CFD simulation with LS-DYNA
**R7** first release:
- Fluid structure interaction (FSI)
- LES turbulence model
- Basic support for freesurface
- Conjugate heat transfer

**R8**
- New turbulence models including k-e, k-w, relizable k-e, Spalart-Allmaras and WALE with several new laws of the wall and advanced boundary layer mesh control tools
- Added isotropic/anisotropic generalized flow in porous media.
- Pre and post processing in LS-PrePost has major improvements
- Several new control and database options.
- Non-inertial reference frames
- Wave generator
- Coupling with DEM
- Added non-linear conjugate heat transfer and new solver.
What is the main barrier for large scale CFD simulations?

It's not the software. It's not the hardware. It is the rising cost of licenses/core of traditional commercial CFD packages.*

*IDC Automotive Update 2014

LS-DYNA license cost is competitive and has not changed in years.
What are the reasons for such a stiff interaction between CFD, crash & safety and NVH departments?

1. There is no interest in collaboration.  
   - Incorrect

2. There is not enough volume of problems to justify a more permanent interaction.  
   - Correct

3. The software does not make it easy to collaborate.  
   - Correct
LS-DYNA is an excellent option for CAE consolidation

**Thermal Solver**
- Implicit
- Heat Transfer
- Radiation
  - Double precision

**EM Solver**
- Implicit
- Air (BEM)
- Conductors (FEM)
  - Double precision

**Mechanical Solver**
- Implicit/Explicit
- Double precision/
  - Single precision

**Fluid Solver**
- Implicit/Explicit
- Incompressible (ICFD)
- Compressible (CESE/ALE)
  - Double precision

**Particle methods**
- Explicit
- SPH
- DEM
  - Double precision
Accurate CFD analysis will require structural coupling

The mechanical behavior of structural parts subject to heat and fluid pressure loads have been neglected.

Until the road test
Accurate CFD analysis will require structural coupling.
Some areas of applications
Some areas of applications
Aeroelasticity analysis of spoilers

- Prevent flutter. A broken spoiler flying at a 100 MPH is dangerous.
- Provide insight for structural designs
- Drag and downforce aerodynamics performance
Flutter is a catastrophic event that is very difficult to predict.

Only coupled CFD+structural simulations can produce flutter.

LS-OPT is currently being used to identify the flutter velocities of a coupled system.
Aeroelasticity and flutter using LS-OPT

- LS-OPT can help identify flutter modes
- The analysis of data is done automatically
- A neural network could be fed with design parameters to create an envelope of configurations that will not flutter.
Some areas of applications
Conjugate heat analysis with radiation

- Complex thermal analysis involving fluid and structural parts
- Couple to MAT_4 or MAT_106 allow structural parts to respond to thermal changes.
The conjugate heat solver allows the solution of coupled CFD-structural thermal problems. The main applications include: cooling in metal stamping, heat management inside engine compartment.
Some areas of applications
Fluid Structure analysis

- CFD analysis of full vehicle.
- Couple parts of the structure to analysis the response in a realistic environment.

Von Mises stress
Some areas of applications
Free surface flows

- Free surface flows are used in sloshing analysis, hydraulic problems, etc.
- New developments have added support for wave generation.
Some areas of applications
The discrete element method is good at solving particle interaction and brittle material.

Coupled to CFD allows a large range of applications that include erosion, fracture and particle interaction.

Coupled DEM+CFD solutions

- Powder simulation
- Drug delivery
- Mud deposition
- Water management: Rain Simulation
Some areas of applications
Heart valve simulations

- Hemodynamics represent some of the most challenging Fluid Structure Interaction problems
- New Windkessel model used as boundary condition to represent the load undertaken by the heart during a cardiac cycle.

The introduction of the catheters to the vessel for repairing abdominal aortic aneurysm (velocity isosurface representation). Courtesy of Hossein Mohamed, McGill University.
Some areas of applications
Generalize flow in Porous media

- Modeling of Newtonian/Non-Newtonian flows over the whole range of variation of porosity material parameters: \{porosity → 0, permeability → 0\} and \{porosity → 1, permeability →\infty\}, i.e. all Reynolds numbers.
- Heat Transfer and Free-Surface flow capabilities.
- Permeability tensor mapping for moving nd deformable solids

Ex 1) Mold filling analysis

Ex 2) High Reynolds Flow through a car radiator:

Ex 3) Heat transfer in a hybrid channel.
LS-PrePost
Pre and Post treatment of a CFD run in LS-Prepost

- The new CFD GUI is accessible through the MS button. Its main feature is the introduction of an object oriented Tree structure which allows the user to handle each part separately and to apply several options on it.

- In the present case, the surface parts composing a road vehicle are displayed, colored by the pressure field, along with the domain floor, displayed in Wire mode.
Several new objects can be created, each one of them with their own independent properties.

In the present case, a section plane, streamlines and an isosurface have been applied on the fluid volume. Note that the Streamline and Section Plane have been colored by velocity and that they both have their own independent fringe bar.
Pre and Post treatment of a CFD run in LS-Prepost

- Here is another example, this time, Vectors and Streamlines objects have been created by applied on the Section Plane rather than on the volume. Again, the streamlines have been colored by velocity.
More advanced tools are available for a always better comprehension of the flow. In the present case, the Vortex Cores are captured and displayed allowing the user to very quickly identify regions of interest, where flow separation or turbulent effects may occur.

Pre and Post treatment of a CFD run in LS-Prepost
Data extraction is also made easier. In this case for example, the Pressure along the vehicle body is plotted as a function of the distance. This is a very typical CFD post-treatment.
It is also possible to post treat all ASCII files (See *ICFD_DATABASE family) dumped by the ICFD solver (forces, flux, point data etc).
A specific interface in order to set up a CFD input deck is also under development. It follows a similar Tree structure and it aims at providing the CFD user with a friendly environment to define his problem and allowing him to easily check his models for error and inconsistencies between keywords.
Validation

External Aerodynamics

Internal Aerodynamics

Free surface flow

Fluid Structure Interaction

Thermal Problems

http://lstc.com/applications/icfd
- Designed by TUM, Inst. For Aerodynamics.
- The objective is to perform automotive aerodynamics validation.
- It is a generic reference model with a modern car geometry.
- There is wind tunnel experimental data for comparison.

Configuration used in the study F_D_wM_wW
Fastback_Detailed underbody_with Mirrors_with Wheels
- 3 hours of pre-processing using ANSA.
- Automatic volume mesh using LS-Dyna CFD.
- The results presented are from the first run.
- 22 Million element mesh.
- $Re = 4.8 \times 10^6$. LES turbulent model.

Error = 4.2%
We are constantly testing new ideas that we think can benefit the applications of our users.

We are currently exploring:

New advection techniques based on particles ideas for a more conservative transport of properties.

Testing AMG preconditioning ideas to compare with our current solver technology.

We are exploring the field of shape optimization using LS-OPT and pseudo-FSI analysis using FEM or IGA.

Post-processing CFD results using immersive VR technology with real analysis results.
Conclusions

LS-DYNA’s CFD accuracy and scalability are as good as any other commercial CFD solver.

Fluid structure interaction and thermal coupling provides insights on material behaviors that are otherwise overlooked by CFD only simulations.

A single license and a single model greatly simplifies the coupled solutions.
Thank You