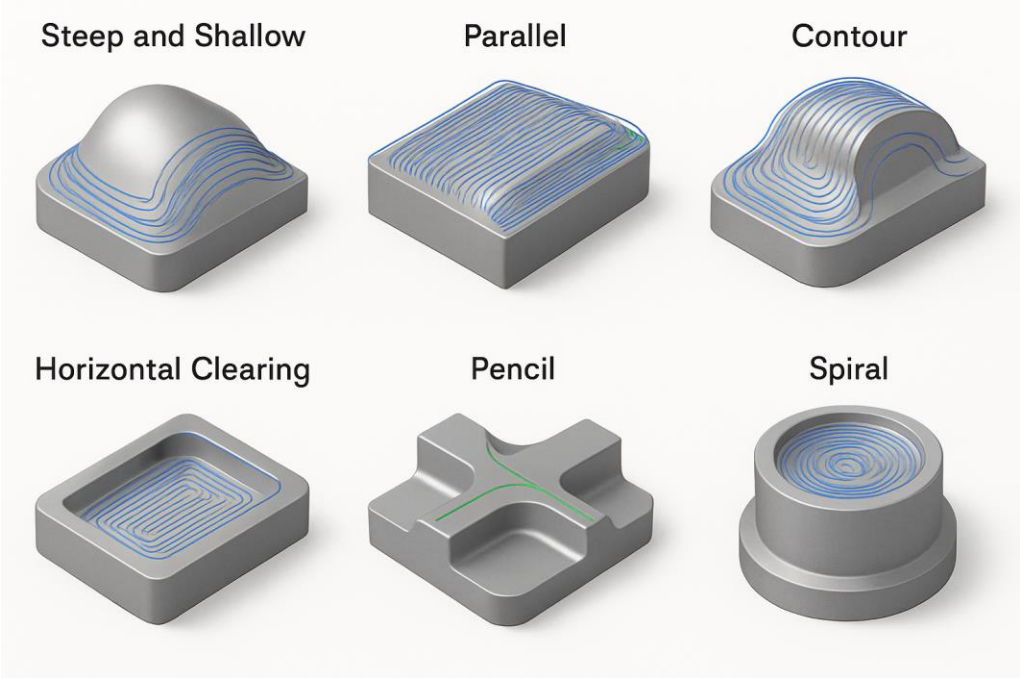
Understanding Toolpaths: Contour, Pocketing, Drilling, and Profiling Explained

Toolpaths are digital instructions that guide how machine tools move. In CAM systems, picking the right toolpath affects how long machining takes, the finish on the part, and how quickly tools wear out. Toolpaths are mostly used for cutting or removing material, but they also support trimming and finishing work after forming or bending.

Toolpaths guide the process to improve accuracy, reduce mistakes, and keep machines working without interruptions.

What Are Toolpaths in CAM?

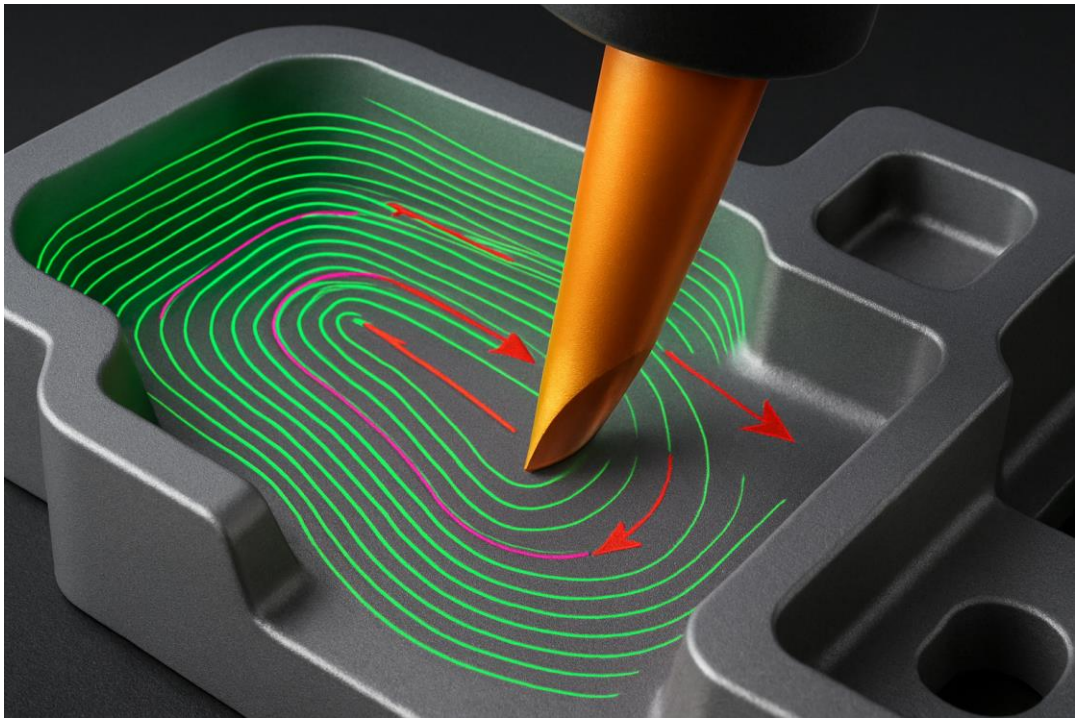


In CAM software, toolpaths control the machine's movements. They're programmed routes based on a 2D or 3D model. CAM software processes the model and generates G-code, which tells the machine what path to follow. The CAM system uses a post-processor to translate toolpaths into G-code, which is customized to match the specific CNC machine's controller.

Toolpaths help shape, cut, or finish parts according to the design. While they're used mainly for subtractive tasks like milling and drilling, they also help with trimming parts that

have already been bent or formed. These trimming operations are typically carried out in secondary setups using CNC routers or milling machines, especially for complex or curved sections.

Contour Toolpaths (2D/3D Perimeter Cutting)



Contour toolpaths follow the shape of the part. These paths cut along the outside or inside of a profile and are used in a wide range of applications.

Common uses:

- Trimming bent or flat aluminum parts
- Cutting outlines from metal plates
- Finishing edges after the forming process

What to consider in CAM:

- Add entry and exit moves to protect the finished surface
- Set the correct tool offset for accurate paths
    - Some machines also use dynamic compensation systems that adjust tool paths based on wear, tool diameter, or sensor feedback.
- Slow down the tool at corners to avoid chatter or overcutting

These toolpaths work on both external and internal features, cutting along any defined curve.

Pocketing Toolpaths (Cavity Machining)

Pocketing toolpaths remove material inside a set boundary. Instead of cutting all the way through, the tool clears out a shallow or deep recess.

Common uses:

- Creating cavities for brackets or fasteners
- Shaping jig bases or forming fixtures
- Opening up sections for welded parts

What to consider in CAM:

- Choose a proper step-over and depth based on the tool size
- Use adaptive clearing if you need smoother tool motion
    - This method works best with high-efficiency tools and machines that can handle constant engagement angles and rapid movements.
- Make sure chips evacuate easily, especially in soft materials

This toolpath isn't used during bending. It helps when preparing parts or setting up workpieces that will go through the forming process later.

Drilling Toolpaths (Axial Operations)

Drilling toolpaths are designed to plunge a tool vertically into the part. The tool enters and exits the material at a controlled speed and depth.

Common uses:

- Adding mounting holes in machine parts
- Drilling pin locations for alignment
- Creating paths for lubrication or cooling

What to consider in CAM:

- Break deep cuts into smaller steps to protect the drill
- Adjust your speed and feed rate for the material
- Organize the drilling order to reduce tool travel

These toolpaths apply after forming when holes need to line up accurately with other parts or systems.

Profiling Toolpaths (Advanced Edge Machining)

Profiling toolpaths follow curved or angled shapes and are often used when cutting along a 3D edge.

Common uses:

- Trimming formed aluminum parts
- Cleaning up edges after bending
- Shaping non-uniform profiles

What to consider in CAM:

- Make sure your machine has the right axis setup for the shape
- Use tool compensation when the part may shift slightly
- Set tolerances to match the final fit or appearance

Profiling is used both before and after bending if you're either prepping extrusion ends or cleaning up the edges of a finished part.

Why Toolpath Strategy Matters

Choosing the right toolpath affects how well the machine runs and how the part turns out.

Toolpaths influence:

- How fast a part can be made
- How close the cut stays to the correct size
- How long the tool will last before wearing out
- How smoothly the machine runs with less vibration
- What adjustments are needed for each material type

Some toolpaths are better for aluminum because it cuts faster with lighter passes, while others work better on harder metals. Aluminum tends to favor high-speed, shallow passes, while harder materials may require slower, deeper cuts with reduced step-over. The correct strategy depends on the part, the material, and the machine.

Conclusion

Each toolpath type is built for a different task. Contour paths shape the edges. Pocketing clears out space. Drilling adds holes. Profiling helps finish bent or shaped parts. When you choose the right toolpath, you reduce cutting time, avoid unnecessary wear, and get more consistent results.

Cutting, shaping, and trimming each rely on paths that match the material and the final shape of the part. These paths guide how the tool moves and what result it produces. Even when the forming process is already finished, toolpaths still play a part in cleaning up and finalizing each component.

At Inductaflex, we support every part of the bending and finishing process—right down to the last cut.