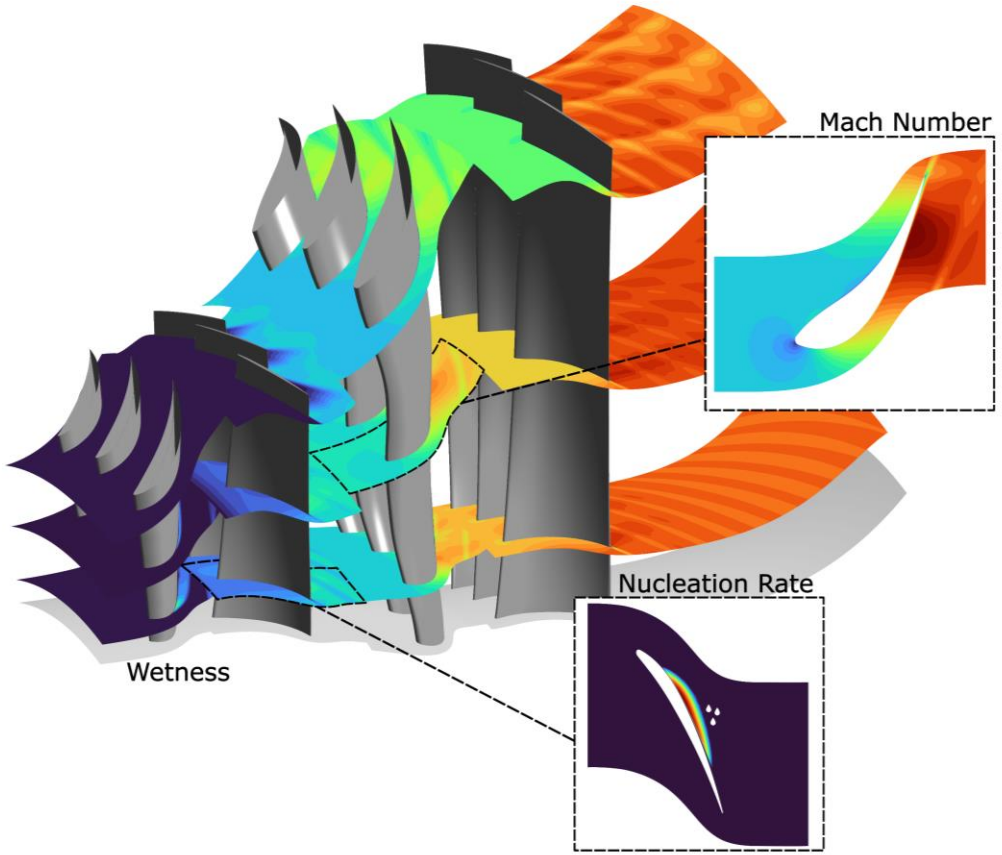
Best Practice for Steam Turbine Simulation on the Example of the STAC Test Case

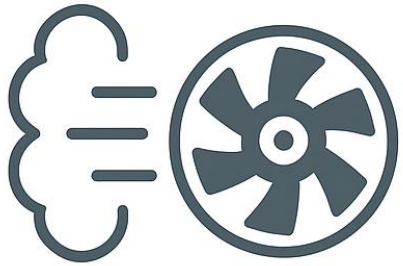
Guidelines for Ansys CFX

Table of Contents



Ansys part of SYNOPSYS®

Steam Turbines

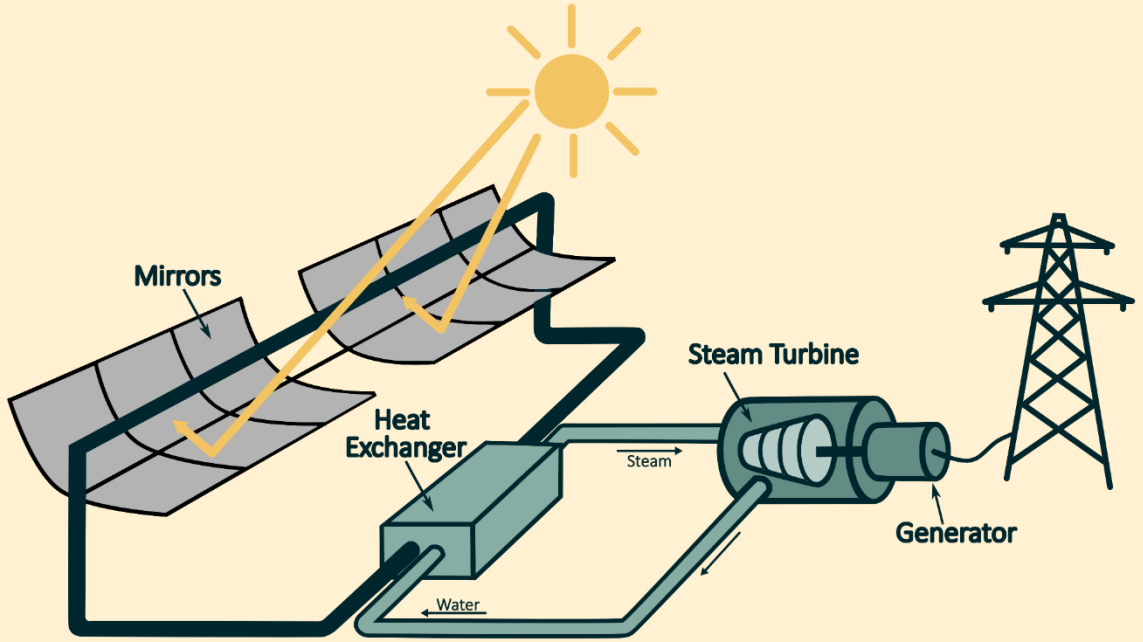


Thermal Energy of Steam → Mechanical Work

- 42.5% of the USA's total electricity generation (2022) [1]
- Core component in thermal power Plants
  - Coal, nuclear, concentrated solar power (CSP), biomass, geothermal, etc.

- Within a steam turbine condensation can occur leading to wet steam being present
- Water droplets can cause severe erosion to turbine blades at low-pressure stages (LP-stages)

→ Wet Steam Model for accurate modeling of these processes



[1] *https://www.eia.gov/energyexplained/electricity/how-electricity-is-generated.php*

Steam Turbine Test Case for Aeromechanics and Condensation (STAC)

Case Introduction

STAC: Steam Turbine Test Case for Aeromechanics and Condensation

The STAC case is a two stage low-pressure steam turbine case provided by the ITSM (Institute of Thermal Turbomachinery and Machinery Laboratory) at the University of Stuttgart. With permission to share both case information and geometry, this case offers the opportunity to demonstrate the principles, capabilities, and challenges of modeling steam turbines in Ansys Fluid Solutions.

University of Stuttgart
Institute of Thermal Turbomachinery and Machinery Laboratory

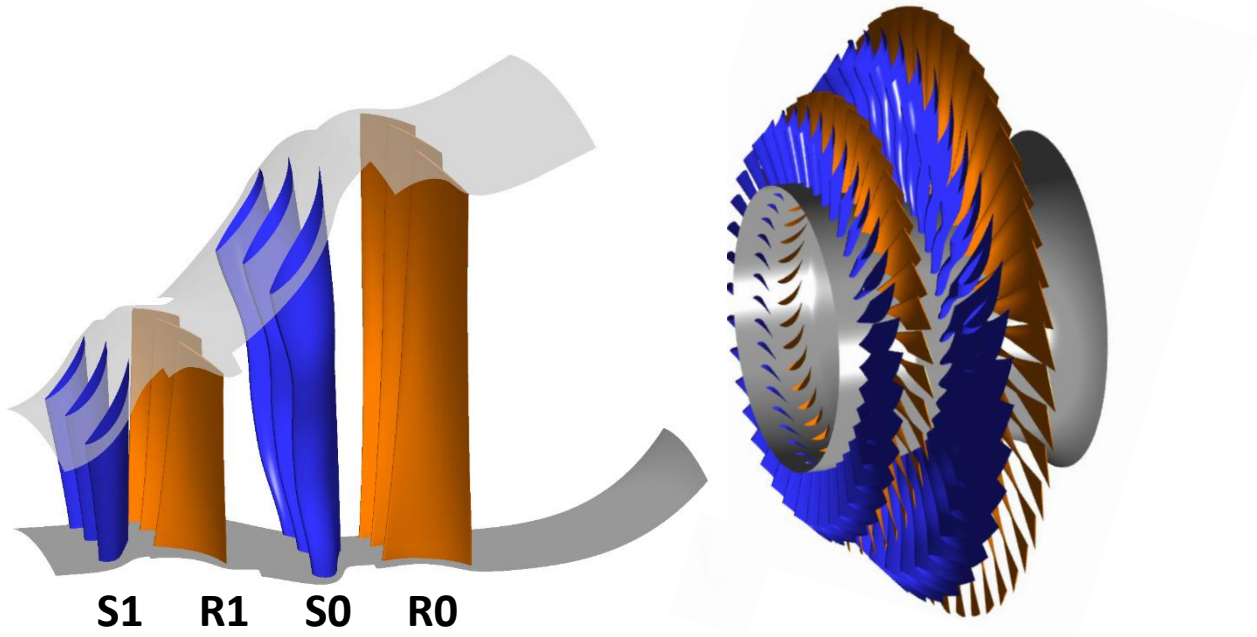
Public Case & Geometry

Case Details

- Numerical Test Case (no experimental data available)
- Last two stages of low-pressure steam turbine
- Blade lengths exceeding 1 meter
- Rapid expansion leading to spontaneous condensation

Setup files for this case accompanying this document are provided.

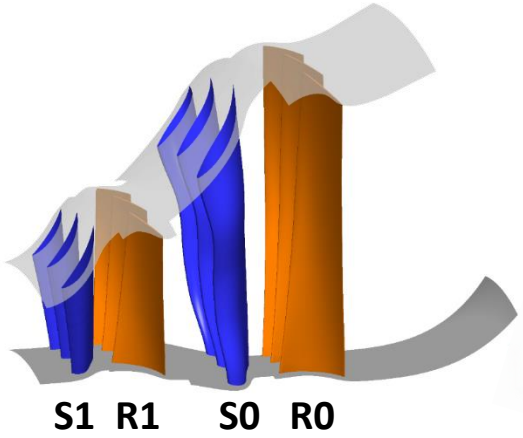
More Information on the case can be found at:
STAC | **Institute of Thermal Turbomachinery and Machinery Laboratory** | University of Stuttgart



STAC Case Setup Information

Technical Details

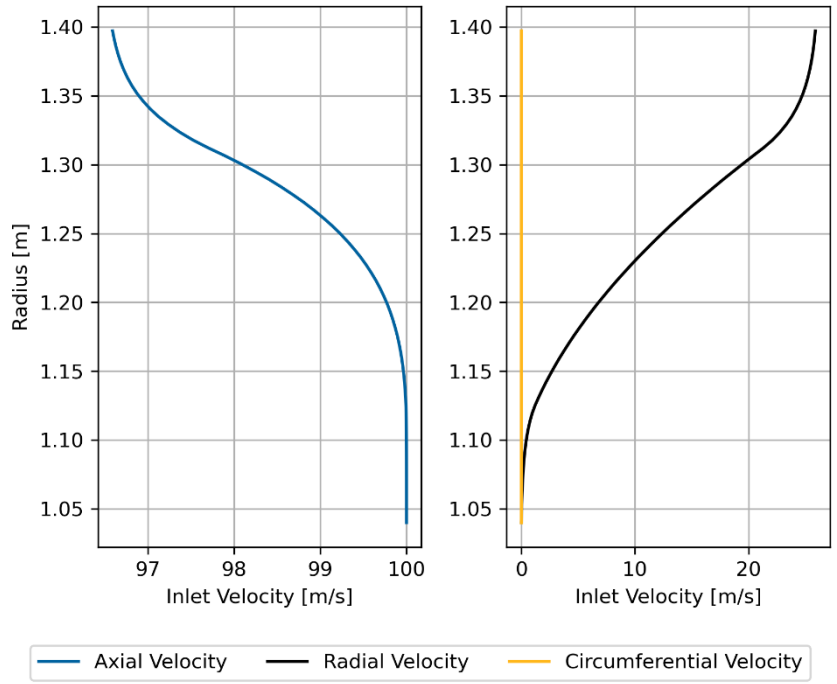
- 4 rows named as S1, R1, S0, and R0 for rotors and stators
- At the pressure inlet velocity directions are prescribed as profile data
  - Inlet velocity components are provided as CSV file



Boundary Conditions for Design Operating Point	
Inlet Total Temperature [K]	360
Inlet Total Pressure [Pa]	41 230
Outlet Static Pressure [Pa]	6500
Inlet Wetness Fraction [%]	0

Geometrical Information	S1	R1	S0	R0
Blade Count	39	41	37	45
Blade Length [m]	0.40	0.64	0.86	1.27
Rotational Speed [Hz]	-	50	-	50
Tip Clearance [m]	-	0.0064	-	0.0127

Inlet Velocity Components



Inlet Profile Data
01_STAC_Inlet_Boundary_Conditions.csv

Meshing of STAC Case

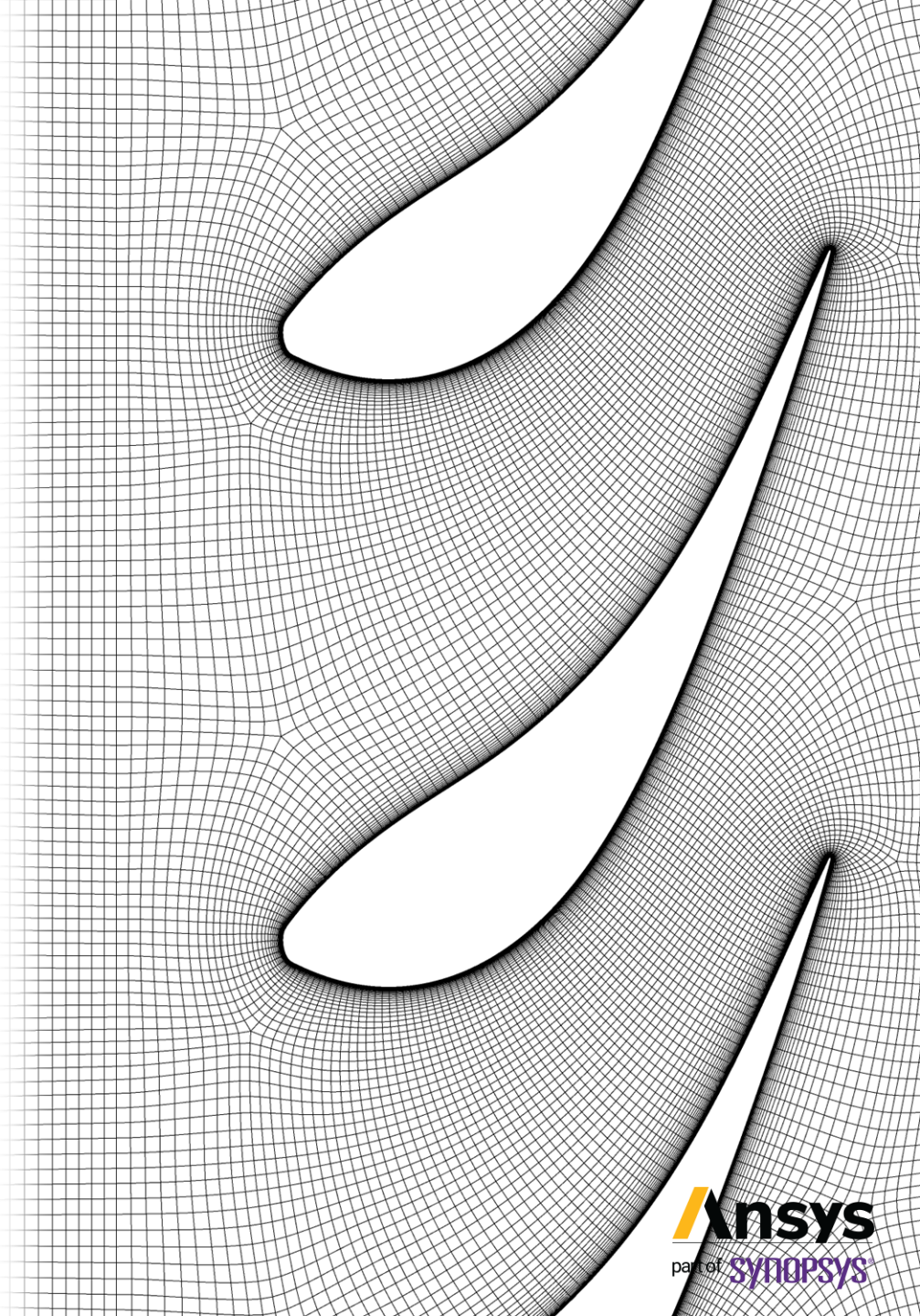
Using Ansys TurboGrid

STAC Mesh Study

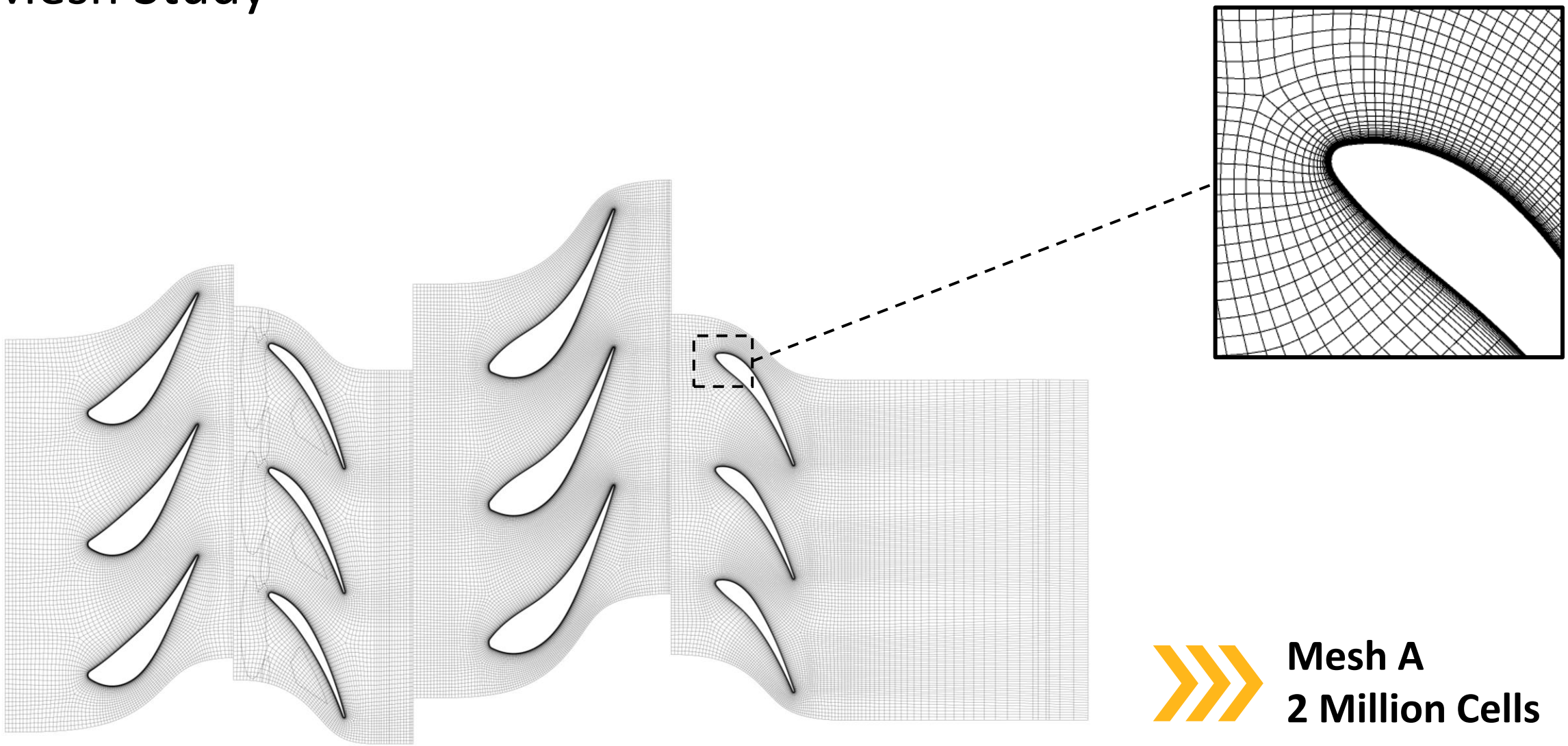
Using the geometrical information provided by the ITSM four meshes, each refined progressively, were generated and a mesh study was conducted. An overview of these meshes is given by the table below and on the following slides. The meshing was done using Ansys TurboGrid.

Mesh Information	Mesh A	Mesh B	Mesh C	Mesh D
Number of Cells	2 079 850	4 296 175	8 480 010	36 546 740
Min. Orthogonality Angle	19.92°	19.91°	19.91°	20.16°
Max. Edge Length Ratio	1387.08	2733.16	4452.21	8501.94
Average y+	5.84	1.93	0.94	0.47

Meshing TurboGrid Files → 08_TurboGridFiles



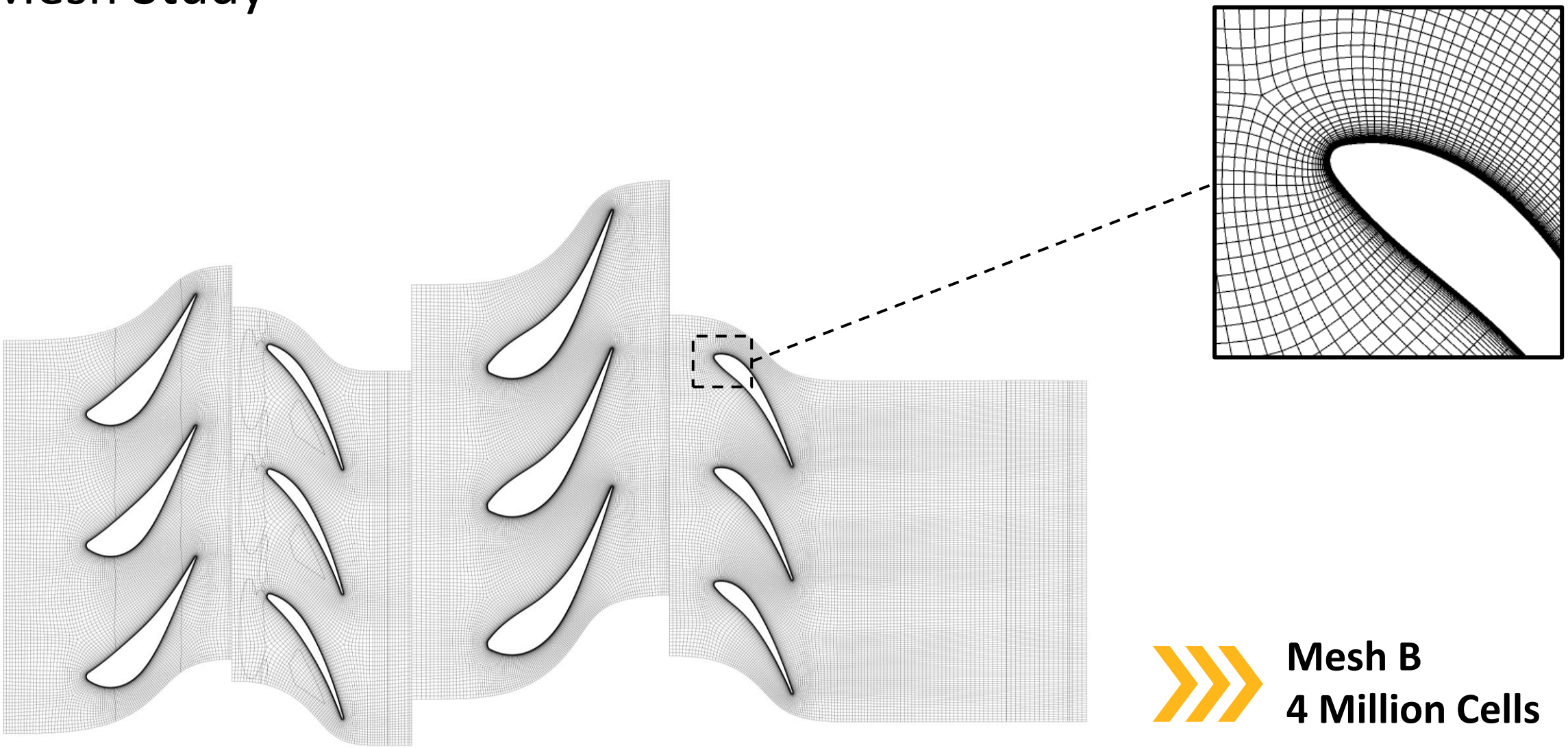
Mesh Study



Mesh A
2 Million Cells

Plot at 50% Span

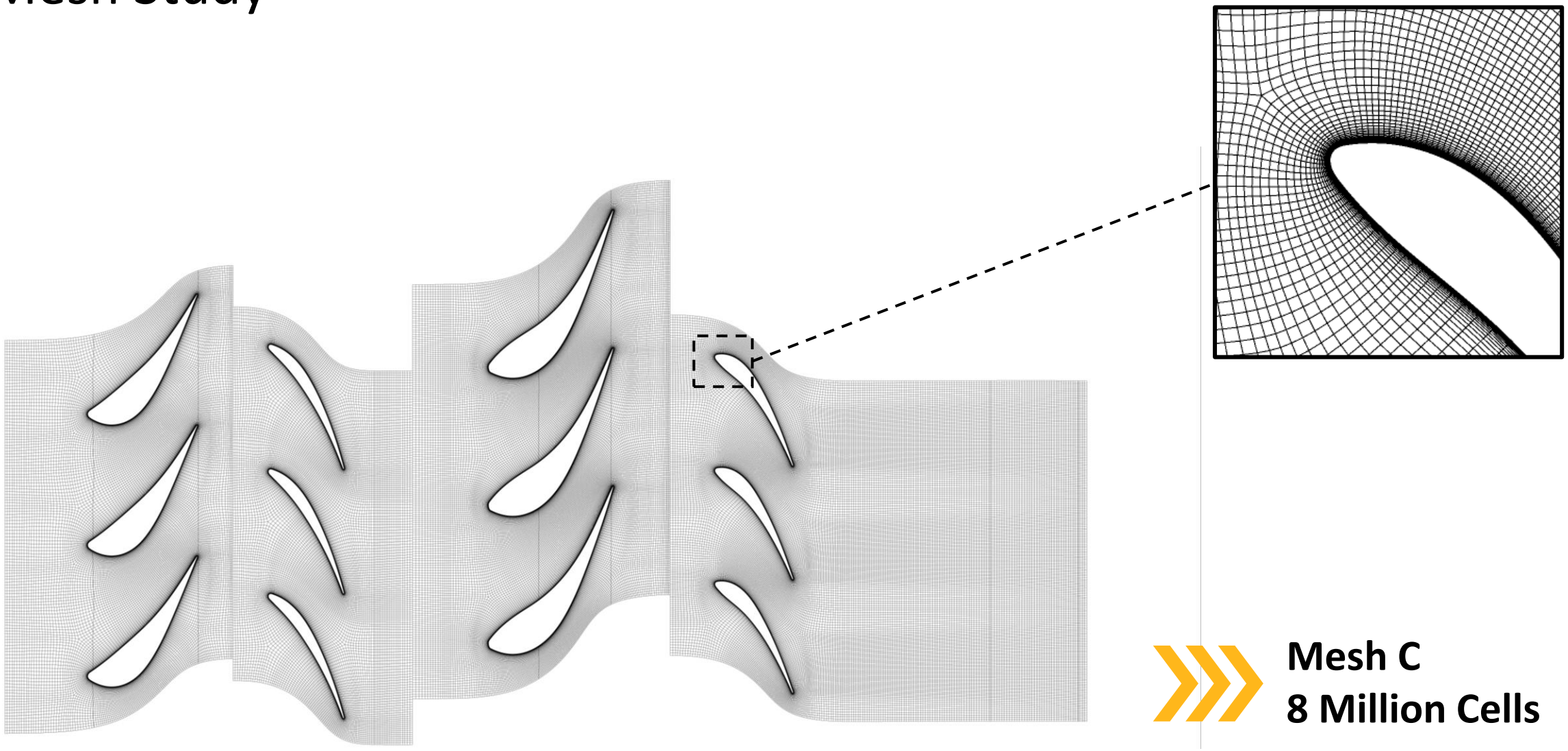
Mesh Study



Mesh B
4 Million Cells

Plot at 50% Span

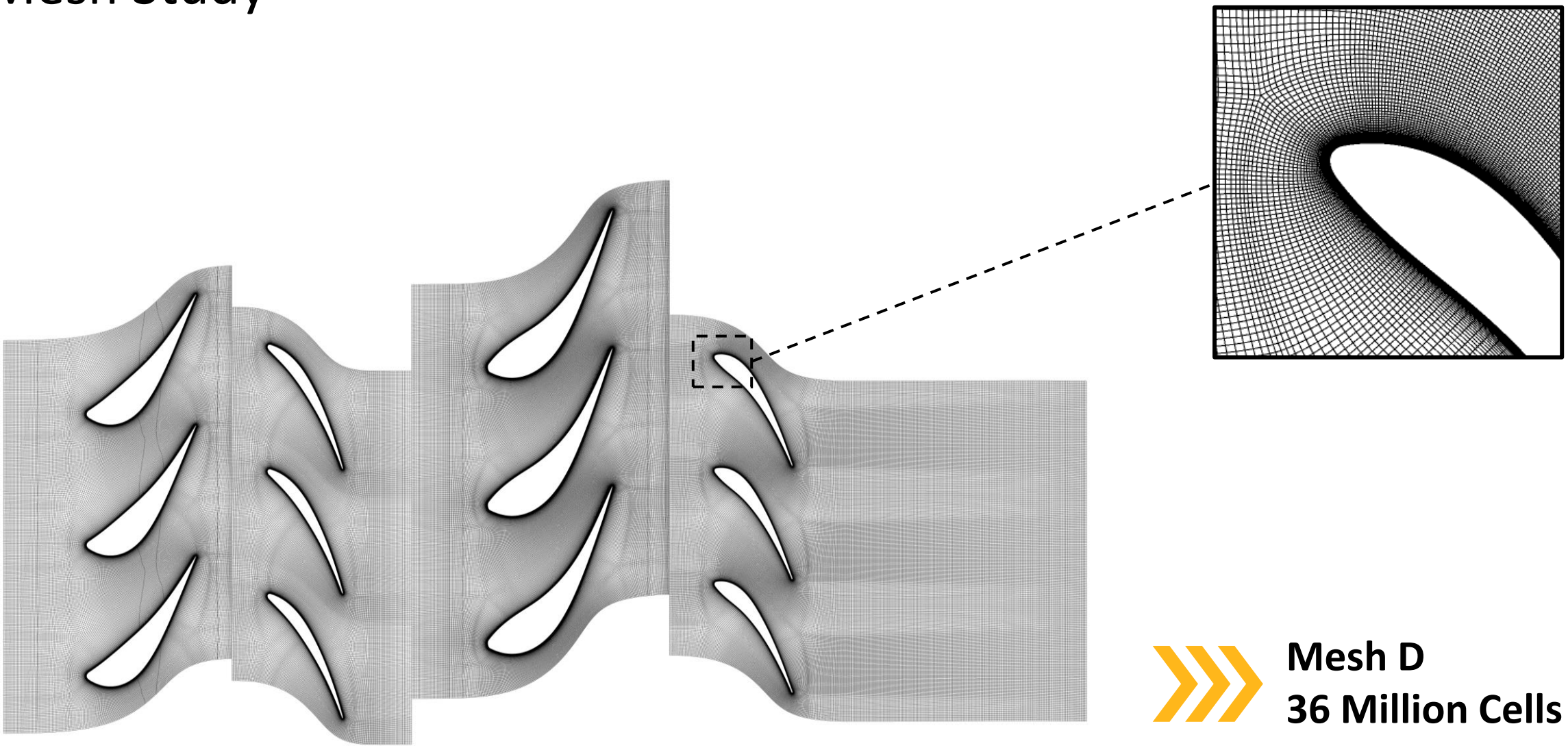
Mesh Study



Mesh C
8 Million Cells

Plot at 50% Span

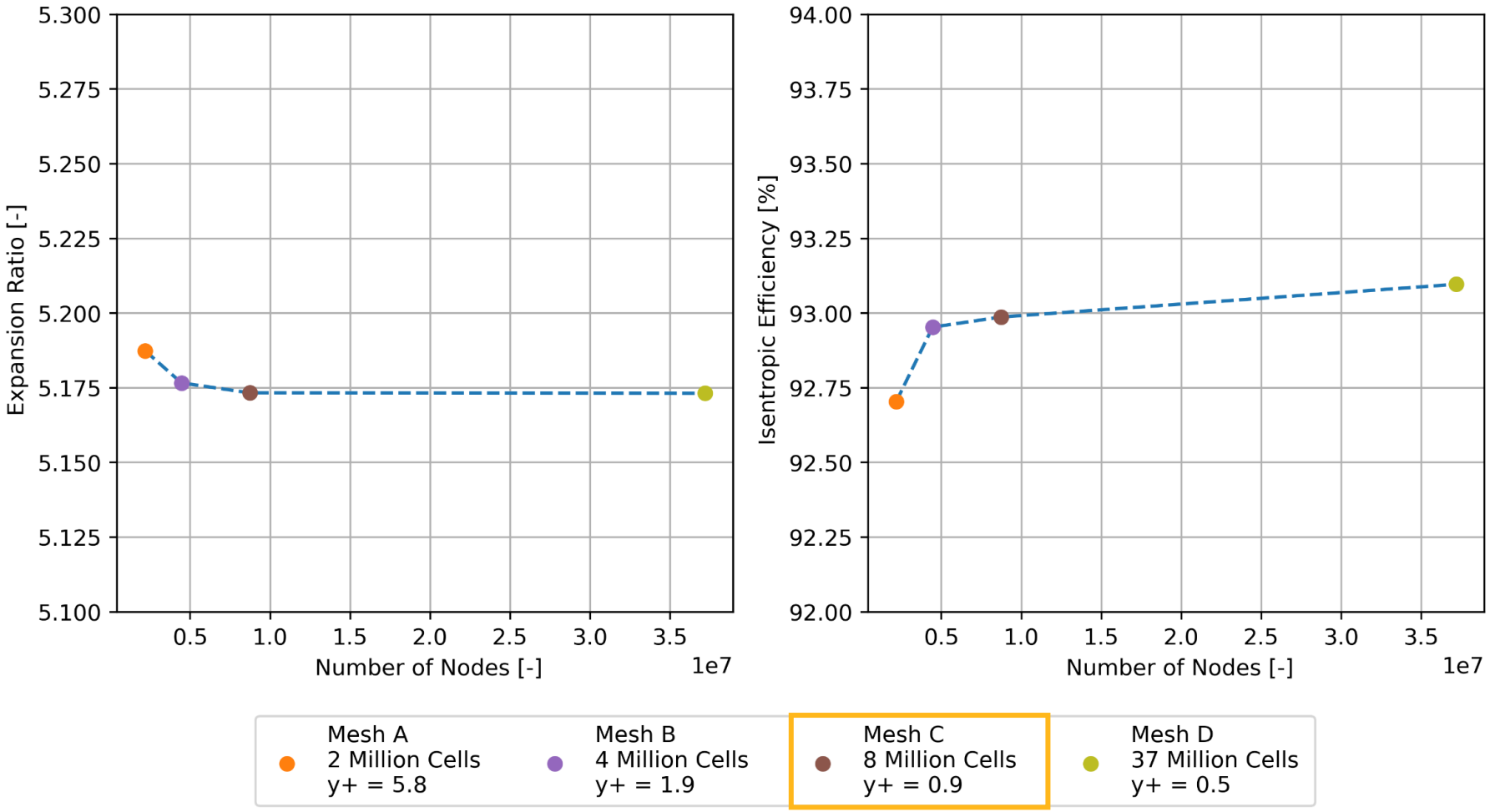
Mesh Study



Mesh D
36 Million Cells

Plot at 50% Span

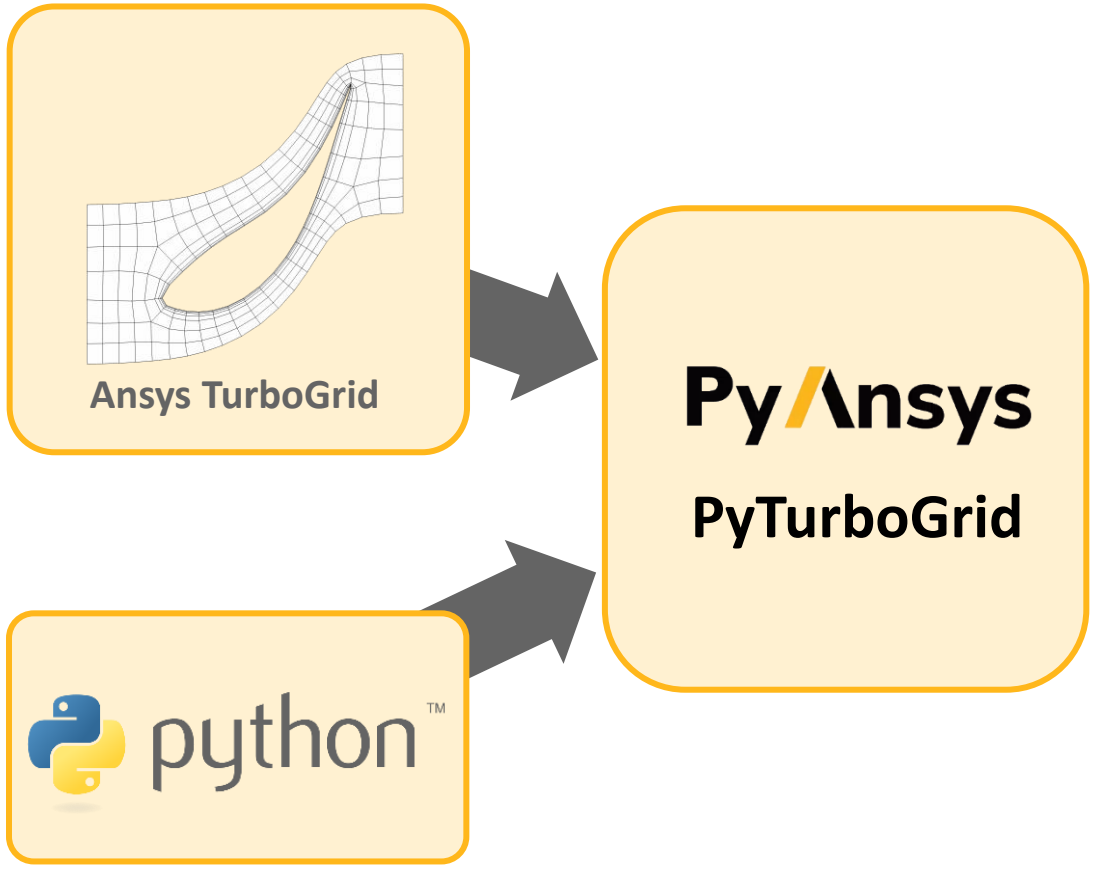
Mesh Study – Ideal Gas



Calculations based on the ideal gas assumption were performed in Ansys CFX using these four meshes. While full mesh independence has not been achieved, the differences between Mesh C and Mesh D are acceptable and can most likely be attributed to flow differences in the tip region (e.g. due to the different resolution of the tip gap vortex, etc.)

In comparison to industry standards, Mesh D, with over 36 million cells, is overly refined. To align more closely with industry practices, Mesh C is used as the basis for all following calculations.

Meshing – Ansys PyTurboGrid



An alternative method to access the meshing capabilities of Ansys TurboGrid is now available through PyTurboGrid.

PyTurboGrid allows the usage of Ansys TurboGrid within a Python environment
- Requires interaction only through Python interface
- Easier automatization of meshing process

Multi Blade Row Module
- Allows meshing of multiple blade rows at once
- A meshing strategy is implemented which adjusts the size factors of all rows to match overall
  → similar elements sizes in all blade rows
  → smoother mesh transitions without manual adjustments

Example Script for STAC Case provided within PyTurboGrid GitHub Page

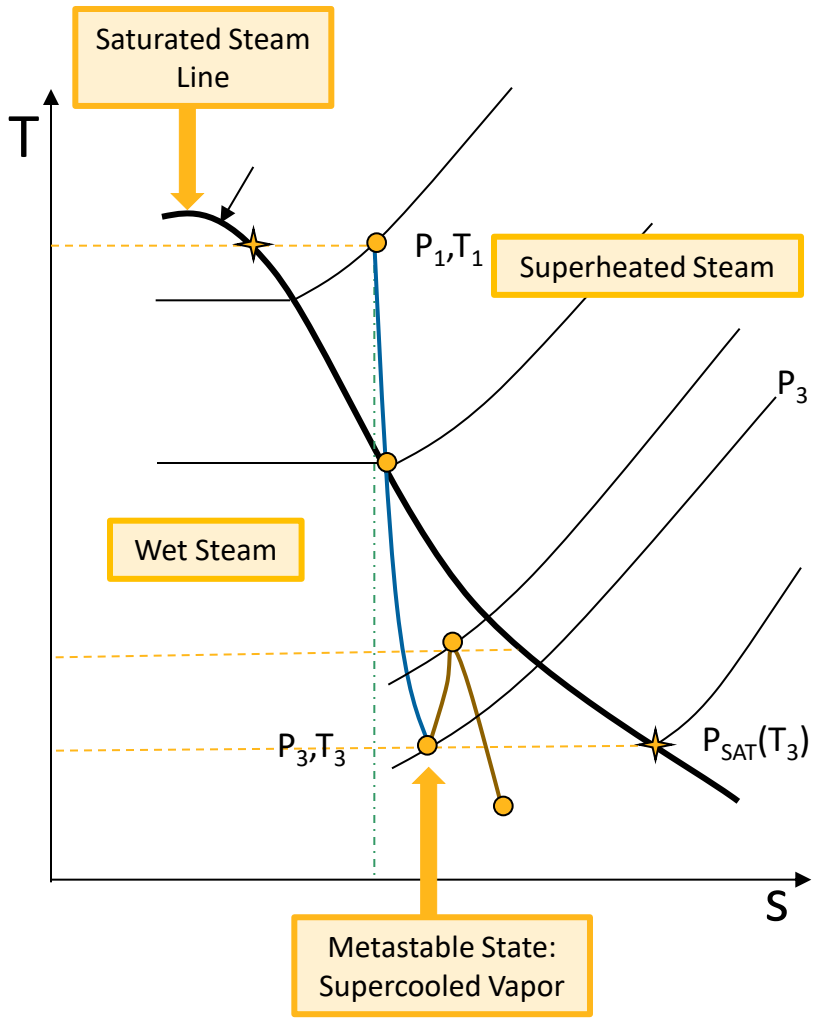
→ Explanation of Process / Code
→ https://github.com/ansys/pyturbogrid → In Examples folder

Prerequisite: pip install the ansys-turbogrid-core package (within virtual environment)

More Information: PyTurboGrid Documentation

Steam Modelling Approaches with Ansys CFD Solutions

Wet Steam in Steam Turbines

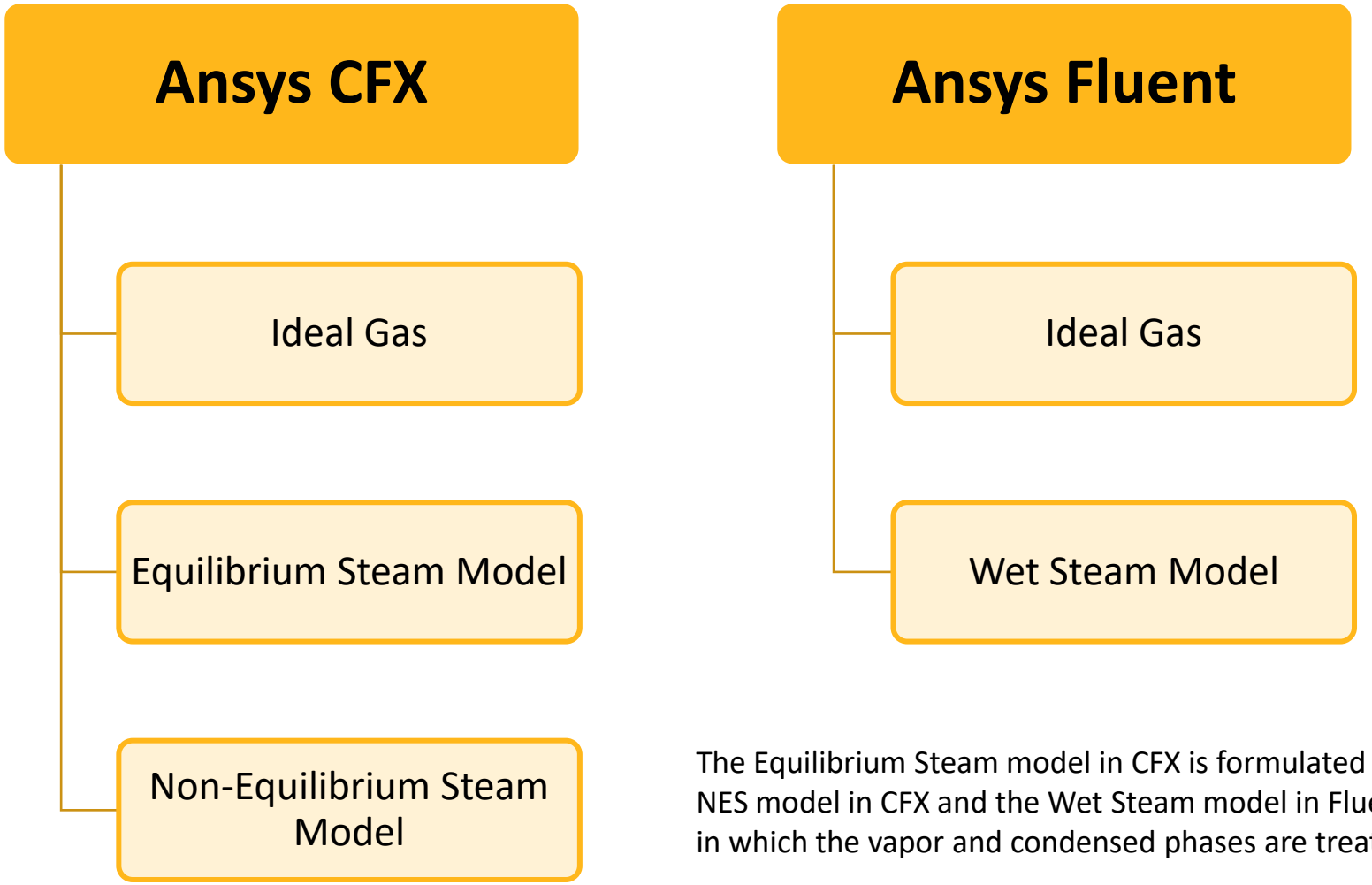


Wet steam is a two-phase mixture consisting of fine water droplets suspended in water vapor. Unlike dry or superheated steam, which is entirely in the gaseous phase and contains no liquid, wet steam has both vapor and liquid components.

In rapid expansion processes (such as pressure drops through nozzles or turbines), steam may briefly cross the saturation line into the two-phase region without immediate condensation. In this situation, the vapor becomes metastable: this steam is supercooled — it is “colder” or at a lower pressure than the equilibrium saturation condition. This metastable state is temporary.

Once a critical level of instability is reached, homogeneous nucleation begins. Clusters of molecules form and grow into fine droplets, leading to the onset of wet steam which will relax toward thermodynamic equilibrium, where liquid and vapor phases coexist

Steam Turbine Modelling Approaches with Ansys CFD Solutions



The Equilibrium Steam model in CFX is formulated as a single-fluid, multicomponent approach. The NES model in CFX and the Wet Steam model in Fluent employ multiphase Euler–Euler formulations, in which the vapor and condensed phases are treated as separate interpenetrating continua.

This document of guidelines will focus on steam turbine modelling in Ansys CFX.

Steam Turbine Modelling Approaches – Ansys CFX

Ideal Gas

H_2O can be set up as ideal gas, meaning it is approximated using the ideal gas law. As it is a pure gas phase no condensation occurs, and it is not an accurate representation of wet steam. However, it may serve as a good starting point for calculations.

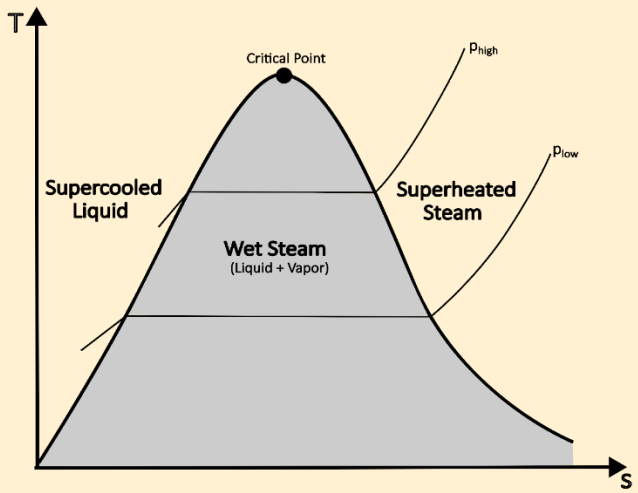
$$pV = nRT$$

Equilibrium Steam Model (EQS)

A single fluid made up of multiple components (gas and liquid phase) is created. The mixture of the two phases is assumed to always be in local thermodynamic equilibrium. As both phases have the same temperature, phase change occurs instantaneously, and the mass fractions are determined from the phase diagram.

→ Suitable for flows with small liquid mass fractions

Phase Diagram



Steam Turbine Modelling Approaches – Ansys CFX

Detailed Information in Ansys Theory Guide

Non-Equilibrium Steam Model (NES)

This setup involves multiple fluids: a continuous gas phase and one or more dispersed liquid phases. Mass conservation equations are solved for both the continuous and dispersed phases, along with an additional equation for the droplet number in the dispersed phase.

Mass transfer between the phases accounts for interfacial mass transfer (droplet growth) and droplet formation via nucleation, with the latter also contributing as a source term in the droplet number equation. Nucleation is modeled using classical homogeneous nucleation theory, and mass transfer at the interface is calculated using the Young model.

Unlike the Equilibrium Steam Model, this model does not assume instantaneous thermodynamic equilibrium. As a result, it also includes losses associated with thermodynamic irreversibility.

Mass Conservation Continuous Phase:

$$\frac{\partial \rho_c r_c}{\partial t} + \frac{\partial}{\partial x_i}(\rho_c u_i r_c) = -\sum_{i=1}^{nd}(S_d + m^* r_c J_d)$$

Mass Conservation Dispersed Phase:

$$\frac{\partial \rho_d r_d}{\partial t} + \frac{\partial}{\partial x_i}(\rho_d u_i r_d) = S_d + m^* r_c J_d$$

Droplet Number:

$$\frac{\partial \rho_d N_d}{\partial t} + \frac{\partial}{\partial x_i}(\rho_d u_i N_d) = \rho_d r_c J_d$$

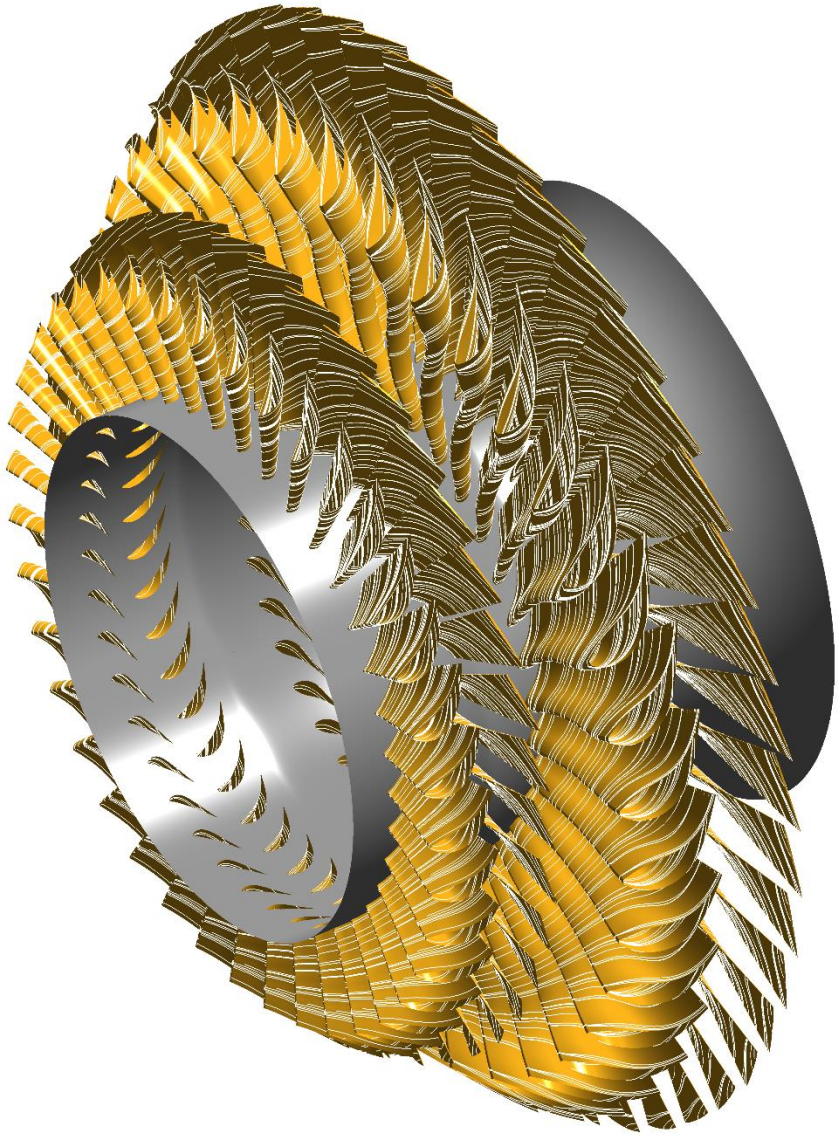
→ Suitable for flows that undergo rapid pressure reduction leading to nucleation

J_d - Nucleation Rate
S_d - Interfacial Mass Transfer

General Tips for Turbine Simulation in Ansys CFX

Steam Turbine in Ansys CFX

STAC Ideal Gas Set Ups
Mesh A → 02_MeshA_Ideal_Gas.def
Mesh C → 03_MeshC_Ideal_Gas.def



Ideal Gas as Starting Point

When modeling steam turbines in CFX, using H_2O as an ideal gas is a good starting point. This simplified model can help identify potential challenges that should be addressed when modeling the turbine. Additionally, the ideal gas solution serves as a useful initialization for the Non-Equilibrium Steam Model.

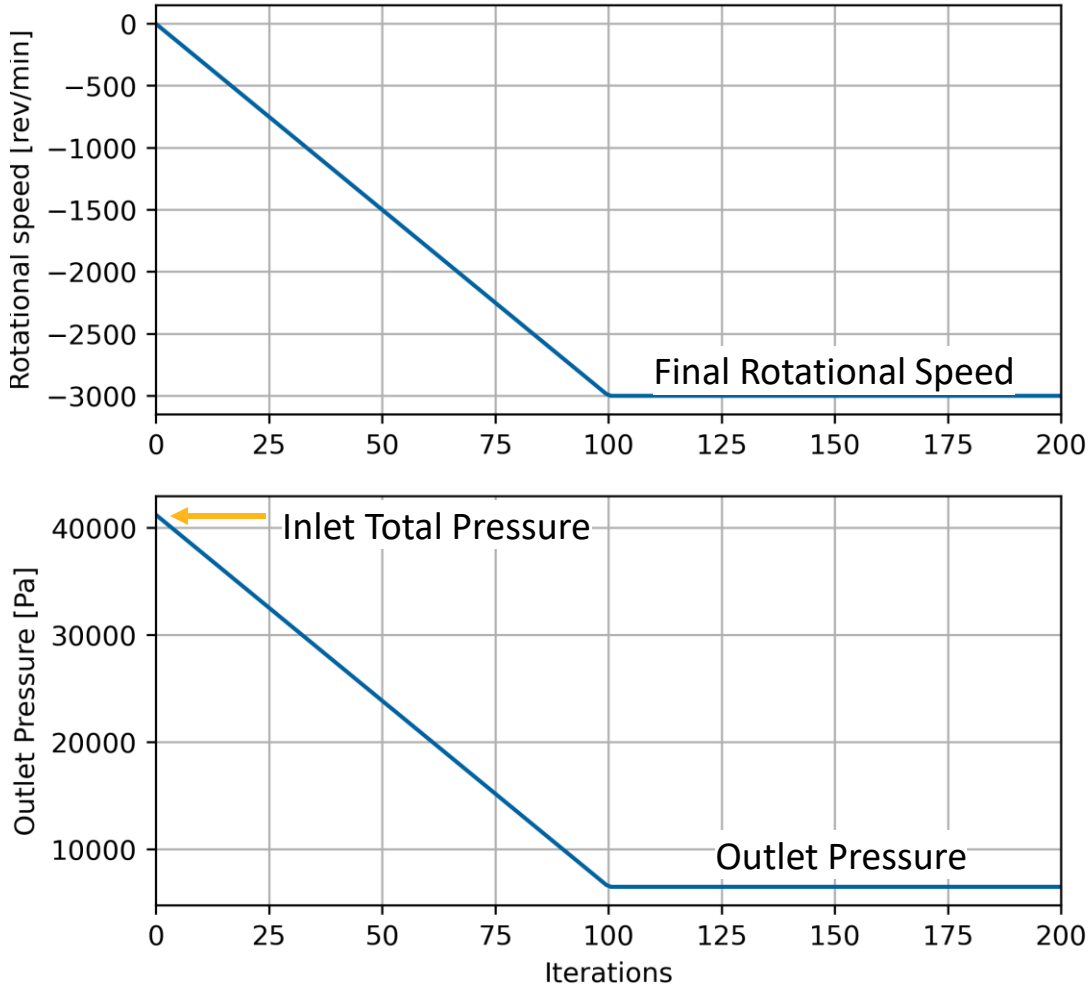
The following sections outline example settings and measures that proved helpful or were essential for the STAC case ideal gas or general calculations such as:

- Ramping Boundary Conditions
- High Speed Numerics Settings
- Velocity Blending at Wall Velocity Jumps
- Convergence Check Criteria
- Mixing Plane Settings to Reduce Reflections

Ramping Boundary Conditions

When starting a calculation directly at full load the sudden changes can cause numerical instabilities in the solver. This might lead to a solver crash or convergence issues. Alternatively, the load can gradually be increased over iterations slowly approaching full load. This is done by “ramping up” certain boundary conditions.

For the STAC case (ideal gas), it was necessary to ramp both the rotational speed and the outlet pressure. A linear ramp over 100 iterations was used to approach the full load boundary conditions. Since the STAC case rotates in the mathematical negative direction, the rotational speed is decreased from zero to the final negative rotational speed. The outlet pressure is also reduced, starting initially at the inlet total pressure. As a result, the expansion ratio and, consequently, the mass flow rate, are gradually increased.



STAC Case - CEL Expressions used:

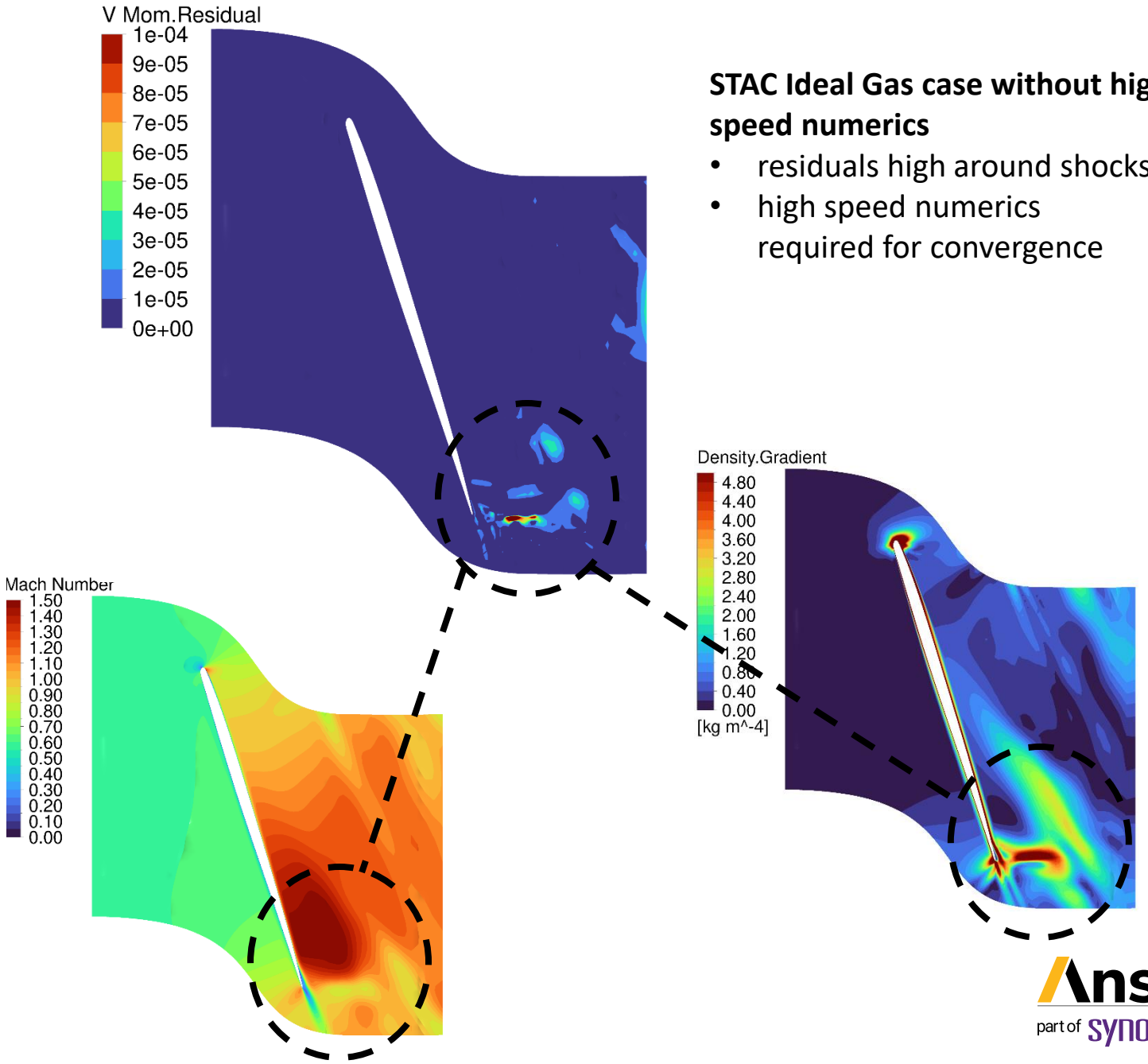
BC RPM Ramp = BC RPM [rev/min] * min (aitern/100, 1)

BC OUT p Ramp = max(BC Out p, ((BC Out p - BC In p)/100) * aitern + BC In p) [Pa]

High Speed Numerics

The high speed numerics setting activates special numerics that can be helpful in flows with shocks with convergence issues. An indication that high speed numerics can lead to improvements is that the maximum residuals are found in regions of shocks.

High Speed Numerics allows blending from higher order to lower order schemes near extrema, thereby reducing wiggles in regions of shocks.



STAC Ideal Gas case without high speed numerics

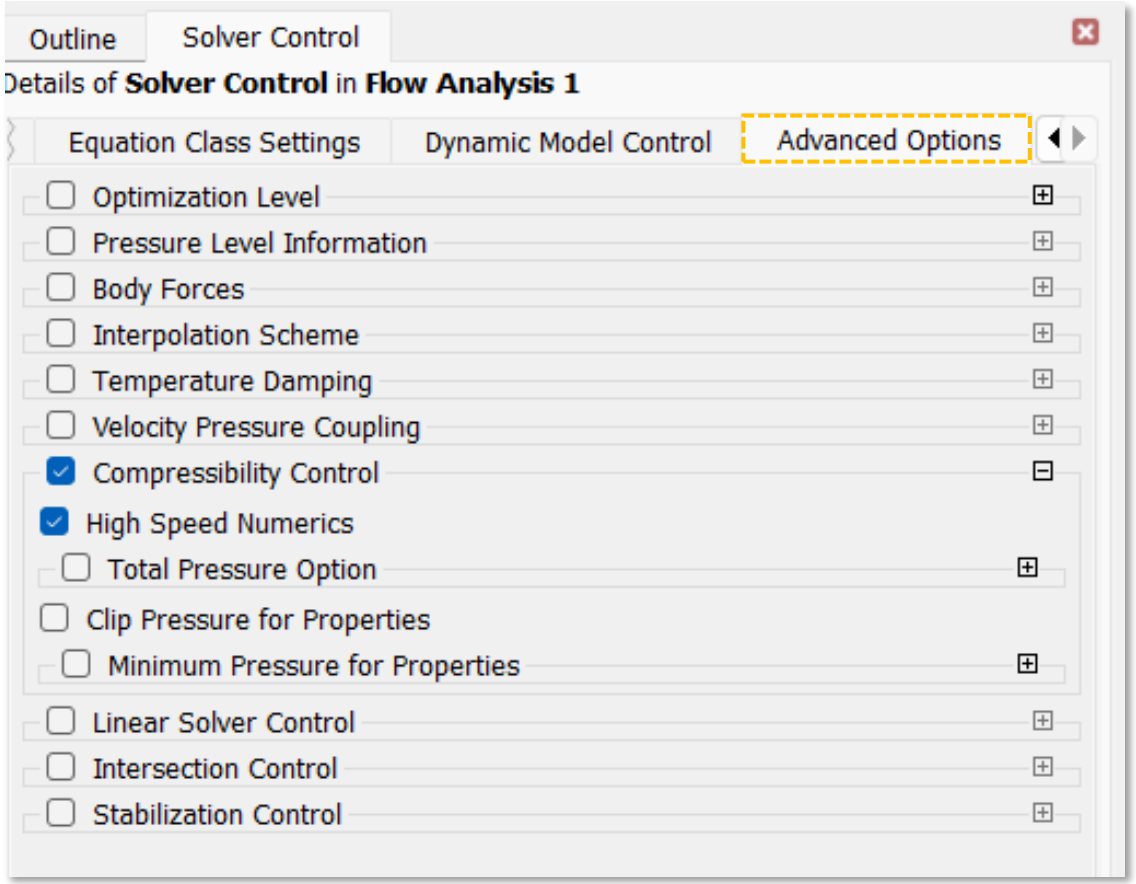
- residuals high around shocks
- high speed numerics required for convergence

High Speed Numerics

The high speed numerics setting activates special numerics that can be helpful in flows with shocks with convergence issues. An indication that high speed numerics can lead to improvements is that the maximum residuals are found in regions of shocks.

High Speed Numerics allows blending from higher order to lower order schemes near extrema, thereby reducing wiggles in regions of shocks.

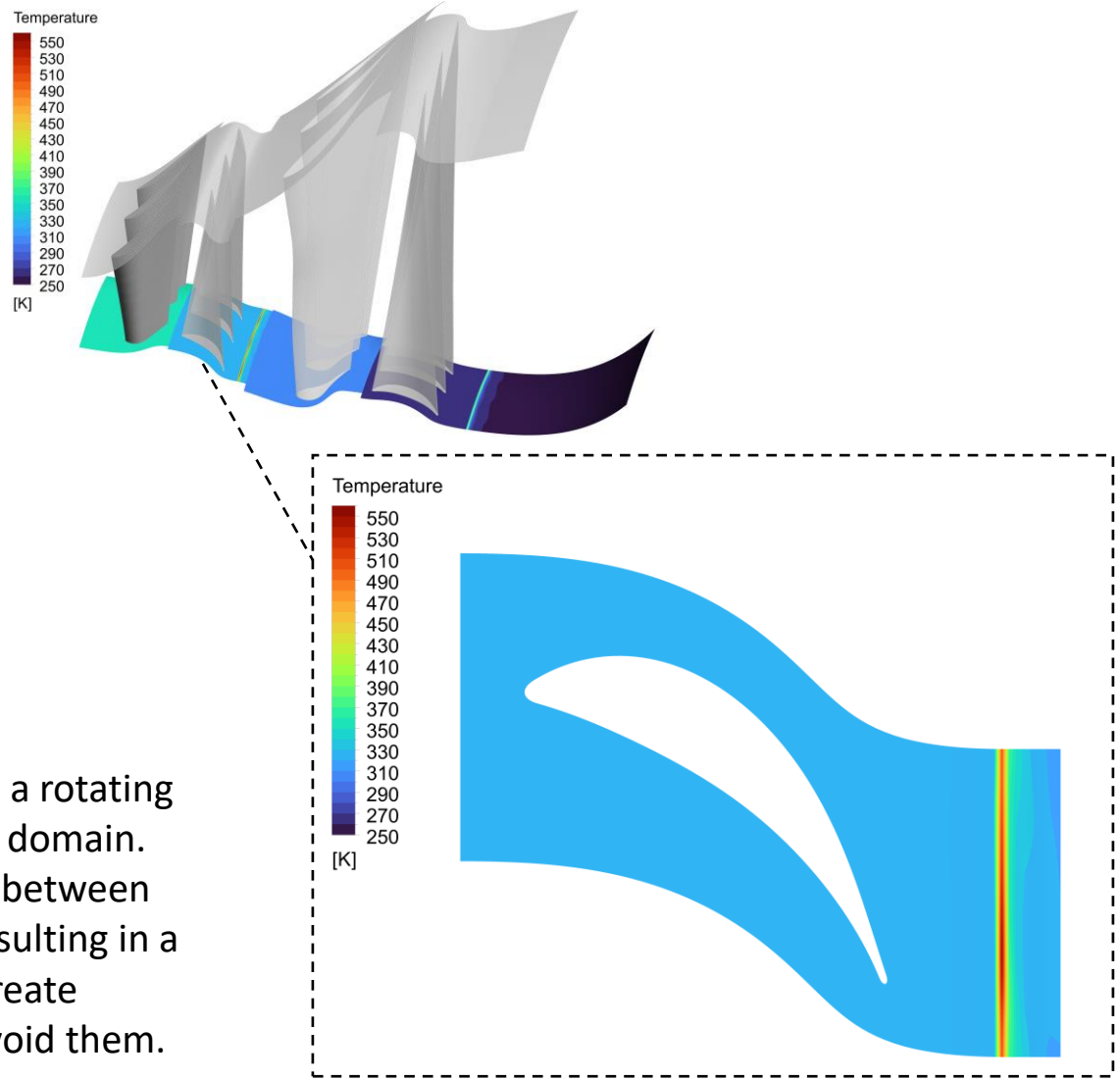
More Information on the effect of High Speed Numerics settings can be found in the CFX-Pre Users Guide: 22.6. Advanced Options Tab



Setting High Speed Numerics

Solver → Solver Control → Advanced Options Tab → Compressibility Control → High Speed Numerics

Velocity Blending at Jumps



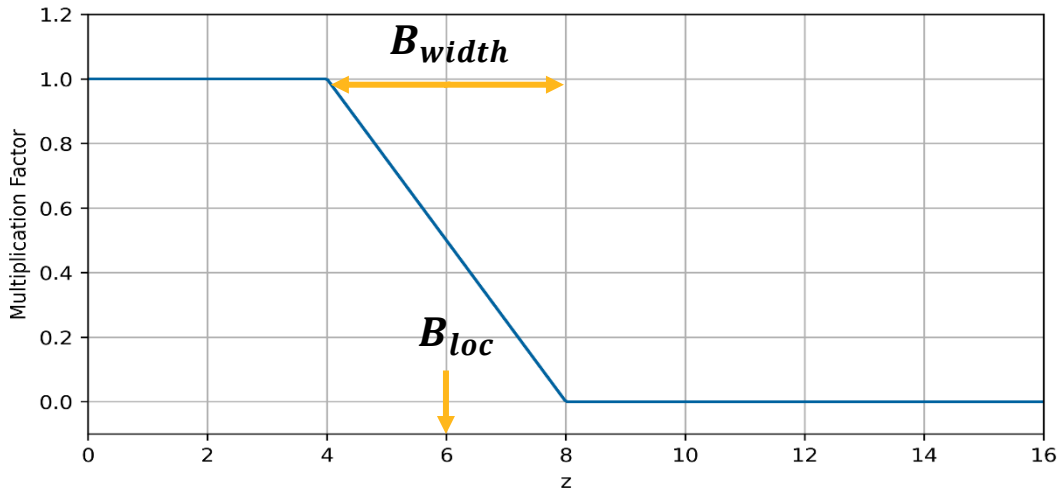
When modeling the STAC turbine, the rotor hubs were divided into a rotating main part and a static (non-rotating) section toward the end of the domain. This division creates a significant velocity gradient at the transition between these two regions, leading to work done by viscous stresses and resulting in a temperature spike at the transition. Such temperature peaks can create challenges in multiphase flow simulations, and it is preferable to avoid them.

Velocity Blending at Jumps

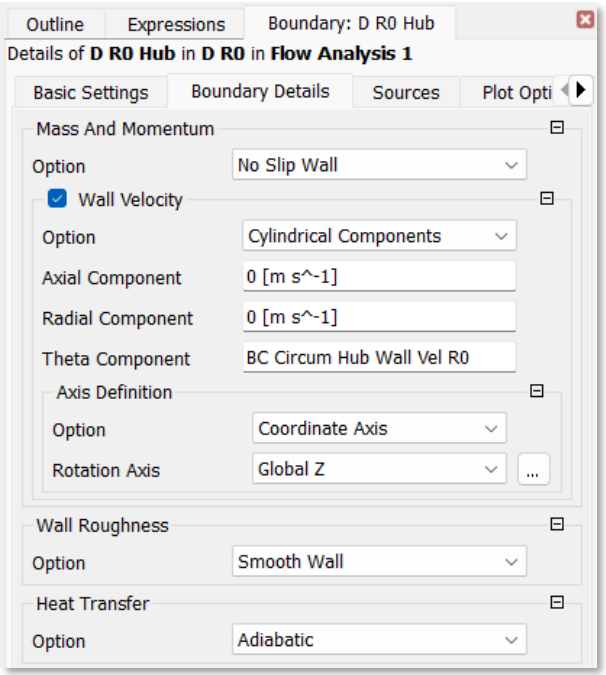
Reduce velocity gradient at transition by introducing a velocity function with blending:

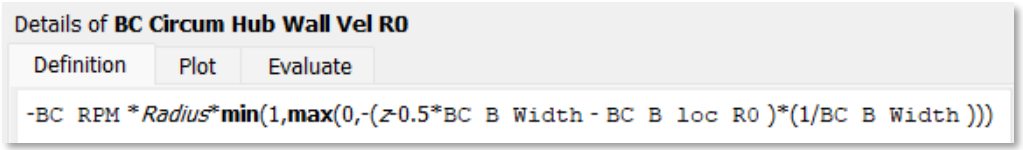
- A tangential wall velocity is assigned to the hub of the rotor → includes the transition from the rotating to the static hub
- Wall velocity is defined as an expression and a function of the axial coordinate
- Tangential speed defined via radius and rotational speed and multiplied with a factor to perform the blending

$$\text{Circumferential Wall Velocity} = \text{RPM} \cdot \text{Radius} \cdot \boldsymbol{max}\left(0, \boldsymbol{min}\left(1, \frac{-z + 0.5\boldsymbol{B_{width}} + \boldsymbol{B_{loc}}}{\boldsymbol{B_{width}}}\right)\right)$$



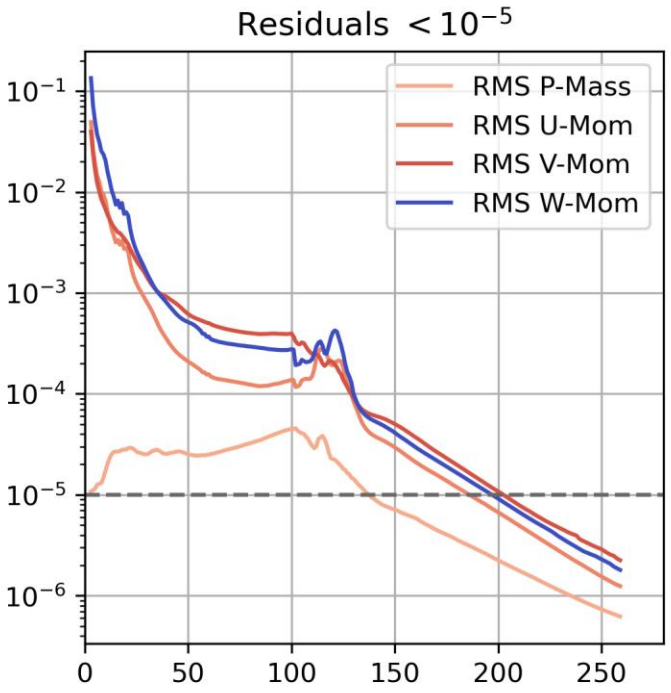
$\boldsymbol{B_{loc}}$ - Location of blending center
$\boldsymbol{B_{width}}$ - Width of blending



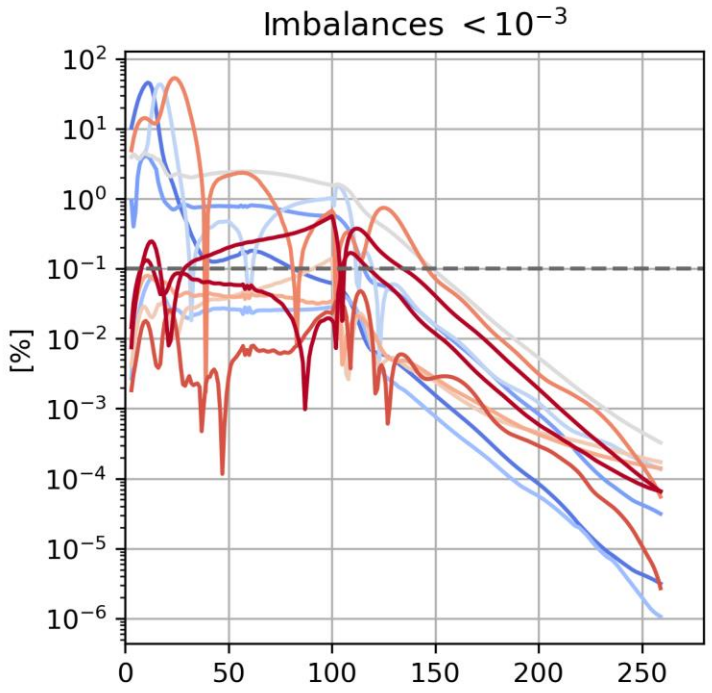


Convergence Criteria

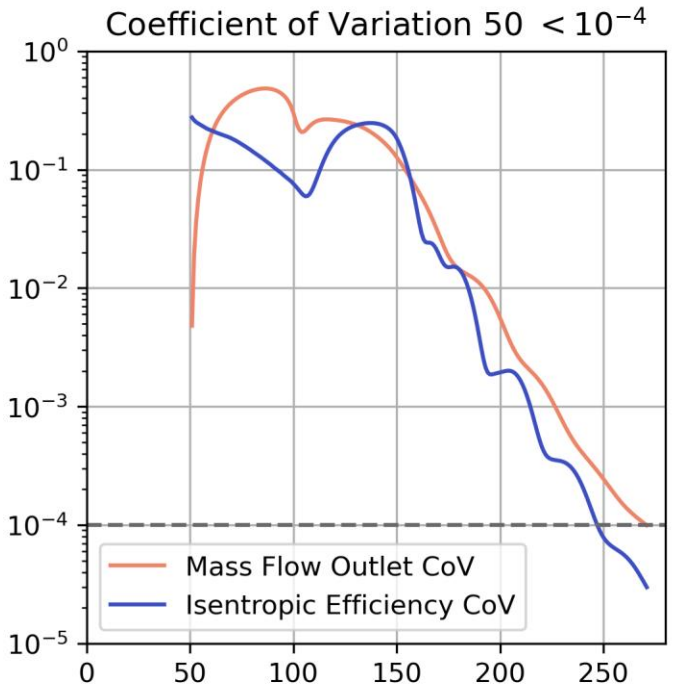
To determine whether a simulation has converged, three types of criteria are typically evaluated. Once all criteria fall below a specified threshold, the simulation is generally considered to have converged.



All RMS Residuals below **10^{-5}**



All Imbalances of all domains below **10^{-3}**



Check the Coefficient of Variation for some integral quantities (details on following slides). These should be below **10^{-4}** if the CoV is over 50 iterations.

Convergence Criteria – Coefficient of Variation

The Coefficient of Variation (CoV) is a dimensionless parameter that combines multiple characteristic values. It measures the relative variability of a dataset, making it an effective indicator of how consistent a quantity remains over a series of iterations. The CoV is particularly useful for assessing the consistency of integral quantities.

$$\boldsymbol{CoV(i) = \sigma(i) / \overline{\phi(\mathrm{i})}}$$

Moving Average $\overline{\phi(\mathrm{i})} = \frac{1}{N}\sum_{j=i-N+1}^{i} \phi_j$

→ Average of the most recent N values

Standard Deviation $\sigma(i) = \sqrt{\frac{1}{N}\sum_{j=i-N+1}^{i}\left(\phi_j - \overline{\phi_j}\right)^2}$

→ Variation of the previous N values from the mean

→ By normalizing the standard deviation with respect to the mean, quantities of different scales can be compared using the CoV as a common value

Convergence Criteria as Stopping Criteria

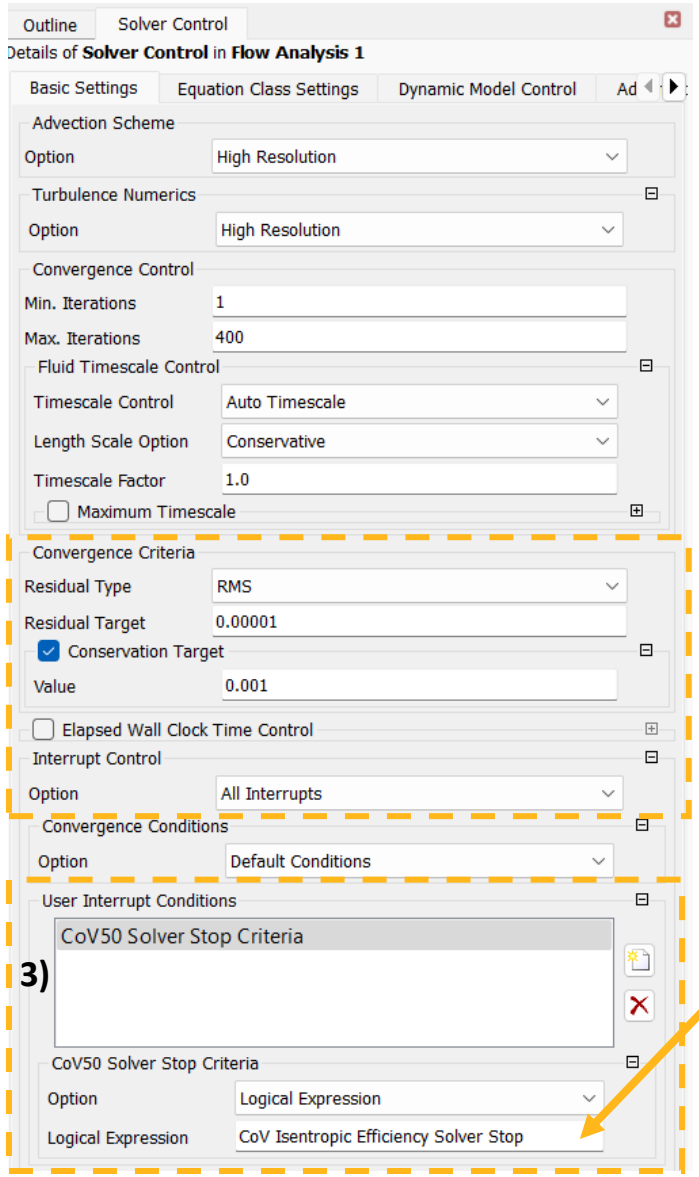
The three criteria can be set in CFX Pre, allowing the calculation to automatically stop once all criteria are met. The main settings for this are done in the Solver Control Settings:

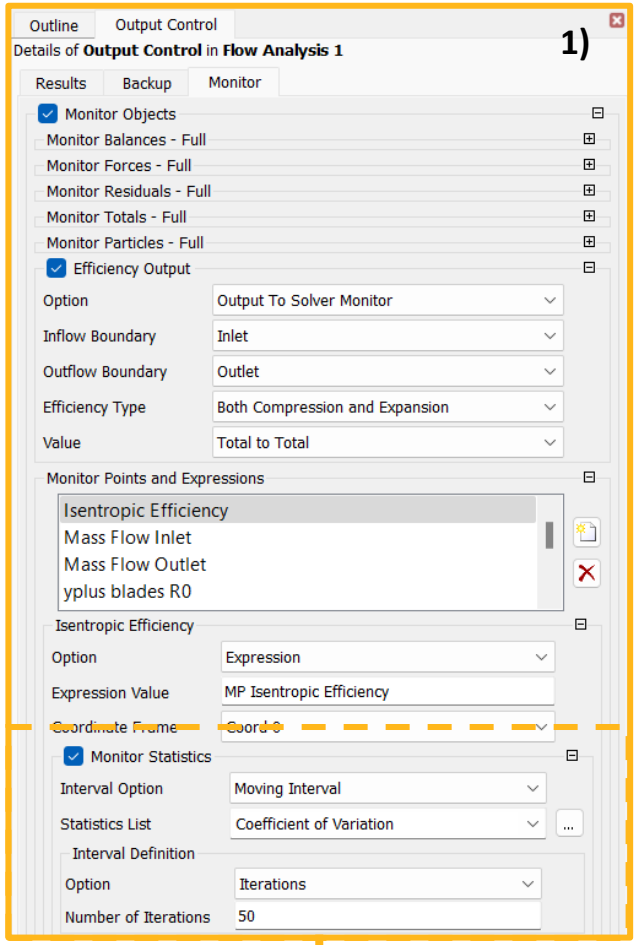
- Set Residual Type to RMS and Residual Target to 10^{-5}
- Select Conservation Target and the Value to 10^{-3}
- Set “All Interrupts” as Interrupt Control Option → The calculation will only stop once all criteria are met or the max. number of iterations or calculation time is reached.
- Add the Coefficient of Variation for integral quantities as User Interrupt Condition:

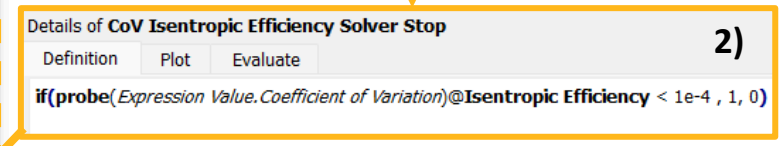
1) In Output Control → Monitor:
   - Create a monitor point for an integral quantity (based on an expression for this quantity)
   - Select Monitor Statistics
   - Chose Moving Interval, Coefficient of Variation and set the number of Iterations (we usually suggest 50)

2) Create a logical Expression which checks if the CoV for this integral value is below the threshold of 10^{-4} for a CoV over 50 iterations. If the CoV is over less iterations a lower threshold is required.

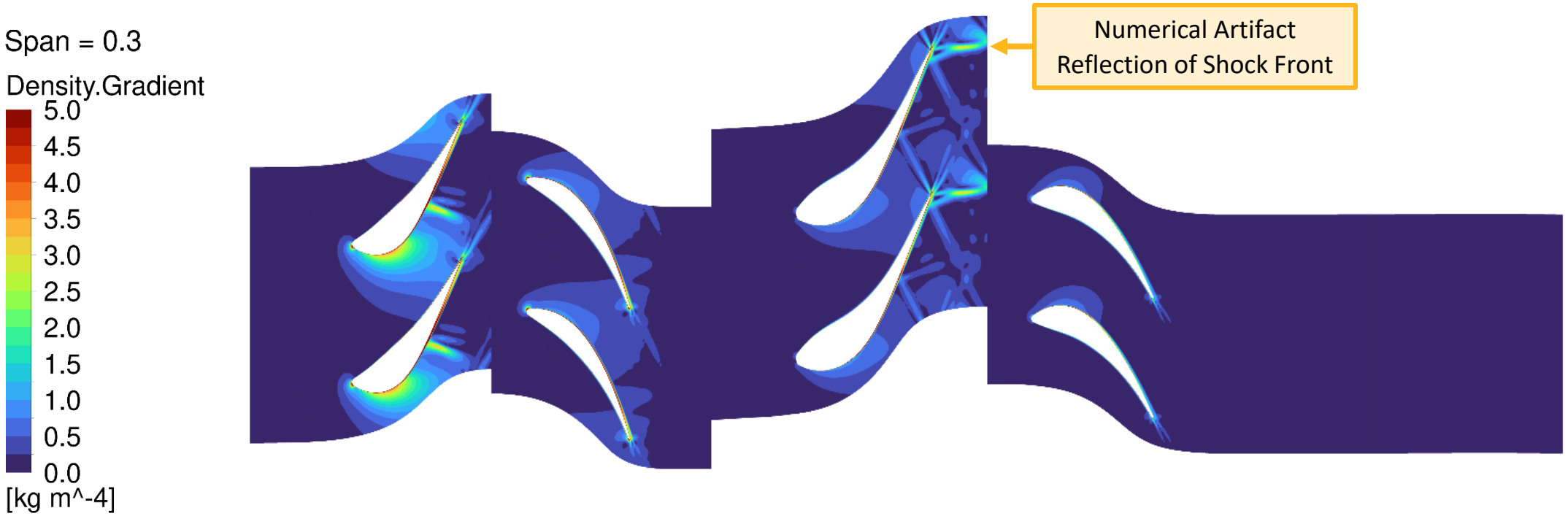
3) Add the respective User Interrupt Condition based on the expression created







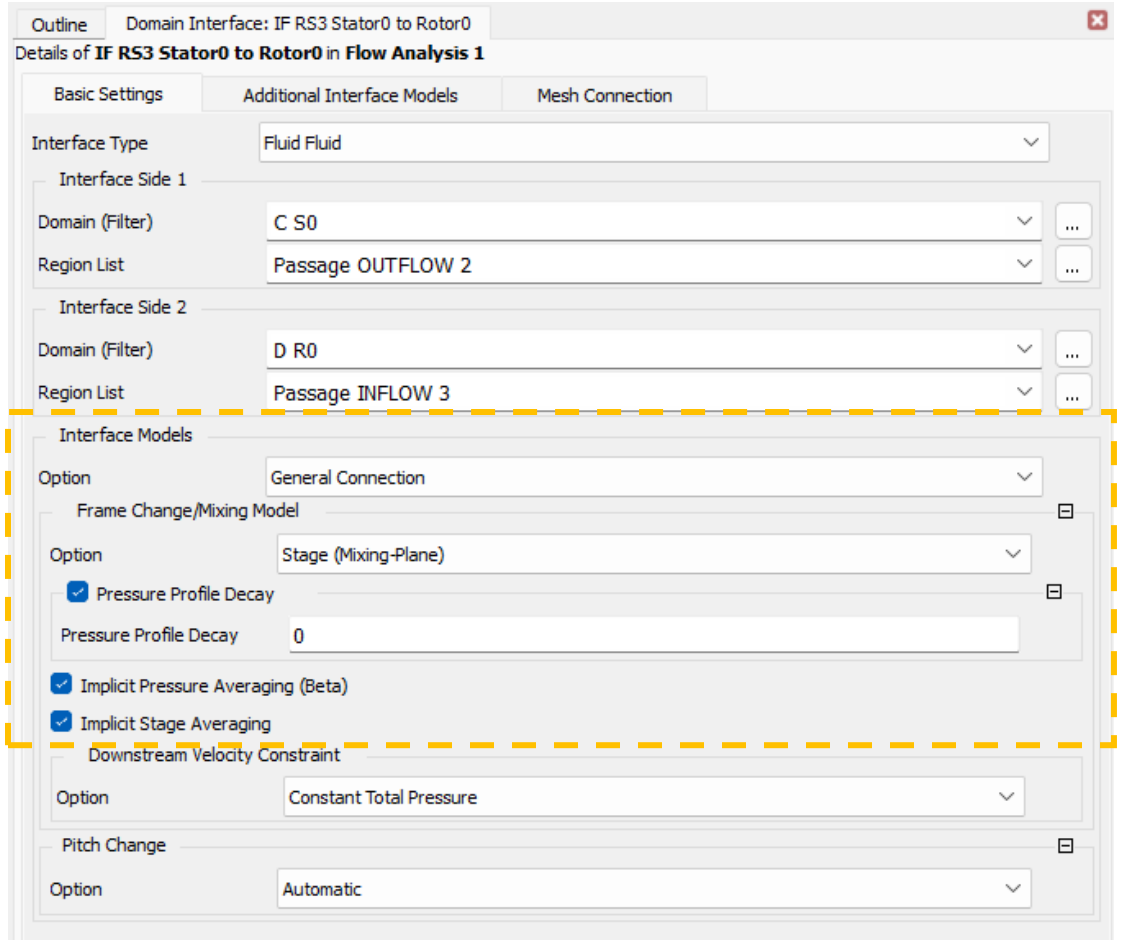
Reflections at Mixing Planes



When using the default mixing plane settings, the averaging process can result in the reflection of a shock front at the mixing plane. This creates an unphysical numerical artifact that may need to be eliminated.

→ Improvement when using implicit mixing plane settings (see next slide)
→ Improves results when using Ideal Gas, the Equilibrium Steam Model and the Non-Equilibrium Steam Model

Reflections at Mixing Planes – Implicit Mixing Plane Settings



Implicit Mixing Plane Settings

- Activate Implicit Pressure Averaging (beta features must be enabled)
- Activate Implicit Stage Averaging
- Set the Pressure Profile Decay to 0.0

  → Alternatively use / adapt provided ccl file

Implicit Mixing Plane Settings

GUI

Interfaces → Specific Interface → Interface Models → Frame Change / Mixing Model

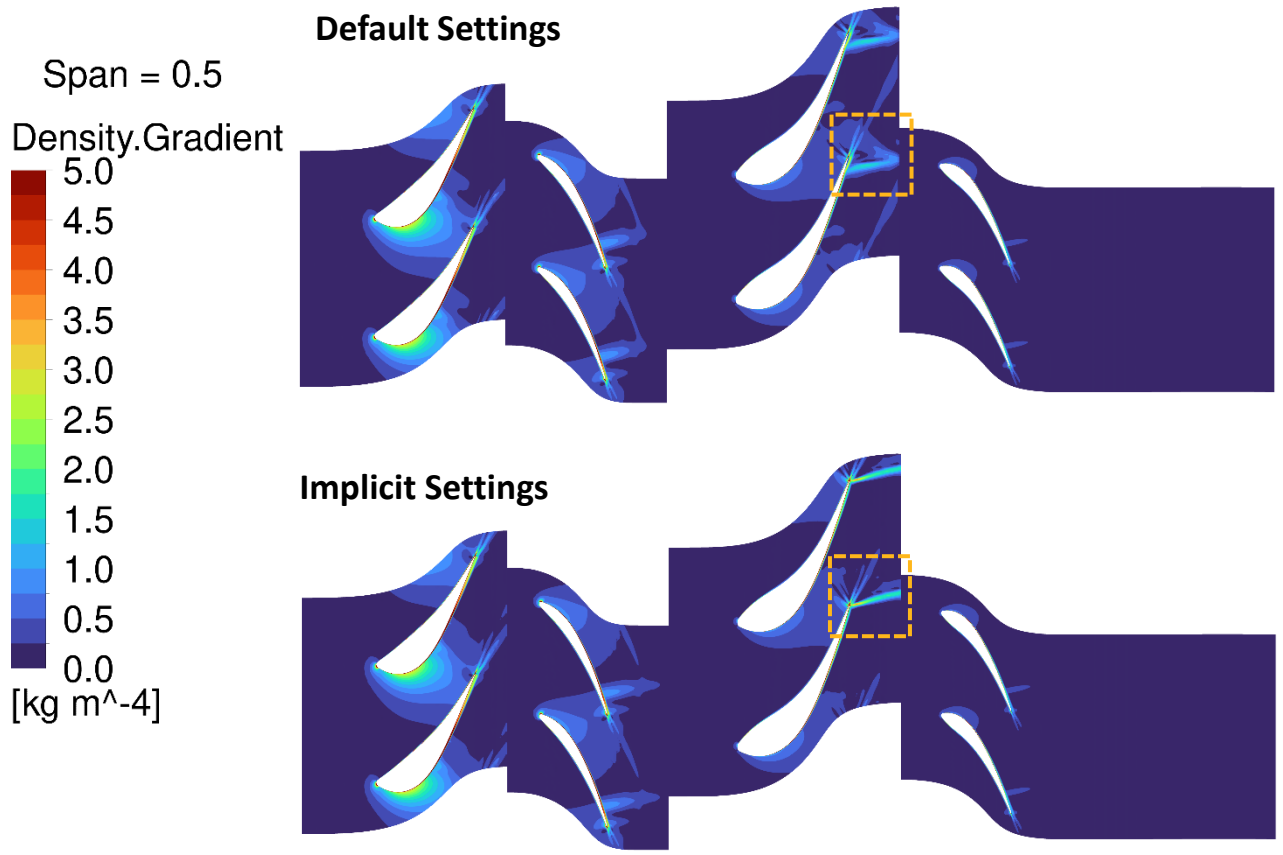
STAC Implicit Settings
07_Implicit_Mixing_Plane_settings.ccl

```
INTERFACE MODELS:
      Option = General Connection
      FRAME CHANGE:
        Implicit Pressure Averaging = On
        Implicit Stage Averaging = On
        Option = Stage
        Pressure Profile Decay = 0
```

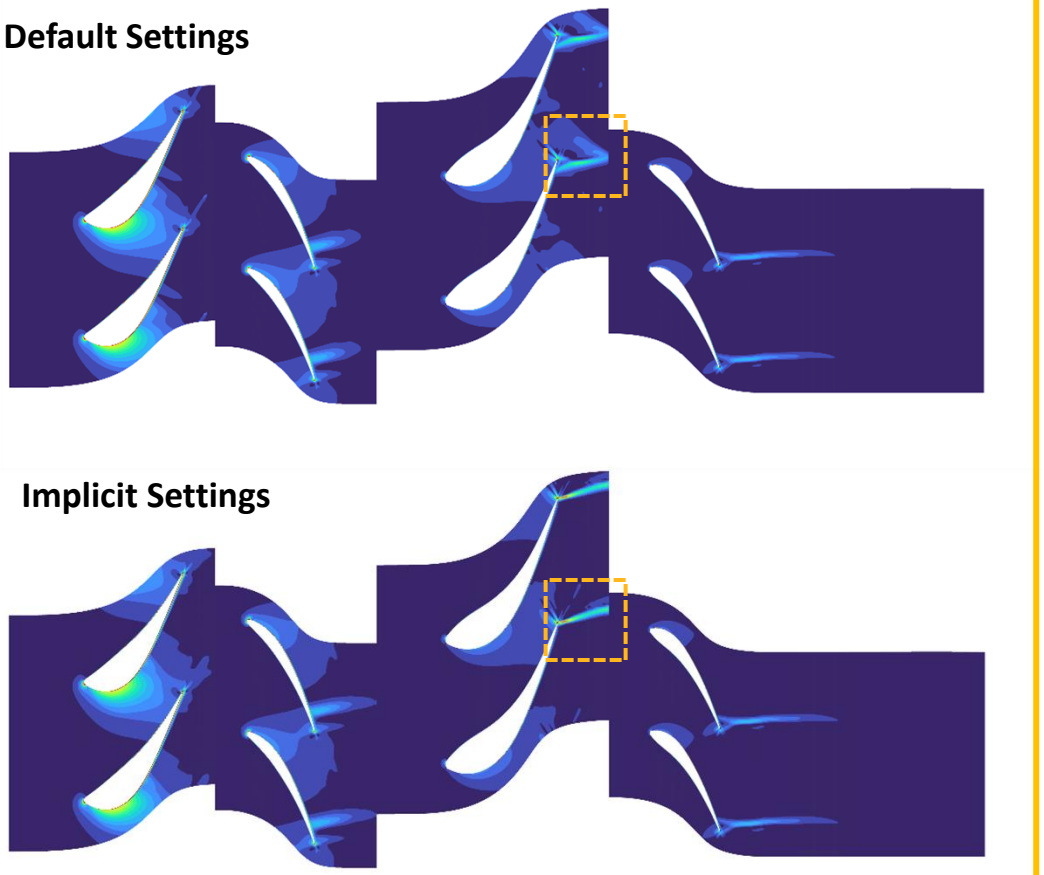
CCL snippet

Reflections at Mixing Planes – Results STAC Case

Ideal Gas



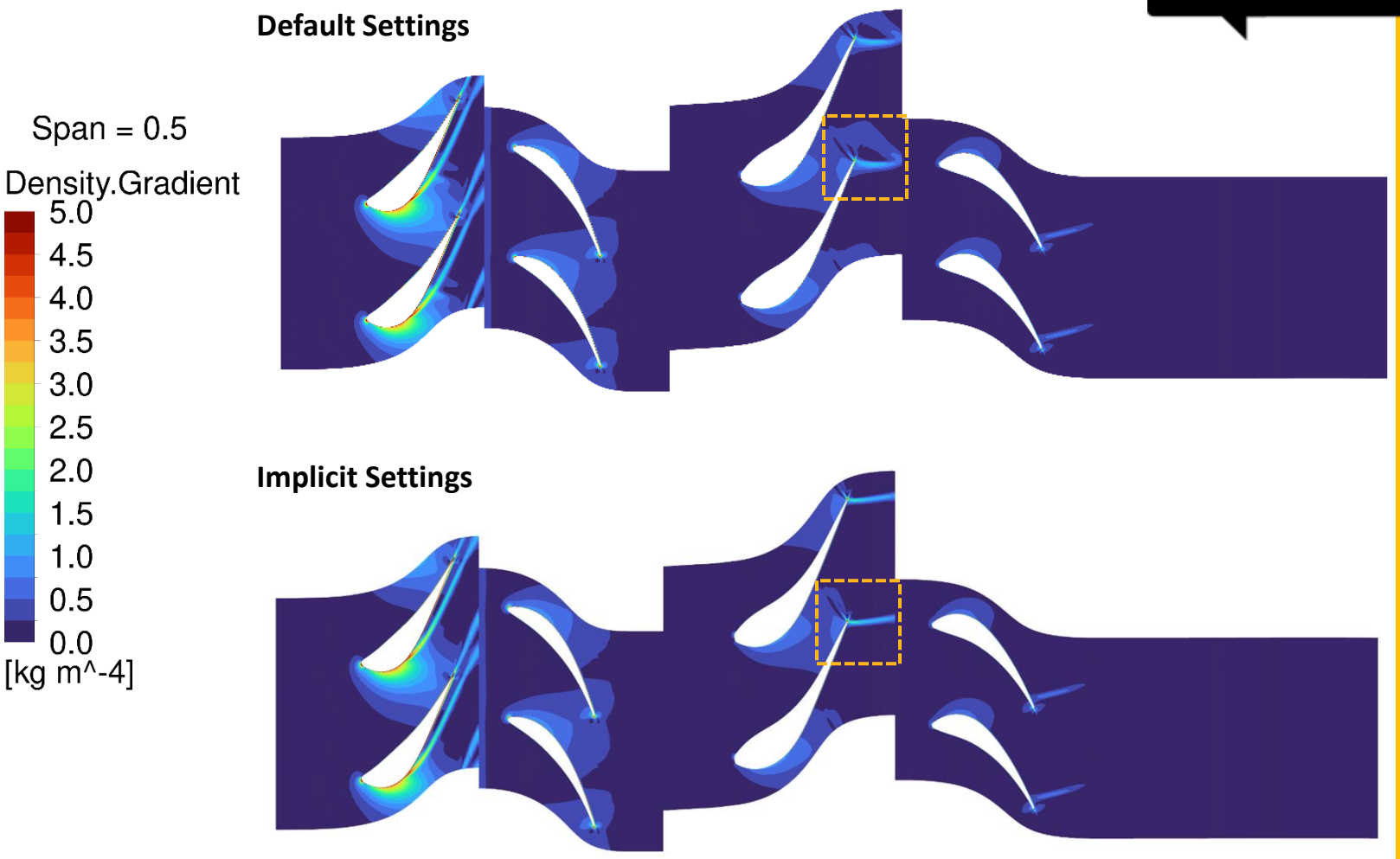
Equilibrium Steam Model (EQS)



Reflections at Mixing Planes – Results STAC Case

Non-Equilibrium Steam (NES)

Will be possible in 2026 R1



The implicit settings for mixing planes have also shown improvements for reflection behavior for the Non-Equilibrium Steam Model.

To run the set up with the implicit settings for mixing planes a much smaller timestep compared to the default settings is needed and hence it takes more iterations for the solution to converge.

STAC Case:
dt default = 1e-03 vs. dt implicit = 1e-05

It is recommended to start with the default settings and to only move to implicit settings if reflections at the mixing planes impact the solution. Default setting solutions can be used for initialization of the calculations using implicit settings at the mixing planes.

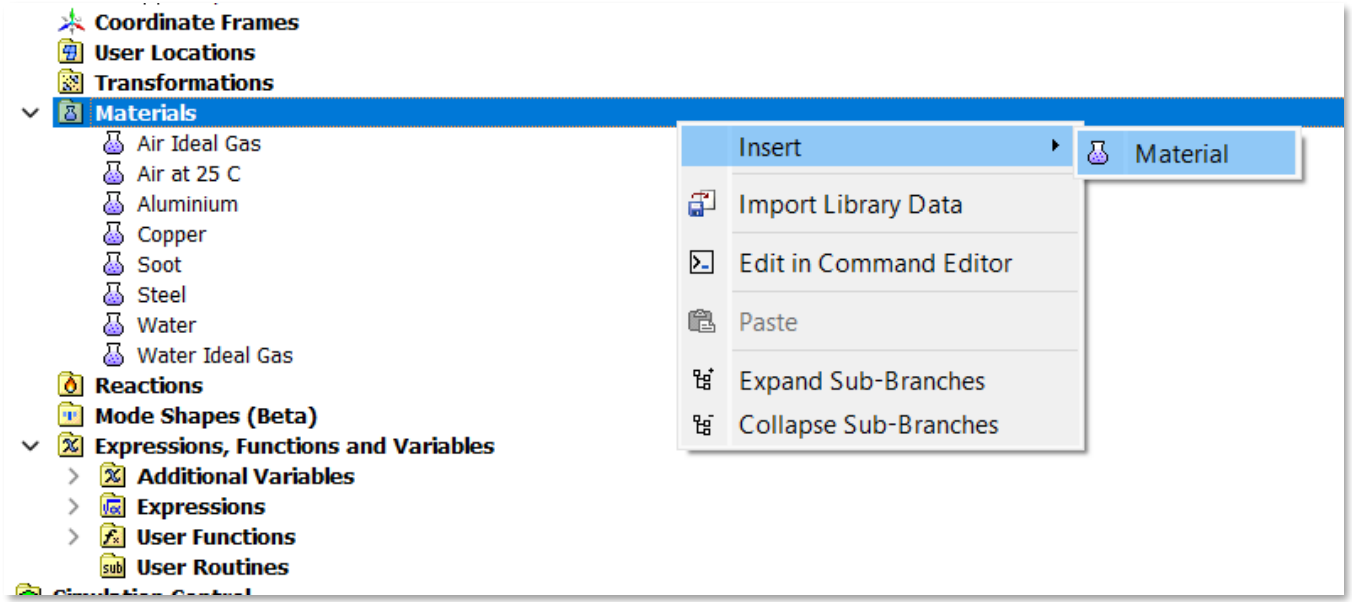
Equilibrium Steam Model (EQS)

Set Up

STAC EQS Set Up
04_Equilibrium_Steam.ccl

Equilibrium Steam Model – Setup in CFX Pre

The Equilibrium Steam Model (EQS) can be applied to a turbine setup in CFX-Pre by defining two distinct materials: one representing the liquid phase of water and the other for the gaseous phase. These two materials are then combined into a homogeneous binary mixture, which is assigned as the domain fluid. A prerequisite for implementing the Equilibrium Steam Model is a turbine setup that successfully operates with water modeled as an ideal gas as the domain fluid.

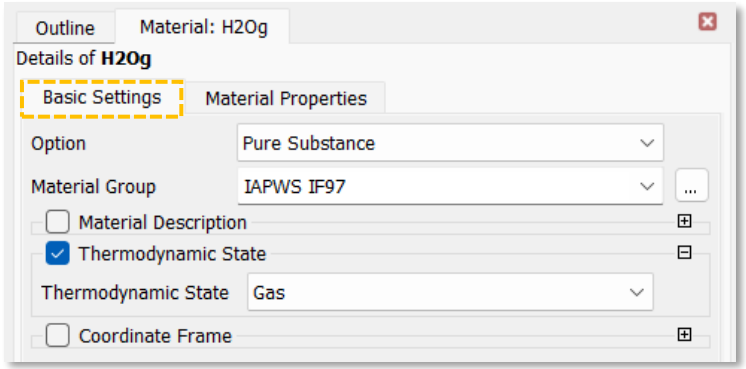


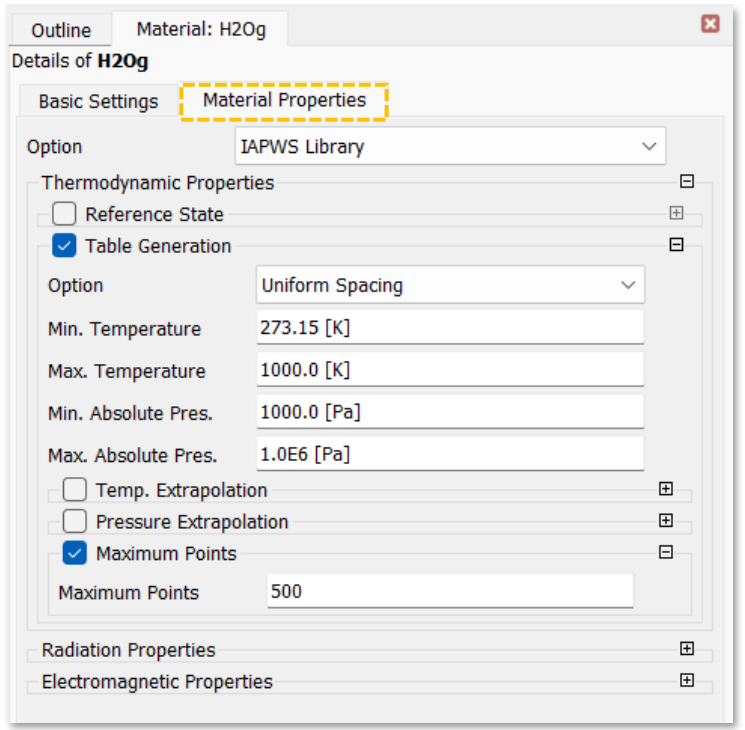
1) Create New Materials: H2Og & H2Ol

→ RMB on Materials → Insert → Material → give name H2Og and H2Ol respectively → OK

Additional Tutorial - Axial Turbine Equilibrium and Non-Equilibrium
Chapter 24: Axial Turbine Equilibrium and Non-Equilibrium Steam Predictions

Equilibrium Steam Model – Setup in CFX Pre





2) Setting Material Properties for H2Og

Basic Settings Tabs

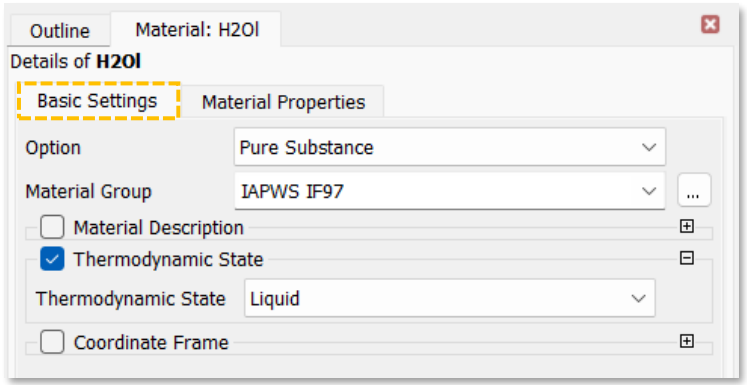
- Option → Pure Substance
- Material Group → IAPWS IF97
  → The IAPWS-IF97 represents an accurate equation of state for water and steam properties
- Activate Thermodynamic State and set it to “Gas”

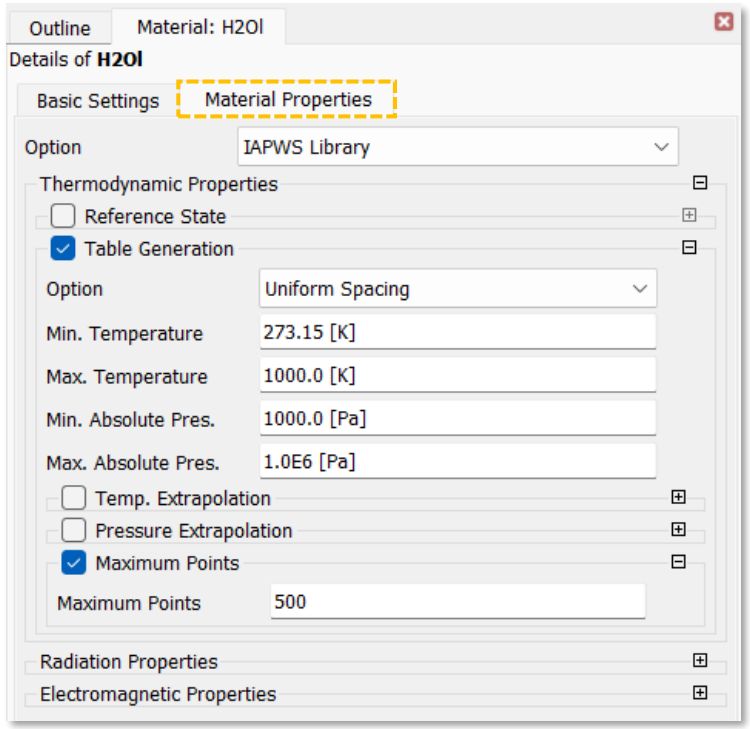
Material Properties Tabs

- Option → IAPWS Library
- Activate Table Generation
  → Will create a real gas property table (RGP) based on the equations of state of IAPWS-IF97 database
  → Table format can be efficiently evaluated during the CFD calculation
  → The generated RGP tables are used to evaluate properties at the current state
- Chose Uniform Spacing as Option
- Setting Property Range: Set a minimum and maximum temperature and pressure
  → If necessary, this range can be adapted later
- Select and set maximum points: around 500 points is a good starting point
- For the treatment of table bounds Temp. and Pressure Extrapolation can be activated so that table bounds are extended if necessary
  - If not activated, then the CFX-Solver will clip properties at the bounds you have supplied

Additional Explanations: 1.4.6. Table Generation Pressure and Temperature Limits

Equilibrium Steam Model – Setup in CFX Pre





3) Setting Material Properties for H2Ol

The settings for the liquid phase are mostly the same as for H2Og.

- Set Thermodynamic State to Liquid
- All other settings are equal to H2Og

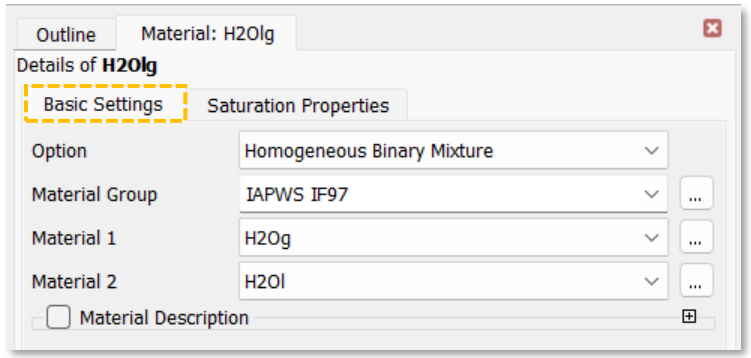
Basic Settings Tab

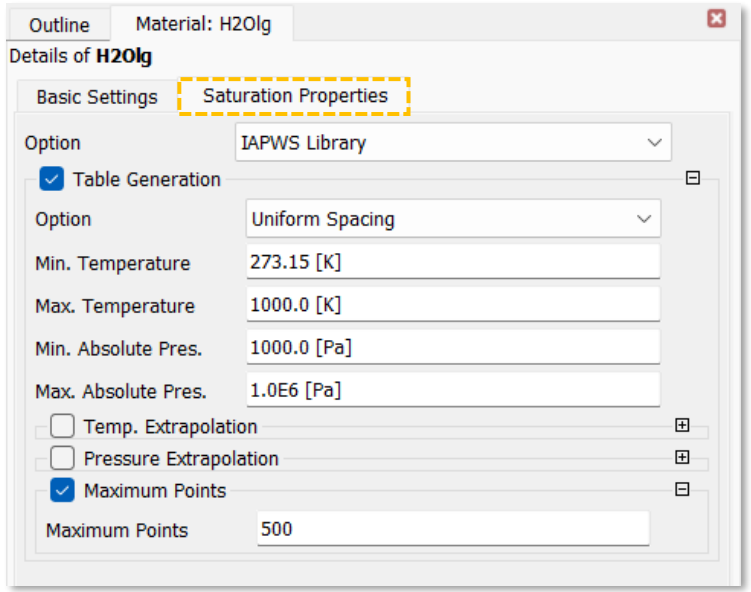
- Option → Pure Substance
- Material Group → IAPWS IF97
- Set Thermodynamic State to Liquid

Material Properties

- Option → IAPWS Library
- Activate Table Generation
- Chose Uniform Spacing as Option
- Setting Property Range: Set a minimum and maximum temperature and pressure
- Select and set maximum points: around 500 points is a good starting point

Equilibrium Steam Model – Setup in CFX Pre





4) Create H2Olg as new Material

Basic Settings Tab

- Option → Homogeneous Binary Mixture
- Material Group → IAPWS IF97
- Material 1 as H2Og
- Material 2 as H2Ol

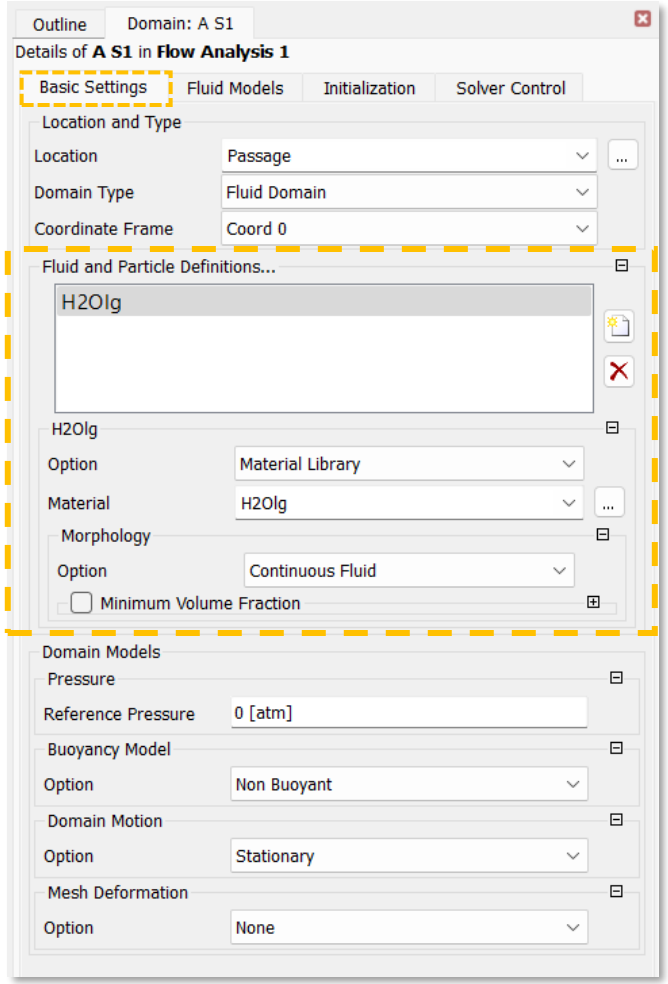
Saturation Properties Tab

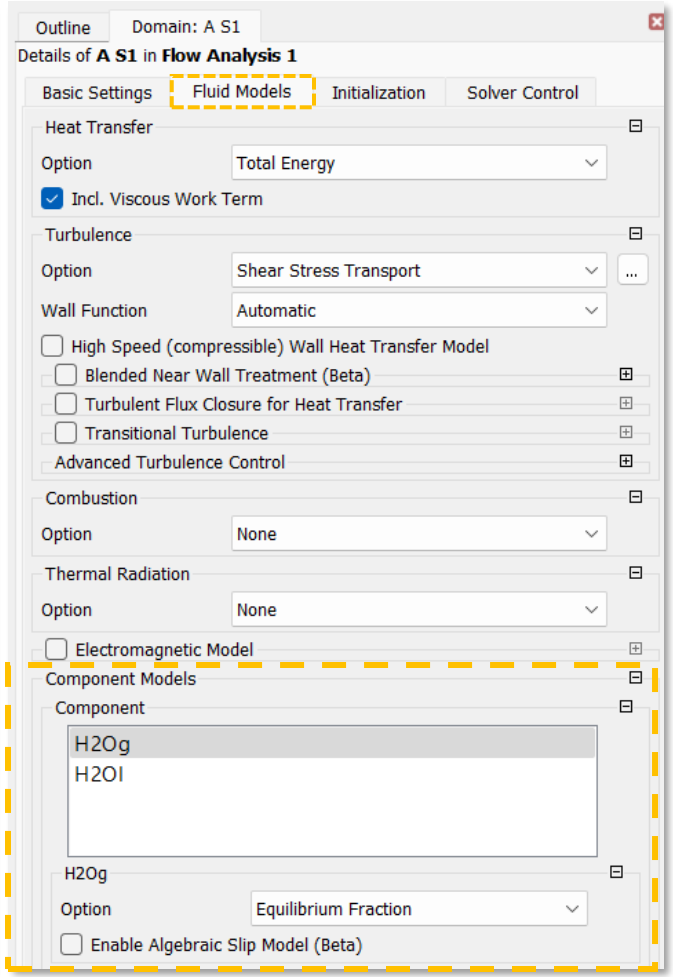
- Option → IAPWS Library
- Activate Table Generation
- Chose Uniform Spacing as Option
- Setting Property Range: Set a minimum and maximum temperature and pressure
- Select and set maximum points: around 500 points is a good starting point

→ This homogeneous binary mixture is a material that will always be in thermodynamic equilibrium

→ The generated table for H2Olg defines the saturation line between the superheated and wet steam region

Equilibrium Steam Model – Setup in CFX Pre





5) Adapting Domain Fluid

- Open one of the domains within the turbine

Basic Settings Tab

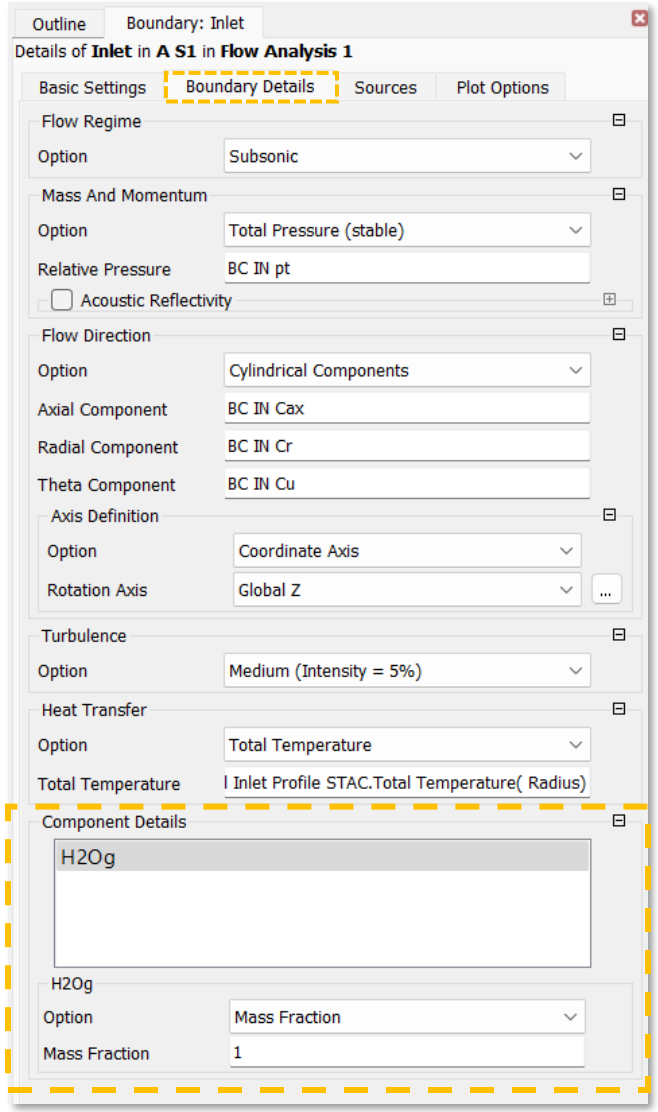
- Fluid and Particle Definitions:
  - Remove current fluid
  - Create a new Fluid and name it H2Olg
  - Assign H2Olg as the Material

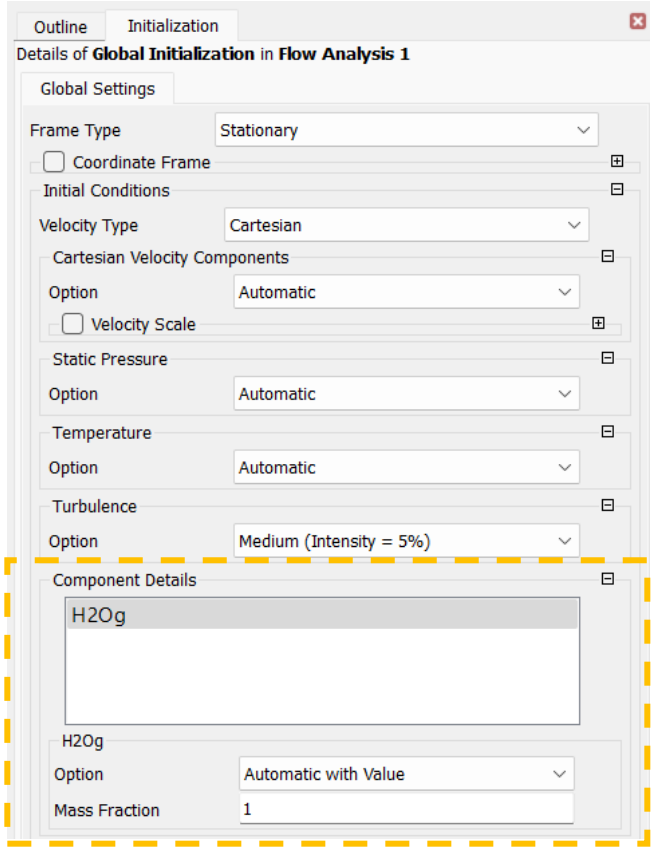
Fluid Models Tab

- Component Models → Chose H2Og
  - Set Option to Equilibrium Fraction
    → This way the Equilibrium Fraction will be the steam quality (if H2Ol is set to Equilibrium Fraction it will be the wetness)
- Click OK

For the default settings the domain settings are automatically taken over for all domains. Therefore, the fluid only needs to be set in one domain (unless constant domain physics is disabled in Case Options → General → Physics Beta Features).

Equilibrium Steam Model – Setup in CFX Pre





6) Setting Inlet Conditions

Chose the inlet boundary

Boundary Details Tab → Component Details

- Chose H2Og
- Set Mass Fraction to 1 (or respective value if wet rather than dry steam enters through the inlet)
- Click OK

7) Setting Initial Values

Click Global Initialization

- Set Initial value for mass fraction of H2Og to 1

8) Run Calculation

- For difficult cases (e.g. the STAC Case) ramping up boundary conditions can be helpful when using the Equilibrium Steam Model

Equilibrium Steam Model – Clipping behavior

During the calculation, the solution properties are evaluated based on the generated RGP tables. Since both the temperature and pressure ranges are defined with upper and lower bounds, the tables cover only a specific range. If, during the calculation, the temperature or pressure falls outside this range, the solver will either clip or extrapolate the affected value, triggering a warning in the output file. If this warning disappears as the run progresses, it indicates that the solution has settled within the defined ranges. However, if the warning persists by the end of the run, it suggests that the ranges of the generated tables need to be extended.

The table bounds warnings can indicate how the tables must be adapted:

- Check Bound → Increase Min. or Max value depending on which bound is affected
- Check if Independent variable is pressure or temperature and adapt according table bounds
- Its best to always adapt the tables of all three materials (H2Ol, H2Og, H2Olg) so that all have an equal range

→ Increase the table bounds step by step as overly large tables can cause convergence issues. The specified range should be somewhat wider than expected but should not be wider than the expected range by a factor much greater than 2.

```
+-------------------------------------------------------------------+
|              Table bounds warnings at: END OF RUN                 |
+-------------------------------------------------------------------+

+-------------------------------------------------------------------+
|                                                                   |
| Independent variables went out of bounds while computing the      |
| variables listed below using table interpolation. In each case    |
| the bounds error was handled by clipping or extrapolation.        |
| If this situation persists, consider increasing the table range.  |
|                                                                   |
+-------------------------------------------------------------------+
|                                                                   |
| Location Name  : D R0                                             |
| Mesh location  : VERTICES                                         |
| Routine        : cal_TTOT                                         |
| Process        :         18                                       |
| Variable Name  : H2Ol.Saturation Entropy                          |
| Ind. Variable  : Absolute Pressure                                |
| Bound          : Upper                                            |
| Max Value      :  5.4962E+06                                      |
| Handled By     : Clipping                                         |
+-------------------------------------------------------------------+
|                                                                   |
| Location Name  : D R0 Shroud                                      |
| Mesh location  : BELG1/IP                                         |
| Routine        : SU_HYB_VAR                                       |
| Process        :         18                                       |
| Variable Name  : H2Og.Density                                     |
| Ind. Variable  : Absolute Pressure                                |
| Bound          : Upper                                            |
| Max Value      :  5.5746E+06                                      |
| Handled By     : Clipping                                         |
+-------------------------------------------------------------------+
|                                                                   |
| Location Name  : D R0                                             |
| Mesh location  : VERTICES                                         |
| Routine        : cal_TTOT                                         |
| Process        :         18                                       |
| Variable Name  : H2Ol.Density                                     |
| Ind. Variable  : Absolute Pressure                                |
| Bound          : Upper                                            |
| Max Value      :  5.4962E+06                                      |
| Handled By     : Clipping                                         |
|                                                                   |
+-------------------------------------------------------------------+
```

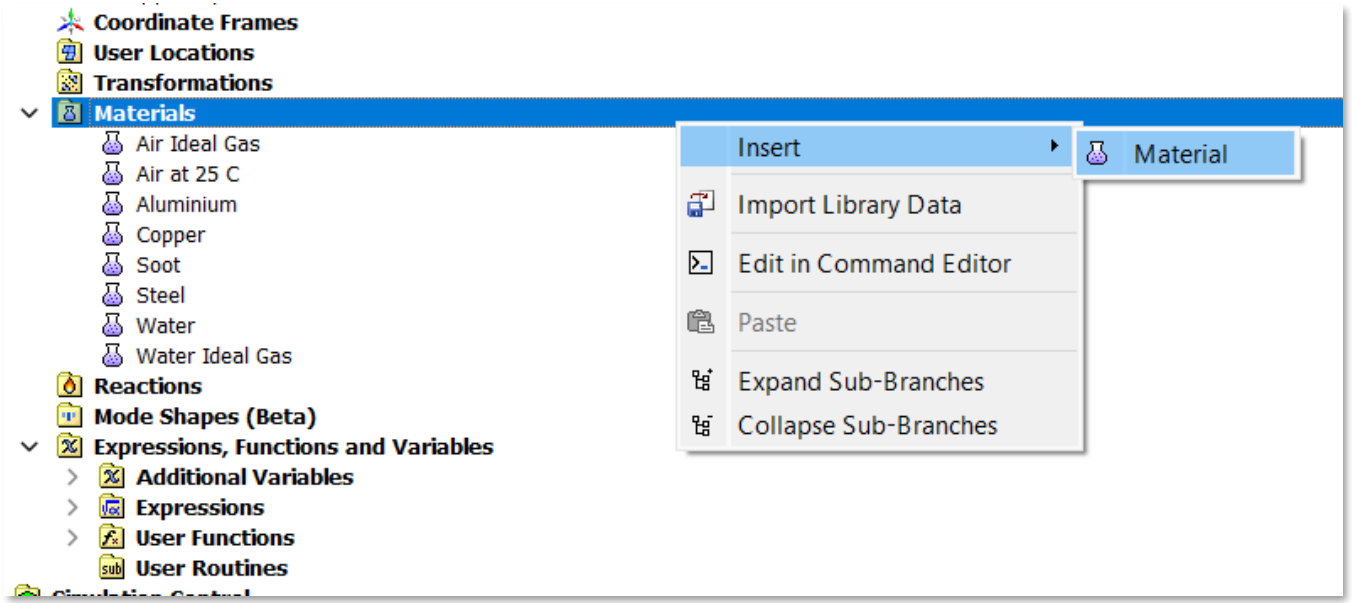
Non-Equilibrium Steam Model (NES)

Set Up

STAC NES Set Up
05_Non_Equilibrium_Steam.ccl

Non-Equilibrium Steam Model – Setup in CFX Pre

For the Non-Equilibrium Steam Model (NES) two distinct materials, one representing the liquid phase of water and the other for the gaseous phase, are defined in CFX Pre. The settings of these materials are the same as for the Equilibrium Steam model, however, both materials are then introduced as fluids to the domain instead of creating a binary mixture of two components. A prerequisite for implementing the Equilibrium Steam Model is a turbine setup that successfully operates with water modeled as an ideal gas as the domain fluid.

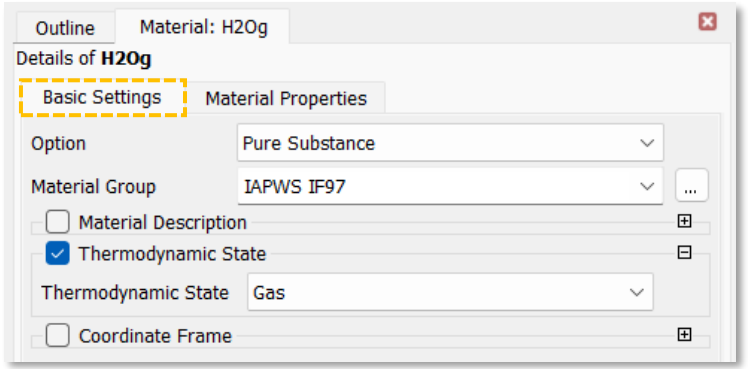


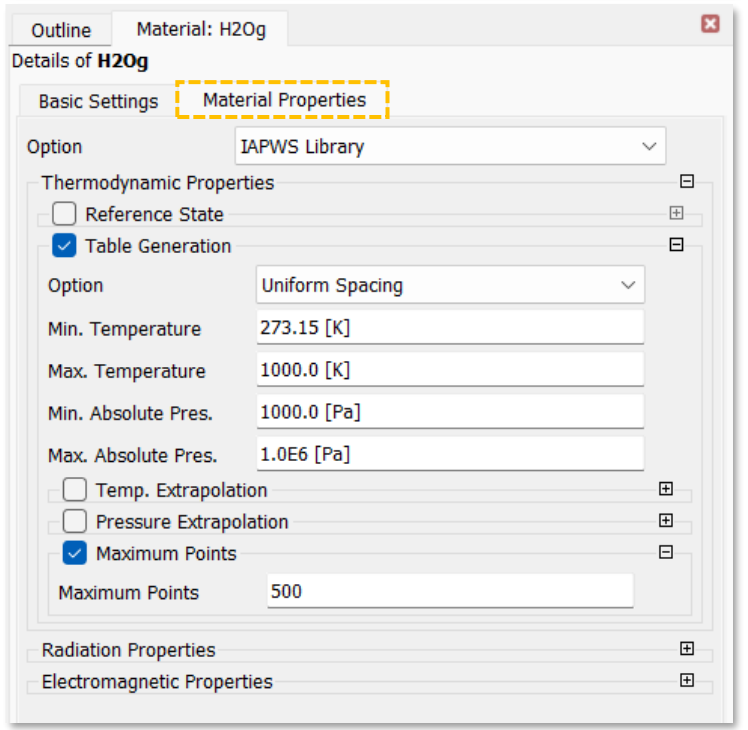
1) Create New Materials: H2Og & H2Ol

→ RMB on Materials → Insert → Material → give the name H2Og or H2Ol → OK

Additional Tutorial - Axial Turbine Equilibrium and Non-Equilibrium
Chapter 24: Axial Turbine Equilibrium and Non-Equilibrium Steam Predictions

Non-Equilibrium Steam Model – Setup in CFX Pre





2) Setting Material Properties for H2Og

Basic Settings Tabs

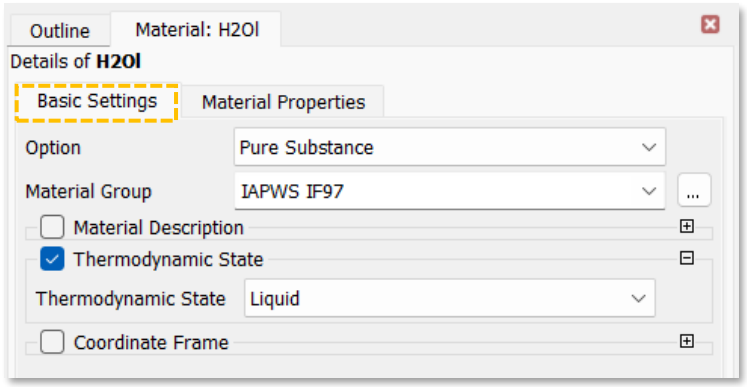
- Option → Pure Substance
- Material Group → IAPWS IF97
  → The IAPWS-IF97 represents an accurate equation of state for water and steam properties
- Activate Thermodynamic State and set it to “Gas”

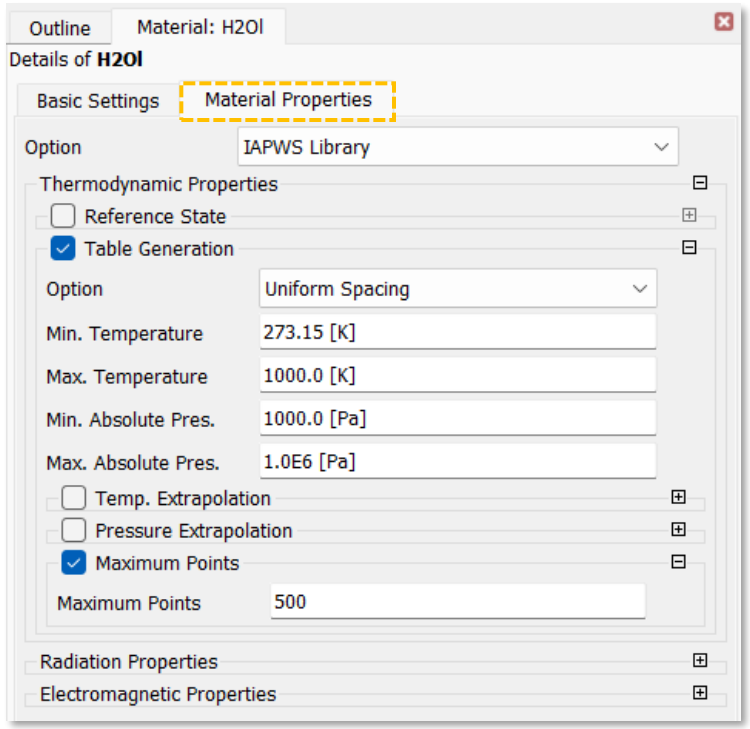
Material Properties Tabs

- Option → IAPWS Library
- Activate Table Generation
  → Will create a real gas property table (RGP) based on the equations of state of IAPWS-IF97 database
  → Table format can be efficiently evaluated during the CFD calculation
  → The generated RGP tables are used to evaluate properties at the current state
- Chose Uniform Spacing as Option
- Setting Property Range: Set a minimum and maximum temperature and pressure
  → If necessary, this range can be adapted later
- Select and set maximum points: around 500 points is a good starting point
- For the treatment of table bounds Temp. and Pressure Extrapolation can be activated so that table bounds are extended if necessary
  - If not activated, then the CFX-Solver will clip properties at the bounds you have supplied

Additional Explanations: 1.4.6. Table Generation Pressure and Temperature Limits

Non-Equilibrium Steam Model – Setup in CFX Pre





3) Setting Material Properties for H2Ol

The settings for the liquid phase are mostly the same as for H2Og.

- Set Thermodynamic State to Liquid
- All other settings are equal to H2Og

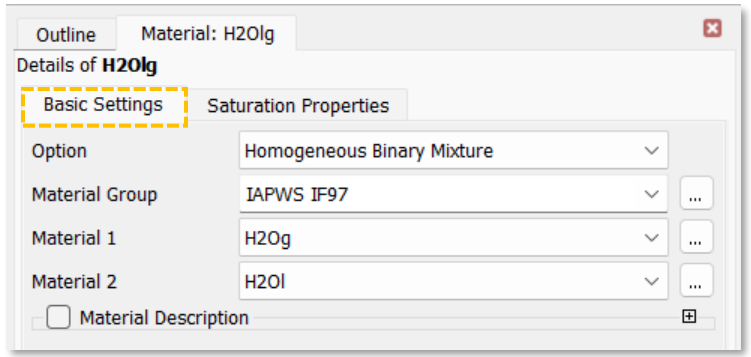
Basic Settings Tab

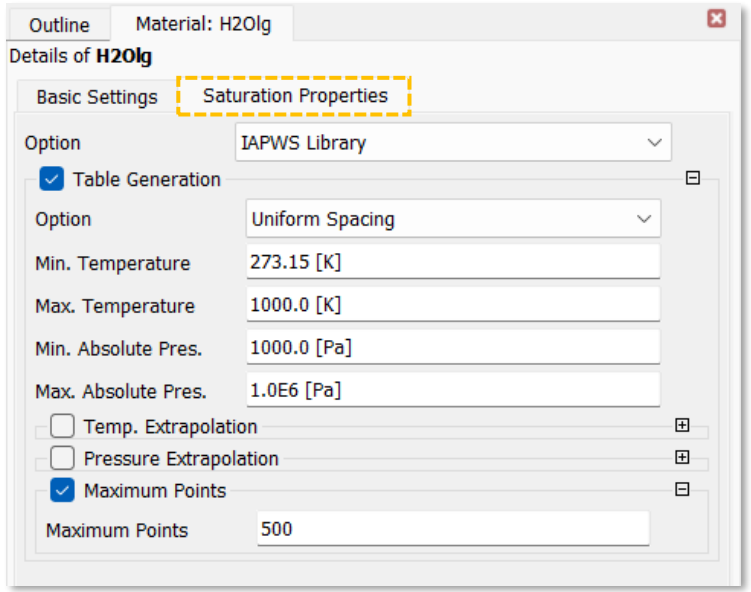
- Option → Pure Substance
- Material Group → IAPWS IF97
- Set Thermodynamic State to Liquid

Material Properties

- Option → IAPWS Library
- Activate Table Generation
- Chose Uniform Spacing as Option
- Setting Property Range: Set a minimum and maximum temperature and pressure
- Select and set maximum points: around 500 points is a good starting point

Non-Equilibrium Steam Model – Setup in CFX Pre





4) Create H2Olg as new Material

Basic Settings Tab

- Option as Homogeneous Binary Mixture
- Material Group as IAPWS IF97
- Material 1 as H2Og
- Material 2 as H2Ol

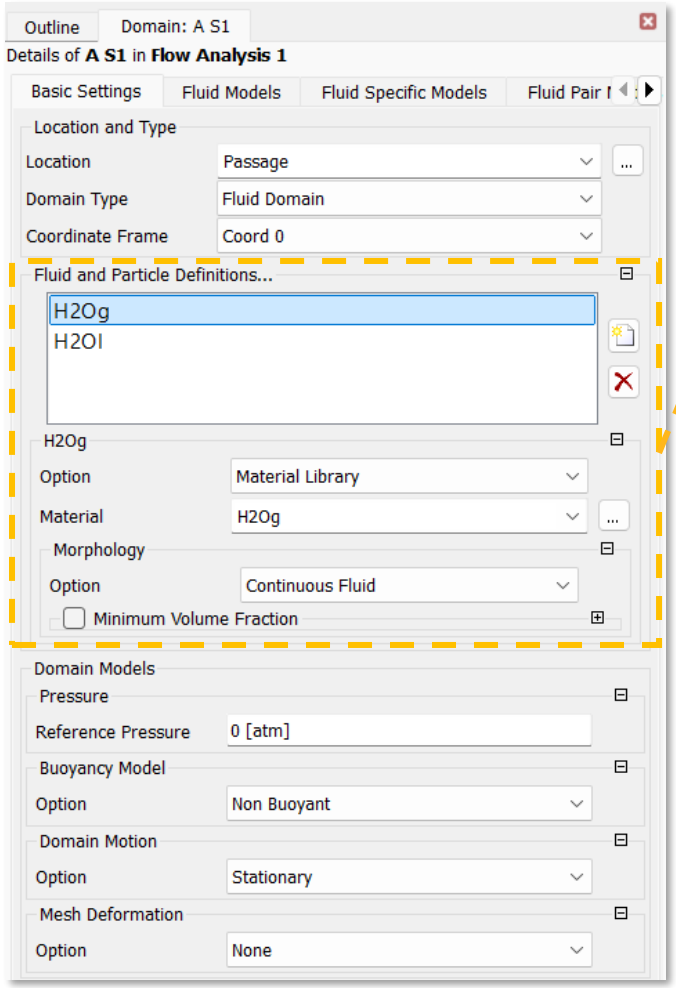
Saturation Properties Tab

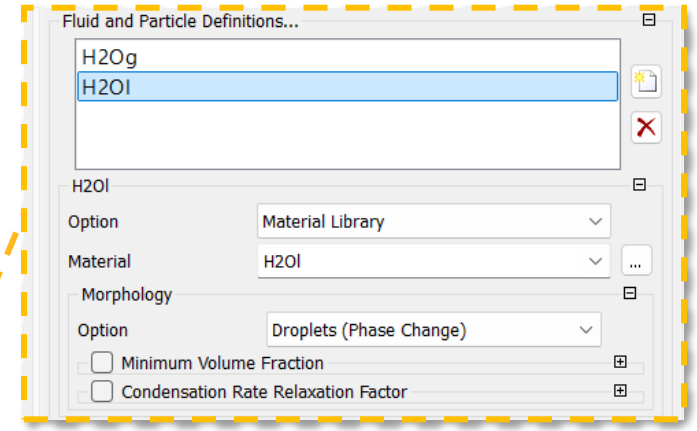
- Option → IAPWS Library
- Activate Table Generation
- Chose Uniform Spacing as Option
- Setting Property Range: Set a minimum and maximum temperature and pressure
- Select and set maximum points: around 500 is a good starting point

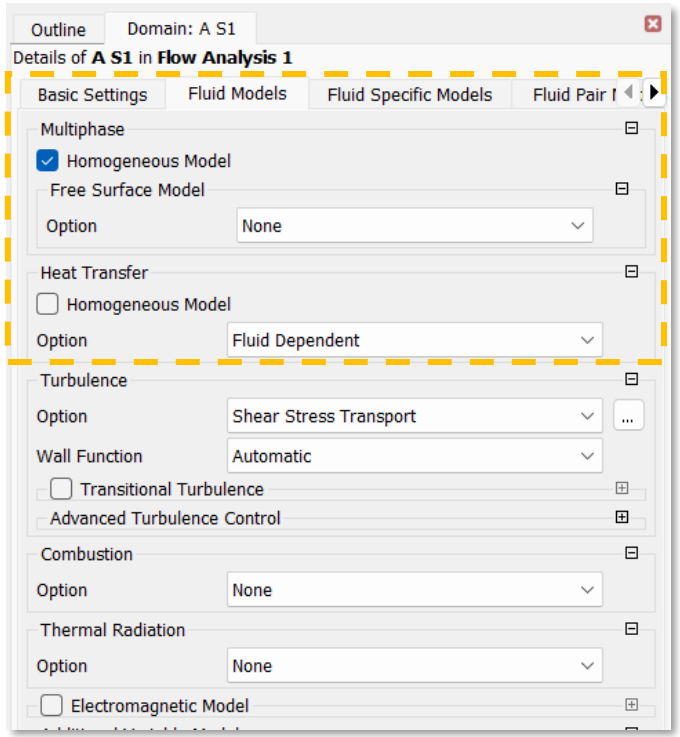
→ This homogeneousbinary mixture is a material that will always be in thermodynamic equilibrium

→ The generated table for H2Olg defines the saturation line between the superheated and wet steam region

Non-Equilibrium Steam Model – Setup in CFX Pre







5) Adapting Domain Fluid

- Open one of the domains within the turbine

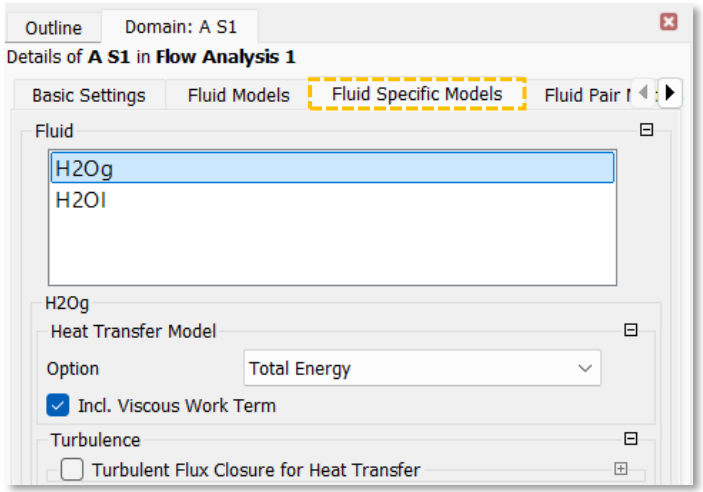
Basic Settings Tab

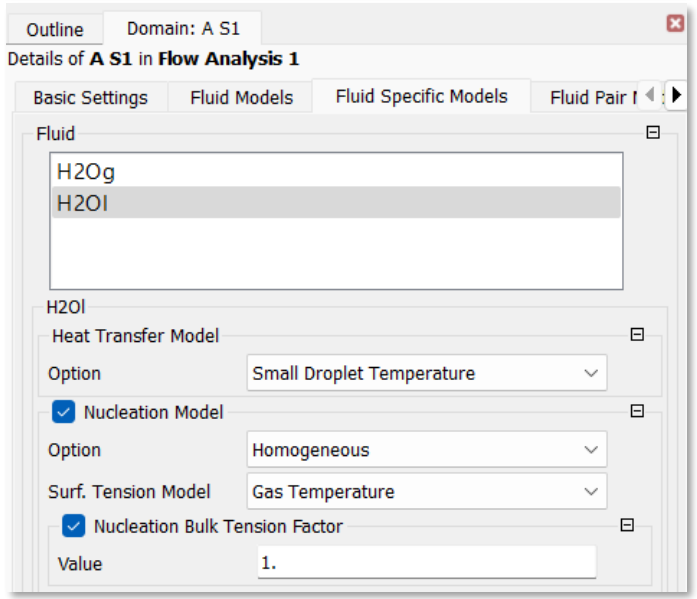
- Fluid and Particle Definitions:
  - Remove current fluid
  - Create a new Fluid and name it H2Og
    - Assign H2Og as the Material
    - Set the Morphology → Option: Continuous Fluid
  - Create an additional Fluid named H2Ol
    - Assign H2Ol as the Material
    - Set the Morphology → Option: Droplets (Phase Change)
    - Optionally a default Minimum Volume Fraction can be set. When not set manually a default of 10^{-15} is set – this is done for numerical robustness as volume fractions can not be machine zero
  - Click Apply

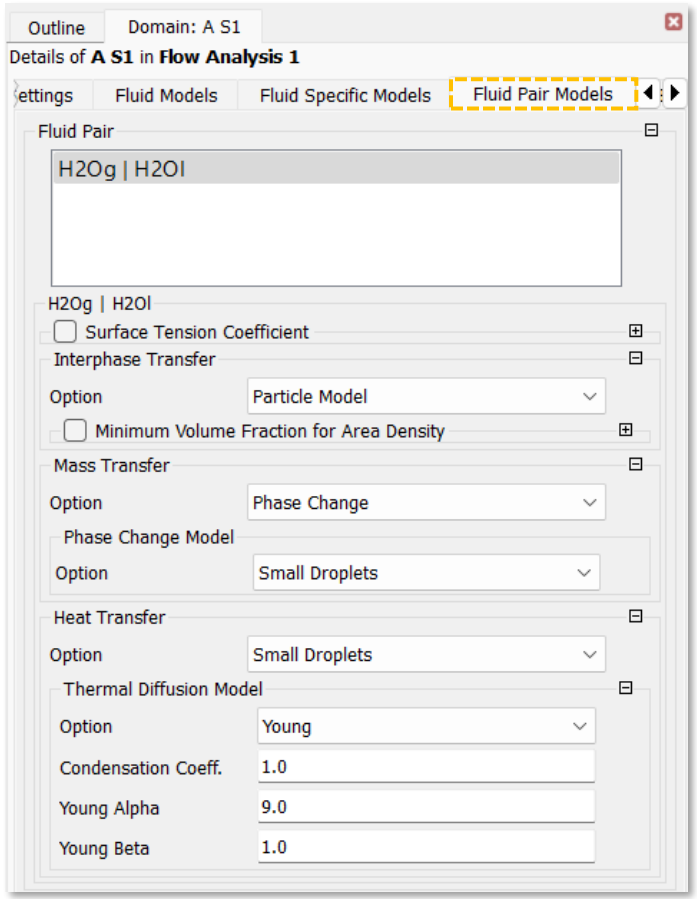
Fluid Models Tab

- Select Multiphase → Homogeneous Model
- Deselect Heat Transfer option “Homogeneous Model”
- Set Heat Transfer → Option: Fluid Dependent

Non-Equilibrium Steam Model – Setup in CFX Pre







The domain settings will automatically transfer to all domains except for the Nucleation Model. The Nucleation Model settings must be set explicitly for all domains.

5) Adapting Domain Fluid

Fluid Specific Models Tab

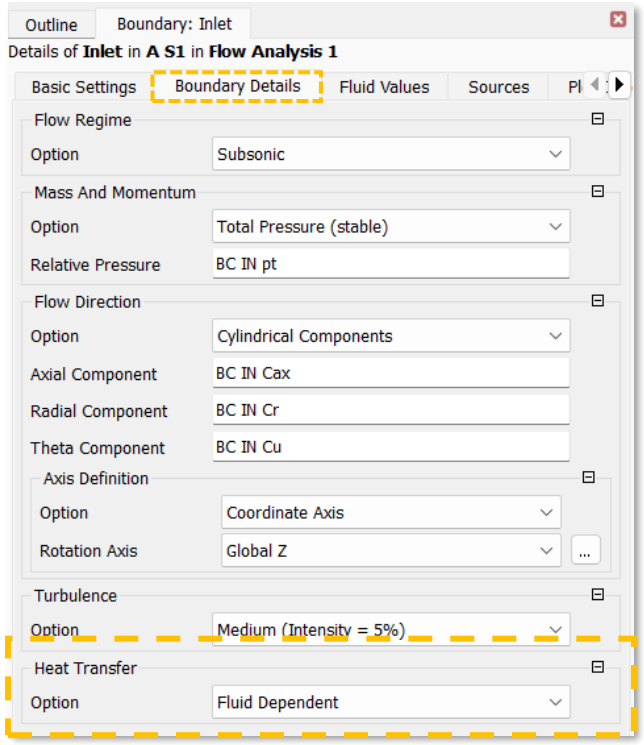
- Select H2Og
  - Set Heat Transfer Model → Total Energy
- Select H2Ol
  - Set Heat Transfer Model → Small Droplet Temperature
  - Select Nucleation Model
  - Set Nucleation Model → Option: Homogeneous
  - Select Nucleation Bulk Tension Factor
  - Set Nucleation Bulk Tension Factor to 1

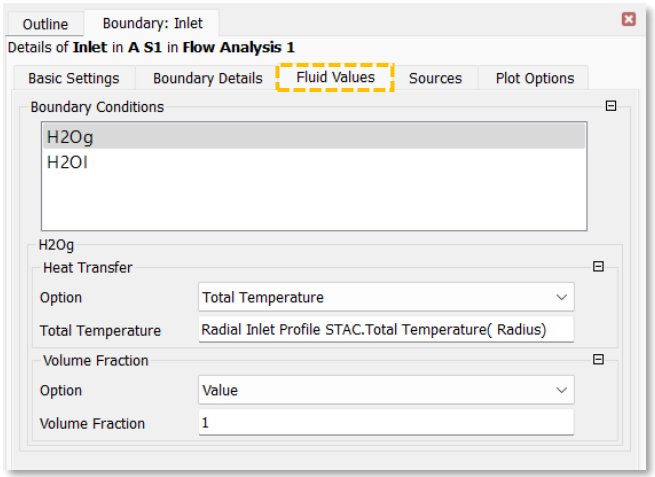
  → Caution: The Nucleation must be set up for all domains individually

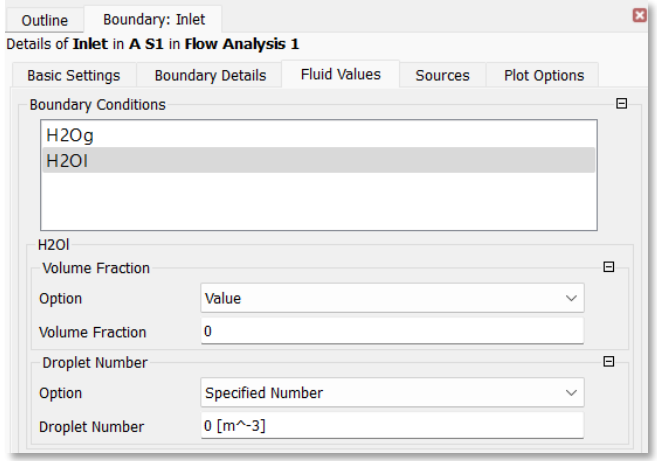
Fluid Pair Models Tab

- Set Interphase Transfer → Option: Particle Model
- Set Mass Transfer → Option: Phase Change
- Set Mass Transfer → Phase Change Model: Small Droplets
- Set Heat Transfer → Option: Small Droplets
- Suggested to keep the “Thermal Diffusion Model” as the Young Model with default values

Non-Equilibrium Steam Model – Setup in CFX Pre







6) Setting Inlet Conditions

Chose the inlet boundary

Boundary Details Tab

- Set Heat Transfer → Option: Fluid Dependent

Fluid Values Tab

- Select H2Og
  - Heat Transfer → Option: Total Temperature
  - Set Total Temperature Value
  - Volume Fraction → Option: Value
  - Set Volume Fraction to 1
- Select H2Ol
  - Volume Fraction → Option: Value
  - Set Volume Fraction to 0
  - Droplet Number → Option: Specified Number
  - Set Droplet Number to 0

→ These are the settings for dry steam entering the inlet, hence no liquid component. If the wet steam enters the domain through the inlet the volume fraction and droplet number values need to be set accordingly.

7) Initialization and Run

Details on initialization shown in the following sections

Non-Equilibrium Steam Model – Clipping behavior

The clipping behavior for the Non-Equilibrium Steam Model (NES) is the same as for the Equilibrium Steam model (described on slide 39). If clipping warnings are still printed at the end of the run and pressure and temperature values lie outside the table ranges, the respective tables should be adapted.

The table bounds warnings can indicate how the tables must be adapted:

- Check Bound → Increase Min. or Max value depending on which bound is affected
- Check if Independent variable is pressure or temperature and adapt according table bounds
- Its best to always adapt the tables of all three materials (H2Ol, H2Og, H2Olg) so that all have an equal range

→ Increase the table bounds step by step as overly large tables can cause convergence issues. The specified range should be somewhat wider than expected but should not be wider than the expected range by a factor much greater than 2.

```
+-------------------------------------------------------------------+
|              Table bounds warnings at: END OF RUN                 |
+-------------------------------------------------------------------+

+-------------------------------------------------------------------+
|                                                                   |
| Independent variables went out of bounds while computing the      |
| variables listed below using table interpolation. In each case    |
| the bounds error was handled by clipping or extrapolation.        |
| If this situation persists, consider increasing the table range.  |
|                                                                   |
+-------------------------------------------------------------------+
|                                                                   |
| Location Name  : D R0                                             |
| Mesh location  : VERTICES                                         |
| Routine        : cal_TTOT                                         |
| Process        :         18                                       |
| Variable Name  : H2Ol.Saturation Entropy                          |
| Ind. Variable  : Absolute Pressure                                |
| Bound          : Upper                                            |
| Max Value      :  5.4962E+06                                      |
| Handled By     : Clipping                                         |
+-------------------------------------------------------------------+
|                                                                   |
| Location Name  : D R0 Shroud                                      |
| Mesh location  : BELG1/IP                                         |
| Routine        : SU_HYB_VAR                                       |
| Process        :         18                                       |
| Variable Name  : H2Og.Density                                     |
| Ind. Variable  : Absolute Pressure                                |
| Bound          : Upper                                            |
| Max Value      :  5.5746E+06                                      |
| Handled By     : Clipping                                         |
+-------------------------------------------------------------------+
|                                                                   |
| Location Name  : D R0                                             |
| Mesh location  : VERTICES                                         |
| Routine        : cal_TTOT                                         |
| Process        :         18                                       |
| Variable Name  : H2Ol.Density                                     |
| Ind. Variable  : Absolute Pressure                                |
| Bound          : Upper                                            |
| Max Value      :  5.4962E+06                                      |
| Handled By     : Clipping                                         |
|                                                                   |
+-------------------------------------------------------------------+
```

Non-Equilibrium Steam Model

Calculation Startup

Using a Dynamic Variable for NES

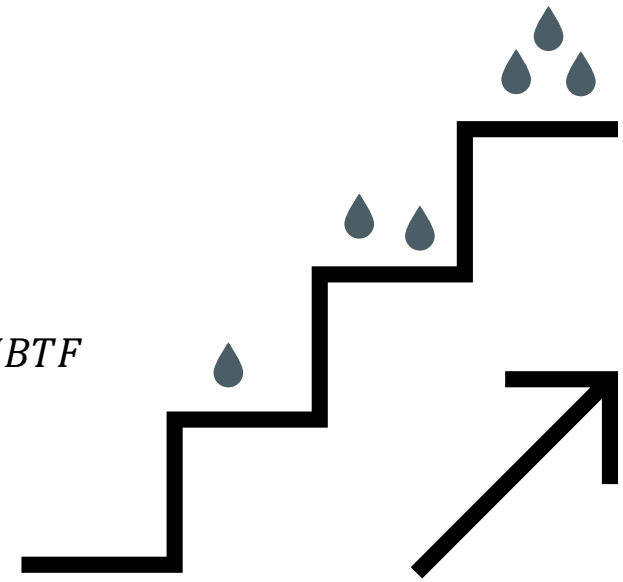
Introducing a nucleation model and thereby allowing droplet formation can cause instabilities and it can be difficult to start a stable calculation. To stabilize the solver, the nucleation rate can be initially limited and gradually increased step by step towards the final value using a dynamic variable.

Nucleation Rate

This is the rate at which critical droplets form per unit volume and time. Droplets form from the supersaturated vapor in absence of impurities.

$$J_d = \frac{q_c}{(1+\theta)} \left(\frac{\rho_v^2}{\rho_l}\right) \sqrt{\frac{2\sigma}{M_m^3}}\, e^{-\left(\frac{4\pi r_*^2 \sigma}{3K_b T}\right)}$$

- Depends on the surface tension σ (negative exponential relationship)
- The surface tension can be corrected using the Nucleation Bulk Tension Factor (NBTF): $\sigma \cdot NBTF$
- High NBTF → Low Nucleation Rate

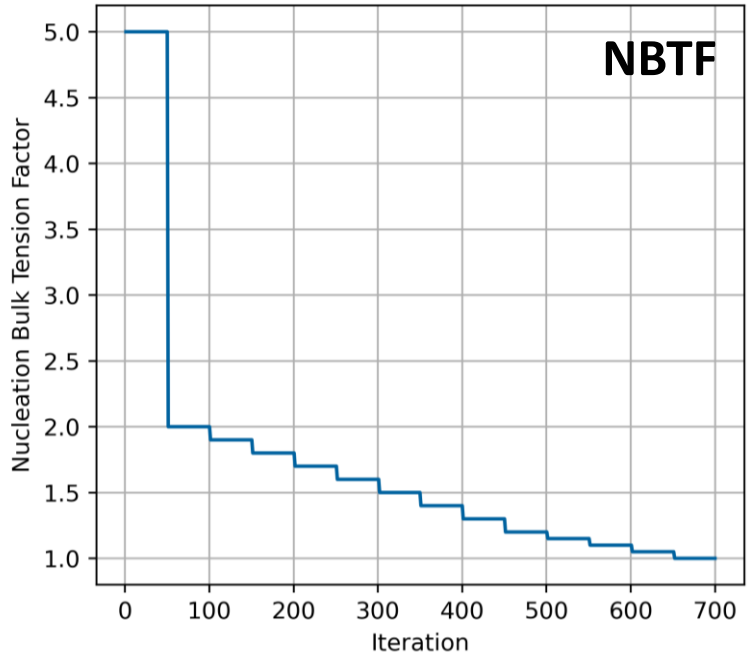


Using a Dynamic Variable for NES

Nucleation Rate

$$J_d = \frac{q_c}{(1+\theta)} \left(\frac{\rho_v^2}{\rho_l}\right) \sqrt{\frac{2\sigma}{M_m^3}}\, e^{-\left(\frac{4\pi r_*^2 \sigma}{3K_b T}\right)}$$

$\sigma \cdot NBTF$



Nucleation Bulk Tension Factor (NBTF)

- To stabilize calculations, start with a low nucleation rate
- Hence, start with a high NBTF at first and reduce it step by step to allow more nucleation

★ **New Feature in 2025 R1**

Dynamic Variable

- A parameter that automatically adjusts its value during the solution convergence process in discrete steps (control steps) until it reaches a value of unity
- The specified convergence conditions must be met before advancing to the next control step
- Functions as a dimensionless scaling factor
- Aids in stabilizing the solution process
- Once final step of unity is reached the dynamic variable no longer has an effect

→ Introduce the Nucleation Bulk Tension Factor as Dynamic Variable

Using a Dynamic Variable for NES

Nucleation Rate

$$J_d = \frac{q_c}{(1+\theta)} \left(\frac{\rho_v^2}{\rho_l}\right) \sqrt{\frac{2\sigma}{M_m^3}}\, e^{-\left(\frac{4\pi r_*^2 \sigma}{3K_b T}\right)} \rightarrow \sigma \cdot NBTF$$

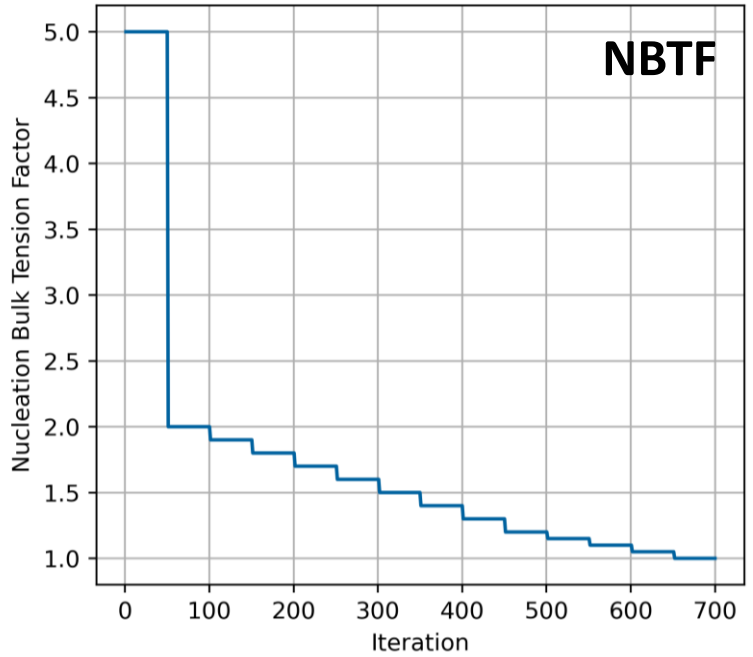
Nucleation Bulk Tension Factor (NBTF)

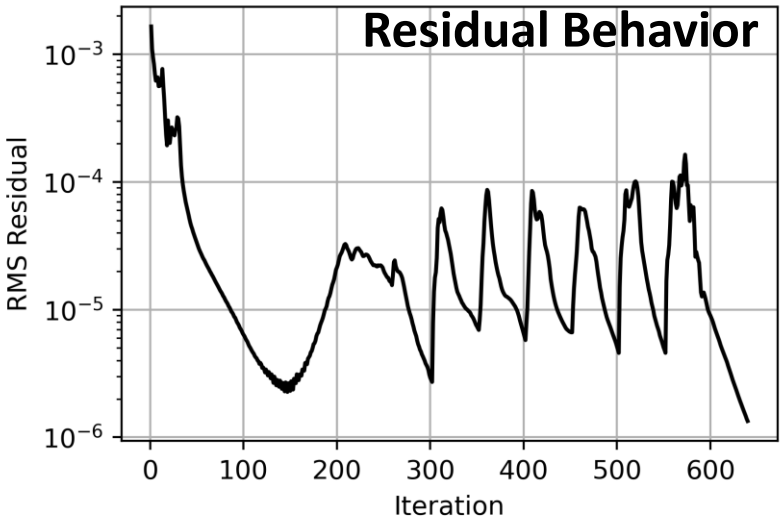
- To stabilize calculations, start with a low nucleation rate
- Hence, start with a high NBTF at first and reduce it step by step to allow more nucleation

Introduce a Dynamic Variable for NBTF

- Step by step reduce NBTF towards one
- The NBTF decreases once the residuals have decreased below a threshold, or a maximum number of iterations is reached
- The dynamic variable NBTF is assigned to nucleation model to limit the nucleation rate

New Feature in 2025 R1





Dynamic Variable for NBTF – How to set it up?

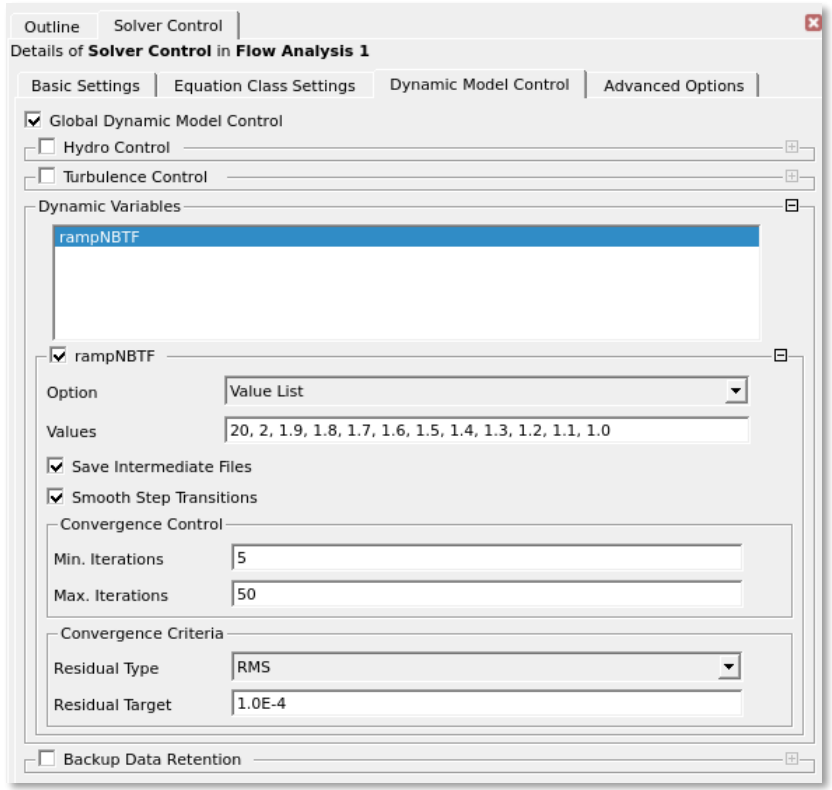
1) Add an additional variable as dynamic variable:

Expressions, Functions and Variables → RMB on Additional Variables: Insert → Additional Variable → Variable Type: Dynamic Variable

2) Specify the Dynamic Variable

Solver Control → Dynamic Model Control

- Activate the dynamic variable that was created
- Set the option for the dynamic variable to a values list or an exponential scale and specify steps or number of steps
- Set Min. and Max. iterations
- Set Residual Target



Dynamic Variable for NBTF – How to set it up?

Exponential Scale

```
DYNAMIC MODEL CONTROL:
    Global Dynamic Model Control = On
    SMALL DROPLETS CONTROL:
      Option = Exponential Scale
      Number of Control Steps = 10
      Initial Factor = 20
      CONVERGENCE CONTROL:
        Maximum Number of Iterations = 15
        Minimum Number of Iterations = 5
        Physical Timescale = 1E-4 [s]
        Timescale Control = Physical Timescale
      END
      CONVERGENCE CRITERIA:
        Residual Target = 1E-3
        Residual Type = RMS
      END
    END
  END
```

Value List

```
DYNAMIC MODEL CONTROL:
    Global Dynamic Model Control = On
    SMALL DROPLETS CONTROL:
      Option = Factor List
      Factors = 20, 2, 1.8, 1.7, 1.6, 1.5, 1.4, 1.3, 1.2, 1.1, 1
      CONVERGENCE CONTROL:
        Maximum Number of Iterations = 15
        Minimum Number of Iterations = 5
        Physical Timescale = 1E-4 [s]
        Timescale Control = Physical Timescale
      END
      CONVERGENCE CRITERIA:
        Residual Target = 1E-3
        Residual Type = RMS
      END
    END
  END
```

The dynamic variable for the NBTF should be defined either as an exponential scale or as a list of values.

For both approaches, the initial value should be set relatively high to minimize nucleation and allow the calculation to stabilize initially, with little to no condensation.

Subsequent step sizes should decrease as the value approaches 1. This reduction occurs automatically with the exponential scale. For a value list, this must be manually implemented, but it offers greater flexibility in adding or removing steps at specific points.

The residual target specifies the threshold that the residuals must reach before the dynamic variable progresses to the next value.

The maximum number of iterations indicates the number of iterations after which the dynamic variable will move to the next value, even if the residual threshold has not been reached.

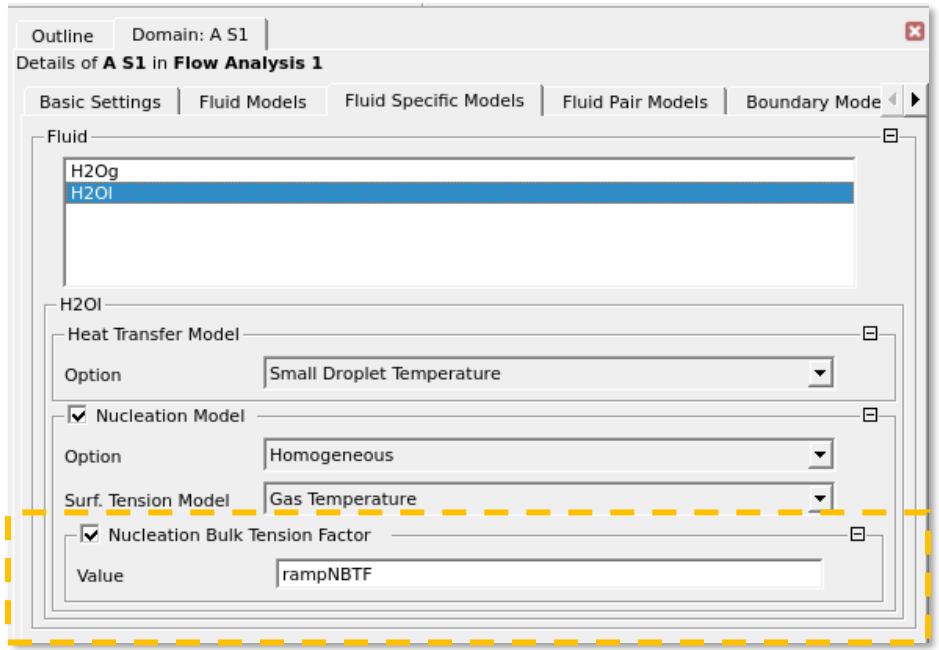
Dynamic Variable for NBTF – How to set it up?

3) Assign Dynamic Variable to Nucleation Model

Domain → Fluid Specific Models → Liquid Phase

- Set the Nucleation Bulk Tension factor to the dynamic variable created
- Might require name to be typed (may not be shown in list)

CAUTION: Nucleation needs to be set for every domain separately. Settings are not taken over from one to all domains.



4) To **monitor the current value of NBTF** create a monitor point for the dynamic variable

Methods to Initialize NES Calculations

Getting a NES calculations started without the solver crashing may not always be straightforward. It might be necessary to incorporate intermediate steps before running the NES calculations on a fine mesh. Initializing with an ideal gas solution and possibly run the NES setup on a coarse mesh first can be helpful.

For the STAC test case 2 methods were successful. The results of the previous step are used to initialize the next.

1) **Ideal Gas Coarse Mesh** → **NES Coarse Mesh** → **NES Refined Mesh**

Start by ramping up boundary conditions

Set NBTF as dynamic variable and reduce step by step until 1 is reached

Set NBTF = 1

2) **Ideal Gas Refined Mesh** → **NES Refined Mesh**

Start by ramping up boundary conditions

Set NBTF as dynamic variable and reduce step by step until 1 is reached

Note: For this second approach smaller steps were required for the NBTF approaching 1 compared to method 1

NBTF: Nucleation Bulk Tension Factor
→ allows limiting the nucleation rate

Tips for Problems with NES Calculations

Early Solver Crash

→ Begin with higher NBTF Value

A higher initial factor means the nucleation rate is further limited, enabling the solver to stabilize with minimal to no condensation.

→ Test different timesteps: smaller and larger

In general, we recommend to start with a timestep around one magnitude smaller than a timestep which proved successful for initial gas calculations with the same set up. However, if this leads to a solver crash, start by testing if a smaller or a larger timestep works.

```
DYNAMIC MODEL CONTROL:
  Global Dynamic Model Control = On
  SMALL DROPLETS CONTROL:
    Option = Factor List
    Factors = 20, 2, 1.8, 1.7, 1.6, 1.5, 1.4, 1.3, 1.2, 1.1, 1
    CONVERGENCE CONTROL:
      Maximum Number of Iterations = 15
      Minimum Number of Iterations = 5
      Physical Timescale = 1E-4 [s]
      Timescale Control = Physical Timescale
    END
    CONVERGENCE CRITERIA:
      Residual Target = 1E-3
      Residual Type = RMS
    END
  END
END
```

Tips for Problems with NES Calculations

Late Solver Crash

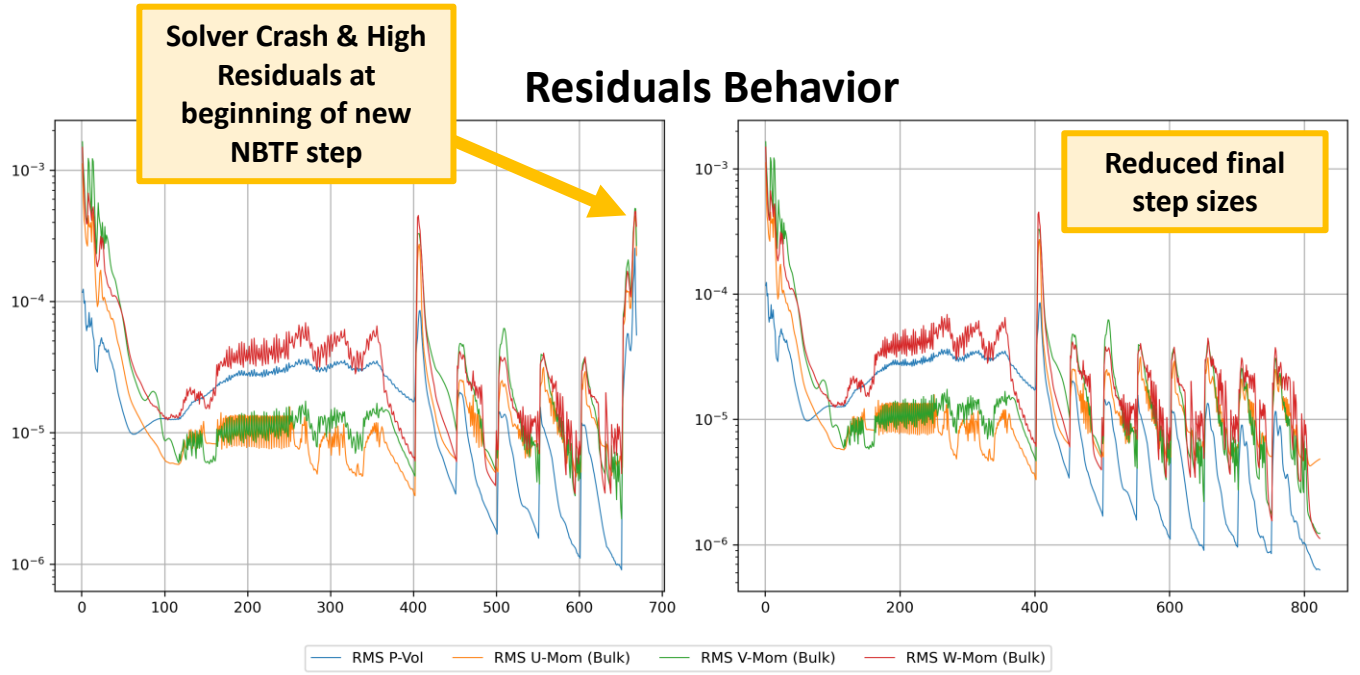
→ Increase the max. number of iterations for dynamic variable if residuals are not decreasing sufficiently within one step

→ If residuals become unexpectedly high after new NBTF step decrease step size to next NBTF value. Add an additional factor in-between (factor list) / increase the number of control steps (exponential scale).

```
DYNAMIC MODEL CONTROL:
  Global Dynamic Model Control = On
  SMALL DROPLETS CONTROL:
    Option = Factor List
    Factors = 20, 2, 1.8, 1.7, 1.6, 1.5, 1.4, 1.3, 1.2, 1.1, 1
    CONVERGENCE CONTROL:
      Maximum Number of Iterations = 15
      Minimum Number of Iterations = 5
      Physical Timescale = 1E-4 [s]
      Timescale Control = Physical Timescale
    END
    CONVERGENCE CRITERIA:
      Residual Target = 1E-3
      Residual Type = RMS
    END
  END
END
```

1.05

Residuals Behavior



NBTF value list: 20, 1.9, 1.8, …. **1.1,1**
→ Solver crash when NBTF = 1 reached

NBTF value list: 20, 1.9, 1.8, …. 1.1, **1.05, 1.025**, 1

Non-Equilibrium Steam Model

Efficiency Calculation

Efficiency Calculations for NES Calculations

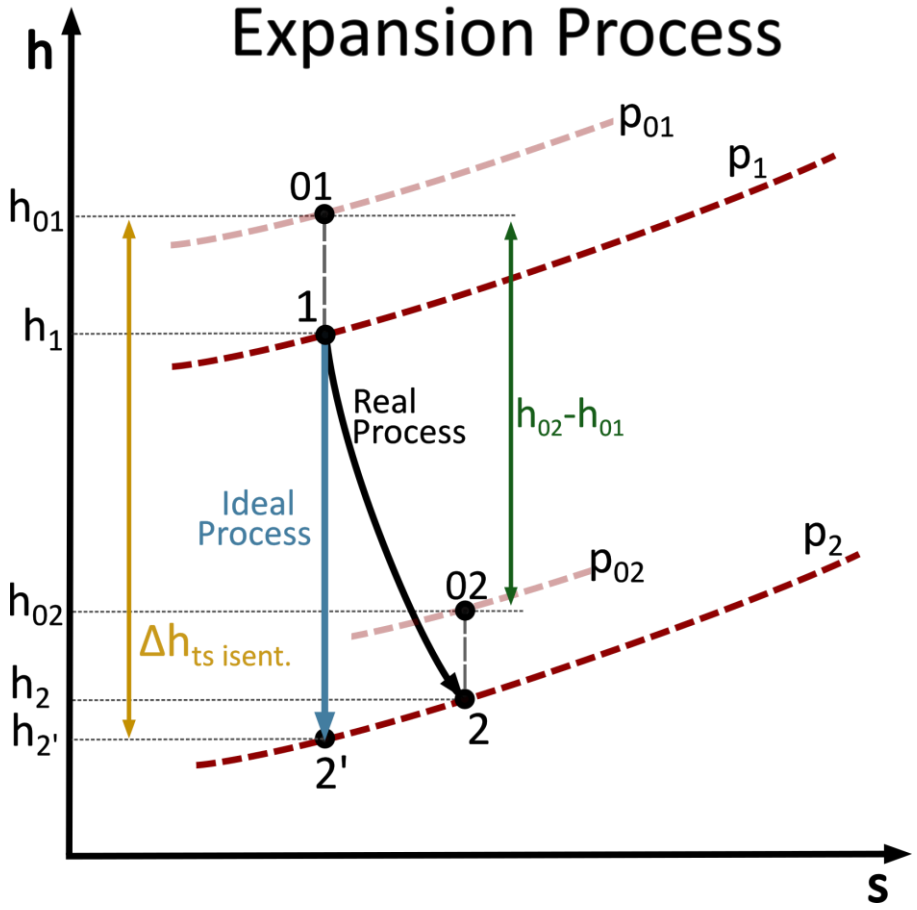
The efficiency monitors provided by CFX are calculated based on the continuous phase, while neglecting the dispersed (liquid) phase. To obtain a meaningful efficiency value for steam turbines modelled with the NES model, it is necessary to consider the enthalpy values of all phases, which is the mass-weighted average of the liquid and the vapor enthalpies

Total to Static Isentropic Efficiency

$$\eta_{isen,ts} = \frac{Real\ Turbine\ Work}{Isentropic\ Turbine\ Work} = \frac{Total\ Torque\ \cdot 2\pi\ \cdot \omega}{\Delta h_{ts_{isentropic}} \cdot \dot{m}} = \frac{h_{01} - h_{02}}{h_{01} - h_{2'}}$$

Multiple methods exist for calculating the isentropic efficiency, one of which involves using the torque as an indicator of the actual turbine work. The required calculations can be done in Python.

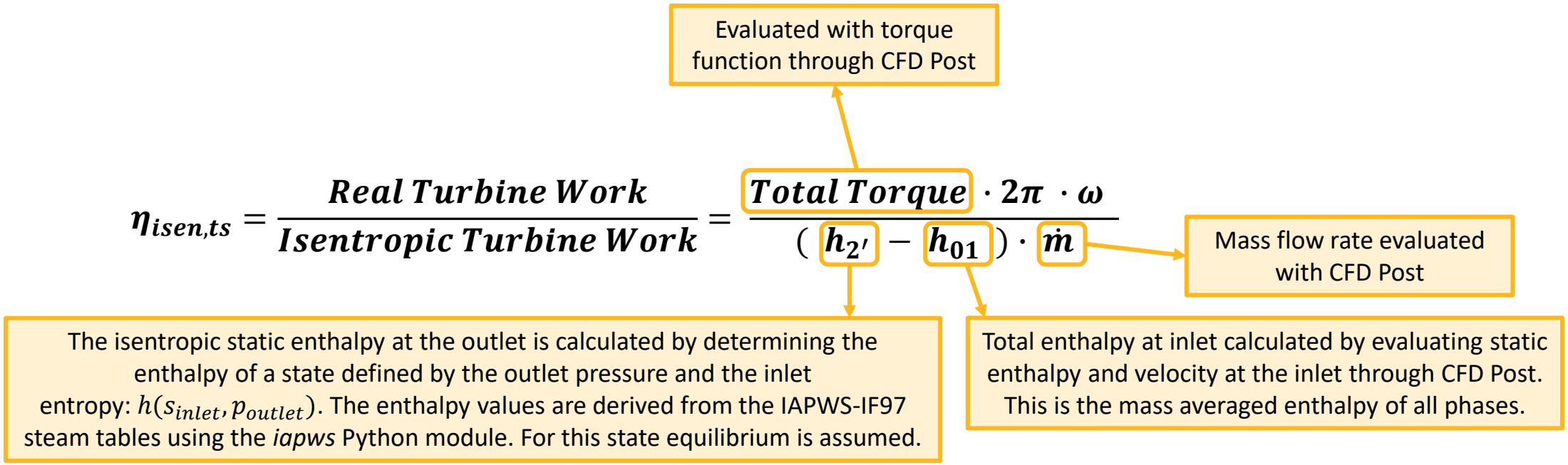
A Python example script is provided, which shows how the efficiency can be calculated considering all phases (one for NES and one for EQS results).



NES Efficiency → 09_Efficiency_Calculations/Calc_ts_isen_efficency_NES.py
EQS Efficiency → 09_Efficiency_Calculations/Calc_ts_isen_efficency_EQS.py

Python Script Functionality

The example Python script uses CFD Post and the Python module *iapws* to compute the isentropic efficiency:



To use the provided Python Scripts the following steps must be completed:

- Pip install the iapws Python module
- Place the respective Eff.cse file in the same directory as the Python script (Eff_NES.cse or Eff_ES.cse)
- Within Python script adapt path to respective .res file
- Within Python script adapt path to CFD Post
- Adapt value for ω in Python script (rotational speed)
- Provide number of blades for required rows within Python script

NES Efficiency → 09_Efficiency_Calculations/Calc_ts_isen_efficency_NES.py
EQS Efficiency → 09_Efficiency_Calculations/Calc_ts_isen_efficency_EQS.py

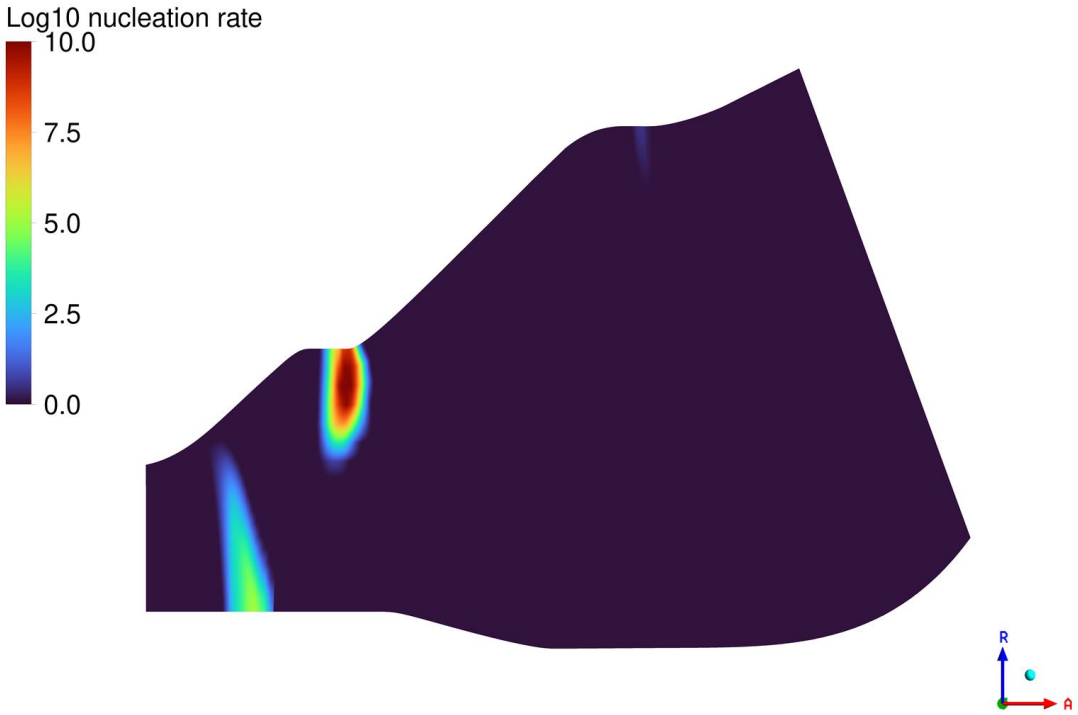
Non-Equilibrium Steam Model

Multiple Nucleation Phases

STAC NES Multiple Nucleation Phases Set Up
06_NES_multiple_nucleation_phases.ccl

Non-Equilibrium Steam – Multiple Nucleation Phases

As the standard method for setting up the NES one continuous phase is combined with one dispersed liquid phase. Alternatively, one could introduce multiple liquid phases.



Nucleation in STAC turbine

→ The nucleation rate indicates at which locations in the turbine droplets are formed

→ This meridional view shows that nucleation occurs in the first two rows and slightly in the last row (domains S1, R1 and R0)

Single Liquid Phase

- One H2O liquid phase for droplets in the entire domain
- Nucleation to form H2O droplets is allowed in all domains
- When all droplets are of the same liquid phase the computed droplet size is an average of all the present droplet sizes

→ The droplet diameter is calculated from the volume fraction of the liquid phase and the droplet number (these two equations are solved).

→ If only one liquid phase exists, droplets forming further downstream add to the volume fraction and droplet number of droplets formed at an earlier stage.

→ The droplet diameter is calculated based on the overall volume fraction and droplet number.

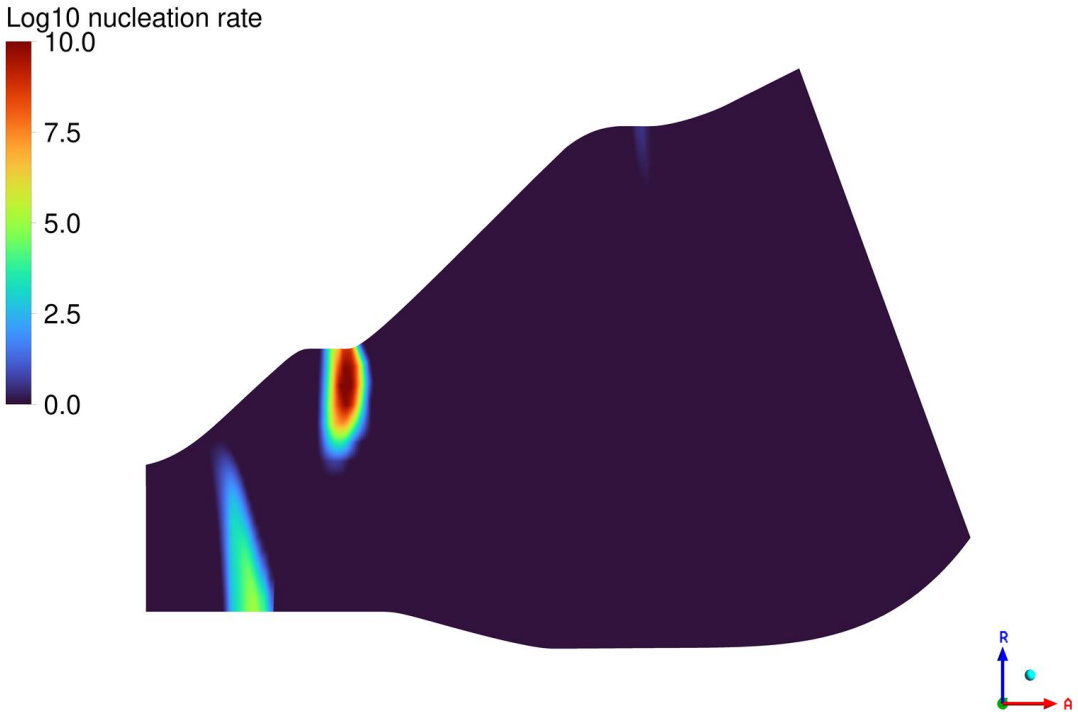
➡ **No prediction of actual droplet size**

Droplet Radius

$$R_d = \left(\frac{3\, r_d}{4\pi\, N_d}\right)^{\frac{1}{3}}$$

r_d - Volume Fraction of Dispersed phase
N_d - Droplet Number

Non-Equilibrium Steam – Multiple Nucleation Phases



Nucleation in STAC turbine

→ The nucleation rate indicates at which locations in the turbine droplets are formed

→ This meridional view shows that nucleation occurs in the first two rows and slightly in the last row (domains S1, R1 and R0)

Multiple Liquid Phases

- Alternatively, one can introduce one liquid phase for each domain in which nucleation occurs (domains S1,R1,R0 for the STAC case)
- This means that droplets formed at different locations do not add up to the same volume fraction and droplet number, but each liquid phase has a separate volume fraction and droplet number

→ Separate droplet diameters are calculated

→ Early formed droplets are not size averaged with later formed droplets

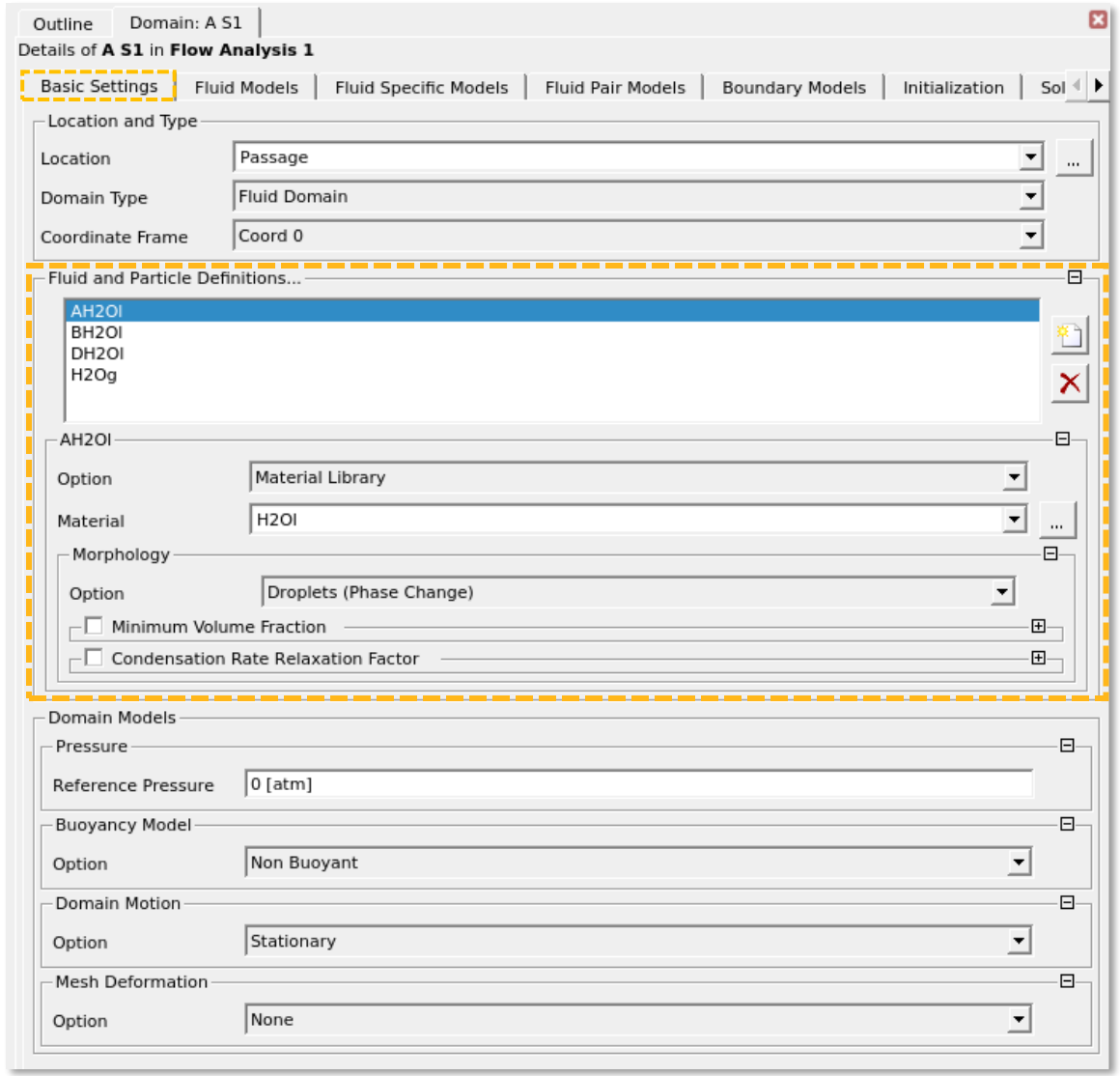
➡ **Better prediction of actual droplet size**

Droplet Radius

$$R_d = \left(\frac{3\, r_d}{4\pi\, N_d}\right)^{\frac{1}{3}}$$

r_d - Volume Fraction of Dispersed phase
N_d - Droplet Number

Non-Equilibrium Steam – How to Include Multiple Nucleation Phases



1) In the Domain → Basic Settings:

- Add a fluid for each domain in which nucleation occurs in addition to one single gas phase (H2Og)
- Assign the liquid water phase material to all these fluids

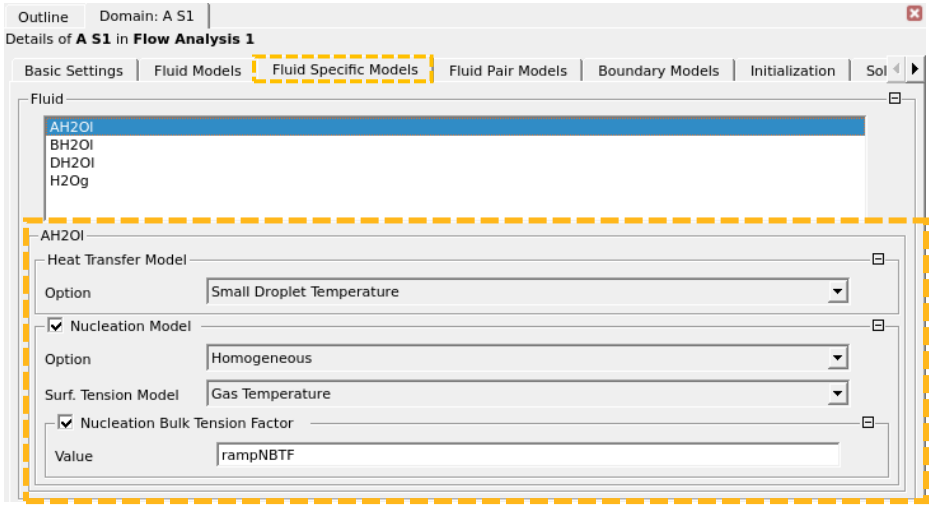
  Material → H2Ol

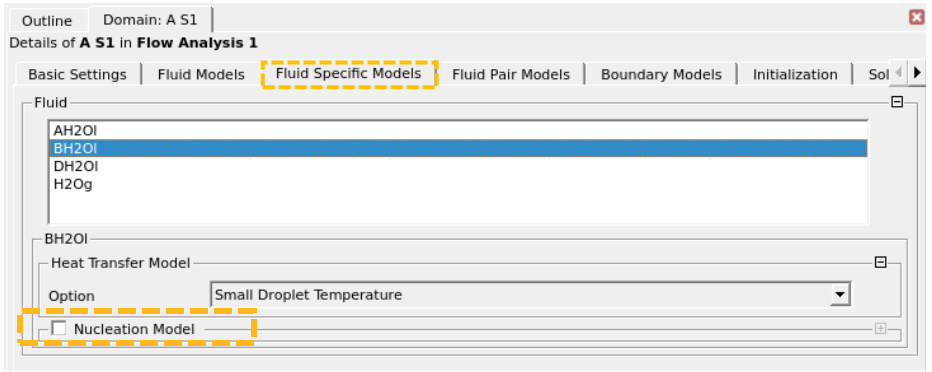
  (Only create one liquid water phase material, not one for each phase. Otherwise, this will cause problems while solving.)
- Assign **Morphology → Droplets** for all liquid phases (continuous fluid for gas phase)
- Important: All fluids (AH2Ol, BH2Ol, CH2Ol, H2Og) must be present in all domains, otherwise the solver will crash

2) In First Domain → Inlet → Fluid Values

- Initial values (Volume fraction = 0, Droplet Number = 0) must be set for all liquid phases

Non-Equilibrium Steam – How to Include Multiple Nucleation Phases





3) In the Domain → Fluid Specific Models:

- In the respective domain of each phase activate Nucleation Model and if needed assign a dynamic variable to the Nucleation Bulk Tension Factor.

  STAC Case:
  Domain A S1 → Activate nucleation for AH2Ol
  Domain B R1 → Activate nucleation for BH2Ol
  Domain D R0 → Activate nucleation for DH2Ol

- Do not activate nucleation for any other liquid phase

4) Other settings are the same as in the regular Non-Equilibrium Steam Model

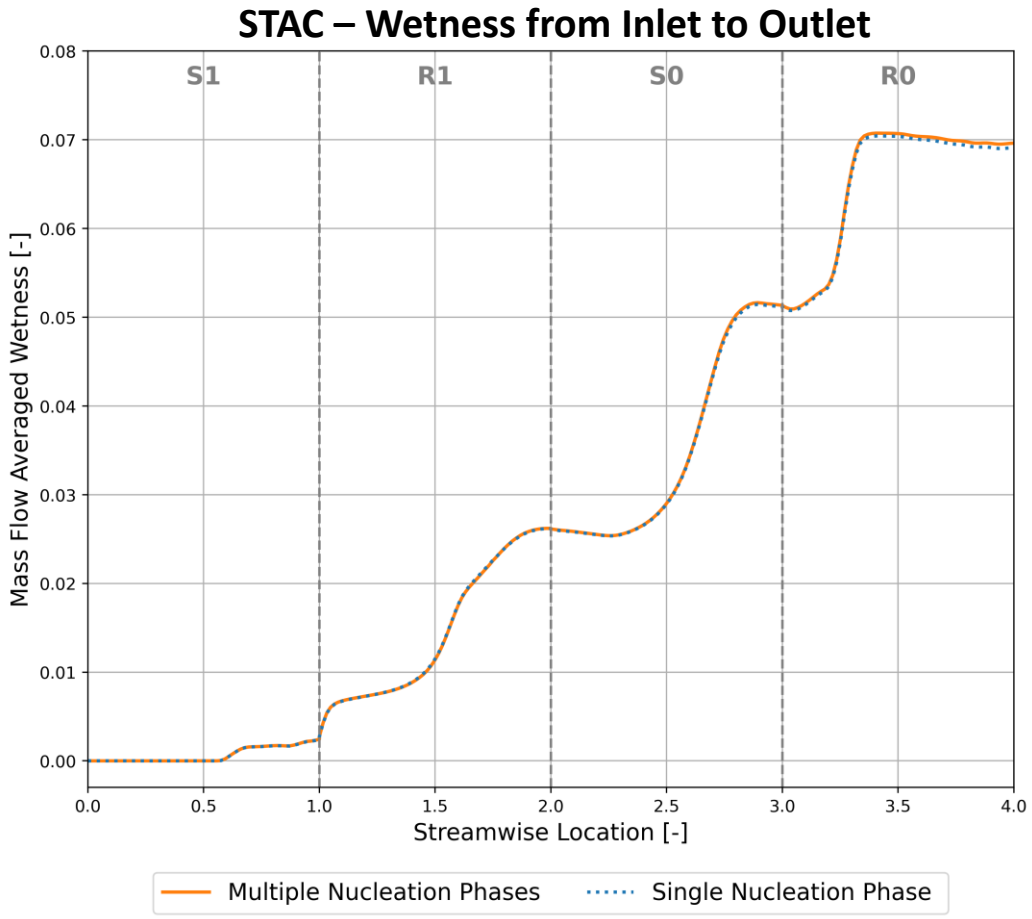
5) Initialization:

Possibilities:

- Initialize with an Ideal Gas solution and use a dynamic variable to reduce the Nucleation Bulk Tension Factor step by step
- Initialize with an NES case using a single nucleation phase and set the Nucleation Bulk Tension Factor to one

- It might be necessary to test varying timestep sizes

STAC - Impact of Multiple Nucleation Phases



Total to Static Isentropic Efficiency	
NES Single Nucleation Phase:	82.96 %
NES Multiple Nucleation Phases:	82.95 %

Mass Fraction Liquid Phase – Wetness

If the main interest is wetness and how much condensation occurs, then introducing multiple nucleation phases is not necessary. The STAC case results reflect that the wetness along the domain is just slightly impacted by the number of nucleation phases.

(The slight difference in mass fraction towards the end of the domain are likely due to the droplet diameter R_d impacting the interfacial mass transfer. For details see the Ansys CFX Solver Theory Guide: 5.13.6.The Droplet Condensation Model)

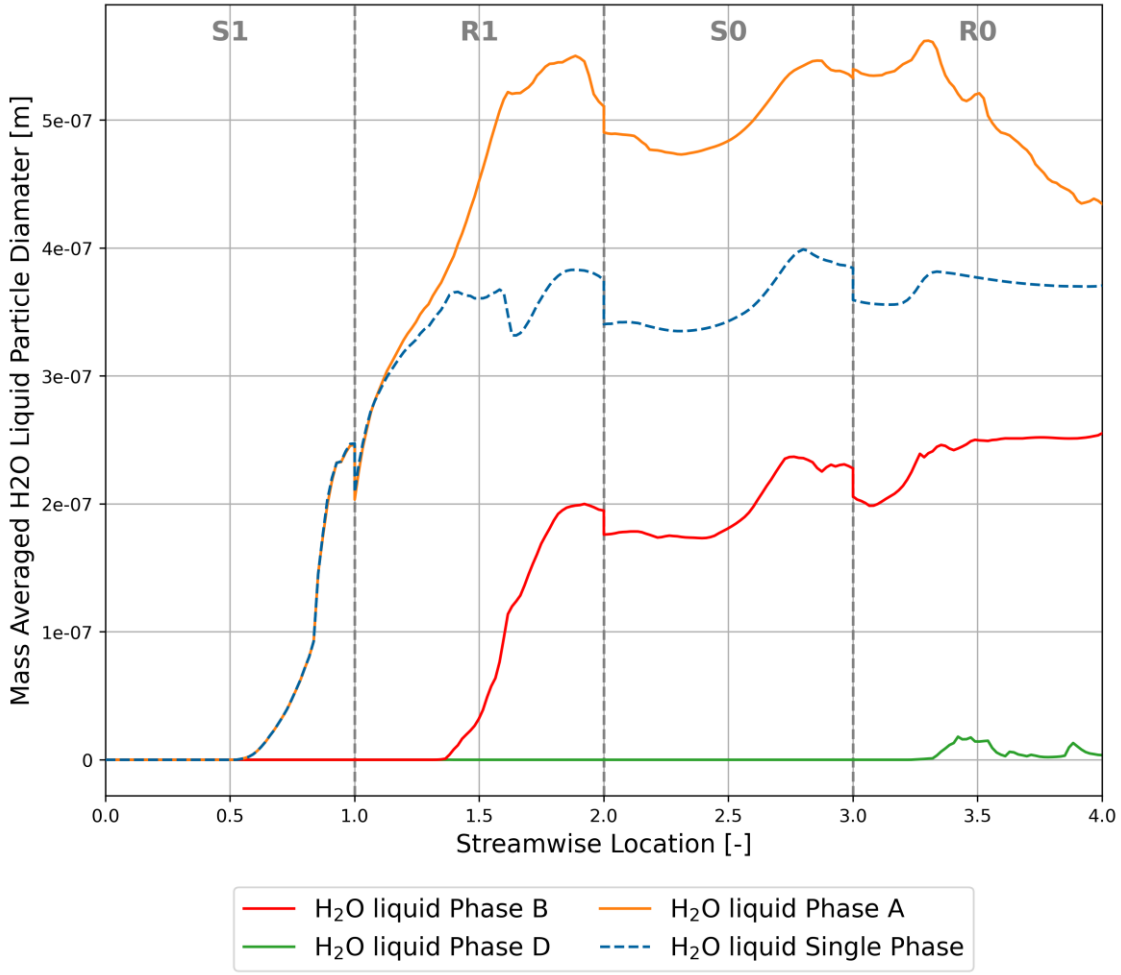
Efficiency

Likewise, the performance values remain similar for both cases and an interest in efficiency does not require the introduction of multiple nucleation phases.

To calculate efficiency for a case with multiple nucleation cases a separate Python script is provided.

STAC - Impact of Multiple Nucleation Phases

STAC – Droplet Diameter from Inlet to Outlet



Droplet Diameter

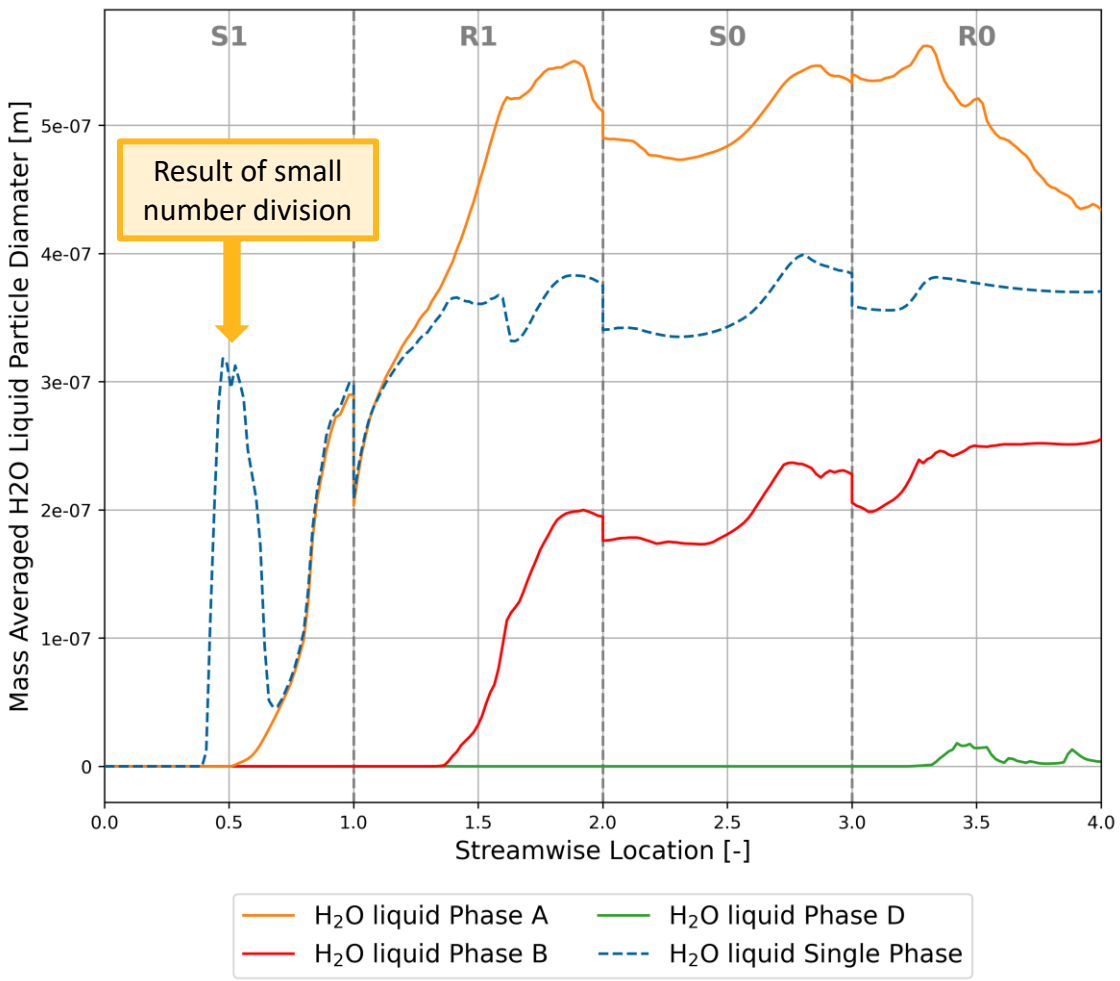
The size of droplets might be of interest as it impacts for example the erosion of turbine blades. For a better insight into which droplet sizes are present it is useful to introduce multiple nucleation phases.

STAC Case

The plot on the left shows the comparison of the droplet diameter from inlet to outlet for the case of only one liquid phase (dashed line) and the various liquid phases of the case with multiple nucleation phases.

It is clearly shown, that for the single nucleation case one averaged droplet diameter is computed. When multiple liquid phases are introduced one can see the distinct droplet sizes present within the domains.

Droplet Diameter – Numerical Issues



Droplet Radius

$$R_d = \left(\frac{3\, r_d}{4\pi\, N_d} \right)^{\frac{1}{3}}$$

r_d - Volume Fraction of Dispersed phase

N_d - Droplet Number

Since the droplet radius is not directly determined by a transport function but is instead dependent on the volume fraction and droplet number, numerical irregularities may arise when both values are small. Dividing by a small number can cause the droplet radius to show unexpectedly high values, even when the liquid volume fraction is near zero.

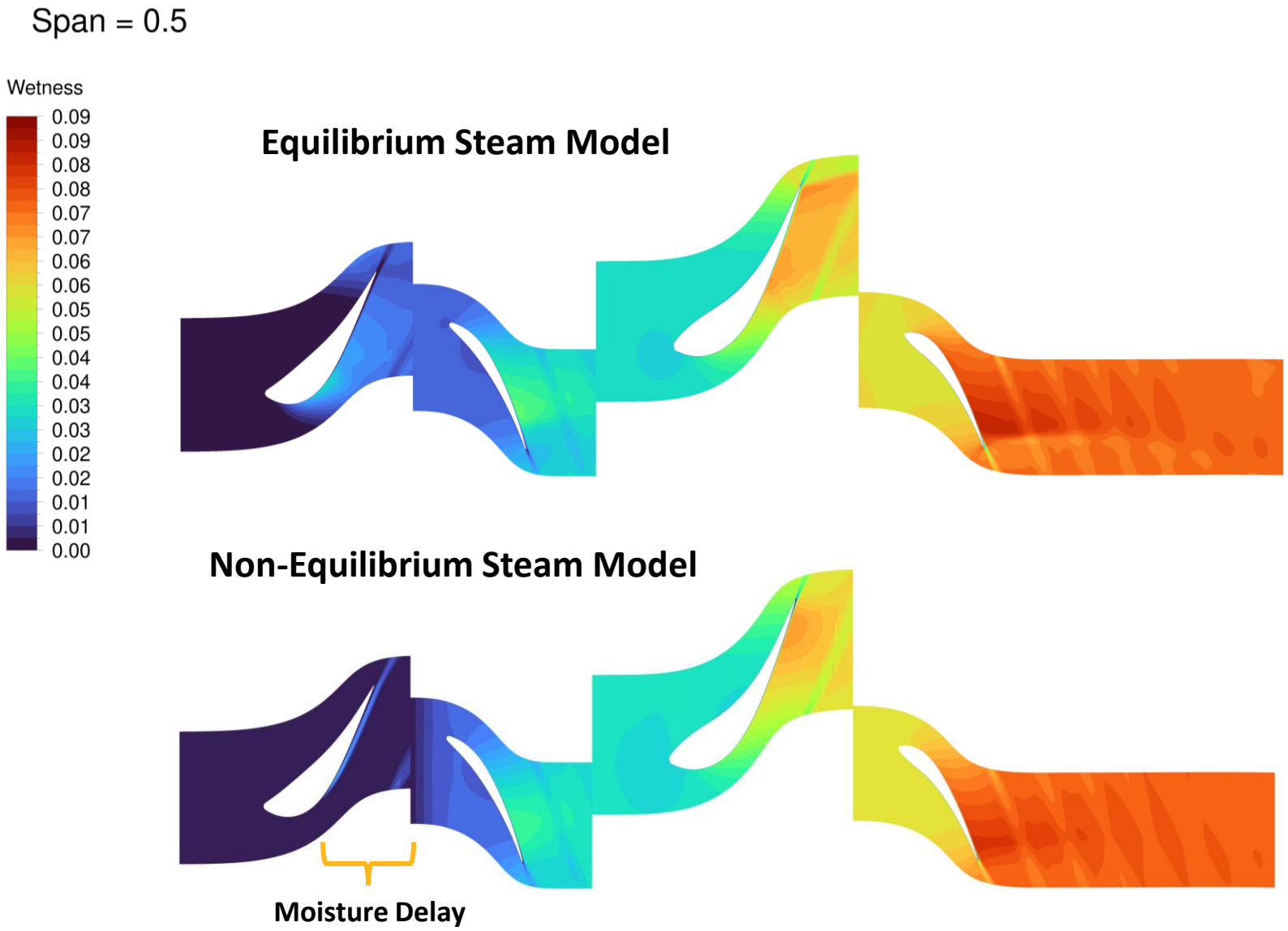
To remove unphysically high droplet diameters, introduce an expression/variable setting the droplet diameter to zero when the volume fraction or droplet number are below a threshold, e.g.

```
if (H2Ol.Volume Fraction < 1e-10, 0 [m],
    H2Ol.Particle Diameter)
```

Model Comparisons

Equilibrium Steam and Non-Equilibrium Steam

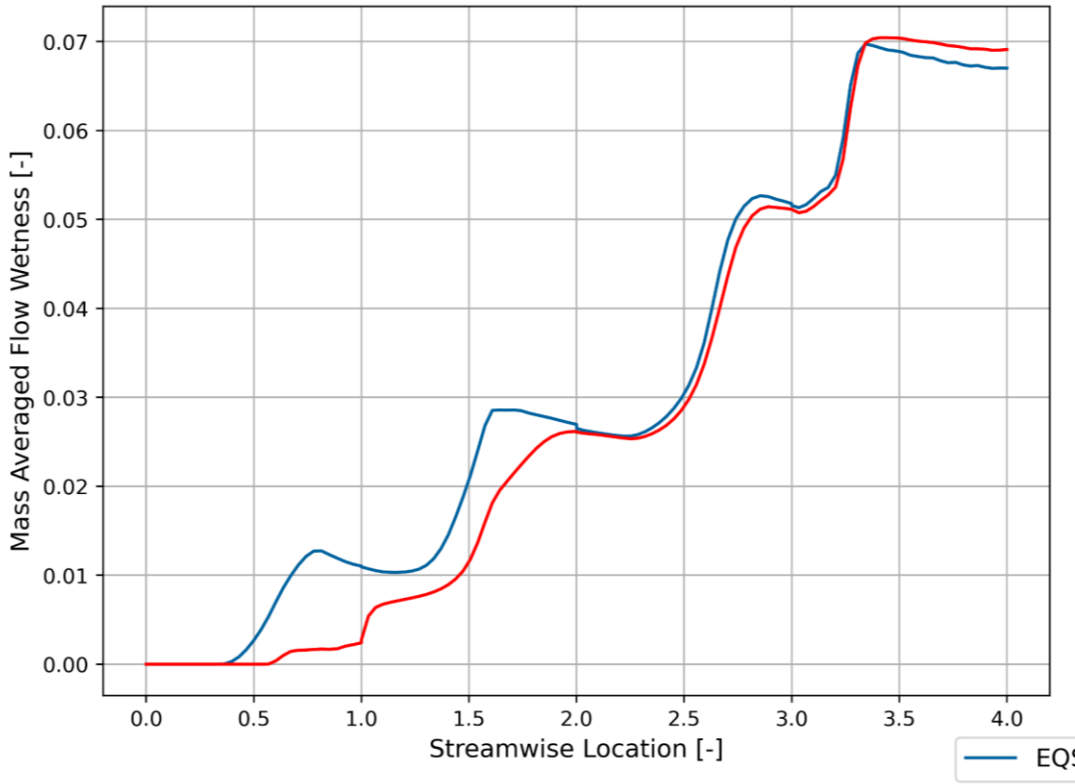
NES and EQS Results Comparison



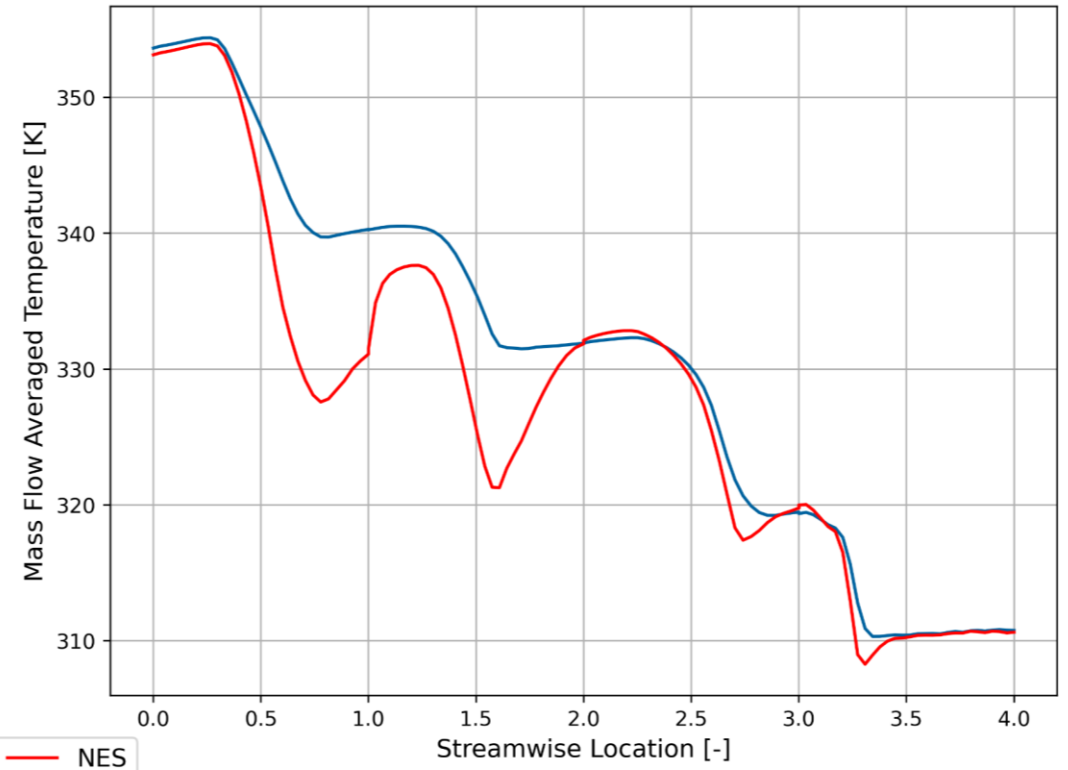
The results from the STAC case emphasize the differences between the Equilibrium Steam (ES) and Non-Equilibrium Steam (NES) models. The EQS model assumes instantaneous thermodynamic equilibrium, meaning a phase change occurs immediately when the saturation temperature is reached. In contrast, the NES model demonstrates a delayed condensation process. This is reflected in the plot, where wetness increases further from the inlet for the NES results compared to the EQS results.

NES and EQS Results Comparison

Inlet to Outlet – Liquid Mass Fraction



Inlet to Outlet – Temperature



The mass averaged wetness from inlet to outlet shows the delayed condensation of the NES model compared to the EQS model.

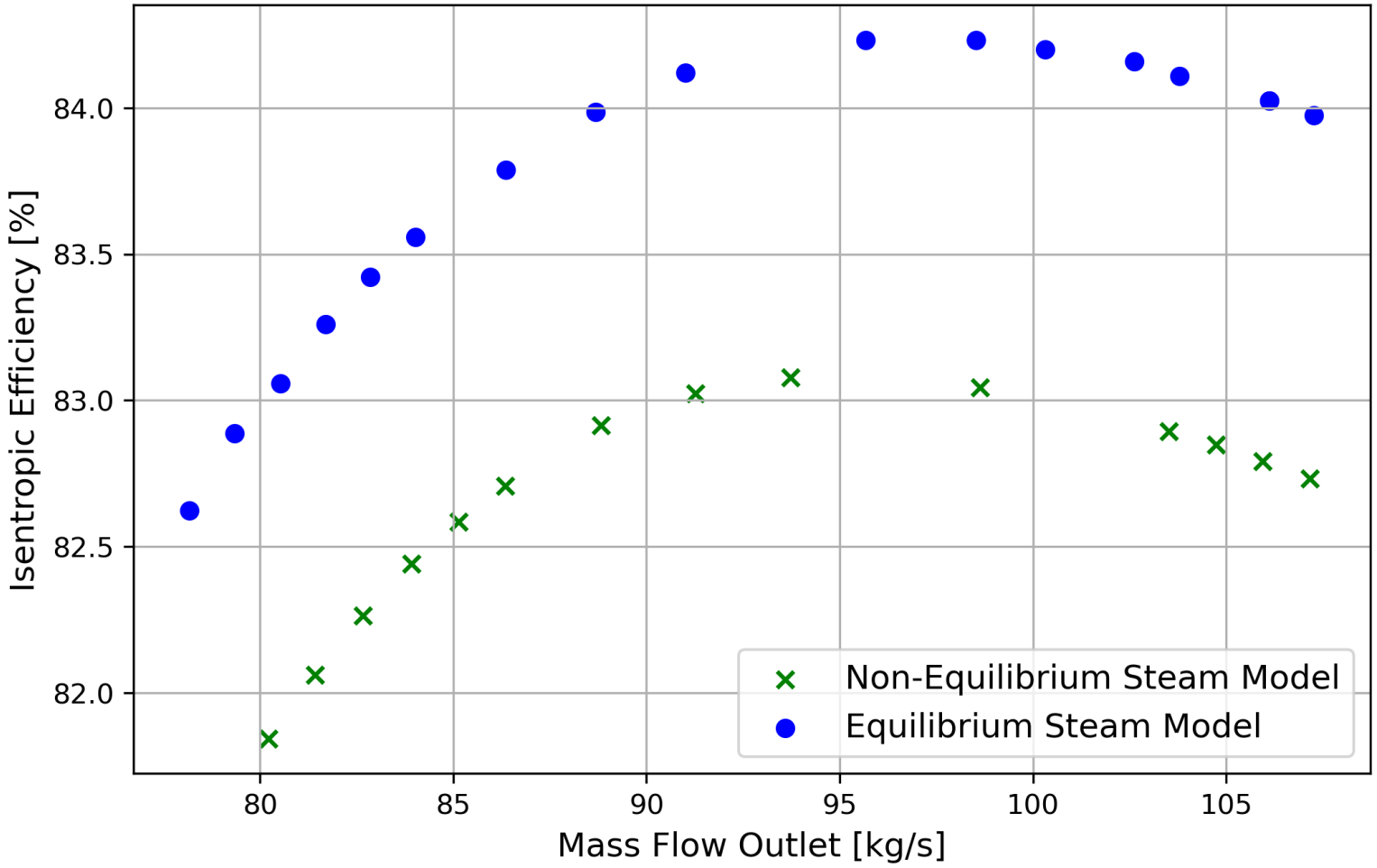
The dips in temperature profile from inlet to outlet demonstrate that the NES model allows the gas phase to temporarily drop below the saturation temperature, whereas in the EQS model, reaching the saturation temperature results in an instantaneous phase change.

Efficiency Characteristics

The differences between the Non-Equilibrium Steam Model (NES) and the Equilibrium Steam Model (EQS) are also evident in the resulting efficiencies.

Unlike the Equilibrium Steam Model, the Non-Equilibrium Steam Model does not assume the flow to reach equilibrium conditions instantaneously and therefore implicitly includes losses due to thermodynamic irreversibility.

As a result, the Equilibrium Steam Model tends to yield higher efficiencies than the Non-Equilibrium Steam Model, which is more likely to produce realistic values.



Provided Files

How to use STAC Case Files

STAC Case files

With these guidelines we provide a set of files which show the setups for the different models and approaches presented within these guidelines. These allow users to run and test the STAC case themselves. For running the STAC Case a def-file with the desired mesh and the appropriate ccl files need to be passed to the solver.

File Overview:

01_STAC_Inlet_Boundary_Conditions.csv	Csv file which holds information on the inlet velocity profile.
02_MeshA_Ideal_Gas.def	Def-file with STAC setup using ideal gas based on Mesh A (see slide 8).
03_MeshC_Ideal_Gas.def	Def-file with STAC setup using ideal gas based on Mesh C (see slide 8).
04_Equilibrium_Steam.ccl	CCL containing STAC set up using the Equilibrium Steam Model. Chose a def file based on ideal gas and import this ccl setup.
05_Non_Equilibrium_Steam.ccl	CCL containing STAC set up using the Non-Equilibrium Steam Model. Chose a def file based on ideal gas and import this ccl setup. Initialize with an Ideal Gas Solution.
06_NES_multiple_nucleation_phases.ccl	CCL containing STAC set up using the Non-Equilibrium Steam Model with 3 liquid phases (see slide 64). Chose a def file based on ideal gas and import this ccl setup. Initialize with an Ideal Gas Solution.
07_Implicit_Mixing_Plane_settings.ccl	CCL containing the mixing plane implicit settings. Apply to any model set up. Apply to improve reflections at mixing planes (see slide 30)
08_TurboGridFiles	TurboGrid state files and required curve files for all rows for meshes A and C. Allows editing of mesh in Ansys TurboGrid.
09_Efficiency_Calculations	Contains Script for calculating Efficiencies as described on slides 62/63. Scripts for efficiency calculations for NES and EQS setups. Each requires a respective .cse file that must be placed within the same directory. For use see instructions Slide 63.

List of Abbreviations

CoV	*Coefficient of Variation*
EQS	*Equilibrium Steam Model*
ITSM	*Institute of Thermal Turbomachinery and Machinery Laboratory*
NBTF	*Nucleation Bulk Tension Factor*
NES	*Non-Equilibrium Steam Model*
RMB	*Right Mouse Button*
STAC	*Steam Turbine Test Case for Aeromechanics and Condensation*