



Glenn Research Center

Physical and Zonal Boundary Conditions

Boundary Conditions

The governing equations are solved within a finite flow domain and zone.

A solution of the equations requires accurate specification of conditions at the boundaries of the domain and zones \Rightarrow **Boundary Conditions**.

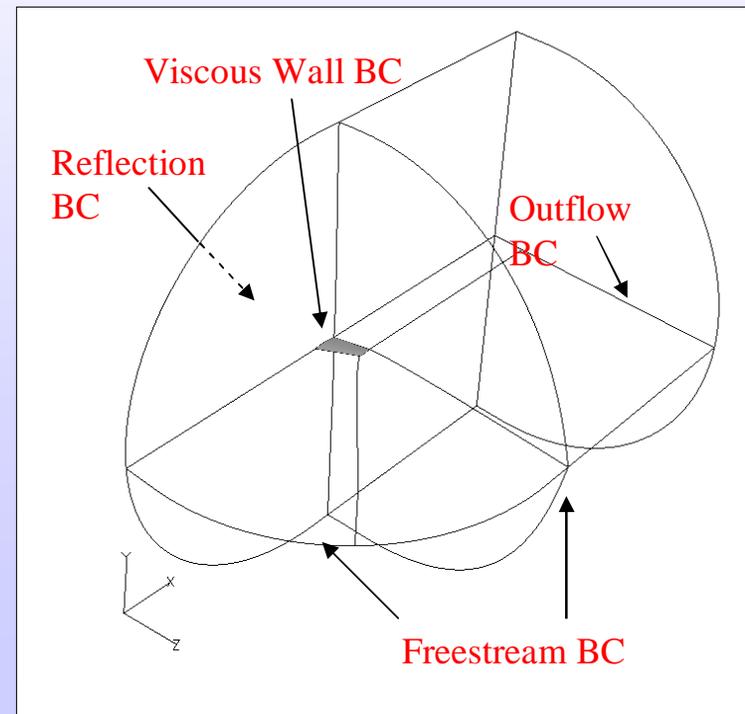
Distinguish between two types:

Physical Boundary Conditions

at the boundary of the flow domain

Zonal Boundary Conditions

at the boundary between two zones





Physical Boundary Conditions

Physical boundary conditions can be of several *Types* (as set in GMAN) and can be grouped:

FROZEN
FREESTREAM
ARBITRARY INFLOW
OUTFLOW

} Flow Interface BCs
Boundaries at which
the flow crosses freely

VISCOUS WALL
INVISCID WALL
REFLECTION
BLEED
BLOWING
MOVING WALL

} Wall BCs
Boundaries with
a solid surface
(relatively small
amounts of flow
may cross)

Boundary conditions also
needed for turbulence,
chemistry, and MFD
equations also.

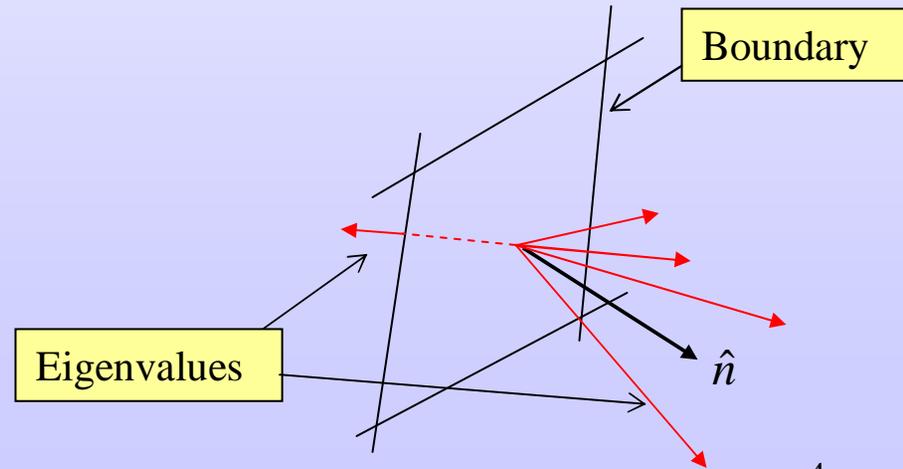
Characteristic Nature at Boundaries

The *characteristic (acoustic wave)* nature of the flow at the boundaries provides critical information for specifying boundary conditions.

Eigenvalues of the flow equations at the boundaries indicate the speed of the acoustic waves and whether they are entering (positive) or leaving (negative) the flow domain at the boundary.

This indicates how much information should come from outside the domain (*physical*) and how much information should come from inside the domain (*numerical*).

Boundary normal \hat{n} is positive directed into the flow domain





Flows and Eigenvalues

The flow at the boundary is:

Inflow when $v'_n > 0$ and so $\lambda_1, \lambda_2, \lambda_3 > 0$

Outflow when $v'_n < 0$ and so $\lambda_1, \lambda_2, \lambda_3 < 0$

Subsonic when $|v'_n| < c$

Supersonic when $|v'_n| > c$

Combinations are:

Subsonic Inflow $\lambda_1, \lambda_2, \lambda_3, \lambda_4 > 0$ and $\lambda_5 < 0$

Subsonic Outflow $\lambda_4 > 0$ and $\lambda_1, \lambda_2, \lambda_3, \lambda_5 < 0$

Supersonic Inflow $\lambda_1, \lambda_2, \lambda_3, \lambda_4, \lambda_5 > 0$

Supersonic Outflow $\lambda_1, \lambda_2, \lambda_3, \lambda_4, \lambda_5 < 0$

	Physical	Numerical
Subsonic Inflow	4	1
Subsonic Outflow	1	4
Supersonic Inflow	5	0
Supersonic Outflow	0	5



Subsonic Inflow BC

The **Subsonic Inflow** condition has eigenvalues

$$\lambda_1, \lambda_2, \lambda_3, \lambda_4 > 0 \quad \lambda_5 < 0$$

which indicates that 4 physical and 1 numerical conditions need to be imposed at the boundary. These conditions can be imposed when using the **FREESTREAM** or **ARBITRARY INFLOW** BC types at flow interfaces or the **BLEED** or **BLOWING** BC types at walls.



Subsonic Outflow BC

The **Subsonic Outflow** condition has eigenvalues

$$\lambda_1, \lambda_2, \lambda_3, \lambda_5 < 0 \quad \lambda_4 > 0$$

which indicates that 1 physical and 4 numerical conditions need to be imposed at the boundary. These conditions can be imposed when using the **FREESTREAM** or **OUTFLOW** BC types at flow interfaces or the **BLEED** or **BLOWING** BC types at walls.



Supersonic Inflow / Frozen BC

The **Supersonic Inflow** condition has all positive eigenvalues

$$\lambda_1, \lambda_2, \lambda_3, \lambda_4, \lambda_5 > 0$$

which indicates that all information should come from outside the flow domain. The **FROZEN** BC type is appropriate for supersonic inflow because the boundary condition simply keeps the flow conditions at the boundary unchanged. These conditions can be also imposed when using the **FREESTREAM** or **ARBITRARY INFLOW** BC types at flow interfaces.



Supersonic Outflow BC

The **Supersonic Outflow** condition has all negative eigenvalues

$$\lambda_1, \lambda_2, \lambda_3, \lambda_4, \lambda_5 < 0$$

which indicates that all information should come from inside the flow domain. An extrapolation of the solution to the boundary is the typical numerical method. The **FREESTREAM** or **OUTFLOW** BC types are both capable of imposing the correct conditions.



Freestream BC

The **FREESTREAM** BC type is used at boundaries where *flow conditions are expected to be fairly uniform and close to freestream conditions*. The BC is flexible in that it checks whether the flow is inflow or outflow and subsonic or supersonic at the boundary and imposes the correct conditions.

For **supersonic inflow** or **supersonic outflow** the previous two slides indicate the conditions that are imposed.

For **subsonic inflow** and **subsonic outflow**, a characteristic-based method is used to determine conditions. Assuming uniform conditions, two Riemann invariants R^+ and R^- can each be evaluated either numerically or from freestream conditions as appropriate and then used to compute the other flow properties.

$$R^{\pm} = v'_n \pm \frac{2c}{\gamma-1} \quad v'_n = \frac{1}{2} (R^+ + R^-) \quad c = \frac{\gamma-1}{4} (R^+ + R^-)$$



Glenn Research Center

Arbitrary Inflow BC 1 of 2

The **ARBITRARY INFLOW** BC type is used to impose a subsonic inflow condition, for internal flows (i.e. plenum). BC requires 4 physical conditions and 1 numerical condition. The BC has the capability to:

- Hold total pressure and total temperature at the inflow.
- Specify the flow angle at the inflow.
- Hold total pressure, total temperature, and flow angles of input solution.
- Specify an inflow profile (IJK_RANGE).
- Specify inflow turbulence levels.
- Specify inflow chemistry composition.
- Specify inflows with a solid-body rotation.
- Specify time-varying inflows.



Arbitrary Inflow BC 2 of 2

HOLD_TOTALS

- 4 physical conditions: Specify and hold: p_t , T_t , α , and β
- 1 numerical condition: Riemann invariant R^-
- Uniform or vary (i.e. boundary layer) over boundary
- Newton iteration on T of the form: $T^{m+1} = T^m - f / f'$

$$f = T_t - T - \frac{\gamma - 1}{2} \frac{V^2}{\mathcal{R}} \quad f' = - (1 + V / A c)$$

$$V A = \vec{v} \cdot \hat{n} \quad V = \frac{1}{A} \left(R^- + \frac{2c}{\gamma - 1} \right)$$

A is constant

V is magnitude of velocity

c is speed of sound and function of T

```

ARBITRARY INFLOW
TOTAL
HOLD_TOTALS
ZONE 3
UNIFORM 0.3 14.0 600.0 5.0 0.0
ENDINFLOW
    
```



Outflow Boundary Conditions

The **OUTFLOW** BC type is used mainly to impose a *subsonic outflow* condition, especially for internal flows. The four numerical conditions are imposed through extrapolation. The one physical condition is imposed by determining the static pressure at the boundary in some way. Options include:

Uniform Static Pressure (constant or time-varying)

Variable Static Pressure (reference pressure with spatial variation over the boundary)

Mass Flow (for boundary and then compute the local static pressures)

Average Mach Number (for boundary and then compute the local static pressures)

Compressor Face (reflection coefficients for wave interactions with compressors)

Non-Reflecting (reference Mach number with non-reflecting constraints)

In the case of *supersonic outflow*, the Outflow BC extrapolates all flow variables.



Subsonic Outflow BCs 1 of 3

a) Specified Static Pressure Model

- Directly specify static pressure
- Spatially uniform or variable
- Steady or time-varying

b) Mass Flow Model

- Match specified outflow mass flow
- Relaxation over iterations

$$p^{n+1} = p^n \left[1 + \theta (\dot{m}_B - \dot{m}^n) / \dot{m}_B \right]$$



Subsonic Outflow BCs 1 of 3

c) Mach Number Model (*Chung-Cole, 1994; Mayer-Paynter, 1994*)

- Specify a uniform or average Mach number, M_{cf}

$$p = \bar{p}_t \left(1 + \frac{\gamma-1}{2} M_{cf}^2 \right)^{-\gamma/(\gamma-1)}$$

d) Nozzle Model

- Use nozzle section attached to outflow
- Choked flow at nozzle throat
- Supersonic outflow at exit of nozzle (extrapolate flow)
- Adjust mass flow by varying nozzle throat area



Wall Boundary Conditions

Wall boundary conditions involve a solid surface which may be porous. The fundamental relation at a wall boundary is

$$\rho (\vec{v} - \vec{g}) \cdot \hat{n} S_B = \rho v_n' S_B = \dot{m}$$

S_B is the area of the cell face over which the BC is applied.

The **Bleed** or **Blowing** BC will have $\dot{m} \neq 0$, and so, $v_n' \neq 0$.

The **Moving Wall** BC will have $\vec{g} \neq 0$.

For all the wall boundary conditions, the value of v_n' is based on the presence of bleed or blowing. If $\dot{m} = 0$ then $v_n' = 0$.



Wall BCs: Viscous Wall

The **VISCOUS WALL** BC type is used at solid boundaries for which a velocity no-slip condition is required. A boundary layer forms along the wall. The no-slip condition is that the tangential flow relative to the boundary is zero:

$$v'_{t_1} = (\vec{v} - \vec{g}) \cdot \hat{t}_1 = 0 \qquad v'_{t_2} = (\vec{v} - \vec{g}) \cdot \hat{t}_2 = 0$$

where the two tangential vectors are perpendicular, $\hat{t}_1 \cdot \hat{t}_2 = 0$.

Classical boundary layer theory indicates that the pressure gradient in the direction normal to the wall is approximately zero, this is used as a physical condition at the viscous wall

$$\frac{\partial p}{\partial n} = 0$$

Another physical condition used at the wall assumes an adiabatic condition for

which $\frac{\partial T}{\partial n} = 0$ or a constant temperature at the wall $T_B = T_{wall}$.



Wall BCs: Inviscid Wall / Reflection

The **INVISCID WALL** BC type is used at solid boundaries for which a velocity slip condition is desired. Thus the tangential velocity components v'_{t_1} and v'_{t_2} must now be computed in some manner. One approach is to simply extrapolate the tangential velocity components from the interior cells to the boundary.

The **REFLECTION** BC type uses the same methods as the inviscid wall BC type; however, the intent of the reflection BC is to apply it at a planar boundary of flow symmetry.



Bleed Boundary Condition 1 of 2

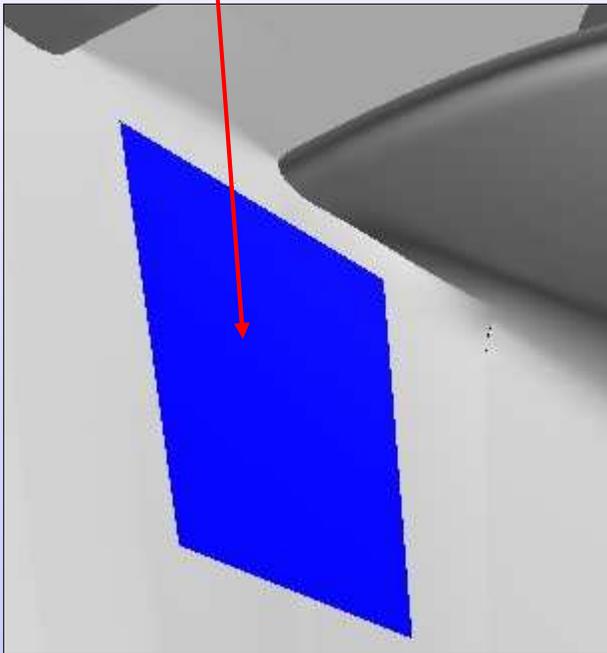
The **BLEED** BC type is used at solid boundaries for which there is a small amount of flow across a porous boundary. The purpose of bleed is to remove the low energy portion of a boundary layer to leave the higher energy boundary layer flow to reduce the chance of boundary layer separation.

- Subsonic outflow
 - 4 numerical conditions
 - 1 physical condition (velocity component of bleed)
- Models bleed \dot{m}_{bleed} over boundary area A_{region}

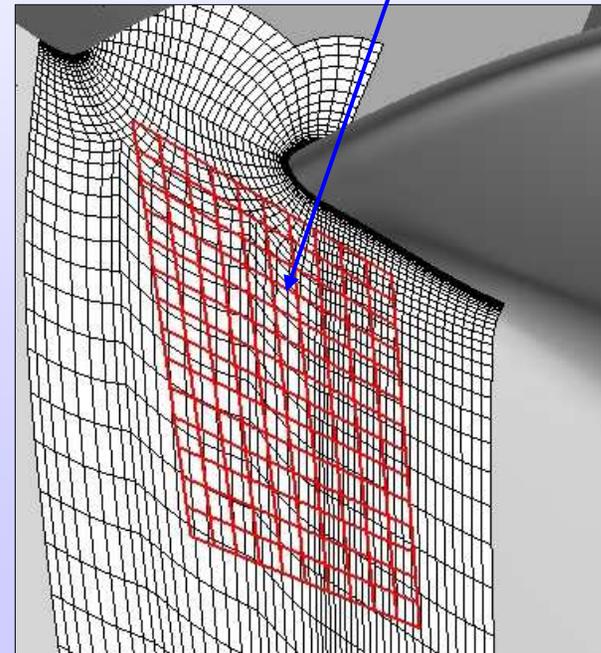
$$\vec{u}_{bleed} = -u_{bleed} \hat{n} \qquad u_{bleed} = \frac{\dot{m}_{bleed}}{\rho A_{region}}$$

Bleed Boundary Condition 2 of 2

The bleed region A_{region} is the outline of the porous bleed region.



A dummy grid zone is created in the shape of the outline of the porous bleed region and then coupled to the inlet surface grid.





Bleed Models 1 of 3

Model 1) Directly specify the mass flow over the region \dot{m}_{bleed}

$$u_{bleed} = \frac{\dot{m}_{bleed}}{\rho A_{region}}$$

The mass flow is specified as a fraction `blv1` of the captured freestream flow

$$\dot{m}_{bleed} = blv1 \left(\rho_{\infty} V_{\infty} A_{cap} \right)$$

where A_{cap} is the capture area defined in GMAN. The keyword for WIND is

`BLEED ibrg blv1`

However, this model imposes uniform and fixed bleed rate that can not respond to local flow conditions.

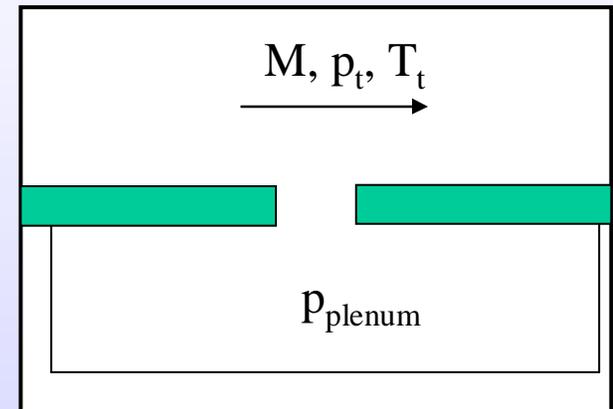
Bleed Models 2 of 3

Model 2) Porous bleed with a specified discharge coefficient C_d

$$\dot{m}_{bleed} = C_d \dot{m}_{ideal}$$

$$\dot{m}_{ideal} = p_t \Phi M \left(\frac{\gamma}{RT_t} \right)^{1/2} \left[1 + \frac{\gamma-1}{2} M^2 \right]^{-[(\gamma+1)/2(\gamma-1)]}$$

$$\Phi = \frac{A_{bleed}}{A_{region}}$$



Where Φ , porosity of the bleed region (blv2)

A_{bleed} , total area of all the bleed holes

BLEED POROSITY ibrg blv1 blv2 blv3

Bleed Models 3 of 3

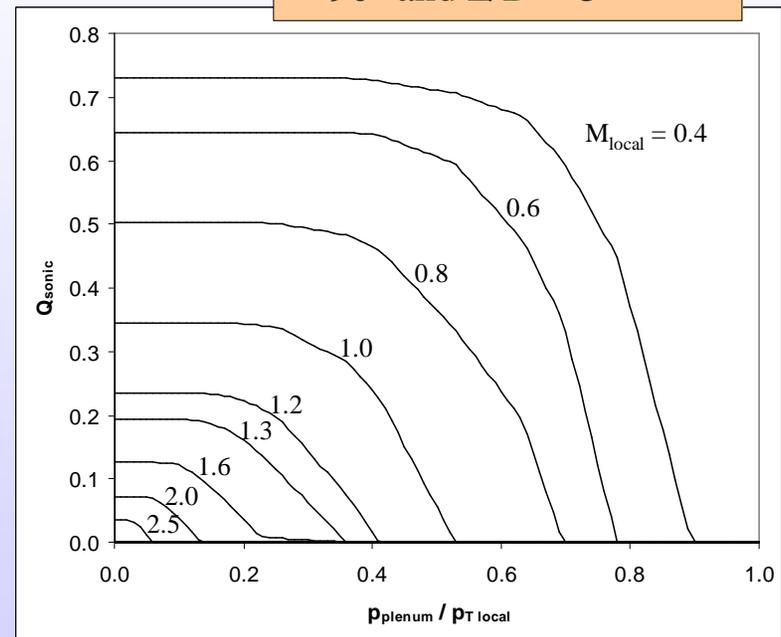
Model 3) Porous bleed with a table lookup of Q_{sonic}

Empirical data for $\alpha_{bleed} = 90^\circ$ and $L/D = 3$

$$\dot{m}_{bleed} = Q_{sonic} \dot{m}_{max}$$

$$\dot{m}_{max} = p_t \Phi \left(\frac{\gamma}{RT_t} \right)^{1/2} \left[1 + \frac{\gamma-1}{2} \right]^{-[(\gamma+1)/2(\gamma-1)]}$$

$$Q_{sonic} = f \left(\alpha_{bleed}, M_{local}, \frac{P_{plenum}}{P_{tlocal}} \right)$$



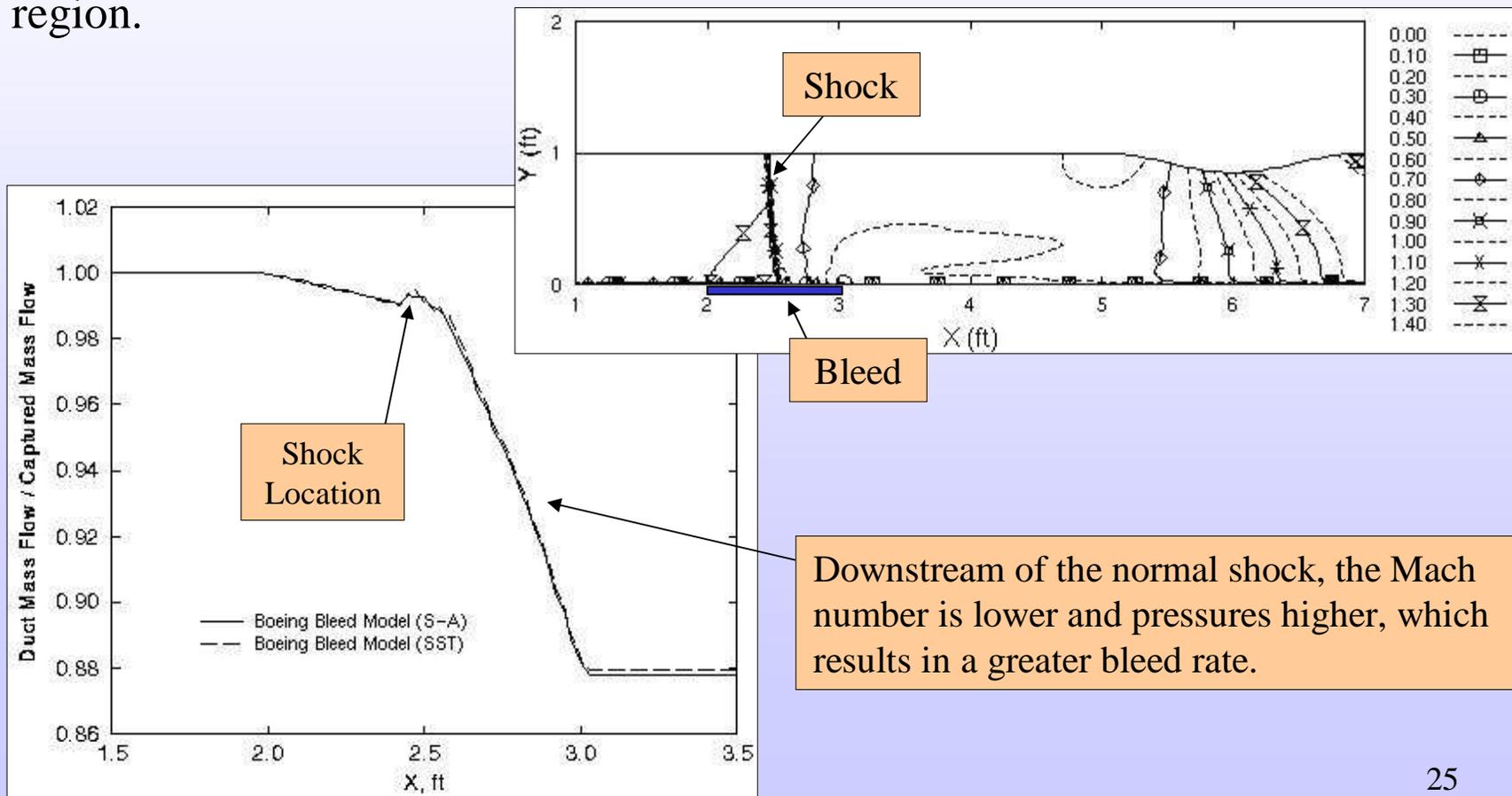
\dot{m}_{max} corresponds to a choked bleed hole ($M = 1$)

M_{local}, P_{tlocal} are at the edge of the boundary layer

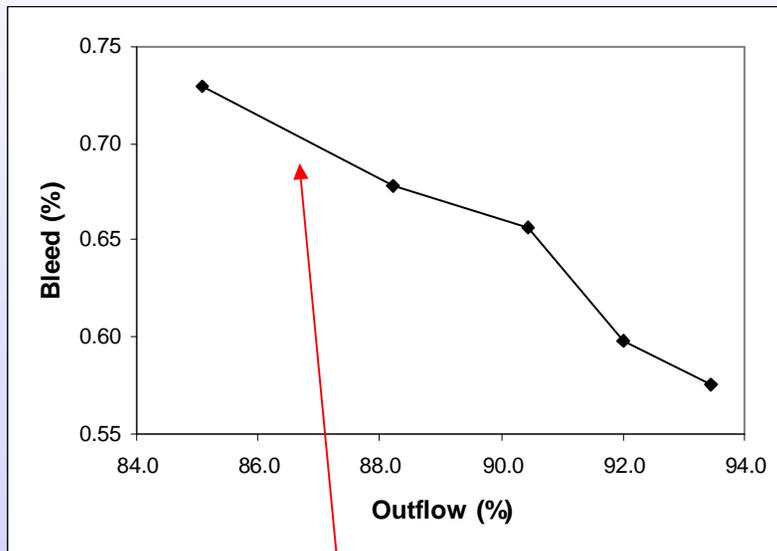
BLEED MODEL ibrg mode blv1 blv2 blv3 blv4

Porous Bleed BC Application 1 of 2

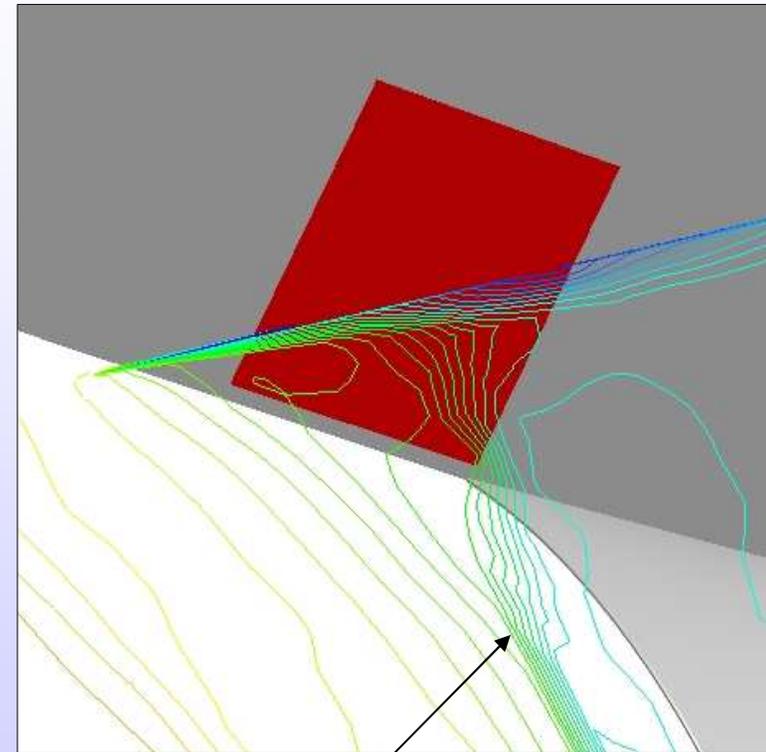
Model 3 with varying Q_{sonic} and consideration of local conditions allows the bleed flow to vary, as it should, when shock waves interact with the bleed region.



Porous Bleed BC Application 2 of 2



With decrease inlet outflow, the shock moves forward and exposes the bleed region to higher local pressure, which results in greater bleed.



Shock



Wall BCs: Moving Wall

The **MOVING WALL** boundary condition is used at solid boundaries that have uniform translation or rotation. An example of a uniform translation is an airplane going down a runway. An example of a uniform rotation is the airplane spinning about its x -axis. This BC type can be used with the viscous or inviscid wall BC types.



Boundary Condition Numerical Methods

- For a cell-vertex finite-volume cell, the boundary condition is applied at the solution point on the boundary (approach of WIND).
- For a cell-centered finite-volume cell, the boundary condition is applied at the cell face on the boundary as a flux.
- Boundary conditions can be imposed *explicitly* at boundary solution points (this is the default method in WIND).
- Boundary conditions can be imposed *implicitly* by including the conditions into the implicit matrices of the numerical method for the inner flow field. Wall BCs can be solved implicitly in WIND.
- Boundary conditions are imposed at the end of each iteration of the flow equations.



Flow Property Extrapolations

Most of the boundary conditions use an extrapolation of flow field properties to the boundary as part of imposing a numerical condition. A simple zero-order extrapolation of a property ϕ (which can represent static pressure or density) is

$$\phi_B = \phi_A$$

Where ϕ_B is the boundary value and ϕ_A is the value at a solution point adjacent to the boundary. A first-order extrapolation has the form

$$\phi_B = (1 + \alpha)\phi_A - \alpha\phi_{AA} \quad \alpha = \delta s_A / \delta s_{AA}$$

Where ϕ_{AA} is the value at a solution point adjacent to the adjacent point to the boundary. The “adjacent” direction is usually approximately normal to the boundary and along the grid line from the boundary. The δs are the spacings between the solution points. The **ORDER** keyword option sets extrapolation.



Zonal Boundary Conditions

- Zonal Boundary Conditions are needed at zonal boundaries that are in contact with other zonal boundaries.
- Flow information is transferred across zonal boundaries.
- Zonal BCs can be classified as:
 - o Single-Zone Zonal BCs
 - o Multi-Zone Zonal BCs

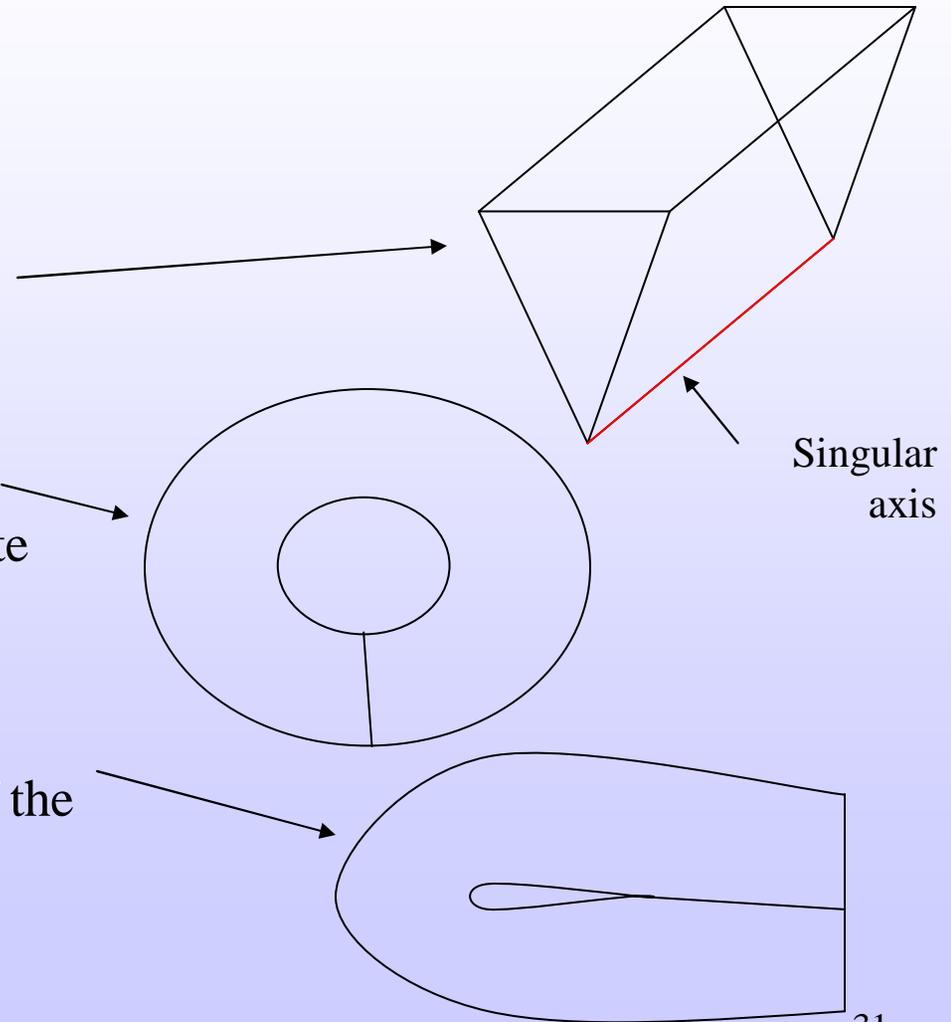
Single-Zone Zonal BCs

A zonal boundary of a zone may be topologically connected to itself or other boundaries of the same zone:

Singular Axis A zonal boundary collapses to a line.

Self-Closing A zonal boundary connects to its topologically opposite zonal boundary.

Self-Coupled A zonal boundary is coupled to itself on some portion of the zonal boundary.



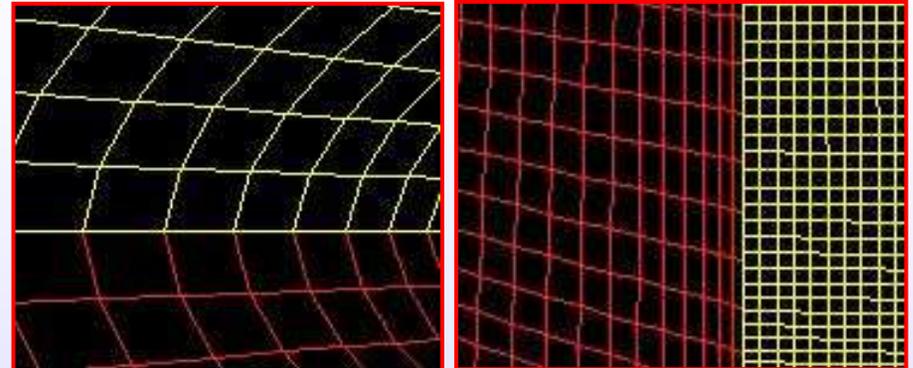
Multi-Zone Zonal BCs

A zonal boundary of a zone may be topologically connected to other boundaries of other zones.

Connectivity defines how a zonal boundary is connected to other zonal boundaries.

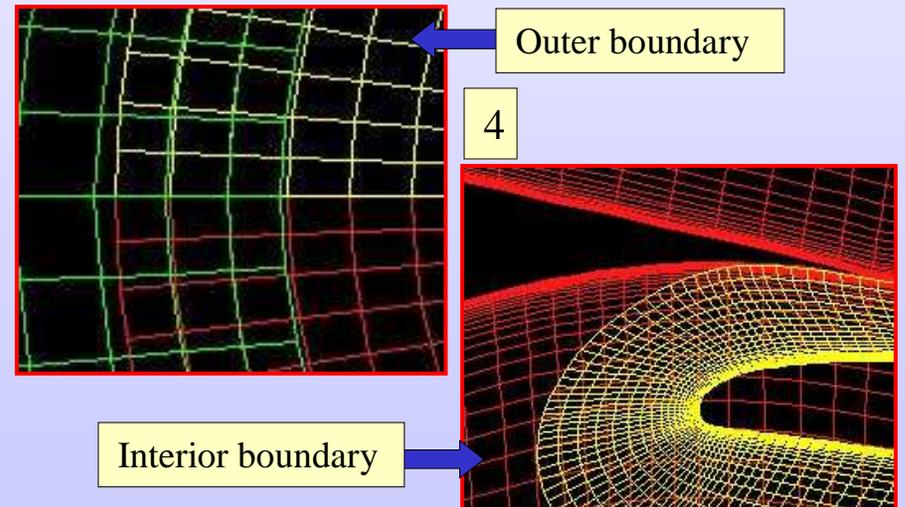
Types of zonal connectivity:

- 1) Abutting, Point-to-point match
- 2) Abutting, Non point-to-point match
- 3) Overlapping, Point-to-point match
- 4) Overlapping, Non point-to-point match



1 & 3

2



Outer boundary

4

Interior boundary



Zone Coupling Methods

Information is transferred across the zonal boundaries by a couple of methods:

Characteristic Coupling. This method uses one-dimensional characteristic flow theory to set boundary flow field variables based on local flow direction and strength. These boundary variables are then transferred between zones using a tri-linear interpolation.

Roe Coupling. This method uses Roe's flux-difference splitting to compute flux cell "interface states" and the zonal boundary is considered a "cell interface" in the Roe scheme. A higher-order zone coupling involves the transfer of solution derivatives. This method only operates on the inviscid component of the flux. The exchange of turbulence information uses the tri-linear interpolation.

Both methods operate as explicit operations. In WIND, these are done at the end of a cycle rather than the end of an iteration.



Zone Coupling Issues

Some issues regarding zone coupling:

- Avoid placing zonal boundaries in regions with large flow gradients (shocks, boundary layers, shock / boundary layer interaction).
- Best if zonal boundaries are place *normal* to the flow direction.
- Placing zonal boundaries parallel to the flow in regions of high gradients is generally not good, but often unavoidable.
- Point-to-point matching across zones is best since inviscid flux information can be directly transferred and errors are less in interpolation of viscous and turbulent information.
- Best if zones overlap by at least two grid points, especially in turbulent flows. This improves the interpolation. However, this may be difficult for grids with lots of zones and complicated geometry.



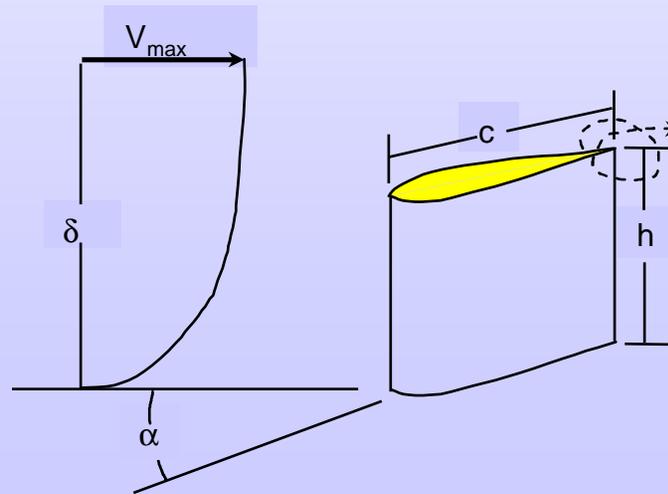
Physical Models Applied at Zonal Boundaries

Several physical models can be applied at zonal boundaries to affect the flow as it crosses the zonal boundary. Those in WIND were implemented to have some specific capabilities for the analysis of propulsion systems:

- **Actuator Disk.** This model imposes a rotational motion to the velocity field as the flow goes through the zonal boundary. It is meant to simulate the addition of energy to the flow due to a rotor (compressor fan).
- **Screen.** This model imposes a loss in total pressure across the zonal boundary.
- **Vortex Generators.** This model imposes a number of small vortices that simulate the vortices produced by an array of vane-type vortex generators.

Vortex Generator Model 1 of 2

- **Objective:** To give the Inlet Branch the capability to model vane-type vortex generators in inlet CFD calculations
- **Description:** Derived using conservation of momentum & inviscid theory correlated with experimental data
 - Data taken one chord length downstream of trailing edge
 - Relates the vortex strength, Γ , and peak vorticity, ω_{max} to:





Vortex Generator Model 2 of 2

Description (cont'd):

- The cross-flow velocity of one vortex is then given by:

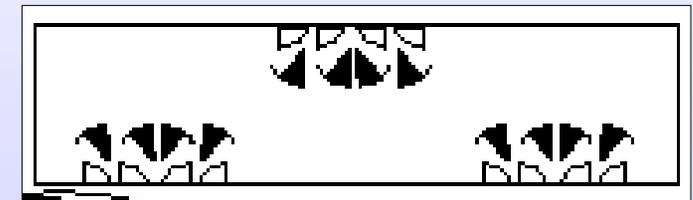
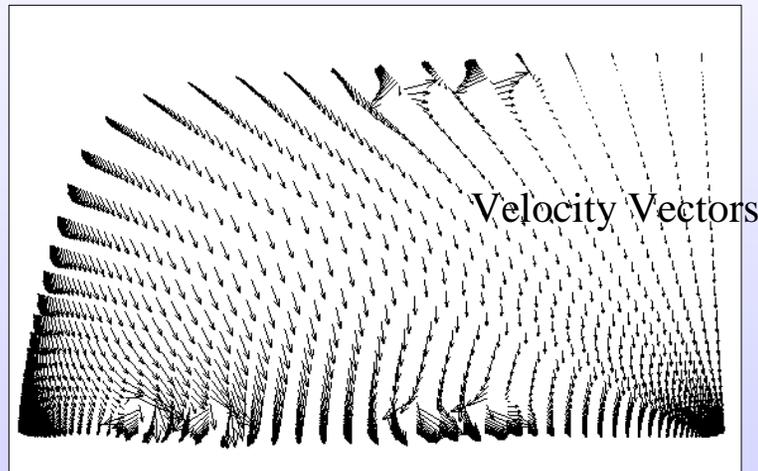
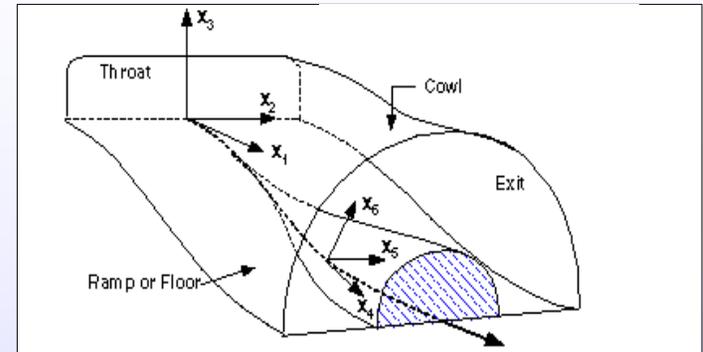
$$v_{\theta} = \Gamma_i \left[1 - \exp\left(-\frac{\pi \omega_i^{\max} R_i^2}{\Gamma_i}\right) \right]$$

- An array of generators is represented by a summation (superposition) of terms
- Patterned after existing Wind actuator disk/screen boundary conditions
- Modification to a coupled zonal interface boundary
- Vortices produced by the generators are simulated by a step change in the cross-flow velocity
- The user inputs the VG parameters: c , h , α , V_{max} , and δ . Recommended values:

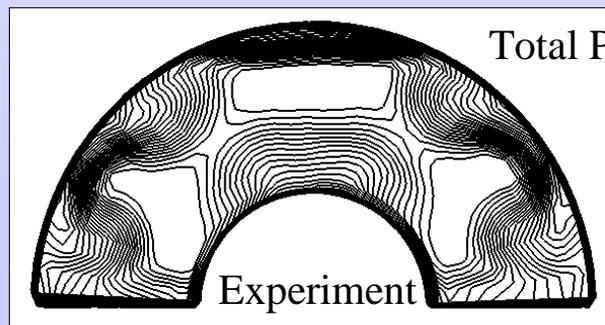
$0.13 < h/c < 2.62$	$8 \text{ deg} < \alpha < 20 \text{ deg}$
$0.12 < h/\delta < 2.60$	$0.2 < M < 0.6$

Vortex Generator Model Validation: Td118 Subsonic Diffuser

- Objective: Demonstrate the use of the WIND vortex generator model for a strong adverse pressure gradient diffuser flow.



Total Pressure Contours



Total Pressure Contours

