Learning CAD Training Videos

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Introduction

Philosophy ("Don't just tell me, show me")

Welcome to *Learning CAD Training Videos* CD pack. It is my earnest desire that you maximize your productivity using the tools introduced in this study guide and training videos. This training package is not intended to replace the technical manual that you received with the software. The first video is on using the *Help* command for getting further information and details on specific tools and or options.

Just as in building a home or any other structure, the foundation is the most critical. As an apprentice, one must first learn the tools, how they work, then learn the different methods for building the structure. With that training, the apprentice gains knowledge and skill sets that enable him or her to more efficiently construct the building.

In learning a new CAD system it is crucial that one becomes familiar with the environment and how to navigate within that environment. The video sequence is designed to familiarize you with and transition you from the basic workspace environment and templates, to basic 2D construction, geometry modification and transformation, levels, attributes, then into 3D modeling, Layout mode, dimensioning and printing. There is a variety of different construction techniques that will enable you to eventually determine which combinations work best for you as you create drawings and models. As you view the videos, there are several videos showing construction techniques that you can follow along with. The video will direct you to which file to open and work along with. There are a several challenge 2D and 3D problems in the appendix which you can create. You can then compare your work with the masters for accuracy that are included in this CD pack.

Intended Audience

The *Learning CAD Training Videos* study guide is for new users and pre-Cadkey 19 users who wish to migrate to KeyCreator. It is assumed that you have basic Windows skills.

Study Guide Structure

This study guide is intended to be used with the videos on this 2 CD pack. You are introduced to concepts, processes, and techniques for developing complete documentation for products and systems. If you are a new user view the videos in sequence for best results. As you gain confidence, you may want to move on to different modules, i.e. 3D modeling or Assemblies.

The appendix will have the 2D problems for you to generate. The **Video List** shows the time duration for each video. Hovering over the topic on the video menu list will also display the duration time. The **Hotkeys** (Keyboard Shortcuts) list will display in **bold** the ones used in the videos.

The **index** is a cross reference to the videos. This will get you directly to a specific video topic that you choose.

Do This First

Before viewing the videos:

1. **Print** this study guide for reference as you view the videos.
2. **Create** two folders named TRAINING and TRAINING 2 in the C: drive
3. **Copy** the **DEMO** and **FILE** folder from CD 1 into your TRAINING folder on the C: drive
4. **Copy** the **Training.wsp** and **Training.kbd** files from the FILES folder on this CD to the KEYCREATOR 3.0 folder on your computer. (It is assumed KEYCREATOR 3.0 folder is located at the root directory C :)
5. **Copy** the **Training.ckt** file from the FILES folder on this CD to the CKT folder located in the KEYCREATOR 3.0 folder on your computer. (It is assumed KEYCREATOR 3.0 folder is located at the root directory C :)
6. Launch the KeyCreator program
7. Select **Tools, Options**, then select **Startup** tab. (See Figure 1)
8. In the "Keyboard File" field, select "load" then set the path as shown in figure 1
9. In the "Workspace File" field, select the browse button (right side of field) then set the path as shown in Figure 1
10. Select **Tools, Options,** then select **Misc** tab. (See Figure 2)
11. In the "New Part Template" field, select the browse button then set the path as shown in Figure 2.
12. Exit KeyCreator then launch again, this will open with the new settings.
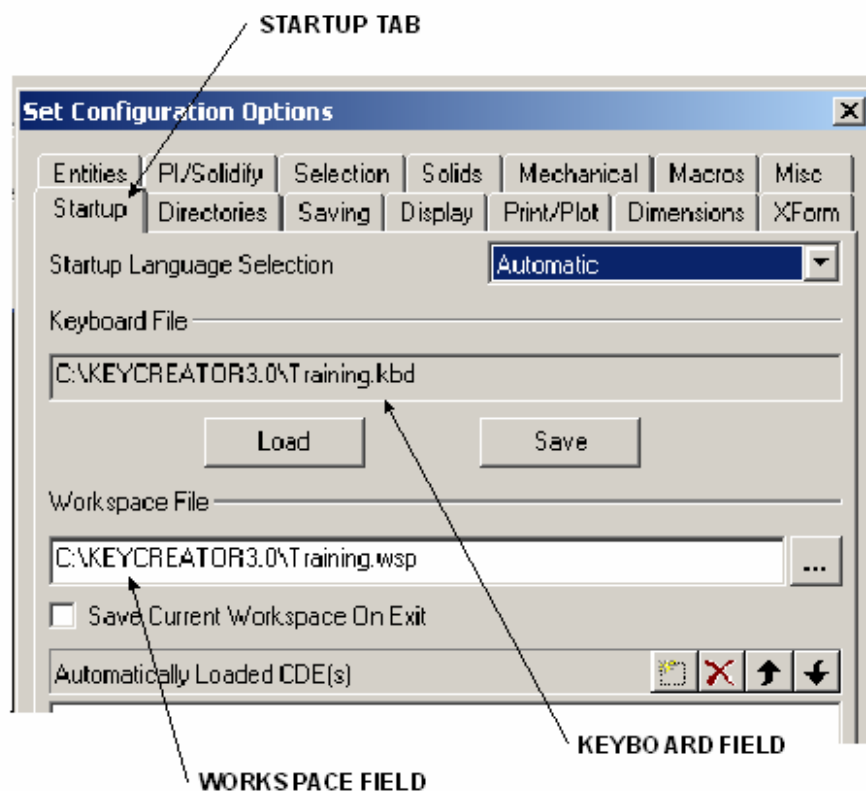
Figure 1
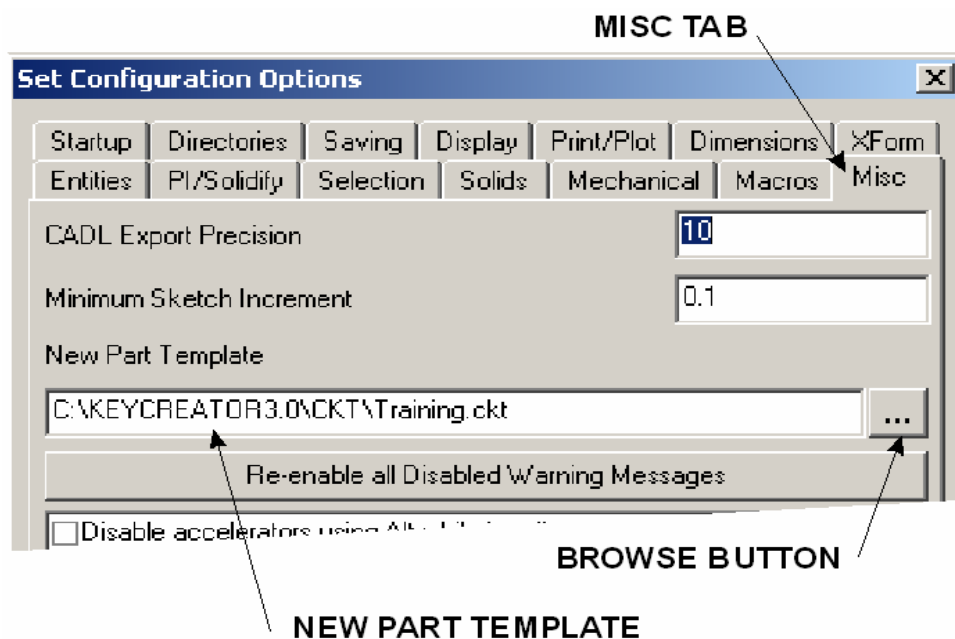
Figure 2

All the videos will be using the same settings. If you choose to create additional hotkeys to the existing .kbd file, save it with a different name (i.e. Training 2.kbd)

Note: For the best maximized screen display set your PC resolution to 1024x768 pixels. A lower resolution will not show the full recorded display window. A higher setting will only reduce the size of the recorded display window. You will determine the resolution that suits you best. To change the resolutions do the following:

On the task bar select the **Start** button, and then **Control Panel, Display,** then the **Settings** tab. On the "Display" scroll bar, click and drag the bar for changing the display resolution. Select **Apply**, the screen will temporarily go blank, then select **OK.** The screen display will then take on the new resolution.

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Fundamentals

Learning Process

You can either view the videos from the CDs or **copy** the following folder and files from CD 1 to TRAINING on the c: drive: **Media**, **autorun.inf**, and **Learning Cad Training Video 1.exe**. Then copy the following folder and files from CD 2 to TRAINING 2 on the c: drive: **Media**, **autorun.inf**, and **Learning Cad Training Video 2.exe.** Select the executable files to launch the video player menu.

When you insert the CD in the CD player, Camtasia player will launch the Menu page. Launch the CAD software. You can then toggle between software using the **Alt+Tab** keys. This will allow you to immediately practice what you have just viewed after viewing, pausing, or rewinding the video.

To end a selected video while viewing, press the **Esc** button to display the video controller, and then select the close button in the upper right corner of the video.

HELP

V1-**Help:** *After Viewing this video and applying the demonstration you will be able to:*

- *Use the help command to seek further information on the different features*
- *Use the Context Help for immediate information on feature selected*

The demonstration is on using the **Help** option on the menu line which can be accessed using the **Alt+F1** keys. This is a comprehensive system explaining every available feature and function. There are several demonstrations in this video including "**Context Help**" tool on the end of the default toolbar.

USER INTERFACE

V2-Interface Screen-Toolbars: *After Viewing this video and applying the demonstration you will be able to:*

- *Identify the Title Bar, Menu Line, and the different standard Toolbars*
- *Toggle "off/on" the standard toolbars*
- *Dock and float the different toolbars*

Upon initial startup, a default workspace will be loaded. This workspace will contain a default set of toolbars. The demonstration identifies the **Title Bar** and **Menu line**. It then goes on to introduce the 13 different standard **Toolbars**. The

standard toolbars are displayed by right mouse clicking anywhere in the grey area producing a "fly-out" which displays the toolbars and the "**Custom**" option. The display of each toolbar will be toggled "off/on" by left clicking on its name on the "fly-out" list. Each and every toolbar is toggled "off/on" showing its default location in the workspace and a brief explanation is given on the toolbar features. The process of "docking" and "floating" toolbars as with any other Windows program is also demonstrated.

V3-Workspaces: *After Viewing this video and applying the demonstration you will be able to:*

•   *Deselect the "Cool Look" option*
•   *Reconfigure the workspace*
•   *Save the new workspace*
•   *Select other workspaces that come with the program*

The demonstration shows how to rearrange the location of the toolbars and then save the workspace. It begins by selecting "Customize" option on the toolbar "fly-out" then turning off the "cool look" option. After rearranging some toolbars, the workspace is then saved. This is the same workspace (Training.wsp) you saved as instructed in the "Do This First" section of this study guide. There will be no need for you to save the workspace since it is the default. You can practice rearranging the workspace; just don't save it unless you want to save a new workspace arrangement. Other workspaces are opened to show the different workspaces that come with the program.

V4-Templates: *After Viewing this video and applying the demonstration you will be able to:*

•   *Open pre-existing templates*
•   *Set user preferences via **File, Properties***
•   *Configure template to be the default at startup*

This video demonstrates how to open pre-existing templates and where they are located. Changing user preferences (i.e. background color, units, etc.) and saving those changes as a template is also demonstrated. The video then shows the process of then designating the new template to be the default and startup template upon startup of the program.

V5-Custom Toolbars: *After Viewing this video and applying the demonstration you will be able to:*

•   *Create custom toolbars*
•   *Populate toolbars*
•   *Delete toolbars*

The demonstration is on how to create custom toolbars. **Right click,** select **Custom,** select **New** this creates a new toolbar after entering a new name. Select the **Commands** tab to display all the functions. Select a function which displays the sub-functions from which you can populate your toolbar using the click and drag method. The process of saving the workspace and making it the startup default workspace is again demonstrated as in the **V3** video. The process of deleting a toolbar in then demonstrated.

V6A-Intro to Macros: *After Viewing this video and applying the demonstration you will be able to:*

- *Define and create a "macro"*
- *Use the "record" and "pause" macro creation features*
- *Identify the three different file (.bmp, .txt, .cdl) created when making a macro*
- *Identify the three different types of macros: Startup, Root, and Immediate Mode*
- *Assign a keyboard shortcut to the macro*
- *Change the description of the macro*

This demonstration walks you through the process of creating a macro using **Tools, Options, Macro Record** or you can use the **Ctrl+Alt+J** keyboard shortcuts. In the process of creating a macro you are introduced to the three different types of macros available and the three different types of files created when you make a macro. You are then shown how to assign a keyboard shortcut to play the macro. Finally, you are shown how and where you can change the description and the graphics representation for the macro.

V6-Keyboard Shortcuts: *After Viewing this video and applying the demonstration you will be able to:*

- *Identify where the Keyboard files (.kbd) are located*
- *Create a keyboard shortcut for a function*
- *Save the new keyboard shortcuts to the keyboard file*
- *Save a new keyboard file*

This demonstration familiarizes you with where the keyboard files are located. You are then shown how to assign a keyboard shortcut to a function. (**Note:** If you have performed steps 7 and 8 in the "Do This First" section of this study guide you are now able to use the keyboard shortcut keys listed in the Appendix and demonstrated throughout these videos). The video refers to a "Word document containing all the hotkeys, it is now the "Keyboard Shortcut" document in the Appendix. To create a keyboard shortcut **Right Mouse Click,** select **Customize,** select the **Commands** tab, select a function. In the "Press New Shortcut field enter a key, then select the "Assign Key" button. Then select the "OK" button. To save as new or to update the present keyboard select **Tools, Options,** in the "Keyboard File" file select the "**Save"** button.

NAVIGATING THE VIEWPORT

V7-Zooming Options: *After Viewing this video and applying the demonstration you will be able to:*

•   *Perform zoom Autoscale using **Alt+A** hotkeys or toolbar or menus pull down option*
•   *Perform zoom Window using **Alt+W** hotkeys or toolbar or menus pull down option*
•   *Perform zoom Half scale using **Alt+H** hotkeys*
•   *Perform zoom Double scale using **Alt+D** hotkeys*
•   *Perform zoom dynamic using the mouse wheel*
•   *Perform zoom option via the menu pull down options*

This demonstration introduces the various methods for zooming in and out of your Viewport. Selecting **View,** on the menu bar will get you to the **Zoom** pull down and fly-out options. You are introduced to and encouraged to use the following hotkeys: **ALT+A** Autoscale, **Alt+W** zoom Window, **Alt+H** zoom Half Scale, **Alt+D** zoom Double scale. To be a productive user it is best to use the keyboard shortcut keys when performing common functions like zooming. You are encouraged to fine tune using the mouse wheel.

V8-Panning Options: *After Viewing this video and applying the demonstration you will be able to:*

•   *Perform pan using **Alt+P***
•   *Perform pan using the arrow keys*
•   *Perform pan option via the menu pull down options*

This demonstration introduces how to use the Pan option via the menu pull down and fly-out options. You are then shown how to pan using the **Alt+P** hotkeys. The cursor becomes the "center of Viewport" with a rectangle representing the boundaries of the Viewport. As you position the cursor in the Viewport that location then becomes the new center location for the Viewport. Finally you are introduced to the pan method using the keyboard arrow keys.

2D Basics

EDIT

V9-Deleting & Undeleting 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Delete entities via the menu pull down options*
- *Delete entities using the **Ctrl+Q** hotkeys*
- *Delete entities using the **Delete Multiple** tool on the default toolbar*
- *Delete entities using the keyboard **delete** key*
- *Undelete entities*
- *Delete using the Selection Menu options*
- *Right mouse click to "accept" or "enter" after a command*

This demonstration walks you through the different options for deleting entities (elements). You are shown how to using the menus pull down options by selecting **Edit, Delete** then selecting the various options. You are introduced and encouraged to use the **CTRL+Q** or **Delete** keyboard key to bring up the **Selection Menu**. The following options in the selection menu are shown: **Single, Window,** and **AllDsp.** You are also shown the selection menu sub choices under "window", these include: **All in, All Out, Part In,** and **Part out**. The different options are demonstrated in the Viewport. Finally you are introduced to the **Undo** option for undeleting the entities.

V10-Deleting & Undeleting 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Delete entities using the selection menu sub options*
- *Delete entities using the masking options*
- *Mask entities by type*

This demonstration continues from the previous video using the selection menu sub options. You are then introduced to the masking or filtering options under **AllDsp, By Type** options. When selected, the **Masking by Type** dialog screen is displayed. You are shown how to then select your entities by the various masking options presented by the dialog screen.

V11-Generic Move: *After Viewing this video and applying the demonstration you will be able to:*

- *Move single entities using the Generic Move tool*
- *Move single entities using the menu pull-down option*
- *Indicate Base Position using the "snap" (Universal Position) options*
- *Use the F keys for selecting the "snap" options*
- *Toggle the "Position snap" toolbar*

This demonstration introduces you to the generic move menu option for moving single entities. On the main menu select**, Edit, Move** options. You can also select the **Generic Move** tool. When prompted for the base position you are introduced to the "Position" or "Universal Position" options. Using the **F** keys is an option for selecting the snap options or mouse click on the selection. Snapping to the end and center of the entities is demonstrated. Emphasis is placed on selecting **cursor** if you wish to have the "Position Snap" toolbar toggle on.

V12-Generic Edit: *After Viewing this video and applying the demonstration you will be able to:*

- *Edit geometry of single entities using menu pull-downs or the **Generic Edit** tool*
- *Edit attributes of single entities using menu pull-downs or the **Generic Edit** tool*

This demonstration uses the following methods for editing single entities: On the main menu select **Edit, Entities,** and **Edit** or select the **Edit** tool on the default toolbar. Select the entity you wish to edit then a dialog screen displays two tabs allowing you to edit either the geometry or the attributes. Experiment with the different options to become more familiar with the options.

CREATE LINES

V13-Snap Options: *After Viewing this video and applying the demonstration you will be able to:*

- *Indicate Base Position using the "snap" (Universal Position) options*
- *Use the F keys for selecting the "snap" options*
- *Toggle the "Position snap" toolbar*
- *Use the "Position snap" toolbar*

This demonstration gives an overview of the variety of "position" options available when prompted to indicate the start position when creating geometry. The Position snap toolbar will be the default if toggled to "on" and if the snap options located above the conversation bar is set to **Cursor**.

V14-Enpoints, String and Polygon: *After Viewing this video and applying the demonstration you will be able to:*

•   *Create lines using the Modeling Toolbar*
•   *Create lines Endpoints, String, and Polygon using the **L** hotkey*
•   *Create polygons using the different size options*

This demonstration introduces the create line options by either selecting the **Create, Line** buttons in the Modeling toolbar or the **L** hotkey. The sub-menu options will appear wherever the cursor is in the Viewport. The create line command is shown how it works in conjunction with position options. The five different size options are explored when prompted for a size during creation of a polygon.

V15-Parallel through a Pt, Parallel at a Distance, At a Specified Angle: *After Viewing this video and applying the demonstration you will be able to:*

•   *Create parallel lines through a point*
•   *Create parallel lines at a distance using the **P** hotkey*
•   *Create a line at an angle to another line*

This demonstration explains the two methods for creating parallel lines. You can either create a parallel line through a point (i.e. center of a circle) or at a distance from the reference line. Using the **P** hotkey is used for the create line parallel at a distance. There are a variety of ways to create a line at a specific angle to an existing line, however, using the create line **angle** command requires that you become aware of the importance of where the reference line is selected. Much of this demonstration covers this topic.

V16-Horizontal, Vertical, Both: *After Viewing this video and applying the demonstration you will be able to:*

*Create horizontal lines using the **H** hotkey*
*Create vertical lines using the **V** hotkey*
*Create both vertical and horizontal lines using the create line **Both** command*

This demonstration introduces the tools for creating horizontal, vertical, and both types of lines using the **L** line command then selecting the options on the third row of the line menu. Horizontal lines can be created using the **H** hotkey. Vertical lines can be created using the **V** hotkey. Emphasis is placed on the fact that any of these options will only create the line to the extent of the Viewport.

V17-Tan to 2 Entities, Tan through a Point, Tan to 1st, Perp to 2nd: *After Viewing this video and applying the demonstration you will be able to:*

•   *Create a line tangent to two entities using the **T** hotkey*

- *Create a line tangent to an entity and through a specified point*
- *Create a line perpendicular to the first entity and tangent to the second entity*

This demonstration introduces the fourth row of create line tools. These will allow you to create lines tangent to two entities, you can also use the **T** hotkey. You can create lines tangent to the first entity and through a specified point and perpendicular to the first entity and tangent to the second entity. By experimenting you will see that the tangent point is at the approximate location of the cursor.

V18-Perpendicular to 2 Entities, Perp through a Point, Perp to 1st, tangent to 2nd: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a line perpendicular to 2 entities*
- *Create a line perpendicular to an entity and through a specified point*
- *Create a line perpendicular to the first entity and tangent to the second entity*

This demonstration introduces the fifth row of the create line tools. These will allow you to create lines perpendicular to 2 entities. You can create lines perpendicular to an entity and through a specified point. You can create lines perpendicular to the first entity and tangent to the second entity.

CREATE CIRCLES

V19-Center & Radius, Center & Diameter, Center & Edge, and Three Point: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a circle by specifying the center location and a radius*
- *Create a circle by specifying the center location and a diameter*
- *Create a circle by specifying the center location and an edge location for size*
- *Create a circle by locating 3 points on the circumference*

This demonstration introduces the create circle command by selecting the **Create, Circle** tool on the modeling toolbar or using the **C** hotkey. The first four options are demonstrated including the four listed above. After viewing the video these options are pretty self explanatory. Specifying the center and edge determines the size of the circle. Creating a circle using the 3 points determines the size of the circle.

V20-2 Tangents, 3 Tangents, Center & Tangent, and Diameter 2 Points: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a circle tangent to 2 entities*
- *Create a circle tangent to 3 entities*
- *Create a circle specifying the center point and tangent to an entity*
- *Create a circle specifying 2 points*

This demonstration uses the create circle command by selecting the **Create, Circle** tool on the modeling toolbar or using the **C** hotkey. The last four options are demonstrated including the four listed above. Create a circle tangent to 2 entities will prompt for a radius and will not create the circle if the radius is too small. The other options will determine the size of the circle based on the entities selected.

CREATE ARCS

V21-Center and Radius, Center and Diameter, Center and Edge, 3 Positions: *After Viewing this video and applying the demonstration you will be able to:*

- *Create an arc specifying the center point and the radius*
- *Create an arc specifying the center point and the radius*
- *Create an arc specifying the center point and an edge location*
- *Create an arc specifying 3 positions*

This demonstration uses the create arc command by selecting the **Create, Arc** tool on the modeling toolbar or using the **A** hotkey. Creating an arc using the first three options listed above will prompt for a beginning angle, end angle, and a center point. A positive angle for both will create the arc in a counterclockwise direction. The size is determined by the radius or diameter entered and by the location of the edge point. The size and location for the fourth option will determine the center location and the size.

V22-2 Tangents, 3 Tangents, Center and Tangent, Tangent & End Pt: *After Viewing this video and applying the demonstration you will be able to:*

- *Create an arc tangent to 2 entities*
- *Create an arc tangent to 3 entities*
- *Create an arc specifying the center point and tangent to an entity*
- *Create an arc tangent to an entity and specifying the end point*

This demonstration continues with the **Create, Arc**, tool on the modeling toolbar or using the **A** hotkey. Creating an arc tangent to 2 or 3 entities prompts for the beginning and end angles and the radius of the arc. When creating an arc specifying the center point and the tangent entity the size is determined by the location of the center point. Creating an arc tangent to an entity and specifying the end point will determine the size.

V23-Tan, Start and End, Radius and 2 Positions, Offset Distance, offset Position: *After Viewing this video and applying the demonstration you will be able to:*

- *Create an arc by specifying the beginning and ending position and the included angle*
- *Create an arc by specifying a radius and 2 positions*
- *Create an arc by offsetting from an existing arc a specific distance*
- *Create an arc by offsetting from an existing arc through a point*
- *Verify angle between two lines*

This demonstration continues with the **Create, Arc**, tool on the modeling toolbar or using the **A** hotkey. Creating an arc by specifying the beginning and ending position and included angle determines the size by the points picked. How to verify the angle between two lines is also demonstrated. The size of the arc and the beginning and ending locations are determined by the arc being offset from.

CREATE RECTANGLES

V24-Line Corners, Line Width & Height: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a rectangle by specifying the corners*
- *Create a rectangle by entering the width and height*
- *Create a rectangle using polyline*

This demonstration uses the create rectangle command by selecting **Create, Rectangle** tool on the modeling toolbar or using the **R** hotkey. Creating a rectangle specifying the corners determines the size and the location is determined by the two points picked. Creating a rectangle by entering the width and height determines the size but the location is determined by where the anchor point is picked. There are 9 options for anchor point of the rectangle. You have the same options for creating rectangles using polyline, however, the polyline rectangles are single object instead of 4 different lines.

V25-Rounded Line Corners, Rounded Line Width & Height: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a rectangle with rounded corners by specifying the corner radius and the corners*
- *Create a rectangle with rounded corners by specifying the corner radius, width, and height*

This demonstration continues with the **Create, Rectangle** command on the modeling toolbar or using the hotkey **R** to create rectangles with rounded corners. Creating a rectangle with rounded corners by specifying the corner radius and the corners will dynamically display the results. Creating a rectangle with rounded corners by specifying the corner radius, width, and height will require the location of the anchor point.

MODIFY FILLETS

V26-Fillet and Trim, Fillet and No Trim: *After Viewing this video and applying the demonstration you will be able to:*

- *Create fillets with trim function*
- *Create fillets with no trim function*
- *Create a fillet using the quick trim tool*
- *Create a chamfer with trim function*
- *Create a chamfer with no trim function*

This demonstration uses the create fillet command by selecting **Modify, Fillet**, command on the modeling toolbar. Select either the **Trim** or the **No Trim** option. Specify the radius then select the two entities the fillet will be tangent to. Activating the **Quick Trim** tool on the switches toolbar will allow you to pick the fillet corner, not the individual tangent entities. Use the create chamfer command by selecting **Modify, Chamfer** command on the modeling toolbar. Select either the **Trim** or **No Trim** option. Specify distances for each leg to create the chamfer.

TRIM EXTEND

V27-Trim First and Trim Both: *After Viewing this video and applying the demonstration you will be able to:*

- *Trim or extend an entity to another entity at their intersection*
- *Trim two entities at their intersection*

This demonstration uses the trim first command by selecting **Modify, Trim First** command on the modeling toolbar or use the **F** hotkey. The **Trim First** and the **Trim Both** commands will either trim or extend the entities at their intersections.

V28-Trim Divide and Trim Double: *After Viewing this video and applying the demonstration you will be able to:*

- *Trim an entity to two other entities at their intersection, removing the portion between the trimming entities*
- *Trim an entity to two other entities at their intersection, leaving the portion between the trimming entities*

This demonstration uses the trim divide command by selecting **Modify, Trim Divide** command on the modeling toolbar. Or use the **Shift+D** hotkey. Select the entity crossing two entities you wish to divide out between those two trimming entities, and then select the two trimming entities, this will remove or divide the first entity selected. Use the **Modify, Trim double** command on the modeling toolbar or use the **D** hotkey. Select the entity crossing two other entities that you wish to trim off at the intersection of those two trimming entities. You must use

the trim double command for circles. Once the circle is no longer closed (arc) you would then treat it as you would a line.

V29-Trim Multiple and Trim Position: *After Viewing this video and applying the demonstration you will be able to:*

- *Trim multiple entities to one entity*
- *Trim or extend an entity to a position*

This demonstration uses the trim multiple command by selecting **Modify, Trim Multiple** command on the modeling toolbar. Using this trim function requires that you select the trimming entity first, and then select the entities you wish to trim that intersect the trimming entity. Use the trim to position command by selecting **Modify, Trim to Position** command on the modeling toolbar or use the **Shift+P** hotkey. Using this command allow you to trim or extend an entity to a position.

XFORM

V30-Delta Move, Copy, and Join: *After Viewing this video and applying the demonstration you will be able to:*

- *Transform entity location by moving using the X and Y coordinates*
- *Create a copy of an entity or entities using the X and Y coordinate*
- *Transform an entity or entities by creating a copy and joining the ends.*

This demonstration uses the Xform delta command by selecting **Xform, Delta** command on the modeling toolbar. This command gives three options for transforming entities using the X and Y Cartesian Coordinate system. These options are move, copy, or join. Use the selection menu for selecting entities and the position menu for base point and new location.

V31-Old New Move, Copy, Join: *After Viewing this video and applying the demonstration you will be able to:*

- *Transform move location of entities*
- *Transform copy entities*
- *Transform copy and Join entities*
- *Using mask option for selecting entities*

This demonstration uses the Xform old new command by selecting the **Xform, Old to New** command on the modeling toolbar or use the **M** hotkey. Use the selection menu for selecting the entities. You are prompted for the base point, follow up with **Skip** or **Enter** for the second question, then select the new position using the position menu choices. Transform **Copy** works the same as **Move** only the original entities remain. Transform **Join** works the same as **Copy** with the addition of lines joining the ends of the entities.

V32-Rotate Move, Copy, Join: *After Viewing this video and applying the demonstration you will be able to:*

- *Transform rotate entities clockwise or counterclockwise*
- *Transform rotate and copy entities clockwise or counterclockwise*
- *Transform rotate, copy, and join entities clockwise or counterclockwise*
- *Transform and rotate entities using Circular Array*

This demonstration uses the Xform rotate command by selecting the **Xform, Rotate** command on the modeling toolbar. Use the selection menu for selecting the entities. You are prompted for the "first position" on axis, select a position, then for the second prompting "second position" on axis select **Enter**. Enter an angle of rotation (positive angles are counterclockwise, negative angles are clockwise). You are prompted for number of copies; enter the amount the select **Enter** to accept. You can perform the same steps for the **Rotate Copy** and **Rotate, Copy, Join**. Use the **Xform, Circular Array** command on the modeling toolbar for performing a rotate copy operation without rotating the resulted copies as they are arrayed about the axis, the orientation remains the same.

V33-Mirror Move, Copy, Join: *After Viewing this video and applying the demonstration you will be able to:*

- *Move entities about a mirror plane*
- *Create a mirror copy of entities about a mirror plane*
- *Create a mirror copy of entities about a mirror plane and joining the end of the entities*

This demonstration uses the Xform mirror command by selecting the **Xform, Mirror** command on the modeling toolbar. Select from the **Move, Copy,** or **Join** options. Use the selection menu for selecting the entities. You are prompted to select a type of mirror plane then position it using the position menu. The result is either a move, copied, or joined result depending on which sub-menu choice you selected.

LEVELS

V34-Level Introduction: *After Viewing this video and applying the demonstration you will be able to:*

- *Display the levels*
- *Use the Toggle Splitter to toggle the level display*
- *List the level properties*
- *Activate a level*
- *Create parent levels and sub-levels*
- *Reorder the levels*

- *Rename levels*

*Toggle the level display using the **Alt+Right Arrow** and **Alt+Left Arrow** keys*
This demonstration introduces the levels. The level screen is displayed via the **Toggle Splitter** tool or using the **Alt+Right Arrow** keys. Depressing the **Alt+Right Arrow** key again will expand the level display to include all the columns. To collapse and remove the display use the **Alt+Left Arrow** keys. You are introduced to the different column headings and shown how to create parent levels and sub levels, reorder the level display and activate the levels.

V35-Setting up Levels: *After Viewing this video and applying the demonstration you will be able to:*

- *Create levels*
- *Rename levels*
- *Expand level display using the splitter bar*
- *Assign attributes to levels*
- *Set attributes by level*

This demonstration continues with the level functions. You are shown how to create levels, rename levels, and then assign attributes to levels so that the entities reflect the attributes assigned to the level the geometry is created on.

V36-Entity Transfer: *After Viewing this video and applying the demonstration you will be able to:*

- *Transfer entities to other levels using the **Ctrl+B** hotkeys*
- *Toggle the level display*
- *Change attributes via the **Shift+A** hotkeys or Change Attribute Select tool*
- *Transfer entities to other levels via the Change Attributes" dialog screen*
- *Use mask option for selecting entities*

This demonstration continues with the level functions. You are shown how to transfer entities to different levels using the **Ctrl+B** hotkeys. Entities do not take on the attributes of the new level. Selecting the **Shift+A** hotkeys or the **Change Attributes Select** tool invokes the "Change Attributes" dialog screen. This "wish list" is where you select the attributes that you wish to change to. Toggling the display of levels displays what is on each level.

V37-Level Display: *After Viewing this video and applying the demonstration you will be able to:*

- *Minimize the level display using the splitter bar*
- *Apply transparency to solid bodies*
- *Toggle the level display*
- *Spin the model in the Viewport via the Spin tool*

This demonstration continues with the level functions. You are shown how to minimize the level display using the splitter bar then toggle "off/on" the display of each level. The process of changing the transparency of a model is introduced using the **Face, Change Transparency** tools in the Advanced Modeling toolbars. The **Spin** tool is used for revolving the model in the display. **Use the file shown below to follow along with the video.**
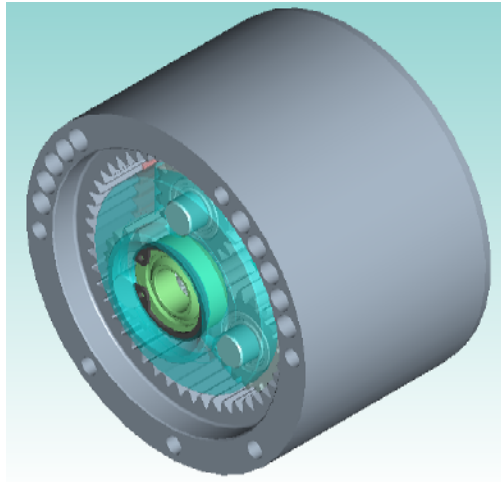
Open **levels sample.ckd** located in **C:\DEMO\3DModeling**

ATTRIBUTES

V38-Setting Attributes: *After Viewing this video and applying the demonstration you will be able to:*

Set attributes using the Set Attributes tool
*Set attributes using the **Alt+X** hotkeys*

This demonstration introduces two methods for setting the attributes for creating geometry. The two methods are using the **Set Attribute** tool or using the **Alt+X** hotkeys. This will bring up the **System Attributes** dialog screen. Select the attributes you wish to use for creating geometry.

V39-Changing Attributes: *After Viewing this video and applying the demonstration you will be able to:*

- *Change attributes using the menu*
- *Change attributes using the Change Attributes tool*
- *Change attributes using the **Ctrl+R** or **Shift+A** hotkeys*

This demonstration introduces three methods for changing attributes on existing entities. The three methods are: From the menu line select **Edit, Entities,** and **Attributes by Selection**. You can also use the hotkeys **Ctrl+R** or **Shift+A**. These options will bring up the **Change Attributes** dialog screen referred to as the "wish list". Select the attributes you want to change to, and then select the geometry you wish to take on the attributes selected from the dialog screen.

IMPORTING BORDERS & DRAWINGS

V40-Importing Borders 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Import a border into existing file*
- *Preview files before importing*
- *Select importing options*
- *Select option from "Units Mismatch" dialog screen*
- *Import into root level*
- *Import into active level*
- *Group the imported file*
- *Using the "Key in" option for placing at the origin*

This demonstration introduces importing a border into the existing file. On the menu line select **File, Import, and keyCreator file** this will bring up the "select file to import" dialog screen. Make sure the Preview button is depressed. The default "look in" is the "CKD" folder, select the "Borders" folder, select the border file, and then select the **Options** button. The different options are explored by importing the border using the various options. Once you have selected the preferred options select **Open** to bring the file in. Position the "base" position at the "origin" using the "key in" option. Get in the habit of "grouping" the imported file.

V41-Importing Borders 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Move entities by masking*
- *Move entities by group*
- *Use levels to mask entities*

This demonstration goes through the process of importing a border into the active design file then showing the different ways of masking for the geometry to be moved.

V42-Merging Drawings 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Edit text using Generic Edit tool*

- *Merge design files using Cut & Paste Windows function*
- *Display all open design files using the Window option*
- *Reorganize the level column heading*
- *Change the size of the level column display*

This demonstration begins with editing and re-formatting text using the **Generic Edit** tool. Using the **Window** command on the menu line you are shown how to display the open files using several methods. Select **Tile Vertical** to display the files side by side. Use the **Cut** and **Paste** tools on the default toolbar to merge geometry from one file with the other. Reorganize the level column heading and change the column widths. The demonstration then demonstrates the process of changing the attributes and verifying entity levels by toggling their display.

V43-Merging Drawings 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Display all open design files using the Tree Window tool*
- *Expand the Tree Window display to view the design file path*
- *Merge design file geometry using Cut & Paste Windows function*

This demonstration introduces the Tree Window option for displaying open files. Right click in the grey area, select **Tree Window**; this will display all the open design files, the one with the red check mark is the active file. Right click on the "part" block of the design file you want to activate, this will display the options. Select **Edit Part**, the file becomes the active file. Use the **Cut** & **Paste** tools on the default toolbar for merging the geometry. Finally you are shown how to save the file then expanding the Tree Window display to view the file path.

CREATING TEXT

V44-Creating Text: *After Viewing this video and applying the demonstration you will be able to:*

- *Set Active level using the **Alt+L** hotkeys*
- *Create Text using Detail, Notes, and Key-in options in the Modeling Toolbar or from the menu line*

This demonstration introduces the process for creating text. Use **Alt+L** hotkeys to activate the level you wish to work on, and then enter that level number when prompted. To create text, select **Detail**, **Notes,** and **Key-In** options from the menu line or in the Modeling Toolbar. This brings up the "Create a New Note" dialog screen. Select the **Format** tab, set your format preferences then select the **Text** tab to enter the text. Position the text in the drawing.

SAVING A FILE

V45-Saving a File: *After Viewing this video and applying the demonstration you will be able to:*

•   *Save a design file (part file) using the **Ctrl+F** hotkeys*
•   *Save the file using the save button on the default toolbar*
•   *Enable "Automatic Time Save"*
•   *Save different generations of the file*

This demonstration shows how to save a file by selecting **File, Save As** on the menu line when saving for the first time, or, use **Ctrl+F** hotkeys. This brings up the "Save As" dialog screen. Select the location in the "Save In" field, then enter the name of the file in the "File Name" field, and then select the **Save** button. Once the file has been save for the first time, you can select the **Save** button on the default toolbar to continually update the file as you work on it. To enable the "Automatic Time Save", select **Tools, Options,** and then select the **Saving** tab. Select the option preferences then select **OK**.

PRINTING

V46-Printing: *After Viewing this video and applying the demonstration you will be able to:*

•   *Modify the Printer Configuration*
•   *Preview the print*

This demonstration introduces printing by first setting the printer configuration. Select **Tools, Options,** select the **Print Plot** tab, and then select the **Modify Configuration** button. Set configuration preferences. Select the **Color** tab and select **Black**. Select the **Width Scale** tab then set the preference. Select **OK** to accept the changes and exit the print plot configuration screen. Select **File**, **Print Preview** select **Print**, this will bring you to the **Print/Plot Setup** dialog screen. Set preferences then select **OK**. This will bring you back to the drawing area with a rectangle representing the paper centered about the cursor. Select **Skip** or **Enter** on the keyboard to center on your drawing. This will display the print preview.

2D Construction Techniques

Do the following before you open the practice drawings listed below: Select Tools, Options, select the Dimension tab then uncheck the last two boxes

2D Basics Construction Techniques

V50-Geometric Construction 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Toggle "off/on" grid and snap display*
- *Setting the grid and snap options*
- *Create a drawing using the grid and snap*

This demonstration is on using the grid and snap functions for creating geometry. You can set the grid and snap by selecting **View, Grid and Snap, Set Grid and Snap** to bring up the "Grid Snap Dialog" screen where you can set or change the settings. **Ctrl+G** hotkeys will also bring up the dialog screen. Select the **Grid** and **Snap** tools on the "Switches Toolbar" to toggle "off/on" the grid and snap display. **Use the file shown below to follow along with the video.**
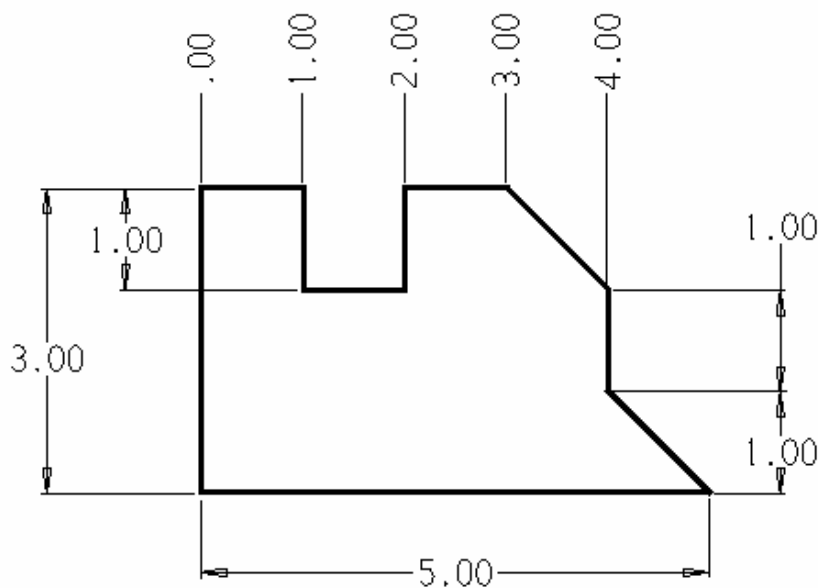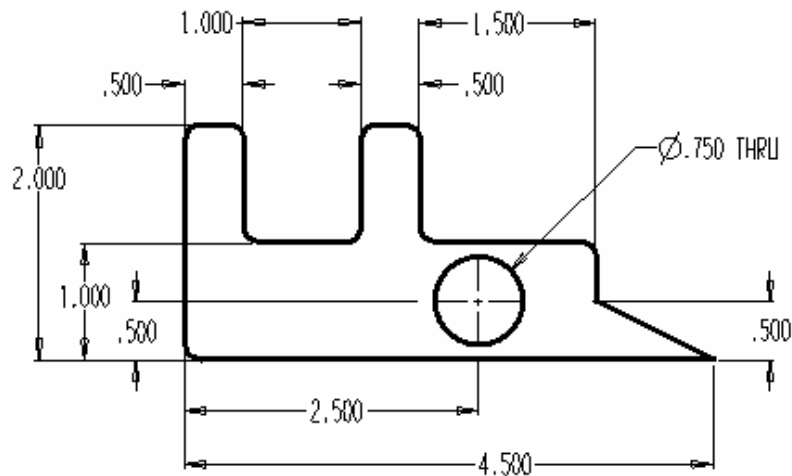
2D BASICS
PRACTICE PROBLEM NO. 1

1. Open **PRAC-1B.ckd** located in**: C:\DEMO\2D Construction**
2. Import **PRAC-1-KEY.ckd** located in: **C:\DEMO\2D Construction**
3. Match the "key" to your drawing

V51-Geometric Construction 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a drawing using the create line string command*
- *Create fillets*
- *Create circles*
- *Create centerlines*

This demonstration show how to create a drawing using the **create line string** command and using the **KeyIn** option from the position menu to enter the end locations for the X and Y coordinates. This creates the basic profile of the part, a circle and fillets are then added to complete the drawing visible lines. Centerlines are then created using the **Detail, Detail Lines,** tools from the modeling toolbar, then set the preferences and select the arc or circle to apply the centerlines to. **Use the file shown below to follow along with the video.**

1.000 ⊢— 1.500 —⊣
.500⊣
.500
2.000
⌀.750 THRU
1.000
.500
.500
2.500
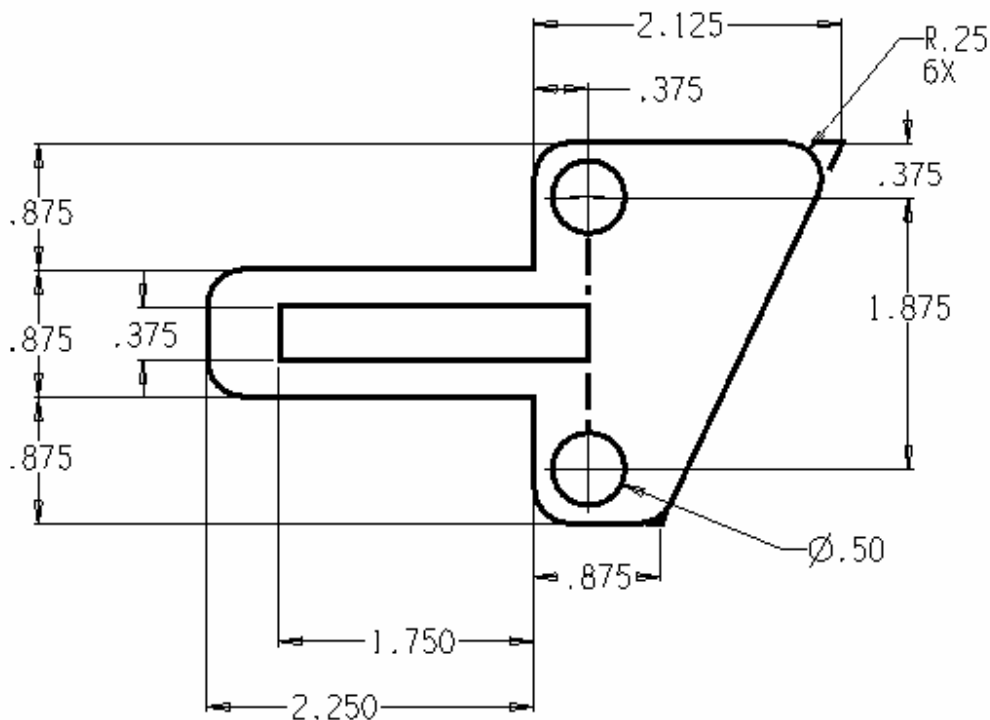4.500

NOTES:
1. ALL ROUNDS AND FILLETS R.125

2D BASICS
PRACTICE PROBLEM 2

1. Open **PRAC-2B.ckd** located in**: C:\DEMO\2D Construction**
2. Import **PRAC-2-KEY.ckd** located in: **C:\DEMO\2D Construction**
3. Match the "key" to your drawing

V52-Geometric Construction 3: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a drawing beginning with construction lines*
• *Use trimming functions to "clean up" construction lines*
• *Create slotted feature using arc and mirror join*

This demonstration develops a drawing using construction lines for the general shape of the part, then trimming away the excess using various trim/extend functions. You then add the slotted feature using the arc and mirror functions. Finally the circle and centerlines are added for completion. **Use the file shown below to follow along with the video.**

3.000
2.375
1.625
.750
.500
2.250
1.625
1.125
1.000
.500
.625
R .250
Ø .500
.875

2D BASICS
PRACTICE PROBLEM NO.3

1. Open **PRAC-3B.ckd** located in**: C:\DEMO\2D Construction**
2. Import **PRAC-3-KEY.ckd** located in: **C:\DEMO\2D Construction**
3. Match the "key" to your drawing

V53-Geometric Construction 4: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a drawing beginning with horizontal and vertical construction lines*
- *Developing additional construction lines using parallel function*
- *Use trimming function to "clean up" construction lines.*

This demonstration develops a drawing beginning with horizontal and vertical construction lines then offsetting additional lines with the parallel function for the general shape of the part. Finishing off by trimming away the excess using various trim/extend functions. The fillets, holes, and centerlines are then added to complete the drawing. **Use the file shown below to follow along with the video.**
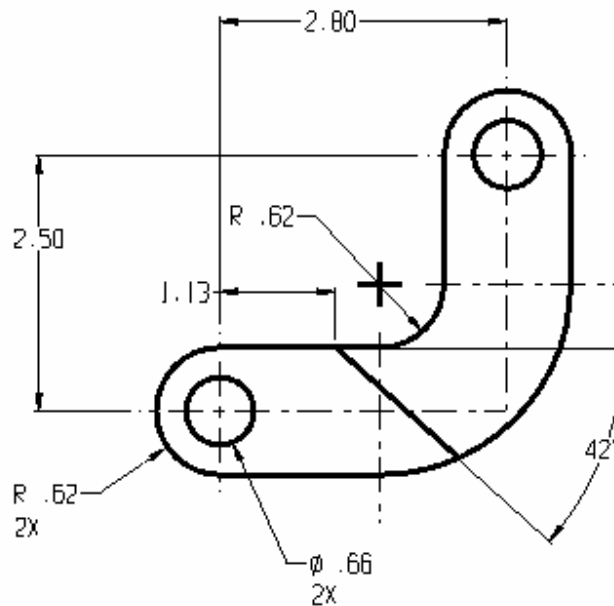
2D BASICS
PRACTICE PROBLEM NO.4

1. Open **PRAC-4B.ckd** located in**: C:\DEMO\2D Construction**
2. Import **PRAC-4-KEY.ckd** located in: **C:\DEMO\2D Construction**
3. Match the "key" to your drawing

V54-Geometric Construction 5: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a drawing beginning with rectangle*
- *Create a slot using the create rectangle with rounded corner*
- *Create an angled line using Xform rotate*
- *Break entities using the break function*

This demonstration develops a drawing beginning with a rectangle, then creating a slotted feature using create **rounded corner rectangle**. The break first function is introduced using the **B** hotkey. The angle line is created using **Xform, Rotate**, and **Move**. The angle line is again created using the **Offset, Polar** command from the position menu. The right side view is then added using rectangle. The drawing is finalized by adding the hidden feature in the right side view. **Use the file shown below to follow along with the video.**
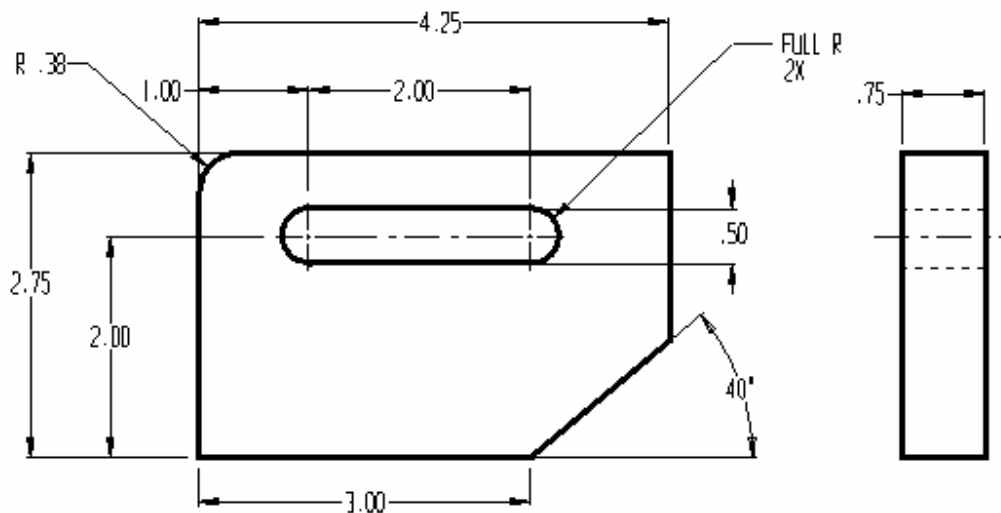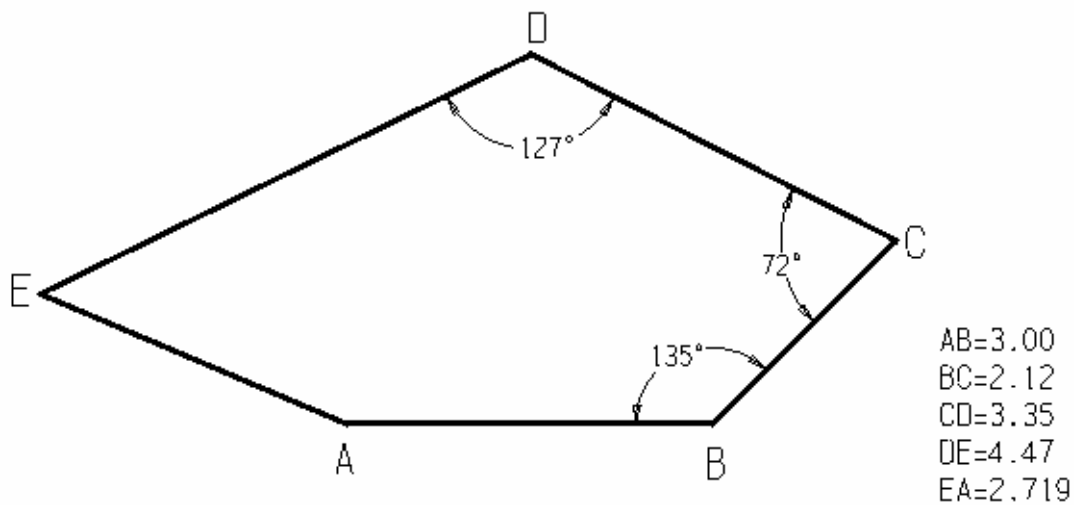
KEYCREATOR 2D BASICS
PRACTICE PROBLEM NO.5

1. Open **PRAC-5B.ckd** located in**: C:\DEMO\2D Construction**
2. Import **PRAC-5-KEY.ckd** located in: **C:\DEMO\2D Construction**
3. Match the "key" to your drawing

V55-Geometric Construction: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a drawing using 5 different methods*

This demonstration creates the drawing using 5 different methods to give a better understanding of how you can combine different tools to come up with the same drawing. The following hotkeys are used: **J** for Xform Copy Rotate, **X** for Xform Rotate, **T** for Tangent, **D** for Trim Double, **F** for Trim First, and **Alt+L** for Level Set. **Use the file shown below to follow along with the video.**

2D BASICS
PRACTICE PROBLEM NO. 6

1. Open **PRAC-6B.ckd** located in**: C:\DEMO\2D Construction**
2. Import **PRAC-6-KEY.ckd** located in: **C:\DEMO\2D Construction**
3. Match the "key" to your drawing

V56-Geometric Construction 7: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a drawing of a closed traverse*
• *Utilize Xform Rotate in conjunction with AlongE position menu to close a traverse*
• *Utilize the Offset Polar command to create a closed traverse*

This demonstration show two ways for creating a closed traverse. The first utilizes the combination of **Xform, Rotate** commands from the modeling toolbar and **AlongE** from the position menu to create the drawing. The second method uses the **Offset, Polar** commands from the position menu to create the drawing. Each method shows the advantage of using the **Grid** and **Snap** options. **Use the file shown below to follow along with the video.**

2D BASICS
PRACTICE PROBLEM 7

1. Open **PRAC-7B.ckd** located in**: C:\DEMO\2D Construction**
2. Import **PRAC-7-KEY.ckd** located in: **C:\DEMO\2D Construction**
3. Match the "key" to your drawing

V57-Geometric Construction 8: *After Viewing this video and applying the demonstration you will be able to:*

• *Use verify to measure perimeter, area, distance, etc.*
• *Use the calculator*
• *Create crosshatching*

This demonstration uses the **Tools, Verify**, commands on the menu line to extract information from a drawing. Use **Ctrl+I** command to bring up the calculator. Finally you are shown the various ways to create and edit crosshatching. In the last example for selecting entities to cross hatch, you can also use the **Window, Part in**, command to select the boundary for hatching. **Use the file in the previous video (Practice Problem 7b.ckd) to follow along with the video.**

Open **PRAC-7B.ckd** located in**: C:\DEMO\2D Construction**

Dimensioning and Annotation

Dimensioning and Annotation

V58-Basic Dimensioning 1: *After Viewing this video and applying the demonstration you will be able to:*

Set attachment and associativety
Create associative and non associative dimensions
Box move geometry

This demonstration introduces you to setting the attachment and associativety of dimensions to geometry. Use **Tools**, **Options,** and then select the **Dimension** tab, select preferences then select **Ok.** Create dimension using **Detail** on the menu line, this presents pull-down options, or, select **Detail** on the modeling toolbar for the sub-menu choices. Box move is using **Xform, Box Move,** then select geometry to drag, thereby stretching or shrinking the drawing. Associative and non-associative dimensions are then created. **Use the file shown below to follow along with the video.**
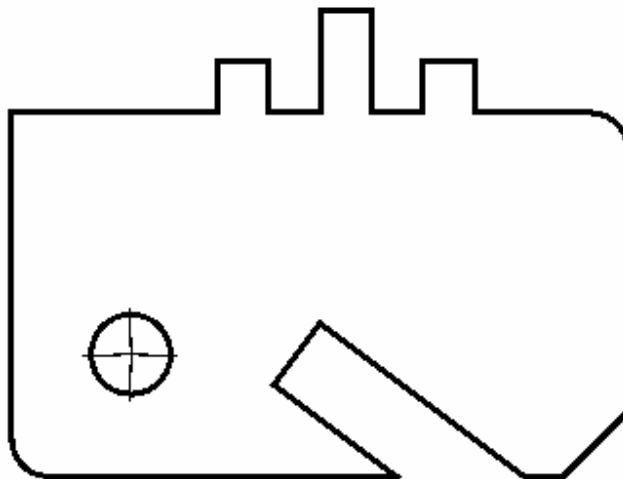
Open **Dimensioning.ckd** located in**: C:\DEMO\2D Construction**

V59-Basic Dimensioning 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Set dimension attributes*
- *Apply dimensions to a drawing*

This demonstration introduces the detail settings using **Detail, Detail settings, Set Attributes** commands on the modeling toolbar. Dimensions are then applied to the drawing using **Detail,** then selecting the different dimensioning tools. **Use the drawing from the previous video (Dimensioning.ckd) to follow along with the video.**

V60-Dimension Edit: *After Viewing this video and applying the demonstration you will be able to:*

- *Change dimension attributes*
- *Re-associate dimensions*
- *Turn off display of disassociated dimensions*
- *Update dimensions*

This demonstration introduces the detail dimension change attribute tool by selecting **Detail, Detail Settings,** and **Change Attribute** commands on the modeling toolbar use the **Generic Edit** tool for single editing. **Use the file in the previous video (Dimensioning.ckd) after applying the dimensions, to follow along with the video.**
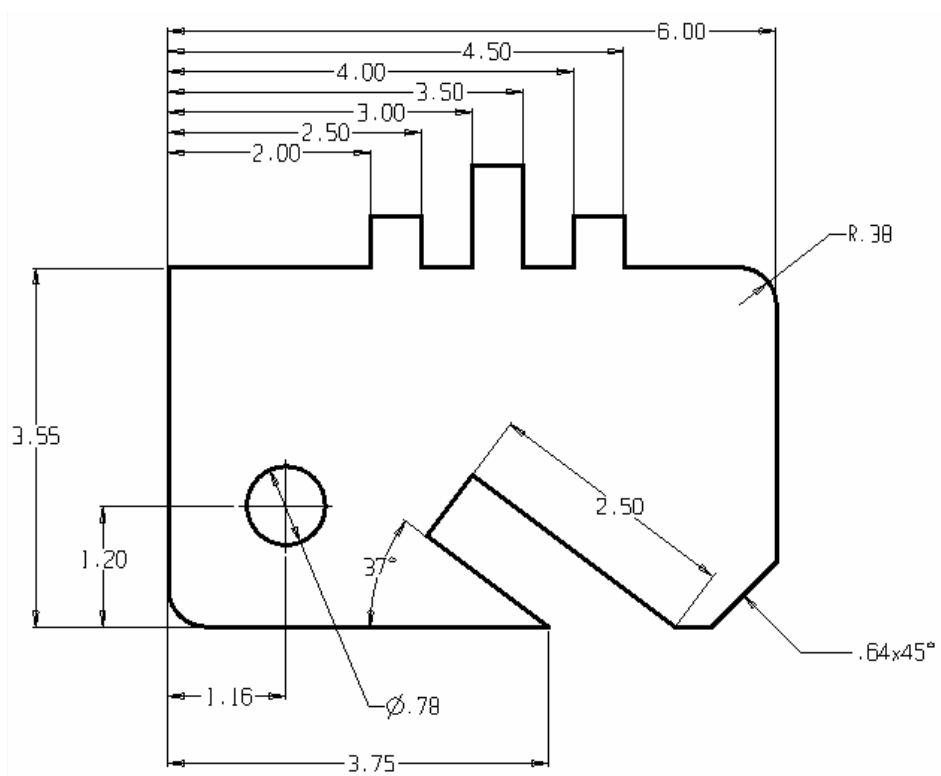
(Dimensioned results)

V61-Ordinate Dimensioning: *After Viewing this video and applying the demonstration you will be able to:*

- *Delete dimensions*
- *Create ordinate dimensioning*
- *Align and offset ordinate dimensions*

This demonstration illustrates how to delete dimension then goes on to create ordinate dimensions using **Detail, Baselinear Dim** in the modeling toolbar, this displays the ordinate dimensioning options. **Use the file in the previous video (Dimensioning.ckd) to follow along with the video.**

V62-Basic Dimensioning 3: *After Viewing this video and applying the demonstration you will be able to:*

- *Create running dimensions*
- *Create chain dimensions*
- *Create offset dimensions*
- *Align existing dimension*
- *Create parallel linear dimensions*
- *Create coordinate dimension*

This demonstration introduces running dimensions found in **Detail, Baselinear Dim** command in the modeling toolbar, this displays the **Running** options. Chain dimensioning is also used via **Detail, Linear Dimension** command in the modeling toolbar, this displays the **Chain Dimension** options. Offset and aligned dimensioning is also introduced by selecting **Detail, Linear Dimension** command in the modeling toolbar, this displays the **Offset** and **Aligned** options. To create parallel aligned dimensioning select **Detail, Linear Dimension,** And **Parallel Linear**. Finally, to create coordinate dimensioning select **Detail, Coordinate Dimension**, this brings up the dialog box to enter preferences. **Use the file in the previous video (Dimensioning.ckd) to follow along with the video.**

V63-Notes and Symbols: *After Viewing this video and applying the demonstration you will be able to:*

- *Create detail lines*
- *Create local notes (labels)*
- *Move local notes*
- *Create balloon notes*
- *Create a note linked to a file*

This demonstration shows how to create detail lines by selecting **Detail, Detail Lines,** then select from the four options shown. Creating a local note and balloons is done by selecting **Detail, Note,** then choosing the **Create Label** or

Create Balloon commands. **Use the file in the previous video (Dimensioning.ckd) to follow along with the video.**

V64-Geometry from Notes: *After Viewing this video and applying the demonstration you will be able to:*

- *Create note using symbols*
- *Create geometry from note*

This demonstration creates a local note using with inserted symbols. Geometry is then created from an existing note by using the **Burst** command on the default toolbar. The demonstration goes on to create a solid model of the geometry by extrusion in the isometric view. **Use the file in the previous video (Dimensioning.ckd) to follow along with the video.**

V65-Auto Dimensioning 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Set horizontal auto dimension configuration*
- *Set vertical auto dimension configuration*
- *Set arc auto dimension configuration*
- *Use the mode and selection tab options*
- *Apply auto dimension to a drawing*

This demonstration sets different configurations for horizontal, vertical and arcs dimensions in conjunction with the **Mode and Selection** tab options. Use **Detail, Auto Dim, Configure Auto Dim** commands from the modeling toolbar. The video offers various dimensioned results from the different configuration settings. **Use the file shown below (Auto Dim.ckd) to follow along with the video**
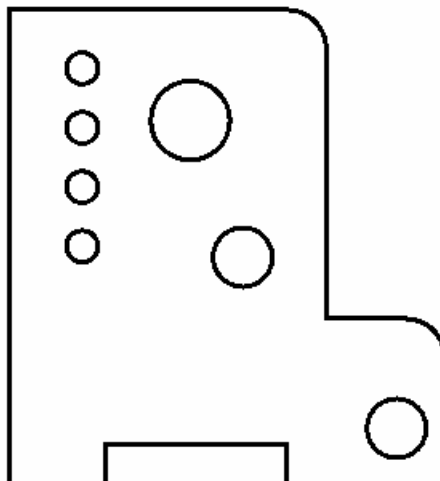
Open **Auto Dim.ckd** located in**: C:\DEMO\2D Modeling**

V66-Auto Dimensioning 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Set circle auto dimension configuration*
- *Set hole table auto dimension configuration*
- *Apply auto dimensioning to a drawing*

This demonstration sets different configurations for circles and hole tables in conjunction with the **Mode and Selection** tab options. Use **Detail, Auto Dim, Configure Auto Dim** commands from the modeling toolbar. The video offers various dimensioned results from the different configuration settings. **Use the file from the previous video (Auto Dim.ckd) to follow along with the video.**

3D Basic Tools

3D Basic Tools

V70-3D View Tools: *After Viewing this video and applying the demonstration you will be able to:*

- *Change the views*
- *Toggle construction from 2D to 3D*
- *Toggle from CPlane and World construction mode*
- *Change Cplanes*

This demonstration is an introduction to the 3D tools on the **View Toolbar**. You are shown the standard 8 views and Cplanes available in the **View** and **Cplane** tools. It also familiarizes you with the **Cplane** and **World** construction modes. Finally you are shown how to toggle between **2D** and **3D.**

V71-3D Environment: *After Viewing this video and applying the demonstration you will be able to:*

- *Cycle through default 3D views*
- *Draw in "display view" (CP=DV)*
- *Change construction plane*
- *Create text and geometry in different construction planes*
- *Set construction depth*
- *Rotate the view dynamically*

This demonstration shows how to set or change the views using the **View** command on the **View** toolbar or use **Alt+V** hotkeys. You are shown how to change the construction plane and set the depth using the **Cplane** and **Construction Depth** commands on the **View** toolbar or use **Ctrl+K** hotkeys. Finally text and geometry is created on the using the different construction planes then rotating the view using the **Spin** tool.

V72 Construction Planes: *After Viewing this video and applying the demonstration you will be able to:*

- *Create geometry in 3D using the 2D tools*
- *Create 3D "wireframe" using Xform, Delta, Join*
- *Create Cplane using the "3 PTS command*
- *Set Cplane axis display*
- *Add new Cplane to Cplane list*

This demonstration continues with creating in 3D space on a front construction plane. The geometry is then "extruded" using **Xform, Delta, Join** commands form the modeling toolbar. A new Cplane (construction plane) is created using the **3 Pos** command after selecting the **Cplane** command. Displaying the Cplane axis indicator is done by selecting **Tools, Options,** and then the **Display** tab, check the box "Display Cplane axis indicator in model". Finally, the new cplane is added to the cplane list. **Use the file shown below to follow along with the video.**
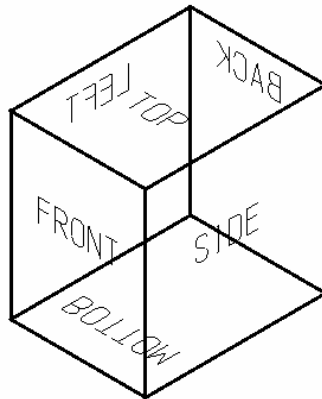
Open **3D Basics.ckd** located in**: C:\DEMO\3D Modeling**

V73-Viewports: *After Viewing this video and applying the demonstration you will be able to:*

• *Change the Viewport display*
• *Rotate objects in view about a vertical or horizontal axis*
• *Resize the viewports*
• *Change the multiple Viewport combinations*
• *Autoscale all viewports*

This demonstration introduces changing the Viewport from single to multiple viewports by selecting **view, Viewport layout,** then select a combination option. **Quad** was select in the demonstration. Using the combination of **Shift+Arrow** keys will rotate the model about either a vertical or horizontal axis. You can use the zoom tools in each individual viewport or to autoscale all viewports select **Alt+A** then **Enter.** You are also shown how to resize the viewport sizes. **Use the file from the previous video (3D Basics.ckd) to follow along with the video.**

V74-Creating Primitives 1: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a primitive block*
• *Render the block*

This demonstration introduces creating a primitive block by selecting **Create, Primitive**, on the advanced modeling toolbar, then selecting the **Create Block** command. Several primitive blocks are created using the various options available in the preferences dialog screen. The options are similar with all primitives that include the following options: "Create By", "Parameters", and "Axis". Finally, the different rendering options are introduced by selecting **View, Render** on the menu line. You can also use **Shift** and a combination of the number keys 1 thru 7. **Use the file from the previous video (3D Basics.ckd) to follow along with the video.**

V75-Creating Primitives 2: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a solid cylinder, cone, pyramid, prism, torus, and sphere*
• *Extract wireframe from a solid*

This demonstration continues with creating primitives solids. A wireframe is created by extracting from a solid body by selecting **Tools, Solid Tool Extract,** and **Create Wireframe from Solid Body** from the advanced modeling toolbar. **Use the file from the previous video (3D Basics.ckd) to follow along with the video.**

V76-Create Extrude: *After Viewing this video and applying the demonstration you will be able to:*

• *Create an extruded solid*
• *Extrude to a solid body*

This demonstration introduces the process for extruding a closed planer profile. Select **Create, Create Swept Solid** on the advanced modeling toolbar, and then select the extrusion tool. The video shows the various options in the "Extrude Planer Profile" dialog box including: "Method", "Direction", "Draft", and "Twist". Finally you are shown how to extrude to a solid body. **Use the file shown below to follow along with the video.**
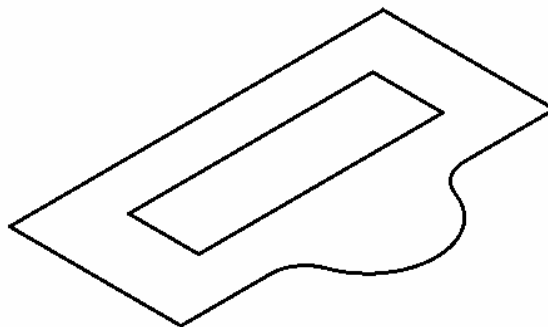
Open **Extruding.ckd** located in**: C:\DEMO\3D Modeling**

V77-Boolean Operations: *After Viewing this video and applying the demonstration you will be able to:*

• *Create solid model using Boolean operations*

This demonstration introduces creating solid models using the Boolean operations. Select **Modify, Modify Boolean** on the advanced modeling toolbar. This will bring you to the Boolean operations which include "Union", "Difference", and "Intersect". You are shown demonstrations of the different operations. **Use the file shown below to follow along with the video.**
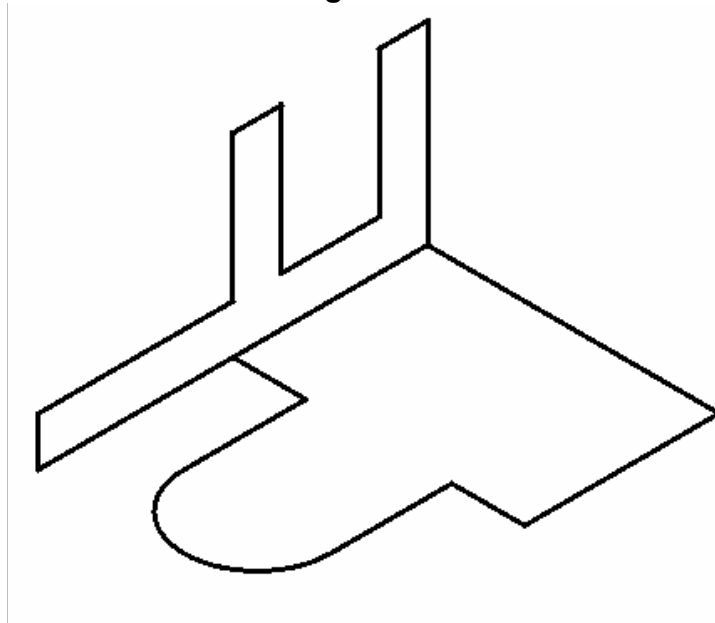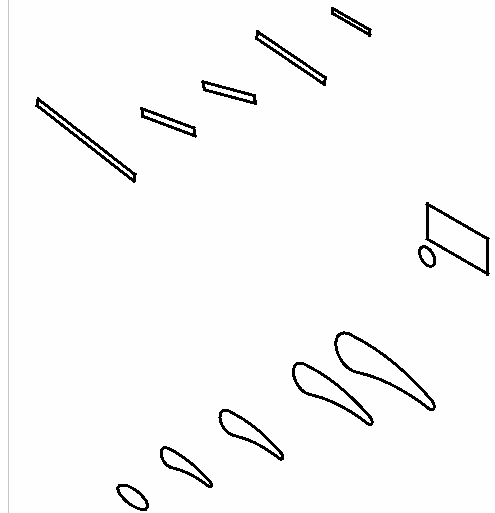
Open **Intersect.ckd** located in**: C:\DEMO\3D Modeling**

V78-Create Revolve: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a revolved solid body*
• *Revolved to a solid*

This demonstration introduces the process for revolving a closed planer profile. Select **Create, Create Swept Solid** on the advanced modeling toolbar then select the revolve tool. The video shows the various options in the "Revolve Planer Profile" dialog box including: "Method", "Options", "Draft", and "Steps". Finally you are shown how to revolve to a solid body. **Use the file shown below to follow along with the video.**
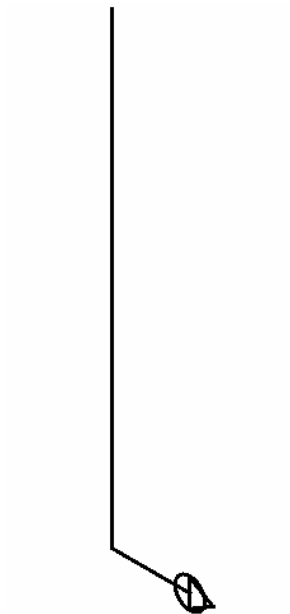
Open **Revolve.ckd** located in**: C:\DEMO\3D Modeling**

V79-Create Revolve 2: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a multi-faceted revolved solid model*

This demonstration is a continuation from the previous video and offers another option in creating a revolved solid, a "multi-faceted solid model. **Use the file from the previous video (Revolve.ckd) to follow along with the video.**

V80-Creat Swept: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a swept solid*

This demonstration introduces the process for sweeping a closed planer profile. Select **Create, Create Swept Solid** on the advanced modeling toolbar then select the sweep tool. The video shows the various options in the "Revolve Planer Profile" dialog box including: "Method", "Miter", "Draft", and "Twist". **Use the file shown below to follow along with the video.**
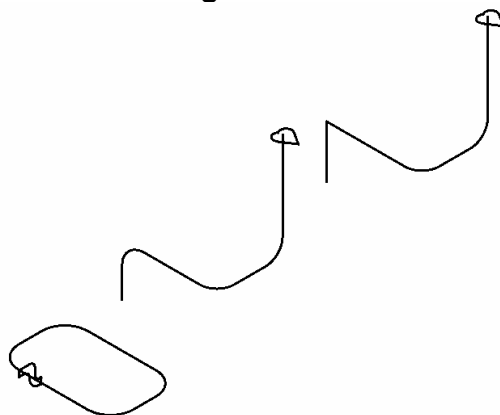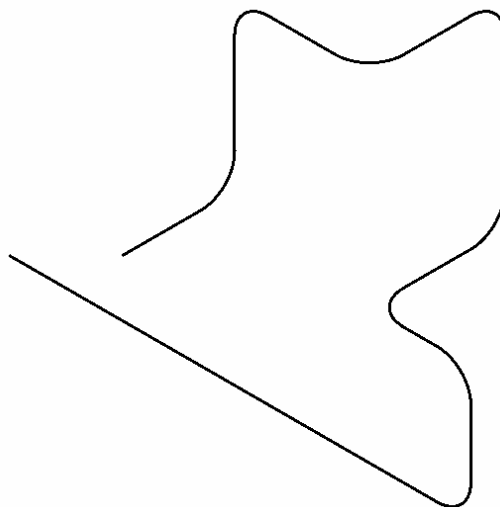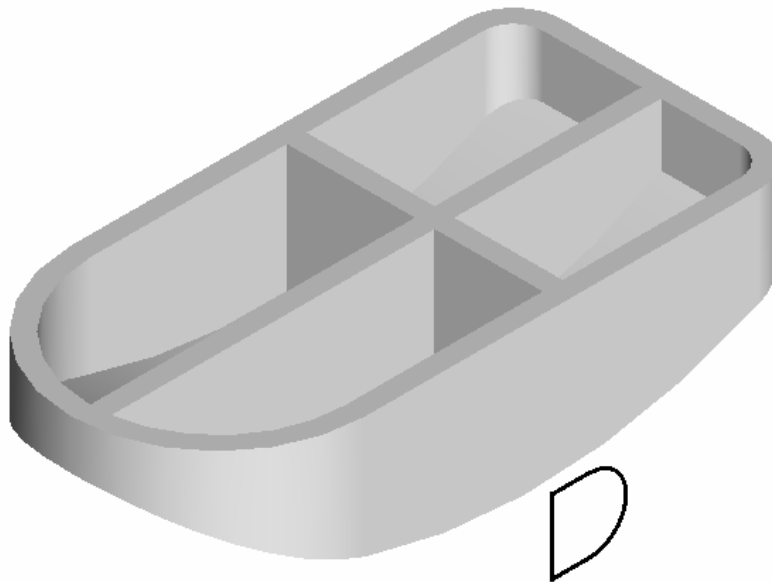

Open **Swept.ckd** located in**: C:\DEMO\3D Modeling**

V81-Create Loft: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a loft solid*

This demonstration introduces the process for sweeping a closed planer profile. Select **Create, Create Swept Solid** on the advanced modeling toolbar then select the loft tool. The video shows the various options in the "Skin Loft Profile 1 of 1" dialog box then after selecting the "done" button and checking the appropriate boxes in the "Skin/Loft Options" dialog box. **Use the file shown below to follow along with the video.**

Open **Loft.ckd** located in**: C:\DEMO\3D Modeling**

V82-Create Helix: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a helix solid*

This demonstration introduces the process for creating a helix solid by sweeping a closed planer profile. Select **Create, Create Swept Solid** on the advanced modeling toolbar then select the helix tool. The video shows several results by entering different values in the dialog screen and using different profiles. **Use the file shown below to follow along with the video.**

Open **Helix.ckd** located in**: C:\DEMO\3D Modeling**

V83-Create Pipe: *After Viewing this video and applying the demonstration you will be able to:*

• *Create a pipe solid*

This demonstration introduces the process for creating a pipe solid. Select **Create, Create Swept Solid** on the advanced modeling toolbar then select the **Create Pipe** tool. Enter the values in the "Create Pipe" dialog screen. The video shows several results by entering different values in the dialog screen. **Use the file shown below to follow along with the video.**

Open **Helix.ckd** located in**: C:\DEMO\3D Modeling**

V84-Punch & Protrude: *After Viewing this video and applying the demonstration you will be able to:*

• *Modify a solid using the punch tool*
• *Modify a solid using the protrude tool*

This demonstration introduces the process of modifying a solid model using the "cut" tool. Select **Modify, Modify Feature,** and **Cut Away Material from a Solid Body** command form the advanced modeling toolbar. This brings up the "Cut" dialog screen where you can select "Cut Direction", "End Conditions", "Draft Properties". The video shows several results from entering different end conditions in the dialog screen. **Use the file shown below to follow along with the video.**

Open **Punch & Protrude.ckd** located in**: C:\DEMO\3D Modeling**

V85-Drilled Holes: *After Viewing this video and applying the demonstration you will be able to:*

• *Extrude a face of a solid*
• *Create a hole into a solid*
• *Edit size, type, and location of hole in a solid*
• *Create a hole into a solid using the Features option*

This demonstration introduces the process of "extruding" or extending the face of a solid by selecting **Modify, Modify Faces,** and **Extrude Outward** command from the advanced modeling toolbar. Enter the parameters in the "Extrude a Face of a Solid" dialog screen. Create a hole in a solid by selecting **Modify,**

Modify Features, then select the drill a hole command from the advanced modeling toolbar. This brings up the "Drill a Hole into a Solid" dialog screen where you can select "Cut Direction", "End Conditions", and "Draft Properties". The video shows several results from entering different end conditions in the dialog screen. You are then shown how to modify holes by selecting **Modify, Modify Features,** and **Edit a Feature Hole** command. This brings up the "Solid Hole Properties" that will allow you to change the size, type, and location of the hole. Finally you are shown how to create holes by selecting **Tools** on the modeling toolbar. Select **Features** which gives you a variety of hole types to create. **Use the file shown below to follow along with the video.**
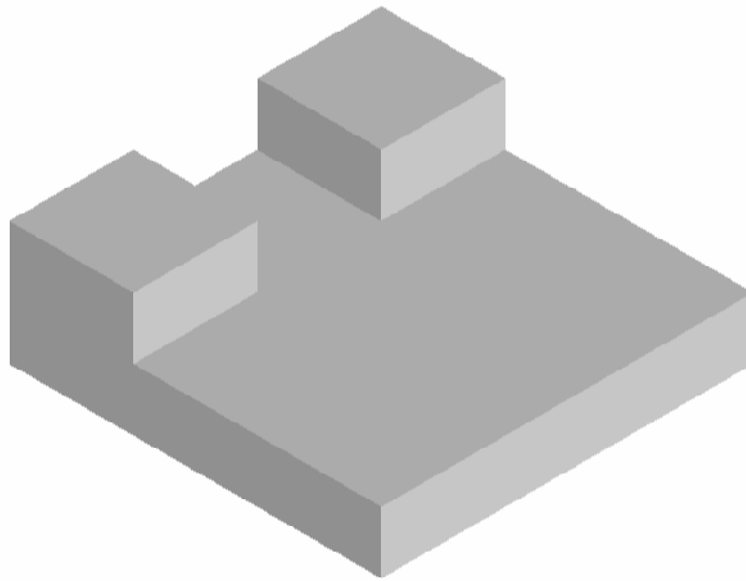
Open **Fillets & Rounds.ckd** located in**: C:\DEMO\3D Modeling**

V86-Fillets & Rounds: *After Viewing this video and applying the demonstration you will be able to:*

• *Create blended edges using Constant blend*
• *Create blended edges using Variable blend*
• *Create blended edges between faces of a solid*
• *Delete and modify blends*

This demonstration introduces the blending command by selecting **Modify, Modify Solid Blend,** in the advanced modeling toolbar, then select one of the four options available depending on what type of blend preferred. This brings up a dialog screen where you can select your preferences. The dialog will differ depending on which type of blend is selected. The video demonstrates the different types of blends including creating constant and variable blends and blends between faces of a solid. You are also shown how to delete and modify blends. **Use the file from the previous video (Fillets & Rounds.ckd) to follow along with the video.**

V87-Shelling: *After Viewing this video and applying the demonstration you will be able to:*

• *Perform shell operation*
• *Perform shell operation with multiple open faces*
• *Perform shell operation with lip*

This demonstration introduces the shelling command by selecting **Modify, Modify Feature**, command in the advanced modeling toolbar, then select "Select the Thin Walled Shell Option" which now brings up the "Shell Dialog Screen" dialog screen where you can select your preferences. The video then demonstrates performing a shell operation with multiple faces and an open face with a lip. **Use the file shown below to follow along with the video.**
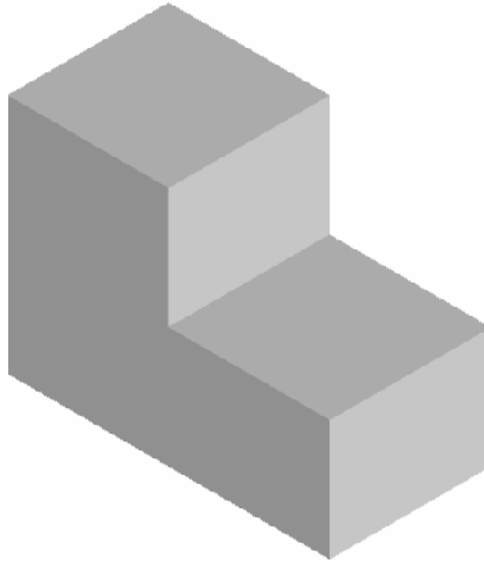
Open **Shelling Sample.ckd** located in**: C:\DEMO\3D Modeling**

V88-Trim to Curves: *After Viewing this video and applying the demonstration you will be able to:*

• *Trim or split a solid to a curve*
• *Trim or split a solid to a sheet body*

This demonstration introduces the trim/split command by selecting **Modify, Modify Advanced Trim/Split** command in the advanced modeling toolbar, then select "Trim/Split a Body to a Set of Curves" command which now brings up the "Trim or Break….." dialog screen where you can select your preferences. The video then demonstrates trimming the solid to open curve entities and to a spline.

Select **Modify Advanced Trim/Split** command again then select "Trim/Split a Body to a Sheet Body or Surface" command which now brings up the "Trim or Break…." Dialog screen where you can select your preferences. This operation is then demonstrated. **Use the file shown below to follow along with the video.**
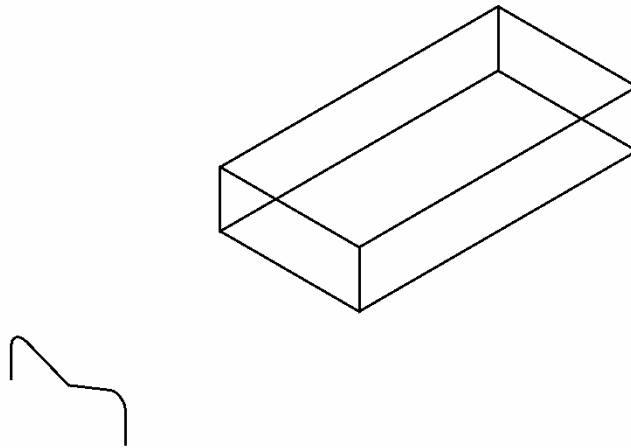
Open **Trim Solids.ckd** located in**: C:\DEMO\3D Modeling**

V89-Trim to Solids: *After Viewing this video and applying the demonstration you will be able to:*

- *Create solids using extrude*
- *Perform a Boolean operation*
- *Create a plane*
- *Trim or split a body to another body*
- *Break a solid to a curve*
- *Trim or break a Solid to a plane*

This demonstration performs extruding (see V76) and Boolean (see V77) then continues with the trim/split command by selecting **Modify, Modify Advanced Trim/Split** command in the advanced modeling toolbar, then select "Trim/Split Solids to One Another" command which now brings up the "Trim a Body to a Body" dialog screen where you can select your preferences. The video then demonstrates the different options for trimming one solid to another. Trimming to a plane is introduced by first creating the plane. Select **Create, Create a Plane,** then select the "Create a Plane using the Plane Definition Menu" command. Select the **Modify, Modify Advanced Trim/Split** command then select "Trim/ Split Body along a Plane" which brings up the dialog screen for selecting preferences. **Use the file from the previous video (Trim Solids.ckd) to follow along with the video.**

V90-Create Surfaces: *After   Viewing   this   video   and   applying   the demonstration you will be able to:*

- *Create a custom color for the color palette*
- *Create a ruled surface between 2 curves*
- *Create a surface between 3 or 4 curves*
- *Create a surface through a mesh of curves*

This demonstration begins with creating a custom color for the color palette. The video then demonstrates creating three different types of surfaces. Access to the three options is done by selecting **Create, Create Surfaces** command. The first example selects "Create a Ruled Surface Between two Curves" command which brings up the "Ruled Surface" dialog screen for selecting preferences. The second example selects "Create a Surface between 3 or 4 Curves" command which brings up the "Edge Curve Surface" dialog screen for selecting preferences. The third example selects "Create a Surface through a Mesh of Curves" which brings up the "Curve Mesh Surface" dialog screen for selecting preferences. **Use a blank file to follow along with the video.**

3D Construction Techniques

3D Basic Modeling Techniques

V91-Solid Modeling 1: *After Viewing this video and applying the demonstration you will be able to:*

Create a completed solid model using various approaches

This demonstration develops the model shown below using various approaches that begins with the a 2D drawing then uses a variety of tools including Boolean intersect, trim/split solids to curves, shell multiple faces, and blend tools. **Use the file shown below to follow along with the video.**
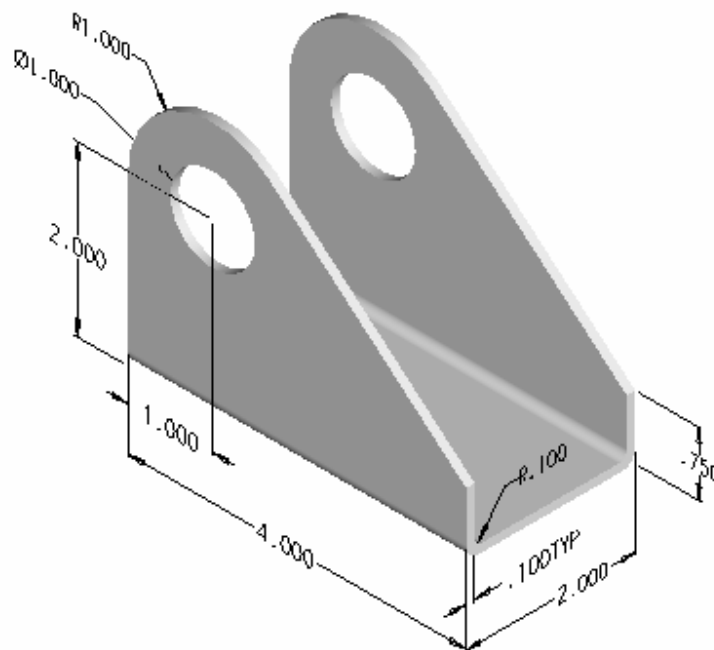
Open **3D Modeling 1.ckd** located in**: C:\DEMO\3D Modeling**

V92-Solid Modeling 2: *After Viewing this video and applying the demonstration you will be able to:*

* *Use the protrude tool in creating a solid model*
* *Create the solid model shown below*

This demonstration develops the model shown below using various tools that begin with the a primitive block then uses a variety of tools including Xform

rotate, create plane, protrude, create hole, and edit features. **Use the file shown below to follow along with the video.**
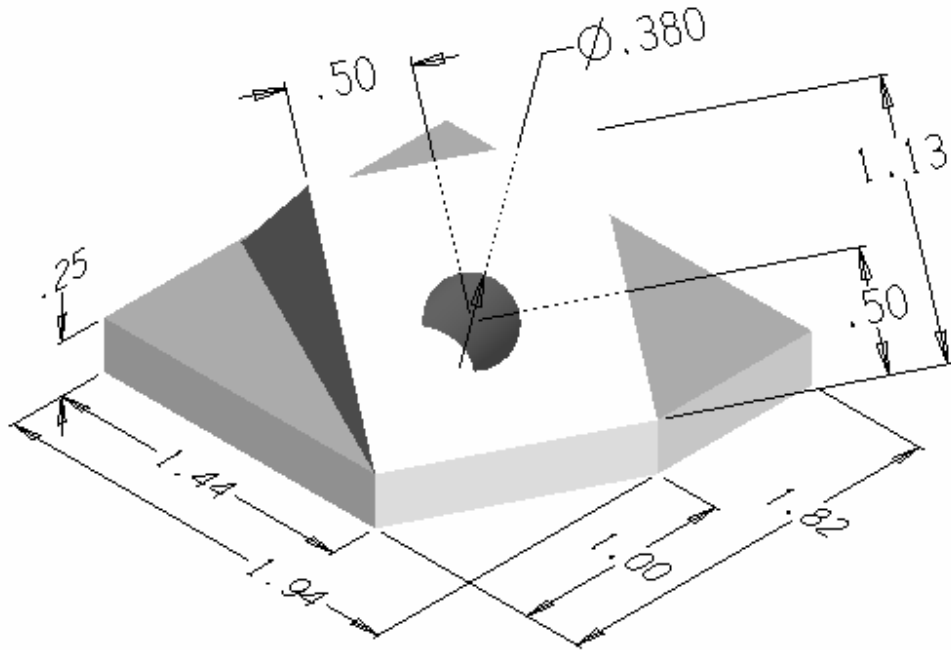
Open **Angle Bracket.ckd** located in**: C:\DEMO\3D Modeling**

V93 Solid Modeling 3: *After Viewing this video and applying the demonstration you will be able to:*

• *Create the previous model using different approach*

This demonstration develops the model from the previous video using various tools that begin with the a primitive block then uses a variety of tools including Xform rotate, create plane, protrude, trim solid to a curve, and create hole using tool feature. To check you work import **Angle Bracket Master.ckd. Use the file from the previous video (Angle Bracket.ckd) to follow along with the video.**

V94 Solid Modeling 4: *After Viewing this video and applying the demonstration you will be able to:*

• *Create the model shown below using a variety of tools*
• *Bend a model using the modify warp tool*
• *Offset a face of a solid*

This demonstration develops the model shown below using various tools that begins with the a 2D drawing then uses a variety of tools including extrude, blend, face offset, Boolean union, trim solid to a curve, and modify warp. To

bend the model, use the modify warp tool. Select **Modify, Modify Warp,** select the "Bend a Body about an Axis….." tool. This will bring up the "Bend" dialog screen for setting the parameters. To check you work import **Adjusting Key Master.ckd.** **Use the file shown below to follow along with the video.**
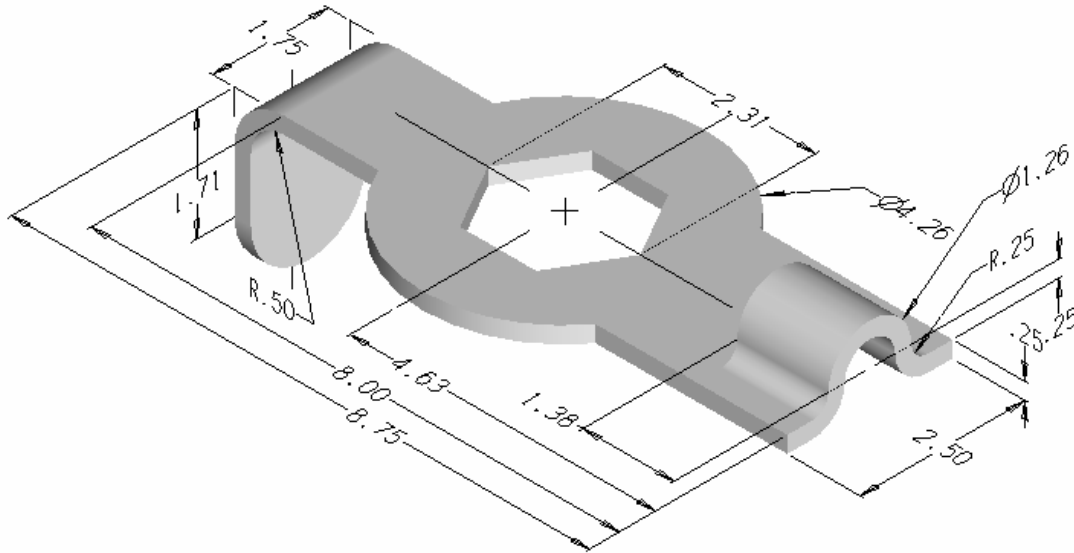
Open **Adjusting Key.ckd** located in**: C:\DEMO\3D Modeling**

V95-3D Razor 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Create complex solid model*
- *Verify mass properties of a solid*

This demonstration gives an overview of the process in developing the model shown below. Some of the foundation is already developed for you on the levels not displayed but you can create the same geometry if you wish. You will create the solids portion of the model. The completion is on the next video. You will use various tools that begin with the a 2D drawing then uses a variety of tools including spline, points, break, planes, cplanes, loft, primitive cylinder, Boolean union, and blend. You are introduced to analyzing the mass properties of a solid. Select **Tools, Verify Distance, Compute mass properties of a body** command from the advanced modeling toolbar. Once you select a solid or solids the mass properties will be displayed based on the density entered. **Use the file shown below to follow along with the video.**
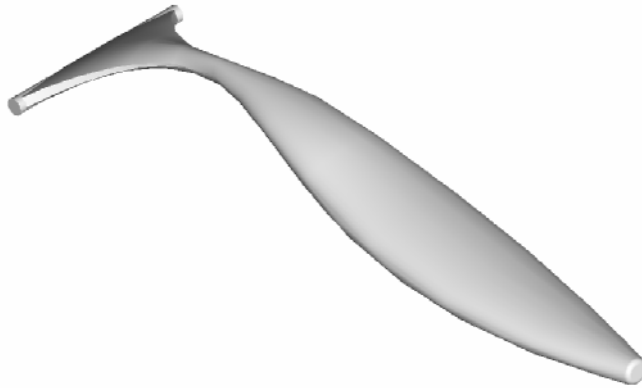
Open **Razor.ckd** located in**: C:\DEMO\3D Modeling**
(The display above is the finished model when completed; the actual file will
open with the level display and nothing in the viewport displayed)

V96-3D Razor 2: *After Viewing this video and applying the demonstration you
will be able to:*

•   *Create the model shown below*
•   *Create a duplicate body*
•   *Unstitch a body*
•   *Change face color*

This demonstration continues from the previous video in applying the raised text.
You will use various tools that begin with the creating text then using a variety of
tools including burst, Boolean union, and trim solid to surface, extract a face, and
change the face color. To duplicate a body select **Tools, Duplicate Body** in the
advanced modeling toolbar. To extract a face select **Modify, Modify Topology,
Unstitch**, on the advanced modeling toolbar then select the face to extract. **Use
the file shown below to follow along with the video.**
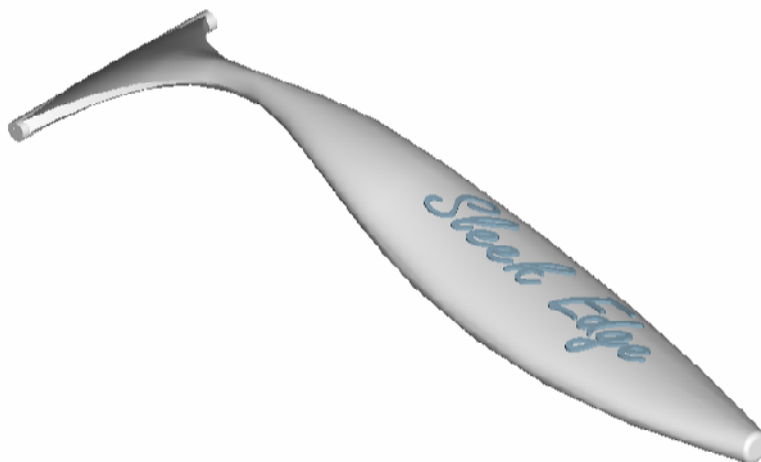
Open **Razor 2.ckd** located in**: C:\DEMO\3D Modeling**

V97-Texture Mapping: *After Viewing this video and applying the demonstration you will be able to:*

- *Apply textures to faces*
- *Imprint a curve onto a solid*

This demonstration continues using the same file from the previous video in applying textures to faces. To apply textures to a face select **Tools, Face Tools,** then "Change the Texture of Face or Solid" tool in the advanced modeling toolbar. This will bring up the "Apply Texture…." Dialog screen then select your preferences including browse and select a texture from the .bmp list. To imprint a curve onto a solid body select **Modify, Imprint, Curve onto Solid Body**, then select the body and the curve to imprint. **Use the file from the previous video (Razor 2.ckd) to follow along with the video.**

V98-3D Agitator 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Create the model shown below*
- *Apply body warping-bend and twist*

This demonstration creates the Agitator shown below. The process is completed on the next video. A variety of tools are used to create the model including but not limited to the following: primitive cylinder, helix sweep, create spline, extrude midplane, body warping-bend and twist. **Use the file shown below to follow along with the video.**
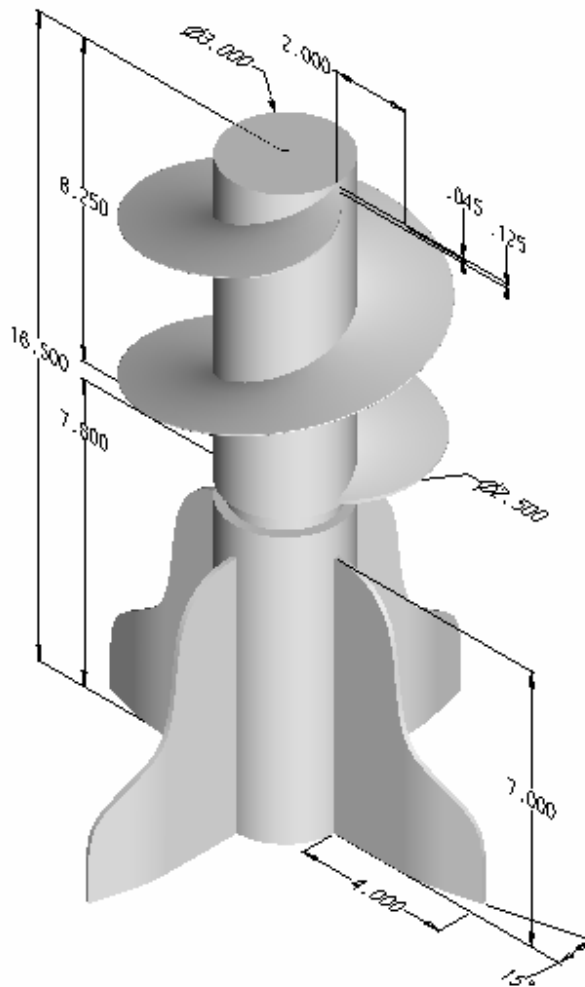
Open **Agitator.ckd** located in**: C:\DEMO\3D Modeling**

V99-3D Agitator 2: *After Viewing this video and applying the demonstration you will be able to:*

• *Trim helix using trim solid using sheet body*

This demonstration continues from the previous video to finish by applying the taper to the helix. You will create a sheet body from an extruded arc then a sheet body from the extrusion. This will then be copied and mirrored. Finally you trim the solid to the sheet body. **Use the file from the previous video (Agitator.ckd) to follow along with the video.**

Layout Mode and Sectioning

Layout Mode

V101-Layout 1: *After Viewing this video and applying the demonstration you will be able to:*

- *Create a layout*
- *Toggle between layout and model mode*
- *Create instances*
- *Display and create levels in layout mode*
- *Display model mode entities in layout mode*

This video introduces the layout mode (2D drawing environment). Select **Layout, Layout Dialog** command on the modeling toolbar to open the layout dialog screen. The parameters determine title, paper size, and scale.etc. This gets you into the layout environment. Select **Layout Toggle** or the **Alt+Shift+L** hotkeys. In layout mode, create instances (views) by selecting **Inst Util, Create a new Instance,** command**,** this opens the instance dialog screen. Once the parameters have been set, select a view to bring into the layout. Align the instances. Model specific geometry was imported to show the levels in layout mode differ from levels in model mode. **Use the file shown below to follow along with the video.**
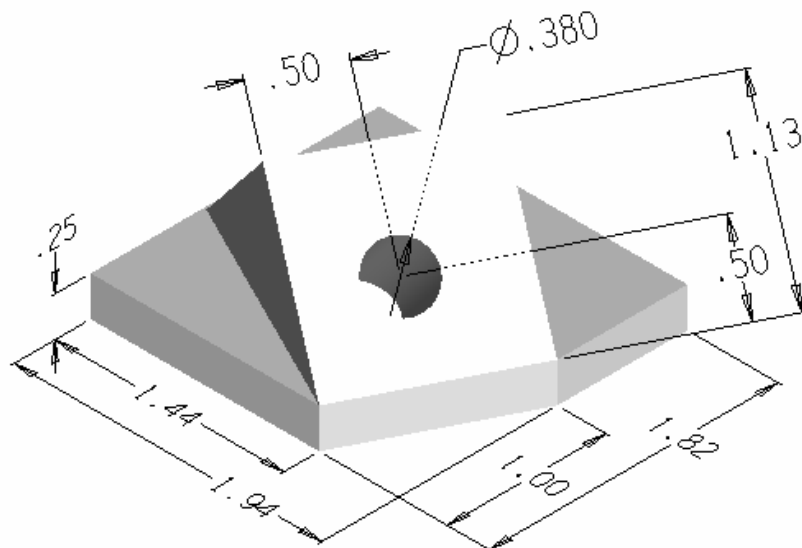
Open **Angle Bracket.ckd** located in**: C:\DEMO\3D Modeling**

V102-Layout 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Work with associativety between instances and model*
- *Edit instance display*
- *Create levels in instance*
- *Import border in layout (**Border Annotation.ckd**)*
- *Toggle off/on paper display*
- *Edit imported text on imported border*
- *Create a copy of layout*
- *Edit instances (scale, resize border, pan, change rendering, etc.)*

This demonstration continues with layout mode using the file in the previous video. You are shown the associativety between the instances and the model. A 2D document is in the process of being created by importing a border and title block complete with general notes and tolerance block. A copy of the layout is created so that changes could be made to the original without affecting the copy. You are shown the different options for editing the instances. **Use the same file from the previous video (Angle Bracket.ckd) to follow along with this video.**
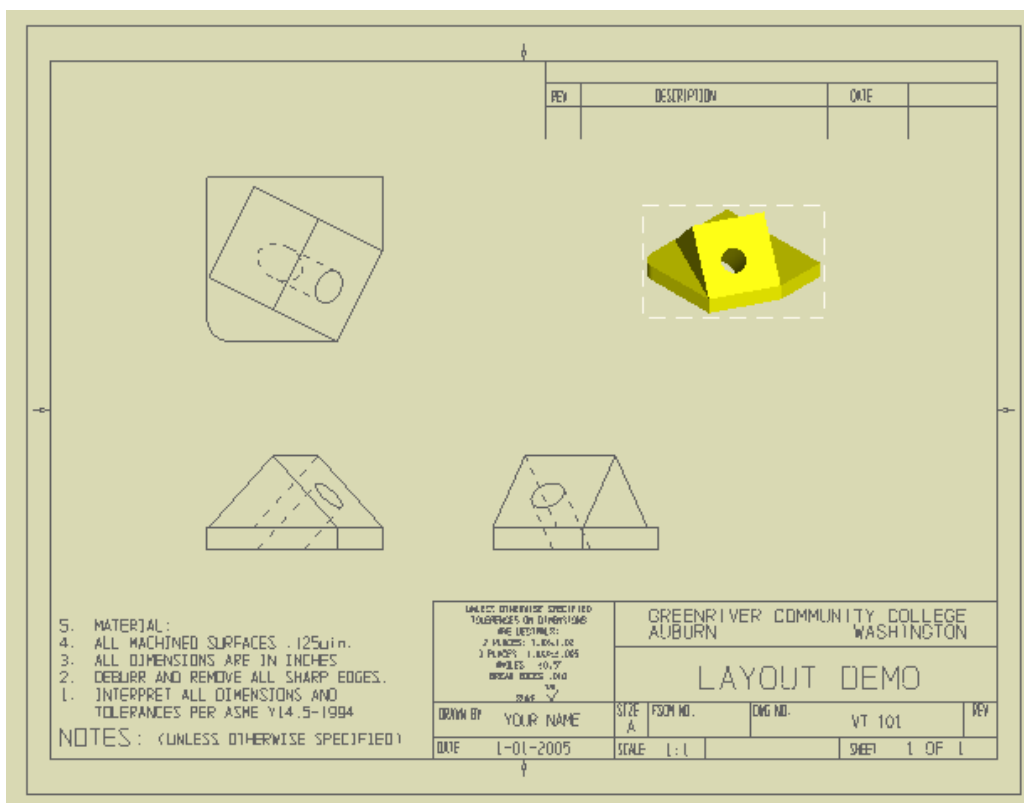
Import **Border Annotation.ckd** located in: **C:\Demo\2D Construction**

V103-Layout 3: *After Viewing this video and applying the demonstration you will be able to:*

• *Burst instances*
• *Create and save a view and cplane in model mode*
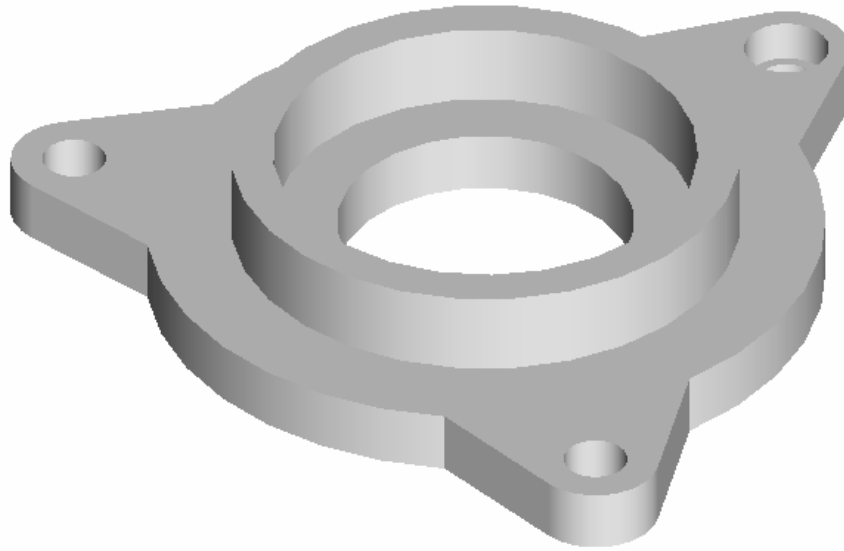• *Create a fastener*
• *Auto fit instances*

This demonstration continues with layout mode using the file in the previous video. The associativety between the instances and the model is reinforced. The process of creating a new view and new cplane in the model mode is demonstrated. The instances in one of the layouts are **burst (**modelized) to show that there is no longer a link between the instances and the model. A fastener is then created in model mode using the **Tools, Fasteners** command in the modeling toolbar. The fastener dialog screen allows for selecting and editing the fastener. The addition of the fastener is reflected in the layout that has the instances (linked) whereas the layout without the instances (burst) does not reflect the addition of the fastener. **Use the same file from the previous video (Angle Bracket.ckd) to follow along with the video.**

V104-Layout 4: *After Viewing this video and applying the demonstration you will be able to:*

• *Create and save a new view*
• *Create an auxiliary view*
• *Change display of instance*

This demonstration continues with the layout mode from the previous video. The associativety between the instances and the model is reinforced. The video shows how to create and save a new view that are used to create an auxiliary view. Finally, you are shown how to change the display of an instance. **Use the same file from the previous video (Angle Bracket.ckd) to follow along with the video.**

V105-Sectioning 1: *After Viewing this video and applying the demonstration you will be able to:*

• *Create horizontal, vertical, and auxiliary section views in layout*
• *Configure sectioning options*
• *Locate cutting plane in instance*
• *Place section view*
• *Redisplay a section view by moving the cutting plane*

This demonstration introduces creation of section views in layout mode. Select **Detail, Section View,** Then you are prompted to "select instance to be cut". Once the instance is selected the "Section View" dialog screen is presented. This

is where you set the configuration for the display of the cutting plane and section view. You are then shown a variety of options for placement of the cutting plane and the section view. Finally you are shown when you move the cutting plane it updates the section view display. **Use file shown below to follow along with the video.**

Open **Sectioning.ckd** located in**: C:\DEMO\3D Modeling**

V106-Sectioning 2: *After Viewing this video and applying the demonstration you will be able to:*

- *Delete cutting plane and section views in layout*
- *Create a fastener from the features option*
- *Create aligned and offset aligned section views of assemblies in layout*

This demonstration is a continuation from the previous video. To delete a cutting plane and section view simply select the cutting plane after the delete command, the section view will also be deleted because of the associativety between the cutting plane and the section view. A fastener is created using the **Tools,** and **Fastener** command for creating the demonstrated feature. You are then shown how to create aligned and offset aligned section views of the assembly. **Use the file from the previous video (Sectioning.ckd) to follow along with this video.**

Assemblies

V110-Merging Design Files: *After Viewing this video and applying the demonstration you will be able to:*

- *Combine parts from different design files into the active file*
- *Delete levels*
- *Work using the Tree Window*
- *Combine, duplicate, and align parts*

This demonstration introduces a method using the **Tree Window** for combining different parts from different files into the active file. The parts are then copied, rotated, move, and aligned with the base part. **Use the following files to follow along with this video: 0Body.ckd, 0-Hex Hd fastener.ckd,** and **0-Set Screw.ckd**
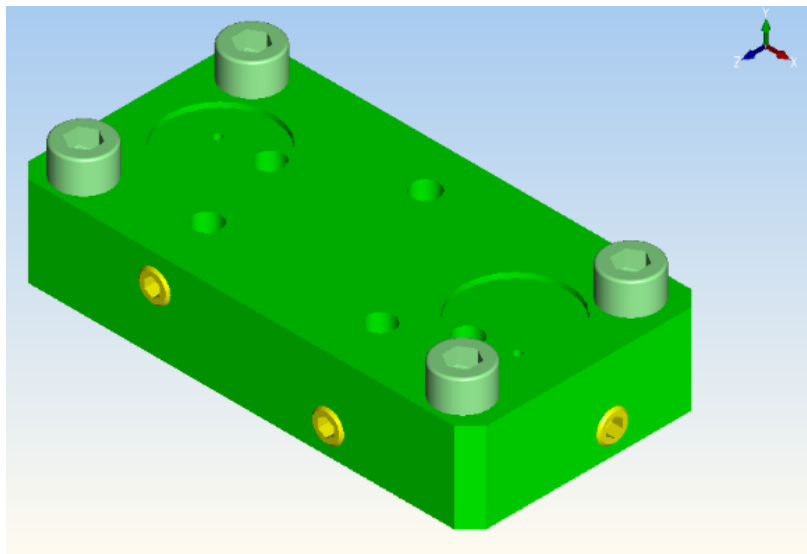
(View of combined parts)

V111-Exporting Images: *After Viewing this video and applying the demonstration you will be able to:*

- *Create an "exploded" view of an assembly*
- *Copy a raster or vector image to the clipboard*
- *Create a hidden line rendering (HLR) in the model mode*
- *Edit (HLR) to add centerlines*

This demonstration continues from the previous video and introduces a method for creating "exploded" assemblies then copying the raster or vector image to the clipboard for use in other programs like Microsoft Word, Paint, etc. You are then introduced to how to create a hidden line rendering of the drawing by selecting from the advanced modeling toolbar: **Tools, Solid Tools Extract,** and then "Extract a Hidden line…" commands on a different level, then edit it for adding centerlines.
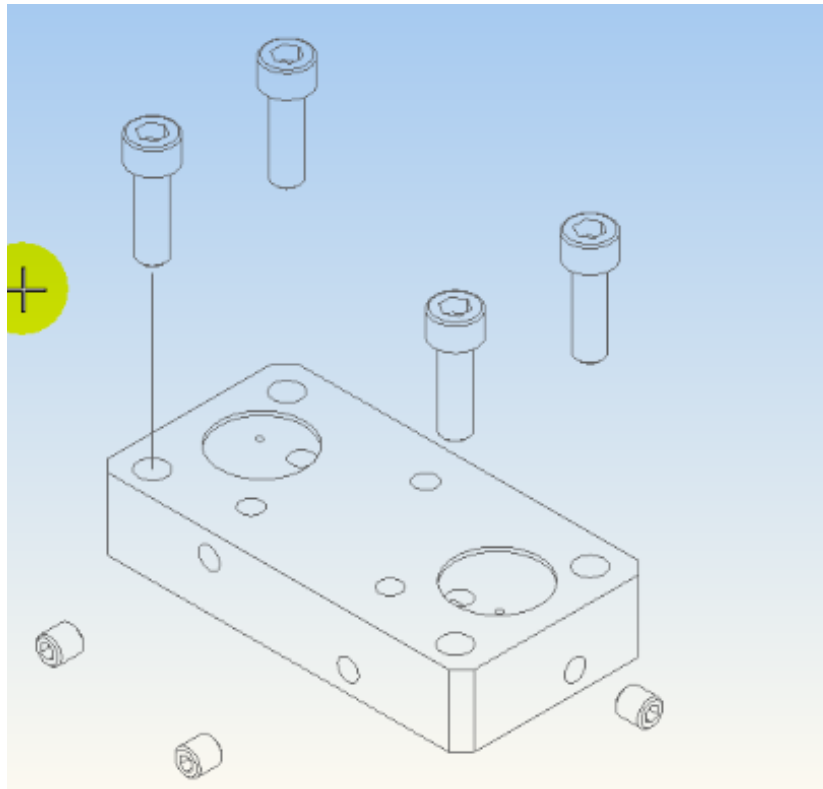
(Example of exported vector image)

V112-Editing HLR: *After Viewing this video and applying the demonstration you will be able to:*

• *Add thread representation to fasteners in HLR*

This demonstration continues from the previous video and shows how to add thread representation to the fasteners in hidden line rendering.
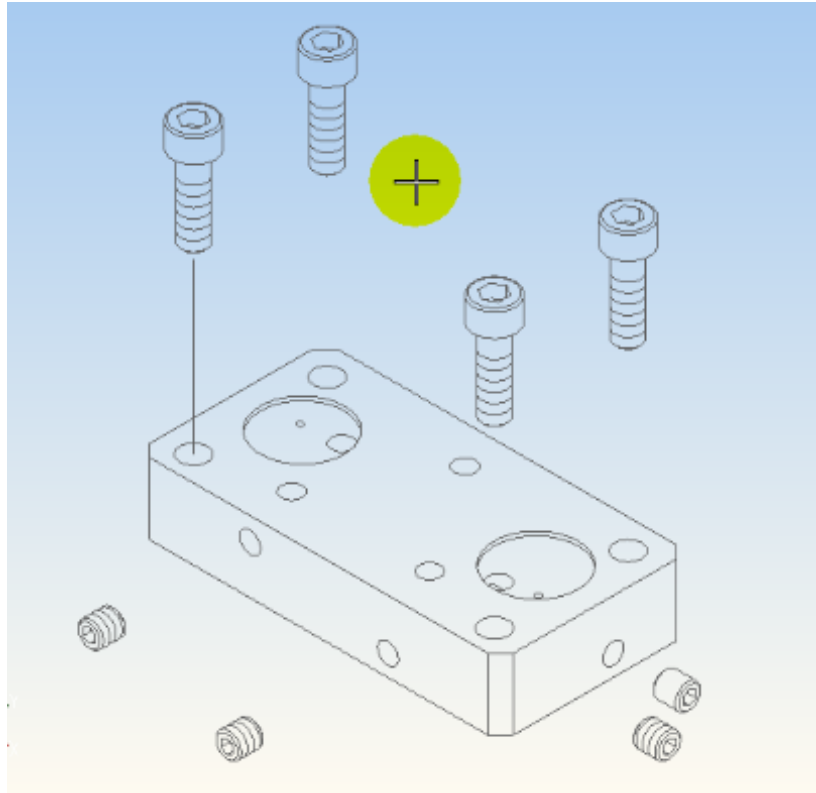
(Example of threads added to vector image)

V113-Assemblies: *After Viewing this video and applying the demonstration you will be able to:*

- *Reference an external part to an assembly*
- *Create planes*
- *Reference an external part using "associative" option*
- *Reference an external part using "current cplane"*

This demonstration introduces another method for creating assemblies by referencing external parts into the active file. To do this select on the modeling toolbar; **Tools, Assemblies, Create a New Part Reference** commands. This will bring up the "Creating a New Reference Part" dialog screen. Follow the example for referencing the initial "base part" as "non-associative". Planes are then created on their own level to be reference to. You are then shown how to reference a part using the "associative" option for referencing the part to the planes constructed in the previous step. Finally you are shown how to reference a part using the "current cplane" option for referencing the part to the active construction plane. **Use the "American.ckt" template file for this video.**

V114-Assemblies 2:   *After Viewing this video and applying the demonstration you will be able to:*

- *Rename a reference part*
- *Display and edit level display of a reference part*
- *Change color of a reference part*
- *Change orientation of a reference part*
- *Rotate planes*
- *Change location of a reference part*
- *Rotate a reference part*
- *Edit a reference part*
- *Hide and unhide a reference part*
- *Suppress and unsuppress a reference part*

This demonstration continues from the previous video and introduces the level display of referenced parts by selecting the **Part References** tab at the bottom of the part splitter display. You are shown how to change the name and color of the each reference part using this list. You are then shown the various tools available for changing the orientation, rotation, and location of the reference parts. Select **Tools, Assemblies,** this will display the tools. This is the same way to access the tools that will allow you to hide and suppress the reference parts and to edit a reference part. **Use the same part from the previous video to follow along with this video.**

V115-Assemblies 3:   *After Viewing this video and applying the demonstration you will be able to:*

- *Create "exploded' assemblies by offsetting reference parts*
- *Paste parts into assembly by "snapping to "edges" of reference parts in the assembly*
- *Create geometry using edges of reference parts in the assembly*

This final demonstration exemplifies the flexibility of working in assemblies consisting of referenced parts. You are shown how to 'explode" the reference parts using the offset tool. The video shows how you can create new geometry in the assembly file and work of the edges of the reference parts. **Use the file from the previous video to follow along with this video.**

Appendix

2D Assignments

Create the assignments in this section. Check your work by importing the master located in: **C:\Training\Demo\2D Masters**

Note to Instructors:

1. You may want to transfer the files contained in the 2D Assignments folder to a folder only you have access to. Use these files as masters to compare student's work.

2. To complete the assignments you may have the students import the borders and apply dimensions and annotation or they can import **Border Annotated.ckd** located in: **C:\Training\DEMO\2D Construction**

GUIDE – Assignment No. 1

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5
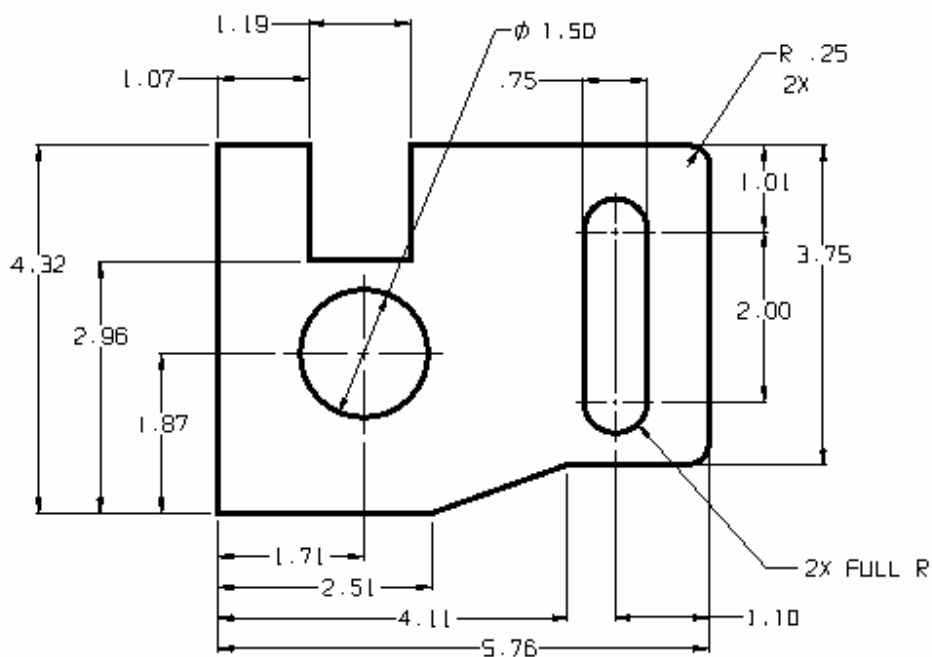
Create the drawing of the GUIDE. Check your work with the master located in:
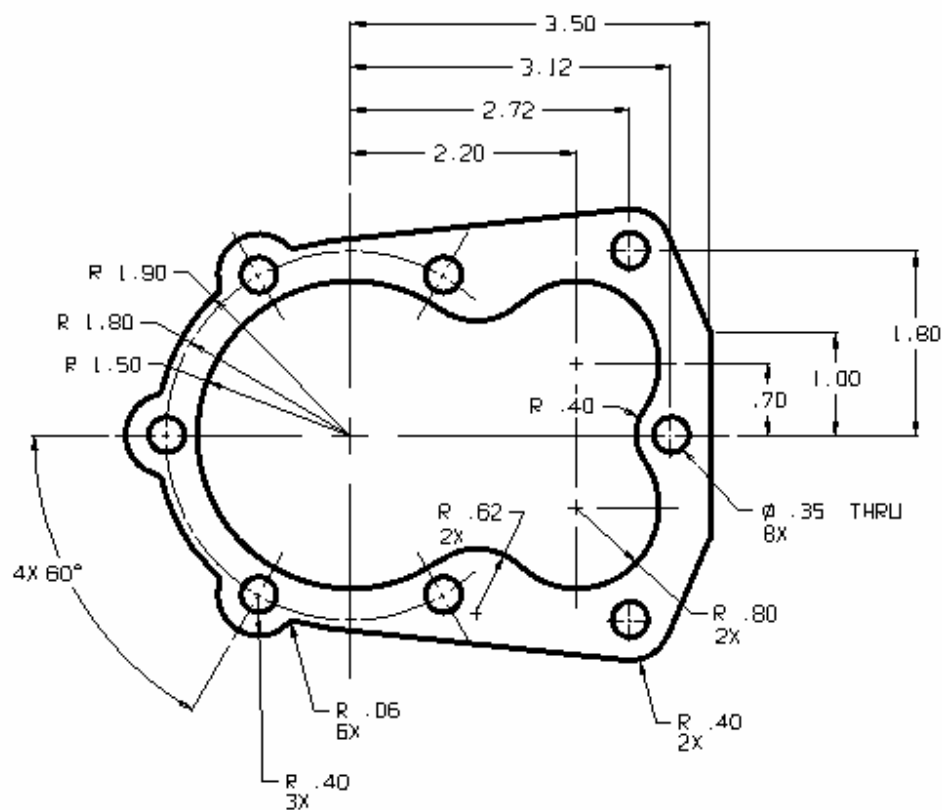**C:\Training\Demo\2D Masters\Guide.ckt**
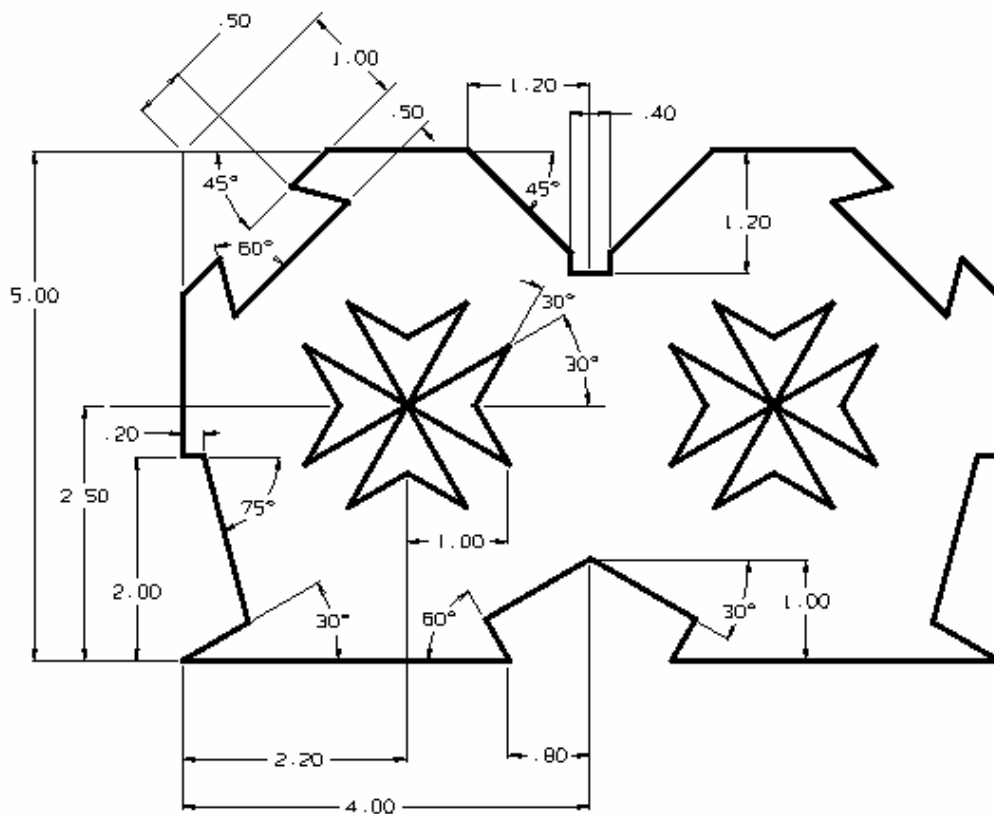
LATCH PLATE – Assignment No. 2

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5
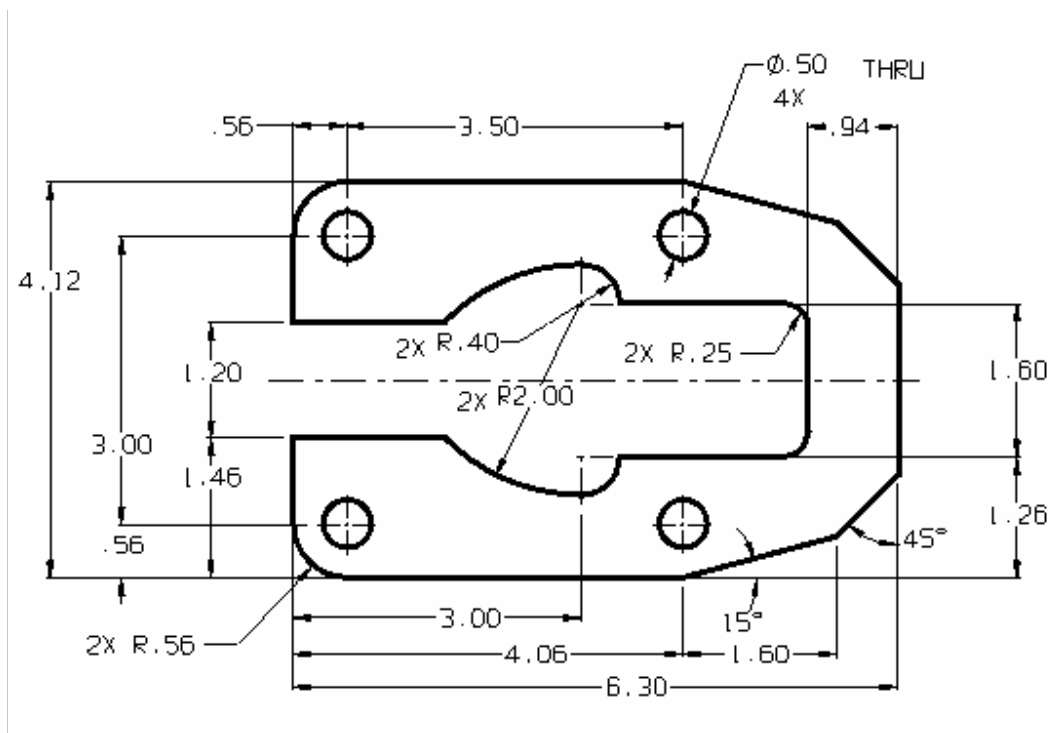
Create the drawing of the LATCH PLATE. Check your work with the master located in: **C:\Training\Demo\2D Masters\Latch Plate.ckt**

SHIFT LEVER – Assignment No. 3

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5
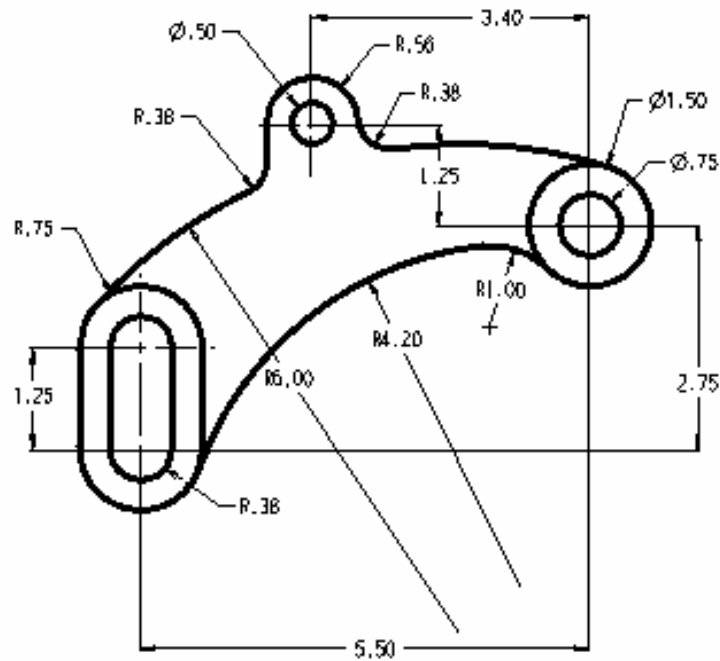
Create the drawing of the SHIFT LEVER. Check your work with the master located in: **C:\Training\Demo\2D Masters\Shift Lever.ckt**

PUMP GASKET – Assignment No. 4

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5
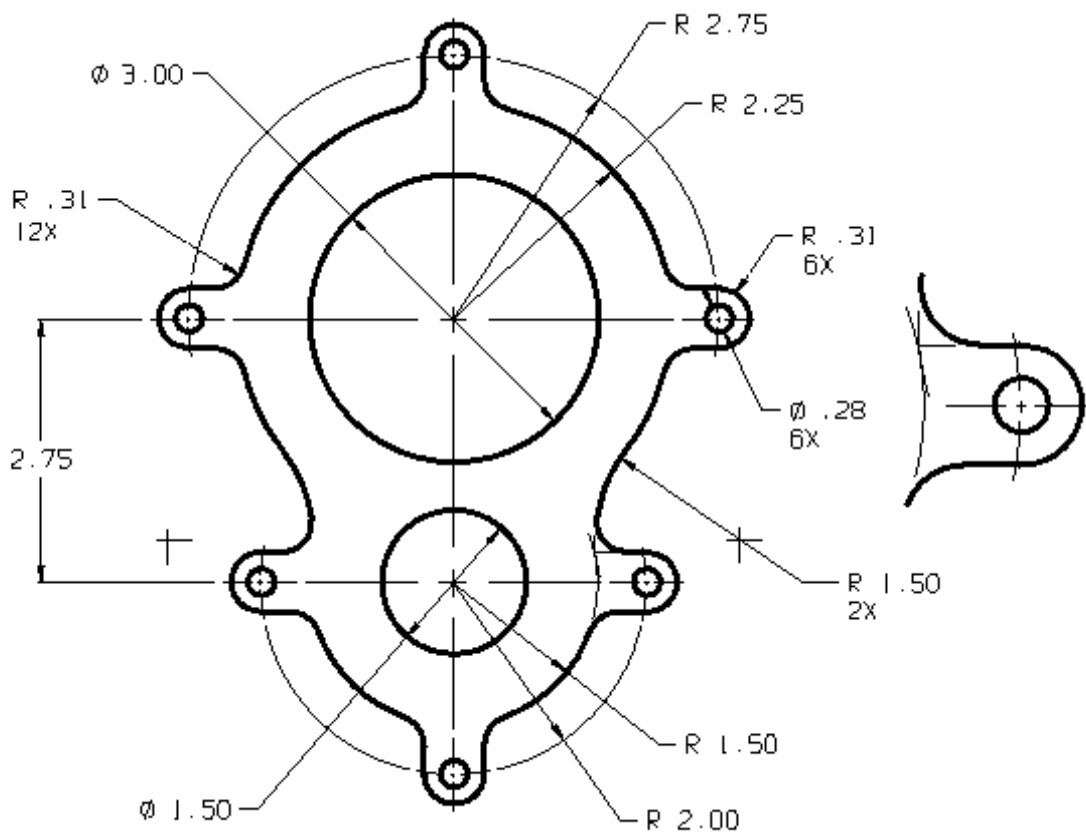
Create the drawing of the PUMP GASKET. Check your work with the master located in: **C:\Training\Demo\2D Masters\Pump Gasket.ckt**

GASKET – Assignment No. 5

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5
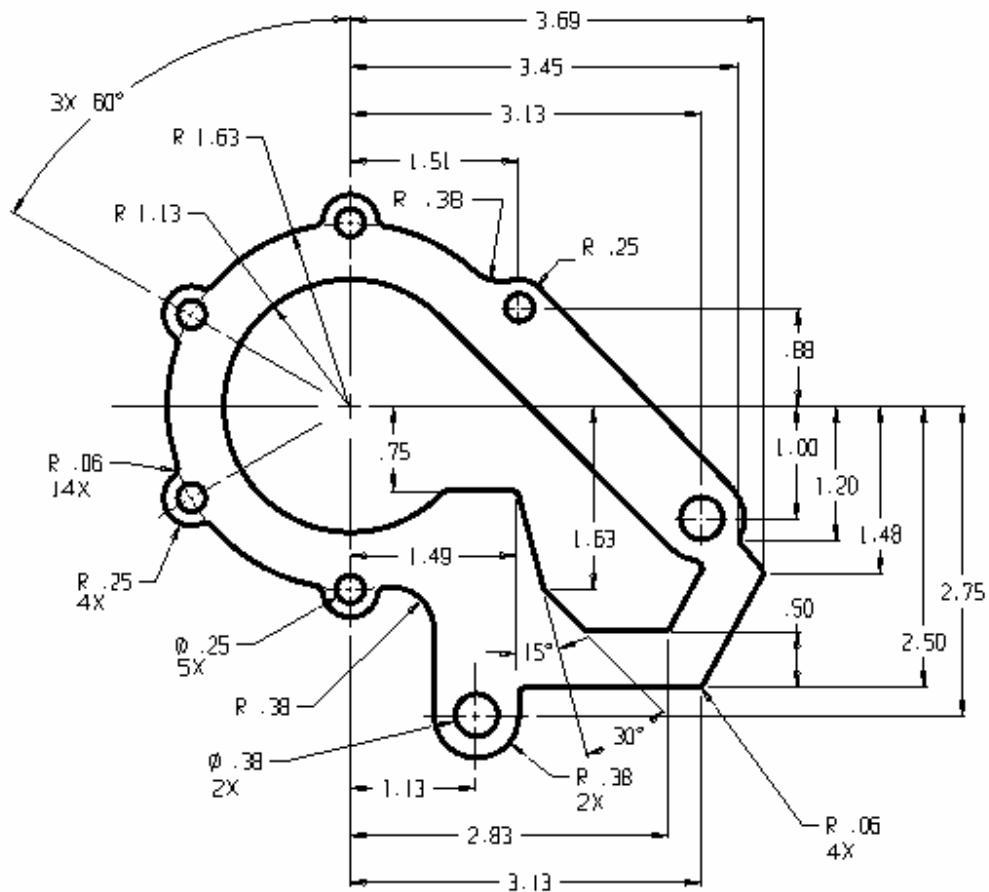
Create the drawing of the GASKET. Check your work with the master located in:
C:\Training\Demo\2D Masters\Gasket.ckt

GASKET 2 – Assignment No. 6

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5

Create the drawing of the GASKET 2. Check your work with the master located in: **C:\Training\Demo\2D Masters\Gasket 2.ckt**

STAR PLATE – Assignment No. 7

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5

Create the drawing of the STAR PLATE. Check your work with the master located in: **C:\Training\Demo\2D Masters\Star Plate.ckt**

RATCHET WHEEL – Assignment No. 8

ATTRIBUTES AND LEVEL TABLE				
Line Type	Color	Color No.	Level	Pen No.
Center	Red	2	1	1
Text	Lt. Blue	5	2	2
Hidden	Purple	6	3	3
Visible	Green	1	4	4
Border	Purple	8	5	5
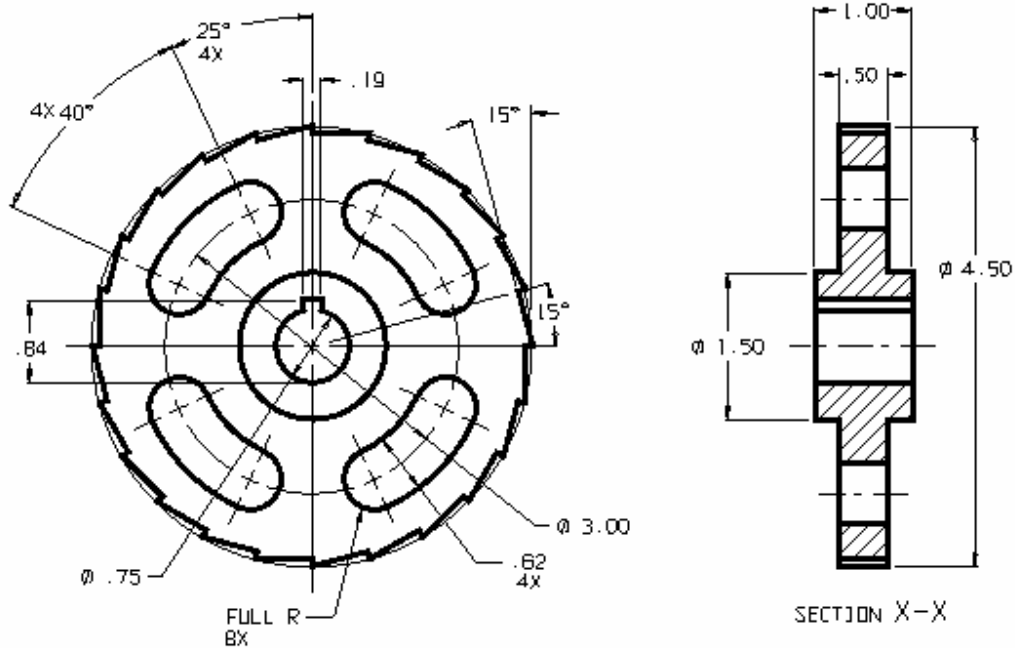
Create the drawing of the RATCHET WHEEL. Check your work with the master located in: **C:\Training\Demo\2D Masters\Ratchet Wheel.ckt**

Video List

HELP

V1-Help   1:52   3.76mb

USER INTERFACE

V2-Interface Screen-Toolbars   5:14   9.61mb
V3-Workspaces   3:25   6.80mb
V4-Templates   4:05   8.22mb
V5-Custom Toolbars   3:04   6.30mb
V6A-Intro to Macros   7:02   12.06mb
V6-Keyboard Shortcuts   4:03   7.37mb

NAVIGATING THE VIEWPORT

V7-Zooming Options   3:21   6.20mb
V8-Panning Options   2:03   3.74mb

2D BASIC TOOLS

EDIT

V9-Deleting & Undeleting   2:43   4.84mb
V10-Deleting & Undeleting   1:55   3.75mb
V11-Generic Move   3:28   6.36mb
V12-Generic Edit   1:39   3.10mb

CREATE LINES

V13-Snap Options   2:02   3.75mb
V14-Endpoints, String, and Polygon   3:00   5.43mb
V15-Para through a Point, Para at a Distance, At a Specified Angle   2:50   5.18mb
V16-Horizontal, Vertical, Both   1:27   2.61mb
V17-Tan to 2 Entities, Tan through a Point, Tan to 1st, Perp to 2nd   1:52   3.39mb
V18-Perp to 2 Entities, Perp through a Point, Perp to 1st, Tang to 2nd   1:23   2.55mb

CREATE CIRCLES

V19-Center & Radius, Center & Diameter, Center & Edge, Three Point   1:35   2.91mb
V20-2 Tangents, 3 Tangents, Center & Tangent, Diameter End Point   2:29   4.55mb

CREATE ARCS

V21-Center and Radius, Center and Diameter, Center and Edge, 3 Positions   3:02   5.20mb
V22-2 Tangents, 3 Tangents, Center and Tangent, Tangent & End Pt   2:34   4.65mb
V23-Included Angle, Radius and 2 Positions, Offset Distance, Offset Position 3.51   6.96mb

CREATE RECTANGLES

V24-Line Corners, Line Width & Height   2:05   3.60mb
V25-Rounded Line Corners, Rounded Line Width & Height   1:23   2.42mb

MODIFY FILLETS

V26-Fillet and Trim, Fillet and No Trim, Chamfer   3:30   6.08mb
TRIM EXTEND

V27-Trim First and Trim Both   4:12   7.20mb
V28-Trim Divide and Trim Double   3:24   6.10mb
V29-Trim Multiple and Trim Position   2:05   3.60mb

XFORM

V30-Delta move, copy, and join   3:03   5.24mb
V31-Old to New move, copy, and join   3:04   5.48mb
V32-Rotate move, copy, join, Array   3:38   6.39mb
V33-Mirror move, copy, and join   3:44   6.66mb

LEVELS

V34-Level Introduction   3:22   6.18mb
V35-Setting up Levels   5:05   9.56mb
V36-Entity Transfer   4:37   8.93mb
V37-Level Display   3:00   10.36mb

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V38-Setting Attributes   1:54   3.36mb
V39-Changing Attributes   3:32   7.33mb

IMPORTING BORDERS & DRAWINGS

V40-Importing Borders   4:55   9.44mb
V41-Importing Borders 2   4:23   8.54mb
V42-Merging Drawings   4:42   9.20mb
V43-Merging Drawings 2   2:59   5.64mb
CREATING TEXT

V44-Creating Text   3:52   7.01mb

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V45-Saving a File   3:09   5.81mb

PRINTING

V46-Printing   4:54   8.90mb

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V51-Geometric Construction 2 5:28 9.24mb

V52-Geometric Construction 3 6:06 10.85mb
V53-Geometric Construction 4 5:11 9.09mb
V54-Geometric Construction 5 5:45 10.20mb
V55-Geometric Construction 6 7:11 12.96mb
V56-Geometric Construction 7 8:49 15.47mb
V57-Geometric Construction 8 9:08 17.33mb

DIMENSIONING & ANNOTATION

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V59-Basic Dimensioning 2   3:59 7.10mb
V60-Dimension Editing   6:30 12.17mb
V61-Ordinate Dimensioning   6:03 10.90mb
V62-Basic Dimensioning 3   5:47 10.58mb
V63-Notes and Symbols   4:54 9.59mb
V64-Geometry From Notes   3:18 6.29mb
V65-Auto Dimensioning 1   4:30 8.45mb
V66-Auto Dimensioning 2   4:42 8.91mb

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V75-Creating Primitives 2 6:34 13.07mb
V76-Create Extrude   6:45 13.69mb
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V83-Create Pipe   2:04 3.95mb
V84-Punch & Protrude   6:53 16.34mb
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V87-Shelling   2:34 5.79mb
V88-Trim to Curves   4:00 10.01mb
V89-Trim to Solids   4:25 9.87mb
V90-Create Surfaces   7:17 15.00mb

3D BASIC MODELING TECHNIQUES

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V92-Solid Modeling 2   7:39   16.78mb
V93-Solid Modeling 3   3:56   8.83mb
V94-Solid Modeling 4   12:03 26.12mb
V95-3D Razor   7:44   17.28mb
V96-3D Razor 2   10:00 23.44mb
V97-Texture Mapping   3:42   16.11mb
V98-3D Agitator   10:05 22.02mb
V99-3D Agitator 2   4:46   12.93mb

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V101-Layout 1   8:39   16.75mb
V102-Layout 2   8:33   17.24mb
V103-Layout 3   5:29   11.15mb
V104-Layout 4   5:11   10.55mb
V105-Sectioning 1   5:13   10.14mb
V106-Sectioning 2   6:03   12.82mb

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V110-Merging Design Files   7:12   14.86mb
V111-Exporting Images   6:22   13.58mb
V112-Editing HLR   2:25   5.05mb
V113-Assemblies 1   6:13   13.41mb
V114-Assemblies 2   8:55   18.01mb
V115-Assemblies 3   4:27   9.73mb

Keyboard Shortcuts (Hotkeys)

Ctrl+A	Autoscale the active viewport
Ctrl+B	Move entities from one level to another
Ctrl+Shift+B	Move forward one in the view history
Ctrl+C	Copy the selection and put it on the Clipboard
Ctrl+D	Double the scale of the active viewport
Ctrl+E	Set the entity selection mask
Ctrl+G	Toggle the grid display
Alt+G	Grid and Snap Settings
Ctrl+H	Halve the scale of the active viewport
Ctrl+I	On-line calculator
Ctrl+K	Select a new CPlane
Ctrl+L	
Ctrl+N	Create a new design file
Ctrl+O	Open an existing part in a design file
Ctrl+P	Print the active document
Alt+P	Pan the active viewport
Ctrl+Q	Delete entities using multiple selection
Ctrl+S	Grid and Snap Settings
Ctrl+V	Insert Clipboard contents
Alt+Shift+V	Rotate the active viewport
Ctrl+X	Toggle the cursor snap
Ctrl+Z	Undo the last action
Alt+Shift+Z	Set the 3D/2D construction mode
Ctrl+Alt+J	Start recording a macro
Ctrl+Alt+K	Pause macro recording
Alt+F8	Play back a macro
Alt+Shift+F	Export an image file
Ctrl+F4	Close the active part
Ctrl+Shift+P	Display full pages
Ctrl+Y	Redo the previously undone action
Ctrl+Shift+Z	Redo the previously undone action
Ctrl+T	Edit Entities
Ctrl+M	Move Entities
Ctrl+R	Change the attributes of entities
Alt+X	Set the attributes for the active part
Alt+K	Select Part Color
Alt+S	Select Part Line Style
Alt+Y	Select Part Line Width
Alt+Shift+C	Set the World/CPlane construction mode
Alt+Shift+Q	Trim and Fillet with only a single selection
Alt+Shift+I	Toggle the line limit switch
Alt+Shift+D	Toggle Restrict Details
Ctrl+Shift+J	Set Arrow Direction
Ctrl+F	Save the active design file with a new name
Ctrl+W	Window in on the active viewport
Alt+V	Select a new Display View
Alt+Z	Set the construction depth
Alt+L	Change the active level for creating entities
Ctrl+Shift+L	Move entities from one level to another
Alt+J	Set the current default attributes
Ctrl+J	Change attributes of detail entities
Alt+Shift+L	Toggle between Model and Layout modes
Alt+Shift+N	Toggle the cursor snap

```
Alt+Shift+G          Toggle the grid display
Alt+Shift+A          Change the active instance
Alt+Shift+P          Toggle display of the paper border
Alt+Shift+S          Highlight instance-specific geometry
Ctrl+Alt+X           Toggle the part splitter
Ctrl+Alt+W           Toggle Magnify Window
Ctrl+Alt+T           Show or hide the docking tree window
Ctrl+Alt+O           Show or hide the conversation bar
Ctrl+Alt+A           Show or hide the settings window
Delete               Delete entities using multiple selection
Tab
Space
Ctrl+Up              Zoom in on the active viewport
Shift+Up             Rotate the active viewport up
Ctrl+Down            Zoom out on the active viewport
Shift+Down           Rotate the active viewport down
Alt+Left             Close the part splitter
Shift+Left           Rotate the active viewport left
Alt+Right            Open the part splitter
Shift+Right          Rotate the active viewport right
F1                   Conversation Bar Button 1
F2                   Conversation Bar Button 2
F3                   Conversation Bar Button 3
F4                   Conversation Bar Button 4
F5                   Conversation Bar Button 5
F6                   Conversation Bar Button 6
F7                   Conversation Bar Button 7
F8                   Conversation Bar Button 8
F9                   Conversation Bar Button 9
F11                  Conversation Bar Button 11
F12                  Conversation Bar Button 12
End                  Conversation Bar Button 12
Enter                Conversation Bar Accept
F10                  Conversation Bar Backup
Backspace            Conversation Bar Backup
Esc                  Conversation Bar Escape
Alt+F1               List Help topics
Shift+F1             Display help for a selected button, menu or
window
Ctrl+F6              Switch to the next window pane
Ctrl+Shift+F6        Switch back to the previous window pane
Ctrl+`               Advanced Modeling
Alt+`                Modeling
Up                   Pan the active viewport up
Right                Pan the active viewport right
Down                 Pan the active viewport down
Left                 Pan the active viewport left
Shift+Page Up        Rotate the active viewport clockwise
Shift+Page Down      Rotate the active viewport counter-clockwise
A                    Create an arc
L                    Create a line
C                    Create a circle
R                    Create a rectangle
S                    Create a spline
P                    Create a line parallel at a distance
T                    Create a line tangent to two entities
B                    Break an entity to another entity
```

F	Trim first entity to a second
D	Double trim an entity on both ends
I	Create a radial or circular dimension
O	Create a Baselinear dimension
N	Break an entity to a position
X	Transform using Delta
W	Transform using Old To New
J	Transform using Rotate
G	Transform using Mirror
M	Move selected entities using a base position
E	Delete entities using multiple selection
V	Create a vertical line
Q	Get information about the entity
Shift+P	Trim an entity to a position
Shift+E	Create Extrusions, Revolves and Sweeps & lofts
Shift+S	Create Surfaces
Shift+B	Perform boolean union, subtract and intersect
Shift+T	Trim Solids
Shift+C	Shell, Cut, Drill and Protrude Solids
Shift+M	Modify a solid body
Shift+L	Display a Level
Shift+F	Create a fillet arc, trim entities at tangency
Shift+H	Round sharp edges and vertices of a solid body
Shift+Q	Remove Features
Shift+1	Set viewport rendering to wireframe
Shift+2	Set viewport rendering to HLR
Shift+3	Set viewport rendering to HLD
Shift+4	Set viewport rendering to flat shaded
Shift+5	Set viewport rendering to smooth shaded
Shift+6	Set viewport rendering to smoothshade wireframe
Shift+7	Set viewport rendering to smooth shaded w/o wireframe
Shift+8	Set viewport rendering to silhouette
Alt+1	Change display view to view 1, top
Alt+2	Change display view to view 2, front
Alt+3	Change display view to view 3, back
Alt+4	Change display view to view 4, bottom
Alt+5	Change display view to view 5, right
Alt+6	Change display view to view 6, left
Alt+7	Change display view to view 7, isometric
Alt+8	Change display view to view 8, axonometric
Ctrl+1	Change construction plane to cplane 1, top
Ctrl+2	Change construction plane to cplane 2, front
Ctrl+3	Change construction plane to cplane 3, back
Ctrl+4	Change construction plane to cplane 4, bottom
Ctrl+5	Change construction plane to cplane 5, right
Ctrl+6	Change construction plane to cplane 6, left
H	Create a horizontal line
U	Undo the last action
Shift+D	Divide an entity by trimming out a section in the middle
Alt+H	Halve the scale of the active viewport
Alt+A	Autoscale the active viewport
Alt+W	Window in on the active viewport
Alt+D	Double the scale of the active viewport

Shift+X	Create a Centerline
Shift+A	Change the attributes of entities
Z	Default Macro Description

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-V58
X Form Delta Copy –V30, 91
X Form Delta Join –V30, 71, 93
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Zoom –V7