AutoCAD Dimensioning Commands
## Dimensioning Commands

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The **DIMLINEAR** Command
DIMLINEAR Command

- Use DIMLINEAR command for
  - Length
  - Width
  - Depth

- Use DIMLINEAR for
  - Linear dimensions – (X,Y,Z)
    - Horizontal
    - Vertical
    - Slanted surfaces
**DIMLINEAR Command**

- All dimension components are placed automatically by AutoCAD.
  - Extension lines.
  - Dimension lines.
  - Dimension text and arrowheads.
DIMLINEAR Command

To DIMLINEAR command:

Pick the **Linear Dimension** button on the Dimension toolbar.

**OR**

Pick **Linear** from the **Dimension pull-down menu**.

**OR**

Type **DLI** or **DIMLINEAR** at the **Command** prompt.
**DIMLINEAR Command**

- **Command Sequence:**
  - Command: DLI or DIMLINEAR
  - Pick First extension line origin.
  - Pick Second extension line origin.
  - Pick Location of dimension.
First Extension Point

10’
7’
18’
Second Extension Point

DIMALIGNED
Specify first extension line origin or <select object>: <Osnap on>
Specify second extension line origin: <Ortho off>

10'  7'  18'
Dimension Location

Extension: 2'-0" < 90°
**DIMLINEAR Command**

- When picking extension line origins:
  - Use one of the **OSNAP modes** for **accuracy**.
  - **ENDpoint** is a good one to use.
**DIMLINEAR Command**

- The **DIMLINEAR** command allows you to generate dimensions that are:
  - Horizontal.
  - Vertical.
  - Aligned with a slanted surface.
  - Rotated at a specified angle.
DIMLINEAR Command

◆ Command: DIMLINEAR

◆ The following prompt appears:

Dimension line location (Mtext/Text/Angle/Horizontal/Vertical/Rotated)

◆ Notice submenu choices

◆ Mtext
◆ Text
◆ Angle
◆ Horizontal
◆ Vertical
◆ Rotated
**DIMLINEAR Command**

- **Dimension line location**
  - Is the default option selection.
  - Click the location where you want the dimension line.
**DIMLINEAR Command**

- **Mtext**
  - Accesses the Multiline Text Editor.
  - The `< >` (Chevrons) indicate the current dimension value.
  - The `< >` is an intelligent value.
  - Normally, you should **NOT** delete the `< >`.
  - If you want the dimension changed, delete `< >`. 

![Multiline Text Editor](image)
MTEXT Option

Multiline Text Editor

Modify character properties.
DIMLINEAR Command

- **Text** *(Uses DTEXT)*
  - Uses the Command Line to change the dimension text.

- **Angle**
  - Allows you to change the dimension text angle.

- **Horizontal**
  - Sets dimensions to horizontal distances only.

- **Vertical**
  - Sets dimensions to vertical distances only.

- **Rotated**
  - Allows you to change the angle for dimension line.
Text Option chosen

Text added

Command:
Dimension text = 7'-0"
Angle

Dimension text = 7'-0"
Vertical

5'

Horizontal

7'
Rotated

8'-5 13/16"
The DIMALIGNED Command
Dimensioning Angled Surfaces

- **DIMALIGNED**
  - Use for *dimensioning a surface drawn at an angle*.

- **DIMALIGNED**
  - Functions similarly to the **DIMLINEAR** command
  - **aligns** dimension lines with the **surface**.
Dimension text = 8' - 7 1/4"
The QDIM Command
The QDIM Command

- For DLI and DAL,
  - You must pick the extension line origins.

QDIM
- Lets you pick a geometry
  - Circle, Rectangle, etc.
- Extension line origins are AUTOMATICALLY selected.
The DIMANGULAR Command
**DIMANGULAR Command**

- Use **DIMANGULAR** command to
  - Dimension the **included angle** of an **arc**.
  - The arc’s **center point** becomes the **angle vertex** and
  - the **two arc endpoints** are the **origin**.
Origin

Endpoint

42°

Origin

Endpoint
**DIMANGULAR Command**

- Use **DIMANGULAR** command to dimension part of a circle.
  - First pick point on a circle becomes the **endpoint** of the **first extension line**.
  - Second pick point becomes the **endpoint** of the **second extension line**.
Origin

Endpoint

Origin

Endpoint

Dimension text = 70
Command: <Snap on>
Command:
DIMANGULAR Command

Access the DIMANGULAR command by:

- Pick the Angular Dimension button on the Dimension toolbar
- Pick Angular in the Dimension pull-down menu
- Type DAN or DIMANGULAR at the Command: prompt.
**DIMANGULAR Command**

- **If there is enough room**, AutoCAD will
  - Place the
    - Dimension text
    - Dimension line arc, and
    - Arrowheads *inside* the the extension lines.
Specify second angle endpoint: <Ortho off>
Specify dimension arc line location or [Mtext/Text/Angle]:
**DIMANGULAR Command**

- **If there is not enough room** AutoCAD will
  - Place the **arrowheads outside** and
  - Place the **number outside** the dimension lines.

- **If space is very small, ALL will be placed outside.**
Specify second angle endpoint: <Ortho on>
Specify dimension arc line location or [Mtext/Text/Angle]:

66°
Specify dimension arc line location or [Mtext/Text/Angle]:
Dimension text = 36
Command:
Basic Dimensioning Practices

36°
Datum Dimensioning
DIMBASELINE
Datum Dimensioning

In Datum or Baseline dimensioning, dimensions originate from common surfaces, centerlines or center planes.
Datum Dimensioning

Datum Dimensioning is commonly used in mechanical drafting because each dimension is independent of the others.

AutoCAD refers to Datum Dimensioning as DIMBASELINE or DBA
Chain Dimensioning
DIMCONTINUE
Chain Dimensioning

- Chain dimensioning
  - Is rarely used in mechanical drafting.
  - has less accuracy, since each dimension is dependent on the previous dimensions.
Chain Dimensioning

- If chain dimensioning is used, include an overall dimension to help insure accuracy.

- AutoCAD refers to chain dimensioning as DIMCONTINUE.
Basic Dimensioning Practices
Chain Dimensioning

Chain Dimensioning is commonly used in architectural drafting.

Architectural drafting commonly also shows an overall dimension with the chain dimensioning.
Datum Dimensioning

- AutoCAD refers to **datum dimensioning** as **Baseline** dimensioning.
- Use the **DIMBASELINE** command.
Datum Dimensioning

- Use the DIMBASELINE command to perform baseline dimensioning.
  - Pick the Baseline Dimension button in the Dimension toolbar
    - OR
  - Pick Baseline in the Dimension pull-down menu
    - OR
  - Type DBA or DIMBASELINE at the Command: prompt
Datum Dimensioning

- A dimension must exist first before the DIMBASELINE command can be used.

- AutoCAD will use the most recently drawn dimension unless you specify another one.
Datum Dimensioning

- In datum dimensioning
- AutoCAD automatically spaces and places
  - Extension lines.
  - Dimension lines.
  - Arrowheads.
  - Numbers.
Chain Dimensions

Create chain dimensions by:

- Picking the Continue Dimension button in the Dimension toolbar
  - OR
  - Picking Continue in the Dimension pull-down menu
  - OR
- Typing DCO or DIMCONTINUE at the Command: prompt
The QDIM Command
The QDIM Command

- The **QDIM command**
  - makes chain and datum dimensioning even easier.
  - It eliminates the need to define the exact points being dimensioned.
  - Automates the process of point selection.
The QDIM Command

QDIM can be selected by:

- Selecting QDIM from the Dimension pull-down menu
  - OR
- Picking the Quick Dimension button from the Dimension toolbar
  - OR
- Typing QDIM at the Command: prompt.
The **QDIM Command**

- **Single Polyline:**
  - QDIM draws linear dimensions to every vertex.

- **Single arc or circle:**
  - QDIM draws a radius or diameter dimension.
The QDIM Command

- Multiple Objects: QDIM draws
  - **Linear dimensions** to the **vertex** of every line or polyline.
  - **Linear dimensions** to the **center** of every **arc** or **circle**.

- **AutoCAD** finds these points automatically.
Basic Dimensioning Practices

- Dimensions: 10'-0" and 13'-0"
- Circles of different sizes.
Basic Dimensioning Practices
Inserting Dimensioning Symbols
Inserting Symbols

- After you select a feature to dimension, AutoCAD responds with the measurement or dimensioning number.

- If the object is a circle or radius, AutoCAD automatically places the radius \((R)\) or diameter \((\phi)\) symbols before the dimension number.
Inserting Symbols

- AutoCAD does **not** place a symbol automatically in **linear** dimensioning.
Inserting Symbols

◆ Use the Mtext editor to insert other symbols.

◆ Place the text cursor in the location where you want the symbol placed
  ◆ Open the Symbol drop-down list
  ◆ Select the symbol
  ◆ Click OK.
Creating Your Own Symbols

- If a needed symbol does not exist, you can draw it and insert it on the drawing.

- It is faster to create your own symbols, save them as a block and insert the block on the drawing as needed.
The Circle Center Mark
and
DIMCEN
Circle Center Dashes

- Circles are dimensioned to the CENTER of the circle.
- AutoCAD will easily draw a centermark that can be “dimensioned to”.
  - The command is **DIMCENTER**
  - The mark size is set by the **DIMCENS** variable
**Circle Center Dashes**

- To draw the center dashes:
  - Pick the **Center Mark** button on the **Dimension** Toolbar
  - Pick **Center Mark** in the **Dimension** pull-down menu
  - Type **DCE** or **DIMCENTER** at the Command: prompt

- When the arc or circle is picked, the center marks are automatically drawn.
**DIMCEN Variable**

- The **DIMCEN** variable controls the center dashes.
- The default length is 0.09” long.
DIMCEN Variable

◆ If DIMCEN = 0; center marks are not drawn.

◆ If DIMCEN < 0; AutoCAD draws
  ◆ Complete centerlines that extend past the edge of the circle.
  ◆ Center dashes.
DIMCEN=1/16
DIMCEN=(-)0.3
DIMCEN=(-)1/16
 DIMCEN Variable

◆ If DIMCEN > 0; AutoCAD draws
  ◆ Center dashes ONLY.
  ◆ No extension lines outside the circle.
**DIMCEN Variable**

- The value of the **DIMCEN** variable sets
  - The length of the **center dashes** and
  - The length of the **extension lines past the edge of the circle**.

- **DIMCEN = 0.09.** Dashes are 0.09 units long.
- **DIMCEN = 0.5.** Dashes are 0.5 units long.
DIMCEN = 1/16
DIMCEN = (-) 0.3
DIMCEN = (-) 1/16
Summary - Dimensioning Commands
Dimensioning Commands

- DLI DIMLINEAR
- DAL DIMALIGNED
- DAN DIMANGULAR
- DBA DIMBASELINE
- DCE DIMCENTER
- DCO DIMCONTINUE
- DED DIMEDIT
- DRA DIMRADIUS
- DDI DIMDIAMETER
- DOR DIMORDINATE
- DOV DIMOVERRIDE
- DST DIMSTYLE
- LE LEADER
Using the Dimensioning Mode
Using the Dimensioning Mode

- Entering `DIM` or `DIM1` enters the "dimensioning mode"
  - You can enter as many dimensions as desired in this mode
  - Enter `E` or `EXIT` or press the `[Esc]` key to exit this mode
Transparent Dimensioning
Transparent Commands

- Dimensioning variables can be entered transparently while in another command.
Transparent Commands

- At most dimensioning prompts, the variable name can be entered without the DIM prefix.
  - Horizontal
  - Vertical
  - Aligned
Transparent Commands

- **DIM: prompt**
  - ALIGNED
  - ANGULAR
  - BASELINE
  - CONTINUE
  - DIAMETER
  - HOMETEXT
  - HORIZONTAL
  - LEADER
  - LEADER
  - NEWTEXT

- **Command: prompt**
  - DIMALIGNED
  - DIMANGULAR
  - DIMCENTER
  - DIMCONTINUE
  - DIMDIAMETER
  - DIMEDIT Home
  - DIMLINEAR
  - Horizontal
  - DIMEDIT Text
Transparent Commands

- **DIM: prompt**
  - OBLIQUE
  - ORDINATE
  - OVERRIDE
  - RADIUS
  - RESTORE
  - ROTATED
  - SAVE

- **Command: prompt**
  - DIMEDIT Oblique
  - DIMORDINATE
  - DIMOVERRIDE
  - DIMRADIUS
  - DIMSTYLE Restore
  - DIMLINEAR
  - DIMSTYLE Save
Transparent Commands

**DIM: prompt**
- STATUS
- TEDIT
- TROTATE
- UPDATE
- VARIABLES
- VERTICAL

**Command: prompt**
- DIMSTYLE Status
- DIMTEDIT
- DIMEDIT Rotate
- DIMSTYLE Apply
- DIMSTYLE Variables
- DIMLINEAR Vertical
Transparent Commands

Exception: DIMCEN

Is interpreted as CEN object snap override if not preceded by DIM.
Drawing Leader Lines
The **DIMDIAMETER** and **DIMRADIUS** commands automatically place leaders on the drawing.

The **LEADER** command allows you to begin and end a leader line where you desire.
Drawing Leader Lines

Leader lines are ideal for:

- Adding specific notes to a drawing.
- Situations where a leader line must be staggered to go around other drawing features.
- Where a double leader line is required.
- Where custom leaders are needed.
Chamfer
LEADER Command

- The leader line characteristics, such as arrowhead size and text positioning are controlled by the dimension variable settings.

- Other features are controlled by options within the LEADER command.
The **LEADER** command is accessed by:

- Picking the **Leader** button in the Dimension Toolbar.
- **OR**
- Picking **Leader** in the Dimension pull-down menu.
- **OR**
- Typing **LE** or **LEADER** at the Command: prompt.
LEADER Command

- The LEADER command prompts look like the LINE command prompts.
  - Pick the starting and ending points of the leader.
  - The Annotation option is the default.
  - Press [Enter] and type in the desired note.
  - The Mtext prompt appears.
  - Type as many lines of text as desired.
  - Terminate with the [Enter] key.
LEADER Command Options

- The Annotation sub-options
  - Tolerance.
  - Copy.
  - Block.
  - None.
LEADER Command Options

- The Annotation sub-options
  - Tolerance
    - Allows creation of the feature control frame
    - Discussed in Chapter 21 of text.
  - Copy
    - Copies text, mtext, feature control frame or block, and connects the new object to the leader being created.
    - Refer to Figure 18-43B.
Framed text

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LEADER Command Options

✦ The Annotation sub-options
  
 ✦ **Block**
    ✦ Inserts a specified block at the end of the leader.
  
 ✦ **None**
    ✦ This option ends the leader with no annotations of any kind.
LEADER Command Format Options

- The LEADER command allows you to modify the way a leader line is presented.
- Accomplished by accessing the Settings option.
LEADER Command Format Options

The **Settings Options**:

- Spline.
- Straight.
- Arrow.
- None.
- Exit.
This is the MTEXT editor for use with leaders

Modify character properties.
Framed text

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Leader with an unlimited number of spline points

This is leader text using MTEXT.
Framed text

Chamfer

Leader with an unlimited number of spline points

Enter next line of annotation text: spline points
Enter next line of annotation text:
The UNDO Option

- The Undo option removes the last leader segment that was drawn.
**Using Multiple Leaders**

- Multiple Leaders can be drawn in AutoCAD.
  - Use the **LEADER** command to place the first leader and note.
  - Enter the **LEADER** command again and pick the beginning of the previous leader shoulder as the second point.
  - Use the **Annotation None** option to terminate the command.