

# Boundaries and Boundary Conditions



# CONVERGE Studio Workflow

- **Case Setup module**

- Begin a project
- Import the surface geometry
- Prepare the surface
- **Configure case setup**
  - **Boundary conditions and region definitions**
  - Initialization
  - Grid control
  - Physical models (turbulence, spray, combustion, sources, CHT, VOF, etc.)
  - Advanced options
- Export input and data files to the Case Directory

-----Run CONVERGE simulation-----

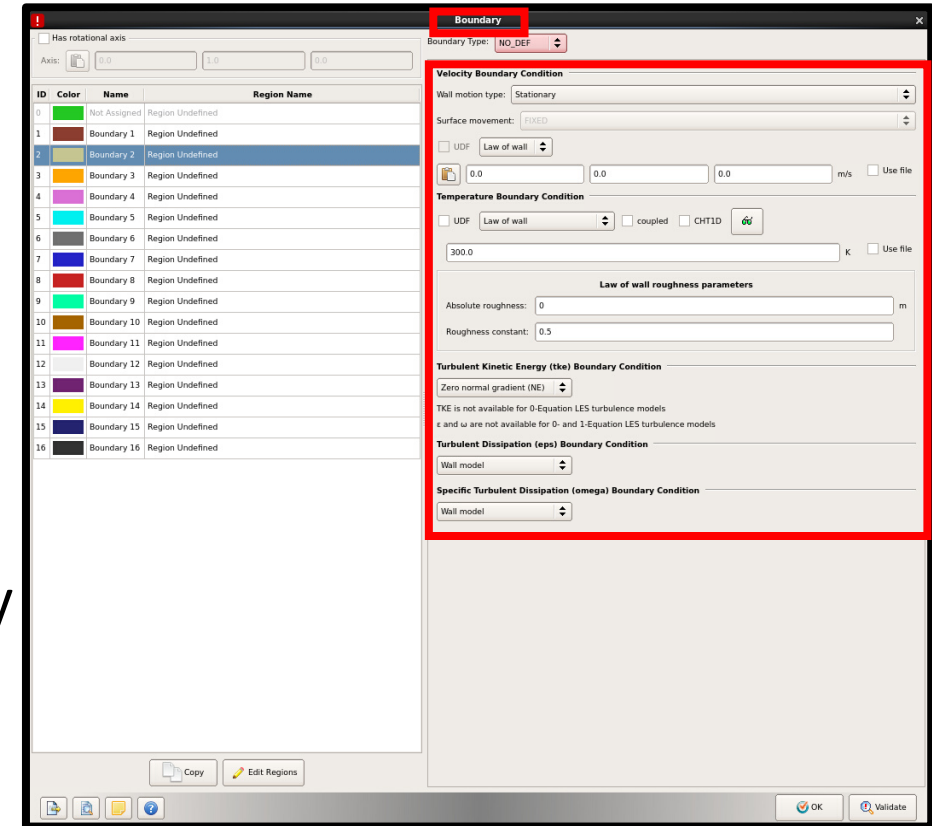
- *Line Plotting module*
- *Post-Processing 3D module*

# Boundary Conditions

- To solve the governing transport equations, a boundary condition for each equation must be specified
- A boundary condition applies constraints to a differential equation when solving for a unique solution
- To set a boundary condition, specify the location and constraints of the boundary
- Improperly defined boundary conditions can negatively impact your simulation results

# Boundary Type

- All flagged boundaries must have a *Boundary Type*
- CONVERGE offers the following boundary types
  - INFLOW
  - SYMMETRY
  - OUTFLOW
  - TWO\_D
  - WALL
  - GT-SUITE
  - PERIODIC
  - INTERFACE
- After you select the boundary type, CONVERGE may require the following parameters to be defined
  - Pressure, temperature, velocity, species mass fraction, passive fraction, and/or turbulence



# CONVERGE Boundary Conditions

- CONVERGE offers two ways to express the parameters ( $\phi$ ) as boundary conditions

- Dirichlet (specified value)

$$\phi = f$$

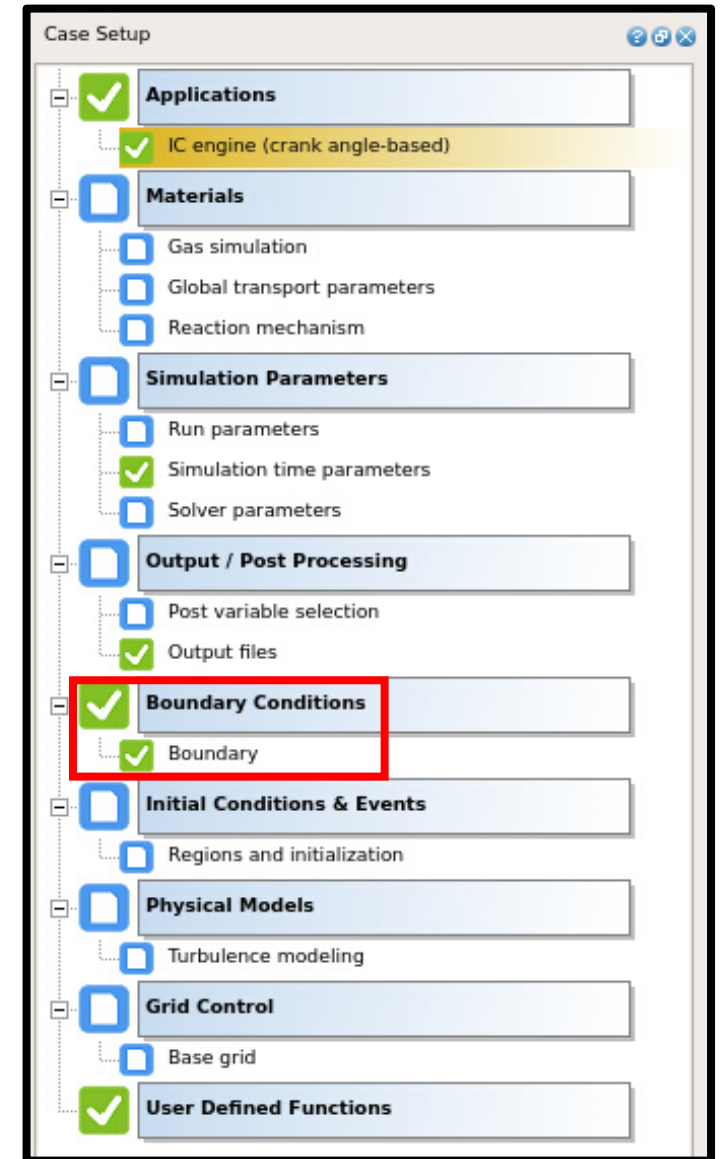
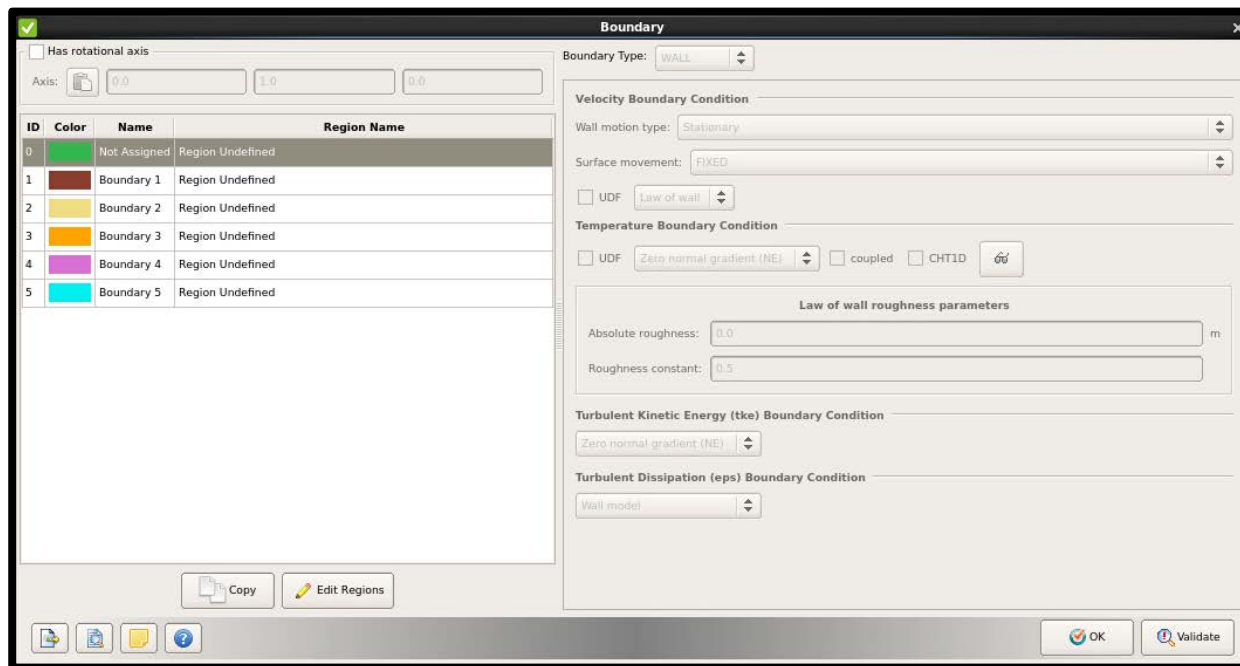
- Neumann (specified first derivative)

$$\frac{\partial \phi}{\partial x} = f$$

- Additional boundary conditions are available but they are special cases of Dirichlet or Neumann (e.g., slip, law-of-the-wall, etc.)

# Setting Up Boundary Conditions

- To set up the boundary conditions for a simulation, go to *Case Setup* > *Boundary Conditions* > *Boundary*

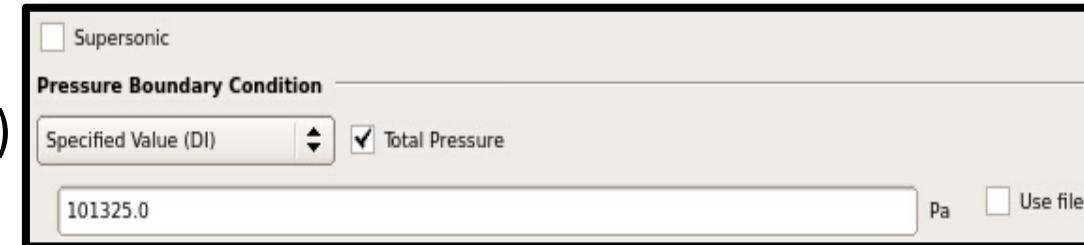


# INFLOW and OUTFLOW Boundary Types

- An INFLOW or OUTFLOW boundary specifies the flow going into or out of the computational domain, respectively
- An INFLOW/OUTFLOW boundary requires you to specify the following parameters
  - Pressure, velocity, and temperature
  - If solving for species or passives, you must specify the species or passive fractions
  - If using a turbulence model, you must specify turbulent kinetic energy and either turbulent dissipation rate or specific dissipation rate, depending on the model

# INFLOW/OUTFLOW: Pressure (1/3)

- There are three types of INFLOW/OUTFLOW pressure boundary conditions
  - Dirichlet (specified pressure value)
  - Zero normal Neumann (zero pressure gradient)
  - Transonic
- You may specify a static pressure or total pressure for the boundary
- For an INFLOW boundary, you can specify if the flow is subsonic or supersonic
  - For a subsonic flow, use the total Dirichlet pressure and Neumann velocity conditions  
OR the Neumann pressure and Dirichlet velocity conditions
  - For a supersonic flow, use the static Dirichlet pressure and Dirichlet velocity conditions



The screenshot shows a software interface for setting a pressure boundary condition. At the top, there is a checkbox labeled "Supersonic" which is currently unchecked. Below this, the title "Pressure Boundary Condition" is displayed. Under the title, there is a dropdown menu set to "Specified Value (DI)" and a checked checkbox labeled "Total Pressure". Below these options is a text input field containing the value "101325.0". To the right of the input field is the unit "Pa" and a checkbox labeled "Use file" which is unchecked.



# INFLOW/OUTFLOW: Pressure (2/3)

- For a Dirichlet OUTFLOW pressure boundary condition, enter a distance (*presdist*) downstream from the boundary pressure ( $P_{static}$ ) to a known user-specified pressure ( $P_{specified}$ )
  - Using the pressure at the cell nearest to the boundary ( $P_{fluid, cell}$ ), CONVERGE calculates  $P_{static}$
  - Using *presdist* will help dampen reflecting pressure waves in the domain by reducing the difference between  $P_{fluid, cell}$  and  $P_{static}$

$$P_{static} = \left( \frac{1}{1 + presdist} \right) P_{specified} + \left( \frac{presdist}{1 + presdist} \right) P_{fluid, cell}$$

# INFLOW/OUTFLOW: Pressure (3/3)

- For the transonic OUTFLOW boundary condition
  - Specify minimum and maximum Mach numbers ( $M_{min}$  and  $M_{max}$ ), which dictate the transition from subsonic to supersonic flow
  - CONVERGE will automatically adapt the static pressure ( $P_{static}$ ) calculation at the outlet based on the local Mach number

$$P_{static} = \begin{cases} P_{static} & M \leq M_{min} \\ \left(\frac{M_{max} - M}{M_{max}}\right) P_{static} + \left(\frac{M}{M_{max}}\right) P_{fluid,node} & M_{min} < M < M_{max} \\ P_{fluid,node} & M \geq M_{max} \end{cases}$$

# INFLOW/OUTFLOW: Velocity (1/5)

- Choose one of seven types of INFLOW/OUTFLOW boundary conditions available for the velocity equation
  - Dirichlet (specified velocity value)
  - Zero normal Neumann (zero velocity gradient)
  - Specified mass flow rate (special Dirichlet condition)
  - Pump flow rate (special Dirichlet condition)
  - Normal Neumann (special Neumann condition)
    - This condition forces the velocity direction to be normal to the boundary surface
    - Only for INFLOW boundaries
  - Average velocity
  - UDF

# INFLOW/OUTFLOW: Velocity (2/5)

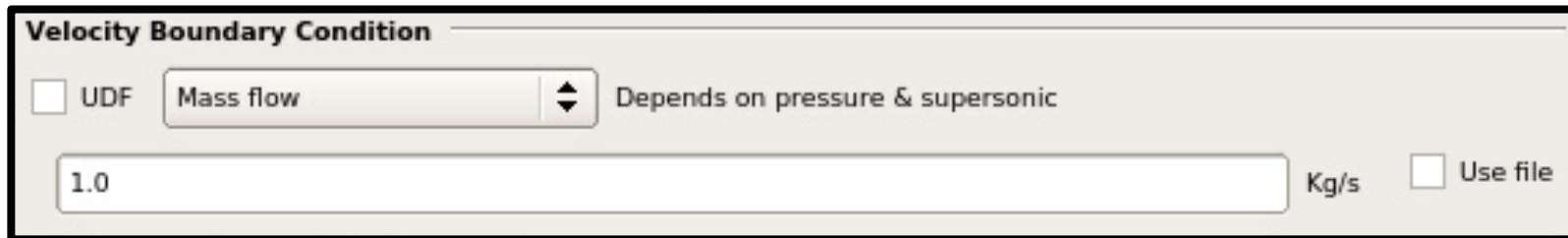
- Dirichlet
  - Enforces the three velocity components at the boundary location
- Zero normal Neumann
  - Enforces a zero velocity gradient at the boundary

The screenshot shows the 'Boundary' dialog box in ANSYS Fluent. The 'Boundary Type' is set to 'INFLOW'. The 'Velocity Boundary Condition' section is highlighted with a red box. It shows the 'Specified Value (DI)' option selected, with three input fields for the velocity components (U, V, W) all set to 0.0 m/s. The 'Pressure Boundary Condition' section shows 'Zero normal gradient (NE)' selected. The 'Temperature Boundary Condition' section shows 'Specified Value (DI)' selected with a value of 0.0 K. The 'Species Boundary Condition' section shows 'Specified Value (DI)' selected with a mass fraction sum of 0.0000. The 'Passive Boundary Condition' section shows 'Specified Value (DI)' selected. The 'Turbulent Kinetic Energy (tke) Boundary Condition' section shows 'Intensity' selected with a value of 0.02. The 'Turbulent Dissipation (eps) Boundary Condition' section shows 'Length scale' selected with a value of 0.003 m. The 'Has rotational axis' checkbox is unchecked. The 'Axis' fields are set to 0.0, 1.0, and 0.0. The 'ID' table lists boundaries 0 through 5, with boundary 1 highlighted. The 'Copy' and 'Edit Regions' buttons are at the bottom left. The 'OK' and 'Validate' buttons are at the bottom right.

ID	Color	Name	Region Name
0	Green	Not Assigned	Region Undefined
1	WAL	Boundary 1	Region Undefined
2	Yellow	Boundary 2	Region Undefined
3	Orange	Boundary 3	Region Undefined
4	Purple	Boundary 4	Region Undefined
5	Cyan	Boundary 5	Region Undefined

# INFLOW/OUTFLOW: Velocity (3/5)

- Mass flow rate
  - Sets the velocity ( $u_i$ ) for all cells at the INFLOW/OUTFLOW boundary based on the user-specified mass flow rate



$$u_i = - \frac{Mass\ flow}{\rho_{ave} A} n_i$$

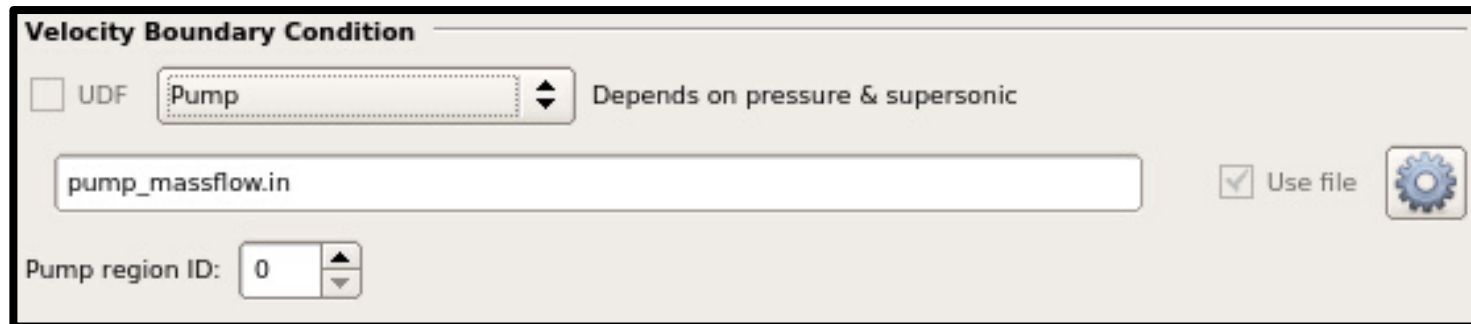
$\rho_{ave}$  is the average density at the boundary surface

$A$  is the total surface area

$n_i$  is the outward-pointing surface normal

# INFLOW/OUTFLOW: Velocity (4/5)

- Pump boundary condition
  - Sets the mass flow based on the pressure, as specified in a user-provided file (*e.g.*, *pump\_massflow.in*)
  - The pressure values in this file correspond to the average pressure of the region in which the boundary lies
  - List the region ID of the pump boundary



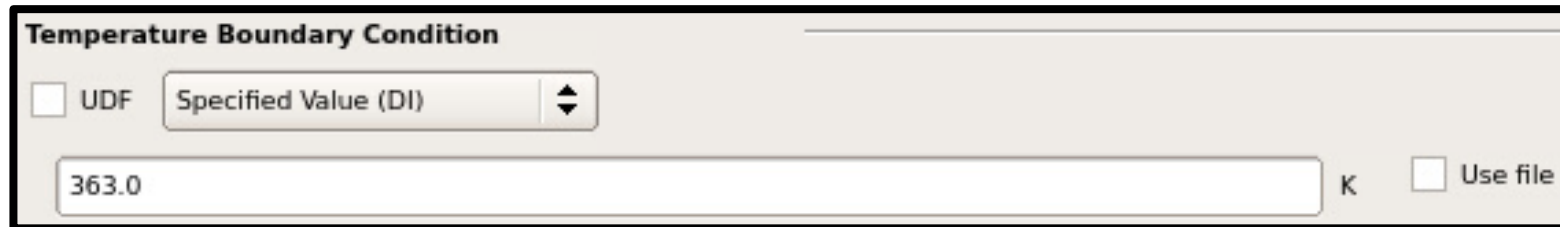
The screenshot shows the 'Velocity Boundary Condition' dialog box. It features a 'UDF' checkbox which is unchecked. Next to it is a dropdown menu currently set to 'Pump'. To the right of the dropdown is the text 'Depends on pressure & supersonic'. Below these elements is a text input field containing 'pump\_massflow.in'. To the right of this field is a checked 'Use file' checkbox and a gear icon. At the bottom left, there is a label 'Pump region ID:' followed by a spinner box showing the value '0'.

# INFLOW/OUTFLOW: Velocity (5/5)

- Normal Neumann
  - Ensures the velocity direction is normal to the INFLOW boundary
  - Only for INFLOW boundaries
- Average velocity
  - Ensures the velocity profile starts uniformly and then transiently develops throughout the simulation
  - Conserves average velocity
- UDF
  - Sets velocity profile based on user-specified file
  - Allows complex velocity boundary conditions

# INFLOW/OUTFLOW: Temperature

- There are two types of INFLOW/OUTFLOW temperature boundary conditions
  - Dirichlet (specified temperature value)
  - Zero normal Neumann (zero temperature gradient)
- For complex temperature boundary conditions, you may specify a UDF file
- We recommend a Dirichlet INFLOW condition and a Neumann OUTFLOW condition
- If you specify total pressure for the pressure boundary condition, CONVERGE assumes the temperature boundary condition is total temperature

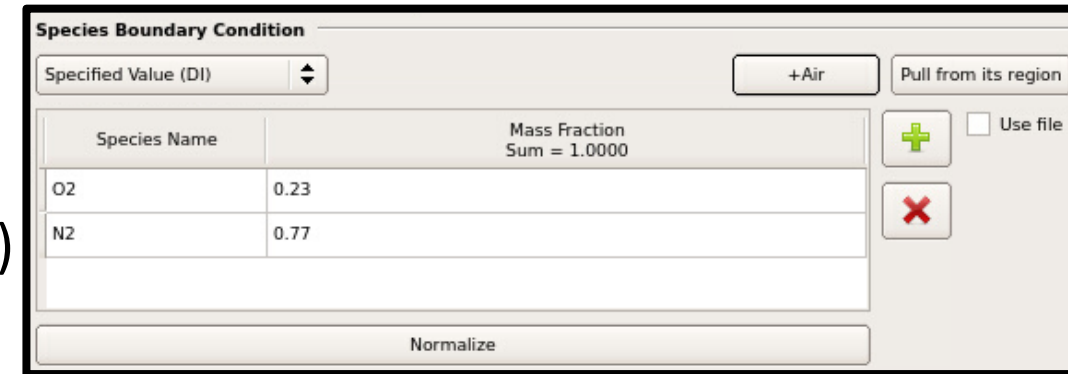


The screenshot shows a dialog box titled "Temperature Boundary Condition". It contains a checkbox labeled "UDF" which is unchecked. To its right is a dropdown menu currently showing "Specified Value (DI)". Below these is a text input field containing the value "363.0". To the right of the input field is a unit selector showing "K". Further to the right is another checkbox labeled "Use file", which is also unchecked.



# INFLOW/OUTFLOW: Species

- There are two types of species boundary conditions
  - Dirichlet (specified species value)
  - Zero normal Neumann (zero species gradient)
- Specify species mass fractions
  - You must import a reaction mechanism or define species in order for the species to be available for boundary conditions
  - Any species not set in the boundary condition will have a mass fraction of 0.0
  - Mass fractions must add up to 1.0
    - Use the Normalize button

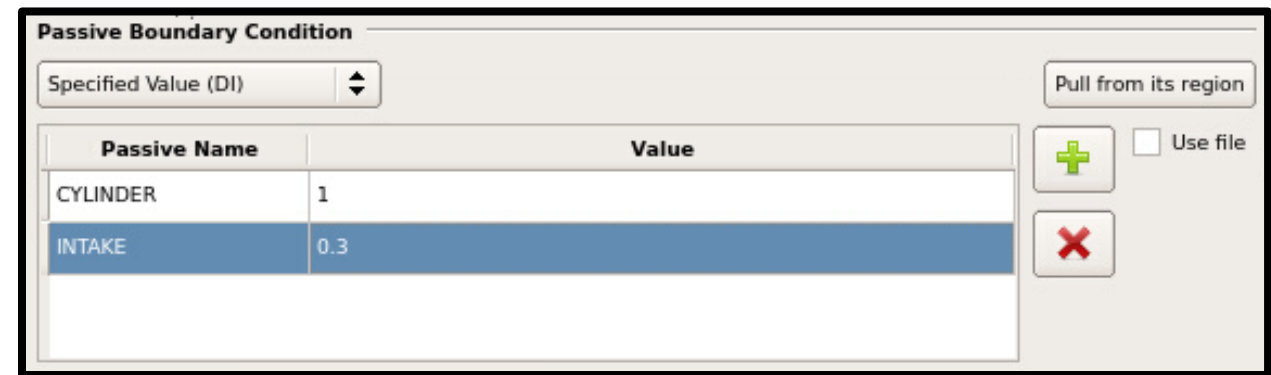


The screenshot shows the 'Species Boundary Condition' dialog box. At the top, there is a dropdown menu set to 'Specified Value (DI)'. To the right are buttons for '+Air' and 'Pull from its region'. Below these is a table with two columns: 'Species Name' and 'Mass Fraction'. The table contains two rows: 'O2' with a mass fraction of 0.23, and 'N2' with a mass fraction of 0.77. To the right of the table are a green '+' button, a red 'X' button, and a checkbox labeled 'Use file'. At the bottom of the dialog is a 'Normalize' button. The text 'Mass Fraction Sum = 1.0000' is displayed above the table.

Species Name	Mass Fraction
O2	0.23
N2	0.77

# INFLOW/OUTFLOW: Passive

- You can specify a tracer (“passive”) at a boundary and monitor its movement through the domain
  - The passive is defined in *Case Setup > Materials > Species > Passives*
  - Most simulations do not require passives; however, for some combustion models, passives are essential
- There are two types of INFLOW/OUTFLOW boundary conditions available for the passive equation
  - Dirichlet (specified passive value)
  - Zero normal Neumann (zero passive gradient)



The image shows a software dialog box titled "Passive Boundary Condition". At the top, there is a dropdown menu set to "Specified Value (DI)". Below this is a table with two columns: "Passive Name" and "Value". The table contains two entries: "CYLINDER" with a value of "1", and "INTAKE" with a value of "0.3". The "INTAKE" row is highlighted in blue. To the right of the table, there are two buttons: a green plus sign and a red minus sign. Above the plus sign is a button labeled "Pull from its region". Below the minus sign is a checkbox labeled "Use file".

Passive Name	Value
CYLINDER	1
INTAKE	0.3

# INFLOW/OUTFLOW: Turbulent Kinetic Energy (1/2)

- If using a turbulence model, you must specify a boundary condition for turbulent kinetic energy (tke)
- There are three types of INFLOW/OUTFLOW boundary conditions for tke
  - Dirichlet (specified tke value)
  - Zero normal Neumann (zero tke gradient)
    - Only for OUTFLOW boundaries
  - Turbulent intensity
    - This option is a special case of the Dirichlet boundary condition

Turbulent Kinetic Energy (tke) Boundary Condition

Specified Value (DI) ▾

1.0  $\text{m}^2/\text{s}^2$  ☐ Use file

Turbulent Kinetic Energy (tke) Backflow

Zero normal gradient (NE) ▾

Turbulent Kinetic Energy (tke) Boundary Condition

Intensity ▾

0.02| fraction ☐ Use file

# INFLOW/OUTFLOW: Turbulent Kinetic Energy (2/2)

- When using turbulence intensity, CONVERGE calculates tke ( $k$ ) from the turbulence intensity ( $I$ ) and local velocity ( $u$ ) via

$$k = \frac{3}{2} u_i^2 I^2$$

- Typical turbulence intensity values depend on Reynolds Number
  - High turbulence flow : 0.05 to 0.20 (*e.g.*, engines and turbines)
  - Medium turbulence flow: 0.01 to 0.05 (*e.g.*, low speed internal flow)
  - Low turbulence flow: < 0.01 (*e.g.*, low speed external flow)

# INFLOW/OUTFLOW: Turbulent Dissipation Rate (1/2)

- If using a k- $\epsilon$  turbulence model, there are three types of INFLOW/OUTFLOW boundary conditions for the turbulent dissipation rate ( $\epsilon$ )
  - Dirichlet (specified  $\epsilon$  value)
  - Zero normal Neumann (zero  $\epsilon$  gradient)
    - Only for OUTFLOW boundaries
  - Turbulent length scale
    - This option is a special case of the Dirichlet boundary condition

**Turbulent Dissipation ( $\epsilon$ ) Boundary Condition**

Specified Value (DI)   $\text{m}^2/\text{s}^3$  ☐ Use file

**Turbulent Dissipation ( $\epsilon$ ) Backflow**

Zero normal gradient (NE)

**Turbulent Dissipation ( $\epsilon$ ) Boundary Condition**

Length scale  m ☐ Use file

# INFLOW/OUTFLOW: Turbulent Dissipation Rate (2/2)

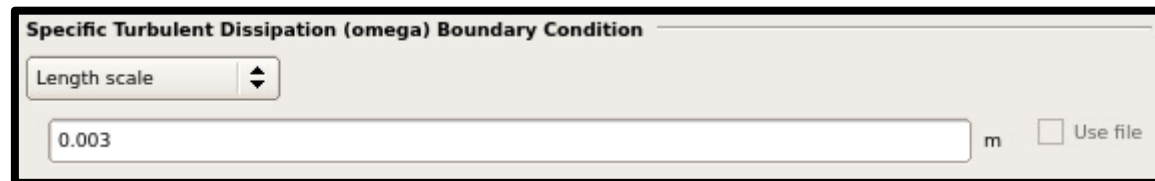
- When using the turbulent length scale boundary condition, CONVERGE calculates  $\epsilon$  from a modeling constant  $c_\mu$ , the  $(k)$ , and the user-specified *length scale*

$$\epsilon = \frac{c_\mu^{3/4} k^{3/2}}{\text{length scale}}$$

- $c_\mu$  is typically 0.09
- Typically the turbulent length scale depends on the hydraulic diameter
  - We recommend starting with a *length scale* that is less than 10% of the hydraulic diameter

# INFLOW/OUTFLOW: Specific Turbulent Dissipation (1/2)

- If using a k- $\omega$  turbulence model, there are three types of INFLOW/OUTFLOW boundary conditions for the specific turbulent dissipation ( $\omega$ )
  - Dirichlet (specified  $\omega$  value)
  - Zero normal Neumann (zero  $\omega$  gradient)
    - Only for OUTFLOW boundaries
  - Turbulent length scale
    - This option is a special case of the Dirichlet boundary condition



Specific Turbulent Dissipation ( $\omega$ ) Boundary Condition

Length scale

m ☐ Use file

# INFLOW/OUTFLOW: Specific Turbulent Dissipation (2/2)

- When using the turbulent length scale boundary condition, CONVERGE calculates omega ( $\omega$ ) from the turbulent modeling constant  $\beta_{star}$ , tke, and the user-specified *length scale*

$$\omega = \frac{k^{1/2}}{(\beta_{star})(length\ scale)}$$

- The tke, eps, and omega parameters are related with the turbulent modeling constant ( $\beta_{star}$ ) via the following relationship

$$\omega = \frac{\varepsilon}{(\beta_{star})k}$$



# OUTFLOW: Backflow


- INFLOW and OUTFLOW boundaries are similar with the main exception being that OUTFLOW boundaries require input for reverse flow (*i.e.*, flow back into the domain)
- Specify realistic reverse flow inputs to ensure convergence
- The boundary conditions can be either Dirichlet or zero normal Neumann
- You must define backflow boundary conditions for the following
  - Temperature
  - Species
  - Passive
  - Turbulence (only if a turbulence model is active)

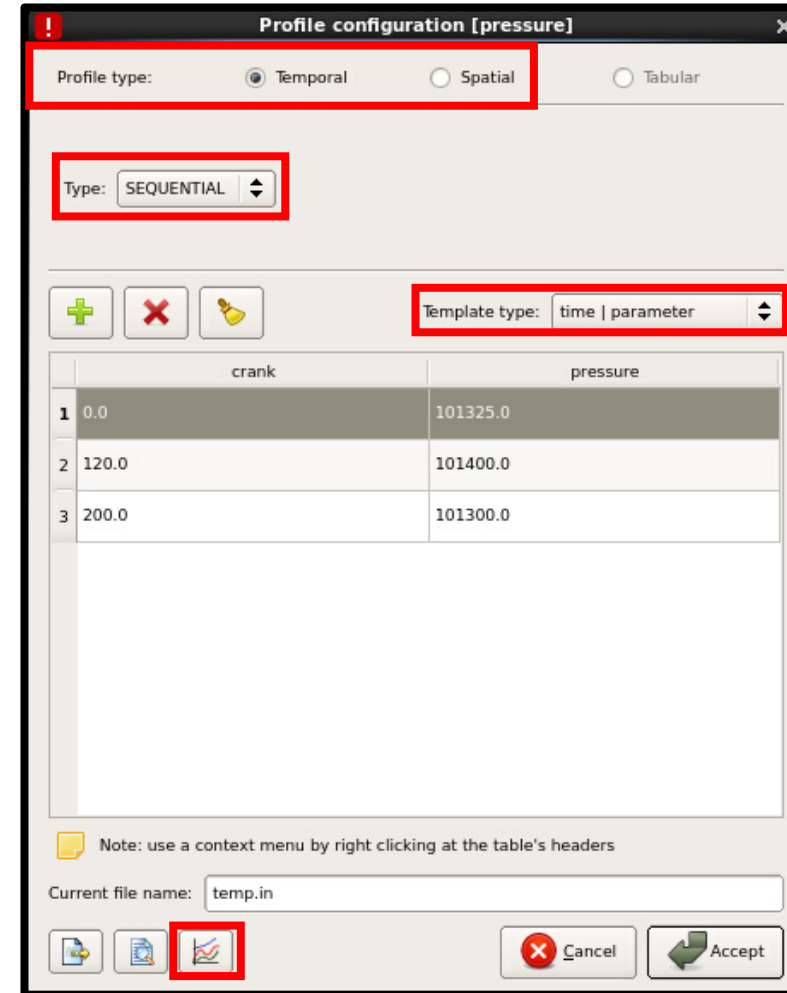
# INFLOW/OUTFLOW: Varying Boundary Conditions (1/2)

- For an INFLOW/OUTFLOW boundary, you can temporally and/or spatially vary any of the previously described boundary conditions by clicking on the Use file box and then the gear button
  - Manually enter values using CONVERGE Studio or
  - Import a \*.in file
- For transient simulations, boundary conditions can vary temporally and/or spatially
- For steady-state simulations, boundary conditions can only vary spatially



# INFLOW/OUTFLOW: Varying Boundary Conditions (2/2)

- If you choose temporal variation, specify the type as SEQUENTIAL or CYCLIC
- If you choose spatial variation, specify the type of movement as a scale, translation, or rotation
- The Template Type menu offers the suggested format for the tabulated variation
- If your tabulated variation is in a two-column format, you can plot the variation by clicking the  button



Profile configuration [pressure]

Profile type: ☒ Temporal ☐ Spatial ☐ Tabular


Type: SEQUENTIAL

Template type: time | parameter

	crank	pressure
1	0.0	101325.0
2	120.0	101400.0
3	200.0	101300.0

Note: use a context menu by right clicking at the table's headers

Current file name: temp.in

Buttons:  Cancel Accept

# How to Set Up Both Temporal and Spatial Variation

- 1) Click the Spatial profile type
- 2) Specify the spatial variation
- 3) Select the [time | vector XYZ] template type
- 4) Right click on the column header and insert a column for the parameter name (*e.g.*, temperature)
- 5) Fill in the table manually or by importing a file
- 6) Click Accept

Profile configuration [temperature]

Profile type: ☐ Temporal ☒ Spatial ☐ Tabular

Scale: 1.0 Translation X/Y/Z: 0.0 0.0 0.0

Rotation axis/angle: x 30.0 deg.

Template type: time | vector XYZ

	crank	temperature	x	y	z
1	100.0	350.0	.5	.5	0.0
2	200.0	360.0	1.0	0.0	1.0
3	300.0	400.0	0.0	.3	.1

Note: use a context menu by right clicking at the table's headers

Current file name:

Cancel Accept

# WALL Boundary Type

- A WALL boundary prohibits flow through it
- A WALL boundary requires you to specify the following parameters
  - Velocity, temperature, and roughness
  - If using a turbulence model, you must also specify turbulent kinetic energy and either turbulent dissipation rate or specific dissipation rate

# WALL: Velocity (1/5)



- There are two components to the WALL velocity boundary condition
  - Wall motion type: this is the boundary condition
  - Surface movement: this is the physical movement of the surface triangles
- WALL motion type can be Stationary, Translating, Rotating, Rotating and Translating, Arbitrary, Dependent, FSI, or User-specified
  - For each, specify Law-of-the-wall, slip (a special Neumann condition), No-slip (a special Dirichlet condition), or Dirichlet (specified velocity value)
- Surface movement can be FIXED or MOVING

# WALL: Velocity (2/5)

- Stationary
  - Specifies a fixed, non-moving wall boundary
- Translating
  - Imposes a translational motion
    - This can be user-specified or calculated as piston motion
  - If MOVING, the boundary moves according to a user-specified velocity vector and influences the adjacent fluid
  - If FIXED, the boundary remains motionless while CONVERGE applies a non-zero velocity boundary condition to mimic the surface translation

# WALL: Velocity (3/5)

- Rotating
  - Imposes a rotational motion
  - If MOVING, the boundary rotates according to a user-specified rotation rate, center of rotation, and axis of rotation
  - If FIXED, the boundary remains motionless while CONVERGE applies a non-zero angular velocity condition to mimic the surface rotation
- Translating and Rotating
  - Imposes a translational and rotational motion
  - This assumes a MOVING surface where the boundary will translate and rotate according to user-specified quantities



# WALL: Velocity (4/5)

- Arbitrary
  - Specifies the position and orientation of the boundary according to a user-supplied input file
- Dependent
  - Forces each vertex on a boundary to move with the adjacent boundary that shares that vertex
  - Appropriate for simulations in which adjacent fluid cells can be in different regions at different times

# WALL: Velocity (5/5)

- FSI
  - Specifies a boundary whose movement depends on FSI forces (FSI model must be active)
- User
  - Specifies a boundary that follows a user-specified deformation of individual vertices

# WALL: Velocity: Piston Motion (1/4)

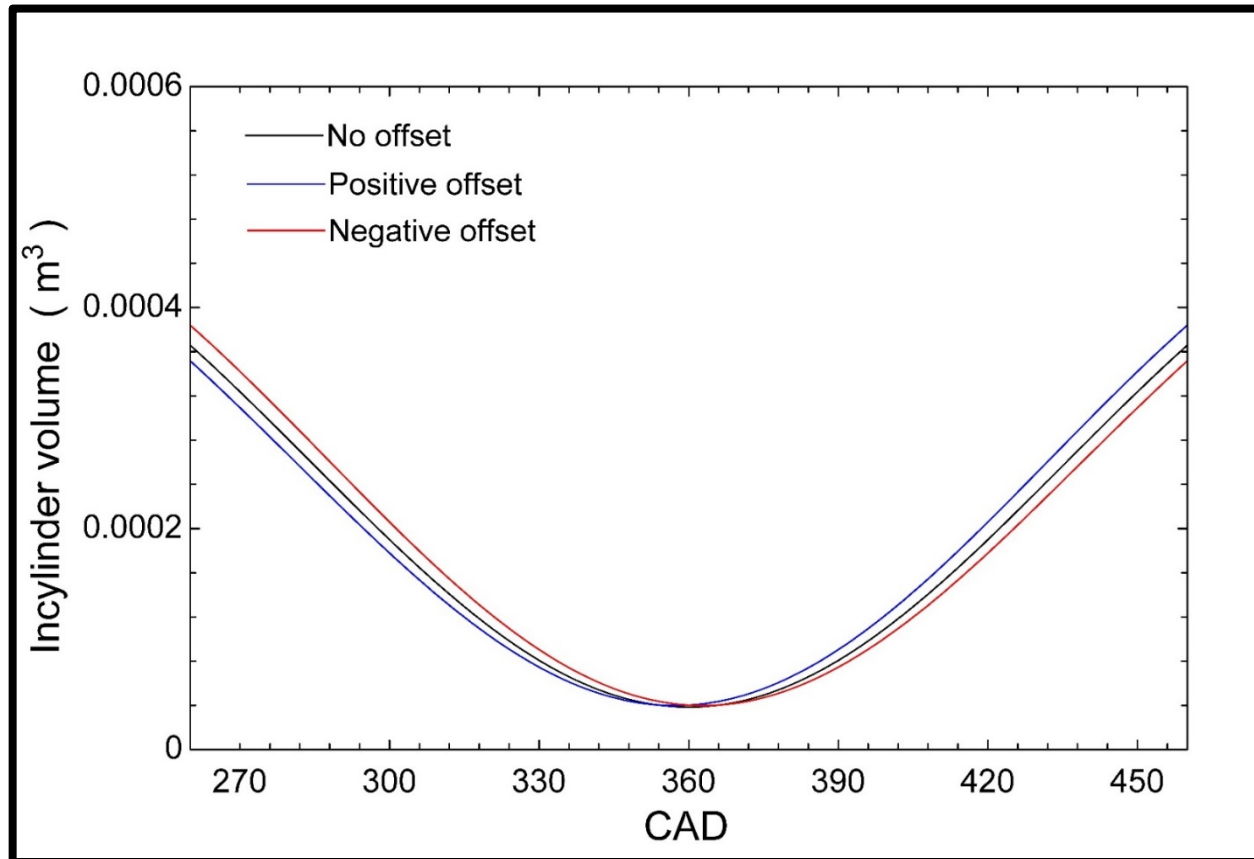
- To move a boundary with the same velocity and direction as the piston motion, click the Piston motion button
  - CONVERGE will use the engine parameters to internally generate position tables for the motion using a slider-crank mechanism
  - This option is only available for a Translating WALL boundary
- You can output the piston motion file
- You can set a phase-lag to offset the piston motion ( $z_{piston}$ )
$$z_{piston} = f(\theta + \phi)$$
  - Here  $f$  is the function describing piston position,  $\theta$  is the crank angle, and  $\phi$  is the phase-lag

The screenshot shows the 'Velocity Boundary Condition' dialog box. It has the following fields and options:

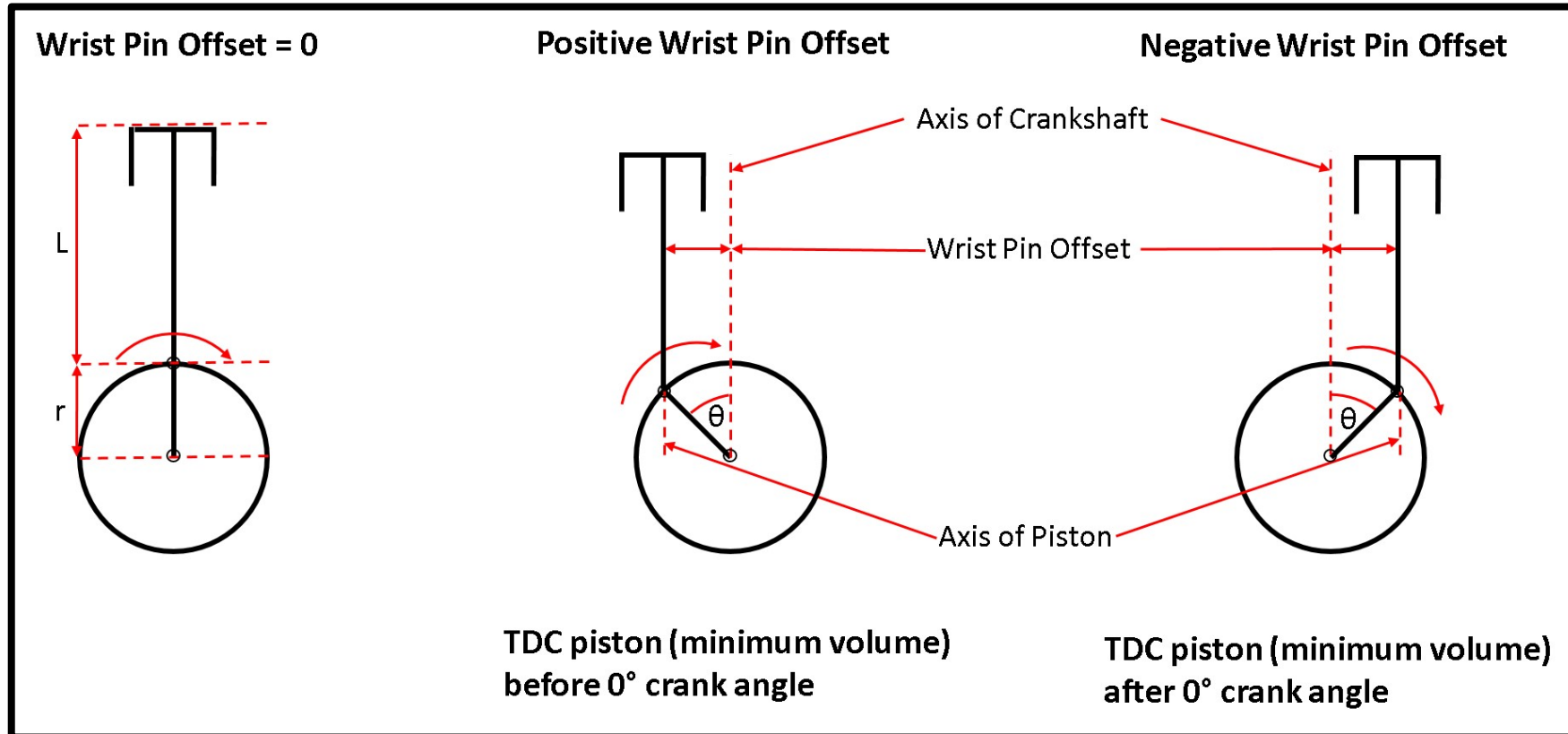
- Wall motion type: Translating
- Surface movement: MOVING
- UDF: ☐ (disabled), Law of wall (dropdown), ☐ User specify, ☒ Piston motion (highlighted with a red rectangle)
- Output piston motion file 'piston\_profile#.out': ☐
- $\Phi$ : 10 (text input field)

# WALL: Velocity: Piston Motion (2/4)

- If the engine has a wrist pin offset, specify it in *Case Setup > Engine application > IC Engine*
- Apply a non-zero wrist pin offset when the axes of the piston and crankshaft are not aligned
  - Otherwise, top dead center (TDC) occurs at 0 *crank angle degrees*



# WALL: Velocity: Piston Motion (3/4)



# WALL: Velocity: Piston Motion (4/4)

- CONVERGE calculates the piston position in the z direction ( $z_{piston}$ ) using the following formula

$$z_{piston} = 2a - l \left[ 1 - \cos \left\{ \sin^{-1} \left( \frac{a \sin \theta + \Gamma}{l} \right) \right\} \right] - a(1 - \cos \theta)$$

$a$  is the crank radius

$\Gamma$  is the wrist pin offset

$l$  is the connecting rod length

$\theta$  is the crank angle in *radians*

# WALL: Temperature (1/2)

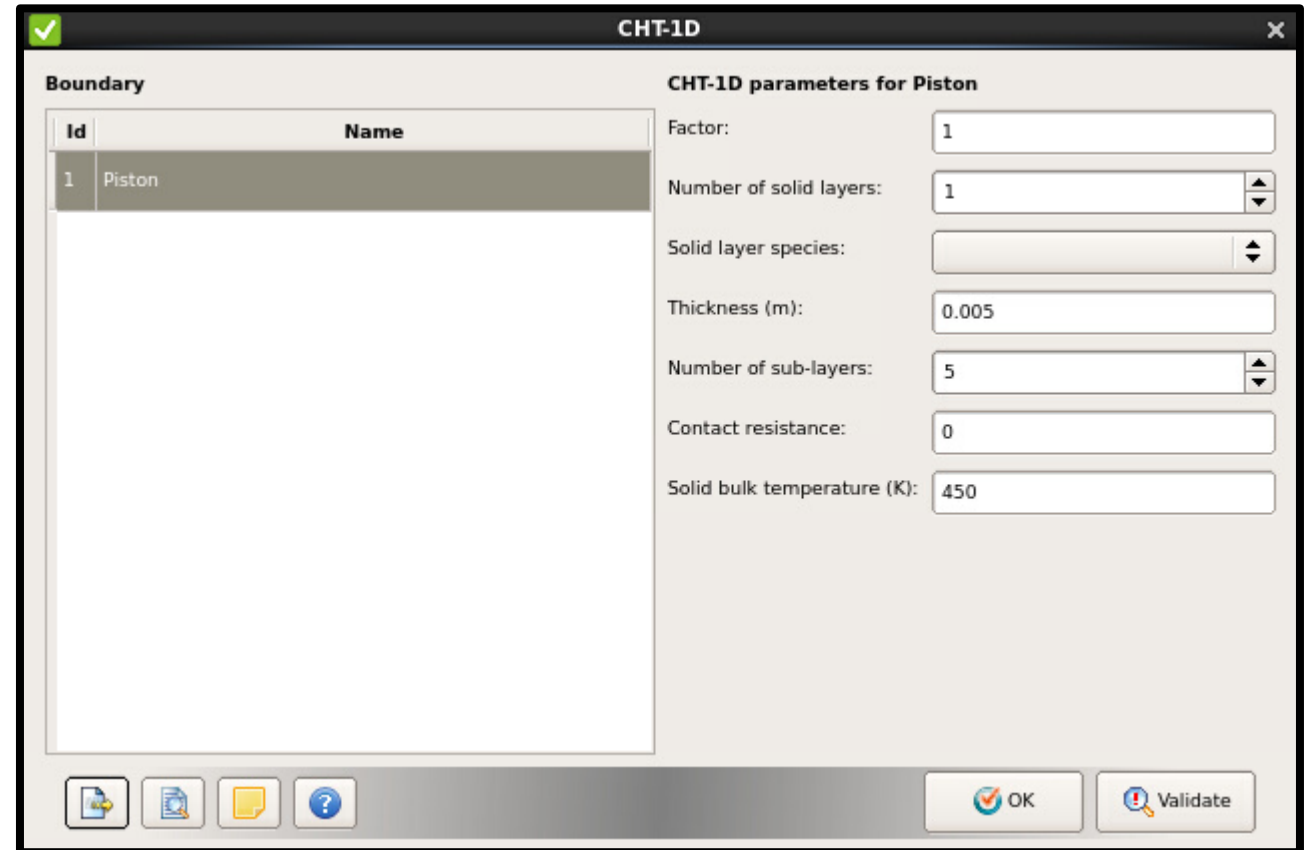
- There are five types of WALL temperature boundary conditions
  - Law-of-the-wall
  - Dirichlet (specified temperature value)
  - Zero normal Neumann (zero temperature gradient)
  - Heat flux
  - Convection
- CONVERGE calculates the heat transfer across interface boundaries if you check the coupled box
  - Check this option for both WALL boundaries associated with the interface
  - Enter in the contact resistance value
    - A value of 0.0 implies no contact resistance between two solids



The screenshot shows the 'Temperature Boundary Condition' dialog box. It features a title bar with the text 'Temperature Boundary Condition'. Below the title bar, there are several options: a checkbox for 'UDF' (unchecked), a dropdown menu for 'Specified Value (DI)' (selected), a checkbox for 'coupled' (unchecked), a checkbox for 'CHT1D' (unchecked), and a green icon with a plus sign. Below these options, there is a text input field containing the value '300.0', followed by a unit selector 'K'. At the bottom right, there is a checkbox for 'Use file' (unchecked).

# WALL: Temperature (2/2)

- The CHT1D boundary condition models heat transfer in thin solid layers between a fluid and a solid
  - This condition is only compatible with law-of-the-wall and Dirichlet temperature boundary conditions



The screenshot shows the 'CHT-1D' dialog box with a green checkmark icon in the top-left corner. The dialog is divided into two main sections: 'Boundary' on the left and 'CHT-1D parameters for Piston' on the right.

**Boundary Section:** A table with two columns, 'Id' and 'Name'. The first row is highlighted and contains the values '1' and 'Piston'.

Id	Name
1	Piston

**CHT-1D parameters for Piston Section:** A list of parameters with input fields:

- Factor: 1
- Number of solid layers: 1
- Solid layer species: (empty dropdown)
- Thickness (m): 0.005
- Number of sub-layers: 5
- Contact resistance: 0
- Solid bulk temperature (K): 450

At the bottom of the dialog, there are four icons on the left (a folder, a document, a yellow square, and a question mark) and two buttons on the right: 'OK' and 'Validate'.



# WALL: Roughness (1/2)

- Model the wall roughness effects with two roughness parameters
  - Absolute roughness
  - Roughness constant

Law of wall roughness parameters	
Absolute roughness:	<input type="text" value="4.5e-05"/> m
Roughness constant:	<input type="text" value="0.5"/>

- The default absolute roughness of 0.0 corresponds to a smooth wall
- The default roughness constant is 0.5
- This information assumes the roughness is constant across the entire surface

# Wall: Turbulence (1/2)

- If using a turbulence model, you must specify a boundary condition for turbulence parameters (tke and either eps or omega)
- There is only one type of tke boundary condition
  - Zero normal Neumann (zero tke gradient)
- There are two types of eps/omega boundary conditions
  - Zero normal Neumann (zero eps/omega gradient)
  - Wall model
    - This option is a special case of the Dirichlet boundary condition

# Wall: Turbulence (2/2)

- When using the turbulent models, CONVERGE calculates eps ( $\varepsilon$ ) from a modeling constant  $c_\mu$ , tke ( $k$ ), the distance from the wall to the middle of the cell ( $y$ ), and the Von Karmen's constant ( $\kappa$ )

$$\varepsilon = \frac{c_\mu^{3/4} k^{3/2}}{\kappa y}$$

- The tke, eps, and omega parameters are related with the turbulent modeling constant ( $\beta_{star}$ ) via the following relationship

$$\omega = \frac{\varepsilon}{(\beta_{star})k}$$

# WALL: Varying Boundary Condition

- For a WALL boundary, you can vary the temperature and/or velocity conditions by clicking on the Use file box and then the gear button
- You can vary the temperature boundary conditions both temporally and spatially
- For spatially varying temperature
  - Measure temperature at any *CAD*
  - The z location in the spatially varying temperature file must correspond to the piston at BDC
  - Start the simulation at any *CAD*
- You can only vary the velocity boundary conditions temporally

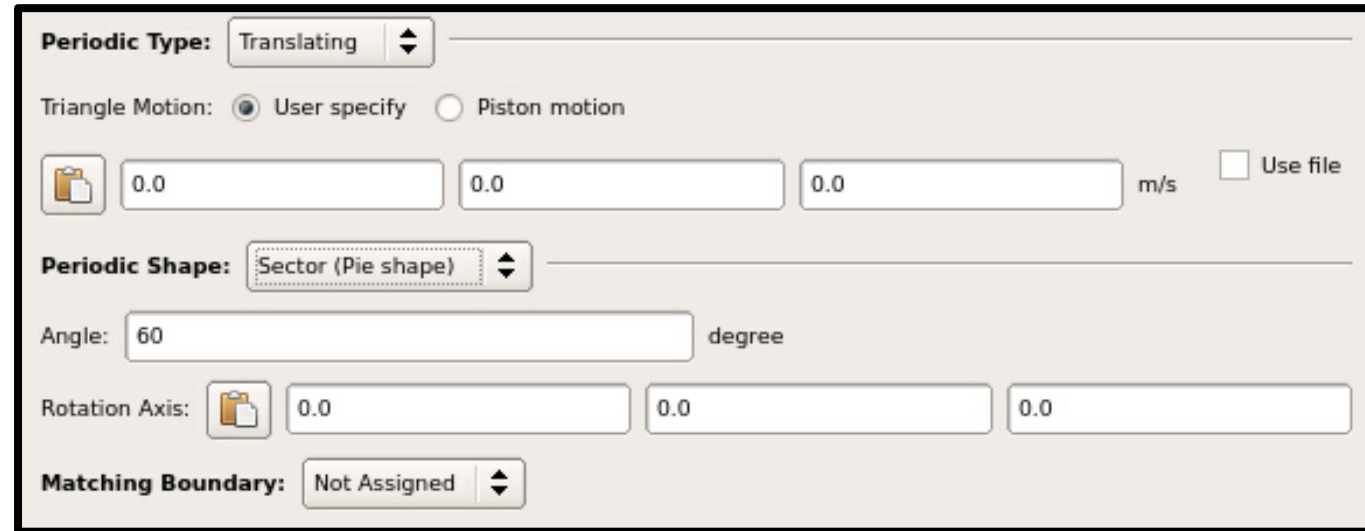


# PERIODIC Boundary Type (1/2)

- A PERIODIC boundary must occur with its pair
  - The two boundaries must be geometrically identical and planar
  - CONVERGE copies the values from the first PERIODIC boundary to the matching PERIODIC boundary
    - Choose the matching boundary from the drop-down menu
- Specify if the boundary is Stationary or Translating
  - If a boundary is Translating, vertices contained entirely within the boundary will move according to the specified velocity
- Specify the shape as either Sector (Pie shape) or Planar (Box shape)
  - If Sector, the PERIODIC boundaries must rotate about the z axis and be symmetric about the x axis

# PERIODIC Boundary Type (2/2)

- To specify a Sector shape, specify an angle of rotation and vector about which the first PERIODIC boundary would need to rotate to match its partner
  - The vector originates from the origin and the sign of the angle of rotation is determined from the right-hand rule
- To specify a Planar shape, specify the vector for which the first PERIODIC boundary would need to translate to match its partner

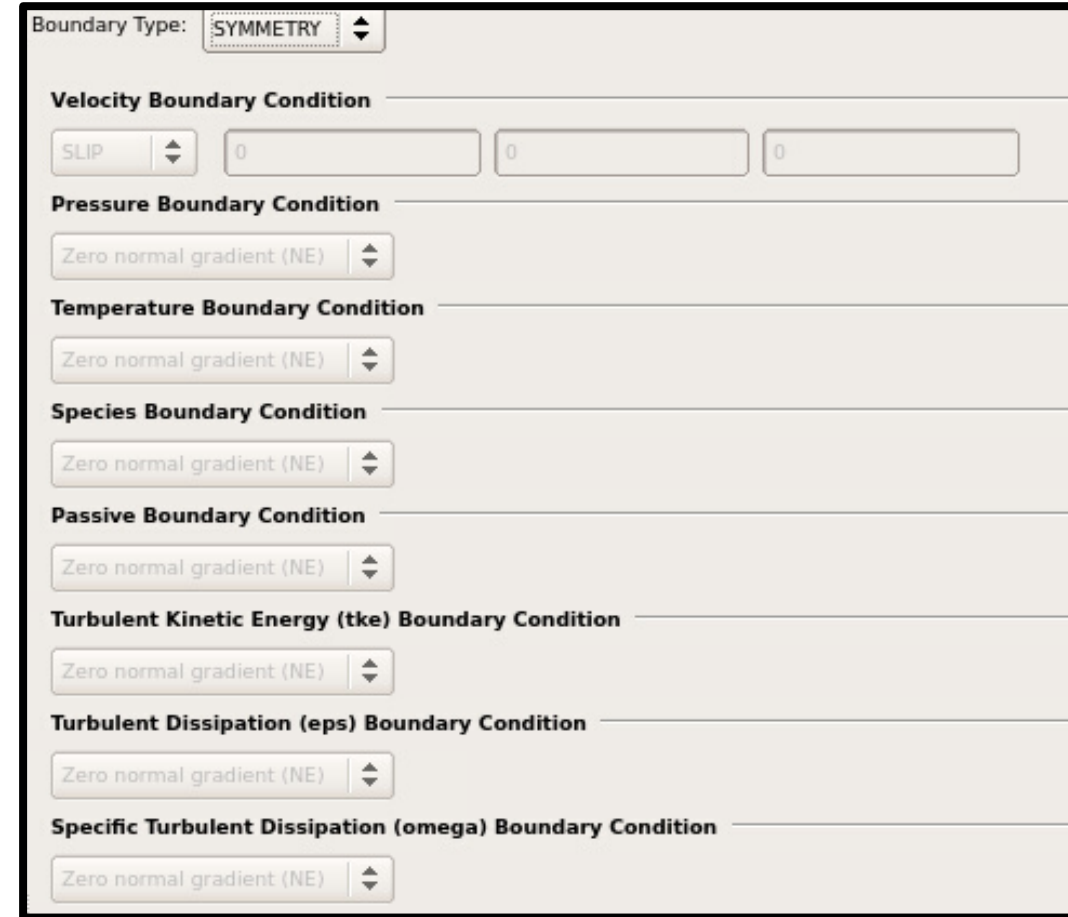


The screenshot shows the 'Periodic Type' dialog box in ANSYS Fluent. The 'Periodic Type' is set to 'Translating'. Under 'Triangle Motion', 'User specify' is selected. The velocity components are set to 0.0, 0.0, and 0.0 m/s, with a 'Use file' checkbox. The 'Periodic Shape' is set to 'Sector (Pie shape)'. The 'Angle' is set to 60 degree. The 'Rotation Axis' components are set to 0.0, 0.0, and 0.0. The 'Matching Boundary' is set to 'Not Assigned'.

Field	Value
Periodic Type	Translating
Triangle Motion	User specify
Velocity (m/s)	0.0, 0.0, 0.0
Use file	<input type="checkbox"/>
Periodic Shape	Sector (Pie shape)
Angle (degree)	60
Rotation Axis	0.0, 0.0, 0.0
Matching Boundary	Not Assigned

# SYMMETRY Boundary Type

- A SYMMETRY boundary specifies the computational domain be symmetric
  - The velocity boundary condition defaults to SLIP
  - All other boundary conditions default to zero normal Neumann
  - This is useful to model a portion of the geometry
- You can use a SYMMETRY boundary to perfectly reflect spray parcels

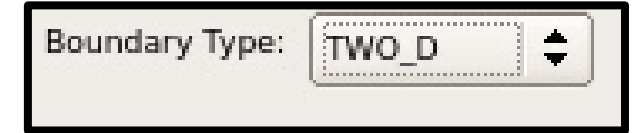


The screenshot displays the configuration panel for a SYMMETRY boundary type. At the top, the 'Boundary Type' is set to 'SYMMETRY'. Below this, several sections define the boundary conditions for different physical quantities:

- Velocity Boundary Condition:** Set to 'SLIP', with three input fields for velocity components, all containing the value '0'.
- Pressure Boundary Condition:** Set to 'Zero normal gradient (NE)'.
- Temperature Boundary Condition:** Set to 'Zero normal gradient (NE)'.
- Species Boundary Condition:** Set to 'Zero normal gradient (NE)'.
- Passive Boundary Condition:** Set to 'Zero normal gradient (NE)'.
- Turbulent Kinetic Energy (tke) Boundary Condition:** Set to 'Zero normal gradient (NE)'.
- Turbulent Dissipation (eps) Boundary Condition:** Set to 'Zero normal gradient (NE)'.
- Specific Turbulent Dissipation (omega) Boundary Condition:** Set to 'Zero normal gradient (NE)'.

# TWO\_D Boundary Type

- A TWO\_D boundary type specifies a two-dimensional simulation
- You must still configure a three-dimensional surface in CONVERGE Studio
  - Create a pair of identical parallel boundaries and designate them both as TWO\_D boundaries
  - Ensure that the corresponding TWO\_D boundaries are on either side of the z axis
- You cannot use the TWO\_D boundary for moving geometries or spray modeling





# GT-SUITE Boundary Type

- A GT-SUITE boundary type couples CONVERGE with GT-SUITE to send and receive boundary condition values
- Choose the boundary ID (from the GT-SUITE file) at which CONVERGE - GT-SUITE coupling occurs

The screenshot shows the 'Boundary Type' configuration window for a GT-SUITE boundary. The 'Boundary Type' dropdown is set to 'GT-SUITE'. Below it, the 'Boundary ID in GT-SUITE' is set to '1'. The 'Passive Boundary Condition' section has 'Zero normal gradient (NE)' selected, with a 'Pull from its region' button. The 'Turbulent Kinetic Energy (tke) Boundary Condition' section has 'Intensity' selected, with a value of '0.02' entered in the text box, followed by the unit 'fraction' and a 'Use file' checkbox. The 'Turbulent Dissipation (eps) Boundary Condition' section has 'Length scale' selected, with a value of '0.003' entered in the text box, followed by the unit 'm' and a 'Use file' checkbox.

Boundary Type: GT-SUITE

Boundary ID in GT-SUITE: 1

**Passive Boundary Condition**

Zero normal gradient (NE) Pull from its region

**Turbulent Kinetic Energy (tke) Boundary Condition**

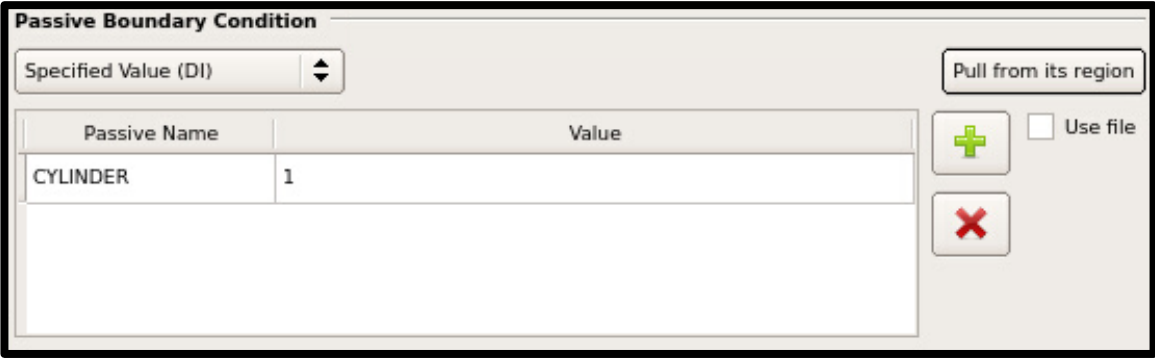
Intensity 0.02 fraction Use file

**Turbulent Dissipation (eps) Boundary Condition**

Length scale 0.003 m Use file

# GT-SUITE: Passive

- There are two types of GT-SUITE boundary conditions available for the passive equation
  - Dirichlet (specified passive value)
  - Zero normal Neumann (zero passive gradient)
- If you do not specify the value of any passives, the values of all passive scalars will be coupled with GT-SUITE



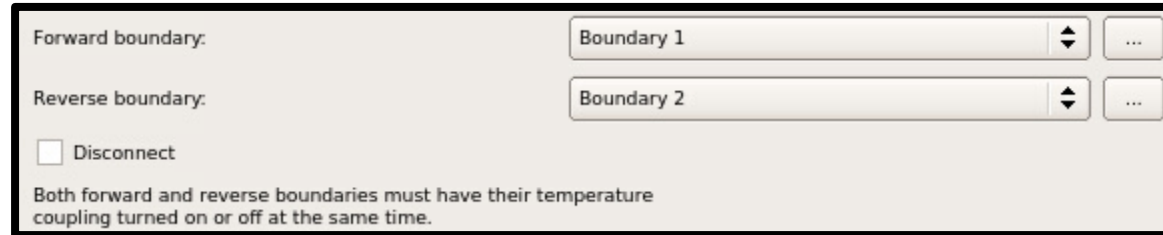
The screenshot shows the 'Passive Boundary Condition' dialog box. At the top, there is a dropdown menu set to 'Specified Value (DI)'. To the right of this is a button labeled 'Pull from its region'. Below the dropdown is a table with two columns: 'Passive Name' and 'Value'. The table contains one row with 'CYLINDER' in the first column and '1' in the second column. To the right of the table are two buttons: a green plus sign and a red X. Below these buttons is a checkbox labeled 'Use file'.

Passive Name	Value
CYLINDER	1

# GT-SUITE: Turbulence

- If using a turbulence model, you must specify a boundary condition for turbulence parameters (tke and either eps or omega)
- There are two types of GT-SUITE boundary conditions for tke
  - Dirichlet (specified tke value)
  - Intensity
- There are two types of WALL boundary conditions for eps and omega
  - Dirichlet (specified eps/omega value)
  - Turbulent length scale
    - This option is a special case of the Dirichlet boundary condition

# INTERFACE Boundary Type



The screenshot shows a dialog box for defining an INTERFACE boundary. It contains two dropdown menus: 'Forward boundary:' set to 'Boundary 1' and 'Reverse boundary:' set to 'Boundary 2'. Below these is a checkbox labeled 'Disconnect' which is currently unchecked. At the bottom, a note states: 'Both forward and reverse boundaries must have their temperature coupling turned on or off at the same time.'

- An INTERFACE boundary demarcates the boundary between two different materials or phases
  - Use this type of boundary to simulate conjugate heat transfer
- You must define a forward and reverse boundary for each INTERFACE
  - The boundary facing the direction of the normal vectors is the forward boundary
  - The boundary facing the opposite direction of the normal vectors is the reverse boundary
- CONVERGE controls the interface triangles between the two phases
  - If you check the Disconnect box, CONVERGE disables the interface triangles and allows flow between regions during OPEN events; otherwise, interface triangles will remain for the entire simulation

# THANK YOU!

CONVERGECFD.COM



© 2015 Convergent Science. All Rights Reserved.