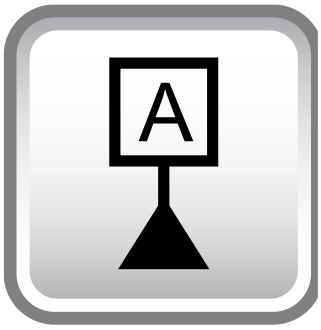
True Position - Position Tolerance

Position is one of the most useful and most complex of all the symbols in GD&T. The two methods of using Position discussed on this page will be RFS or Regardless of Feature Size and under a material condition (Maximum Material Condition or Least Material Condition). Position is always used with a feature of size.

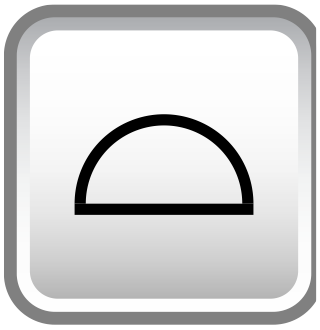
Learn Symbol →



Datum Feature

A datum is theoretical exact plane, axis or point location that GD&T or dimensional tolerances are referenced to. You can think of them as an anchor for the entire part; where the other features are referenced from. A datum feature is usually an important functional feature that needs to be controlled during measurement as well.

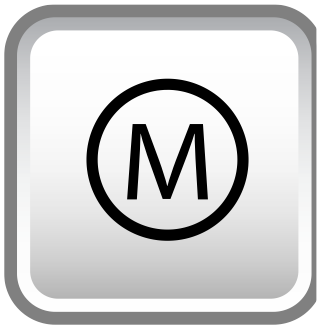
Learn Symbol →



Profile of a Surface

Profile of a surface describes a 3-Dimensional tolerance zone around a surface, usually which is an advanced curve or shape...

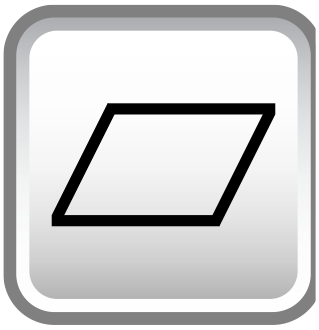
Learn Symbol →



Maximum Material Condition (MMC)

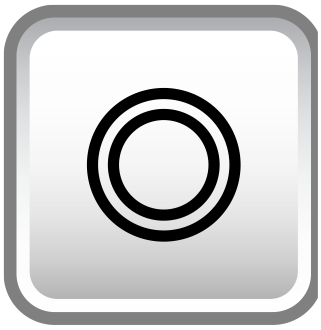
Maximum Material Condition (MMC), is a feature of size symbol that describes the condition of a feature or part where the maximum amount of material (volume/size) exists within its dimensional tolerance.

Learn Symbol →



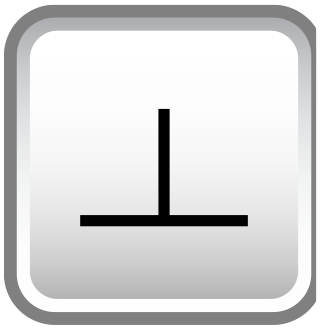
Flatness

GD&T Flatness is a common symbol that references how flat a



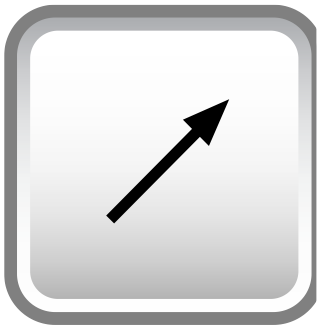
Concentricity

Concentricity, is a tolerance that controls the central



Perpendicularit

Perpendicularity is a fairly common symbol that requires the



Runout

Runout is how much one given reference feature or features

surface is regardless of any other datum's or features. It comes in useful if a feature is to be defined on a drawing that needs to be uniformly flat without tightening any other dimensions on the drawing. The flatness tolerance references two parallel planes (parallel to the surface that it is called out on) that define a zone where the entire reference surface must lie.

Learn Symbol →

derived median points of the referenced feature, to a datum axis. Concentricity is a very complex feature because it relies on measurements from derived median points as opposed to a surface or feature's axis.

Learn Symbol →

referenced surface or line to be perpendicular or 90° from a datum surface or line...

Learn Symbol →

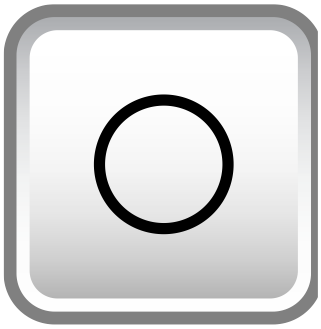
vary with respect to another datum when the part is rotated 360° around the datum axis.

Learn Symbol →

Parallelism

Parallelism is a fairly common symbol that describes a parallel orientation of one referenced feature to a datum surface or line...

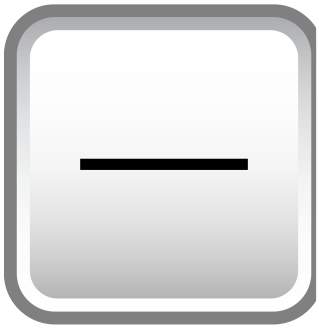
Learn Symbol →



Circularity

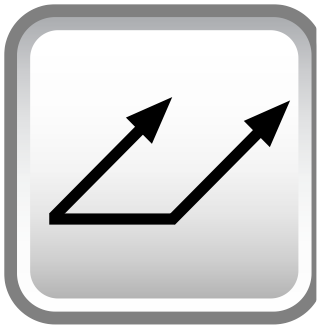
The circularity symbol is used to describe how close an object should be to a true circle...

Learn Symbol →



Straightness

The standard form of straightness is a 2-Dimensional tolerance that is used to ensure that a part is uniform across a surface or feature. Straightness can apply to either a flat feature such as the surface of a block, or it can apply to the surface of a cylinder along the axial direction. It is defined



Total Runout

Total Runout is how much one entire feature or surface varies with respect to a datum when the part is rotated 360° around the datum axis...

Learn Symbol →

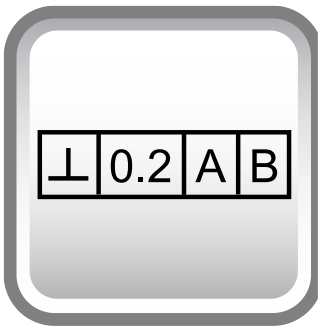
as the variance of the surface within a specified line on that surface.

Learn Symbol →

Cylindricity

The Cylindricity symbol is used to describe how close an object conforms to a true cylinder...

Learn Symbol →



Feature Control Frame

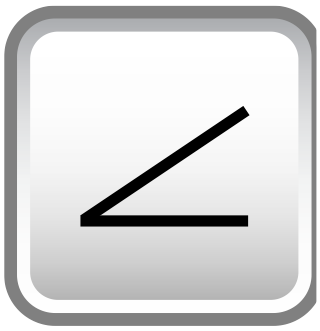
In GD&T, a feature control frame is required to describe the conditions and tolerances of a geometric control on a part's feature...

Learn Symbol →

Symmetry

GD&T Symmetry is a 3-Dimensional tolerance that is used to ensure that two features on a part are uniform across a datum plane...

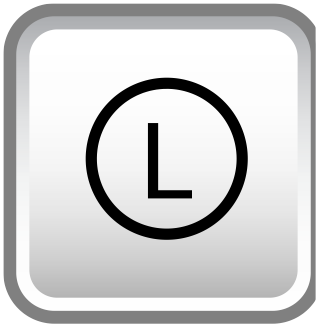
Learn Symbol →



Angularity

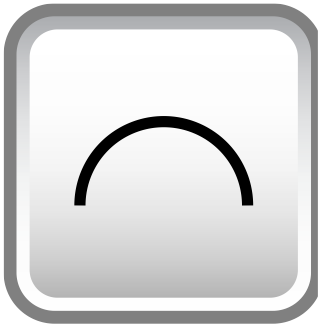
Angularity is the symbol that describes the specific orientation of one feature to another at a referenced angle...

Learn Symbol →



Least Material Condition (LMC)

Least material condition is a feature of size symbol that describes a dimensional or size condition where the



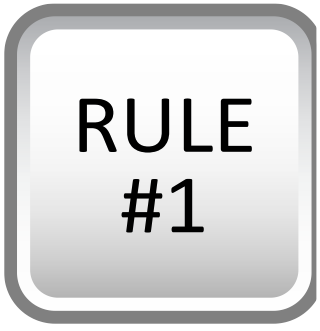
Profile of a Line

Profile of a line describes a tolerance zone around any line in any feature, usually of a curved shape...

Learn Symbol →

Regardless of Feature Size

Regardless of feature size simply means that whatever GD&T callout you make, is controlled independently of the size dimension of the part. RFS is the



GD&T Rule #1: Envelope Principle

GD&T Rule #1, also known as the Envelope principle, states that the form of a regular feature of size is controlled by

least amount of material (volume/size) exists within its dimensional tolerance...

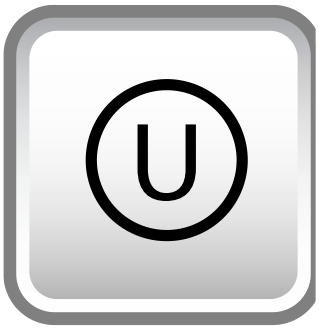
Learn Symbol →

default condition of all geometric tolerances by rule #2 of GD&T and requires no callout.

Learn Symbol →

its “limits of size." Limits of size, or otherwise known as size tolerances, can be seen in many forms. A few of them are symmetric, unilateral, and bilateral.

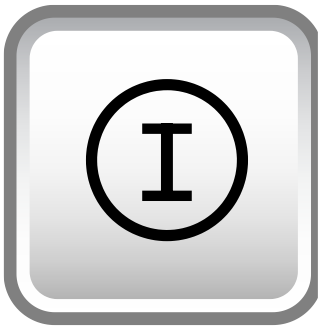
Learn Symbol →



Unequally Disposed Profile

The unequally disposed profile tolerance symbol is used to apply unilateral or unequal tolerance zones to a profile of a part.

Learn Symbol →



Independency

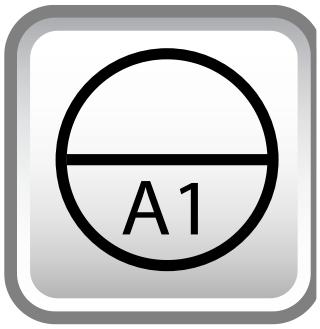
The Independency Symbol is used on drawings to declare that the requirement for perfect form at MMC or LMC is removed and the form tolerance may be larger than the size tolerance. This symbol only exists in the ASME Y14.5 standards, not the ISO GPS standards.

Learn Symbol →

Envelope Requirement (E Symbol) - ISO Only

The Envelope Requirement Symbol is used on ISO drawings to declare that size is to control form with respect to perfect form at MMC.

Learn Symbol →



Datum Target

The Datum Target Symbol is used to define a specific point, line, or area to be used in order to establish a datum on a drawing.

Learn Symbol →

Continuous Feature

The Continuous Feature Symbol is used to indicate that a group of disjointed features or surfaces are to be considered as one continuous feature.

Learn Symbol →

Projected Tolerance Zone

The Projected Tolerance Symbol is in used GD&T to indicate the tolerance zone of a feature is to be assessed beyond the surface extents of the feature.

Learn Symbol →

Free State Symbol

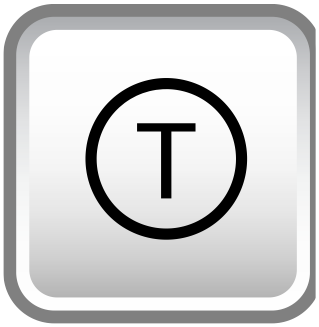
The Free State Symbol is in used GD&T to indicate that a specific dimension and its associated tolerance are to be assessed in an unrestrained state.

Learn Symbol →

Restrained Condition Note

Some parts, such as those made of rubber or sheet metal, may fail inspection due to deformation from gravity if left in their "free state." This can be prevented by invoking a "restrained state."

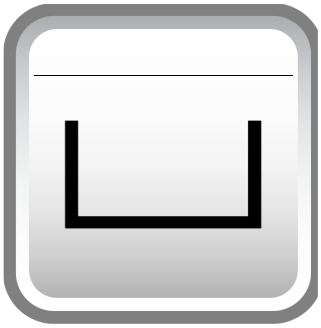
Learn Symbol →



Tangent Plane

The Tangent Plane Symbol is used in conjunction with GD&T surface controls to indicate that a specific control is applied to a theoretical tangent plane simulated by the high points of irregular tangible surface rather than the actual elements of the surface themselves.

Learn Symbol →



Counterbore

The counterbore symbol is used on a drawing to indicate that a counterbore hole feature is required. A counterbore is a flat-bottomed cylindrical hole that is larger than and coaxial to another cylindrical hole.

Learn Symbol →

Spotface

A spotface is a very shallow flat-bottomed cylindrical hole that is larger than and coaxial to another cylindrical hole. Its purpose is to provide a flat mounting surface for mating parts, such as washers or pan head screws.

Learn Symbol →

Countersink

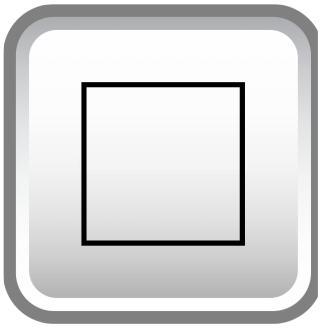
A countersink consists of a conical hole that is coaxial to a cylindrical hole, where the angle of the cone is determined by the fastener to be used. The purpose of a countersink is to allow a fastener, typically a flathead screw, to sit slightly below the surface of the part.

Learn Symbol →

Diameter

The diameter symbol is used to indicate that the size of a circular feature is being dimensioned using the diameter of that feature.

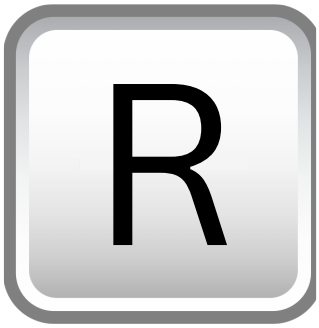
Learn Symbol →



Square

The Square symbol is used to indicate a square feature on a drawing.

Learn Symbol →



Radius

The Radius symbol is used when a circular feature is dimensioned using the radius length.

Learn Symbol →

Controlled Radius

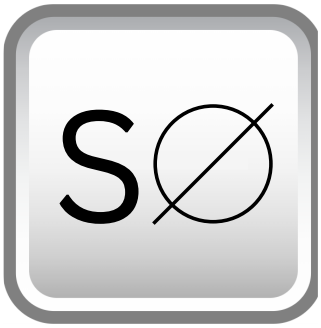
The Controlled Radius symbol is a variation of the Radius, and is used when the surface of the contour is required to be a "fair" curve.

Learn Symbol →

Spherical Radius

The Spherical Radius is used to indicate that the Radius is of a spherical, rather than circular, feature.

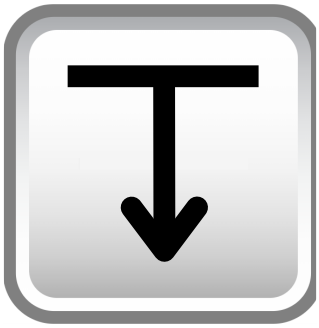
Learn Symbol →



Spherical Diameter

The Spherical Diameter symbol is used to indicate that the Diameter is of a spherical, rather than circular, feature.

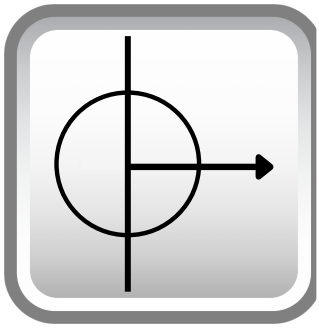
Learn Symbol →



Depth

The depth symbol is used to indicate a measurement from the bottom of a feature to the outer surface of a part.

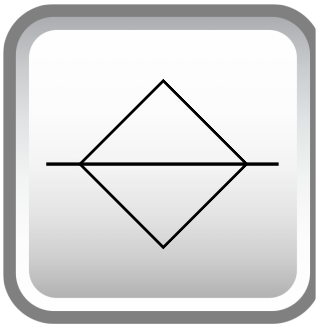
Learn Symbol →



Dimension Origin

The dimension origin symbol is used to indicate where a dimension must be measured from. This symbol is used in place of a dimensional arrow leader to indicate the origin for measurement.

Learn Symbol →



Parting Line

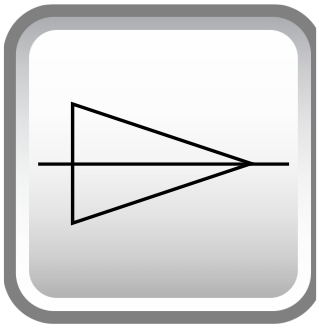
A parting line is the location where separate parts of a mold or die come together. If the parting line is not dimensioned on the drawing, the location of the parting line is up to the manufacturer's discretion.

Learn Symbol →

Arc Length

The Arc Length symbol, also known as the Arc Modifying symbol, is placed above a value to indicate that the arc's length is being dimensioned on a curved outline.

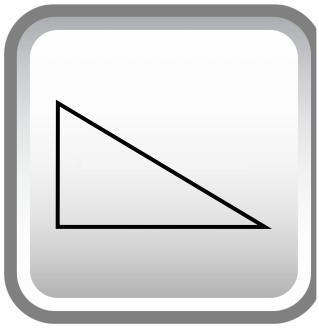
Learn Symbol →



Conical Taper

The conical taper symbol is placed with a dimension value to indicate that the value is controlled as a standard taper. The value listed with the symbol is the ratio of the diameter change to length change.

Learn Symbol →



Slope

The slope modifying symbol is placed with a value to indicate that this value is controlled as a slope, or a "flat taper." The value listed with the symbol is the ratio of the height change to length change.

Learn Symbol →