New developments and future road map for the ICFD solver in LS-DYNA

Facundo Del Pin, Rodrigo R. Paz, Peggy Huang and Inaki Caldichoury

Ansys, Inc.

1 Abstract

This paper will discuss some of the new additions that will be part in the R13.0 release for the Incompressible CFD (ICFD) solver in LS-DYNA. The paper will also cover some of the highlights of R12. In the past year there has been an increased interest in the model of problems that involve the simulation of free surfaces, Fluid Structure Interaction (FSI) and porous media flow. These topics will be discussed, and the new features/improvements will be presented. The road map is a collection of feature requests from LS-DYNA distributors, Ansys ACE organization, academic collaborators and customers. Based on this a brief discussion the top topics will be presented including immersed interface techniques, gap closure models, multi-species transport and tighter integration with Ansys tools.

2 Introduction

Multiphysics simulations continue to be the focus of development and inspiration for new features in LS-DYNA. There are five clear industrial sectors were LS-DYNA ICFD can provide insights and solutions to assist in the simulation of innovative products: healthcare, aerospace, automotive, manufacturing and energy. In healthcare there are several research topics in the area of cardiac simulation in collaboration with academia and industry. In recent years a special focus has been devoted to the improvement in accuracy, stability and robustness for the fluid structure interaction (FSI) simulation of prosthetic heart valves. In the aerospace industry there has been recent development in the area of porous membrane materials which is assisting in the simulation of parachutes. For the automotive industry there has been improvements and additions to the solver for high pressure resin transfer molding which can assist in the manufacturing of composite materials, and we are advancing in the simulation of processes like Ro-Dip and the conjugate heat transfer fluid structure simulation of clutch problems. For the manufacturing sector cooling applications have been the center of attention for problems of metal stamping and battery cooling. In terms of the energy industry recent developments in the simulation of fluid structure interaction simulation for wind turbines with dynamic loads has been developed using sliding mesh techniques and periodic boundary conditions.
In the following sections a more detailed introduction to each of these new features will be presented, starting with feature from R12 in section 3 and then introducing some of the new features of R13 in section 4. In section 5 a roadmap will be discussed.

3 Release 12 new features

3.1 Sliding mesh

Sliding mesh is a technique that prevents excessive re-meshing in problems that involve rotating parts. It is used for the simulation of turbomachinery. It is typically built using a cylindrical body which contains the blades of the machine. The cylinder is not topologically connected to the rest of a domain but constrains are used to transfer values across the interfaces (see Fig. 1). The blades and the cylinder rotate together as a rigid body which eliminates distortions in the mesh due to the rotating motion. In the case where the blades are flexible mesh distortion could occur due to FSI.

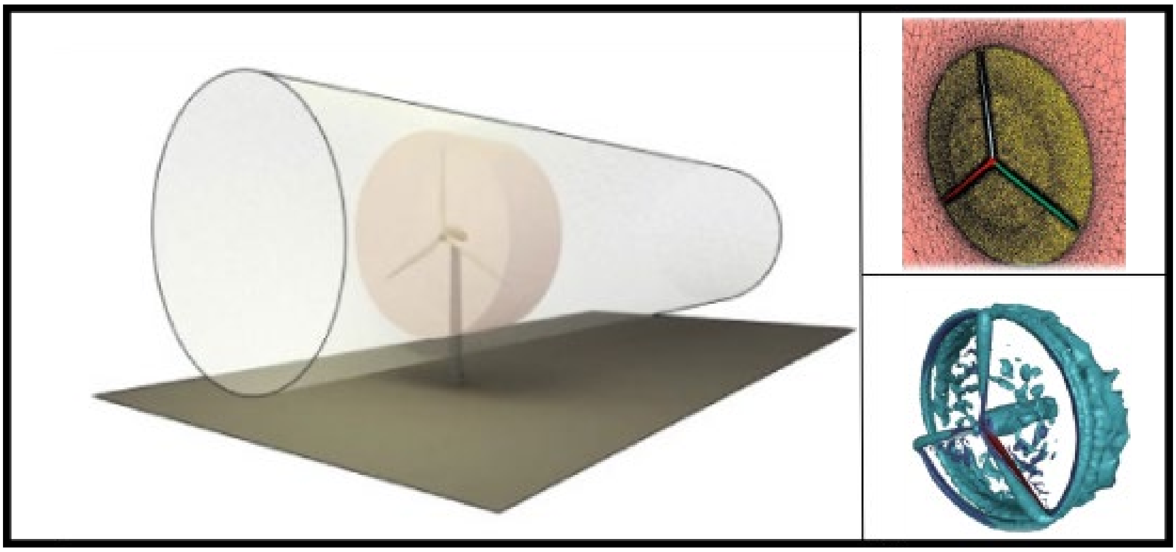

Fig.1: Wind turbine with sliding mesh domain shown in red. Left: details of the mesh used for the simulation (top) and results (bottom).

3.2 Periodic boundary conditions

Periodic boundary conditions allow a domain reduction of the areas with a repeating fluid pattern. It is widely used in the simulation of turbomachinery (see Fig.2). Non-inertial reference frames are used to represent the fluid rotation.

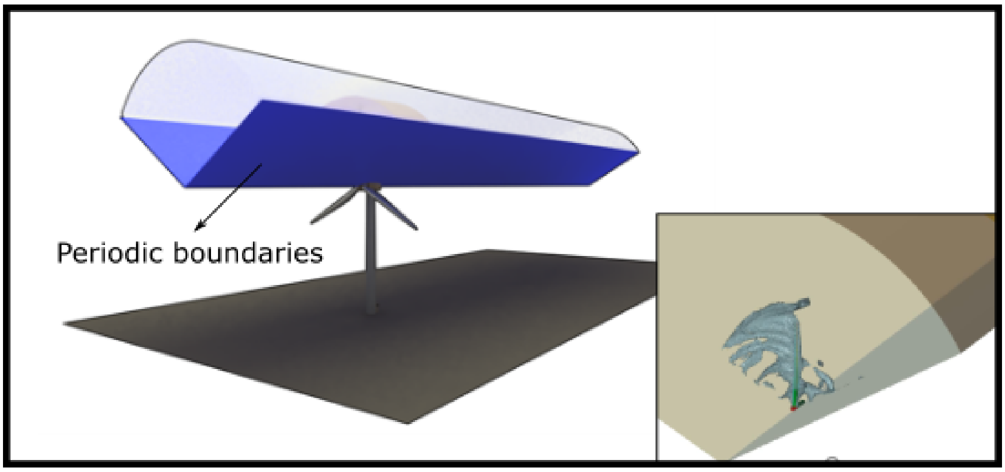

Fig.2: Wind turbine with a perodic boundary domain. Left: iso-surface of fluid velocity for the one blade simulation result.

3.3 Porous media model on membranes

Porous media models for volumetric domains were already available in ICFD for the simulation of isotropic/anisotropic porous matrices. In R12 there is a new model that allows the simulation of porous membrane materials. The membrane could be represented using shell elements in the structure which could be flexible allowing the simulation of FSI. Typical applications involve parachute and facemasks simulation (see Fig.3).

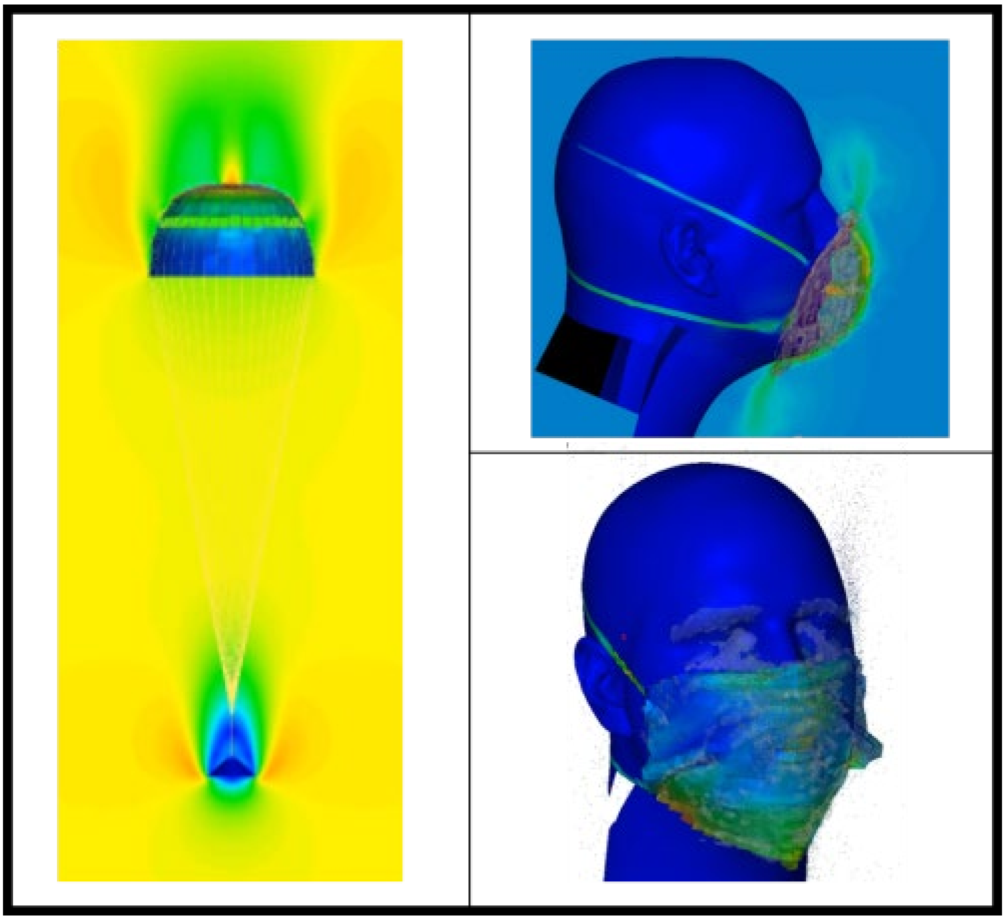

Fig.3: FSI simulation of a parachute (left) and face masks using porous models on membranes.

3.4 Wave generator for free surfaces

The ICFD solver has a complete set of 2D and 3D regular and irregular wave shapes for deep/intermediate/shallow water flows.

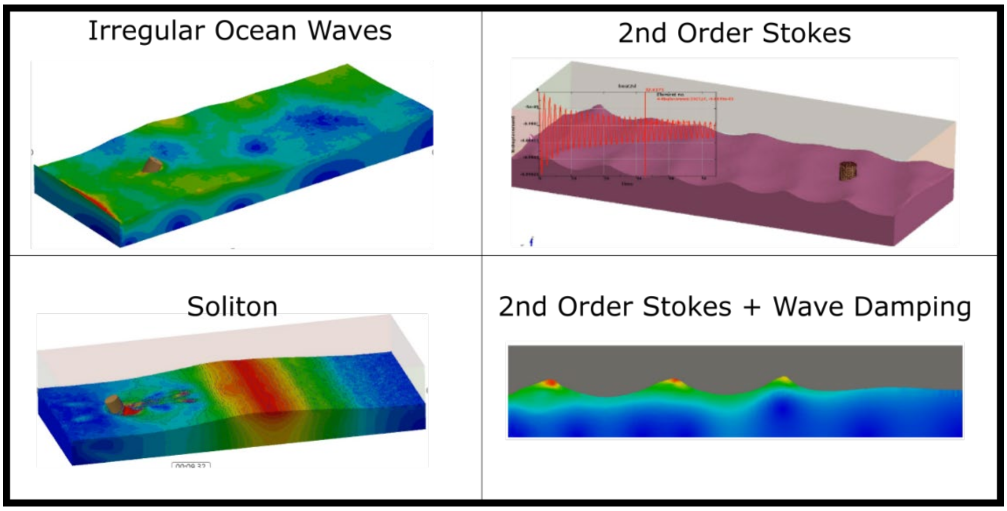

Fig.4: Example of a few wave models and features like wave damping.

3.5 Conjugate heat transfer: battery cooling

Conjugate heat transfer continues to be an active development area. Metal stamping applications for tool cooling presents challenges which the current version of LS-DYNA tries to tackle. The main objective in this release was to provide the user with tools that allow a custom balance between accuracy and computational time. A new option to switch from a direct solver to an iterative solver was added. The user can also control the frequency used to update the thermal matrix (see Table 1 for a comparison of running time). A new option also allows for a steady state CFD solution followed by a thermal analysis which will automatically use the fluid steady results in the thermal problem. Alternatively, the transient Navier-Stokes solution may be shut down and a subsequent thermal analysis may continue.

Tool Cooling (constraints contact)	MF2 direct solver	MF2+matrix update every ten timesteps	GMRES iterative solver	GMRES +matrix update every ten timesteps
Time per CHT iteration	22.0 seconds	0.25 seconds	1.7 seconds	0.4 seconds
Total run time	13 hours	4 hours 30 minutes	4 hours 20 minutes	3 hours 40 minutes

Table 1: Running time difference for a benchmark problem by changing the the linear solver from direct to iterative and increasing the frequency for the matrix update.

These developments allow the solution of challenging models like battery pack cooling (see Fig. 5).

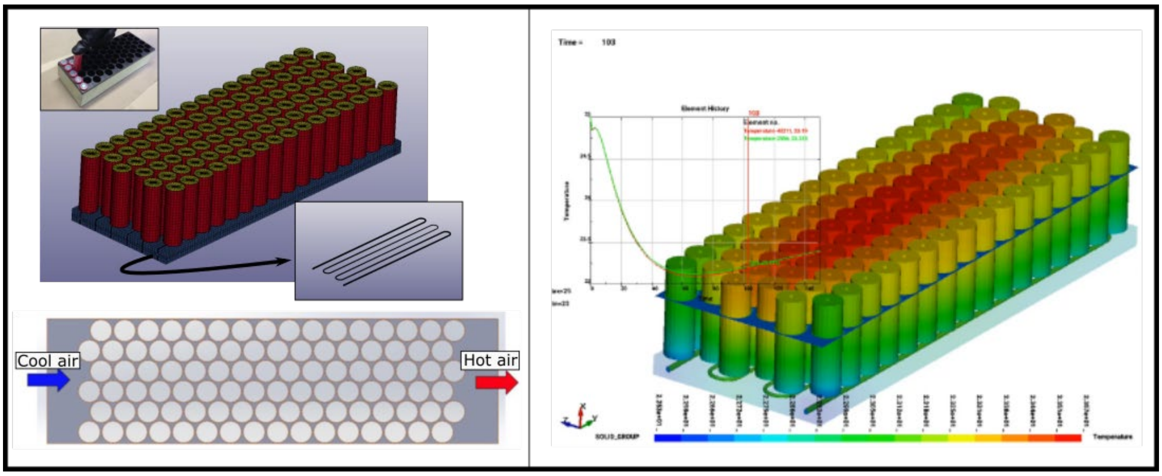

Fig.5: Battery ack cooling using air and a liquid flowing inside the serpentine.

Finally, a different time step between the thermal solver and Navier Stokes was implemented allowing sub-stepping during the coupling process.

4 Release 13 new features

4.1 Meshing

In this new release a new version of the meshing library is available. It is not yet the default meshing tool. The user will have to active it through the control cards. It is expected that in a typical application an improvement on quality robustness and performance is observed.
There is also a new strategy for mesh motion aimed at rotating problems. This new technique will allow the solution of sliding mesh problems where the blades of the rotor can be considered as flexible.

4.2 Free Surface Improvements

An increased interest in the free surface solver for sloshing and dam break problems has shown the need to improve the fidelity of this functionality. In the current version an improved interface capturing method is presented with reduced numerical viscosity and smoother free surface (see Fig.6).

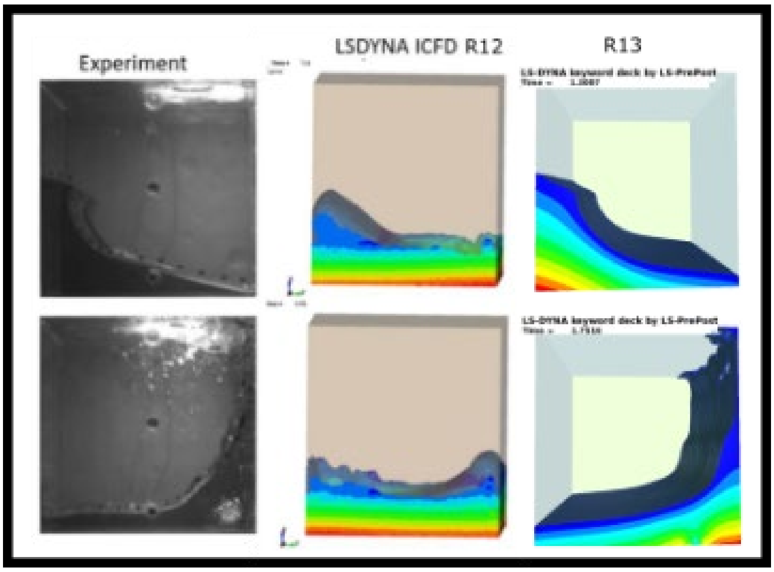

Fig.6: Comparisson between experiment, R12 and the improved R13 solution for the level set.

4.3 Fluid Structure Interaction

There is a new feature in R13 that allows the simulation of solid parts that are sliding and interacting with fluid. Such is the case of a clutch system with cooling channels (see Fig. 7).

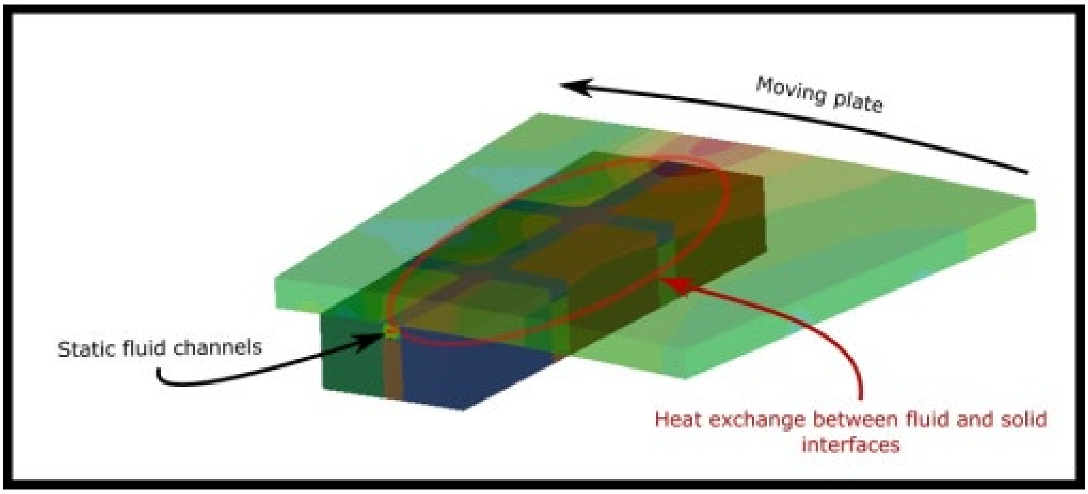

Fig.7: Clutch cooling simulation. The heat generated through friction between the moving plate and the pad is cooled down using fluid channels.

There is a new development aimed at improving the fidelity of non-slip boundary conditions in cases where embeded shells intersect with each other or walls in the domain. The intersection of embedded shells creates triple points wich results in non-watertight meshes where fluid can spill through the mesh. The new approach prevents this effect isolating the mesh effectively and preventing any fluid escapement.
Another useful new features allows an FSI problem to be initialized with a steady state fluid solution which helped achive better results in some models.

4.4 Multi-species Transport

Multi-Species transport is added for users interested in FSI simulations of intradermic injections. A one specie transport model coupled with FSI, porous media flow solver and heat transfer was added.

5 Roadmap and future development

The roadmap for the current and coming year has been inspired by user feedback as usual. The ACE organization in Ansys interacts closely with developers to better understand the features that we offer and to match the user needs with the best internal tool. From this feedback it is clear that high fidelity strongly coupled FSI solutions are a priority with a strong emphasis in biomedical applications. The current line of work aims at providing solutions in the field of prosthetic heart valve simulations, intradermal injection simulation and other biomedical devices. There is an increased interest in the free surface solver for problem like sloshing and dam break and this is a line of work where resources are working to improve the solution. There are also several applications in the automotive industry that will benefit from these developments as is the case of Ro-Dip simulation, where free surface and fluid structure interaction play an important role. Finally better integration with some of Ansys tools is being explored with emphasis on pre-processing. One approach is a workflow where a geometry manipulated and meshed in Ansys SpaceClaim and the mesh is automatically imported by LS-PrePost MS-explorer to set up the model.