ROHINI COLLEGE OF ENGINEERING AND TECHNOLOGY

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Department of Mechanical Engineering

VALUE ADDED COURSE ON GEOMETRIC DIMENSIONING AND TOLERANCING

GEOMETRIC DIMENSIONING AND TOLERANCE

OBJECTIVE:

This course is designed to instruct students and introduce basic geometric dimensioning and tolerancing principles. Topics include symbols and notation and applications. Upon completion students should be able to enter interpret and apply basic geometric dimensioning and tolerancing principles to drawings.

- To provide an overview of how computers are being used in mechanical component design
- To understand the application of computers in various aspects of Manufacturing viz., Design, Proper planning, Layout & Material Handling, Dimensioning and tolerance.

UNIT I GETTING STARTED WITH CREO ELEMENTS/DIRECT DRAFTING

The Creo Elements/Direct Drafting User Interface-Basic Working Methods-What are Viewports? -Positioning Menus-How to Delete Elements-How to Store a Work file-Autosave-Getting Help-Switching Languages-Switching Between Flat and Raised Menu Buttons-Switching the Beeper on or Off-Further Reading

UNIT II DRAWING METHODS

Drawing Construction Geometry-Drawing Construction Lines-Drawing Construction Circles-Drawing a Lever Using Construction Geometry-Using the Line Drawing Set-Creating Points-Using Overdraw-Drawing Lines-Drawing Circles-Drawing Arcs-Creating a Centerline-Creating Symmetric Elements-Indicating the Projected Reference Point-Creating a Reference Line-Splitting Elements-Merging Elements-Drawing Fillets-Drawing Chamfers-Keeping Corners-Creating a Polyline-Drawing Ellipses-Drawing Splines-Drawing Parallel Contours

UNIT III DIMENSIONING A DRAWING

Dimensioning Fundamentals- Dimensioning with Creo Elements/Direct Drafting- Preparing to Dimension- Single Dimensioning- Chain Dimensioning- Datum Dimensioning- Radius, Diameter, Tangential, and Arc Dimensioning- Angle and Chamfer Dimensioning- Geometry-Sensing Assignment- Dual Dimensioning- Hatching a Drawing Area- Converting Hatch to Geometry-Entering Text- Editing and Deleting Text

UNIT IV GD&T FRAMEWORK

GD&T Framework- Size tolerances- Datums- Position- Orientation (Axis/Midplane)- Material modifiers

UNIT V SURFACE STANDARDS

Surface profile- Surface orientation- Surface form- Runout controls- Derived element controls

TEXT BOOKS:

1. Simplified GD&T: Based on ASME-Y 14.5-2009 (Edition) Paperback – 28 April 2018

2. Geometric Dimensioning and Tolerancing: Applications and Techniques for Use in Design Manufacturing and Inspection (PB) Paperback – 1 January 2017

3. Simplified GD&T: Based on ASME-Y 14.5-2009 (Edition) Paperback – April 28, 2018

REFERENCES:

1. Chris McMahon and Jimmie Browne "CAD/CAM Principles", "Practice and Manufacturing management "Second Edition, Pearson Education, 1999.

2. Donald Hearn and M. Pauline Baker "Computer Graphics"'. Prentice Hall, Inc,1992.

Geometric Dimensioning and Tolerancing

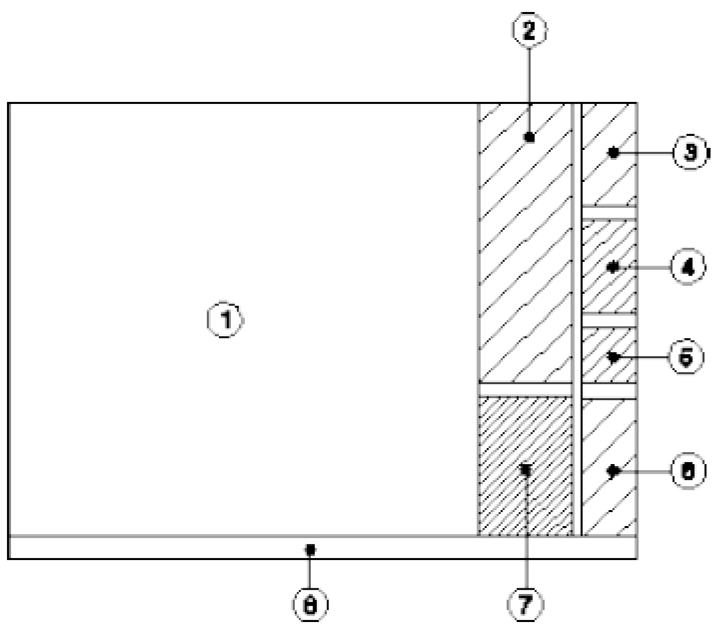
UNIT I

GETTING STARTED WITH CREO ELEMENTS/DIRECT DRAFTING

The Creo Elements/Direct Drafting User Interface

The Creo Elements/Direct Drafting user interface is designed to ensure a short learning curve and to provide maximum efficiency in 2D CAD projects. All of the menus, commands, functions, settings and tools specified in this manual are acccessible via the Creo Elements/Direct Drafting user interface. You can use Creo Elements/Direct Drafting either in screen-only. In screen-only mode, you will access the entire Creo Elements/Direct Drafting functionality by means of a mouse and your keyboard. The following figure indicates the screen layout of the Creo Elements/Direct Drafting screen-only version.

Screen-Only Display Layout



1 Drawing area. This is the area where you will create and modify drawings.

2 Command menu. Displays the commands and options available through the current menu.

3 Command section. Click any of these menu commands to display the associated menu.

4 Command section. Contains frequently used commands.

5 Additional Modules section. Enables you to access further modules integrated with Creo Elements/Direct Drafting.

6 Interrupt functions, viewports, and Toolbox.

7 Window area for interrupt functions, viewports, and Toolbox

8 System status bar, system prompt line, user input line. These provide general user guidance and feedback.

Optionally, you can run multiple Creo Elements/Direct Drafting instances (in separate Creo Elements/Direct Drafting windows) at the same time.

General User Guidance and Feedback

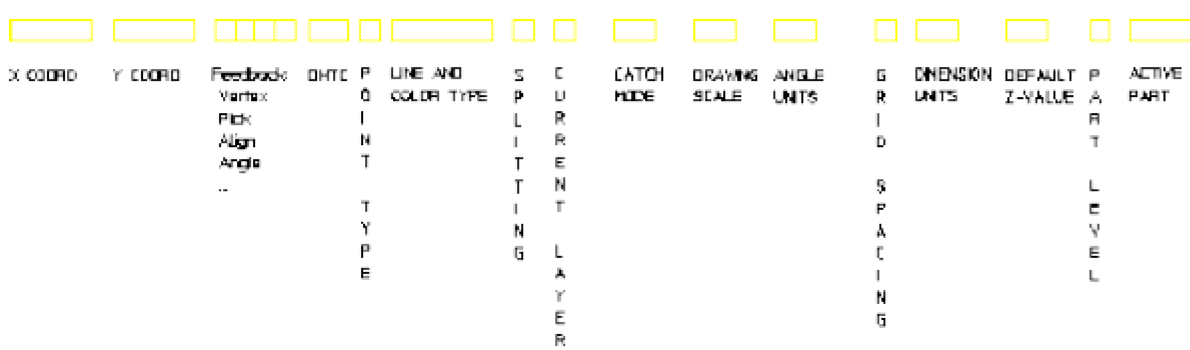
The System Status Bar, the System Prompt Line, and the User Input Line at the bottom of the Creo Elements/Direct Drafting display provide general guidance and feedback.

• System Status: Bar provides information on the current state of Creo Elements/Direct Drafting.

• System Prompt Line: displays messages to guide you through the commands.

• User Input Line: displays the data you enter via the keyboard.

System Status Bar



Changing the Status Bar Colors

Optionally, you can change the status bar colors by entering the following in the user input line:

• CMD_BG_COLOR followed by a color name to change the background color of the user input line.

• CMD_TXT_COLOR followed by a color name to change the color of the text on user input line.

Using the Dynamic Mouse

To use the dynamic mouse, press and hold the [Ctrl] key and press one of the following mouse buttons:

• The full functionality is available on a three-button mouse. If a two-button mouse is used, press the left and right button simultaneously to emulate the middle button.

• This functionality is available on screen-only versions of Creo Elements/ Direct Drafting.

• Whenever the user definable buttons are pressed, the corresponding string value for that button is displayed on the user input line.

• If your drawing contains images (pixmaps), you can hide these using the SHOW Box command before doing any dynamic operations. This improves system performance.

Additional Information:

• The full functionality is available on a three-button mouse. If a two-button mouse is used, press the left and right button simultaneously to emulate the middle button.

• This functionality is available on screen-only versions of Creo Elements/ Direct Drafting.

• Whenever the user definable buttons are pressed, the corresponding string value for that button is displayed on the user input line.

• If your drawing contains images (pixmaps), you can hide these using the SHOW Box command before doing any dynamic operations. This improves system performance.

Basic Working Methods

It will not take you very long to become familiar with Creo Elements/Direct Drafting, but a few hints may help you to get started.

The best way to learn is to practice, and we recommend you try the examples given in this manual. If you need further explanation of any of the 2D functions or commands, refer to this manual. It contains a description of how to use each command or function.

An index is included at the back of the manual; this contains not only the command and function names but also a task reference. Many operations require similar user interaction, For example, you will often be prompted to identify an element, enter a value, enter a filename, or indicate a point. Some of the most commonly used terms are listed below:

Identifying an Element or a Point

This implies that the element or point already exists. Identify it by:

• Pressing the cursor at the appropriate position.

• Entering the coordinates separated by a comma.

Indicating a Point

This implies that the point chosen could be anywhere on the drawing and may or may not be marked by an existing point. Again, the point required is indicated by:

• Pressing the cursor at the appropriate position.

• Entering the coordinates separated by a comma.

Entering a Value

Entering a value means entering the value and pressing [Enter]. For certain commands and functions, you must enter a numerical value, and for others a string enclosed in quotes ' ' or"". Additionally, values to be entered can be first calculated by typing the appropriate mathematical expression in parentheses:
(15 * tan 30)

Undo and Redo

In Creo Elements/Direct Drafting, you can undo mistakes with **Undo** / **Redo**.
Click to undo an operation (before you click **Confirm**)
Click to redo an operation (after you have clicked **Undo**).
Each time you click **UNDO** or **REDO**, you will undo or redo all the changes that were created by the last command you entered.

You can set a limit on the number of steps to keep in the **UNDO** or **REDO** history in the System Setting dialog.

1. In the main menu, click **Setup** ▶ **Edit Environment** ▶ **System**.
2. You can select:

• **-1**: no limit.
• **0**: disable the Undo/Redo option.
• **+[number]**: the maximum number of steps you can undo or redo when no command is active. If you already have more steps in your history than the number you selected as the limit, Creo Elements/Direct Drafting will automatically delete the extra steps.

Limitations:

• You can undo or redo complete macros, but you cannot use **UNDO** or **REDO** inside macros.
• **UNDO** or **REDO** cannot undo or redo:

• Commands related to parametric design, hidden lines, PIXMAPs, or OLEs. Once you have executed one of these commands, you can no longer undo or redo any of your previous commands.

• The RESET_SYSTEM command.

• Commands for default settings (although these commands do not affect the undo/redo history).

Undo and Redo with state history

Each time you enter a command and change your drawing, Creo Elements/Direct Drafting adds a new state to the undo/redo history. Creo Elements/Direct Drafting marks the new state with a unique ID. With this information, you can:

• Use UR_GET_CURRENT_ID to see the ID of the current undo/redo state.

• Use UR_ID_EXISTS to see if a specified ID exists in the undo/redo history.

• Use UR_GET_FIRST_ID to see the ID of the first state in the undo/redo history.

• Use UR_GET_LAST_ID to see the ID of the last state in the undo/redo history.

• Use UR_MOVE to undo/redo the drawing to a specified state.

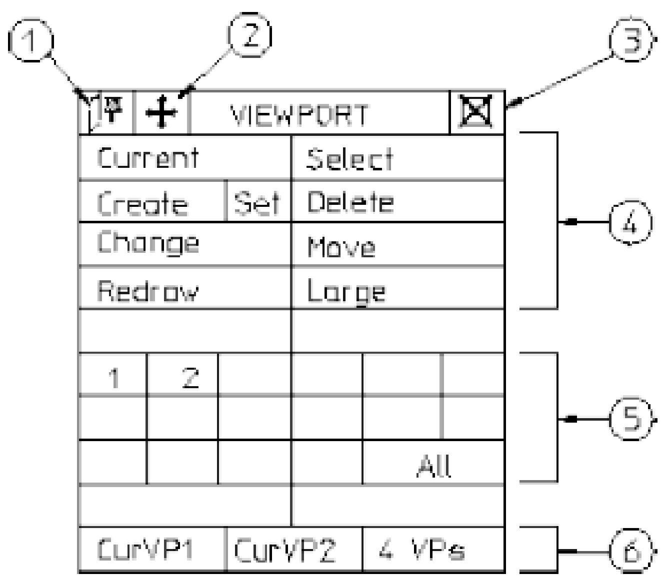
What are Viewports?

A viewport is a rectangular area on the screen whose position and size you can define. Specific parts of your drawing can be displayed in the viewports. This then enables you to study the drawing or add information with greater ease and accuracy. You can create up to 16 viewports but usually between 2 and 6 are sufficient. The viewports are identified by a number in the top right-hand corner. You will notice one is in inverse video – this is the current viewport. All window functions operate on the current viewport, for example, if you press **NEW** and make a box in any viewport the new view will be drawn in the current viewport. Any viewport can be made current by either picking the identification number on the screen or by pressing the **CURRENT** function in the **VIEWPORT** menu.

In Creo Elements/Direct Drafting, you can overlap viewports with other viewports, menus, and programmable tables, without any limitations in functionality. All overlapping viewports are permanently updated during your drawing session.

The VIEWPORT menu of the screen-only version includes the command buttons **Select**, **Move**, and **Set**. **Set** allows you to define the viewport's color and its border width. Additionally, the screen-only viewport menu provides a status area for the viewports. Click a button in this status area to define a single or all viewports. The bottom row of the viewport menu provides combined commands. **CurVP1** and **CurVP2** create two viewports and set either viewport 1 or viewport 2 current. **4 VPs** creates 4 predefined viewports.

Viewport Menu



The icons marked 1 to 3 can also be found in all other menus.

1. Pin the menu
2. Move the menu
3. Close the menu

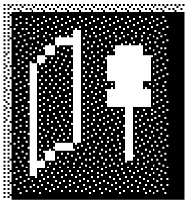
4. Commands

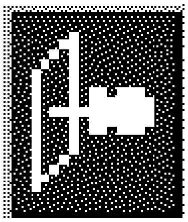
5. Status area

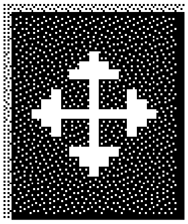
6. Combined commands

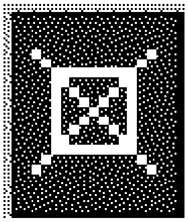
Positioning Menus

You can position Creo Elements/Direct Drafting menus anywhere on the screen and pin them so that when they are recalled, they open in the same position. The title bar of each menu provides the following icons:

 This icon indicates that the menu is currently unpinned. Pick this icon to pin the menu.

 This icon indicates that the menu is currently pinned. Pick this icon to unpin the menu.

 Pick this icon to move a menu to a new position.

 Pick this icon to close a menu.

How to Delete Elements

As you learn to work with the system you are sure to make mistakes. To delete an element or piece of text:

1. Press the **DELETE** block.
2. Identify the element or text to be deleted. (If you make a mistake, you can restore the element or text by pressing **UNDO**.)

How to Store a Workfile

To prevent loss of your drawing (due to power failure, for example) you should store the workfile regularly (every 10 to 15 minutes is recommended).

To store your workfile:

1. Press **FILE**.
2. Pick STORE.
3. Pick 'workfile'.

Autosave

Creo Elements/Direct Drafting provides an autosave feature to automatically save a drawing at routine intervals. (Autosave is set by the command AUTO_STORE_ TIME.)

• The autosaved file is me_backup.mi. The file is put in the working directory as specified by ME_WORKING_DIR. If ME_WORKING_DIR is not defined, the file is stored in \temp on Windows based systems.

• The autosave interval is a time interval in minutes. Possible values are 10 and 0 (0 is off, this is the default).

• While the autosave is running, a message pops up and the blue store bar displays the progress.

Getting Help

The help file contains technical reference information about all the commands and functions that you can use within Creo Elements/Direct Drafting.

To use the help system:

1. Press **HELP**.

2. Input the command or function on which you require help by either:

• Picking a command or function from the screen menu.

• Entering the command or function name.

Creo Elements/Direct Drafting responds by displaying the required help information in a separate browser.

Switching Between Flat and Raised Menu Buttons

The MEPELOOK variable in the me10.ini file (located in the Creo Elements/ Direct Drafting installation directory) enables you to select either the "flat" look or the "raised menu buttons" look of the Creo Elements/Direct Drafting Classic user interface:

Switching the Beeper on or Off

Optionally, you can switch the "beeper" on or off in Creo Elements/Direct Drafting. Simply enter BEEPER and then ON or OFF into the user input line.

Further Reading

You can now begin to use Creo Elements/Direct Drafting to produce high quality 2D drawings. When you are ready to learn more about the system and its command structure refer to your Writing Macros with Creo Elements/Direct Drafting, which contains advice on macro writing and command syntax.

UNIT II

DRAWING METHODS

Drawing Construction Geometry

Construction geometry is used for positioning and as an aid to drawing. This section shows how to draw using construction lines and circles, just as you would normally do when working on a drawing board. Construction lines differ from real lines in that they are considered to be of infinite length. Only full construction circles can be drawn. When creating construction lines, you can indicate a point for the line or enter the exact coordinates via the keyboard. Enter coordinates using the following format:

2.5,3

12,4.8

5.35,6.1

The x and y coordinates are separated by a comma. When you have typed in the coordinates, press [Enter].

Drawing Construction Lines

Ten commands allow you to draw construction lines.

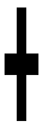
Construction Line by Two Points

Press this option, then indicate two points for the construction line required or enter the coordinates of two points.

Horizontal Construction Line

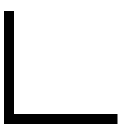
Enter the y coordinate of the line required, or use the cursor to identify a point through which the line must pass. To create several horizontal construction lines, simply enter the appropriate y coordinates separated by spaces.

Vertical Construction Line



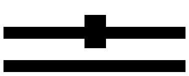
Enter the x coordinate of the line required, or use the cursor to identify an existing point through which the line must pass. To create several vertical construction lines, simply enter the appropriate x coordinates separated by spaces.

Perpendicular Construction Line



You can create a construction line perpendicular to any existing line. Press the option, identify the existing line, and then indicate a point through which the perpendicular construction line must pass.

Parallel Construction Line



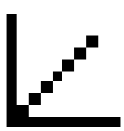
You can create construction lines parallel to any existing line in the following ways:

• Identify any existing line. Enter, or indicate a point for the parallel construction line to pass through. Multiple parallel construction lines are drawn by entering or indicating further points for the lines to pass through.

• Enter the required separation of the line. Now identify any existing line. Finally, indicate on which side the parallel construction line should be drawn. Multiple lines can be drawn at the previously set separation by indicating the original lines and the desired sides.

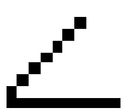
• A set of parallel lines can be drawn from an original line. For example, if you enter line separations in the form: 10 20 30 40, four parallel lines will be drawn at the given distances from the indicated line.

Construction Line Bisecting an Angle



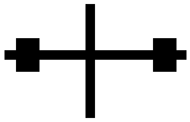
You have the option of entering a value for the ratio of the division, otherwise, the angle will be bisected. Indicate the apex, and then two points (Usually on existing lines) to define the angle.

Construction Line at an Angle



When you select this option, Creo Elements/Direct Drafting will prompt you to indicate or enter the coordinates of a point through which the construction line must pass. Next enter the angle of the line with respect to the x axis. Note that a positive input will result in a line drawn counterclockwise from the x axis. The angle units will be those defined in the SETUP menu.

Bisecting Construction Lines

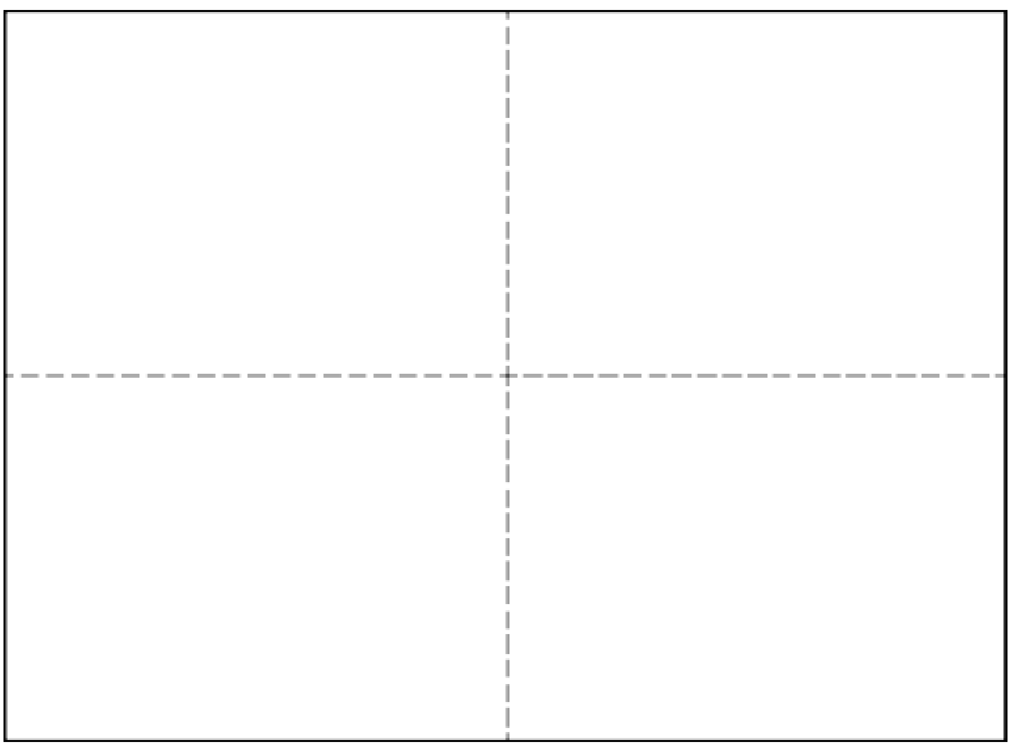


If you do not want the line concerned divided into two equal parts, enter a value for the ratio of the division. Then identify the two end points of the line. A construction line will be drawn perpendicular to the line at the appropriate position.

Drawing a Lever Using Construction Geometry

Use the following tutorial example to familiarize yourself with the various techniques used when producing a drawing with the aid of construction geometry.

Horizontal and Vertical Construction Lines



1. Press **CREATE**.
2. Press **CATCH**.
3. Press **SHOW**, **ALL** and **ON**.

4. Press **SHOW**, **VERTEX** and **OFF**.

5. Press C_LINE HORIZONTAL in the **CONSTRUCT** block.

6. Enter 0,0.

7. Press C_LINE VERTICAL in the **CONSTRUCT** block.

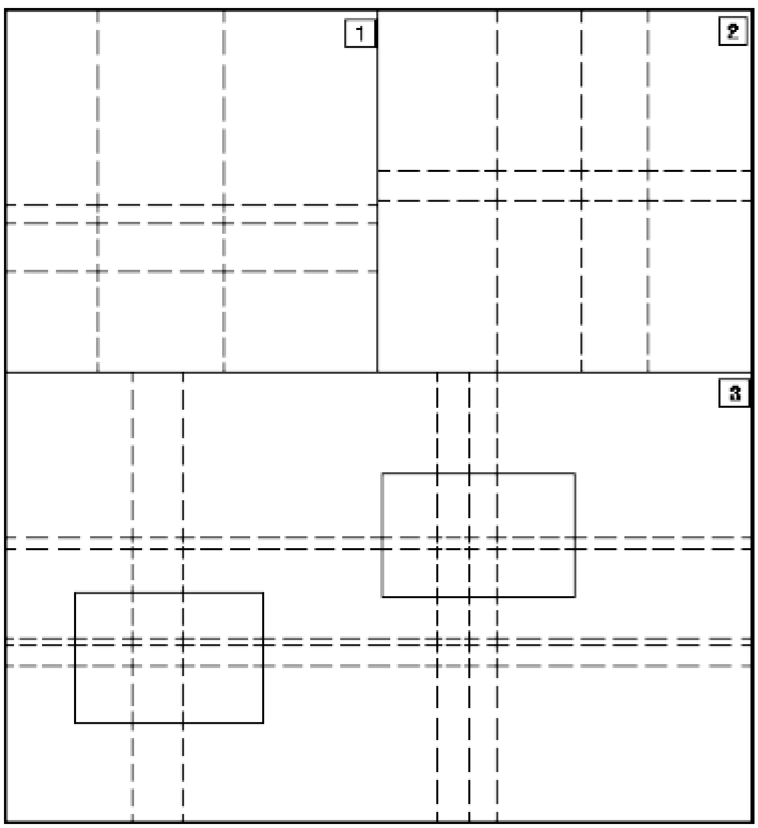
8. Enter 0,0.

9. Press **FIT** in the **WINDOW** block.

10. Press **END**.

Now draw some parallel horizontal construction lines.

Three Viewports



1. Press **DELETE** in **VIEWPORT**.

2. Press **ALL** in **SELECTION**.

3. Press **CREATE** in the **VIEWPORT** block.

4. Create viewport 1 as shown above by indicating 2 corner points of the viewport.

5. Repeat step 2 and create viewports 2 and 3.

Viewports 1 and 2 give you a closer look at certain areas of your drawing where you will do more complex work. Viewport 3 displays the entire drawing.

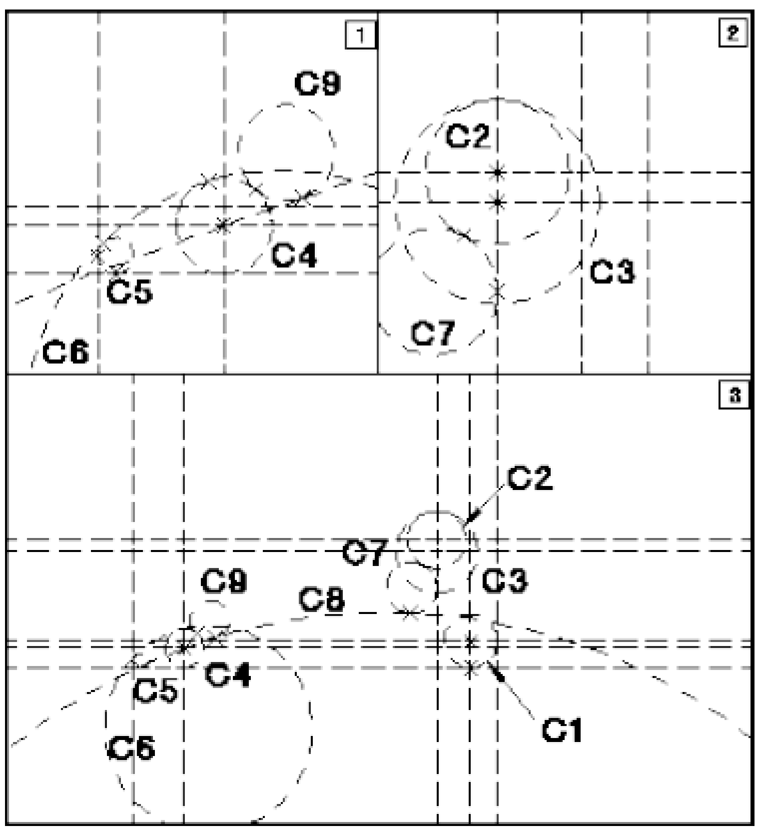
6. Press **CURRENT** in **VIEWPORT** and identify viewport 1.

7. Press **NEW** in **WINDOW** and indicate 2 points in viewport 3 as defined by the left-hand rectangle.

8. Press **CURRENT** in **VIEWPORT** and identify viewport 2.

9. Press **NEW** in **WINDOW** and indicate 2 points in viewport 3 as defined by the right-hand rectangle.

Construction Circles



1. Press **ALL** in the **CATCH** block.

2. C1 — Press C_CIRCLE CENTER in **CONSTRUCT** and identify the center

point and lower circumference point (in viewport 3).

3. C2 — Press C_CIRCLE CENTER in **CONSTRUCT** and identify the center

4. C3 — Press C_CIRCLE CENTER in **CONSTRUCT** and identify the center point and enter 17 (in viewport 2).

5. C4 — Press C_CIRCLE CENTER in **CONSTRUCT** and identify the center point and enter 8 (in viewport 1).

6. Press C_CIRCLE TAN2_PT in **CONSTRUCT**.

7. C5 — Indicate the two tangent points and enter the radius 3 (in viewport 1).

8. C6 — Indicate the two tangent points and enter the radius 43 (in viewport 1).

9. C7 — Indicate the two tangent points and enter the radius 10.5 (in viewport 2).

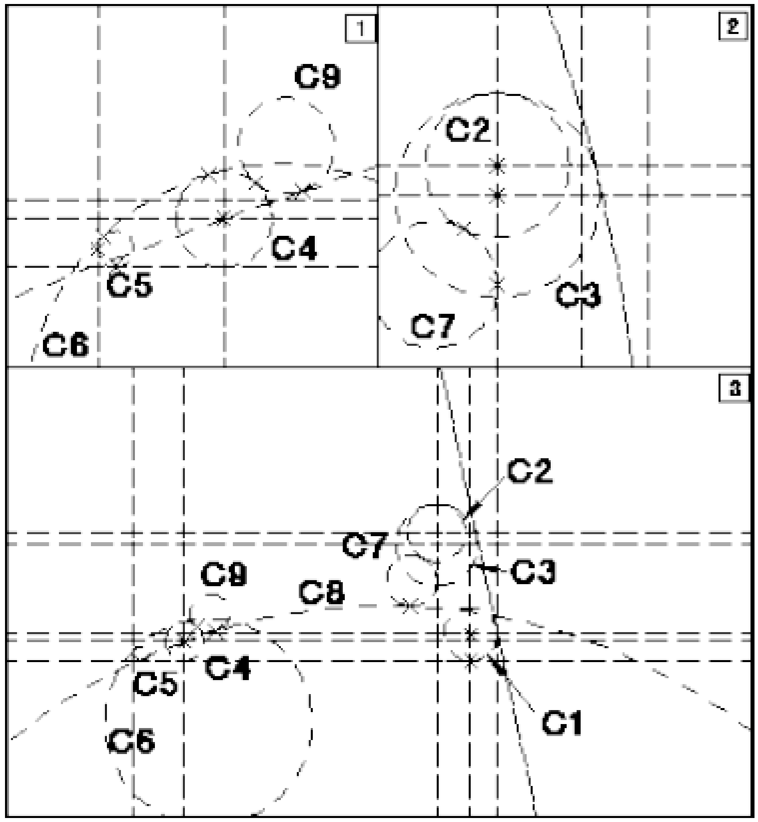
10. Press C_CIRCLE TAN_PT_PT in the **CONSTRUCT** block.

11. C8 — Indicate the tangent point to C7 and the center point of C4. Enter the radius 246 (in viewport 3).

12. Press C_CIRCLE TAN2_PT in the **CONSTRUCT** block.

13. C9 — Indicate the two tangent points and enter the radius 8 (in viewport 3). Draw a construction line tangent to C1 and C3.

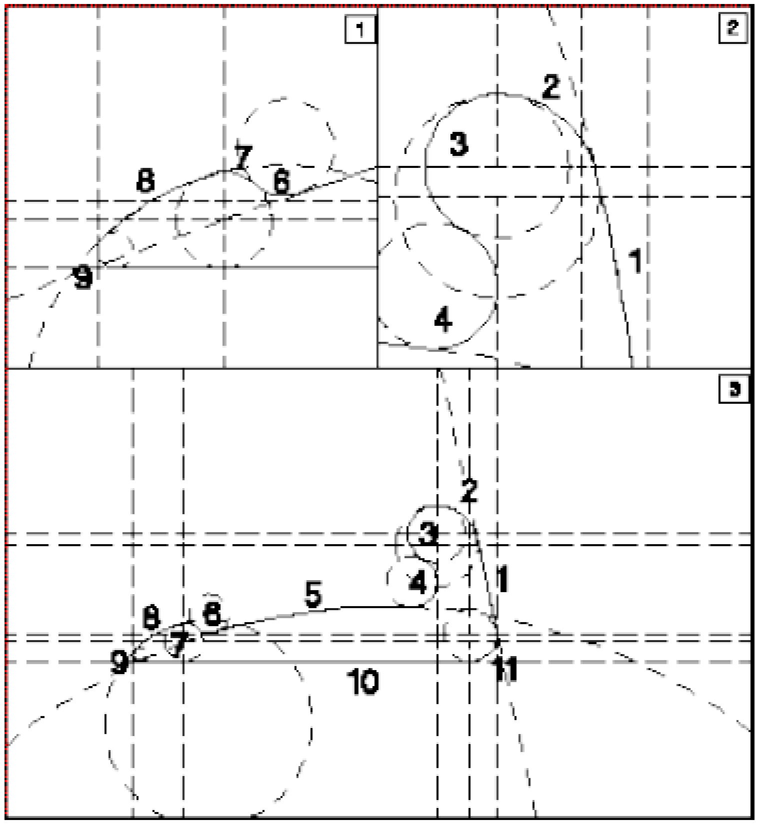
Construction Line Tangent



1. Press C_LINE TAN2 in the **CONSTRUCT** block.
2. Indicate the tangent points on C1 and C3 (in viewport 3).

Now pick OVERDRAW in **CREATE** to draw the lever itself.

Drawing the Lever



1. Press **CREATE**.
2. Pick OVERDRAW.
3. Indicate point 11/1 as the start point.
4. Indicate a point on element 1 and indicate point 1/2.
5. Indicate a point on element 2 and indicate point 2/3.
6. Indicate a point on element 3 and indicate point 3/4.
7. Indicate a point on element 4 and indicate point 4/5.
8. Indicate a point on element 5 and indicate point 5/6.
9. Continue with this procedure for elements 6, 7, 8, 9, 10, and 11.

Creating Points

Points can be created anywhere on your drawing simply by picking the POINT command and indicating the position with the cursor, or entering the required coordinates. If you have difficulty deleting points, try enclosing them in a box for deletion.

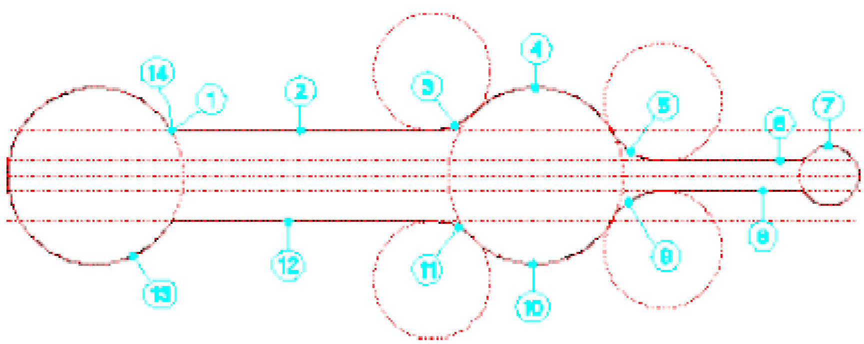
Using Overdraw

Use OVERDRAW to draw a contour between intersection points in the construction geometry. The lines will be drawn over the existing construction lines and circles just as you would line in a contour by hand. Additionally, it is now not necessary to pick each construction geometry intersection point before identifying the line or arc you want to draw over. You simply pick the next construction geometry element and Creo Elements/Direct Drafting calculates the intersection point. This saves one pick for every overdraw action.

To use the OVERDRAW command:

1. Draw the construction geometry as shown in Figure
2. Click OVERDRAW
3. Identify the start point (1)
4. Identify points 2 to 14 (end point)

Using the OVERDRAW Command



The OVERDRAW command also supports multiple UNDO operations.

Drawing Lines

Ten commands allow you to draw lines:

• 2 Pts

• Polygon

• Horizontal

• Vertical

• Parallel

• Perpend

• Tan Arc &Pt

• Tan 2 Arcs

• Ang & Len

• Rectangle

Line by Two Points

This is the default option.

1. Press **CREATE**.

2. Pick 2 Pts or the LINE command.

3. Indicate the start and end point of the line required.

Polygon

To draw a polygon:

1. Press **CREATE**.

2. Pick Polygon.

3. Indicate the start point, and then a sequence of end points.

Note that the result does not have to be a closed polygon. The lines are treated as individual elements and must therefore be deleted individually or enclosed in a box for deletion. Once you have closed a polygon, you can continue by indicating the first point of a new polygon.

Horizontal Line

This option is similar to line by two points but only draws a horizontal line.

To draw a horizontal line:

1. Press **CREATE**.
2. Pick Horizontal.
3. Indicate the start point of the line.
4. Indicate the end point of the line, or enter the length of the line.

If you indicate an end point, the line is drawn through that point.

Vertical Line

This option is similar to line by two points but only draws a vertical line.

To draw a vertical line:

1. Press **CREATE**.
2. Pick Vertical.
3. Indicate the start point of the line.
4. Indicate the end point of the line, or enter the length of the line. If you indicate an end point, the line is drawn through that point.

Parallel Line

To draw a line parallel to an existing line:

1. Press **CREATE**.
2. Pick **Parallel**.
3. Identify the existing line.
4. Indicate a point on the new line.

Perpendicular Line

To draw a perpendicular line:

1. Press **CREATE**.
2. Pick **Perpend**.

3. Identify the existing line.

4. Indicate the point from which the perpendicular should be dropped.

Alternatively:

1. Pick **Perpend**.

2. Identify the existing line.

3. Indicate a point on the existing line.

4. Enter the length of the perpendicular. If the perpendicular drawn is on the wrong side of the line, repeat the operation using the opposite sign for the entered value.

Tangent from Point to Arc or Circle

This option enables you to draw a tangent to a circular element from any given point.

1. Press **CREATE**.

2. Pick **Tan Arc &Pt**.

3. Indicate the approximate point of tangency on the arc or circle.

4. Indicate the point from which the tangent should be drawn.

The system will inform you if such a tangent is not possible.

Line Tangent to Two Arcs or Circles

To draw a tangent from one circular element to another:

1. Press **CREATE**.

2. Pick **Tan 2 Arcs**.

3. Indicate the approximate point of tangency on each of the two arcs or circles. The tangent drawn will be the one with points of tangency closest to those indicated.

Line with Given Length and Angle

To draw a line with a given length and angle:

1. Press **CREATE**.

2. Pick Ang & Len.

3. Indicate the start point of the line.

4. Enter the angle of the line with respect to the x-axis.

5. Enter the length of the line.

The line will be drawn accordingly.

Rectangle

Rectangles can be created as whole entities, but the system treats them as four individual line elements. This means the sides of a rectangle can be deleted or manipulated individually.

To draw a rectangle:

1. Press **CREATE**.
2. Pick **Rectangle**.
3. Indicate two diagonal corner points.

You will see that the rectangle moves with your cursor until the second point is defined.

Drawing Circles

The five commands used for drawing circles are:

• Cen & Pt/R

• 3 Pts

• Tan2 &Pt/R

• Concentric

• Diameter

Circle by Center and Circumference Point

This is the default option. Therefore, you can either pick Cen & Pt/R or the CIRCLE command itself. To draw a circle by indicating the center and point:

1. Press **CREATE**.

2. Pick **Cen & Pt/R**.

3. Indicate the center of the circle.

4. Indicate a point on the circumference, or enter the length of the radius. The circle moves with the cursor until the circumference point is defined. You can create additional concentric circles by indicating additional points or radii.

Circle by Three Points

To draw the circle by indicating three points on the circumference:

1. Press **CREATE**.

2. Pick **3 Pts**.

3. Indicate three circumference points.

The circle drawn will vary with the movement of the cursor until the third point is defined.

Circle by Radius and Two Elements

To draw a circle of given radius tangential to two existing elements:

1. Press **CREATE**.

2. Pick **Tan2 &Pt/R**.

3. Indicate the approximate tangent points on the two elements.

4. Enter the radius required.

The elements can be either lines, arcs or circles. Creo Elements/Direct Drafting will calculate the best solution or, if no solution is possible, it will suggest a valid radius.

Concentric Circles

To draw concentric circles:

1. Press **CREATE**.

2. Pick Concentric.

3. Identify the existing circle.

4. Indicate the desired circumference point and the circle will be drawn.

To draw further concentric circles, simply indicate circumference points.

Alternatively:

1. Pick Concentric.
2. Identify the existing circle.
3. Enter the radial offset.

Circle by Diameter

To draw a circle by diameter:

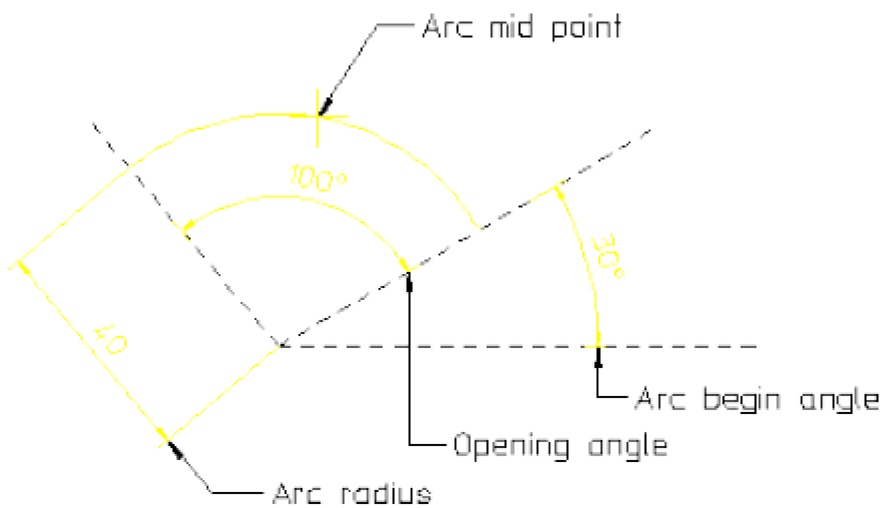
1. Press **CREATE**.
2. Pick Diameter.
3. Indicate the two diameter end points for the circle required.

Drawing Arcs

This section shows how to use the seven commands for drawing arcs. Arcs are normally drawn in a counterclockwise direction. The options are:

- 3 Pts
- Cen R Angs
- Cen & Ends
- Concentric
- Pt R Angs
- Diameter
- Smooth

Figure Arc Geometry



Arc by Three Points

To draw an arc through three points:

1. Press **CREATE**.
2. Pick **3 Pts** or **ARC**.
3. Indicate the two end points of the required arc.
4. Indicate a third point on the arc.

The arc drawn will vary with the movement of the cursor until the third point is defined.

Arc by Center and Radius

To draw an arc with a given center and radius:

1. Press **CREATE**.
2. Pick Cen R Angs.
3. Indicate the center point.
4. Enter the radius required.
5. Enter the start angle and the end angle of the arc. Alternatively, you can define the angles by indicating two points.

Arc by Center and End Points

1. Press **CREATE**.
2. Pick Cen & Ends.
3. Indicate the center of the arc.
4. Indicate the radius (arc start point).
5. Indicate a third point. It does not need to lie on the arc plane, but its line defines the end of the arc.

Concentric Arcs

To draw concentric arcs:

1. Press **CREATE**.
2. Pick Concentric.
3. Identify the existing arc.
4. Indicate a point on the required arc.

Simply indicate points to draw further concentric arcs. You can also enter the radial offsets.

Arc by Mid-Point and Radius

To draw an arc by specifying the arc mid-point and radius:

1. Press **CREATE**.
2. Pick Pt R Angs.
3. Enter or indicate the mid-point of the desired arc.
4. Indicate the arc radius.
5. Enter the arc begin (or end) angle (angles are measured counterclockwise from the horizontal).
6. Enter the arc opening angle.

Arc by Diameter

1. Press **CREATE**.

2. Pick Diameter.

3. Indicate the diameter end points and an arc will be drawn between them.

The resulting position of the arc will depend upon which diameter point is indicated first.

Smooth Connecting Arc

To draw a smooth connecting arc:

1. Press **CREATE**.

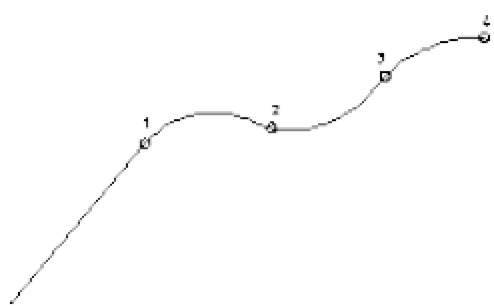
2. Pick Smooth.

3. Identify an element end point (line, circle, and so on).

4. Indicate the end point of the arc.

The arc is drawn tangential to the element you have identified. With each further point indicated; another arc is drawn tangential to the previous one.

Figure Smooth Connecting Arcs

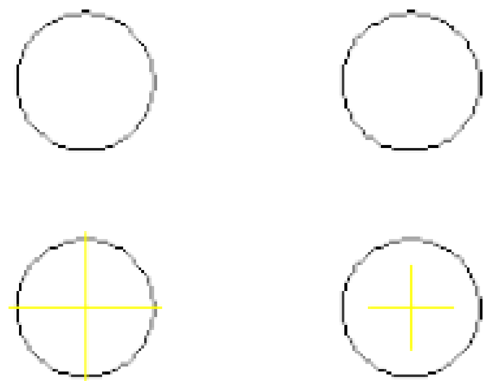


Creating a Centerline

This command lets you create a centerline for a circular element. This command is particularly useful because it speeds-up the early design phase. The centerline becomes part of the circular element and is moved or deleted if the associated circular element is moved or deleted.

To create a centerline, pick CENTERLINE in **CREATE 2** and identify the circle. The different appearance of the centerlines is done by changing absolute and relative values. Refer to the online help on how to do this (help centerline).

Creating a Centerline



Creating Symmetric Elements

The SYM LINE command lets you create elements that are:

• Symmetric with existing elements

• Associated (or anchored) to the defining elements.

When you modify the defining elements, the associated element is modified accordingly.

To draw an associated element:

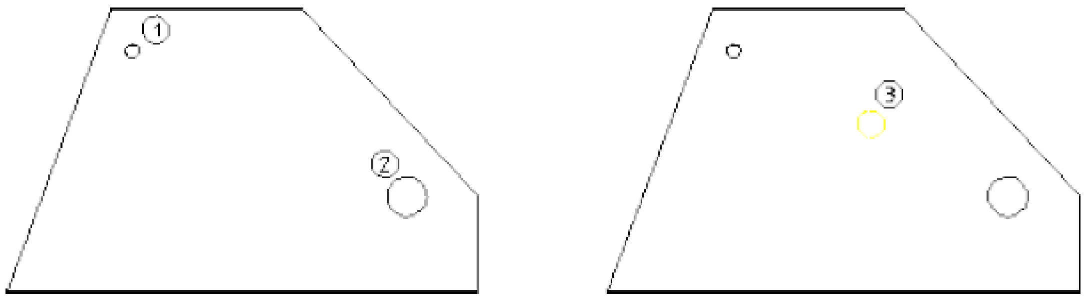
1. Draw a polygon similar to that shown in FIG.
2. Create circle 1 having diameter 1 mm.
3. Create circle 2 having diameter 3 mm.
4. Pick SYM LINE in **CREATE 1**.
5. Pick circle at point 1.
6. Pick circle at point 2.

Creo Elements/Direct Drafting draws the associated circle (point 3) that is:

• Midway between circles 1 and 2.

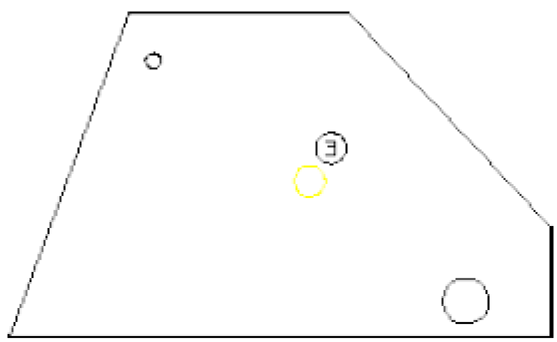
• Is 2 mm in diameter (half the sum of the defining diameters).

Associated Symmetric Elements

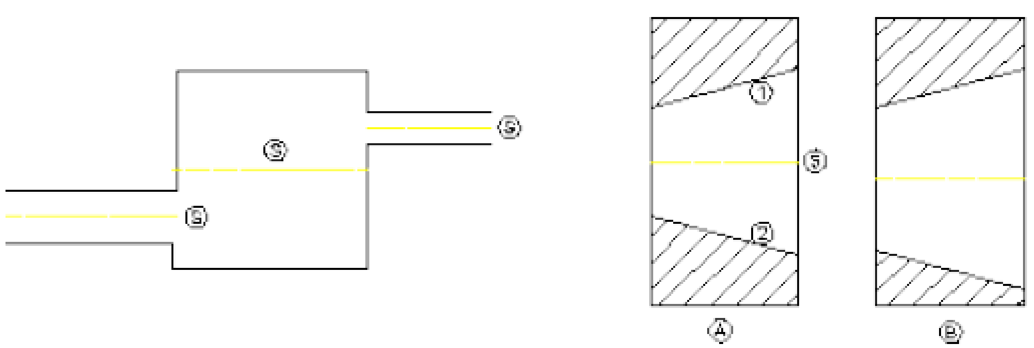


Use Chg SymLin to change the position of the centerline. Now try moving the circle at position 2 and notice how the associated element also moves.

Effect of Moving the Defining Circle



Finding Lines of Symmetry



Splitting Elements

It is often useful to split an element (line, circle, and so on) into two or more parts. By doing this, the segments of lines that intersect can be treated as separate elements and deleted individually.

Splitting Elements Automatically

Before you begin to draw, pick Splitting on in **CREATE** to automatically split elements at their intersection points. This technique keeps contours clean, which is important for automatic hatching.

When a new element is created, Creo Elements/Direct Drafting checks to see if one of the end points of the new element lies on other elements. If so, it splits the other element at that point if possible. When the new element crosses the old element, the old element will not be split.

Splitting Single Element/Elements in a Box

To split a single element:

1. Press **CREATE**.
2. Pick Elem/Box in SPLIT.
3. Identify the element to be split.

Creo Elements/Direct Drafting highlights the chosen element.

4. Indicate the point where the element should be split.

Alternatively, to split several elements automatically at their intersection points:

1. Pick Element/Box.
2. Box the elements to be split (be sure to fully enclose the elements in the box).

To see that all elements have been split, press SHOW POINTS. All existing points will be highlighted.

Dividing an Element Fractionally

To split an element at a point along a fraction of its length:

1. Press **CREATE**.

2. Pick Split On.

3. Pick Divide Len in SPLIT.

4. Identify the ends of the element to be split.

5. Enter a mixing factor (for example, 0.2) or indicate the desired split point.

Merging Elements

To merge elements that have been split:

1. Press **CREATE**.

2. Pick **MERGE**.

3. Identify the elements to be merged.

The elements will then be returned to their original state. Intersection points will no longer be marked when you press **SHOW POINTS**.

Drawing Fillets

To draw a fillet radius:

1. Press **CREATE**.

2. Pick **FILLET**.

3. Enter the radius.

4. Identify the two elements between which the fillet should be drawn, or identify the vertex.

The fillet radius will be drawn and by default the corner point will be deleted. If you do not want the corner to be deleted, pick Keep in CORNER before identifying the elements or vertex.

Changing Fillets

To change fillet radii you have already created:

1. Press **CREATE**.

2. Pick **CHANGE FIL**.

3. Enter the new fillet radius.

4. Identify the individual fillets to be changed or enclose them in a box. You also can change all existing fillets by pressing the option **ALL** in the **SELECTION** block. To remove all fillets, enter a radius of zero.

Drawing Chamfers

Four methods of drawing a chamfer are described in this section. By default, the corner point will be deleted after the chamfer is drawn. If you do not want the corner to be deleted, pick Keep in CORNER before you choose a chamfer option.

Chamfer by Two Points

This is the default option. To draw a chamfer by two points:

1. Press **CREATE**.
2. Pick **2 Pts** or the **CHAMFER** command itself.
3. Indicate the start point and end point of the required chamfer.

Chamfer by Two Distances

To draw a chamfer by specifying two different distances:

1. Press **CREATE**.
2. Pick **Dist Dist**.
3. Enter the distance for chamfer along the first element.
4. Enter the distance for chamfer along the second element.
5. Identify the first element.
6. Identify the second element.

Chamfer at Angle to First Line

To draw a chamfer at a given angle to the first line and at a given distance from the vertex:

1. Press **CREATE**.
2. Pick **Dist Ang**.
3. Enter the angle between the chamfer and the first element.

4. Enter the chamfer distance.
5. Identify the first element.
6. Identify the second element.

Chamfer at Distance from Vertex

To draw a chamfer at a given distance from the vertex along each line:

1. Press **CREATE**.
2. Pick Vertex.
3. Enter the chamfer distance.
4. Identify the vertex (the point where the two lines intersect).

Keeping Corners

When drawing fillets or chamfers, you can keep or delete the corners:

1. Press **CREATE**.
2. Pick **Keep** or **No Keep** (default) in **CORNER**.

Creating a Polyline

Polylines can be created with the POLYELEM command. Polylines are useful during the modification phase (for example, moving furniture around room).

The following example shows some of the differences of working with polyline elements and non-polyline elements. Drawing A is a standard polygon created with Polygon in **CREATE**. To modify the pen sizes of drawing A to produce drawing A1, you need to pick points 1, 2, and 3 (for the purpose of this example, it is assumed that only individual elements can be picked).

Drawing B is similar to drawing A but has been converted into a polyline. Because drawing B is a polyline, it acts like a single element and you need only pick point 4 to modify the complete shape (drawing B1).

Drawing Ellipses

The three commands used for drawing an ellipse are:

• 2 Verts & Pt

• Cen Ang R

• Cen & Pts

• 2 Foc & Pt

• FcAngExMaj

Ellipse by Vertices and Point

This is the default option. To draw an ellipse by vertices and point:

1. Press **CREATE**.

2. Pick **2 Verts&Pt**.

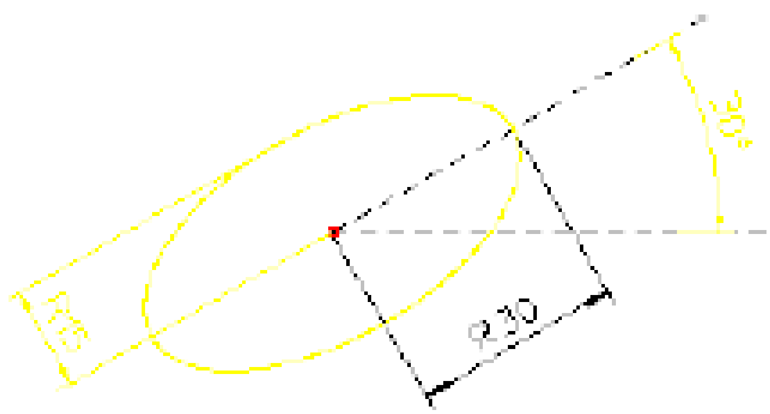
3. Indicate the two vertices of the major axis of the required ellipse.

4. Indicate one point on the circumference.

Ellipse by Center, Angle and Radii

To draw an ellipse by center, angle and radii:

1. Press **CREATE**.



1. Pick Cen Ang R.

2. Indicate the center of the required ellipse.

3. Enter the angle the major axis makes with the x axis

4. Enter the major radius

5. Enter the minor radius

Ellipse by Center and Two Points

To draw an ellipse using the center and two points:

1. Press **CREATE**.
2. Pick **Cen & Pts**.
3. Indicate the center point of the ellipse.
4. Indicate the first point on the circumference.
5. Indicate the second point on the circumference.

Ellipse by Focus and Points

Use the **ELLIPSE** command options 2 Foc&Pt and FcAngExMaj to create elliptical shapes by defining the focus point and major radius.

To create an ellipse by specifying the focus point (drawing A):

1. Pick 2 Foc & Pt
2. Pick first focal point (1)
3. Pick second focal point (2)
4. Pick a point on the circumference (3)

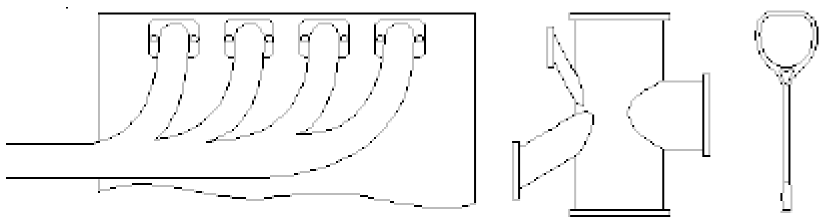
To create an ellipse by specifying the focus and direction (drawing B):

1. Pick **FcAngExMaj**
2. Pick first focal point (point 1)
3. Pick the direction of the major axis (point 2)
4. Enter an Ex-value (for example, 0.8)
5. Enter a value for the major radius (for example, 15)

Drawing Splines

This section describes how to draw spline curves. Spline curves are special types of smooth curves that pass through given data points. When you have specified the data points, Creo Elements/Direct Drafting uses a powerful technique (Bspline mathematics) to produce the shape. The following diagram shows some possible uses of spline curves.

Figure Typical Spline Curves: Engine Manifold, Pipe Tree, Racquet Head



UNIT III

DIMENSIONING A DRAWING

Dimensioning Fundamentals

To clarify how the dimensioning commands work, it is necessary to define the terminology used when discussing dimensioning. Generally, a dimension consists of two parts:

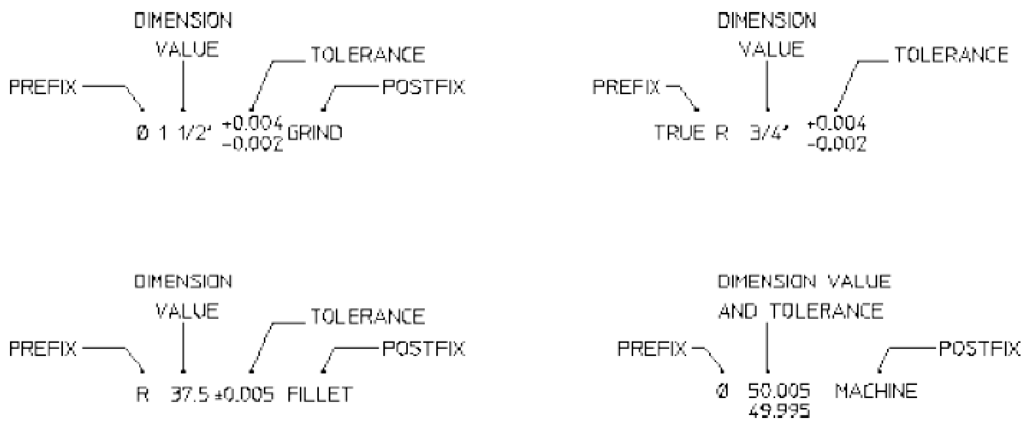
- Dimension Text.
- Dimension Geometry.

What is Dimension Text?

Dimension text describes the geometry feature that is marked by the dimension geometry. Dimension text is divided into the fields:

- Prefix
- Dimension Value
- Tolerance
- Postfix
- Subfix
- Superfix

Dimension Text Examples



Prefix, tolerance, postfix, subfix and superfix are optional.

What is a Prefix?

Use a prefix to include additional information before the dimension value. Examples of prefixes are: DIA (diameter), TRUE R (true radius), and R (radius).

What is a Dimension Value?

A dimension value is the numerical value assigned to the geometry feature you are dimensioning. The value can be imperial, metric or angular depending on the geometry feature and the standard units you use.

What is a Tolerance?

A tolerance is used to include information about the tolerance allowed in the dimension value. Tolerance is the total amount by which a specific dimension is permitted to vary. Types of tolerances are:

• Limit tolerances

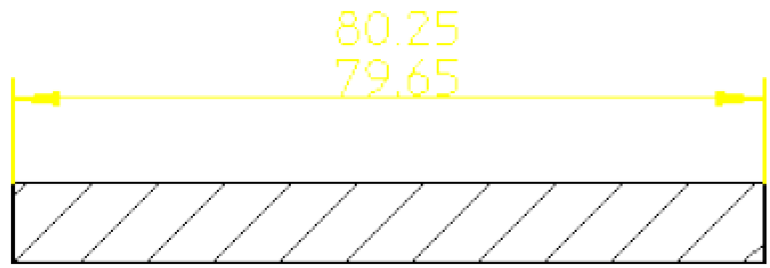
• Plus, and minus tolerances

• Upper and lower tolerances

Limit Tolerances

Limit tolerancing is used to give the maximum and the minimum dimension values. The high limit (maximum value) is placed over the low limit (minimum value).

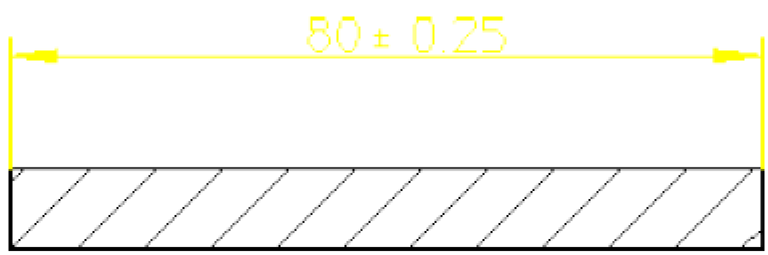
Limit Tolerancing



Plus, and Minus Tolerances

Plus, and minus tolerancing is used to place the tolerance to the right of the dimension value as a plus and minus expression of the tolerance.

Plus and Minus Tolerancing



Upper and Lower Tolerances

Upper and lower tolerancing is used to place the tolerances to the right of the specific dimension value. They appear as upper and lower values of the permissible variation of the size of the feature.

What is Dimension Geometry?

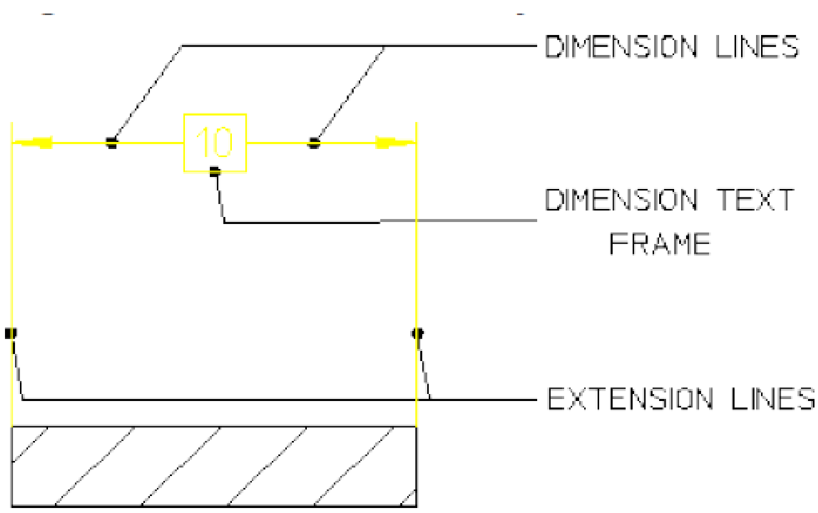
Dimension geometry marks the geometry feature corresponding to the dimension text. Dimension geometry consists of:

• Dimension lines

• Extension lines

• A dimension text frame

Dimension Geometry



Dimension Lines

Dimension lines show the direction and extent of a dimension. The dimension line may be broken and the dimension text inserted (referred to as on the dimension line), or the line may be a full unbroken line with the dimension text above or below the dimension line. The dimension line will normally end with a terminator.

Extension Lines

Extension lines show the extension of a surface or point to a location outside the part outline. In most technical drawings, you draw them perpendicular to the dimension line with a visible gap from the geometry feature.

Dimension Text Frames

The dimension text frame surrounds all dimension text relating to the dimension geometry. Normally, the frame is switched off. You can make the frame appear as a box-shaped frame or a balloon-shaped frame or a flag-shaped frame or a basic frame by selecting the appropriate option.

Setting the Dimension Geometry and Text Distances

As previously described, a dimension is made up of several parts:

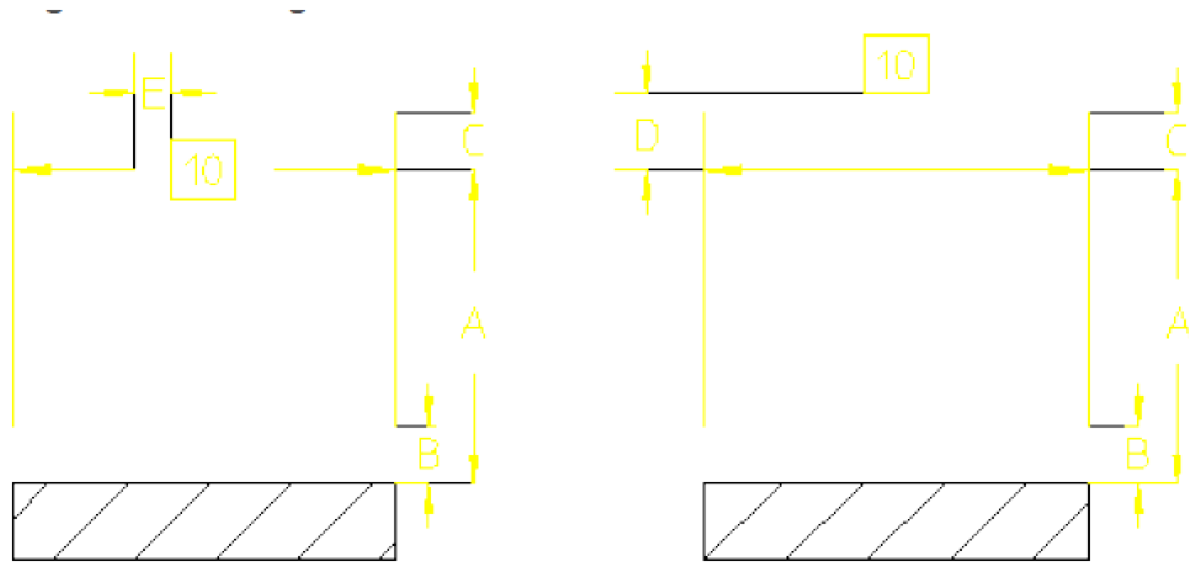
• Dimension text

• Dimension lines

• Extension lines

You can set the distance between some of the parts making up a dimension.

Setting Dimension Distances



Dimensioning With Creo Elements/Direct

Drafting

The dimensioning module has the following features to make dimensioning your drawing quick and easy:

• A full-featured user interface that includes:

○ An efficient screen menu system.

○ Combination of dimensioning commands and functions.

○ Integration with Creo Elements/Direct Drafting common and selection commands.

• Powerful dimension assignment features:

○ A range of dimensioning assignment methods.

○ Multiple elements can be dimensioned at once.

○ Selection aids speed the assignment process.

○ Dimensioning between parts and on layers.

• Tools for dimension style setting and modification:

○ Standards-based dimensioning styles.

○ Style saving and re-assignment via the STYLE table.

○ Simultaneous modification of dimensioning geometry and text.

The dimensioning module is sometimes referred to as the Dimension Advisor (DA).

Dimensioning Commands and Functions

The dimensioning menus provide both commands and functions. Commands and functions can be combined to perform complex operations in a short amount of time. The current dimension style is indicated in a separate display area in the dimensioning menus.

Dimensioning Commands

Generally, a command remains active until:

• You press **END** to end the command.

• You end the command by starting another command.

This means you can repeat the command without having to press or pick the command again.

For example:

1. Press **DIMENSION 1**.
2. Pick the **SINGLE** command.
3. Identify and dimension more than one geometry feature.
4. When you have finished using **SINGLE**, press **END** to end the command.

Dimensioning Functions

Use functions to interrupt commands. Once you have completed the function, Creo Elements/Direct Drafting will return to the active command.

Combining the Dimensioning Commands and Functions

A powerful feature of the Creo Elements/Direct Drafting dimensioning module is the ability to combine dimensioning commands and functions to perform complex operations efficiently. The following procedure illustrates the importance of this feature.

Preparing to Dimension

Before you assign dimensions to your drawing, you may want to set up the basic dimensioning style that will be used for your dimensions. You can manually set dimensioning attributes with the SET LINES and SET TEXTS commands on **DIMENSION 3**, which are described later in this chapter. In addition, you can set up dimensioning to emulate a particular standard.

Selecting a Dimensioning Standard

Creo Elements/Direct Drafting can emulate the dimensioning styles specified in various drawing standards as well as allow you to define your own company or site-wide standard. The standards available are:

- ANSI
- DIN
- ISO
- JIS
- Company (Creo Elements/Direct Drafting default)

To select a dimensioning standard:

1. Press **DIMENSION 1**.
2. Pick the field adjacent to Standard to toggle between standards.

Parallel Dimensions

This is the default option. To dimension along an inclined plane:

1. Press **DIMENSION 1**.
2. Pick **SINGLE** and then **Parallel**.

3. Select **MANUAL** or **AUTO** placement.

4. Select the dimension points or elements to be dimensioned.

5. If MANUAL placement was selected, indicate the desired location for the dimension text(s), using the dimension trace for guidance.

Horizontal Dimensions

To produce dimensions in the horizontal plane:

1. Press **DIMENSION 1**.

2. Pick **SINGLE** and then **Horizontal**.

3. Select **MANUAL** or **AUTO** placement.

4. Select the dimension points or elements to be dimensioned.

5. If MANUAL placement was selected, indicate the desired location for the dimension text(s), using the dimension trace for guidance.

Vertical Dimensions

To produce dimensions in the vertical plane:

1. Press **DIMENSION 1**.

2. Pick **SINGLE** and then **Vertical**.

3. Select **MANUAL** or **AUTO** placement.

4. Select the dimension points or elements to be dimensioned.

5. If MANUAL placement was selected, indicate the desired location for the dimension text(s), using the dimension trace for guidance.

Perpendicular Dimensions

To produce dimensions perpendicular to a reference line:

1. Press **DIMENSION 1**.

2. Pick **SINGLE** and then **Perpend to**.

3. Select **MANUAL** or **AUTO** placement.

4. Identify a reference line. You can identify any line on the drawing as the reference line.

5. Select the dimension points or elements to be dimensioned.

6. If MANUAL placement was selected, indicate the desired location for the dimension text(s), using the dimension trace for guidance.

Parallel to a Reference Line

To produce dimensions Parallel to a reference line:

1. Press **DIMENSION 1**.
2. Pick **SINGLE** and then **Parallel to**.
3. Select **MANUAL** or **AUTO** placement.
4. Identify a reference line. You can identify any line on the drawing as the reference line.
5. Select the dimension points or elements to be dimensioned.
6. If MANUAL placement was selected, indicate the desired location for the dimension text(s), using the dimension trace for guidance.

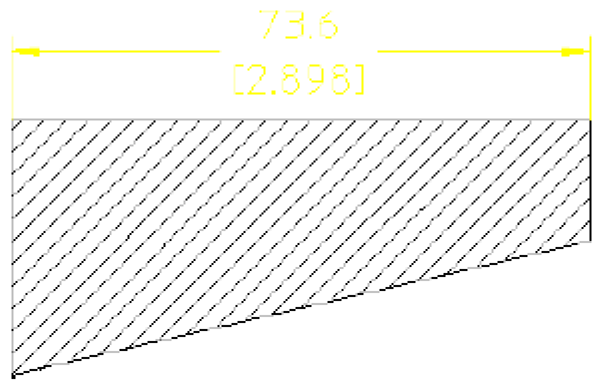
Dual Dimensioning

Optionally, two different units of measure can be used in the same dimension. One of these serves as the primary dimension unit and the other as the secondary dimension unit. To avoid confusion, it is a good idea to enclose the secondary dimension in brackets.

The following example shows you how to set the secondary unit to inches. The example assumes that the primary unit is set to mm.

1. In the SETUP 1 menu, pick Dimensions.
2. In DIMENSIONS ENV, set CURRENT_DIM_TEXTS SEC_ALL BRACKETS to ON.
3. In DIMENSIONS ENV, set Current_secondary_dim_linear_unit to Inches.

Dual Dimensioning



Hatching a Drawing Area

This section shows how to hatch your drawing.

The two command options are:

• Auto

• Manual

Automatic Hatching

This is the default option. You can hatch any area fully enclosed by drawing lines or already hatched. To automatically hatch a drawing area:

1. Press **HATCH**.
2. Pick **Auto** in **START**.
3. Indicate a point inside the area to be hatched.

The cursor will automatically track around the enclosing polygon and then search for any inner holes. Having established the inner and outer contours, Creo Elements/Direct Drafting will hatch the bounded area. If Creo Elements/ Direct Drafting finds any gaps or incorrectly split lines in the enclosing polygon, the command is aborted and Creo Elements/Direct Drafting marks the gap or split line.

Manual Hatching

Manual hatching is used to hatch any area not fully enclosed by drawing lines. To manually hatch a drawing area:

1. Press **HATCH**.
2. Pick **Manual** in START.
3. Indicate the start point on the existing outer contour of the area to be hatched.
4. Continue by indicating successive points around the outer contour.
5. Pick the start point again to close the contour. When you have closed the contour, you can indicate any inner holes that are not to be hatched in a similar manner.
6. Press **END** and Creo Elements/Direct Drafting will hatch the area.

To abort the command press **CANCEL**. Use **UNDO** to undo erroneous points. If a side consists of a complete drawing line, simply indicate a mid-point on the line using the element (**ELEM**) catching mode. Creo Elements/Direct Drafting will accept the line. If you wish to identify only part of a line, mark the desired end point with a construction line, use the intersection (**INTERS**) catching mode, and indicate the desired end point. Creo Elements/Direct Drafting will then accept only part of the line.

A series of connected drawing lines can be indicated by the end of one line and the mid-point of the last line. Creo Elements/Direct Drafting will not accept a series of successive lines if there is a branch at one of the vertices.

Automatic Splitting

You can use automatic splitting options for extended contour creation functionality. To enable these options, click **Setup** ▶ **Hatch** ▶ **HATCH_AUTO_SPLIT**.

Turn this option on. Use these options to automatically detect element intersections and add only parts of elements to contours.

• Use **Next Point** to automatically create a split point at the pick point on a selected element and use that element as the contour element if it is connected to previously selected element. All elements connecting currently picked and previously picked elements are included in the contour. This option remains active until you select another option (Intersect, Next Point).

• Use **Intersect** to automatically find the next intersection of elements on the contour and splits them both at that point. Drafting starts searching for an intersection along the contour in the same direction as between the previous and current pick points. All elements connecting the previously picked element and the intersection are included in the hatch contour. If Drafting doesn’t find an intersection, this option behaves the same as the No Split option. This option remains active until you select another option (Intersect, Next Point).

• Use **No Split** to restore default behavior. The command behaves as if the Automatic Split for Hatch Manual setting is off. This option remains active until you select another option (Intersect, Next Point).

• Use **Direct** to automatically split element at the pick point, and directly connect it to your previous pick point without following the contour. This option remains active only for one pick.

Converting Hatch to Geometry

Hatches are handled differently by different applications. In some cases, hatch that is passed from OSD to AutoCAD can change appearance. To make sure your hatch does not change appearance, convert it to geometry:

1. Click **HATCH**.
2. Click **HATCH->GEO**.
3. "Identify hatch to convert it to the geometry" prompt appears. Click in your viewport window or use the SELECT function to select hatches in the current part. GLOBAL selection is not available.
4. Click **CONFIRM**.
5. Click **END**.

The appearance of your hatching does not change, but it is now geometry. For example, you can delete a single line of the hatch. Completely filled hatches will not convert. In this case, you will see the error message, "Completely filled hatches cannot be converted to geometry" or "Some hatches will not be converted to geometry." You can use UNDO and REDO in combination with Hatch-Geo.

Deleting Hatching

This section shows how to delete areas of hatching.

To delete an area of hatching:

1. Press **HATCH**.
2. Pick **DELETE** in **HATCH**.
3. Identify individual areas of hatching to be deleted, (or enclose a group of hatched areas in a box). Alternatively:

• Use the **SELECTION** block as you would for a normal deletion.

• Finish by pressing **CONFIRM** to delete the selected groups of hatching.

4. If you delete an area by mistake, press **UNDO** to undo the last delete operation.
5. Press **END** to end the command.

Delete hatch that has been converted to geometry the same way you would delete geometry, using the DELETE command.

Setting Hatch Parameters

This section shows how to set hatch parameters before you begin to hatch.

The three command options are:

• Angle

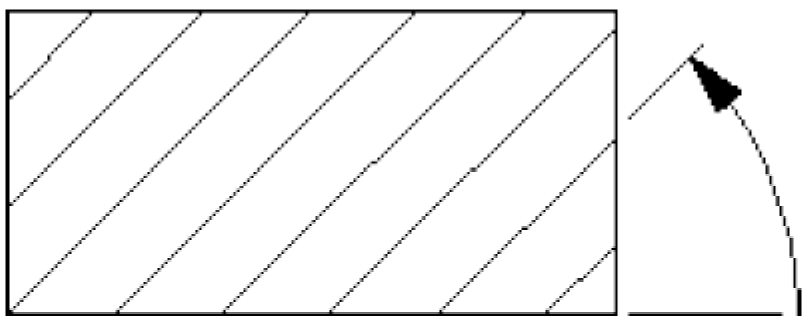
• Distance

• Match Pt (Match Point)

Hatch Angle

This is the default option. The angular units will be those you selected in the **SETUP** module. A zero value is horizontal and a positive angle results in counterclockwise rotation.

Positive Hatch Angle



To set the hatch angle:

1. Press **HATCH**.
2. Pick **Angle** in **SET**.

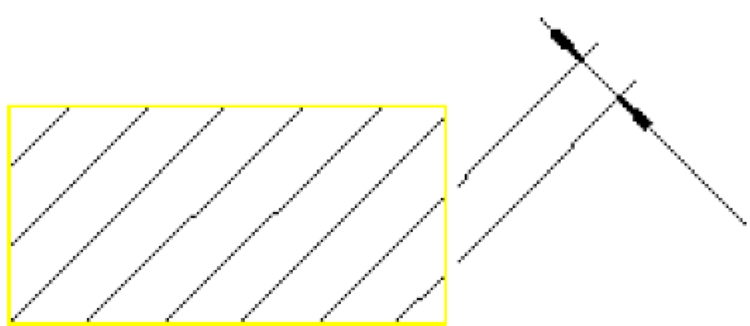
The current hatch angle will be displayed in the System Prompt Line.

3. Enter the new value and all subsequent hatch lines will be drawn at the new hatch angle.

Hatch Distance

You can set the distance between each hatch line for all subsequent hatching.

Hatch Pattern Line Distance



To set the hatch distance:

1. Press **HATCH**.

2. Pick **Distance** in **SET**. The current line distance will be displayed in the System Prompt Line.

3. Enter the new value. All subsequent hatch lines will be drawn with the new hatch line distance.

Entering Text

This section shows how to:

• Enter and position text on a drawing.

• Plant text on a drawing.

ENTER has the following two options:

• Line

• In place

• Screen

Line

This is the default option.

To add single lines or simple blocks of text to your drawing:

1. Press **TEXT 1**.

2. Pick Line in ENTER.

3. Enter the desired text in the User Input Line. The text must be enclosed in quotes (' ' or " ").

4. Indicate the location for the text. Creo Elements/Direct Drafting draws the text.

If you require the same text at several locations on the drawing, simply continue to indicate locations and the text will be repeated.

Inplace

To enter text directly in your drawing:

1. Press **TEXT 1**.
2. Pick Inplace in ENTER.
3. Use the edit keys and keyboard to enter the required text. It is not necessary to use quotes. To force a new line use either the [Enter] key or the edit keys.
4. When you have finished entering text, press [Ctrl] [D].

Screen

To add large blocks of text to your drawing:

1. Press **TEXT 1**.
2. Pick Screen in ENTER.

The normal display will be replaced by a blank screen.

3. Use the edit keys and keyboard to enter the required text. It is not necessary to use quotes. To force a new line use either the [Enter] key or the edit keys.
4. When you have finished entering text press [Ctrl] [D]. The normal display will reappear.
5. Indicate the desired location for the block.

Creo Elements/Direct Drafting will draw the block of text.

Cutting and Pasting with the Mouse

Text displayed in a non-Drafting window can be copied (by dragging the mouse over it) and pasted on the user input line or into the Creo Elements/Direct Drafting text editor. You must first copy the text to the clipboard.

To copy text:

1. Display the (terminal) window containing the text
2. Click the beginning of the text string and drag the cursor over the text
3. Release the mouse

To paste text:

1. Click ENTER in **TEXT 1**

2. Type in the start-quote character (')

3. Click the middle mouse button

4. Type in the end-quote character (')

5. Press [Enter]

6. In the Creo Elements/Direct Drafting drawing area, click the left mouse button to place the text

Text can be pasted onto the user input line or in the Creo Elements/Direct Drafting editor only. It cannot be pasted directly into the drawing area.

Editing and Deleting Text

To edit existing text in a separate screen:

1. Press **TEXT 1**.

2. Pick EDIT.

3. Identify the text to be edited.

The normal display will be replaced by a blank screen containing the text to be edited.

4. Use the keyboard and edit keys to make the required changes to the text.

5. Press [Ctrl] [D].

The normal display will reappear and Creo Elements/Direct Drafting will change the identified text.

To edit text directly in the drawing:

1. Press **TEXT 1**.

2. Pick Inplace in EDIT.

3. Identify the text to be edited.

4. Use the keyboard and edit keys to make the required changes to the text.

5. Press [Ctrl] [D]. The EDIT CONV command is included in the **TEXT 1** menu for convenience. It is the same command as displayed in the **SYMBOLS 1** menu and is used to allow editing of symbol-type fonts.

Creo Elements/Direct Drafting deletes text in the same way as normal drawing lines. To delete text without using the selection block:

1. Press **DELETE**.
2. Box the text to be deleted, or indicate a single piece of text. Creo Elements/Direct Drafting will now delete the identified text. Press **UNDO** to correct mistakes and then repeat the operation or end the command. To delete text using the selection block:
1. Press **DELETE**.
2. Select text for deletion using the **SELECTION** block options just as you would for a normal **DELETE** command. You can press **UNDO** to correct any mistakes.
3. Press **CONFIRM**. Creo Elements/Direct Drafting deletes the selected text.

UNIT IV

GD&T FRAMEWORK

Dimensioning

Before an object can be built, complete information about both the size and shape of the object must be available. The exact shape of an object is communicated through orthographic drawings, which are developed following standard drawing practices. The process of adding size information to a drawing is known as dimensioning the drawing.

- **Geometrics** is the science of specifying and tolerancing the shapes and locations of features on objects. Once the shape of a part is defined with an orthographic drawing, the size information is added also in the form of **dimensions**.
- Dimensioning a drawing also identifies the tolerance (or accuracy) required for each dimension.
- If a part is dimensioned properly, then the intent of the designer is clear to both the person making the part and the inspector checking the part.
- A fully defined part has three elements: graphics, dimensions, and words (notes).

Size and Location Dimensions

- A well dimensioned part will communicate the size and location requirements for each feature. Communications is the fundamental purpose of dimensions.
- Parts are dimensioned based on two criteria:
- Basic size and locations of the features.
- Details of a part's construction and for manufacturing.

Unit of measure

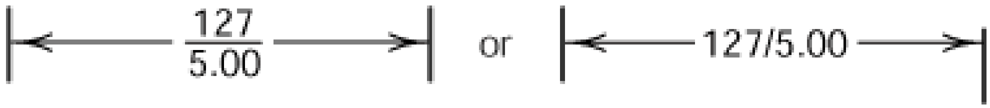
 On a drawing used in American industry, all dimensions are in inches, unless otherwise stated.

 Most countries outside of the United States use the metric system of measure, or the international system of units (SI), which is based on the meter.

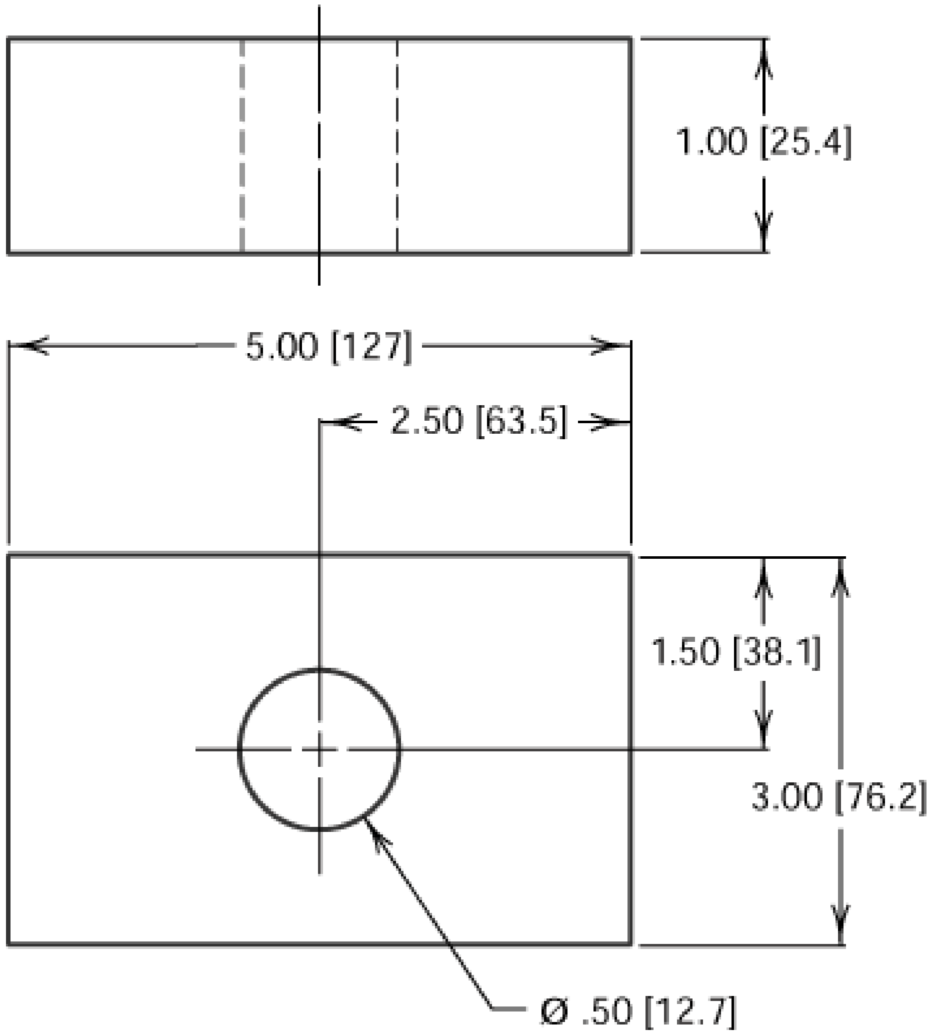
 The SI system is being used more in the United States because of global trade and multinational company affiliations.

 Occasionally, a company will used dual dimensioning, that is, both metric and English measurements on a drawing.

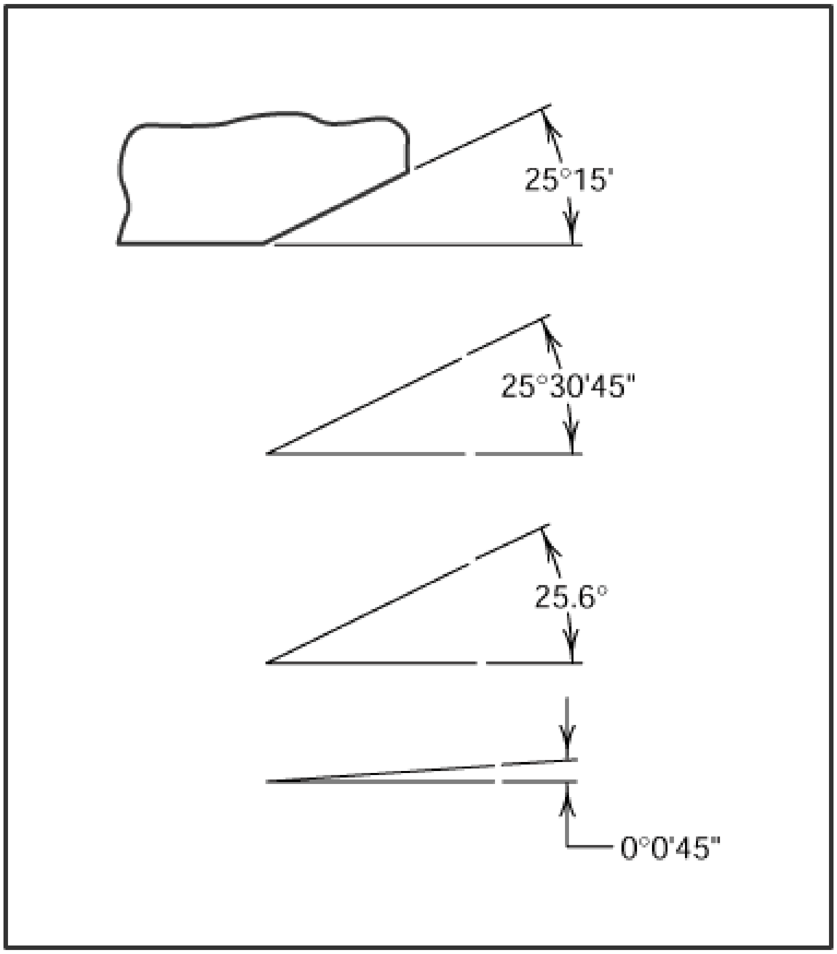
 Angular dimensions are shown either in decimal degrees or in degrees, minutes, and seconds.



(A) Position Method



(B) Bracket Method

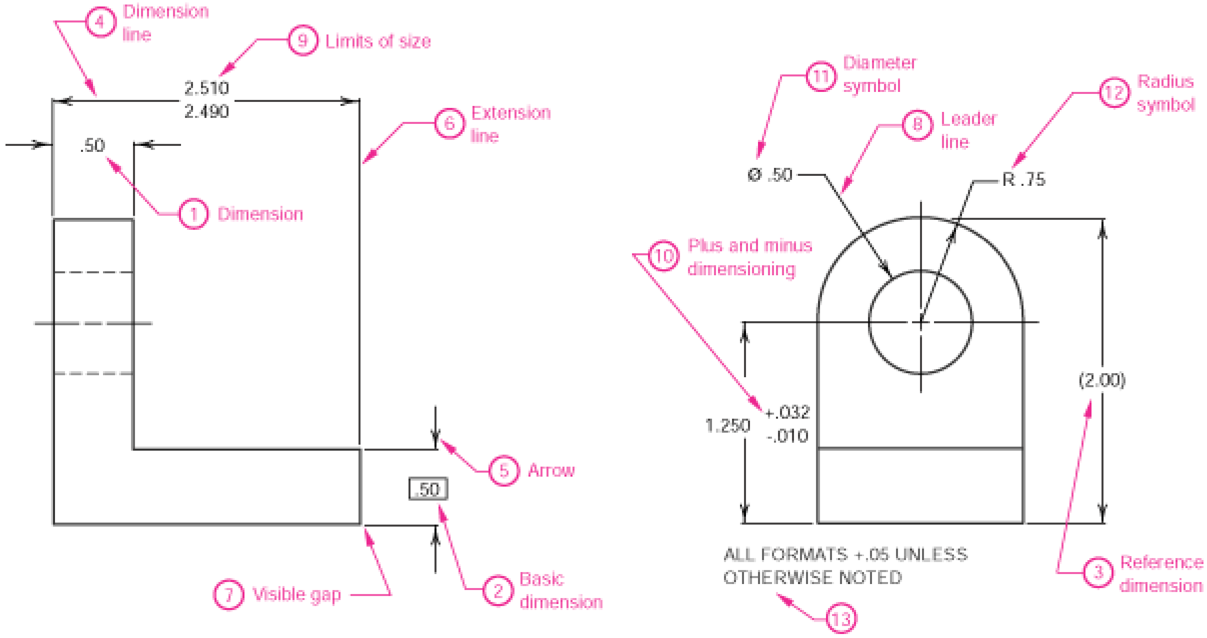


Terminology

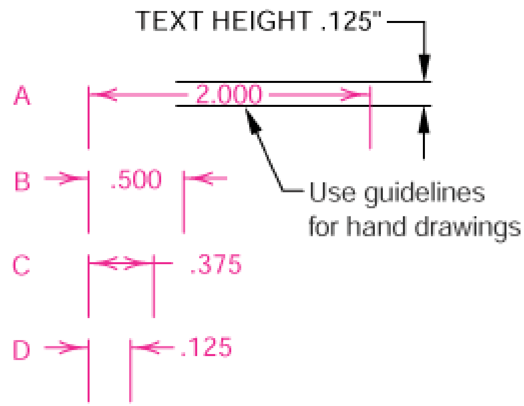
 Dimension is the numerical value that defines the size or geometric characteristic of a feature.

 Basic dimension is the numerical value defining the theoretically exact size of a feature.

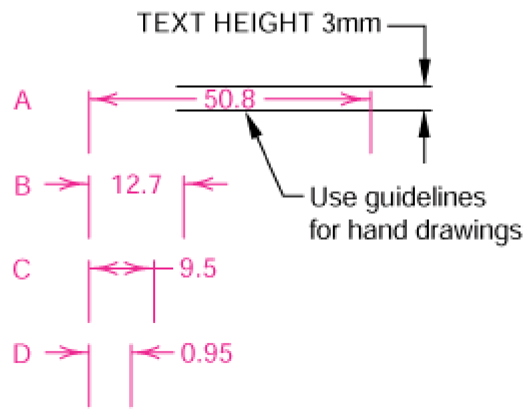
 Reference dimension is the numerical value enclosed in parentheses provided for information only and is not used in the fabrication of the part.



4 Dimension line
9 Limits of size
2.510
2.490
6 Extension line
.50
1 Dimension
5 Arrow
.50
2 Basic dimension
7 Visible gap
11 Diameter symbol
8 Leader line
12 Radius symbol
Ø .50
R .75
10 Plus and minus dimensioning
1.250
+.032
-.010
(2.00)
ALL FORMATS +.05 UNLESS OTHERWISE NOTED
13
3 Reference dimension



Decimal dimensioning



Millimeter dimensioning

- Leader line is the thin solid line used to indicate the feature with which a dimension, note, or symbol is associated.

- Tolerance is the amount a particular dimension is allowed to vary.

- Plus, and minus dimensioning is the allowable positive and negative variance from the dimension specified.

- Limits of size is the largest acceptable size and the minimum acceptable size of a feature.

- The largest acceptable size is expressed as the maximum material condition (MMC)

- The smallest acceptable size is expressed as the least material condition (LMC).

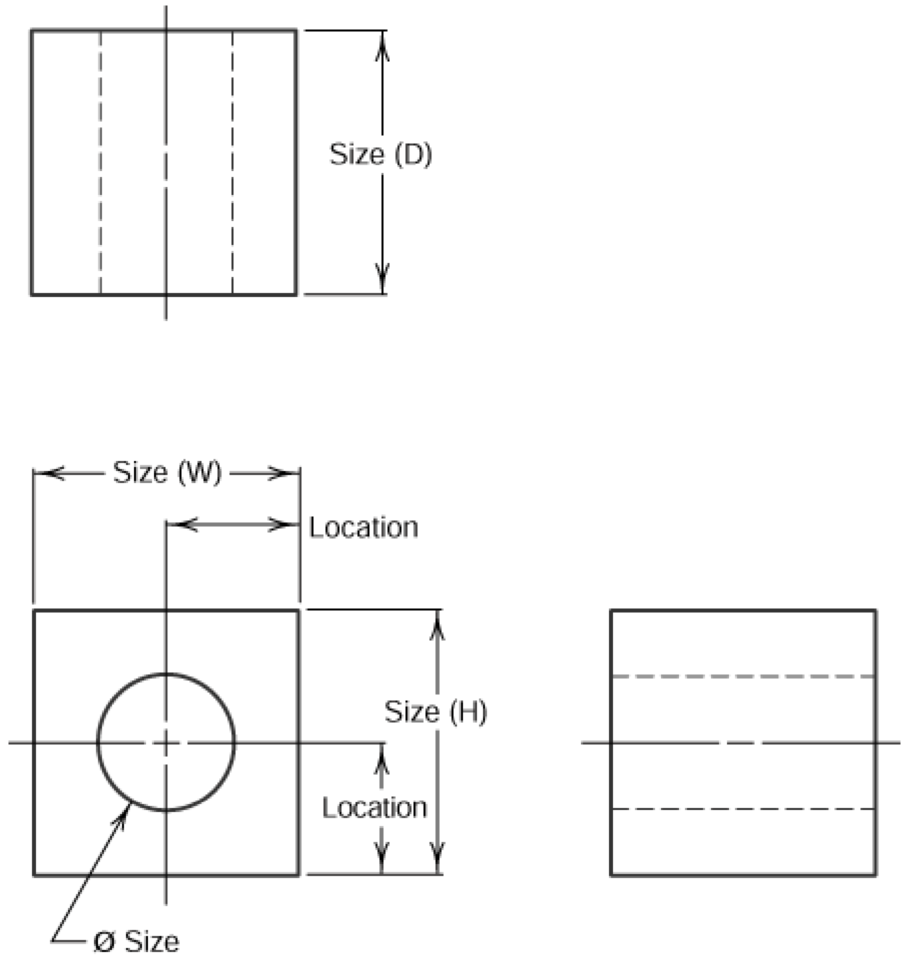
- Diameter symbol is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the diameter of a circle. The symbol used is the Greek letter phi.

- Radius symbol is the symbol which is placed preceding a numerical value indicating that the associated dimension shows the radius of a circle. The radius symbol used is the capital letter R.

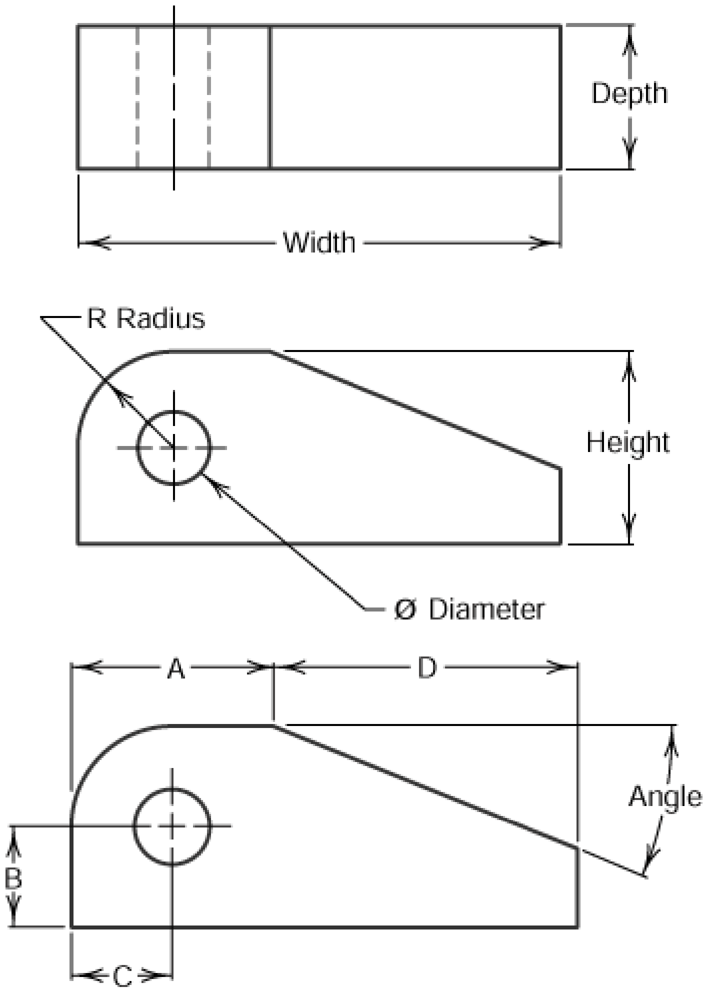
- Datum is the theoretically exact point used as a reference for tabular dimensioning.

Basic Concepts

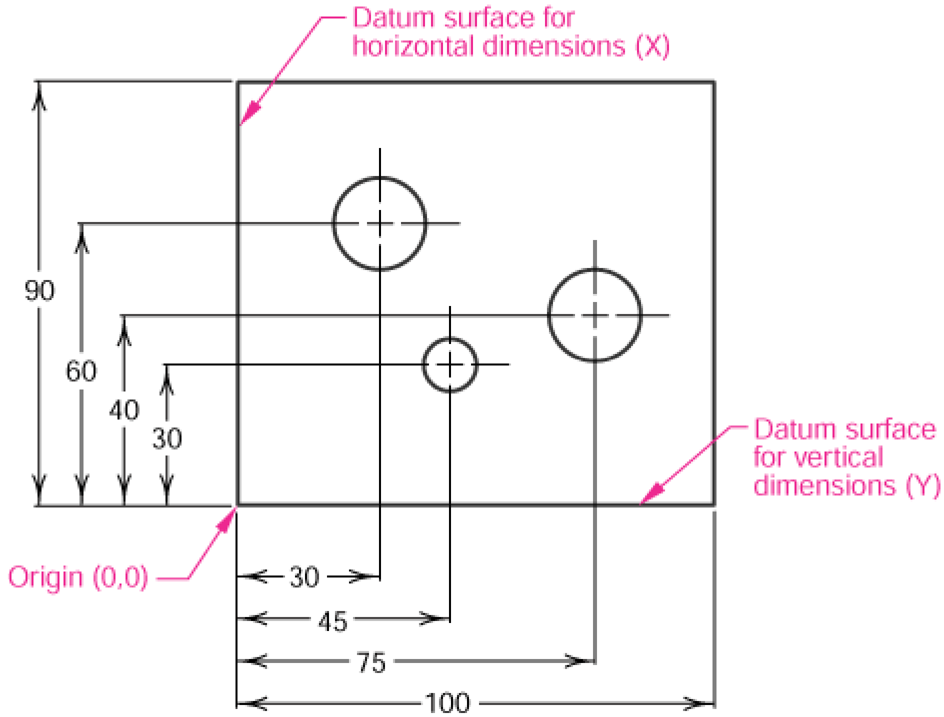
- Size dimension might be the overall width of the part or the diameter of a drilled hole.
- Location dimension might be length from the edge of the object to the center of the drilled hole.



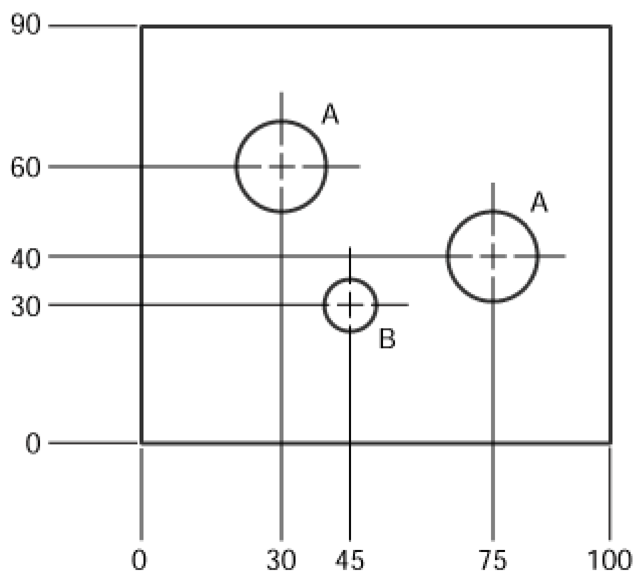
- Size dimensions
- Horizontal
- Vertical
- Diameter
- Radius
- Location and Orientation
- Horizontal
- Vertical
- Angle



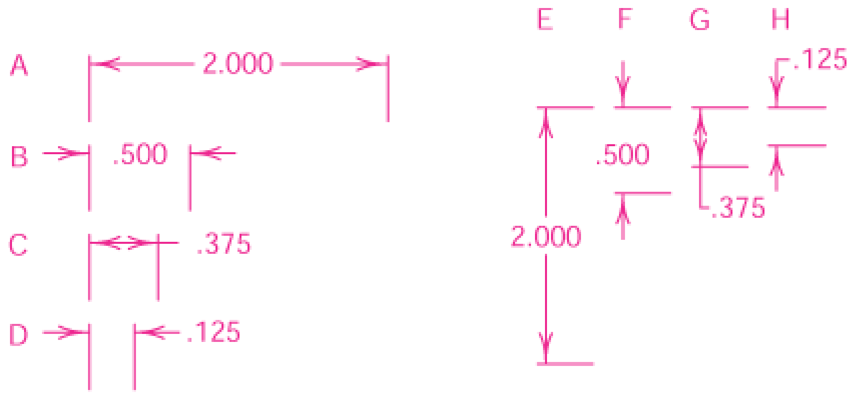
 Rectangular coordinate dimensioning, a base line (or datum line) is established for each coordinate direction, and all dimensions specified with respect to these baselines. This is also known as datum dimensioning, or baseline dimensioning. All dimensions are calculated as X and Y distances from an origin point, usually placed at the lower left corner of the part.



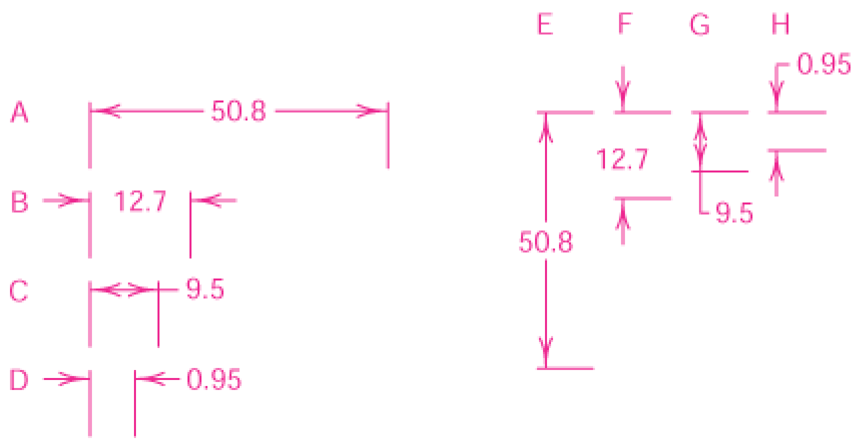
Symbol	A	B
Hole diameter	20	10



 Dimension placement depends on the space available between extension lines. When space permits, dimensions and arrows are placed between the extension lines.



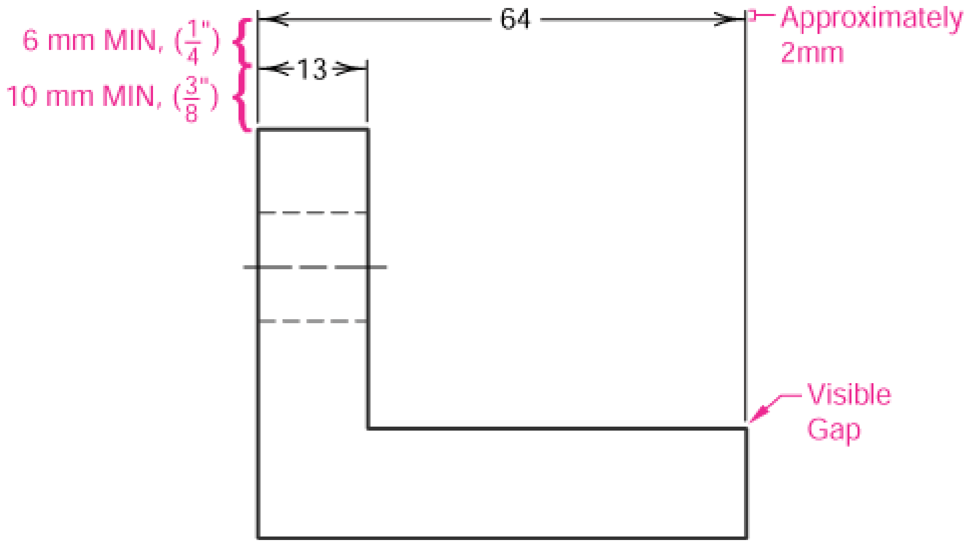
Decimal dimensioning



Millimeter dimensioning

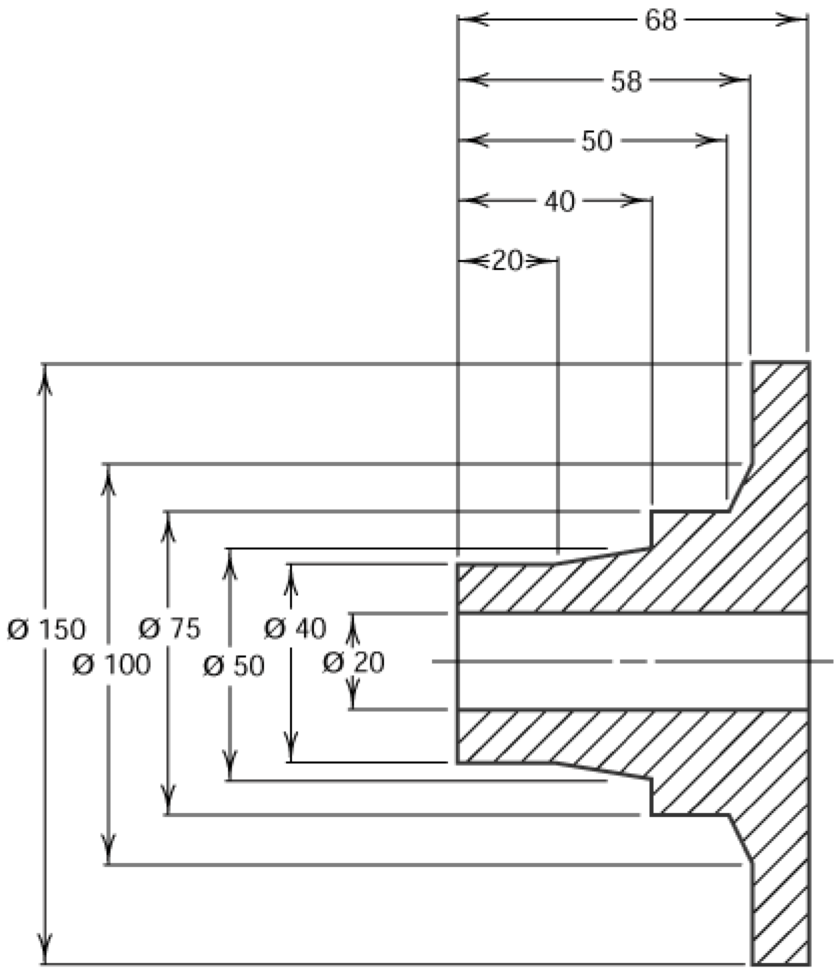
Standard Practices- Spacing

- The minimum distance from the object to the first dimension is 10mm (3/8 inch). The minimum spacing between dimensions is 6mm (1/4 inch).
- There should be a visible gap between an extension line and the feature to which it refers.
- Extension lines should extend about 1mm (1/32 inch) beyond the last dimension line.



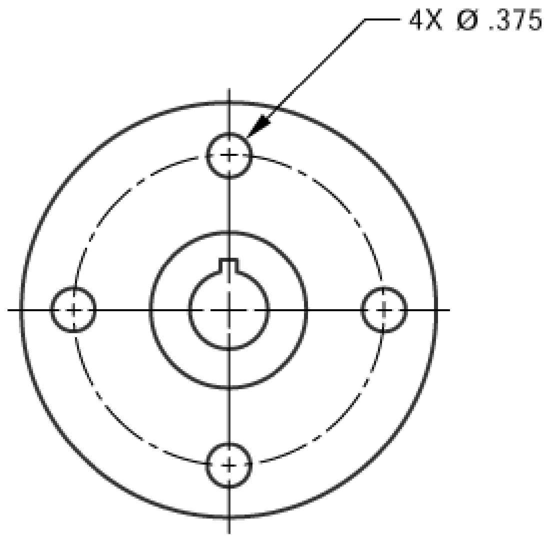
Standard Practices-Staggering

 Where there are several parallel dimensions, the values should be staggered.



Repetitive Features

 The symbol X is used to indicate the number of times a feature is to be repeated. The number of repetitions, followed by the symbol X and a space precedes the dimension text.



Dimension Guidelines

 The primary guideline is that of clarity and whenever two guidelines appear to conflict, the method which most clearly communicates the size information shall prevail.

 Every dimension must have an associated tolerance, and that tolerance must be clearly shown on the drawing.

 Avoid over-dimensioning a part. Double dimensioning of a feature is not permitted.

 Dimensions should be placed in the view which most clearly describes the feature being dimensioned.

 A minimum spacing between the object and dimensions and between dimensions must be maintained.

- A visible gap shall be placed between the end of extension lines and the feature to which they refer.
- Manufacturing methods should not be specified as part of the dimension unless no other method of manufacturing is acceptable.
- Placing dimensions within the boundaries of a view should be avoided whenever practicable.
- Dimensions for materials typically manufactured to gages or code numbers shall be specified by numerical values.
- Unless otherwise specified, angles shown on drawings are assumed to be 90 degrees.
- Dimensioning to hidden lines should be avoided whenever possible. Hidden lines are less clear than visible lines.
- The depth of blind, counterbored, or countersunk holes may be specified in a note along with the diameter.
- Diameters, radii, squares, counterbores, spotfaces, countersinks, and depth should be specified with the appropriate symbol preceding the numerical value.
- Leader lines for diameters and radii should be radial lines.

Tolerancing

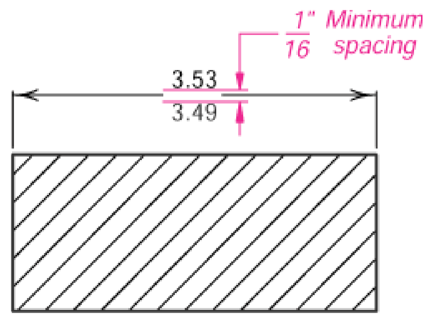
- Tolerance is the total amount a dimension may vary and is the difference between the upper (maximum) and lower (minimum) limits.
- Tolerances are used to control the amount of variation inherent in all manufactured parts. In particular, tolerances are assigned to mating parts in an assembly.
- One of the great advantages of using tolerances is that it allows for interchangeable parts, thus
permitting the replacement of individual parts.
- Tolerances are used in production drawings to control the manufacturing process more accurately and control the variation between parts.

UNIT V SURFACE STANDARDS

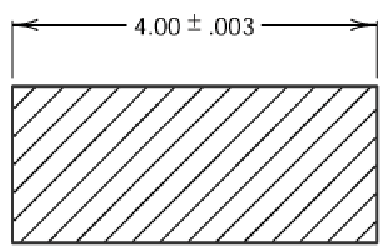
Tolerancing

Tolerance representation

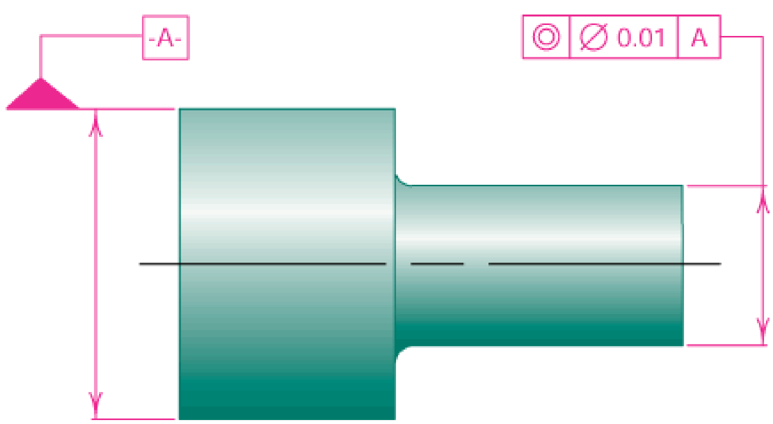
- Direct limits or as tolerance values applied directly to a dimension.
- Geometric tolerances
- Notes referring to specific condition.



(A) Direct limits

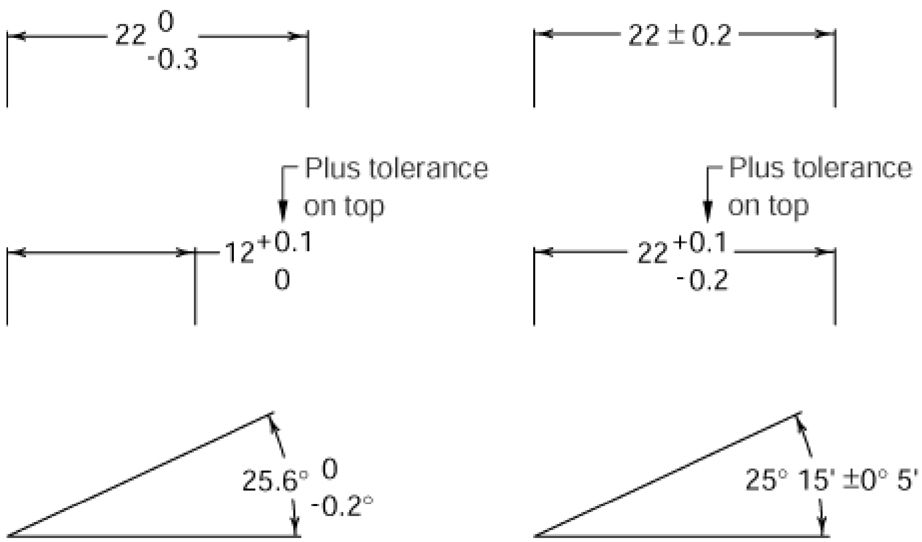


(B) Tolerance values



 Tolerance representation

 Plus/Minus



(A) Unilateral tolerancing (B) Bilateral tolerancing

 Important terms

 Nominal size a dimension used to describe the general size usually expressed in common fractions.

 Basic size the theoretical size used as a starting point for the application of tolerances.

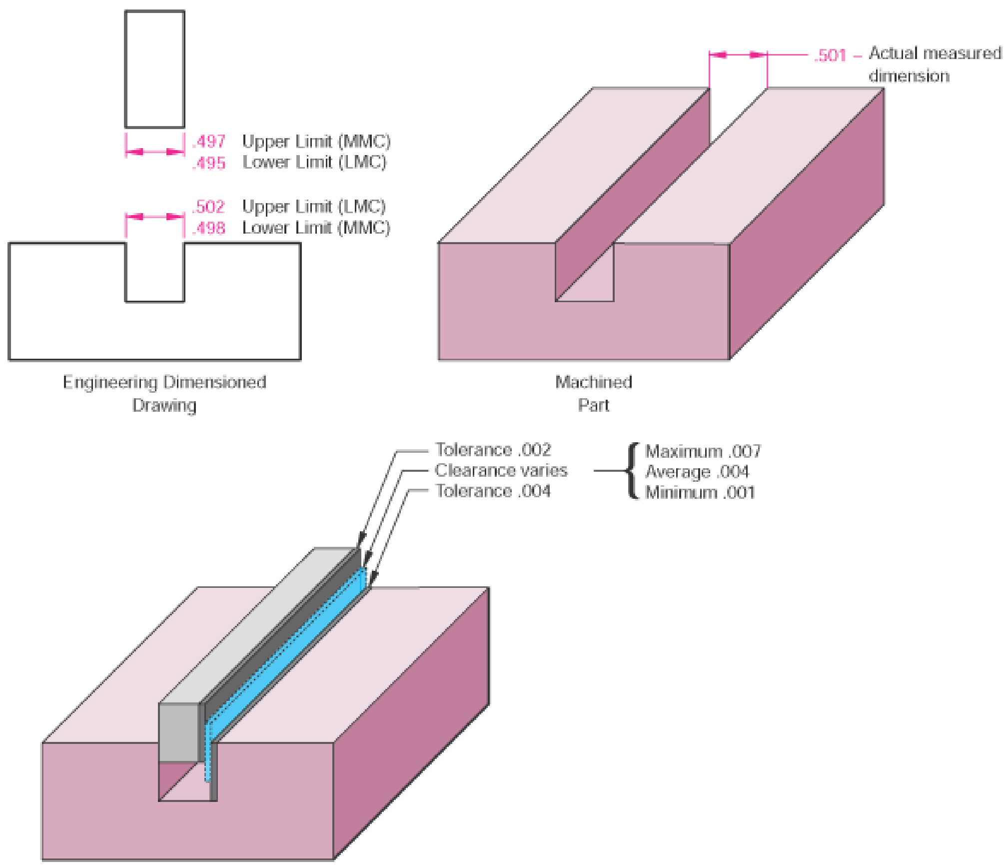
 Actual size the measured size of the finished part after machining.

Limits the maximum and minimum sizes shown by the toleranced dimension.

 Allowance is the minimum clearance or maximum interference between parts.

 Tolerance is the total variance in a dimension which is the difference between the upper and lower limits. The tolerance of the slot in Figure 14.50 is .004" and the tolerance of the mating part is .002".

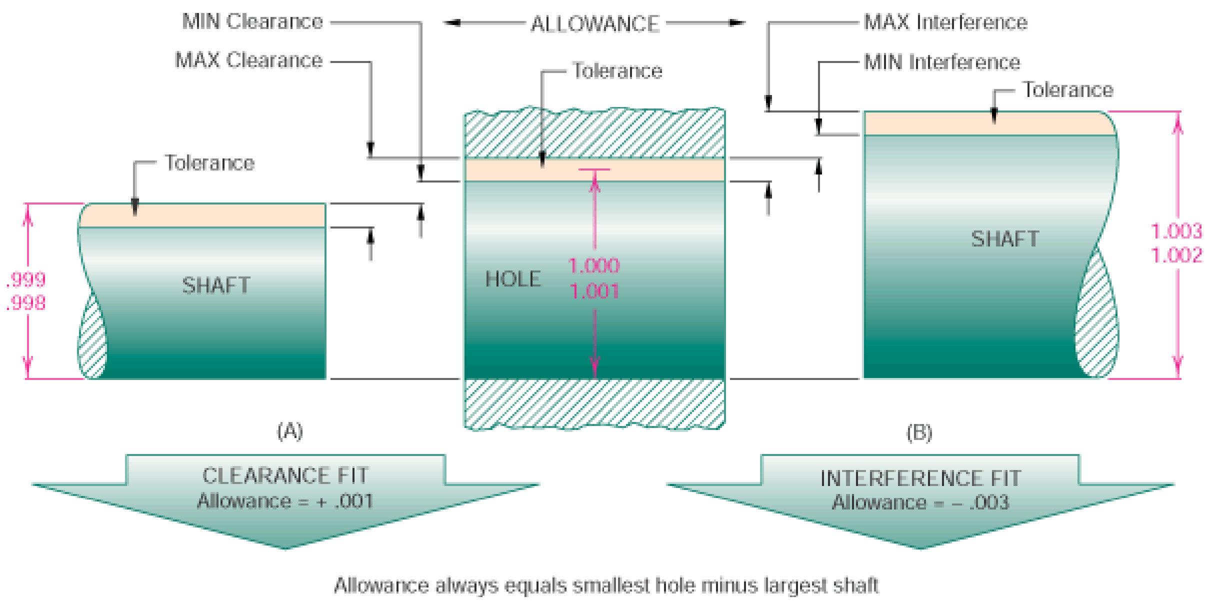
- **Maximum material condition (MMC)** is the condition of a part when it contains the most amount of material. The MMC of an external feature such as a shaft is the upper limit. The MMC of an internal feature such as a hole is the lower limit.
- **Least material condition (LMC)** is the condition of a part when it contains the least amount of material possible. The LMC of an external feature is the lower limit of the part. The LMC of an internal feature is the upper limit of the part.

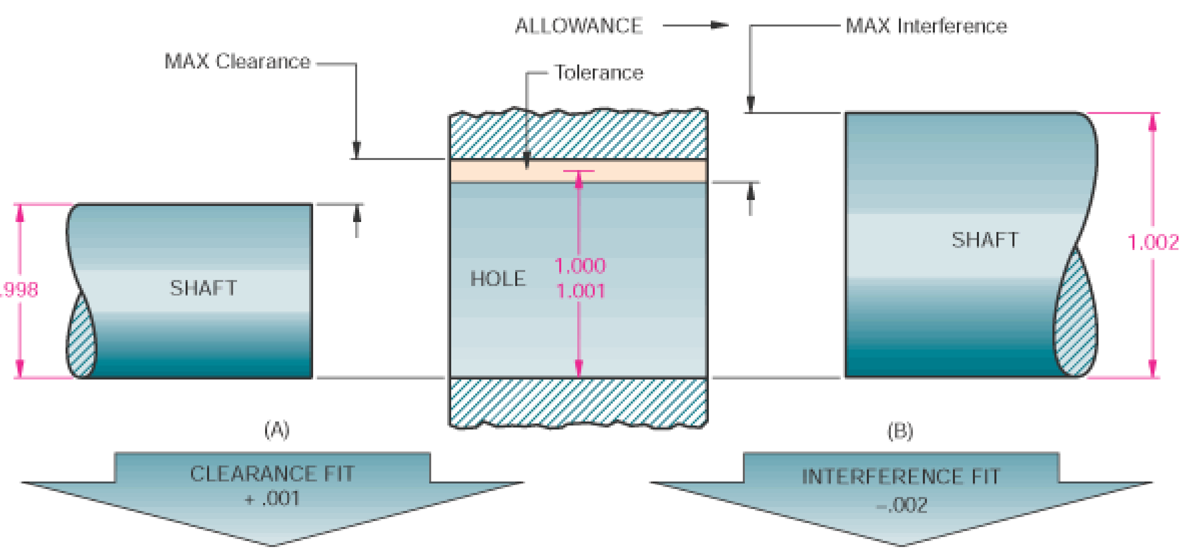


- **Fit types**
  - **Clearance fit** occurs when two tolerance mating parts will always leave a space or clearance when assembled.

 Interference fit occurs when two tolerance mating parts will always interfere when assembled.

 Transition fit occurs when two toleranced mating parts will sometimes be an interference fit and sometimes be a clearance fit when assembled.



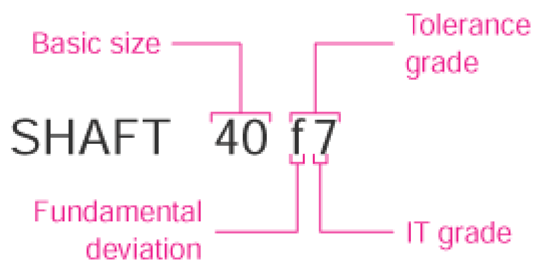


- **Metric Limits and Fits**
    - Basic size
    - Deviation
    - Upper Deviation
    - Lower Deviation
    - Fundamental Deviation Tolerance
    - Tolerance zone
    - International tolerance grade
    - Hole basis
    - Shaft basis

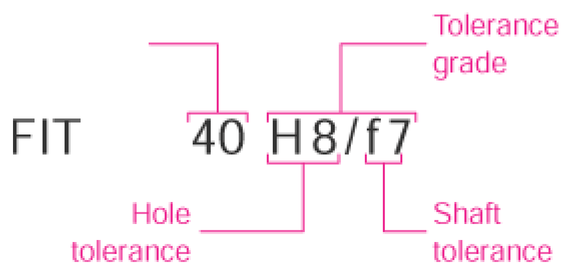
- **Symbols and Definitions**
    - Methods

(A)	(B)	(C)
40H8	$40H8 \begin{pmatrix} 40.039 \\ 40.000 \end{pmatrix}$	$\begin{pmatrix} 40.039 \\ 40.000 \end{pmatrix} 40H8$

(A)



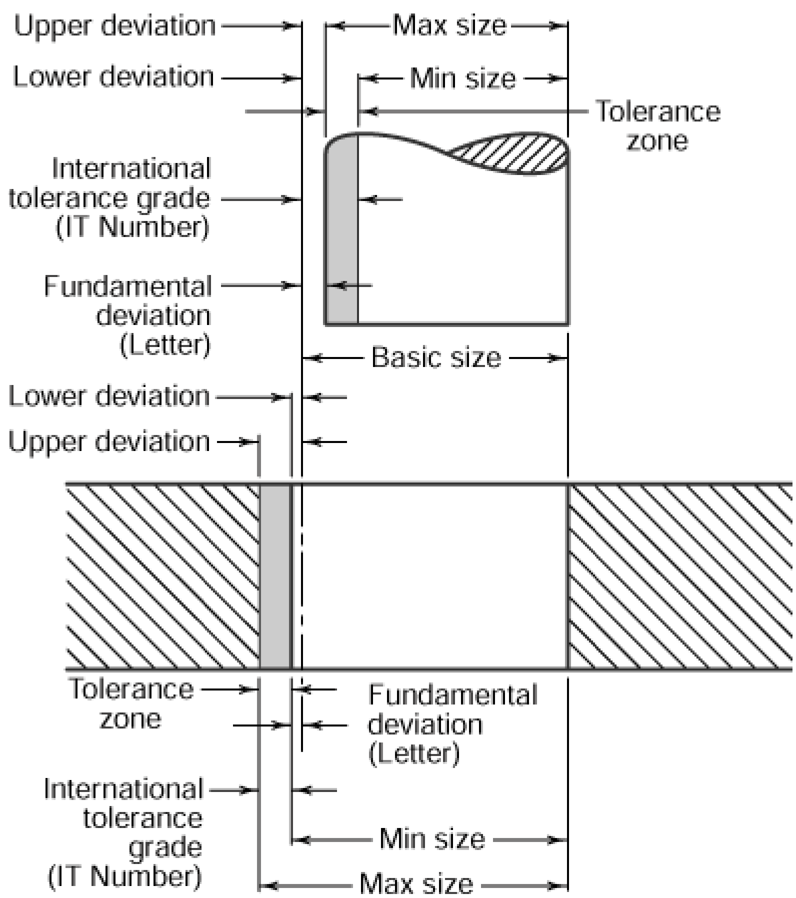
(B)



(C)

 Standard Hole basis table; limits

BASIC SIZE		LOOSE RUNNING			FREE RUNNING			CLOSE RUNNING			SLIDING			LOCATIONAL CLEARANCE		
		Hole H11	Shaft c11	Fit	Hole H9	Shaft d9	Fit	Hole H8	Shaft f7	Fit	Hole H7	Shaft g6	Fit	Hole H7	Shaft h6	Fit
40	MAX	40.160	39.880	0.440	40.062	39.920	0.204	40.039	39.975	0.029	40.025	39.991	0.050	40.025	40.000	0.041
	MIN	40.000	39.720	0.120	40.000	39.858	0.060	40.000	39.950	0.025	40.000	39.975	0.009	40.000	39.984	0.000
50	MAX	50.160	49.870	0.450	50.062	49.920	0.204	50.039	49.975	0.089	50.025	49.991	0.050	50.025	50.000	0.041
	MIN	50.000	49.710	0.130	50.000	49.858	0.080	50.000	49.950	0.025	50.000	49.975	0.009	50.000	49.984	0.000
60	MAX	60.190	59.860	0.520	60.074	59.900	0.248	60.046	59.970	0.106	60.030	59.990	0.059	60.030	60.000	0.049
	MIN	60.000	59.670	0.140	60.000	59.826	0.100	60.000	59.940	0.030	60.000	59.971	0.010	60.000	59.981	0.000
80	MAX	80.190	79.550	0.530	80.074	79.900	0.248	80.046	79.970	0.106	80.030	79.990	0.059	80.030	80.000	0.049
	MIN	80.000	79.660	0.150	80.000	79.826	0.100	80.000	79.940	0.030	80.000	79.971	0.010	80.000	79.981	0.000
100	MAX	100.220	99.830	0.610	100.087	99.880	0.294	100.054	99.964	0.125	100.035	99.988	0.069	100.035	100.000	0.057
	MIN	100.000	99.610	0.170	100.000	99.793	0.120	100.000	99.929	0.036	100.000	99.966	0.012	100.000	99.978	0.000
120	MAX	120.220	119.820	0.620	120.087	119.880	0.294	120.054	119.964	0.125	120.035	119.988	0.069	120.035	120.000	0.057
	MIN	120.000	119.600	0.180	120.000	119.793	0.120	120.000	119.929	0.036	120.000	119.966	0.012	120.000	119.978	0.000
160	MAX	160.250	159.790	0.710	160.100	159.855	0.345	160.063	159.957	0.146	160.040	159.986	0.078	160.040	160.000	0.065
	MIN	160.000	159.540	0.210	160.000	159.755	0.145	160.000	159.917	0.043	160.000	159.961	0.014	160.000	159.975	0.000
200	MAX	200.290	199.760	0.820	200.115	199.830	0.400	200.072	199.950	0.168	200.046	199.985	0.040	200.046	200.000	0.075
	MIN	200.000	199.470	0.240	200.000	199.715	0.170	200.000	199.904	0.050	200.000	199.956	0.015	200.000	199.971	0.000
250	MAX	250.290	249.720	0.860	250.115	249.830	0.400	250.072	249.950	0.168	250.046	249.985	0.090	250.046	250.000	0.075
	MIN	250.000	249.430	0.280	250.000	249.715	0.170	250.000	249.904	0.050	250.000	249.956	0.015	250.000	249.971	0.000
300	MAX	300.320	299.670	0.970	300.130	299.810	0.450	300.081	299.944	0.189	300.052	299.983	0.101	300.052	300.000	0.084
	MIN	300.000	299.350	0.330	300.000	299.680	0.190	300.000	299.892	0.056	300.000	299.951	0.017	300.000	299.968	0.000
400	MAX	400.360	399.600	1.120	400.140	399.790	0.490	400.089	399.938	0.208	400.057	399.982	0.111	400.057	400.000	0.093
	MIN	400.000	399.240	0.400	400.000	399.650	0.210	400.000	399.881	0.062	400.000	399.946	0.018	400.000	399.964	0.000
500	MAX	500.400	499.520	1.280	500.155	499.770	0.540	500.097	499.932	0.228	500.063	499.980	0.123	500.063	500.000	0.103
	MIN	500.000	499.120	0.480	500.000	499.615	0.230	500.000	499.869	0.068	500.000	499.940	0.020	500.000	499.960	0.000



	ISO Symbol		Description	
	Hole Basis	Shaft Basis		
Clearance fits	H11/c11	C11/h11	Loose running fit for wide commercial tolerances or allowances on external members	More clearance
Clearance fits	H9/d9	D9/h9	Free running fit not for use where accuracy is essential, but good for large temperature variations, high running speeds, or heavy journal pressures	
Clearance fits	H8/f7	F8/h7	Close running fit for running on accurate machines and for accurate location at moderate speeds and journal pressures	
Clearance fits	H7/g6	G7/h6	Sliding fit not intended to run freely but to move and turn freely and locate accurately	
Transition fits	H7/h6	H7/h6	Locational clearance fit provides snug fit for locating stationary parts but can be freely assembled and disassembled	
Transition fits	H7/k6	K7/h6	Locational transition fit for accurate location; a compromise between clearance and interference	
Transition fits	H7/n6	N7/h6	Locational transition fit for more accurate location where greater interference is permissible	
Interference fits	H7/p6*	P7/h6	Locational interference fit for parts requiring rigidity and alignment with prime accuracy of location but without special bore pressure requirements	
Interference fits	H7/s6	S7/h6	Medium drive fit for ordinary steel parts or shrink fits on light sections; the tightest fit usable with cast iron	
Interference fits	H7/u6	U7/h6	Force fit suitable for parts that can be highly stressed or for shrink fits where the heavy pressing forces required are impractical	More interference

*Transition fit for basic sizes in range from 0 through 3 mm

- **Standard Precision Fit; English Units**
  - Running and Sliding (RC)
  - Clearance Locational (LC)
  - Transition Locational (LT)
  - Interference Locational (LN)
  - Force and Shrinks (FN)

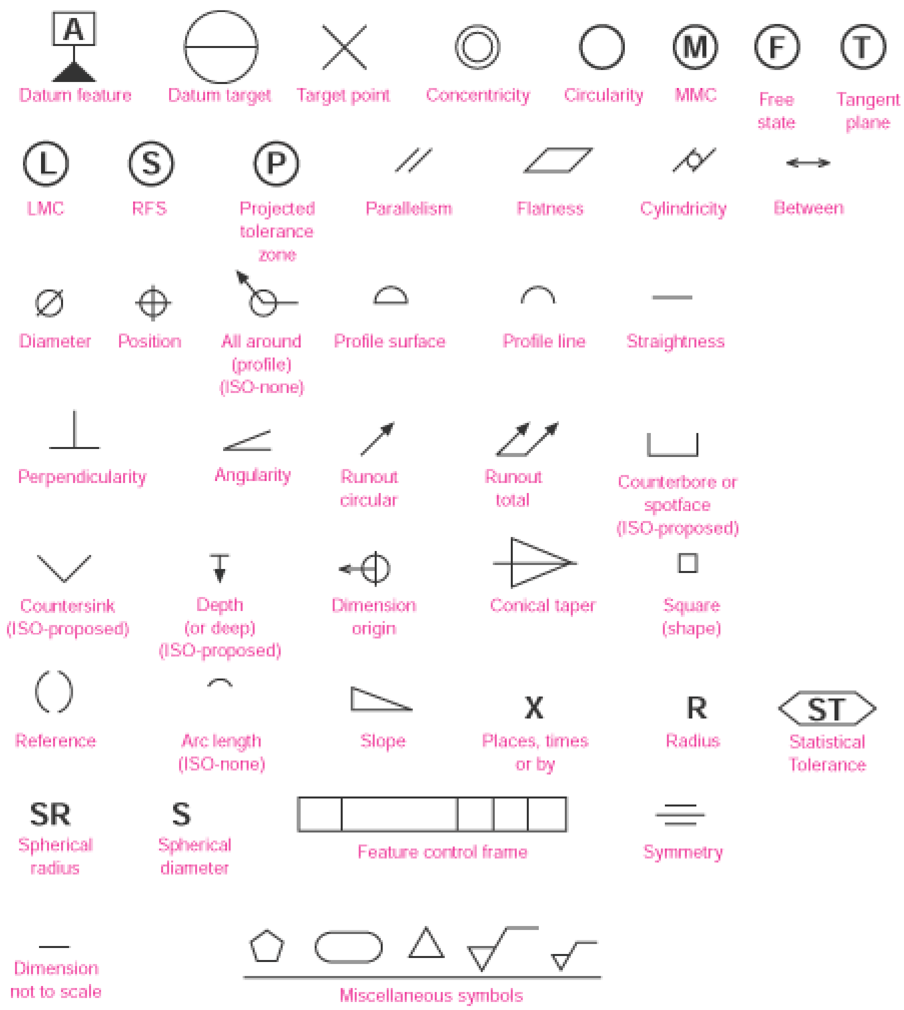
Geometric Dimensioning and Tolerancing

- GDT is a method of defining parts based on how they function, using standard ASME/ANSI symbols.
- Within the last 15 years there has been considerable interest in GDT, in part because of the increased popularity of **statistical process control**. This control

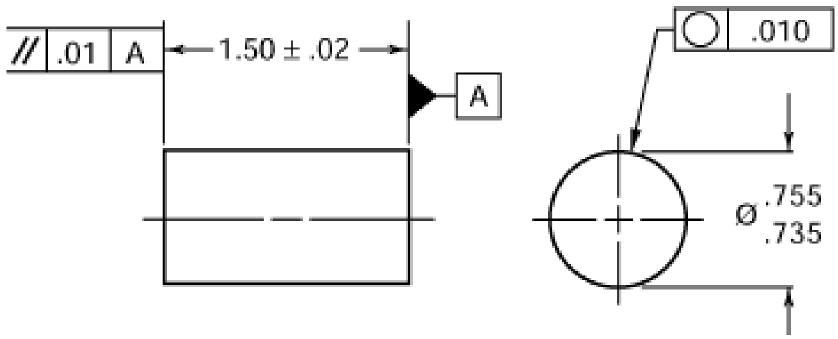
process, when combined with GDT, helps reduce or eliminate inspection of features on the manufactured object. The flipside is

that the part must be tolerance very efficiently; this is where GDT comes in. Another reason for the increased popularity of GDT is the rise of worldwide standards, such as ISO 9000, which require universally understood and accepted methods of documentation.

GDT-Symbols



 Feature control frames



 MMC/LMC

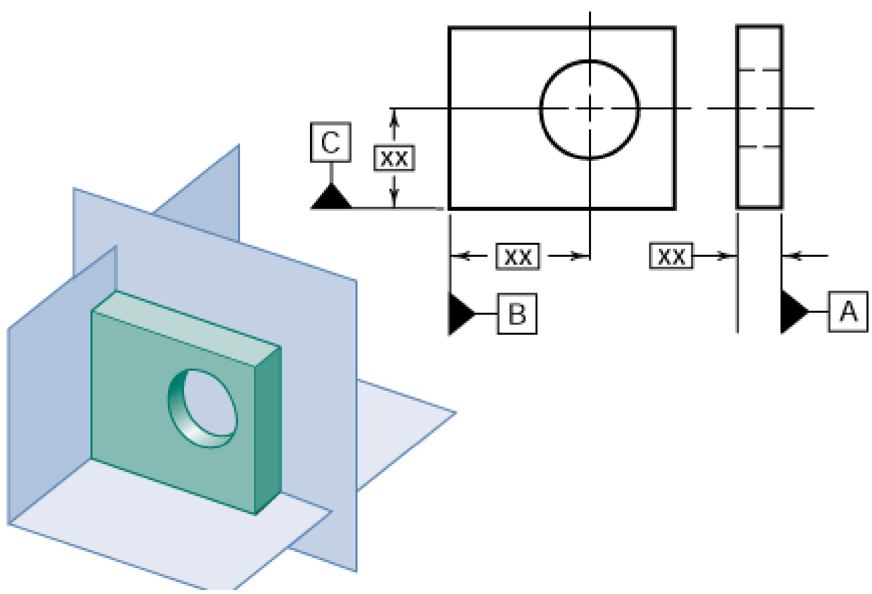
 Datums

 Geometric Controls

 Form

 Orientation

 Position

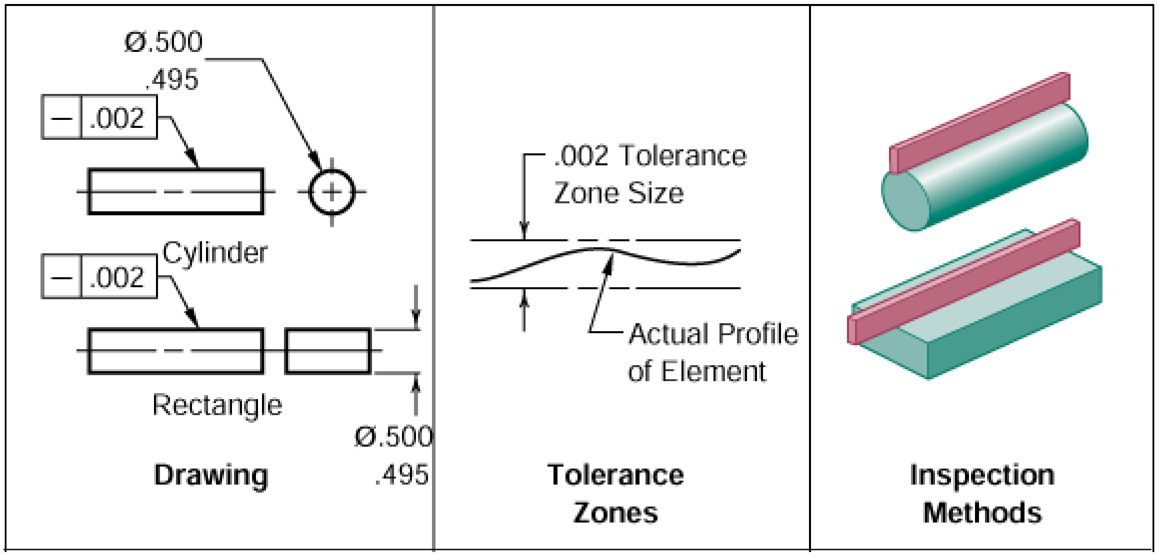


 Forms

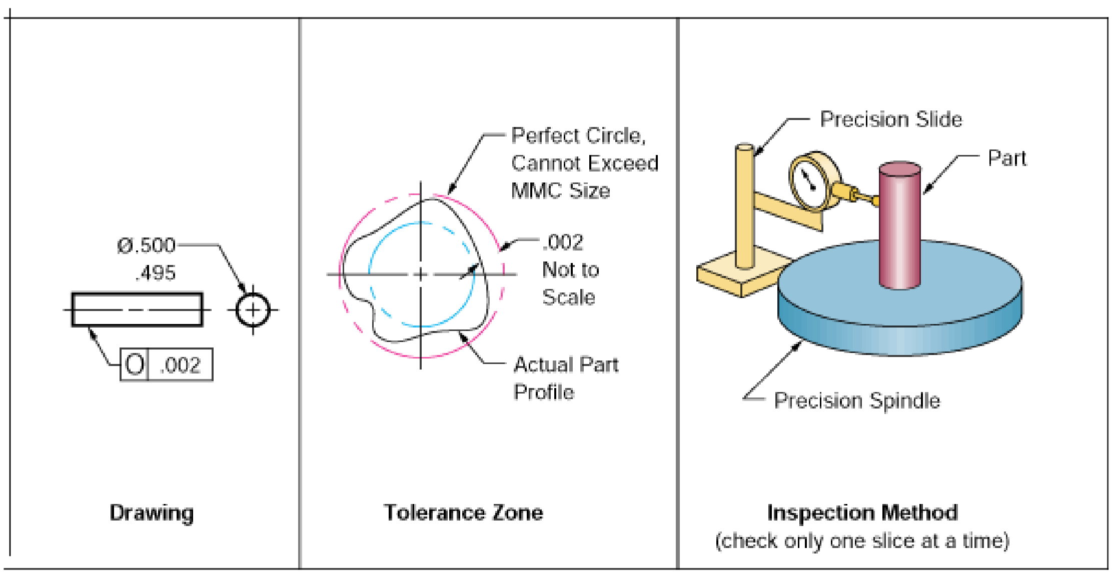
 Straightness

• Line element

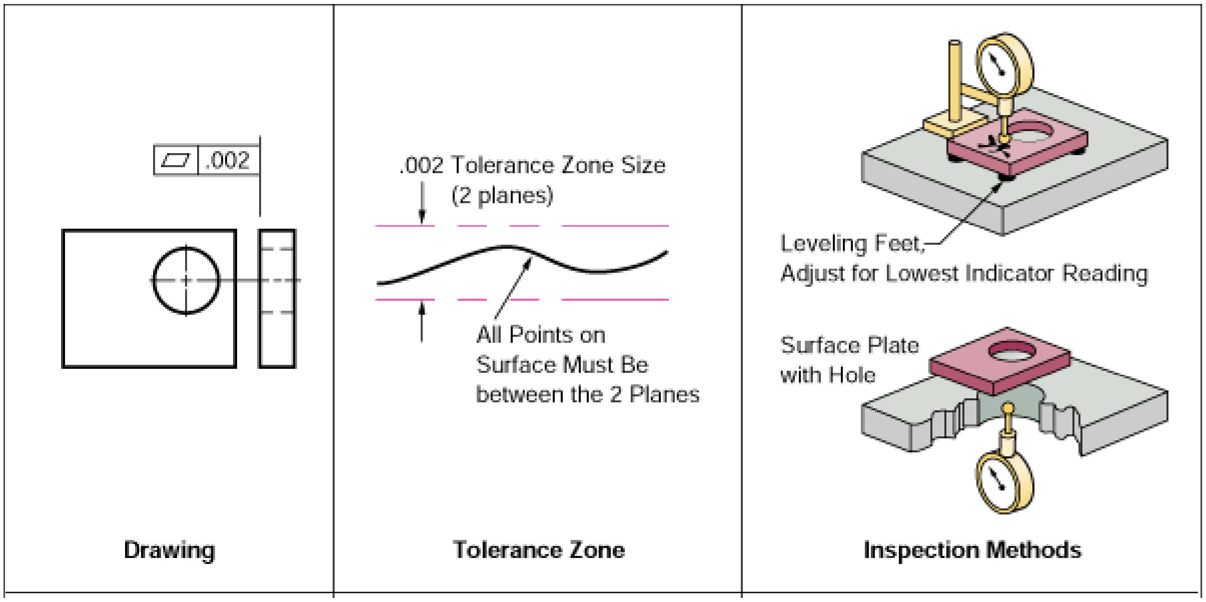
• Axis



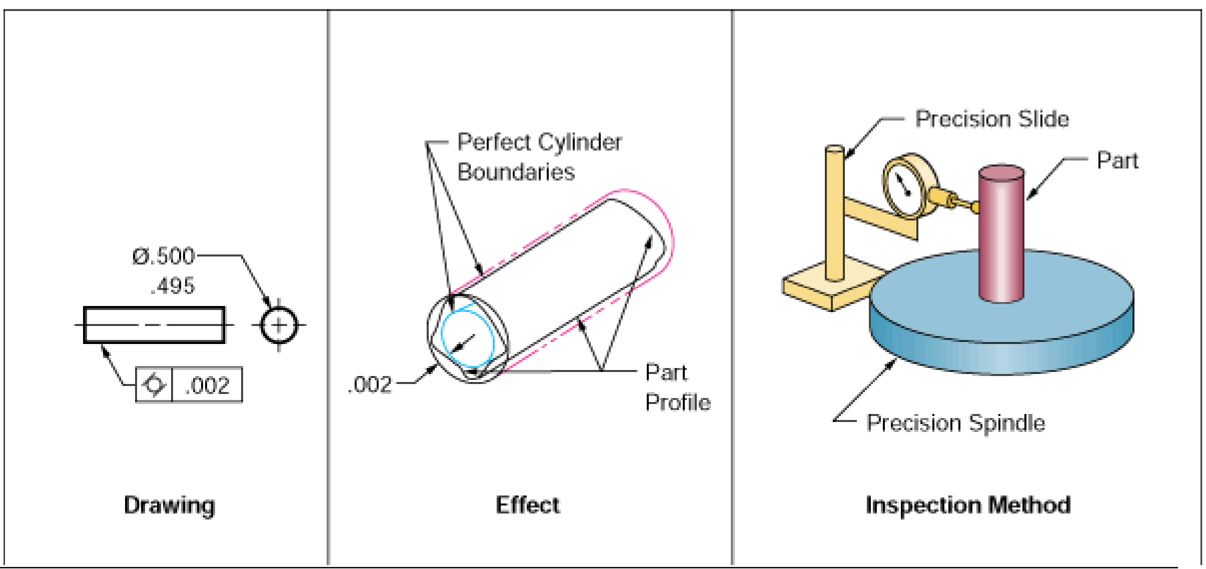
- **Forms**
  - Circularity



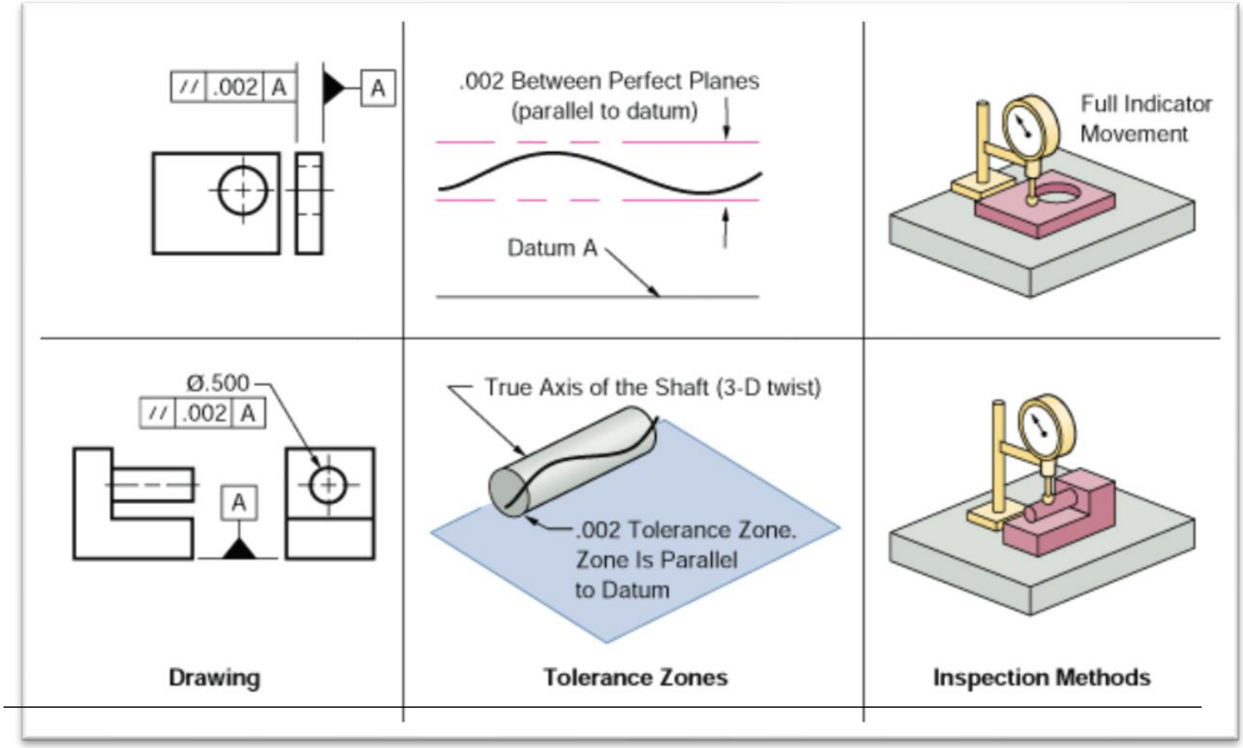
- **Forms**
  - Flatness



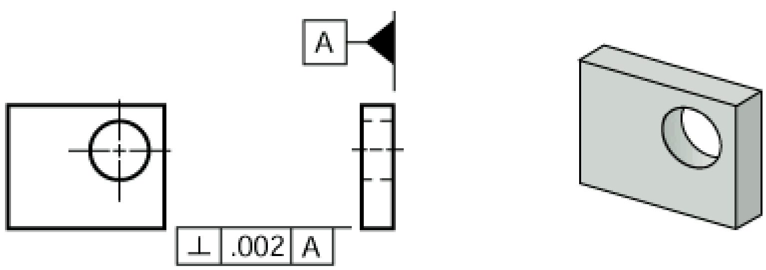
- **Forms**
  - Cylindricity

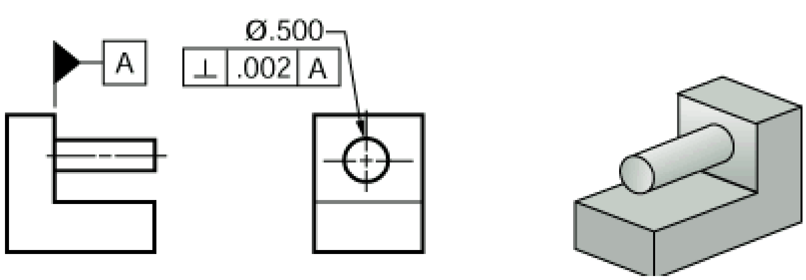


- **Orientation**
  - Parallelism



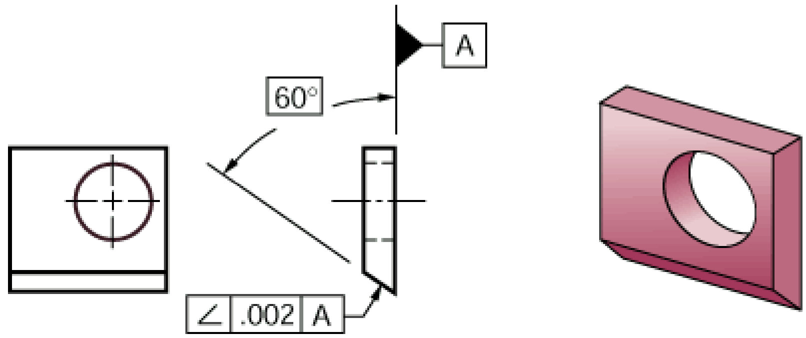
- **Orientation**
    - Perpendicularity

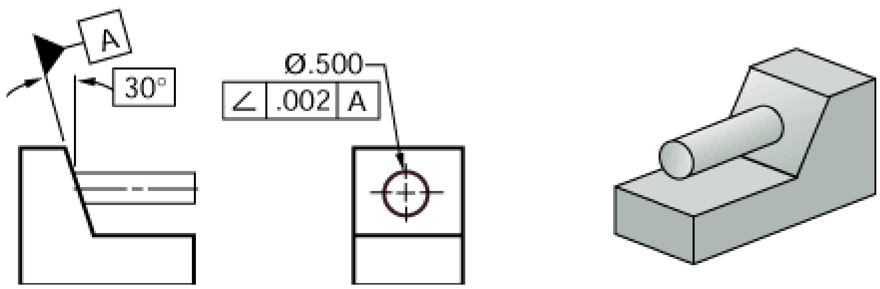




Orientation

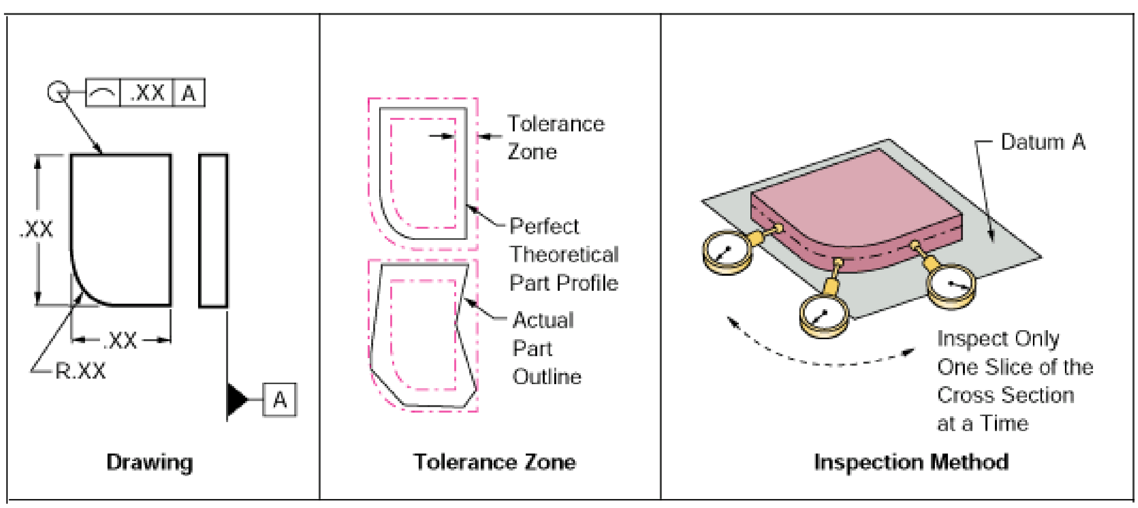
- Angularity





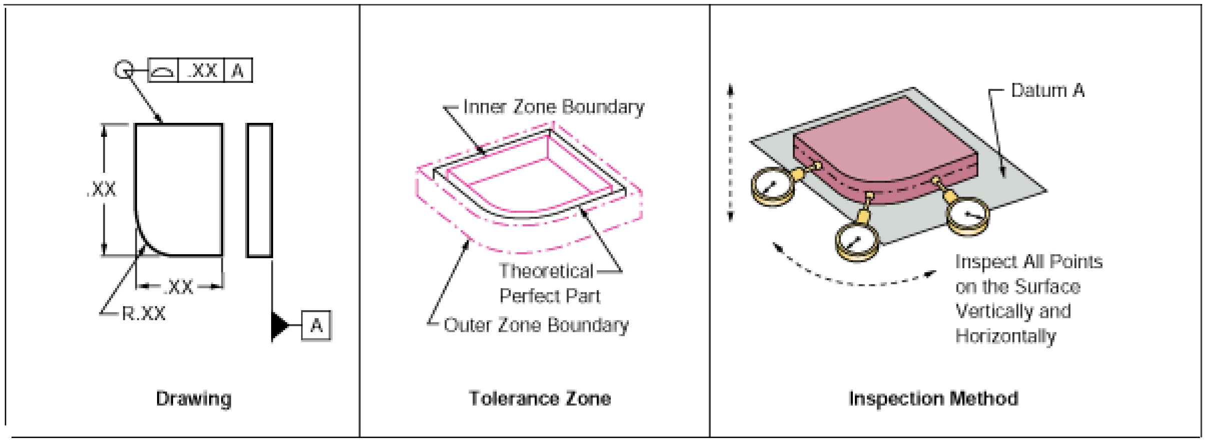
Orientation

- Line profile

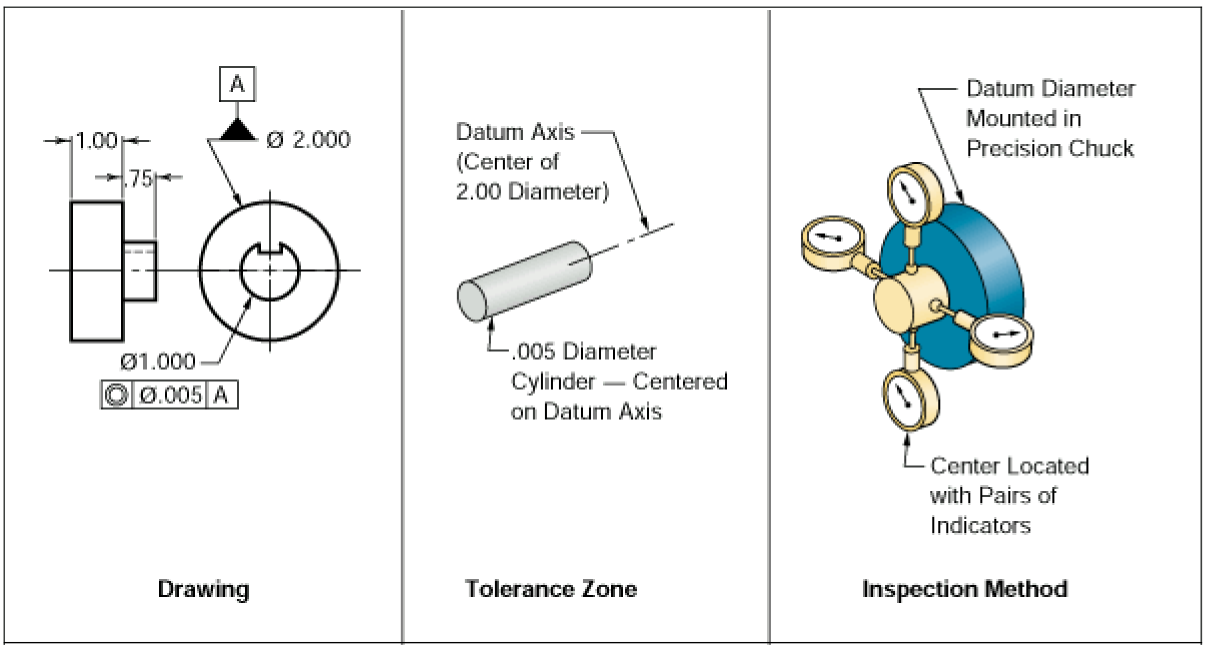


Orientation

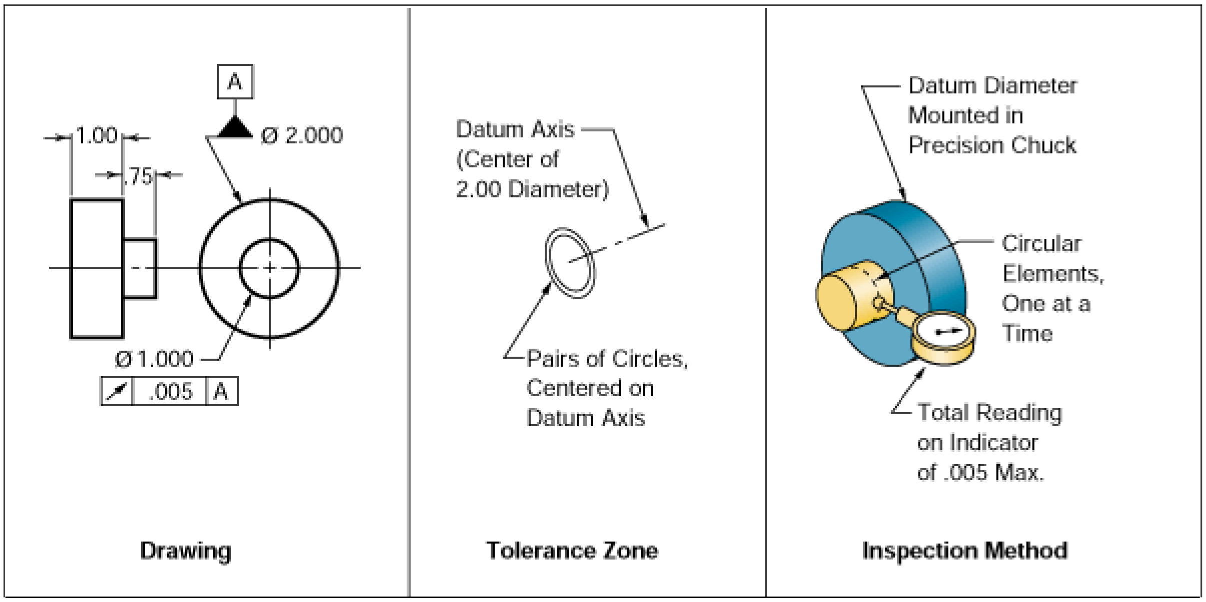
- Surface profile



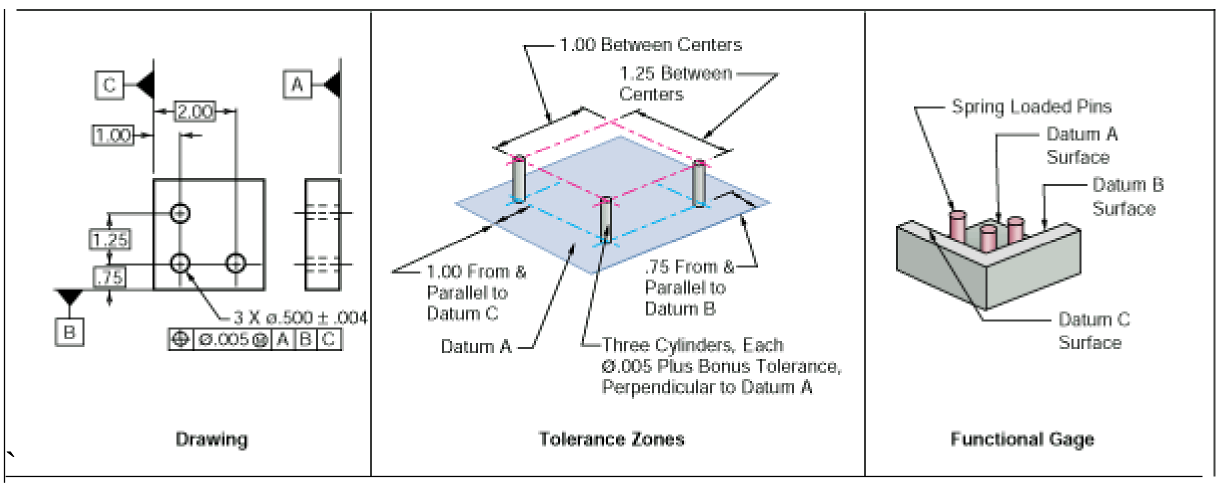
- **Location**
  - Concentricity

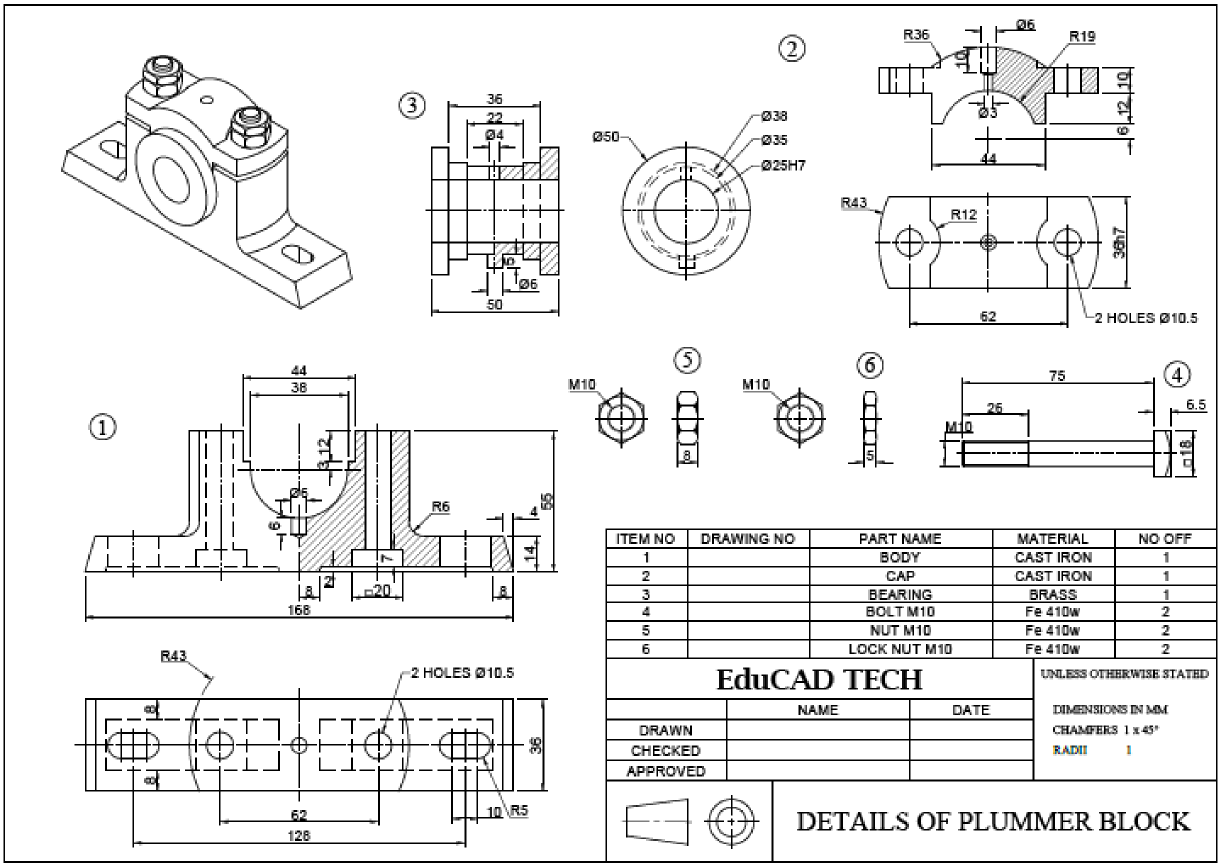


- **Location**
  - Runout

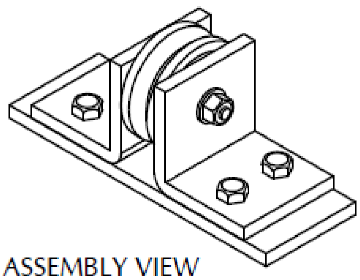


- **Location**
  - Position

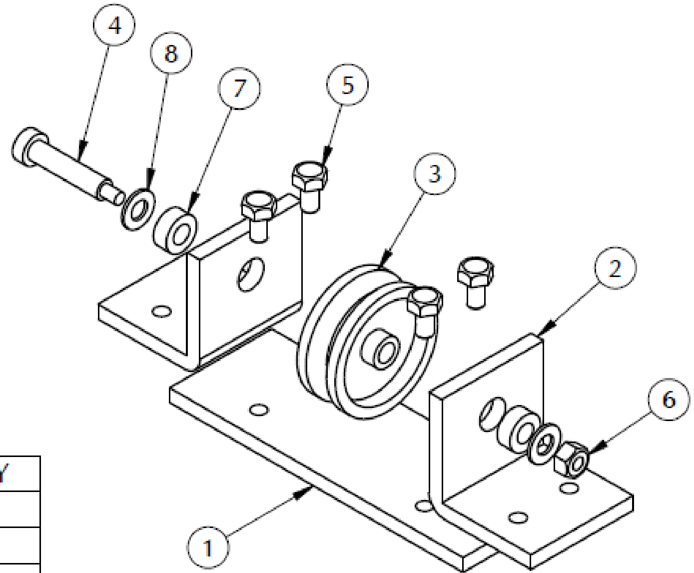



ITEM NO
DRAWING NO
PART NAME
MATERIAL
NO OFF
1 BODY CAST IRON 1
2 CAP CAST IRON 1
3 BEARING BRASS 1
4 BOLT M10 Fe 410w 2
5 NUT M10 Fe 410w 2
6 LOCK NUT M10 Fe 410w 2
EduCAD TECH
UNLESS OTHERWISE STATED
DIMENSIONS IN MM
CHAMFERS 1 x 45°
RADII 1
NAME
DATE
DRAWN
CHECKED
APPROVED
DETAILS OF PLUMMER BLOCK
2 HOLES Ø10.5
R43
R36
R19
R12
R6
R5
Ø6
Ø50
Ø38
Ø35
Ø25H7
Ø4
Ø3
M10
168
128
75
62
55
50
44
38
36
26
22
20
18
14
12
10
8
7
6.5
6
5
4
2

WHEEL SUPPORT ASSEMBLY



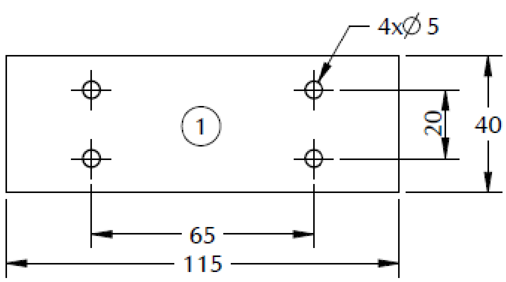
ASSEMBLY VIEW

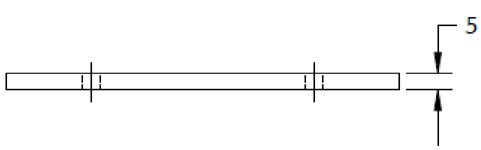


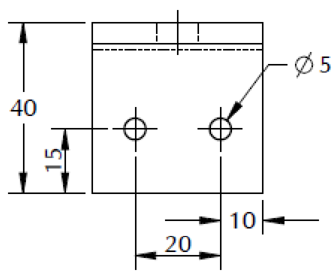
EXPLODED VIEW

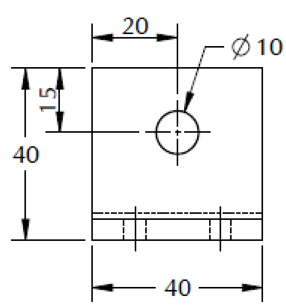
ITEM.NO	PART NAME	QTY
1	BASE	1
2	SUPPORT	2
3	WHEEL	1
4	SHOULDER SCREW	1
5	BOLT	4
6	NUT	1
7	BUSHING	2
8	WASHER	2

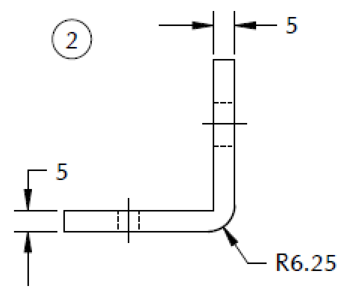
DETAILS OF WHEEL SUPPORT ASSEMBLY

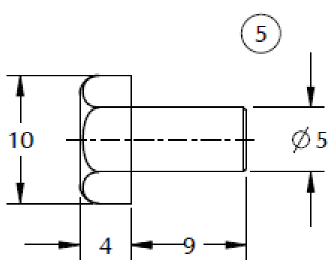




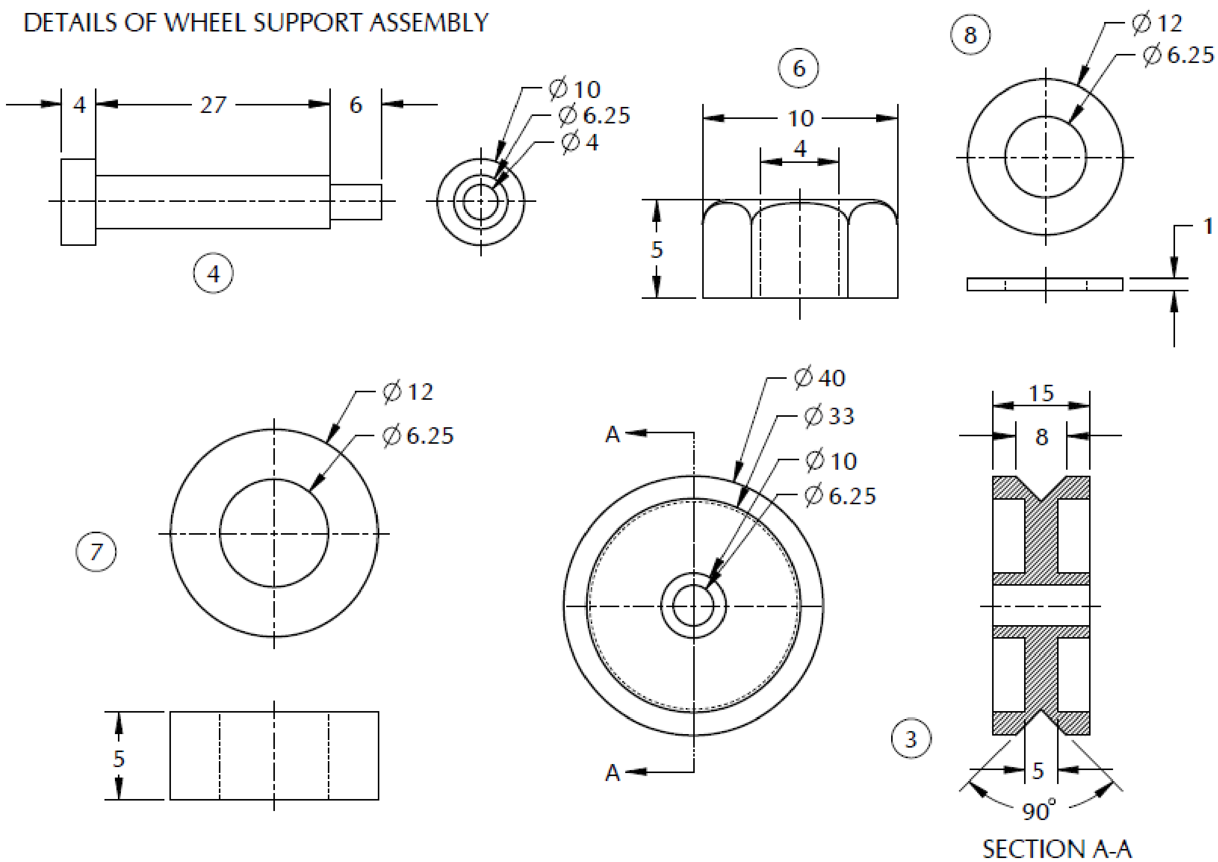




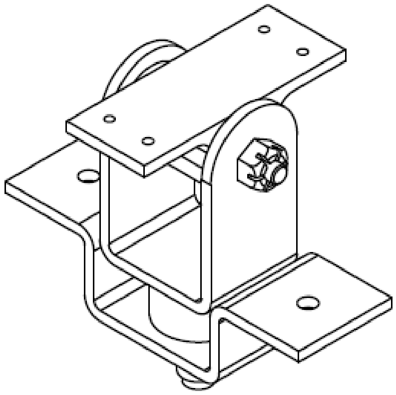




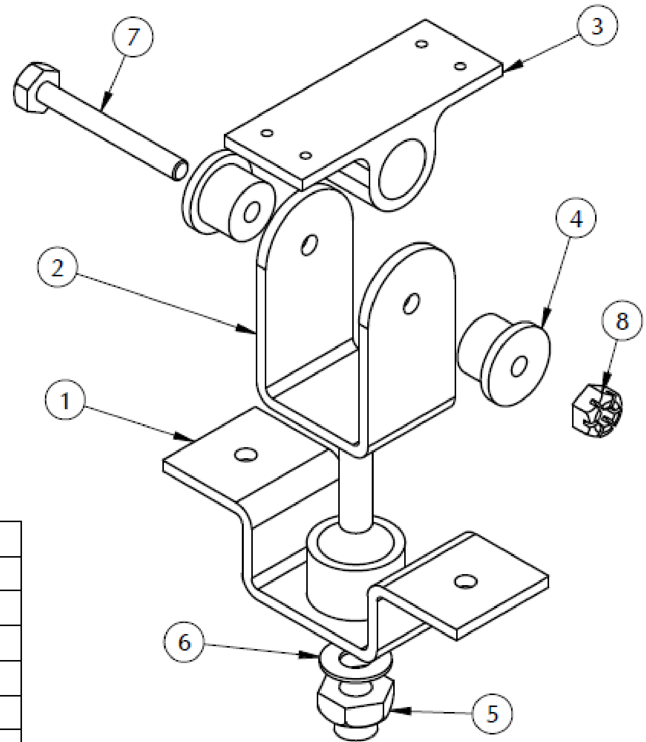
DETAILS OF WHEEL SUPPORT ASSEMBLY



SHOCK ASSEMBLY



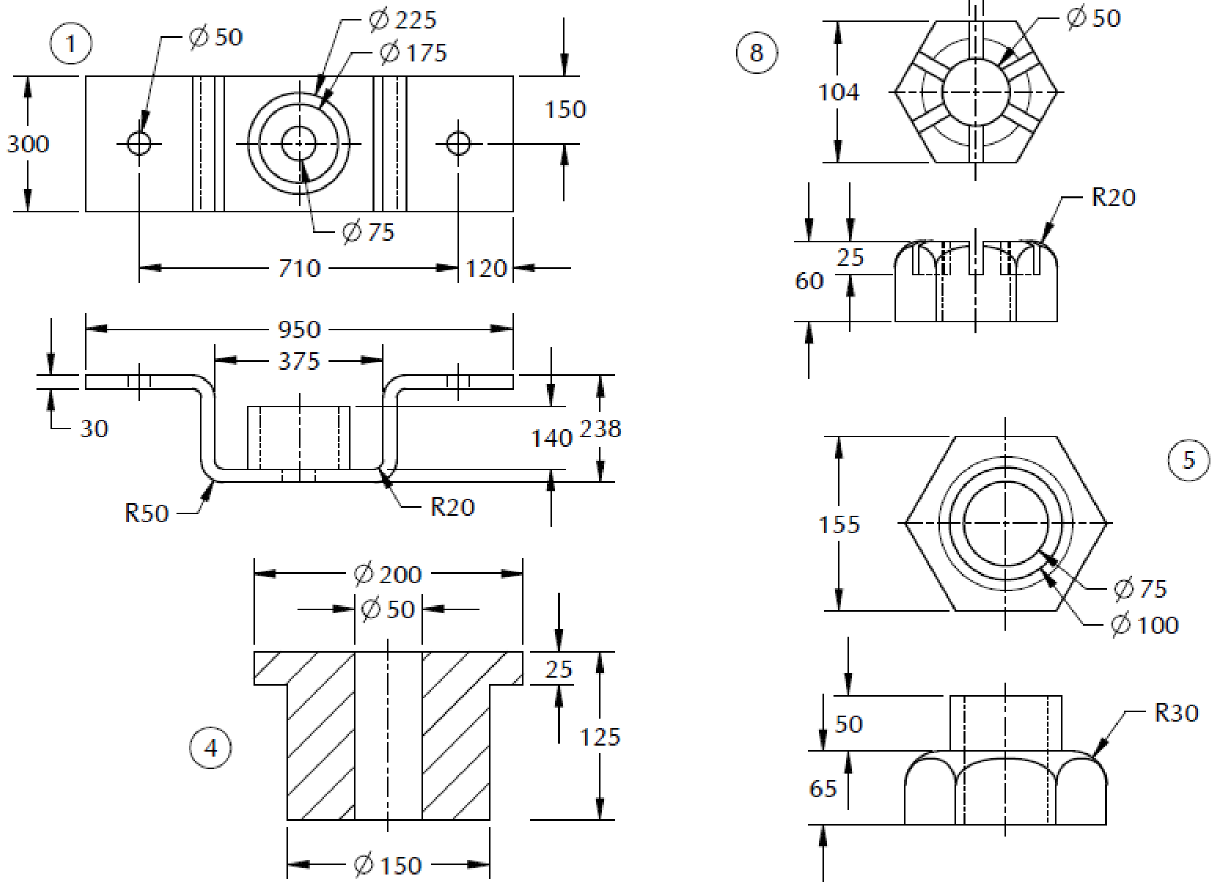
ASSEMBLY VIEW



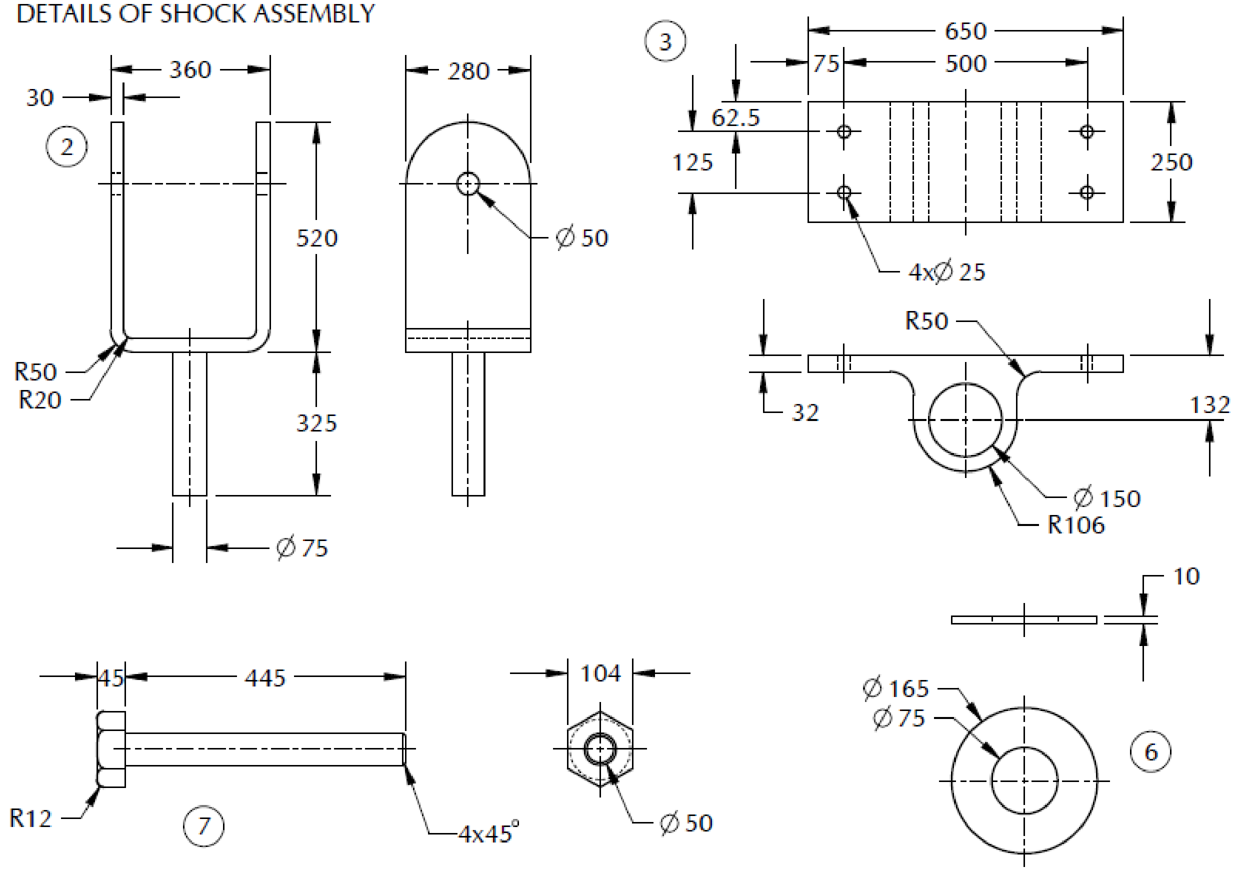
EXPLODED VIEW

ITEM NO	PART NAME	QTY
1	BRACKET	1
2	U-SUPPORT	1
3	PIVOT	1
4	BUSHING	2
5	SELF LOCKING NUT	1
6	WASHER	1
7	HEXAGONAL BOLT	1
8	CASTLE NUT	1

DETAILS OF SHOCK ASSEMBLY



DETAILS OF SHOCK ASSEMBLY



THANK YOU….