

5.1.3 Vertical cut (V) vs. Horizontal cut (H) — what they mean physically

Vertical cut (V) vs. Horizontal cut (H) — what they mean physically

5.1.3 concept

When making oval cuts at the drill press, the operator physically moves the ball relative to the drill bit along one of two axes to remove material and elongate the pilot hole. These two directions of movement are referred to throughout pro shop practice — and throughout Book 05 — as **vertical cuts (V)** and **horizontal cuts (H)**. Understanding what each means physically is essential before working with the x/y offset values Spectre Cloud calculates.

↕ Vertical Cut (V)

A **vertical cut** is made by moving the ball along the **y-axis** — toward or away from the fingers — while the drill bit remains in the hole. The bit removes material along the vertical plane, elongating the pilot hole in the direction of movement.

- □ A V cut toward the fingers moves the hole opening in the direction of the span — relevant to the span allowance discussion in 4.5.4.

- A V cut away from the fingers moves the hole opening toward the back of the ball — the most common direction for single-direction oval cuts.
- For larger ovals requiring more material removal than a single pass can cleanly achieve, multiple V cuts are made in sequence — each pass removing an incremental amount of material until the full y-axis offset is reached.
- A V cut does not move the ball side to side — any lateral component of the oval requires an H cut.

In plain terms: A vertical cut is the drill press equivalent of sliding the ball forward or backward while the bit is running — the hole gets longer in the direction you move it.

↔ Horizontal Cut (H)

A **horizontal cut** is made by moving the ball along the **x-axis** — side to side, left or right — while the drill bit remains in the hole. The bit removes material along the horizontal plane, elongating the pilot hole laterally.

- An H cut to the right or left elongates the hole in the corresponding lateral direction.
- Like V cuts, larger ovals may require multiple H cut passes to reach the full x-axis offset cleanly without taking too much material in a single movement.
- An H cut does not move the ball toward or away from the fingers — any vertical component of the oval requires a V cut.

In plain terms: A horizontal cut is the drill press equivalent of sliding the ball left or right while the bit is running — the hole gets longer in the direction you move it.

☐☐ How V and H Work Together

Most oval thumb holes require **both** a V cut and an H cut — because most oval degree values are neither perfectly vertical (/) nor perfectly horizontal (). Spectre Cloud's x/y calculation output tells the operator exactly how much V movement and how much H movement is needed to produce the oval at the correct angle.

Oval degrees	V cut required?	H cut required?	Relationship between V and H
<input type="text" value="0°"/> / <input type="text" value="180°"/>	<input type="checkbox"/> Yes — full travel	<input type="checkbox"/> No	Pure V cut — all movement is vertical
<input type="text" value="90°"/>	<input type="checkbox"/> No	<input type="checkbox"/> Yes — full travel	Pure H cut — all movement is horizontal

Oval degrees	V cut required?	H cut required?	Relationship between V and H
45° / 135°	<input type="checkbox"/> Yes	<input type="checkbox"/> Yes	V and H travel are equal — movement split evenly between axes
Any other angle	<input type="checkbox"/> Yes	<input type="checkbox"/> Yes	V and H travel differ — proportions determined by the degree value

Note: The combined result of the V and H movements traces a straight line through the pilot hole center at exactly the oval degrees angle. Spectre Cloud calculates the correct V and H distances so the operator does not need to resolve the angle manually — the output tells you precisely how far to move in each direction.

Multiple Passes on Larger Ovals

For ovals with a large difference between the starting bit diameter and the oval width, attempting to reach the full V or H offset in a single pass risks tearing the ball material or producing an uneven cut. In these cases the total offset is divided across multiple passes — each pass removing a controlled amount of material until the cumulative movement reaches the calculated V or H distance.

- Divide the total V or H offset into equal increments — for example, a total V offset of might be executed as three passes of each.
- Take the same number of passes on both V and H axes to keep the oval shape developing evenly — an oval that is cut fully on one axis before the other is started can produce an uneven or asymmetric finished hole.
- Smaller increments per pass produce a cleaner edge on the finished oval — particularly important for urethane and reactive resin ball surfaces.
- Do not attempt to reach the full offset in one pass on a large oval — the risk of material tearout or a ragged hole edge outweighs the time saved.

Verify with Spectre team: confirm whether Spectre Cloud's oval cut output includes recommended pass increments for larger ovals, or whether the number of passes is left to the operator's discretion based on the total offset distance.

Tips for Clean V and H Cuts

- Zero your drill press lateral and forward/back adjusters after drilling the pilot hole — this gives you a clean reference point from which to measure the V and H offsets precisely.

- ☐ Make V cuts before H cuts as a consistent practice — establishing a standard sequence reduces the risk of losing track of which axis has been cut and by how much.
- ☐ After completing all V and H passes, return the ball to the zero position and check the hole visually — the oval should be centred on the pilot hole with clean, even edges on all sides.
- ☐ If the finished oval looks off-centre or asymmetric, check whether the V and H offsets were applied in both directions (bidirectional) or only one direction — a missing return pass is the most common cause of an oval that looks right on one side and wrong on the other.
- ☐ Do not mix up V and H directions mid-sequence — if you lose track of which axis you are on, stop, return to zero, and restart the pass sequence from the beginning rather than guessing.

Related Sections

- 5.1.1 — What is an oval thumb hole and why is it used?
- 5.1.2 — Oval terminology: Starting Bit, Oval Width, Degree, Taper
- 5.2 — Oval degrees — understanding hole orientation
- 4.5.3 — How the system calculates the oval cuts from your inputs
- 4.5.4 — Entering the span with an oval thumb
- Book 06 — Drilling Your First Ball

Tip: The V and H cut sequence becomes second nature after a handful of oval drillings. Until it does, keeping the Spectre Cloud spec sheet visible at the drill press — with the calculated V and H values clearly in view — and calling out each movement aloud before executing it is a simple habit that prevents the most common oval drilling errors.

...

Revision #2

Created 11 May 2026 16:04:41 by Admin

Updated 27 May 2026 20:28:31 by Art