Boundaries

Advanced course

Content

- Patch type
- Base type
- Primitive type
- Derived type

Patch type

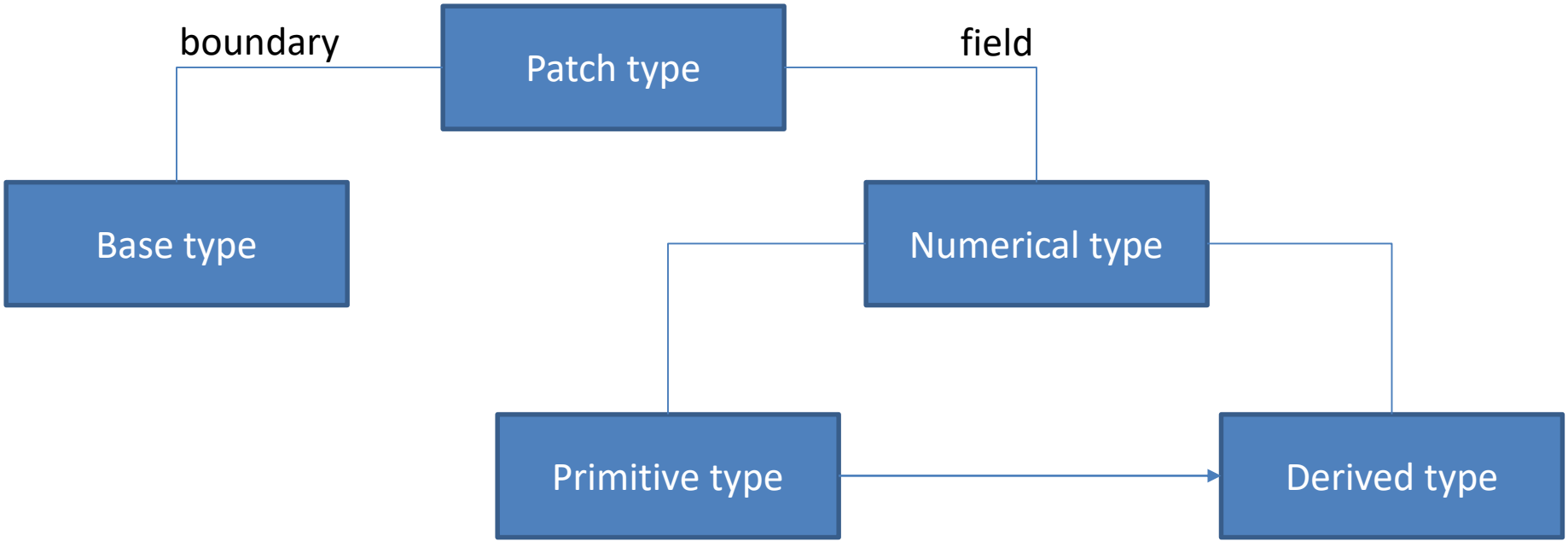
```
rotor
{
    type            wall;
    inGroups        1(wall);
    nFaces          3179;
    startFace       355492;
}
```

- Base type
  - Specifies the patch type and the associated faces
  - Location: \constant\polyMesh\boundary
- Numerical type:
  - Specifies the numerical type and the associated value
  - Can be primitive or derived
  - Location: field files under the time directories, e.g. \0\U, 0\P, ...

```
rotor
{
    type            fixedValue;
    value           uniform (0 0 0);
}
```

Patch type

- Overview



- Web: http://www.openfoam.org/docs/user/boundaries.php

Base type

- patch
  - generic patch, no physical entity (except for “wall” variant)
  - E.g. Inlet, outlet patches
- wall
  - patch that coincides with a wall
  - represents physical surface to which wall functions can be applied
- Symmetry
  - For the faces located on a symmetry plane
- empty
  - used for front and back of 2D simulations
  - e.g. Cavity flow tutorial
- wedge
  - The one-cell-thick slice used in axi-symmetric setups
- cyclic
  - 2 physically connected patches
- Processor
  - Boundary between neighbouring cells that are computed on different processors

Primitive type

- fixedValue
  - Specify the value of ϕ (scalar or vector)
- fixedGradient
  - Specify the normal gradient of ϕ (scalar)
- zeroGradient
  - Normal gradient of ϕ is zero
- calculated
  - Boundary field ϕ derived from other field
- mixed
  - Mixed fixedValue/fixedGradient condition
- directionMixed
  - Mixed with difference between normal & tangential direction

Derived type

- Patch types based on the primitive types
- Overview: