AutoCAD Course Book Update

By

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BRAE 462
BioResource and Agricultural Engineering
Cal Poly, San Luis Obispo
2013
Acknowledgments

First, I would to thank my project advisor, Dr. Andrew Holtz, for his help and guidance throughout the entire process of this project.

Second, I would like to thank Dr. Andrew Holtz, Dr. Mark Zohns, and Professor Harry Hamilton for their advice and input in the book format, command selection, and lab assignment selections.

Finally, I would like to thank all the BRAE and ASM students who participated in the evaluation of the current course book and the evaluation and input of my proposed book.
Abstract

This project is a proposed updated version of the course book used in the BRAE 151 course. The book that is currently used is a product of Doug Phillips (BRAE ’03). Since Mr. Phillips designed the book currently used, many new features have been added and the user interface is drastically different. This purpose of this book is to bring the course book used in BRAE 151 up the current version.
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Introduction

AutoCAD is a computer-aided drafting program that is used by all BioResource and Agricultural Engineering (BRAE) and Agricultural Systems Management (ASM) students, as well as other engineering students and professionals alike. In the BRAE department, this program is taught in a three hour lab class, which uses a book that teaches the students how to use the primary features of the program. The problem with the current book is that it is based off of AutoCAD 2002. Since the book was written some features have been added or changed. Another problem is that students typically take this class within a year of arriving at Cal Poly and forget a substantial portion of the skills learned before implementing the skills in a later BRAE class. While this is not a reflection of the current curriculum, the way in which it is taught can be changed.

While the current assignments in the book do teach the students how to effectively use AutoCAD, it is missing many new features that have been developed since the book’s printing. Some of these features include: surface modeling, new background, and new file saving format. AutoDesk has also released AutoCAD programs for Mac, as well as iPhone and Android phones. The problem that will be addressed is the implementation of new features into the BRAE 151 course book. A new book will incorporate these new features while still keeping the drawing functions that are necessary to learn how to effectively use AutoCAD.

The significance of this project is that BRAE and ASM students will learn how to use all of the most modern features of AutoCAD, which will be extremely helpful in projects, using the CNC Plasma Cutter, and for use in internships and in careers after graduation. BRAE 151 and AutoCAD are a critical part of the BRAE and ASM curriculum because most, if not all, subsequent BRAE classes use AutoCAD in some way. By having the book reflect the most current version of AutoCAD ensures that our students will be properly prepared for future classes, as well as internships and careers.
Literature Review

A search was conducted for published AutoCAD training manuals as well as effective teaching methods.

The current course book for BRAE 151 is effective in teaching the material; however, there are some redundancies that can be removed and some new features that need to be added. Assignments can be modified to reflect assignments and applications that will be seen later in the BRAE and ASM curriculum.

AutoDesk, the company that makes AutoCAD, has a website that does not have published material, but they do provide several videos that highlight the new features that are added to each new version. (AutoDESK, 2013)

*AutoCAD for Dummies* provides the easiest explanations about AutoCAD 2013 for someone one is not familiar with the program. It also is the easiest way to understand the new features. The drawings are simple enough to follow and the steps are generally laid out in a way that is easy to teach; however, it is too simple for a college level course, and takes a lot of time explaining nuances that are not stressed as hard in a ten week course. (Byrnes & Fane, 2012)

The AutoCAD course book used at Carnegie Melon was written by Kristen Kurland. It is a very in-depth book that is over 300 pages and covers a variety of topics as simple as opening and starting a new drawing. It also covers topics that are covered in the current BRAE 151 course book like polar arrays and using layers. The book, which is clearly used for sessions longer than 10 weeks, goes much more in depth showing features like linking to Excel. This book, however, is only in 2D. (Kurland, 2008)

Professor Randy Shih of Oregon Institute of Technology compiled an AutoCAD tutorial that is incredibly helpful, but very long (200+ pages). This book has so many helpful trick and tips as well as lays out what will be on the screen. This book is most similar to what the new course book should resemble. (Shih, 2010)

“Strategies for Effective Teaching,” a book prepared by engineering teachers at University of Wisconsin has many ideas about the best way to relay information as well as helping students retain this information. The book has several chapters about what helps student learn and a few of these ideas are: practical examples, guided design projects, open ended labs, brainstorming, and the question-and-answer method. (Balaraman, Fleming, Lacey, Kahn, & Nowicki, 1996)
Vanderbilt University a website for effective teaching methods and featured on this website is a book titled *Creating Significant Learning Experiences: An Integrated Approach to Designing College Courses*. This book, written by L.D. Fink, is filled with helpful tips and insightful questions. There are a few questions that are particularly worth contemplating; one good question is: “where do you want to go?” This question is answered by what is the desired outcome of the course. Another good question is “how will the students and you know if they get there?” This is an important question because one of the goals is to make sure that the students retain this information for use in future classes and careers. One more example of a relevant question is “what are you going to do to get there?” This question is essentially the goal of the project; the assignments need to be easy enough for the students to understand yet challenging enough that the skills can be retained. (Fink, 2003)
Procedure

The first step to updating the BRAE 151 course book was to review the old book and get input on how it is received by students and faculty. This was done by meeting with past BRAE 151 instructor, Dr. Andrew Holtz, and current BRAE 151 instructor, Professor Harry Hamilton, as well as current BRAE students. Both student and faculty input were used in the format and descriptions of the commands.

One of the major changes was the way commands are called. In the previous book, the commands were called using the drop down menus. In the new version, the commands are called using the shortcut key commands. Using the keyboard commands is beneficial in several ways. One advantage of using typing the commands is that the shortcuts for the commands never change; the icons and toolbar locations are constantly being changed with each version. Another benefit is that the commands save time.

The next step to creating a new course book was to compile a list of features that have been added since the original book was published. This was done by accessing a list of new features that AutoDESK publishes every year, with each new version of AutoCAD, which can be found at the following website: http://www.autodesk.com/products/autodesk-autocad/features, or by entering “AutoCAD new features” in an internet search engine.

A new icon list was compiled with updated icons. After consulting with BRAE faculty members, Dr. Holtz, Dr. Zohns, and Professor Hamilton, it was decided that the icons were not as vital to the book as the quick key commands, because the commands never change, but the icons are frequently updated. As previously stated, the individual assignment sections of the updated book, the old selection method used dropdown menus, which were replaced with the quick key commands in the new version.

Attending the BRAE 151 lab each week provided a refresher on how the class was taught and what the assignment sheets looked like. The assignment sheets were completed each week and the commands used in the respective drawing were noted and used in determining which commands will go with which section.
Meetings with project advisor and former BRAE 151 instructor, Dr. Andrew Holtz, and former BRAE 151 instructor, Dr. Mark Zohns, helped shape the list of nine assignments that would be required for this class. After meeting with Dr. Holtz and Dr. Zohns, as well as current BRAE 151 instructor Professor Harry Hamilton, a list of drawings was decided upon. Below is a list of the new assignments, with assignment five being a continuation of assignment four:

1. Drawing Basics
2. Drawing Editing
3. Layers
4. Retaining Wall
5. Blocks
6. 3-D Modeling
7. 3-D Editing
8. Problem Solving
9. Plasma Cutter

The two labs that were added were the problem solving assignment and the plasma cutting assignment. The problem solving assignment serves to teach the students how to use AutoCAD’s properties tools, such as area and volume calculators, which have seen significant change and improvement since the 2002 version of AutoCAD, which is what the current AutoCAD course book is based off of. The current version of AutoCAD has many new features under the mass properties and geometry properties umbrella commands.

The plasma cutter assignment allows the students create a drawing that would be suitable for use on a computer numerical control, or CNC, machine. Drawings that are to be used on a CNC machine, like the BRAE department’s plasma cutter, need to be done perfectly; namely, all lines must connect and there can be no overlapping lines. The drawings will then be checked to see whether or not they are suitable for use on the CNC machine. The course instructor will choose whether or not the drawings will be cut out. The current proposed drawing is a speed bottle opener like the one shown below. The instructor may choose a different object to be drawn, if he/she chooses to do so. The ninth lab could be added to the end of the eighth lab if there are not enough lab periods in a quarter.

![Figure 1: Bottle Opener](image-url)
Results

The result of this project is a complete, updated BRAE 151 course book, current with the latest version of AutoCAD (as of May 2013). The major change of this book is found in the individual lab sections; the call for the commands uses the shortcut keys rather than the toolbar menus. Another major change is the order and content of the labs. The majority of labs were left intact, with the exception of the three-dimensional wireframe. This lab was supplanted by the problem solving lab.

The appendices have also been changed to be more user-friendly. The commands are listed by function grouping (drawing, editing, annotating, etc.) rather than in alphabetic order. The command icons have also been updated for the users who prefer to use the toolbar menus.

The completed course book can be found in Appendix A.
Discussion

The primary command method was changed from menu toolbars to shortcut keys mostly because the shortcut keys have never changed over the many AutoCAD versions that have been released. Since the last version of this course book was released many commands icons have changed and the layout of the menu toolbars is very different. Dr. Holtz and Professor Hamilton assisted in making the decision to make this change. Another reason for the transition is that the shortcut keys save time as compared the menu toolbars.

The three-dimensional wireframe lab was removed in part due to BRAE 152, a Solidworks course that deals primarily in three-dimensions; this course will still retain two three-dimensional labs. This decision was made with the help of Dr. Holtz and Dr. Zohns. The removal of this lab made space for the problem solving lab. This lab is designed for the students to use the various property commands to solve problems that may be seen in subsequent classes or in industry, such as pipe flow or the area of an odd-shaped field.

The final lab, the plasma cutter lab, is a shorter lab to accommodate varying quarter schedules. At the instructor’s discretion, the lab can be supplemented to the end of the eighth lab if time does not permit a ninth lab period. Currently, the proposed drawing for this lab is a beverage bottle opener of the speed opener variety (as shown in figure 1). The instructor may choose to have the students draw another object. The purpose of this lab is to familiarize students with the process of taking a drawing to the plasma cutter, something that will be done multiple times in a student’s BRAE or ASM curriculum.

This course book is not a complete manual for the AutoCAD 2013 software. A complete comprehensive manual would not be feasible for a ten week introductory class. This course book is meant to teach students the fundamentals and some more advanced commands to aid and assist BioResource and Agricultural Engineering and Agricultural Systems Management students in future course assignments.
Recommendations

It is recommended that this course book be updated in five to seven years, to accommodate new commands in features that may be released in future versions of AutoCAD, or the BRAE department switches software for this course.

It is also recommended that the course book be updated if Cal Poly were to ever switch from quarters to semesters.

When updating this course book it is recommended that each lab assignment be drawn and the commands used are written down or the command history (F2) is copied to determine which labs should be used. This was the method used for this update of the course book and it provided thorough lists of commands for each individual lab.

A final recommendation is that the ninth lab, the plasma cutter lab, be a simpler drawing and relatively small if it is to be cut out on the plasma cutter.
References


Appendix A- Course Book
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Introduction

AutoCAD is a computer-aided drafting program that releases a new version every year. Every year new features are added and the graphic user interface is changed. In the 2013 version of AutoCAD, there are two ways to perform commands:

1. Using the tab menus and toolbar buttons on the top of the screen
2. Typing in the command or the shorthand version of the command

Typing in the command, or the shorthand (line or L for example), is the recommended method. The views of AutoCAD will constantly be changing, but the command codes will never change. These commands can be typed into the command land or the user can just start typing and the program will try to assist by showing suggestions that match the letters typed.

The first thing to check when drawing are the units (if they are needed). Typing units will bring up the “Drawing Units” dialogue box. In this box the user may choose between a variety of English and Metric scales. The annotation style and precision are also controlled.
in this box. The style choices are Architectural, Decimal, Engineering, Fractional, and Scientific. When a style is chosen, an example will appear in the Precision section.

Once a command is entered and completed, pressing Enter will repeat the command. For example, if a circle is drawn, pressing enter will enable the user to draw another circle without having to type the command or press the icon. Some commands, such as line, will continue on after each point until the user ends the command by pressing Esc. Esc can also be used to cancel a command.

In order to choose multiple items to delete, copy, move, etc. a user may click on each individual piece of the drawing, or, if a whole region is to be selected, a user may click and drag a transparent green box over all parts that are to be selected.

Once drawings start to fill up the whole screen the user can zoom out in two ways. One way is to use the scroll ball on the mouse, scrolling up will zoom in and scrolling down will zoom out. Another way to zoom out would to be type Z (Enter) E (Enter). The command is the Zoom: Extents command; it zooms out (or in) so that the drawing covers the whole screen. This command is useful if a drawing is mysteriously missing or for viewing the largest whole view.

If a mistake is made or something is accidently deleted, pressing Ctrl+Z will undo the action. This can also be done by typing UNDO and then telling the program how many actions to undo.

Leaving the mouse on an icon (without clicking) for approximately three seconds will bring up an animation that shows what that icon does.

<table>
<thead>
<tr>
<th>Command</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enter</td>
<td>Repeat Last Command</td>
</tr>
<tr>
<td>Esc</td>
<td>End/Cancel Current Command</td>
</tr>
<tr>
<td>Z (enter) E (enter)</td>
<td>Shows the Entire Drawing</td>
</tr>
<tr>
<td>Units</td>
<td>Brings Up Unit Dialogue Box</td>
</tr>
<tr>
<td>Undo</td>
<td>Undo Multiple Actions</td>
</tr>
<tr>
<td>Ctrl+Z</td>
<td>Undo Last Action</td>
</tr>
<tr>
<td>Ctrl+S</td>
<td>Save</td>
</tr>
<tr>
<td>F2</td>
<td>Opens Past Command Box</td>
</tr>
<tr>
<td>Mouse on an icon and press F1</td>
<td>AutoCAD help on that feature</td>
</tr>
</tbody>
</table>
Lab #1: Drawing Basics

Objective: This lab is designed to familiarize the users with the basic commands of AutoCAD. This commands learned in this lab will serve as a building block and all subsequent labs and most drawings will use most of these commands.

Comments: The icons for the commands are constantly changed with each new version of AutoCAD; however, the shortcut keys never change. The shortcut keys are in bold right before the description of the command. The commands do not need to be entered in the command line at the bottom of the screen, the user can type and command will appear wherever the cursor is. AutoCAD will sometimes be able to auto-finish the command. If the command is typed incorrectly, AutoCAD will output a message saying “Unknown Command...Press F1 for help.” Pressing F1 will bring up the help box and commands can be searched in the search bar.

Commands:

grid F7: Pressing F7 will toggle the gridlines that appear in the background of the drawings.

line I: The line command will allow the user to draw a straight line. A line can start at any point or be “snapped” onto an existing part of the drawing. Once the line is started, the user is given the option to dictate and angle and type in a distance, or manually pick the end point of a line. After a line ending point is chosen, the program will continually begin a new line from the end of the last line until the user presses ENTER or ESC.

osnap F3: Pressing F3 toggles the OSnap mode which assists the user in choosing start and end location. Right clicking the OSnap (which is a square with an orange circle on the corner) in the status bar, will allow the user to choose which OSnap properties to turn on or off.

circle c: AutoCAD currently has six ways to draw a circle

- Center, Radius: Choose center, then enter a radius
- Center, Diameter: Choose center, then enter a diameter
- 2p: Choose two points
- 3p: Choose three points
- ttr: Choose two tangent points, then a radius
- ttt: Choose three tangent points
offset: Offset will create a line or curve parallel to an existing object. The user must select a line or curve to offset, the distance to offset, and which direction.

erase: Objects can be selected one at a time to be erased or the selection box can be used to select multiple items. Another way to erase objects is to select an object, then press the “Delete” button on the keyboard.

undo: This will undo the previous action. If the whole command “undo” is typed out (as opposed to “u”), AutoCAD will ask how many actions to undo. Another option for undoing an action is Ctrl+Z.

fillet: This will fillet two connecting lines. After entering the fillet command, type r (for radius) and specify the fillet radius, then select the first and second lines. Fillet radius default is set for 0.000, so a radius must be entered.

trim: This command will cut intersecting lines. After the command is initiated, choose the line that will act as the “cutting edge,” then select the part of the line that is to be cut.

extend: This command will connect a line to another line. After the command is initiated, select the surface that the line will extend to, then choose the line that is to be extended.

poly line: This command functions very similarly to “line,” but when poly line is closed on itself, it forms a polygon, that is recognized as a single object, whereas a square made from the “line” command would be recognized as four lines.

arc: This command draws an arc. AutoCAD provides several ways to draw an arc based on what information is known. These options are start, end, center, angle, length, radius, and direction.
Lab #2: Editing

Objective: The purpose of this lab is to teach the user how to use the editing features that make AutoCAD such a useful tool in drafting.

Comments: The commands in this lab, like mirror and array, allow the user to create many details in short amounts of time. The commands also enable the user to edit objects quickly.

Commands:

**polygon**  
**pol:** This command will draw a polygon that can inscribed or circumscribed on a circle of a selected radius. The command will bring up a prompt that will ask for a number of sides. Generally anything with more than 12 sides will look like a circle.

**mtext**  
**mt:** This command allows the user to input a multiline text box on the drawing. Once this command is called, the program will ask for a starting point and for the opposite corner. The text can be copied, moved, and arrayed just like any object. Double-clicking the text will allow the user to edit the text and change the size, color, etc.

**regenerate**  
**re:** The regenerate command recomputes the screen coordinates for the objects in the current viewport. This command will show a more accurate representation of the drawing because AutoCAD approximates the coordinates of shapes to save time on regeneration.

**move**  
**m:** This command moves an object or any number of selected objects. Objects are chosen in the same way as if they were being erased, then a base point is selected and the objects are moved to the new location.

**copy**  
**co:** This command will copy an object/objects. Once an object is chosen to be copied, the copies can be placed using the mouse or a specified distance can be entered.

**rotate**  
**ro:** This command will rotate an object or group of objects. The user must select the object(s) and a base point. After these are chosen, the user can type in an angle of rotation or rotate the object(s) manually using the mouse.

**scale**  
**sc:** This command allows the user to enlarge or reduce an object or group of objects. The user must select the object(s) and a base point. The user
then can type in an enlargement factor (i.e. 2) or a reduction factor (i.e. 0.5).

**mirror**

mi: This command will mirror an object or objects. The user must select the object(s) and the direction. After these are chosen AutoCAD will ask: “delete source object?” Entering “Y” will delete the object that the mirror image was taken from; then this function basically acts as a cut, paste, and flip. Entering “N” will keep both the source object and the mirrored image, leaving the object and the mirror image.

**array**

ar: There are three different array commands: rectangular, polar, and path.

- **Rectangular**: Distributes the object(s) into a user specified combination of rows, columns, and levels
- **Polar**: Distributes the object(s) into a circular pattern of any angle up to 360°
- **Path**: Distributes the object(s) along a line, polyline, arc, etc.
Lab #3: Layers and Annotating

**Objective:** The purpose of this lab is to allow the users to become familiar with the commands that are necessary for future BRAE and ASM drawings that require hatching, dimensioning, and annotating.

**Comments:** When a drawing becomes complex with internal objects, hatching can make it easier to differentiate between objects, as well as make finding areas simpler. Using the boundary or region command to convert lines into objects allows the user to hatch the object and it makes using the editing commands from the previous lab easier. The dimension command is an important command that will be used in almost every drawing.

**Commands:**
- **region** `reg:` This command allows the user to convert lines into a region. It functions similarly to the poly line command. The user chooses lines that are connected and closed, and AutoCAD will convert those individual lines to a single region. To use this command the lines must form a closed entity and the lines cannot have an internal intersection.
- **boundary** `bo:` The boundary command creates a boundary or a polyline from a closed entity. After entering the command, the boundary creation dialogue box will appear. Under the boundary retention section, the user can choose between creating a polyline or a region. After choosing the preferred type, hitting enter will bring the user back to the drawing. AutoCAD will then ask to pick an internal point. Choose a point inside the desired object and AutoCAD will automatically create the polyline or region, or display an error message. **Some commands will not work with a polyline (like massprop)**
- **bhatch** `bh:` This command allows a user to fill in a region with a hatch pattern with a variety of color and pattern options. After the shortcut is entered, AutoCAD will ask for an internal point; this internal point must be a closed region like one created with the region command. Before the internal point is chosen, the user should choose the color, pattern, and other property options that appear where the toolbar icons used to be. After the properties are chosen, clicking an internal point will fill the region with the hatch pattern. If the user wants to change the hatch pattern later on, clicking on the pattern will allow the user to change the properties.
layer  

**la:** This command brings up the layer properties manager dialogue box. Using layers allows the user to keep a drawing looking organized by having different line properties. In this dialogue box the user can create a new layer and edit layer properties such as line color, line weight, and line style. The default setting is a white, continuous line. Each layer has several properties that are adjustable in this dialogue box and this is how they appear in order:

- **On/Off:** A layer that is “on” will be visible, and “off” layer is invisible and will not be plotted
- **Thawed/Frozen:** Thawed layers are shown and plotted, frozen layers are not shown, plotted, regenerated. This saves refresh time during drawing. The drawing is regenerated when the frozen layer is thawed.
- **Unlock/Lock:** Objects in unlocked layers can be edited. Objects in locked layers can be seen but cannot be edited or deleted.
- **Line Color:** Choose a line color for a layer
- **Line Type:** Choose solid, dashed, or patterned line
- **Line Weight:** Choose the thickness of a line
- **Transparency:** Choose a percent transparency
- **Plottable/Not Plottable:** Plottable layers will be show up when the drawing is printed or exported to a PDF file. Not Plottable layers will not show up.

chprop  

**ch:** This shortcut brings up the properties dialogue box. Click on an object to see and edit its properties. From this dialogue box, the user can change the object’s properties such as color, layer, and line weight. It also gives geometry properties such as area, perimeter, radius (where applicable), etc.
dim This command is best used without a shortcut command. The top icon under the **annotation** section of the toolbar menu area is the dimension command. The default is a linear dimension, other dimensions can be chosen by clicking the dropdown arrow of the icon. The dimension types are:

- **Linear**: Shows the distance along the X or Y axis. The user must choose two points and AutoCAD will find the distance.
- **Aligned**: This functions the same way as linear, but will find the actual distance between two points, not just the difference along the X or Y axis.
- **Angular**: Choose two connected lines and AutoCAD will find the angle between.
- **Arc Length**: Finds the length of an arc or polyline arc segment.
- **Radius**: Select a circular and AutoCAD automatically finds the radius.
- **Diameter**: Select a circular and AutoCAD automatically finds the diameter.
Lab #4: Retaining Wall

**Objective:** The purpose of this lab is to draw a retaining wall and use the new table command, as well as learning the different plot options.

**Comments:** The retaining wall is a drawing that will appear in upper division BRAE classes. The plot to PDF is a good way to check what will be printed before actually printing the drawing. Leaders are another command that will be used on most drawings because they are an easy way to annotate a drawing.

**Commands:**

- **dimension style manager**
  - `d`: This command allows the user to edit the dimension style. The command brings up a dialogue box that allows the user to create a new dimension style or modify the current style. The lines, text, units, and tolerances are all controlled in this dialogue box.

- **donut**
  - `do`: This command creates two concentric circles. One way to use this command is to specify an inner diameter and an outer diameter, and a donut shape will appear that can be placed any number of times on the drawing. The other way to use this command is to choose two points and specify the outside diameter.

- **page setup**
  - `page setup`: This command brings up the page set up manager dialogue box. The manager will show the current set up details; the user can create a new page or modify the current one by pressing the New or Modify buttons on the right. While modifying a page set up there are many things that can be changed including:
    - Paper Size: Letter, ANSI D, etc.
    - Plot Area: what will be plotted or printed
    - Pot Scale: Fit to paper or the user can choose a scale
    - Plot Orientation: Portrait or Landscape
    - Printer/Plotter: The user can choose a connected printer to send the drawing to or can send the file to another program like Adobe PDF
**multileader**  
*mleader:* This command creates and annotative arrow with a text box at the end of its line. This feature is used for specifying parts of drawing like rebar size. This command will ask for a starting location, which is where the arrow will point. The user then chooses where the line will end/text box will start. The text in this box can be edited just like the *mtext* command.

**quickleader**  
*le:* The *quickleader* command allows the user to edit the setting before putting in the annotative leaders. After typing the command, entering *settings,* will allow the user to change the annotation styles, leader line and arrow styles and the text attachment location. Placing the *quick* leader is the same as placing the multileader. Once the leader is placed, the user can enter the text. Hitting enter after each line will start a new line of text, hitting enter after no text will end the command.

**table**  
*table:* The *table* command will bring up a table set up dialogue box where a user can create a table that will be inserted in the drawing. In the dialogue box, the user can choose to start a new table or link to an Excel table, under the insert option section. Under the column and row settings the user can specify how many columns and rows are needed. The cell styles section allows the user to customize the first and second rows, as well as all subsequent rows. The default is first row title, second row headers, and all other rows data.

**table edit**  
*tabledit:* The *table edit* command allows the user to edit the data in the cells. Pressing enter will move the cursor down a cell and pressing tab will move the cursor to the right. Clicking on the table will bring up blue squares on every cell. Clicking and dragging the cells will adjust their height and width.

**plot**  
*plot:* This command brings up a dialogue very similar to the page set up box. Just like the page set up box the user can choose a printer or to save a copy as a PDF. The user can also choose between a wide variety of paper sizes. Perhaps two of the most important options are “center the plot” under plot offset and “fit to paper” under plot scale. Center the plot will take everything that is to be plotted, find the center, and put that point in the center of the chosen paper size. Fit to paper will adjust the scale so that the entire drawing will fit on the chosen paper size.
Lab #5: Blocks

Objective: The objective of this lab is to familiarize students with the process of creating and editing blocks, as well as the benefits of using blocks.

Comments: A block is a group of individual objects that are combined into one object, similar to the group function in Microsoft Word. Using blocks can speed up the process of a drawing, if an object is to be used many times, like in an electrical schematic of a circuit. Blocks can be copied, rotated, and scaled.

Commands:

**block**: This command allows the user to convert multiple objects into a singular object; it is basically the region or boundary command for objects that do not close on themselves. Enter block to bring up the block definition dialogue box. The first thing needed is a name for the block. Next the user can choose between using objects or a base point. Checking the box that says “pick on screen” allows the user to choose the objects on screen as if it were to be copied. After the command is completed, the multiple items will be converted to a single entity. This can be checked be holding the cursor over the newly created object. Blocks make copying objects easier; there is a branch of block commands.

**insert**: This command is used to bring in previously created blocks. These blocks can be created in the drawing or be imported from a file. After entering the command the insert dialogue box will appear. Any blocks created in the drawing will appear in the dropdown menu, while any blocks from a file can be accessed by pressing “Browse.” In this dialogue box the user can edit the insertion point, scale, and rotation.

**explode**: This command is essentially an undo button for blocks. After entering this command, AutoCAD will ask for an object to explode. The chosen object will return to being a series of individual objects, just like it was before it was turned into a block.
**block editor**  
**bedit:**  This command brings up the block editor dialogue box; all the blocks will appear on the left hand side. Under the parameters tab the user can choose the desired block and edit the points, linear dimension, polar dimension, rotation, and alignment. Under the actions tab, the user can edit the location and scale, as well as stretch, rotate, flip, and array the block. The constraints can be controlled under the constraints tab.

**define attributes**  
**attdef:**  This command allow the user to store data in a block by creating an attribute definition. Creating an attribute allows the user to store data in a block, such as a part number. Entering –attdef will bring up the attribute dropdown option menu that allows the user to toggle modes which include invisible, constant, verify, preset, lock position, annotative, and multiple lines.

**block attribute manager**  
**battman:**  This command allows users to edit the attribute definitions in blocks. Entering the command will bring up the block attribute manager dialogue box. The properties that will show up the dialogue are tag, prompt, default, mode, and annotative. All of these properties can be edited in this dialogue box by double clicking an attribute.
Lab #6: 3-D Modeling

Objective: This lab is designed to teach the users how to use the three dimensional drawing tools.

Comments: The three-dimensional toolbar is not part of the default toolbars. The user must right click on the menu tabs (for example on the annotation tab) and go to “show tabs” then “3D tools.” This will pin the three-dimensional tab to the menu bar. Three dimensional objects will appear as wireframe until they are converted to mesh by using the “convert to mesh” command.

Commands:

- **box**: This command will create a three dimensional box or cube. After typing box, AutoCAD will ask for a starting point and then the opposite corner and finally the height. If the user wants a perfect cube then entering in box (enter) c will create a cube.

- **cylinder**: This command creates a three dimensional cylinder. After entering the command, AutoCAD will ask for a starting point, radius, and height, and then it will create the cylinder. After choosing a starting point the user can change the specified dimension from radius to diameter, 2p, 3p, ttr, or elliptical just like the circle command from lab one.

- **cone**: This command will create a three dimensional cone. AutoCAD will ask for a center location, a radius, and a height. A negative height will create an inverted cone. Just like the cylinder command, instead of specifying a radius, the user can choose diameter, 2p, 3p, ttr, or elliptical.

- **sphere**: This command will create a three dimensional sphere. AutoCAD will ask for a starting point and radius. Again, alternatives to specifying a radius are diameter, 2p, 3p, ttr.

- **pyramid**: This command creates a three dimensional pyramid. The user must choose a starting location, then a radius followed by a pyramid height. A negative height will create an inverted pyramid.
wedge  we:  This command creates a three dimensional wedge, which is a right triangle. To create a wedge, the user must choose a starting location and the opposite corner. Next AutoCAD will ask for a height. **The raised part of the wedge will be on the starting location side**

torus  tor:  This command creates a three dimensional donut. The user must choose a center location and a radius (typing d will let the user specify a diameter, instead of a radius). The next step is to choose a tube radius. In addition to the option to specify a diameter, the user could type in 2point, on the last step and choose to points to determine the thickness.

poly solid  psolid:  Poly Solid is a relatively new command that allows users to create three dimensional walls. Clicking the poly solid icon or entering the command will give the users many options on drawing the wall. Typing in any of these commands before choosing the starting location will enable the user to customize the wall.
  
  - **Object:** this command enables the user to convert a 2D object into a 3D wall
  - **Height:** this command enables the user to change the wall height
  - **Width:** this command enables the user to change the wall width
  - **Justify:** this command enables the user to change justification

  After all the parameters are selected the user will use the mouse to click the wall on the drawing. The wall will continue until the user presses Esc or commands close. The close command will close the wall on itself, provided there are no self-intersecting points. Will using this command, entering arc, will create a curved three dimensional section of the wall.

extrude  ext:  This command turns a two dimensional object into a three dimensional object. After entering the command, the user selects the object(s) to be converted. AutoCAD will then ask for a height, which can be entered manually or chosen with the mouse.

revolve  rev:  This command creates a three dimensional object by revolving a line, arc, or ellipse about an axis. After entering the command, the user selects the object that is to be revolved, then the axis that it will be revolved around. AutoCAD will then ask for the angle of revolution, entering 360 will result in a complete revolution.

loft  loft:  This command creates a three dimensional solid by connecting a series of at least two cross sections. The entities must be on different planes; the objects may be lofted include lines, circles, arcs, and regions.
After entering the command, the user chooses the objects that will be lofted in order. AutoCAD will automatically create the loft.

**sweep**

**sw:** This command creates a three dimensional object by sweeping a profile along a path. The user must first choose the profile that will be swept and then the path that it will follow. Objects that can be swept or be used as a path include arcs, circles, ellipses, and lines.

**convert to meshsmooth:** This command is more easily used by pressing the “convert to mesh” button on the mesh section of the 3D toolbar. This command converts 3D wireframe objects to mesh. The smoothness of the mesh can be increased or decreased by using the buttons to the right of the “convert to mesh” button; the plus (+) increases the smoothness and the minus (-) decreases the smoothness.

**mesh**

**mesh:** This command is a short cut to drawing a mesh 3D object. Instead of drawing a 3D wireframe object and converting it to mesh, typing mesh will bring up a box that allows the user to choose from the seven basic three dimensional shapes: box, cone, cylinder, pyramid, sphere, wedge, and torus. Below all these options is the setting option where the user can adjust the smoothness.
Lab #7: 3-D Editing

Objective: The purpose of this lab is to teach the user how to use the three-dimensional editing tools.

Comments: Three-dimensional drawings in AutoCAD can be tricky, especially when it comes to viewing the drawings. The best way to keep the drawing clear is to use the color face/color lines command (if the drawing is hard to see). The “View Cube” in the top right corner is also very useful in getting different views. The user can choose any of the faces of the cube (front, top, back, etc.) or choose the edges and corners which is helpful when drawing in three-dimensions.

Commands:

press/pull presspull: This command functions similarly to extrude. After entering the command, the user must choose a face and then can choose a new location for the face with the mouse (either pulling it away from its original location or pushing it closer to its original location). The opposite side of the face will not be affected by the relocation of the press/pulled wall.

offset edge offsetedge: This command will create a copy of a face and lock the center points together, while allowing the user to choose the dimensions of the new face. The result is a scaled up or scaled down copy of the original face with a matching center location. The new face can be either larger or smaller than the original face.

fillet edge filletedge: The fillet edge command performs the same function as the normal fillet command from lab one, but does so on three dimensional objects. After entering the command, the user can enter radius, to change the radius (default is 1.000), then choose an edge to see a preview and then press enter to accept the fillet. If the fillet is not right during the preview stage, the user can enter radius again to change the radius.

shell solidedit: The shell command is part of the default three-dimensional toolbar and can be chosen by pressing the icon or by entering solidedit (enter) body (enter) shell (enter); either way brings the same result. After calling the command, the user must select a three-dimensional body. The next step is to click anywhere on the edge of the object and then choose the thickness.
**chamfer edge chamferedge:** The chamfer edge command functions with the same basic principles as the fillet edge command. The user must enter the command then chose an edge; the default slope is 1/1. To change the slope the user must enter **distance** before choosing an edge. AutoCAD will then ask for a distance one, which is the vertical component of the slope, and a distance two, which is the horizontal component of the slope. A preview will appear and pressing enter will accept the chamfer. The chamfer can be edited the same way as the fillet.

**slice**  
**slice:** The slice command can turn a single object into multiple objects. The first thing the user must do after entering the slice command is choose an object to be sliced. AutoCAD will then ask for a starting point and ending point of the slice (the slicing plane must intersect the selected solid). The user then must choose a side to keep or hit **enter** to keep both sides. Keeping both sides will turn the object into two connected, sliced objects. If the user does not press enter and clicks on a side of the side of the slice, that object will remain and the other side of the slice will be deleted.

**union**  
**union:** The union command will turn two three-dimensional objects into a single, composite three-dimensional object. After entering the command the user can select any number of objects to join (as long as they all intersect). After all the objects are chosen, hitting enter will create the composite solid.

**intersect**  
**intersect:** The intersect command will turn multiple three-dimensional objects into a single three-dimensional object by removing the areas that the objects do not have in common. After calling the command, the user must select all the objects are desired for the command. After the objects have been selected, pressing enter will remove the non-intersecting areas, leaving only the new object.
**subtract**

*subtract*: The subtract command will turn multiple three-dimensional objects into a single three-dimensional object by subtracting out selected objects. To use this command the user must enter `subtract` (enter), select the object to subtract from (the object that will remain) press enter and then select the object to subtract (the object that will disappear. The objects in this command must be intersecting or the command will not work.

**solid edit**

*solidedit*: The solid edit command is an umbrella command that has many command underneath it. Entering this command will bring up a selection menu that has the options of face, edge, or body. Each choice has different functions under its command:

- **Face**:
  - Extrude: extends a face in the X, Y, or Z direction
  - Move: moves a face
  - Rotate: rotates a face about an axis
  - Offset: offsets all faces equally
  - Taper: creates a taper at a specified angle
  - Delete: deletes a face
  - Copy: copies a face
  - Color: change a color of a face (overrides layer)
  - Material: choose a material for a face
  - Undo: undoes all actions in current solidedit session

- **Edge**
  - Copy: copies an edge while keeping geometry the same
  - Color: changes the color of an edge
  - Undo: undoes all actions in current solidedit session

- **Body**
  - Imprint
  - Separate Solids
  - Shell
  - Clean
  - Check
  - Undo: undoes all actions in current solidedit session
Lab #8: Problem Solving

Objective: This lab is designed to allow the user to use the mass properties and geometric properties, as well as dimensioning commands, to solve problems such as missing lengths, angles, and areas.

Comments: In addition to the new commands used in this lab, the dimension commands from lab three will be useful.

Commands:

**Mass Properties**

massprop: This command will find the mass properties of three dimensional objects including mass, volume, centroid, and moment of inertia. After entering the command, the user must select a region and press enter. A text box will appear with mass properties including area, perimeter, centroid, and moment of inertia.

**Geometric Properties**

measuregeom: This is a multi-faceted command that allows the user to find several properties including:

- D: Distance
- R: Radius
- A: Angle
- AR: Area
- V: Volume

To find the area and/or volume of an object, type “object” before clicking any points. This will then allow the user to choose a whole object rather than points. For example, to find the area of a circle: type measuregeom→ar→object then click on the circle.

**Adding Areas**

area: This command allows the user to add areas of regions. To perform this command this user must use these following steps:

1. Type area (enter)
2. Type add (enter) object (enter) and then click on the regions needed
3. The net area will appear in the command area

**The objects must be closed regions***
Subtracting Areas

**area**: This command allows the user to subtract areas of objects that are inside other objects. To perform this command this user must use these following steps:

1. Type `area` (enter)
2. Type `add` (enter) `object` (enter) and then click on the larger area (enter)
3. Type `subtract` (enter) `object` (enter) and then click on the internal object (enter)
4. The net area will appear in the command area

**The objects must be closed regions**

NOTE: An alternative way to calculate a subtracted area would be to use the hatch command:

1. Type `bh` (enter) and choose an internal point in the object (enter)
2. Type `area` (enter) `object` (enter) and select the hatching
3. The area and perimeter will appear in the command box
Lab #9: CNC Cutting Pattern Creation

Objective: The objective of this lab is to introduce the students with the process of transferring AutoCAD drawings into a file that is acceptable to be cut on the BRAE department’s CNC plasma cutting machine in lab six, or to be sent out to a vendor for cutting.

Comments: Drawings that are being used on the plasma cutting machine must not have any lines that are on top of any lines, must not have lines that extend through other lines, and all lines must be completely connected to their respective end points. Below are examples of common problems with using AutoCAD drawings with the plasma cutter.

- The line is not long enough to connect with its endpoint. The user may have to zoom in all the way to see this error.
- The line is too long and goes past its endpoint. The user may have to zoom in all the way to see this error.

**Another frequent problem that occurs is having lines or objects directly on top of other lines or objects**
Below is an example of a simple bottle opener that can be designed for this lab.

Commands:

**save**  **saveas:** To use an AutoCAD drawing on a CNC machine, it must be saved as a .dxf file. Entering saveas will bring up the save as dialogue box; underneath the file name line is the “files of type” dropdown list. About half way down the list is AutoCAD [year] DXF (*.dxf), choose this option and press enter to save the file.

**explode**  **text**  **txtexp:** This command explodes multiline text (mtext) from text to lines, so it can be cut using a CNC machine. First the user must have multiline text on the document, then enter the command and click on the text to explode it.
Appendix A- Shortcut Keys
**Standard**

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**Leaders**

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**Modify**

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