CFD Basic Module

How to Run Various Test Cases on Shaheen?

Rooh Khurram

CFD Services

- Software: Maintaining and Upgrading Software and Test Cases
- Training: Basic, Intermediate and Advanced
- Helpline: cfd-helpline@hpc.kaust.edu.sa
- CFD Consulting Projects
- ANSYS Workshop Series
- CFD User Group Meetings
- Student Certification
- 1-week Summer & Winter CFD School

Core Labs and
Research Infrastructure

جامعة الملك عبدالله
للعلوم والتقنية
King Abdullah University of
Science and Technology
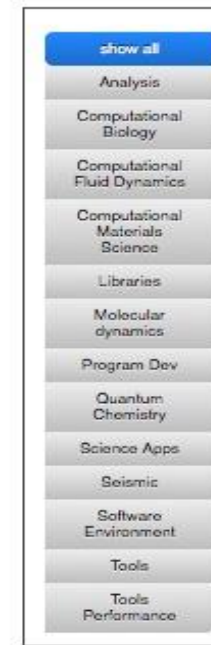
Basic CFD Module

- Quick Overview of Applications on Shaheen
- Typical CFD workflow
- CFD Example Cases
    - Ansys/Fluent
    - CONVERGE
    - OpenFOAM
    - Paraview
    - Tecplot
    - Strong Scalability
    - Interactive Computing in Batch Mode
- Shaheen users can find all material here:

/sw/ex111genoa/openfoam/cfd_module
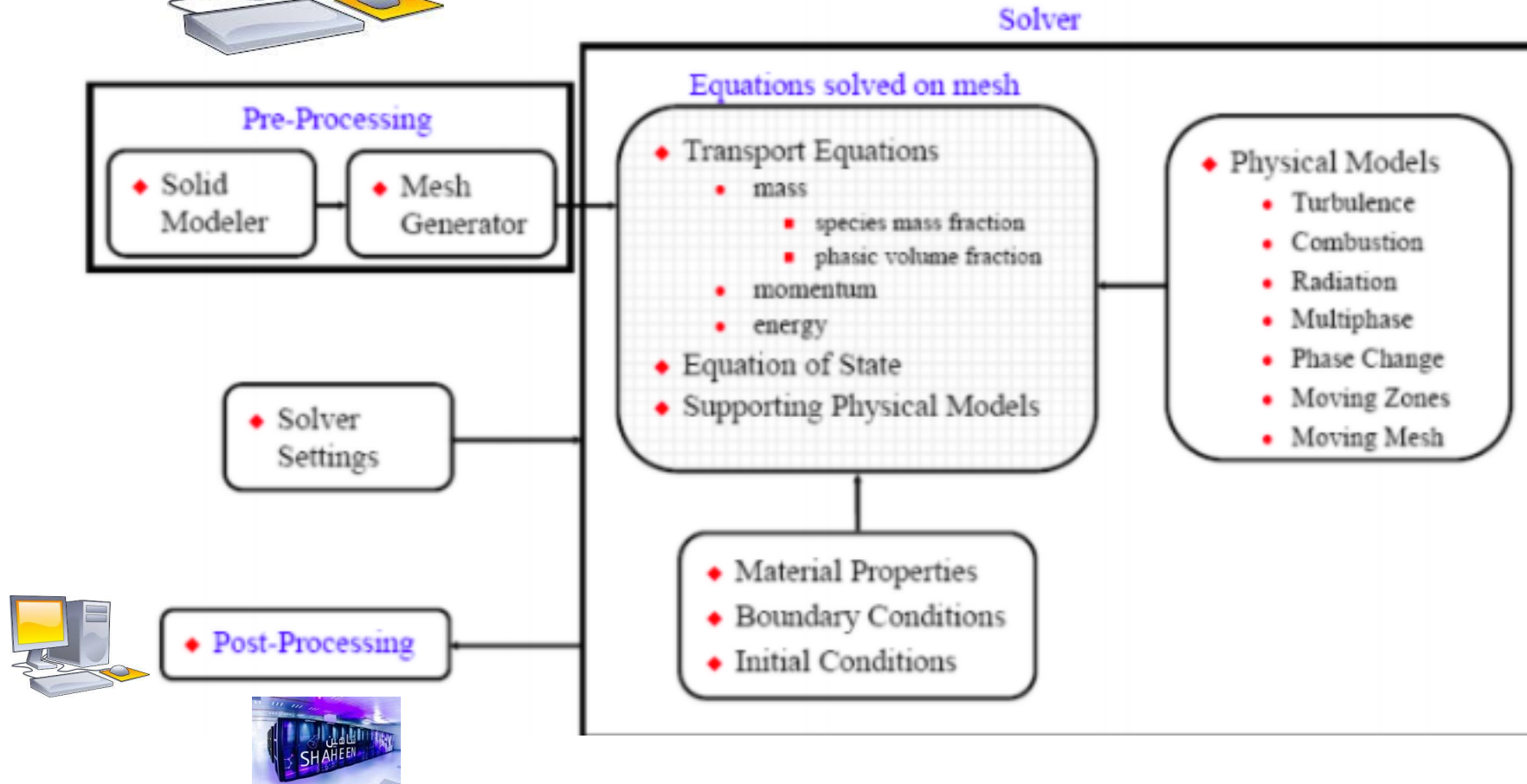
Applications on Shaheen

CFD Applications:

- Commercial codes (Fluent, CONVERGE)
- Open source codes (OpenFOAM)
- Code development tools (dealii, FEniCS, OpenFOAM)
- Mesh generation (ANSYS)
- Post processing (Tecplot, visit, paraview, Ensight)

Typical CFD Workflow

Hands-on Demo

- Log into Shaheen
- cd /scratch/$USER
- cp –r **/sw/ex111genoa/openfoam/cfd_module .**
- cd cfd_module