

```

/*-----* C++ *-----*/
|=====|
|  \ \ /  F i e l d      | OpenFOAM: The Open Source CFD Toolbox
|  \ \ /  O p e r a t i o n | Version: 4.1
|  \ \ /  A n d           | Web:      www.OpenFOAM.org
|  \ \ /  M a n i p u l a t i o n |
/*-----*

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

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

internalField   uniform 0;

boundaryField
{
    inlet
    {
        type          freestreamPressure;
    }

    outlet
    {
        type          freestreamPressure;
    }

    airfoil
    {
        type          zeroGradient;
    }

    frontAndBack
    {
        type          empty;
    }
}

// *****

```