

```

/*-----*- C++ -*-----*\
/ ===== /
/ \ \ / F i e l d / OpenFOAM: The Open Source CFD Toolbox /
/ \ \ / O p e r a t i o n / Version: 2.1.1 /
/ \ \ / 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
{
    top
    {
        type      zeroGradient;
    }

    bottom
    {
        type      zeroGradient;
    }

    left
    {
        type      zeroGradient;
    }

    right
    {
        type      zeroGradient;
    }

    front
    {
        type      zeroGradient;
    }

    back
    {
        type      zeroGradient;
    }
}
// *****

```