CAM PROGRAMMING
(CAD CAM Applications)

Created by: **Muhammad Kevin Fahlevi**

Supervised by: **Dr. Norbert Geier**
Magyar Gergely
Lukács Tamás

Budapest University of Technology and Economics

Faculty of Mechanical Engineering
Department of Manufacturing Science and Engineering

CONTENTS

1. Introduction

CAD/CAM Programming is a vital aspect of modern manufacturing, integrating the design and manufacturing processes into a seamless workflow. CAD (Computer-Aided Design) software allows engineers to create precise 2D and 3D models, while CAM (Computer-Aided Manufacturing) software translates these designs into machine-readable instructions (G-code) for manufacturing.

Edgecam is one of the leading CAM software solutions, widely used for programming CNC (Computer Numerical Control) machines. It provides a comprehensive suite of tools for machining, turning, milling, and more. Edgecam's advanced features and intuitive interface make it a preferred choice for industries aiming to enhance production efficiency and precision.

Edgecam offers a range of advanced features that make it a powerful tool in CAD/CAM programming. It seamlessly integrates with various CAD software, allowing users to import designs from platforms like SolidWorks, Autodesk Inventor, and PTC Creo. Its toolpath creation capabilities enable precise machining strategies for operations such as contouring, pocketing, and drilling, with 3D simulation ensuring error-free execution. Edgecam's intuitive interface is designed to simplify workflows, making it accessible for both beginners and experienced programmers. The software supports advanced machining processes, including 2D and 3D milling, turning, and multi-axis machining, with options like wave-form roughing and high-speed machining. Additionally, it offers customizable post-processors and macro support, enabling users to tailor their G-code generation and automate repetitive tasks. With robust simulation and verification tools, Edgecam ensures collision-free machining and high accuracy, making it a comprehensive solution for modern manufacturing needs.

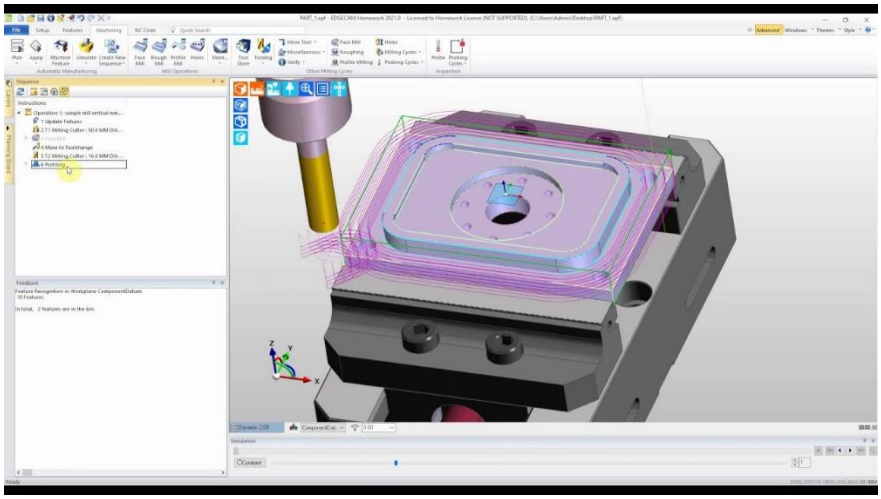


Figure 1. Edgecam layout

2. Assignment Lists

CAD/CAM applications	[BMEGEGTBG65]
Homework Assignment	CAM programming

2.1 Part I: CAD

1. Check whether the machining operation can be conducted on an **NCT/EmL-510** machine tool centre. If not, find another one which is able to manufacture the part or propose an engineering solution, how the machining task could be carried out!
2. Create a 3D solid model of the part and the stock[1] using an arbitrarily chosen CAD software based on the drawings! Take care of the appropriate modelling of the dimensional tolerances!
3. Create or download a simple fixture model for the machining operation! The default fixture models in the EdgeCAM's database can be also applied.

2.2 Part II: CAM

4. Create the CAM model for the part machining based on the operation sheet using the latest student version[2] of the **EdgeCAM** software (or older version)!
5. Create the roughing and rest-roughing tool paths! Apply the given cutting parameters (see the operation sheet)! Optimise the roughing tool paths to minimise the machining time!
6. Create the finishing tool paths to finish the functional surfaces ($Ra \leq 3{,}2$ μm)! All of the functional surfaces have to be finished by climb milling (also known as down milling) strategy. Apply the given cutting parameters (see: operation sheet)! Be aware of the surface roughness and tolerance requirements of the part!
7. Create the drilling, reaming and threading cycles. Apply the given cutting parameters (see: operation sheet)!
8. Simulate the CAM program and make sure it does not contain any collisions!
9. Optimise the roughing cycles to minimise tool path, *i.e.* minimise unnecessary tool movements and select appropriate strategy (circular, lace *etc.*). Collect the total machining time for each machining strategy.

[1] Offset contour of the part is cut from sheet metal by thermal or waterjet cutting (allowances can be found on the drawing)
[2] Download from here: https://enterprisegroup.hu/plm/edgecam/edgecam-diakv

2.3 Part III: DOCUMENTATION

10. Create a pdf documentation about the homework assignment (1st page is a cover page, 2nd and 3rd pages are this document, 4th page is the drawing, 5th page is the operation sheet, 6th and 7th page is the textual-documentation)! The textual-documentation has to include the explanation of the first task (machine-tool selection), the manufacturing time of the different roughing tool paths, the full manufacturing time and the reasons for them.
11. Create a short (no longer than 5 minutes!) MS PowerPoint presentation to present your homework assignment! It has to include the definition of the tasks and a short video of the simulation of tool path (advanced simulation with the part, stock and cutting tools).

2.4 Cutting tools that can be used:

Ø50x20	Roughing face mill	M8x15	Thread drill
Ø60x10	Finishing face mill	Ø10H7x15	Reamer
Ø20x50	End mill	Ø9,8x15	Twist drill
Ø10R1x8	Radius end mill	Ø5,9x20	Twist drill
Ø10	End mill	Ø10	Twist drill
Ø8	End mill	Ø8	Twist drill
Ø3R0,5x5	Radius end mill	Ø6H7x15	Reamer
Ø6,75x15	Twist drill	M6x15	Thread drill
Ø15/Ø5	Chamfer drill	Ø5x20	Twist drill

Submission

The deadlines and the requirements regarding the homework submission can be downloaded from the official MS Teams group.

3. Given CAD Drawing

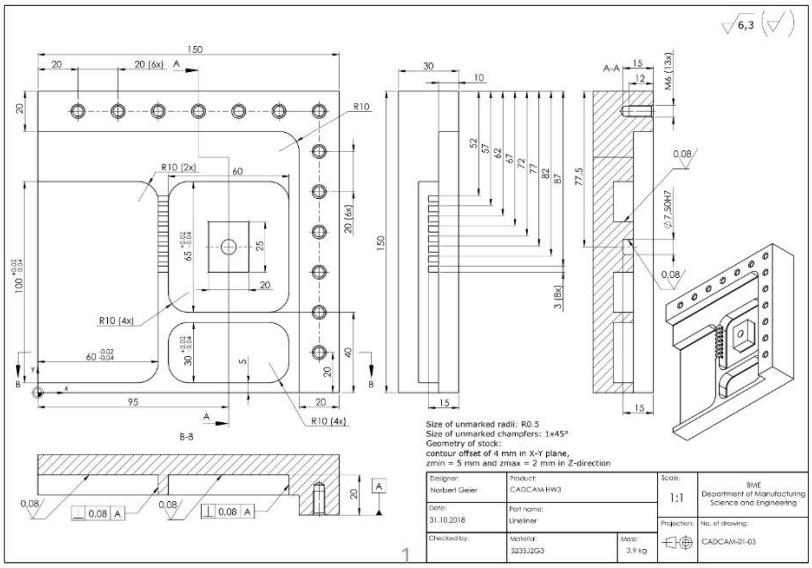
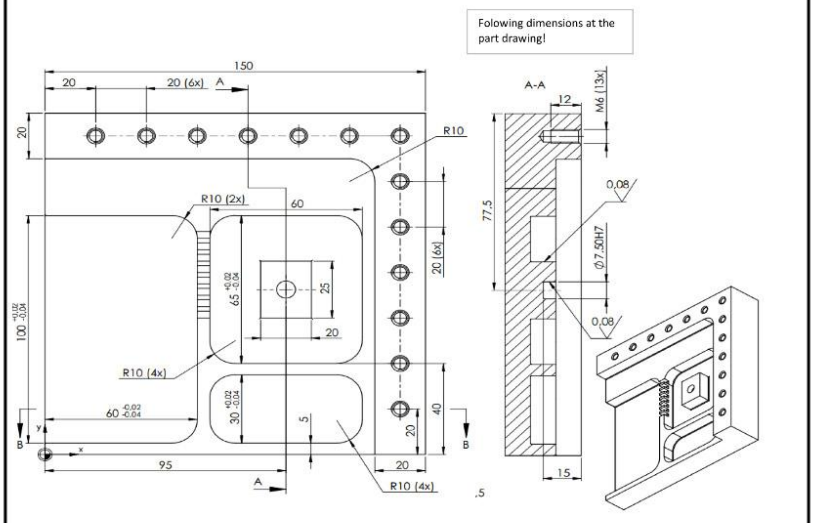


Figure 2. Technical drawing of the model

BME Department of Manufacturing Science and Engineering	*Manufacturing programming* **NC machine-tool centre**		NC program:	O1234
			Created:	Norbert Geier
			Signature:	
Machine-tool: **NCT/EmL-510**	**Operation sheet**		Part no.	CADCAM-01-01
Control: **NCT 201**			Date: 22.09.	Pages: 1/1
Part name: Presser	Part material: S235J2G3		Fixture:	Clamp
Operation no.: 2.	Max. stock size: 158x158x37	Operation name:		Milling

Sketch:



No.	Si.	Name of the operation element	Tool number	v_c m/min	a_e mm	a_p mm	f_z mm/z	f mm/rot
2.1.	-	Clamping	-	-	-	-	-	-
2.2.	-	Face rough-milling	**T1**	300	15	3	0,1	0,6
2.3.	-	Face finish-milling	**T2**	320	20	2	0,1	0,8
2.4.	-	Roughing	**T3**	180	2	3	0,06	0,36
2.5.	-	Rest material roughing	**T4**	190	2	1	0,06	0,36
2.6.	-	Finishing of functional surfaces	**T5**	190	1	1	0,06	0,36
2.7.	-	Drilling	**T14**	115	-	-	0,08	0,16
2.8.	-	Chamfering	**T7**	95	1	1	0,1	0,2
2.9.	-	Thread drilling	**T13**	15	-	-	-	1

Operation time	20 min	Piece time	60 min	Series nu.:	16 pcs.	Get: QM

Figure 3. Given operation of the model

4. Designed CAD Model - Solidworks

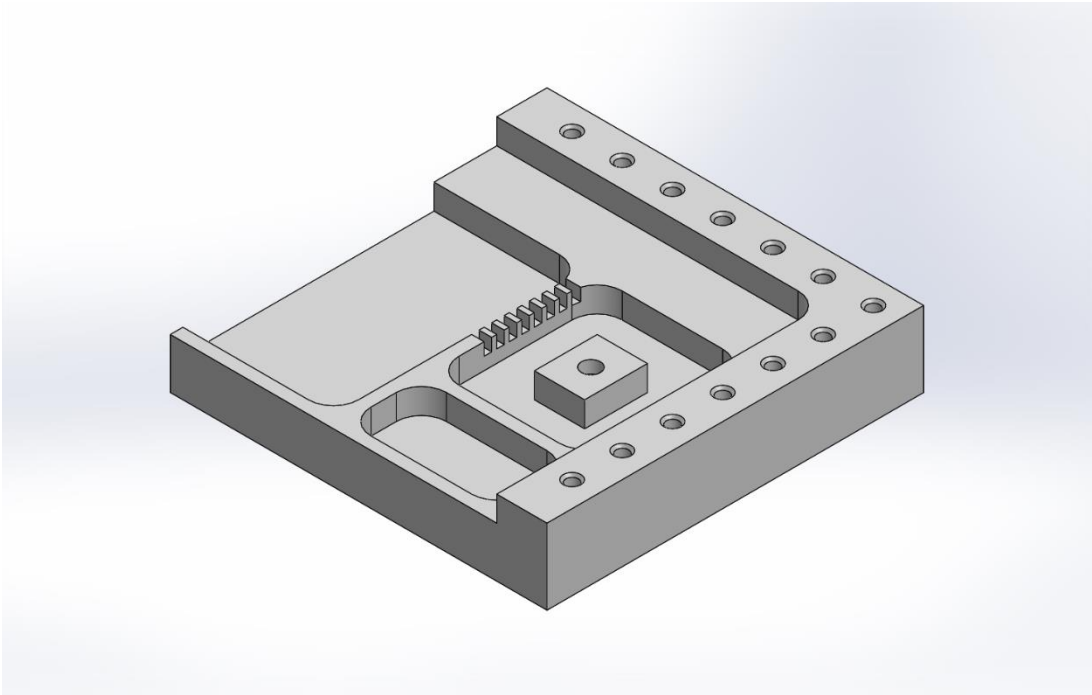


Figure 4. CAD model of the part

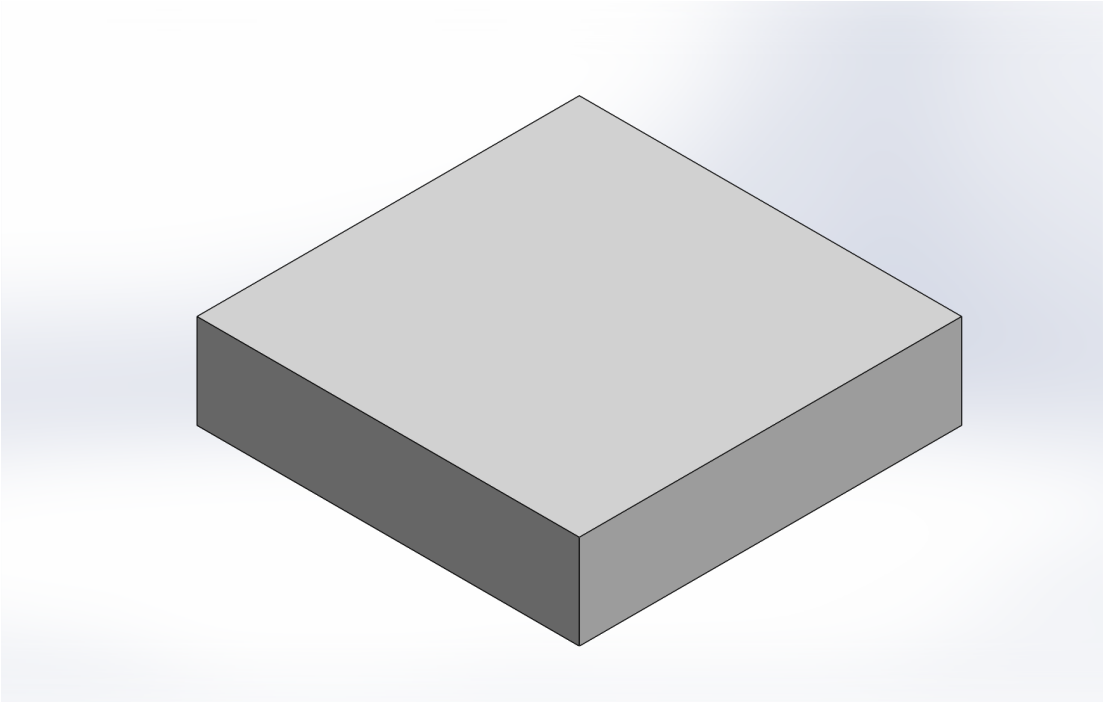


Figure 5. CAD Model of the stock

5. CAM Processing Data

Manufacturing data:

- Machine: NCT/Eml-510
- Chosen clamping: KURT 3600
- Total time: 35 min 29 sec
- Total operation: 34

Machine tool centre data:

- Table size: 600 x 320 mm
- Table load capacity: 300 kg
- X-axis travel: 510 mm
- Y-axis travel: 410 mm
- Z-axis travel: 460 mm
- Spindle speed: Standard up to 8,000 RPM; optional up to 12,000 RPM
- Spindle taper: #40, compatible with DIN or BT tooling
- Rapid traverse speed (X/Y/Z): 36/36/30 m/min
- Tool magazine capacity: Standard 12 tools; optional up to 20 tools.

Figure 6. NCT EmL – 510 machine

These specifications suggest that the EmL-510 is an excellent match for the CAM process described in your document. Its travel ranges are suitable for the part dimensions, and the spindle speeds are aligned with the specified cutting parameters. Furthermore, the tool magazine capacity and tool size limitations are compatible with the tools detailed in the operation sheet. As a result, the NCT EmL-510 machine is well-equipped to and carry out the required machining operation.

6. Operational Manufacturing Parameters

Tool Number	Main Operation	n [rpm]	Stepover [%]	Tool Reasoning	Manufacturing time [hour:min:s]
T1: 50.0 MM DIA X 90 DEGREE FACE MILL	Face rough-milling	1909.85	30	To efficiently and accurately remove material while leaving a 0.5 mm layer for the finishing process	00:06:02
T2: 60.0 MM DIA X 90 DEGREE FACE MI LL	Face finish-milling	1697.65	33.33	Complete the work initiated by T1, ensuring a smooth surface finish	00:01:05
T3:10.0 MM DIA MULTI-FLUTE END MILL	Roughing	5729.58	20	Its diameter is optimally sized to enable a fast and efficient roughing cycle	00:09:04
T4:2.0 MM DIA X 0.5 MM RAD ENDMILL	Rest material roughing	10000	100	This step prepares the surface for finishing, and using a smaller tool helps minimize stress and potential damage	00:00:49
T5: 10 MM DIA X 1 MM RAD END MILL	Finishing of functional surfaces	6047.89	-	Climb mills and finishes	00:15:09
T14: 6 MM DIA DRILL	Drilling	6100.94	-	According to the standard M6 thread	00:00:35
T15: 7.5 MM DIA MACHINE REAMER	Simple drilling	4880.75	-	To create a 7,5H7 hole	00:00:01
T7: 15.0 MM DIA X 90 COUNTERSINK - 3.0MM MIN DIA	Chamfering	2015.96	-	Chamfer tool, reduces the stress on the edges	00:01:06
T13: M6 X 1.0 RH SPIRAL FLUTE TAP	Thread drilling	795.77	-	Specified by the standard for creating an M6 threaded hole	00:00:35

7. Conclusion

The document demonstrates the integration of CAD and CAM processes in manufacturing, focusing on using the NCT EmL-510 machine for machining operations. Through the comprehensive application of EdgeCAM software, the assignment explores efficient material removal, surface finishing, and thread creation, ensuring compliance with technical standards and achieving precise tolerances.

The selected NCT EmL-510 machine aligns with the operational requirements due to its optimal table size travel ranges, spindle speeds, and tool magazine capacity. The document also highlights a systematic approach to machining, involving face milling, roughing, finishing, drilling, and threading operations, with specific tools chosen for their efficiency and precision.

The overall manufacturing process is streamlined and optimized, achieving high-quality outcomes within reasonable total machining time of 35 minutes and 29 seconds. This approach underscores the effectiveness of modern CAD/CAM integration in achieving precision and efficiency in manufacturing workflows.