



EuroCC workshop

CFD on HPC: OpenFoam

Leon Kos, Faculty for mechanical engineering, University of Ljubljana

Date: 26-06-2023

About EuroCC project

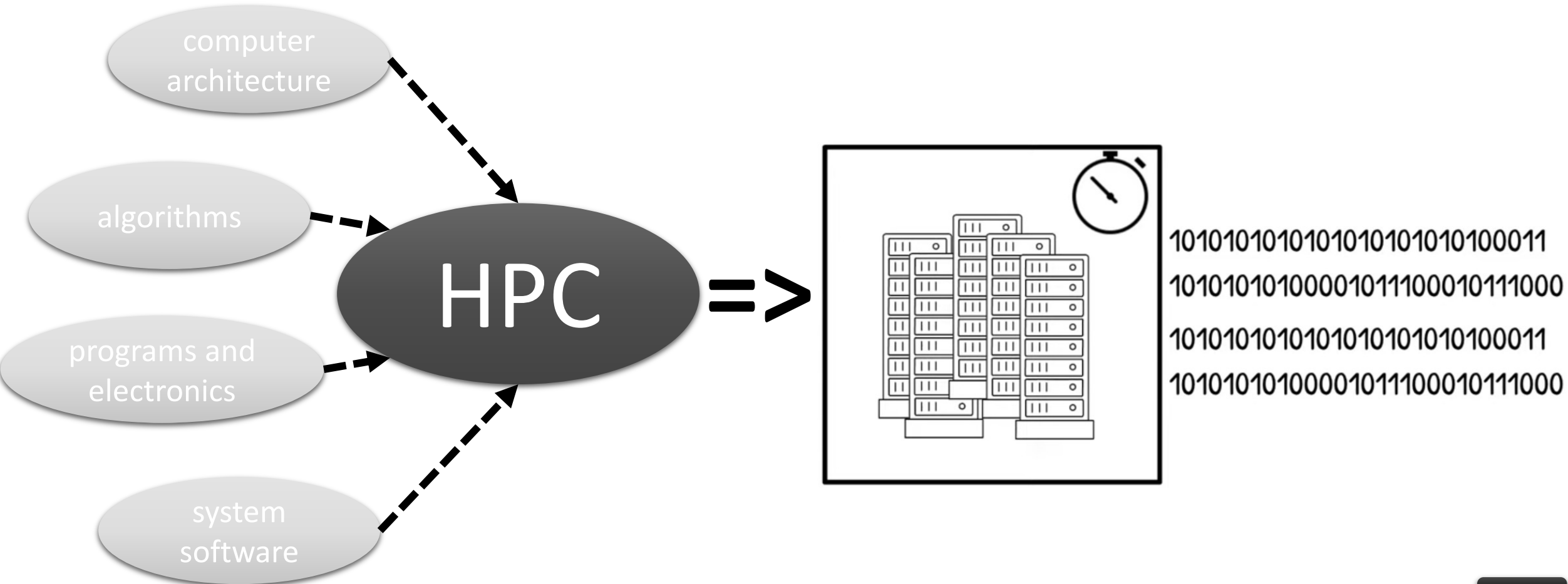


- National Competence Centres for HPC in 33 countries
- to fill the gaps in HPC knowledge in different areas:
 - Use in industry (SME),
 - School education,
 - Higher education institutions and
 - Enthusiastic individuals.



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**

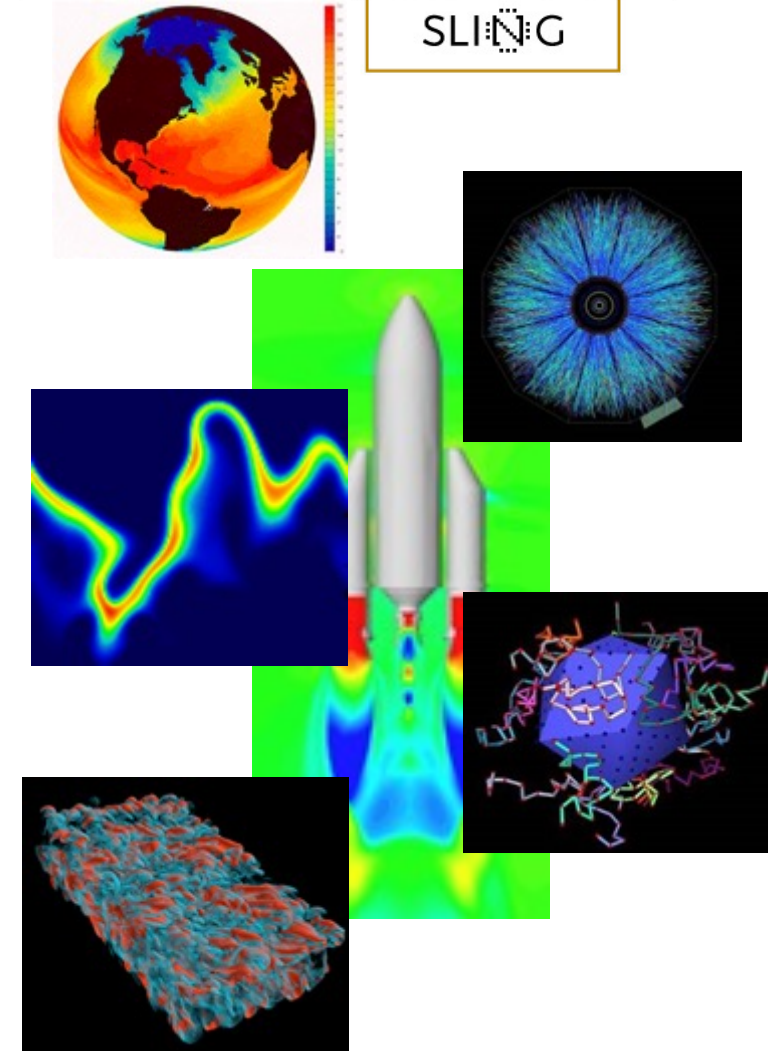
- understanding complex materials, complex chemistry, nanoscience
- the determination of electronic and transport properties

- **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, ...



Agenda

MONDAY, 26 JUNE



09:00 → 10:30 **CFD on HPC: Introduction to supercomputers, supercomputing world, benefits for the research and industrial needs.**
Convener: Pavel Tomšič (Faculty for Mechanical Engineering, University of Ljubljana)

10:30 → 12:00 **CFD on HPC: Introduction to Computational fluid dynamics (CFD)**
Convener: Aleksander Grm

12:00 → 13:00 **CFD on HPC: Setting up environment for OpenFOAM, description of the basic program environment and running of basic programs.**
Convener: Aleksander Grm

TUESDAY, 27 JUNE



09:00 → 10:30 **CFD on HPC: Basic usage of OpenFOAM I**
Convener: Aleksander Grm

10:30 → 11:30 **CFD on HPC: Hands-on with OpenFOAM I**
Convener: Aleksander Grm

WEDNESDAY, 28 JUNE



09:00 → 10:30 **CFD on HPC: Basic usage of OpenFOAM II**
Convener: Aleksander Grm

10:30 → 12:00 **CFD on HPC: Hands-on with OpenFOAM II**
Convener: 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