CADMEN 虎門科技 | Ansys精英合作夥伴

Ansys

2022 R1 新技術線上研討會

Engineering What's Ahead.

4/7 (四) 14:00-14:40	Ansys CFD 進階應用_TurboWorkflow 新技術線上研討會	李奕璋
4/7 (四) 15:00-15:40	Ansys Rocky 新技術線上研討會 (Ansys Fluent、Motion & OptiSLang 耦合應用技術)	林健文
4/7 (四) 16:00-16:40	Ansys GRANTA 碳足跡應用與產業成功案例分享-新技術線上研討會	李易軒

Ansys 2022R1

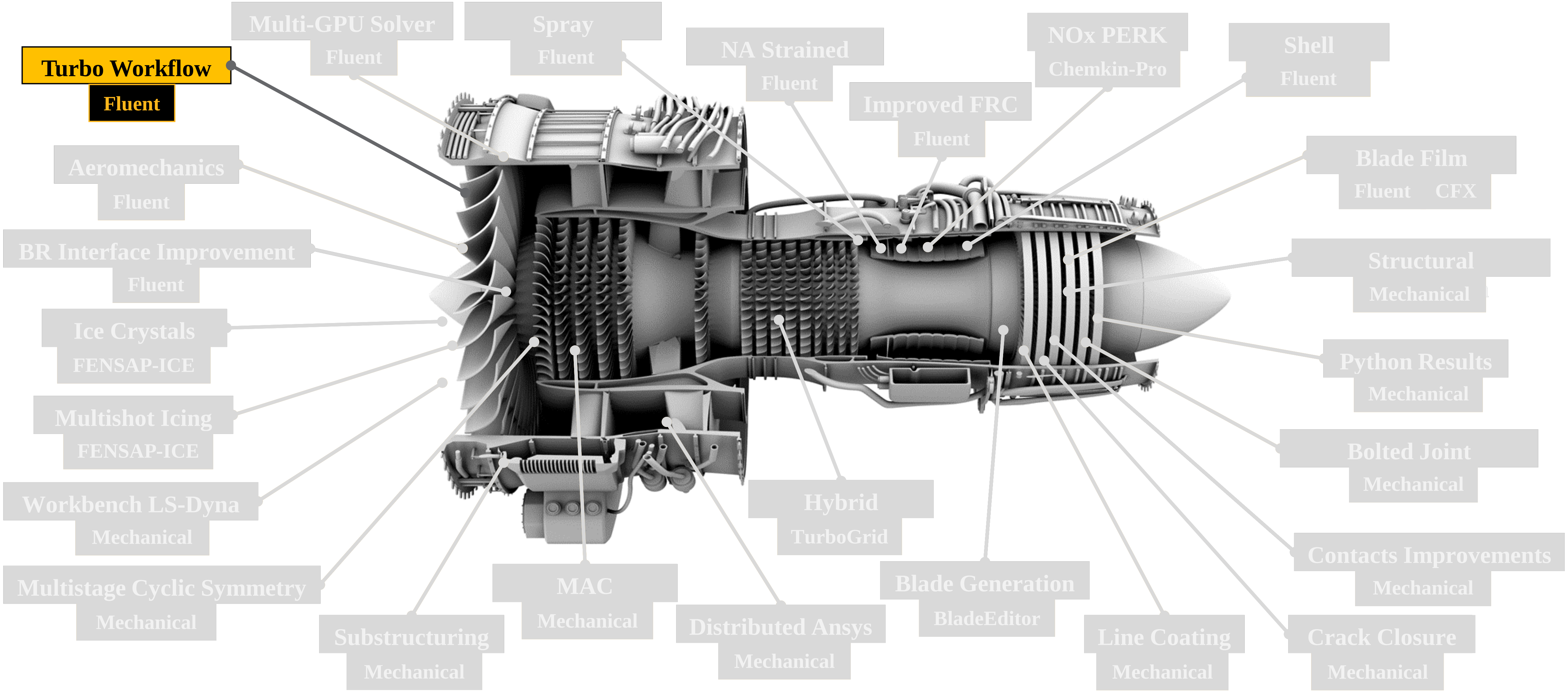
New Turbo Tools

Fluent TurboWorkflow

虎門技術工程師 Haruto
haruto.lee@cadmen.com

2022 R1 Features by Gas Turbine Modules - Dashboard

Speed
Accuracy
Robustness



Ansys

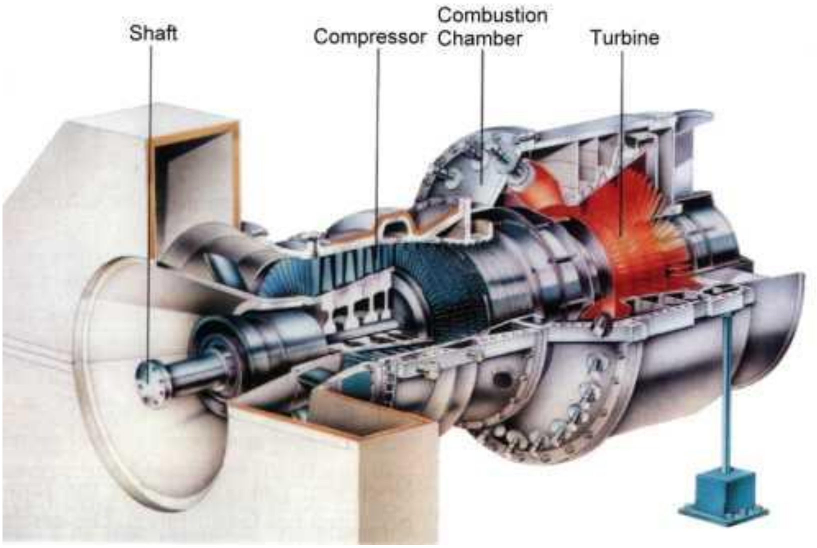
Fluent Turbo Methods and Workflow

- 2022 R1 functionality
  - **Pre-processing tool** with Turbo workflow
  - **Post-processing tool** with Periodic instancing + Turbo workflow + Simulation Report
  - **Base Turbo methods** to cover the tree pillars of Gas Turbine analysis
    - Aerodynamic analysis

      Steady & Transient , main and secondary path, axial, radial or mix components
    - Aeromechanics analysis

      Aerodynamic damping & Forcing load for Forced Response analysis
    - Aerothermodynamic analysis

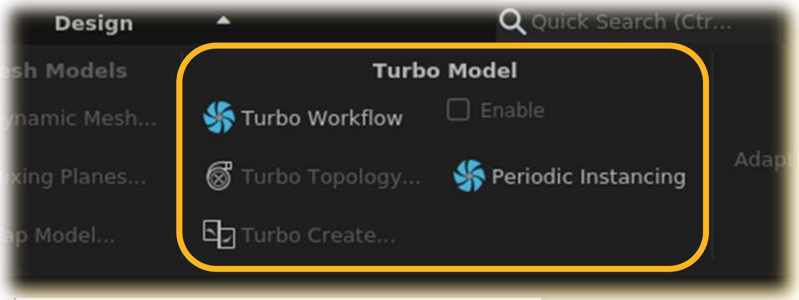
      Detailed and reduced model Blade Film Cooling

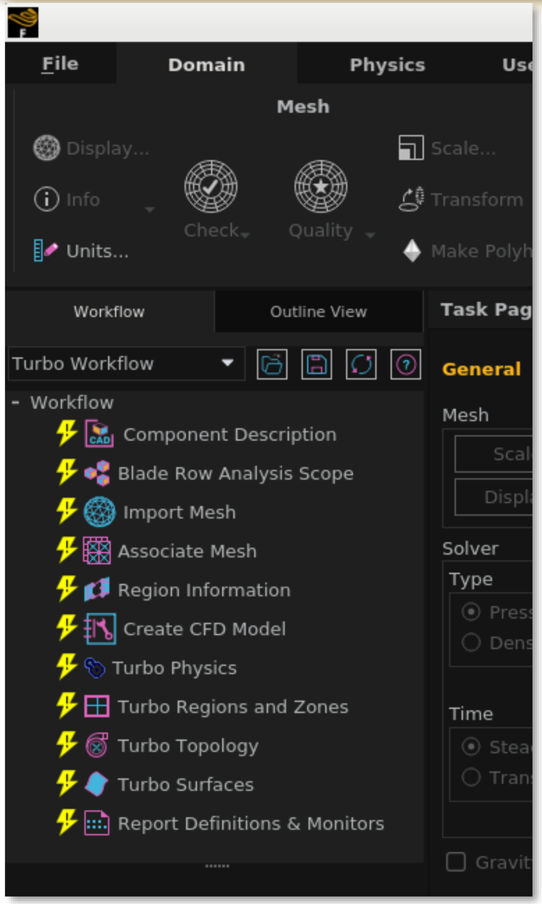


Turbo Workflow

Quickly set up turbomachinery flow problems and instrument primary performance parameters and post processing

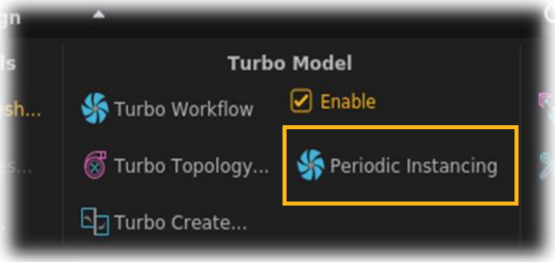
- Guided task-based approach
  - Eliminates repeated user inputs and minimize user error during setup
  - Supports reverting and editing the setup
- Features:
  - Setup for Axial/Radial Compressors & Turbines
  - Can read multiple mesh files (.msh, .def, .gtm)
  - Copy, rotate, and stitch fluid zones
  - Turbo coordinates for turbo post processing
  - Performance monitors

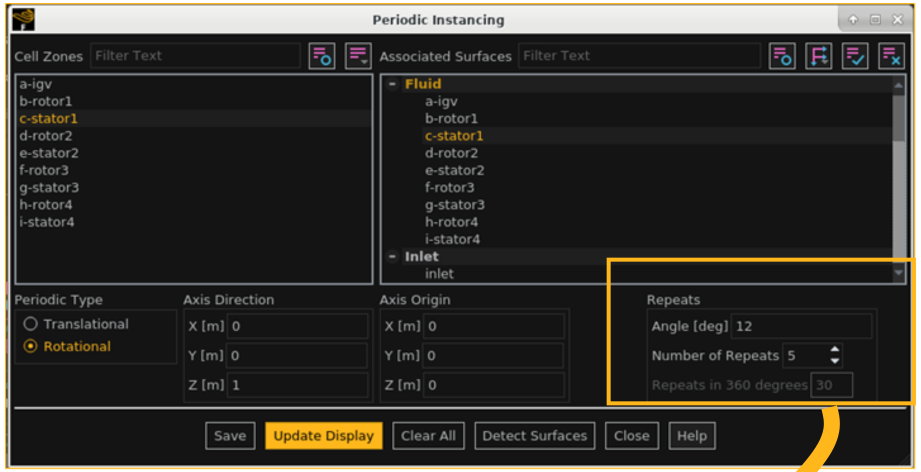


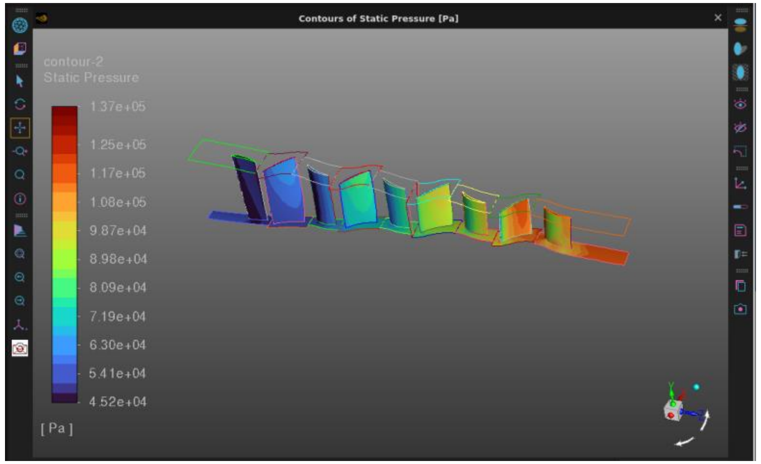


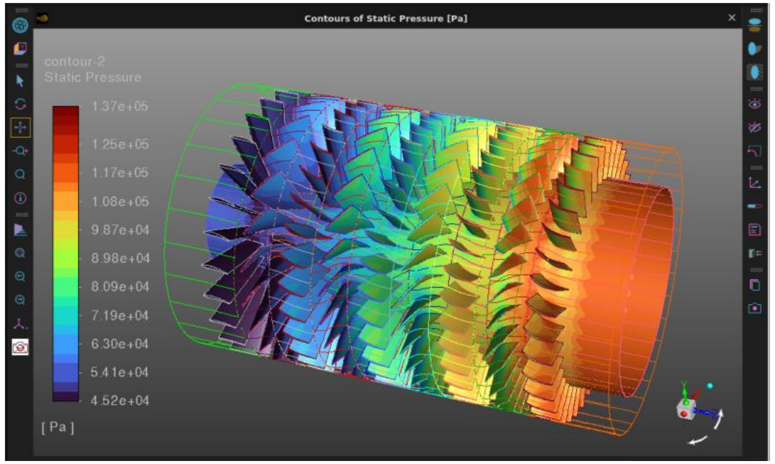
Periodic Instancing

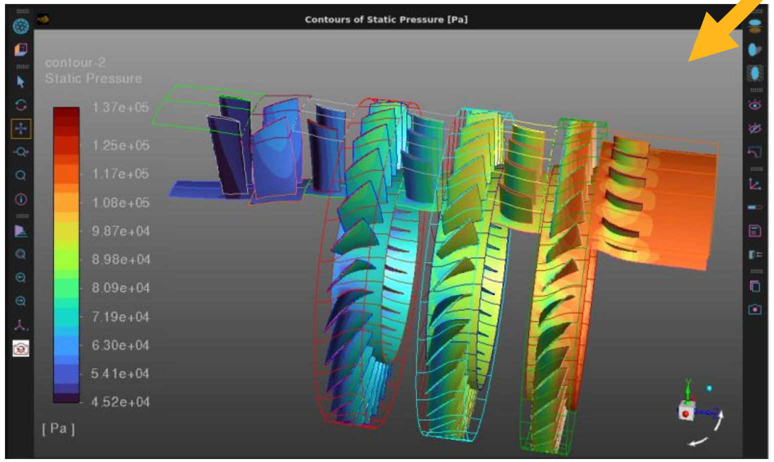
- Enhanced (and renamed) Periodic Repeats feature
- Improve post-processing for most turbomachinery applications











Clear All → **Single Repeat**

- Detect surfaces associated with each zone*
- Sets the view model to ***single repeat***

Detect Surfaces → **Full Wheel**

- Detect surfaces associated with each zone*
- Sets the view model to ***full wheel***

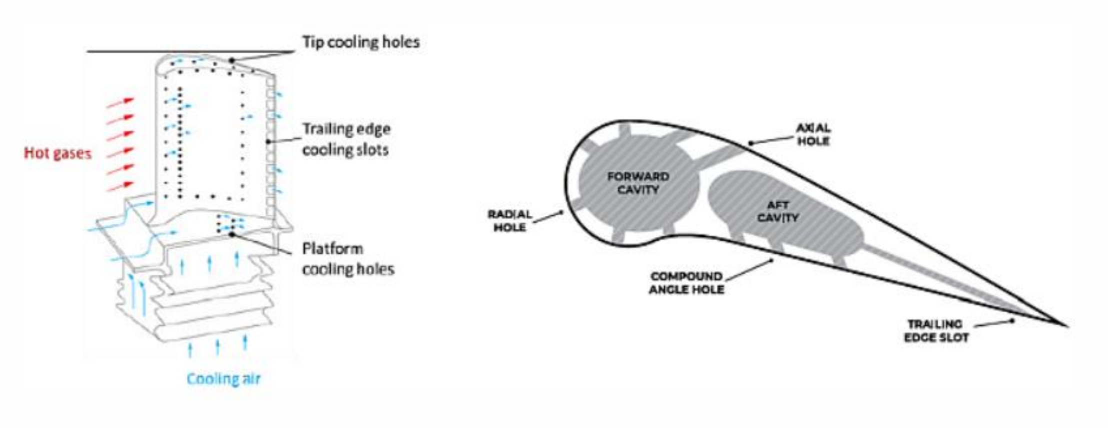
Or use the Repeats settings

- Select cell zone and enter the number of repeats
  - Positive value: forwarded instancing
  - Negative value: backward instancing

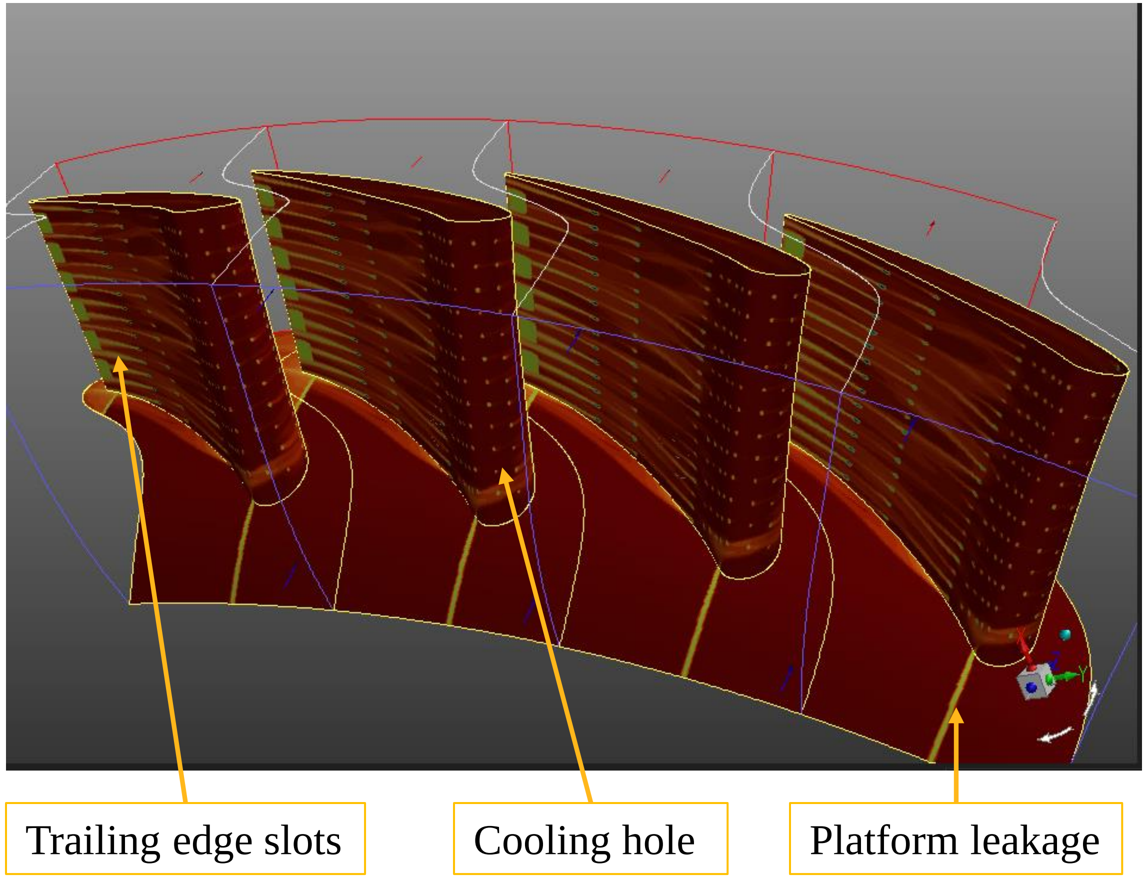
**If turbo surface cut spans more than one zone it will be ignored*

Blade Film Cooling Model for Gas Turbines

- Pre 2022 R1
  - Blade film cooling model available in CFX but not in Fluent
- 2022 R1 functionality
  - Hole coordinates and parameters are imported from profile files to create virtual geometry
    - Rectangular or circular holes oriented normal-to-surface or along specified direction

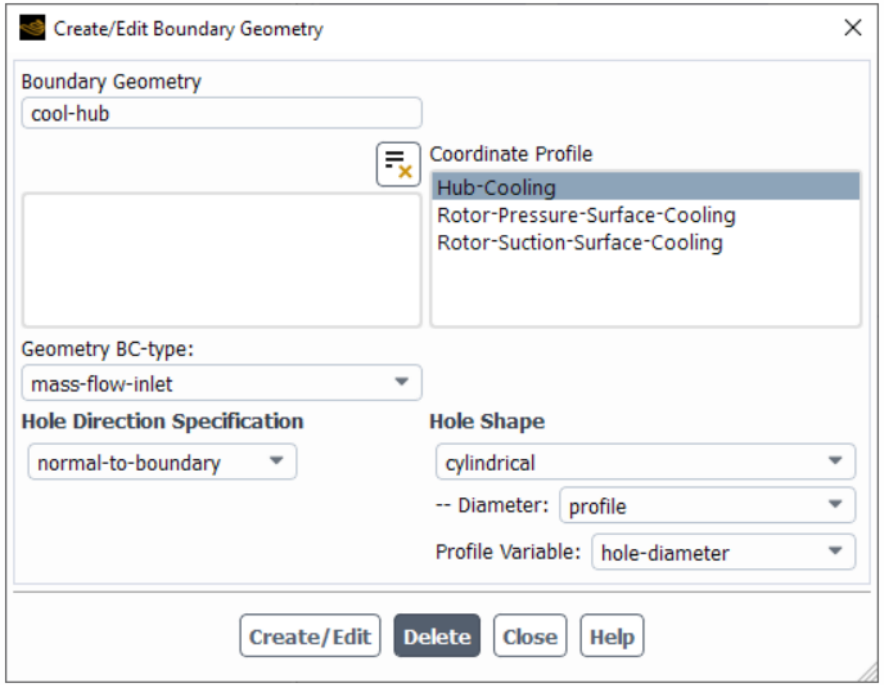


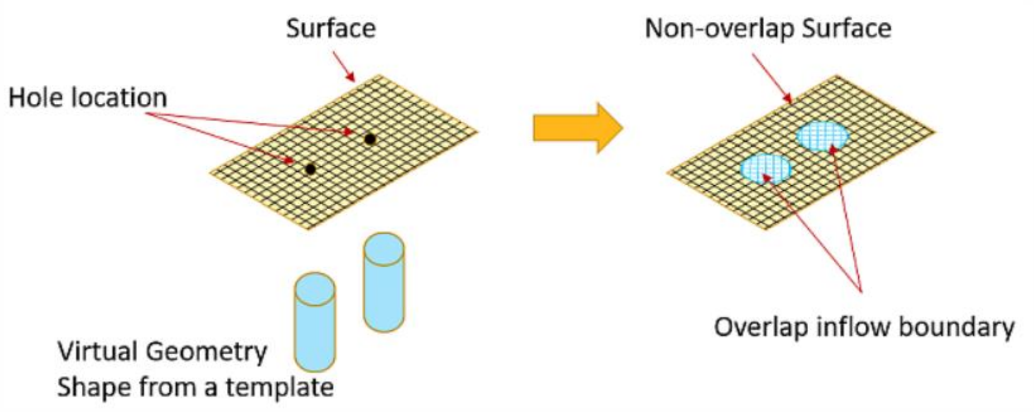
Use Virtual Geometry & Boundary Conditions to Simulate Turbomachinery Blade Cooling

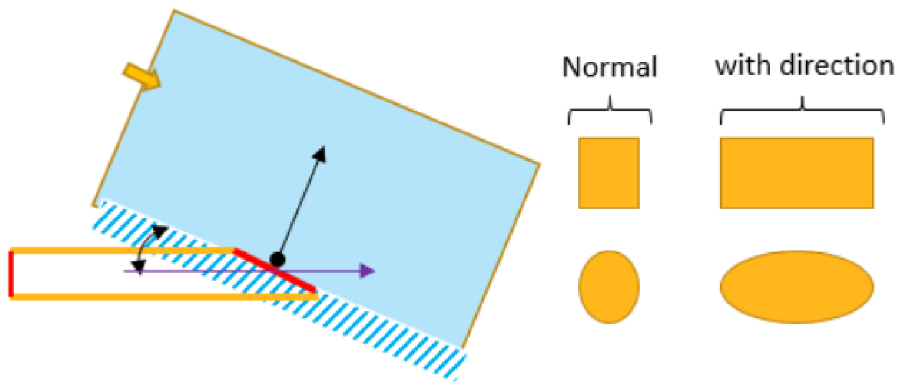


Create virtual boundary conditions

- New **Boundary Interface** intersects virtual geometry with boundary surface to form virtual boundary conditions
    - Similar to creating conventional mesh interfaces
    - Overlapped intersection becomes mass-flow-inlet or mass-flow-outlet
    - Non-overlapped intersection retains underlying surface settings
- Requires Turbo Models to be turned on



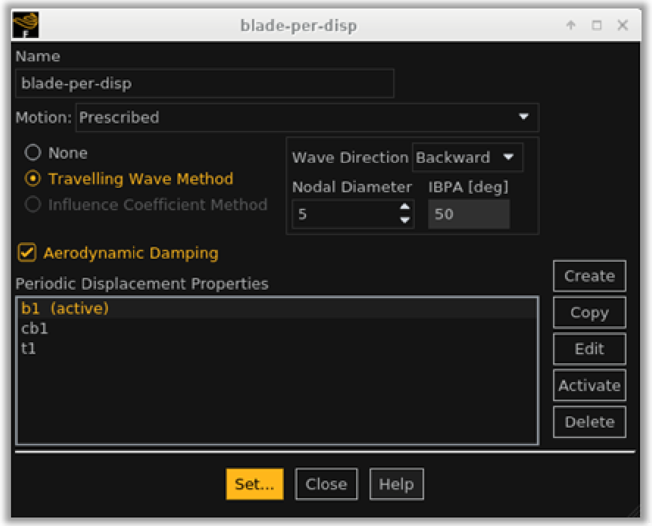


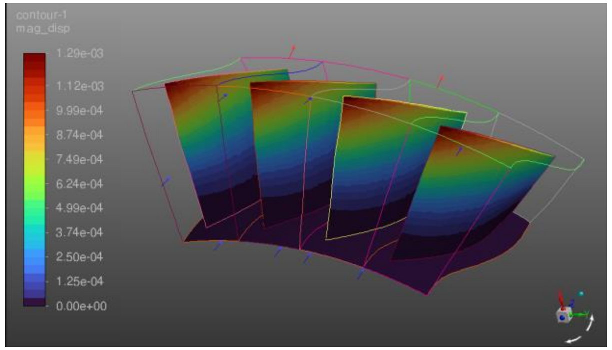


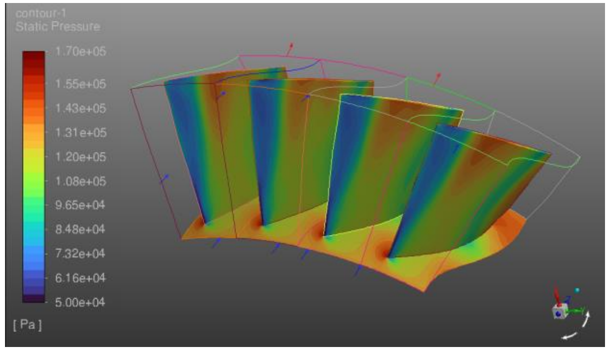
Aeromechanics: Aero Damping/Periodic Displacement

- Pre 2022 R1
  - Aerodynamic Damping computed using Energy Method
  - Single simulation per Nodal Diameter
- 2022 R1 functionality
  - Read and set up multiple mode-shape profiles which can be activated selectively to test aero-damping of different modes
  - Simulate a range of 1-way FSI problems
    - **Traveling Wave Method (TWM)** for blade-row aerodamping
    - **None** option for turbo or non-turbo applications
      - E.g. transient simulations using periodic displacements

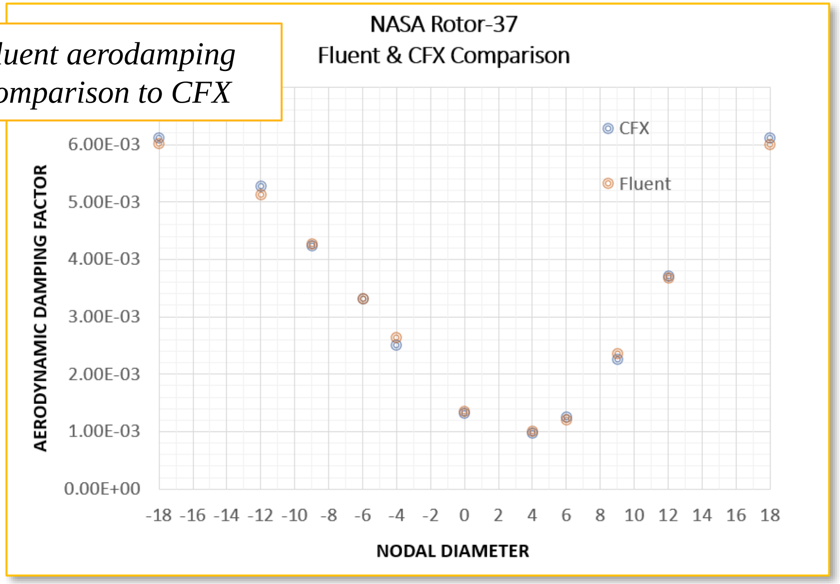
Usability Improved With New Periodic Displacement GUI







Fluent aerodamping comparison to CFX



Help Documentation

- Fluent Turbo Workflow
- Ansys Help Documentation
  - Fluent User's Guide
    - 12.1. Using the Turbomachinery Guided Workflow (ansys.com)
  - Coming soon!
    - Tutorial
    - Video

Chapter 12: Modeling Turbomachinery Flows

This chapter provides details about the turbomachinery capabilities in Ansys Fluent.

The information in this chapter is divided into the following sections:

- Using the Turbomachinery Guided Workflow
- Frozen Gust / Inlet Disturbance Flow Modeling
- Blade Row Interaction Modeling
- Aerodynamic Damping (Blade Flutter Analysis)
- Non-equilibrium Wet Steam Model for Steam Turbines
- Blade Film Cooling for Gas Turbines
- Turbomachinery Postprocessing

PART III: SOLUTION MODE | CHAPTER 12: MODELING TURBOMACHINERY FLOWS |

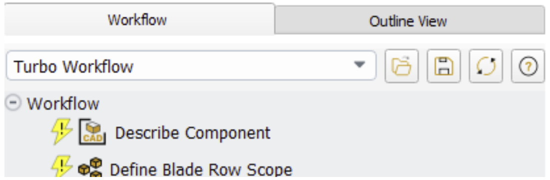
12.1. Using the Turbomachinery Guided Workflow

The **Turbo Workflow** provides you with the basic steps to defining and refining the setup and solution of your turbomach

You can access the turbomachinery guided workflow by selecting the **Enable Workflow** option in the **Turbo Workflow** dr

Domain → Turbomachinery → Turbo Workflow → Enable Workflow

This exposes a Workflow tab, alongside the Outline View, where you can use the **Turbo Workflow** to begin your work. Th through each task in order to more easily analyze your turbomachinery problem and generate an accurate simulation.



Other Fluent improvements will affect turbo users

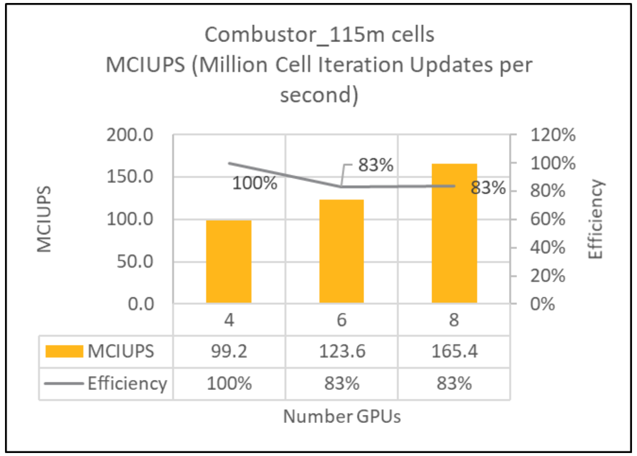
- Performance improvements
- UI enhancements
  - View synchronization
  - Materials for mesh rendering
  - Expression enhancements
- Combustion enhancements
- Acoustics
- Turbulence
- Adjoint
- More…

Multi-GPU Solver (Beta)

- Pre 2022 R1
  - GPU used in offload mode
- 2022 R1 functionality
  - New native multi-GPU Solver in Fluent platform accelerates steady-state CFD simulations
  - Single/multi-GPU (shared / distributed memory)
    - Subsonic compressible flows
    - Ideal gas
    - Material with constant properties
    - Turbulence: standard k-epsilon, GEKO k-omega
    - CHT
    - Porous media
  - Revolutionary speed-up
    - Ball park number from external aerodynamics tests

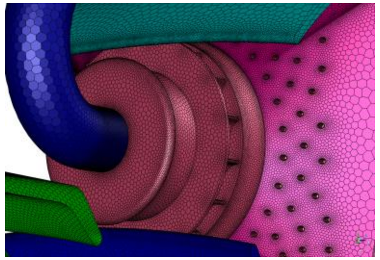
1 GPU ≈ 640 cores (on 5 nodes)
8 GPUs ≈ 3840 cores (on 32 nodes)

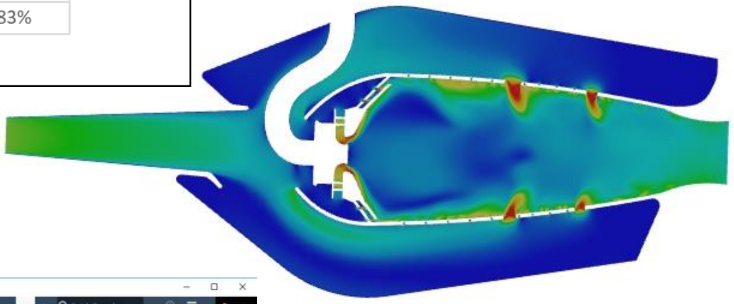
- *Nvidia A100 card vs AMD Milan cores (very similar to Xeon Gold 6142)*

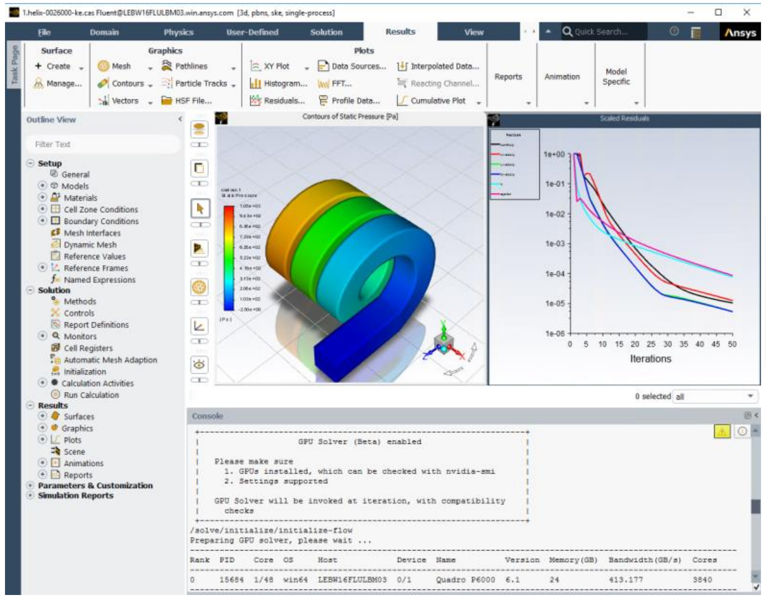


	4	6	8
MCIUPS	99.2	123.6	165.4
Efficiency	100%	83%	83%

~83% parallel efficiency with 8 GPUs





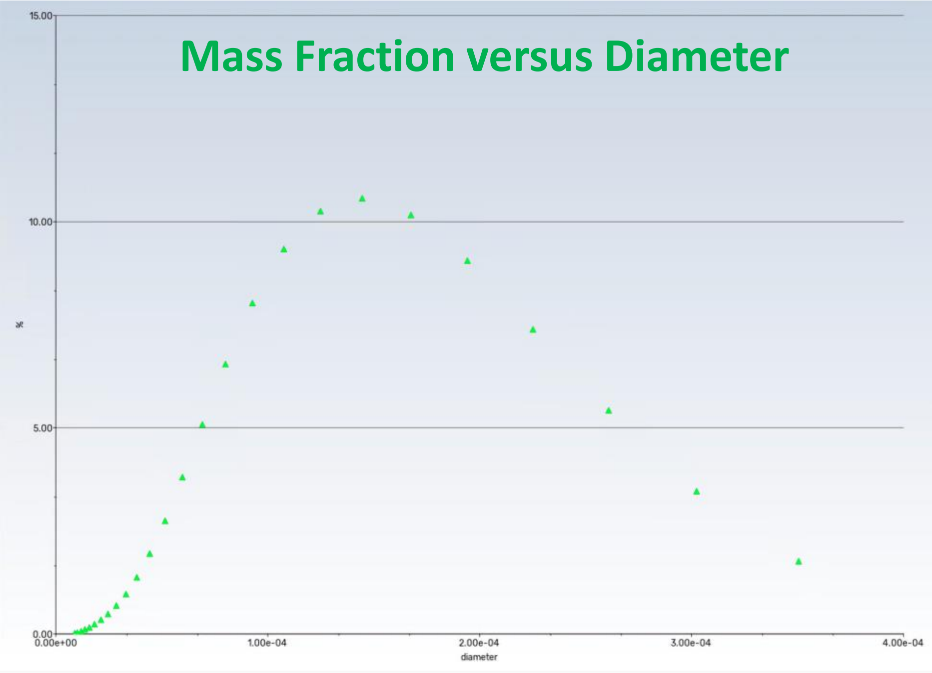


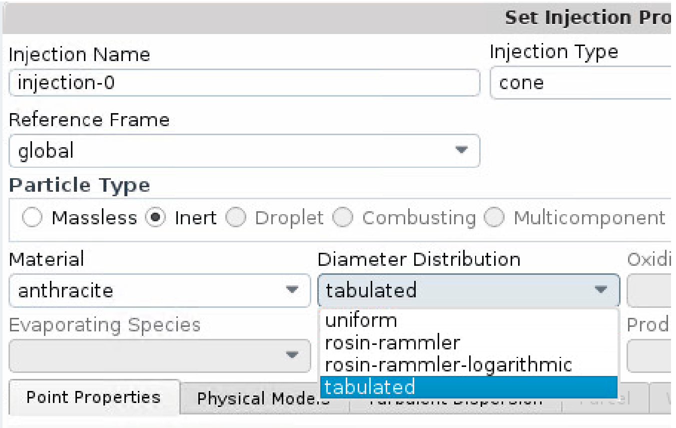
*7x cheaper hardware purchase cost and 4x lower power consumption**

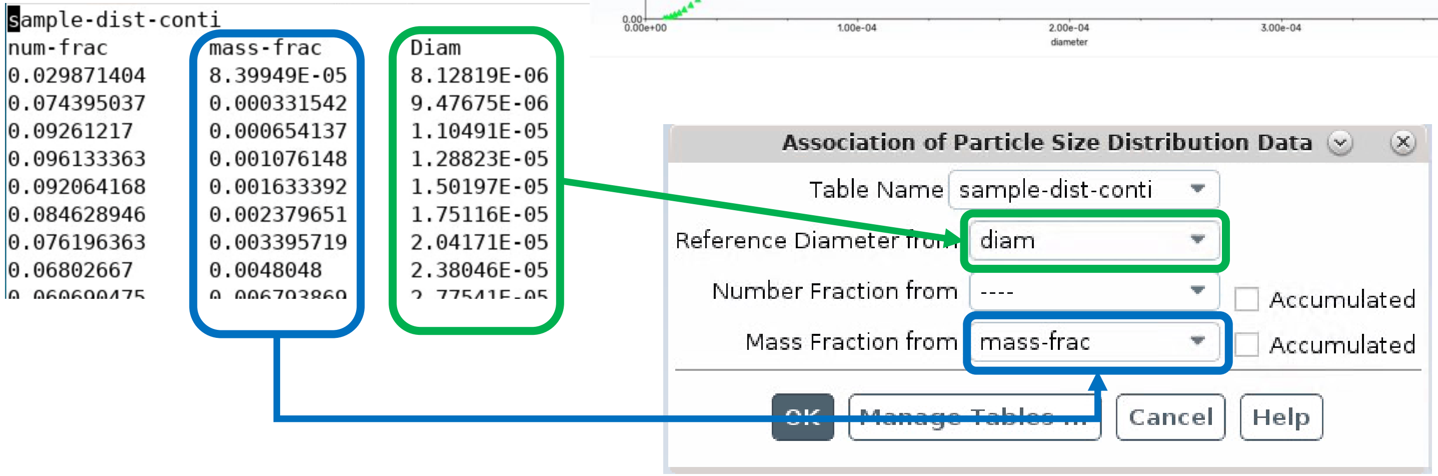
** 1024 core CPU cluster using 9600 W versus 6*V100 server using 2400 W*

Tabulated Particle Size Distribution

- Pre 2022 R1
  - Uniform and Rosin-Rammler particle diameter distributions available
- 2022 R1 Functionality
  - Import **tabulated** data for injection size distributions
  - For surface and cone injections
  - Specify mass and / or number frequency distribution of measured particle sizes
    - Either as frequency or cumulative distributions

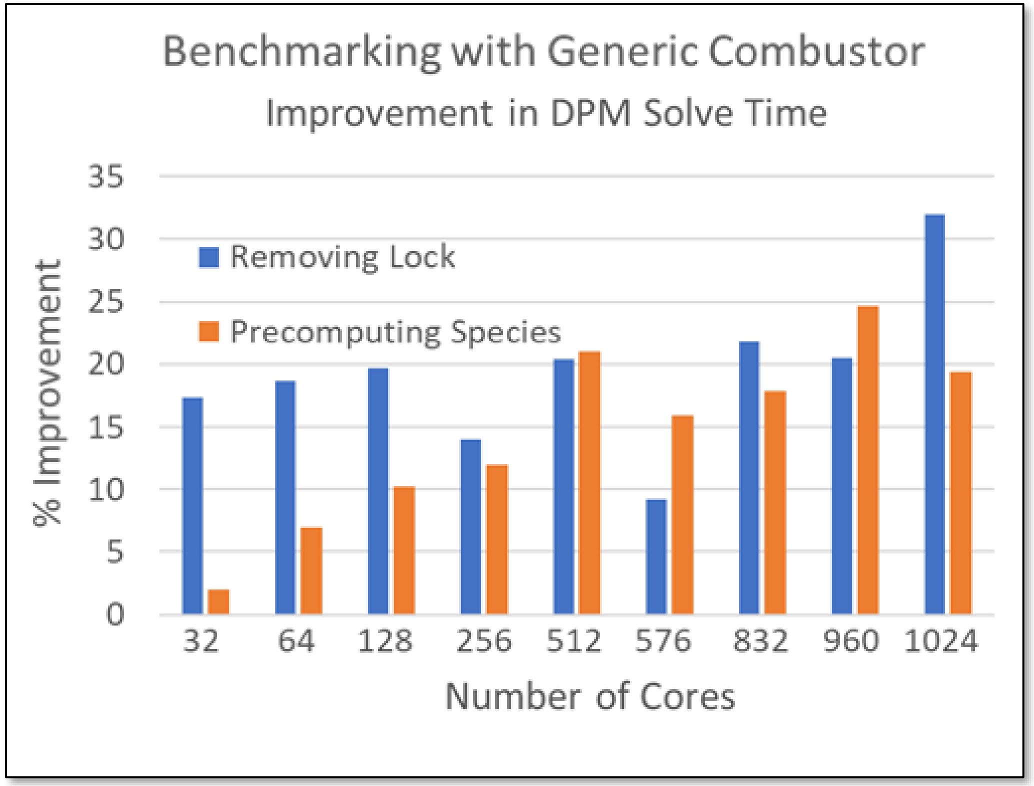






Improved Particle Tracking

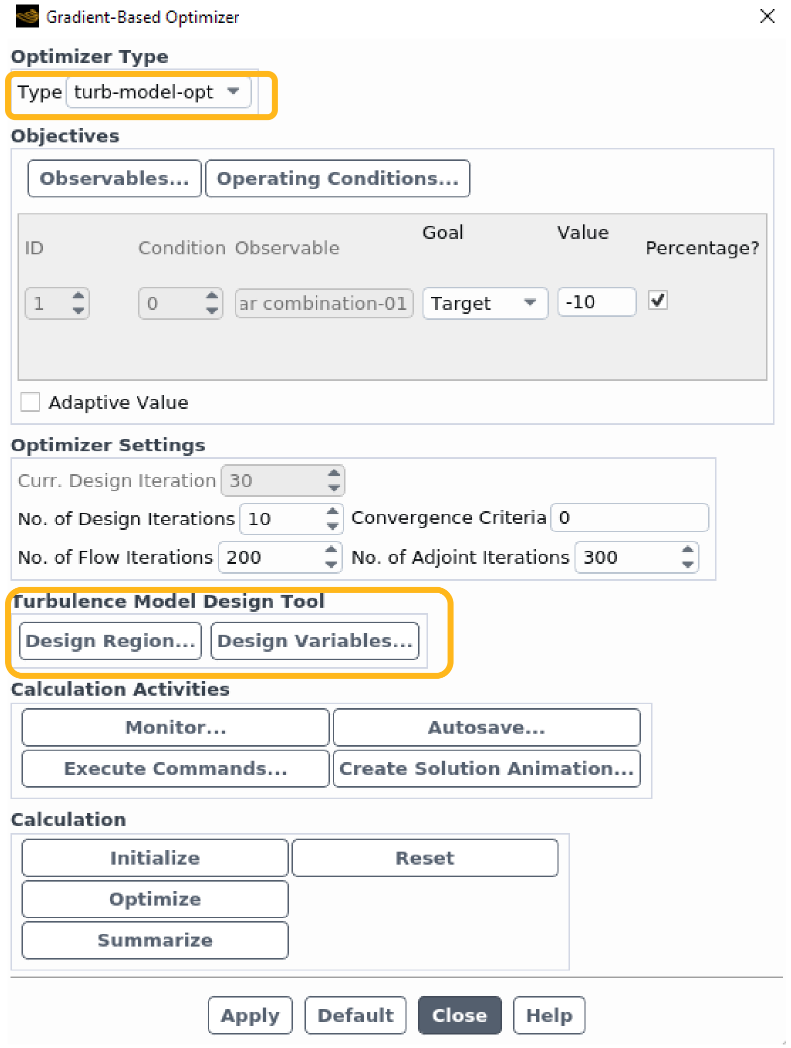
- 2022 R1 Functionality
  - Performance improvements
    - Thread lock removal improves core hybrid tracking performance
    - Precompute species option
      - Improves performance for combustion cases with all available tracking approaches and parallel modes
    - Further performance options for GTC simulations
      - Barycentric intersections
      - Timestep adaption of Darmofal and Haimes
  - Accuracy and Robustness improvements
    - Improved variable interpolation for
      - Multiphase models
      - Steep gradients within a cell
      - Lagrangian wall film particles



Significant reduction in DPM solve time
Improved robustness and accuracy in various scenarios

Turbulence Model Optimizer: GEKO

- Pre 2022 R1:
  - The GEKO model consolidates 2-equation turbulence modeling into a single model
  - Can be adjusted to different flow regimes with 4 tunable coefficients: CSEP, CMIX, CNW, Blending function
- 2022 R1 Functionality
  - Turbulence model optimizer to enhance GEKO accuracy
  - TMO Workflow:
    - Training:
      - Optimize turbulence coefficients to match data from high fidelity simulation/experiment
      - Devise Neural Network to generalize the correlation between the optimized GEKO coefficients and flow features
    - Deployment:
      - Incorporate the devised neural network into other similar simulations to reproduce tuned-GEKO behavior



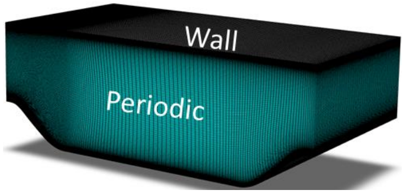
Train the GEKO turbulence model to yield results more like a high-fidelity, high-cost scale-resolving simulation

Turbulence Model Optimizer Example

Training: **3D Periodic Hills Case**

Deployment

2.5D Hump Case



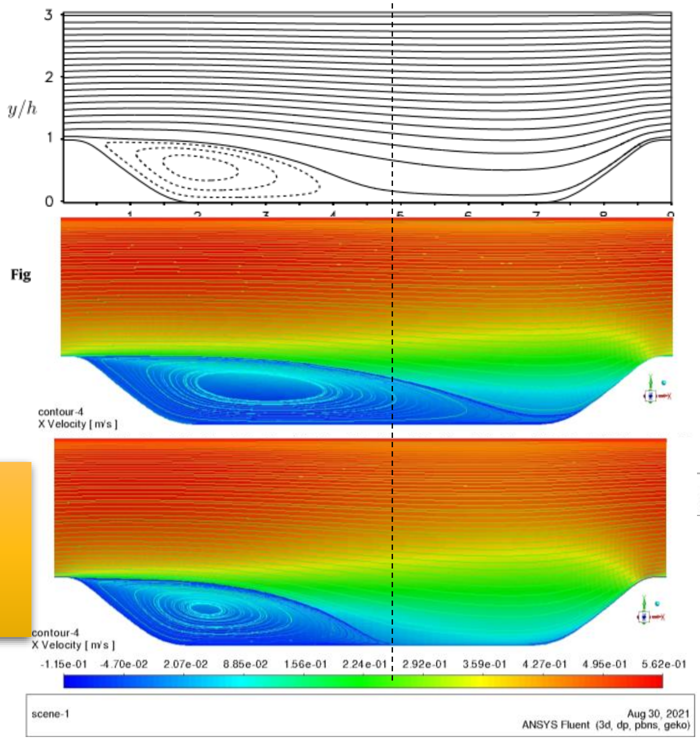
$Re_H = 10400$

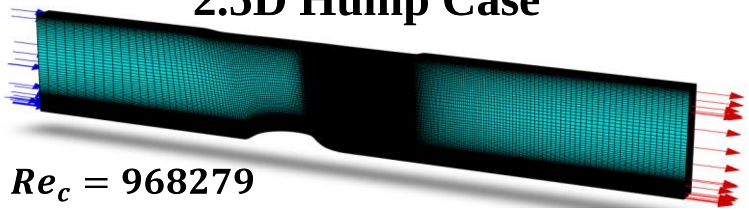
M. Breuer et al

LES Reference

GEKO Default

Neural Network Augmented





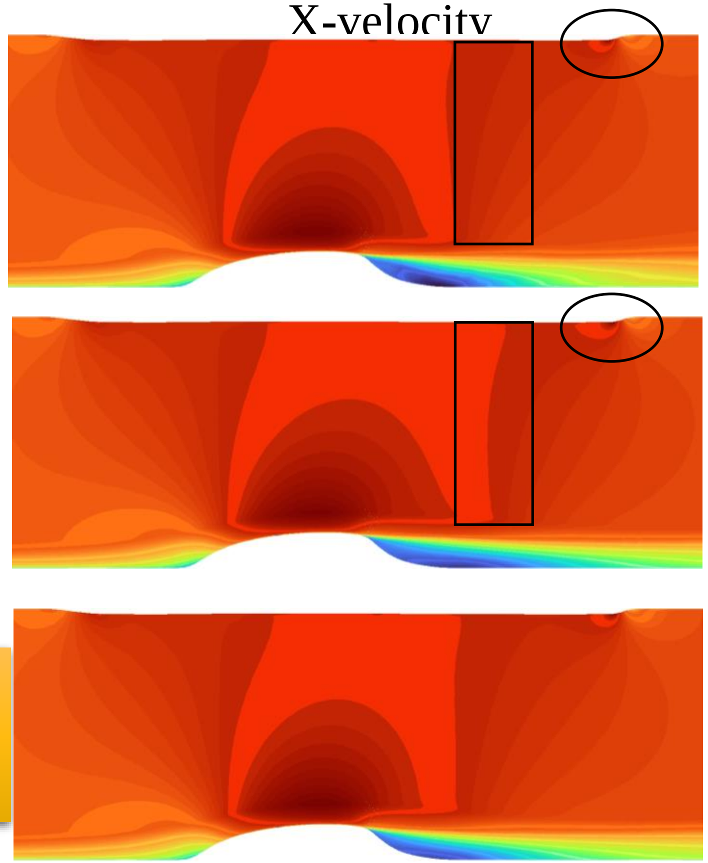
$Re_c = 968279$

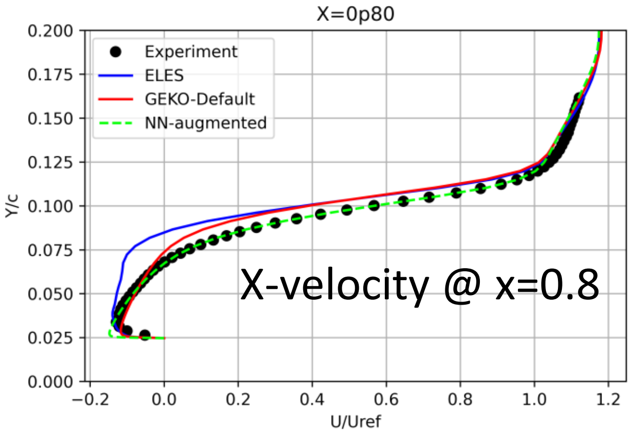
X-velocity

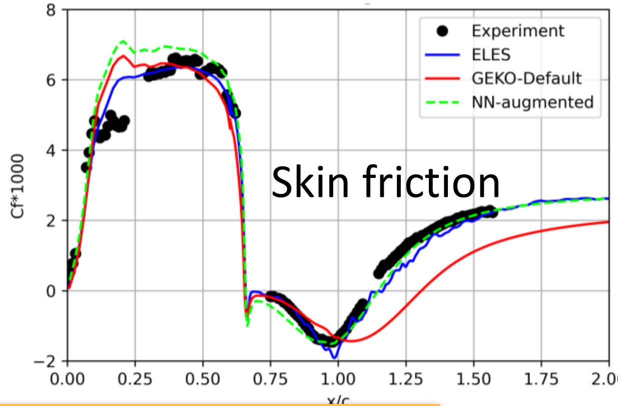
ELES Reference

GEKO Default

Neural Network Augmented





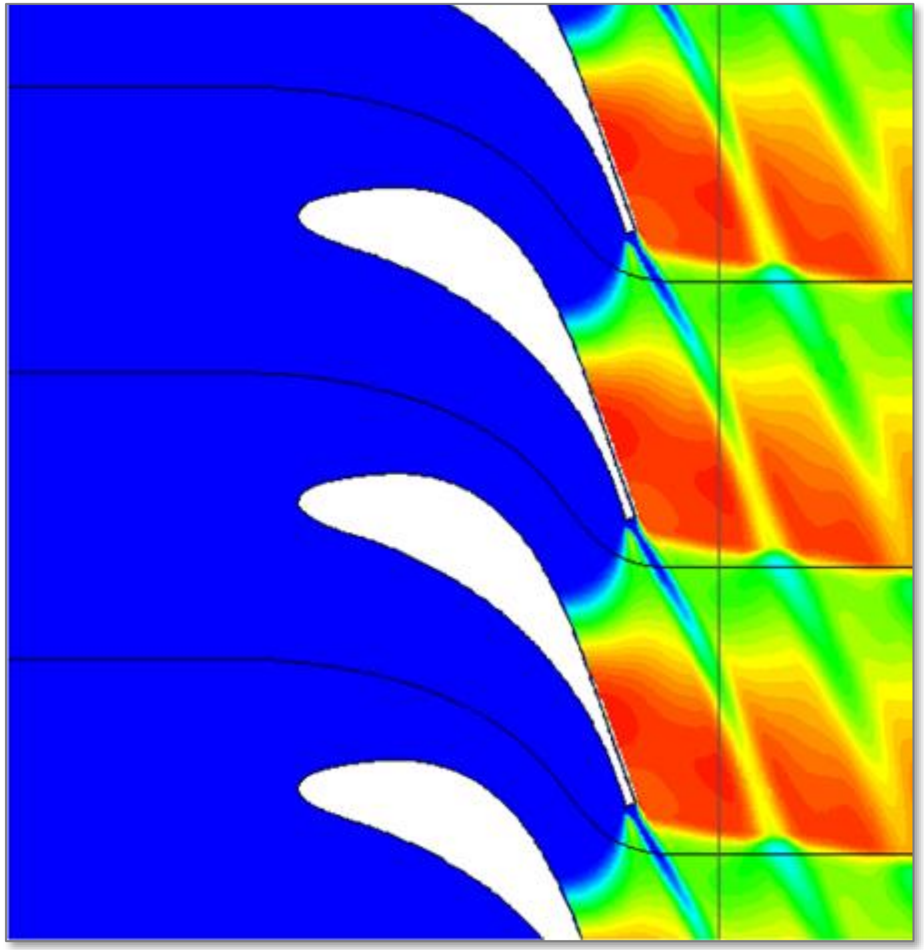


NN augmented GEKO velocity nearly match ELES/Exp.

Fluent: Non-Equilibrium Wet Steam Model

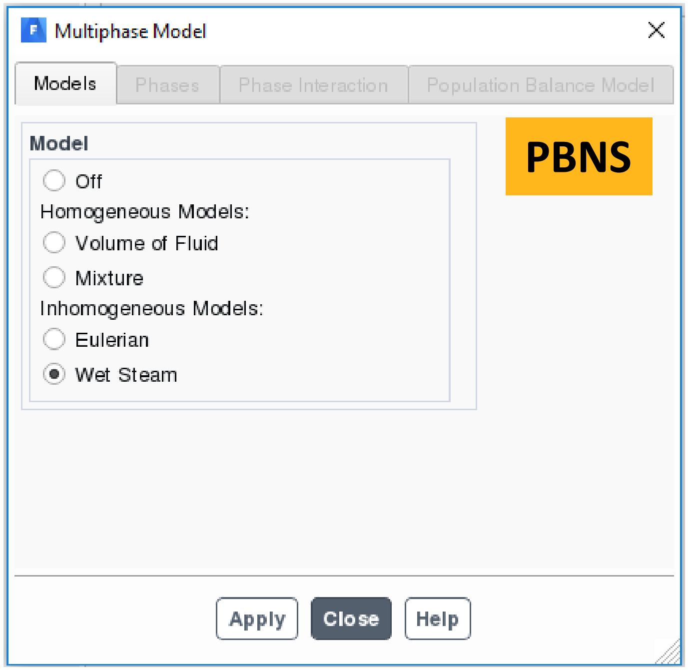
Access to IAPWS97 provides industry standard steam properties

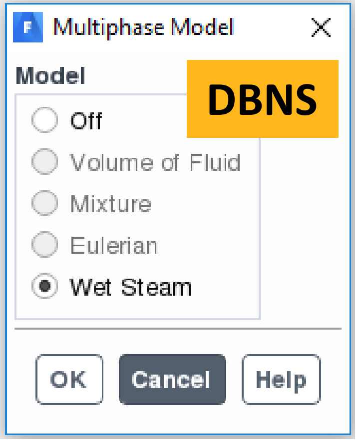
- Real Gas Property (RGP) tables for wet steam model
  - Alternative to the built-in thermodynamic wet steam properties, using the industry standard IAPWS97
    - Same format as used in the CFX real gas model
  - Built-in RGP file for steam available:
    - Turn on Turbo mode, turn on wet steam model
    - Read RGP file: file/table-manager/read-rgp-file
    - Link the RGP to wet steam model: define/models/multiphase/wet-steam/set/rgp-tables
- Enhancements to Non-Equilibrium Wet Steam
  - Convergence improvements for 2nd-order wet steam model
  - Alternate stagnation condition computation available:
    - The default method is based on mixture of vapor and liquid droplets
    - The alternative is based on vapor gas phase (similar to CFX)
    - Accessed via TUI: define/models/multiphase/wet-steam/set/stagnation-conditions



Liquid mass fraction in a stationary cascade of steam turbine blades

Wet Steam Model: User Interface



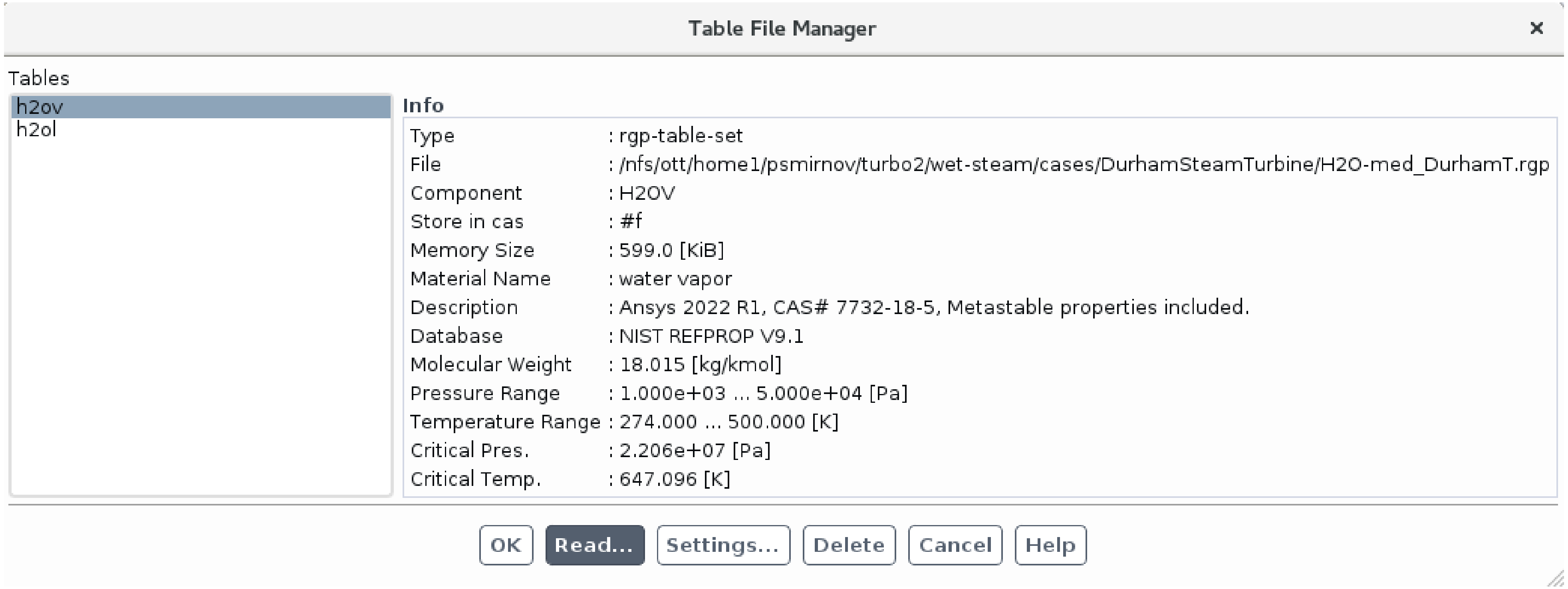


Text User Interface

```
/define/models/multiphase/wet-steam/set>
droplet-growth-rate      rgp-tables                virial-equation
max-liquid-mass-fraction stagnation-conditions

/define/models/multiphase/wet-steam/set> droplet-growth-rate
Use Hill's formula for the droplet growth rate? (if no, Young's formula will be used) [no] no
Enter modeling parameter, alpha [9] 9
Enter modeling parameter, beta [1] 1

/define/models/multiphase/wet-steam/set> virial-equation
Use Young's virial equation of state? (if no, equation from Vukalovich will be used) [no] no

/define/models/multiphase/wet-steam/set> max-liquid-mass-fraction
maximum value for liquid-phase mass-fraction? [0.1] 0.1

/define/models/multiphase/wet-steam/set> stagnation-conditions
Compute stagnation conditions using the gas phase only? (if no, mixture will be used) [no] no

/define/models/multiphase/wet-steam/set> rgp-tables
Use RGP tables for the wet steam model? [no] no
```

User Interface to link the RGP table to WS model

Step 1: load the file with the table using Table File Manager, File-> Table File Manager ...



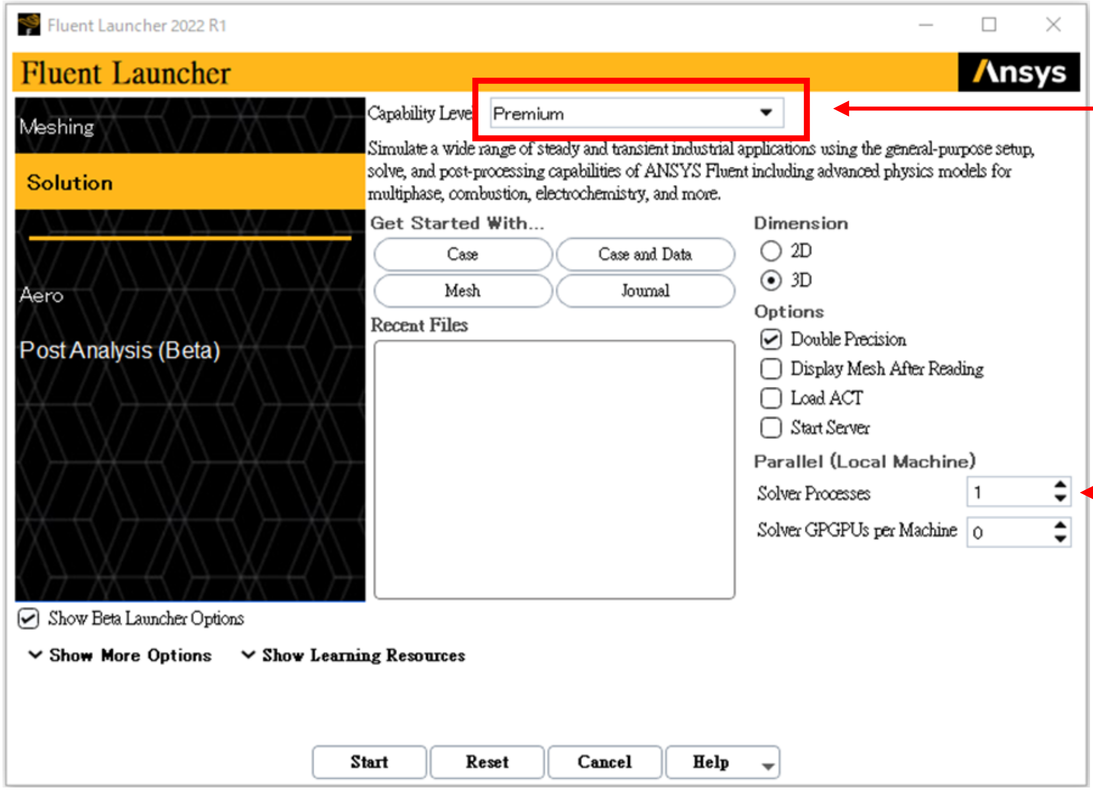
Step 2: in the TUI for the Wet Steam model, link the loaded table as follows:

```
/define/models/multiphase/wet-steam/set>
droplet-growth-rate      rgp-tables              virial-equation
max-liquid-mass-fraction stagnation-conditions
```

```
/define/models/multiphase/wet-steam/set> rgp-tables
Use RGP tables for the wet steam model? [no] yes
Available tables: (h2ov h2ol)
Select table [] h2ov
Linked table: h2ov
```

Fluent TurboWorkflow

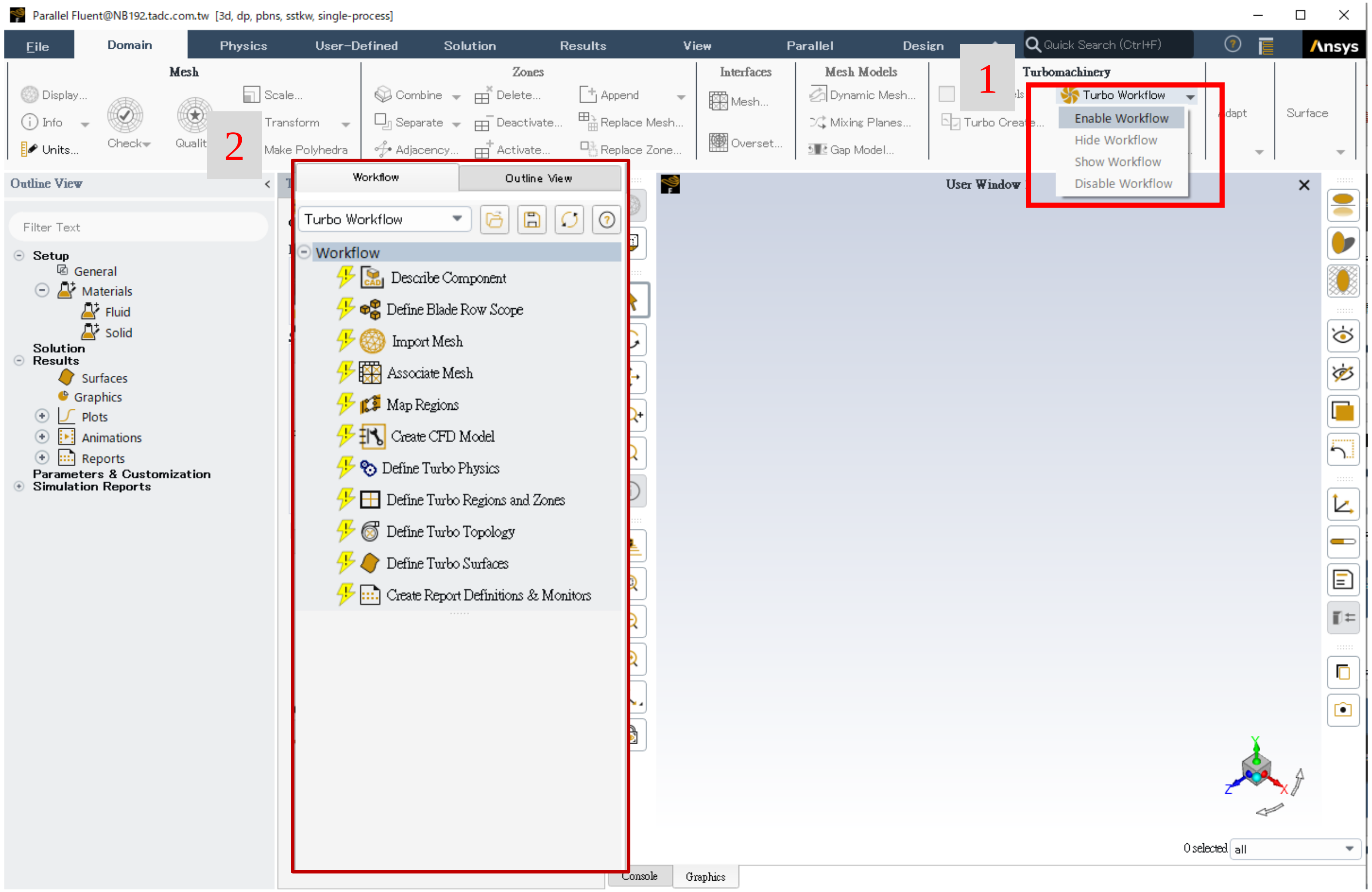
開啟Fluent



License授權必須是Premium以上(Premium或Enterprise)

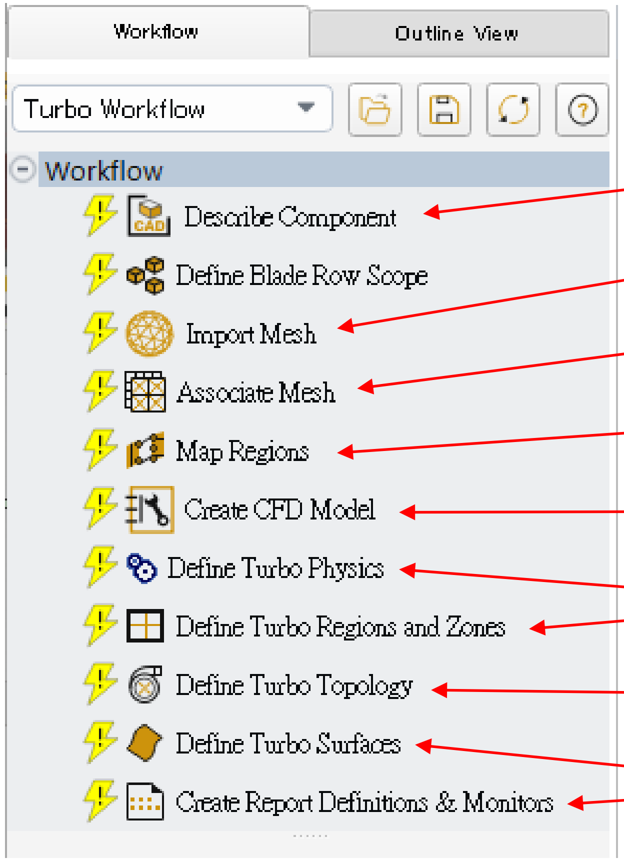
啟用多核心

開啟Fluent TurboWorkflow



1. 進入Fluent介面後，於上方Ribbon區域開啟Turbo Workflow。
2. 開啟Turbo Workflow後會自動進入到Turbo model的模式，此時可看到流程式的設定模式進程。

TurboWorkflow



分析的類型問題定義

匯入葉片的網格模型

將網格與定義的問題進行關聯的匹配

將網格命名的邊界進行定義匹配(如定義Blade、Hub…的命名匹配)

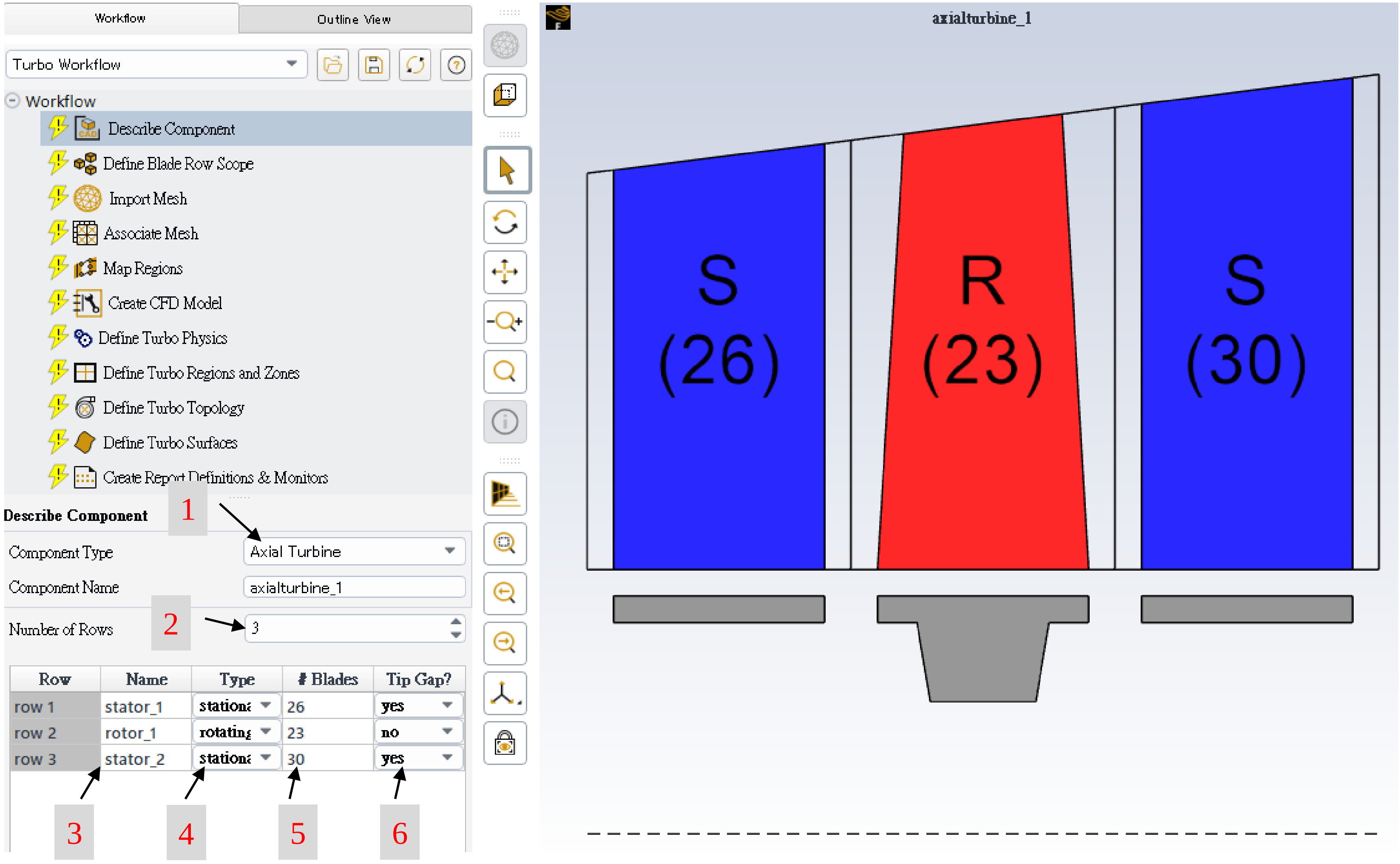
建構出CFD分析的模型定義

定義分析的邊界條件等問題

邊界條件上的整合定義(如將s1與r1的hub進行合併)

自動化的後處裡定義

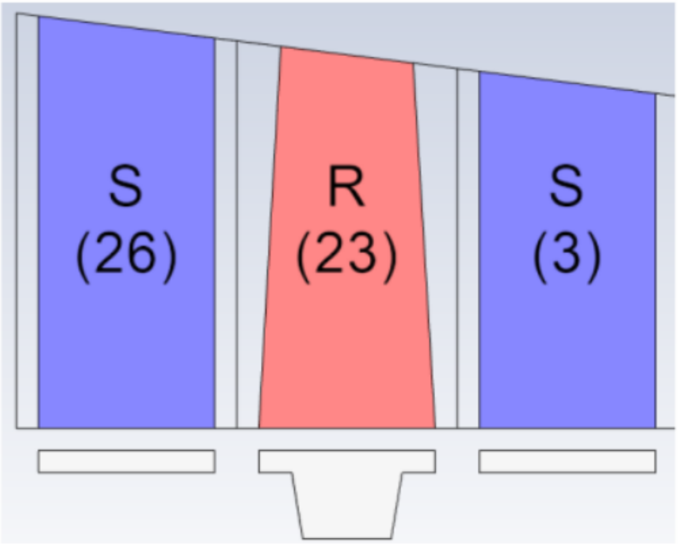
Describe Component



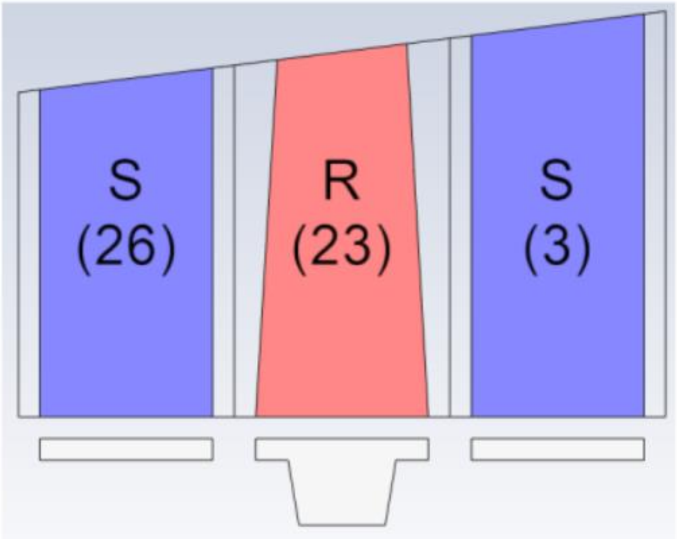
1. 描述分析的問題類型，清單下拉可選擇Compressor及Turbine
2. 描述問題Row的數量
3. 每個Row的命名
4. 每個Row的類型定義，定子/轉子
5. 葉片的數量
6. 是否於翼尖處含有Gap空間

Component type

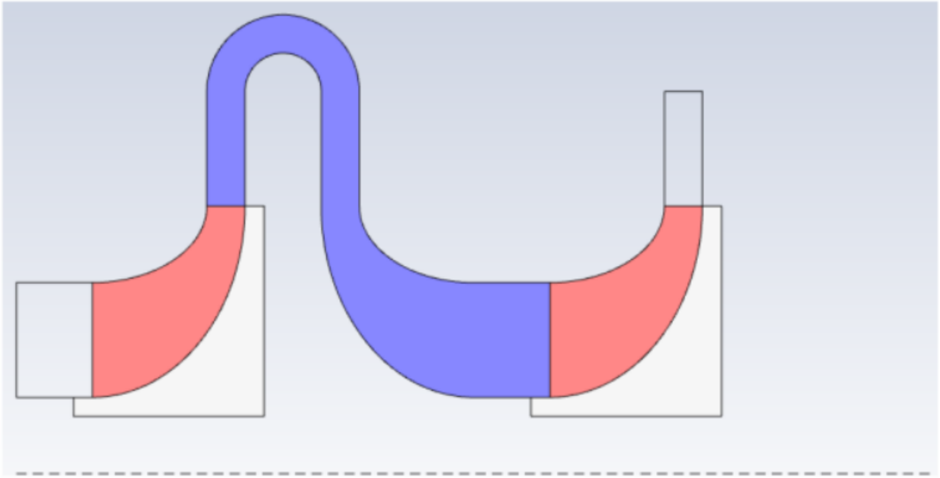
Axial Compressor



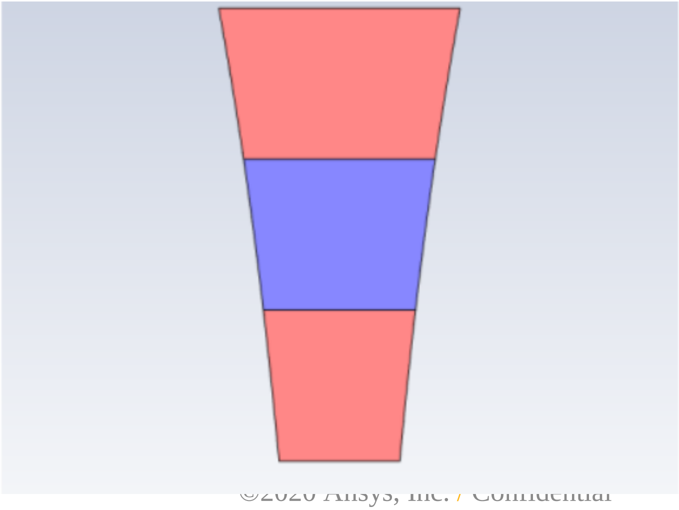
Axial Turbine



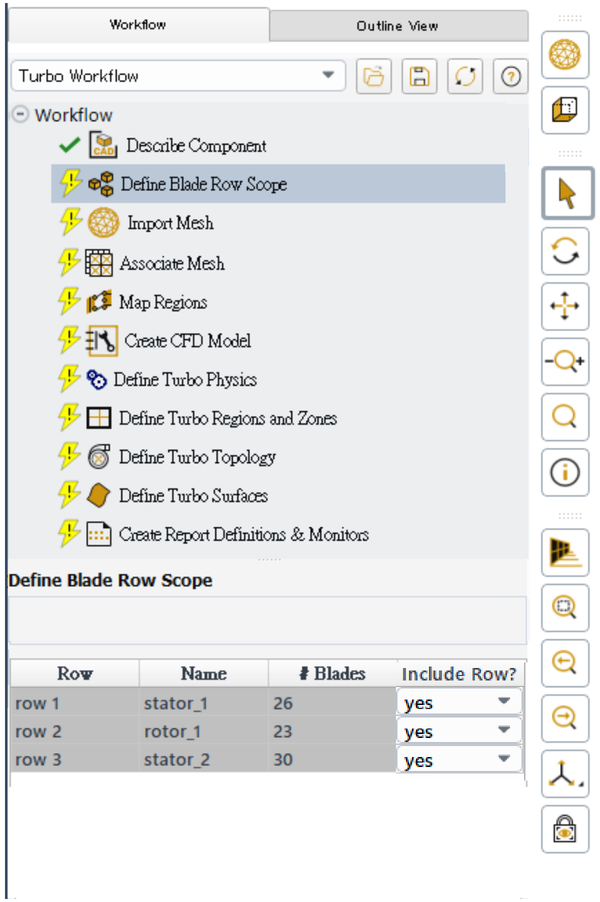
Radial Compressor



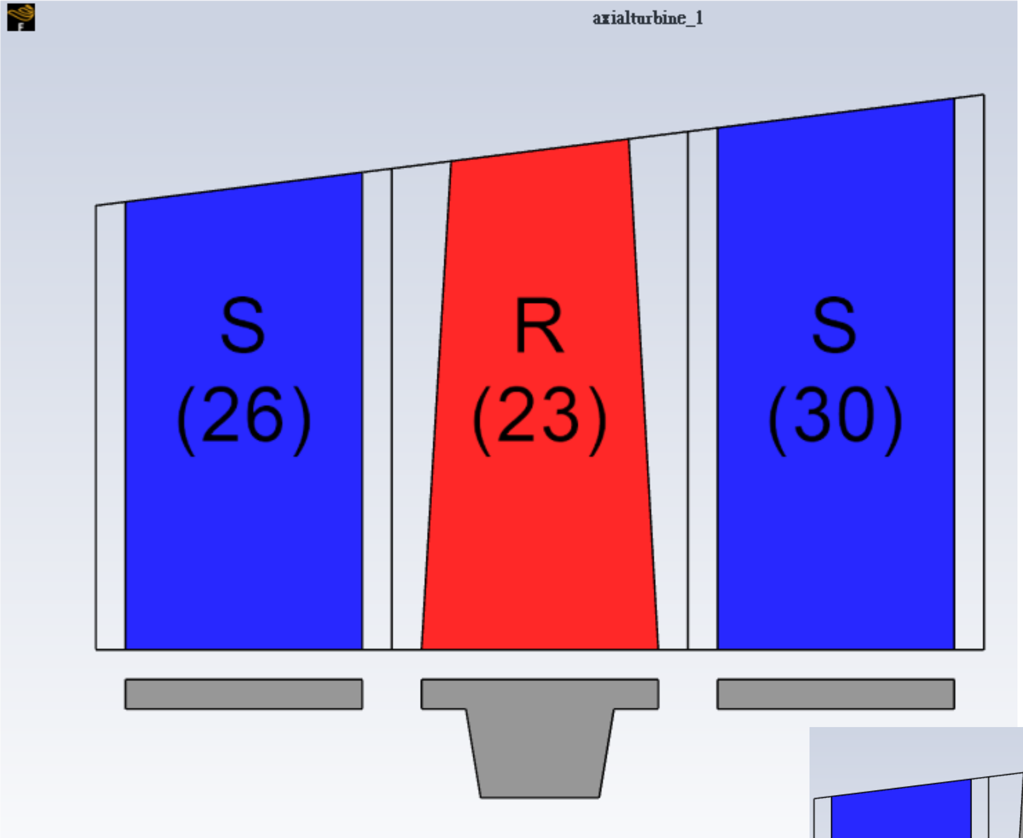
Radial Turbine



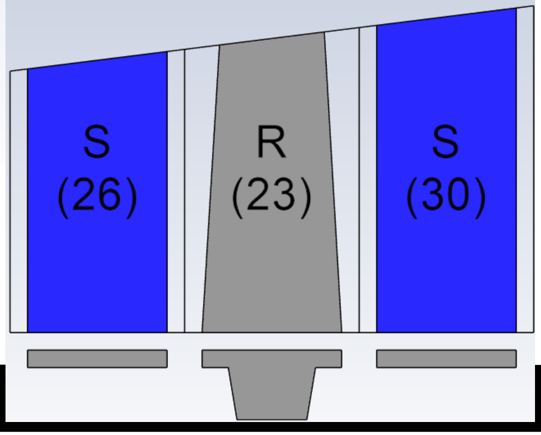
Define Blade Row Scope



Row	Name	# Blades	Include Row?
row 1	stator_1	26	yes
row 2	rotor_1	23	yes
row 3	stator_2	30	yes



Row	Name	# Blades	Include Row?
row 1	stator_1	26	yes
row 2	rotor_1	23	no
row 3	stator_2	30	yes

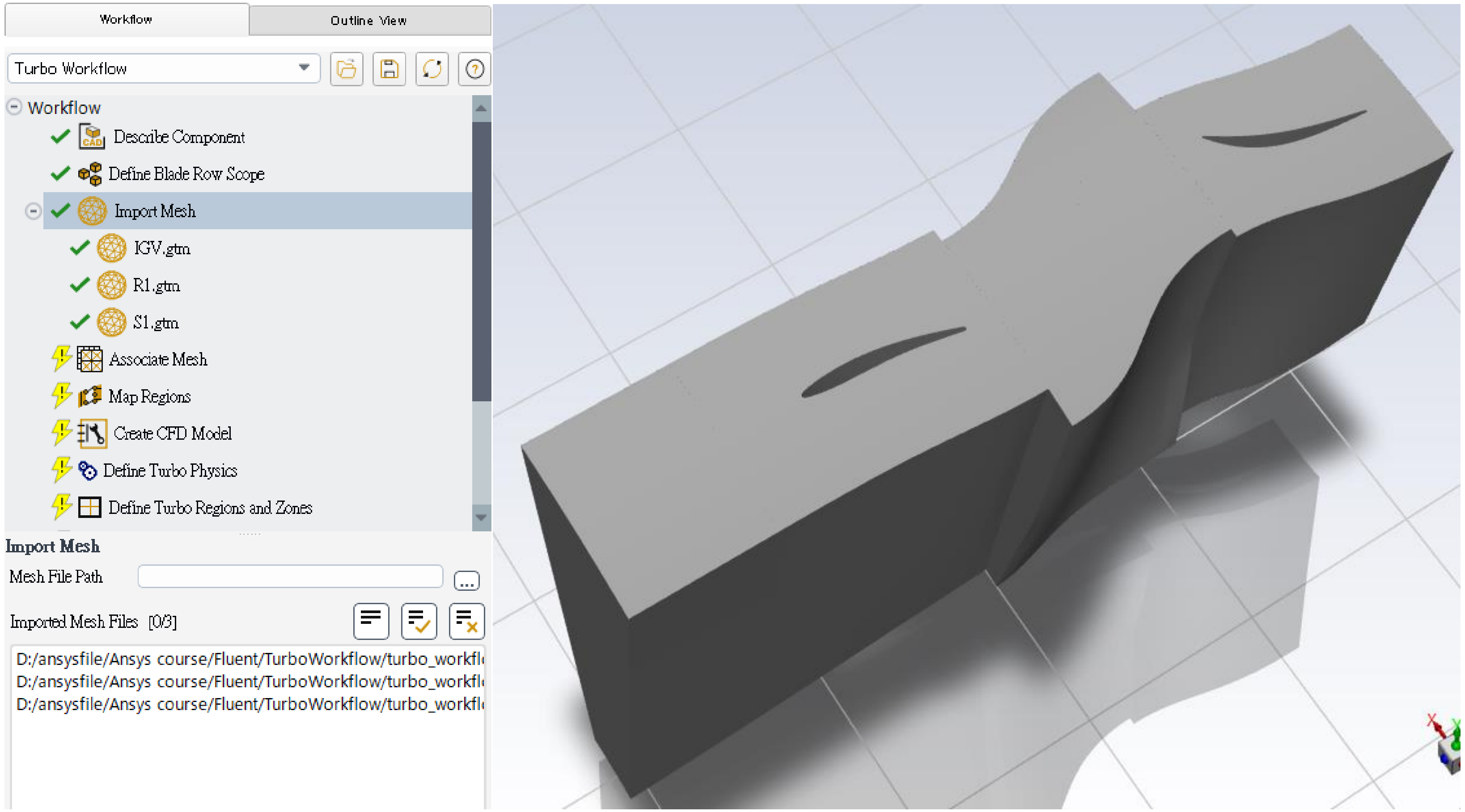


對於上一步驟所描述的Turbine或Compressor，可考慮是否要進行後續的問題定義與更進一步的分析

若只想對某幾個Row進行分析，則可以在Include Row中改成no，Row跟Row中請注意需是相連的

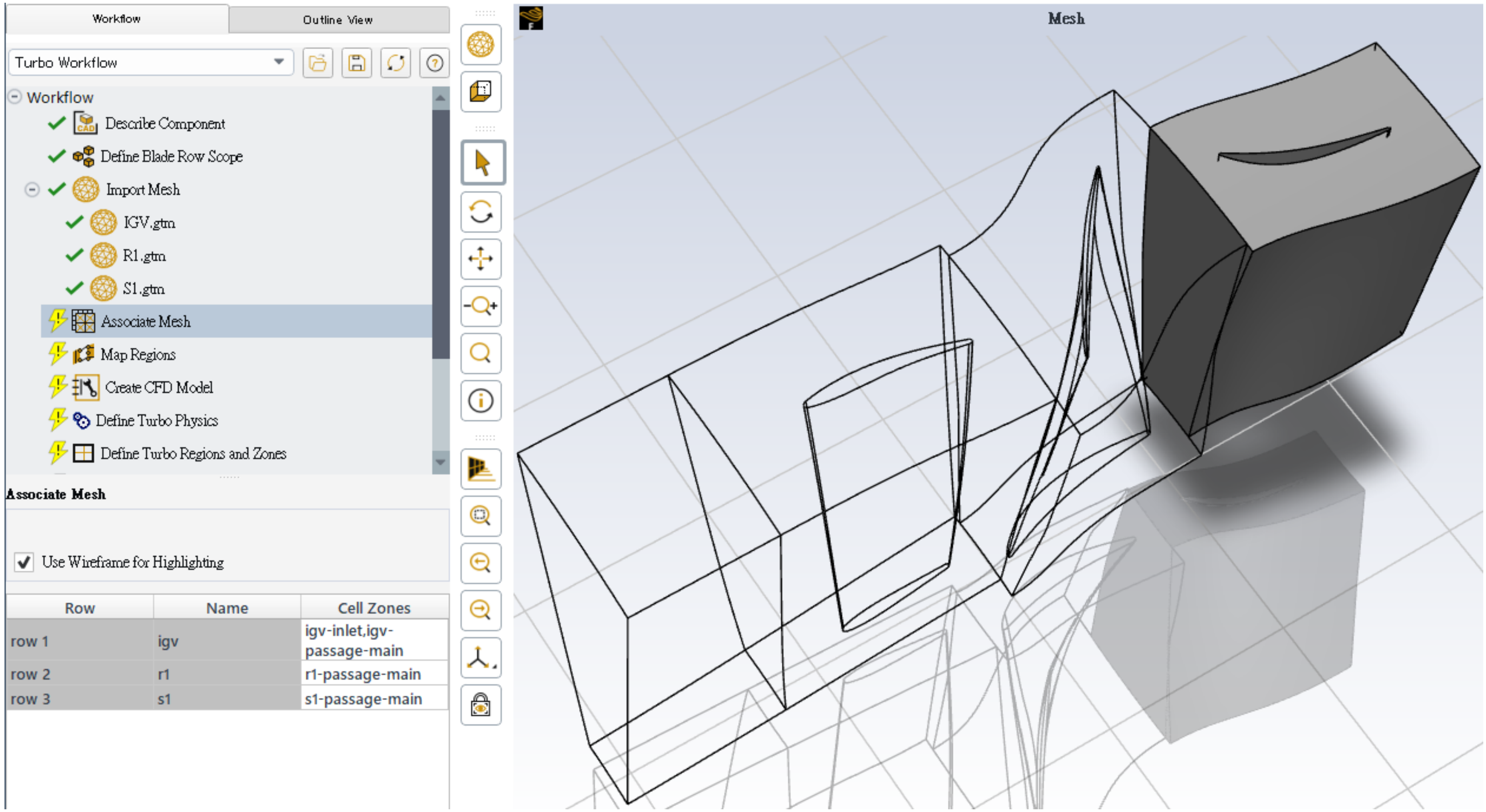
Ansys

Import Mesh



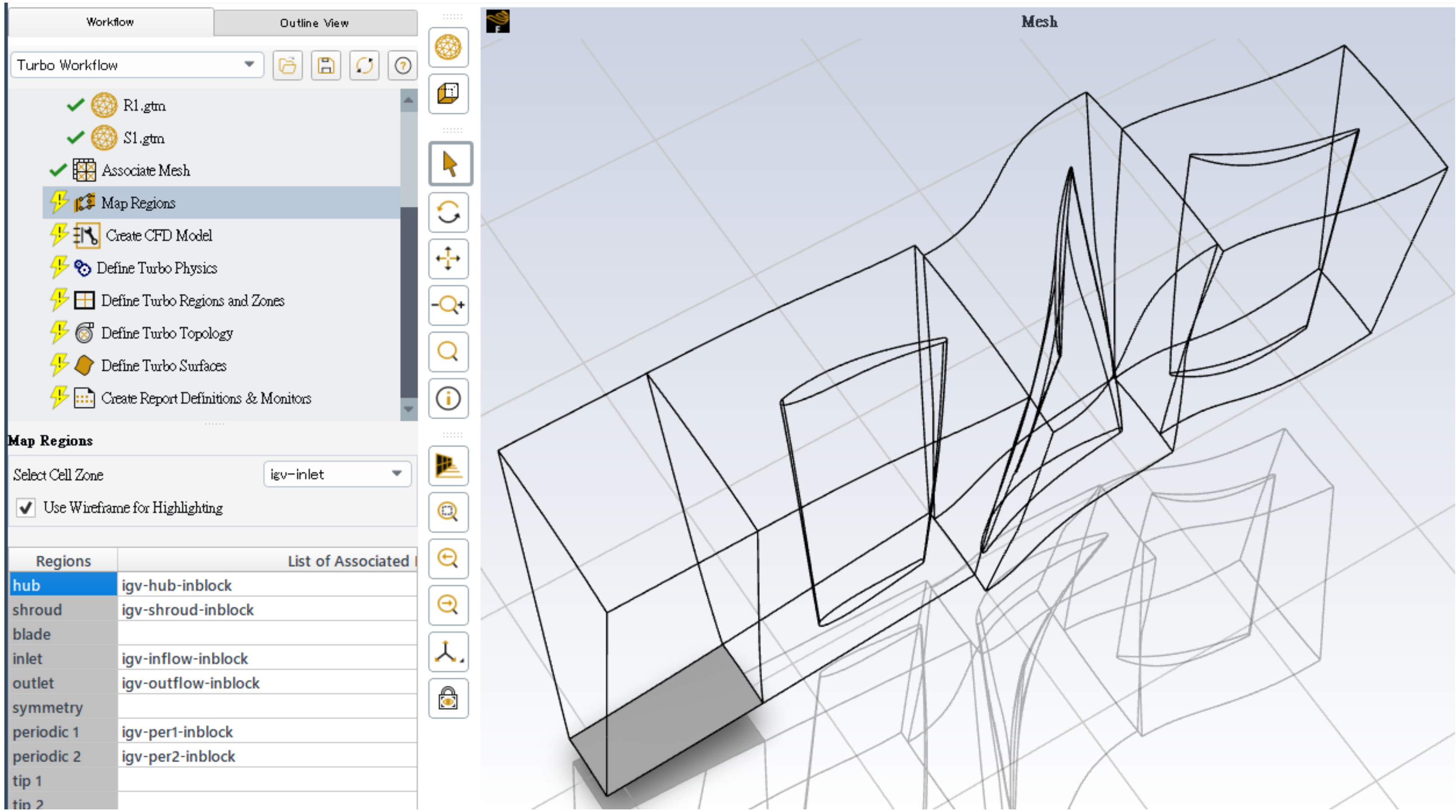
將網格匯入其中，其檔案格式可讀取.msh、.def、.cgns、.gtm

Associate Mesh



將匯入的網格與所描述的問題定義進行相關性，即將各網格定義到各個Row區域中

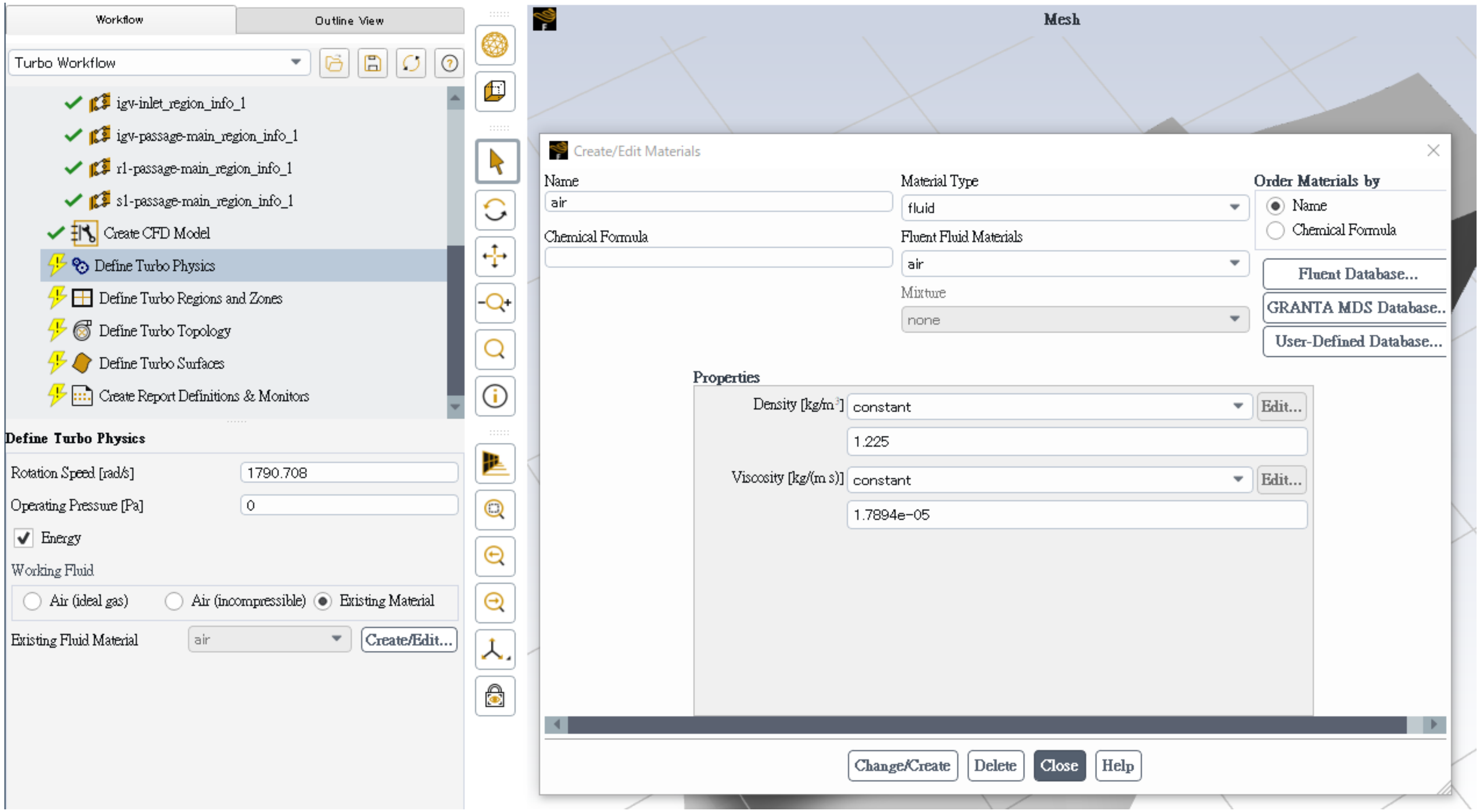
Map Regions



匯入的網格中所命名的邊界與進行Mapping的動作，例如將網格hub處的邊界區域mapping到程式中hub的區域中

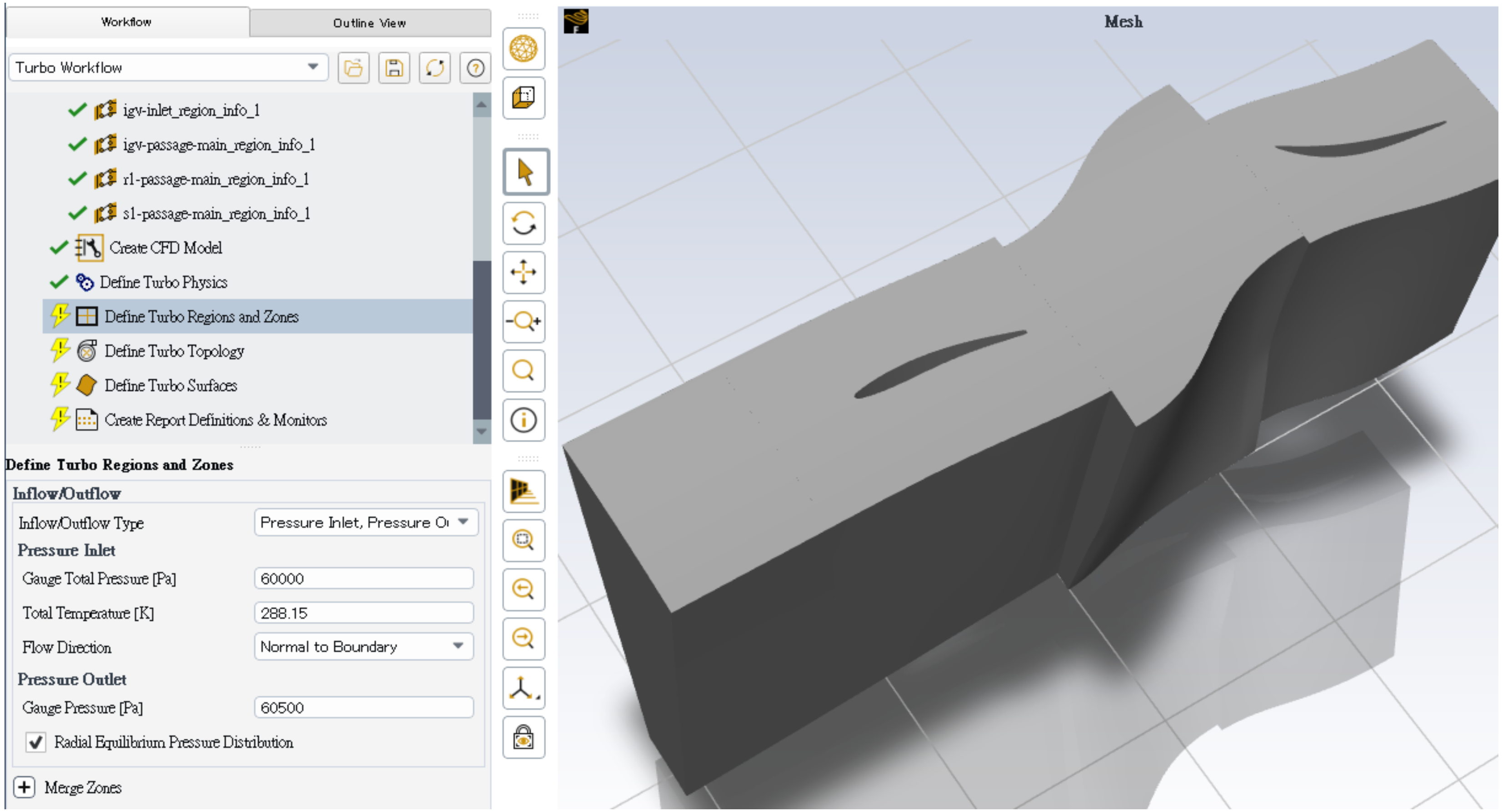
Ansys

Define Turbo Physics



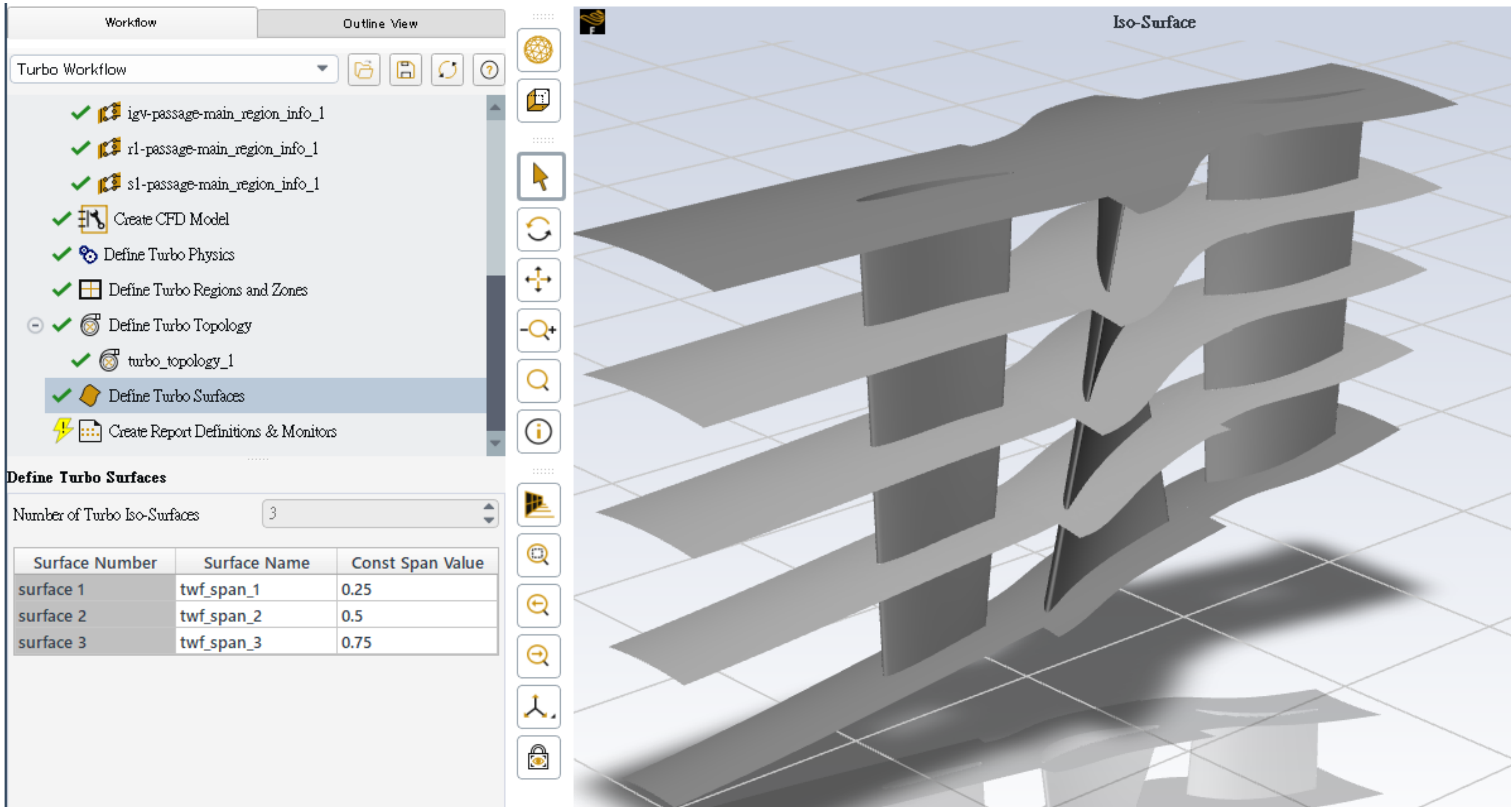
設置分析的物體條件，如轉速、材料性質等等，其材料性質可自行定義

Define Turbo Regions Zones



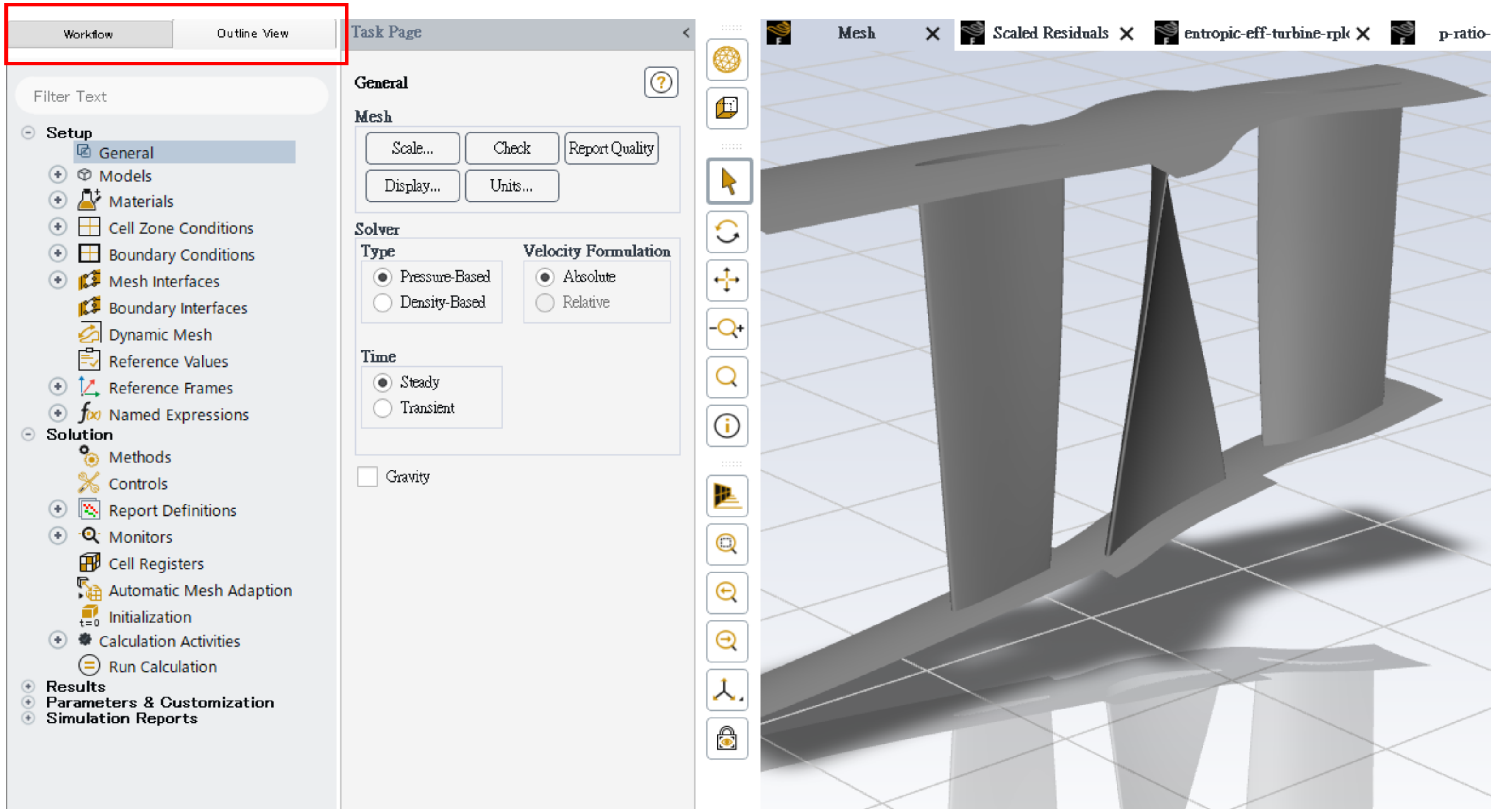
設置分析的邊界條件，可設置壓力出/入口或是值流量出/入口

Define Turbo Surface / Create Report



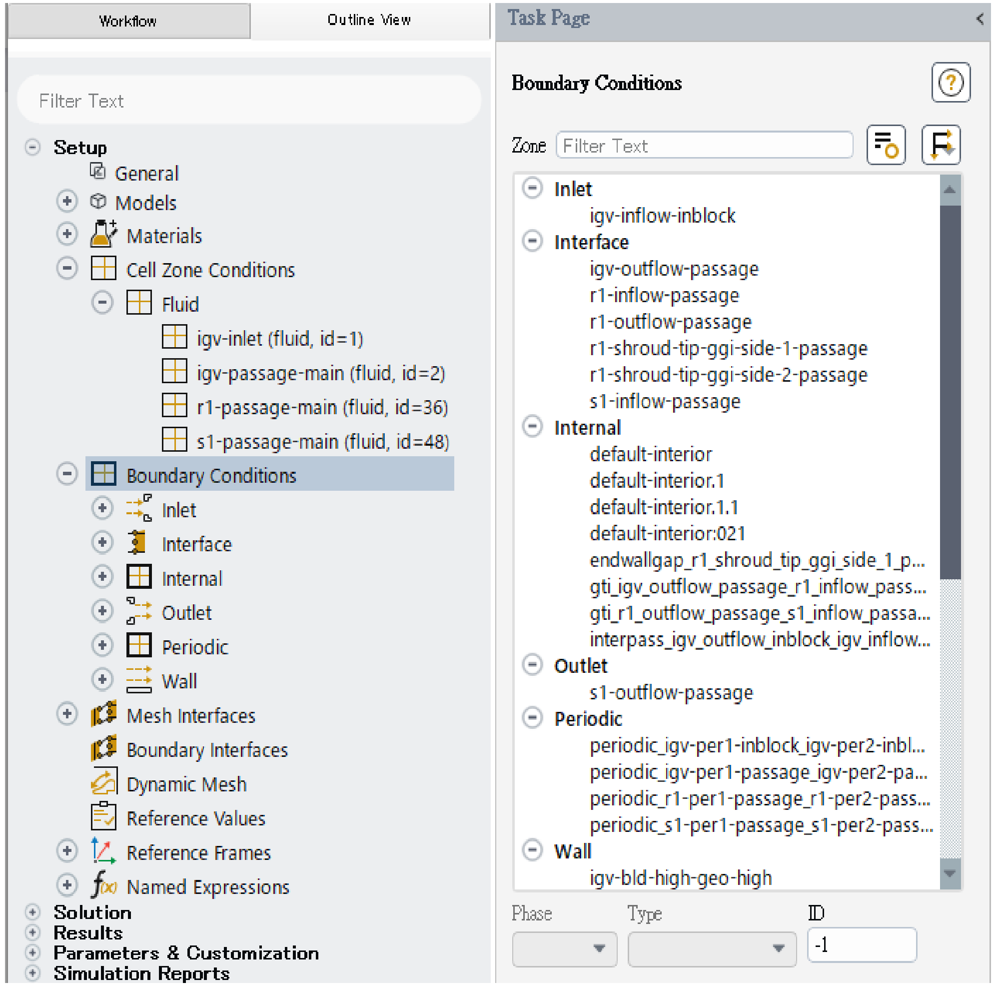
自動化建構出等值的切面位置與報告觀測值，以便於使用者在收斂上的監控與後處理的觀察

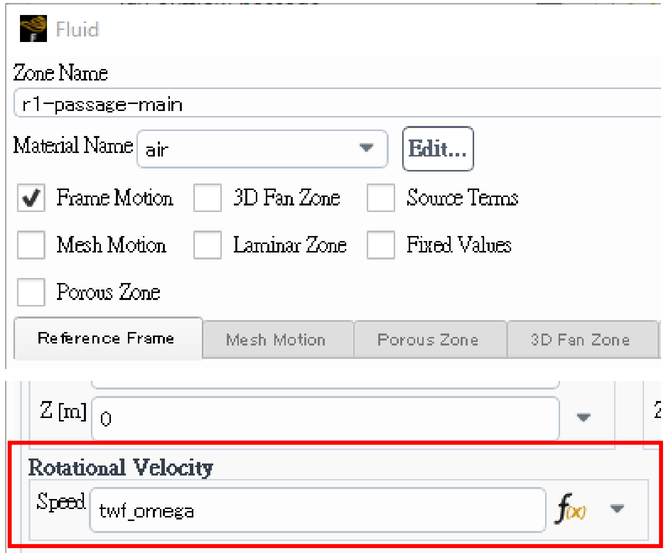
Fluent Outline

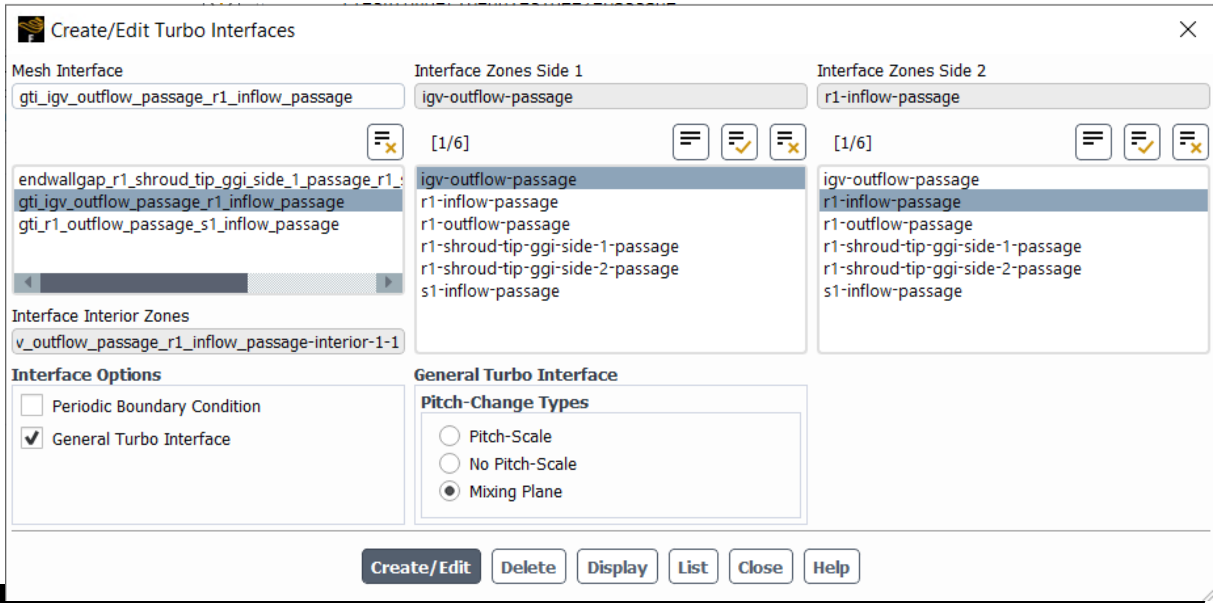


設定流程完成後，切換回到Fluent的Outline中，可發現大部份的的設定都在Turbo Workflow中完成了

Boundary Condition Setting

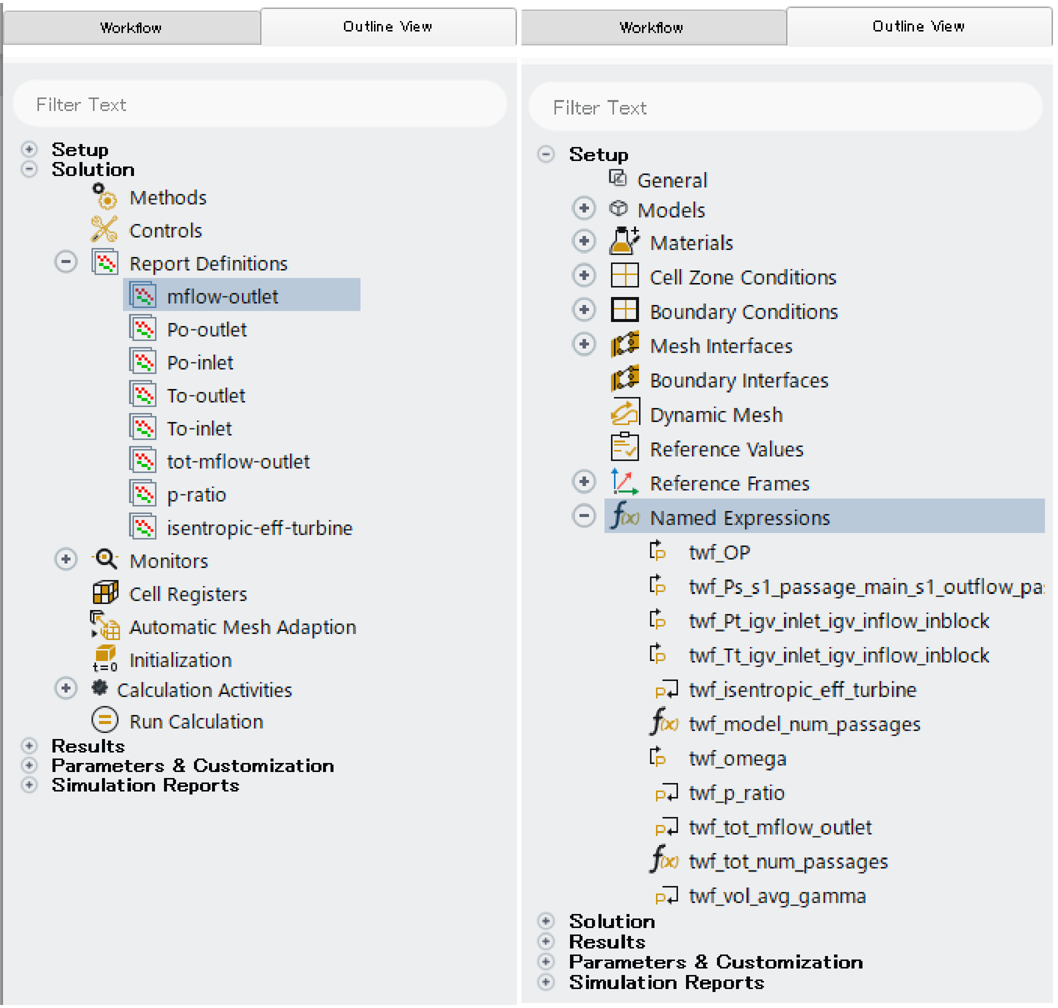


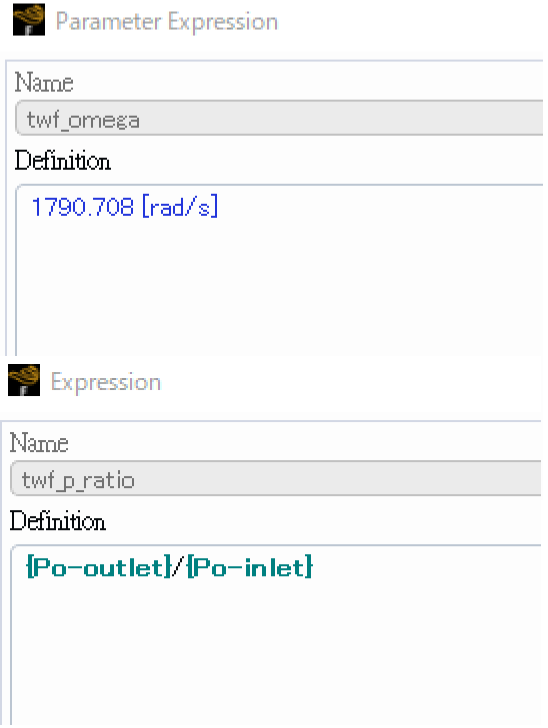




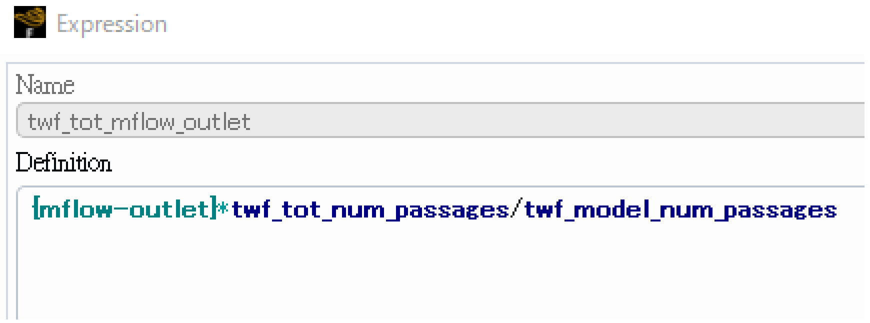
邊界條件根據在Turbo Workflow中的設定自行設置到各條件內，並且包含了Interface的接口也自行設置完成

Expression

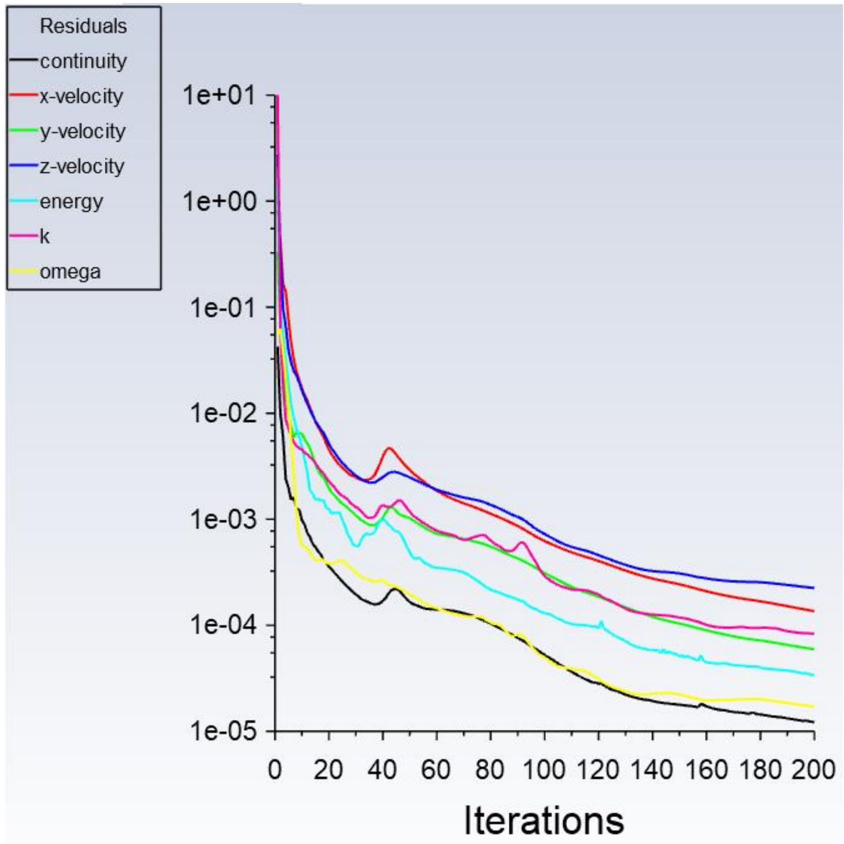


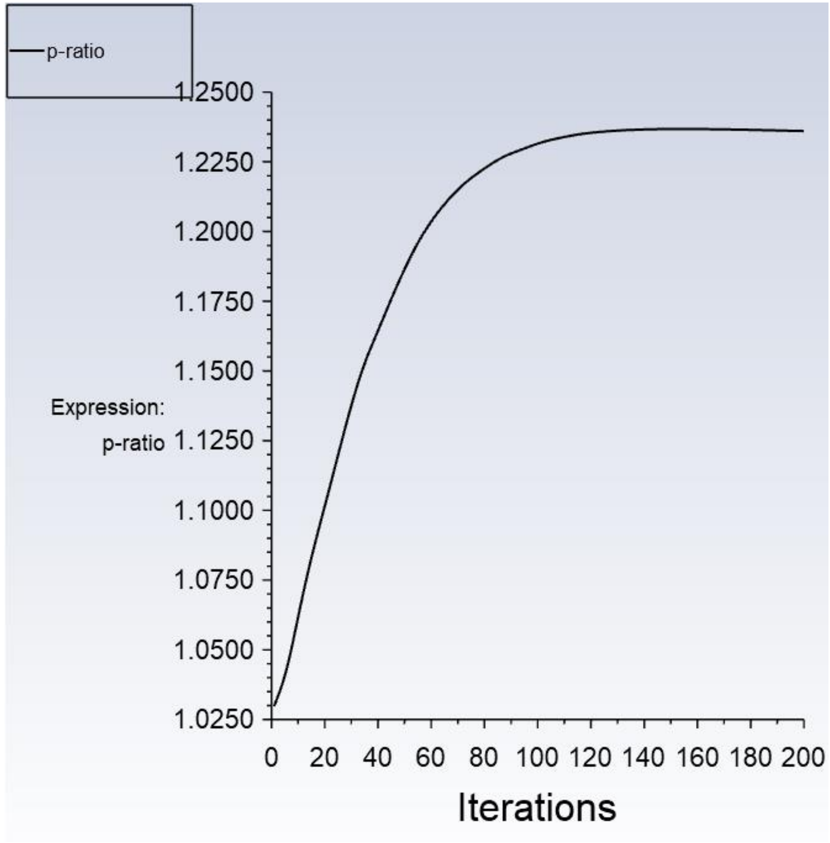


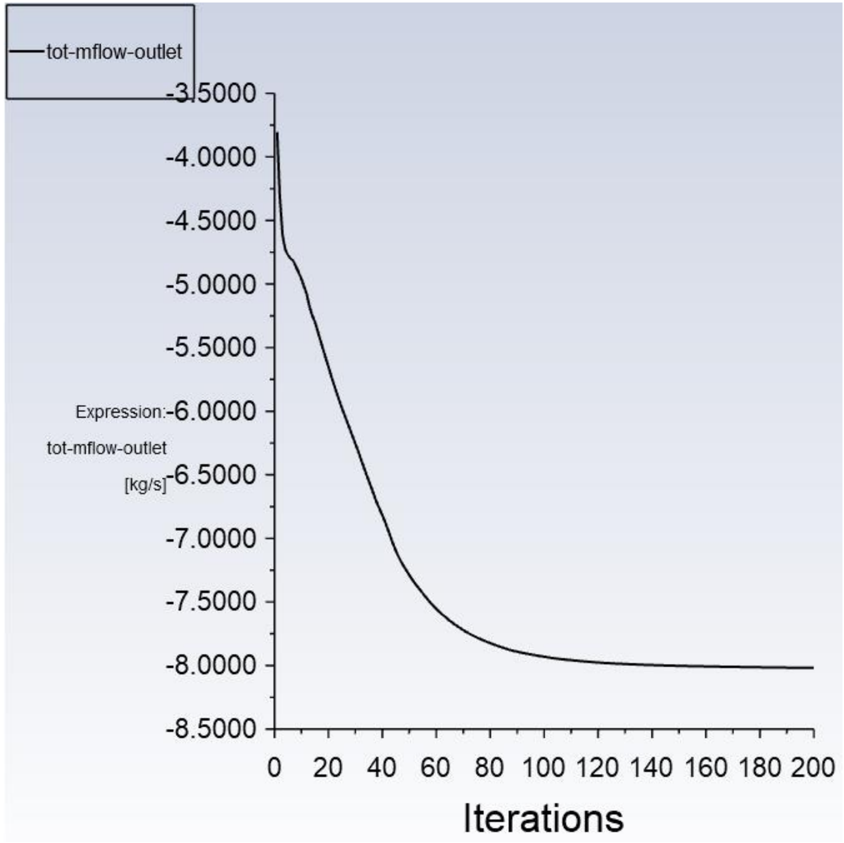
Expression中除了邊界條件外，Workflow也定義了在後處理上常會觀測到的數值，如值流量、壓縮比等等



Calculation

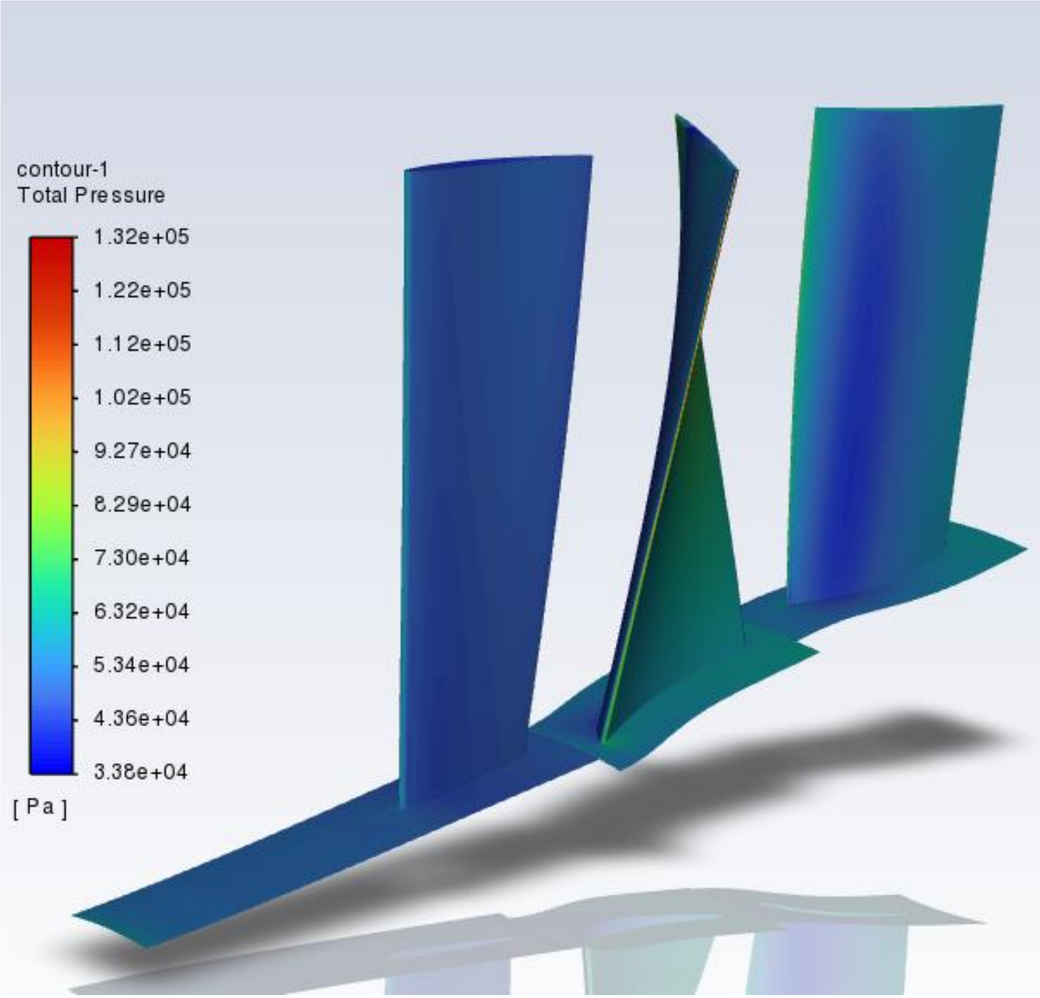




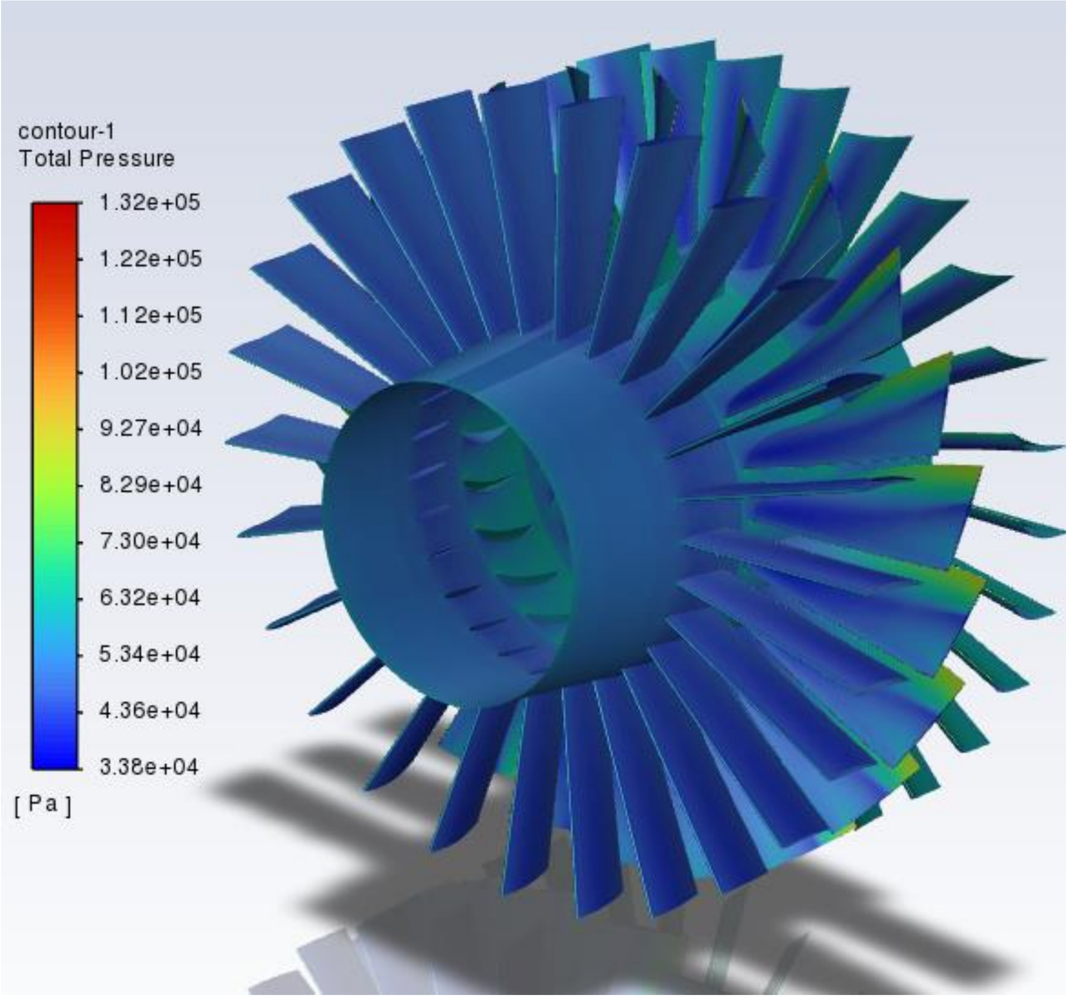


Result

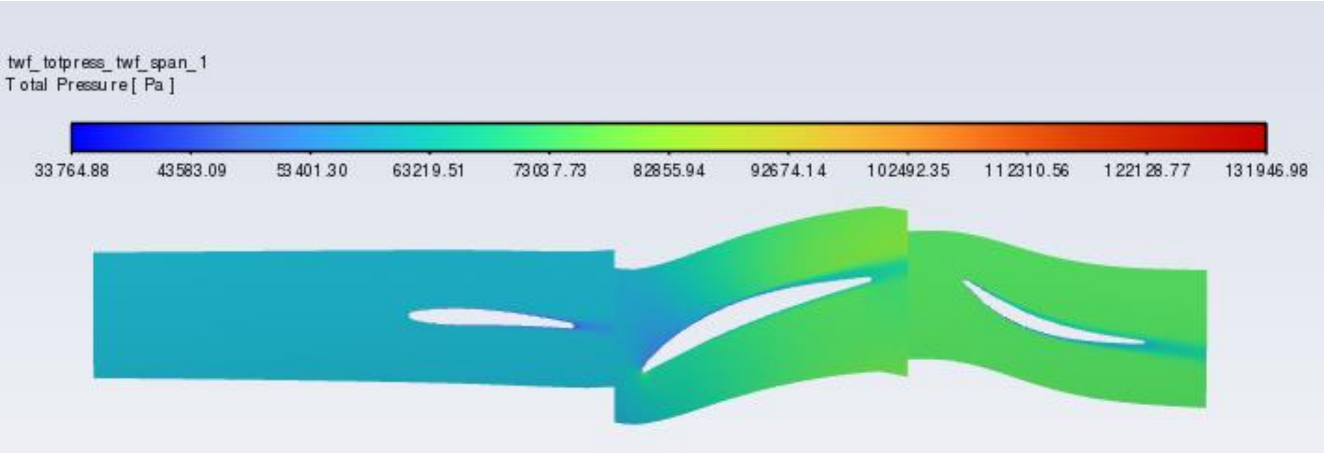
Single Passage

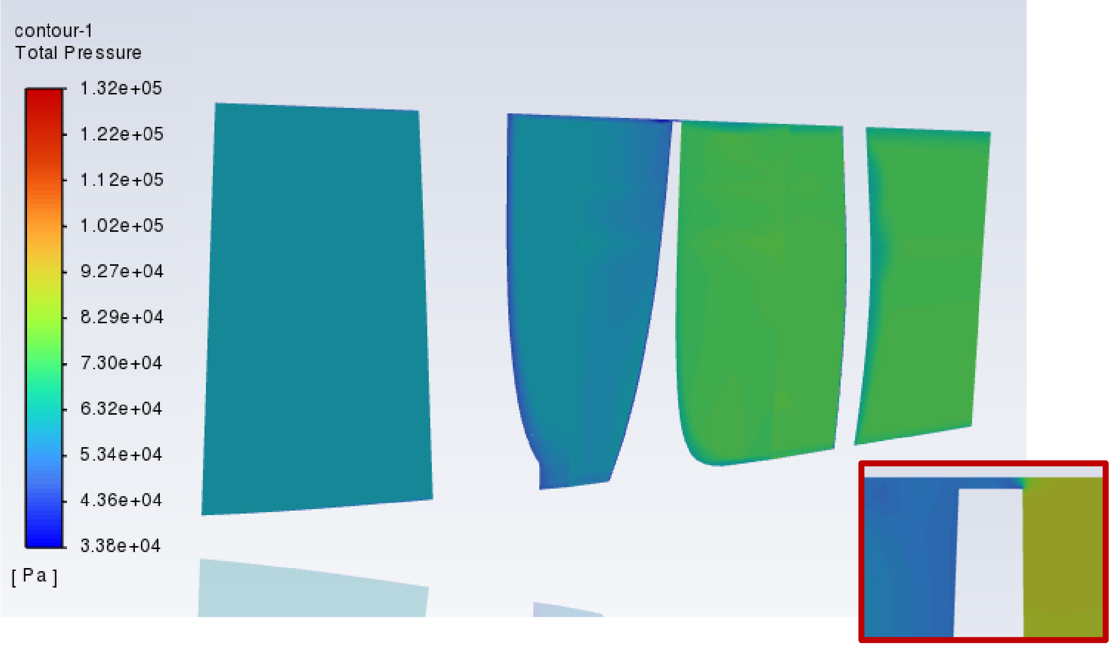


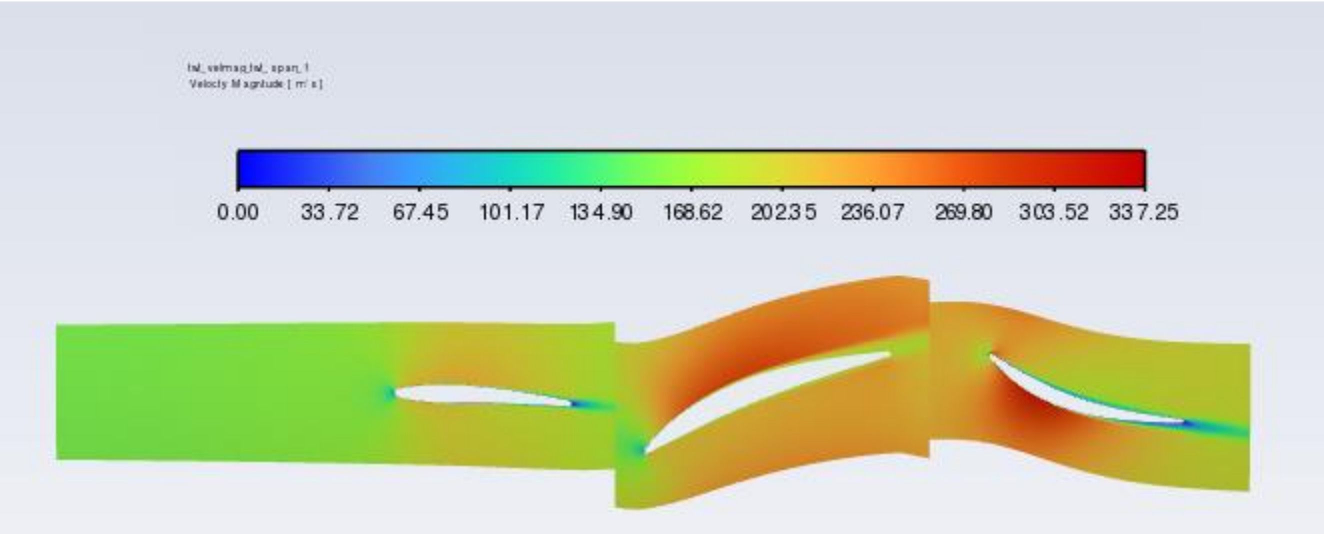
Show Full Model

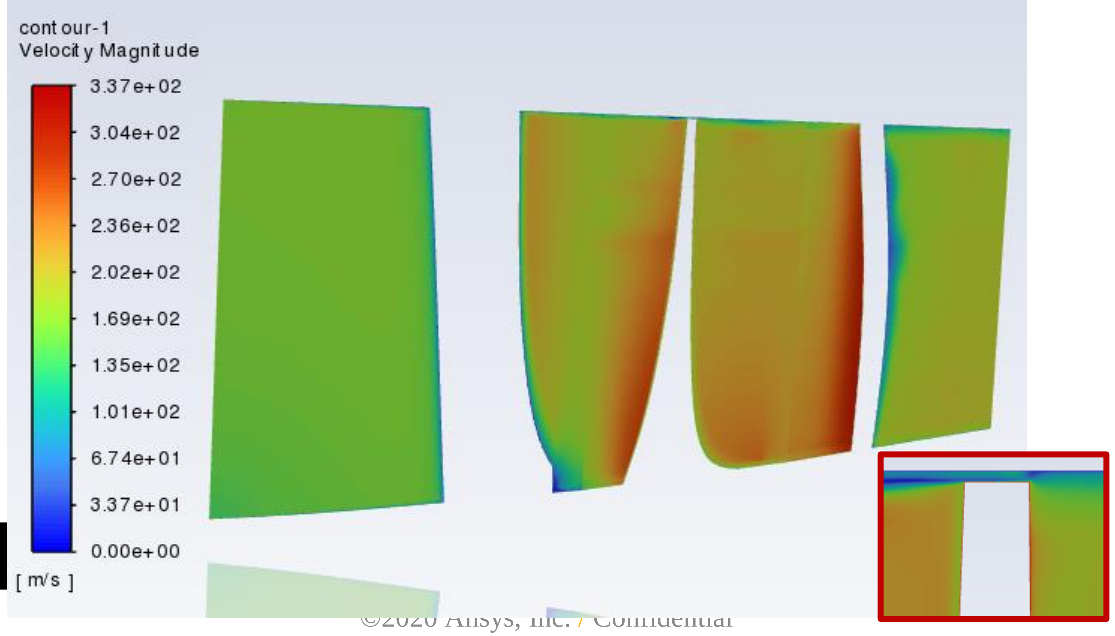


Result


twf_totpress_twf_span_1
Total Pressure [Pa]
33764.88
43583.09
53401.30
63219.51
73037.73
82855.94
92674.14
102492.35
112310.56
122128.77
131946.98


contour-1
Total Pressure
1.32e+05
1.22e+05
1.12e+05
1.02e+05
9.27e+04
8.29e+04
7.30e+04
6.32e+04
5.34e+04
4.36e+04
3.38e+04
[Pa]


0.00
33.72
67.45
101.17
134.90
168.62
202.35
236.07
269.80
303.52
337.25


contour-1
Velocity Magnitude
3.37e+02
3.04e+02
2.70e+02
2.36e+02
2.02e+02
1.69e+02
1.35e+02
1.01e+02
6.74e+01
3.37e+01
0.00e+00
[m/s]

Thanks!!

Question & Answer