A novel approach to a complex multi-physics problem - high speed impact

*A.K. Slone, A.J. Williams, T.N. Croft and M. Cross

The University of Greenwich, London, U.K.

Traditionally high impact interaction amongst solid structures is modelled within the computational solid mechanics(CSM) community using contact analysis, particularly when modelling the behaviour of metals. For example, conventional finite element(FE) modelling of contact between two solid bodies requires the definition of a "contact-pair", where the surface of one body is taken as the contact surface and the other is taken as the target surface. There are a number of difficulties associated with the modelling of impact for large scale problems in the traditional, Lagrangian, CSM contact-pair manner:

- Penetration is not permissible, which is an essential feature of high impact amongst solid structures, e.g. a projectile hitting a target.
- For large problems run on parallel clusters the aim is to achieve good load balancing, as this facilitates high efficiency. On impact the entire target surface must be searched and even if good balancing is achieved, this global search will, unfortunately, lead to poor scalability.
- Re-meshing may be required periodically to overcome element distortion, which is expensive in CPU time and degrades parallel scalability.
- Modelling inelastic behaviour, of a fluid for example, using existing contact algorithms is difficult.

However, the modelling a projectile hitting a target involves computational fluid dynamics(CFD) as well as CSM. The approach presented here is a mixed Eulerian/Lagrangian approach, where the projectile and the projectile/target interface are modelled as as a non- Newtonian fluid within an Eulerian framework and the target is modelled using a conventional CMS approach. The use of the Eulerian framework for the projectile and projectile/target interface avoids the difficulties associated with remeshing as it uses a mesh that is fixed in space for all time through which the materials flow. The boundaries of the target are modelled using a free surface technique, where the free surface of the target material is modelled using the Scalar Equation Method(SEM). This method uses a fluid marker variable ϕ to track the interface between the two fluids. This work has been carried out using the multi-physics code PHYSICA$^+$, where the three dimensional fluid flow, heat transfer and solidification algorithms are approximated using cell-centred finite volume unstructured mesh techniques on a collocated mesh.