15th Australasian Fluid Mechanics Conference
The University of Sydney, Sydney, Australia
13-17 December 2004

An Investigation of Flow over a Two-Dimensional Circular Cavity

C.C. Beves, T.J. Barber, E. Leonardi

School of Mechanical & Manufacturing Engineering
The University of New South Wales, Sydney, NSW, 2052 AUSTRALIA

Abstract

The flow structure within a two-dimensional spherical cavity on a flat plate has been studied numerically. A verification analysis of the grid spacing has been conducted; and a validation of the turbulence schemes has been performed by comparing with experimental results obtained using flow visualisation techniques.

Nomenclature

δ depth of the cavity

D effective diameter of the cavity

R_e Reynolds Number

T.I. Turbulence Intensity

Introduction

The application of spherical cavity wells to enhance heat transfer by turbulent mixing of the boundary layer has been investigated both numerically [2, 4] and experimentally [1, 6]. The benefits of using spherical cavity wells, as opposed to more conventional techniques e.g. vortex generators, comes from the improved hydrodynamic characteristics which result from the reduced drag since the cavities do not protrude into the boundary layer.

Previous work [1-7] focused primarily on the benefits to heat transfer by using an array of such cavities. Several main findings were consistent among the literature:

- Flow separation exists within a cavity of $\delta/D > 0.2$
- A vortex leaving the cavity pushes oncoming external flow away from the cavity
- Rounding the edges of the cavity reduced hydraulic friction.

An inconsistency was reported by [3], which refers to the limited understanding of the detailed structural characteristics of the vortices shed from the cavities. A detailed verification and validation analyses on the work conducted may lead to a better understanding of this.

In this paper, two major factors affecting numerical results, namely grid spacing and turbulence modelling are considered. Numerical simulations are conducted using the commercially available CFD package Fluent 6.1.22. A simplified two-dimensional representation of the cavity will be used as a preliminary step, with the aim of building on this knowledge to a three dimensional representation.

Flow Structure Characteristics

A region of flow recirculation exists for cavities with $\delta/D>0.22$. For the most part the centre of this recirculation exists (at approximately 0.5δ) below the surface within which the cavity sits and towards the upstream face of the cavity, as shown in figure 1. For the problem studied Reynolds number is in the order of $3.0x10^3$ with respect to the cavity diameter (D).

The location of this recirculation zone is largely dependent on Reynolds number. As R_e is increased to the values near $3.0x10^5$, the recirculation zone moves closer to the free surface but does not protrude above the free surface and shifts towards the downstream face of the cavity.

There exists a point at which the flow separates from the cavity which is reasonably consistent for the range of Reynolds number solved ($2.7x10^3$ to $3.0x10^5$). For the most part this coincides with the point at which the edge radius and the upstream face of the cavity become tangent.

As the edge radius decreases this clearly impacts on the flow separation point by bringing it closer to the surface within which the cavity is located. This affects the flow structure within the cavity as the flow recirculation zone and the flow separation point are heavily linked, therefore is within the wake of the flow separation that the recirculation zone exists.

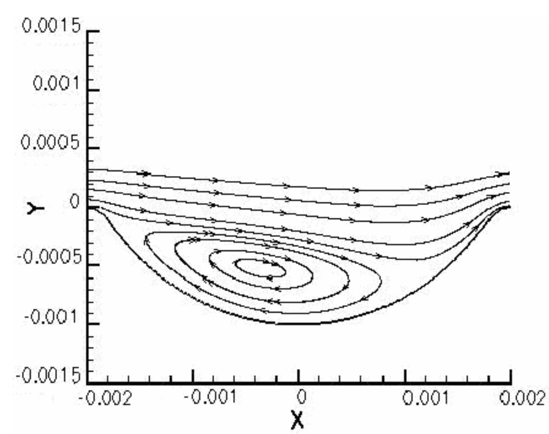

Figure 1 Velocity Streamline plot of the flow within the cavity $\delta/D=0.22$ $R_e = 2.7x10^3$ κ-ε realisable turbulence model at T.I. 10%.

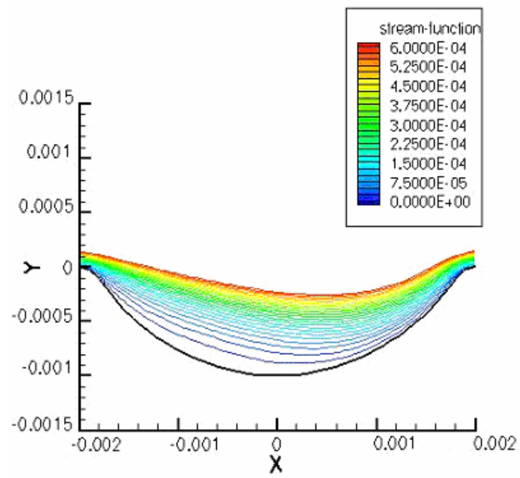

Figure 2 Stream function plot of the flow in the cavity δ/D=0.22 $R_e = 2.7x10^3$ for grid spacing 0.1 mm κ-ε realisable turbulence model at T.I. 10%.

Grid Spacing within the Cavity

Initial steps were taken to ensure the accurate modelling of the flow within the cavity; this focused on how fine the grid spacing within the cavity should be.

Five different grid spacings were used within the cavity (4 mm in diameter and 1 mm in depth): 0.02 mm, 0.04 mm, 0.06 mm, 0.08 mm and 0.1 mm. This corresponded to 192 064, 107 996, 71 571, 49 276 and 45 958 elements respectively throughout the domain (measuring 50 mm x 244 mm).

Changing the grid spacing along the face of the cavity yielded some interesting results; the first of which was that for the coarsest spacing (0.1 mm) there was no presence of any flow recirculation within the cavity, see figure 2. As the grid spacing was further refined to 0.08 mm, this yielded a stalled region of flow within the upstream face of the cavity. From 0.06 mm to 0.02 mm the flow recirculation clearly exists, however as there are more cells located within the cavity the centre of the flow recirculation migrates towards the central part of the cavity, see figure 3. The change in location and size of the recirculation zone did not vary greatly between the 0.04mm and 0.02mm cases, thus it was concluded that sufficient grid convergence was achieved.

The intensity of the recirculation increases for finer grids, as the recirculation region becomes clearly defined and the flow that travels back up the upstream face of the cavity is accelerated locally compared to the coarser grids.

Investigation of the static pressure profile within the cavity shows that for the coarsest grid spacing, there is a larger region of positive pressure that covers most of the surface of the cavity and extends out of the cavity. As the grid spacing is refined, the size of the high static pressure region retracts to where there is only a localised region on the downstream face of the cavity where the edge radius and the cavity are tangent.

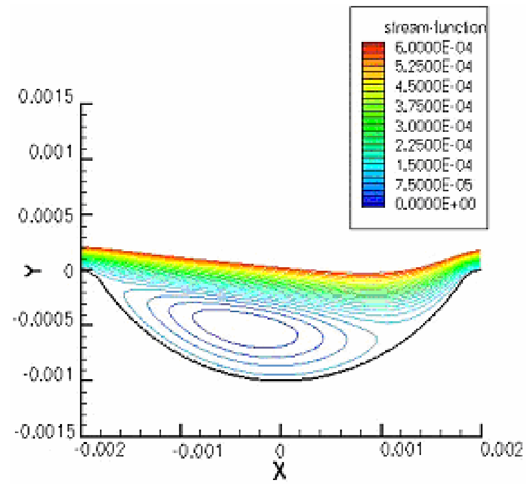

Figure 3 Stream function plot of the flow in the cavity δ/D=0.22 $R_e = 2.7x10^3$ for grid spacing 0.02mm κ-ε realisable turbulence model at T.I. 10%.

Not only does the size of the high static pressure region retract but the strength of it increases (from 2.61 Pa to 5.7 Pa). This mirrors the increase in recirculation strength due to local flow acceleration as the grid spacing becomes finer.

Turbulence Modelling

Considering the complex flow structure and the low level of mass transfer within the cavity, a turbulence model validation analysis was conducted using the commonly available turbulence models, namely κ-ε Realisable, RNG κ-ε and Reynolds stress. This was to ensure that when validating the results with those obtained experimentally the most correct model would be used.

A cavity of δ/D = 0.5 was chosen both in the experiments and computational analysis due to ease of manufacture, and also because the flow separation and recirculation zones would be more pronounced for lower experimental R_e numbers (as a smoke will be used for visualisation in this initial study).

κ-ε Realisable

Ranges of turbulence intensity (T.I.) levels were used, from 5% to 20% in 5% intervals (R_e = 8.2 x 10^2). The results showed no discernable change in the location of the centre of the flow recirculation zone for all turbulence intensity values.

The location of this centre was approximately 0.36δ and 0.6D for the height as a percentage of cavity depth from the free surface and the longitudinal placement as a percentage of the cavity diameter from the upstream edge point tangent to the free surface, see figure 4.

With a change in R_e = 2.9 x 10^4 for T.I. = 10% the centre of recirculation zone migrated to 0.38δ and 0.57D, that is to say it shifted down and towards the upstream face.

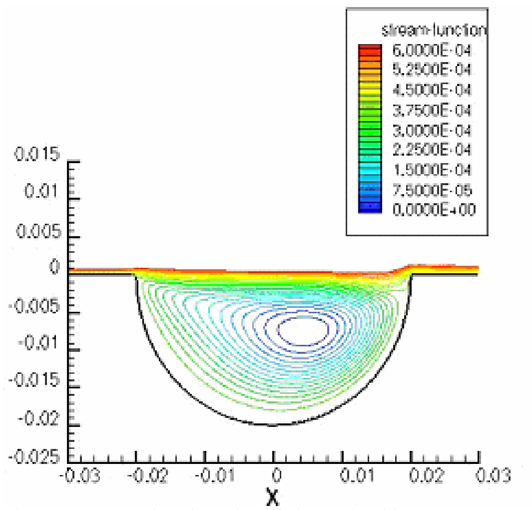

Figure 4 Stream function plot of the cavity δ/D = 0.5 R_e = 8.2 x 10^2 κ-ε realisable turbulence model at T.I. 10%.

κ-ε RNG

A 10% turbulence intensity level was used to compare it to the same T.I. 10% for the κ-ε realisable model as a conservative compensation for the turbulence in the wind tunnel was used. The result exhibited a fairly similar solution to the K- ε realisable 10% T.I.

The centre of flow recirculation existed at 0.43δ and 0.6D from the same point, that is to say the recirculation zone migrated 0.13δ away from the free surface for the RNG T.I. 10% model, but the flow separation point and stagnation points remained relatively unchanged. Thus there is little difference between κ-ε RNG and Realisable κ-ε turbulence models for the flow within a cavity of δ/D = 0.5 for low Reynolds Numbers (8.2 x 10^2).

Reynolds Stress (Re-σ)

A 10% turbulence intensity level was again used to compare with the 10% RNG and 10% Realisable model. The results show the recirculation zone at 0.34δ and 0.40D, which indicates that for the same R_e the Reynolds stress turbulence model predicted the recirculation zone to be slightly higher and significantly further towards the upstream face of the cavity compared with the with κ-ε realisable and κ-ε RNG models, which gave fairly similar results. These solutions must now be validated with experimental results.

Experimentation

Initial experimentation was centred on flow visualisation techniques to validate the numerical models. The information to be retrieved included the flow structure and location of the recirculation position. Also there was additional task to determine if the centre of rotation was changing a position within the cavity with time.

For the experimentation an existing wind tunnel was used, which was of open circuit, closed test section design (see figure 5). The tunnel was made from Perspex to allow the laser sheet to pass into the test section.

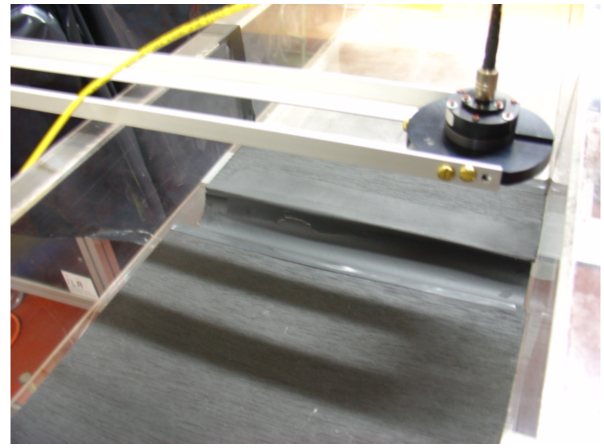

Figure 5 Experimental set-up, showing laser line generator on arm and channel for the two-dimensional cavity

A 25mW JDS ‘Uniphase’ Model 1145 He-Ne laser was used to generate the laser sheet and a Sony DSC-V1 digital camera used to capture the pictures. A digital manometer was used to determine the free stream velocity in the tunnel. For visualisation a Le Maitre G100 smoke generator was used to seed the tunnel.

The test piece consisted of a channel running across the tunnel. A dimple depth to effective dimple diameter ratio of δ/D = 0.5 was chosen due to the ease of manufacture and the visualisation benefits resulting from the more pronounced flow recirculation. The image in figure 6 shows the instance at which the recirculation zone forms within the cavity.

Several observations were made:

- The recirculation zone took quite a while to develop, as the momentum within the cavity is quite small.
- The recirculation zone did slightly reciprocate around itself.
- The interaction of the fluid ejecting from the rear of the cavity and the boundary layer on the surface (within which it sits) caused an oscillation within the boundary layer even at low Re.
- Once the smoke developed in the cavity it remained locally for several minutes without the introduction of additional smoke.

Figure 6 Flow visualisation within cavity δ/D = 0.5 R_e = 8.2 x 10^2.

Turbulence model	Δδ with respect to experiments	ΔD with respect to experiments
Realisable 5%	+0.05	+0.03
Realisable 10%	+0.05	+0.03
Realisable 15%	+0.05	+0.03
Realisable 20%	+0.05	+0.03
RNG 10%	-0.12	-0.03
R_e-σ	+0.03	-0.14

Table 1 Change in the position of recirculation centre with respect to experimental result.

The recirculation zone of the experiment and the numerical result are overall fairly consistent with each other. For the K-ε Realisable turbulence model the results were the most consistent with the experimental result with the deviation in the recirculation centre shown in Table 1.

Conclusions

The flow structure within a two-dimensional spherical cavity on a flat plate has been studied numerically. A two-dimensional configuration was used to develop a consistent turbulence model and ensure that sufficient grid spacing for the three-dimensional case is used. Vortex shedding was not observed, possibly due to a two-dimensional nature of the experiment. However the oscillation of the flow exiting the 'two-dimensional' cavity may cause a similar phenomenon three-dimensionally.

The stability of the flow within the cavity over time suggests that the assumption of a steady state analysis (for such low Reynolds numbers) is appropriate. Although an unsteady solution was obtained, it was observed that once the recirculation zone had formed off the upstream edge of the cavity it stabilised to a position consistent with the steady-state case.

The grid spacing has a significant effect on the flow structure within the cavity, as seen in figures 2 and 3. There need not be too much refinement in the grid for flow recirculation to exist, from meshed edge length 0.1mm to 0.8mm a recirculation zone was established, and the overall flow field within the cavity was approximate to the finer spaced grids, albeit the static pressure profile and recirculation centre changed. For accuracy in modelling the flow within the cavity this requires fairly strong computing power, especially as the simulation progresses from two-dimensional to three-dimensional cases.

The turbulence model validation suggests that a κ-ε realisable scheme is the most consistent to the experimental situation. The turbulence intensity levels did not have too much of an effect within the numerical cases, although more accurate modelling of the turbulence levels within the tunnel may yield a more accurate representation of the recirculation centre for the numerical solution.

References

[1] Ekkad S.V., Nasir H., Dimple Enhanced Heat Transfer in High Aspect Ratio Channels, Proceed. of IMECE: 2001 ASME International Mechanical Engineering Congress & Exposition Nov 1-16, 2001, New York

[2] Isaev S.A., Leont'ev A.I., Baranov P.A., Identification of Self-Organised Vortexlike Structures in numerically Simulated Turbulent Flow of a Viscous Incompressible Liquid Streaming around a Well on a Plane, Technical Physics Letters. **26** (1), 2000.

[3] Jongmyung Park, Desam P.R., Ligrani P.M., Numerical Predictions of Flow Structure above a Dimpled Surface in a Channel, *Journal of Numerical Heat Transfer*, Part A, **45** Part A, 2004, 1-20.

[4] Lin, Y.-L., Shih T.I.-P. Chyu M.K., Computations of Flow and Heat Transfer in a Channel with rows of Hemispherical Cavities, *ASME Paper* No. 99-GT-263.

[5] Mahmood G.I., Hill M.L., Nelson D.L., Ligrani P.M. Moon H.K., Glezer B., *ASME Trans. Turbomachinery* **123**, 2001, 115.

[6] Mahmood G.I., Ligrani P.M., Heat Tracnfer in a Dimpled Channel: Combined Influences of Aspect Ratio, Temperature Ratio, Reynolds Number and Flow Structure, *International Journal of Heat and Mass Transfer*, **45**, 2002, 2011-2020.

[7] Schukin A.B., Kozlov A.P., Chudnovskii Ya.P., Agachev R.S., Intensification of Heat Exchange by Sperical Depressions. A Survey, *Applied Energy: Russian Journal of Fuel, Power, and Heat Systems,* **36 (**3), 1998, 45-62.