Computer Aided Design and Computer Aided Manufacturing (CAD/CAM)

Christopher Saldana, Ph.D.
Edward Nguyen, Derrick Mu
Woodruff School of Mechanical Engineering
Georgia Institute of Technology
Atlanta, Georgia USA
Learning Objectives

• Understand major considerations in CAD/CAM for rapid prototyping
• Understand principles of machine-based processing
• Identify technologies used in computer numerical control
• Identify toolpath types and tooling to be used for particular part features
CAM with Laser Cutting

- Invention Studio equipment
  - Trotec 400 (Goji) – MS 39”x24”x0.5”
  - Trotec 300 (Elderberry, Fig, Honeydew) – MS 29”x17”x0.5”

- Process
  - Machine setup, Inkscape, JobControl, Machine cleanup

- Considerations
  - Kerf (0.005-0.010”), max size (MS), material limits

- Considerations / use cases
  - Manufacturing boxes
  - Manufacturing gears
  - Part nesting approaches
Laser Cutting: Boxes

Makercase

- Box shape (basic/polygon/bent)
- Box sizes (width/height/depth)
- Material thickness
- Box type (open/closed)
- Edge joints (flat/finger/t-slot)
- Finger size / # bends

Links

https://www.festi.info/boxes.py/index.html
https://makeabox.io/
http://jeromeleary.com/laser/

https://www.makercase.com/
Laser Cutting: Gears

Gear Generator

- Involute spur gear generator
- DXF / SVG format output
- Multiple gear systems
- Input/output ratio and rotation speeds
- Internal/external, # teeth, pitch dia., diametral pitch, pressure angle
- Rack and pinon in other generators

Links

https://geargenerator.com/
http://hessmer.org/gears/InvoluteSpurGearBuilder.html
http://www.jeromeleary.com/gears/

Using gear generator
Working in inkscape to prepare jobs
Laser Cutting: Part Nesting

What is part nesting?
• Rearrange individual components on raw material layout so to limit bounding box and to share cut lines between them
• Save time on fabrication
• Save material (e.g., trim loss)
• Manual nesting vs optimized nesting

Links
https://deepnest.io/ (Mac, Windows, Linux)
https://svgnest.com/ (web-based)
Manual Machining

Milling Machine Characteristics

- **Primary use:** making precision prismatic parts
- **Fixture:** part is clamped in vise
- **Primary motion:** tool
- **DOF:** 3-axis motions (X, Y, Z)
- **Table axes:** X (left/right), Y (front/back)
  - Control: handwheels
  - Position: digital read out
- **Head axis:** Z (vertical)
  - Control: hand crank (coarse), handwheel (fine)
  - Position: digital read out
- **Caution:** tool/vise crashes, report any issues
- **Training:** optional training sessions upcoming

Milling machine (geared, 1 hp, 65-1550 RPM)
https://www.mscdirect.com/product/details/64162779
CNC-Based Machining

IDEA Laboratory CNC Mill

3-axis CNC mill (Siemens 828D controller)  
https://emco.co.uk/emcomill-e350/

G-code file

Training: optional training sessions upcoming

Autodesk Fusion 360 (free!)
3 Axis Machining Capabilities
Computer Numerical Control - Machining

CNC Overview
• What is CNC
• Brief History
• Advantages/Disadvantages
• Preparing for CAM

CNC Fundamentals
• Toolpaths
• Tool Types
• Cutting Direction & Speeds/Feeds
• Coordinate Systems
CNC Milling

Computer Numerical Control
- Computer Motion Control

Subtractive Process
- More complex than 3D Printing
- Still dominates industry (~90%)
CNC Overview – Brief History

Then

Now
Advantages and Disadvantages of CNC

Advantages
- Improved Automation
- Consistent/Accurate
- Speed
- Mass Production
- Flexibility

Disadvantages
- Price
- Programming Skillset
- Space, Electricity, Maintenance
- Time intensive for simple projects
Preparing for CAM

CAD

CAM

- Workpiece Setup
- Tools & Toolpaths
- Simulate

Post Process

Transmit to Machine

- Dry Run
- Cut Part
CNC Fundamentals – Toolpaths

Toolpath Types

2D
2.5D
3D
4-Axis
3+2 Axis
5-Axis
Machine Coordinate System: G53

Right hand rule convention
2D Toolpaths

Laser Cutter

Waterjet

2.5D Toolpaths

• Multiple Depths
• Certification Part

3D Toolpaths

• Curved and contoured surfaces
4-axis, 3+2, and 5-axis Toolpaths

Involves Rotational Axes

4-Axis (A axis usually)

3+2 Axis

5-Axis
Toolpaths: 2.5D Terminology

- Clearance Height
- Rapid Height
- Feed Height
- Top of Stock
- Depth
- Stepdown Depth
- XY Stock Allowance
- Z Stock Allowance
- Stepover
- Toolpath Centerline
- Cut Direction
- Clearance Height
Toolpaths: Facing

Usually the first operation

Generates clean, smooth reference top surface
Toolpaths: Contouring

Sometimes known as profiling

Follows outlines of shapes

• Creates outside walls

Special Cases

• Chamfering
• Radius Milling
Toolpaths: Pockets

Creates interior cavities

Removes material in shapes

Special Considerations

• Ramping entry

Special Cases

• Boring
• Slotting
Toolpaths: Holemaking

Spot & Center Drills

• Short & Rigid
• Used to Start Holes

Drilling

Reaming

Special Considerations

• Peck Drilling
• Tapping
Toolpaths: Adaptive Toolpaths

Relatively new concept

Seeks constant tool engagement

Better material removal rates & longer tool life
Toolpaths: REST (REmaining STock)

Sometimes known as re-machining

Uses a smaller diameter tool

Covers areas not reached by larger tool
Toolpaths: Roughing vs Finishing

Applies to all toolpath types

Rough to remove as much material as quickly as possible.
Finish to create smooth surfaces and final dimensions.
3D Toolpaths: Roughing vs Finishing
3D Toolpaths: Stepover effects

Narrow Stepover  Wide Stepover
CNC Fundamentals – Milling Tools
Tool Materials and Tool Holders

Materials
HSS
Carbide
Coatings

Holders
Collet Holders
Weldon Holders (1)
Drill Chucks (2)
Arbors (3)

End Mill
Flange
Pull Stud
Taper
Collet & Collet Nut
Milling Tools: End Mill Types

Dictates whether or not you can plunge

- Non-center cutting requires ramping
Milling Tools: Face (Shell) Mills
Milling Tools: Hole Making

Spot Drill

Countersink

Center Drill (Combination)

Twist Drill

Tip Angle
Hole Making: Spot, then Drill
Hole Making: Tapping (Form Vs Cut)

Cutting material vs deforming it

Form/Roll Tap

Cutting Tap
CNC Fundamentals – Machining Concepts

Rotation Direction
Chip Formation
Chip Load
Milling Direction
Conventional vs Climb Milling

- **Climb Milling**
  - Cut Direction
  - Spindle Rotation
  - Material to be removed by the next tooth (chip)

- **Conventional Milling**
  - Cut Direction
  - Spindle Rotation
  - Feed per tooth

**CONVENTIONAL CUTTING**

**CLIMB CUTTING**

39
Speeds and Feeds Formulas

\[
\text{Speed} \left( \frac{\text{rev}}{\text{min}} \right) = \frac{\text{SFM} \left( \frac{\text{ft}}{\text{min}} \right) \times 3.82}{\Phi(\text{in})}
\]

\[
\text{Feed} \left( \frac{\text{in}}{\text{min}} \right) = \text{Speed} \left( \frac{\text{rev}}{\text{min}} \right) \times CL \left( \frac{\text{in}}{\text{rev}} \right) \times \#\text{Flutes}
\]
Toolpath Terminology

- **Lead-In**
- **Retract**
- **Rapid**
- **Rapid Feed**
- **Cutting (feed)**
- **Depth of cut**
- **Spindle Speed (RPM)**

**Spindle Speed (RPM)**

- **Ramp**
Toolpath Terminology
The EMCO E350
Basic Machine Anatomy
Machine Coordinate System: G53
Work Coordinate System: G54
Setting Work Offsets
Tool Length Offsets

1. Tool Tip at Machine Home
   - Part Datum (Easiest)

2. 1-2-3 Block
   - 1-2-3 Block (Better)

3. Tool Probe
   - Tool Probe (Best)
Setting Tool Length Offsets
## CNC Programming Language

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
<th>Code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>Rapid Positioning</td>
<td>M01</td>
<td>Optional Stop</td>
</tr>
<tr>
<td>G01</td>
<td>Linear Interpolation</td>
<td>M03</td>
<td>Spindle on (clockwise)</td>
</tr>
<tr>
<td>G02</td>
<td>Circular Interpolation (clockwise)</td>
<td>M04</td>
<td>Spindle on (counterclockwise)</td>
</tr>
<tr>
<td>G03</td>
<td>Circular Interpolation (counterclockwise)</td>
<td>M05</td>
<td>Spindle off</td>
</tr>
<tr>
<td>G04</td>
<td>Dwell (pause)</td>
<td>M06</td>
<td>Tool change</td>
</tr>
<tr>
<td>G09</td>
<td>Exact Stop Check</td>
<td>M07</td>
<td>Coolant on (mist, air)</td>
</tr>
<tr>
<td>G10</td>
<td>Programmable Data Input</td>
<td>M08</td>
<td>Coolant on (flood)</td>
</tr>
<tr>
<td>G20</td>
<td>English Units</td>
<td>M09</td>
<td>Coolant off</td>
</tr>
<tr>
<td>G21</td>
<td>Metric Units</td>
<td>M30</td>
<td>End of Program/Reset</td>
</tr>
<tr>
<td>G28</td>
<td>Machine Zero Return (Machine Home)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G40</td>
<td>Cutter radius comp: cancel</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G41</td>
<td>Cutter radius comp: left</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G42</td>
<td>Cutter radius comp: right</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G53</td>
<td>Machine Coordinate System</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G54</td>
<td>1st workshift offset</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G90</td>
<td>Absolute Coordinates</td>
<td></td>
<td></td>
</tr>
<tr>
<td>G91</td>
<td>Incremental Coordinates</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### G00 and G01 Examples

```
N130 G01 Z-.374 F4.2 S700
N140 X3.6 Y8.5
N150 Y3.1
N160 X2.4
N170 Y1.9
N180 X4.125
N190 Y3.625
N200 G00 Z1.0
```
Part Programs

Machine Instructions

- Workpiece
- Spindle
- Miscellaneous

Dry Run

Simulate

N50 G00 X3.0 Y2.5 Z-1.0 T2 M06
N60 G01 X12.0 Y2.0 F4.2 S700 M03

G – Go
F – Feed
S – Speed
T – Tool
M – Miscellaneous
N – Name
X – X Coordinate
Y – Y Coordinate
Z – Z Coordinate
CAD/CAM Setup and Programming

Install Fusion 360
Create Profile/Account
Upload Part
Open Manufacturing Workspace
Create Setups
Manage Tool Library
Create Toolpaths
Simulate
Post Process
Edits (SUPA)
CNC Setup and Operation

Pre-Start Activities
Start Up & Warm Up
Load Tools
Set Tool Length Offsets
Set Workpiece Offset XY
Set Workpiece Offset Z
Load CNC Program
Simulate
Run
Adjust & Rerun
Shutdown & Cleanup
Learning Objectives

• Understand major considerations in CAD/CAM for rapid prototyping
• Understand principles of machine-based processing
• Identify technologies used in computer numerical control
• Identify toolpath types and tooling to be used for particular part features