Postgraduate Certificate in Hydroinformatics in Civil Engineering

Computational Fluid Dynamics

Acceleration is the rate of change of velocity of a fluid, it is a vector quantity and can be expressed in terms of the material derivative of the velocity vector. In Computational Fluid Dynamics, acceleration is an important quantity as it is used to describe the motion of fluids and the forces that act upon them. Related terms include deceleration, which is the rate of change of velocity in the opposite direction, and velocity, which is the rate of change of position.

Advection is the process by which a quantity, such as heat or a tracer, is transported through a fluid due to the motion of the fluid itself. In Computational Fluid Dynamics, advection is an important process as it can dominate the transport of quantities in high-velocity flows. Related terms include convection, which is the process by which heat is transported through a fluid due to the motion of the fluid, and diffusion, which is the process by which a quantity is transported through a fluid due to random motions of the fluid molecules.

Algorithm is a set of rules or procedures that are used to solve a particular problem or perform a particular task. In Computational Fluid Dynamics, algorithms are used to solve the governing equations of fluid motion, such as the Navier-Stokes equations. Related terms include numerical method, which is a method that uses algorithms to solve a problem, and discretization, which is the process of approximating a continuous problem using a discrete set of values.

Anisotropy is the property of a material or fluid that exhibits different properties in different directions. In Computational Fluid Dynamics, anisotropy is important as it can affect the behavior of a fluid in different directions. Related terms include isotropy, which is the property of a material or fluid that exhibits the same properties in all directions, and homogeneity, which is the property of a material or fluid that exhibits the same properties at all points in space.

Applied mathematics is the branch of mathematics that deals with the application of mathematical techniques to solve real-world problems. In Computational Fluid Dynamics, applied mathematics is used to develop models and solve the governing equations of fluid motion. Related terms include pure mathematics, which is the branch of mathematics that deals with the development of mathematical theories and techniques for their own sake, and engineering mathematics, which is the branch of mathematics that deals with the application of mathematical techniques to solve engineering problems.

Artificial viscosity is a numerical technique used to stabilize the solution of the governing equations of fluid motion. In Computational Fluid Dynamics, artificial viscosity is used to prevent the growth of unphysical oscillations in the solution. Related terms include numerical diffusion, which is the process by which a numerical method spreads out a sharp front or interface, and dissipation, which is the process by which energy is lost from a system due to friction or other dissipative processes.

Aspect ratio is the ratio of the length of a computational cell to its width. In Computational Fluid Dynamics, the aspect ratio is an important quantity as it can affect the accuracy and stability of the solution. Related terms include grid size, which is the size of the computational cells used to discretize the problem, and mesh quality, which is a measure of how well the computational cells are shaped and sized.

Boundary condition is a condition that is applied to the borders of a computational domain. In Computational Fluid Dynamics, boundary conditions are used to specify the behavior of the fluid at the boundaries of the domain. Related terms include initial condition, which is a condition that is applied to the entire computational domain at the start of a simulation, and interface condition, which is a condition that is applied to the interface between two different materials or fluids.

Boundary layer is a region near a surface where the velocity of a fluid changes rapidly. In Computational Fluid Dynamics, the boundary layer is an important region as it can affect the behavior of the fluid and the forces that act upon it. Related terms include viscous sublayer, which is a region near a surface where the viscosity of the fluid is important, and turbulent boundary layer, which is a region near a surface where the fluid is turbulent.

Boussinesq approximation is an approximation that is used to simplify the governing equations of fluid motion. In Computational Fluid Dynamics, the Boussinesq approximation is used to neglect the variation of density with temperature or salinity. Related terms include hydrostatic approximation, which is an approximation that is used to neglect the vertical acceleration of a fluid, and shallow water approximation, which is an approximation that is used to neglect the vertical structure of a fluid.

Cavity flow is a type of flow that occurs in a cavity or enclosure. In Computational Fluid Dynamics, cavity flow is an important type of flow as it can be used to study the behavior of fluids in confined spaces. Related terms include lid driven cavity flow, which is a type of flow that occurs in a cavity with a moving lid, and natural convection, which is a type of flow that occurs due to buoyancy forces.

Cell-centered is a type of numerical method that uses the center of a computational cell to store the values of the solution. In Computational Fluid Dynamics, cell-centered methods are used to solve the governing equations of fluid motion. Related terms include node-centered, which is a type of numerical method that uses the nodes of a computational mesh to store the values of the solution, and face-centered, which is a type of numerical method that uses the faces of a computational cell to store the fluxes of the solution.

Chaotic is a type of behavior that is exhibited by a fluid when it is turbulent. In Computational Fluid Dynamics, chaotic behavior is important as it can affect the predictability of the solution. Related terms include unsteady, which is a type of flow that changes with time, and transient, which is a type of flow that changes with time but eventually reaches a steady state.

Characteristic is a curve or surface that is used to analyze the behavior of a fluid. In Computational Fluid Dynamics, characteristics are used to study the waves that propagate through a fluid. Related terms include method of characteristics, which is a numerical method that uses characteristics to solve the governing equations of fluid motion, and Riemann problem, which is a problem that is used to study the behavior of a fluid at a discontinuity.

Column is a type of data structure that is used to store the values of a solution. In Computational Fluid

Dynamics, columns are used to store the values of the solution at different locations in space. Related terms include row, which is a type of data structure that is used to store the values of a solution at different times, and array, which is a type of data structure that is used to store the values of a solution in a compact form.

Computational domain is the region of space where a problem is solved. In Computational Fluid Dynamics, the computational domain is an important quantity as it can affect the accuracy and efficiency of the solution. Related terms include boundary, which is the edge of the computational domain, and mesh, which is a discretization of the computational domain into smaller cells.

Computational fluid dynamics is the branch of engineering that deals with the simulation of fluid flow and heat transfer using numerical methods. In Computational Fluid Dynamics, the goal is to develop models and solve the governing equations of fluid motion to predict the behavior of fluids in different situations. Related terms include fluid mechanics, which is the branch of physics that deals with the behavior of fluids, and heat transfer, which is the branch of engineering that deals with the transfer of heat between different bodies.

Conservation law is a principle that states that a certain quantity, such as mass or energy, is conserved in a system. In Computational Fluid Dynamics, conservation laws are used to develop the governing equations of fluid motion. Related terms include continuity equation, which is a conservation law that states that mass is conserved in a system, and momentum equation, which is a conservation law that states that momentum is conserved in a system.

Control volume is a region of space that is used to analyze the behavior of a fluid. In Computational Fluid Dynamics, control volumes are used to develop the governing equations of fluid motion. Related terms include system, which is a collection of control volumes, and subsystem, which is a part of a system.

Convergence is the process by which a numerical method approaches the exact solution of a problem. In Computational Fluid Dynamics, convergence is an important quantity as it can affect the accuracy of the solution. Related terms include stability, which is the ability of a numerical method to resist disturbances or perturbations, and consistency, which is the ability of a numerical method to approximate the governing equations of fluid motion.

Coordinate system is a frame of reference that is used to describe the position of a point in space. In Computational Fluid Dynamics, coordinate systems are used to develop the governing equations of fluid motion. Related terms include Cartesian coordinate system, which is a coordinate system that uses rectangular coordinates to describe the position of a point, and cylindrical coordinate system, which is a coordinate system that uses polar coordinates to describe the position of a point.

Courant number is a dimensionless number that is used to analyze the stability of a numerical method. In Computational Fluid Dynamics, the Courant number is an important quantity as it can affect the accuracy and stability of the solution. Related terms include CFL condition, which is a condition that must be satisfied for a numerical method to be stable, and time step, which is the size of the time increment used in a numerical method.

Data structure is a way of organizing data in a computer. In Computational Fluid Dynamics, data structures are used to store the values of the solution and to perform operations on the data. Related terms include array, which is a type of data structure that is used to store the values of a solution in a compact form, and linked list, which is a type of data structure that is used to store the values of a solution in a dynamic form.

Differential equation is a mathematical equation that describes the behavior of a quantity that changes with space or time. In Computational Fluid Dynamics, differential equations are used to develop the governing equations of fluid motion. Related terms include ordinary differential equation, which is a differential equation that involves a single independent variable, and partial differential equation, which is a differential equation that involves multiple independent variables.

Diffusion is the process by which a quantity, such as heat or a tracer, is transported through a fluid due to random motions of the fluid molecules. In Computational Fluid Dynamics, diffusion is an important process as it can affect the behavior of a fluid and the transfer of quantities. Related terms include conduction, which is the process by which heat is transported through a fluid due to collisions between fluid molecules, and advection, which is the process by which a quantity is transported through a fluid due to the motion of the fluid itself.

Discretization is the process of approximating a continuous problem using a discrete set of values. In Computational Fluid Dynamics, discretization is an important step as it can affect the accuracy and efficiency of the solution. Related terms include grid, which is a discretization of the computational domain into smaller cells, and mesh, which is a discretization of the computational domain into smaller elements.

Divergence is a mathematical operation that is used to describe the behavior of a vector field. In Computational Fluid Dynamics, divergence is an important quantity as it can affect the behavior of a fluid and the transfer of quantities. Related terms include curl, which is a mathematical operation that is used to describe the rotation of a vector field, and gradient, which is a mathematical operation that is used to describe the rate of change of a scalar field.

Domain is the region of space where a problem is solved. In Computational Fluid Dynamics, the domain is an important quantity as it can affect the accuracy and efficiency of the solution. Related terms include boundary, which is the edge of the domain, and mesh, which is a discretization of the domain into smaller elements.

Double precision is a format for storing numbers in a computer. In Computational Fluid Dynamics, double precision is an important quantity as it can affect the accuracy of the solution. Related terms include single precision, which is a format for storing numbers in a computer that uses fewer bits than double precision, and integer, which is a format for storing whole numbers in a computer.

Eigenvalue is a value that is used to describe the behavior of a system. In Computational Fluid Dynamics, eigenvalues are used to analyze the stability of a numerical method. Related terms include eigenvector, which is a vector that is used to describe the direction of a system, and spectrum, which is a set of eigenvalues that are used to describe the behavior of a system.

Element is a small part of a domain that is used to discretize a problem. In Computational Fluid Dynamics, elements are used to develop the governing equations of fluid motion. Related terms include node, which is a point that is used to store the values of a solution, and face, which is a surface that is used to store the fluxes of a solution.

Energy is a quantity that is used to describe the ability of a system to do work. In Computational Fluid Dynamics, energy is an important quantity as it can affect the behavior of a fluid and the transfer of quantities. Related terms include kinetic energy, which is the energy of motion of a fluid, and potential energy, which is the energy of position of a fluid.

Equation is a statement that is used to describe the behavior of a system. In Computational Fluid Dynamics, equations are used to develop the governing equations of fluid motion. Related terms include algebraic equation, which is an equation that involves only algebraic operations, and differential equation, which is an equation that involves differential operations.

Error is a difference between the exact solution of a problem and the approximate solution obtained using a numerical method. In Computational Fluid Dynamics, error is an important quantity as it can affect the accuracy of the solution. Related terms include accuracy, which is the ability of a numerical method to approximate the exact solution of a problem, and stability, which is the ability of a numerical method to resist disturbances or perturbations.

Finite difference is a numerical method that is used to solve the governing equations of fluid motion. In Computational Fluid Dynamics, finite difference methods are used to discretize the computational domain into smaller cells. Related terms include finite element, which is a numerical method that is used to solve the governing equations of fluid motion using a variational formulation, and spectral method, which is a numerical method that is used to solve the governing equations of fluid motion using a Fourier decomposition.

Finite element is a numerical method that is used to solve the governing equations of fluid motion. In Computational Fluid Dynamics, finite element methods are used to discretize the computational domain into smaller elements. Related terms include finite difference, which is a numerical method that is used to solve the governing equations of fluid motion using a grid-based formulation, and boundary element, which is a numerical method that is used to solve the governing equations of fluid motion using a boundary-based formulation.

Flow is the motion of a fluid. In Computational Fluid Dynamics, flow is an important quantity as it can affect the behavior of a fluid and the transfer of quantities. Related terms include laminar flow, which is a type of flow that is smooth and continuous, and turbulent flow, which is a type of flow that is chaotic and random.

Flux is a quantity that is used to describe the rate of transfer of a quantity, such as mass or energy, through a surface. In Computational Fluid Dynamics, fluxes are used to develop the governing equations of fluid motion. Related terms include convection flux, which is the flux of a quantity due to the motion of the fluid, and diffusion flux, which is the flux of a quantity due to random motions of the fluid molecules.

Force is a quantity that is used to describe the influence of one object on another. In Computational Fluid Dynamics, forces are used to develop the governing equations of fluid motion. Related terms include pressure force, which is the force exerted by a fluid on a surface, and viscous force, which is the force exerted by a fluid on a surface due to friction.

Fourier analysis is a mathematical technique that is used to decompose a function into its component frequencies. In Computational Fluid Dynamics, Fourier analysis is used to analyze the behavior of a fluid and the transfer of quantities. Related terms include Fourier series, which is a representation of a function as a sum of sinusoidal functions, and Fourier transform, which is a representation of a function as a function of frequency.

Grid is a discretization of the computational domain into smaller cells. In Computational Fluid Dynamics, grids are used to develop the governing equations of fluid motion. Related terms include mesh, which is a discretization of the computational domain into smaller elements, and node, which is a point that is used to store the values of a solution.

Heat transfer is the process by which heat is transferred from one body to another. In Computational Fluid Dynamics, heat transfer is an important process as it can affect the behavior of a fluid and the transfer of quantities. Related terms include conduction, which is the process by which heat is transferred through a fluid due to collisions between fluid molecules, and convection, which is the process by which heat is transferred through a fluid due to the motion of the fluid.

Hybrid is a combination of different methods or techniques. In Computational Fluid Dynamics, hybrid methods are used to combine the advantages of different numerical methods. Related terms include finite difference, which is a numerical method that is used to solve the governing equations of fluid motion using a grid-based formulation, and finite element, which is a numerical method that is used to solve the governing equations of fluid motion using a variational formulation.

Hydrodynamics is the branch of physics that deals with the behavior of fluids in motion. In Computational Fluid Dynamics, hydrodynamics is an important field as it can affect the behavior of a fluid and the transfer of quantities. Related terms include hydrostatics, which is the branch of physics that deals with the behavior of fluids at rest, and fluid mechanics, which is the branch of physics that deals with the behavior of fluids in motion.

Hydrostatic is a condition that is used to describe the behavior of a fluid at rest. In Computational Fluid Dynamics, hydrostatic conditions are used to develop the governing equations of fluid motion. Related terms include hydrodynamics, which is the branch of physics that deals with the behavior of fluids in motion, and fluid mechanics, which is the branch of physics that deals with the behavior of fluids in motion.

Incompressible is a condition that is used to describe the behavior of a fluid that does not change density with pressure. In Computational Fluid Dynamics, incompressible conditions are used to develop the governing equations of fluid motion. Related terms include compressible, which is a condition that is used to describe the behavior of a fluid that changes density with pressure, and barotropic, which is a condition that is used to describe the behavior of a fluid that has a constant density.

Initial condition is a condition that is used to describe the behavior of a fluid at the start of a simulation. In Computational Fluid Dynamics, initial conditions are used to develop the governing equations of fluid motion. Related terms include boundary condition, which is a condition that is used to describe the behavior of a fluid at the borders of the computational domain, and interface condition, which is a condition that is used to describe the behavior of a fluid at the interface between two different materials or fluids.

Instability is a condition that is used to describe the behavior of a fluid that is unstable. In Computational Fluid Dynamics, instability is an important quantity as it can affect the behavior of a fluid and the transfer of quantities. Related terms include stability, which is a condition that is used to describe the behavior of a fluid that is stable, and bifurcation, which is a condition that is used to describe the behavior of a fluid that is sensitive to small changes in the parameters.

Interface is a surface that separates two different materials or fluids. In Computational Fluid Dynamics, interfaces are used to develop the governing equations of fluid motion. Related terms include boundary, which is the edge of the computational domain, and contact line, which is the line that separates two different materials or fluids.

Interpolation is a technique that is used to estimate the value of a function at a point that is not a node. In Computational Fluid Dynamics, interpolation is used to estimate the values of the solution at points that are not nodes. Related terms include extrapolation, which is a technique that is used to estimate the value of a function at a point that is outside the range of the data, and regression, which is a technique that is used to estimate the value of a function using a model.

Iteration is a process that is used to solve a problem by repeating a sequence of steps. In Computational Fluid Dynamics, iteration is used to solve the governing equations of fluid motion. Related terms include convergence, which is the process by which a numerical method approaches the exact solution of a problem, and residual, which is the difference between the exact solution of a problem and the approximate solution obtained using a numerical method.

Kinematics is the branch of physics that deals with the motion of objects without considering the forces that cause the motion. In Computational Fluid Dynamics, kinematics is an important field as it can affect the behavior of a fluid and the transfer of quantities. Related terms include dynamics, which is the branch of physics that deals with the motion of objects and the forces that cause the motion, and statics, which is the branch of physics that deals with the behavior of objects at rest.

Laminar is a type of flow that is smooth and continuous. In Computational Fluid Dynamics, laminar flow is an important type of flow as it can affect the behavior of a fluid and the transfer of quantities. Related terms include turbulent, which is a type of flow that is chaotic and random, and transitional, which is a type of flow that is between laminar and turbulent.

Linear is a type of equation that can be written in the form of a straight line. In Computational Fluid Dynamics, linear equations are used to develop the governing equations of fluid motion. Related terms include nonlinear, which is a type of equation that cannot be written in the form of a straight line, and quadratic, which is a type of equation that can be written in the form of a parabola.

Lithosphere is the outer layer of the Earth that is composed of rock. In Computational Fluid Dynamics, the lithosphere is an important component as it can affect the behavior of a fluid and the transfer of quantities.

Related terms include atmosphere, which is the layer of gases that surrounds the Earth, and hydrosphere, which is the layer of water that covers the Earth.

Mach number is a dimensionless number that is used to describe the speed of a fluid. In Computational Fluid Dynamics, the Mach number is an important quantity as it can affect the behavior of a fluid and the transfer of quantities. Related terms include Reynolds number, which is a dimensionless number that is used to describe the viscosity of a fluid, and Prandtl number, which is a dimensionless number that is used to describe the diffusivity of a fluid.

Mass is a quantity that is used to describe the amount of matter in an object. In Computational Fluid Dynamics, mass is an important quantity as it can affect the behavior of a fluid and the transfer of quantities. Related terms include density, which is the mass per unit volume of a fluid, and momentum, which is the product of the mass and velocity of a fluid.

Matrix is a mathematical object that is used to describe the behavior of a system. In Computational Fluid Dynamics, matrices are used to develop the governing equations of fluid motion. Related terms include vector, which is a mathematical object that is used to describe the direction and magnitude of a quantity, and tensor, which is a mathematical object that is used to describe the relationships between different quantities.

Mesh is a discretization of the computational domain into smaller elements. In Computational Fluid Dynamics, meshes are used to develop the governing equations of fluid motion. Related terms include grid, which is a discretization of the computational domain into smaller cells, and node, which is a point that is used to store the values of a solution.

Method of characteristics is a numerical method that is used to solve the governing equations of fluid motion. In Computational Fluid Dynamics, the method of characteristics is used to solve the hyperbolic equations that describe the behavior of a fluid. Related terms include finite difference, which is a numerical method that is used to solve the governing equations of fluid motion using a grid-based formulation, and finite element, which is a numerical method that is used to solve the governing equations of fluid motion using a variational formulation.

Mixing is the process by which a fluid is mixed with another fluid or substance. In Computational Fluid Dynamics, mixing is an important process as it can affect the behavior of a fluid and the transfer of quantities. Related terms include diffusion, which is the process by which a quantity is transported through a fluid due to random motions of the fluid molecules, and advection, which is the process by which a quantity is transported through a fluid due to the motion of the fluid itself.

Model is a mathematical representation of a system. In Computational Fluid Dynamics, models are used to develop the governing equations of fluid motion. Related terms include simulation, which is a computer program that is used to solve the governing equations of fluid motion, and experiment, which is a physical test that is used to validate the results of a simulation.

Momentum is the product