Unit One Engineering Graphics and CAD

Lesson 1 Engineering Graphics in the Third-angle Projection

教学目的和要求

通过本文的学习，了解第三角画法的概念、投影原理和应用。要求掌握工程图学的各种视图及投影规律的英文专业术语和表达习惯，了解第三角投影和第一角投影画法的区别和变换。

重点和难点

（1）了解第三角投影的原理和投影规律。
（2）掌握第三角画法的组合体、零件图、装配图的专业词汇和表述。
（3）难点在于第三角投影和第一角投影画法的变换。

Part A Text

Preface

Graphics comes to our vocabulary from the Greek word grapho, whose extended meaning is *drafting* or *drawing*, the drawing is the primary medium for developing and communicating technical ideas[1]. Engineering drawings provide an exact and complete description of objects. In addition to a description of the shape of an object, an engineering drawing gives all further information needed to manufacture the object drawn, such as dimensions, tolerances, and so on. So, it is often said the engineering drawing is the common language of engineering. Every engineer must master this language. The main tasks of engineering graphics: to learn the knowledge of the projections to cultivate the drawing making and drawing reading abilities to cultivate and develop the spatial analysis and spatial visualization abilities.

As a common language of engineering，the drawing is used to direct the production and make the technical interchange. So, it is necessary to specify, in a unified way, drafting practices, such as the layout of drawings, dimensioning, and so on.

★ **Formation of three-projection views**

Three projection planes system: Since it is impossible to determine the position of a point with its only one projection, more projection planes are added. Usually, three projection planes perpendicular to each other are used in orthographic projection. They are horizontal projection plane, frontal projection plane and profile projection plane, denoted by H, V, W, respectively.

The third-angle projection: Three projection planes divide space into eight parts or quadrants numbered from 1 to 8, as shown in Fig.1.1. According to the Chinese National Standard of Technical Drawings, the first-angle projection is used to make engineering drawings while in some other countries, such as in the USA and Canada, the third-angle projection is used[2]. In this paper we focus on the third-angle projection.

Formation of three-projection views: In first angle, an object is placed in quadrant 1, and observer always looks through the object towards the projection plane. But in third-angle, the object is placed in quadrant 3, and observer always looks through the projection plane towards the object. In third-angle projection, projection plane is assumed transparent, so form into views.

The symbols of the first -angle and the third -angle projection in the following Fig.1.2

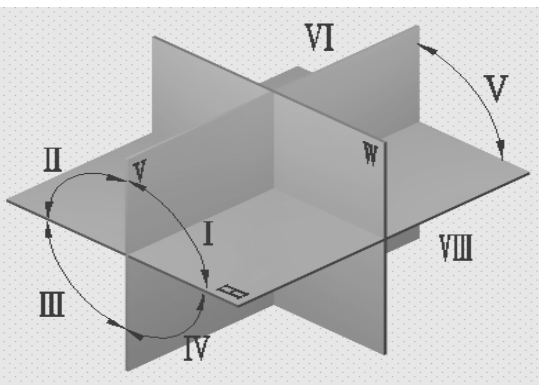
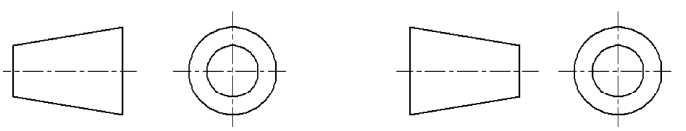


Fig.1.1　Projection planes



（a）The first-angle projection　　（b）The third-angle projection

Fig.1.2　The symbols of the different angle projection

★ **Composite Objects**

Projection rules of an object: The front and top views are aligned vertically to show the width of the object. The right and left views are aligned horizontally to show the height of the object. The

top and right views have the same depth of the object.

Drawing three views:

(1) Analyzing-shape method: Any composite object can be broken into a combination of some primary geometric object. Any of these basic shapes can be positive, classified to the superposition style and the cutting style.

(2) Select the projection: Because the front view is the most important one in the three views, it is very important to select adequate projection direction to form the front view.

(3) Drawing steps: Locate the axis lines, center-lines of symmetry and base lines; Draw the base with H pencil, check the drawing and darken the lines.

Reading the composite views:

Points of reading views: break the object into its individual basic shapes; when reading views at least two views should be read simultaneously, master the meanings of lines and areas in views.

Methods to read views:

(1) Analyzing shape method: Break the object down into its basic geometric solids.

(2) Analyzing lines and planes method: Break the object into various surfaces and lines.

For example, like the following Fig.1.3, the three views of a Composite Object.

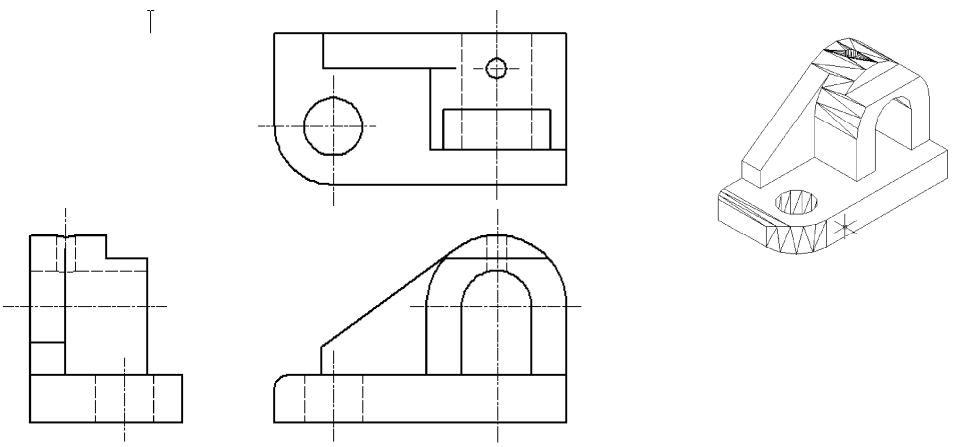


Fig.1.3 Three Views of Composite Object

★ Detail drawings

Detail drawings: it is a drawing that indicates the construction, size, and technical requirements of a part. It describes its shape, gives the dimensions, provides all the information needed to make the part.

The contents of a detail drawing:

(1) A sufficient number of views to give a complete shape description of the exterior and interior

constructions of the part.
(2) All the dimensions needed for manufacturing the part.
(3) Technical requirements including tolerances, geometric tolerances, surface roughness, material specification, heat treatments, and so on.

Selection of views: To meet the requirements of making a clear and complete shape description of the object, the first thing to decide before starting to draw is which views are needed and the best way to position the part on the drawing.
(1) Selection of main view: the characteristic shape principle, the functioning position principle, the machining position principle.
(2) Selection of other views: to limit the number of views to the minimum necessary and sufficient principle, to avoid the use of hidden lines principle, to avoid unnecessary repetition of details principle.

★ Assembly drawings

Assembly drawings: A drawing that shows the parts of a machine or machine unit assembled in their relative working position is called an assembly drawing.
The contents of an assembly drawing are as follows:
(1) A set of views showing the positional relationship and mutual operation of the parts being assembled.
(2) A few dimensions that are needed to show the positional relationship between critical parts, the positioning of the product at site, etc.
(3) Technical requirements including all the information necessary for assembling, checking, and maintaining the machine.
(4) Item numbers for each part, the item list, and the title block.

Conventions in assembly drawings:
(1) General conventions: In an assembly drawing there is no gap shown between contact surfaces or mating surfaces. On the other hand, a gap should be shown between non-contact surfaces or non-mating surfaces. Section lines of adjacent parts should be carried out with different directions or spaces. In an assembly drawing, solid parts cut along their axis are shown without section lines. Such as shafts, axles, rods, handles, pins, keys, etc. Screws, bolts, nuts, and their washers also keep its shape.
(2) Special conventions: Representation of making the cut along joint face or taking some parts apart, Representation of showing parts separately, Representation of using phantom lines, Exaggerated representation, Simplified representation[3].

Words

graphics	*n*. 制图，图学
drafting	*n*. 草图，制图
drawing	*n*. 绘图，制图，图样
projection	*n*. 投影
dimension	*n*. 尺寸；*v*. 给……标注尺寸
spatial analysis	空间分析
spatial visualization	空间想象
horizontal projection	水平投影
frontal projection	正投影
profile projection	侧投影
quadrant	*n*. 象限
center-lines of symmetry	对称中心线
composite object	组合体
detail drawing	零件图
assembly drawing	装配图
phantom line	假想线

Notes

[1] Graphics comes to our vocabulary from the Greek word grapho, whose extended meaning is *drafting* or *drawing*, the drawing is the primary medium for developing and communicating technical ideas.
【译文】“图学”一词来源于希腊字“grapho”，其延伸意义为“绘图”或“图样”。图样是开发和交流技术思想的主要工具。

[2] According to the Chinese National Standard of Technical Drawings, the first-angle projection is used to make engineering drawings while in some other countries, such as in the USA and Canada, the third-angle projection is used.
【译文】依据中国机械制图国家标准，制图采用第一角投影，而其他一些国家如美国和加拿大则采用第三角投影。

[3] Special conventions: Representation of making the cut along joint face or taking some parts apart, Representation of showing parts separately, Representation of using phantom lines, Exaggerated representation, Simplified representation.
【译文】特殊规定：沿结合面剖切或把某些零件拆开的画法，单独表示零件画法，使用假想

线画法，夸大画法，简化画法。

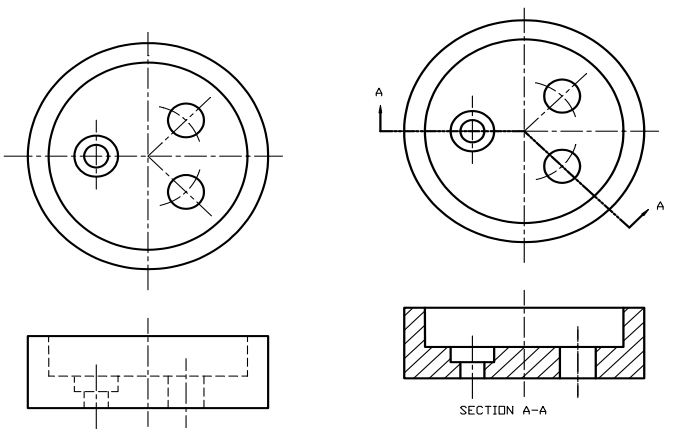
Part B Reading Materials

Similar to an offset in that the cutting-plane line staggers, however, it differs in that the cutting-plane line is offset at some angle other than 90°. When the section is taken the sectional view is drawn as if the cutting—plane is rotated to the plane perpendicular to the line of sight. This is why the right-side sectional view may sometimes be elongated (depending on the shape).

Exercise：

Drawing the front view to aligned section.

The Fig.1.4(a) is the topic and Fig.1.4(b) is the answer.



(a) (b)

Fig.1.4 The Topic and the Answer

Part C Exercise

A brief introduction to the difference between the first projection and the third projection.

Lesson 2　CAD Application

教学目的和要求

本文简要介绍了计算机辅助设计（CAD）的软件用途及分类，以 AutoCAD 为例，介绍了 AutoDesk 的绘图界面。计算机辅助绘图软件是机械工程专业的通用软件，其操作命令都是以英文输入的。通过本文的学习，可以了解有关 CAD 的英文表达和绘图界面的常见图标的英文名称，有助于学习和应用英文版本 CAD；要求掌握文中所涉及的绘图界面图标的英文名称，并能在实际应用中不断积累绘图命令的英文名称及快捷命令，并能根据文后所附的练习绘制出同样的图形。

重点和难点

（1）重点掌握 CAD 相关的专业术语及表达。

（2）掌握 AutoDesk 绘图界面各部分的名称和图标的名称。

Part A　Text

CAD refers to Computer Aided Drafting. CAD has replaced the traditional drawing instruments such as triangles and pencils, but it still has not, and probably never will, practically replace the ideal freehand sketch or the designer's experience with geometry and graphical conventions and standards.

CAD gives us increased accuracy, productivity and transferability. It affords us the flexibility to change drawings with minimal effort. Before CAD, a minor mistake made on a manual drawing could mean extensive time and cost to correct a major mistake resulted in a recreation of the entire drawing. CAD also eliminates the need to frequently redraw standard equipment, components, and details.

AutoCAD was first released in 1982. AutoCAD is undoubtedly the world's leading CAD software for 2D drafting, detailing, design documentation, and basic 3D design. It is a vector graphics drawing program that uses primitive entities—such as lines, polylines, circles, arcs, and text—as the foundation for blocks and more complex objects. AutoCAD can be customized for the various uses/disciplines by adding interfaces such as Architectural Desktop, AutoCAD Electrical and Mechanical Desktop. AutoCAD has a full set of basic solid modelling and 3D tools, but lacks advanced capabilities of solid modelling applications compared to such software as Pro/ENGINEER and Solid Works[1].

There are many Computer Aided Drafting programs on the market today with each having its own strengths and weaknesses.

CAD distinguishes itself into **three** main categories:

- 2D,
- mid-range 3D solid Modelling，
- high-end 3D hybrid systems.

Some programs are customized for specific disciplines (e.g. ECAD for Electrical Engineering design and MCAD for Mechanical Engineering design). Some programs are designed for entry level drafting, others for advanced design. For example, ArchiCAD has the ability to design virtual tours of building models and also estimate monthly/annual energy cost[2]. Pro/ENGINEER is designed for various platforms, such as UNIX, Windows, and Linux and is exchangeable between platforms with noticeable conversions. AutoDesk Raster Design can aid you in turning scanned images into vector drawing with some editing that you can open with another computer aided drafting program for editing[3].

Interactive third-party software enhances more generic software by allowing information created from one software program to be viewed and edited by another. For example, SolidWorks is a program that can create a 2D drawing from a 3D image and vice versa.

Note:

If this dialog box does not appear when you start AutoCAD your drawing will be set for either imperial or metric, possibly without you knowing which. Therefore, you should set the system variable to a value of 1 so that it will appear when you choose to open a new drawing, as shown in Fig.2.1. To do so, at the command prompt, type **STARTUP** and enter a value of 1.

Another option to ensure you are working in a metric prototype drawing is to select File, New and choose **acadiso.dwt** (metric template).

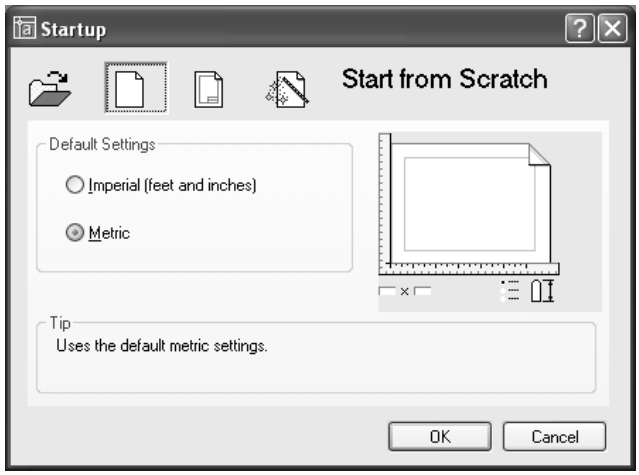


Fig.2.1 Start Up Dialog Box

Open a Drawing Opens an existing drawing, as shown in Fig.2.2. An AutoCAD drawing has a .dwg filename extension.

Start from Scratch Begins a drawing quickly. You will be prompted to use one of two measurement systems (English or Metric).

Use a Template Starts a drawing/setting based on a template that contains preset values. A template drawing would have a. dwt filename extension. There are over 60 predefined templates.

Use a Wizard Leads you through setting up a drawing.

Note:

Other common filename extensions in addition to those already mentioned include:

- .bak (backup drawing—see saving a drawing).
- .plt (plot file—see plotting).
- .ctb (see plot style table).

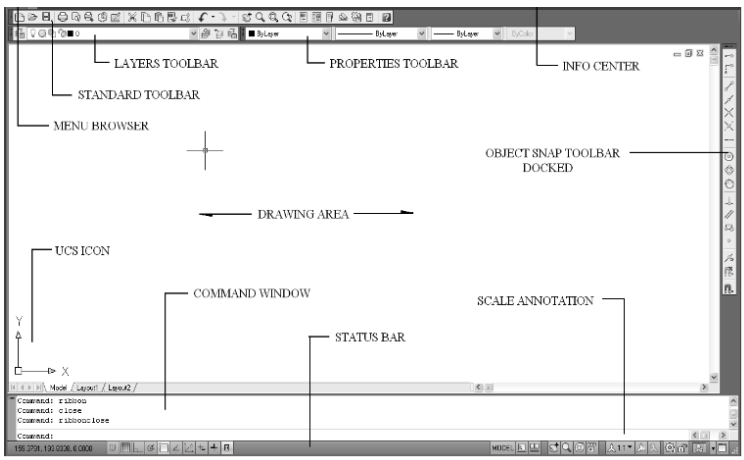


Fig.2.2 Interface (without Ribbons)

Menu Browser Menu commands that are organized in groups such as File, Edit, View, Insert, Format, Tools, Draw, Dimension, Modify, etc.

Status Bar Displays X, Y Co-ordinates and Drawing Aids (SNAP, GRID, ORTHO, POLAR, OSNAP, OTRACK, MODEL). To the right of the Status bar you will see a series of icons used to control the drawings '**Scale Annotations**' to simplify size of annotation

	in relation to actual size of the drawing. It affects dimensions, linetype definitions and hatch patterns. Scale annotation can be activated in the Dimension dialog box setup.
Drawing Area	Commonly referred to as Drawing Editor. It displays the drawing.
Command Window	Displays history of commands.
UCS	**U**ser **C**o-ordinate **S**ystem icon is a standard co-ordinate system showing the direction of the X and Y axes. The icon can be turned off by typing the command: **UCSICON**.
Toolbars	Displays icons to initiate commands. To display a toolbar, right-click on an existing toolbar and select from the drop-down list. Example of a toolbar in AutoCAD: **Standard Toolbar** includes Open, Save, Print, Zoom, Check, Cut, Copy, Paste, etc.

Fig.2.3 Interface (with Ribbons)

Ribbons are frequently used commands and options that are visually grouped together, as shown in Fig.2.4. If ribbons are not visible you may type RIBBON at the command prompt to turn them on. To turn ribbons off, type RIBBONCLOSE at the command prompt.

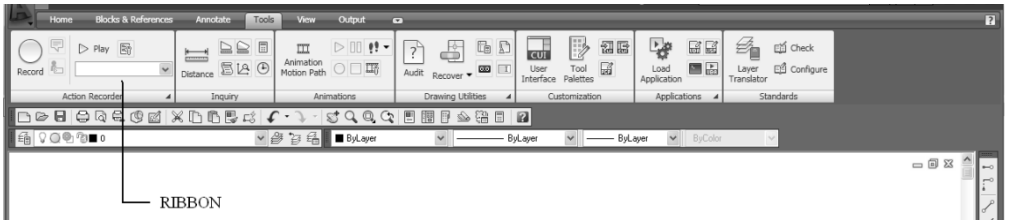


Fig.2.4 Ribbons

Words

vector	*n*. 矢量
graphics	*n*. 制图学；制图法；图表算法；
polylines	*n*. 多段线
circle	*n*. 圆
arc	*n*. 圆弧
block	*n*. 块
imperial	*adj*. 英制的

metric	*adj*. 公制的
dimension	*n*. 尺寸；*vt*. 标出尺寸，尺寸标注
modify	*n*. 修改
linetype	*n*. 线型
activate	*vt*. 激活
dialog box	*n*. 对话框
co-ordinate	*n*. 坐标
icon	*n*. 图标
paste	*vt*. 粘贴

Notes

[1] AutoCAD has a full set of basic solid modelling and 3D tools, but lacks advanced capabilities of solid modelling applications compared to such software as Pro/ENGINEER and SolidWorks。

【注释】

①Pro/ENGINEER（简称 Pro/E）：该操作软件是美国参数技术公司(PTC)旗下的 CAD/CAM/CAE 一体化的三维软件。Pro/E 第一个提出了参数化设计的概念，并且采用了单一数据库来解决特征的相关性问题。另外，它采用模块化方式，用户可以根据自身的需要进行选择，而不必安装所有模块。应用 Pro/E 提供的基于特征方式，能够把从设计到生产全过程集成，实现并行工程设计。它不但可以应用于工作站，而且也可以应用到单机上。Pro/E 采用了模块方式，可以分别进行草图绘制、零件制作、装配设计、钣金设计、加工处理等，保证用户可以按照自己的需要进行选择使用。

②SolidWorks：SolidWorks 是达索系统(Dassault Systemes S.A)下的子公司，专门负责研发与销售机械设计软件的视窗产品，公司总部位于美国马萨诸塞州。SolidWorks 软件功能强大，组件繁多，有功能强大、易学易用和技术创新三大特点，这使得 SolidWorks 成为领先的、主流的三维 CAD 解决方案。SolidWorks 能够提供不同的设计方案、减少设计过程中的错误以及提高产品质量。SolidWorks 不仅提供如此强大的功能，而且对每个工程师和设计者来说，操作简单方便、易学易用。在目前市场上所见到的三维 CAD 解决方案中，SolidWorks 是设计过程比较简便而方便的软件之一。美国著名咨询公司 Daratech 评论：“在基于 Windows 平台的三维 CAD 软件中，SolidWorks 是最著名的品牌，是市场快速增长的领导者。”

[2] Some programs are customized for specific disciplines (e.g. ECAD for Electrical Engineering design and MCAD for Mechanical Engineering design). Some programs are designed for entry level drafting, others for advanced design. For example, ArchiCAD has the ability to design virtual tours of building models and also estimate monthly/annual energy cost.

【注释】

① ECAD：电气工程计算机辅助设计。

② MCAD：机械工程计算机辅助设计。

③ ArchiCAD：虚拟建筑设计，ArchiCAD 不同于原始的二维平台及其他三维建模软件，其

中最重要的一点就是能够利用 ArchiCAD 平台创建的虚拟建筑信息模型，进行高级解析与分析，如绿色建筑的能量分析、热量分析、管道冲突检验、安全分析等。在 2D 环境下，每一张图样都是建筑师一笔一笔地以线条形式画出来的：先从平面图开始绘制，然后画立面图、剖面图，最后再按照项目进展更改所有的图样。然而，在“虚拟建筑”中，ArchiCAD 虚拟建筑设计平台将彻底地改变这个工作过程，使建筑师能够从各种繁杂的图样绘制工作中抽身。在这里，设计的核心工作不再是绘制施工图样，而是以虚拟建筑信息模型为工作中心，所有的图样都直接从模型中生成。建筑师将集中更多的精力用于建筑物的设计，而图样将彻底成为设计的副产品。

[3] AutoDesk Raster Design can aid you in turning scanned images into vector drawing with some editing that you can open with another computer aided drafting program for editing.

【注释】

①AutoDesk：中文名称为欧特克，是全球最大的二维、三维设计和工程软件公司，旗下拥有 AutoCAD、3DS Max 等知名软件。

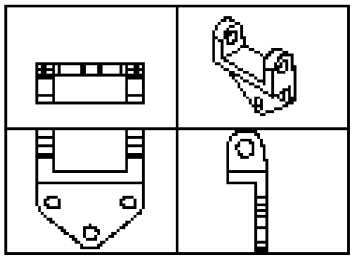
②Raster Design：译为光栅设计软件。在 AutoCAD 环境中，通过使用 Autodesk Raster Design 软件的光栅编辑、操纵和矢量化工具，扩展光栅数据的价值。在项目中添加扫描的图纸、地图、航拍照片、卫星图像和其他形式的光栅数据，并使用功能强大的清理工具来提高图像质量。可以仅对那些需要修改的光栅图元进行矢量化，从而节省编辑时间。

Part B Reading Materials

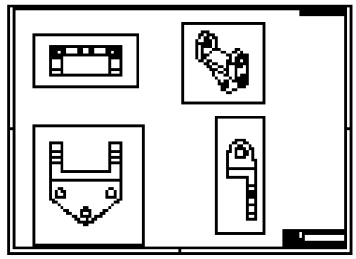
Viewports

When working in Model space you draw geometry in tile viewports [Fig.2.5(a)] which are represented by Viewport Table Record objects. You can display one or several different viewports at a time. If several tiled viewports are displayed, editing in one viewport affects all other viewports. However, you can set the magnification, viewpoint, grid, and snap settings individually for each viewport.

In paper space, you work in floating viewports [Fig.2.5(b)] which are represented by viewport objects and can contain different views of your model. Floating viewports are treated as objects that you can move, resize, and shape to create a suitable layout. You also can draw objects, such as title blocks or annotations, directly in the Paper space view without affecting the model itself.



（a）Tiled Viewports



（b）Floating Viewports

Fig.2.5 Tiled Viewports and Floating Viewports

Part C Exercises

Can you draw the figure of hollow pipe below according to the drawing process below? Please explain the meanings of the following commands.

1. surfab 2:
2. tabsurf:
3. move:
4. revsurf:
5. hide:

Command: surftab 2
New current value for surftab 2 (2 to 32766) <6>: 10
Command: _tabsurf
Select entity to extrude:
Select line or open polyline for extrusion path:
Command: move
Select objects:
Specify base point or [Displacement] <Displacement>:0,0,1000
Command: _revsurf
Select a linear entity to revolve:
Select the axis of revolution:
Angle to begin surface of revolution <0>:
Degrees to revolve entity (+ for ccw, - for cw <360>: 90
Command: _revsurf
Select a linear entity to revolve:
Select the axis of revolution:
Angle to begin surface of revolution <0>:
Degrees to revolve entity (+ for ccw, - for cw <360>: -90
Command: _move
Select objects:
Specify base point or [Displacement] <Displacement>:0,0,1000
Command: hide

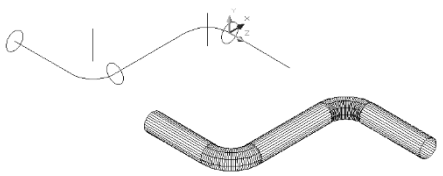


Fig.2.6 Hollow Pipe

Lesson 3　UG Application

教学目的和要求

本文介绍 UG 软件提供的混合实体/曲面建模的功能优势，首先说明实体建模和曲面建模的优缺点，以及为了实现这两种建模的优势组合而产生的混合建模；其次说明混合建模的应用场合并举例说明其特点。在熟练应用 UG 软件的基础上，再通过本文的学习，读者可以了解 UG 软件中关于混合实体/曲面建模的应用及相关术语的英文表达。

重点和难点

（1）重点掌握 UG 软件中关于建模的相关专业术语及表达。

（2）本文的学习首先建立在对 UG 软件熟悉的基础上，有了基础才能更好地理解本文，建议有选择地学习。

Part A　Text

What is hybrid solid/surface modeling and what are the benefits?

Almost all Modelers today use a boundary representation to represent solids. In a nutshell, this means that the object is represented by the surfaces that define the outer boundary of the volume. Thus, the axiomatic principle that is key to an understanding of solid versus surface modeling is that given enough time and the proper set of tools, any object that can be modeled with solids can be also modeled with surfaces[1]. The converse however is not true. In order to provide a reasonable set of operations, solid Modelers typically restrict the user to a finite set of shapes (i.e. spheres, boxes, etc.) and/or procedures (i.e. extrude, revolve, sweep, fillet etc.) that can be used to define the geometry of the object. This inherently limits the range of objects that can be described. In addition, they do not allow the representation of zero thickness partitions, such as the parting surface of a Mold. Because solid Modelers require that a true bounded volume be described at all times, manipulation and *fine-tuning* of the individual bounding faces is generally restricted.

Why then, with these limitations, has the use of solid modeling grown so dramatically? The primary reason is productivity. Since solid Modelers can perform in a single step what would generally take numerous operations with surfaces, most objects can be described much more efficiently. Also, these solid operations can generally be accompanied by describing dimensions and constraints in a fashion that allow them to be modified quickly and easily.

So, we see that there are advantages to both methodologies that can be summarized as speed (solids) versus flexibility (surfaces). It would seem then that the ideal Modeler would provide both techniques, in an environment that allows the user to easily switch back and forth between them. This is known as hybrid solid/surface modeling. In practice, the synergy of this combination

has proven to exceed the sum of its parts. A well-designed hybrid modeling system allows the user to leverage the power and efficiency of solids whenever possible, yet never force a design to be compromised for lack of an appropriate tool to achieve the desired geometry.

What applications is hybrid solid/surface modeling best suited for?

The ability to have a single CAD/CAM system that can be fast enough to design a gearbox on Monday and flexible enough to design a body panel on Tuesday is perhaps a compelling enough reason to have integrated solid and surface design. However, to truly realize the benefits of integrated solid/surface modeling one must look beyond this either/or paradigm. Consider, for example, the design of a camera or cellular phone. A true hybrid modeling system provides the functionality to design a beautiful, ergonomic body for the appliance with surface modeling tools and in the same environment apply a powerful and intelligent set of solid modeling tools to convert it into a thin shelled case, split it in half and add all the required bosses, ribs and flanges. If requirements dictate a geometric feature that cannot be achieved with one of the standard solid feature tools, in a hybrid modeling system surfaces can be used to *knife and fork* the desired shape[2]. Beyond the design of products that don't easily fit the standard feature-modeling paradigm, there are several other applications where the integration of solids and surfaces yields great benefits. One of these is for the repair or modification of models that have geometry problems, most often as a result of data translation or Modelers that allow geometry inaccuracies to accumulate.

Another is for the design of Molds & Dies, tools or fixtures where a shape needs to be partitioned into pieces along irregular boundaries, or faces of an object need to be extended or manipulated. Quite often supplemental geometry needs to be created for the purposes of NC tool path creation. A CAM system with an integrated hybrid Modeler is a tremendously powerful tool.

Will your organization benefit from Hybrid modeling?

Does your company do NC programming?

If yes, you need hybrid modeling. If no, continue.

Does your company work with imported solid or surface data via IGES, STEP or VDA[3]? If yes, you need hybrid modeling.

If no, continue.

Does your company design parts that have free-form shapes? Any feature of the part should be considered and not just the overall shape. For example, your design is 98% prismatic in shape but has one feature such as a cam that requires a free form or variably swept face.

If yes, you need hybrid modeling.

If no, continue.

Does your company design Molds, Dies, tools, or fixtures? If yes, you need hybrid modeling.

If no, continue.

Does your company use a solid Modeler that is limited in the functions available to create your designs? For example, does your parts have complex fillets or corner blends that your *state-of-the-art* solid Modeler is not capable of creating?

If yes, you need hybrid modeling.

If no, continue.

Are your designs primarily surface modeling based but you would like to realize the benefits of parametric, dimension driven modeling?

If yes, you need hybrid modeling.

Hybrid modeling example 1. Open the file *Advanced_Modeling* which is located on the training CD. Edit the part *01_Hybrid1*. Step through the part history to see how the model was constructed. Notice how the Bottom Face has been deliberately removed in order to make this a surface model or open shape.

In spite of this you can see that the extruded cut operations work even though the part is not a solid. In wire frame mode, some of the edges of the faces are shown as blue and dotted to indicate that they are not attached to other face edges. As shown Fig.3.1(a).

(a) (b)

Fig.3.1 Wire Frame Mode

In shaded mode ZW3D will show you the inside (or negative normal) faces clearly in a pink color. When creating faces that share common edges, the normals will be oriented in the same direction

automatically. If the faces eventually form an enclosed volume (i.e. a solid) then all the faces will be orientated so that the positive normal points away from the solid body.

Rotate the view so that you are looking underneath the part as shown below Fig.3.1(b). Note: open the face attribute option under Attributes pull-down menu, then set the color of the front and back from the pop-up dialog box.

You can clearly see that the extruded faces and the main body need to be trimmed to each other. Although there are many powerful trimming options in ZW3D that allow the user to trim the faces, it is nevertheless a time-consuming process. Instead we will use hybrid modeling techniques to quickly join the Shapes to each other.

From the Shape Tab, using the Combine > Add command , join Pick the main body as the base shape. Middle-click to advance the menu. With ZW3D hybrid modeling the user can use commands, normally associated with Solid Modelers, on surfaces and or solids.

For example. Using the fillet command let's put a 5mm fillet on all of the edges (window pick the entire part). Remember all of the commands shown in this part were done on a surface model. No need to save the file.

Words

hybrid modeling	*n*. 混合模型
modeler	*n*. 模型师；建模程序
nutshell	*n*. 简言之，一言以蔽之
axiomatic	*adj*. 不证自明的; 原则的
extrude	*v*. 拉伸
revolve	*v*. 旋转
sweep	*v*. 扫掠
fillet	*v*. 圆角
inherently	*adv*. 固有地
manipulation	*n*. 操作
fine-tuning	微调
paradigm	*n*. 范式
ergonomic	*adj*. 人类环境改造学的; 符合人机工程学的
boss	*n*. 凸台
rib	*n*. 筋
flange	*n*. (机械等的)凸缘，(火车的)轮缘

Notes

[1] In a nutshell, this means that the object is represented by the surfaces that define the outer boundary of the volume. Thus, the axiomatic principle that is key to an understanding of solid versus surface modeling is that given enough time and the proper set of tools, any object that can be modeled with solids can be also modeled with surfaces.

【注释】In a nutshell: 简言之，一言以蔽之。

【译文】简言之，即由确定了该体积外边界的多个曲面来表达某一对象。因而，对实体曲面建模的认识较为关键的原则，就是在时间充分、工具组合适的前提下，对那些能利用实体进行建模的对象，都可以用若干曲面进行建模。

[2] If requirements dictate a geometric feature that cannot be achieved with one of the standard solid feature tools, in a hybrid modeling system surfaces can be used to *knife and fork* the desired shape.

【注释】knife and fork：分割并构建。

【译文】若所要求的几何体特征无法通过其中一种标准实体特征工具达成，则可通过混合建模系统，利用若干曲面来“分割并构建”所需的造型。

[3] Does your company work with imported solid or surface data via IGES, STEP or VDA?

【注释】

① IGES：全称是 The Initial Graphics Exchange Specification，即初始化图形交换规范，它基于 Computer-Aided Design (CAD)&Computer-Aided Manufacturing (CAM) systems (计算机辅助设计和计算机辅助制造系统)不同计算机系统之间通用的 ANSI 信息交换标准。

② STEP：全称是 Standard for the Exchange of Product Model Data，即产品模型数据交互规范。这个标准是国际标准化组织制定的用于描述整个产品生命周期内产品信息的标准，是一个正在完善中的“产品模型数据交换标准”。它是由国际标准化组织(ISO)工业自动化与集成技术委员会(TC184)下属的第四分委会(SC4)制定的，ISO 正式代号为 ISO-10303。它提供了一种不依赖具体系统的中性机制，旨在实现产品数据的交换和共享。它不仅适合于交换文件，也适合于作为执行和分享产品数据库和存档的基础。发达国家已经把 STEP 标准推向工业应用，它的应用显著降低了产品生命周期内的信息交换成本，提高了产品研发效率，成为制造业进行国际合作、参与国际竞争的重要基础标准，是保持企业竞争力的重要工具。

③ VDA：全称是 Video Display Adapter，即视频显示适配器。VDA 文件是一个 VDA-FS CAD 文档，而 VDA-FS 是一种用于将表面模型从一个 CAD 系统传输到另一个 CAD 系统的 CAD 数据交换格式。

【译文】贵公司是否使用通过 IGES、STEP 或VDA 导入的实体或曲面数据？

Part B Reading Materials

Shell (Shape Tab)

Use this command to create an offset shell feature (Fig.3.2) from a solid. The offset will apply to all faces of the solid except those that have Face offset attributes. You can also specify which faces of the shell if any should remain open.

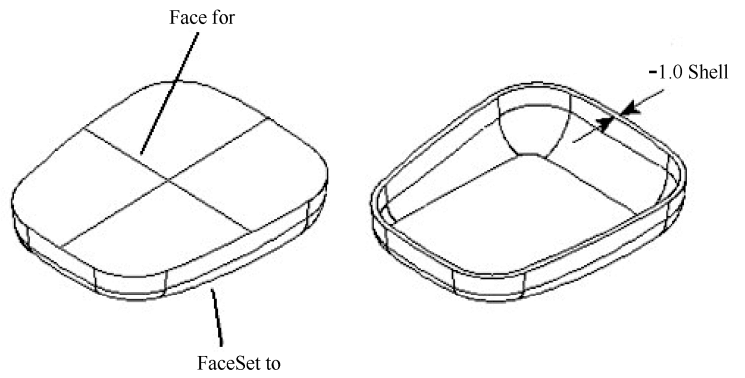


Fig.3.2 Shell

1) Shell Solid

Shells can also be applied to open shape features.

A negative value for thickness creates the wall thickness inside the original shape (on the pink side).

Firstly, Optional inputs.

Secondly, Create side faces.

Use this command option to control the creation of side faces during the Offset Face and Shell commands. When you select side faces the shape will be reconnected to form a closed solid (recommended).

Finally, Intersections.

Use this command option to check for self-intersecting faces during the Shell commands. This option will also detect disappearing fillet faces, fillet corners, and chamfers. These checks require some time to complete depending on part complexity.

Part C Exercises

Please translate the English annotation in the Fig.3.3 below.

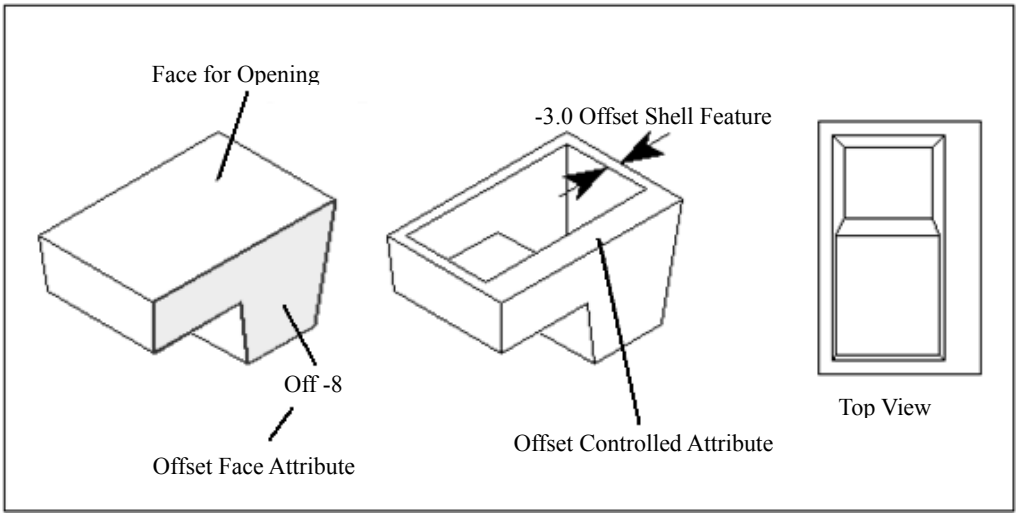


Fig.3.3 Drawing Process