FINITE POINTSET METHOD (FPM): A MESHFREE APPROACH FOR INCOMPRESSIBLE FLOW SIMULATIONS APPLIED TO COMPOSITE MATERIALS

Alain Trameçon[1], Patrick de Luca[1], Christophe Binetruy[2] and Jörg Kuhnert[3]

[1] *ESI-GROUP*
Parc d'affaire SILIC, 99 Rue des Solets, BP 80112 F-94513 RUNGIS, FRANCE:
atr@esi-group.com

[2] *Polymers and Composites Technology & Mechanical Engineering Department*
Ecole des Mines de Douai, 941 Rue Charles Bourseul, F-59508 DOUAI, France :
binetruy@ensm-douai.fr

[3] *Fraunhofer Institute for Industrial Mathematics (ITWM)*
Gottlieb-Daimler-Straße, D-67663 KAISERSLAUTERN, GERMANY:
kuhnert@itwm.fhg.de

ABSTRACT: A meshfree particle method is used to simulate resin flow through a complex network of fibers. Flows are modeled by the incompressible Navier-Stokes equations. The particle projection method is used to solve the Navier-Stokes equations. The spatial derivatives are approximated by the weighted least squares method (WLS). One application is presented regarding the application of the method to numerical permeability prediction, related to LCM processes.

KEYWORDS: Mesh-free method, incompressible Navier-Stokes equations, least squares (LSQ) approximation, LCM, permeability prediction

INTRODUCTION

Meshfree techniques

In this paper we present a mesh-free method called FPM (Finite Point Method) for simulations of resin transfer moulding through a complex network of fibers. This method was developed by Dr Jorg Kuhnert at ITWM in Germany. A fluid domain is first replaced by a discrete number of points, which are referred to as particles. Each particle carries all fluid information, like density, velocity, temperature etc. and moves with fluid velocity. Therefore, particles themselves can be considered as geometrical grids of the fluid domain. This method has some advantages over grid based techniques, for example, it can handle fluid domains, which change naturally, whereas grid based techniques require additional computational effort. Or they are able to simulate the flow in very complicated domains, which would be impossible or difficult to mesh.

A classical grid free Lagrangian method is Smoothed Particle Hydrodynamics (SPH), which was originally introduced to solve problems in astrophysics (Lucy 1977, Gingold et al. 1977). It has since been extended to simulate the compressible Euler equations in fluid dynamics and applied to a wide range of problems, see (Monaghan 92, Monaghan et al. 1983, Morris et al. 1997). The method has also been extended to simulate inviscid incompressible free surface flows (Monaghan 94). The implementation of the boundary conditions is the main problem of the SPH method.
Another approach for solving fluid dynamic equations in a grid free framework is the moving least squares or least squares method (Belytschko et al. 1996, Dilts 1996, Kuhnert 99, Kuhnert 2000, Tiwari et al. 2001 and 2000) which derived in the Finite Pointset Method (FPM). With this approach boundary conditions can be implemented in a natural way just by placing the particles on boundaries and prescribing boundary conditions on them (Kuhnert 99). The robustness of this method is shown by the simulation results in the field of airbag deployment in car industry. Here, the membrane (or boundary) of the airbag changes very rapidly in time and takes a quite complicated shape (Kuhnert et al. 2000).

FPM fluid code

FPM is a meshfree CFD finite difference code, mainly designed to overcome several drawbacks of classical CFD methods (Finite Element Method (FEM) , Finite Volume Method (FVM)). The main drawback of the classical methods (FEM,FVM) is the relatively expansive geometrical mesh-grid required to carry out all numerical computations. The computational cost to establish and maintain these grids becomes more dominant as the considered geometry becomes complex or moves in time. For several applications, the effort for grid maintenance is beyond acceptance, the computations take too long or fail completely. Thus, FPM opens up new fields of application in computational structural and fluid mechanics, or makes the handling of several problems much more easy.

General Equations

FPM is a mesh-free thermal CFD code for incompressible and compressible flows. FPM includes newtonian viscosity, natural convection, heat conduction, heat exchange at the boundaries.
FPM solves the general Navier Stokes equation as written below in a Lagrange form :

$$\begin{aligned} &\tfrac{d}{dt}\rho + \rho \cdot \nabla \mathbf{v} = 0 \\ &\tfrac{d}{dt}(\rho \mathbf{v}) + (\rho \mathbf{v}) \cdot \nabla \mathrm{v} + \nabla \mathrm{p} - \nabla \mathbf{S} = \rho \cdot \mathbf{g} \\ &\tfrac{d}{dt}(\rho \mathrm{E}) + (\rho \mathrm{E}) \cdot \nabla \mathbf{v} + \nabla(\mathrm{p} \cdot \mathbf{v}) - \nabla(\mathbf{S} \cdot \mathbf{v}) = (\rho \cdot \mathbf{g} \cdot \mathbf{v}) + \nabla \cdot \left(\mathrm{k} \nabla^{T} \mathrm{T}\right) \end{aligned} \tag{1}$$

v : fluid velocity
ρ : density
p : pressure
S : deviatoric stresses
g : gravity
T : temperature
E : specific total energy per unit mass

For incompressible flows, these equations can be simplified as follows :

$$\begin{aligned} &\tfrac{d}{dt}\rho = 0 \quad \Rightarrow \quad \nabla \mathbf{v} = 0 \\ &\tfrac{d}{dt}\mathbf{v} = -\frac{1}{\rho}\nabla p + \frac{\eta}{\rho}\cdot\Delta\mathbf{v} + \mathbf{g} \\ &\tfrac{d}{dt}T = \frac{1}{\rho\cdot c}\cdot\nabla(k\cdot\nabla T) \end{aligned} \qquad (2)$$

Moving Least Square (MLS) approximation

FPM does not require the support of a mesh and therefore values are known at discrete "interpolation" points, which do not have a fixed connection like finite elements between them. The list of neighbor points is determined for each point at each time step in order to construct afterwards a proper interpolation function as described in Fig.1. For this purpose, a smooth interpolation of the discrete function values is constructed using polynomial functions, best fitted to the discrete values using a moving least square method (Fig.2).

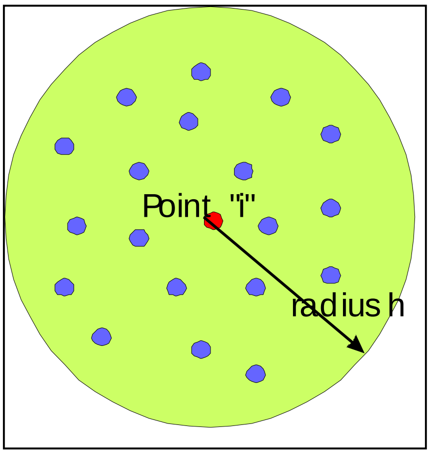

Fig. 1: Smoothing length

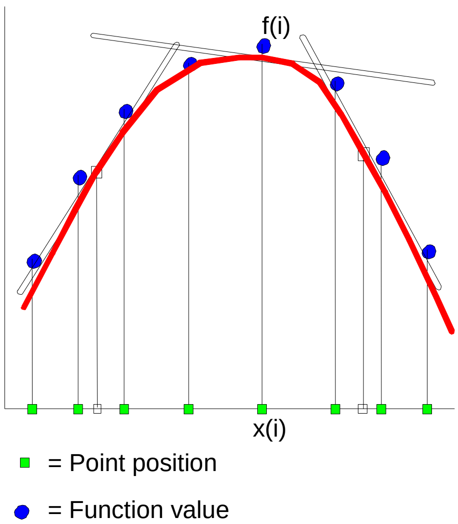

Fig. 2 : Moving least square method : f(i) stands for the value of point I of the function to be interpolated : pressure, density, velocities, temperature.

The interpolation function that is constructed is based upon the Moving Least Square (MLS) approximation. The idea is to find the local polynomial which minimizes the distance between the values at the discrete points and the approximated values on the function. This is done using a least square fit method as follows:

$$\sum_{i=1,N} W(\mathrm{x}-\mathrm{x_i})\left(f_i - \mathrm{p}_d(\mathrm{x},\mathrm{x_i})\right)^2 \overset{!}{=} \min \tag{3}$$

where d is the degree of the polynom, d>=2 for Navier-Stokes as second order derivatives are required.

The weight function W(x-xi) decreases with respect to the distance between the location x of the central point and the neighbor points xi, so that points which are far away from the central point will have less influence than points which are closer. The domain of interpolation is limited by a sphere of radius h, called the smoothing length, so that points which have a distance greater than h will have no interaction.
The interpolation function can afterwards be derived. For Navier Stokes incompressible cases, second order derivatives have to be computed. In order to maintain an even distribution, points are generated or removed automatically during the simulation.

APPLICATION TO NUMERICAL PERMEABILITY PREDICTION

In order to perform accurate LCM filling simulations, physical parameters such as fabric permeability are needed. It is well known that the experimental measurement of that fabric property is very delicate.

Usually a fluid is injected at constant pressure (or constant flow rate) through several layers of fabric of cross section *A* and length *L*. Then pressure loss Δp and flow rate *Q* are measured, and the permeability *K* is calculated using Darcy's law:

$$K = \frac{QL\mu}{A\Delta p} \tag{4}$$

The simulation of flow through a periodic cell should provide a reliable solution and avoid that experimental procedure. Numerically, an injection at constant flow rate could be performed. Then the pressure loss can be computed by the FPM code and the permeability is also given by Eqn. 4. Fig.3 shows the fabric cell considered here for benchmark purpose (Belov et al, 2004). It is bounded by a box in contact with yarns that defines the computational domain. Shell elements surrounding the fiber tows prevent the particles from escaping the domain with proper contact. Also particles are not allowed to penetrate the tows. Finally the tows are considered to be rigid and fixed in space. The porosity of such a domain is 0.36.
Starting from an initially unfilled cavity, the domain is automatically filled by particles. The inflow velocity is constant and equal to 0.01 m/s. The fluid viscosity is constant and equal to 0.01 Pa.s. The computed pressure loss is ΔP= 417,6 Pa giving a saturated permeability of 2.28×10^{-10} m^2. Computations last around 20 minutes with a state of the art PC.

According to Belov et al., the saturated permeability of a single layer of such fiber reinforcement is in between 2.6 $\times 10^{-10}$ m^2 and 3.5 $\times 10^{-10}$ m^2 using the Lattice Boltzmann Method (LBM). Measurements provided values between 1.34 $\times 10^{-10}$ m^2 and 1.49 $\times 10^{-10}$ m^2.
The FPM code provides results in good agreement with experimental data.

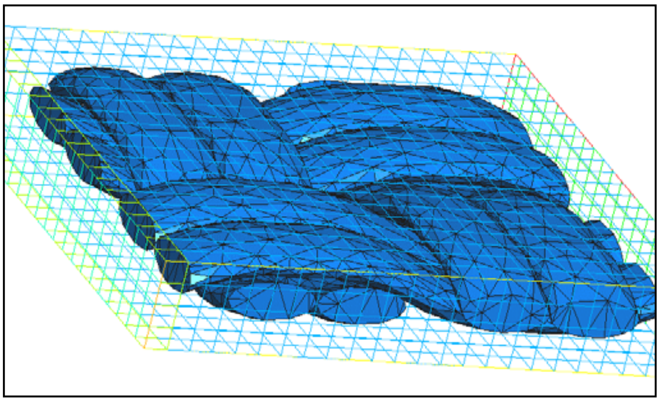

Fig. 4 : Fabric unit cell and domain used for the calculations

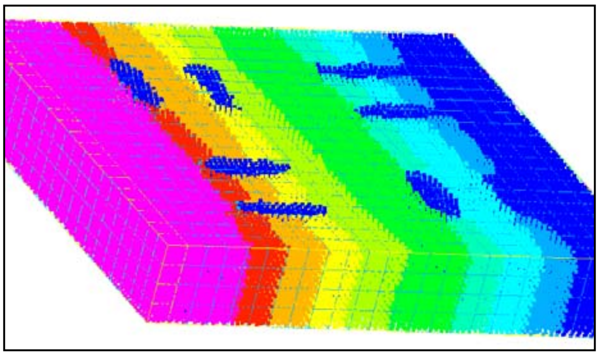

Fig.5 : Computed flow patterns and pressure field (blue spots are contact zones not wetted by the fluid)

CONCLUSION

FPM solves for compressible and incompressible flow problems and can be coupled to structural FE codes. An important feature regarding polymer composites manufacturing science is that chemical reactions, heat transfer, temperature dependent viscosity can be handled. To illustrate the potential of the code, the case of numerical permeability prediction of a fabric unit cell has been presented and compared with existing experimental data. The FPM code provides results in good agreement with experiments. Current work focuses on flow-induced fiber deformation.

REFERENCES

ASH N., POO J. Y., Coalescence and separation in binary collisions of liquid drops, *J. Fluid Mech*., vol. 221, 1990, p. 183 - 204.

Belov E. B., Lomov S. V., Verpoest I. , Peters T. , Roose D. , Parnas R. S., Hoes K. , and Sol H. . Modelling of permeability of textile reinforcements : lattice boltzmann method. *Comp. Sci. Tech.*, 64 :1069–1080, 2004.

BELYTSCHKO T., KRONGAUZ Y., FLEMMING M., ORGAN D., LIU W.K.S., Smoothing and accelerated computations in the element free Galerkin method, *J. Comp. Appl. Maths,*. vol. 74, 1996, p. 111-126.
CHORIN A., Numerical solution of the Navier-Stokes equations, *J. Math. Comput*,. vol. 22, 1968, p.745-762.

DILTS G. A., Moving least squares particle hydrodynamics. I: consistency and stability, *Hydrodynamics methods group report*, Los Alamos National Laboratory, 1996

GINGOLD R. A., MONAGHAN J. J., Smoothed particle hydrodynamics: theory and application to non-spherical stars, *Mon. Not. Roy. Astron. Soc.*, vol. 181, 1977, p. 375-389.

GINZBURG I., WITTUM G., Two-phase flows on interface refined grids modeled with VOF, staggered finite volumes, and spline interpolants, *J. Comp. Phys.,*. vol. 166, 2001, p. 302-335.

HANSBO P., The characteristic streamline diffusion method for the time dependent incompressible Navier-Stokes equations, *Comp. Meth. Appl. Mech. Eng*., vol. 99, 1992, p. 171-186.

HARLOW F. H., WELCH J. E., Numerical study of large amplitude free surface motions, *Phys. Fluids*, vol.8, 1965, p. 2182.

HIRT C. W., NICHOLS B. D., Volume of fluid (VOF) method for dynamic of free boundaries, *J. Comput. Phys.,* vol. 39, 1981, p. 201.

KELECY F. J., PLETCHER R. H., The development of free surface capturing approach for multi dimensional free surface flows in closed containers, *J. Comput. Phys*., vol. 138, 1997, p. 939.

KOTHE D. B., MJOLSNESS R. C., RIPPLE: A new model for incompressible flows with free surfaces, *AIAA Journal*, Vol. 30, No 11, 1992, p. 2694-2700.

KUHNERT J., *General smoothed particle hydrodynamics*, Ph.D. thesis, Kaiserslautern University, Germany, 1999.

KUHNERT J., An upwind finite pointset method for compressible Euler and Navier-Stokes equations, *preprint, ITWM, Kaiserslautern*, Germany, 2000.

KUHNERT J., TRAMECON A., ULLRICH P., Advanced Air Bag Fluid Structure Coupled Simulations applied to out-of Position Cases, *EUROPAM Conference Proceedings 2000*, ESI group, Paris, France

LANDAU L. D., LIFSHITZ E. M., *Fluid Mechanics*, Pergamon, New York, 1959.

LAFAURIE B., NARDONE C., SCARDOVELLI R., ZALESKI S., ZANETTI G., Modelling Merging and Fragmentation in Multiphase Flows with SURFER , *J. Comput. Phys.,* vol. 113, 1994, p. 134 - 147.

LUCY L. B., A numerical approach to the testing of the fission hypothesis, *Astron. J.,* vol. 82, 1977, p. 1013.

MARONNIER V., PICASSO M., RAPPAZ J., Numerical simulation of free surface flows, *J. Comput. Phys*. vol. 155, 1999, p. 439.

MARTIN J. C., MOYCE M. J., An experimental study of the collapse of liquid columns on a liquid horizontal plate, *Philos. Trans. Roy. Soc. London*, Ser. A 244, 1952, p. 312.
MONAGHAN J. J., Smoothed particle hydrodynamics, *Annu. Rev. Astron. Astrop*, vol. 30, 1992, p. 543-574.

MONAGHAN J. J., Simulating free surface flows with SPH, *J. Comput. Phys.*, vol. 110, 1994, p. 399.

MONAGHAN J. J., GINGOLD R. A., Shock Simulation by particle method SPH, *J. Comp. Phys*., vol. 52, 1983, p. 374-389.

MORRIS J. P., FOX P. J., ZHU Y., Modeling Low Reynolds Number Incompressible Flows Using SPH, *J. Comput. Phys*., vol. 136, 1997, p. 214-226.

TIWARI S., KUHNERT J., Grid free method for solving Poisson equation, *Berichte des Fraunhofer ITWM*, Kaiserslautern, Germany, Nr. 25, 2001.

TIWARI S., KUHNERT J., Finite pointset method based on the projection method for simulations of the incompressible Navier-Stokes equations, *to appear in M. Griebel, M. A. Schweitzer (Eds.), Springer LNCSE: Meshfree Methods for Partial Differential Equations,* Springer-Verlag, 2003.

TIWARI S., KUHNERT J., Particle method for simulations of free surface flows, *preprint Fraunhofer ITWM*, Kaiserslautern, Germany, 2000.

TIWARI S., A LSQ-SPH approach for compressible viscous flows, to appear in: *Proceedings of the 8th International Conference on Hyperbolic Problems Hyp2000.*

TIWARI S., MANSERVISI S., Modeling incompressible Navier-Stokes flows by LSQ-SPH, *Berichte des Fraunhofer ITWM*, Kaiserslautern, Germany, 2000.