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ABSTRACT

Reverse Engineering: An Evaluation of Contact Technology and Application in Manufacturing Systems.

**by
Shailesh V. Jadhav**

There are many instances where one-of-a-kind parts, such as prototypes or custom-built parts, need to be reproduced. When a Computer Aided Design (CAD) of an existing part is not available, reverse engineering is involved in recreation of such part. Currently, geometric measurements from the surface of a prototype are extracted manually and sent to a CAD system. It is typically time consuming, tedious, and potentially a source of error. This study outlines a methodology, for the development of a CAD model, rapid prototype, and subsequently manufacturing of the part. The method developed uses a contact technique to extract 2-D or 3-D data from the surface of parts using a coordinate measurement machine. Procedures for processing the data into a polygon mesh representation and orthographic projections are detailed. CAD solid modelling to Computer Aided Manufacturing (CAM), which is an important step in manufacturing industries are described by CNC code generations and use of ethernet to send the machine code file to the CNC machine tool on the shopfloor.

**REVERSE ENGINEERING: AN EVALUATION OF CONTACT TECHNOLOGY
AND APPLICATION IN MANUFACTURING SYSTEMS.**

**by
Shailesh V. Jadhav**

**A Thesis
Submitted to the Faculty of
New Jersey Institute of Technology
in Partial Fulfillment of the Requirement for the Degree of
Master of Science**

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*To
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CHAPTER 1

INTRODUCTION

1.1 Introduction to Reverse Engineering

In the normal manufacturing sequence, a part is first designed and then manufactured. Reverse engineering is the process by which an existing part is created or cloned. In order to perform this (within the context of an automated manufacturing environment), a CAD model of the existing part must be drawn or otherwise acquired. Then, the traditional, manufacturing sequence can be followed. The difference in process between the traditional manufacturing sequence and a sequence that uses reverse engineering is shown in Figure 1 (Inglesby, Tom 1991). The traditional manufacturing sequence starts with the designer's conception of functional constraints, performance specifications and associated factors. It ends with a finished batch of the part.

1.2 Need for Reverse Engineering in Manufacturing

There are many instances where one-of-a-kind parts such as prototypes or custom-built parts, need to be reproduced. Moreover, the design of the existing manufactured parts may require periodic modifications to update and improve performance. CAD models of these parts are often not available. However, the creation of a CAD model is desirable and often necessary. The flow chart in Figure 2 shows the reverse engineering within the larger framework.

The other application areas are (Bidanda and Motavalli 1991) as follows:

1) New design: The design process of a new product does not always start from a CAD model. A prototype is often built first. Once the design is approved, measurements are made (either manually or with the use of a contact probe). The

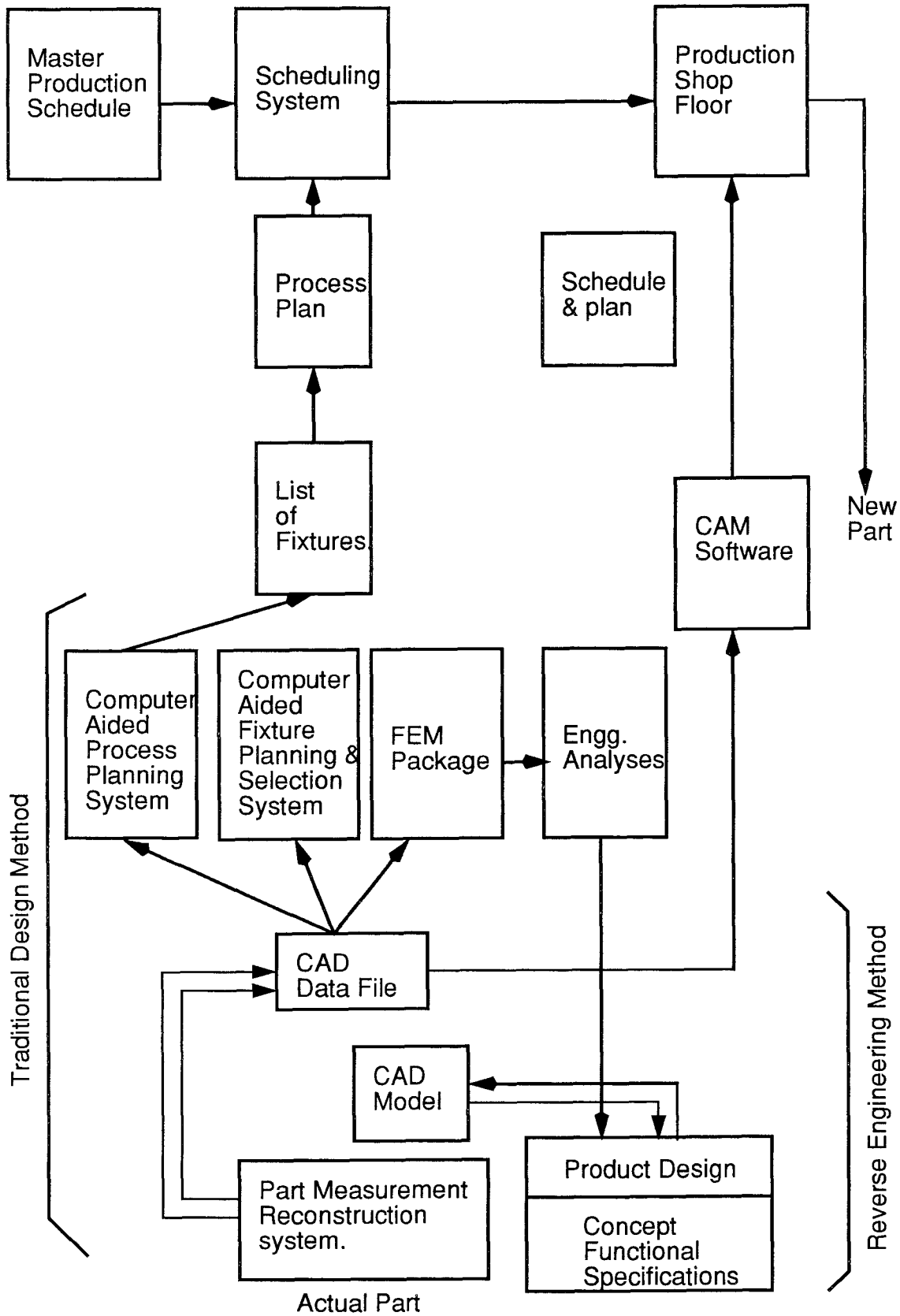


Figure 1 Traditional vs. Reverse Engineering Product Design Sequence

extracted data is then manually entered into a CAE system for further analysis. This process has two disadvantages: it is time consuming and is a potential source of measurement errors.

2) Modify existing design: In some instances the design of an existing product has to be modified. The modification process and design improvements are best performed on a CAD model. However, CAD models of many existing products are not available. Here, a part measurement reconstruction system can play an important role in reducing design time.

3) Design of large items: Precise measurements of large parts are often not possible with traditional metrological equipment. Here, a reverse engineering - based part reconstruction system can help by mapping the part surface as a CAD model. This CAD model can now be scaled and modified as needed.

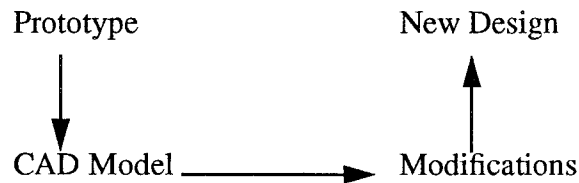
4) Worn or broken parts: When a one-off part breaks or wears out, and the engineering drawing is no longer available, a part measurement reconstruction system can be used to create the CAD model. The CAD model can now be used to manufacture the clone of the worn or broken part.

5) Industrial inspection: When a part is to be compared to its existing CAD model, a reverse engineering system can acquire an actual map of the part surface and deviations, if any, can be identified.

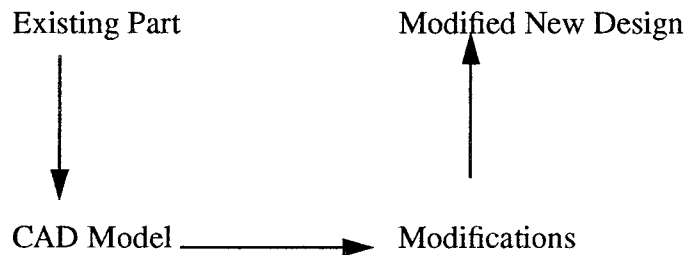
6) Biomedical applications: The crucial bone implants of various types can be manufactured as the original ones by using this technique. In fact, a dedicated system can be developed to make the CAD models and the part subsequently.

7) Non-traditional applications: In production of rings in the jewelry industry or trumpet mouth piece in the musical instrument industry, the master pattern is often handcrafted by highly skilled craftsman from sketches provided by jewelry designers. Any modifications, including just a change in size, requires recrafting of

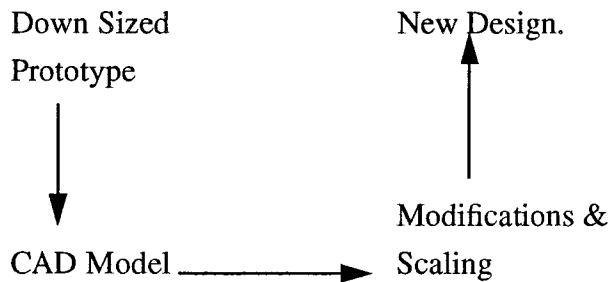
* New Design:



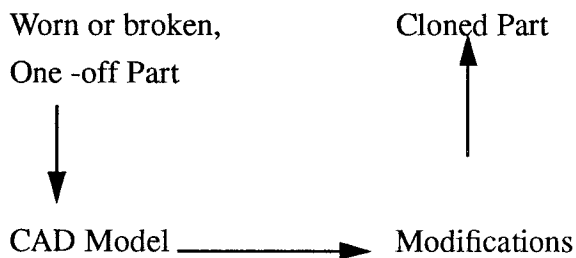
* Modify Existing Design:



* Design of Large Items:



* Worn Parts:



* Inspection:

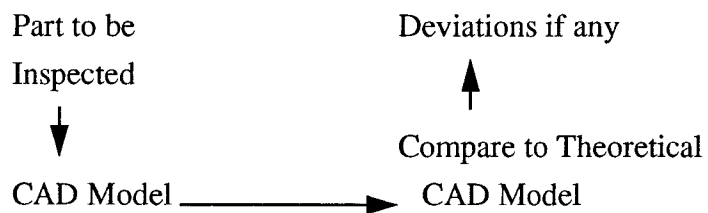


Figure 2 Reverse Engineering Applications in Manufacturing

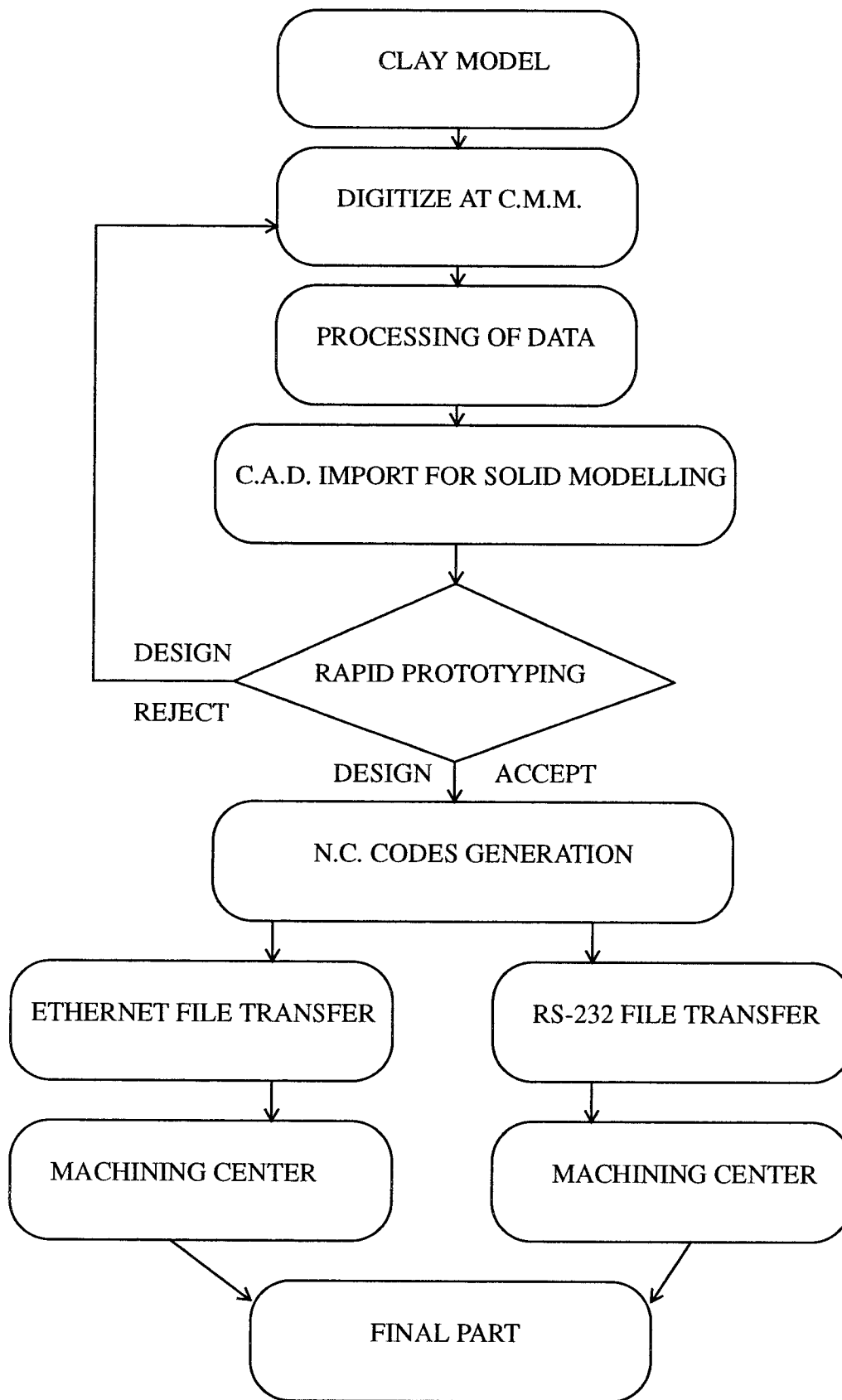


Figure 3 Flow Chart of Reverse Engineering Application at NJIT

the pattern, which is not the best use of the craftsman's skills or time. The use of a part measurement reconstruction system will reduce the time required to apply design modifications and improve manufacturing processes. The first step in this aspect of reverse engineering involves the extraction of geometric data from the surface of existing parts or prototypes to input into a CAD system. The design of the part is then analyzed and improved using the CAD system. The improved design is stored in the CAD database. The geometric information contained in the CAD database, along with other manufacturing information (i.e. materials, roughness etc.), is then used to create a process plan. An NC part program is also created using the part geometry. The NC code, along with the process plan, is then used to manufacture the part.

1.3 Scope and Purpose of This Thesis

The purpose of this work is to apply the reverse engineering techniques at the "Factory of the Future" facility at New Jersey Institute of Technology. This work concentrates on two important areas in modern manufacturing, CAD/CAM. The factory floor equipment integration emphasizes the importance of Computer Aided Engineering in today's rapidly changing environment. Here the machining of a complex geometry part is discussed. The flowchart in Figure 3 shows the concept. Digitized data is extracted from the available part using a coordinate measuring machine. A CAD model is then made by processing the data file. An important breakthrough technology of rapid prototyping is also discussed here. The further processing of the CAD model, which is still a difficult area, is handled and codes for the machining the part are made. Although there are numerous reverse engineering applications, the scope of this work will be restricted to physically producing the part at different levels. The metrological, metallurgical and physical aspects, which are of equal consideration, will not be discussed in this work.

CHAPTER 2

DIGITIZATION ON COORDINATE MEASURING MACHINE

2.1 Introduction

Coordinate measuring machines (CMM) are widely used tools for performing fast accurate three-dimensional measurements of mechanical components. A typical CMM uses highly precise linear transducers and a microcomputer based controller to achieve measurement accuracies of ± 0.005 millimeters (± 0.0002 inch). These devices can simplify otherwise complex three-dimensional inspection tasks. Important applications for CMMs can be realized by interfacing them with CAD software. Connecting the digital measurements capabilities of a CMM with the graphical communications capabilities of a CAD system enables the user to perform reverse engineering. A coordinate measuring machine is designed to record the position of a spherical probe as it is moved along three coordinate axes. The probe is typically placed in tangential contact with a component when a feature is being measured. Various calculations are automatically performed by the machine to determine the relationship between the probe and the actual geometry of the feature.

A typical CMM measurement is shown in Figure 4 (Source: Brown and Sharpe Mfg. Co). A measurement probe makes successive hits to record a particular measurement in the typical fashion as shown in the figure. Note the part being measured is asymmetric in appearance. Accurate measurements of the probe positions are made by the use of linear position transducers (e.g. graduated glass scales) mounted on the machine axes. These measurements are recorded by a microcomputer and are printed, stored, or transmitted to another computer for further processing. Various geometric calculations can be performed using the

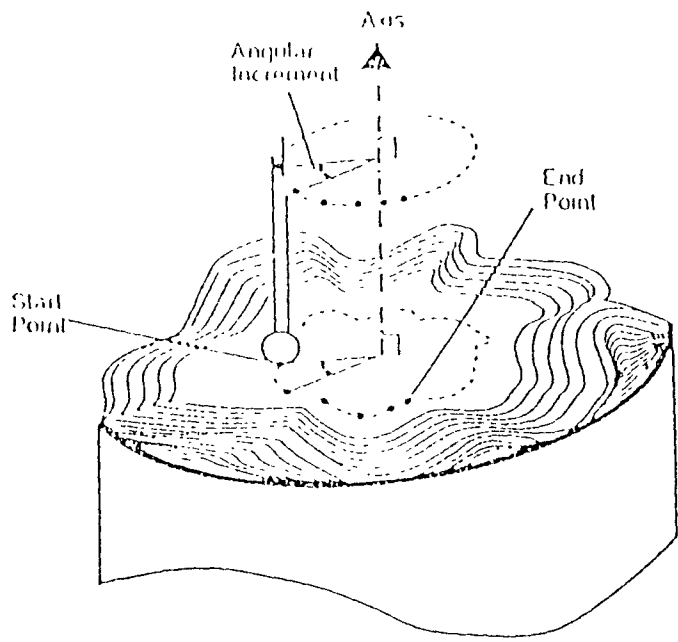


Figure 4 Measurement On a Coordinate Measuring Machine

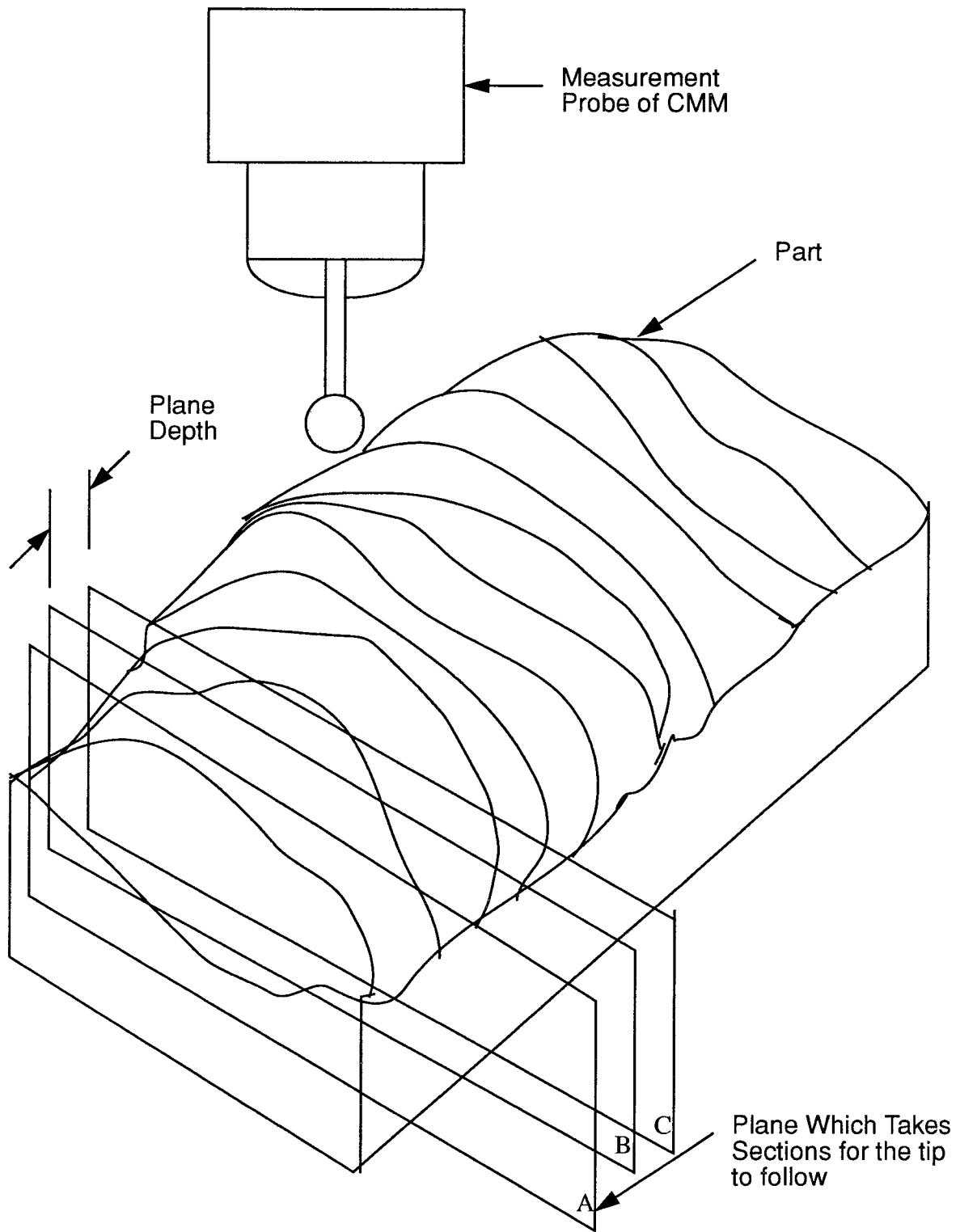


Figure 5 Slicing of a Typical Complex Geometry Part by Planes

measurements such as determining angles, diameters, centers of circle, and form deviations such as roundness or perpendicularity.

2.2 Aquisition of 3-D geometrical data

The Brown and Sharpe coordinate machine was used to provide a fast and accurate way of capturing the required 3-D geometrical data from the part. This allows one to digitize 3-D data using a force sensitive probe. A series of points can be digitized across a non-mathematical contour, and complex curves can be reconstructed from these by concatenation. With a sufficient number of points, a set countours of the part can be developed and an accurate geometrical model reconstructed. The extracted data is in the form of chopped *slices*. Every *slice* is reconstructed in I-DEAS using consturction geometry mode. Thus every *slice* of the part has a profile associated in the I-DEAS.

2.2.1 Advanced Validator Interface Language [AVAIL]

The Brown & Sharpe measuring machine uses a measurement program called Advanced Validator Interface Language (Avail). Avail follows a set of measurement instructions to produce measurements reports of the part to be measured. The Learn mode is used to create the measurement instructions to be used during the execute mode. AVAIL supports the following operational modes: Learn, Execute, Inspect and Append.

Learn Mode: The learn mode is used to create the mesurement insturctions for the part to be measured. One uses the avail menus to create a permanent list of measurement instructions to be associated with the current part. This list of instructions is saved in the current part folder in a file named llf (Learn List File).

Execute Mode: The execute mode is used to mesure previously learned part. The llf generated during the learn mode can be executed for any number of parts

of the same family.

Inspect Mode: The inspect mode is used when you need to check a few dimensions of a part. The inspection mode produces the measurement reports, but does not save the measurement instructions.

Append Mode: This mode allows for the continuation of the learn mode on a different day. It uses the partially learned IIf to add instructions.

Mixing Modes: A powerful feature of AVAIL is mixing of modes. One can jump between learn, execute and edit modes, all in the same session. One can start the learn mode mode and jump to the edit mode. While in the edit mode, one can make modifications in programs, execute the new changes and then continue with the learn mode. Avail uses an english-like language as the programming language. The typical syntax will be as follows.

LABEL = COMMAND/OPTION; ARGUMENT LIST

Avail has many macros for the ease of measurements. Unlike typical NC machines, the Brown and Sharpe CMM with AVAIL, can be used very effectively in measurements. Few of the features are: angularity, coaxiality, concentricity, conicity, cylindricity, parallelism, roundness, straightness etc.

2.2.2 Part Orientation

Part orientation is a very important aspect in the CMM measurements. AVAIL can automatically relate the axes of the part with those of the machine. The user needs to choose and define three part features: a point feature, a plane or line feature; and another plane or line feature. The system uses these features to determine the part's axes and its point of origin. If these features are not measured first, the system cannot align the part. Also to capture the maximum points of an intricate or uncommon geometry part, the part should be oriented such that maximum

surface is accessible for the measuring probe. This step is of very much importance as it gives more true geometry capture in CAD. The various methods of orientation should be checked for these two aspects. The first plane or line (plane are most commonly chosen) is called the level and establishes one of the axes. It also provides the system with the information needed to compensate for any misalignment between the part's reference plane and the table called the axis and establishes the second axis. It also provides the information needed to compensate for misalignment between the axis of the part and the table coordinate system. (The third axis is automatically perpendicular to the level and axis). The point is used as the part's point of origin and is the reference point for all later measurements.

2.2.3 Silcing of Part

The part, to be measured on the CMM, has to be measured in a particular way. The I-DEAS import can be done in two dimensional form only. The part should be sliced by some imaginary planes as shown in the figure 5. Then each plane cut can be addressed by a section. In other words, these sections grouped together form the part. The sections are two dimensional measurements by CMM. The import of this data into I-DEAS generates a series of profiles, each of which represents a corresponding section. These profiles can be skinned using the techniques in solid modelling. This is how a geometry transfer takes place using CMM measurement data.

2.3 Measurement output file - a data set

The reverse engineering aspect we are emphasizing is used to create intricate and complex geometry shapes in CAD. The parts which are handmade by skilled craftsman (e.g. clay model of automobile) have a very unique geometry. These

shapes cannot be made using conventional CAD packages. To capture these shapes, the measurement output file from a CMM is very helpful. This process is also called 'digitization of the part'. The part is measured using usual measuring techniques. This data file serves as an input to the CAD system. Thus, this measurement output file is a very important data set.

2.3.1 CMM output file:

The AVAIL output of the measurement is written to a file. This file contains the X and Y coordinate point positions and has two columns as the output. These two columns are the profile points, which are contours of the sliced part. This file output is designed to have its particular format. Usual measurement output, which has the headers, is eliminated. Thus a file which has only two columns of data points is created. This file is used for further processing. The extremely fine measurement of the given part resulted in 186 points to represent the intricate profile.

2.3.2 Uploading File to SUN Workstation

The Brown and Sharpe coordinate measuring machine has a xenix operating system. A hardware connector was required to convert the machine's port from RS232 to RS422. A convertor card was installed to solve this problem. RS422 communication protocol is required for SUN system compatibility. The coordinate measuring machine was connected via an ethernet node. The wiring was done within the allowable distance limits. Now the coordinate measuring machine could be connected to the SUN system. The uploading of the files could be done by using 'long block kermit' file transfer protocol. This method was used to receive a data file from the PC which controls the coordinate measuring machine.

2.3.3 Compilation to I-DEAS format

The uploaded data file is in table column format of X and Y coordinate positions. This file should be glanced at for possible transfer errors and cannot be imported into I-DEAS as is. To convert the file into an I-DEAS executable format, a C language program was written. The program reads the data file step by step and writes it into an I-DEAS compatible execution file. This file is now called program (.PRG) file. The C program which generates a (.PRG) file, which in turn draws a profile in I-DEAS, is as follows:

```
# CMM to I-DEAS conversion in construction geometry
#Program by Shailesh Jadhav
#include <stdio.h>
#include <string.h>
main ()
{
    FILE *fp,*fopen();
    static float x=0.0,y=0.0,z=0.0;

    fp = fopen("input.in", "r");
    y = 0.1;

    printf("K : !\n");
    printf("K : !\n");
    printf("K : CR\n");
    printf("K : PR\n");
    printf("K : K\n");
    while (1) {
        fscanf(fp, "%f", &x);
        if (x == -999.) break;
        fscanf(fp, "%f", &z);
        printf("K : %6.4f,%6.4f,%6.4f\n", x,y,z);
```

```
}  
printf("K : f\n");  
printf("K : d\n");  
printf("K : \n");  
printf("K : MF\n");  
printf("K : PR\n");  
printf("K : E\n");  
}
```

2.3.4 Program Files

A program file is an external file of I-DEAS commands that can be built from within I-DEAS, or as a text file outside of I-DEAS and executed at a later time. Program and user-defined variables can be included here.

Once created, one can run the program file to repeat the commands recorded in the file. The file performs the commands in sequence. Because the file is saved as a text file, one can edit it with a host text editor like: vi, muccoin etc. At this time the data file from the CMM is run through program file. The data file is compiled via a C-language program. The program reads the data file and writes the (.PRG) file. Program files can be very useful when one uses a long sequence of commands. One can define and manipulate program files in model file menu. The control characters in a program file are shown in the following TABLE 1. The signal characters are added automatically by I-DEAS as a program file is generated.

Table 1 Automatic Signal Characters

Signal Character	Meaning
CL :	Output to the list region
CM :	Output to the menu region
CP :	Output to the prompt region
E :	Logical end-of-file
K :	Normal keyboard input
L :	Screen position for picking or locating
FS :	Start input from form
FA :	Apply form changes
FC :	Cancel form, do not apply changes
FD :	Done with form, apply changes
FR :	Reset form contents

Nesting of Program Files:

The nesting technique of the program file can be used to “call” other program file as part of the command sequence programmed. A typical looping can be shown as:

```

/model_file Program_file create Test
Run program_file_1
/Create
/Profile_from_points
/Key_in_X_Y
/(data set values)
/fit
/done
/dr
/
Manage
/Store

```

Begins 1st program file

```

/Model_file Program_file End
    Run program_file 2
        /Create
        Profile_from_points
        Key_in_X_Y
        /(data set values)
        /fit
        /done
        /dr
        /
        Manage
        /Store

```

Calls 2nd program file

```

/Model_file Program_file End

```

```

    Run program_file 3
        /Create
        Profile_from_points
        Key_in_X_Y
        /(data set values)
        /fit
        /done

```

Calls 3rd program file

```
/dr  
/
```

```
Manage  
/Store
```

```
/Model_file Program_file End  
End
```

Ends program file example

The program file execution can be interrupted many ways. The pause signal (P:) stops the automatic execution of program file commands. It allows one to work interactively before continuing with program file execution.

The program file can also consist of an interactive input signal. The interactive input signal (?) also stops the automatic execution of program file commands. One has to use an editor to enter the interactive input signal. When a program file is run, it stops at the point where the (?) shows up in the program. After one enters the required information, the program file continues automatically.

CHAPTER 3

PART REGENERATION IN COMPUTER AIDED DESIGN

3.1 CAE Package of I-DEAS

I-DEAS is a comprehensive Computer Aided Engineering package (IDEAS -V by SDRC "*User's manual*" 1990-91). It is also a complete package in itself which can be used right from the concept to the final production of the product. In the following are listed some of the major features of I-DEAS:

- (1) Solid Modeling (Geomod)
- (2) Drafting (Geodraw)
- (3) Engineering Analysis (Supertab)
- (4) Graphic Numerical Control (G.N.C.)

The major advantage with I-DEAS is that it is interactive and menu driven . Thus it is very user friendly and convenient. The whole software has been divided into families, such as Solid Modeling or Engineering Analysis. Each family has been further divided into tasks, and each task has its own subdivisions. Here our study will be restricted to the Solid Modeling family. The geometry import and processing steps in I-DEAS is shown in the Figures 14 through 18 in the appendix.

3.1.1 Geometry Modeling Task

Geometry modeling task is used to create the finite element model of the object or to modify an existing finite element model. It is similar to the construction geometry task of the solid modeling family but less powerful. The finite element model of an object can be created two ways;

- (1) Transfer from object modeling: This method is used when the geometry is simple and easily transferable.
- (2) Creation of a new model: This method is used in two cases.

- a) When the geometry is too complex to be transferred from object modeling.
- b) When the geometry is too simple (especially for 2-D modeling).

In most cases, there is a partial transfer of the geometry from object modeling and the rest of the geometry is completed in the geometry modeling task. The major part of the geometry modeling is done using create, wire, copy, orient, modify and delete menus.

3.1.2 Mesh Creation Task

Mesh creation task is used to create mesh areas and mesh volumes, to generate mesh, to create and modify nodes and elements, and to define the material and physical properties of the object.

Creation of Mesh Areas : The geometry created in the geometry modeling task is used to form mesh areas. Formation of mesh area decides the type of element and the type of mesh to be used for mesh generation. There are two types of mesh :

- (1) Manual or Mapped Mesh
- (2) Automatic or Free Mesh

In manual mesh, the user has to define the number and size of the elements along each curve, while in automatic, or free mesh, the user has only to define the size of the element and the I-DEAS package automatically generates the mesh. The automatic mesh can be refined at a particular point or area by giving a new local element size at that particular point. I-DEAS also has the capability of identifying a hole while defining the mesh area. For that, the mesh area is created using auto_create option.

Once the mesh is formed, it can be checked by exploding it, using the explode option to check that the mesh area curves are correct.

Creation of Mesh Volumes : Mesh volumes are needed when we are deal-

ing with 3-D objects. The mesh volume for a particular piece of object is made up of the mesh areas bounding that piece. Thus the mesh volume for a cube would be made up of six mesh areas which are defined by its six faces. Once created, the mesh volumes can be checked by exploding them.

Generation of Mesh: Generation of mesh is done using the generate mesh menu. Nodes and elements are generated on the specified mesh areas and mesh volumes using the generate mesh menu. The generate mesh menu can generate mesh directly from the solid model of the object for simple shapes and thus save the trouble of creating geometry, forming mesh areas and mesh volumes and defining mesh size. Thus, for simple objects, all one has to do is make a solid model of the object in the solid modeling family and then generate the mesh in finite element family just by giving one command.

Nodes and Elements: The nodes and elements are created by generating the mesh. They can also be created independently by creating nodes and then creating elements from nodes. I-DEAS also allows the option of modifying nodes and elements once they have been generated. If the user doesn't like the generated mesh, he can generate a new mesh, but before that, he has to delete all the existing nodes and elements in that particular mesh area or volume.

Material and Physical Properties: At the time of creation of mesh areas, I-DEAS gives the user the option of defining the material and physical properties of the object or accept the default values. Most of the time, user accepts the default values to concentrate more on mesh areas at that time. Default values are actually the values for ordinary steel. There are two separate menus in the mesh creation task that manage the physical and material properties which can be changed anytime by the user.

3.1.3 Boundary Conditions Task

The boundary conditions task in I-DEAS is used to define the constraints and restraints on an object. It is also used to define the magnitude and type of loads acting on an object. Since the object may be under different loading conditions at different times, I-DEAS provides case management. A case set defines one particular condition of loads and restraints on an object, and more than one case set can be defined for an object.

3.1.4 Model Solution Task :

The finite element problem is solved in model solution task. Model solution task has further subtasks :

- (1) Linear Statics Task
- (2) Normal Mode Dynamics Task
- (3) Constraint Mode Dynamics Task
- (4) Heat Transfer Task
- (5) Forces Response Task
- (6) Potential Flow Task

For static loading, linear statics task is used therefore only linear statics task will be discussed in detail here.

Linear Statics Task: Linear statics task is used for solving finite element problems with static loading. First, the user has to specify the case set he wants to use for the solution. Then select the executive options, whether the solution should be in interactive mode or batch mode, etc. Then the user has to select the method of solution correctly. The `Verification_Only` method is used when the user just wants to check for any error in the model. Most of the time, `Solution_No_Restart` method is used . In this method, the solution stops the moment some error is found and thus avoids unnecessary calculations. It is most important, before the

problem is solved, to select the types of output needed. Output_Selection menu is used to select the type of output needed. For example, displacement, stresses, reaction forces, strain energies, etc. Ultimately when all the formalities are done, Solve_Linear_Statics provides the solution. Depending upon the complexity of the assigned problem, I-DEAS might take from one to twenty minutes to get the solution.

3.1.5 Post Processing Task

Once the solution has been prepared in the model solution task, the results are viewed, interpreted and processed in the post processing task. The first step is to go to analysis dataset selection menu to choose the type of result which the user needs to view. Further, depending upon the requirement, there are Contour, Criterion and Deformed Geometry options to view the results. A plot of stresses can be taken using XYZ_Plot menu. The most important option and most commonly used is Contour. It gives the distribution of stresses, forces or strain energies over the selected group of elements and works best to visualize the results.

3.1.6 Adaptive Meshing

Adaptive meshing task is used to get an optimal mesh for getting best results.

Adaptive meshing can be done on the following two basis :

- (1) Elemental Distortion
- (2) Analysis Results

(1) Elemental Distortion :

Sometimes the mesh generated in the mesh creation task has elements with distortion which exceeds the allowable limit. Adaptive meshing technique analysis shows the distortion summary of the existing mesh, locates the elements to be modified and then modifies the mesh to bring the elemental distortion down

to the allowable level.

(2) Analysis Results :

This is the most common use of adaptive meshing to get a better solution. Most of the time, the refinement is done based upon the strain energy distribution. This technique analyzes the strain energy distribution over the object and identifies the regions which need refinement for getting a better solution, and then refines the selected regions. The desired refinement may not take place in a single step, so it is an iterative process and can be continued until acceptable results are obtained.

Method of Modification :

The method of adaptive meshing can be chosen depending upon the type of problem. I-DEAS uses the following methods for adaptive meshing :

- (1) by moving nodes
- (2) by splitting elements
- (3) complete remesh

The first method just shifts the node positions to get optimal mesh and is used only when small changes in mesh are required. The second method splits the current elements to get more elements in the required region to refine the mesh, and the third method generates a completely new mesh. The user can combine these methods to get better results. Most of the time, method one and method two are used in tandem to attain the benefits of both. The second method works only when a surface is attached to the selected mesh area. Because of this, the second method is mostly used when we start with mapped meshing, as mapped meshing generates a surface on the mesh area using Coon's Patch Technique. Complete remesh method is usually used when the user starts with automatic free mesh.

3.2 Iges Output for Codes Generation

The CAD design, prepared with imported data in I-DEAS, needs to be transferred into useful manufacturing information. A CAD model loses its importance unless it is transferred to a CAM software package. The software package in turn converts the geometry into A.P.T. (Automatic Programming Tools) codes or EIA/ISO (Electronic Industries Association/International Standards Organization) format of output. IGES, which is a non-proprietary neutral format for exchanging design data between different CAD systems, is the better (Chatterjee, A., and M. A. Cohen., and W.L. Maxwell, and L.W. Miller 1984) of available methods to pass the part information.

IGES was originally published by the National Bureau of Standards (changed to, The National Institute of Standards and Technology) in 1980. A consortium of government and private sector representatives created the specification as a means of representing geometry and other design data (dimensions, layers, and annotation) in a file format. The IGES exchange format is often a convenient means of exchanging data between CAD system vendor to conceal proprietary information regarding the internal representation of geometry, data base structure, or algorithmic techniques.

An existing drawing on a CAD system can be preprocessed using a software program provided by the CAD system vendor to translate the drawing into an IGES format file. The IGES file can be post-processed by a software program provided on a completely different CAD system to convert it into a drawing. Although there are many cases where conversion problems exist, the neutral file format is useful as a practical means of exchanging design data.

Thus I-DEAS output can be imported into SmartCam for further processing. It can be received into Pro/ENGINEER software for further processing. The IGES output of the profile generated in I-DEAS is shown on the following page.

GEOMETRY CREATED BY SDRC I-DEAS V SOLID MODELING S 1

„4Hnone,9Hfirst.igs, G 1

50HSDRC I-DEAS V: Solid_Modeling, VAX ,TYPE-IGES30,,32,8,24,8,56,9HfiG 2

rst.igs,1.0,2,2HMM,,13H920824.145120,1.0E-03,1.4E+01,,4,; G 3

112 1 1 1 0 0 0 000000000D 1

112 0 4 80 0 0D 2

112 81 1 1 0 0 0 000000000D 3

112 0 4 15 0 0D 4

112 96 1 1 0 0 0 000000000D 5

112 0 4 19 0 0D 6

112 115 1 1 0 0 0 000000000D 7

112 0 4 22 0 0D 8

112 137 1 1 0 0 0 000000000D 9

112 0 4 19 0 0D 10

112 156 1 1 0 0 0 000000000D 11

112 0 4 38 0 0D 12

112, 2, 1, 2, 23, 1P 1

0.0000000E+00, 0.2211945E+00, 0.3003883E+00, 0.3653910E+00, 1P 2

0.4201323E+00, 0.4680489E+00, 0.5097067E+00, 0.5474008E+00, 1P 3

0.5814685E+00, 0.6118678E+00, 0.6408173E+00, 0.6694784E+00, 1P 4

0.6961699E+00, 0.7208970E+00, 0.7446927E+00, 0.7676361E+00, 1P 5

0.7899650E+00, 0.8120545E+00, 0.8336463E+00, 0.8544562E+00, 1P 6

0.8745803E+00, 0.8940842E+00, 0.9141304E+00, 0.1000000E+01, 1P 7

0.1413281E+02,-0.1267573E+00,-0.1521879E+01, 0.0000000E+00, 1P 8

0.3746500E+01,-0.4159723E+01, 0.2172603E+00, 0.0000000E+00, 1P 9

0.0000000E+00, 0.0000000E+00, 0.0000000E+00, 0.0000000E+00, 1P 10

0.1403032E+02,-0.8000165E+00,-0.5218725E+00, 0.0000000E+00, 1P 11

0.2837022E+01,-0.4063612E+01, 0.9040011E-01, 0.0000000E+00, 1P 12

0.0000000E+00, 0.0000000E+00, 0.0000000E+00, 0.0000000E+00, 1P 13

0.1396369E+02,-0.8826556E+00,-0.3622539E+01, 0.0000000E+00, 1P 14

0, 0; 11P 193

S 1G 3D 12P 193 T 1

3.3 SLA Output for Rapid Prototyping

I-DEAS Rapid Prototyping Interface links I-DEAS solid modeling to stereolithography equipment. Stereolithography allows designers to create a preproduction prototype of a new product within hours of completing the design. Typically, the process involves the solidification of liquid using heat from UV light. Geometric information is taken from the I-DEAS Solid Model and transferred to the rapid prototyping system. The first step in creating a prototype of product design is to create a model using I-DEAS solid modeling, then the next steps are:

- * triangulate the facets
- * transfer the model to a Pearl Database.

The rapid prototyping interface produces an output file. This is in particular (.STL) format required by 3D Systems, Inc., the makers of stereolithography apparatus. The objects position could be controlled in the device with the Rapid Prototyping Interface, and selected from a list of supported devices for size considerations. One could add to the list of supported devices as new rapid prototyping devices which support the (.STL) file format become available. The (.STL) file could be transferred to any device the reads in this file format. The (.STL) file is then transferred to the Stereolithography controller computer. This file transfer takes a lot of time. The CAD files are typically big in sizes. In order to send a file, without corrupting it in the transfer, it needs to be split into small sizes. Then the FTP (File Transfer Protocol) is used to send the file to a workstation near the machine. The controller is an IBM-DOS based machine, so again FTP is used to receive the files. The files are then concatenated to reassemble into one.

CHAPTER 4

RAPID PROTOTYPING OF A CAD MODEL

4.1 Stereolithography Definition

Stereolithography (3 D Systems Inc. "*SLA users Guide*" 1991) is a three-dimensional printing process which produces copies of solid or surface models in plastic. The process uses a moving laser beam, directed by the computer, to print or draw cross sections of the model onto the surface of photo-curable liquid plastic.

Using a Stereolithography Apparatus (SLA), solid or surface data is sliced by software into very thin cross sections. A Helium Cadmium (HeCd) laser then generates a small intense spot of ultraviolet (UV) light, which is moved across the top of a vat of liquid photopolymer by a computer-controlled optical scanning system. The laser changes the liquid photopolymer into a solid where it touches, precisely printing each cross section. A vertical elevator system lowers the newly formed layer, while a recoating and leveling system establishes the next layers thickness. Successive cross sections, each of which adheres to the one below it, are built one on top of another to form the part from the bottom up. After the last layer is made, the part is removed from the SLA and flooded with high intensity ultraviolet light to complete the polymerization process. The part can then be finished by sanding, sand blasting, painting or dyeing.

4.2 Applications

Design Engineering:

SLA produces models in much less time than conventional methods. Parts that take days, weeks or months to prototype with present methods can be made in a matter of hours with stereolithography. Conceptual design models are used widely throughout a variety of industries, including: automotive, aerospace, medical,

computer, electronics, and consumer products. By quickly forming three-dimensional conceptual models from design ideas, engineers can evaluate a design, demonstrate its feasibility and sell the concept. "What if" scenarios can also be quickly evaluated to iterate a design toward an optimum. Up-to-date, design review prototypes provided for a quick understanding of the state of the current design, resulting in valuable feedback. The physical model can be used to check the form of a part, such as the new styling for an automobile part, or industrial design of consumer product. Fit can be verified by placing or assembling the prototype with other components in a subassembly, checking clearances, and identifying interference problems. Function tests can range from aerodynamic testing in wind tunnels to using the model in human engineering studies. The impact of changes in other related components can be quickly studied. CAD/CAM/CAE databases, now gaining popularity as the primary definition of the product, can also be verified, since the data is used directly to create the part. In addition, the important and time consuming task of verifying engineering changes can be facilitated.

Manufacturing Engineering Link:

SLA aids in the link between engineering and manufacturing , allowing for optimizing the part for function, while simultaneously reducing manufacturing costs and schedules. Producibility studies can utilize the prototype to provide valuable input earlier in the design process. This could reduce assembly time or eliminate unnecessary tooling or machining. By accurately describing complex geometry, stereolithographic parts can help contract shops bid on jobs and reduce problems with interpreting engineering drawings on the shop floor. Visualizing the final product also assists the design of tooling on a CAM system, as well as aiding in programming toolpaths and optimizing machine set up and tool changes. Models are also used to convey complex design geometry and formulate manufacturing

CAD Process

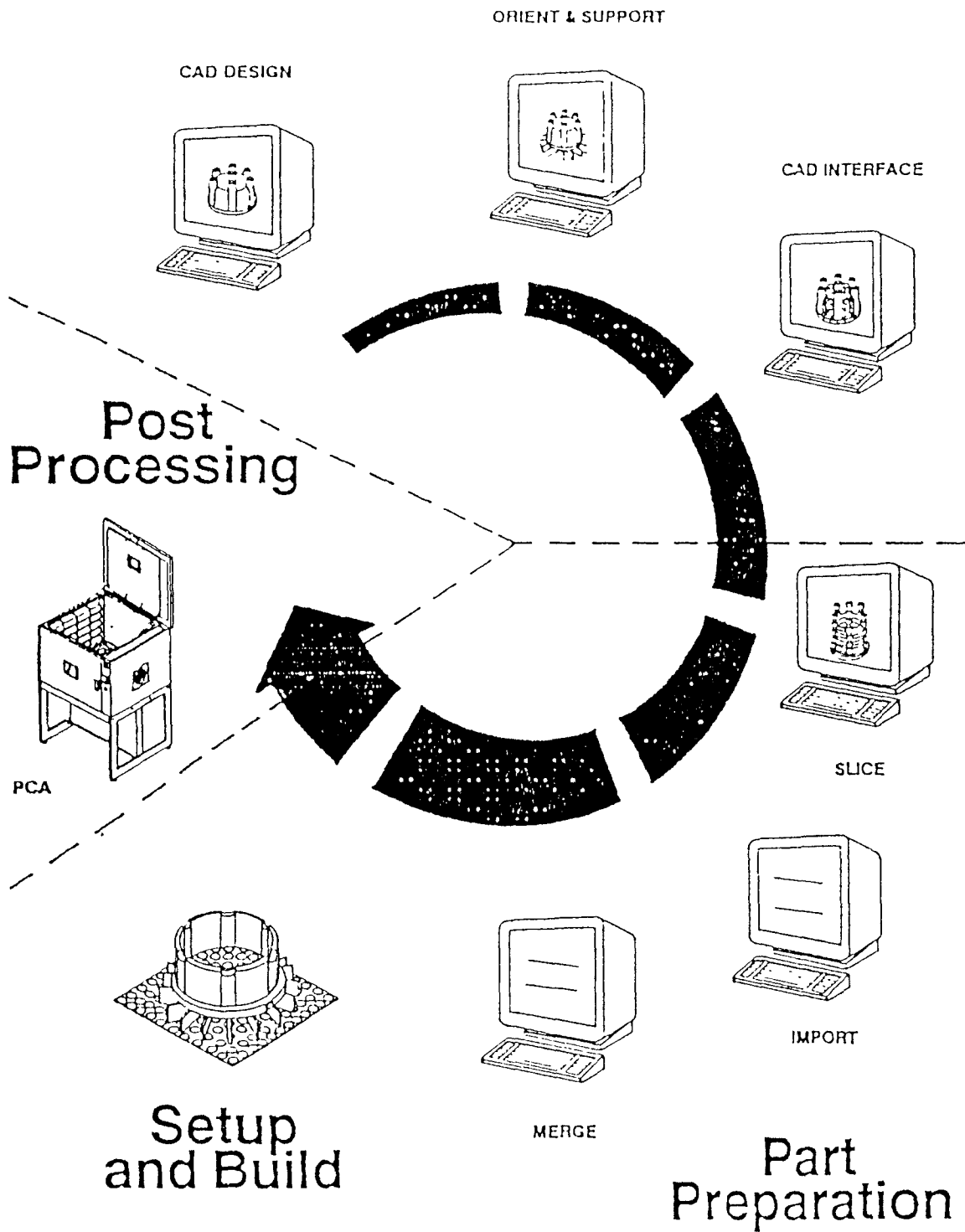


Figure 6 The Stereolithography Process

production processes and tooling. Prototypes, especially with intricate internal detail, are also used as master patterns for castings.

4.3 Benefits

Design Optimization:

One of the benefits of stereolithography is the fast turn-around time for conceptual models, hours instead of the conventional weeks or months. Design review cycles can also be reduced by evaluating the three-dimensional design instead of studying engineering drawings.

Product Quality:

Form, fit, function tests can be performed much earlier in the design cycle with a physical model, reducing the number of costly engineering change orders issued after the product has entered production. This can reduce the amount of lost production time, retooling, rework, and scrap waste incurred in manufacturing. A physical model can reduce errors from incorrect interpretation of the design.

Time to market:

With conceptual three-dimensional models, designs can be finalized much faster. Producibility requirements can be brought into the design process before a commitment to one design is made, reducing manufacturing costs. Utilizing the plastic part, production tooling design times can also be reduced.

4.4 The Stereolithography Process

CAD SYSTEM

Using a CAD system, the first three steps give structure to the object data for use on the SLA. Figure 11 (Source: 3 D systems Inc.) shows the process.

CAD Design: (Step 1) A surface or solid modeling CAD system is made. This data must represent a clearly defined enclosed volume.

Orient and Support: (Step 2) The model is oriented in three -dimensional space for optimum building. A support is then added to hold the part in place while the layers are being built, to maintain layers registration and the structure integrity of the part. (Thin vertical webs usually comprise the supports.) Various help support offers are shown in figure 11 and 12 (Source: 3 D System Inc.).

CAD Interface: (Step 3) The design model and support data are converted to a stereolithography format through an interface supplied by the CAD/CAM/CAE system vendor. This format is similar to a finite element model consisting of triangular surface elements or patches. The user controls how accurately these triangles define the original surfaces. The output from this process is a stereolithography (.STL) file.

SLA: Using the Stereolithography Apparatus, the next six steps process the object and support data and control the part building process.

Slice: (Step 4) Using a sophisticated, proprietary algorithm, each file is sliced into cross sections, typically between 0.005 to 0.020 inch(0.12 to 0.50 mm). The user can select the layer thickness and vary it throughout the part. In addition, the user controls the creation of an internal cross hatch structure between inner and outer surface boundaries (vertical walls). This serves to support the vertical walls and entrap liquid which will later be solidified with a flood of UV light. Horizontal "skins" for top and bottom surfaces are also created. The output is a Slice file (.SLI).

Network: (Step 5) After slicing, each database is transferred to the computer using a local area network. Here we are sending files from SUN to the SLA apparatus lab.

Merge: (Step 6) All the objects (support and part) are combined into one build part database. In this process, the data is reformatted into SLA machine code contained in four separate files: vector files (.V), range file (.R), layer file (.L), machine parameter file

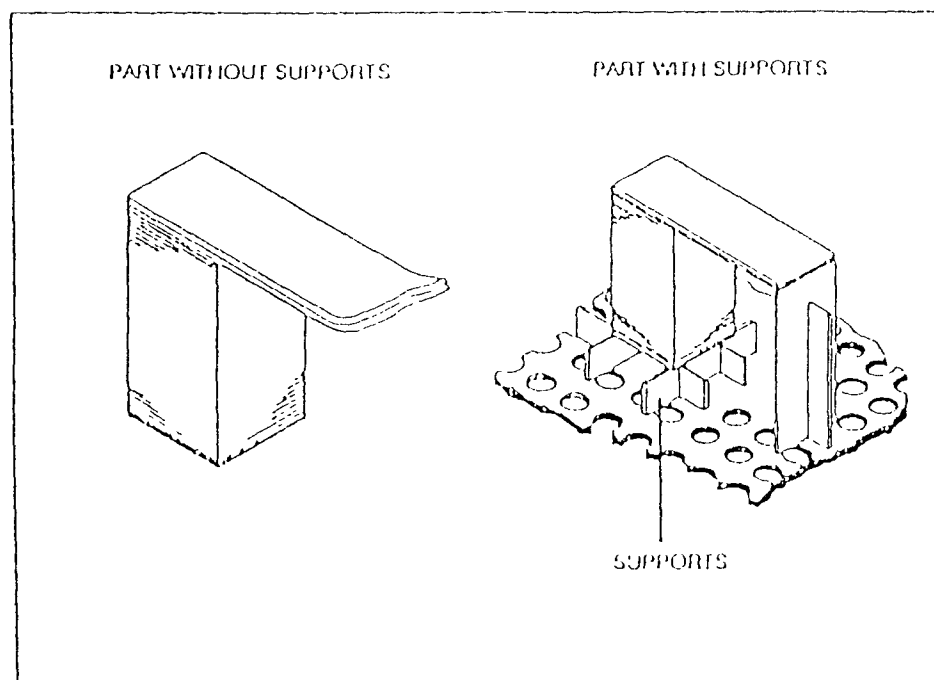


Figure 7' Supports Preventing Deformation and Curl

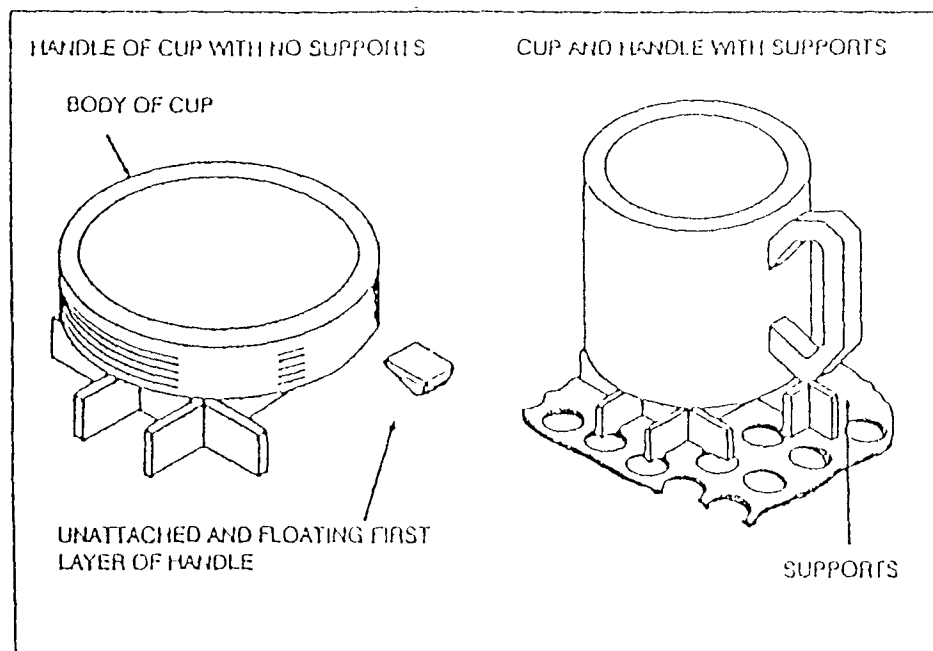


Figure 8 Supports anchoring Unattached Layer Sections

(.PRM).

Utility: (Step 7) The laser power is measured and the SLA is initialized and set up to build parts. A platform is attached and lowered into the vat filled with liquid resin.

Prepare: (Step 8) The users sets building parameters, including the amount of adhesion between layers, and the number and location of parts to be built as well as recoating parameters based on the models's geometry and end use. Each phase of the part building can be customized by varying building parameters within vertical ranges. The parameter setting is important here for the part strength required.

Build: (Step 9) The resin level is adjusted by adding photopolymer. The user activates the Build Software to control the part building process which creates one layer at a time by moving a focused laser beam across the surface of photo curable resin, solidifying the liquid where it strikes. The layer draws the first cross section, which adheres to the platform. When creating each cross section, the inside and exterior boundaries are solidified along with the internal crosshatch. Resin is then trapped between the solid areas, as the object is built up. The elevator dips below the surface, coating the cross section with resin, and a recoater system maintains the correct amount of liquid on top of the three-dimensional object. When the part has been completed, the trapped liquid still requires UV exposure to become solid. The SLA, designed for round-the-clock operation, can be left unattended while the part is building. When the last layer has been printed, the completed part is raised out of the vat. Building times are then logged into a history file and the laser is automatically powered off, if desired, unless an on time is scheduled within four hours.

PCA: The last step uses the Post Curing Apparatus as follows.

Post Process: (Step 10) The part, after building has been completed, requires final ultraviolet curing. First, the part is raised out of the vat and excess liquid is

drained off the part. Next, the remaining excess liquid is removed with solvents, ultrasonics or wicking. The part is then placed, by the user, into the PCA to undergo several exposures. The part is later removed from the platform and the support structure is clipped from the part. Finishing techniques, such as sanding, sandblasting, painting, or dyeing, can then be applied.

CHAPTER 5

CNC MACHINE CODE GENERATION AND TRANSFER

5.1 Smartcam

SmartCam (Point Control Co. "*SmartCam (ver 4.51) Reference Manual*" 1990) is the easiest method for getting part print information into CNC programming code. The different modules of SmartCam working together form an integrated system. It can solve complex part geometry problems and is a powerful system having many functions like roughing, group manipulations, tool path display and plotting. One can either enter dimensions from the part print or transfer data from a CAD system. In our case an IGES file output from I-DEAS CAD package was imported. The job plan file provides important information to the shop floor about the part, tooling, material, and the CNC program itself. Smartcam supports a wide variety of CNC machines, but in our case we modified the postprocessor to suit the Mazak turning machine at NJIT. Job plan module has a built-in speed and feed calculator. The shape module modifies itself for the machine type and tool types one specifies. SmartCam also computes estimated run time as it generates the CNC code. CAM connection translates data from a CAD drawing to a CAM tool path by automatically ordering the CAM data. The shape module will do complex roughing routines for mills, lathes, punch presses and profilers. The system is integrated as a whole.

First step is to prepare a job plan where you make tooling, setup, and process decisions. Once this is done, information for the machine as well as part is available. Next step will be to go to the shape module to describe the part geometry and define the tool path. Once the definition of part is done, one specifies which machine is to produce the part, and the machine appropriate CNC code is generated. Edit Plus menu to make any final changes to the code.

```
Job Plan
Files - access files on disk
New   Job_Info   Tools   CAM_Layers   Print   Files   Leave

Filename: lathe.jsf
Date: 11/19/92
Units: Inch
Machine type: Lathe
Machine: MAZAK CNC TURNING CENTER
Customer: CENTER FOR MANUFACTURING SYSTEMS
Dwg#:
Part: UNDEFINED PART AS PER DRAWINGS
Operation: 1 OF 1
Material Blank: MILD STEEL 4 INCH ROUND
Notes: CLAMP IN THREE JAW CHUCK, EXTEND 6 INCHES.
       Check for drill offsets for compensation.
```

Figure 9 Job Plan Showing the Details of the Machining Data

Job Plan												
Edit - change data for existing tool												
Edit Insert Delete Move Quit												
TL#	Type	ID#	CAD	Z_Set	X_Set	CD	Insert	Rad/Di	LA/WD	Speed	Feed	
1	Face	A1	1	0.0000	0.6250	DL	TNMG433	0.0469	-15.00	275 S	0.010 R	
Comment: IC = .5, Rad = 3/64												
2	Turn	B3	2	0.0000	0.5000	DL	DNMG322	0.0312	-5.000	275 S	0.012 R	
Comment: Rough Turn IC = ..375 Rad = 1/32, LA -5												
3	Drill		3	0.0000	1.0000	L		0.5000	118.00	350 R	0.018 R	
Comment:												
4	DD Grv	O88	4	0.0000	0.2500	DL		0.1000	0.2500	120 R	0.003 R	
Comment: Wd = .25, Rad = .005												

Figure 10 Job Plan Showing the Tool Layout at the CNC

The SmartCam Modules:

Job Plan: The Job Plan module serves several important functions. In addition to documenting the CNC program and job setup for the CNC machine operator, a job plan provides information about cutting tool size, speed, feed, etc. to the shape module. If one is using a CAM connection, Job Plan assigns the tools to be used to cut geometry on separate CAD layers (or by color depending on the CAD system). Job Plan files should be set up in advance to include the common cutting tools that one will use regularly, along with standard CAD non-tool-path layer assignment. A job plan encourages proper planning and documentation by describing the job, material, fixturing, tooling, speeds and feeds, and setup procedures. Figures 13 and 14 show the job plan information.

Shape: Shape transforms part drawing information into CNC toolpath code. To create geometry, simply enter the dimensional information from the drawing into Shape. Shape generates the coordinate data and draws the tool path on the screen as one enters dimensions. One can then manipulate the data graphically with "group" functions such as Copy, Move, Rotate, Image and Scale. One can specify tool path information such as lead-in, tool offset, and depth of cut. Display and use of an optional plotter to plot the tool path to check for interferences is also done. Geometry driven by a particular tool is easy to spot: a new color is assigned to each tool. Both tool and non-tool-path information can be mixed selectively so that you can view the part in different ways to anticipate tool movements and obstructions. Shape's automatic Blend function allows one to indicate the corners one want to blend and the radii. Shape also does the tedious work of figuring arc centers, tangencies, and intersection points. You can program for roughing and threading and watch simulations on the screen. When finished defining the tool path, Shape converts the tool path into the CNC code designed specifically for the machine. As it generates code, Shape reads its own geometry data base and gets tooling

information from the Job Plan file. Shape also incorporates information from the machine file that defines CNC controller, and from the template file for initialization codes, orientation code, and tool change routines. Shape produces a finished CNC program. When it has finished generating code. Shape reports the estimated cycle time. Shape will also accept part program geometry from a CAD system via the optional CAM Connections module. This integration of CAD/CAM system further speeds up the print-to-part process and reduces the potential for errors.

Edit Plus: Edit Plus is a powerful CNC part program editor that includes many valuable tools that make the programmer's job faster. The end result is more accurate programs in less time. The tool-path code generated by Shape can be read into Edit Plus so that you can use its special features, such as arithmetic, search and replace operations, absolute to incremental conversion, and repeat with axis increments. Edit Plus also helps you edit and upgrade existing programs and archive them on floppy disks. When the part program is completed, it can be saved on a disk and a "hard copy" can be printed out.

Utilities:The Communicate module on SmartCam's Utilities menu allows one to either punch a paper tape or send code to and from the CNC machine via RS-232 communication. SmartCam has "Shape Machine Files" (.SMF files) that define the codes and formats for several popular CNC machines. If a machine is not represented among the files already defined, Machine Define on the utilities menu allows one to define their own "post processor" for Shape's code generating function.

Tape-to-Shape:Tape-to-shape will read an existing CNC code file and create a Shape file (.SH2 or .SH3 extension) for use with the SmartCam system. Tape-to-Shape can be described as a "code interpreter" that processes an existing CNC code file and then re-creates the original source file from which an equivalent

CNC code file could be generated.

CAM Connection: The CAD/CAM Standard, CAM Connection is included with the software. CAM Connection reads the CAD drawing files and converts and organizes the information to CAM data that SmartCam can use. The conversion process automatically orders the drawing information into a CAM oriented tool path while assigning tools to selected geometry.

Drafting System:The Drafting System selection on SmartCam's main menu allows to access your CAD system straight from the SmartCam menu. By editing the file XDRAFT.SU, customization is possible.

5.2 Post Processing

The exported geometry in IGES is received in SmartCam by using the cam-connection module. The settings need a change in order to receive the geometry without any restrictions. First job plan is prepared as per the tooling available at the CNC turning center. The job plan is shown in Figure 13 and Figure 14. The imported profiles are shown in Figures 15 and 16. And Figure 17 shows the simulation operation.

Then the profile shape is operated with the tools on the screen. The simulation operation is very important to visualize the tool path. The design and manufacturing engineers can discuss together about optimum toolpath, economics of machining, and quality of products at this stage. This new method reduces time, energy, expensive tooling and rework required at later stages if the product needs rework. The tool path simulation is followed by codes generation. The code file is produced separately as per the machine configuration defined.

5.3 MASP (Manufacturing Automation Software Project)

After the generation of a code file, it needs to be sent to the machine. Various

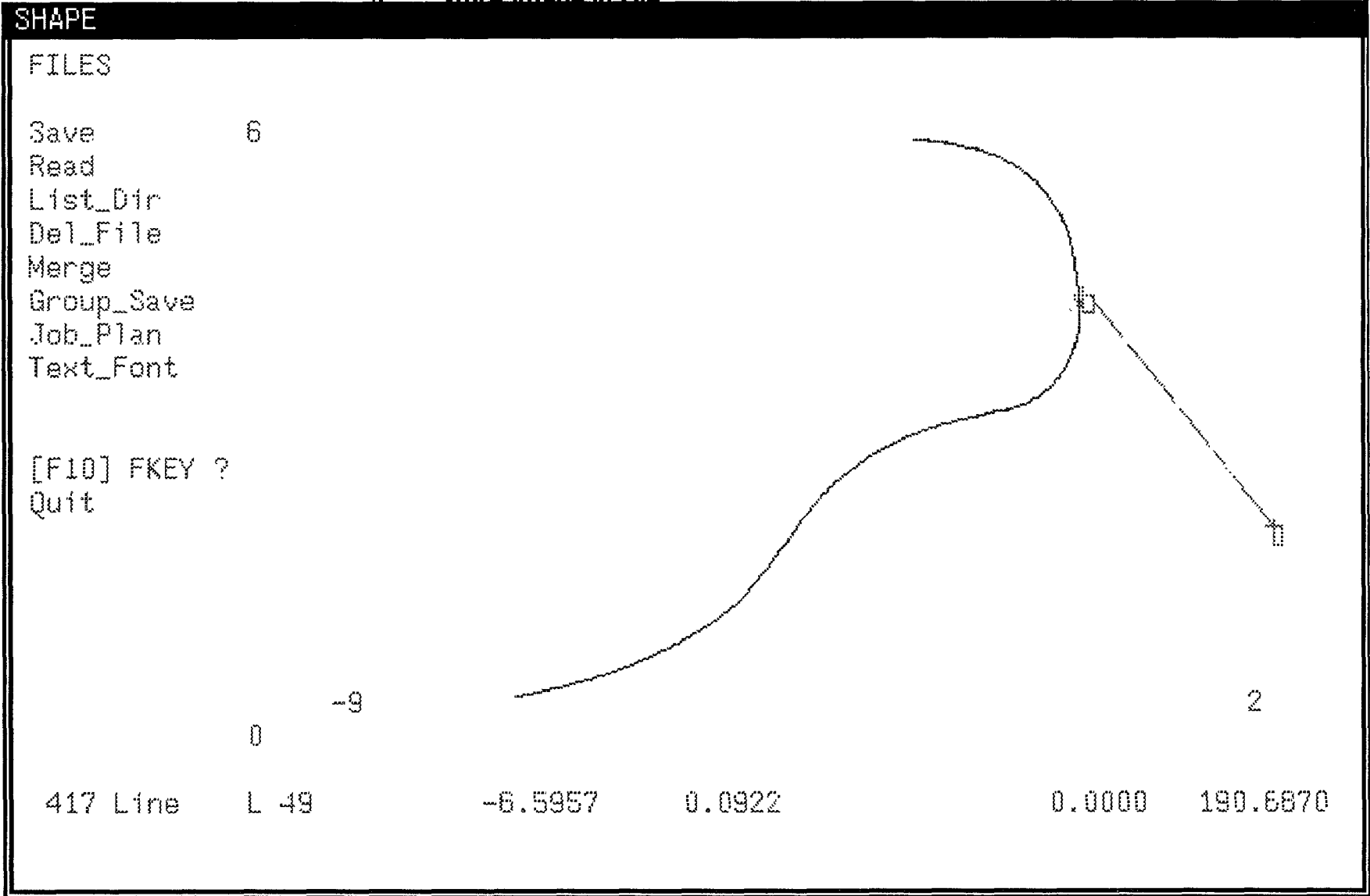


Figure 12 incorrect Way for Cutting Tool Approach

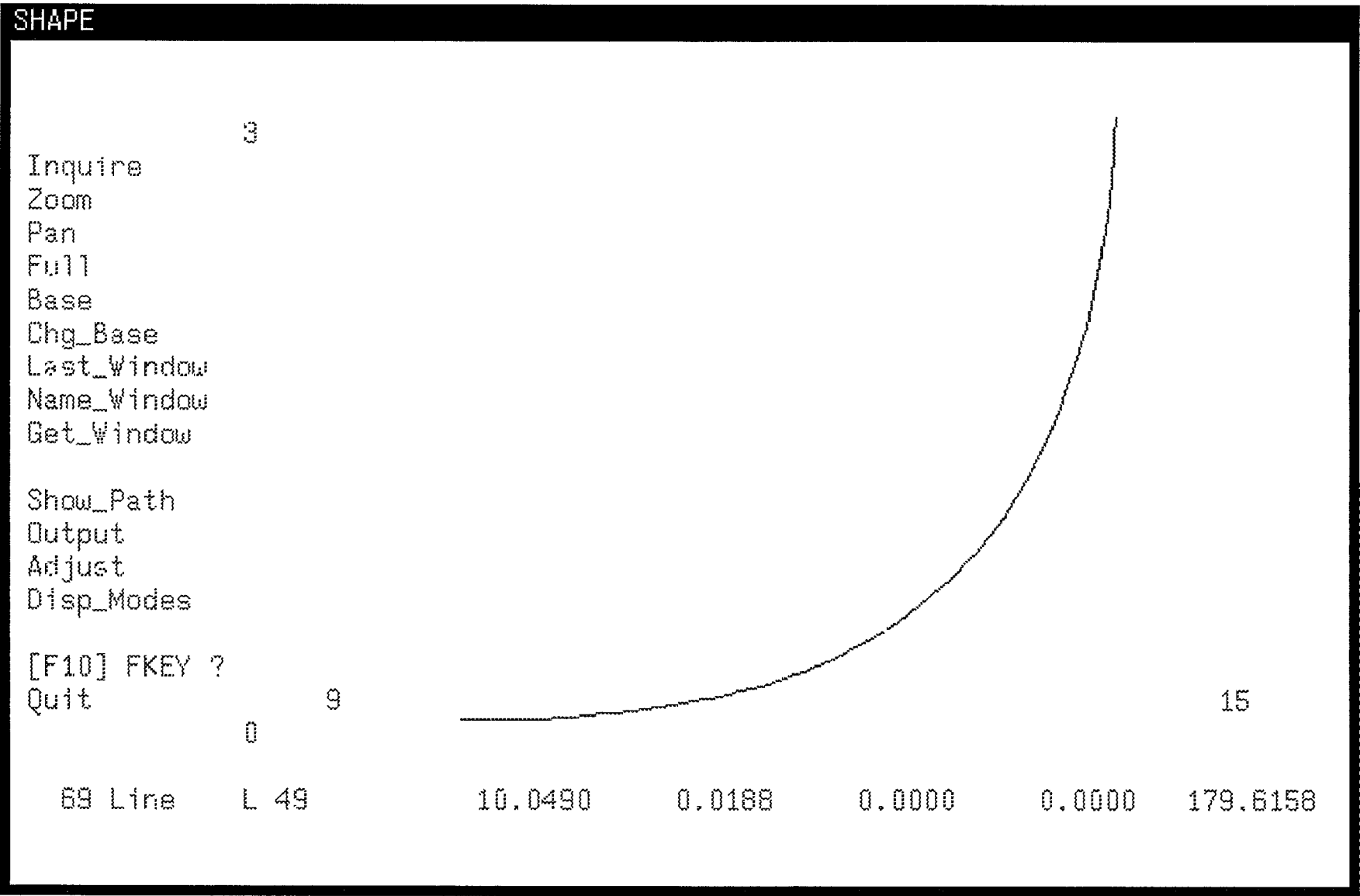


Figure 17 The Upper side Cutting Profile

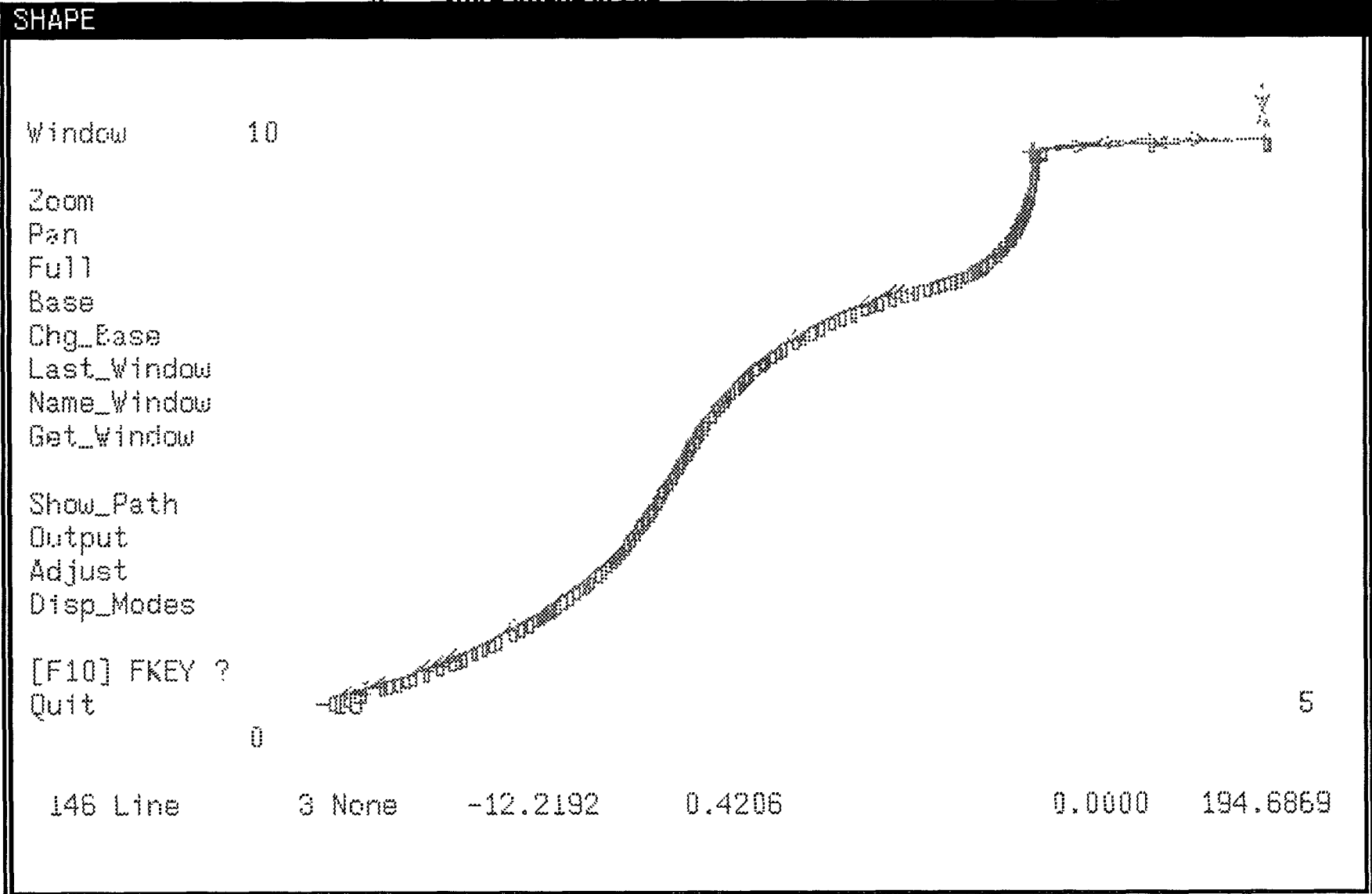


Figure 13 lower side Cutting Profile with Tool path

network communication methods are used for the file transfer operation. NJIT has developed a communication module under Manufacturing Automation Software Project (MASP). The file from any SUN spark workstation can be sent to the shopfloor machine via the ethernet connections. This operation is similar to a direct numerical control (DNC). The SUN system uses Andrew File System (AFS) designed by Carnegie Mellon University. This file system allows user to access code file in his/her own account. The file is renamed as per the machine to which it has to go (ex: xx.mzk for mazak turning center). Changes in the .cshrc are required for hoops software operation. The file is sent to a server named "truth". This server is connected to the machines on the shopfloor. The machine then activates the file server and receives the file belonging to its extension. For every file request there is a password accompanied so as to have restricted access to an individual's file.

5.4 Manufacturing at CNC Machine

The file received at the machine is inspected for toolpath, as there is a possibility of data getting corrupted. The "check" function on the MAZAK CNC turning machine shows the toolpath of the program. The MAZAK CNC turning machine works on point-to-point co-ordinate system. The sample part needed to be manufactured in two sections. One the inner curve, second the outer curve. First a dry run was made to check for safety. Then the file was kept as masterfile. The masterfile can be run repeatedly to make the desired quantity of parts. The generated file output using SmartCam is shown in the appendix.

CHAPTER 6

CONCLUSIONS

6.1 Evaluation of the Factory of Future for Reverse Engineering

1) The real power of reverse engineering based part reconstruction system lies in the ability to be readily integrated with other manufacturing software. After the raw data has been acquired by the Brown and Sharpe coordinate measuring machine, it must be *messed* before it can be used by the CAD system. The data gets converted into a CAD interpretable program using a C-program translator. This is a CAD interface to transform the data.

2) The data acquired by reverse engineering must be smoothed and outlier points must be eliminated. But data collected by coordinate measuring machine is a contact technique. There are no noise patterns in the data as in the case of image reconstruction system which uses a CCD camera (Chua, Chee Chai., and Robert Gay 1991). In that case, the camera's picture is not distinguishable if light intensity changes.

3) The language of CAD is geometry based. The drawings are stored in data files as collections of entities such as lines, arcs, points, and so on. A CAM database is, however, based on part features such as face, taper, chamfer, groove and so on. These features are used in process planning for interpretation of the manufacturing process.

The geometry export takes place from CAD to CAM package as a chained profile. The transfer works in IGES format. All the communication is done using RS232/RS422 communication ports and ethernet network to connect machines. The file transfer is done using "long block kermit" and "file transfer protocol (ftp)". The system together works very effectively and can be used repeatedly for the conceptual use of the factory of the future.

6.2 Conclusion

The integrated reverse engineering system makes it possible to shorten the product development cycle by going from a preliminary prototype part to a final manufactured product. This is especially important because of the trend towards shorter life cycles for new products. Further development of these integrated systems will have a substantial impact in the process of integration of CAD and CAM. Reverse engineering based part reconstruction systems are custom designed and beyond the reach of many small and medium sized manufacturing environments. However, a reverse engineering system can become economically feasible and will begin to play a major role in enhancing productivity.

6.3 Scope for Future

These advancements in CAD/CAM and reverse engineering areas are well out of the reach of small scale manufacturers and job shops in terms of costs. The hardware and software required is expensive and usually not accessible to small shops. But day by day the computing power is increasing rapidly with decreasing prices. Small jobshop which usually have a PC, is getting cheaper. The CAD packages like AutoCad, VersaCad, and Cadkey are getting more sophisticated. Some manufacturers have introduced CAM packages like MasterCam and, SmartCam which run on PC based system.

A detailed statistical study of tolerances of the reverse engineering based part reconstruction system should be made. Analysis of dimensional changes and variation due to various CAD/CAM system integration should be studied. This work will further increase the accuracy and reliability of the part reconstruction system.

The shop of the 90's will be equipped with these smart and new tools. A strong need of integration of these softwares is needed just like flexible manufacturing cells in order to have "one single" PC based system. Equipment is

less effective if they are not integrated. With these modern tools, these shops will be quicker and responsive to the changing market situations and product development cycles will decrease rapidly. This will bring a big productivity boost to them.

APPENDIX

@SmartCam output for Mazak CNC Turning Center

@file name: SHAILESH.MZK

N100 G90
N105 G00 X20.0 Z2.0
N110 T0300 M06
N115 G97 S150
N120 M03
N125 M08
N130 G00 X20.0 Z2.0 T0303
N135 G01 X19.7 Z0.1 F0.02
N140 G00 X19.6 Z-0.05
N145 X20.0 Z2.0
N150 Z4.0
N155 X21.9985 Z3.9616
N160 X20.02 Z3.9948
N165 G01 X19.72 Z0.0948
N170 G00 X19.6 Z0.0
N175 X19.6223 Z-0.0619
N180 G01 X19.6025 Z-0.0612
N185 X19.3964 Z-0.0547
N190 X19.1858 Z-0.0508
N195 X19.0409 Z-0.052
N200 X18.8912 Z-0.057
N205 X18.7508 Z-0.065
N210 X18.6162 Z-0.076
N215 X18.4724 Z-0.0887
N220 X18.3987 Z-0.0966
N225 X18.3258 Z-0.1061
N230 X18.1868 Z-0.1243
N235 X18.0369 Z-0.1446
N240 X17.8859 Z-0.1698
N245 X17.7436 Z-0.1965
N250 X17.5981 Z-0.2242
N255 X17.5241 Z-0.2392
N260 X17.4518 Z-0.256
N265 X17.3017 Z-0.2922
N270 X17.1564 Z-0.3292
N275 X17.0147 Z-0.3707
N280 X16.8647 Z-0.4173
N285 X16.7881 Z-0.4425
N290 X16.714 Z-0.4695
N295 X16.5625 Z-0.5276
N300 X16.4413 Z-0.5777
N305 X16.3274 Z-0.6321

N310 X16.2148 Z-0.6884
N315 X16.1088 Z-0.7477
N320 X15.9555 Z-0.8431
N325 X15.875 Z-0.8974
N330 X15.7978 Z-0.953
N335 X15.7139 Z-1.0182
N340 X15.6355 Z-1.085
N345 X15.5481 Z-1.1668
N350 X15.4668 Z-1.2503
N355 X15.3732 Z-1.3582
N360 X15.2906 Z-1.4683
N365 X15.2396 Z-1.5449
N370 X15.1707 Z-1.6607
N375 X15.1106 Z-1.7778
N380 X15.045 Z-1.9129
N385 X14.94 Z-2.1145
N390 X14.8271 Z-2.3151
N395 X14.6947 Z-2.5391
N400 X14.5502 Z-2.7611
N405 X14.4692 Z-2.8763
N410 X14.3838 Z-2.9907
N415 X14.2145 Z-3.2023
N420 X14.046 Z-3.3978
N425 X13.8739 Z-3.5857
N430 X13.7058 Z-3.7591
N435 X13.5401 Z-3.9183
N440 X13.3756 Z-4.0674
N445 X13.2146 Z-4.2062
N450 X12.9715 Z-4.4043
N455 X12.8561 Z-4.4946
N460 X12.7316 Z-4.5817
N465 X12.5854 Z-4.6824
N470 X12.5086 Z-4.7356
N475 X12.4286 Z-4.7876
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N485 X12.1209 Z-4.9729
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N495 X11.9676 Z-5.0592
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N515 X11.3478 Z-5.3725
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N525 X11.054 Z-5.5024
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N635 X7.4914 Z-6.6856
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N645 X7.1852 Z-6.7908
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N710 X4.9869 Z-7.7744
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N765 X3.7445 Z-8.6487

N770 X3.6659 Z-8.7075
N775 X3.5099 Z-8.8339
N780 X3.3454 Z-8.9757
N785 X3.1838 Z-9.1205
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N795 X2.807 Z-9.4765
N800 X2.6891 Z-9.5971
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N810 X2.3518 Z-9.9698
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N825 X2.0131 Z-10.3831
N830 X1.8418 Z-10.613
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N880 M02

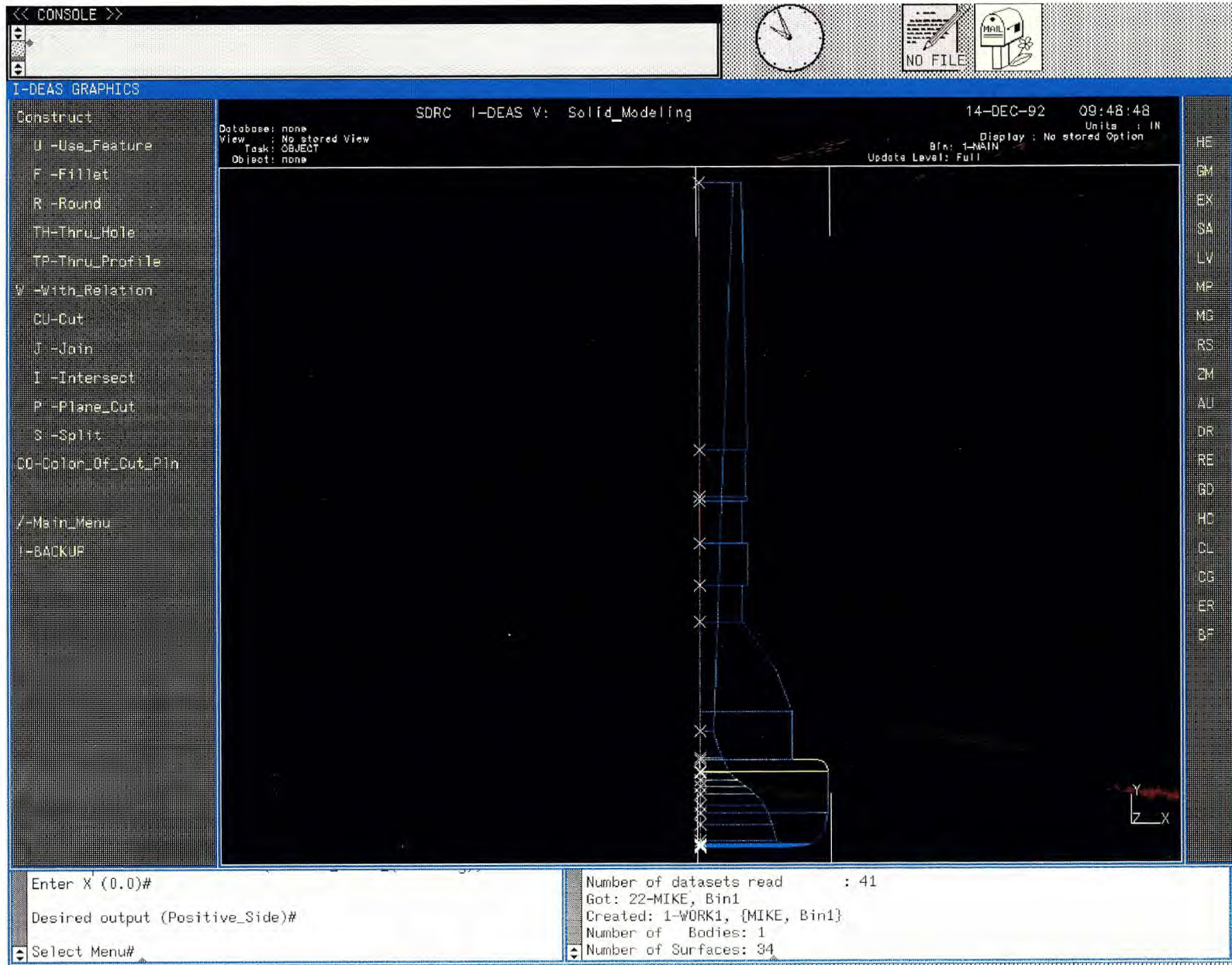


Figure 14

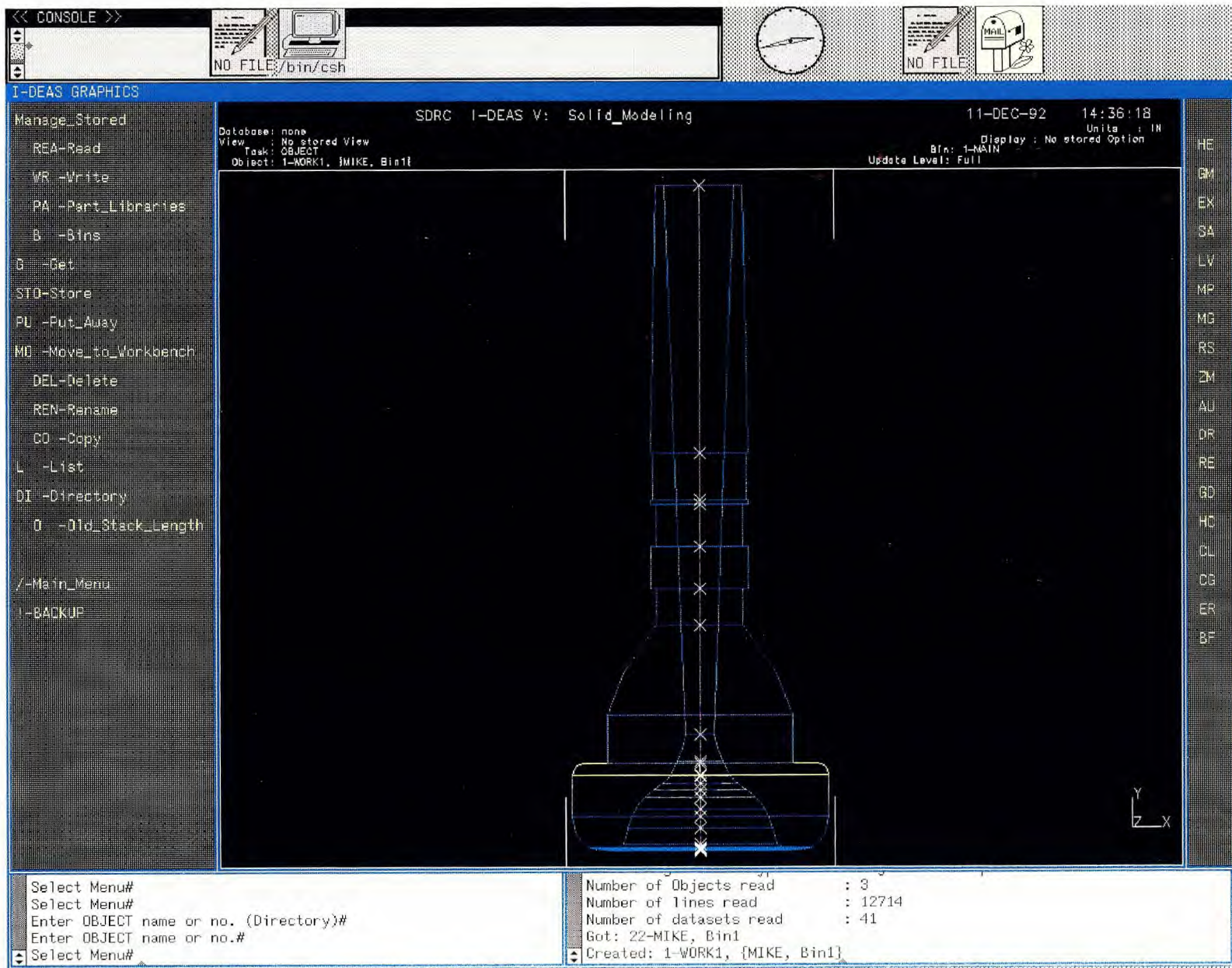


Figure 15

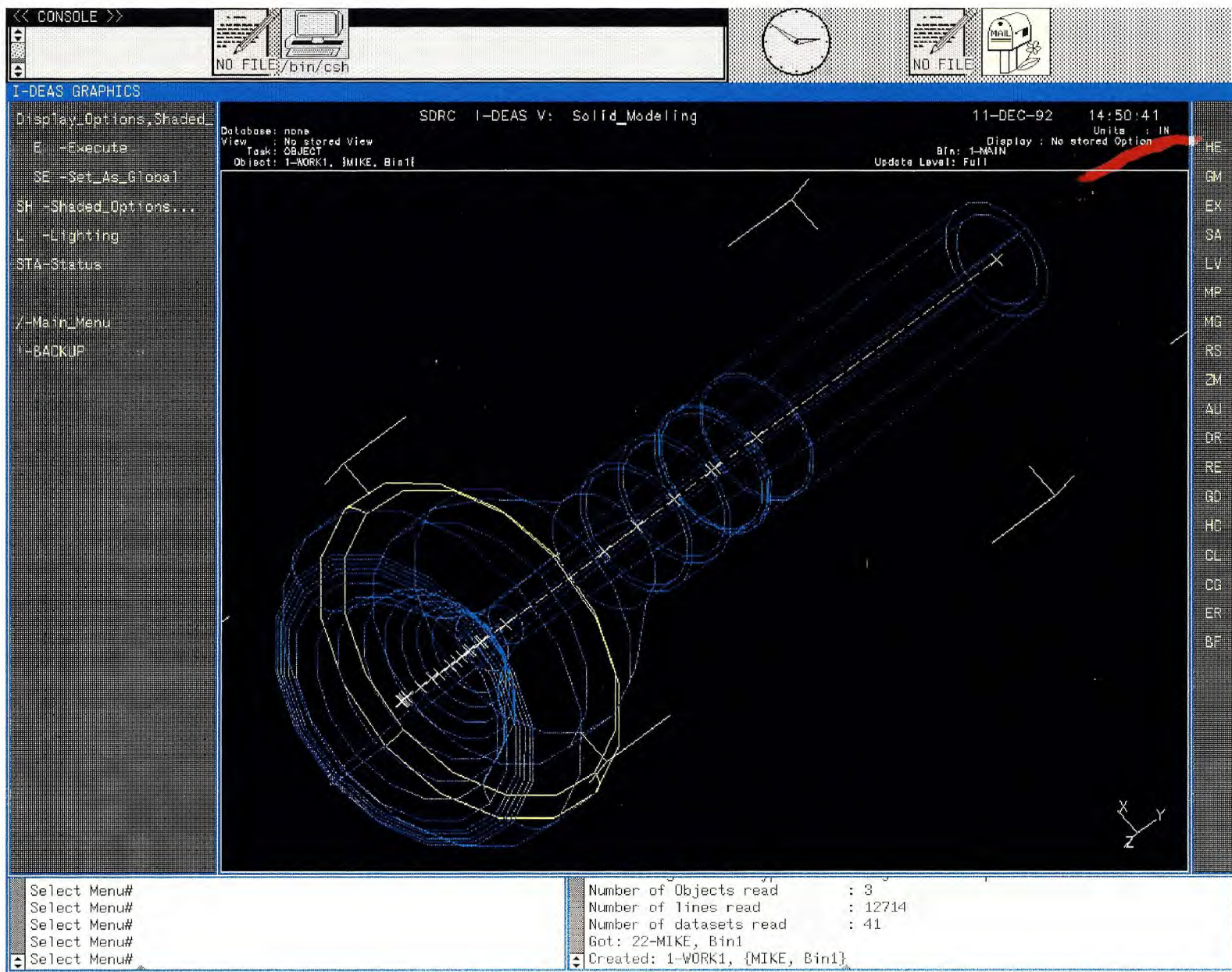


Figure 16

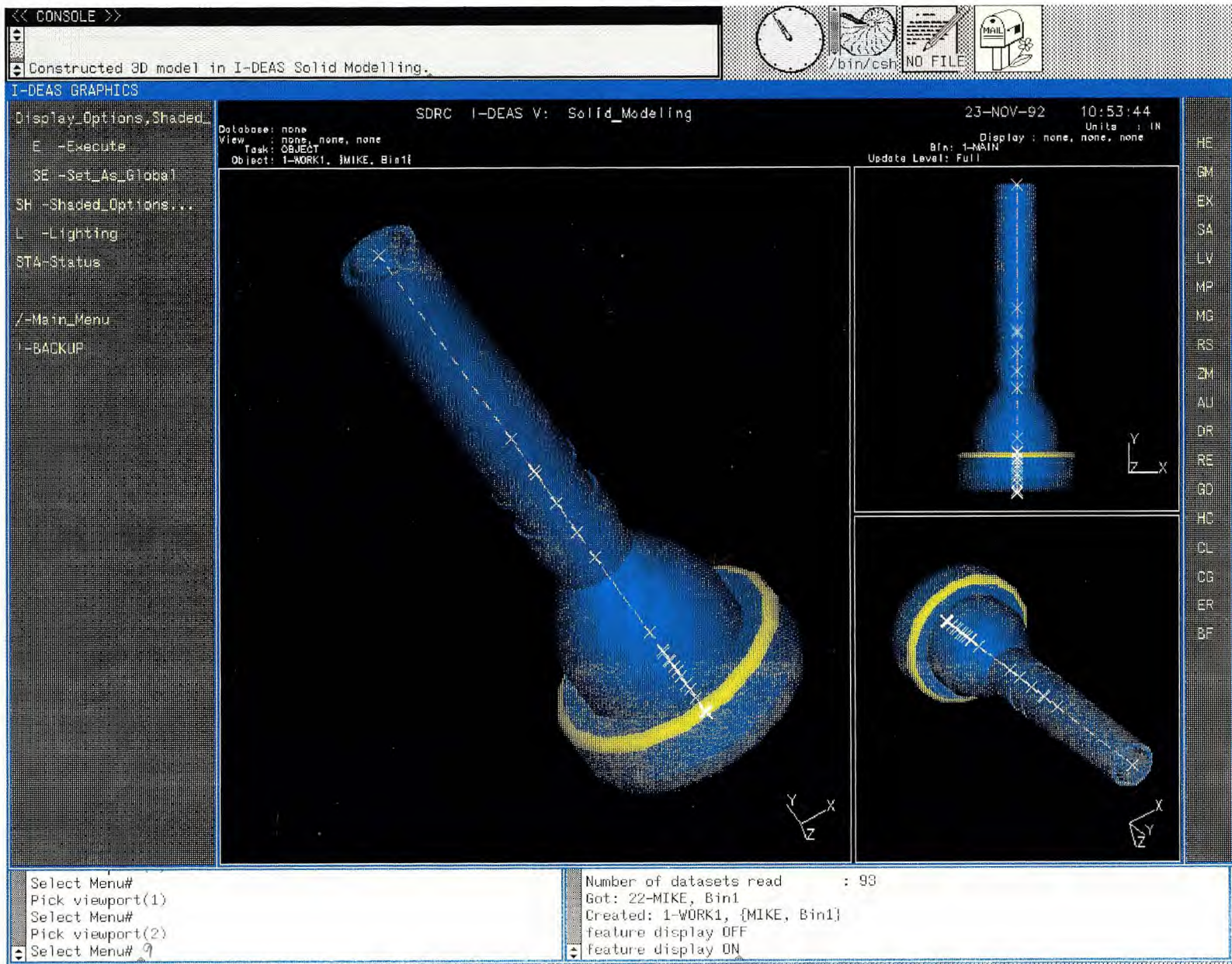


Figure 17

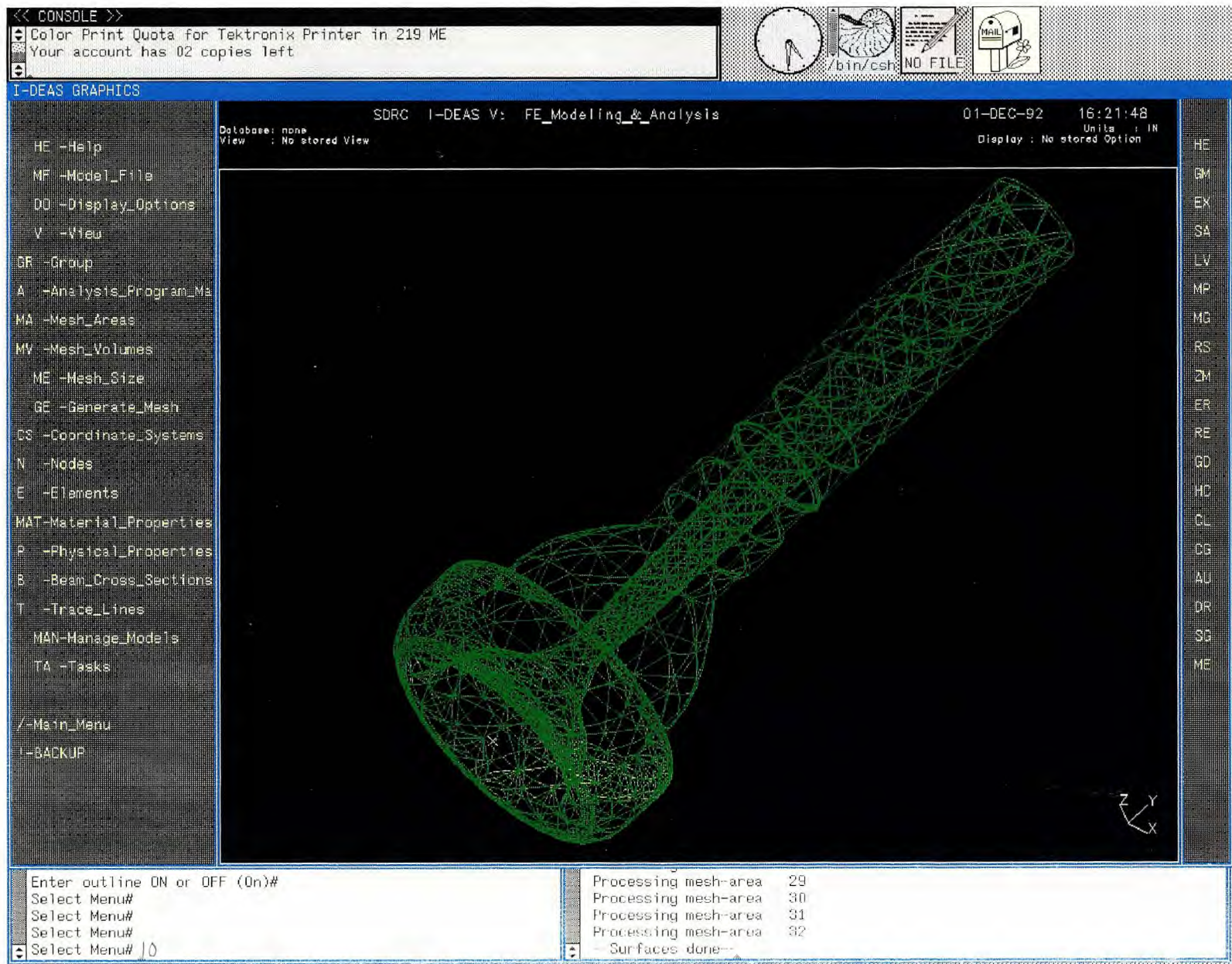


Figure 18

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