

# Part One

# INTRODUCTION

## Module 1A

### The Basic Concepts & Application of Descriptive Geometry

#### *Section 1*

#### *Basic Concepts of Descriptive Geometry*

**Descriptive geometry** is the science of graphical representation and solution of spatial relationships of points, lines, and planes by means of projections, it makes analysis of spatial relationship (“geometry,” or “location measurement”) of the three basic geometric elements (points, lines, and planes) in any three-dimensional objects (called "solids"). Engineers and designers use the knowledge of descriptive geometry to project three-dimensional figures onto a two-dimensional plane of paper in such a way that allows the determination of lengths, angles, shapes and other information.

**Please use the following props, and refer to the drawings on the next page to try out the concepts of descriptive geometry:**

- Two pencils (representing lines);
- Three candies (representing points); and
- Two pieces of cardboard (representing planes);

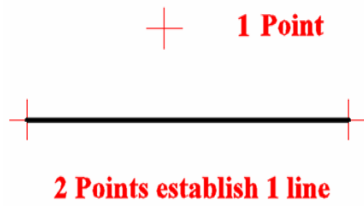


Figure 1-1A: Point and line (left); AutoCAD Point Styles dialog window (right).

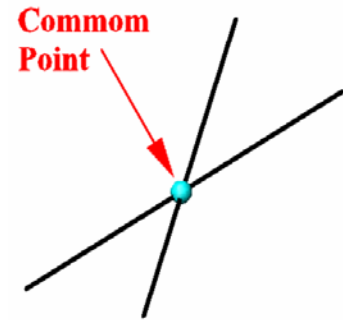
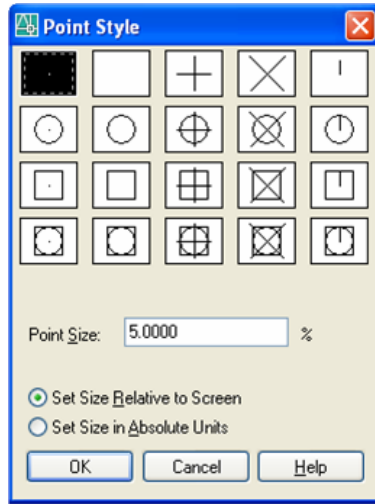
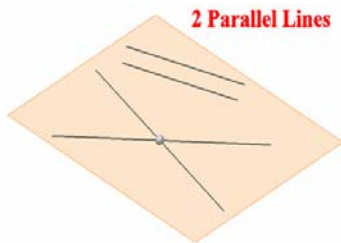


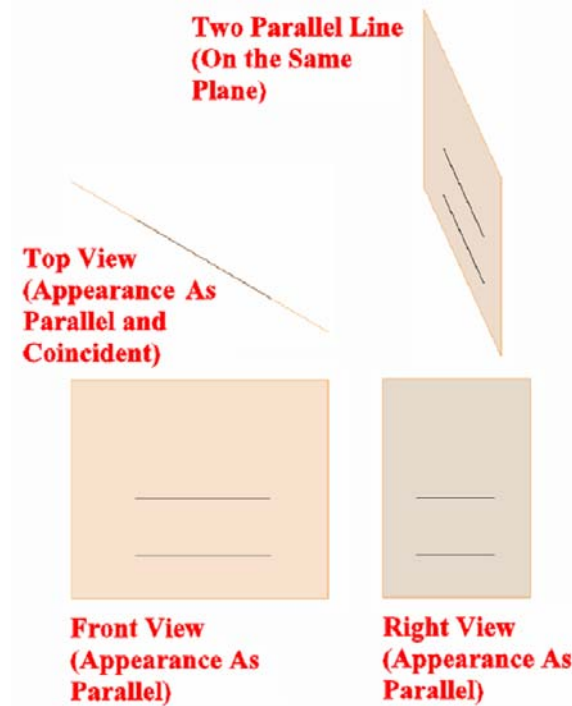
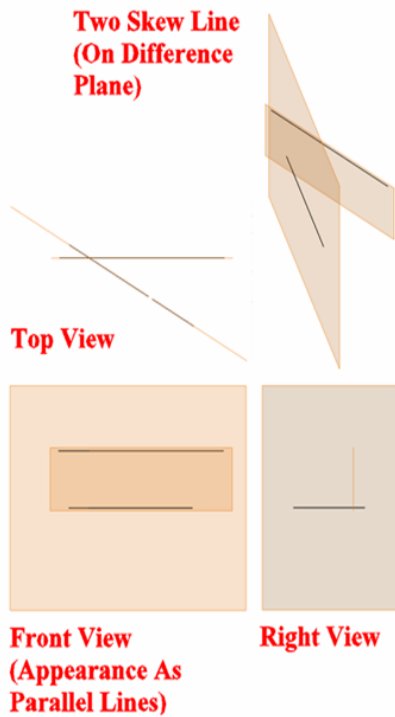
Figure 1-1B: Two intersecting lines sharing one common point.



2 Intersecting Lines

Figure 1-1C: Two parallel straight lines (top) and two intersecting lines (bottom) on the same plane.

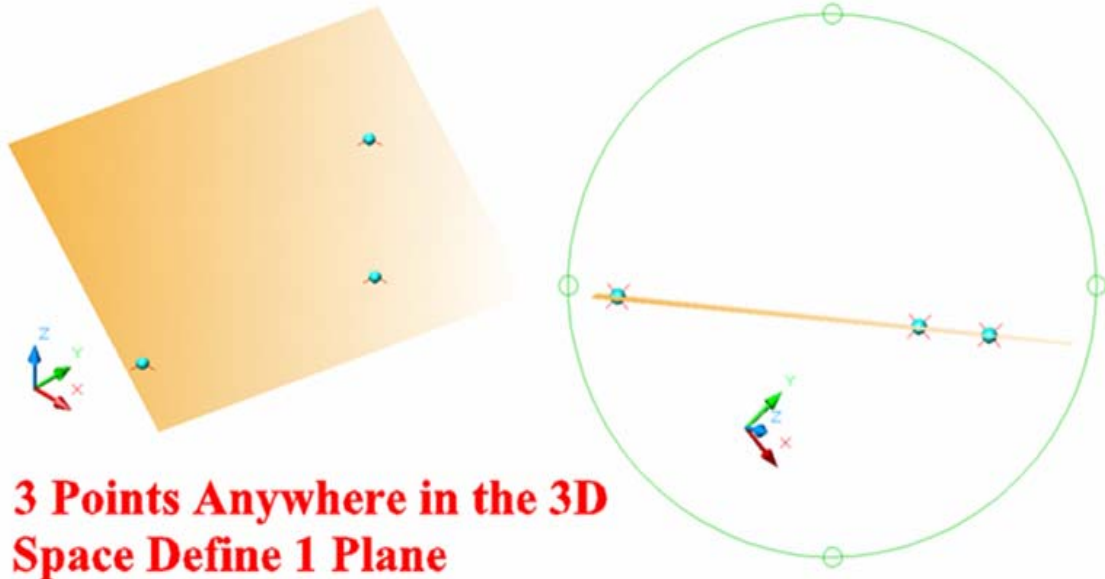
Figure 1-1D (Bottom): Two skew lines on different planes, appearing "parallel" in one view (bottom left); and two parallel lines on the same plane, appearing in at least two principal views (bottom right).



***Basic concepts used in descriptive geometry:***

**Points, lines, planes (surfaces) and solids**

- **Point**: A point is a location in space with no dimensions other than its spatial location. Two points establish a line or line segment; a point has location coordinates but no dimensions; a line has length as its dimension; in traditional drafting practice, a point is usually represented by a + or × symbol; in AutoCAD, points are created with the Point tool and many styled symbols can be used by going to the **Format→Point Style** menu (*Figure 11-1A*).
- **Intersecting lines**: Two line segments having one point in common are intersecting lines. Example: line 1-2 (a line that is defined both two endpoints, point 1 and point 2) and line 2-3 (a line that is defined both two endpoints, point 2 and point 3) are intersecting lines, which share the point 2 as the point in common; notice that points 1, 2, and 3 can have any sets of x, y, z-coordinates in three-dimensional space (*Figure 1-1B*).
- **Lines on the same plane**: Two straight lines on the same plane must intersect unless the lines are parallel (*Figure 1-1C*).
- **Skew lines**: Nonintersecting and nonparallel lines are called skew lines; notice that two lines may appear to be parallel in a particular view, but as long as they do not lie on the same plane, they are actually skew lines (*Figure 1-1D*).
- **Plane**: A plane is a “flat” 2D geometric entity defined by three points, or by one line (which is defined by two points) plus one point, anywhere in the three-dimensional space (*Figure 1-1E*); theoretically, a plane can be considered as extending to infinity in all directions without borders.
- **Surface**: A surface is a 2D geometric entity with a finite area but without thickness or volume, and it can be represented by: (a). intersecting lines; (b). parallel lines; (c). three points not in a straight line; (d). a line and a point not on the line; a surface is the tangible part of an object; it can be flat, curved (single-curved or double-curved) or warped (*Figure 1-1F*).



### 3 Points Anywhere in the 3D Space Define 1 Plane

Figure 1-1E: A plane defined by three points.

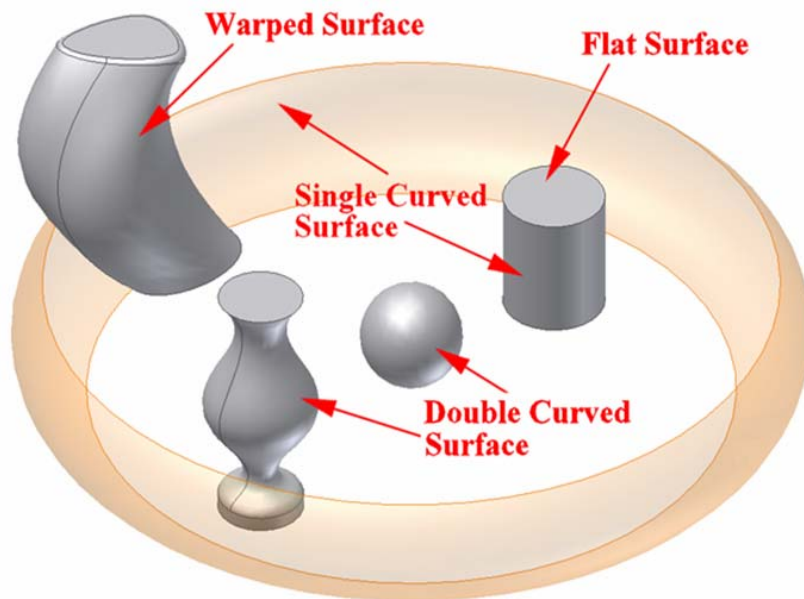


Figure 1-1F: Flat surface, single- and double-curved surfaces, and warped surface.

- **Piercing Points:** if a straight line is not parallel to a plane, it must intersect that plane in a single point called a piercing point (*Figure 1-1G*).

- **Ruled Surface**: A ruled surface is any surface generated by a moving straight line, and can be a plane, a single-curved surface, or a warped surface. A plane can be created by moving a straight line (the “generatrix”) along a straight line path (called the “**directrix**”). A non-planar, curved or warped surface can be created by moving a straight line or curved line “generatrix” in the three-dimensional space along a path that is not straight. The straight line that generates the ruled surface is called the **generatrix**; and any position of the generatrix on the surface that is created by the motion of the straight line “generatrix” is called an **element**. In AutoCAD, a ruled surface can be created, for example, by first using 2D drawing tools (such as **Line** and **Arc**) from the **Draw** tool bar to create two lines, straight or curved, on different planes created with various options of the **UCS** tool; and then by using the **Ruled Surface** tool from the **Surface** tool bar (*Figure 1-1H*).
- **The intersection of two planes**: It is a straight line with all points common to two planes (*Figure 1-1J*).

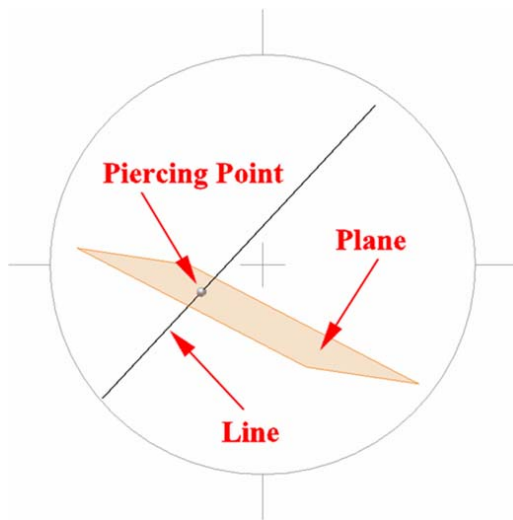


Figure 1-1G: A piercing point.

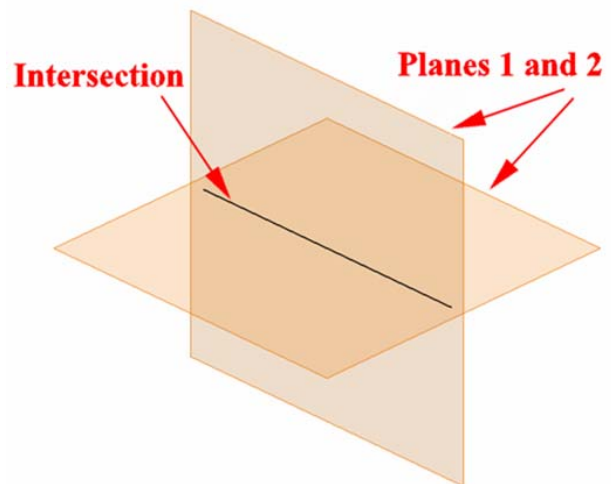


Figure 1-1J: Intersection of two planes (a straight line).

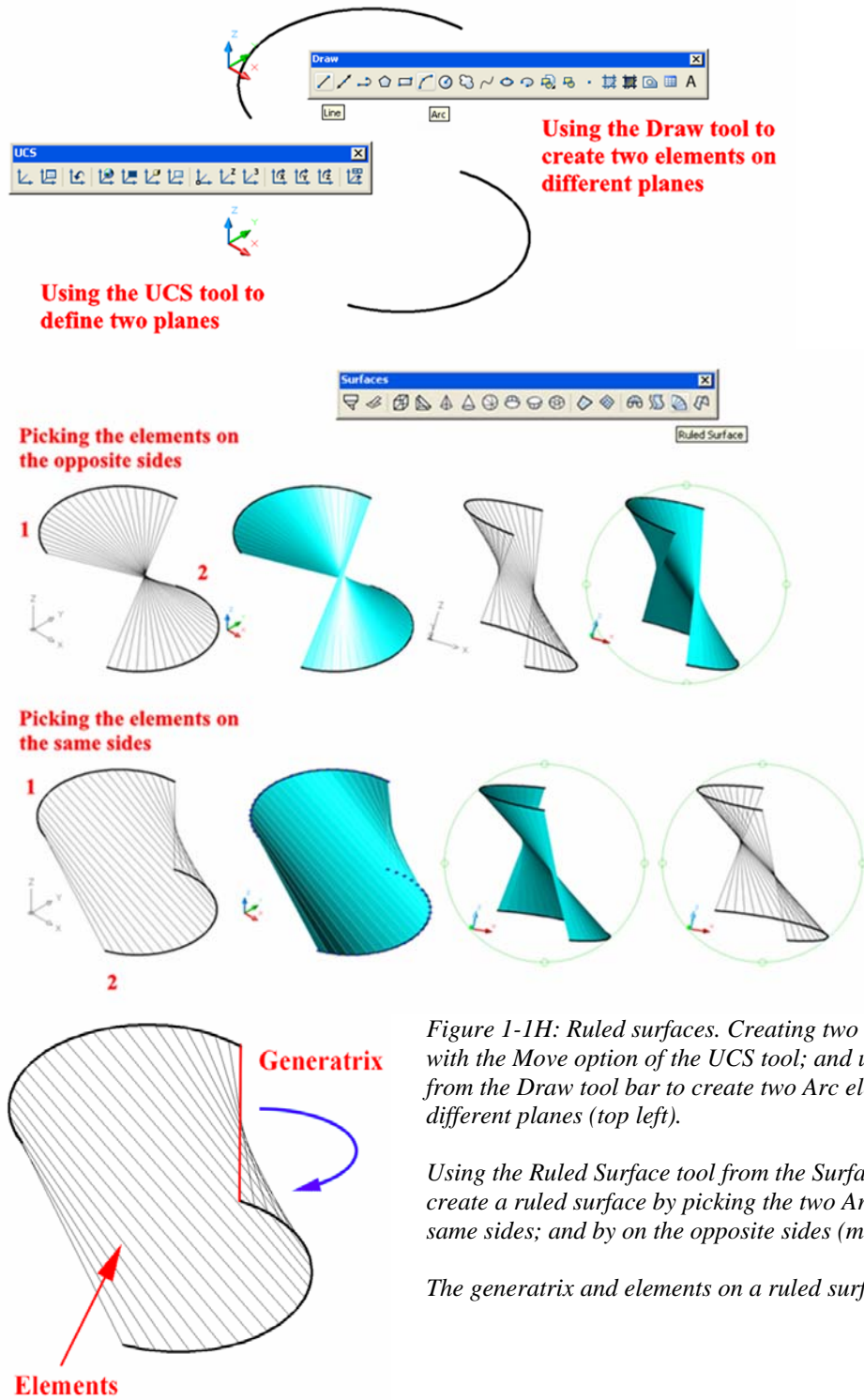
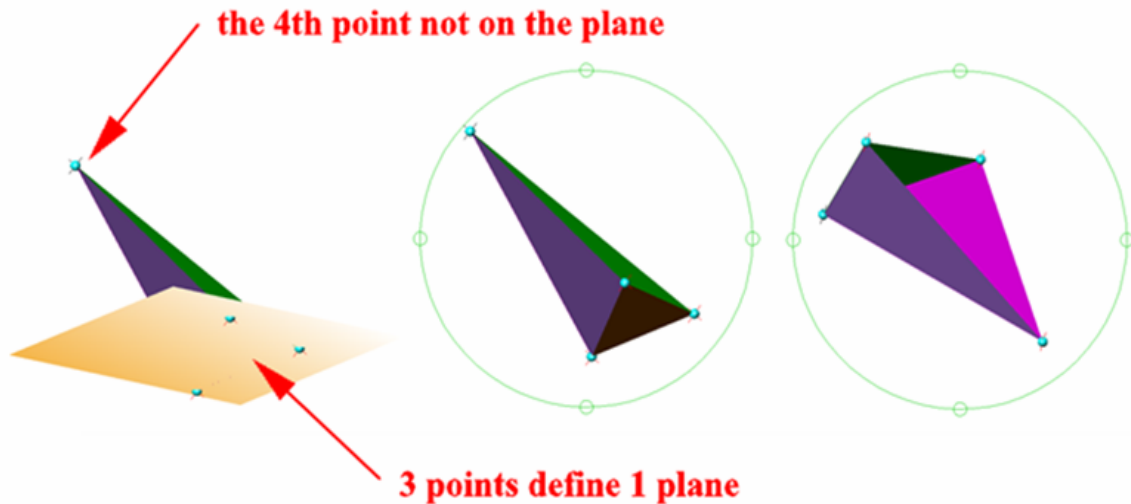


Figure 1-1H: Ruled surfaces. Creating two different planes with the Move option of the UCS tool; and using the Arc tool from the Draw tool bar to create two Arc elements on different planes (top left).

Using the Ruled Surface tool from the Surface tool bar to create a ruled surface by picking the two Arc elements on the same sides; and by on the opposite sides (middle).

The generatrix and elements on a ruled surface (bottom left).

- **Solid:** A solid can be defined by four points in the three-dimensional space; among these four points, three points defined a plane, and the fourth point does not lie on the plane defined by the other three points. Solids are bounded by surfaces (flat, curved or warped); and in practice, the term solid refers to any 3D closed form, which may be either solid or hollow (*Figure 1-1K*).



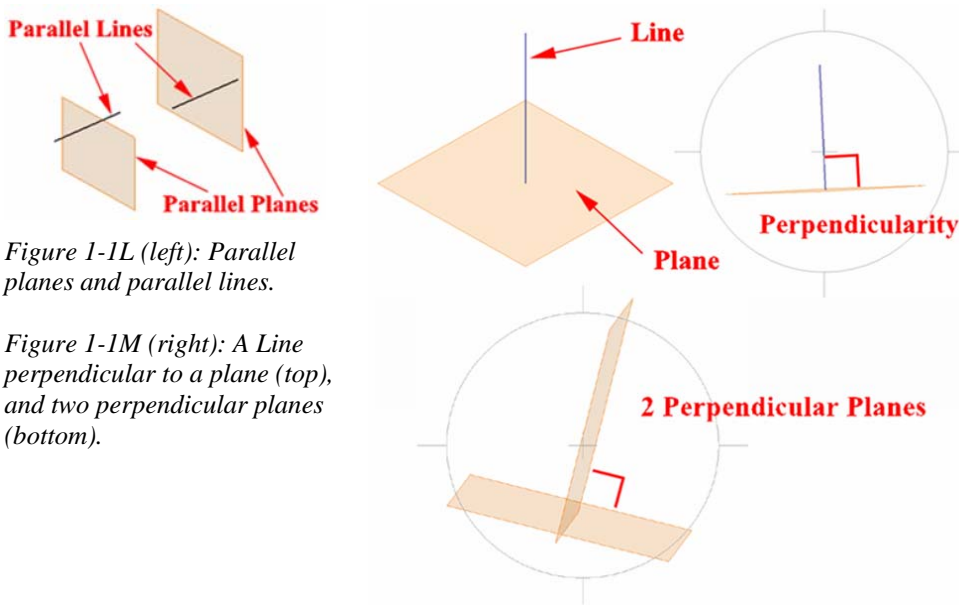
*Figure 1-1K: A solid defined by four points, and bounded by flat surfaces.*

### **Parallelism, perpendicularity and angularity**

- **Parallelism:** In any view, parallel lines in space will be projected as parallel lines, or as points when they are perpendicular to the viewing plane, or coincide as a single line when they are parallel to the viewing plane and lined up one behind the other. **Parallel planes:** If a pair of intersecting lines in one plane is parallel to a pair of intersecting lines in a second plane, then the planes are parallel. **Line parallel to plane:** A line is parallel to a plane if it is parallel to a line in the plane; a plane is parallel to a line if it contains a line that is parallel to the given line (*Figure 1-1L*).
- **Perpendicularity:** A  $90^\circ$  angle appears in true size in any view showing one leg true length, provided that the other leg does not appear as a point in the same view. **A Line perpendicular to a plane:** A line perpendicular to a plane is perpendicular to all lines in the plane. For a simpler statement, we can say that: A line perpendicular to a plane is perpendicular to at least two nonparallel lines in the plane. This line will also appear perpendicular and in true length to the edge view of the plane. **Perpendicular planes:** If a line is

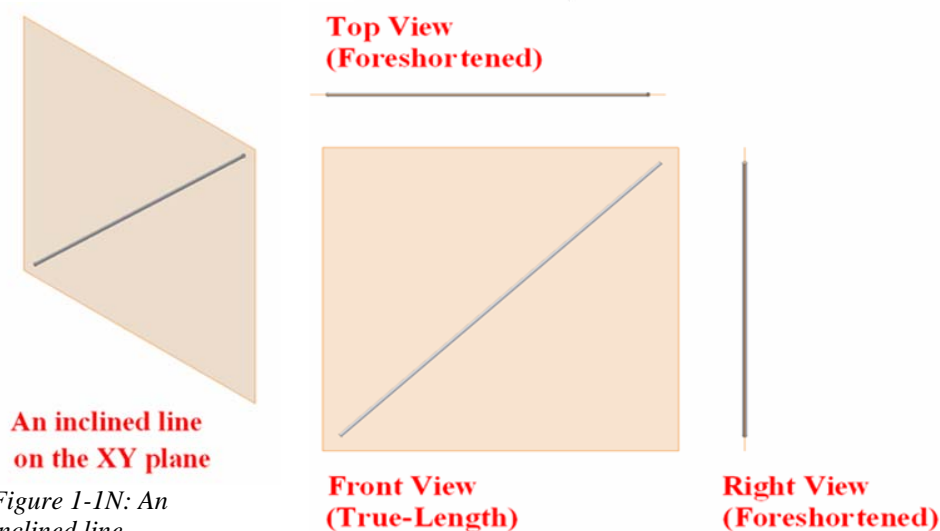
perpendicular to a given plane, any plane containing that line is perpendicular to the given plane (*Figure 1-1M*).

- **Inclined line:** An inclined line is also called an **angled line**; and it appears true length on the plane to which it is parallel. Inclined lines are often classified as frontal (corresponding to front view), horizontal (corresponding to top view) or profile (corresponding to side view), see *Figure 1-1N*.



*Figure 1-1L (left): Parallel planes and parallel lines.*

*Figure 1-1M (right): A Line perpendicular to a plane (top), and two perpendicular planes (bottom).*



*Figure 1-1N: An inclined line.*

*Figure 1-1P: True-length and foreshortened views of a line.*



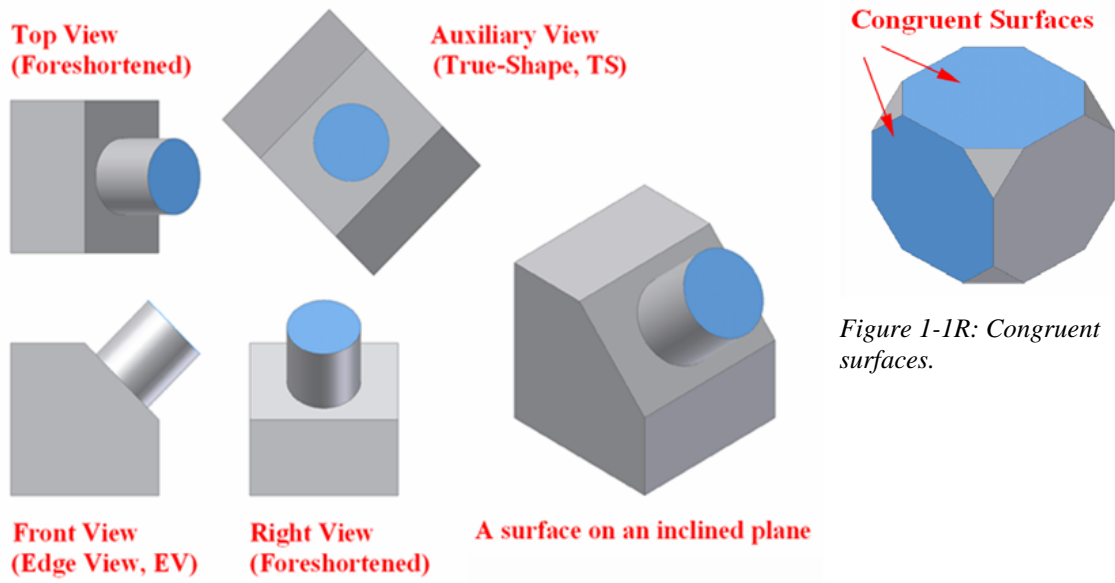


Figure 1-1Q: True-shape view of a surface.

### **Direction of sight, true-length, true-shape and congruity**

- **True-length**: The true-length (abbreviated as TL) view of an inclined line is always in an inclined position, and the foreshortened views are always in either vertical or horizontal positions (*Figure 1-1P*).
- **True-shape**: The true-shape view of a surface is always on the view where the surface is perpendicular to the line of sight (*Figure 1-1Q*).
- **Congruence**: Congruence or congruity: two neighboring faces with similar shapes are congruent or congruous (*Figure 1-1R*).
- **Line of sight and views**: If a direction of sight (also called line of sight and abbreviated as LOS) is parallel to a true-length view of a line, then that line will appear as a point (point view or PV) in the resulting view. If a direction of sight is perpendicular to a true-length (TL) view of a line, then that line will appear as a true-length line in the resulting view. If a direction of sight is neither parallel nor perpendicular to a true-length view of a line, then that line will appear as a foreshortened line in the resulting view (*Figure 1-1S*).

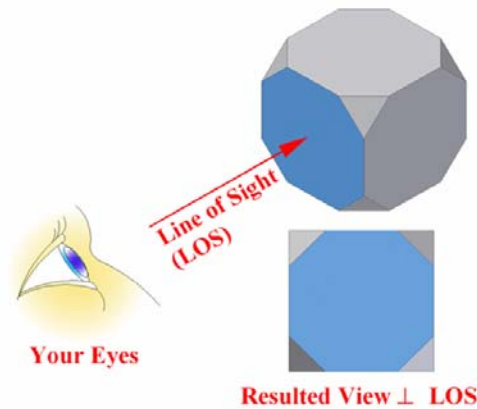
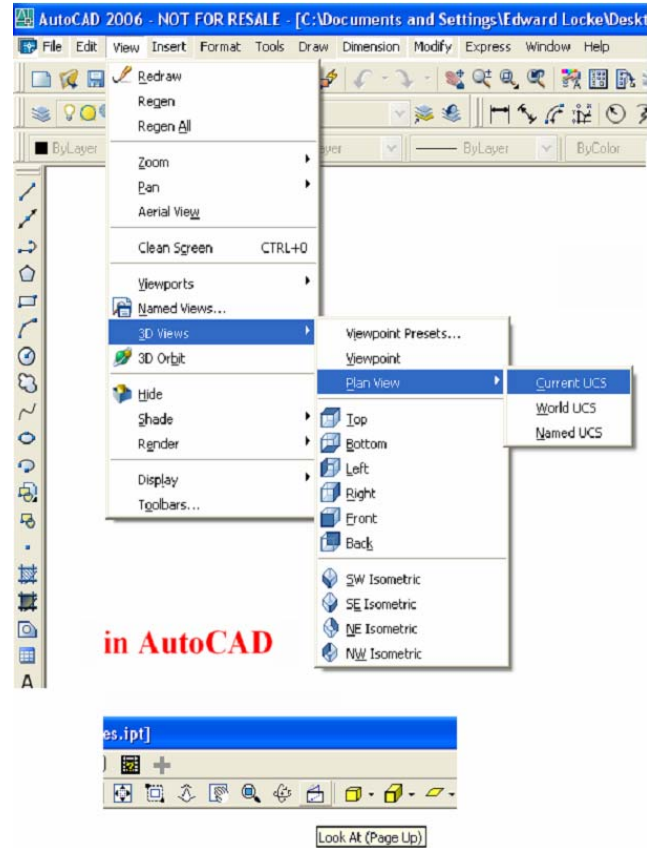


Figure 1-1S: Line of sight (Left). In AutoCAD, if the XY Plane of the UCS is on the surface of any 3D model to be perpendicular to the line of sight (LOS), then the true-shape (TS) view can be obtained on the screen by going to the View → 3D Views → Plan View → Current UCS menu (top right). In Inventor, the true-shape (TS) view of any surface can be obtained on the screen by click-selecting the surface and then the Look At tool button on the Standard tool bar (bottom right).



in Inventor

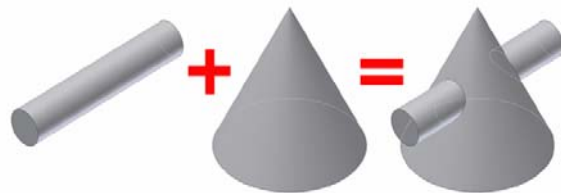


Figure 1-1T: Intersection of two solids.



Figure 1-1U: Folded model (left) and development (right).

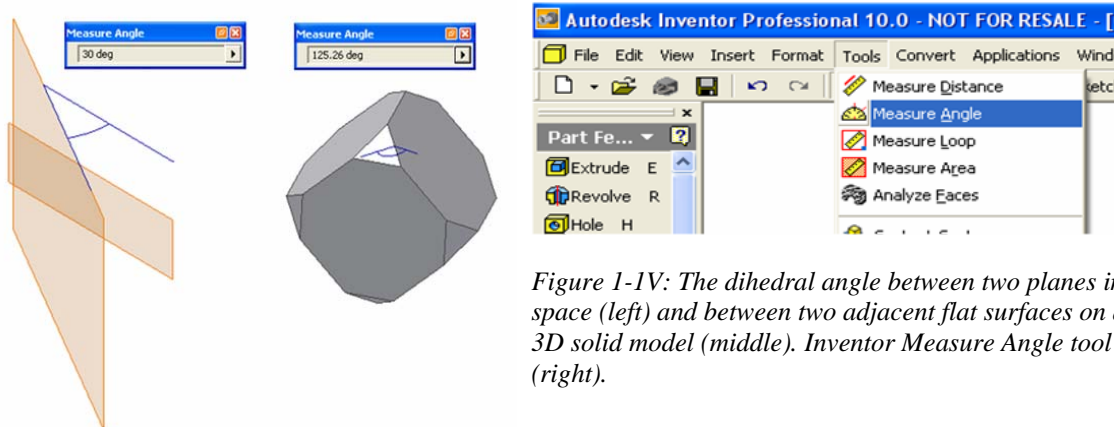
### Intersection and development

- **Intersection:** An “intersection” is a point or a line that is formed when two (or more) geometric forms (such as two planes or a prism and a cylinder) meet or pass through each other (Figure 1-1T).

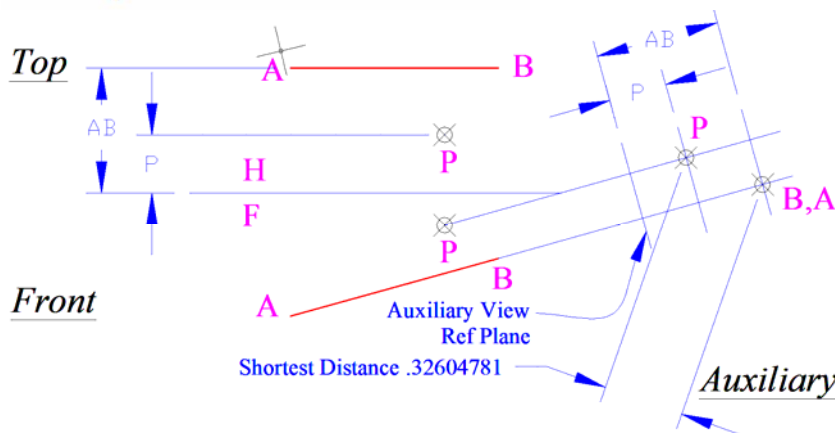
- **Development:** The term “development” in engineering descriptive geometry refers to an unfolded (or “developed”), flat surface (also called “surface development,” “stretch-out,” “flat pattern” or “templates”) of a three-dimensional object, and can be either a precise development (for any “single-curved” object) or approximate development (for any “double-curved” object, such as warped surfaces and sphere (*Figure 1-1U*)).

**Graphical mathematics**

- **Dihedral angle:** The angle between two adjacent flat surfaces on a three-dimensional solid or two planes in space. In AutoCAD, using the **Angular Dimension** tool to measure the dihedral angle on a 3D solid is a complicated process; therefore, in the subsequent learning modules, the focus will be placed on using two-dimensional orthographic projection methods to obtain the dihedral angle. In Inventor, it can be accomplished easily with the **Measure Angle** tool (*Figure 1-1V*).



*Figure 1-1V: The dihedral angle between two planes in space (left) and between two adjacent flat surfaces on a 3D solid model (middle). Inventor Measure Angle tool (right).*



*Figure 1-1W: Using two-dimensional orthographic projections to obtain the shortest distance in AutoCAD.*

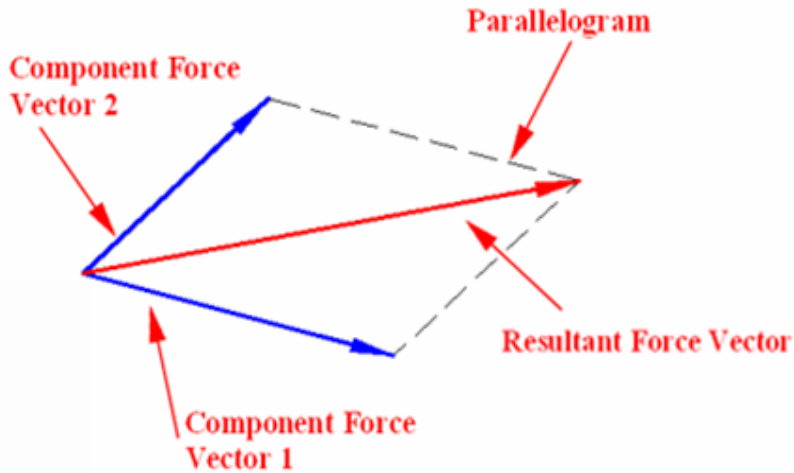


Figure 1-1X: Vector force analysis.

- **Shortest distance**: For two non-intersecting lines in the three-dimensional space, the distance defined by the shortest possible line that can be drawn perpendicular to both non-intersecting lines (*Figure 1-1W*).
- **Vector force**: The method of obtaining the resultant force of two or more component forces by drawing a parallelogram using the component force vectors as sides and by drawing the diagonal line as the resultant force vector (*Figure 1-1X*).

## ***Section 2*** ***Types of Surfaces and Solids***

The motion of a straight or curved line creates a **geometric surface**. Geometric surfaces can be used to define volumes of solids.

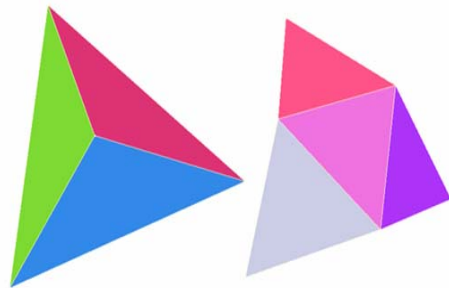
In practice, the term solid refers to any 3D closed form, which may be either solid or hollow. There are basically three types of solids:

### **1. Polyhedrons**

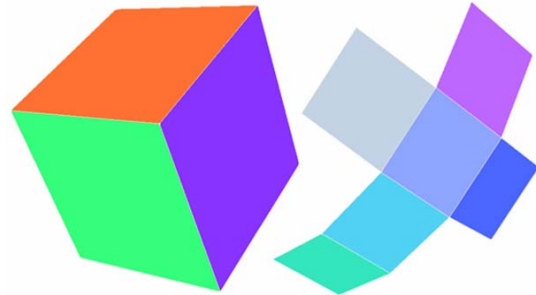
Polyhedrons are solids bounded by plane surfaces called “faces” with lines of intersection of faces called “edges.” **Plane (flat) surfaces** are found on all faces of polyhedrons, such as cube, prism, tetrahedron, pyramid, as well as on the top and/or

bottom faces of cone and cylinder. They can be precisely developed. The following Five Regular or “Plutonic Solids” have plane surfaces only:

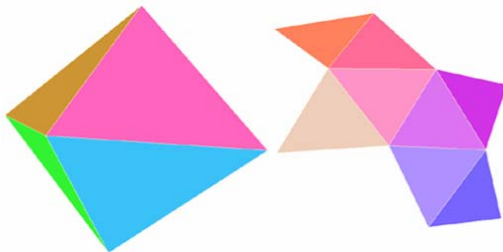
- **Tetrahedron**: with four equilateral triangular faces (*Figure 1-2A*).
- **Hexahedron (cube)**: with six square faces (*Figure 1-2B*).
- **Octahedron**: with eight equilateral triangular faces (*Figure 1-2C*).
- **Dodecahedron**: with twelve pentagonal faces (*Figure 1-2D*).
- **Icosahedron**: with twenty equilateral triangular faces (*Figure 1-2E*).



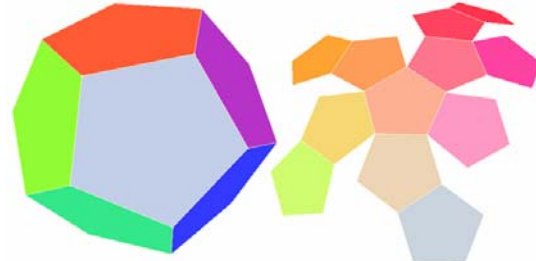
*Figure 1-2A: Tetrahedron.*



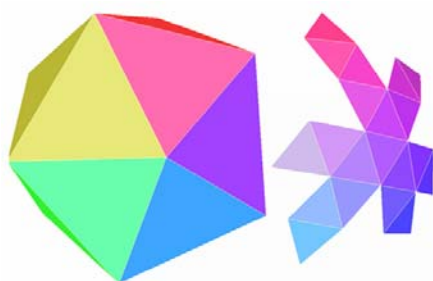
*Figure 1-2B: Hexahedron (cube).*



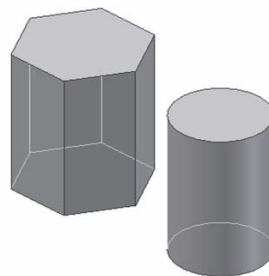
*Figure 1-2C: Octahedron.*



*Figure 1-2D: Dodecahedron.*



*Figure 1-2E: Icosahedron.*



*Figure 1-2F: Prism (left) and cylinder (right).*

## 2. Solids of revolution:

They are generated by revolving a plane figure about an axis in the plane of the figure. They include:

- **Single-curved surfaces:** They are found on the lateral surfaces of cone and cylinder. They are ruled surfaces and can be precisely developed. Two types of relationship between the generatrix and the directrix may exist on a single-curved surface: (1) **Cylinder or prism** (Figure 1-2F): a single-curved surface can be generated by a straight line generatrix one point of which moves along a plane-curve directrix, and the generatrix always remain parallel to its original position as for a cylinder; for example, in a cylinder, all elements of a cylinder are parallel to each other and to the axis of the cylinder. A plane tangent to a cylinder contains only one straight-line element of its surface, and is parallel to the remaining elements and to the axis); or (2) **Cone and pyramid** (Figure 1-2G): a single-curved surface can be generated by a straight line generatrix one point of which moves along a plane-curve directrix, and the generatrix keeps passing through a fixed point (the vertex) as for a cone. In a cone, all elements of a cone pass through its vertex. A plane tangent to a cone contains only one straight-line element to its surface, and this straight-line must pass through the vertex of the cone. **Right-circular cones:** All elements of a right-circular cone form the same angle with the base plane of the cone. A plane tangent to a right-circular cone contains one element and forms the same angle as does any element with the base plane of the cone. If two right-circular cones with the same vertex and elements of the same length intersect, the common element or elements of the two cones form the same angles with the two base planes as do the respective sets of elements.

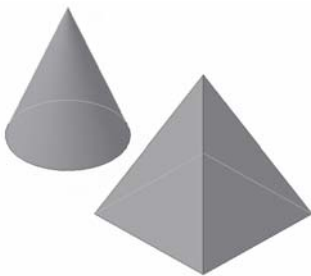


Figure 1-2G: Cone and pyramid.



Figure 1-2H: Ellipsoid, torus, and sphere.

- **Double-curved surfaces:** They are found on the surfaces of **sphere**, **oblate spheroids** or **ellipsoids**, **paraboloids**, **torus**, etc. They can be approximately developed only. They can be created by revolving a curved line about a straight line. **Sphere** is a typical example of a solid bound by a double-curved

surface, which contains no straight-line elements but only infinite number of points equidistant from the center of the sphere. A plane tangent to a sphere contains one and only one point in that surface. Double-curved surfaces can be developed only approximately by dividing them into sections and substituting for each section a developable, plane or single-curved surface (*Figure 1-2H*).

### 3. Solids bounded by warped surfaces

**Warped surface:** They include automobile hood and others, and cannot be developed by regular method; however, approximate development of a warped surface can be made by dividing the usually smooth warped surface into sections and substituting for each section a developable, plane or single-curved surface such as with a series of triangles. A warped surface is a non-developable ruled surface generated by straight lines moving in various patterns. In a warped surface, no two adjacent positions of the generatrix lie in the same plane. Warped surfaces include conoid helicoids, hyperboloid, streamlined surfaces of aircrafts and automobiles. Screw threads are examples of solids bound by warped surfaces (*Figure 1-3*). In practical fabrication, objects with warped surfaces are made by stretching, pressing, stamping, spinning, or vacuum or explosive processes, which are applied to flat sheet materials if they are pliable enough.

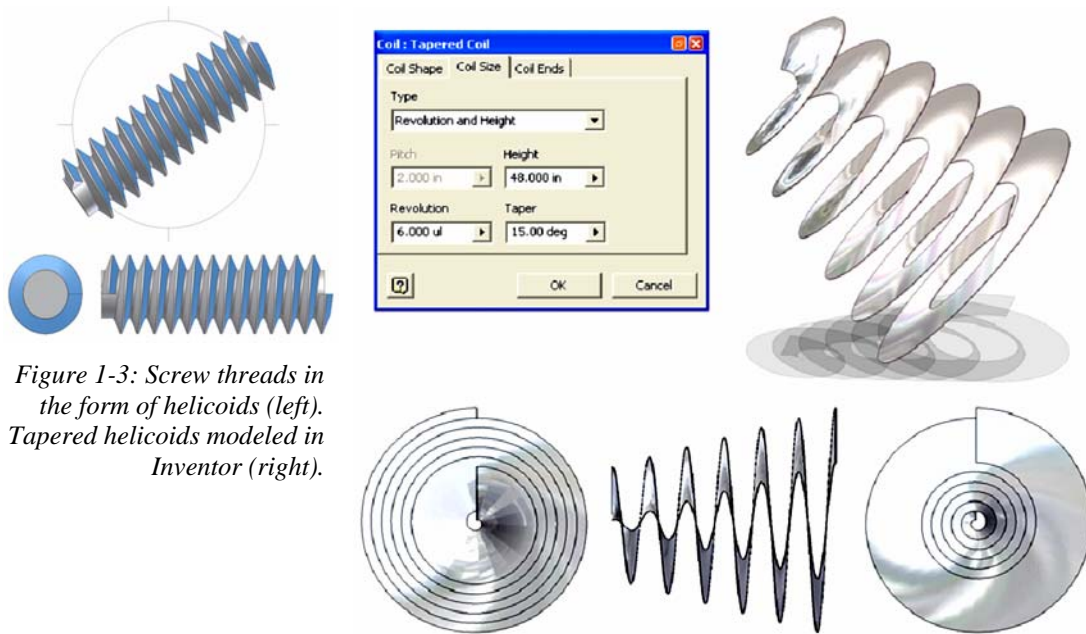


Figure 1-3: Screw threads in the form of helicoids (left). Tapered helicoids modeled in Inventor (right).

We have explored so far various concepts and terminologies used in descriptive geometry, with text and figure illustrations. To better understand these concepts and terminologies before proceeding to hands-on learning of the subjects with AutoCAD 2D tools, we can open the relevant digital files inside the *Descriptive Geometry Concepts Illustration* folder of the enclosed *Student Descriptive Geometry Learning Resource CD* (Figure 1-4A). The files have been created in either AutoCAD or Inventor. The AutoCAD files have been saved in **AutoCAD 2000/LT 2000 Drawing [\*].dwg** format; and be opened with any version of AutoCAD 2000/LT 2000 or later. The Inventor files are created in Inventor 10 version. It might be more convenient to launch the AutoCAD and Inventor programs first and to open the files from the programs' **File→Open** menu, rather than double-click the files' icon to open them.

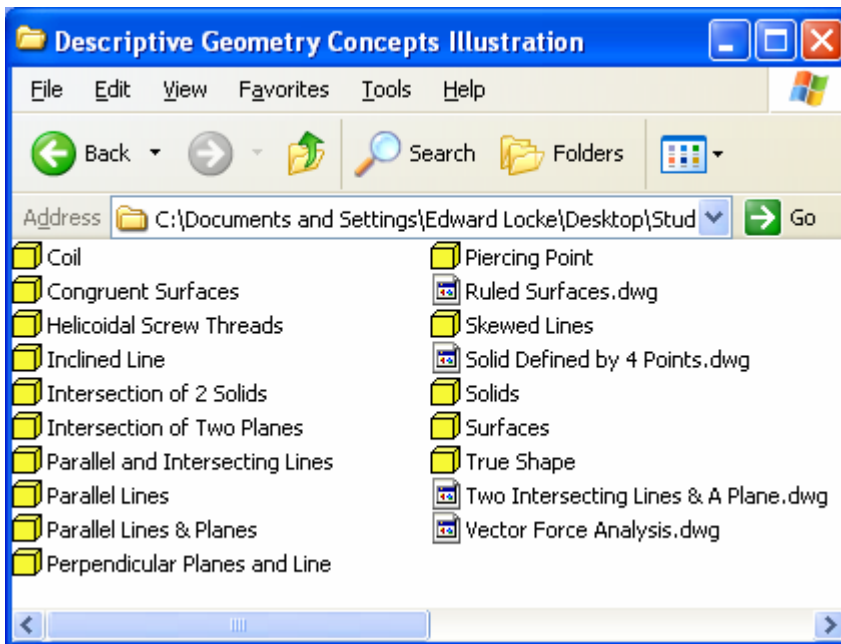


Figure 1-4: The digital files.

Tools that help to move the models in these files are explained in *Figure 1-4B* (for AutoCAD) and *Figure 1-4C* (for Inventor).

- In AutoCAD, various options in the **View→3D Views** menu can help to view the model from different view perspectives. The **3D Orbit** tool (from the **3D Orbit** tool bar) can be used to rotate the model; to use this tool, drag on the handles at the four quadrants of the “circle” to rotate around X or Y axes; drag inside the “circle” to freely rotate the model in 3D space; and drag outside the “circle” to rotate circularly on the XY plane.



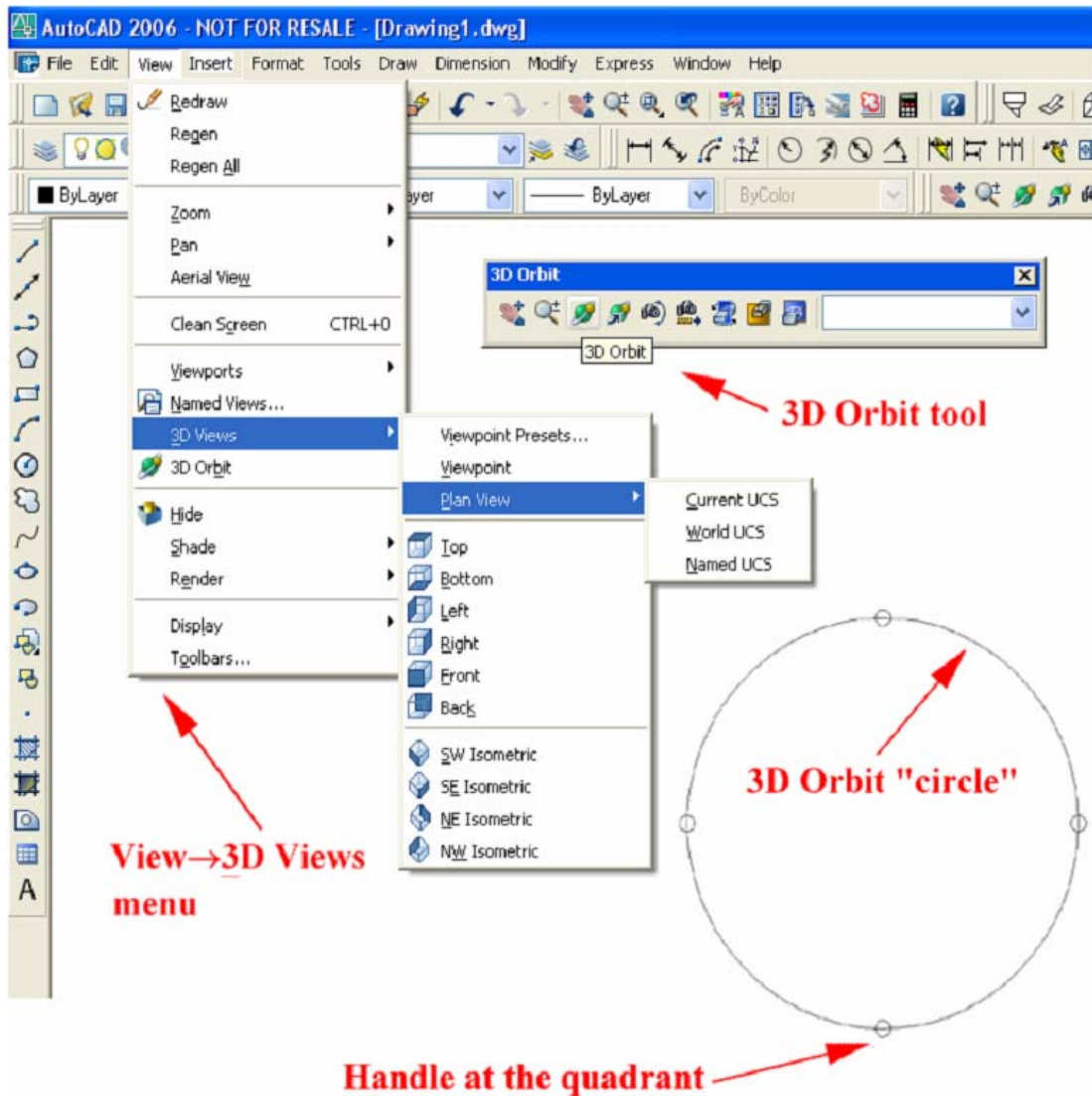


Figure 1-4B: AutoCAD tools. The View → 3D Views menu for viewing the model from different view perspectives (left). The 3D Orbit tool for rotating the model (right).

- In Inventor, the **View → Isometric** menu helps to view the model in isometric view. The Rotate tool (from the **Inventor Standard** tool bar) helps to rotate the model; this tool contains both **Free Rotate [SPACE]** (with a “circle” symbol) and **Common View [SPACE]** options (with a “cube” symbol), which can be changed by right-clicking for the shortcut menu and choosing the appropriate option; to use the **Free Rotate [SPACE]** option of this tool, drag on the handles at the four quadrants of the “circle” to rotate around X, Y or Z axes; drag inside the “circle” to freely rotate the model in 3D space; and

drag outside the “circle” to rotate circularly on the XY plane (right); to use the **Common View [SPACE]** option of this tool, click on an arrow at any corner of the “cube” to rotate the model to an isometric view; click on an arrow at any edge of the “cube” to rotate the model to an orthographic view. To use the **Look At** tool (from the **Inventor Standard** tool bar) to rotate the any flat surface of a model or any **Work Plane** feature to the normal view (which is perpendicular to the lone of sight or LOS), click select the tool’s icon and the surface or **Work Plane** (the order of selection does not matter).

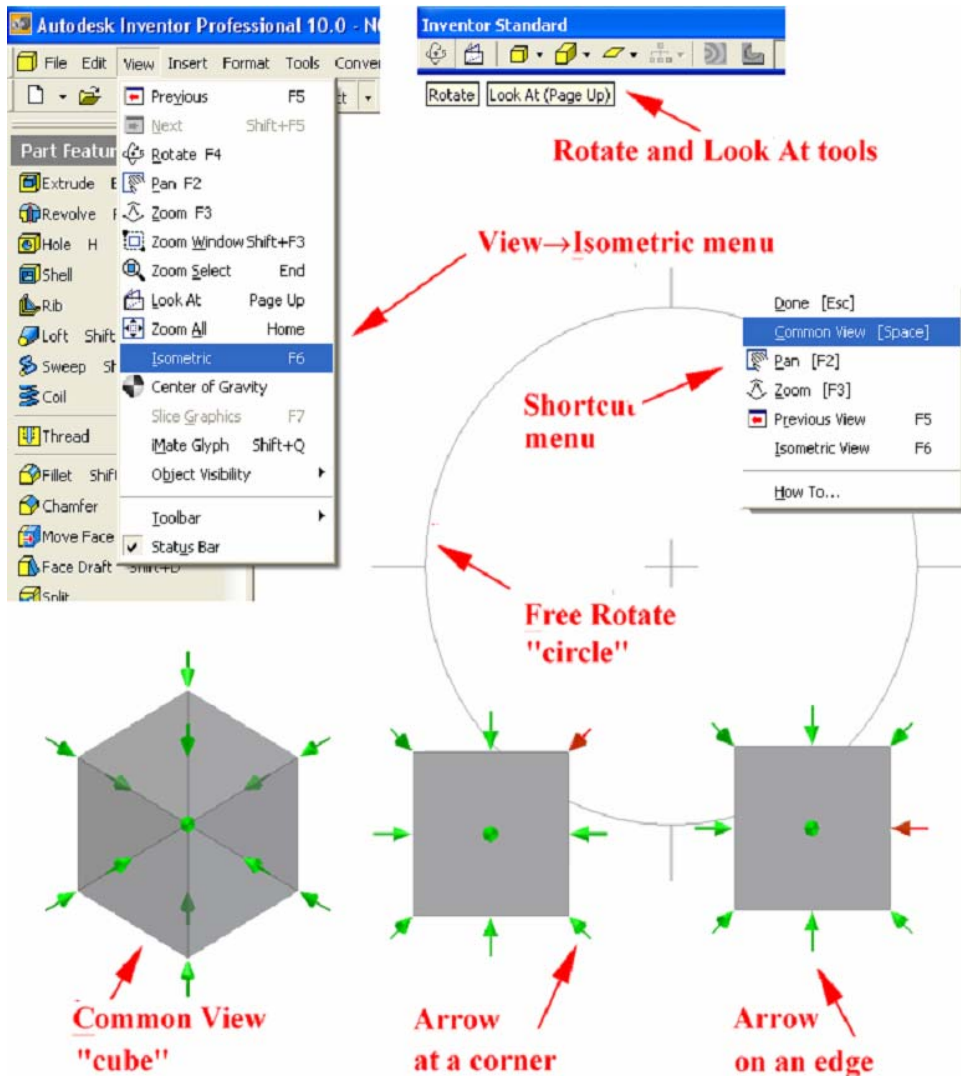


Figure 1-4C: Inventor tools. The View → Isometric menu for viewing the model in isometric view (left). The Rotate and Look AT tools for rotating the model in 3D space (top right). The shortcut menu for choosing the Common View option (middle right). The “cube” symbol for the Common View option (bottom)

### ***Section 3***

## ***The Application of Descriptive Geometry in Sheet Metal Trade with Intersection and Development***

### ***Application of intersection and development in sheet metal products***

The knowledge and skills related to intersection and development of solids can be applied to the following trades:

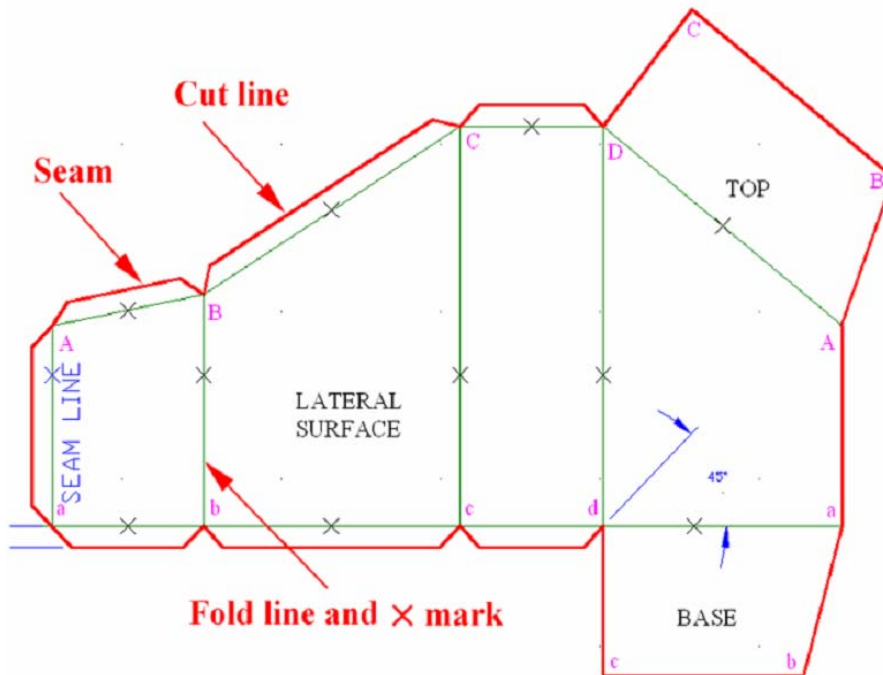
- Aerospace, ship construction and automobile industries: The knowledge of intersection and development is used to determine various shapes of instrument panels, window opening, and body sections (such as the skin of an aircraft).
- Architectural design: The intersection of surfaces is needed for the design and specification of building fixtures (such as a pipe that goes through a wall).
- HVAC (heating, ventilation, air conditional and cooling): Sheet metal products, used primarily in ductwork all use the knowledge of intersection and development. The design and construction of boilers, smokestack breaching, storage tanks, office furniture, cabinets, and of refineries in the oil and chemical industries need the knowledge of intersections.
- Packaging: The knowledge of intersection and development of patterns can be used in the design of containers (electrical equipment housing, computer case, milk carton, cardboard cereal box, and other packaging products).

### ***The Process of Design and Fabrication of Sheet Metal Products***

There are basically three steps in the process of design and fabrication of sheet metal products:

1. Pattern development: The engineer or designer first draws a development pattern together with a pictorial drawing (usually isometric) of the finished product. Development pattern can be either an **inside** or an **outside pattern**. An **inside pattern** shows the inside surfaces of the object, and is traditionally the preferred method because the fold lines marked on the surface will not be seen when the object is folded, and because the folding equipments are usually designed to bend the patterns inward. An **outside pattern** shows the outside surfaces. Traditionally, designers indicate on the drawing whether the

pattern is inside or outside so as to avoid any chance of misunderstanding. A pattern development should include cutting line (lines along which the flat pattern should be cut) and fold line (lines that show the location where the development is to be folded). Fold line can be indicated by a × mark across them (*Figure 1-5A*).



*Figure 1-5A: Cut line, fold line with × mark, and seam.*

2. **Pattern cutting:** The pattern is cut from flat sheet of material, such as sheet metal (tin, steel, copper, brass, aluminum, etc), paper, cardboard, corrugated boards, plastics, films, wood, fiberboard, fabrics.
3. **Shape-forming and assembling:** The cut pattern is folded or rolled into desired 3D shapes and fastened (or glued, or welded) together with specially designed machines.

Students who want to learn more about sheet metal fabrication may check out related books (such as those listed in Section 2.2.2.1 *Textbooks on sheet-metal trade of Chapter 2 Review of The Literature* of this textbook project, in local technical libraries; or refer to the following technical books (available at technical bookstore such as OPAMP Technical Books (1033 N. Sycamore Ave., West Hollywood, California; telephone number: 323-464-4322):

- *Techniques of Pressworking Sheet Metal An Engineering Approach To Die Design 2nd Edition*: published by Prentice Hall, ISBN 0-13-90096-6.
- *Sheet Metal*: written by Leo A. Meyer, published by American Technical Publishers Inc., Homewood, Illinois 60430, ISBN 0-8269-1907-3.
- *Mechanics of Sheet Metal Forming*: written by Z. Marciniak, J. L. Duncan, S. J. Hu, published by Butterworth-Heinemann, An Imprint of Elsevier Science, Linacre House, Jordan Hill, Oxford OX2 8DP, 225 Wildwood Avenue, Woburn, MA 01801-2041, www.bh.com; ISBN 0-7506-5300-0,.

***Important principle to remember:***

For effective design and fabrications of sheet metal products, which shall save labor and cost in fabrication and make the products more functional, the following principles are helpful:

1. **Feasibility**: In the drafting of surface development, each plane must be drawn in a correct position next to another plane, or to make it simple, two adjacent planes on the 3D objects need to be drawn on the surface pattern in such a way that when folded up, they will be adjacent again.
2. **Cost effectiveness**: In order to reduce the amount of work needed in welding, in applying adhesives, or in using rivets, patterns should be developed in such a way that the ends to be joined are the shortest possible. The drafter also needs to take the stock sizes into consideration so as to save materials and labor.
3. **Aesthetics**: As much as possible, the patterns should be developed in such a way that on the finished products, the joints are in the most invisible places.

***A good way to check for errors in the drafting of development:***

Cut a scaled-down paper pattern and secure the joints with tape to see if the development works!

***Some real world design parameters***

In this collection of learning modules for the solution of descriptive geometry problems using the 2D tools of AutoCAD, the focus of our study is on the application of

orthographic projection methods. In the real world of design and fabrication, however, some technical parameters are taken into consideration, although they shall not be a part of our study in the Modules. These technical parameters include but are not limited to the following:

- **Bend allowances**: In the real world application of development, **bend allowances** or **thickness allowance**: calculation for bending and stretching for sheet metals, as well as **thickness allowance** for paper materials, are needed depending on the material used. However, this collection of Modules is primarily concerned with the 2D orthographic projection methods related to intersection and development of patterns, technical parameters such as **bend allowances** and **thickness allowance** will be ignored in most of the Modules. Parametric CAD programs such as Autodesk Inventor (which is explored in this textbook project), CATIA, SolidEdge, SolidWorks, Pro-Engineer, etc., can generate flat patterns from the 3D folded part with calculation for these parameters built in.
- **Seams and laps**: Extra material called “seams and laps” is added to the cut edge of pattern (the shortest edge is preferred) for the fabrication of the 3D folded parts. Throughout this collection of Modules, students are encouraged to use heavy card stock paper (110 lb) to build models of their projects.
- **Wiring**: A way of strengthening open ends of sheet metal material by enclosing a wire in the edge; a band of material with a width equal to 2.5 times the diameter of the wire is added to the pattern. This parameter shall be ignored in most of the Modules.
- **Hemming**: a way of stiffening the edges sheet-metal parts with bended and interlocked extra material on both end edges; its includes single and double-hemming (*Figure 1-5B*). This parameter shall be ignored in most of the Modules.
- **Seaming**: similar to hemming and includes soldering on lap seams, flat lock seams, grooved seams and other types of seams (*Figure 1-5C*). This parameter shall be ignored in most of the Modules.

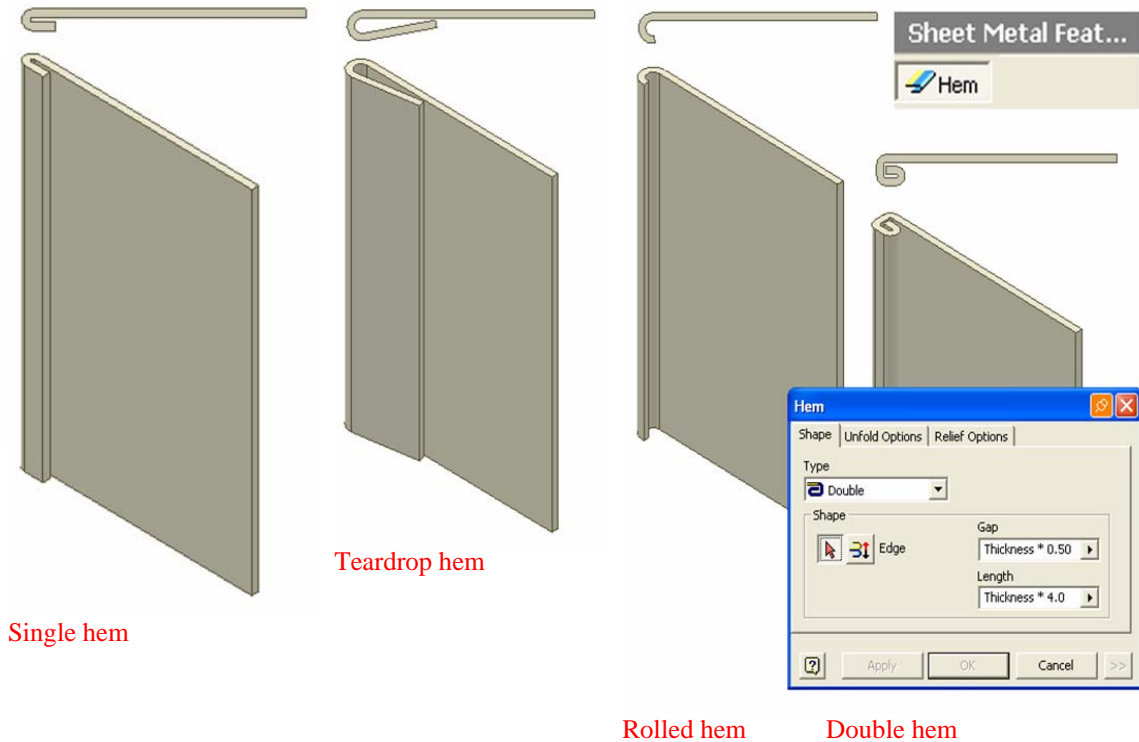


Figure 1-5B: Various types of hems (3D models created in Autodesk Inventor).

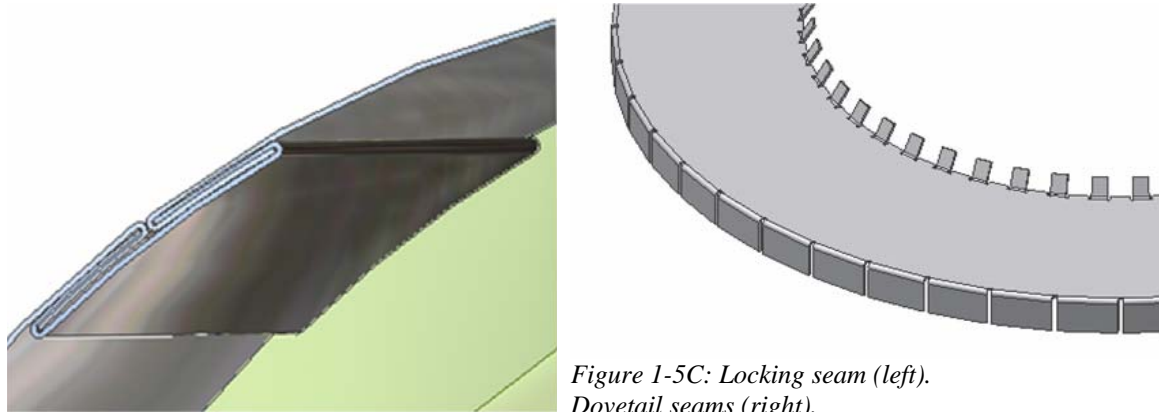


Figure 1-5C: Locking seam (left).  
Dovetail seams (right).

The creation of packages needs both engineering and artistic skills because packages need to protect the goods and also look attractive for sales appeal. Some packages are used for a long time while others are used briefly and recycled. Packages need to be designed in such a way that they can be mass-produced easily and at a reasonable cost. Almost all packages are printed. Most packages need glue on their tabs while others are designed in a way that no glue is needed.

### *Various types of development*

Developments can be classified under the following three categories:

- **Parallel line development**: used for objects having parallel elements, such as prisms and cylinders.
- **Radial line development**: used for objects having elements "radiating" from a vertex point, such as pyramids and cones.
- **Triangulation development**: used for transition pieces by breaking a surface that can not be developed by the methods of **Parallel line development** and **Radial line development**, into a series of triangles.

Details of the above three methods will be covered in the subsequent Modules.

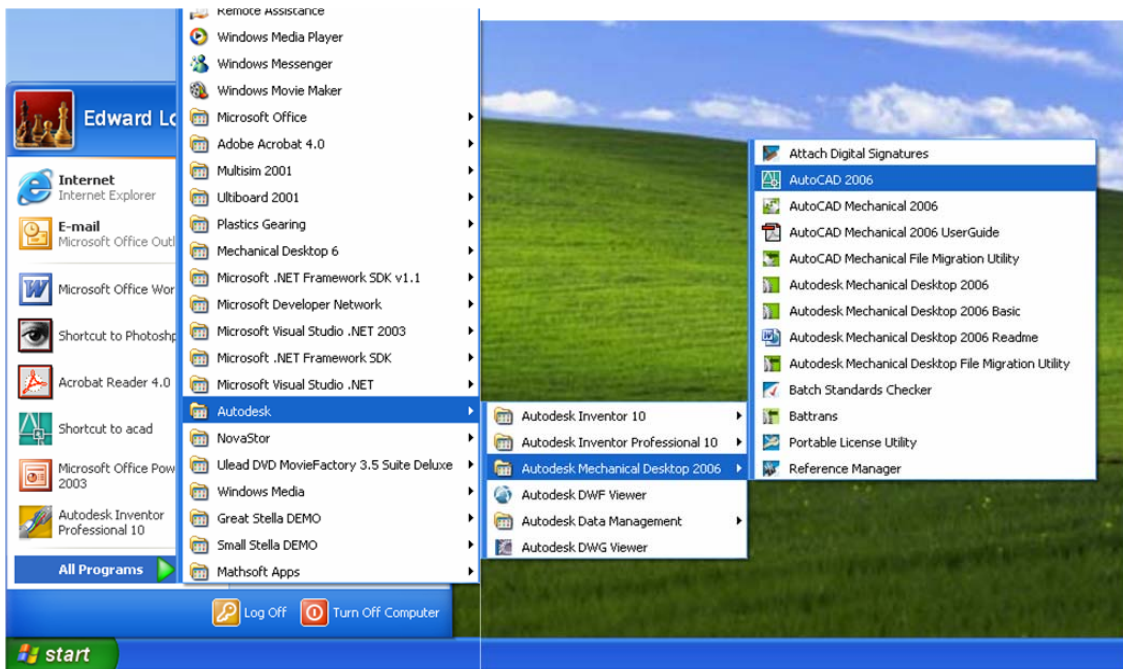


Figure 1-5D: Launching AutoCAD 2006 from the Start menu.



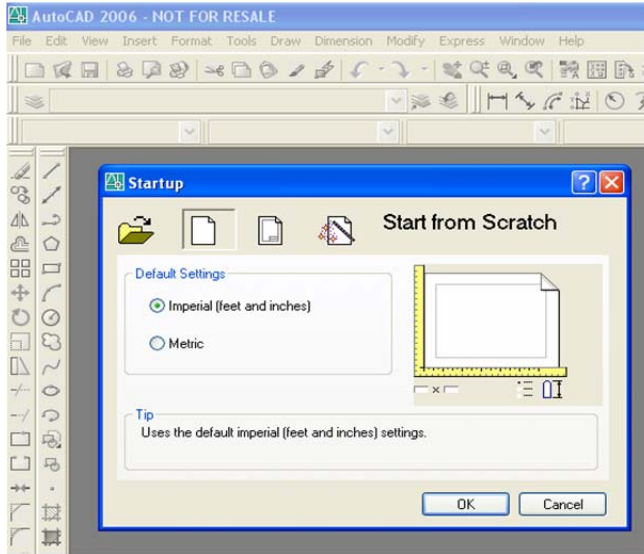


Figure 1-5E: Selecting either Imperial [feet and inches] or Metric, and clicking the OK button to start a new file.

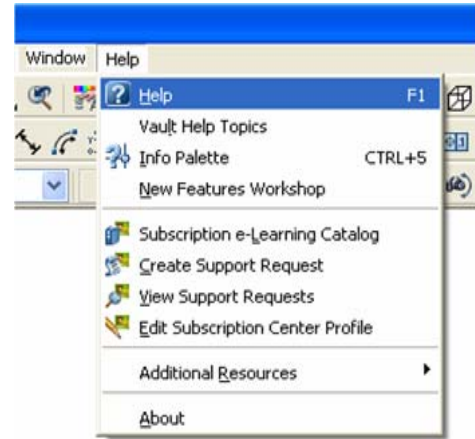


Figure 1-5F: Clicking-open the Help pull-down menu, or pressing the F1 key on the keyboard.

## **Section 4**

### ***Getting Ready To Solve Descriptive Geometry Problems in AutoCAD 2D Environment***

#### ***Foundation in AutoCAD for use of this collection of learning modules***

This collection of descriptive geometry learning modules using the 2D tools of AutoCAD program has been written in such a way that each individual tool with its particular options and settings, as well as each step in the completion of any particular project are thoroughly described in great details. Therefore, there is practically little need for students to possess great skills in AutoCAD in order to use the learning modules contained in this collection. However, a beginning course in AutoCAD (covering basic settings and 2D tools) is helpful. For those students who has never taken any course in AutoCAD before, any of the following is recommended:

- Concurrently enroll in a beginning AutoCAD course.
- Running through the Lessons in either of these two textbooks: 1. *AutoCAD 2006 Tutorial First Level: 2D Fundamentals* written by Randy H. Shih and published by Schroff Development Corporation ([www.schroff.com](http://www.schroff.com)); 2.

*AutoCAD 2006 Exercise Workbook for Beginning AutoCAD* written by Cheryl R. Shrock, Chair of Drafting Technology at Orange Coast College, Costa Mesa, CA, and published by Shrock Publishing (714-557-2555).

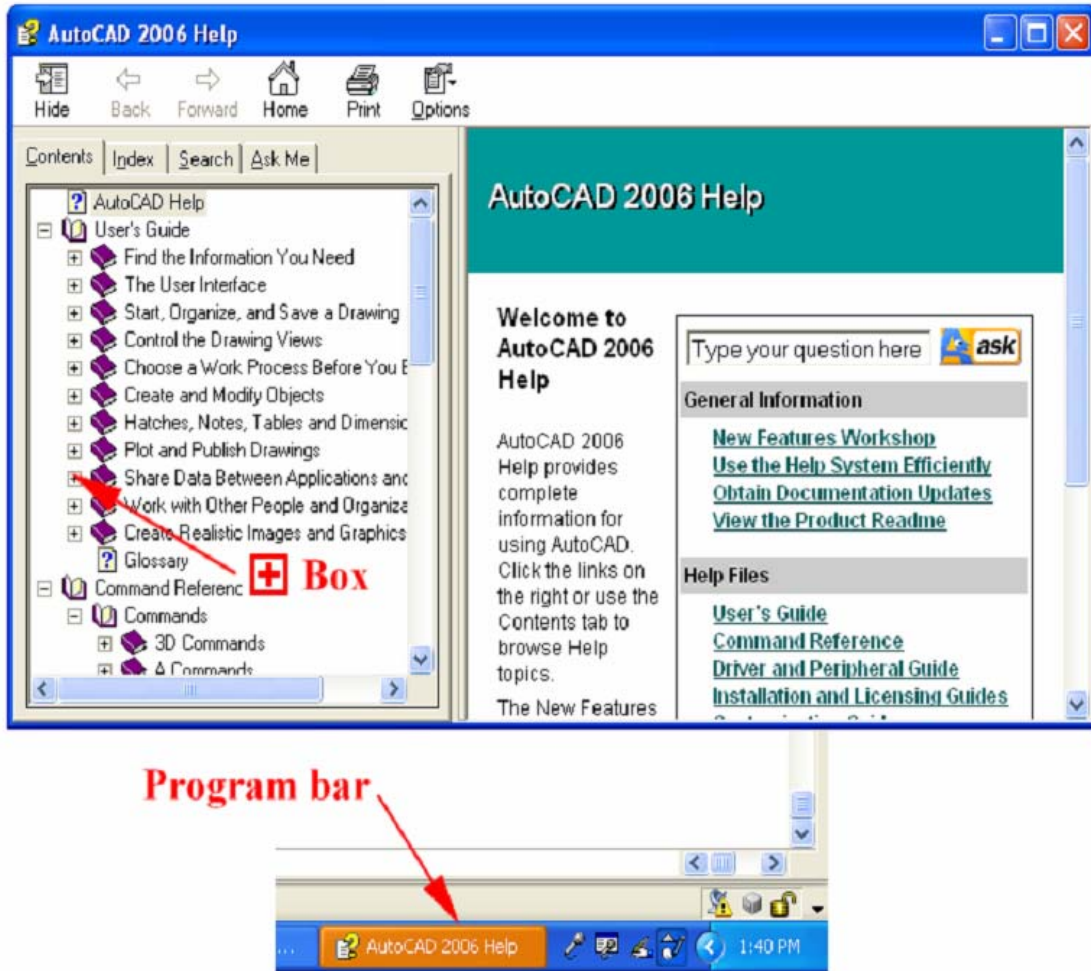


Figure 1-5G: The AutoCAD 2006 Help window with its bar docked on the Start menu bar at the bottom of the screen.

### ***AutoCAD on-line User's Guide and Search for topics***

AutoCAD 2006 program contains an on-line *User's Guide* that provides a wealth of information on all tools and settings. To access the on-line **User's Guide**, first launch the AutoCAD program (Figure 1-5D); next, in the **Startup** dialog window, click-select the bullet button on the left of either **Imperial [feet and inches]** or **Metric** text, and click the **OK** button to start a new file (Figure 1-5E), a new file opens; next, click-open the

**Help** pull-down menu (the rightmost on the menu bar), and choose the **Help** option (Figure 1-5F), or press the **F1** key on the keyboard to select the same menu, the **AutoCAD 2006 Help** window opens with its bar docked on the **Start** menu bar at the bottom of the screen (Figure 1-5G), if the **AutoCAD 2006 Help** window is hidden, then click bar to bring the window to the front. The **AutoCAD 2006 Help** window has two main panels; the left panel has several tabs; the **User's Guide** is under the **Contents** tab; the **User's Guide** is organized by topics; and to access the topics, click-open the boxed + sign on the left of the related topic listing. The **Search** tab allow user to type the text on requested information and obtain a list of related topics; to use this tool, click-open the **Search** tab, type a text string for requested information and click the **List Topics** button, a list of related topics appears in the left panel; click-select a listing, double-click on it or click the **Display** button, the information appears immediately on the right panel (See Figure 1-5H).

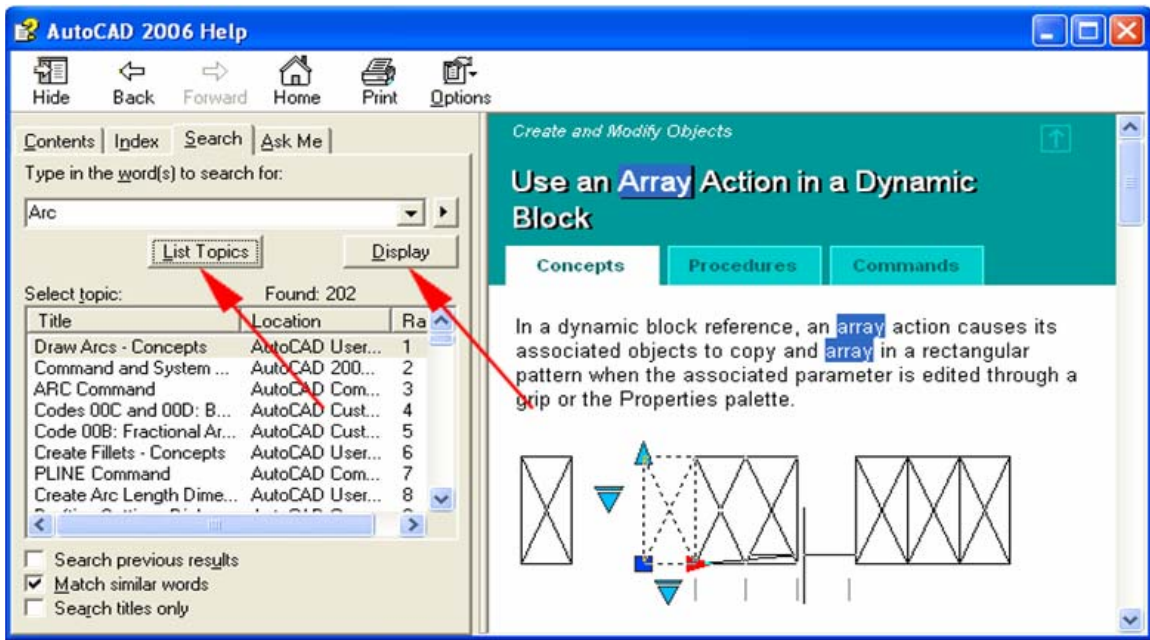


Figure 1-5H: The Search engine for AutoCAD program information.

### ***The most important tools and settings used in the learning modules***

The most frequently used tools in this collection of learning modules are listed in Figure 1-5J. There are three basic methods for accessing a tool (Figure 1-5K):

- **Pull-down Menu**: Tools are grouped under various pull-down menus; some options of a particular tool (such as **Arc**) can be accessed only through the

pull-down menu. However, it is usually more time-consuming to access a tool this way; therefore, this method is generally not recommended.

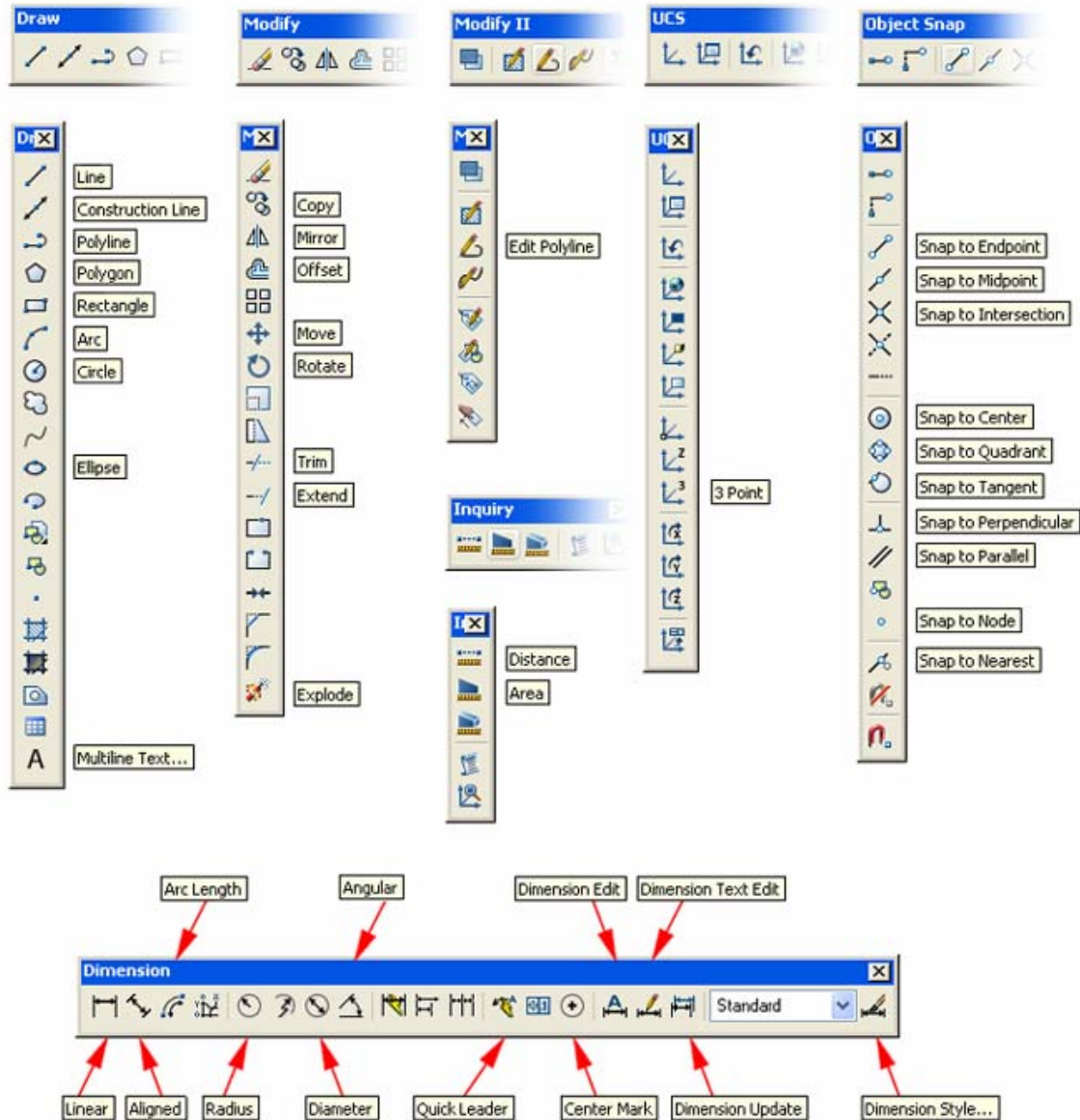


Figure 1-5J: The most frequently used tools in this collection of learning modules.

- **Tool Icon:** Tools can be accessed by clicking a tool icon on a related tool bar; however, in most cases, tool icons do not provide access to the options of the tools (in AutoCAD, options of tools are selected by typing the capitalized

letter of the name of the option listed in the **Command Line** and then pressing the **Enter** key on the keyboard); therefore, this method is not recommended either.

- **Command Line**: AutoCAD started as a code-based program and still carries this tradition, although it has evolved over time into a program based on GUI (graphical user interface); most of the tools and settings in AutoCAD can be invoked by typing the full name or the code (usually the first letter, or a few letters from the names of the tools), and then pressing the **Enter** key on the keyboard. This is the fastest way of invoking a tool or a setting, and is almost the only way to select the options of most tools. Therefore, this method is recommended and is used in the subsequent modules. To help the students familiarize with and memorize **Command Line** codes for tools and settings used in the subsequent learning modules, a list of codes and key names is provided as follows:

<b><u>Name of Tools or Settings</u></b>	<b><u>Keyboard Shortcut</u></b>
Open	Ctrl+O
Cut	Ctrl+X
Copy	co
Copy and Paste	Ctrl+C and Ctrl+V
Line	l
Polyline	pl
Polyline Edit	pe
Multiline	ml
Construction Line	xl
Polygon	pol
Rectangle	rec
Arc	a
Ellipse	el
Circle	c
Multiline Text	mt
Explode	x
Erase	e, or Delete key (on the keyboard)
Mirror	mi
Offset	o
Array	ar
Move	m

<u>Name of Tools or Settings</u>	<u>Keyboard Shortcut</u>
Rotate	ro
Scale	sc
Trim	tr
Extend	ex
Properties	ch, mo, or double-click the entity
Measurement	me (do not type measure)
Division	div
Drafting Settings	ds
Options	op
Distance	di
Layer Properties Manager	la

### *Achieving accuracy in drafting*

Accuracy in drafting is very important in the solution of descriptive geometry problems. In AutoCAD, such accuracy can be achieved by the following factors:

- **Correct Command Line data entry:** generally speaking, after a tool (such as **Line** and **Circle**) is invoked, the **Command Line** prompts the user to select an option by typing the capitalized letter of the name of the option (*Figure 1-5L*), or to type a requested value for a parameter (such as the center and radius of the circle, or the endpoints of a line), and then and pressed the **Enter** key on the keyboard to confirm the option and to proceed to the next option. The entry of values can be:
  1. “**Absolute coordinates:**” Type the x,y,z values of a point in the three-dimension space counted from the 0, 0, 0 origin point of the **WCS**, the default **World Coordinate System** or the user-defined **User Coordinate System (UCS)**, with locations on the right, above and toward user from the origin as positive values, and locations on the left, below and away from the user from the origin as negative values, following the “right-hand rule,” as shown in *Figure 1-5M*. For example, typing 3,5,-6 (no space between the comma punctuation “,” and the next value) and press the **Enter** key on the keyboard designates a point 3 units on the right, 5 units above and 6 units away from the user, all counted from the 0,0,0 origin point of the default **World Coordinate System** or the user-defined **User Coordinate System**.

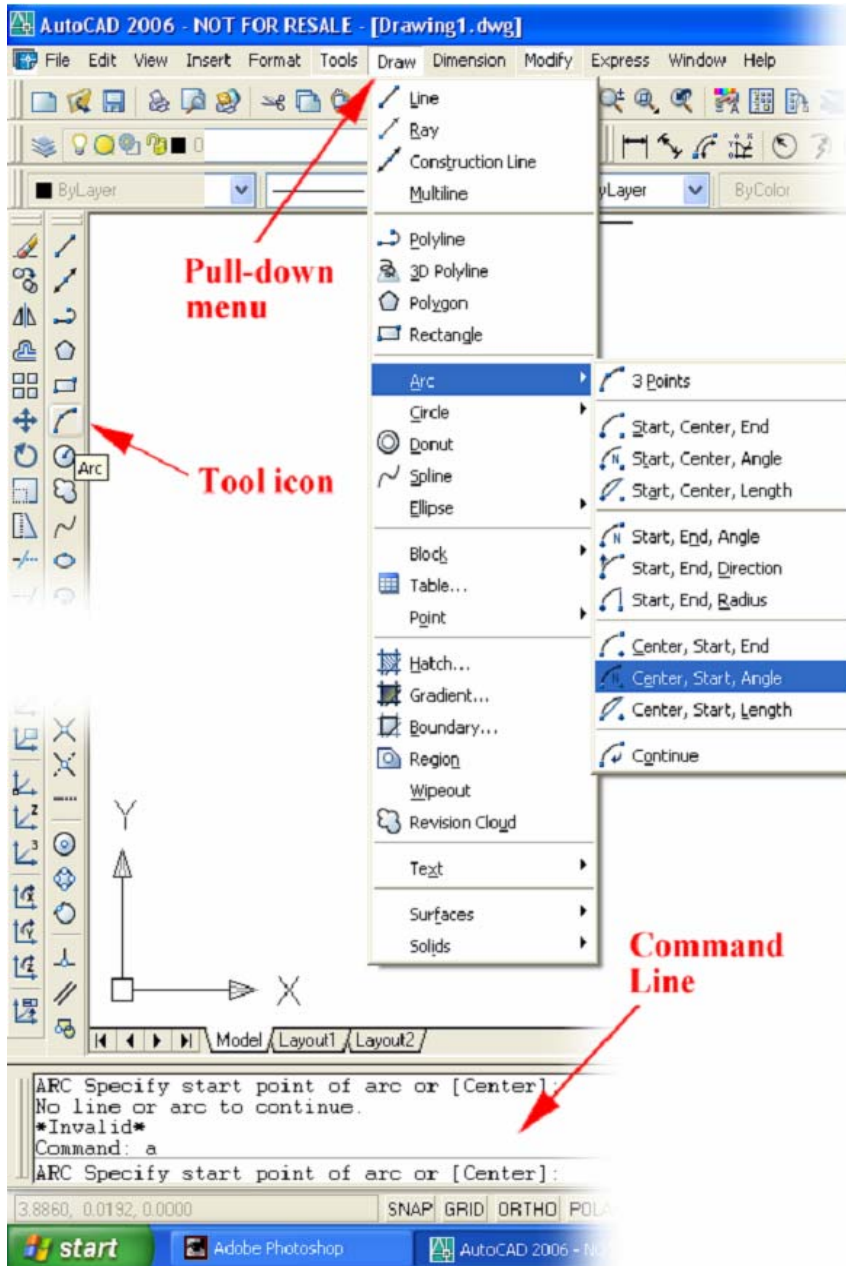


Figure 1-5K: Three basic methods for accessing a tool. Pull-down menu (top); tool icon (left); and Command Line entry (bottom).

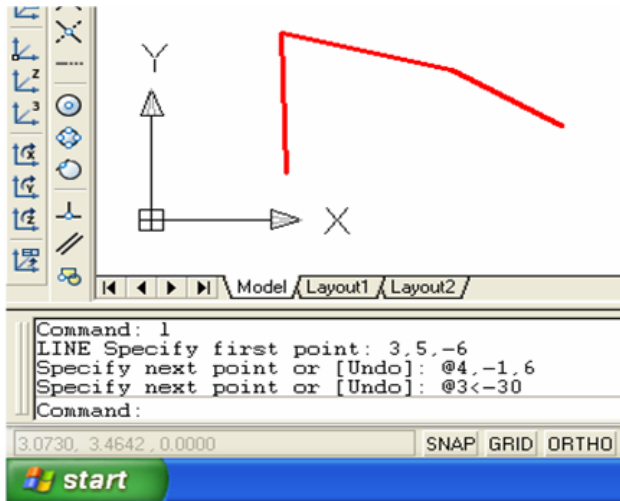


Figure 1-5L: Entry of option or value in the Command Line. “Absolute coordinates” entry (1<sup>st</sup> line); “relative coordinates” entry (2<sup>nd</sup> line); “polar coordinates” entry (3<sup>rd</sup> line).

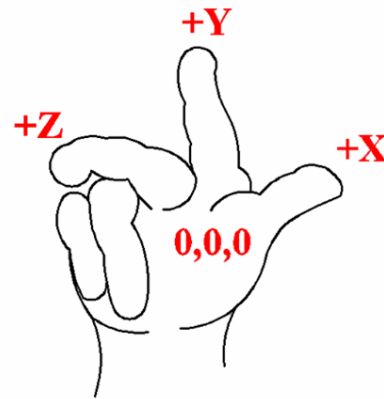
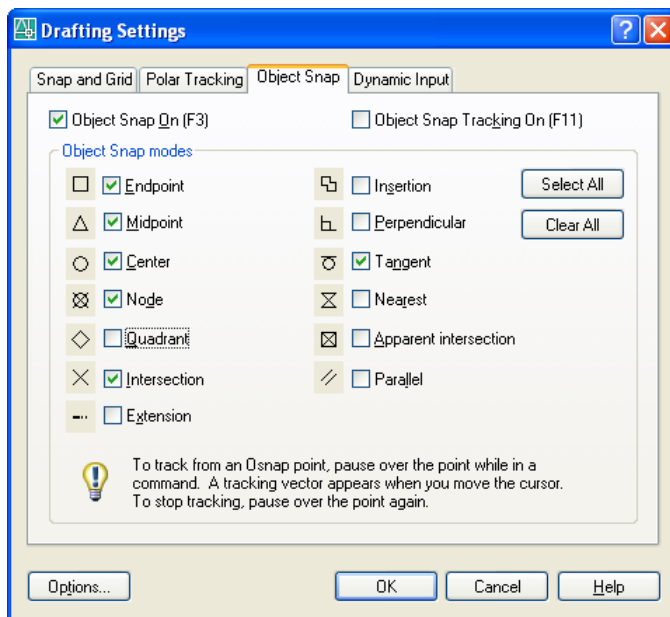


Figure 1-5M: The right-hand rule (the palm represents the 0,0,0 origin, the thumb pointing to +x, the index pointing to +y, and the middle finger pointing to +z).

2. “**Relative coordinates:**” Type the x, y, z values of the next point in the three-dimension space counted from the previous point, in the format of @x,y,z, following the same sign conventions as in “**absolute coordinates**”). For example, typing @4,-1, 6 (no space between the comma punctuation “,” and the next value) and press the **Enter** key on the keyboard designates a point 4 units on the right, 1 units below and 6 units toward the user, all counted from the last point entered.
  3. “**Polar coordinates:**” Type the distance value, always positive, and the angular value, in the format of @distance<angle-value, with any angle formed CCW or counter-clockwise from the positive x-axis of the **User Coordinate System** or **UCS** being a positive value, and any angle formed CW or clockwise from the positive x-axis of the **User Coordinate System** or **UCS** being a negative value. For example, typing @3<-30 and press the **Enter** key on the keyboard designates a point with a 3 units distance and 30° counted CCW from the positive x-axis of the **UCS**, counted from the last point entered.
- **Correct usage of Object Snaps:** The Object Snaps (OSNAP) allow the user to accurately set the coordinate points at certain strategic points of an entity (such as the **Center** and the four **Quadrants** of a **Circle**). There are two ways to access the **Object Snaps**:



1. “**Running Object Snaps:**” Too many Object Snaps turned on will cause confusion when picking up a needed Object Snap; therefore, turn on only the most frequently used **Object Snaps** as “running **Object Snaps**” in the **Drafting Settings** window, under the **Object Snap** tab. To open this window, type *ds* in the **Command Line** and press the **Enter** key on the keyboard; click the  check box between the symbols and the names of the Object Snaps (check marks are placed in the boxes as ), and on the left of **Object Snap On (F3)** text to turn “running **Object Snaps**” on; click the **OK** button to set the parameters and close the window (See *Figure 1-5N*).

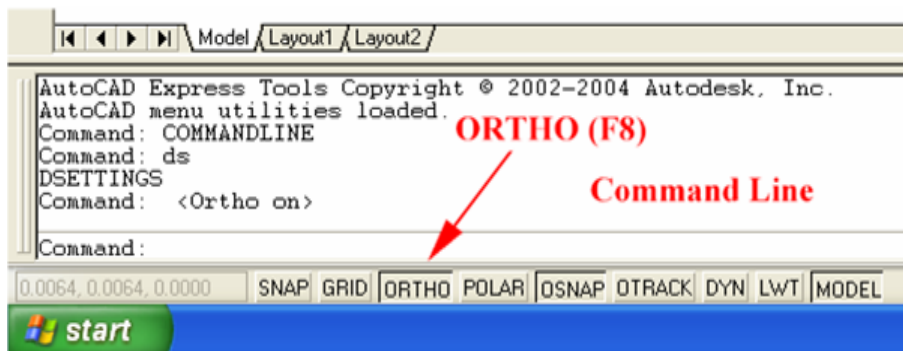


*Figure 1-5N: Turning running Object Snaps on in the Drafting Settings window.*

2. “**Temporary Object Snaps:**” For **Object Snaps** that are not used frequently, invoke them when needed by typing the first three letters of their name during the creation of a geometry entity.

<u>Osnap</u>	<u>Code</u>	<u>Osnap</u>	<u>Code</u>	<u>Osnap</u>	<u>Code</u>
Endpoint	end	Intersection	int	Nearest	nea
Midpoint	mid	Extension	ext	Apparent Intersection	app
Center	cen	Insetion	ins		
Node	nod	Perpendicular	per	Parallel	par
Quadrant	qua	Tangent	tan		

- Correct usage of ORTHO and POLAR functions: The **ORTHO** and **POLAR** function allow “**direct distance entry**” (**DDE**) during the creation and editing of a geometric entity. This method is very important in the solution of descriptive geometry problems through orthographic projection to be covered in the subsequent learning modules. Using this method, the users can move the mouse cursor to a certain direction, type a number that represents a distance along the constraint direction, and press the **Enter** key on the keyboard to establish a point on the geometric entity to be created:
  1. **ORTHO**: When the user click the **ORTHO** button beneath the **Command Line** or press the F8 key on the keyboard (*Figure 1-5P*), the **ORTHO** is turned on (the button looks “recessed”). When the user again click the **ORTHO** button beneath the **Command Line** or press the F8 key on the keyboard, the **ORTHO** is turned off (the button “jumps back to the front”). The **ORTHO** allows the user to draw only straight lines along the x- and y-axes according to the **World Coordinates System** or **User Coordinates System** chosen with the **UCS** tool. Since multi-view orthographic projection lines are drawn along these axes, the **ORTHO** function plays an important role in the solution of multi-view drafting problems. To use the **ORTHO** function as constraints for creating orthographic projection lines, while using the **Line** tool, first invoke the **Line** tool and press the **F8** key to turn **ORTHO** on, then click-select a point to be projected with the help of an appropriate **Object Snap**, then drag the mouse cursor in the desired direction and click once at a convenient location (*Figure 1-5Q*). To use the “**direct distance entry**” (**DDE**) method during the creation of a geometric entity, after a tool (such as **Line**) is invoked and the first endpoint is established, press the **F8** key to turn **ORTHO** on, drag the mouse cursor to the desired side of the first point to (left, right, top, or bottom), type a number for the distance and press the **Enter** key to finish (*Figure 1-5R*).



*Figure 1-5P: The ORTHO and POLAR on-and-off button beneath the Command Line.*

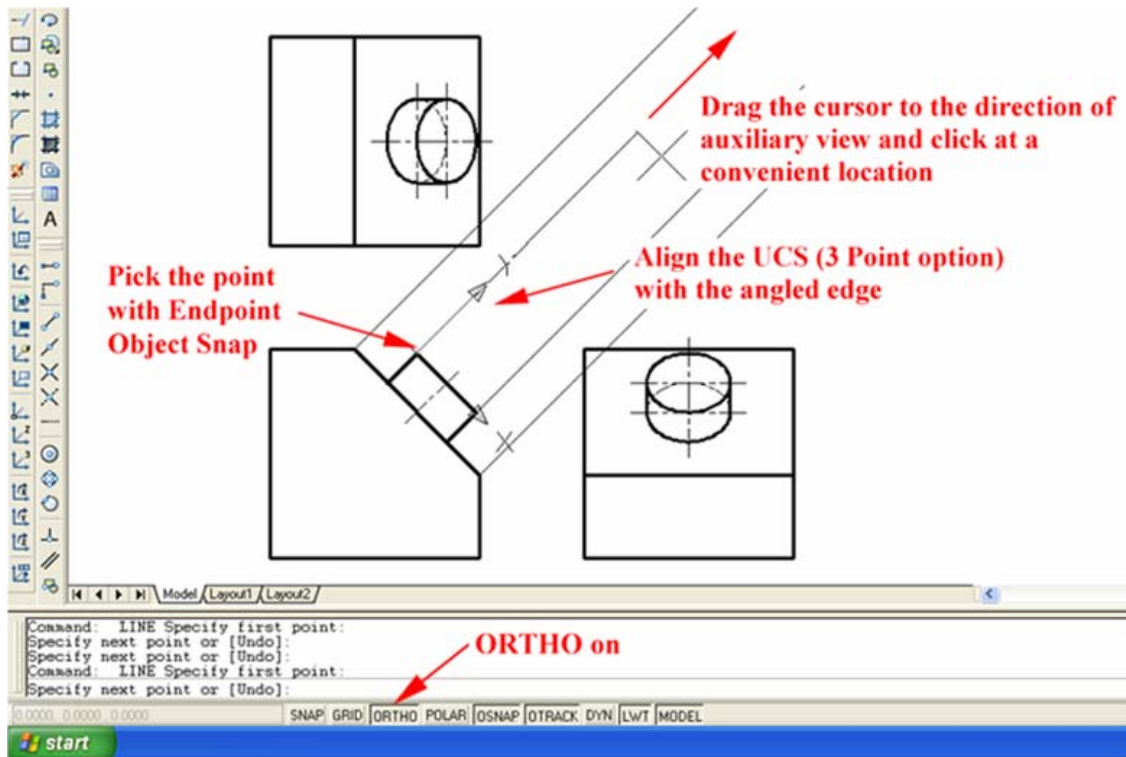


Figure 1-5Q: Multi-view orthographic projection lines drawn along the x- and y- axes established by the UCS, with the ORTHO constraint turned on.

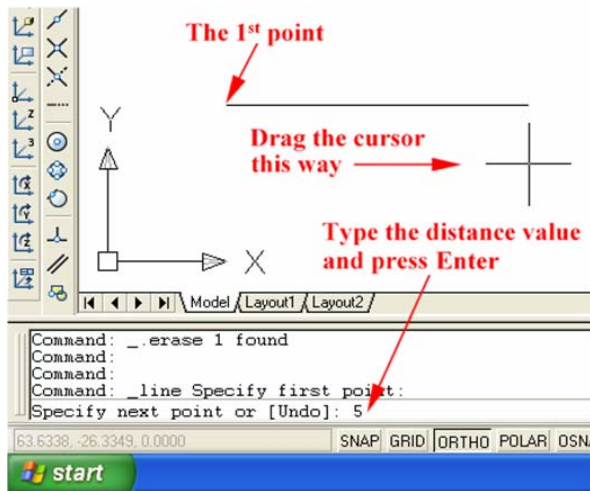
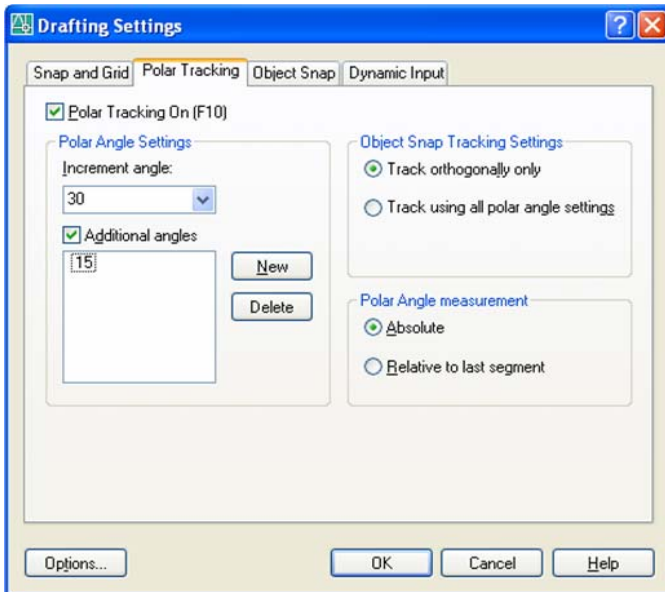
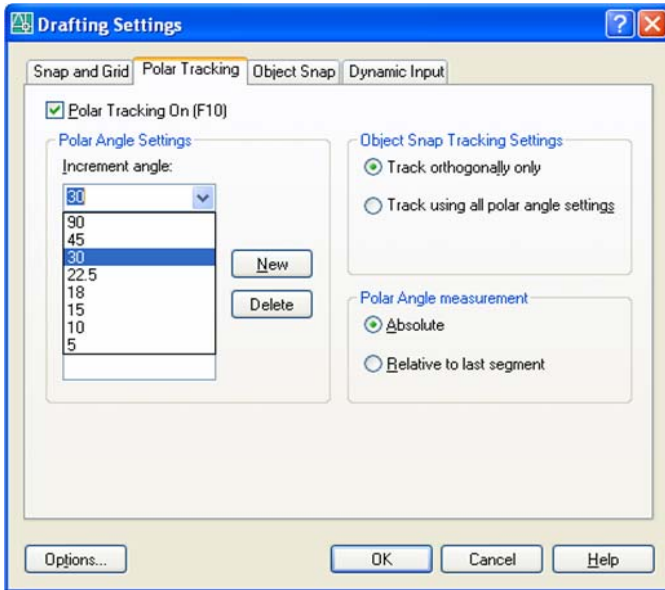


Figure 1-5R: "Direct Distance Entry" with ORTHO turned on when using the Line tool.

2. **POLAR (Polar Tacking)**: When the user click the **POLAR** button beneath the **Command Line** (on the right of the **ORTHO** button) or press the F10 key on the keyboard, the **POLAR** is turned on (the button looks "recessed"), and the **ORTHO** is automatically turned off. When the

user again click the **POLAR** button beneath the **Command Line** or press the F10 key on the keyboard, the **POLAR** is turned off (the button “jumps back to the front”). The **POLAR** allows the user to draw straight lines at a suggested angle with the x-axes according to the **Increment angle** values at in the **Drafting Settings** window under the **Polar Tracking** tab. To setup **Polar Tracking** parameters, type *ds* in the **Command Line** and press the **Enter** key on the keyboard to open the **Drafting Settings** window, and select the **Polar Tracking** tab; in the **Polar Angle Settings** section, click the downward arrow  $\nabla$  button for a pull-down list of available increment angles and select one (“increment angles” are multiples of a selected angle; for example, if 30 is selected from the list, then **Polar Tracking** will apply to 30°, 60°, 90°, 120°, ...); to add a particular angle needed, click the **New** button, a boxed text field appears in the  **Additional angles** white space, type a number for an additional angle (**Additional angles** applies only to the particular angles established, not to their multiples); then click the  check box on the left of both **Polar Tracking On (F10)** and **Additional angles** texts, check marks are placed inside the boxes (); click the **OK** button to set the parameters and close the window (*Figure 1-5S*). To use the **POLAR** function as hints for creating angled lines with the “**direct distance entry**” (**DDE**) method, first invoke the **Line** tool and press the F10 key to turn **POLAR** on, then select the first endpoint of the line, then move the mouse cursor in the desired angular direction until a dotted “tracking line” and a boxed “polar tracking tip” appear on the screen; then type a number representing the distance along the angular direction from the first point, and press the **Enter** key on the keyboard to finish (*Figure 1-5T*).

The **ORTHO** and **POLAR** functions are mutually exclusive since the **ORTHO** constraints the direction parallel to the x-axes, while the **POLAR** constraints the direction at an angle to the X- and y-axes; in other words, they can not be turned on at the same time.



*Figure 1-5S: Turning the Polar Tracking on in the Drafting Settings window. Selecting an Increment angle (top), creating Additional angles (bottom).*

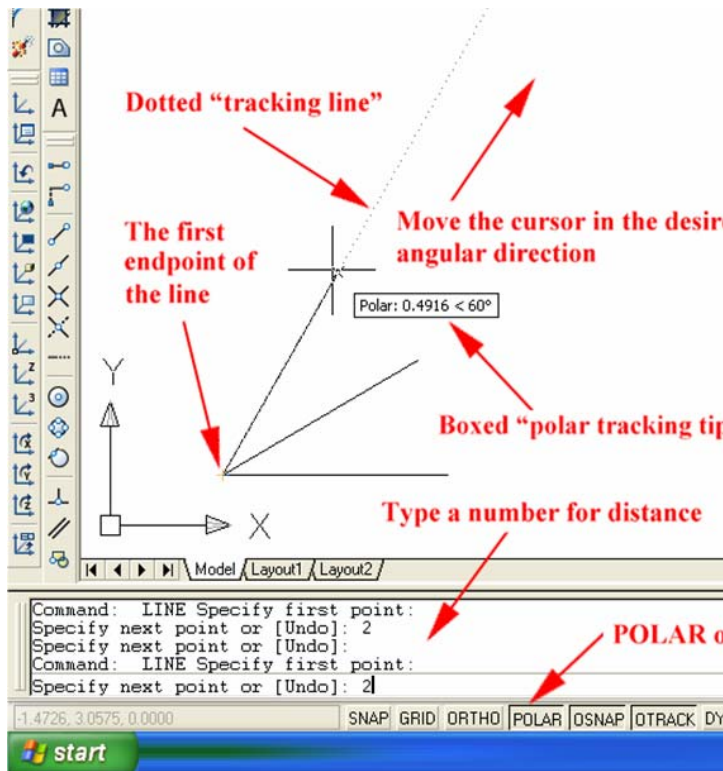


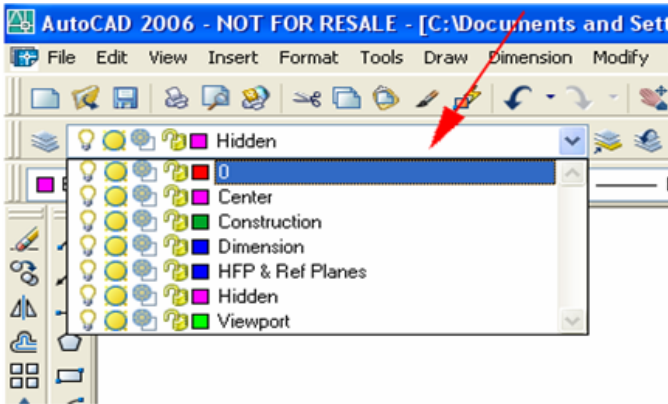
Figure 1-5T: Using the Polar Tracking function for creating angled lines with the “direct distance entry” (DDE) method. Dotted “tracking line” (top left); and boxed “polar tracking tip” (middle right).

### Layer management in AutoCAD files

The last part of this module will explore the concept of **layer** in AutoCAD. Layers in AutoCAD allow collective management of the properties (such as color, line weight, line type, etc.) of objects created in particular layers; and if any of the properties of a particular is changed in the **Layer Properties Manager**, then the relevant property of all objects created in the layer (except the ones whose relevant property has been individually changed before) will change accordingly. In a typical drawing, different types of objects (for example, object lines, hidden lines, center lines, dimensions, etc.) are created with different properties for convenient identification. In the subsequent learning modules, it is recommended that students create different types of objects in different layers, by first selecting the appropriate layer from the **layer** pull-down list (Figure 1-6A), and then start to create object on the selected **layer**; all objects created on the selected **layer** will assume the properties of the **layer**. By default, all objects create in any layer have their properties listed as **ByLayer** in **Color**, **Linetype** and **Lineweight** pull-down list. Occasionally, any of the properties of one or several objects created on a layer needs to be changed; to accomplish this task, select the objects first, then go to the **Color**, **Linetype** and **Lineweight** pull-down list and select the appropriate item (Figure 1-6B);

when the task is accomplished, press the **Esc** key on the keyboard to exit the editing process.

We will now proceed to creating several layers for the subsequent learning modules. Click the **Layer Properties Manager** tool button on the Properties tool bar; the **Layer Properties Manager** window opens (*Figure 1-6C*).

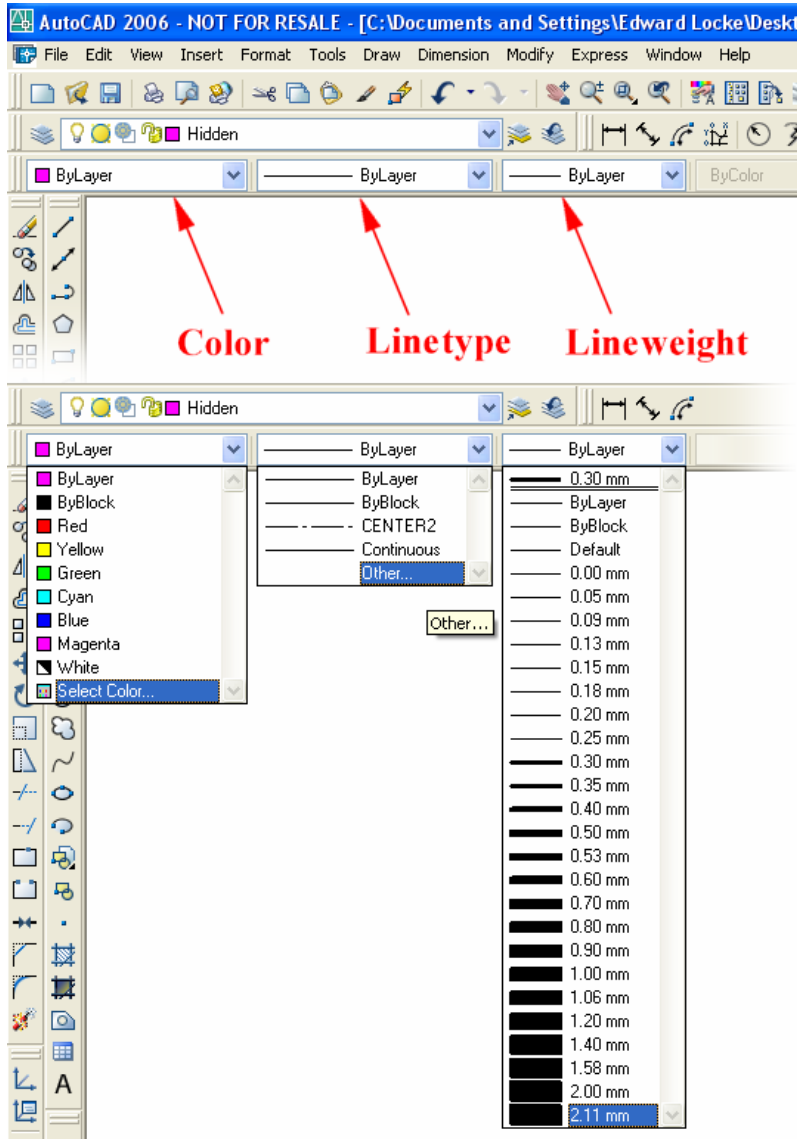


*Figure 1-6A: Selecting the appropriate layer from the layer pull-down list.*

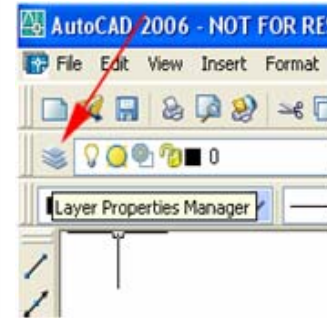
On each **layer** in the **Layer Properties Manager** window, there are several options that we can change by clicking the item listed on each **layer** to open the related window, and choosing the appropriate item in the window and then clicking the **OK** button. These options and related windows include:

- **Select Color window:** Click-select any color in the samples well and click the **OK** button to return to the **Layer Properties Manager** window (See *Figure 1-6C*).
- **Select Linetypes window:** By default, this window lists only one line type (continuous), as shown in *Figure 1-6C*. To add more line types, click the **Load...** button; the **Load or Reload Linetypes** window opens (*Figure 1-6D*); select a line type and click the **OK** button to return to the **Select Linetypes** window, and click the **OK** button to return to the **Layer Properties Manager** window.
- **Lineweight window:** Notice that all line weights are measured in metric unit (millimeter or mm). Select appropriate line weight (*Figure 1-6C*) and click the **OK** button to return to the **Layer Properties Manager** window. In the drawing window, the line weight of the objects can be displayed when the **LWT** button is turn on (*Figure 1-6E*).

- **On button:** A bright yellowish light bulb symbol is by default displayed under the **On** option name, indicating that the objects to be created on the layer will be visible; to make the objects invisible, click the bright yellowish light bulb symbol; it turns to a grayish bulb, indicating that the objects to be created on the layer will be invisible (*Figure 1-6F*).



*Figure 1-6B (left): Changing the properties of one or several objects on a layer from the Color, Linetype and Lineweight pull-down list. Clicking the Select Color option in the Color pull-down menu will open the Select Color window; clicking the Other... option in the Linetype pull-down menu will open the Select Linetype window(as shown in Figure 1-6C).*



*Figure 1-6C (right): The Layer Properties Manager tool icon on the Properties tool bar.*



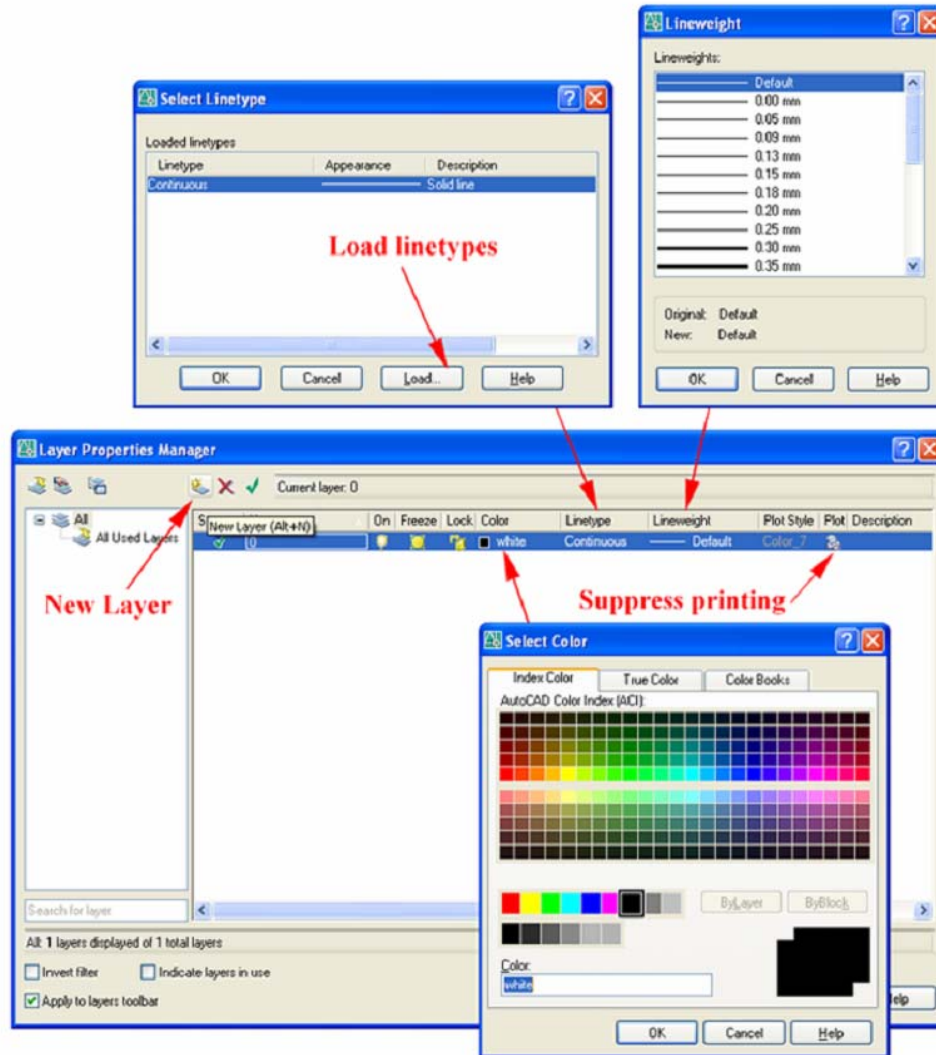


Figure 1-6C: Layer Properties Manager with various property options.

- **Plot** button: A printer symbol is by default displayed under the **Plot** option name, indicating that the objects to be created on the layer will be printed; to make the objects not printable (but still visible on the layer unless the **On** button is turned off), click the printer symbol; a red forward slash/ is placed on the printer symbol (in the previous version of AutoCAD, a red stop sign Ø was used), indicating that the objects to be created on the layer will not be printed (Figure 1-6F).

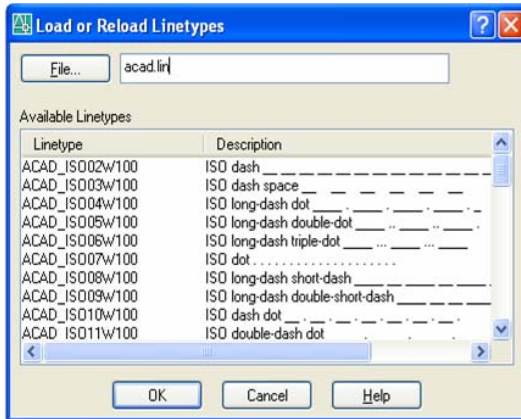


Figure 1-6D: The Load or Reload Linetypes window.

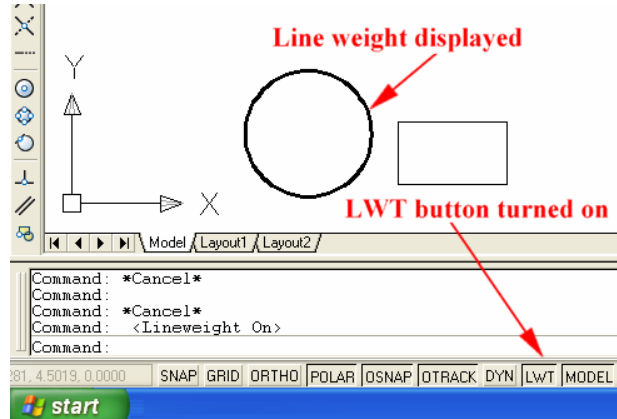


Figure 1-6E: The LWT button.

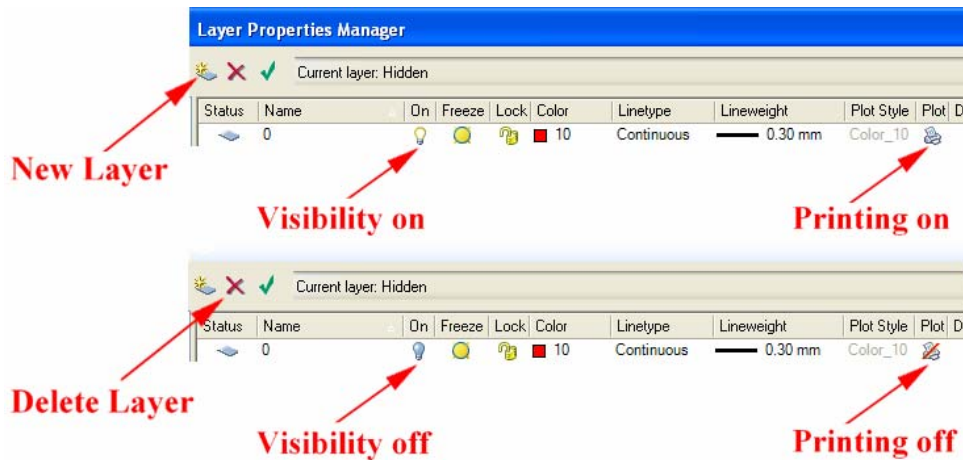


Figure 1-6F: The On (visibility) and Plot (printing) options.

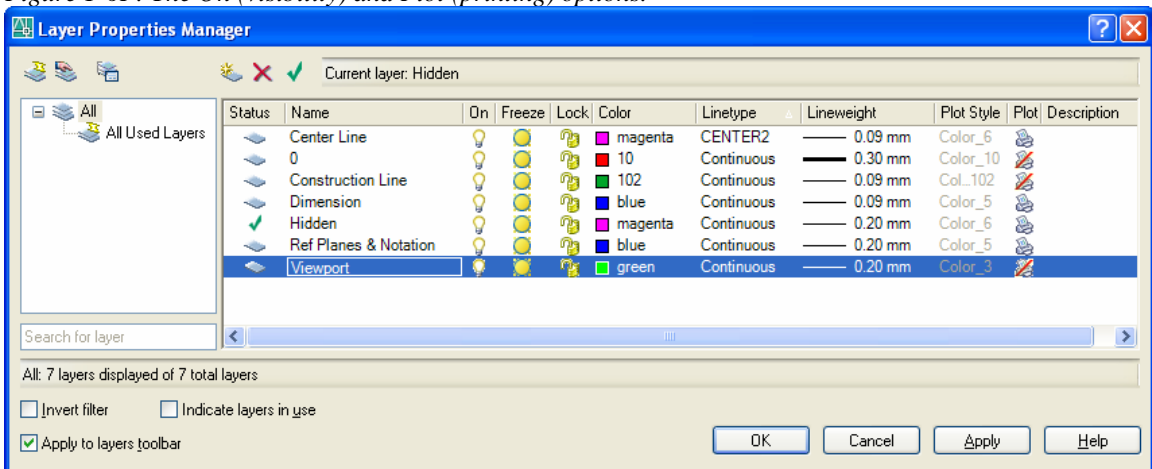


Figure 1-6G: The Layers created for the subsequent modules.

To create a new layer, click the **New Layer** button (*Figure 1-6F*); a new layer appears in the list with the name highlighted, type a new name and perform desired changes of properties (Notice that the name of the first layer that is created by default will stay **0** and cannot be changed). To delete a layer, select the layer first and then click the **Delete Layer** button, with a red **x** symbol, as shown in *Figure 1-6F* (Notice that the first layer that is created by default, or the layer **0**, cannot be deleted). It is recommended to create several **layers** with properties as shown in *Figure 1-6G*. Notice that the **Layer 0** will be used to create visible (“object”) lines (especially the **cut lines** in development patterns) throughout the subsequent learning modules. When all layers are created with desired properties, click the **OK** button to exit the **Layer Properties Manager** window.

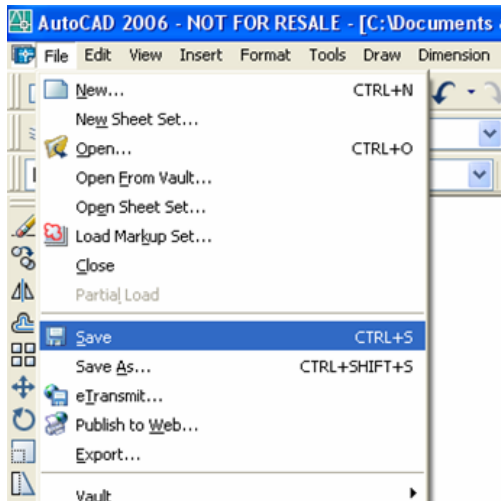


Figure 1-6H: The File → Save menu (highlighted).

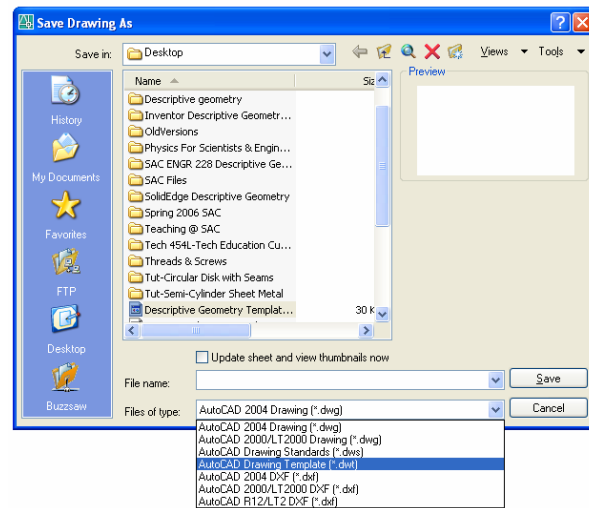


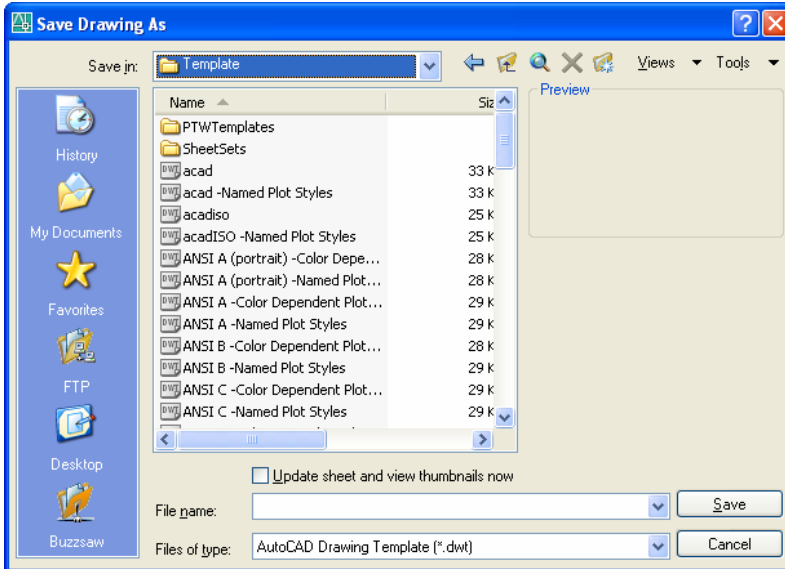
Figure 1-6J: Selecting AutoCAD Drawing Template [\*.dwt] file type.

### Saving the file as an AutoCAD .dwt template file

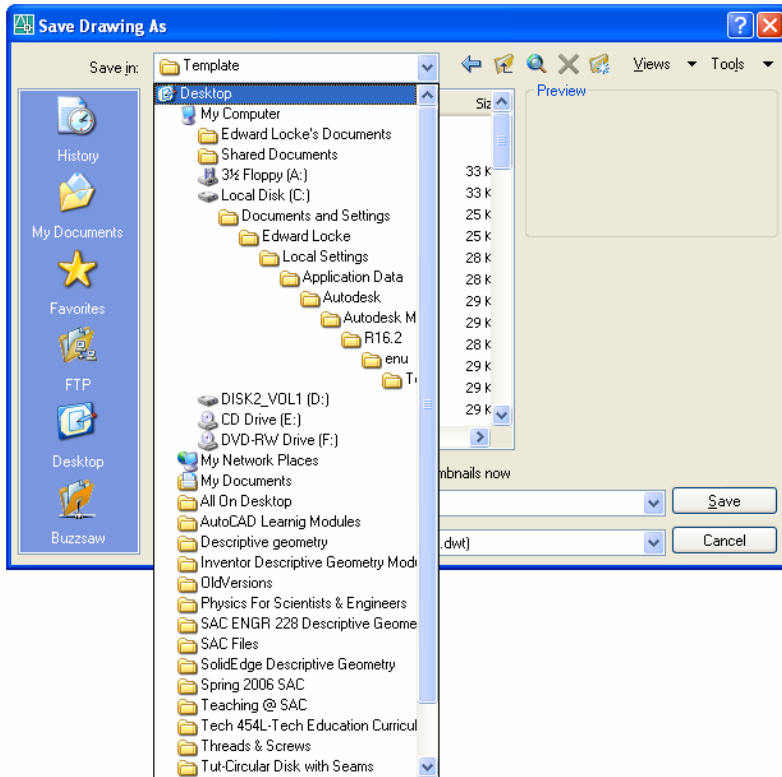
We will now proceed to save the file as an AutoCAD .dwt Drawing Template file, which allows the file to be used many times and saved as other AutoCAD .dwg Drawing files. Go to the File→Save menu (*Figure 1-6H*) or press the **Ctrl** and **s** keys on the keyboard simultaneously, to open the **Save Drawing As** window.

In the **Save Drawing As** window, select **AutoCAD Drawing Template [\*.dwt]** file type from the **Files of type** pull-down list (*Figure 1-6J*); the AutoCAD Save file directory jumps to the **Template** folder (*Figure 1-6K*); go to the **Save in** pull-down menu to select a desired directory, in this case, we choose the **Desktop** (*Figure 1-6L*); click the

**Create New Folder** icon to create a new folder (*Figure 1-6M*); a new folder opens with the default name highlighted (*Figure 1-6N*).



*Figure 1-6K: The AutoCAD Save file directory jumping to the Template folder.*



*Figure 1-6L: Selecting a desired directory.*

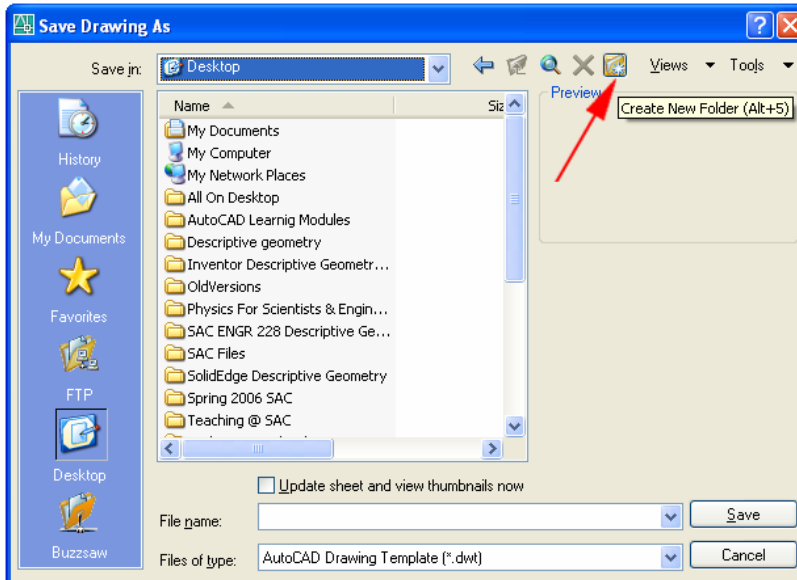


Figure 1-6M: Clicking the Create New Folder icon to create a new folder.

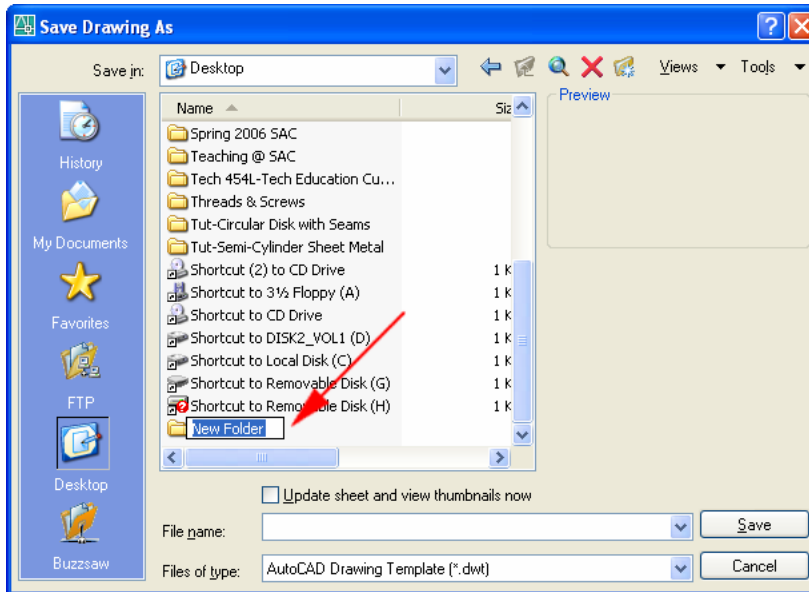


Figure 1-6N: The new folder with the default name highlighted.

Next, type *AutoCAD Descriptive Geometry Learning* as the name for the folder and double-click the icon of the folder to open it (*Figure 1-6P*); in the **File name** text field, type *AutoCAD Descriptive Geometry Template* as the name for the file and click the **Save** button to save the file as a template (*Figure 1-6Q*); the **Template Description** window opens; type a description text if desired and click the **OK** button to save the file

(Figure 1-6R). The template's icon appears in the folder just created, on the **Desktop** (Figure 1-6S).

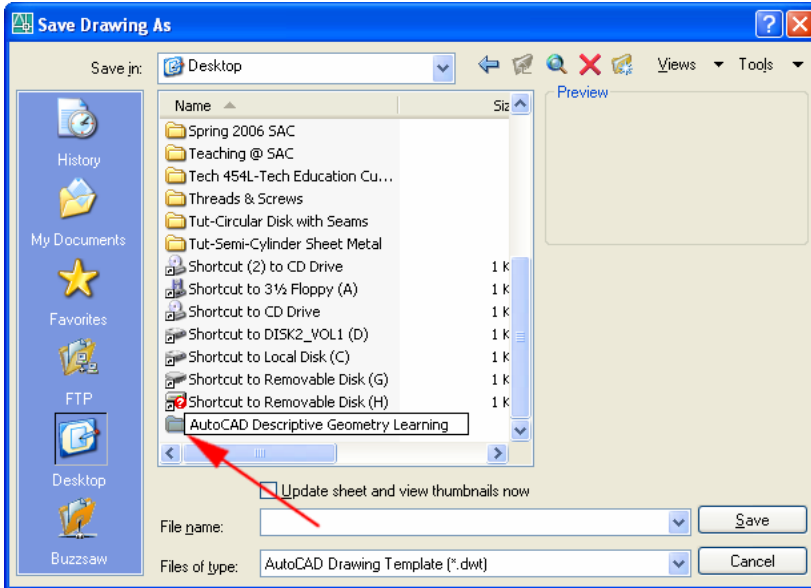


Figure 1-6P: Typing a new name for the folder and double-clicking the icon of the folder to open it.

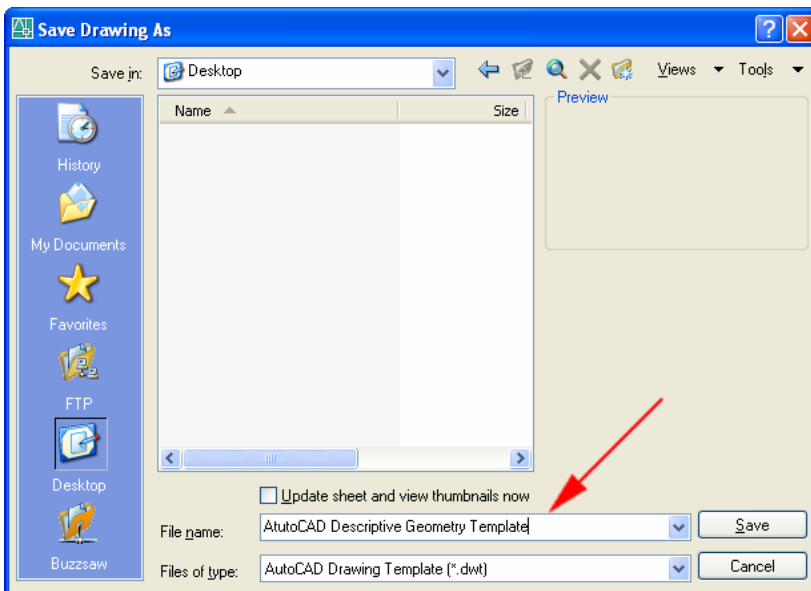


Figure 1-6Q: Typing a new name for the file and clicking the Save button to save the file as a template.

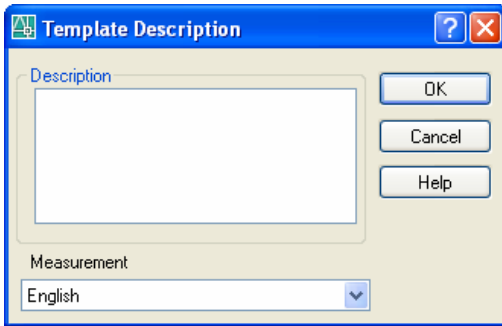


Figure 1-6R: Typing a description text if desired and clicking the OK button in the Template Description window to save the file.

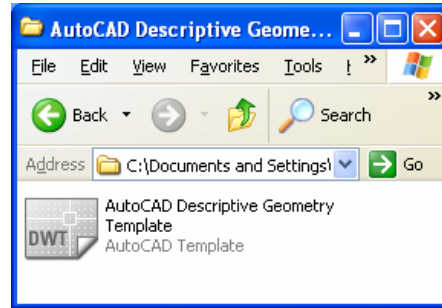


Figure 1-6S: The template's icon in the folder just created.

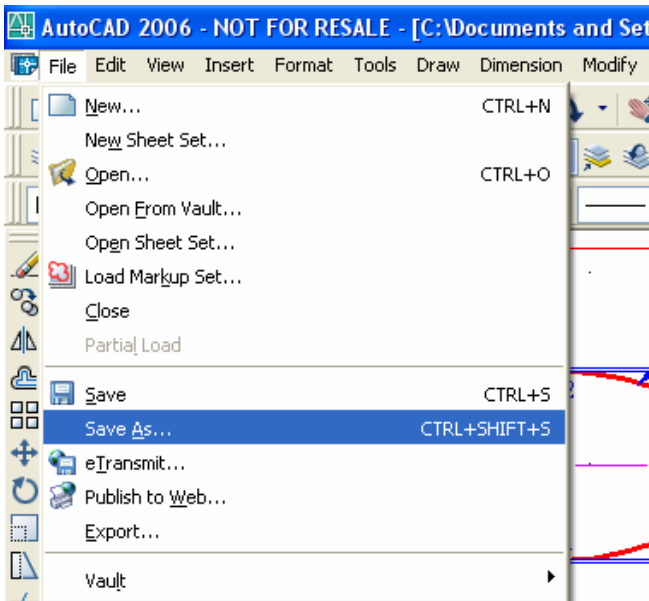


Figure 1-6T: The File → Save As... menu (highlighted).

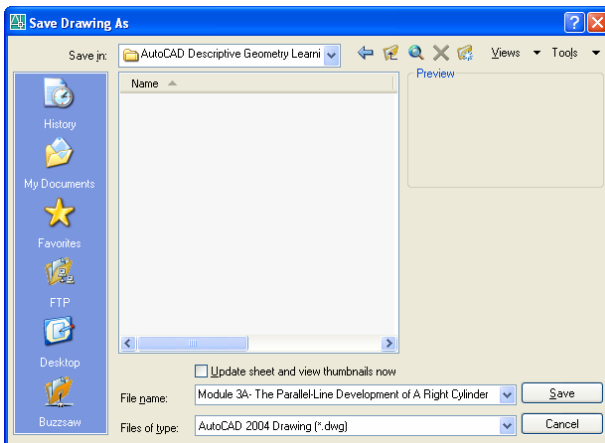


Figure 1-6U: The AutoCAD Drawing 2004 [\*].dwg file format from the Files of type pull-down list.

For your convenience, when you start a new module during this course, you can first launch the AutoCAD program; then open the *AutoCAD Descriptive Geometry Template* file from the *AutoCAD Descriptive Geometry Learning* folder; and then go to the **File**→**Save As...** menu (*Figure 1-6T*) to save it under another name and as an **AutoCAD 2004 Drawing [\* .dwg]** file (this file format can be accessed from the **Files of type:** pull-down list, as shown in *Figure 1-6Q*), in the same folder; and then start your drafting session.

Congratulations! In this module, you have explored

- The basic concepts and terms of descriptive geometry; and
- The application of the knowledge of and skills in solving descriptive geometry problems in industry, especially the sheet-metal design and fabrication;

In addition, you have

- Reviewed the basic tools and settings of AutoCAD programs that will be used in the subsequent learning modules; and
- Saved an AutoCAD .dwt Drawing Template file with useful layers for future use in the subsequent learning modules.