

What's New in Ansys 2025R1 Fluent Webinar

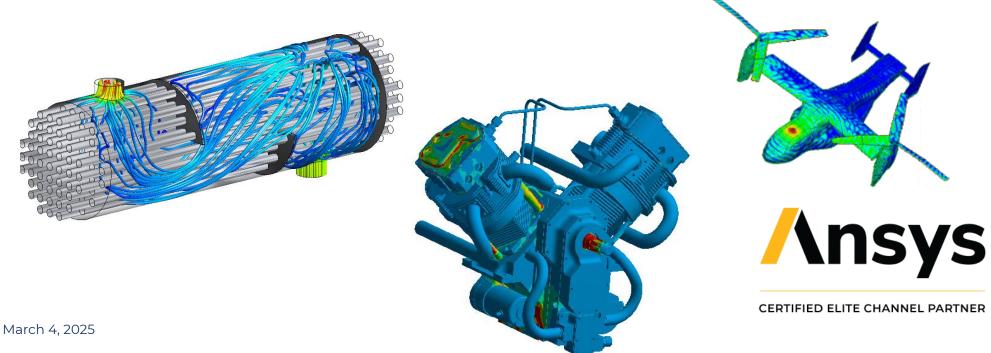
Adam Kroll– DRD's Fluids Team Lead March 4, 2025



- Fluent Native GPU Solver
- Fluent User Experience
 - Desktop Interface
 - Web Interface
- Fluent Meshing
- Fluent CPU Solver Physics Improvements

Mission Statement

DRD Technology helps engineering teams accelerate product development. With in-house expertise spanning the entire range of physics, we ensure customers succeed when using Ansys simulation tools for virtual prototyping and design verification.



Technical Support Contact Coordinates

SIMULATION PRODUCTS CONSULTING TRAINING COURSES ~ ABOUT ~ CONTACT US RESOURCES TECHNOLOGY (918) 743-3013 x1 support@drd.com Submit a Technical Support Question Or through our website at www.drd.com As part of DRD's customer services, we encourage you First name¹ Last name to send us questions and development requests regarding the software products we represent. The guestion/enhancement will be emailed immediately to Email^{*} Phone number the technical support personnel at DRD.

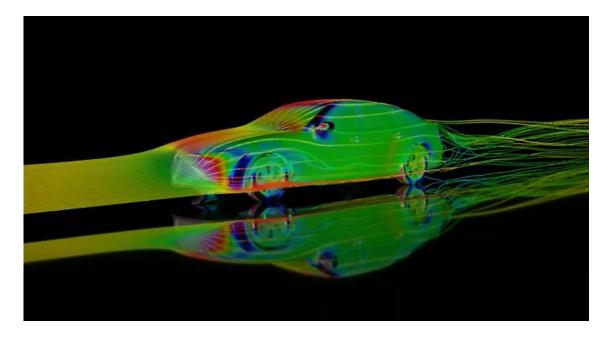
For more than five years, I have worked closely with DRD Technology to execute tactical and strategic initiatives here at EaglePicher due to our unprecedented growth. We've been very happy with DRD and will continue to work with them as our business partner for using Ansys tools effectively and efficiently.

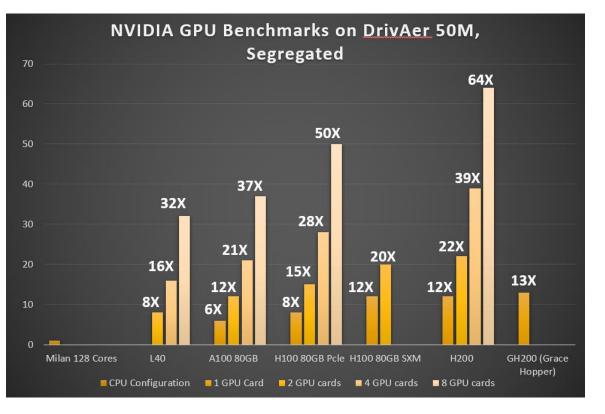
> Doug Austin Director of Research and Development EaglePicher™ Technologies, LLC

Support:

Fluent Native GPU Solver

GPU Solver Performance





Multi-GPU CFD Solver: Release Timeline

R

2023 **R2**

20**22 RI**

20**22 R2**

β releases with support for :

- Single/multi-GPU (shared / distributed memory)
- Steady & transient simulations
- Incompressible & subsonic compressible flows
- All mesh types
- Ideal Gas and Materials with variable properties
- Turbulence: standard k-e, SST, GEKO, RKE, SBES
- Solid conduction and CHT
- Porous media
- Transient scale-resolving simulations
- Non-conformal mesh interfaces
- Moving walls & Moving Reference Frame
- Windows and Linux
- ...

First full releases :

- Various numerics improvements
- Enhanced RANS
- Enhanced LES numerics
- Species transport
- Non-stiff reacting flows
- Compressible flows
- Sliding meshes
- Improved UX and launcher enhancement
- .

Advanced physics and models :

- Coupled solver
- Lagrangian particles
- Reacting flows
- Acoustics
- (<u>.</u>...

024 **RI**

024 **R2**

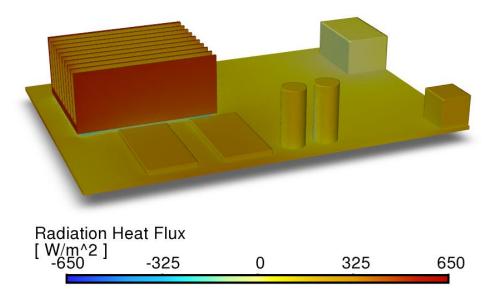
7

AMDE

NIDIA

Surface-to-Surface (S2S) Radiation Model

- S2S model now supported with GPU
 - Simplified workflow on GPU compared to CPU
 - No need to calculate the view factors explicitly:
 - Enable the radiation model •
 - Define the emissivity on the various surfaces •
 - Initialize and calculate the solution
 - Support for Periodic BC together with radiation

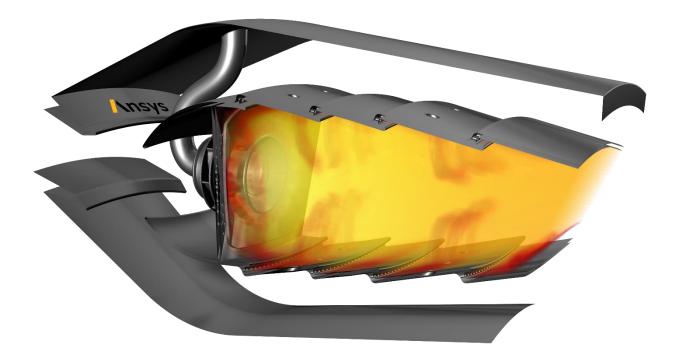


DRD

Flamelet Generated Manifold (FGM)

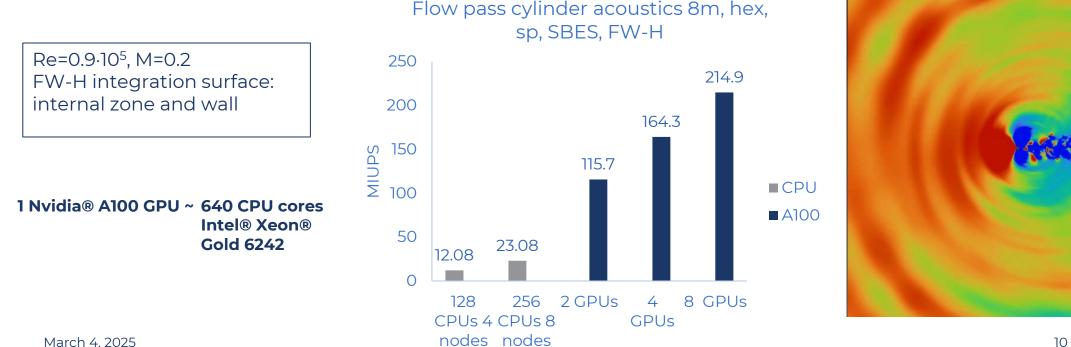
• Support for adiabatic FGM

- Assumes adiabatic flame temperature
- Pre-tabulated detailed chemistry
- Four transport variables:
 - Mixture fraction [f]
 - Variance of mixture fraction [f"]
 - Progress Variable [C]
 - Variance of progress variable [C"]
- Non-adiabatic FGM (Beta)
 - One additional transport equation
 - Total (sensible + combustion) enthalpy
 - Supported Wall Boundary Conditions
 - Heat flux and adiabatic
 - Specified temperature
 - Coupled walls
 - Support for Discrete Ordinates (DO) radiation and Discrete Phase Model (DPM) coupling
 - Conjugate Heat Transfer (CHT) is not yet supported
 March 4, 2025

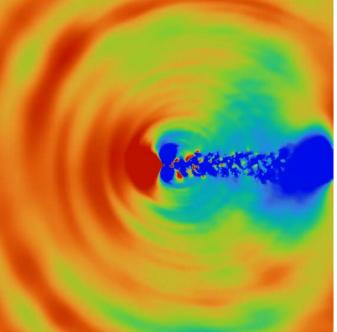


Ffowcs Williams & Hawkings (FWH) (Acoustics)

- Release of hybrid GPU (flow) / CPU (sound signal) solution •
- Similar workflow as with CPU solver
 - Requires export of asd and ard files •

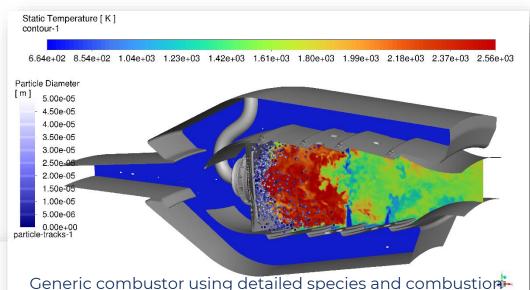


dp/dt, far-field BC



Discrete Phase Model: Two-Way Coupling for Mass-Transfer

- Sources for detailed species transport equations
- Mass transfer from particles to coupled gas phase supports single-component evaporating / boiling particles

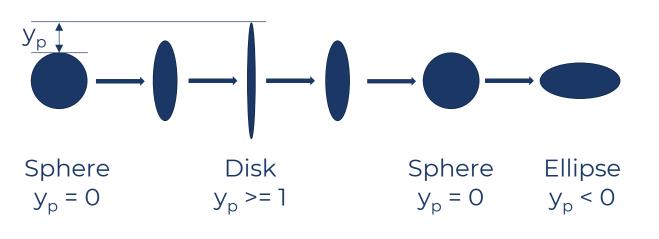


<section-header><list-item><list-item>

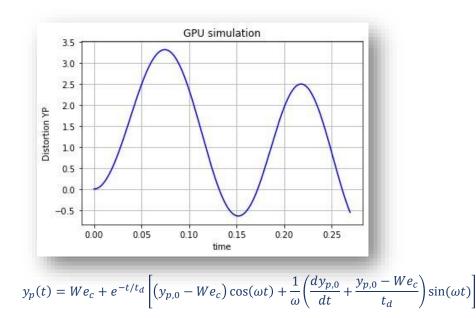
March 4. 2025

Discrete Phase Model: Dynamic Drag Law

- Solving equation for particle distortion assuming a damped spring-mass system
- New drag option available for particles undergoing droplet breakup
 - Injection specific
 - Scales drag coefficient as function of the particle distortion, $y_{\rm p}$



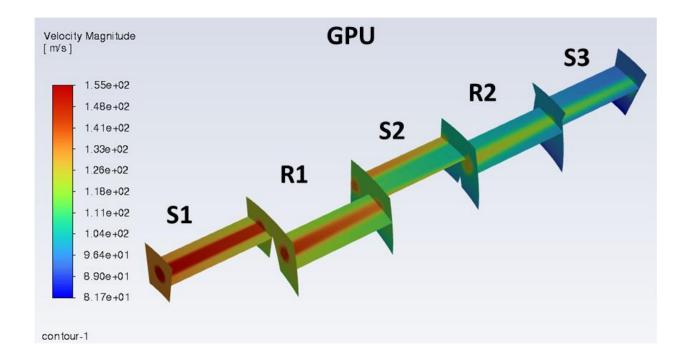
March 4, 2025



Set Injection Properties				
Injection Name	Injection Type		Number of Streams	
injection-0	group		10	¢
Reference Frame				
global 👻				
Particle Type				
O Massless Inert Droplet 				
Material Diam	eter Distribution			
anthracite 🔹 linea				
Evaporating Species				
				
Point Properties Physical Models Turbule	nt Dispersion			
Momentum Exchange Drag Law dynamic-drag dynamic-drag dynamic-drag				
	OK File C	Cancel Help		

Turbomachinery

- Periodic repeat option now supported with GPU
 - Pitch change and other turbo interface models will be supported in a future release



Material Properties: Specific Heat

• Specific heat capacity can now depend on temperature when using the compressible ideal gas density formulation

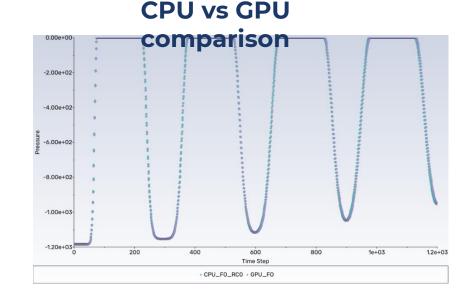
🧐 Create/Edit Materials		×
Name	Material Type	Order Materials by
air	fluid 🗸	• Name
Chemical Formula	User-Defined Fluid Materials	Chemical Formula
	air 🔹	Fluent Database
	Mixture	
	none 🔻	GRANTA MDS Database
		User-Defined Database
Properties		
	m³] ideal-gas 🔹 Edit	
Cp (Specific Heat) []/(kg	K)] piecewise-polynomial Edit	
Thermal Conductivity [W/(m	K)] kinetic-theory	
Viscosity [kg/(m	s)] kinetic-theory	
Cha	nge/Create Delete Close Help	

DRD

Material Properties: Compressible Liquid Water Hammer

Pipe flow with steady state profiles is suddenly closed at one end, which produces water hammer effect.

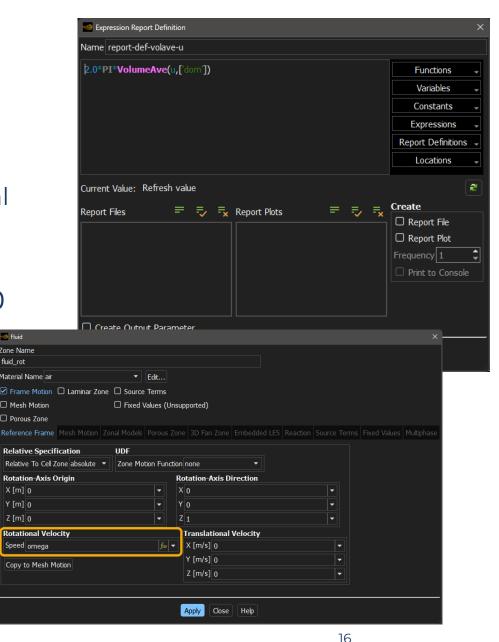
- Density is assumed as a function of pressure alone using Tait's law
- Allows both compression and expansion
- Useful from physical and numerical perspective
- Current Limitations:
 - Supported only for single phase flows
 - Potentially less stable with second order time



Area weighted average pressure at the outlet

Expanded Expression Support

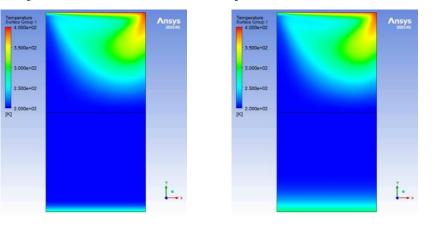
- Run time monitoring of quantities of interest defined as expressions and create plots
- Support for flow variables (u, v, w, P, T) and mathematical operators like sin, cos, exp, sqrt
 - Examples: sqrt(u^2 + v^2+w^2) Time*Sum(T, ['fluid']) * 2 [kg/(m^3*s)] AreaAve(u^2+v^2+w^2, ['pressure-outlet 7'])
- Moving reference frames can be customized with expressions
 - Time-dependent expressions for rotational speed, axis origin, direction and translational velocities
 - No spatially varying expressions
 - Ability to mix and match inputs between expressions and constants
 - Example: 3.0*sin(2.0*PI*t/1[s])*1[rad s^-1]



Transient Numerics

- Solid time-stepping
 - Independent timestep for CHT solids
 - Reduces calculation time when only final state is of interest

$$dt_f = dt_s = 10$$
 $dt_f = 10, dt_s = 100$



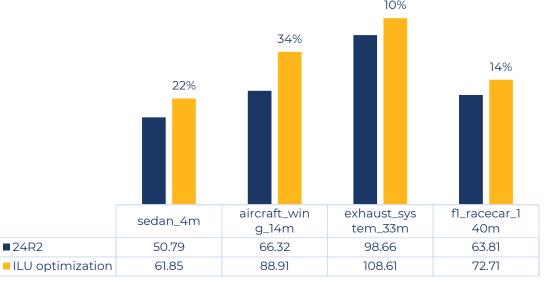
Check Case		Preview Mesh Motion	
Fime Advancement			
Туре		Method	
Fixed	•	User-Specified	
Parameters			
Number of Time Steps		Time Step Size [s]	
1000	¢	10	
Max Iterations/Time Step		Reporting Interval	
20	¢	1	
Options			
Report Simulation Status			
Solid Time Stepping			
🗹 Specify Solid Time Step	Size		
Method		Time Step Size [s]	
User-Specified	-	100	

- Bounded Second Order now available for Transient Formulation
 - Improves stability for large time step size
 - Limiter-in-time for second order transient schemes (GPU only)
 - Improves boundedness for energy

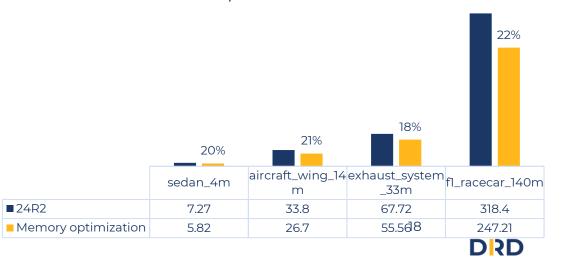
Energy	
Second Order Upwind	-
Pseudo Time Method	
Off	~
Transient Formulation	
Bounded Second Order Implicit	•
🗹 Use limiter in time	
Reduced Rank Extrapolation	Options

Coupled Solver Improved Performance Coupled solver performance optimization, MIUPS

- Optimizations provide improved performance and reduced memory for the coupled GPU solver
- Calculation time reduces by up to 30%
- Memory consumption decreases by up to
 - 20% in single precision
 - 25% in double precision

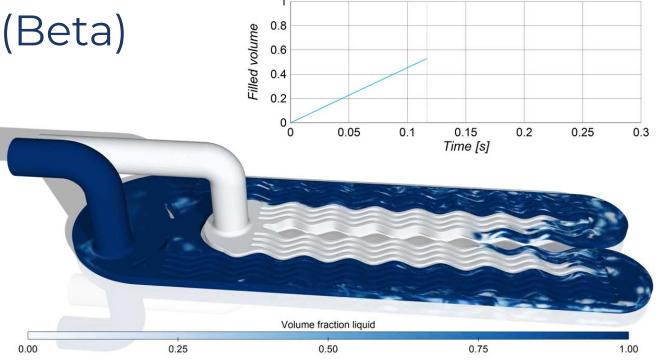


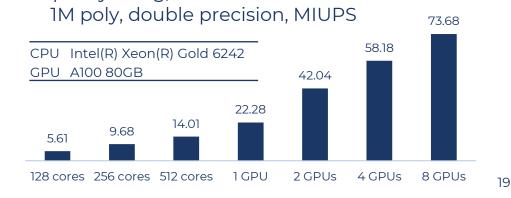
Memory usage (GB) optimization of single precision



Volume of Fluid (VOF) (Beta)

- VoF method now available on GPU
 - Excellent performance and scalability, with 2 x A100 GPU's ~10x faster than 128 CPU cores
 - Most calculations on GPU can use single precision, while CPU requires double precision
 - Reduces memory cost of GPU by a factor of 2 and improves speed up to 40%
 - GPU solver often works with higher URFs compared to CPU solver
 - Reduces required number of iterations

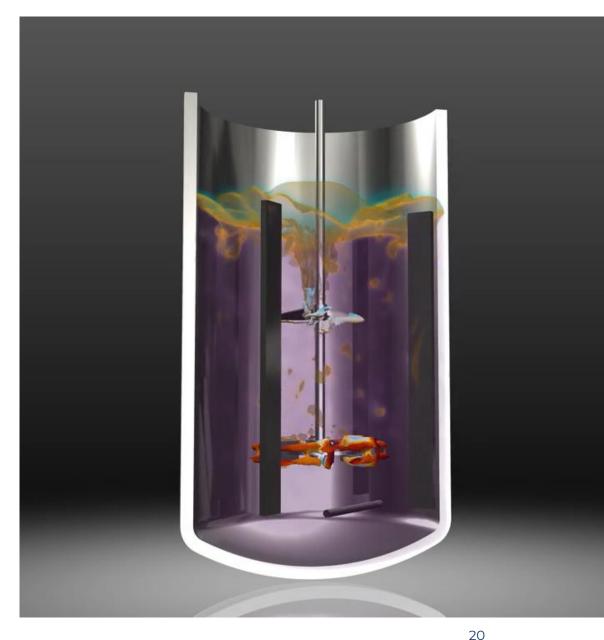




Capillary filling, VOF + surface tension

VOF (Beta) Limitations

- Support for sliding meshes, laminar flow and RANS models
- Limitations of initial beta release:
 - Only isothermal calculations (energy equation must be disabled)
 - Constant material properties
 - Not compatible with other models like
 - Species transport
 - Discrete Phase Model (DPM)
 - Large Eddy Simulation (LES)
 - Only implicit VOF with Compressive scheme



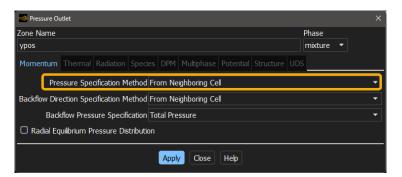
VOF (Beta) Other

• Surface tension for VOF

- Continuum surface force model with constant surface tension coefficient
- Optional boundary-zone-based wall adhesion
- Modified Body Force Weighted
- From Neighboring Cell
 - Volume Fraction
 - Pressure

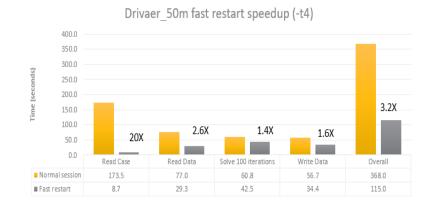
Solution Methods	?
Pressure-Velocity Coupling	
Scheme	
SIMPLEC	-
Skewness Correction	
0	¢
Spatial Discretization	
Gradient	
Least Squares Cell Based	-
Pressure	
Modified Body Force Weighted	•
Momentum	
Second Order Upwind	-
Volume Fraction	
Compressive	-

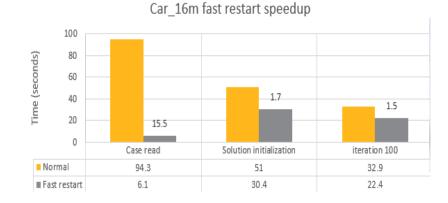


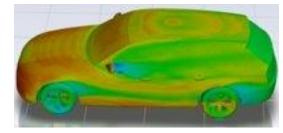


Fast Restart/Native GPU Post-Processing (Beta)

- Reduce the case/data loading time when restart the GPU solver calculation and enabling native GPU post-processing
 - Command line option: -lite
 - Unsupported objects like iso-surfaces are disabled
- Work for both first-time and restart runs
- Once saved with lite, cas/dat can only be read in this mode







March 4, 2025

DRD

Fluent User Experience: Desktop Interface

Transient Post-Processing

	<u>F</u> ile	Solution	Results Vi	w Paralle	l De	sign Pa	arametric	Learning and Sup	port	Transient	Postprocessing	• •
			Timestep Selection	n Tra	nsient Disp	olay	Transi	ent Animation	s	ave Reports	Result Comparison	
	✓ Transient Postprocessing	<u>í</u> E.		Display		뜸 Create.	🔲 Playback	5	Monitor			
		Timestep Selector	C Reverse	► Play	🖒 Reset	Stop Ci	reating Animation	•	Stop Writing Files	Compare		

1 192-00 1 192-00 1 192-00 1 092-00 1 092-00 1 092-00 1 092-00 5 78e 61 9 56e 61 denety contaut

🥌 Timestep Selector		×
Filter 🚽 Filter Text		
Time-Step	Flow-Time	File-Name
10	0.001000	box-1-00010.dat.h5
20	0.002000	box-1-00020.dat.h5
30	0.003000	box-1-00030.dat.h5
40	0.004000	box-1-00040.dat.h5
50	0.005000	box-1-00050.dat.h5
	OK Cancel Help)

- Transient post-processing of multiple time steps within Fluent
 - Only for static meshes in the first release, recommended for small cases
 - First release can only read the full Fluent *.dat* files, which requires more time compared to reduced data sets in CFD Post or EnSight
- Read the last dat file of a sequence to access all previous steps with the Timestep Selector
- Create transient animations after the calculation is finished

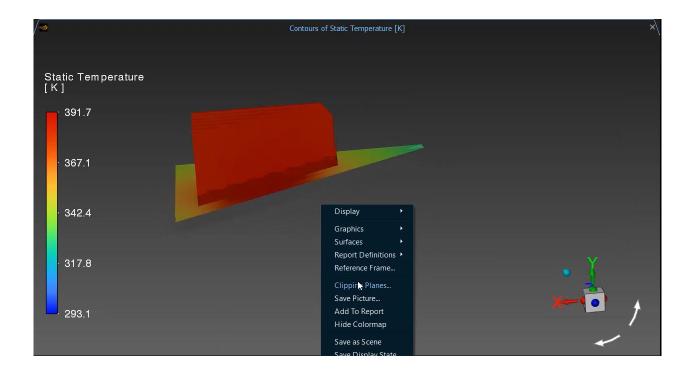
24

DSD

March 4, 2025

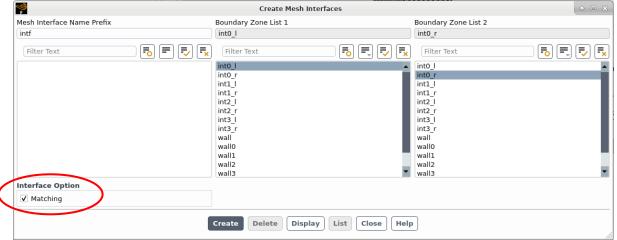
Clipping Plane

- Clipping plane available in context menu of a graphics window
 - Visual clip only, not stored in an object
 - Use clip surface objects for persistent clips
 - Limited to one active clipping plane
 - Interactive handle to change position/orientation



Non-Conformal Interfaces

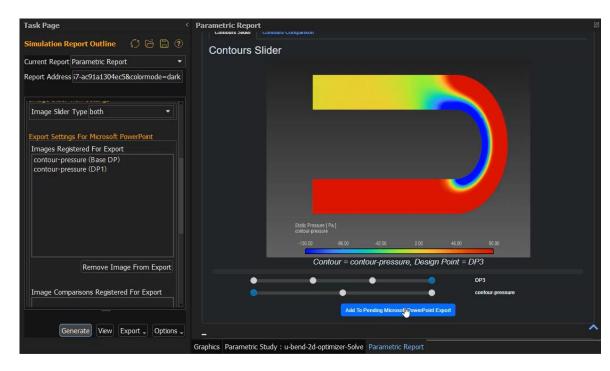
- New Matching option for 1-to-1 interfaces
 - Forces zero area non-overlapping boundaries
 - Cannot pair interface boundary participating in matching interface in another 1-to-1 interface
- Cell zone pairing for 1-to-1 interfaces
 - Allows interface creation by selecting cell zones
 - All boundaries of selected cell zones (of type wall and interface) participate in auto pairing
 - Can mix and match face and cell zones
- NCI exclusion list
 - Allows to specify boundary pairs not to be paired by auto pairing (TUI and GUI)
 - /define/mesh-interfaces/autooptions/set-exclusion-pairs

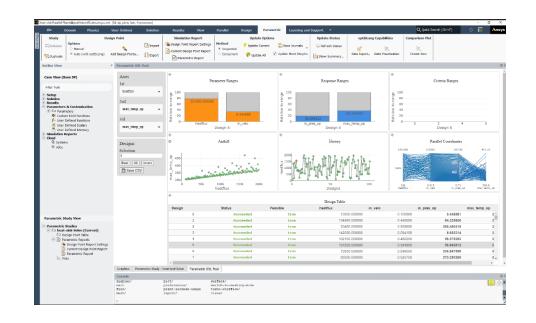


*	Mesh I	nterfaces			•
Zones		Mesh Interface	s		
Filter Text		Filter Text) = =
⊖ Fluid					
fluid0 [0]					
fluid1 [0]					
fluid2 [0]					
Interface					
int0_1 [0]					
int0_r [0] int1 [0]					
int1_r [0]					
int2_1 [0]					
int2_r [0]					
ints_i [0]					
int3_r [0]					
Solid					
solid0 [0]					
solid1 [0]					
Wall wall0 [0]					
wallo [0] wall1 [0]					
wall2 [0]					
wall3 [0]					
wall4 [0]					
wall5 [0]					
Interface Name Prefix intf		Edit	List	Delete	Display
Create	Ianual Create	Preview M	esh Motion	Optio	ons
	Cle	se Help			

Parametric Workflow

- Parametric Report in PPT format
 - Based on the DP results objects
 - DP-specific images and comparisons of DPs
 - Exported content defined after the parametric run to keep the size of the report manageable





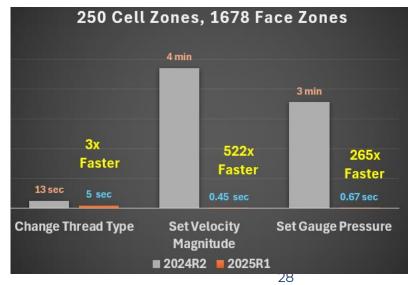
User Interface Performance

- Performance is dramatically improved for cases with many zones
 - Test cases show UI operations are 3x to 500x faster
 - Overall setup time reduced by 4x to 15x
 - 1-to-1 auto-pairing for Non-Conformal Interfaces:

Cores	Boundaries	Interfaces	24R1 [mins]	24R2 [mins]	25R1 [mins]	Speedup 24R1 > 25R1
128	332	660	97	32	16	6x
64	127	369	71	14	7	10x

• Performance also improved for 1-to-1 interface deletion





Graphics Performance

- New preference to improve performance of rendering large surfaces with millions of triangles
 - Large surfaces are splits into multiple shells of 160k facets
 - Need to redisplay after setting the preference
 - Applies to both Meshing and Solve modes
- Improved interactivity: more responsive panning, zooming, and rotating
 - Best for low number of zones with high facet count
 - Example case: 1 zone, 150M faces
 - Rotate: FPS 0-1 (24R2) vs 55-65 (25R1)
 - Zoom: FPS 0-1 (24R2) vs 15-25 (25R1)
 - Pan: FPS 0-1(24R2) vs 55-65 (25R1)
 - Initial mesh display time: 760s (24R2) vs 350s (25R1)

General Appearance	Automatically embed domain outline (mesh display) upon initialization	D
Graphics Meshing Workflow	· Performance	
Navigation	Optimize for	Automa
Turbo Workflow	- Optimize Input Data	
Simulation Parametric Study	Enabled - Minimum Frame Rate	
, , , , , , , , , , , , , , , , , , ,	Enabled	
	- Fast Interactive Display	
	Status	Automa
	Show faces	
	Show perimeter edges	0
	Show nodes	
	Show model silhouette	0
	Transparency	Ο
	OK Default Cancel Help	×
	scene-1	
	JES ER ST	
0		6
*		8
Q		5
BEFORE		
		[=
1.	N	
	2	
	J.	
		4
-	scene-1	2
0		6
*		2
<u>a</u>		
		5
		K
AFTER		-
Q		E
@ @		
<u>م</u> ۲.		
6		
	phics interaction	

Fluent User Experience: Web Interface

Fluent Web Interface

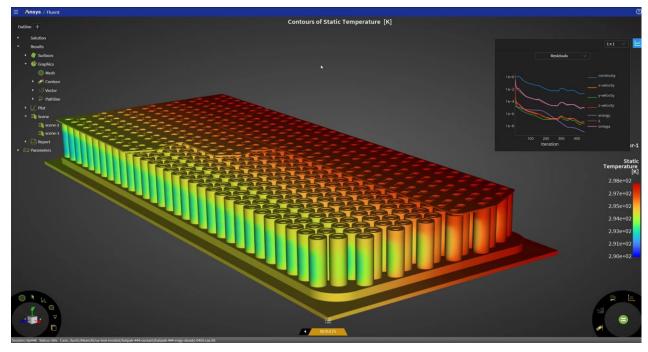
WHAT IS IT?: A single workflow interface from meshing (β) to solving to post-processing available directly in a web browser

WHO IS IT FOR?: Available to any Fluent user of any license type.

WHAT CAN IT DO?:

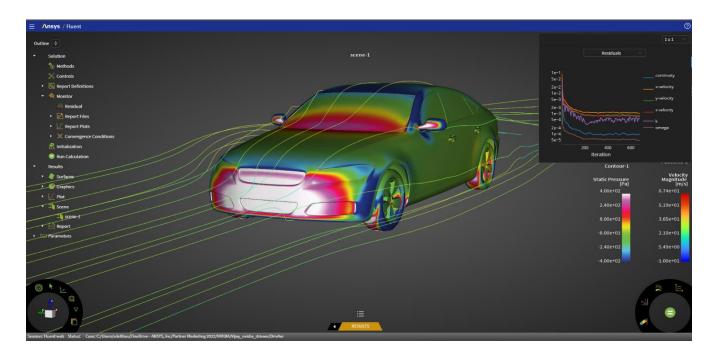
- Mesh (β), physics setup solve and post-process within a single interface
- Monitor simulation results in real-time, before the problem has been solved
- Available to connect with batch as well as interactive Fluent sessions
- Collaborate with colleagues on the same project at the same time
- Modernized UI/UX with a familiar look and feel to Fluent desktop
- While not all models are available for setup, additional models are included with each release

March 4, 2025



Additional Physics (Beta)

- Eddy dissipation, finite rate
- Partially premixed combustion with FGM
- Volume of fluids multiphase method (excluding mass transfer and reactions)
- Discrete phase model and particle post-processing
- Adjoint solver, shape optimization, parameterize and explore (parametric mesh morphing)



Fluent Meshing Watertight Workflow (Beta)

- Interactive cutting planes
- Mesh diagnostics, cell clusters
- Wildcards, improvements for large models
- FM-style shortcuts
- Desktop preferences
- Display options
- Some options require Fluent Meshing to run with graphics support and do not work when running in batch without graphics

Ξ Ansys / Fluent		0
Watertight Geometry Y 🖻 🖥 C	Mesh	
Mashing		
✓ Simport Geometry		
 ✓ Z Add Local Sizing 		
V 🌋 sensor		
Generate the Surface Mesh		
🔹 🛷 📐 Describe Geometry		
🝷 🧹 📋 Enclose Fluid Regions (Capping)		
🗸 🚡 inlet		
✓ → outlet		
Vpdate Boundaries		
Create Regions		
→ D Update Regions		
 		
Setup		
	Diagnostics Tools ×	
Results		
Parameters	Issue	
	Cell Quality	
	Measure Orthogonal V Compute	
	Average Minimum Maximum	
	0.9023735 0.1101592 0.9999006	
	Range Min Max	
	Set Max V 0.1101592 0.8	
	Mark	
	Details	
	Total 10	
	Cluster Navigation	
	Selected All	
	Count 10 List Ignore	✓ ★ ⊠
	Restore	a
	E Draw Histogram	
	MESHING	
Session: FluentWeb		

Shareable Links with Permissions

- View only, results edition only, or full access
- Accessed from web interface share icon in the upper right corner
 - Links remain active for the current session
 - Creating a link does not require an email





Share a link to this session with others via email or copy the link directly

O View Only

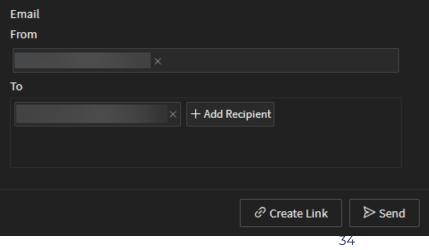
Can view all the settings and display results but not edit them. The Python Console and the file browser are not available.

Edit - Results Only

Can edit only results and report definitions. The Python Console and the file browser are not available.

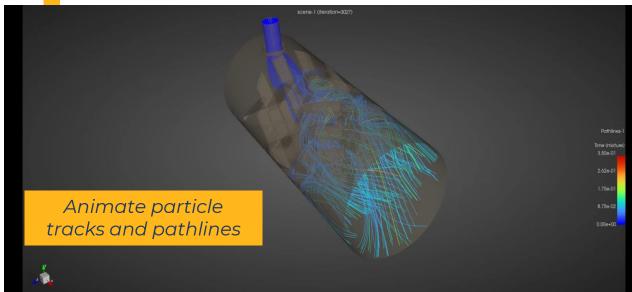
Edit

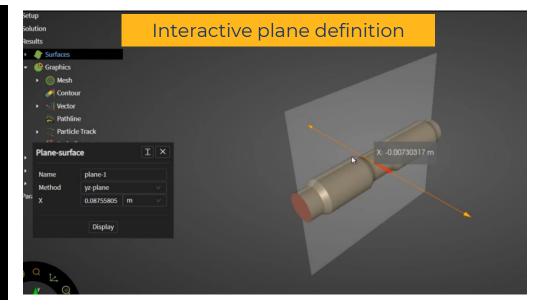
Can edit all the settings. The Python Console and the file browser are not available.

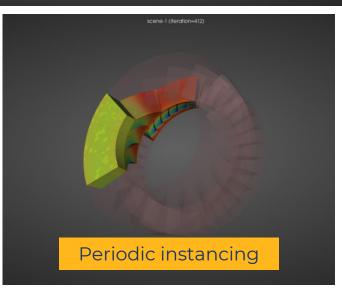


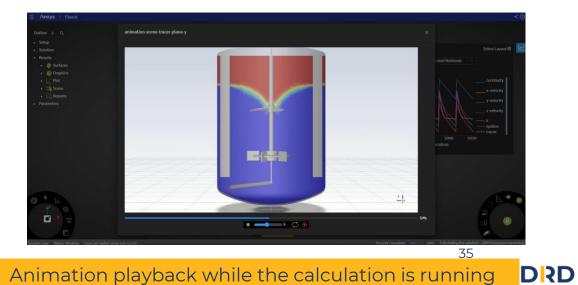
March 4, 2025

Post-Processing & Animations





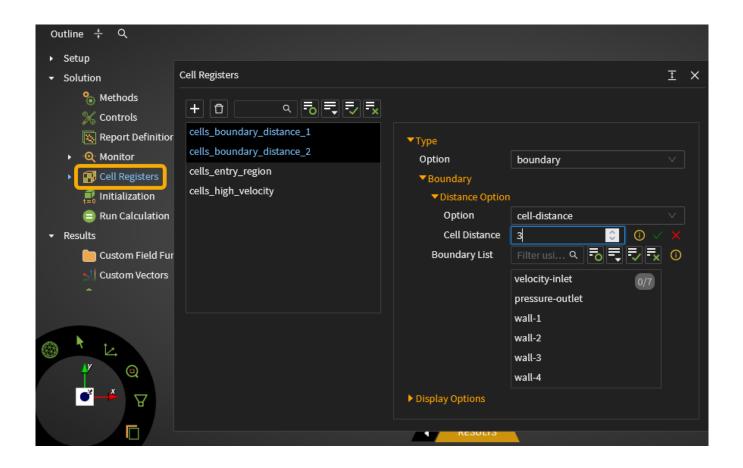




March 4, 2025

Manager Panels

- Introduction of manager panels to edit multiple objects simultaneously
- Available for almost all grouped objects in the Outline tree



Notification of Invalid or Incomplete Settings

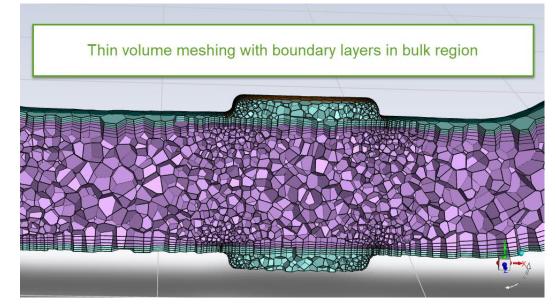
• When the settings of an object are either invalid or incomplete, a warning message inside the panel indicate what must be corrected

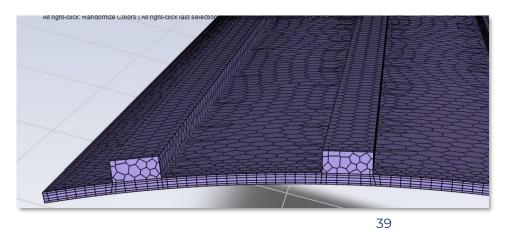
utline + Q	Contour		Ξ×
Setup			
Solution	Name	contour-1	
Results	Contours of	Static Pressure	\sim
🛅 Custom Field Functions		at is undefined	
Store Custom Vectors	Surfaces	Filter using wi ۹ 🗖	€₹
🕨 🧳 Surfaces			0/17
🝷 🕒 Graphics		velocity-inlet	
🛞 Mesh		Internal	
🕨 🥖 Contour	►	interior-1-2	
Vector		interior-2-3	
Pathline		interior-3-4	
Particle Track			
a failucie flack	Danga	No Rendering surfaces have Global Auto	
	Range	O O Pa	Clip
	▶ Options		
t L			
	Coloring		
	Colormap Op	tions	
		Display	
			RESUL

Fluent Meshing

Thin Meshing

- New capability to create layered mesh inside thin regions
 - Algorithm can automatically transition to unstructured mesh in locally-bulky regions
 - Conformal connection with all other volume fill methods supported
- Thin Meshing controls are similar to Multizone controls
 - Selection of parameters (number of layers, biasing, etc)
 - Selection of regions
 - Selection of source and target faces
 - Advanced options
- Stacked Plates option
 - For models with stacked planar plates aligned in Global X, Y, Z the "Stacked Plates" option can be used for semi-automatic assignment of source and target faces based on selection of one or a few key seed sources





Auto-Thin Meshing (Beta)

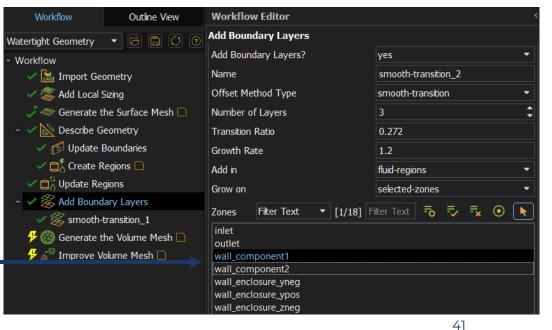
- Automatically detect and configure thin mesh regions and source/target selections
 - Targeted use cases: Solid thin plates and decomposed thin fluid regions
 - Cannot be used together with Stacked Plates
 - Additional option to specify a target cell size within the thin gap in addition to the min number of intervals
 - Separates faces automatically if needed
 - Limited to serial meshing for the thin regions

Workflow Editor	<		
Add Thin Volume Meshing Controls			
Name	thin-meshing_2		
Assign Size Using	Size 🔹		
Minimum Number of Intervals	1		
Size [mm]	0.3125		
Growth Rate	1		
Double Biasing			
Stacked Plates	Auto Control Creation		
Max Gap Size [mm]	1		
Local Unstructured Fill	Aggressive 🝷		
Regions Filter Text 💌 [4/10] F	ilter Text 🛛 🗟 🕏 🖏 🔍		
fluid solid_capacitor1 solid_capacitor2 solid_component1 solid_component2 solid_gpu solid_heat_sink solid_memory1 solid_memory2 solid_pcb			
Advanced Options			
	Draw Thin Volume Regions		
Add Thin Volume Meshing Controls Revert and Edit 💂			
	40		

Boundary Layer Usability

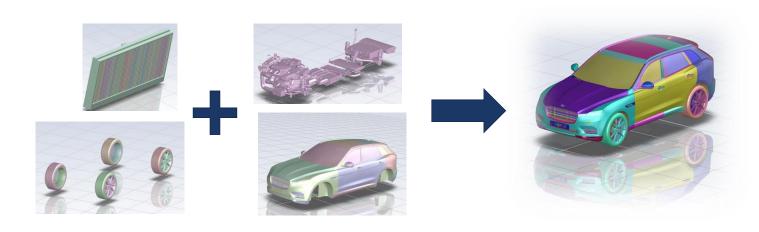
- The list of available zones/labels is filtered automatically to show only items without boundary layer definition
 - Release for WTM, beta for FTM
 - Simplifies setup for complex models

Workflow	Outline View	Workflow Editor <		
Watertight Geometry	- 6 6 0	Add Boundary Layers		
- Workflow		Add Boundary Layers?	yes 🔹	
🗸 🔝 Import Ge	ometry	Name	smooth-transition_1	
🗸 촗 Add Local	Sizing	Offset Method Type	smooth-transition 🔹	
🧳 🛷 Generate t	the Surface Mesh 🗋	Number of Layers	3	
- 🗸 📐 Describe G	Beometry	Transition Ratio	0.272	
🗸 💋 Update	Boundaries	Growth Rate	1.2	
✓ 🗖 Create F		Add in	fluid-regions 🔹	
🗸 🗖 🖓 Update Re		Grow on	selected-zones 🔹	
✓ 🛞 Add Bound	lary Layers the Volume Mesh 🗋	Zones Filter Text 🔻 [2/20]] Filter Text 🙃 🐬 🛼 💿 📐	
	/olume Mesh	utlet wall_capacitor1	ered	
			omatically for t BL definition	



Fault-Tolerant: Modular Wrapping

- For very large model it is more efficient to simplify subassemblies before meshing the full model
 - Sub-assemblies are skin-wrapped
 - Tasks updated accordingly (Generate volume mesh removed...)
 - Faster as no remesh nor improve operations done
 - Easier manipulation/inspection
 - Resulting sub-assemblies are imported into a new workflow for global wrap and volume mesh

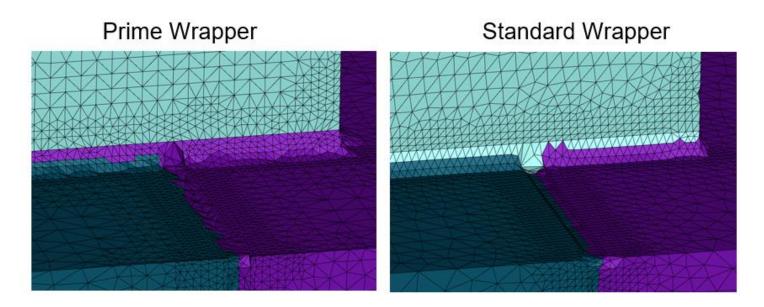


Workflow		Outline View
Fault-tolerant Meshing		• 6 8 0 0
- Workflow		
🖉 🖉 Import CAD and Part Mana	gement	
🧚 🕅 Describe Geometry and Flov	N	
🧚 📑 Identify Regions		
🧚 🕕 Define Leakage Threshold		
$\frac{7}{2} \square_{0}^{2}$ Update Region Settings		
- 🧚 🛄 Choose Mesh Control Option	s	
🧚 🛄 Add Local Sizing		
🧚 🎯 Generate the Skin Surface [
✓ Skinned Surface Mes	h	
Workflow Edar		
Describe Georetry and Flow		
Modeling Objectre:		
O External floor around object		
O Internal float through the object		
 Both external and internal flow Skin assemblies by wrap 		
		<u> </u>
Standard Options	<u></u>	<u>.</u>
Create large caps?	○ Yes	⊙ No
 Advanced Options 		
Identify regions?	 Yes 	○ No
Close leakages?	 Yes 	○ No
Add thickness to any walls?	○ Yes	• No
Extract edge features?	○ Yes	No

Fault-Tolerant: Prime Wrapper (Beta)

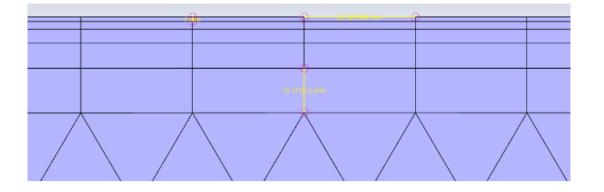
- An alternate wrapper is available as beta for the modular wrapping approach
 - Speed-up of mesh generation
 - Faceting will be different

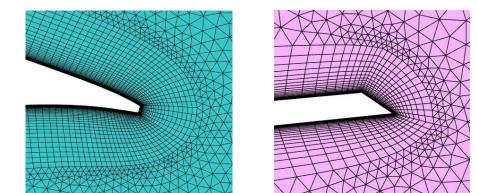
Workflow Editor			
Describe Geometry and Flow			
Modeling Objective:			
C External flow around object			
O Internal flow through the object			
 Both external and internal flow Skin assemblies by wrap 			
🕞 Standard Options			
Create large caps?	 Yes 	○ No	
Advanced Options			
Identify regions?	• Yes	○ No	
Close leakages?	• Yes	○ No	
Add thickness to any walls?	○ Yes	• No	
Extract edge features?	⊖ Yes	• No	
Enable prime wrapper?	⊙ Yes	○ No	
Describe Geometry and Flow Revert and Edit			



2D Meshing: Boundary Layers

- Various improvements for smoother boundary layers
 - Normal smoothing
 - Shrinkage
- Introduction of Last Ratio method



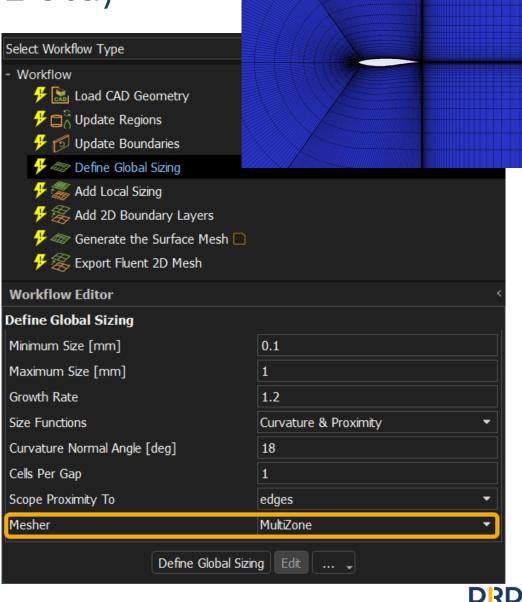


Workflow Editor	<
Add 2D Boundary Layers?	yes 🔹
Add 2D Boundary Layers	
Name	last-ratio_1
Offset Method Type	last-ratio 🔹
Number of Layers	3
Transition Ratio	0.272
First Layer Height [mm]	0.01
Max Layer Height [mm]	1
Add in	fluid-regions 🔹
Grow on	only-walls 🔹
Advanced Options	
Add 2D Boundary Layers	Revert and Edit 🗸

2D Meshing: Mapped Quad (Beta)

- 2D MultiZone allows generation of quad mesh
 - Used for the full domain
- Local edge sizing options with biasing

Workflow Editor	<			
Would you like to add local sizing?	yes 🗸			
Add Local Sizing				
Name	edgesize_1			
Size Control Type	Edge Size 🗸			
Assign Size Using	Interval 🔹			
Number of Layers	3			
Growth Pattern	`			
Assign Growth Using	Growth Rate 🔹			
Growth Rate	1.2			
Select By	label 🗸			
Edge Zone Labels Filter Text 🔹 [0/	1] Filter Text 🗟 🗟 🤿 🖡 💿 📐			
part-boundary				
Reverse Edge Zone Orientation Draw Size Boxes				
Add Local Sizing Revert and Edit C	ear Preview Draw Bodies 🖕 🖕			



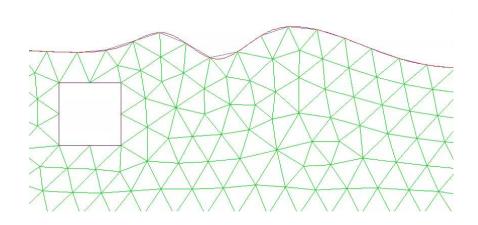
Fluent CPU Solver Physics Improvements

Polyhedral Unstructured Mesh Adaption (PUMA)

- 2D polyhedra support
 - Hanging node conversion to polyhedral by default when reading older case file
 - Can be disabled using /file/convert-hanging-nodes-during-read?
- 2D PUMA
 - Default adaption method
 - Can be changed to legacy hanging node adaption using /mesh/adapt/set/method
- Refinement levels from Fluent Meshing imported during I/O and mode transfer when reading Rapid Octree meshes
 - Allows for more intuitive PUMA adaption and better mesh quality in Fluent

Dynamic Remeshing

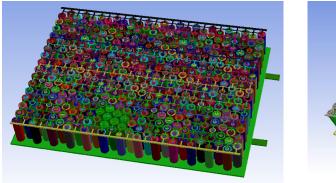
- Previously released Geometry Reconstruction extended to dynamic mesh remeshing
 - Nodes are projected onto the smooth reconstructed surface



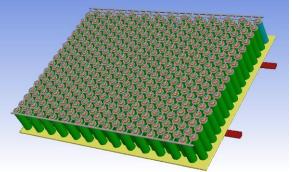
Battery Model: Variable-Based Connection

- Variable-based connection replaces zone-based connection to mitigate bottlenecks for connecting a large number of zones
 - Requires a single zone for all modules, tabs, and busbars

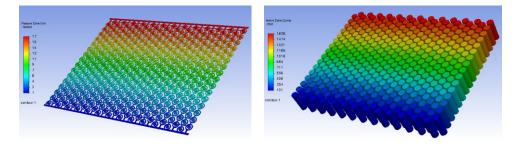
Total cell count: 2,189,189 Battery connection: 16s26p (416 batteries)



Cell threads: 1682 Face threads: 8343

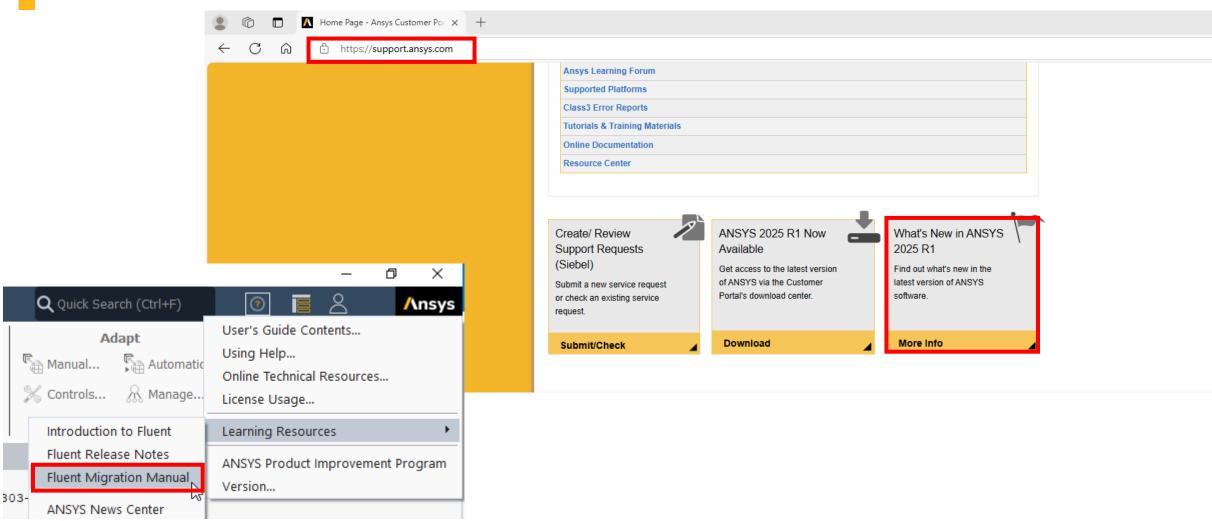


Cell threads: 6 Face threads: 28



	Old Method	New Method
Case reading	30 min	40s
Data reading	10 min.	7s
Solution initialization	1 min	10s
Running 20 iterations	4 min.	60s
Case writing	3 min.	13s
Data writing	20 s	5s

More Updates





The recording and slides for this webinar are in our Technical Resources Library.



SIMULATION PRODUCTS CONSULTING TRAINING COURSES - SUPPORT RESOURCES - ABOUT - CONTACT US 💭

Technical Resources Library

IN-DEPTH RESOURCES TO ENHANCE YOUR SIMULATION SKILLS

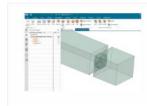


If you are not on our mailing list, or are unsure if you are, please let us know at <u>support@drd.com</u> and we can add you!



WHITE PAPER

Six Considerations for Selecting Engineering

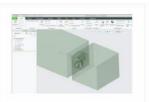


WEBINAR Full CAD Associativity Between NX and Ansys - (June 22,



WEBINAR

Full CAD Associativity Between Autodesk



WEBINAR

Full CAD Associativity Between Creo Parametric and

Wrap Up

← → C ☆ @ drd.com/training/



Whether you're onboarding with the Ansys platform or looking to take your simulation proficiency to the next level, we have a training course carefully designed to fit your needs. With frequent introductory and advanced courses conducted live virtually and in-person or on-demand, we offer many opportunities for you to get the training experience that best suits your needs. Additionally, since our trainings are conducted by our in-house engineering and physics experts, we have the unique opportunity to carefully listen to your requirements and further refine our custom training materials to help you continually meet your goals.

Explore our training center below.



Thank you for your attention!

May I address any questions?

