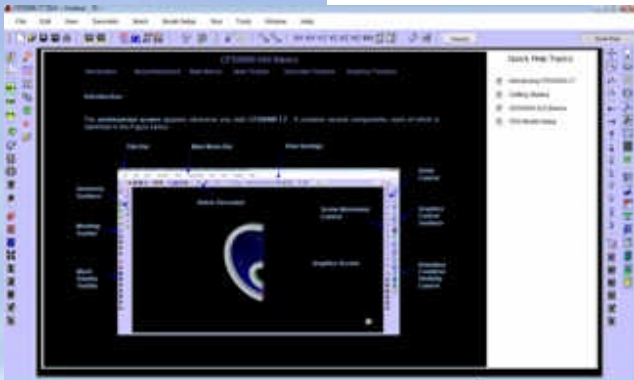


Adaptive Research would like to introduce an exciting offer to acquire CFD2000 LT 2014, an advanced Computational Fluid Dynamic software system designed for education.

CFD2000 LT 2014 is a fully integrated software system that allows you to harness the power of an advanced Navier Stokes flow solver and grid generation system with minimal effort.



## CFD2000 for Academic Research

### CFD2000: A Research Tool

CFD2000 uses a fast and efficient 3-D finite-volume Navier-Stokes solver that has been extensively validated against experimental data. By utilizing the integral form of the governing equations, conservation is enforced exactly and the PISO algorithm produces superior steady-state and transient solutions.

### CFD2000: An Instructional Tool

More than 120 CFD models are provided with the software, ranging from traditional 2-D validation problems (flow around cylinder, backward-facing step, and thermally-driven cavity) to 3-D applications such as hypersonic missile flow.

### Advanced Functions

A powerful pre-defined function capability is available to provide added control over fluid properties and boundary conditions. This capability eliminates the requirement for user customization of the STORM flow solver.

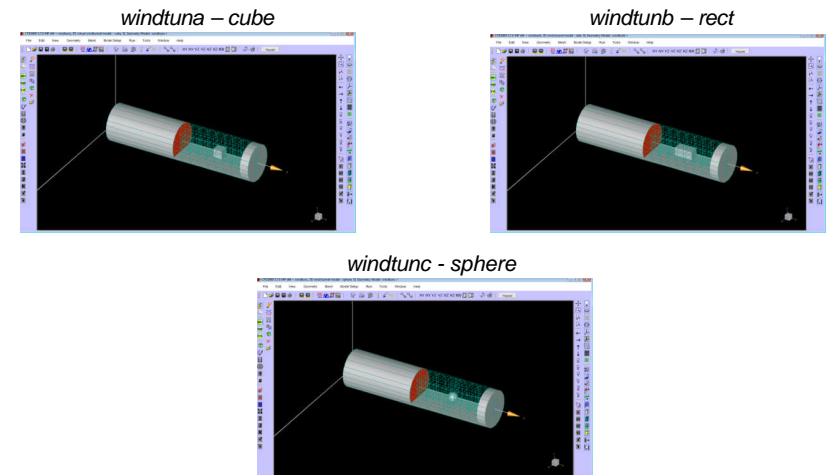
[arsales@adaptive-research.com](mailto:arsales@adaptive-research.com)

Computational Fluid Dynamics (CFD) models have been developed for three basic shapes placed in a virtual wind tunnel.

### Loading a CFD Model

Start the CFD2000 program by double-clicking on the **cfdbat** batch file located in the working directory.

From the **File** menu, select the **Open CFD Project** menu item. An **Open CFD2000 Project** dialog window will appear, select the **windtuna.usr** file. The CFD model will appear in the CFD2000 viewport window.

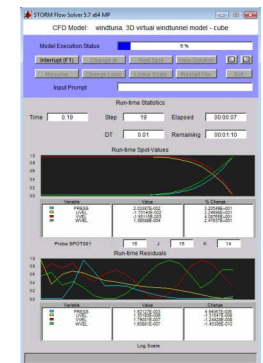


### Executing the CFD Models

The CFD models can be immediately executed by clicking the STORM execution icon located on the top toolbar:



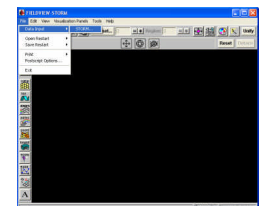
The STORM Solution monitoring program will appear (see next page).



### Visualizing Simulation Result(s)

Once the CFD model has completed execution, a series of output files will have been created in the local working directory. These are binary files with the file extension **“.svu”**.

The first step in the solution visualization process is to load the transient data set into the visualization program. Start the visualization program by double-clicking on the **svbat** batch file located in the working directory. Minimize the program size by clicking on any corner of the window, holding down the left mouse button and moving mouse to resize the window.



## Modeling Application Example – Virtual Wind Tunnel

To load the transient data, click the **File** menu and select **Data Input->STORM...**

Click the **Read Grids & Results Data...** button. Then from the **Open Grids & Results File** dialog window, select the **windtuna0001.svu** file of choice. Click the **Open** button

The Function Subset Selection window will appear.

Click OK.

The **Transient Set** window will appear. Click **Yes**.

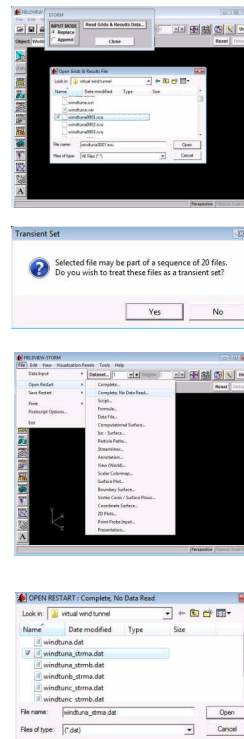
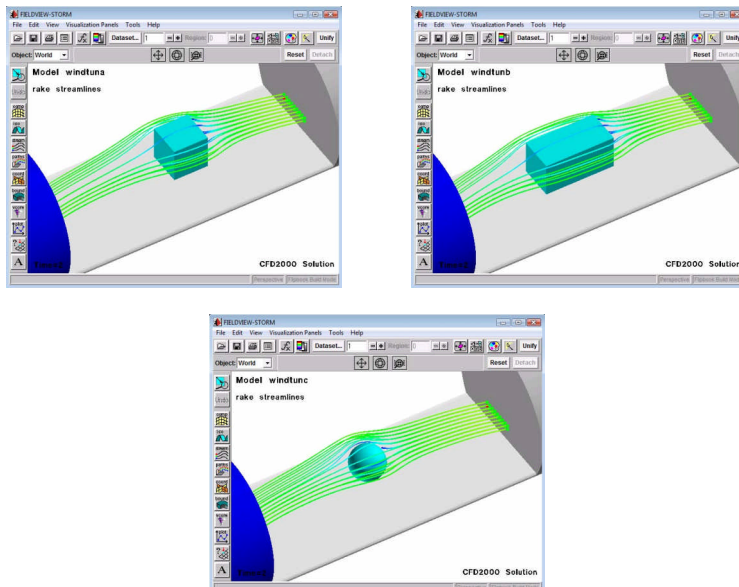
### Loading a Visualization Restart File

To load and apply a predefined visualization restart view to a loaded data set, from the **File** menu select the **Open Restart -> Complete, No Data Read ...** menu item.

The **Open Restart** dialog window will appear.

Select the **windtuna\_strma.dat** file and click the **Open** button. The initial transient file with the applied visualization view will appear in the Fieldview-STORM viewport window. Use the **Tools->Transient Data** to move the solution to the last transient file.

### Simulation Results



## CFD2000 LT 2014 - Features & Capabilities

Steady-state and transient processes

Laminar flows

Turbulence models

- LES
- k-epsilon STD
- k-epsilon RNG
- k-epsilon Chen-Kim
- low RE number k-epsilon

Newtonian and non-Newtonian fluids

Compressible and incompressible flows

Subsonic through supersonic flows

Heat transfer

- Conduction
- Natural, forced, and mixed convection

Fluids and solids library

Switch-on physical model

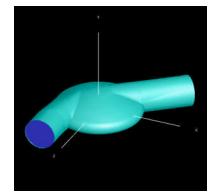
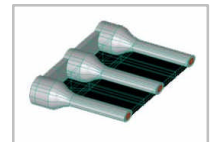
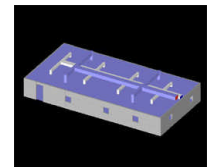
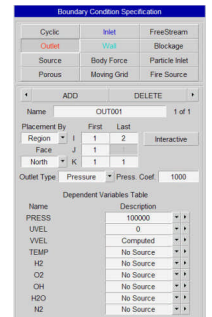
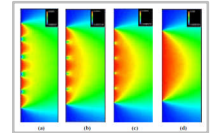
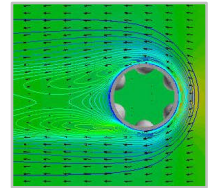
- Filters, screen, solids
- Porous media
- Walls (solid, porous, fixed temperatures, heat, flux)
- Symmetric boundaries
- Cyclic boundaries in any coordinate direction
- Swirl flow (axisymmetric flow)
- General source (or sink) boundary

Easy to use Graphical User Interface

- Menu-driven input
- User customizable
- Point-and-click BCs
- Geometry display tools
- Flow-field initialization tools
- Real-time convergence
- Error diagnostics
- Example library
- Prompt line help

Geometry and Mesh

- CAD data import
- Primitive object library
- Grid-Independent geometry
- Geometry creation and modification tools
- Copy, scale, translate, and rotate functions
- Individual edge control for node distribution
- Automatic grid generation/optimization
- Surface of revolution
- Transfinite interpolation



## Contact Adaptive Research

Fax: 808-356-0128  
 E-mail: [arsales@adaptive-research.com](mailto:arsales@adaptive-research.com)  
 Web: [www.adaptive-research.com](http://www.adaptive-research.com)

