

PRACE Autumn School 2013 - Industry Oriented HPC Simulations

Report of Contributions

Contribution ID: 0

Type: **Presentation**

RBF morph theory and applications case-history

Friday, 27 September 2013 10:00 (45 minutes)

An overview about mesh morphing and its benefits will be given with a specific focus on Radial Basis Functions (RBF) methods and the industrial tool RBF Morph, currently available as an add-on for the CFD solver ANSYS Fluent.

Standard applications of mesh morphing will be first explained; the tool is here used as an effective way to make the original CFD model parametric with respect to the shape. Details on how such parametric models can be introduced in typical industrial workflow (shape or set-up optimization) will be given including CAD connection (STL targets and back to CAD) and flow sculpting (using adjoint solver of Fluent).

Advanced applications will be covered as well including: fluid structure interaction using 2-ways FSI and modal superposition, ice/snow accretion modeling, transient morphing with desired time histories.

Presenter: Prof. BIANCOLINI, Marco Evangelos ("Tor Vergata" University of Rome)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 1

Type: **Industry case study**

High Performance Computing of gas turbine flows: current and future trends

Friday, 27 September 2013 09:00 (1 hour)

Nicolas Gourdain presents elsA for aerodynamics used by many by industrial partners such as Airbus, SAFRAN, etc. and AVBP for combustion, used by SAFRAN, IFP, etc.

The use of Computational Fluid Dynamics (CFD) is mandatory today both for scientific investigation of complex flows and industrial design of gas turbines and aircraft. Further improvement and optimization of such systems currently present one of the most formidable challenges in modern engineering research due to the very stringent requirements for efficiency, pollutant emissions, reliability and safety. Efficient numerical tools coupled with high performance computers, have become a key element of the design process in the fields of energy supply and transportation. However flow phenomena that occur in complex systems such as gas turbines and aircrafts are still not understood mainly because of the models that are needed. In fact, most CFD predictions as found today in industry focus on a reduced or simplified version of the real system (such as a periodic sector) and are usually solved with a steady-state assumption. This course discusses how to overcome such barriers and how this challenges can be addressed by developing flow solvers running on high-end computing platforms, using concurrently thousands of computing cores. Parallel strategies used by modern flow solvers are presented with particular emphases on mesh-partitioning, load balancing and communication.

Presenter: Dr GOURDAIN, Nicolas (CERFACS)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 2

Type: **Hands-on training**

RBF morph hands-on

Friday, 27 September 2013 11:00 (1h 30m)

The first session will cover basic exercises to set-up shape modifications on simple geometries (a cube immersed in a wind tunnel and a straight pipe). A step by step set-up will be demonstrated to gain confidence with the GUI of RBF Morph (set-up stage). An advanced session addressed to industrial meshes and complex set-up will follow. Pre computed set-up will be available so that the students can deepen their knowledge about advanced feature of RBF Morph for check trouble shooting the set-up (preview, mesh quality).

Presenter: Prof. BIANCOLINI, Marco Evangelos ("Tor Vergata" University of Rome)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 3

Type: **Hands-on training**

Multi-block structured code (elsA) and an unstructured code (AVBP) hands-on

Friday, 27 September 2013 13:30 (5 hours)

The aim will be to test some of the requirements presented in the course (such as mesh-partitioning and load balancing) to achieve high-scalability on massively parallel computers (maybe also on GPUs).

Two examples are used to illustrate these concepts: a multi-block structured code (elsA) and an unstructured code (AVBP). Parallel computing strategies used with both flow solvers are detailed and compared. This comparison indicates that mesh-partitioning and load balancing are more straightforward with unstructured grids than with multi-block structured meshes. However, the mesh-partitioning stage can be challenging for unstructured grids, mainly due to memory limitations of the newly developed massively parallel architectures. Finally, detailed investigations show that the impact of mesh-partitioning on the numerical CFD solutions, due to rounding errors and block splitting, may be of importance and should be accurately addressed before qualifying massively parallel CFD tools for a routine industrial use.

Presenters: Dr GAZAIX, Michel (ONERA); Dr GOURDAIN, Nicolas (CERFACS)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 4

Type: **Presentation**

Opening and Welcome address

Monday, 23 September 2013 13:00 (20 minutes)

Welcome from University of Ljubljana as prace PRACE coordinator for Slovenia.
Industry challenges tackled with HPC and research targeted for products with simulations and optimizations that give concrete answers to many open questions.

Presenter: Prof. DUHOVNIK, Jožef (University of Ljubljana, PRACE coordinator for Slovenia)

Contribution ID: 5

Type: **Presentation**

HPC – the Perspective of a CFD Practitioner

Monday, 23 September 2013 13:20 (40 minutes)

The lecture will address the trends of the relation between Computational Fluid Dynamics (CFD), which is a subsection of Simulation-Based Engineering Science, and High Performance Computing (HPC) applications.

See attached extended abstract.

Presenter: Prof. SOUSA, Antonio C.M. (University of Aveiro and New Brunswick)

Contribution ID: 6

Type: **Presentation**

Discover your design quicker as before with HPC

Monday, 23 September 2013 14:00 (1 hour)

An overview over current technology, recent benchmark results from a user perspective, as well as new ways of calculating large models and extensive design variations will be presented.

- High Performance Computing (HPC). approaches overview
- Success stories from structural mechanics and electromagnetics
- Overview of the solvers across all physics
- Ongoing HPC initiatives

Presenter: DOPF, Alexander (CADFEM)

Contribution ID: 7

Type: **Industry case study**

Robust Design Optimization – from the idea to the optimized product

Tuesday, 24 September 2013 09:00 (2h 30m)

Introduction

Motivation for parametric variations

Parametric workflows

Introduction into optiSLang

Systematic variation using optiSLang

a) Sensitivity Analysis and Metamodeling

b) Tutorial: Sensitivity analysis of a notch

Presenter: VEIZ, Andreas (CADFEM)

Session Classification: Structural Mechanics

Contribution ID: 9

Type: **Industry case study**

Examples of Solving Industrial Cases with Fluent

Thursday, 26 September 2013 09:00 (1h 15m)

Several industrial cases will be presented, executed by SimTec or its customers with the aid of ANSYS Fluent general-purpose 3D CFD package. The examples are taken from industrial applications in SE Europe countries but emphasis will be given on the Slovenian cases. The applications cover the industry sectors of marine, constructions (HVAC), chemical, pharmaceutical, hydraulic, metal, power generation, amongst others. Each case presentation includes: (a) modelling targets, (b) description of the physical model, (c) results, (d) modelling conclusions.

Presenter: Dr SOFIALIDIS, Dimitrios (SimTec)

Track Classification: Computational Fluid Dynamics

Contribution ID: 11

Type: **Hands-on training**

Second Fluent workshop (hands on)

Thursday, 26 September 2013 14:30 (30 minutes)

Each participant will choose 1 among 3 alternative workshops. These workshops will be formed after the list of participants is closed, in order to cover as much as possible application fields declared by the participants.

Presenter: Dr SOFIALIDIS, Dimitrios (SimTec)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 12

Type: **Industry case study**

BIO-CFD Case-history: Introduction on CFD in hemodynamics

Wednesday, 25 September 2013 11:00 (1 hour)

Wide range of haemodynamics CFD application is presented (medical implantable devices, blood filters, multi-scale-models, non-Newtonian rheology,...) to perform in silico health-care research and medical device design.

The main expected benefits of the lecture+tutorial for attendants are:

- enlarge their knowledge on state of art methods and algorithms;
- apply best practices on state-of-the-art software deployment;
- experience technical tips-and-tricks throughout the different phases of the tutorial;
- learn from case history and practical applications told by CFD expert.

Presenter: Dr PONZINI, Raffaele (CINECA)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 13

Type: **Hands-on training**

BIO-CFD Tutorial

Wednesday, 25 September 2013 13:30 (3 hours)

- a) Basic modeling using Fluent for carotid bifurcation (GUI in fluent for case setup from mesh import to first steady state, Newtonian simulation running; TUI and case journaling in Fluent for batch execution; post-processing in Fluent)
- b) Intermediate modeling using Fluent for carotid bifurcation haemodynamics (unsteady BC in Fluent; non-Newtonian custom models; multi-scale models coupling 3D and 0D in Fluent; customized post-processing)

Presenter: Dr PONZINI, Raffaele (CINECA)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 14

Type: **Industry case study**

Sailing Yachts CFD

Wednesday, 25 September 2013 10:15 (45 minutes)

Case-history on the balance between computational costs & insight benefits obtained by using advanced CFD modeling (Delayed Detached Eddy Simulations vs Reynolds Averaged NS) in sailing yachts aerodynamics

Presenter: Dr PONZINI, Raffaele (CINECA)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 15

Type: **Hands-on training**

EM Example 2: WLAN-Antenna Optimization with HFSS

Wednesday, 25 September 2013 13:00 (1h 30m)

Problem Description

Analysis Setup and Basic Simulation Results

Design Optimization

Presenter: Dr RÖMELSBERGER, Christian (CADFEM)

Session Classification: Multiphysics: Electromechanical and Mechatronic Systems

Track Classification: Multiphysics: Electromechanical and Mechatronic Systems

Contribution ID: 16

Type: **Hands-on training**

EM Example 3: Starter-Generator Design with Maxwell 3D/ Mechanical

Wednesday, 25 September 2013 14:50 (1h 40m)

Problem Description
Analysis Setup and Basic Simulation Results
Efficiency Calculation and Loss Assessment
Temperature Evaluation

Presenter: OTTO, Jens (CADFEM)

Session Classification: Multiphysics: Electromechanical and Mechatronic Systems

Track Classification: Multiphysics: Electromechanical and Mechatronic Systems

Contribution ID: 17

Type: **Industry case study**

EM: Summary and discussion

Wednesday, 25 September 2013 16:30 (30 minutes)

Presenters: Dr RÖMELBERGER, Christian (CADFEM); OTTO, Jens (CADFEM)

Session Classification: Multiphysics: Electromechanical and Mechatronic Systems

Track Classification: Multiphysics: Electromechanical and Mechatronic Systems

Contribution ID: **18**

Type: **Presentation**

EM Simulation Overview (tools and numerical methods)

Wednesday, 25 September 2013 09:00 (1 hour)

Presenters: Dr RÖMELSBERGER, Christian (CADFEM); OTTO, Jens (CADFEM)

Session Classification: Multiphysics: Electromechanical and Mechatronic Systems

Track Classification: Multiphysics: Electromechanical and Mechatronic Systems

Contribution ID: 19

Type: **Hands-on training**

EM Example 1: Position Sensor Design (Hall based) with Maxwell 3D

Wednesday, 25 September 2013 10:15 (1h 15m)

Presenter: OTTO, Jens (CADFEM)

Session Classification: Multiphysics: Electromechanical and Mechatronic Systems

Track Classification: Multiphysics: Electromechanical and Mechatronic Systems

Contribution ID: 20

Type: **Hands-on training**

Sensitivity analysis of a notch (cont.)

Tuesday, 24 September 2013 13:00 (4 hours)

Getting started in design improvement:

a) How to improve a design

b) Tutorial: Improve the notch

Typical Questions in postprocessing results

Outlook: Efficient performance of extensive design variation

Presenter: VEIZ, Andreas (CADFEM)

Session Classification: Structural Mechanics

Track Classification: Structural Mechanics

Contribution ID: 21

Type: **Hands-on training**

Hands On Training with HPC

Monday, 23 September 2013 15:00 (1h 30m)

Prerequisites: MPI – HP or Intel

How to set up HPC: solver handler, number of CPUs, and DMP

Handling of large models: DM Prep in v14.5; mechanical filters, tags and named selections

Presenter: Dr HÖSSL, Bernhard (CADFEM)

Session Classification: Structural Mechanics

Track Classification: Structural Mechanics

Contribution ID: 22

Type: **Hands-on training**

Hands On Training with HPC (cont.)

Monday, 23 September 2013 16:45 (2h 15m)

Mesh controls for large models: curvature and proximity

How to cut analysis time: Proper contact and analysis settings

Evaluation of results: results tracker; post command snippets; max tag adjust to visible

Presenter: Dr HÖSSL, Bernhard (CADFEM)

Session Classification: Structural Mechanics

Track Classification: Structural Mechanics

Contribution ID: 23

Type: **Hands-on training**

HPC architectures and MPI, OpenMP programming hands-on

Monday, 23 September 2013 09:00 (2h 30m)

HPC architecture overview with examples of Parallel Programming with

- distributed MPI,
- shared OpenMP, and
- hybrid MPI+OpenMP, memory models.

Presenter: Dr KOS, Leon (University of Ljubljana)

Session Classification: HPC and parallel programming

Contribution ID: 24

Type: **Presentation**

Boundary Conditions and Solver Settings (Convergence & Accuracy)

Thursday, 26 September 2013 13:30 (30 minutes)

The presentation will cover and explain the basic steps of a CFD analysis. Important topics such as Boundary Conditions, Convergence of the solution and increasing the Accuracy of the results will be discussed in depth.

Presenter: Dr SOFIALIDIS, Dimitrios (SimTec)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 25

Type: **Presentation**

Turbulence Modelling, Heat Transfer & Transient Calculations

Thursday, 26 September 2013 14:00 (30 minutes)

As most flow problems exhibit turbulent nature and most engineering applications involve heat transfer, these two subjects will be covered. In addition, as many fluid flows are not steady-state phenomena, transient simulations and solution and relative issues like time step size will be discussed.

Presenter: Dr SOFIALIDIS, Dimitrios (SimTec)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 26

Type: **Hands-on training**

High-Performance Computing with Fluent (hands-on)

Thursday, 26 September 2013 15:00 (3 hours)

An industrial CFD case will be executed as a workshop. The setup, solution and postprocessing will span through the first half of the session. The second half will be spent with experimentation of alternative solutions (e.g. changing BCs, material properties, mesh size, etc.) and discussion regarding best practices, error reduction and accuracy, tips and tricks, parallel computing efficiency, etc.

Presenter: Dr SOFIALIDIS, Dimitrios (SimTec)

Track Classification: Computational Fluid Dynamics

Contribution ID: 27

Type: **Presentation**

Introduction to CFD

Thursday, 26 September 2013 10:30 (45 minutes)

Presenter: Dr SOFIALIDIS, Dimitrios (SimTec)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics

Contribution ID: 28

Type: **Hands-on training**

Fluent Workshop 1: Fluid Flow & Heat Transfer in a Mixing Tee

Thursday, 26 September 2013 11:15 (1h 15m)

Presenter: Dr SOFIALIDIS, Dimitrios (SimTec)

Session Classification: Computational Fluid Dynamics

Track Classification: Computational Fluid Dynamics