ANSYS CFD 2022R1 進階應用

（GPU Solver、Interface、PCB & Thin Gap Model）

虎門科技 CAE 事業部 技術部

工程師 劉怡萱Gina, Mar 2022

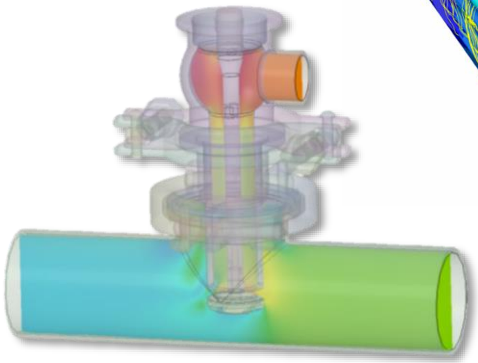
Ansys

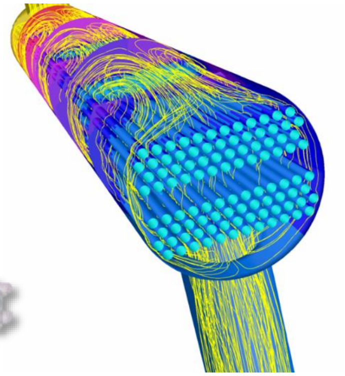
Multi-GPU Solver in Fluent
多張GPU求解器

Fluent Multi-GPU Solver (Beta)

Utilize the power of multiple GPUs to accelerate your CFD simulations

Target applications for first release:
- External aerodynamics
- Internal flows / ducting / ventilation
- Heat transfer





Supported Capabilities:
- Single/multi-GPU (shared / distributed memory)
- Supports all mesh types (poly, hex, tet, pyramid, prism, hanging node)
- Subsonic compressible flows
- Ideal gas
- Material with constant properties
- Turbulence: standard k-epsilon and GEKO k-omega
- Solid conduction and CHT
- Moving walls
- Porous media

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

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

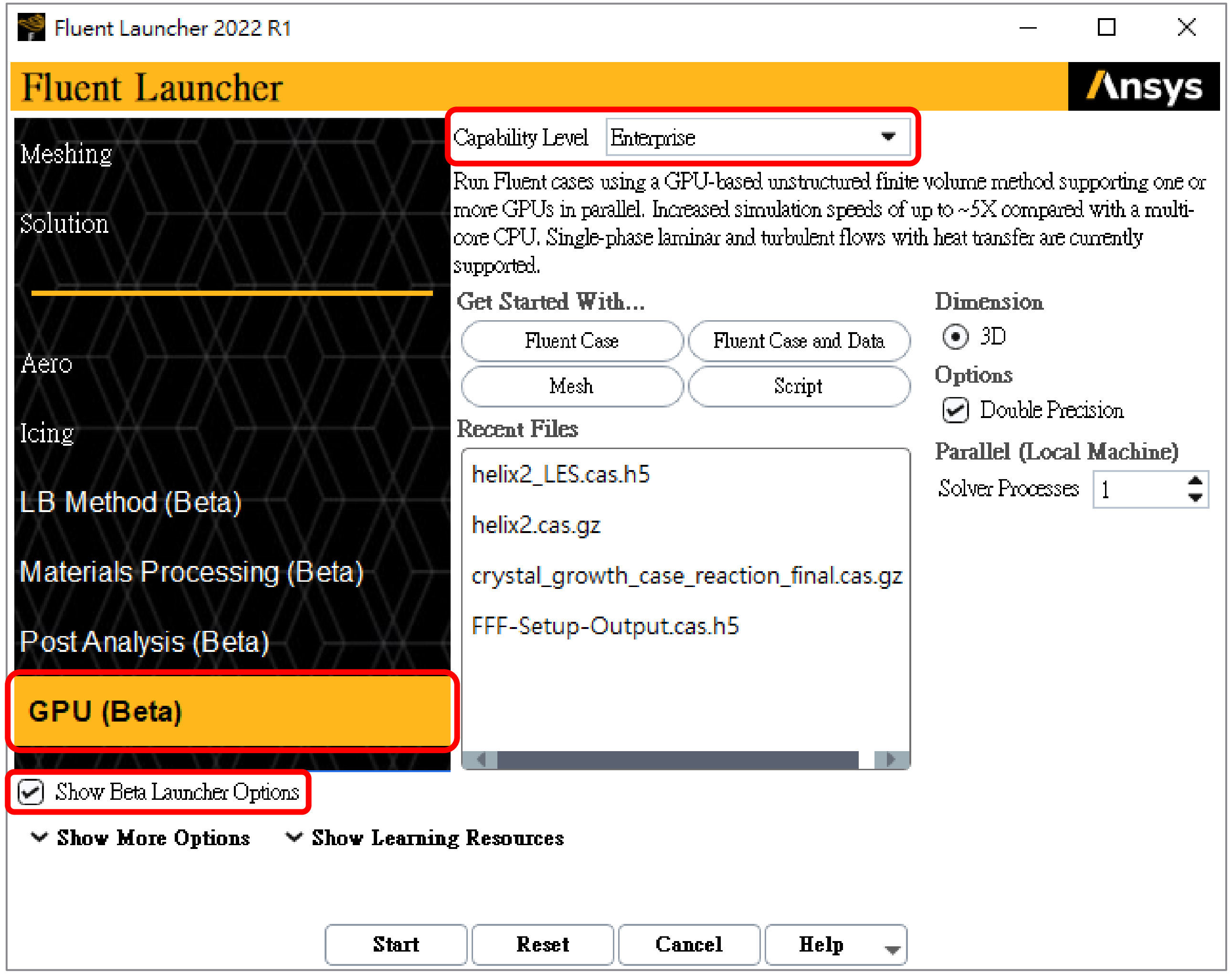
Program Capabilities and Supported GPUs

- The Fluent GPU Solver supports the following NVIDIA GPUs:
  - Quadro P6000
  - Quadro GV100
  - Quadro RTX 6000
  - Tesla P100
  - Tesla V100
  - Tesla A100

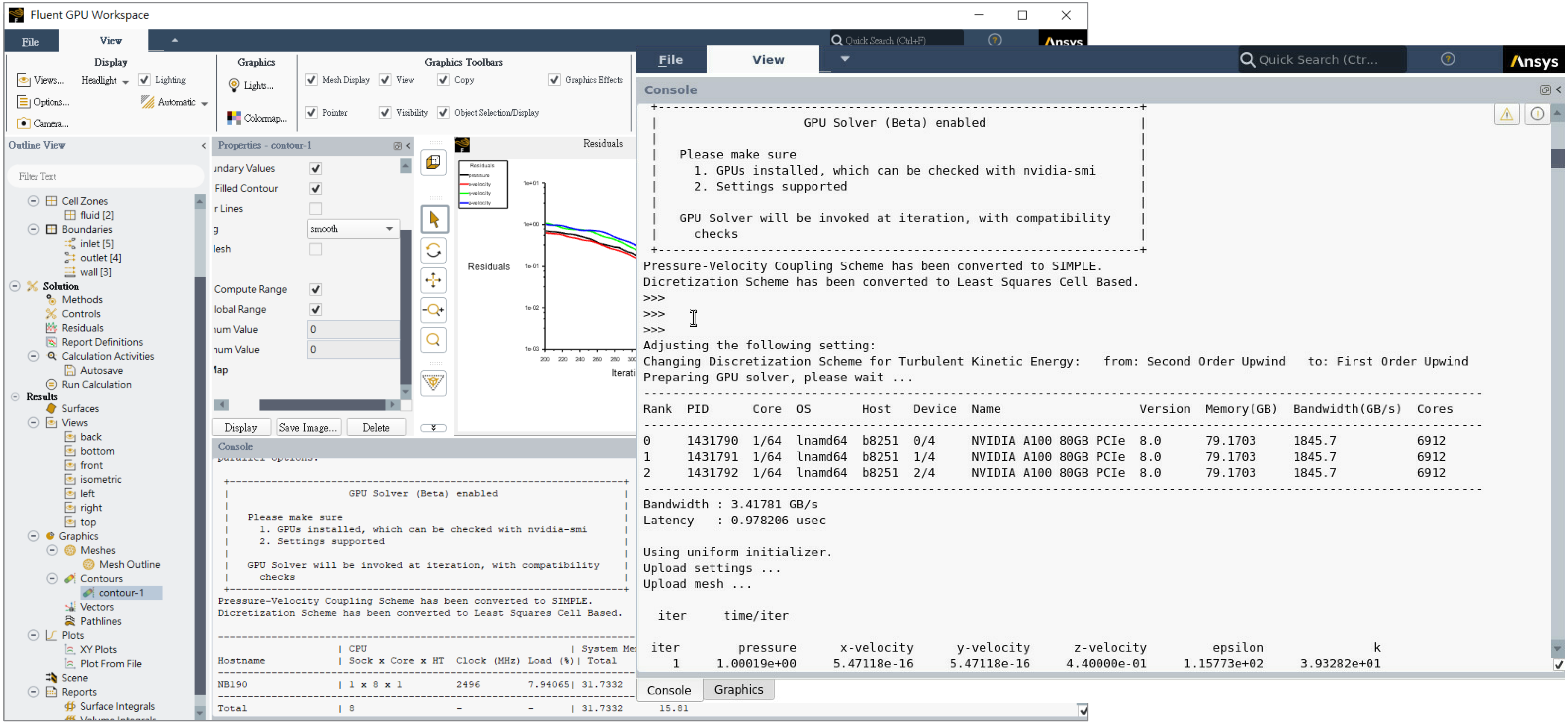
Note that GPUs used by the GPU Solver must be compatible with CUDA version 11.0 or newer

Fluent Multi-GPU Solver (Beta) - GPU Solver Workspace

- To start the Fluent GPU Solver from the Fluent Launcher:
  1. Open the Fluent Launcher.
  2. Select **Enterprise** from the **Capability Level** drop-down list.
  3. Enable **Show Beta Launcher Options**.
  4. Select the **GPU (Beta)** workspace in the Fluent Launcher.

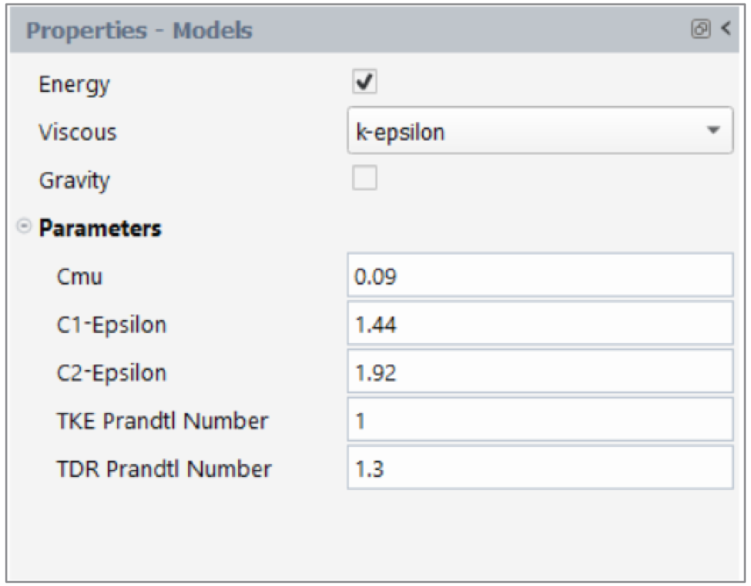


Fluent Multi-GPU Solver (Beta) - GPU Solver Workspace

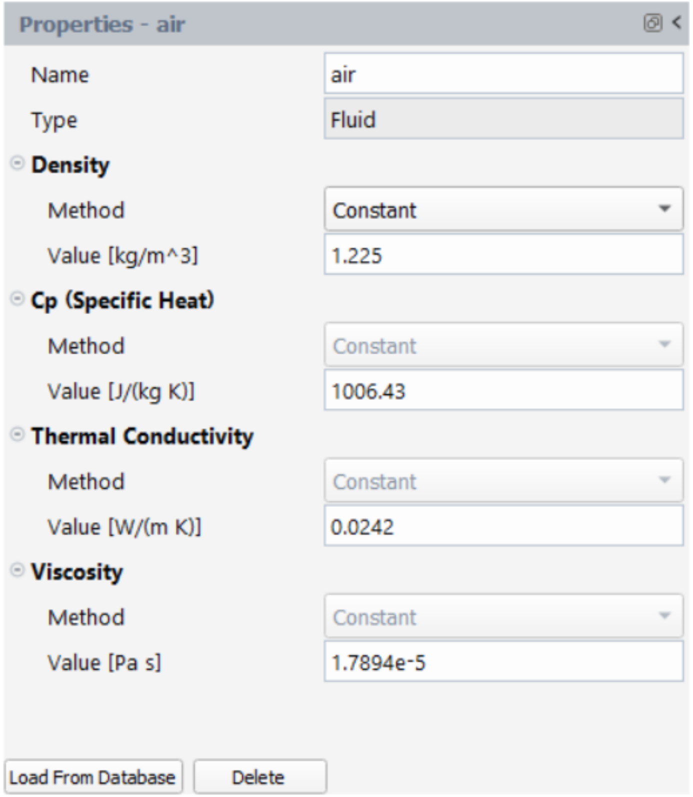


Fluent Multi-GPU Solver (Beta) - GPU Solver Workspace

- You can specify whether the flow is to be treated as laminar or turbulent by selecting from the **Viscous** drop-down list in the **Properties - Models** window. For modeling turbulence, the **k-omega GEKO** or **k-epsilon** turbulence models can be selected.



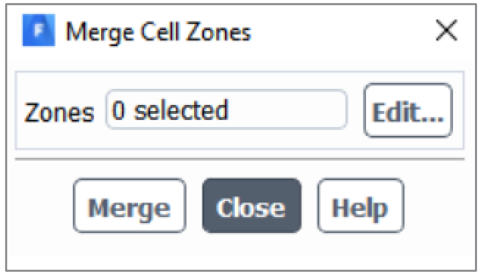
- Open the properties table for the material specified from your imported case by expanding **Materials** in the tree and selecting the material.

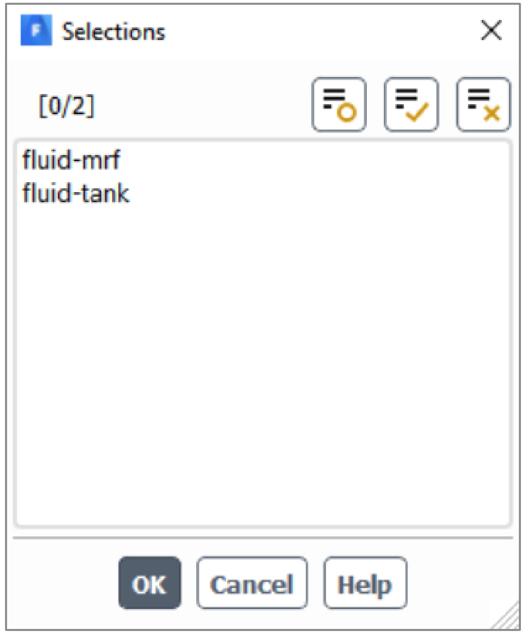


Ansys

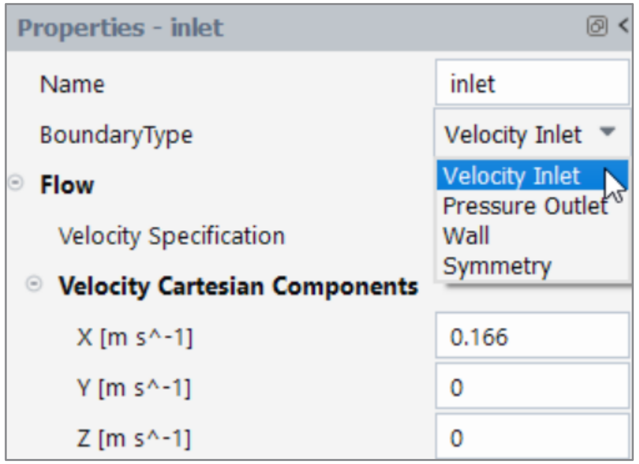
Fluent Multi-GPU Solver (Beta) - GPU Solver Workspace

- Merging and Deleting Cell Zones
  - Although the Fluent GPU Solver only allows for a **single fluid cell zone**, if you import a Fluent case, it may contain more than one fluid cell zone. All of the solid cell zones will automatically be removed and all of the fluid zones will be imported. At this point, you must **merge** or **delete** all of the additional fluid zones, so that you have a single, uniform region.



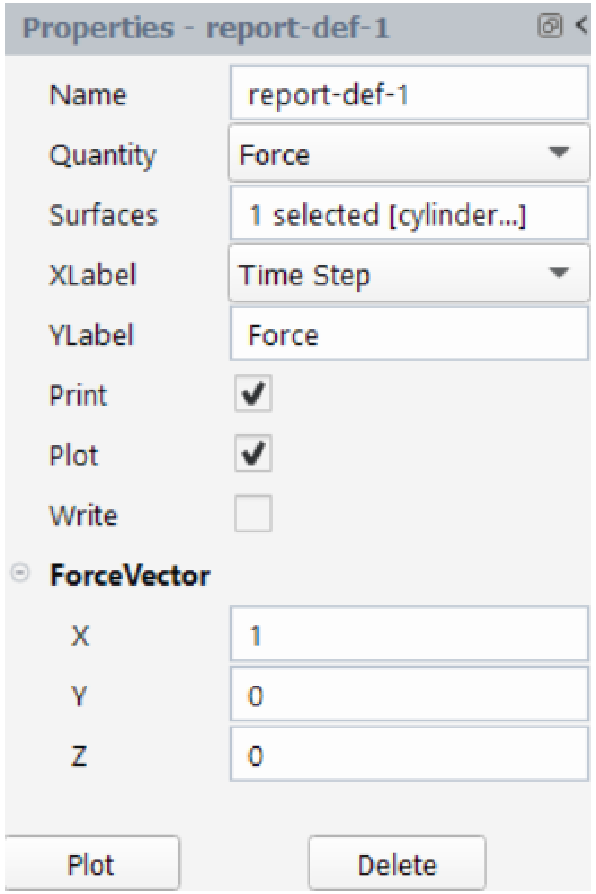


- The following boundary types are available in the Fluent GPU Solver:
  - Velocity inlet (incompressible)
  - Pressure outlet
  - Wall
  - Moving/rotating walls
  - Symmetry
  - Pressure Inlet
  - Mass-Flow Inlet (incompressible)
  - Mass-Flow Outlet (incompressible)

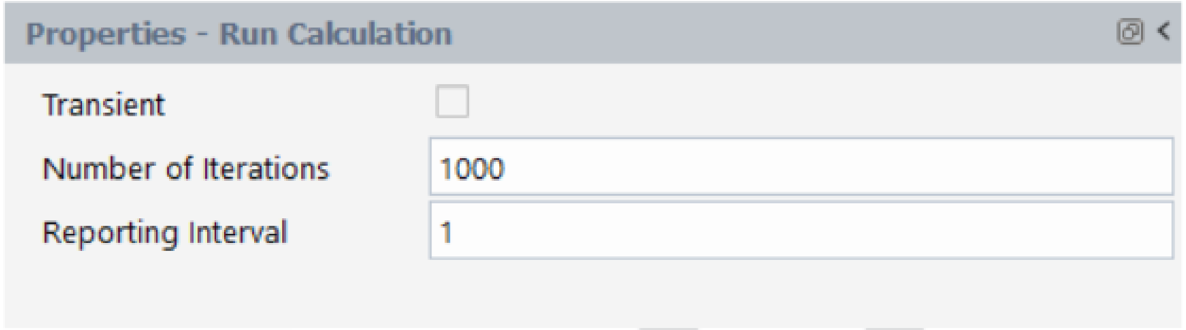


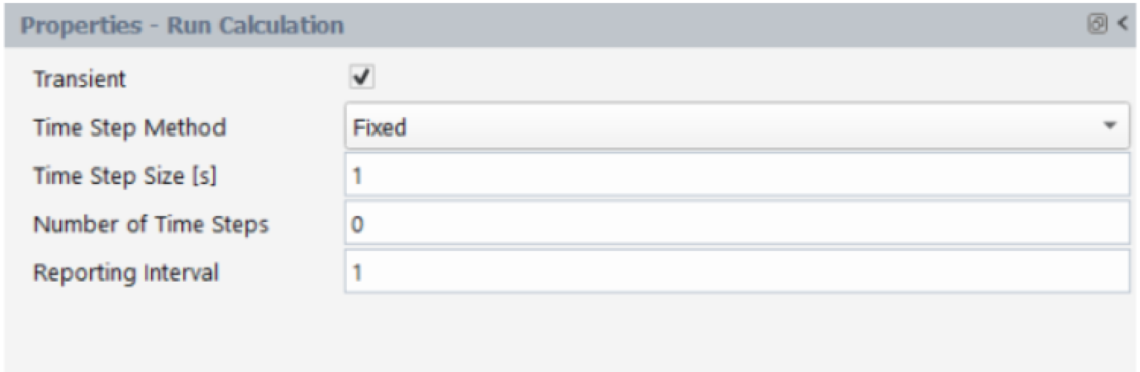
Fluent Multi-GPU Solver (Beta) - GPU Solver Workspace

- The Fluent GPU Solver supports the following Report Definitions:
  - Surface Reports:
    - Mass Flow Rate…
  - Force Reports:
    - Lift…
    - Drag…
  - Flux Reports:
    - Total Heat Transfer Rate…
    - Radiation Heat Transfer Rate…
    - Mass Flow Rate…



- After initializing and setting up your calculation activities, you can specify the settings for running the calculation.





Fluent Multi-GPU Solver (Beta) - in Fluent Solution Mode

Important: Using the Fluent GPU Solver Workspace is not recommended and should only be used as a reference for features supported by the GPU Solver. All users are encouraged to use the Fluent GPU Solver within ***Fluent solution mode****.*

- For Windows, executing the fluent.exe program located in the Ansys Fluent directory (for example, **C:\Program Files\ANSYS Inc\v221\fluent\ntbin\win64**).
- Once your working directory path is specified, you can execute the ***nvidia-smi*** command in the command prompt window. This will display the GPUs available on your machine as well as their current usage. Each GPU on your machine will have an ID of 0 to 7 after executing nvidia-smi.

```
C:\Program Files\ANSYS Inc\v221\fluent\ntbin\win64> nvidia-smi
+-----------------------------------------------------------------------------+
| NVIDIA-SMI 496.13       Driver Version: 496.13       CUDA Version: 11.5     |
|-------------------------------+----------------------+----------------------+
| GPU  Name            TCC/WDDM | Bus-Id        Disp.A | Volatile Uncorr. ECC |
| Fan  Temp  Perf  Pwr:Usage/Cap|         Memory-Usage | GPU-Util  Compute M. |
|                               |                      |               MIG M. |
|===============================+======================+======================|
|   0  Quadro P4000       WDDM  | 00000000:D5:00.0  On |                  N/A |
| 46%   35C    P8    11W / 105W |   3463MiB /  8192MiB |      1%      Default |
+-------------------------------+----------------------+----------------------+

+-----------------------------------------------------------------------------+
| Processes:                                                                  |
|  GPU   GI   CI        PID   Type   Process name                  GPU Memory |
|        ID   ID                                                   Usage      |
|=============================================================================|
|  No running processes found                                                 |
+-----------------------------------------------------------------------------+
```

- fluent 3d -t1 -gpuapp
- fluent 3d -t2 -gpuapp=1,2
- fluent 3d -t3 -gpuapp=1,2,4

Multi-GPU Licensing

- A CFD Enterprise license is required to use the solver within Fluent
- Running on a single GPU is included
  - I.e., no additional HPC licenses are required
- Each subsequent GPU requires 64 HPC tasks
  - 4 HPC tasks are included with CFD Enterprise, so 60 additional HPC tasks are required to enable the 2nd GPU, then 64 further HPC tasks for the 3rd GPU, etc.

# GPUs	Number of HPC Licenses	
	Workgroup tasks	HPC Packs, 2022R1
1	0	0
2	60	3
3	124	3
4	188	4
5	252	4
6	316	4
7	380	4
8	444	4

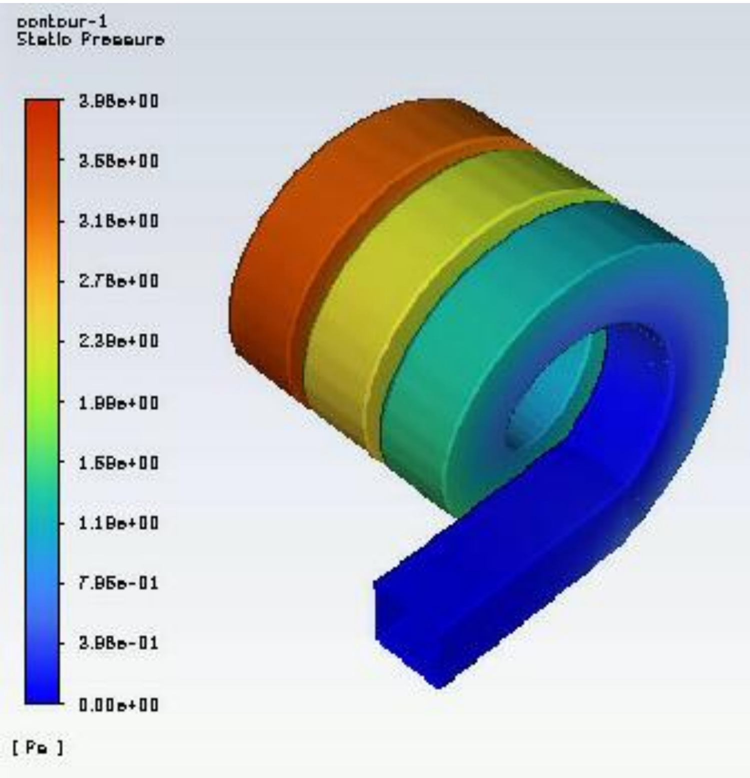
Ansys

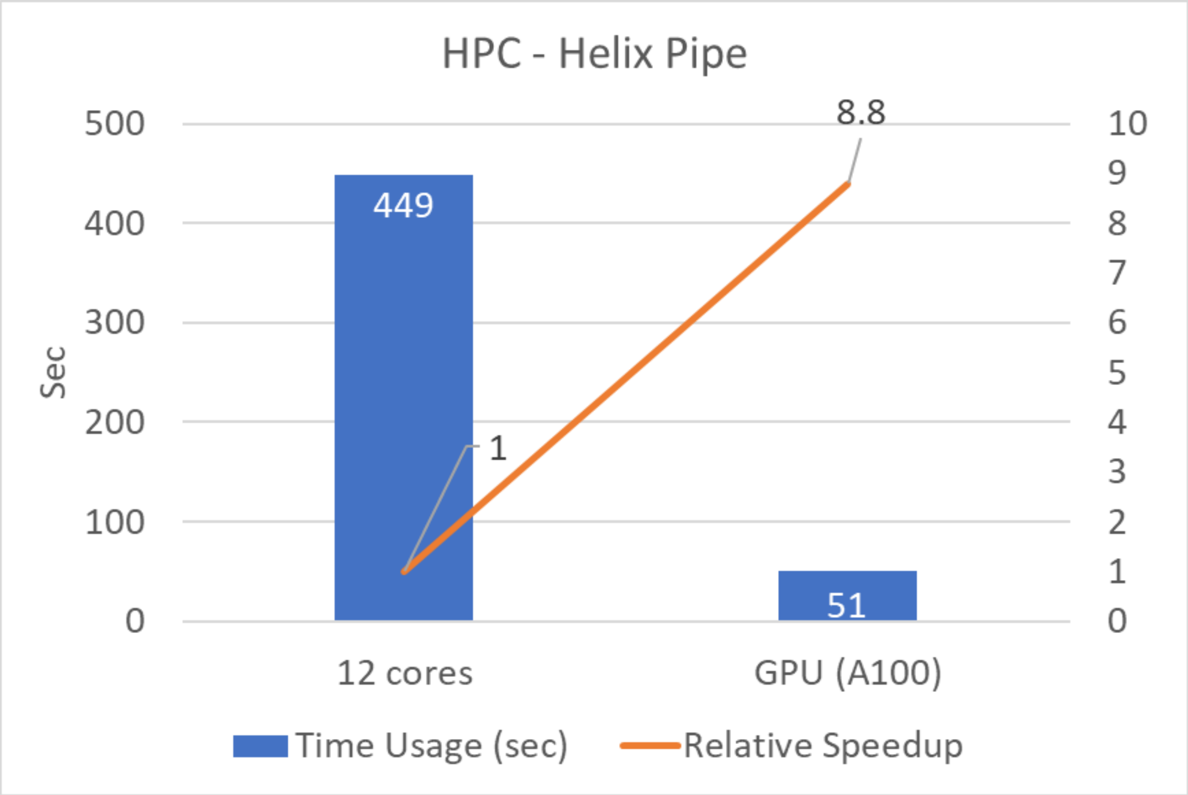
Fluent Multi-GPU Solver (Beta) - Performance

Helix Pipe

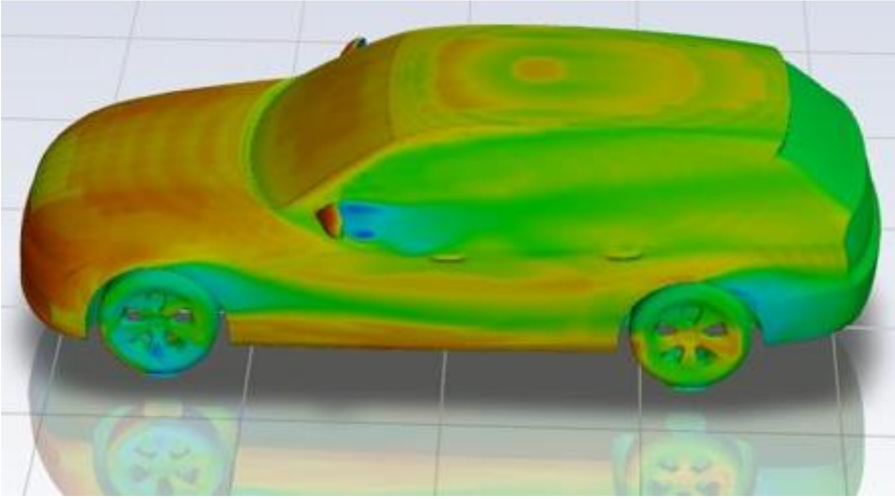
- Mesh Type: hexcore
- Mesh Size: 208,000

- Steady
- K-epsilon Standard
- Inlet: 1 m/s
- Number of Iterations: 10000



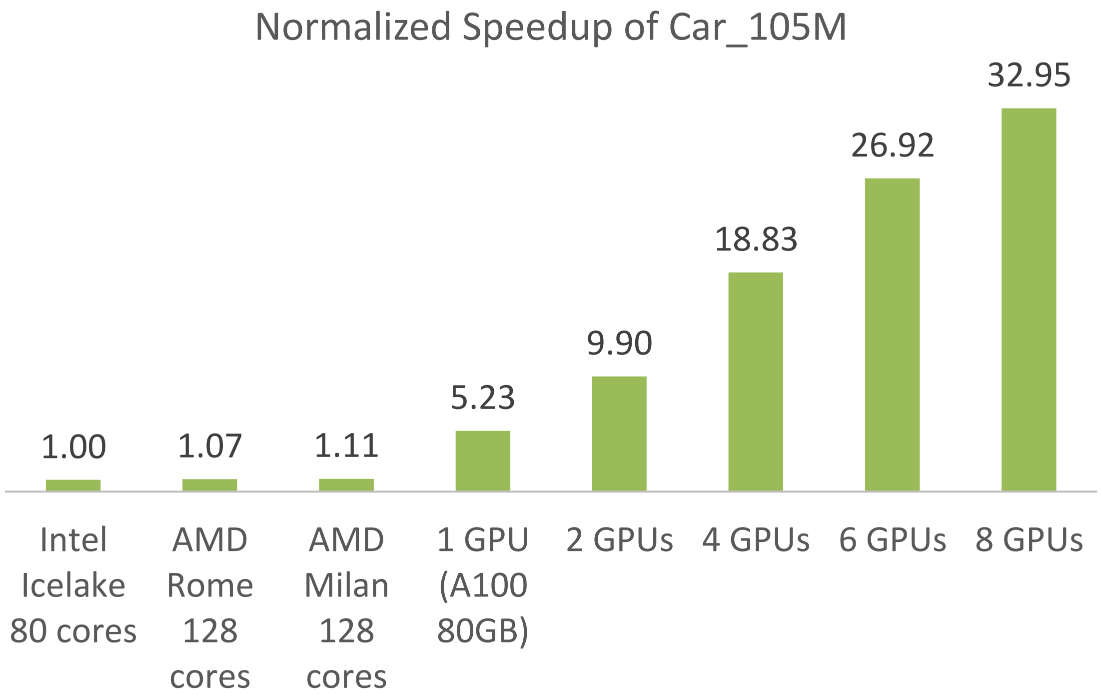


Fluent Multi-GPU Solver (Beta) cont.



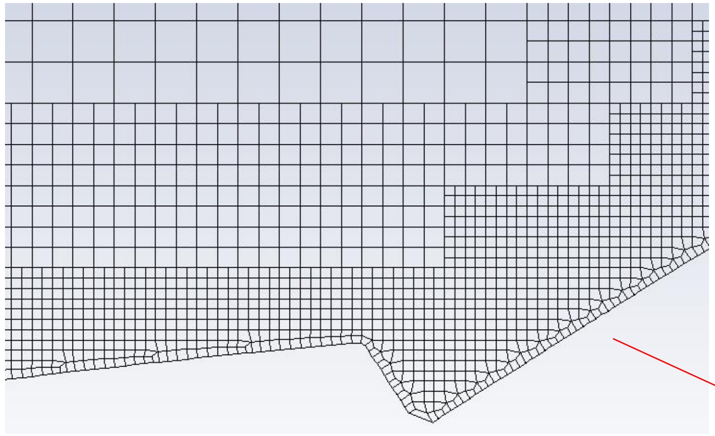
Strong scaling with parallel efficiency of 80% on automotive external aerodynamics case

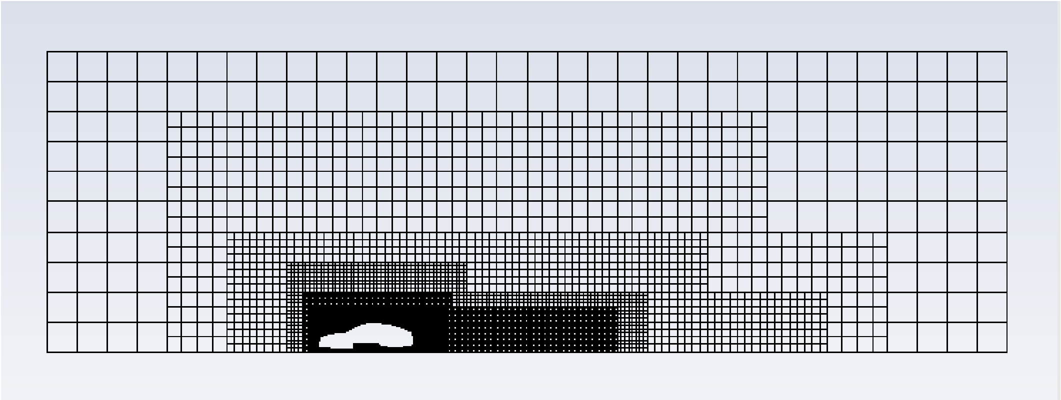
- 105 million cell case, single precision with GEKO, runs on 1 A100 80GB card
  - ~1 iteration per 2 seconds, 20 minutes to converge such a case from initialization
- 1 A100 GPU ≈ 640 AMD Milan cores on 5 nodes
- 8 A100 GPUs ≈ 3840 AMD Milan cores on 32 nodes
- Parallel efficiency is 80% from 1 to 8 GPUs

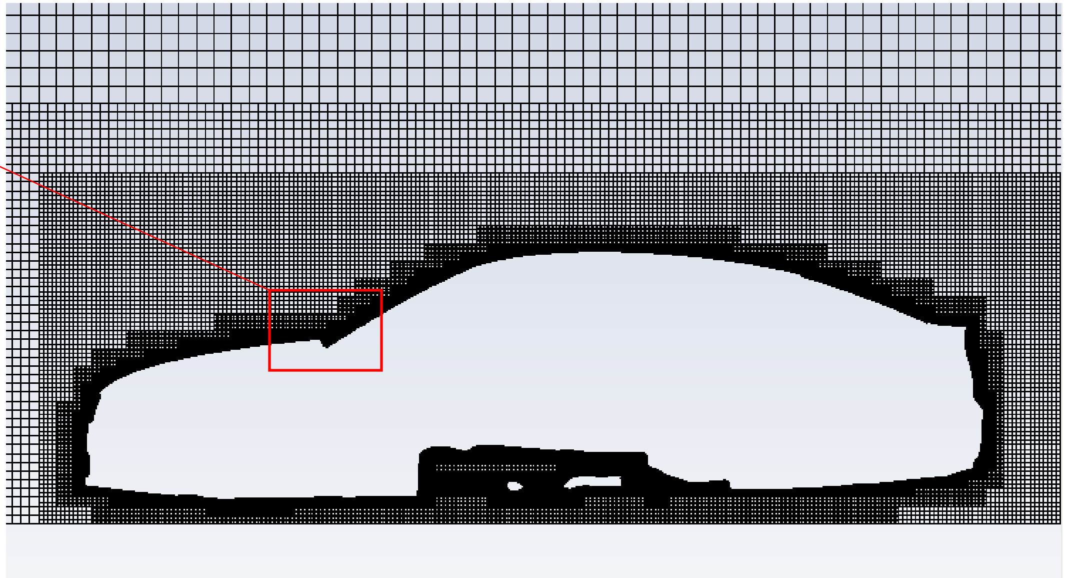


DrivAER Car - Rapid Octree Mesh

- Sizing:
  - Background ~1.4m
  - Body ~5.6mm
  - Levels = 9
  - Prisms = 1 layer
- BOIs
  - Rectangular around body and in wake
- Mesh size
  - 19.43M

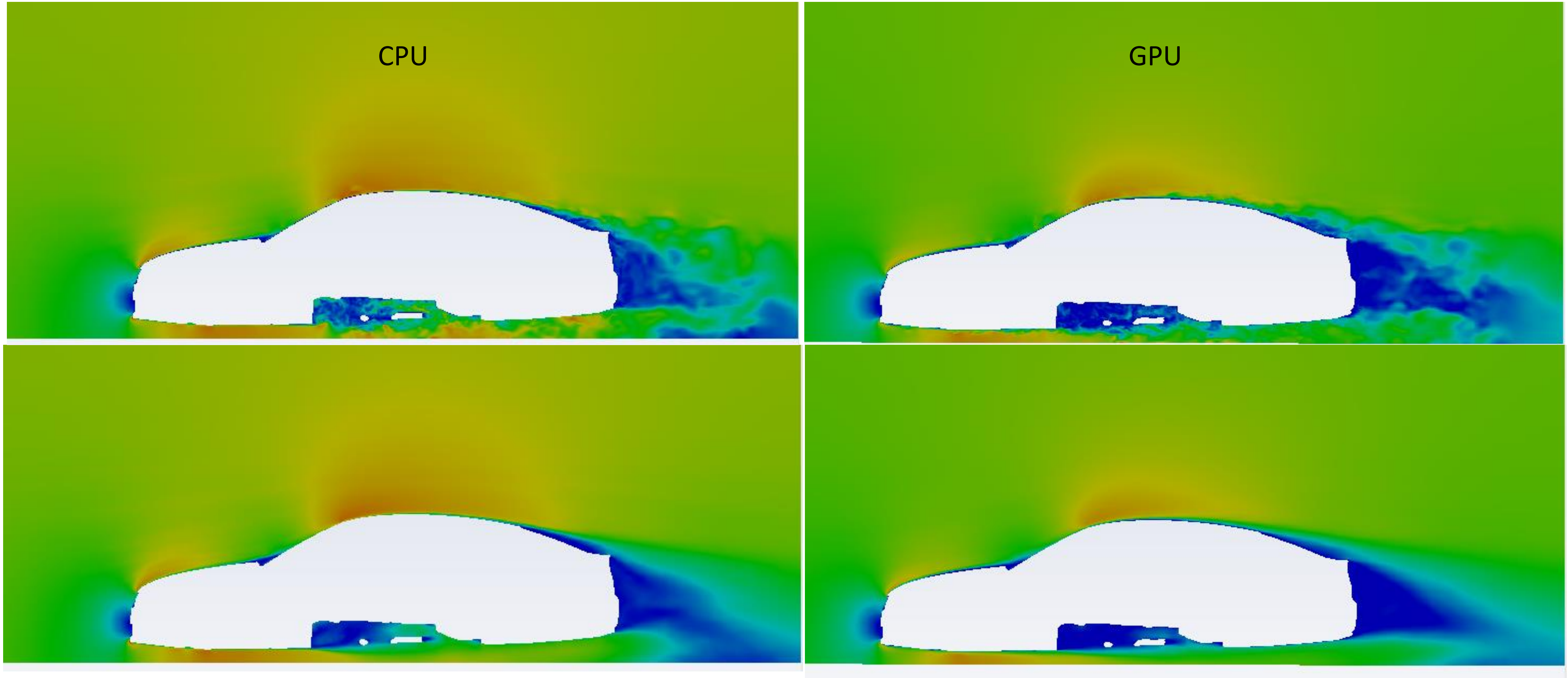






DrivAER Car - CPU/GPU

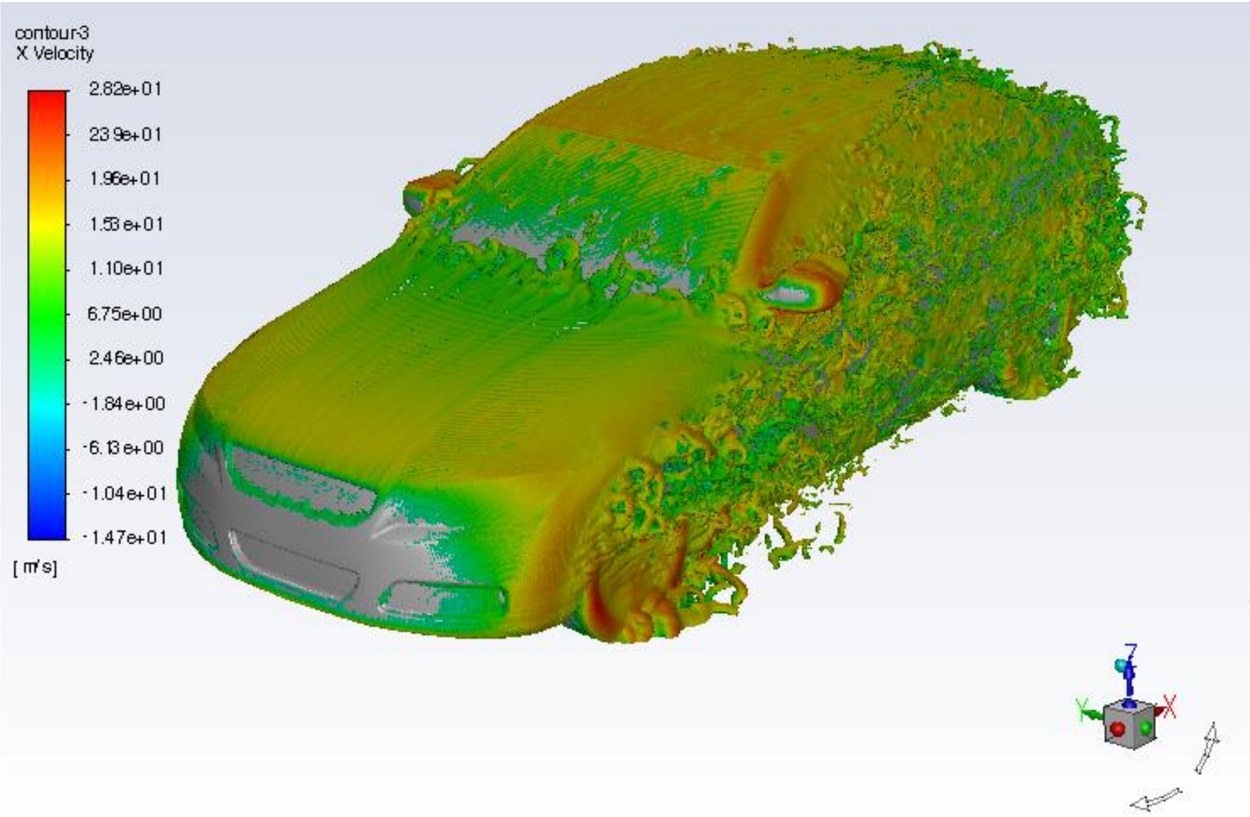
Velocity Magnitude (0-25 m/s)



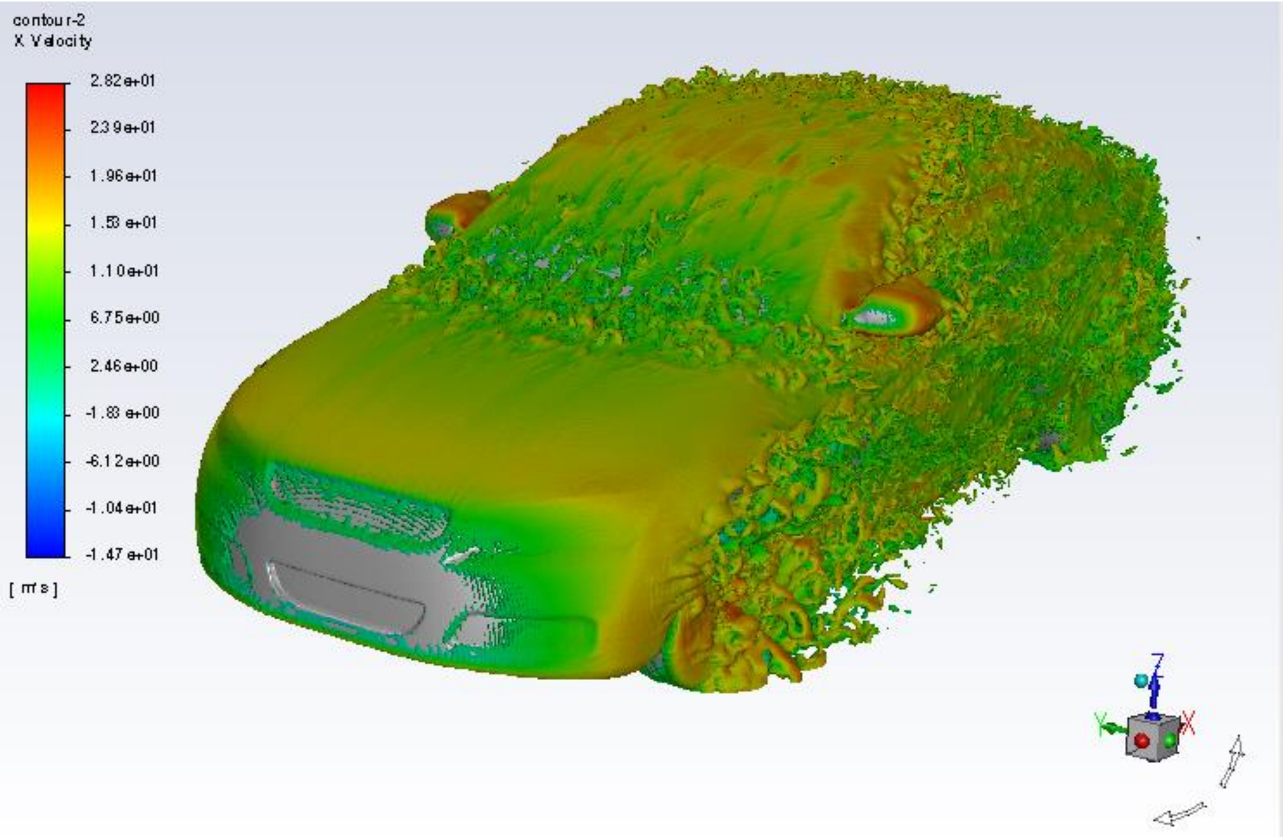
DrivAER Car - CPU/GPU

Iso. Vort. Mag = 500 [s^-1]

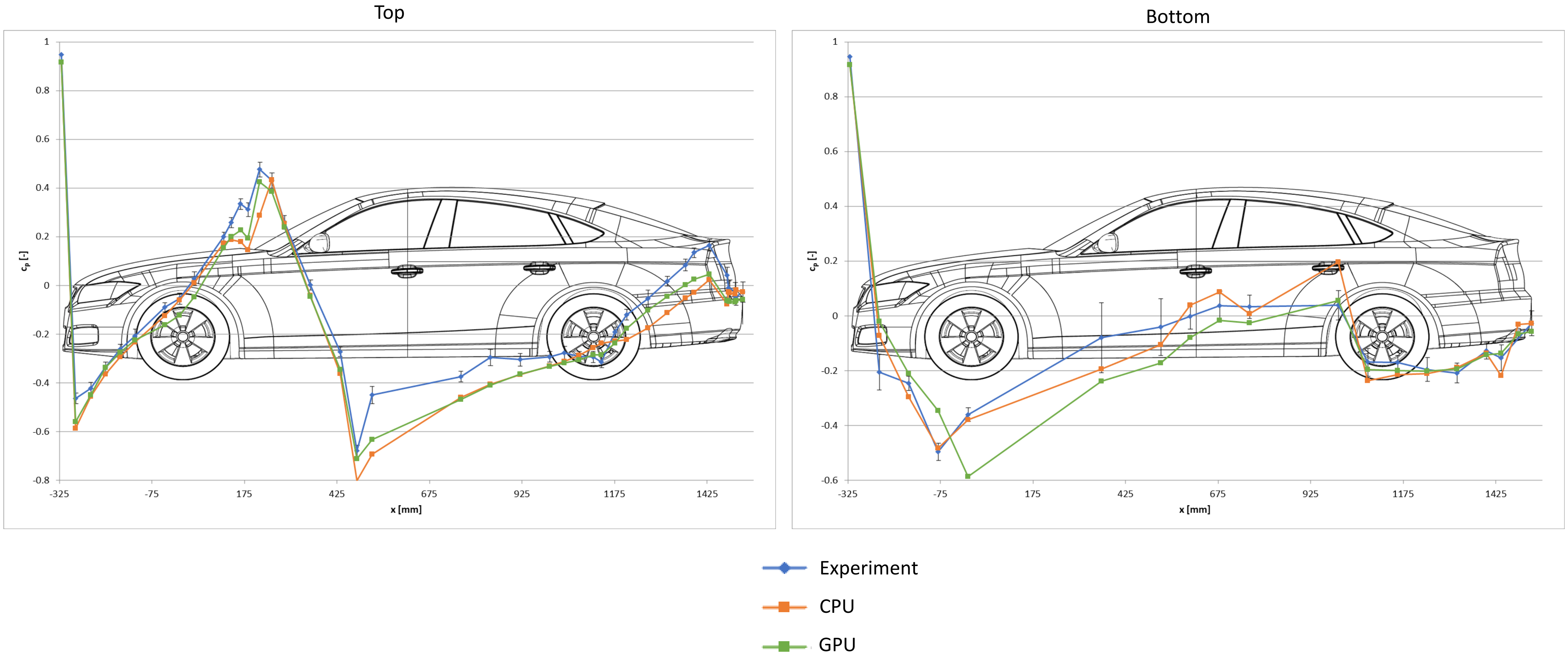
CPU



GPU



DrivAER Car - CPU/GPU – Mean surface Cp values

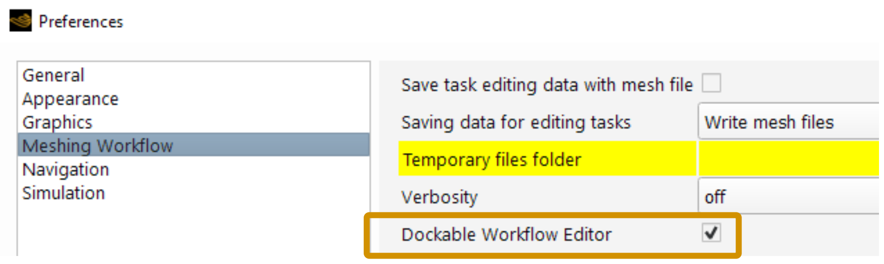


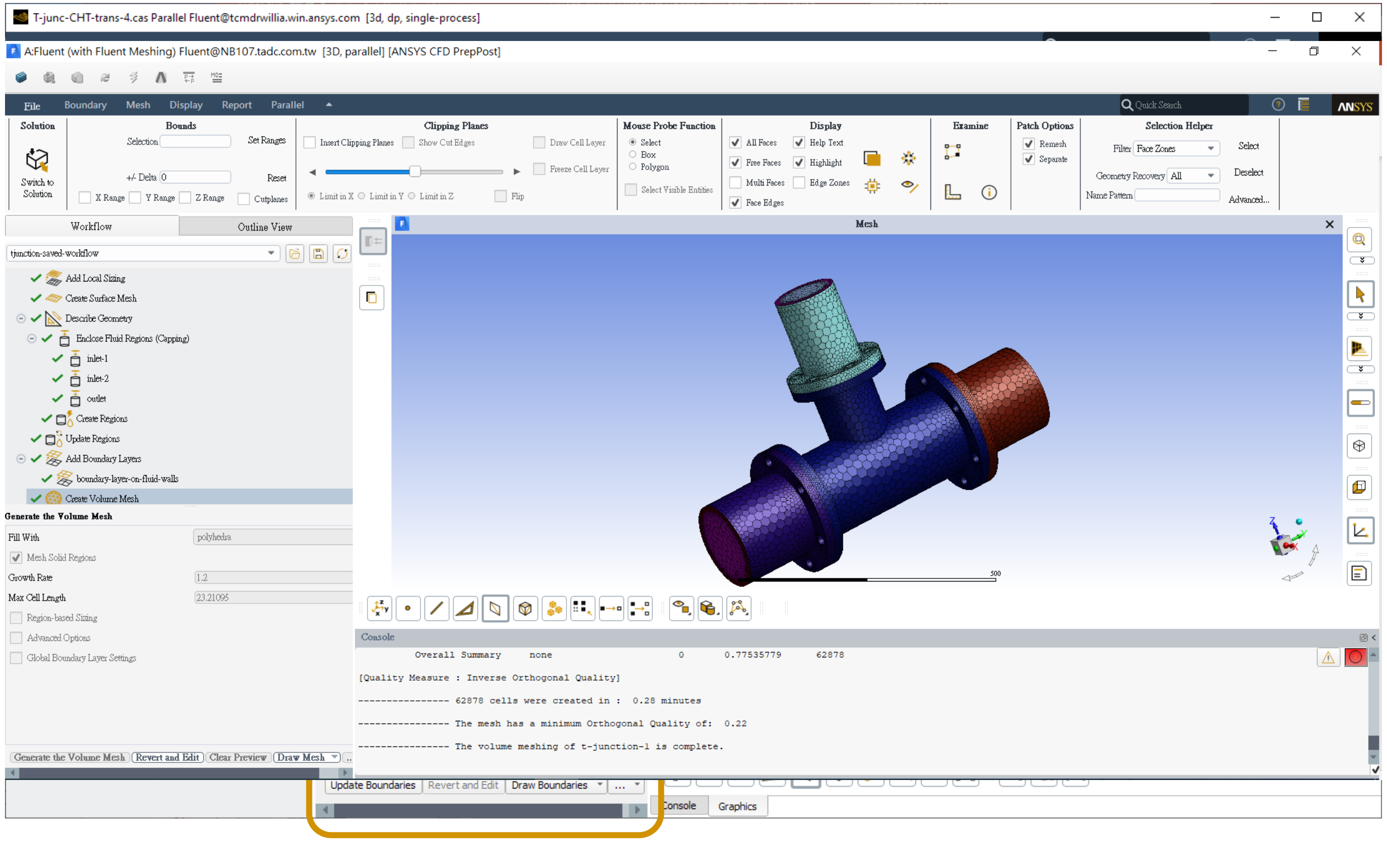
Fluent Meshing Workflows
流體網格處理流程

Dockable Workflow Editor

2021R2

- Optionally separate workflow task editor from task list
- More space to work with in task editing for cases with many regions, etc.
- Enable through Preferences (requires restart)





Ansys

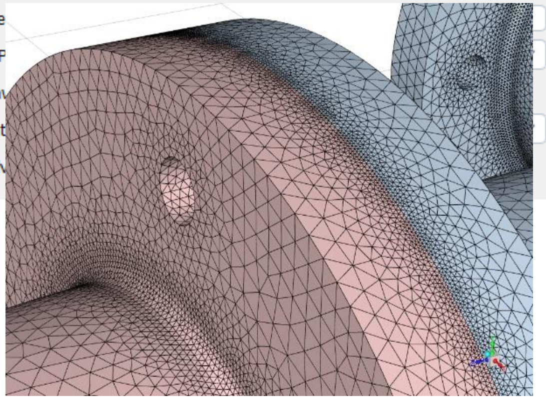
Watertight Meshing – Sizing Functions Improvements

- Ignore Proximity Across Objects
  - Only visible if assemblies are imported without full ST in SCDM/DM
  - Avoid over-refinement when Share Topology in workflow or NCI

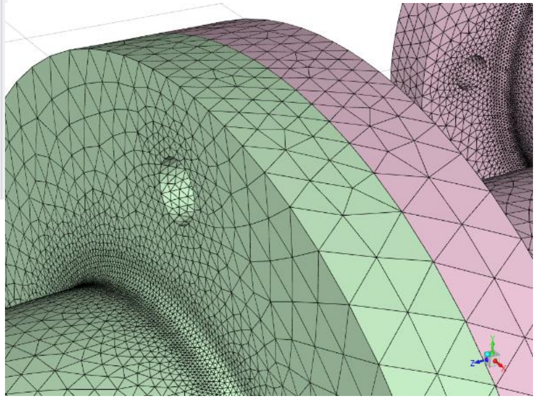
Workflow Editor

Generate the Surface Mesh

Use Custom Size Field/Control Files?	No
Minimum Size [mm]	0.2871093
Maximum Size [mm]	7.349989
Growth Rate	1.2
Size Functions	Curvature & Proximity
Ignore Proximity Across Objects?	Yes
Curvature Normal Angle [deg]	18
Cells Pe	
Scope F	
☑ Draw	
Separat	
+ Adv	



No (default)



Yes

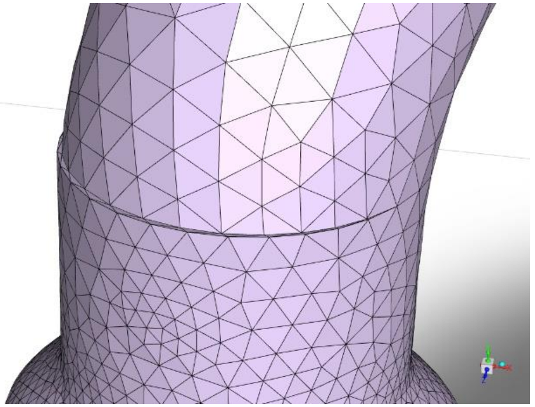
- Ability to remove thin geometry details
  - Globally only
- Improves quality
- Available at 2 places
  - In Surface Mesh task

− Advanced Options	
Check Self-Intersection?	yes
Smooth Folded Faces/Repair Free Nodes Limit	10
Auto Assign Zone Types?	yes
Invoke Quality Improve?	no
Remove Steps?	yes
Max Step Width [mm]	0.7

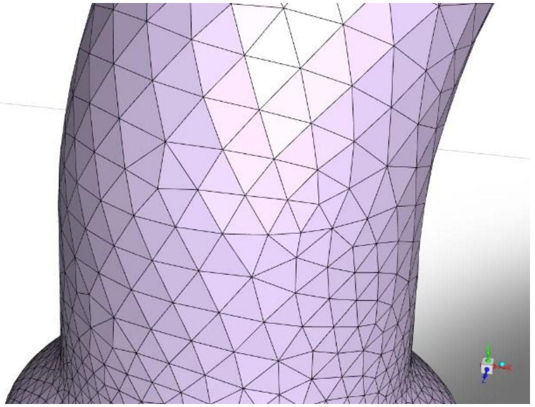
  - In added Surface Mesh Improve task

Improve Surface Mesh

Face Quality Limit	0.7
− Advanced Options	
Quality Improve Max Angle [deg]	160
Quality Improve Iterations	5
Quality Improve Collapse Skewness Limit	0.85
Remove Steps?	yes
Max Step Width [mm]	0.7
Step Skewness Quality Limit	0.6



No Improve , Max skew = 0.97



***Improve** only, Max skew = 0.92*

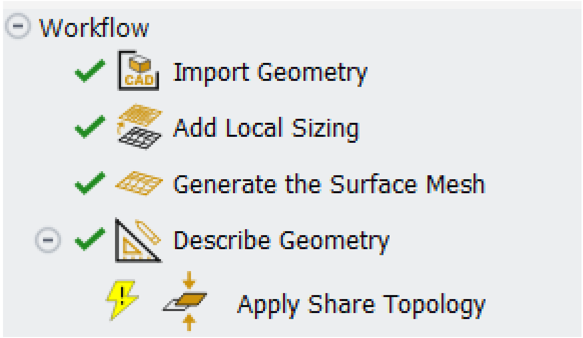
***Improve** + Step, Max skew = 0.44*

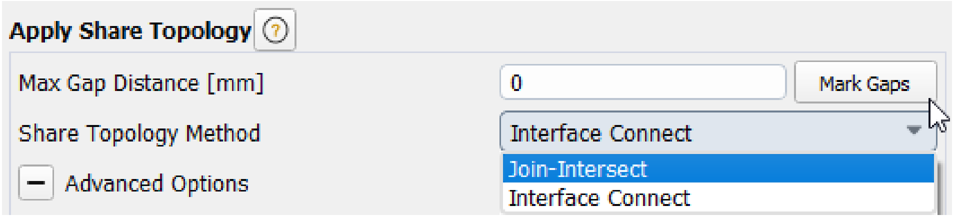
Ansys

New Share Topology Method in Watertight Workflow

Offers potentially faster and more robust method than Join-Intersect

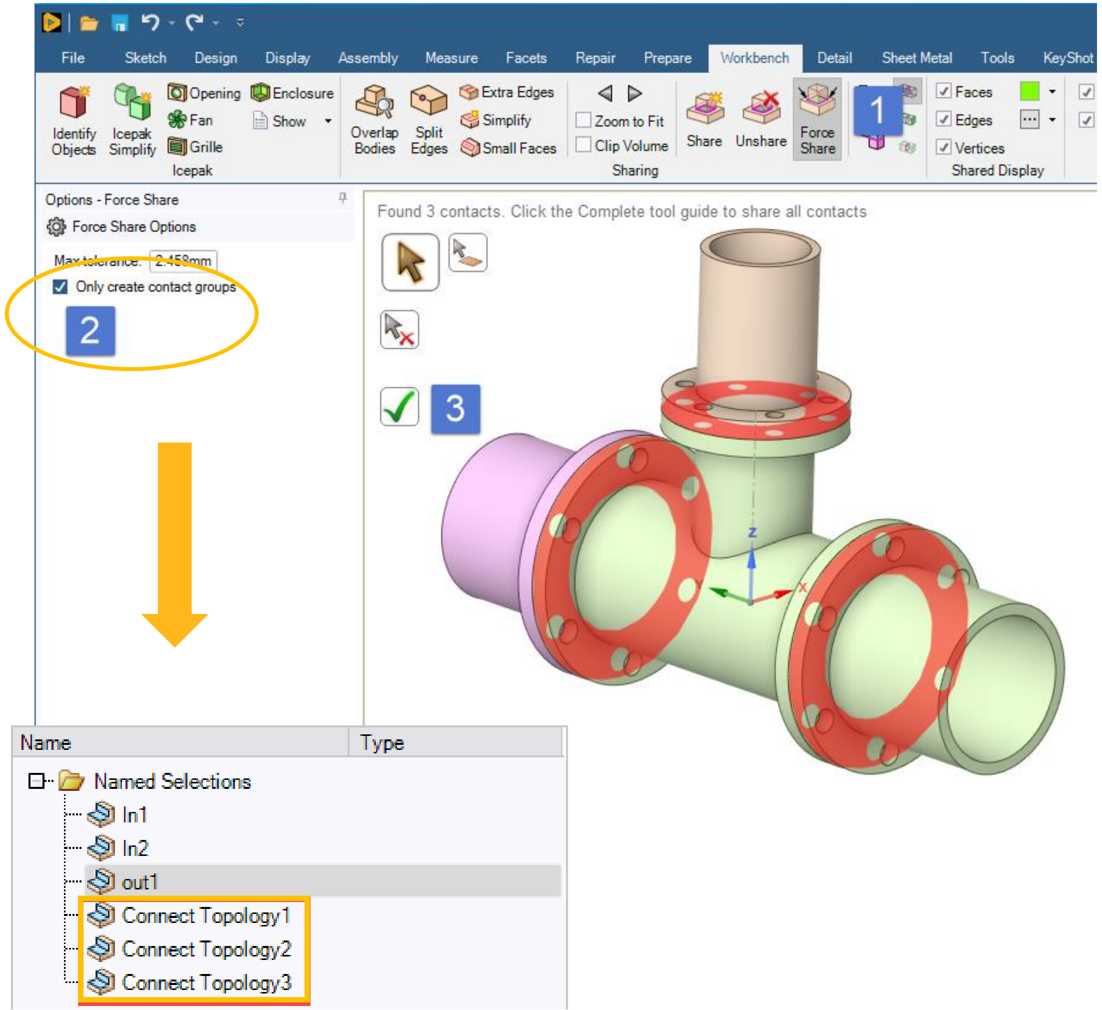
- New **Interface Connect** method for **Apply Share Topology** task
  - Connects edges of overlapping face pairs (rather than intersecting faces like **Join-Intersect**) which can be faster / more robust
  - Three modes:
    - **Automatic - Using Connect Topology:** use the labels created by SpaceClaim if the Force Share feature was used.
    - **Manual:** user manually selects the interface labels from the list of available labels.
    - **Automatic:** automatically separate face zones, identify overlapping faces, and assign the interface connect labels. Useful when connect topology has not been utilized in SpaceClaim or if the mesh was obtained from another source.

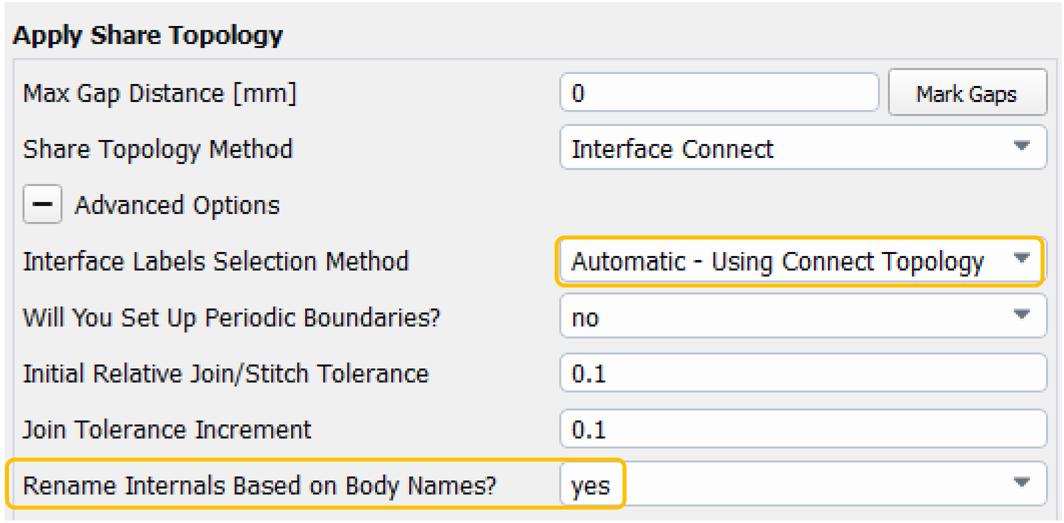




Note : "Ignore proximity across objects" option strongly recommended

WTM – Interface Connect – Automatic Using Connect Topology

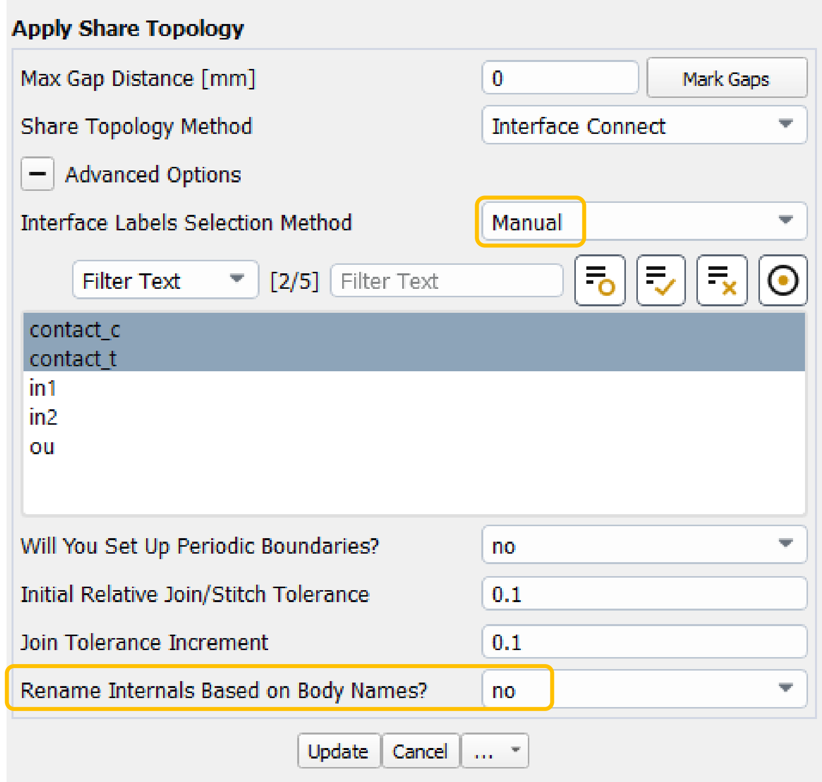




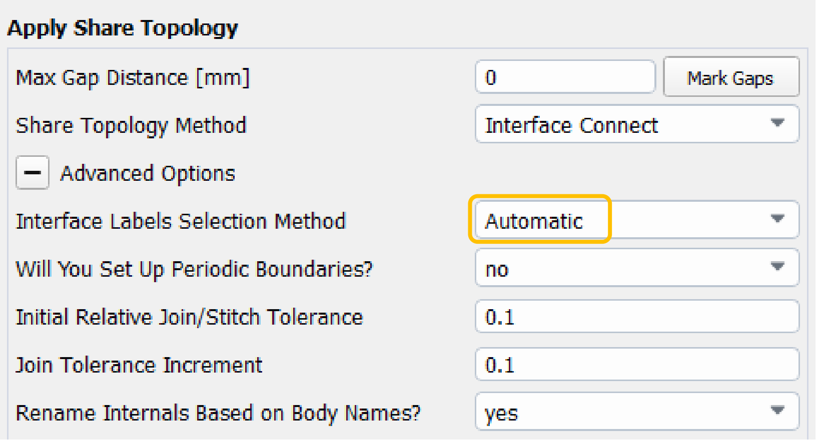
- SpaceClaim
  - Define contact groups with Force Share
- WTM
  - “Automatic – Using Connect Topology” is automatically set
  - No label selection needed
  - « Connect topology » contacts renamed by default using adjacent body names

WTM – Interface Connect – Manual & Automatic

- Manual
  - Fastest option
  - Contact surfaces must be defined in the CAD
  - Switch off Rename to keep given names



- Automatic
  - Virtually no speedup compared to normal share topology
  - Zones are separated before contact check
    - Not merged back for now
  - Identification of overlapping zones and assign labels, then execute



Fault-Tolerant Meshing Part Replacement

Efficiently add, remove or replace geometry objects of your CAD model without having to re-mesh the complete model

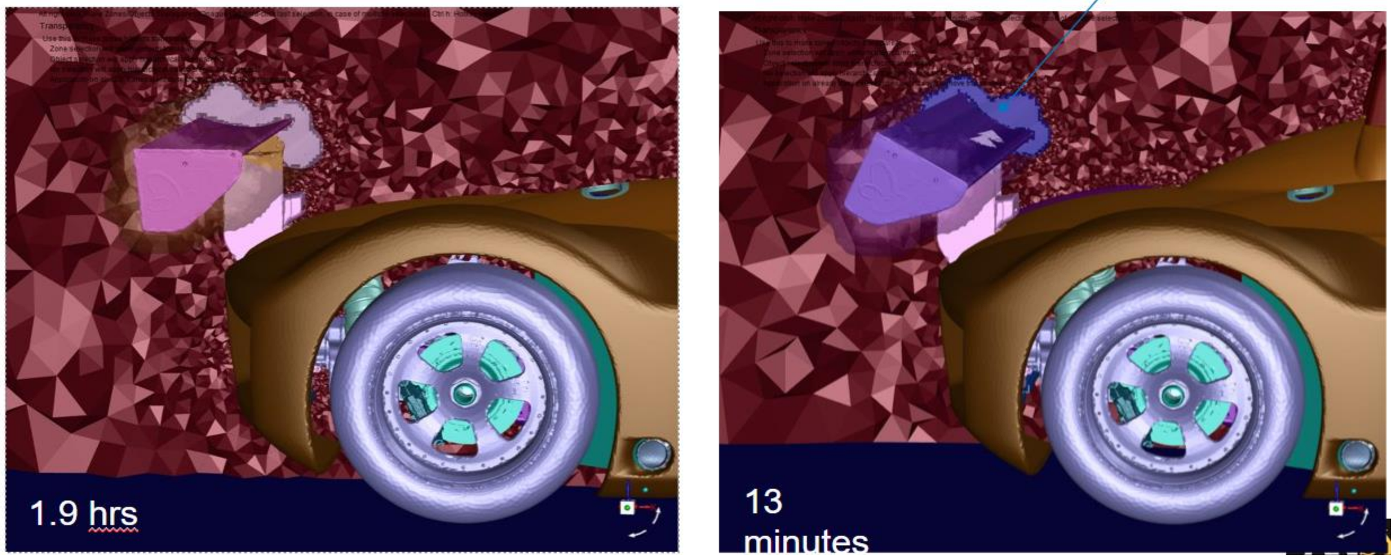
- Choose Part Replacement Options
  - Append CAD and Part Management
  - Part Replacement Settings
  - Update the Volume Mesh

- Users can replace/add/remove parts of CAD model
- Two approaches are implemented which reduces the time to study the design variation by many times
  - Volume-mesh level
  - Surface-mesh level
- Supports all mesh types

Part Replacement Settings

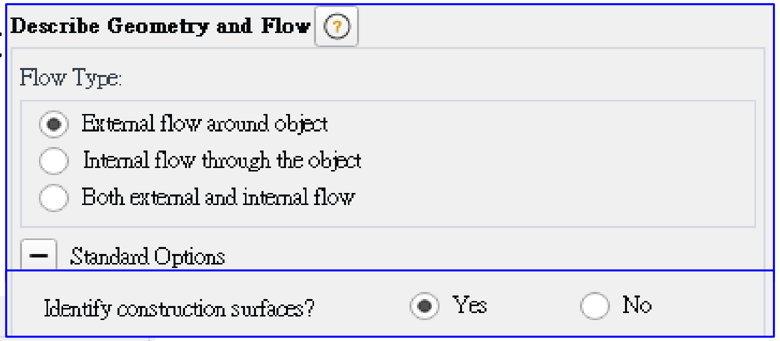
Name	replacing-mirror
Operation	Replacement
Method	Volume Mesh Based

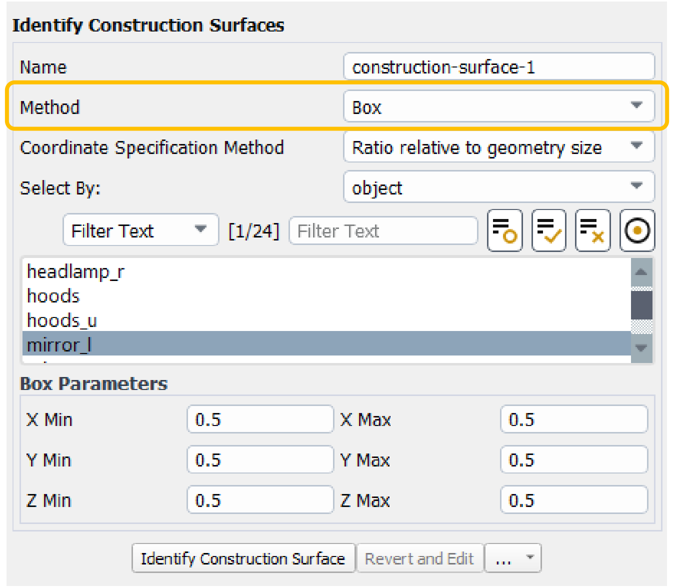
Volume Mesh Level: PTC World car, Replacing the rear spoiler

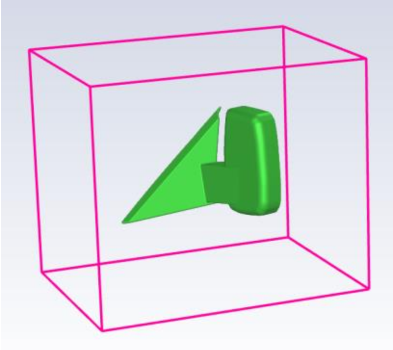


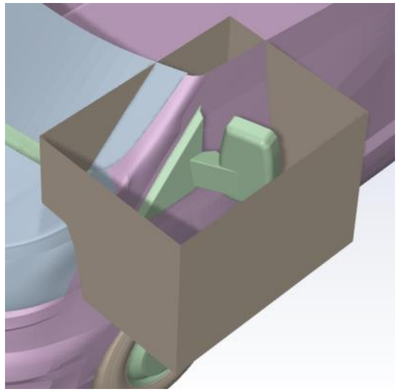
FTM – Add Offset surface/Box to create construction surface

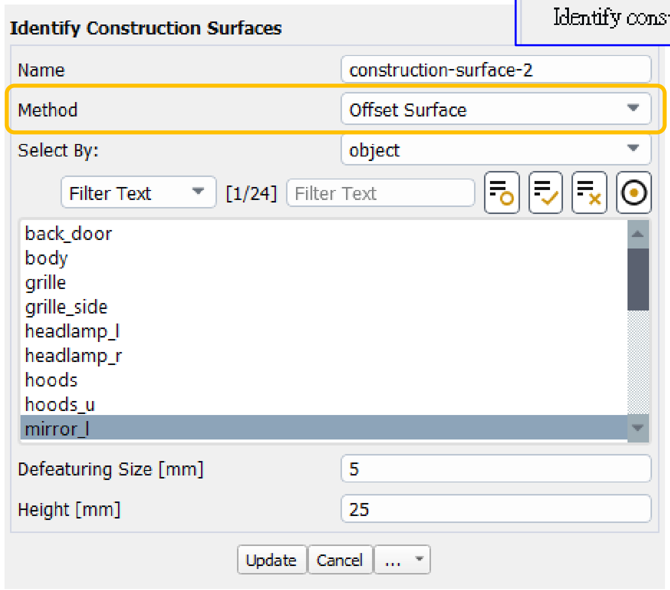
- Construction surfaces are remeshed and intersected with wrapper surface
- 2 new options to create them instead of importing them : Box and Offset
  - No transformation possible, box is aligned with global coordinate
- Useful for part replacement

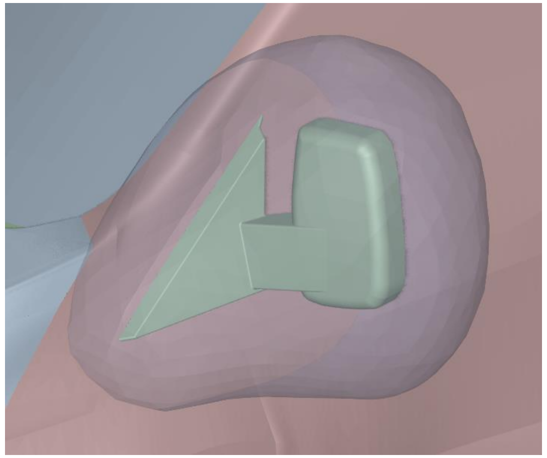








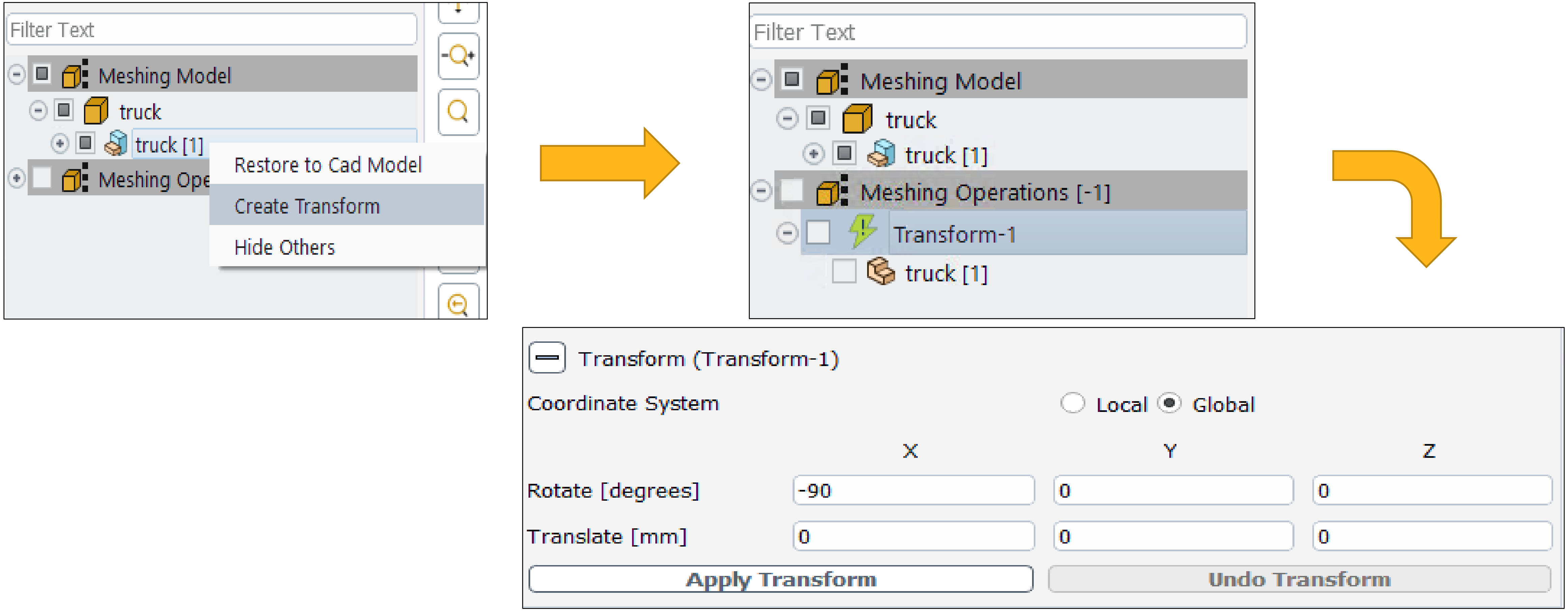




Defeature size : size of surface mesh
Height : distance to the object

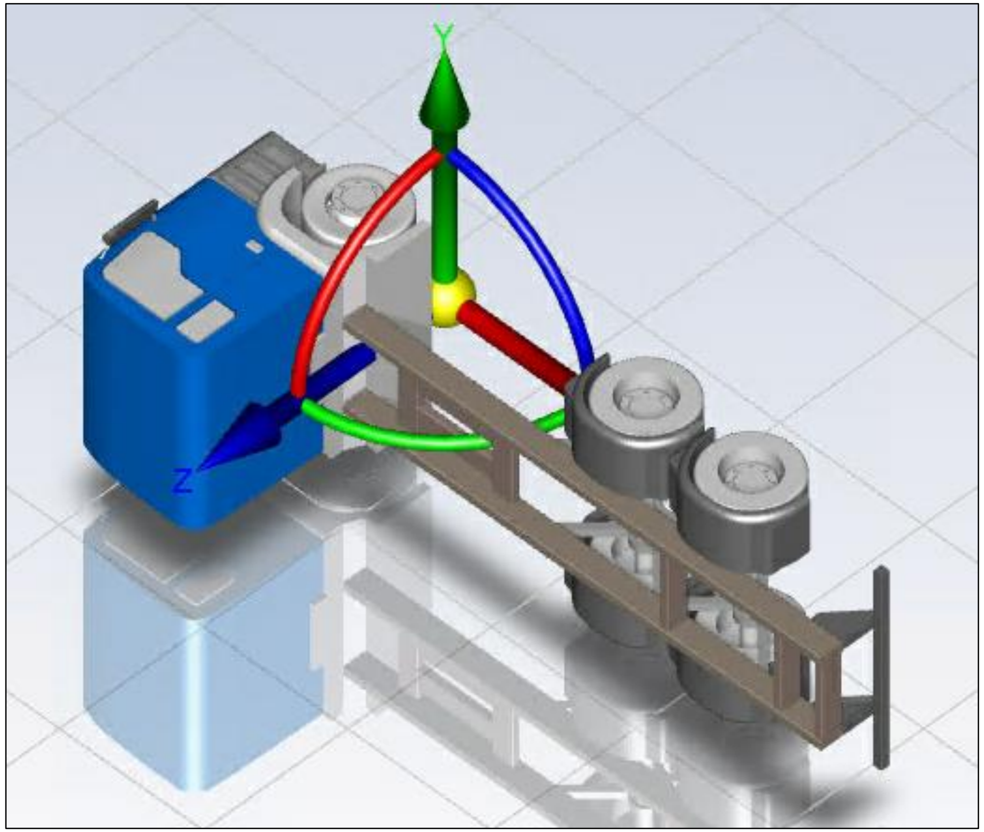
FTM: Append CAD and Part Management Transformations

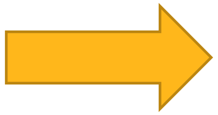
- This feature facilitates transforming cad objects by rotating/translating about local or global coordinate system.

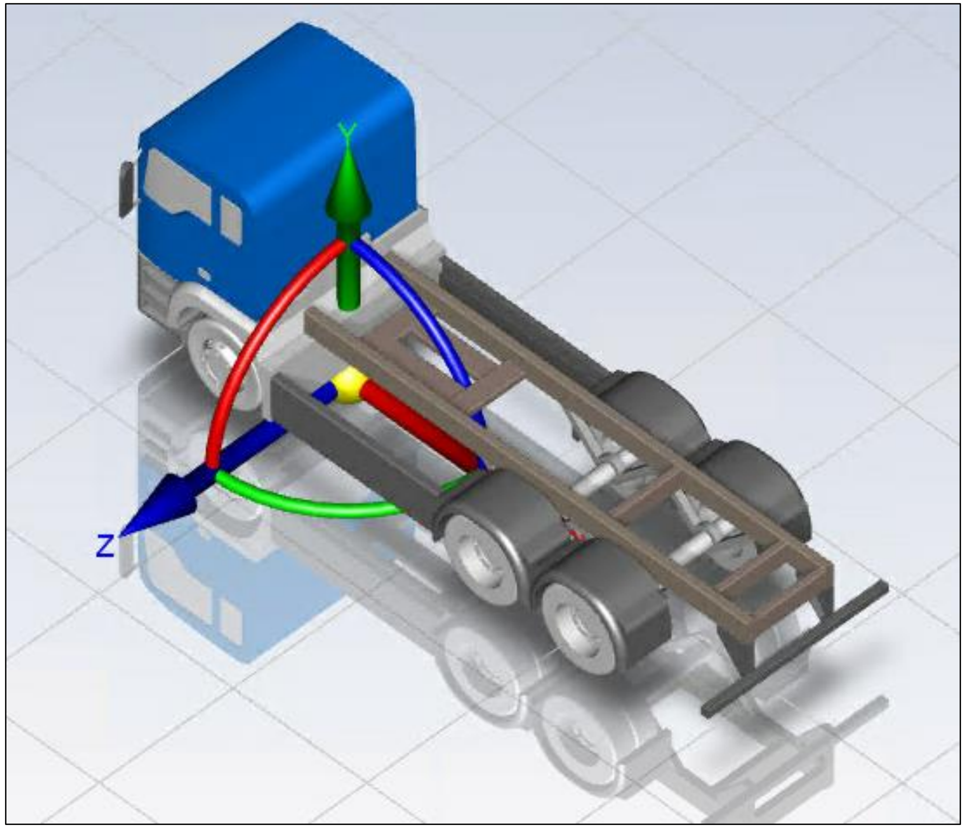


FTM: Part Management Transformations

- ❑ Rotating by –90 degrees about global X-axis







Fluent User Interface
使用者操作介面

Performance for Large Cases With Many Zones

Large improvements in user interface and I/O performance when 10k's of zones are present

- Example: Battery case - 24M cells, 32k face zones
  - Graphics Display ***up to 3X Faster***
  - Various command executions ***up to 20X Faster***
  - Bottlenecks removed in display of many dialog boxes
- Example: CHT – 64M cells, 22k face zones
  - Case read 2X faster compared to 2021 R2
- Combined, some scripted case-setup workflows are ***up to 6X Faster***

Operation	Percentage Improvement (Average of serial and parallel on 8 cores)
Mesh Display	73%
Contour Display	83%
Vector Display	80%

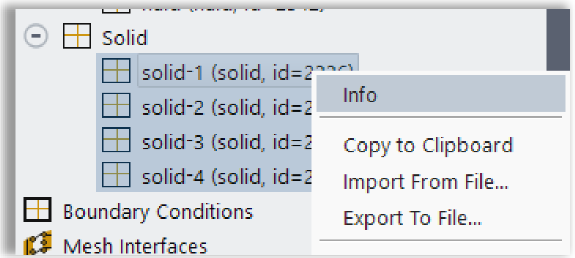
Battery test case: 24M cells, 32k face zones

	Time (s)		Improvement	
	21.2	**22.1**	**(seconds)**	**(%)**
Time taken to turn on energy	12.72	5.89	6.83	53.67
Time taken to open Turbulence panel and apply change	26.41	16.61	9.8	37.13
Time taken to open Battery Model and OK	224.76	201.04	23.72	10.55
Time taken to open Fluids material panel and change/create	0.77	0.7	0.07	8.7
Time taken to open Solids material panel and change/create	0.26	0.22	0.04	14.89
Time taken to open fluid cell zone panel and apply	74.16	64.83	9.33	12.58
Time taken to open solid cell zone panel and apply	20.56	12.32	8.24	40.1
Time taken to open wall BC panel and apply	20.48	14.67	5.81	28.4
Time taken to create solid zone surfaces	18.37	17.33	1.04	5.63
Time taken to open mesh object panel	12.86	6.28	6.58	51.14
Time taken to open contour object panel	3.98	0.26	3.72	93.58
Time taken to open vector object panel	7.65	0.21	7.44	97.31
Time taken to open pathline object panel	7.72	4.71	3.01	38.99
Time taken to open particle object panel	0.31	0.28	0.03	7.27
Time taken to open surface area report definition panel	19.69	0.73	18.96	96.3
Time taken to open surface custom vector based flux report definition panel	8.09	0.32	7.77	96.02
Time taken to open surface custome vector weighted avergae report definiti	10.97	0.29	10.68	97.32
Time taken to open surface facet average report definition panel	8.15	0.32	7.83	96.11
Time taken to open surface facet maximum report definition panel	11.92	0.81	11.11	93.19
Time taken to open surface flow rate report definition panel	8.26	0.34	7.92	95.83
Time taken to open surface integral report definition panel	11.34	0.25	11.09	97.8

User Interface Performance and Usability

Easier access to cell-zone details including quality statistics

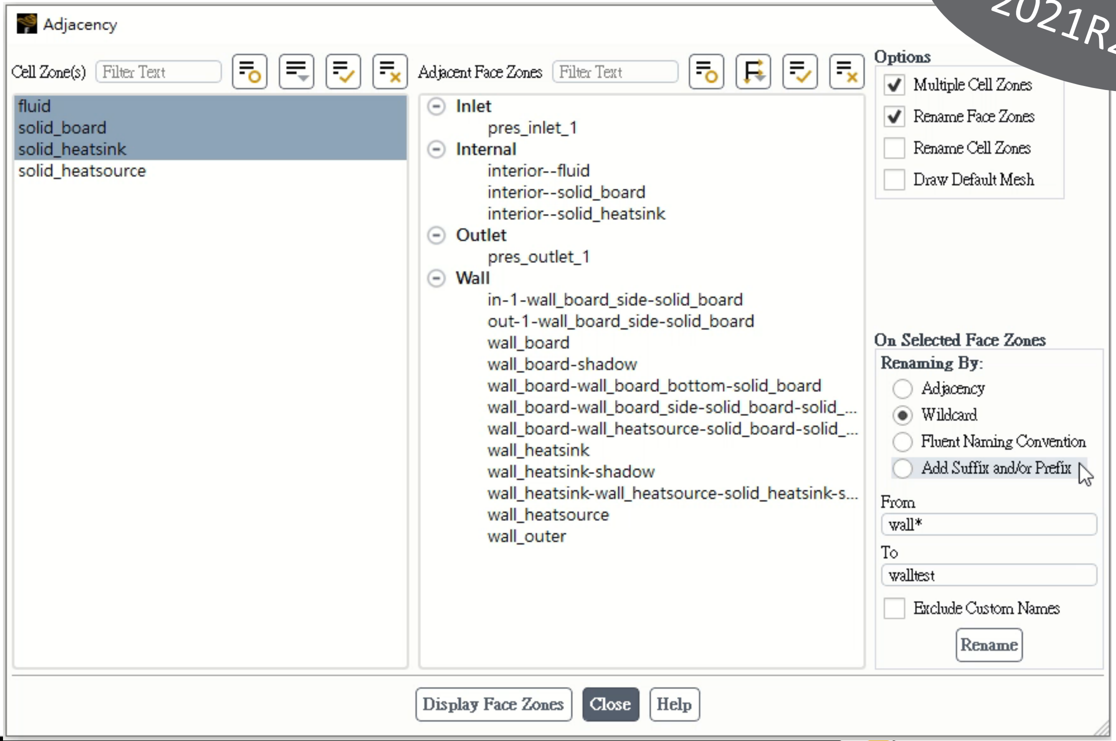
- RMB > Info on cell & face zones
  - You can easily print mesh face and cell counts by zone using the Info context menu option for the Cell Zone Conditions and Boundary Conditions branches in the Outline View tree (accessed via right-click).



Cell Zone Name	Cell Count	Minimum Orthogonal Quality	Orthogonal Quality Below 0.1
solid-4	8083	0.20356136	0
solid-3	6293	0.32538144	0
solid-2	20388	0.20182542	0
solid-1	6322	0.27628726	0

- Adjacency panel add suffix/prefix
  - Bulk renaming of cell zones is now available in the **Adjacency** dialog box. You can rename cell zones by adding suffixes and/or prefixes, by wildcard, and by converting to the Fluent naming convention.

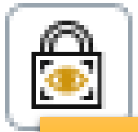
2021R2



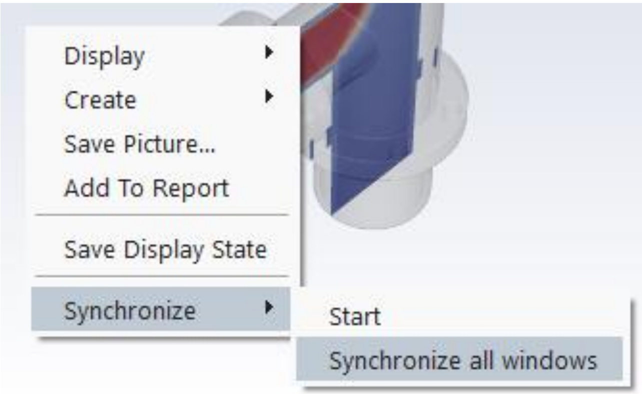
View Synchronization

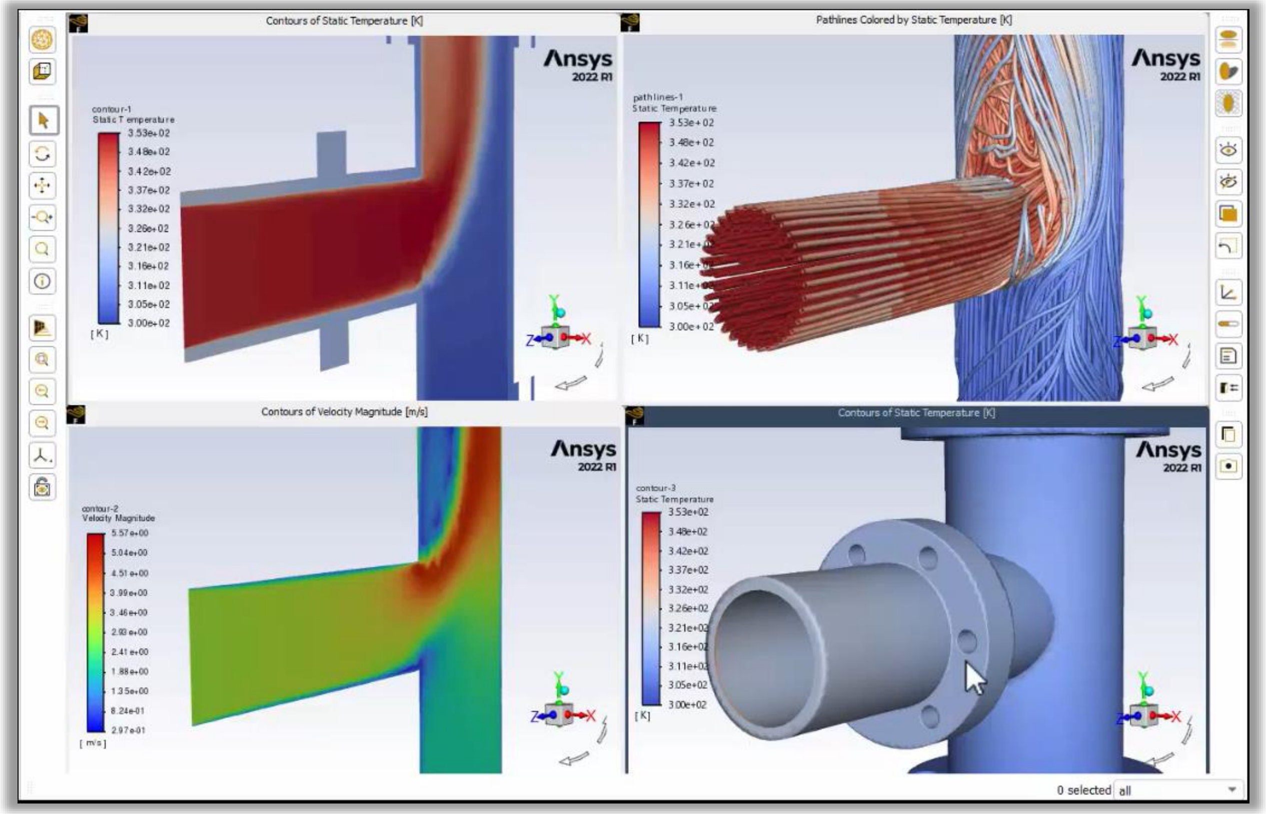
Lock view orientation among multiple windows for visual comparison

- Simultaneously view multiple visualizations from a consistent viewpoint
- Synchronize all sub-windows, or only selected sub-windows
- Accessible from toolbar icon or graphics window context menu:



Click to synchronize all windows





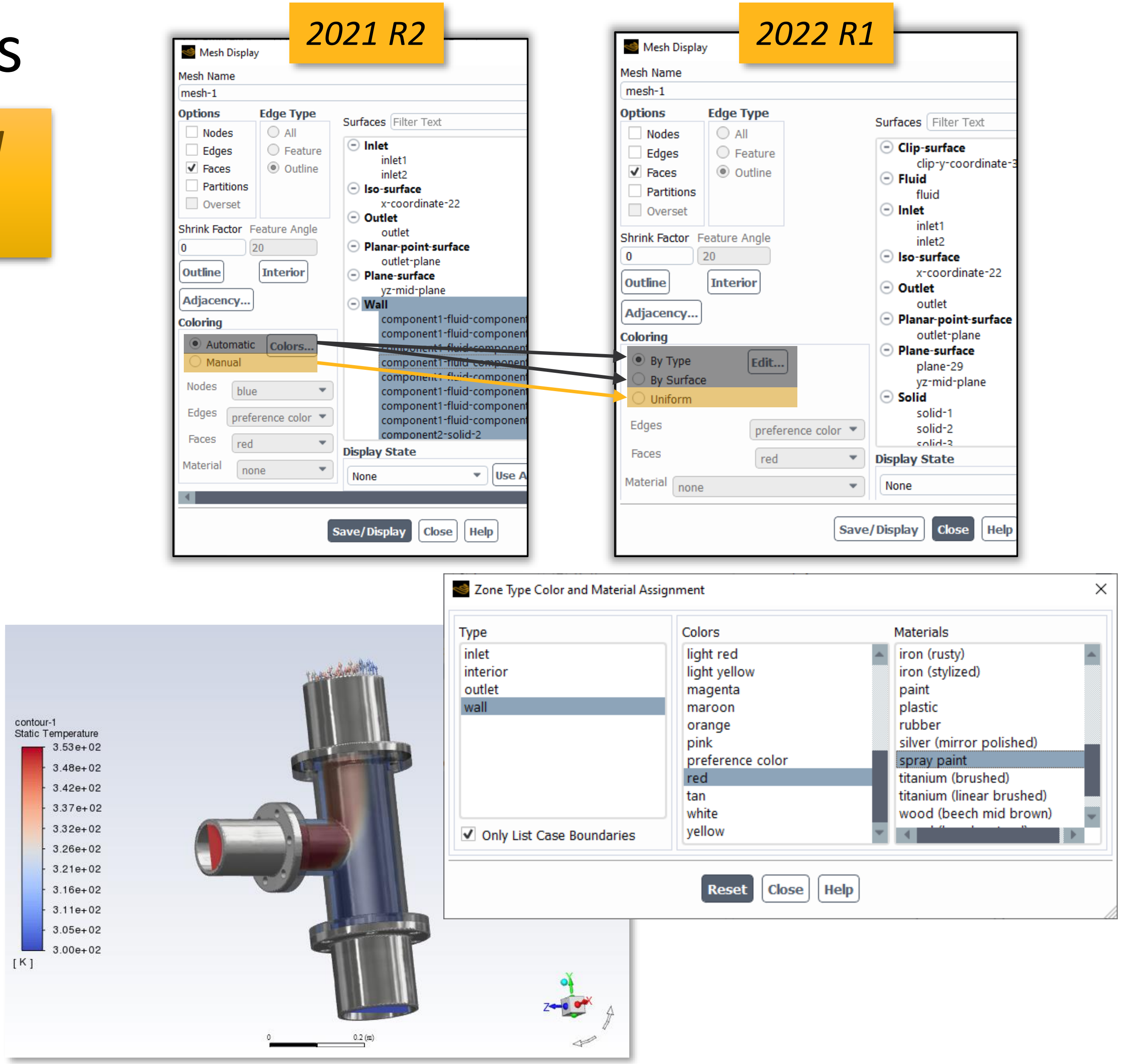
Video showing viewport synchronization

Mesh Rendering Enhancements

New materials for rendering and added flexibility for Mesh Display objects

- Many new solid materials available for rendering (brick, concrete, etc.)
- Choice of color **by ID** vs **by Type** are now stored for each Mesh Object
  - **Automatic** option split into **By Type** and **By Surface** (analogous to by ID)
  - **By Type** can be used to configure both color and material choices

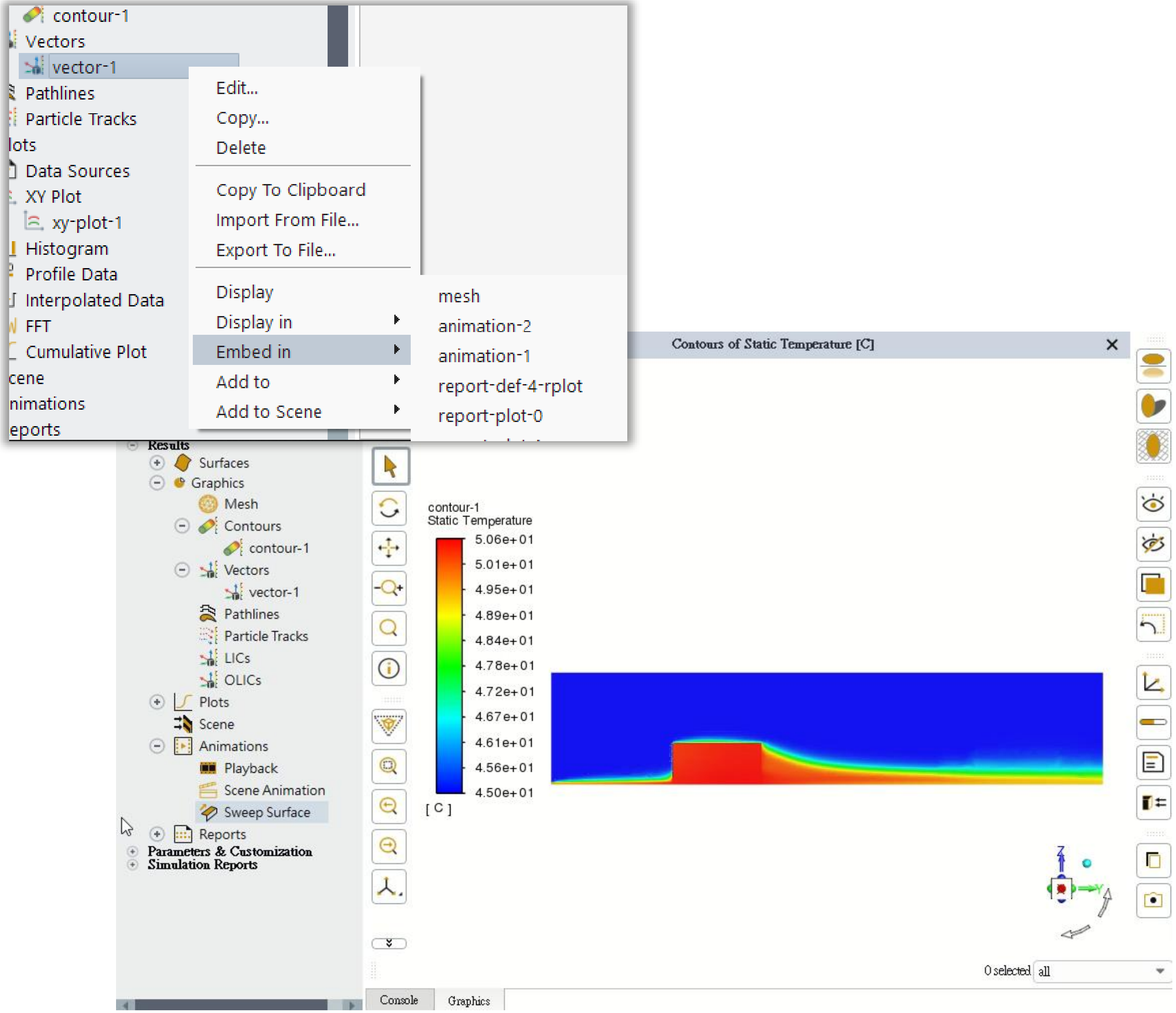
    ⇒ **Removes requirement that same material must be used for all surfaces in a mesh display object**
  - **Manual** renamed to **Uniform**



Embedded Window and Animation Enhancements

Workflow improvements

- Embedded Windows
  - Exposure in Outline View context menus
  - Not limited to Reserved windows
  - Placeholder frames can be embedded before start of simulation
  - Journal support
  - Animation frames not stored by default when using **Automatically embed residuals during calculation**

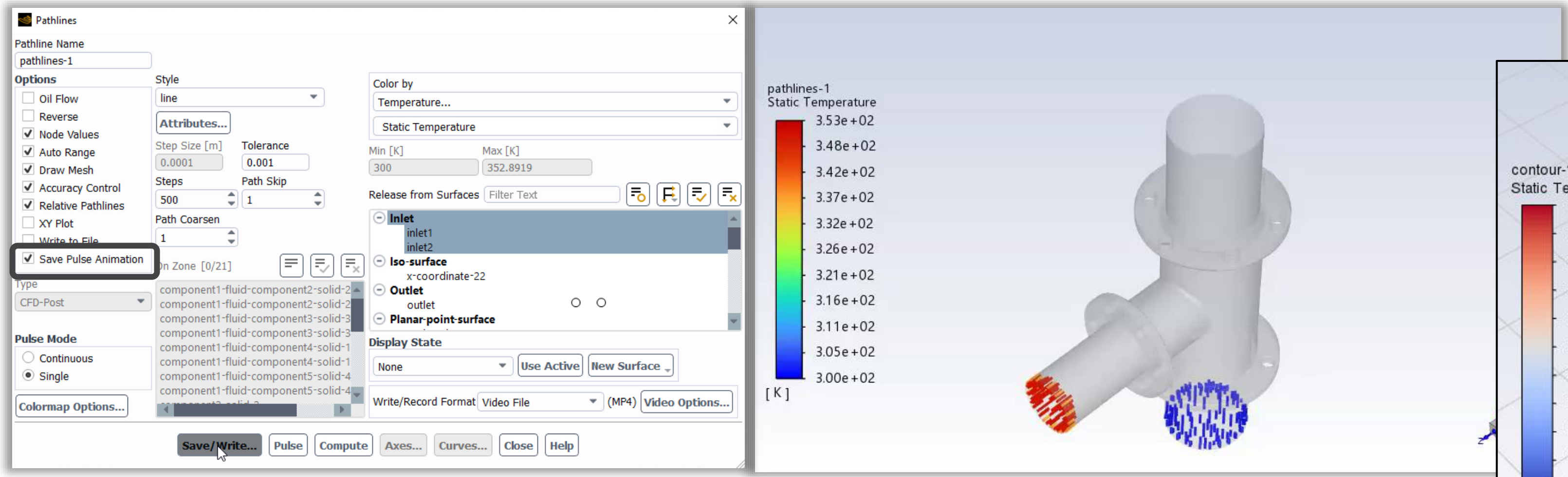


Miscellaneous Usability Enhancements

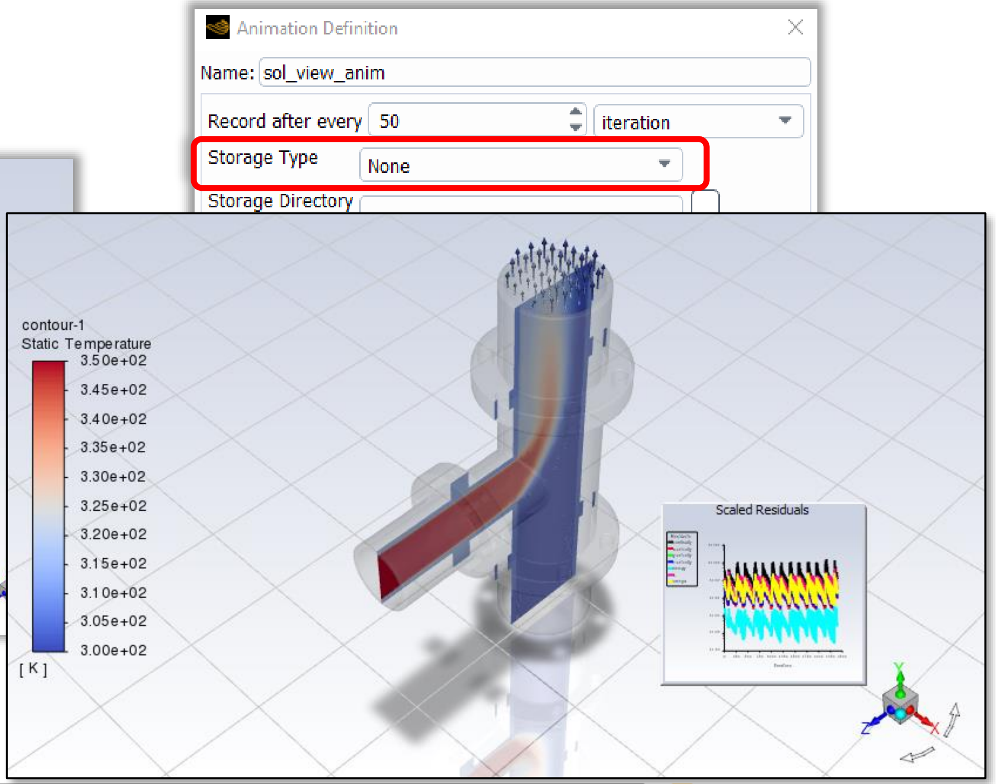
- *Recordable pathline animations*

- Pulsed pathline animations can be saved in common video formats (MP4, AVI, FLV, MOV, MPEG)

- Animation options
  - **None** option for **Storage Type:** render periodic visualization updates during solution without any memory / file use
    - Precludes later playback / recording



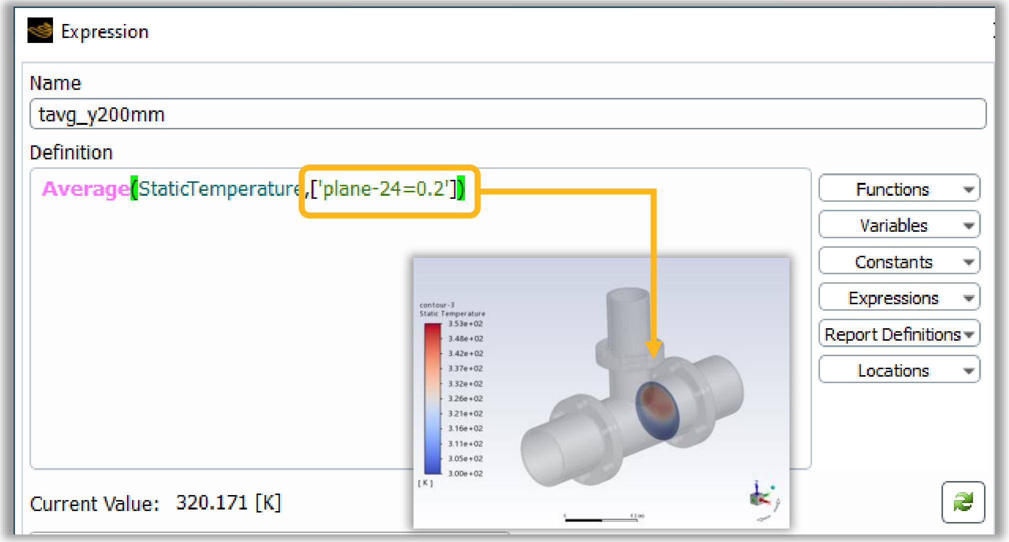
Video showing pulsed pathlines

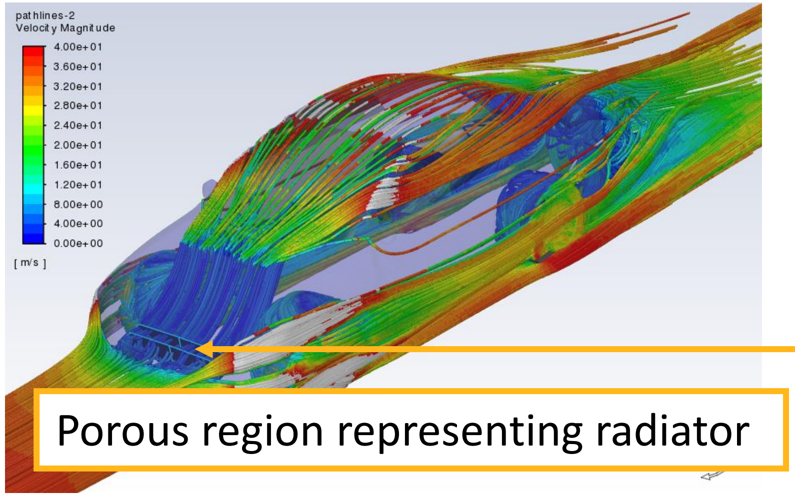


Expressions Enhancements

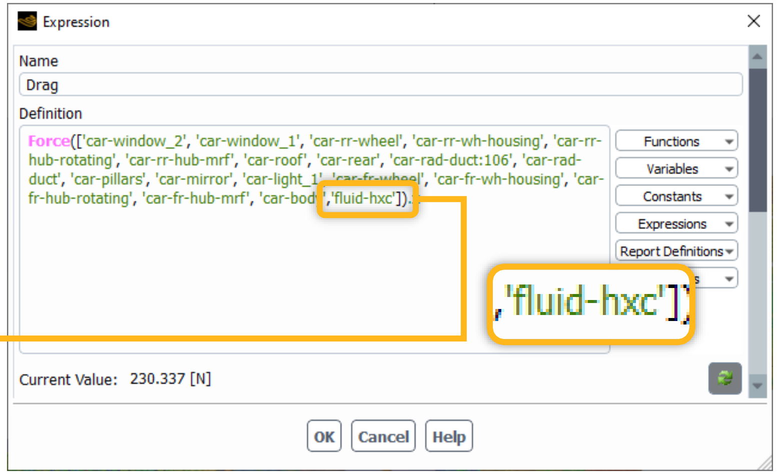
Support for User-Defined Surfaces and Porous Region Contribution to Forces

- User-defined surface(s) can be included in the **Location** for reduction functions (Average, Minimum, etc)
- Porous region contributions to forces
  - E.g., aero cases with radiators





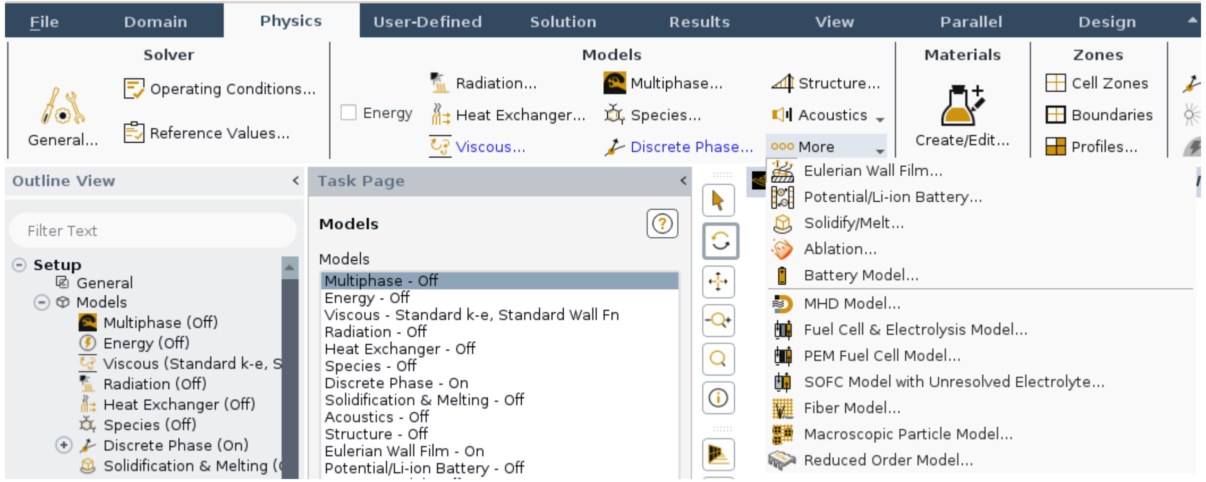
Porous region representing radiator



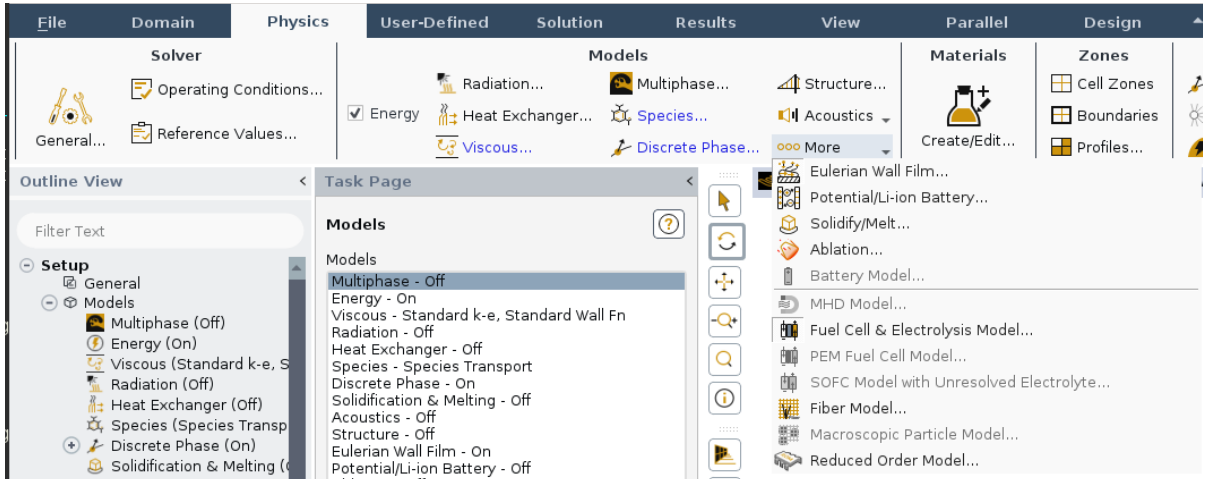
Fluent Add-on Model
新模組

Exposure of Add-On Modules in Ribbon

Easier access to add-on modules from GUI



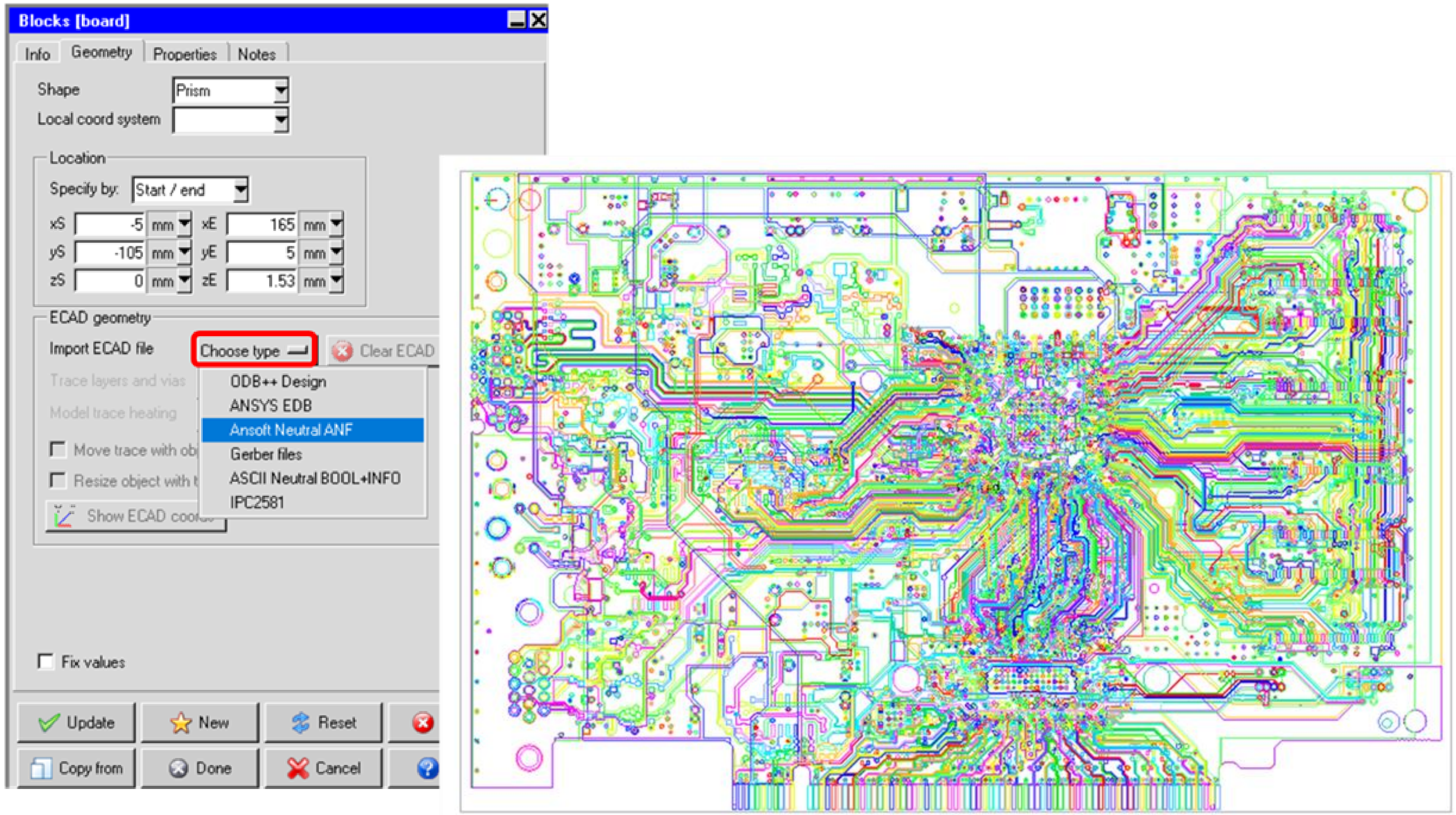
- Provide access to supported add-on modules in the ribbon
  - Simplifies access
  - Includes new defined icons
  - TUI is maintained for scripting
- Added compatibility checks of add-on modules where possible
  - Incompatible options are grayed out



Thermal Analysis of Printed Circuit Boards

Thermal analysis of a PCB requires knowledge of the orthotropic components of thermal conductivity (that is, X, Y, and Z components of thermal conductivity).

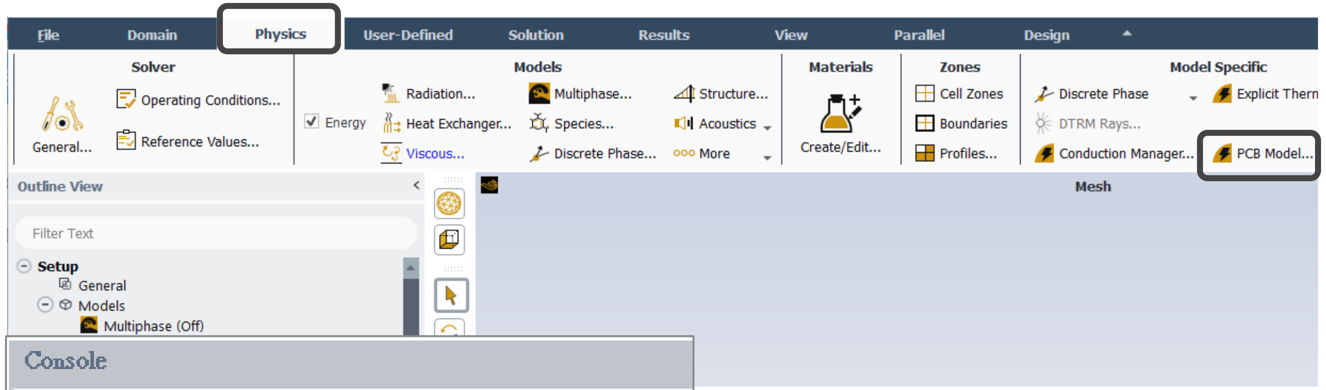
- **Pre-requisite Files**
  - Electronic CAD mesh file (ECAD for Metal and di-electric layers, generated in Ansys Icepak)
  - Board configuration file from Ansys Icepak ("board_config.dat")
  - Metal fraction information from Ansys Icepak (.cond file)
  - Power profiles from Ansys Icepak (.prof files)

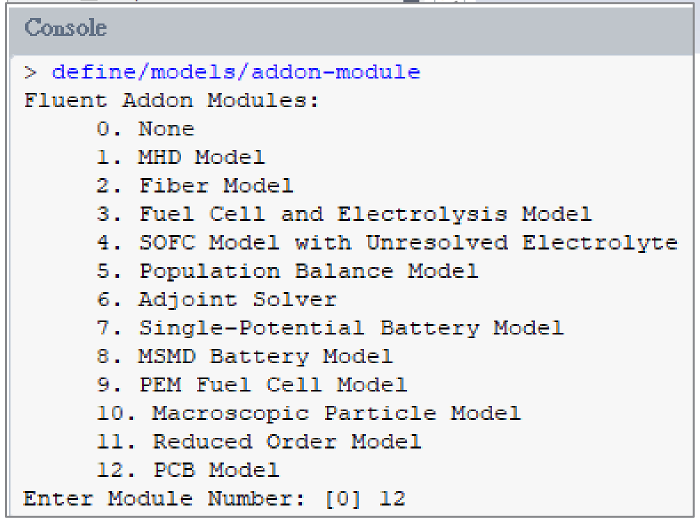


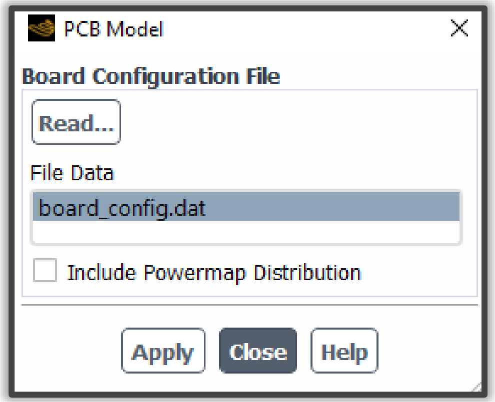
IcePak → Fluent Workflow for Printed Circuit Boards

Include an IcePak PCB analysis in a Fluent case, for example LED headlamps and automotive sensor assemblies, where the complex geometry and/or physics handling of Fluent is needed

- Append a PCB-only *.cas file from IcePak into a Fluent case with other geometry / physics
- Use Fluent's Mesh interfaces to connect PCB to rest of the domain
- Use the IcePak *board_config* file* to apply an accurate PCB thermal conductivity profile in Fluent



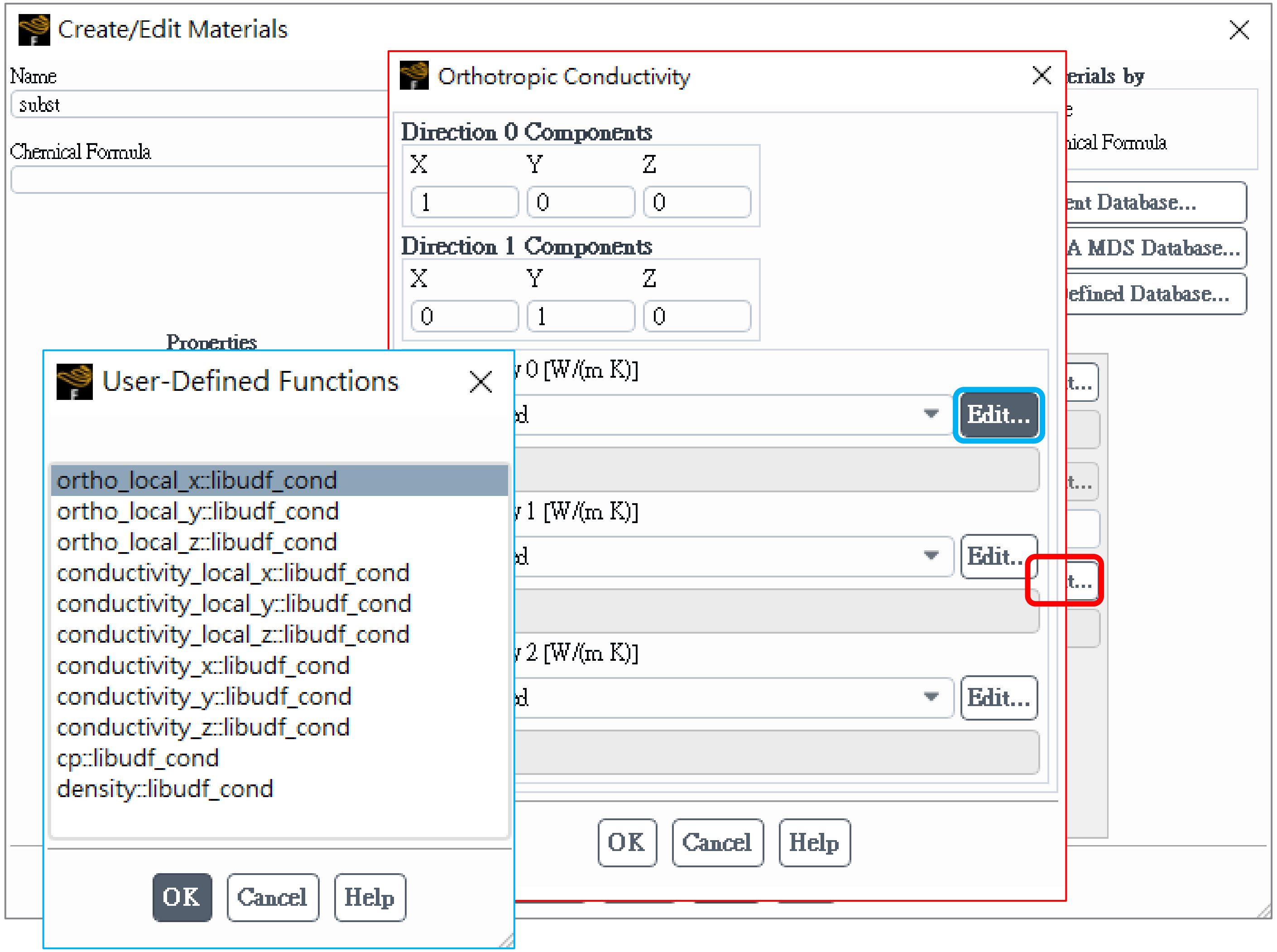




Note: It may be necessary to edit the thread_id in the board_config file to match the correct zone in the full Fluent simulation

Material Properties for PCB Model Material

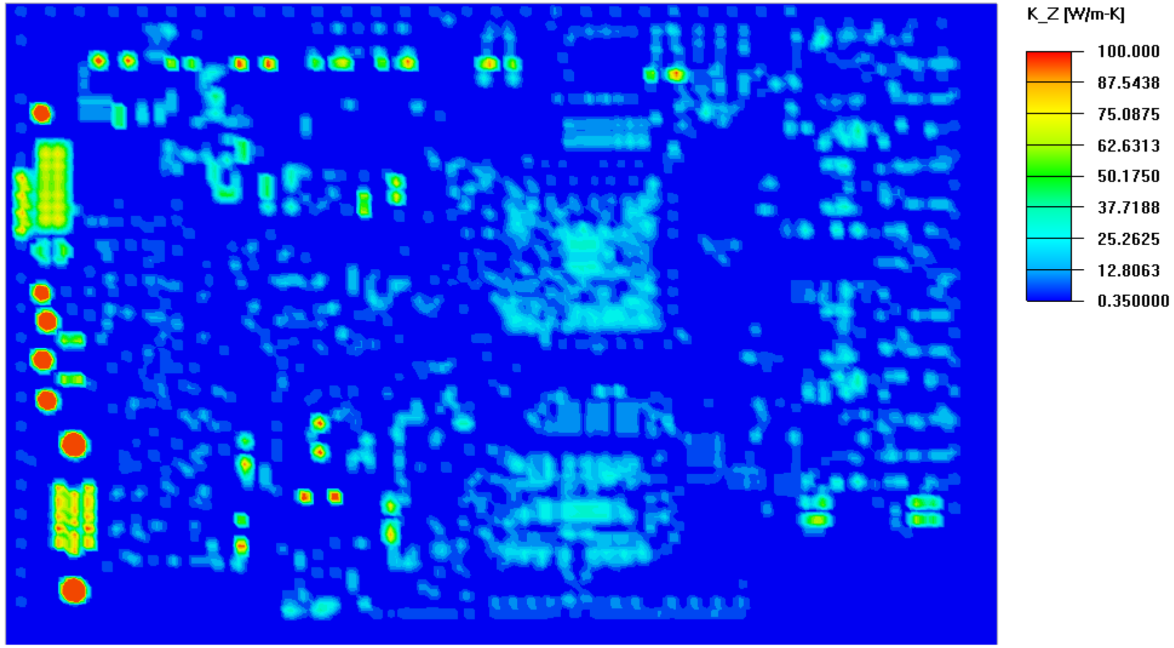
- The material properties of subst are automatically defined:
- Density is defined via UDF that is automatically set by the PCB model.
- Specific heat is assigned a constant value of 1 because the density defined by the UDF is itself multiplied by the specific heat.
- Thermal Conductivity is defined as orthotropic. Clicking Edit... will open the Orthotroptic Conductivity dialog box.



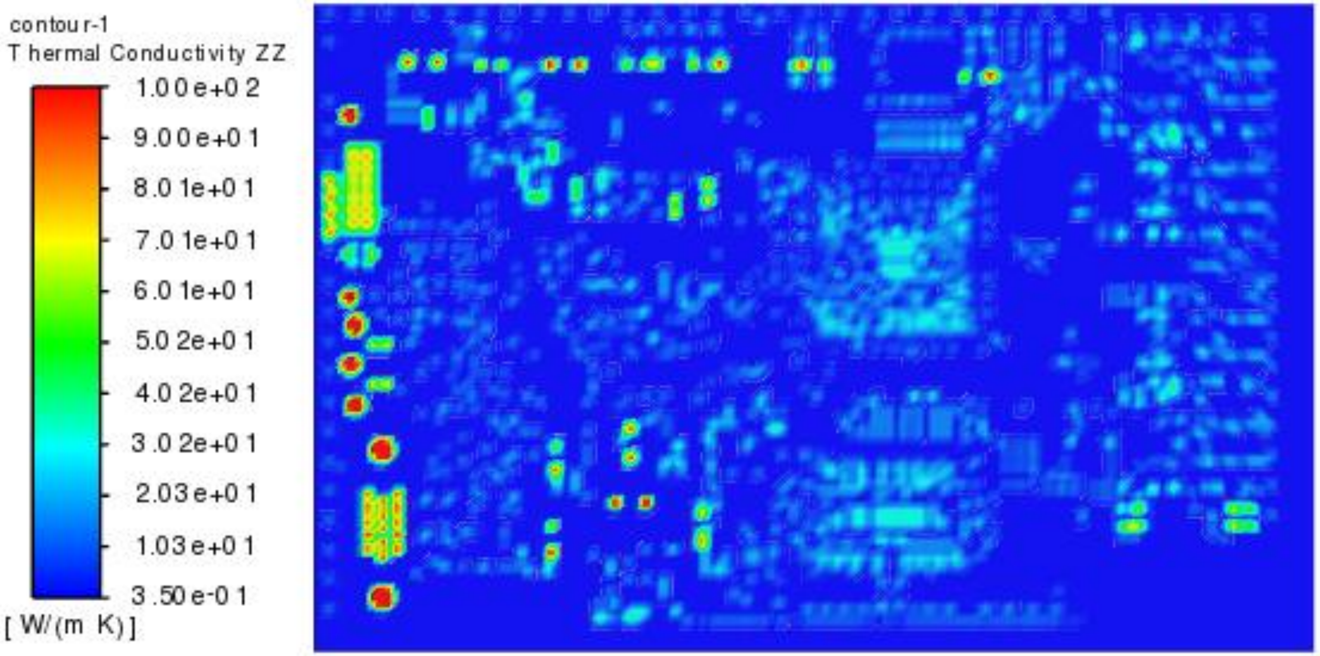
Postprocessing for the PCB Model

- The following additional variables are available for postprocessing with the PCB model:
  - Thermal Conductivity X
  - Thermal Conductivity Y
  - Thermal Conductivity Z
- Note that these orthotropic thermal conductivities are computed in the global coordinate system.

Icepak



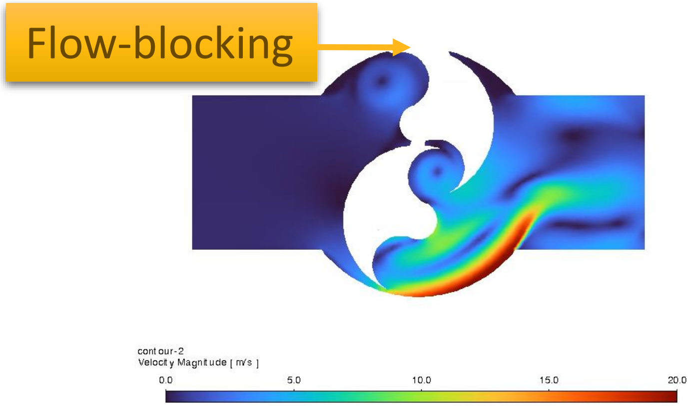
Fluent

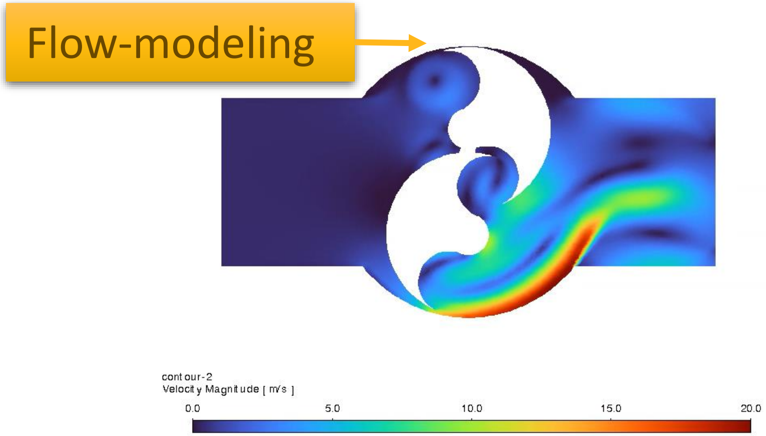


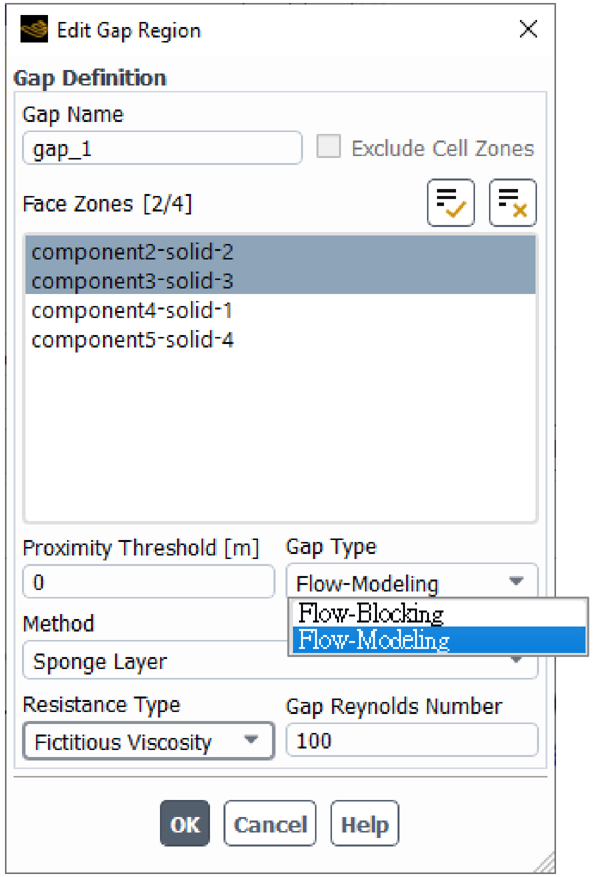
Flow Modeling in Thin Gaps

Provides a modeling solution for flow through narrow gaps in moving mesh simulations

- Alternative to fully-blocking flow in narrow gaps
- Sponge-layer method artificially increases viscosity based on user-supplied Reynolds number in the gap region
  - **Fictious Viscosity:** scaled viscosity in discretization of momentum equations only. Other equations use scaled velocities from momentum equations.
  - **Real Viscosity:** scaled viscosity effects are observable in all equations







- User-Defined Source method with DEFINE_GAP_MODEL_SOURCE

Ansys