Impingement jet flows for cooling using LS-DYNA®: an introduction to ISPH and ICFD approaches

Edouard Yreux[1], Iñaki Çaldichoury[2]

[1]Ansys
[2]Ansys

1 Introduction

Cooling jet flows are commonly encountered in many industrial applications where fast and strong heat dissipation is required such as in pistons, gears, electrical engines and so forth. With the rapid growth and acceptance of simulation as a companion tool intervening directly in the design process, there is a need to provide fast and robust numerical solutions that can provide information on flow patterns and cooling efficiency.

LS-DYNA® includes several solvers capable of representing fluids and solve Multiphysics simulations. This presentation will focus on the ISPH solver and the ICFD solver. The ISPH solver is based on LS-DYNA's SPH solver which has been recently extended to handle incompressible flows for fluid injection, wading and similar splashing applications. Heat transfer coefficient can be displayed on wetted surfaces thanks to the use of empirical laws based on flow patterns and then passed on to the structure. The ICFD solver is a finite element incompressible CFD solver, which includes a robust and accurate monolithic approach for conjugate heat transfer applications.

2 The ISPH solver

The Incompressible SPH (ISPH) solver is a recent addition to LS-DYNA. It was originally developed for analyzing automotive water wading events [1], and has more recently seen interest in other applications such as gearbox modeling and oil cooling. For the latter type of simulation, capturing accurate fluid flow patterns is often not enough to carry out an informative analysis, and some thermal quantities also need to be evaluated.

A common approach in this field is to run a first fluid flow analysis, and extract heat transfer coefficients on the surface of the solid to be cooled once the fluid has reached a steady state. These coefficients are then mapped onto the solid mesh of a secondary, purely thermal analysis. The efficiency of the design at cooling the system can then be assessed by examining temperature evolutions at various locations of the domain.

Because ISPH lacks any form of boundary layer, the discretization of the fluid is typically too coarse to directly estimate the heat transfer coefficient from temperature gradients at the wall. Empirical expressions of the Russelt number Nu_x as a function of the Reynolds number Re_x and Prandtl number Pr are used instead [2]. When the flow is considered laminar, the following empirical law is used:

$$Nu_x = 0.332 Re_x^{\frac{1}{2}} Pr^{\frac{1}{3}},$$

where $Re_x = vx/\nu$ with v the tangential fluid velocity, x is the total distance traveled along the surface, and ν is the fluid kinematic viscosity.

For local Reynolds numbers larger than 10^5, the flow is considered turbulent, and a Dittus-Boelter equation is used instead, yielding :

$$Nu_x = 0.023 Re_x^{4/5} Pr^{0.3}.$$

The heat transfer coefficient at each fluid particle in contact with the structure is then calculated using $h = Nu_x k/x$, and interpolated back on the surface geometry, where k is the fluid's thermal conductivity.

Due to the particle-based nature of the SPH method, instantaneous readings of the heat transfer coefficients on the solid surface can sometimes be noisy. To help alleviate this limitation, the heat transfer coefficients can be averaged out over a user-specified time interval before being output. LS-

DYNA will output these coefficients directly in a D3PLOT format in the SPH interface file and will also output a keyword file that contains all the convection boundary conditions to be used in the secondary thermal analysis.

3 The ICFD solver and conjugate heat transfer (CHT)

The ICFD solver is part of the Multiphysics solvers included in LS-DYNA. As it is based on a traditional CFD approach and capable of solving Conjugate heat transfer problems (CHT), its results will serve as a guideline against which the ISPH approach and results will be compared and discussed.

The ICFD solver uses a finite element approach to solve the Navier Stokes equations using a fractional step approach [3]. For Jet impingement applications, a free surface approach is adopted with the introduction of a levelset function to track the intersection between void and fluid. Thanks to the incompressibility hypothesis, the heat equation is solved separately. Its expression in the fluid adds a convection term representing the fluid's velocity effects. When thermal transfer between fluid and solid occurs, a single system is assembled for the two domains, where the heat equation is solved simultaneously in a monolithic approach [4]. The interface between the fluid and solid is typically handled by a constraint method which forces an equal temperature at the interface (See Fig. 1).

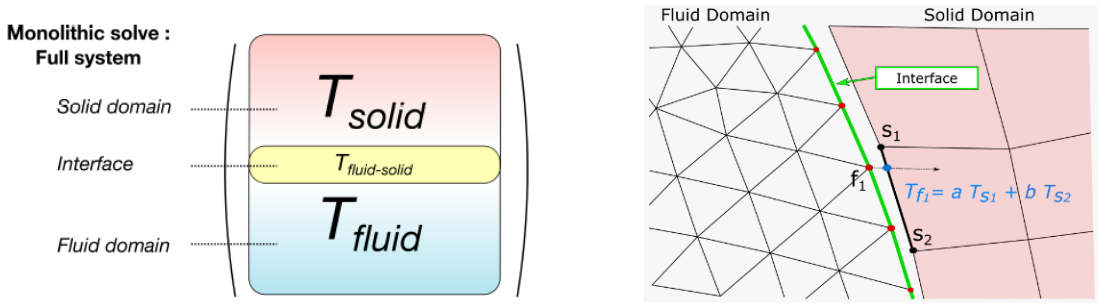
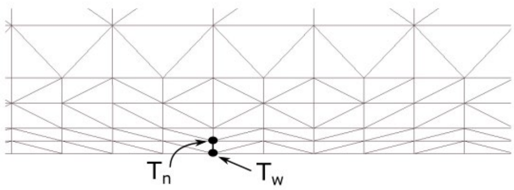


Fig.1: The fluid structure monolithic domain and constraint handling at the interface for conjugate heat transfer problems

To calculate the heat transfer coefficient (HTC), the ICFD solver typically defines it as the ratio between the heat flux and an expression of the driving force for the flow of heat (often a temperature difference between the temperature at the wall T_w and a "bulk" temperature T_b commonly defined as the external fluid temperature).



$$q = -k\frac{\partial T}{\partial y} = -k(\frac{T_n - T_w}{y})$$

$$HTC = \frac{q}{T_w - T_b}$$

Fig.2: HTC at the wall

Due to its strong dependence to the temperature gradient at the wall, its values can sometimes vary based on the choice of boundary layer mesh size. For the ICFD solver, when the constraint approach is used it is not a quantity that is used during the conjugate heat transfer solve itself but is available as a post treatment value. It can then be output in a keyword format for a subsequent thermal only analysis. This approach is used in certain applications where the cooling time scales are far longer than the typical time it takes for the flow to reach a form of steady state. In those cases, an option available to users is to stop the fully coupled Conjugate heat transfer and to run a subsequent pure thermal analysis by using the previously estimated heat transfer coefficient values as a convection boundary condition similarly to the ISPH workflow. We will place ourselves in such a configuration for the present analysis to better compare the ICFD and ISPH approaches.

4 Jet impingement model and workflow

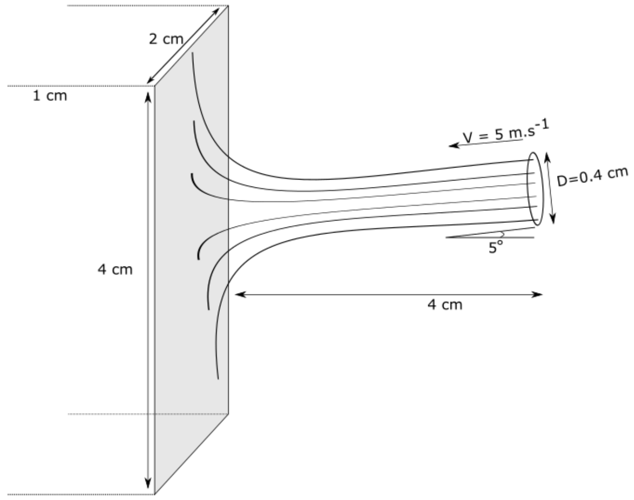


Fig.3: Jet impingement model

The chosen model is a typical representation of an application for which the ISPH method is well suited. An impinging oil jet impacts a flat steel plate at a slight angle (Fig. 3). Cooling is then expected to transfer to an underlaying solid block. The initial splashing pattern is captured within 0.1 seconds while the expected time for the block of steel to reach its equilibrium temperature is expected to be in the range of several seconds or minutes. The final objective of this analysis is to estimate the time it takes to cool down different portions of the solid.

The workflow for both approaches can be described as follows: the ISPH or ICFD simulation is run first with the impinging jet hitting the plate. Once a satisfactory steady state has been achieved, the calculation is stopped, and the heat transfer coefficient mapped on the surface mesh is output in a keyword format. These results are then included in a subsequent thermal only simulation where the heat transfer coefficient is being used as convection boundary condition. A summary of the different material characteristics is offered on Table 1.

To further alleviate the computational costs, the solid block in the thermal only simulation is made significantly coarser than the surface mesh in the initial ISPH/ICFD simulation (Fig. 4). Mapping of the temperature results from the fine shell to the coarser solid can be easily done thanks to the recent introduction in LS-DYNA of a constraint based thermal contact (Appendix A).

Finally, it is worth noting that this approach does not consider the cooling effects which occurred during the initial splashing event and establishment of flow pattern. However, for this application, it is an acceptable approximation due to steel's low thermal conductivity and the short time scales involved (around 0.1 seconds).

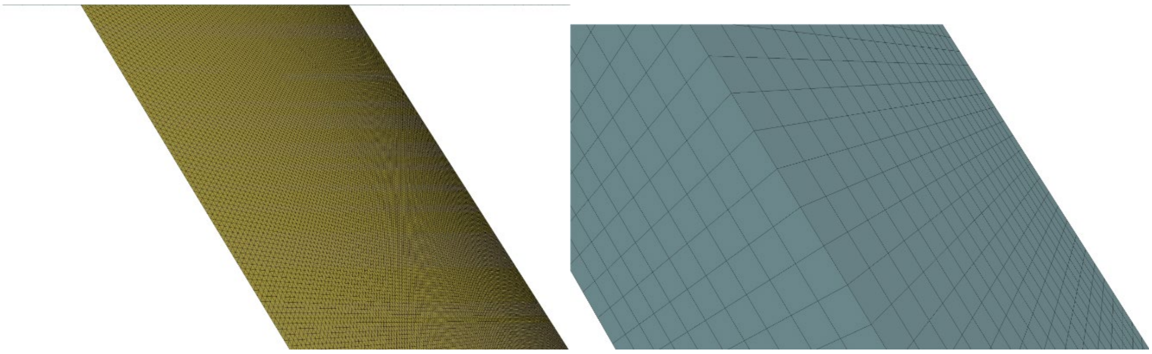


Fig.4: Left: Finer mesh used for the fluid flow analysis. Right: Coarser solid mesh used in the thermal analysis.

ISPH/ICFD simulation	
Inlet Velocity	$5\ m \cdot s^{-1}$
Fluid Density	$791\ kg \cdot m^{-3}$
Fluid Viscosity	$0.055\ Pa \cdot s$
Surface Tension	$0.0241\ N \cdot m^{-1}$
Bulk Temperature	$293.15\ K$
Fluid heat capacity	$2050\ J \cdot kg^{-1} \cdot K^{-1}$
Fluid thermal conductivity	$0.141\ W \cdot m^{-1} \cdot K^{-1}$
Timestep	Automatic, around $5 \cdot 10^{-6}\ s$
Thermal simulation	
Initial Temperature	$573.15\ K$
Density	$8000\ kg \cdot m^{-3}$
Heat Capacity	$500\ J \cdot kg^{-1} \cdot K^{-1}$
Thermal Conductivity	$45\ W \cdot m^{-1} \cdot K^{-1}$
Timestep	$0.2\ s$

Table 1: Parameters for the ISPH, ICFD and Thermal simulations.

5 Results and comparison

5.1 SPH Simulation

The incompressible SPH solver is employed with an interparticle distance of $0.1\ \mathrm{mm}$ and using the parameters listed in Table 1. A list of relevant keywords is also available in Appendix A. A general view of the oil flow during and after impact is illustrated in Fig. 5. At each timestep, HTC values are automatically calculated at fluid particles in contact with the surface, as detailed in Section 2. These HTC values are then interpolated on the surface. To minimize numerical noise in the results, values were averaged out over a period of $10^{-2}\ \mathrm{s}$, and the final values at $t = 0.1\ \mathrm{s}$ are shown in Fig. 6. Along with a d3plot output, a keyword file is also created to serve as an input file that contains corresponding boundary conditions at each surface element for the thermal-only analysis.

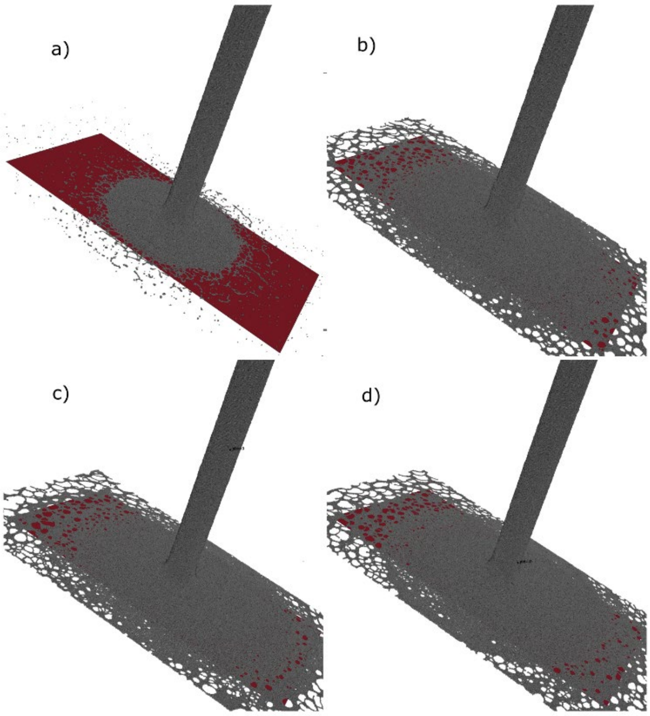


Fig.5: SPH simulation: Jet impacts the plate at a) $t = 0.01$ s*, b)* $t = 0.015$ s*, c)* $t = 0.03$ s*, d)* $t = 0.1$ s

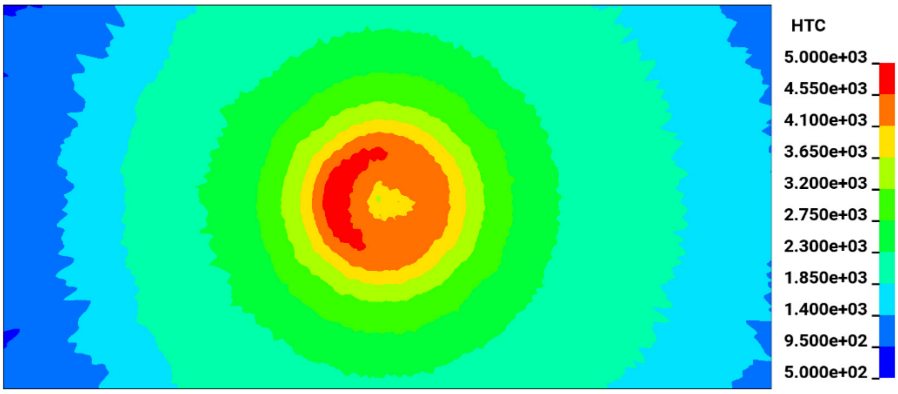


Fig.6: SPH simulation: Heat transfer coefficient output on the fluid surface mesh after t=0.1 s

5.2 ICFD Simulation

We adopt a mesh size of 0.1 mm and a timestep of 10^{-5} seconds. A couple of elements and a power law growth has been adopted for the boundary layer mesh. Fig. 7 shows the flow pattern at different instants. Most of the plate has been wetted after 0.03 seconds, validating the assumption of an established flow at the 0.1 second mark. Fig. 8 shows the Heat transfer coefficient mapped on the fluid surface. Compared to Fig. 6, the ICFD output appears slightly more diffusive but the pattern is remarkably similar, especially considering the distinct solving techniques used, as well as the different HTC calculation methods.

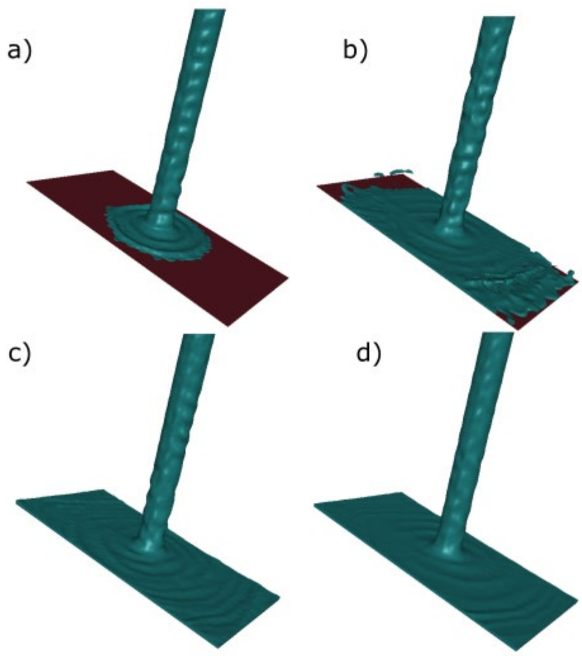


Fig.7: ICFD simulation: Jet impacts the plate at a) $t = 0.01$ s*, b)* $t = 0.015$ s*, c)* $t = 0.03$ s*, d)* $t = 0.1$ s

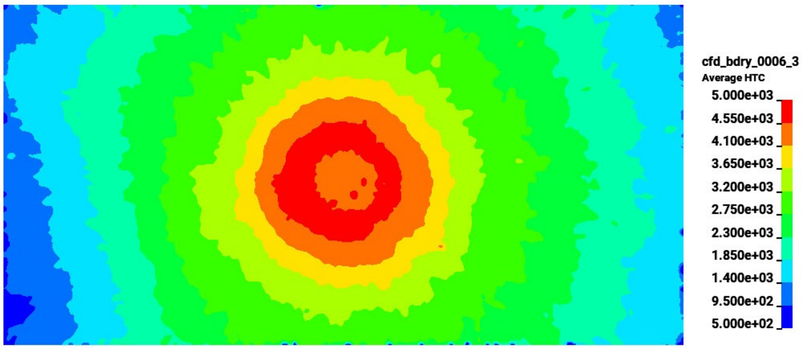


Fig.8: ICFD Simulation: Heat transfer coefficient output on the fluid surface mesh after t=0.1 seconds

5.3 Thermal Simulation and Comparison

The thermal-only analysis is then performed using the Heat Transfer Coefficients obtained from both ISPH and ICFD runs, for comparisons. In both instances, the solver generated a keyword file containing the HTC values at each shell element of the plate, which is used as a convection boundary condition on this second analysis. A constraint based thermal contact is employed to transfer these HTC values to the coarser solid mesh, see Appendix A.

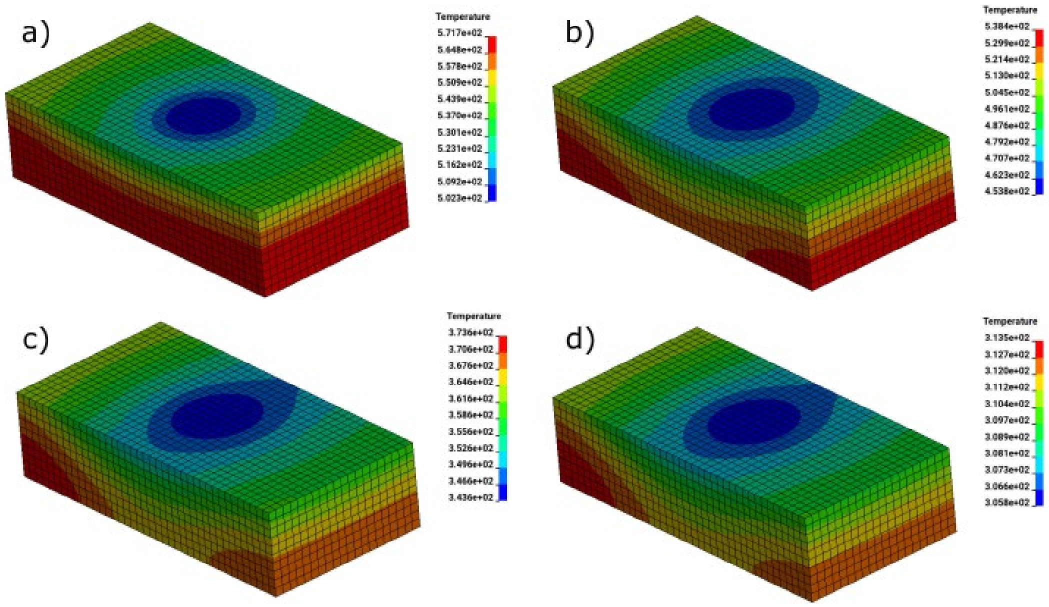


Fig.9: Cooling solid block at a) t=1s, b) t=5s, c) t=30s, d) t=60s

Fig. 9 shows the thermal results of the subsequent thermal-only simulation using the HTC results coming from the ICFD solver. Fig. 10 offers some comparison on the temperature behaviour between ISPH and ICFD simulations, at a few points along the solid's central axis. The behaviour of the two cases is consistent with only a few degrees differences occurring. Fig. 11 shows the average temperature on the top and bottom surfaces respectively. Again, results are consistent and agree very well.

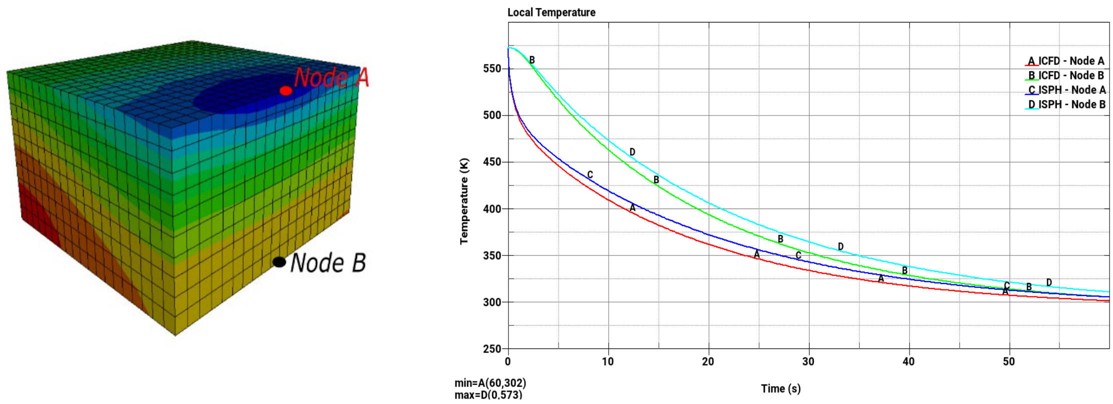


Fig.10: Thermal simulation. Comparison of local temperatures obtained using the ICFD and ISPH Heat transfer coefficients.

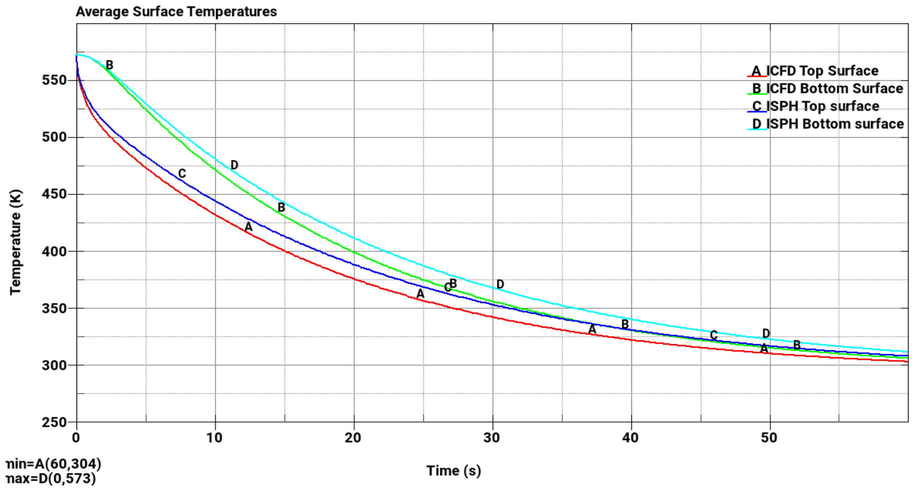


Fig.11: Thermal simulation. Comparison of average temperatures on the top and bottom surfaces using the ICFD and ISPH Heat transfer coeffcients

6 Summary

While the incompressible SPH has proven valuable for automotive applications such as water wading analysis, applications in the field of gearbox or oil cooling analysis are still to be explored. Because heat management is often a central aspect of the design, it was necessary to develop capabilities for the ISPH solver to calculate heat transfer coefficients in an efficient manner, in the absence of any boundary layer mesh in the fluid. The current approach relies on provided empirical laws, and future developments will allow users to provide their own correlation law. Comparisons with the ICFD solver, more established in the area of thermal management, provide an encouraging assessment of this approach.

7 Literature

[1] Yreux, E "Implicit SPH in LS-DYNA® for Automotive Water Wading Simulations", 2019 12th European LS-DYNA Conference.
[2] Bergman, Theodore L., et al. "Fundamentals of heat and mass transfer", 2011, John Wiley & Sons
[3] Del Pin, F. Caldichoury, I. "ICFD Theory Manual", 2014
[4] Caldichoury, I. Paz, R. " Conjugate Heat Transfer in LS-DYNA®: An Update of the ICFD-Structure Coupling Capabilities for Hot Stamping", 2020 16th International LS-DYNA Conference

Appendix A: relevant keywords

ISPH solver	
`*CONTROL_SPH`	Formulation type 13 is selected to activate the incompressible SPH solver.
`*DEFINE_SPH_MESH_SURFACE`	The impact plate gets automatically sampled with SPH particles using this keyword.
`*DEFINE_SPH_INJECTION`	The injection speed and direction of the oil jet are defined using this keyword.
`*MAT_SPH_INCOMPRESSIBLE_{FLUID/STRUCTURE}`	The fluid properties (density, viscosity, surface tension coefficient, thermal capacity, thermal conductivity) and structural properties (roughness and adhesion) are defined here.
`*DATABASE_BINARY_ISPHFOR`	This option, coupled with the "isph=" command line parameter, activates the SPH interface file output, to visualize HTCs in a d3plot format and activate output of the *BOUNDARY_CONVECTION keywords for secondary analysis.

`*DEFINE_SPH_AMBIENT_DRAG`	Add drag forces to moving SPH particles based on given ambient parameters.
Thermal Solver	
`*BOUNDARY_TEMPERATURE_PERIODIC_SET`	This keyword is a recent addition to R13. It allows to define various thermal boundary conditions, or a constraint based thermal contact. In this analysis, it is used to transfer the temperatures from the surface shell to the solid.
`*BOUNDARY_CONVECTION`	The heat transfer coefficient at each surface shell element is extracted from the first analysis and imposed on the second, thermal analysis using this keyword.
ICFD solver	
`*ICFD_BOUNDARY_FSI/CONJ_HEAT/FSI_EXCLUDE`	Those are the three keywords that were used to define the conjugate heat transfer problem at the interface. The first one was applied to the fluid surface mesh to detect the faces in contact. The second was needed to build the temperature boundary condition between fluid and solid. The last one was used here to exclude the solid block from the fluid solid interface and ensure that heat transfer would occur in the following order: fluid -> solid thin shell -> solid block.
`*ICFD_DATABASE_HTC`	This is the keyword which was used to trigger the calculation of the heat transfer coefficient by the ICFD solver and output it in a keyword format.
`*CONTROL_THERMAL_SOLVER`	As typically done for larger CHT applications, an iterative approach was selected for the solver. Note that since R13, the GMRES solver (solver type 17) is also available in pure thermal simulation and no longer restricted to CHT problems.
`*MESH BL`	This keyword controls the boundary layer mesh generation for the ICFD solver. For this analysis, a couple of elements and a simple Power law (default behavior) has been used.