Technology of Machining and CAM Systems

učební text

Robert Čep
Marek Sadílek

Ostrava 2012
Recenze: xxx

Název: Technology of Machining and CAM Systems
Autor: Robert Čep, Marek Sadílek
Vydání: první, 2011
Počet stran: 121
Náklad:

Studijní materiály pro studijní obor Strojírenství Fakulty strojní
Jazyková korektura: nebyla provedena.

Určeno pro projekt:
Operační program **Vzdělávání pro konkurenceschopnost**
Název: Rozvoj jazykových kompetencí pracovníků VŠB-TU Ostrava: InterDV
Číslo: CZ.1.07/2.2.00/15.0132
Realizace: VŠB – Technická univerzita Ostrava
Projekt je spolufinancován z prostředků ESF a státního rozpočtu ČR

© Robert Čep, Marek Sadílek
© VŠB – Technická univerzita Ostrava

ISBN xx
CONTENT

1. BASICS OF METAL CUTTING ................................................................. 5
   1.1 GENERAL CONCEPTS AND TERMINOLOGY ........................................ 5
   1.2 CLASSIFICATION OF MACHINING METHODS ...................................... 5
   1.3 WORKPIECE .................................................................................. 7
   1.4 CUTTING TOOL ............................................................................ 8
   1.5 MOVEMENTS IN MACHINING (METAL CUTTING) ............................. 9

2. TECHNOLOGY OF METAL CUTTING (MACHINING) .............................. 12
   2.1 TURNING .................................................................................... 12
      2.1.1 Cutting conditions ................................................................. 13
      2.1.2 Turning forces .................................................................... 15
      2.1.3 Unit machining time ............................................................. 16
      2.1.4 Basic works on lathe ............................................................. 17
      2.1.5 Cutting tools – turning tools .................................................. 19
      2.1.6 Machine tools – lathes .......................................................... 23
      2.1.7 Cutting tool clamping ........................................................... 29
   2.2 MILLING ...................................................................................... 29
      2.2.1 Cutting Conditions ................................................................. 33
      2.2.2 Milling Force and its Components ........................................... 35
      2.2.3 Unit Machining Time ............................................................. 36
      2.2.4 Cutting tools – Milling Cutters ............................................... 37
      2.2.5 Milling Machines ................................................................. 41
      2.2.6 Dividing Machines ............................................................... 45
      2.2.7 Tool Clamping ...................................................................... 47
      2.2.8 Workpiece Clamping ............................................................. 49
   2.3 DRILLING, CORE-DRILLING, REAMING, BORING ............................ 50
      2.3.1 Drilling ............................................................................... 50
      2.3.2 Calculation of Chip Cross-Section ......................................... 54
      2.3.3 Cutting Force and its Components ......................................... 55
      2.3.4 Unit Machining Time ............................................................. 56
      2.3.5 Tools - Drills ........................................................................ 57
      2.3.6 Core-Drilling ........................................................................ 66
      2.3.7 Reaming .............................................................................. 67
      2.3.8 Recessing ............................................................................. 70
      2.3.9 Drilling Machines ................................................................. 71
3. COMPUTER AIDED MACHINING – INTRODUCTION AND STRUCTURE OF PRODUCTION PROCEDURE .................................................................................. 74

3.1 PROGRAMMING OF NUMERICALLY CONTROLLED MACHINE TOOLS ......................... 76

4. WORKING PROCEDURE IN CAM SYSTEM .................................................................. 77

4.1 GEOMETRIC PART OF CAD / CAM SYSTEM .............................................................. 78
4.2 TECHNOLOGICAL PART OF CAD / CAM SYSTEM ..................................................... 78

5. IMPORTING GEOMETRICAL DATA INTO CAM SYSTEM ........................................... 81

5.1 LOCATION OF OBJECTS INTO THE APPROPRIATE POSITIONS ................................... 81

6. CREATING OF STOCK AND FIXTURES ...................................................................... 85

7. TURNING IN CAM SYSTEM ....................................................................................... 88

7.1 2 - AXIS TURNING .................................................................................................. 88
7.1.1 Basic menu of machining cycles for 2 – axis turning ............................................. 88
7.1.2 Basic setting data for individual machining cycles .................................................. 89
7.1.3 Straight (rectangular) turning ............................................................................... 89
7.1.4 Rough turning ...................................................................................................... 90
7.1.5 Profile turning ..................................................................................................... 90
7.1.6 Pocket turning .................................................................................................... 90
7.1.7 Groove turning ................................................................................................... 91
7.1.8 Thread turning ................................................................................................... 92
7.1.9 Hole machining .................................................................................................. 92
7.1.10 Parting off ......................................................................................................... 92
7.1.11 Residual turning ............................................................................................... 93

8. MILLING IN CAM SYSTEM ........................................................................................ 94

8.1 BASIC SETTING DATA FOR INDIVIDUAL MACHINING CYCLES ................................. 98
8.2 2.5-AXIS MILLING .................................................................................................. 99
8.2.1 Plane milling and face milling .............................................................................. 99
8.2.2 Roughing - 2.5 D milling .................................................................................... 100
8.2.3 Profile milling .................................................................................................... 101
8.2.4 Pocket milling ................................................................................................... 102
8.2.5 Circular milling – hole pocketing ........................................................................ 104
8.2.6 Thread milling ................................................................................................... 105
8.2.7 Groove milling (Slot cycles) ................................................................................ 105
8.2.8 Chamfer milling .................................................................................................. 106
8.2.9 Text milling ....................................................................................................... 107
8.2.10 Engraving milling ............................................................................................. 107
8.2.11 Manual Milling .................................................................................................. 108
8.2.12 2.5 D rest machining ................................................................. 108
8.2.13 Plunge machining ................................................................. 108

9. SIMULATION AND VERIFICATION .................................................. 110

10. POSTPROCESOR - POSTPROCESING ......................................... 113

11. CREATION OF ACCOMPANYING DOCUMENTATION .................. 114

12. CONCLUSION ............................................................................. 119

REFERENCES ............................................................................... 120
1. **BASICS OF METAL CUTTING**

Machining technology occupies a significant position in the mechanical engineering production. In many technological processes, it enables to create a product of the required shape, required dimensional accuracy and quality of the machined surfaces from a semi-product. This chapter introduces basic terms and definitions regarding the theory of the cutting process.

### 1.1 General concepts and terminology

**Machining** is a technological process in which the surfaces of a workpiece of the defined shape, size and quality are generated by removing particles of the material through the mechanical, electrical, chemical and other effects, eventually through their combinations.

**Cutting** is a machining process in which particles of the material in the shape of a chip are removed by means of the cutting tool edge.

Machining is carried out in the **machining system (SNOP)**. The system consists of the following components (only three parts missing the fixture are also considered):

- Machine tool (S);
- Cutting tool (N);
- Workpiece (O);
- Fixture (P).

The **workpiece** is an object of the machining process. It is a part or a component which is being machined or has been already machined. In terms of geometry, the workpiece is characterized by the dimensions and shapes of the respective surfaces.

The **allowance** is a layer of material occurring between the the workpiece surface being machined and machined workpiece surface which needs to be removed by machining.

**Removal layer** is a part of the allowance inclined to the surface of the cut which is cut by one cutting edge in the shape of a chip.

The **chip** is a cut-off and deformed removal layer of the workpiece material.

### 1.2 Classification of machining methods

Machining methods can be classified according to various criteria. Classification of the machining methods uses various characteristics, such as the mutual contact of the tool and the workpiece, combination of variants of the machine, tool and workpiece movements.

Depending on the nature of the performed work, machining methods can be divided into:

- manual;
- mechanical.
Manual machining represents the work performed by a human with hand tools, such as chopping, sawing, scraping, etc. It also includes works done by manually operated machines, such as hand-held electric grinders, drilling machines, etc. Physical strength and manual skills of workers are used in the process. Considering the technical progress, the level of the manual machining productivity is low at present. The manual machining has its importance in the field of maintenance and repairs.

Regarding the mechanical machining, energy is necessary for the process. It is usually supplied in the form of electrical energy to the machine tool. There it is transformed into mechanical energy which is used for the realization of the machining process.

Basic classification of the machining methods according to the characteristics of the cutting edge geometry:

- machining with the defined cutting edge geometry (e.g. turning, milling, drilling, reaming, boring, planning and slotting, broaching, etc.);
- machining with the undefined cutting edge geometry (e.g. grinding, lapping, honing, etc.);
- unconventional machining methods (e.g. electrical discharge, chemical, ultrasonic, laser, plasma, etc.);
- adjustments of the machined surfaces (e.g. rolling, polishing, burnishing, shot blasting).

Classification of the machining methods according to the nature of cutting engagement:

- continuous cutting during which the cutting edge is engaged throughout the cutting;
- interrupted cutting in which the cutting edge alternately goes in and out of engagement. A typical example of the interrupted cutting is milling.

Classification of the machining methods according to the movement direction of material particles with regard to the cutting edge:

- free machining in which the movement direction of the chip particles is the same in all points of the cutting edge;
- bounded machining in which the chip particles move in different directions, though basically in perpendicular direction to the cutting edge.

Plunge turning or cutting off turning is a typical example of free cutting as shown in Figure 2.1, or tangential turning with skew movement of the cutting tool to the workpiece axis, i.e. tangential to the machined surface.

The example of the bounded cutting on Fig. 2.1 shows the longitudinal turning using the cutting tool with a rounded tip. Concerning the bounded cutting, real direction of the chip outgoing is determined by the resultant of movements of the respective chip particles.
1.3 Workpiece

In terms of geometry, workpiece is characterized with a surface being machined (1), machined surface (2) and cut surface (3) (also known as cutting area).

Surface being machined is a part of the workpiece surface being removed by machining. In Fig. 2.2 is indicated with number 1.

Machined surface is a surface on the workpiece created by the cutting tool. In Fig. 2.2 is indicated with number 2.

Cut surface is an immediate area of the workpiece created by machining using the cutting edge of the cutting tool in the course of a revolution or stroke. In Fig. 2.2 is indicated with number 3.
1.4 Cutting tool

Cutting tool is an active part in the machining system. It is the cutting tool in interaction with the workpiece which enables the realization of the cutting process. To carry out the cutting, its working part, i.e. the cutting edge, enters the workpiece material and it subsequently removes particles in the form of chips from it. Cutting tool consists of the following parts:

Cutting part is a operational part of the cutting tool which contains the elements forming the chip (see Fig. 2.3, indicated with 3). It includes the edge, face and back of the cutting tool. If the cutting tool has more edges (teeth), each edge (tooth) has its own cutting part.

Cutting edge is an element of the cutting tool bounded by a face and a back. It is a part of the cutting tool in the shape of a wedge that enters the workpiece. Surfaces forming the cutting wedge are shown in Fig. 2.3.

Clamping part (tool shank) is a part of the cutting tool which serves to clamping into the machine tool (see Fig. 2.3, indicated with 1).

Cutting tool base is a flat element of the tool shank and it is used for cutting tool placement and orientation during manufacturing, quality control and sharpening. Not all the cutting tools have their base clearly defined (see Fig. 2.3, indicated with 2).

Surfaces on cutting tools are specified by symbols that consist of letters A and Greek alphabet index indicating the type of surface (e.g. \( A\gamma \) indicates the cutting tool faces). Surfaces assigned to the secondary edge are marked with a comma (e.g. \( A\alpha' \) indicates the secondary back).

Cutting tool flank \( A\alpha \) is a surface or a complex of surfaces that are oriented to the workpiece surface in the cutting process. The main back \( A\alpha \) is oriented to the workpiece transition surface while the minor flank \( A\alpha' \) is oriented to the workpiece machined surface (see Fig. 2.3.)

Cutting tool face \( A\gamma \) is a surface or a complex of surfaces along which the chip leaves. The shape of the tool face is determined by a curve created by the intersection of the tool face surface \( A\gamma \) with the required plane, while this shape is usually defined and measured in the cutting edge plane \( Pn \).

Chip breaker is a part of the tool face surface destined for the chip breaking or rolling. They can be molded or attached to the tool face.
Cutting edge is a part of the cutting part by which the respective cutting process is realized. It is an intersection of the cutting tool face and back.

Main cutting edge $S$ is a part of the cutting edge which is to create the transition area on the workpiece.

Minor cutting edge $S'$ carries out the finishing work on the machined area, though it does not create the transition area.

Considered cutting edge point is a point occurring in any part of the main or minor cutting edge and in which the origin of the coordinate system is.

Cutting tool corner (tip) is a relatively small part of the cutting edge which is situated on a join of the main and minor cutting edge. It can be direct (chamfered) or rounded.

Movements in machining (metal cutting)

Main cutting motion (primary cutting motion) is a movement between the tool and the workpiece which is realized by the machine tool. Direction of the main cutting motion is defined as a direction of the immediate main motion of the considered cutting edge point.

Cutting speed $v_c$ is expressed as an immediate speed of the main cutting motion of the considered cutting edge point with regard to the workpiece.
Feed motion (secondary cutting motion) is realized as an additional relative movement between the cutting tool and workpiece. This movement does not occur within some machining methods.

Feed speed $v_f$ is defined as an immediate speed of the feed motion in the considered edge point with regard to the workpiece.

Resulting cutting motion is a movement resulting from the synchronous movement of the main and secondary motion. It is realized by the vector sum of both movements.

Speed of resulting cutting motion is an immediate speed of the resulting cutting motion in the considered cutting edge point with regard to the workpiece.

Cutting motion angle $\eta$ is an angle between the direction of the main cutting motion and direction of the resulting cutting motion in a work plane $P_{fe}$.

Feed motion angle $\varphi$ can be expressed as an angle between the direction of the current feed motion and direction of the main cutting motion in the work plane $P_{fe}$. For some machining processes, this angle is not defined (e.g. planning or broaching).

Infeed is a movement of a cutting tool or a workpiece by which the cutting tool is set to operating position according to the required cutting width $a_p$. (in technical practice, it is still used the cutting depth $h$).

![Fig. 2.5 Movements of cutting tool and workpiece in feed turning (left) and cutting up milling by cylindrical cutter with straight teeth (1 – direction of main cutting motion, 2 – direction of feed motion, 3 – direction of resulting cutting motion, 4 – considered cutting edge point)](image-url)
Fig. 2.6. Main cutting motion ($v_c$) and feed cutting motion ($v_f$) regarding selected machining methods.
2. TECHNOLOGY OF METAL CUTTING (MACHINING)

Machining is a working process in which a workpiece gets the required shape and size of final machine parts by removing material from the surface layer. Machining methods can be divided according to different aspects. Within this learning text, we will deal only with turning, milling, drilling and reaming.

2.1 Turning

Turning is a conventional machining method destined to a production of mainly rotating shapes, above all, by means of various single point cutting tools - turning tools. It is one of the simplest and most common methods of machining (about 30-40% of turning is done on lathes). On the lathes with manual or automatic control, workpieces with weight ranging from a few milligrams to several tons are turned on manual and automatically controlled lathes.

In the course of turning, an excess layer (working allowance) is cut by a cutting part of cutting tool with the defined geometry. Cutting layer goes out of the workpiece in the form of chips. In order to separate the chips from the workpiece, the active part of cutting tool must have the cutting edge in the shape of a wedge which is supposed to be substantially harder than the machined material. The workpiece subsequently obtains the required shape, size, surface roughness and also some mechanical qualities. In machining, it is necessary to establish and maintain the respective cutting conditions.

Main cutting motion \((v_c)\) is a rotating motion and it is executed by the workpiece. From the surface of the rotating workpiece, chips are cut by means of the cutting edge.

Feed motion \((v_f)\) is a secondary motion, it is usually straight and it is executed by the cutting tool. This motion is necessary to ensure subsequent cutting of chips of the required cross-section.

If the turning tool moves in the direction of the workpiece rotation axis (axial), it refers to the feed turning and resulting cutting motion \((v_e)\) is in the shape of a screwline. If the cutting tool moves in the perpendicular direction to the rotation axis (radial), it refers to the direct (face) turning (cutting off, grooving ...) and resulting cutting motion is in the shape of the Archimedes spiral. Infeed is perpendicular to the feed, i.e. in case of the feed turning it is radial and in case of the face turning is axial.
Fig. 1.1 Turning types a) feed turning, b) face turning

### 2.1.1 Cutting conditions

During establishing the cutting conditions, it is essential to define: cutting speed $v_c$, feed $f$ and thickness (cutting depth) $a_p$ of the machined layer.

- **Cutting speed** is a speed of the main cutting motion and it is defined as a peripheral velocity measured on the machined surface. Different cutting speeds are used for different types of materials of the tools and workpieces. They range from 5 m.min$^{-1}$ up to 1000 m.min$^{-1}$.

  Periphery cutting speed of the workpiece in the place of turning can be defined according to the following equation:

  $$
  v_c = \frac{\pi \cdot D \cdot n}{1000} \quad \text{[m.min}^{-1}] \quad (1.1)
  $$

  - $D$ – diameter of machined surface [mm]
  - $n$ – number of spindle revolutions [min$^{-1}$]

- **Feed** is a path executed by the cutting tool during one workpiece revolution. In roughing, its value ranges from 0.4 up to 3.5 mm, in finishing from 0.06 up to 0.3 mm and in fine turning from 0.03 up to 0.05 mm. In the course of turning, cutting tool moves by a feed value during one revolution, and therefore, it is possible to define $v_f$, feed speed depending on the spindle speed as follows:

  $$
  v_f = f \cdot n \quad \text{[mm.min}^{-1}] \quad (1.2)
  $$

  - $f$ – feed per revolution [mm]
  - $n$ – number of spindle revolutions [min$^{-1}$]

  $$
  v_r = \sqrt{v_c^2 + v_f^2} \quad \text{[m.min}^{-1}] \quad (1.3)
  $$

  **Cutting depth** $a_p$ ranges from a few tenths of mm to several mm within turning. The feed rate, cutting depth, primary cutting tool edge angle and shape of the cutting edge
section, which is in engagement, affect the size and shape of the chip cross-section. For basic types of turning, the cross-sectional dimension of chips according to Fig. 1.2 can be calculated as follows:

for feed turning \[ a_p = 0.5 \cdot (D - d) \quad [\text{mm}] \] (1.4)

D – diameter of surface being machined [mm]
d – diameter of machined surface [mm]

for face turning \[ a_p = L - l \quad [\text{mm}] \] (1.5)

L – length of surface being machined [mm]
l – length of machined surface [mm]

Specific chip width \( b_D \) and specific chip thickness \( h_D \) are calculated according to the following equations:

\[ b_D = \frac{a_p}{\sin \kappa_r} \quad [\text{mm}] \] (1.6)

\( \kappa_r \) – cutting tool edge angle

\[ h_D = f \cdot \sin \kappa_r \quad [\text{mm}] \] (1.7)

Specific chip cross-section \( A_D \) is calculated according to the following equation:

\[ A_D = b_D \cdot h_D = a_p \cdot f \quad [\text{mm}^2] \] (1.8)

Fig. 1.2 Identification of chip cross-section in turning a) round area b) face area

\( a_p \) – cutting depth, \( b_D \) – specific chips width, \( h_D \) – specific chip thickness, \( \kappa_r \) – cutting tool edge angle, \( D \) – diameter of surface being machined, \( d \) – diameter of machined surface, \( L \) – length of surface being machined, \( l \) – length of machined surface
2.1.2 Turning forces

In turning, total machining force $F_e$ has 3 components – cutting component of machining force $F_c$, feed component of machining force $F_f$ and passive component of machining force $F_p$. In feed turning, we can calculate these force components according to the following equations:

\[
F_c = c_{Fc} \cdot a_p^{x_{Fc}} \cdot f^{y_{Fc}} \quad \text{[N]} \quad (1.9)
\]

\[
F_f = c_{Ff} \cdot a_p^{x_{Ff}} \cdot f^{y_{Ff}} \quad \text{[N]} \quad (1.10)
\]

\[
F_p = c_{Fp} \cdot a_p^{x_{Fp}} \cdot f^{y_{Fp}} \quad \text{[N]} \quad (1.11)
\]

c_{Fc}, c_{Ff}, c_{Fp} – material constants [-]

x_{Fc}, x_{Ff}, x_{Fp} – feedback index of $a_p$ [-]

y_{Fc}, y_{Ff}, y_{Fp} – feedback index of $f$ [-]

$a_p$ – cutting depth [mm]

$f$ – feed per revolution [mm]

![Diagram of forces and resistances in feed turning](image)

*Fig. 1.3 Components of machining forces and resistances in feed turning*
Total machining force is defined by the following equation:

\[ F = \sqrt{F_c^2 + F_f^2 + F_p^2} \quad \text{[N]} \quad (1.12) \]

Ratio of individual machining forces in conventional feed turning is as follows:

\[ F_c : F_f : F_p = 1 : 0.4 : 0.25 \quad \text{at } \alpha_r = 45^\circ \]

Cutting force related to the unit surface is called \textbf{specific cutting force} and it is indicated with \( k_c \). It can be expressed as the ratio of the cutting components of machining force \( F_c \) and surface of the specific chip cross-section \( A_D \).

\[ k_c = \frac{F_c}{A_D} \quad \text{[MPa]} \quad (1.13) \]

Required performance of the machine tool is defined by the following equation:

\[ P = \frac{F_c \cdot v_c}{\eta} \quad \text{[kW]} \quad (1.14) \]

\( \eta \) – mechanical efficiency of machine tool [%]

\[ \text{2.1.3 Unit machining time} \]

Ratios shown in Fig. 1.4 are important for defining the unit machine time of the turning processes. In respect of the feed turning (Fig. 1.4a), unit machining time is based on the following equation:

\[ t_{AS} = \frac{L}{v_f} = \frac{L}{n \cdot f} \quad \text{[min]} \quad (1.15) \]

\( L_n \) – cutting tool path [mm]

\( v_f \) – feed speed [mm.min\(^{-1}\)]

\( n \) – workpiece revolution [min\(^{-1}\)]

\( f \) – feed per revolution [mm]

Total of individual elements is equal to the cutting tool path in the feed direction \( L \) [mm]:

\[ L_n = l + l_n + l_p \quad \text{[mm]} \quad (1.16) \]

\( l \) – length of surface being turned [mm]

\( l_n \) – length of pre-travel [mm]

\( l_p \) – length of over-travel [mm]
Regarding the face turning (Fig. 1.4b), it is important to calculate the unit machine time at constant revolution of workpiece $t_{ASn}$ and constant cutting speed $t_{ASv}$. To calculate the $t_{ASn}$, we apply the equation (1.15). Distance $L$ (for conditions shown in Fig. 1.4b) can be defined by the following equation:

$$L = \frac{[D_{max} + 2l_n] - [D_{min} - 2l_p]}{2} \quad [\text{mm}] \quad (1.17)$$

Value of the $t_{ASv}$ (for conditions shown in Fig. 1.4b) can be defined by the following equation:

$$t_{ASv} = \frac{\pi [D_{max} + 2l_n]^2 - [D_{min} - 2l_p]^2]}{4000 \cdot v_c \cdot f} \quad [\text{min}] \quad (1.18)$$

$v_c$ – cutting speed [m.min$^{-1}$]

$f$ – feed per revolution [mm]

### 2.1.4 Basic works on lathe

By means of turning, it is possible to machine outer and inner cylindrical surfaces, spherical and general rotating surface, plane (face) surfaces and threads. On lathes, we can also carry out drilling, boring, reaming, cutting threads, indenting, rolling, smoothing, polishing, producing flank of form cutters by relieving, etc. Single operations are shown in Fig. 1.5.
<table>
<thead>
<tr>
<th>Operation</th>
<th>Diagram 1</th>
<th>Diagram 2</th>
<th>Diagram 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>feed turning</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>face turning</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>chamfer</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>turning of cone</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>necking</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>face-off</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>drilling</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>boring</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>threading</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>turning with shape tool</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>copying</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>indenting</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Fig. 1.5 Basic operations on lathe
2.1.5 Cutting tools – turning tools

The most common cutting tools used for machining are turning tools. They are single point cutting tools of simple shape and they are relatively cheap and easy to maintain. Basic characteristics of the turning tools are:

- cutting edge shape,
- material of cutting part,
- cross-section of cutting tool.

In respect of technology, we can divide the turning tools into:

- radial (the most common used group of cutting tools),
- prismatic,
- circular tool,
- tangential.

![Fig. 1.6 Form turning tools a) prismatic, b) circular, c) tangential](image)

**Radial turning tools**

These cutting tools can be further divided according to their construction as follows:

- solid – base and cutting part are made of tool steel or high speed steel and they build 1 unit,
- with soldering cutting inserts – an insert made of cutting material is soldered with hard solder to the base of turning tool made of structural steel,
- with indexable inserts (VBD) – an insert is mechanically chucked into a tool holder of structural steel using the ISO clamping systems (fig. 1.7).

Mechanical clamping into the tool holder is a principle of the turning tools with indexable inserts. There are many clamping systems whose aim is to ensure the same solidness as the soldering inserts have.
Radial turning tools can be further divided as follows:

According to the feed direction during machining:
- right – cutting tool moves from tailstock to spindle,
- left – cutting tool moves from spindle to tailstock.

According to the machining method:
- for machining of outer surfaces (fig. 1.8),
- for machining of inner surfaces (fig. 1.9).

Each of these groups can be subdivided as follows:
- rougher tool,
- necking tool,
- cut-off tool,
- copying tool,
- threading tool,
- form tool.

According to the body shape of the turning tool:
- facing tool,
- straight tool,
- side tool,
- bent tool,
- corner tool.

Fig. 1.8 Outer turning tools,
a – rougher facing tool, b – rougher straight tool, c – rougher straight tool, d – rougher bent tool, e – rougher double-sided tool, f – corner tool, g – corner tool, h – rougher side tool, i – finishing tool, j – circular tool

Fig. 1.9 Inner turning tools,
1 – inner rougher tool, 2 – inner corner tool, 3 – inner copying tool, 4 – inner rougher tool, 5 – inner rougher tool, 6 – inner corner tool

Suitable materials for production of the indexable inserts are the following: sintered carbide, ceramic, cermets, cubic boron nitride and diamond (fig. 1.10). Inserts can be manufactured as one-sided (there is one tool face) or as double-sided (there are two faces). Insert faces are either plain or they have prepressed chip breakers (in case of very hard tool material, they are grinded). At present, almost all inserts are produced as multi-edge cutting tools. The advantage is that, after the tool wearing, these inserts can be indexed into a new position to use the following cutting edge (e.g. triangle double-sided insert has 6 usable edges). Clamping of the inserts must ensure directing the cutting resistance to the tool holder stands so that the clamping system is not overloaded. Changing of inserts is very quick and easy and it is usually not necessary to adjust the cutting edge position.
Fig. 1.10 Indexable cutting tool inserts
1 – sintered carbide, shape S, double-sided, prepressed chip breaker
2 – sintered carbide, shape W, double-sided, prepressed chip breaker
3 – sintered carbide, necking, prepressed chip breaker
4 – sintered carbide, threading inserts
5 – sintered carbide, cut-off, prepressed chip breaker
6 – solid cubic boron nitride, shape R
7 – diamond gag, shape C, one-sided
8 – diamond gag, shape T, one-sided, chip breaker
9 – diamond gag, shape S, double-sided
10 – solid cubic boron nitride, shape S, one-sided, grinded chip breaker

Fig. 1.11 Other shapes of indexable inserts
The advantages of the cutting tools with indexable inserts:

- body of the cutting tool has substantially longer tool life compared to the cutting tools with soldering inserts because of the possibility to slew the cutting insert or to replace the whole insert,
- regarding the soldering cutting inserts, there might occur residual stresses arising during the insert grinding or soldering. On this account, cutting tools with indexable inserts have longer tool life of the cutting edge,
- instead of replacement of the whole cutting tool, we can change only the cutting insert which is timesaving,
- cutting tools with indexable inserts reduce costs of warehousing.
The disadvantage of the cutting tools with indexable inserts is that they need more space for mechanical clamping than the soldering cutting inserts as the cutting tools with indexable inserts are bigger than the soldering cutting tools. This disadvantage is perceptible within the production of small parts on automatic machine tools and turret lathes where small cutting tools are used.

Form tools (prismatic and circular tools) are used to cut smaller form parts. To use the form turning tools is convenient because of high productivity, accuracy and possibility of high number of sharpening.

Turning tools and indexable cutting inserts are standardized in the unified system of ISO. This system is used by all important producers of the cutting tools.

By using higher cutting speeds and feeds and by means of new and progressive cutting tools, we can increase the productivity up to 60%. In connection with the productivity increasing, the machined surface quality and reliability of the cutting tools are supposed to improve as well as the machining time and product price are supposed to be reduced.

With respect to the product costs and quality and the expected development, we can assume that modernization of the cutting tools will be reflected in the total price reduction as follows:

- at increasing the cutting tool life by 50%, final price of product will be reduced by 1%,
- at decreasing the cutting tool price by 30%, final price of product will be reduced by 1%,
- at increasing the cutting speed and feed by 20%, final price of product could be reduced by 15%.

2.1.6 Machine tools – lathes

Machine tools destined for turning (lathes) form the most numerous group of the machine tools. There are many types and sizes of lathes.

According to construction, we can divide lathes into:

- centre lathe,
- vertical lathe (carousel),
- facing lathe,
- turret lathe,
- special lathe.

According to degree of automations, we can divide lathes into:
- hand controlled,
- semiautomatic – they work in an automatical cycle (operator intervenes only in case of the cycle repeating and workpiece clamping and changing),
- automatic – they enable selfactive repeating of the working cycle after machining of one component.

Regarding the semiautomatic and automatic working cycles, hard of flexible automation is applied.

2.1.6.1 Centre lathes

Centre lathes are used in single-piece and small-lot production for turning of shaft and flange parts of various sizes and shapes. Two basic types of centre lathes are manufactured – universal and simple (production) lathes.

Universal lathes have a lead-screw and anything can be manufactured on them. They are suitable for machining of outer and inner surfaces and they are also used for cutting of the face surfaces, necking within the feed or face turning, threading, turning of conuses or contoured surfaces. They can be equipped with a stepless speed variation. Simple lathes have no lead-screw, they are equipped with a stronger electromotor and are used for roughing works.

Size of lathes is defined by maximal diameter of the machine tool (which is possible to be machined) and by cutting length (distance between the centres).

Fig. 1.14 Universal centre lathe
2.1.6.2 Vertical lathes (carousels)

Vertical lathes are used in single-piece and small-lot production of medium and large rotating parts. They are manufactured as single-column (Fig. 1.15) and double-column (Fig. 1.16). Their main parts are rotating table, frames and traverses with lathe carriage. They are sometimes equipped with a grinding inclinable equipment (for grinding of inner and outer surfaces), workpiece position indication and numerical control (NC).

Principle of a carousel is that there is a chunk with workpiece fixed on a spindle. There are one or two traverses with a tool post located on a stand with vertical leading. Cutting tools can move in vertical as well as in horizontal direction.

On vertical lathes, we can turn an outer and inner cylindrical surface, conic surface (with carriage indexing) or threading. If the carrousel is equipped with a copying device, we can turn shape surfaces, too.

![Fig. 1.15 Vertical lathe single-column](image1.png)  ![Fig. 1.16 Vertical lathe double-column](image2.png)

2.1.6.3 Facing lathes

Facing lathes (fig 1.17) are suitable for cutting of platy parts of large diameters and small length. They are equipped with one or two carriages and they generally have no tailstock to support the workpiece. To clamp the workpiece, vertical clamping plates fixed on the spindle end are used. They are not very common in production as they are often replaced by vertical lathes.
2.1.6.4 Turret lathes

Turret lathes are suitable for small and medium workpiece series that demand higher number of cutting tools. Subsequently, more cutting tools machine more surfaces in the course of one clamping. These machine tools enable feed turning, face turning, drilling, boring, reaming, threading, etc. A tool turret in which the cutting tools are clamped is revolving around the vertical or horizontal axis. Mainly, cutting tools for machining of a surface and cutting tools for machining of holes are used. Setup of the cutting tool is carried out only for the first workpiece and then the whole serie is machined. The whole cycle is automatized and it is very often numerically controlled. The tool turret is exchangeable.

Resulting semiproducts can be bars (clamped into collets), casts, forgings, etc. According to the rotation axis, turret lathes are divided into lathes with horizontal (fig. 1.18), vertical or oblique axis of the tool turret.
2.1.6.5 Workpiece clamping

The correct clamping must fulfil several conditions. It must be reliable, simple, fast, with high tenacity and it must provide a single valued position of the workpiece with regard to the cutting tool. There are many clamping systems and their combinations. **Clamping types depends especially on the workpiece shape and weight.**

The most common clamping system is the **universal chuck** (fig. 1.19). The most frequent chucks are those that have three jaws, though the four-jaw chunks also exist. Movement of individual jaws can be bounded (self-centering chuck) or they can move independently. Clamping is usually done by hands (by a socked wrench) or pneumatically, hydraulically or electrically which saves the time needed for clamping.

Self-centering chucks are used for turning on semiautomatic and automatic lathes. When the jaws start to rotate, they grip the workpiece automatically. Carrier plates with a spring loaded center are used on this type of the machine tools. Torque is transmitted by cutting resistance of cutters which are fitted in the workpiece face. This alternative enables to turn the workpiece on its whole length.

![Fig. 1.19 Universal chuck with exchangeable jaws](image)

Workpieces with the length-to-diameter ratio higher than 2/3 are **centered** (fig. 1.21). Centres are slipped to a centre dot which is drilled in the workpiece face. There is a fixed
centre in headstock and a rotating centre in tailstock. This type of clamping is used in case of demand on higher accuracy (when the minimal concentricity between the machined surface and rotation axis is defined). Torsion moment is transmitted by the carrier plate with the lathe carrier.

As the very thin workpieces are easy to deflect during the machining, they must be supported by **steadies** (fig. 1.20). These steadies are mounted in the middle of the workpiece. We differ a steady rest (fixed on lathe bed) or a sliding steady (it moves on lathe bed).

Bars of small diameter (cylindrical or hexagonal cross section) are clamped into accurate clamping adapters – **collets** (fig. 1.22). Collets are split with several circumferential splines and material is clamped by pulling into a conical pocket. The disadvantage is that they are graded by 0.5 mm or by 1 mm and, therefore, the whole kit of collets is always necessary. It is a very precise clamping method (concentricity of 0.01 up to 0.05 mm).

Heavy and shorter workpieces of irregular shape (i.e. casts) are clamped into **clamping plates** (fig. 1.23) with separately adjustable moldings. Within the serial production, clamps are used for the workpiece clamping. They are individually designed for the complicated shapes of components. They provide the accurate clamping as well as simple and quick handling.
2.1.7 Cutting tool clamping
Clamping of the turning tools into the lathes should be single valued, reliable and with minimal tool overhang and it should be easy to adjust the height. Turning tools are clamped into the different rotating tool posts (up to 4 turning tools at once) or into clamps.

Regarding the turret and special lathes, the turning tools are clamped into special tool holders which are placed in a toolholder magazine. Adjusting of the toolholder magazines is carried out outside of the working space. From this space, they are automatically transferred to the working process.

2.2 Milling
Milling is one of the most frequent methods of machining. The great advantage of milling is relatively high performance at very good quality of machining. Milling is used for machining of prismatic plane, form and rotating surfaces. It is also used for machining of grooves of various profiles as well as for machining of threads and gears.

During milling, chips are removed by edges of a rotating tool – milling cutter. The main milling motion is a rotary motion and it is made by the tool. The secondary motion is a feed motion which is usually direct and it is made by the workpiece. Concerning the modern machines, feeds can be changed continuously and they can be carried out in different directions at the same time (multi-axis machining centres). The cutting process is interrupted because each tooth usually cuts off short chips of variable depth.

In respect of technology, there are different types of milling according to the position of the tool axis to the surface being machined:

- **Cylindrical (Fig. 1.24)** – tool circumference - the tool axis is parallel to the surface being machined.
- **Face (Fig. 1.25)** – tool face - the tool axis is perpendicular to the surface being machined and the cutting depth is set in the direction of the tool axis.

There are also other types of milling which are derived from the following basic types:

- **Circular (Fig. 1.26)** – rotary tool circumference – on the workpiece, also of rotary shape – the tool and workpiece axes are usually mutually inclined and the cutting depth is set in the perpendicular direction to the workpiece axis. Circular milling can be used for machining of external as well as internal cylindrical surfaces,
- **Planetary (Fig. 1.27)** – it is used for machining of cylindrical external and internal surfaces.
Fig. 1.24 Cylindrical milling a) cutting-up milling b) cutting-down milling

Fig. 1.25 Face milling

Fig. 1.26 Circular milling a) external b) internal
For cylindrical milling, cylindrical and form mills are used. Teeth are placed only around the tool circumference and the cutting depth is set in the surface perpendicular to the mill axis and to the feed direction. The machined surface is parallel to the tool rotation axis.

According to the position of the tool axis, there are two types of cylindrical milling:
- cutting-up milling (Fig. 1.24a)
- cutting-down milling (Fig. 1.24b)

Within the cutting-up milling, a cutting edge of a rotary tool moves in the opposite direction of the workpiece feed direction in the contact point. The machined surface is generated when the tool penetrates the workpiece. The variable chip depth changes from the minimum (zero) value to the maximum value. When the tool is penetrating the material, at the beginning, the material is only compressed by the edge. The chip is removed in the moment when the layer cutting depth reaches the respective size because the mill edge can not be a sharp edge but it can be a facet of the radius of $R = 8$ to $30 \, \mu m$. Force effects and deformation resulting from the cutting-up milling cause the increased edge wear.

The advantages of cutting-up milling in conventional chip machining:
- lower tool (screw and nut) wear,
- initial engagement of teeth does not depend on the cutting depth,
- tool life is not significantly effected by the workpiece surface.

The disadvantages of cutting-up milling:
- impaired quality of the machined surface,
- direction of milling force with respect to clamping.
In terms of **cutting-down milling**, a cutting edge of the rotary tool moves in the workpiece feed direction in the contact point. The variable chip depth changes from maximum value to minimum (zero) value. To carry out the cutting-down milling on a machine, the conventional mill has to be adjusted in such way that clearance and pre-load between the feed screw and table nut are specified. If this condition is not met, the tool (or even the machine) can be damaged.

The advantages of cutting-down milling:
- higher quality of tools,
- higher cutting speed and feed can be used,
- lower cutting power for machining is needed,
- simpler clamping (workpiece is pressed to the table by cutting force),
- lower inclination to oscillating,
- higher quality of the machined surface.

The disadvantages of cutting-down milling:
- unsuitable for machining of semi-products with hard and polluted surface,
- force load of each tooth in engagement.

Edges of the **face milling** tool are placed not only on the mill circumference but also on the mill face (on the surface perpendicular to the mill axis). According to the position of mill rotation axis with regard to the surface being machined, there are two basic methods:

- **symmetrical milling (Fig. 1.28a)** – the tool axis goes through the centre of the surface being milled,
- **asymmetrical milling (Fig. 1.28b)** – the tool axis is outside the centre of the surface being milled.
In Fig. 1.25, there is clearly shown that during the face milling, cutting-down milling and cutting-up milling are in progress at the same time.

![Diagram of Face Milling](image)

**Fig. 1.28 Face milling a) symmetrical b) asymmetrical**

### 2.2.1 Cutting Conditions

To simplify the calculation, we consider the tool circumferential speed as the cutting speed $v_c$:

$$v_c = \frac{\pi \cdot D \cdot n}{1000} \quad [\text{m.min}^{-1}] \quad (1.19)$$

- $D$ – tool diameter [mm]
- $n$ – tool revolutions [min$^{-1}$]

The **feed per revolution** $f_n$ (mm) is a length of the workpiece path per one tool revolution. The **feed per tooth** $f_z$ (mm) is a basic unit of feed motion. It is a length of the workpiece path per one tool revolution divided by the number of tool teeth.

$$f_n = f_z \cdot z \quad [\text{mm}] \quad (1.20)$$

- $z$ – number of tool teeth (edges) [-]

To calculate the feed speed $v_f$, we use the following equation:

$$v_f = f_n \cdot n = f_z \cdot z \cdot n \quad [\text{mm.min}^{-1}] \quad (1.21)$$

- $n$ – number of tool revolutions [min$^{-1}$]
Cutting depth

During cylindrical cutting-up milling, the chip depth \( h_i \) changes from zero to the maximum value. During cutting-down milling, it changes from maximum value to zero (Fig. 1.24). The chip depth \( h_i \) is expressed in any phase by the following equation:

\[
h_i = f(\varphi_i) = f_z \cdot \sin \varphi_i \quad \text{[mm]}
\]

\( f_z \) – feed per tooth [mm]

\( \varphi_i \) – feed angle [°]

The feed angle changes depending not only on the respective tooth position but, regarding the mills with oblique teeth or with helical teeth, also along the respective edge.

A\textsubscript{di} indicates the specific chip cross-section for the mill position \( i \). For the conditions shown in Fig. 1.29a and 1.29b, we can calculate it according to the following equation:

\[
A_{Dh} = a_p \cdot h_i = a_p \cdot f_z \cdot \sin \varphi_i \quad \text{[mm}^2]\]

\( a_p \) – cutting depth [mm]

If \( \varphi_i = \varphi_{\text{max}} \), the maximum value of the specific chip cross-section is:

\[
A_{D_{\text{max}}} = a_p \cdot h_{\text{max}} = a_p \cdot f_z \cdot \sin \varphi_{\text{max}} \quad \text{[mm}^2]\]

\[
\sin \varphi_{\text{max}} = \frac{2}{D} \sqrt{D \cdot H - H^2}
\]

In terms of face milling, the chip depth changes depending on the feed angle \( \varphi_i \). It is also affected by the setting angle of the main edge \( \kappa \). (Fig. 1.30, value of \( \kappa \) = 90 °). The instantaneous value can be calculated from the following equation:

\[
h_i = f_z \cdot \sin \varphi_i \cdot \sin \kappa \quad \text{[mm]}
\]

Fig. 1.29a The chip cross-section in a cylindrical mill

Fig. 1.29b The chip cross-section in a face mill

\[
\sin \varphi_{\text{max}} = \frac{2}{D} \sqrt{D \cdot H - H^2}
\]
The specific chip width \( b_i \) is for any \( \varphi_i \) constant and it can be calculated according to the following equation:

\[
b = \frac{a_p}{\sin \kappa_i} \quad [\text{mm}] \quad (1.26)
\]

The specific chip cross-section \( A_{Di} \) for \( \kappa_i = 90^\circ \)

\[
A_{Di} = b \cdot h_i = a_p \cdot f_z \cdot \sin \varphi_i \quad [\text{mm}^2] \quad (1.27)
\]

The maximum value of the specific chip cross-section at \( \varphi_i = 90^\circ \) is:

\[
A_{D\text{max}} = a_p \cdot f_z \quad [\text{mm}^2] \quad (1.28)
\]

### 2.2.2 Milling Force and its Components

To determine individual components of the milling force, we consider the force ratio on an edge angled in \( \varphi_i \). In Fig. 1.30, there is a cylindrical milling tool with straight teeth. The total cutting force affecting the cutting edge \( F_i \) is further divided to \( F_{ci} \) and \( F_{cNi} \), respectively to \( F_{fi} \) and \( F_{fNi} \) components.

![Fig. 1.30 Decomposition of cutting force on a tooth of cylindrical mill in working plane \( P_{fe} \)
Fi – total milling force, \( F_{ci} \) – cutting component of cutting force, \( F_{cNi} \) – perpendicular cutting component of cutting force, \( F_{fi} \) - feed component of cutting force, \( F_{fNi} \) - perpendicular feed component of cutting force](image)

Cutting component of cutting force \( F_{ci} \) is effected by the specific cutting force \( k_{ci} \) and chip cross-section \( A_{Di} \) as follows:

\[
F_{ci} = k_{ci} \cdot A_{Di} = k_{ci} \cdot a_p \cdot f_z \cdot \sin \varphi_i \quad [\text{N}] \quad (1.29)
\]
Specific cutting force $k_{ci}$ can be expressed as follows:

$$k_{ci} = \frac{C_{Fi}}{h_i^{1-x}} = \frac{C_{Fi}}{(f_z \cdot \sin \phi_i)^{1-x}} \quad \text{[MPa]} \quad (1.30)$$

$C_{Fi}$ – constant expressing the influence of material being machined [-]

$x$ – exponent expressing the influence of chip depth [-]

If introduced into the 1.29 and 1.30 equations, then:

$$F_{ci} = C_{Fi} \cdot a_p \cdot f_z^{x} \cdot \sin \phi_i \quad \text{[N]} \quad (1.31)$$

### 2.2.3 Unit Machining Time

In Fig.s 1.31 and 1.33, there are shown ratios for the basic examples of milling resulting from which it is possible to determine the unit machining time.

In general, the unit machining time $t_{As}$ is expressed as follows:

$$t_{As} = \frac{L}{v_f} \quad \text{[min]} \quad (1.32)$$

$L$ – tool path in feed direction [mm]

$v_f$ – feed speed [mm.min$^{-1}$]
In terms of cylindrical milling, the path \( L \) is expressed as follows: (Fig. 1.31):
\[
L = l + l_n + l_p + l_{nf} \quad [\text{mm}] \quad (1.33)
\]
\[
l_{nf} = \sqrt{H(D - H)} \quad [\text{mm}] \quad (1.34)
\]

In terms of asymmetric rough face milling, the path \( L \) is expressed as follows:
\[
L = l + l_n + l_p + \frac{D}{2} - l_{pf} \quad [\text{mm}] \quad (1.35)
\]
\[
l_{pf} = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{B}{2} + e\right)^2} \quad [\text{mm}] \quad (1.36)
\]

In terms of asymmetric finishing face milling, the path \( L \) is expressed as follows:
\[
L = l + l_n + l_p + D \quad [\text{mm}] \quad (1.37)
\]

### 2.2.4 Cutting tools – Milling Cutters

Mills are multi-edge tools. There are edges arranged on a cylindrical, conical or other form surface of a mill. Regarding the face mills, there are edges arranged also on the face surface. In respect of large range of technology, it is possible to use many types of mills most of which are standardized.

The mills can be classified according to different aspects:
- **Cylindrical mills** (the edge is placed on cylindrical surface – \( a_1 \)),
- **Face mills** (teeth are placed on face surface – \( a_2 \)),
- **Cylindrical face mills** (the edge is placed on face surface as well as on cylindrical surface – \( a_3 \))

Note: Indication in parentheses (letter and number: eg. \( a_1 \), \( d_3 \), \( f_2 \), \( g_5 \), etc.) are used for mill identification in Figures 1.34 - 1.46.
According to the tool material, there are mills of high-speed steel (b1), sintered carbides (b2), cermet (b3), cutting ceramics (b4), KNB (b5) and PKD (b6).

According to the method of teeth production, we differentiate mills with milled teeth (c1) and mills with undercut teeth (c2). Regarding the milled teeth, the face and flank are formed by plane surface. The thin facet on the flank of 0.5 - 2 mm width firms up the edge. Sharpening is carried out on the flank. Concerning the undercut teeth, their flank surface is formed as a part of the Archimedean spiral. The tooth face is a plane surface and sharpening is made on the face. While sharpening the undercut teeth, the profile changes only slightly therefore it is mainly used for form mills.
According to the direction of teeth with regard to the rotation axis, we differentiate mills with straight teeth (D1) and mills with helical teeth (D2), right or left. The advantage of the helical grooves is that the teeth gradually enter the gearing and therefore the cutting process is fluent and quieter. The helix twist is of 10° and 45°, sometimes more.

According to the number of teeth with regard to the rotation axis, we differentiate soft-teeth mills (e1), half-rough-teeth mills (e2) and rough-teeth mills (e3). To ensure the steady operation of a mill, there should be such number of teeth so that at least 2 teeth are in engagement.

According to the constructional organization, there are solid mills (teeth and body are of the same material f1), mills with inserted teeth (f2) and mills with indexable cutting inserts mechanically clamped in the mill body (f3).
According to the geometric shape of the functional part, there are cylindrical mills (g1), circular mills (g2), angular mills (g3), groove mills (g4), copy mills (g5), radius mills (g6), mills for gear production (g7), etc.

According to the method of clamping, there are plug mills (clamped to the central hole - h1) and shank mills (clamped to the cylindrical shank – h2 or cone shank - h3).

According to the direction of rotation viewed from the machine spindle, there are right-cutting mills (i1) and left-cutting mills (i2).
2.2.5 Milling Machines

Milling machines are manufactured in a large number of models, sizes and performances with various accessories. They can be divided into 4 groups:

- console,
- table,
- planar,
- special.

The size of milling machine is determined by various technical parameters:

- width of the table clamping surface,
- size of the cone in the spindle for the tool clamping,
- movement length of the work table or headstock,
- feed range and spindle revolutions,
- electromotor performance for the spindle rotation,
- high quality parameters of the machined surface.

In terms of operation, there are:

- manually operated mills,
- programme controlled (with hard or flexible automation).

2.2.5.1 Console Milling Machines

Console mills are the most common milling machines. They are characterized by a height-adjustable console moving in the vertical direction along the stand leading. There is a mobile transverse table with a longitudinal work table on the console. Therefore, it is possible to adjust the workpiece clamped to the work table in three rectangular coordinates with regard to the tool. They are used for machining of plane and form surfaces of small and medium-size workpieces within unit and small-lot production.

They are produced in three variants:

- horizontal,
- vertical,
- universal.
**Horizontal console mills** (Figure 1.49) have a horizontal axis of working spindle which is parallel with the surface of longitudinal table and perpendicular to the direction of the movement of the longitudinal table. Mainly the cylindrical, disc and form milling cutters are used for machining. The milling centre can be supported by one or two supporting bearing points. Mills with a cylindrical shank and milling heads are used rarely. On the horizontal mills, mainly the surface parallel to the clamping surface of the table, grooves and form surface are machined. In contrast with the horizontal milling machines, the longitudinal table of the universal console milling machines is in the horizontal plane rotating around the vertical axis by ± 45 °.

![Figure 1.49 Horizontal console mill](image)

1 – base stand
3 – console
4 – shoulder
5 - cross table
6 - longitudinal work table
7 – spindle
8 - control panel

**Vertical console mills** (Figure 1.50) have an axis of working spindle vertical to the clamping surface of table. The working spindle is placed in a vertical head mounted on the mill stand or, regarding the mills with a cylindrical shank, it is placed directly in the stand. The vertical head can be indexed by ± 45 ° on both sides. The spindle is vertically adjustable. For machining, face milling cutters set on a short centre or milling cutters with cylindrical shank which are clamped directly into the spindle taper are used. Milling cutters with a cylindrical shank are clamped directly into the chuck. Larger mills also use milling heads. On the vertical console mills, especially the plane surfaces parallel to the clamping surface of a table, grooves in these surfaces and form surfaces are milled.
Universal console mills combine the advantages of both types of the mills mentioned above. They are equipped with an extendable shoulder for clamping of the centre with the cylindrical milling cutter and with a vertical head for clamping of the face mills and milling heads. They can be profitably used for milling of helical grooves on drills, gear cutters and core-drills with helical teeth, etc.

Universal suitability of the console mills are significantly extended with special equipment which includes the following:

- **Universal milling head** – It is fixed to the face surface of the stand of the universal horizontal mill. It can be rotated around two axes so the milling cutter can be adjusted to any position and machine the inaccessible areas.

- **Vertical milling head** – it rotates around the axis of the working spindle and it complements the horizontal mill for works that otherwise would require a vertical mill. The torque moment is transferred from the spindle in very similar way as in case of the universal head.

- **Rotary table** – It is fixed on a mill work table and it enables milling of rotary shapes with shank milling cutters. It can be rotated manually or rotating is derived from the longitudinal movement of the work table by the telescopic shaft.

- **Shaping head** – It is very often an integral part of the equipment of tool-milling machines or it is a high quality equipment of universal and horizontal mills. It is clamped similarly as the vertical milling head. Due to the ability to turn/rotate it, it enables machining in the horizontal, vertical and inclined direction.
- **Dividing machine** – It enables turning the workpiece (dividing) by the defined angle or pitch. It is used for milling of squares and hexagonals, multi-groove shafts, gears, multi-edge tools, notches on the front surfaces, etc. (Figure 1.54).

### 2.2.5.2 Table Milling Machines

In contrast with the console mills, they do not include any console, but a longitudinal cross table. The vertical movement is performed by moving the headstock along the machine stand. On the table milling machines (Figure 1.51), larger and heavier components can be machined very productively and of very high quality. There are both horizontal as well as vertical table milling machines.

![Fig. 1.51 Table Milling Machine](image)

1 - base desk  
2 – stand  
3 – headstock  
4 – spindle  
5 – work table  
6 - control panel

### 2.2.5.3 Planar Milling Machines

They are sturdy machines which belong to the most efficient mills. Using them, it is possible to machine workpieces of large dimensions and weight. The planar mills (Figure 1.52) most frequently use milling heads for machining of horizontal, vertical and inclined surfaces and shank milling cutters for milling of narrow surfaces and grooves. The work table has one degree of freedom (allowing movement in one direction). It is also produced with multiple headstocks as portal (Figure 1.53).
2.2.6 Dividing Machines

Dividing machines are used for indexing of a workpiece (for dividing) by an angle or a pitch for machining of polygons, gears, etc. They are divided as follows:

- **simple** – a dividing disc having on its circumference notches or holes is used to divide the workpiece circumference. It is used for direct dividing method. This method is based on the divider spindle rotation of the spindle cutting machine by the required part of the circumference and locked position. According to the number of notches on the disc, it is possible to divide the workpiece circumference on multiples of 1/24, 1/36 or 1/48 of the circumference. Workpiece can be either clamped on a board with clamping slots or into a universal chuck.
- **universal** - (Figure 1.54) - it allows dividing with a direct, indirect and differential method. For the direct dividing, a dividing disc mounted and fixed on the front end of the dividing spindle. The wheel has 24, 36 or 48 holes into which fits the pin stored in machine.

![Universal dividing machine](image)

Fig. 1.54 Universal dividing machine

The dividing process is the same as in the simple dividing machine. Indirect and differential dividing is carried out by a disc that has different numbers of holes in concentric circles on the front surface. For example, it is 15, 16, 17, 18, 19, 20, 21, 22, 23, 24 ... The machine spindle is indexed by a handle through the gear mechanism $z_1$, $z_2$ with the transmission of 1:1 and the worm gear usually of 1:40.

**An example of indirect dividing** (Figure 1.55) - for example, if the dividing spindle is supposed to be indexed by $\frac{1}{26}$ of the circumference, it is necessary to index the handle by

\[
\frac{40}{26} = 1 + \frac{14}{26}.
\]

Because there are not 26 holes on the dividing disc, it is necessary to adjust the fraction of $\frac{14}{26}$ to $\frac{21}{39}$. Using the pin in the handle, we fasten down the position in one of the holes on a circle with 39 holes. Then we index the handle once around by 21 pitches. Indirect dividing can be also carried out as comprised of pitches on the two circles with different numbers of holes.
An example of differential dividing (Fig. 1.56) - differential dividing is used when the indirect dividing is insufficient or when it is necessary to divide circumference into numbers given by some prime numbers. The machine is equipped with additional transfers of \( z_1 \) to \( z_2 \). With rotating of the dividing handle, the spindle of dividing machine is indexed through the worm and worm gear as an indirect cutting. Simultaneously, the dividing disc is rotating through the reverse transfers of \( z_1 \) to \( z_4 \). If the transfer between the dividing handle and working spindle is formed by \( z_1 \) to \( z_4 \) wheels, the dividing disc rotates in the same direction as the dividing handle. If another idle wheel is added, the dividing disc will rotate in the opposite direction of the dividing handle.

There are elaborated tables for dividing machines. They help to solve different examples of dividing. Universal dividing machines are also used for milling of helical grooves. In this case, the spindle engine is derived from a working screw of longitudinal table of the universal milling machine. The table is indexed by the helix twist angle.

### 2.2.7 Tool Clamping

Cylindrical plug milling cutters are clamped on milling centres. The clamping cone of milling centres and working spindle are produced as metric with conicalness of 1:20, Morse of 1:19 to 1:20 or steep of 1:3.5. Because the metric cone and Morse cone are self-locking, they transmit the torque moment from the spindle to the centre. For the perfect torque moment transmission, there is a rectangular recess on the spindle end in which fits the flattened shoulder at the milling centre end. Regarding the steep cone, the torque moment is transmitted by two stones fixed on the spindle face which fits directly into the shoulder of milling centre. Steep cone only centres the centre in the working spindle. The freely strung expander ring ensures the milling cutter position on the long centre (Fig. 1.57).
There is also a guide bushing on the centre which is a part of the sliding supporting bearing located on the telescopic arm of a horizontal mill. It is important to set up the guide bushing in such a position so the bearing centre is supported. To ensure the firm tool clamping, it is important to clamp the milling cutters as close to the spindle as possible and to pull the telescopic arm as close to the mill as possible. To clamp the shell end milling cutter and milling heads, we use short clamping centres clamped to the machine spindle (Fig. 1.58).

![Fig. 1.57 Long milling centre](image)

1 – spindle  
2 - centre  
3 - expander rings  
4 – supporting bearing  
5 - clamping nuts  
6 - clamping screw

![Fig. 1.58 Short milling centre](image)

1 – spindle  
2 - centre  
3 - clamping screw  
4 - parallel slip feather  
5 - cross slip feather

To clamp the **milling cutters with conical shank**, we use reducters located directly in the mill spindle. Reducters are also used if the milling centre cone does not correspond to the spindle cone. To clamp the **milling cutters with cylindrical shank** to the mill spindle, it is necessary to use a chuck with clamping adapter.

To clamp the **milling cutters with cylindrical shank** (diameter of 3-50 mm) we frequently use special thermal or hydraulic fixtures (Fig. 1.59). When using the thermal fixture, the tool inserted into the fixture is heated by the magnetic field of a coil of a high-frequency generator. The heating process is so fast that the tool temperature due to heat conduction increases minimally. Subsequently, the clamped tool is usually cooled using the air stream. The process of material shrinking ensures reliable clamping. The tool unlocking is carried out by heating in the same device.
2.2.8 Workpiece Clamping

During milling, there are more teeth in engagement at the same time, which causes great cutting force. Therefore it is important to clamp the workpiece properly. For clamping, it is necessary to meet the following conditions:

- workpiece may not be deformed during clamping,
- clamping must be firm and reliable,
- clamping surface must be as close to the spindle as possible.

Smaller workpieces are usually clamped into common mechanical vices, rotary and hinged vices (Fig. 1.60), special vices for clamping of cylindrical components (Fig. 1.61), etc. All these vices can be controlled manually, pneumatically or hydraulically.
Larger workpieces are clamped using various clamping equipments, such as clamps, steady bars, supports, etc. (Fig. 1.62). Clamping equipment is clamped into the T-grooves of the milling cutter table using special screws with a square-shape head.

For precise machining on the numerically controlled milling machines, we use technological pallets with which the workpiece can be moved between the respective machining machines.

![Fig. 1.62 Basic clamps and workpiece supports](image)

2.3 Drilling, Core-drilling, Reaming, Boring

It is a technology which is used to cut holes. This operation is very common in mechanical engineering. Machining of holes is a method during which the inner surfaces of the machine parts are machined. Their shapes can be various and their function depends on them. Production of rotary holes is the least demanding therefore they are the most common in mechanical engineering.

2.3.1 Drilling

It is a manufacturing method which is used to make holes into solid material or to ream the holes. As a tool, a drill is used to perform the main motion – rotating motion. In some special cases, a workpiece can perform the main motion. The secondary motion is rectilinear and sliding (in the axis direction) motion which is also performed by a tool. During machining, the drill axis is mostly perpendicular to the surface being machined.

In drilling, we differentiate through holes or blind holes. From a technological point of view, the through holes can be machined very easily. Regarding the blind holes, it is necessary to focus on the hole ending ensuring the accurate drilling depth, cutting-off the remainder of the chip on the hole bottom, etc. The drill rotates several more turns after the feed stops to cut off the chips from the hole bottom.
The characteristic feature of the tools used to make holes is that the cutting speed decreases from the circumference towards the centre of the tool whereas the cutting speed is zero in the tool axis. Circumferential speed on the maximal diameter of tool edge is considered as a cutting speed and we define it, similarly to turning, as follows.

\[
v_c = \frac{\pi \cdot D \cdot n}{1000} \quad \text{[m.min}^{-1}] \quad (1.38)
\]

\[
v_f = f \cdot n \quad \text{[mm.min}^{-1}] \quad (1.39)
\]

\[
v_c = \sqrt{v_c^2 + v_f^2} \quad \text{[m.min}^{-1}] \quad (1.40)
\]

\(v_c\) – cutting speed [m.min\(^{-1}\)]
\(v_f\) – feed speed [mm.min\(^{-1}\)]
\(v_e\) – speed of resulting cutting motion [m.min\(^{-1}\)]
\(D\) – diameter of hole being machined (maximal diameter of tool edge) [mm]
\(n\) – drill speed (or workpiece) [min\(^{-1}\)]
\(f\) – cutting tool feed per revolution [mm]

We define the feed per tooth according to the following equation:

\[
f_z = \frac{f}{z} \quad \text{[mm]} \quad (1.41)
\]

\(z\) – number of tool teeth [-]

According to the drilling technology, construction type and geometry of the used drill we divide drilling as follows:

- Collaring of the hole beginning by a centering drill into solid material (Fig. 1.63)
- Short-hole drilling into solid material when the ratio of \( \frac{D}{L} = 1/5 \pm 1/10 \) (\( D \) – hole diameter, \( L \) – hole length): twist drills, lanceolate drills, drills with indexable tips and drills with indexable cutting inserts are used for these operations.

- Drilling holes into the pre-drilled holes: the same tools as for the short-hole drilling into the solid material are used. Rarely, gun (barrel) drills are used.

- Drilling deep holes into solid or premachined material when the ratio of \( \frac{L}{D} > 1/10 \): gun drills, ejector drills, BTA, STS are used. Twist drills can be also used to drill holes of small diameter.

- Core drilling – it is a cutting off the material being machined in the form of an annular area using a single-edge or multi-edge coronal drill (Fig. 1.64). It is used to drill through holes of larger diameters.

- Special types of drilling, e.g. drilling holes into sheet metal by a thermal forming drill (Fig. 1.65), drilling stepped holes by a stepped drill (Fig. 1.66), drilling holes with synchronous reaming, threading, recessing (Fig. 1.67) or smoothing by associated tools.
Fig. 1.65 Thermal drilling holes in thin-walled steel profile

Fig. 1.66 Stepped drill

Fig. 1.67 Associated tool for drilling and double recessing

- Drilling into hard-machinable, composite and non-metallic material such as rubber, concrete, stone, bricks by drills with a special construction or geometry.
2.3.2 Calculation of Chip Cross-Section

To calculate a chip cross-section $A_D$ for drilling into solid material and to ream the pre-drilled holes, we use the parameters showed in Fig. 1.68, when the chip is cut off with one edge of a twist drill.

$$A_D = b_D \cdot h_D = a_p \cdot \frac{f}{2} \quad [\text{mm}^2] \quad (1.42)$$

$b_D$ – specific chip width [mm]
$h_D$ – specific chip depth [mm]
$a_p$ – cutting depth [mm]
$f$ – feed per revolution [mm]

![Fig. 1.68 Chip cross-section within drilling by double-edged twist drill](image)

Regarding the drilling into solid material, the cutting depth is $a_p = D/2$, regarding the drilling into pre-machined hole (pre-drilled etc.), the cutting depth is $a_p = (D - d) / 2$. Based on these facts, the equation for calculating the chip cross-section with a single-edge tool for drilling into solid material can be defined as follows:

$$A_D = \frac{D \cdot f}{4} \quad [\text{mm}^2] \quad (1.43)$$

For drilling into the pre-machined hole, the equation is as follows:

$$A_D = \frac{(D-d) \cdot f}{4} \quad [\text{mm}^2] \quad (1.44)$$

For two-edge tool, the total chip cross-section for drilling into solid material is as follows:

$$A_D = \frac{D \cdot f}{2} \quad [\text{mm}^2] \quad (1.45)$$

For drilling into pre-machined hole:
$A_D = \frac{(D - d) \cdot f}{2}$ \hspace{1cm} [mm$^2$] \hspace{1cm} (1.46)

To calculate the parameters of the chip cross-section for machining by a twist drill:

- specific chip width for drilling into solid material:

$$b_D = \frac{D}{2 \cdot \sin \kappa}$$ \hspace{1cm} [mm], \hspace{1cm} (1.47)

- specific chip width for drilling into pre-machined hole:

$$b_D = \frac{D - d}{2 \cdot \sin \kappa}$$ \hspace{1cm} [mm], \hspace{1cm} (1.48)

- specific chip depth

$$h_D = \frac{f}{2 \cdot \sin \kappa}$$ \hspace{1cm} [mm], \hspace{1cm} (1.49)

### 2.3.3 Cutting Force and its Components

In order to overcome the resistance generated during machining, cutting force and its components work upon the drill. Standard twist drill or lanceolate drill have 2 edges which are located symmetrically towards its axis. To obtain the resulting cutting force, we need to consider components affecting both cutting tool edges (Fig. 1.69).

The feed component of drilling force:

$$F_r = F_{f1} + F_{f2}$$ \hspace{1cm} [N] \hspace{1cm} (1.50)

The passive component of drilling force:

$$F_p = F_{p1} + F_{p2}$$ \hspace{1cm} [N] \hspace{1cm} (1.51)

The cutting component of drilling force:

$$F_c = F_{c1} + F_{c2}$$ \hspace{1cm} [N] \hspace{1cm} (1.52)

The forces on both cutting edges are equal in case of correct and accurate sharpening:

$$F_{f1} = F_{f2} = \frac{F_f}{2}$$ \hspace{1cm} [N] \hspace{1cm} (1.53)

$$F_{p1} = F_{p2} = \frac{F_p}{2}$$ \hspace{1cm} [N] \hspace{1cm} (1.54)

$$F_p$$ is 0 \hspace{1cm} $$F_{c1} = F_{c2} = \frac{F_c}{2}$$ \hspace{1cm} [N] \hspace{1cm} (1.55)

Fig. 1.69 Components of cutting force during drilling
During drilling, as well as during turning, we can determine individual components of the drilling force for the whole tool using the following empirical equation:

$$ F_f = C_{F_f} \cdot D^{x_{F_f}} \cdot f^{y_{F_f}} \quad [N] \quad (1.56) $$

$$ F_c = C_{F_c} \cdot D^{x_{F_c}} \cdot f^{y_{F_c}} \quad [N] \quad (1.57) $$

- $C_{F_f}$, $C_{F_c}$ – constants expressing the effect of material being machined [-]
- $x_{F_f}$, $x_{F_c}$ – exponents expressing the effect of drill diameter [-]
- $y_{F_f}$, $y_{F_c}$ – exponents expressing the effect of the feed per revolution [-]
- $D$ – drill diameter [mm]
- $f$ – feed per revolution [mm]

If we use equation 1.55 and 1.57 for the calculation of torque moment, we will get the following equation:

$$ M_k = 2 \cdot \frac{F_c}{2} \cdot \frac{D}{4} = \frac{1}{4} \cdot F_c \cdot D = \frac{1}{4} \cdot C_{F_c} \cdot D^{x_{F_c}} \cdot f^{y_{F_c}} \cdot D = \frac{1}{4} \cdot C_{F_c} \cdot D^{(x_{F_c} + 1)} \cdot f^{y_{F_c}} \quad (1.58) $$

The simplified equation:

$$ \frac{1}{4} \cdot C_{F_c} = C_M, \quad x_{F_c} + 1 = x_M $$

$$ M_k = C_M \cdot D^{x_M} \cdot f^{y_{F_c}} \quad [Nmm] \quad (1.59) $$

To determine the cutting power in drilling, we can use the following equation:

$$ P_c = \frac{F_c \cdot v_c}{2 \cdot 60 \cdot 1000} = \frac{F_c \cdot v_c}{1,2 \cdot 10^3} \quad [kW] \quad (1.60) $$

$\nu_c$ – cutting speed can be calculated according to the equation (1.38) [m.min$^{-1}$]

### 2.3.4 Unit Machining Time

According to the Fig. 1.70, the unit machining time at drilling of through hole is expressed by the following equation:

$$ t_{AS} = \frac{L}{\nu_f} = \frac{l_n + l + l_p}{n \cdot f} \quad [\text{min}] \quad (1.61) $$

- $l_n$ – length of drill pre-travel [mm]
- $l$ – length of drilled hole [mm]
- $l_p$ – length of drill over-travel [mm]
- $\nu_f$ – feed speed [mm.min$^{-1}$]
\( n \) – drill revolutions [min\(^{-1}\)]
\( f \) – feed per revolution [mm]

For standard drills with the tip angle: \( 2\alpha_r = 118^\circ \):
\[
l_p = 0,5D \cdot \tan 31^\circ + (0,5 \div 1,0) \simeq 0,3D + (0,5 \div 1,0) \quad [\text{mm}]
\]

\( l_p = (0,5\div1,0) \quad [\text{mm}] \quad (1.62) \)

**Fig. 1.70 Drill path**

### 2.3.5 Tools - Drills

According to technology and the type of drilling, construction and geometry of the used drill, drills can be divided into several groups.

**Centering drills** (Fig. 1.68), known as well as drills for centre dots, are used to collar the hole beginning into solid material. Centering drills are often used for collaring the centre hole so that the material could be clamped into the turning lathe (tailstock).

**Twist drills** (Fig. 1.76) are the most common tools used for short-hole drilling. There are two opposite spiral grooves on the cylindrical tool body which are used to remove chips or to supply process fluid to the cutting area. Regarding the drills constructed for drilling steel and cast irons of standard strength and hardness, the spiral angle is \( 27^\circ \pm 5^\circ \). To drill into materials with high toughness (mild steel, aluminium alloy, thermoplastics), the spiral angle of the drills is greater (\( 42^\circ \pm 5^\circ \)). To drill into harder materials, the spiral angle of the drills is \( 12^\circ \pm 5^\circ \) (bronze, brass, hard rubber, plexiglass).

The main cutting edges of the drills are connected by a cross edge which has negative impact on working conditions (it increases the torque moment and feed force). This
is the reason why we try to remove the cross edge by various modifications (e.g. relief grinding) or the tool is constructed in such a way that the cross edge is removed.

1 – axis
2 – tool shank (conical, smooth cylindrical, smooth cylindrical with tappet)
3 – shedder
4 – taper
5 – body
6 – neck
7 – total length
8 – helical groove length
9 – helical groove
10 – second secondary tool flank
11 – width of secondary tool flank
12 – core
13 – core thickness
14 – first side tool flank (facet)
15 – facet width
16 – secondary edge
17 – reduce of secondary tool flank
18 – depth of reduce
19 – heel
20 – main tool flank
21 – head
22 – main edge
23 – tool edge
24 – external tip
25 – cross edge
26 – length of lateral edge
27 – length of main edge
28 – nominal diameter of drill
29 – diameter of reduce
30 – backward conicalness
31 – twist of spiral
32 – angle – helical groove
33 – angle of inclination – lateral edge
34 – tool tip angle
35 – tool flank angle

Fig. 1.71 Basic parameters of a twist drill

Twist drill core (0,25÷0,5 D) provides strength in torsion or in buckling. To ensure lower drill friction in the holes being drilled, the secondary tool flanks are reduced to smaller diameter and the drill body is slightly conically tapered towards the tool shank. Original size remains only on narrow surface (facet) at edge of each groove. Drills with a tip angle of $\varepsilon_r = 2\varepsilon_r = 118^\circ$ are used for ordinary non-alloyed steel of medium strength and cast iron of medium strength. Drills with a tip angle of $140^\circ$ are used for hard-machinable materials and drills with a tip angle of $90^\circ$ are used for plastics and hard rubber. It is also possible to
sharpen the drill with a double angle, e.g. 90° and 120°. It reduces the tool wear as heat stress is reduced. It is mainly used for materials with worse machinability.

Variable tool angles of the tool flank and tool face along the main edge indicate that twist drills have relatively complicated geometry of the edges. Angles of the tool flank and tool face along the main edge is affected by relieving of the main tool flank which can be carried out according to conical, cylindrical, helical or plane surface (Fig. 1.72).

![Fig. 1.72 Ways of relieving the flank surfaces](image)

Material for the twist drills usually is: high-speed steel, with soldered inserts from sintered carbide for harder machining conditions, sintered carbide for drills of monolithic material without coats or with coats against the wear resistance, mostly based on TiN (Fig. 1.73). Twist drills can have a central hole for coolant (Fig. 1.74) and they are also produced with three edges (Fig. 1.75).

![Fig. 1.73 Monolithic drill with TiN coat](image)

![Fig. 1.74 Drill with central supply of cutting fluid](image)
Lanceolate drills (Fig. 1.81) are double-edge tools with transverse cutting edge and with external chip ejector. These drills have high rigidity and they allow drilling of holes into solid material with diameter of 10 ÷ 128 mm. It is important to keep the ratio of length and diameter – the maximum ratio $L/D = 3/1$. At present, most of these drills are manufactured with internal supply of process liquid.

Current types of these drills are formed by a body into which the indexable cutting inserts of special shapes of high-speed steel or sintered carbide are clamped. The tool angle of the main edge is usually $\alpha = 66^\circ$. Chip removal is ensured by separating grooves on the both main tool ridges. There are facets created on the main tool flanks of inserts which reduce the friction. Roughness of the machined surface is worse than after using the twist drill.

Drills with interchangeable tips are produced in the form of cutting insert (Fig. 1.77) or head (Fig. 1.78). Sintered carbide (usually coated) is usually used as material for production of the cutting inserts and head. The heads are available with various geometry depending on the material being machined and other technological requirements.
Drills with indexable cutting inserts (Fig. 1.79) – inserts of sintered carbide are clamped in the holder body by screws directly or using cassettes (for larger drills) that facilitate the exchange and protect the tool bed against the tool wear. Because of different cutting speed, coated cutting inserts of sintered carbide and centred inserts of non-coated cemented carbide are used for some drills. Tool life of the selected inserts is approximately similar and the used inserts are exchanged at the same time.

Adverse working conditions (resulting from drilling) can be improved by using different inserts having chip formers (it solves the problems with chip division, thermal and mechanical load, etc.). Most of drills with indexable cutting inserts have central coolant and can be used for turning internal and external cylindrical surfaces.
Gun-drills and barrel drills are suitable for machining of less deep holes as the tool has to be taken out after cutting certain depth so the chip can be removed from the hole. Barrel drills (Fig. 1.81) are suitable for drilling the more accurate holes. Regarding the tool construction: the cutting part of high-speed steel or sintered carbide is soldered to a pipe or a rod of the required length. There are also tools with the soldered cutting inserts. Reliable drill centering is ensured by leads soldered to a windlass body. Process liquid is fed to a cutting place by holes in the drill body and it provides leaching of the created chips (Fig. 1.82). Specially modified turning lathes (Fig. 1.83) are used for machining by gun-drills and barrel drills. These drills can drill holes of diameter of 4 ÷ 250 mm and length up to several meters.
Ejector drills are suitable for drilling holes whose depth exceeds quintuple of diameter. They consist of a drilling head (Fig. 1.84) screwed into the outer drill pipe. Ejector heads are produced with indexable inserts. Process liquid comes to the cutting point through annulus between the outer and inner tube. A small amount of it goes through slots in the rear part of inner tube which causes the ejector effect – liquid is sucked away from the drill edges.
(Fig. 1.85). For machining, we use conventional machine tools (Fig. 1.86), CNC turning-lathes as well as machining centers.

**Fig. 1.84 Ejector drills - heads**

**Fig. 1.85 Principle of ejector drilling**

**Fig. 1.86 Equipment for ejector drilling**

**BTA or STS drills** (Fig. 1.87) can be used for a wide range of drilled diameters than ejector drills. They can be used for machining of the pre-machined hole, drilling into solid material as well as for the “on kernel” method which is their advantage. Holes into solid material can be drilled up to the diameter of 180 mm and “on kernel“ method can be applied up to the diameter of 120 ÷ 300 mm. The principle of chip removing from the cutting area is in bringing the process liquid through the space between the wall of drilled hole and the drill tube. The process liquid, together with the chip, is taken away through the center of tube (Fig. 1.87). The mechanism must be equipped with seal (Fig. 1.89).
There are also special types of drills such as the drill sheet (Fig. 1.65), stepped drills for drilling two or more diameters simultaneously (Fig. 1.66) and special associated tools (Fig. 1.67).
2.3.6 Core-Drilling

After drilling, the holes usually have a high surface roughness and unusable geometric parameters: e.g. inaccurate roundness and cylindricity, infringement of specific diameter, and other. Holes are improved during finishing operations such as core-drilling and reaming. These operations refine on the hole shape and decrease the surface roughness.

Reaming is used only for small holes (up to 10 mm). Larger holes are core-drilled in advance and then reamed which implies that core-drilling usually is not the last machining operation as reaming follows. Core-drilling is used to specify the geometric requirements. Reaming is used to complete the accurate hole with all the required geometric parameters and hole surface roughness. While manufacturing, we have to respect the allowance of core-drilling and reaming. Their size depends on type of the machined material, required quality of the machined hole and tool construction.

1 – cutting cone
2 – body
3 – clamping shank
4 – clamping hole
5 – brazed cutting inserts from sintered carbide

Fig. 1.90 Core-drills a) Shank-type b) Shell-type

Fig. 1.91 The shape and geometry – tooth of core-drill from high-speed steel
Conventional core-drills are three-edge or four-edge (exceptionally five-edge) tools with edges in a form of a helix (Fig. 1.90). Shank drills are used for core-drilling of diameters up to 30 mm and shell-type core-drills are used for larger diameters (Fig. 1.90b). The tool body is usually made of structural steel and the cutting part is of high-speed steel and it is welded to the tool body. There are also core-drills with soldered cutting inserts of sintered carbide.

Fig. 1.91 demonstrates the edge geometry of a core-drill of high-speed steel. Recommended values of the edge geometry are shown in Table 1.1. The length of cutting cone is selected on the basis of tool diameter which is \( l_1 = 1+3 \text{ mm} \), length of lead part \( l_2 = (0,75+0,8)l \) and facet width \( b_a = 1+3 \text{ mm} \). The edge geometry for a core-drill of sintered carbide is \( \omega \) (angle of helix inclination) = 10\(^\circ\), \( \gamma_o \) (tool face angle in orthogonal plane) = 5\(^\circ\). Other parameters are the same as for core-drills of high-speed steel.

### Tab. 1.1 Recommended geometry for core-drills of high-speed steel

<table>
<thead>
<tr>
<th>Machined material</th>
<th>Angle</th>
<th>( \omega )[(^{\circ})]</th>
<th>( \alpha )[(^{\circ})]</th>
<th>( \kappa )[(^{\circ})]</th>
<th>( \kappa' )[(^{\circ})]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steels Rm=600÷800 MPa</td>
<td>8</td>
<td>12</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Steels Rm=800÷1200 MPa</td>
<td>0</td>
<td>5</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Grey cast irons</td>
<td>6</td>
<td>8</td>
<td>5</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>Chilled cast irons</td>
<td>0</td>
<td>5</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Aluminium alloys</td>
<td>2</td>
<td>30</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

At core-drilling and reaming on the drilling machines, the main motion is performed by the tool and feed is performed by the workpiece. On the turning lathes, the main motion is performed by the workpiece and feed is performed by the tool.

### 2.3.7 Reaming

It is a finishing operation used for manufacturing of precise holes. It ensures the required surface roughness and geometric parameters. For high-quality reaming, it is necessary to ensure sufficient allowance to prevent extrusion of material instead of cutting it. In this case, the hole would not reach the desired circular diameter and the tool life would be
shorter. To calculate the diameter with allowance for reaming, we can use the following equation:

\[ p = 0,1 + 0,005 \cdot D \]  \hspace{1cm} [\text{mm}]  \hspace{1cm} (1.63)  

\[ D \] – specific diameter of hole being reamed [mm]

To ensure high accuracy of reamers, the edges have to be as sharp as possible with radius of edge curvature smaller than \( r = 10 \, \mu m \). It can be achieved by careful grinding and lapping of edges.

There are mechanical and manual reamers. The teeth are straight or helical and they are composed of cutting part and cylindrical part. Regarding the \textbf{manual reamers} whose tool shank is ended by a foursquare for wrench, a cone cuts with inclination of \( 1 \div 3 \, ^\circ \). The \textbf{machine reamers} have a conical or cylindrical tool shank of larger diameter. There are stem tools and shell tools. Material is cut off by a cone of \( \kappa_r \) angle. To meet demanding requirements for roundness and surface quality, reamers are produced with irregular distance of teeth. The number of teeth is usually 4 – 18 (it depends on the diameter).

![Fig. 1.92 Manual reamer – shape and geometry](image1)

![Fig. 1.93 Machining reamer – shape and geometry](image2)
Expanding reamers are designed for repairs and renovation of mechanical parts. The body of reamers is hollow and cut up between individual teeth in the longitudinal direction. While sinking the cone into the cone hole in the body, there occurs expansion and the diameter of teeth circle increases. There are also adjustable reamers with teeth sliding in grooves on the conical surface of the body. Increasing and decreasing of the teeth circle is carried out by moving teeth in one or the other direction. On the body of single-edge reamers, there is a mechanically attached insert and 2 or 3 guides, the both of sintered carbide. Holes with the diameter of 15 ÷ 80 mm are manufactured using the peeling reamers allowing the cutting speed of $v_c = 15 + 20$ m.min$^{-1}$ and feed per revolution of $f = 0,4 + 1$ mm.

Tab. 1.3 Recommended values – flank angle and face angle for machining reamers of high-speed steel

<table>
<thead>
<tr>
<th>Machined material</th>
<th>$\alpha$ [']</th>
<th>$\gamma$[']</th>
<th>Inclination of teeth</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steels $R_m = 700$ MPa</td>
<td>5 ÷ 7</td>
<td>0</td>
<td>straight teeth</td>
</tr>
<tr>
<td>Steels $R_m = 1000$ MPa</td>
<td>5 ÷ 6</td>
<td>3 ÷ 5</td>
<td>left helix $\omega = 6^\circ$</td>
</tr>
<tr>
<td>Austenitic steels</td>
<td>5 ÷ 6</td>
<td></td>
<td>right helix $\omega = 6^\circ$</td>
</tr>
<tr>
<td>Aluminium alloy, wrought</td>
<td>8 ÷ 10</td>
<td>8 ÷ 10</td>
<td>left helix $\omega = 10^\circ$</td>
</tr>
<tr>
<td>Aluminium alloy, casting</td>
<td>10 ÷ 12</td>
<td>0</td>
<td>straight teeth</td>
</tr>
</tbody>
</table>
2.3.8 Recessing

Recessing is used to machine holes for heads of recessed screws and to machine coaxial cylindrical or conical recess. It also includes the face alignment where we can use flat double-edge countersinks. According to the surface shape, we differentiate cylindrical, conical and flat countersinks. There are cylindrical countersinks with a tool shank or shell-type countersinks. Conical countersinks have a self-centering effect. Cylindrical and flat countersinks are kept in a pre-drilled hole by a pilot pin. Countersinks are produced / manufactured with straight or helical teeth and there usually are 4 in a helix. Countersinks are made of high-speed steel or sintered carbide (uncoated or coated) or are produced with indexable cutting inserts.

Fig. 1.95 shows a reverse countersinks which are used for machining in inaccessible surfaces. Their cutting part is located eccentrically towards the clamping shank. The tool is inserted into the pre-drilled hole. The workpiece is moved in the perpendicular direction to the spindle axis in the value. The required machining is created by reverse axial feed.
Fig. 1.96 shows reverse countersink which uses the principle of hinged arm with a mechanically clamped indexable cutting insert.

### 2.3.9 Drilling Machines

Drilling machines are often used for drilling, reaming, core-drilling and recessing. Holes can be also carried out on turning lathes, horizontal boring machines and machining centres. Size of drilling machines is divided according to the maximum diameter which can be made on the respective machine into solid material of medium strength steel. According to their construction, drilling machines can be divided as follows:

- manual,
- bench,
- pillar, rotary,
- horizontal for deep holes,
- special.

**Bench drilling machines** (Fig. 1.97) are of the simplest structure. Spindle feed on the short pole is manual and the work-table height is easily adjustable. The stepped pulley is used for changing the speed. Bench drills can drill holes of up to 20 mm diameter.
Pillar drilling machines (Fig. 1.98) usually have a vertical headstock sliding on worktable as well as on the pillar which is their basic structural element. Drill includes a gearbox by which the spindle speed can be regulated. Its movement is mechanical. Smaller workpieces are clamped on the table; bigger workpieces are clamped directly on the base plate of drilling machine.

Rotary drilling machine (radial) (Fig. 1.99) can be used for almost all drilling operations, mostly for drilling into bigger and larger parts. Their characteristic feature is a shoulder on which the work spindle moves in horizontal direction.

Mounted drilling machines are portable and their great advantage is that they can machine holes into heavy components. In principle, it is a special type of rotary drilling machines. It is possible to set an arbitrary angle on the spindle shoulder and to rotate the shoulder by 360°. It allows the drilling operation in a large space around the machine drill.
Fig. 1.99 Rotary drilling machine

**Special drilling machines** are designed for special drilling operations. There are included: drill for deep holes drilling, multi-spindle drilling machines, modular drilling machines, CNC drilling machines, etc.
3. COMPUTER AIDED MACHINING – INTRODUCTION AND STRUCTURE OF PRODUCTION PROCEDURE

Computer Aided Machining with the use of CAD / CAM systems for programming computer-controlled machine tools. The textbook describes basic principles and methods of manufacturing on CNC machines.

**CAD** (*Computer Aided Design*)

It is a design of new components when the whole geometry is interactively modeled and displayed in a real form. Therefore, it is a sum of means for creating geometric models. The information representing the geometric model is stored in the application compiled database which is the basis for further steps in complex engineering solution of the new model design.

**CAM** (*Computer Aided Manufacturing*)

CAM refers to a system that prepares data and programs for the management of numerically controlled machines for the automatic production of parts. This system uses geometric and other information created in the CAD system. The main application of CAD / CAM systems in manufacturing are in the area of producing molds, dies and other components of complex shapes in various sectors of machinery industry (mainly space, aerospace and automotive). This well-known fact about the use of the privilege earlier era is the period of early introduction of CNC systems and CAM systems. Today CAD / CAM systems can also be applied in normal engineering production. Please see the following chart for the most frequently used terms:

This chapter briefly outlines what this script deals on. It describes a basic process of manufacturing components for a CNC machine from the very idea to the finished product.

![Fig. 3.1 Process of the product development using the CAD / CAM systems](image-url)
Table 1 The most frequently used terms

<table>
<thead>
<tr>
<th>Marking</th>
<th>Meaning of the abbreviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAA</td>
<td>Computer Aided and Assembly</td>
</tr>
<tr>
<td>CAD</td>
<td>Computer Aided Design</td>
</tr>
<tr>
<td>CAE</td>
<td>Computer Aided Engineering</td>
</tr>
<tr>
<td>CAQ</td>
<td>Computer Aided Quality</td>
</tr>
<tr>
<td>CAM</td>
<td>Computer Aided Manufacturing</td>
</tr>
<tr>
<td>CAPE</td>
<td>Computer Aided Production Engineering</td>
</tr>
<tr>
<td>CAPP</td>
<td>Computer Aided Process Planning</td>
</tr>
<tr>
<td>CAPPS</td>
<td>Computer aided part programming and production control</td>
</tr>
<tr>
<td>CARC</td>
<td>Computer Aided Robot control</td>
</tr>
<tr>
<td>CATS</td>
<td>Computer Aided Transport and Store</td>
</tr>
<tr>
<td>CIM</td>
<td>Computer Integrated Manufacturing</td>
</tr>
<tr>
<td>EDM</td>
<td>Economic Development Manager</td>
</tr>
<tr>
<td>PDM</td>
<td>Programming Development Manager</td>
</tr>
<tr>
<td>PLM</td>
<td>Product Lifecycle Management</td>
</tr>
<tr>
<td>PPS</td>
<td>Production Planning Systems</td>
</tr>
<tr>
<td>PPC</td>
<td>Production Planning and Control</td>
</tr>
<tr>
<td>PMS</td>
<td>Production Management System</td>
</tr>
<tr>
<td>MRP II</td>
<td>Manufacturing Resource Planning</td>
</tr>
</tbody>
</table>

The structure of components in the CAD/CAM systems that accompany creating of the product from initial design phase to the final stage of production resulting in a specific product. The sequence of these activities is shown schematically on the following figure.
3.1 Programming of numerically controlled machine tools

Programming of CNC machine tools can be done in the following ways - literature and practice show some possible breakdown:

- Online - programming directly to the CNC machine shop floor programming (SFP - Shop Floor Programming).
- Offline - creating NC program outside of the control system:
  - by hand (writing with the ISO code)
  - using the CAM system.

- Hand programming,
- Automatic programming.

- Direct writing of NC code,
- Using the geometric programming languages,
- Using the CAM system,
- Using the CAD / CAM system.
4. WORKING PROCEDURE IN CAM SYSTEM

The machining process is a gradual removal of material from a workpiece until the final shape of the designed components is reached.

- Import of parts, clamps and workpiece
- Rough strategies
- Pre-finish strategy
- Finishing strategy
- Strategy - machining of the residual
- Output of the toolpaths into the machine

Simulation of removing the material

*Fig. 4.1 Overview of steps in creating a technological process in CAD / CAM system*
4.1 Geometric part of CAD / CAM system

The basic operations used in the geometric part:

- treatment of corrupted data transmission,
- treatment of inaccuracies caused by entry form,
- determine face and back of machined surfaces,
- suitable orientation and continuity of geometrical shape of the machined features → eliminate unnecessary tool crossings,
- digital adjustment of the model in terms of machining technology - such as defining the parting line, addition of technological benefits for clamping, etc.,
- orientation of the model in the workspace (in the coordinate system), set the system origin,
- definition of stock,
- definition of clamping elements,
- location of the workpiece model to the semi-product model – definition of machining allowances,
- definition of the coordinate system bond of the machine tools and the workpiece,
- suitable utilization of the machine tools,
- measuring of the model geometry to define machining.

4.2 Technological part of CAD / CAM system

The basic operations used in the technology part are the following:

- setting of initial conditions (selection and setting of the machine (postprocessor option),
- definition of a reference point, selection of the workpiece material, selection of the clamping method (clamps, vise, chuck, etc.),
- selection of the first tool, i.e., usually roughing tool, and definition of its parameters and tool holder,
- definition of the operating cycle with the necessary parameters,
- selection of an additional tool and definition of its tool path and cutting conditions,
- continuation of the previous step,
- simulation of machining and subsequent setting correction,
- generation of the NC program,
- creation of accompanying documents: sheet of operating cycles, machine setting and tool sheet (with tool corrections - compensations) etc.,
- debugging of the NC program in simulation software,
- debugging of the NC program on the CNC machine tool.
Work in the CAM system is determined by an order of individual tasks and instructions into a tree, as they actually follow. The individual items are recorded in the instruction window. Examples include the following instructions:

- tool movement to the reference position,
- tool movement to replacement,
- selection of instruments,
- spindle on / off,
- selection of cooling type
- chuck open/close,
- chuck pressure,
- inputting and canceling of the tool corrections (Length Offset),
- input of machining cycles,
- input of speed-feeds,
- moves the tool at the rapid rate,
- moves the tool at the specified feed rate,
- input of delays (program Dwell),
- rotation of the table or the tool head,
- movement of the tailstock and quill,
- Insert a comment,
- bar feed cycle,
- selection of clamps,
- Insert NC Code,
- safe distance,
- synchronize turrets,
- part catcher,
- putting exact stop,
- selection of other machine functions (M-functions), etc.

All of these instructions can be edited, copied, pasted, deleted (erased), moved, as in other commonly used software. It is also possible to simulate them separately.
Fig. 4.2 Example of individual instructions in a tree in EdgeCAM system

Fig. 4.3 Example of individual instructions in a tree in MasterCAM system
5. IMPORTING GEOMETRICAL DATA INTO CAM SYSTEM

More complicated geometric shapes are preferably imported into the CAM system from CAD systems that are user-friendly software. The most frequently used CAD systems are AutoCAD, Inventor, SolidWorks, Pro-Engineer, CATIA (used by many companies in the automotive industry) and other.

Another option is to create part geometry (geometric elements representing individual component, futures, individual surface or solid model) directly in the CAM system. The advantage is to eliminate potential errors when importing the components from the other software.

Note

Geometric data may not represent only the 3D part models. They can form a variety of geometric shapes (lines, circles, arcs, curves), surfaces, features, profiles, contour lines and components, etc.

5.1 Location of objects into the appropriate positions

After importing the objects into the CAM, it is necessary to place the object or objects in the appropriate location. For this, transformation commands such as translation, rotation, etc. are used. Every CAD/CAM system has its own universe. They all have a World Zero, Master Coordinate System, System Origin, etc. [1]. Just like machine tools, all these locations use different names.

When working with solid models, it is advantageous to use special commands for the model positioning, model turning and milling.

It is necessary to provide a suitable location for turning and positioning of the parts to the workpiece zero point (W). This zero point is placed according to certain rules and conventions of gender stereotypes. The most frequent location of the workpiece zero point during turning is at the face (forefront of components) in the workpiece axis.
The most frequent location of the workpiece zero point during milling is the highest point (in the corner, in the riser axis, etc.) of the machined parts.

Reference points of CNC machines:

- $R$ - reference point,
- $W$ - system origin (workpiece zero point),
- $C$ - starting point of the program,
- $A$ - stop-point,
- $F$ – tool zero point,
- $M$ - machine zero point.
Fig. 5.3 Coordinate system (the axis orientation) during turning

Fig. 5.4 Location of the model to the starting position for milling
Fig. 5.5 Location of the model to the starting position for milling

To learn

Zero point and other reference points on the CNC machines belong to basic knowledge of programming CNC machine tools.
6. CREATING OF STOCK AND FIXTURES

Semi is the default form for the following machining. The basic principle is to draw a stock or fixtures (clamping elements), and then to import them into the CAD system. Simpler forms can be quickly drawn directly in the CAM system, however, more complex shapes such as castings and forgings should be imported from a powerful CAD system. Real stock and fixtures (clamping elements) allow detecting any collision between them and the cutting tools.

Creating of a stock and fixtures can be done in the following ways:

a) to import them from the CAD / CAM system,
b) to draw them directly into the CAM system,
c) to create a stock "automatically",
d) to create a stock from an already created toolpath
e) to use a stock model from machining simulation.

ad a) When importing fixtures from the CAD / CAM system, it is advantageous to build a database for each machine tool.

Fig. 6.1 Stock - Import Model
Fig. 6.2 Stock-casting

Where:
1 - feature for turning,
2 - starting point for machining feature,
3 - cycle starting point,
4 - feature of a stock.

ad b) Simple shape of a stock or fixtures (clamping elements) can be quickly drawn directly in the CAM system.

ad c) Creating stock can be very easily automatically installed such as just typing an addition to working to the turned shaft or the milled surface, etc.

Fig. 6.3 Creating a stock automatically - turning operation
ad d) Another way how to create a stock is using already created toolpaths, i.e. using the tool paths from the previous operation cycle. This method can minimize tool motions idle.

ad e) In cutting simulation, there can be created a stock as a result of the simulation end. The machined part can be saved as the stl file.
7. TURNING IN CAM SYSTEM

Most of the CAM systems can generate tool paths of unpowered and powered tools at turning of the assigned geometric shapes. Classification of turning in CAM systems is not generally defined. However, at least a basic classification of turning in CAM systems can be introduced as follows:

- 2–axis turning,
- 4–axis turning (turning with the second, eventually with the third tool head),
- turning with powered tool (with C, Y and B axis),
- turning with secondary spindle (eventually with more spindles).

7.1 2-axis turning

In the 2-axis turning, there are used lathes with the drive control in two axes (mostly z and x) with a tool head.

7.1.1 Basic menu of machining cycles for 2–axis turning

Basic types of the used menu of machining cycles and operating sections for the 2-axis turning are the following:

- straight (rectangle) turning,
- face turning,
- rough turning,
- finish turning,
- profile turning,
- groove turning,
- pocket turning,
- thread turning,
- hole machining,
- parting off (cutting off),
- rest cycle (residual turning),
- hand setting of tool motion.

Tool paths created according to the above mentioned menu can be edited (mirrored, moved, copied, rotated, etc.).
7.1.2 Basic setting data for individual machining cycles

Basic settings in operating sheets of machining cycles are the following:

- inputting of cutting conditions and their editing in case they are automatically selected by the system from the database of cutting conditions,
- depth of cut (depth of cut in % of cutting edge, setting of constant depth of cut or variable depth of cut),
- start-up of the cut (angle, length, radius, percentage of feed, etc.),
- getting out of the cut (angle, length, radius, the percentage of feed, etc.),
- delay of spindle speed, etc.
- specification of the distance between cuts (cut increment, stepover).
- typing of crossing between tool paths, etc.

Standard sequence for turning operation

- coolant (off/on),
- spindle (on/stop),
- type of movement to the tool exchange (rapid to tool exchange), etc.

7.1.3 Straight (rectangular) turning

Straight (rectangular, orthogonal) turning is a turning in the radial or axial direction.

Machining methods:

- face (along X axis),
- longitudinal (along Z axis),
- at specified angle relative to the Z axis.

This cycle usually requires two opposite corners of the rectangular area.

Fig. 7.1 Method of setting of the place of the material removal
In the EdeCAM, it is easy to use Operation. Menu “operations” are for beginners. How to use the Turn Operation:

1. Click on the Turn Operation button.
2. Digitize the profile to be turned and then perform a Finish.
3. Digitize the new start point for the active profile (if necessary). Press the Ctrl key while holding down the left mouse button to toggle between selecting a new start point and a new end point for the profile.
4. Perform a Finish.
5. Digitize the cycle start point.
6. Digitize a toolpath or a continuous entity to act as a billet for the cycle (if necessary).
7. Perform a Finish to complete the billet selection.
8. Complete the dialog box for the operation and click OK.

The EdgeCAM now generates the toolpaths for the turning operation including any movements to the tool position.

7.1.4 Rough turning

Rough machining is used to remove material from an irregular workpiece. Roughing can be defined by the coordinates (X, Z), geometric elements or a window.

7.1.5 Profile turning

Profile turning means roughing and finishing of the workpiece profile according to a specified profile (contour of the part).

7.1.6 Pocket turning

Pocket turning is a special menu for roughing and finishing of the pocket shape. It is not available in some CAM systems and, therefore, to turn various types of pockets, the menu of a rough cycle, finishing cycle or profile turning is used.
7.1.7 Groove turning

Groove turning allows roughing and finishing of a groove. A variety of methods with the unpowered tool can be used.

It is necessary to correct the cycle – to insert one or two spindle revolutions as the part must finish its cut at depth to ensure all cutting has finished. If the cut is carried out within only one revolution, the material is left uncut as the tool moves to depth, as shown in this end-on view:
Using the Groove Operation in the EdgeCAM:
1. Click on the Groove Operation button.
2. Digitize the profile to be grooved.
3. Perform a Finish to complete digitizing of the profile.
4. Digitize the new start point for the active profile (if necessary). Hold down the Ctrl key while selecting the left mouse button to toggle between selecting a new start point and a new end point for the profile.
5. Perform a Finish to complete digitizing of the new start point.
6. Digitize the cycle start point.
7. Complete the dialog box for the operation.
8. Click on OK.

The EdgeCAM now generates the toolpaths for the groove operation, including any movements to the tool position.

7.1.8 Thread turning

Thread turning also enables to rough and finish different types of external and internal threads. Thread can be produced on a cylinder and on a cone. A line segment is the most common setting geometry for the thread turning. Other setting geometry can also be a profile or two points.

7.1.9 Hole machining

Hole machining means either only a hole drilling or a sum of operating segments comprised of hole drilling, pre-drilling, drilling, countersinking, core-drilling and reaming or thread making (tapping).

A point is the most common setting geometry for drilling, though it can be a feature, too.

7.1.10 Parting off

Through this menu, the workpiece is cut off from the stock. A line segment is the most frequently used setting geometry, though it can also be a feature or two points. It is necessary to set driving points on the edge. The set point of the tool is configured so that the rear face of the component can be parted off. The set point of the tool is configured to part off on the front face of the component, as shown on the next figure.
Other setting parameters within the parting off are the following:

- possibility of chamfers,
- crossing the axis,
- diameter for moving on, etc.

### 7.1.11 Residual turning

Within this menu, material that remained after the previous operation cycle is removed. A feature is the most frequently used setting geometry for residual turning (in the EdgeCAM indicated as roughing). The toolpath of the previous roughing cycle is marked as a stock. The CAM system connects the residual turning to the previous roughing cycle.

Please, see the next figure: in the left part, you can see the toolpath at roughing and, in the right part, you can see the following residual turning.
8. MILLING IN CAM SYSTEM

A classification of milling strategies, operating segments and menu of machining cycles in CAM systems is not generally defined. However, we can introduce at least the basic terminology used in the CAM systems:

**Machining cycles for 2.5-axis milling:**
- milling of flat surfaces and facet surfaces,
- roughing,
- milling of feature, contour and casting (offsetting, parallel routing),
- pocket milling,
- finishing of flat surfaces,
- hole machining,
- drilling on the perimeter,
- pre-drilling of holes for plunge milling,
- chamfer milling,
- groove milling,
- text milling,
- gravering,
- bore jet,
- manual milling,
- 2.5-axis residual milling,
- plunge milling, etc.

**Note**
At the 2.5-axis machining, the tool moves in two axes (e.g., X and Y), while the remaining axis (e.g., z-axis) is fixed.

**Machining cycles for 3-axis milling**
- roughing (in layers),
- finishing,
- cycle profiling
- surface milling,
- parallel lace cycle
- Constant Cusp Finishing (milling at constant roughness),
- milling in z layers,
- orbital milling, spiral milling,
- projection cycles,
- milling according to the control curve,
- text milling,
- rest finishing cycle (milling of residual material),
- pencil milling - cutting corners,
- groove milling,
- hole pocketing,
- 3-axis drilling
- thread milling, etc.

**Note**
At the 3-axis machining, the tool can move simultaneously in all three directions.

**Multi-axis machining cycles:**
- 4-axis rotary machining,
- 5-axis circuit,
- 5-axis finishing,
- 5-axis surface milling,
- 5-axis drilling,
- 5-axis projected milling
- 5-axis groove milling, 5-axis slot milling, 5-axis grooving on the surface,
- 5-axis milling along the curve,
- and more.

**Note**
Compared to the 3-axis machining, at the 4-axis machining, the tool movement is complemented with a rotary table or tilting tool.

**Note**
At 5-axis machining, the tool can move simultaneously in five axes. The surface can be machined in any direction. This can be carried out through continuous indexing of a headstock, workpiece, or if we divide the turning between the headstock and the workpiece.
Depending on the type of the selected machining cycle, it is necessary (before or after completion of the operating certificate) to gradually choose:

- machined geometry (points, line segments, profiles, contours, surfaces, solid models, etc.)
- semi-product
- machining limits
- position of auxiliary elements (start and end of cycles, etc.)
- limiting elements, etc.

On the workpiece surface, there can be indicated (draw) the machining limits in which the machining will be or will not be carried out.

The order of machining cycles can be changed, they can be edited, it is possible to insert new cycles into the already created, or to insert various instructions and commands into them. The toolpaths created according to the above mentioned menu of machining cycles can be edited (mirrored, moved, rotated, copied to a different depth, etc.)

The following figures show examples of kinematics milling centers.

![Fig. 8.1 The 3-axis and 5-axis milling center](image-url)
Fig. 8.1 The Prima - TOS Varnsdorf horizontal milling center

Fig. 8.2 The Mori Seiki GV 5035AX vertical machining center

Fig. 8.2 The MCU 630 - 5x milling center
Basic setting data for individual machining cycles

Basic setting data in operating sheets include the following:

- inputting of cutting conditions and their editing in case they are automatically selected by the system from the database of cutting conditions,
- milling method: climb milling, conventional milling, combined milling (spacing),
- setting of the tool movement angle,
- setting of various milling strategies,
- determining of depths,
- setting of the crossings,
- start-up into the cut (angle, length, radius, feed percentage, etc.),
- getting out of the cut (angle, length, radius, feed percentage, etc.),
- delay of spindle speed, etc.

![Diagram of depth determining methods](image)

Fig 8.3 Methods of depth determining

- Clearance plane (Crossing) - an absolute value in which (or above which) there is no collision possibility of the tool and workpiece (clamp) at the tool rapid feed.
- Level Absolute (Measuring plane) - an absolute value that indicates the position from which the machine starts the machining process (default level).
- Retract plane – an incremental value measured from the measuring plane. It indicates the plane position into which the tool goes within the engagements.
- Depth – an incremental value measured from the measuring plane. It specifies the hole depth.
### 8.2 2.5-Axis Milling

In the 2.5-axis (2-1/2-axis) machining, the tool can move in two directions simultaneously (X and Y - axis) and a movement in another axis (Z-axis) is limited to setting of a fixed level. Basic strategies in 2.5-axis milling will be described in the following text.

![Fig. 8.4 2.5-axis milling simulation](image)

#### 8.2.1 Plane milling and face milling

Plane and face milling are used to remove material from the workpiece plane area. Setting geometry for defining the place from where the material will be removed can be surfaces, profiles, simple 2D geometry (e.g. rectangles), etc.

![Fig. 8.5 Toolpath - concentric and spacing](image)
8.2.2 Roughing - 2.5 D milling

Roughing cycle is used for comprehensive cutting of the material. It is carried out in successive engagements in the Z axis (in the Z-layers), while the toolpath in the engagements is derived from the shape of the machined area or stock.

A solid model is the most often used setting geometry. Also various combinations of pockets (cavities) and risers can be selected.
8.2.3 Profile milling

With the profile milling, open and closed regions can be machined so that the tool moves along one side of their contour (or along its center).

Features, contours and various components of 2D geometry (simple and complex shapes in the plane), etc. are the usual setting geometry. The selection is mostly defined by the contour shape and closed 2D boundary of the stock.

Profiling is usually used as a finishing operation, e.g. after pockets milling or face milling.

Fig. 8.8 Example of tool path - 2.5 D roughing

Fig. 8.9 Profile milling
8.2.4 Pocket milling

A cycle of pocketing usually comprises functionality of pocketing and profiling. It is possible to carry out Pocketing - Roughing and Profiling - Completing in one menu of the operating section.

The residual material after roughing can be automatically removed during finishing without setting how much material is necessary to cut.

Methods of pocket milling (cavities):
- one-way (one way),
- zig-zag,
- toward the center (concentric IN),
- away from the center (concentric OUT).

Fig. 8.10 Pocket milling - toolpath one way strategy

Fig. 8.11 Pocket milling - toolpath Zig - Zag strategy
At pocket milling, it is very important to define how the tool enters the material.

**Fig. 8.14** Entering the material at an angle

CAM systems offer the possibility to set the definition of side wall bevel.

**Fig. 8.15** Definition of chamfer side walls
**Must remember**

In milling of pockets (cavities) in the CAM system, it is possible to specify the pocket shape (profile), which will be applied to the side walls. I.e., it is not necessary to prepare a model with a complicated cavity shape.

![Image](image1.png)

*Fig. 8.16 Determination of the cavity profile (pockets)*

In the same way, it is possible to specify a riser profile (shape), as shown below.

![Image](image2.png)

*Fig. 8.17 Determination of the riser profile (pocket)*

### 8.2.5 Circular milling – hole pocketing

This strategy is also known as a hole pocketing. It is used to create holes of large diameters instead of drilling. The entering tool movement can have a shape of a helix with a specified angle or pitch.

![Image](image3.png)

*Fig. 8.18 Hole pocketing*
8.2.6 Thread milling

This machining cycle is used to cut the threading using thread milling tool. The thread pitch is a part of the tool definition - screw cutter. It is possible to cut inner, outer, right or left thread. Threads can be cut in one pass or more passes with the specified value of the side step.

8.2.7 Groove milling (Slot cycles)

The menu of groove milling is used to slot grooves of different types in a plane and in a space. Setting geometry for defining the place from where the material will be removed can be geometry of the tool center movement or solid model. For the groove milling, it is also possible to use the menu of Profile milling (Contour milling).
Using sloting operations in EdgeCAM

1. Click on the Slot Operation button.
2. Digitize slot. Perform a Finish to stop selecting entities to slot.
3. Digitize a new start and end point for the active profile (if necessary).
4. Hold down the Ctrl key while selecting the left mouse button to toggle between selecting a new start point and a new end point.
5. Perform a Finish.
6. Complete the dialog box for the operation. Click on OK.

### 8.2.8 Chamfer milling

The menu of chamfer milling (also known as folding) is used for chamfering on solid models. Setting geometry for defining the place from where the material will be removed mostly is a solid model.

![Fig. 8.22 Simulation of chamfer milling](image)

For chamfer milling, it is also possible to use the menu of profile milling, as can be seen on the following figure.

![Fig. 8.23 Chamfer with the use of profile milling](image)
8.2.9 Text milling

It is used to mill a text where the setting geometry is a text element. A text element can be created directly in the CAM system menu "create a text".

![Fig. 8.24 Text milling](image1)

Text can be machined using more methods (concentrically, spacing, or toolpath by profile).

8.2.10 Engraving milling

Engraving is typically used to create a text or a logo on the finished model. It is possible to machine open, closed, 2D or 3D regions. This type of machining is similar to cutting along a curve when the curve is projected onto component surface that lying below the region which is being machined.

<table>
<thead>
<tr>
<th>Must remember</th>
</tr>
</thead>
<tbody>
<tr>
<td>Conical engraving tool is used for engraving.</td>
</tr>
</tbody>
</table>

The following figure shows a typical tool path for engraving. In the corners, the tool sets off upwards to fully use the smaller tool diameter.

![Fig. 8.25 Engraving](image2)

Many templates for engraving are made in specialized systems that use multiple text fonts. There are special features for editing or generating these patterns from scanned documents. These specialized systems can be transferred to the CAM data using the vector DXF file.
8.2.11 Manual Milling

The menu of manual milling serves to simple and fast specifying the working place which is of simple geometry. Usually, there is no option of advanced menus (i.e., without entry, exit, ramps, etc.).

Material is removed mostly in one shot, i.e., in one cutting depth. Setting geometry is a 2D geometry (points, lines, circles, arcs, etc.).

8.2.12 2.5 D rest machining

This strategy is used to remove residual material that remained on the workpiece from the previous machining. It is necessary to select smaller tool diameter compared to the previous cycle.

Fig. 8.26 2.5 D rest machining

8.2.13 Plunge machining

Plunge machining is a machining away stock in a series of Z axis plunges. Plunge machining is carried out using an appropriate tool where the tool performs a series of rotary motion (motion in the z-axis and not in the x-axis or y-axis) in a regularly arranged network of points.

This strategy is used when demands of the machining process are higher than in common milling. Plunge machining is primarily used to machine pockets and external stock - in deep moulds or dies for example. Long tools gain stability from a cutting force directed up the tool axis rather than as a side force. The tool then performs a series of overlapping movements that resemble drilling and gradually removes cylindrical volumes of material. Grooving tool axis requires special tools.
Fig. 8.27 Plunge milling

The advantage of this method is the effect of cutting forces mainly in the axial direction (i.e., in the tool axis). Size removal is small because of small feed per tooth and due to necessary pre-finishing operating cycles.

<table>
<thead>
<tr>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>In the 3-axis machining, this strategy is called a BORE JET or vertical machining.</td>
</tr>
</tbody>
</table>
9. SIMULATION AND VERIFICATION

Another broad area of CAM systems, which largely contributes to increasing the machining efficiency is the high quality visualization and verification of the created NC program. Simple visualization and verification of the created NC program is usually built into most of the CAM systems. Using this verification, a collision with the workpiece or tool clamps can be controlled. Verification also analyzes any residual material or gouges.

Some CAM systems can simulate and verify the collision with material and clamps not only for the tool holder and spindle but the complete machine tool, including the full geometry and motion. Such programs allow to create any shape of tools, holders, clamps, vices, etc., and save and store them in libraries. Using special modules, it is possible to define dimensions and kinematics of a particular machine, to import its control system and to perform simulation and verification of the machine movements.

Fig. 9.1 Residual materials - simulation in NX5

Fig. 9.2 Example of machining simulation in NX
Summary of the machining simulation options:

- checking the collision between the tool shank, tool holder, tool spindle and the workpiece and clamps,
- analyzing the undercuts,
- analyzing the geometry and motion of CNC machine,
- creating a part model for individual operations:
  - as a blank for other operations
  - as an outline for further modeling,
- analyzing the size of the residual material (accuracy of the machined surface),
- comparing the machined components to the model - the possibility of a numeric or color display of the difference size,
- carrying out the cuts on the machined shape and measuring,
- sorting and typing a collision,
- possibility of creating features,
- saving the simulation to *.exe file, and more…

The following figure shows the display traces of tools without any need of advanced simulations, where, in the corners of the cavity (in the circle), unremoved material can be seen.

*Fig. 9.3 Depiction of a Tool Path (EdgeCAM - Simple simulation)*
One of the largest suppliers of verification for the respective CAM modules is the MachineWorks company with its product called CNC Simulation & Verification in 6.0. It uses, as a module for simulating, many CAM systems, such as: Catia, Climatron E, Edge CAM, Esprit, hyperMILL, MasterCAM, and others. Other programs for the production visualization are VERICUT and Predator. These systems can load the G code (ISO code, dialogue) and CAD / CAM system creates the NC toolpath (APT CL data).

Fig. 9.4 EdgeCAM - Advanced simulation module
10. POSTPROCESSOR - POSTPROCESSING

For the CAM programming, post processing is a very important issue. Postprocessing is a translation of the INC file (that has already generated tool paths) in the language understood by the relevant machine control system. In the world, there are many control systems and their variants. Requirements for postprocessing are always based on the particular machine.

The processor generates the APT or CL data (Cutter Location Data) – a "software" for the "ideal" NC machine.

CL data can be adapted to technical and formal possibilities of entered program for a specific pair of the control system and CNC machine.

Postprocessor translates the generated CL data (i.e., the already generated tool paths) into a language comprehensible for the respective control system of the machine.

In other words, after completion of all machining cycles and after machining simulation without any collision, an NC code is generated simply by pressing the appropriate icon. To create the NC code, the CAM system uses a NC code generator, which converts the created technological procedure to the instructions of the respective machine and control system. The NC code generator writes the instructions into an ASCII text file. This file can be modified before its sending off to the machine. To the modification, a special editor supplied with the CAM product package or a simple program available on any computer such as Notepad can be used.

There are many control systems and their variants. Requirements for postprocessing are always based on the particular machine. It is important for users to have the option to edit and configure the postprocessors, so that the work could be adapted to individual local customs and mutations.
11. CREATION OF ACCOMPANYING DOCUMENTATION

In the CAM systems, there can be created the accompanying documentation. This documentation is intended for monitoring of the order status, materials, tools, etc., further, for commercial organizations and suppliers who want to participate on the created work process (to monitor order statuses, to approve the created work process, etc.). The accompanying documentation is also used by CNC machine operators to obtain better orientation in the created NC program.

EdgeCAM menu modules such as the Job Manager and Order Assistant designated for work with orders.

In the accompanying documentation, there can be found, for example, the following information:

- who created the NC program,
- who created the model components,
- who approved the contract,
- description of the contract,
- used tools (description of the tools, position, etc.),
- machining time,
- set-up time,
- workpiece material,
- used strategies - possible graphic preview,

where the individual files are located and their names (NC program, the CAM file, model file), etc.

Accompanying documentation can be easily printed out and handed over to the NC operator or, in case of partner organizations, it can be sent by an e-mail. The advantage is that these partners do not need to possess any CAM system license as the documentation can be stored in a common executable file (*.txt, htm *, etc.).

One of the possibilities of the accompanying documentation is to create a listing of the respective instructions in a text file. The created instructions can be listed in a text file, as shown on the following figure.
The following examples are of the so called adjusting protocols developed in various CAM systems.
Fig. 11.3 Example of the accompanying documentation (production EDM electrode EM13/8)

– HSC technology - DelCAM

Fig. 11.4 Toolbar - MasterCAM
**SHOP FLOOR DOCUMENTATION**

**PROGRAM NAME : NC_PROGRAM**

<table>
<thead>
<tr>
<th>TOOL NAME</th>
<th>TOOL TYPE</th>
<th>DIAMETER</th>
<th>COR RAD</th>
<th>NOSE RAD</th>
<th>ADJ REG</th>
</tr>
</thead>
</table>

**PROGRAM NAME : CAST_BRZDY_F2V**

<table>
<thead>
<tr>
<th>TOOL NAME</th>
<th>TOOL TYPE</th>
<th>DIAMETER</th>
<th>COR RAD</th>
<th>NOSE RAD</th>
<th>ADJ REG</th>
</tr>
</thead>
<tbody>
<tr>
<td>FR.HRUB_10HR_DL</td>
<td>Milling Tool-5 Parameters</td>
<td>10.0000</td>
<td>0.0000</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>FR.TIZIT_AL_10X22</td>
<td>Milling Tool-5 Parameters</td>
<td>10.0000</td>
<td>0.0000</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>NAVRTAVAK_DL25</td>
<td>Drilling Tool</td>
<td>2.0000</td>
<td>0.0000</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>VRTAK_4.2</td>
<td>Drilling Tool</td>
<td>4.2000</td>
<td>0.0000</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>VRTAK_6.3</td>
<td>Drilling Tool</td>
<td>6.3000</td>
<td>0.0000</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>ZAVITNIK_M5</td>
<td>Drilling Tool</td>
<td>5.0000</td>
<td>0.0000</td>
<td>-</td>
<td>0</td>
</tr>
</tbody>
</table>

*Fig. 11.5 Example of the accompanying documentation*
MO-LN-1732_SONE1.NC

ZÁKLADNÍ INFORMACE

Název výrobců: C:\DOCUMENTS AND SETTINGS\XAVIER\SOTEXT\MASTERCAM\SOME\NC
NC soubor: C:\DOCUMENTS AND SETTINGS\XAVIER\SOTEXT\MASTERCAM\SOME\LN-1732_SONE1.NC
Cílové programy: 0
Datum: 07-01-09
Nízor: Forma
Popis operace: F15001H1-073593
Zákazník: SONEKTECH
Programátor: FK

PART IMAGE

POLOTOVAR

Material: ALUMINUM, 6061 - 2014

NÁSTROJE

<table>
<thead>
<tr>
<th>NÁSTROJ</th>
<th>POPIS NÁSTROJ</th>
<th>DÍRA OFF (M)</th>
<th>LEN OFF (M)</th>
<th>DIAM (M)</th>
<th>C-Bar (M)</th>
<th>MIN Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>FREZA PK 5/65 SECO, SUP R/3 F 63246, D 63236</td>
<td>D0</td>
<td>H1</td>
<td>60.000mm</td>
<td>-</td>
<td>2-19.95000mm</td>
</tr>
<tr>
<td>T2</td>
<td>FREZA PK 5/65 SECO, OCTODRILL, SUP R/2 D 32747 F 51740</td>
<td>D2</td>
<td>H2</td>
<td>40.000mm</td>
<td>-</td>
<td>2-20.00000mm</td>
</tr>
<tr>
<td>T3</td>
<td>FREZA PK 5/65 SECO, SUP R/3 F 63218, D 63156</td>
<td>D3</td>
<td>H3</td>
<td>40.000mm</td>
<td>-</td>
<td>2-24.00000mm</td>
</tr>
</tbody>
</table>

OPERATION LIST

| OP 1: G54 | NFINISH | HHM | STOCK LEFT WALLS: 25 | STOCK LEFT FLOORS:.5 | SEVICE-3 |
| OP 2: G54 | NFINISH | HHM | STOCK LEFT WALLS: 25 | STOCK LEFT FLOORS:.5 | SEVICE-3 |
| OP 3: G55 | CONTOUR | SEVICE-3 |

ESTIMATED OPERATING TIME = 00 HOURS, 32 MINUTES, 09 SECONDS

Fig. 11.6 Example of the accompanying documentation - MasterCAM

118
12. CONCLUSION

Recently, a production of models, molds, dies and other parts of complex shapes in various sectors of machinery industry has developed. For this purpose, the most up-to-date numerically controlled CNC machine tools are used, as well as more obsolete machines used by companies that do not have such a possibility to establish the latest technology.

The application of CAD / CAM systems provides:

- Significant reduction of time from the preparation of NC programs, through their simulation, to the production itself.
- Higher efficiency and rationalization of work characterized by decrease in rejects.
- Cost reduction in development and production of new parts where the gradual deployment of CAD / CAM systems provide a short payback period of these software products.

As a result of increasing competition on the world market and our growing need for introducing modern technology into our businesses, these new technologies deployed in the area of machining (e.g., electro-machining, laser and water jet machining, dry machining, high speed machining - HSC) or application of new cycles of cutting tools, require the deployment of machines numerical controlled by a computer based on the application of available CAD / CAM systems.

Development workers programing the CAM systems strive to simplify and facilitate the work for programmers of the CNC machines by creating software with a user-friendly and intuitive operation.

The purpose of this script is the summary of terms, general laws, procedures and information related to the CAM system. The purpose is not to show work in the particular CAM system, but describe how to proceed in any CAM system, what a technologist of any CAM system should expect and what he should be aware of. The objective of this script is to complement and expand information on the subjects taught in the Department of Machining and Assembly, Faculty of Mechanical Engineering, VŠB – Technical University of Ostrava, "CAD / CAM systems in machining and CAM systems in Machining II."

I would be very happy if these objectives of the script are fulfilled.
REFERENCES


[7] www pages of producers and dealers of machinetools and cutting tools


[16] MRKVICA, I., UHLÁŘ, V. Mathematic Model of the Milling Head First Contact. Technologické inženierstvo, 2/2007, s. 49-51. ISSN 1336-5967.


