TEST CASE DOCUMENTATION
AND TESTING RESULTS

TEST CASE ID ICFD-BENCH-2.1

Driven Cavity Flow

Tested with LS-DYNA® v980 Revision Beta

Friday 1st June, 2012

Document Information	
Confidentiality	**external use**
Document Identifier	LSTC-QA-LS-DYNA-ICFD-BENCH-2.1-1
Author(s)	Iñaki Çaldichoury, Facundo Del Pin
Number of pages	11
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-2.1. 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	Cavity: formation of vortexes in a 2D Cavity
Test Case ID	ICFD-BENCH-2.1
Test Case Status	Under consideration
Test Case Classification	Benchmarking
Metadata	INTERNAL FLOW

Table 1: Test Case Summary

3 Test Case Specification

3.1 Test Case Purpose

The purpose of this test case is to study the apparition and the location of vortexes in a two dimensional driven cavity with a velocity transversal boundary condition applied at the top of the cavity.

3.2 Test Case Description

The driven cavity problem has long been used as a benchmarking test case for incompressible CFD solvers. The problem geometry is simple and two-dimensional, and the boundary conditions are also simple. The standard case is a fluid contained in a square domain with Dirichlet boundary conditions on all sides, with three stationary sides and one moving side (with velocity tangent to the side). A sketch figure of the problem can be seen at Figure (1) where the location and the associated number of the different vortexes are shown. For this test case, the presence and the positions of the different vortexes will be studied as well as some velocity profiles for different Reynolds numbers and mesh sizes. The results will be compared to different results available in literature from several different codes.

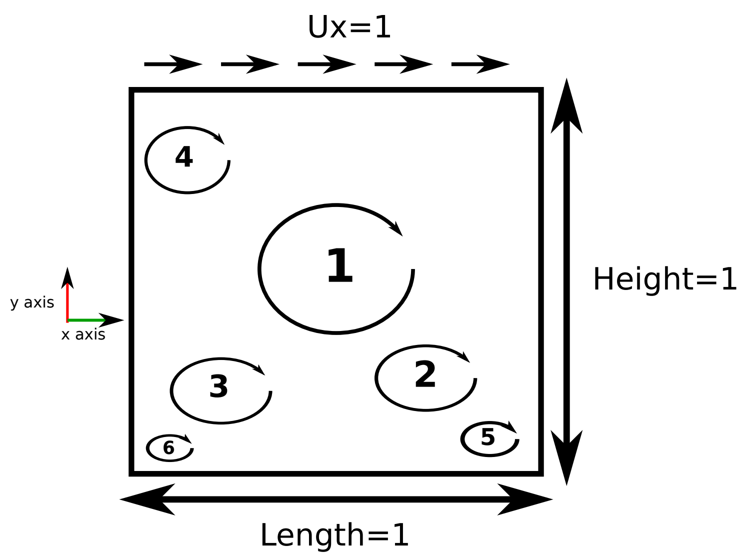

Figure 1: Test Case Sketch

The test case's primary objective is therefore to study the (x, y) position of all the vortexes that can be spotted for the various mesh sizes and Reynolds numbers and to compare them to results found in literature. A second objective is to analyze the mesh convergence by studying the velocity profiles along the vertical and horizontal axes, $U_x(x = 0.5)$ and $U_y(y = 0.5)$ and by comparing them to results found in literature.

3.3 Model Description

For this test case, three mesh sizes will be studied namely 40*40, 80*80 and 160*160 elements detailled in Table (2). The different resulting volume meshes are shown in Figure (2). Table (3) gives the physical parameters that will be used with the viscosity depending on the Reynolds number.

Model information	
Surface Element size(40*40)	0.025
Volume Nodes(40*40)	1686
Volume Elements(40*40)	3210
Volume Nodes(80*80)	6570
Volume Elements(80*80)	12818
Volume Nodes(160*160)	25943
Volume Elements(160*160)	51244

Table 2: Test Case Mesh

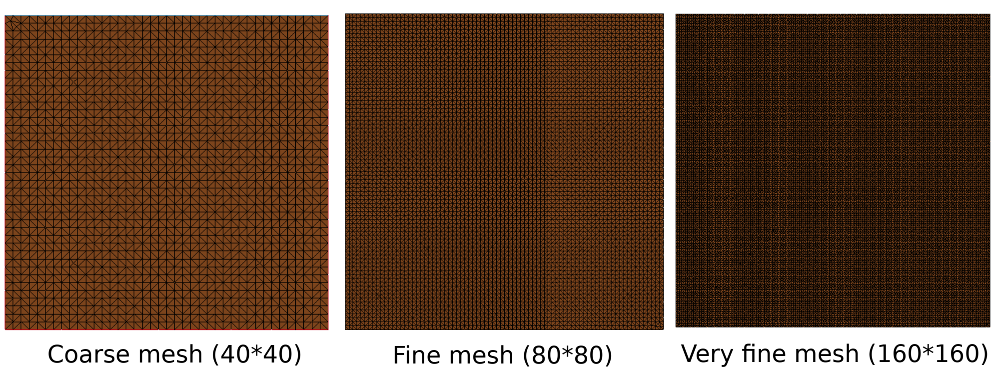

Figure 2: Test Case volume meshes

Model physical parameters	
Fluid Density	1
Incoming velocity	1
Viscosity	0.001,0.0002,0.0001

Table 3: Test Case Parameters

4 Test Case Results

4.1 Test Case observations

Figure (3) shows the velocity fringes after the different cases have run. The main vortex can be clearly identified. The following figures (see Figures (4),(5),(6)) consist in zooming on the different zones in order to better identify the other vortexes. All results shown in the figures are for the finest mesh (160*160) for each Reynolds number. For $Re = 1000$, only three vortexes can be captured even for a fine mesh (see Figure (4)). This result disagrees with some of the references found in literature but seems to be in accordance with others(See Table (5)). For $Re = 5000$, the first four vortexes are captured even for a coarse mesh. For a finer mesh, vortex five can be captured but with an inaccurate location. For the finest mesh, vortex number five can be captured accurately. Vortex number six also starts to appear for the finest mesh but inaccurately. The same observations can be made $Re = 10000$. It is likely that for vortex number six to be correctly represented, a finer mesh is needed. In order to validate this hypothesis, a mesh of 320*320 elements was tested for this two Reynolds numbers. The results are shown in Table (4). The positions of the vortexes agree very well for the finer meshes with those of several references shown in Table (5).

Figures (7),(8) and (9) show the convergence of the results for the velocity profiles for the different mesh sizes. For $Re = 1000$, the results match those of [3] with a 80*80 elements mesh whereas for higher Reynolds numbers, a finer mesh is needed thus further justifying the use of finer meshes as the Reynolds number grows.

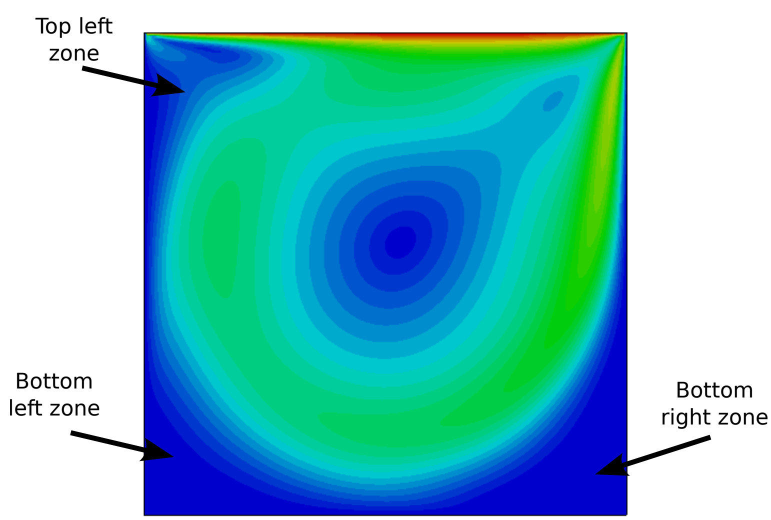

Figure 3: Test Case velocity fringes

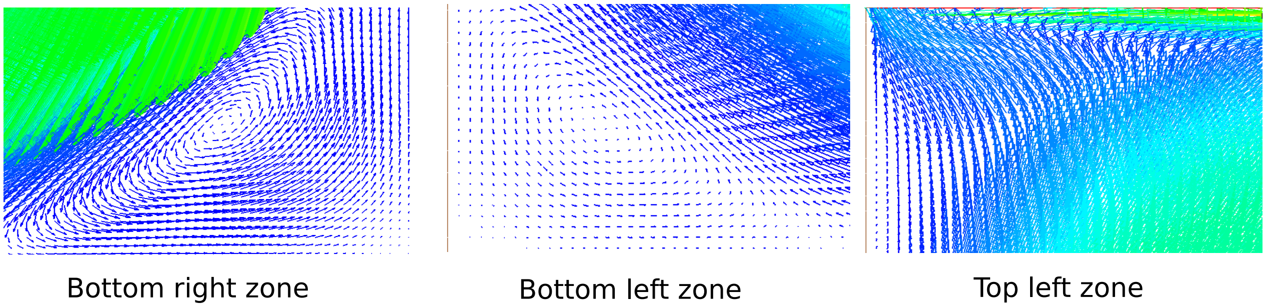

Figure 4: Test Case velocity vectors for $Re = 1000$

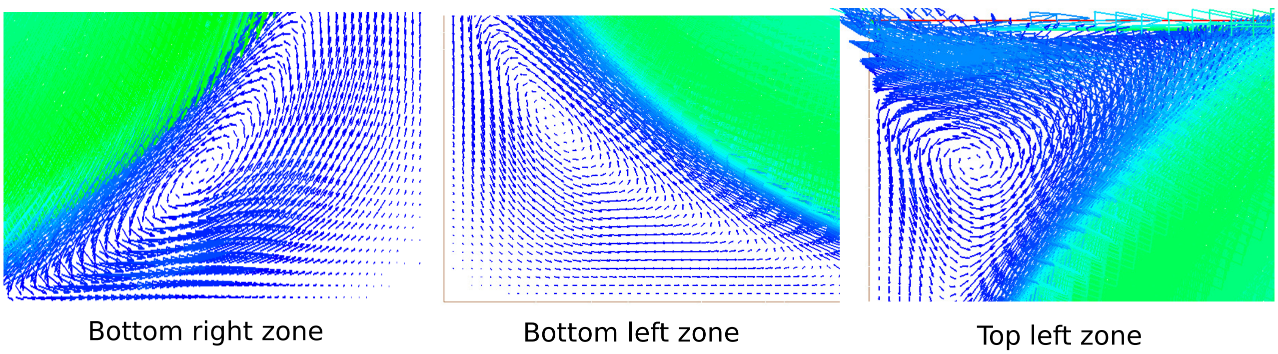

Figure 5: Test Case velocity vectors for $Re = 5000$

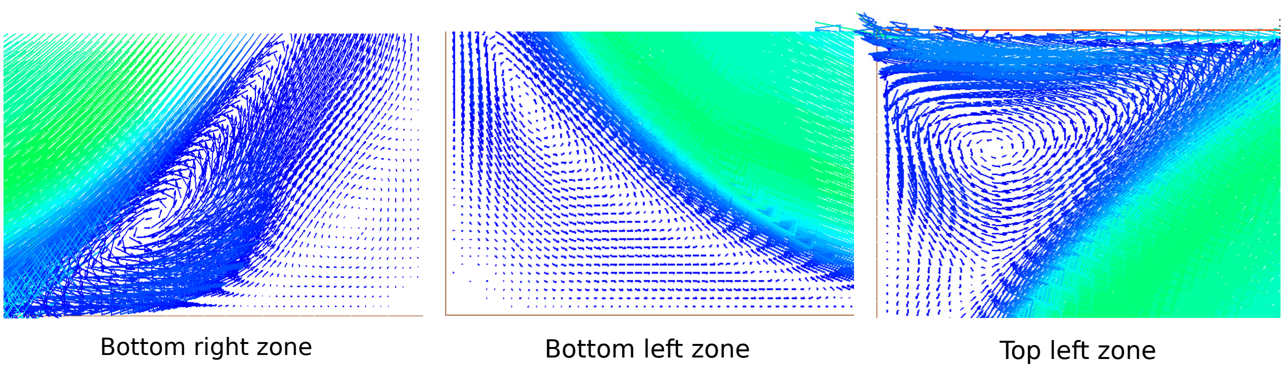

Figure 6: Test Case velocity vectors for $Re = 10000$

	Vortex 1	**Vortex 2**	**Vortex 3**	**Vortex 4**	**Vortex 5**	**Vortex 6**
Re=1000(40*40)	0.5375/0.5875	0.85/0.125	0.075/0.075			
Re=1000(80*80)	0.5313/0.5688	0.8625/0.1125	0.0875/0.075			
Re=1000(160*160)	0.5313/0.5656	0.8625/0.1125	0.0844/0.0781			
Re=5000(40*40)	0.5250/0.5500	0.85/0.10	0.0875/0.1125			
Re=5000(80*80)	0.5125/0.5375	0.800/0.0813	0.075/0.125	0.0625/0.906	0.9912/0.0088	
Re=5000(160*160)	0.5156/0.5344	0.8030/0.0750	0.0750/0.1359	0.0656/0.9094	0.9813/0.0188	0.0045/0.0045
Re=5000(320*320)	0.5156/0.5359	0.8030/0.0735	0.0734/0.0781	0.0640/0.9109	0.9797/0.01875	0.0078/0.0078
Re=10000(40*40)	0.5375/0.5875	0.8625/0.1375	0.075/0.075			
Re=10000(80*80)	0.5310/0.5690	0.8625/0.1125	0.0875/0.075	0.06875/0.906	0.956/0.0375	
Re=10000(160*160)	0.5125/0.5313	0.7656/0.5938	0.0563/0.1563	0.0688/0.9125	0.9313/0.0563	0.0125/0.0125

Table 4: Numerical results for vortexes positions

	Vortex 1	**Vortex 2**	**Vortex 3**	**Vortex 4**	**Vortex 5**	**Vortex 6**
Re=1000([3])	0.5313/0.5625	0.8594/0.1094	0.0859/0.0781	0.9922/0.0078		
Re=1000([4])	0.5308/0.5660	0.8643/0.1115	0.0832/0.0775	0.9941/0.0066		
Re=1000([2])	0.5313/0.5654	0.8643/0.1123	0.0830/0.0781			
Re=1000([1])	0.5409/0.5855	0.8684/0.1072	0.0760/0.0754			
Re=5000([3])	0.5117/0.5352	0.8086/0.0742	0.0703/0.1367	0.0625/0.9102	0.9805/0.0195	0.0117/0.0078
Re=5000([4])	0.5148/0.5362	0.7959/0.0706	0.0728/0.1365	0.0624/0.9076	0.9728/0.0223	0.0070/0.0073
Re=5000([2])	0.5146/0.5355	0.8057/0.0732	0.0732/0.1367	0.0430/0.8896	0.9785/0.0186	0.0078/0.0078
Re=5000([1])	0.5029/0.5420	0.8012/0.0638	0.0754/0.1345	0.0585/0.9130		
Re=10000([3])	0.5117/0.5333	0.7656/0.0586	0.0586/0.1641	0.0703/0.9141	0.9336/0.0625	0.0156/0.0195
Re=10000([4])	0.5064/0.5284	0.7548/0.0555	0.0578/0.1659	0.0709/0.9092	0.9266/0.0791	0.0138/0.0163
Re=10000([2])	0.5117/0.5303	0.7754/0.0596	0.0586/0.1621	0.0703/0.9111	0.9355/0.0674	0.0166/0.0205
Re=10000([1])	0.5000/0.5420	0.7573/0.0551	0.0676/0.1536	0.0676/0.9130		

Table 5: Reference values for vortexes positions

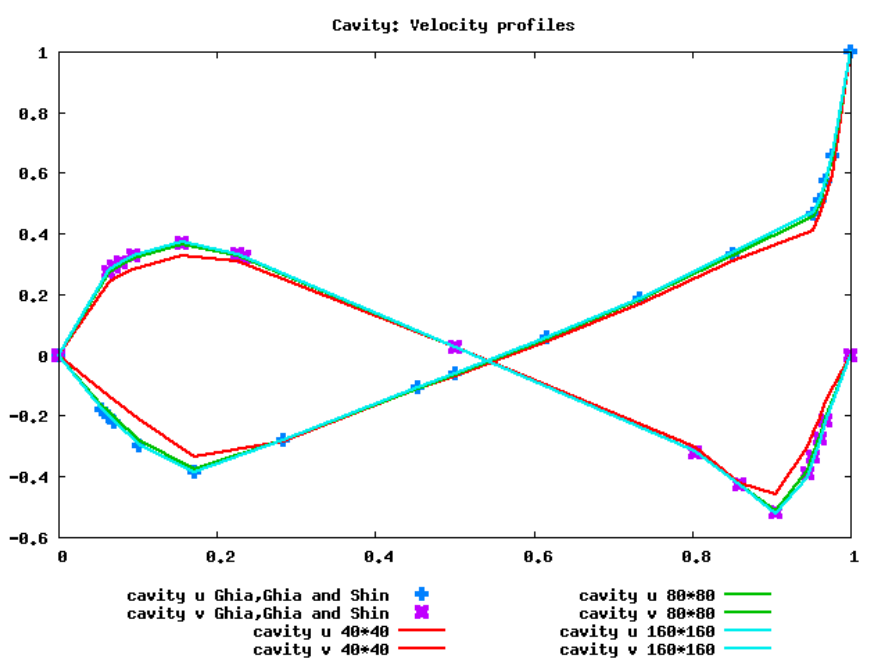

Figure 7: Test Case velocity profiles, $U_x(x = 0.5)$ and $U_y(y = 0.5)$, for different mesh sizes at $Re = 1000$. Comparison with reference result points by [3]

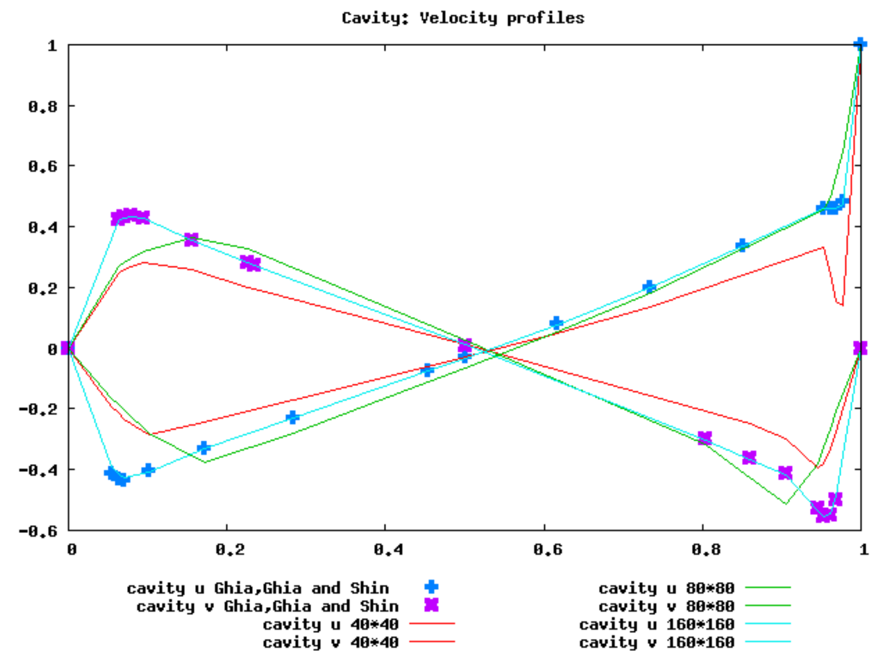

Figure 8: Test Case velocity profiles, $U_x(x = 0.5)$ and $U_y(y = 0.5)$, for different mesh sizes at $Re = 5000$. Comparison with reference result points by [3]

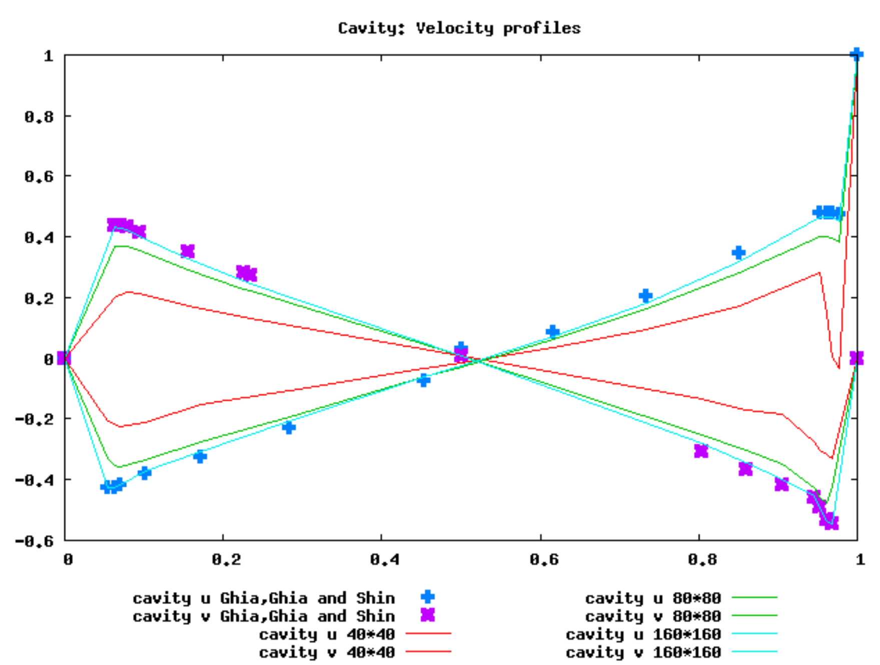

Figure 9: Test Case velocity profiles, $U_x(x = 0.5)$ and $U_y(y = 0.5)$, for different mesh sizes at $Re = 10000$. Comparison with reference result points by [3]

References

[1] O. Cruchaga, M.A., *A finite element formulation for incompressible flow problems using a generalized streamline operator*, Computer Methods in Applied Mechanics and Engineering, 143 (1997), pp. 49–67.

[2] E. Erturk, *Discussions on driven cavity flow*, INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, 60 (2009), pp. 275–294.

[3] U. GHIA, K. N. GHIA, and C. T. SHIN, *High-re solutions for incompressible using the navier-stokes equations and a multigrid method**, COMPUTATIONAL PHYSICS, 48 (1982), pp. 187–411.

[4] W. A. Wall, *Fluid Struktur Interaktion mit stabilisierten Finiten Elementen*, PhD thesis, Universität Stuttgart Institut für Baustatik, 1999.