LS-DYNA® R7: The ICFD Solver for Conjugate Heating Applications

Iñaki Çaldichoury
Facundo Del Pin
Rodrigo R. Paz

Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, CA 94551

Abstract

LS-DYNA version R7 introduced an incompressible flow solver (ICFD solver) which may run as a standalone CFD solver for pure thermal fluid problems or it can be strongly coupled using a monolithically approach with the LS-DYNA solid thermal solver in order to solve complex conjugate heat transfer problems. Some validation results for conjugate heat transfer analyses have been presented at the 9th European LS-DYNA Conference (2013) [1].

This paper will focus on a new output quantity, the heat transfer coefficient or 'h' which has recently been implemented in the ICFD solver. Its description, calculation and uses will be presented as well as some validation results.

1- Presentation of the problem

Heat transfer is a discipline of thermal engineering that concerns the generation, use, conversion and exchange of thermal energy and heat between physical systems. The ICFD solver offers the possibility to solve and study the behavior of temperature flow in fluids. Potential applications are numerous and include refrigeration, air conditioning, building heating, motor coolants, defrost, or even heat transfer in a human body. However, the industrial application which has seen a rising number of users involves internal forced convection flow i.e stamping or forming applications where a fluid flowing through a tube is being used to cool the dye or the tool down (See Figure 1).

Such analyses may be conducted by solving the complete coupled problem with the fluid and the solid interacting together and exchanging temperature information at the interface between the two domains. The coupling between the structure and the fluid makes use of a monolithic approach and is therefore very tight and strong at the fluid structure interface. It can also become costly from a computational point of view especially when models involve several millions of elements.

In fact, if the temperature differences between the fluid inlet and the fluid outlet do not vary too much, the effect of the fluid cooling on the structure is often approximated by a convection boundary condition where a heat transfer coefficient "h" is directly applied on the solid. It may therefore be interesting from an analysis perspective to "uncouple" the thermal problem by solving the fluid thermal problem independently and by then calculating the heat transfer coefficient before displaying it on the pipe surface in order to better understand where local heat losses may occur (due to turbulent effects or abrupt geometry changes for example).

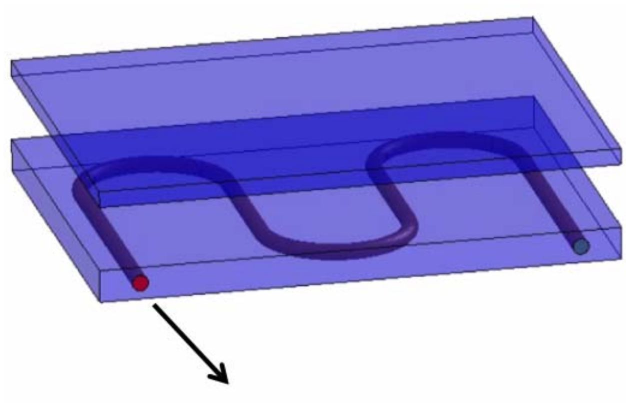

Figure 1 Stamping application: fluid flowing through a circular pipe and used as dye coolant.

2- Calculation of the heat transfer coefficient

The heat transfer coefficient is expressed as the ratio between the heat flux i.e the rate of heat energy transfer through a given surface and an expression of the driving force for the flow of heat (often a temperature difference):

$$h = \frac{q}{T_s - T_b}$$

where q is the heat flux in W/m^2, T_s the temperature at the surface and T_b a "bulk" temperature. The difficulty of extracting the h lays in the calculation of T_b. For external forced convection flows, a constant freestream temperature may be used but for an internal flow an expression of a mean temperature must be used. The mean temperature of the fluid at a given cross section can be defined in terms of the thermal energy transported with the bulk motion of the fluid as it passes the cross section. The rate at which this transport occurs $\dot{E}_t$ may be obtained by integrating the product of the mass flux (ρu) and the internal energy per unit mass ($C_v T$) over the cross section [2]. That is:

$$\dot{E}_t = \int \rho u C_v T dA_c$$

With ρ the density, u the velocity, C_v the specific heat at constant volume and A_c, the surface through which the fluid flows.

If we apply the conservation of energy principle and consider that the energy transported by the fluid through a cross section in actual flow must be equal to the energy that would be transported through this same cross section if the fluid were at a temperature T_b, we obtain:

$$\dot{E}_t = \rho C_v u_{avg} T_b A_c$$

Which yields:

$$T_b = \frac{\int uT dA_c}{u_{avg} A_c}$$

So for each node on the pipe surface, the ICFD solver will track the nodes in the volume mesh that are in the same plane, compute a weighted average temperature T_b and finally compute and display the heat transfer coefficient. In the next section some validation results will be presented.

3- 2D and 3D validation results

3-1 Test Case description

The problem considered here is that of a smooth laminar fluid flowing through a smooth tube. Both the 2D and the 3D problems will be studied i.e the infinite rectangle case and the circular tube geometry. If an inflow temperature boundary condition has been given and in absence of any work interactions, the conservation of energy for the steady state yields [3]:

$$\dot{Q}(x) = \dot{m} C_p (T_b(x) - T_i)$$

With $\dot{m} = \rho u_{avg} A_c$ the mass flow rate and T_i the inflow average temperature.

The thermal conditions at the surface can usually be approximated as constant surface temperature (Dirichlet boundary condition) or constant surface heat flux (Newman Boundary Condition). Either $T_s = cte$, or $q_s = cte$ but not both. Figure 2 offers a sketch of the problem.

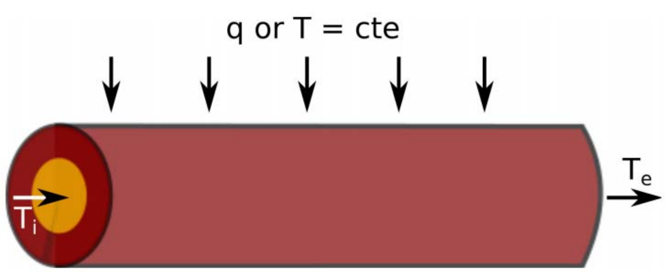

Figure 2 Sketch of heat transfer problem in pipe

In the case of a constant surface heat flux, the rate of heat transfer can also be expressed as:

$$\dot{Q}(x) = q_s P_s x$$

Where P_s is the surface perimeter where the heat flux $q_s = -k\,\frac{\partial T}{\partial r}$ is applied.

Then, in the fully developed region the fluid mean temperature is expected to vary linearly as:

$$T_b = T_i + \frac{q_s P_s x}{\rho U_{avg} A_c C_p}$$

The surface temperature T_s in the case of constant surface heat flux can then be determined from:

$$q_s = h(T_s - T_m)$$

In the case of a constant surface temperature, the energy balance on a differential control volume gives:

$$d\dot{Q}(x) = \dot{m} C_p dT_b = h(T_s - T_b) P_s dx$$

Since:

$$dT_s = -d(T_s - T_b)$$

Therefore:

$$\frac{d(T_s - T_b)}{T_s - T_b} = -\frac{hP_s}{\dot{m} C_p} dx$$

Which means that the temperature difference between the fluid and the surface decays exponentially in the flow direction and the rate of decay is expressed as:

$$\alpha = \frac{hP_s}{\dot{m} C_p}$$
$$d(T_s - T_b) = e^{-\alpha x}$$

The heat rate coefficient h will be compared against values found in the literature (See [2]).

The behavior of the flow in the entrance region will also be studied, more specifically the distance from the inflow needed for the adimensional temperature and velocity profiles to become constant. Empirical considerations for the velocity and temperature profiles for laminar flows give:

$$L_{vel} \approx 0.05 R_e D$$
$$L_{temp} \approx 0.05 R_e D P_r$$

Where D is a typical section length, R_e the Reynolds number and P_r the Prandtl number.

3-2 Model Description

Both the 3D and the 2D models will have the same surface mesh size. Table 1 gives some information on the surface mesh sizes chosen and their corresponding total number of volume nodes. The viscosity ($= 0.01$), density ($= 1$), heat capacity ($= 100$), thermal conductivity ($= 1$), channel height ($= 1$) and fluid inlet velocity ($= 0.5$) are chosen so that the flow remains in the laminar regime ($R_e = 0.25, P_r = 2$) and the channel length (=20) is chosen such as to be long enough to ensure that a fully developed flow can be solved. The inflow temperature will be held constant at 20 degrees with either a constant heat flux of 100 or a constant temperature of 100 imposed on the exterior wall of the channel.

Table 1 Mesh information

	2D model	3D Cylinder
Surface element size	0.05	0.05
Volume Nodes	11000	150000
Volume Elements	21000	800000
Elements added to the BL mesh	2	2

3-3 Results-2D Model

Figure 3 and Figure 4 show the temperature and velocity profiles for the constant heat flux and temperature cases. It can be observed that the behavior of the velocity in both cases is similar but the behavior of the temperature differs. The hydrodynamic entrance region can visually be observed and its length can be estimated to be around X=1.25 which is in good agreement with the expected result. The temperature entrance region can be identified in Figure 5 which shows the behavior of the heat transfer coefficient along the channel. It decreases exponentially in the entrance region and reaches a constant value in the fully developed region. Again the thermal entrance region length agrees well with the expected result of X=2.5.

Figure 6 show the behavior of the bulk temperature and the surface temperature along the channel as calculated by the solver. For the constant heat flux, it can be observed that the bulk temperature increases linearly with the surface temperature once it has reached the fully developed region. For the constant temperature case, it seems to rise exponentially which is again in accordance with the expected behavior.

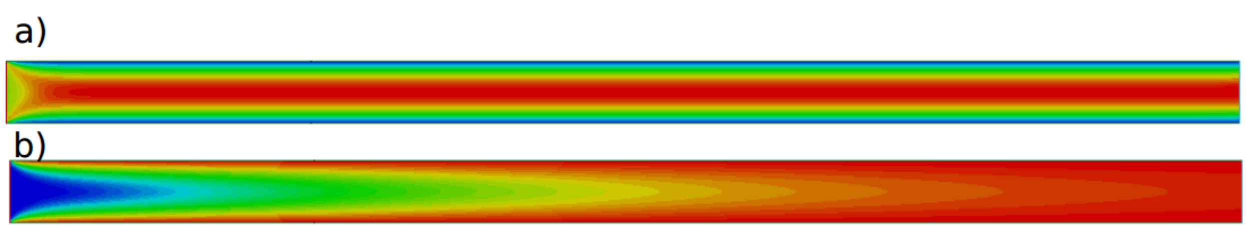

Figure 3 Constant surface temperature case: a) Velocity profile b) Temperature profile

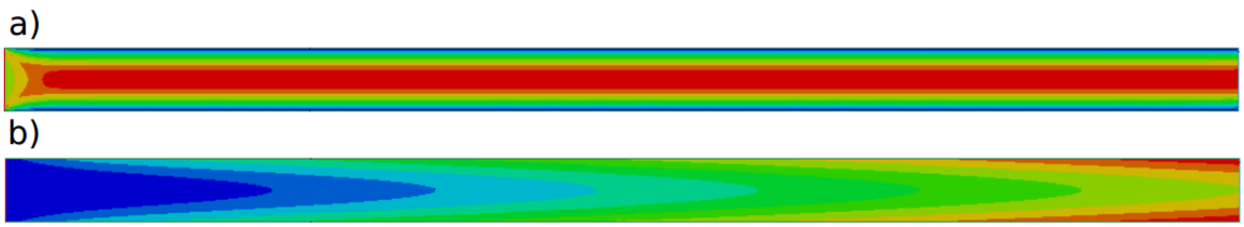

Figure 4 Constant surface heat flux case: a) Velocity profile b) Temperature profile

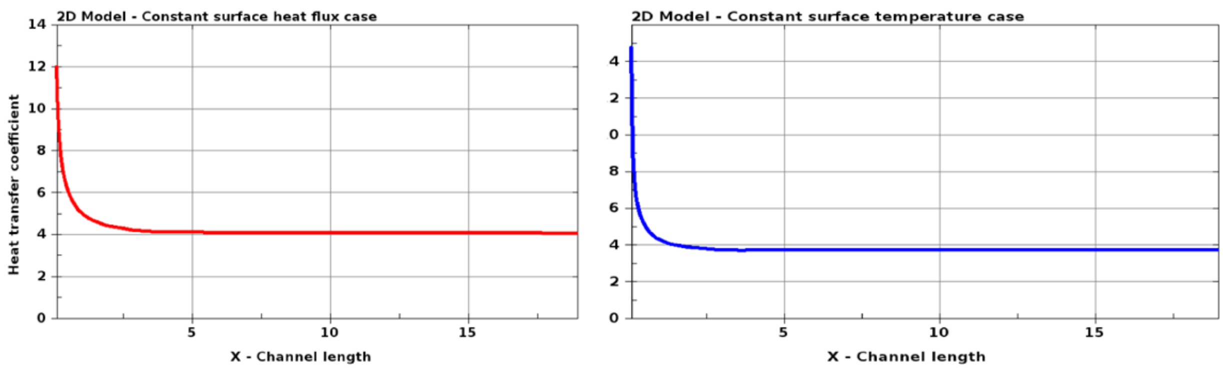

Figure 5 Behavior of the heat transfer coefficient along the channel exterior wall surface

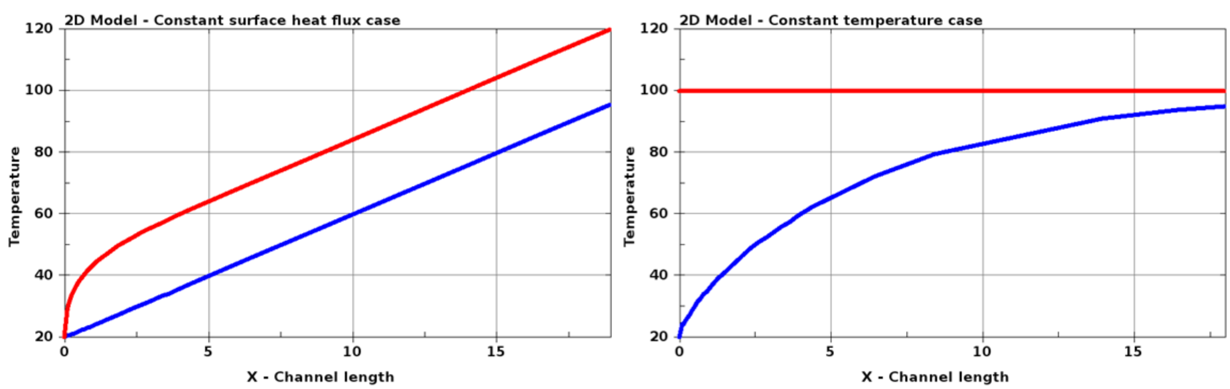

Figure 6 Comparison between the bulk temperature Tb (in blue) and the channel surface temperature Ts (in red)

3-3 Results-3D Cylinder

Figure 7 shows the temperature profile across a cross section in the fully developed region as well as the heat flux calculated on the pipe surface. Again the entrance and fully developed regions can be fully identified. Figure 8 further confirms this behavior while Figure 9 shows again the behavior of Tb along the channel.

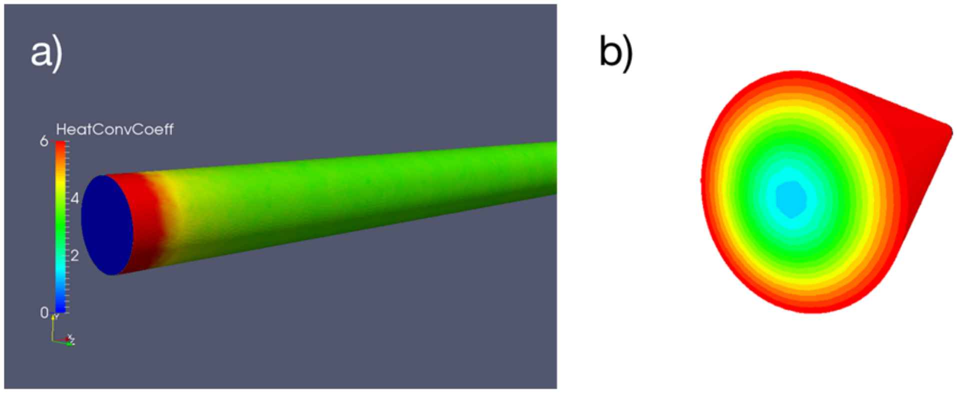

Figure 7 a) Heat transfer coefficient on the cylinder surface b) Temperature fringes through the section

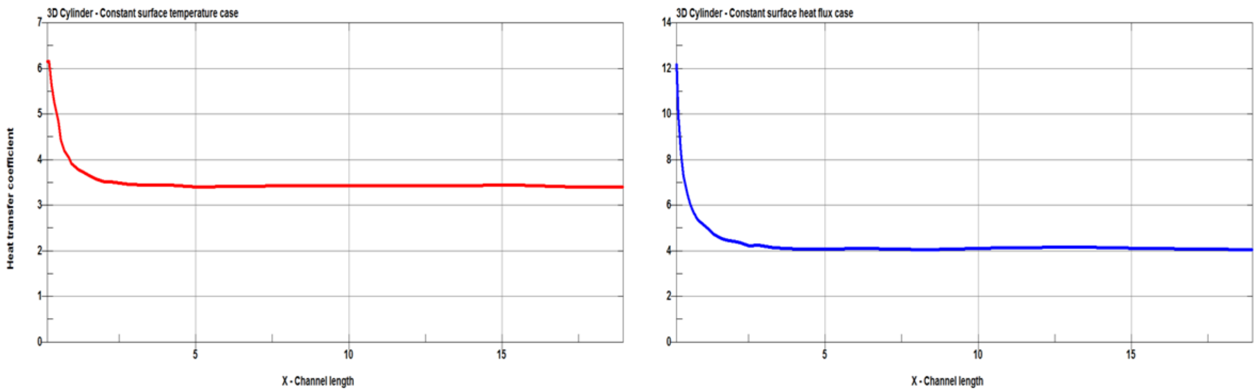

Figure 8 Behavior of the heat transfer coefficient for both the constant surface temperature case and the constant surface heat flux case

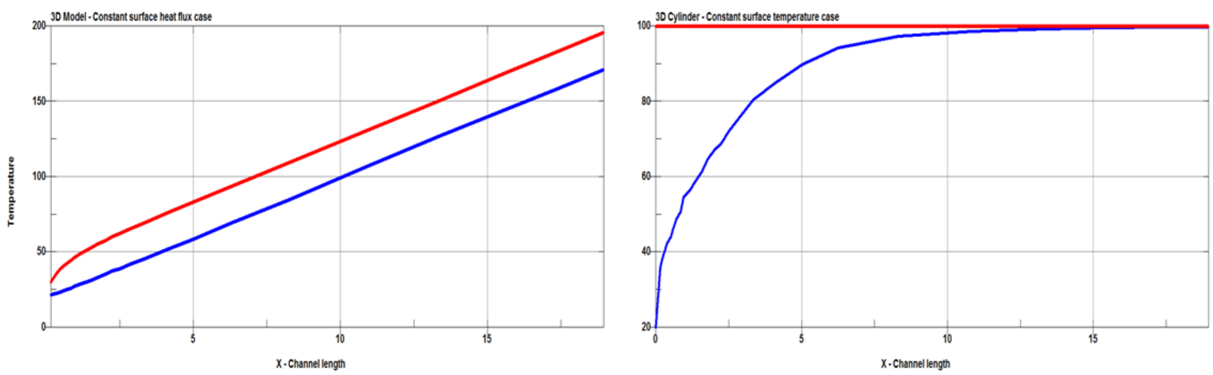

Figure 9 Comparison between the bulk temperature Tb (in blue) and the channel surface temperature Ts (in red)

The results regarding the heat transfer coefficient are summed up in Table 2 for all cases.

Table 2 Comparison between reference and numerical results

	Reference heat transfer coefficient [2]	Numerical heat transfer coefficient	Error (%)
2D Cte Temp	3.74	$\approx 3.74 - 3.82$	$\approx 0\% - 2.5\%$
2D Cte Heat Flux	4.12	$\approx 4.10 - 4.16$	$\approx -0.5\% - 1\%$
3D Cte Temp	3.66	$\approx 3.40 - 3.60$	$\approx -7\% - -2\%$
3D Cte Heat Flux	4.36	$\approx 4.05 - 4.30$	$\approx -7\% - -1\%$

4- Conclusion

The calculation and display of the heat transfer coefficient has been implemented and validated in the ICFD solver. The objective of this post treatment is to offer engineers more tools to understand and fully analyze the behavior of the flow and thermal quantities in internal aerodynamic applications and more specifically for cooling flows in pipes and channels. It is also an alternative to solving the complete conjugate heat transfer problem which aims to offer precise and robust solutions but which is also computer costly.

References

[1] D. P. De Witt et F. P. Incropera, Fundamentals of heat transfer, 1986.

[2] W. M. Kays, Convective Heat and Mass Transfer, McGraw-Hill, 1966.

[3] I. Caldichoury et F. Del Pin, «Conjugate heat transfer problems and...,» chez *LS-DYNA European Conference*, Manchester, 2013.