CONJUGATE HEAT TRANSFER AND MULTI-PHASE FLOW ANALYSIS FOR INCOMPRESSIBLE FLOWS

F. Del Pin*, I. Caldichoury*, R.R. Paz*, B. Boll, N. Karajan****

***Livermore Software Technology Corporation (LSTC)**
****DYNAmore GmbH, Industriestraße 2, 70565 Stuttgart, Germany**

ABSTRACT

1 Motivation

In the past, the simulation capabilities in computational fluid dynamics, solid mechanics or thermal analysis have reached a very sophisticated level. Backed by the steadily increasing computer power, the boundaries of simulation are currently pushed further towards coupled simulations between fluids and solids, which might even be superimposed with temperature or even electromagnetic fields.

In this contribution, two scenarios will be presented. The first is a coupled cooling simulation where a fluid flows through cooling channels to cool a forming tool that is used for hot stamping and quenching. The second case is about the fluid-fluid interaction for multi-phase fluid analysis as well as its extension to fluid-structure interaction with a solid structure. Both cases will be compared with experimental results.

2 Conjugate heat transfer

It is an essential requirement in the press hardening process, to get a particular amount of heat in pa particular amount of time out of the blank. The cooling of the blank occurs almost completely through the heat transfer from the blank to the tools. Therefore, the cooling of the tools is an essential need and the way to do this at the press hardening lines of Volkswagen is to run a fluid through them.

In this presentation, the coupled simulation of the flow through the cooling duct and the thermal conditions of blank, tool and fluid in a complete forming cycle is described. A completely shaped blank is used just from the beginning of the simulation. The contact surface between blank and tool is split into regions with different heat transfer coefficient. The distribution of the heat transfer coefficient is determined in a previously thermal-mechanical forming simulation, which is not part of this presentation.

Subject of the current investigation is the simulation of the transient, turbulent and viscous flow and conjugate heat transfer problem covering the heat transfer inside the blank, from blank to tool, inside the tool, from tool to fluid

and inside the fluid using LS-DYNA's monolithically coupled ICFD solver and thermal solver. The fluid is assumed to be incompressible with flow properties of water. The initial temperature distribution of the tool is determined in previous thermal-only simulations of multiple succeeding forming cycles where the temperature at the end of one cycle is used to initialize the tool temperature of the subsequent cycle. Simulations are performed with all available turbulence models in LS-DYNA's ICFD solver and results are compared.

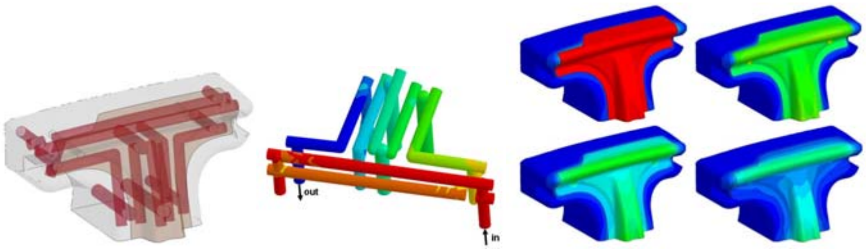

Fig. 1: Cooling channels in the tool, pressure distribution oft the fluid and temperature distribution over time during the quenching process due to the cooling channels

.

3 Multi-phase analysis

A common problem that arises in industrial design is involving immiscible fluids that can collide, exchange heat, impact into structures and, generally speaking, change their interface geometry and topology as the flow develops. These kinds of problems have been widely studied in numerous scientific applications and there exists well known methodologies that provide robust and accurate tracking of the interfaces. A common approach which was implemented in the incompressible CFD (ICFD) solver in LS-DYNA is the one that involves the tracking of a level set function. The level set method uses a scalar field which provides an implicit representation of the interface where the value of the level set $\phi = 0$. The level set function is advected using the Eikonal equation

$$\frac{\partial \phi}{\partial t} + u \frac{\partial \phi}{\partial x} = 0$$

where u is the fluid velocity. In the ICFD solver, the level set ϕ is the distance function to the interface. Clearly a maximum of two fluids can be represented with this approach, one taking the positive side of the level set and the other the negative.

In this contribution a brief introduction of the method will be presented. Moreover, two validation problems are presented for two-phase problems without and with fluid-structure interaction, cf. Fig 2,3.

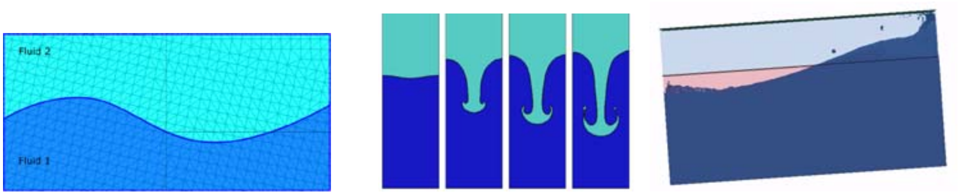

Fig. 2: Two-phase flow, Rayleigh–Taylor instability and two-phase sloshing problem.

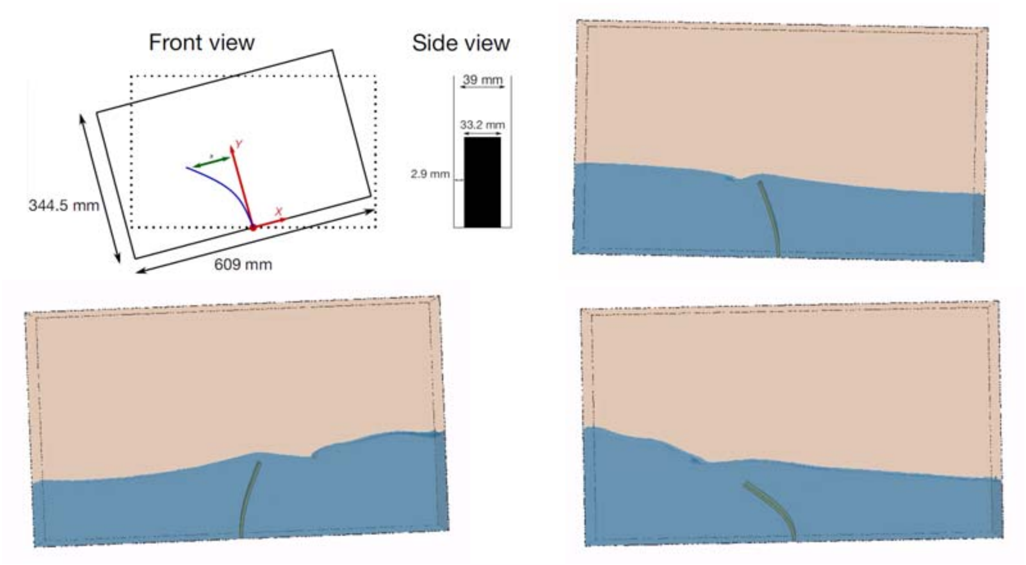

Fig. 3: Two-phase flow with fluid-structure interaction.