### 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



#### 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

### Supported Capabilities:

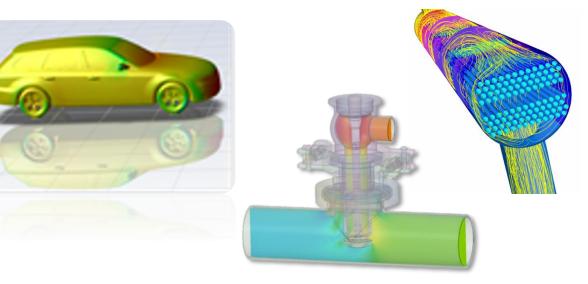
- Single/multi-GPU (shared / distributed memory)
- Supports all mesh types (poly, hex, tet, pyramid, prism, hanging node)
- Subsonic compressible flows
- Ideal gas
- Material with constant properties
- Turbulence: standard k-epsilon and GEKO k-omega
- Solid conduction and CHT
- Moving walls
- Porous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

#### Target applications for first release:

- External aerodynamics
- Internal flows / ducting / ventilation
- Heat transfer



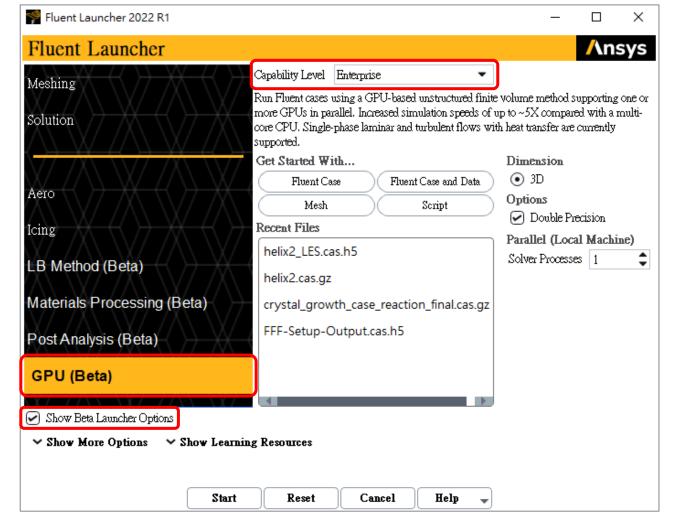
### Program Capabilities and Supported GPUs

- 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



- 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 GPU Workspace											- 0	×					
File View 🔺						_			uick Search (Ct	Ctrl+F)	(?)	\nsvs					
Display	Graphics		Graphics Toolbars		<u>F</u> ile	•	View	-						<b>Q</b> Quid	k Search (Ctr	0	/\nsys
💽 Views Headlight 🚽 🗹 Lighting	🍳 Lights	🖌 Mesh Display 🖌 Vi	iew 🖌 Copy	✓ Graphics Effects	Conso	ole											0 <
Dptions 🥢 Automatic -		V Pointer Vi	isibility 🗸 Object Selection/Disp	alasr	+								-+				
Camera	Colormap			,,				GPU Sol	ver (Be	eta) en	abled		1				
Outline View	< Properties - contor undary Values		Residuals	Residuals		Please ma 1. GPUs			ch can	be che	cked with nvic	lia-smi	i				
	Filled Contour	<b>v</b>	-svelocity 1	<sup>e+01</sup>	i	2. Sett	ings su	pported					i				
Cell Zones     fluid [2]	r Lines							ha invalu		tonti	en vith eeme		1				
Boundaries	9	smooth 💌	_ <b>\</b>	e+00		checks	rWill	De invok	ed at 1	teratio	on, with compa	tibility	-				
inlet [5]	lesh				+								-+				
arr outlet [4] i wall [3]				1e-01 -							converted to						
😑 🔀 Solution	Compute Range	✓	<b>·</b> <u>+</u> <b>·</b>	1		tization	Scheme	has been	conver	ted to	Least Squares	Cell Based	1.				
⁰ Methods ⅔ Controls	lobal Range	✓	- <b>Q</b> +	1e-02 -	>>>	Ŧ											
🖄 Residuals	num Value	0		1	>>>	T											
<ul> <li>Report Definitions</li> <li>Q Calculation Activities</li> </ul>	านm Value	0	Q,	1e-03		ting the								1 125 051 110172 07 3	11 11 1200 17 1800 400	1 12 12 12	
Autosave	lap			200 220 240 280 3 Itera	chung.	ing Discr ring GPU					nt Kinetic Ene	ergy: from	n: Second	Order Upwind	l to: First Orde	er Upwind	
<ul> <li>Run Calculation</li> <li>Results</li> </ul>			- 🔍			ring GPU											
Surfaces		•			Rank	PID	Core	0S	Host	Devic	e Name		Version	Memory(GB)	Bandwidth(GB/s)	Cores	
<ul> <li>Views</li> <li>back</li> </ul>	Display Sav	/e Image] Delete	<b></b>		-												
j back	Console				0			lnamd64 lnamd64			NVIDIA A100 NVIDIA A100			79.1703 79.1703	1845.7 1845.7	6912 6912	
[€] front	paratter oper				2			lnamd64			NVIDIA A100			79.1703	1845.7	6912	
isometric igi left	+		r (Beta) enabled	+													
ight						idth : 3.											
top		nake sure Js installed, which	can be checked with	nvidia-smi	Laten	cy : 0.	978206	usec									
<ul> <li>Graphics</li> <li>Meshes</li> </ul>	2. Set	ttings supported			Using	uniform	initial	izer									
Mesh Outline			l at iteration, with o	ompatibility	-	d setting		1261.									
Contours	checks	3		ا +		d mesh											
Contour-1			me has been converted														
Pathlines	Dicretization	i scheme nas been co	converted to Least Squ	ares Cell Based.	ite	r tim	e/iter										
Plots     XY Plots		I CPU		System M	- e iter	D	ressure	x-	velocit	v	y-velocity	z-veloc:	tv	epsilon	k		-
S. Plot From File	Hostname		re x HT Clock (MHz)	Load (%)  Total	1		019e+00		7118e-1	-	5.47118e-16	4.40000e	-	.15773e+02	3.93282e+01		1
Scene	NB190	1 x 8 x 1	2496	7.94065  31.7332	Contra	la Crast	nice										
<ul> <li>Reports</li> <li>Surface Integrals</li> </ul>	 Total				_ Conso 15.8		lics										
Wolume Integrals	IOCAL	1 0	-	-   31./332	12.0	1						<b>v</b>					



 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.

Energy	✓	
Viscous	k-epsilon	•
Gravity		
<sup>⊙</sup> Parameters		
Cmu	0.09	
C1-Epsilon	1.44	
C2-Epsilon	1.92	
TKE Prandtl Number	1	
TDR Prandtl Number	1.3	

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

Properties - air		0 <				
Name	air					
Туре	Fluid					
Density						
Method	Constant	•				
Value [kg/m^3]	1.225					
ි Cp (Specific Heat)						
Method	Constant					
Value [J/(kg K)]	1006.43					
O Thermal Conductivity						
Method	Constant	¥				
Value [W/(m K)]	0.0242					
• Viscosity						
Method	Constant	*				
Value [Pa s]	1.7894e-5					

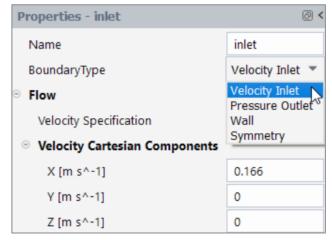


#### • Merging and Deleting Cell Zones

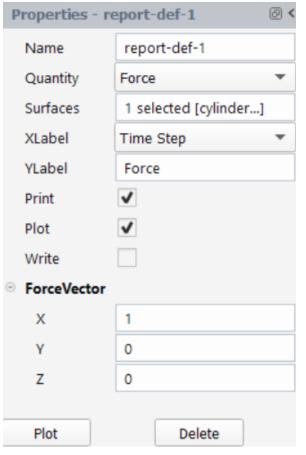
- Although the Fluent GPU Solver only allows for a **single fluid cell zone**, if you import a Fluent case, it may contain more than one fluid cell zone. All of the solid cell zones will automatically be removed and all of the fluid zones will be imported. At this point, you must **merge** or **delete**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

	Selections ×
Merge Cell Zones ×	[0/2] 🗟 🔁
Zones 0 selected Edit	fluid-mrf fluid-tank
Merge Close Help	
Delete Cell Zones X	
Zones 0 selected Edit	
Delete Close Help	OK Cancel Help

- 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)



- 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.

Properties - Run Ca	lculatio	n	0
Transient			
Number of Iterations	;	1000	
Reporting Interval		1	
Properties - Run Calculatio	n		0 <
Transient	✓		
Time Step Method	Fixed		*
Time Step Size [s]	1		
Number of Time Steps	0		
Reporting Interval	1		



### 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.

	·		NSYS Inc\v221\f				
NVIDI	A-SMI	496.1	3 Driver	Version:	496.13	CUDA Versio	on: 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.
			I			1	MIG M.
======	=====	=====	===============+			=+=========	
0	Quadro	o P400	Ø WDDM	0000000	0:D5:00.0 On	1	N/A
46%	35C	P8	11W / 105W	3463M	iB / 8192MiB	1%	Default
			+			-+	
Proce	esses:						
GPU	GI	CI	PID Typ	e Proc	ess name		GPU Memory
		ID					Usage
		=====		=======			
No n	unnin	g proc	esses found				

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



© 2021 ANSYS, Inc.

### Multi-GPU Licensing

- A CFD <u>Enterprise</u> 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 2<sup>nd</sup> GPU, then 64 further HPC tasks for the 3<sup>rd</sup> GPU, etc.

<b>" 0</b> 011	Number of HPC Licenses						
# GPUs	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					



### Fluent Multi-GPU Solver (Beta) - Performance

#### Helix Pipe

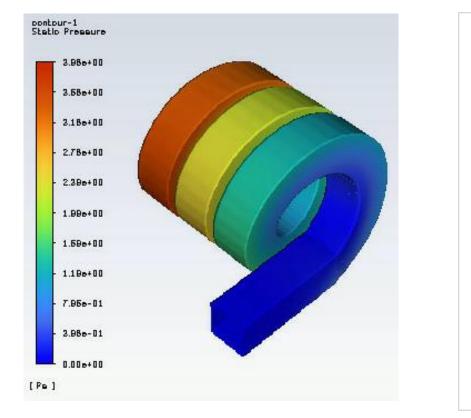
۲

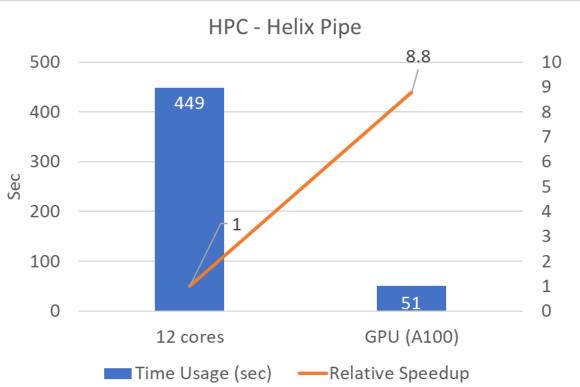
۲

Mesh Type: hexcore

Mesh Size: 208,000

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





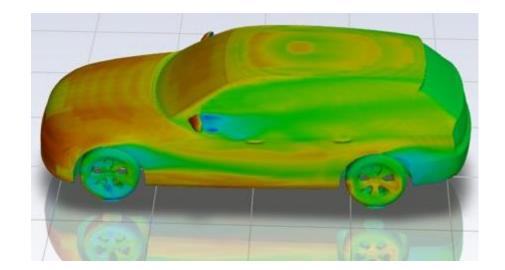


©2021 ANSYS, Inc.

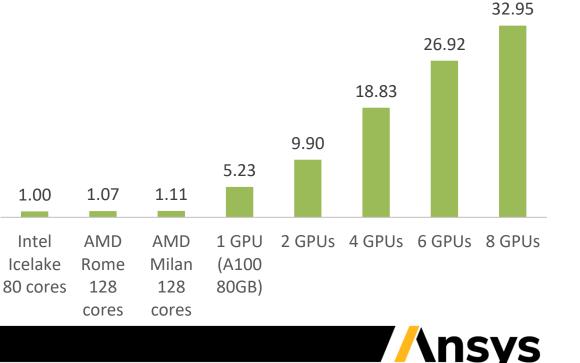
### Fluent Multi-GPU Solver (Beta) cont.

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

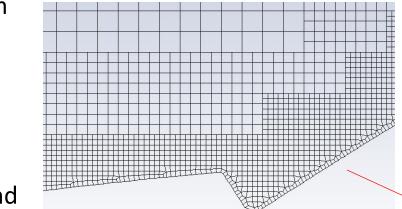


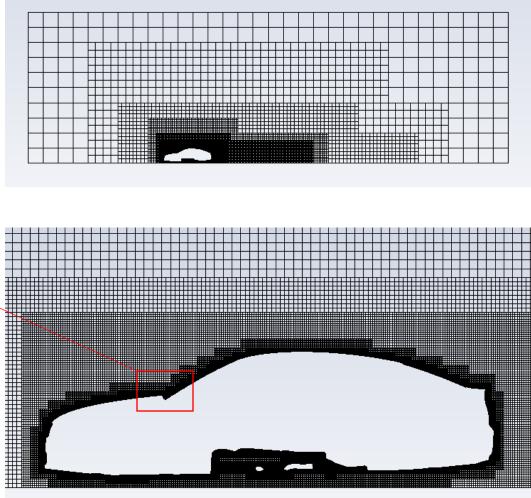




### DrivAER Car - Rapid Octree Mesh

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

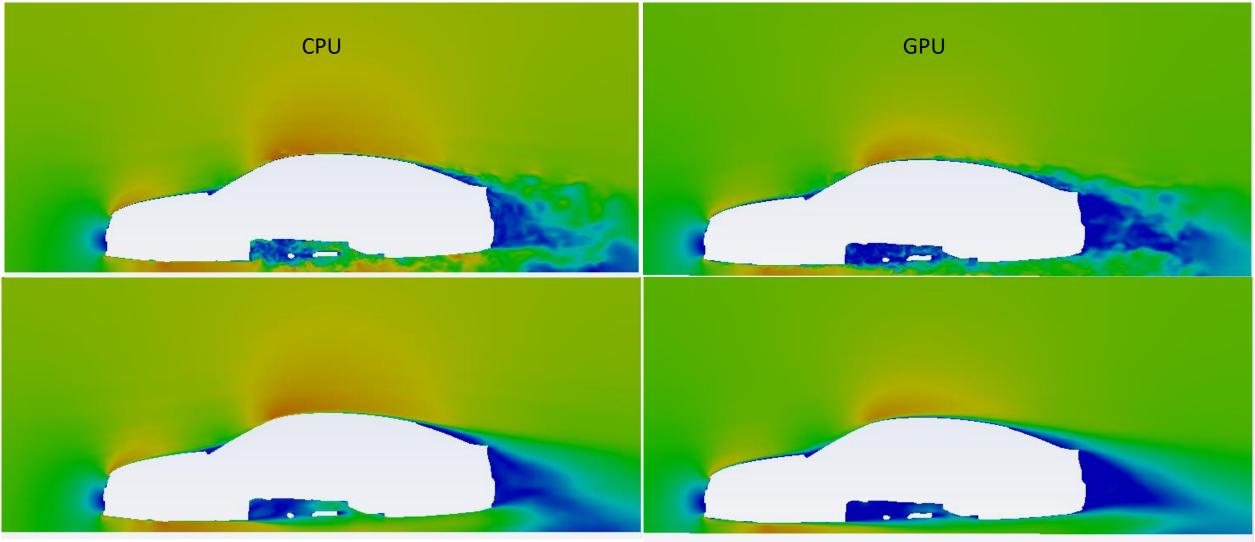






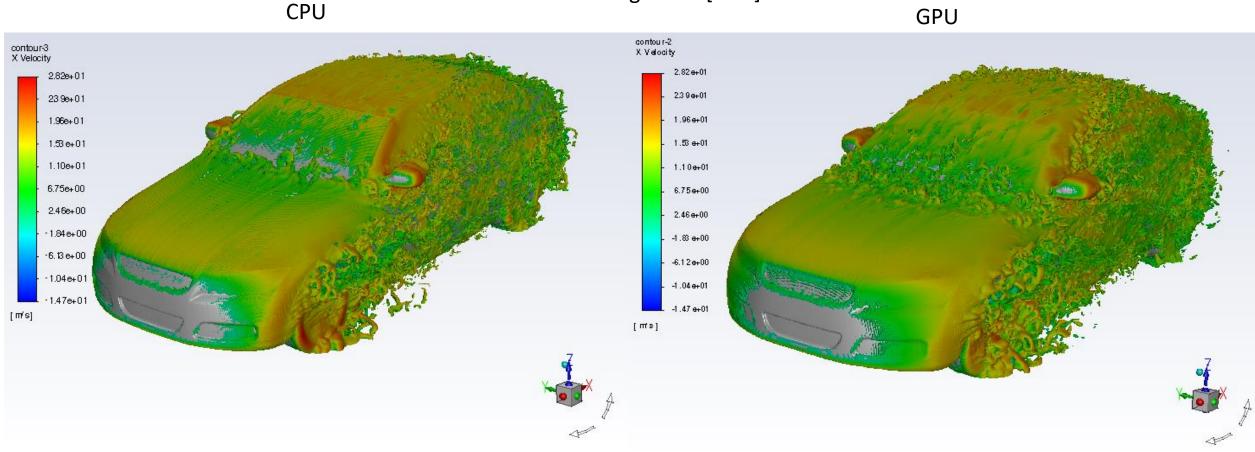
### **DrivAER Car - CPU/GPU**

#### Velocity Magnitude (0-25 m/s)





DrivAER Car - CPU/GPU

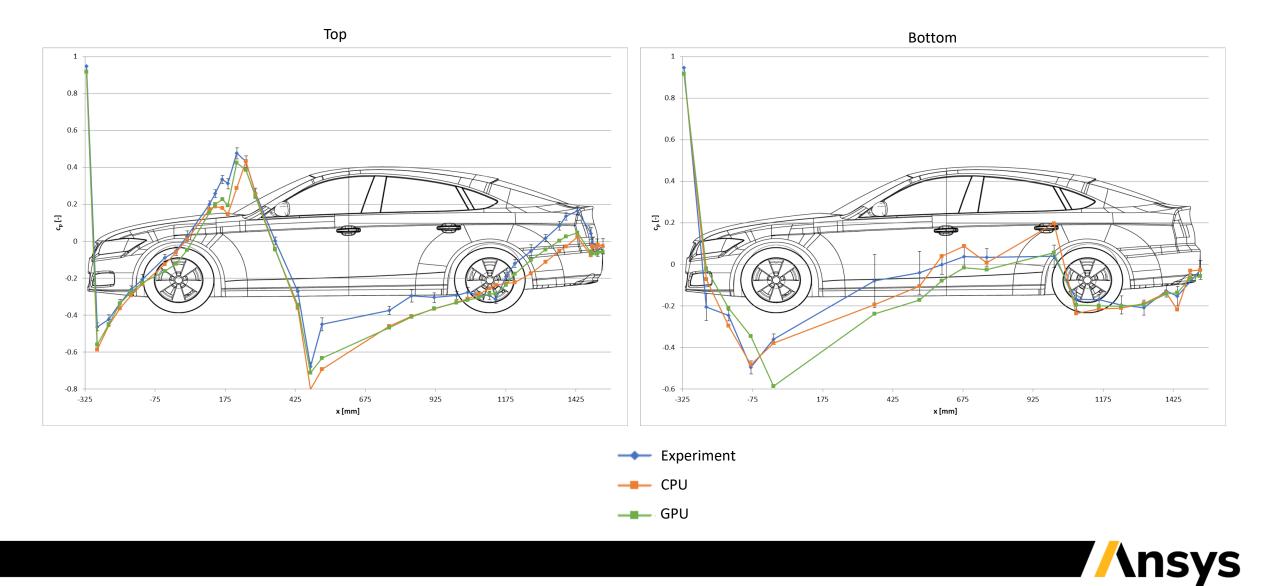


Iso. Vort. Mag = 500 [s^-1]



©2021 ANSYS, Inc.

### DrivAER Car - CPU/GPU – Mean surface Cp values



©2021 ANSYS, Inc.

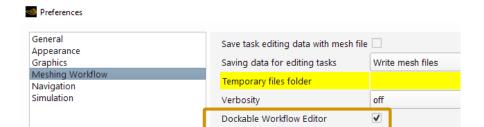
#### Fluent Meshing Workflows 流體網格處理流程



### Dockable Workflow Editor

- 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)

File Boundary Mesh Display Report	Parallel 🔺							Q Quick Search	7	/
Switch to +/ Delta 0	t Ranges Chypring Planes Sho Reset & Limit in X O Limit in Y O Lim	Freeze Cell Layer	Mouse Probe Function	✓ All Fees     ✓ Help Text       ✓ Free Faces     ✓ Highlight       Multi Faces     Edge Zones       ✓ Face Edges	■ * † •	Eramine	Patch Options	Selection Helper Filter Face Zones  Filter Face Zones  Filter Face Zones  Filter Face Zones  Filter Automation Constrained Face Automatic  Automatic  Automatic  Filter Face Zones  Filt		
Workflow Outl	ine View		1		Mesh	1		1		×
junction-saved-workflow	• 6 <b>6</b> 0 <sup>11</sup>									
🗸 촗 Add Local Sizing										
🖌 🧇 Create Surface Mesh										
💿 🖌 📐 Desarbe Geometry				BBBBBB						
<ul> <li></li></ul>										
✓ indet-1										
✓ indet-2 ✓ indet-2				Carl Carl Carl			1993			
✓ □ Could ✓ □ <sup>5</sup> Create Regions							2020			
✓ □ Update Regions										
<ul> <li></li></ul>										
<ul> <li>koundary-layer-on-fluid-walls</li> </ul>										
🗸 阙 Create Volume Mesh					338					
enerate the Volume Mesh									<b>z</b> •	
polyhedra polyhedra									Tex A	
V Mesh Solid Regions						(0)			- /	r
howth Rate 1.2 Max Cell Length 23.21095									200	
Region-based Sizing	●	/ 🔟 🕲 🐎 🔍 🕨	📑 📲 🛸	<b>A</b>						
Advanced Options	Console			, <u> </u>						
Global Boundary Layer Settings		rall Summary none	0	0.77535779 62878						Δ
	[Quality Meas	ure : Inverse Orthogonal Qualit	2Y]							
		62878 cells were created in	1 : 0.28 minutes							
		The mesh has a minimum Orth	ogonal Quality of:	0.22						
			action-1 is complete.							





2021R2

© 2021 ANSYS, Inc.

### Watertight Meshing – Sizing Functions Improvements

Yes

- 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 & Pr	oximity			
Ignore Proximity Across Objects?	Yes				
Curvature Normal Angle [deg]	18				
Cells Pe					C. A.
Scope F				MAS	
			AXA	TATA	
Separat					
+ Adv			XXXX		
			XXXXX		
			14		***
	THAY &				XXXXXXXX
	HANK I				
	XXXXX	XXXX	XX		

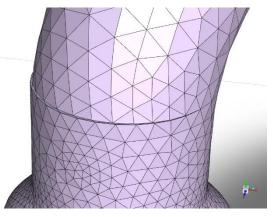
No (default)

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

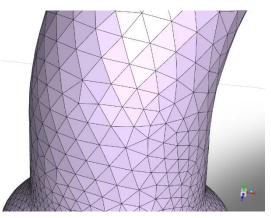
- Advanced Options		
Check Self-Intersection?	yes	•
Smooth Folded Faces/Repair Free Nodes Limit	10	
Auto Assign Zone Types?	yes	•
Invoke Quality Improve?	no	*
Remove Steps?	yes	•
Max Step Width [mm]	0.7	

In added Surface Mesh Improve task

Improve Surface Mesh 🕜		
Face Quality Limit	0.7	
<ul> <li>Advanced Options</li> </ul>		
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



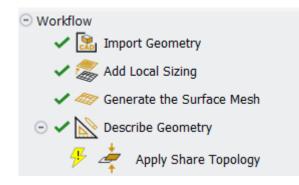
Improve testlyp, Maaxs kkew -00924



### 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.

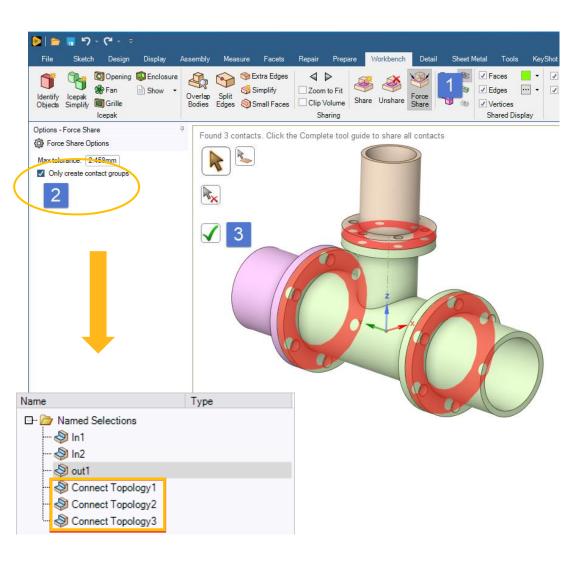




<u>Note</u> : "Ignore proximity across objects" option strongly recommended



### WTM – Interface Connect – Automatic Using Connect Topology



Apply Share Topology	
Max Gap Distance [mm]	0 Mark Gaps
Share Topology Method	Interface Connect
- Advanced Options	
Interface Labels Selection Method	Automatic - Using Connect Topology 💌
Will You Set Up Periodic Boundaries?	no 💌
Initial Relative Join/Stitch Tolerance	0.1
Join Tolerance Increment	0.1
Rename Internals Based on Body Names?	yes

- 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

Max Gap Distance [mm]	0 Mark Gaps
Share Topology Method	Interface Connect 🔹
<ul> <li>Advanced Options</li> </ul>	
Interface Labels Selection Method	Manual
Filter Text   [2/5] Filter Text	₹, ₹, ⊙
contact_c	
contact_t in1	
in2	
in2 ou	
ou	no
ou Will You Set Up Periodic Boundaries?	no 💌 0.1
ou Will You Set Up Periodic Boundaries? Initial Relative Join/Stitch Tolerance	(
	0.1

- 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

apply Share Topology	
Max Gap Distance [mm]	0 Mark Gaps
Share Topology Method	Interface Connect
- Advanced Options	
Interface Labels Selection Method	Automatic
Will You Set Up Periodic Boundaries?	no 💌
Initial Relative Join/Stitch Tolerance	0.1
Join Tolerance Increment	0.1
Rename Internals Based on Body Names?	yes 💌



### Fault-Tolerant Meshing Part Replacement

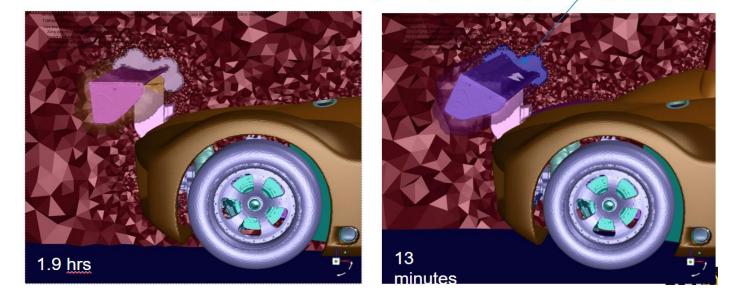
Efficiently add, remove or replace geometry objects of your CAD model without having to re-mesh the complete model

- 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 Offset-construction

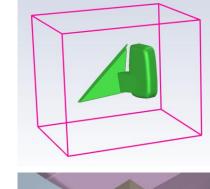


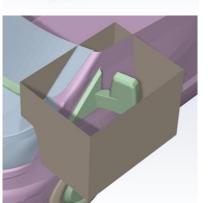


### FTM – Add Offset surface/Box to create construction surface

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

lame		construction	-surface-1
Method		Box	•
Coordinate Spe	cification Method	Ratio relativ	e to geometry size 🔹 🔻
Select By:		object	~
Filter Tex	t 🔻 [1/24] (	Filter Text	_ =₀ =₂ =₂ ⊙
headlamp_r			A
hoods hoods u			
mirror_l			-
Box Paramete	ers		_
X Min	0.5	X Max	0.5
	0.5	Y Max	0.5
Y Min			





			flow through the object temal and internal flow Options	
tify Construction Surfaces		Identify con:	struction surfaces?	Yes
ne	construction-surface	-2		
hod	Offset Surface	-		
ect By:	object	•		
Filter Text Tilter	Text 🗾			
ck_door dy lle lle_side adlamp_l adlamp_r ods ods_u rror_l		*		11
eaturing Size [mm]	5			
ght [mm]	25			
Update Can	ncel *			
ofosturo cizo : cizo c	of curfaco m	ach	ALA NELLE	

Flow Type:

External flow around object

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



No

© 2021 ANSYS, Inc.

Ider

Met

ba gri gri hea hea

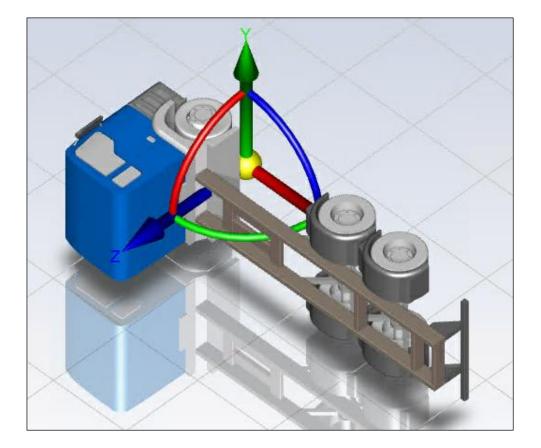
### **FTM:** Append CAD and Part Management Transformations

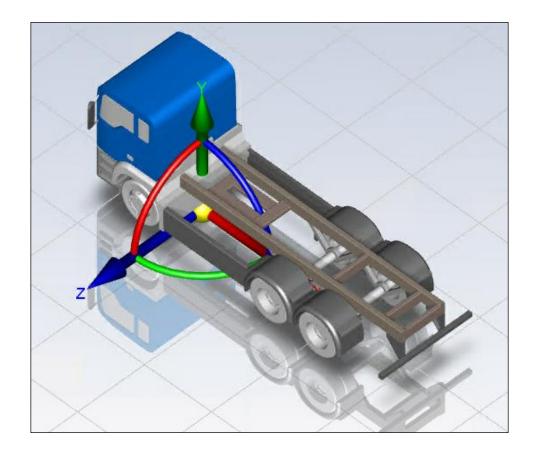
□ This feature facilitates transforming cad objects by rotating/translating about local or global coordinate system.

Filter Text		F	Filter Text			
🖂 🗖 Meshing Mod		e	🕘 🗖 Meshing Model			
🖸 🔲 🗂 truck	Q		🗩 🔲 🗂 truck			
<ul> <li>         •          •          •</li></ul>	Restore to Cad Model		💿 🔲 🍣 truck [1]			
	Create Transform		Meshing Operati	ions [-1]		
	Hide Others		⊙ □ 🎋 Transform-1			
			🗌 🗳 truck [1]		•	
		Transform (Trans	form-1)			
		Coordinate System		🔘 Local 🖲 Global		
			x	Y	Z	
		Rotate [degrees]	-90	) [0	0	
		Translate [mm]	0	0	0	
		( Apply	Transform	Unde	o Transform	

### FTM: Part Management Transformations

□ Rotating by –90 degrees about global X-axis







©2021 ANSYS, Inc.

#### 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	<b>Percentage Improvement</b> (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).

1	⊖ ⊞ Solid					
	solid-1 (solid, id=2 solid-2 (solid, id=2	Info				
	<ul> <li>➡ solid-3 (solid, id=2</li> <li>➡ solid-4 (solid, id=2</li> <li>➡ Boundary Conditions</li> <li>Mesh Interfaces</li> </ul>	Copy to Clipboa Import From File Export To File				
1	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
1	solid-l	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.

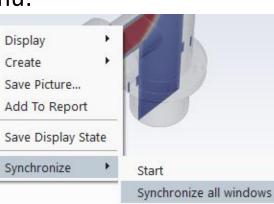
	<021R2
<ul> <li>Inlet         pres_inlet_1         </li> <li>Internal         interiorsolid_board         interiorsolid_heatsink         </li> <li>Outlet         pres_outlet_1         </li> <li>Wall         in-1-wall_board_side-solid_board         out-1-wall_board_side-solid_board         wall_board-wall_board_side-solid_board         wall_board-wall_board_side-solid_board         wall_board-wall_board_side-solid_board-solid         wall_board-wall_board_side-solid_board-solid         wall_heatsink         wall_heatsink         wall_heatsink-shadow         wall_heatsink-sull_board_solid_heatsink-s         wall_heatsource         solid_board_solid_heatsink-s      </li> </ul>	Options  Multiple Cell Zones  Rename Face Zones  Rename Cell Zones  Draw Default Mesh  On Selected Face Zones  Renaming By: Adjacency Wildcard Fluent Naming Convention Add Suffix and/or Prefix From wall*
wall_heatsource wall_outer	
	<ul> <li>Inlet         pres_inlet_1         </li> <li>Internal         interiorsolid_board         interiorsolid_heatsink         </li> <li>Outlet         pres_outlet_1         </li> <li>Wall         in1wall_board_side-solid_board         out-1-wall_board_side-solid_board         wall_board         wall_board-side-solid_board         wall_board-wall_board_side-solid_board-solid_moard-solid_moard-solid_moard-solid_moard-solid_moard-solid_board         wall_board-wall_board_side-solid_board-solid_moard-solid_moard-solid_moard-solid_moard-solid_moard-solid_moard-solid_moard-solid_moard-solid_moard-solid_</li></ul>

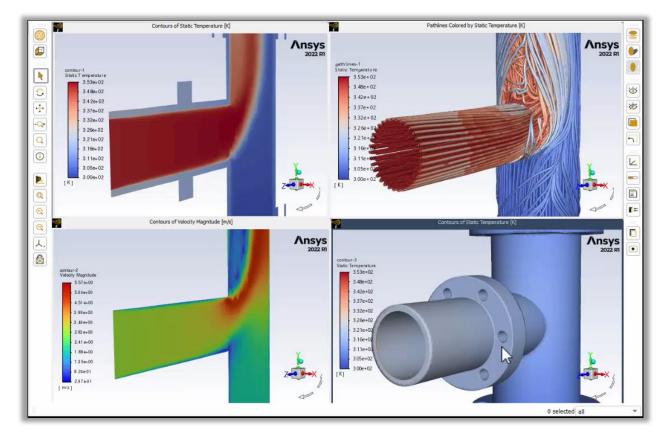
### View Synchronization

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:







Video showing viewport synchronization



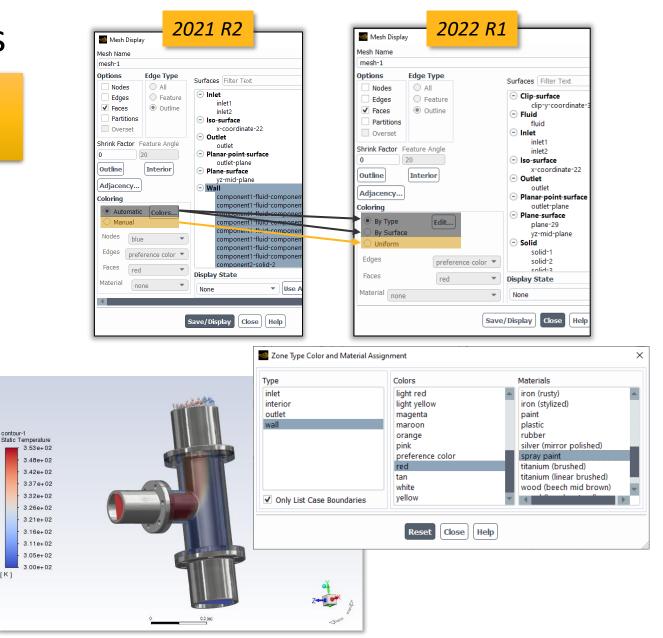
### **Mesh Rendering Enhancements**

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

 $\Rightarrow$  Removes requirement that same material must be used for all surfaces in a mesh display object

Manual renamed to Uniform \_



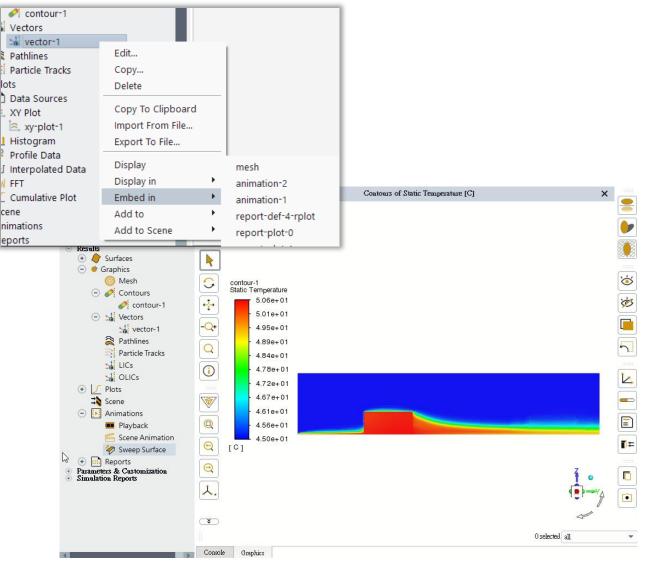


LK1

### 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

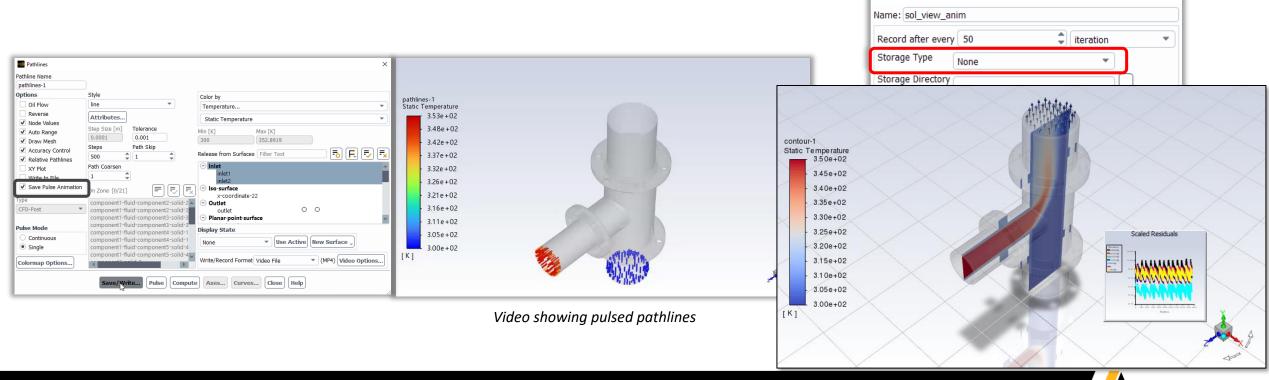
#### Recordable pathline animations

 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

Animation Definition

4

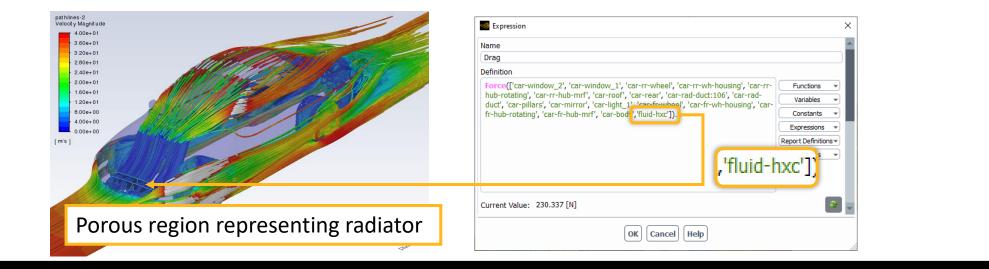


### Expressions Enhancements

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

Expression		:
Name tavg_y200mm Definition		
Average Static Temperature, ['plane-2	Extense State Tensentare 139+02 140+02 140+02 137+02 137+02 137+02 137+02 137+02 137+02 120+02 120+02 151+02 111+02 151+02	Functions         Variables         Constants         Expressions         Report Definitions         Locations
Current Value: 320.171 [K]	(K) 300+02	2





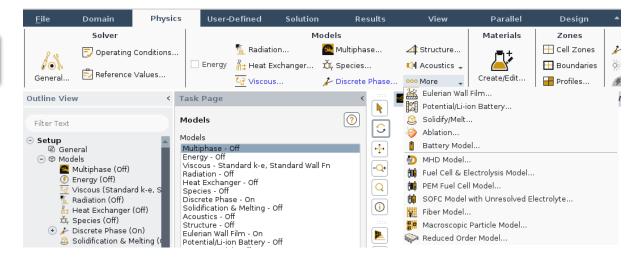
© 2021 ANSYS, Inc.

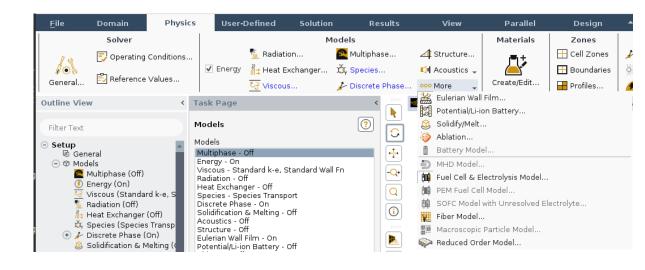
#### Fluent Add-on Model 新模組

### Exposure of Add-On Modules in Ribbon

#### Easier access to add-on modules from GUI

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



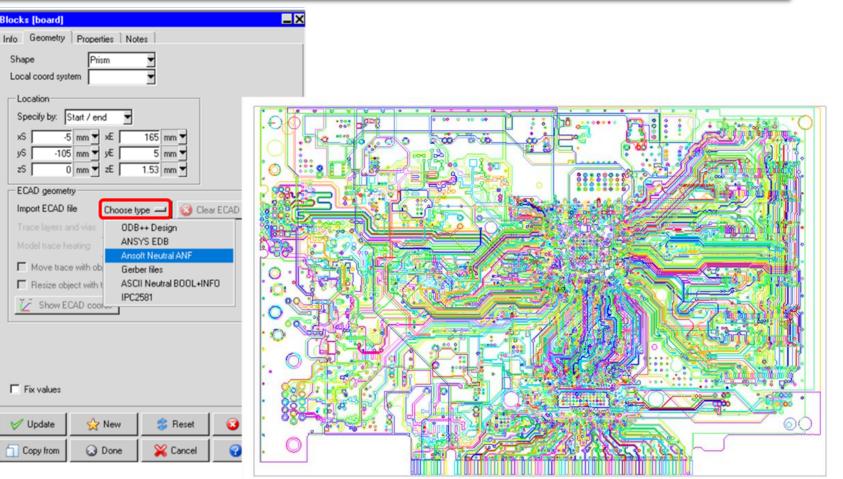




### Thermal Analysis of Printed Circuit Boards

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 dielectric layers, generated in Ansys Icepak)
  - Board configuration file from <u>Ansys Icepak</u> ("board\_config.dat")
  - Metal fraction information from <u>Ansys Icepak</u> (.cond file)
  - Power profiles from Ansys Icepak (.prof files)

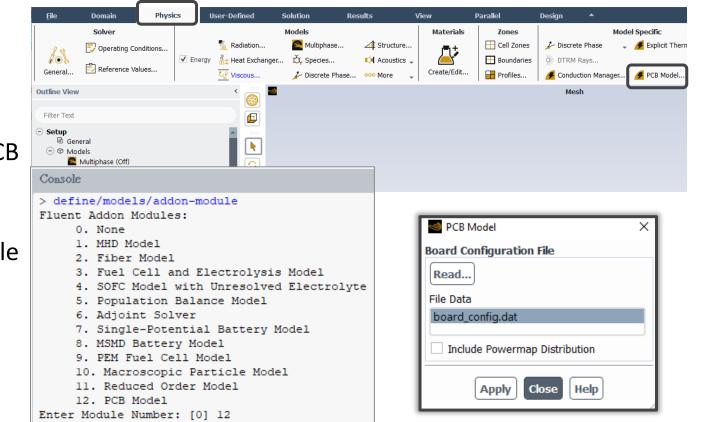




### IcePak → Fluent Workflow for Printed Circuit Boards

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.

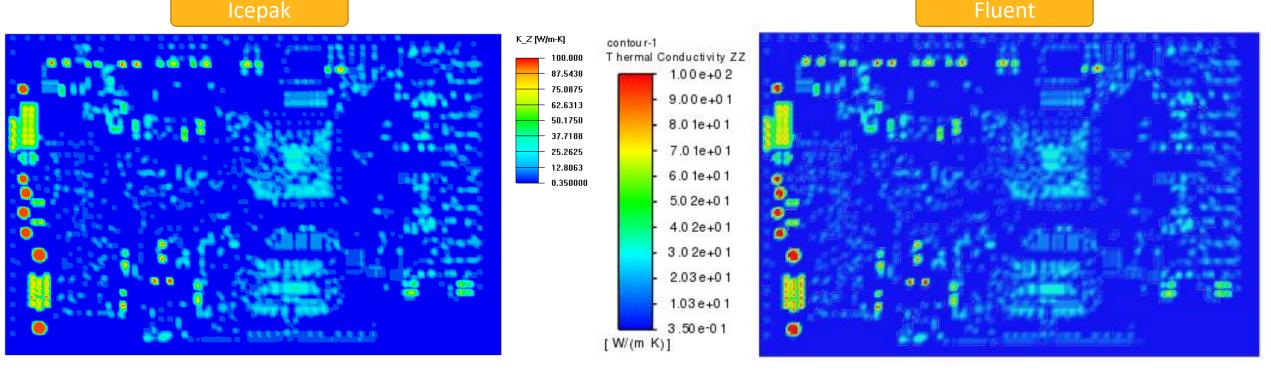
🌳 Create/Edit Materials		
Varne	🎇 Orthotropic Conductivity	× erials by
subst	Direction 0 Components	P
Themical Formula	X Y Z	hical Formula
		ent Database
	Direction 1 Components X Y Z	A MDS Database
		efined Database
Properties	0 1 0	
Ser-Defined Function	s $\times \frac{10 \left[ W / (m K) \right]}{1}$	<b>t</b>
	а 	Edit
ortho_local_x::libudf_cond		t
ortho_local_y::libudf_cond	71 [W/(m K)]	
ortho_local_z::libudf_cond conductivity_local_x::libudf_co	d d	▼ Edit.
conductivity_local_y::libudf_co		t
conductivity_local_z::libudf_co	nd	
conductivity_x::libudf_cond	y 2 [W/(m K)]	
conductivity_y::libudf_cond	rd.	▼ Edit
conductivity_z::libudf_cond cp::libudf_cond		
density::libudf_cond		
	OK Cancel Help	
OK Cancel He	ר — <u>— —</u> —	



### 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

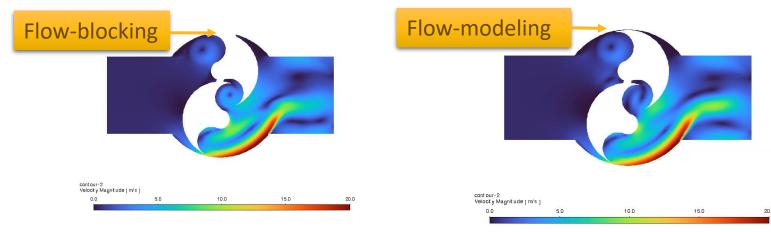
• Note that these orthotropic thermal conductivities are computed in the global coordinate system.



### 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



ap Definition	
Gap Name	
gap_1	Exclude Cell Zones
Face Zones [2/4]	=_ =_
component2-solid-2	
component3-solid-3	
component4-solid-1	
component5-solid-4	
Proximity Threshold [n	n] Gap Type
Proximity Threshold [n 0	Flow-Modeling
0	Flow-Modeling  Flow-Blocking
0 Method	Flow-Modeling
Method Sponge Layer	Flow-Modeling  Flow-Blocking Flow-Modeling
0 Method Sponge Layer Resistance Type	Flow-Modeling Flow-Blocking Flow-Modeling Gap Reynolds Number
0 Method	Flow-Modeling  Flow-Blocking Flow-Modeling

• User-Defined Source method with DEFINE\_GAP\_MODEL\_SOURCE





