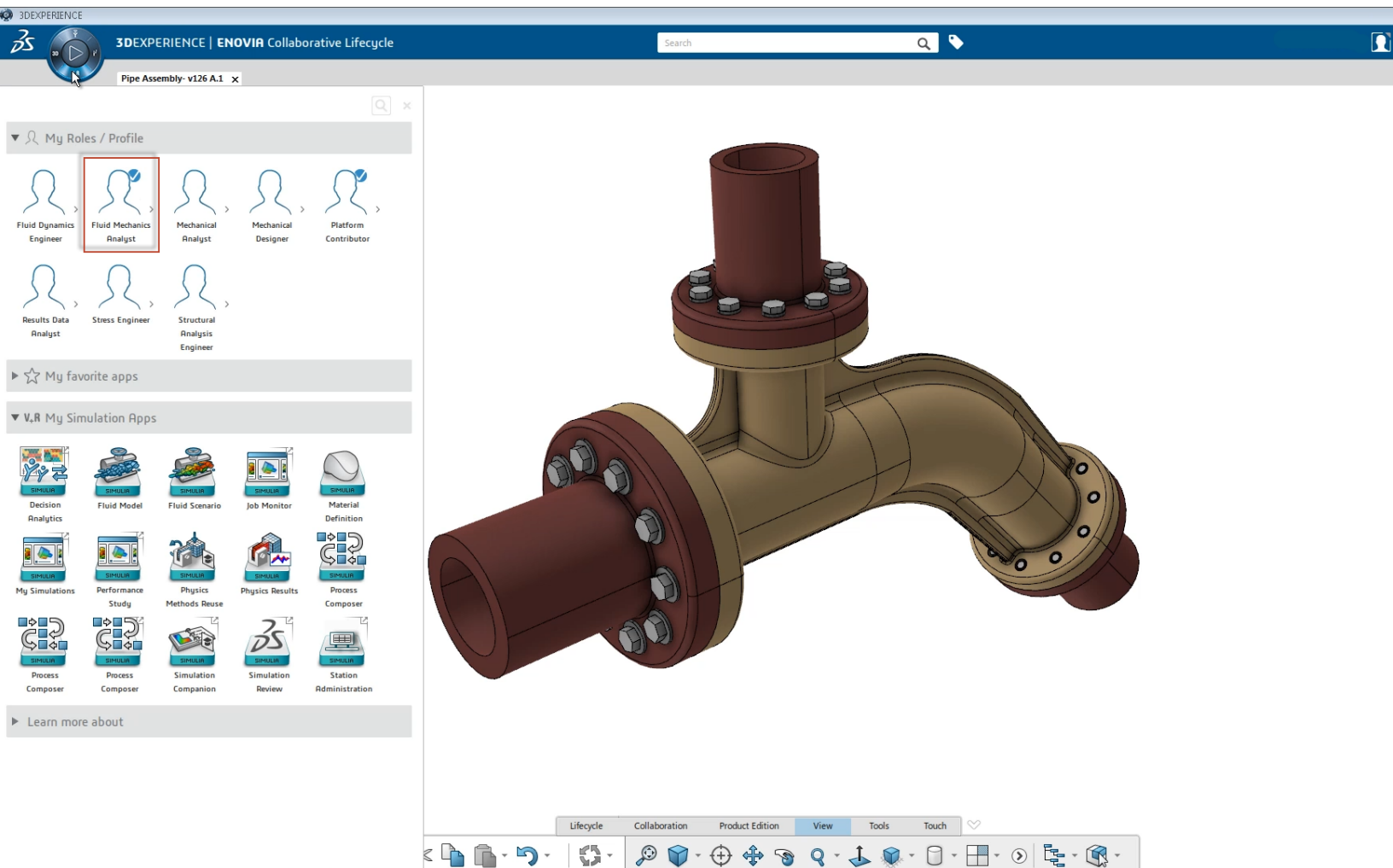




FLUID MECHANICS ANALYST

3DEXPERIENCE USER ROLE



**BRINGING
COMPUTATIONAL
FLUID DYNAMICS
(CFD) CAPABILITIES TO
THE 3DEXPERIENCE
PLATFORM**

Extend simulation capabilities to modeling fluid flow and heat transfer, both convective and conductive

Fluid Mechanics Analyst permits a seamless switch between applications, allowing users to be product-focused: applying the right tool at the right time in their workflow processes. This allows a much greater degree of flexibility and collaboration in the design and simulation process, where geometry changes are automatically updated and made immediately available on the same platform for the analyst to obtain updated results.

There are two key components to Fluid Mechanics Analyst which capture the CFD work flow processes: Fluid Model Creation and Fluid Scenario Creation.

- Fluid Model Creation allows the user to easily build high-fidelity meshes for fluid flow calculations, which accurately capture boundary layers and automatically extract the fluid domain, thus reducing the need for most manual preparation.
- Fluid Scenario Creation allows the user to define the fluid properties and the physics/boundary conditions that define the simulation scenario.

Reduced complexity of CFD workflows on a variety of flow scenarios

The dedicated workflow guides users through the principle steps in a CFD and Conjugate Heat Transfer (CHT) simulation giving novice users the ability to set-up and successfully run such a calculation. Flow Scenarios include internal flows such as pipes, valves, porous media and ducts with automatic fluid domain extraction, and external flows such as buildings and vehicles with automatic creation of bounding surface of fluid domain.

Accurate results for complex boundary layer problems and turbulent flows

Hex dominate meshing with body fitted prism layers provide a higher fidelity simulation for convective flows with accurate capture of the boundary layer for accurate wall shear stress and drag calculations. A suite of standard turbulence models are available to capture the effects of turbulence, as well as a hybrid near-wall treatment to accurately predict boundary layer flows independent of near-wall mesh density.

Reduce computational time for steady state and transient problems

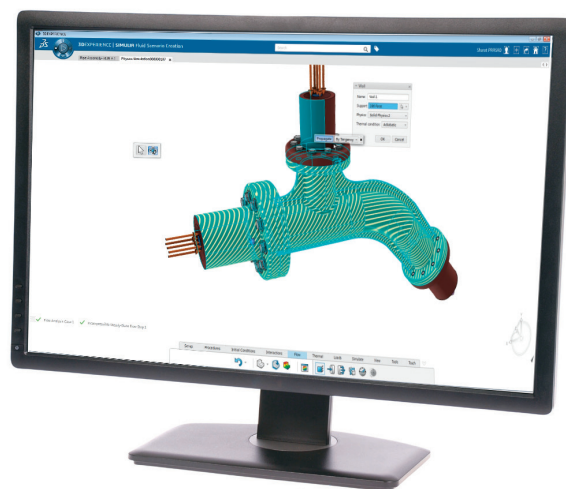
The steady state solver significantly reduces computational time for inherently steady state flow conditions and conversely, the transient solver accurately captures time varying flow behavior such as vortex shedding.

User Role Highlights

- Integration on the **3DEXPERIENCE®** platform
- Tight integration with CAD and PLM
- Shorter time to results for high production level fluid flow and heat transfer simulations

Our 3DEXPERIENCE® platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3DEXPERIENCE®** Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 210,000 customers of all sizes in all industries in more than 140 countries. For more information, visit www.3ds.com.



Different selection methods are available to facilitate better selection of supports.



The Velocity Vector contour plot shows the direction of fluid flow in the simulation.

