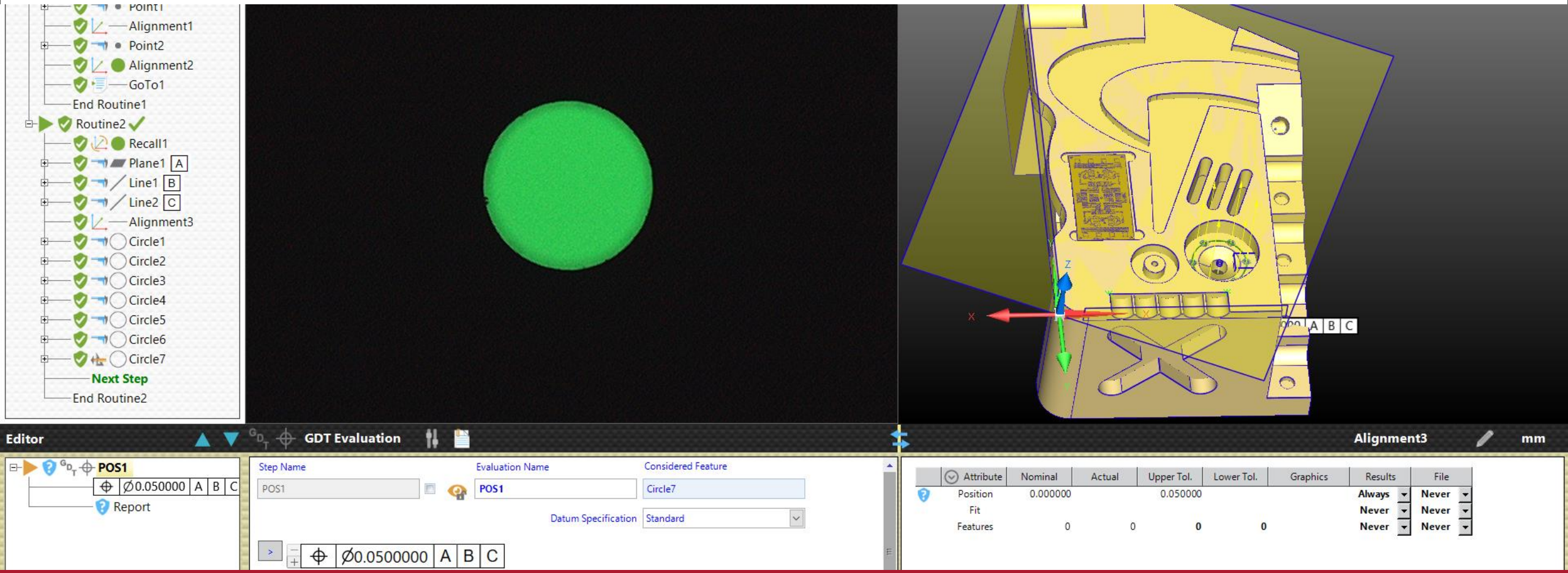
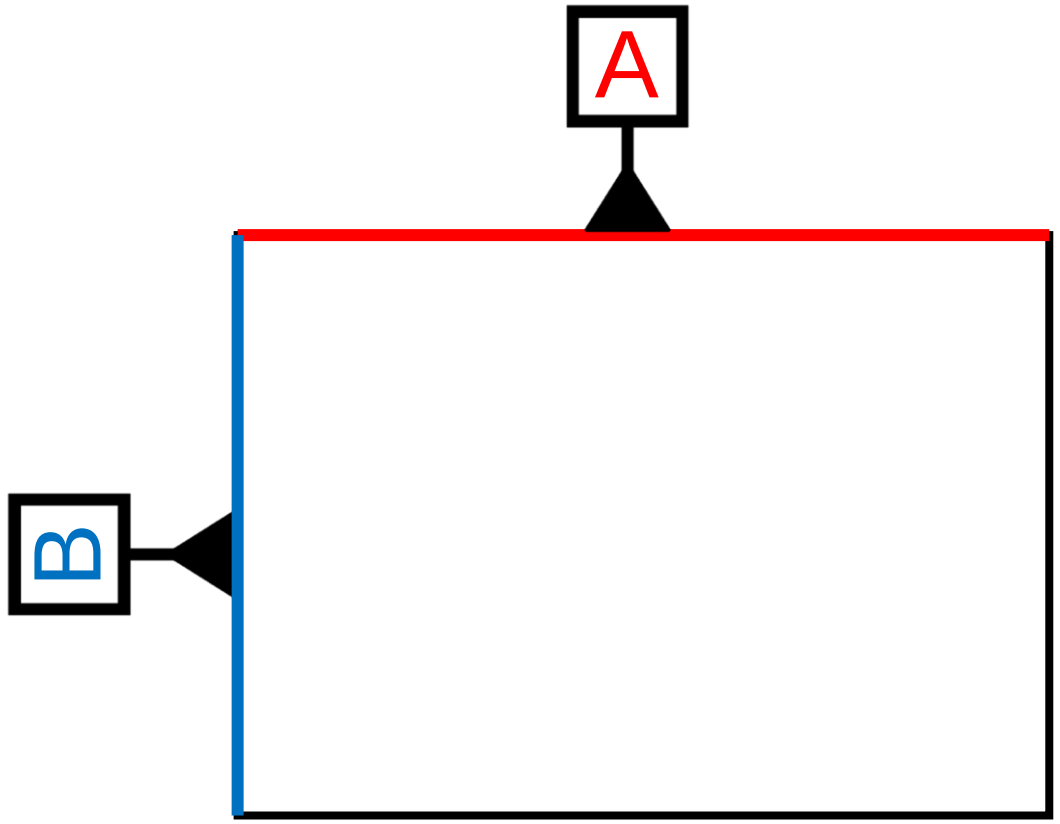
# GD&T in ZONE3

## Reference Slides

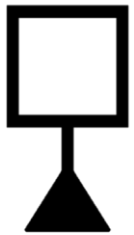


# Datums

- Theoretical entities that may be referenced when measuring features
- Established from selected features on the part
- May be used with all GD&T characteristic types except form




**Datum symbol:**

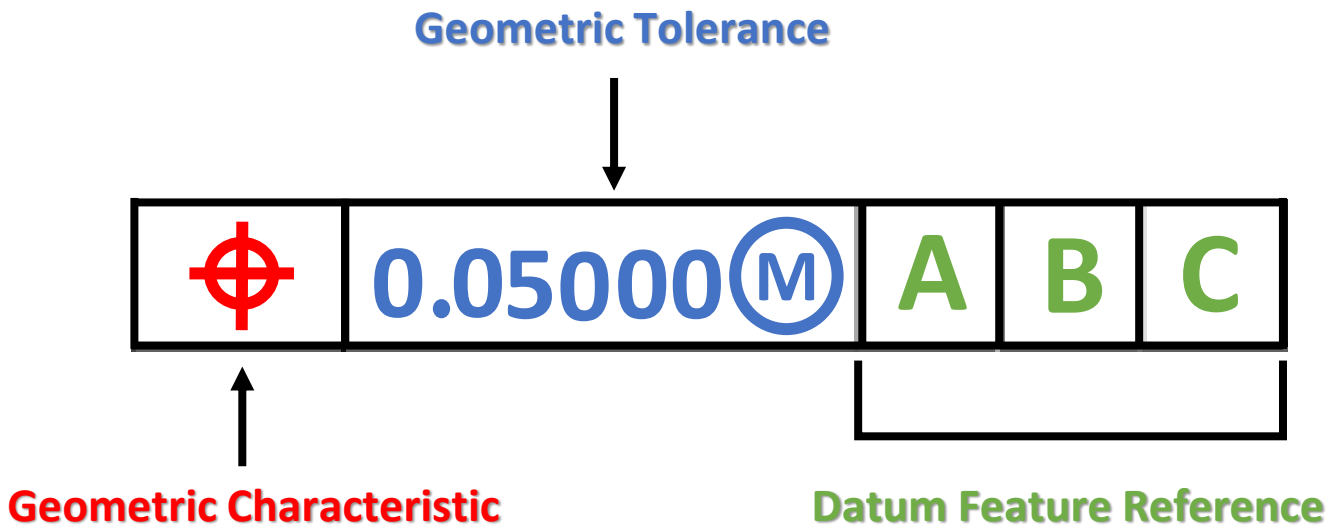


# Feature Control Frame

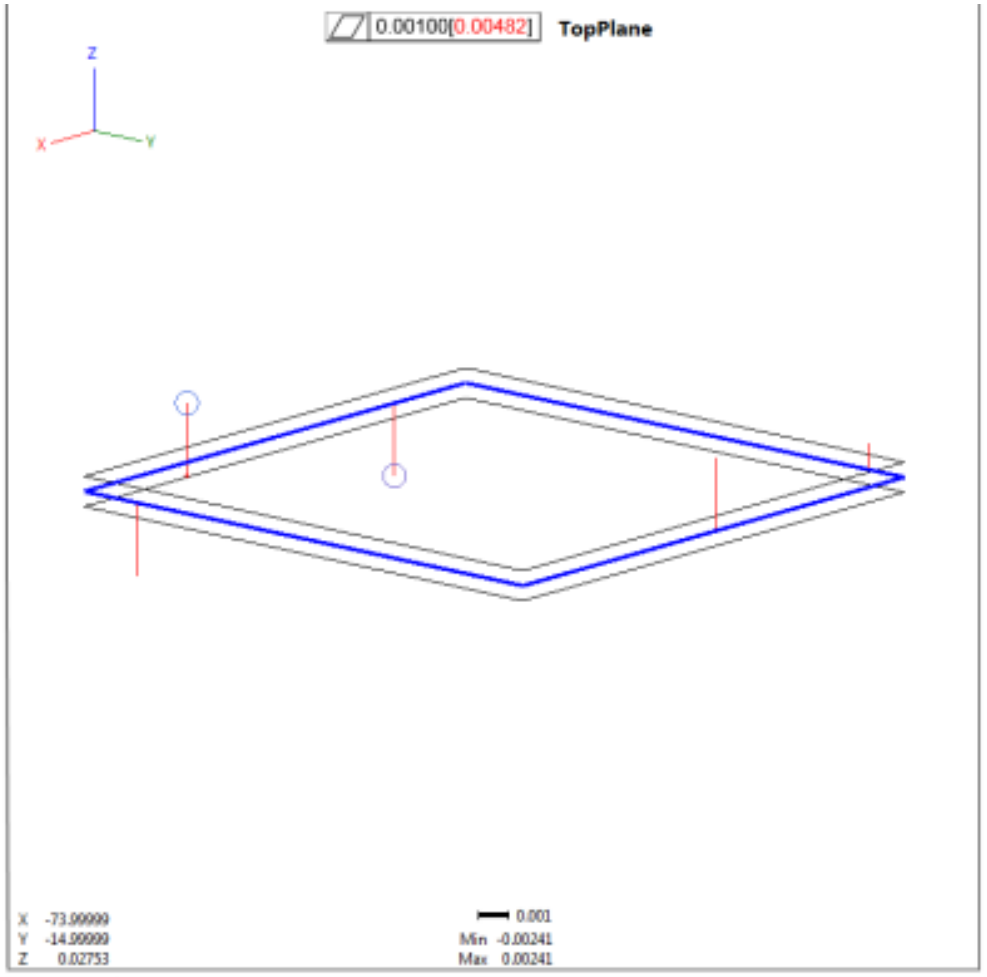
- Indicates a geometric tolerance for an individual feature

- Has three components:
  1. Geometric characteristic
  2. Geometric tolerance
  3. Datum feature reference (optional depending on characteristic)

- May include material conditions



# Form Plots




- Shows:
  - Orientation of the feature relative to the current alignment
  - Tolerance zone relative to the nominal feature
  - Deviation whiskers for measured points
  - Maximum and minimum deviations
- Available for:
  - Circularity
  - Cylindricity
  - Flatness
  - Profile of a line
  - Roughness
  - Straightness
- Clicking + in reporting sub-step of GD&T step will display form plot within step and single-run report
- Can be displayed in custom reports

# GD&T: Key Terms

**Bonus Tolerance:** When a maximum material modifier is applied, the extra tolerance that is available because of the modifier. It is calculated as the difference between the actual feature size and the MMC of the feature. In ZONE3, if an MMC has been applied in a GD&T step, the bonus tolerance value will be shown under the Features attribute in the report sub-step.

**Datum:** A referenced theoretical feature that is established from measured features on a part. Datums may be used with all GD&T characteristic types except form and can be assigned in ZONE3 in the feature editor window of a measured or constructed feature.

**Evaluate Tab:** Ribbon group that contains evaluation steps, including all the available GD&T characteristic steps. The number of available evaluation steps depends on the current level of ZONE3 (express, prime, or pro).

**Feature Control Frame:** A rectangular box that indicates geometric tolerance for an individual part feature and is divided into left, middle, and right compartments that hold: the symbol of the geometric characteristic to be evaluated, the shape and size of the tolerance zone, and optionally, the datum reference features.

**Form Plot:** A plot that shows the orientation of the feature relative to the current alignment, the tolerance zone relative to the nominal feature, deviation whiskers for measured points, the magnification of the tolerance zone, and the maximum and minimum deviations. Form plots are available for circularity, cylindricity, flatness, profile of a line, roughness, and straightness and can be displayed by selecting the + in the report sub-step of the GD&T step.

**Form Tolerance:** Defines the allowable variation in the shape of a feature, specifically its circularity, cylindricity, flatness, or straightness.

**Geometric Characteristic:** Specific element of a feature's geometry that is used to describe how a part varies from its intended design. This can be chosen from the Evaluate tab in ZONE3.

**Geometric Tolerance:** Specifies the maximum amount a part's geometry can vary from its intended design. This can be set in the feature control frame within a GD&T step in ZONE3.

# GD&T: Key Terms

**GD&T:** Geometric dimensioning and tolerancing; sets tolerance zones for geometric variation using standard language and symbols.

**GD&T Standard:** A group of standards for GD&T. These are published by the International Organization of Standardization (ISO) and the American Society of Mechanical Engineers (ASME). ZONE3 conforms to multiple standards; the current standard in ZONE3 can be modified in the System properties or Project properties.

**Location Tolerance:** Defines the allowable variation in the location or placement of a feature, specifically its concentricity, position, or symmetry.

**Material Condition Modifier (MMC):** Sets the material condition for application of the tolerance for features of size. The modifier options include Maximum Material Condition, Least Material Condition, and Regardless of Feature Size. You can choose a modifier in ZONE3 by clicking on the empty circle next to the geometric tolerance.

**Orientation Tolerance:** Defines the allowable variation in the direction of a feature, specifically its angularity, parallelism, or perpendicularity.

**Profile Tolerance:** Simultaneously defines the allowable variations in the size, location, orientation, and form of the line profile or surface profile of a feature.

**Runout Tolerance:** Simultaneously defines the allowable variations in the form, location, and orientation of cylindrical parts, referred to the parts' circular runout and total runout.