

# Hydrofill

## Support

<b>Category:</b>	Soluble
<b>When to use:</b>	Cases where soluble support material like Hydrofill really shine are with prints that feature internal geometry or extremely delicate features.

Print Settings	
<b>Extruder Temperature</b>	234°C
<b>Bed Temperature</b>	70°C
<b>Bed Preparation</b>	Use corresponding WolfBite adhesive that the primary build material calls for.
<b>Cooling Fan</b>	Yes
<b>Closed Chamber</b>	Yes
<b>Heated Chamber</b>	No
<b>Chamber Temperature</b>	-
<b>Notes</b>	

Material Management	
<b>Compatible Materials</b>	<ul style="list-style-type: none"> <li>• ABS</li> <li>• PLA</li> <li>• Nylon</li> <li>• TPU</li> </ul>
<b>Support Removal Suggestions</b>	<ul style="list-style-type: none"> <li>• Separate support material with hands, nails, picks, pliers, etc.</li> <li>• Soak the part in hot water for several hours. Agitation and heat work very well to speed up the process.</li> </ul>
<b>Storage</b>	<ul style="list-style-type: none"> <li>• Moisture-free environment.</li> </ul>
<b>Drying</b>	<p><b>Temperature</b> 70°C – 80°C</p> <p><b>Dry Time</b></p> <ul style="list-style-type: none"> <li>• Heat Only → 2 – 3 hours</li> </ul>

Slicer Settings	APEX		
<b>Material Profile Name</b>	Hydrofill	ABS with Hydrofill	Nylon with Hydrofill PLA with Hydrofill TPU with Hydrofill
<b>Extruder Temperature</b>	234°C	245°C	235°C

Available at the [TRAK](#) and [Airwolf 3D](#) stores for purchase.