

## Tolerance and Fit Systems

**Technical Drawing / Terms:** If producer and quality supervisor can speak the same language, the products will be perfect.

**Tolerance Definition:** Error margins of workpieces in standard dimensions in the meaning of tolerance (indulgence).

Tolerance is the error margins within the lower and upper limit specified by taking the main dimensions as a reference. A standardization has been done around the world and certain standards have been determined due to negative situations such as difference of the products in the past, rejects and economic / labourship losses.

Selecting small tolerances in the production method improve the quality but increases the cost. Selecting large tolerances decreases both costs and quality. Therefore, tolerance values in the Machine and Die / Mould production has been standardized.

We pay attention to products that do not have different features. This system is important for product replacements and repairs. Guvenal - GTH branded Die & Mould Components are designed according to DIN / ISO norms. The different standard types of other local producers are to create problems and difficulties in the production phase. These dies and moulds shipped to other countries will create the same problems and difficulties at the destination. For this reason, we recommend you to prefer Guvenal - GTH brand products for your dies and moulds, especially systems used as sets.

**There are two types of tolerance depending on the type of writing:**

**1- Numerical Tolerance:**

Written nearby principal dimensions (basic dimension).

**Example:**  $40^{+0.05}_{-0.01}$      $40_{\pm 0.05}$

**2- Letter Tolerance:**

The letters and numbers which classified according to DIN and ISO systems are written to right hand side of the principal dimension and read according to type of micron.

**Example:**  $40^{H7}$      $40_{h6}$

**There are two types of tolerance according to their practice:**

**1- Manufacturing Tolerance:** It is based on the difference between the largest dimension and the smallest dimension.

**2- Fit Tolerance:** It is made according to shaft and hole system that indicated by DIN and ISO system. Beside, it is the relation due to difference between dimensions of two simple components (hole and shaft) to be joined each other.

**Recommendation:**

It is easy to machining external surfaces of workpieces. Since it will be hard to machining internal surfaces, hole dimensions should be kept fixed and a precise machining is performed by giving tolerance to the shaft.

**Comparing of DIN and ISO Systems**

- Both systems have been classified according to normal hole and normal shaft system.
- Normal temperature has been accepted as 20°C in both systems.
- Tolerance differences of shafts and holes, often tolerance comparement and keeping the basis for tolerance of difference to a zero base line is the same in both.
- At DIN System there are 4 types of fit which are Precise, Fine, Medium and Roughing. At ISO System there are 18 kind different quality.
- At DIN System the product diameter were found to be arithmetic average, at ISO System it is found by diameter's geometric average.

