Jurnal Teknologi

Full Paper

NUMERICAL ANALYSIS OF SWIRL INTENSITY IN TURBULENT SWIRLING PIPE FLOWS

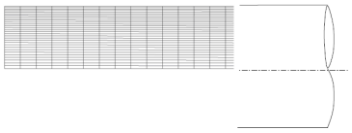
Khairul Fikri Tamrin[a], Nadeem Ahmed Sheikh[b], Bahbibi Rahmatullah[a*]

[a]Computing Department, Universiti Pendidikan Sultan Idris, 35900 Tanjung Malim, Perak, Malaysia
[b]Department of Mechanical Engineering, Muhammad Ali Jinnah University, Islamabad, Pakistan

Article history
Received
1 February 2015
Received in revised form
24 March 2015
Accepted
1 August 2015

*Corresponding author
bahbibi@fskik.upsi.edu.my

Graphical abstract



Abstract

Swirling flows are often observed in nature such as weather systems, cyclones and tornados. A number of applications use swirling nature of flow for enhanced mixing, heat transport and other transport phenomena. Naturally occurring swirls as well as induced swirls are often usually turbulent in nature. Understanding the flow physics of turbulent swirling flow is important for better understanding and control of processes involving swirling flows. With the increase of computational resources and advancements in turbulent flow modelling, it is now possible to simulate highly complex flow structures. Here turbulent swirling flow induced by guide vanes is studied using Computational Fluid Dynamics (CFD) simulations in a two-dimensional axisymmetric channel. The results for the variation of velocity components are compared with the work of an earlier research. The results are initially compared for the evaluation of best discretisation scheme. It was observed that the second-order and third-order schemes produced similar results. To simulate the turbulent flow two equations (*k-ε)* model and the five equations Reynolds Stress Model (RSM) are used. The comparison of both models with higher order discretisation schemes shows that the standard k-ε model is incapable of predicting the main features of the flow whilst RSM yields result close to the experimental data.

Keywords: Swirling flow, numerical simulation, turbulence model, standard *k-ε* model, RSM model

Abstrak

Aliran berpusar sering diperhatikan dalam alam semula jadi seperti dalam sistem cuaca, siklon dan puting beliung . Beberapa aplikasi menggunakan aliran berpusar telah digunakan untuk meningkatkan hasil percampuran, pengangkutan haba dan fenomena pengangkutan yang lain. Aliran berpusar juga sering terjadi dalam alam semula jadi. Memahami fizik aliran aliran berpusar adalah penting untuk memahami dan mengawal proses yang melibatkan aliran berpusar. Dengan peningkatan sumber pengiraan dan kemajuan dalam model aliran bergelora, ia kini mungkin untuk meniru struktur aliran yang sangat kompleks. Aliran berpusar yang dihasilkan oleh bilah pandu dikaji menggunakan Dinamik Bendalir Komputeran (CFD) simulasi dalam saluran simetri sepaksi dua dimensi. Keputusan bagi mengubah komponen halaju dibandingkan dengan kerja sebelum ini. Keputusan pada mulanya dibuat perbandingan untuk menilai skim pendiskretan terbaik. Diperhatikan bahawa skim tertib kedua dan ketiga menghasilkan keputusan yang sama. Untuk mensimulasikan aliran berpusar dua persamaan (k-ε) model dan lima persamaan Reynolds Tekanan Model (RSM) digunakan. Perbandingan kedua-dua model dengan skim pendiskretan yang lebih tinggi menunjukkan bahawa model k-ε standard tidak mampu meramalkan ciri-ciri utama aliran manakala keputusan RSM lebih dekat dengan data eksperimen.

Kata kunci: Aliran berpusar, simulasi numerical, model turbulen, model asas *k-ε*, model RSM