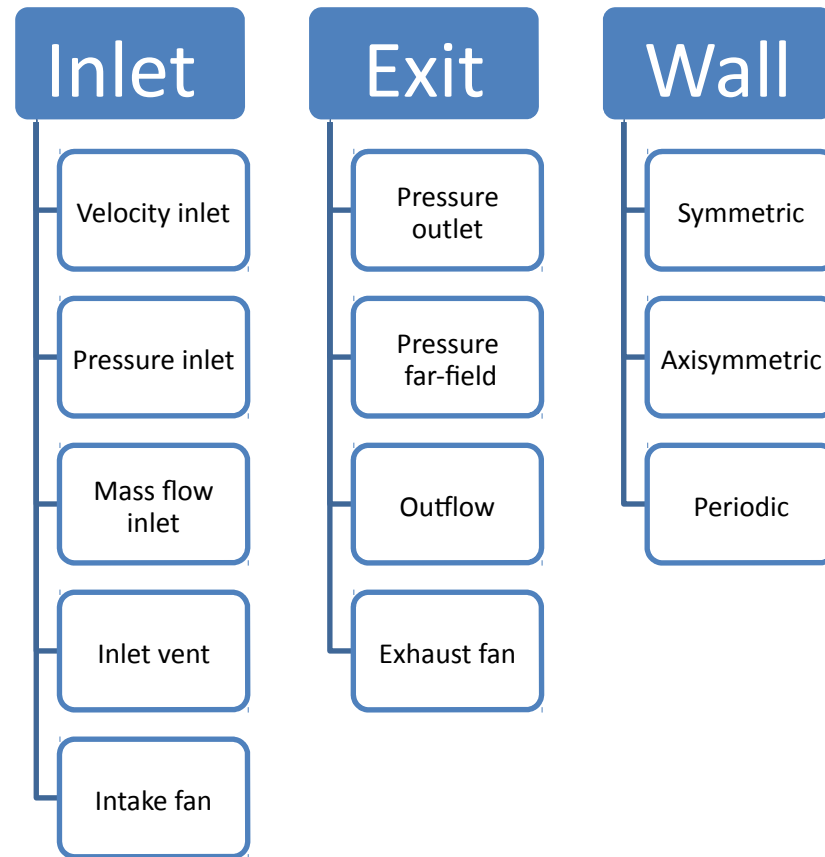


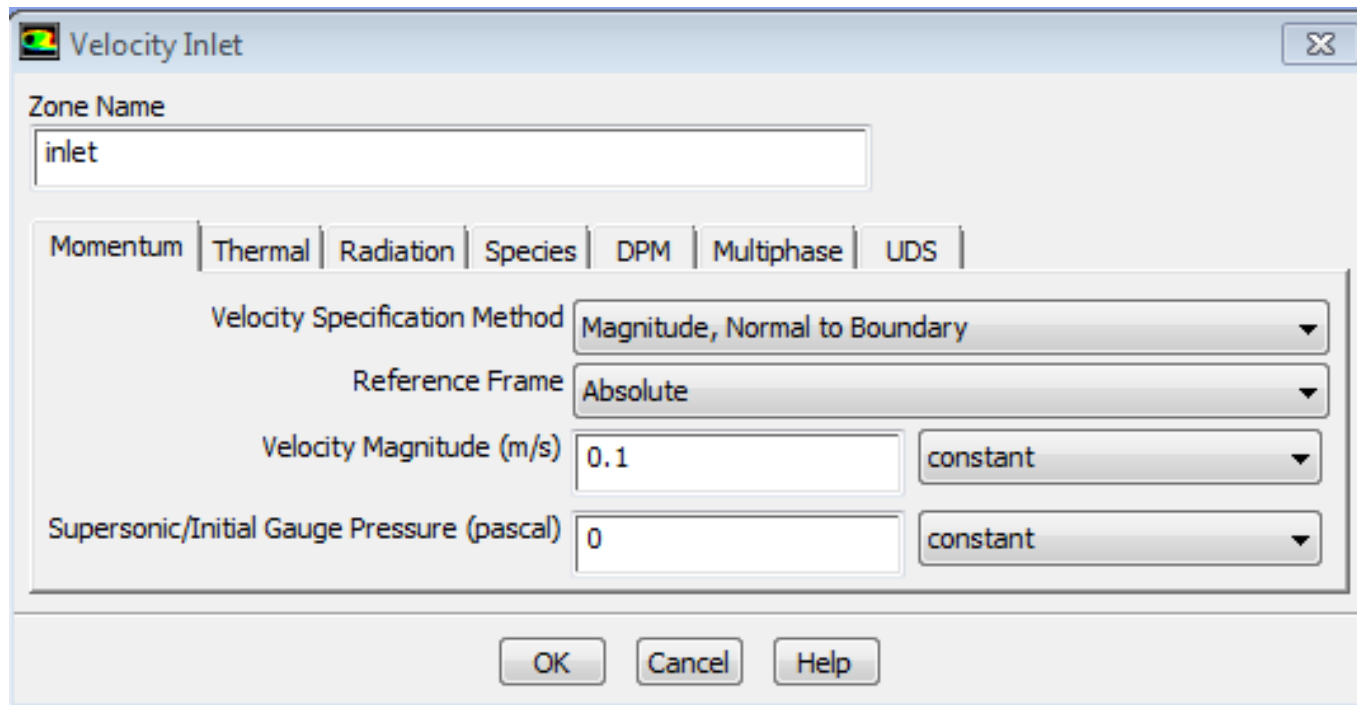
Boundary Conditions

Available BC types in Fluent



Velocity inlet

- Define the velocity and scalar properties of the flow at the boundary



The image shows a software dialog box titled "Velocity Inlet". It has a "Zone Name" field containing the text "inlet". Below this is a tabbed interface with tabs for "Momentum", "Thermal", "Radiation", "Species", "DPM", "Multiphase", and "UDS". The "Momentum" tab is selected. Inside this tab, there are two dropdown menus: "Velocity Specification Method" set to "Magnitude, Normal to Boundary" and "Reference Frame" set to "Absolute". Below these are two rows of input fields and dropdowns. The first row has "Velocity Magnitude (m/s)" with a text field containing "0.1" and a dropdown set to "constant". The second row has "Supersonic/Initial Gauge Pressure (pascal)" with a text field containing "0" and a dropdown set to "constant". At the bottom of the dialog are three buttons: "OK", "Cancel", and "Help".

Velocity Inlet

Zone Name
inlet

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

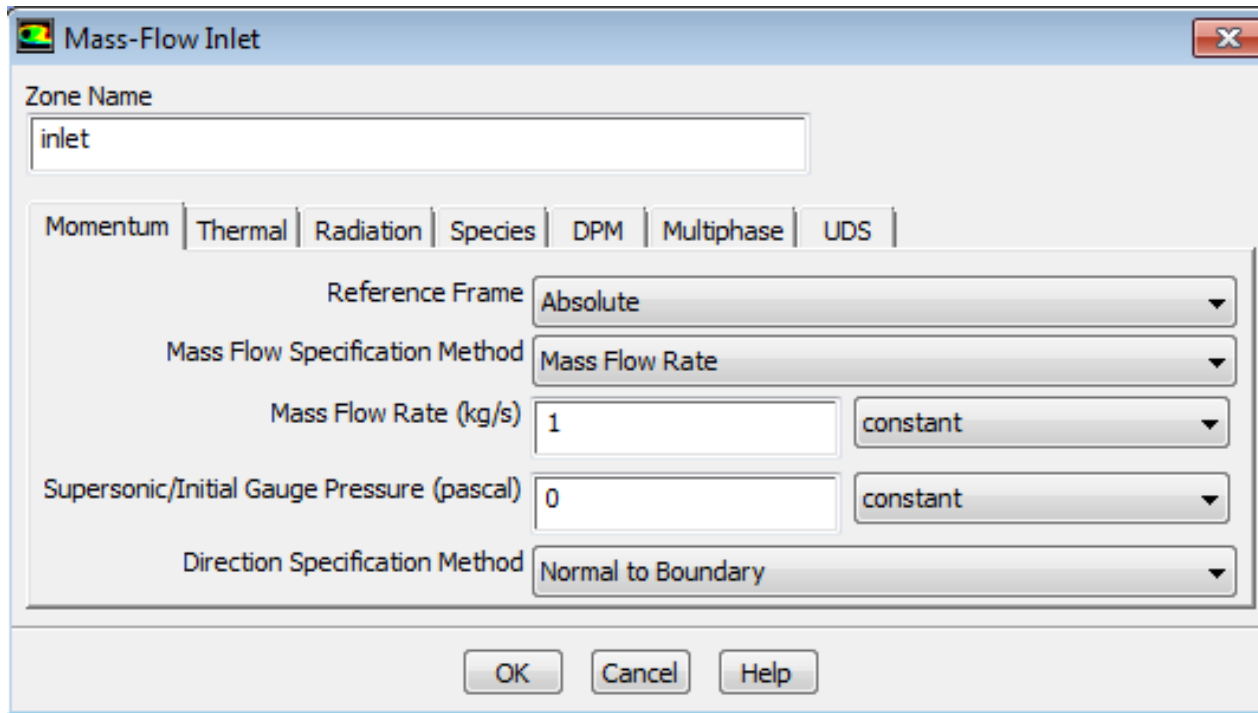
Velocity Magnitude (m/s): 0.1 constant

Supersonic/Initial Gauge Pressure (pascal): 0 constant

OK Cancel Help

Mass flow inlet

- Define the mass flow rate for compressible flows at the boundary



The image shows a software dialog box titled "Mass-Flow Inlet". It has a standard Windows-style title bar with a close button (X) in the top right corner. The dialog is divided into several sections. At the top, there is a "Zone Name" label followed by a text input field containing the word "inlet". Below this is a tabbed interface with tabs for "Momentum", "Thermal", "Radiation", "Species", "DPM", "Multiphase", and "UDS". The "Momentum" tab is currently selected. Within this tab, there are several configuration options: "Reference Frame" is set to "Absolute" via a dropdown menu; "Mass Flow Specification Method" is set to "Mass Flow Rate" via a dropdown menu; "Mass Flow Rate (kg/s)" is set to "1" in a text field, with a "constant" dropdown menu to its right; "Supersonic/Initial Gauge Pressure (pascal)" is set to "0" in a text field, with a "constant" dropdown menu to its right; and "Direction Specification Method" is set to "Normal to Boundary" via a dropdown menu. At the bottom of the dialog, there are three buttons: "OK", "Cancel", and "Help".

Mass-Flow Inlet

Zone Name
inlet

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Reference Frame: Absolute

Mass Flow Specification Method: Mass Flow Rate

Mass Flow Rate (kg/s): 1 constant

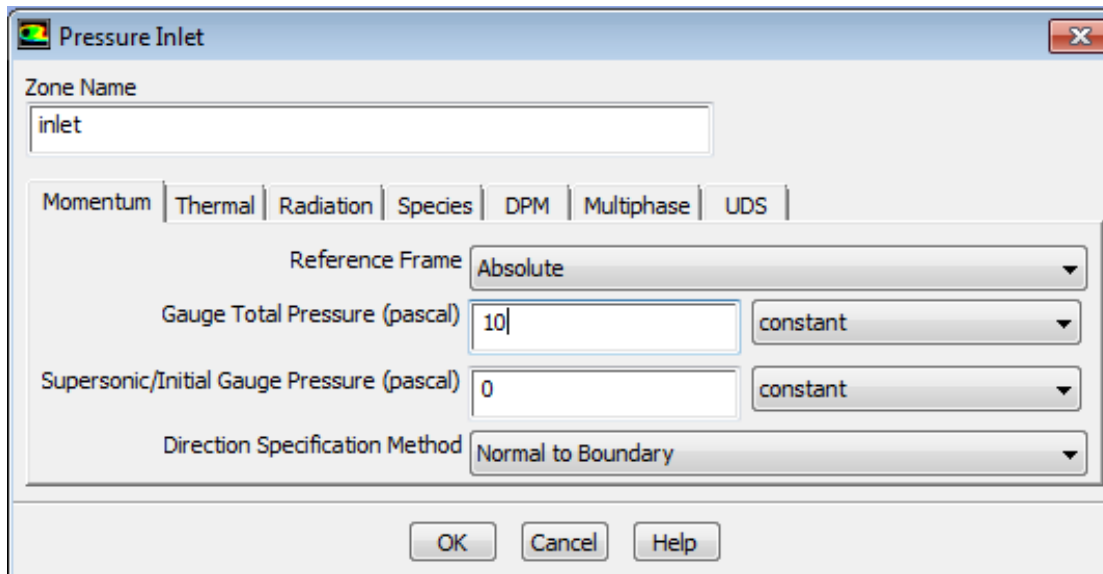
Supersonic/Initial Gauge Pressure (pascal): 0 constant

Direction Specification Method: Normal to Boundary

OK Cancel Help

Pressure inlet

- Define the total pressure and other scalar properties at the boundary
- $p_{\text{total}} = p_{\text{static}} + 0.5\rho v^2$

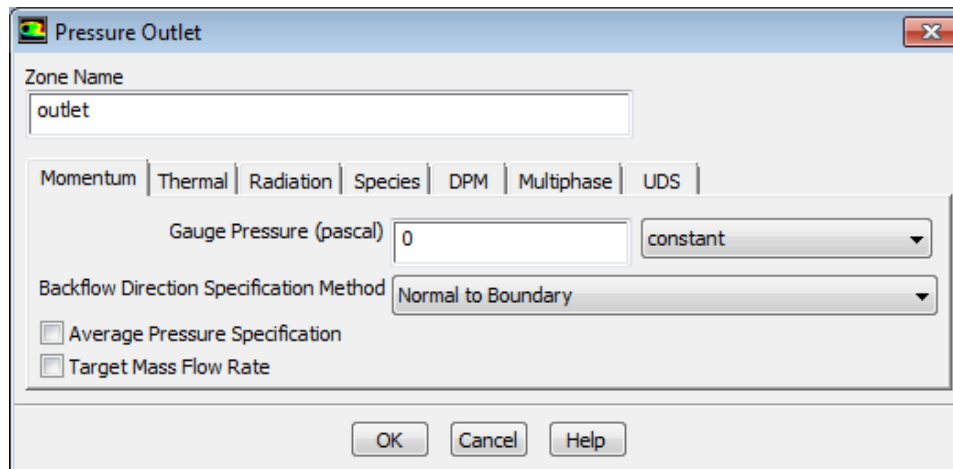


The screenshot shows a "Pressure Inlet" dialog box with the following fields and options:

- Zone Name:** inlet
- Reference Frame:** Absolute
- Gauge Total Pressure (pascal):** 10 | constant
- Supersonic/Initial Gauge Pressure (pascal):** 0 | constant
- Direction Specification Method:** Normal to Boundary
- Buttons:** OK, Cancel, Help

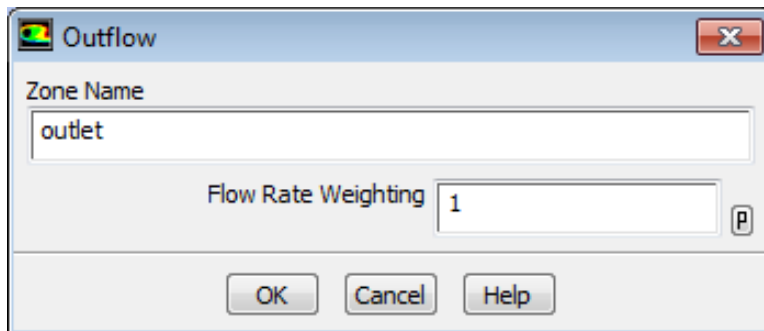
Pressure outlet

- Define the static pressure (and also other scalar variables, in case of backflow) at the boundary



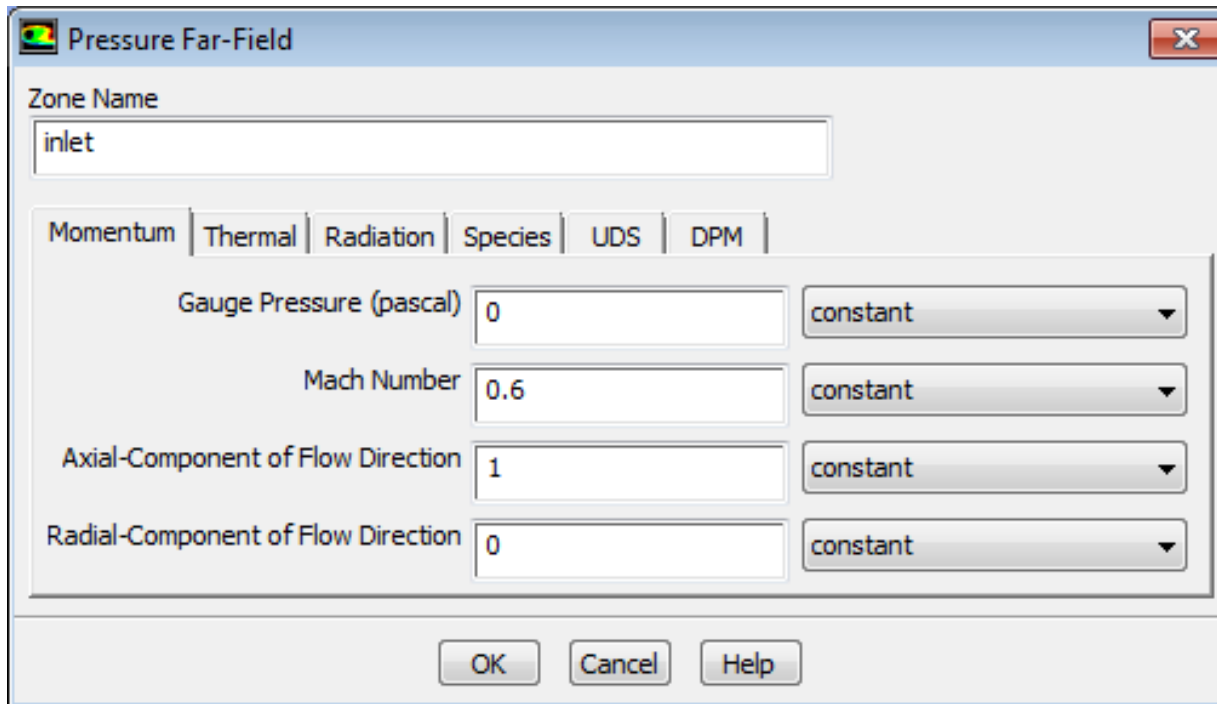
Outflow

- Assume zero streamwise gradient for all flow variables except pressure
- Not appropriate for compressible flows
- Cannot use with pressure inlet



Pressure far-field

- Model a free-stream compressible flow at infinity



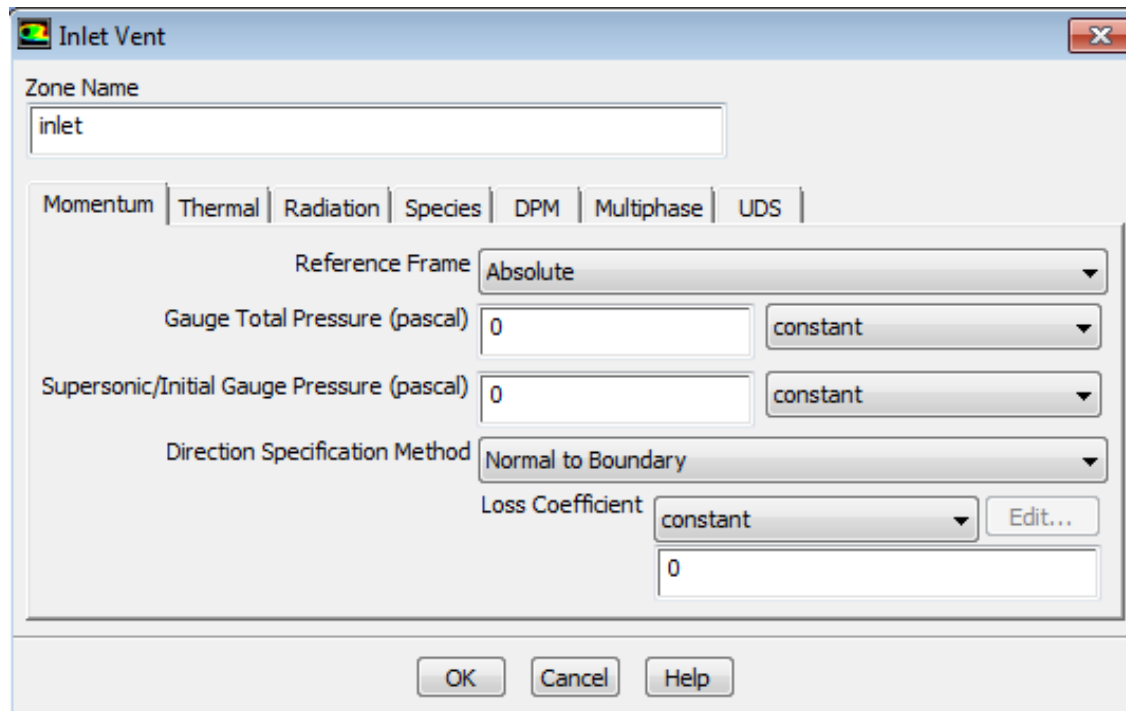
The screenshot shows a dialog box titled "Pressure Far-Field" with a standard Windows-style title bar (minimize, maximize, close buttons). The "Zone Name" field contains the text "inlet". Below this is a tabbed interface with six tabs: "Momentum", "Thermal", "Radiation", "Species", "UDS", and "DPM". The "Momentum" tab is currently selected. Under the "Momentum" tab, there are four rows of input fields, each with a text box and a dropdown menu set to "constant":

Parameter	Value	Condition
Gauge Pressure (pascal)	0	constant
Mach Number	0.6	constant
Axial-Component of Flow Direction	1	constant
Radial-Component of Flow Direction	0	constant

At the bottom of the dialog box are three buttons: "OK", "Cancel", and "Help".

Inlet/outlet vent

- Model an inlet/outlet vent with a specified loss coefficient



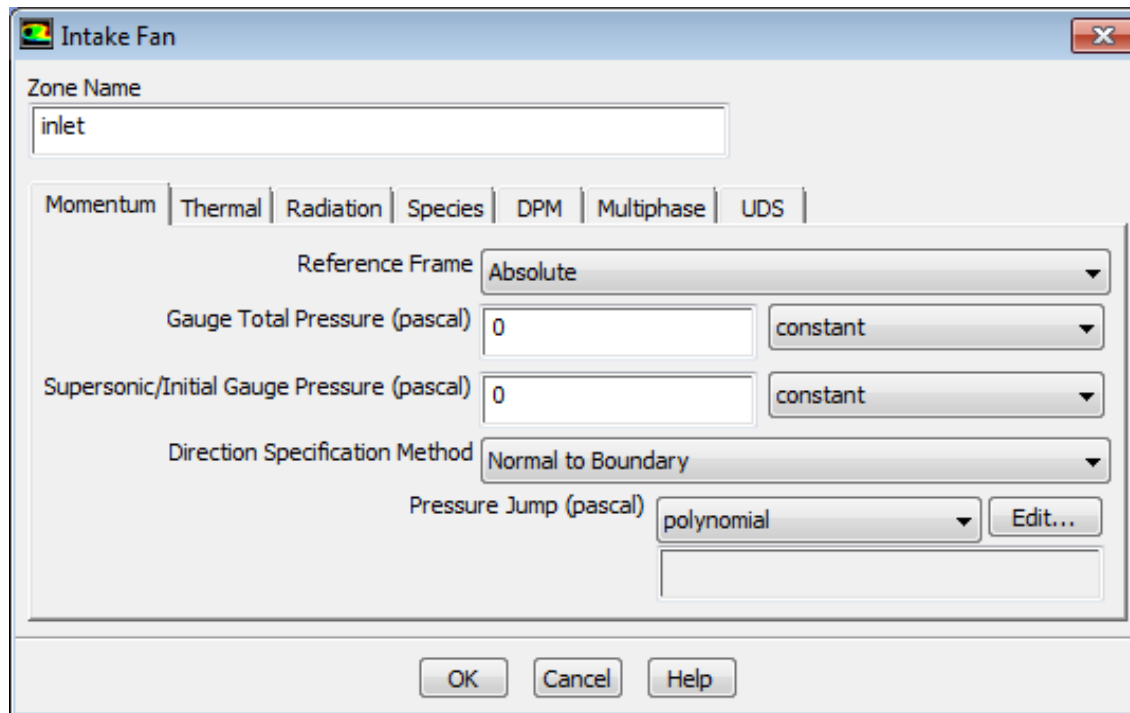
The screenshot shows the 'Inlet Vent' dialog box with the following settings:

- Zone Name:** inlet
- Reference Frame:** Absolute
- Gauge Total Pressure (pascal):** 0, constant
- Supersonic/Initial Gauge Pressure (pascal):** 0, constant
- Direction Specification Method:** Normal to Boundary
- Loss Coefficient:** constant, 0

Buttons at the bottom: OK, Cancel, Help.

Intake/exhaust fan

- Model an external intake/exhaust fan with a specified pressure jump



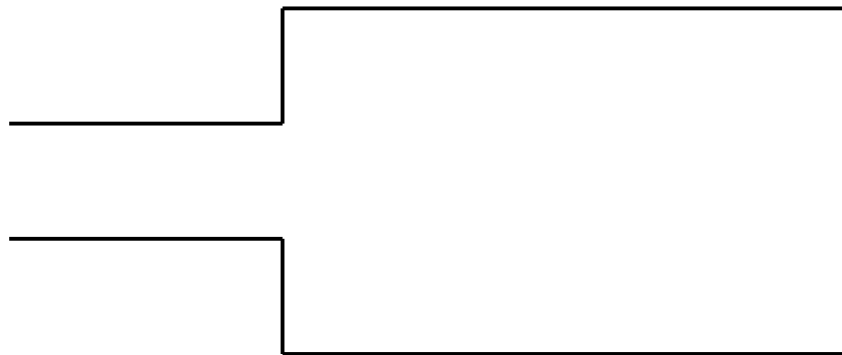
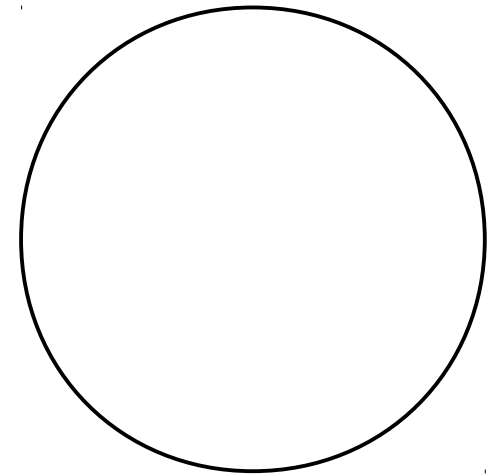
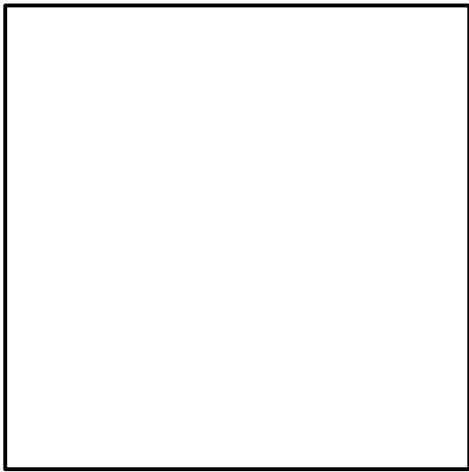
The screenshot shows the 'Intake Fan' dialog box with the following settings:

- Zone Name:** inlet
- Reference Frame:** Absolute
- Gauge Total Pressure (pascal):** 0, constant
- Supersonic/Initial Gauge Pressure (pascal):** 0, constant
- Direction Specification Method:** Normal to Boundary
- Pressure Jump (pascal):** polynomial, Edit...

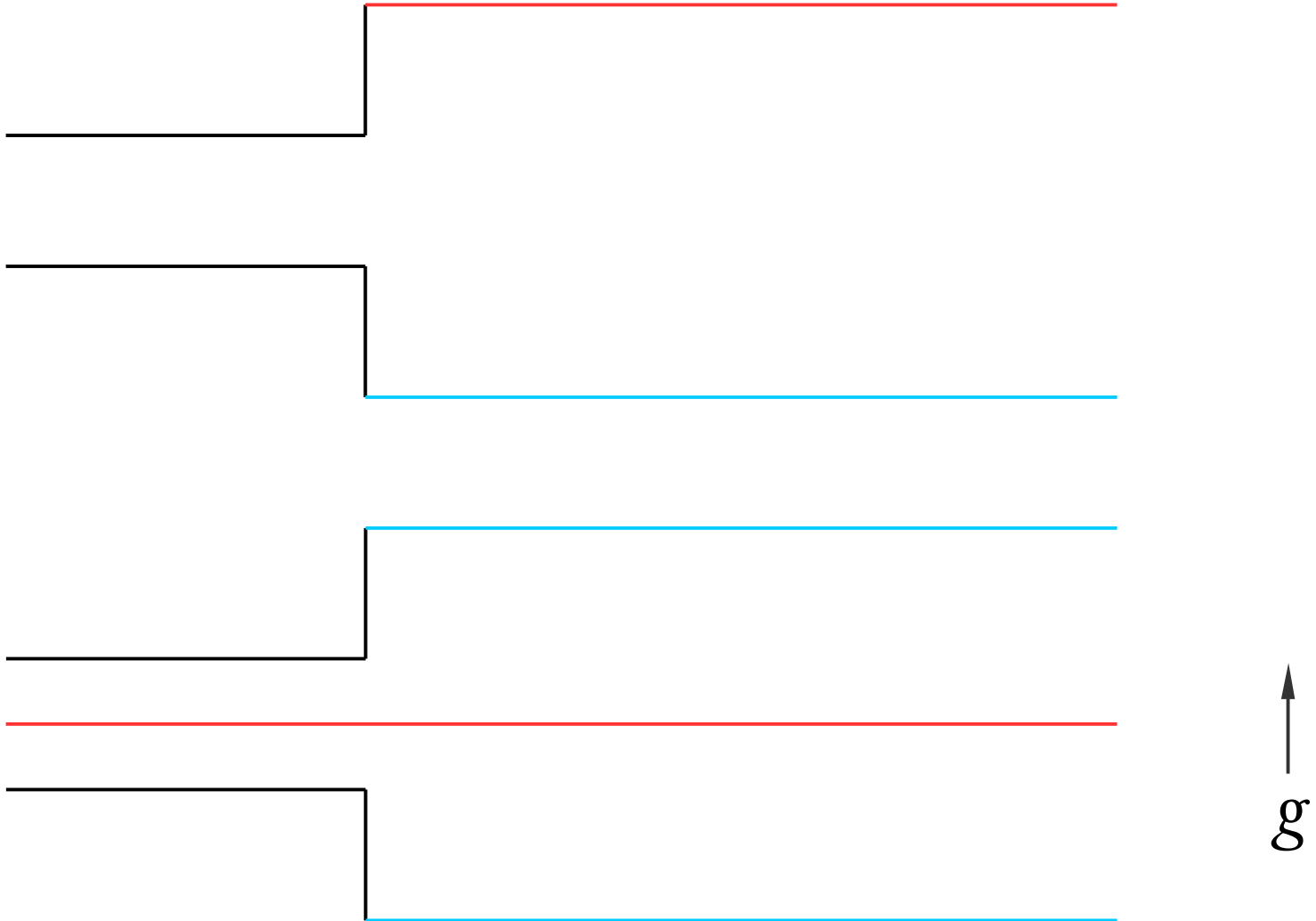
Buttons at the bottom: OK, Cancel, Help.

Symmetry BC

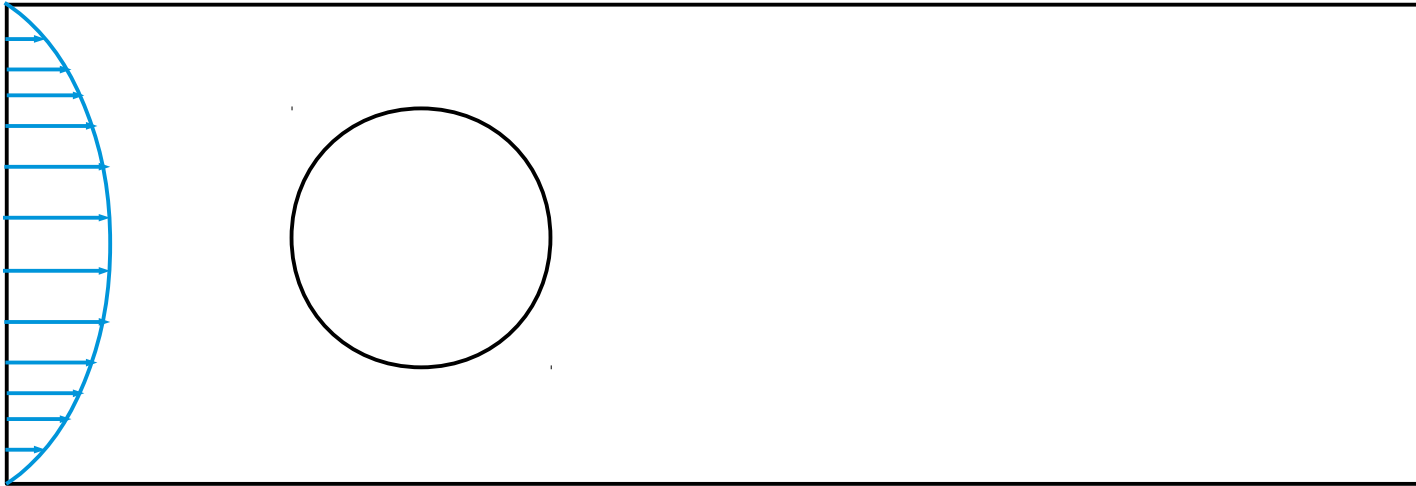
Are these domains symmetric?



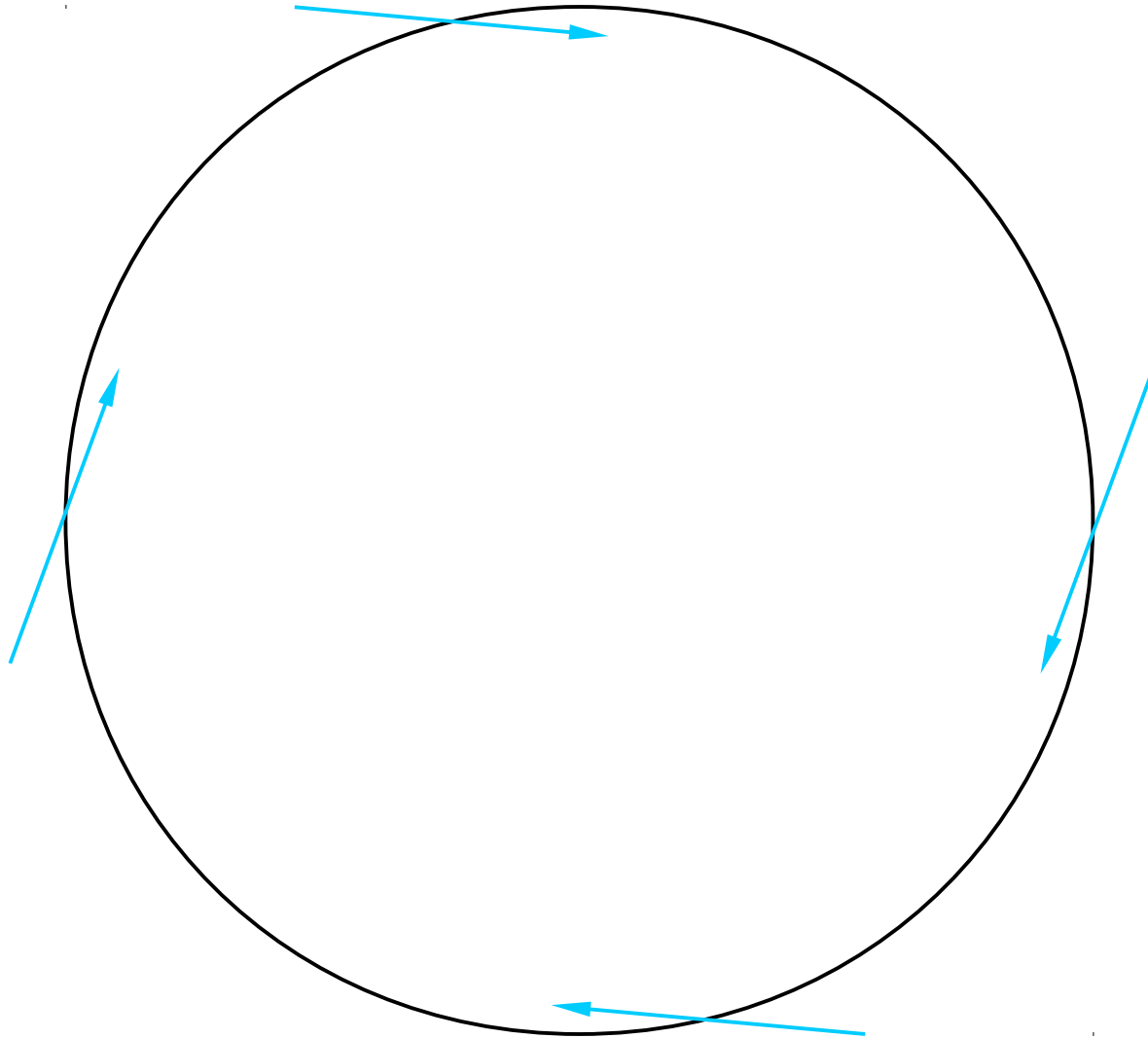
Are these domains symmetric?



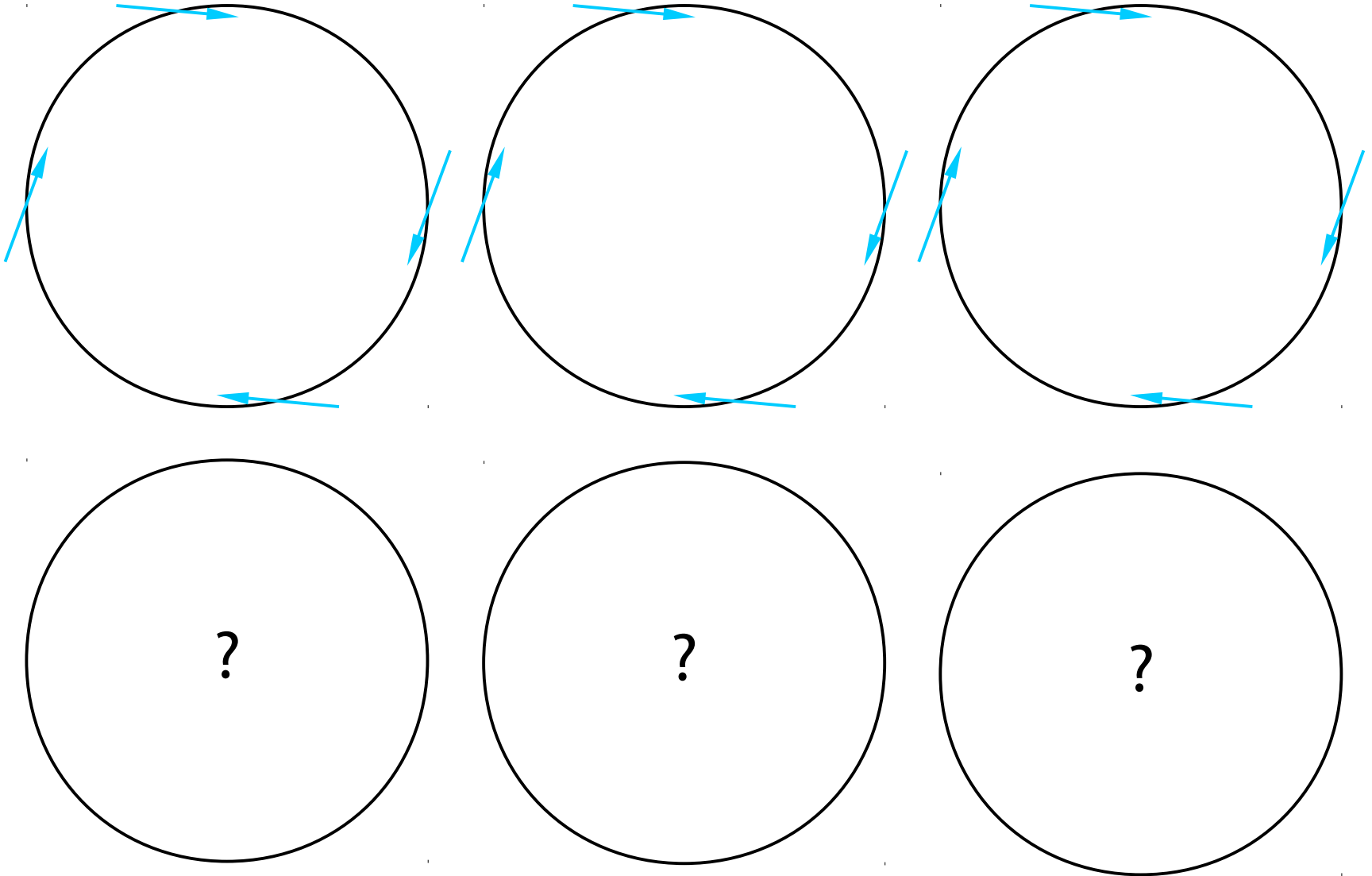
Are these domains symmetric?



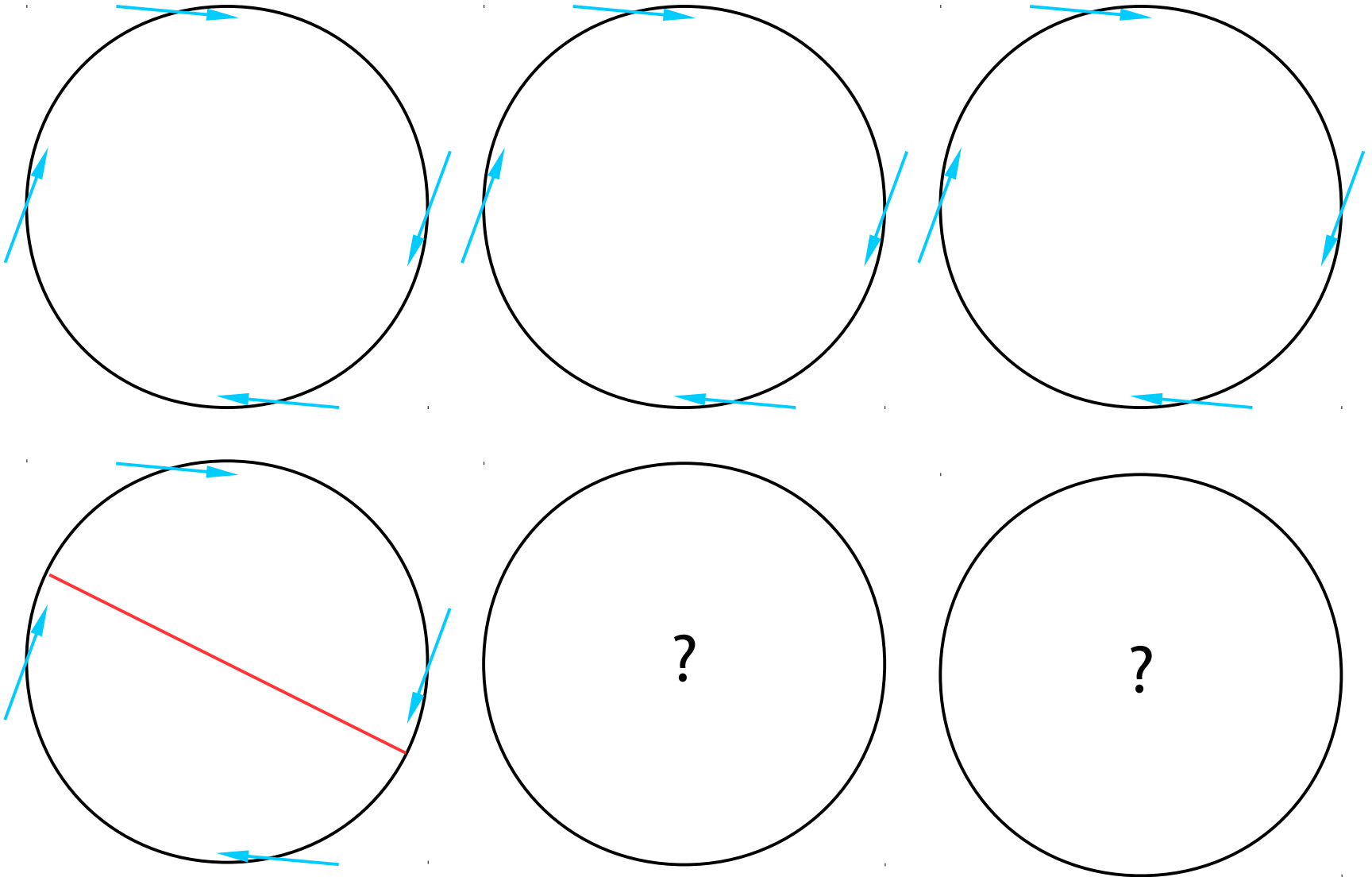
How do we symmetrize this domain?



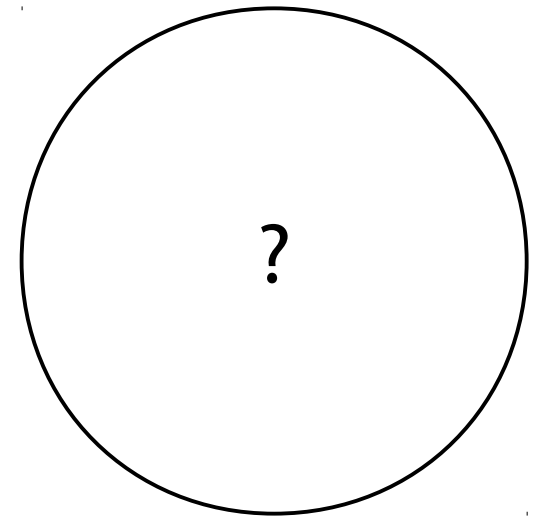
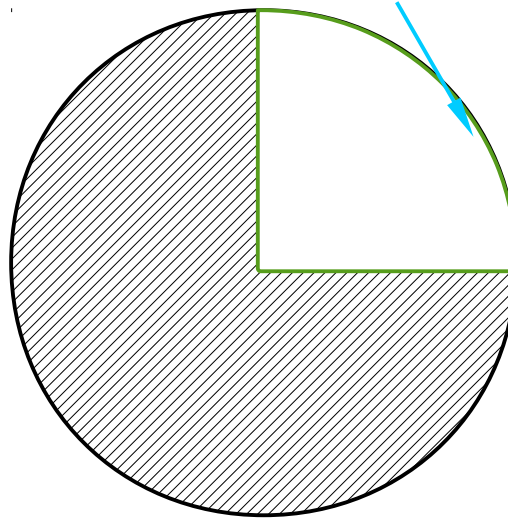
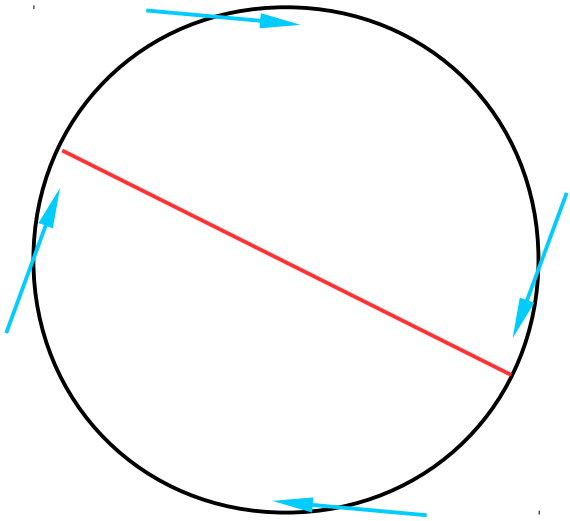
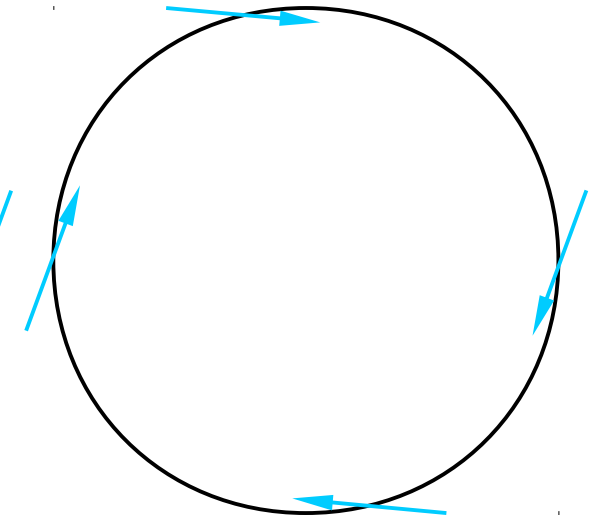
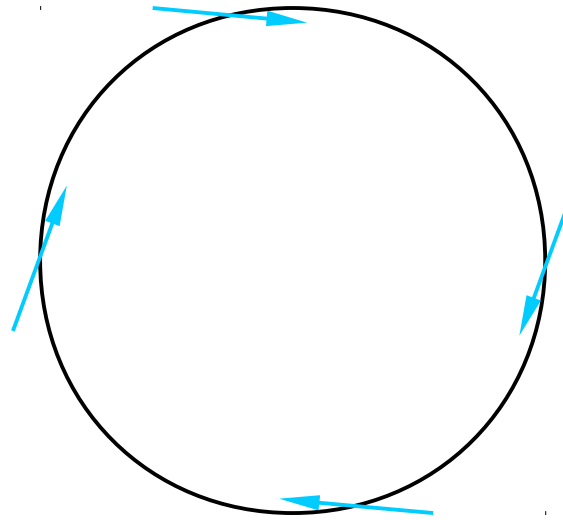
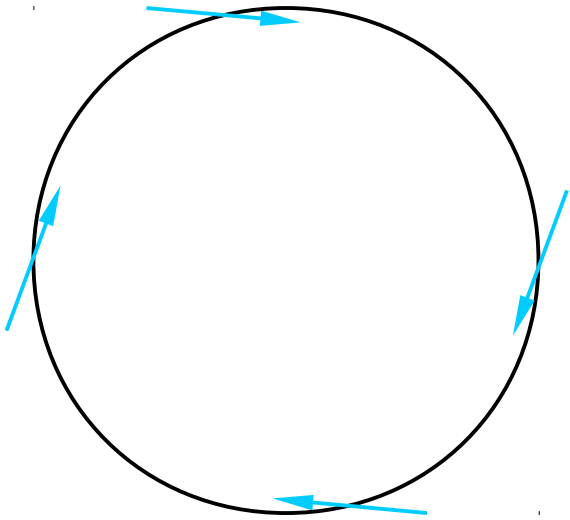
How do we symmetrize this domain?



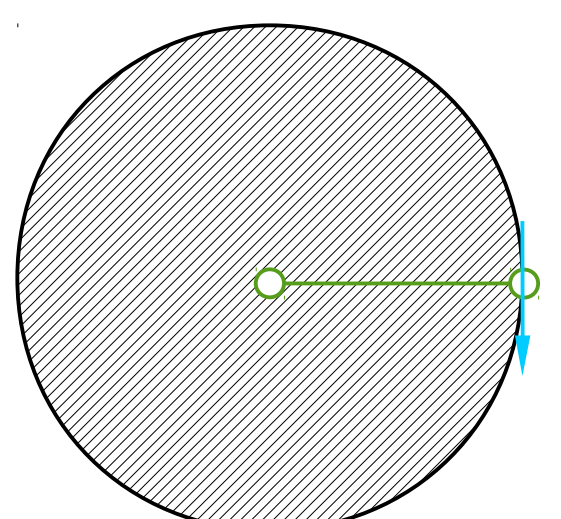
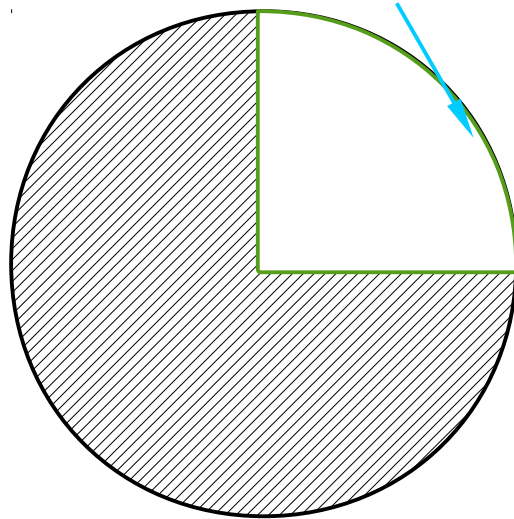
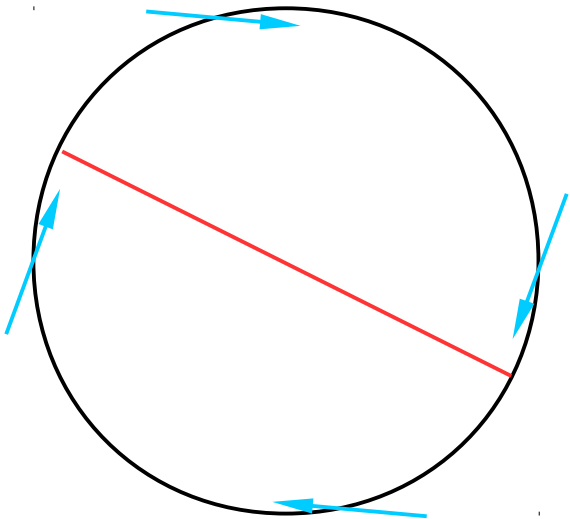
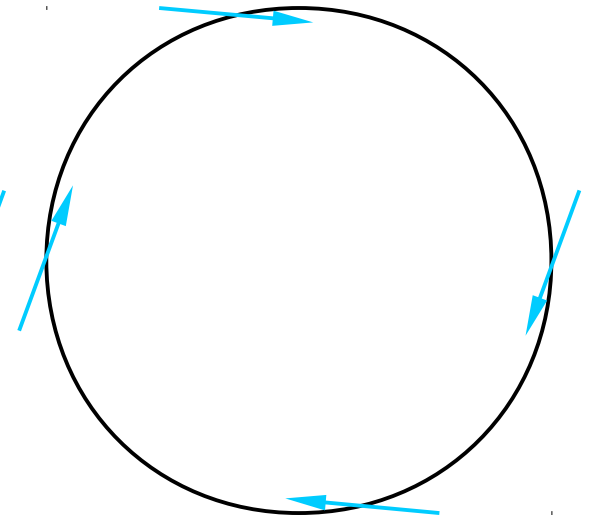
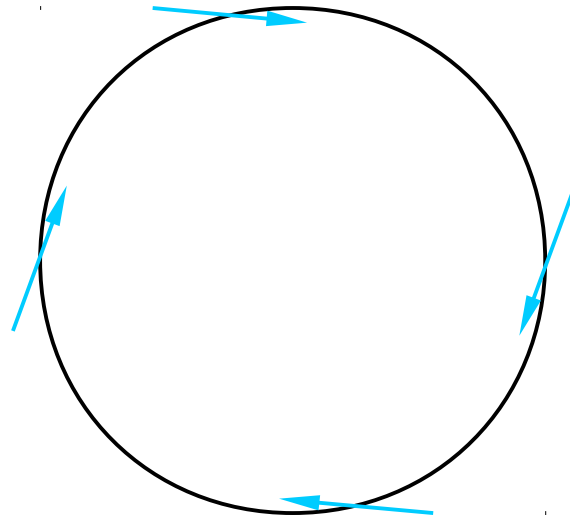
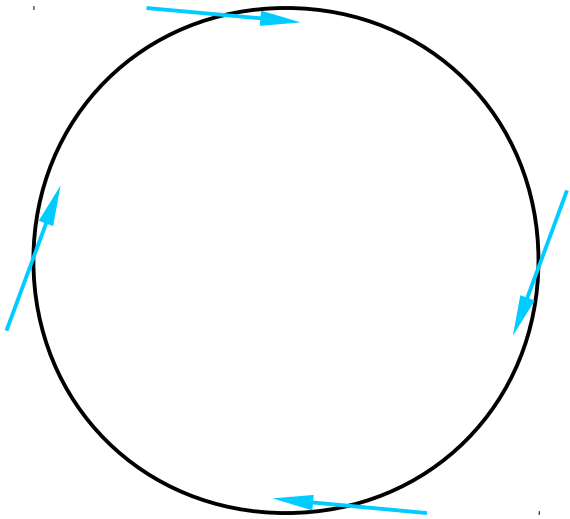
How do we symmetrize this domain?



How do we symmetrize this domain?



How do we symmetrize this domain?



UDF C Macro Language

- DEFINE_PROFILE—specify profile condition on boundary face zone
- DEFINE_TRANSIENT_PROFILE—specify profile condition on boundary face zone as function of time
- DEFINE_PROPERTY—specify material property (density, viscosity, thermal conductivity, rate of strain, mixing laws, ...)
- DEFINE_SOURCE—specify custom source term (mass, momentum, energy, species mass fractions, radiation model, user-defined scalars); written for *single* cell

UDF: DEFINE_PROFILE

$$p(y) = 1.1 \times 10^5 - 0.1 \times 10^5 \left(\frac{y}{0.0745} \right)^2$$

```
/*  
  UDF for specifying steady-state parabolic pressure profile boundary  
  profile for a turbine vane  
  */
```

```
#include "udf.h"  
DEFINE_PROFILE(pressure_profile,t,i)  
{  
  real x[ND_ND];  
  real y;  
  face_t f;  
  begin_f_loop(f,t)  
  /* this will hold the position vector */  
  {  
    F_CENTROID(x,f,t);  
    y = x[1];  
    F_PROFILE(f,t,i) = 1.1e5 - y*y/(.0745*.0745)*0.1e5;  
  }  
  end_f_loop(f,t)  
}
```

UDF: DEFINE_PROPERTY

```

/*****
  UDF that simulates solidification by specifying a temperature-
  dependent viscosity property
  *****/
#include "udf.h"
DEFINE_PROPERTY(cell_viscosity,c,t)
{
  real mu_lam;
  real temp = C_T(c,t);
  if (temp > 288.)
    mu_lam = 5.5e-3;
  else if (temp > 286.)
    mu_lam = 143.2135 - 0.49725 * temp;
  else
    mu_lam = 1.;
  return mu_lam;
}

```

UDF: DEFINE_SOURCE

$$source = -0.5 C_2 \rho y |v_x| v_x$$

```

/*****
  UDF for specifying an x-momentum source term in a spatially
  dependent porous media
  *****/
#include "udf.h"
#define C2 100.0
DEFINE_SOURCE(xmom_source,c,t,dS,eqn)
{
  real x[ND_ND];
  real con, source;
  C_CENTROID(x,c,t);
  con = C2*0.5*C_R(c,t)*x[1];
  source = -con*fabs(C_U(c, t))*C_U(c,t);
  dS[eqn] = -2.*con*fabs(C_U(c,t));
  return source;
}

```