VSCALER FOR CFD

Computational fluid dynamics simulations in the cloud

Computational fluid dynamics (CFD) is a branch of mechanics that predicts the behaviour of fluids or gasses as they move around an object or structure. as well as the interactions of those fluids and gases with the objects' surface.

These calculations are best run on supercomputers with parallel processing, a feat which typically beyond the budgetary means of smaller companies. Recent advances in workstations' computing power have make it possible to run simulations on desktops, but the computations can take hours, and sometimes days.

While it is possible to run CFD workloads on powerful workstations (albeit slowly), engineers are finding that by leveraging the superior power of high-performance computing (HPC) along with containerised software applications, they can deliver high quality results far faster and easier.

vScaler can offer CFD users an application specific, cloud environment with the performance benefits of a dedicated supercomputer but at the fraction of the cost. Supporting industry leading software (ANSYS, OpenFOAM, Pam-Crash and STAR-CCM+), vScaler for CFD is specifically designed to improve access and development efficiency. With ready-to-execute application containers, you can simplify your software setup while delivering faster analysis than legacy desktop workstations.

vScaler's private cloud solution for CFD, is built on open source technology that enables you to create a secure, scalable, cost-effective, flexible IT infrastructure. No vendor lock-in, no hidden costs, just your cloud, your way.

Faster Analysis

Our ready-to-execute applications deliver faster analysis compared to legacy desktop workstations.

Lower Costs

vScaler is an application specific cloud environment with all the performance benefits of a dedicated supercomputer but at the fraction of the cost.

Scalable

Scale-up or down with the touch of button, under a single customisable management portal.

VSCALER

WHAT DO I GET FROM VSCALER?

Open Source

Built on open source cloud technology, and using open source CFD technologies such as OpenFOAM, vScaler offers researchers access to the tools and HPC parallel-processing power they need to perform CFD analyses on their own.

Minimise your Product Development Lifecycle

Rely less on physical testing and speed up your product development lifecycle by running your CFD simulations on our open source cloud platform rather than legacy hardware and wind tunnels.

CFD Specialists

As an added service, vScaler's in-house highly specialised team of Formula 1 CFD and aerodynamics experts-can also aid in tuning meshing and solver performance through, for example: process and memory bindings, to deliver staggering results.

INDUSTRY LEADING SOFTWARE

vScaler for CFD supports industry leading software such as ANSYS, OpenFOAM, Pam-Crash and STAR-CCM+ (among others) out of the box.

HOW CAN I MOVE TO VSCALER?

The switch from your old system can be done in stages with little to no impact on your daily operations. vScaler offers full service on-boarding and service off-boarding assistance, as well as a modular Managed Services options, with no-vendor lock-in.

SUPPORTED APPLICATIONS

OpenFOAM
STAR-CCM+
ANSYS
Pam-Crash
Others available on request

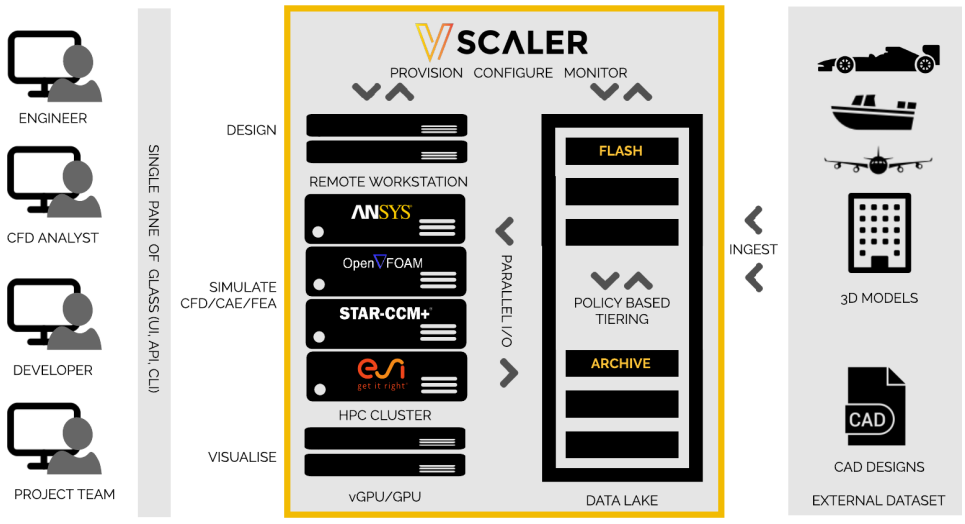
STORAGE

Unified Storage
Containerisation

SECURITY

Disaster Recovery
Rolling Upgrades
Zero Downtime

VSCALER FOR CFD WORKFLOW



For More information,

Contact vScaler

info@vscaler.com | www.vscaler.com | +44 (0)20 3889 0662

©2018 vScaler. All rights reserved. vScaler trademark is registered in the UK. All other brand names mentioned herein are for identification purposes only and may be the trademarks of their respective holder(s).