In the unlikely event of a pressurised component failing, it is possible that missiles could be generated and potentially strike other components in the plant. This paper presents a multi-physics model which enables calculation of missile velocity based on a coupling between Smooth Particle Hydrodynamics (SPH) and Finite Element Method (FEM) in ABAQUS. This is to provide accurate estimations of both the energy of plant generated missiles and the energy required to penetrate steel targets. Investigations were performed using air as the fluid at different pressures; this enabled optimisation of the model so that it could be used to assess high pressure steam as the process fluid. A comparison between a simple hand calculated maximum acceleration, and the peak obtained from the simulation, provided a means of validating the model. In addition to this, the impact of a missile on a target was simulated using FEM with a special material model based on strain rate hardening and shear failure. This methodology provides a means of obtaining missile velocity and penetration energy without being overly conservative. The implications of this work for the treatment of indirect consequences of failure in the UK nuclear industry will also be briefly discussed.

1. Introduction

The assessment of plant generated missiles in the nuclear industry is currently based on best estimate approaches. It is desirable to reduce the uncertainty associated with these assessments, and models have been developed in ABAQUS using Smoothed Particle Hydrodynamics and Finite Element Analysis to achieve this. These models were compared to the industry approach and it was found that the SPH gave significantly lower predicted missile velocities. This backs up the claim that they are conservative and also creates the opportunity for less pessimism in future assessments.

2. Theoretical Background

Smoothed Particle Hydrodynamics is a method to solve Partial Differential Equations (PDEs) in a Lagrangian environment. The method works by dividing the continuum into a set of particles where the solution is approximated between particles with a smoothing function.

The method’s Lagrangian nature, associated with the absence of a fixed mesh, is its main strength. Difficulties associated with fluid flow and structural problems involving large deformations and free surfaces are resolved in a relatively natural way.
At its core, the method is not based on discrete particles colliding with each other in compression or exhibiting cohesive-like behavior in tension as the word particle might suggest. Rather, it is simply a clever discretization method of continuum PDEs. In that respect, smoothed particle hydrodynamics is quite similar to the finite element method. SPH uses an evolving interpolation scheme to approximate a field variable at any point in a domain. The value of a variable at a particle of interest can be approximated by summing the contributions from a set of neighbouring particles, denoted by subscript \( j \), for which the “kernel” function, \( W \) is not zero, i.e.

\[
\langle f(x) \rangle \approx \sum \frac{m_j}{\rho_j} f_j W(|x - x_j|, h)
\]  

An example of the kernel function is shown in Figure 1 (Simulia, 2014).

2.1 Artificial Viscosity

Artificial viscosity in SPH has the same meaning as bulk viscosity for finite elements. Similar to other Lagrangian elements, particle elements use linear and quadratic viscous contributions to dampen high frequency noise from the computed response (Simulia, 2014). In all the cases considered in the paper, the default artificial viscosity is used.

2.2 Particle Volume

The method to calculate particle volume is based on a characteristic length that defines a small cube centred at the particle. This is then used to compute the mass associated with each particle. When stacked together these cubes fill the overall volume of the body with some minor approximation at the free surface of the body. In ABAQUS, the initial body of particles is defined by conventional nodes and elements, which are then converted to particles (PC38) using the conversion tool. The characteristic length is then half the distance between the nodes in this mesh.

2.3 Material and Fluid Models

The FE and SPH models both have sophisticated material models to most accurately represent the behaviour of the solid and fluid parts respectively. For the FE part, the model is based on an elastic plastic analysis with a damage criteria to account for when the material no longer bears any
load. This is based on a critical failure strain of 0.5, which is modified using a triaxiality factor (ASME, 2011). This factor increases the failure strain when the elements are in compression, and reduces to 1 when the elements are in pure uniaxial loading, this is described mathematically by equation 2 and plotted in Figure 2.

\[ \varepsilon_{f}^{tri} = \varepsilon_{f}^{uni} \exp \left\{ -\frac{\alpha_{SL}}{1+m_{2}} \left(M - \frac{1}{3}\right) \right\} \]  

(2)

where \( \alpha_{SL} \) is a material factor for the multiaxial strain limit, \( m_{2} = 0.75(1-R) \) for Stainless Steels, and \( R \) is the ratio of minimum specified yield stress to UTS. \( M \) is the ratio of pressure stress to Mises stress:

\[ M = \frac{(\sigma_{1} + \sigma_{2} + \sigma_{3})}{\sqrt{\frac{1}{2}[(\sigma_{1} - \sigma_{2})^{2} + (\sigma_{2} - \sigma_{3})^{2} + (\sigma_{3} - \sigma_{1})^{2}]} \]  

(3)

where \( \sigma_{1} \), \( \sigma_{2} \) and \( \sigma_{3} \) are the principle stresses at a location.

\[ \text{Figure 2. ASME triaxiality curve (ASME, 2011)} \]

The fluid model uses an ideal gas law to give the pressure-density relationship. Given that SPH is a compressible formulation, it is possible to account for the drop in density when the gas escapes from the vessel. This is an important aspect of the physics to capture as it has a strong influence on the missile velocity. It is a novel application of SPH in ABAQUS, as previously it was common practice to enforce incompressibility by specifying a large speed of sound. The ideal gas law is modified with a compressibility factor, \( Z \), which accounts for the deviation from ideal behaviour under realistic in service conditions. This is included in equation (4):

\[ pv = ZRT \]  

(4)
3. Results

The problem considers a pressure vessel containing pressurized air between 1 and 10MPa. The fluid is modeled with a body of particles that are given an initial pressure. A small opening was defined to simulate a break in the wall of the vessel, and a missile of the same size and shape of the rupture was modelled with finite elements. At the start of the analysis, the pressurized fluid can escape through the opening and accelerate the missile as shown in Figure 3. The body of particles is defined with material properties of an ideal gas as discussed in Section 2.3. Therefore, the analysis starts with the fluid containing internal energy, which is then relieved as the analysis progresses. Some of this energy is transferred to the missile, causing it to accelerate until it reaches a steady velocity some distance away.

![Figure 3. Pressure vessel with break and missile](image)

3.1 Sensitivity study

Experiments were performed to establish the optimum particle density. The results of this are shown in Figure 4.
It can be seen that there is a slight increase in fluid velocity with particle density, however the computational time increases exponentially. Therefore, as a conservative approach, the highest density was chosen. However, it is still possible to use lower particle densities and get reasonable results. The line denoted ‘F=ma’ is calculated from a simple hand calculation which uses the initial pressure to calculate the acceleration of the missile, which is then multiplied by time to give velocity. This will overpredict missile velocity apart from at the very first moment of rupture.

### 3.2 Comparison with R3

The method to calculate missile velocity was compared with the R3 procedure (EDF Energy, 2003). This method uses hand calculations to predict missile velocity in a conservative manner. Figure 5 demonstrates that the R3 procedure gives significantly higher predictions of missile velocity than SPH. However, the overall shape of the curves are the very similar as the pressure increases, which gives confidence in the validity of the SPH method.
3.3 Finite element impact analysis

In order to assess the damage caused by plant generated missiles, finite element models using damage material models were developed. The damage model is based on the concept of a critical failure strain, where elements are removed from the calculation when this strain is reached. This is dependent on the mesh so it is important to have sufficiently refined and regular mesh to ensure reliable results. Figure 6 shows a typical mesh and element removal for an impact simulation. The stress strain curves are derived from yield and tensile data at high strain rates using the Ramberg Osgood law. This ensures that there is enhanced strength at higher strain rates.
The results indicate that this methodology is a good way of analysing missile impacts for steel components. Comparison with hand calculated penetration energies showed good agreement for cylindrical shaped missiles. For non cylindrical missiles, such as the one shown in Figure 6, the agreement was not so good, and for assessments FEA analysis is recommended over hand calculations.

4. Conclusions

The methodology presented in this paper demonstrates how missile impacts can be evaluated in the ABAQUS software. These results indicate that current methods are conservative, and the new method could provide a platform for more accurate assessments in the future.

5. References