

# Finite volume options

## Table of Contents

[Introduction](#)

[Options](#)

[Usage](#)

[Selecting the region](#)

## Introduction

OpenFOAM solver applications typically include core functionality such as turbulence modelling, heat transfer, and buoyancy effects.

Further flexibility is offered via `fvOptions`—a collection of run-time selectable finite volume options to manipulate systems of equations by adding sources/sinks, imposing constraints and applying corrections.

These are specified in the `fvOptions` file located in the `$FOAM_CASE/system` or `$FOAM_CASE/constant` directories.

## Options

- [Sources](#)
- [Constraints](#)
- [Corrections](#)

## Usage

### Selecting the region

The majority of options are applied to collections of mesh cells. These can be selected according to the entry `selectionMode`, e.g.

```
selectionMode    all;
```

Valid `selectionMode` entries include:

- `all`: all cells
- `cellZone`: cells defined by a cell zone. This requires an additional entry to specify the name of the cell zone, e.g.

```
selectionMode    cellZone;  
cellZone         myCellZone;
```

where `myCellZone` is the name of the cell zone

- `cellSet`: cells defined by a cell set. This requires an additional entry to specify the name of the cell set, e.g.

```
selectionMode    cellSet;  
cellSet          myCellSet;
```

where `myCellSet` is the name of the cell set.

- `points`: a list of points. This requires an additional entry to list the points, e.g.

```
selectionMode    points;  
points           ((0 0 0) (1 1 1) (2 2 2));
```


---

Would you like to suggest an  
improvement to this page?

Create an issue

---

Copyright © 2016-2018 OpenCFD Ltd.

Licensed under the Creative Commons License BY-NC-ND  [Creative Commons License](#)