



Figure 5.2: Axi-symmetric geometry using the `wedge` patch type.

5.2.1 Geometric (constraint) patch types

The main geometric types available in OpenFOAM are summarised below. This is not a complete list; for all types see `$FOAM_SRC/finiteVolume/fields/fvPatchFields/constraint`.

- **patch**: generic type containing no geometric or topological information about the mesh, *e.g.* used for an inlet or an outlet.
- **wall**: for patch that coincides with a solid wall, required for some physical modelling, *e.g.* wall functions in turbulence modelling.
- **symmetryPlane**: for a planar patch which is a symmetry plane.
- **symmetry**: for any (non-planar) patch which uses the symmetry plane (slip) condition.
- **empty**: for solutions in 2 (or 1) dimensions (2D/1D), the type used on each patch whose plane is normal to the 3rd (and 2nd) dimension for which no solution is required.
- **wedge**: for 2 dimensional axi-symmetric cases, *e.g.* a cylinder, the geometry is specified as a wedge of small angle (*e.g.* 1°) and 1 cell thick, running along the centre line, straddling one of the coordinate planes, as shown in Figure 5.2; the axi-symmetric wedge planes must be specified as separate patches of **wedge** type.
- **cyclic**: enables two patches to be treated as if they are physically connected; used for repeated geometries; one **cyclic** patch is linked to another through a **neighbourPatch** keyword in the *boundary* file; each pair of connecting faces must have similar area to within a tolerance given by the **matchTolerance** keyword in the *boundary* file.
- **cyclicAMI**: like **cyclic**, but for 2 patches whose faces are non matching; used for sliding interface in rotating geometry cases.
- **processor**: the type that describes inter-processor boundaries for meshes decomposed for parallel running.