TEST CASE DOCUMENTATION
AND TESTING RESULTS

TEST CASE ID ICFD-VER-5.2

Heat transfer problem

Tested with LS-DYNA® R7 Revision Beta

Tuesday 13th May, 2014

Document Information	
Confidentiality	**external use**
Document Identifier	LSTC-QA-LS-DYNA-ICFD-VER-5.2-1
Author(s)	Iñaki Çaldichoury, Facundo Del Pin
Number of pages	14
Date created	Tuesday 13th May, 2014
Distribution	External

Disclaimer:

Contents

1 Introduction

1.1 Purpose of this Document

This document specifies the test case ICFD-VER-5.2. It provides general test case information like name and ID as well as information to the confidentiality, status, and classification of the test case.

A detailed description of the test case is given, the purpose of the test case is defined, and the tested features are named. Results and observations are stated and discussed. Testing results are provided in section 4.1 for the therein mentioned LS-DYNA® version and platforms.

2 Test Case Information

Test Case Summary	
Confidentiality	external use
Test Case Name	Heat Transfer: Laminar flow inside smooth tube
Test Case ID	ICFD-VER-5.2
Test Case Status	Under consideration
Test Case Classification	Benchmarking
Metadata	THERMAL PROBLEM

Table 1: Test Case Summary

3 Test Case Specification

3.1 Test Case Purpose

In this test case, we shall study the heat exchange between a fluid and a heated boundary. Its purpose is to verify the correct implementation of the temperature and heat flux boundary conditions.

3.2 Test Case Description

The problem considered here is that of smooth laminar fluid flowing through a circular smooth tube and has been widely studied [2], [1]. If an inflow temperature boundary condition has been given and in absence of any work interactions, the conservation of energy equation for the steady flow gives ([2]) :

$$\dot{Q}(x) = \dot{m}c_p(T_m(x) - T_i) \tag{1}$$

with $\dot{m} = \rho V_{avg} S_{flow}$ the mass flow rate, c_p the fluid's heat capacity, T_m the tube's section average temperature 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 $T_s = constant$ or $q_s = constant$ but not both. Technically, constant heat problems arise in a number of situations such as electric resistance heating, radiant heating, nuclear heating and so forth. The constant surface temperature condition is also very common and can be found in heat exchangers such as evaporators, condensers and any heat exchanger where one fluid has a very much higher capacity rate than the other ([2]). Figure (1) offers a scheme of the problem. For this test case, both 2D and 3D models will be set up and studied.

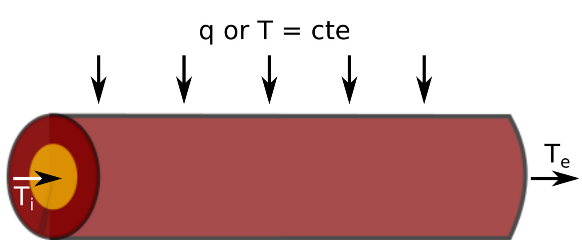

Figure 1: Heat transfer in pipe sketch

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

$$\dot{Q}(x) = q_s P_s x \tag{2}$$

where P_s is the surface perimeter where the heat flux $q_s = k\frac{\partial T}{\partial r}_{r=r_0}$ is applied.

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

$$T_m(x) = T_i + \frac{q_s P_s x}{\rho V_{avg} S_{flow} c_p} \tag{3}$$

where S_{flow} is the surface are the flow goes through and V_{avg} is the average velocity.

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

$$q_s = h(T_s - T_m) \tag{4}$$

where h is the convective heat transfer coefficient.

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

$$d\dot{Q}(x) = \dot{m}c_p dT_m = h(T_s - T_m)P_s dx \tag{5}$$

Since $dT_m = -d(T_s - T_m)$ therefore :

$$\frac{d(T_s - T_m)}{T_s - T_m} = -\frac{hP_s}{\dot{m}c_p}dx \tag{6}$$

which means that the temperature difference between the fluid and the surface decays exponentially in the flow direction and the rate of decay depends on :

$$\alpha = \frac{hP_s}{\dot{m}c_p} \tag{7}$$

$$d(T_s - T_m) = e^{-\alpha x} \tag{8}$$

In order to evaluate h, the Nussel number is often given in literature ([2]) :

$$N_u = \frac{4hS_{flow}}{kP_s} \tag{9}$$

where k is the thermal conductivity.

Several values extracted from reference empirical values for different geometrical shapes are given in Table (2) ([2]) :

For the constant surface temperature, we will also study the entry length a.k.a the distance from the inflow needed for the adimensional temperature and velocity profiles to become constant. Analytical considerations give for the velocity and temperature profiles for laminar flows :

Tube Geometry	Cross section ratio (Length/Width)	Nussel number ($T_s = cte$)	Nussel number ($q_s = cte$)
Circle		3.66	4.36
Rectangle	1	2.98	3.61
	2	3.39	4.12
	4	4.44	5.33
	8	5.60	6.49
	∞	7.54	8.24

Table 2: Test Case Mesh Information

$$L_{vel} \approx 0.05 R_e D \tag{10}$$

$$L_{temp} \approx 0.05 R_e D Pr \tag{11}$$

where D is a typical section length, R_e the Reynolds number and Pr the Prandtl number :

$$P_r = \frac{\mu}{\rho\alpha} \tag{12}$$

3.3 Model Description

Both the 3D and the 2D models will have the same surface mesh size. However, the 3D models will generate volume meshes with a lot more elements. Table (3) gives some information on the surface mesh sizes chosen and their corresponding total number of volume nodes. For the 2D model, the section will be considered unitary and we will choose a tube length of 20, long enough so that a fully developed flow can be solved. For the 3D cylinder model, the radius will be chosen as unity. The parameters chosen are given in Table (4) as well as the corresponding Reynolds and Prandtl numbers.

Model information		
	2D Model	3D Cylinder
Surface Element size	0.05	0.05
Volume Nodes	12 000	235 000
Volume Elements	23 000	872 000
Anisotropic elements added to the boundary layer mesh	2	2

Table 3: Test Case Mesh Information

Model physical parameters	
Fluid Density	1
Viscosity	0.02
Inflow velocity ($= V_{avg}$)	0.5
Inflow temperature	20
Constant surface temperature	100
Constant heat flux	100
Heat capacity	100
Thermal conductivity	1
Reynolds number	25
Prandtl number	2

Table 4: Test Case Parameters

4 Test Case Results

4.1 Test Case observations

4.1.1 The 2D Model

Figure (2) and Figure (3) show the temperature and velocity profiles for the constant heat flux and temperature cases. It can be observed that the behavior of the velocity is similar in both cases but the behavior of temperature differs. Figure (4) and Figure (5) focus on the constant surface temperature case and show the dimensionless velocity and temperature profiles at different distances from the point of entry. It can be observed that the velocity profile can be considered fully established between $X = 1.25$ and $X = 1.5$ between $X = 2$ and $X = 2.5$ for the temperature profile. This is consistent with the approximate expected results of $X \approx 1.25$ and $X \approx 2.5$ respectfully based on our choice of parameters. The temperature entrance region can also be identified in Figure (6) which shows the behavior of the heat transfer coefficient along the channel.

Figure (7) 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.

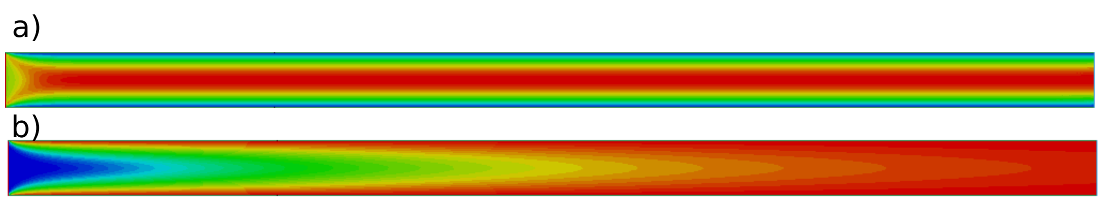

Figure 2: Constant surface temperature case : a) Velocity profile, b) Temperature profile

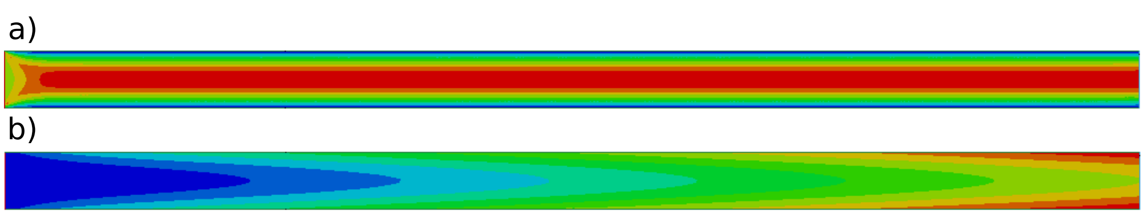

Figure 3: Constant surface heat flux case : a) Velocity profile, b) Temperature profile

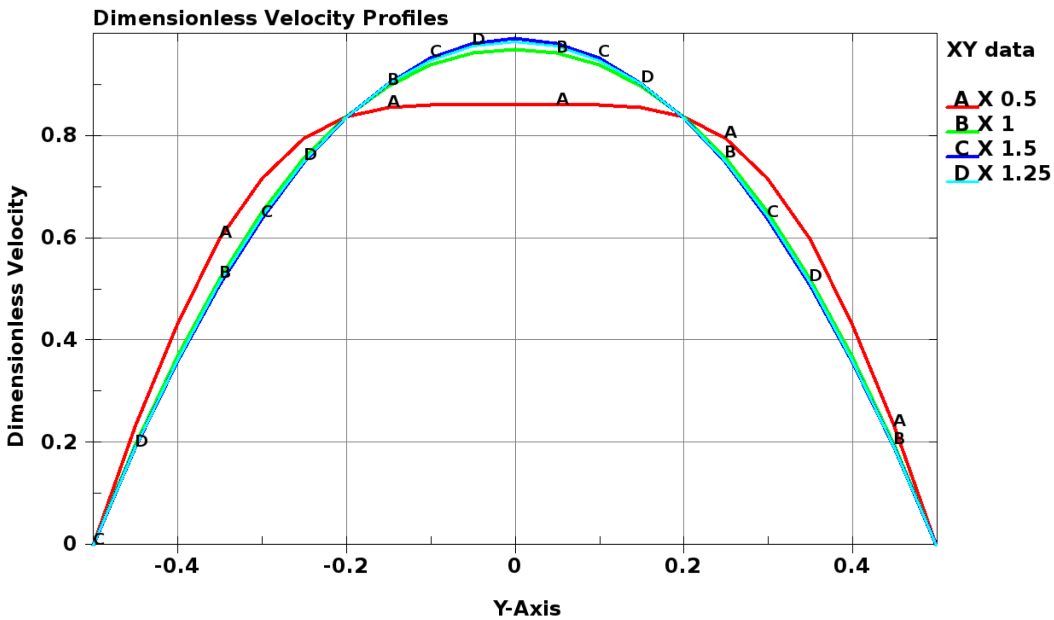

Figure 4: Dimensionless Velocity profiles ($\frac{V(y)}{V_{max}}$)

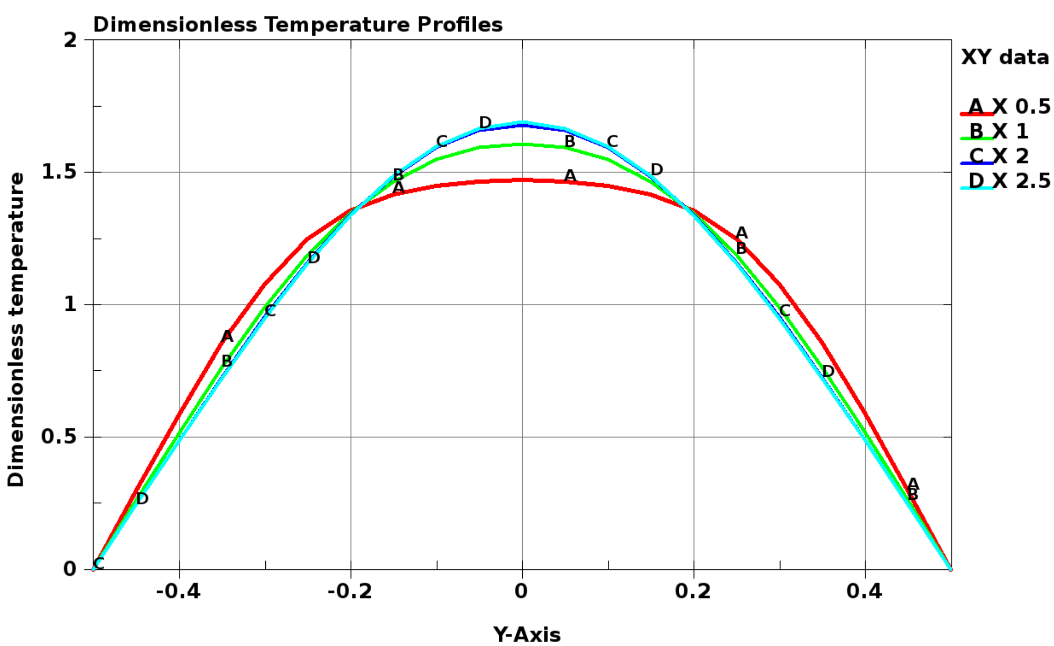

Figure 5: Dimensionless Temperature profiles ($\frac{(T_s-T(y))}{(T_s-T_m)}$)

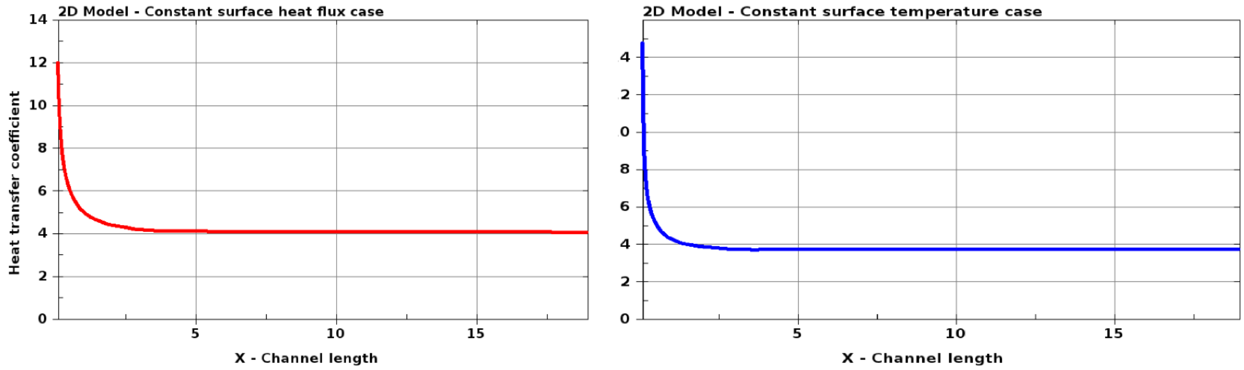

Figure 6: 2D Model : Behavior of the heat transfer coefficient along the channel exterior wall surface.

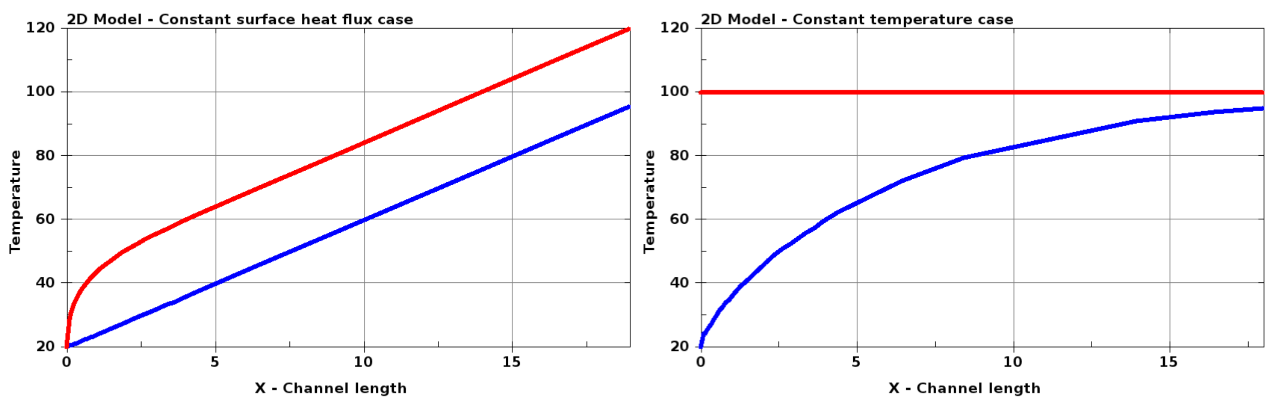

Figure 7: 2D Model : Comparison between the bulk temperature Tb (in blue) and the channel surface temperature Ts (in red).

4.1.2 The 3D Cylinder model

Figure (8) 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 (9) further confirms this behavior while Figure (9) shows again the behavior of Tb along the channel.

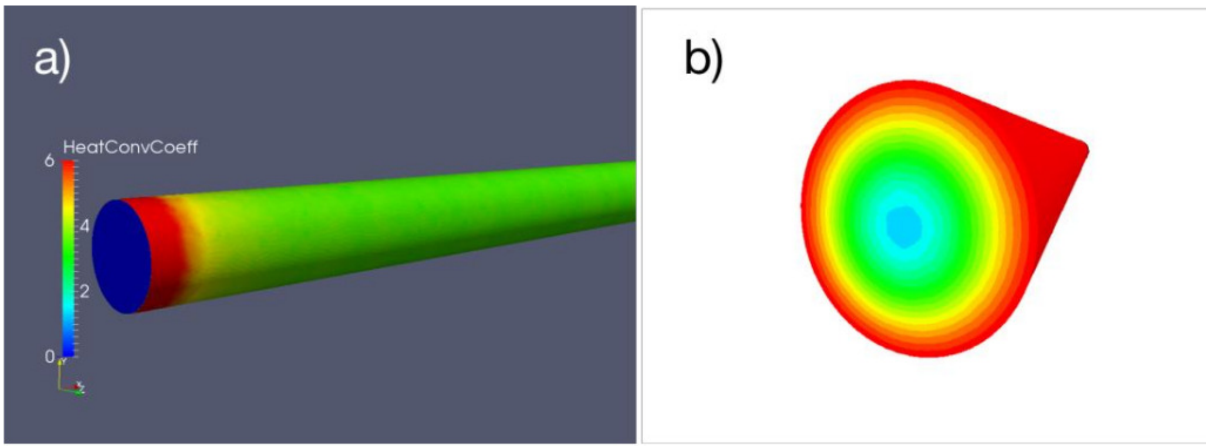

Figure 8: 3D Cylinder Model : a) Heat transfer coefficient on the cylinder surface b) Temperature fringes through the section.

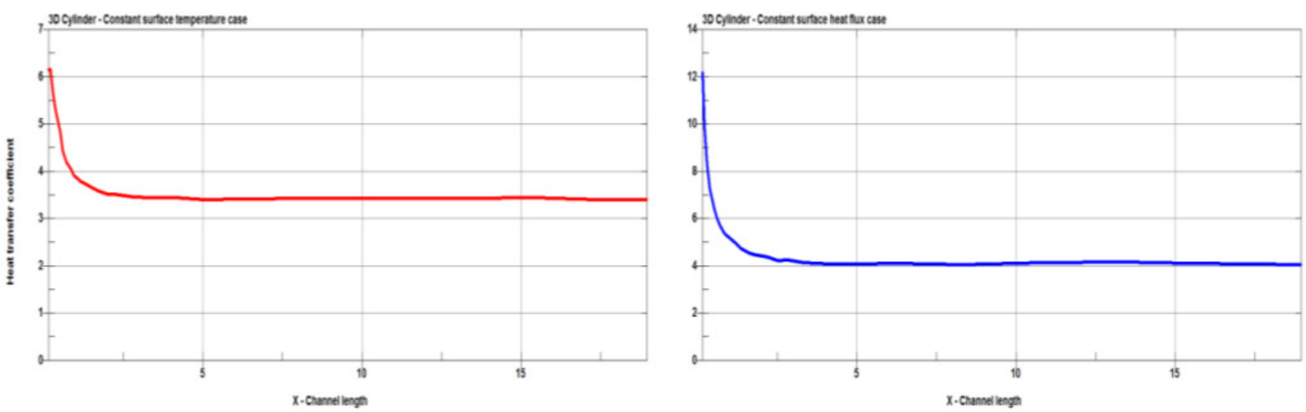

Figure 9: 3D Cylinder Model : Behavior of the heat transfer coefficient for both the constant surface temperature case and the constant surface heat flux case.

4.1.3 Summary of results

The results regarding the heat transfer coefficient are summed up in Table (5) for all cases. The results are in good agreement with the reference results of [1].

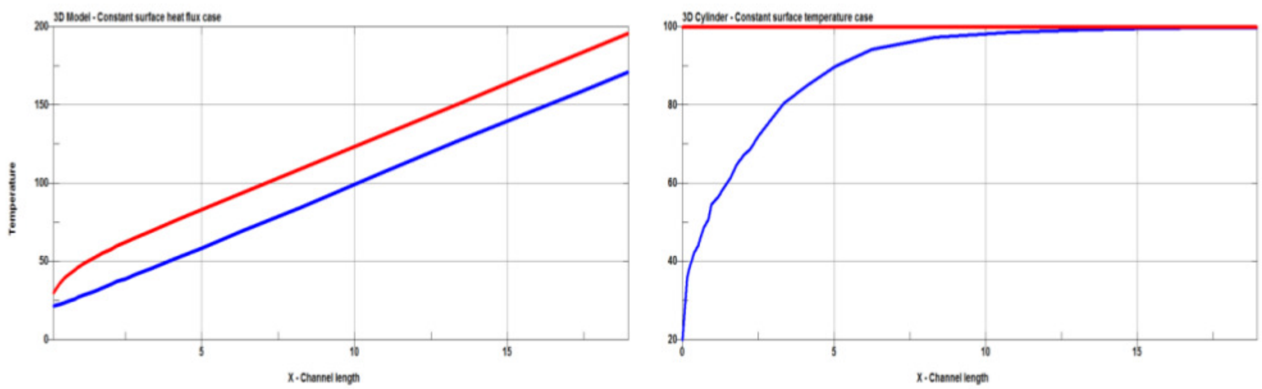

Figure 10: 3D Cylinder Model : Comparison between the bulk temperature Tb (in blue) and the channel surface temperature Ts (in red).

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

Table 5: Heat transfer coefficient summary

References

[1] F. Incropera, D. Dewitt, T. Bergman, and A. Lavine, *Principles of Heat and Mass transfer*, Wiley, 1982.

[2] W. Kays, *McGraw-Hill series in mechanical engineering*, McGRAW-HILL Book Company, 1966.