1. UNITS

- **Description**: The UNITS command allows you to set the drawing units in AutoCAD. This includes selecting the type of unit (e.g., architectural, decimal, engineering, etc.), setting precision, and configuring angles for your design.

2. LIMITS

- **Description**: The LIMITS command sets the boundaries of the drawing area. It establishes a rectangular space within which you can work and set a maximum drawing size. It's helpful in defining the area for drafting in larger-scale projects.

3. NAVIGATION

- **Description**: Navigation tools in AutoCAD allow you to move around and zoom in/out on your drawing. Common navigation tools include the ZOOM, PAN, and 3DORBIT commands to control the view and perspective of your workspace.

4. OSNAP (Object Snap)

- **Description**: OSNAP is used to snap to exact points on objects while drawing or editing. For example, you can snap to endpoints, midpoints, centers, intersections, and other significant geometric locations to ensure precision.

5. ORTHO

- **Description**: ORTHO restricts cursor movement to horizontal and vertical directions only. This ensures that lines are drawn perfectly parallel to the x- and y-axes. It's particularly useful for creating straight-line drawings.

6. UCS (User Coordinate System)

- **Description**: The UCS command allows you to define a custom coordinate system. This can be especially useful in 3D modeling when you want to rotate or move objects in specific directions. It controls how the x, y, and z axes are oriented in your drawing.

7. F-KEYS (Function Keys)

- **Description**: Function keys (F1 to F12) in AutoCAD provide shortcuts for common commands:
    - F1: Help
    - F2: Toggle text window
    - F3: Toggle OSNAP
    - F4: Toggle 3D OSNAP
    - F5: Toggle Isoplane
    - F6: Toggle coordinate display
    - F7: Toggle grid
    - F8: Toggle ORTHO
    - F9: Toggle snap mode
    - F10: Toggle polar tracking
    - F11: Toggle object snap tracking
    - F12: Toggle dynamic input

8. LINE

- **Description**: The LINE command allows you to draw straight lines between two or more points. It is one of the most basic and frequently used commands for creating geometric shapes.

9. CIRCLE

- **Description**: The CIRCLE command allows you to draw circles. You can create a circle by specifying its center point and either the radius or diameter.

10. ARC

- **Description**: The ARC command is used to create an arc, a part of a circle, by specifying three points: a start point, a second point on the arc, and an endpoint.

11. POLYLINE

- **Description**: The POLYLINE command creates a connected sequence of straight or curved line segments as a single object. It allows for easier editing and manipulation compared to individual lines.

12. DIMENSIONING STYLE

- **Description**: The DIMSTYLE command allows you to create and modify dimension styles. It lets you control how dimensions appear in your drawings (e.g., text size, arrow size, units, etc.).

13. LINE TYPE AND ITS PROPERTIES

- **Description**: Line types control the appearance of lines in AutoCAD (e.g., solid, dashed, dotted). You can assign different line types to different objects and manage their properties such as color, thickness, and pattern scale.

14. MOVE

- **Description**: The MOVE command moves selected objects from one location to another within the drawing. It is typically used with precision by specifying base and destination points.

15. ROTATE

- **Description**: The ROTATE command allows you to rotate selected objects around a base point by a specified angle.

16. TRIM

- **Description**: The TRIM command cuts objects along the edges of other objects. It allows you to remove parts of an object that extend beyond the boundaries defined by other intersecting objects.

17. COPY

- **Description**: The COPY command creates a duplicate of selected objects and places them elsewhere in the drawing. It's used for creating copies of objects without moving the original.

18. ERASE

- **Description**: The ERASE command deletes selected objects from the drawing. It's the equivalent of the delete function.

19. MIRROR

- **Description**: The MIRROR command creates a mirrored copy of selected objects across a specified axis. It's helpful for creating symmetrical drawings.

20. SCALE

- **Description**: The SCALE command enlarges or reduces the size of selected objects based on a scale factor. It allows you to change the proportion of objects while keeping their original shape.

21. FILLET

- **Description**: The FILLET command creates a rounded corner between two objects (usually lines or arcs) by specifying a radius for the arc that joins them.

22. CHAMFER

- **Description**: The CHAMFER command creates a beveled edge between two objects. It works similarly to FILLET, but instead of rounding the corner, it creates a straight slanted line.

23. ARRAY

- **Description**: The ARRAY command creates multiple copies of objects in a specified pattern. There are three types of arrays: rectangular, polar (circular), and path arrays. They allow for repetitive patterns, such as rows of objects or circular arrangements.