ANSYS **Fluent CFD** Boundary Conditions

FLOW BOUNDARY CONDITIONS

Velocity

Inlet—define velocity and scalar flow properties at inlet

Pressure

Inlet—define total pressure and other scalar quantities at flow inlet

Outlet—define static pressure at flow outlet (provides better convergence with backflow)

Mass Flow—prescribe mass flow rate and scalar flow properties at inlet (suitable for compressible flow)

Pressure Far-Field—model a free-stream compressible flow at infinity by Mach number and static conditions (requires compressible flow)

Outflow—model flow exits where details of flow velocity and pressure are unknown prior to solution of problem (appropriate for fully developed flow; requires incompressible flow)

Vent

Inlet—model inlet vent with specified loss coefficient, flow direction, and ambient (inlet) total pressure and temperature

Outlet—model outlet vent with specified loss coefficient and ambient (discharge) static pressure and temperature

Fan

Intake—model external intake fan with specified pressure jump, flow direction, and ambient (intake) total pressure and temperature.

Exhaust—model external exhaust fan with specified pressure jump and ambient (discharge) static pressure.

Degassing—model free surface through which dispersed gas bubbles are allowed to escape but the continuous liquid phase is not (requires two-phase liquid-gas flows using Eulerian multiphase model)

OTHER BOUNDARY CONDITIONS

Wall—bound fluid and solid regions (no-slip enforced by default, but user can specify tangential velocity component or model "slip" wall by specifying shear; can specify roughness, surface tension, motion, species, film conditions, flux, etc.)

Symmetry—mirror system (reduce the extent of your computational model to a symmetric subsection of the overall physical system) [N.B. Symmetry doesn't equate to mirroring; it mathematically represents a zero-shear-force wall.]

Axis—mirror system at globale coördinate system axis

Periodic—used when physical geometry of interest and expected pattern of flow/thermal solution have periodically repeating nature (may be *cyclic* or *periodic*)

Fan—allows user to input empirical fan curve governing head and flow rate across fan element; user can also specify radial, tangential components of fan swirl velocity

Radiator—specify both pressure drop and heat transfer coefficient as functions of velocity normal to the radiator using lumped-parameter model

Porous Jump—model thin membrane of known pressure-drop characteristics

Non-Reflecting—eliminate spurious wave reflections in artificially truncated domain

Extracted from ANSYS Fluent User's Guide v16.2.