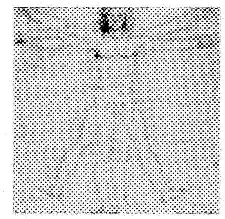
PARAMETRIC DESIGN

Chapter 5



LEARNING OBJECTIVES

Upon completion of this chapter you will be able to:

1. Understand the basic processes involved in parametric feature-based design.
2. Describe the steps in the design process when using a parametric CAD/CAM/CAE system.
3. Understand the function and application of the sketcher in creating base and construction features.
4. Explain how features are used to create a design.
5. Show how parts are designed and then combined into assemblies.
6. Explore the automated generation of details and drawings using parametric design.
7. Understand the use of datum features to model a part.
8. Discover several methods of capturing design intent in parametric modeling.
9. See how a feature-based CAD/CAM system can successfully incorporate engineering knowledge into the solid model.

5.1 INTRODUCTION

This chapter introduces the basic parametric design concepts for creating and documenting individual parts and assemblies. **Parametric** can be defined as *any set of physical properties whose values determine the characteristics or behavior of something.* **Parametric design** enables you to generate a variety of information about your design—its mass properties, a drawing, or a base model. To get this information, you must first model your part design (Fig. 5.1). The part in this figure was modeled with **Pro/ENGINEER™** from **Parametric Technology Corporation.** The part is a solid model and is displayed in two windows on the screen. The primary screen displays the model with a mesh on its surfaces. The mesh functions in variety of engineering applications and capabilities, including engineering analysis of force, moment, and displacement.

This chapter is intended to acquaint you with parametric modeling philosophies. Throughout the remainder of the

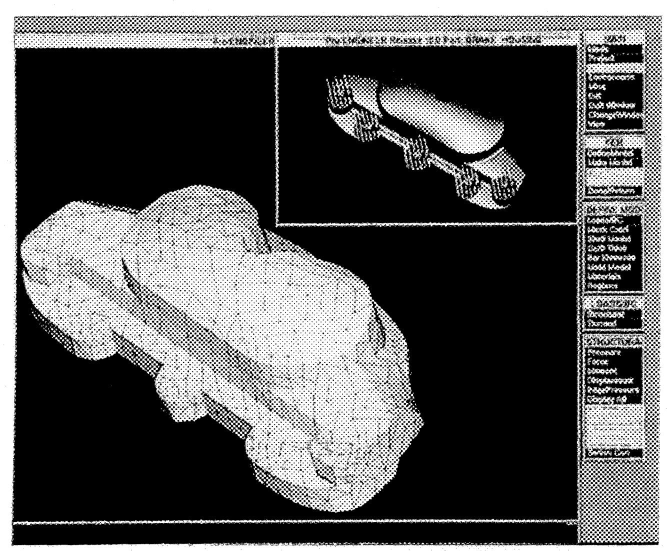


FIGURE 5.1 Part Modeled Using Pro/ENGINEER and Displayed with Pro/MESH

text, a series of **Applying Parametric Design** boxes are introduced to illustrate parametric design ideas for each chapter. *With the exception of this chapter,* all Applying Parametric Design boxes are based entirely on Pro/ENGINEER. This chapter will use an **AutoCAD's Designer™** example in its Applying Parametric Design box.

The following methodologies are the principal aspects of successful parametric solid modeling.

Feature-Based Modeling. Parametric design represents solid models as combinations of engineering features (Fig. 5.2). This chapter introduces the various types of features, along with an example of their combination into a part model.

Creating Assemblies. Just as features are combined into parts, parts may be combined into assemblies, as shown in Figure 5.3, where the race car design incorporates a large number of parts, subassemblies, and a final assembly. This chapter will briefly discuss the hierarchical relationships of assembled parts, as well as the creation of new parts within an assembly model.

Capturing Design Intent. Several methods of capturing design intent in parametric modeling are presented in this chapter. To be able to incorporate engineering knowledge successfully into the solid model is an essential aspect of parametric modeling (Fig. 5.4). This ensures that critical parameters are satisfied as your design evolves.

The diagram in Figure 5.5 illustrates the role of each of the methodologies in the modeling process.

5.1.1 Modeling vs. Drafting

A primary and essential difference between parametric design and traditional computer-aided drafting systems is that parametric design models are three-dimensional. Designs increas-

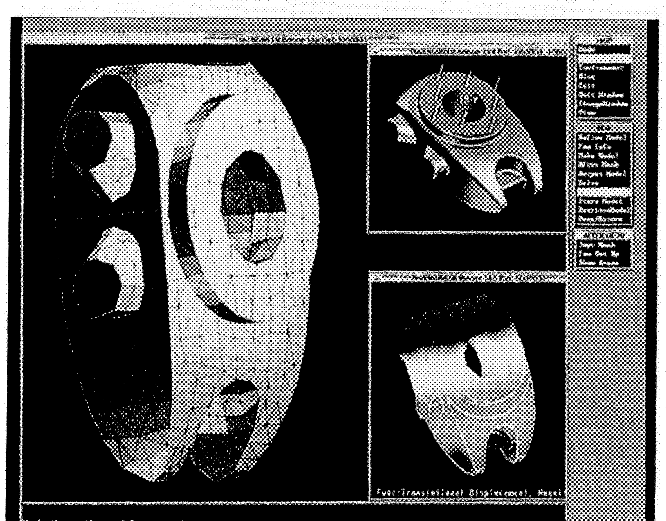
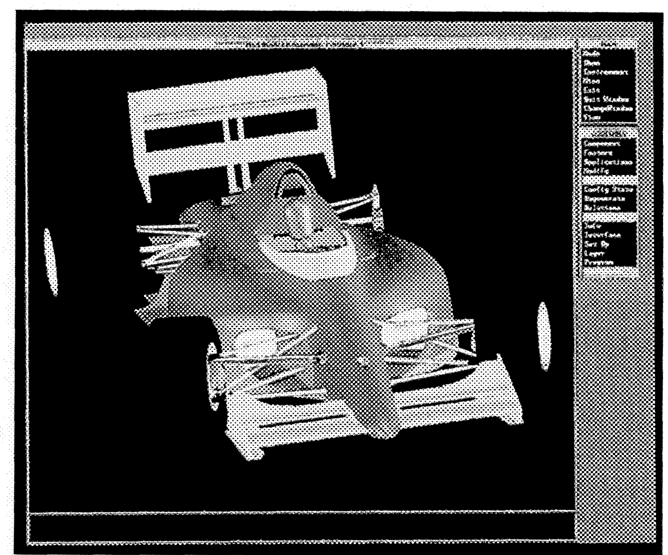
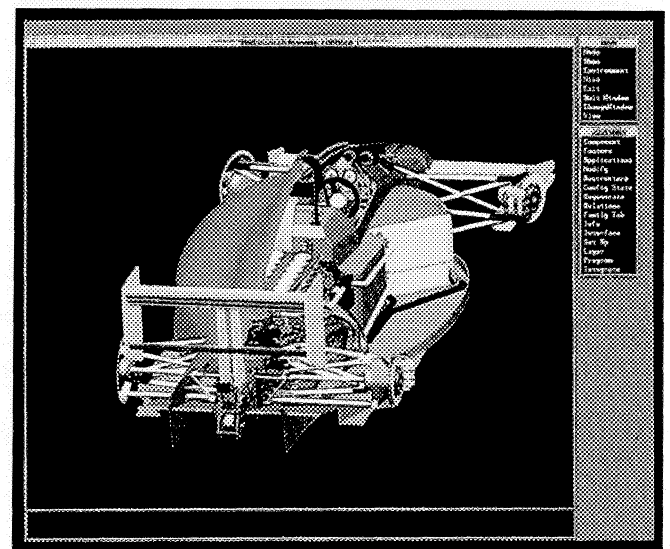


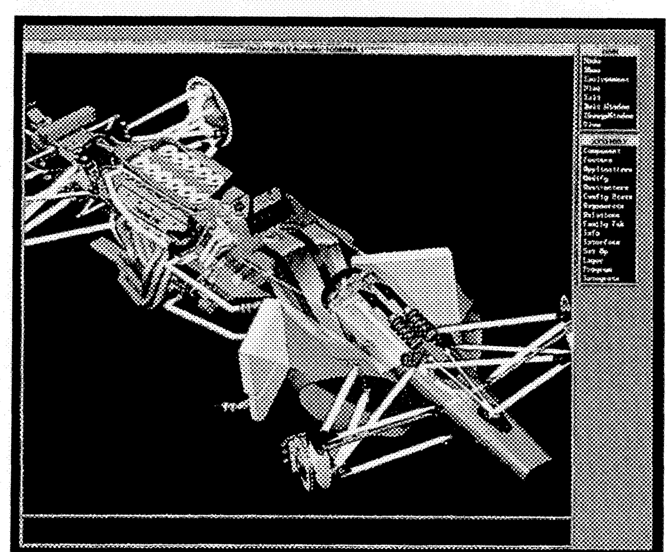
FIGURE 5.2 Analysis Performed on a Part Model Using Pro/ENGINEER



(a) Complete race car

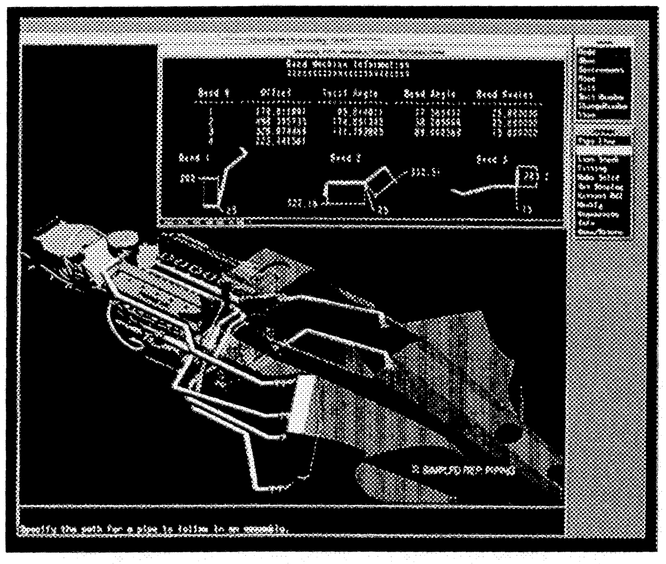


(b) Half of car body removed to see interior

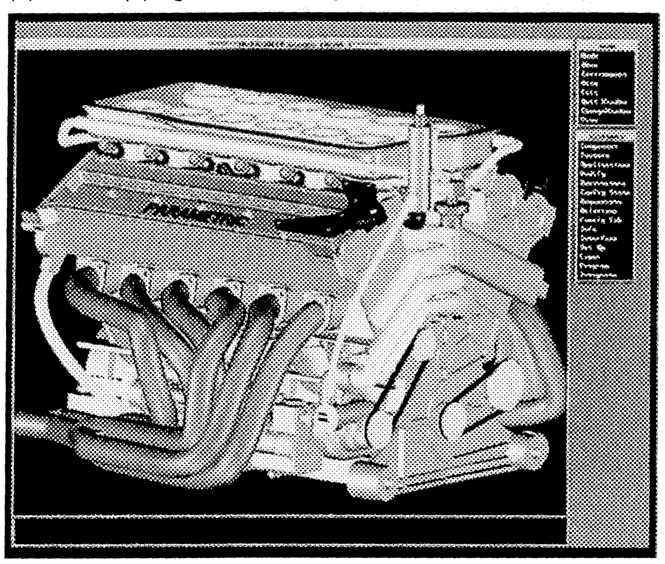


(c) Subassembly

FIGURE 5.3 Race Car Modeled with Pro/ENGINEER



(d) Exhaust piping and bend analysis



(e) Engine

FIGURE 5.3 Race Car Modeled with Pro/ENGINEER—*Continued*

ingly are represented in the form of solid models that capture design intent as well as design geometry. Engineering designs today are frequently constructed as mathematical solid models instead of as 2D drawings. A solid model is one that represents a shape as a 3D object having mass properties.

There are two main reasons for the move to solid models. First, solid-modeling packages can serve as an easy means of portraying parts for study by cross-functional concurrent-engineering teams. The solid model can be understood by even nontechnical members of the team, such as those from the marketing and sales departments.

Second, the capabilities of solid modelers have been upgraded so the model can represent not only the geometry of part being designed, but also the intent of the designer. This is of most significance when the designer needs to make changes to the part geometry. Far fewer changes will be necessary in later-generation parametric solid models that capture design intent than in previous CAD/CAM modeling software.

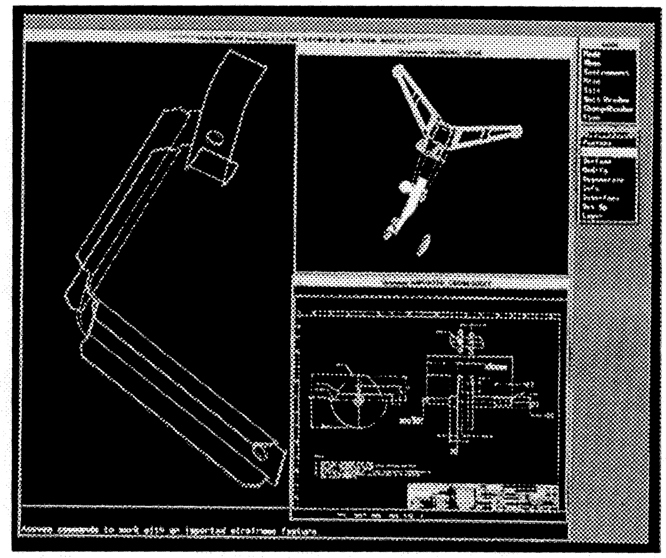


FIGURE 5.4 Parametric Design Using Pro/ENGINEER

In parametric design, drawings are produced as views of the model, rather than the other way around. Parametric design models are not so much drawn as *sculpted* from solid volumes of materials.

5.1.2 Parametric Design Overview

To begin the design process, analyze your design. Before any work is started, take time and *tap* into your own knowledge bank and others that are available. The acronym **TAP** (**t**hink, **a**nalyze, **p**lan) can remind you of these three steps so essential to any engineering design process in which you may be involved.

Break your overall design down into its basic components, building blocks, or primary features. Identify the most fundamental feature of the part as the first feature, or base feature, to sketch. A variety of **base features** can be modeled using the commands *protrusion-extrude, revolve, sweep,* and *blend.* **Sketched features** (*neck, flange,* and *cut*) and pick-and-place features, called **referenced features,** complete the design (*holes, rounds,* and *chamfers*). With the **sketcher,** you use familiar 2D entities (points, lines, circles, arcs, splines, ellipses). There is no need to be concerned with the accuracy of the sketch. Lines can be of differing angles, arcs and circles can have unequal radii, and dimensions and relationships can be sketched without regard to the actual part dimension. In fact, exaggerating the difference between features that are similar but not exactly the same is actually a better practice when using the sketcher.

The system helps to apply logical geometric constraints to the sketch. **Constraints** clean up the sketch geometry according to the system assumptions. **Geometry assumptions** and constraints close ends of connected lines, align parallel lines, and snap sketched lines to the horizontal and vertical. Additional constraints are added through **parametric dimensions** to control the size and shape of the feature. For parts with more than one feature, these steps can create additional parametric features.

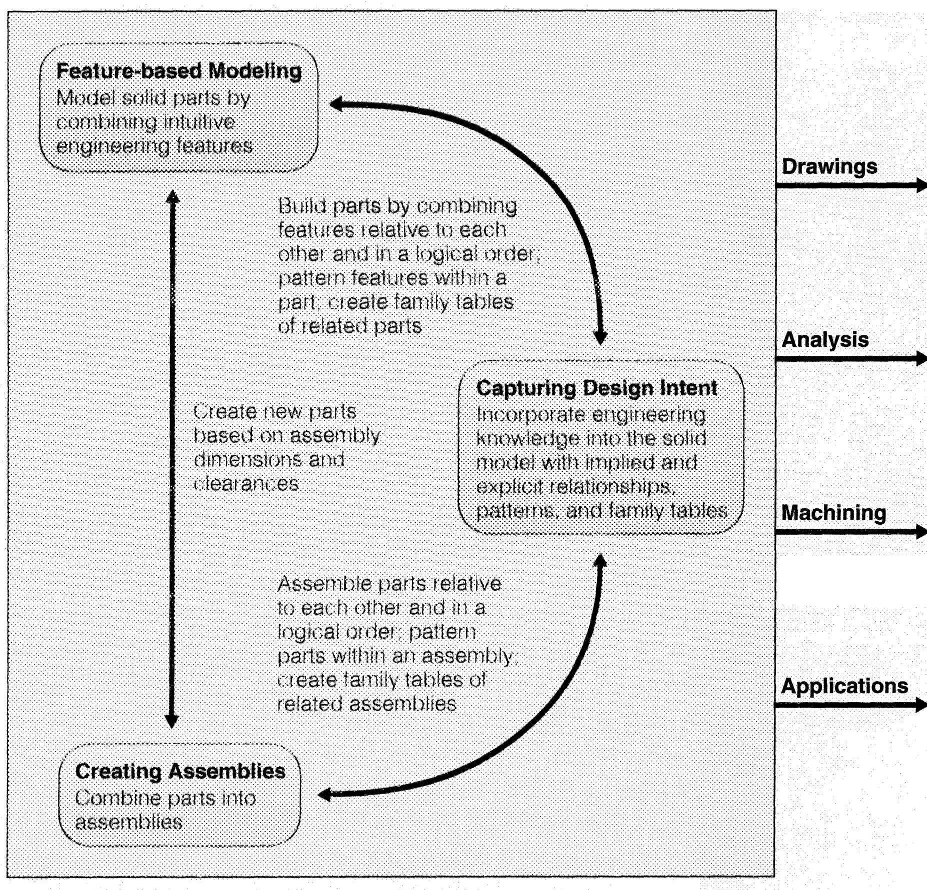


FIGURE 5.5 Parametric Solid Model

5.2 FEATURE-BASED MODELING

Features are the basic building blocks for building a part. Features "understand" their fit and function—they have "smarts" built in. For example, a hole or neck or cut feature knows its shape and part location and that it has a negative volume. As you modify a feature, the entire part automatically updates after regeneration. The idea behind feature-based modeling is that the designer construct a part so it is composed of individual features that describe how the geometry is supposed to behave in the event its dimensions change, which happens quite often in industry, as in the case of a design change.

The easiest way to explain feature-based modeling is to contrast it with older solid-modeling methods. One example is that of creating a hole in a part. Older solid modelers used **constructive solid geometry (CSG)** to define such a feature. Here, the designer would define a simple cylinder having the diameter of the desired hole and long enough to extend through the part. Then the system would be told to perform a **Boolean** difference operation between the part and the cylinder. The result would be a hole in the part having the diameter of the cylinder (Fig. 5.6).

A problem arises with this approach if the part is later modified, for instance, if the part must be thicker. If the designer did not make the cylindrical space long enough to extend through the new thicker part, the result will be a model of a blind hole (not all the way through the part feature). The model captured the geometry specified by the designer, but it did not capture the intent, which in this case is a thru-hole.

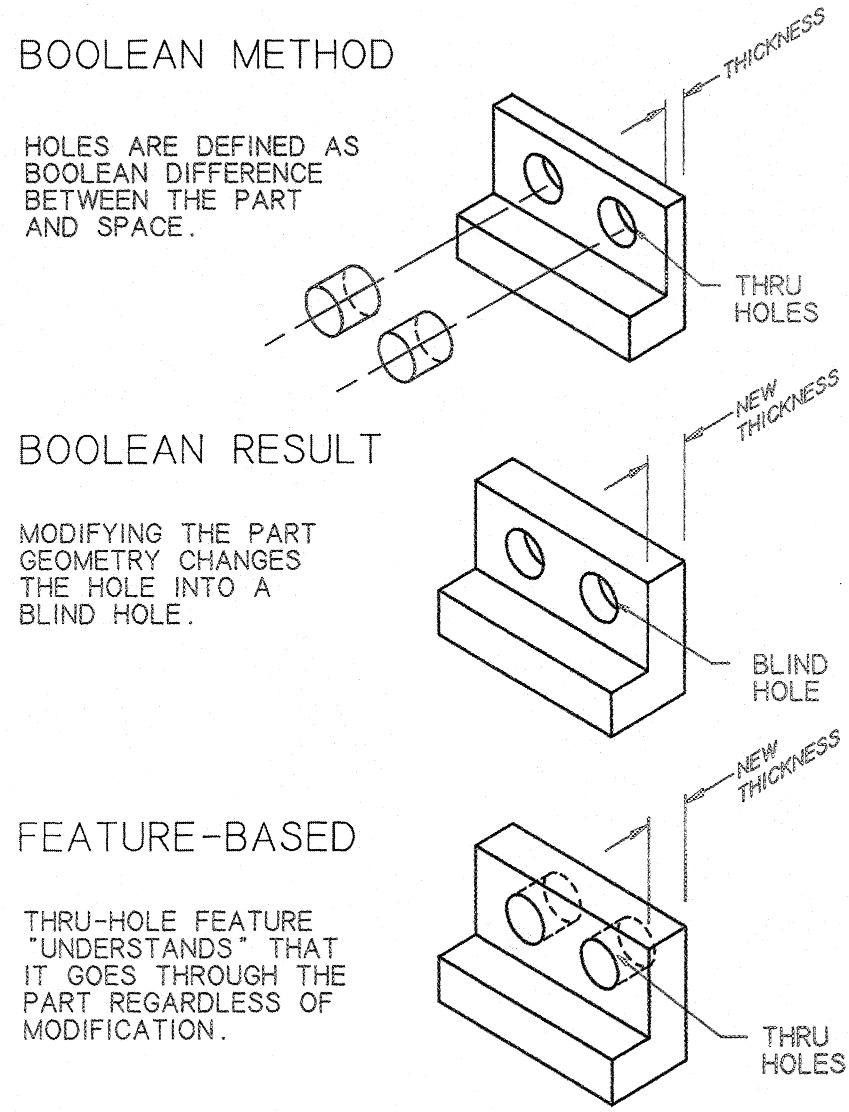
A designer working in a feature-based modeler would approach the thru-hole differently. A feature called a thru-hole would be defined such that no matter what the thickness dimension of the part, the hole extends completely through it.

Modelers also let the designer suppress features temporarily, as a means of making it easier to change the part geometry. It is also possible to modify previously defined features, or to define new features using combinations of old features. Systems prompt the designer for input during the defining of the feature. Inputs may include positional constraints, relational definitions, and other factors. Feature-based modelers also allow designers to define features pertaining not only to geometry, but also to steps involved in downstream analysis and manufacturing.

The term **parametric modeling** describes the capturing of design operations as they take place, as well as future modifying and editing that takes place on the design. The order of the design operations is significant. Suppose a designer specifies that two surfaces—surface 1 and surface 2—are parallel to each other. Then, if surface 1 moves, surface 2 moves along with it to maintain the specified design relationship. Surface 2 is a **child** of surface 1 in this example. Parametric modelers allow the designer to **reorder** the steps in the part's creation.

The "chunks" of solid material from which parametric

FIGURE 5.6 Feature-Based Modeling



design models are constructed are called features. Features generally fall into one of the following categories.

Base Feature. The base feature may be either a sketched feature or datum plane(s) referencing the default coordinate system. The base feature is important because all future model geometry will reference it directly or indirectly; it becomes the root feature. Changes to the base feature will affect the geometry of the entire model (Fig. 5.7).

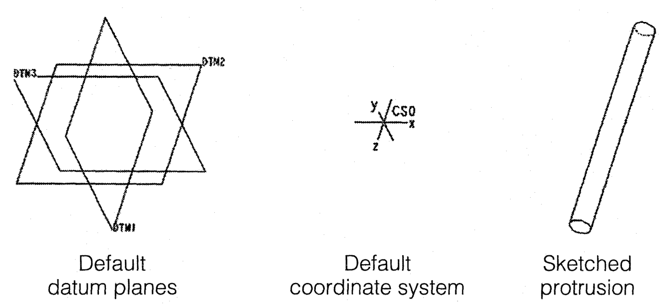


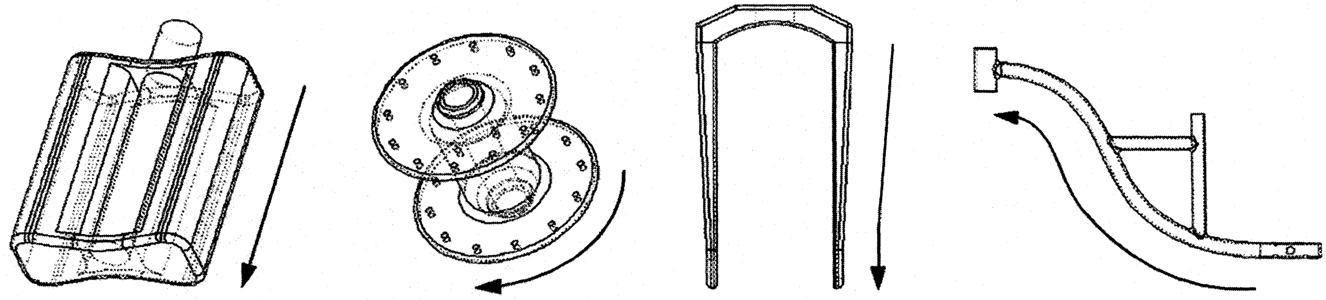
FIGURE 5.7 Base Features

Sketched Features. Sketched features are created by extruding, revolving, blending, or sweeping a sketched cross section. Material may be added or removed by protruding or cutting the feature from the existing model (Fig. 5.8).

Referenced Features. Referenced features reference existing geometry and employ an inherent form; they need not be sketched. Some examples of referenced features are rounds, drilled holes, and shells (Fig. 5.9).

Datum Features. Datum features, such as planes, axes, curves, and points, generally help provide sketching planes and contour references for sketched and referenced features. Datum features do not have physical volume or mass, and may be visually hidden without affecting solid geometry (Fig. 5.10).

The various types of features are used as building blocks in the progressive creation of solid parts. Figure 5.11 demonstrates this process.



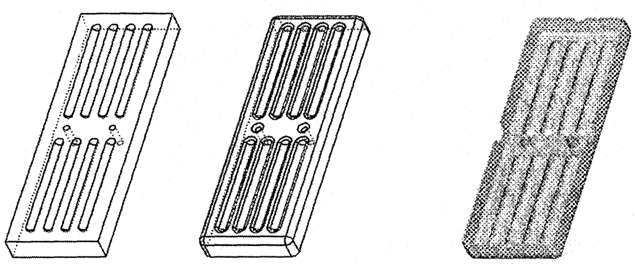
Extrude:
Pedal created by extruding bow-shaped section

Revolve:
Hub created by revolving sections

Blend:
Fork created by blending several cross sections

Sweep:
Frame created by sweeping cross section along shown trajectory

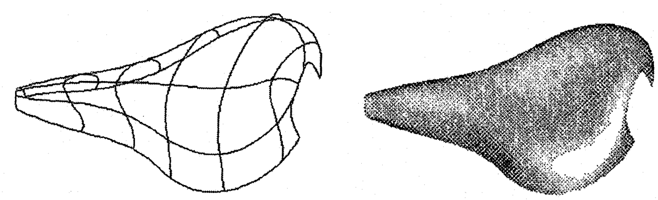
FIGURE 5.8 Sketched Features



Rounds feature references edges and surfaces, removing material to a specified radius

Shell feature references outer surfaces, reducing thickness to a specified value

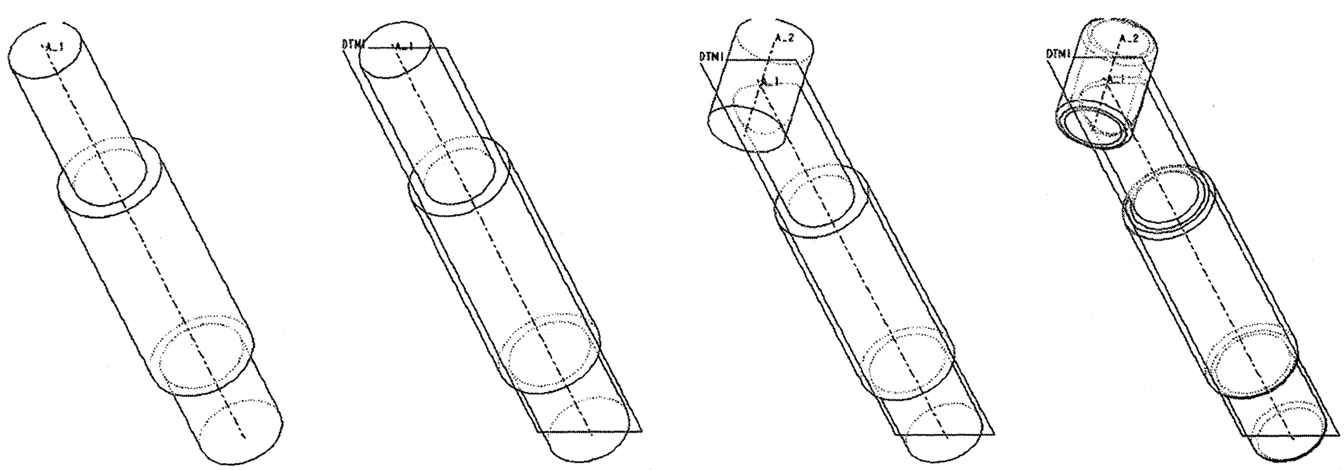
FIGURE 5.9 Referenced Features



Web of datum curves used to control surface contour

Seat created by enclosing volume with additional surfaces and filling with solid material

FIGURE 5.10 Datum Curves



Base feature:
Revolved protrusion from sketched cross section

Datum features:
Datum plane created at zero offset normal to Z axis of default coordinate system

Sketched feature:
Extruded protrusion sketched on datum plane with center aligned to top of base feature

Referenced features:
Hole drilled coaxially through top protrusion; rounds created along sharp edges

FIGURE 5.11 Features

5.2.1 Establishing Part Features

The design of any part requires that the part be *confined, restricted, constrained,* and *referenced.* In parametric design the easiest method to establish and control the geometry of your part design is to use three datum planes (see Chapters 15 and 16). The system allows you to use the **primary datum** to start your base feature. By creating the default datum planes (DTM1, DTM2, and DTM3 in Pro/ENGINEER) you can constrain your design in all three directions (see Fig. 5.7). Try a simple exercise: Put a book on the floor of a room in your house or school. This establishes **datum A,** or the primary datum plane (DTM3 for Pro/ENGINEER in most cases). Now slide the book up to a wall near the corner of the room. This establishes **datum B,** the **secondary datum** plane (DTM2 for most cases in Pro/ENGINEER). Choose the longest or second most important surface. Lastly, shove the book up against the other wall. You have now established **datum C,** or the **tertiary datum** plane (DTM1 in most cases with Pro/ENGINEER). The book is now constrained by three plane surfaces (walls). With a couple of clamps you can secure the part and machine it as if it were on a milling table (Figs. 5.12 and 5.13).

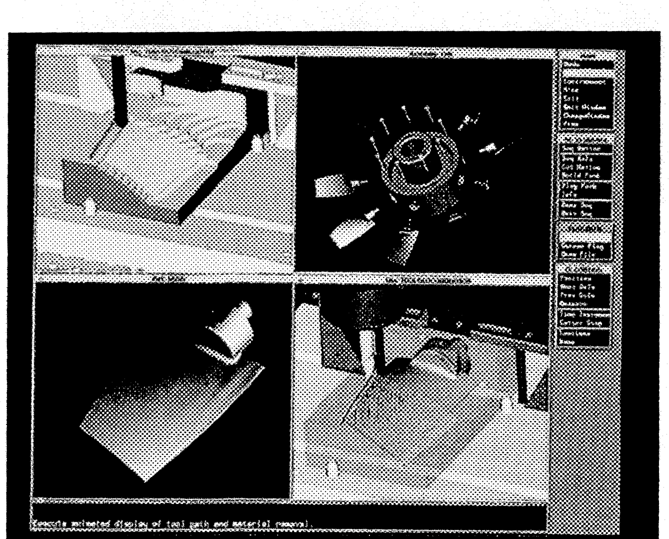


FIGURE 5.12 Machining Using Pro/ENGINEER and Pro/MANUFACTURING

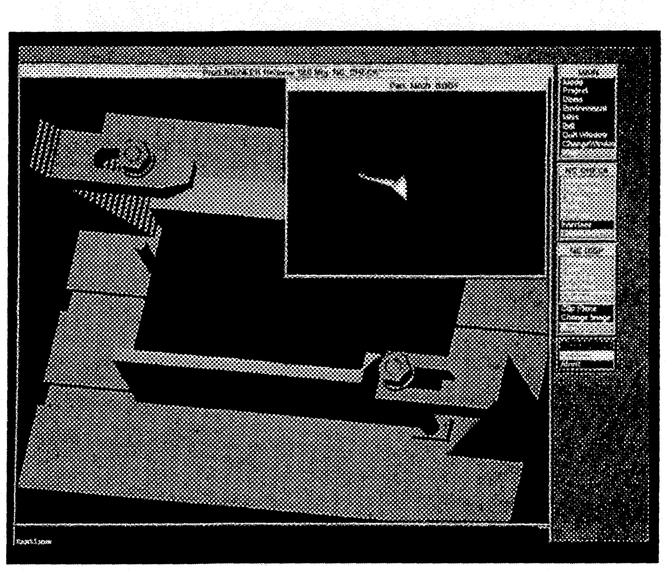


FIGURE 5.13 Pro/MANUFACTURING and Machining Fixtures

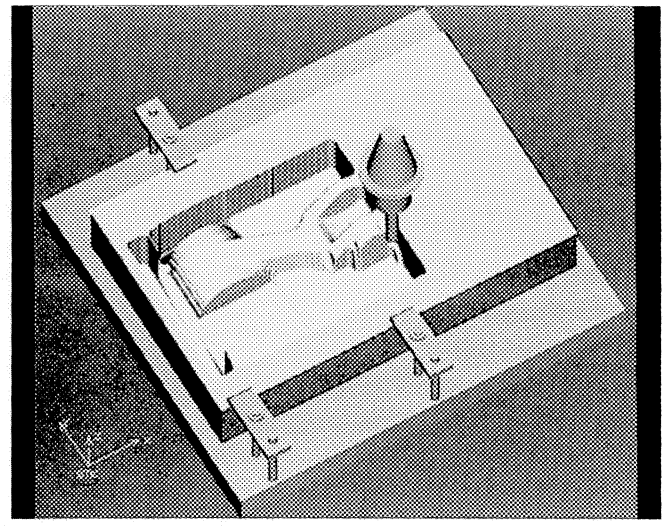


FIGURE 5.14 Visually Machining a Part Using CGTECH™ Software

Although this exercise and description is simplified and will not work for some parts, it does demonstrate how to establish your part in space using datums. You may use any of the datums as sketching planes or for that matter any of the part surfaces for construction geometry. Any number of other datums can be introduced into the part as required for feature creation, assembly operations, or manufacturing applications (Fig. 5.14).

5.2.2 Datum Features and Datum Planes

Datum features are planes, axes, and points by which you place geometric features on the active part. There are three types of datum features: *datum planes, datum axes,* and *datum points.* You can display all types of datum features, but they do not define the surfaces or edges of the part or add to its mass properties.

Datum planes are infinite planes located in 3D model mode and associated with the part that was active at the time of their creation. To select a datum plane, you can pick on its name or pick anywhere on the planar square.

Datum planes are *parametric*—geometrically associated with the part. Parametric datum planes are associated with and dependent on the edges, surfaces, vertices, and axes of a part. For example, a datum plane placed parallel to a planar face and on the edge of a part moves whenever the edge moves and rotates about the edge if the face moves. As you create parametric datum planes, you determine the relationship to the active part by defining combinations of a placement option that link the datum plane to the part.

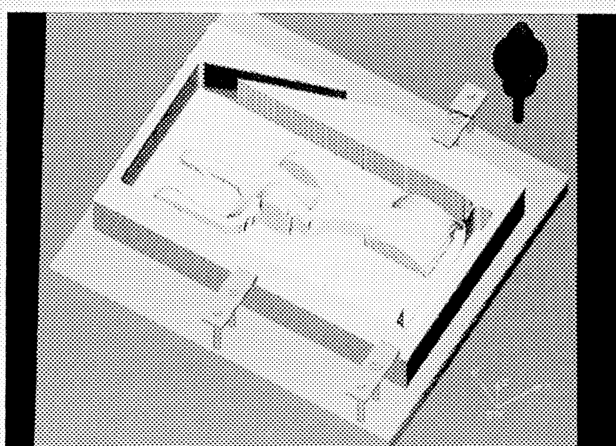
Focus On . . .

DISPLAYING PART MACHINING

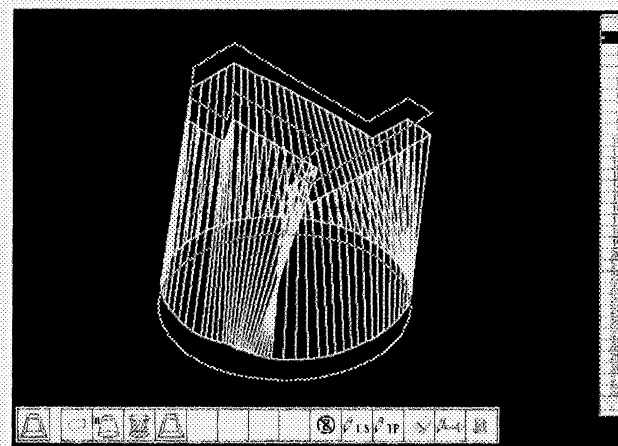
Before a part is manufactured on a machine using expensive and time-consuming methods, we can now display the cutter, cutter path, machining sequence, and material removal of a part directly on the CRT (see figure). Multiple-axis milling modules for cylindrical parts like roller dies, and drum cams (see figure), EDM wire capabilities (see figure), and multiaxis turning for lathe parts (see figure) are all available in today's high-technology manufacturing.

The volume and local milling of a part on a CAD/CAM system can be displayed and plotted for verification (see figure). In this last figure we can see how easy it is to analyze the programmed cutting sequence and the removal of material for a typical part. Here, a large tool is used to remove a majority of the material through volume milling. The next step involves having a smaller cutting tool finish the pocket and remove the leftover material.

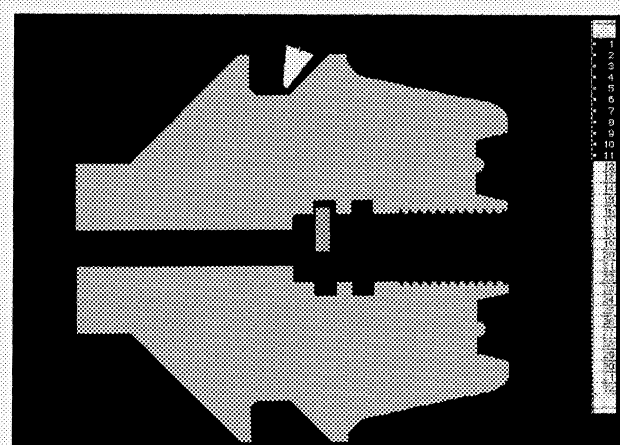
Full visual verification and control of all tool motion are now possible. These new tools are making the process of designing and manufacturing a part quicker, highly efficient, and extremely precise.



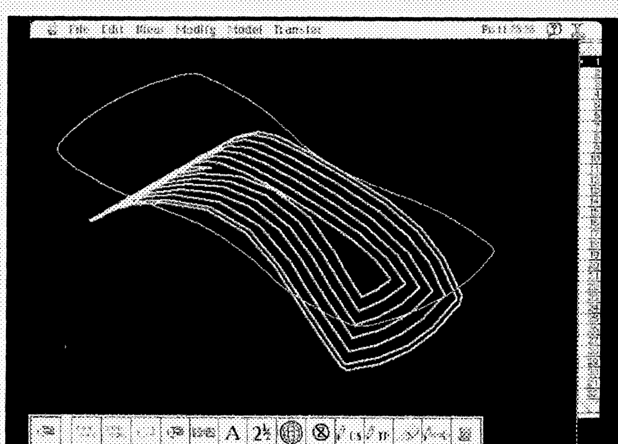
Displaying cutter, workpiece, and material removal using CGTECH™ software.



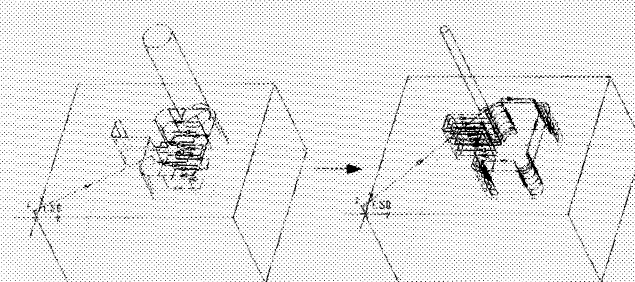
Wire EDM using Gibbs CAM software.



Multi-Axis turning using Gibbs CAM software.



Four-axis milling using Gibbs CAM software.



Milling.

Datum planes can create a reference on a part where one does not already exist. For example, you can sketch or place features on a datum plane when there is no appropriate planar surface. You can also dimension to a datum plane as if it were an edge. When you are constructing an assembly, you can use datums with assembly commands.

A datum is created by specifying constraints that locate it with respect to existing geometry. For example, a datum plane might be made to pass through the axis of a hole (Fig. 5.15) and parallel to a planar surface. Figure 5.15(a) shows a model of a clamp without hidden lines. Figure 5.15(b) displays the same part with hidden lines. And Fig. 15.15(c) displays the datum planes. DTM1, 2, and 3 are default datums used to create the base feature. DTM4 and 5 were introduced parallel to DTM1 and 3, respectively, and placed at the center of the part. These datums were used to locate the countersunk hole. Chosen constraints must locate the datum plane relative to the model without ambiguity.

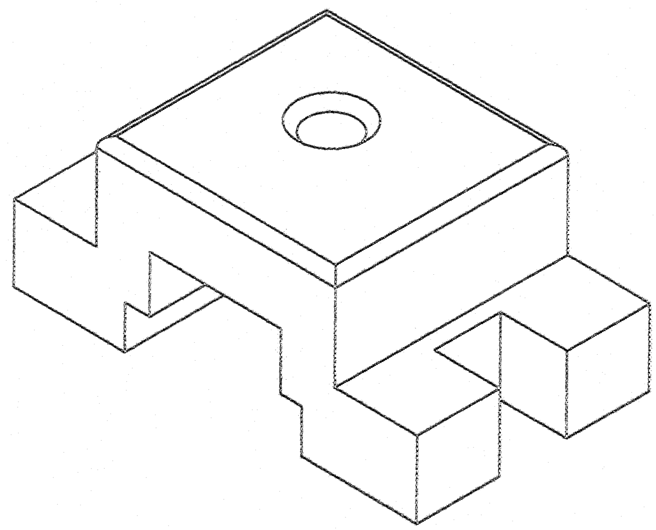


FIGURE 5.15(a) Clamp

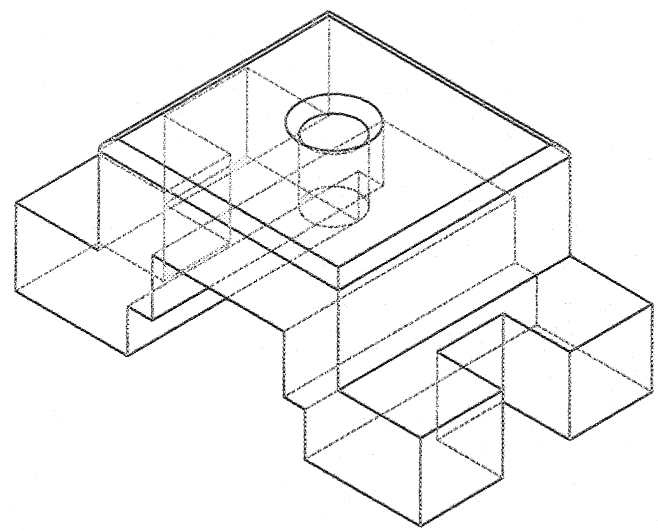


FIGURE 5.15(b) Clamp Displayed with Hidden Lines

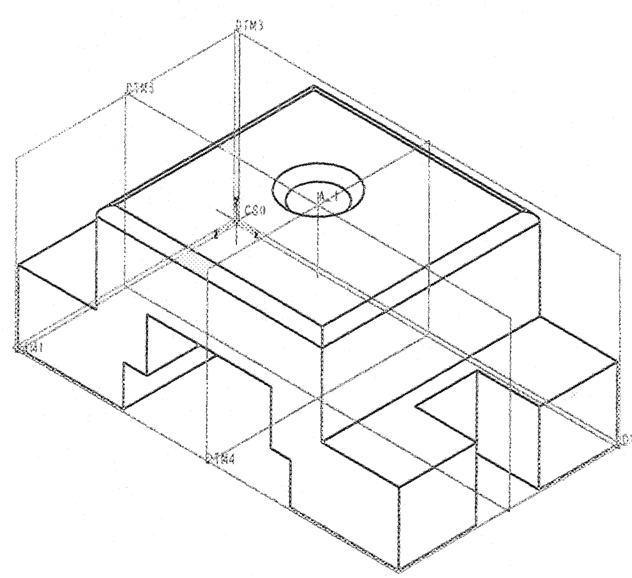


FIGURE 5.15(c) Clamp with Datum Features Displayed

5.3 PARENT–CHILD RELATIONSHIPS

Because solid modeling is a cumulative process, certain features must, by necessity, precede others. Those that follow must rely on previously defined features for dimensional and geometric references. The relationships between features and those that reference them are termed **parent–child relationships** (Fig. 5.16). Because children reference parents, features can exist without children, but children cannot exist without their parents.

The parent–child relationship is one of the most powerful aspects of parametric design. When a parent feature is modified, its children are automatically recreated to reflect the changes in the parent feature's geometry. It is therefore essential to reference feature dimensions so that design modifications are correctly propagated through the model. As an example, the modification to the length of a part is automatically propagated through the part and will affect all children of the modified feature.

5.4 ASSEMBLIES

Just as parts are created from related features, so **assemblies** are created from related parts [Fig. 5.17(a)]. The progressive combination of parts and features into an assembly creates parent–child relationships based on the references used to assemble each component [Fig. 5.17(b)].

Just as features can reference part geometry, parametric design also allows creation of parts referencing assembly geometry. Assembly mode (Fig. 5.18) allows the designer to both fit parts together and design parts based on how they should fit together. Figure 5.18(a) shows the completed assembly of the motorcycle. In Figure 5.18(b) the outer

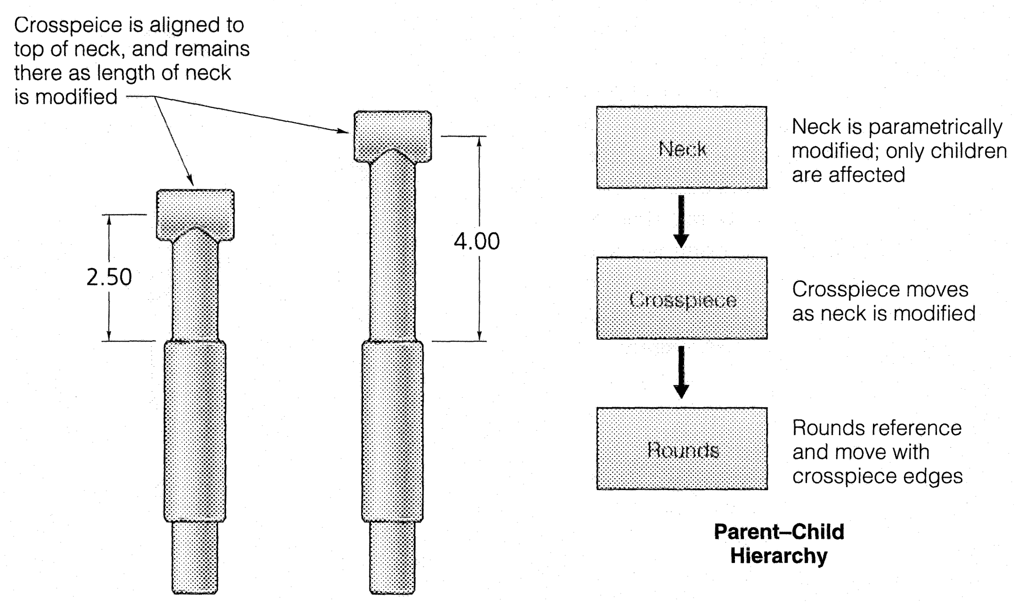


FIGURE 5.16 Parent–Child Hierarchy

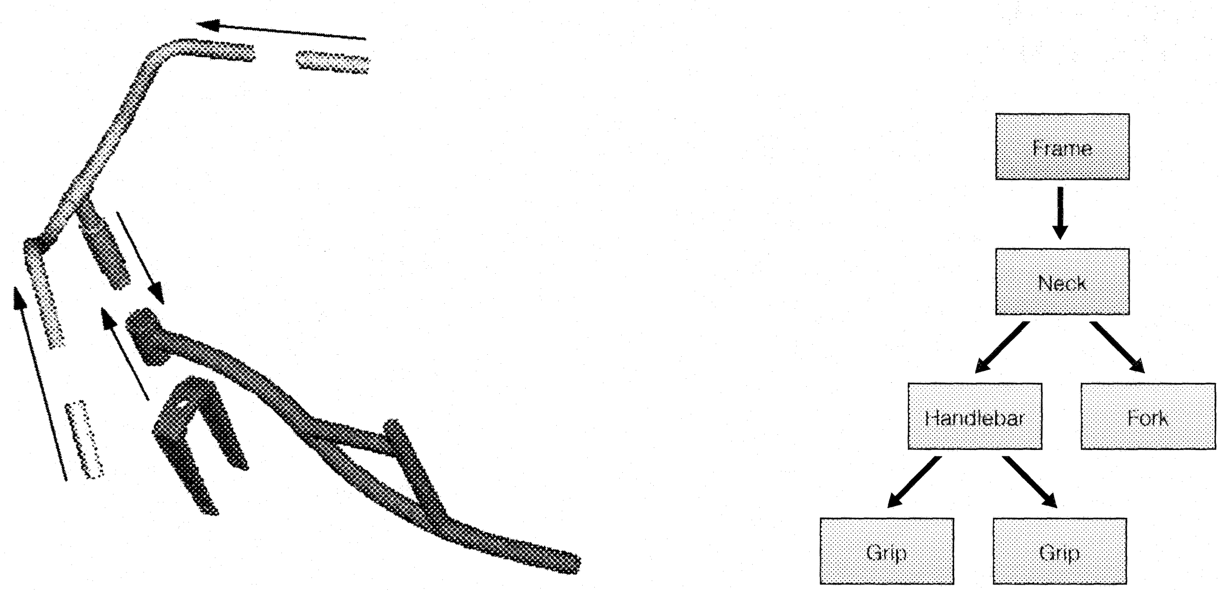


FIGURE 5.17(a) Assembly Hierarchy

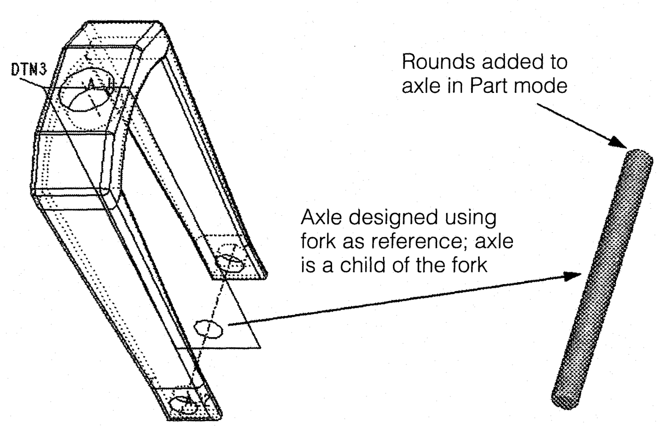


FIGURE 5.17(b) Assemblies and Components

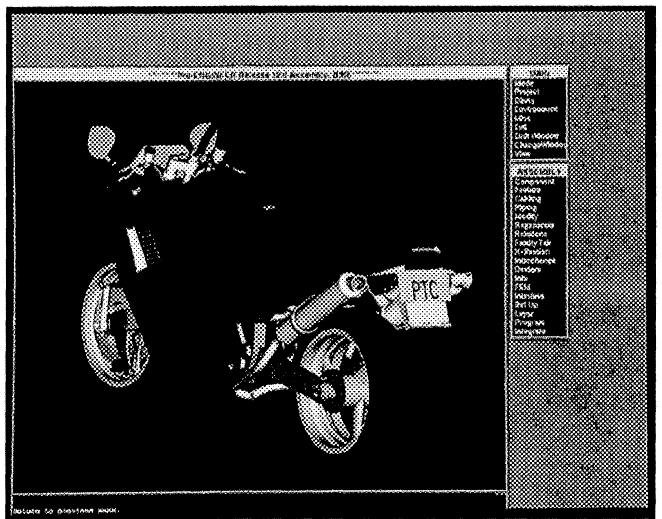


FIGURE 5.18(a) Motorcycle Assembly

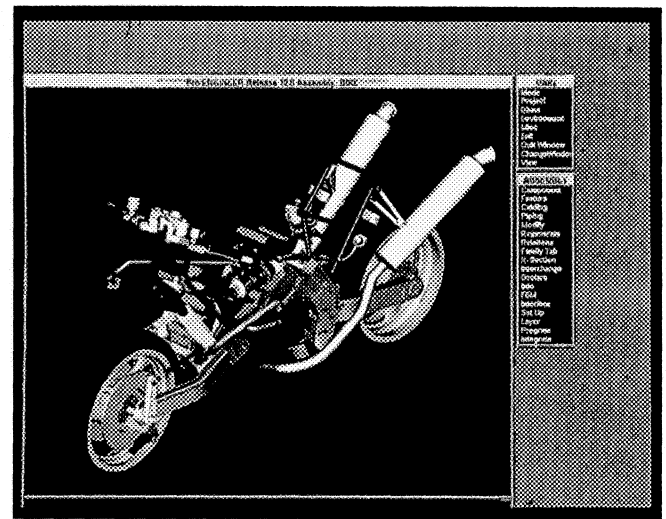


FIGURE 5.18(b) Motorcycle with Body Shell Removed

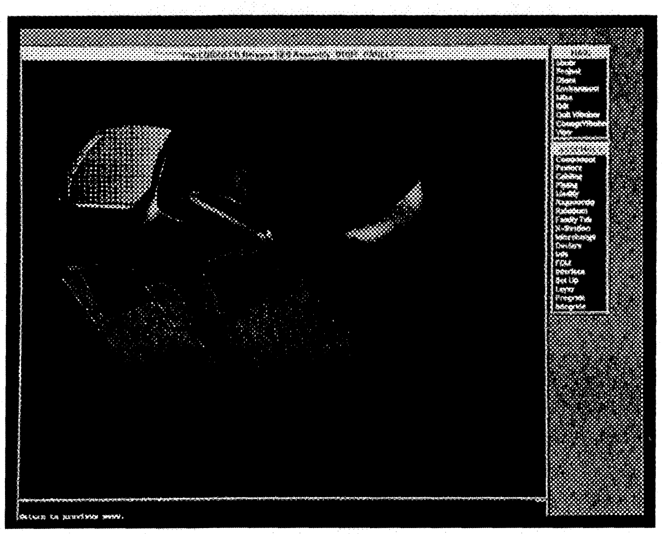


FIGURE 5.18(c) Body Panel Assembly

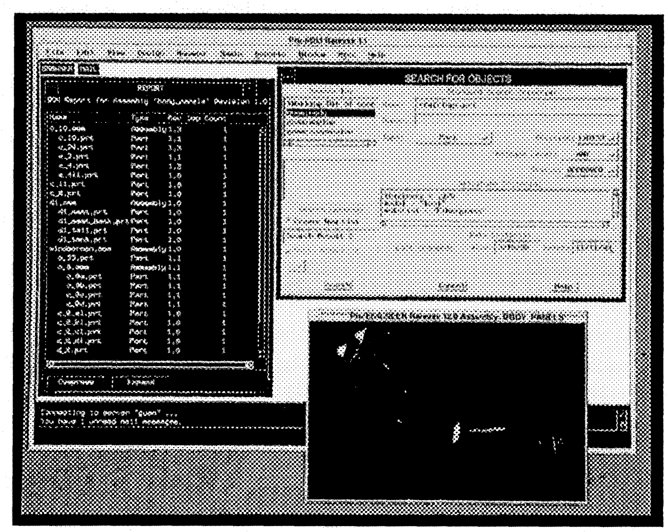


FIGURE 5.18(d) Bill of Materials

molded body panel subassembly [Fig. 5.18(c)] has been removed to view the engine and other components. In Figure 5.18(d) a report is being generated on the removed subassembly. Here, a bill of materials (BOM) is generated for the body panels.

5.5 CAPTURING DESIGN INTENT

A valuable aspect of any design tool is its ability to **render** the design and at the same time capture its **intent.** Parametric methods depend on the sequence of operations used to construct the design. The software maintains a *history of changes* the designer makes in specific parameters. The point of capturing this history is to keep track of operations that depend on each other. Whenever the system is told to change a specific dimension, it can update all operations that are referenced to that dimension.

For example, a circle representing a bolt hole may be constructed so that it is always concentric to a circular slot. If the slot moves, so does the bolt circle. Parameters are usually displayed in terms of dimensions or labels, and serve as the mechanism by which geometry is changed. The designer can change parameters manually by changing a dimension or by referencing them to a variable in an equation (**relation**) that is solved either by the modeling program itself or by external programs such as spreadsheets.

Parametric modeling is particularly useful in modeling whole **families** of similar parts and in rapidly modifying complex 3D designs. It is most effective in working with designs where changes are likely to consist of dimensional changes rather than radically different geometries.

Feature-based modeling, as already discussed, refers to the construction of geometry as a combination of **form features.** The designer specifies features in engineering terms, such as holes, slots, or bosses, rather than geometric

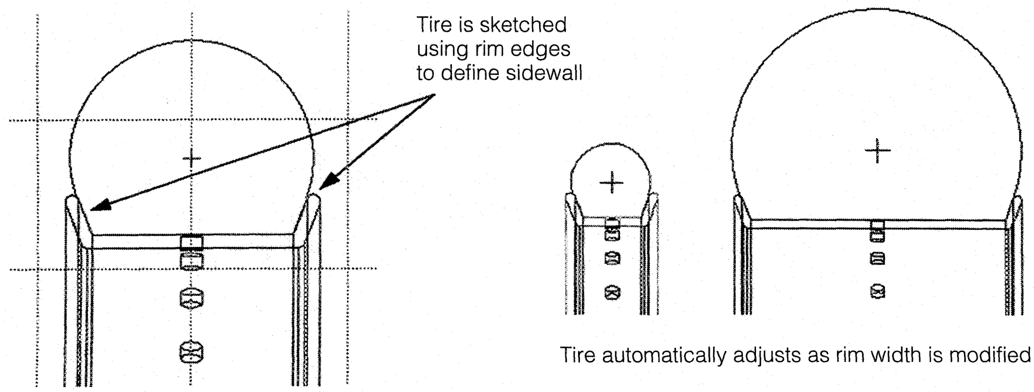


FIGURE 5.19 **Section Sketch of Tire and Rim**

terms, such as circles or boxes. Features can also store nongraphic information useful in activities such as a drafting, NC, finite-element analysis, and kinematics analysis.

The concept behind capturing design intent is to incorporate engineering knowledge into a model by establishing and preserving certain geometrical relationships. The wall thickness of a pressure vessel, for example, should be proportional to its surface area and should remain so even as its size changes. Parametric design captures these relationships in several ways.

Implicit Relationships. Implicit relationships occur when new model geometry is sketched and dimensioned relative to existing features and parts. An implicit relationship is established, for instance, when the section sketch of a tire (Fig. 5.19) uses rim edges for reference.

Patterns. Design features often follow a geometrically predictable pattern. Features and parts are patterned in parametric design by referencing either construction dimensions or existing patterns. One example of patterning is a wheel hub with spokes (Fig. 5.20). First the spoke holes are radially patterned. The spokes can then be strung by referencing this pattern. Any modification of a pattern member affects all members of that pattern. This helps capture design intent by preserving the duplicate geometry of pattern members.

Explicit Relations. Whereas implicit relationships are merely implied by creating a feature, an explicit relation is entered mathematically by the user. This equation relates part and feature dimensions in the desired manner. An explicit relation might be used, for example, to ensure that any number of spoke holes will be spaced evenly around a wheel hub (Fig. 5.21).

Family Tables. Family tables can create part families from generic models by tabulating dimensions or the presence of certain features or parts. A family table might be used, for example, to catalog a series of wheel rims of varying width and diameter (Fig. 5.22).

Modeling serves to incorporate the features and parts of a complex design while properly capturing design intent to provide flexibility in modification. Each parametric design model may be seen as a careful synthesis of physical and intellectual design (Fig. 5.23).

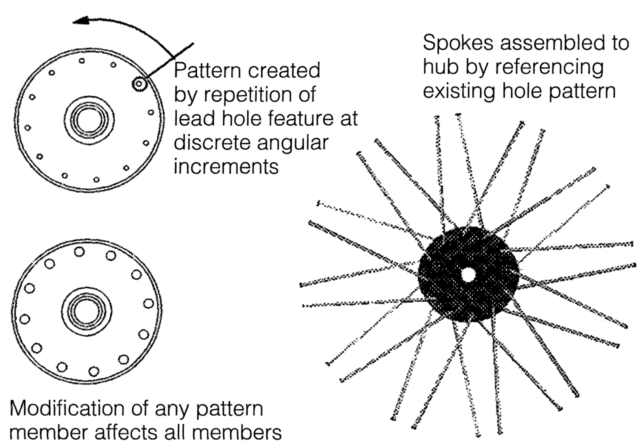
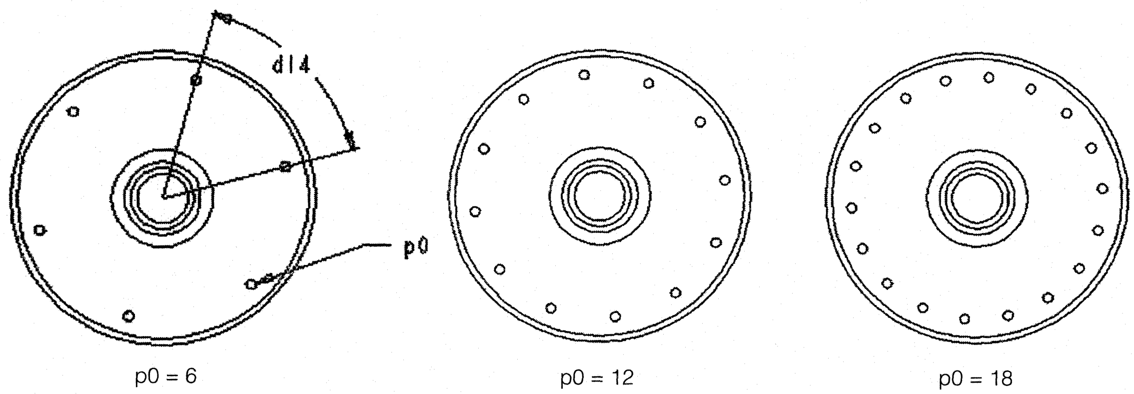


FIGURE 5.20 **Wheel Hub with Spokes Designed via a Pattern**

5.6 THE SKETCHER ENVIRONMENT

Sketcher techniques are used in many areas of parametric design. The aim of the **sketcher**, like that of parametric design, is to enable the quick and simple creation of geometry for your model. The sketcher requires you to create and dimension this geometry, but during the sketching process you do not have to be concerned with dimensional sizes or the creation of perfect and accurate geometry. Creating sections in sketcher mode is easy to do, and there are only a few steps to remember.

1. **Sketch the section geometry.** Use sketcher tools to create the section geometry.

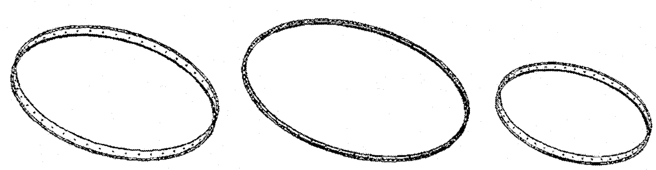


Relation: `d14 = 360/p0`

Where: `d14` = angular separation between pattern instances
`p0` = number of pattern instances

FIGURE 5.21 Relations and Feature Creation

2. **Dimension and align the section.** Use a dimensioning scheme that you want to see in a drawing or that makes sense for controlling the characteristics of the section. Align the section geometry to a datum feature or to a part feature.
3. **Regenerate the section.** Regeneration solves the section sketch based on your dimensioning scheme.
4. **Add section relations.** Add relations to control the behavior of your section.



Name	Diameter	Width
MOUNTAIN	24.00	1.25
ROAD	26.00	0.50
DIRT	18.00	1.00

FIGURE 5.22 Family Table

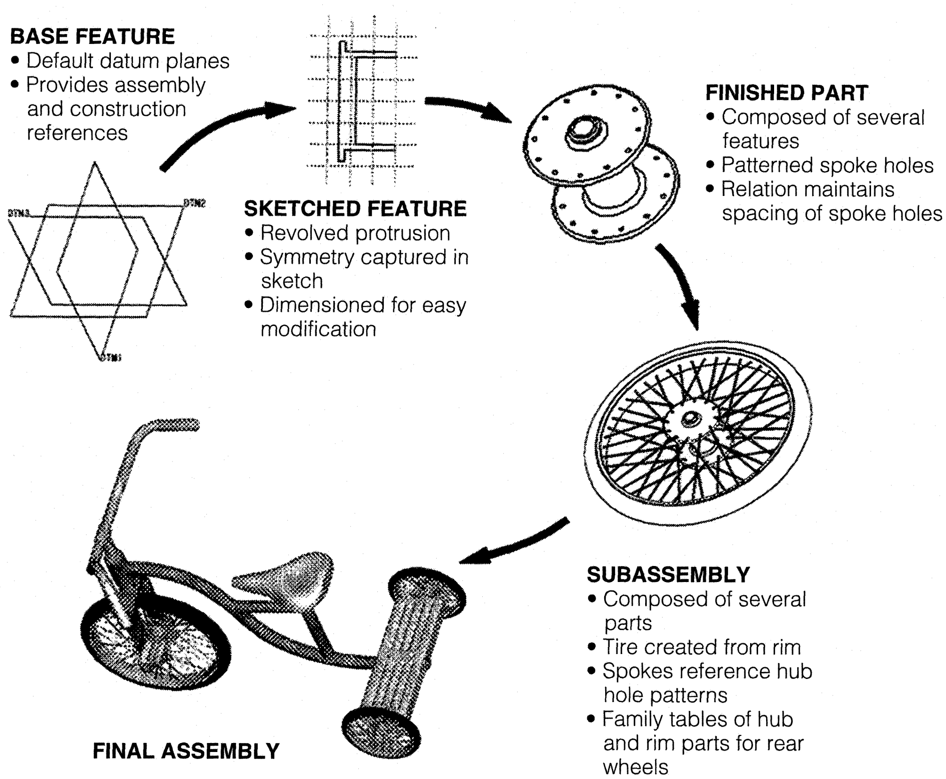


FIGURE 5.23 Parametric Design

5.6.1 The Sketcher

Sketcher mode serves to establish 2D sections that are the basis for the 3D feature being created. In order to understand just how powerful the sketcher is on a parametric design system, you need only look at what the sketching process has been throughout the ages. *Sketching is a way, simply and efficiently, to establish the basic design and intent of a designer-engineer* on paper, and now possible on a CRT.

In the past, designers have sketched on paper, showing lines, arcs, circles, and other geometric shapes in rough, simplified outline and internal forms. The sketched shapes are assumed to be what they *sort of* look like. Round shapes approximating a circle are assumed by the sketch reader to be circles, curved shapes are assumed to be arcs, and lines drawn straight up or down are assumed to be vertical. Lines drawn left to right are assumed to be horizontal. Lines sketched at an angle are straight lines that are angled. Dimensions roughly sketched on a less than perfect drawing of a part are assumed to represent the exact perfect shape desired by the person sketching. All this seems obvious to most people involved in engineering design. But now we have a whole new tool. With the introduction of parametric design we can sketch on the screen and allow the system to make all the assumptions that were traditionally made by a person creating a sketch or reading a sketch. These assumptions include, but are not limited to, the following: symmetry, tangency, parallelism, perpendicularity, equal angles, same-size arcs and circles, coincident centers, colinerarity.

AutoCAD Designer compares your sketch with a set of **rules.** AutoCAD Designer cleans up the sketch after it finds a group of rules that fit the sketch. If AutoCAD Designer exhausts all possible rules without solving the sketch, your sketch is underconstrained. As an example, AutoCAD Designer parametric-based system applies the following rules to a sketch, in the following order.

1. A line sketched nearly horizontal is horizontal.
2. A line sketched nearly vertical is vertical.
3. Two arcs or an arc/circle and a line sketched nearly tangent are tangent.
4. Two arcs or circles whose centers are sketched nearly coincident are concentric.
5. Two lines sketched nearly overlaying along the same line as each other are collinear. Lines sketched nearly parallel are parallel. Lines sketched nearly perpendicular are perpendicular. The lines must be attached for perpendicularity to be inferred automatically.
6. Any arcs and circle sketched with nearly the same radius have the same radius.
7. Geometric forms (lines, arcs, etc.) are attached using the endpoint of one form and the near point of the other.

All parametric design systems make similar assumptions when sketching.

5.6.2 Regenerating a Section Sketch

During regeneration, the system checks to make sure it understands your dimensioning scheme and that you have created a complete and independent set of parameters. It analyzes your section based on the geometry you have sketched and the dimensions you have created. In the absence of explicit dimensions, implicit information based on the sketch may be used. Here is a table of implicit information that Pro/ENGINEER uses to regenerate a section (you can see the similarity to AutoCAD's Designer).

Rule: Equal radius/diameter
Description: If two or more arcs or circles are sketched with approximately the same radius, they are assigned the same radius value.

Rule: Symmetry
Description: Entities sketched symmetrically about a centerline are assigned equal values with respect to the center line.

Rule: Horizontal and vertical lines
Description: Lines that are approximately horizontal or vertical are considered to be exactly so.

Rule: Parallel and perpendicular lines
Description: Lines that are sketched approximately parallel or perpendicular are considered to be exactly so.

Rule: Tangency
Description: Entities sketched approximately tangent to arcs or circles are assumed to be tangent.

Rule: 90°, 180°, 270° arcs
Description: Arcs are considered to be multiples of 90° if they are sketched with approximately horizontal or vertical tangents at the endpoints.

Rule: Collinearity
Description: Segments that are approximately collinear are considered to be exactly so.

Rule: Equal segment lengths
Description: Segments of unknown length are assigned a length equal to that of a known segment of approximately the same length.

Rule: Point entities lying on other entities
Description: Point entities that lie approximately on lines, arcs, or circles are considered to be exactly on them.

Rule: Centers lying on the same horizontal
Description: Two centers of arcs or circles that lie approximately along the same horizontal direction are set to be exactly so.

Rule: Centers lying on the same vertical
Description: Two centers of arcs or circles that lie approximately along the same vertical direction are set to be exactly so.

These rules are applied to all Pro/ENGINEER sketches.

The following section provides an example of the sequence of steps involved in modeling with Pro/ENGINEER. Though different systems use different command names, the capabilities, and in many cases the steps, are the same. Note that the following part would take an experienced designer 3–5 minutes to create.

5.6.3 Creating a Part Using Sketched and "Pick and Place" Geometry

The premise of the sketcher in parametric feature-based design is to create quick-and-simple geometry for your model. The sketching process enables you to create and dimension the geometry for a feature or set of features on your design. Remember, during the sketching process you need *not* concern yourself with creating perfect geometry or accurate dimension values. You might modify your dimensions later in the design process.

In Figure 5.24 the creation of a simple part is described. After the **units of measurement, default datums,** and other **setups** are performed, the sketcher is entered by selecting a feature type. Here the feature to be used as the base feature is an **extruded protrusion.** The default datums [Fig. 5.24(a)] are used to select the sketching plane and orient the sketch. DTM3 is the sketch plane. A sketched section can start the model. The following commands were given.

Figure 5.24(a). Using the datum planes to set up and orient the sketch.

Feature--Create--Datum--Default
Feature--Create--Datum--Coordinate System--Default
Feature--Create--Solid--Protrusion--Extrude--Solid--Single--One Side--Blind--Plane (pick DTM3)**--Okay--Top** (pick DTM2)

Figure 5.24(b) and Figure 5.24(c). Drawing lines with the sketcher. Note that the lines are not perfectly oriented (vertically or horizontally). They are drawn in approximately the desired shape of the base protrusion, with no regard to actual size.

Sketch--Line (sketch the geometry as shown)
Regenerate (system asks you to locate with respect to the part)
Alignment (align the ends of lines to the datums and the lines along the datums to the datums themselves)

Figure 5.24(d). Dimensioned sketch has **symbols** for dimensions, since it has not been regenerated yet.

Dimension (add required dimensions to locate the geometry to the datum planes and describe the features of the geometry)

Figure 5.24(e). Regenerated and aligned sketch has sketch dimension values (which are not correct at this stage).

Regenerate (system responds with "Regeneration completed successfully")

Figure 5.24(f). Modify the dimension values to reflect the correct design requirements. Notice that after the regeneration the section geometry looks quite different from the original sketch!

Modify (pick each dimension in succession and change the value to the required size)
Regenerate

Figure 5.24(g) and Figure 5.24(h). Enter depth of the protrusion (here 5.00). The system responds with "**PROTRUSION** has been created successfully." Show the part in a pictorial view and change it to isometric projection.

View--Default (displays the part in a rotated projection)
Environment--Isometric
View--Default

Figure 5.24(i) and Figure 5.24(j). The base feature is now complete. You may modify any dimension at this stage of the design. Let's change the depth of the part from 5.00 to 6.00.

Modify (pick a line on the part—all the protrusions features will be displayed)
Modify (pick the depth dimension 5.00 and type the new value 6.00)
Regenerate (the model will update itself using the new depth)

Figure 5.24(k). Now let us create a thru-hole on the angled surface. The hole will have a 1.25 diameter and will be placed at the center of the plane. The hole command can be a pick-and-place feature or a sketched feature. We will use the pick-and-place version. Note that the hole will be a child of the surface on which it is created and of surfaces from which it is located.

Feature--Create--Hole--Single--Linear--Straight--One Side--Thru All (enter diameter: 1.25)
(select placement plane—choose the angled surface)
(select two edges, axes, planar surfaces, or datums for dimensioning—select DTM3 and give a value of 3.00)
(select the second reference—select the front lower edge line of the angled surface and give a value of 2.00. Note that 2.00 will only place the hole near the plain center. Later you can modify this value, after you get the system to provide the length of the angled edge line.)

Figure 5.24(l) and Figure 5.24(m). Using the system **Info** command we can get the exact length of the angled edge line and use this dimension to modify the placement of the hole. At the same time, change the size of the hole to ∅ 1.500.

Info--Measure--Vertex (pick the ends of the angled line—the system gives the distance as 3.60555)
Modify (pick the hole—the system displays the diameter and the location dimensions of the hole. Pick the ∅ 1.25 dimension and change its value to ∅ 1.500. Next pick the 2.00 dimension and change it to 3.60555/2. This will make it half the length of the edge line.
Regenerate

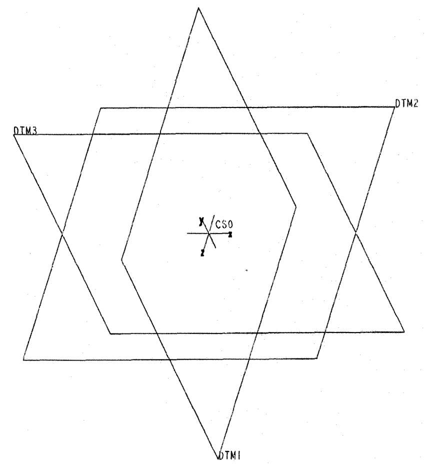


FIGURE 5.24(a) Default Datums

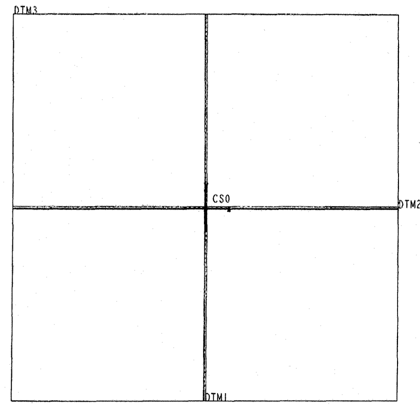


FIGURE 5.24(b) Sketching Plane

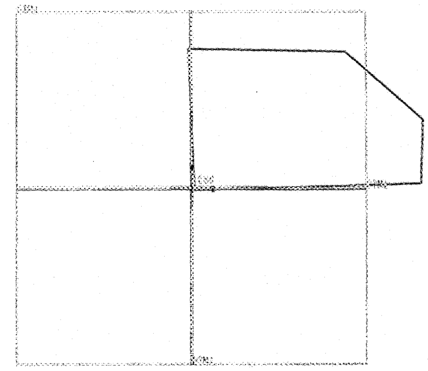


FIGURE 5.24(c) Sketching the Base Feature

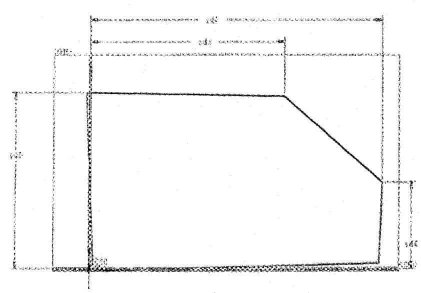


FIGURE 5.24(d) Dimension Symbols

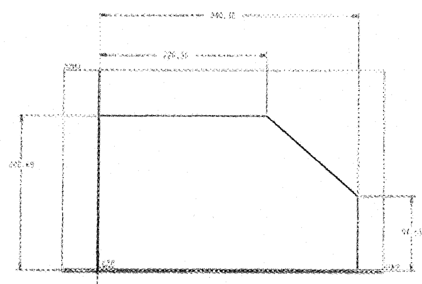


FIGURE 5.24(e) Aligned and Regenerated Sketch with Sketch Dimension Values

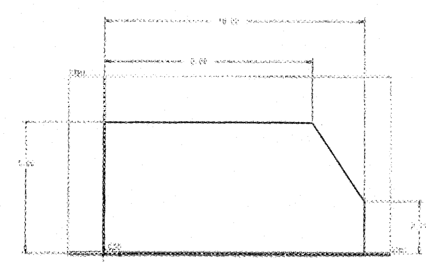


FIGURE 5.24(f) Regenerated Sketch with Modified Dimension Values

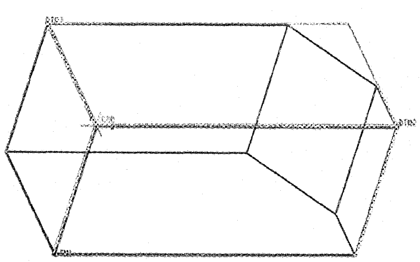


FIGURE 5.24(g) Default Pictorial View

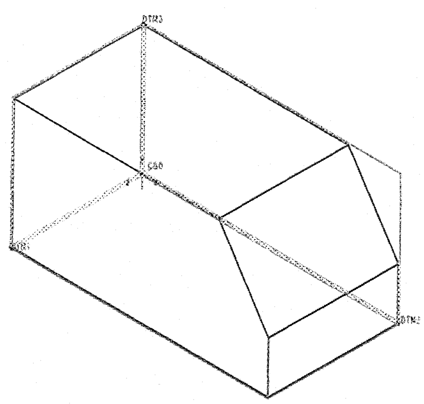


FIGURE 5.24(h) Isometric View

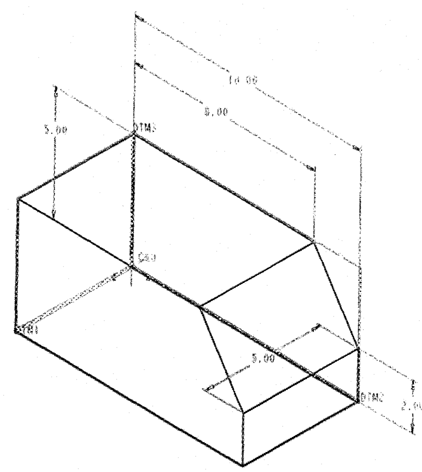


FIGURE 5.24(i) Base Feature with Dimensions Shown and Available to Modify

Figure 5.24(n) and Figure 5.24(o). The hole needs a chamfer of 45° × .20. Add the chamfer using the chamfer feature. The chamfer is a pick-and-place feature. The chamfer is a child of the hole.

Feature--Create--Chamfer--Edge--45xd (enter chamfer dimension for d:.2) (select one or more edges to chamfer—pick the holes edge twice—both sides, since the hole is split in half when using Pro/ENGINEER)

Figure 5.24(p). Lastly, we will shade the part.

View--Cosmetic--Colors--Define--Set (define new colors and set the surfaces)
Shade--Display

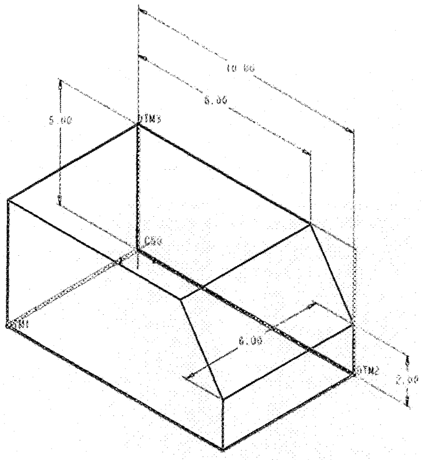


FIGURE 5.24(j) Modifying the Depth Dimension from 5.00 to 6.00

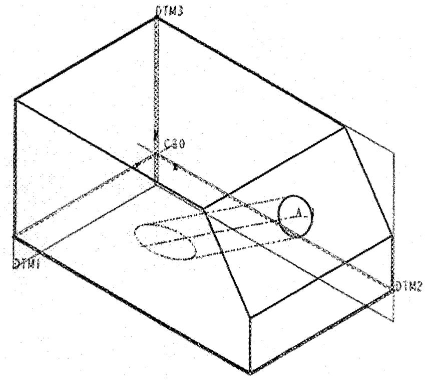


FIGURE 5.24(k) Adding a Hole to the Angled Surface

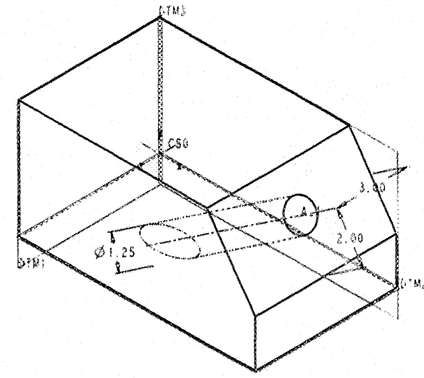


FIGURE 5.24(l) Hole Dimensions Available for Modification

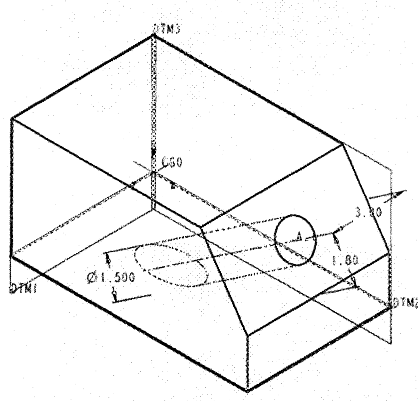


FIGURE 5.24(m) Modifying the Hole Diameter from ∅ 1.25 to ∅ 1.50

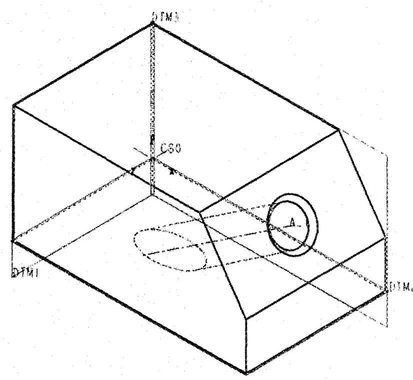


FIGURE 5.24(n) Adding a Chamfer Feature

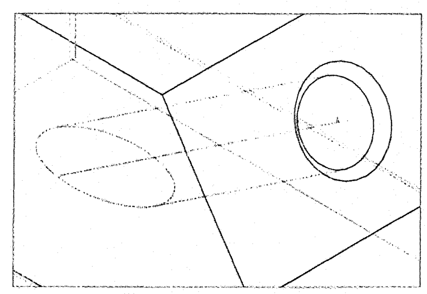


FIGURE 5.24(o) Enlarged View of Chamfer

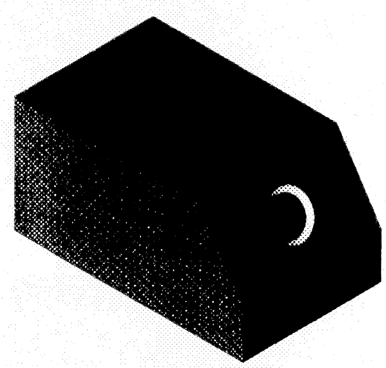


FIGURE 5.24(p) Shaded Part

5.7 GENERATING DRAWINGS

You can create **drawings** of all parametric design models (Fig. 5.25) or by importing files from other systems. All model views in the drawing are **associative:** If you change a dimensional value in one view, other drawing views update accordingly. Moreover, drawings are associated with their parent models: Any dimensional changes made to a drawing are automatically reflected in the model; any changes made to the model (i.e., addition of features, deletion of features,

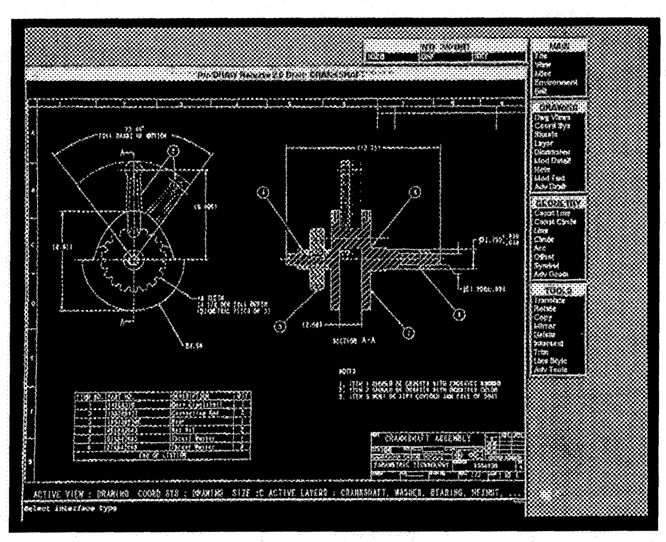


FIGURE 5.25 Dimensioned Drawing Created with Pro/DRAW

Applying Parametric Design . . .

PARAMETRIC DESIGN ON PC'S

This chapter will overview Pro/ENGINEER™ and the Applying Parametric Design boxes from Chapters 6–29. But there are many high-quality parametric design systems available on the market. AutoCAD Designer is a low-cost parametric-based system that can be added on to the latest version of AutoCAD. It is affordable and will allow you to experience parametric design on a PC. Computervision, SDRC's IDEAS, IBM, and Pro/ENGINEER are a few of the dominant high-end parametric design systems found throughout industry. Ford, Chrysler, John Deere, FMC, Boeing, and Lockheed are some of the many companies switching to high-end parametric design systems. If training on these systems is unavailable at a local college, AutoCAD Designer will provide all of the basics that can be learned at home on your PC.

AutoCAD's Designer™ lets you generate many types of information about your design—its mass properties, a drawing, or a base model (Fig. A is a solid model of the Dremel Gun body). To get this information, you must first model the parts that make up the design. To begin the design process, analyze your design. Then break it down into its basic components, or building blocks. Next, identify the most fundamental feature of the part as the first feature to sketch.

Features are the basic building blocks you use to build a part. AutoCAD Designer features "understand" their fit and function. For example, a hole feature knows its shape and part location and that it has a negative volume. So to create a hole in a solid, all you have to do is input a specific position as the place where you want to put the hole. Also, when you edit a feature, the entire part automatically updates in a logical way. The Dremel Gun body has a number of such holes incorporated into its design.

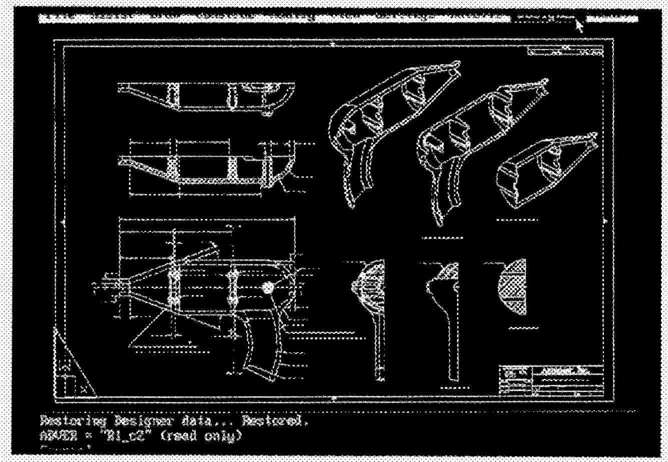


FIGURE B Detail Drawing of Dremel Body with Multiple Dimensions and Sections

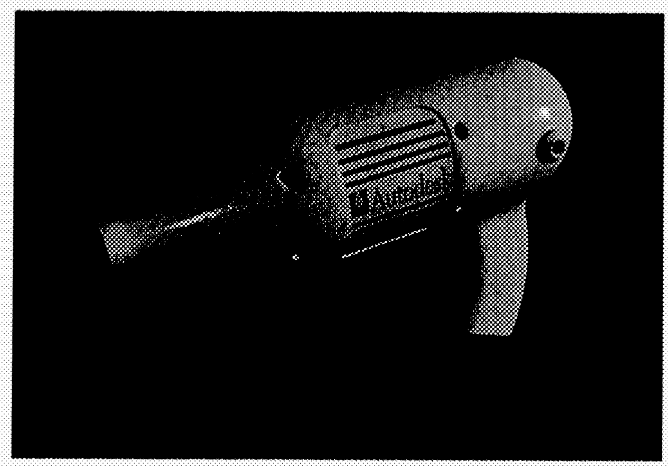


FIGURE C Assembly of Dremel Gun

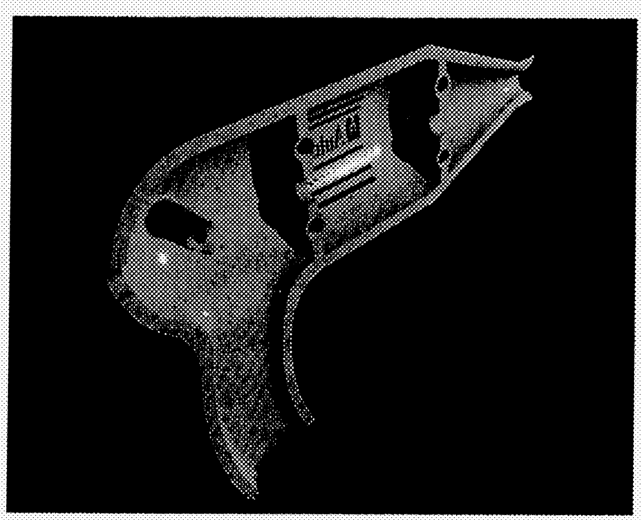


FIGURE A Solid Parametric Model of the Dremel Gun Body

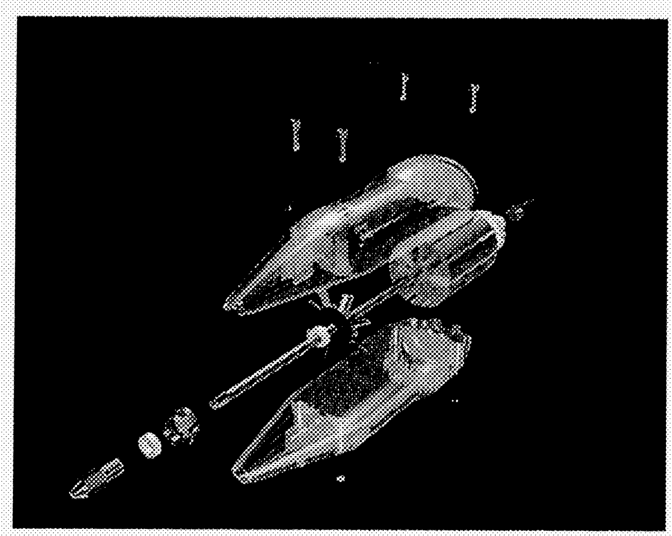


FIGURE D Exploded Assembly of Dremel Gun

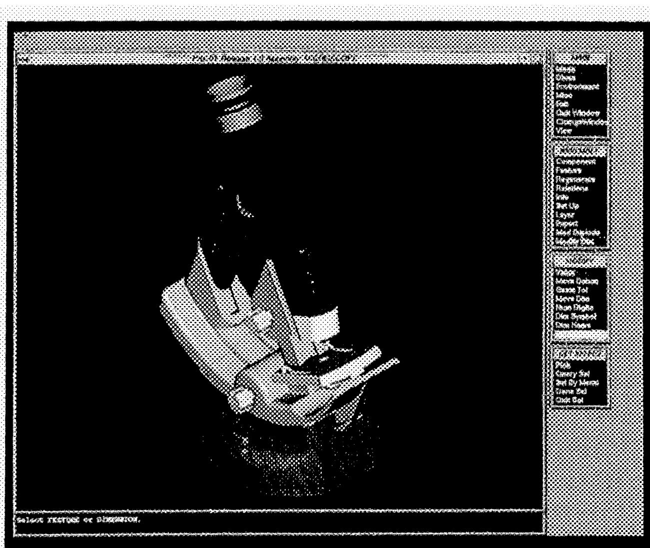


FIGURE E Solid Model of Microscope

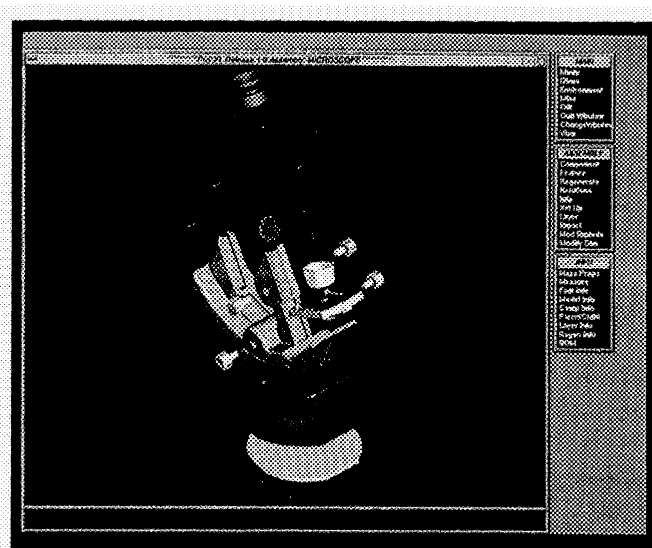


FIGURE H Exploded Solid Model of Microscope

FIGURE F Shelled Model of the Microscope Base

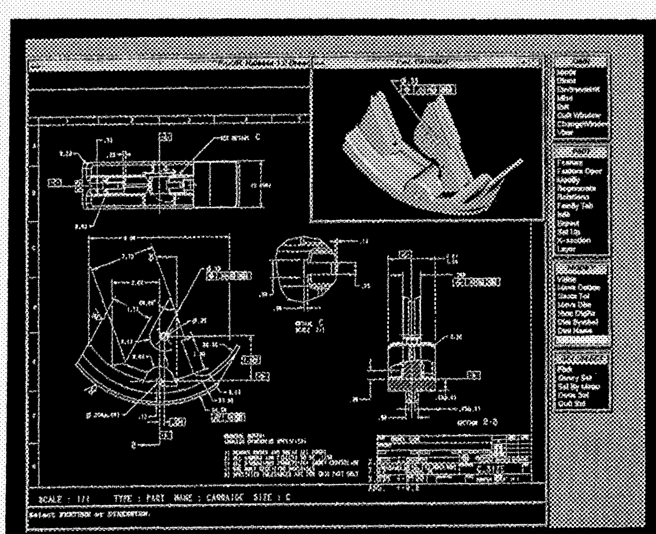


FIGURE G Detail Drawing of Microscope Component

In sketching your features, you can use many of the AutoCAD 2D entities as you normally would. However, you don't have to be concerned with the accuracy of the sketch. Lines can be off angle, arcs can have unequal radii, and overall dimensions and relationships can be incorrect. Simple commands help apply logical geometric constraints to the sketch. These constraints clean up the sketch geometry according to the current system settings. The constraints close endpoints, align parallel lines, and snap entities to horizontal and vertical angles. Additional constraints and parametric dimensions are applied to control the size and shape of the feature.

You can begin documentation of a finished part at anytime using the AutoCAD Designer drawing commands. By identifying the drawing views, AutoCAD Designer automatically removes hidden lines and places the feature dimensions on the drawing. A series of sections of the Dremel Gun body is shown in Figure B, along with dimensions required for manufacture. If you make changes to the part and add dimensions and features, AutoCAD Designer updates the drawing along with the part.

Multiple parts are combined to build an assembly that is tied together through global parameters. Figure C shows a completely assembled Dremel tool. You can also create assembly drawings the same way you create parts drawings, and as shown in Figure D, exploded views of the assembly are easily displayed.

Pro/JR.™, from **Parametric Technology Corp.**, is a parametric, feature-based solid-modeling system for the design through documentation of mechanical parts and assemblies (see Fig. E, a solid model of a microscope). The Pro/JR.'s full associativity ensures that a change made anywhere is reflected in all engineering stages of the design-through-manufacturing sequence. This is an entry-level version of Pro/ENGINEER. Pro/JR. is a more expensive PC-based parametric design system than AutoCAD's Designer. But for companies wishing to move from drafting-orientated 2D-3D systems to the more advanced capabilities of 3D parametric modeling, this is an excellent midrange system.

(Continues)

(Continued)

Pro/JR. enables engineers and designers to design parts quickly and easily by selecting "pick and place" features from a menu or by creating sketched features directly on the part. The shelled component of the microscope base (see Fig. F) is an example of a part created via the shell command in Pro/JR.

After a part is modeled with Pro/JR., fully associative drawings are created directly from the design model. Any changes made to the drawing will automatically be reflected in the solid model. Likewise, changes to the model are captured in the drawing. Figure G shows a detail drawing of one piece of the microscope. Drawings are created from the design model without the necessity of drafting views and creating traditional 2D drawings of the part. Pro/JR., as with Pro/ENGINEER, has built-in ANSI or ISO standards for displaying appropriate design-driven dimensions and geometric tolerancing feature specifications. Assembly models and drawings created with Pro/JR. can be displayed in an exploded state to expose hidden components, as shown in Figure H in the exploded view of the microscope model.

Regardless of the parametric design system available to you as an engineering student or as an employee of a design engineering company, this technology will be the primary form of engineering communication and documentation in the near future. Any available training on a parametric design system will greatly enhance your marketability as a beginning engineer or designer.

dimensional changes, etc.) in Part, Sheet Metal, Assembly, or Manufacturing mode are also automatically reflected in the corresponding drawings.

This section describes drawing mode for Pro/ENGINEER, and how to create and manage drawings.

5.7.1 Drawing Mode and Basic Parametric Design

Drawing mode in parametric design provides you the basic ability to document solid models in drawings that share a two-way associativity with the model. *Any changes that are made to the model in Part or Assembly mode will automatically cause the drawing to update and reflect the changes. Any changes made to the model in Drawing mode will be immediately visible on the model in Part and Assembly modes.* Basic Pro/ENGINEER (without the optional module Pro/DETAIL) allows you to create drawing views of one or more models in a number of standard view types and to dimension them. In addition, you can annotate the drawing, manipulate the dimensions, and use layers to manage the display of different items on the drawing.

The optional module **Pro/DETAIL** may be used to extend the drawing capability, or as a stand-alone module for creating, viewing, and annotating models and drawings (Fig. 5.26). Pro/DETAIL supports additional view types and multisheets, and offers commands for manipulating items in the drawing and adding and modifying different kinds of textural and symbolic information. In addition, the abilities to customize engineering drawings with sketched geometry, create custom drawing formats, and make numerous cosmetic changes to the drawing are available.

Drawing parameters are saved with each individual drawing and drawing format. Drawing parameters determine the height of dimension and note text, text orientation, geometric tolerance standards, font properties, drafting standards, and arrow lengths. Parameter values are given defaults by the system.

When you regenerate a drawing, the drawing and the model that it represents are recreated, not simply redrawn. This means that if any of the model's dimension values were changed while in drawing model, regenerating the drawing causes the model to update these changes. The regenerated drawing displays the updated model and any changes that were made to it.

5.7.2 Storing Drawings

Some of the entities and information that you can create or modify in a drawing are saved with the model, rather than with the drawing. This is important to be aware of, since such changes made to the drawing may affect the model that it documents. An example of this occurs when you set a dimension as basic. The dimension becomes theoretically exact, and any tolerances this dimension had are removed.

Strictly *cosmetic* information is saved with the drawing; this information includes:

- All draft entities
- The view in which an entity is displayed
- The placement of an entity on the sheet

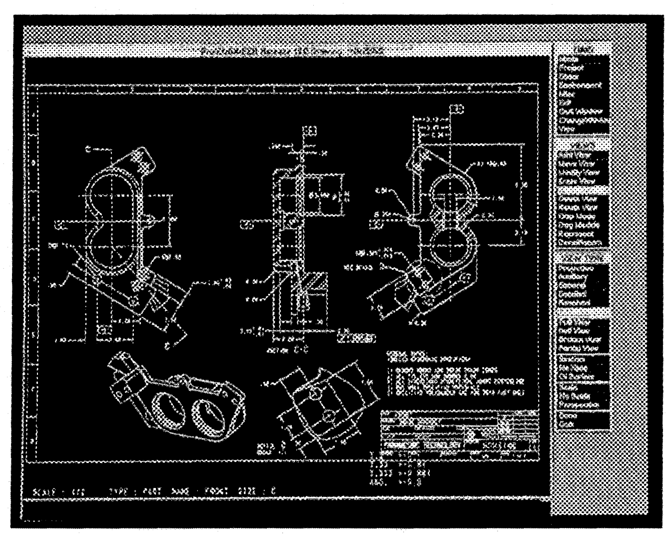


FIGURE 5.26 Detail Drawing Created with Pro/DETAIL

- Jogs and breaks in leaders and dimension lines
- The insertion of dimensions in notes
- The font, height, width, and slant angle of text

However, much of the information added to a drawing is saved with the model; this includes:

- Geometric tolerances (can also be saved in the drawing)
- Dimension information (reference and driven dimensions also), including:
  - Additional text
  - Standard/ordinate dimension type
  - Attached geometric tolerance list
  - Attached set datum or axis reference
  - Value and tolerance information
  - The difference between the primary and secondary units, when explicitly set by the user
  - Basic and inspection attributes
- Set datum and axis information
- Datum target point information
- Surface finishes, including type and value
- Layer membership information for all entities in the model

Whenever you save a drawing after making changes that affect the model, the model is saved with the drawing.

Reports can also be generated from the part or assembly. The race car in Figure 5.27(a) and Figure 5.27(b) is an example of a complex product design documented with a parametric design system. The body design [Fig. 5.27(c)], engine [Fig. 5.27(d)], and internal systems [Fig. 5.27(e)] contain the complete design database required to produce the vehicle. Information about the design can be extracted graphically using drawings and through the generation of reports [Fig. 5.27(f)].

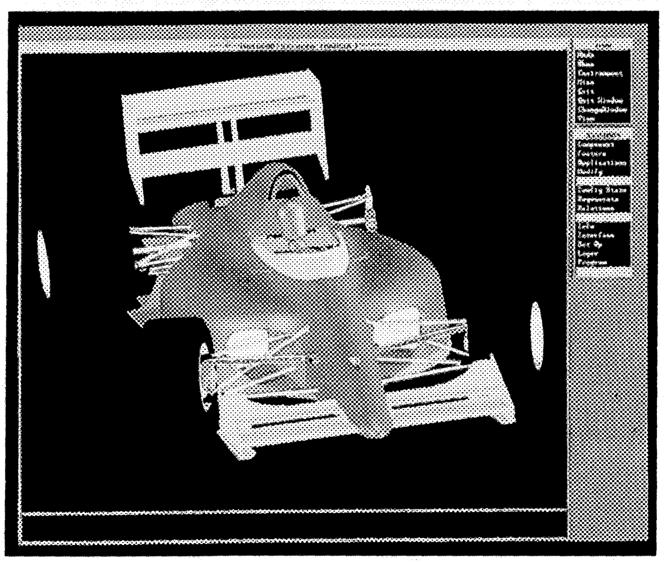


FIGURE 5.27(a) **Indy Race Car Designed with Pro/ENGINEER**

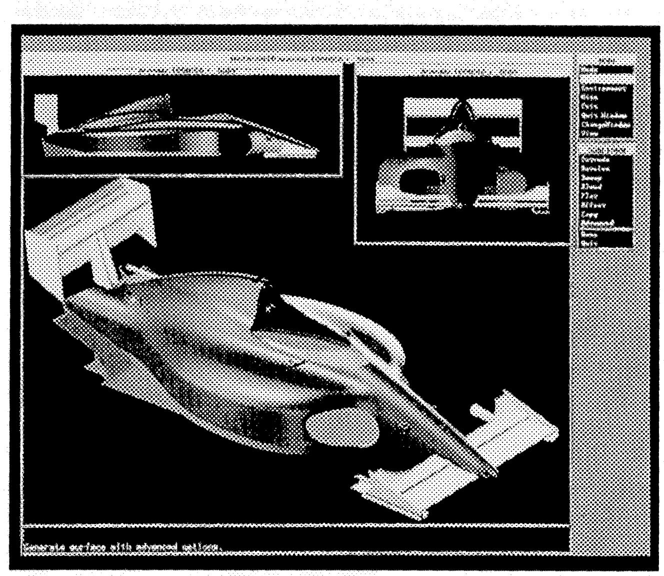


FIGURE 5.27(c) **Indy Car Body Design**

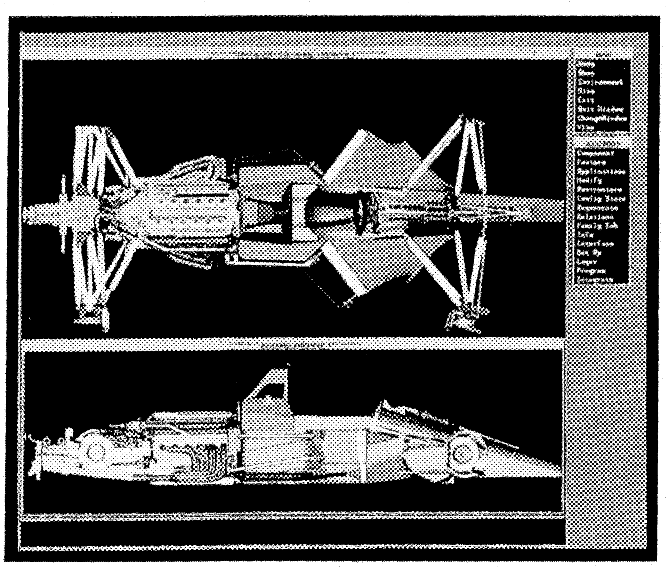


FIGURE 5.27(b) **Internal View of Indy Car**

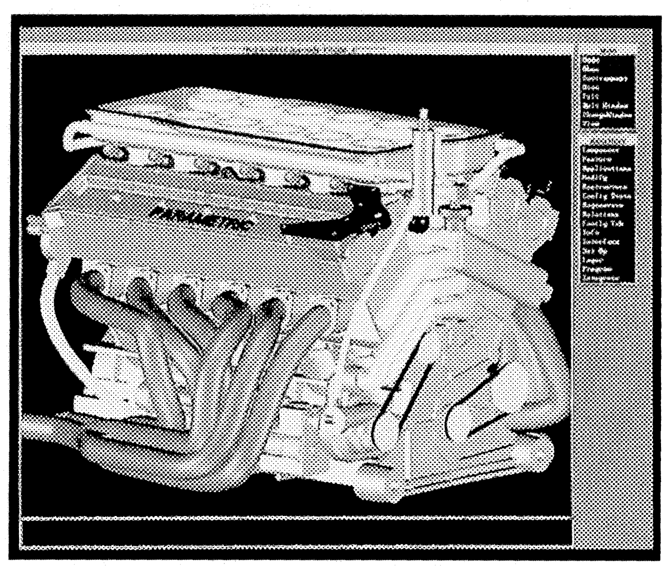


FIGURE 5.27(d) **Indy Car Engine**

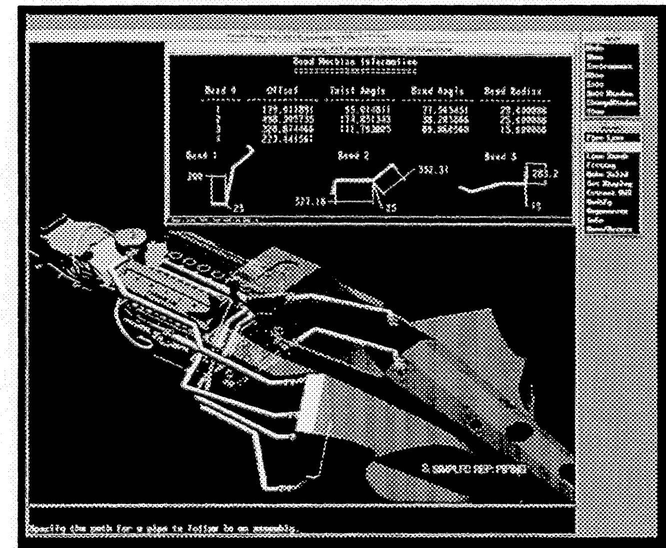


FIGURE 5.27(e) Indy Car Internal Systems

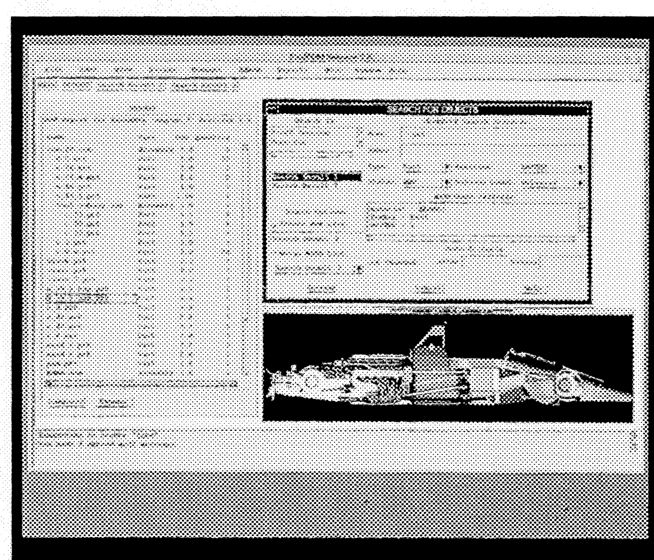


FIGURE 5.27(f) Generating Design Reports for Indy Car

5.8 MANUFACTURING AND PARAMETRIC DESIGN

Parametric design systems also provide the tools to program and simulate numerical control manufacturing processes (Fig. 5.28). The information created can be quickly updated should the engineering design model change. NC programs in the form of ASCII CL data files, tool lists, operation reports, and in-process geometry can be generated.

Pro/MANUFACTURING software will create the data necessary to drive an NC machine tool to machine a part (Fig. 5.29). It does this by providing the tools to let the manufacturing engineer follow a logical sequence of steps to progress from a design model to an ASCII CL data file that can be postprocessed into NC machine data (Fig. 5.30).

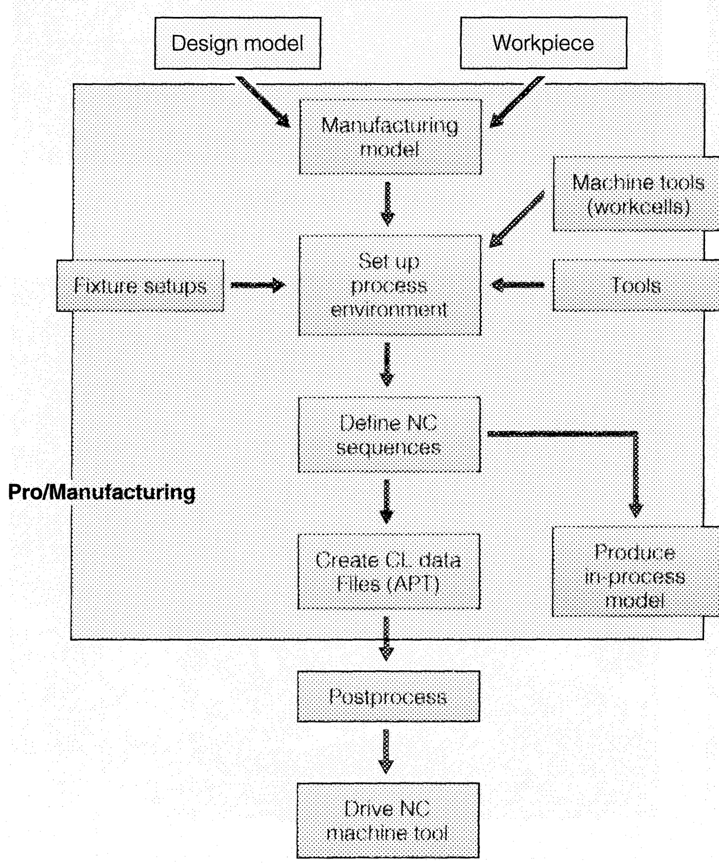


FIGURE 5.28 Pro/MANUFACTURING

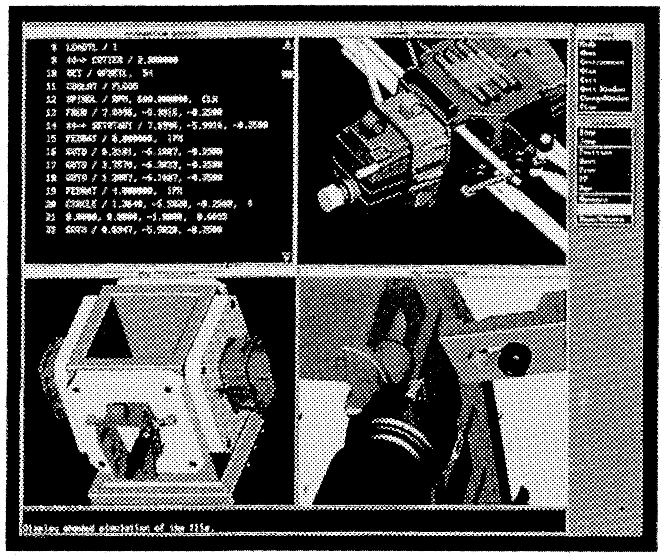


FIGURE 5.29 Fixturing and Part Machining

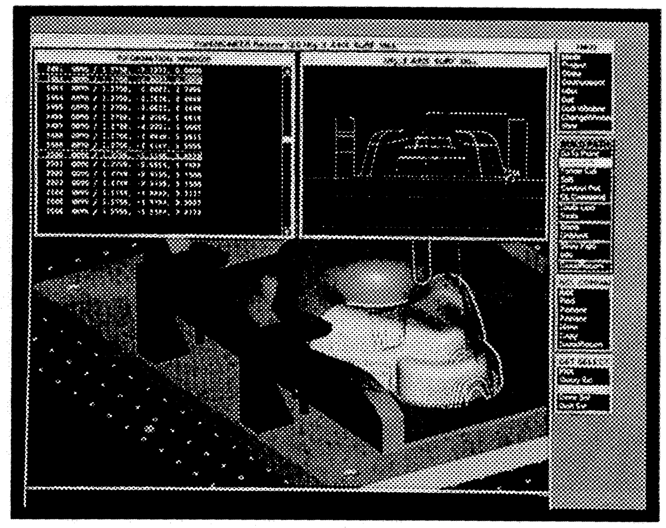


FIGURE 5.30 Part, Fixture, and NC Program

5.8.1 Design Model

The Pro/ENGINEER **design model**, representing the finished product, is the basis for all engineering (Fig. 5.31) and manufacturing (Fig. 5.32) operations. Features, surfaces, and edges are selected on the design model as references for each manufacturing operation (Fig. 5.33). Referencing the geometry of the design model sets up a parametric relationship between the design model and the workpiece. Because of this relationship, when the design model is changed, all associated manufacturing operations are updated to reflect the change. Parts, assemblies, and sheet metal parts may be used as design models.

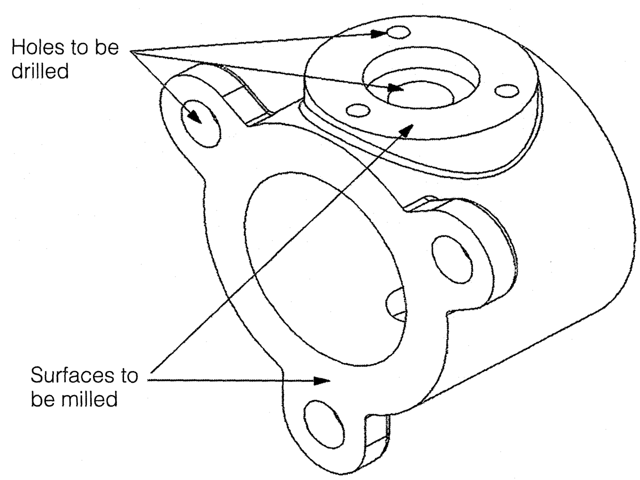


FIGURE 5.32 The Design Model—A Valve Housing

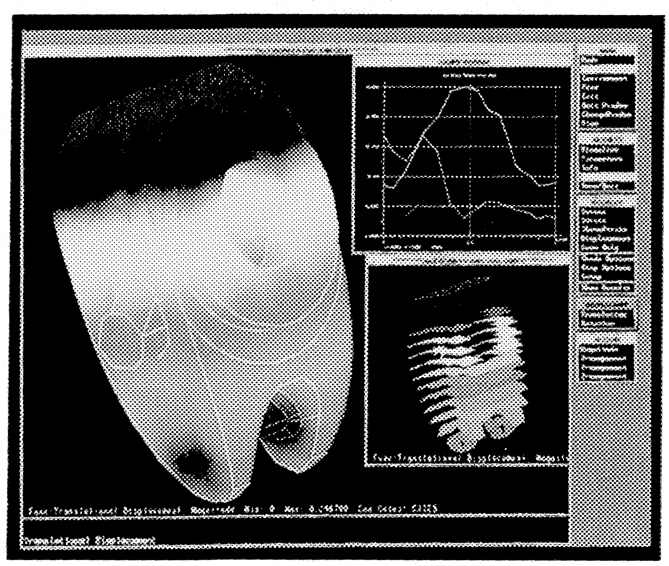


FIGURE 5.31 Engineering Analysis and the Design Model

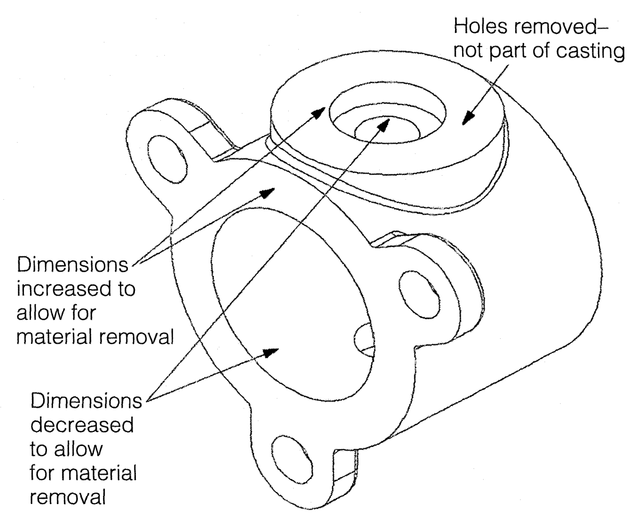


FIGURE 5.33 The Workpiece—A Casting

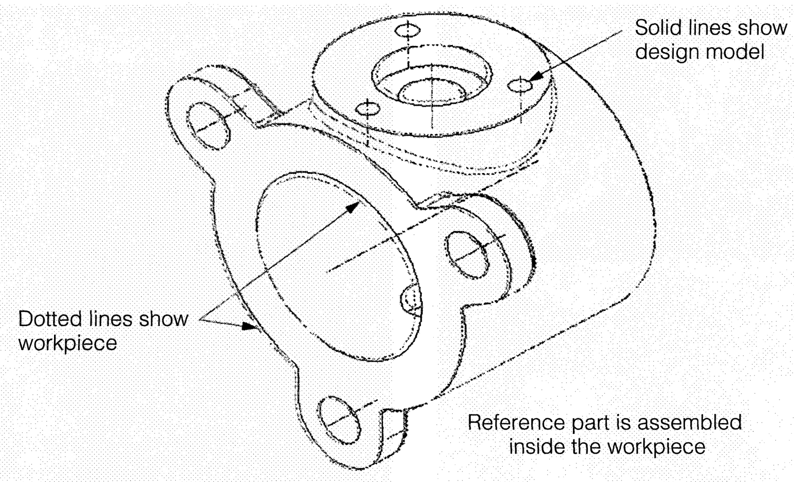


FIGURE 5.34 The Manufacturing Assembly

5.8.2 Workpiece

The **workpiece** represents the raw stock that is going to be machined by the manufacturing operations. The workpiece can represent any form of raw stock: bar stock, a casting (Fig. 5.33), etc. It may easily be created by copying the design model and modifying the dimensions or deleting/suppressing features to represent the real workpiece. As a part model, the workpiece can be manipulated as any other; it can exist as an instance of a part family table; it can be modified, redefined, etc.

5.8.3 Manufacturing Model

A regular **manufacturing model** consists of a *design model* (also called "reference part" since it is used as reference for creating NC sequences) and a *workpiece* assembled together (Fig. 5.34). As the manufacturing process is developed, the material removal simulation can be performed on the workpiece. Generally, at the end of the manufacturing process the workpiece geometry should be coincident with the geometry of the design model.

5.8.4 Part and Assembly Machining

These are the two separate types of **Pro/MANUFACTURING.**

Part Machining. Acts on the assumption that the manufacturing model contains one reference part and one workpiece (also a part). Multipart manufacturing (Fig. 5.35)

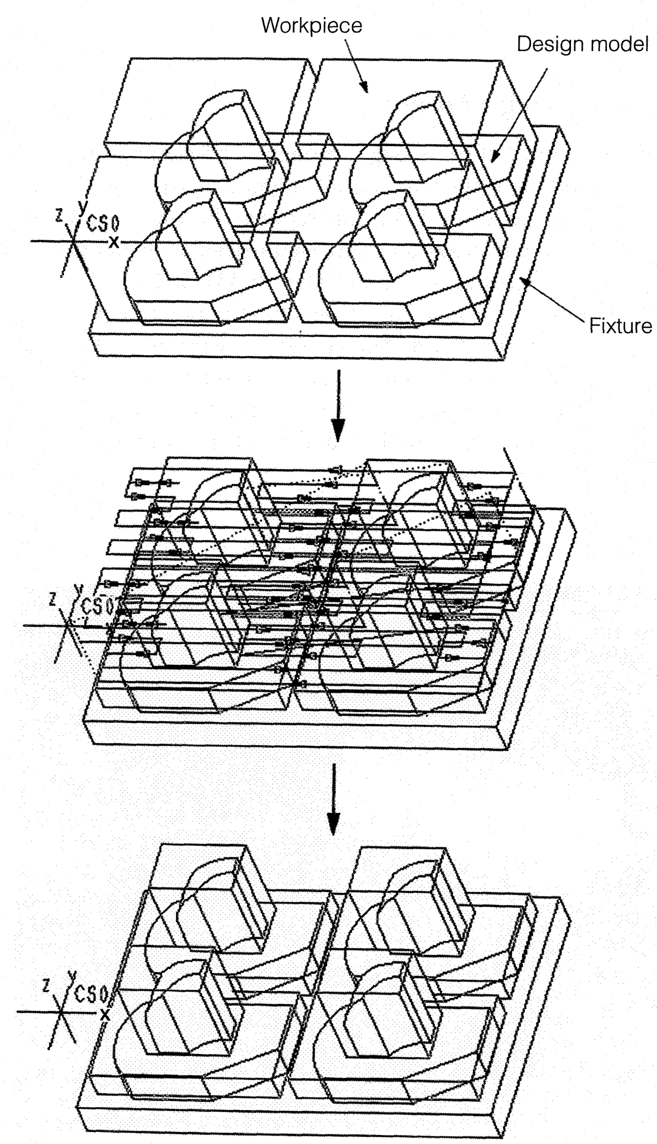


FIGURE 5.35 Machining Multiple Workpieces

allows you to assemble multiple design models and workpieces, but they are automatically merged upon assembly so that the manufacturing model still consists of one reference part and one workpiece.

Assembly Machining. No assumptions are made by the system as to the manufacturing model configuration. The manufacturing model can be an assembly of any level of complexity (with subassemblies, etc.), and can contain any number of independent workpiece and/or reference models. It can also contain other components that may be part of the manufacturing assembly but have no direct effect on the actual material removal process (i.e., the turntable, clamp, etc.).

Once the manufacturing model [Fig. 5.36(a)] is created, part and assembly machining use similar techniques to develop the manufacturing process [Fig. 36(b)]. The major difference between part and assembly machining is that in part machining all the components of the manufacturing process(operations, workcells, NC sequence, etc.) are *part features that belong to the workpiece,* while in assembly machining these are *assembly features that belong to the manufacturing assembly* (Fig. 5.36(c)].

Besides machining, a variety of other manufacturing processes can be accomplished with help of the original design model database. Die design (Fig. 5.37), mold design (Fig. 5.38), and casting design (Fig. 5.39) are all available on Pro/ENGINEER, along with many other high-level systems.

While working through the various chapters of this text, try to keep in mind the capabilities of parametric design and how they differ from traditional manual drafting and design and from traditional CAD/CAM systems. However, regardless of the system, method, or level of sophistication of the design and engineering process, the knowledge contained in the chapters covering dimensioning, springs, fasteners, gears, and other standard engineering and design applications will remain important to learn.

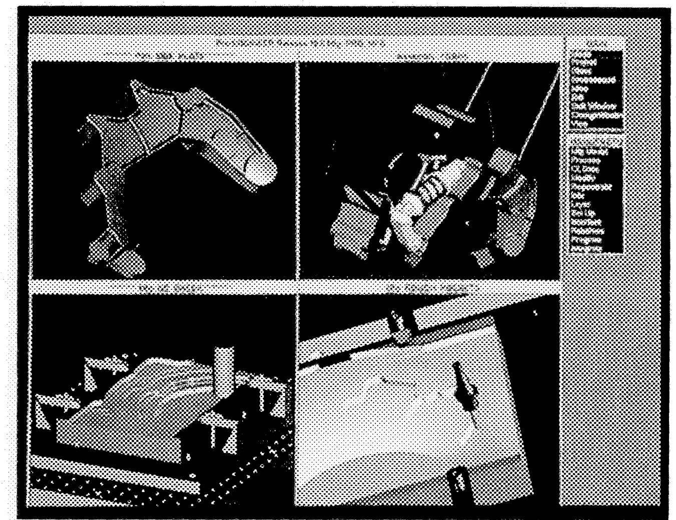


FIGURE 5.36(b) Manufacturing Processes

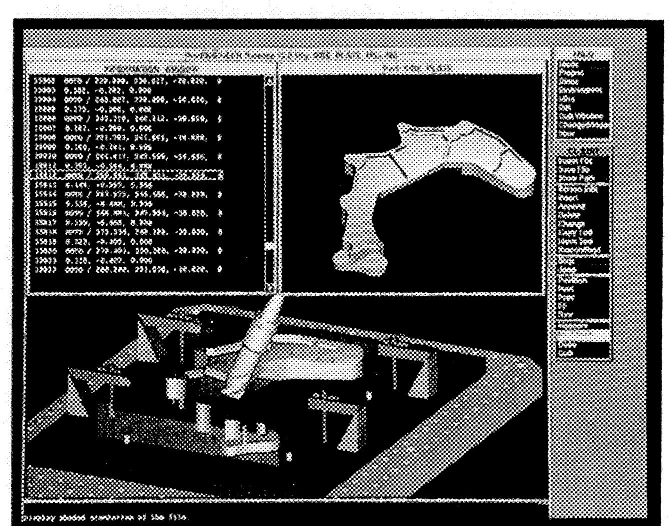


FIGURE 5.36(c) Manufacturing Assembly

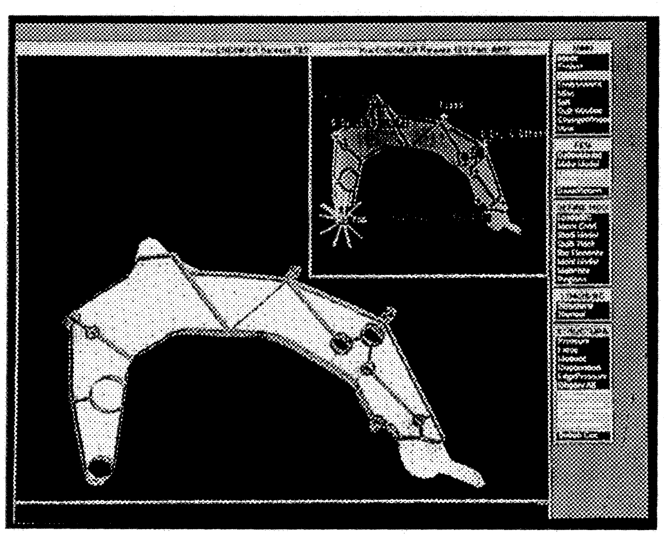


FIGURE 5.36(a) Manufacturing Model

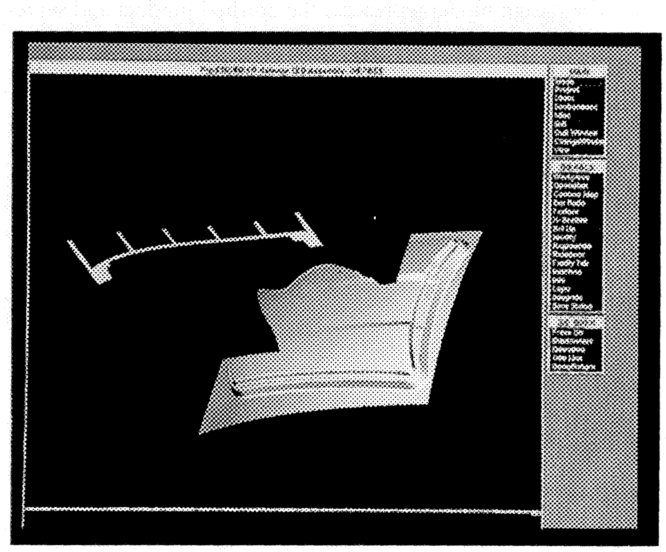


FIGURE 5.37 Pro/DIEDESIGN

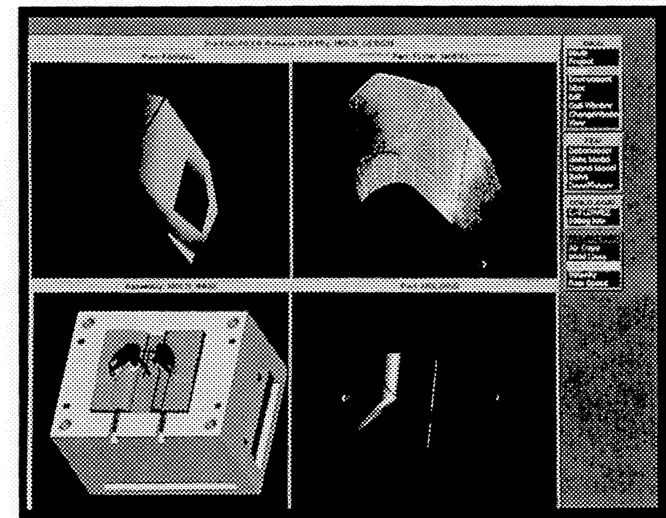


FIGURE 5.38 Pro/MOLDESIGN

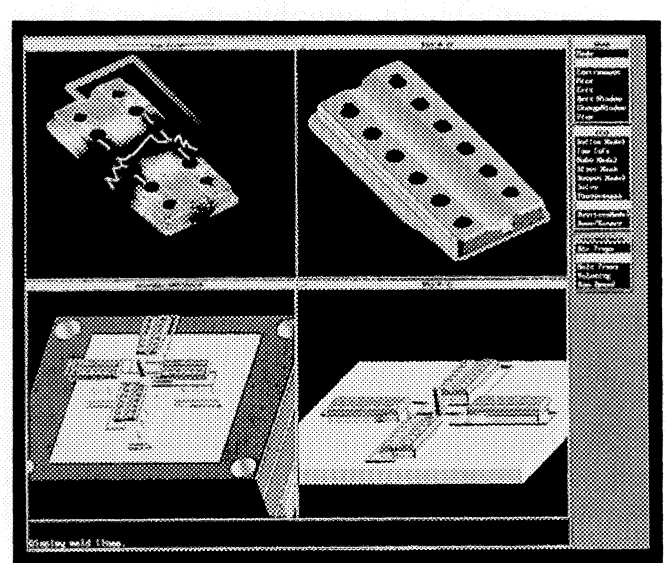


FIGURE 5.39 Pro/CASTING

QUIZ

True or False

1. Parametric design starts with the creation of drawings that represent the 3D shape of the part and proceeds toward a manufacturing 3D model.
2. Sketching in parametric design is very similar to sketching on paper.
3. Extrusions, revolved protrusions, and sketched outlines of parts are just three of many "pick and place" geometry items available with parametric modeling.
4. *Collinearity* describes a situation in which two outlines are exactly the same, because they are divided by a centerline shared by both.
5. Any arcs and circles sketched with nearly the same radius have the exact same radius.
6. The design model represents the finished product and serves as the basis for all engineering and manufacturing operations.
7. Parametric modeling is the capturing of design operations as they take place, as well as future modifying and editing that take place on the design.
8. *Feature-based modeling* refers to the construction of geometries as a combination of form features.

Fill in the Blanks

9. __________ __________ are used to create a reference on a part where one does not already exist.
10. __________ features are created by extruding, revolving, blending, or sweeping a cross section.
11. The design of any part requires that the part be __________, __________, constrained, and __________.
12. There are three types of datum features: __________, __________ __________, and __________ __________.
13. __________ __________ allows you to assemble multiple design models, and workpieces are automatically merged upon __________ so that the manufacturing model still consists of one reference part and one workpiece.
14. __________ can also be generated from the part or __________.
15. The __________ __________ can be an assembly of any level of complexity and can contain any number of independent workpiece and reference models.
16. __________-__________ relationships are an important factor in parametric modeling.

Answer the Following

17. Give the primary reasons for using solid models.
18. What are *referenced features* in parametric modeling?
19. Explain what is meant by a *workpiece.*
20. Discuss in your own words *parent–child* relationships.
21. What does *capturing design intent* mean in parametric modeling?
22. Define *drawing parameters.*
23. What is *feature-based modeling?*