Chapter 1

CAD Modeling Overview

Chapter Highlights

- Wireframe, surface, and solid
- Feature-based solid modeling
- Primitive solids
- Sketched solid modeling functions
- Boolean operation functions

Overview

CADKEY is a very powerful CAD software that can create 2D and 3D wireframe or solid models. You can use it to model a part in wireframe, surface, or solid mode. This chapter introduces the basic concepts of CAD modeling. It begins with three modes of modeling parts in CAD programs, followed by the feature-based solid modeling concept, solid modeling techniques, and Boolean operation functions.

1.1 CAD Modeling Modes

A Computer-Aided Design or Drafting (CAD) system is the indispensable tool to design, analyze, and manufacture parts in this modern world. There are hundreds of commercially available CAD systems and they can be classified into one of four categories: **2D CAD, wireframe CAD, Surface CAD,** and **Solid CAD**. In fact, most modern CAD systems are capable of modeling parts in at least three or even all four modes. Very few CAD systems only do wireframe modeling.

2D CAD Modeling

2D CAD is the most primitive CAD modeling technique. It only creates geometry entities on a 2D plane. The basic 2D geometry entities are points, lines, arcs, circles, and splines. 2D CAD can only present the part model in a plane view. Figure 1.1 shows a plane-view drawing of a part. It is possible to use three plane-views to describe a part. Commonly the three plane-views used are the top view, front view, and side view (Figure 1.2).

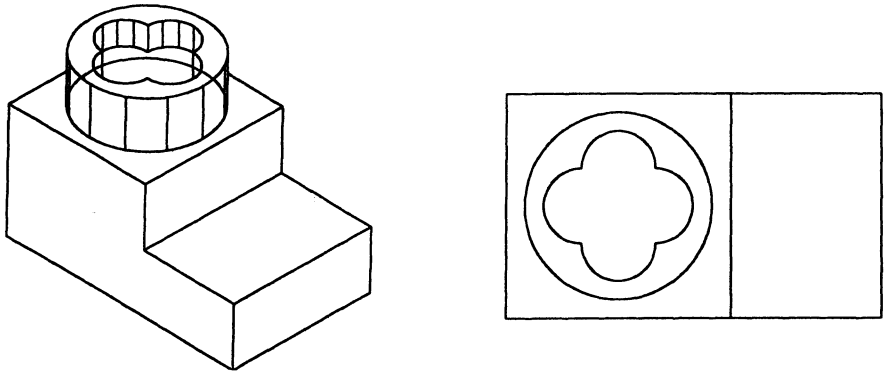


Figure 1.1 A plane-view drawing

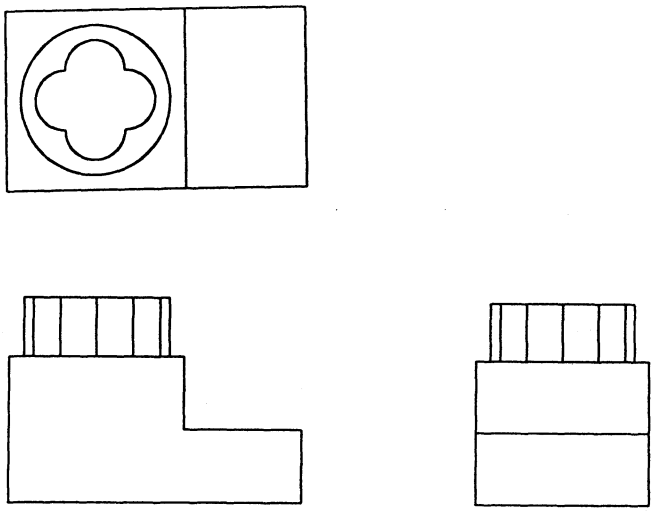


Figure 1.2 Three plane-views

A 2D CAD system offers the following advantages:

- Less expensive and affordable
- Simple, easy to operate
- Can create three-view drawings of a part
- Suitable for 2D and 2-1/2D machining

2D CAD systems are simple CAD programs that can be developed easily. Therefore, they are less expensive. A typical 2D CAD system costs less than $200. They are easy to operate because all of the geometry entities lie only on a 2D-plane and their available entity types are limited to point, line, arc, and spline. You only need to know a number of commands to create 2D drawings. Two typical applications of 2D CAD systems are creating drawing views of parts and generating their 2D and 2-1/2D toolpaths (Figure 1.3). Most CNC machining projects only involve 2D and 2-1/2 D cutting. Early CAD/CAM systems only had a 2D CAD capacity, but they were sufficient for creating geometry entities necessary for generating toolpaths.

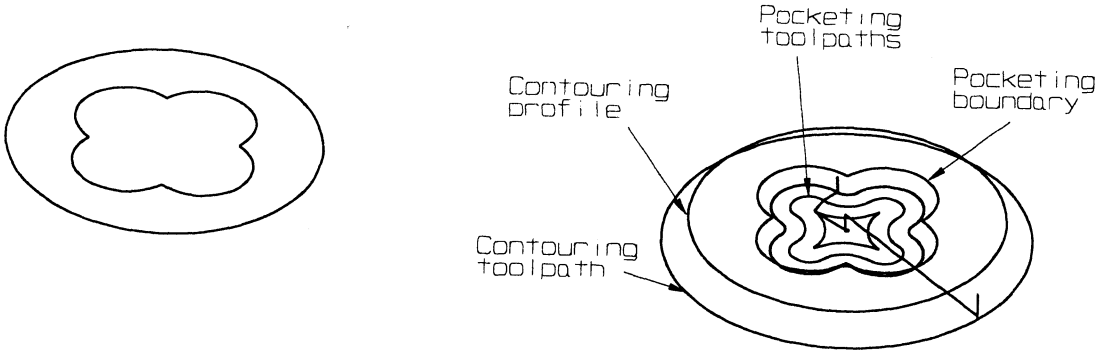


Figure 1.3 Generating 2D and 2-1/2D toolpaths

2D CAD systems have the following two drawbacks:

- No 3D representation of object
- Can't do 3D linear or circular toolpaths

2D CAD systems can not show the third dimension of the part. In other words, 2D CAD drawings show no thickness of the objects. The depth of the toolpaths must be specified using a depth parameter. Since the geometry entities are drawn in a 2D-plane, there is no way to model any 3D geometry entities. Similarly, no 3D toolpaths can be cut from designs generated in a 2D CAD system. Surface skins of a part must be modeled in a 3D CAD system.

Wireframe CAD Modeling

A wireframe model represents an object by defining its edges with a series of lines and curves (Figure 1.4). The term "wireframe" is derived from the fact that one may imagine a wire that is bent to follow the object's edges to generate the model. The basic constructing elements of wireframe models are points, lines, arcs and circles, conics, and curves. Current wireframe modeling systems are capable of constructing three-dimensional representations and have considerably enhanced interactive procedures.

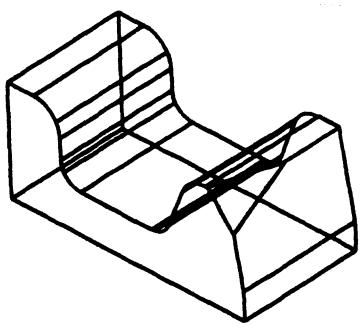


Figure 1.4 Wireframe model

Wireframe models are widely used in generating 2D toolpaths for contouring and pocketing (Figure 1.5). 3D contouring toolpaths can also be generated from a wireframe model (Figure 1.6).

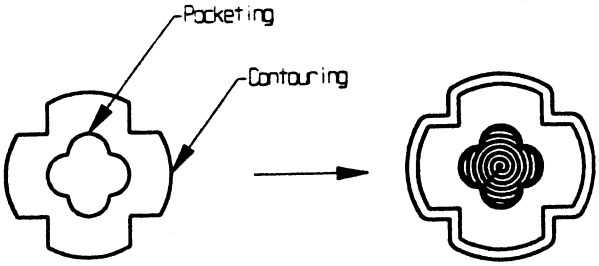


Figure 1.5 2D toolpaths

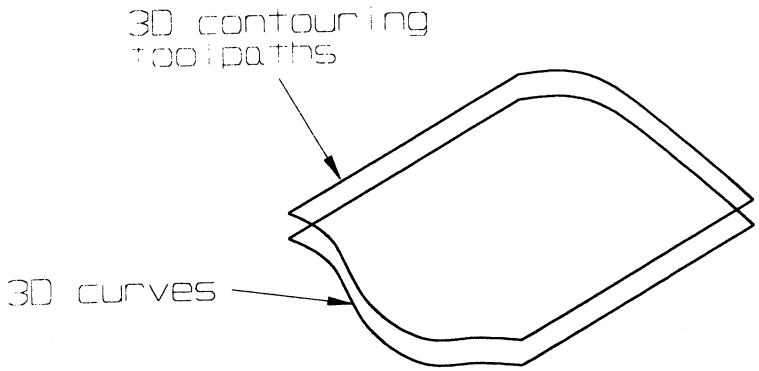


Figure 1.6 3D contouring toolpaths

Wireframe CAD systems have the following advantages:

- Display the 3D profile of the part
- Have small storage requirements and can be accessed and displayed quickly
- Resulting line drawings constitute an acceptable visualization aid in most cases for designers
- Can generate toolpaths to cut along the 3D geometry contours
- Are used as surface boundaries and cross-sectional features for developing surface models

All pure wireframe CAD systems exhibit the following deficiencies:

- There may be ambiguities in interpreting the representation
- Can't represent the shape and size of the part's surface skin
- Require a significant number of geometry entities to show the approximated shape of the model
- Can't be directly used to cut the surface skin of the model

Three-dimensional wireframe models are frequently ambiguous in interpreting the entity representation. Figure 1.7 is an example of this ambiguity. As shown there are three equally likely passages through the object. It is also possible to create unrecognizable objects, since these systems usually have no internal logical tests to detect it. Figure 1.8 shows that implied faces interpenetrated in such a way that makes the interpretation of a physically realizable solid impossible. Another problem is the lack of contour or profile information for surfaces inferred between the wireframe lines and curves (Figure 1.9).

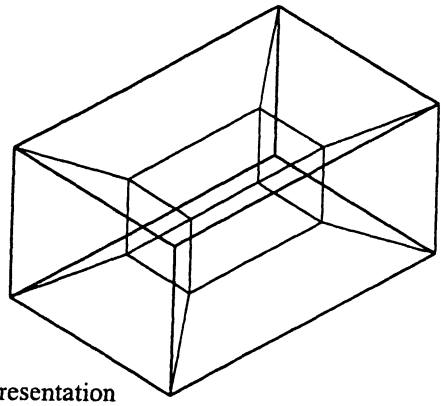


Figure 1.7 Ambiguous representation

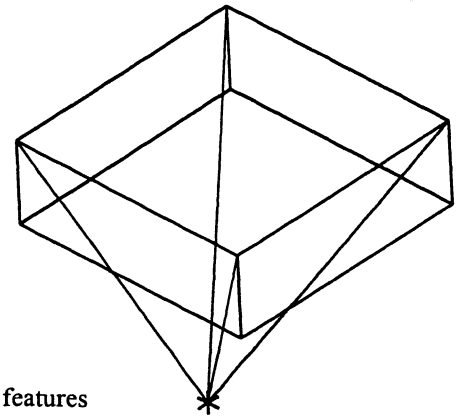


Figure 1.8 Unrecognizable features

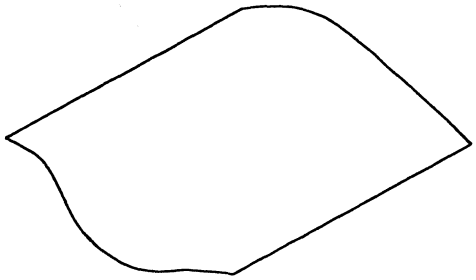


Figure 1.9 Lack of contour information

Surface CAD Modeling

A surface model defines the surface skin of the part, including the edges and each surface. Models show not only the edges of surfaces, but also the shapes (Figure 1.10). A surface actually is the mathematical representation of a part's skin. It has no thickness or volume. To create a surface model, you begin by constructing wireframe entities and then connecting them appropriately with the proper surfacing technique.

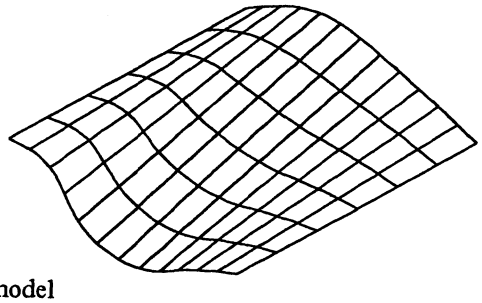


Figure 1.10 A surface model

The advantages of surfacing techniques are:

- Accurately model the true shape and size of part's skin
- Ability to model very complicated sculptured shapes (Figure 1.11)
- Flexibility for modifying and manipulating surface shapes
- Can generate 3D toolpaths to cut the accurate surface skins of a part (Figure 1.12)

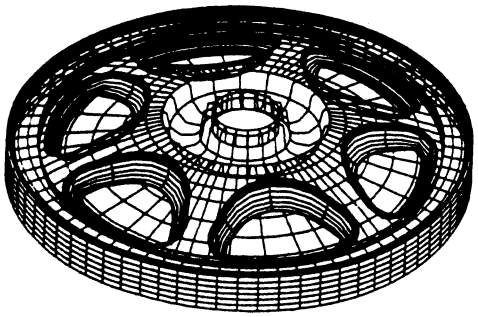


Figure 1.11 Complicated sculptured surfaces

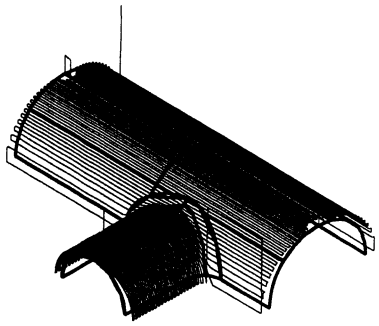
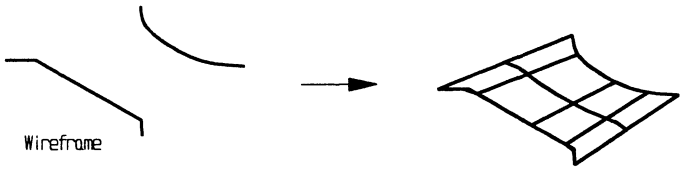


Figure 1.12 3D toolpaths from a surface model

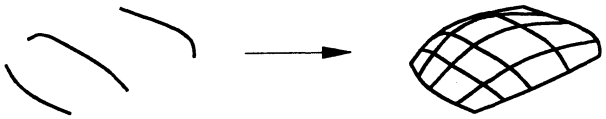
The drawbacks of surfacing systems are:

- Can't provide physical properties of the part such as thickness, volume, weight, centroid, etc.
- Require a full or partial wireframe model to create surfaces
- Often require multiple surfaces for fully defining the shape and size of a part
- Relatively difficult to construct surfaces

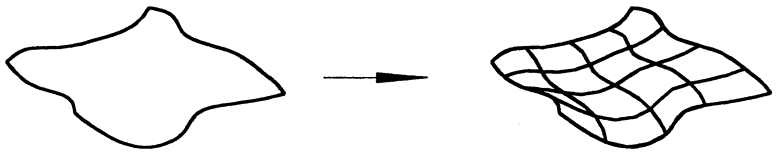
There are six basic surfacing techniques used to model various surfaces. They are **ruled surface, loft surface, Coons surface, revolved surface, draft surface** and **swept surface** (Figure 1.13).



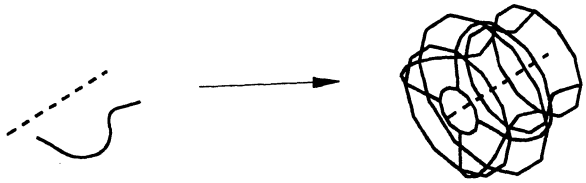
a. Ruled surface



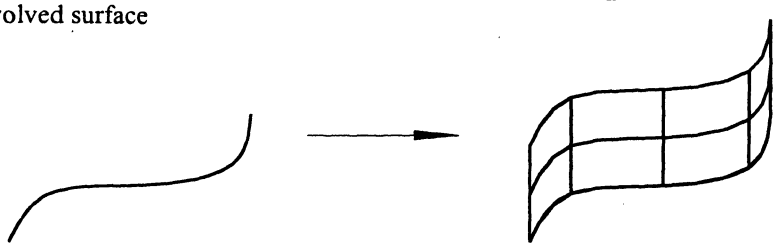
b. Loft surface



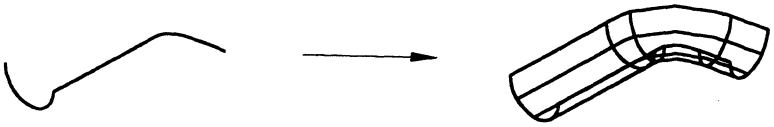
c. Coons surface



d. Revolved surface



e. Draft surface



f. Swept surface

Figure 1.13 Various surfacing techniques

Solid CAD Modeling

A solid model fully describes the shape, size, and volume of the object (Figure 1.14). It provides a complete set of geometric data about the object for further applications. These may include mass properties calculation, kinematic analysis, stress analysis, manufacturing, etc. Solid modeling uses a feature-based concept that a part geometrically is composed of a combination of several identifiable features such as blocks, holes, slots, etc., to model parts (Figure 1.15).

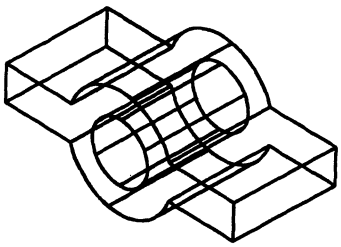


Figure 1.14 A solid model

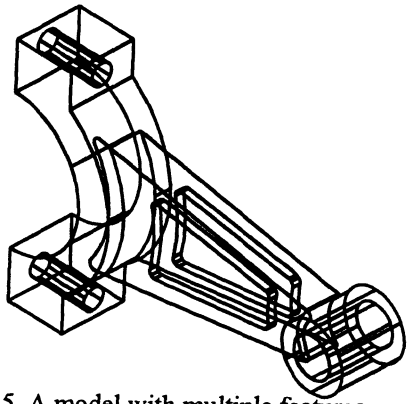


Figure 1.15 A model with multiple features

The solid modeling offers the following benefits:

- Easy to model a part
- Reduction in using wireframe geometry
- Readily construct 3D model
- Render complete physical information about a part
- Easy to modify a part

The solid modeling CAD systems have the following limitations:

- Relatively expensive
- Difficult to model complicated sculptural shapes

Solid modeling techniques can be grouped into three categories; primitive solids, sketched solids, and edited solids. Primitive solids include those basic geometric elements that can be fully defined by specifying a number of geometric parameters (Figure 1.16). Sketched solids are those that require the use of one or more pre-existing sketched sections as the profile or path to create solids (Figure 1.17). Solid features can be edited by adding fillets, chamfers, or shells (Figure 1.18).

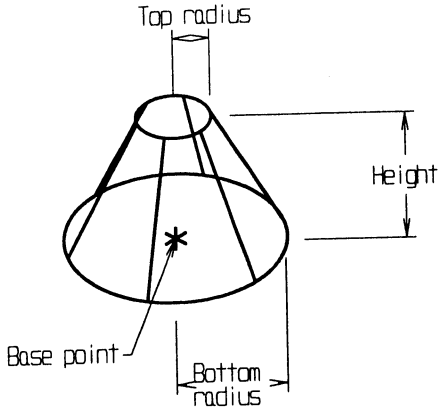


Figure 1.16 Primitive solids

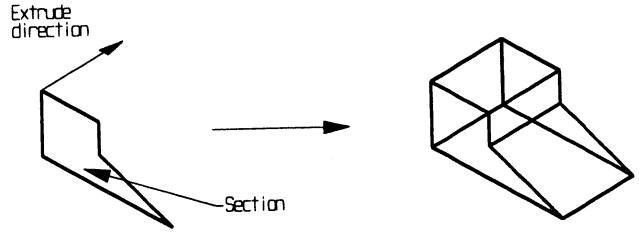


Figure 1.17 Sketched solids

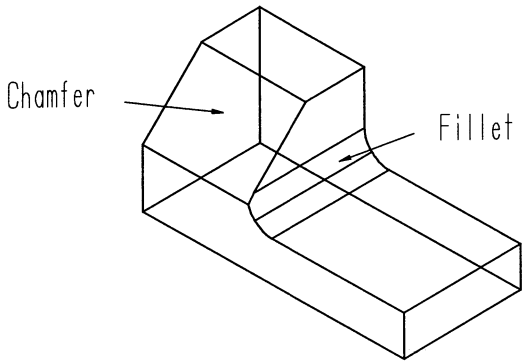


Figure 1.18 Edited solid features

Most CAD programs today are capable of modeling parts in any one of these four modes; 2D CAD, wireframe, surface, and solid. Each modeling mode has its strengths and limitations. Solids give you the speed, surfaces for power, and 2D CAD and wireframe offer simplicity. We can mix and match these modeling techniques to effectively model parts. Also, we can add surface or wireframe elements to a solid, or add solid components to a surface model.

1.2 Feature-Based Solid Modeling

CADKEY uses a feature-based construction approach to create parts. Features are the building blocks of a part. There are two types of features: **base features** and **added features**. An object or a part normally consists of a combination of a base feature and several added features (Figure 1.19). The base feature is the initial workblock that defines the basic shape of the part. In other words, the base feature is the basic form of the part. Figure 1.20 shows some typical examples of base features. The base feature is analogous to raw stock.

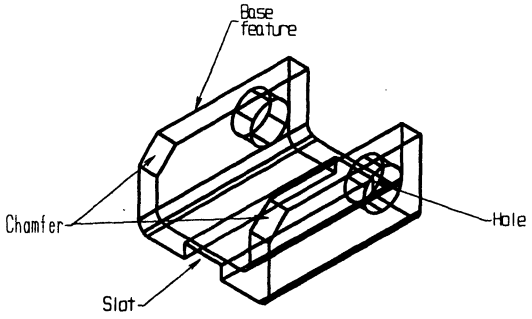


Figure 1.19 Based feature and added features

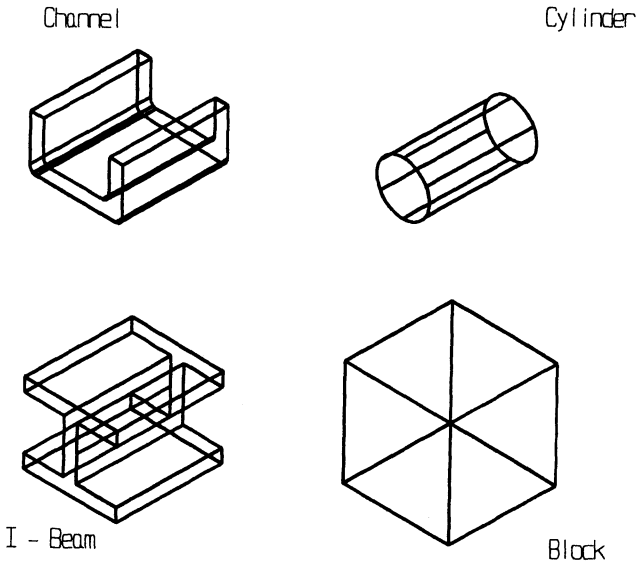


Figure 1.20

The base feature is the most basic element in the part design and should be represented in its simplest shape and form. The added features are added to the base feature to define the details of a part. Figure 1.21 shows two examples of using added features with the same base features to further define the parts.

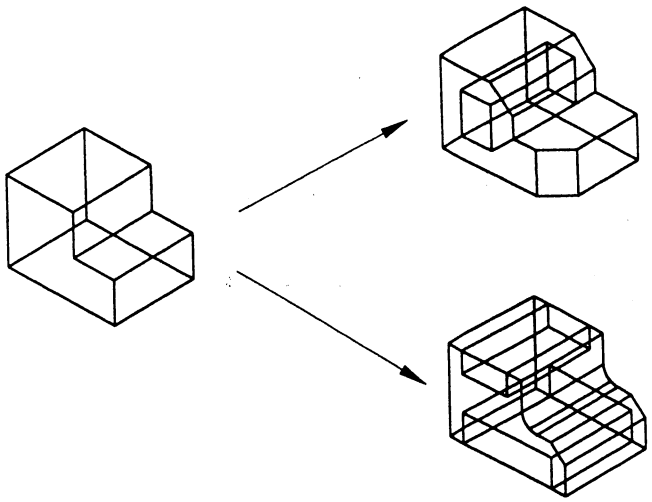


Figure 1.21

Added features can be formed in two ways: **adding** and **removing** material. As the name implies, adding material features add the material of the feature to the existing model (Figure 1.22). Any primitive solids or sketched solids can be used as adding material features. Removing material features remove the material of the feature from the existing model (Figure 1.23). Any primitive solid and sketched solid can be used as removing material features. In addition, CADKEY has three solid editing functions: **fillet**, **chamfer**, and **shell**, for removing material from the model (Figure 1.24).

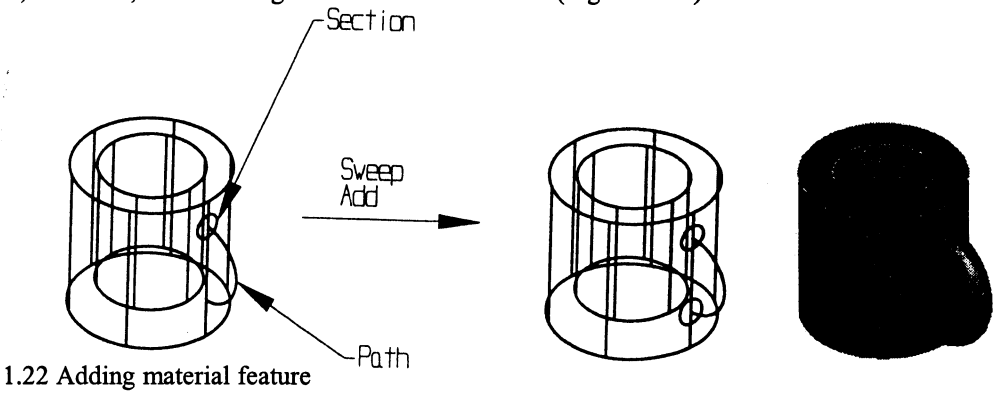


Figure 1.22 Adding material feature

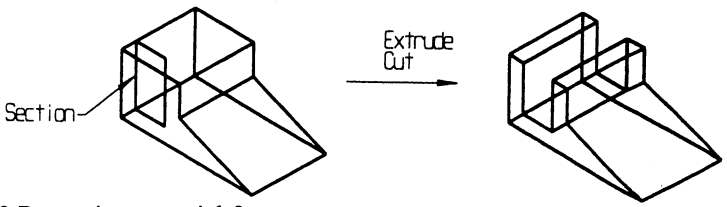


Figure 1.23 Removing material feature

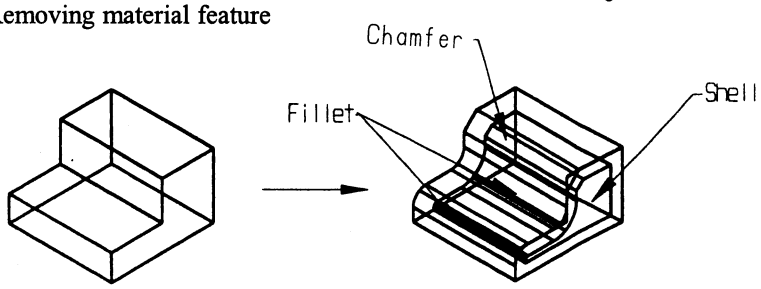


Figure 1.23 Solid editing function

1.3 Solid Modeling Functions

CADKEY provides three groups of solid modeling functions: **primitive solids, construction solid functions** and **solid modifying functions.** Primitive solids are those solid features that can be readily defined by specifying a number of parameters. A solid block, for example, can be fully defined by its height, length, width, and base point. There are **seven** primitive solid functions including **cylinder, cone, prism, pyramid, block, sphere** and **torus** (Figure 1.25).

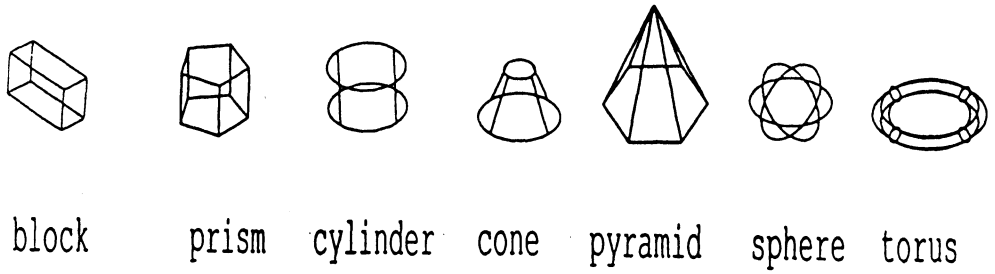


Figure 1.25 Primitive solids

There are six construction solid modeling functions in CADKEY: **extrude, revolve, sweep, loft, helix and pipe.** They all require the use of pre-existing geometry entities as the basic sections or profiles for creating solid features. The extrude function creates a solid feature by extruding selected sections in the specified direction by a given distance (Figure 1.26). The revolve function creates a solid feature by revolving selected sections about a revolving axis by a given angle span (Figure 1.27). The sweep function generates a solid feature by smoothly translating and rotating a section along a sweeping path (Figure 1.28). The loft function smoothly blends two or more planar sections to form a solid feature (Figure 1.29).

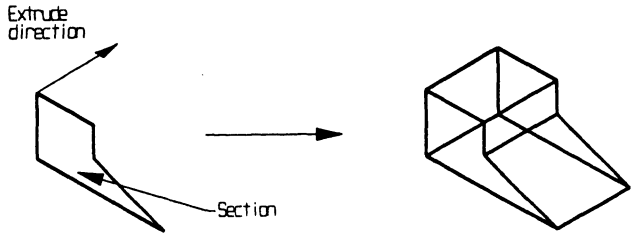


Figure 1.26 Extrude feature

Figure 1.27 Revolve feature

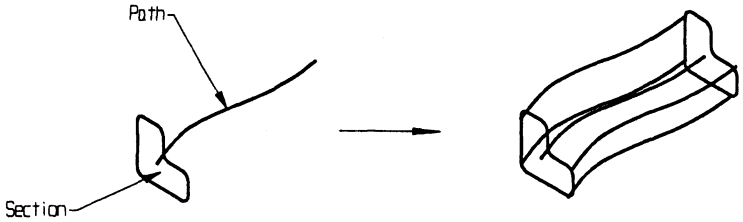


Figure 1.28 Sweep feature

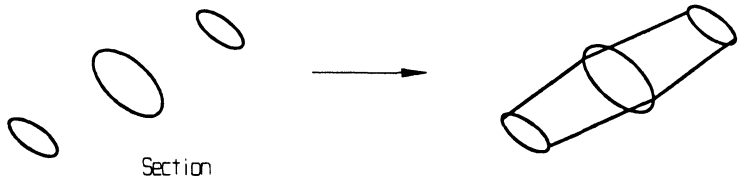


Figure 1.29

CADKEY has several solid modification functions that are used to modify existing solid features. **Fillet, chamfer** and **shell** are three commonly used features. The fillet function adds a round transition between two adjacent faces (Figure 1.30), while the chamfer function adds beveled edges to the selected edges of the model (Figure 1.31). The shell function hollows solid bodies and optionally leaves selected faces open. The remaining faces are thickened by a specified amount (Figure 1.32).

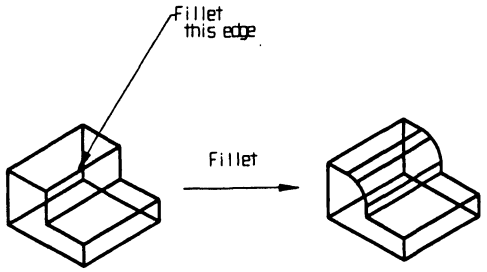


Figure 1.30 Fillet

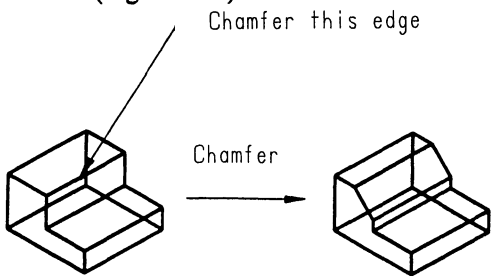


Figure 1.31 Chamfer

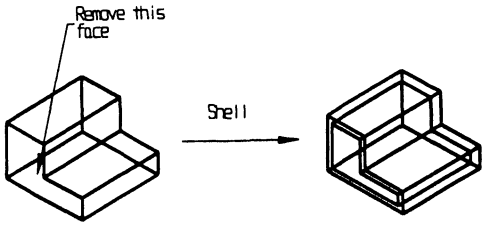


Figure 1.32 Shell

1.4 Boolean Functions

Solid features can be created as separate solid entities. Boolean functions are used to determine the relationship between those solid features. There are three basic Boolean operation functions in CADKEY that allow you to construct a solid by using a combination of two or more existing solids. The first selected solid feature in performing a Boolean operation is designated as the **target body**. Each subsequent solid that you select for the operation is referred to as a **tool body**. These three Boolean functions are **Unite, Subtract** and **Intersect**. These three Boolean operation functions are summarized in the following Table 1.1.

Guidelines of applying Boolean operation functions:

Primitive solids are created as independent, separate solid entities. They must be combined to form a solid by using one Boolean operation function.

Primitive solids or sketched solids that are to be copied to other locations must be created as new solids. New solids are regarded as independent, separate solids. Use one of the transform functions to copy the solid(s) to proper locations, then use one of the Boolean operation functions to combine them with the mode (Figure 1.33).

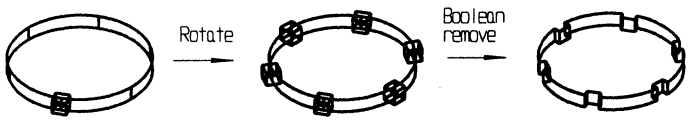
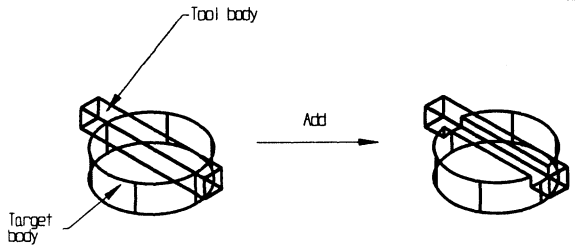
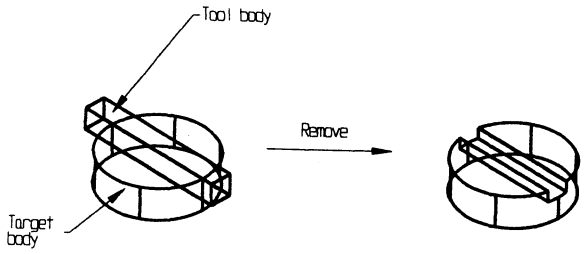
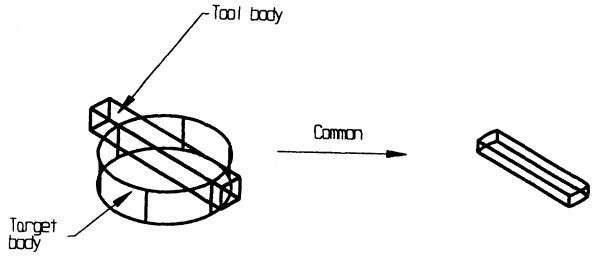


Figure 1.33

Table 1.1 Boolean Operation Functions

Operation	Description	Illustration
Unite	Add the selected tool bodies to the target body. It adds the material of the tool bodies to the model.	
Subtract	Remove the selected tool bodies from the target body. It removes the material of tool bodies from the model.	
Intersect	Keep only the material that is commonly shared by the tool bodies and target body and removes all other regions of the solids.	

1.5 Review Questions

1. The the four categories of CAD systems.
2. Describe the main features of 2D CAD modeling.
3. Describe the main features of wireframe CAD modeling.
4. Describe the main features of surface CAD modeling.
5. Describe the main features of solid CAD modeling.
6. What are features?
7. What are the two types of featues?
8. Describe the two ways of adding features?
9. Describe the three basic Boolean operation functions.