

```

/*-----*- 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        volVectorField;
    object       U;
}
// *****

dimensions      [0 1 -1 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    top
    {
        type      fixedValue;
        value     uniform (1 0 0);
    }

    bottom
    {
        type      fixedValue;
        value     uniform (0 0 0);
    }

    left
    {
        type      fixedValue;
        value     uniform (0 0 0);
    }

    right
    {
        type      fixedValue;
        value     uniform (0 0 0);
    }

    front
    {
        type      fixedValue;
        value     uniform (0 0 0);
    }

    back
    {
        type      fixedValue;
        value     uniform (0 0 0);
    }
}
// *****

```