How to design parts for CNC machining

Written by Alkaios Bournias Varotsis

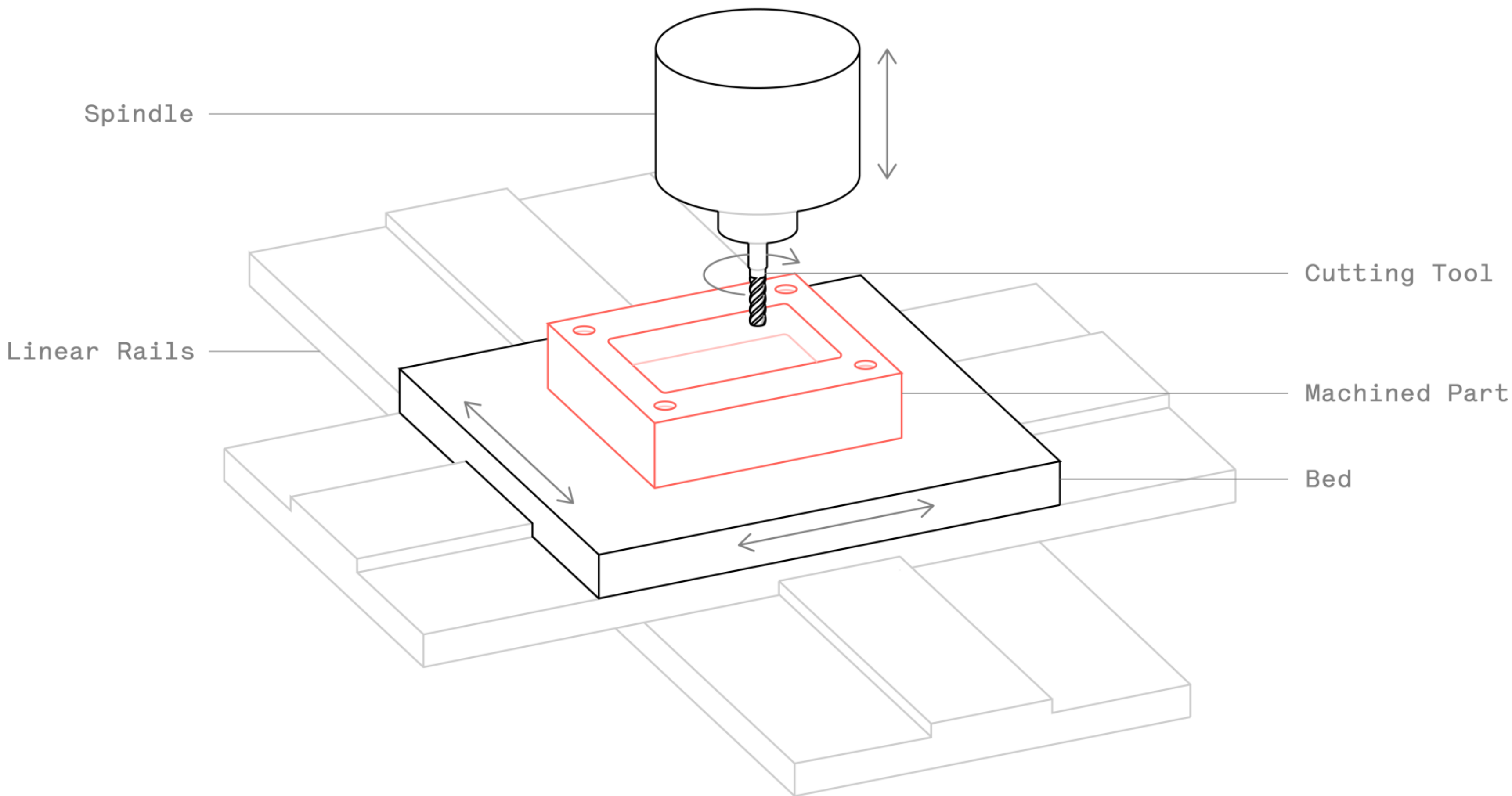
Introduction

To take advantage of the capabilities of CNC machining to the fullest, a designer must follow certain *Design for Manufacturing* rules. This can be challenging though, as specific industry-wide standards do not exist.

In this article, we compiled comprehensive guidelines with the best design practices for CNC machining. We asked for feedback from industry experts and CNC machining service providers to compile an extensive list of up-to-date information.

We focused on describing what is feasible with modern CNC systems, neglecting the associated cost. For a guide on designing cost-effective parts for CNC, please refer to this article.

The CNC machining process



Schematic of a CNC machine

CNC machining is a subtractive manufacturing technology. In CNC, material is removed from a solid block using a variety of cutting tools that rotate at high speed (thousands of RPM) to produce a part based on a CAD model. Both metals and plastics can be CNC machined.

CNC machined parts have **high dimensional accuracy** and **tight tolerances**. CNC is suitable for both **high volume production** and **one-off jobs**. In fact, CNC machining is currently the most cost-effective way of producing metal prototypes, even compared to 3D printing.

An introduction to the basic principle of CNC machining can be found here.

Video of a CNC machine, milling the outline of a cavity. SourceFinishing pass, removing minimal material, to ensure high dimensional accuracy and good surface finish. Source

Main CNC design restrictions

CNC offers great design flexibility, but certain design restrictions exist. These restrictions are related to the basic mechanics of the cutting process and are related mainly to tool geometry and tool access.

Tool geometry

Most common CNC cutting tools (end mill tools and drills) have a **cylindrical shape** and a **limited cutting length**.

As the material is removed from the workpiece, the geometry of the tool is transferred to the machined part. This means, for example, that the internal corners of a CNC part will always have a radius, no matter how small a cutting tool was used.

Tool access

To remove material, the cutting tool approaches the workpiece directly from above. Features that cannot be accessed this way, cannot be CNC machined.

There is an exception to this rule: **undercuts**. We will see how you can use undercuts in your designs in latter section.

A good design practice is to align all features of your model (holes, cavities, vertical walls etc) to one of the 6 main principal directions. See this rule as a recommendation and not a restriction though, as 5-axis CNC systems offer advanced workpiece holding capabilities.

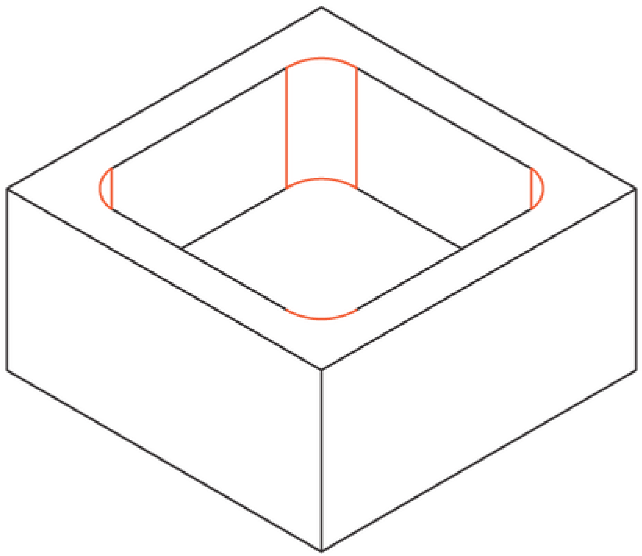
Tool access is also an issue when machining features with large depth to width ratio. To reach the bottom of a deep cavity, for example, special tools with long shafts are required. This lowers the stiffness of the end effector, increasing the vibrations and lowering the achievable accuracy.

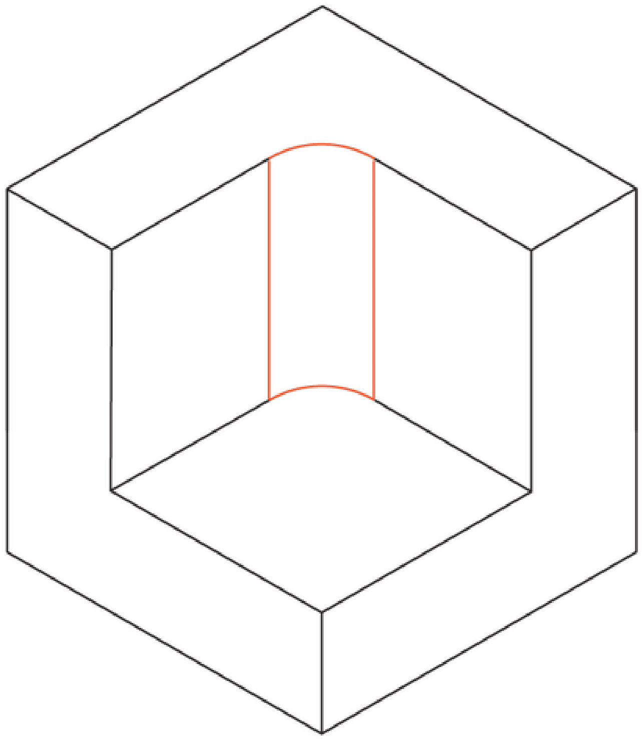
CNC experts recommended to design parts that can be machined using a tool with the **largest possible diameter** and the **shortest possible length**.

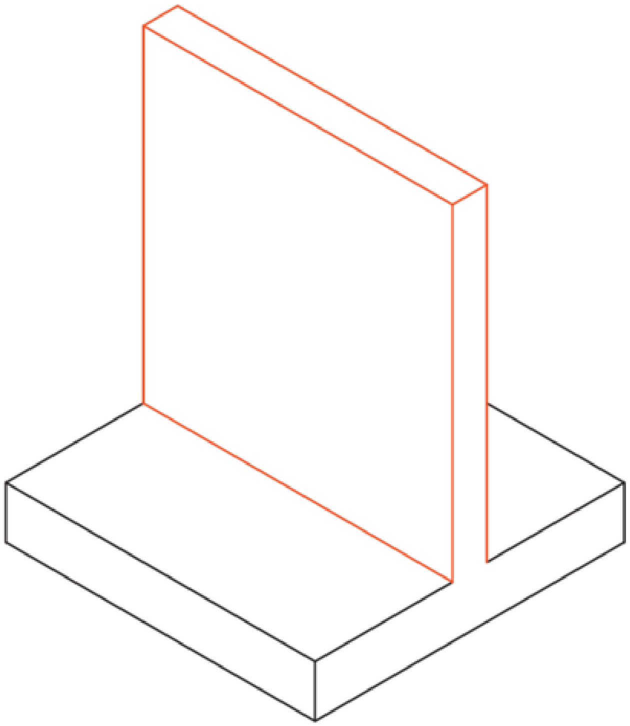
CNC design rules

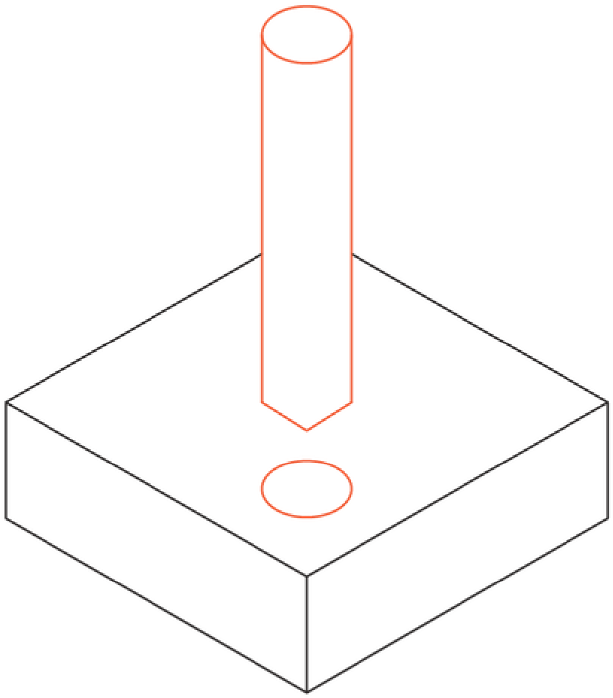
A challenge that frequently comes up while designing a part for CNC machining is that no specific industry-wide standards exist: the CNC machine and tool manufacturers continuously improve the capabilities of the technology, expanding the limits of what is possible.

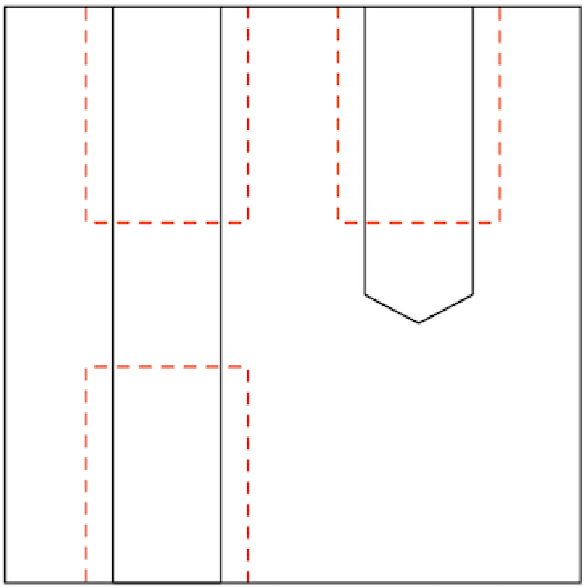
In the table below, we summarized the recommended and feasible values for the most common features encountered in CNC machined parts.

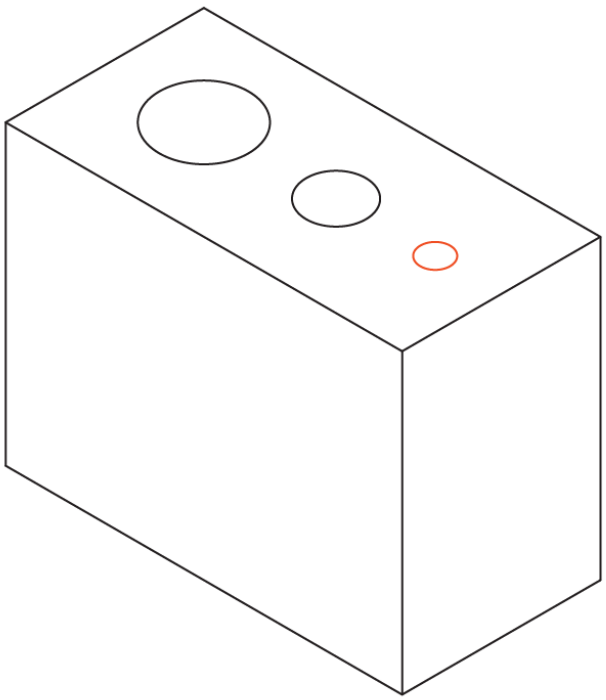
Feature	Description
	**Cavities & Pockets** **<u>Cavity depth:</u>** **Recommended:** 4 x cavity width End mill tools have a limited cutting length (typically 3-4 times their diameter). Tool deflection, chip evacuation, and vibrations are becoming more prominent when cavities with smaller depth to width ratio. Limiting the depth of the cavity to 4 times its width ensures good results. If larger depths are required, consider designing parts with a variable cavity depth (see the image above for an example). **Deep cavity milling:** cavities with depth greater than 6x the tool diameter are considered deep. Ratios of tool diameter to cavity depth of up to 30:1 are possible using specialized tooling (maximum depth: 30 cm with a 1" diameter end mill tool).

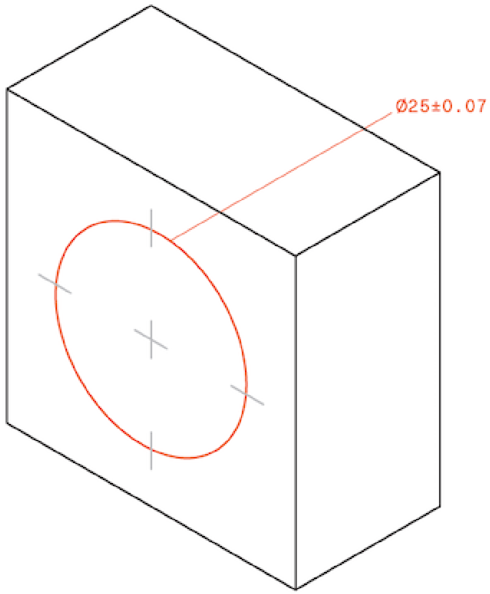
Feature	Description
	**Internal Edges** **<u>Vertical Corner Radius:</u>** **Recommended:** ⅓ x cavity depth (or larger) Using the recommended value for internal corner radii ensures that a suitable diameter tool can be used and aligns with the guidelines for the recommended cavity depth. Increasing the corner radii slightly above the recommended value (for example by 1 mm), allows the tool to cut following a circular path instead of a 90° angle. This is preferred as it results in a higher quality surface finish. If sharp 90 degree internal corners are required, consider adding a T-bone undercut instead of reducing the corner radius. **<u>Floor Radius:</u>** **Recommended:** 0.5 mm, 1 mm or no radius **Feasible:** any radius End mill tools have a flat or slightly rounded lower cutting edge. Other floor radii can be machined using ball end tools. It is a good design practice to use the recommended values, as it is preferred by the machinists.

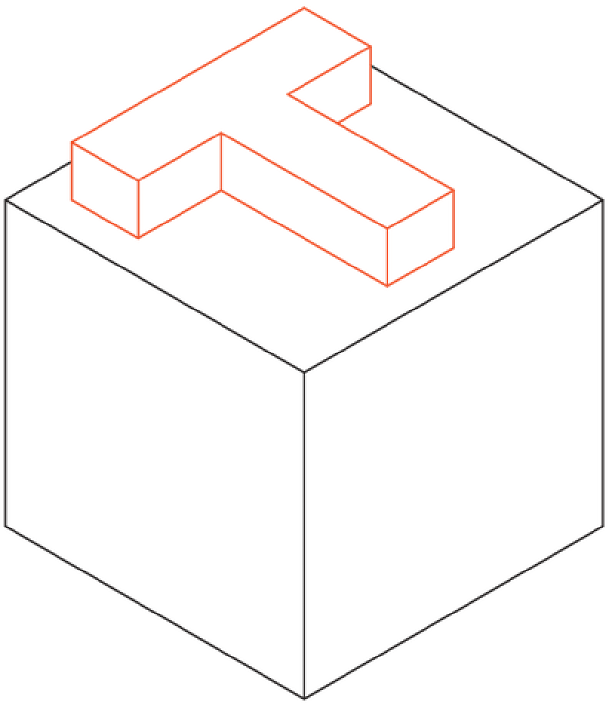
Feature	Description
	**Thin walls** **<u>Minimum wall thickness:</u>** **Recommended:** 0.8 mm (metals), 1.5 mm (plastics) **Feasible:** 0.5 mm (metals), 1.0 mm (plastics) Decreasing the wall thickness reduces the stiffness of the material, which increases vibrations during machining and lowers the achievable accuracy. Plastics are prone to warping (due to residual stresses) and softening (due to temperature increase), so a larger minimum wall thickness is recommended. The feasible values stated above must be examined in a case-by-case basis.

Feature	Description
	**Holes** **<u>Diameter:</u>** **Recommended:** standard drill bit sizes **Feasible:** any diameter larger than 1 mm Holes are machined using either a drill bit or an end mill tool. The size of the drill bits is standardized (in metric and imperial units). Reamers and boring tools are used to finish holes that require tight tolerances. For high accuracy holes smaller than ⌀20 mm, using a standard diameter is recommended. **<u>Maximum Depth:</u>** **Recommended:** 4 x nominal diameter **Typical:** 10 x nominal diameter **Feasible:** 40 x nominal diameter Holes with non-standard diameter must be machined with an end mill tool. In this case, the maximum cavity depth restrictions apply and the recommended maximum depth value should be used. Holes deeper than the typical value are machined using specialized drill bits (minimum diameter 3 mm). Blind holes machined with a drill have a conical floor (135° angle), while holes machined with an end mill tool will be flat. There is no particular preference between through holes or blind holes in CNC machining.

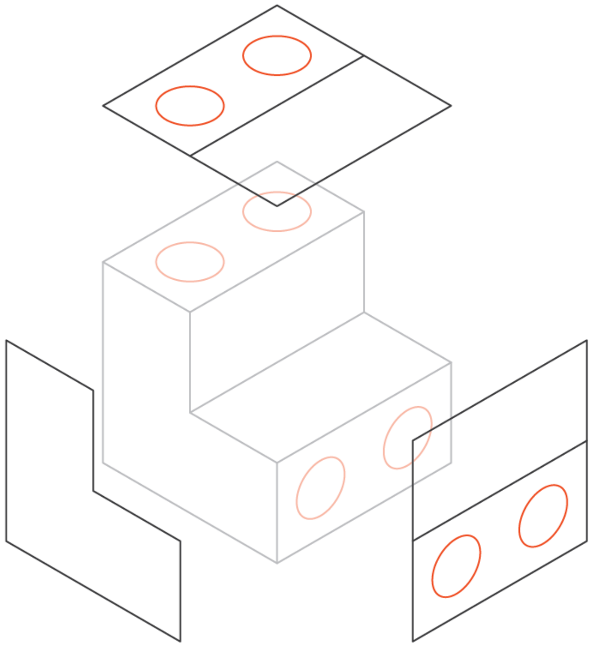
Feature	Description
	**Threads** **Thread size:** **Minimum:** M2 **Recommended:** M6 or larger Internal threads are cut with taps and external threads with dies. Taps and dies can be used to cut threads down to M2. CNC threading tools are common and they are preferred by machinists, as they limit the risk of tap breakage. CNC threading tools can be used to cut threads down to M6. **Thread length:** **Minimum:** 1.5 x nominal diameter **Recommended:** 3 x nominal diameter The majority of the load applied to a thread is taken by the few first teeth (up to 1.5 x the nominal diameter). Threads longer than 3 x nominal diameter are thus unnecessary. For threads in blind holes cut with taps (i.e. all threads smaller than M6), add an unthreaded length equal to 1.5 x the nominal diameter at the bottom of the hole. When a CNC threading tool can be used (i.e. threads larger than M6), the hole can be threaded throughout its length.

Feature	Description
	**Small Features** **<u>Minimum hole diameter:</u>** **Recommended:** 2.5 mm (0.1") **Feasible:** 0.05 mm (0.005") Most machine shops will be able to machine accurately cavities and holes using tools down to 2.5 mm (0.1") in diameter. Anything below this limit is considered micro-machining. Speciality tools (micro-drills) and expert knowledge are required to machine such features (the physics of the cutting process change in this scale), so it is recommended to avoid them unless absolutely necessary.

Feature	Description
	**Tolerances** **Standard:** ± 0.125 mm (0.005") **Typical:** ± 0.025 mm (0.001") **Feasible:** ± 0.0125 mm (0.0005") Tolerances define the boundaries for an acceptable dimension. The achievable tolerances vary depending on the base dimension and the geometry of the part. The values above are reasonable guidelines. If no tolerances are specified, most machine shops will use the standard ± 0.125 mm (0.005") tolerance.

Feature	Description
	**Text & Lettering** **Recommended:** font size 20 (or larger), 5 mm engraved Engraved text is preferred over embossed text, as less material is removed. Using a Sans-Serif font (like Arial or Verdana) with a size of at least 20-points is recommended. Many CNC machines have pre-programed routines for these fonts.

Machine setups & Part orientation



Schematic of a part thast requires multiple setups

It was mentioned previously that tool access is one of the main design limitations in CNC machining. To reach all surfaces of the model, the workpiece has to be rotated multiple times.

For example, the part of the image above must be rotated three times in total: two times to machine the holes along the two main principal directions and a third time to gain access to the back side of the part.

Whenever the workpiece is rotated, the machine has to be re-calibrated and a new coordinate system has to defined.

It is important to think about machine setups while designing for two reasons:

- **The total number of machine setups affects the cost.** Rotating and re-aligning the part requires manual work and increases the total machining time. This is often acceptable if the part needs to be rotated up to 3-4 times, but anything above this limit is excessive. More design tip on reducing the cost of CNC can be found here.
- **To achieve the maximum relative positional accuracy**, two features must be machined in the same setup. This is because the new callication step introduces a small (but non-negligible) error.

5-axis CNC machining

The need for multiple machine setups can be eliminated when using a 5-axis CNC machining. Multi-axis CNC machining can manufacture parts with complex geometries, as they offer 2 additional rotational axes.

5-axis CNC machining allows the tool to remain constantly tangential to the cutting surface. More intricate and efficient tool paths can be followed, resulting in parts with a better surface finish and lower machining times.

Of course, 5-axis CNC has its limitations. Basic tool geometry and tool access limitations still apply (for example, parts with internal geometries cannot be machined). Also, the cost of using such systems is higher.

Video of a 5-axis CNC machine in action. Source

Designing undercuts

Machining a one-sided and a double-sided dovetail undercut. Source

Undercuts are features that cannot be machined using standard cutting tools, as some of their surfaces are not accessible directly from above.

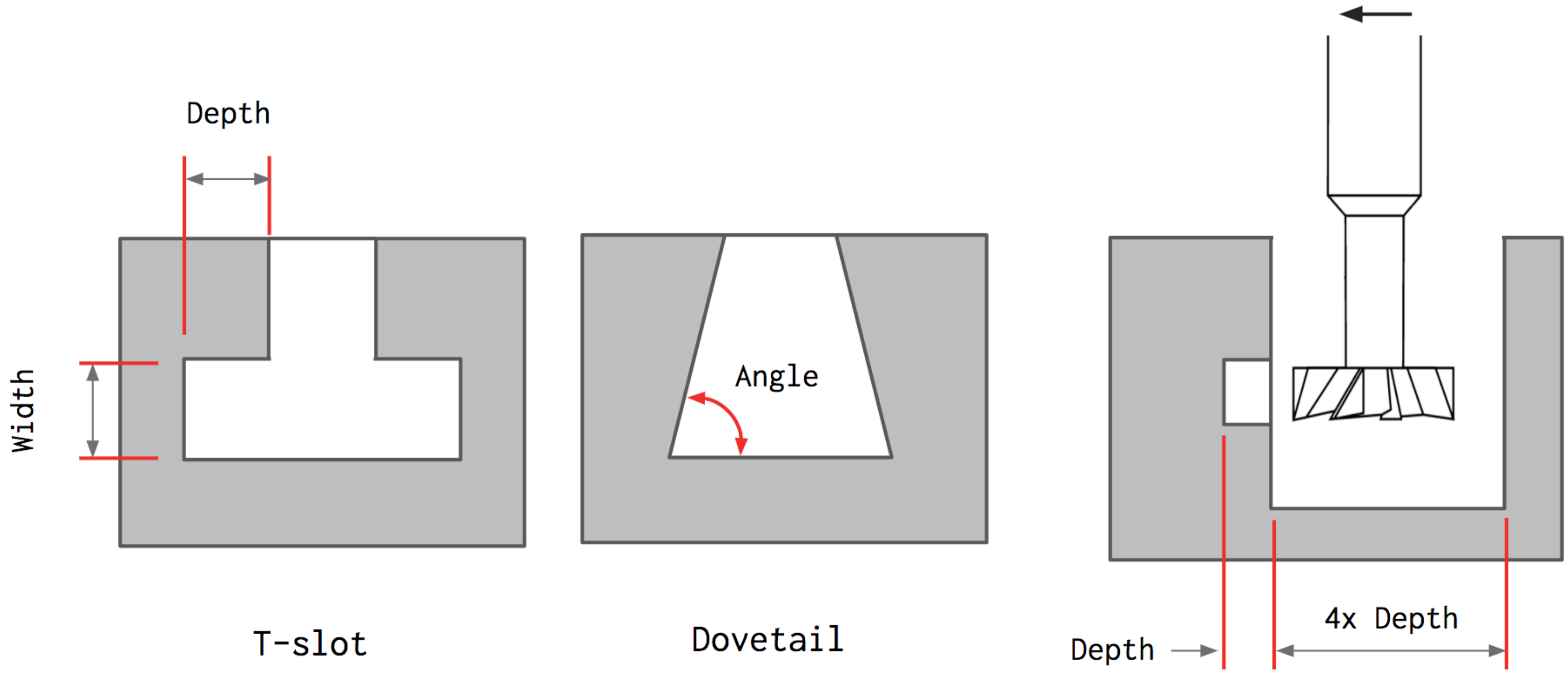
There are two main types of undercuts: **T-slots** and **dovetails**. Undercuts can be one-sided or double-sided and are machined using special tools.

T-slot cutting tools are essentially made of a horizontal cutting blade attached to a vertical shaft. The width of an undercut can vary between 3 mm and 40 mm. Using standard sizes for the width (i.e. whole millimeter increments or standard inch fractions) is recommended, as it more likely for a tool to be already available.

For **dovetail** cutting tools, the angle is the defining feature size. 45° and 60° degree dovetail tools are considered standard. Tools with an angle of 5°, 10° and up to 120° (at 10° increments) also exist, but are less commonly used.

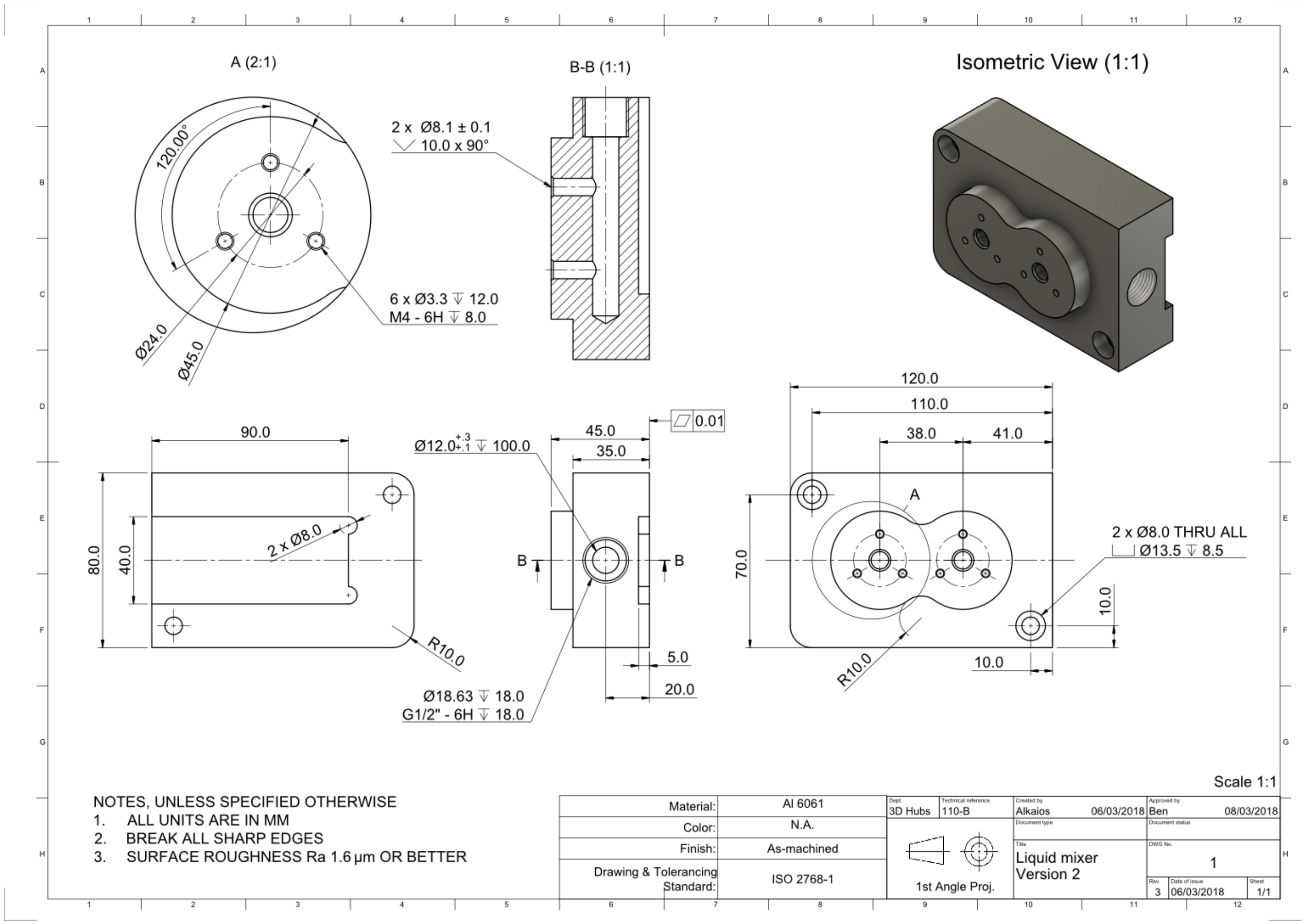
When designing parts with undercuts on internal walls, remember to **add enough clearance** for the tool. A good rule of thumb is to add space equal to at least 4 times the depth of the undercut, between the machined wall and any other internal wall.

For standard tools, the typical ratio between the cutting diameter and the diameter of the shaft is 2:1, **limiting the cutting depth**. When a non-standard undercut is required, it is a common practice for machine shops to manufacture their own custom undercut tools. This can add to the lead time and cost, so it should be avoided if possible.



A T-slot (left), a dovetail undercut (middle), and a one-sided undercut on an internal wall (right).

Drafting a technical drawing



A well-drafted technical drawing

Even though a technical drawing is not always required to place a CNC order (a STEP or IGES file is sufficient), it is recommended to also submit a drawing, as it improves the communication between the designer and the CNC machine operator.

Be aware that certain design specifications cannot be included in a STEP or IGES file. If your model includes one or more of the following, providing a 2D technical drawing is mandatory:

- **Threaded** holes or shafts.
- Dimensions with **tolerances**.
- Specific **surface finish** requirements.
- **Notes** for the CNC machine operator.

An extensive article on preparing a technical drawing for CNC machining can be found here.

Rules of Thumb

- Design parts that can be machined with the tool of the largest possible diameter.
- Add the large fillets (at least ⅓ x cavity depth) to all internal vertical corners.
- Limit the depth of cavities to 4 times their width.
- Align the main features of your design along one of the six principal directions. If that is not possible 5-axis CNC machining is an option.
- Submit a technical drawing with your drawing when your design includes threads, tolerances, surface finish specifications, or other notes for the machine operator.