# Advanced Features

Your task is to recreate the blades of this vintage screw propeller. Create a solid sweep based on two existing profiles, add some fillets and then pattern the new feature to complete the driving gear.

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1. Step 1: Sweep the Profiles

To create the propeller blade, you will sweep a new solid feature from two existing profiles using **Generic Solid**.
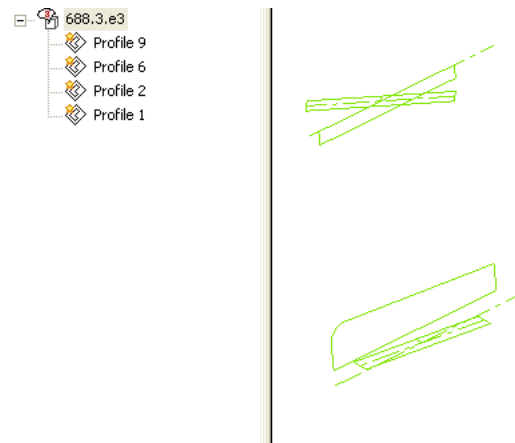
Some of the profiles we need for this step are created.

Now, we can create the blade with **Generic Solid**.

- Select **Insert**↘**Solid**↘**Sweep**↘**Generic Solid** from the menu.

- Click on the two rectangular profiles to select the boundaries for the command.
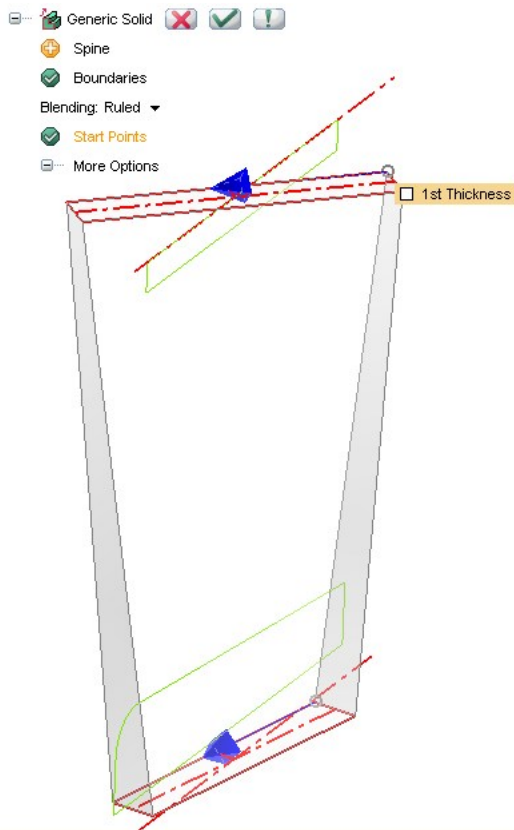
Notice the grey preview lines in the image above. They are straight, and you would like them to allow water to flow across them smoothly, so we can change the Blending option from Ruled to Hermite too. Switch to various blending options and observe the result.
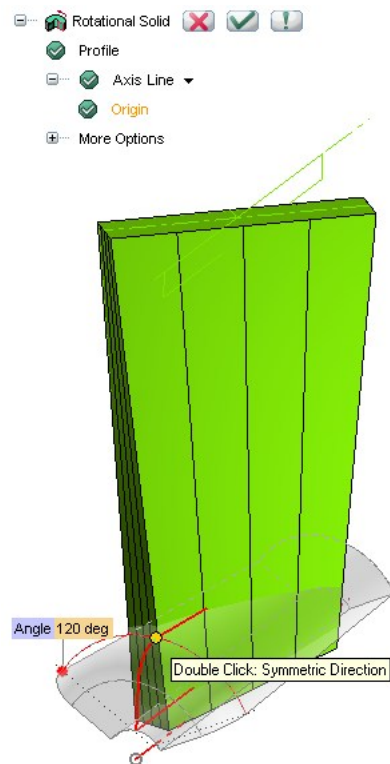
- Keep the Blending to Ruled in the Selection List.

- Hit ✅ OK to finish the command.

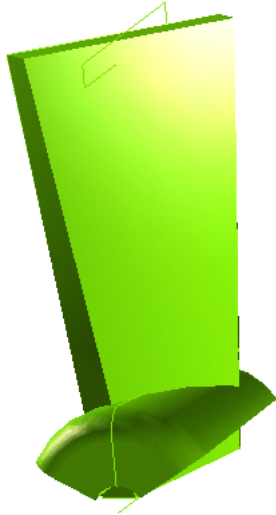Using Profile2, create a **Rotational Solid**.

- Activate **Rotational Solid**.

- Select Profile2.

- Pick the bottom most reference line for the axis selection.
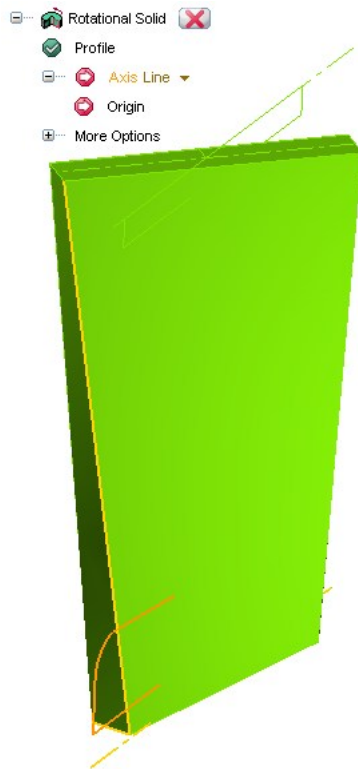
- Change the angle to 120 deg.

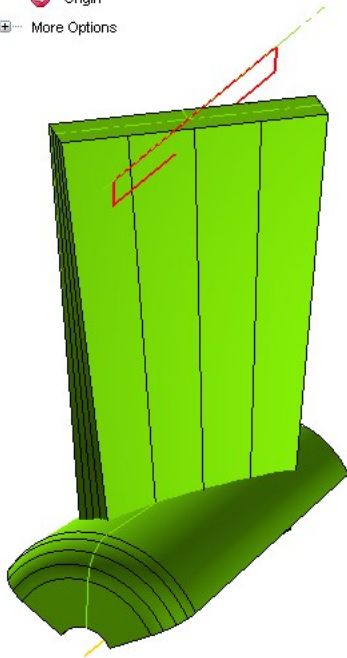- Right click on the minidialog box and say Symmetric.
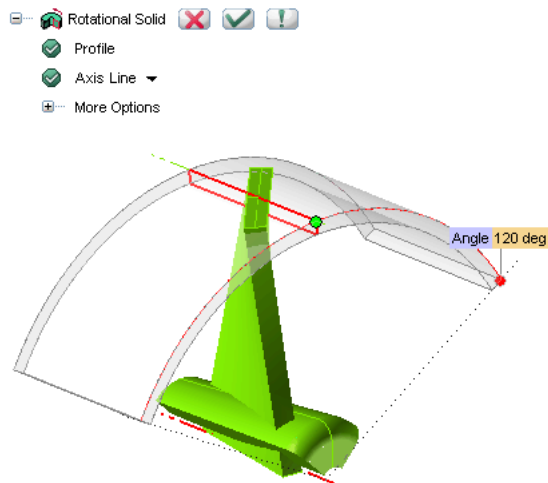

- Hit ☑ OK.

Follow the above procedure to create another rotational solid using Profile6.

- Select **Rotational Solid**.

- Pick Profile6 in the history tree.

- Pick the reference line from Profile1 as axis line.

- Set angle to 120 deg and Symmetric.

- Click ✔ OK.

Hide Entities all the dimension lines.

# 2. Step 2: Union of the Ring and the Shaft.

The blades should be attached to the center solid and the outer ring. So, using the Local Boolean **Solid Union** command, you will attach the outer ring to the shaft and blades to create a single entity.

Let's attach blade to the shaft now.

- Activate the **Solid Union** command.

- Change the Boolean option to Local.

- Select blade and shaft as Solids.
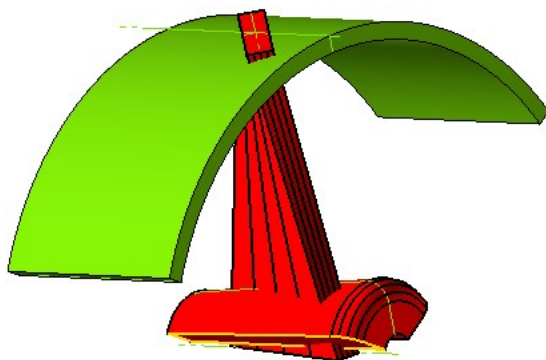
- Select the outer surface of the shaft for Faces selection.
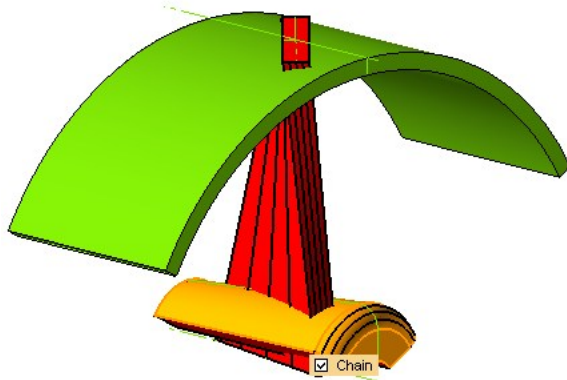
NOTE

Starting from this version, a new enhancement is done in the Solid Boolean command. This enhancement allows you to retain the original solids on which the Boolean command is applied. This enhancement is applicable for both Global and Local option. Refer to Release Notes for more information.
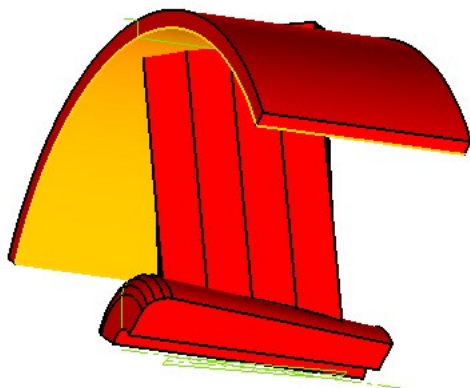

- Hit ✔ OK.
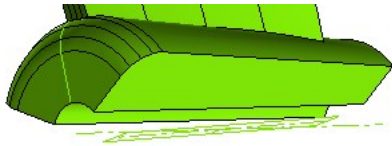
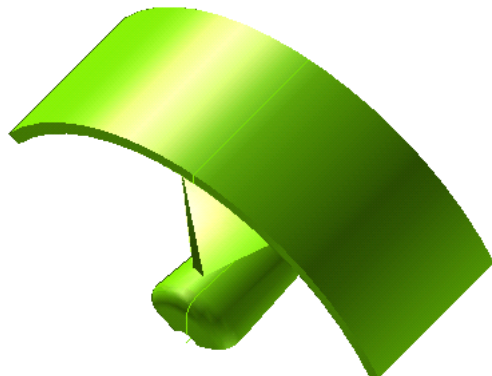Now again use **Solid Union** to union the ring to the rest of the propeller solid.

- Select **Insert↘Solid↘Boolean↘Union** from the menu.

- Change the type of Boolean to Local.

- Click on each of the two solids.
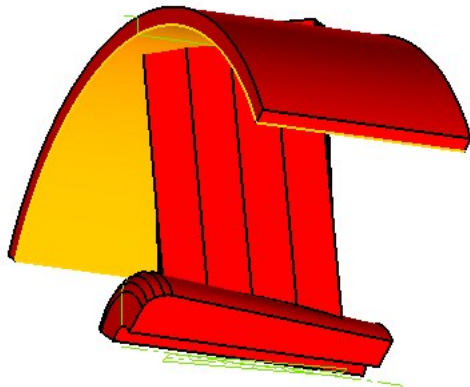
- Now select the inner surface of the ring.

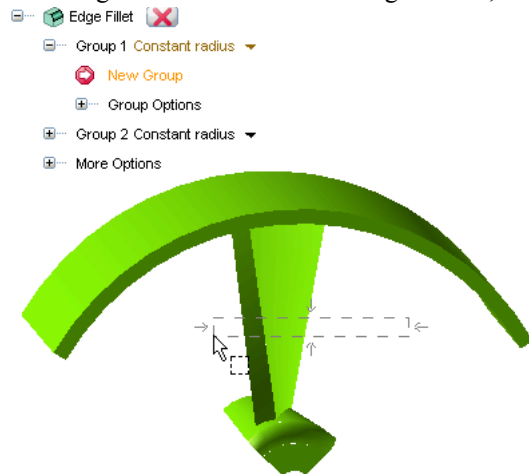- Hit ☑ OK to complete the command.

Now, by rounding the corners and edges of the blade, we can make it more hydrodynamic to make the propeller

more efficient.

Let's add those fillets.

- Select **Fillet Edges** command from the toolbar.

- Using window selection from right to left, select all the 4 edges of the blade.
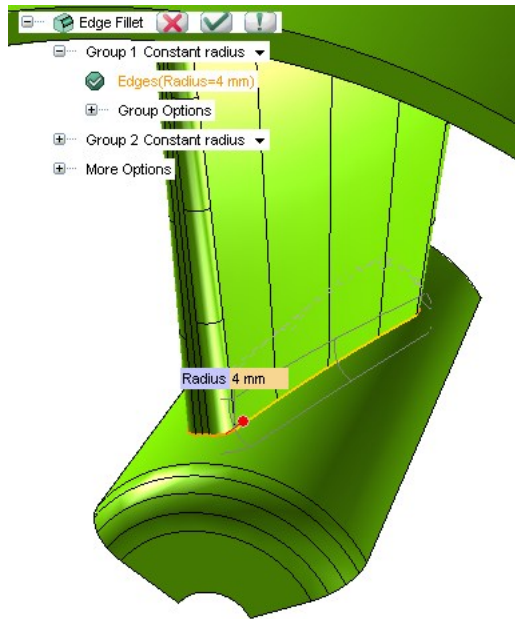

- Set Radius2 mm

- Hit ✓ OK

You have selected the **Fillet Edges** command and now each area where the blades intersect the outer ring and shaft should be filleted.

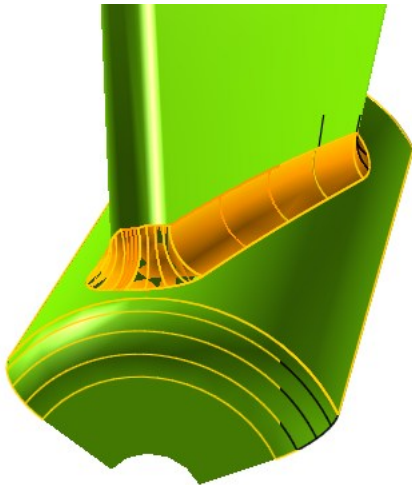# 3. Step 3: Fillets - Constant arc length & Face - to - Face.

Let's add fillets at the edges of the blade, where it touches the shaft and the Outer ring.

- Start **Fillet Edges**.

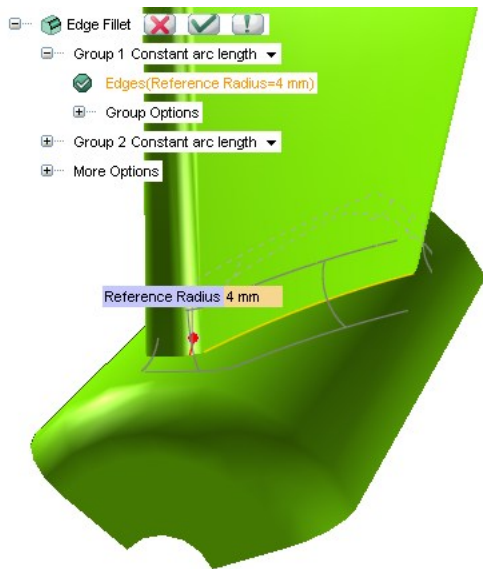- Select the edges of the blade and shaft.
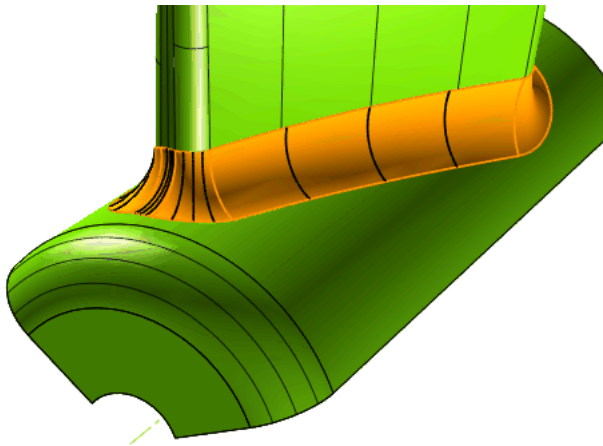
- Set the Radius4 mm

- Hit ✅ OK.

You may need to zoom in and/or dynamically rotate the model to observe the fillet.

- Right click on the last fillet created and say Redefine Feature.

- Change the Group1 option to Constant arc length.

- Hit ✓ OK

Observe the difference between these 2 options.

This version introduces a new type of fillet called the **Fillet Face-to-Face**. By using this command it is possible to apply a constant radius fillet by selecting two faces.

- Start **Fillet Face-to-Face**.

- Select any one face of the Blade and Inner face of the Propeller as shown below.
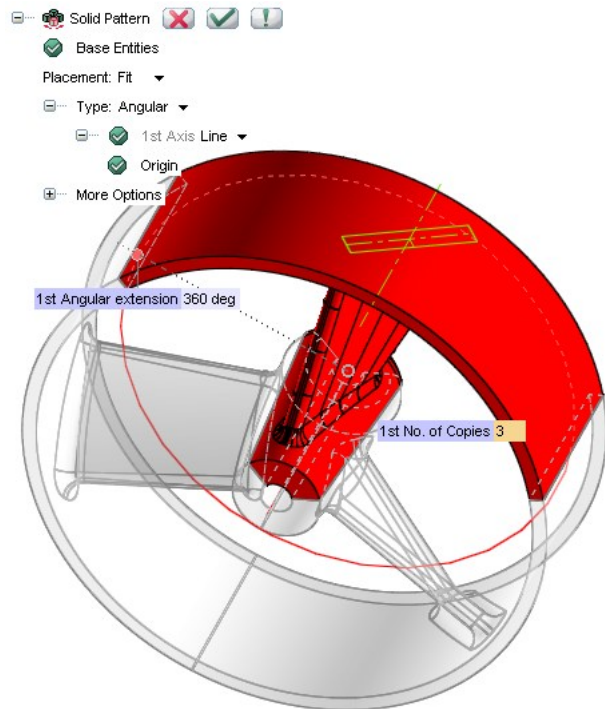
- Set the Radius4 mm

- Hit ✓ OK.
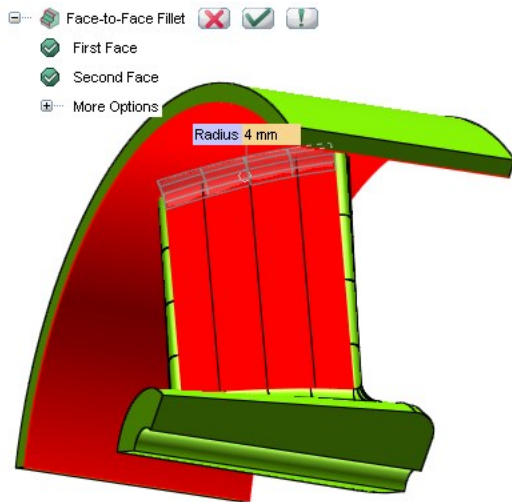
Since the adjacent faces of the blade are tangential to the selected face, Fillet is applied on all the faces as shown below.
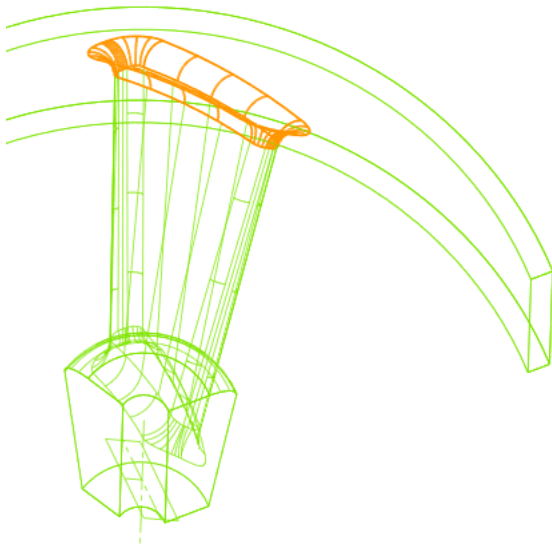
4. Step 4: Pattern and union.

You will be needing at least three of these blades, so use the **Pattern Solid** command.

• Click the **Pattern Solid** icon.

• Set the Type to Angular in the Selection List.

• Set the Placement option to Fit.

- Set 1st No. copies3 .

- Set 1st Angular step360 deg

- Hit ✅ OK to complete the command.

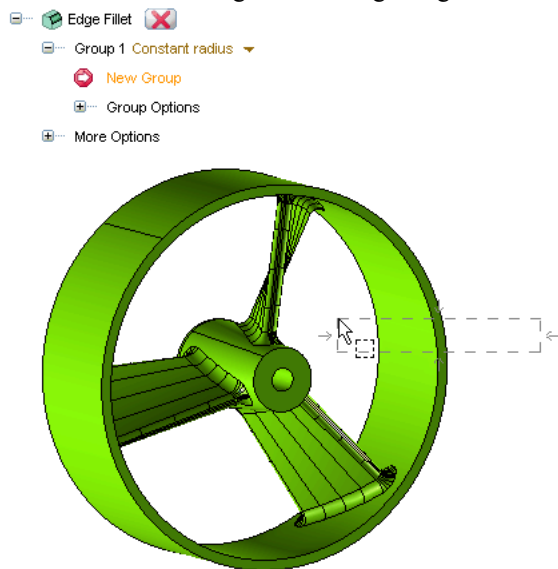Pretty cool, isn't it? Now you need to union this solid.

- Hit **Fit View**.

- Start **Solid Union**.
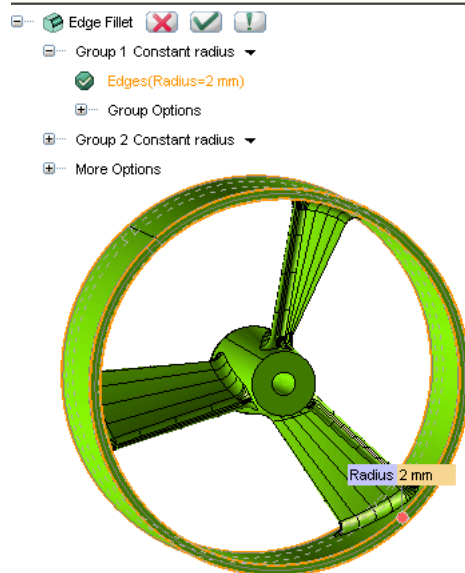
- Select all the 3 solids.
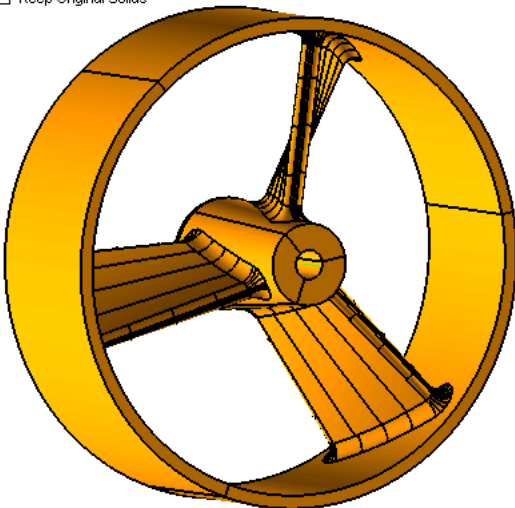
- Hit ✔ OK.

Let's add fillets to the edges of the ring.

- Start **Fillet Edges**.

- Change the Group 1 option to Constant radius.

- Select the four edges of the ring using window selection from right to left.

- Set the Radius2 mm .

- Hit ✔ OK.

- Hide all the profiles and dimensional lines using **Hide Entities**.

If you look at the History Tree, you will notice that only the solid objects are visible. Good job!