NuFD
Numerical Flow Designing

Numerical Flow Designing Co., Ltd.
1-10-7 Higashi-Gotanda, Shinagawa-ku,
Tokyo 141-0022 JAPAN
E-mail: info@nufd.jp URL: http://www.nufd.jp

Next-generation IT program of MEXT/Revolutionary Simulation Software project
Entity Developer: Institute of Industrial Science, the University of Tokyo

Fluid Dynamics Simulation software: The Next Generation

NuFD/FrontFlowRed

Large Eddy Simulation (LES) can be conducted by using high performance computing (HPC) techniques for turbulence simulation in engineering fields. The unstructured-grid LES code "NuFD/FrontFlowRed" based on the finite volume method (FVM) is optimized for the execution on HPC. "NuFD/FrontFlowRed" provides not only turbulence simulation, but also simulation for the complex multi-physics problems, such as turbulent combustion, multi-phase flow and so on.

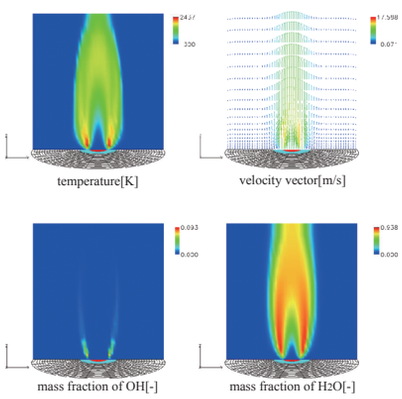
Details reaction model

For accurate analysis with combustion, two or more reaction models are needed. A sample simulation of diffuse flame of a co-axial jet burner is shown, considered with 9 chemical species and 21 elementary reaction formulae.

Temperature/velocity vector
General axial-symmetric laminar diffusion flame is recognized.

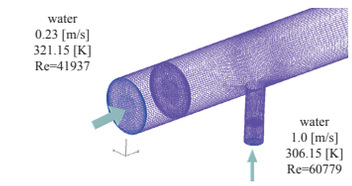
Radical OH
Radical-OH exists mainly around interface of oxidizer and fuel of diffusion flamelet

Products H2O
Without changing chemically, H2O is spreading from reaction zone.



Flow oscillation

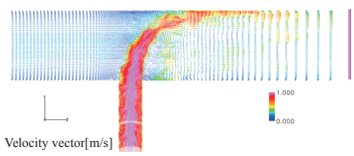
Turbulence analysis result of T-junction composed of unstructured-grid is shown. Fully-developed turbulence flows from the main pipe and the branch.

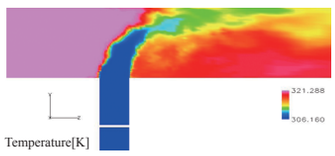


It is well known that the flow of jet from the branch into the mixing region can be classified to 4 following patterns trough flow visualizing experiments.

1) wall jet flow
2) reattaching jet flow
3) deflected jet
4) impinging jet

This analysis result is consistent with the experimental results corresponding to impinging jet.

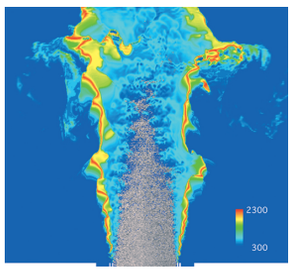




Large-Scale Simulation of Spray Combustion

Right figure shows the snapshot of temperature and droplets in spray combustion.
The simulation is done by using 50 million grids mesh and 2 million droplet particles with flamelet model.

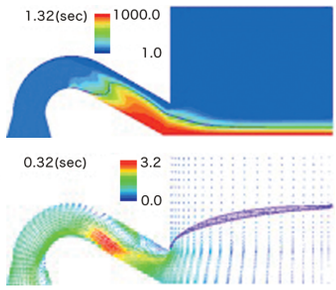
(Cooperative development with Dr. Eng. Ryoichi Kurose, Associate Professor, Dept. of Mechanical Engineering & Science, Kyoto University.)



Flame temperature and droplet distribution in spray combustion

Flamelet model

By adopting 2-scalar flamelet model (mass fraction equation and G-equation for premixed flame model), "NuFD/FrontFlowRed" provides complex unstable flow analysis with diffusion and premixed flame. For examples, "NuFD/FrontFlow Red" can compute lifted flame in turbulence and flame stabilization mechanism by interference from vortex and flame



simulation for problem of siphon (for DMW CORPORATION)

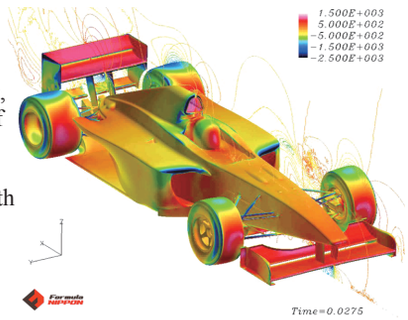
Two-phase flow/cavitation

As to two-phase flow model, "NuFD/FrontFlowRed" will provide two-fluid flow model based on phase separation method and VOF/level-set method by interface-tracking method. With demands from analysis targets, "NuFD/FrontFlowRed" can compute gas-liquid two phase flow, free surface flow, compressible cavitation and so on.

High Performance Computation

The main purpose of this simulation is to evaluate the validity of LES, as an alternative to the conventional wind tunnel testing as well as the RANS method for the foreseeable future, to the aerodynamic assessment of automobiles.
"NuFD/FrontFlowRed" was intensively optimized for the Earth Simulator, and finally we could successfully conduct the simulation on the parallel environment using 100 nodes/800 CPUs.

(Institute of Industrial Science, the University Tokyo)



Features

- high parallelization/vectorization for large-scale computations
- modern turbulence models (LES/RANS)
- flexible with compressible/incompressible scheme
- pulverized coal/ spray combustion with particle tracking method
- LES flamelet models for diffusion, premixed and partial-premixed flames
- sound pressure level predictions with turbulence noise models
- coupling radiative and convective heat transfer using real gas model
- two phase flow with complicated physical phenomenon
- assessment of wind environment and air-diffusion simulation based on meteorological model
- polyhedral mesh support
- data import/export for many meshes and visualization tools