**Bonus Tolerance Calculation Formula**

The bonus tolerance formula is:

Bonus Tolerance=(Actual Feature Size−Basic Size)+Feature ToleranceBonus Tolerance=(Actual Feature Size−Basic Size)+Feature Tolerance

If the actual feature size exceeds the basic size, a "bonus" in tolerance is applied, effectively giving more tolerance to the feature. If the actual size is at or below the basic size, only the feature tolerance is considered.

**Example Calculation**

For example, if:

* The **Basic Size** is 50 mm,
* The **Feature Tolerance** is 0.5 mm, and
* The **Actual Feature Size** is 51 mm,

The bonus tolerance would be:

Bonus Tolerance=(51−50)+0.5=1.5 mmBonus Tolerance=(51−50)+0.5=1.5 mm

**Importance and Usage Scenarios**

Bonus tolerance is particularly useful in manufacturing when precision parts must fit together. By allowing additional tolerance if the size deviates in a favorable direction, it reduces the risk of part rejection and helps keep production costs under control. This added tolerance is critical when dealing with mating parts in assemblies, as it can significantly reduce the number of parts that must be scrapped or reworked.

1. **What is GD&T?**
   * Geometric Dimensioning and Tolerancing (GD&T) is a symbolic language used in engineering drawings to define the geometry and tolerances of parts.
2. **Can bonus tolerance be negative?**
   * No, bonus tolerance is not negative. If the actual feature size is less than or equal to the basic size, the bonus tolerance is zero, and only the feature tolerance applies.