

11. Transferring 3D Parts To Drafting Detailing

The complete documentation of a product includes more information than just the 3D geometry. Other information required to completely document a product design includes the following.

- Geometry (Topology)
- Dimensions
- Tolerances for each dimension
- Geometric tolerances of features (*Datums, GD&T, feature control frames*)
- Manufacturing information (*Surface finish, welding notation, etc.*)
- inspection information (*key location points*)
- Assembly instructions
- Product information *Materials, Suppliers, Part numbers, Bill of Materials, Revision history, etc.*)

11.1. 2D VS. 3D DOCUMENTATION

The product documentation listed above has traditionally been recorded on 2D paper drawings. The trend is toward both an electronic paperless environment and using a 3D representation rather than 2D drawings. The speed of this transition varies with different companies in different industries.

Companies tend toward either a “model-centric” or a “drawing-centric” philosophy of how their products are documented. The difference is not whether they use 3D models or 2D drawings or both. The question is which form contains the true released information. For example, a “drawing-centric” company may perceive of 3D part models as a preliminary step to create 2D drawings, but the released 2D drawings are archived to document the final design. A “model-centric” company may hold a view that the 3D part model contains the master information, and the 2D drawings are only an intermediate form of communication to transmit information to suppliers who need printed drawings.

Which view a company holds will affect how they use 2D and 3D documentation. For example, manufacturing information such as GD&T feature control frames (which will be described later in this chapter) can be placed either on the 3D part model or added as annotation to the 2D drawing.

Although there are times when a 2D approach is necessary, there is a trend toward more “model-centric” companies, because any information attached to the 3D part model is available for multiple uses.

11.1.1. Individual Roles

Just as companies have a model-centric or a drawing-centric view, individuals within these companies have a different view, depending on their uses for product notation. This will lead to different practices in how additional manufacturing and product information is created, documented and used.

Individuals with an Industrial Design role tend to focus on the form and aesthetics of a design. They may work entirely with 3D models and not be interested in 2D drawings or manufacturing information. Individuals in this role are more interested in the nominal geometry than in tolerance deviations.

Those in an Engineering role focus on the function of the product. They may mainly use 3D models, but occasionally generate an informal 2D drawing to get a prototype made. When defining allowable tolerances, they will understand this information in a 3D context. In their view, because additional product information defines the function of a product, they will tend to want it available with the 3D models.

Manufacturing users tend to focus on manufacturing processes and assembly fit. They work with surface finish, dimensions, and tolerance. This information is traditionally presented in 2D drawing views. Drafting users typically focus on the appearance of 2D documentation. They may create 2D drawings from existing parts and assemblies, or create drawings from scratch. In their view, dimensions and other notation belong to the drawing view instead of the 3D part.

Because users in each of these roles have different uses of product information, they may prefer to create it at different phases of the development process and store it in different ways. The software allows for different methods of creating and presenting this information to accommodate different needs.

Once you've designed and created your part, you need to be able to present it to others. The typical way to do this is to generate a detail drawing of the part. From this drawing, the physical part can be manufactured.

11.2. CREATING DRAWING VIEWS

CADD software may create the drawing views automatically from your 3D part. You start from a base view and then add the different types of views that you desire such as orthographic, auxiliary, or isometric. You simply indicate where the views will go. The dimensions come from the specifications you used to create the part. You may find that all you have to do is alter the placement of the dimensions to complete the drawing. Even the hidden lines can be added automatically

Another great feature is that the drawings will update automatically as you change your part. The information contained in the part and drawing are linked. If you change the part, the drawing will change to reflect your alterations. Conversely, if you change the drawing, the part will change. This allows for easier and quicker modifications. A typical process for creating 2D drawings is

1. get a 3D part,
2. create the 2D drawing and,
3. add annotation to it.

This order may depend on the practices of the company and the roles of the individuals involved. The drawing can be stored electronically for retrieval, or plotted on paper. There are a variety of types of views that can be used to represent your part, allowing you a huge amount of flexibility in the layout of your drawing. You can create single views one at a time or multiple views all at once.

In addition to the necessary views and the annotation, you may need to include additional "special" views common to engineering drawings: details, sections, and auxiliary views. Detail views are included on a drawing to show a small portion of the part so that it is "zoomed" or enlarged. If you have a feature on a part that is small relative to the overall size of the object, you will probably want to show this feature as a detail view for clarity. Typically, dimensions for the small feature will be added to the detail view rather than the view of the entire part, in order to remove clutter from the drawing.

There are six basic view types: base, ortho, aux, iso, detail, and broken. You can add hidden lines and sections to these views.

Base. The base view is the first view to be created since it is used to create other views. They are based on their location and viewpoint. You can pick from a top, front, or side view for the base view.

Ortho. This is the typical orthographic view that is projected from another view, either horizontally or vertically.

Aux. This is an auxiliary view that is created perpendicular to a selected edge on another view. It is typically used to view a slanted surface without distortion, as might happen with an orthographic view.

Iso. This type of view represents an isometric view at the standard 30 degree angle. You can pick from one of the four quadrants to generate the view. Isometric views can help others understand your design when the part is a complex shape.

Detail. The detail view is used to focus on a particular area of an existing view. For example, it is used to enlarge an area to be viewed more clearly, making it easier to refine a part's dimensions.

11.2. EXTRACTING DATA FROM 3-D MODELS

Three-dimensional computer tools have revolutionized the engineering design process. In some cases, engineers can work directly from 3-D computer models in the manufacture of parts or assemblies. In many instances, however, a drawing is still required as the design documentation necessary for the manufacture of a part. It is relatively easy to extract a drawing from a 3-D solid model, and most currently available computer-aided design software packages have this capability. Other data that you may wish to extract from your 3-D model include on-screen measurements and the physical properties of a model. Finally, some applications, such as stereo lithography and rapid prototyping, require that you extract cross sections from models at various locations. Most modern-day software has this ability. Being able to extract cross sections from parts is particularly useful if you have created the part by lofting or sweeping. The remainder of this chapter will cover these types of data extraction.

11.2.1. Extracting Physical Data from 3-D Parts

In some instances, you may wish to compute the physical properties of your 3-D models. For example, you may wish to minimize an object's volume or weight. If you check the physical properties of the object as you make changes to it, you should be able to optimize your design through an iterative process. The types of properties that can be typically calculated from a 3-D part definition include the surface area, volume, mass, density, and center of gravity of the part. In addition, the software can also often calculate inertial properties, such as radii of gyration and moments of inertia. The physical properties of a part can be computed about any set of axes—either the global, the local, or some other set of user-defined axes. Properties can be input by material type for computational purposes. For example, you can specify that a given part is composed of steel, and the software will automatically use the standard density of steel in its calculation of physical properties.

11.2.2. Creating 2-D Drawings from 3-D Models

In most cases, engineering drawings are drawn to *scale* because it is virtually impossible to always make drawings that are true size. You are familiar with the concept of scaled drawings from your experience. Engineers use scales in drawings that are based on either the metric or the English system of units (inches, feet, etc.).

When creating a drawing from a 3-D model, you first typically specify the size of the paper on which you wish to create the drawing. In the metric system of units, there are five standard drawing sizes that are roughly similar to the English paper sizes: A4 (210 mm x 297 mm), A3 (297 mm x 420 mm), A2 (420 mm x 594 mm), A1 (594 mm x 841 mm), and AO (841 mm x 1,189 mm).

The scale you use for your drawing will depend on the size of the part relative to the size of the paper you are using. Some typical scales are: true size, half size, quarter size, and eighth size. In setting up your drawing from your 3-D part, make sure that you have enough room on the page for dimensions. Note that

the drawing scale does not change the size of the part itself. Specifying a drawing scale only controls your viewing scale of the object.

You may need to specify before you create your drawing from an object is whether you wish to include a title block on the drawing. If you are working for a company, you will probably include the standard corporate title block on any drawing you create. Otherwise, most software packages have a standard title block for your use in creating drawings.

Engineering drawings typically contain one or more orthographic views of an object and may also contain an isometric pictorial for clarity's sake. The views that are typically found on a drawing are the top, front, and right-side views, although others may be included as necessary. You are already familiar with being able to view a 3-D object from several different viewpoints as you are working with your model in 3-D space. The difference when you are creating a drawing is that you will display the object from multiple viewpoints simultaneously.

Figure 11.1 shows a 3-D part and a four-view drawing that was created from the part. Note that the drawing shows the top, front, right-side, and isometric views of the object. Note also that the drawing contains dimensions in addition to the geometric information.

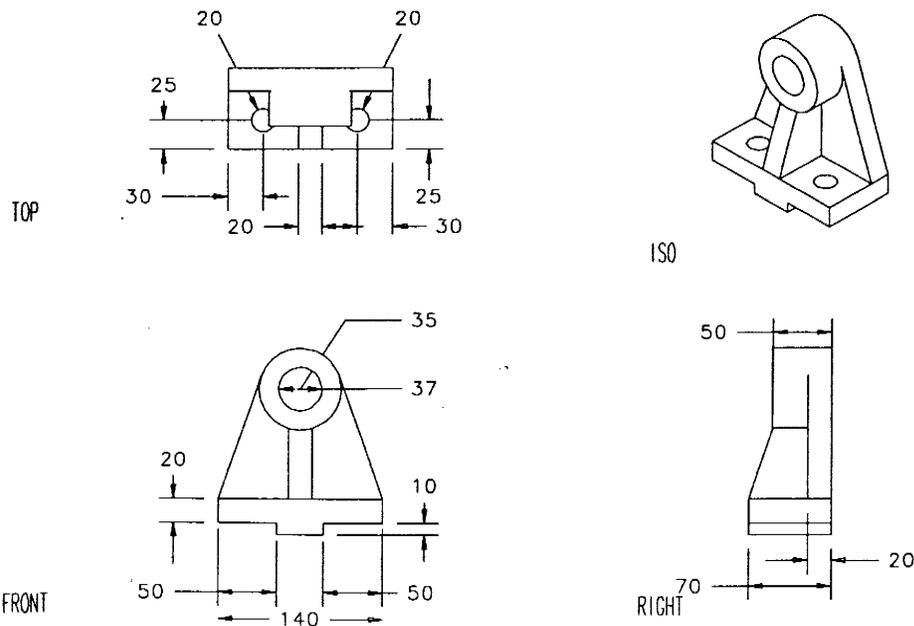


Figure 11.1. Drawing Extracted from Part Geometry

Annotation in the form of dimensions, notes, and labels is commonly found on engineering drawings. Notes are merely text that appears on a drawing, whereas labels consist of text plus an arrow pointing to some specific feature on the drawing. Although dimensions are typically included in the definition of the object, many times you will need to edit a drawing generated from a 3-D model to include either more or different dimensions than those that were used to define the part. This type of annotation is easily added to such a drawing.

Slicing through an object to expose its interior creates sectional views. In this way, numerous hidden lines will not be necessary on the drawing itself: Once you expose the interior features, they are no longer hidden. Cross-hatching is included on the drawing where the cutting plane passes through material.

Typically, cross-hatching is considered to be a type of annotation inserted when you work with computer-aided design software tools. Many types of cross-hatching patterns and many types of sectional views can be defined. Because our focus is on solid modeling, these topics are beyond the scope of the text. Figure 11.2 shows a drawing with a top view and a front sectional view of a piston. Also shown are a detail view of the region near the location of the grooves for the piston rings and an isometric view of the object.

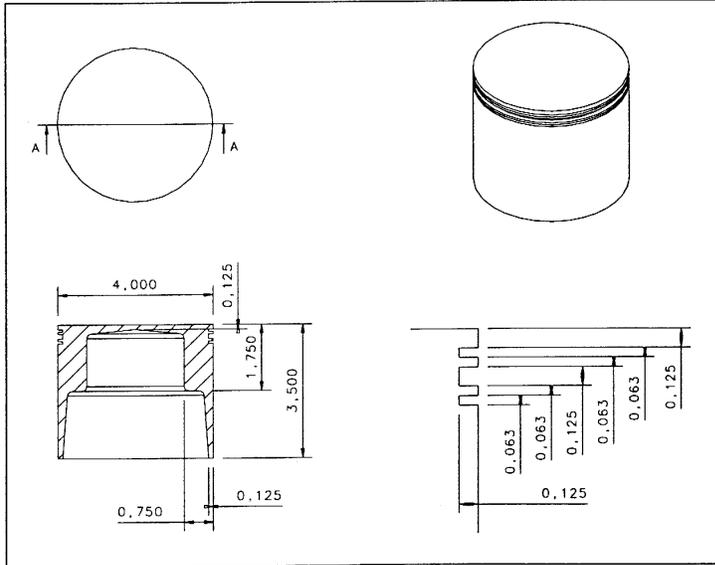


Figure 11.2. 2-D Drawing of Piston

Auxiliary views are defined as any view other than one of the six primary views, which consist of the front, top, right-side, left-side, back, and bottom views. A typical use of an auxiliary view is to show an inclined surface of an object in true size and shape, especially when the surface has a feature on it such as a hole that must be located accurately for manufacturing purposes. Since an inclined surface will not appear in true size or shape in any of the primary views, a true-size auxiliary view is necessary to define the location of the hole. Auxiliary views can be constructed to show the true-size view of the inclined surface only or to show the entire object from this viewpoint, depending on the capabilities of the software.

11.2.3. Associativity

Most modern-day computer-aided design tools have either uni- or bi-directional associativity. With unidirectional associativity, if you make a change to the 3-D part definition, any drawings you have created from the part will automatically be updated to reflect the changes made to the model. With bi-directional associativity, if you change the part, the drawing will change, but in addition to this, if you change a dimension on the drawing, the 3-D part will change. Associativity is a useful feature of computer-aided design software in that you do not have to continuously recreate drawings from a part as you modify or change it.

References

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