

```
/*-----*- C++ -*-----*\
=====
\\ / F i e l d   | OpenFOAM: The Open Source CFD Toolbox
\\ / O p e r a t i o n   | Website: https://openfoam.org
\\ / A n d   | Version: 6
\\ \ M a n i p u l a t i o n   |
/*-----*/
```

FoamFile

```
{
  version 2.0;
  format  ascii;
  class  dictionary;
  object  blockMeshDict;
}
// ***** //
```

convertToMeters 1;

vertices

```
(
  (0 0 0)
  (10 0 0)
  (10 5 0)
  (0 5 0)
  (0 0 10)
  (10 0 10)
  (10 5 10)
  (0 5 10)
);
```

blocks

```
(
  hex (0 1 2 3 4 5 6 7) (20 10 20) simpleGrading (1 1 1)
);
```

edges

```
(
);
```

boundary

```
(
  floor
  {
    type wall;
    faces
    (
      (1 5 4 0)
    );
  }
  ceiling
  {
    type wall;
    faces
    (
```

```
        (3 7 6 2)
    );
}
fixedWalls
{
    type wall;
    faces
    (
        (0 4 7 3)
        (2 6 5 1)
        (0 3 2 1)
        (4 5 6 7)
    );
}
);

mergePatchPairs
(
);

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