Proceedings of PVP2006-ICPVT-11
2006 ASME Pressure Vessels and Piping Division Conference
July 23-27, 2006, Vancoucer, BC, Canada

PVP2006-ICPVT-11-93430

EULER-LAGRANGE COUPLING WITH DEFORMABLE POROUS SHELLS

N. Aquelet
Livermore Software Technology Corporation
7374 Las Positas Road, Livermore, CA 94550, USA
aquelet@lstc.com

J. Wang
Livermore Software Technology Corporation
7374 Las Positas Road, Livermore, CA 94550, USA
jason@lstc.com

B.A.Tutt
Irvin Aerospace Inc.
3701 West Warner Avenue,
Santa Ana, CA 92704, USA
Btutt@irvinaerospace.com

I.Do
Livermore Software Technology Corporation
7374 Las Positas Road,
Livermore, CA 94550, USA
ian@lstc.com

H.Chen
Livermore Software Technology Corporation
7374 Las Positas Road,
Livermore, CA 94550, USA
hao@lstc.com

ABSTRACT

A newly developed approach for tridimensional fluid-structure interaction with a deformable thin porous media is presented under the framework of the LS-DYNA software. The method presented couples a Arbitrary Lagrange Euler formulation for the fluid dynamics and a updated Lagrangian finite element formulation for the thin porous medium dynamics. The interaction between the fluid and porous medium are handled by a Euler-Lagrange coupling, for which the fluid and structure meshes are superimposed without matching. The coupling force is computed with an anisotropic Ergun porous flow model. As test case, the method is applied to an anchored porous MIL-c-7020 type III fabric placed in an air stream.

INTRODUCTION

Many important engineering applications such as airbags, parachute, to name a few, involve transient flows in deformable porous media [1]. In general the deformable porous media problem should be described at both the micro and macro scale. The problem at the microscopic level has a deformable skeleton surrounded by one or several fluids. At the macroscopic level the solid is usually described by the Lame equations of linear elasticity and the fluid by the Navier-Stokes equations. The first simple model of a mechanical system comprised of a deformable porous solid matrix filled with a fluid has been developed by Biot [2] who formulated the macroscopic equations for the effective medium. The application of asymptotic homogenization methods [3,4] has lead to theoretical justification of Biot's equation [1,5,6] along with appropriate pore problems from which the macroscopic parameters can be computed numerically. The macroscopic equations are derived under the assumption that the solid-fluid interface displacements are small compared to the pore size. For a large class of poroelastic problems it is not possible to derive macroscopic equations. In general the upscaling problem for poroelasticity medium is not separable even when the pores are well separated. This is due to the fact that the skeleton can deform arbitrarily large due to different parameters such as macroscopic displacements, pressure and velocity. However it is not practical to model the flow at the pore scale and undesirable to have to gather the tremendous amount of fine scale data that is required to model an entire parachute for example. Moreover present computational resources are not able to handle flow simulations of this size. Hence the models in this paper describe the essential physical behaviour in an averaged sense at the mega scale without modelling finer scale details. This assumption for thin porous media such as parachute seems reasonable. The focus of this paper is the case when, due to problem parameters such as macroscopic pressure and velocity field for a MIL-c-7020 type III fabric, the deformation of the fluid-solid interface is not considerable at the pore level. This interface could be approximated by a rigid motion of its initial position. Thus the porous coupling force is computed by using the Ergun equation [7] with a constant porosity.

In this work, the governing equations for fluid and thin porous medium problem are first formulated together with boundary conditions. Then a description of the porous Euler-Lagrange coupling algorithm is presented. Further, this

numerical method is applied to a porous Fluid Structure Interaction problem by comparing the numerical results to experimental data.

DESCRIPTION OF FLUID AND STRUCTURE PROBLEMS

The fluid is solved by using an Eulerian formulation [8] on a Cartesian grid that overlaps the porous structure, while this latter is discretised by Lagrangian shells based on the Belytschko-Lin-Tsay formulation [9].

Eulerian description of Navier-Stokes equations

For simplicity, the numerical simulations in this paper have been restricted to an Eulerian formulation for the fluid, although the formulation can be extended to an ALE formulation. The Eulerian formulation is a particular case of the ALE finite element formulation. Thus a general ALE point of view is first adopted to solve the Navier-Stokes equations before presenting the Eulerian formulation.

In the ALE description of motion, an arbitrary referential coordinate is introduced in addition to the Lagrangian and Eulerian coordinates[11]. The total time derivative of a variable f with respect to a reference coordinate can be described as Eq.(1):

$$\frac{df(\vec{X},t)}{dt} = \frac{\partial f(\vec{x},t)}{\partial t} + \vec{c}.\overrightarrow{grad} f(\vec{x},t) \tag{1}$$

$$\vec{c} = \vec{v} - \vec{w} \tag{2}$$

where $\vec{X}$ is the Lagrangian coordinate, $\vec{x}$ is the ALE coordinate, $\vec{v}$ is the particle velocity and $\vec{w}$ is the velocity of the reference coordinate, which will represent the grid velocity for the numerical simulation, and the system of reference will be later the ALE grid. In some papers [10, 11] $\vec{c}$ is referred as the convective velocity.

Thus substituting the relationship between the total time derivative and the reference configuration time derivative derives the ALE equations.

Let $\Omega^f \in R^3$, represent the domain occupied by the fluid, and let $\partial\Omega^f$ denote its boundary. The equations of mass, momentum and energy conservation for a Newtonian fluid in ALE formulation in the reference domain, are given by:

$$\frac{\partial \rho}{\partial t} + \rho div(\vec{v}) + (\vec{v} - \vec{w}) grad(\rho) = 0 \tag{3}$$

$$\rho \frac{\partial \vec{v}}{\partial t} + \rho(\vec{v} - \vec{w}).\overline{\overline{grad}}(\vec{v}) = \overrightarrow{div}(\overline{\overline{\sigma}}) + \vec{f} \tag{4}$$

$$\rho \frac{\partial e}{\partial t} + \rho(\vec{v} - \vec{w}).\overrightarrow{grad}(e) = \overline{\overline{\sigma}} : \overline{\overline{grad}}(\vec{v}) + \vec{f}.\vec{v} \tag{5}$$

where ρ is the density and $\overline{\overline{\sigma}}$ is the total Cauchy stress given by:

$$\overline{\overline{\sigma}} = -p.\overline{\overline{Id}} + \mu(\overline{\overline{grad}}(\vec{v}) + \overline{\overline{grad}}(\vec{v})^T) \tag{6}$$

where p is the pressure and μ is the dynamic viscosity. Equations (3)-(5) are completed with appropriate boundary conditions. The part of the boundary at which the velocity is assumed to be specified is denoted by $\partial\Omega_1^f$. The inflow boundary condition is:

$$\vec{v} = \vec{g}(t) \quad on \quad \partial\Omega_1^f \tag{7}$$

The traction boundary condition associated with Eq.(4) are the conditions on stress components. These conditions are assumed to be imposed on the remaining part of the boundary.

$$\overline{\overline{\sigma}}.\vec{n} = \vec{h}(t) \quad on \quad \partial\Omega_2^f \tag{8}$$

One of the major difficulties in time integration of the ALE Navier-Stokes equations (3)-(5) is due to the nonlinear term related to the relative velocity $(\vec{v} - \vec{w})$. For some ALE formulations, the mesh velocity can be solved using a remeshing and smoothing process. In the Eulerian formulation, the mesh velocity $\vec{w} = \vec{0}$, this assumption eliminates the remeshing and smoothing process, but does not simplify the Navier-Stokes equations (3)-(5). To solve equations (3)-(5), the split approach detailed in [8],[12] and implemented in most hydrocodes such as LS-DYNA3D is adopted in this paper.

Operator splitting is a convenient method for breaking complicated problems into series of less complicated problems. In this approach, first a Lagrangian phase is performed, using an explicit finite element method, in which the mesh moves with the fluid particle. In the CFD community, this phase is referred to as a linear Stokes problem. In this phase, the changes in velocity, pressure and internal energy due to external and internal forces are computed. The equilibrium equations for the Lagrangian phase are:

$$\rho \frac{d\vec{v}}{dt} = \overrightarrow{div}(\overline{\overline{\sigma}}) + \vec{f} \tag{9}$$

$$\rho \frac{de}{dt} = \overline{\overline{\sigma}} : \overline{\overline{grad}}(\vec{v}) + \vec{f}.\vec{v} \tag{10}$$

The mass conservation equation is used in its integrated form Eq.(11) rather than as a partial differential equation [11]. Although the continuity equation can be used to obtain the density in a Lagrangian formulation, it is simpler and more accurate to use the integrated form Eq.(11) in order to compute the current density ρ :

$$\rho J = \rho_0 \tag{11}$$

where ρ_0 is the initial density and J is the volumetric strain given by the Jacobian:

$$J = \det\left(\frac{\partial x_i}{\partial X_j}\right) \tag{12}$$

In the second phase, called advection or transport phase, the transportation of mass, momentum and energy across element boundaries are computed. This may be thought of as remapping the displaced mesh at the Lagrangian phase back to its initial position. The transport equations for the advection phase are:

$$\begin{aligned} &\frac{\partial \phi}{\partial t} + \vec{c}.\overrightarrow{grad}(\phi) = 0 \\ &\phi(\vec{x},0) = \phi_0(x) \end{aligned} \tag{13}$$

The hyperbolic equation system (13) is solved by using a finite volume method. Either a first order upwind method or second order Van Leer advection algorithm [13] can be used to solve Eq.(13). The advection method is successively applied for the conservative variables: mass, momentum and energy with initial condition $\phi_0(x)$, which is the solution from the Lagrangian calculation of equations (9)-(10) at the current time. In Eq.(13), the time *t* is a fictitious time: in this paper, time step is not updated when solving for the transport equation. There are different ways of splitting the Navier-Stokes problems. In some split methods, each of the Stokes problem and transport equation are solved successively for half time step. The following paragraph presents the description of the structure.

Macroscopic governing Equations for porous thin media

In this paragraph the porous structure problem is described at the macroscopic scale and the Belytschko-Lin-Tsay shell formulation [9] employed to model the thin porous medium is compared to the Hughes Liu shell formulation [14].

Let $\Omega^s \in R^3$, the domain occupied by the porous structure, and let $\partial\Omega^s$ denote its boundary. An updated Lagrangian finite element formulation is considered: the movement of the thin porous medium Ω^s described by $x_i(t),(i=1,2,3)$ can be expressed in terms of the reference coordinates $X_i(t),(i=1,2,3)$ and time t:

$$x_i = x_i(X_\alpha, t) \tag{14}$$

The momentum equation is given by Eq.(16) in which $\overline{\overline{\sigma}}$ is the Cauchy stress, ρ is the density, f is the force density, $\frac{d\vec{v}}{dt}$ is acceleration and $\vec{n}$ is the unit normal oriented outward at the boundary $\partial\Omega^s$:

$$\rho\frac{d\vec{v}}{dt} = \overrightarrow{div}(\overline{\overline{\sigma}}) + \vec{f} \tag{15}$$

The solution of Eq.(15) satisfies the displacement boundary condition Eq.(16) on the boundary $\partial\Omega_1^s$ and the traction boundary condition Eq.(17) on the boundary $\partial\Omega_2^s$.

$$\vec{x}(\vec{X},t) = \vec{D}(t) \quad \text{on} \quad \partial\Omega_1^s \tag{16}$$

$$\overline{\overline{\sigma}}.\vec{n} = \vec{\tau}(t) \quad \text{on} \quad \partial\Omega_2^s \tag{17}$$

In this paper, the shell formulation used to model the canopy in the channel is the Belytschko-Lin-Tsay formulation [9]. The Belytschko-Lin-Tsay shell 4-node element is based on a co-rotational coordinate system and a constitutive computation using a rate of deformation. The embedded element coordinate system that deforms with the element is defined in term of four corner nodes. As the element deforms, an angle may exist between the fiber direction and the unit normal of the element coordinate system. The magnitude of this angle is limited in order to keep a plane shell geometry. In this local system, the Reissner-Mindlin theory gives the velocity of any point in the shell according to the velocity of mid-surface and the rotations of the element's fibers.

Then, the rates of deformation are computed at the center of the element. The new Cauchy stresses are computed by using the material model and by accounting for the incremental rotation, $\overline{\overline{\Delta R}}$. For the Hughes-Liu family of shell elements [14], $\overline{\overline{\Delta R}}$ is estimated by using an approximation of the Jaumann rate. Therefore, in every integration points, the instantaneous rotation field is computed. Moreover, since the Jaumann rate update is performed in the global system, the stresses and the rates of deformations are rotated from the global coordinate system to the local coordinate system and, after the update, the new stresses are rotated back to the global system. Thus, the Jaumann rate rotation requires the most operation cost in the Hughes-Liu shell process. For the Belytschko family of elements, the incremental rotation is obtained by expressing the element base vectors at t(n+1) in the local system at t(n). Since the material rotation is equal to the rotation of the local system, $\overline{\overline{\Delta R}}$ is the identity matrix. This involves the Belytschko-Lin-Tsay shell element is a computationally efficient alternative to the Hughes-Liu shell element. Then, the element-centered resultant forces and moments are obtained by integrating the stresses through the thickness of the shell. The relations between these forces and moments and the local nodal forces and moments are obtained by performing the principle of virtual power with one point quadrature. Finally, the global nodal forces and moments are derived by using the transformation relations defined by the global components of the corotational unit vectors.

The following section presents the porous Euler-Lagrange coupling method, which handles the fluid - porous structure problem.

FLUID – POROUS STRUCTURE INTERACTION

The Lagrangian finite element formulation uses a computational mesh that follows the material deformation. This approach is efficient and accurate for problems involving moderate deformations like structure motions or flows that are essentially smooth. When this latter departs from this kind of smoothness, the ALE or Eulerian formulation must be used because the finite element mesh is allowed to move independently from the material flow. This takes away all problems associated with distorted mesh that are commonly encountered with a Lagrangian. In this paper the Euler Lagrange coupling using Eulerian formulation for the fluid, is more suitable for solving fast transient porous fluid-structure interaction problems. First, the Eulerian formulation is able to simulate fluid large deformations and second, the coupling can handle the interaction between the fluid and thin porous medium. This method can be described as Eulerian contact. The following paragraph presents the principle of the coupling .

In an explicit time integration problem, the main part of the procedure in the time step is the calculation of the nodal forces. After computation of fluid and structure nodal forces, we compute the forces due to the coupling, these will only affect nodes that are on the fluid - porous structure interface. For each structure node, a depth penetration $\vec{d}$ is incrementally updated at each time step, using the relative velocity $\overrightarrow{v_{rel}}$ at the slave and master node. For this coupling, the slave node is a structure mesh node, whereas the master node is not a fluid mesh node, it can be viewed as a fluid particle within a fluid element, with mass and velocity interpolated from the fluid element nodes using finite element shape functions. The location of the master node is also computed using the isoparametric coordinates of the fluid element. If $\vec{d}^{\,n}$ represents the penetration depth at time $t = t^n$, it is incrementally updated in Eq.(18):

$$\vec{d}^{\,n+1} = \vec{d}^{\,n} + \overrightarrow{v_{rel}}^{\,n+1/2}.\Delta t \tag{18}$$

In Eq.(18) $\overrightarrow{v_{rel}}^{\,n+1/2} = \overrightarrow{v_s}^{\,n+1/2} - \overrightarrow{v_f}^{\,n+1/2}$ in which the fluid velocity $\overrightarrow{v_f}$ is the velocity at the master node location and the structure velocity $\overrightarrow{v_s}$ is the velocity at the slave node location. The coupling acts only if penetration occurs, $\overrightarrow{n_s}.\vec{d}^{\,n} < 0$, where $\overrightarrow{n_s}$ is built up by averaging normals of structure elements connected to the structure node. The porous coupling forces are derived from the integration of the Ergun Equation [7] on the shell volume:

$$\frac{dp}{d\hat{z}} = a(\mu,\varepsilon)\overrightarrow{v_{rel}}.\overrightarrow{n_s} + b(\rho,\varepsilon)\left(\overrightarrow{v_{rel}}.\overrightarrow{n_s}\right)^2 \tag{19}$$

in which $\hat{z}$ is the local position along the fiber direction of the shell element and ε is the porosity. The coefficient $a(\mu,\varepsilon)$ is the reciprocal permeability of the porous shell or viscous coefficient. $b(\rho,\varepsilon)$ represents the inertia coefficient. For flows under very viscous conditions the second term in Eq.(19), which represents the inertia effects drops out and the Blake-Kozeny equation for laminar flows in porous media is obtained. At high rates of flow it is the first term or viscous term, which drops out and the Blurke-Plummer equation for turbulent flows in porous media is obtained.

The force F derived from Eq.(19) is applied to both master and slave nodes in opposite directions to satisfy force equilibrium at the interface coupling, and thus the coupling is consistent with the fluid-structure interface condition namely the action-reaction principle. At the structure coupling node, we applied a force:

$$F_s = -F \tag{20}$$

whereas for the fluid, the porous coupling force is distributed to the fluid element nodes based on the shape functions, at each node i (i=1,..,8), the fluid force is scaled by the shape function N_i :

$$F_f^i = N_i.F \tag{21}$$

Where N_i is the shape function at node i. Since $\sum_{i=1}^{8} F_f^i = F$, the action-reaction principle is satisfied at the coupling interface.

The following paragraph presents the application of this approach to a porous membrane in a channel flow.

NUMERICAL APPLICATION

A channel problem with a porous membrane is modeled by the porous Euler-Lagrange coupling method. First the viscous and inertia coefficients must be deterrmined by using the experimental permeability curve of the MIL-c-7020 type III fabric [15] (see Fig.1). Second the value of these parameters are checked with the channel model.

Determination of the viscous and inertia parameters

As indicated in the introduction the porosity of the canopy is assumed constant. At the steady state the air density and dynamic viscosity are supposed uniform. Under these assumptions the viscous and inertia parameters in Eq.(19) are constant. The experimental permeability curve of the MIL-c-7020 type III fabric gives the rate of flow through the nylon canopy versus the pressure drop. To determine the viscous and inertia parameters in Eq.(19), the Ergun theoretical permeability should be a parabolical fit of the experimental one. Thus the coefficients ax and bx were computed by solving the following system:

$$\begin{cases} dp_1 / e = a.v_1 + b.v_1^2 \\ dp_2 / e = a.v_2 + b.v_2^2 \end{cases} \tag{22}$$

where $e = 0.1016mm$ is the shell thickness and the couple of points (v_1, dp_1) and (v_2, dp_2) was chosen on the experimental plot so that the Ergun equation fits it as close as possible. The values of the viscous and inertia parameters are $a = 1599174kg.m^{-2}.s^{-1}$ and $b = 480514kg.m^{-3}$.

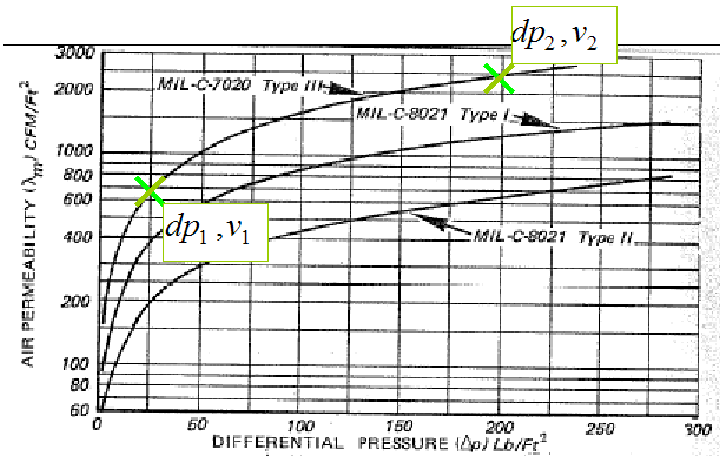

Figure.1 EXPERIMENTAL POROSITY CURVES [15]

A better approach to determine these coefficients should be to employ the following equations derived from the Ergun theory:

$$a = \frac{150\mu(1-\varepsilon)^2}{D^2\varepsilon^3} \tag{23}$$

$$b = \frac{1.75\rho(1-\varepsilon)}{D\varepsilon^3} \tag{24}$$

where ε is the porosity : $\varepsilon = \frac{v_{void}}{v_{total}}$ and D is a characteristic length defined by: $D = \frac{6(1-\varepsilon)V}{S}$ with V, the volume of the canopy and S, the "wetted" surface. However it is tricky to get the porosity of the MIL-c-7020 type III fabric in the literature. The following application is dedicated to the validation of these parameters.

Channel model

The model is a channel with a constant prescribed rate of flow at the inlet. The channel sketched on Fig.2 is a Eulerian mesh of 3000 cubic solid elements based on the fluid formulation described previously. The first layer of solid elements on Fig.2 is composed of ambient or reservoir elements with a constant pressure. The square section of the channel is $100m^2$. A deformable nylon (MIL-c-7020 type III fabric, $\rho = 533.77kg.m^{-3}$, $E = 0.4309GPa$) shell occupies all one section of the channel, which is located at $2m$ from the inlet. This membrane is meshed by 100 Lagrangian Belytschko-Lin-Tsay square shell elements. The simulation time is enough large to reach the steady state. It is $20sec$. The run takes about $2h$ on a AMD Opteron Processor 248 (CPU: 2GHZ, cache size:1Mb) because of the time step is scaled down to avoid instability of the computation: $\Delta t = 55\mu s$. Actually the timestep needs to be adapted to prevent the run from crash. The higher the velocity is, the lighter the fabric is, the lower the time step should be.

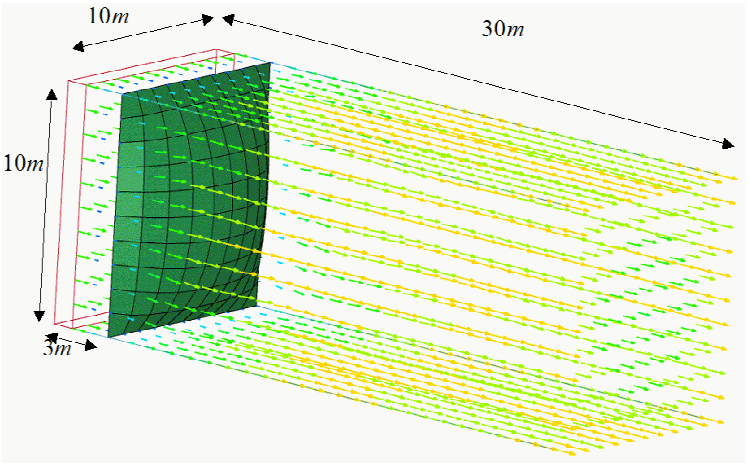

Figure 2. CHANNEL MODEL

The average pressure on the canopy at the steady state is post-treated for different inflow velocities. The purpose is to check if the porous behaviour of the fabric is well modeled. The rate of air flow through the deformed nylon shell enables to compute an average permeability velocity, for which the experimental and numerical pressure drops through the fabric are compared on Tab.1.

The slower the velocities are, the larger the relative errors on Tab.1 are. However the relative errors are acceptable. Thus the Ergun equation with $a = 1599174kg.m^{-2}.s^{-1}$ and $b = 480514kg.m^{-3}$ approximates well the porous behaviour of the MIL-c-7020 type III fabric which will make up the parachute canopy of a further analysis.

Table 1. NUMERICAL AND EXPERIMENTAL PRESSURE DROPS

Inflow velocity (*m/s*)	Permeability Velocity (*m/s*)	Experimental Pressure Drop (*Pa*)	Numerical Pressure Drop (*Pa*)	Relative Errors (%)
10	2.7	862	794	9%
20	4	1628	1478	10%
30	5.4	2490	2316	7%
40	6.4	2969	3104	4%
50	7	3735	3653	2%

CONCLUSION

This paper has described a method to solve fluid-structure interaction problem between a thin porous media and a fluid. These method was successfully applied to the problem of a porous membrane in a channel flow. The numerical and experimental results agree well. The prospective goals of this ongoing research will be to apply the porous coupling to a parachute problem and implement Ergun coefficients what will depend on the porosity, density and dynamic viscous.

REFERENCES

[1] Bear, J., 1972, "Dynamics of fluids in porous media," Dover, New York.

[2] Biot, M.A., 1941, "General theory of three dimensional consolidation," J. Appl. Phys., **12**, pp.155–164.

[3] Bensoussan, A., Lions, J.L., Papanicolaou, G., 1978, "Asymptotic analysis for periodic structures," Studies in Mathematics and Its Applications, **5**, North-Holland, Amsterdam.

[4] Zhikov, V.V., Kozlov, S.M., Oleinik, O.A. 1994, "Homogenization of differential operators and integral functionals," Springer-Verlag, Berlin.

[5] Sanchez-Palencia E., 1981, "Non-homogeneous media and vibration theory," Lecture Notes in Physics, **127**, Springer-Verlag, Berlin.

[6] Mei, C.C., Auriault, J.-L., 1989, "Mechanics of heterogeneous porous media with several spatial scales," Proc. Roy. Soc. Lond., **A 426**, pp.391–423.

[7] Ergun, S., 1952, "Fluid flow through packed beds," Chem. Eng. Prog., **48**(2), 89-94.

[8] Benson D.J., 1992, "Computational methods in Lagrangian and Eulerian hydrocodes," Computer Methods in Applied Mechanics and Engineering, **99**(2), pp.235-394.

[9] Belytschko T., Lin J., Tsay C.S., 1984, "Explicit algorithms for nonlinear dynamics of shells," Comp. Meth. Appl. Mech. Engrg., **42**, pp-225-251.

[10] Souli, M., Ouahsine, A., Lewin, L., 2000, "ALE and Fluid-Structure Interaction problems," Comput. Methods Appl. Mech. Engrg., **190**, pp.659-675.

[11] Belytschko T., Liu W.K., Moran B., "Nonlinear Finite Elements for Continua and Structures," John Wiley & Sons, LTD.

[12] Hughes, T.J.R., Liu, W.K., Zimmerman, T.K., 1981, "Lagrangian Eulerian finite element formulation for viscous flows," J. Comput. Methods Appl. Mech. Engrg., **21**, pp.329-349.

[13] Van Leer B., 1977, "Towards the Ultimate Conservative Difference Scheme.IV. A New Approach to Numerical Convection," Journal Computational Physics, **167**, pp.276-299.

[14] Hughes, T.J.R., Liu,W. K., 1981, "Nonlinear Finite Element Analysis: Part I. Two-Dimensional Shells, " Comput. Methods Appl. Mech. Engrg., **27**, pp.167-181.

[15] Air Force Flight Dynamics Laboratory Technical Report (AFFDL-TR-78-151), Recovery Systems Design Guide, June 1978.