The Move command will cut out the drawing entities and allows you to paste it to a new location.

Steps:
- Type MOVE at the command prompt;
- Select objects:
- Specify base point or [Displacement]
- Specify second point or <use first point as displacement>: 

The COPY command is used to duplicate one or more existing drawing entities to another location without erasing the original.

The Move command will cut out the drawing entities and allows you to paste it to a new location.

Steps:
- Type COPY at the command prompt
- Select objects: Select the objects using the desired selection method
- Specify base point or displacement, [Multiple]: (M for multiple)
- Specify base Point: (Pick a point)
- Second point of displacement: (Pick second point)
**FILLET**

- The Fillet command connects two lines, arcs, or circles with a smooth arc of specified radius.
- It adjusts the lengths of the original lines or arcs so they end exactly on the Fillet arc.
- If the Polyline option is used, you can apply fillets to an entire polyline or remove the fillets from a polyline.

**Steps**:
- Type `FILLET` at the command prompt
- Polyline/Radiu3/<select two objects>: (enter R to change the fillet radius)
- Fillet Radius: (enter 1)
- Polyline/Radiu3/<select two objects>: (select the first object)
- Select second object: (select the second object)
- Enter "P" to fillet an entire Polyline. Enter "R" to set the fillet radius. Otherwise select two entities to fillet.

**CHAMFER**

- This command trims two intersecting lines with a specified distance from the intersection and connects the trimmed ends with a new line segment.

- Chamfering Two Lines - This option trims two intersecting lines.
  
  **Steps**:
  - Type `CHAMFER` at the command prompt
  - Polyline/Distances/<Select first line>: (type D to select the chamfer distance)
  - Enter first chamfer distance <current>: (enter distance)
  - Enter second chamfer distance <current>: (enter distance)
  - Polyline / Distances / <Select first line>: (select the first line)
  - Select second line: (select the second line)
OBJECT SNAP [F3]

- Lists object snaps
  - **Endpoint** - Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, spline, region, or ray, or to the closest corner of a trace, solid, or 3D face.

  ![Endpoint Object Snap Diagram]

  selection point
  snap point

  To track from an Snap point, pause over the point while in a command. A tracking indicator appears when you move the cursor. To stop tracking, pause over the point again.

- **Midpoint** - Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, region, solid, spline, or xline.

  ![Midpoint Object Snap Diagram]

  selection point
  snap point
LIST OF LAYERS

- Displays layers and their properties.
- To modify a property, click its icon.
- Names
  - Displays the names of the layers.
- On/Off
  - Turns layers on and off. When a layer is on, it is visible and available for plotting.
- Freeze/Thaw in All Viewports
  - Freezes selected layers in all viewports.
  - freeze layers to speed up ZOOM, PAN, and many other operations; improve object selection performance; and reduce regeneration time for complex drawings.
- Lock/Unlock
  - Locks and unlocks the layers. You cannot edit objects on a locked layer.
  - For reference
- Color
  - Changes the color associated with the selected layers.
- Linetype
  - Changes the linetype associated with the selected layers.
- Lineweight
  - Changes the lineweight associated with the selected layers.
- Plot Style
  - Displays a list of plot styles available to apply to the selected layers.

CREATING LAYERS

- Lines, circles and text are examples of drawing objects.
- In addition to its geometric positions, each object has an associated layer, linetype and colour.
- Normally, the colour and linetype are inherited from the layer on which the object is drawn.
- Therefore, creating of layers is the most important step in organising your drawing.
- These four object properties when creating a layers:
  - Colour;
  - Linetype;
  - Line weight; and
  - Plot style.
EXERCISE7F****

- Draw object using different layer
  - Object (Colour: Black, lineweight: 0.75, linetype: continuous)
  - Dimension line (Colour: red, lineweight: default, linetype: default)
  - Centerline (colour: yellow, lineweight: 0.5, linetype: ISO dash)

OUTCOMES

- Applying drawing tools and setting
- Explain the use of edit commands
- Prepare to make drawings using drawing aids command
- Explain all characteristics about layers
- Explain all characteristics about block

8.5 BLOCK
TO CREATE A BLOCK DEFINITION

- From the Draw menu, choose Block Make.
- In the Block Definition dialog box, enter a name for the block.
- Under Objects, choose the Select Objects button to use the pointing device to select objects for the block definition.
- The dialog box closes temporarily while you select objects for the block. Press ENTER when you are done selecting blocks.
- The dialog box reopens.
- If you want to create a selection set, use the Quick Select button to create or define a filter for your selection set.
- Under Objects, specify whether to retain, convert to a block, or delete the selected objects.
  - Retain: Keeps selected objects in the current drawing, in their original state.
  - Convert to Block: Replaces selected objects with an instance of the block.
  - Delete: Removes selected objects after the block is defined.
- Under Base Point, enter the coordinate values for the insertion base point or choose the Specify Insertion Base Point button to use the pointing device.
- Under Description, enter text to help identify the block for easy retrieval.
- Under Icon, specify whether to create an icon from the block definition.
  - Do not include an icon: Omits preview image from block definition.
  - Create icon from block geometry: Saves preview image with the block definition.
- Choose OK.
- The block definition is saved in the current drawing.
You can insert blocks or entire drawings into the current drawing with `INSERT`. When you insert a block or drawing, you specify the insertion point, scale, and rotation angle. When you insert an entire drawing into another drawing, AutoCAD treats the inserted drawing like any other block reference. Subsequent insertions reference the block definition (which contains the geometric description of the block) with different position, scale, and rotation settings. If you change the original drawing after inserting it, the changes have no effect on the inserted block. If you want the inserted block to reflect the changes you made to the original drawing, you can redefine the block by reinserting the original drawing. If you want to insert a drawing but you don't want the file name to be the same as the block name, you can insert it using a `blockname = filename` syntax at the Block Name prompt.

From the Insert menu, choose Block. In the Insert dialog box, specify the block name, where you want to insert it in the current drawing, and whether you want to explode it after insertion. If you have modified the original drawing file for a block, you can redefine the block in the current drawing by choosing File to locate the file for the block. Choose OK. The block reference is updated in the current drawing.