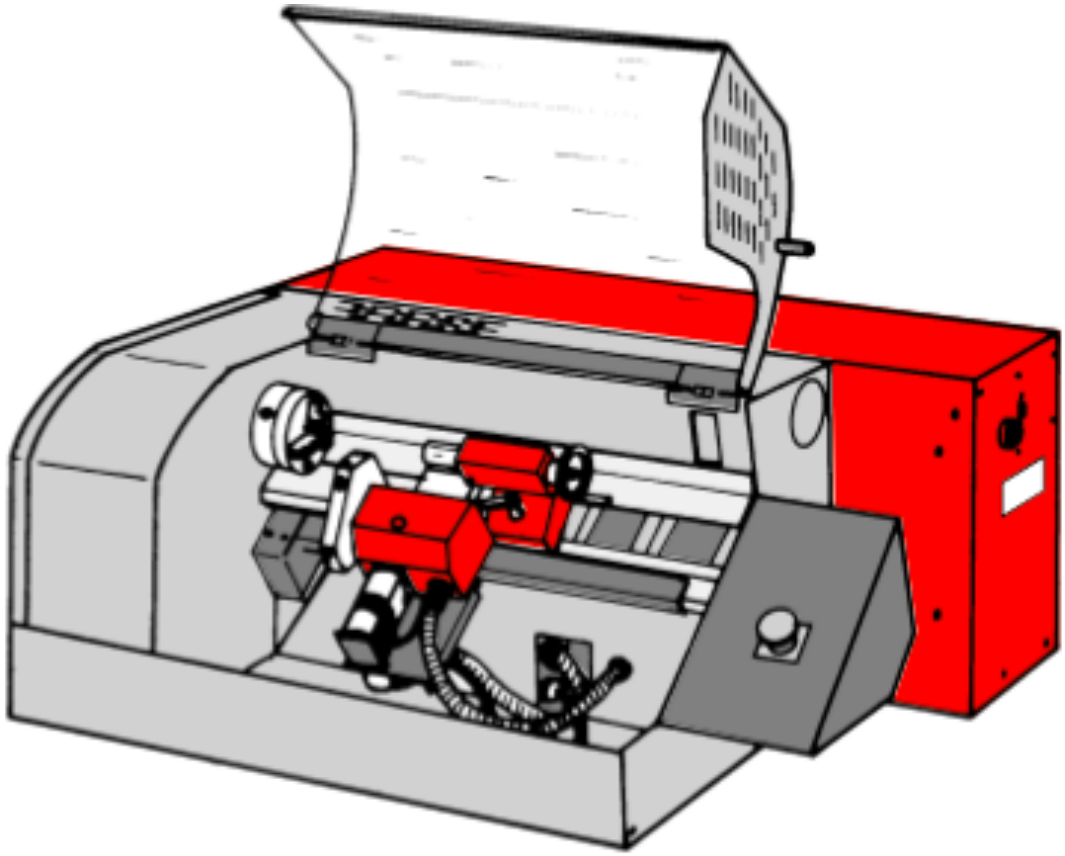
EMCO PC TURN 55 – SINUMERIK 810/820 T – FUSION 360 CAM
USER MANUAL



Autor: Sebastián Hudák
Student number: 679255
Date: 28-09-2023
Version: 1.0

innovative machine tools
industrial training systems

Preface

This manual provides essential guidance on running NC programs and utilizing the Fusion 360 CAM environment for the transfer of NC programs to the EMCO Turn 55 CNC Lathe, which is owned by Inholland University of Applied Sciences in Delft. It is imperative to note that this manual is exclusively applicable to this specific machine and should be employed at your own discretion and risk.

Furthermore, it is assumed that the reader possesses a knowledge of the software for the SINUMERIK 810/820 T EMCO CNC Lathe, as provided by EMCO (a link to which can be found in Appendix A).

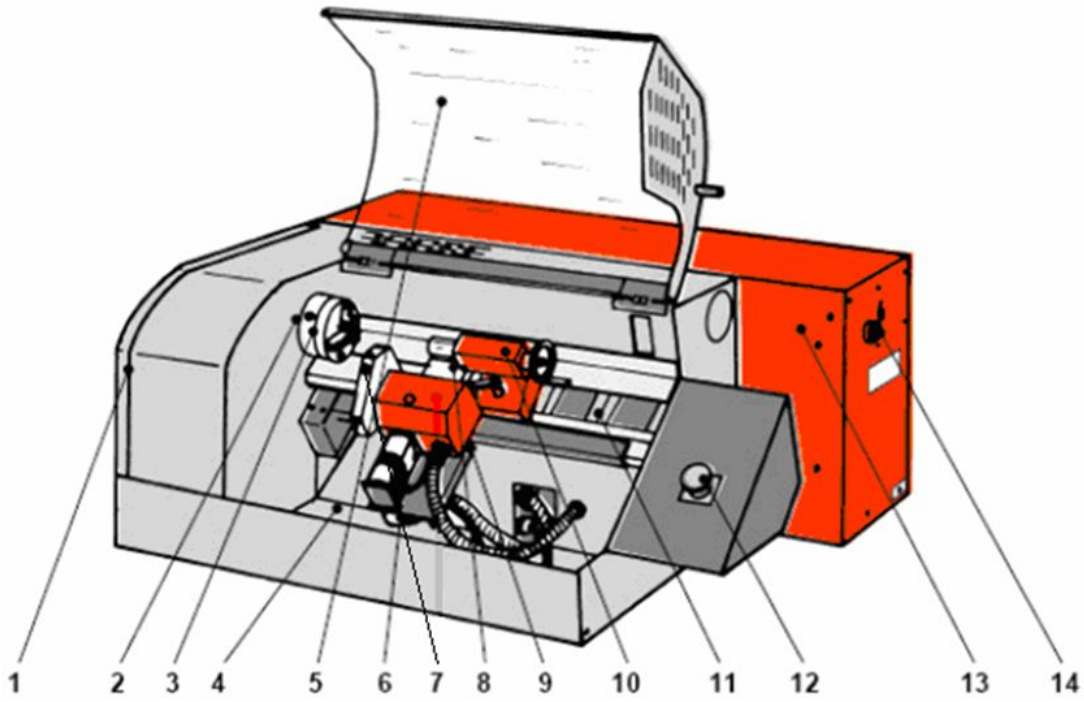
For comprehensive information on using the WINCAM software, please refer to APPENDIX C (Dutch version) within this documentation.

CNC machines are expensive and valuable tools, designed for operation solely by individuals with a substantial level of expertise. Under no circumstances should you attempt to operate this machine without proper supervision if you lack experience with it. Always seek guidance from the staff regarding procedures and safety precautions before use.

It is mandatory to wear Personal Protective Equipment (PPE) when operating the machine!

Table of Contents

Main parts of the Turning (Lathe) machine:



1. Spindle belt transmission case
2. Casing
3. Spindle
4. Tray
5. Doors
6. X-axis stepper motor
7. Tool turret (6 tools)
8. X-axis rail
9. Tool turret mechanism
10. Tailstock
11. Z-axis rail
12. Emergency button
13. Electrical enclosure
14. Master switch

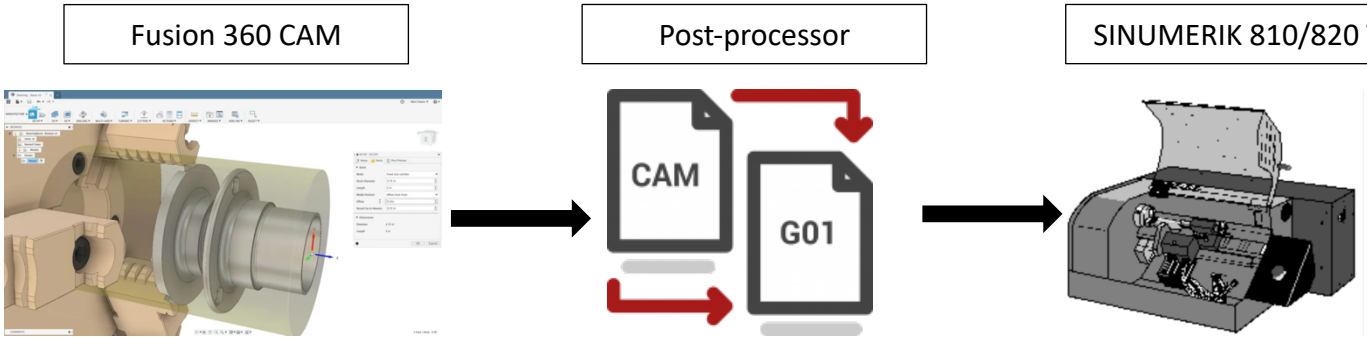
Understanding of problematics

In order to effectively utilize Fusion 360 software for creating NC (Numerical Control) programs, it is important to understand the fundamental principles of NC programming. Fusion 360's CAM (Computer-Aided Manufacturing) functionality is a powerful tool that offers a comprehensive suite of functions for generating the necessary toolpaths required in the production process. Users have the capability to define every facet of the manufacturing process while visualizing a simulation of the actual toolpaths. Furthermore, considering that users typically have pre-designed parts at their disposal, these designs can be seamlessly imported into Fusion 360's CAM environment within a matter of seconds.

The conventional process of creating toolpaths and CAD (Computer-Aided Design) design can be a cumbersome and antiquated procedure when using software supplied with CNC (Computer Numerical Control) machines. For this reason, Fusion 360 serves as an ideal solution, merging functionality with practicality within a user-friendly environment.

However, for the CNC machine's software to comprehend the NC program generated in Fusion 360, it is essential for the program's structure and language to align with the specific requirements of the machine, in our case, the SINUMERIK 810 T. This alignment is achieved through the use of a post-processor.
A post-processor is a form of code that takes input in the form of CAM data and produces output results in the form of G-code, the CNC machine's language.

The machine to which this manual pertains utilizes a customized post-processor, which will be detailed in the subsequent chapter.



Post-processor

To begin, please ensure your familiarity with Fusion 360's CAM design workspace as this manual assumes your proficiency in this area.

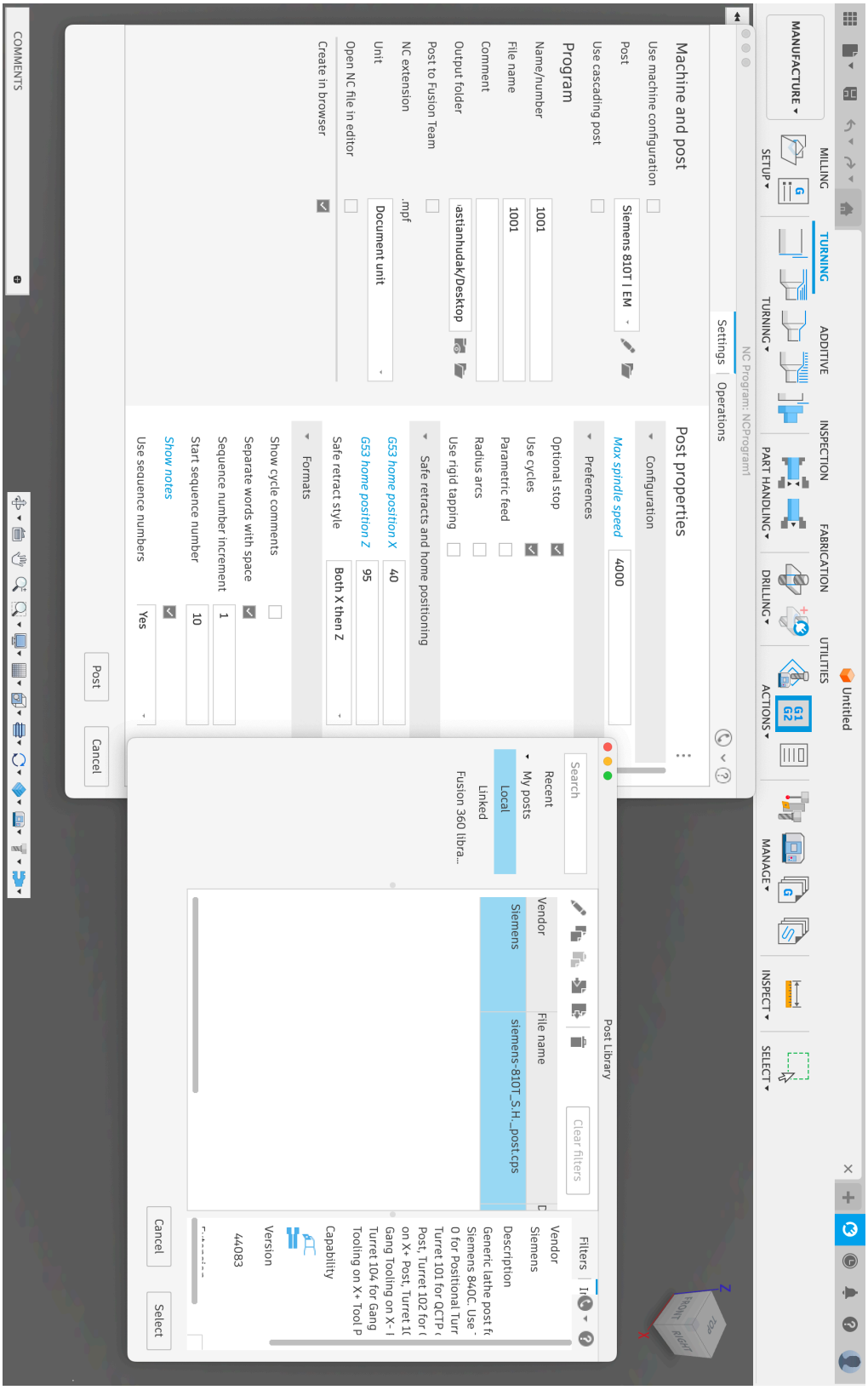
To initiate the process, copy the post-processor file (siemens-810T_S.H._post.cps) located on the EMCO CNC computer's desktop within the 'Get started' folder. Paste this file into a directory on your computer and proceed to load it into your Fusion 360 post-processor library.

Once the provided post-processor has been successfully loaded into Fusion 360, you can commence the creation of your CAM (Computer-Aided Manufacturing) designs. As a recommended practice, it is advisable to perform a brief review of the generated NC (Numerical Control) program prior to its actual deployment on the machine. You can reference the list of G-codes specific to the SINUMERIK 810 T in APPENDIX B.

It is essential to note that the maximum permissible stock diameter is 52mm.

After a successful post, you have to rename your file to %[Number of programs] and save it into a desktop folder named „G-CODES“ on the EMCO CNC computer.

ALWAYS ENSURE THAT THE MAXIMUM SPINDLE SPEED IS SET TO 4000 RPM IN ALL SETTINGS!!!


MANUFACTURE
MILLING
TURNING
ADDITIVE
INSPECTION
FABRICATION
UTILITIES
SETUP
TURNING
PART HANDLING
DRILLING
ACTIONS
MANAGE
INSPECT
SELECT
Untitled
Machine and post
Use machine configuration
Post
Siemens 810T | EM
Use cascading post
Program
Name/number
1001
File name
1001
Comment
Output folder
astianhudak/Desktop
Post to Fusion Team
NC extension
.mpf
Unit
Document unit
Open NC file in editor
Create in browser
Post properties
Configuration
Max spindle speed
4000
Preferences
Optional stop
Use cycles
Parametric feed
Radius arcs
Use rigid tapping
Safe retracts and home positioning
G53 home position X
40
G53 home position Z
95
Safe retract style
Both X then Z
Formats
Show cycle comments
Separate words with space
Sequence number increment
1
Start sequence number
10
Show notes
Use sequence numbers
Yes
Settings
Operations
NC Program: NCProgram1
Post
Cancel
COMMENTS
Post Library
Search
Recent
My posts
Local
Linked
Fusion 360 libra...
Vendor
File name
Siemens
siemens_810T_S.H._post.cps
Clear filters
Filters
Vendor
Siemens
Version
44083
Description
Generic lathe post f
Siemens 840C. Use
0 for Positional Turr
Turret 101 for QCTP
Post, Turret 102 for
on X+ Post, Turret 1
Gang Tooling on X- I
Turret 104 for Gang
Tooling on X+ Tool P
Capability
Cancel
Select

Issues

This machine had previous problems with the spindle controller.

Do not press the manual spindle on the button when in JOG setup!

This will give the error message 6014 (no spindle speed) followed by 6013 (main drive not ready) messages.

Solution: to solve this issue, turn OFF the machine, wait 120 seconds, and turn ON again. This should erase the error messages and you can continue with your work.

APPENDIX A

Links: https://www.emco-world.com/fileadmin/user_upload/_Group/pics_products/Training_Software/software_manuals/Sinumerik_810_820/Sinumerik810820Turn_en.pdf

APPENDIX B

APPENDIX C