Fusion 360 CAM Handout

Step-by-step: Facing ? Profile ? Spot Drill (Fusion 360)

Preparations (before creating operations)

1. Open your design in **Fusion 360** and switch to **Manufacture**.
2. Confirm units (mm or in) — set to what your CNC expects.
3. Save the file (version your CAM changes if needed).
4. Quick checklist: material selected, correct model orientation, correct stock size, tooling on hand.

Part of learning to be a machinist is understanding how to use a given tool to perform a given job on a given material. This is a lot of variables: we want to be careful with out setup to make sure that making our part is successful.

Make sure you understand the tool, and how the tool will help or hinder making your part:

- Limitations (speeds and feeds, torque, strength)
- Tolerances (precision, runout, flex)

Understand your material: what are the general properties of the material, how the material behaves at different speeds and feeds, how the material reacts to various tooling.

Knowing the characteristics of your tools and materials will help you design the job that makes your part.

1. Create a Setup

1. **Manufacture → Setup → New Setup**.

2. **Operation Type:** Milling.
3. **Model:** Confirm the body/component you will machine is selected.
4. **Work Coordinate System (WCS / Origin):**
   - Choose a logical origin — commonly top-front-left corner, or bottom (z zero on top of parallels).
5. **Stock:**
   - Mode: **From solid** or **Fixed size box** (e.g., measure the size of the material stock the part is going to be milled from).
   - Confirm orientation matches how the part will sit on the machine.
6. Click **OK**.

---

2. Facing Operation (2D Adaptive / 2D Face)

> Goal: remove top material to create a clean, flat face and establish final Z height.

A. Choose Operation

1. **2D → 2D Face**

B. Tool Selection

1. Click **Tool** → **Select**. Pick a face cutter or flat endmill (example: 1/4" flat endmill).
2. Example starting values (adjust to your machine/tool/material):
   - **Spindle speed:** 8,000–10,000 RPM for aluminum with carbide; lower for steel. (These numbers vary depending on the quality of end mills that are being used)
   - **Feedrate:** ~30–60 ipm for 1/4" endmill

> For the class, use a spindle speed of 8,000–10,000 RPM.

C. Heights & Passes

1. **Heights tab:** Set **Top Height** to stock top (usually Stock top). Set **Bottom Height** to final face Z (usually model top).
2. **Passes:** Step-over ~20–50% of tool diameter for finishing; step-down small if face is shallow. If heavy stock, use multiple passes or Adaptive clearing first.

E. Generate & Simulate

1. Click **Generate**.
2. **Actions → Simulate**. Watch for gouging or collisions. Use **Stock** visualization to confirm material removal.
3. If safe, proceed. If not, adjust heights/clearance or origin.

---

3. Profile Cut (2D Contour)

> Goal: cut the outside profile (or an interior profile) to final shape or rough to near-net.

A. Choose Operation

1. **2D → 2D Contour**.

B. Select Geometry

1. Select the face or sketch edge(s) that define the contour to cut.
2. Pay attention whether you want **Outside**, **Inside**, or **On path.**
   This choice affects where the tool will cut in relation to the tool path.
3. Choose **Multiple passes** for deep cuts.

> For the class, select MUMBLE and MUMBLE.

C. Tool Selection

1. Select an appropriate tool: for finishing profile choose the same flat endmill or a smaller endmill (e.g., 1/8") for tighter radii on inside corners.
2. Example params (finish profile in aluminum): spindle 8,000-10,000 RPM; feed ~20-40ipm depending on tool size.

D. Heights & Passes

1. **Heights:**
   - **Top Height:** Stock Top (or model top if already faced).
   - **Bottom Height:** Set to final cut depth (eg: part bottom). Use caution to not contact the vise jaws.
2. **Passes:**
   - **Multiple depth passes:** set **Maximum roughing stepdown** appropriate to tool (e.g., 1–2 × tool diameter for carbide roughing, or smaller for finishing).
   - For finishing, use single pass at final depth or small stepdown (.010") for surface quality.

3. **Stock to Leave:** If you plan a separate finishing pass, set small radial/axial stock to leave (e.g., .010").

F. Quick Retracts & Clearance

1. Ensure **Retract Height** clears clamps and features.
2. Use **Ramp In** or **Helical Lead-in** if cutting into a closed profile.

G. Generate & Simulate

1. Generate toolpath and simulate.
2. Use **Toolpath Color** and **Stock** display to check full travel and final geometry. Confirm no collisions.

---

4. Spot Drill (Drilling Operation or CAM Drill Cycle)

> Goal: create a small conical center or guide hole before using a twist drill.

A. Choose Operation

1. **Drilling**

B. Select Geometry

1. Select the points where you want to spot (hole centers). Use sketch points, hole features, or pick model faces.
2. Verify XY coordinates match intended locations.

C. Tool Selection

1. Pick a spot-drill (or countersink) with the desired included angle.
2. Typical parameters: slow plunge feed, shallow depth (.050"), spindle speed for material and drill diameter.

D. Heights & Pecking

1. **Heights:**
   - **Top Height:** Stock top (or model top).
   - **Bottom Height:** Small negative value to create a conical dimple (e.g., −.020 to .050").

2. **Pecking:** Not needed for spot drill — single plunge sufficient.

E. Feed & SFM

1. Adjust feed to match tool size and material (small tools = lower feed).
2. Use conservative plunge feed to avoid tool breakage.

F. Generate & Simulate

1. Generate. Simulate full sequence including facing and profiling to ensure the spot-drill toolpath aligns after previous operations.
2. Verify that the spot drill does not collide with clamps and that Z retracts clear.

---

5. Run Order & Stock Updates

1. Confirm operation order in the **Setup** tree: Facing should be first, then Profile, then Spot Drill (or Spot Drill before profile if you need holes established before cutting the outer profile). Drag operations to reorder if needed.

---

6. Final Simulation & Verification

1. Run a **Full Simulation** of all operations together.
2. Use **Play** to confirm final shape and check for collisions.
3. Verify tool changes and tool holder clearances.

---

7. Post Process & Export G-Code

1. **Actions → Post Process** in the Setup or individual operation (If using the free version of Fusion, you will need to output one file per tool type)
2. Select the correct **post-processor** for the controller (Mach3mill or CNCRouterPartsMach3Mill).
3. Set file name.
4. Click **Post** and transfer G-code to the machine per shop procedures.

Tips & Troubleshooting

- If facing leaves scallops: increase step-over or use a larger face cutter, or add a finishing pass with lighter cut.
- If profile chatter occurs: reduce feed, increase spindle speed, or use a shorter tool stick-out.
- Use **Stock to Leave** on the profile if you want a final finish pass to get accurate dims.

Revision #5
Created 28 October 2025 15:57:39 by J N
Updated 16 November 2025 16:46:45 by Bookstack Admin