining. The savings in machining time are due to higher material removal rates and a larger step over with five axis.

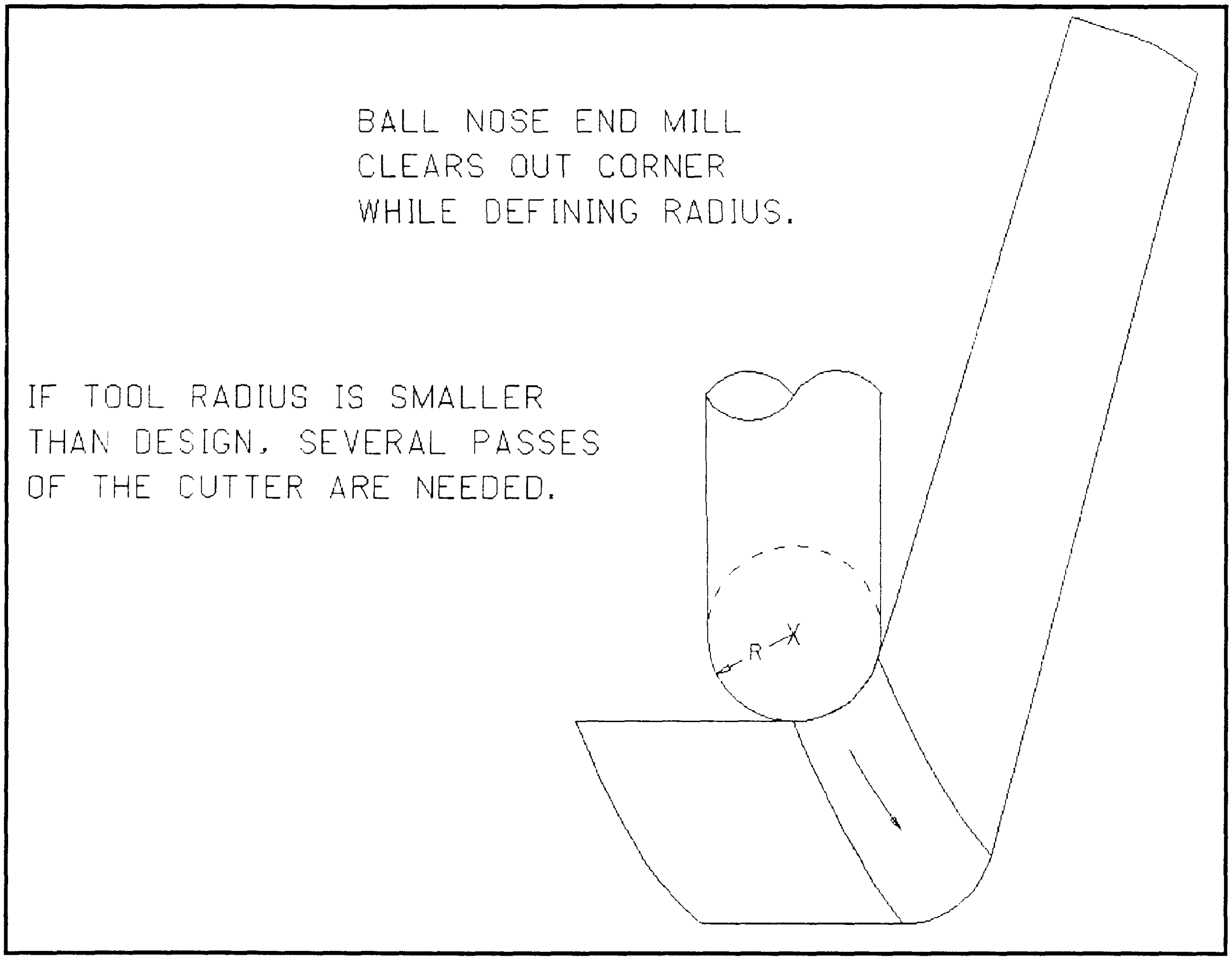


Figure C4. Flow cut.

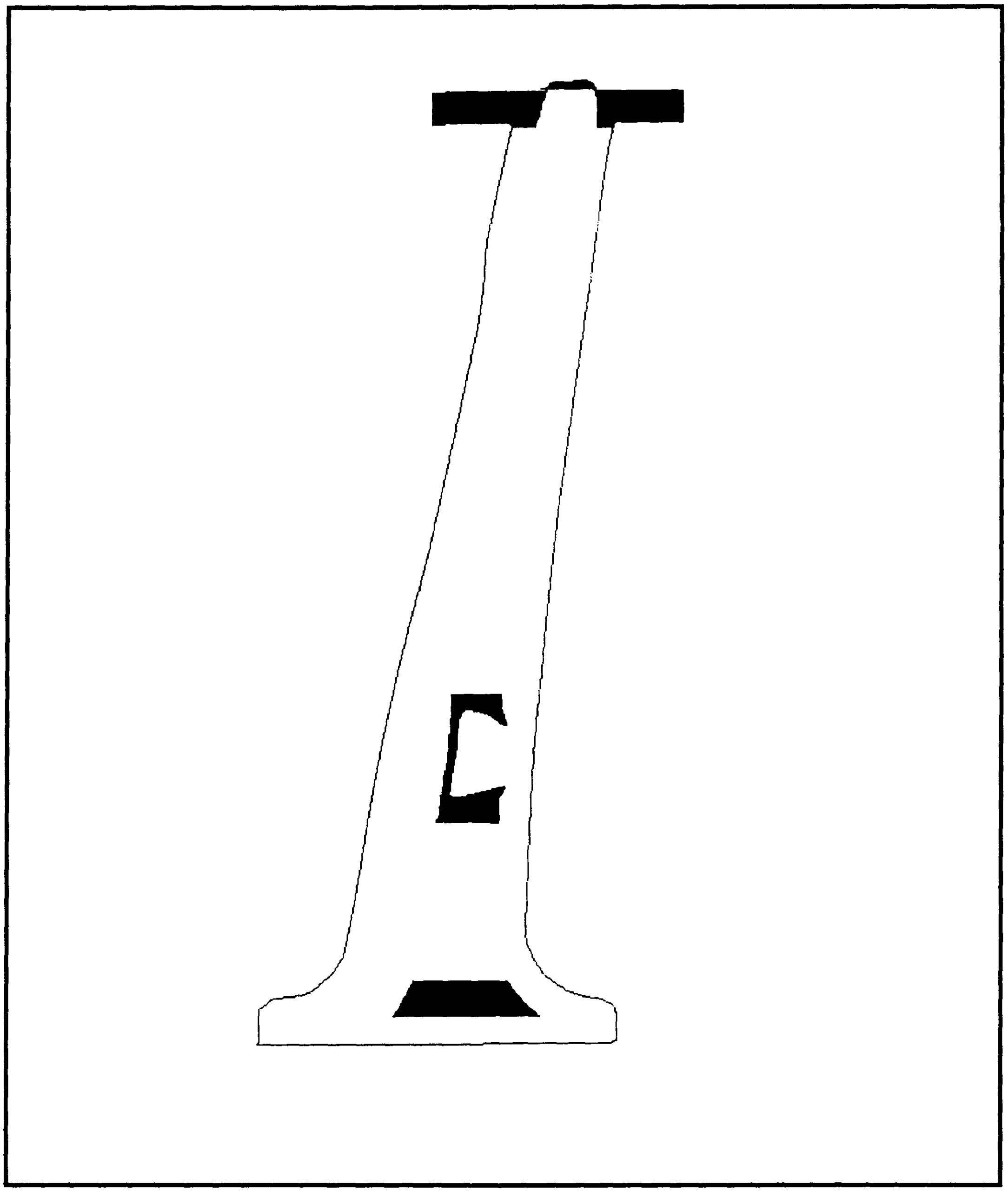


Figure C5. Center pillar regions not attempted in five axis.

Discussion

Difficulties encountered with the hood die

Programming the lace cut (contour) for the hood die took approximately 14 hours longer in five axis than three axis. The five axis contour was defined with four programs. The largest program defined the surface of the hood up to the horizontal radii. The remaining three programs were written to cut the front, rear, and side horizontal radii. If the entire surface (including radii) were cut in one program, the tool vector would start at -90 degrees, trace a path to near 0 degrees, cut across the hood surface at a diagonal, then rotate around the opposite radius to a 90 degrees tool vector. The contour was divided into four separate programs to:

1. Keep the cusp height low,
2. Avoid gouging the part surface, and
3. Avoid software problems caused with vertical cuts.

1. Cusp height consideration.

Wen the tool travels diagonally across the surface and then down the horizontal radius, the effective step-over increases substantially. This results in an unacceptably high cusp height. The three axis tool paths are so tightly spaced that the increase in step over results in a negligible increase in cusp height.

2. Gouge considerations.

Five axis TCS contains a "gouge detect and update" function that was designed to detect gouges and correct for them automatically. The package fails to detect gouges in many applications. The gouge correction software defines a tool as a series of discrete points at the tools periphery. It drags this representation over the generated tool path to check and optionally correct for gouging. Unfortunately the function does not adequately define an alternate tool path when a gouge is detected. It updates gouged points by backing the tool away from the part along the tool vector. If the gouge occurs away from the leading edge of the tool, the result is an undercut that may be quite substantial and lead to additional bench work. A better solution is to rotate the tool vector or shift the point on the surface, neither of which the function capable of doing. Were the "gouge detect and update" function able to accurately detect gouge points in a five axis move, it would not be able to adequately correct for the gouge.

3. Software deficiencies.

The final reason for this sectioned approach was to avoid software problems when the tool transitions from a horizontal to vertical cut. TCS will occasionally misunderstand surface data and cut on the wrong side of a vertical wall.

Difficulties encountered with the center pillar

Contouring (lace cutting and pocketing) took 3.5 hours to program in three axis; the equivalent work took 21 hours to program in five axis. Most of the time difference between the three and five axis contour programming is attributed to the number of programs required. To contour the center pillar required 8 programs using five axis and 3 programs using three axis.

In three axis one program can trace out the cross section (flange/rib/top/rib/flange) over the entire die (excluding the ends). In five axis, programming was divided into three flange programs and three programs to contour the top. This does not include swarf cutting which requires 2 more five axis programs.

Tool lines over the entire cross section (flange/rib/top/rib/flange) could not be programmed in five axis due to software problems. As with the hood die, TCS five axis package has difficulties dealing with horizontal to vertical transitions. Each five axis program required that tool lines be developed and tool paths be gouge checked. Again, the "gouge detect and update" feature proved inadequate.

The ends of the center pillar were programmed using conventional three axis machining since it was too difficult to program in five axis. Even if we were to program the ends with five axis, we would have to repeat much of the cutting with a three axis flow program to clean out the radii missed by the five axis cut.

The webs of the center pillar seemed ideal candidates for five axis swarf cutting. However, it took longer to program the swarf cuts for the center pillar webs than it did to contour program the entire part in three axis (6 hours versus 3.5 hours). This was the most disappointing result of the experiment. The swarf results were so poor because:

1. Generating tool lines with the TCS five axis swarf software is very time consuming,

2. The software contains no gouge checking function,
3. The product (math) data was not designed with five axis swarf cutting in mind.

1. Generating tool lines

Generating tool lines for a five axis swarf cut is time consuming with TCS. There are two general approaches for generating these tool lines;

- i. Use a tool with a smaller corner radius than that on the part and clean the missed stock later with a series of flow passes.
- ii. Use a tool with a corner radius equal to the part corner radius and machine the entire rib and flange down to net shape.

The former approach was used in this experiment due to problems with the product data (see below). To generate a tool line required that an augmented line be generated at the tangent point of the rib. This augmented line was offset normal to the rib an amount equal to half the tool diameter. The line contained vectors that were parallel to the rib surface. A tool path was defined to follow this augmented line with the tool parallel to the web surface (Figure C6).

The second approach would yield a cut that simultaneously swarf cuts the rib and cleans out the radius, eliminating the need to flow cut the radius. To generate this tool line required that an augmented line be defined that was offset normal from both the drive and limit surfaces. This line was then offset normal to the drive surface an amount equal to half the tool diameter. A tool path was defined to follow this line with the tool vector parallel to the drive surface (Figure C7).

2. Gouge checking

Unlike the TCS three and five axis contour software, the swarf cutting software contains no "gouge detect and update" function. Once the swarf paths were generated, the programmer must step through each point in the program to visually check for and manually correct gouging. Approximately 300 points had to be visually checked for the center pillar swarf programs.

3. Product data difficulties

The product data caused additional difficulties when trying to swarf cut the center pillar ribs. The ribs contain surfaces whose linear segments are not in the plane formed by the flange surface and flange surface normal (see Figure C8). That is, to successfully swarf cut requires that the rib surfaces have straight elements in the plane perpendicular to the flange surface.

This not being the case with the center pillar caused the tool to rock back and forth over the flange surface during the swarf cut which in turn caused the leading or trailing edge to gouge the flange surface. To prevent gouging the tool must be lifted off the flange surface. Lifting the tool off the flange surface leaves material uncut (see Figure C9) and requires subsequent flow cuts to clean out the radius. The result was that either of two approaches to swarf cutting mentioned above would require subsequent flow cuts. The first method was used since it required less programming effort and resulted in less gouging of the flange and web.

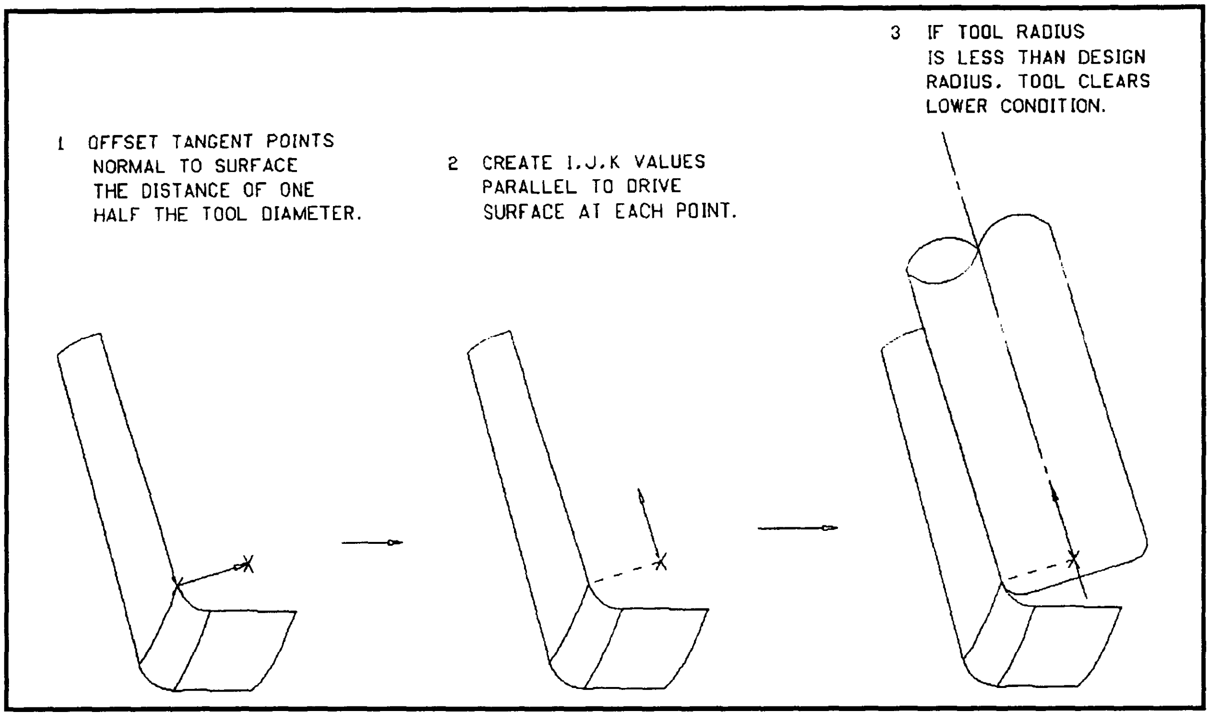


Figure C6. Generating tool lines for a swarf cut using method i.

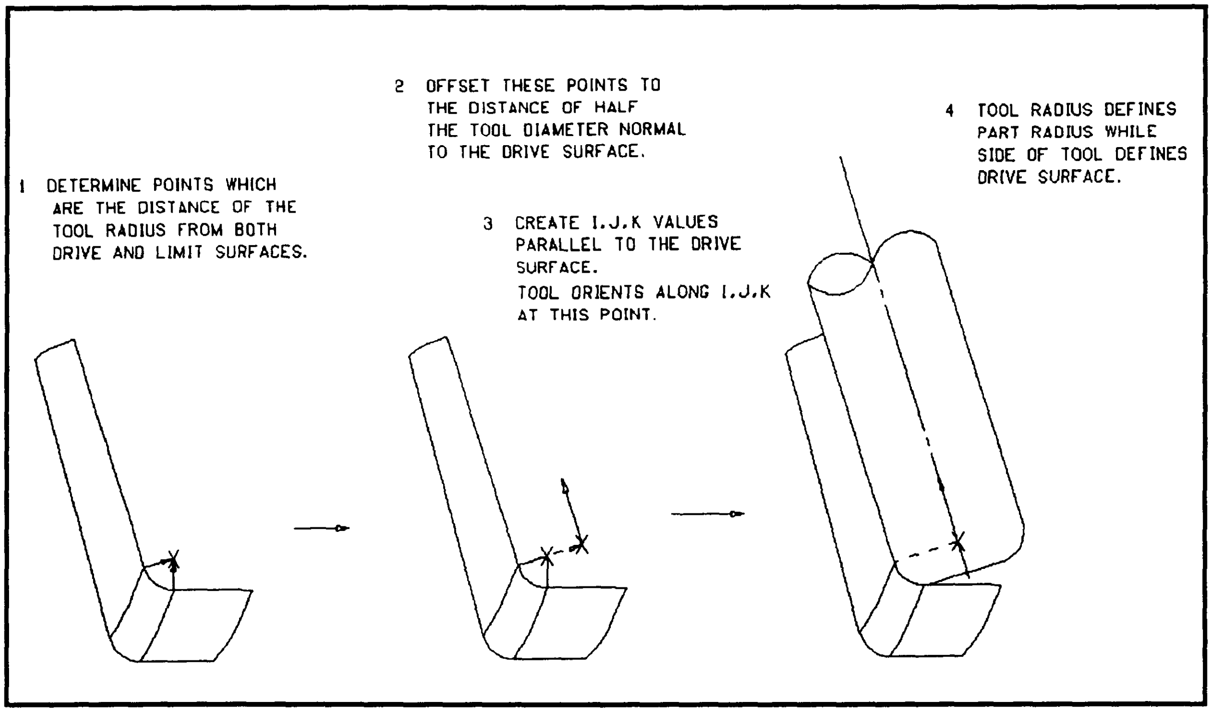


Figure C7. Generating tool lines for a swarf cut using method ii.

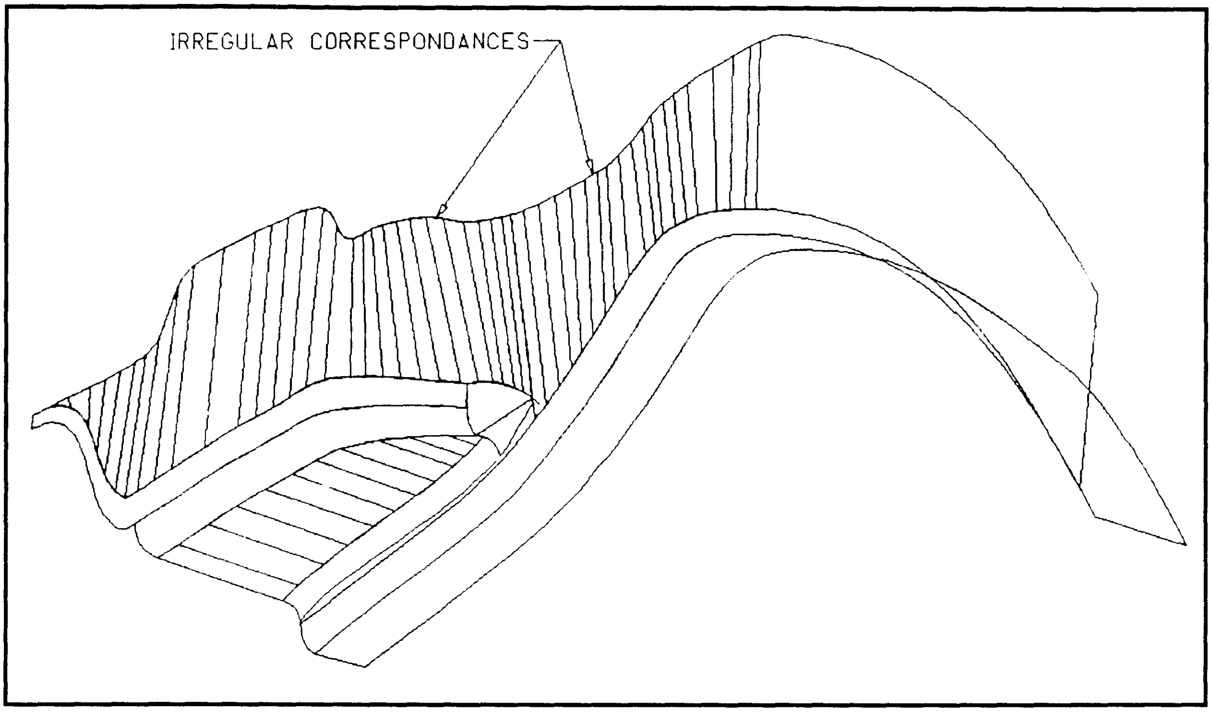


Figure C8. Product data difficulties.

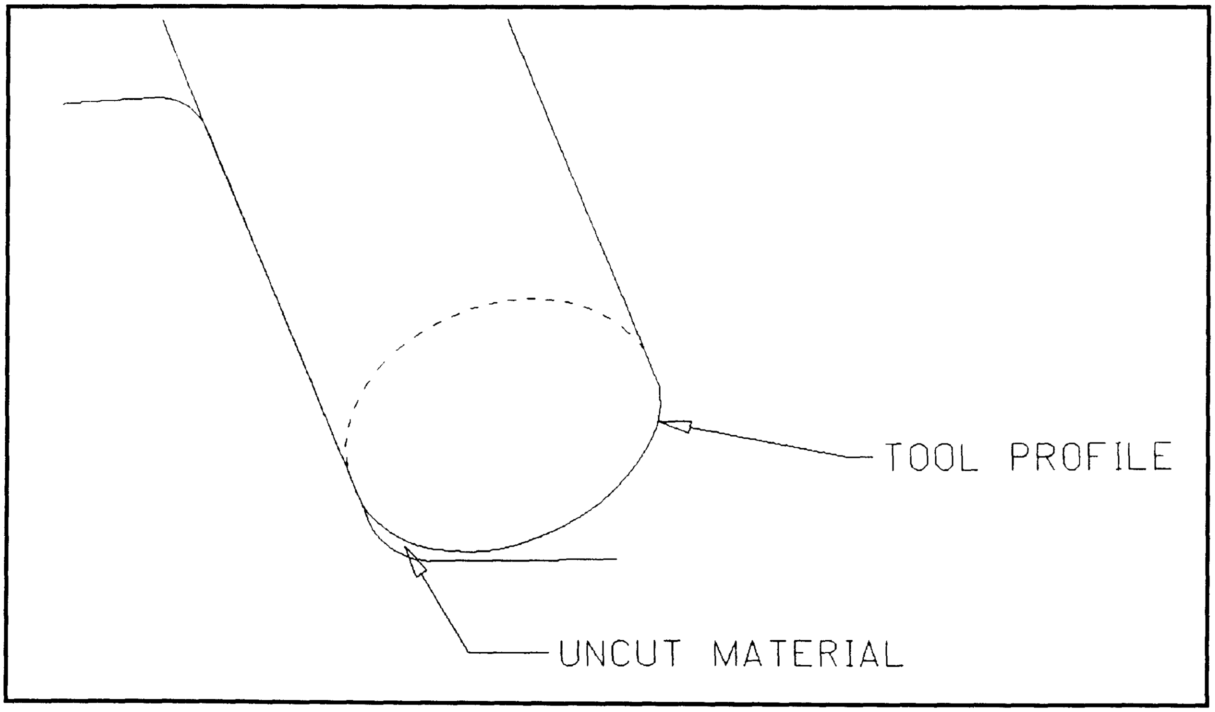


Figure C9. Uncut material during swarf cut.

Data

Hood outer, male draw.

Three Axis Programming

Feeds and Speeds:

Profile: 2" endmill, 50 ipm, 1000 rpm. 1/2" step-down

Flow: 1" ballnose, 3800 rpm, 61 ipm, 0.060" pick, box cut.

Contour: 1" ballnose, 3800 rpm, 61 ipm, 0.060" pick, lace cut.

Programming time:		*CPU Processing time* (in background):	
Profile	2hrs, 25min.	Profile	07min.
Flows (R&F)	5hrs, 40min.	Flows (R&F)	19min.
Lace	4hrs, 30min.	Lace	47min.
Working around bugs	20min.		

Size of cutter path file:		*Estimated machining time:*	
Rough profile	2127 Points.	Rough profile	19.83 minutes.
Finish profile	2131 Points.	Finish profile	19.57 minutes.
Flows	3141 Points.	Flows	12.06 minutes.
Clear detail	152 Points.	Clear detail	01.87 minutes.
Lace (Area 1)	86543 Pts.	Lace (Area 1)	406.4 minutes.
Lace (Area 2)	86433 Pts.	Lace (Area 2)	446.9 minutes.
Detail Flows	1554 Points.	Detail Flows	08.08 minutes.

Verification time (checking for gouging and tolerance checks): Negligible.

Hood outer, male draw.

Five Axis Programming

Feeds and Speeds:

Flow: 2" endmill with 1/8" corner radius, 1078 rpm, 65 ipm, 0.280 pick, 5 degree lead angle, lintol 0.002", box cut.

Contour: 2" endmill with 1/8" corner radius, 1078 rpm, 65 ipm, 0.280 pick, 5 degree lead angle, lintol 0.002", lace cut.

Programming time:		*CPU Processing time* (in background):	
Profile	02hrs, 25min.	Profile	07 min.
Flows (R&F)	02hrs, 30min.	Flows (R&F)	00 min.
Lace	17hrs, 50min.	Lace	10 hrs, 17 min.
Swarfs	01hrs, 40min.	Swarfs	00 min.
Working around bugs	45min.		

Size of cutter path file:

Rough profile	2127 Points.
Finish profile	2131 Points.
Lace	39151 Pts.
Front scans	7831 Points.
Outboard scans	12719 Pts.
Rear scans	16748 Pts.
Swarfs	147 Points.

Estimated machining time:

Rough profile	19.83 min.
Finish profile	19.57 min.
Flows	4.352 min.
Lace	177.3 min.
Front scans	19.58 min
Outboard scans	55.59 min.
Rear scans	31.36 min.
Swarfs	2.541 min.

Verification time

(gouging and tolerance checks):

Flows (R&F)	15min.
Lace	02hrs, 30min.
Swarfs	15min.

Center pillar, male draw.

Five Axis Programming

Feeds and Speeds:

Profile: 2" endmill, 50 ipm, 1000 rpm, 1/2" step-down

Swarf: 2" endmill with 1/8" corner radius, 1000 rpm, 50 ipm, 0.5" stepdown, lintol 0.002", box cut.

Contour (Lace): 1" endmill with 1/8" corner radius, 1078 rpm, 65 ipm, 0.280 pick, 5 degree lead angle, lintol 0.002".

Contour (Lace): 1/2" endmill with 1/16" corner radius, ??? rpm, ?? ipm, ??? pick, 5 degree lead angle, lintol 0.002".

Contour (Flow): 1/2" endmill with 1/16" corner radius, 1078 rpm, 65 ipm, 0.280 pick, 5 degree lead angle, lintol 0.002".

Flanges: 1/2" endmill with 1/16" corner radius, 3000 rpm, 200 ipm, 5 degree lead angle, lintol 0.002".

4 Axis: 3/8" endmill at 5 degree lead angle, 3000 rpm, 200 ipm, lintol 0.002".

3 Axis Lace: 3/8" ballnose, 2400 rpm, 150 ipm, 0.177 pick, lintol 0.002".

Programming time: Profile 01hr. Flows on radii 08hrs. Contour (w/flange) 11hrs, 30min. Swarfs 06hrs. 3 Axis 03hrs, 30min. Pocketing 30min. Toolkit 03hrs, 30min.	*CPU Processing time* (in background): Profile negligible Flows on radii 38min. Contour lace 20min. Flange flows 30min. Swarfs negligible. 3 Axis 12min. 4 Axis negligible.
Size of cutter path file: Rough profile 373 Points. Finish profile 375 Points. Flows on radii 11881 Pts. Swarfs 1412 Points. Contour lace 7704 Points. Flange contour 2332 Points. 3 Axis 8168 Points. 4 Axis 788 Points.	*Estimated machining time:* Rough profile 03.94 min. Finish profile 03.94 min. Swarfs 10.84 min. Contour lace 17.42 min. Flange flows 06.69 min. 3 Axis 16.61 min. 4 Axis 05.58 min. Flows on radii 82.54 min.

Verification time

(gouging and tolerance checks):

Contour (w/flange)07hrs, 30min.

Swarfs 01hrs, 30min.

Center pillar, male draw.

Three Axis Programming

Feeds and Speeds:

Flow: 1" or 3/8" Balinese, 3800 rpm, 61 ipm, 0.060" pick, box cut.

Contour: 1" Ball nose, 3800 rpm, 61 ipm, 0.060" pick, lace cut (perpendicular to length of center pillar).

Programming time:		*CPU Processing time* (in background):	
Profile	1 hr.	Profile	Negligible
Flows (1" & 3/8")	8hrs, 15min.	Flows (1")	08min.
Lace	2hrs, 20min.	Lace	36min.
Pocketing	1hr.	Flows (3/8")	35min.
Toolkit	2hrs, 30min.		

Size of cutter path file:		*Estimated machining time:*	
Rough profile	373 Points.	Rough profile	3.94 min.
Finish profile	375 Points.	Finish profile	3.95 min.
1" Flows	989 Points.	1" Flows	4.45 min.
3/8" Flows	11100 Pts.	3/8" Flows	71.88 min.
Lace (center section)	41538 Pts.	Lace (center section)	87.74 min.
Lace (ends)	9634 Points.	Lace (ends)	21.88 min.
Pocketing	608 Points.	Pocketing	3.46 min.

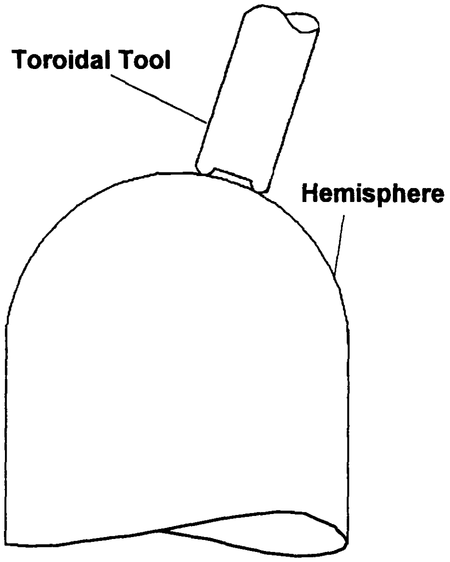
Verification time (gouging and tolerance checks): Negligible

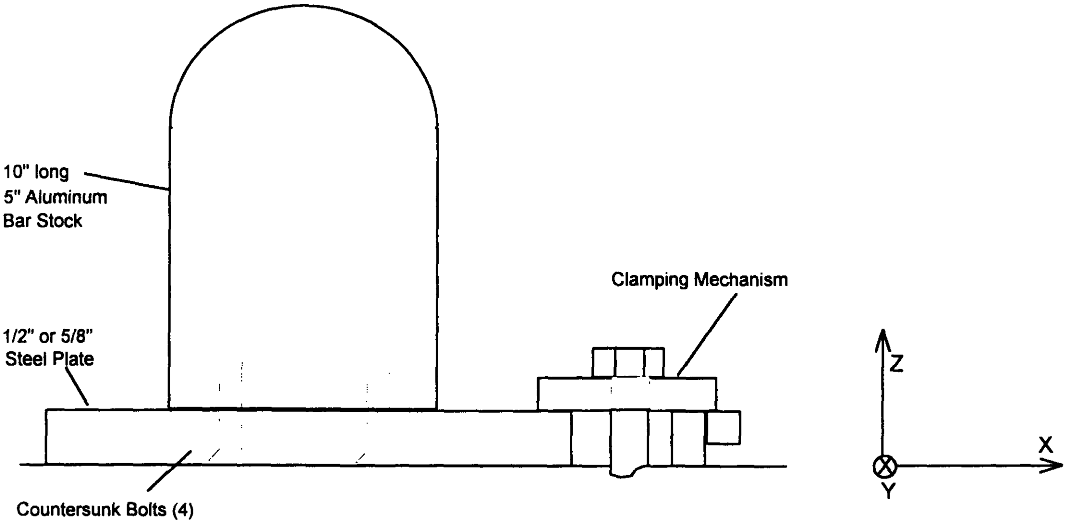
Appendix D. Five axis spindle unit verification

Identifying
5-Axis
Machining
Errors
with a
Nutating Spindle Unit

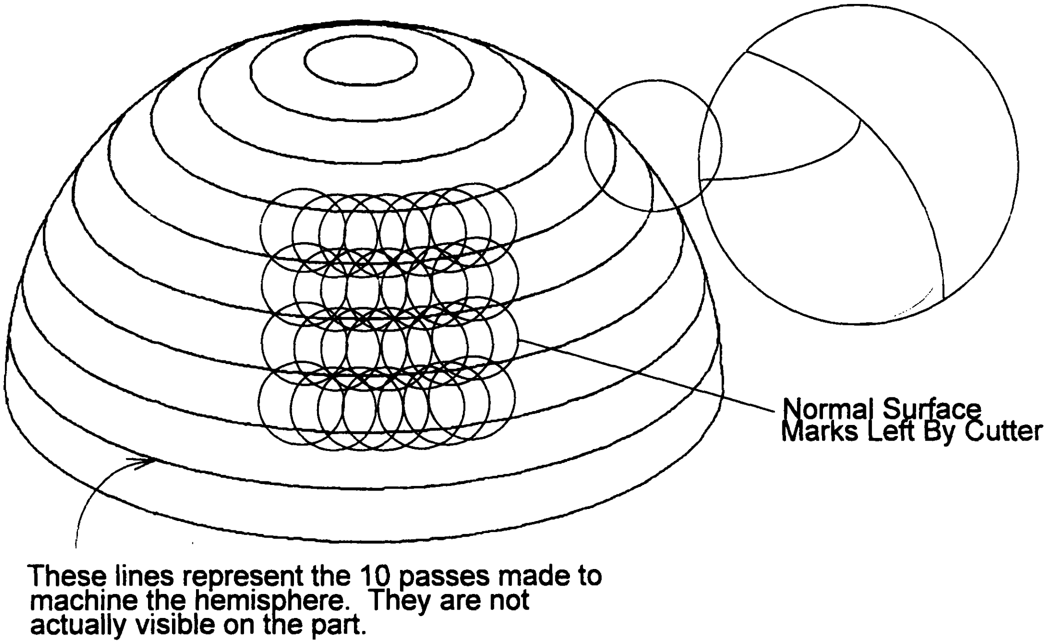
Using a cutting tool which is toroidal in shape, and approximately one inch in diameter, a hemisphere of five inches in diameter can be completely cut with 10 passes and no cusps.

The finished hemisphere would show faint circular marks of the tool at each tool path position. The circles would overlap each adjacent scan by about one third.





Set-up of Aluminum Bar Stock on Machining Center



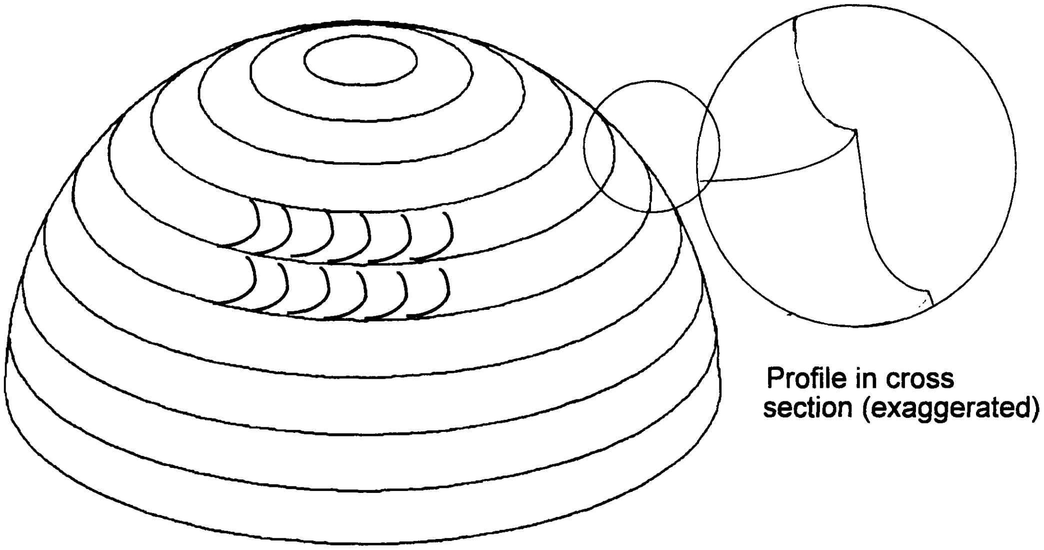
Slight errors in axis positioning would cause the surface patterns to be significantly different than shown above. These positioning errors could be detected by an examination of the different surface patterns shown on the hemisphere. The axis positioning errors would not be clearly evident if the hemisphere were cut by a flat end mill on tool centers, and only slightly evident if the hemisphere were cut with a tool at the leading edge.

B Axis Positioning Errors

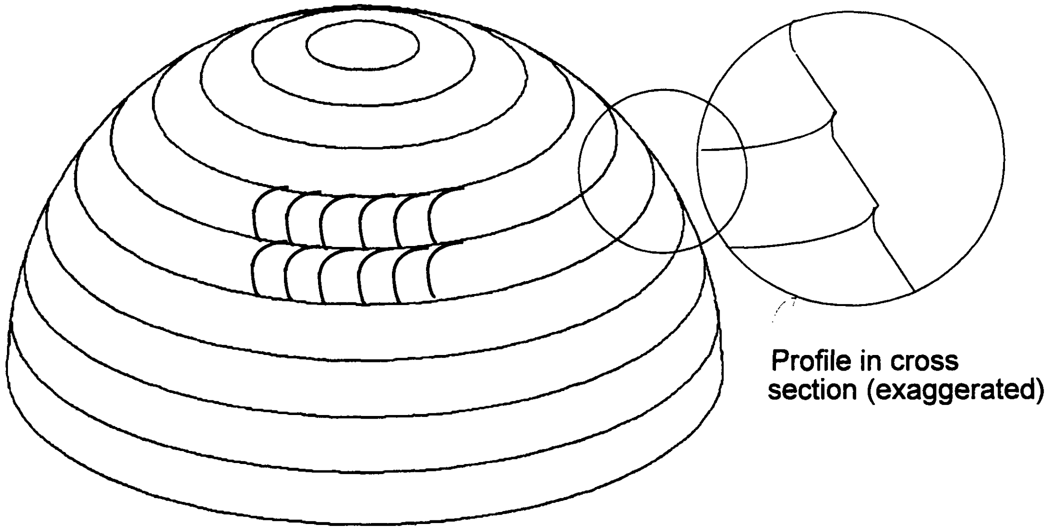
The following two figures show the effect on the hemisphere when the B axis is out of position; either over rotated, or under rotated.

This indicates a problem with the B axis (null) offset.

Over Rotation



Under Rotation

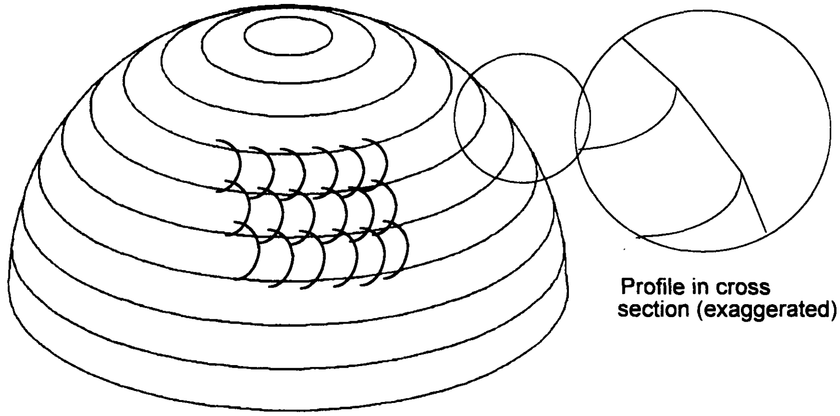


C Axis Positioning Error

The following two figures show the effect on the hemisphere where the C axis is out of position; either over rotated (+C direction), or under rotated (-C direction).

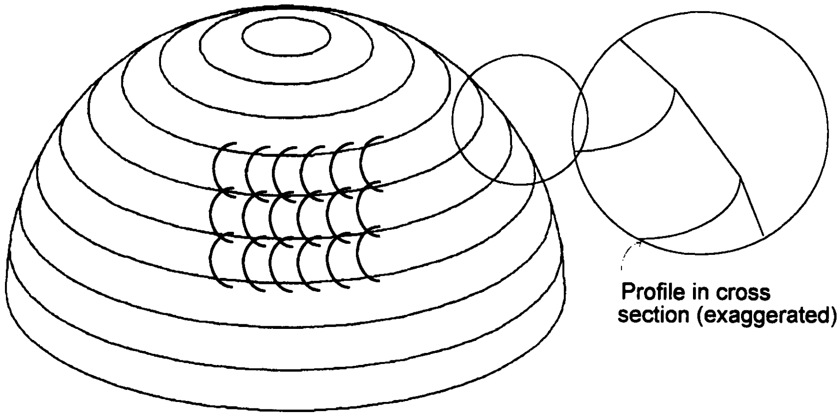
This indicates a problem with the C axis offset.

+ C Axis Error



Profile in cross section (exaggerated)

- C Axis Error

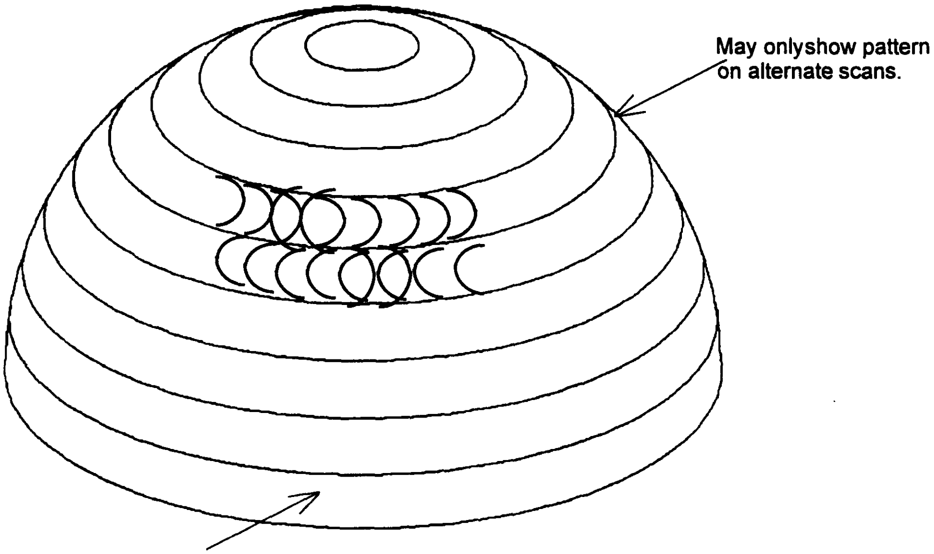


Profile in cross section (exaggerated)

C Axis Backlash

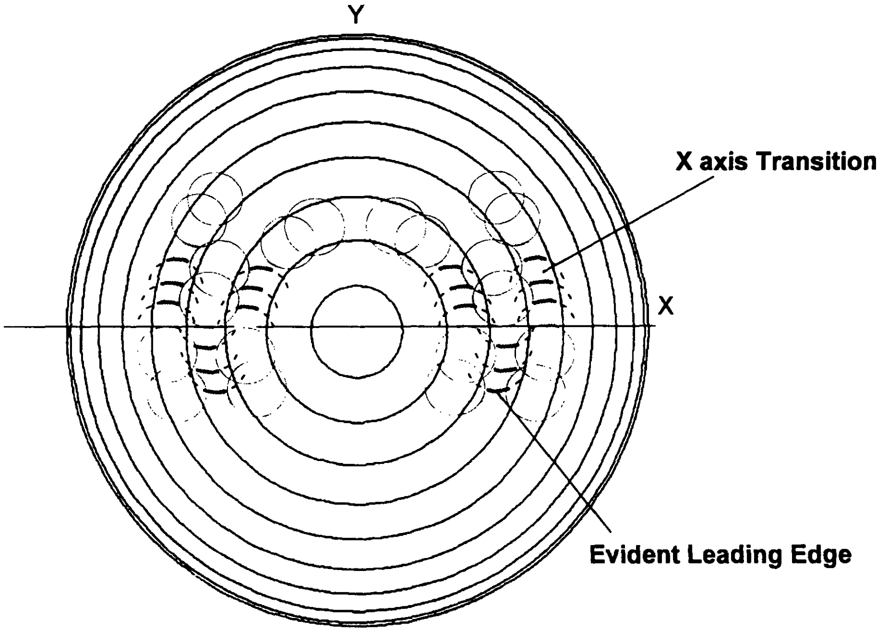
The following figure shows the effect on the hemisphere when the C axis contains backlash.

This may also indicate a problem with the C axis gain (a C axis gain problem will not show double gouging at the transition).

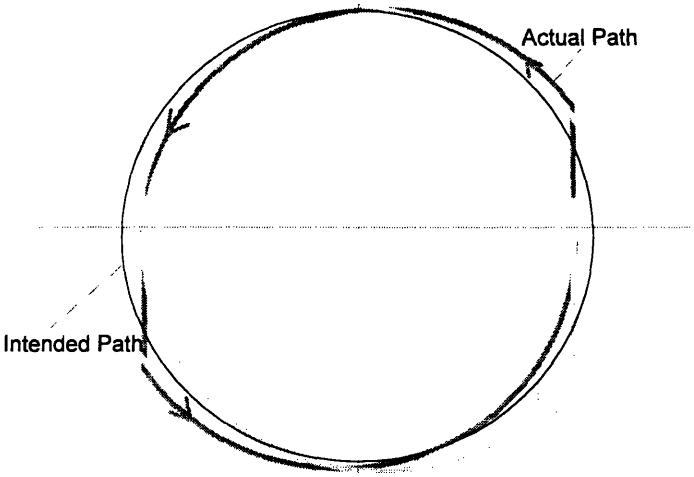


X Axis Backlash

The following figure shows the effect on the hemisphere when the X axis contains backlash. This error can be confirmed with a ball bar test.

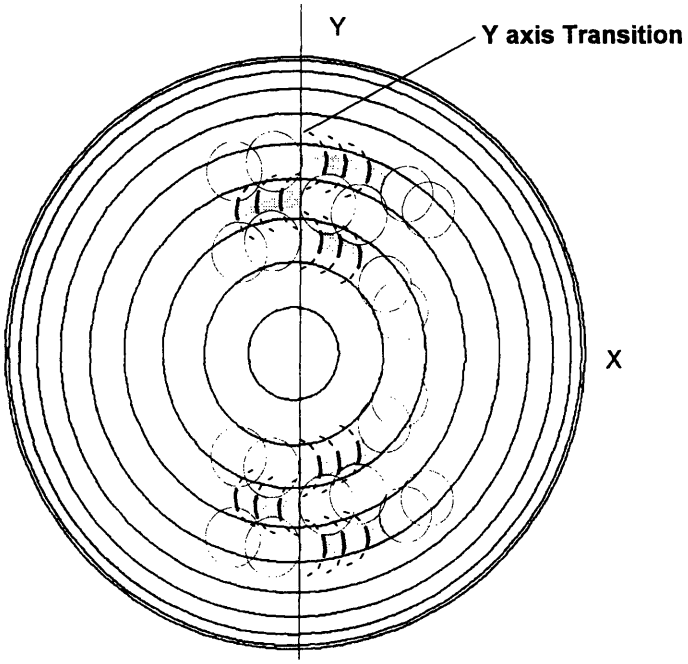


Toolpaths:

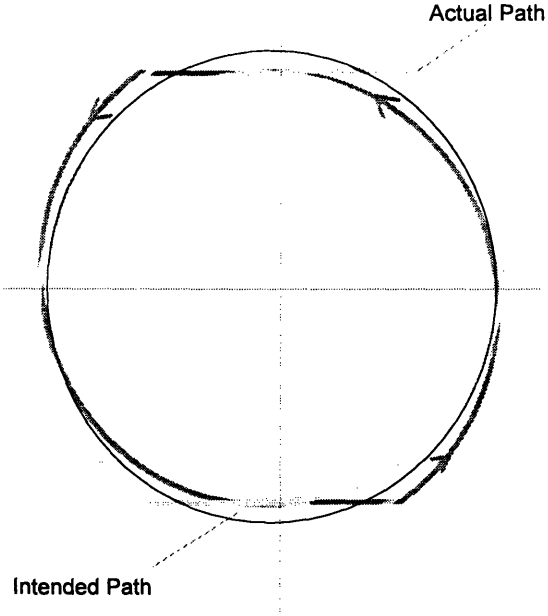


Y-Axis Backlash

The following figure shows the effect on the hemisphere when the Y axis contains backlash. This error can be confirmed with a ball bar test.

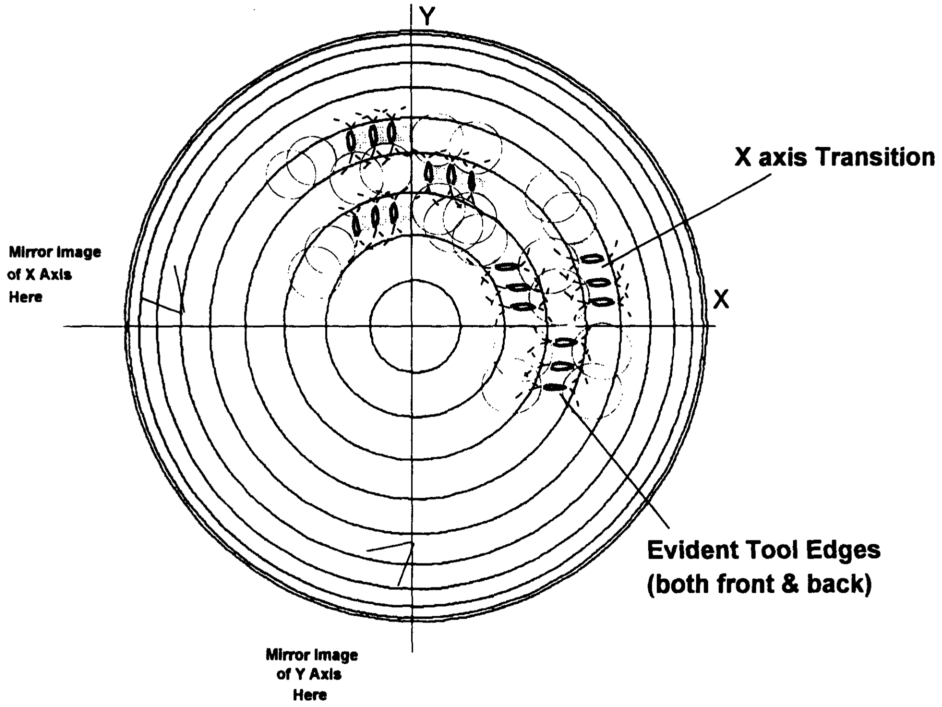


Toolpaths:



X & Y Servo Gains Not Matched

The following figure shows the effect on the hemisphere when there is an X and Y servo mismatch. This error can be confirmed with a ball bar test.



Toolpaths: