

# OpenFOAM Programming Tips

OpenFOAM Programming Tips

```
label outletPatchID = mesh.boundaryMesh().findPatchID("outlet");  
scalar outFlux = gSum(phi.boundaryField()[outletPatchID]);  
Info << "Volumetric flux = " << outFlux << " [m^3/s]" << endl;
```

Keywords :

- OpenFOAM
- findPatchID
- gSum
- faceCells

Fumiya Nozaki

Last Updated: 1 June 2014

# 1. How to get patch's label from patch's name

```
label patchID = mesh.boundaryMesh().findPatchID("NAME_OF_PATCH");  
  
Info << "patchID = " << patchID << endl;
```

## Example

```
5  
(  
0 inlet  
{  
    type            patch;  
    nFaces          30;  
    startFace       24170;  
}  
1 outlet  
{  
    type            patch;  
    nFaces          57;  
    startFace       24200;  
}  
2 upperWall  
{  
    type            wall;  
    inGroups        1 (wall);  
    nFaces          223;  
    startFace       24257;  
}  
3 lowerWall  
{  
    type            wall;  
    inGroups        1 (wall);  
    nFaces          250;  
    startFace       24480;  
}  
...  
)
```

```
label patchID = mesh.boundaryMesh().findPatchID("upperWall");  
  
Info << "patchID = " << patchID << endl;
```

⇒ patchID = 2

## 2. How to calculate the sum over the specified patch

We can calculate the total outlet flux by summing the field `phi` over the patch named `outlet`:

```
label outletPatchID = mesh.boundaryMesh().findPatchID("outlet");  
scalar outFlux = gSum(phi.boundaryField()[outletPatchID]);  
Info << "Volumetric flux = " << outFlux << " [m^3/s]" << endl;
```

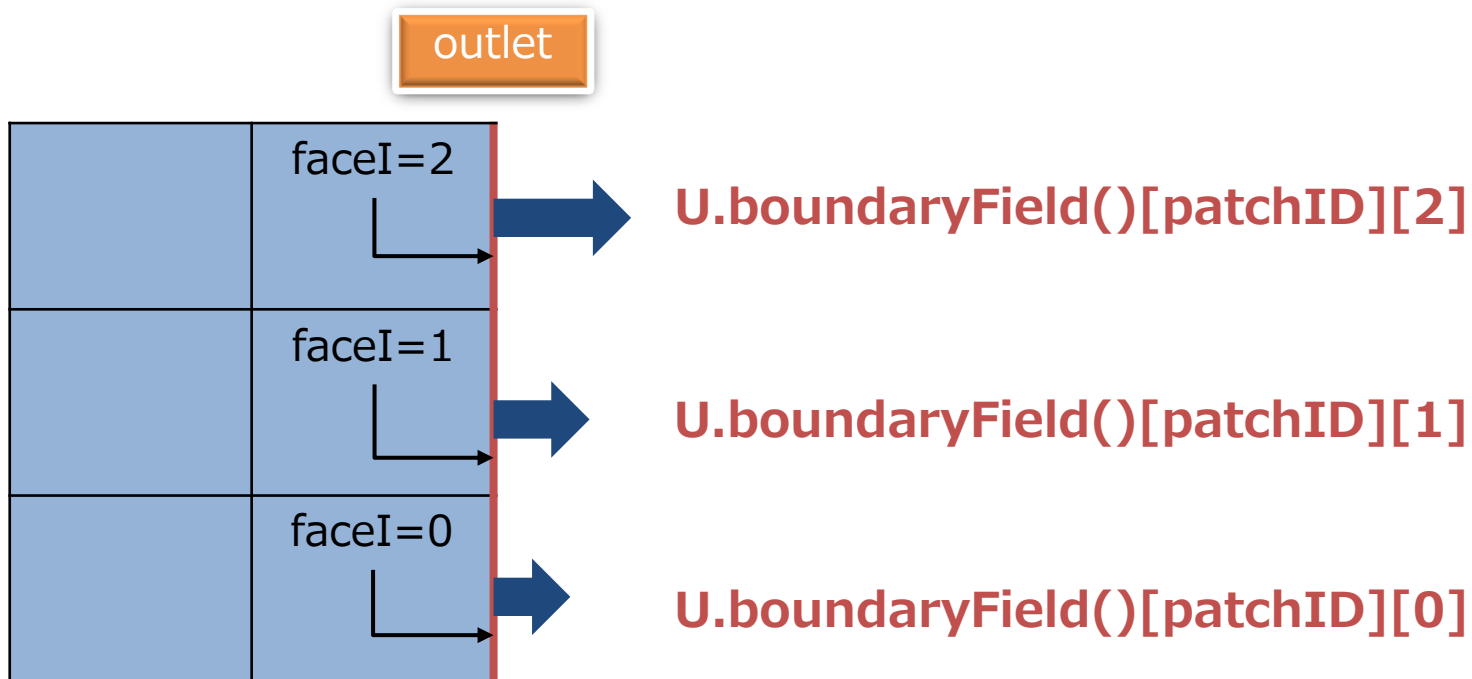
- ✓ `gSum()` sums over all the processors in a parallel run
- ✓ If you calculate the total "inlet" flux using the above code, it takes the **negative** value because the face normal vectors point in the opposite direction from the inlet velocities.



### 3. How to get a boundary value of a variable

We can get the velocity on the outlet patch using the following code:

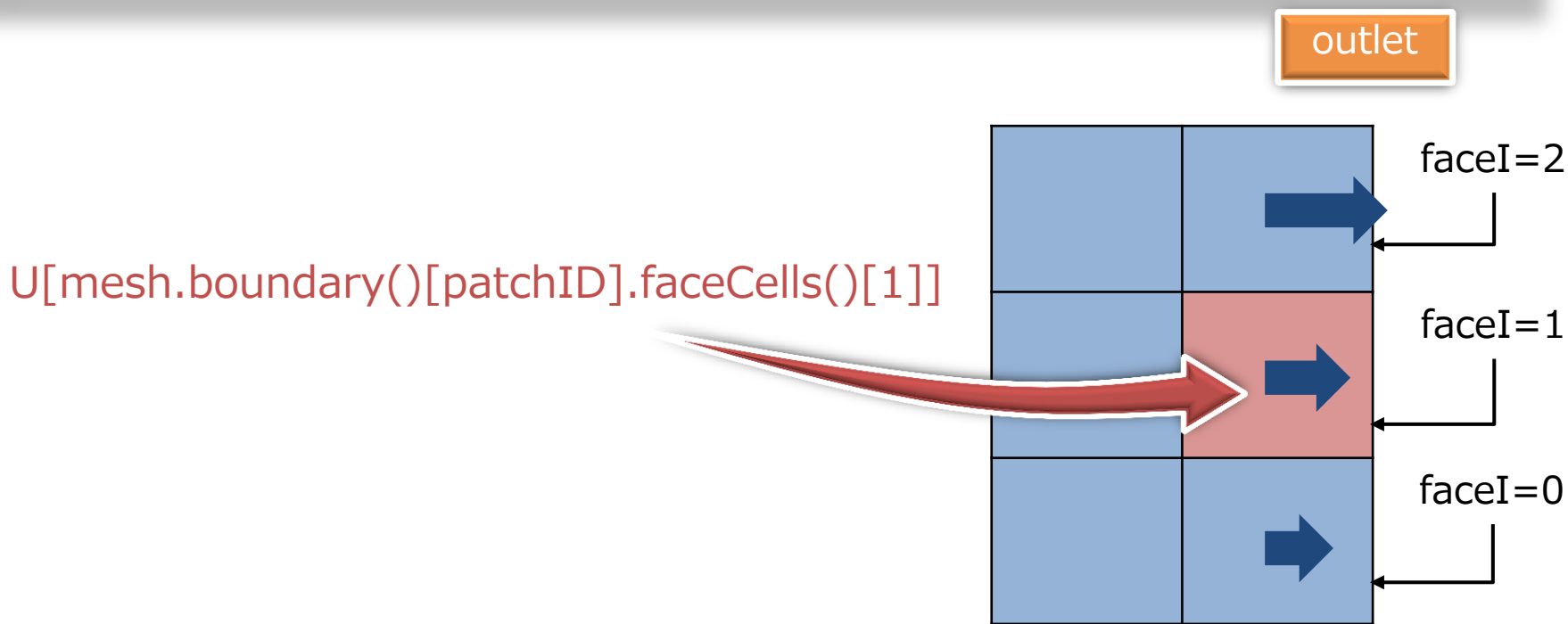
```
label patchID = mesh.boundaryMesh().findPatchID("outlet");  
  
forAll(mesh.boundary()[patchID], faceI)  
{  
    Info<< U.boundaryField()[patchID][faceI] << endl;  
}
```



## 4. How to get variable values in the cells adjacent to a patch

We can get the label list of cells adjacent to patch using `faceCells()`:

```
label patchID = mesh.boundaryMesh().findPatchID("outlet");  
  
forAll(mesh.boundary()[patchID], faceI)  
{  
    Info<< U[mesh.boundary()[patchID].faceCells()[faceI]] << endl;  
}
```



---

I will continuously update this slide in the future.

Kindly let me know  
if you have any ideas about what topics to cover.



Thank  
You!