NAFEMS UK Regional Conference 2018 - Abstract Submission

Submission Date	2018-02-08 11:47:45
Name	Mr. Mi chael Clapp
Job Title	Technical Director
Company	80/20 Engineering Ltd
Please identify the event for which your submitting?	NAFEMS UK Conference 2018
Will you be the presenting author?	Yes
Presentation Title	NAFEMS Publication - A Designers' Guide to CFD
Relevant Themes / Keywords	CFD, Democratization, Application Templates, Best Practices within the CAE Process, Verification and Validation

Abstract (plain text)

Introduction

This abstract submission is in reference to the forthcoming NAFEMS publication of a book entitled a "Designers Guide to CFD" to be published shortly. An overview of the book will be presented based on the following preface.

Introduction.

The author was convinced of the need for a publication that introduces today's multi-tasking engineers to the new breed of CFD tools that are developed for "design engineers". These software packages promise to spread the use of Computational Fluid Dynamics (CFD) techniques to a far wider audience by making a step change in the ease of use. This may have been achieved in many cases but are there still issues that these engineers need to be aware of to get the best out of these tools and avoid computer aided disasters?

The main purpose of the proposed book is to address these issues and provide guidance to "design engineers", and their managers, as to how to make best use of such CFD software. The first few chapters will do this by discussing the technology available and providing examples of how it has been used successfully and unsuccessfully. Then the basic simulation lifecycle will be worked through to highlight what the software user needs to be aware of at each stage of the process. Common mistakes will be highlighted at each stage. In the final sections the use of the results to drive design changes will be highlighted as well as why to make use of more advanced expertise in the field of CFD simulation.

Scope.

In the authors opinion there are currently three types of software that "Design Engineers" may encounter. Firstly there are the high end, "traditional" packages which attempt to solve most known problems that could be encompassed by the term Computational Fluid Mechanics (e.g. FLUENT, STAR, CFX, FLOW-3D, Numeca). They tend to do this at the expense of becoming complex to use, mainly due to the sheer number of user options. Secondly there are the newer breed of design focused CFD packages that integrate closely with mainstream 3D CAD packages and while trying to be general purpose tend to focus on a subset of the physics encompassed by the first group. These are sometimes special versions of the software developed by the "traditional" CFD vendors (e.g. Flowizard from FLUENT, STAR CAD Series) and sometimes newcomers to the market (e.g. FloEFD, Autodesk CFD (formerly CFdesign)). A third group are those devised for very specific applications. These range between the group that target the electronics industry (e.g. Flotherm, Icepak) to very specialist packages that are designed for a single application (e.g. es-ice, shellflo).

Although mentioning all these groups of software this book will concentrate on the user of the second group of packages, because this is where users are most likely to gain from an additional publication from NAFEMS. The use of the first group of packages is already covered by existing publications, while the third group contains such a wide range of diverse applications and procedures that it would be impossible to make sense of them in a single publication.

For ten of the last fifteen years he has worked for one of the first companies to advocate spreading the use of Computational Fluid Dynamics to a wider audience of engineers. During this time he has supported many users and companies through the process of implementing such software and has therefore encountered most of the benefits and pitfalls of using such software. Through his new connection with 8020 Engineering, he will also be able to call on many other experiences from the world of CFD.

abstract id

UK18-52