TEST CASE DOCUMENTATION AND TESTING RESULTS

TEST CASE ID ICFD-BENCH-1.1

Slamming

Tested with LS-DYNA® v980 Revision Beta

Friday 1st June, 2012

Document Information	
Confidentiality	**external use**
Document Identifier	LSTC-QA-LS-DYNA-ICFD-BENCH-1.1-1
Author(s)	Iñaki Çaldichoury, Facundo Del Pin
Number of pages	10
Date created	Friday 1st June, 2012
Distribution	External

Disclaimer:

Contents

1 Introduction

1.1 Purpose of this Document

This document specifies the test case ICFD-BENCH-1.1. 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	Slamming: Impact between a ship hull and the water
Test Case ID	ICFD-BENCH-1.1
Test Case Status	Under consideration
Test Case Classification	Benchmarking
Metadata	FREE SURFACE

Table 1: Test Case Summary

3 Test Case Specification

3.1 Test Case Purpose

The purpose of this test case is to study the water entry of a two dimmensional body of conical shape with different deadrise angles.

3.2 Test Case Description

The slamming effect occurs during impact between a blunt body and the water. It is therefore of one the great issues that have to be addressed by the research community in Naval shipbuilding ([1]). Other crucial applications include the landing of space capsules into the sea. Figure (1) offers a sketch of the problem. The entering body will be modeled as a wedge with varying deadrise angles α, entering an initially calm free surface with a constant velocity.

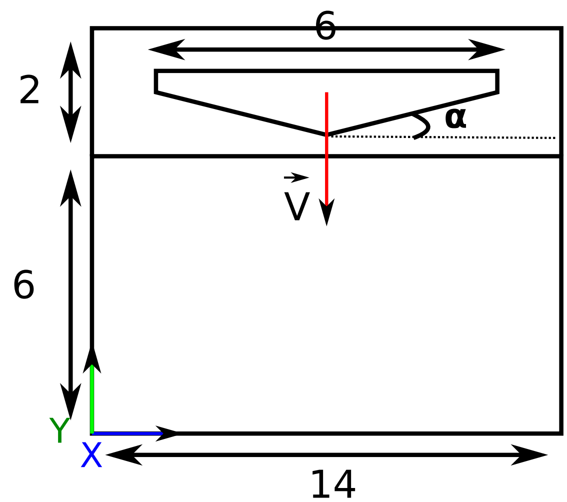

Figure 1: Test Case Sketch

The test case's objective is to study the pressure behavior along the wedge as the it enters the water for the chosen deadrise angles. The maximum pressure coefficent ($Cp_{max} = \frac{P_{max}-P_o}{0.5\rho V^2}$) values will then be compared to those of [2].

3.3 Model Description

The description of the model's geometry is shown in Figure (2) for a deadrise angle of 10° while Table (2) provides some information on the mesh. For this study the deadrise angle will be varied from 4° to 40°. Table (3) gives the physical parameters that will be used. The model consists of two phases, the liquid column and the air or vaccum. Therefore the physical parameters given correspond to the fluid's parameters. In this case, the gravity is neglected and the fluid is considered inviscid.

Model information	
Surface Wedge Element size (Approx)	0.01
Volume Nodes (Approx)	126500
Volume Elements (Approx)	251250
Elements added to the wedge boundary layer	4

Table 2: Test Case Mesh Information

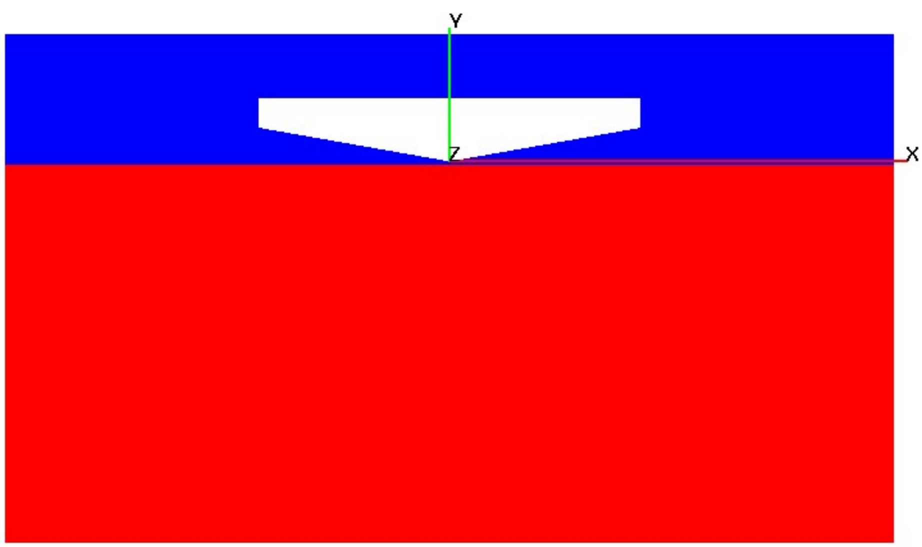

Figure 2: Test Case Geometry

Model physical parameters	
Fluid Density	2
Wedge constant velocity	1

Table 3: Test Case Parameters

4 Test Case Results

4.1 Test Case observations

The pressure behavior along the wedge for the different dearise angles can be observed in Figure (4). The results for the maximum pressure coefficient are summed up in Table (4). As similarly done in [2] the values for the maximum pressure have been obtained by averaging the values in the time interval $T_1 < t < t_{max}$ where T_1 corresponds to some instants after the wedge enters the water and the initial oscillations have been smoothed out and t_{max} corresponds to the moment when the water jet reaches the end of the wedge (see Figure (3)) (Different for the various deadrise angles). Figure (5) shows a good agreement between the numerical solution obtained by [2] and the present analysis. It also shows good agreement with the previous results obtained by using the asymptotic expansion of Wagner's(1932) ([2]) local jet flow analysis and Dobrovol'skaya's(1969) similarity solutions ([2]).

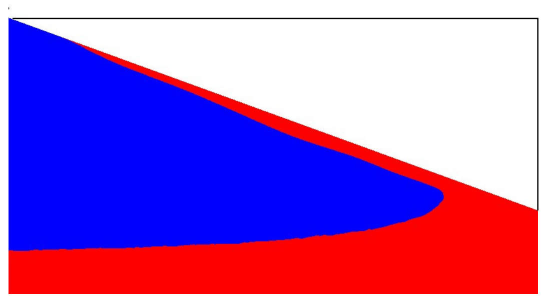

Figure 3: Visualisation of the jet flow "running" up the wedge as it enters the water

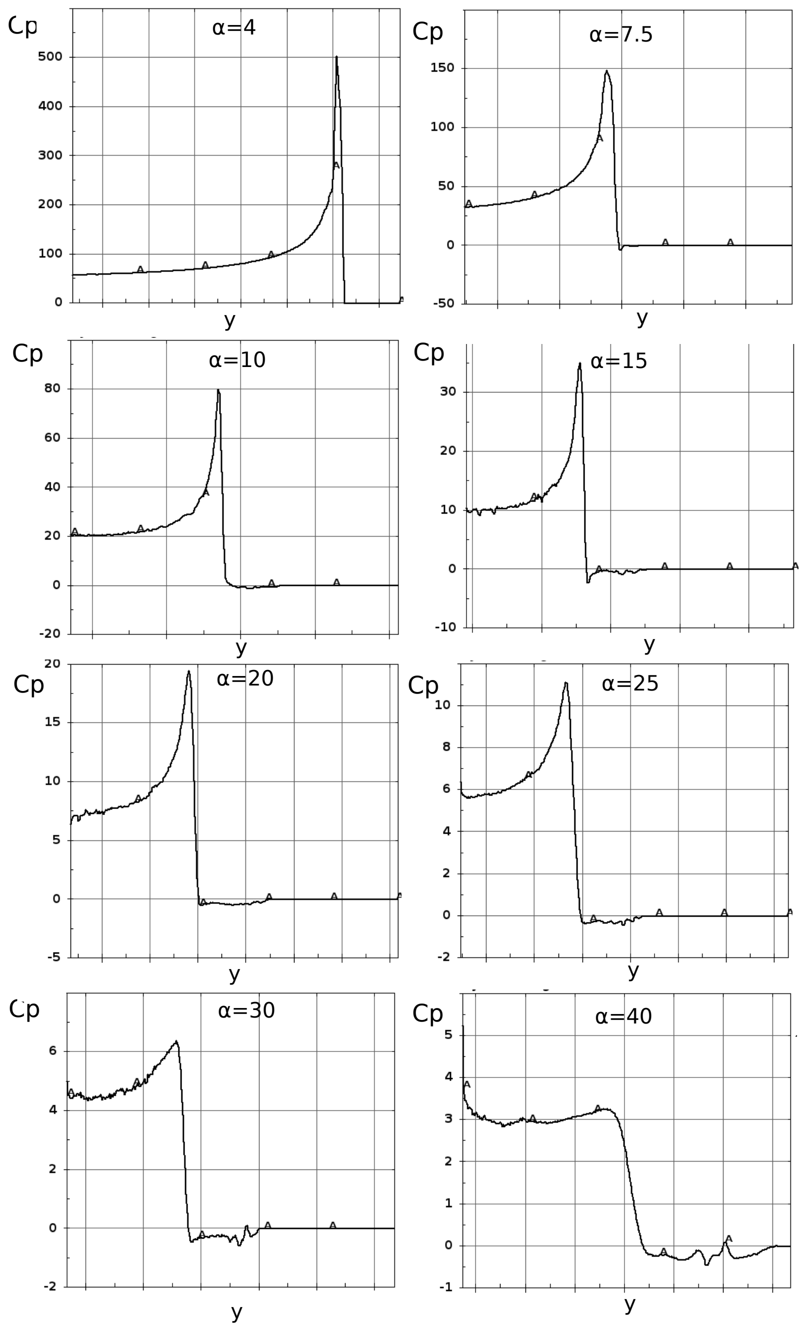

Figure 4: Pressure distribution along the wedge function of the y coordinate.

Maximum Pressure Coefficients	
Wedge angle α	Cp_{max}
4	506.9
7.5	147.7
10	81.6
15	34.2
20	18.8
25	11.2
30	7.35
40	4.0

Table 4: Test Case Presure results

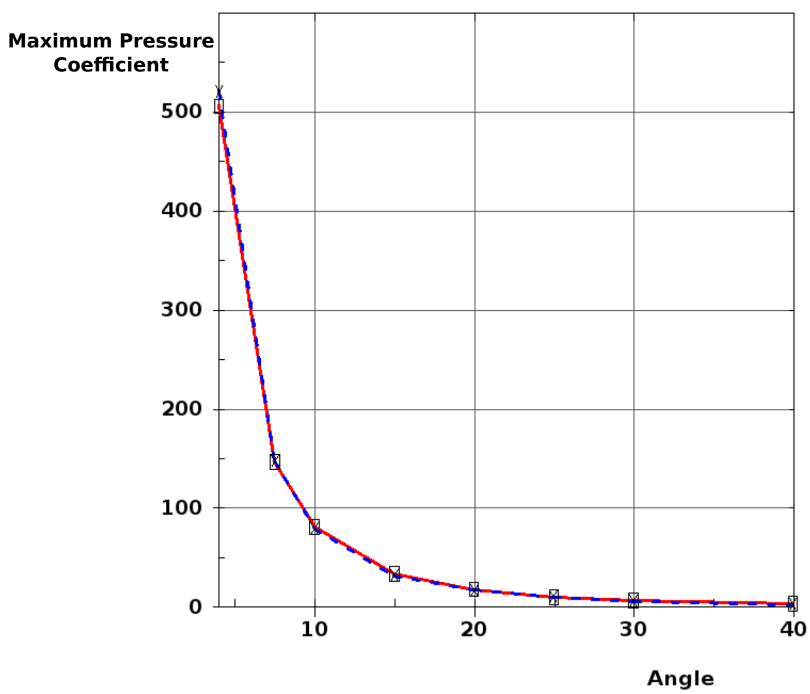

Figure 5: Superimposition of R.Zhao's (see [2]) numerical boundary element method results (dotted, in blue) with the present analysis (in red)

References

[1] C. d. J. N. M. L. R. Marcer, C. Berhault, *Validation of cfd codes for slamming*, European Conference on Computational Fluid Dynamics, (2010).

[2] O. F. R. Zhao, *Water entry of two-dimensional bodies*, J. Fluid Mesh, vol 246.