

```

/*-----*- C++ -*-----*/
| ===== |
|  \ \ /  F ield      | OpenFOAM: The Open Source CFD Toolbox |
|  \ \ /  O peration | Version: 4.1 |
|  \ \ /  A nd       | Web: www.OpenFOAM.org |
|  \ \ /  M anipulation |
/*-----*-*/

FoamFile
{
    version      2.0;
    format       ascii;
    class        volScalarField;
    object       nut;
}
// *****

dimensions      [0 2 -1 0 0 0 0];

internalField   uniform 0.14;

boundaryField
{
    inlet
    {
        type          freestream;
        freestreamValue uniform 0.14;
    }

    outlet
    {
        type          freestream;
        freestreamValue uniform 0.14;
    }

    airfoil
    {
        type          nutUSpaldingWallFunction;
        value         uniform 0;
    }

    frontAndBack
    {
        type          empty;
    }
}

// *****

```