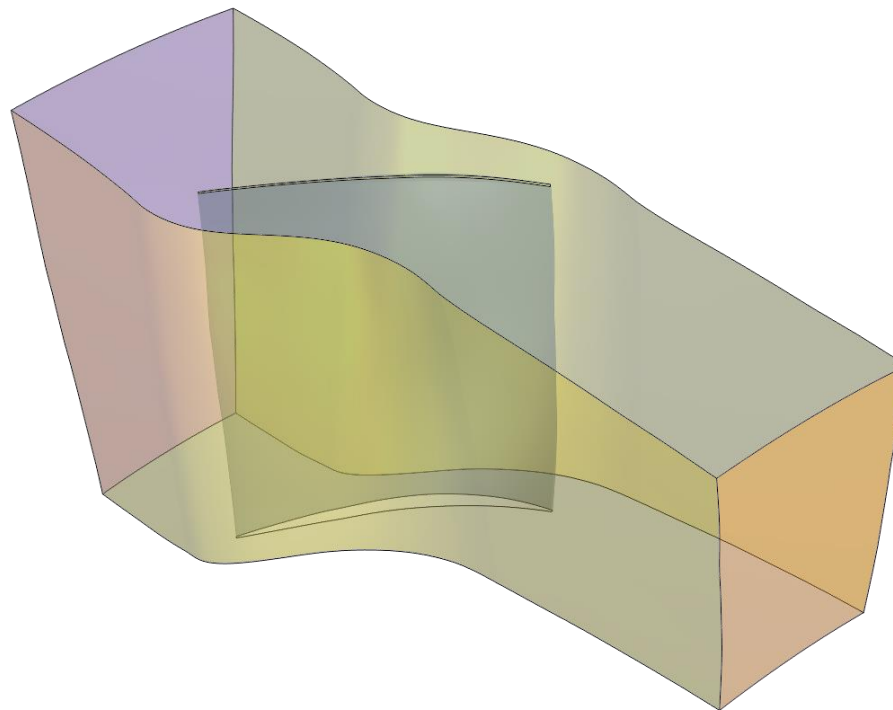


CFD applied to Turbomachinery

Francesco Romanò

`francesco.romano@ensam.eu`

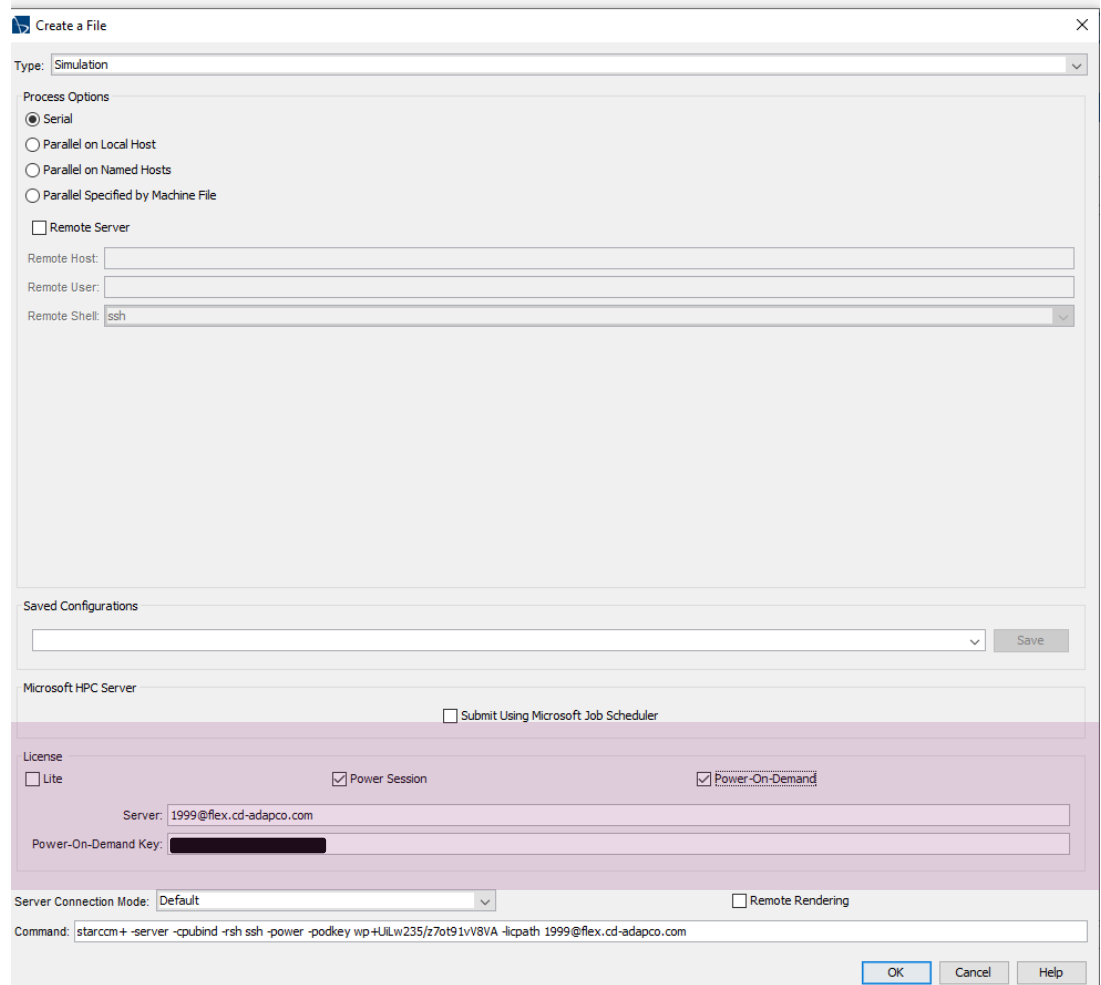
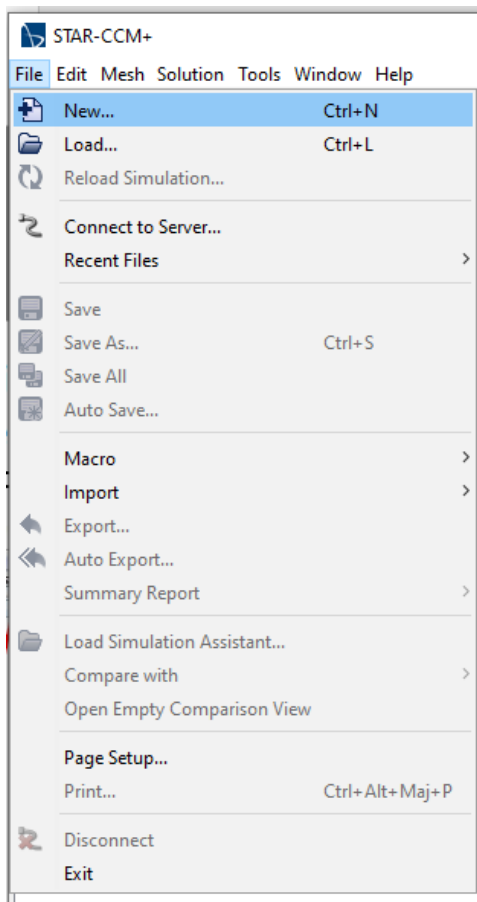
Laboratoire de Mécanique des Fluides Lille, Arts et Métiers Institute of Technology, 59800, Lille, France



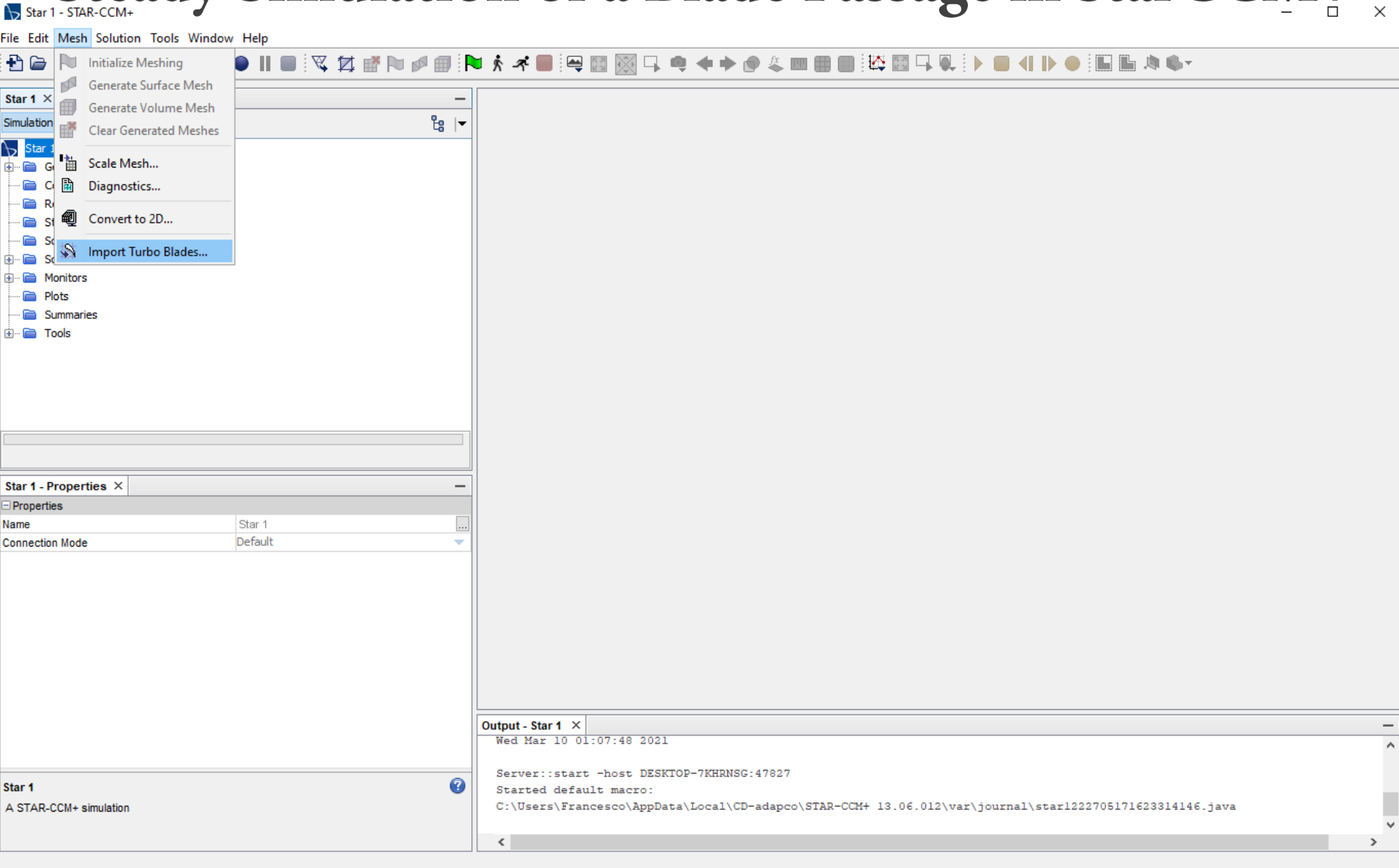
March 11, 2021, International Master of Turbulence



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+

The screenshot shows the 'Import turbo wizard geometry options' dialog box. The 'Geometry' tab is selected. The 'Number of blade rows' is set to 1, and 'Select current blade row' is also set to 1. The 'Current blade row is rotating' checkbox is checked, with a value of 17200.0 rpm. The 'Hub surface is rotating' checkbox is also checked. The 'Shroud surface is rotating' checkbox is unchecked. The 'Select Flow Axis' is set to +Z, and the 'Mesh Arrangement' is set to Annular. The 'Blade Geometry Definition/Surface Option' is set to 'Import blade geometry from external file'. The 'Select Blade Grid Type' is set to O-Grid. The 'Blade geometry file for blade row 1' is set to 'seTurbulence\sources\SingleRow\Rotor37.estg'. The 'File geometry in actual location (Not default +Z flow axis)' checkbox is unchecked. The 'Use open bottom of blade surface as hub section' and 'Use open top of blade surface as shroud section' checkboxes are both checked. The 'Number of blades around annulus' is set to 36. The 'Blade Geometry Units' are set to cm. At the bottom, there are buttons for 'Load Turbo Wizard Settings', 'Save Turbo Wizard Settings', 'OK', 'Cancel', and 'Help'.

1. Make sure that Number of blade rows is set to 1
2. Make sure that Select current blade row is set to 1
3. Activate Current blade row is rotating
4. Change the rotational speed to 17 200 rpm
5. Activate Hub surface is rotating
6. Set Select Flow Axis to +Z
7. Import the geometry in estg format
8. Select the two options about the BC
9. Select the amount of blades
10. Select the units in cm



Steady Simulation of a Blade Passage in StarCCM+

Import turbo wizard geometry file options

Number of blade rows: 1

Select current blade row: 1

☒ Current blade row is rotating: 17200.0 rpm

☒ Hub surface is rotating: Note: Turning on any surface rotation automatically turns on the default turbulence model settings and cannot be turned off, unless all are off

☐ Shroud surface is rotating

Select Flow Axis: +Z

Mesh Arrangement: Annular

Geometry | Meshing | Flowpath | Grid Settings | Boundary Conditions / Turbulence | Initial Conditions

Mesh generation method: Build 3D volume mesh using elliptical griddler

Load Turbo Wizard Settings Save Turbo Wizard Settings

OK Cancel Help



Steady Simulation of a Blade Passage in StarCCM+

The screenshot shows the 'Import turbo wizard geometry file options' dialog box. The 'Geometry' tab is selected. The 'Number of blade rows' is set to 1, and 'Select current blade row' is also 1. The 'Current blade row is rotating' checkbox is checked, with a value of 17200.0 rpm. The 'Hub surface is rotating' checkbox is also checked, with a note: 'Note: Turning on any surface rotation automatically turns on the default turbulence model settings and cannot be turned off, unless all are off'. The 'Shroud surface is rotating' checkbox is unchecked. The 'Select Flow Axis' is set to '+Z'. The 'Mesh Arrangement' is set to 'Annular'. The 'Geometry' tab is active, showing fields for 'Inlet boundary file for blade row 1' (seTurbulence\sources\SingleRow\inlet.bndy), 'Exit boundary file for blade row 1' (rseTurbulence\sources\SingleRow\exit.bndy), 'Generate clearance grid for blade row 1' (Shroud clearance grid), 'Define clearance streamline by:' (Setting fixed distance value), 'Number of clearance streamlines' (5), and 'Clearance distance' (0.0356 cm). The 'Hub boundary file' is cescio\Desktop\CourseTurbulence\sources\SingleRow\hub.bndy, and the 'Shroud boundary file' is ncesco\Desktop\CourseTurbulence\sources\SingleRow\tip.bndy. The 'Number of total streamlines' is 33, 'Continuous streamlines across blade rows' is unchecked, 'Initial hub streamline spacing' is 0.015 cm, and 'Initial shroud streamline spacing' is 0.015 cm. The 'Flow is in negative axis direction (Flip boundaries)' checkbox is unchecked. At the bottom, there are 'Load Turbo Wizard Settings', 'Save Turbo Wizard Settings', 'OK', 'Cancel', and 'Help' buttons.

1. Include the boundary file for the inlet
2. Include the boundary file for the outlet
3. Select “Shroud clearance grid”

Reduces y^+ values near the blade tip, which improves convergence of the turbulence model

4. Select “Setting fixed distance value”
5. Set “Number of clearance streamlines” to 5
6. Set “Clearance distance” to 0.0356 cm
7. Include the boundary file for the hub
8. Include the boundary file for the shroud
9. Set “Number of clearance streamlines” to 33

Controls the number of streamlines from the inlet to outlet boundaries, on which the mesh layers between hub and shroud are constructed

10. Set “Initial hub streamline spacing” to 0.015 cm
Defines the thickness of the mesh layers next to the hub surfaces

11. Set “Initial shroud streamline spacing” to 0.015cm

Defines the thickness of the mesh layers next to the shroud surfaces



Steady Simulation of a Blade Passage in StarCCM+

Import turbo wizard geometry file options

Number of blade rows: 1

Select current blade row: 1

☒ Current blade row is rotating: 17200.0 rpm

☒ Hub surface is rotating: Note: Turning on any surface rotation automatically turns on the default turbulence model settings and cannot be turned off, unless all are off

☐ Shroud surface is rotating

Select Flow Axis: +Z

Mesh Arrangement: Annular

Geometry | Meshing | Flowpath | **Grid Settings** | Boundary Conditions / Turbulence | Initial Conditions

Distribution | Number Of Points | Block Boundary Settings

User Defined

☒ Initial Clustering, Leading Edge (DS_LE): 0.003 cm

☒ Initial Clustering, Trailing Edge (DS_TE): 0.003 cm

☒ Near Wall (Airfoil) Spacing (DS_IN): 0.0015 cm

☒ Periodic Boundary Spacing (DS_OUT): 0.03 cm

☐ Clearance grid apex point % spacing, leading edge (DS_LECLR)

☐ Clearance grid apex point % spacing, trailing edge (DS_TECLR)

☒ Allow turbo wizard to move airfoil surface points to preserve grid orthogonality

Number of Dynamic Orthogonality Iterations: 8

Parameter Information

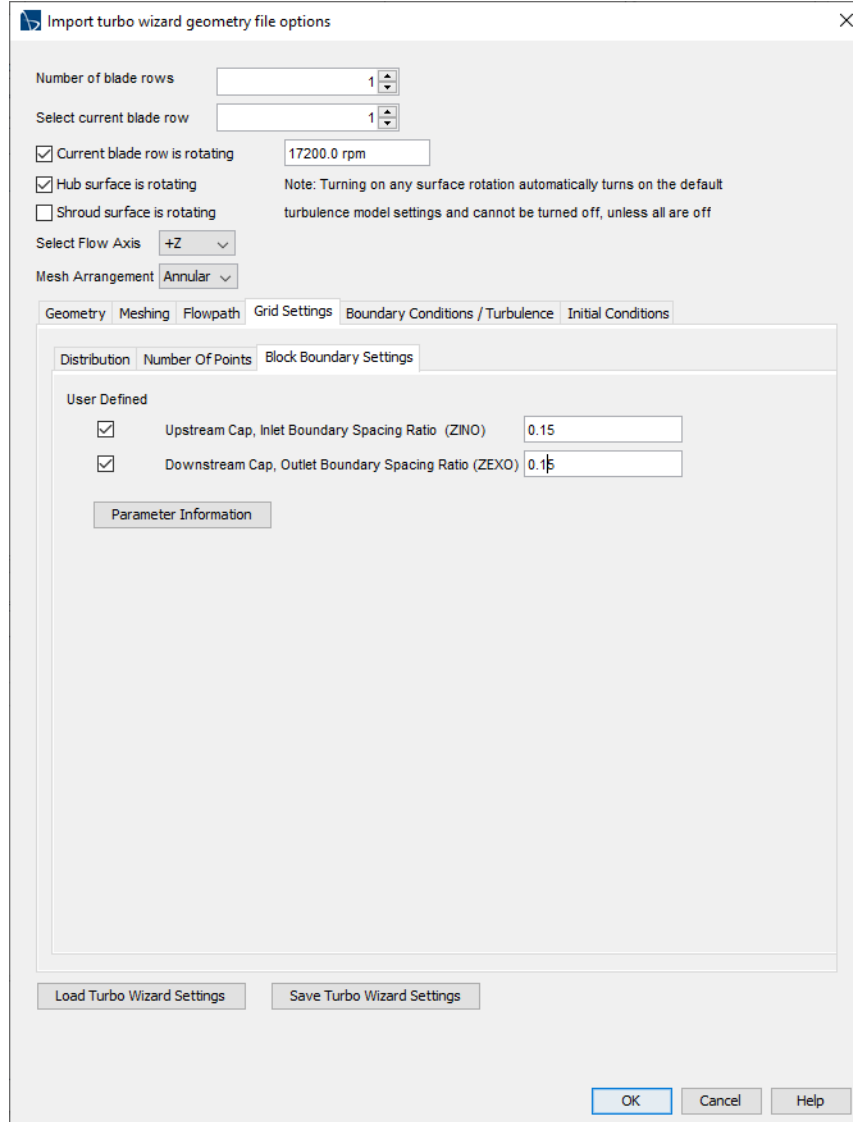
Load Turbo Wizard Settings | Save Turbo Wizard Settings

OK | Cancel | Help

1. Activate Initial Clustering, Leading Edge and set it to 0.003 cm
2. Activate Initial Clustering, Trailing Edge and set it to 0.003 cm
3. Activate Near Wall (Airfoil) Spacing and set it to 0.0015 cm
4. Activate Periodic Boundary Spacing and set it to 0.03cm
5. Activate the grid orthogonality option and set "Number of Dynamic Orthogonality Iterations" to 8



Steady Simulation of a Blade Passage in StarCCM+



The image shows the 'Import turbo wizard geometry file options' dialog box in StarCCM+. The 'Geometry' tab is selected, and the 'Block Boundary Settings' sub-tab is active. The 'User Defined' section is expanded, showing the 'Upstream Cap, Inlet Boundary Spacing Ratio (ZINO)' and 'Downstream Cap, Outlet Boundary Spacing Ratio (ZEXO)' both set to 0.15. The 'Current blade row is rotating' checkbox is checked, and the 'Hub surface is rotating' checkbox is also checked. The 'Shroud surface is rotating' checkbox is unchecked. The 'Select Flow Axis' is set to '+Z', and the 'Mesh Arrangement' is set to 'Annular'. The 'Number of blade rows' is set to 1, and the 'Select current blade row' is set to 1. The 'Current blade row is rotating' speed is set to 17200.0 rpm. The 'Load Turbo Wizard Settings' and 'Save Turbo Wizard Settings' buttons are at the bottom left, and the 'OK', 'Cancel', and 'Help' buttons are at the bottom right.

Import turbo wizard geometry file options

Number of blade rows: 1

Select current blade row: 1

☒ Current blade row is rotating: 17200.0 rpm

☒ Hub surface is rotating: Note: Turning on any surface rotation automatically turns on the default turbulence model settings and cannot be turned off, unless all are off

☐ Shroud surface is rotating

Select Flow Axis: +Z

Mesh Arrangement: Annular

Geometry | Meshing | Flowpath | Grid Settings | Boundary Conditions / Turbulence | Initial Conditions

Distribution | Number Of Points | Block Boundary Settings

User Defined

☒ Upstream Cap, Inlet Boundary Spacing Ratio (ZINO): 0.15

☒ Downstream Cap, Outlet Boundary Spacing Ratio (ZEXO): 0.15

Parameter Information

Load Turbo Wizard Settings | Save Turbo Wizard Settings

OK | Cancel | Help

1. Activate ZINO and set it to 0.15
2. Activate ZEXO and set it to 0.15



Steady Simulation of a Blade Passage in StarCCM+

Import turbo wizard geometry file options

Number of blade rows: 1

Select current blade row: 1

☒ Current blade row is rotating: 17200.0 rpm

☒ Hub surface is rotating: Note: Turning on any surface rotation automatically turns on the default turbulence model settings and cannot be turned off, unless all are off

☐ Shroud surface is rotating

Select Flow Axis: +Z

Mesh Arrangement: Annular

Geometry | Meshing | Flowpath | Grid Settings | Boundary Conditions / Turbulence | Initial Conditions

☒ Automatically set-up default turbulence models?

Default turbulence model uses K-Omega Turbulence with SST (Menter) K-Omega and All y+ Wall Treatment

Stagnation Inlet | Pressure Outlet

Constant static pressure: 5000.0 Pa

Constant static temperature: 300.0 K

Turbulence Intensity: 0.01

Turbulent Viscosity Ratio: 10.0

Load Turbo Wizard Settings | Save Turbo Wizard Settings

OK | Cancel | Help

Import turbo wizard geometry file options

Number of blade rows: 1

Select current blade row: 1

☒ Current blade row is rotating: 17200.0 rpm

☒ Hub surface is rotating: Note: Turning on any surface rotation automatically turns on the default turbulence model settings and cannot be turned off, unless all are off

☐ Shroud surface is rotating

Select Flow Axis: +Z

Mesh Arrangement: Annular

Geometry | Meshing | Flowpath | Grid Settings | Boundary Conditions / Turbulence | Initial Conditions

Constant static pressure: 0.0 Pa

Constant static temperature: 300.0 K

Initial velocity (m/sec): 0.0 X 0.0 Y 150.0 Z

Turbulence Intensity: 0.01

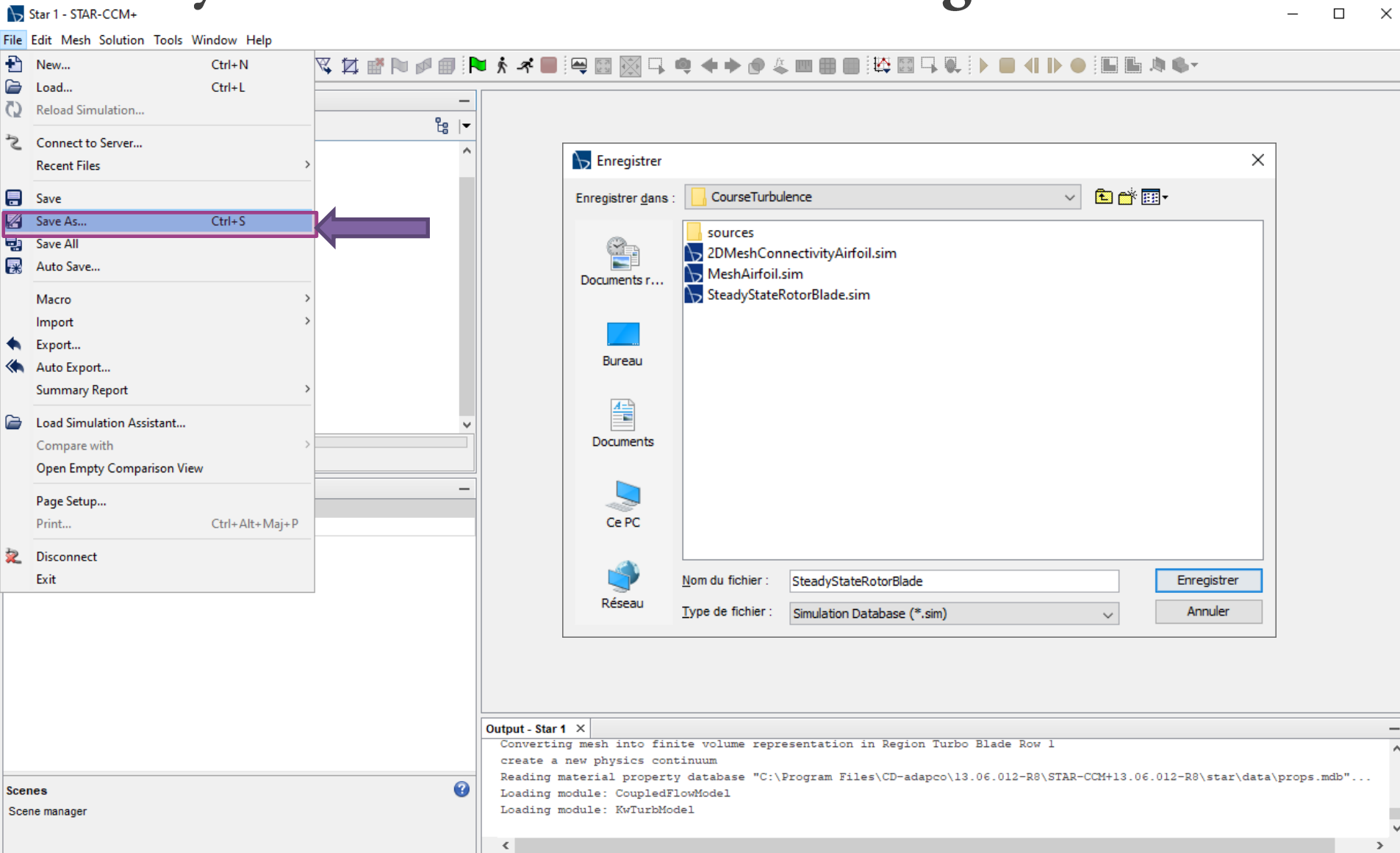
Turbulent Viscosity Ratio: 10.0

Save Turbo Wizard Settings

OK | Cancel | Help



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+

The image displays three screenshots of the StarCCM+ software interface, illustrating the setup for a steady simulation of a blade passage.

Left Screenshot: Shows the **Regions** tree structure. The **Turbo Blade Row 1** region is selected, and its **Physics Conditions** are being configured. The **Reference Frame Specification** property is highlighted.

Middle Screenshot: Shows the **Motion Specification - Properties** dialog box. The **Motion** is set to **Stationary**, and the **Reference Frame** is set to **Rotating in Turbo Blade Ro...**.

Right Screenshot: Shows the **Tools** tree structure. The **Rotating in Turbo Blade Row 1** tool is selected, and its **Properties** are displayed in the **Rotating in Turbo Blade Row 1 - Properties** dialog box.

Reference Frame Specification - Properties

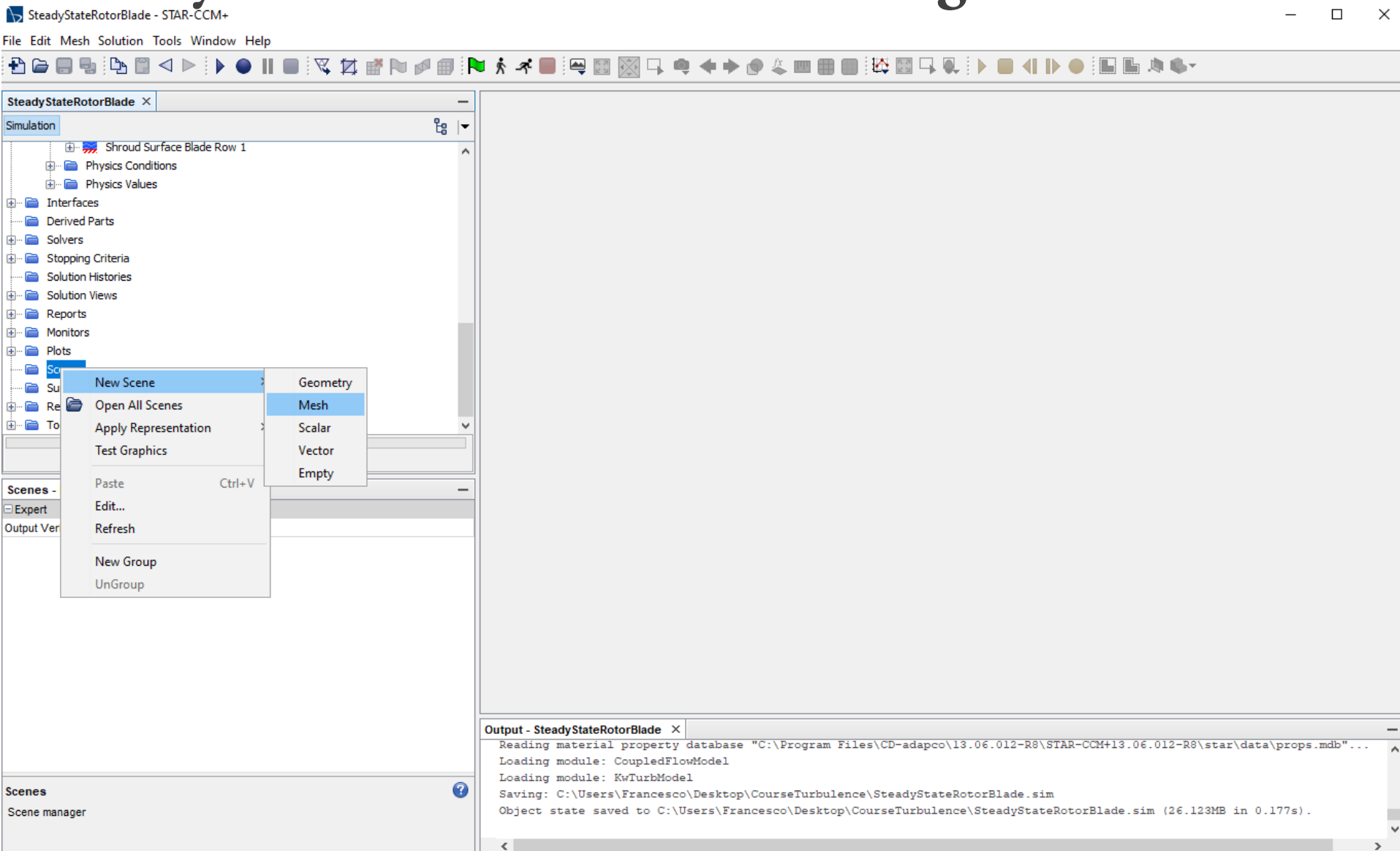
Properties	
Option	Region Reference Frame
	Lab Frame
	Region Reference Frame
	Local Reference Frame

Rotating in Turbo Blade Row 1 - Properties

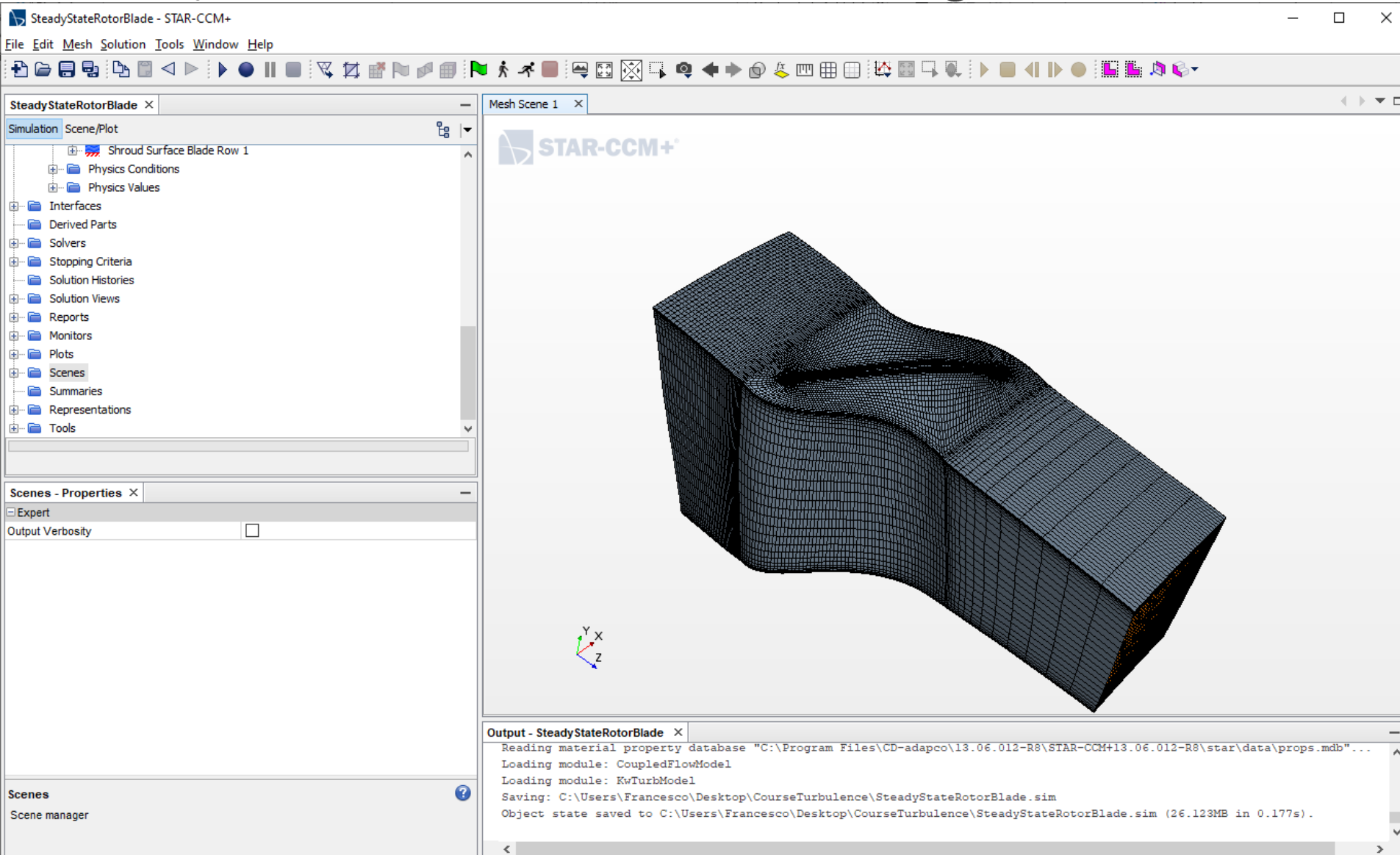
Properties	
Axis Direction	[0.0, 0.0, 1.0]
Coordinate System	Laboratory
Axis Origin	[0.0, 0.0, 0.0] m
Rotation Rate	1801.1798095703125 ra...



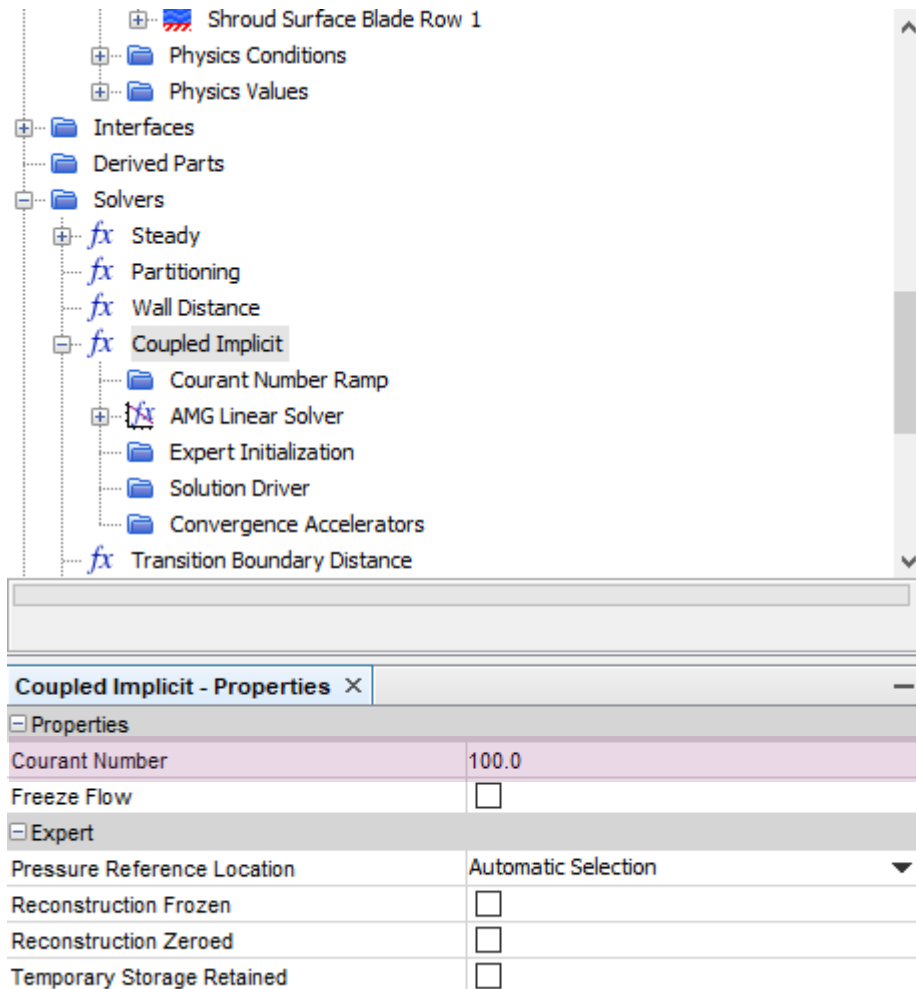
Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+

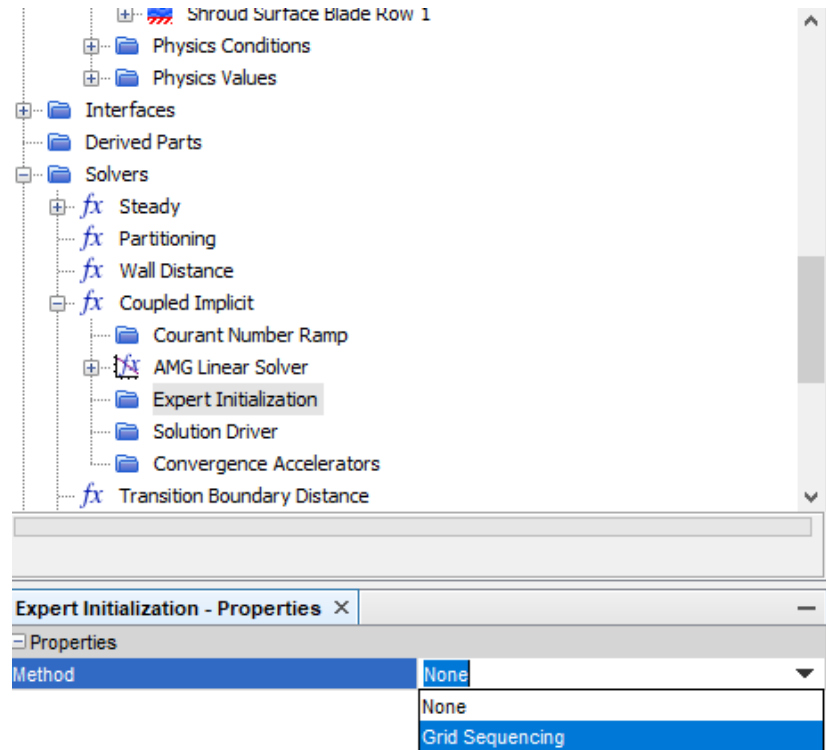


The left panel shows the StarCCM+ tree view with the following structure:

- Shroud Surface Blade Row 1
- Physics Conditions
- Physics Values
- Interfaces
- Derived Parts
- Solvers
 - Steady
 - Partitioning
 - Wall Distance
 - Coupled Implicit
 - Courant Number Ramp
 - AMG Linear Solver
 - Expert Initialization
 - Solution Driver
 - Convergence Accelerators
 - Transition Boundary Distance

The bottom panel shows the **Coupled Implicit - Properties** dialog box:

Coupled Implicit - Properties	
Properties	
Courant Number	100.0
Freeze Flow	☐
Expert	
Pressure Reference Location	Automatic Selection
Reconstruction Frozen	☐
Reconstruction Zeroed	☐
Temporary Storage Retained	☐



The right panel shows the StarCCM+ tree view with the following structure:

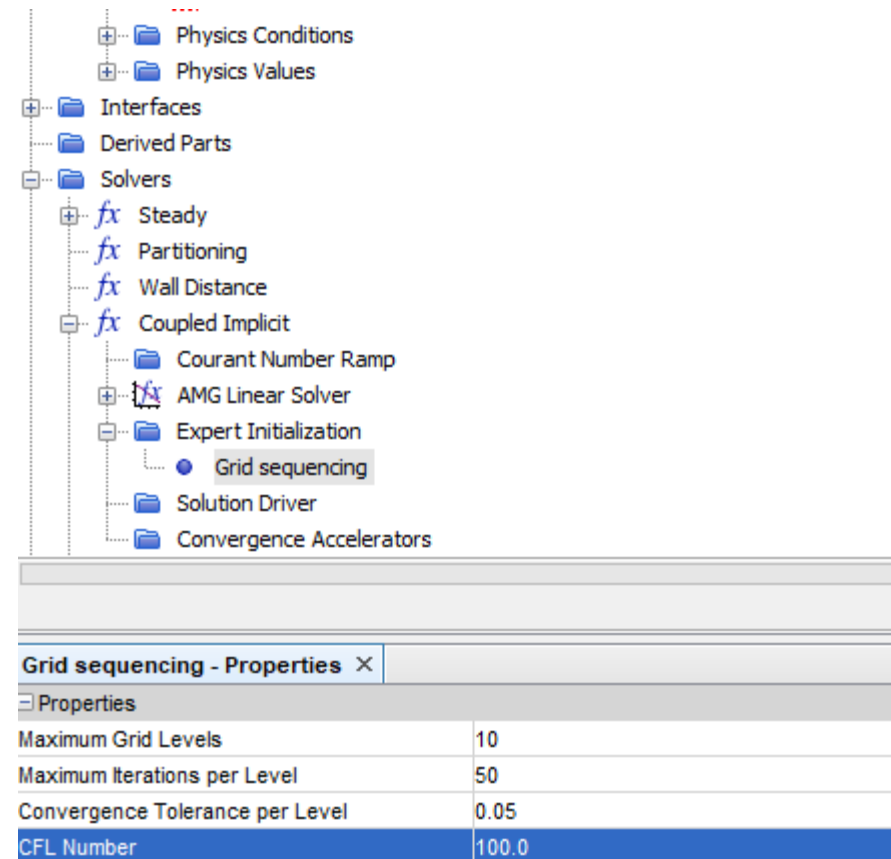
- Shroud Surface Blade Row 1
- Physics Conditions
- Physics Values
- Interfaces
- Derived Parts
- Solvers
 - Steady
 - Partitioning
 - Wall Distance
 - Coupled Implicit
 - Courant Number Ramp
 - AMG Linear Solver
 - Expert Initialization
 - Solution Driver
 - Convergence Accelerators
 - Transition Boundary Distance

The bottom panel shows the **Expert Initialization - Properties** dialog box:

Expert Initialization - Properties	
Properties	
Method	None
	None
	Grid Sequencing



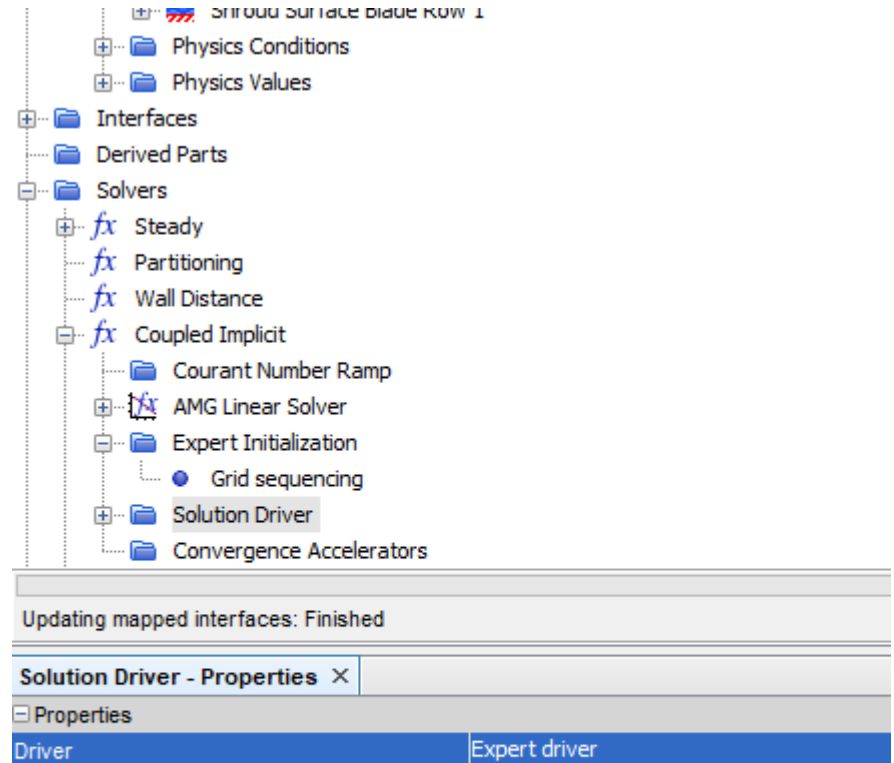
Steady Simulation of a Blade Passage in StarCCM+



The left panel shows the StarCCM+ tree view. The 'Solvers' folder is expanded, showing 'Steady', 'Partitioning', 'Wall Distance', 'Coupled Implicit', 'Courant Number Ramp', 'AMG Linear Solver', 'Expert Initialization', 'Grid sequencing', 'Solution Driver', and 'Convergence Accelerators'. The 'Grid sequencing' node is selected.

Below the tree view, the 'Grid sequencing - Properties' dialog is open, showing the following settings:

Grid sequencing - Properties	
Properties	
Maximum Grid Levels	10
Maximum Iterations per Level	50
Convergence Tolerance per Level	0.05
CFL Number	100.0



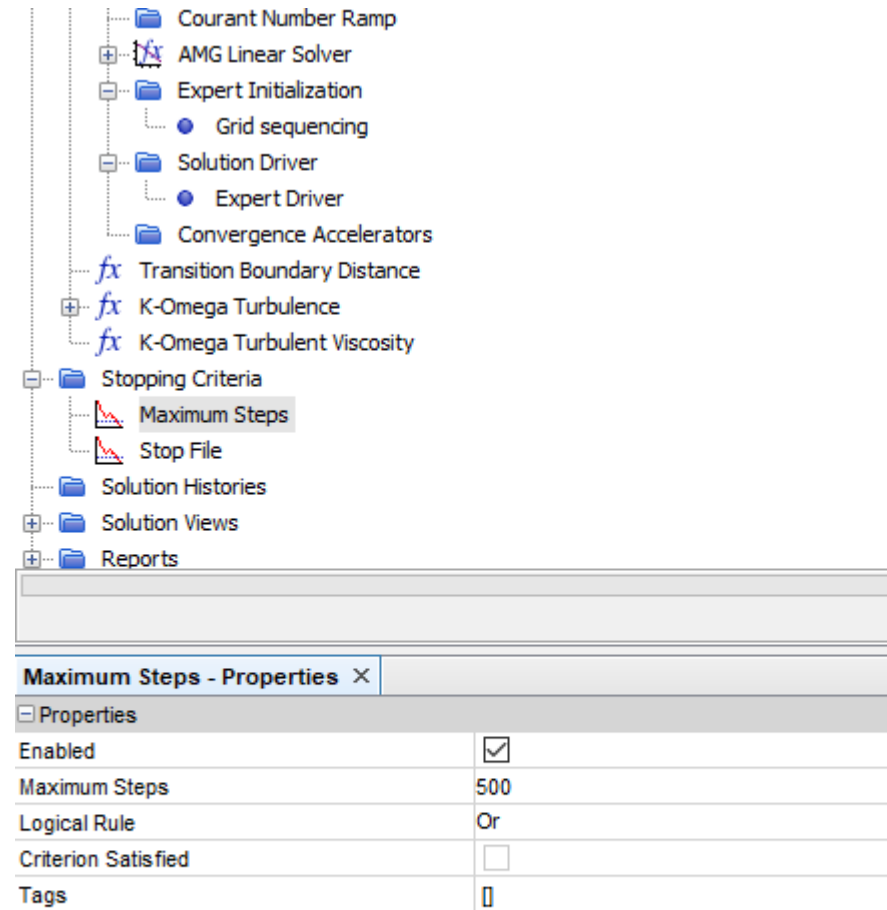
The right panel shows the StarCCM+ tree view. The 'Solvers' folder is expanded, showing 'Steady', 'Partitioning', 'Wall Distance', 'Coupled Implicit', 'Courant Number Ramp', 'AMG Linear Solver', 'Expert Initialization', 'Grid sequencing', 'Solution Driver', and 'Convergence Accelerators'. The 'Solution Driver' node is selected.

Below the tree view, the 'Solution Driver - Properties' dialog is open, showing the following settings:

Solution Driver - Properties	
Properties	
Driver	Expert driver

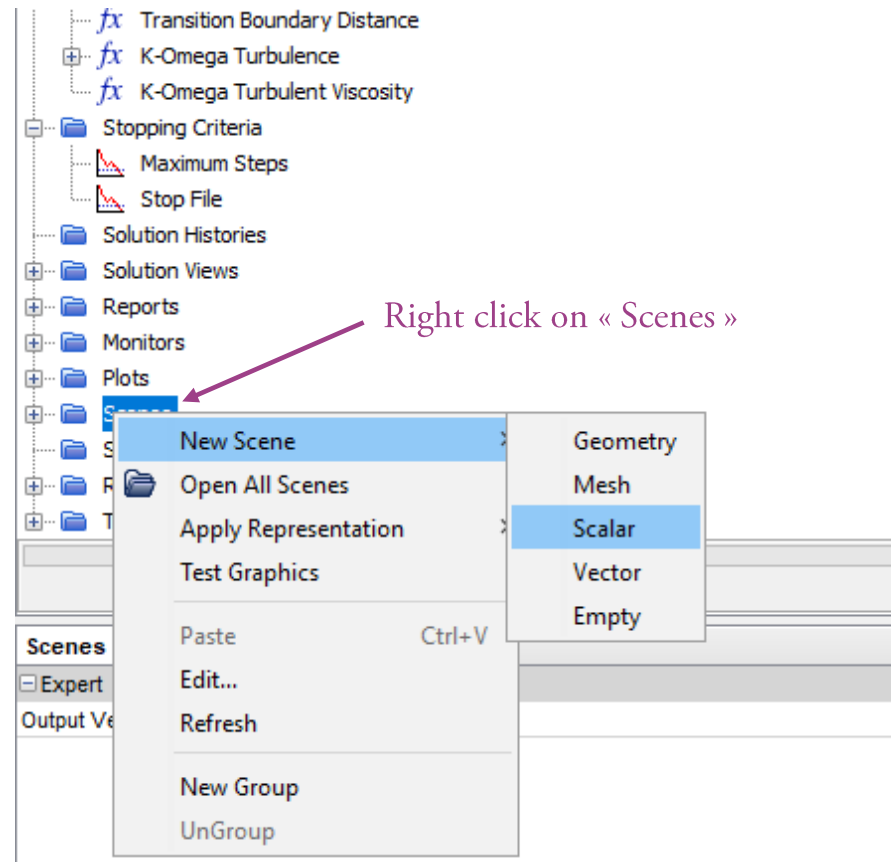


Steady Simulation of a Blade Passage in StarCCM+



The left panel shows the StarCCM+ simulation tree. The 'Stopping Criteria' folder is expanded, showing 'Maximum Steps' and 'Stop File'. The 'Maximum Steps' property is highlighted. Below the tree, the 'Maximum Steps - Properties' panel is visible, showing the 'Properties' tab.

Maximum Steps - Properties	
Properties	
Enabled	☒
Maximum Steps	500
Logical Rule	Or
Criterion Satisfied	☐
Tags	



The right panel shows the StarCCM+ simulation tree. The 'Scenes' folder is highlighted, and a context menu is open. A purple arrow points to the 'Scenes' folder with the text 'Right click on « Scenes »'. The context menu options are: New Scene, Open All Scenes, Apply Representation, Test Graphics, Paste (Ctrl+V), Edit..., Refresh, New Group, and UnGroup. A secondary menu is also visible, showing: Geometry, Mesh, Scalar (highlighted), Vector, and Empty.



SteadyStateRotorBlade - STAR-CCM+



Steady Simulation of a Blade Passage in StarCCM+

SteadyStateRotorBlade - STAR-CCM+

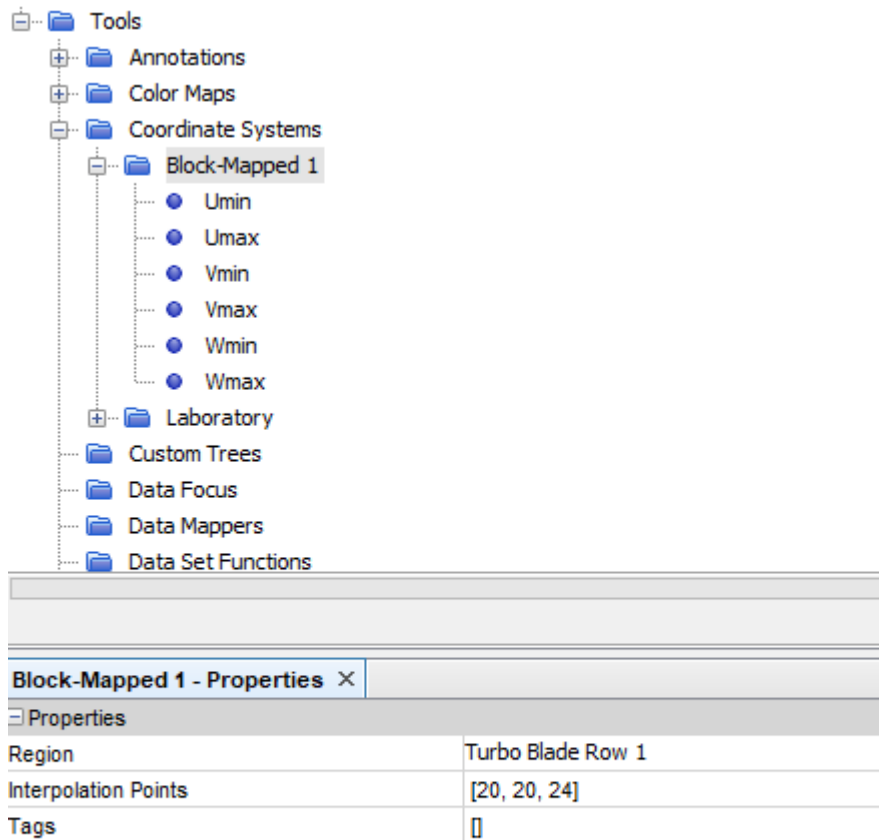
File Edit Mesh Solution Tools Window Help

The screenshot displays the StarCCM+ software interface. On the left, the 'Tools' tree is expanded, and the 'Coordinate Systems' menu is open, showing options like 'New Block-Mapped', 'Paste', and 'Refresh'. A red arrow points to the 'Coordinate Systems' menu with the text 'Right click on « Coordinate Systems »'. Below this, the 'Coordinate Systems - Properties' panel is visible, showing '<No Properties>'. The main 3D view shows a wireframe model of a blade passage. A coordinate system (X, Y, Z) is visible in the bottom left of the 3D view. A blue bar at the bottom of the 3D view is labeled 'Relative Mach Number'. The bottom panel shows the 'Output - SteadyStateRotorBlade' window with the following text:

```
Reading material property database "C:\Program Files\CD-adapco\13.06.012-R8\STAR-CCM+13.06.012-R8\star\data\props.mdb"...  
Loading module: CoupledFlowModel  
Loading module: KwTurbModel  
Saving: C:\Users\Francesco\Desktop\CourseTurbulence\SteadyStateRotorBlade.sim  
Object state saved to C:\Users\Francesco\Desktop\CourseTurbulence\SteadyStateRotorBlade.sim (26.123MB in 0.177s).
```

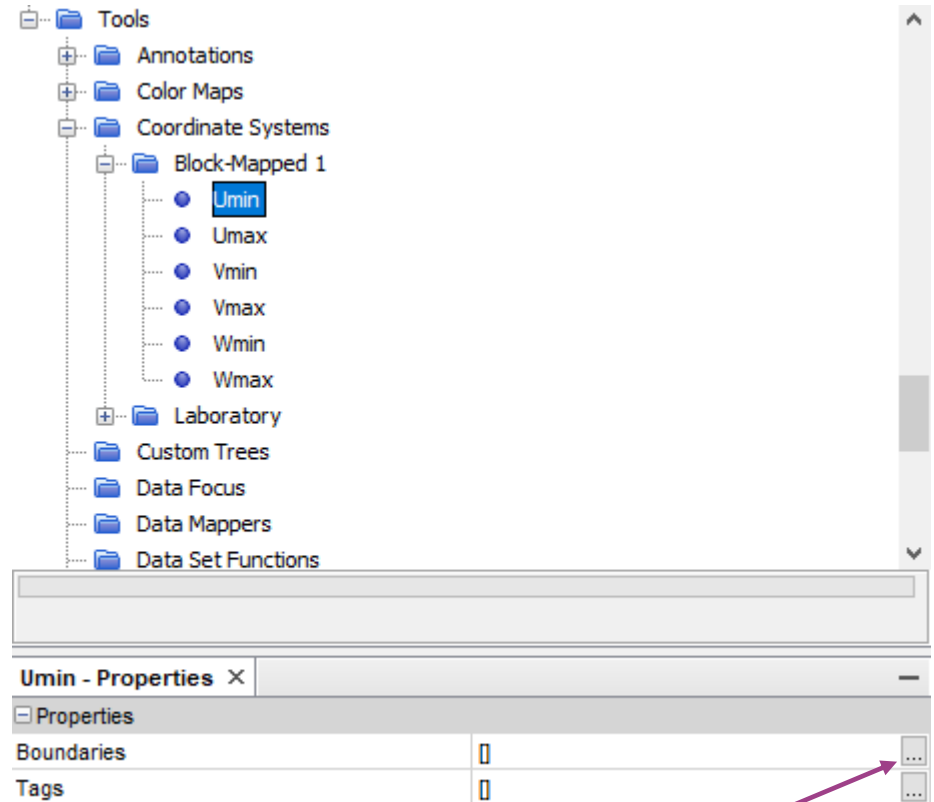


Steady Simulation of a Blade Passage in StarCCM+



Increases the resolution in the meridional direction

Umax: Right Blade Cyclic Blade Row 1
Vmax: Shroud Surface Blade Row 1
Wmax: Blade Outlet Blade Row 1

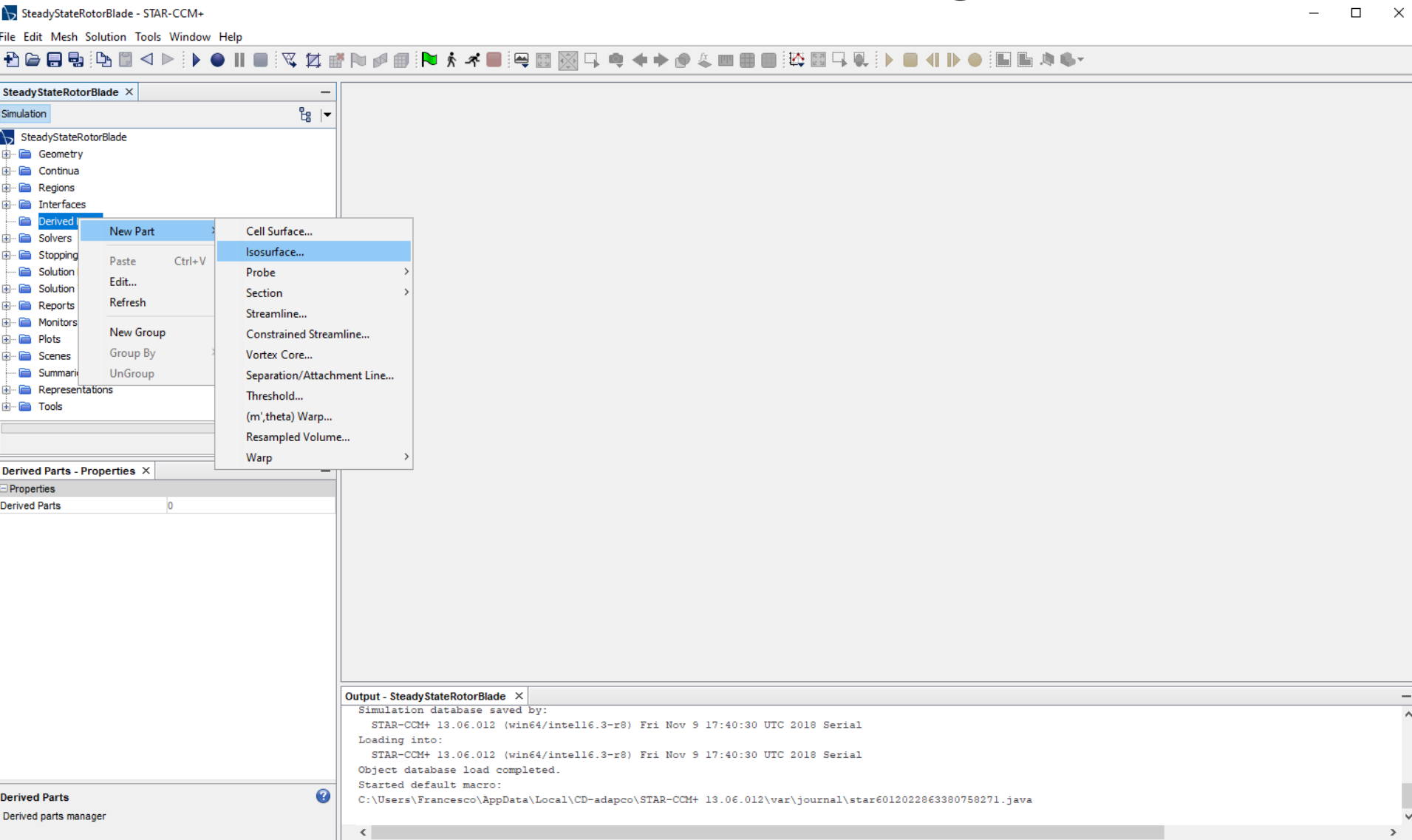


Click on the dots of « Boundaries »
and select « Left Blade Cyclic Blade Row 1 »

Umin: Left Blade Cyclic Blade Row 1
Vmin: Hub Surface Blade Row 1
Wmin: Blade Inlet Blade Row 1



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ software interface for a steady simulation of a blade passage. The main window is titled 'SteadyStateRotorBlade - STAR-CCM+'. The left sidebar shows the project hierarchy, including 'Simulation', 'Geometry', 'Continua', 'Regions', 'Interfaces', 'Derived Parts', 'Isosurface', 'Solvers', 'Stopping Criteria', 'Solution Histories', 'Solution Views', 'Reports', 'Monitors', 'Plots', 'Scenes', and 'Summaries'. The 'Isosurface' node is expanded, showing 'Value' and 'Properties'. The 'Isosurface - Properties' panel is open, showing 'Parts' (empty), 'Scalar Field' (set to 'Block-Mapped 1: Wmin to Wmax'), 'Mode' (set to 'Single'), and 'Tags' (empty). A purple arrow points to the 'Block-Mapped 1: Wmin to Wmax' selection. The 'Isosurface - Scalar Field' dialog box is open, showing a list of 'Field Functions'. The 'Block-Mapped 1: Wmin to Wmax' function is selected. The 'Output - SteadyStateRotorBlade' window at the bottom shows the simulation progress, including the message 'Simulation database saved by: STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial' and 'Object database load completed.'

Isosurface - Scalar Field

Field Functions

- ☐ f_{x_s} Absolute Pressure
- ☐ f_{x_s} Absolute Total Pressure
- ☒ f_{x_s} Area
- ☐ f_{x_s} Axial Velocity
- ☐ f_{x_s} Block-Mapped 1: Umin to Umax
- ☐ f_{x_s} Block-Mapped 1: Vmin to Vmax
- ☒ f_{x_s} Block-Mapped 1: Wmin to Wmax
- ☐ f_{x_s} Boundary Advection Heat Flux
- ☐ f_{x_s} Boundary Advection Heat Transfer
- ☐ f_{x_s} Boundary Circumferential Bin Coordinate
- ☐ f_{x_s} Boundary Circumferential Bin Index
- ☐ f_{x_s} Boundary Circumferential Bin Pitch
- ☐ f_{x_s} Boundary Circumferential Bin Theta
- ☐ f_{x_s} Boundary Conduction Heat Flux
- ☐ f_{x_s} Boundary Conduction Heat Transfer
- ☐ f_{x_s} Boundary Heat Flux
- ☐ f_{x_s} Boundary Heat Flux on External Side

1 of 231 selected

OK Cancel Help

Output - SteadyStateRotorBlade

Simulation database saved by:
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Loading into:
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Object database load completed.
Started default macro:
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java



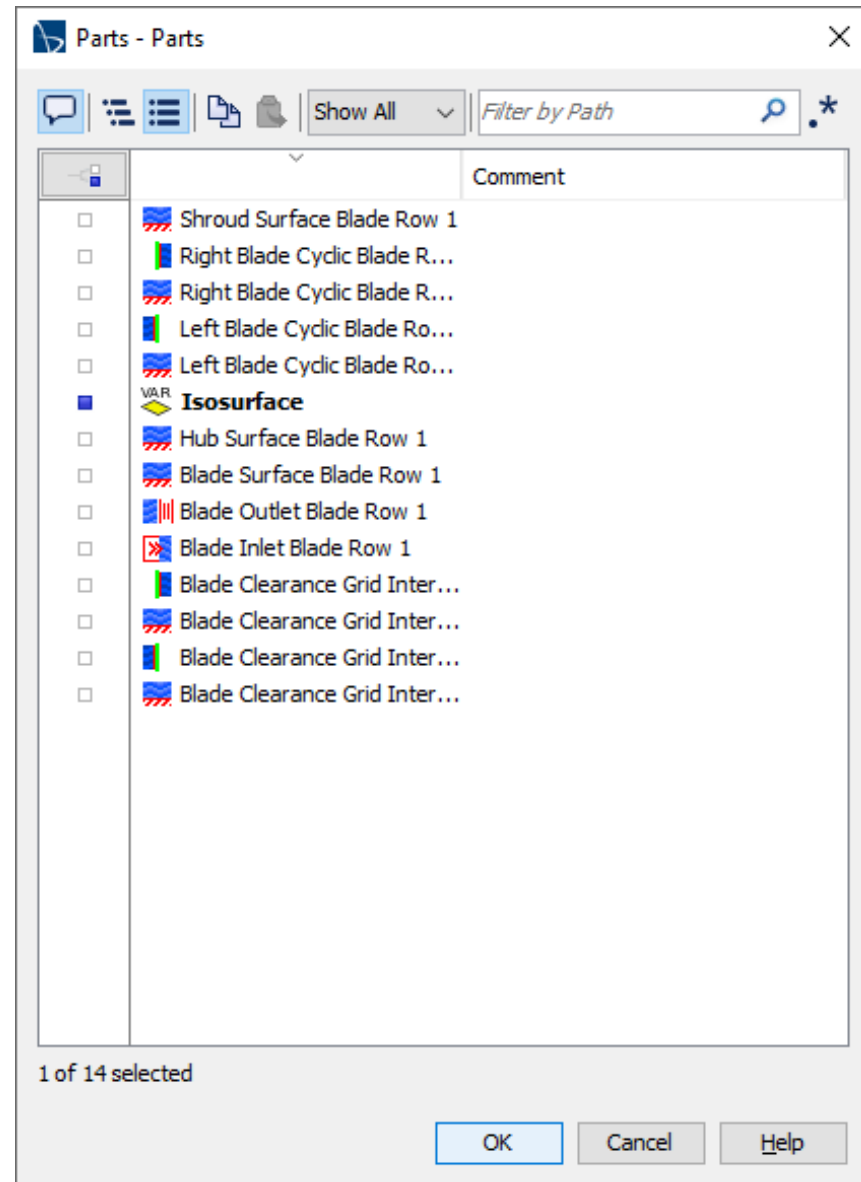
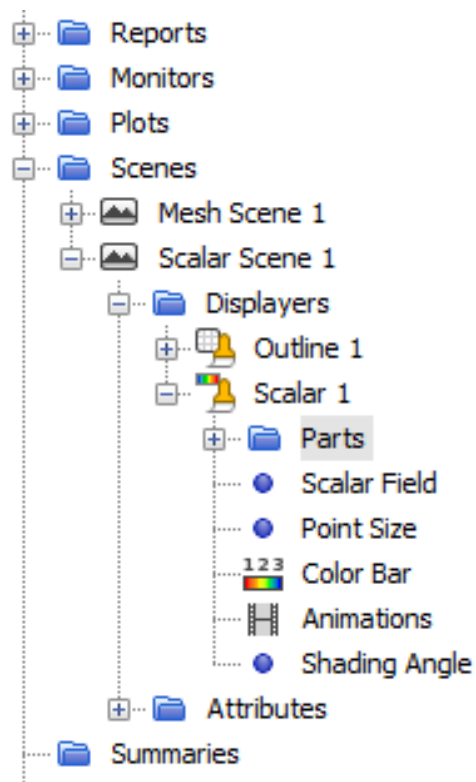
Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ user interface for a steady simulation of a blade passage. The main window is titled "SteadyStateRotorBlade - STAR-CCM+". The left sidebar shows the project hierarchy under the "Simulation" tab, including Geometry, Continua, Regions, Interfaces, Derived Parts, Isosurface, Value, Solvers, Stopping Criteria, Solution Histories, Solution Views, Reports, Monitors, Plots, Scenes, and Summaries. The "Value" property is selected, showing an "Isovalue" of 0.01. The bottom panel displays the "Output - SteadyStateRotorBlade" window, which contains the following text:

```
Simulation database saved by:  
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial  
Loading into:  
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial  
Object database load completed.  
Started default macro:  
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java
```



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ software interface for a steady simulation of a blade passage. The main window shows the 'Simulation' tree on the left, with 'Derived Parts' expanded. A context menu is open over 'Derived Parts', showing options like 'New Part', 'Cell Surface...', 'Isosurface...', 'Probe', 'Section', 'Streamline...', 'Constrained Streamline...', 'Vortex Core...', 'Separation/Attachment Line...', 'Threshold...', '(m',theta) Warp...', 'Resampled Volume...', and 'Warp'. The 'Isosurface - Scalar Field' dialog box is open in the foreground, showing a list of field functions. The 'Block-Mapped 1: Vmin to Vmax' function is selected. The 'Output - STAR-CCM+' window at the bottom shows the simulation progress and error messages.

SteadyStateRotorBlade - STAR-CCM+

File Edit Mesh Solution Tools Window Help

Simulation

Continua
Regions
Interfaces
Derived Parts
VAR
AxialV
Solvers
Stopping
Solution H
Solution V
Reports
Monitors
Plots
Scenes
Mesh Scene 1
Scalar Scene 1
Displayers
Outline 1

New Part
Paste Ctrl+V
Edit...
Refresh
New Group
Group By
UnGroup

Cell Surface...
Isosurface...
Probe
Section
Streamline...
Constrained Streamline...
Vortex Core...
Separation/Attachment Line...
Threshold...
(m',theta) Warp...
Resampled Volume...
Warp

Derived Parts - Properties

Properties
Derived Parts 1

Isosurface - Scalar Field

Show All Filter by Path

Field Functions

- ☐ f_x Absolute Pressure
- ☐ f_x Absolute Total Pressure
- ☒ f_x Area
- ☐ f_x Axial Velocity
- ☐ f_x Block-Mapped 1: Umin to Umax
- ☒ f_x Block-Mapped 1: Vmin to Vmax
- ☐ f_x Block-Mapped 1: Wmin to Wmax
- ☐ f_x Boundary Advection Heat Flux
- ☐ f_x Boundary Advection Heat Transfer
- ☐ f_x Boundary Circumferential Bin Coordinate
- ☐ f_x Boundary Circumferential Bin Index
- ☐ f_x Boundary Circumferential Bin Pitch
- ☐ f_x Boundary Circumferential Bin Theta
- ☐ f_x Boundary Conduction Heat Flux
- ☐ f_x Boundary Conduction Heat Transfer
- ☐ f_x Boundary Heat Flux
- ☐ f_x Boundary Heat Flux on External Side

1 of 231 selected

Output - STAR-CCM+

Loading...

STAR-CCM+ 13.06.012 (win64/intell16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Object database load completed.
Started default macro:
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java
Error in NodeCommunicator::flush()

Derived Parts
Derived parts manager



Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ user interface for a simulation titled "SteadyStateRotorBlade - STAR-CCM+". The interface is divided into several panels:

- Simulation Panel (Left):** Shows the project hierarchy. Under "Derived Parts", there are "AxialView" and "Isosurface", each with a "Value" property. The "Value" property for "Isosurface" is currently set to 0.5.
- Value - Properties Panel (Bottom Left):** Shows the "Isosurface Value" property.
- Output Panel (Bottom Right):** Displays the simulation log. The log shows the following text:

```
STAR-CCM+ 13.06.012 (win64/intell6.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Loading into:
STAR-CCM+ 13.06.012 (win64/intell6.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Object database load completed.
Started default macro:
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java
Error in NodeCommunicator::flush()
```



Steady Simulation of a Blade Passage in StarCCM+

SteadyStateRotorBlade - STAR-CCM+

File Edit Mesh Solution Tools Window Help

Simulation

Solution Views

Reports

Monitors

Plots

Scenes

Mesh Scene 1

Scalar Scene 1

Displayers

New Displayer

Edit...

Paste Ctrl+V

Geometry

Scalar

Vector

Streamline

Point Size

Color Bar

Animations

Shading Angle

Attributes

Summaries

Displayers - Properties

<No Properties>

Displayers

Displayer manager

Output - SteadyStateRotorBlade

STAR-CCM+ 13.06.012 (win64/intell16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial

Loading into:

STAR-CCM+ 13.06.012 (win64/intell16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial

Object database load completed.

Started default macro:

C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java

Error in NodeCommunicator::flush()



Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ interface for a steady simulation of a blade passage. The main window shows the 'SteadyStateRotorBlade - STAR-CCM+' project. The 'Simulation' tree on the left includes 'Mesh Scene 1', 'Scalar Scene 1', 'Displayers', 'Outline 1', 'Scalar 1', 'Parts', 'Scalar Field', 'Point Size', 'Color Bar', 'Animations', 'Shading Angle', 'Scalar 2', and 'Parts'. The 'Parts - Properties' panel at the bottom left shows the 'Parts' icon, which is highlighted by a purple arrow. The 'Parts - Parts' dialog box is open, showing a list of parts with checkboxes. The parts listed are: 'Shroud Surface Blade Row 1', 'Right Blade Cyclic Blade R...', 'Right Blade Cyclic Blade R...', 'Left Blade Cyclic Blade Ro...', 'Left Blade Cyclic Blade Ro...', 'Hub Surface Blade Row 1', 'Blade2BladeView' (VAR), 'Blade Surface Blade Row 1', 'Blade Outlet Blade Row 1', 'Blade Inlet Blade Row 1', 'Blade Clearance Grid Inter...', 'Blade Clearance Grid Inter...', 'Blade Clearance Grid Inter...', and 'AxialView' (VAR). The '1 of 15 selected' status is shown at the bottom of the dialog. The 'Output - SteadyStateRotorBlade' panel at the bottom right shows the simulation progress, including the text: 'STAR-CCM+ 13.06.012', 'Loading into:', 'STAR-CCM+ 13.06.012', 'Object database load co', 'Started default macro:', 'C:\Users\Francesco\AppData', and 'Error in NodeCommunicat'.

SteadyStateRotorBlade - STAR-CCM+

File Edit Mesh Solution Tools Window Help

SteadyStateRotorBlade x

Simulation

Mesh Scene 1

Scalar Scene 1

Displayers

Outline 1

Scalar 1

Parts

Scalar Field

Point Size

Color Bar

Animations

Shading Angle

Scalar 2

Parts

Scalar Field

Point Size

Color Bar

Animations

Shading Angle

Parts - Properties x

Properties

Parts

Output - SteadyStateRotorBlade

STAR-CCM+ 13.06.012

Loading into:

STAR-CCM+ 13.06.012

Object database load co

Started default macro:

C:\Users\Francesco\AppData

Error in NodeCommunicat

Parts - Parts

Show All Filter by Path

Comment

Shroud Surface Blade Row 1

Right Blade Cyclic Blade R...

Right Blade Cyclic Blade R...

Left Blade Cyclic Blade Ro...

Left Blade Cyclic Blade Ro...

Hub Surface Blade Row 1

VAR Blade2BladeView

Blade Surface Blade Row 1

Blade Outlet Blade Row 1

Blade Inlet Blade Row 1

Blade Clearance Grid Inter...

Blade Clearance Grid Inter...

Blade Clearance Grid Inter...

VAR AxialView

1 of 15 selected

OK Cancel Help

Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ software interface for a steady simulation of a blade passage. The main window shows the 'Isosurface - Scalar Field' dialog box, which is used to define the scalar field for the isosurface. The 'Field Functions' list includes various options, with 'Block-Mapped 1: Umin to Umax' selected. A purple arrow points to the 'Isosurface' property in the left sidebar, indicating the selected field function.

Isosurface - Properties

Property	Value
Parts	
Scalar Field	<Select Function>
Mode	Single
Tags	

Isosurface - Properties

Isosurface Part

Output - SteadyStateRotorBlade

```
STAR-CCM+ 13.06.012 (win64/intell16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Loading into:
STAR-CCM+ 13.06.012 (win64/intell16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Object database load completed.
Started default macro:
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java
Error in NodeCommunicator::flush()
```



Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ software interface for a steady simulation of a blade passage. The main window is titled "SteadyStateRotorBlade - STAR-CCM+". The left sidebar shows the project tree with the following structure:

- SteadyStateRotorBlade
 - Geometry
 - Continua
 - Regions
 - Interfaces
 - Derived Parts
 - VAR
 - AxialView
 - Value
 - Blade2BladeView
 - Value
 - MeridionalView
 - Value
 - Solvers
 - Stopping Criteria
 - Solution Histories
 - Solution Views
 - Reports
 - Monitors

The "Value - Properties" panel shows the "Isovalue" property set to 0.025. The "Output - SteadyStateRotorBlade" panel displays the following log:

```
STAR-CCM+ 13.06.012 (win64/intell6.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Loading into:
STAR-CCM+ 13.06.012 (win64/intell6.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Object database load completed.
Started default macro:
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java
Error in NodeCommunicator::flush()
```

The bottom of the interface shows the "Isovalue" and "Isosurface Value" sections.



Steady Simulation of a Blade Passage in StarCCM+

SteadyStateRotorBlade - STAR-CCM+

File Edit Mesh Solution Tools Window Help

Simulation

- Plots
- Scenes
 - Mesh Scene 1
 - Scalar Scene 1
 - Display
 - New Displayer
 - Geometry
 - Scalar
 - Vector
 - Streamline
 - Edit...
 - Paste Ctrl+V
 - Point Size
 - Color Bar
 - Animations
 - Shading Angle
 - Scalar 2
 - Parts
 - Scalar Field
 - Point Size
 - Color Bar

Displays - Properties

<No Properties>

Output - SteadyStateRotorBlade

```
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Loading into:
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9 17:40:30 UTC 2018 Serial
Object database load completed.
Started default macro:
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+ 13.06.012\var\journal\star6012022863380758271.java
Error in NodeCommunicator::flush()
```



Steady Simulation of a Blade Passage in StarCCM+

The screenshot displays the StarCCM+ software interface for a steady simulation of a blade passage. The main window is titled "SteadyStateRotorBlade - STAR-CCM+". The left sidebar shows a tree view of the simulation setup, including "Simulation", "Parts", "Scalar 2", "Scalar 3", and "Parts". The "Parts" panel is active, showing a list of parts. A purple arrow points to the "Parts" panel, with a tooltip that reads: "Click, or select and press CTRL-SPACE to open custom editor; right click for menu".

The "Parts - Parts" panel on the right lists the following parts:

- Shroud Surface Blade Row 1
- Right Blade Cyclic Blade R...
- Right Blade Cyclic Blade R...
- MeridionalView**
- Left Blade Cyclic Blade Ro...
- Left Blade Cyclic Blade Ro...
- Hub Surface Blade Row 1
- Blade2BladeView
- Blade Surface Blade Row 1
- Blade Outlet Blade Row 1
- Blade Inlet Blade Row 1
- Blade Clearance Grid Inter...
- Blade Clearance Grid Inter...
- Blade Clearance Grid Inter...
- Blade Clearance Grid Inter...
- AxialView

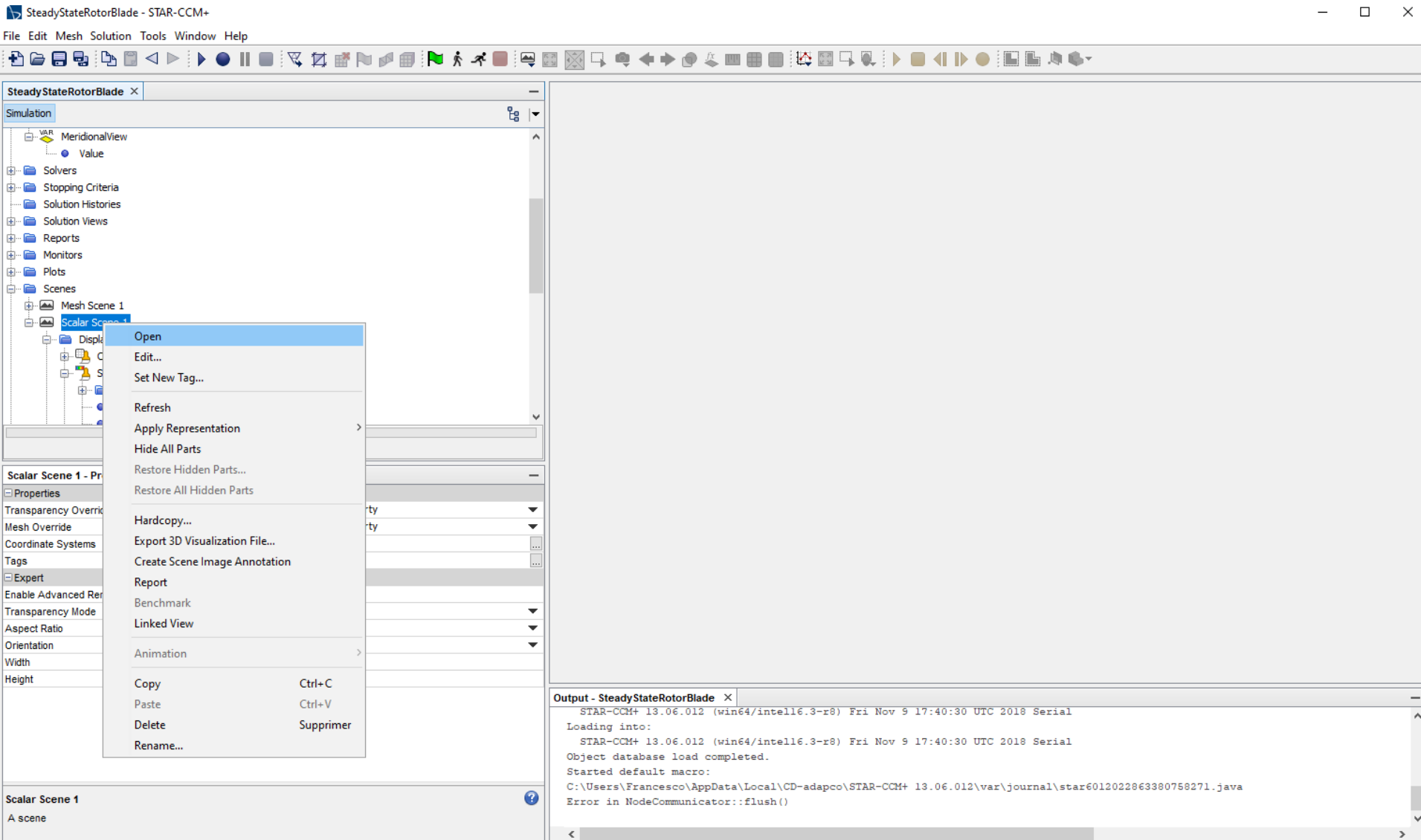
The "Output - SteadyStateRotorBlade" panel at the bottom shows the following text:

```
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9
Loading into:
STAR-CCM+ 13.06.012 (win64/intel16.3-r8) Fri Nov 9
Object database load completed.
Started default macro:
C:\Users\Francesco\AppData\Local\CD-adapco\STAR-CCM+
Error in NodeCommunicator::flush()
```

The "Parts" panel at the bottom left shows "Parts" and "Group of parts."



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+

SteadyStateRotorBlade - STAR-CCM+

File Edit Mesh Solution Tools Window Help

Simulation Scene/Plot

Scalar Scene 1

Displays

- Outline 1
 - Scalar 1
 - Parts
 - Scalar Field
 - Point Size
 - Color Bar
 - Animations
 - Shading Angle
- Scalar 2
- Scalar 3

Attributes

- Model Window
- Background Color
- View
- Update

Scalar 1 - Properties

Properties

Opacity	1.0
Contour Style	Automatic
Display Mesh	Automatic
Tags	Filled
Expert	Smooth Filled
Line Width	Smooth Filled + Lines
Point Size	Line
Smooth Shade	Smooth Blended
Ignore Clip	☐
Face Culling	None
Part Visibility	Use Part Visibility Property
Specular Highlight Mode	Material
Lighting	[0.0, 1.0, 0.0, 100.0]
Data Focus	None
Transform	Identity
Point Scale Mode	Pixels
Representation	Volume Mesh

Contour Style

Contour fill style

Scalar Scene 1 x Mesh Scene 1 x

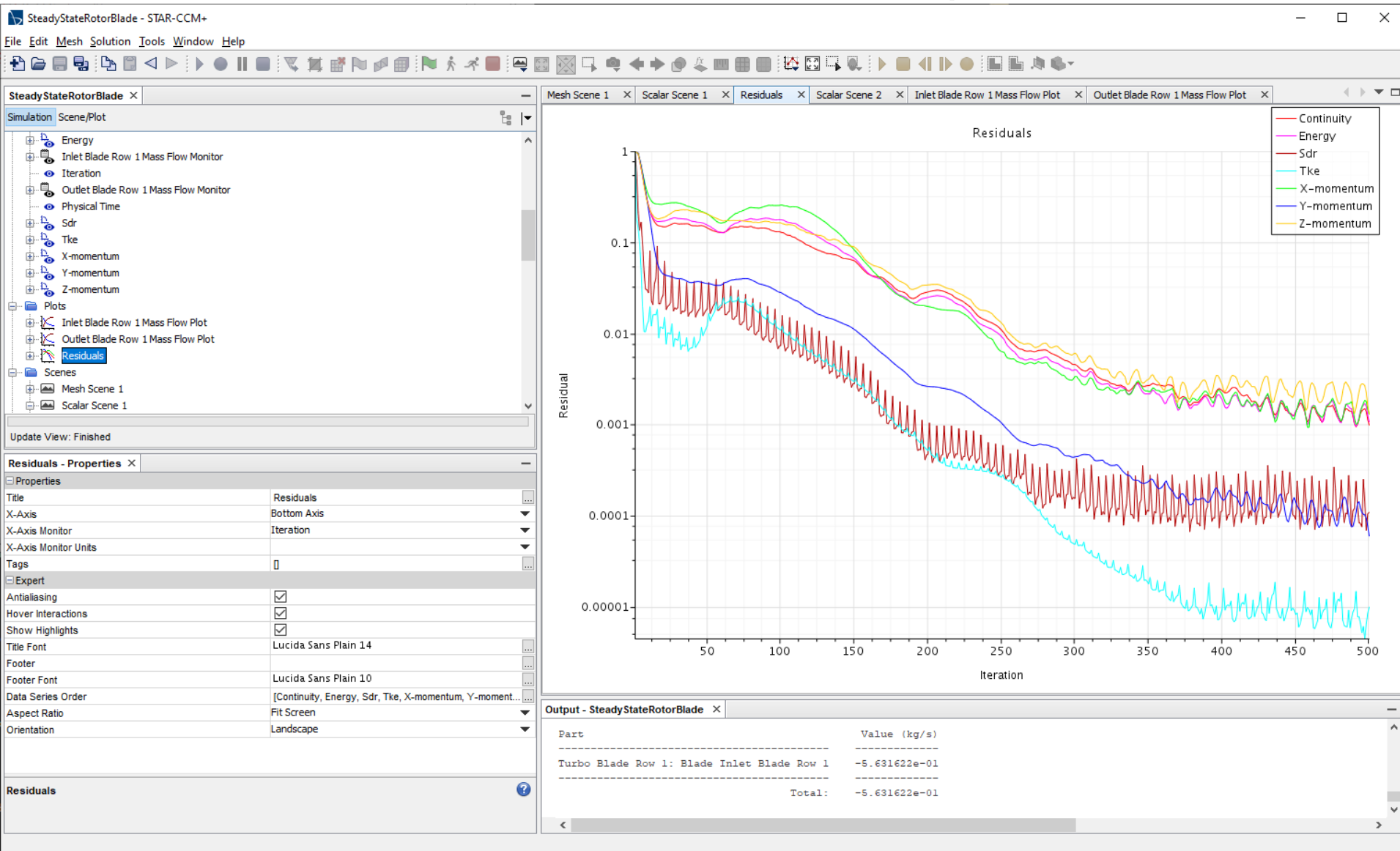
Initializing scene contents! Please wait...

Output - SteadyStateRotorBlade

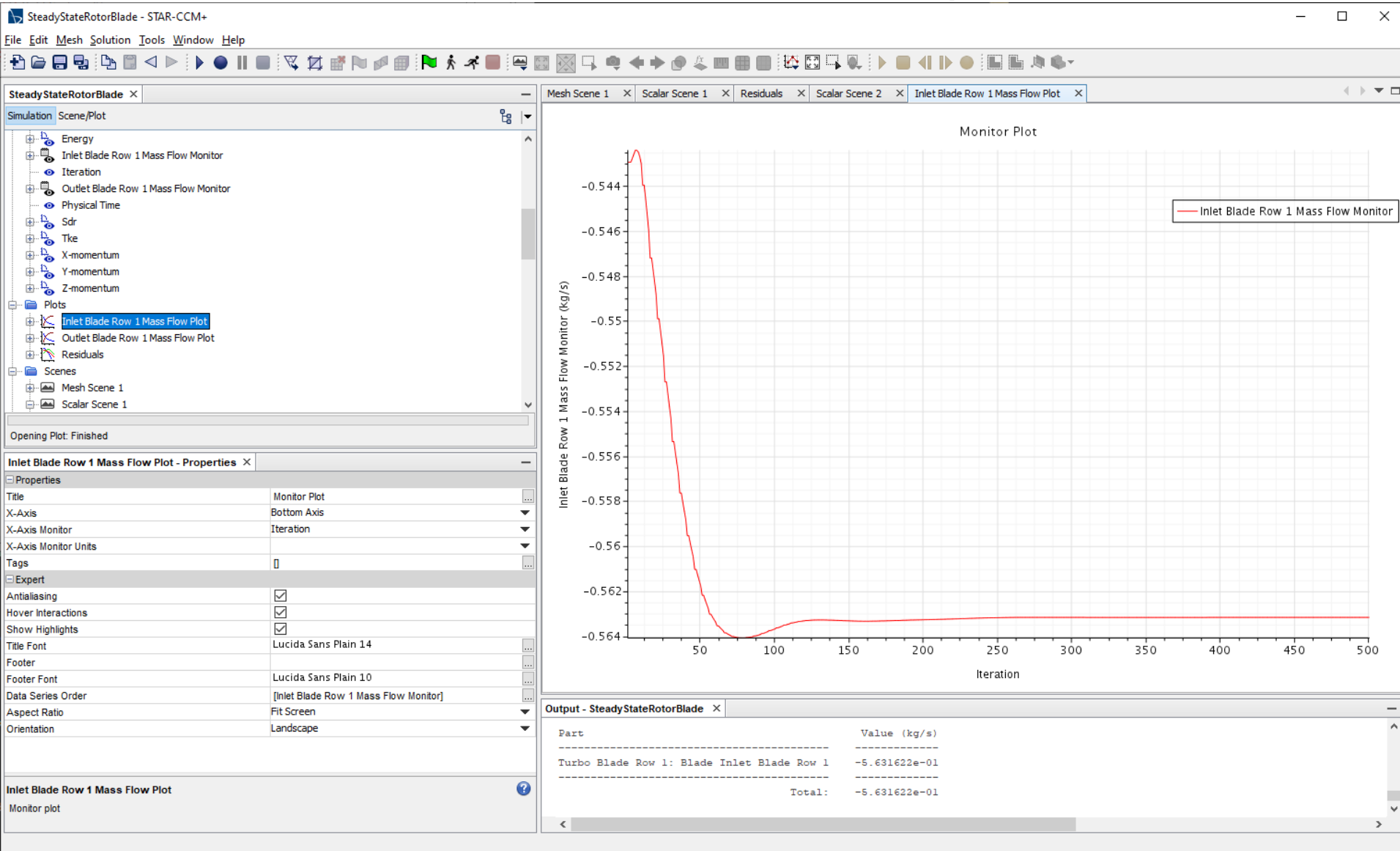
```
Error in NodeCommunicator::flush()
Loading/configuring connectivity (old/new partitions: 1|1)
Turbo Blade Row 1 (index 0): 231840 cells, 679304 faces, 248209 verts.
Configuring finished
Reading material property database "C:\Program Files\CD-adapco\13.06.012-R8\STAR-CCM+13.06.012-R8\star\data\props.mdb"...
Saving: C:\Users\Francesco\Desktop\CourseTurbulence\SteadyStateRotorBlade.sim
Object state saved to C:\Users\Francesco\Desktop\CourseTurbulence\SteadyStateRotorBlade.sim (26.23MB in 0.177s).
```



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+



Steady Simulation of a Blade Passage in StarCCM+

