International Journal of Heat and Fluid Flow 48 (2014) 52–68

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

journal homepage: www.elsevier.com/locate/ijhff

Formation of vortex breakdown in conical–cylindrical cavities

Diego Alves de Moro Martins *, Francisco José de Souza, Ricardo de Vasconcelos Salvo [1]

School of Mechanical Engineering, Federal University of Uberlandia, Av. João Naves de Ávila, 2121 Bloco 5P, Uberlandia, Minas Gerais 38400-902, Brazil

ARTICLE INFO

Article history:
Received 2 August 2013
Received in revised form 28 April 2014
Accepted 2 May 2014
Available online 24 May 2014

Keywords:
Vortex breakdown
CFD
Rotating flow
Turbulence modeling

ABSTRACT

Numerical simulations in confined rotating flows were performed in this work, in order to verify and characterize the formation of the vortex breakdown phenomenon. Cylindrical and conical–cylindrical geometries, both closed, were used in the simulations. The rotating flow is induced by the bottom wall, which rotates at constant angular velocity. Firstly the numerical results were compared to experimental results available in references, with the purpose to verify the capacity of the computational code to predict the vortex breakdown phenomenon. Further, several simulations varying the parameters which govern the characteristics of the flows analyzed in this work, i.e., the Reynolds number and the aspect ratio, were performed. In these simulations, the limits for the transitional regime and the vortex breakdown formation were verified. Steady and transient cases, with and without turbulence modeling, were simulated. In general, some aspects of the process of vortex breakdown in conical–cylindrical geometries were observed to be different from that in cylinders.

1. Introduction

Rotating flows occur in various natural phenomena, like in hurricanes, waterspout and tornados. In industry, it is often used in combustion chambers, turbines, pumps, and cyclone separators. As in other types of flow, rotating flow displays specific features. A typical phenomenon in rotating flow is the vortex breakdown, which can be defined as an abrupt change in flow direction in certain points along the rotation axis, creating one or more recirculation zones with stagnation points, spiral or helical structures. The first scientific record of the vortex breakdown was made by Peckham and Atkinson (1957). By running experiments with delta wings, they observed the breakdown in the leading edge vortices at high angles of attack. After this proof of vortex breakdown, many researchers explored the causes and characteristics of the phenomenon. Ludweig (1962) has proposed that vortex breakdown, with a local stagnation of the axial flow, is a direct consequence of hydrodynamic instability with respect to spiral disturbances. Benjamin (1962) proposed that vortex breakdown can be explained as "a transition between two steady states of axi-symmetric swirling flow, being much the same in principle as the hydraulic jump in open-channel flow". Sarpkaya (1971) carry out experiments in a diverging duct and identified three forms of vortex breakdown: axisymmetric bubble, spiral and double helix.

Due to the easy manipulation, several studies about vortex breakdown phenomenon in confined rotating flow were performed. Vogel (1968, 1975) and Escudier (1984) carried out experiments in cylindrical containers, in order to define the stability limit at which a vortex breakdown bubble occurs on the axis of symmetry. Lopez (1990) examined the vortex breakdown phenomenon using the Navier–Stokes equations with the streamfunction–vorticity formulation. The phenomenon was confirmed for certain values of the governing parameters for flows in an enclosed cylinder, and in agreement with the experimental results (Escudier, 1984). The author verified bubbles evolution in function of the governing parameter and the transition to an oscillatory regime.

Several analyses of the vortex breakdown influence on the transition of rotating flows were carried out. Serre and Bontoux (2002) performed numerical simulations in a cylindrical rotor–stator cavity of large aspect ratio using a highly accurate spectral method. Different topologies of the breakdown bubble, including the spiral shape, were verified. The transition to a periodic regime was observed to occur through an axisymmetric Hopf bifurcation in the sidewall layers. Gelfgat et al. (2001) analyzed the three-dimensional instability of the axisymmetric flow between a rotating lid and a stationary cylinder. They verified the instabilities regime for different aspect ratios and concluded that the onset of the instability is not connected with the vortex breakdown. Blackburn and Lopez (2000) studied the symmetry breaking of the flow and of vortex breakdown in a cylindrical container. They confirmed the three-dimensionality of the flow through rotating waves in the azimuthal velocity component and attributed this symmetry breaking to an instability of the swirling jet produced

* Corresponding author. Tel.: +55 (34) 3239 4040 (617).
E-mail address: moromartins@gmail.com (D.A.d.M. Martins).
[1] Current address: School of Mechanical Engineering, Universidade Tecnológica Federal do Paraná, Avenida dos Pioneiros, 3131, Bloco B, Londrina, Paraná 86036-370, Brazil.

http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.05.001
0142-727X/

by the corner where the rotating bottom meets the stationary sidewall.

Gauthier et al. (1999) investigated experimentally the primary destabilization of the flow between a stationary and a rotating disk when the cell is closed and of rather large aspect ratio. They show that this destabilization occurs in the Bödewadt layer and leads to propagative circular waves. Sørensen et al. (2006) studied experimentally the swirling flow between a rotating lid and a stationary cylinder. They extracted wave modes to map in details the route of transition from steady and axisymmetric flow to unsteady and three-dimensional flow. Pereira and Sousa (1999) performed experiments and numerical simulations on a rotating cone within a closed cylindrical container. They verified the flow behavior according the main parameters of the flow. As Escudier (1984), they defined a map of stability limit in which a vortex breakdown occurs. More recently, Escudier et al. (2007) analyzed the rotating flow inside a truncated cone through numerical simulation. They identified the conditions of the vortex breakdown and Moffatt eddies.

In industry, vortex breakdown can be observed in a number of situations, and it may have a positive influence, like in swirl combustors, in which it acts as an efficient mixer for the combustion process. On the other hand, in high-speed aircrafts, for instance, the breakdown can reduce the lift causing the wing stall. In industrial cyclones the bubble can retain particles in the recirculation zones, thus hindering the separation process.

Since many flows of practical interest take place in complex geometries, it is of interest to investigate the vortex breakdown in geometries other than cylinders. The objective of this paper is to carry out numerical simulations of flows similar to that in cyclones and combustion chambers, in order to characterize the vortex breakdown regimes and the stability limits. Rotating flows at different Reynolds numbers were investigated in two conical–cylindrical cavities. A cylindrical cavity was used to validate the numerical results by a comparison with experimental ones (Escudier, 1984). The mechanics involved in the vortex breakdown formation in such cavities was explored, and an explanation for the difference with respect to the cylindrical geometry was proposed, based on the flow physics.

2. Modeling

2.1. Mathematical Model

The conservation of mass and the Navier–Stokes equations for a general incompressible, Newtonian flow can be written, adopting the Einstein convention, respectively as:

$$\frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

By applying a filtering process to the above equations, it is possible to separate the larger scales of motion, which are related to the lowest frequencies, from the smallest scales, which are related to the higher frequencies. Eq. (2) may then be rewritten as:

$$\frac{\partial \rho \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho \bar{u}_i \bar{u}_j) = -\frac{\partial \bar{p}^*}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right] \tag{3}$$

In Eq. (3), the overbar denotes the filtered quantity, the asterisk denotes the modified pressure and μ_t is the turbulent viscosity. This term represents the energy dissipation present in the smallest scales of flow, which are not resolved in LES and must be modeled:

$$\mu_t = \rho(C\Delta^2)\bar{S} \tag{4}$$

Δ is the grid filter length and $\bar{S}$ is the filtered shear strain rate:

$$\bar{S} = \sqrt{\bar{S}_{ij}\bar{S}_{ij}}, \text{ and } \quad \bar{S}_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) \tag{5}$$

$\bar{p}^*$ is the modified pressure:

$$\bar{p}^* = \bar{p} + \frac{2}{3}\rho\kappa \tag{6}$$

κ is the subgrid turbulence kinetic energy.

Aiming at a precise turbulence modeling, the dynamic Smagorinsky model was employed (Germano et al., 1990). In this model, the eddy viscosity coefficient is locally calculated to reflect closely the state of the flow. This is done by sampling the smallest resolved scales and using this information to model the sub-grid scales. Two different filters are utilized:

- The grid filter, $\bar{\Delta}$, in which the grid dimensions are used to calculate its characteristic length.
- The test filter, $\tilde{\Delta}$, in which a multiple of grid size, normally two, is used to calculate the larger characteristic length.

A summary of the formulation used in this model for incompressible flows, with the modifications proposed by Lilly (1992), is presented below:

$$C = -\frac{1}{2}\frac{L_{ij}M_{ij}}{M_{ij}M_{ij}} \tag{7}$$

where:

$$M_{ij} = \tilde{\bar{\Delta}}^2|\tilde{\bar{S}}|\tilde{\bar{S}}_{ij} - \bar{\Delta}^2\widetilde{|\bar{S}|\bar{S}_{ij}} \tag{8}$$

and:

$$L_{ij} = \widetilde{\bar{u}_i\bar{u}_j} - \tilde{\bar{u}}_i\tilde{\bar{u}}_j \tag{9}$$

In the above equations the overbar denotes the grid filter process whereas the overtilde denotes the test filter process.

The model parameter produced by Eq. (7) is a rapidly varying function of spatial coordinates and time so the eddy viscosity may take large values of both signs. This can lead to numerical divergence, so a usual technique is clip the negative turbulence viscosity, as proposed by Ferziger and Peric (2002). For more information on this model the interested reader is referred to the original paper by Germano et al. (1990) and the work of Lilly Pereira and Sousa (1999).

The numerical solution of the conservation Eqs. (1) and (2) is accomplished by the computational code UNSCYFL3D (Unsteady Cyclone Flow 3D). This in-house tool is based on the finite volume method on unstructured three-dimensional grids, which enables the faithful representation of complex geometries. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used to couple the velocity and pressure fields. In all the simulations carried out in this work the three-time-level scheme was used for time-advancement and the second-order centered differencing scheme was employed for the advective and diffusive terms of the momentum equations. Further validation of this code can be found in Souza et al. (2012, 2014).

In most cases investigated in this work, it was not necessary to use turbulence modeling, since some of the flows were steady or did not display turbulence features. In some unsteady cases at higher Reynolds numbers, the mesh resolution was not enough to resolve all the structures of the flow, requiring a turbulence model. The LES approach was chosen to accurately model the influence of these unresolved scales. Nevertheless, the turbulence viscosity was seen to represent a small fraction of the effective viscosity, which suggests that the relevant scales for the problem have been captured.