Chapter Goals

- Interpret the perpendicularly control
- Interpret the angularity control
- Interpret the parallelism control
Introduction

- We are going to explore controls that apply to the orientation of a part.
- These controls are for the perpendicularly, angularity and parallelism of the part.
- Orientation controls are used when other drawing limits are inadequate to control the assembly or fit for function of a part.
Implied right angles

Two lines shown at 90 degrees create an implied 90 degree angle with a tolerance usually noted in the title block or a note on the drawing.

Shortcomings of implied 90 degree angles

- Tolerance zone is fan shaped that is gets bigger the farther you are from the origin of measurement.
- No datum to inspect from? What side is the inspection done from?
Coordinate Tolerancing and Perpendicularity

A

10 ± 0.5

50 ± 0.5

UNLESS OTHERWISE SPECIFIED ANGLES ± 2°

B

Should the part be inspected from the short side or from the long side?

4° tolerance zone

The tolerance zone becomes larger as the distance from the surface plate increases

Surface Plate

FIGURE 7-2 Implied Right (90°) Angles
The condition that results when a surface, axis or centerplane is exactly 90 degrees to a datum.

Perpendicularly Control

A geometric tolerance that limits the amount a surface, axis or centerplane is permitted to vary from being perpendicular to the datum.
Two Parallel Planes

A Cylinder
Applied to a surface

- Shape of the zone is two parallel planes perpendicular to the datum plane.
- Tolerance value of the perpendicularity control defines the distance separating the two planes.
- All elements of the surface must be within the tolerance zone.
- The perpendicular tolerance zone limits the flatness of the tolerance surface.
Add Picture of Perp to a surface

FIGURE 7-4 Perpendicularity with Two Datum References
Perpendicularly Applications

- Applied to a planar FOS
  - Used to ensure assembly.

- Certain Rules Apply
  - Tolerance zone is two parallel planes perpendicular to the datum plane.
  - Tolerance value of the perpendicularity control defines the distance separating the two planes.
  - Centerplane of the AME of the FOS must be within the tolerance zone.
  - Bonus tolerance applies
  - Fixed gage may be used to verify perpendicularity control.
Perpendicular Control to a planar Surface

FIGURE 7-5 Perpendicularity Applied to a Planar FOS

<table>
<thead>
<tr>
<th>Slot AME</th>
<th>tol.</th>
<th>Bonus tol.</th>
<th>Tolerance zone width</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.4</td>
<td>0.1</td>
<td>0.0</td>
<td>0.1</td>
</tr>
<tr>
<td>6.5</td>
<td>0.1</td>
<td>0.1</td>
<td>0.2</td>
</tr>
<tr>
<td>6.6</td>
<td>0.1</td>
<td>0.2</td>
<td>0.3</td>
</tr>
<tr>
<td>6.7</td>
<td>0.1</td>
<td>0.3</td>
<td>0.4</td>
</tr>
<tr>
<td>6.8</td>
<td>0.1</td>
<td>0.4</td>
<td>0.5</td>
</tr>
</tbody>
</table>
Perpendicularity Applications

- Applied to a cylindrical FOS
  - Controls the axis of the FOS.
  - Tolerance zone is a cylinder perpendicular to the datum plane
  - Tolerance value of the perpendicularity control defines the diameter of the tolerance zone cylinder.
  - Axis of the diameter must be within the tolerance zone (when FOS is at MMC).
  - Bonus tolerance allowed.
  - WCB of the diameter is affected.
  - Fixed gage may be used to verify.
Perpendicular Control on a Cylindrical FOS

![Diagram of a cylindrical FOS with a table showing diameter, tolerance, and zone values.](image)

**FIGURE 7-6** Perpendicularity Applied to a Diometrical FOS
More Perpendicular Control Notes

- A perpendicular control applied to a surface does not affect the WCB but a perpendicular control applied to a FOS does.

- Note that the WCB of a FOS that is tolerated with an orientation is oriented relative to the datum's specified.
Several controls indirectly affect perpendicularity.
- Tolerance of position
- Runout
- Profile

Indirect controls are not measured. If you want perpendicularity inspected, apply the control.
Review flow chart pg. 185 fig. 7-7.

Note that this applies only to RFS datum references only.

Note that if the control is applied to a surface then the projected tolerance zone, diameter, MMC and LMC may not be used in the tolerance portion of the feature control frame.
There are three separate checks needed to verify the control.

- Size of FOS
- Rule One Boundary
- Perpendicularity requirement

The inspection of the perpendicularity requirement is a reverse go gage.
Insert Inspection Picture

FIGURE 7-8 Inspecting Perpendicularity
The condition of a surface, centerplane or axis being exactly at a specified angle.

The control is a geometric tolerance that limits the amount a surface or centerplane is allowed to vary from its specified angle.
Need Pictures

Two Parallel Planes

Cylinder
Angularity Applied to a Surface

- WCB of the tolerated surface is not affected.
- Tolerance zone is two parallel planes.
- Angularity control tolerance defines distance between zone planes.
- All elements of surface must lie between zone planes.
- Tolerance zone is oriented relative to the datum plane by a basic angle.
- Angularity tolerance zone limits flatness of tolerated surface.
Angularity Applied to a Surface
The WCB of the FOS is affected.

If FOS is toleranced with an orientation control, the WCB is oriented relative to the datums specified.

If an angularity control is applied to a cylindrical FOS, some new rules apply.
Tolerance zone is a cylinder.

Angularity control tolerance defines diameter of tolerance cylinder.

Axis of toleranced feature must be within tolerance zone.

Tolerance zone is oriented relative to the datum plane by a basic angle.

An implied 90 degree basic angle exist in the other direction.
Angularity Control to a Diametrical FOS

Note the two datums.
Several geometric controls indirectly affect angularity of a part feature.

- Tolerance of position
- Total runout
- Profile

Remember that indirect controls are not inspected. If you need the angularity checked, use an angularity control.

Also note that the tolerance of angularity control should be less than the tolerance value of any indirect control.
Legal Test of an Angularity Control

Review Flow sheet page 190, fig. 7-11.
Verifying Angularity

FIGURE 7-12 Verifying Angularity

Dial indicator reading is the angularity error of the surface

0.4 wide angularity tolerance zone

Gage blocks

Surface plate

Part from Figure 7-9

Sine plate used to orient part surface to basic angle
Implied Parallelism

If two surfaces are shown parallel on a drawing, the size dimensions of the surfaces controls the parallelism between the surfaces.

Poor inspection due to lack of datums and parallelism has same dimensions as size.

Not a good way to design.
Implied Parallelism Picture
Parallelism is the condition that results when a surface, axis or centerplane is exactly parallel to a datum.

The control is a geometric tolerance that limits the amount a surface, axis or centerplane is permitted to vary from being parallel to the datum.
Parallelism Tolerance Zones

Two Parallel Planes

Cylinder/Diameter
If a parallelism control is applied to a surface, the following applies:

- Tolerance zone is two parallel planes parallel to the datum plane.
- Tolerance zone is located within the limits of the size dimension.
- Tolerance value of the parallelism control defines the distance between tolerance zone planes.
- All elements of the surface lie within the tolerance zone.
- Parallelism tolerance zone limits flatness of the tolerated feature.
If parallelism control, containing the MMC modifier, is applied to a cylindrical FOS, the following applies:

- Tolerance zone is a cylinder parallel to the datum plane.
- Tolerance value of the parallelism control defines the diameter of the tolerance zone cylinder.
- Axis of the cylinder must be within tolerance zone, when FOS is at MMC.
- Bonus tolerance is permissible.
- Fixed gage may be used.
- WCB or VC of the hole is affected.
When a surface is to be parallel to a datum, the feature control frame is either connected by a leader to the surface or directly connected to the extension line of the dimension.
- When parallelism is applied to an axis the axis of the hole may be specified within a tolerance zone that is parallel to a given datum.
  - The feature control frame is placed with the diameter dimension.
Parallelism with Tangent Plane Modifier

- Tangent plane modifier, $T$, denotes that only the tangent plane established by the high points of the controlled surfaces must be within the parallelism tolerance zone.

- If the tangent plane modifier is used, the flatness of the tolerance surface is not controlled.
Parallelism with Tangent Plane Modifier

Other rules apply:

- Tolerance zone is two parallel planes.
- Tangent plane is established by the high points of the surface and must be within the tolerance zone of the control but is not a flatness control.
- Tolerance zone will “float”.
Tangent Plane Modifier Application

FIGURE 7-16 Parallelism with the Tangent Plane Modifier

Tangent plane of the tolerated surface must be within the tolerance zone.

Parallelism tolerance zone is two parallel planes 0.1 apart. Zone may float between the max and min dimensional limits.

Datum plane A

Flatness of surface is not controlled by the parallelism control.
Several geometric controls indirectly affect parallelism of a feature:

- Tolerance of position
- Total runout
- Profile

Remember that indirect controls are not measured. To measure parallelism one must use a control.
Review fig. 7-17, page 197.
Inspecting Parallelism

FIGURE 7-18 Verifying Parallelism
### Summary

A summarization of orientation control information is shown in Figure 7-19.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Datum reference required</th>
<th>Can be applied to a Surface</th>
<th>Can be applied to a FOS</th>
<th>Can affect WCB</th>
<th>Can use M or L modifier</th>
<th>Can be applied at RFS</th>
<th>Can use T modifier</th>
</tr>
</thead>
<tbody>
<tr>
<td>⊥</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes*</td>
<td>Yes*</td>
<td>Yes**</td>
<td>Yes*</td>
</tr>
<tr>
<td>&lt;</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes*</td>
<td>Yes*</td>
<td>Yes**</td>
<td>Yes*</td>
</tr>
<tr>
<td>//</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes*</td>
<td>Yes*</td>
<td>Yes**</td>
<td>Yes*</td>
</tr>
</tbody>
</table>

* When applied to a FOS  ** Automatic per Rule #2  • When applied to a surface

**FIGURE 7-19 Summarization of Orientation Controls**
Questions?
Chapter Eight

Tolerance of Position, Part One
Chapter Goals

- Understand fundamental concepts of tolerance of position; definitions, conventions, advantages and basic theories.
- Interpret RFS and MMC tolerance of position applications.
- Draw cartoon gages for tolerance of position (MMC) applications.
True position: the theoretically exact location of a FOS as defined by basic dimensions.

Tolerance of Position Control: a geometric tolerance that defines the location tolerance of a FOS from its true position.
A TOP control, specified at RFS, defines a tolerance zone that the center, axis, or centerplane of the AME of a FOS must be within.

If specified on an MMC or LMC basis, TOP control defines a boundary, VC, that may not be violated by the surface(s) of the considered feature.
If specified at RFS, the TOP feature control frame will have no modifiers. Remember that RFS is the default condition for all geometric tolerances.
TOP Feature Control Frames

1. **TOP with RFS implied**
   - \( \varnothing 0.2 \quad A \quad B \quad C \)

2. **TOP with MMC specified**
   - \( \varnothing 0.2 \quad M \quad A \quad B \quad C \)

3. **TOP with LMC specified**
   - \( \varnothing 0.2 \quad L \quad A \quad B \quad C \)

**FIGURE 8-2 TOP Feature Control Frames**
When a TOP control is specified, the theoretically exact location of the axis or centerplane of the FOS must be defined with basic dimensions.

This exact location is called the true position of the FOS.
TOP Tolerance Zone and True Position

![Diagram of TOP Tolerance Zone and True Position]

**Figure 8-3** TOP Tolerance Zone

- True position of $\Omega_{10.6}^{10.0}$
- $\Omega_{0.2}$ cylindrical tolerance zone perpendicular to datum A
- (From datum B)
- (From datum C)
Basic dimensions define the true position relative to the referenced datums.

If basic dimensions are not specified, they are implied.

- Implied basic 90 degree angle: applies where centerlines of features are located and defined by basic dimensions and no angle is specified.

- Implied basic zero dimension: a centerline or centerplane of a FOS is shown in line with a datum axis or centerplane, the distance between the centerlines or centerplanes is an implied basic zero.
Implied Angles and Zero Dimensions

The 90° angles for the holes are an implied basic 90° from datum plane A.

The centerline of this diameter is an implied basic zero from datum axis A.

FIGURE 8-4 Implied Basic 90° Angles and Implied Basic Zero Dimension
Advantages of Tolerance of Position

- Provides larger tolerance zones; cylindrical tolerance zones are 56% larger than square zones.
- Permits bonus tolerance and datum shift.
- Prevents tolerance accumulation.
- Permits use of functional gages.
- Protects part function.
- Lowers manufacturing costs.
Commonly used with TOP controls.

57% larger tolerance zone?

- How?

Bonus and Datum shift add additional tolerance, up to 100%.

Help clarify drawing to manufacturing

Reduces overall cost.
CT and TOP Tolerance Zone Comparison

FIGURE 8-5 Comparison of Coordinate Tolerancing and TOP Tolerance Zones
What Can Be Controlled With TOP?

- Distance between features of size such as holes, bosses, slots, tabs, etc.
- Location of features of size, or patterns of features of size, such as holes, bosses, slots, tabs, etc.
- Coaxiality between features of size.
- Symmetrical relationship between features of size.
If TOP controls are used, you must specify under which material condition the control is to apply. (MMC, LMC or RFS).

The function of the FOS being tolerated is the primary dictate for the material condition.

Cost of production is a secondary consideration but still important.

Do not forget inspection cost.

Note that the MMC modifier is most common and least costly.
# Guide For Selecting TOP Control Modifiers

<table>
<thead>
<tr>
<th>Modifier</th>
<th>Commonly used in these functional applications</th>
<th>Bonus or datum shift permissible</th>
<th>Relative cost to produce and verify</th>
</tr>
</thead>
</table>
| M        | • Assembly  
          • Location of a non-critical FOS         | Yes                             | Lowest                            |
| L        | • Minimum wall thickness  
          • Minimum part distance  
          • Minimum machine stock  
          • Alignment              | Yes                             | Greater than MMC; less than RFS   |
| RFS invoked by showing no modifier | • To control a symmetrical relationship  
                                           • When the effects of bonus or datum shift will be detrimental to the function of the part  
                                           • To control minimum machine stock.  
                                           • Centering  
                                           • Alignment | No                              | Highest                           |

**FIGURE 8-6 Guide for Selecting Modifiers in TOP Controls Based on Product Function**
Virtual Condition Boundary Theory:
- Theoretical boundary that limits the location of the surfaces of a FOS.

Axis Theory:
- Axis or centerplane of a FOS must be within the tolerance zone.
Virtual Condition Boundary Theory

- TOP at MMC
  - Hole must be specified limit of size.
  - Located so no element of its surface will be inside a theoretical boundary.
  - Theoretical boundary centered about true position of the hole.
  - Theoretical boundary is equal to the MMC of the FOS minus the TOP tolerance.
    - Theoretical boundary is the VC of the hole (gage pin).
TOP VCB for Internal FOS

VC = MMC - TOP tol value
\[ \varnothing 3.8 = \varnothing 4.0 - 0.2 \]

Hole location is limited by theoretical boundary
Orientation variation is limited by the theoretical boundary

The virtual condition boundary is a basic 90° to datum plane A
A TOP is also an indirect orientation control.

The theoretical boundary is orientated relative to the primary datum referenced in the TOP callout.

The gage pin also limits the orientation of the FOS.
Virtual condition boundary theory also applies to an external FOS.

Theoretical boundary for a TOP, at MMC, of an external FOS is the MMC of the FOS plus the TOP tolerance.

Location and orientation of the FOS is limited by the TOP control.
Axis Theory

- Used when TOP is applied at RFS.
- Specified TOP applies at whatever size the FOS is produced at.
- Axis of the hole AME must be within the TOP tolerance zone cylinder.
- The TOP tolerance zone cylinder is centered around the true position of the hole.
- The diameter of the TOP tolerance zone is equal to the tolerance value specified in the TOP callout.
TOP Axis Theory: Internal FOS

FIGURE 8-9 TOP Axis Theory (Internal FOS)
A TOP is also an indirect orientation control.

The theoretical boundary is orientated relative to the primary datum referenced in the TOP callout.

Tolerance zone that controls the location of the FOS also limits the orientation of the FOS.
Tolerance zone for a TOP, applied at RFS, of an external planar feature of size is two parallel planes spaced apart at a distance equal to the TOP tolerance.

The orientation and location of the centerplane of the AME of the FOS is limited by the TOP tolerance zone.
TOP Axis Theory: External Planar FOS

Datum centerplane B

Two parallel planes contact the part to establish datum centerplane B

Datum plane A

The centerplane of the tab is located and oriented by the TOP tolerance zone

0.2 tolerance zone width

Outer boundary of tab = 6.8

FIGURE 8-10 TOP Axis Theory (External FOS)
Three conditions are present when a TOP control is applied at RFS:

- Tolerance zone applies to the axis or centerplane of the FOS.
- Tolerance value applies regardless of size of the tolerated FOS.
- Requirement must be verified with a variable gage.
- Review Chart on page. 216.
In TOP, when it is an RFS situation, there are two tolerance zones commonly found:

- Fixed diameter cylinder
- Two parallel planes a fixed distance apart

The diameter of the cylinder or the distance between planes is equal to the tolerance value specified in the TOP callout.

Location of the tolerance zone is always centered around the true position of the FOS.
Figure 8-9, page 219 has a diameter symbol in the tolerance portion of the FCF. This says a cylindrical tolerance zone.

Figure 8-10, page 220, there is no diameter modifier in the tolerance portion of the FCF. This says the tolerance zone is two parallel planes.
Hole location may be controlled with a TOP at RFS. Figure 8-11 shows such a situation. The axis of the hole is controlled relative to the outside surfaces of the part.
Following rules apply here:

- Shape of tolerance zone is a cylinder
- Tolerance zone is located by basic dimensions relative to the datum planes
- Tolerance zone is RFS
- Dimension between centerline of hole and datum A is an implied 90 degrees
- No datum shift is possible
- Tolerance zone controls orientation of hole relative to the primary datum from TOP callout
- WCB of hole is affected...6.0-0.2=5.8
Hole Controlled With TOPS @ RFS
When more than one hole, a pattern, is located by a TOP at RFS, certain conditions will apply. An example is fig. 8-12, pg. 225.

The following applies:

- Shape of tolerance zone is cylindrical
- Tolerance zone is located by basic dimensions
- Tolerance zones are at RFS
- Tolerance zones control orientation of holes relative to the primary datum
- Tolerance zones are at an implied 90 basic degrees to datum A
- Rule One applies
The axis of the AME of each hole (RFS) must be within the tolerance zone.

Datum plane C

Datum plane B

Datum plane A

The orientation of the axis of the AME of each hole (RFS) is also limited by the tolerance zone cylinder.

FIGURE 8-12 Pattern of Holes Controlled with TOP Using RFS
When the location of coaxial diameters is controlled by a TOP @ RFS, the following applies:

- Tolerance zone is a cylinder
- Tolerance zone is applied at RFS
- Dimension specifying the location of the diameter relative to the datum feature is an implied zero
- Tolerance zone limits the orientation of the tolerated diameter relative to the datum A axis
- No datum shift allowed
- Rule one applies
Coaxial Diameters Controlled by TOPs @ RFS

The axis (of the AME) of the tolerated diameter (RFS) must lie within the tolerance zone cylinder centered about datum axis A.
Three separate checks are needed to verify part:

- Size of hole
- Rule one boundary
- TOP requirements

Hole size and rule one was explained before.

TOPS inspecting requires a variable gage.

Coordinate Measurement Machine is the ideal
One way the TOP requirement on the part from Figure 8-11 could be inspected is shown in Figure 8-14. First, the part is rested on the surface plate and the gage elements that simulate the datum reference frame. The first, second, and third part surfaces to contact the inspection equipment are defined by the datum sequence of the TOP callout. Once the part is located in its datum reference frame, the location of the hole is established. A “best fit” gage pin is placed in the tolerated hole. The gage pin represents the actual mating envelope of the hole. Next, the location of the center of the gage pin—relative to the datum reference frame—is determined. The center of the gage pin must be within the tolerance zone cylinder that is defined by the TOP callout.

FIGURE 8-14 Inspecting TOP Applied
A TOP applied at a max material condition is specified when function, assembly or the effects of a bonus tolerance/datum shift would not affect the actual function/assembly of the part.

If a TOP @ MMC is used, the following applies:
- Tolerance zone is considered a boundary zone
- Bonus tolerance and datum shift is allowed
- Requirement can be verified with a functional gage
MMC and RFS Comparison

MMC Tolerance Zones
In tolerance of position MMC applications, two tolerance zone shapes are common: a virtual condition cylindrical boundary and a virtual condition parallel plane boundary. The virtual condition boundary is often considered the gage pin (or width) size. The location of the tolerance boundary is always centered around the true position of the FOS.

<table>
<thead>
<tr>
<th></th>
<th>MMC</th>
<th>RFS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tolerance zone</td>
<td>A boundary zone</td>
<td>An axis zone (or centerplane)</td>
</tr>
<tr>
<td>Bonus tolerance permissible</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Gaging</td>
<td>Functional (fixed)</td>
<td>Variable</td>
</tr>
</tbody>
</table>

FIGURE 8-15 TOP MMC/RFS Comparison
Hole Location Controlled with TOP @ MMC

In Figure 8-16, the maximum allowable bonus is equal to the difference between the MMC and LMC of the AME of the tolerated diameter.
If a hole pattern is controlled by TOP @ MMC, the location of the holes is controlled relative to the part edges and the following applies:

- Tolerance zone shapes are Virtual Condition boundaries
- Tolerance zones are located by basic dimensions from the datum planes
- Relationship between centerlines of holes and datum plane A is implied basic 90 deg.
- Bonus tolerance is permissible
- Rule one applies
- Tolerance zone controls orientation of the holes relative to the primary datum referenced in TOP callout
FIGURE 8-17 Pattern Location Controlled with TOP MMC

The surface of the holes cannot be within the virtual condition boundaries.

Datum plane C

The orientation of the axis of the AME of each hole is also limited by the VC boundary.

Datum plane A

Ø 7.7 VC boundary located at the true position of each hole.
## Coaxial Diameter Applications

<table>
<thead>
<tr>
<th>Configuration</th>
<th>Method</th>
<th>Example</th>
<th>Max Permissible Bonus/Datum Shift</th>
</tr>
</thead>
</table>
| Opposed Diameters                    | Use one diameter to establish the datum axis. Locate the second diameter relative to the datum axis. | ![Diagram](image1.png) | Bonus = 0.2  
Datum shift = 0.6 |
| Non-opposed Diameters (same size)    | Implied self-datum or Specified self-datum                             | ![Diagram](image2.png) | Bonus = 0.04  
Datum shift = 0 |
| Non-opposed Diameters (different size) | Use both diameters as datums, and relate each diameter to the common axis. | ![Diagram](image3.png) | Bonus = 0.2  
Datum shift = 0 |

**FIGURE 8-18** Coaxial Diameters Controlled with TOP MMC
For a TOP to be a legal callout, the following applies:

- TOP must be applied to a FOS
- Datum references are **required**
- Basic dimensions must be used to establish the true position of the tolerated FOS from the datums referenced and between FOS in a pattern

All of these conditions must be met or callout is not legal.

Refer to fig.8-19, pg.234
Multiple types of gages may be used to verify a TOP applied at MMC. These include:

- CMM
- Variable gages
- Other functional gages

A functional gage is one that verifies functional requirements of part features as defined by the geometric tolerances.

- This means that a functional gage mimics the actual function of the part being measured.
Benefits of a Functional Gage

- Represents worst case mating part
- Parts may be verified quickly
- Economical to produce
- No special skills needed to read or interpret results
- Functional gage may be able to check several part characteristics at the same time
Gage should represent the virtual condition of the tolerated FOS

Note that the TOP applied at MMC may be verified by other devices, such as an CMM.

Note that a functional gage must also be verified as to dimensional correctness
A sketch of a functional gage. A cartoon gage defines the same part limits that a functional gage would, but it does not represent the actual gage construction of a functional gage.

A cartoon gage construction is described in figure 8-20, page 237.
STEP 1
Determine the size of the gage feature.

STEP 2
Establish the simulated datums.

STEP 3
Locate the gage features relative to their respective datums.

FIGURE 8-20 Steps for Drawing a Cartoon Gage
The cartoon gage for this application was drawn using the designer’s judgment. The effects of the TOP callout are added to the MMC of the tolerated diameter to produce the virtual condition. The gage must also be built to conform to the virtual condition of datum B.

Note that the gage only checks orientation and location. Form and size must be checked separately.
Summary of TOP Information

- Review figure 8-22, page 239.

<table>
<thead>
<tr>
<th>TOP control</th>
<th>Datum reference required</th>
<th>Can be applied to a Surface</th>
<th>Can affect WCB</th>
<th>Can use M or L modifier</th>
<th>Can be applied at RFS</th>
<th>Overrides Rule #1</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Yes*</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes**</td>
</tr>
</tbody>
</table>

* Coaxial diameter exception

** Is automatic per Rule #2

FIGURE 8-22  Summarization of TOP
Questions?
Chapter Nine

Tolerance of Position: Part Two
Chapter Goals

- Interpret tolerance of position special applications.
- Calculate distances on a part dimensioned with tolerance of position.
- Calculate tolerance of position tolerance values using the fixed and floating fastener formulas.
There are times that holes are not parallel or perpendicular to the datum axis. In order to control the location and orientation of these holes, a TOP control is used.

Figure 9-1 is one example.
There are some rules or conditions that apply in this type of instance:

- Tolerance zone is the cylindrical virtual condition boundary.
  - What is the VC?
- Tolerance zones are located by basic dimensions relative to the referenced datums.
- Angle of the hole, relative to datum B, is limited by the TOP.
- Bonus tolerances are permitted.
**TOP Applied To Non-Parallel Holes**

**FIGURE 9-1** TOP Applied to Holes that are Non-Parallel

<table>
<thead>
<tr>
<th>Hole AME</th>
<th>φ Tol. Dia</th>
<th>Bonus Tol.</th>
<th>Total Tol. Dia</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.0</td>
<td>0.4</td>
<td>0</td>
<td>0.4</td>
</tr>
<tr>
<td>6.1</td>
<td>0.4</td>
<td>0.1</td>
<td>0.5</td>
</tr>
<tr>
<td>6.2</td>
<td>0.4</td>
<td>0.2</td>
<td>0.6</td>
</tr>
</tbody>
</table>

The VC boundary tolerance zone = 5.6 dia.
There are times we need to provide a hole a tolerance in more than one direction. This is where a bi-directional control is used.

Two feature control frames are used and each one is attached to a dimension line of the hole in the direction you need controlled.

Figure 9-2 is an example of this instance.
Bi-Directional TOP

The following applies to this bi-directional TOP application:

- Tolerance zones are parallel boundaries in the direction of the TOP control.
- Shape of the tolerance zones is irregular.
- Tolerance zones are located by basic dimensions relative to the datums referenced.
- Bonus tolerances are permitted.
What is the VC formula?
Elongated holes are unique in the application of TOP. One would think that this type of hole needs to have bi-directional TOP controls. In fact this is the best way of design but you could still use one TOP.

Again the feature control frames are attached to the dimension lines and the word BOUNDARY is added beneath each feature control frame.

Figure 9-3 is an example of this application.
The following conditions apply for this application:

- Tolerance zone is a boundary of the identical shape as the elongated hole, minus the position tolerance value in each direction.
  - Why?
- Tolerance zones are located by basic dimensions relative to the datums referenced.
- Bonus tolerances are permitted.
- Elongated hole must meet size requirements.
What happens if only one TOP control is used?
Threaded hole dimensioning is not easy. One must consider just what is being dimensioned and how the rest of the design is affected.

Figure 9-4 is an example of this issue.
The squareness error of the pin or fastener has resulted in an interference fit with the hole. This prevents proper torque being applied or fastener being fully seated.

To eliminate this condition, one should use a projected tolerance zone.

Back to Figure 9-4.
TOP and Projected Tolerance Zone

**Interference condition**

**Without projected tolerance zone**

**With projected tolerance zone**

FIGURE 9-4 Fixed Fastener Interference Condition
A tolerance zone that is projected above the part surface. There is a specific symbol for a PTZ, this would be placed in the tolerance section of the feature control frame.

Several conditions apply to a PTZ:

- Height is specified after the symbol in the FCF.
- Should be equal to the thickness of the mating part.
- Orientation of the fastener is fixed.
- Figure 9-5 is another example.
If symmetry is critical relationship on a part, the centerplane of the part (AME) may be controlled with a TOP control.

Figure 9-6 is an example of this application.

These conditions would apply:

- Tolerance zone shape is two parallel plates.
- Tolerance zone is located by implied basic zero dimensions.
- Bonus tolerance is allowed.
- Datum shift is allowed.
Symmetry Controlled With TOP

Can also be used at LMC and RFS
If one needs to control a minimum distance on a part, say a wall thickness, then a TOP control with an LMC modifier will work.

Figure 9-7 is an example of this application.

Some conditions apply:
- Tolerance zone shape is a cylindrical boundary.
- Dimension between centerline of diameter and datum axis is an implied zero.
- Bonus tolerance is allowed.
- Perfect form at LMC applies.
MW = 1.6 \[(24.2 - 20.8 - 0.2)/2 = 1.6\]
If one wants to only control the spacing and orientation of holes in a pattern, a TOP control with a single datum may be used.

In figure 9-8, we have an example of a TOP control limiting the spacing between holes and the orientation about datum A but there is no control over location.

Further, the gage only checks perpendicularly and spacing.
TOP & Hole Pattern

FIGURE 9-8 TOP with a Single Datum Reference Applied to a Hole Pattern
Multiple Single-Segment TOP Controls

- Sometimes you may want to control location, spacing and orientation of a pattern of features of size. Then multiple single segment TOP controls will be used.
- Figure 9-9 is an example of this application.
- Note that the upper segment tolerances location while the lower tolerances spacing and orientation.
Multiple TOP Segments

FIGURE 9-9 Multiple Single Segment TOP Control with a Single Datum Reference in the Lower Segment
Zero tolerancing is a method of helping manufacturing produce a quality part with the lowest cost.

ZT @ MMC simply states a zero tolerance at MMC but places the geometric tolerance with the FOS tolerance.

Figure 9-10 illustrates this application.
ZT @ MMC

Positional tolerance
Bonus tolerance
Total tolerance

Positional tolerance
Bonus tolerance
Total tolerance

Virtual condition of holes
MMC hole
Positional tolerance
Virtual condition

A Conventional Tolerancing

B Zero Tolerance at MMC

FIGURE 9-10 Conventional and Zero Tolerance at MMC Comparison
A Tolerance Analysis Chart

FIGURE 9-11 Conventional and Zero Tolerance Analysis Chart at MMC
Tolerance stacks at RFS

This method will allow you to calculate the minimum and maximum distance between the edges of two holes.

The process uses the basic dimensions between holes, TOP tolerance value and the MMC or LMC hole size.
FIGURE 9-12  Tolerance Stacks Using TOP at RFS
Here we are going to use something called the gage method. This method involves using a cartoon gage to calculate distance.

An advantage is that bonus tolerances and datum shifts are automatically included in the calculations.

The five steps are illustrated in figure 9-13.
Five steps:
1. Draw the cartoon gage.
2. Draw the part on the gage in the position that gives the extreme condition being calculated.
3. Label start and end points of the distance calculated.
4. Establish a path of continuous known distances from the start point to the end point.
5. Calculate the answer.
Another Example

FIGURE 9-14  Tolerance Stacks Using TOP at MMC

- **A** - 3.5 Gage pin radius
- **B** - 60.0 Gage pin location
- **C** + 70.4 Max width dimension from datum plane B
  + 6.9 Max distance X

**STEPS 2, 3, & 4**

- **A** - 8.6 LMC hole dia.
- **B** + 3.5 Gage pin radius
- **C** - 60.0 Gage pin location
- **D** + 69.6 Min width dimension from datum plane B
  + 4.5 Min distance X
This is where the fastener is fixed or restrained by/into one of the components of the assembly.

- Typically a screw, stud or other threaded fastener.

Figure 9-15 is an example of screws as fixed fasteners.
Fixed Fastener

FIGURE 9-15 Example of a Fixed Fastener Assembly

Cover

Housing

4X M14 X2 Screws

Cover
This formula determines the amount of tolerance for a fixed fastener and needs to have the projected tolerance modifier used on the threaded hole.

The formula is:

\[ T = \frac{(H - F)}{2} \]

- \( T \) is position tolerance diameter
- \( H \) is MMC of the clearance hole
- \( F \) is MMC of the fastener
**Fixed Fastener Example**

**Fixed Fastener Formula**

\[
T = \frac{H - F}{2}
\]

\[2T = 14.4 - 14\]

\[T = 0.2\]

**Figure 9-16** Using the Fixed Fastener Formula to Determine TOP Tolerance Values
In figure 9-16, the MMC modifier is used as the function of the holes is assembly. The MMC allows for additional tolerance for assembly.

Now the projected tolerance zone may not be specified, if that happens the tolerance formula changes to:

- $T = F + 2T$
- Symbols still mean the same.
A floating assembly is where the components are held together by a combination of a bolt and nut or some other type of fastener where both of the fastened components have clearance holes for the fastener.

Figure 9-17 is an example of such an application.
Floating Fasteners

FIGURE 9-17 Examples of a Floating Fastener Assembly
The formula for floating fasteners tolerance is:

\[ T = H - F \]

- \( T \) is position tolerance
- \( H \) is MMC of clearance hole
- \( F \) is MMC of fastener

The tolerance when determined applies to each part in the assemble.

The MMC modifier is needed.

Figure 9-18 is an example of this application.
Floating Fastener Example

Use M6 x 1.5 fasteners with a MMC of 6.0

Floating Fastener Formula

\[ T = H - F \]

\[ T = 6.8 - 6.0 \]

\[ T = 0.8 \]

FIGURE 9-18 Using the Floating Fastener Formula
Floating Fastener Notes

Floating fasteners seem to be simple. Drill holes, keep tolerances adequate and check the formula. But there are other items to consider:

- Clearance of the threaded portion where the nut goes. Is the clearance for the threaded end?
- Can you get the wrench or socket on the flat end? Is there vertical and horizontal clearance?
- When the assembly is put into the system, will there be room for service or is this a blind assembly?
Questions?
Chapter Ten

Concentricity and Symmetry Controls
Chapter Goals

- Interpret the concentricity control
- Interpret the symmetry control
Concentricity is a condition where the median points of all diametrically opposed elements of a cylinder or surface of revolution are congruent with the axis of a datum feature.

Note: Median points are at the midpoint of a two point measurement.
Concentricity Tolerance Zone

FIGURE 10-2 Example of a Concentricity Control Tolerance Zone
The concentricity control is a geometric tolerance that limits concentricity error specifically for a part feature.

Tolerance zone is a 3 dimensional cylinder coaxial with the datum axis.

Diameter of the cylinder is equal to the concentricity tolerance value.

Concentricity is only applied at RFS and a datum needs to be specified.
Concentricity Examples

Opposing median points must fall in the cylindrical tolerance zone of .004.
All of the coaxiality controls are intended to control concentric features. For that reason, many designers and engineers choose concentricity. Unfortunately, concentricity ignores the size, roundness and cylindricity of the feature. It requires that the inspector derive a median line. In a situation where you don't care about the size, roundness or cylindricity of the feature, concentricity may be specified. I do know of a design where this is truly the case. The closest application, perhaps, is when dynamic balance is needed. In such a case, measuring a part statically does not assure dynamic balance if the material is not homogeneous. If dynamic balance is required, a dynamic balancing note is probably in order rather than concentricity. For that reason, When in Doubt, Use Runout.
Concentricity example

Ø0.3 concentricity tolerance zone

Datum axis A

Derived Median Line
Concentricity Example

Axis of feature must be in position within a 0.3 diameter cylinder.

Smallest circumscribed cylinder contacting feature to establish axis.

The entire surface must not runout (Full Indicator Movement) more than 0.3.
Runout Review

FIM at 6 slices of the feature

-0.02 to +0.01
+0.01 to +0.02
-0.02 to -0.04
-0.05 to -0.08
-0.07 to -0.09
-0.04 to -0.05

Circular Runout is 0.03
Total Runout is 0.11
But first some notes:

- Diameter must meet the size and rule 1 requirements.
- Concentricity control tolerance zone is a cylinder that is coaxial with a datum axis.
- Tolerance value defines the diameter of the tolerance zone.
- All median points of the toleranced diameter must be within the tolerance zone.
- The maximum distance between median points is half the concentricity tolerance value.
Concentricity Example

FIGURE 10-3 Concentricity Application
**Concentricity, Runout and TOP @ RFS**

<table>
<thead>
<tr>
<th>Concept</th>
<th>Geometric Control</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Concentricity</strong></td>
<td><strong>Total Runout</strong></td>
</tr>
<tr>
<td>Cylinder</td>
<td>Two coaxial cylinders</td>
</tr>
<tr>
<td>Median points of</td>
<td>Surface elements of a tolerated diameter</td>
</tr>
<tr>
<td>tolerated diameter</td>
<td></td>
</tr>
<tr>
<td>Relative cost to produce</td>
<td>$$</td>
</tr>
<tr>
<td>Relative cost to inspect</td>
<td>$$$</td>
</tr>
<tr>
<td>Part characteristics</td>
<td>Location and orientation</td>
</tr>
<tr>
<td>being controlled</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Design Tip**

As a rule of thumb, runout and concentricity should only be considered on parts that rotate.

---

**Figure 10-4 Differences Between Concentricity, Runout, and TOP**
TECHNOTE 10-2 Differences Between Concentricity and Runout

Two differences between runout and concentricity are:

1. The shape of the tolerance zone
2. Runout affects form

One difference between TOP (RFS) and concentricity is:

- With TOP the axis of the AME must be within the tolerance zone. With concentricity, the median points of the tolerated diameter must be within the tolerance zone.
Total runout is the difference between the highest and lowest readings found over the entire feature. The highest reading was +0.02 and the lowest reading was -0.09. Therefore, the total runout for the feature is 0.11, the difference between +0.02 and -0.09.
Page 286 has the flowchart for the test to determine if a concentricity control specification is legal. Note the conditions stated:

- FCF must be applied to a surface of revolution coaxial to the datum axis.
- Datum references are required.
- The concentricity symbol must be in the tolerance portion of the FCF.
- MMC, LMC, Tangent Plane and Projected Tolerance Zone may not be used in the FCF.
Inspecting Concentricity

- Remember that rule one and size requirements apply so the feature must meet size requirements.
- The rule one boundary must also be met.
  - What is the boundary definition?
- The concentricity requirement must be met.
- Does concentricity control feature size?
Inspecting Concentricity

Must calculate and locate the median points.

<table>
<thead>
<tr>
<th>Figure 10-6 Inspecting Concentricity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Datum axis A</td>
</tr>
<tr>
<td>Chuck or collet</td>
</tr>
<tr>
<td>Datum axis A</td>
</tr>
<tr>
<td>0.3 Tolerance zone</td>
</tr>
<tr>
<td>Median points of the tolerated diameter must be within the tolerance zone</td>
</tr>
</tbody>
</table>

X = Distance from datum axis to part surface
Y = Distance from datum axis to part surface
X+Y = Distance of two-point measurement
W = Midpoint = \( \frac{X + Y}{2} \)
Z = Distance between midpoint and datum axis
Z = X - W

Each distance Z must be within the cylindrical tolerance zone.
# Five Ways To Control Coaxiality of Features

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Used to Locate:</th>
</tr>
</thead>
<tbody>
<tr>
<td>⊙</td>
<td>The axis of a feature of size</td>
</tr>
<tr>
<td>→</td>
<td>Circular elements that are contained within a cylindrical feature that is shown centered on the datum axis.</td>
</tr>
<tr>
<td>↑</td>
<td>Cylindrical features shown centered on the datum axis. Controls all geometric characteristics except size.</td>
</tr>
<tr>
<td>⊙</td>
<td>Cylindrical features shown centered on the datum axis. Similar to total runout but also controls size if the size dimension is BASIC</td>
</tr>
<tr>
<td>○</td>
<td>Derived median points of a surface of revolution</td>
</tr>
</tbody>
</table>
# GD&T Symbols

<table>
<thead>
<tr>
<th>TYPE OF TOLERANCE</th>
<th>CHARACTERISTIC</th>
<th>SYMBOL</th>
<th>DATUM REFERENCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>FORM</td>
<td>STRAIGHTNESS</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>FLATNESS</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>CIRCULARITY (ROUNDNESS)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>CYLINDRICITY</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PROFILE</td>
<td>PROFILE OF A LINE</td>
<td></td>
<td>INDIVIDUAL OR RELATED</td>
</tr>
<tr>
<td></td>
<td>PROFILE OF A SURFACE</td>
<td></td>
<td>INDIVIDUAL OR RELATED</td>
</tr>
<tr>
<td>ORIENTATION</td>
<td>ANGULARITY</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PERPENDICULARITY</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>PARALLELISM</td>
<td></td>
<td></td>
</tr>
<tr>
<td>LOCATION</td>
<td>POSITION</td>
<td></td>
<td>RELATED</td>
</tr>
<tr>
<td></td>
<td>CONCENTRICITY</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>SYMMETRY</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RUNOUT</td>
<td>CIRCULAR RUNOUT</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>TOTAL RUNOUT</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Symmetry is the condition where the median points of all opposed elements of two or more feature surfaces are congruent with the axis or centerplane of a datum feature.

What this means is that opposed points must be equally spaced or apart and meet the conditions of the control.
Symmetry Control

- This is a geometric tolerance that limits the symmetry error of a part feature.
  - This control only works when applied to part features shown symmetrical to the datum centerplane.
  - The tolerance zone is centered about the datum centerplane and the width is equal to the tolerance value.
  - All median points must lie within the parallel plane tolerance zone regardless of feature size.
  - Symmetry controls are always applied at RFS.
Symmetry Example

FIGURE 10-7 Symmetry Control Tolerance Zone
Symmetry Example

Datum center plane A

All Derived Median Points of the feature
A symmetry control is a special application that requires work on the part of the designer or inspector to determine. Remember that one must determine the datum plane and then the tolerance zone locations in order to measure symmetry.
Often designers need to create symmetrically shaped parts. Symmetrical parts are usually easier to assemble, look better and help maintain balance in a design. Features shown symmetrical must be controlled to avoid incomplete drawing requirements (2.7.3 of ASME Y14.5M-1994). Symmetry is an option in these situations but it is difficult to measure since it requires deriving the features' median points to determine if they are contained within the specified tolerance zone which is centered on the datum axis or datum center plane.
Position may also be used to assure a symmetrical relationship. The advantages of using Position include the ability to modify the tolerance and datum reference at RFS (implied in 1994 Standard), MMC or LMC. In addition, verification is usually easier for Position than that required for Symmetry since it is the center plane of the Actual Mating Envelope (simulated by the inspection equipment) that must be within the tolerance zone.
Positional Control of Symmetry

Center plane

Datum center plane A

Processing equipment used to establish the center plane of the Actual Mating Envelope
When a symmetrical control is specified, the part in question usually needs to have a specific attribute of a part controlled. Specific areas may include wall thickness, functional appearance and other factors that include balance and fit.

If a symmetry control is applied, the following applies:
Controlled feature must meet size and rule one requirements.

Tolerance zone is two parallel planes centered about the datum centerplane.

Tolerance value of the symmetry control dictates distance between parallel planes.

All median points of the toleranced feature must be within the tolerance zone.
Symmetry Example

FIGURE 10-8 Symmetry Applications
# Symmetry and TOP @ RFS Differences

<table>
<thead>
<tr>
<th>CONCEPT</th>
<th>GEOMETRIC CONTROL</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>SYMMETRY</td>
</tr>
<tr>
<td>Tolerance zone</td>
<td>Two parallel planes</td>
</tr>
<tr>
<td>Tolerance zone applies to . . .</td>
<td>Median points of tolerated FOS</td>
</tr>
<tr>
<td>Types of part characteristics being controlled</td>
<td>Orientation and location</td>
</tr>
<tr>
<td>Relative cost to produce</td>
<td>$$$</td>
</tr>
<tr>
<td>Relative cost to inspect</td>
<td>$$$</td>
</tr>
</tbody>
</table>

**FIGURE 10-9 Differences Between Symmetry and TOP**
Figure 10-10 on page 291 is the flowchart of questions one needs to ask to ensure that the control you want to use is properly written.

Take note of the questions. This is the same flow we have always seen and will see in the next few chapters.
Inspecting Symmetry

1. Using the height gage, measure the opening of the variable jaws:
   - Establish the location of the datum centerplane from the surface plate.
   - Locate the dial indicators to be at the datum centerplane.

2. Take a two-point measurement on opposing part surfaces:
   - The difference in the gage reading—divided by two—is the symmetry error of the median point.

3. Repeat as many times as necessary.

FIGURE 10-11 Inspecting Symmetry
Symmetry may also be controlled by position.

**Table: Summary of Concentricity and Symmetry Controls**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Datum reference required</th>
<th>Can be applied to</th>
<th>Can affect WCB</th>
<th>Can use modifier</th>
<th>Can override Rule #1</th>
<th>Tolerance zone shape</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Two or more planar feature surfaces</td>
<td>Cylindrical or surface of revolution</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>○</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Cylindrical</td>
</tr>
<tr>
<td>─</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
<td>No</td>
<td>No</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Parallel planes</td>
</tr>
</tbody>
</table>

*FIGURE 10-12 Summarization of Concentricity and Symmetry Controls*

What do concentricity and symmetry indirectly control?
Chapter Eleven

Runout Controls
Chapter Goals

- Interpret the circular runout control
- Interpret the total runout control
Runout is a composite control. That is the form, location and orientation of a part feature is controlled simultaneously.

Runout controls may also control coaxiality of diameters.

Runout must always be applied with a datum reference.

Runout may be applied to any feature that surrounds or intersects a datum axis.

Runout is measured as the max indicator reading on the feature when rotated 360 degrees.
First we must establish a datum axis.

We do so by:

- Using a single diameter
- Two coaxial diameters sufficiently apart
- A surface and a diameter at right angles
- Figure 11-2 illustrates these concepts.
Establishing a Datum Axis

FIGURE 11-2 Establishing a Datum Axis for Runout
Circular runout is a composite control, that is a control for form, orientation and location of circular elements of a part feature.

A circular runout control is a geometric tolerance that limits the amount of circular runout of a part surface.

Circular runout may control location or if applied to a diameter, the form and location of a datum axis.
The tolerance zone for circular runout is two coaxial circles whose centers are located on the datum axis.

The outer tolerance zone circle is established by the feature element farthest from the datum axis. The inner circle is offset by the tolerance amount.

All surface elements must reside within the tolerance zone circles.

Figure 11-3 illustrates this
Circular Runout Tolerance Zone

**Tolerance zone shape**

- **Datum axis**
- **Part surface**
  - Two coaxial circles originate from the datum axis
- **Radius to surface element farthest from the datum axis**
- **Radial distance between circles equal to the runout tolerance value**

**Figure 11-3 Circular Runout Tolerance Zone**

- **A**
- **Ø 26.0 25.4**
- **8.6 8.4**
- **WCB = 27**
Circular runout, being a composite control, measures several different errors with a single measurement. Figure 11-4 illustrates these various errors.

Each error is measured by a single reading.
Circular Runout As Composite Control

A

Indicator reading (runout value) equal to zero

B

Circular element is perfectly round

Datum axis and feature axis are coaxial
Circular Runout As Composite Control

**Figure 11-4** Circular Runout as a Composite Control

- **C**
  - Datum axis and feature axis are coaxial.
  - Indicator reading (runout value) equal to 0.2.
  - Circular element is out-of-round within size limit.
  - Circular element rotated 90°.

- **D**
  - Axis of circular element.
  - Indicator reading (runout value) equal to 0.3.
  - Circular element rotated 180°.
  - Axis of tolerated diameter must be located and oriented within this zone.
  - (Max offset or eccentricity possible)
The following applies:

- Diameter must meet size requirements
- WCB is affected
- Runout control is applied RFS
- Runout applies at each circular element of the tolerated diameter
- Tolerance zone is two coaxial circles separated by the tolerance
- Maximum possible axis offset is one half of the runout tolerance
Circular Runout Applied To A Diameter

FIGURE 11-5 Circular Runout Applied to a Diameter
Circular Runout

Applied to a surface, the following applies:

- Control is applied at RFS
- Runout applies to each circular element of the surface
- Tolerance zone is two coaxial circles offset by the tolerance value
- Control does not control orientation of the surface
Circular Runout Applied To A Surface

FIGURE 11-6 Circular Runout Applied to a Surface
Figure 11-7 is the flowchart for answering the questions of a legal specification for runout.
Verifying Circular Runout

1. Must meet rule 1 boundary
2. Must meet size limits
A composite control that affects form, orientation and location of all surface elements, either a diameter or a surface, relative to a datum axis.

If applied to a diameter, the tolerance zone is two coaxial cylinders with centers located on the datum axis.

Figure 11-9 illustrates this principle.
Total Runout

![Diagram of a part with dimensions and an indicator moving parallel to the datum axis as the part is rotated.](image)

**FIGURE 11-9 Total Runout**

- Diameter: 6.2 mm to 6.0 mm
- Diameter: 12.8 mm to 12.6 mm
- Radial distance between cylinders: 0.1 mm

Indicator is moved parallel to the datum axis as the part is rotated.
Total Runout As A Composite Control

- Total runout limits cylindricity, orientation and axis offset of a diameter.
- TR does affect worst case boundary.
- Again, the single reading will read three types of part errors. Figure 11-10 illustrates this principle.
What happens if we have combined axis and form error?

Error could be form.

Entire error could be axis offset.

The dial indicator would measure:
- Cylindricity
- Orientation
- Axis offset

These types of part errors are combined into a single indicator reading.
Total Runout Application

- Diameter must meet size requirements
- WCB is affected
- Control is applied at RFS
- Runout applies simultaneously to all surface elements
- Tolerance zone is two coaxial cylinders
- Maximum axis offset is one half the tolerance value
Total Runout Applied To A Diameter
Control is applied at RFS

Runout applies simultaneously to all surface elements

Tolerance zone is two parallel planes perpendicular to datum axis

Runout controls the angular (orientation) aspect of the surface to the datum axis

Runout also controls the flatness
Total Runout Applied To A Surface

Why is flatness also controlled?
Runout may be applied to a diameter or a surface.

Legal specification for Total runout is the same as runout.

Figures 11-13 and 11-14 illustrate how to verify total runout.
Verifying Total Runout

**FIGURE 11-13 Verifying Total Runout**

- **Chock or Collet**: Max permissible dial indicator reading is the runout tolerance value.
- **Part rotated 360° and the gage is moved along the surface**: Indicator covers a helix on the surface of the diameter.
- **Datum axis A**:
Verifying Total Runout

FIGURE 11-14 Verifying Total Runout
Comparison between Circular and Total Runout

- Figures 11-15 and 11-16 illustrate how each runout control should be used and what could happen if not applied properly.
### Comparison Chart

<table>
<thead>
<tr>
<th>Concept</th>
<th>Circular Runout</th>
<th>Total Runout</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Tolerance zone</strong></td>
<td>Two coaxial circles</td>
<td>Two coaxial cylinders</td>
</tr>
<tr>
<td><strong>Relative cost to produce</strong></td>
<td>$</td>
<td>$ $</td>
</tr>
<tr>
<td><strong>Relative cost to inspect</strong></td>
<td>$</td>
<td>$ $</td>
</tr>
<tr>
<td><strong>Part characteristics being controlled</strong></td>
<td>Location Orientation Circularity</td>
<td>Location Orientation Cylindricity</td>
</tr>
</tbody>
</table>

**FIGURE 11-15** Comparison of Circular and Total Runout
Comparison of Controls

FIGURE 11-16 Comparison of Circular and Total Runout

Circular runout error = 0
Total runout error = 0.1

Circular runout error = 0
Total runout error = 0.1
This is a calculation to find the max/min distance on a part. When we calculate these values all tolerances must be used.

This calculation in figure 11-17 is one example and may be used for circular or total runout.
Runout Calculations

**Maximum distance X**

1. - 11.0  Min radius (ID)
2. + 0.2   Runout (ID) to datum axis A
3. + 0.15  (OD) Runout to datum axis A
4. + 12.4  Max radius (OD)

**Minimum distance X**

1. - 11.1  Max radius (ID)
2. - 0.2   Runout (ID) to datum axis A
3. - 0.15  (OD) Runout to datum axis A
4. + 12.2  Min radius (OD)

**FIGURE 11-17 Tolerance Stacks Using Runout**
Summary
A summarization of runout control information is shown in Figure 11-18.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Datum reference required</th>
<th>Can be applied to a</th>
<th>Can affect WCB</th>
<th>Can use M L T P modifier</th>
<th>Can be applied at RFS</th>
<th>Can override Rule #1</th>
<th>Tolerance zone shape **</th>
</tr>
</thead>
<tbody>
<tr>
<td>⧔</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>Yes*</td>
<td>No</td>
</tr>
<tr>
<td>⧔</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
<td>No</td>
<td>Yes*</td>
<td>No</td>
</tr>
</tbody>
</table>

* Is automatic per Rule #2
** When applied to a diameter

FIGURE 11-18 Summarization of Runout Controls
Questions?
Chapter Twelve

Profile Controls
Chapter Goals

- Understand profile tolerancing
- Interpret the profile of a surface control
- Interpret profile of a line control
There are two basic profile controls used today:
- Profile of a surface
- Profile of a line

Each has special characteristics to consider.

Profile is considered one of the most useful tools in the designers toolbox. Any surface, of any shape, may use a profile control.
Profile controls may be specified with or without a datum reference.

- If a datum is referenced, we have a related feature control
- If no datum is referenced, we have a form control
- Form controls apply specifically where the surface exist.
A profile is the outline of a part feature in a given plane.

- A true profile is the exact profile of a part feature as shown by basic dimensions.

A profile control is a geometric tolerance that specifies a uniform boundary along the true profile that the elements of the surface must lie within.
A profile line control applies to line elements of the tolerated surface.

The true profile must be located with basic or tolerated dimensions relative to the datums referenced in the profile control.

Figure 12-2 illustrates these issues.
Profile Control

A profile control specifies a uniform boundary within which the part surface must be.

An outline that is defined within basic dimensions is called a true profile.

The outline of a part is called its profile.

FIGURE 12-2 Profile Control
Profile of a surface has a tolerance zone for the width, length and depth of the control.

Profile of a line tolerance zone is length and width. Applies for full length of surface.

All profile controls, whether surface or line, are bilateral with equal distribution.

Figure 12-3 has examples and exceptions.
# Profile Tolerance Zone

<table>
<thead>
<tr>
<th>PROFILE TOLERANCE ZONE SPECIFICATION</th>
<th>INTERPRETATION</th>
<th>COMMENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bilateral - Equal distribution</td>
<td>Tolerance zone is a 1.0 uniform boundary centered around the true profile</td>
<td>The most common application of profile</td>
</tr>
<tr>
<td>Bilateral - Unequal distribution</td>
<td>Tolerance zone is a 1.0 uniform boundary offset 0.8 outside the true profile</td>
<td>Use of phantom lines and basic dimension specify the amount of unequal distribution</td>
</tr>
<tr>
<td>Unilateral - Outside</td>
<td>Tolerance zone is a 1.0 uniform boundary offset outside the true profile</td>
<td>Use of a phantom line denotes the direction the tolerance zone is offset</td>
</tr>
<tr>
<td>Unilateral - Inside</td>
<td>Tolerance zone is a 1.0 uniform boundary offset inside the true profile</td>
<td>Use of a phantom line denotes the direction the tolerance zone is offset</td>
</tr>
</tbody>
</table>

*FIGURE 12-3 Profile Tolerance Zone*
A profile control, when pointing to a surface, applies to the entire length and width of the surface.

There are ways of extending or moderating the control.

Figure 12-4 shows these ways.
Profile Tolerance Zone Coverage

Symbols are preferred over notes. Option C is better.
Advantages of Profile

- Clear definition of the tolerance zone.
- Communicates datums and datum sequence.
- Eliminates accumulation of tolerances.
- Figure 12-5 illustrates this.
Advantages of profile

A

PROFILE TOLERANCING

C

0.4

A

B

D

R

2X φ 4.8

4.6

φ 0.1 M

A

B

C

20

D

Radius
Tolerance zone 0.4 wide uniform boundary

Datum sequence specified

No tolerance accumulation

B

COORDINATE TOLERANCING

R

10.2

9.8

2X φ 4.8

4.6

30.1

29.9

10.1

9.9

Radius
Tolerance zone shape unclear

Must assume a datum sequence

Hole tolerances accumulate and may affect radius location

FIGURE 12-5 Advantages of Profile
A geometric tolerance that limits the amount of error a surface can have relative to its true profile.

Applications include controlling size, location, orientation, and form for:

- Planar, curved or irregular surfaces
- Polygons
- Cylinders, surfaces of revolution, or cones
- Coplanar surfaces
Figure 12-6 is an example of this type of use for a profile control.

- Profile control is applied to a true profile.
- True profile is related to the datums referenced with basic dimensions.
- Tolerance zone is a uniform boundary centered around the true profile.
- All elements of the surface must be within the tolerance zone simultaneously.
- Tolerance zone limits location, orientation, and form of the surface.
Profile & Tolerance Of A Surface Location

FIGURE 12-6 Profile Used to Tolerance a Surface Location
Profile and Polygons
In Figure 12-8, the profile controls the form and orientation of a conical feature. In this application, the following conditions apply:

- The profile callout is applied to a true profile. (The basic angle establishes the true profile.)
- The true profile is related to the datums referenced with basic dimensions. (An implied basic angle equal to half the specified angle exists.)
- The profile tolerance zone applies all around the cone. (When profile is applied to a surface of revolution, it automatically applies all around.)
- The tolerance zone is a uniform boundary centered around the true profile. (The equal bilateral default is in effect.)
- All elements of the surface must be within the profile tolerance zone. (The profile tolerance zone floats within the size tolerance zone.)
- The profile tolerance zone limits the orientation and form of the conical surface. (The size of the cone is limited by the tolerated dimension.)
If a basic dimension was used to size the cone, the profile would limit the size of the cone.
A profile control may be used when it is intended to treat two or more coplanar surfaces as a single surface. In this type of application, the profile control is a form control and does not use datum references; it simulates a flatness control. When profile is used as a form control, the tolerance zone is unilateral (away from the implied self-datum).

When profile of a surface (as a form control) is applied to coplanar surfaces, it controls the form of the surfaces as if they were a single surface. Figure 12-9 shows an example of profile applied to coplanar surfaces. In this application, the following conditions apply:

- The profile callout is applied to a true profile. (An implied basic zero between the surfaces establishes the true profile.)
- The number of surfaces being controlled is designated next to the profile callout.
- The tolerance zone is a unilateral boundary extending away from the implied datum. (The unilateral tolerance zone is automatic with an implied self-datum.)
- All elements of the surfaces to which the tolerance applies must be within the profile tolerance zone.
- The profile tolerance zone limits the form and coplanarity of the surfaces.
Profile and Coplanar Surfaces
In an application where there are several coplanar surfaces, it may be desirable to use two surfaces to create a single datum plane and tolerance the remaining surfaces in reference to that datum plane. Figure 12-10 shows an example of an application of multiple coplanar surfaces tolerated with profile.

- The profile callout is applied to a true profile. (An implied basic zero between the surfaces establishes the true profile.)
- The number of surfaces being controlled is designated next to each profile callout.
- The tolerance zone for the profile control of the outer two surfaces is a unilateral boundary extending away from the implied datum. (The unilateral tolerance zone is automatic with a self implied datum.)
- The tolerance zone for the profile control of the inner two surfaces is a bilateral boundary equally centered about the true profile.
- All elements of the surfaces (which each profile control applies to) must be within the profile tolerance zone.
- The profile control on the outer surfaces limits the form and coplanarity of the surfaces.
- The profile control on the inner surfaces limits the location, orientation, and form of the surfaces.
Profile and Multiple Coplanar Surfaces

FIGURE 12-10 Profile Used to Tolerance Multiple Coplanar Surfaces
Sometimes the form, size, orientation, and location tolerances need to be at different levels. Multiple single-segment controls are then used. We have seen them before.

If a surface is toleranced differently relative to different datums, multiple controls must be used.
Multiple Single-Segment Controls

- Controls location
- Controls orientation
- Controls size & form

Applied to true profile
Multiple Single-Segment Controls

0.2 zone controls size and form; this zone is not related to any datums and floats with the 0.4 zone.

0.4 zone refines the orientation relative to datum A; this zone floats within the 1.0 zone.

1.0 zone controls location and orientation relative to datums A, B, & C.

Datum plane C

Datum plane A

Datum plane B
Legal Specification For Profile

- Figure 12-12 is the flowchart for use.
Figure 12-6 illustrates one way to inspect surface profile. There are several others.

- CMM
- Optical
- Surface profilometer
Inspecting Surface Profile

**FIGURE 12-13 Inspecting Profile of a Surface**
Profile Of A Line

- Similar to profile of a surface except tolerance zone is two dimensional for a line whereas it is three dimensional for a surface.
- Profile of a line control is a geometric tolerance that limits the amount of error for line elements relative to their true profile.
- Profile of a line controls in one direction. May be used with multiple single-segment controls or with profile of surface control.
Profile of Line Example

- The profile callouts are applied to a true profile.
- The profile of a surface control limits the size, location, and orientation relative to the datums specified.
- The profile of a line control refines the form of the line elements.
- The profile of a line control refines the orientation of the line elements in one direction. (Note that the profile of a line tolerance value is less than the profile of a surface tolerance value.)
Profile of Line Example

FIGURE 12-14 Profile of a Line Used in a Multiple Single-Segment Profile Control
Profile of Line and a CT to Control Form and Location

- The profile callout is applied to a true profile.
- The coordinate tolerance locates the surface.
- The profile of a line control refines the form and orientation of the line elements in one direction. (Note that the profile of a line tolerance value is less than the coordinate tolerance value.)
Profile of Line and a CT to Control Form and Location

**FIGURE 12-15** Profile of a Line Used with a Coordinate Tolerance
Legal Specification For Profile of a Line

Figure 12-12 is the flowchart from which one can answer the question of whether or not one has used the profile of a line control correctly.
Inspecting Profile of a Line

Datum plane B

Profile template of large size of profile tolerance zone

Measure the height of the part

Datum plane A

Use a gage wire to measure gap between template and part line element

FIGURE 12-16 Inspecting Profile of a Line
More Tolerance Stacks

1. Label the start and end points of the distance to be calculated. On the start point of the calculation, draw a double-ended arrow. Label the arrow that points towards the end point of the calculation as positive (+). Label the other arrow as negative (-). Each time a distance that is in the direction of the positive arrow is used in the calculation, the distance will be a positive value. When a distance is used in the negative direction, it will be a negative value.

2. Establish a loop of part dimensions or gage distances from the start point to the end point of the calculation.

3. Calculate the answer.

When solving for a min. distance, half the profile tolerance value (for a bilateral-equal distribution control) is subtracted from the calculation. When solving for the max. distance, half the profile tolerance value is added to the calculation.
More Tolerance Stacks

FIGURE 12-17 Part Calculations Involving Profile

MAXIMUM DISTANCE X

1. + 0.3  ½ TOL
2. - 12.0 BASIC DIM TO B
3. + 18.0 FROM B TO SLOT
4. + 0.5  ½ TOL

MINIMUM DISTANCE X

1. - 0.3  ½ TOL
2. - 12.0 BASIC DIM TO B
3. + 18.0 FROM B TO SLOT
4. - 0.5  ½ TOL

MAXIMUM = 6.8
MINIMUM = 5.2
Summary

A summarization of profile control information is shown in Figure 12-18.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Datum reference required</th>
<th>Can be applied to a Surface</th>
<th>Can use M or L modifier</th>
<th>Can be applied at RFS</th>
<th>Override Rule #1</th>
<th>Can use bonus tolerance concepts</th>
<th>Can use datum shift concepts</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Yes*</td>
<td>Yes</td>
<td>No</td>
<td>Yes**</td>
<td>No*</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td></td>
<td>Yes*</td>
<td>Yes</td>
<td>No</td>
<td>Yes**</td>
<td>No*</td>
<td>No</td>
<td>Yes</td>
</tr>
</tbody>
</table>

* Can be used with or without a datum reference
** Is automatic per Rule #2
* Must be applied to a true profile therefore Rule #1 doesn't apply
* These modifiers may be used in the datum portion of the feature control frame

FIGURE 12-18 Summarization of Profile Controls
Questions?
Next Week

Total Semester Review. Bring Questions. I am going to highlight important principles but you need to ask the questions.