

December 19, 2016

PropCad: Tips for root fillets with Rhinoceros3D

Root fillets are often the most time-consuming process during CAD/CAM migration.

PropCad is often used as an intermediate step to the final CAD model of the propeller. There are often additional details included in the CAD model such as puller holes, stern seal recesses, keyway reliefs, etc. As a result, the blades and hub geometry from PropCad is often exported to a native CAD format for direct manipulation within that environment.

PropCad data is exported to native CAD formats using “macros” that call the specific CAD commands. One of the most difficult surfaces to recreate in a native CAD format is the root fillet. Fillets are highly-specialized routines and have different inputs for each native CAD environment – some fillets must be created as surfaces, while other must be derived from solid geometry.

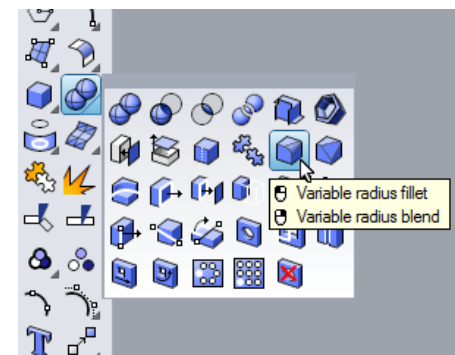
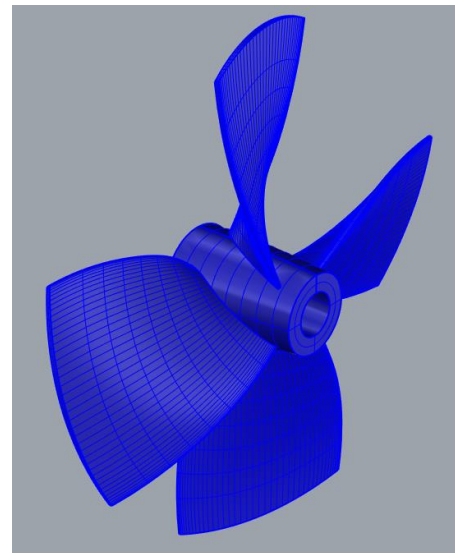
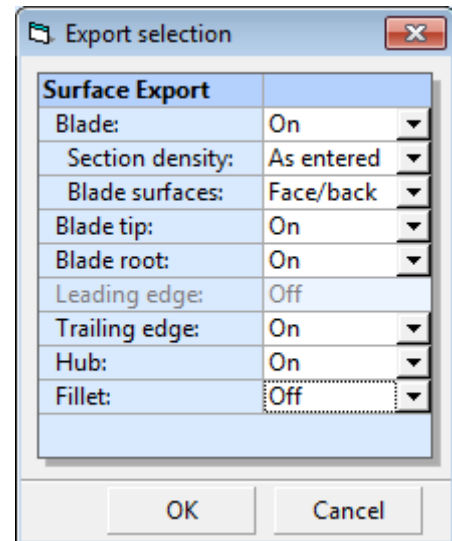
The Rhino3D command file (*.txt) is one of the most popular CAD exports for PropCad projects. Rhino is very useful because it has sufficient tools to handle both solids and surfaces. Recently, HydroComp staff found a new, Solid-based approach to developing root fillets. This approach is considered the most-effective and most-expeditious method for adding root fillets to a propeller model.

The propeller below is a 4-blade Kaplan developed in PropCad. First, we export the propeller using the *Rhino 3/4/5 macro* (*.txt). The Surface Export options are used to export the blade surfaces as well as the root, tip, and edge surfaces. The hub is also exported, but the fillet (generated from PropCad) is disabled.

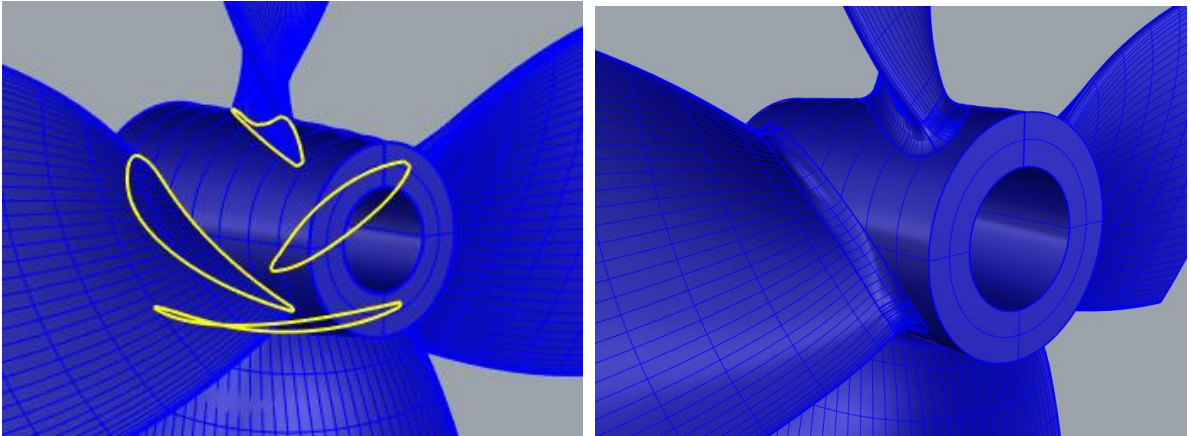
The Rhino macro is imported to Rhinoceros 3D (v5.12.50810) via *Tools > Commands > Read from File*. The first step is to join all blade surfaces (including the root, tip, and edge surfaces) into a solid using the *Join* (🔗) command. This solid blade should be rotated and copied using *Transform > Rotate* to create the correct number of blades. The individual solid blades are combined with the solid hub using the *Solid > Union* command.

To create the root fillet, use the Solid variable radius fillet command. This command is located under the Solid fly-out context panel. The solid variable radius fillet allows you to select multiple edges, which makes creating a full-round fillet much easier than manually intersecting individual fillet surfaces for each edge.

The steps for creating the fillet are shown below. First, all applicable edges for the root of each blade are selected. The



radii can be defined using a single value in the command window.



The resulting geometry is already a valid closed polysurface, so no further editing or sewing of edges is required.

About HydroComp

Celebrating its 30th year of operation in 2015, HydroComp provides software and services for resistance and propulsion prediction, propeller sizing and design, and forensic performance analysis. HydroComp is proud to have served over 700 industry, research, academic, and government clients from more than 60 countries.

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