THESIS FOR THE DEGREE OF LICENTIATE OF ENGINEERING
IN
THERMO AND FLUID DYNAMICS

Analytical and Numerical Investigations of Steady and Unsteady Turbulent Swirling Flow in Diffusers

WALTER GYLLENRAM

Department of Applied Mechanics

CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden, 2006

Analytical and Numerical Investigations of Steady and Unsteady Turbulent Swirling Flow in Diffusers

WALTER GYLLENRAM

THESIS FOR LICENTIATE OF ENGINEERING no. 2006:05
ISSN 1652-8565

Department of Applied Mechanics
Chalmers Tekniska Högskola
SE-412 96 Göteborg, Sweden
Phone +46-(0)31-7721400
Fax: +46-(0)31-180976

This document was typeset using LaTeX

Printed at Chalmers Reproservice, Göteborg, Sweden 2006

Analytical and Numerical Investigations of Steady and Unsteady Turbulent Swirling Flow in Diffusers

by

Walter Gyllenram

Division of Fluid Dynamics
Department of Applied Mechanics
Chalmers University of Technology
SE-412 96 Göteborg, Sweden
walter.gyllenram@chalmers.se

Abstract

Swirling flows are found in many technical applications, e.g. turbines, pumps, fans, compressors and combustors. The objective of the present work is to acquire an understanding of the physics of swirling flow in general and unsteady swirling flow in draft tubes of water turbines in particular. Analytical studies of axisymmetric swirling flow were carried out using a quasi-cylindrical approximation of the Navier-Stokes equations. It is shown that there are no quasi-cylindrical solutions to the Navier-Stokes equations for certain critical levels of swirl. It is argued that this property of the equations is related to a vortex breakdown phenomenon. In draft tubes of hydraulic power plants, the vortex breakdown phenomenon gives rise to a highly unsteady, oscillating pressure field that can endanger the machine. Numerical three-dimensional and unsteady simulations of swirling flows in simplified draft tube geometries were carried out to investigate this dynamic behaviour. The numerical methods include both LES (large eddy simulations) and RANS (Reynolds averaged Navier-Stokes) simulations. An LES is not an option for a full-scale draft tube simulation but provides valuable information for simplified cases. The RANS simulations in combination with the standard two-equation turbulence models are more applicable for industrial purposes but do not provide enough information about the unsteadiness of the flow. A dynamic filtering procedure of the turbulent length and time scales is generalised, employed and evaluated in order to remedy this shortcoming in the two-equation models. It is shown that the filtering procedure can yield solutions that contain more information about the flow dynamics as well as better time-averaged results compared to an ordinary RANS simulation.

Acknowledgements

This work was conducted at the Division of Fluid Dynamics, Department of Applied Mechanics, Chalmers University of Technology.

The research presented in this thesis has been part of the "Water turbine collaborative R&D program" which is financed by the Swedish Energy Agency, Hydro Power companies[1] (through Elforsk AB), GE Energy (Sweden) AB and Waplans Mekaniska Verkstad AB. Their financial support is gratefully acknowledged.

I would like to thank my supervisors, Dr. H. Nilsson and Prof. L. Davidson, for their support and guidance during this work. I would also like to thank my colleagues and all other employees at the Division of Fluid Dynamics for creating a stimulating working atmosphere.

---

[1] Vattenfall AB Vattenkraft, Fortum Generation AB, Sydkraft Vattenkraft AB, Skellefteå Kraft AB, Graninge Kraft AB, Jämtkraft AB, Sollefteåforsens AB, Karlstads Energi AB, Gävle Energi AB, Öresundskraft AB

Contents

Background

Approximately 50% of the electrical power produced in Sweden is generated by water turbines. Consequently, even a small improvement of their hydrodynamic design can contribute a great deal to the supply of energy. An overview of a hydraulic power plant is shown in Fig. 1 (top). The available (potential) energy is proportional to the static head, and the potential energy is converted to kinetic energy by letting gravity work on the water. The kinetic energy is in turn converted to electrical energy by a runner that is connected to a generator. Downstream of the runner in hydraulic power plants of the Francis or Kaplan type, the flow exits the water turbine through a draft tube. The purpose of the draft tube is to reduce the exit velocity with a minimum loss of energy. This will induce a relatively lower static pressure in the region just downstream of the runner. Hence, the pressure difference over the runner and thereby also the output power will increase. The efficiency of the draft tube is very important for a water turbine working at low head and it is determined by how well the flow responds to the geometry. The design of many draft tubes in use today is far from optimal and, when old hydro-power plants are refurbished, possibilities exist for modifying the draft tubes.

A swirling flow is created in the wicket gate just upstream of the runner, see Fig. 1 (bottom right). The runner rotates in the same direction as the flow, and the runner blades will counteract and neutralise the tangential velocity component if the turbine is working at its design point. Usually a small part of swirl is allowed to enter the draft tube in order to stabilise the flow and to prevent flow separation. However, at part load, a strongly swirling flow will exit the runner in the form of a large vortex. Because of the strong adverse pressure gradient in the draft tube, this vortex may break down into a precessing asymmetric shape and give rise to an oscillating pressure field. The pressure fluctuations in the draft tubes of Francis turbines can cause vibrations of a magnitude that endangers the supporting structure of the machine. While the pressure fluctuations in Kaplan turbines are usually not large enough to cause structural damage, the draft tube of

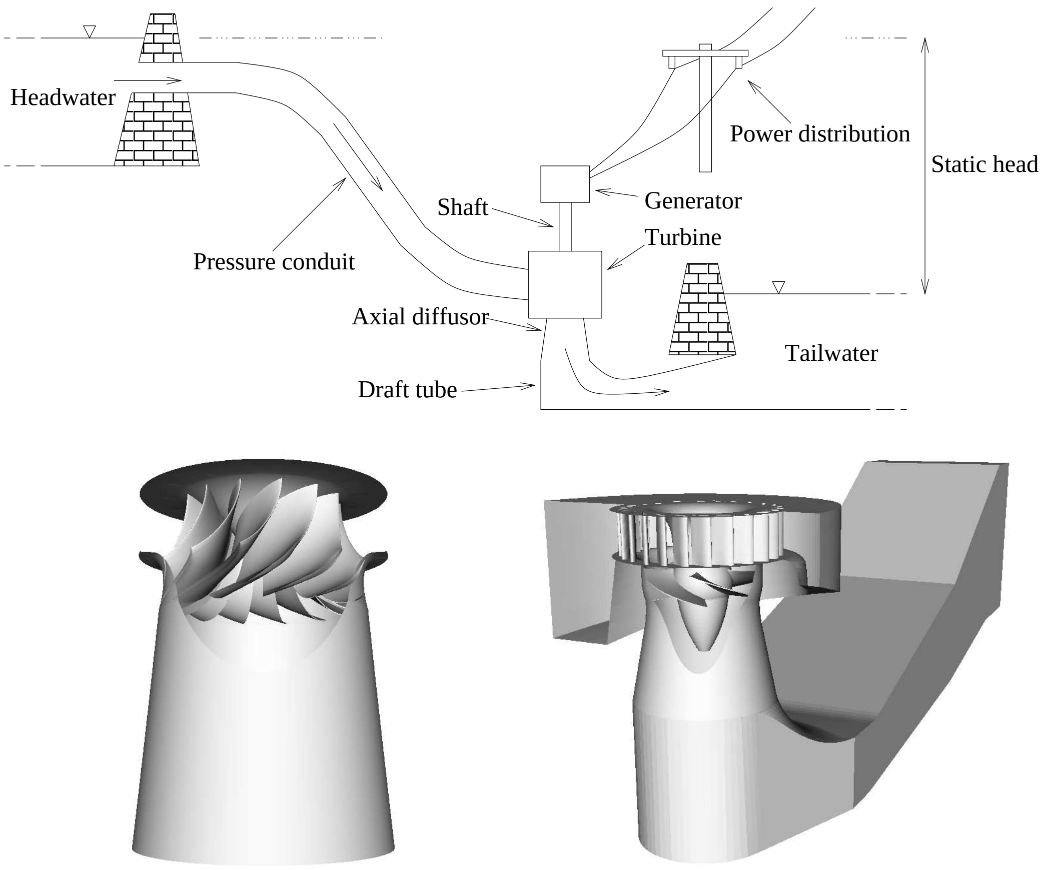

FIGURE 1: Top: An overview of a hydraulic power plant. Bottom left: The blades of a Francis runner and the entrance of a draft tube. Bottom right: The geometry of the Kaplan turbine at Hölleforsen including the spiral casing, the wicket gate, the runner and the draft tube.

a Kaplan turbine is very sensitive to flow separation. Flow separation can be triggered by pressure fluctuations and have a serious impact on the efficiency of the power plant. To improve the hydrodynamical design of the turbine it is necessary to be able to accurately predict these features of the flow.

The flow in draft tubes is complicated. It is a swirling, unsteady, partly separated flow at curved surfaces, flowing against an opposing pressure gradient. These features are all difficult to predict numerically. In this thesis, the flow in simplified draft tube geometries is considered in order to isolate and study the most important physics of real draft tube flow.

Summary of Papers

Paper I

On the failure of the quasi-cylindrical approximation and the connection to vortex breakdown in turbulent swirling flow

Paper I reports an analytical study of turbulent swirling flow in a straight (or slightly diverging) pipe. A quasi-cylindrical approximation of the time-averaged Navier-Stokes equations is derived and analysed. The result is a method that can be used to determine the radial distribution of the radial velocity component at any cross-section where the other velocity components are known. This is valuable because the radial velocity component of an internal swirling flow is usually very difficult to measure because of its low magnitude and other practical reasons such as visual access. The method can also be used to estimate whether a given swirling flow is near a critical level of swirl (vortex breakdown) because it is found that the quasi-cylindrical approximation is singular for certain swirl levels. The singularities give rise to unphysical solutions that violate the assumptions that were used in the derivation of the approximation. It is argued that this is connected to the phenomenon of vortex breakdown because, if the assumptions are not valid, the flow must either be highly unsteady or have large axial gradients.

Paper II

Large eddy simulation of turbulent swirling flow through a sudden expansion: Analysis of periodic unsteadiness caused by vortex breakdown

The analysis in paper I does not give information about the dynamic behaviour of swirling flow as it is based on the time-averaged Navier-Stokes equations. The need of detailed information about the dynamics and the properties of turbulence in swirling flow was the main motivation for the work described in paper II. A swirling flow through a sudden expansion was examined using LES. The flow resembles the flow in a draft tube at a water turbine that is working at part load. The results of the study agree well with the experimental data that were used for validation of the method. They also provide additional information about the oscillating vortex core and the properties of turbulence in swirling flow. A frequency analysis of the flow was carried out and it is shown that the rotational rate of the vortex core is not sensitive to the numerical order of the numerical discretization scheme.

Paper III

Very large eddy simulations of draft tube flow

The method used in paper II gives detailed information about the oscillating vortex core and about the more randomly distributed turbulence but it is not yet suitable for industrial (full-scale) applications. However, it will serve in the future as a benchmark case for less time-consuming turbulence modeling. Unsteady RANS simulations are better candidates for industrial purposes. The disadvantage of RANS simulations in combination with the standard two-equation turbulence models is that they do not provide enough information about the unsteadiness of the flow. In paper III it is argued that this shortcoming derives from the fact that the standard turbulence models are tuned for very simple test cases in which all unsteadiness of the flow can be regarded as turbulence. A dynamic filtering procedure of the turbulent length and time scales is generalised, employed and evaluated in order to remedy this shortcoming in the two-equation models. The filter sets a limit for the influence of the modeled turbulent length and time scales on the unsteady mean flow in order to reduce the damping effect of the turbulence model. The test case chosen in this work is a swirling flow through a conical diffuser. The filtering procedure is found to yield solutions that contain more information about the flow dynamics as well as better time-averaged results as compared to an ordinary RANS simulation.

Appended Papers

Paper I *On the failure of the quasi-cylindrical approximation and the connection to vortex breakdown in turbulent swirling flow*

Paper II *Large eddy simulation of turbulent swirling flow through a sudden expansion: Analysis of periodic unsteadiness caused by vortex breakdown*

Paper III *Very large eddy simulations of draft tube flow*

Additional Publications of Interest

Numerical investigations of swirling flow in a conical diffuser, W. Gyllenram and H. Nilsson, Proc. 22nd IAHR Symposium, Stockholm, Sweden, June 2004

Modeling of Swirling Flow in a Conical Diffuser, W. Gyllenram, Thesis for the degree of Master of Science, Chalmers University of Technology, Göteborg, Sweden, 2003