9. Extruding, Modifying and Redefining

Most modern solid modelers therefore are not generally constrained to use on one modeling method but are a hybrid of both B-Rep and CSG methods. This gives greater flexibility and the ability to define more complex objects. The use of CSG alone can limit the complexity of the model created, even though the range of primitives may be extensive and include such as torus, cone, pipe, pyramid and sections of these such as cylinder section and hemisphere.

In addition to profile construction and operations, primitive creation and Boolean operations which can be classified as synthesis type functions, many solid modelers provide useful modification functions for the creation of chamfers, fillets and rounded corners. Although they are modification tools from the user's point of view, they mainly involve the creation of geometry and Boolean operations, which are transparent to the user. An edge can be chamfered by the creation of an angled block or cutting plane followed by a subtraction, cut or split operation. A fillet, for an internal corner, is generated by the addition of a section created by a triangular prism from which a cylinder has been subtracted. Chamfering to the correct size and adding a cylinder can round an external edge.

Parametric design packages are often referred to as feature-based modelers. A feature is a subcomponent of a part that has its own parameters, references, and geometry as shown in Figure 9.1.
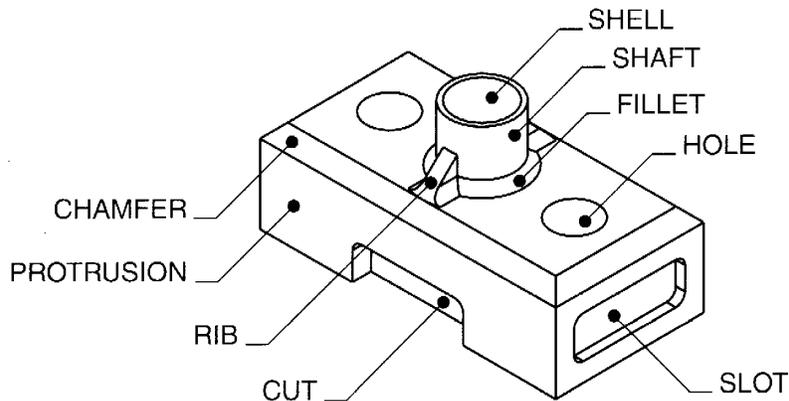

Figure 9.1. Features of a part

Geometry is the graphic description of a feature. Geometry can be sketch-defined or predefined. Sketch-defined features consist of sketched sections that are pro-minded or cut to form either positive or negative space. Predefined geometry has a common section such as a hole, round, or chamfer. *Parameters* are the dimensional values and definitions that define a feature. A hole may have a diameter of 1 inch and can be extruded completely through all existing features. The diameter is a parameter, as is the through-all definition. Parametric modeling packages allow users to modify parameters after the feature has been modeled. This is one of the unique properties that separate parametric modelers from Boolean-based modelers.

9.1. FEATURE-BASED MODELING

An important advancement in 3-D solids modeling is the introduction of feature-based modeling. Like constraint-based modeling, feature-based modeling is an attempt to make modeling a more efficient process more in tune with how designers and engineers actually work. Feature-based modeling bundles

commands together to automate the process of creating and modifying features that represent common manufacturing operations. Usually implemented in modelers that also have constraint capabilities, feature-based modeling systems use special dialog boxes or other interface elements that allow users to input all of the variables needed to create a common manufactured feature. (Figure 9.2)
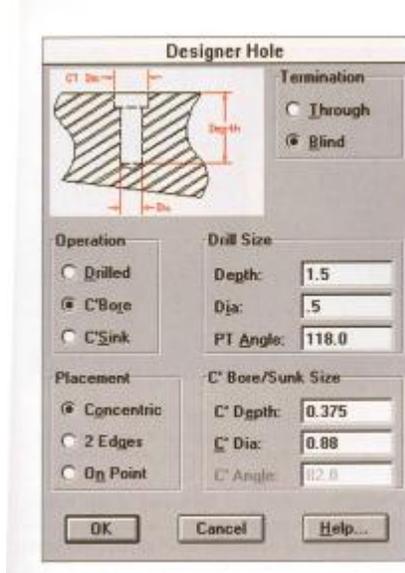

Figure 9.2. Example of a Dialog Box in a Feature-Based Modeler

Examples of manufactured features created through special feature-based dialog boxes include the following:

- Blind holes.
- Counterbores and countersinks. Slots.
- Bosses.

The hole dialog box shown in Figure 9.2 is a good example of automating the process of creating features in a model. The feature is broken down into its essential variables, with each variable represented by an input in the dialog box. Variables such as the hole's diameter have a value typed in, while the depth can be set to "through" by clicking a button or set to a finite value.

Notice that all of the essential variables defining the feature are represented as inputs in the dialog box. Typically, many of these variables can also be defined as parameters linked to other features on the model.

The variables entered through the dialog box largely define the *shape* and *size*. Once these variables of the feature are defined, the *location* is defined. By convention, features usually don't constitute an entire part. For that reason, the feature is typically located somewhere on a *base part*. A feature such as a blind hole is located by indicating its orientation to a face and distance from two edges. (Figure 9.3) In a constraint-based modeler, all of the variables of the feature—its shape, size, and location—are parametrically controlled and can be updated at any time. In addition, the parameters defining the feature can also be linked to other parameters defining the part. So, for example, the depth of a hole might be related to the overall thickness of the base part.
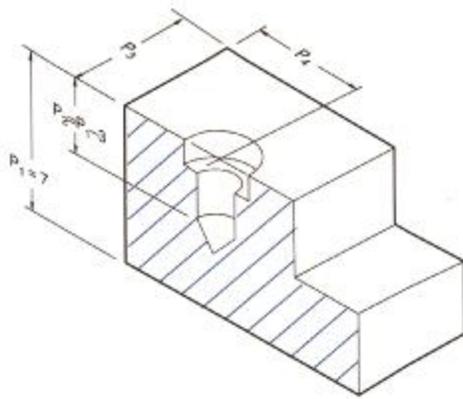
2

Figure 9.3. Locating a Feature on the Base Part

The blind counterbore is located on the base part relative to a face on the part. The overall depth of the counterbore is linked to the overall height of the part through a parametric equation. Boolean operations, which can be carried out between primitives, are:

Addition, union, join. Primitives are joined together to form one solid and a prerequisite for the success of this operation is that the two primitives are touching or partially intersecting one another. An application of this operation would be the addition of a projecting feature on the main body of a component; the addition of a circular boss would require the generation and the positioning of a cylinder followed by a union operation.

Subtraction, cut. One primitive is subtracted from another to leave a void or hole which requires complete or partial intersection between the two solids. The subtracted primitive or solid is used as a template for the cavity desired in that which it is subtracted from. An application of subtraction would be the creation of a hole or cavity in a component. The cut operation can also be carried out by the creation of a cutting plane as opposed to another solid primitive. The solid is cut or split in two by the cutting plane and the option given to the user as to which side to keep or remove.

Intersection. The two primitives or solids are intersected and the resulting solid is the common solid material between the two. The operation obviously requires complete or partial intersection between the two solids in the operation. This operation has a useful application in the creation of a solid model of complex geometry where the profiles when viewed from two or more orthogonal planes are known. Each profile can be generated in each plane and extruded to the correct depth or height and the two extrusions intersected to create the final model.

9.2. PARENT-CHILD RELATIONSHIPS

Parametric models are composed of features that have established relationships. Features are built upon other features in a way that resembles a family tree, hence the phrase *parent-child relationship.* The first feature created in a part is the center of the web and is the parent feature for all features. Child features branch of the base feature and themselves become parent features. Unlike a typical family tree, a child feature may have several parent features.

Parent-child relationships can be established between features implicitly or explicitly. Implicit relationships can be established through the adding of a numeric equation using the relations option. An example of this would be making two dimensions of equal value. In this process, one dimension governs

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the value of another. The feature with the governing dimension is the parent feature of the feature with the governed dimension. Care should be taken when modifying a feature that has a dimension that governs another.

Explicit relationships are created when one feature is used to construct another. An example would be selecting a plane of one feature as the sketch plane for a second feature. The new feature will become a child of the feature that is being sketched upon. Another similar example of an explicit relationship occurs when using existing feature edges within the sketcher environment to create a new feature. By specifying references while sketching, these selected references will create a relationship between the feature being sketched and the existing feature being referenced. The new-sketched feature becomes a child of any referenced feature.

9.3. CONSTRUCTION METHODS

The construction of solid models by the assembly of instances of primitives or by the identification of faces, edges and vertices is quite a tedious process, and therefore a number of alternative methods of generating the model have been developed. The most important of these are the sweep techniques, which involve taking two-dimensional profiles (generally comprising lines and arcs) and generating solids by projecting them either normal to the plane in which they are constructed, or along a curve, or by revolving them around a center line to form a solid of revolution.

The primary difference between the protrusion command and the cut and slot commands are that a protrusion is a positive space feature and cut and slot are negative space features. When you protrude a feature, you are actually creating a solid object; an extruded feature removes material from existing features. Figure 9.4 illustrates how one section can be used to create an extrude, revolve, sweep, or blend feature.
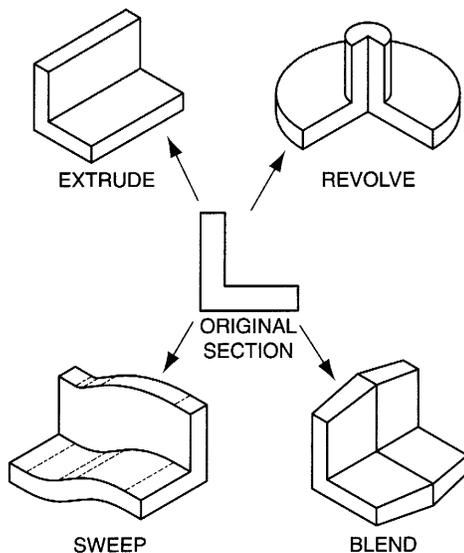
Figure 9.4. Variations in menu option features

9.3.1. Extrude

The Extrude option sweeps a sketched section along a straight trajectory. The user draws the section in the sketcher environment and then provides an extrude depth. The depth entered by the user protrudes the section. In computer-aided design software, extrusion means that a 2-D shape is pushed back (or forward) a parts specified amount in a direction perpendicular to the plane in which the shape was sketched. As the

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shape is pushed then back, the endpoints of the lines, arcs, etc., that make up the shape are connected across the third dimension, forming a solid object.

A 2-D shape can be extruded to create a new part as shown in Figure 9.5, or it can be used to modify an existing part by either protrusion or cutting out. With protrusion, the 2-D shape is drawn on a face of an existing object, and the shape is extruded out from that face. This results in an increased volume for the original part because material is added to it. Figure 9.5a shows a block with three circles sketched on one face, and Figure 9.5b shows the same block after extrusion of the circles to form protrusions on the object. (Note that the smaller circles were not protruded the same distance as the larger one.)
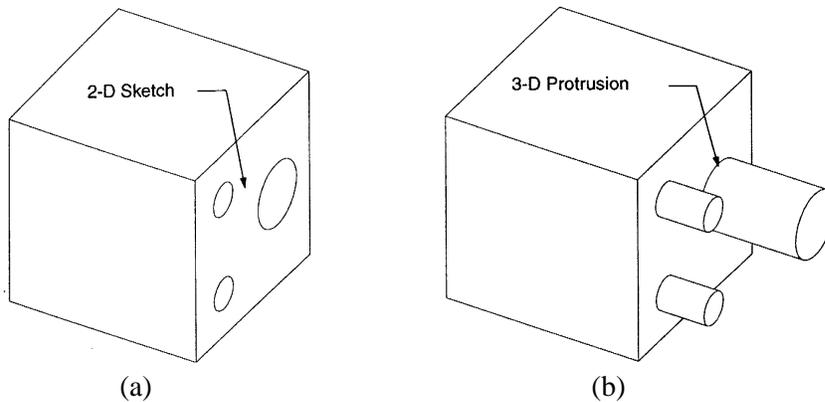
Figure 9.5. Protruding 2-D Geometry from a 3-D Part

Extrusion to cut out a shape from an existing part is the opposite of protrusion. A 2-D shape is sketched on the face of an existing object and is extruded *into* the part. As it is extruded, material is removed from the existing part in the shape of the 2-D sketch. Thus, the volume of the original object decreases. When cutting out the material, you can choose to cut all the way through the part or only partially through. Figure 9.6 shows an object with a sketch on one face and the result of extruding the indicated 2-D sketch through the pan by cutting out material.
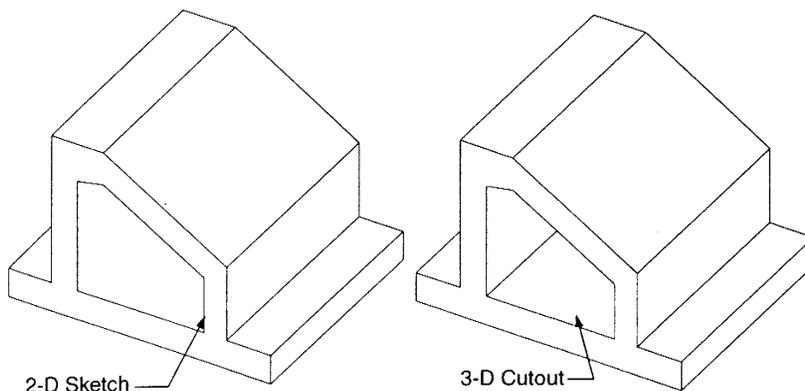

Figure 9.6. Cutting Out 2-D Geometry by Extrusion

A 2D profile is created and is extruded or swept in a direction perpendicular to the plane of the profile to produce a solid object. The solid will have constant cross section if no taper is specified (taper angle is zero). By specifying a nonzero taper angle, various tapered solids can be modeled.

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Figure 9.7 shows some examples of the original 2D image and the resulting solid that is produced by extrusion. In all of these cases, the extrusion or sweep is perpendicular to the 2D profile. Some of the objects have a nonzero taper angle. The particular taper angle would have to be calculated and could be dependent on the extrusion height to create the desired solid.
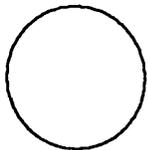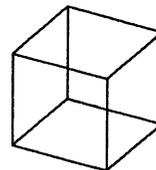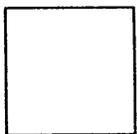
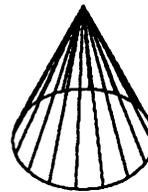**Two Dimensional Sketch**   **SOLID**
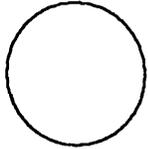
Extrusion ⊥ to paper
(Taper angle = 0)
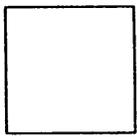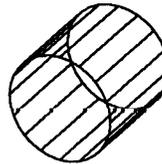
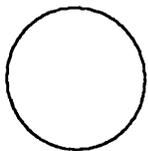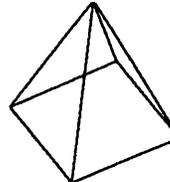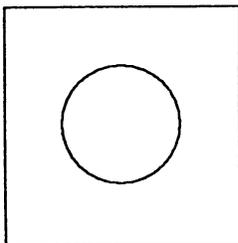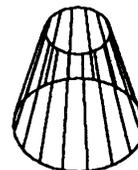Extrusion ⊥ to paper
(Taper angle = 0)

Extrusion ⊥ to paper
(Taper angle related to
extrusion height)

Extrusion ⊥ to paper
(Taper angle related to
extrusion height)

Extrusion ⊥ to paper
(Taper angle related to
extrusion height)

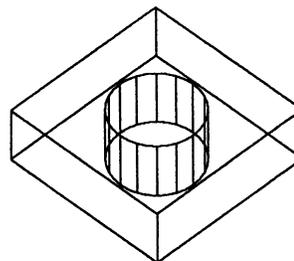Extrusion ⊥ to paper
(Taper angle = 0)

Figure 9.7. Extrusion variety

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9.3.2. Revolve

The Revolve option sweeps a section around a centerline. The user sketches a profile of the revolved feature and a centerline from which to revolve. The user then inputs the degrees of revolution. A revolved feature is a section that is rotated around a centerline. For any type of revolved feature within the sketcher environment, the user sketches the profile of the section to be revolved and the centerline from which to revolve. Revolved features cam is positive space or negative space. A revolved protrusion is an example of a positive space feature. A revolved protrusion's negative space counterpart is the revolved cut. The flange command is another example of a revolved positive space feature. Its counterpart negative space feature is the neck command.

As shown in Figure 9.8 the geometry of a revolved feature must be sketched on one side of the centerline and the section must be closed. One centerline must be sketched. Entities that lie on the axis of revolution do not serve as a replacement for the lands, centerline.
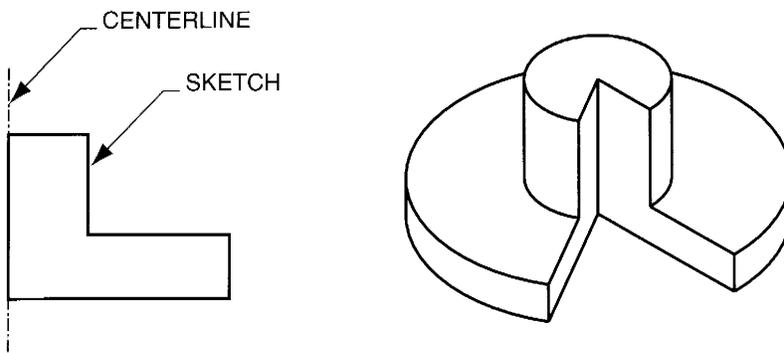

Figure 9.8. Revolved Feature

Revolved features are often used to create cylindrical objects such as shafts. Drafting standards require cylindrical objects to be dimensioned with a diameter value. This creates a unique situation within the sketcher environment.

As with extruded protrusions and cuts, a variety of options are available for defining revolved feature parameters. The revolve direction attribute is similar to the extrude direction attribute.

Revolving a 2-D shape to form a 3-D solid is another method that is commonly used to create a part, especially a part that has a basic cylindrical shape. When creating a solid in this manner, you typically specify the number of degrees through which you wish to revolve the shape, or profile. You also must specify the axis about which you want the profile to revolve. Figure 9.9a shows a 2-D profile, and Figures 9.9b and Figure 9.9c show the result of revolving the profile about the axis labeled Y by 180 degrees and 360 degrees, respectively. Figure 9.9 shows the object that results by revolving the profile 360 degrees about the axis labeled X. Notice that each of these operations results in parts that are very different from one another.
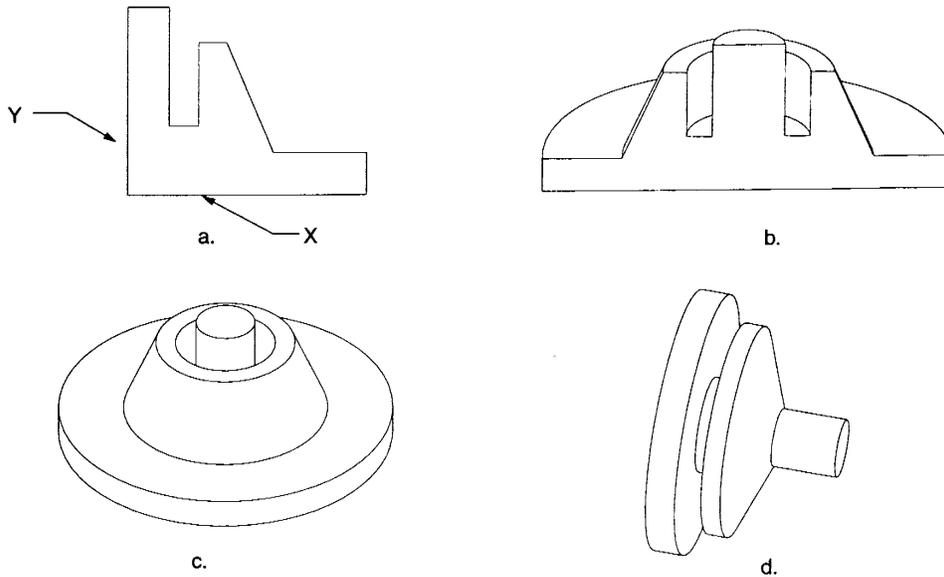
Figure 9.9. Revolution of 2-D Geometry

One problem that you may encounter in creating objects by revolution is what happens when you attempt to create self-intersecting parts. Typically, this happens when you try to revolve a portion of the profile onto the 3D solid that you are in the process of creating.

9.3.3. Sweeping And Sweeping Operations

The Sweep option protrudes a section along a user-sketched trajectory. The user sketches both the trajectory and the section. Solids can be generated by sweeping techniques as well as CSG methods. In sweeping, a 2D profile is first drawn and this outline is either extruded (also called a linear sweep) perpendicular to the plane of the 2D profile or is revolved about an axis (also referred to as circular sweep) to produce an object of revolution. Some modelers allow oblique sweeping (extrusion at other than 90° to the 2D profile). Profiles can also be swept along specified curves. This process could produce a helix, for example.

In B-rep modelers, prismatic or transnational objects can be created by a compound Euler operation termed *sweeping* and rotationally symmetric ones by *swinging* or rotational sweeps. Both these operations use a two-dimensional description as the definition of the plan or profile of the desired result. In the case of sweeping, the two-dimensional profile represents a cross-section; for swinging, it represents the axial profile. Simple examples of these operations are illustrated in Figure 9.10.
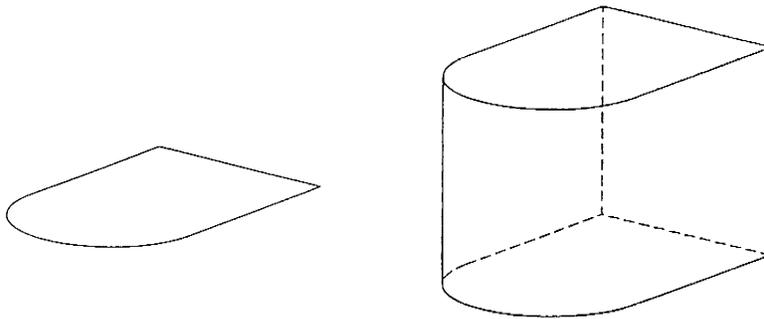

Figure 9.10. An example of a sweep operation

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9.3.4. Combining CSG and Sweeping Techniques

Sweeping or CSG methods may be used wherever each is effective. They may be combined in creating a model. Because there are often several different ways of producing a given object, you must map out a strategy prior to using the solid modeler. You need a plan for creating each entity and for combining them into a solid. Some entities can be conveniently created with CSG methods while others lend themselves to sweeping techniques. Figure 9.11 shows a simple object that was formed by some sweeping and some CSG methods.
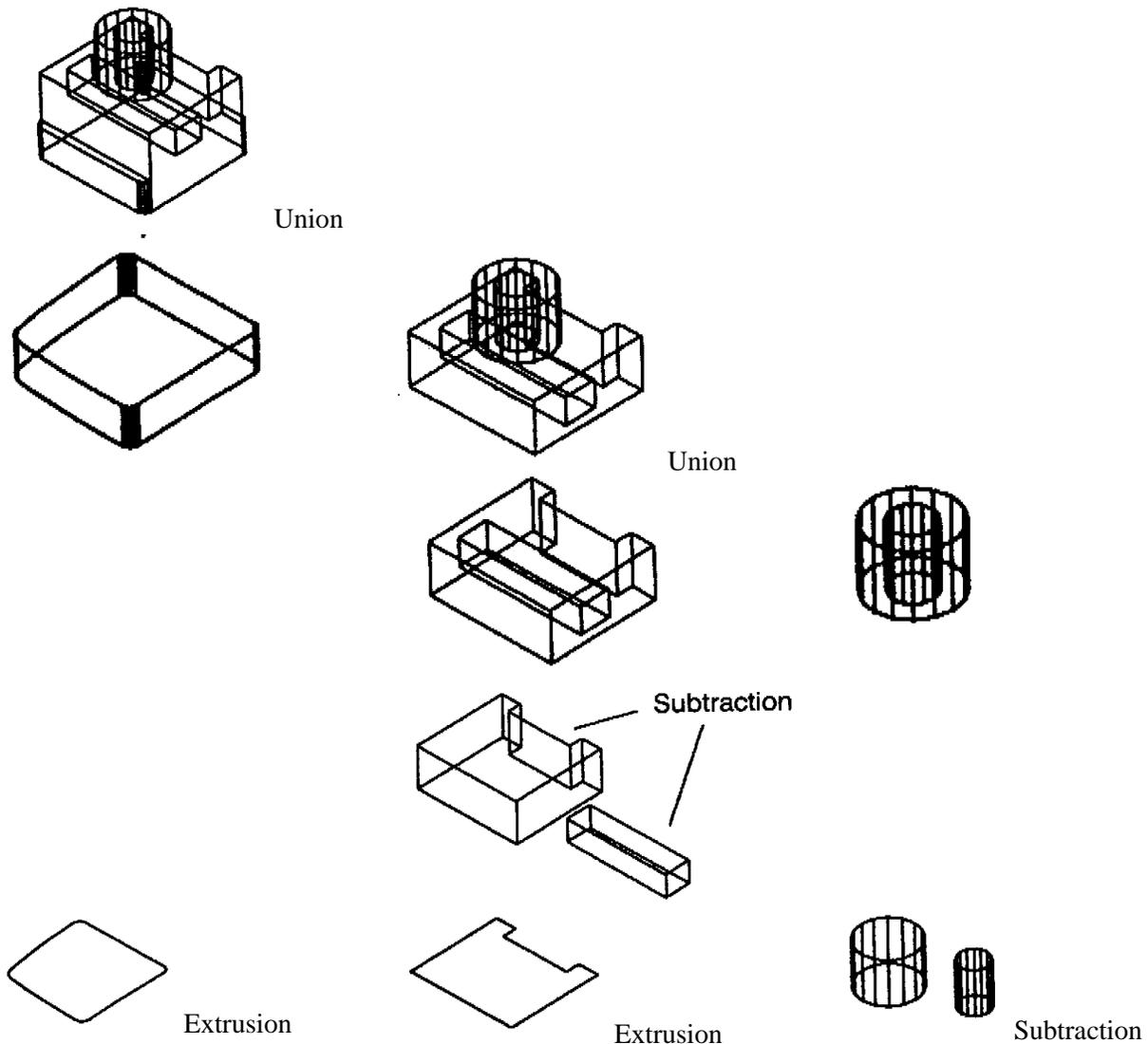
Union

Union

Subtraction

Extrusion

Extrusion

Subtraction

Figure 9.11. Combining CSG and Sweeping Techniques

9.3.5. Blend

The Blend option joins two or more sketched sections. The trajectory may be straight or revolved. The blend option can also be compared to the extrude option. Primarily, the blend option creates a feature by protruding along a straight trajectory between two or more user-defined sections (see Figure 9.12). A

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partially revolved blend can be created also. Three types of blends are available: parallel, rotational and general.
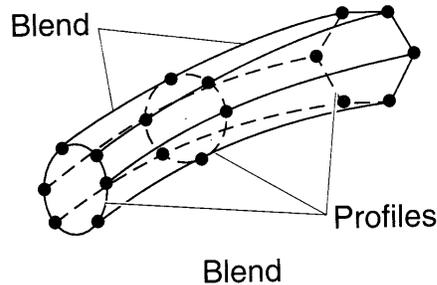

Figure 9.12. Blend function

9.4. MODIFYING FEATURES

What separate parametric modeling packages from Boolean based modeling packages are their feature modification capabilities? Examples of parameters include parametric dimensions; extrude depth, and material side. Parameters such as these are established during feature construction. These feature parameters and other feature definitions, such as a section's sketch and sketch plane, can be modified later in the part modeling process. Parametric dimensions are dimensions that are used to define a feature. They can be modified at any time. Modifying a dimension value is the most common dimension modification function, but other modification tools do exist. The number of decimal places in a dimension can be modified along with the tolerance format.

9.5. SOLID MODEL DATABASES

Solid models can be stored in different forms by modelers depending on the use to be made of the data. Commonly, models are stored as CSG trees or as boundary representations (B-reps). Figure 9.13 illustrates the concepts of a CSG tree database and a B-rep database for a very simple model. You can imagine the complexity of the databases for objects that require many entities to be combined to create them.
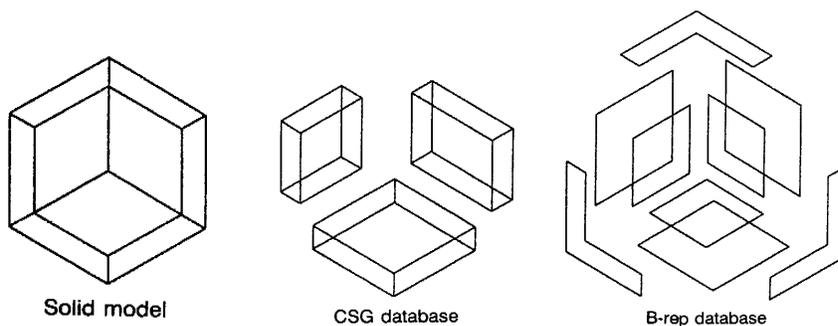

Figure 9.13. CSG and B-rep database

As an example, AutoCAD stores a composite model in CSG form as well as storing B-rep information. The model stored in a CSG tree allows the model to be "taken apart" since the process of creation is documented. It also provides the opportunity to edit an entity that was used in the creation. For instance, a cylinder that was used to produce a hole could be moved or its diameter changed in the database. When

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the model is rebuilt, the changes would be evident. The existence of a CSG tree is also very useful in geometric calculations such as surface area or volume.

On the other hand, the B-rep information is much more valuable for display purposes. After all, it is the pot various surfaces that one sees when looking at an object. B-rep information is used for hidden surface and line removal and for various rendering schemes. It should be pointed out that B-rep information could be derived from a CSG tree. It is usually impossible to create a CSG tree from a B-rep database, however.

9.5.1. Invalid Solids

Care is needed to avoid creation of invalid solids such as solids that have a face or surface with no area, a solid with no volume, or a wall with zero thickness. The software detects some of these problems and you will be told of them in the fort of an error message. Other times, the model will b created with a defect, at least as far as the physics world is concerned. Figure 9.14 depicts a CSG process where two wedge-shaped solids are subtracted from the prism. The resulting model is treated as a single object by some software but physically represents two objects that just touch.
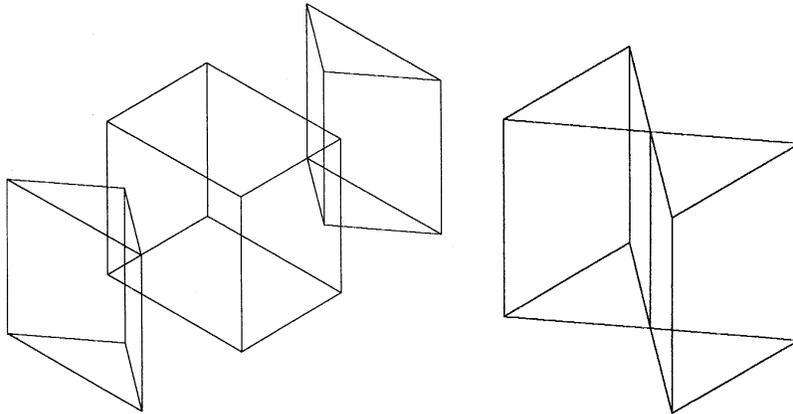

Figure 9.14. A physical invalid solid

An object with zero wall thickness can be generated via CSG methods as in Figure 9.15. To exactly construct this object physically would be impossible.
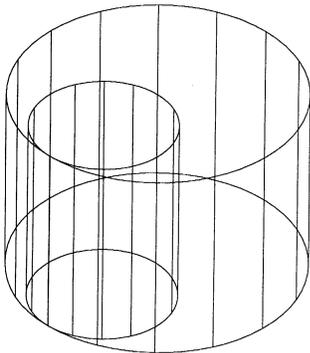

Figure 9.15. Zero wall thickness problems

Sweeping or extrusion processes can also result in problems. The objects in Figure 9.16, if extruded, will produce zero wall thickness. Visual inspection of the model from several viewpoints is a good test to guard against creation of invalid solids.
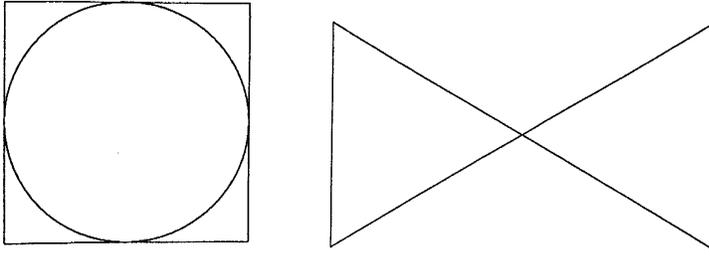
11

Figure 9.16. Unextruded shapes into valid solids

9.5.2. Sketch Planes

When you are working with solid-modeling software, you must remember that you are working in 3-D space. Sketches are typically made on a 2-D plane somewhere in space. Therefore, one of the first things you will need to do before making most sketches is to define the plane on which you are sketching. Otherwise, the computer will not know exactly where your sketch is located with respect to other geometric entities that may be on the screen or stored in memory.

You can move this working plane around in 3-D space by rotating or translating it so that you are not confined to always sketching from the same planar location in space. Sometimes you can attach this working plane to surfaces on an existing object such that you will sketch directly onto that surface. Figure 9.17 shows a block with a 2-D shape sketched onto one of its surfaces. Note that, from this view, it is really impossible to tell exactly where the shape is located in 3-D space; however, since you know that it was drawn on the indicated surface, its location is fixed in space.
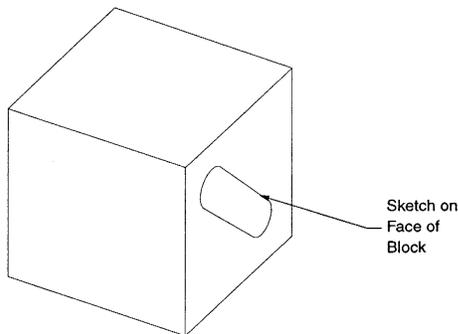

Sketch on
Face of
Block

Figure 9.17. Two-Dimensional Geometry sketched on a surface

9.5.3. Modeling Strategy

The methods that you choose to create 3-D solids will depend on the shape of the geometry you are trying to model. Many times, you can use more than one method to achieve the desired result. It is up to you to choose the method with which you are the most comfortable. Figure 9.18 shows two methods for creating a simple cylinder. With the first method, a circle was drawn and extruded to form a 3-D solid. This method, which involves sketching on a planar surface and extruding to cut out a cylindrical feature, is especially useful when you are creating circular holes in an existing object. With the second method, a rectangle was drawn and revolved about one of its edges to form a 3-D object. Both methods, of course, resulted in the same object.
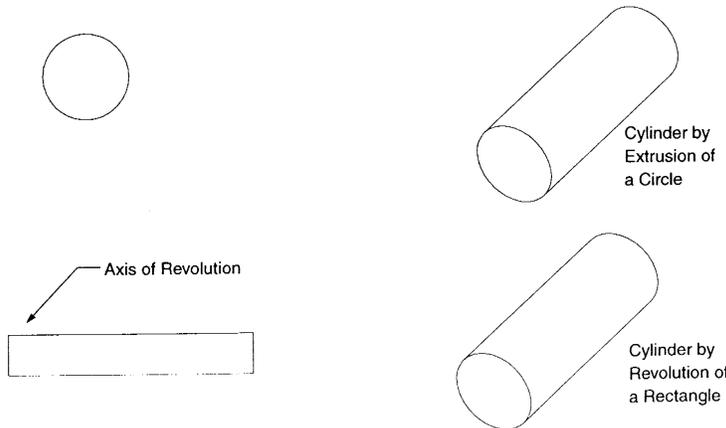
Figure 9.18. Creating a Cylinder by Extrusion or by Revolution

Most times when you start creating a model, it is important that you plan what you are going to do before you sit down in front of the computer to begin your actual work. Basically you can start with a larger object and remove material from it, or you can start by always building up or adding to an initial shape. Sometimes it is easiest to use a combination of these strategies. The method you choose will depend on your personal preferences, among other things.

Questions

1. Draw or describe a dialog box, which would create bosses in a feature-based modeler.
2. Explain what is meant by the parent-child relationship that exists between parametric features.
3. List and describe various revolved features available within CADD packages
4. Describe the necessary requirements for sketching a revolved feature's section.
5. Describe the procedure for creating a diameter dimension within a section that will be revolved.
6. What revolved features require a vertical centerline?
7. Describe the difference between a hole placed with the radial option and a hole placed with the linear option.
8. Describe each of the feature creation options found under the protrusion, cut, and slot options.
9. What is the difference between a solid feature and a thin feature?
10. How does the slot option differ from the cut option?
11. Describe methods available within part mode to modify a dimension's value.

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