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## Chapter 1 : AutoCAD Tutorial First Level 2D Fundamentals - Download Free EBooks

*The primary goal of AutoCAD Tutorial - First Level: 2D Fundamentals is to introduce the aspects of Computer Aided Design and Drafting (CADD). This text is intended to be used as a training guide for students and professionals.*

This text is intended to be used as a training guide for students and professionals. This text covers AutoCAD and the lessons proceed in a pedagogical fashion to guide you from constructing basic shapes to making multiview drawings. The lessons are further reinforced by the video presentations found on the enclosed multimedia DVD. It takes a hands-on, exercise-intensive approach to all the important 2D CAD techniques and concepts. This text is also helpful to AutoCAD users upgrading from a previous release of the software. The new improvements and key enhancements of the software are incorporated into the lessons. The 2D-CAD techniques and concepts discussed in this text are also designed to serve as the foundation to the more advanced parametric feature-based CAD packages such as Autodesk Inventor. The basic premise of this book is that the more designs you create using AutoCAD, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons. This book is intended to help readers establish a good basis for exploring and growing in the exciting field of Computer Aided Engineering. Special reference guides show students where the performance tasks are covered in the book. These videos cover the first three chapters and selected end-of-chapter exercises from throughout the book. Students see the AutoCAD interface, menu picks, and models that are created while a voice overlay explains the design activities. Students, while at home, will become acquainted with the AutoCAD interface and design process. Basic Object Construction Tools 3. Geometric Construction and Editing Tools 4. Object Properties and Organization 5. Orthographic Views in Multiview Drawings 6. Basic Dimensioning and Notes 7. Templates and Plotting 9.

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*AutoCAD® Tutorial: 2D Fundamentals AutoCAD Certified Associate Examination Objectives Coverage This table shows the pages on which the objectives of the Certified Associate Examination are covered in.*

Controlling the Display in Drawings Precision Creating Basic Drawings Format It is necessary to begin with the basic alphabet and learn how to use it correctly and effectively through practice. This will require learning some new concepts and skills as well as learning a different vocabulary. Today, the majority of the Mechanical CAD systems are capable of creating three-dimensional solid models. Nonetheless, all CAD systems create designs using basic geometric entities and many of the constructions used in technical designs are based upon two-dimensional planar geometry. The method and number of operations that are required to accomplish the basic planar constructions are different from one system to another. In order to become effective and efficient in using a CAD system, we must learn to create geometric entities quickly and accurately. In learning to use a CAD system, lines and circles are the first two, and perhaps the most important two, geometric entities that one should master the skills of creating and modifying. Straight lines and circles are used in almost all technical designs. In examining the different types of planar geometric entities, the importance of lines and circles becomes obvious. Triangles and polygons are planar figures bounded by straight lines. Ellipses and splines can be constructed by connecting arcs with different radii. As one gains some experience in creating lines and circles, similar procedures can be applied to create other geometric entities. In our example, AutoCAD opened the graphics window using the default system units and assigned the drawing name Drawing1. We should determine the system of units within the CAD system before creating the first geometric entities. In the Menu Bar select: Click on the Length Type option to display the different types of length units available. Confirm the Length Type is set to Decimal. On your own, examine the other settings that are available. This will set the measurement to the default English units, inches. Set the Precision to two digits after the decimal point as shown in the above figure. Pick OK to exit the Drawing Units dialog box. Setting the Drawing Limits controls the extents of the display of the grid. It also serves as a visual reference that marks the working area. Note that this setting does not limit the region for geometry construction. AutoCAD Fundamentals 4. On your own, move the graphic cursor near the upper-right corner inside the drawing area and note that the drawing area is unchanged. The Drawing Limits command is used to set the drawing area, but the display will not be adjusted until a display command is used. Inside the Menu Bar area select: If no objects are constructed, the Drawing Limits are used to adjust the current viewport. Move the graphic cursor near the upper-right corner inside the drawing area and note that the display area is updated. Hit the function key [F7] once to turn off the display of the Grid lines. Move the graphics cursor to the first icon in the Draw panel. This icon is the Line icon. Note that a brief description of the Line command appears next to the cursor. Select the icon by clicking once with the left-mouse-button, which will activate the Line command. AutoCAD expects us to identify the starting location of a straight line. Move the graphics cursor inside the graphics window and watch the display of the coordinates of the graphics cursor at the bottom of the AutoCAD drawing screen. The three numbers represent the location of the cursor in the X, Y, and Z directions. We can treat the graphics window as if it was a piece of paper and we are using the graphics cursor as if it were a pencil with which to draw. Coordinates of the graphics cursor. Do not be overly concerned with the actual size or the accuracy of your freehand sketch. We will start at a location about one-third from the bottom of the graphics window. Left-click once to position the starting point of our first line. This will be point 1 of our sketch. Next move the cursor upward and toward the right side of point 1. Notice the rubber-band line that follows the graphics cursor in the graphics window. Left-click again point 2 and we have created the first line of our sketch. Move the cursor to the left of point 2 and create a horizontal line about the same length as the first line on the screen. Repeat the above steps and complete the freehand sketch by adding three more lines from point 3 to point 4, point 4 to point 5, and then connect to point 5 back to point 1. Notice

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that the Line command remains activated even after we connected the last segment of the line to the starting point point 1 of our sketch. Inside the graphics window, click once with the right-mouse-button and a popup menu appears on the screen. Select Enter with the left-mouse-button to end the Line command. Move the cursor near point 2 and point 3, and estimate the length of the horizontal line by watching the displayed coordinates for each point. This method is perhaps the fastest way to specify locations on the screen. However, it is rather difficult to try to create a line of a specific length by watching the displayed coordinates. It would be helpful to know what one inch or one meter looks like on the screen while we are creating entities. Note that the buttons in the Status Bar area serve two functions: When the corresponding button is highlighted, the specific option is turned ON. Using the buttons is a quick and easy way to make changes to these drawing aid options. Another aspect of the buttons in the Status Bar is these options can be switched on and off in the middle of another command. Move the cursor inside the graphics window, and estimate the distance in between the grid lines by watching the coordinates display at the bottom of the screen. Using the grid is similar to placing a sheet of grid paper under a drawing. The grid helps you align objects and visualize the distance between them. The grid is not displayed in the plotted drawing. The default grid spacing, which means the distance in between two lines on the screen, is 0. We can see that the sketched horizontal line in the sketch is about 4. Move the cursor inside the graphics window, and move the cursor diagonally on the screen. Observe the movement of the cursor and watch the coordinates display at the bottom of the screen. When SNAP mode is on, the screen cursor and all input coordinates are snapped to the nearest point on the grid. The default snap interval is 0. Click on the Line icon in the Draw toolbar. Use the right-mouse-button and select Enter in the popup menu to end the Line command if you have not done so. We will erase two of the lines using the Erase command. Pick Erase in the Modify toolbar. The icon is a picture of an eraser at the end of a pencil. Select any two lines on the screen; the selected lines are displayed as dashed lines as shown in the figure below. Right-mouse-click once to accept the selections. The selected two lines are erased. Inside the graphics window, click once with the right-mousebutton to bring up the popup option menu. Pick Repeat Erase, with the left-mouse-button, in the popup menu to repeat the last command. Notice the other options available in the popup menu. Move the cursor to a location that is above and toward the left side of the entities on the screen. Left-mouse-click once to start a corner of a rubber-band window. First corner Second corner 4. Move the cursor toward the right and below the entities, and then left-mouse-click to enclose all the entities inside the selection window. Notice all entities that are inside the window are selected. Inside the graphics window, right-mouse-click once to proceed with erasing the selected entities. Experiment with using the different commands we have discussed so far. In most CAD systems, the world space is defined using a three-dimensional Cartesian coordinate system. Three mutually perpendicular axes, usually referred to as the X-, Y-, and Z-axes, define this system. The intersection of the three coordinate axes forms a point called the origin. Any point in world space can then be defined as the distance from the origin in the X-, Y- and Zdirections. In most CAD systems, the directions of the arrows shown on the axes identify the positive sides of the coordinates. A CAD file, which is the electronic version of the design, contains data that describes the entities created in the CAD system. Information such as the coordinate values in world space for all endpoints, center points, etc. Knowing that AutoCAD stores designs by keeping coordinate data helps us understand the inputs required to create entities. By default, the user coordinate system is aligned to the world coordinate system WCS.

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