The Application of Parametric Software into the Undergraduate Computer-Aided Manufacturing Environment

by

Richard John Cournoyer

A Thesis

Submitted to the Faculty

of the

WORCESTER POLYTECHNIC INSTITUTE

in partial fulfillment of the requirements for the

Degree of Master of Science

in

Manufacturing Engineering

May 1999

Approved:

Mustapha S. Fofana, Advisor
Manufacturing Engineering

Shaukat Mirza,
Director Manufacturing Engineering Program

Mohammed N Noori,
Mechanical Engineering Department Head

Abstract

This thesis presents an in depth study of Pro/Engineer's manufacturing module and its application into the Computer-Aided Manufacturing (CAM) undergraduate education environment. Mechanical Engineering has a lot to gain by incorporating computers into the undergraduate curriculum in comparison to only the traditional classroom surroundings. Today, complex problems can be solved in mere seconds thanks to the power and speed of current computers. Likewise within today's manufacturing sector, numerical controlled (NC) machines are no longer programmed manually. In today's globally competitive manufacturing environment, integrated systems such as CAD/CAM help reduce the ever-shrinking time to market. This thesis contains the background as well as the curriculum material necessary to teach undergraduate students CAM using Pro/Engineer's manufacturing module. The curriculum material starts with the tutorials to teach and reinforce Pro/Engineer basic sketcher skills, which are necessary background information. Followed with in-depth click tutorials to teach the manufacturing module for 2 axes turning, and 3 axes hole drilling and milling. It also includes the necessary lab manuals that reinforce the class lecture material, an electronic manufacturing exam, and the students' evaluations from 2 terms when the CAM course (ME3820) was offered.

Acknowledgments

I would like to knowledge and express my appreciation to the following people and corporations who have supported and guided me in the completion of this thesis.

First I would like to thank my adviser Mustapha S. Fofana for numerous entities. He spotted what he thought was a diamond "in the rough" and sequentially provided me the encouragement and financial support to start this project. Additionally he has granted me the freedom to use my many years of manufacturing background combined with my recently acquired education to develop this thesis. Change is not always been easy thing to bring about.

Secondly I would like to thank Parametric Technology Corporation for the generous contribution of Pro/Engineer and the numerous modules that have been provided to Worcester Polytechnic Institute (WPI). Additionally I am thankful for them donating the software training for my adviser and me. This was imperative to learn the proper techniques of the software and essential to the overall success of the course.

Lastly and unequivocally as important, I want to thank my daughter Heather, who sat patiently through many of her precious weekends and never uttered "I'm bored" while watching her father read, write and research.

Table of Contents

List of Figures

Chapter 1 The Need for CAD/CAM in Education

Many companies are anxious to incorporate CAD/CAM methods into their design and manufacturing processes to improve productivity and to lower manufacturing costs. Because of the increase of CAD/CAM in today's industry, students should be introduced to these methods during their engineering education (Turcic, 1987). There is a need for quality trained manufacturing engineers in today's workplace. Quality apprenticeship trained machinists and craftsmen are becoming a rare commodity in today's work environment, and business owners are taking advantage of Computer Numerically Controlled (CNC) machines operated by mostly unskilled labor to fill their needs. These machines are often programmed by a qualified and knowledgeable manufacturing engineer to offset the lack of skilled labor. Therefore engineers with knowledge in CAD/CAM and solid modeling related concepts are in great demand because industry cannot find enough skilled people with the proper technical background or experience. Manufacturing still plays a vital role in today's economy and the opportunities for the individuals who are well prepared are nearly boundless. Using tools that are presently widely utilized in industry, will provide students at WPI an opportunity to properly prepare for this manufacturing challenge.   This thesis is an in-depth study of Pro/Engineer's manufacturing module and its application into the undergraduate course (ME 3820) Computer-Aided Manufacturing (CAM).

It is the objective of this thesis to provide the students an opportunity to design and manufacture part geometry using Pro/Engineer's manufacturing module and CNC machining. The remaining parts of this thesis are categorized as follows: in Chapter 2,

the role of computer software in the undergraduate environment is examined through, its history and needs. Chapter 3 investigates the background of Pro/Engineer (parametric software) and what makes it stand out from other CAM software on the market such as Catia, AutoCAD, and Unigraphics. The process of implementing Pro/Engineer's manufacturing module into WPI's Mechanical Engineering course (ME3820) is discussed in Chapter 4. It contains the early work from the first term when the class was taught, and the student and peer evaluation. Chapter 5 discusses the thought process and objective of the tutorials and laboratory manuals. The final class structure that resulted from nearly a year's worth of research and training is presented in Chapter 6. The actual education material (i.e. tutorials, lab manuals, quizzes, and exams) is contained in Appendices (A-E). Chapter 7 contains conclusions and future recommendations.

Chapter 2 Literature Review of CAM Software in the Undergraduate Curriculum

The world is an ever-changing environment, and there are times when we must adapt to changes or be left behind. The computer revolution is still alive and strong, and forever making its impact on the academic world (Tsatsakis, 1994). Some examples of software application into the educational environment are, circuit analysis, assembly languages, multimedia interfaces and finite element analysis.

New powerful Computer-Aided Design (CAD) tools are available and allow students to design products and determine forces, stresses, and motion using computers. Students can also manufacture these designs in a plastic compound directly from the computer with rapid prototyping machines. This new CAD software can also be used to teach basic engineering technology such as strengths of materials and mechanics. Thus a curriculum can be envisioned where undergraduates learn how to create solid models within the CAD environment. Throughout the remainder of their education, advanced analysis tools can then be used to teach and reinforce specific course material including design, strength of materials, and mechanics (Cole, 1998).

Computer-Aided Manufacturing (CAM), the name itself implies the use of computers in the manufacturing environment. Today, it would be difficult to locate a product that was manufactured without the use of a computer-controlled machine. Nowadays computers are fast, cheap and are commonly used to operate, monitor and control, manufacturing machines around the world. The days of programming computer-controlled machines manually has gone in the same direction as our earlier desktop computers that operated on DOS and now operate through the use of a graphical user interface, (GUI) such as Windows 95. There is a variety of parametric software that can

be used to graphically program CNC machines, and WPI owns the license to Pro/Engineer and most of its modules. Manufacturing is one of the more powerful modules. This module for programming CNC machines is available throughout the Mechanical Engineering (ME) department to all of its students and professors. It was seldom used because of the lack of training, and its high-level of difficulty to operate without proper training.

Like other disciplines, ME has a lot to gain by incorporating computers into the curriculum in comparison to only the traditional classroom environment. Complex problems can be solved in mere seconds thanks to the power and speed of today's computers. Students can now tackle complete sets of complex problems and quickly solve them compared to only using a calculator. The advancement of computers is an important step in the education of today's students and this should not be overlooked. An example of this would be the nonlinear finite element analysis of a complex shaped part for junior and senior level projects, and this part could be analyzed in a fraction of time compared to hand calculations. This advancement of computer technology for ME is similar to the introduction of computers in Electrical Engineering (EE). The EE students design and analyze complex digital circuits nearly instantaneously rather than the tedious and sometimes impossible task of attempting to solve them using traditional long hand methods. Likewise, ME can analyze a complex three-dimensional part quickly and fairly simply with the use of a computer.

A number of universities and colleges have adopted computers in undergraduate education. The University of Notre Dame has successfully used computers in the Civil

Engineering program since 1986 (Hamill, 1986). Some of the benefits they cited were as follows:

- Present data and knowledge in a clearer and more structured form.

- Enable students to work at their own pace.

- Helps develop expertise in computer usage.

- Introduce students to the type of software packages used professionally.

- Make education more interesting and more pleasurable.

Given these benefits, there was little doubt whether computer aided learning (CAL) should be used extensively throughout the engineering curriculum. University of Wisconsin (Turcic, 1987) has used a particular software package called Microcomputer Aides Design Analysis and Manufacturing software (MADAM) successfully for years to teach their undergraduate CAD/CAM. Students quickly mastered the software and then applied powerful capabilities to solve engineering design problems.

2.1 The Application of CAD/CAM at Western Washington University

Western Washington University's manufacturing engineering technology program (Oberoi, 1995) teaches a modified CAM class called Numerical Control Operations. Students are exposed to a real shop-floor environment, containing industrial size CNC equipment, cutting tools, and commercial CAM software packages. A variety of manufacturing techniques are taught to the students. They are, CNC programming, fixturing, selection and application of cutting tools, speeds and feeds, and machine optimization for large and small and quantity batches.

The laboratory contains five CNC machines and they are,

- Bridgeport Series 1, 2 1/2 Axes

- DYNA 2400 Vertical Milling machine, 3 Axes (10,000 RPM spindle)

- HAAS VF-2 Vertical Milling Center, 4 Axes (20 tool changer)

- MAZAK SQT -10M, 3 Axes turned-mill center (12 tool turret)

- Wire EDM, 2 Axes.

At Western Washington University, cutting tools throughout the laboratory are primarily indexable insert type, using a variety of geometries and insert materials ranging from standard carbide to ceramic, including diamond coated. The majority of the tooling holders and inserts used in the laboratory are on loan from various tooling companies for research. The manufacturing computer laboratory contains 13 terminals running AutoCAD for the engineering graphics, and three different CAM software packages which include the following:

- EZCAM

- MasterCAM

- AUTOSURF/AUTOMILL

The students work with a variety of materials including aircraft grade aluminum, stainless steel, alloy steel, copper alloys, and composite material. The Boeing Corporation provides most of the material used in the laboratory. Students are encouraged to use a variety of materials on their projects to familiarize themselves with some of the material unique properties.

The theoretical section of the course teaches many concepts of CNC manufacturing from fundamental equipment to final production runs. Some of the topics covered are fundamental of manufacturing and concepts of CNC equipment. Basic

construction of CNC machines, including, spindle construction according to international standards, and drive accessory devices. Industrial tours of the Boeing Corporation's machining facility are also conducted so that student can get a better view of modern industrial applications.

The selection of tooling and fixturing is a critical step in CNC machining, and therefore tooling applications and fixturing are taught in detail. Emphasis on fixturing for the CNC application is taught with concentrations on reducing setup time, and the automation of work holding devices. Students design their own fixtures, and are evaluated on the basis of setup time, accuracy of location, clamping forces and ergonomics. Students are encouraged to work with local industry in redesigning their fixtures to improve the company's productivity.

Process planning is also taught to students. Prior to any actual machining, each project is required to provide the speeds and feed calculations, horsepower requirements, and a detailed process sheet for each part within the project. Because of the number of projects versus machines, each student is also required to analyze part cost, required setup time and time scheduling for each production run.

Several programming techniques are taught to the students. Starting with basic G code or NC code, and then introducing them to subroutines, loops, macros, and parametric programs. Once the basic programming is understood, the students are then taught several CAM software packages. Each part's cutter path is checked using program verification or virtual simulated machining within the CAM software. The process of checking the cutter's path by simulated machining helps to reduce setup time and in addition, helps to eliminate any possible machine crashes.

Part production is emphasized to the students, rater than single piece manufacturing. After the first piece is machined, the part is then thoroughly inspected using a coordinate-measuring machine (CMM). Once accepted, the students are then required to machine their workpiece using automatic modes. The complete process is then checked for optimization by monitoring overall production time and per piece spindle run time. The students are then required to increase productivity by reviewing unnecessary and inefficient cutter paths. Additionally, tool wear and life is also monitored during the part optimization process.

This comprehensive manufacturing class also encourages students to look outside their normal classroom environment (i.e. local industry) for the class project. Having an outside source not only helps to enhance the student's practical knowledge of industry, but simultaneously helps to offset the laboratory costs by obtaining a sponsor. Western Washington University is also a participant in the formula SAE racecar, and their class assists in the manufacturing of some of the car's components. The author concludes the paper by stating that the intent of the course is to expose the students to actual industrial conditions. These conditions contain commercial machines, software, cutting tools, and advanced materials.

2.2 The Application of CAD/CAM at Christian Brothers University

Christian Brothers University (CBU) in Memphis, Tennessee offers an intermediate manufacturing course (ME445), where Pro/Engineer's manufacturing module is used to generate NC code during the manufacturing portion of the class (Beard, 1996). Successful manufacturing companies rely on modern technology to automate

8

their production process, and CBU feels it is crucial that students are also exposed to this process. This curriculum was created with the help of a National Science Foundation grant, which funded an Integrated Laboratory for Manufacturing Education (ILME). This laboratory includes a CNC vertical machining center, and seven SGI workstations running Pro/Engineer CAD/CAM software.

The semester where ME435 is taught is divided into three portions. The first phase is devoted to familiarizing the students with a basic understanding of Pro/Engineer's major modules, and proceeds with concentrating on the sketcher (CAD) module. The second phase of the course concentrates on the manufacturing module (CAM). Before the CAM software is explored, students are given a basic but solid understanding of NC code that enables then to manually write a G code program. This is necessary information when editing a program that is generated through the post processor of Pro/Engineer's manufacturing module. The last portion of the course concentrates on the thermal and structural analysis properties of Pro/Engineer's Mechanica module. Students are exposed to advanced techniques of integrated analysis. A final project is assigned to the students where all three phases of concentration (CAD, CAM, and analysis) are applied to a part. The syllabus for this course at CBU is as follows:

1. Basic to the computer aided design

    a) Basic Part creation, datum planes

    b) Slots, rounds, chamfer, etc.

    c) Blends, patterns, etc.

    d) Assemblies

e) Relations, parent/child dependencies, family tables

f) Drawings and other graphical output

2. Interface to numerical-controlled machining

    a) Part specification and design

    b) Manufacturing model

    c) CNC code practice

    d) CNC code development

    e) Machine shop

3. Structural Analysis

    a) Part development

    b) Distress and defamation analysis

    c) Optimization and sensitivity studies

4. Thermal Analysis

    a) Part development

    b) Temperature and heat blocks analysis

    c) Optimization and sensitivity studies

5. Field trip

6. In-class rapport

7. Final design project

The author of this paper concludes that this course is highly successful in meeting the educational goals set by the ILME. Students gain valuable manufacturing knowledge in integrated manufacturing, commercial CAD/CAM software (Pro/Engineer), design, and development within a modern manufacturing environment.

Clearly, students can benefit by introducing parametric software into the CAM manufacturing program curriculum. It will enhance their practical manufacturing knowledge that is always beneficial in the competitive manufacturing job market. Likewise, the manufacturing program at WPI can benefit from modeling its CAM course in a similar approach to other successful programs, in particular, Western Washington University.

Chapter 3 Computer Software in Undergraduate Classrooms

3.1 Computer Assisted Learning

There are many terms used to describe the technique of learning a subject with the assistance of a computer, Computer Assisted Learning (CAL), Computer-Aided Engineering (CAE), Computer-Aided Instruction (CAI), and Computer-Assisted Curriculum (CAC). Once CAD was called Computer Aided Drafting, today its Computer Aided Design, regardless of what it is named or called, the meaning is the same. Learning with the assistance of a computer. The computer and specialized software is used to teach, train, assist or any combination of all three. The idea of education with the assistance of a computer is not a new approach, it was introduced in the late '50s. Since then it has come through many stages of success and failure and has faced criticism as well as praise. Computer-aided instruction is no longer a novelty, and can be found across the continent at almost any learning institution (Solveig, 1985). It is no longer considered experimental, but moreover a necessity in today's demanding and competitive environment (Tsatsakis, 1994).

Computers are used in education primarily for three reasons, to learn software, to enhance theoretical course material, or a combination of the two. This thesis is an attempt to combine CAL into CAM. It will embark on the process of educating undergraduate students with the theoretical knowledge of CAM, and in addition to providing the students with a robust knowledge of Parametric Technology's Pro/Engineer's manufacturing module. Once the students have acquired some basic skills

and are knowledgeable of this commercial software, they can expand them in their remaining senior level courses where Pro/Engineer is taught as well.

It is now becoming a common practice for corporations to expect their prospective employees to possess some knowledge of software standard to that industry. However commercially available CAD/CAM software programs are written for production purposes and not for education, thus they are very complicated, and tedious to learn. Presently the sales of Pro/Engineer software is outperforming all the competitors, such as Unigraphics, and IBM's Cascade and is a number one selling high end CAD/CAM software according to Daratech's *Industry Update* (Daratech, 1998) and is well accepted as industry standard.

However there is another side of teaching students Pro/Engineer. This thesis-designed class together with its laboratory sections can offer students some practical hands-on experience, by using commercial software that is in demand in today's manufacturing environment. A common complaint about recent engineering undergraduates is that they lack practical experience (Furman, 1995). Worcester Polytechnic Institute's laboratories will continue to grow more dependent on CNC machines, and programming complex surfaces into these machines manually with NC Code is a difficult and tedious programming process. When you couple this with WPI's short seven-week terms, it becomes nearly impossible and highly impractical for students to grasp a complete understanding of this language, in additional to absorbing the theoretical information of CAM. Today it is rare in industry to program CNC machines manually. Software such as Pro/Engineer, and MasterCAM that is used to programs CNC machines have gone in the same direction as desktop computer operating systems,

meaning they have turned to a graphical user interfaces (GUI). These new programs, such as Pro/Engineer, allow parts to be programmed with a point-and-click method, similar to operating some of today's software.

Teaching students to program CNC machines with graphical based programs such as Pro/Engineer better prepares them for modern work environment. Product development and manufacturing have undergone some radical changes in recent years. This is due primarily to the advancement of solid modeling software and rapid prototyping, both that has drastically cut the time to market for a new product. Corporations are now expecting their employees to possess these computer skills before they can be hired (Palvia, 1996).

With the onset of cheap and powerful computers, the transfer of international graphics exchange specification (IGES) data from solid and wire frame models into graphical based CNC programming, has made manufacturing software an extremely popular item in today's work environment. It is no longer necessary to toil away for hours to reestablish a part's geometry once it has been drawn in CAD. An added bonus to the graphical based manufacturing software available today, is the ease in which complex surfaces can be quickly programmed for multiple axes CNC machines with minimal programming knowledge. There are only a few complete manufacturing software packages available today. The term complete refers to the software's ability to model, analyzes, and manufacture a part all within one central piece of software. Packages such as MasterCAM, EZ FeatureMill, EZCAM, SmartCAM, are separate standalone modules, and usually need to import the part's surfaces to be manufactured from another CAD file, or to recreate the part's surfaces before it can write the NC code. A common error in the

file transfer process is that some critical surfaces or features can be lost during the data transfer. The seamless transition between the solid modeling CAD file, part analysis, and the manufacturing within Pro/Engineer is a significant feature that should not be overlooked. It is a major reason for its popularity in industry.

Unfortunately, there appears to be some unwritten rule about the complexity of software and the inverse of its ease-of-use. Meaning that powerful and complex software is always difficult to operate. Pro/Engineer is an extremely powerful and complex package of software and unfortunately it is not user friendly, and thus it is nearly impossible to use correctly without proper training. This training is difficult and often extremely expensive to acquire. Before a new user can attempt to use any particular manufacturing module (mill, turn, drill, etc.), a good understanding of Pro/Engineer's basic sketcher module is essential. If this knowledge is not present, training must first begin with Pro/Engineer's sketcher.

3.2 Computers in Mechanical Engineering

In most universities, the course curriculum is overloaded (Bjorke, 1992). They are in a constant struggle with new topics that are becoming increasingly important from an industrial point of view, while still trying to maintain an academic balance. If a new course is created in the curriculum, normally something has to be removed to make room for this new class, and this generates conflict in maintaining balance between fundamental material and more recently applied subject matter.

Mechanical Engineering in general, has a lot to gain by incorporating computers into the traditional classroom environment (Furman, 1996). Today, there are times when

the standard technique of black board lecturing is inadequate for demonstrating particular

subjects. Examples of these are dynamics, vibrations, advance fluid dynamics and CAM.

Unfortunately the most prevalent approach to teaching Mechanical Engineering is still

the traditional black board lecture. Creating a new course where computers will be used

to enhance the learning process necessitates rethinking of the learning steps. It would be

less demanding for a teacher to present the same material in the same lecture mode

without change, but this does not allow the student to be given the opportunity of

advancing technology. This raises the point that maybe the teacher's role as well as the

instructional format may need to redefine when integrating Computer-Aided material into

the curriculum. Traditional methods for comprehending lecture materials has always

been by conventional methods: a verbal expression of the underlying theory, assisted by

analytical solutions of simple to understand examples, followed by laboratory

experiments to illustrate real world situations. Proven method of teaching using

traditional ways should not be discarded to communicate more effective through the use

of computers in an instructional mode, but the methods may need to be modified

(Grosman, 1985).

3.3 The Aspect of Parametric Software

Parametric Technology Corporation (PTC) is the CAD/CAM/CAE industry's

leading supplier of software tools used to automate the mechanical development of a

product from its conceptual design through production. Worldwide, more than 22,000

companies employ Pro/Engineer's integrated software technologies to reduce time to

market, improve engineering processes, and optimize product quality. Parametric

Technology's growth strategy emphasizes technological leadership, aggressive price/performance, hardware independence, worldwide distribution, and extensive customer support.

Pro/Engineer is a cluster of programs that are used to design, analyzes, and manufacture a magnitude of products. This thesis will utilize the manufacturing module whereas the sketcher module has the capability to create the 3D solids, and engineering drawings. Other modules within the Pro/Engineer will allow the construction of piping systems, wiring harnesses, sheet metal operations, and mold designs. Currently there are nearly eighty modules for the Pro/Engineer environment, and one of the popular one is Pro/Mechanica. This module performs the structural, thermal, optimization, and motion analysis of the design components.

3.3.1 Parametric Software

Pro/Engineer is a feature based parametric-based modeling system. Feature based means that the parts characteristics are produced on a base structure also called the foundation. Some of these features include datum planes, slots, cuts, holes, protrusions, rounds, and chamfers. A part is started usually with simple geometry, and by adding or removing material using Pro/Engineer's pick and place features the part achieves its final shape. This sequence or method allows the part to be modified easily at anytime. Something most other solid based programs can't accomplish.

Parametric based software allow the user to mathematical define relationships between dimensions and features. These relationships are in place to capture the design intent of the product. An example of a parametric relationship would be to control the locations of a pattern of holes, and the number of holes in proportion to the part's size.

This relationship can accomplish several design intents and will allow the user to perform major modifications to a part by simply changing one dimension. It also allows many similar parts to be designed using one parametric based model. This can be important when computer performance or disk space is a critical factor.

Pro/Engineer is currently in use at over 300 colleges and universities throughout the United Stated, and Canada. Presently here at WPI, Pro/Engineer is used in two undergraduate courses, Advanced Computer Aided Design (ES3323), and Computer Aided Manufacturing (ME3820).

Chapter 4 Pro/Engineer's Manufacturing Module in ME3820

The objective for the Pro-Manufacturing thesis designed course was fairly straightforward. Provide the necessary information to educate students on the correct use of the Pro/Engineer manufacturing module, but before this could take place it was necessary to provide the students with a solid understanding of Pro/Engineer's 2-D Sketcher. This is necessary for two reasons. First, because the class where Pro/Engineer's 2-D Sketcher is taught in detail (Advanced Computer Aided Design CAD, ES 3323) presently is not a prerequisite for CAM and second, there is still a large amount of center line (CL) sketching that is necessary to correctly use the software. This is an unfortunate problem since the process of introducing the students to sketcher involves nearly two weeks of seven-week term, resulting in a reduced CAM information content. Next is to introduce the students to the basic fundamentals of the manufacturing module. This process starts with an in-class demonstration of samples of what they will be doing for nearly the next six weeks. This is accomplished by using the overhead LCD display and some selected pre-manufactured turning and milling CAM samples.

The required equipment needs for this CAM course were presently on-hand at WPI. This was not a coincidence. The course was attempted because the equipment was available. The basic requirements were; computer workstations, software, and a CNC machine. The Higgins Design studio has eighteen Silicon Graphics workstations with working licenses for Pro/Engineer. The Washburn Robotics Laboratory has a CNC Series 1 Bridgeport attached to a DNC computer. Additionally, The Washburn Design studio located near the Robotics Laboratory has three additional Silicon Graphics workstations running Pro/Engineer also.

4.1 Methodology

The process for creating this Computer Aided Manufacturing (CAM) class for this thesis started with two weeks of corporate training from Parametric Technology Corporation (PTC) in Waltham, Massachusetts. The first week of training occurred in early May of 1998 and was for the Pro/Engineer's basic sketcher module. Before the second week of training could commence a minimum 100 hours of practice was required. This was not a problem since at the time the author of this thesis had been actively using the Pro/Engineer sketcher module for approximately 14 months. The second week of training for the fundamentals of milling module didn't occur until late August 1998 because of scheduling problems at PTC.

The objective of Parametric's training classes is to familiarize the user with the proper techniques of their software (Parametric, 1999). This is an important statement, because prior to this opportunity of factory training of Pro/Engineer's software, many students who drudged through the countless hours necessary to become self-taught, often ended up using the software incorrectly and almost always inefficiently. The inaccurate use of this software was a strong motivating factor on the concentration topic for this thesis. A quality set of easy-to-understand tutorials would provide the students with the sound basis of training, that they could use to learn this complex software. Once the software was comprehended, it could be applied to program and fully utilize the CNC machines presently in the laboratory, as well any forthcoming machines.

4.2 CAD/CAM Experience in A Term 1998

The first computer-assisted CAM class of was taught during 1998 A Term while the author's Pro/Engineer training was still in progress. This first term was primarily an experimental class, and feedback from the students was welcomed to help improve the information content of the course. Working very close with the professor, basic syllabus was developed that contained: two one-hour lectures, one hour of Pro/Engineer practice, and three hours of hands-on laboratory work nearly every week. It was decided that this term would consist of four laboratory sessions containing projects and followed up with formal lab reports. The Pro/Scan lab is an overview of reverse engineering. A complex surface is scanned using the lab's coordinate measuring machine (CMM) and the data is imported into Pro/Engineer. Two of the other labs are directly related to Pro/Engineer's manufacturing module, and this was complimented by five, Friday one-hour Pro-Manufacturing tutorial scheduled sessions.

The students received their Pro/Engineer lesson plans at the beginning of the week, which contained tutorials and class assignments for that week. The tutorials that were prepared for the students are modeled similarly to the "click by click" method practiced by Dr. Roger Toogood in his books and tutorials (Toogood, 1998).

The Pro/Engineer lecture tutorials for A Term 1998 are as follows:

Week 1:   Pro/Engineer overview, basic sketcher set-up sheet

Week 2:   Four sketcher tutorials (Crank-Throw, Rubber Foot, Bushing, Pulley).

Week 3:   Lab tutorial (Creation of milling lab part geometry, and NC Code))

21

Week 4:   Lab tutorial (Creation of milling part workpiece and Pro/Milling)

Week 5:   Review and question-and-answer period.

The above tutorials are located in Appendix A1, and A3. Essentially the tutorials for A1998 term are identical to C1999 term except for the dates listed on the cover pages.

The CNC Bridgeport was connected to a stand-alone IBM-compatible 386 computer running on a DOS operating system. This outdated computer lacked the capability to multitask, and this was not very beneficial if there was a minor problem with the program. Additionally, the only method to enter a file into the Bridgeport was through a floppy drive, and while this method worked, it was not very productive.

A Direct Numerical Control (DNC) system was implemented with an updated computer running Windows 95 that provided the necessary multitasking option. Now it is possible to send programs to the CNC server from anywhere within the world via the Internet. The DNC's computer IP address is 130.215.73.13 and its server properties allow for a two-way conservation. Meaning that once the programs are edited and corrected the server can then forward the programs back to their designated storage location.

At the end of this term the students were handed a Pro/Engineer tutorial and lab evaluation form which also provided an area for comments on improving the new lesson plan. Overall the results were quite favorable, but generally the students wanted more Pro/Engineer manufacturing content included into the course. It was concluded that the students could easily accomplish the tasks in the tutorial, but did not feel comfortable that they could produce quality data outside the realm of the information that was contained within the tutorials.

4.3 CAD/CAM Experience in C Term 1999

From the insights gained from A term, some new changes were made to the existing class syllabus. The class content was then re-evaluated in a committee meeting that included the professor, the two teaching assistants, and myself. What resulted from this meeting was a radical oval haul of the existing course content. The course work flowchart is shown in Figure 1. The existing structure from A term was not substandard, but the author's knowledge of the Pro/Engineer manufacturer's module had increased and the students desired more manufacturing knowledge also. For these reasons, two Pro/Engineer lecture sessions per week, a group-milling project, and an electronic online Pro/Engineer CAM simulation test were included as course material for C term 1999.
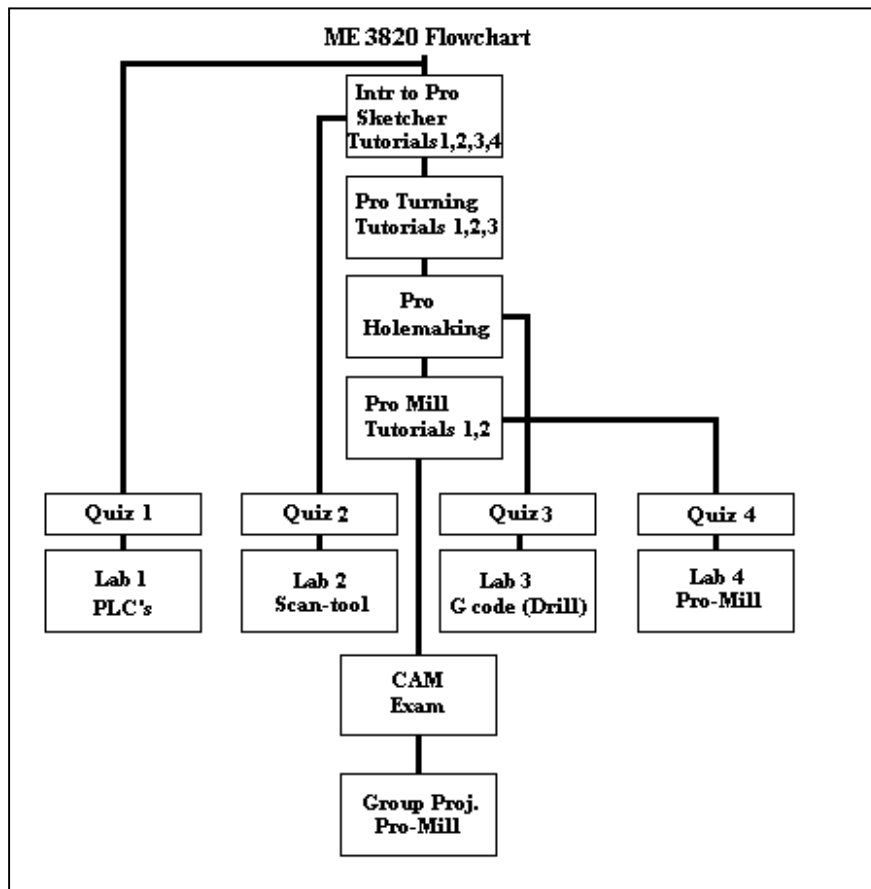
Figure 1   The finalized work flowchart for CAM ME3820.

The group milling project was assigned mid term because of its workload. The project was estimated to take:

- 3 - 6 hours to complete the design process,

- 3 - 4 hours to manufacture it within Pro/Engineer,

- 1 - 2 hours to post, and complete the necessary edits, and setup sheet,

- 1 - 2 hours to produce the engineering drawing,

- 1 - 2 hours to machine the part.

This project was intended to allow the students to demonstrate their overall knowledge of Pro/Engineer (sketcher, and manufacturing modules), as well as manufacturing techniques taught to them during the theoretical lecture portion of the class. The students are allowed to pick nearly any shape to represent their milling part. It could contain any geometry with the only restriction that it fit onto a 4-inch square by one-half inch thick plate. This was selected to help contain the cost of the class.

The online CAM simulation test is a new concept (See Appendix E). At the end of the term, students were given an exam that appeared no different than any other exam they had taken, but there was a difference. This was an electronic exam, in the sense that rather than the standard fill in the blank, or multiple choice test, the work sheet for this exam was the actual Pro/Engineer module, and the saved files were used for grading. Because of the proximity of the seating, two different versions of the exam were created, and each contained similar but different parts. The exam consisted of one turning and milling problem, and they could choose either one or both, but would only be graded on the best scored part. The students were given one hour to complete a two sequence manufacturing process. The milling part contained a profile and either slotting or hole

making sequence, whereas the turning parts similarly contained a profile and either a grooving or facing sequence. The students were instructed to save two essential files with particular names, and this was used to identify the exam work on the computer's server. At the end of the examination period, the student's exam files were transferred to an Iomega Zip disk for storage and grading. The grading scheme was as follows:

Assembly of parts	1/10
NC Sequence # 1	2/10
NC Sequence # 2	2/10
CL File (Center Line)	3/10
Manufacturing Process	2/10

Some of the additional tutorials for the new class' framework contained simplified instructions for the first time user of Pro/Engineer's turning module. This provides an opportunity for students to obtain a well rounded CAM education, and the ability to program nearly any two through four axes milling or turning machine currently available. It is not the intention of educational institutions like WPI to produce manufacturing engineers that are computer operators; moreover, the exposure of WPI students to industry recommended software will give them an opportunity to gain some practical industrial experience within a laboratory setting.

Once students were familiar with Pro-Manufacturing from a turning perspective, which was essentially contained within the 2-D environment, they were then introduced into the 3-D environment of Pro-Manufacturing. This was accomplished by having the students drill holes in various surfaces using Pro-Holemaking. Finally, the students were

introduced to the milling segment. Additional milling tutorials helped to strengthen their skills before completing the laboratory session in milling. Complete tutorials and laboratory manuals used for this term can be seen in Appendixes A1 through A3.

The Pro/Engineer lecture tutorials for C Term 1999 are as follows:

(Monday lecture = a, Friday lecture = b)

Week 1a: Pro/Engineer overview, basic sketcher set-up sheet

Week 1b: Five sketcher tutorials (Block, Crank-Throw, Rubber foot,

               Bushing, Pulley).

Week 2a: Electronic demonstration of Pro/Engineer's manufacturing module.

               Turning 1 tutorial

Week 2b: Turning 2 tutorial

Week 3a: Turning 2 review, Turning 3 tutorial and Lab 3 (G Code)

Week 3b   Turning 3 review

Week 4a: Holemaking tutorial.

Week 4b   Milling tutorial/Lab 4

Week 5a: Milling 2 tutorial

Week 5b: Milling review and project help session.

Week 6a: Exam review, and conclusion of Pro/Engineer manufacturing.

Week 6b: Electronic CAM simulation exam

Chapter 5 Tutorial and Lab Methodology

The process of teaching students a new software package requires more insight and thought process than just learning the software and writing about it. Particularly if the students are simultaneously being taught a theoretical course material process. Computer-Aided Manufacturing software such as Pro/Engineer allows the user to manufacture parts in many formats. Some of these formats can be quite complex even though the actual manufacturing process is quite simple. With proper training and experience, both in the manufacturing sector and with the software, it is possible to provide a simple technique for nearly all machining processes. The object of these tutorials is to provide correct format and technique from the start of the most basic commands, through the process of complex machining. The purpose of the laboratory manual is to reinforce the newly learned tutorial techniques. The utilization of these tutorials is separated into two sections, sketcher and manufacturing. The sketcher module is first taught to the students followed by the manufacturing section that is divided into three sub-components, turning, hole-making, and milling. It is critical that the training proceeds in a methodical manner, such that there is a complete comprehension of the sketcher features before proceeding onto the manufacturing tutorials. Likewise, the six tutorials within the manufacturing section are designed to be used in a particular and logical order. All subsequent tutorials continue to build on the user's knowledge and assumes complete comprehension of the prior tutorial.

5.1 Pro/Engineer Sketcher

The tutorials that are contained within Appendix A1 are a complete training package for teaching an undergraduate student the Pro/Engineer manufacturing module

even with no prior parametric software training. The format of the training package is to first familiarize the students with the basic setup commands of Pro/Engineer. Unlike some software that is commonly used today, Pro/Engineer does not simply default to the users requirements. Meaning, while Microsoft Word opens with several defaults (i.e. white background, font style and size, margins, etc.) the defaults within Pro/Engineer need to be setup every time the user starts the software. Once the basic setup of Pro/Engineer is understood, the next course of action is to familiarize the student with the basic sketcher functions. Although all models and workpieces necessary for these tutorials are pre-drawn and supplied, cutter paths or CL files need to be established using the sketcher module. Therefore, a good understanding of Pro/Engineer sketcher is necessary. There are five exercises or tutorials in this section that teach the students three of the four possible extrusion method. The first tutorial is named "Pulley," and familiarizes the student with common mouse commands and the revolve feature. The revolve feature is a very powerful tool within Pro/Engineer; it permits complex parts to be drawn in one step as opposed to many. Although both processes would achieve the same final design, the multistep process takes considerably longer. Within the manufacturing environment, time is money and it is never too early for the students to learn this important and crucial concept.

The next sketcher tutorial is for a part named "Block." This familiarizes the students with the most common solid extrusion method. The next tutorial is called "Crank Pin Bushing." It is another revolve featured based part and is intended to simply strengthen this important revolve feature. The blend feature is similarly another powerful extrusion method that allows complex curve and rectangular surfaces to be rapidly

sketched extruded. It is taught as the forth sketcher tutorial named "Rubber Foot." The last sketcher tutorial is called "Crank Throw." This tutorial combines the basic solid and blend extrusion method in addition to the hole-making feature.

5.2 Manufacturing Process

The above five sketcher tutorials presented in Appendix A1, provide the necessary knowledge and information content to proceed onto Pro/Engineer's manufacturing module. Training for the manufacturing section of this course involves three lathe (i.e. turning), one hole-making, and two milling tutorials. Each phase (turning, hole-making, milling) is reinforced with a minimum of four additional parts for the students to practice their newly acquired skills.

5.2.1 Turning Process

There are three turning tutorials (See Appendix A2) in this section, the first tutorial starts out with a simple turning part. There are two objectives for this tutorial. They are to introduce the students to the assembly features necessary for all work within the Pro/Engineer manufacturing module, and second to familiarize them with the basic menu setup. The tutorial also emphasizes the need for superior design intent. Although Pro/Engineer allows the designer or engineer to rapidly make changes to any of the feature-based components, if these components are not correctly and properly implemented within the part, the changes are difficult, cumbersome, and sometimes impossible to make. This is a critical concept that needs to be taught very early in the learning phase. This concept is related to the manufacturing module principally during the assembly phase. Before the simulated machining process can begin, the model and the workpiece must be precisely and accurately assembled together, so that the extra

material (if any) or critical surfaces are properly aligned. If the part and the workpiece are designed with the proper intent the assembly element becomes trivial, as opposed to difficult and time-consuming.

The second turning tutorial continues to build upon the information acquired from the first, adding more features and complexity. The intent of this tutorial is to illustrate the process of cutter path optimization, tool creation and continued reinforcement of the concepts gained from the first tutorial. Using known techniques, the first portion of the tutorial is completed. Next the user is instructed to zoom into a detailed region of the part to notice that there is still material left as a consequence of a large tool nose radius. The student is then given two options, one to redo the prior work using a tool with no nose radius, or selecting an additional tool for removing just this small region of excess material. A brief explanation about tool stress as related to nose radius is illustrated, and as a reason for proceeding with the next section of the tutorial that will remove that small amount of material. In the last part of the tutorial, the students are instructed on how to optimize the tool's cutter paths to produce efficient and productive CL files.

The last turning tutorial again starts similar to the first two, and continues to grow in complexity. The objective within this tutorial is to add new features such as grooving, threading, indexing the part for second side machining, and coordinate system selection. Upon the completion of these tutorials, the information content gained is adequate to program nearly any turned part. Additionally the part within this tutorial requires machining on both ends. Therefore the part needs to be indexed to perform the second side. One side was machined in class with the students, and the machining of the second side was assigned as homework. Three extra models and their complementary

workpieces are provided for additional class work. Although these extra parts are not

required work, they are recommended as study tools for the final CAM exam.

5.2.2 Hole-Making (Drilling) Process

The process of drilling, centerdrilling, boring, counterboring, and contersinking a

part is an extremely simple process within Pro/Engineer manufacturing environment and

is the objective of this tutorial. Part placement that was very critical in the turning

tutorials is less important in the drilling process since the extra material is simply within

its bore. The process of drilling, centerdrilling, or any of the other features listed above

are possible by simply choosing the desired feature from a menu and choosing the proper

depth. The depth feature is relatively simple since a finish model lies just beneath the

workpiece, and Pro/Engineer's intuition is to automatically choose a depth to a mating

finish surface (i.e., counterbore, to a finished counterbored surface). The most difficult

portion of this tutorial is choosing the correct holes to drill. There are six techniques to

choosing holes on a part model, which are clearly defined in the tutorial. Depths can

often be over looked due to the fact that Pro/Engineer will automatically and accurately

choose a depth that correctly represented on the finish model. Likewise with the turning

portion, extra practice models and workpieces are provided.

5.2.3 Milling Process

The last segment to the manufacturing tutorials concludes with milling because of

its complexity. A strong understanding of the previous segments is necessary before the

milling tutorials are attempted. The cutter path in the turning segment was strictly

contained within a 2-D environment, and the cutter's path within the hole-making

segment was automatically chosen; however, the milling segment will require generation of 3-D sketches and extrusions for all surfaces, except profiles. Added to this complex feature, is the fact that Pro/Engineer will not allow the cutter to extend past the workpiece's outer profile without carefully drawn window volumes. Window volumes are three-dimensional shapes that describe the material to be removed. There is no alternative to this method, and although first thought of as a hindrance it is actually a tool that allows the user tremendous freedom and flexibility when designing detailed and complicated cutter paths. Most competitive software available on the market, and sampled during the early phase of this thesis uses featured-based assumptions. A feature is an object known as a slot, hole, rib etc. A severe drawback to feature based software is that not all surfaces can be clearly defined as a feature, which can result in nonproductive and inefficient cutter paths.

The actual objective to the milling tutorials is to guide the student through the necessary steps for three-axis milling. The Robotics Laboratory's CNC Series 1 Bridgeport has the capability of two and a half axis milling. Pro/Engineer has the ability of post processing 3 axes data into 2.5 axes instantaneously. The computing power of the Bridgeport cannot produce code for the three servo motors necessary in nonlinear movements which is necessary for full 3 axes movement. This can be avoided through Pro/Engineer automatically by programming small linear moves that accomplishes the same task (i.e. a 3-D circle can be drawn as a group of small dots connected in a linear move). The tutorials also include such features as slots, pockets and complex inner and outer shapes.

5.3 Laboratory Manuals

The intent of the laboratory sessions is to reinforce the information taught during the Pro/Engineer lecture. Each laboratory session begins with a short quiz that test for pre-reading knowledge. At the conclusion of each lab, the groups must produce a thorough and complete formal engineering report. The first Laboratory session is an exercise in programmable logic controllers (PLC) and was not the focus of this thesis, but was directed from the professor. It is scheduled first while the students are gaining Pro/Engineer experience. The second laboratory session partakes on a technique known as reverse engineering. A finished part is scanned using a Coordinate Measuring Machine (CMM). The data points in a Cartesian coordinate system (X, Y, Z) taken from the part's outer surface are then imported into Pro/Engineer to create a surface. The surface can then be modeled as a solid for reproduction or tooling purposes.

Laboratory three's objective is to introduce the students to manual NC code. Although not commonly practiced in industry today, it does provide students with the necessary background for editing and error checking the NC code format produced by Pro/Engineer. In this laboratory session students write a program to drill a minimum of 12 holes whose depth cannot exceed 0.75 inches deep, within a block that is 2.12 square by 1.56 inches deep. The group is also given an extra bonus of 5 points (based on a score of 100) if they use a macro or canned cycle in the program. A canned cycle is a pre-programmed NC code function commonly used in industry to shorten the length of a program. For example, it allows many holes to be drilled with only a few lines of code. A complete explanation of canned drilling cycles is incorporated at the end of laboratory three's manual found within Appendix A3.

The final Laboratory is an accumulation of all the techniques and information gathered since the beginning of the term. In addition to the compilation of information, students complete the Pro/Engineer manufacturing process by producing an NC program by posting it through a post processor. Once the NC code is posted they perform the necessary edits and electronically forward the program to the DNC computer near the Bridgeport. The next task is to actually machine the part using the laboratories' CNC Bridgeport. To help distinguish one group's part from another, each group is given unique sizes. The finished machined part is then compared against the unique sizes assigned to that group. If changes are necessary the group is instructed to either perform this edit manually or allow Pro/Engineer to make the corrections and produce a new NC program. The Laboratory Session for the group is complete only when it passes the inspection process.

Chapter 6 Results and Discussions

The computer aided manufacturing course (ME3820) that has resulted from nearly a year of training, work, and preparation, is a class that prepares an engineering student in several areas. First, it provides the students with the necessary training to program and machine nearly any part that they could conceived within the realm of their MQP. This includes parts that are too large for our present machines by simply placing multiple coordinate systems within the component. Second, it also provides the students with knowledge of one of the finest solid modeling CAD/CAM programs available today.

The class that resulted from several iterations constitutes a workflow as shown in Chapter 4. Again the information content of the course was evaluated for C term 1999. The results were very favorable, and produced only a few negative comments. However there is still room for improvement. The need to have CAD (ES3323), as a prerequisite would solve several suggestions identified from the comment section of the evaluation form. This will emanate the 2 weeks presently spent on reviewing Pro/Engineer sketcher, and which in turn will strengthen the student's manufacturing knowledge. It would also mean that the students would be more familiar with Pro/Engineer in general, and allow a higher confidence level. A second prerequisite is the Materials Selection and Manufacturing Processes (ME 1800) class. The knowledge gained from this course would provide some basic machine and manufacturing knowledge that would be beneficial when setting the manufacturing parameters within Pro/Engineer. Some of these parameters are the speeds and feeds for various materials, spindle speed, feed rate, depth of cut, cutter material and the number of flutes on the cutter. Additionally, it would

provide the students with some basic information about proper machine setups, and available fixturing.

Comparing the results from the two terms when this class was taught is problematical. The classes differed greatly in their Pro/Engineer manufacturing course content, which makes a numerical comparison of the evaluation forms found in Appendices B and C, an asymmetrical comparison. At an in-depth look, the numbers look more favorable in the first term (A, 1998), but the amount of Pro/Engineer work was minimal and imbalanced. Moreover, a comparison of the student's comments in A 1998 term clearly showed that they did not feel comfortable with their Pro/Engineer knowledge. They did not feel qualified to produce NC code for a part, whose geometry strayed ever so slightly from original geometry contained within the tutorial that they received. As opposed to the students from the C 1999 term who produced some complex shapes for their group-milling project. The comments also supported this fact that they were well equipped to produce additional NC code, as long as they retained their class notes.

Chapter 7 Conclusions and Recommendations

The greatest impact of using Pro/Engineer software in CAM (ME3820) will be the simplicity to program existing and future CNC machines at WPI. It is sometimes difficult if not impossible to retain the knowledge necessary to program the many varieties of controllers that operate today's CNC machines, but Pro/Engineer offers an alternative. Gain knowledge of a single piece of software and be capable of programming nearly any CNC machine. It is not uncommon to see many pieces of laboratory equipment sitting idle most of the time. This is usually due to the complex characteristic associated with programming the piece of equipment. Personally I find it a waste of funds to have a machine that could be better utilized, then simply sitting idle. Pro/Engineer as the ability to reduce, if not eliminate this idle time.

Although the students in this course were only taught the basic fundamentals of the Pro/Engineer manufacturing module, it is a good foundation for the students to build upon. The information gained is adequate to program nearly any part that WPI presently has the capability to machine. Another advantage of using Pro/Engineer is its full bi-directional associativity between modules. These processes allows a part to be updated or modified within any of its modules (i.e. sketcher) and instantly have the CNC centerline (CL) files updated to represent these new changes. Likewise changes to a part's geometry made within the manufacturing module will update the lower level part instantly. Additionally, students who possess knowledge of Pro/Engineer and the manufacturing module can now designed and build an MQP with precision and

craftsmanship. Previously this would have been possible only with several years of machine shop hands-on experience.

Complex software such as Pro/Engineer, will require a fairly high degree of understanding from the teaching assistance (T/A), the professors, or both to continue to be successful in the future. One example of why this is necessary is the fact that Parametric Technology Corporation creator of Pro/Engineer, updates their software a minimum of annually and often biannually and this will require the tutorials to be reviewed for possible updating, or rewriting. Presently Pro/Engineer's current version is 20, with version 21 (Pro/Engineer 2000i) due out mid June 1999.

The Christian Brothers University's (CBU) fourteen week intermediate manufacturing course was the only publication located during the literature review search that used the Pro/Engineer's manufacturing module to teach CAM. While they have had great success with this program, there are two fundamental problems for adopting their approach to WPI's seven weeks curriculum. This seven weeks term employed here makes it difficult if not impossible to teach the three main modules of Pro/Engineer. Secondly, CBU uses verbal commands to instruct their students on Pro/Engineer. This does not allow students with prior experience to progress at a faster pace. The author of the CBU paper even mentions that this is one of the drawbacks associated with oral instructions. The alternative to oral instructions was a detailed set of tutorials that CBU felt was too time-consuming to prepare.

The direction and success of Western Washington University's presentation of its manufacturing course is an excellent example for future recommendations for ME3820. However, it should be noted that Western Washington University teaches an engineering

technology program as opposed to WPI's engineering program. Technology programs have a tendency to be a little more "hands-on," and standard engineering programs such as WPI, are more theoretical. There needs to be a clear understanding of both theoretical information and laboratory practical experience to comply with WPI's standards.

The fact that WPI has a project based undergraduate program could create a near endless opportunity for student projects. These projects could be for the individual student, or it is possible that this course could be a machining manufacturing center for the entire institute. It can be envisioned where the manufacturing program actually receives funding, and new machines to facilitate this new expanded manufacturing program.

References

1.   Bjorke, O., A New Approach to Manufacturing Engineering Education, *Proceedings Annuals of the CIRP,* Aix-en Provence, France, August 23 - 29 (1992), pp 573 - 576.

2.   Beard, B., Yeu-Sheng, S. *Intermediate Manufacturing Course for Undergraduate Education* ASEE Annual Conference Proceedings of the 1997, Milwaukee, WI, 6/15 - 6/18, pp 90-198.

3.   Cole, W., Using CAD analysis tools to teach mechanical engineering technology, *Proceedings of the 1998 Annual ASEE Conferenc,* (1998), Seattle, WA, June 28 - July1, pp 1042 - 1052.

4.   Furman, B., The Un-Lecture: A Computer-Assisted Curriculum Delivery Approach For The Effective Teaching of Mechanical Design, *Proceedings of the 26th Annual Conference on Frontiers in Education,*(1996), pp 1379 - 1382.

5.   Furman, B., Towards a More Hands-on, Design-oriented Course on Mechanisms, *ASEE Annual Conference* (1995), Anaheim, CA, (1996), June 25 - 28, pp 2763 - 2771.

6.   Grosman, A. D., Launder, B. E., Reece, G. J., Computer-Aided Engineering, Heat Transfer and Fluid Flow, Ellis Horwood Limited, England, 1985.

7.   Hamill, L., Computer Aided Learning in Civil Engineering Education, *Civil Engineering Education,* Vol. 8, No 2, Fall 1986, pp 9-21.

8.   Oberoi, H., Teaching Concepts with Real Shop-Floor Environment Laboratory, *Proceedings of the ASEE Annual Conference* (1995), Anaheim, CA, June 25 - 28pp 1443-1448.

9.   Palvia, S. C., Jalajas, D., Computer Work Education: Alternative Conceptual Approaches, *Proceedings 27th Annual Meeting of the decision Sciences Institute* (1996), pp 610 - 612.

10. Parametric Technology Corporation. Professional Services, Training. 09 May 1999. http://www.ptc.com/services/edserv/classes/1.htm

11. Solveig, O., Computer Aided Instruction in the Humanities, The Modern Language Association of America, New York, NY, 1985.

12. Toogood, R., Pro/ENGINEER Tutorial and Design Modeling with Pro/Engineer, Schroff Development Corporation, Mission, Kansas, 1998.

13. Tsatsakis, I., E., Kayafas, E., Cambourakis, G., Education in Electronics: A Multimedia Metaphor, *Proceedings IEEE; Middle East Technical University; Bilkent University; Chamber of Electrical Engineers of Turkey IEEE Piscataway NJ USA* (1994), pp. 1166 - 1169.

14. Turcic, D. A., Hamerslag, J., Jablokow, A. G., Dorst, S., Computer Aided Design and Computer Aided Manufacturing software for an Engineering Education Environment, *Proceedings of the ASME International Computers in Engineering Conference and Exhibition ,* (1987), pp. 233 - 240

15. "Banner Year for CAD/CAM, CAE in 98" *CAD/CAM, CAE Industry Update,* Daratech, Inc., April, 1998

Appendix A

Tutorials and Lab Manuals C 1999

Appendix A1

C 1999

Pro/Engineer Sketcher Tutorials

Pro/Engineering Tutorial
Dept. of Mechanical Engineering
Worcester Polytechnic Institute

- File   New (Ref this can be substituted with clicking on the "New" Icon.

"New" Menu Window:

Part (Insure that "Part " is selected in the "type" Column.
Enter the part name in the "Name" Window. (You do not need to remove the default name).
From the PART menu
- Part   Setup   Units
- Principal Sys
From the Principal System of Units Window
- Major System (click on the arrow) Pick the system of units that you desire (Ref: Pro/Engineer Default).
- Ok   Done.

- Feature   Create   Datum   Plane   Offset (Enter 3 times to accept the 0.00 offset)

There are FOUR methods of creating parts on Pro/Engineer. Pick **ONE** of the following lines. **Pick the first (Extrude) unless otherwise noted)**
- Create   Solid   Protrusion   **EXTRUDE**   Solid   Done

- Create   Solid   Protrusion   **REVOLVE**   Solid   Done

- Create   Solid   Protrusion   **SWEEP**   Solid   Done   Sketch Traj.

- Create   solid   Protrusion   **BLEND**   Solid   Done   Parallel   Regular Sec   Sketch Sec
    Done   Straight   Done

Side menu:
- One side   Done

Sketch Plane Preference:

- Click on DTM3 first. Accept the arrow direction (Okay), then click on DTM2 last.
    We are now in "2D" sketcher mode. Here you draw in 2D and then extract into 3D.

Note 1: Standard parts such as screws and washers (if necessary) can be obtained from the standard parts library, **(disk7/ptc20/objlib)** and will not be drawn in this tutorial.

Note 2: Tutorials, Tips and a vast amount of Information is available at:

http://www.geocities.com/CapeCanaveral/Lab/2549/proe.htm

PULLEY

Use the Basic setup, and note the following changes:
- Major System (click on the arrow) Pick Millimeter Newton Second (mmNs).
- Create Solid Protrusion **REVOLVE** Solid Done

This part is created by **revolving** a small modified rectangle 360°. The rectangle is actually ½ of a centerline sectional view. This process will create the outside diameter, all the geometry for the "vee" groove, and the through hole. This accomplished in **one** step, as opposed to **six**, if the blend feature was used. Once the piece is revolved, the next process will be to install a hole through *one side* of the pulley's body for the setscrew.

The first step is to create the shape as shown in Figure 1 below.

Sketcher Menu:
- Sketch Line. Draw a shape similar to Figure 2. Remember that Pro/E does not require you to draw to scale.
- Sketch Line Centerline. Draw a centerline on the horizontal datum, length is not a critical factor.
- Dimension. This part has three (3) diameters, and they will be dimensioned as diameters. This is accomplished as follows: First, click (LMB) on the left top (12.70) surface of the rectangle: second, click on the centerline; third, click back on the left top surface; forth, place the cursor where you want the leader placed and click the (CMM). Repeat this process for the top right (12.70) surface; the small flat on the bottom of the "vee" groove; and last the through bore (also the bottom of the rectangle).
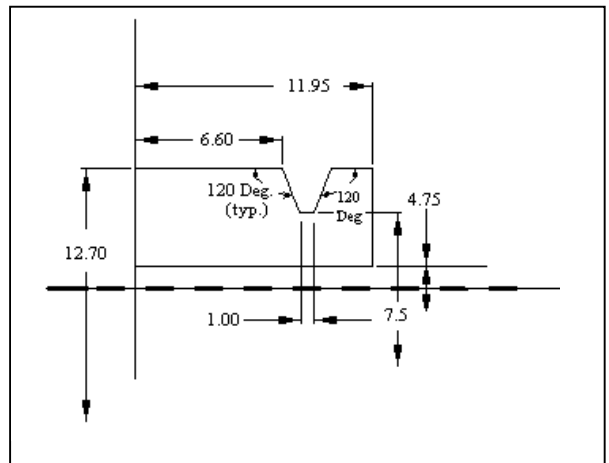
Figure 2

- Dimension. Now dimension the remaining surfaces as shown in the figure. There should be a total of eight (8) dimensions.
- Modify. Modify all eight dimensions to the above figure.
- Align. There are two surfaces that can be aligned here. Can you see them? They are the vertical edge of the rectangle that in adjacent to the datum, the horizontal centerline.

44

- Regenerate   Done.
- Revolve. Choose 360 degrees   Done.
- OK.

Now the small hole will be installed through one side of the pulley.

- Create   Hole
- Straight   Done
- Linear   Done.

Choose DTM 2 as the placement plane of the hole. Click somewhere on the pulley, and accept the arrow (Okay). The direction is not critical here because we picked one of the center datums. Therefore, any direction here would still place a hole through one wall of the pulley. Next, pick DTM 3 as the first reference location, and enter "0" offset or align "Y." Last pick DTM 1 for the second reference location. Enter 6.60/2. Sure we could have easily accomplished the math in our head, but feel free to use the computer to accomplish some of your calculations.
- One Side   Done.
- Thru all   Done.
When prompted for the diameter, Enter 2.36.
OK. From the "Hole window
Rotate the part, Shade it, and enjoy your fine work!

- Save the part (enter).

BLOCK

Use the Basic setup

Sketcher menu:
- Sketch   Mouse sketch   Rectangle. Draw a rectangle in the central on the vertical and horizontal datums. See Figure 3
- Sketch   Line   Centerline. Draw centerlines on both the vertical and horizontal datums.
- Dimension it vertically and horizontally.
- Alignment Align the Centerlines to the datums
- Regenerate.
- Modify. The dimensions are: 2.00 by 2.00.
- Regenerate   Done
- Blind   Done. Enter a depth of 1.00.
- OK

Next, a circular protrusion will be extruded top of this square feature.
- Create   Solid   Protrusion   Extrude   Solid   Done
- One Side   Done.

Choose the side opposite DTM3 as the sketch plane, and point the arrow **away** from the piece.
And pick any side as the top reference surface.

Sketcher Menu:
- Sketch . Using the middle mouse bottom, draw a circle at the intersection of the vert. And horz. DTM's. See Figure 4.
- Dimension it as a diameter (Ref double click on the circle).
- Alignment Align the Circle to the datums
- Regenerate.
- Modify. The diameter is 0.625
- Regenerate   Done.
- Blind   Done. Enter a depth of 0.50.
- OK.

Feature   Modify. From the Model Tree pick the first or second protrusion depending on what dimension need to be changed to suit your groups predetermined sizes. Click on the dimension that needs to be changed, and type in the new size.
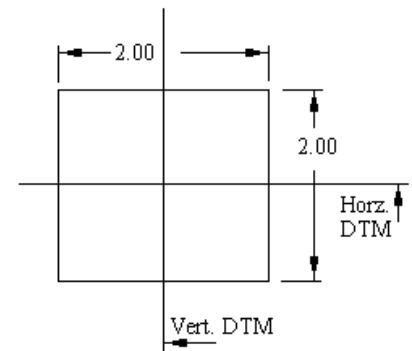
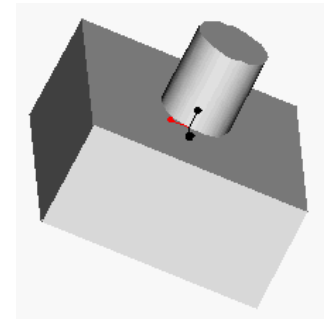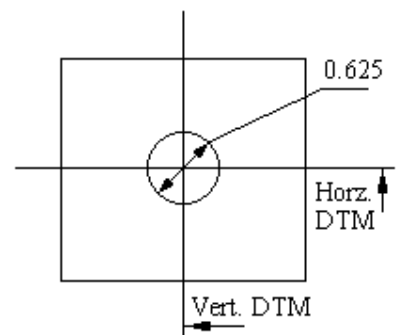- Regenerate   Done
- File   Save (enter).

Figure 3

Figure 4

CRANK PIN BUSHING

Use the Basic setup, and note the following changes:

- Major System (click on the arrow) Pick Millimeter Newton Second (mmNs).
- Create   Solid   Protrusion   **REVOLVE**   Solid   Done

This part consists again of two protrusions and a hole, and all these features are

accomplished in one small revolve feature.

Sketcher menu:

- Sketch   Line   2 Point. Draw the shape of the shaded box as shown in Figure 1.

- Dimension. Place leader lines similar to Figure 5

- Modify. The Dimensions to Figure 5. Note: The dimensions on the right side of the

  figure are "Diameters."

- Regenerate   Done.

- Revolve. Choose 360 degrees   Done.
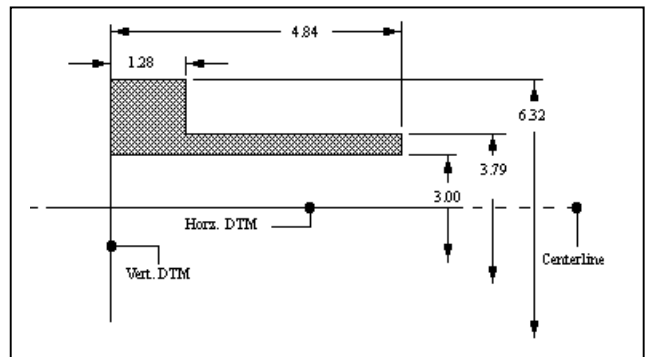
- Preview   OK.

Dbms   Save (enter)   Done.

This completes this part.

Figure 5

RUBBER FOOT

Use the Basic setup, and note the following changes:
- Major System (click on the arrow) Pick Millimeter Newton Second (mmNs).
- Create   Solid   Protrusion   **BLEND**   Solid   Done
- Parallel   Regular sec   Sketch sec   Done
- Straight   Done

Sketcher Menu:
- Sketch   Circle   Center Pt. Draw a circle that originates from the origin.
- Dimension. the diameter.
- Modify. the diameter to 12.7 mm
- Align circle to vertical and horizontal datums.
- Regenerate. After a successful regeneration, proceed to Sec tools as below.
- Sec tools   Toggle. Circle's outer edge should now turn gray.

Next the second and smaller circle will be created. Last the two surfaces will be blended together.
- Sketch   Circle   Center Pt. Draw another circle also originating from the origin.
- Dimension. the diameter.
- Modify. the diameter to 11.13 mm
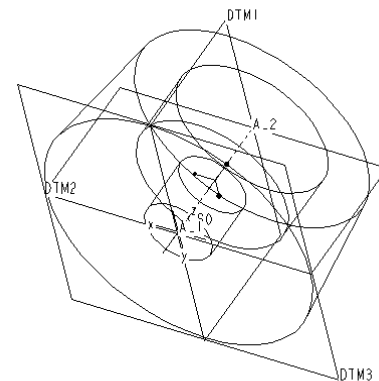
- Align as before.
- Regenerate   Done.

Enter depth of 6.15 mm.

- Preview   OK.

Create a hole through the rubber foot, starting from the top face of the protrusion.
- Create   Hole   Straight   Done
- Linear   Done. Choose DTM3 as placement plane of the hole . **(CLEARLY CLICK ON THE PART'S SURFACE)** Flip arrow to point inwards into the part. Click 'OK' when done. Now select DTM2 and DTM1 . Align to DTM2 and DTM1. (REF: offset = 0)
- One side   Done .

Figure 6

48

- Thru All   Done
  Enter diameter as 3.2 mm.
- OK.

Create another hole (Counter-Bore) from the bottom surface up towards the top surface. This will actually be the counter-bore for the screw's head. Note: The bottom is the smaller diameter of the above blend surface. (See Figure 6.)
- Create   Hole   Straight   Done
- Coaxial   Done.
Pick the "A1" axis
Choose the bottom of the foot as Placement Plane of the hole.
- One Side   Done.
- Blind   Done
  Enter depth as 3.33.
  Enter diameter as 7.2.

- Save the part!

CRANK THROW.

Use the Basic setup, and note the following changes:
- Major System (click on the arrow) Pick Millimeter Newton Second (mmNs).

<u>Sketcher menu:</u>
- Sketch   Mouse sketch   Rectangle. Draw a rectangle in the first quadrant, with two edges adjacent to the vertical and horizontal datums. See Figure 7
- Dimension it vertically and horizontally.
- Modify. The dimensions are:19.31 by 9.9.
- Align one edge to DTM2 and the other to DTM1.
- Regenerate   Done
- Blind   Done. Enter a depth of 4.14.
- OK

Next, we create a **protrusion** of type **'blend'** on the top of this feature.
- Create   Protrusion   Blend   Solid   Done.
- Parallel   Reg. Sec   Sketch Sec   Done.
- Straight   Done.
Choose DTM3 as the sketch plane, and point the arrow **away** from the piece.
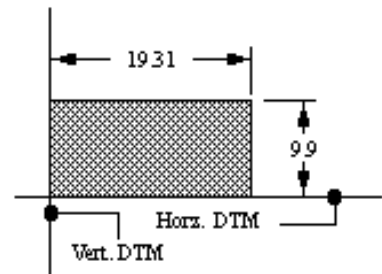And pick DTM2 as the horizontal reference.

Figure 7

<u>Sketcher Menu:</u>
- Sketch   Circle   Center pt. Draw a circle somewhere on the rectangle. See Figure 8.
- Dimension it from DTM 3 and DTM 1
- Modify   The numbers are: DTM2 to the center of the circle as 4.95 and DTM1 to the center of the circle as 14.36. The diameter is 9.9.
- Regenerate
- Sec tools   Toggle. **Circle should turn Grey**.
- Sketch   Circle   Center pt. Draw another circle that originates from the other circle.
- Dimension the diameter only this time.
- Modify. The Diameter is 8.34.
- Regenerate   Done.
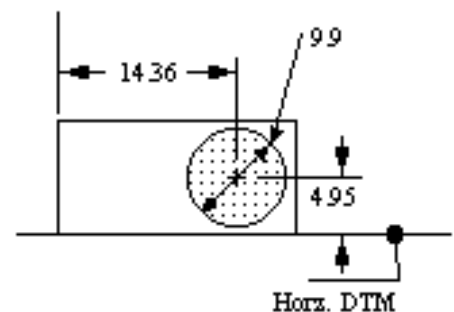- Blind   Done.
Enter depth of 4.85 mm .
- OK.

Figure 8

Now we create a hole through both protrusions.
- Create   Hole
- Straight   Done
- Linear   Done . Choose DTM3 as the reference plane for the hole. Rotate the part if necessary, and CLEARLY click ON the part. The arrow direction does not matter since the hole will be installed using "Both sides". (Click Okay)

Select DTM1 for the first reference. The distance is 4.95 from DTM1.
Now select DTM2 for the second reference. The distance is 4.95 from DTM2.
- **Both** sides   Done .
- Thru all   Done   Done.
  Enter diameter as 4.75.

Her we create another hole. This is used for mounting the connecting rod.
- Create   Hole
- Straight   Done
- Linear   Done . Choose DTM3 as the reference plane for the hole. Rotate the part if necessary, and CLEARLY click ON the part. The arrow direction does matter here, and must point into the part. (Click Okay, when it is in the correct direction.)

Select DTM1 for the first reference. The distance is 14.60 from DTM1.
Now select DTM2 for the second reference. The distance is 4.95 from DTM2.
- **One** side   Done .
- Thru all   Done.
  Enter diameter as 4.75.

Repeat the above hole procedure for the connecting rod placement hole. Several changes are required and they are: change the 4.95 (first one) to 5.00. The arrow should point **into** the part for this hole, and likewise pick One side this time. The last change is the hole diameter, it will be 2.86.

Last, a side hole is installed perpendicular to the 4.75 hole. This will be for the set screw to fasten it to the axle
Now to place the hole.
- Create   Hole.
- Straight   Done.
- Linear   Done.

Choose the new datum for the placement plane (DTM1). Click on the part. The arrow needs to point **into** the part.
First reference is to DTM 3, align or enter "0" for distance.
Second reference is from DTM 2 and the offset is 4.95.
- One side   Done.
- Thru **Next.**
- When prompt, enter 2.36 for the diameter.

Save the part. This part is now complete.

Appendix A2

C 1999

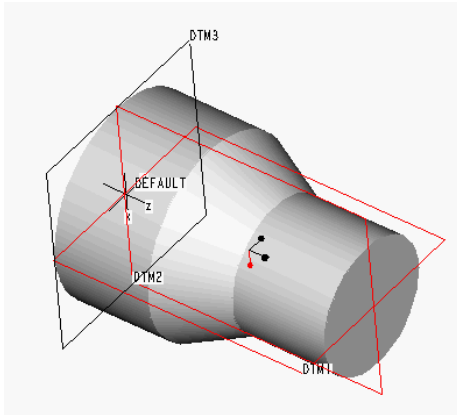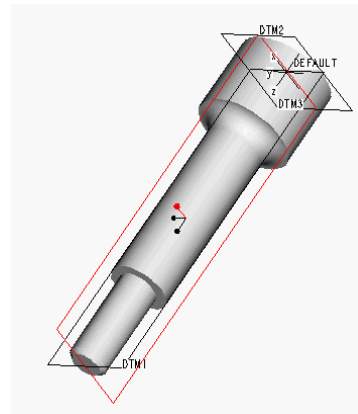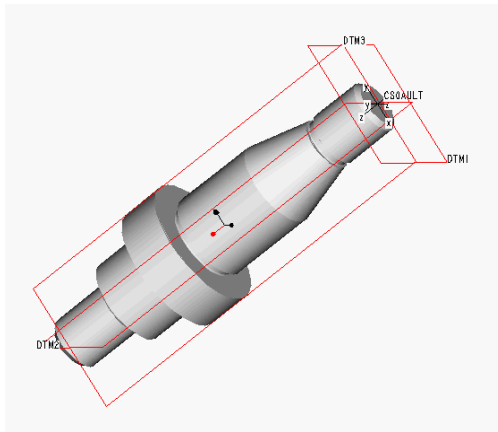**Manufacturing Tutorials**

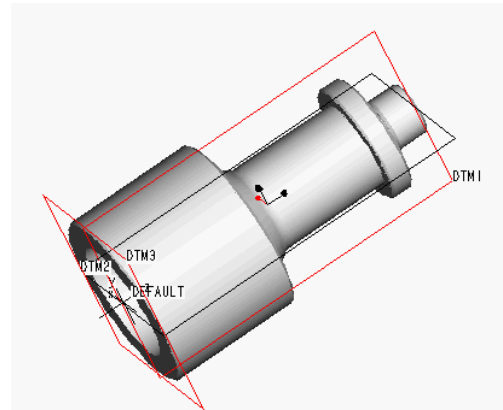# ME3820 CAM Pro/Turning Tutorials

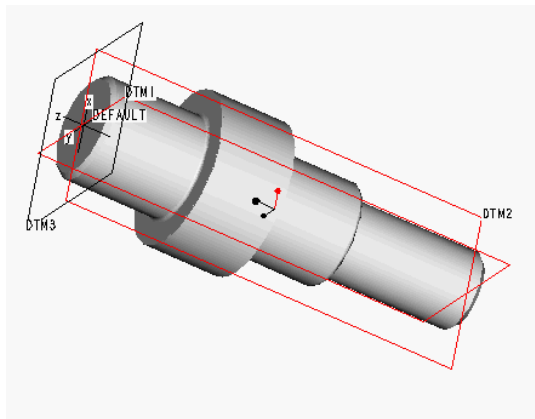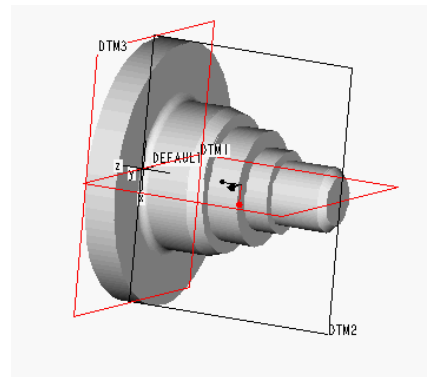TURNING 1 (TUTORIAL)

TURNING 2 (TUTORIAL)

TURNING 3 (2ND SIDE IS CLASS WORK)
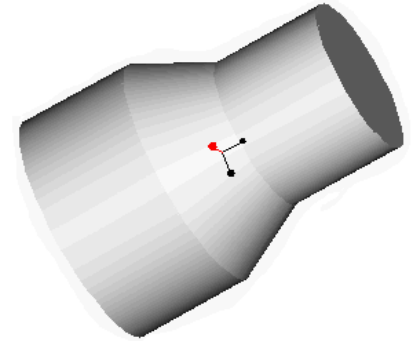(1ST SIDE TUTORIAL)

TURNING 4 (CLASS WORK)

TURNING 5 (CLASS WORK)
WORK)

TURNING 6 (CLASS

Pro/Manufacturing Turning Tutorial 1

By now you should be familiar with the Basic Commands
Pro/Engineer's Sketcher. This first part is extremely basic in its
design shape. It has one turned and tapered diameter. It will be
a good introductory part to become more familiar with the
Basic Commands of Pro/Manufacturing's Turning Module.
Before the actual machining of a part can commence, the
finished part (Model) must be assembled on to the stock part (work piece). There are two
degrees of freedom on each axis, one rotational and one translation to which results into
six degrees of freedom.   Four of the six axes must be contained before the part can be
fully constrained. Later in this course you will be designing your own models and work
piece, and will see in the next three tutorials that the best position to place the coordinate
system is such that a simple constraint placement will result in a properly designed
manufacturing model (i.e. extra material). Let's begin.

- Click on the *new* icon. Click on manufacturing and enter LATHE1 for the new name.
  Click OK. The open part window will appear, locate LATHE1 on the screen and
  double click on the correct part.

- Click on Mfg Model   Assemble   Workpiece (From the "Open Window') locate
  LATHE1_WP in double click on.

The Component Placement menu window appears. Under Constraint Type click on
Coord System and ensure that Pick is highlighted. Next clicked on the coordinate system
for each part. To click on a Coordinate System, you must click on the word (i.e. Default,
CS0). The part should now be probably placed showing extra material on the one side
where the part needs machining. When satisfied click

- OK   Done/Return.

From a menu, click on

- Machining   NC Sequence   Done Oper.

- Cell type   Lathe   2 Axis   Done.

- Horizontal   Done.

- Cell Setup   Done.

- Mach Coord System   Pick.

It is common for either the work part (WP) and the model to have several coordinate systems placed upon them, as you will see in future tutorials. The reason for multiple coordinate systems is that Pro/Engineer uses them to orient the part probably to the machine's axis of importance. In other words, if a part needs to be machined on opposite ends (for a turned part) it must have a minimum of 2 coordinate systems. Normally one on each end, with the Z-axes placed in opposite directions. It is critical to recognize the difference between the coordinate system on the work part in the one on the model. Remember the work part is highlighted in green, whereas the part is not. Use the mouse functions to move the part into clearer view before selecting the "Default" Coordinate System on the workpiece. (Note the text dialog box at the bottom of the screen.)

- Done Oper.

- Machine   Area   Face   Done.

- Seq Setup   (take defaults)   Done.

- Tool Setup   (The standard Facing Tool is acceptable) Apply   File   Done.

Manufacturing Parameters. The actual manufacturing parameters for the tool and the machine are set here. The speeds and feeds of the machine as well as the cutter's path are chosen here. This area should be experimented with after the first successful manufacturing tutorial. Speeds feed, Scan types, and depth of cut should be changed and the play path and NC check rerun to view the differences. A lot can be learned for this technique.

- MFG Param   Set (At this level of Pro/Manf. experience, only the "-1" need to be changed at first.)

    Cut_Feed = 0.1   (Change the -1 to 0.1)

55

Step_Depth = 0.1

Spindle_Speed = 1500

File   Exit   Done.

Once the Workcell is completed, and all parameters are chosen probably, the last item is to create profile in which the cutter will travel.

- Create Profile   (Name it "Face") Enter.
- Sketch   Done. (Close the Intent Manager if it appears.)The vertical line must be drawn on the right face of the model. Because the facing tool has a 1/16 nose radius, the line (i.e. Cutter's path) must extend past the centerline to completely clean the face. Also the line should extend above the green colored workpiece for proper tool entrance, and allow for varying stock sizes. See Figure 9.
- Align. Align the vertical line to the front edge of a part
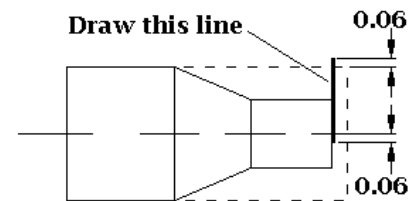
  by double clicking on the line. However, the cursor must be somewhere on the line in on the workpiece not above it.

Figure 9

- Dimension. Dimension to the Figure 1.
- Regenerate. The part should have successfully regenerate, if not recheck the above steps.
- Modify the dimensions to 0.06 two places.
- Regenerate   Done   OK.

The next two questions that you'll answer will be essentially to control the path of the tool entering and exiting the part. The default is

- Parallel (Highlighted in black)   Done. Repeat this again,
- Parallel   Done. This completes the first NC sequence, the next process is to play the cutter's path.
- Play Path   Done.

56

The generation of tool paths (in red) should be seen on the screen. If this was not clear it is possibly because the CPU is too fast. The animation can slowed down by selecting with a numerical value of something less than 1. (Click on Time Increment   Enter, type "0.1") When satisfied with tool path the next process is to view the machining process in a 3D virtual environment. During this process several colors will be apparent. The colors represented here are; green, raw stock; yellow, in process machine surfaces; magenta, finished size; light blue, under sized (or gouged) part. Unfortunately some video cards on these machines are somewhat antiquated and round surfaces are highlighted in blue although they are not undersize. This is something that will have to be tolerated on round surfaces, but does not appear on flat surfaces (i.e. milling tutorials).

- NC Check   Run. If satisfied with the manufacturing process for this N C. Sequence continue by selecting
- Done Seq.
- Material Removal   #1Face   Auto   Done.
- Save the part (Enter).

Let's begin the next NC Sequence.
- Machining   NC Sequence   New Sequence   Area   Outside   Done.
- Seq Setup   Done (Take defaults)
- Set. Change the following parameters,
     Cut _ feed = 0.1,
     Step depth = 0.1,
     Spindle Speed = 1500.
- File   Exit   Done.
- Create profile   name it "OD" Enter. Or click on the Green Icon
- Sketch   Done.
- Geom Tools   Use Edge   Sel Edge. Select the 2

Select these 2 edges
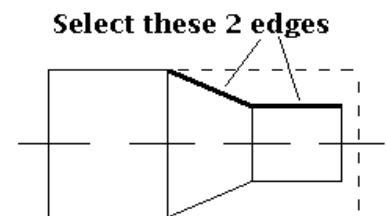
Figure 10

edges shown in Figure 10.

- Regenerate   Done   OK

- Parallel   Done

- Parallel   Done

- Play Path   Done. If the tool path is acceptable, continue on.

- NC Check   Run.

- Done Seq.

- Save the part (Enter).

This completes the machining portion of operation 10, but the two individual NC sequences (1 Face, 2 Profile) must be packaged together to complete the operation. This is accomplished by the following. From the mean menu

- CL Data   Output   Select One   Operation   OP 010. (CL stands for centerline.)

- File   Done. (If a NC tape file were required the MCD option would be checked).

- Save As   New Name "Lathe1" OK. Now it's run the entire operation consisting of the Face and the Profile sequence. This is accomplished by:

- Display   Done. Remember to change the Time Increment if the cutter path was to fast for observation.

- Done   Done Output.

- NC Check   Run. If needed, rotate the part around using the Mouse features to gain a better view of the virtual machining to insure the complete machining process. From the "Open" window locate (LATHE1) and double click on it. Note the NCL extension.

- Run. This completes the first Pro turn manufacturing tutorial.

- Save the part (Enter).

Pro/Manufacturing Turning Tutorial 2

The second tutorial begins very similar to the first one (facing, profiling); however, the second NC Sequence, will be slightly more complex requiring the use of several sketcher functions. This tutorial ends with some editing commands to improve the productivity and quality of the overall CL data file. Let's begin.
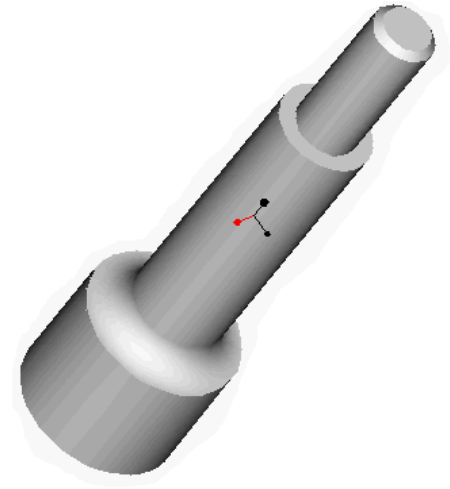
Click on the new icon. Click on manufacturing and enter a LATHE2 for the new name. Click OK. The "open part" window will appear, locate LATHE2 on the screen and double click on the correct part name.

- Click on Mfg Model   Assemble   Workpiece (From the "Open Window') locate LATHE2_WP and double click on.

The Component Placement menu window appears, under "constraint type" click on Coordinate System and ensure that Pick is highlighted. Next click on the coordinate system for each part. To pick on a Coordinate System, you must click on the word and (i.e. Default, CS0). The part should now be probably placed allowing extra material on the one end that needs machining. When satisfied click

- OK   Done/Return.
- From a menu, click on Machining   NC Sequence   (Take defaults) Done Oper.
- CELL TYPE   Lathe   2 Axis   Done.
- Horizontal   Done.
- CELL Setup   Done.

- MACH CSYS   Select   Pick. Pick the "Default" Coordinate System on the workpiece. (Note the text dialog box at the bottom of the screen. " Showing COORDINATE SYSTEM in feature 1 in model LATHE2")
- Done Oper.

- Machine   Area   Face   Done.

- Seq Setup   (Take defaults)   Done.

- Tool Setup   (The standard Facing tool is acceptable) Apply   File   Done.

- MFG Param   Set. Change the parameters with a -1 as follows.

     Cut_Feed = 0.1   (Change the -1 to 0.1, etc.)

     Step_Depth = 0.1

     Spindle_Speed = 1500

- File   Exit   Done.

- Create Profile   (Name it "Face") Enter.

- Sketch   Done. (Close the Intent Manager if it appears.) Draw a vertical line the right most face of the finished part. See Figure 11.

- Align. Align this vertical line to the front edge of the part by double clicking on the line. Note the cursor must be somewhere on the line and on the work piece not above it.

- Dimension. Dimension to the Figure.

- Modify the dimensions to the figure.

- Regenerate. The part should have successfully regenerate, if not recheck the above steps.

- Done   OK.

- Parallel   Done. Repeat this again

- Parallel   Done. This completes the first N C sequence, the next process is to play the path. To view the tool path in a 2 D. format, complete the following.

Figure 11

- Play Path   Done.
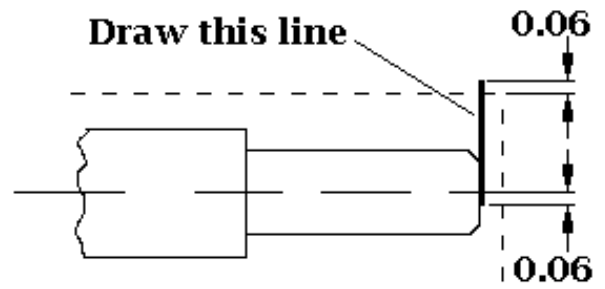
- NC Check   Run. If satisfied with the manufacturing process for this N C Sequence, to continue by selecting
- Done Seq.
- Matrl Remove   1 NC Sequence   Automatic   Done
- Save the part (Enter).

Let's begin the next NC Sequence.
- MACHINING   NC Sequence   New Sequence.
- Machining   Area   Outside   Done.
- Seq Setup   Done (Take defaults)

    Set. Change the following parameters,

    Cut_Feed = 0.1

    Step_Depth = 0.1

    Spindle_Speed = 1500
- File   Exit   Done.
- Create profile   name it "OD". (Enter)
- Sketch   Done.
- Geom Tools   Use Edge   Sel Edge. Select the 4 edges shown in Figure 2
- Sketch. Using the left mouse button draw a short vertical line connecting the first in the second diameter. See Figure 12 is the if assistance is required.
- Alignment   Align the 5 lines to the part.
- Regenerate   Done   OK
- Parallel   Done
- Parallel   Done.
- Play Path   Done. If the tool path is acceptable, continue on.
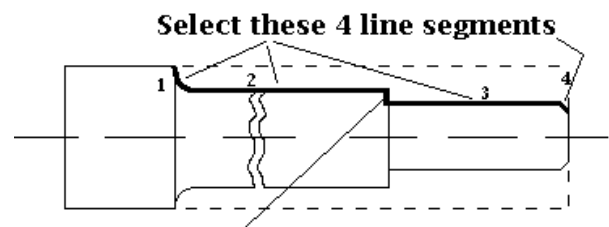
Select these 4 line segments

Sketch this line

Figure 12

- NC Check   Run.

- Done Seq.

- Matrl Remove   2 NC Sequence   Automatic   Done

- Save the part (Enter).

This completes the machining portion of operation 10, but the two individual NC sequences must be packaged together to complete the operation. This is accomplished by the following. From the mean menu

- CL Data   Output   Select One   Operation   OP 010. CL, stands for center line, and the path in which the cutter travels upon is called a centerline file.

- PATH   File   Done. (Rem: If a NC tape file were required the MCD option would be checked).

- Save As   New Name "LATHE2" OK. Now run the entire operation consisting of the Face and the Profile sequence and this is accomplished by:

- PATH   Display   Done. Remember to change the Time Increment if the cutter path was to fast for observation.

- Done   Done Output.

- NC Check   Run. If needed, rotate the part around using the mouse features to gain a better view of the virtual machining to insure the complete machining process. From the "Open" window locate the file (LATHE2) and double click on it. Note the NCL extension.

- Run.

- Done/Return.

During the NC Check, if you were to zoom in on the corner between the first and second diameter you will see that there is still material in the corner from using a tool with a radius, yet the finished part requires a

sharp corner. This could be corrected two ways. First you

0.10
2 Places

Figure 13

62
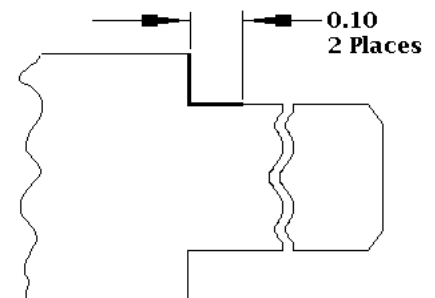
could edit the tool's radius to zero, or second add an additional NC Sequence using a tool with no radius to remove only the material left in this corner. The first choice is unacceptable because the tool's life is directly proportion to the strain that is applied to the tool tip (i.e. a tool with no nose radius usually means a drastically shortened tool life. The correct approach.   Create a new third NC Sequence using all the same parameters as NC sequence two, except to eliminate the front radius on the tool. Name the second tool T0003 (Or "Face _R_0), and change the pocket to 2. (Note you'll have to physically check the tool box at the Seq Setup portion). Name the profile "Corner". During the Sketch phase of this NC Sequence, Sketch a small one-tenth by one-tenth set of lines as shown in Figures 13. After a successful regeneration chose

- **Perpendicular**   Done rather than parallel**.** (Two times).
- **Perpendicular**   Done

- Play Path   Done
- NC Check   Run
- Done Seq.

During the NC Check you should have noticed that the tool took many cuts of before actually reaching actual material. This wasted cutter motion is unacceptable in a production environment and needs to be edited. It is accomplished by:

- Machining (From the main menu).
- Utilities   Redefine   NC Sequence   No. 3.
- Check the reference box   Done   Seq Setup   Parameters   Done   Set. Change the Rough_Only to Prof _ Only. (Note: these options can be found from the top pulled on menu).
- File   Exit

Redo the CL Data and rerun the Play path and NC Check tool path. It should be acceptable now.

This completes the first Pro turn manufacturing tutorial. Save the part (Enter).

Pro/Manufacturing Turning Tutorial 3.

This third tutorial begins very similar to the first two tutorials
with a facing and a profile sequence. However this part will be
more complex, also requiring a grooving sequence, a threading
sequence. Then the part will be indexed to have the second
side machines, (included in a second operation). Also, new
tools will be designed to meet these new machining sequences. A special note, while it is
possible to represent the work piece and model in Pro/Engineer generated tooling (i.e.
three jaw chuck), it is not advisable because the additional geometry would strain the
CPU, resulted in a slower system.
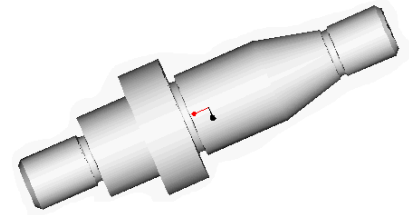
Click on the new icon. Click on manufacturing and enter LATHE3 or lathe3 (The
program is not case sensitive) for the new name. Click OK. The "open" part window
will appear, locate the part "lathe3" and double click on the correct part name. (Or single
click on the part name, or type the part name in the window, followed by clicking OK.)

- Click on MFG Model   Assm   Work Piece (From the "Open Window") locate
  lathe3_WP and double click on it to open.

The Component Placement menu window should appear. Under "constraint type" click
on **Align** and ensure that Pick, or Query Sel is highlighted. The assembly of these two
parts for this manufacturing sequence must be done differently from what was performed
in the previous tutorials. This finished model was drawn within the Pro/Engineer
environment using the revolver feature and was placed on incorrect datum planes. This
does not render the part useless, although it does make the assembly process slightly
more difficult, but only minimally. Furthermore, a detailed analysis into this part reveals
that there are actually 3 coordinate systems rather than the normal two; more about this
will made clear later in the tutorial.

The parts will be assembled using the following technique.

- Align. Using the mouse pick DTM2 on the green workpiece (When prompt, choose the yellow side). Next pick DTM 2 on the finish model, and again choose the yellow side.

- Align. For the second constraint, pick DTM 1 on the workpiece (Yellow side), and DTM 3 on the model (Yellow side).

- Align Offset. This last constraint will place the work piece properly on the model allowing extra material on each end for the facing sequence. Start this procedure by picking DTM 3 on the work part and choosing the yellow side, followed by picking DTM 1 on the model, and this time choosing the **Red** side. Enter the offset (4.625 in.)

- OK   Done/Return.

- From a menu, click on Machining   NC Sequence   New.
- Define operation (the defaults are adequate here)   Done.
- Cell type   Lathe   2 Axis   Done.
- Horizontal   Done.
- Cell Setup   Done (In Cell Setup the user can change several parameters, and other functions, but this will the not be required for these tutorial.)

This next segment of this tutorial is quite perhaps the most critical. Unfortunately after 20 version updates of Pro/Engineer, it still does not have a backup command and this is unfortunate to new users. Do not just jump into the next procedure without thoroughly understanding what is required because continuing on past point with an
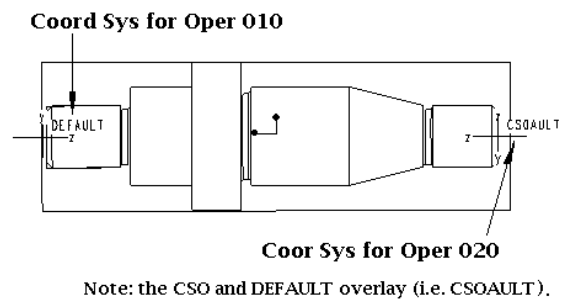

Coord Sys for Oper 010

Coor Sys for Oper 020

Note: the CSO and DEFAULT overlay (i.e. CSOAULT).

Figure 14

65

incorrect datum chosen will render this tutorial useless. By now you should be familiar with the difference between the work part, highlighted in green and the finished model, normally highlighted in white. Using the mouse control functions rotate the part so it looks similar to Figure 14 below. While it is true that Pro/Engineer does not have a backup command, the operator can use "Query Select" to ensure that the correct surface, or feature, is selected **before** accepting the selection. If not familiar with query select it can be activated by clicking on the right mouse button (RMB) when in the pick option. Once Query Select is activated, left click on the item that needs to be selected using the mouse. Read the note that is now active at the bottom of the monitor to check if it is the correct features has been selected. If not the correct feature, use the RMB to activate additional features until the correct one is highlighted. When the correct feature is highlighted both visually on the CAD screen and through the text note at the bottom, use the middle mouse button (MMB) to accept this feature, or click on accept. Those unfamiliar with this mouse feature may find it cumbersome at first, however it can drastically reduced the errors so often evident in new users work. Become more familiar with this feature if not already. Now let's attempt to pick the correct coordinate system.

- Mach Coord System   Query Sel. Select the "Default" Coordinate System on the workpiece. (Note: Pick the Coordinate system for Oper 010 as shown in Figure 1, also the text dialog box at the bottom of the screen should state "Showing COORDINATE SYSTEM in feature 1 in **model** LATHE3_WP. Confirm selection.")
- Done Oper. This completes the part set up.

- Machine   NC Sequence   Face   Done.
- Seq Setup   (Take the system defaults)   Done.
- Tool Setup   (The default standard Facing tool is acceptable) Apply   File   Done.
- MFG Param   Set. (Only the parameters with a -1 need to be changed, but more experience user can click on the advanced features and find a plethora of settings that can be updated or changed to suit your need. As you become more familiar with

Pro/Manufacturing, return to some previously acceptable tool paths and change some of the advanced features to note the new changes in the tool path.)

- Feed = 0.1 (This feed is quite high, but is acceptable for demonstration purposes)
- Step Depth = 0.1
- Speed = 1500
- File   Exit   Done.

- Create Profile   (Name it **"Face1"**) Enter.
- Sketch   Done. (Close the Intent Manager if it appears.) A vertical line must be drawn on the right most face of the finished part extending pass the surface of the work part. Because the facing tool has a 1/16 radius the tool path must extend past the centerline to completely face this surface. Refer to Figure 15 if necessary.
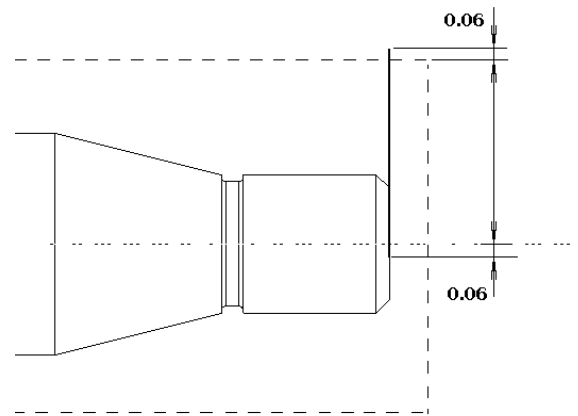
Figure 15

- Align. Align this vertical line to the front edge of the part by double clicking on the line. Note the cursor must be somewhere on the line and the model.
- Dimension. Dimension the line properly.
- Regenerate. The part should have successfully regenerate, if not recheck the above steps.
- Done   OK.
- Parallel   Done. Repeat this again,
- Parallel   Done.

View the tool path in a 2D format.
- Play Path   Done.
- NC Check   Run.

67

- Done Seq.

This is a complex part compared to the first two tutorials, and it would be helpful to remove the material after each machining operation, although this process this is not a compulsory step it will be helpful for viewing later tool paths. This is accomplished by:

- Matrl Remove   1 NC Sequence   Automatic   Done
- Save the part (Enter).

- First Profile. Let's begin the next NC Sequence.
- Machining   NC Sequence   New Sequence   Area   Outside   Done.
- Seq Setup   Done (Take defaults)

    Set. Change the following parameters, Cut _ feed = 0.1, Step depth = the.1, Speed = 1500, Rough option = Rough and Prof. (Profile options can be selected from the pull down menu)

- File   Exit   Done.

Unlike the profiles on earlier tutorials, there are areas on this profile (i.e. grooved surfaces) where extra material will be left intentionally since the existing facing tool will not function properly. This will make the creation of this profile slightly more involved that previous tutorials.

- Create profile   name it "OD1".
- Sketch   Done.
- Geom Tools   Use Edge   Sel Edge. Select the four following edges, front chamfer, threaded, taper, and the one inch diameter outside surface as shown in Figure 16.
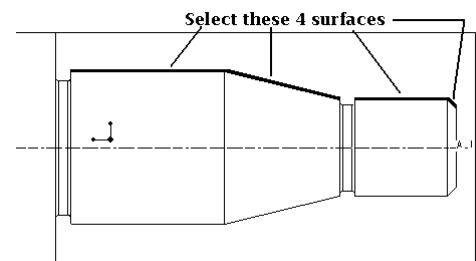
Figure 16

- Sketch. Using the left mouse button draw a short horizontal from the end of the thread surface to the

68

beginning of the tapered surface. (This line will allow the facing tool to ignore the groove) Next draw a line over the second groove and then vertically to the corner of the main diameter. See Figure 17

- Alignment   Align all lines to the part.(The no dimensions are required, but as many as 10 and as few as 7 alignment are required)

- Regenerate   Done   OK

- Parallel   Done.

- Parallel   Done.

- Save the part (Enter).

- Play Path   Done. If the tool path is acceptable, continue on.

Figure 17

- NC Check   Run.

- Done Seq.

- Matrl Remove   2 NC Sequence   Automatic   Done
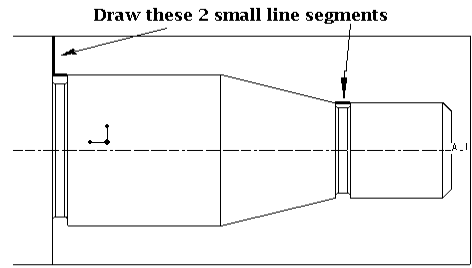
The next NC Sequence will be a grooving one. Unfortunately Pro/Engineer will not allow both grooves to be accomplished in one sequence, therefore to individual sequences must be constructed. After the first NC Sequence is complete, returned to this part of the tutorial and complete the second grooving sequence.

- NC Sequence   New

- Mach   Groove   Outside   Done.

- Seq Setup   Done

Tool Setup: make the following changes to the Tool Window. (Note the tool is now called a "grooving" tool, verses turning tool as in earlier sequences.)

Pocket = 2, Tool_ID =T0002, Nose Rad = 0.02, Side Angle = 90, End Angle = 90, Length = 3.0, Side _ width = 0.085.

- Apply   File   Done.

- MFG Param   Set. Make the following changes to the Param Tree.
  Cut feed = 0.1, Step over = 0.1, Spindle speed = 1500, Rough Opt = Prof_Only.
  File   Exit.

- Create Profile. Name it "Groove1" (Ref. Groove2 for the second sequence)
- Sketch   Done
- Sketcher   Geom Tools   Use Edge. The two grooves are only slightly different in
  that the one nearest the large diameter results in a longer left line segment. The first
  groove manufactured will be the one nearest the threads.
- Select the bottom edge of this group, and
  the two small radiuses is adjacent to it. See
  Figure 18
- Sketch   Mouse Sketch. Using the left
  mouse button draw 2 vertical lines from the
  top of the radiuses to the top of the groove.
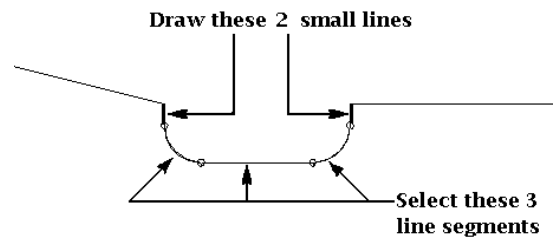
  These lines that have just been drawn will be aligned
  vertically as well as the end to the adjacent surfaces.

  Draw these 2 small lines

  **Select these 3
  line segments**

  Figure 18

- Align. Align the bottom line once, align each radius once, and the two vertical lines
  as described above.
- Regenerate. If you do not get a successful regeneration you may have to align each
  line to its adjacent line (i.e. Corner radiuses to adjacent vertical lines.)
- Done   OK.

- EXT DIR   Perpendicular   Done.
- EXT DIR   Perpendicular   Done.

- Play Path   Done
- NC Check   Run

70

- Done sequence.
- Matrl Remove   3 (or 4) NC Sequence   Automatic   Done

This completes the first groove, repeat his procedure for the second groove. Remembering that the left vertical line will be slightly longer the right line, but again no dimensions will be required, just alignments prior to regeneration.

The last NC Sequence for the first turned side will be a threading sequence. The major diameter of the thread was turned, but not the thread itself. Because of the complex geometry required to visually produce threads graphically, and would slow down the workstation's performance, NC Check is not a viable option for the threading sequence. However, the tool path will still be seen during play path process.

- Machining NC Sequence   New
- Machining   Thread   Outside   Done.
- Thread type   Unified   ISO   Done.
- Seq Setup   Done (Take defaults)
- Tool Setup. Make the following changes for threading tool.
    Tool ID = T0004, Pocket = 4, Type = Turning, Angle (Side and End) = 60, Nose
    Rad = 0.01, Side Width = 0.5, Tool Width = 1.0.
- Apply   File   Done.

- MFG Parameter   Set
- Cut feed = 0.01, Thread feed = 11, Speed = 500, Clear Dist = 0.1.
- File   Exit   Done.
- Create profile (Name it "Thread1")
- Sketch   Done.

- Geom tools   Use Edge   Sel Edge   Pick. Pick the top of the threaded surface.
  See Figure 19.
- Done/Return
- Align. Align the selected line to the top
  surface.
- Regenerate   Done   OK.
- Play Path   Done
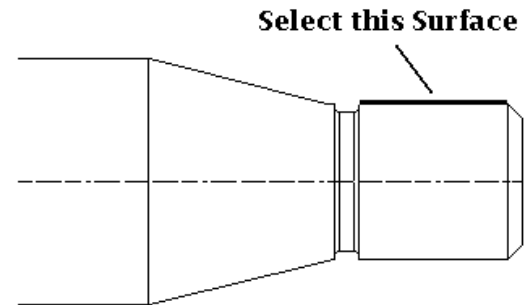
Done Seq. This completes the manufacturing
sequence for the first side of shaft. The CLData
(outlined) for operation 10 needs to be compiled.

Figure 19

- CLData   Output   Select One   Operation   Op010
- Path   File space or   Done. Name the file "Lathe3_First_ Side.   OK.
- Display   Done. Remember if the tool path display is to fast it can slowed down by
  changing the time increment to something smaller than 1. This completes the first
  end of the shaft.
- NC Check   Run. Run the entire operation 010 machining sequence.

Note the message at the bottom of the text screen " Cannot perform NC Check on NC
Sequence TURNING (id = 180). It was skipped. "

- Save the part.

Machining the Opposite End.

- Machining   Operation   Defined Oper (take defaults)   Done.
- Workcell   Use Previous   010. (Ref: Lathe, 2 Axis, Horizontal)
- Mach Coord Cys   Select   Query Sel.

Pick the **CS0** coordinate system (Refer to Figure 1 for location if necessary). Using
query select and assorted mouse functions zoom in on both coordinate systems at the
newly machined end of the model. Note there are two coordinate systems and they are

superimposed. By using query select, click on the coordinate systems, and read the text screen at the bottom of the monitor. When the correct choice is made the text note will read "Showing COORDINATE SYSTEM in feature 10 in model LATHE3. Confirm this selection. Use the right mouse button or "next" feature until the correct Coordinate System is highlighted and then accept it (middle mouse button).

The second side is similar to the first side, in that it will require the same technique and tools used to manufacture the first side. Refer to earlier tutorials, or prior sections of this tutorial for assistance.

This completes the Turning3's Pro/Turning manufacturing tutorial.

- Save the part (Enter).

ME3820 CAM Pro/Drilling Tutorial
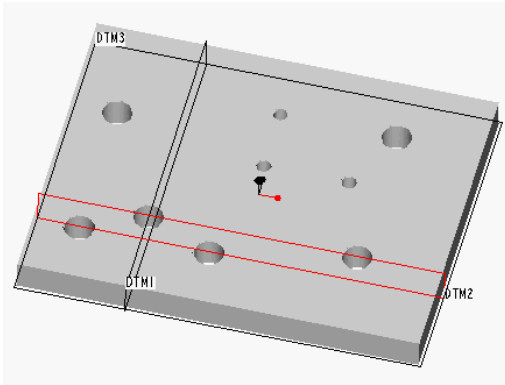
DRILL1
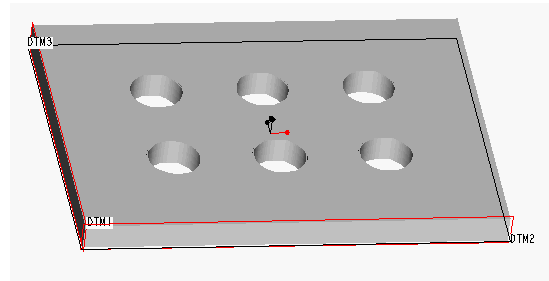
DRILL2
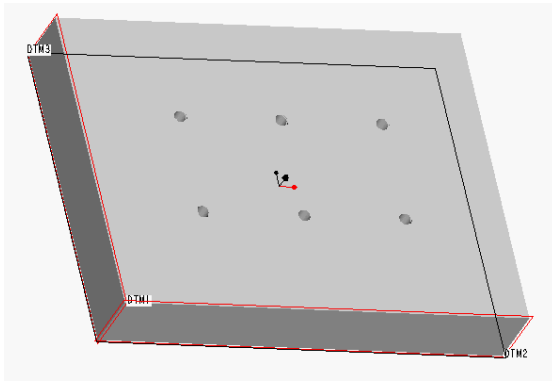
DRILL 2
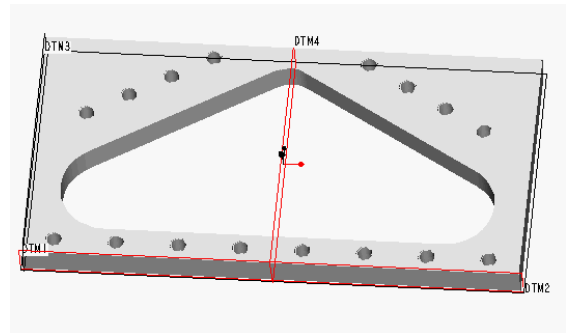
DRILL3
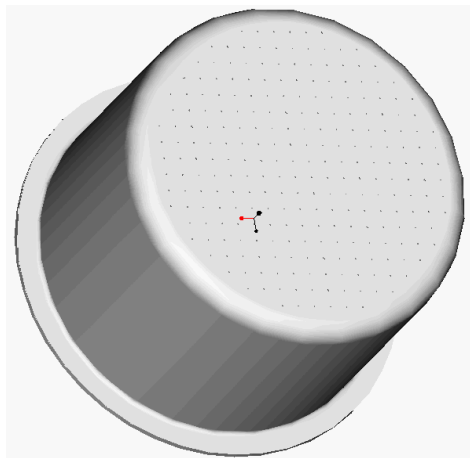
DRILL5
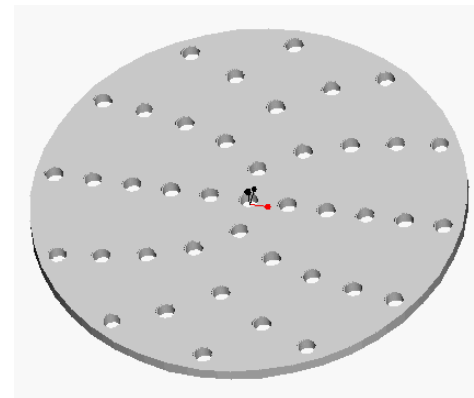
DRILL6

Pro/Manufacturing Drilling Tutorial 1.

This tutorial will guide you through the process for drilling numerous holes in a flat plate. It will also prepare the user with the necessary tools to manufacture various parts using Pro/Engineer's Hole making process. The machining process for drilling will be accomplished on a vertical-machining center since there is no machine as a CNC drill press.
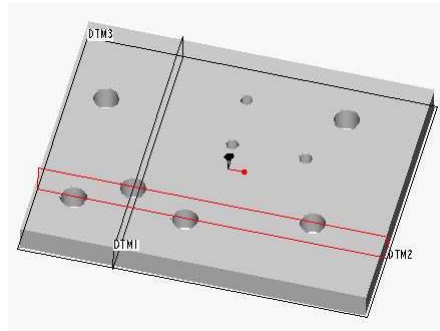
- Open Pro/Engineer.
- File ⟶ New (Or click on the "New" icon). Select manufacturing in the "Type" column, and check to see that NC Part is selected in the Sub-Type column. Enter the name "DRILL1". Next, the "Open" part window will appear. Locate and double click on the DRILL1 part to open it.

The Model and Work Piece need to be assembled first before proceeding. From the Manufacture menu choose:

- Mfg Model ⟶ Assemble ⟶ Workpiece. Another "Open" window appears. Locate and double click on the Work Piece (ref: DRILL1_WP).

The finished model and work piece will both appear in the active window. A Component Placement Window also will open. The model and the work piece need to be assembled together in a position that would properly represent actual production. There are several constraints that could work for this example, (Mate offset, Align offset, etc.) however the simplest method will be the method chosen (Coord Sys. constraint).

- Constraint Type ⟶ Coord Sys ⟶ Select ⟶ Pick. Choose the coordinate systems on the model and on the work piece (in any order). Note to pick a Coord Sys, you must click on the name (i.e. Default) and not on the axes themselves.
- OK ⟶ Done/Return

Now that the initial part setup is complete, the next process is to set up the machine

workcell.

- Machining   DEFINE OPER. (Workcell, Machine Csys, and Activate should be checked)   Done Oper.

The first parameter to select is the machine or workcell (i.e. Bridgeport)

- Create (From the WORKCELL menu)   Mill   3 Axis   Done   Done

The second parameter is to pick the coordinate system for the Bridgeport. There is only one on the block, and if it did not satisfy our requirements (i.e. "Z" axes was on the wrong plane or if the direction was opposite) another coordinate system could be created now. However, the present coordinate system is adequate for our use.

- MACH CSYS   Select   Pick. (Or Query Sel) Click on the Default coordinate system.

This finishes the define operation portion.

- Done Oper.

Next we need to define the "Type" of machining process that needs to be done.

- Machining   Holemaking   Done
- Drill   Standard   Done
- Seq Setup (Tool, Parameters, Retract, and Holes, should be checked)   Done.

The Tool Setup Window appears. (Note: After you have manufactured several parts you will have built up a library of tools, and will not have to make them each time; however, it is important to save the tool upon *exiting* Pro/Engineer.) Set the following parameters:

- Tool_ID (Drill _0_500) This is an optional piece of information that will prove useful for future manufacturing projects. This column sorts the tools, and if there is a large collection of tools with only nonsense information (i.e. T0001), you end up redesigning tools uselessly, whereas a few minutes of definition can save hours of tool redesign.
- Apply   File   Done
- MFG PARAMS   Set. A Param Tree opens. Not all parameters need to be changed only the ones with "-1" in the Right column. Adjust the information to read:

CUT_FEED: 50 (This represents 5.0 inches a minute for the Robotics Lab's
Bridgeport)

BREAKOUT _ DISTANCE :   0.06 (The distance that the body of the drill will
                        protrude to through the plate.

SPINDLE_SPEED: 1500 (Useless information, which can't be used on our Bridgeport
                        controller, but can't be left blank either.)

CLEAR_DIST: 0.1 (This will be used to control the rapid approach to the piece.)

PULLOUT_ DIST   0.1 (The distance that the drill bit will be above the work piece
                        before moving to the next hole. (This is used to clear such as
                        clamps, fixtures, etc.)

- File   Exit.

- Done (From the MFG PARAMS menu).

The retract selection menu appears. Normally we would want the cutter to retract along
the Z-axis; however, if the cutter was in a pocket (i.e. "T" slot) we might want it to move
in the X or Y-axis first, and Pro/Engineer allows us this option.

- Along Z Axis   Enter Z Depth. (Enter 1.0 here.)   OK.   This option actually creates
  an offset datum.

Hole Set. The next prompt is for the surface that we want drilled. Pro/Engineer allows
the user to select holes using six different techniques:

Axes   Specify holes by selecting individual hole axes.
Groups   Select predefined drill hole groups.
Points   Specify hole locations by selecting datum points or reading in a file with
              datum point.
Diameters   Specify holes by entering diameter values. The system automatically
              includes all Hole or round Slot features of specified diameter.
Surfaces   Specify holes by selecting surfaces of the reference part or workpiece. The
              system automatically includes all Hole or round Slot features located on
              selected surfaces
Parameters   Select holes with a certain parameter value.

- Axes. Click on the axes tab in the whole set window. (Repeat this tutorial, and
  choose diameter next time.)

- Single   Add   Pick. Click on axis A1 through A6 in any order, Pro/Engineer will

automatically create the shortest route.

- Done   Sel

- Options   Depth   Auto   Shoulder   OK   Done/Return.

- Play Path   Done (This may take a few seconds to appear because Pro/Engineer is calculating the cutter paths). Changing the Time Increment to something other than 1.0 can slow the speed of the cutter down. (Click on Enter, then renter the new time.)

- NC Check   Run. Finally the virtual manufacturing process. There are four colors during this process. Green (extra stock), Yellow, (in-process extra stock), Magenta (Finish surfaces), Blue (Gouge Surfaces). We should not see ANY blue surfaces here, if there is some present, go back and recheck your work.

- Done Seq.

- MACHINING   Matrl Remove   NC SEQ LIST. Choose NC Sequence No. 1

- Automatic   Done

- Save the part.

This completes the first sequence.

Let's begin a new sequence for the second set of holes.

- NC Sequence   New sequence.

- Machining   Holemaking   Done.

- Drill   Standard   Done

- SEQ SETUP   (Check Tool, along with the defaults) Done.

- Tool Setup Design a tool to the following parameters.

    Tool_ID (Drill _0_250)

    Pocket = 1 (Bridgeport has no Tool changer)

    Cutter Dia = 0.25

- Apply   File   Done

- MFG PARAMS   Use Prev (Select NC Sequence #1)   Done.

Hole Set

- Axes. Click on the axes tab in the whole set window.

- Single   Add   Pick. Click on axis A7 through A9 in any order.
- Done Sel   (Note: No depth was selected this time, Pro/Engineer will still assume it to automatic unless told differently.)
- OK   Done/Return.
- Play Path   Done
- NC Check   Run..
- Done Seq.
- MACHINING   Matrl Remove   NC SEQ LIST. Choose NC Sequence No. 2
- Automatic   Done
- Save the part.

This completes the machining portion of operation 10, but the two individual NC must be packaged together to complete the operation. This is accomplished by the following. From the mean menu

- CL Data   Output   Operation   OP 010. (CL stands for centerline.)
- File   Done. (If a NC tape file were required the MCD option would be checked).
- Save As   New Name " DRILL1"   OK.   Run the entire operation
- Display   Done. Remember to change the Time Increment if the cutter path was to fast for observation.
- Done   Done Output. (If needed, rotate the part around using the mouse features to gain a better view of the virtual machining to insure the complete machining process.)
- NC Check   Run. From the "Open" window locate (DRILL1) and double click on it. Note the NCL extension.
- Save the part (Enter).

This completes the first Pro Drill manufacturing tutorial.

ME3820 CAM Pro/Milling
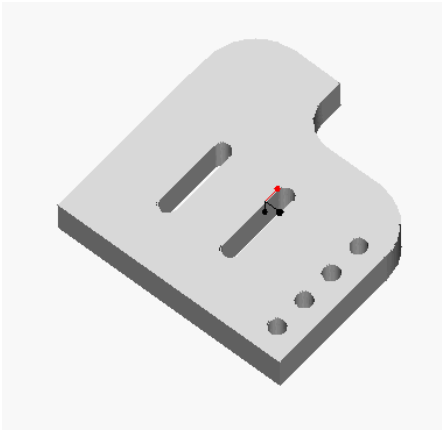
MILL1
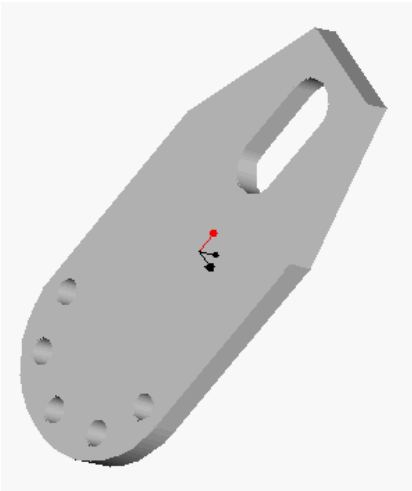
MILL2
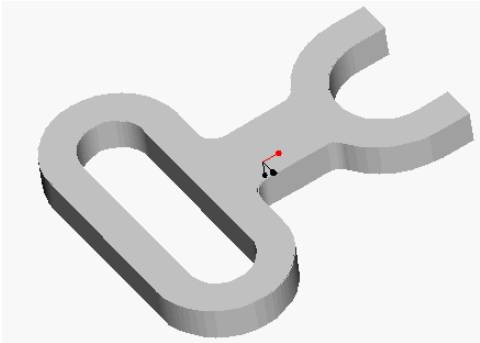
MILL3
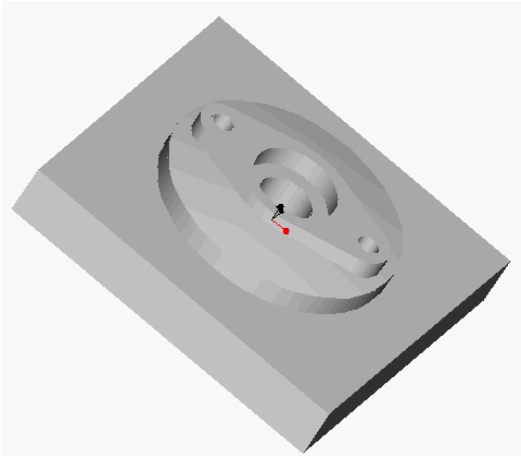
MILL 4
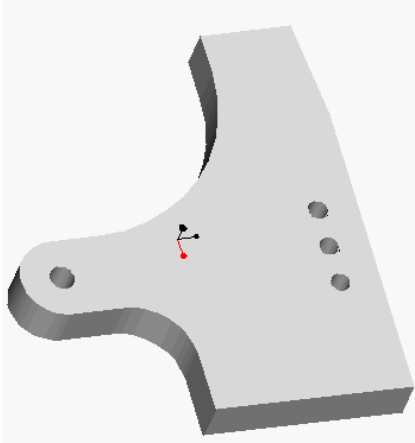
MILL5
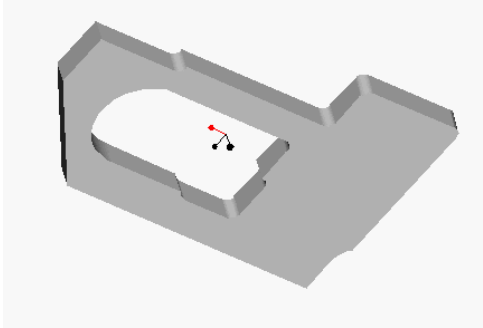
MILL6

Pro/Manufacturing Milling Tutorial

- Open Pro/Engineer.

- File   New (Or click on the "New" icon). Select manufacturing in the "Type" column, and check to see that NC Part is selected in the Sub-Type column. Enter the name "Block." Next, an "Open" part window will appear. Double click on the MILL1 part to open it.
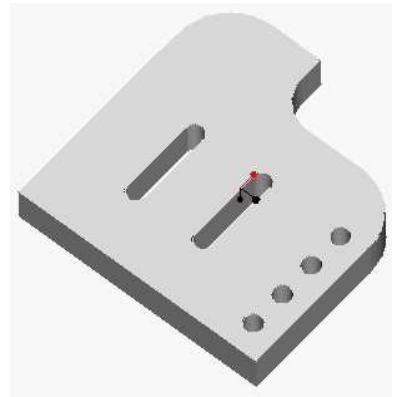
The Model and Work Piece need to be assembled first before proceeding. From the Manufacture menu choose:

- Mfg Model   Assemble   Workpiece. Another "Open" window appears. Locate and double click on the Work Piece (ref: MILL1_WP).

The finished model and work piece will both appear in the active window. A Component Placement Window also will open. The model and the workpiece need to be assembled together in a position that would properly represent actual production conditions. Here is a perfect example, when the finished part (model) and the stock (work part) have different coordinate systems and datum placements. The coordinate system on the model should take precedence for this tutorial. When properly placed, the workpiece should have equal amounts of extra stock all around the perimeter. The thickness does not require any additional machining.

- Constraint Type   Align.. Using the mouse Pick DTM3 on the green workpiece (When prompt, choose the yellow side). Next pick DTM 3 on the finish model, and again choose the yellow side.

- Align Offset. Start this procedure by picking DTM 2 on the workpiece in choosing the yellow side, followed by picking DTM 2 on the model, also choosing the Yellow side. Enter the offset (1.75 Inches)

- Align Offset. Repeat the above procedure, except choosing DTM 1 on the workpiece and choosing the yellow side, followed by picking DTM 1 on the model, and also choosing the Yellow side. Enter the offset (1.75 Inches

- OK   Done/Return.

Now that the initial part setup is complete, the next process is to set up the machine Workcell.

- Machining   Define Oper. (Workcell, Machine Csys, Activate should be checked)
- Done Oper.

The first parameter to set is a Workcell (i.e. Bridgeport)

- Create (From the WORKCELL menu)   Mill   3 Axis   Done   Done

The second parameter is to pick the coordinate system. Since the model and the work piece was assembled using offset functions the coordinate system for model and Work Piece are also offset. It is therefore critical the correct Coordinate System must be chosen in accordance with this tutorial or it will be difficult to follow.

- Select   Query Sel. Click on the Default coordinate system on the model. (Ref: The text box should respond with a "Showing COORDINATE SYSTEM in feature 1 in model MILL1. Confirm selection.".
- Accept   Done ("Done" may not be necessary.)

This finishes the "Define Oper" part.

- Done Oper.

Next we need to define the "Type" of machining process that needs to be done. The type of machining that will be accomplished on the part is fairly simple, but the sequence order is critical. Since the workpiece is a square piece aluminum plate it will need to be probably clamped in place. If the perimeter were machined first the part would move as it was cut from the outside stock and we would lose our orientation for the slot and holes. The direction that this tutorial will take will be to machine the slot and the holes in no particular order as operation 10, and then continue with a second operation (20) that will machine the perimeter. If this was a production part alternatively we could be the place a pause within the program alert the machine operator to secure the part before machining the outside perimeter.

- Machining   Holemaking   Done

- Drill   Standard   Done

- Seq Setup (Tool, Parameters, Retract, and Holes should be checked)   Done.

- Tool_ID (Drill _0_250) This is an optional piece of information that will prove useful for future manufacturing projects. This column sorts the tools, and if there is a large collection of tools with only nonsense information, you end up redesigning tools uselessly, whereas a few minutes of definition can save hours of confusion. Change the

    Cutter diameter, = 0.25 inches.

- Apply   File   Done

- MFG PARAMS   Set. A Param Tree opens. Not all parameters need to be changed only the ones with "-1" in the Right column. Adjust the information to read:

CUT_FEED: 50 (This represents 5.0 inches a minute for the Robotics Lab's
Bridgeport)

BREAKOUT _ DISTANCE :   0.06 (The distance that the body of the drill will
                                protrude to through the plate.

SPINDLE_SPEED: 1500 (Useless information, which can't be used on our Bridgeport
                                controller, but can't be left blank.)

CLEAR_DIST: 0.1 (This will be used to control the rapid approaches to the
                                piece.)

PULLOUT_ DIST   0.1 (The distance the Drill will be above the work piece before
                                moving to the next hole.

- File   Exit.

- Done (Form the MFG PARAMS menu).

The Retract selection menu appears. Normally we would want the cutter to retract along the Z-axis; however, if the cutter was in a pocket we might want it to move in the X or Y-axis first, and Pro/Engineer gives us this option.

- Along Z Axis   Enter Z Depth. Enter 1.0 here. This option actually creates an offset

datum.

Hole Set

Axes. Click on the axes tab in the whole set window.

Single   Add   Pick. Click on axis A1 through A4 in any order, remember

Pro/Engineer will automatically create the shortest route.

- Done /Return

This next sequence, can be used change how deep the drill bit will go. If skipped Auto

Depth will be selected, therefore you can skipped This and just click OK.

- Options   Depth   Auto   Shoulder   OK

- OK   Done/Return.

- Play Path   Done ("Done" may take a few seconds to appear because Pro/Engineer is
  calculating the cutter paths). Changing the Time Increment to something other than
  1.0 can slow the speed of the cutter down. (Click on Enter, then renter the new time.)

- NC Check   Run. Finally the virtual manufacturing process. Remember that there
  are four colors during this process. Green (extra stock), Yellow, (in-process extra
  stock), Magenta (Finish surfaces), Blue (Gouge Surfaces). We should not see ANY
  blue surfaces here, if there is some present, go back and recheck your work.

- Done Seq.

- MACHINING   Matrl Remove   NC SEQ LIST. Shoes NC Sequence No. 1

- Automatic   Done

- Save the part.

This completes the first sequence.

Let's begin the first new sequence for the slots. Individual sequences (2,3) must be

produced for each lot. The process below is for the first slot, repeated this process for the

second slot naming it accordingly.

- NC Sequence   New sequence.

- Machining   Volume   Done.

- SEQ SETUP   (Include Tool, along with the defaults) Done.

- Tool Setup

- Tool_ID (FEM_0_187)

- Pocket = 1 (Bridgeport has no Tool changer)

- Cutter Dia = 0.187

- Apply   File   Done

MFG Param   Set.

CUT_FEED: 100 (This represents 10 inches a minute for our Bridgeport)

STEP_DEPTH: 0.1 (Depth of cut)

STEP_OVER: 0.15 (This is the maximum amount this that the cutter would step

over if making multiple passes.)

ROUGH_ OPTION: PROF _ ONLY (Rem: Different options are located on pull down

bar)

SPINDLE_SPEED: 1500 (More useless information, which can't be used on our

Bridgeport controller, and again can't be left blank.)

CLEAR_DIST: 0.5 (This will be used to control the rapid approach

- File   Exit   Done.

- Create Vol   Name (Slot1).

- Sketch   Extruded   Solid   Done.

- One side   Done.

- Sketch Plane (Use query select to pick the top of the model Mill 1) Accept. Note:
  The red Arrow should be pointing down into the part. If this is not clear use the
  mouse functions to rotate and zoom the part to gain a better view. (Okay the arrow's
  direction if correct).

- SETUP SK PLANE   Top   Query Sel. (Pick DTM one on the model Mill 1)

- Geom Tools   Use Edge   Sel Edge. Pick the four edges of one slot.

- Regenerate   Done.

- Depth   Thru All   Done

- OK   Done/Return

- PATH   Display   Done (Ref: Change the time increment id the cutter is moving to fast).

- CL CONTROL   Done.

- Play Path   Done (If happy with the cutter's tool path continue with NC Check.)

- NC Check   Run

- Done Seq.

- Matrl Remove   #2 NC Sequence   Auto   Done.

Repeat the above NC Sequence and create sequence No. 3 for the second slot.

- Save the part.

This completes operation 10 once all the interior holes are finished. The preceding 3 NC Sequences need to be compiled into one tape.

- CLData   Operation   Pick   OP 010.

- Path Menu   File   Done. (If a NC tape file were required the MCD option would be checked).

- Save As   New Name "Mill1_Holes" OK. Now lets run the entire operation consisting of the drilled holes and the slotting sequences. This is accomplished by:

- Display   Done. Remember to change the Time Increment if the cutter path was to fast for observation.

- NC Check   Run. If needed, rotate the part around using the model features to gain a bitter view of the virtual machining to insure the complete machining process. From the "Open" window locate (Mill1_Holes) and double click on it. Note the NCL extension.

- Save the part (Enter).

With the interior holes completed the machine operator can now install fasteners in some of these holes to secure the work piece for milling the perimeter. The operation for milling the perimeter is as follows.

- Machining   Operation   New Operation   DEFINE OPER (take defaults)   Done Oper.

- Workcell   Use Previous   010. (Ref: Mill, 3Axis,etc.)

- MACH CSYS   Select   Query Sel. Click on the Default coordinate system on the model. (Ref: The text box should respond with a "Feature 1 in Model Mill 1".

- Accept   Done Oper.

- NC Sequence   New sequence.

- Machining   Profile   Done.

- SEQ SETUP   (Include **Tool**, along with the defaults) Done.

- Tool Setup. Make the following changes:

  Tool_ID (FEM_0_375)

  Pocket = 1 (Bridgeport has no Tool changer)

  Cutter Dia = 0.375

- Apply   File   Done

- MFG Param   Set. (Make the following changes)

     CUT_FEED: 200 (This represents 20 inches a minute for our Bridgeport)

     STEP_DEPTH: 0.25 (Depth of cut)

     SPINDLE_SPEED: 1500

     CLEAR_DIST: 0.5 (This will be used to control the rapid approach

- File   Exit   Done.

- Retract Selection   Along Z Plane   Enter 1.0   OK

- Select Srfs   Model   Done

- Add   Surface   Pick. (This is a tedious method for choosing the edges that make up the perimeter however it is far more accurate than choosing edge loop which seldom works. You will need to use the mouse functions to rotate the part as well as zoom to gain better access to all these surfaces. They will be highlight in red once they are selected.)

- Done Sel   Done   Done/Return   Done/Return. (You would think one would be

87

enough)

- Play Path   Done (If happy with the cutter's tool path continue with NC Check.)

- NC Check   Run

- Done Seq.

- Matrl Remove   #1 NC Sequence   Auto   Done.

This completes operation 20, (Profile milling).

To prepare the CL file:

- CLData   Output   Select One   Operation   Op010

- Path   File space or   Done. Name the file " Mill1_Profile"   OK.

- Display   Done. Remember if the tool path display is to fast it can slowed down by changing the Time Increment to something smaller than 1. This completes this Mill manufacturing sequence.

- Save the part. This completes this Mill tutorial.

Appendix A3

Tutorials and Lab Manuals C 1999

Laboratory Manuals

WPI

WORCESTER POLYTECHNIC INSTITUTE

Department of Mechanical Engineering

ME 3820 – Computer-Aided Manufacturing

Instructor: Professor M. S. Fofana   **Term:** C –1999

Office: Washburn Room 311C   **Telephone:** (508) 831-5966

e-mail: msfofana@wpi.edu

Lab Manager: R. Cournoyer   **e-mail**: rjc@wpi.edu

Title: CMM SCANTOOL

Laboratory Session Number: **ME 3820 – A99 – 302**

Date Assigned: January 27, 1999   **Due: February 5,1999**

Statement of the Problem: A local sports manufacturer with a vast selection of products has asked the WPI manufacturing program to produce several molds of their existing products. The CAM Lab (WB108) will produce a sample mold surface. These surfaces must be exact reproductions of their product line; therefore, they have provided the laboratory with a complete set of products. The process of producing these molds will require the outer surfaces of each part to be scanned using a Coordinate Measuring Machine, and then importing this data into CAD/CAM software for the manufacturing.

This lab will consist of several components, but only the first two parts will be

performed during lab time. The components of Lab1 are:

1. Scan Part
2. Backup Data
3. Data Manipulation
4. File Transfer
5. Pro/Engineer Scantool

This lab involves the complex process of taking a finished part and importing the solid surface data into a CAD system. It is one example or method of reverse engineering that is common in many industries such as the automotive, where a full size clay mockup or prototype is first produce to a size or shape that is satisfactory to the appropriate personal. This is followed by importing the surface data to an appropriate CAD/CAM program where tooling or additional parts are then fabricated using the scanned data.

This lab will involve taking a fairly simple piece of geometry (i.e. football, hockey puck, etc.), and physically scanning the part's upper surface using a Coordinate Measuring Machine (CMM). Once sufficient data points are measured (aprox. 150-400 points) the file is then copied to a floppy disk. The file called "manual.v##," where ## is represented as a 2 digit number. The groups file will be the largest (or last) file to exist in the proper directory. This text file can be imported into any word processor (MSWord) or spreadsheet (Excel) for viewing or data manipulation. The CMM stores the X, Y, and Z coordinates in an arrangement that also contains additional information that Pro/Engineer will not allow and must be cleaned before importing it.

At the end of this lab is a set of process sheets that will guide you step by step in the process to rearrange this data into the correct form. How you as a group go about rearranging this data is not important, but what is important, is that the data be in the EXACT format (See Page 9.) before importing it into Pro/Engineer.

The group should have received the hand out called the "Short Guide to Operating the CMM." Please read this manual and become familiar with the first four sections, Basic Commands, Start-up Procedure, Shutdown Procedure, and Calibration.

Procedure:

The process of scanning your object will require you to fasten your object to the CMM's granite table. Since the CMM probe exerts minimal pressure on the part it can simply be taped down to the table (ensuring the tape is not in the scanning area). If the Flywheel fixture is on the table, please DO NOT remove it, but work around it. Once the start up procedure is completed, and the tip is calibrated, (Refer to the proceed with following set of instructions:

From the Main Screen follow this step by step procedure.

Tab over to Program (enter)

H) Directory (enter)

F1 (Change Directory) Locate and highlight your groups directory under the "mustapha" directory. (enter)

ESC

A) Manual   (A new window will appear. Change the settings to read as below, using the
    right/left arrows)

      Header: (Enter the name of your part)

      Units   >IN<

      Ang.   >Decimal<

      Driver Mode   >Man<   (This is the manual mode)

      Sams Output   >Term/Stor<   (This allows the info to be stored and displayed)

F10

Tab over to "Feature" (enter)

Tab down to "Points" (enter)

Here is where the actual measurement taking process will commence. Using the joysticks move the probe near your part taking extreme care not to bump the probe on anything on the table. Since only half of the part will be scanned, you can start at either

the top, or the halfway down. The scanning of points will be done in constant "Z" curves. That is, the CMM will be moved using only the X/Y joystick till one revolution of your part is complete, then the Z will be moved (either up or down) some small amount (i.e. 0.125-0.250) the actual amount will depend on the size and complexity of your sample.

When you are properly positioned, move the probe in either the X or Y direction and touch the probe to the part. The probe should emit a beep if the point was correctly taken. Next the "F10 key needs to be pressed TWICE to prepare the CMM for the next point. Continue around the next curve taking points until the probe has completed one revolution, next move the Z and start the next set of points for the second curve. Once the part has been completely scanned, use the ESC key to exit the point menu. From the Program menu, tab to END, and enter to exit the manual mode.

Next your recorded point file will need to be copied over to a floppy disk, and this is accomplished in the UTILITY menu. The SAMS program doesn't have a "Copy" function, but the "Backup" function will accomplish this task nevertheless. Tab to UTILITY, then down to the "BACKUP" command, and press enter. A window will appear. Change the headers to read as follows:

ScrDir: 3:/sams/mustapha/(your group)

Mask: Vendor Format

Files: manual.v## (pick the file with the largest number)

Dest Dir   <A>

ESC.   Place a disk (3.5") into the drive.

F10 to start the backup command.

Copy the program cmm2proe.exe from the lab computer onto your disk. This program will read through the file created by the CMM and remove the extra characters and white space, outputting a file which contains just the columns of numbers that you want for the next step of the lab. Run the program cmm2proe.exe. The program will prompt you to enter the name of your manual.v## file and a file name to save to.

Next open your new file in the word processor of your choice, it should look like the following Sample file sheet attached (Step One). Using the editing commands, edit the file until it is modified to Step Two attached. A few notes, enter a "Begin curve ! #" between curves. Different curves are signified by a change in the "Z" coordinates. Cut and paste the first line of a curve to the last line of the curve. This ensures that your curve is a closed curve, however before leaving that curve, look to see that you did not overshoot the endpoint. Once all the editing is complete, the file is ready to be transferred as **ASCII** data into the Moosehead hard drive located in the Higgins Design Studio (HL234). This is accomplished using an FTP (File Transfer Program). The Dell PC in the Robotics Laboratory Design studio has this software installed, and is very easy to use. See the Lab instructor if further instruction if necessary

After a successful file transfer, this data may still need two more modifications done to it. First, MSWord places a carriage return at the end of each line and depending on the FTP program they will need to be edited out if present, and can be done so using the UNIX editor "Jot." When the returns are removed, save the file as a new name with an "ibl" extension (example: group8.ibl).

Open Pro/Engineer (Ver. 20) in either Higgins (HL234) or the Robotics Lab and continue with the click by click tutorial below. Please take care to not miss a step since Pro/Engineer does not have a "back-up" command. If you do have an error, see the instructor for assistance.

1. File
2. New
3. Click on "part" and enter a name
4. Application
5. Scantools
6. Scan Crv Set
7. Create Set
8. Low Density
9. Create (From "Get Coord" menu)
10. (Locate your file in the Pop-up Window)

11. Done/Return

12. Style Crv Set

13. Create Set (name it)

14. Create

15. Automatic

16. Copy Scan

17. Pick (click on all the curves)

18. Done Sel

19. Done

20. Done/Return

21. Style Surface

22. Create

23. Style Crv Surf

24. First Dir

25. Add Item

26. Curve

27. Pick – click the curves in the same order as step "17" above.

28. Done Sel

29. Done Curves

30. Preview

31. OK

32. Click on the SHADE icon

33. Click on the SAVE icon (enter)

Report

Prepare a report according to the guidelines set forth in the course syllabus. Remember, it is imperative that the front page is the same format as page 7 (NOT a copy). The order of contents follows page 6, with the understanding that not all items listed on that page will be included on all reports if there was no activity (i.e. manufacturing process).

Questions (incorporate the answers in the report)

1. What other methods are there available to import solid surfaces into CAD software?
2. Name 5 industries that regularly scan and import data.
3. What was the size of your scan file, and how many points were taken?
4. What would your group suggest to improve or simplify this lab.

Remember to give proper credit and cite information resources.

Initial condition of data from the manual.v## file direct from the Starrett CMM.

DATE: 08/14/98

TIME: 01:13pm

PROGRAM: MANUAL

OPERATOR: STUDENT

Header: tape4

```
Units:        Distance in INCH Angle in DECIMAL

Define Machine Coordinate System as MCS

Select Sensor:    SNS0

Measuring POINT  POINT1:      1 Points

FEATURE ACTUAL               POINT          POINT1
Position           x  :    12.1769   y :      3.6491    z :    -10.7283
_____
_____

Measuring POINT  POINT2:      1 Points

FEATURE ACTUAL               POINT          POINT2
Position           x  :    12.1676   y :      3.9211    z :    -10.7283
_____
_____

Measuring POINT  POINT3:      1 Points

FEATURE ACTUAL               POINT          POINT3
Position           x  :    12.1553   y :      4.1251    z :    -10.7283
_____
_____

Measuring POINT  POINT4:      1 Points

FEATURE ACTUAL               POINT          POINT4
Position           x  :    12.1372   y :      4.4194    z :    -10.7282
_____
_____

Measuring POINT  POINT5:      1 Points

FEATURE ACTUAL               POINT          POINT5
Position           x  :    12.1232   y :      4.6131    z :    -10.7282
```

Step One

1. Run the program cmm2proe.exe

What remains is nothing but XYZ coordinates separated by tabs.

```
80.2844    88.0580     30
74.2996    88.1233     30
65.7593    92.0884     30
57.2721    101.058     30
54.1674    106.885     30
49.8533    116.567     30
47.9856    127.414     30
47.8588    137.806     30
80.2844    88.0580     30
87.5227    90.5291     36
80.7437    89.6162     36
70.4187    91.6599     36
64.7492    96.1422     36
57.9387    103.244     36
50.3239    117.199     36
48.9823    125.875     36
48.9180    136.275     36
87.5227    90.5291     36
87.6395    93.6402     42
79.6438    93.2608     42
66.8769    98.9293     42
62.2199    103.479     42
52.7688    117.409     42
51.5612    133.334     42
51.7050    146.357     42
51.3358    161.927     42
87.6395    93.6402     42
```

Step Two.

1. List is numbered.
2. Header is placed above first curve .
3. Beginning of each curve is noted.

(Note 1: Notice that the first and last line of each curve is the same, this completes or closes each curve. Cutting and pasting the first line of the curve can place this last line there.)

```
Closed      Index      Arclength
Begin section ! 1
Begin curve ! 1
1. 80.2844 88.0580     30
2. 74.2996 88.1233     30
3. 65.7593 92.0884     30
4. 57.2721 101.058     30
5. 54.1674 106.885     30
6. 49.8533 116.567     30
7. 47.9856 127.414     30
8. 47.8588 137.806     30
9. 80.2844 88.0580     30
Begin curve ! 2
1. 87.5227 90.5291     36
2. 80.7437 89.6162     36
3. 70.4187 91.6599     36
4. 64.7492 96.1422     36
5. 57.9387 103.244     36
6. 50.3239 117.199     36
7. 48.9823 125.875     36
8. 48.9180 136.275     36
9. 87.5227 90.5291     36
Begin curve ! 3
1. 87.6395 93.6402     42
2. 79.6438 93.2608     42
3. 66.8769 98.9293     42
4. 62.2199 103.479     42
5. 52.7688 117.409     42
6. 51.5612 133.334     42
7. 51.7050 146.357     42
8. 51.3358 161.927     42
9. 87.6395 93.6402     42
```

Department of Mechanical Engineering

ME 3820 – Computer-Aided Manufacturing

Instructor: Professor M. S. Fofana   **Term:** C –1999

Office: Washburn Room 311C   **Telephone:** (508) 831-5966

e-mail: msfofana@wpi.edu

Lab Manager: R. Cournoyer   **e-mail**: rjc@wpi.edu

Title:

Computer Numerical Control (CNC) Part Programming

("G Code" Drill Program)

Laboratory Session Number: **ME 3820 – C99 – 303**

Date Assigned: February 8, 1999   **Due: February 19, 1999**

Statement of the Problem:

Before Lab 4 (Automated Manufacturing Procedure of Pro/Engineer) is utilized, it is critical that some basic understanding of Machine "G" code programming first be obtained. This Lab Report contains several separate manual, sample programs, and miscellaneous appendices that will give your lab group sufficient information to correctly write their own program **PRIOR** to attending the Wednesday's laboratory session. The Lab time on Wednesday is to have the program checked for errors and machining the part using the CNC Bridgeport.

This lab will consist of several components:

❑ Write a "G" code program for drilling.

❑ Successfully machine a part from the above code using the Lab's Bridgeport CNC.

❑ Write a report and include the questions below within the report (Refer to the syllabus for the correct format).

Questions (Incorporate the answers in the report)

1. For each member of the group, identify a "G" or "M" code used in today's modern CNC machines that is not allowed in the Lab's Bridgeport. (i.e. 3 members equals 3 G codes)

2. Now that the group has successfully manufactured the block by manually programming the CNC, what changes would you like to see on the next version, and why?

3. What are the G codes for cutter compensation, and when would you need them.

4. What are the speeds and feeds for a 0.750 diameter High Speed Steel, 2 flute end mill cutting mild steel plate; T6 aluminum. (This may not have been covered in class, so research it lightly)

Program Requirements and Critical Information:

• A minimum of 12 holes must be drilled.

• The depth cannot exceed 0.75 inches deep.

• The drill size is pre-selected (11/64 " dia.).

• The pattern of holes can be in any shape or depth (3/4 max.)

• Block size is 2.12' (square), by 1.56" deep. Refer to drawing in Appendix (B) for detailed "home" information and sizes.

• Your program must be a text file (i.e. lab3_drill.txt)

• All holes be must be contained within the block's material.

• **Extra credit** 5 bonus points added to the report's grade if a "Canned Cycle" is used.

(Section from the Bridgeport Operating Manual)

Bridgeport Series I CNC

The Series I CNC machine is a specially designed version of the popular Standard Series I Bridgeport Vertical Milling Machine to handle the needs of Numerical Control. It is a 2-1/2 axis milling machine with three DC stepper motors providing motion for the X,Y, and Z axes. The knee has additional way area at the top to carry the extended deep saddle, and the table is designed for an automatic machine tool. The ballscrew X and Y axis drives are suspended clear of the operator's working area and in such a way that they eliminate any bending moment on the table. The modified (2J) head is rated with a 2HP motor and includes the Z axis drive arrangement through the rotation of the nut in a preloaded recirculating ballscrew. This ballscrew axis is concentric with the quill and, therefore, concentric with the axis of rotation of the spindle to give precise, repeatable control and additional support to the quill.

The Bridgeport is programmed to automatically machine complex workpart shapes using a software language called BOSS 5.0, a numerical machine code consisting of a few basic commands. The code can be written into an ASCII file using any text editor on a personal computer. The code in the ASCII file is transferred to the Bridgeport's on-board micro-computer via a serial communications (RS-232C) line, and then is executed by controller which in turn controls the stepper motors to mill the workpart.

Bridgeport CNC Control System

The Bridgeport CNC System uses a microcomputer-based (8088) machine tool controller. The control program text may be input to the Bridgeport computer memory storage from an optical tape reader or from an external CRT, teleprinter or remote computer (as done in ME3820). The storage capacity is equivalent to 80 ft (aprox 8K). of pre-punched tape and can contain program text in the form of Word Address data to RS-247C. This text can include up to 16 macro subroutines, each having variables that can be modified in the call statement. The control permits 3 axis linear and 2 axis circular motion in switchable planes with vector velocity control. The text editor residing in the

115

microcomputer memory enables the entering, altering or outputting of the text through the use of 16 command characters. In Lab #2 you will use a text editor, such as MS Word, Word Perfect, on a separate PC. Remember to save your work as a text (txt) file on a 3.5" floppy disk.

BOSS 5.0

Programming using the BOSS 5.0 language is a relatively simple process. It consists of writing lines of code that give the Bridgeport milling machine instructions as to what path it should follow when machining a work part. Each line of code (also called an NC block) is made up of two parts: *commands* and *data*. The commands specify which motion or operation to perform and how to perform it. For example, commands define process parameters such as the dimension system (mm or inches), the kind of coordinate system, tool selection, the use of linear or circular interpolation, etc. The data provide the physical pattern (size and depth) and direction in which the part is to be cut. An NC block is an appropriate combination of these commands and data.

To generate a useful program, the programmer must send the cutting tool to the starting point of the cut. The following line of code will specify the way in which the cut will be done and the endpoint of the path. This procedure is followed for both linear and circular paths. However, circular paths require additional information. To make a circular path, it is necessary to specify the center of the circle (using I and J codes). Refer to the example code presented below for the correct format of a linear cut and a circular arc cut. Below is a summary of the BOSS command. For a complete guide of the BOSS 5.0 code please ask for the CNC User's manual available in the Robotics Laboratory.

Code   Command Name Comment

N   **Line Number** This is the first component of any NC Block. After it, the line number is given (N10, N20). The lines should be numbered by tens allow room to add additional lines.

X   **X Axis Command** Provides the x-coordinate data. All coordinates (X, Y, Z, I, J, K) should be given to at least one, and no more than four decimal places.

Y   **Y Axis Command** Provides the y-coordinate data.

Z   **Z Axis Command** Provides the z-coordinate data.

I   **Arc Center Offset, X Axis** When milling arcs, this code provides the x-coordinate of the center of the arc.

J   **Arc Center Offset, Y Axis** When milling arcs, this code provides the y-coordinate of the center of the arc.

K   **Arc Center Offset, Z Axis** When milling arcs, this code provides the z-coordinate of the center of the arc.

F   **Feedrate** This indicates the feedrate at which the workpart moves in relation to the cutter. The feedrate range is between 0.2 and 32.0 in/min given in 0.1 in/min increments (i.e. F100 = 10 in/min). A feedrate must be declared before any cutting occurs.

T   **Tool Select** Indicates which tool is being used. Up to 24 tools can be used in any given program. The Tool Length Offset (TLO) for each tool must be declared before the program is run.

M00   **Program Stop** Stops program to allow operator to perform some manual function. To continue operation, press the START/CONTINUE button.

M02   **Program End** Indicates the program is finished. The quill retracts to the Home position before motion occurs, and the controller loops to the first line of code. To repeat the par t program, press the START/CONTINUE button.

M06   **Tool Change** After the Z axis retracts to its UP position, X and Y motion will occur. The TLO of the new tool is automatically sent to the register. After the operator changes the tool and re-starts the spindle, the program is resumed by pressing the START/CONTINUE button.

G00   **Rapid Transverse** This command the mill to move at a speed much faster than that usually specified by the feed rate and is only to be used when not cutting (translating from point to point before cutting).

G01   **Linear Interpolation, feed** This command causes the machine to travel in a straight line at the specified feedrate.

G02   **Circular Interpolation Mode Clockwise** Indicates that the axes are to generate an arc in the clockwise direction.

G03 **Circular Interpolation Arc Counterclockwise** Indicates that the axes are to generate an arc in the counter-clockwise direction.

G17 **XY Plane Selection** Selects the XY plane of operation (top or plan elevation) for circular interpolation. This is the default plane selection at machine startup.

G18 **ZX Plane Selection** Selects the ZX plane of operation (front elevation) for circular interpolation.

G19 **YZ Plane Selection** Selects the YZ plane of operation (LH side elevation) for circular interpolation.

G70 **Select Inch Dimension System** Allows the user to input English units (inches). This is the default selection at machine startup.

G71 **Select Metric Dimension System** Allows the user to input units in metric dimensions (mm).

G75 **Multi-quadrant Circular Interpolation** Allows the user to mill circles that span more than one quadrant.

See Appendix (Z) for Canned Cycles

G90 **Absolute Programming Mode** When in this mode, an origin (0, 0, 0) is established on the workpiece and it remains there throughout machining. When entering data, it is given with this origin as a reference point.

G91 **Incremental Programming Mode** When in this mode, a new origin is established every time a move is made. The last point the mill moves to becomes the origin. Coordinates (data) for any operation are then given with this last point as a reference frame (origin).

NC Code Format

The format of a NC block of code is as follows: Block Number, Move Type, Data Type, Data, Feed Rate, and Tool. By combining the information in this order, the CNC controller is able to perform the milling operations. For example, the line of code below

will cause the milling machine to move from its current position to the point (1,1,0) with a feed of 20 IPM and tool 1:
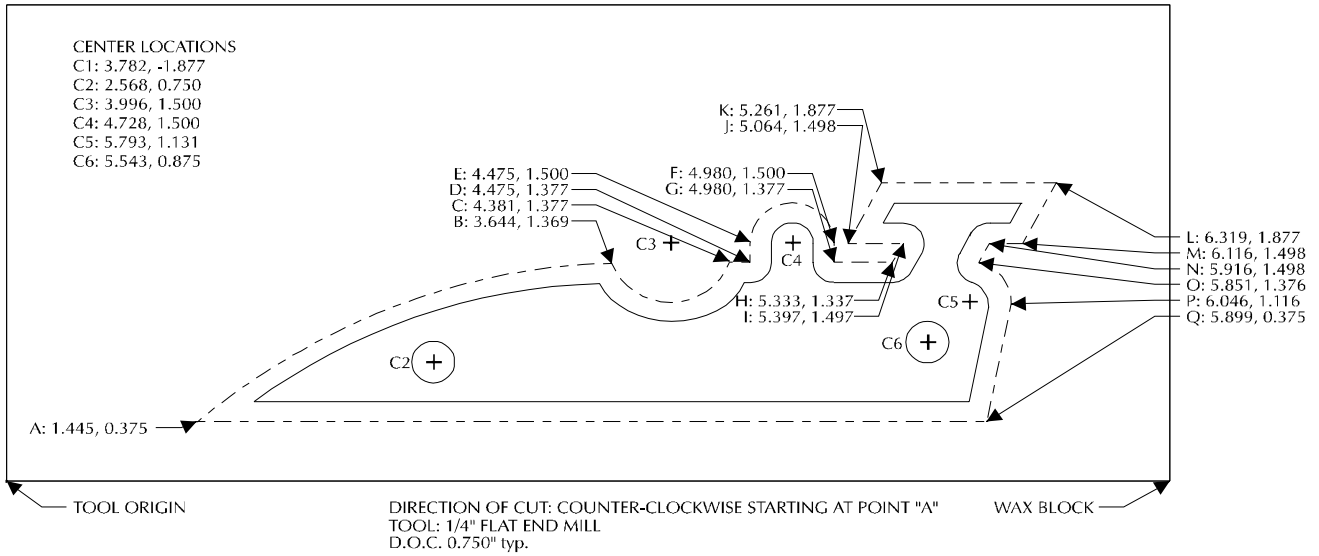
N10G1G90X1.00Y1.00Z0.00F200T1

If this code were to be broken up into its components, we would have the following:

Block #	Move Type	Data Type	Data Type	Data	Data Type	Data	Data Type	Data	Feed Rate	Tool
N10	G1	G90	X	1.00	Y	1.00	Z	0.00	200	1

Machine Drawing

The Figure below is a machine drawing of a race car chassis that is to be machined out of a block of wax similar to the one used in. The drawing is presented in the required format that includes:

- An outline of the wax block.
- Work part drawing (solid line).
- Tool path (dashed line) and tool path origin.
- Endpoints of tool path including a letter designation, and the X and Y coordinates.
- Coordinates of the arc center points.
- Tool designation.
- Depth of cut.
- Direction of cut.

CENTER LOCATIONS
C1: 3.782, -1.877
C2: 2.568, 0.750
C3: 3.996, 1.500
C4: 4.728, 1.500
C5: 5.793, 1.131
C6: 5.543, 0.875

K: 5.261, 1.877
J: 5.064, 1.498

E: 4.475, 1.500
D: 4.475, 1.377
C: 4.381, 1.377
B: 3.644, 1.369

F: 4.980, 1.500
G: 4.980, 1.377

L: 6.319, 1.877
M: 6.116, 1.498
N: 5.916, 1.498
O: 5.851, 1.376
P: 6.046, 1.116
Q: 5.899, 0.375

H: 5.333, 1.337
I: 5.397, 1.497

A: 1.445, 0.375

TOOL ORIGIN

DIRECTION OF CUT: COUNTER-CLOCKWISE STARTING AT POINT "A"
TOOL: 1/4" FLAT END MILL
D.O.C. 0.750" typ.

WAX BLOCK

C1 +

Machine Drawing.

Note that the tool path designates the path of the **center** of the tool. Therefore it is located 1/2 of the diameter of the tool away form the desired perimeter of the part (0.125" for a 0.250" end mill in the above example).

Example Code

The following is an example of how commented code should appear. Note that the points are referenced off of the engineering drawing by letter only.

N10G0G90X0Y0Z0.25T1M6   ;Start of program. Rapid transverse move to tool

;origin. 0.25 is clearance plane height.

M6 selects   ;Tool 1.

N20X1.445Y0.375   ;Rapid transverse move to point "A"

N30G1G90Z-.750F200 ;Linear interpolation move to cutting
depth of 0.75. ;Feed 20 IPM
N40G75G2G90X3.644Y1.369I3.782J-1.877 ;Multi-quadrant, clockwise arc. Endpoint
at "B"

;center at "C1"
N50G75G3G90X4.381Y1.377I3.996J1.500 ;Multi-quadrant, counter-clockwise arc.
Endpoint at "C" center at "C3"
N60G1G90X4.475Y1.377 ;Linear interpolation move to point "D"
N70Y1.500 ;Linear interpolation move to point "E".
Since last ;command was given as linear
interpolation, ;absolute (G1G90), the command does
not have to ;be repeated.
N80G75G2G90X4.980Y1.500I4.728J1.500 ;Multi-quadrant, clockwise arc. Endpoint
at "F"

;center at "C4"
N90G1G90X4.980Y1.377 ;Linear interpolation move to point "G"
N100X5.333 ;Linear interpolation move to point "H"
N110X5.397Y1.497 ;Linear interpolation move to point "I"
N120X5.064 ;Linear interpolation move to point "J"
N130X5.261Y1.877 ;Linear interpolation move to point "K"
N140X6.319 ;Linear interpolation move to point "L"
N150X6.116Y1.498 ;Linear interpolation move to point "M"
N160X5.916 ;Linear interpolation move to point "N"
N170X5.851Y1.376 ;Linear interpolation move to point "O"
N180G75G2G90X6.046Y1.116I5.793J1.131 ;Multi-quadrant, clockwise arc. Endpoint
;at "P" center at "C5"
N190G1G90X5.899Y0.375 ;Linear interpolation move to point "Q"
N200X1.445Y0.375 ;Linear interpolation move to point "A"
N210Z0.25 ;Linear interpolation move to clearance
plane

N220G0G90X2.568Y0.750	;Rapid transverse move to point "C2"
N230G1G90Z-0.75	;Linear interpolation move to cutting
depth of 0.75"	
N240Z0.250	;Linear interpolation move to clearance
plane	
N250G0G90X5.543Y0.875	;Rapid transverse move to point "C6"
N260G1G90Z-0.75	;Linear interpolation move to cutting
depth of 0.75"	
N270Z0.250	;Linear interpolation move to clearance
plane	
N280G0G90X0Y0	;Rapid transverse move to tool origin
point	
N290M2	;End program

General Information

- Type code in ALL CAPITALS, save as ASCII (Text only) file on 3.5 " disc.
- All commands are modal. That is, they act as toggle switches; once they are turned "ON", they remain "ON" for the rest of the program or until they are changed.
- When the CNC is turned on, it is initialized using its default settings. If the programmer does not specify otherwise, the default commands will be used.
- Do not use more decimal places than necessary, four is the limit, three is recommended. Use at least one decimal place everywhere, except for zeros; a single digit with no decimal place is interpreted as 0.00X.
- The tool path designates the path of the center of the tool. Therefore it is located 1/2 of the diameter of the tool away form the desired perimeter of the part (0.125" for a 0.250" end mill in the above example) Remember Lab2 will use a 0.75 Dia. Cutter.
- Comments and spacing should be added to code so that it is readable. This helps in debugging or later modifications. A comment begins with a semicolon and continues to the end of that line. It can be placed before a block of code to summarize a complex operation, or on the same line as a BOSS 5.0 command. Comments, as well

as blank lines and spaces, will be removed before it is downloaded to the CNC with RCOMMENT, a comment removing program.

- The coordinate measurement system on the CNC has been established as follows: When facing the CNC, positive X is to the right; positive Y is away from the person, and positive Z is towards the ceiling.

- A feedrate must be declared before any cutting occurs. Typically, the feedrate is defined on the first block of code containing a "G1".

- This machine is capable in milling in 2 1/2 axes. That is, arcs can only be machined in the xy, yz, or zx planes, but linear feeds can use the full 3 axes simultaneously

- A clearance plane is used to ensure that the tool does not collide with fixtures or other material on the table.

- To implement tool changes, the workpiece must be programmed to move away from the spindle. A tool change code (M6) must be included. After the milling machine pauses, the operator will be able to turn the spindle off and change the tools. One the tools have been changed, the milling process may be continued by pressing the START/CONTINUE button.

Setting Up the CNC

To set up the CNC, the X and Y origin of the part, and the vertical distance of the tool form the stock must be established. The latter is done by entering the Tool Length Offset.

Main Control Panel

The main control panel is located on the right side of the Bridgeport. It houses the various operator interfaces divided into five the columns described below (left to right):

OVERRIDE:

Limit: Resets the axes when they move beyond their range of motion.

% Feedrate: Allows the user to manually control the feedrate as a percentage of the rate

programmed into the BOSS 5.0 code.

ABS/TLO:

Upper selector switch:

Zero: Used to enter the X and Y zero coordinates, and to go to the Z zero

coordinate.

Goto: Moves table to the X and Y zero coordinates.

Lower selector switch:

XY/T1:   Selects the X and Y zero.

Z/TNO: Selects the Z zero.

AXIS MOTION:

Upper left switch: Selects increment of motion from step (0.001") to jog.

Upper right switch: Select which axis to move.

Move switch: Performs actual move in either the + or - direction.

FUNCTION:

Upper selector switch:

Restart: Restarts the current program from the beginning.

Start/Continue: Starts the current program, or restarts the program after a hold or

pause.

Lower Selector switch:

Edit: Calls for a program to be downloaded from PC (sends a "handshake" signal).

MODE:

Upper selector switch:

Auto: Allows program to be run automatically.

Block: Executes program one line (block) at a time.

Set-up: allows set-up procedure to be carried out (i.e. zeroing, TLO, downloading).

Hold button: Manual pause button.

Keypad:

Used to enter TLO data and select which coordinates to display.

Start up procedure

Unless otherwise specified, the user will be entering information at the main control panel

1. Turn on the PC located adjacent to the CNC
2. Turn on main CNC power. Switch lever located on the rear right side of the CNC.
3. Switch the CNC to set-up (mode column)
4. Reset the limit (OVERRIDE column)

Zeroing the X and Y coordinates (Part "Home" Position)

This procedure sets the center of the tool at the tool path origin. A device called an Edge Finder is used to locate the edges of the stock being cut. There is no preset or mandatory Home position, and for this lab, XY Home will be the center of the fixture (Ref 0.375 Diameter Hole).

125

Setting the TLO

Use this procedure for setting the TLO for 1 tool only. Multiple tools require a different procedure. See you lab instructor for assistance.

1. Move the spindle to the Z origin by selecting Zero and selecting and pressing Z/TNO (ABS/TLO column).
2. Move the stock under the end mill.
3. Lower the knee so the stock is approximately 1.5" below the end mill.
4. Lower the tool 1.000".
5. Hold a strip of paper between the end mill and the top of the stock. **DO NOT** start spindle
6. Use the knee adjustment lever to move the stock up to the end mill.
7. Slide the paper under the end mill. Stop moving the knee when the paper grabs.
8. Move the table up 0.003"
9. Re-zero the knee dial by loosening the knurled locknut and spinning the dial until zero aligns with the hash mark.
10. Retighten the knee dial locknut.
11. Move the spindle to the Z origin by selecting Zero and selecting and pressing Z/TNO (ABS/TLO column).
12. At the keypad press the T button located in the green strip.
13. Enter 1 (for tool 1)
14. Press the TLO button located in the green strip.
15. Enter 1000 (for 1").
16. Press the <enter> key (unmarked key in the lower right corner).

The TLO should not be greater than 1.5 inches due to a misalignment in the vertical screw. Additionally it must always be greater than the clearance plane. When using more than one tool, be sure that the shortest tool has a TLO less than 1.5 inches.

Downloading

Prior to sending your program to the Bridgeport's memory module, the program first needs to be placed into the DNC (Direct Numerical Control) PC's memory. There are several possibilities or methods to accomplish this task.

The program can be

- Stored on a 3.5"floppy. (known as "Sneaker-Net")
- FTP the file to the IP address 130.215.73.13 (fab5.wpi.edu) Anonymous login with correct email address required
- ICQ File transfer. (ICQ # 19 601 631)

To transfer your program to the CNC Bridgeport follow the procedure outlined below:

1. From the PC located near the Bridgeport, start the Procomm software (Click the icon from the hidden MSOffice tool bar) Proceed to the next step after the "blue" screen appears.
2. Clear and Reset the CNC machine by using the clear/reset toggle switch located on the left side of the machine in the control cabinet. Make sure x and y coordinates are at (0,0) before you reset the machine so that the axes do not need to be re-zeroed.
3. Zero the X, Y and Z coordinates by selecting Zero and selecting and pressing XY/T1 (ABS/TLO column)
4. Reenter the Tool Length Offset for the tool or tools you are using. (Ref: "Setting the TLO" section above.)
5. Press the Edit button on the Bridgeport control panel (FUNCTION column).
6. If an asterisk (*) shows up on the PC screen, repeat steps 3 through 6.
7. At the PC press the <Page Up> key followed by "7". Press <Enter>.
8. Type the path and filename of your un-commented program and press <Enter>. You should see your program being sent (line by line) on the screen. (i.e. A:\Test.txt, C:\CNC\Test.txt)
9. Once the computer finishes downloading the program, it will stop. Press the <Ctrl> and "Z" keys simultaneously to finish the downloading sequence. If the word "EXIT" is not displayed, press <Ctrl> <Z> again.

If you wish to return to the DOS prompt, press the <ALT> and "X" keys simultaneously.

Machining

1. After the program has been downloaded to the CNC, return to the Bridgeport control panel and do the following:
2. Move the upper selector switch located in the MODE column from set-up to auto.
3. Start the spindle (HIGH SPEED ONLY).
4. Move the % feedrate dial to 20 %.
5. Move the restart start/continue switch located in the FUNCTION column to start/continue. Press the button.
6. The machining process will begin. Adjust the feedrate accordingly to allow for chip removal.

If pauses or tool changes are included in the program, the start/continue button can be used to continue milling. To manually pause the process, press the hold button. The start/continue resumes the process.
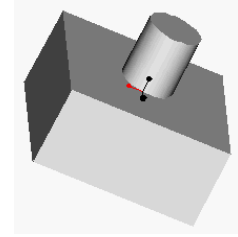
MOST IMPORTANTLY:

See the Lab Instructor if there are ANY questions.

Appendix A (Lab 3)

Sample Milling Program

Here is another sample of a "G" code program. This one for
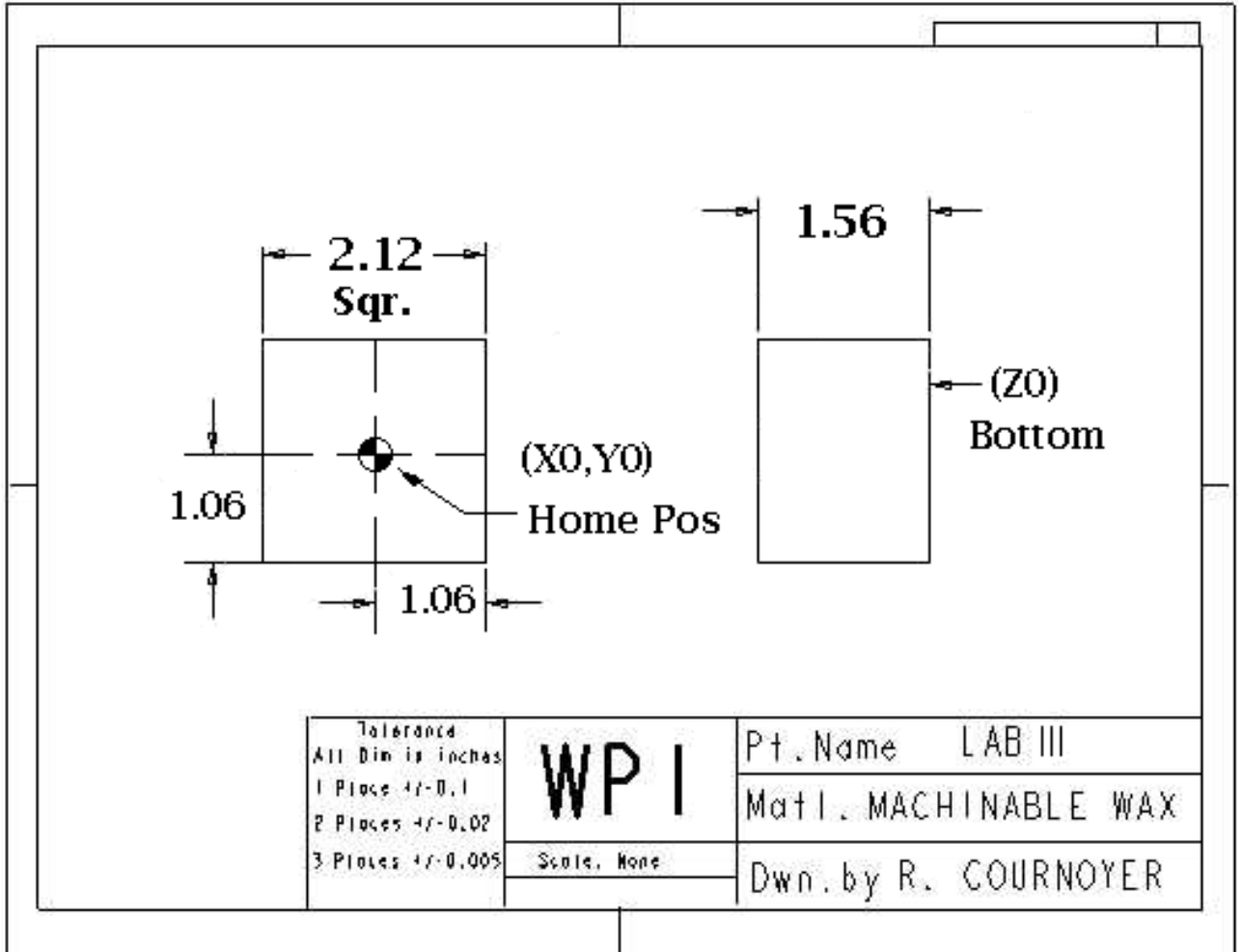that for the block as shown.

```
N10G0G90X0Y0T1M6
N20G1G90X-1.40Z0.5F100
N30Z0                   ;Initial cut across the top
N40X1.10
N50Z-0.25               ;Z movement for first milling trip
N60Y1.0
N70X-0.6875
N80Y-1.0
N90X0.6875
N100Y0
N110G75G3G90X0.6875Y0I0J0    ;Cutting the extruded shaft
N120G1G90Z-0.5               ;Z movement for second mill
N130Y1.0
N140X-0.6875
N150Y-1.0
N160X0.6875
N170Y0
N180G75G3G90X0.6875Y0I0J0    ;Cutting for the extruded shaft
N190G1G90X1.375              ;Move to the outside walls
N200Z-0.75              ;Z Movement for outside milling
N210Y1.375
N220X-1.375
N230Y-1.375
N240X1.375
N250Y0
N260Z-1.00              ;Z movement for outside milling
N270Y1.375
N280X-1.375
N290Y-1.375
N300X1.375
N310Y0
N320Z-1.25              ;Z movement for outside milling
N330Y1.375
N340X-1.375
N350Y-1.375
N360X1.375
N370Y0
N380Z-1.50              ;Z Movement for outside milling
N390Y1.375
N400X-1.375
N410Y-1.375
N420X1.375
N430Y0
N440Z2.0F200                 ;Z movement to zero position
N450G0G90X1.5Y1.5        ;X and Y movement out of the way
N460M2                   ;End Program
```

Appendix B (Lab 3)

Part Drawing

Here is a drawing of the basic block size before machining:

2.12
Sqr.

1.56

(Z0)
Bottom

1.06

(X0,Y0)
Home Pos

1.06

Tolerance All Dim is inches 1 Place +/-0.1 2 Places +/-0.02 3 Places +/-0.005	WPI	Pt.Name LAB III
		Matl. MACHINABLE WAX
	Scale. None	Dwn.by R. COURNOYER

Appendix C (Lab 3)

Canned Drilling Cycles and Miscellaneous Information

From the Bridgeport BOSS5 Operating Manual

(Complete manual on reserve in the Robotics laboratory.)

INSERT CANNED
CYCLE PAGES
HERE (18 pages)

Department of Mechanical Engineering

ME 3820 – Computer-Aided Manufacturing

Instructor: Professor M. S. Fofana   **Term:** C –1999

Office: Washburn Room 311C   **Telephone:** (508) 831-5966

e-mail: msfofana@wpi.edu

Lab Manager: R. Cournoyer   **e-mail**: rjc@wpi.edu

Title: **Basic Part Generation with Pro/Engineer
and Computer Numerical Control Part Programming**

Laboratory Session Number: **ME 3820 – C99 304**
99 – 304

Date Assigned: February 15,1999   **Due: February 26,
1999**

Statement of the Problem:

During Lab 3, the surfaces of several sports related products were scanned and solid surfaces produced within Pro/Engineer's PT/Modeler. The next procedure hypothetically would be to take these surfaces and produce molds from this surface data. This is often a very difficult and tedious process to perform manually by writing an APT (G Code) programs, but Pro/Engineer's manufacturing module can easily accomplish this task. This lab is an exercise to help the student become more familiar with this module and allows a seamless transition from solid geometry to virtual manufacturing to actual part machining.

This lab will consist of several components::

1. Modify the part's size.
2. Create the workpiece (WP) by modifying the finish part.
3. Pro/Manufacture the Block by:
    a) Assemble the Model and Work Piece.
    b) Setup a Machine workcenter.
    c) Setup Manufacturing parameters
    d) Virtual machine the Block until satisfied with the resulting cutter paths.
4. Post the manufacturing information, which will write the "G" code program for this block.
5. Edit the code for Bridgeport controller compatibility.
6. Manufacture a test block from the Pro/Engineer derived code using the Lab's CNC Bridgeport.
7. Write a lab report.
    a) Refer to the syllabus for the correct format.
    b) The lab report must include print outs of the "TAP" and LST files created during the NC Post process)
    c) Include any edits (technical errors) that are found within this tutorial as an appendix to your report.

General Information:

The finish model (the block) was first created using Pro/Engineer's solid geometry sketcher. Before the manufacturing procedure can begin, a Work Piece must be created. The size of the work piece is directly related to the stock size that the finished part will be manufactured from on the shop floor. It is critical that Pro/Engineer has this information to properly calculate the tool paths. Therefore, some research into the available material is necessary prior to building the work piece. Since this lab is a controlled, and the work piece is actually a small casting measuring 2.12 square by 1.56 tall (The research has been done).

There are two thoughts to creating a part's solid geometry within the Pro/Engineer platform. Starting from scratch or modifying existing similar geometry. The later is usually always faster when possible. Therefore, the geometry from the Lab two's block will be modified below to produce the work piece.

Pro/manufacturing creates several new files during the process of operating (i.e. assembly files) and it is imperative that these files all remain within the same directory and user. Furthermore, on a personal level, I can't stress enough on the fact that practice produces Pro/Engineer experts. I would advise you to run through this lab exercise several times to become more familiar with it.

The Pro/Engineer manufacturing information is stored within the work piece, and can't be reused without deleting all this new data. This can be easily accomplished by opening the work piece, and using the modify   delete command to remove it. Then accomplish a "save as" procedure calling the new file name (Some name_WP) and go through the exercise again.

Individual Group's Sizes

Group Number	Existing Pro/E Size	Modified Size
1.	2.00	1.95 (One Side)
2.	2.00	1.95 (Both Sides)
3.	0.50	0.45
4.	0.63	0.68
5.	0.50	0.55
6.	2.00 & 0.50	1.95 (One Side) & 0.55

Questions (Incorporate the answers in the report)
1. What are the advantages to virtual manufacturing?
2. What changes in computers have made virtual manufacturing possible?
3. What other virtual manufacturing programs are available on the commercial market?

4. What changes would your group have wanted in the Pro/Engineer produced cutter paths, and why?

5. What is does APT stand for? (other than a programming language)

6. What is the theoretical (undeformed) chip thickness at the present speeds and feeds? (Ref: 0.750 2 flute HSS end mill, 10.0 in per minute, 1500 rpm)?

The Block's dimensions are modified, and Work Piece is created here.

- File   Open (Locate the BLOCK1 within disk6/independent/parts directory). Double click on the part from the "File Open" window to open it.
- Save as. Save the block as "Block"
- Open. Locate BLOCK in your directory, and open it.

- Feature   Modify. Click on the blocks geometry and modify the necessary dimensions to your group individule sizes listed above.
- Regenerate the part.
- Save it.

Now the finish part will be modified to make the workpiece (WP),

- Feature   Delete   Pick. Pick the last protrusion from the model tree. If you have closed the model tree it can be retrieved by clicking on the Utilities   Environment menu. Place a check on the Model Tree option, then click on Apply, followed by OK.
- Done. The round protrusion should now be deleted.
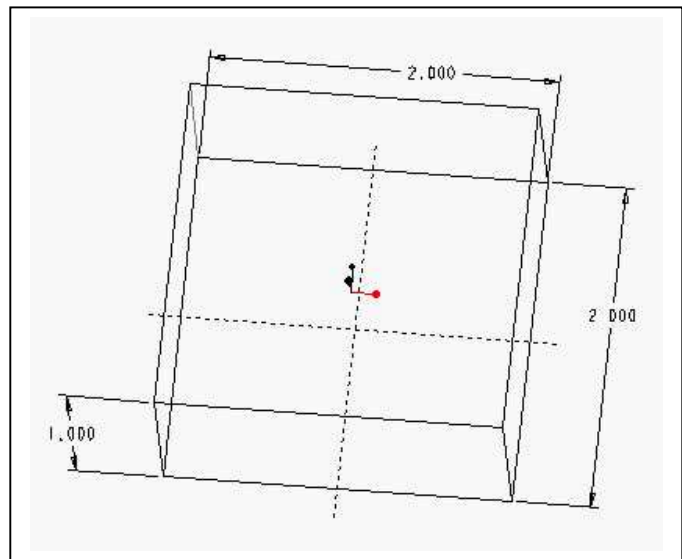
Next the square block's size will be modified to suit the   **Figure 20**
needs of the work piece.
- Part   Modify. Pick the protrusion from the model tree. The screen should look similar to Figure 20. Pick the 2.00 in dimensions and change it to 2.18. Next change the 1.00 to 1.56.
- Regenerate   Done.

This NEW part must be saved as a NEW name.
- File   Save as. (or choose the save as icon)

- Model Name [this should have the EXISTING part name here]
- New name   [Type "BLOCK_WP"]

This completes the work piece preparation portion.

Pro/Manufacturing

Background:

PT/Products manufacturing applications provide a single-source solution to a company's design and manufacturing requirements. Full associativity between design and manufacturing disciplines means no more translations from one database to another and eliminates concerns about lost or compromised geometry, data integrity is ensured. This dynamic concurrent engineering environment permits manufacturing engineers to influence design decisions early in the development process when their input can dramatically reduce cost.

Two manufacturing software modules, PT/Mill and PT/Turn, provide the tools to define the manufacturing processes required to cut parts. These modules use a parametric, feature-based approach to define and sequence the machining processes. As each feature is created, PT/Mill and PT/Turn derive the toolpaths and display the results in the manufacturing model. Total integration with the design cycle allows manufacturing engineers and designers to work in a concurrent environment where model changes automatically update the manufacturing toolpaths.

Virtual Manufacturing using Pro/Engineer's PT/Mill:

- Open Pro/Engineer.
- File   New (Or click on the "New" icon). Select manufacturing in the "Type" column, and check to see that NC Part is selected in the Sub-Type column. Enter the name "Block." Next, an "Open" part window will appear. Double click on the block part to open it. (The name might be different, depending what you called it in lab 2.)

The Part and Work Piece need to be assembled first before proceeding. From the Manufacture menu choose:

- Mfg Model   Assemble   Workpiece. Another "Open" window appears. Locate and double click on the Work Piece part (ref: BLOCK_WP).

The finish block and the work piece will both appear in the active window. A Component Placement Window also will open. The block and the work piece need to be assembled together in the same position that would appear in the CNC machine. The are several constraints that would work for this example, (Mate offset, Align offset) however what will be used here is a Coord Sys. Constraint.

- Constraint Type   Coord Sys   Pick. Choose the coordinate systems in the block and the work piece (in any order). Note to pick a Coord Sys, you must click on the name (i.e. default) and not on the axes themselves.
- OK   Done/Return

Now that the initial part setup is complete, the next component is the actual machining of the part. This area will require several preliminary definitions to be selected before a tool path can be generated.

- Machining Define Oper. (Workcell, Machine Csys, Activate should be checked)
- Done Oper.

The first parameter to set is a Workcell (i.e. Bridgeport)

- Create (From the WORKCELL menu)   Mill   3 Axis   Done Done (yes, twice).

The second parameter is to pick the coordinate system for the Bridgeport. There is only one on the block, and if we were not happy with it (i.e. "Z" was on the wrong plane) another could be created now. However, since the this part was created with good design intent (mine) the coordinate system is adequate for our use.

- Select   Pick. Click on the one cood sys present.

This finishes the "Define Oper" part.

- Done Oper.

Next we need to define the "Type" of machining process that needs to be done. This perhaps is the most difficult part, because if the incorrect process is chosen it is impossible for Pro/Engineer to accomplish this task because of the parameters. In this manufacturing exercise we will be using Face for the first sequence, Volume for the second, and profile for the third.

- Face   Done.

- Seq Setup (Tool, Parameters, Retract, and Surfaces, should be checked) Done.

The Tool Setup Window appears. After you have manufactured several parts you will have built up a library of tools, and will not have to make them each time; however, it is important to save the tool upon exiting Pro/Engineer. Set the following parameters:

- Cutter_Diam (0.750)

- Length (3.0)

- Tool_ID (FEM_0_750) FEM stands for Flat End Mill. This is an optional piece of information that will prove useful for future manufacturing projects. This column sorts the tools, and if there is a large collection of tools with only nonsense information, you end up redesigning tools uselessly, whereas a few minutes of definition can save hours of confusion.

- Apply   File   Done

- MFG PARAMS   Set. A Param Tree opens. Not all parameters need to be set, only the ones with "-1" in the Right column. Adjust the information to read:

CUT_FEED: 100 (This represents 10 inches a minute for our Bridgeport)

STEP_DEPTH: 0.06 (The amount of material that the machine will take on each pass, here it also represents the material that is left on the top of the block (Ref: 1.56-1.50)

STEP_OVER: 0.6 (This isn't required for our Facing operation, but Pro/Engineer will not allow us to leave it blank.) It is however, the amount that the cutter would theoretically move over if a larger face was selected.

Figure 21

SPINDLE_SPEED: 1500 (More useless information, which can't be used on our
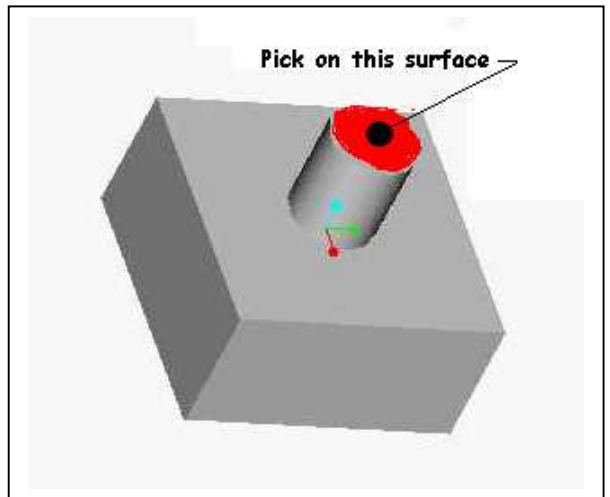
Bridgeport controller, and again can't be left blank.)

CLEAR_DIST: 0.1 (This will be used to control the rapid approach to the piece.)

- File   Exit. Yes, exit. Notice there is no "Done" sequence. One of the problems with Pro/Engineer Ver 20 is the inconsistency between menus.

- Done (Form the MFG PARAMS menu).

The retrace selection menu appears. Normally we would want the cutter to retract along the Z-axis; however, if the cutter was in a pocket we might want it to move in the X or Y-axis first, and Pro/Engineer gives us this option.

- Along Z Axis   Enter Z Depth. Enter 2.0 here. This option actually creates a offset datum, and since the bottom of work piece is "Z" zero it will create a datum 0.4 off the top face.

The next prompt is for the surface that we want to Face.

- Model   Done

- SELECT SRFS   Seq Setup   Add   Pick. This is a critical part for Pro/Engineer. The correct surface MUST be chosen. Refer to Figure 21.

- Done Sel   Done/Return

- Play Path   Done ("Done" may take a few seconds to appear because Pro/Engineer is calculating the cutter paths). Changing the Time Increment to something other than 1.0 can slow the speed of the cutter down. (Click on Enter, then renter the new time.)

- NC Check   Run. Finally the virtual manufacturing process. There are four colors during this process. Green (extra stock), Yellow, (in-process extra stock), Magenta (Finish surfaces), Blue (Gouge Surfaces). We should not see ANY blue surfaces here, if there is some present, go back and recheck your work.

- Done Seq.

This completes the first sequence.

Let's begin a new sequence. This one will machine the top of the block and leave the round portion (hopefully). Pro/Engineer does not contain enough information to

successfully machine this top portion of the block (it will leave material in the four corners). To eliminate this problem, a volume window will be created that is larger than the work piece to force the cutter to remove all the material that is required. Unfortunately there is (at present) no easier method to accomplish this task.

- NC Sequence   New sequence.
- Volume   Done.
- SEQ SETUP   (Parameters, and Volume should be checked) Done.
- Use Prev (since we are using the same tool, most of the parameters are the same for this sequence, with a minor adjustment or two.
- NC SEQ List. Pick the Face Mill sequence (it's the only chose).
- Set. Here we make the necessary adjustments.

    STEP_DEPTH: 0.25
    SCAN_TYPE: TYPE_SPIRAL. Click on the down arrow from the Input
                            bar to view the options, and select SPRIAL.
    ROUGH_OPTION: PROF_ONLY
- File   Exit
- Done (Form the MFG PARAMS menu).
- DEFINE VOL   Create Vol. Name it "Top_Surface"
- CREATE VOL   Sketch
- SOLID OPTS   Extrude   Solid   Done.
- ATRIBUTES   One Side   Done.
- SETUP SK PLANE   Setup New   Plane   Pick. The prompt is asking for us to select a

                    SKETCHING PLANE. Another critical surface. What is required
                    here is the top surface of the WORK PIECE, not the MODEL, and
                    opposite of DTM3 to be selected. A QUERY SEL is the best
                    choice, keep clicking on NEXT till the correct surface is
                    highlighted, and accept it when it appears.
- DIRRECTION The arrow MUST point INTO the part, select Okay when the

direction is correct.

- SKET VIEW   Top. . The prompt is asking for us to select a horizontal or vertical reference

   for sketching. Choose any of the sides for this reference.

Close the INTENT MANAGER is it appears. The 2D sketcher in now active. Several items need to be defined here. The first will be a rectangle that is larger than the work piece, next the finished surfaces of the model will be chosen to complete this window. Finally, after a successful regeneration, a depth will be assigned to the part. This dept might be different if your group was assigned a different size.

- SKTECHER   Sketch   Rectangle. Using the rectangle manager, draw a square around the block that is about a quarter of an inch larger. (Remember if it is drawn as a square, only one dimension will be required)
- Sketch   Line   Centerline. Draw them both vertically and horizontally.
- Alignment. Align the centerlines to their appropriate datums.
- Geom Tools   Use Edge   Sel Edge. Select the circle (both halves) that is present in the middle of the block (ref: Round post)
- Done Sel   Done/Return.
- Dimension. Place a dimension on one side of the square.
- Regenerate. If the error is "Under dimensions part", place a second dimension on an adjacent side, and regenerate.
- Modify. Change the dimension of the square to 2.75.
- Regenerate   Done
- Blind   Done. When prompted enter the correct depth. Normally it will be 0.50 inches.
- OK From the PROTRUSION: Extrude window.
- CREATE VOL   Done/Return. Pro/Engineer will now create the cutter paths for this sequence. Depending on processor speed this may take a few minutes.

149

- Play Path   Done.
- NC Check.   Run. Move or rotate the part to gain a top view of the part, and virtual machine it, then remachine it from a side view to completely check for gouges.   If you want to run this NC Check again, select Done/Return and check the Refresh box after running.
- Done Seq.
- Matrl Remove. Select the Volume sequence   Automatic   Done.

This completes two thirds of the machine setup, leaving only the perimeter. Start the final sequence.

- NC Sequence   New Sequence.
- MACH AUX   Profile   Done.
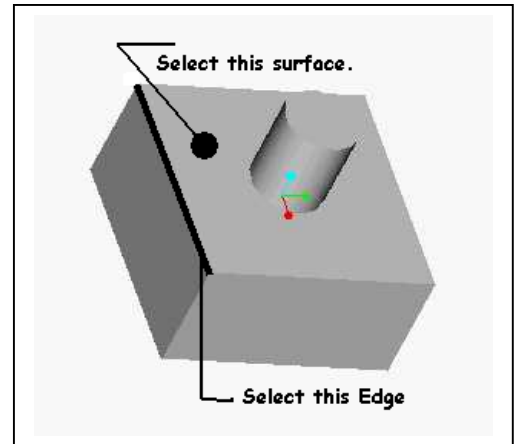
Figure 22

- SEQ SETUP (Again accept the defaults)   Done.
- MFG PARAMS   Use Prev   Pick #2 (Volume). No changes are necessary.   DONE.
- SURF PICK   Model   Done.
- SURF/LOOP   Loop. We are prompted to pick a face (surface). Pick the top of the block adjacent to the round post. Next we are prompted to select an edge. See Figure 22.
- Done Sel   Done   Done/Return   Done/Return.
- Play Path   Done.
- NC Check   Run.
- Done Seq.
- Matrl Remove. Select the Profile sequence   Automatic   Done.

- File   Save (Enter).

150

This completes machining portion of the Pro/Manufacturing exercise. Before the file is
sent to a NCPost and converted into G code, the entire operation should be run to
ensure a smooth cutter path. To accomplish this task, follow the next set of
exercises.

- CLData   Output   Select One   Operation.
- SEL MENU   OP010
- PATH   File
- OUTPUT TYPE   (Accept defaults) Done.
- Save As   New Name (Enter "BLOCK010") Enter.
- PATH   Display   Done (Ref: Change the time increment id the cutter is moving to
  fast).

Once the cutter path has been displayed the NC Check can be run to check for additional
gouging.

- CL CONTROL   Done.
- PATH   Done Output.
- CL DATA   NC Check   Display   Run. The OPEN window appears, locate the
  "BLOCK010.ncl" file and double click on it. The entire Pro/Manufacturing process
  should run, and no BLUE surface should appear. When satisfied with the
  manufacturing process, the operation is ready to Post.
- NC CHECK   Done/Return
- CL DATA   Output   Operation   SEL MENU   OP010.
- PATH   File.
- OUTPUT TYPE   (Check the MCD File) Done.
- Save As   New Name (Save it as "Mach_Block_010").
- PP OPTIONS (Take the defaults) Done.
- PP List   Mill302. A Pro/NCPOST Generator Window opens, and the NC Control
  Code and APT Input will appear. Any key will close this window, then close the
  INFORMATION WINDOW.

Pro/Engineer will generate a "tap" file containing the G code program within the PTC_Data directory. There are a few edits that are presently required to prepare it for CNC Bridgeport and they are as follows. The newly created "TAP" and "LST" files need to be transferred (Ref: WSFTP from lab 1) to a PC that has a disk drive. These are text files, and can be modified using any word-processor.

TAP File

Items to edit from MILL 302 NCPost (Ver 20)

1. Remove the first 3 lines. (starting with the "%" (rewind) symbol), and last two lines (ending with the "%".

2. Remove the :4 line.

3. Remove the single line containing the M10 (Ref N6).

4. Modify the N5 line to read : N5 G0 M6 T1

5. Remove the G17 line.

6. Modify the G2 or G3 multi-quadrant lines to read G75G2 or G75G3.

7. Change the last line M5 to read M02.

Appendix A contains an unedited and an edited file to help in the transition of data manipulation.

Appendix A (Lab 4)

```
N3   G70 G90                          N48  X.6875
N5   G0 M6 T1                         N49  G75G3 X-.6875 Y0. I0. J0.
N7   X.6875 Y0. Z2.5                  N50  G1 X-.95
N8   Z1.7                             N51  Y-.475
N9   G94 G1 Z1.5 F100.                N52  X-.497
N10  X-.6875                          N53  G75G3 X.497 Y-.475 I0. J0.
N11  Z2.5                             N54  G1 X.95
N12  G0 X-.95 Y-.95                   N55  Y-.95
N13  Z1.6                             N56  X-.95
N14  G1 Z1.25                         N57  X-.6875 Y0.
N15  X.95                             N58  G75G2 X-.6875 Y0. I0. J0.
N16  Y-.475                           N59  G1 Z2.5
N17  X.497                            N60  G0 X-.95 Y-.95
N19  G75G2 X-.497 Y-.475 I0.          N61  Z1.35
J0.                                   N62  G1 Z1.
N20  G1 X-.95                         N63  X.95
N21  Y0.                              N64  Y.95
N22  X-.6875                          N65  X-.95
N23  G75G3 X.6875 Y0. I0. J0.         N66  Y-.95
N24  G1 X.95                          N67  Z2.5
N25  Y.475                            N68  G0 X1.375 Y-1.375
N26  X.497                            N69  Z1.1
N27  G75G3 X-.497 Y.475 I0.           N70  G1 Z.75
J0.                                   N71  X-1.375
N28  G1 X-.95                         N72  Y1.375
N29  Y.95                             N73  X1.375
N30  X.95                             N74  Y-1.375
N31  X.6875 Y0.                       N75  Z.5
N32  G75G2 X.6875 Y0. I0. J0.         N76  X-1.375
N33  G1 Z2.5                          N77  Y1.375
N34  G0 X.95 Y.95                     N78  X1.375
N35  Z1.6                             N79  Y-1.375
N36  G1 Z1.25                         N80  Z.25
N37  X-.95                            N81  X-1.375
N38  Y-.95                            N82  Y1.375
N39  X.95                             N83  X1.375
N40  Y.95                             N84  Y-1.375
N41  Z1.                              N85  Z0.
N42  X-.95                            N86  X-1.375
N43  Y.475                            N87  Y1.375
N44  X-.497                           N88  X1.375
N45  G75G2 X.497 Y.475 I0.            N89  Y-1.375
J0.                                   N90  Z2.5
N46  G1 X.95                          N91  M02
N47  Y0.
```

154

Mach302 Post Unedited

```
%
N1  (PGM, NAME="
BLOCK_STK_ME3820")
N2  (PGM, ID=" ")
N3  G70 G90
:4  M6 T1
N5  G0 B.000 S1000 M3 T1
N6  M10
N7  X.6875 Y0. Z2.5
N8  Z1.7
N9  G94 G1 Z1.5 F100.
N10  X-.6875
N11  Z2.5
N12  G0 X-.95 Y-.95
N13  Z1.6
N14  G1 Z1.25
N15  X.95
N16  Y-.475
N17  X.497
N18  G17
N19  G2 X-.497 Y-.475 I0. J0.
N20  G1 X-.95
N21  Y0.
N22  X-.6875
N23  G3 X.6875 Y0. I0. J0.
N24  G1 X.95
N25  Y.475
N26  X.497
N27  G3 X-.497 Y.475 I0. J0.
N28  G1 X-.95
N29  Y.95
N30  X.95
N31  X.6875 Y0.
N32  G2 X.6875 Y0. I0. J0.
N33  G1 Z2.5
N34  G0 X.95 Y.95
N35  Z1.6
N36  G1 Z1.25
N37  X-.95
N38  Y-.95
N39  X.95
N40  Y.95
N41  Z1.
N42  X-.95
N43  Y.475
N44  X-.497
N45  G2 X.497 Y.475 I0. J0.
N46  G1 X.95
N47  Y0.
N48  X.6875
N49  G3 X-.6875 Y0. I0. J0.
```

```
N50  G1 X-.95
N51  Y-.475
N52  X-.497
N53  G3 X.497 Y-.475 I0. J0.
N54  G1 X.95
N55  Y-.95
N56  X-.95
N57  X-.6875 Y0.
N58  G2 X-.6875 Y0. I0. J0.
N59  G1 Z2.5
N60  G0 X-.95 Y-.95
N61  Z1.35
N62  G1 Z1.
N63  X.95
N64  Y.95
N65  X-.95
N66  Y-.95
N67  Z2.5
N68  G0 X1.375 Y-1.375
N69  Z1.1
N70  G1 Z.75
N71  X-1.375
N72  Y1.375
N73  X1.375
N74  Y-1.375
N75  Z.5
N76  X-1.375
N77  Y1.375
N78  X1.375
N79  Y-1.375
N80  Z.25
N81  X-1.375
N82  Y1.375
N83  X1.375
N84  Y-1.375
N85  Z0.
N86  X-1.375
N87  Y1.375
N88  X1.375
N89  Y-1.375
N90  Z2.5
N91  M5
N92  M30
%
```

Appendix B (Lab 4)

Drawing of the Block

2.00
SQR.

Ø.63

1.00

50

Tolerance All Dim is inches	WPI	Pt.Name LAB 2
1 Place +/-0.1		Matl. MACHINABLE WAX
2 Places +/-0.02		
3 Places +/-0.005	Scale. None	Dwn.by R. COURNOYER

Appendix B

Evaluation Form, Results and Comments for A 1998 Term

ME3820 Lab 1-3 Evaluation Results (A98)

		Excel-lent	Very Good	Good	Fair	Poor	Very Poor
1.	The lab on a whole was:	6	9	1	1	0	0
2.	The contents of the lab section was:	3	9	2	0	0	0
3.	The lab manager's contribution to the course was:	6	5	3	1	0	0
4.	The lab manager's effectiveness in teaching the subject mater was:	4	7	3	1	0	0
5.	Explanations by the lab's manager were:	6	6	1	0	1	0
6.	Lab manager's preparedness for the lab session was:	8	5	0	0	1	0
7.	Quality of problems and questions the lab manager raised was:	4	8	2	1	0	0
8.	The lab manager's enthusiasm was:	4	8	2	1	0	0
9.	The students confidence in the managers knowledge was:	5	7	2	1	0	0
10.	The lab manager's ability to solve unexpected problems was:	6	8	0	0	0	0
11.	Interest level of the lab sections was:	6	5	4	0	0	0
12.	Lab manager's interest in whether students learned was:	4	8	2	1	0	0
13.	Amount you learned from Lab1 was: (Ref Scantool)	5	5	1	1	0	0
14.	Amount you learned from Lab2 was: (Ref G Code)	5	7	1	0	0	2
15.	Amount you learned from Lab3 was: (Ref Pro/Manf)	4	7	1	1	1	1
16.	Relevance and usefulness of Lab 1 was:	5	6	2	2	0	0
17.	Relevance and usefulness of Lab 2 was:	5	9	1	0	0	0
18.	Relevance and usefulness of Lab 3 was:	5	8	2	0	0	0
19.	Coordination between class lecture and Lab1 activity was:	1	3	6	0	3	1
20.	Coordination between class lecture and Lab2 activity was:	1	4	3	2	3	2
21.	Coordination between class lecture and Lab3 activity was:	3	2	5	0	1	2
22.	Complexity of assigned work on Lab 1 was:	3	6	3	3	0	0
23.	Complexity of assigned work on Lab 2 was:	3	8	2	2	0	0
24.	Complexity of assigned work on Lab 3 was:	3	8	2	2	0	0
25.	Clarity of student responsibilities and requirements was:	5	5	2	3	0	1
26.	The intellectual challenge presented during the lab was:	4	6	4	1	0	0

27. On average, how many hours per week were spent on the lab portion of this course:

(0) Under 2	(0) 2-3	(2) 3-4	(4) 4-5
(6) 5-6	(2) 6-7	(1) 8-9	(0) 9 or more.

ME3820 <u>Lab 1-3</u> Evaluation

		Excel-ent	Very Good	Good	Fair	Poor	Very Poor
1.	The lab on a whole was:	O	O	O	O	O	O
2.	The contents of the lab section was:	O	O	O	O	O	O
3.	The lab manager's contribution to the course was:	O	O	O	O	O	O
4.	The lab manager's effectiveness in teaching the subject mater was:	O	O	O	O	O	O
5.	Explanations by the lab's manager were:	O	O	O	O	O	O
6.	Lab manager's preparedness for the lab session was:	O	O	O	O	O	O
7.	Quality of problems and questions the lab manager raised was:	O	O	O	O	O	O
8.	The lab manager's enthusiasm was:	O	O	O	O	O	O
9.	The students confidence in the managers knowledge was:	O	O	O	O	O	O
10.	The lab manager's ability to solve unexpected problems was:	O	O	O	O	O	O
11.	Interest level of the lab sections was:	O	O	O	O	O	O
12.	Lab manager's interest in whether students learned was:	O	O	O	O	O	O
13.	Amount you learned from Lab1 was: (Ref Scantool)	O	O	O	O	O	O
14.	Amount you learned from Lab2 was: (Ref G Code)	O	O	O	O	O	O
15.	Amount you learned from Lab3 was: (Ref Pro/Manf)	O	O	O	O	O	O
16.	Relevance and usefulness of Lab 1 was:	O	O	O	O	O	O
17.	Relevance and usefulness of Lab 2 was:	O	O	O	O	O	O
18.	Relevance and usefulness of Lab 3 was:	O	O	O	O	O	O
19.	Coordination between class lecture and Lab1 activity was:	O	O	O	O	O	O
20.	Coordination between class lecture and Lab2 activity was:	O	O	O	O	O	O
21.	Coordination between class lecture and Lab3 activity was:	O	O	O	O	O	O
22.	Complexity of assigned work on Lab 1 was:	O	O	O	O	O	O
23.	Complexity of assigned work on Lab 2 was:	O	O	O	O	O	O
24.	Complexity of assigned work on Lab 3 was:	O	O	O	O	O	O
25.	Clarity of student responsibilities and requirements was:	O	O	O	O	O	O
26.	The intellectual challenge presented during the lab was:	O	O	O	O	O	O

27. On average, how many hours per week were spent   O Under 2   O 2-3   O 3-4   O 4-5

28. on the lab portion of this course: O 5-6   O6-7   O8-9   O 9 or more.

Student Comments

What segment of this lab contributed most to your learning?

What segment of this lab distracted from your learning?

What suggestions do you have for improving the lab portion of this course?

Comments

What suggestions do you have for improving this course?

1. More Pro/Engineer and Pro-Manufacturing.

2. Make it harder.

3. Make sure the T/A's are more knowledgeable in Pro/Engineer.

4. More coverage of specific topics in lecture.

5. The labs should not lead the students through the labs without explaining the purpose of each step executed.

6. For Pro-Manufacturing and G code programming, more time has to be spent somewhere so someone can get something out of it. For me, it was a waste of time and at most I am at a good point to start learning.

What segment of this class contributed most to your learning?

1. Using Pro/Engineer and Pro-Manufacturing.

2. The lab reports.

3. The Pro/Engineer and Pro-Manufacturing portion was very helpful.

4. The two Pro-Manufacturing labs greatly helped.

5. I wish I could say that I can write G code and that I can use Pro-Manufacturing, but I can't

Appendix C

Evaluation Form, Results and Comments for C 1999 Term

ME3820 Evaluation Results C99
(For Pro/E and Labs only)

		Excel-lent	Very Good	Good	Fair	Poor	Very Poor
1.	The lab on a whole was:	3	11	2	1	0	0
2.	The contents of the lab section was:	3	8	5	1	0	0
3.	The lab manager's contribution to the course was:	10	5	1	0	0	2
4.	The lab manager's effectiveness in teaching the subject mater was:	8	7	2	0	0	2
5.	Explanations by the lab's manager were:	7	5	4	0	0	2
6.	Lab manager's preparedness for the lab session was:	8	4	2	2	0	1
7.	Quality of problems and questions the lab manager raised was:	6	4	3	1	2	1
8.	The lab manager's enthusiasm was:	7	5	5	1	0	1
9.	The students confidence in the managers knowledge was:	10	4	1	2	0	1
10.	The lab manager's ability to solve unexpected problems was:	4	9	2	2	0	1
11.	Interest level of the lab sections was:	1	9	5	1	1	0
12.	Lab manager's interest in whether students learned was:	7	4	3	3	0	1
13.	Amount you learned from Lab2 was: (Ref Scantool)	5	4	8	2	0	0
14.	Amount you learned from Lab3 was: (Ref G Code)	5	7	4	3	0	0
15.	Amount you learned from Lab4 was: (Ref Pro/Manf)	6	7	4	1	0	0
16.	Relevance and usefulness of Lab 2 was:	4	5	5	5	0	0
17.	Relevance and usefulness of Lab 3 was:	6	6	5	1	0	0
18.	Relevance and usefulness of Lab 4 was:	7	5	5	1	0	0
19.	Coordination between class lecture and Lab2 activity was:	2	5	3	3	3	2
20.	Coordination between class lecture and Lab3 activity was:	3	5	6	3	1	2
21.	Coordination between class lecture and Lab4 activity was:	3	3	6	3	1	2
22.	Complexity of assigned work on Lab 2 was:	1	5	9	3	0	0
23.	Complexity of assigned work on Lab 3 was:	1	7	8	2	0	0
24.	Complexity of assigned work on Lab 4 was:	1	7	7	3	0	0
25.	Clarity of student responsibilities and requirements was:	2	8	5	2	1	0
26.	The intellectual challenge presented during the lab was:	3	6	6	3	0	0

27. On average, how many hours per week were spent on the lab portion of this course:

(0) Under 2 (0) 2-3 (2) 3-4 (3) 4-5
(3) 5-6 (5) 6-7 (2) 8-9 (2) 9 or more.

ME3820 Evaluation
(For Pro/E and Labs only)

		Excel-lent	Very Good	Good	Fair	Poor	Very Poor
1.	The lab on a whole was:	O	O	O	O	O	O
2.	The contents of the lab section was:	O	O	O	O	O	O
3.	The lab manager's contribution to the course was:	O	O	O	O	O	O
4.	The lab manager's effectiveness in teaching the subject mater was:	O	O	O	O	O	O
5.	Explanations by the lab's manager were:	O	O	O	O	O	O
6.	Lab manager's preparedness for the lab session was:	O	O	O	O	O	O
7.	Quality of problems and questions the lab manager raised was:	O	O	O	O	O	O
8.	The lab manager's enthusiasm was:	O	O	O	O	O	O
9.	The students confidence in the managers knowledge was:	O	O	O	O	O	O
10.	The lab manager's ability to solve unexpected problems was:	O	O	O	O	O	O
11.	Interest level of the lab sections was:	O	O	O	O	O	O
12.	Lab manager's interest in whether students learned was:	O	O	O	O	O	O
13.	Amount you learned from Lab2 was: (Ref Scantool)	O	O	O	O	O	O
14.	Amount you learned from Lab3 was: (Ref G Code)	O	O	O	O	O	O
15.	Amount you learned from Lab4 was: (Ref Pro/Manf)	O	O	O	O	O	O
16.	Relevance and usefulness of Lab 2 was:	O	O	O	O	O	O
17.	Relevance and usefulness of Lab 3 was:	O	O	O	O	O	O
18.	Relevance and usefulness of Lab 4 was:	O	O	O	O	O	O
19.	Coordination between class lecture and Lab2 activity was:	O	O	O	O	O	O
20.	Coordination between class lecture and Lab3 activity was:	O	O	O	O	O	O
21.	Coordination between class lecture and Lab4 activity was:	O	O	O	O	O	O
22.	Complexity of assigned work on Lab 2 was:	O	O	O	O	O	O
23.	Complexity of assigned work on Lab 3 was:	O	O	O	O	O	O
24.	Complexity of assigned work on Lab 4 was:	O	O	O	O	O	O
25.	Clarity of student responsibilities and requirements was:	O	O	O	O	O	O
26.	The intellectual challenge presented during the lab was:	O	O	O	O	O	O

27. On average, how many hours per week were spent
on the lab portion of this course:

O Under 2 O 2-3   O 3-4   O 4-5

O 5-6   O6-7   O8-9   O 9 or more.

Student Comments

What segment of this lab contributed most to your learning?

What segment of this lab distracted from your learning?

What suggestions do you have for improving the lab portion of this course?

Overall comments on Pro/Engineer's Manufacturing tutorials and class work?

Comments from the C term evaluation:

What suggestions do you have for improving this course?

1. I think it would be helpful to take Pro/Engineer first, or have help sessions at the beginning of the term.

2. Incorporate one project where you use all of the material learned in the course.

3. In general it was great, only slight improvements in some write-ups are needed.

4. More G code should be written by hand. It would provide the student with better understanding of what Pro-Manufacturing is outputting, and allow students to optimize cutter paths.

5. More time on labs, or more labs.

What segment of this class contributed most to your learning?

1. Hands-on doing the labs.

2. Group project, generating G code and machining.

3. Using Pro-Manufacturing to generate G code, then seeing it work.

4. Pro-Manufacturing.

5. Utilizing Pro/Engineer and Pro-Manufacturing, understanding G code.

6. The hands-on experience.

7. The variety of machines and also using Pro-Manufacturing for the class and the labs.

Overall comments on Pro-Manufacturing tutorials and class work.

1. I thought the tutorials were excellent for learning Pro-Manufacturing.

2. The tutorials aided a lot in my learning.

3. Very useful and helpful.

4. Too much "click by click." As engineering students we should be able to figure out more individually.

5. Very good, the tutorials made the lectures useless.

6. Excellent, but the lecture time could be used more effectively. The book could be used effectively other than homework.

Appendix D

Quizzes

Name _____   Date 1/20/99

ME3820 CAM <small>Spring 99</small>   Quiz 1

Lab Safety and PLC

Fill in the blanks with the appropriate word (s).

Safety

1. What two things are necessary to remove tools from the laboratory?

    A_____,B_____

2. What is the number to call in case of an emergency? _____

3. During off-hours, persons using the equipment in the Robotics laboratory must have

    proven working knowledge of _____.

4. There must be at least two people in the laboratory to work with machine tools,

    robots, welders or any other potentially hazardous industrial machinery. Two people

    are not needed if only _____ are being used.

5. Hearing protection is needed when?_____

PLC

6. As stated in the Lab reading, name one of the four industries where PLC's have been

    implemented. _____

7. Name an advantage of PLC's over both relays and solid state mechanisms.

    _____

8. The PLC stores information using _____ _____.

Name ___**ANSWER GUIDE**___   Date 1/20/99

ME3820 CAM Spring 99   Quiz 1

Lab Safety and PLC

Fill in the blanks with the appropriate word (s).

Safety

1. What two things are necessary to remove tools from the laboratory?

A___ **Ask permission** ____, B____ **Sign Tool Loan Sheet**____

2. What is the number to call in case of an emergency? ____**5555** _____

3. During off-hours, persons using the equipment in the Robotics laboratory must have proven working knowledge of __ **The machine you intend to use** _____.

4. There must be at least two people in the laboratory to work with machine tools, robots, welders or any other potentially hazardous industrial machinery. Two people are not needed if only ____ **Computers** _____ are being used.

5. Hearing protection is needed when?____ **When necessary** _____.

PLC

6. As stated in the Lab reading, name one of the four industries where PLC's have been implemented. _ **Water / Bakery / Packing / Plastic** _____.

7. Name an advantage of PLC's over both relays and solid state mechanisms.

 __**Easy to program / Little hard wiring**_____

8. The PLC stores information using ___ **Binary** _____ ___ **Numbers** _____.

Name _____   Date 1/27/99

ME3820 CAM Spring 99   Quiz 2

CMM & Scan Tool Lab.

Fill in the blanks with the appropriate word (s).

CMM

1. What is the password that is entered at the LOGIN prompt during the startup

    procedure? _____

2. What FUNCTION key do you press to accept parameters? _____.

3. The first step in a measurement routine is to _____the

    _____ _____.

ScanTool Lab

4. What is the name of the conversion program that modifies the data from the CMM

    and converts into the correct format for Pro/E? _____

5. What does CMM stand for? _____ _____

    _____

6. Approximately how many points are considered sufficient? _____

Name ___**ANSWER GUIDE**___

ME3820 CAM Spring 99   Quiz 2

CMM & Scan Tool Lab.

Fill in the blanks with the appropriate word (s).

CMM

1. What is the password that is entered at the LOGIN prompt during the startup

   procedure? ___ **Sams** _____

2. What FUNCTION key do you press to accept parameters? ___ **F10** _____.

3. The first step in a measurement routine is to ___**Calibrate** _____ the

   ____ **Sensor** _____ ___ **Probe** _____.

ScanTool Lab

4. What is the name of the conversion program that modifies the data from the CMM

   and converts into the correct format for Pro/E? ___ **CMM2proe** _____

5. What does CMM stand for? __ **Coordinate** _____ __ **Measuring** _____

   ___ **Machine** ___

6. Approximately how many points are considered sufficient? __ **150 - 400** _____.

Name _____

ME3820 CAM Spring 99   Quiz 3

G Code Programming and CNC Bridgeport

Fill in the blanks with the appropriate word (s).

G Code

1. The main axes on a CNC Milling machine are X,Y, and Z; however, to machine a circular path, the arc center is described using what other letter designator to describe these axes? _____, _____, and _____.

2. A feedrate of 12.5 inches a minute would be entered as a letter code _____followed a numerical value of _____.

3. All G code commands are "modal," explain. _____

_____

CNC Bridgeport

4. The **Bridgeport** is programmed to automatically machine complex workpart shapes using a software language called _____(Hint: not "G" code.)

5. Briefly describe the steps of the Bridgeport "Startup Procedure" _____

_____

Extra Credit: Name 2 "Canned Cycles (G81--G89) and their functions _____

_____(i.e. G80 = Fixed Cycle Cancel).

Name ___**ANSWER GUIDE**___ Date 2/10/99

ME3820 CAM Spring 99   Quiz 3

G Code Programming and CNC Bridgeport

Fill in the blanks with the appropriate word (s).

G Code

1. The main axes on a CNC Milling machine are X,Y, and Z; however, to machine a circular path, the arc center is described using what other letter designator to describe these axes? __ **I** __, __ **J** __, and __ **K** ___.

2. A feedrate of 12.5 inches a minute would be entered as a letter code __ **F** __followed a numerical value of __ **125** _____.

3. All G code commands are "modal," explain. __**Modal means that once the code is turned on, it remains on till canceled** _____

CNC Bridgeport

4. The **Bridgeport** is programmed to automatically machine complex workpart shapes using a software language called ____ **BOSS 5** _____(Hint: not "G" code.)

5. Briefly describe the steps of the Bridgeport "Startup Procedure" _____ **Turn on PC. Power up CNC, Zero X/Y, set TLO, download program** _____

Extra Credit: Name 2 "Canned Cycles (G81--G89) and their functions _____

_____ **See lab manual** _____(i.e. G80 = Fixed Cycle Cancel).

Name _____   Date 2/17/99

ME3820 CAM Spring 99   Quiz 4

Pro/Engineer' s Manufacturing Module

Fill in the blanks with the appropriate word (s).

1. Full _____ between design and manufacturing disciplines means

    no more translations from one database to another, thus eliminating concerns about

    lost of compromised geometry, data integrity is ensured.

2. Pro/Engineer manufacturing module uses a _____, feature-based

    approach to define and sequence the manufacturing a machining process.

3. The best way to describe a 3/4 inch flat bottom end mill within the Tool Setup

    Window would be: _____.

4. The reason why we want to clearly describe the cutter in question 3, is why?

    _____

    _____

5. The *Workpiece* within the manufacturing environment ends with what suffix?

    (a)_____,

    (b) and why? _____ .

Extra Credit: Name the post processor used (within Pro/Engineer) to generate NC tapes

for the Robotics Laboratory's Bridgeport _____.

(Hint: From the menu item "PP List").

Name ___**ANSWER GUIDE**___   Date 2/17/99

ME3820 CAM Spring 99   Quiz 4

Pro/Engineer' s Manufacturing Module

Fill in the blanks with the appropriate word (s).

1. Full __ **Associativity** _____ between design and manufacturing

    disciplines means no more translations from one database to another, thus eliminating

    concerns about lost of compromised geometry, data integrity is ensured.

2. Pro/Engineer manufacturing module uses a __ **Parametric** _____, feature-

    based approach to define and sequence the manufacturing a machining process.

3. The best way to describe a 3/4 inch flat bottom end mill within the Tool Setup

    Window would be: ___ **FEM_0_750** _____.

4. The reason why we want to clearly describe the cutter in question 3, is why?

    ____ **To better understand the tooling library, rather than (T 0001)** _____

    _____

5. The *Workpiece* within the manufacturing environment ends with what suffix?

(a)__ _**WP** _____,

(b) and why? __ **To clarify the tool list**. _____ .

Extra Credit: Name the post processor used (within Pro/Engineer) to generate NC tapes

for the Robotics Laboratory's Bridgeport _____ **MILL 302** _____.

(Hint: From the menu item "PP List").

Appendix E

Electronic CAM Simulation Exams

Worcester Polytechnic Institute
Department of Mechanical Engineering

ME 3820 – Computer-Aided Manufacturing

Test IIIa CAM Simulation

Instructor: Professor M. S. Fofana　　　　**Term:** C –1999

Office: Washburn Room 311C　　　　**Telephone:** (508) 831-5966

e-mail: msfofana@wpi.edu

Lab Manager: R. Cournoyer　　　　**e-mail**: rjc@wpi.edu

Aids Permitted: Open book, notes, tutorials, etc.

Instructions:

Using Pro/Engineer's Manufacturing module, manufacture **one** of the two parts. If more than one part is manufactured, the best graded part will be marked. The test is finished once the "packaged" CL data file is completed.

Critical Information 1

Your Mfg file must be called " TESTIII ", and the CL file containing all necessary NC sequences must also be called "TESTIII". It is also imperative that you save your work several times during the manufacturing process.

Critical Information 2

Prior to starting the manufacturing process, the necessary part (Model) and workpiece MUST be moved from /disk6/independent/parts folder to your individule ptc_data folder.

Hints:

1. If no file name is specified, than the file name is the operator's choice.
2. Don't waste time running NC Check, view the tool path carefully instead.
3. Standard (default) tooling is adequate (i.e. Don't waste time designing tools).
4. Select "Constraint type," "Coord Sys" for ALL part assembly.
5. Don't perform the material removal process at the end of each sequence.
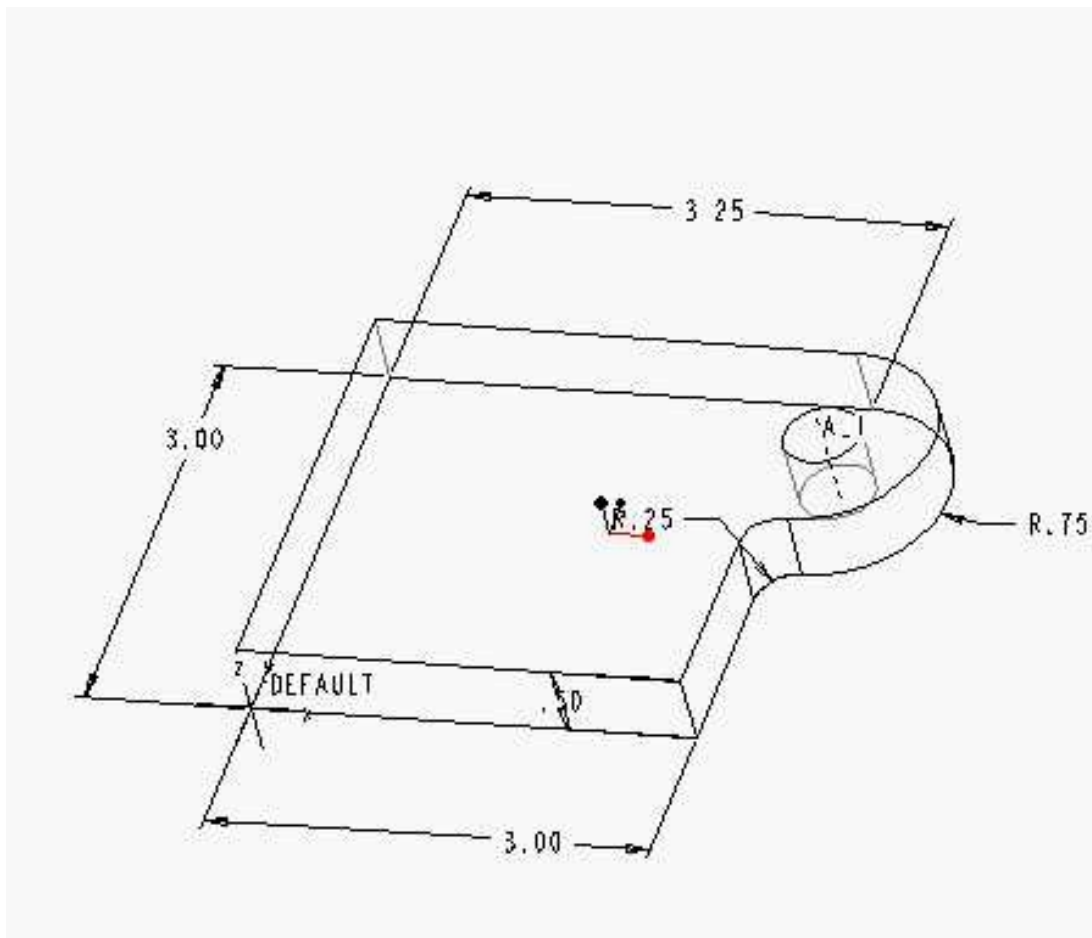
Lathe Part a
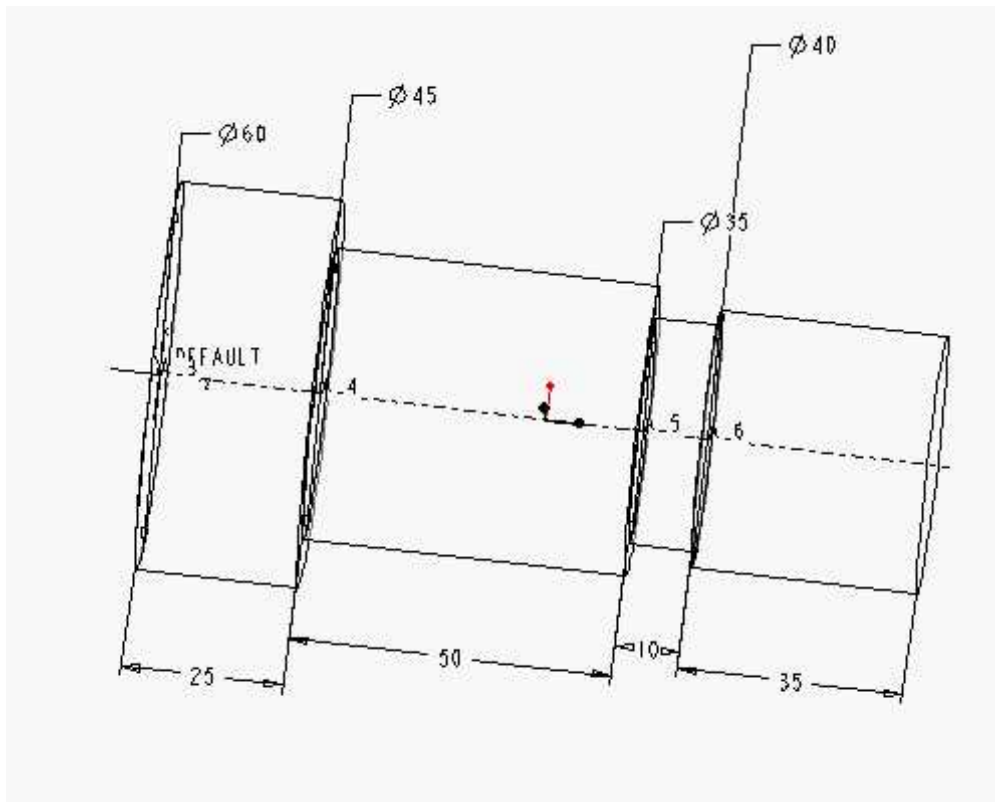
Part name: EXAM_TURN1

Workpiece: EXAM_TURN1_WP

Instructions: Perform two NC sequences on the above part. First sequence, machine the 40 and 45mm diameters. Second sequence, machine the 35 by 10mm groove.

Mill Part a

Part name: EXAM_MILL1

Workpiece: EXAM_MILL1_WP

Instructions: Perform two NC sequences on the above part. First sequence, machine the 1/2 diameter hole. Second sequence, machine the part's profile.

Worcester Polytechnic Institute
Department of Mechanical Engineering

ME 3820 – Computer-Aided Manufacturing

Test IIIb CAM Simulation

Instructor: Professor M. S. Fofana   **Term:** C –1999

Office: Washburn Room 311C   **Telephone:** (508) 831-5966

e-mail: msfofana@wpi.edu

Lab Manager: R. Cournoyer   **e-mail**: rjc@wpi.edu

Aids Permitted: Open book, notes, tutorials, etc.

Instructions:

Using Pro/Engineer's Manufacturing module, manufacture **one** of the two parts. If more than one part is manufactured, the best graded part will be marked. The test is finished once the "packaged" CL data file is completed.

Critical Information 1

Your Mfg file must be called " TESTIII ", and the CL file containing all necessary NC sequences must also be called "TESTIII". It is also imperative that you save your work several times during the manufacturing process.

Critical Information 2

Prior to starting the manufacturing process, the necessary part (Model) and workpiece MUST be moved from /disk6/independent/parts folder to your individule ptc_data folder.

Hints:

1. If no file name is specified, than the file name is the operator's choice.

2. Don't waste time running NC Check, view the tool path carefully instead.

3. Standard (default) tooling is adequate (i.e. Don't waste time designing tools).

4. Select "Constraint type," "Coord Sys" for ALL part assembly.

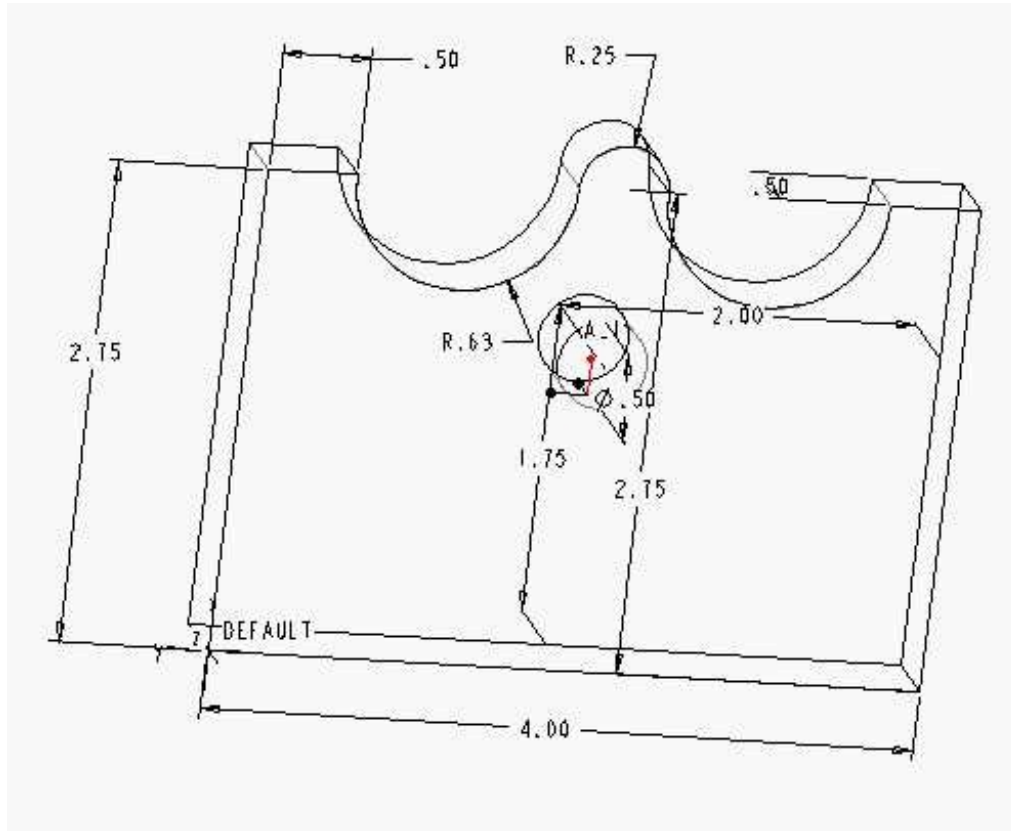5. Don't perform the material removal process at the end of each sequence.

4. Lathe Part b
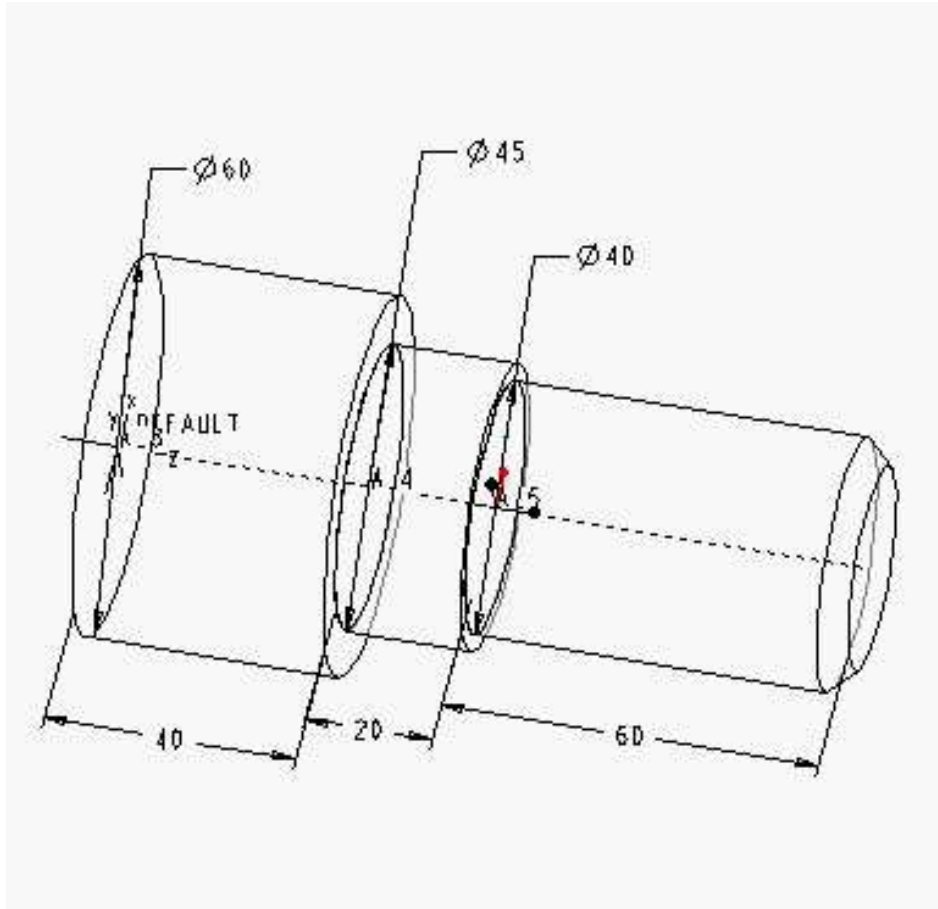
Part name: EXAM_TURN2

Workpiece: EXAM_TURN2_WP

Instructions: Perform two NC sequences on the above part. First sequence, face the front. Second sequence, machine the chamfer, 40, and 45mm diameters

Mill Part b

Part name: EXAM_MILL2

Workpiece: EXAM_MILL2_WP

Instructions: Perform two NC sequences on the above part. First sequence, machine the 1/2 diameter hole. Second sequence, machine the part's profile.