Introduction to OpenFOAM Development

Unit 1: Getting Familiar

Mustafa Bhotvawala, SkillLync

1 Your choices
Your version choices

2 Want to code?
Solvers
Equations

Your version choices

- Often confusing to new users, there are two different versions of OpenFOAM available
- OpenFOAM® is a trademark of ESI-OpenCFD, which distributes it through www.openfoam.com.
- The OpenFOAM Foundation, which has the permission to use the trademark too, distributes the software through www.openfoam.org
- ESI-OpenCFD uses a naming convention like this: "v1806" for the version released in the middle of 2018
- The OpenFOAM Foundation uses a convention like "v5.0". The number is not linked to the date of release

1 Your choices
Your version choices

2 Want to code?
Solvers
Equations

Equations

A scalar transport equation as discretized in OpenFOAM

$$\frac{\partial T}{\partial t} + \nabla \cdot (\mathbf{U} T) - \nabla \cdot (D \nabla T)$$

```
        solve
(
    fvm::ddt(T)
  + fvm::div(phi, T)
  - fvm::laplacian(DT, T)
);
```

References