



User Guide for Version 4.2

Table of Contents

Table of Contents.....	2
1 Overview	3
2 Submarine GUI Overview	4
2.1 Submarine Icon Description	4
3 Submarine Features	5
3.1 Boundary Condition Setup.....	5
3.2 Solver Setup.....	5
3.3 Colour Selection	7
3.4 Clip Planes.....	7
3.5 Iso-Surfaces	8
3.6 Streamlines	8
3.6 Vortex Cores.....	9
4 Submarine Example.....	10

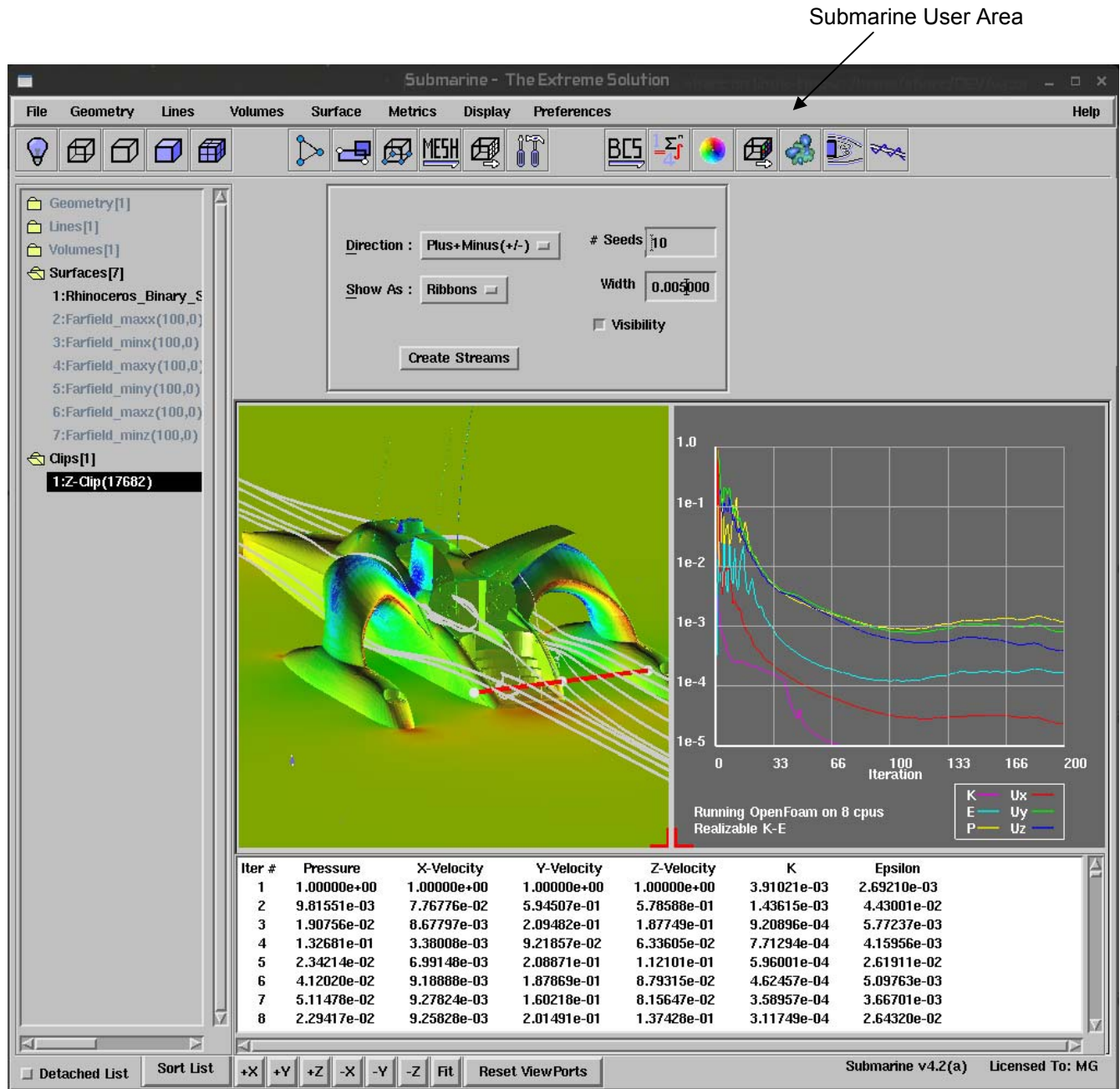
1 Overview

Submarine is a simple generic solver interface within Harpoon. Setup is extremely easy, as is solver choice. Post-processing may be done whilst the case is solving or afterwards.

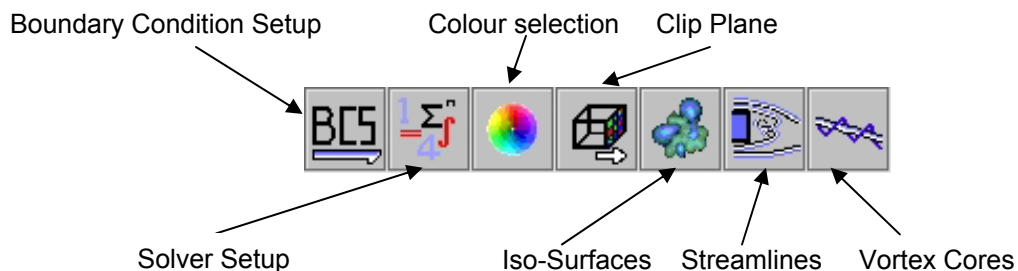
Solvers used currently are OpenFoam and Fluent. These must be installed by the user prior to running Submarine.

This guide is a supplement to the Harpoon user guide and as such basic operations will not be discussed here.

2 Submarine GUI Overview

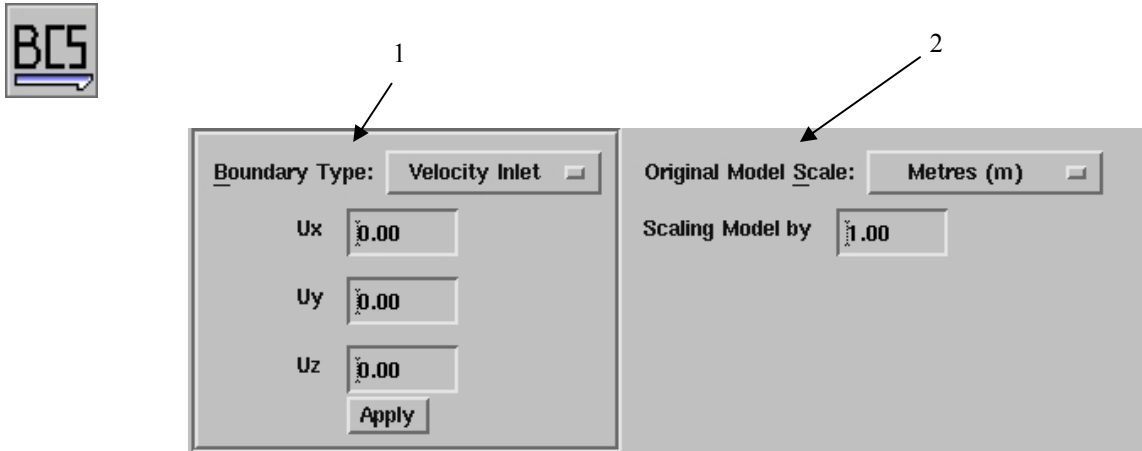


2.1 Submarine Icon Description



3 Submarine Features

3.1 Boundary Condition Setup

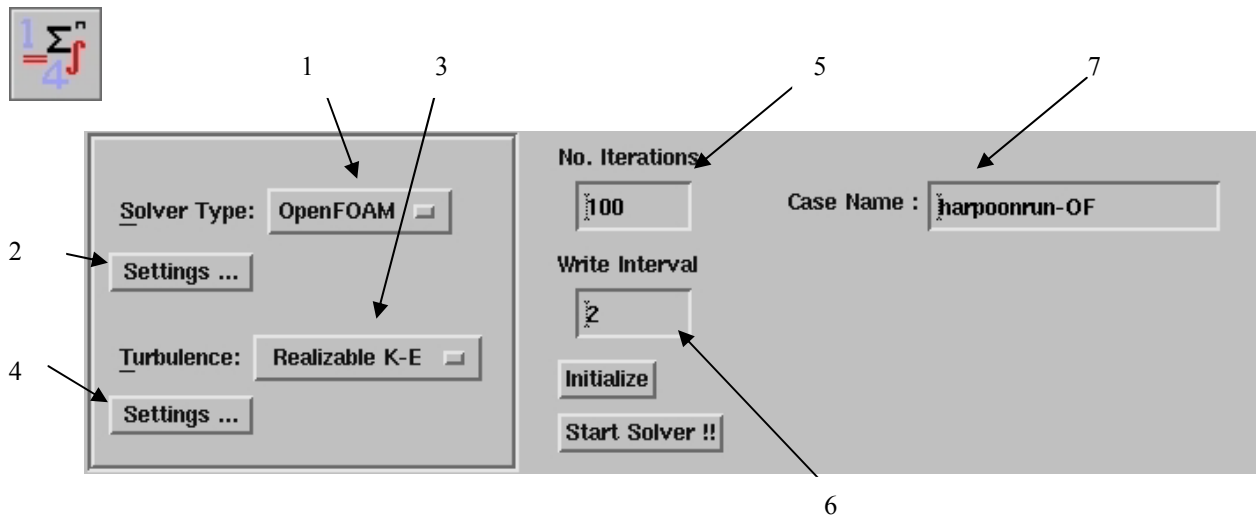


Pressing the **Boundary Condition** icon produces the above user area

Select a surface mesh part and select a **boundary type (1)**. Boundary types are currently: Velocity Inlet, Mass Flow Inlet, Interior, Symmetry, Pressure Outlet or Wall.
Press **Apply** if any numbers have been changed (eg velocity)

If the model is in mm then select the **Original Model Scale (2)** to be mm.

3.2 Solver Setup



Pressing the **Solver Setup** icon produces the above user area.

Select **Solver Type (1)**

To change schemes and parallel setting select **Settings (2)** under **Solver Type**

To change turbulence models select **Turbulence (3)**

To change turbulence setting for OpenFoam, select **Settings (4)**.

Enter number of iterations **(5)** and the write interval **(6)**.

Change name if needed (OpenFoam only) **(7)**

Hit Initialize and then Start Solver

Below is the **Solver Settings** area:

Solver Scheme: 1st Order <input type="checkbox"/>			Parallel Settings		
			# of CPUs	1	
Relaxation Factors					
P	0.30	K	0.70	R	0.70
U	0.70	Ep	0.70	Nu	0.70
Decompose Scheme Simple <input type="checkbox"/>					
dx		dy		dz	
1		1		1	
Apply Settings					

Below is the dialog for the turbulence settings when using OpenFoam

Turbulent Coeffs			
Qmu	0.09	AlphaEps	0.77
C1	1.44	eta0	4.38
C2	1.92	beta	4.38
alphak	1.39		

3.3 Colour Selection



Color Selector

R 0.80 G 0.80 B 0.80 A 1.00

Part Color

Variable Colour

Constant
Pressure
Velocity

Palette Attributes

Min 7.14375e+01
Max 2.14375e+01
Num 5

Display: 3D Border-2D Full

☐ Auto Update Range Per Iteration

Pressing the **Colour Selection** icon produces the above user area

Select surface mesh parts, clip planes of iso-surfaces, then choose variable from the list

3.4 Clip Planes



1

1.399

Direction: X

Value 1.400

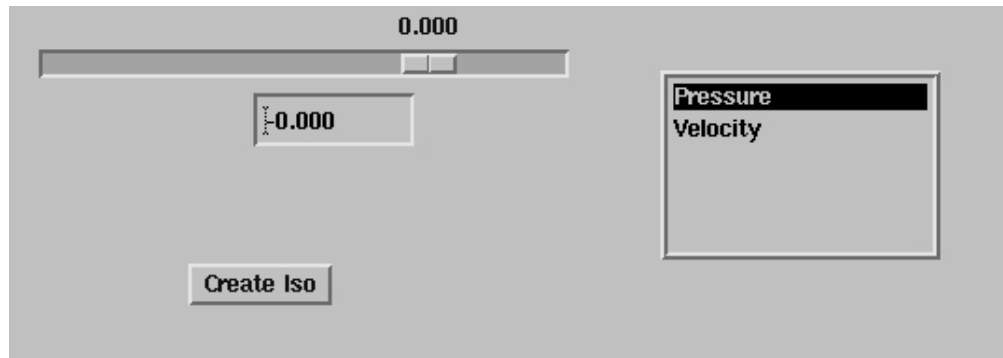
Create Clip

Pressing the **Clip Planes** icon will produce the above user area.

Select the **Direction (1)** of interest (X, Y or Z) and hit **Create clip**

NB The clip must be selected to changed the value

3.5 Iso-Surfaces

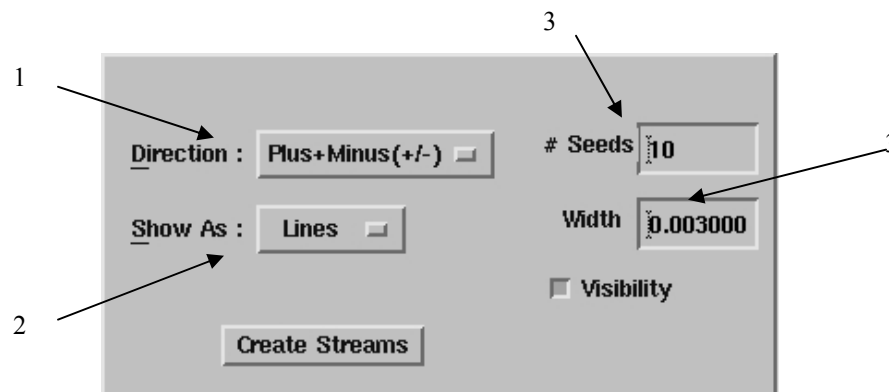


Clicking the **Iso-Surface** icon will produce the above user area.

Select the variable required and hit Create Iso

NB The iso-surface must be selected to changed the value

3.6 Streamlines



Pressing the **Streamline** icon produces the above user area

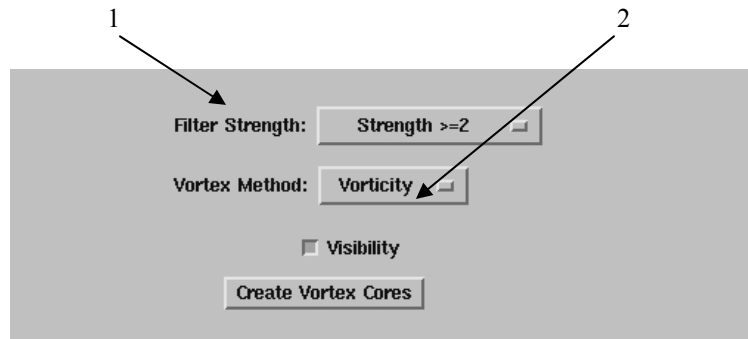
Simply hit Create Streams to a rake and streamlines. The rake can be moved to show how the flow is behaving. The **Direction (1)** can be changed and the streams can be shown as ribbons **(2)**

The number of traces can be changed by changing the number in **#seeds (3)**.
To change the width of the ribbons, enter a different value in **Width (4)**.

Visibility toggles the streamline display on/off.

NB The user must be in the streamline dialog for the streamlines to move.

3.6 Vortex Cores



Pressing the **Vortex Cores** icon produces the above user area

Simply hit **Create Vortex Cores** to calculate the cores.

Change display of cores by using **Filter Strength (1)**

To use a different method of creation select the **Vortex Method (2)** and then rehit **Create Vortex Cores**.

Visibility toggles the core display on/off.

4 Submarine Example

Create the mesh as with example M1 from the Harpoon Users guide.

Go to the **Boundary Condition** area

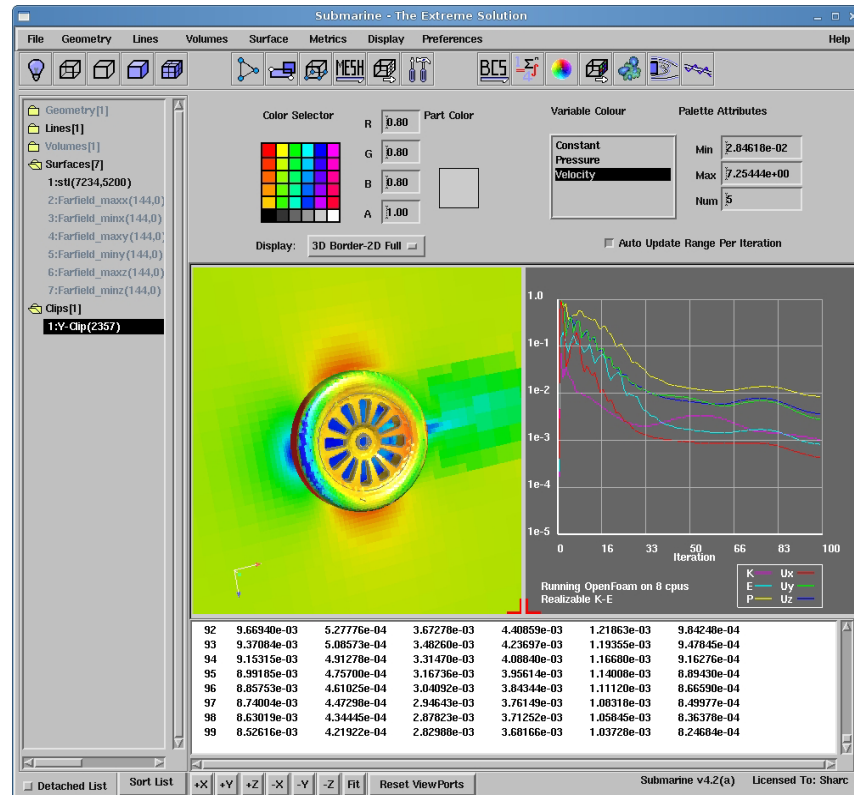
Highlight the surface mesh part Farfield_minx and select velocity inlet as a boundary condition. Only press apply if a new velocity value is required.

Highlight the surface mesh part Farfield_maxx and select pressure outlet as a boundary condition

Go to the **Solver Setup** area

If parallel, then select the settings button under the **Solver Type**. Change **# of cpus** to, say, 8.

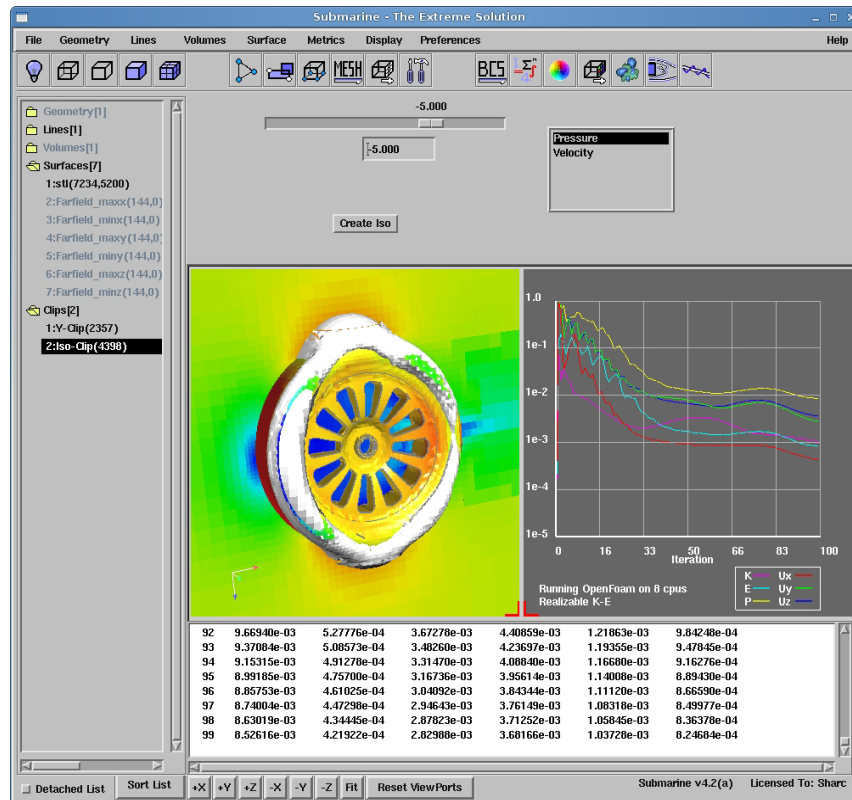
Select the **Solver Setup** icon. Press **Initialize** and then **Start Solving**



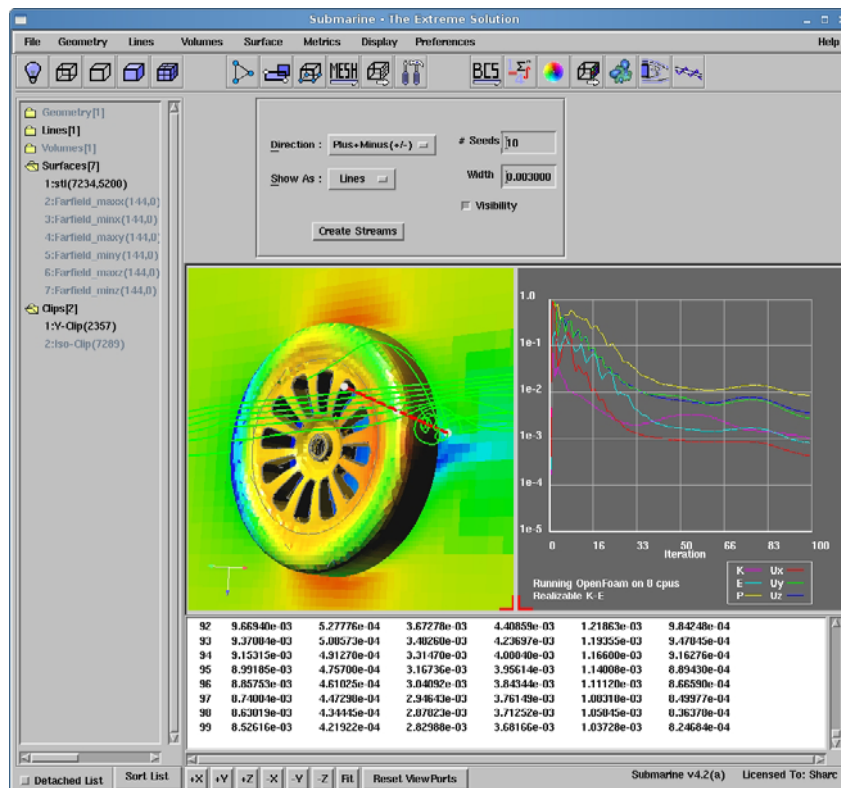
The surface mesh is automatically rendered by the first variable which is pressure.

Go to the **Clip Plane** icon. Select Y and press **Create Clip**. Select the Y-clip in the Parts list and go to the **Colour Selection** icon. Choose velocity.

Go to the **Iso-Surface** icon. Hit **Create Iso**. Select the Iso-clip in the parts list. Enter -5.0 into the text box and hit enter. Below shows the result. User can then colour the Iso-clip by pressure or velocity.



Turn the visibility of the Iso-clip off. Go to the Streamline icon and press **Create Streams**. Interactively move the rake to the middle of the wheel. Select +/- for the **Direction**. Below shows the result.



Go to the **Vortex Core** icon. Hit **Create Vortex Cores**. Select ≥ 2 form the **Filter Strength**. Below is the result

