Numerical investigation of the Couette flow in a duct with an embedded cavity

E. Rossi di Schio, S. Guidozzi

***Abstract*—**In the present paper a numerical solution of the Couette flow is presented. The considered channes displays a cavity, shortly after the inlet section, that enhances the vortex formation. The governing equations are written in a dimensionless form and solved by employing Comsol Multiphysics, a software package based on a Galerkin finite element procedure. Visualizations of vortex, for different values of the Reylonds number, are provided.

***Keywords*—**Couette flow; vortex; forced convection; finite element analysis.

I. Introduction

IN the literature, the vortex formation on cavities has been widely investigated, both numerically [1-4] and experimentally [5]. The topic has many possibile applications, such as, for instance, the investigation of gyres within open lacustrine embayments [6].

Attention has been paid especially with reference to large cavities. In particular, in [3] a lattice Boltzmann method is used to investigate different cavity geometries and different Reynolds number values. The aspect ratio of the cavity varies from 0.1 to 7 and the Reynolds number ranges from 0.01 to 5000. The effects of the aspect ratio and Reynolds number on the size, center position and number of vortices are determined together with the flow pattern in the cavity.

In the literature, also the Couette flow has been widely investigated, but, with reference to the presence of cavities, the attentions has been paid especially to the stability analysis [7-8].

Although the topic does not present particular novelty, no recent analyses on the vortex formation on cavities, due to Couette flow are available in the literature.

In the present paper, a preliminary numerical study of the vortex formation, at low Reynolds numbers, for Couette flow in a place channel with an embedded cavity is presented.

II. Mathematical Model

Let us consider a Newtonian fluid with constant thermophysical properties flowing, in steady conditions, in a channel having variable cross section. Let us assume a two dimensional flow, such that the velocity vector is **U**=(U,V) .

Let us assume that the duct displays a cavity: backward and forward steps.

Let us define the numerical domain as sketched in fig. 1, where the parameters D, σ and λ are defined as well.

Let us assume that the upper walls moves with uniform velocity U_0, and that at the inlet section a liner velocity profile is prescribed.

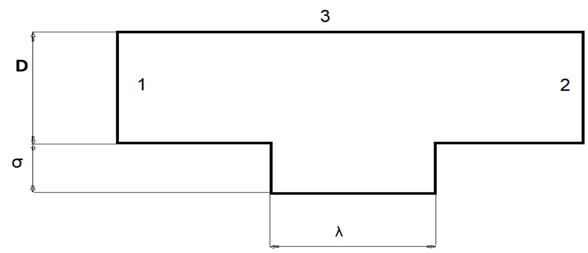

Fig. 1 Sketch of the geometry

The governing equations are the mass and momentum local balance equations. By introducing the following dimensionless quanities,

$$(1)\quad \begin{aligned} &(x,y)=(X,Y)/D,\\ &(u,v)=(U,V)/U_0,\\ &p=\frac{P-P_0}{\rho U_0^2},\\ &\mathrm{Re}=\frac{U_0 h}{\nu}, \end{aligned}$$

and on account of the local mass balance equation,

$$(2)\quad \frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0,$$

one can write the local momentum balance equation, for the steady state, in a dimensionless form as follows:

$$(3)\quad (\mathbf{u}\cdot\nabla)\mathbf{u}=-\nabla p+\frac{1}{\mathrm{Re}}\nabla^2\mathbf{u}.$$

In Eq. (1), p is the dimensioness pressure drop and Re is the

Alma Mater Studiorum – Università di Bologna. Department of Industrial Engineering. Viale Risorgimento 2. I-40136 Bologna. E-mail: eugenia.rossidischio@unibo.it

Reynolds number, ρ is the fluid density and ν is its kinematic viscosity.

Equation (3) will be solved together with the no slip boundary conditions, the inlet boundary condition (on the boundary denoted by “1”in Fig 1) given by a Couette velocity profile, namely

$$u = y, \quad v = 0, \tag{4}$$

and the outlet boundary condition (on the boundary denoted by “2”in Fig 1), namely p=0.

III. Numerical Solution

The numerical solution presented is obtained by employing the software package Comsol Multiphysics, based on a Galerkin finite element analysis.

An unstructured mesh of 14672 triangular elements is employed, and a grid refinement is made for the cavity. The obtained solution is checked to be independent from the particular mesh considered, as well as from the outlet section boundary condition prescribed.

A dimensionless channel having length 15 is defined, ad shown in Fig 2. Since the dimensions of the cavity are clearly defined through the dimensionless quantities λ and σ, simulations are done for different values assumed by the parameters λ, σ and Re.

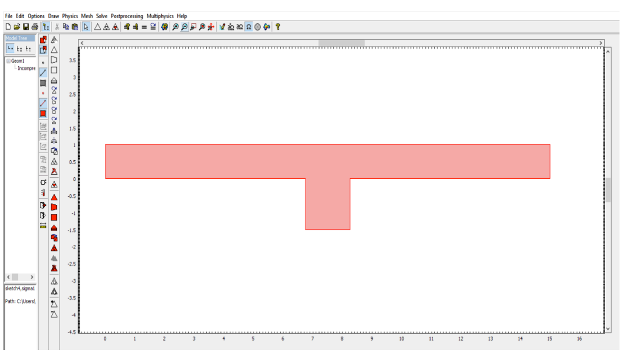

Fig. 2 Sketch of the numerical domain

IV. Discussion of the Results

Let us first consider the case σ=0.5 and λ=1.5. In Fig. 3, the velocity distribution is reported for Re=50.

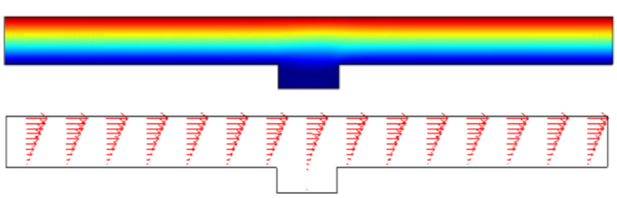

Fig. 3 Velocity distribution for σ=0.5, λ=1.5 and Re=50

A zoom of the velocity streamlines with reference to the cavity is reported in Fig. 4: the presence of a vortex is evident. The position of the vortex center is (7.229;-0.193).

A comparison between Figs 4 to 7, referring all to σ=0.5 and λ=1.5. but to different values of the parameter Re, allows one to investigate how the vortex center varies its position for increasing value of the Reynolds number.

In fact, the vortex center tends to move, for increasing values of the parameter Re, in the direction of the outlet section.

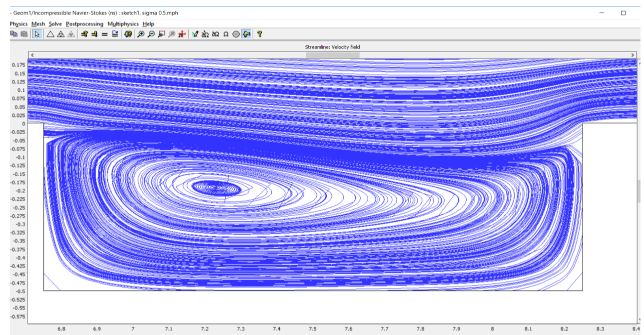

Fig. 4 Velocity streamlines in the cavity, for σ=0.5, λ=1.5 and Re=50

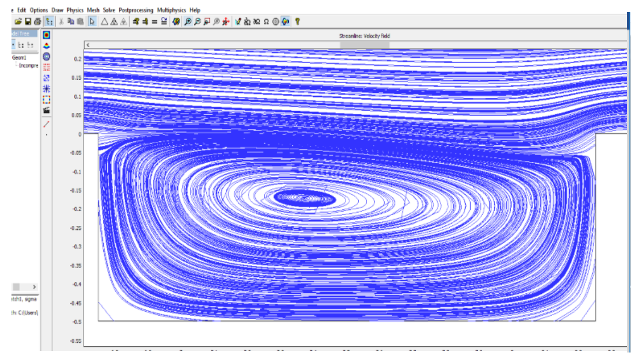

Fig. 5 Velocity streamlines in the cavity, for σ=0.5, λ=1.5 and Re=100

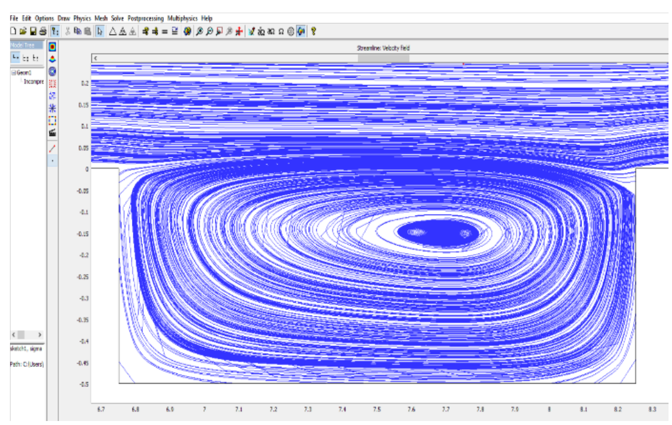

Fig. 6 Velocity streamlines in the cavity, for σ=0.5, λ=1.5 and Re=300

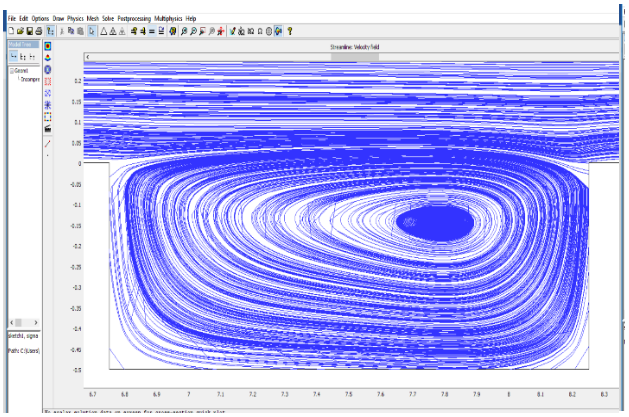

Fig. 7 Velocity streamlines in the cavity, for σ=0.5, λ=1.5 and Re=500

Let us now discuss the influence of the parameter λ. Figures 8 and 9, refer to σ=0.5, Re=25 and λ=3 or λ=4 respectively.

Moreover, Figs. 9 to 13 allow one to investigate the effect of the Reynolds number for a shallw cavity (λ=4). A compartison between the figures

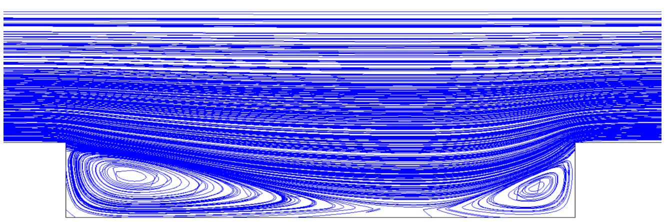

Fig. 8 Velocity streamlines in the cavity, for σ=0.5, Re=25 and λ=3

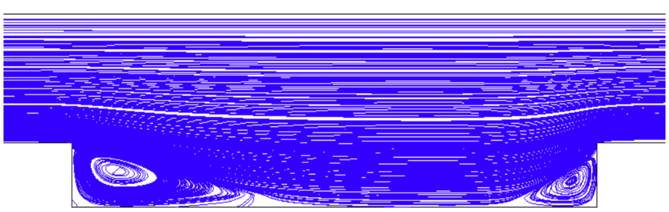

Fig. 9 Velocity streamlines in the cavity, for σ=0.5, Re=25 and λ=4

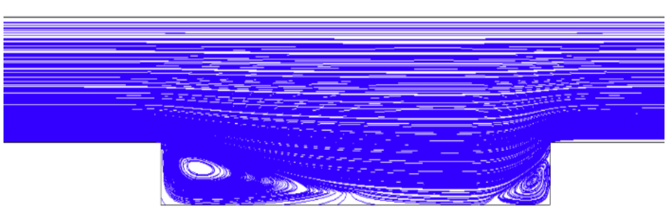

Fig. 10 Velocity streamlines in the cavity, for σ=0.5, Re=40 and λ=4

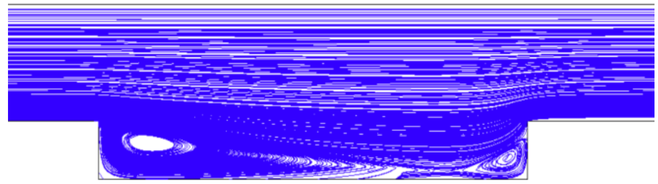

Fig. 11 Velocity streamlines in the cavity, for σ=0.5, Re=65 and λ=4

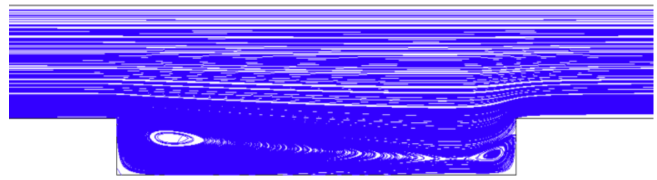

Fig. 12 Velocity streamlines in the cavity, for σ=0.5, Re=100 and λ=4

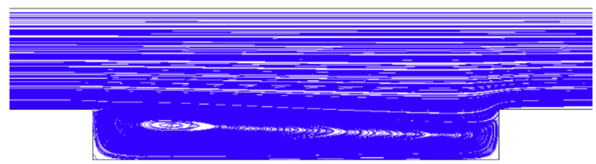

Fig. 13 Velocity streamlines in the cavity, for σ=0.5, Re=200 and λ=4

In fact, the figures show that two vortexes are formed in the edges of the cavity, where wide stagnation regions are presented as well. The vortex on the lefthand side of the cavity is, for both the considered values of λ, larger than the righthand side one: in fact, since the direction of speed is positive, there is less fluid flow in this region than the right one.

We can also observe that for increasing values of the parameter λ, a formation of several vortices for low Re occurs, and the vortex pair will rotate in the same direction, both in clockwise direction.

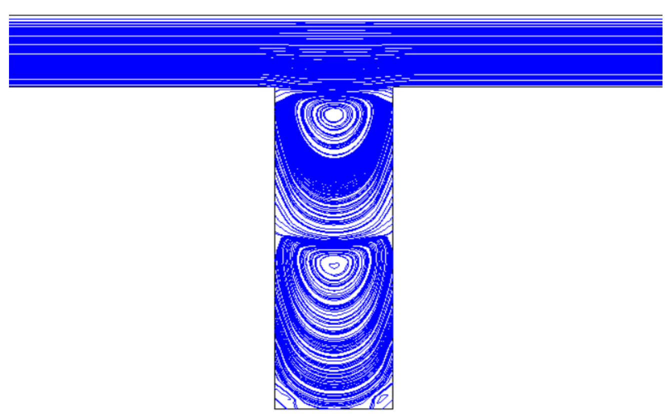

Fig. 14 Velocity streamlines in the cavity, for σ=4.5, Re=1 and λ=1.5

For λ = 4, for increasing values of Re, as shown in Figs. 9-13, the two vortexes on the edges of the cavity are enlarged towards the central region and then come into contact. For Re=100, they melt into a lonely vortex

Finally, the vortex formations for a tall cavity (σ=4.5) is shown in Fig. 14 for Re=1.

V. CONCLUSION

In the present paper, a numerical investigation of the Couette flow in a channel having variable cross section is presented. The governing equations, i.e. the local mass and momentum balance equations, are written in a dimensionless form. The governing dimensionless parameters are the geometrical parameters characterizing the cavity, together with the Rewnolds number. Reference is made to stady laminar flow, and the governing equations are solved by employing the software package Comsol Multiphysics, based on a Galerkin finite element analysis. Features of the obtained vortexes are widely discussed.

This analysis provides a preliminary analysis. Further investigations will be performed, especially with reference to tall cavities and to the exact position of the vortex center, as a function of al the considered parameters.

REFERENCES

[1] C. Shen and J. M. Floryan, “Low Reynolds number flow over cavities” , Physics of Fluids 28, ,1985, pp. 3191; doi: http://dx.doi.org/10.1063/1.865366

[2] K. Gustafson, K. Halasi, “Vortex dynamics of cavity flows” , Journal of Computational Physics 64, 1986, pp. 279-319; doi: https://doi.org/10.1016/0021-9991(86)90035-5

[3] M. Cheng, K.C. Hung, “Vortex structure of steady flow in a rectangular cavity” , Computers & Fluids 35, 2006, pp. 1046-1062; doi: https://doi.org/10.1016/j.compfluid.2005.08.006

[4] M. Rosenfeld, E. Rambod, M. Gharib, “Circulation and formation number of laminar vortex ring”, Journal of Fluid Mechanics 376, 1998 , pp. 297-318

[5] Taneda, “Visualization of Separating Stokes Flows”, Journal of the Physical Society of Japan, 46, 1979, pp. 1935-1942

[6] A. M. Razmi, U. Lemmin, D. Bouffard, A. Wüest, R. E. Uittenbogaard, D. A. Barry, “Gyre formation in open and deep lacustrine embayments: the example of Lake Geneva, Switzerland”, Environmental Fluid Mechanics17, 2017, pp 415–428

[7] E. Serre, M.A. Sprague, R.M. Lueptow , Stability of Taylor–Couette flow in a finite-length cavity with radial throughflow, Physics of Fluids 20, 2008, doi: http://dx.doi.org/10.1063/1.288483

[8] G.A. Brès, T. Colonius, Three-dimensional instabilities in compressible flow over open cavities, Journal of Fluid Mechanics 599, 2008, pp. 309-339, doi: https://doi.org/10.1017/S0022112007009925