PRACE Autumn School 2013
Industry Oriented HPC Simulations

23 - 27 September 2013, University of Ljubljana, Slovenia
Faculty of Mechanical Engineering, Aškerčeva 6, Ljubljana

Resolving Real Industrial Engineering Applications with HPC by ANSYS, optiSLang, AVBP, elsA, RBF morph, etc.

Monday, 23 September 2013

08:30 - 13:00 Registration, Coffee & Poster hanging (Nearby lecture hall IV/2)

Attendees are required to show up and register themselves at the registration desk before the school starts in the afternoon and hang their poster before the introductory poster session at noon.

09:00 - 11:30 HPC and parallel programming: Optional training

This pre-training is optional and will be held if enough participants will register for it. It covers HPC architecture, schedulers and OpenMP and MPI hands on training.

Location: University of Ljubljana, Slovenia (Computing rooms III/1, N17 and II/5)

09:00 **HPC architectures and MPI, OpenMP programming hands-on** *2h30'*

HPC architecture overview with examples of
Parallel Programming with
- distributed MPI,
- shared OpenMP, and
- hybrid MPI+OpenMP,
memory models.

Speaker: Dr. Leon Kos (University of Ljubljana)

12:00 - 13:00 Posters (Hall at the top of the building near the restaurant.)

Attendees communicate their work and meet each other by presenting (past) posters.

13:00 - 13:20 Opening and Welcome address *20'* (Lecture hall IV/2)

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.

Speaker: Prof. Jožef Duhovnik (University of Ljubljana, PRACE coordinator for Slovenia)

13:20 - 14:00 HPC – the Perspective of a CFD Practitioner *40'* (Lecture hall IV/2)

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.

Speaker: Prof. Antonio C.M. Sousa (University of Aveiro and New Brunswick)

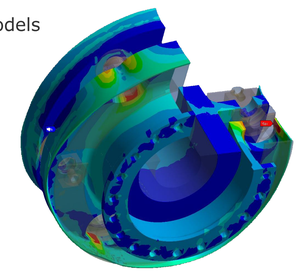
Material: Paper

14:00 - 15:00 Discover your design quicker as before with ANSYS HPC *1h0'* (Lecture hall IV/2)

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.
- The ANSYS Commitment to High Performance Computing (HPC).
- Customer Success Stories
- ANSYS HPC Products and Solver Performance across all physics
- Ongoing HPC initiatives

Speaker: Alexander Dopf (CADFEM)

15:00 - 19:00 Structural Mechanics: Best practices for efficient HPC performance with large models



Location: Computing room II/5

15:00 **Hands On Training with ANSYS HPC** *1h30'*

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

Speaker: Dr. Bernhard Hössl (CADFEM)

16:30 **Break** *15'*

16:45 **Hands On Training with ANSYS HPC (cont.)** *2h15'* (Computing room II/5)

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

Speaker: Dr. Bernhard Hössl (CADFEM)

Tuesday, 24 September 2013

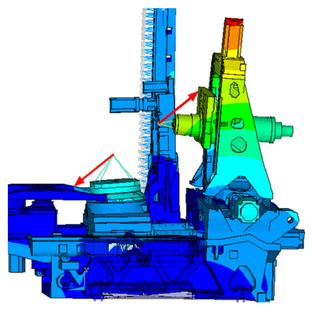
09:00 - 17:00 Structural Mechanics: Robust Design Optimization

Location: University of Ljubljana, Slovenia (Computing room II/5)

09:00 **Robust Design Optimization – from the idea to the optimized product** *2h30'*

Introduction
Motivation for parametric variations
Parametric workflow in ANSYS
Introduction into optiSLang for ANSYS
Systematic variation using optiSLang for ANSYS
a) Sensitivity Analysis and Metamodeling
b) Tutorial: Sensitivity analysis of a notch

Speaker: Andreas Veiz (CADFEM)



11:30 **Lunch break** *1h30'* (Restaurant at the top of the building)

13:00 **Sensitivity analysis of a notch (cont.)** *4h0'*

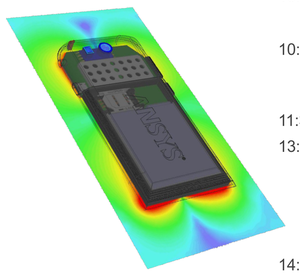
Getting started in design improvement:
a) How to improve a design
b) Tutorial: Improve the notch
Typical Questions in postprocessing results
Outlook: Efficient performance of extensive design variation

Speaker: Andreas Veiz (CADFEM)

Wednesday, 25 September 2013

09:00 - 17:00 Multiphysics: Electromechanical and Mechatronic Systems: Hands on with EM

ANSYS Maxwell and HFSS training



09:00 **EM Simulation Overview (tools and numerical methods)** *1h0'* (Lecture hall IV/2)

Speaker: Jens Otto (CADFEM)

10:00 **Coffe break** *15'* (Lecture room IV/2 and computing rooms II/5, III/1, N17)

Attendees reposition into Computing room for EM tutorials or continue with CFD lectures until lunch.

10:15 **EM Example 1: Position Sensor Design (Hall based) with Maxwell 3D** *1h15'* (Computing room III/1)

Speaker: Jens Otto (CADFEM)

11:30 **Lunch break** *1h30'* (Restaurant at the top of the building)

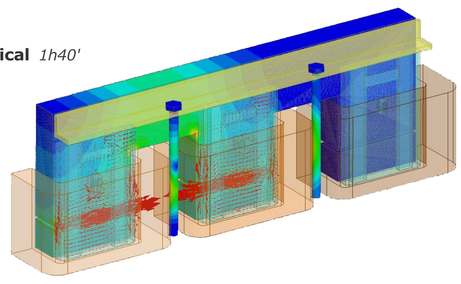
13:00 **EM Example 2: WLAN-Antenna Optimization with HFSS** *1h30'* (Computing room III/1)

Problem Description
Analysis Setup and Basic Simulation Results
Design Optimization

Speaker: Dr. Christian Römelsberger(CADFEM)

14:30 **Break** *20'* (Lecture room IV/2 and computing rooms II/5, III/1, N17)

14:50 **EM Example 3: Starter-Generator Design with Maxwell 3D/ Mechanical** *1h40'* (Computing room III/1)



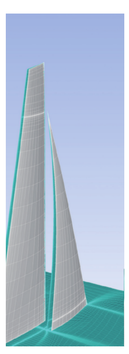
Problem Description
Analysis Setup and Basic Simulation Results
Efficency Calculation and Loss Assessment
Temperature Evaluation

Speaker: Jens Otto (CADFEM)

16:30 **EM: Summary and discussion** *30'*

Speakers: Dr. Christian Römelsberger(CADFEM), Jens Otto (CADFEM)

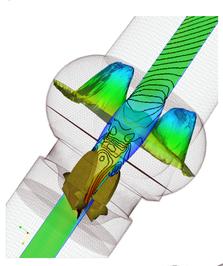
10:15 - 17:00 Computational Fluid Dynamics



10:15 **Sailing Yachts CFD** *45'* (Lecture hall IV/2)

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

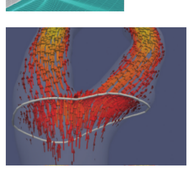
Speaker: Dr. Raffaele Ponzini (CINECA)



11:00 **BIO-CFD Case-history** *1h0'*

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.

Speaker: Dr. Raffaele Ponzini (CINECA)

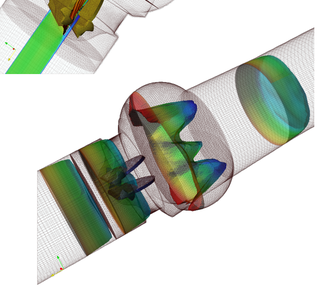


12:00 **Lunch break** *1h30'* (Restaurant at the top of the building)

13:30 **BIO-CFD Tutorial** *3h0'* (Computing room N17)

a) Basic modeling using ANSYS 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)

Speaker: Dr. Raffaele Ponzini (CINECA)



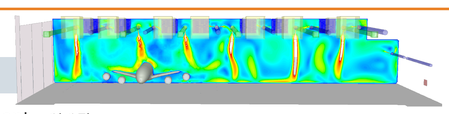
The PRACE Autumn School communicates on and discusses issues and perspectives of HPC targeting industrial applications arising from evaluation performance and/or design of products, e.g. equipment and processes, with a particular emphasis on the automotive, aerospace, and energy fields. The school focuses on resolving real industrial engineering applications with HPC by ANSYS, optiSLang, AVBP, elsA, RBF Morph software in classes given by 12 lecturers (from CADFEM, SimTeC, CERFACS, ONERA, CINECA, Universities of: Rome "Tor Vergata", Aveiro and New Brunswick) from the fields of Structural Mechanics, Computational Fluid Dynamics (CFD and BIO-CFD) and Electromagnetics (EM).

Prerequisites
Applicants are expected to have some experience with the topic of interest. School attendees will be selected based on applications submitted through the registration form, taking into account the order of registrations. Applicants will be informed about their acceptance not later than two weeks after their registration. The number of attendees is limited to about 40–50. All attendees are encouraged to bring a poster on their work related to the topics of the school. The school is offered free of charge to industrial users, researchers and academics residing in PRACE member states and eligible countries. It is the responsibility of the attendees to arrange and cover travel and accommodation.

Short URL: **http://events.prace-ri.eu/event/as13**

Contact prace@hpc.fs.uni-lj.si

The Partnership for Advanced Computing in Europe (PRACE) is an international non-profit association with its seat in Brussels. The PRACE Research Infrastructure provides a persistent world-class high performance computing service for scientists and researchers from academia and industry in Europe. The computer systems and their operations accessible through PRACE are provided by 4 PRACE members (BSC representing Spain, CINECA representing Italy, GCS representing Germany and GENCI representing France). The Implementation Phase of PRACE receives funding from the EU's Seventh Framework Programme (FP7/2007-2013) under grant agreements RI-261557, RI-283493 and RI-312763. For more information, see www.prace-ri.eu.



Thursday, 26 September 2013

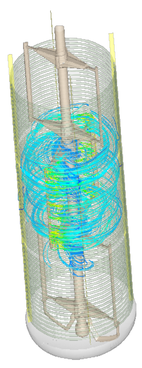
09:00 - 10:15 Examples of Solving Industrial Cases with ANSYS Fluent *1h15'*

Several industrial cases will be presented, executed by SimTec or its ANSYS software 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.

Speaker: Dr. Dimitrios Sofialidis (SimTec)

10:15 - 10:30 Coffee break

10:30 - 15:00 Computational Fluid Dynamics: Express Introductory Training in ANSYS Fluent



Location: Computing room II/5

10:30 **Introduction to CFD** *45'*

Speaker: Dr. Dimitrios Sofialidis (SimTec)

11:15 **Fluent Workshop 1: Fluid Flow & Heat Transfer in a Mixing Tee** *1h15'*

Speaker: Dr. Dimitrios Sofialidis (SimTec)

12:30 **Lunch break** *1h0'* (Restaurant on top of the building)

13:30 **Boundary Conditions and Solver Settings (Convergence & Accuracy)** *30'*

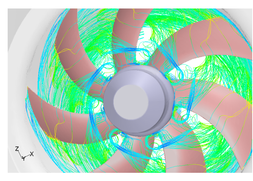
The presentation will cover and explain the basic steps of a CFD analysis. Important topics such as Boundary Conditions, Convergence of the solution and increasing the Accuracy of the results will be discussed in depth.

Speaker: Dr. Dimitrios Sofialidis (SimTec)

14:00 **Turbulence Modelling, Heat Transfer & Transient Calculations** *30'*

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.

Speaker: Dr. Dimitrios Sofialidis (SimTec)



14:30 **Second Fluent workshop (hands on)** *30'*

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.

Speaker: Dr. Dimitrios Sofialidis (SimTec)

15:00 - 18:00 High-Performance Computing with ANSYS Fluent (hands-on) *3h0'*

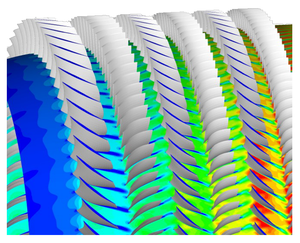
An industrial CFD case will be executed as a workshop. The setup, solution and postprocessing will span through the first half of the session. The second half will be spent with experimentation of alternative solutions (e.g. changing BCs, material properties, mesh size, etc.) and discussion regarding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

Speaker: Dr. Dimitrios Sofialidis (SimTec)

Friday, 27 September 2013

09:00 - 17:00 Computational Fluid Dynamics: Custom CFD codes and plugins

Location: (Computing rooms III/1, N17)



09:00 **High Performance Computing of gas turbine flows: current and future trends** *1h30'* (Lecture hall IV/2)

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.

Speaker: Dr. Nicolas Gourdain (CERFACS)

10:30 **Coffe break** *15'*

10:45 **RBF morph theory and applications case-history** *1h45'* (Lecture room VI/2)

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 makes 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.

Speaker: Prof. Marco Evangelos Biancolini ("Tor Vergata" University of Rome)

12:30 **Lunch break** *1h0'*

13:30 **RBF morph hands-on** *2h0'*

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).

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

13:30 **Multi-block structured code (elsA) and an unstructured code (AVBP) hands-on** *3h30'* (Computing room N17)

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.

Speakers: Dr. Nicolas Gourdain (CERFACS), Dr. Michel Gazaix (ONERA)

http://events.prace-ri.eu/event/as13

University of Ljubljana
Faculty of Mechanical Engineering

e-infrastructure

CAPACITIES

PRACE Projects receive EC funding under grants RI-261557,RI-283493 and RI-312763.