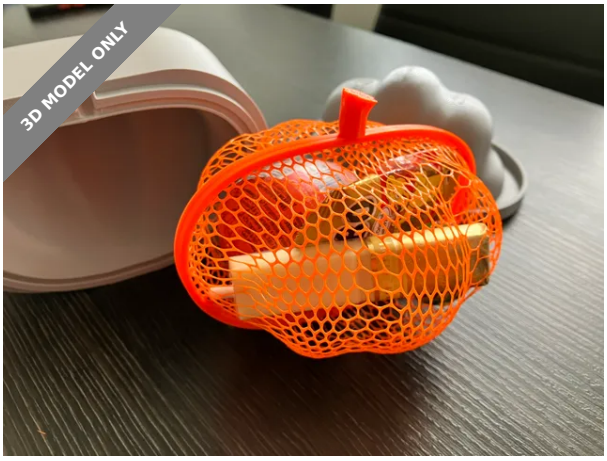


Thermoformed pumpkin - halloween

hvrkh

VIEW IN BROWSER

updated 31. 10. 2026 | published 31. 10. 2026

Summary

A pumpkin-shaped Halloween treat box—perfect for sweets or a little surprise for trick-or-treaters.

Seasonal designs > Autumn & Halloween

Tags: halloween pumpkin diy 3dprinting pla petg candy 3dprinted prusa3d 3dprint trickortreat giftbox handmade moldmaking candybox partyfavor halloweendecor spookyseason giftidea thermoforming falldecor seasonaldecor treatbox trickortreaters halloweengift pumpkindesign pumpkinsurprise sweettreat

Halloween Pumpkin Surprise – Thermoforming Mold

Make Halloween treats extra special with this pumpkin-shaped surprise box. Once formed and assembled, it becomes a unique little gift package for sweets or other small surprises—perfect for trick-or-treaters, party guests, or anyone who deserves a handmade Halloween treat.

This model uses a thermoforming mold made from two parts: Mould Outer and Mould Press. You'll also need to print both different inserts included

with the model. They are not interchangeable: printing the same insert twice will result in mismatched threads at the top.

Forming and assembly

1. Place one insert inside Mould Outer.
2. Use a heat gun to heat the honeycomb surface evenly until it softens enough to form.
3. Press Mould Press into the insert.
4. If the infill still resists, stop pressing and heat the surface some more, then try again.
5. Repeat with the second inser.
6. Fill the pumpkin with sweets or a small gift.
7. Secure the bottom with the clip, then screw the stem on top.

Stem and clip
Print the stem and the small clip. The stem has an internal thread and works like a nut: screw it onto the top to hold the two pumpkin halves together. The clip snaps into the recess on the bottom to secure it in place. The stem can be printed in a contrasting colour.

Recommended print settings

Inserts

- Print both included inserts—one of each.
- Layer height: 0.15 mm
- Supports: off
- Top solid layers: 0
- Bottom solid layers: 0
- Perimeters: at least 4
- Infill: 25% Honeycomb
- The recommended settings should produce three infill layers.
- PLA or PETG can be used. The inserts shown in the photos are printed in PETG.
- Print profile structural - to slow the priner down during printing.

Mould

- Layer height: I printed 0,22mm
- Supports: off
- Perimeters: at least 3
- Use the standard profile for the rest of the settings

Model files



mould-outer-10.stl



mould-press.stl



clip.stl



insert1.stl



insert2.stl



stopka.stl



pumpkin-inserts-print-settings.3mf

License

This work is licensed under a
Creative Commons (4.0 International License)

Attribution—Noncommercial—No Derivatives

✖ | Sharing without ATTRIBUTION
✖ | Remix Culture allowed
✖ | Commercial Use
✖ | Free Cultural Works
✖ | Meets Open Definition