PRODUCTION DIMENSIONING

In addition to a complete *shape description* of an object, a drawing must also give a complete *size description*, that is, it must be *dimensioned.* For dimensions, the international system of units (SI), is based on the *meter*. France adopted the *meter* in 1791, but in America the *decimal-inch system* persists [1 meter = 39.37" (inch); 1" = 25.4 mm]. A tremendous effort is now under way to convert all standards of the American National Standards Institute (ANSI) to the SI units in conformity with the International Standards Organization (ISO) standards.

The need for *interchangeability* of parts is the basis for the development of modern methods of size description. Drawings today must be dimensioned so that production personnel in widely separated places can make mating parts that will fit properly when brought together for final assembly or when used as repair or replacement parts by the customer.

The drawing should show the object in its completed condition and should contain all necessary information to bring it to that final state. Therefore, in dimensioning a drawing, the designer and the drafter should keep in mind the finished piece, the production processes required, and above all *the function of the part in the total assembly.*

6.1 GENERAL PRINCIPLES

Dimensions are given in form of *linear distances, angles,* or *notes*. The student often makes the mistake of giving the dimensions that are used *to make the drawing*. These are not necessarily the dimensions required. The ability to dimension properly requires the following:

- *every feature on the object must be dimensioned for:*

1. **size** (width or length, height and depth)
2. **form** (Φ; R; $\triangleleft$1:k – taper; M - thread; 2x45^o - chamfer);
3. **location** – dimensions between:
   - $\Rightarrow$ **two axes**
   - $\Rightarrow$ **two datum planes**
   - $\Rightarrow$ **an axis and a datum plane**

- *no surface, line or point should be located by more then one dimension in any given direction;*
- *dimensions should not be repeated;*
- ***there is a tolerance for every dimension:***
   - $\Rightarrow$ ***size tolerance;***
   - $\Rightarrow$ ***geometric tolerance;***
- *select three reference or datum planes before beginning the dimensioning; they should be **finished surfaces** or center planes;*
- *do not dimension to invisible (hidden) lines.*

6.2 FINISHED SURFACE

Finished surface is a machined surface and is used to indicate such surface, by a *finish mark*. The basic symbol (Fig. 6.1) is a large capital "V", with the right leg extended. The finish mark is shown only on the edge view (the trac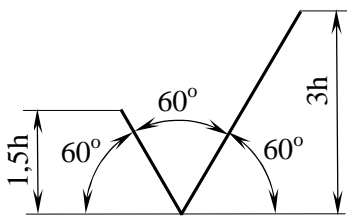e) of a finished surface. Surface finish is intimately related to the functioning of a surface and proper specification of finish of such surface as bearings and seals is necessary. There are two principal reasons for surface finish control: to reduce *friction* and to control *wear*. Surface-quality specifications should be used only where needed, since the cost of producing a finished surface becomes greater as the quality of the surface called for is increased. Generally, the ideal surface finish is the roughest one that will do the job satisfactorily.

Fig. 6.1

Roughness may be defined as the closely spaced surface irregularities produced by machining or grinding operations. Roughness height is the average deviation from the mean plane of the surface and is specified in *micromilimeters*. Its values are according to the various production methods and are shown in table 6.1 (preferred series):

Table 6.1

Process	Roughness average, $R_a (\mu m)$													
	100	50	25	12,5	6,3	3,2	1,6	0,80	0,40	0,20	0,10	0,05	0,025	0,012
Sand casting		░	▓	░										
Perm. mold casting					░	▓	░							
Die casting						░	▓	░						
Forging			░	▓	▓	░								
Flame cutting		░	▓	░										
Sawing			░	▓	▓	░								
Milling			░	▓	▓	▓	▓	░						
Drilling				░	▓	▓	░							
Broaching					░	▓	▓	░						
Reaming					░	▓	▓	░						
Boring			░	▓	▓	▓	▓	▓	░					
Grinding					░	▓	▓	▓	▓	▓	░			
Electrolytic grinding								░	▓	░				
Honing							░	▓	▓	▓	░			
Polishing								░	▓	▓	░			
Super finishing								░	▓	▓	▓	░		

Note that the roughness symbol is read from bottom and/or right side of the drawing. Some examples of the way to put roughness symbol in drawings are shown in Fig. 6.2 – a). – all surfaces and b), not all surfaces have finish marks.
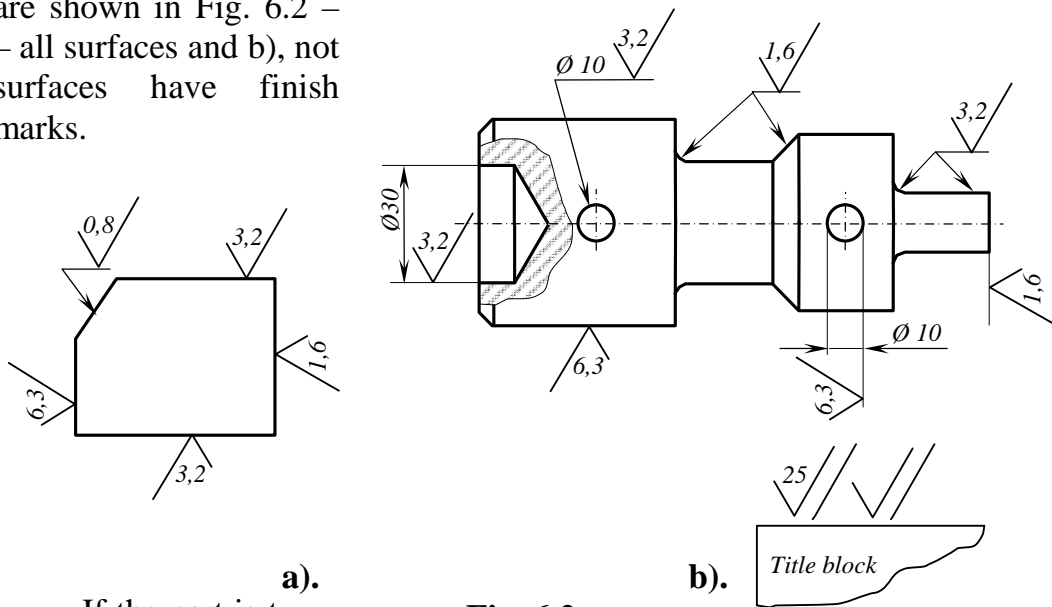

a).

b).

Fig. 6.2

If the part is to finished at different roughness values, but some surfaces are finished to the same roughness, the finish marks should be omitted on the drawing and used over the title block (Fig. 6.2 – b.). The other values of roughness must be put in brackets, or only a blind finish mark, to suggest that other values of surfaces roughness are on the drawing.

6.3 TOLERANCES

I n practice, no **two parts can be made exactly alike**. The objective in interchangeable manufacture is to produce all parts in a given tolerance. Parts can be made to very close tolerances, but manufacturing to unnecessary close tolerances is always costly and should be avoided.

By point of view of tolerances, there are two types of dimensions: **SHAFT** (exterior dimension) and **HOLE** (interior dimension).

6.3.1 SIZE TOLERANCE: it's the maximum permitted variation of a dimension from the perfect size.

Terminology – *Basic size* (result of the projecting calculus – they are selected from a table of preferred sizes) – is a theoretical dimension;

- *Real permitted size* (result of manufacturing);

- *Deviation* (difference between basic and real permitted size); there is an *upper* and a *lower* deviation;

- *Fundamental deviation* (the closest deviation to the basic size) – indicated by cases – *lowercases for shafts* and *uppercases for holes*;

- *Tolerance* or *tolerance zone* (difference between upper and lower deviation);

- *International tolerance grade* – they are 18 IT: 01, 0, 1, 2,… 16, but usually from 4 to 11; there are tables with numerical values for UD(ud) or LD(ld), according to basic sizes.

Examples of size tolerances:

- *tolerance for a hole* - *50H8,* or $\mathbf{\mathit{50H8}}\begin{pmatrix} +0.039 \\ 0 \end{pmatrix}$, or $\mathbf{\mathit{50}}\begin{pmatrix} +0.039 \\ 0 \end{pmatrix}$:

50 - basic size;
H8 – tolerance grade (field);
H – fundamental deviation for a hole (uppercase);
8 – international tolerance grade;
0.0 39 – upper deviation;
0 – lower deviation;

- tolerance for a shaft - **50f7,** or $\boldsymbol{50f7}\left(\begin{array}{c}-0.025\\-0.050\end{array}\right)$, or $\boldsymbol{50}\left(\begin{array}{c}-0.025\\-0.050\end{array}\right)$:

50 *- basic size;*
f7 *– tolerance grade (field);*
f *– fundamental deviation for a shaft (lowercase);*
7 *– international tolerance grade;*
0.025 *– upper deviation;*
0.050 *– lower deviation;*

Fig. 6.3 shows the **tolerance grades for holes,** upper and lower deviation, and fundamental deviation in every case too:
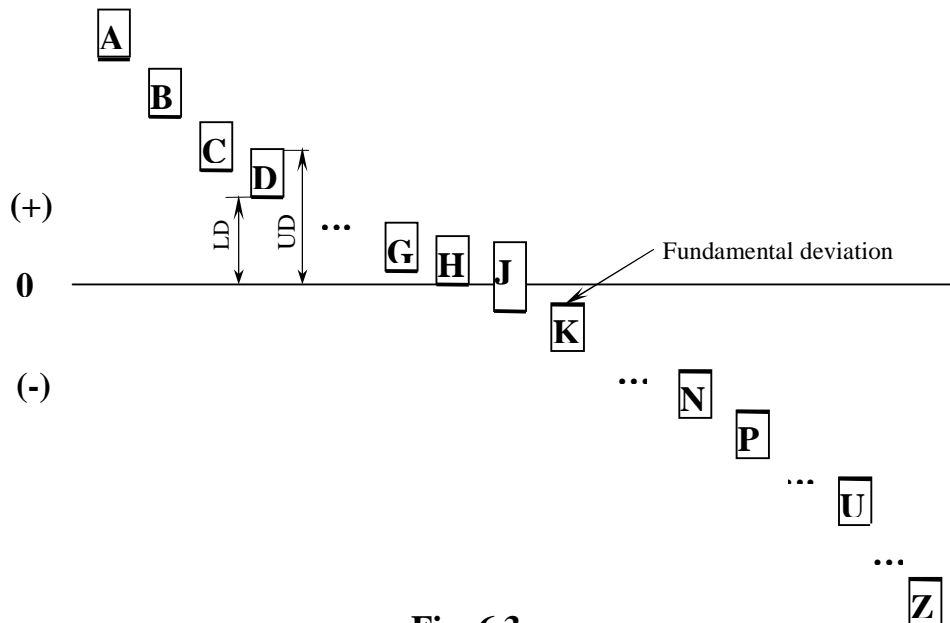
Fig. 6.3

Fig. 6.4 shows the **tolerance grades for shafts,** upper and lower deviation, and fundamental deviation in every case too:
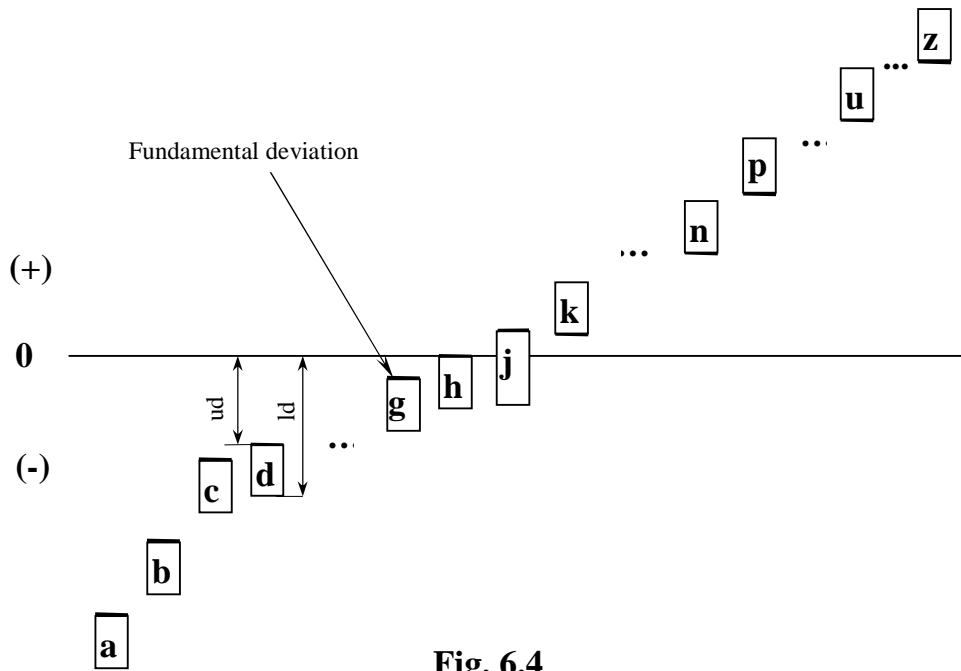
Fundamental deviation

(+)

0

(-)

...

Fig. 6.4

6.3.2   GEOMETRIC TOLERANCE: it's the maximum permitted variation of a form or its position from the perfect geometry (table 6.2 and Fig. 6.5).

Table 6.2 – *Geometric tolerance*

		Type of tolerance	Characteristic	Symbol
For individual features		**Form**	Straightness Flatness Circularity (roundness) Cylindricity	c e g
	c 0.08	**Profile**	Profile of a line Profile of a surface	k d
For related features		**Orientation**	Angularity Perpendicularity Parallelism	´ b f
		Location	Position Concentricity	j r
		Runout	Circular runout Total runout	h hh

b 0.05 A e rules and it's a skill to know **how to use tolerances in a drawing.** For individual features are necessarily two boxes and for related features, three boxes, to inscribe the symbol, the tolerance value and the *datum plane.*
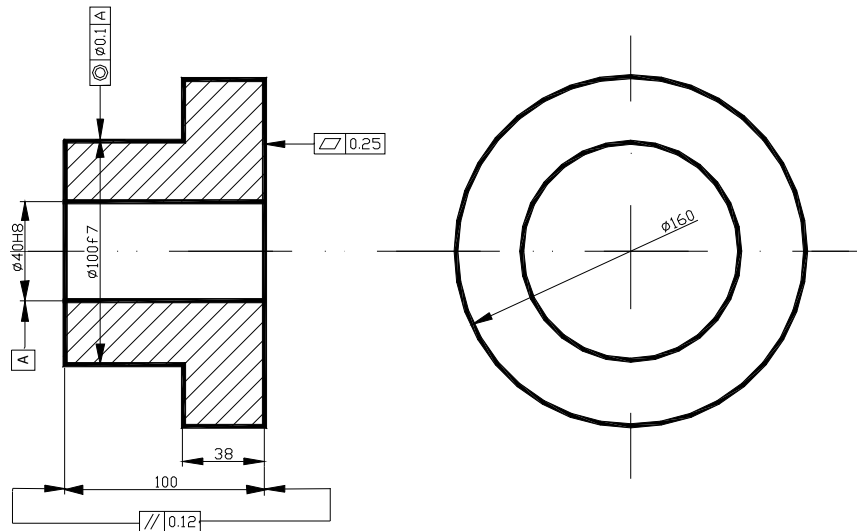
Fig. 6.5

6.4 PRODUCTION DIMENSIONING IN ASSEMBLY DRAWING.

Even the simplest machines have several parts, it is necessary for purposes of both design and production, to know how the various parts fit together. Drawings made to show these relationships are called *Assembly Drawings*. As the name clearly implies, an assembly drawing shows the parts of a machine put together, in their proper working position.

The working parts of any machine have some definite relationship to their *mating parts*, in order to achieve a particular function, such as:

- o Free rotation;
- o Free longitudinal movement;
- o Clamping action;
- o Permanent fixed position.

Since it is not possible to work to absolute sizes, it's necessarily, where interchangeable assembly is required, to give the dimensions of mating parts with *limits*.

There are two cases, when two mating parts, a *shaft* and a *hole,* are putted together:

- The mating surfaces are defined by different basic sizes (Fig. 6.8);
- The mating surfaces are defined by *the same basic size,* which means there is a *FIT* (Fig. 6.9).

Fit is the general term used to signify the range of tightness which may result from the application of a specific combination of allowances and tolerances, in the design of mating parts. There is an infinite number of fits that

can be obtained, for a nominal size dimension. That is the reason to reduce this number, using two *basic systems*:

- **Basic Hole System** – is a system of fits in which the allowance is applied only to the *shaft* and the **hole is in the basic size**, that means (Fig. 6.6), the minimum size of the hole is taken as a base from which all variations are made (allowance 0);
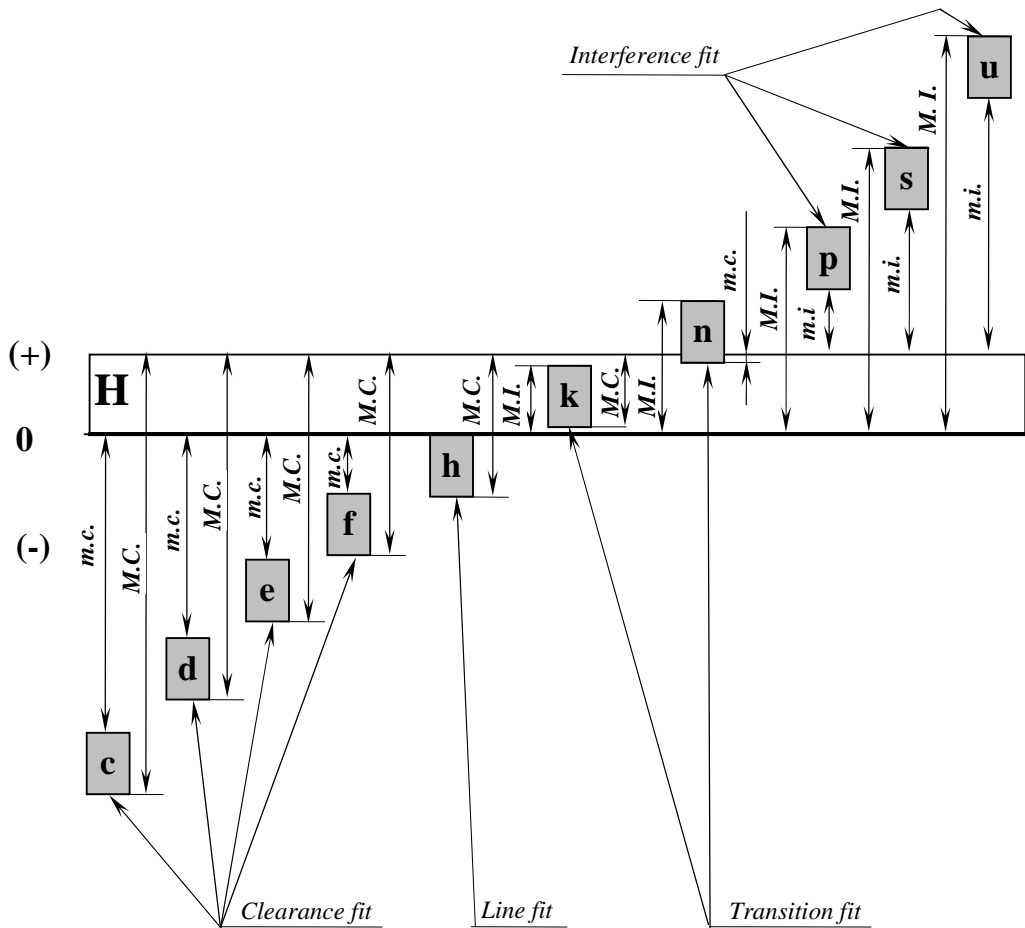
Fig. 6.6

- **Basic Shaft System** – is a system of fits in which the allowance is applied only to the *hole* and the **shaft is in the basic size**, that means (Fig. 6.7), the maximum size of the shaft is taken as a base from which all variations are made (allowance 0).
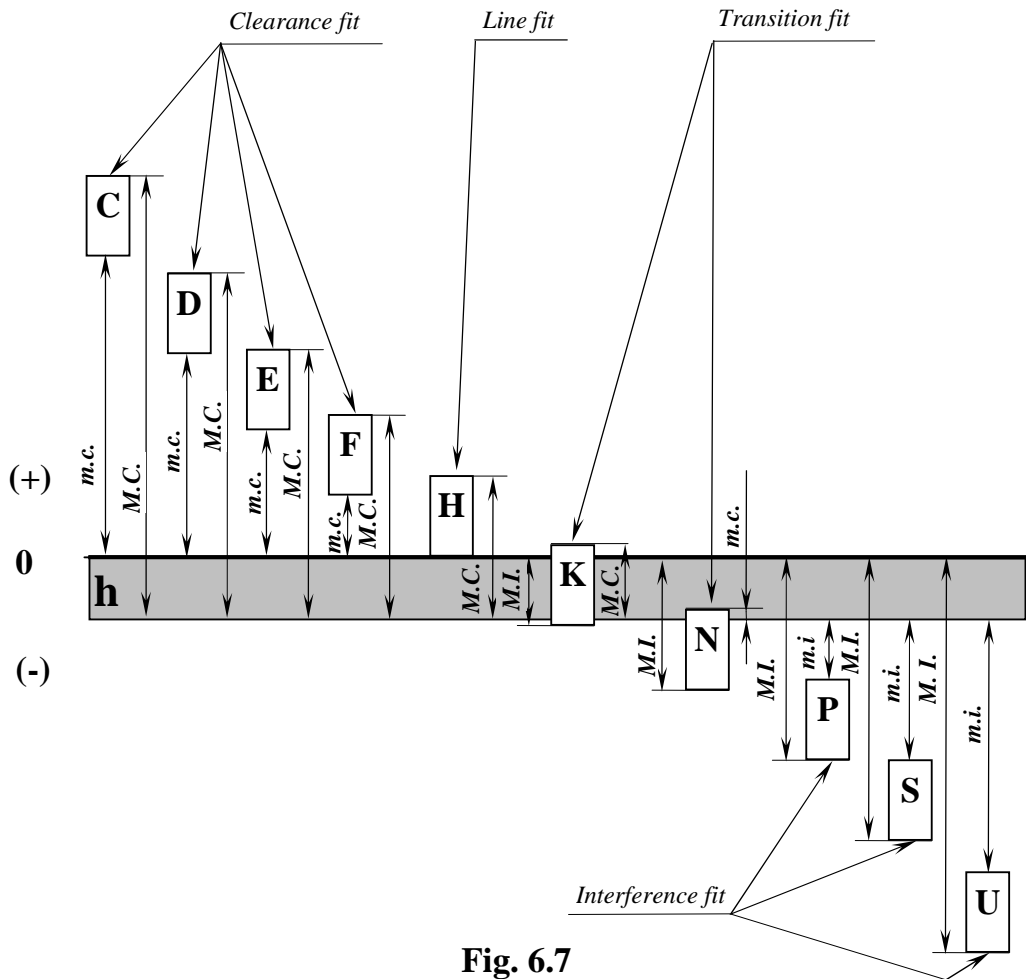
Clearance fit   Line fit   Transition fit

C

D

E

F

(+)   *m.c.*   *M.C.*   *m.c.*   *M.C.*   *m.c.*   *M.C.*   *m.c.*   *M.C.*

H

0

h   *M.C.*   *M.I.*   K   *M.C.*   *m.c.*

(-)

M.I.   N   *M.I.*   *m.i*   *M.I.*   *m.i.*   *M. I.*

P

S   *m.i.*

Interference fit

U

Fig. 6.7

There are four general types of fits:

⇒ ***Clearance fit*** – is one having limits of size so prescribed that a clearance always results (Maximum or minimum), when mating parts are assembled;

⇒ ***Interference*** fit - is one having limits of size so prescribed that an interference always results (Maximum or minimum), when mating parts are assembled;

⇒ ***Transition fit*** - is one having limits of size so prescribed that either a small clearance or a small interference may results, when mating parts are assembled;

⇒ ***Line fit*** - is one having limits of size so prescribed that a clearance or surface contact may results, when mating parts are assembled.

The difference between two mating surfaces which are defined by different basic sizes (Fig. 6.8) or defined by ***the same basic size*** (Fig. 6.9), in drawings is the way to dimension and to represent every surface extreme generatrix: in the first case every surface has its own dimension and generatrix

and in the second, both surfaces have the same dimension with the explain fit and the same generatrix.
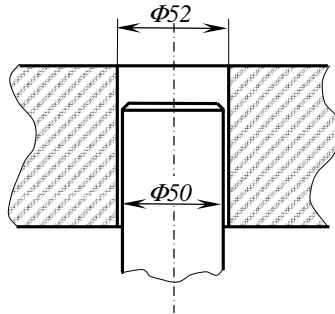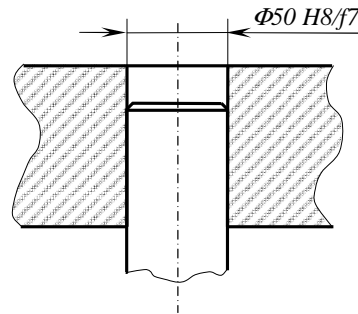

Fig. 6.8   **Fig. 6.9**

6.5 APPLICATIONS:

1. Represent using ***Basic Hole System*** and a scale of 1000:1 the fits that result between the size tolerance of a hole with $H8\left(\begin{smallmatrix}+0,054\\0\end{smallmatrix}\right)$, and the shafts with: $f7\left(\begin{smallmatrix}-0.036\\-0.071\end{smallmatrix}\right)$, $g7\left(\begin{smallmatrix}-0.012\\-0.047\end{smallmatrix}\right)$, $h7\left(\begin{smallmatrix}0\\-0.035\end{smallmatrix}\right)$, $k7\left(\begin{smallmatrix}+0.025\\-0.010\end{smallmatrix}\right)$, $n7\left(\begin{smallmatrix}+0.045\\+0.010\end{smallmatrix}\right)$, $p7\left(\begin{smallmatrix}+0.059\\+0.024\end{smallmatrix}\right)$, $s7\left(\begin{smallmatrix}+0.093\\+0.058\end{smallmatrix}\right)$, determine the fits types and the values of Maximum/minimum clearance or interference.

2. Represent using ***Basic Shaft System*** and a scale of 1000:1 the fits that result between the size tolerance of a shaft with $h7\left(\begin{smallmatrix}0\\-0.035\end{smallmatrix}\right)$, and the holes with: $F8\left(\begin{smallmatrix}+0.090\\+0.036\end{smallmatrix}\right)$, $G8\left(\begin{smallmatrix}+0.066\\+0.012\end{smallmatrix}\right)$, $H8\left(\begin{smallmatrix}+0,054\\0\end{smallmatrix}\right)$, $K8\left(\begin{smallmatrix}+0.010\\-0.044\end{smallmatrix}\right)$, $N8\left(\begin{smallmatrix}-0.010\\-0.064\end{smallmatrix}\right)$, $P8\left(\begin{smallmatrix}-0.024\\-0.078\end{smallmatrix}\right)$, $S8\left(\begin{smallmatrix}-0.058\\-0.112\end{smallmatrix}\right)$, determine the fits types and the values of Maximum/minimum clearance or interference.