



EuroCC workshop

CFD on HPC: OpenFoam

Pavel Tomšič, Faculty for mechanical engineering, University of Ljubljana

Date: 13-06-2022

About EuroCC project



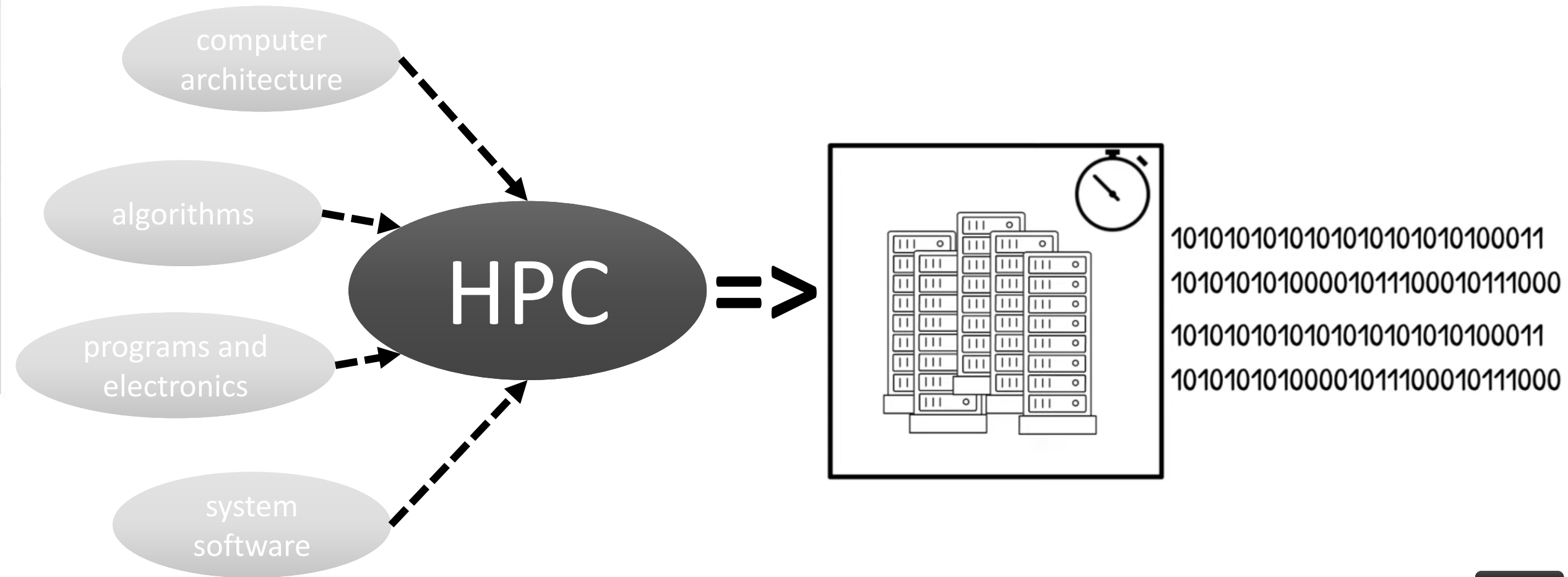
- National Competence Centres for HPC in 33 countries
- to fill the gaps in HPC knowledge in different areas:
 - Use in industry (SME),
 - School education,
 - Higher education institutions and
 - Enthusiastic individuals.



EuroHPC
Joint Undertaking



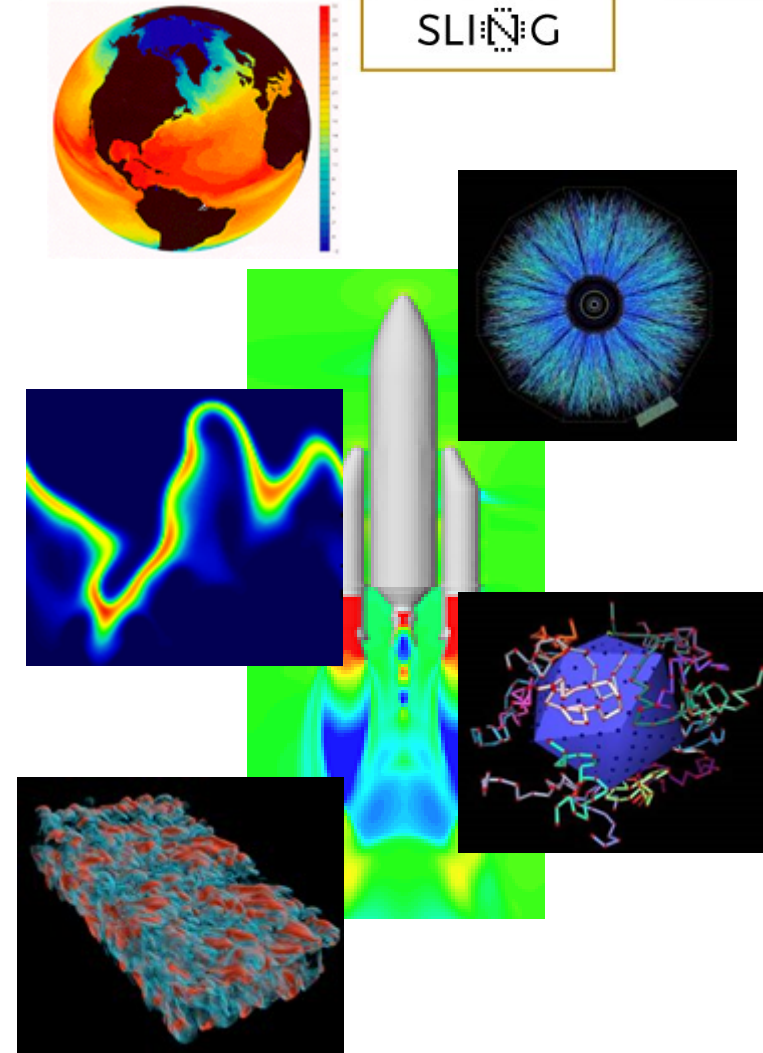
What is High Performance Computing (HPC)



Area of usage



- **Weather, Climatology, Earth Science**
 - degree of warming, scenarios for our future climate.
 - understand and predict ocean properties and variations
 - weather and flood events
- **Astrophysics, Elementary particle physics, Plasma physics**
 - systems, structures which span a large range of different length and time scales
 - quantum field theories like QCD, ITER
- **Material Science, Chemistry, Nanoscience**
 - understanding complex materials, complex chemistry, nanoscience
 - the determination of electronic and transport properties
- **Life Science**
 - system biology, chromatin dynamics, large scale protein dynamics, protein association and aggregation, supramolecular systems, medicine
- **Engineering**
 - complex helicopter simulation, biomedical flows, gas turbines and internal combustion engines, forest fires, green aircraft,
 - virtual power plant



About the workshop



Description:

The three-day workshop includes a presentation of the basic and advanced OpenFOAM usage, with explanation of programming of new sets within the framework. The basic part includes steps of making a mesh, setting boundary / initial conditions, running the problem on the HPC system and reviewing the results. The advanced part covers more complex cases with advanced application usage. What is not covered in the OpenFOAM can be upgraded within the software environment with implementation of your own code. Showcases and hands-on cases will be presented.

About the workshop



Skills to be gained:

- HPC usage,
- Background basics of Computational fluid dynamics (CFD),
- Understanding numerical analysis,
- Basic usage of OpenFOAM,
- OpenFOAM Programming.

About the workshop



Lecturer:

- **Aleksander Grm** began his studies in physics/mathematics at the Faculty of Mathematics and Physics at the University of Ljubljana. In between, he constantly improves his programming skills and uses them in solving scientific computational problems in the field of hydromechanics. Looking for a suitable Master's programme, he gives tutorials in mechanics and mathematics and works as a programmer in some companies. He then continues his studies at the ICTP/SISA in Trieste, Italy, in the Master's course in Applied Mathematics. The course covers the field of modelling and simulation of complex realities. At the end of the Master's course, he received a PhD scholarship from ITWM Fraunhofer, Kaiserslautern, Germany, and completed his PhD in Industrial/Applied Mathematics at the University of Kaiserslautern. After his PhD, he worked partly in academia and entirely in industry. In industry he's involved in many different projects, all related to mathematical modelling of real world problems. In industry, he works as Chief Strategy Officer at C3M, where he leads various industry projects and plans the strategy for the future scientific direction of the company. In 2014, he moves to academia to work in pure and applied research, teaching young people mechanics and mathematics at engineering level.

- Hardware:
 - nadzorni strežniki (master and login nodes)
 - delovna vozlišča (worker nodes)
 - datotečni strežnik (ZFS, NFS, LUSTRE)
 - omrežni del (Infiniband, Ethernet)
- Available software:
 - Ansys Multiphysics
 - Ansys CFX, Fluent, Maxwell, HFSS
 - OpenFOAM CFD + extend
 - VisIt in ParaView postprocesor
 - Intel F90, CC
 - TotalView, Alinea DDT
 - Modelirnik Siemens NX in ProEngineer
 - Octave, R, Mathematica
 - OpenMP, OpenMPI, HPMPI, IntelMPI
 - ATLAS, BLAS, BLACS, FFTW, GOTO, MUMPS, NetCDF, HDF5, Sparsekit, Scalapack, ...



Day 01, 13 June 2022



Beginning	End	Description
13:00	14:30	Introduction to Computational fluid dynamics (CFD) <ul style="list-style-type: none"> About CFD Theoretical background Different methods Pros and cons of the method Historical overview of method development <p>Conveners: Aleksander Grm</p>
14:30	14:45	Coffee break
14:45	15:30	Introduction to supercomputers, supercomputing world, benefits for the research and industrial needs. <ul style="list-style-type: none"> Introducing the participants to modern hardware, HPCs, EuroHPC project Using HPC-FS and NoMachine desktop Linux background <p>Conveners: Leon Kos, Pavel Tomšič</p>
15:30	17:00	Introduction OpenFOAM <p>Setting up environment for OpenFOAM, description of the basic program environment and running of basic programs. Outline:</p> <ul style="list-style-type: none"> Overview of OpenFOAM Workflow and background How to use OpenFOAM Main Components Parallel construct on HPC How to set up OpenFOAM on a local PC <p>Conveners: Aleksander Grm</p>

Day 02, 14 June 2022



Beginning	End	Description
13:00	14:30	Basic usage of OpenFOAM I <ul style="list-style-type: none">• geometry and meshing• mesh manipulations• physical modelling and numerical simulations• initial and boundary conditions Conveners: Aleksander Grm
14:30	14:45	Coffee break
14:45	15:30	Basic usage of OpenFOAM II <ul style="list-style-type: none">• HPC / Cloud• post-processing• basic features of complex simulations• OpenFOAM format conversions Conveners: Aleksander Grm
15:30	17:00	Hands-on with OpenFOAM I <ul style="list-style-type: none">• simple cases for the participants Conveners: Aleksander Grm

Day 03, 15 June 2022



Beginning	End	Description
13:00	14:30	OpenFOAM Programming <ul style="list-style-type: none">• Building blocks of the OpenFOAM library• Solver• Utilities• Boundary conditions Conveners: Aleksander Grm
14:30	14:45	Coffee break
14:45	15:30	Advanced applications <ul style="list-style-type: none">• Mesh manipulation• Multi-phase flows• Thermal simulation• Advanced cases of OpenFOAM usage Conveners: Aleksander Grm
15:30	17:00	Hands-on with OpenFOAM II Advanced cases of OpenFOAM usage Conveners: Aleksander Grm



Thanks!



This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 951732. The JU receives support from the European Union's Horizon 2020 research and innovation programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, United Kingdom, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Switzerland, Turkey, Republic of North Macedonia, Iceland, Montenegro



EuroHPC
Joint Undertaking