Gas explosion modelling with PDRFoam

Jonathan Puttock

Shell Research Ltd, Brabazon House, Concord Business Park, Threapwood Road, Manchester, M22 0RR, U.K.

A gas explosion in petrochemical plant can occur when flow frexpansion following ignition creates turbulence as it passes obstacles such as pipes and vessels. The flame also folds around the obstacles and the increase in flame area further increases the burning rate. Te enhanced combustion rate creates faster flow and thus a positive feedback loop is created; this is known as the Shchelkin mechanism.

In CFD analysis of such an event, for typical plant size it is only feasible to use cells down to a size of about 0.5m. Pipes of 50mm diameter can have a significant effect on the combustion but cannot be resolved. In the CAD description of a large unit there may be 500,000 such objects. Thus, important aspects of the physics have to be included in sub-grid modelling. This is the porosity/distributed – resistance (PDR) approach developed by Prof. Hjertager in the 1980s. Two CFD explosion models using PDR are in active use today. We felt that there would be advantages in using the modern and flexible CFD capabilities of OpenFOAM to develop a new PDR model.

The presentation will describe the approach we have used in this development. A large part of this effort has been in developing a program, CAD_PDR, which takes a file listing the obstacles (pipes, beams etc.) and creates the fields needed by the CFD. These fields include volume blockage, area blockages, surface area, sub-grid drag and turbulence generation. These are then used by the new model PDRFoam. PDRFoam implements porosity-modified momentum and continuity equations with sub-grid source terms. The combustion model is the XiFoam flame area model, with an additional field to track the flame area generated by sub-grid obstacles.

Validation has been performed against a variety of experiments and is continuing.