

Initial and Boundary Conditions

Initial- and boundary conditions are needed

For a steady problems correct initial conditions is important to reduce computational time and reach convergence

Boundary conditions must always be included, and might be of different type depending on the case

Velocities are always set relative to a reference frame (in case of moving regions)

Boundary Conditions

- Inlet
- Pressure outlet
- Walls
- Symmetry

There are also others that may be dealt with later. Example is *periodic*, *mass flow inlet*, *stagnation inlet*, *free-stream*.

Also there may be rotational zones in the interior domain (propeller case, dealt with later). Interior boundaries.

Boundary Conditions

General boundary conditions:

Dirichlet: ϕ is specified at the boundary.

Neumann: $\frac{\partial \phi}{\partial x}$ is specified, usually normal to the boundary.

Boundary Conditions – velocity inlet

Variable	Formulation	Must be specified
Velocity	Inlet face velocity vector specified directly	X
Pressure	Boundary face pressure extrapolated from neighbor cell	
Temperature	The inlet static temperature specified	Only with energy
Turbulence	Constant intensity/viscosity ratio	X

Always use sufficient distance from the inlet to the object. No outflow at the boundary.

Other profiles than uniform possible using field functions.

Boundary Conditions – pressure outlet

Variable	Formulation	Must be specified
Velocity	Boundary face velocity extrapolated from neighbor cell OR normal component	
Pressure	Constant value must be given	X
Temperature	Extrapolation	
Turbulence	Constant intensity/viscosity ratio	X

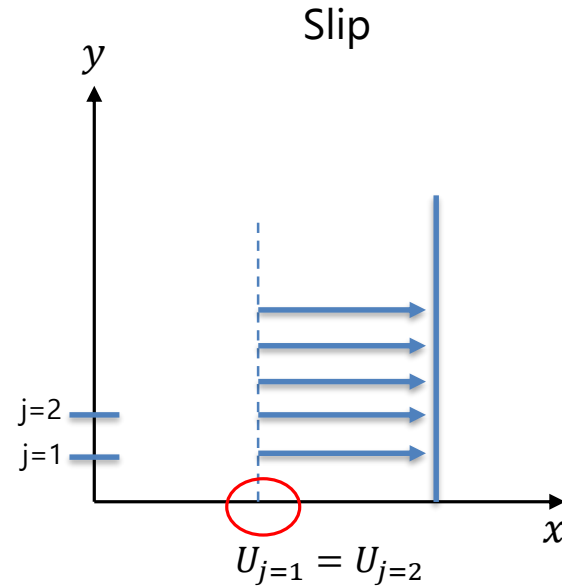
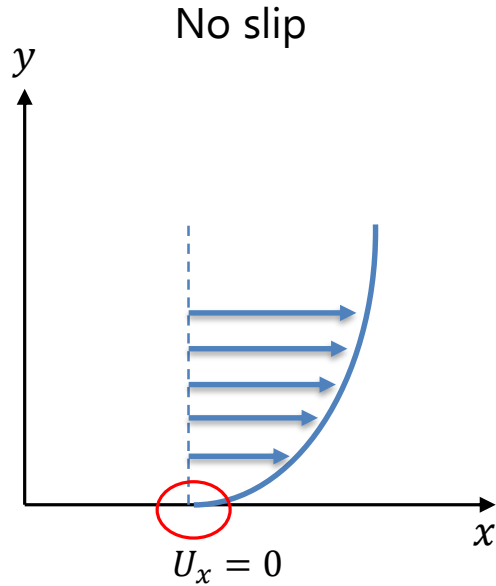
For velocity there is an option to set the flow direction normal to the face. Prevents backflow. Pressure can be specified in different ways inside Star CCM+. Pressure can be taken as the reference pressure.

Boundary Conditions – walls

Variable	Formulation	Must be specified
Velocity	<u>No-slip</u> : tangential velocity zero, or equal to wall velocity. No normal component <u>Slip</u> : face tangential velocity extrapolated. No normal component	
Pressure	Extrapolated	
Temperature	Both temperature and heat flux can be specified	Only with energy
Turbulence	Constant intensity/viscosity ratio	

Can choose smooth or rough wall. In the last mentioned, need to specify some parameters in case it is rough. Rough wall most relevant for turbulent flows.

Boundary Conditions – walls



Boundary Conditions – symmetry

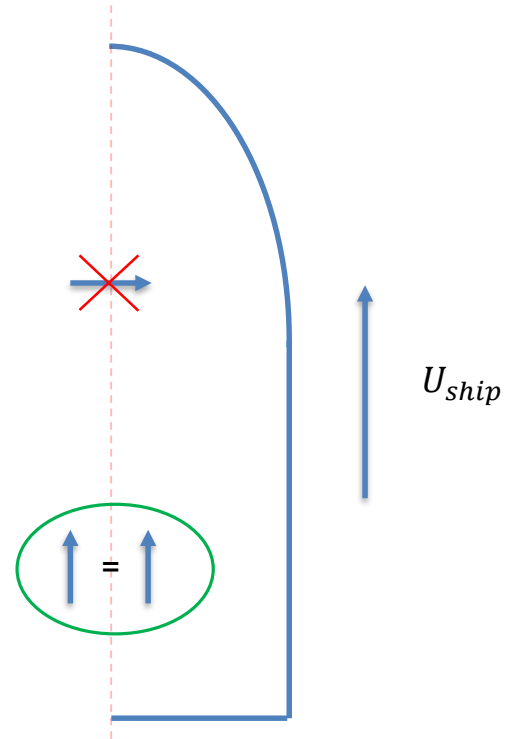
Variable	Formulation	Must be specified
Velocity	Face velocity found by extrapolating parallel component in neighbor cells. No shear stress. No normal components.	
Pressure	Extrapolated	
Temperature	Zero heat flux. Temperature value extrapolated.	
Turbulence	Constant intensity/viscosity ratio	

Example if you model just half the ship and expect the flow to be similar on each side. Save computational effort.

Not to be used if there are significant 3D effect close to the plane (short wing)

Boundary Conditions – symmetry

Normal components enforced to zero



Guidelines – boundary conditions

Remember that boundary conditions set constraints on the interior flow.
Can be under- or over specified

Check for gradients close to the inlet/outlet boundaries, might result in less accuracy or worse no convergence if the domain is too small. Especially relevant if symmetry is used as far field boundary condition

Minimize the cell skewness close to the boundaries.

Treatment and use of boundary conditions should be straight forward in Star CCM+

Boundary conditions -Turbulence

Specified on boundaries and as initial conditions.

Intensity:

$$I = \frac{U'}{U} = \frac{\sqrt{\frac{2}{3}k}}{U}$$

Where k is turbulence energy, U' fluctuating velocity component

Viscosity ratio $\frac{\mu_t}{\mu}$

$I = 0.1$ should be ok in many cases. Viscosity ratio also depend on k . Should be 10

Initial conditions

Star CCM+ initializes the solution according to the values that is set under Physics node. Proper values important to reach convergence faster.

Initialization should be done automatically when push **Run**. But can be wise to manually press the green flag.

If velocity inlet, specify this velocity in the domain. If pressure boundaries (outlet), use the reference pressure.

Do simple predications of the expected flow regime.