Generalized Porous Media Flow in ICFD-LSDYNA: FSI, Free-Surface, RTM and Parachute Modeling

Rodrigo R. Paz[1], Facundo Del Pin[1], Iñaki Caldichoury[1] and Hugo G. Castro[2]

[1]Livermore Software Technology Corporation, USA.

[2]National Council for Scientific and Technological Research, Argentina.

Abstract

Large industries, engineering companies and academia have needs that constantly evolve, requiring models capable to accurately describe problems with increasing complexity. In this regard, the coupling of multiple fields representing different physical models (or phenomena) is of vital importance in these industries and it must be seriously observed by scientist, engineers and developers of well-established numerical codes, like LS-DYNA. Physical and engineering processes involving fluid flows through free and porous mediums are present in a wide range of these kind of problems.

In the ground vehicles industry, understanding aerodynamics phenomena allows us to optimize the operation of a wide spectrum of road vehicles, that ranges from road passenger transport (cars, buses, trains) to road commercial transport (trucks and trains). Road vehicle aerodynamics is a complex topic due to the interaction between the air flow and the ground and some parts (that play an important role in drag and lift development) could be treated as a porous media (e.g. the radiator, the condenser, air filters, etc). Industries like aerospace and those related to oil production have increased their trustfulness on numerical models and codes for the design, research, production and verification of highly critical parts and production processes. Most of these industries have adopted manufacturing procedures involving composites materials in liquid state, like the High Pressure Resin Transfer Molding (HP-RTM) methods, where a Newtonian (or Non-Newtonian) fluid flows through highly anisotropic matrices filling an initially empty container.

Parachute deploying and flight is another example in which the coupling of several fields is needed in order to describe its dynamics and evaluate different designs. This case, where a stream of an incompressible fluid flows through a thin and deformable porous fabric, represents a challenging problem from the numerical point of view that can be tackled considering a strong fluid-structure interaction (FSI) strategy.

This article reviews the numerical modeling of flows through general anisotropic porous media in LS-DYNA and introduces new aspects when considering FSI problems. A generalization of the Navier-Stokes equations that will allow the definition of sub-domains with different permeability/porosity is used. The SUPG|OSS stabilizing Finite Element Method for the spatial approximation and the second-order Fractional Step Method for the time integration were adopted. Also, the paper will provide some examples showing the use of LS-DYNA in a wide range FSI problems. Details about how the users' interface looks like will be given at the conference talk (it also can be found in the LS-DYNA users manual).

Keywords: `computational fluid dynamics, anisotropic/isotropic porous media flow, multi-physics, free surface flows, parachute modeling.`

1 Introduction

The natural generalization of the Navier-Stokes model for the incompressible fluid dynamics is the key point of the multi-physics coupling and porous media flow solver developed in LS-DYNA. This approach allows to user to model problems of 3D fluid infiltration through a porous media matrix with a moving free surface, like in Resin Transfer Molding processes (RTM) and flow through the porous fabric of a parachute in flight. Infiltration process basically depends on the local porosity, the local orientation of the fibers and on their packing density. In LS-DYNA, such a problem can be modeled with the ICFD+Multi-physics solver using an implicit second-order Fractional-Step time integration. Basically, the porous media solver implements the Ergun Correlation and the Darcy-Forchheimer force models in both the anisotropic and the isotropic forms. The fluid constitutive relation, describing the fluid stresses as function of the fluid velocity field could play an important role in RTM processes. For this purposes, LS-DYNA implements not only the classical Newtonian model but also a non-Newtonian model based on the power law. The fluid temperature field transport is also coupled to the structural thermal solver through the Conjugate Heat procedure described in previous articles.

With the multi-physics capabilities of LS-DYNA engineers can couple CFD analysis with a thermal and/or structural solver in order to solve more complex and realistic problems in industry and general engineering. In particular, the generalization of the Navier-Stokes equations allows the definition of sub-domains with different permeability/porosity. The SUPG|OSS stabilizing Finite Element Method for the spatial approximation and the second-order Fractional Step Method for the time integration were adopted.

2 The General Incompressible CFD solver in LS-DYNA.

In recent years LSTC has devoted big efforts in the development of a CFD solver for incompressible flows. The solver is specifically designed to tackle coupled problems where low Mach numbers (M<0.3) are involved and a scalable parallel solution is needed. Some features of the solver include:

1. Implicit solver to allow larger time steps,
2. Optimal MPP scalability,
3. Automatic mesh generation including boundary layer mesh,
4. Weak/Strong Fluid/Structure Interaction capabilities,
5. Turbulence models for RANS/LES, and turbulent inlet boundary conditions,
6. Generalized flow through porous media,
7. Free surface flows,
8. Coupled to the structural and thermal solvers.

Extensive validation has been performed to test the accuracy and robustness of the solver. The tests are documented and available thourgh our website.

3 Generalized Anisotropic/Isotropic Navier-Stokes flows through Porous Media.

3.1 General Model

A generalization of the Navier Stokes equations (see referece [1]) that will allow the definition of sub-domains with different permeability/porosity by means of the ***ICFD_PART** and ***ICFD_PART_VOL** keywords was implemented. Material parameters are introduced via ***ICFD_MAT** and ***ICFD_MODEL_POROUS** keywords (see user's manual). When using coupled Navier-Stokes/Porous Media a ***MESH_INTERF** needs to be defined in the interface between porous and non-porous regions. We use an OSS|SUPG stabilized Finite Element Method for the spatial approximation and an implicit second order Fractional-Step scheme for the time integration.

Basically, nine different models where implemented and tested:

i) the isotropic Ergun correlation,

ii) the Darcy-Forchheimer force model,

iii) a model for the definition of porous parameters using a pressure-velocity curve obtained from experiments,

iv) the General Anistropic Darcy-Forchheimer model with fixed local reference frame,

v) the Anisotropic porous media model with moving local reference frame and permeability vector in local reference frame defined by three Pressure-Velocity curves,

vi) and **vii)** Anisotropic porous media model with moving local referenceframe and constant permeability vector,

viii) the FSI flow through an isotropic porous fabric, used in parachute modeling,

ix) the Anisotropic porous media flow model with variable permeability tensor field which is the result of a solid dynamic problems.

Models **i)** through **vii)** were validated in a previous article **[2]**.

3.2 Resin Transfer Molding in an Anisotropic Multi-Porous Domain.

The following example is a RTM problem where a Newtonian fluid is injected into a mold at high speed. The porous domains consists in two highly anisotropic regions embedded in an isotropic matrix (see Figure 1). In Figure 2 the sequence of solutions for the velocity field and the position of the free surface is shown for several time steps.

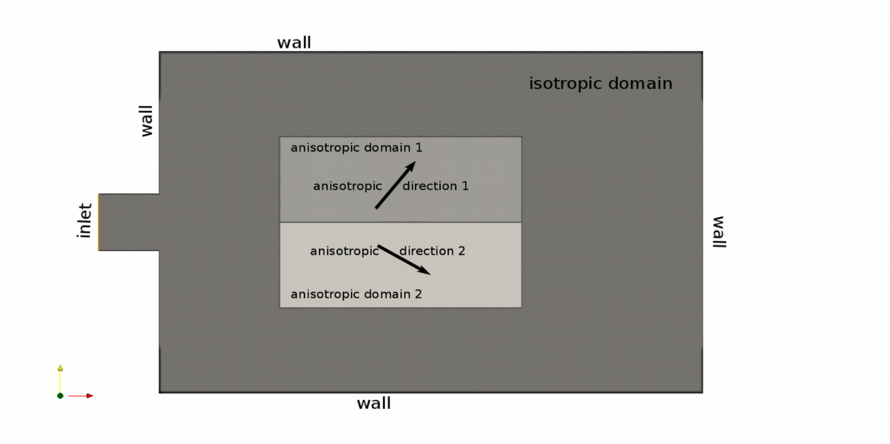

Figure 1: Domain and problem definition.

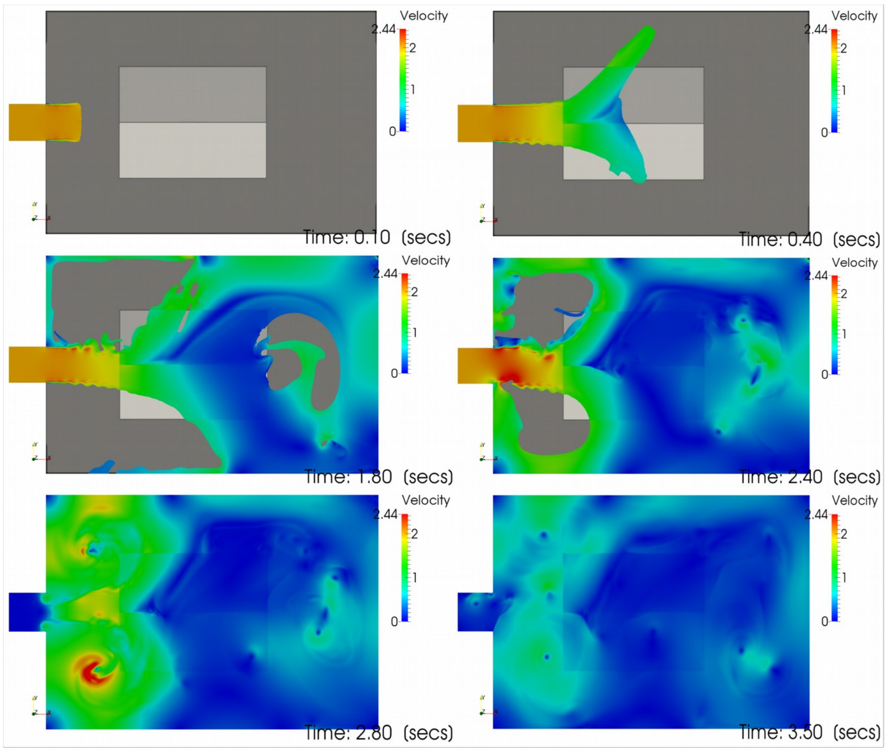

Figure 2: RTM sequence in multi-porous domain.

3.3 Generalized Anisotropic Flow Through Deforming Porous Media Solids.

A new Anisotropic Porous Media flow model (PM model ID=9) is introduced. In this new model, it is possible to use a variable permeability tensor field which could be the result of a solid dynamic problem of a solid porous domain where a fluid is passing through. The model reads the solid mesh and solid state field; and maps elemental permeability tensors and solid displacements to the fluid mesh. At t = 0 the solid mesh and the *MESH_INTERF defining the fluid porous media domain must match. Per-solid-element permeability tensor is computed by the solid dynamics solver and saved to files at needed time steps. Linear interpolation between time steps for mesh displacements and permeabilities if $\Delta t_{fluid} < \Delta t_{solid}$. The LSDYNA file format loaded by the ICFD solver at each time step has the format of "indyna" files; i.e.

```
*NODE
1 0.0 0.0 0.0
2 0.0 0.0 0.5
...
*ELEMENT_SOLID
1 1 6 96 98 33 53 99 179 119
...
*INITIAL_STRESS_SOLID
$for each solid/hexa element (we only used Kij)
1 1 9 1 0 0 0
0.0 0.0 0.0 0.0 0.0
0.0 0.0 K11 K12 K13
K21 K22 K23 K31 K32
K33
...
```

The ICFD user interface is the following:

```
*ICFD_MODEL_POROUS
$ material ID=2, porous media model ID=9
2,9
$porosity, Forchheimer factor, nstepsolid, solidts, file-
basename
1.,0.1,8,0.02,solidst_

where:
nstepsolid: number of time steps for the solidproblem
           (number of files with solid state data),
solidts: time step for the solid problem (fluid time
step<=solidts),
file basename: filename pattern for solid state.
```

Figures 3 and 4 show the solid and fluid states for the incompressible flow through a deformable porous truss at t=0 secs and t=1 secs.

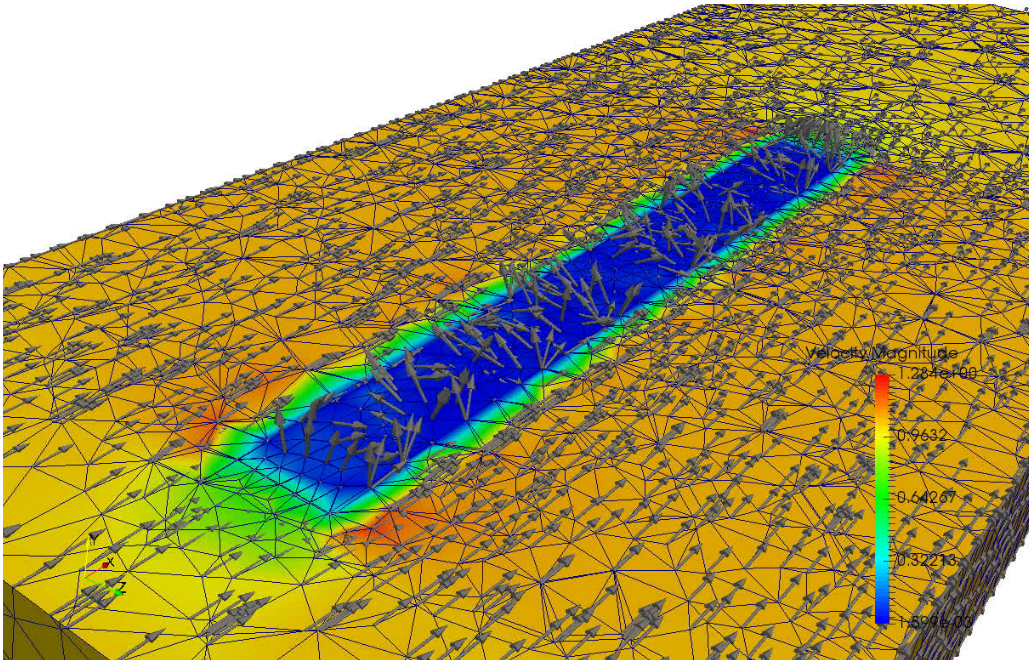

Figure 3

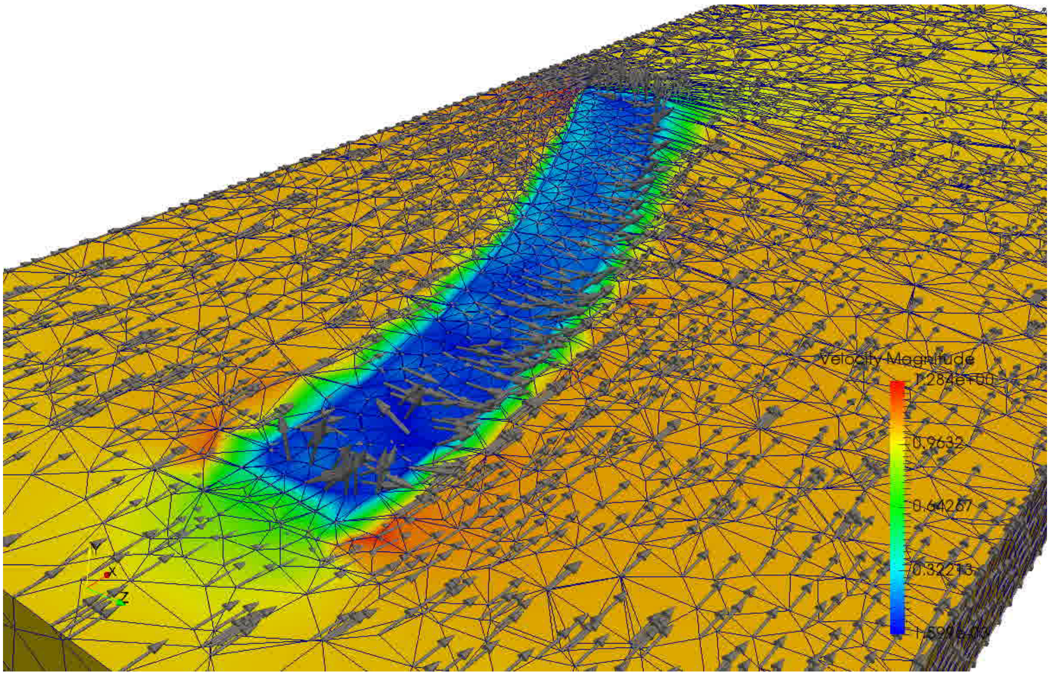

Figure 4

3.4 Parachute modeling in CFD Porous Media Solver: an FSI approach.

The coupling of CFD with structural solvers is generally referred as fluid structure interaction (FSI). The structural mechanics solver in LS-DYNA has two time integration schemes: explicit and implicit. The ICFD solver can couple to both of them but the type of coupling will differ. The coupling between ICFD and the explicit mechanics solver results in a weak coupling. On the other hand the coupling with the implicit solver results in a strong coupling. This will greatly impact the type of problems that each can tackle and thus choosing between explicit or implicit structural solvers is problem dependent. The user has to have good knowledge of the application that they are trying to simulate. A good rule of thumb is that light structures, as parachute fabrics, the strong coupling strategy should be used and thus solve the structure implicitly. The key point of this new model

In order to model the fluid flow through the parachute porous fabric we solve the 1-dimensional Darcy-Forchheimer flow equations along the thickness of the parachute structure allowing the fluid to pass through it. Then, the momentum equation is:

$$\partial p / \partial n = a(\mu, \kappa)(u - u_{mesh}) \cdot dn + b(\rho, \kappa, \epsilon)|u - u_{mesh}|(u - u_{mesh}) \cdot dn$$

where (p,u) is the fluid pressure and velocity fields, u_{mesh} the mesh velocity, n the normal to the parachute fabric (structure), ϵ is the fabric porosity, κ the fabric permeability, $a = \mu / \kappa$ and $b = F \epsilon \rho / \sqrt{\kappa}$.

The parachute CFD domain is defined using an ***EMBEDSHELL** structure (see Figure 5) where its thickness is the fabric thickness. The initial parachute surface mesh is generated as a regular surface shell in LS-PREPOST. The volume parachute mesh will be generated automatically by the ICFD volume and surface mesher at runtime.

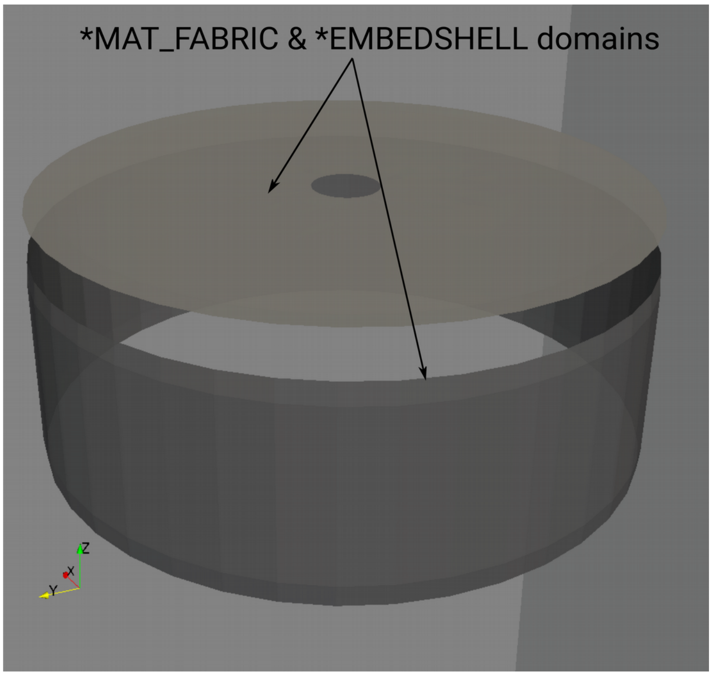

Figure 5

The FSI problem definition is through the following keywords:

```
*MAT_FABRIC
$   mid        ro          ea        eb        ec       prba      prca      prcb
      4 533.772 4.3090E+8   0.000     0.000  0.140000    0.000     0.000
```

```
*MESH_EMBEDSHELL
1
4, 5, 6
```

```
*ICFD_MODEL_POROUS
$ material ID=2, PM ID=8
2,8
$ porosity, permeab, frabric thickness
1., 0.001, 0.005
```

```
*ICFD_CONTROL_FSI
$ two-way coupling
0
*ICFD_BOUNDARY_FSI
$PID's
4,5,6
```

Figure 6 shows the fluid velocity field and the (conceptual) parachute shape of a generic payload is in a stationary free fall.

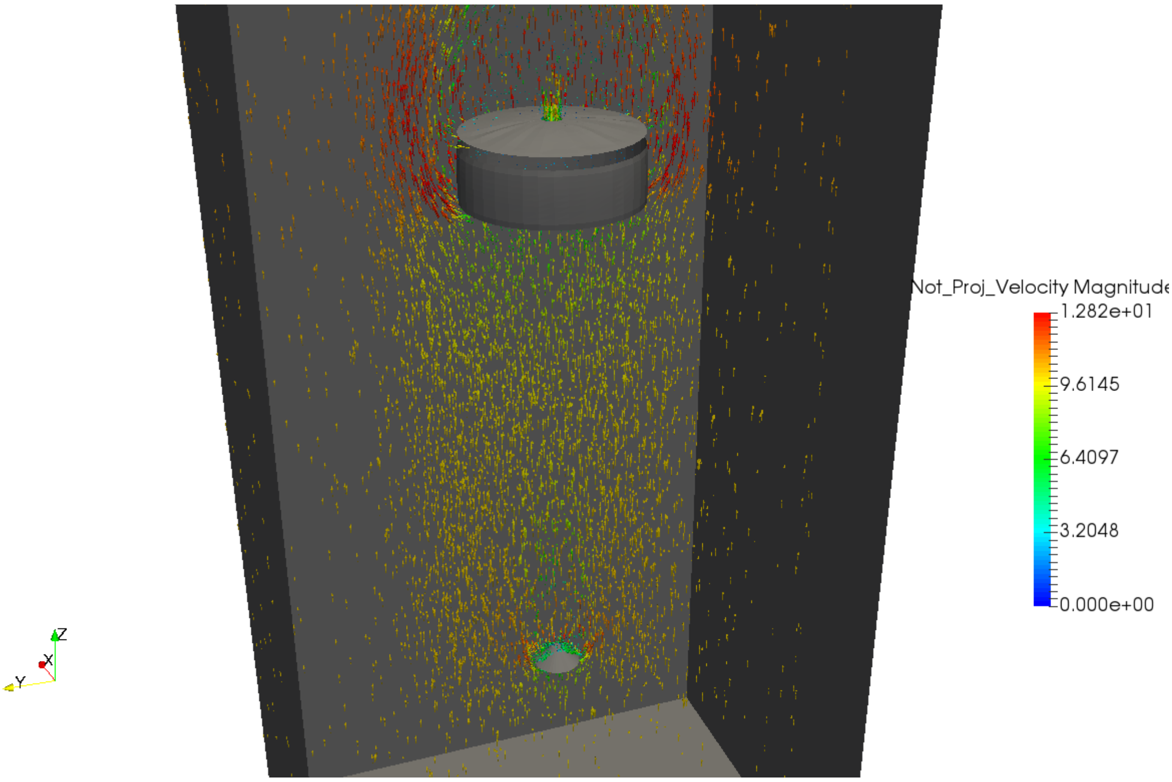

Figure 6

4 Summary.

This paper introduced some of the new features that LS-DYNA offers in the area of multi-physics, including general CFD flows, flows through anisotropic or isotropic multi-porous domains, Free-Surface, and Parachute FSI modeling. Some industry and engineering application problems were presented. Details on the users' interface will be given at the conference talk.

References

[1] The Finite Element Method for Fluid Dynamics, Sixth Edition''. Olek C. Zienkiewicz, Robert L. Taylor, P. Nithiarasu.

[2] Generalized Anisotropic/Isotropic Porous Media Flows in LS-DYNA. Rodrigo Paz, Facundo Del Pin, Iñaki Çaldichoury, Hugo G. Castro. 10th European LS-DYNA Conference 2015, Würzburg, Germany.