

```

/*-----*|
=====
|| / F i e l d      | OpenFOAM: The Open Source CFD Toolbox
|| / O p e r a t i o n |
|| / A n d           | www.openfoam.com
|| V M a n i p u l a t i o n |

```

Copyright (C) 2011 OpenFOAM Foundation

License

This file is part of OpenFOAM.

OpenFOAM is free software: you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation, either version 3 of the License, or (at your option) any later version.

OpenFOAM is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

You should have received a copy of the GNU General Public License along with OpenFOAM. If not, see <<http://www.gnu.org/licenses/>>.

Description

Calculates and outputs the mean and maximum Courant Numbers for the fluid regions

```

|*-----*/

```

```

#ifdef compressibleCourantNo_H
#define compressibleCourantNo_H

```

```

#include "fvMesh.H"

```

```

namespace Foam
{
    scalar compressibleCourantNo
    (
        const fvMesh& mesh,
        const Time& runTime,
        const volScalarField& rho,
        const surfaceScalarField& phi
    );
}

```

```

#endif

```

```

// *****

```