МІНІСТЕРСТВО ОСВІТИ І НАУКИ УКРАЇНИ
Національний аерокосмічний університет ім. М.Є. Жуковського
«Харківський авіаційний інститут»

V. V. Borysevych, O.A. Pavlenko, V.T. Sikulskiy

MODELING OF MACHINING IN CAM "ADEM"

Tutorial For The Yearly Project

МОДЕЛЮВАННЯ МЕХАНООБРОБЛЕННЯ В САМ "ADEM"

Навчальний посібник до виконання курсової роботи

Харків «ХАІ» 2008

УДК 621. 771

Борисевич В. В. Modeling of Machining in CAM "ADEM": навч. посібник до виконання курсової роботи/ В. В. Борисевич, А. А. Павленко, В.Т. Сікульський. –Х.: Нац. аерокосм. ун-т «Харк. авіац. ін-т», 2008. – 42 с.

Описано основні етапи проектування технологічного процесу механообробленння деталей авіаційної техніки, оцінки технологічності деталі, розрахунку припусків, вибору заготівки, наведено основні залежності для розрахунку режимів різання й спрощених методів нормування технологічних процесів. Розглянуто також спосіб комп'ютерного моделювання технологічного процесу в системі ADEM.

.Для студентів, що вивчають дисципліни, пов'язані з технологією літако- та вертольотобудування, із системами автоматизованого проектування, для студентів будь-яких спеціальностей, що вивчають створення тривимірних зображень об'єктів і моделювання процесів, а також при виконанні курсового проектування.

The main stages of the design of technological process of the cutting of the aircraft parts, the estimation of the machinability, calculation of the allowances, selection of the stock, the basic relationship for the calculation of the cutting parameters are given and simplified method of normalization of the technological process. The method of computer designing of the technological process in CAD/CAM system ADEM is given.

For those students who study discipline relating to aircraft and helicopter manufacturing, to CAD/CAM/CAE systems and for those students who study any special subject relating to the solid modeling and to modeling of processes. It can be used during for yearly project

Іл. 19. Табл. 18. Бібліогр.: 7 назв

Р е ц е н з е н т и: канд. техн. наук, проф. А.М. Краснокутський,
канд. техн. наук, доц. Г.К. Крижний

1. PLANNING OF THE MACHINING. CUTTING PARAMETERS CALCULATION

The necessary stages of the development of a technological process

The unit, typical or group technological processes must be worked out in the form of routing card or detailed process plan.

The unit technological process is worked out for a part of complicated shape. In other cases typical or group technological processes are worked out.

The main stages of the development of technological process are follows:

1. The analysis of the input data for the planning of technological process.
2. The choosing of an analog of the unit, typical or group technological process (as a rule the variant from the plant).
3. Selection of the initial stock and of the method of its manufacturing.
4. Selection of the processing datum surfaces.
5. Planning of the technological rout of the machining.
6. Working out of the detailed process plan and elaboration of the technological steps (operations).
7. Rating of the technological process.
8. Establishing of the safety requirements.
9. Calculation of the efficiency of the technological process.
10. Documentation.

1.1. The analysis of the input data for the planning of technological process

1.1.1. Classification of the initial information

The input data for planning of the technological process may include the following information.

The main document which refer to design documentation are follows: drawing of the part, technical requirements, specifications, various lists. Beside the target must be defined for the following manufacturing of the part: yield and terms of the production preparation.

The following documents may be used as guideline: standards concerning technological processes, methods of its control, perspective technological processes documentation, information about environmental protection, safety measures, guideline technological documents, production directions.

Information from reference book, catalogues and concerning cutting parameters, machining allowances may be used as reference information.

1.1.2. Design-technological analysis of the part and detailing of its drawing

Here one must analyse functional purpose of the part, its allocation in assembly and its interaction with mating parts, the shape of the part, dimensioning specifications of all surfaces of the part. All surfaces must have literal or numerical designation. One should define the constructive data of all machined surfaces. The results of analysis should be summarized in a Table 1.1.

Table 1.1

Geometry and Dimension of the part

Designation of the machined surface	Allocation of the surface	Shape of the surface	Dimensions	Class of accuracy or dimension tolerance	Surface finish	Constructive datum	Surface area
Surface A	Outer	Cylindrical	Ø30, L=50	JT10	Rz10	Surface C	471
Surface B	Inner	Flat	20×30	JT12	Rz40	Surface D	600

Here the information about chemical composition and mechanical properties of the material of the part must be, for example of the aluminium alloy AK6 (Table 1.2, 1.3).

Table 1.2

Chemical composition of the AK6 aluminium alloy

Main component (Al the other), %							
Cu	Mg	Mn	Si	Impurities, no more than			
				Fe	Ni	Ti	others
1.8…2.6	0.4…0.8	0.4…0.8	0.7…4.2	0.7	0.1	0.1	0.05

Table 1.3

Mechanical properties of the AK6 aluminium alloy

σ_{uts}, MPa	σ_y, MPa	δ, %	Ψ, %	HB
420	300	13	40	105

The conclusion concerning adequacy of material of the part to its function. The purpose of the technological analysis is to decrease costs and time needed for manufacturing of the part with high quality.

There are two type of estimation of the manufacturability: qualitative and quantitative.

The qualitative estimation is the preliminary one and may describe the manufacturability generally on the base of an experience and may be expressed by follows: good – bad.

The quantitative estimation may be expressed by the main and additional numeric parameters. The main parameters are follows:

– labor expenditures per part E_p (hours);
– manufacturing unit cost C_m.

The additional parameters are the follows:

– unification factor of the structural features;
– coefficient of the application of the standardized machined surfaces;
– material utilization factor;
– part weight;
– the highest degree of surface finish;
– the best class of accuracy;
– coefficient of the application of the standardized technological processes;
– coefficient of the application of the advanced equipment.

Student must define qualitative estimation of the machinability of the part.

During analyse of the part concerning material used it should be estimated its macninability and cost. Beside the possibility of the application of the more light material should be estimated or some hardening technology be offered.

During analysis rationality of the geometry and quality of the surfaces should be taken in mind with consideration of application of the high efficient equipment and rigging. The simplicity of the shape (flat, cylindrical) and big number of the surfaces which hasn't be machined should be referred to favorable factor.

It should be devoted great attention to grounding of application of the dimensional deviation limits because there is great increase in cost due to close tolerances.

There are some particularity of ensuring of machinability for part machined on NC equipment. One should follow the several recommendation:

– high accuracy allows to substitute grinding of cylindrical ledges by turning;
– application of the form tool allows to eliminate cutoff relief grooves;
– the maximum labor volume must be ensured at one positioning of the part (for example key grooves must be from the one side of the shaft);
– the radii must be unified;
– constructive features must be unified.

1.1.3. Approximate calculation of the cutting time

Cutting time may be calculated approximately using information from the Table 1.4.

Table 1.4

Cutting parameters calculation formulas

Operations and operation steps	Class of accuracy/surface finish class, mcm	Cutting time	Cutting parameters	Best possible cutting time
1. Metal cutting				$T_c=kl;\ T_c=kD^2$
1.1. Sawing	$\frac{16...15}{80...40}$	$T_c=\frac{l}{F_m}+\frac{l}{F_{max}}$	F_m=91.7 mm/min F_{max}=5000 mm/min k_1=0.01 $k_2=0.2\cdot10^{-3}$	$T_c=1.1l\cdot10^{-2}$
1.2. Hacksawing	$\frac{16...17}{90...20}$	$T_c=\frac{l}{F}$	F=11.4 mm/min $k=8.77\cdot10^{-2}$	$T_c=8.7l\cdot10^{-2}$
1.3. Cutting off by turning	$\frac{13...12}{40...10}$	$T_c=\frac{\pi D^2}{2000VF}$	F=0.1 mm/rev V=40 m/min $k=3.93\cdot10^{-4}$	$T_c=3.93D^2\cdot10^{-4}$
2. Facing in one step				$T_c=k(D^2-d^2)$
2.1. Rough facing (turning of the ring)	$\frac{13...12}{40...20}$	$T_c=\frac{\pi(D^2-d^2)}{4000VF}$	F=0.5 mm/rev V=70 m/min $k=2.24\cdot10^{-5}$	$T_c=2.24(D^2-$ $-d^2)10^{-5}$
2.2. Finishing facing (turning of the ring)	$\frac{12...11}{2,5...2,0}$	– " –	F=0.41 mm/rev V=174,6 m/min $k=1.1\cdot10^{-5}$	$T_c=1,1(D^2-d^2)\times$ $\times10^{-}$
2.3. Rough facing of the continuous face	$\frac{13...12}{40...20}$	– " –	F=0.5 mm/rev V=70 m/min $k=2.24\cdot10^{-5}$	$T_c=2.24(D^2-$ $-d^2)10^{-5}$
2.4. Finishing facing of the continuous face	$\frac{12...11}{2,5...2,0}$	– " –	F=0.41 mm/rev V=174,6 m/min $k=1.1\cdot10^{-5}$	$T_c=1,1D^2\cdot10^{-5}$
3. Body of revolution turning in one step				
3.1. Rough turning	$\frac{13...12}{80...40}$	$T_c=\frac{\pi Dl}{1000VF}$	F=0.4 mm/rev V=105 m/min $k=7.5\cdot10^{-5}$	$T_c=7.5Dl\cdot10^{-5}$
3.2. Finishing turning	$\frac{11...9}{20...10}$	$T_c=\frac{\pi Dl}{1000VF}$	F=0.15 mm/rev V=120 m/min $k=1.75\cdot10^{-4}$	$T_c=1,75Dl\cdot10^{-4}$
3.3. Outside cylindrical grinding with traverse feed				
3.3.1. Preliminary grinding	$\frac{11...9}{20...2,5}$	$T_c=\frac{\pi Dlhf}{1000VFt}$	F=14 mm/rev V=24 m/min h=0.25 mm t=0.024 mm/stroke f=1.2 $k=1.2\cdot10^{-4}$	$T_c=1,2Dl\cdot10^{-4}$

Continuation of the Table 1.4

Operations and operation steps	Class of accuracy/surface finish class, mcm	Cutting time	Cutting parameters	Best possible value of cutting time
3.3.2. Finishing grinding	$\frac{6}{1,25...0,63}$	-"-	F=10 mm/rev V=30 m/min h=0.1 mm t=0.008 mm/stroke f=1.4 k=1.84·10^{-4}	T_c=1,84DI·10^{-4}
3.3.3. Fine grinding	$\frac{6}{0,32...0,16}$	-"-	F=8 mm/rev V=20 m/min h=0.025 mm t=0.003 mm/stroke f=2.0 k=3.27·10^{-4}	T_c=3,27DI·10^{-4}
3.4. Rolling after finishing turning	$\frac{8}{1,25...0,32}$	$T_c=\frac{\pi DI}{1000VF}$	F=0.3 mm/rev V=100 m/min k=0.1·10^{-3}	T_c=0,1DI·10^{-3}
4. Hole making				
4.1. Drilling of the holes diameter up to 20 mm	$\frac{13...12}{40...10}$	$T_c=\frac{\pi DI}{1000VF}$	F=0.25 mm/rev V=22.4 m/min k=5.6·10^{-4}	T_c=5,6DI·10^{-4}
4.2. Boring of the holes diameter up to 20...70 mm	$\frac{12...11}{20...10}$	-"-	F=0.25 mm/rev V=29.7 m/min k=4.23·10^{-4}	T_c=4,23DI·10^{-4}
4.3. Countersinking	$\frac{11}{20...10}$	-"-	F=1 mm/rev V=15 m/min k=2.1·10^{-4}	T_c=2,1DI·10^{-4}
4.4. Reaming				
4.4.1. Rough reaming	$\frac{10}{2,5...2,0}$	-"-	F=0.6 mm/rev V=12 m/min k=4.36·10^{-4}	T_c=4,36DI·10^{-4}
4.4.2. Finishing reaming	$\frac{6}{1,25...0,63}$	-"-	F=0.6 mm/rev V=6 m/min k=8.76·10^{-4}	T_c=8.76DI·10^{-4}
4.5. Boring				
4.5.1. Rough boring	$\frac{12}{40...20}$	-"-	F=0.36 mm/rev V=63.4 m/min k=1.34·10^{-4}	T_c=1.36DI·10^{-4}
4.5.2. Finishing boring	$\frac{10}{10...2,5}$	-"-	F=0.35 mm/rev V=50 m/min k=1.8·10^{-4}	T_c=1.8DI·10^{-4}
4.6. Inside grinding				
4.6.1. Preliminary grinding	$\frac{11}{2,5...1,25}$	$T_c=\frac{2\pi DIhf}{1000VFt}$	F=12 mm/rev V=27 m/min h=0.2 mm t=0.004 mm/stroke f=1.4 k=1.46·10^{-4}	T_c=1,46DI·10^{-4}

Operations and operation steps	Class of accuracy/surface finish class, mcm	Cutting time	Cutting parameters	Best possible value of cutting time
4.6.2. Finishing grinding	$\frac{7}{0,63...0,32}$	-"-	F=10 mm/rev V=36 m/min h=0.2 mm t=0.009 mm/stroke f=1.5 $k=5.83\cdot10^{-4}$	$T_c=5.83Dl\cdot10^{-4}$
5. Inside broaching				
5.1. Rough broaching	$\frac{9}{10,0...2,5}$	$T_c=\frac{la}{1000V}$	V=7 m/min a=2 $k=2.86\cdot10^{-4}$	$T_c=2.86l\cdot10^{-4}$
5.2. Finishing broaching	$\frac{7}{1,25...1,0}$	-"-	V=4 m/min a=2 $k=2.86\cdot10^{-4}$	$T_c=2.86l\cdot10^{-4}$
5.3. Finishing piercing	$\frac{7}{1,25...0,63}$	$T_c=\frac{la}{1000V}$	V=3 m/min $k=3.3\cdot10^{-4}$	$T_c=3.3l\cdot10^{-4}$
5.4. Fine piercing	$\frac{7...6}{0,63...0,16}$	-"-	V=2 m/min $k=0.5\cdot10^{-4}$	$T_c=0.5l\cdot10^{-4}$
6. Inside smoothing				$T_c=kl$ $T_c=kh$ $T_c=kA_\Sigma$ $T_c=kD$
6.1. Sizing with mandrel	$\frac{8}{1,25...0,63}$	$T_c=\frac{l}{F_m}$	F_m=2000 mm/min $k=0.5\cdot10^{-3}$	$T_c=0.5l\cdot10^{-3}$
6.2. Polishing	$\frac{5...6}{1,25...0,1}$	$T_c=kA_\Sigma$	$k=1.6\cdot10^{-4}$	$T_c=1.6A_\Sigma\cdot10^{-4}$
6.3. Honing	$\frac{6...7}{1,25...0,63}$	$T_c=kh$	k=0.126	$T_c=0.126h$
6.4. Fine honing	$\frac{5...6}{0,32...0,16}$	$T_c=kh$	k=0.121	$T_c=0.126h$
6.5. Superfinish	$\frac{6}{0,4...0,16}$	$T_c=kD$	$k=2.38\cdot10^{-2}$	$T_c=2.38D\cdot10^{-2}$
6.6. Double superfinish	$\frac{5}{0,16...0,08}$	$T_c=kD$	$k=5.1\cdot10^{-2}$	$T_c=5.1D\cdot10^{-2}$
6.7. Mechanical finishing				
6.7.1. Mechanical finishing of the nonhardened steel	$\frac{6}{0,32...0,16}$	$T_c=k\,A_\Sigma$	$k=2.4\cdot10^{-4}$	$T_c=2.4\cdot A_\Sigma\cdot10^{-4}$
7. Flat surfaces machining				$T_c=kl$ $T_c=kDl$ $T_c=kBl$
7.1. Flat face milling				

Continuation of the Table 1.4

Operations and operation steps	Class of accuracy/surface finish class, mcm	Cutting time	Cutting parameters	Best possible value of cutting time
7.1.1. Rough	$\frac{12}{40...20}$	$T_c=\frac{l}{F_m}$	F_m=170 mm/min $k=5.9\cdot10^{-3}$	$T_c=5.9l\cdot10^{-3}$
7.1.2. Finishing	$\frac{10}{20...10}$	- " -	F_m=208 mm/min $k=4.82\cdot10^{-3}$	$T_c=4.82l\cdot10^{-3}$
7.1.3. Fine	$\frac{8}{2,5...1,25}$	- " -	F_m=170 mm/min $k=2.86\cdot10^{-3}$	$T_c=2.86l\cdot10^{-3}$
7.2. Slab milling				
7.2.1. Rough	$\frac{12}{40...20}$	$T_c=\frac{l}{F_m}$	F_m=150 mm/min $k=6.66\cdot10^{-3}$	$T_c=6.66l\cdot10^{-3}$
7.2.2. Finishing	$\frac{10}{20...10}$	- " -	F_m=285 mm/min $k=3.52\cdot10^{-3}$	$T_c=3.52l\cdot10^{-3}$
7.2.3. Fine	$\frac{8}{2,5...1,25}$	- " -	F_m=600 mm/min $k=1.66\cdot10^{-3}$	$T_c=1.66l\cdot10^{-3}$
7.3. Planning				
7.3.1. Rough	$\frac{14}{80...40}$	$T_c=\frac{Bla}{1000V_{ws}F}$	F=1.5 mm/rev V_{ws}=22 m/min a=1.43 $k=4.34\cdot10^{-6}$	$T_c=4.34Bl\cdot10^{-6}$
7.3.2. Finishing	$\frac{11}{10,0...1,5}$	- " -	F=16 mm/rev V_{ws}=26 m/min a=1.43 $k=3.4\cdot10^{-6}$	$T_c=3.4Bl\cdot10^{-6}$
7.4. Flat grinding				
7.4.1.Preliminary	$\frac{9...11}{10,0...2,5}$	$T_c=\frac{lhf}{1000V_bt}$	V_b=12 m/min h=0.3 mm t=0.02 mm/double_str f=1.4 $k=1.5\cdot10^{-3}$	$T_c=1.5l\cdot10^{-3}$
7.4.2. Finishing	$\frac{7}{1,25...0,63}$	- " -	V_b=12 m/min h=0.1 mm t=0.009 mm/double_str f=1.4 $k=1.3\cdot10^{-3}$	$T_c=1.3l\cdot10^{-3}$
7.4.3. Fine	$\frac{6}{0,32...0,16}$	- " -	V_b=8 m/min h=0.04 mm t=0.005 mm/double_str f=1.5 $k=1.5\cdot10^{-3}$	$T_c=1.5l\cdot10^{-3}$
7.5. Polishing	$\frac{5}{1,25...0,04}$	$T_c=k\,A_\Sigma$	$B\times l$ from 30×30 to 200×200 $k=1.6\cdot10^{-4}$	$T_c=1.6\,A_\Sigma\cdot10^{-4}$
8. Machining of the spiral surfaces				$T_c=kD$ $T_c=kDl$

Continuation of the Table 1.4

Operations and operation steps	Class of accuracy/surface finish class, mcm	Cutting time	Cutting parameters	Best possible value of cutting time
8.1. Threading with tap	$\frac{10}{10...2,5}$	$T_c=\frac{\pi D l \alpha}{1000VF}$	F= 2.0 mm V= 9.1 m/min α= 1.85 k= $3.19\cdot10^{-4}$	T_c= $3.19\cdot Dl\cdot10^{-4}$
8.2. Threading with self-opening head	$\frac{9}{10...2,5}$	$T_c=\frac{\pi D l \alpha}{1000VF}$	F= 2.0 mm V= 14 m/min k= $1.12\cdot10^{-4}$	T_c=$1.12\cdot Dl\cdot10^{-4}$
8.3. Threading with multiedge milling cutter	$\frac{10}{10...2,5}$	$T_c=\frac{1.2\pi^2 d_c}{1000VF_t z_t}$	d_c= 65 mm F_t= 0.05 mm/tooth V= 33.3 m/min Z_t= 14 k= $3.3\cdot10^{-2}$	T_c= $3.3\cdot D\cdot10^{-2}$
8.4. Threading by knurling	$\frac{9}{2,5...1,0}$	T_c=kD	k= $3.2\cdot10^{-3}$	T_c= $3.2\cdot D\cdot10^{-3}$
8.5. Threading with cutter				
8.5.1. Rough threading	$\frac{10}{20...10}$	$T_c=\frac{\pi D l i \alpha}{1000VF}$	F= 2.0 mm V= 40 m/min α= 1.85 i= 1.9F k= $2.78\cdot10^{-4}$	T_c= $2.78\cdot Dl\cdot10^{-4}$
8.5.2. Finishing threading	$\frac{6}{0,63...0,32}$	– " –	F= 2.0 mm V = 76.8 m/min α=1.85 i=1.2F k=$9.1\cdot10^{-5}$	T_c= $9.1\cdot Dl\cdot10^{-5}$
8.6. Finishing grinding of the thread	$\frac{6}{0,63...0,32}$	$T_c=\frac{\pi D l}{1000VF}\left(\frac{h}{t}+\alpha\right)$	F= 2.0 mm V= 7.5 m/min t= 0.05 mm/d.str. α=1.1 l= 1.2F k= $4.6\cdot10^{-3}$	T_c= $4.6\cdot Dl\cdot10^{-3}$
9. Machining of the evolute surfaces	T_c= $Bm(k_1 + zk_2)$ T_c= kBz			
9.1. Finishing slotting of the tooth	$\frac{8...7}{2,5...1,25}$	$T_c=Bm\left(\frac{4.4}{1000vF_r}+z\frac{2\pi}{1000vF_{c.r.}}\right)$	F_r= 0.04 mm/d.str. $F_{c.r.}$=0.22 mm/d.str. V=34 m/min k=$1.12\cdot10^{-4}$	T_c=Bm(0.0032+ +$z\cdot$0.0084)
9.2. Finishing milling of the teeth	$\frac{6}{2,5...1,25}$	$T_c=\frac{Bz\pi d_c}{1000vFq}$	d_c= 90 mm F = 1 mm/rev V= 30 m/min q= 1 k= $9.43\cdot10^{-3}$	T_c= $9.43\cdot Bz\cdot10^{-3}$

Continuation of the Table 1.4

Operations and operation steps	Class of accuracy/surface finish class, mcm	Cutting time	Cutting parameters	Best possible value of cutting time
9.3. Finishing shaving	$\frac{7}{1,25...1,63}$	$T_c = \frac{Bzhf}{FnzF_v}$	$h = 0.17$; $n = 280$ rev/min; $f = 1.35$; $z = 73$; $F_v = 0.0045$ mm/str.; $F = 0.25$ mm/rev; $k = 0.1 \cdot 10^2$	$T_c = 0.1 \cdot Bz \cdot 10^2$
9.4. Milling of the worm-gear	$\frac{8...7}{1.25...0.63}$	$T_c = \frac{2.94\pi\sqrt{z}D_c}{1000V_cF_{tan}}$	$D_c = 80$ mm; $F_{tan} = 1.4$ mm/rev; $V_c = 25$ m/min; $k = 0.0212$	$T_c = 0.0212D\sqrt{z}$
10. Machining of the slit	$T_c = klz$			
10.1. Rough milling	$\frac{10}{20...10}$	$T_c = \frac{\pi D_c lz}{1000V_cF}$	$D_c = 100$ mm; $F = 2.1$ mm; $V = 32$ m/min; $k = 4.7 \cdot 10^{-3}$	$T_c = 4.7 \cdot lz \cdot 10^{-3}$
10.2. Finishing milling	$\frac{6}{2,5...1,25}$	–"–	$D_c = 100$ mm; $F = 1.2$ mm; $V = 30$ m/min; $k = 8.7 \cdot 10^{-3}$	$T_c = 8.7 \cdot lz \cdot 10^{-3}$

Where α – coefficient of the idle running; B – width of the stock, mm; D, d – diameters of the stock (of the holes), mm; d_c – diameter of the milling cutter, mm; A_Σ – total machined area, mm^2; f – coefficient of the passes without cross-feed; h – machining allowance, mm; i – number of passes; k – coefficient of the cutting conditions; l – length of the stock, broach strock, mm; l_m – length of the working stroke, mm; m – pitch of gear, mm; n – number of revolution per minute; p – number of passes without cross-feed; F – longitudinal feed, mm/rev; F_t – feed per one tooth, mm; F_m – minute feed, mm/min; F_t – tangential feed, mm/rev; F_r – radial feed, mm/stroke; F_{cr} – critical radial feed, mm/stroke; T_c – cutting time, min; t – cross feed (cutting depth), mm/pass; v – cutting speed, m/min; v_w – circumferential speed of the workpiece, m/min; v_b – speed of the bed, m/min; v_{ws} – speed of the working stroke, m/min; z – number of the tooth of the gear, castle-type shaft; z_c – number of the tooth of the milling cutter; F_v – vertical feed.

Floor-to-floor time for preliminary calculation may be obtained with consideration data for T_c from the Table 1.4 according to the next formula

$$T_f = \Psi_k T_c, \quad (1.1)$$

where Ψ_k is a coefficient from the Table 1.5.

Table 1.5

Coefficient Ψ_k

Type of the equipment	Type of production	
	small-lot production	Large scale production
Lathe	2.14	1.36
Turret lathe	1.98	1.35
Turret multycutter lathe	-	1.5
Vertical drill	1.75	1.3
Radial drill	1.75	1.41
Boring lathe	3.25	-
Round-grinding	2.1	1.55
Planer	1.73	-
Milling	1.84	1.51
Gear-cutting machine	1.66	1.27

1.2. Analysis of the basic variant of the technological process

Here student must analyse the variant which have been obtained during practice at plant. The main purpose of this stage to increase productivity and quality of the production. For this purpose it is necessary to:

-check selection of the rough and finishing datum surfaces to ensure principle of coincidence of the processing and constructive datum surfaces;

-validity of the chosen order of the machining operations and type of the machining;

-estimate the level of the mechanization and automation of production;

-using of advanced processes, tooling and gaging equipment.

In case of big number of dimension the rationality of chosen stock is judged only on a base of dimensions which are defined location of the part in assembling. The validity of the processing datum is checked on a base of Table 1.1.

Degree of mechanization and automation, using of advanced processes, tooling and gaging equipment may be estimated with the help of Table 1.6.

Table 1.6

Information about mechanization of the technological process

Number of operation	Control of the operational cycle of the equipment	Type of the fixturing-clamping device	Method of clamping of the workpiece	Type of the cutting tool	Type of the gaging device
01	Manual	Universal	Manual	Standard	Standard
02	Automatic	Universal with pneumatic actuator	Mechanical	Special purpose	Special
03	Manual mechanized	Universal	Manual	Standard	Special

As a result of analysis of a basic variant student must plan and work out in detail operation which must be changed for increasing of effectiveness of the technological process. In the absence of basic technological process a new one must be designed on the base of unit process, at that only 2–3 operations must be work out in detail.

1.3. Selection of the initial stock and of method of its manufacturing

Selection of the initial stock (rod, tube, section, casting, forging) is determined by a number of factors. The main factors are: the production volume, requirements to the mechanical properties of the part, stock utilization factor. As a rule the working drawing of the part must contain information concerning initial stock. In the absence of such information student must ground this selection. After selection of the stock it is necessary to calculate the machining allowance by its summation for all operation for the surfaces, determining overall dimension of the part.

Unilateral intermediate allowance is calculated according to equation:

$$Z = \left[R_z + h + (\bar{\rho}_a + \bar{\varepsilon}_\delta)\right] + \delta \quad (1.2)$$

where Z is a nominal intermediate allowance;

R_z is a height of the microroughness (roughness coefficient for corresponding surface finish class);

h is thickness of the defect layer produced at previous operation;

$\bar{\rho}_a$ is a vector (spatial) deviations of the related surfaces of the machined workpiece produced at the previous;

$\bar{\varepsilon}_\delta$ is an accuracy of the positioning during machining;

δ is a tolerance during previous machining operation.

During machining of the surface of revolution allowance must be doubled:

$$2Z = \left[R_z + h + (\bar{\rho}_a + \bar{\varepsilon}_\delta)\right] + \delta \quad (1.3)$$

The value of ρ_a includes the error due to drift of the axe of rotation of the part or of the hole, distortion of the prismatic shape due to residual stresses or due to heat treatment.

Accuracy of the positioning is determined on a base of geometry of the part and of scheme of positioning at coinciding of the positioning and constructive data. Parameter is excluded from the equations in case of machining with floating cutter and parameter is excluded in case of calculating allowance for grinding for saving surface layer after surface quenching. The value of the and are determined for certain condition of the machining from the reference literature. Values of R_z, h and δ for rolling products and for forging are given in the Table 1.7, and the greatest machining allowance for the nonferrous casting in the Table 1.8. Calculation of the operational allowance must start from the last step of machining and for every step the allowances must be summarized. After final determination of the type and dimensional of the stock it is necessary to make its drawing.

Table 1.7

The components of the operational allowance, mkm

Type of the machined surface	Type of the machining	R_z	*h*	δ
Outer cylindrical, conical and shaped surfaces of revolution	Lapping	0.05 … 0.5	3 … 5	4 … 11
	Fine turning	1 … 5	15 … 20	8 … 25
	Grinding	1.7 … 15	15 … 25	10 … 40
	Finishing turning	5 … 45	30 … 40	50 … 200
	Rough turning	15 … 100	40 … 60	100 … 400
	Cold metalworking	25 … 100	80 … 100	70 … 340
Cylindrical holes	Lapping	0.05 … 0.5	3 … 5	4 … 13
	Fine boring	1 … 5	15 … 20	15 … 25
	Dimpling with sphere	1 … 5	20 … 25	12 … 18
	Broaching	1.7 … 8.5	10 … 20	18 … 30
	grinding	1.7 … 15	20 … 30	15 … 35
	Finishing boring	2 … 25	3 … 40	100 … 200
	Finishing reaming	15 … 45	10 … 20	20 … 80
	Rough reaming	25 … 100	25 … 30	40 … 150
	Rough boring	25 … 225	30 … 50	200 … 350
	Drilling with bushing jig	45 … 225	50 … 60	70 … 300
	Drilling without bushing jig	45 … 225	50 … 60	120 … 350
	Hot forging	100 … 225	500	600 … 1000
Flat Surfaces	Lapping	0.05 … 0.5	3 … 5	4 … 15
	Grinding	1.7 … 15	15 … 25	10 … 50
	Finishing milling	5 … 45	25 … 50	25 … 100
	Rough milling	15 … 100	40 … 60	70 … 200
	Planing	15 … 100	40 … 50	80 … 200
	Rolling	100 … 225	300	500 … 1600
	Hot forging	100 … 225	500	300 … 1000

Table 1.8

Machining allowance for the casting stock

Overall dimension of the part (length or height), mm	Type of production					
	Mass production		Serial production		Unit production	
	Type of casting stock					
	simple	complex	simple	complex	simple	complex
	Machining allowance					
Up to 100	1.5	2.0	2.0	3.0	2.0	3.0
100…200	1.5	2.0	2.0	3.0	3.0	4.0
200…300	2.0	2.0	4.0	4.0	4.0	5.0
300…800	3.0	4.0	4.0	5.0	5.0	7.0
800…1200	4.0	5.0	5.0	6.0	6.0	8.0
1200…1800	4.0	5.0	5.0	7.0	7.0	9.0
1800…2600	5.0	6.0	6.0	8.0	8.0	10.0

As a result one should calculate stock utilization factor according to the equation:

$$\eta = \frac{G_p}{G_s}, \qquad (1.4)$$

where G_s, G_p are the mass of the stock and of the part correspondingly.

The results of calculation must be represented in Table 1.9.

Table 1.9

Calculation of the operational allowances

Technological operation	Components of the allowance			Calculated allowance, mkm	Calculated nominal operational dimension, mm	Adopted dimension of the stock, mm
Rod (rolling)	60	60	250	–	31.218	32
Turning:						
Rough	50	50	200	490	30.708	
Finishing	30	35	60	400	30.308	
Grinding:						
Preliminary	10	20	50	190	30.118	
Finishing	–	–	–	110	30.008	

1.4. Selection of the processing datum surfaces

Geometrical location of the workpiece relatively cutting tools and reliability of its fixation are ensured by scheme of location and fixturing. Such scheme must be worked out for all operation.

Location of the datum surfaces must ensure convenient positioning, fixturing, unfasten and removing of the part, application of the clamping loads and advance of the cutting tools.

If there are no admissible datum surfaces in part design it is necessary to add such special elements or surfaces which are used only during locating, fixturing and machining. Number of repositioniong should be minimizing.

1.5. Selection of the datum surfaces for rough machining

During selection of the datum surfaces for rough machining (initial locating datum surface) one should keep the following rules:

1. The nonmachined surfaces must be used as datum only for the first operation.
2. The surfaces with sufficient dimension and with highest precision and surface finish parameters are more preferable as datum surfaces. Such surfaces mustn't have parting line, scale, feeder head so on.
3. If there are some nonmachined surfaces at part these surfaces must be used as datum surfaces at the first operation.
4. If there aren't nonmachined surfaces at part the surfaces with least allowance must be used as datum surface.

1.6. Selection of the datum surfaces for finishing machining

The datum surfaces for finishing machining one should select keeping the following rules:

1. During all operations one should ensure principle of constancy of datum surfaces so only one set of datum surfaces for the highest accuracy;
2. The most important factor is coinciding of the machining (locating) and constructive datum surfaces because in this case the given precision may be ensured
3. The locating datum surfaces must have the highest dimensional precision and surface finish parameter, these surfaces mustn't deformed under influence of the cutting forces, clumping and weight.

The typical schemes of the positioning of parts (Fig. 1.1):

a –three planes positioning, **b** – at the face and at outer cylindrical surface; **c** – at face and at inner cylindrical surface; **d** – at outer cylindrical surface and at face; **e** – at inner cylindrical surface and at face; **f** – at face and at conical hole; **g** - at center dimples; **h** – at plane and at two outer cylindrical surfaces; **j** – at plane and at two holes.

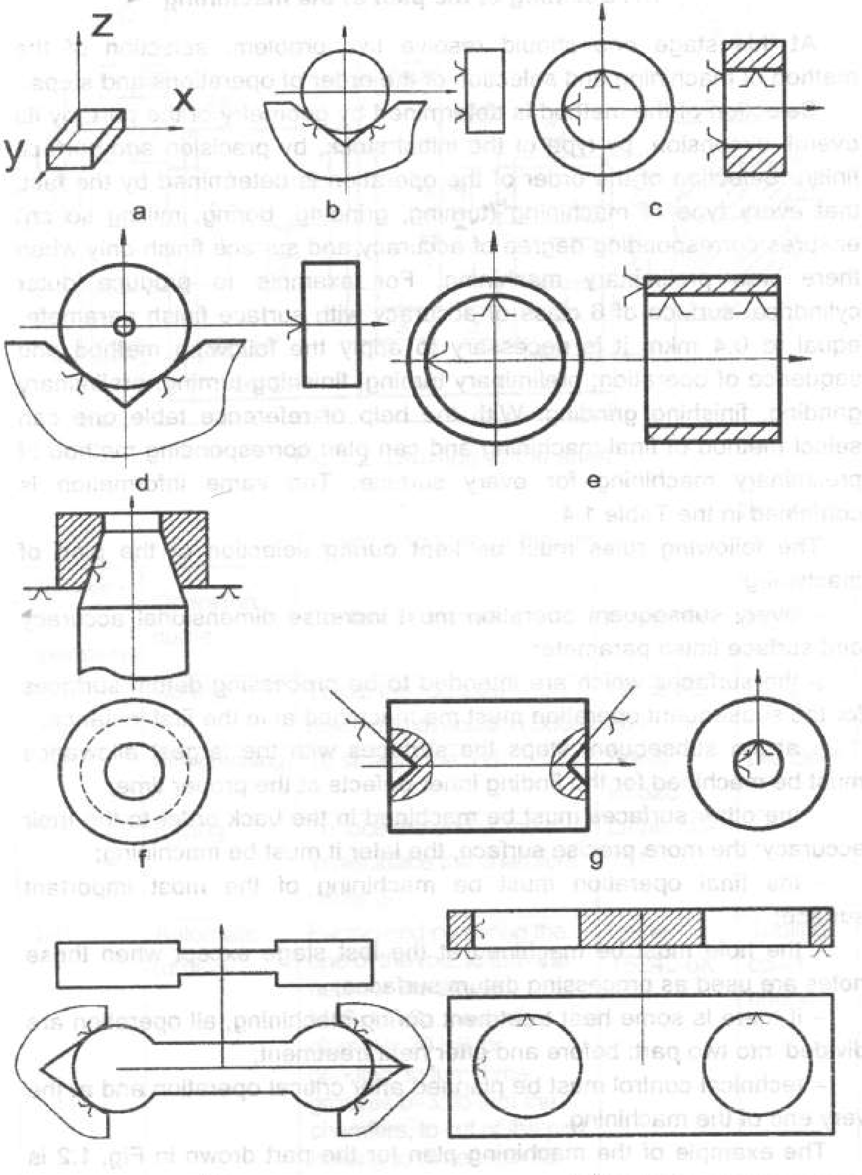


Fig. 1.1. The typical schemes of the positioning of parts

1.7. Desining of the plan of the machining

At this stage one should resolve two problem: selection of the method of machining and selection of the order of operations and steps.

Selection of the method is determined by geometry of the part, by its overall dimension, by type of the initial stock, by precision and surface finish. Selection of the order of the operation is determined by the fact, that every type of machining (turning, grinding, boring, milling so on) ensures corresponding degree of accuracy and surface finish only when there was preliminary machining. For example to produce outer cylindrical surface of 6 class of accuracy with surface finish parameter equal to 0.4 mkm it is necessary to apply the following method and sequence of operation: preliminary turning, finishing turning, preliminary grinding, finishing grinding. With the help of reference table one can select method of final machining and can plan corresponding method of preliminary machining for every surface. The same information is contained in the Table 1.4.

The following rules must be kept during selection of the plan of machining:

– every subsequent operation must increase dimensional accuracy and surface finish parameter;

– the surfaces which are intended to be processing datum surfaces for the subsequent operation must me machined at in the first instance;

– at the subsequent steps the surfaces with the largest allowance must be machined for the finding inner defects at the proper time;

– the other surfaces must be machined in the back order to the their accuracy: the more precise surface, the later it must be machining;

– the final operation must be machining of the most important surface;

– the hole must be machined at the last stage except when these holes are used as processing datum surfaces;

– if there is some heat treatment during machining, all operation are divided into two part: before and after heat treatment;

– technical control must be planned after critical operation and at the very end of the machining.

The example of the machining plan for the part drown in Fig. 1.2 is given in Table 1.10.

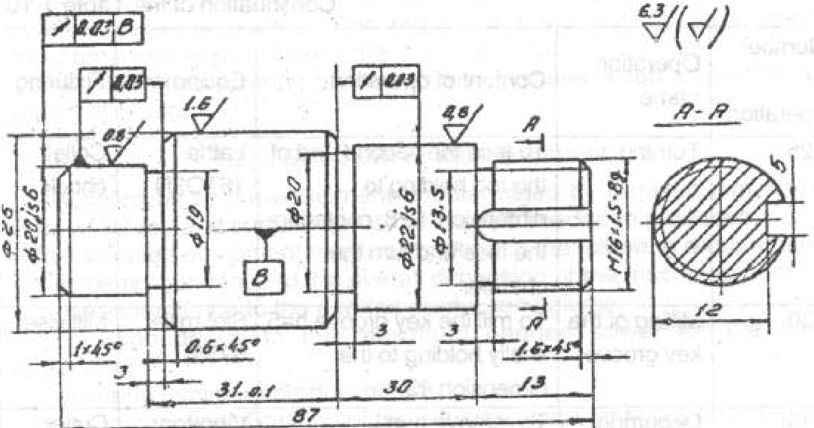


Fig.1.2. Drawing of the shaft

Table 1.10

The working plan of the shaft

Number of operation	Operation name	Content of operation	Equipment	Fixturing
005	Cutting	To cut the rod Ø28 holding to dimension 3000	Press КБ 934	Die block
010	Straightening	To straight the rod	Press И5526	Die block
015	Facing	To face the end of the rod, to resurface the chamfers under 20°	Lathe XC-151	-
020	Automatic turning	Facing end centering the end of the rod, to turn the neck for the thread M16×1.5-8g, neck of diameter 20 js6 for grinding, to turn three grooves b=3; to turn the chamfers, to cut of the part holding to dimension 88	Lathe 1Б240-6К	Tooling, collet chuck

Continuation of the Table 1.10

Number of operation	Operation name	Content of operation	Equipment	Fixturing
025	Turning	To face the second end of the rod holding to dimension 12.8, centering the face and turn the chamfer	Lathe 16ТО2П	Collet chuck
030	Milling of the key groove	To mill the key groove b=5 finally holding to the dimension 13	Slot miller 6930	Mill vise
035	Deburring	To remove burr	Vibratory finishing machine ВМПВ-100	Cutter
040	Thread-knurling	To knurl the thread M16×1.5-8g	Thread-rolling machine A9518	Supporting cutter
045	Round-grinding	To grind the neck of diameter 22 js6 with grinding of the face of diameter 26/22 js6 holding to the dimension 30 finally. After repositioning to grind the neck of diameter 20 js6 with grinding of the face of diameter 26/20 js6	Circular grinder 3У10В	Centers, carrier
050	Rinsing	To rinse the part	Washing machine	–

1.8. Working out of the technological operation

According to the plan of machining for all operations the equipment and tooling must be selected and for the planned operation the required number of steps are assigned (according to the number of surfaces being machined and to the type of the applied cutting tools) and their order is planed. For every step the cutting parameters, cutting time and floor to floor time are calculated.

In aircraft manufacturing processes the computer aided machining are planned as a rule.

1.8.1. Selection of the equipment

The selection of the equipment is determined by required dimension, shape, accuracy and surface quality of the part needed to produced.

The selected equipment must correspond to the following requirement:

– its correspondence to the overall dimension of the machined part;

– correspondence to the needed production volume;

– correspondence to the maximum needed power for the most loaded step;

– ensuring minimum production cost.

Performance specifications of the equipment are represented in [1].

Selection of the technological facility

According standard the different tooling, fixturing, gaging, positioning devices are referred to technological facility. For selection system of lather facility it is necessary to calculate loading of the facility by certain operation and planning period of production of certain parts. The loading parameter of facility is calculated according to the following formula

$$K_l = \frac{T_f O_m}{F}, \tag{1.5}$$

where T_f is a floor-to-floor time per operation, min;

O_m is a number of repetition of operation per month;

F is a fund of the working time per month.

Selection of the cutting tool is done on the base of cutting condition with consideration of type of the equipment, method of the cutting, material of the part, its geometry and dimension, required precision and surface finish [1].

1.9. Calculation of the cutting parameters

Cutting parameters is a combination of the cutting speed, feed and depth of cutting.

The procedure of calculation of the cutting parameters:

1. For the given conditions of cutting one should select the type of tool material, tool geometry and the main dimensions of the cutter. At that it is necessary take into consideration dimensions of the machined part and fixturing device.

2. The depth of the cutting is assigned.

For machining of the cylindrical surfaces (turning, boring, reaming, grinding):

$$t=0/5(D - d), \tag{1.6}$$

where D is a diameter of the workpiece before machining; d – diameter of the workpiece after one stroke of the tool.

For drilling

$$t=0.5D, \quad (1.7)$$

where D is a diameter of the hole.

For milling, planing and plane grinding:

$$t=H - h, \quad (1.8)$$

where H is a dimension of the machined surfaces before machining; h – dimension of the machined surface after one stroke of the cutter.

It is advisable to remove all allowance at one stroke for rough and semifinishing machining or assign depth of the cut equal to allowance. Division of the allowance into several part is advisable for fininishing machining or for machining of the nonrigid workpieces.

3. According to the standard [1] it is necessary to assign the feed S_0. The feed per minute is calculated as:

$$S_{min}=S_o n, \quad (1.9)$$

where n is a speed of rotation of the tool or workpiece rev/min.

Feed per tooth is used for machining by multitoothed cutter:

$$S_t=\frac{S_0}{z}=\frac{S_m}{nz}, \quad (1.10)$$

where S_t is a feed per tooth, mm/tooth; z – number of teeth.

4. According to the standards [1] one should select efficient cutter life depending on machined material, type of the tool material according the following Table 1.11.

Table 1.11

Efficient cutter life

Type of the tool	Machined material	Tool material	Dimension of the tool, mm	Efficient life of the tool, min
Cutter for turning	Steel, nonferrous metals, cast iron	High speed tool steel Hard alloy		30…60 30…120
Driil	Structural and alloyed steel Titanium alloys Cast iron, alluminium alloys	High speed tool steel,	Ø 5…50 Ø 3…50 Ø 5…50	15…90 4…25 20…140
Countersink	Steel, cast iron	–"–	Ø 10…50	30…60
Reamer	structural and alloyed steel Cast iron, alluminium alloys	–"–	Ø6…80 Ø 10…60	25…120 60…180

Continuation of the Table 1.11

Type of the tool	Machined material	Tool material	Dimension of the tool, mm	Efficient life of the tool, min
Milling cutter: face-milling cutter end mill plain milling cutter shaped milling cutter	Steel nonferrous metals, cast iron	–"–		120…400 120…180 120…240 80…180 60…150 120…180
solid plates, rams	Steel nonferrous metals, cast iron	–"–		70…120

5. The next step is calculation of the cutting speed of the corresponding type of machining according to the Table 1.12. Coefficient values and the values of exponents are choosen from reference literature [1].

Table 1.12

Equations for the calculation of the cutting speed

Type of machining	Cutting speed, m/min
Outer longitudinal and transversal turning and boring	$V = \frac{C_v K_v}{T^m t^x S^y}$
Cutting off and shaped turning	$V = \frac{C_v K_v}{T^m S^y}$
Drilling	$V = \frac{C_v D^q K_v}{T^m S^y}$
Boring, reaming	$V = \frac{C_v D^q K_v}{T^m t^x S^y}$
Milling	$V = \frac{C_v D^q K_v}{T^m S^y}$

Where C_v, K_v, m, x, y, q, u, p – constants of proportionality, practice factor and exponents; T – efficient cutter life, min; t – depth of cut, mm; F – feed, mm/rev; F_t – feed per tooth, mm/tooth; B – width of milling, mm; z – number of tooth of milling cutter.

6. The cutting force and horsepower are calculated according to the Table 1.13.

Table 1.13

Cutting force and horsepower

Type of machining	Cutting force	Torque	Cutting horsepower
Outside longitudinal and transversal turning, cutting off and shaped turning, planing	$P_{z,y,x}=10C_p t^x F^y V^n k_p$	$M_{cr}=\frac{P_z D}{200}$	$N=\frac{P_z V}{61200}$
Drilling	$P_0=10C_p D^q F^y k_p$	$M_{cr}=10C_m D^q F^y k_p$	$N=\frac{M_{cr} n}{9750}$
Boring, reaming	$P_0=10C_p D^q F^y k_p$	$M_{cr}=\frac{P_z D}{200}$	$N=\frac{P_z V}{61200}$
Milling		$M_{cr}=\frac{P_z D}{200}$	$N=\frac{P_z V}{61200}$

Where P_z, P_y, P_x – principal (tangent), radial and axial component of the cutting force; P_0 – axial force; C_p, C_m, k_p, k_{mp} – constants of proportionality and practice factors; x, y, q, n, u, w – exponents; D – diameter of the part or of the tool, mm; F – feed, mm/rev; F_t – feed per tooth, mm/tooth; B – width of milling, mm; V – cutting speed, m/min; n – rotational (spindle) speed, rev/min.

7. The required spindle speed is calculated, rev/min:

$$n=\frac{1000V}{\pi D}, \qquad (1.11)$$

where V is the cutting speed, m/min; D is the diameter of the machined workpiece or of the cutting tooll, mm.

8. The calculated cutting parameters are compared with rating of the equipment. There are minimum and maximum value of spindle speed of the cutting equipment in reference literature. The other value are normalized. One should select the nearest lesser value of spindle speed.

1.10. Calculation of the cutting parameters for CAM machining

The cutting parameters for CAM machining are assigned on a base of calculated technological card (CTC) and stock drawing. The CTC must contain the following information:

1. The drawing of the part with OX and OY axes and starting point of the machining.

2. The outline of the part being machined with all necessary dimensions.

3. The tool path in XOY or XOZ coordinate system. The starting point must be both starting and ending point of the machining. The datum points must be defined at the tool path where the geometry of the path or cutting parameters are changed.

4. Control points must be defined where the tool stop must be provided for checking of accuracy or for refixturing.

On a base of CTC the sequence of operation and number of fixturing operation must be assigned. For each fixturing the following information must be assigned:

- the sequence of machining of different zones depending of characteristic features of the parts (inner or outer contour, pockets, ribs);
- the sequence of machining depending on method (rough, finishing machining) in each zone;
- group of the elements of the part, which is machined by one tool in one zone;
- the machining of the several steps by complex program for the one fixturing of the part.

At the following stage cutting parameters are calculated for the every elementary tool path for the zone with highest precision, surface finish class and maximum tool path length.

The values of the feed may be calculated with the help of empiric equations (Table 1.14 – 1.18).

Feed for the rough turning at NC machine, (mm/rev):

$$F_0 = kt^x D^y_{max} D^z_0 . \qquad (1.12)$$

Table 1.14

Parameters for the rough turning

Machined material	*k*	*x*	*y*	*z*
Carbon and alloyed steel	0.15	-0.33	0.19	0.2
Cast iron, copper and aluminum alloys	0.29	-0.3	0.076	0.25
Heat resistant and titanium alloys	0.67	-0.35	0.22	0.2

Where ***k, x, y, z*** – constant of proportionality and exponents; D_{max} and D_0 are the maximum diameter of the part and diameter of the machined surfaces in certain step; ***t*** – cutting depth, mm.

Feed for the finish turning at NC machine (mm/rev):

$$F_0 = k_1 D_{max} + k_2 R_z + b . \qquad (1.13)$$

Table 1.15

Parameters for the finish turning

Machined material	Range of the cutting speed, m/min	k_1	k_2	b
Carbon and alloyed steel	More than 50	0.00012	0.013	0.012
Cast iron, copper and aluminum alloys	Whole	0.00016	0.011	0.036
Heat resistant and titanium alloys	More than 50	0.000085	0.008	0.056

Where k_1, k_2 are the constants, D_{max} is the maximum diameter of the part, R_z – surface finish parameter, mcm.

Feed for the finish turning of the shaped surfaces at NC machine, (mm/rev):

$$F_0 = kD_{max}^x R_z^y \varphi^z. \tag{1.14}$$

Table 1.16

Parameters for the finish turning of the shaped surfaces

Machined material	k	x	y	z
Carbon and alloyed steel	0.012 – 0.067	0.23 – 0.32	0.4 – 0.8	-0.62 – 0.45
Cast iron, copper and aluminum alloys	0.01	0.28 – 0.35	0.5 – 0.73	-0.5 – 0.34

Where k, x, y, z are the constants, D_{max} is the maximum diameter of the part, R_z is the surface finish parameter, mcm, φ is the angle of inclination of the machined contour, degree.

Feed for the drilling at NC machine, (mm/rev):

$$F_0 = k_1 D_d^x. \tag{1.15}$$

Table 1.17

Parameters for the drilling

Machined material	Range of the feed	k	x
Carbon and alloyed steel	1	0.025	0.77
	2	0.025	0.77
	3	0.025	0.77
Cast iron, copper and aluminum alloys	1	0.025	0.77
	2	0.025	0.77
	3	0.025	0.77

Where k, x are the constants, D_d is the diameter of the drill, mm; first range of feeds is for drilling in a rigid part without tolerance and with tolerance according 12 class of accuracy, second range of feeds is for drilling in nonrigid parts; third range of feeds is for drilling of the precise holes.

Feed for milling of the flat surfaces at NC machine, (mm/rev):

$$F_0 = kk_{sz}D_m^x z^y t^p. \quad (1.16)$$

Table 1.18

Parameters for milling of the flat surfaces

Machined material	k	x	y	p
Carbon and alloyed steel	0.007	1.27	-0.64	-0.44
Cast iron, copper and aluminum alloys	0.009	1.30	-0.64	-0.44
Practice factor depending on surface finish				
surface finish parameter R_z, mcm	80	40	20	2.5
Practice factor k_{sz}	1.5	1.0	0.6	0.4

Where k, x, y, p are the constants, k_{sz} is a practice factor; D_m is the diameter of the milling cutter, z is the number of the tooth of the cutter; t is the cutting depth, mcm.

1.10. Rating of the technological operation

In a batch production the floor-to-floor time is calculated as

$$T_f = T_o\left(1 + \frac{\alpha + \beta + \gamma}{100}\right), \quad (1.17)$$

where T_o is machining time, α, β, γ are the constant defining time for technical, organization service and break time ($\alpha = 6$, $\beta = 0.6$–8.0, $\gamma = 2.5$), %.

Machining time is calculated as

$$T_o = \frac{L_l}{Fzn}, \quad (1.18)$$

where F is a feed, mm/min; z is a number of the tooth of the cutter, n is number of the number of revolutions of the spindle.

2. MODELING OF THE MACHINING IN CAM ADEM

Fundamentals

ADEMNC Procedure

The following are steps of the NC program creation:

1. Create a construction entity (pocket, shoulder, plane, hole, etc.).
2. Apply a machining operation (milling, drilling, turning, punching, etc.) to the created construction entity.
3. The result of steps 1 and 2 is a Machining Object (MO).
4. Repeat steps 1-2 for each machining object.
5. If necessary, create machine commands (Home Position, Clear Plane, Stop, etc.).
6. Organize the created machining objects in the right order. This step is optional.
7. Calculate the tool path.
8. Simulate the machining process. This step is optional.
9. Generate, view and save an NC program. Before generating an NC program, you must select the machine tool and make some additional settings. Though you can make them any time during the creation of a machining process, it is recommended to do so at the beginning.

ADEMNC Terminology

Machining Process is a sequence of Machining Objects, which you create to describe what you are going to machine and how you are going to do it. A machining process consists of machining objects.

Machining Object (MO) - Either a Construction Entity with a single Machining Operation, or a Machine Command.

Construction Entity (CE) is the smallest geometrical unit to be machined with one machining operation. In ADEMNC, there are 13 types of construction entities, which enable you to describe any part to be machined. All the construction entities are created in the same way. First, you must select the construction entity you want to create, then specify its parameters and select the contours that define the construction entity boundaries.

Machining Operation (MOp) is a procedure performed by a selected tool on a specified construction entity. All the machining operations are applied in the same way. To apply a machining operation to the active construction entity, select the appropriate command from the Machining Operations toolbar, specify the machining operation parameters and select the appropriate tool.

Machine Command (MC) - The machining process can contain special machine commands, which are not directly connected with machining (material removal). With the help of machine commands you can define some general technological features of the machining process, tool home position, or the plane, on which a tool moves at fast speed between the machining operations. Each machine command is considered to be a separate machining object, so you can manipulate and edit machine commands in the same way as machining operations or construction entities.

CLDATA is a set of instructions generated by the Processor describing the tools, their parameters and the geometrical definitions of the tool motions for all calculated machining objects.

NC Program is a sequence of commands for a specific model of an NC machine. Before generating an NC program, you must calculate the tool path using the Processor command and select an NC machine model.

Starting ADEMNC

The ADEMNC module can be started from ADEM2D.

To start ADEMNC

1. In ADEM2D, click **ADEMNC** on the **Modules** menu.

Opening a project

In this lesson, you will use the file **Sample_2.adm** stored in you training folder. It contains a drawing of the part to be machined.

To open a project

1. Click **File, Open.**
2. Double-click the **Sample_2.adm** file located in the **&ldots;ADEM\Help\Tutorial** folder.

ADEM opens the project and displays the part drawing stored in this project (Fig. 2.1).

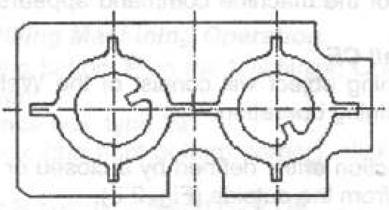


Fig. 2.1

Defining The Machining Process

Defining the Tool Home Position

The tool home position is defined by the Home Position machine command.

The Tool Home Position

Tool Home Position, also called the zero of a part cycle, is the spatial position of the tool's set point defined by ***XYZ*** coordinates set at the beginning or at the end of the machining operation. Either a spindle's or a tool-holder's set point, or the vertex of some operating or no-op tool, is set as a tool's set point.

To define the Tool Home Position

1. Click the **Home Position** button on the **Machine Commands** toolbar. The **Home dialog** box appears.
2. In the **Z coordinate** box, type 25.

3. Click **OK**. A new machine command **"Home"** has been created. The name of the machine command appears on the **Status Line (MO:1 Home)** (Fig. 2.2).

Fig. 2.2

Defining the Clear Plane

The Clear Plane is defined by the **Clear Plane** machine command.

The Clear Plane

A Clear Plane is a plane on which a tool moves at fast speed between the machining operations (MO). The system calculates the tool path according to the following rule: the tool moves from the reference point to the Clear Plane, from that point on the Clear Plane to the new point on the Clear Plane and from the Clear Plane to the end point along the shortest path.

To define the Clear Plane

1. Click the **Clear Plane** button on the **Machine Commands** toolbar. The **Clear Plane** dialog box appears.
2. Select the **Is defined** check box.
3. In the **Z coordinate** box, type **10**.
4. Click **OK**. A new machine command **"Clear Plane"** has been created. The name of the machine command appears on the Status Line (**MO:2 Clear Plane**).

Creating the Wall CE

The third machining object will consist of the Wall construction entity and the Milling machining operation.

The Wall CE

Wall is a construction entity defined by a closed or open boundary that is always machined from the outside (Fig. 2.3).

To create the Wall CE

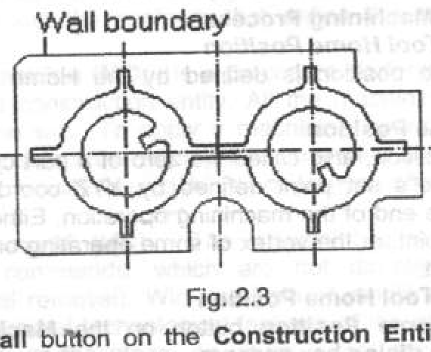


Fig.2.3

1. Click the **Wall** button on the **Construction Entities** toolbar. The **Wall dialog** box appears (Fig. 2.4).
2. In the **Depth** box, type 27 and click **OK**.

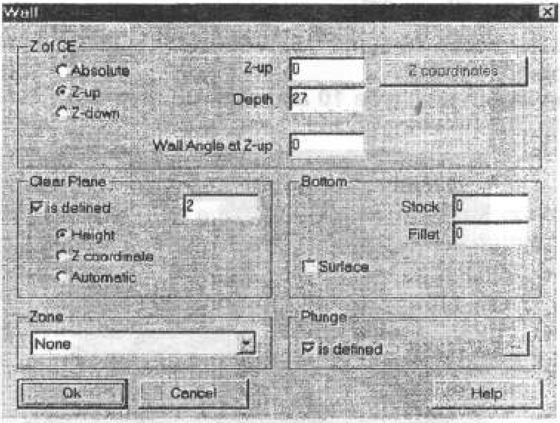


Fig. 2.4

3. Pick the contour highlighted in the illustration above. It will define the wall boundary.

4. Click the **middle** mouse button or press **Esc** to complete wall creation. The name of the construction entity appears on the Status Line (**MO:3 ***/Wall**).

Applying the Milling Machining Operation

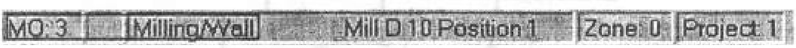
1. Click the Milling button on the Machining Operations toolbar. The Mill dialog box appears.

2. In the **Clearance** box, type **29**.

3. In the **Tool** box (the right-bottom corner), click ... The Tool dialog box appears.

4. In the **Corner R** box, type **5** and click **OK**.

5. Click **OK**. The machining operation has been created. The name of the new machining operation appears on the Status Line (**MO:3 Milling/Wall**).



Creating the Plane CE

The next machining object consists of the Plane construction entity and the Milling machining operation.

The Plane CE

Plane is a construction entity defined by a closed contour. A plane is created by removing the material from the top of the surface defined by the contour. You can create internal entities defined by closed contours, called islands, inside the plane.

To create the Plane CE

1.Click the Plane button on the Construction Entities toolbar. The Plane dialog box appears.

2. In the **Depth** box, type **10** and click OK (Fig. 2.5).

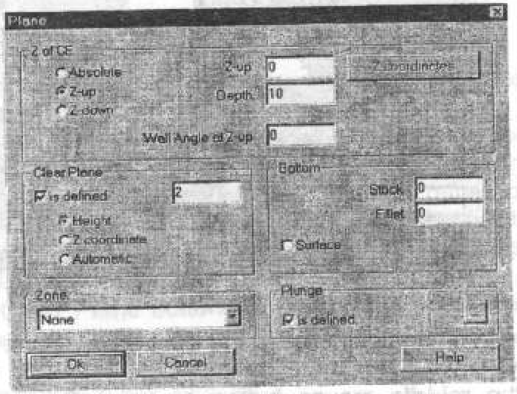


Fig. 2.5

3. Pick the contour **1** (Fig. 2.6) It will define the Plane boundary.

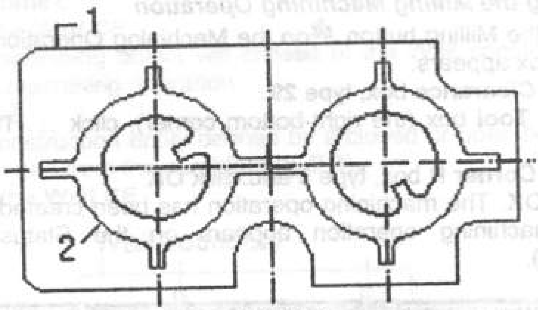


Fig. 2.6

4. Pick the contour **2** (see the illustration above). It will be an island inside the plane.

5. Click the **middle** mouse button or press **Esc** to complete Plane creation. The name of the construction entity appears on the Status Line **(MO:4 ***/Plane).**

Applying the Milling Machining Operation

1.Click the Milling button on the Machining Operations toolbar. The Mill dialog box appears.

2.In the **Clearance** box, type **12.**

3.From the **Type Of Milling** list box, select **Back Offset**.

4. Click **OK**. The machining operation has been created. The name of the new machining operation appears on the Status Line **(MO:4 Milling/Plane)**.

Creating the Window CE

The last machining object consists of the Window construction entity and the Milling machining operation.

The Window CE

Window is a construction entity defined by a closed contour. It has no bottom and is machined inside the boundary contour. You can machine several windows as one construction entity. No islands are allowed inside a window.

To create the Window CE

1. Click the Window button on the Construction Entities toolbar. The Window dialog box appears.

2. In the **Depth** box, type **27** and click **OK** (Fig. 2.7).

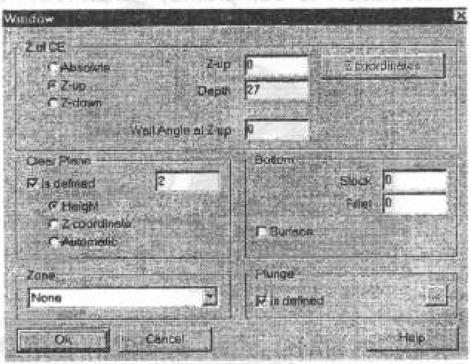


Fig. 2.7

3. Pick contours **1** and **2** (Fig. 2.8). It will define the boundaries of the windows.

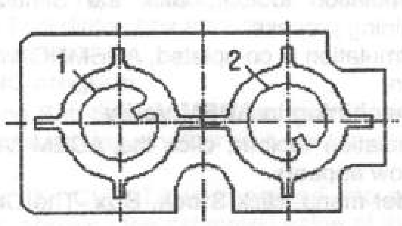


Fig. 2.8

4. Click the **middle** mouse button or press **Esc** to complete Window creation. The name of the construction entity appears on the Status Line **(MO:5 ***/Window)**.

Applying the Milling Machining Operation

1. Click the Milling button on the Machining Operations toolbar. The Mill dialog box appears.

2. In the **Clearance** box, type **29**.

3. Click **OK**. The machining operation has been created. The name of the new machining operation appears on the Status Line (**MO:5 Milling/Window**).

Calculating The Tool Path

Now that all the machining objects are defined, you can calculate the tool path. It is calculated by the Processor command. The result of the Processor operation is a CLDATA file, which contains a set of consecutive process and geometric commands that completely define machining for the intended NC machine.

To calculate the tool path

1. Click the Processor button on the Executing Sub-Systems toolbar.

2. When the calculation is completed, ADEMNC will display the tool path and a message will appear (Fig. 2.9). Click the OK button.

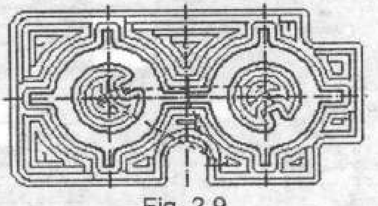


Fig. 2.9

Simulating Machining

After calculating the tool path, you can use built-in ADEMNC Simulation sub-system or the ADEM Verify module to dynamically display the current machining process on the screen.

To simulate machining

1. On the Simulation toolbar, click the Simulation button, to simulate the machining process.

2. When the simulation is completed, ADEMNC will display a message. Click the OK button.

To simulate machining in ADEM Verify

1. On the Simulation toolbar, click the ADEM Verify button. The ADEM Verify window appears.

2. On the **Model** menu, click **Stock, Box**. The Create Box dialog box appears.

3. Enter the following parameters:

4. **Corner 1**: **X=** -133; **Y=** - 68; **Z=** -20;

5. **Corner 2**: **X=** 133; **Y=** 68; **Z=** 0 (Fig. 2.10);

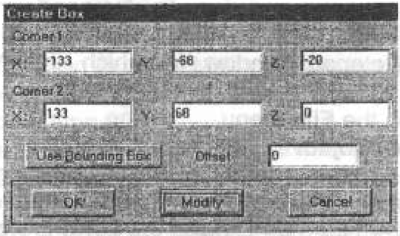


Fig. 2.10

6. Click the **Modify** button.
7. On the **Simulate** menu, click **Simulate.** Now the Simulation Mode is on.
8. On the **Simulate** menu, click **Start**.

The machined part should look like shown at Fig. 2.11.

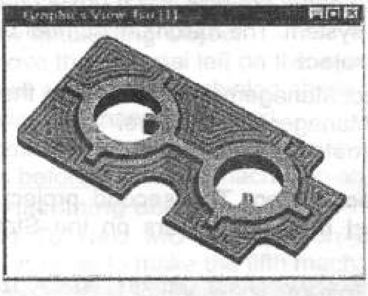


Fig. 2.11

Generating an NC Program

When the simulation is finished, activate the ADEMNC module. Now you will generate an NC program. Before generating an NC program you must select an NC machine type.

To select the machine type

1. Click the Machine Type button on the Project Management toolbar. The Machine Tool dialog box appears.
2. Double-click on **HITACHI**.

To generate an NC program

1. On the Executing Sub-Systems toolbar, click the Adapter button. A message box appears, asking whether you want to change the machine tool in the CLDATA.
2. Click Yes. After the CLDATA is translated into an NC program, a dialogue box appears, showing the estimated value of the time required to machine the part.

To view the NC program

1. Click the View NC Program button on the View toolbar. The Notepad window in which the NC program file is displayed appears.

2. View the NC program and close the Notepad window.

To save the project file

1. Close the Notepad window in which the NC program file is displayed.
2. Click **Save** on the **File** menu

Duplicating the Project

You will now duplicate the created machining process (with all machining operations). The duplicated project will have the same name as the original. However, you can change the current project name using the Information command.

NC Projects

ADEMNC enables you to work on several independent machining processes (projects) based on the same geometric model and store them in the same file. The Project Management feature allows you to work independently on each of the projects. Each project has its own name, scale factor and coordinate system. The maximum number of projects is 10.

To duplicate a project

1. Click the Project Management button on the Project Management toolbar. The Project Manager box appears.
2. Click the **Duplicate** button.

3. Click the **Select** button. The second project is now created and activated. The project number appears on the Status Line (**Project: 2**) (Fig. 2.12).

Fig. 2.12

Now, you will work on the second project.

Modifying the "Milling/Plane" MO Parameters

Now you will change the machining strategy in the 4th machining object ("Milling/Plane"). To change it, you should make it active, and then edit its parameters.

The number of the active machining object and its parameters are displayed on the status line. When a machining object is active, its geometry is highlighted.

To activate the "Milling/Plane" MO

1. Depending on which machining object is active now, click either the Jump To Previous MO button or the Jump To Next MO button on the Machining Object Management toolbar until the desired machining object is active (**MO: 4 Milling/Plane** should appear on the Status Line) (Fig. 2.13).

Fig. 2. 13

You can also use the **Page Up** and **Page Down** keys, to scroll through the machining process

To edit the Milling parameters

1. On the Machining Object Management toolbar, click the Edit Machining Operation button. The Milling dialog box appears.
2. From the **Type of Milling** list box, select **Zigzag**.
3. In the **Angle** box, type **90**.
4. In the **Internal** box, type **2** (this is the thickness of the material to be left on the island after execution of the current machining operation).
5. Click **OK**.

In order to make the changes take effect, you need to recalculate the tool path. You will recalculate the tool path only for the current MO.

To recalculate the tool path for the current machining object

1. Click the Process Current MO button on the Executing Sub-Systems toolbar.
2. When the calculation is completed, ADEMNC will display the tool path and a message will appear. Click the OK button.

3. **Inserting a New Machining Object**

Now, you will remove the material left on the island inside the Plane. In order to do this, you will add a new machining object "Milling/Wall" after the "Milling/Plane" MO using the Insert command.

The Insert command inserts a new machining object into your machining process <u>before</u> the active machining object.

To insert a new machining object

1. Click the Jump To Next MO button on the Machining Object Management toolbar, in order to make the fifth machining object active.
2. On the Machining Object Management toolbar, click the Insert button.
3. Click the Wall button on the Construction Entities toolbar. The Wall dialog box appears.
4. You will use the default parameters, so click OK.

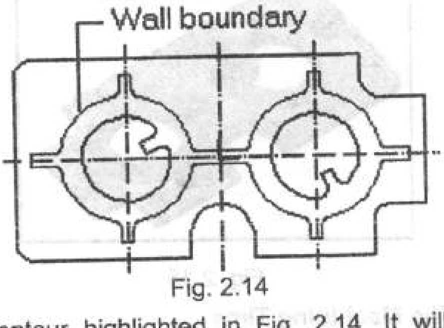


Fig. 2.14

5. Pick the contour highlighted in Fig. 2.14. It will define the wall boundary.
6. Click the **middle** mouse button or press **Esc** to complete wall creation. The name of the construction entity appears on the Status Line (**MO:5 ***/Wall**).

7. Click the Milling button on the Machining Operations toolbar.

8. In the **Clearance** box, type **12**.

9. Click **OK**. The machining operation has been created. The name of the new machining operation appears on the Status Line (**MO: 5 Milling/Wall**).

Recalculating the tool path

In order to make the changes take effect, you need to recalculate the tool path.

To recalculate the tool path

1. Click the Processor button on the Executing Sub-Systems toolbar.

When the calculation is completed, ADEMNC will display the new to Simulating machining

After recalculating the tool path, you can simulate machining in the built-in ADEMNC Simulation sub-system, as well as in ADEM Verify.

To simulate machining in ADEM Verify

1. Click the Machine Type button on the Project Management toolbar. The Machine Tool dialog box appears.

2. Double-click on **APT-CLI**.

3. Click **Yes**.

4. On the Simulation toolbar, click the ADEM Verify button. The ADEM Verify window appears.

5. On the **Model** menu, click **Stock**, **Box**. The Create Box dialog box with the last entered parameters appears.

6. Click the **Modify** button.

7. On the **Simulate** menu, click **Simulate**.

8. On the **Simulate** menu, click **Start**. The machined part is shown in Fig. 2.15.

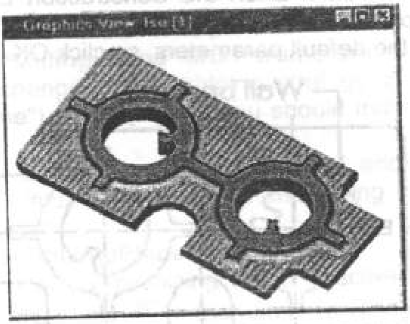


Fig. 2.15

Comparing The Machining Time

You can compare the estimated machining time in the first and second projects. To do so:

1. Click the Machine Type button on the Project Management toolbar. The Machine Tool dialog box appears.

2. Double-click on **HITACHI**.

3. On the Executing Sub-systems toolbar, click the Adapter button. A message box appears, asking whether you want to change the machine tool in the CLDATA.

4. Click Yes. After the CLDATA is translated into an NC program, a dialog box appears, showing the estimated value of the time required to machine the part (Fig. 2.16).

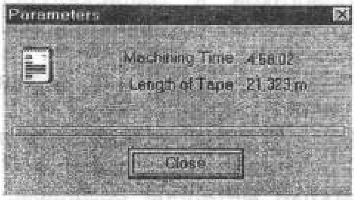


Fig. 2.16

5. Click the Project Management button on the Project Management toolbar. The Project Manager box appears.

6. Double-click on **[1] adem**. The first project is activated. The project number appears on the Status Line (**Project: 1**).

7. On the Executing Sub-systems toolbar, click the Adapter button. A dialog box appears, showing the estimated value of the time required to machine the part (Fig. 2.17).

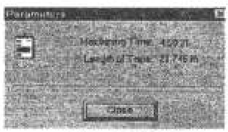


Fig. 2.17

Бібліографічний список

1. ADEM CAD. Руководство пользователя v. 8.
2. Технология самолетостроения: учеб. для авиац. вузов/ А.Л. Абибов, Н.М. Бирюков, В.В. Бойцов и др. – М.: Машиностроение, 1982. – 551 с.
3. Бабук В.В. Проектирование технологических процессов механической обработкой в машиностроении / В.В. Бабук, В.В. Шкред, Г.П. Кривко. – Минск: Вышэйш. шк., 1987. – 225 с.
4. Справочник технолога-машиностроителя: в 2 т.; под ред. А.Г. Косиловой и Р.К. Мещерякова. – М.: Машиностроение, 1985. – Т.1. – 656 с.
5. Балабанов А.Н. Краткий справочник технолога-машиностроителя / А.Н. Балабанов. – М.: Изд-во стандартов, 1992. – 464 с.
6. Панов А.А. Обработка металлов резанием: справ. технолога / А.А. Панов, В.В. Аникин, Н.Г. Бойм. – М.: Машиностроение, 1988. – 736 с.
7. Serope Kalpakjian and Steven R. Schmid, Manufacturing Engineering and Technology, 4th ed., Prentice-Hall, Inc., Upper Saddle River, NJ, 2001. – 1148 p.

В. В. Борисевич, А. А. Павленко, В.Т. Сікульський

MODELING OF MACHINING IN CAM "ADEM"

Редактор Т.А. Ястремська

Технічний редактор Л.О. Кузьменко

Зв. план, 2008
Підписано до друку 04.06.2008
Формат 60×84 1/16. Папір. офс. №2. Офс. друк
Ум. друк. арк. 2,2. Обл.-вид. арк. 2,56. Наклад 100 прим. Замовлення 259.
Ціна вільна

Національний аерокосмічний університет ім. М.Є. Жуковського
«Харківський авіаційний інститут»
61070, Харків-70, вул. Чкалова, 17
http://www.khai.edu
Видавничий центр «ХАІ»
61070, Харків-70, вул. Чкалова, 17
izdat@khai.edu