OpenFOAM® Journal

Volume 00, Pages 000–000 (Xxxx XX, XXXX)
S xxxx-yyyy(XX)0000-0

OpenFOAM® JOURNAL PUBLICATION

AUTHOR1 AND AUTHOR2

Address1
Email address: `Emailaddress1`

Address2
Email address: `Emailaddress2`

DOI: 00.1000/xxx
Version: OpenFOAM® v19xx
Repo: https://github.com/xxx

Abstract. This is the place for an abstract.

1. Introduction

This is the place for introduction.

1.1. **Subsection.** Example text:

We shall consider the specific transport property ϕ and note that its spatial and temporal variation is governed by a second-order partical differential equation (PDE), viz.

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla\cdot(\rho\phi\mathbf{U}) - \Gamma_\phi \nabla^2 \phi - S_\phi(\phi) = 0. \tag{1.1}$$

Herein, $\phi = \phi(\mathbf{x}, t)$ is an arbitrary general intensive physical quantitity, e.g., a fluid property (scalar or tensor of any rank). Thus, (1.1) is often referred to as generic transport equation.

OpenFOAM® (Open Field Operation And Manipulation) is a flexible and mature C++ Class Library for Computational Continuum Mechanics (CCM) and Multiphysics. Its Object-Oriented-Programming (OOP) paradigm enables to *mimic data types and basic operations* of CCM using top-level syntax as close as possible to the conventional mathematical notation *for tensors and partial differential equations*:

```
solve
(
  fvm::ddt(rho,Phi)
  + fvm::div(phi, Phi)
  - fvm::laplacian(Gamma, Phi)
 ==
  Sphi
);
```

Received by the editors xx.xx.202x.

Table 1. Finite Volume Notation

implicit differential operators	
rate of change	$\left[\!\left[\frac{\partial[\rho\phi]}{\partial t}\right]\!\right]$
convection term	$\left[\!\left[\nabla\bullet\left(F[\phi]_{f(F,S,\gamma)}\right)\right]\!\right]$
diffusion term	$[\![\nabla\bullet(\Gamma\nabla[\phi])]\!]$
linear part of source term	$[\![S_p\,[\phi]]\!]$
explicit differential operators	
temporal term	$\frac{\partial\rho\phi}{\partial t}$
divergence term	$\nabla\bullet(\rho\mathbf{U}\phi_{f(\rho\mathbf{U},S,\gamma)})$
laplacian term	$\nabla\bullet(\Gamma\nabla\phi)$
constant part of source term	S_u

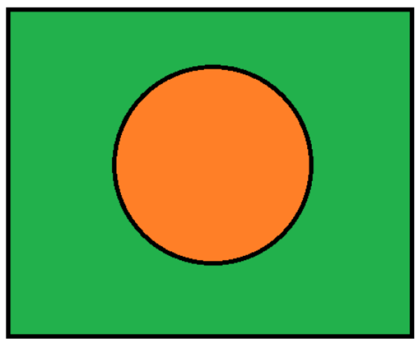

Figure 1. Examplary figure

Beside providing OpenFOAM code itself, spatial and temporal discretisation of Eq. 1.1 can be also described in a precise and concise manner using the finite-volume notation [1] - see Tab. 1.

2. Theoretical backgroud

Text in this section. Here is an examplary figure 1.

3. Conclusion

This is a conclusion.

References

[1] H. Rusche. *Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions*. PhD thesis, Imperial College of Science, Technology & Medicine London, 2002.