

```

/*-----*|
=====
|| / 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) 2011-2016 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/>>.

```

|*-----*/

```

```

#include "compressibleCourantNo.H"
#include "fvc.H"

```

```

Foam::scalar Foam::compressibleCourantNo

```

```

(
    const fvMesh& mesh,
    const Time& runTime,
    const volScalarField& rho,
    const surfaceScalarField& phi
)
{

```

```

    scalar CoNum = 0.0;
    scalar meanCoNum = 0.0;

```

```

{
    scalarField sumPhi
    (
        fvc::surfaceSum(mag(phi))().primitiveField()
    );

```

```

    CoNum = 0.5*gMax(sumPhi/mesh.V().field())*runTime.deltaTValue();

```

```

    meanCoNum =
        0.5*(gSum(sumPhi)/gSum(mesh.V().field()))*runTime.deltaTValue();
}

```

```

Info<< "Region: " << mesh.name() << " Courant Number mean: " << meanCoNum

```

```
<< " max: " << CoNum << endl;  
  
return CoNum;  
}
```

```
// ***** //
```