What Is CFD?

Computational Fluid Dynamics is a powerful engineering tool for simulating all types of fluid flows. CFD simulations produce detailed descriptions of flow characteristics including values for velocities, pressures, and other variables. Thermal characteristics and other advanced physics, like chemical reactions, particle flow, free-surface flow, can also be simulated.

Why use CFD?

Computational Fluid Dynamics can significantly reduce design and development time, provide detailed information otherwise not available from physical experiments, and quickly simulate a wide range of flow conditions. The results are improved designs, lower risk, and shorter time to market for a product or process.
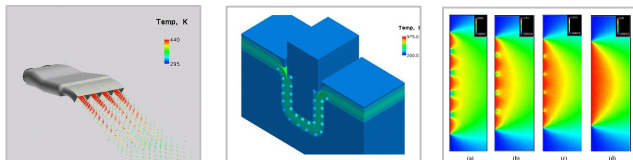
How does CFD work?

CFD simulations involve solving numerically the fundamental laws of physics, called governing equations, which mathematically describe the properties and motion of fluids. The CFD modeling process occurs in thee fundamental steps depicted on the right.
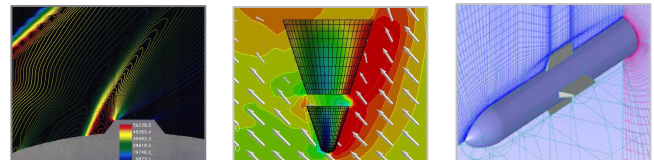
1

Geometric/Mesh Modeling Analysis

2

Numerically Solving Navier-Stokes

3

Solution Analysis Visualization
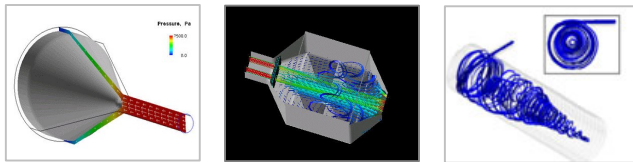
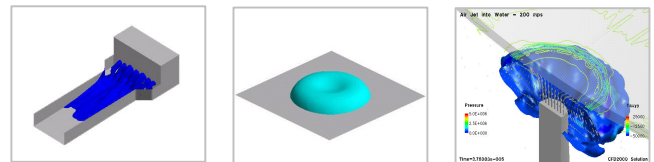### CFD Application Examples

Heat Transfer, CHT, and Radiation

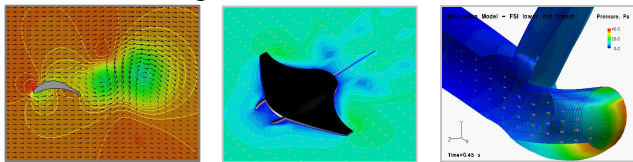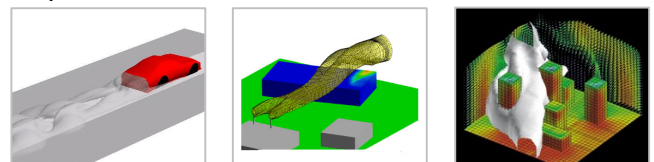*Aerospace Applications*

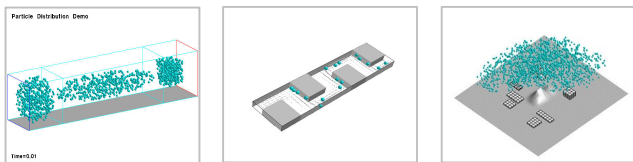*Engineering Systems*

Free Surface Flows

FSI and Moving Bodies

Dispersion Flows

Particle Flows

Bioengineering Applications