

```

/*-----*- 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        polyBoundaryMesh;
    location     "constant/polyMesh";
    object       boundary;
}
// *****

```

```
6
```

```
(
```

```

    top
    {
        type            patch;
        nFaces          16;
        startFace       336;
    }
    bottom
    {
        type            patch;
        nFaces          16;
        startFace       352;
    }
    left
    {
        type            patch;
        nFaces          16;
        startFace       368;
    }
    right
    {
        type            patch;
        nFaces          16;
        startFace       384;
    }
    front
    {
        type            patch;
        nFaces          16;
        startFace       400;
    }
    back
    {
        type            patch;
        nFaces          16;
        startFace       416;
    }
}
)

```

```
// ***** //
```