

CAPSmill™

Software for cycle time reduction and programming



For CNC Machining Centers

CAPSmill cuts your costs

Reduce cycle time

Specially designed to generate minimal cycle time programs

Reduce machine downtime

For dry runs, program entry, program editing and correction

Reduce part rejections

During tryouts

Reduce programming time

First-time-right programs, fast

Reduce tool cost

Efficient and appropriate tool paths

Reduce damage to machine

Eliminate accidents, spindle overuse

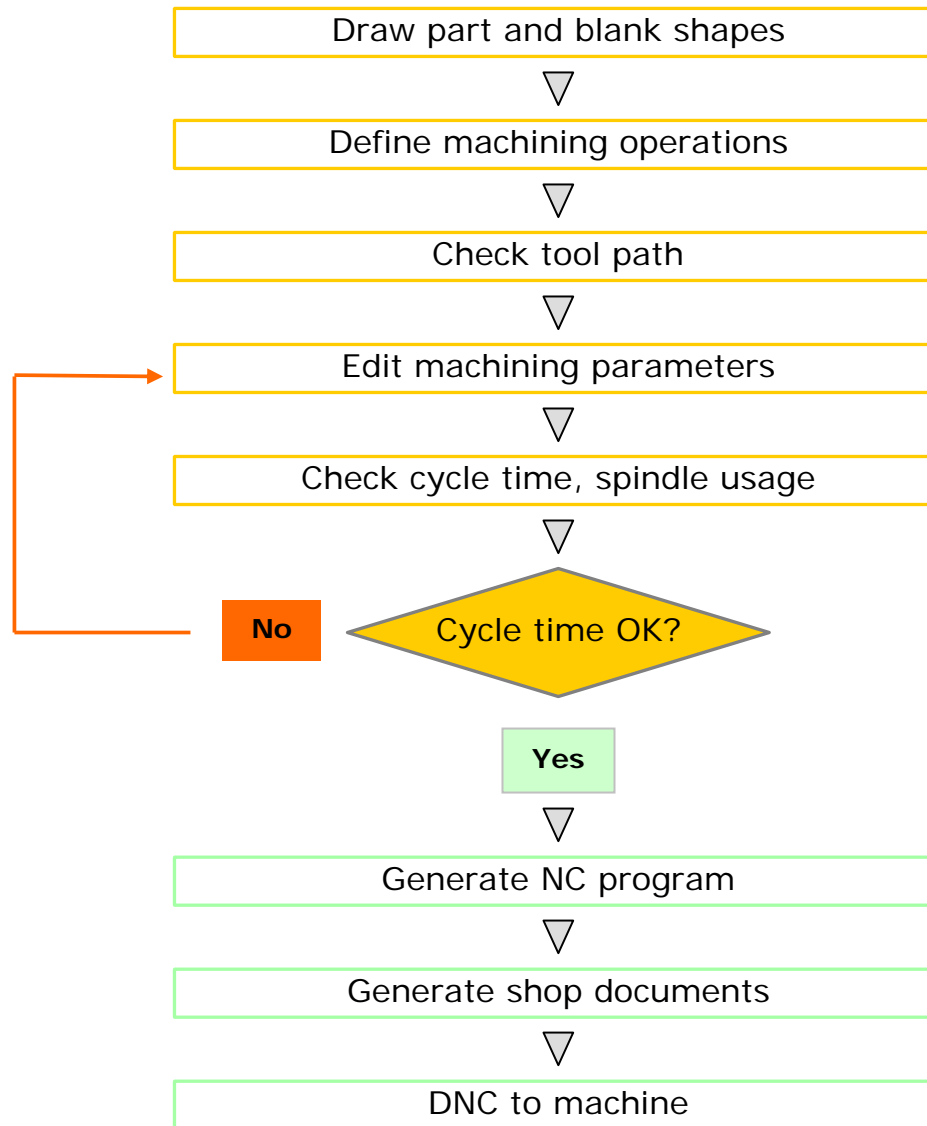
Reduce documentation time

Shop documents generated automatically

Reduce skill level of programmer

Does not need an engineer - a machinist can do the programming

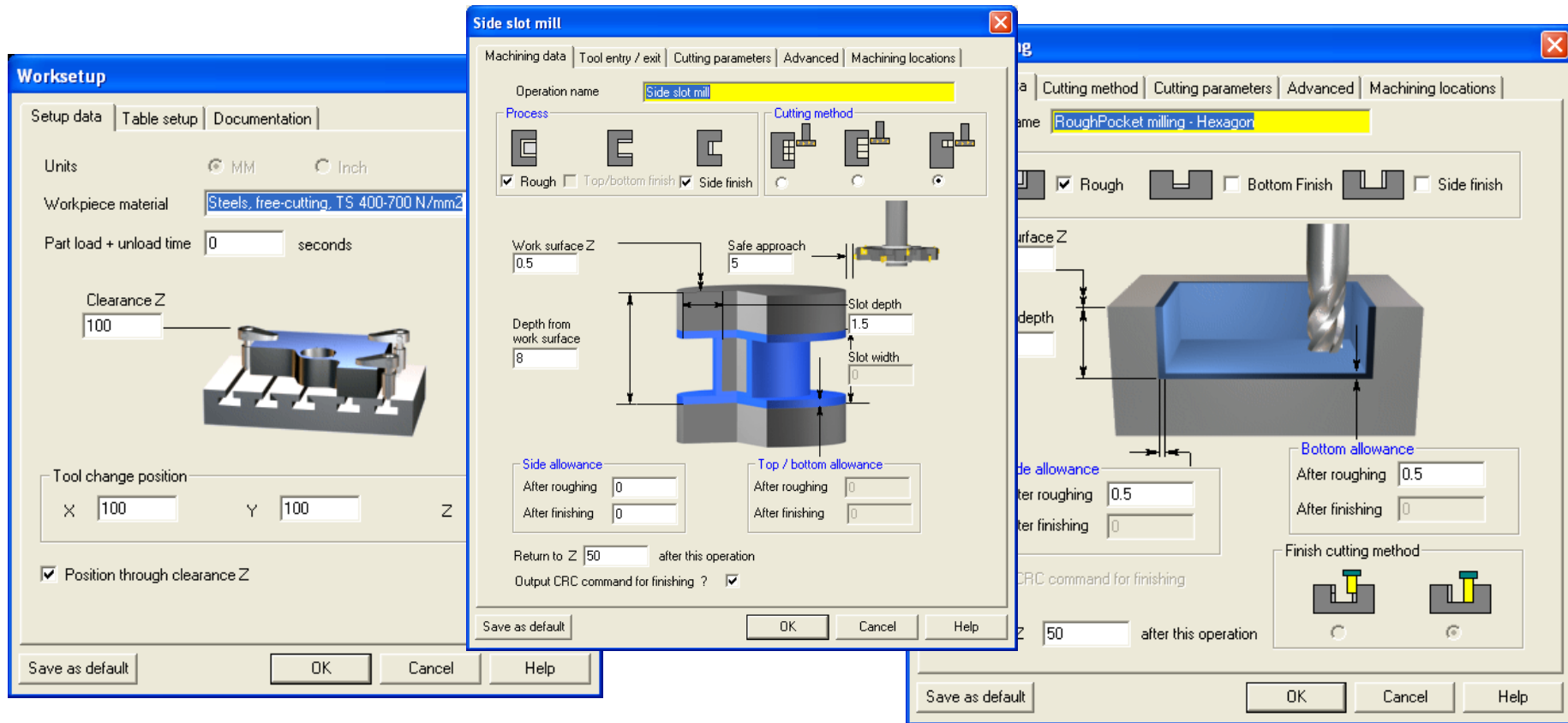
How CAPSmill works



Conversational software

No CNC programming knowledge needed – just machining knowledge.
Training time is typically 2 – 4 hours.

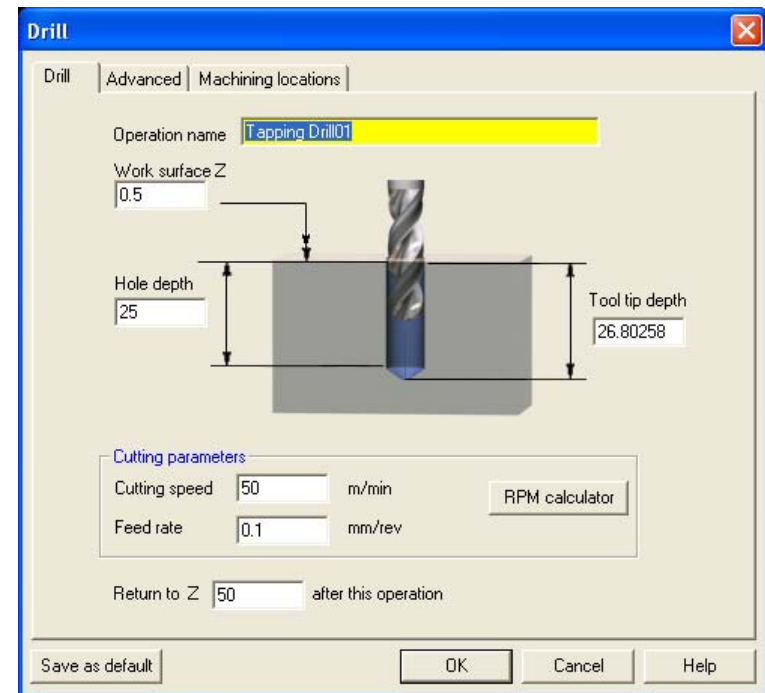
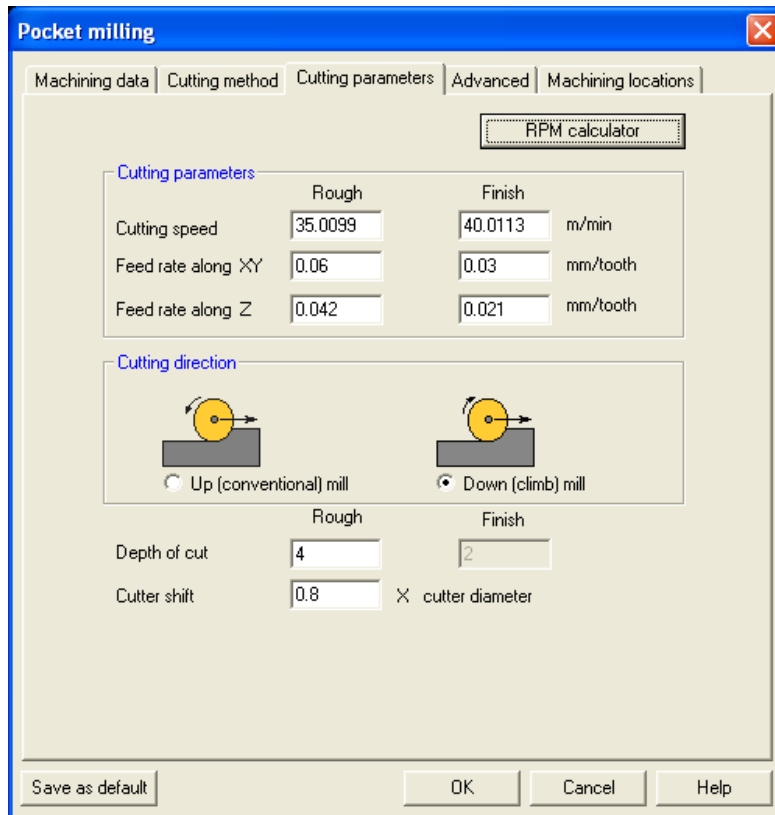
Reduce programming time
Reduce skill level of programmer



Automatic cutting parameters selection

Eliminates a big cause of low cycle times – poor FS selection.
Reduces tool wear caused by poor FS selection.

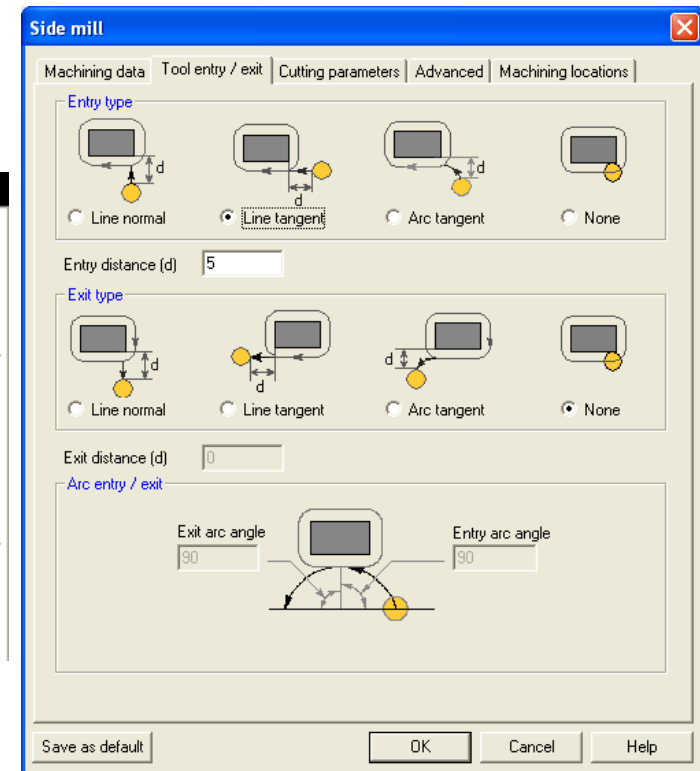
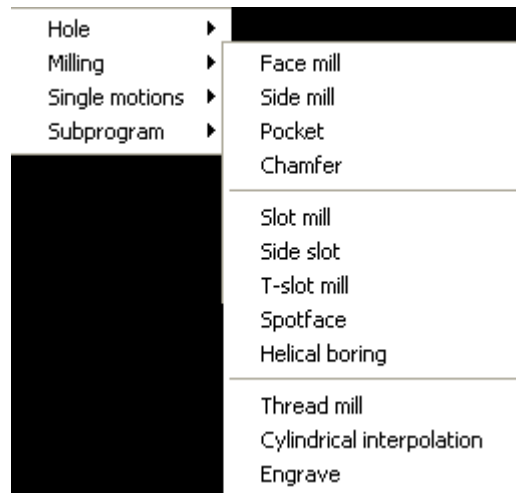
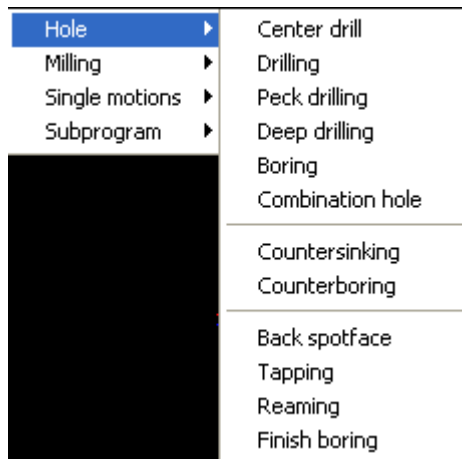
- Reduce cycle time
- Reduce tool cost
- Reduce skill level of programmer



Wide variety of operations with unique tool paths

Eliminates a big cause of low cycle times – inefficient tool paths.
Improves tool life because of proper usage.

- Reduce cycle time
- Reduce tool cost
- Reduce programming time
- Reduce skill level of programmer



Tool selection expert system

Suggests appropriate tools based on operation being performed.
Spindle direction is automatically decided – eliminates a big cause of errors.
Eliminates dry run to check for such errors.

Reduce machine downtime
Reduce programming time
Reduce skill level of programmer

The image displays two overlapping software dialog boxes. The background dialog is titled "Select tool" and has two tabs: "Tool" and "Tool parameters". Under the "Tool" tab, there is a "Tool library" dropdown set to "General (mm)". Below it, a "Select tool type" tree shows "Face mill" expanded, with "Face mill - 45 degree" selected. A list of tools is shown, including "50.00 mm. dia. Face mill" through "250.00 mm. dia. Face mill". The "Selected tool" field contains "50.00 mm. dia. Face mill". Under "Select tool material", "Carbide" is selected. At the bottom, there are input fields for "Tool number" (15), "Length offset number" (15), and "Diameter offset" (15). A "Save as default" button is at the bottom left, and "OK", "Cancel", and "Help" buttons are at the bottom right.

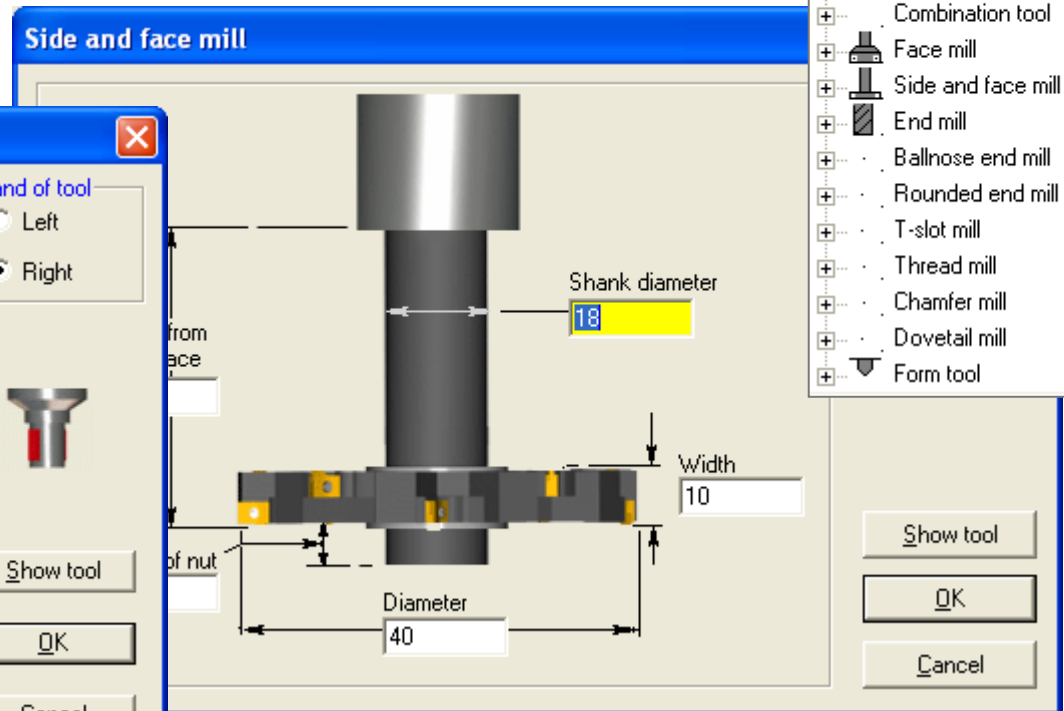
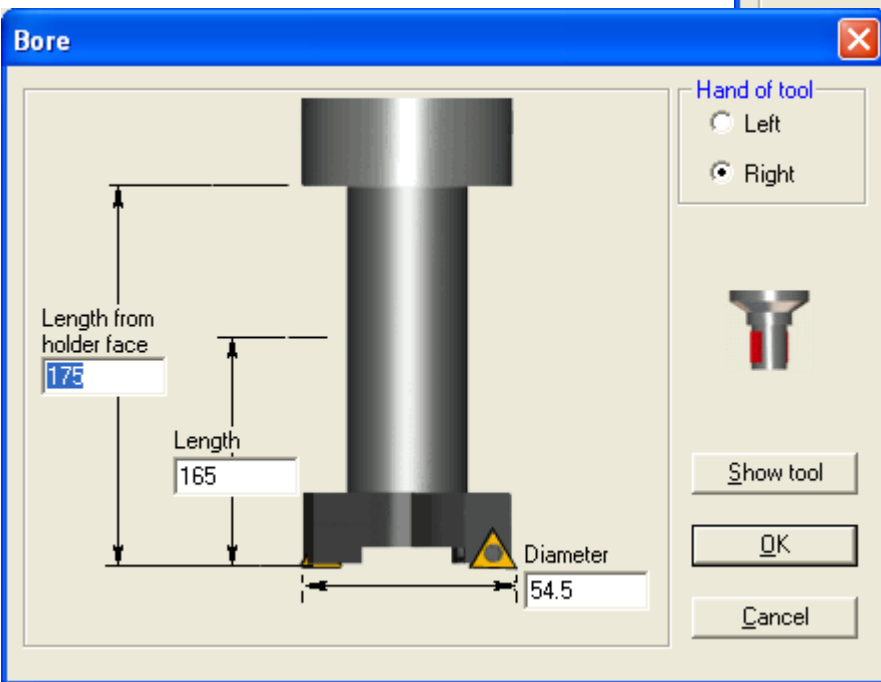
The foreground dialog is titled "Face mill" and shows a 3D model of a face mill tool with various dimensions labeled. The "Number of teeth" is set to 4. The "Hand of tool" is set to "Right". The "Shank diameter" is 22. The "Length from holder face" is 150, and the "Body height" is 40. The "Maximum depth of cut" is 6, the "Inside dia" is 50, and the "Approach angle" is 45. The "Outside dia" is 62.5. There are "Show tool", "OK", and "Cancel" buttons at the bottom right.

Tools database

Extensive library of tools commonly used in machining.
Can be configured by the user.
Tools can be classified according to manufacturer.

Reduce machine downtime
Reduce skill level of programmer

- + Tap
- + Hole mill
- + Trepanning cutter
- + Bore
- + Countersink
- + Back spotface
- + Combination tool
- + Face mill
- + Side and face mill
- + End mill
- + Ballnose end mill
- + Rounded end mill
- + T-slot mill
- + Thread mill
- + Chamfer mill
- + Dovetail mill
- + Form tool



Automatic tool gouge prevention

Tool removes only whatever material it can, does not gouge into part.
Reduces cycle time - you can use roughing tools to the maximum.

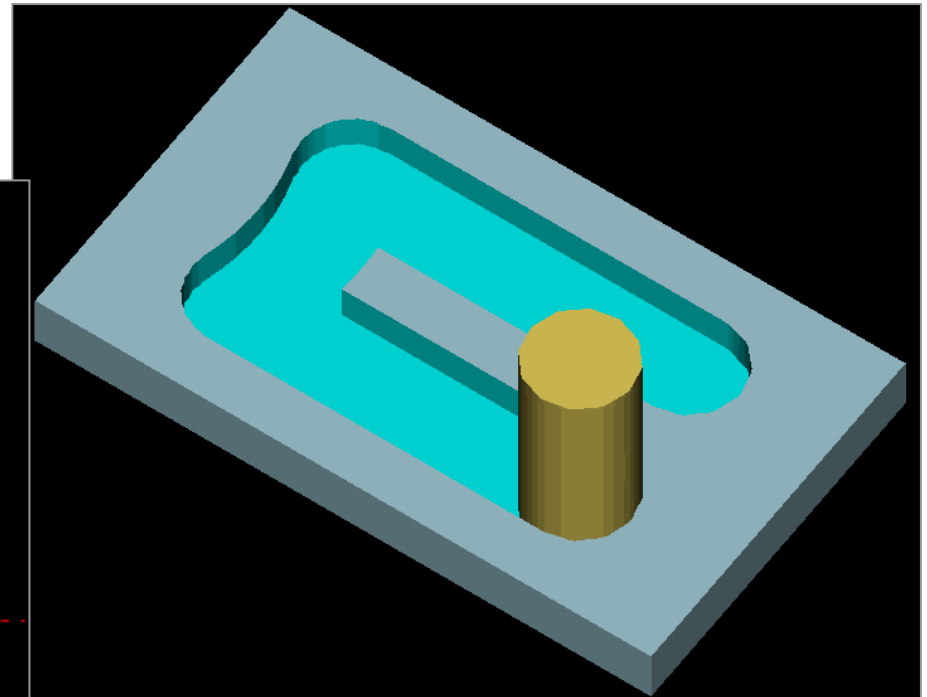
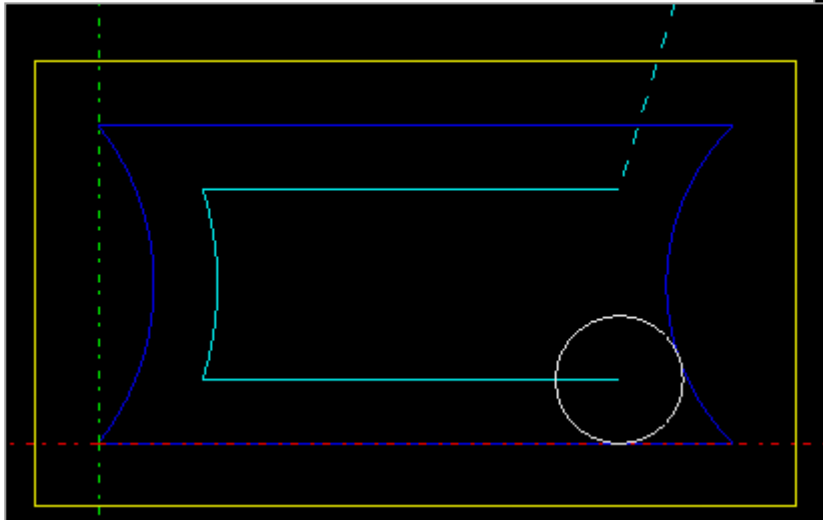
Reduce cycle time

Reduce machine downtime

Reduce tool cost

Reduce damage to machine

Reduce skill level of programmer



Cycle time calculation

Extremely accurate cycle time – less than 1 % error.

Try out many process options, decide on the one with least cycle time.

Generates printable cycle time sheet.

Reduce cycle time

Reduce documentation time

Reduce skill level of programmer

Cycle time sheet

Machine name	Fanuc 18M	Work piece material	Steels, steel-cutting, TG 400-700 M/hr
Partnumber	05	Fixture	
Partname	Sample-mm-01	Programmer	CADEM
Date	11 June 2009	Setup number	1

Sl no.	Operation	Tool	Tool no.	Cutting speed		Feed rate		Cut length	Cutting time	TC time	Rapid time	Total time
				m/min	RPM	mm/min	mm/s					
1	Face mill	100.00 mm. dia. Face mill	4	220.0	700	686.28	0.98	727.0	1.06	.15	.04	1.25
2	Face mill	100.00 mm. dia. Face mill	4	220.0	700	686.28	0.98	966.5	1.58	.00	.03	1.61
3	Side slotmill Rough & finish	4.50 mm. dia. T-slotmill	5	40.0	159	190.99	1.20	664.9	3.43	.15	.04	3.62
4	Center drilling	3.15 mm. dia. Center drill	6	35.0	3536	530.52	0.15	4.9	.01	.15	.03	.19
5	Center drilling	3.15 mm. dia. Center drill	6	35.0	3536	530.52	0.15	39.6	.07	.00	.09	.17
6	Drill	5.00 mm. dia. Twist drill	7	80.0	3183	190.99	0.06	22.5	.12	.15	.03	.30
7	Drill	5.00 mm. dia. Twist drill	7	80.0	3183	190.99	0.06	100.0	.52	.00	.10	.62
8	Counterbore	3.20-16.00 mm. Counterbore	9	60.0	1193	143.24	0.12	4.4	.03	.15	.03	.21
9	Counterbore	3.20-16.00 mm. Counterbore	9	60.0	1193	143.24	0.12	35.1	.26	.00	.09	.34
10	Tap	M5.00 x 1.00 Tap	8	7.0	371	371.36	1.00	40.0	.12	.15	.03	.30
11	Tap	M5.00 x 1.00 Tap	8	7.0	371	371.36	1.00	320.0	1.00	.00	.08	1.07

Summary

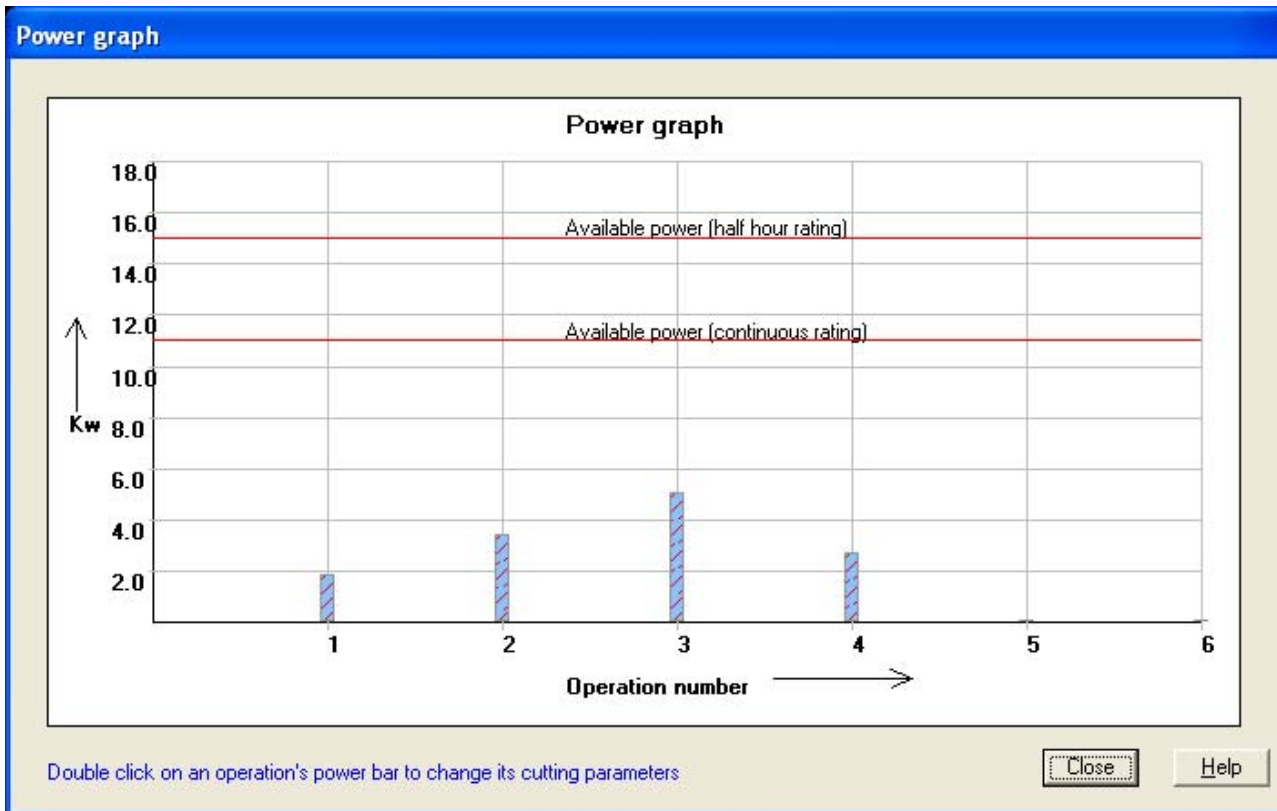
Total cutting time	:	8.21
Total tool change time	:	0.90
Total rapid motion time	:	0.58
Total miscellaneous time	:	0.55
Total cycle time	:	10.25 min.

Spindle power graph

Check if you are using the available spindle power to the maximum.
Check if you are over-using the spindle.

Reduce cycle time

Reduce damage to machine

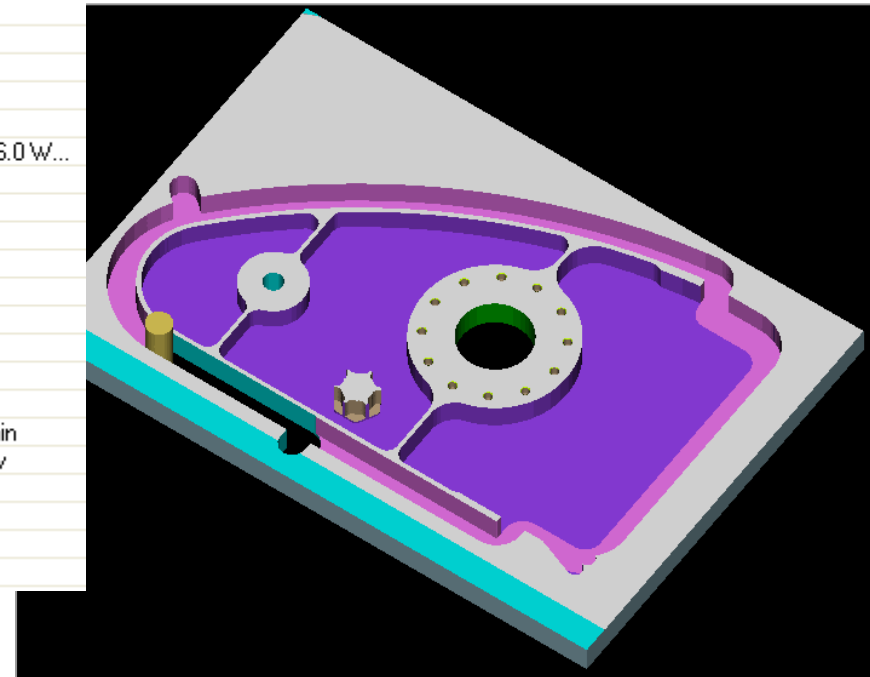
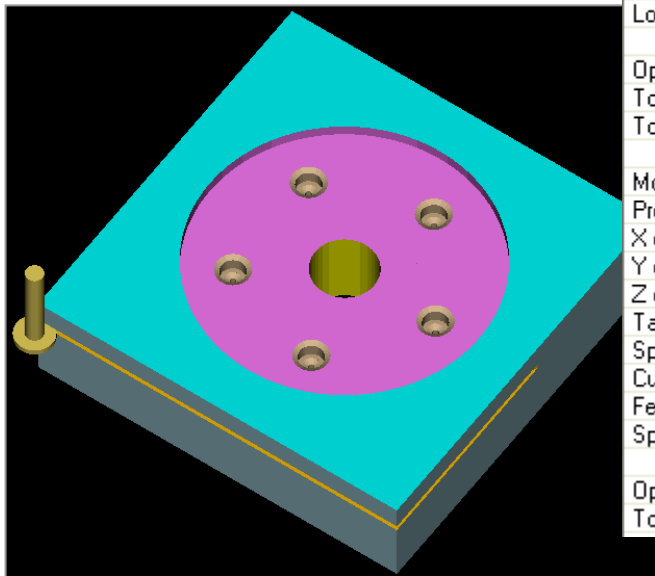


Highly effective tool path simulation

Variety of simulation options - Zoom, pan, single block, tool-wise, auto
Dynamic tool position and parameter display.
Eliminates dry run, rejections and accidents.

- Reduce machine downtime
- Reduce part rejections
- Reduce damage to machine

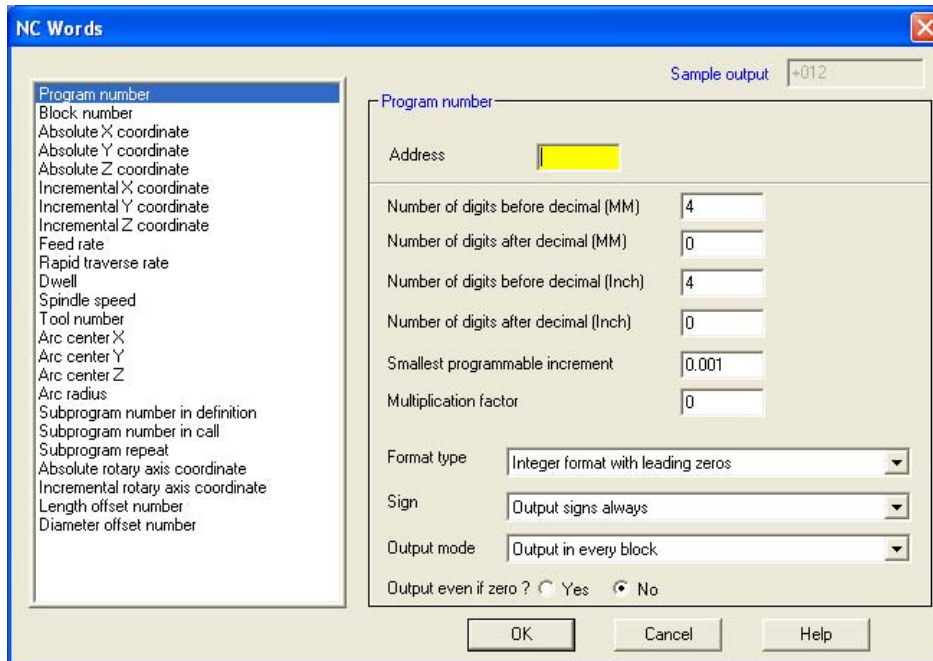
Parameter	Parameter
Machine name	Fanuc 18 M
Machining location	
Work coord system	
Local coord system	
Operation	Side slot mill
Tool name	40.00 mm. dia., 6.0 W...
Tool diameter	40.0000 mm
Motion	Linear
Process	Rough
X coord	-194.0000
Y coord	-214.0000
Z coord	-31.0000
Table position	
Spindle speed	803.7325 rpm
Cutting speed	101.0000 m/min
Feed rate	0.4800 mm/rev
Spindle status	CW
Operation time	1.32765 min
Total time	86.93388 min



Efficient, fully documented programs

Easy to understand because of extensive comments
Has canned cycles, nose radius compensation commands
No editing required on machine.
Build your own machine-specific program format, easily.

Reduce programming time
Reduce machine downtime
Reduce skill level of programmer



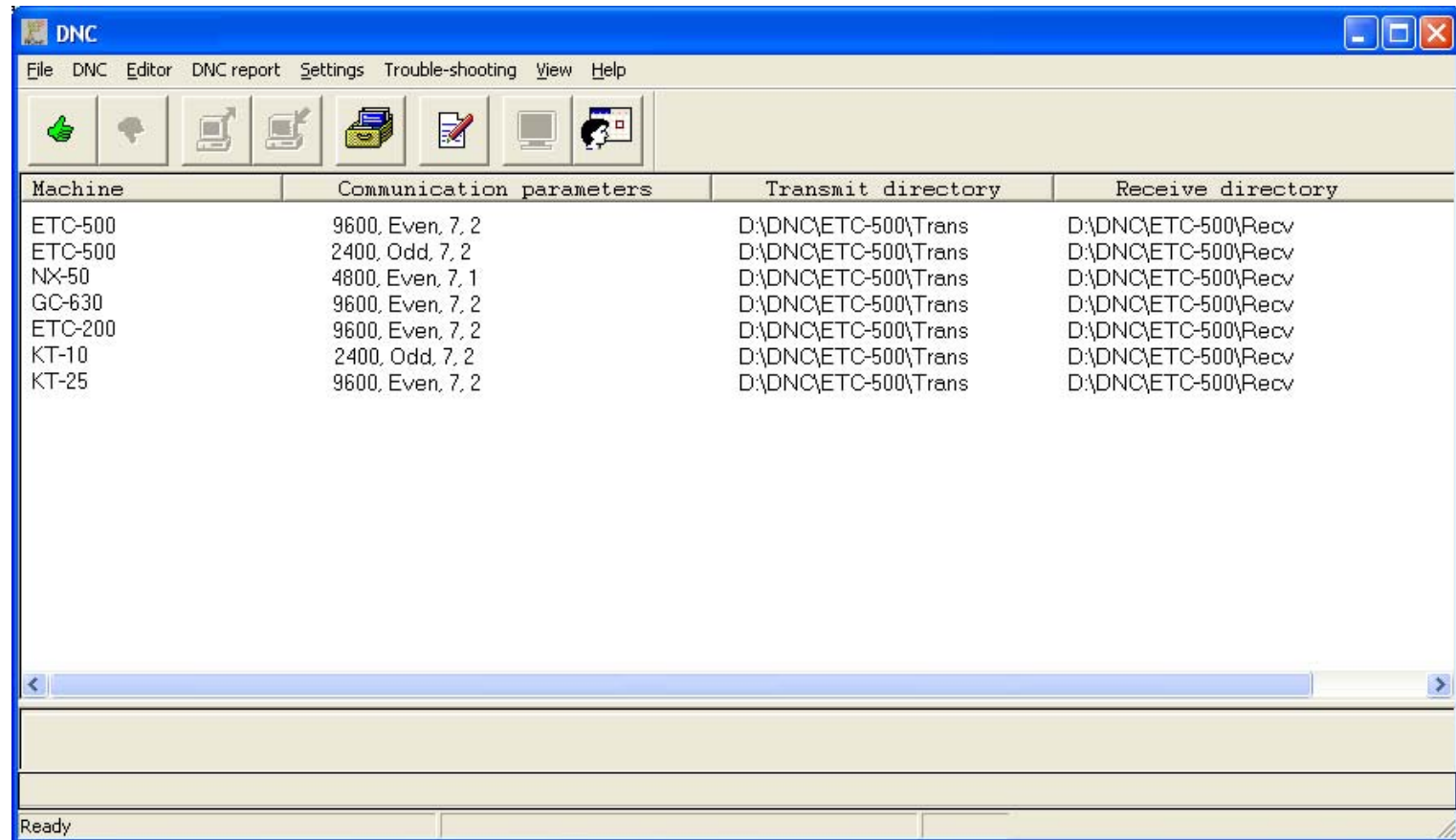
The screenshot shows a text editor window with a menu bar (File, Edit, View, Settings, Utilities, Help) and a toolbar. The text content is a G-code program with extensive comments:

```
%  
O1234  
(STEELS, FREE-CUTTING, TS 400-700 M  
(CLAMPING TEST ITEM)  
(CM57-14)  
(1)  
(04-06-2009)  
G21 G90 G94  
N1 T15 M06 (80.00 MM. DIA. ROUND  
G90  
(ROUGH FACE MILL)  
M3 S1030  
G00 G54 X60. Y85.4 M8  
G44 H15 Z100.  
Z6.  
G01 Z0.5 F605  
M98 P00010055  
G90 G00 X125. Y234.6  
Z1.5  
X60. Y85.4  
S1263  
G01 Z0. F318  
X365. F455  
Y139.8  
X60.
```

DNC

Inbuilt DNC eliminates program entry time

Reduce machine downtime



The screenshot shows the DNC software interface. The window title is "DNC". The menu bar includes "File", "DNC", "Editor", "DNC report", "Settings", "Trouble-shooting", "View", and "Help". The toolbar contains icons for a thumbs up, a thumbs down, a printer, a computer monitor, a folder, a document with a pencil, a computer monitor, and a person icon. The main area displays a table with the following data:

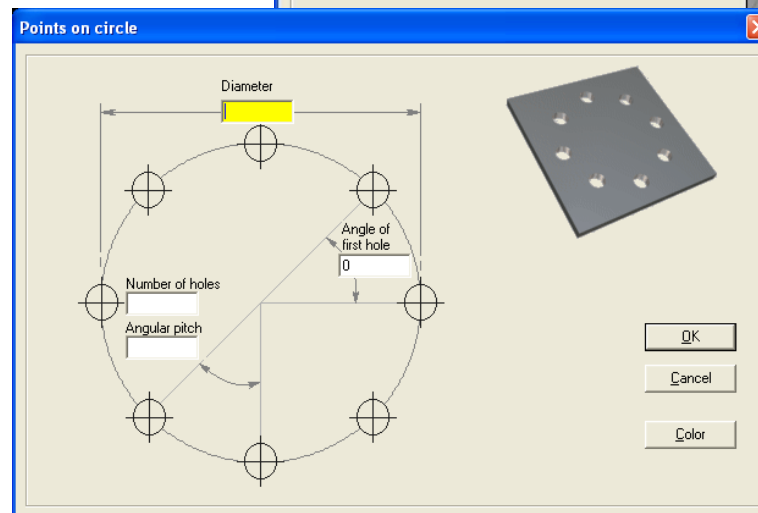
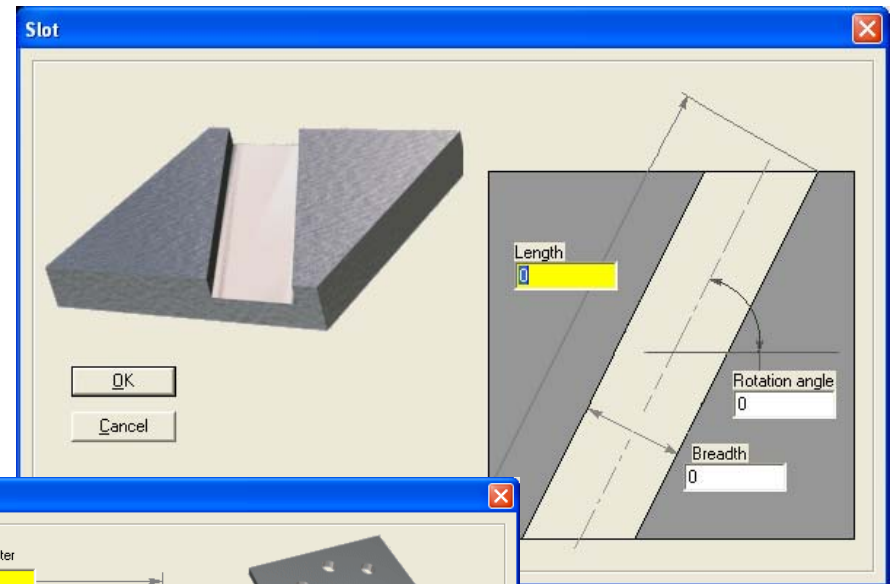
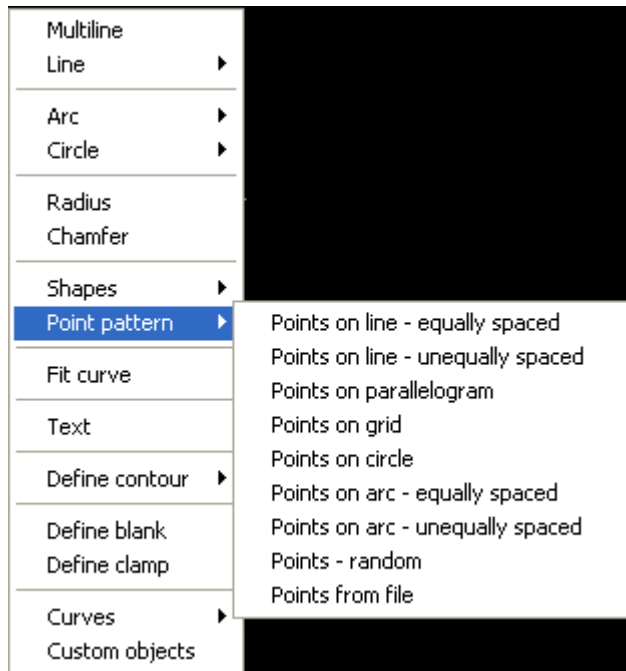
Machine	Communication parameters	Transmit directory	Receive directory
ETC-500	9600, Even, 7, 2	D:\DNC\ETC-500\Trans	D:\DNC\ETC-500\Recv
ETC-500	2400, Odd, 7, 2	D:\DNC\ETC-500\Trans	D:\DNC\ETC-500\Recv
NX-50	4800, Even, 7, 1	D:\DNC\ETC-500\Trans	D:\DNC\ETC-500\Recv
GC-630	9600, Even, 7, 2	D:\DNC\ETC-500\Trans	D:\DNC\ETC-500\Recv
ETC-200	9600, Even, 7, 2	D:\DNC\ETC-500\Trans	D:\DNC\ETC-500\Recv
KT-10	2400, Odd, 7, 2	D:\DNC\ETC-500\Trans	D:\DNC\ETC-500\Recv
KT-25	9600, Even, 7, 2	D:\DNC\ETC-500\Trans	D:\DNC\ETC-500\Recv

The status bar at the bottom left shows "Ready".

CAD for milling

Special functions for quick definition of parts
Can import part and raw material shapes from CAD software

Reduce programming time
Reduce skill level of programmer



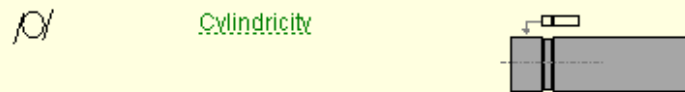
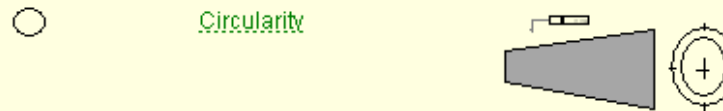
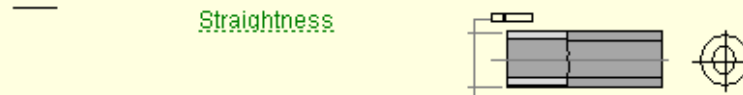
Machining expert

Quick reference for machining data
Diagnostics for machining problems

Reduce programming time
Reduce skill level of programmer

Geometrical accuracy definitions

Symbol	Meaning	Indication on the drawing
—	Straightness	
▱	Flatness	
○	Circularity	
∅	Cylindricity	



Tolerances

Basic hole



Diameter steps in mm		H6	H7	H8	H9	H11
Over	Upto					
1	3	+6 0	+10 0	+14 0	+25 0	+60 0
3	6	+8 0	+12 0	+18 0	+30 0	+75 0

Tool and F-S selection

ISO material classification

P	Steel	Blue
M	Stainless steel	Yellow
K	Cast Iron	Red

Application classification

F: Finishing. Low depth of cut, low feed rate

Typical Depth of cut = 0.2- 0.8 mm., Feed rate = 0.2 mm/rev., Nose radius = 0.2 mm.

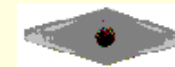
M: Medium rough machining. Single pass machining of casting / forging

Typical Depth of cut = 0.8- 3 mm., Feed rate = 0.3 mm/rev., Nose radius = 0.4 mm.

R: Rough machining.

Typical Depth of cut = 3- 7 mm., Feed rate = 0.5 mm/rev., Nose radius = 0.8 mm.

Procedure



	F	M	R
P			
M			
K			






















Select insert shape
Negative insert for general
Positive insert for turning

Select insert grade and

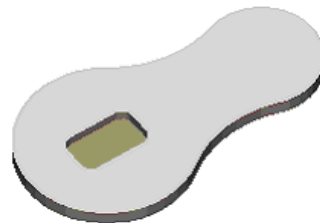
Tutorials

Cover all aspects of the software
Enable you to learn the software yourself, in 2 to 4 hours.

Reduce skill level of programmer

-  **Contents**
-  License agreement
-  Getting started
-  **Tutorials in mm**
-  Tutorial 1
-  **Tutorial 2**
-  Tutorial 3
-  Tutorial 4
-  Tutorial 5
-  Tutorial 6
-  Tutorial 7
-  Tutorial 8
-  Tutorial 9
-  **Customizing CAPSmill**
-  Tutorial 10
-  Tutorial 11
-  Tutorial 12
-  **Tutorials in Inch**
-  Tutorial 1
-  Tutorial 2
-  Tutorial 3

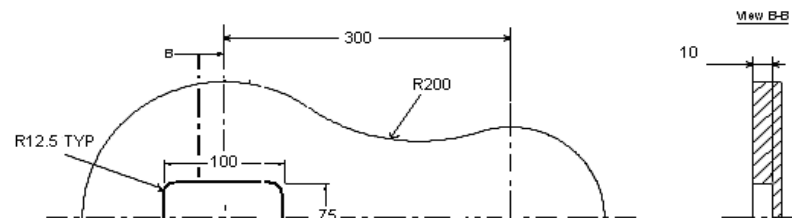
Tutorial 2



This tutorial shows

- Face milling
- Pocket milling
- Side milling

Part drawing



Shop floor documentation – automatic

Use in shop floor, file away for future reference.

Tools list – can be given to operator for tool setting

Tool layout sheet - graphical details of each operation.

Eliminates errors in information flow to shop floor.

Reduce machine downtime

Reduce documentation time

Reduce skill level of programmer

Time study sheet

Cycle time sheet

Machine name: Fanuc 18 M
Part number: 9
Sample-mm-09
Date: 11 June 2009

Work piece material: Non-Fe matls, Al > 16% Si, Alumin
Fixture
Programmer: CADBM
Set up number: 1

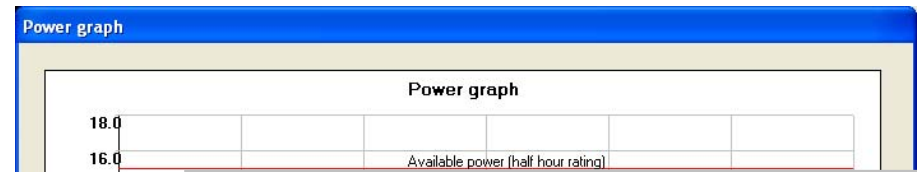
SL no.	Operation
1	Rough Face mill
2	Rough Outer Contour Side m
3	Rough Pocket01milling
4	Rough Pocket02 milling
5	Rough Pocket03 milling
6	Hexagon top surface milling
7	Centre Drill01
8	Centre Drill03
9	Step Drill01
10	Step Drill02

Tools list sheet

Tools list

Machine name: Fanuc 18 M
Part number: 9
Sample-mm-09
Date: 11 June 2009
Fixture
Programmer: CADBM
Set up number: 1

SL no.	Operation	Tool	Tool no.	Length offset no.	Length offset mm.	Radius offset n
1	Rough Face mill	160.00 mm. dia. Face mill	1			
2	Rough Outer Contour Side milling	20.00 mm. dia. End	2			
3	Rough Pocket01milling	12.00 mm. dia. End	3			
4	Rough Pocket02 milling	12.00 mm. dia. End	3			
5	Rough Pocket03 milling	12.00 mm. dia. End	3			
6	Hexagon top surface milling	12.00 mm. dia. End	3			
7	Centre Drill01	3.15 mm. dia. Center drill	5			
8	Centre Drill03	2.50 mm. dia. Center drill	19			
9	Step Drill01	6.50 mm. dia. Twist drill	6			
10	Step Drill02	13.00 mm. dia. Twist drill	7			
11	Step Drill03	20.00 mm. dia. Twist drill	8			



Tool layout sheet Part : KV-8891-H

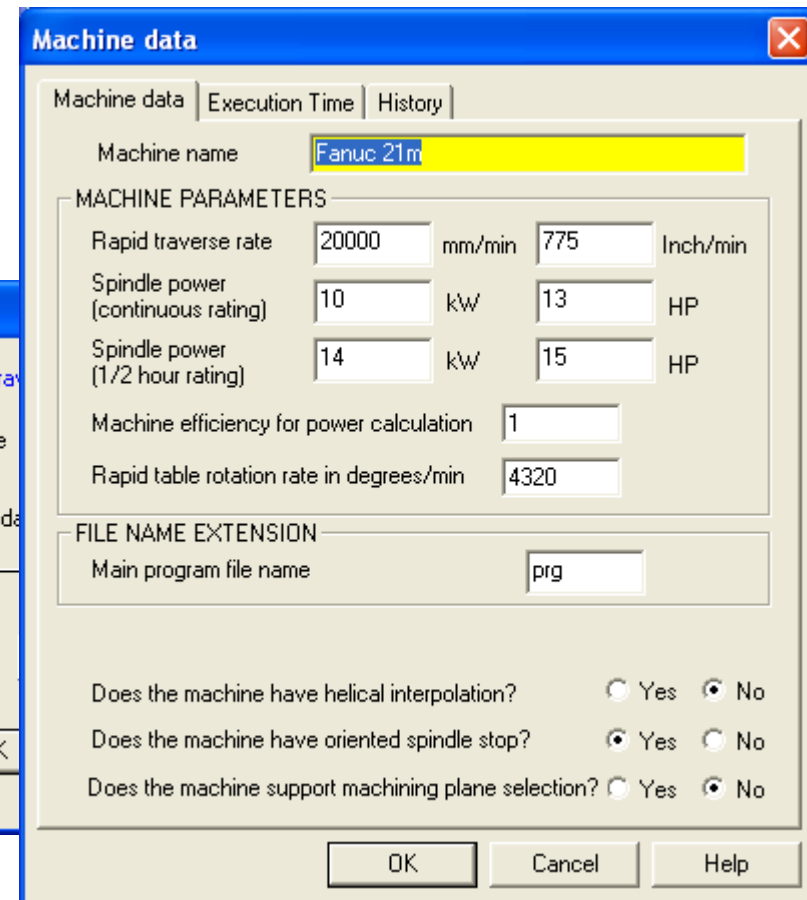
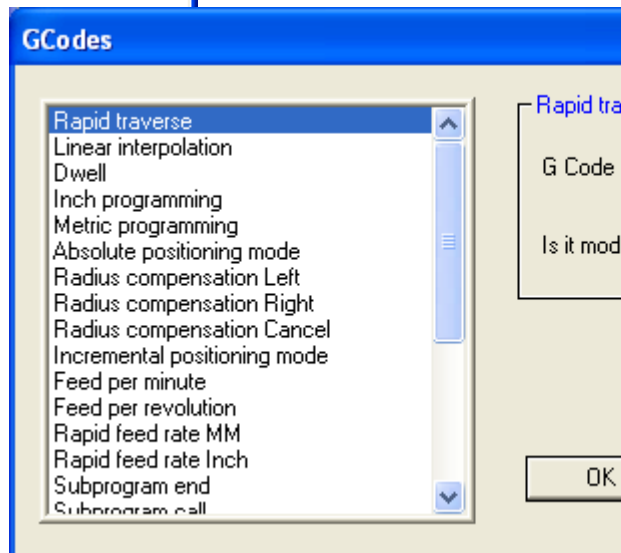
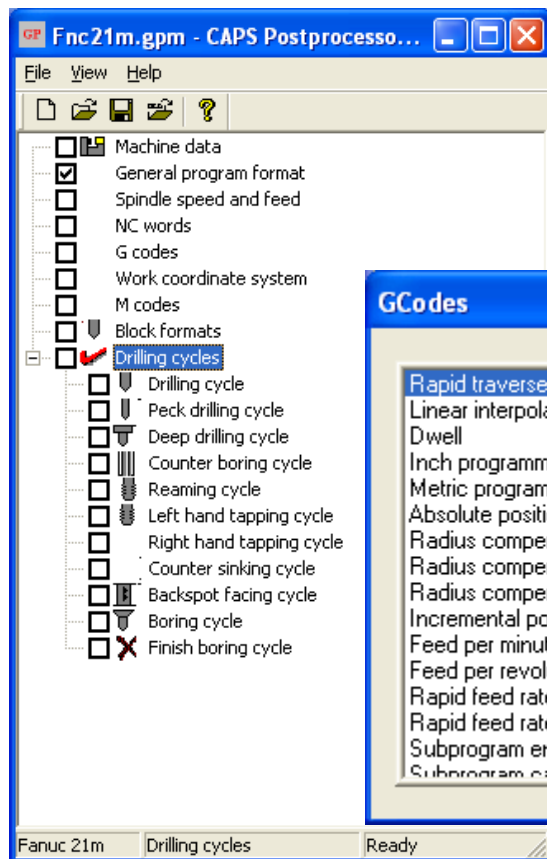
Operation : Finish pocket mill

Tool : 12.00 dia. end mill finish

Generic postprocessor

Makes posts to suit your machine and style of programming.
Conversational, very easy to use.

- Reduce machine downtime
- Reduce programming time
- Reduce skill level of programmer



Summary of features and benefits

Benefit	Features that enable the benefit
Reduce cycle time	Auto FS selection, Unique tool paths, Tool selection expert, Cycle time calculation, Spindle power graph
Reduce machine downtime	Tool path simulation, Tools database, Tool selection expert, Auto documentation, DNC
Reduce part rejections	Tool path simulation, Tool selection expert, Auto FS selection
Reduce programming time	Conversational screens, Tool selection expert, Auto tool gouge prevention, Tools database, Auto FS selection
Reduce tool cost	Auto FS selection, Unique tool paths, Tool selection expert
Reduce damage to machine	Auto tool gouge prevention, Tool selection expert, Tool path simulation, Spindle power graph, Auto FS selection
Reduce documentation time	Auto documentation
Reduce skill level of programmer	Conversational screens, Auto FS selection, Tools database, Unique tool paths, Tool selection expert, Cycle time calculation, Auto documentation, Tutorials

CADEM used by top companies



CADEM endorsed by OEMs

PMT MACHINES LIMITED

LML **LOKESH**
MACHINES LIMITED

ACE
DESIGNERS

hunt

BFW™

AMS

Thank you

For more information, contact
sales@cadem.com

Cadem Technologies Pvt.Ltd.

264/75, 36th cross, 8th Block Jayanagar,
Bangalore – 560 070, INDIA

T: +91 80 2663 4767

F: +91 80 2244 2732

www.cadem.com