

BLUEPRINT READING 131

Class Vocabulary

Term	Definition
American Society Of Mechanical Engineers	ASME. An organization that publishes technical materials and sets industrial and manufacturing standards. Along with International Organization for Standardization, ASME provides written standardization for blueprints.
Arc	A curved feature representing a portion of a circle. Arcs represent a curve or portion of the circumference of a circle.
Auxiliary View	A blueprint view drawn at a right angle to an angled feature of a part. Auxiliary views provide a full straight-on picture of an angled side that cannot be fully represented with a basic orthographic view.
Blueprint	A document that contains all the instructions for a particular part and communicates all requirements necessary to manufacture and inspect a quality part. There are three elements: the views, their dimensions, notes.
Break Line	A line used to define the boundary of an imaginary broken-out section or to shorten dimensions that are excessively long. Break lines are wavy and irregular.
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Broken-Out Section	A section view resulting from an imagined cut that removes a small portion of the part exterior to reveal the features beneath. Broken-out sections are used to highlight small details.
Broken-Out Section View	A section view that removes a small portion of the part exterior to reveal the features beneath. Broken-out section views are used to highlight small details.
Center Line	A line used to define the midpoint of a cylindrical or symmetrical feature or part. Center lines consist of alternating long and short dashes.
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Change Block	The area of the blueprint indicating any changes or revisions made to the part views or dimensions. Change blocks should contain every revision, as well as the dates in which those revisions were made.

Crosshatch	Sets of parallel lines that intersect to form a pattern. Cross hatch patterns are often used in blueprint design.
Cutting Plane Line	A line in a blueprint that represents the path and position of an imaginary cut made to form a sectional view. Cutting planes are made up of a long dash, followed by two short dashes that are slightly thicker than a phantom line.
Deburring	Removing sharp projections left on a workpiece after a machining or grinding operation. Deburring is often done by hand or by robot using coated abrasives.
Decimals	A number representing a portion of a whole. Decimals are indicated by values to the right of a decimal point.
Dimension Line	A line used to define the measurement of a part feature. Dimension lines consist of a solid line with arrows at both ends.
Dimensioning	The process of determining the desired measurement of a feature on a part. The ideal dimensions for a part can never be met, requiring the addition of tolerances.
Dimensions	The desired measurement of a feature on a part. A dimension is listed as a unit.
Drawings	A document that contains detailed instructions to make a part. Drawings, or prints, contain dimensions, directions, and details to guide employees in metal cutting operations to machine a part and its features.
Extension Lines	A line used to visually connect the ends of a dimension line to the relevant feature on a part. Extension lines are solid and drawn perpendicular to the dimension line.
First-Angle Projection	The European standard for engineering drawings. In first-angle projection, the top view is under the front view.
Fractions	A numerical expression representing a part of a larger whole. Fractions appear as one number over the other with a horizontal line or slash between them.
Full Section	A section view in which the part is cut entirely in half. The right side is removed and presented cut-side up.
Full Section View	A section view in which the part is cut entirely in half. On full section views, the right side is removed and the left side is shown cut-side up.
Functionally Complex	Able to correctly and efficiently perform a specific task. The functional complexity of blueprint drawings allows for the creation of parts that can perform tasks immediately after being built.
Geometric Dimensioning And Tolerancing	GD&T. An international standard for communicating instructions about the design and manufacturing of parts. Geometric dimensioning & tolerancing uses universal symbols and emphasizes the function of the part.
Half Section	A section view in which an imaginary cut is made through the part and half of that cut is presented cut-side up. Symmetrical parts often contain half sections.

Half Section View	A section view in which one quarter of a part is shown. Half section views are often used for symmetrical parts.
Head Depth	The measurement of the flat top of a fastener from top to bottom. Head depth is an important measurement in determining the dimensions of a T-slot.
Head Width	The measurement of the distance across the flat top of a fastener. Head width is an important measurement in determining the dimensions of a T-slot.
Hidden Line	A line used to define a part feature that is not visible in a specific view. Hidden lines consist of a series of short dashes.
Inspection	The examination of a part during or after its creation to confirm that it adheres to specifications. During inspection, defects may be identified and corrected.
International Organization For Standardization	ISO. An organization that establishes documented standards, rules, and guidelines to ensure that products, processes, and services are fit for their purpose. Along with American Society of Mechanical Engineers, ISO provides written standardization for blueprints.
Isometric View	A drawing that communicates the shape and size of an object in three dimensions by showing three sides of the object from one perspective. Isometric views are not frequently utilized in blueprints.
Leader Line	A thin line ending in an arrow or dot used to indicate a part feature. Leader lines may be used when there are space limitations in a view.
Notes	An additional instruction or general comment added to a blueprint. Notes contain information about the material, finish, tooling, tolerances, and other miscellaneous information.
Object Line	A line used to define the shape and size of a part feature and represent the visible edges of the part. Object lines are solid.
Oblique	At a slanted angle. Oblique angles are not parallel or perpendicular to a surface or line.
Offset Section	A section view resulting from an imagined cut resulting from a line that has segments which turn at 90 degrees, with both end segments going in the same direction. Offset sections allows features to be shown that don't occur in a straight line.
Offset Section View	A section view resulting from an imagined cut that does not follow a straight line. Offset section views are used to include features that do not reside on a straight line.
Orthographic View	A drawing often used for blueprints that communicates the shape and size of an object in two dimensions. Orthographic views present all the design elements of a part through a series of illustrations, each showing one of its necessary sides from a continuous perspective.

Orthographic Views	A drawing often used for blueprints that communicates the shape and size of an object in two dimensions. Orthographic views allow for a clear image of all the design elements of a part through a series of illustrations each showing one of its necessary sides from a continuous perspective.
Parallel	Running equidistant from each other at all points. Parallel means never intersecting.
Perpendicular	Forming a 90° angle with another plane or object. Perpendicular lines form corners like those found on a piece of paper.
Phantom Lines	A line that indicates the alternate positions of a movable feature, repeated details, or material that will be removed during a stage of the manufacturing process. A phantom line consists of alternating long dashes with two short dashes in between.
Phantom Lines	A line that indicates the alternative positions of a feature that moves, repeated details, or material that will be machined from the part at some stage of the manufacturing process in a blueprint. A phantom line is drawn by alternating a long dash, followed by two short dashes that are thinner than a cutting plane line.
Prints	The examination of a part during or after its creation to confirm that it adheres to specifications. Inspection allows manufacturers to identify and correct product defects.
Ratio	A numerical expression representing a part of a larger whole or proportion. A ratio consists of two numbers separated by a colon.
Reference Dimensions	A dimension that is provided for informational purposes only. Reference dimensions appear in parentheses or are marked with REF.
Removed Section	A section view resulting from an imagined cut that removes and rotates a slice taken from the middle of a part feature. Removed sections are sometimes used to show an enlarged area of a part too small to be illustrated clearly on the original view.
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Revolved Section	A section view resulting from an imagined cut that separates the middle of the part feature and rotates a slice taken from that cut in place. Revolved sections have section lines and show the true shape of a part.
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Scale	The relationship between the size of the drawing on the print and the actual finished part size. The scale for a part drawing is usually listed in the title block.

Section Line	A line used to identify the imaginary cut portion of a part in a section view. Section lines appear as a series of diagonal lines drawn close together at a 45-degree angle.
Section View	A view illustrating a rotated section resulting from an imaginary cut in the part. Blueprints may contain a variety of different section views.
Sum	The total amount resulting from adding two or more numbers together. Sums are useful when showing the total of other dimensions given on a print.
Surface Finishes	The degree of smoothness of a part's exterior after manufacturing. Surface-finish quality will usually be specified on a manufacturing print.
Third-Angle Projection	The American standard for engineering drawings. In third-angle projection, the front view is under the top view.
Three-Dimensional	Having or appearing to have length, breadth, and depth. Three-dimensional drawings show an object's shape better than flat drawings.
Throat Depth	The measurement determined by the thickness of a material and the dimensions allowable for simple machines like a T-slot bolt. Throat depth is an important measurement in determining the dimensions of a T-slot.
Throat Width	The measurement of the threaded neck of a T-slot bolt from side to side. Throat width is an important measurement in determining the dimensions of a T-slot.
Title Block	The area of a blueprint containing information such as company name, part name, part number, designer, scale, and material. Title blocks are unique to each manufacturer.
Tolerances	An acceptable variation or deviation from a given dimension or geometry. Tolerances indicate the allowable difference between a physical feature and its intended design.
T-Slot Nut	A specially-shaped fastener typically used with a threaded clamp to secure a position. T-slot nuts enable engineers to place fixture components anywhere along the component where the T-slot runs.
Two-Dimensional	2D. Having a length and width, but not depth. Flat shapes are two-dimensional.
Typical Dimensions	A dimension that is assumed to be the same for similar features of a part. Typical dimensions are followed by an X or the word PLACES.
Views	All the lines that illustrate the shape of the part. A blueprint often contains multiple views to convey all of a part's design elements.
Windings	Wire wrapped around a core or into a coil that is used to conduct current. Windings form electromagnets and magnetic fields in motors and other devices.
Workholding	A method or device for securing a workpiece for a machining operation. Workholding can include chucks, vises, and bolts.

