```
/*---------------------------------------------------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     |
    \\  /    A nd           | www.openfoam.com
     \\/     M anipulation  |
-------------------------------------------------------------------------------
    Copyright (C) 2011-2014 OpenFOAM Foundation
    Copyright (C) 2020 OpenCFD Ltd.
-------------------------------------------------------------------------------
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/>.

Global
    alphaCourantNo

Description
    Calculates and outputs the mean and maximum alpha Courant Numbers.

\*---------------------------------------------------------------------------*/

#ifndef ddtAlphaNo_H
#define ddtAlphaNo_H

#include "fvMesh.H"
#include "multiphaseSystem.H"

namespace Foam
{
    scalar ddtAlphaNo
    (
        const fvMesh& mesh,
        const Time& runTime,
        const multiphaseSystem& thermol,
        const surfaceScalarField& phi
    );
}

#endif
```