NUMERICAL RESEARCH ON THE PREDICTION OF THE 3-D COMPRESSIBLE FLOW IN TURBOMACHINERY CHANNELS TO ESTIMATE THE LOSSES, TO INCREASE THE FLOW EFFICIENCY AND TO MINIMIZE THE HERMFUL IMPACT ON ENVIROMENT

Fact Sheet

Project Information

CFD TURBOMACHINERY

Grant agreement ID: 505176

Project closed

Start date
20 January 2004

End date
19 January 2005

Funded under
Human resources and Mobility in the specific programme for research, technological development and demonstration "Structuring the European Research Area" under the Sixth Framework Programme 2002-2006

Total cost
€ 13 750,00

EU contribution
€ 14 175,00

Coordinated by
POLITECHNIKA SLASKA
Poland

Objective

Until now the significant part of the world electrical and heat energy is produced by means of steam or gas turbine, which are the main part of the power and on the reduction of the toxins emission. The numerical calculations of the flow through the turbines channels (stages) are very complicated and time consuming, therefore require an efficient numerical code. In these types of flows we have to take into consideration an additional physical phenomena, like condensation in steam turbines or chemical reactions in gas turbines. The first purpose of the research on an improvement of the flow efficiency in turbo machinery is to apply and develop a numerical code elaborated during the two years of the initial Marie Curie Individual Fellowship. This code was related to estimate the losses in the flow of the steam in the stages of LP (low pressure) steam turbine. In this case the numerical simulation was based on the time dependent 3-D Reynolds averaged Nervier-Stokes equations, which were coupled with a two-equation turbulence model (k-o SST model) and additional mass conservation equations for the liquid phase (two for homogeneous and one for heterogeneous condensation). The set of governing equations was closed by a real gas equation of state, so called 'local' gas equation of state. The condensation phenomena were modelled based on the classical nucleation theory of Volume, Freckle and Zeldovich. The Gyarmathy's droplet growth equation was used. The system of governing equations was discredited on a multi-block structured grid using the finite-volume method and integrated in time using an explicit Runge-Kutta method. This numerical code will be developed to the calculations of the flow in gas turbines. The applied 'local' real gas equation of state will be suitable to the

calculations of the high temperature gas flow. The second purpose is to create of an internet page devoted to the elaborated CFD software.

Fields of science (EuroSciVoc)

natural sciences > computer and information sciences > **software**

natural sciences > computer and information sciences > **internet**

natural sciences > physical sciences > classical mechanics > fluid mechanics > **fluid dynamics**

natural sciences > **chemical sciences**

Keywords

CFD SOFTWARE | INTERNET PAGE | LOSSES | STEAM TURBINE | cfd software | efficiency | gas turbine | internet page | losses | steam turbine

Programme(s)

FP6-MOBILITY - Human resources and Mobility in the specific programme for research, technological development and demonstration "Structuring the European Research Area" under the Sixth Framework Programme 2002-2006

Topic(s)

MOBILITY-4.1 - Marie Curie European Reintegration Grants (ERG)

Call for proposal

FP6-2002-MOBILITY-11
See other projects for this call

Funding Scheme

EIF - Marie Curie actions-Intra-European Fellowships

Coordinator

POLITECHNIKA SLASKA

EU contribution

No data

Total cost

No data

Address

Ul. Akademicka 2
GLIWICE
Poland

Last update: 10 March 2023

Permalink: https://cordis.europa.eu/project/id/505176