Republic of the Philippines
Department of Education
REGION III
SCHOOLS DIVISION OFFICE OF NUEVA ECIJA

LEARNING ACTIVITY SHEET
SPECIAL PROGRAM IN ICT 8
FREEHAND DESIGN 8
Fourth Quarter, Week 5

Name of Learner: _____ Date: _____

Grade Level /Section: _____

Terms and definitions relating to CAD
CAD software and Commands

BACKGROUND INFORMATION FOR LEARNERS

Technical Drafting is the process of accurately creating a visual representation of a design to serve as reference for construction or manufacture. Originally, pen and paper are utilized for the creation of technical drawings. To date, workers should also be competent in the use of computer-aided design and drafting software (CADD) for creating technical drawings.

The use of CADD software allows for the production of technical drawings inn a more sophisticated manner. In addition computing dimensions and making corrections is made easier. Knowledge of technical drafting using CADD software is a skill that is in demand among numerous industries. Many offices now employ its use for the production of technical drawings as it increases proficiency and accuracy that in turn leads to maximized profitability.

Skilled technical drafters proficient in the use of CADD software are widely in demand locally and abroad. Among the specializations in this field include two-dimensional design and drafting along with three-dimensional rendering of landscaping, interior and exterior designs for houses to high-rise buildings or skyscrapers.
Examples of Industry Careers for CADD Operator

- Construction
    - Architectural
    - Engineering

- Manufacturing firms
    - Wood manufacturing of furniture
    - Steel firms that create materials as part of buildings, steel bridges, plumbing fixtures for houses and buildings, sewerages and irrigation structures.
    - Automotive
    - Plastic manufacturing
- Information Technology
    - Hardware for mainboard to chipsets for computer and electronics and communication technology.

Introduction of AutoCAD

The commercial software, AutoCAD, created by the Autodesk Company is the industry leader among 2D and 3D CADD applications. It is widely used by architectural, engineering, and manufacturing offices that require technical drawings for construction and production.

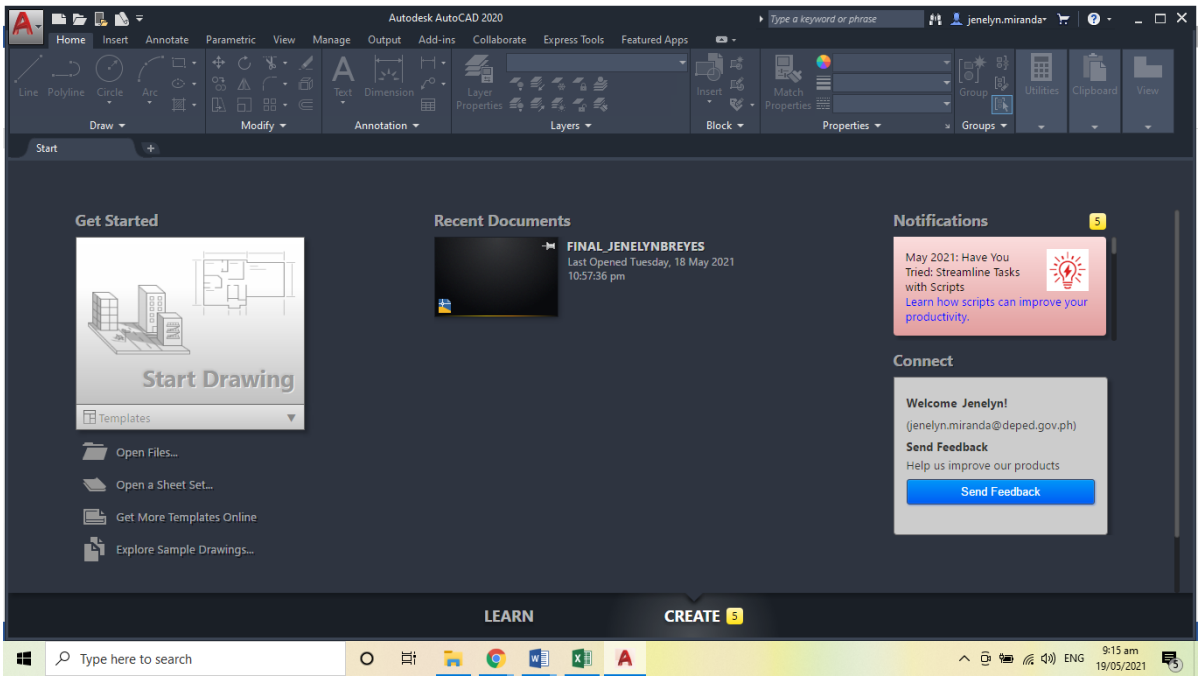
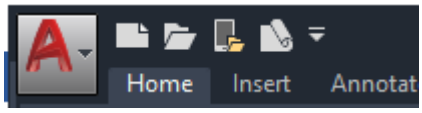


Figure 1.0 AutoCAD 2020 Welcome Screen appears at Startup



In the Welcome Screen, click New to begin a new drawing. A Quick Access toolbar includes familiar commands such as New, Open, Save, Print, Undo, and the like.

Figure 1.1 Quick Access toolbar in the AutoCAD Drawing area

The Command Window (aka Command Line)

At the heart of AutoCAD is the command Window, which is normally docked at the bottom of the application window. The Command Window displays prompts, options, and messages.

Commands can be directly entered at the Command Window instead of using the ribbon, toolbars, and menus.

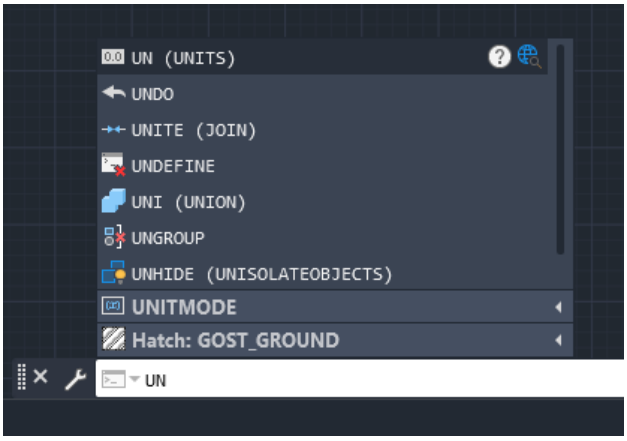
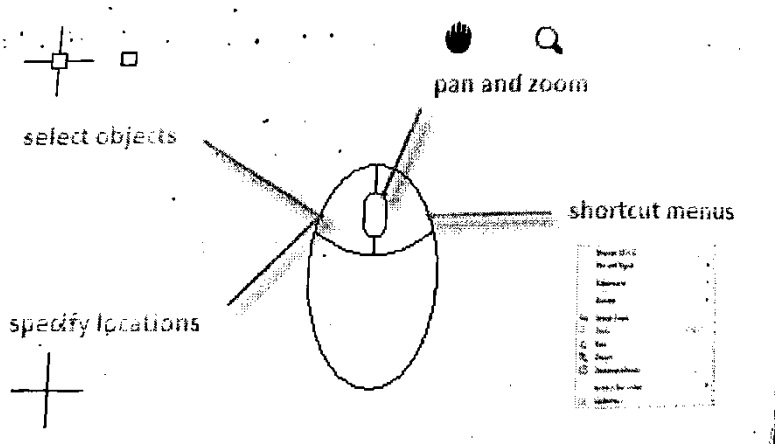


Figure 1.2 Expanded Command Window

This is the preferred method of many long-time AutoCAD users when a user starts to type a command in the window, the program provides adaptive suggestions. When several possibilities are available, click the desired command or use the arrow keys to highlight the command then press ↵ Enter or the Spacebar to execute.

The Mouse

The mouse is the typical pointing device used with AutoCAD though other devices have similar controls. Click the right mouse button to choose an option.
Depending on where the cursor is located, different menus will display relevant commands and options.



Units

Prior to starting the a drawing, it is best to decide what the length of one unit would represent such as an inch, a foot, a centimetre, a kilometre, or some other unit of length.
Unit Display Settings
After choosing the appropriate unit of length for a drawing, use the UNITS command to control unit display settings that include the following:

- Format (or Type). For Example, a decimal length of 6.5 can be set to be displayed as a fractional length of 6-1/2 instead.
- Precision. For example, a decimal length of 6.5 can be set to be displayed as 6.50, 6.500, or 6.500

To use feet and inches, set the unit type using the UNITS command to Architectural. Then when creating objects, be sure to specify lengths in inches. To use metric units, set the unit type using the UNITS command to Decimal. Changing the unit format and precision does not affect the internal precision of a drawing. It only affects how lengths, angles, and coordinates are displayed in the user interface.

If it is necessary to change the UNITS settings, be sure to save the drawing as a drawing template file. Otherwise, changing UNIT settings for each new drawing would be required.

Model Scale
Always create models at full size, i.e., 1:1 scale. The term model refers to the geometry of a design. A drawing includes the model geometry along with the views, notes, dimensions, callouts, tables, and the title block displayed in the layout.

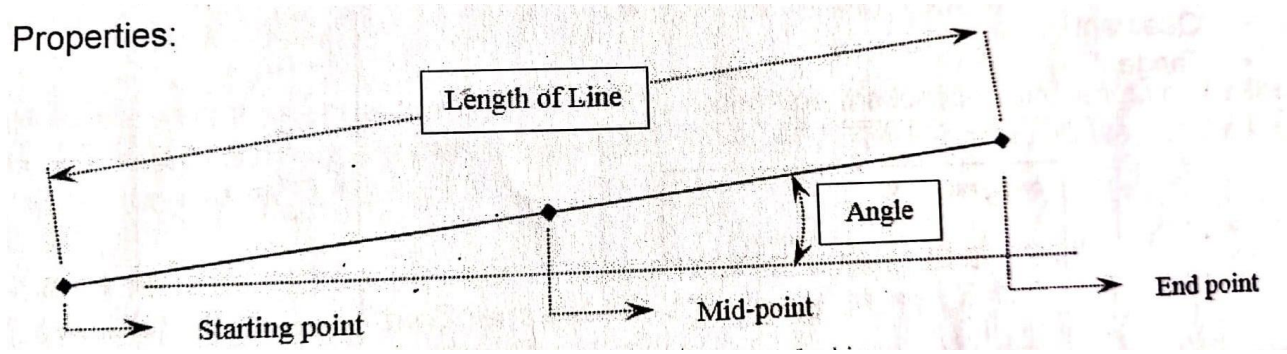
Also, the scaling that is necessary to print a drawing on a standard-sized sheet can be specified later on when the layout is created.

Helpful shortcuts

- Press F1 to open Help for information about the command in progress.
- Press ↵ Enter or the Spacebar to repeat the previous command.
- Select an object and right-click or right-click a user interface element to see varous options related to the selection.
- Press the Escape Key (Esc) to cancel a command in progress.

Geometric Drawing Commands

The **LINE (L) command** is used to draw a single line or a series of lines from one point to another as individual lines.

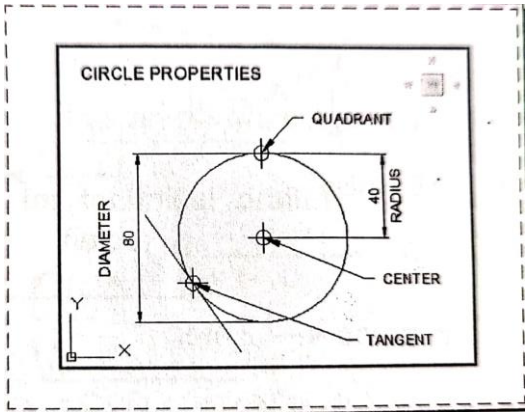


Step by Step:
Type **LINE** or **L** at the Command Windows and press ↵ Enter
Specify first point: x1, y1
Click at any point or use absolute coordinates (x1, y1) to specify a point in the drawing area
Specify next point or [undo]: xn, yn
Click at any point or use absolute coordinates (xn, yn) to specify the next point in the drawing area.
Specify next point or [undo]: xn, yn
Specify another point or press ↵ Enter to terminate the Line command
Note: It is possible to create as many lines as needed in AutoCAD or to undo a given point by typing Undo (U) on the Command Window after each point or undo a drawn line after terminating the Line command.

Circle Command
The **CIRCLE (C)** command is used to draw a circle with specified properties like Center and Radius, Center and Diameter, Two Points, Three Points, Two Tangential Points and Radius, or Three Tangential Points.
Circle Menu Properties:

- Center
- Radius

- Diameter
- Quadrant
- Tangent



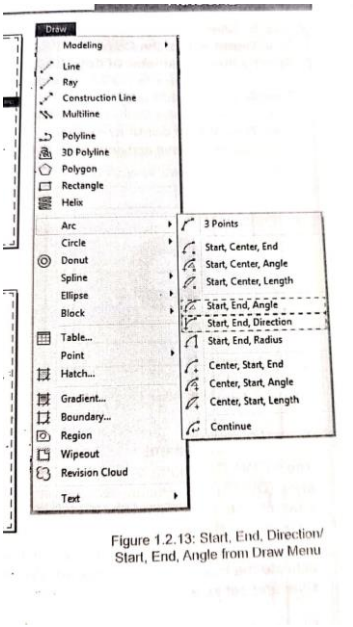


Figure 1.2.13: Start, End, Direction/ Start, End, Angle from Draw Menu

Step by step:

Draw a Circle using CENTER and Radius

Type **CIRCLE** or C at the Command Window and press ↵ Enter

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: x_1, y_1

Click at any point or use absoluate coordinates to specify (x_1, y_1) point in the drawing area.

Specify radius of circle or [Diameter] <39.4231>: 40

Type 40 at the Command Window and press ↵/Enter

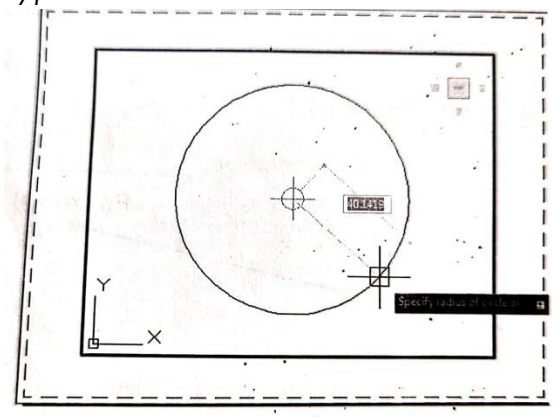


Figure 1.2.5: Sample drawing of a circle

Ellipse Command

The **ELLIPSE** Command creates an ellipse using its geometric properties.

Properties:

- Center
- Minor Radius
- Major Radius
- Quadrants
- Major Axis
- Minor Axis
- Rotational Angle

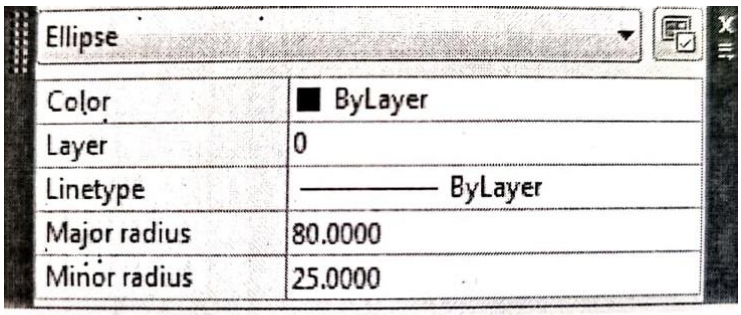
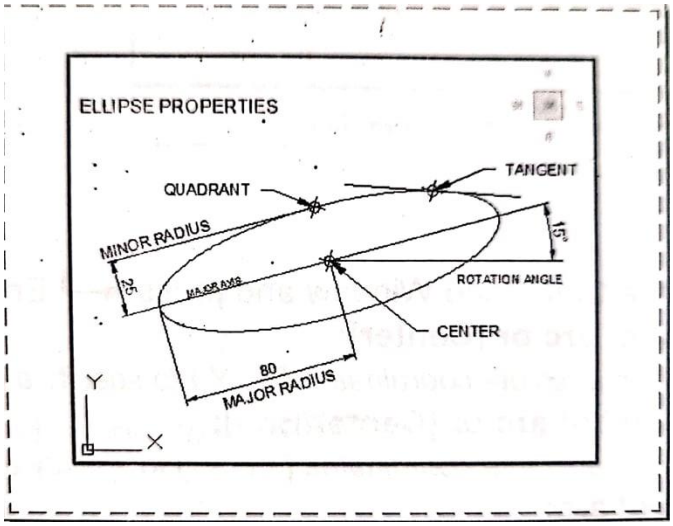


Figure 1.2.6: Ellipse Menu Properties

When creating an ellipse, the first two points determine the location and length of the first axis. The third point determines the distance between the center of the ellipse and the end point of the second axis.



Step by Step:
Type **ELLIPSE** or EL at the Command Window and press ↵ Enter

Specify axis endpoint of ellipse or [Arc/Center]: 80
Click at any point in the drawing area or use absoluate coodinates (x1, y1) to specify a point in the drawing area.

Specify distance to other axis or [Rotation]: 25
Type a value or click a point in the drawing area to specify the distance to the other axis then press ↵ Enter

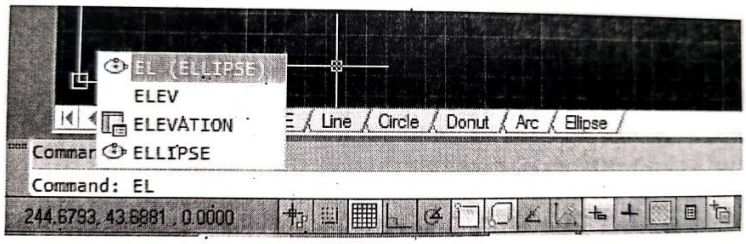


Figure 1.2.8: Ellipse (EL) Command

ARC Command

The **ARC (A)** Command is used to draw an arc as part of a circle using the following properties.

Properties:

- Center
- Start
- End
- Radius
- Angle
- Length

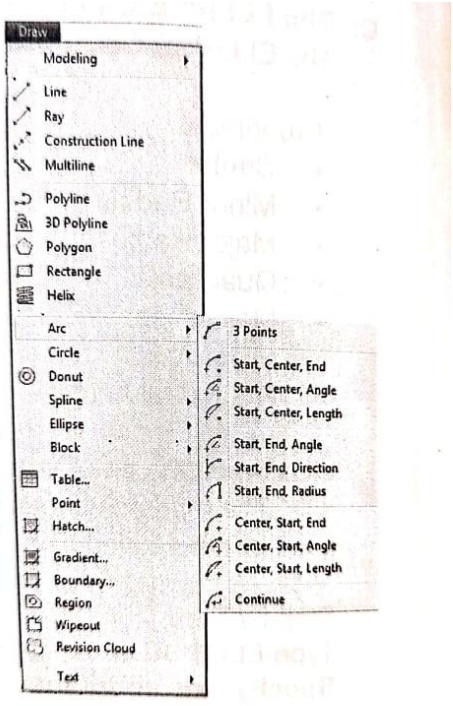


Figure 1.2.9: Arc from Draw Menu

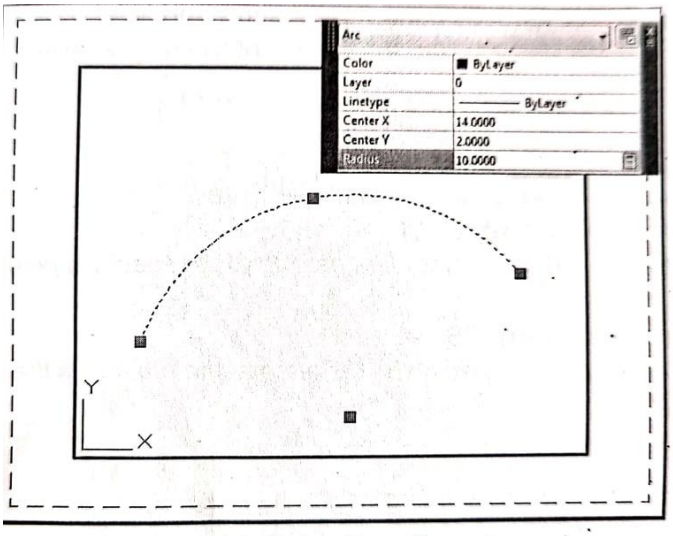


Figure 1.2.10: Arc Menu Properties

Step by Step:

Type ARC or A at the Command Window and press ↵ Enter

Specify start point or arc or [Center]:

Click at any point or use absolute coordinates (x1, y1) to specify a point in the drawing area

Specify end point of arc:

Click at any point or use coordinates to specify a point in the drawing area and press ↵Enter

Using the mouse

Start, End, Direction

- Click Menu > Draw > > Arc > Start, End, Direction
- Click points 1, 2, 3

Donut Command

The **DONUT (D)** Command is used to draw a filled or hollow donut.

Properties:
Center
Inside Diameter
Outer Diameter

Note: If the value of the inside diameter is zero then it is filled.

If the specified value of the inside diameter is greater than the outside diameter, the values will interchange for both dimensions.

Step by Step:

Type **Donut** or D at the Command Window and press ↵ Enter

Specify inside diameter of donut: 10

Type a numerical value for the inside diameter and press ↵Enter

Specify outside diameter:

Type of numerical value for the outside diameter and press ↵Enter

Specify center of donut or <exit>:

Specify center of donut and press ↵/Enter

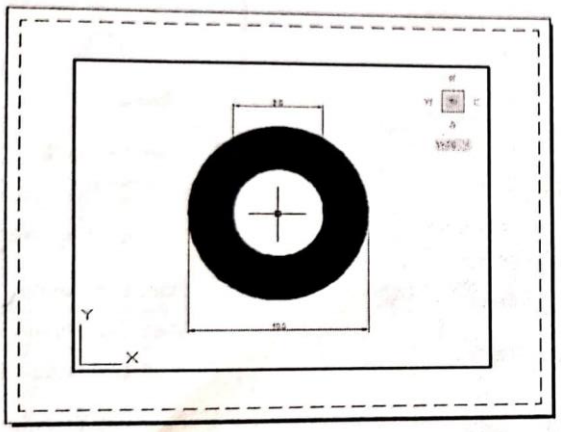


Figure 1.2.14: Donut Properties

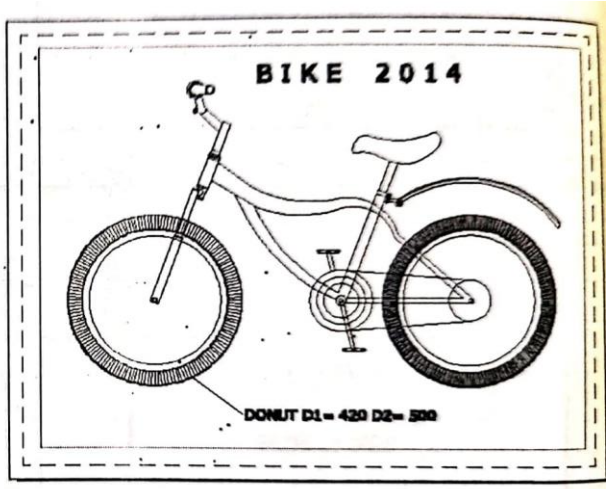


Figure 1.2.15: Sample drawing of a donut

Point Command

The Point command is used to draw a point using a defined point style (DDPTYPE). Points can act as nodes to which a user can snap objects to. A full three-dimensional location can be specified for a point. The current elevation is assumed if the Z coordinate value is omitted. Type DDPTYPE on the Command Window to activate the Point Style Manager from where users can select point style and set size.

Step by Step:

Type **POINT** at the Command Window and press ↵ Enter

Specify a point:

Current point modes: PDMODE=99 PDSIZE=-10.0000

Specify location of point

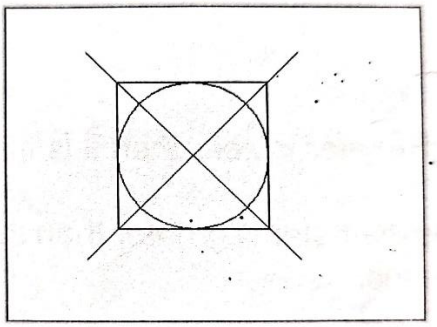


Figure 1.2.17: Sample point in drawing area

RECTANGLE Command

The **RECTANGLE (RECTANG)** command is used to specify the measurements of a rectangle in terms of length, width, and rotation. It also controls the type of corners, i.e., fillet, chamfer, or square.

Properties:

- First Corner
- Other Corner
- Area

- Total Length
- Global Width

Properties:
- First Corner
- Other Corner
- Area
- Total Length
- Global Width

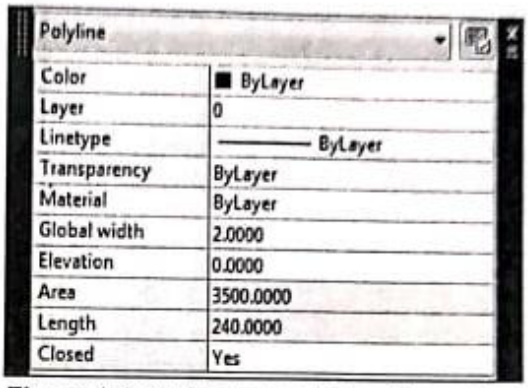


Figure 1.4.12: Rectangle Menu Properties

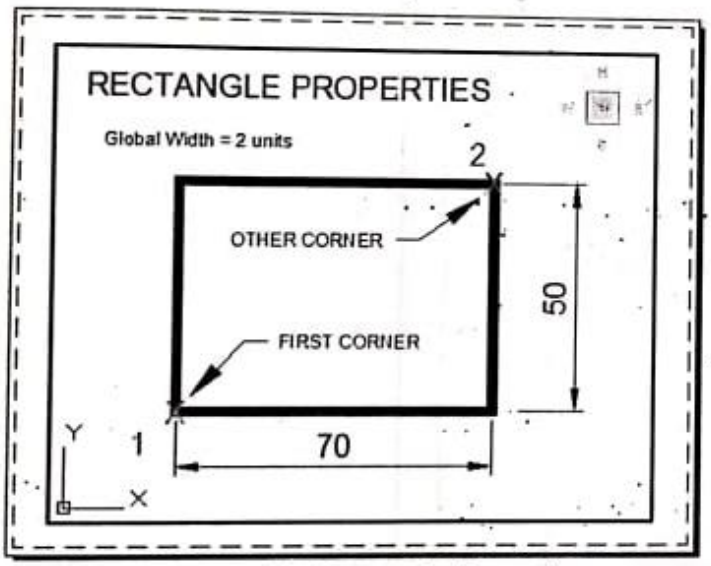


Figure 1.4.13: Rectangle Properties

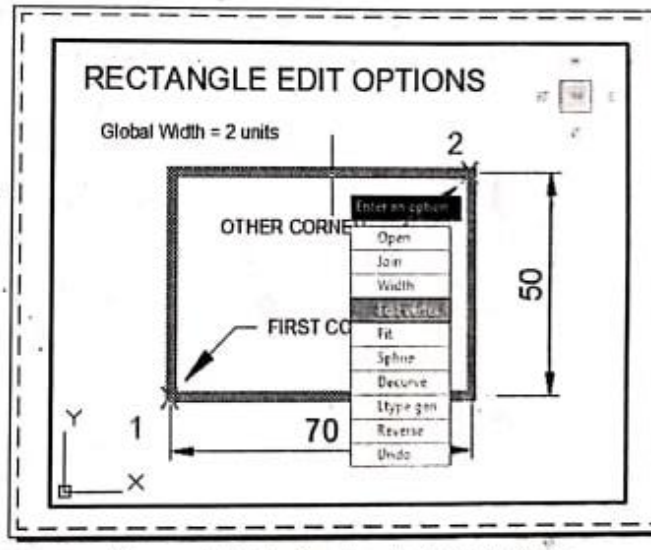


Figure 1.4.14: Rectangle Edit Options

Step by Step:

Type **RECTANGLE or RECTANG at the Command Window and press** ↵ Enter
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]:
Click at any point or use absolute coordinates to specify (x1, y1) point on the drawing area.
Specify other corner point or [Area/Dimensions/Rotation]:

Specify other corner point and then press ↵Enter

Note: Double-click a rectangle to display the Rectangle Edit Options.

Basic Modify Commands

ERASE or DELETE Command

The **ERASE (E)** or DELETE command is used to remove one or more drawing entities.

Step by Step:
Type **ERASE** or **E** in the Command Window and press ↵ Enter.
Select Objects:
Select an object or press CTRL + A to select all objects.
1 found/all object selected
Select objects
Press ↵ Enter to complete the object selection.
Using Mouse and Keyboard Delete Button:
Select or click object/s or CTRL + A, and press the Delete button.

COPY Command

The COPY (CP) command is used to duplicate one or more drawing entities at another location within the drawing without erasing the original entity.
Step by Step:
Type **COPY** or **CP** in the Command Window and then press ↵ Enter
Select objects:
Select the circle inside the box.
1 found
Select objects
Press ↵ Enter to complete the object selection.
<Base point or displacement>/Multiple:
Type M (for multiple copies) at the Command Window and press ↵ Enter
Base point:
Use object snap to select the center point of the circle.
Specify second point or displacement:
Use object snap to select the midpoint of the outer line.
Second point or displacement:
Repeat the preceding step three or more times to create a copy of the circle to the midpoints of the other outer lines as well and the four corners of the Rectangle.

MOVE Command

The **MOVE (M)** makes it possible to relocate an entity from its original position to another point in the drawing area.

Step by step:

Type MOVE or M at the Command Window and press ↵ **Enter**
Select objects:
Select the circle inside the box.
1 found
Select objects
Press ↵ Enter to complete the object selection.
Base Point:
Use subject snap to select the midpoint of the outer line.
Second point or displacement:
Use object snap to select the midpoint of the outer line
Second point or displacement:
Repeat the preceding step three or more times to move the circle to the midpoints of other outer lines as well.

BREAK Command

Use the **BREAK (BR)** command to remove part of an object by indicating two break points. AutoCAD will erase the portion between the first and the second points. If the second point is not on the object, AutoCAD will select the nearest point on the object.

To split an object into two without erasing any portion of it, enter the same point for the first and second points, this can be done by entering @ to specify the second point. AutoCAD converts a circle to an arc by removing a piece of the circle starting counter-clockwise from the first to the second point.

Step by step:

Type **BREAK** or BR at the Command Window and press ↵ Enter
Select objects: (select the object to break)
Select the outer circle from the existing design.
Enter second point (or F for first point):

Type F (for the first point) and press ↵ Enter
Enter second point:
Use object snap (OSNAP) to select the other intersection point of the line and circle.

EXPLODE Command

The EXPLODE (X) command breaks a compound object into its component objects. A compound object actually comprises more than one AutoCAD object. For example, a block is a compound object. The command can be applied to 3D meshes, 3D solids, blocks, bodies, dimensions, Mutlilines, Polyface Meshes, Polygon meshes, Polylines, and Regions. The results would differ depending on the type of compound object to explode.

Step by Step:

Type **EXPLODE** or X in the Command Window and press ↵ Enter
Select objects: (pick a drawing entity)

Select the STAR object and press ↵ Enter

FILLET Command

The **FILLET (F)** command connects two intersecting lines, arcs, or circles using a smooth arc with a specific radius. It adjusts the lengths of the original lines or arcs so that they end exactly on the fillet arc. If the Polyline option is used, the fillet command can be applied to the entire Polyline, or remove the fillet(s) from a Polyline.

Step by Step:

Type **FILLET** or F in the Command Window and press ↵ Enter
(TRIM mode) Current fillet radius = 0.00
Polyines/Radius/Trim/<Select objects>:

Type R (to specify fillet radius) in the Command Window and press ↵ Enter
Enter fillet radius<0.00>:

Type 10 in the Command Window and press ↵ Enter
Command:

Press ↵ Enter to repeat the previous command.
FILLET:
(TRIM mode) Current Fillet radius = 0.00
Polyline/Radius/Trim/<Select objects>:
Select the inner-left vertical line.
Select second object:
Select the inner-top horizontal line.
Command:
Repeat the last three steps to smoothen the other edges of the Star drawing as well.

TRIM Command

Trims objects at a cutting edge defined by other objects or press enter to select all objects as potential cutting edges. Objects that can be trimmed include arcs, circles, elliptical arcs, lines, open 2D and 3D Polyline, rays, and splines.

Step by Step:

Type **TRIM** or **TR** in the Command Window and press ↵ Enter
Select cuttings edges: (Projmode = UCS, Edgemode = No extend)
Select objects:

Type all (to select all lines) at the Command Window and press ↵ Enter
47 found
Select objects:

Press ↵Enter to complete the object selection.
<Select object to trim>/Project/Edge/Undo:
Click the unwanted edges to remove them from your existing design
<Select object to trim>/Project/Edge/Undo:

Once the unwanted edges are removed, press ↵ Enter to exit the Explode command.

OFFSET Command

The OFFSET (O) command is used to duplicate one or more drawing entities at a specified measure of displacement without erasing the original.

Step by Step:

Type **OFFSET** or O in the Command Window and press ↵ Enter
Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0
Specify offset distance or [Through/Erase/Layer] <through>:3

Input value for offset distance (3) and press ↵Enter
Select object to offset or [Exit/Undo]<Exit>:
Select side/s of PENTAGON
Specify point on side to offset or [Exit/Multiple/Undo]<Exit>:
Click outside of PENTAGON

Select object to offset or [Exit/Undo]<Exit>:
Select RECTANGLE
Specify point on side to offset or [Exit/Multiple/Undo]<Exit>:
Click Outsideof RECTANGLE
Select object to offset or [Exit/Undo]<Exit>:
Repeat the last step for all ARC and CIRCLE.
Select HALF CIRCLE
Specify point on side to offset or [Exit/Multiple/Undo]<Exit>:
Click inside of HALF CIRCLE

Repeat the last step for all HALF CIRCLES and press ↵Enter to terminate the command.

LEARNING COMPETENCY

- Terms and definitions relating to CAD
- CAD software and Commands

Activities

I. Identification: Write the correct answer in a one whole sheet of paper.

1. _____ is the process of creating visual representations of designs accurately as reference for construction or manufacture.
2. The use of CADD, which is short for, _____, allows for the production of technical drawings in a more sophisticated manner.
3. Careers for technical drafters who use CADD software is prevalent in the following fields: _____, _____, and _____.
4. _____ is a commercial software created by the AUTODESK Company for CAD applications.
5. The _____ offers easy access to familiar commands like New, Open, Save, Print, Undo, etc.
6. Commands can be directly entered at the _____ instead of using the ribbon, toolbars, and menus.
7. _____ is the file type associated with AutoCAD drawing files.
8. The term _____ refers to the geometry of a design.
9. Press the _____ or _____ to repeat a previous command.
10. Press the _____ to cancel a command in progress.

II. True or False: On the space provided, write I if the statement is True. Write F if the statement if False.

1. The Line Command is used to draw a single line or a series of lines from one point to another.
2. The circle command draws circles even without specified properties.
3. Diameter is a property of ellipses.
4. When drawing ellipses, the third point determines the distance between the center of the ellipse and the end point of the second axis.
5. Angle and length are properties related to drawing an arc.
6. F1 is used to open help for information about the command in progress.

7. The values for the inside diameter and outside diameter of a donut will interchange if the dimension of the inside diameter is greater than the outside diameter.
8. A value of zero for the inside diameter will create a filled donut.
9. The point command draws a point using a defined point style.
10. Point cannot act as nodes to which an object can snap to.

III. Reflection: Write your answer in a one whole sheet of paper.

If you are going to be one of the command in AutoCAD, what would you be and why?

REFERENCES FOR LEARNERS

- MICROCADD Institute Comprehensive AutoCAD Student Manual 2016
- CAL Exploring ICT Technical Drafting using AutoCAD 2015

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