New Ansys Fluent Web UI Supports Access to Massive, Multi-GPU CFD Simulations

February 7, 2024

The web-based technology unlocks the power of cloud computing for faster CFD simulations, reducing reliance on hardware resources

/ Key Highlights

- Ansys Fluent Web user interface™ (UI) enables users to remotely interact with simulations running on the cloud anywhere from any device
- The interface supports large-scale parallelization of massive computational fluid dynamics (CFD) simulations by tapping into multi-GPU and cloud computing capabilities to reduce power consumption and increase simulation speed by up to 10x

PITTSBURGH, Pa., Feb. 7, 2024 /PRNewswire/ -- Ansys (NASDAQ: ANSS) today announced Fluent Web UI, a web-based technology that enables users to remotely access simulations from any device. Fluent Web UI provides an interface for Ansys Fluent™, an industry-leading CFD solver with native GPUs that can speed time-to-result by almost 10x. Users are now able to run, control, and monitor simulations on the cloud or on-premises high-performance computing clusters. The streamlined interface delivers pervasive insights significantly faster for a wide range of CFD applications, including automotive under-hood heat transfer, gas turbine simulations, and external aerodynamics for aerospace applications.

Building on Ansys' cloud initiatives, Fluent Web UI complements other offerings to provide users with a comprehensive suite of tools, from design to validation. For instance, Ansys Discovery™ Burst relies on fast-scaling clusters of GPUs to facilitate expansive, early design exploration that helps designers identify optimal designs faster, reducing the need for physical prototyping. Moreover, users can leverage the power of generative artificial intelligence (AI) with the new Ansys SimAI™ tool to rapidly explore additional designs within a given design envelope, getting results within minutes. SimAI is a pure software-as-a-service offering that supports an open ecosystem and is approachable for users without deep learning expertise.

"Our ability to harness the power of Ansys Fluent native GPU solver has saved us considerable time while using a fraction of the typical hardware resources," said Francesco Manara, aerodynamics specialist, Leonardo Helicopters "For example, we wanted to assess the external aerodynamics for airframe loads — a resource-intensive simulation — with the same reliability as the traditional Ansys CPU solver. With the native GPU solver, we ran our model 2.6 times faster using only one-third of the hardware resources compared with the CPU solver."

"Our customers can leverage our native GPU solvers and AI solutions in the cloud or on premise to realize significant time-saving benefits across industry applications from automotive to aerospace," said Shane Emswiler, senior vice president of products at Ansys. "It's no secret that CFD simulations can be incredibly time and resource-intensive, but our customers are working against tight timelines that have little room for error. The seamless connectivity that Fluent Web UI will help keep complex projects on track, while empowering designers to make better decisions as they monitor the simulation in real time from any device."

/ About Ansys

Our Mission: Powering Innovation that Drives Human Advancement™

When visionary companies need to know how their world-changing ideas will perform, they close the gap between design and reality with Ansys simulation. For more than 50 years, Ansys software has enabled innovators across industries to push boundaries by using the predictive power of simulation. From sustainable transportation to advanced semiconductors, from satellite systems to life-saving medical devices, the next great leaps in human advancement will be powered by Ansys.

ANSS–T

/ Contacts

Media	Mary Kate Joyce 724.820.4368 marykate.joyce@ansys.com
Investors	Kelsey DeBriyn 724.820.3927 kelsey.debriyn@ansys.com

View original content to download multimedia:https://www.prnewswire.com/news-releases/new-ansys-fluent-web-ui-supports-access-to-massive-multi-gpu-cfd-simulations-302055341.html

SOURCE Ansys