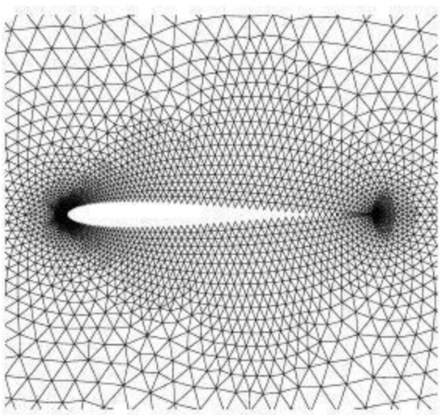
DEPARTMENT OF AEROSPACE ENGINEERING

Faculty Name	:	**Dr.A.Arun Negemiya, ASP/ Aero**	Academic Year	:	**2024-2025 (Even)**
Year & Branch	:	**III AEROSPACE**	Semester	:	**VI**
Course	:	**19ASB304 - Computational Fluid Dynamics for Aerospace Application**			

UNIT I - FUNDAMENTAL CONCEPTS

Meshless Methods

Meshless methods in CFD refer to numerical techniques used to solve fluid dynamics problems without the need for a predefined mesh, relying instead on scattered nodes distributed throughout the domain to approximate the solution, offering advantages for complex geometries and moving boundaries where mesh generation can be challenging; prominent meshless methods include Smoothed Particle Hydrodynamics (SPH), Element-Free Galerkin Method (EFG), and Reproducing Kernel Particle Method (RKPM).



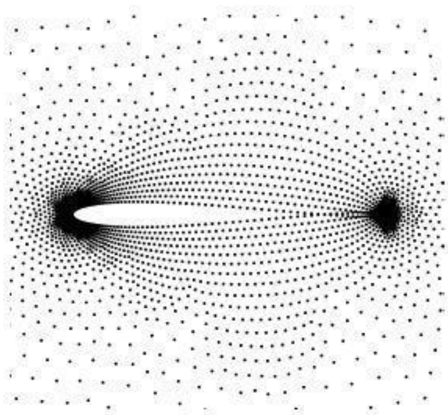


Image shows an airfoil which is meshed (Left) and the same airfoil which is surrounded by points (right) partial differential equations are solved on these points in meshless methods

Key points about meshless methods in CFD:

- **No mesh required:**

  Unlike traditional CFD methods like finite volume or finite element, meshless methods do not require a pre-defined mesh to discretize the domain, allowing for greater flexibility in complex geometries.

- **Node distribution:**

  Instead of a mesh, the domain is represented by a set of scattered nodes distributed across the problem area, including boundaries.

- **Interpolation functions:**

  To approximate the solution at any point, meshless methods utilize interpolation functions based on the values at neighboring nodes, often using kernel functions with a "support domain" to determine which nodes contribute to the calculation.

- **Weak form formulation:**

  Most meshless methods use a weak form of the governing equations, which involves integrating the equations over the domain, allowing for greater numerical stability.

Common Meshless Methods in CFD:

Smoothed Particle Hydrodynamics (SPH) :

SPH, is one of the oldest mesh-less method first used in astrophysics, and thereafter is now increasingly used in fluid flow studies. In this method node points are treated as physical particles with mass and density that can move around over time. Value of any property or its derivative is independent of values in adjacent particles in this method. Particles can be used in any order and doesn't matter if the particles move around or even exchange places. The basic step of the method for domain discretization, field function approximation and numerical solution can be summarized as follows:

1. The continuum is decomposed into a set of arbitrarily distributed particles with no connectivity (meshfree);
2. The integral representation method is adopted for field function approximation;
3. Particle approximation is introduced for converting integral representation into finite summation.

Radial Basis Functions (RBF) :

The development of RBFs into a mesh free method for solving partial differential equations arises from the recognition that a radial basis function interpolant can be smooth and accurate on any set of nodes in any dimension. RBF, is a function whose value depends only on distance from origin or any other specified point and are the means to approximate multivariable functions by linear combinations of terms based on a single univariable function (the radial basis function). These are usually applied to approximate functions or data which are only known at a finite number of points (or too difficult to evaluate otherwise). Some commonly used types of RBF's are

- Gaussian
- Multiquadric
- Inverse quadric
- Inverse multiquadric.

Finite Pointset Method (FPM):

FPM, is a particle method that uses Lagrangian approach and in this method the fluid is replaced by finite number of particles (points) that are non-stationary particles. These particles move with fluid velocities and carry fluid quantities like density, velocity, pressure. Similarly boundaries can be approximated by finite number of boundary particles and boundary conditions can be prescribed on them. As seen in SPH method, FPM also does not use rigid neighborhood list for a certain node/point (as in case of FVM). So all points/ particles are allowed to move and neighborhood list is recomputed at every time step. This method is suitable for flows with complicated geometry, free surface flows, multiphase flows.

FPM method has some advantages over the widely used mesh free method, the SPH method. The main difficulty with SPH method is incorporation of boundary conditions. In FPM method this difficulty is taken care by using moving least square or least square approach in which the boundary conditions can be implemented in a natural way by just placing the particles on boundaries and prescribing boundary conditions on them.

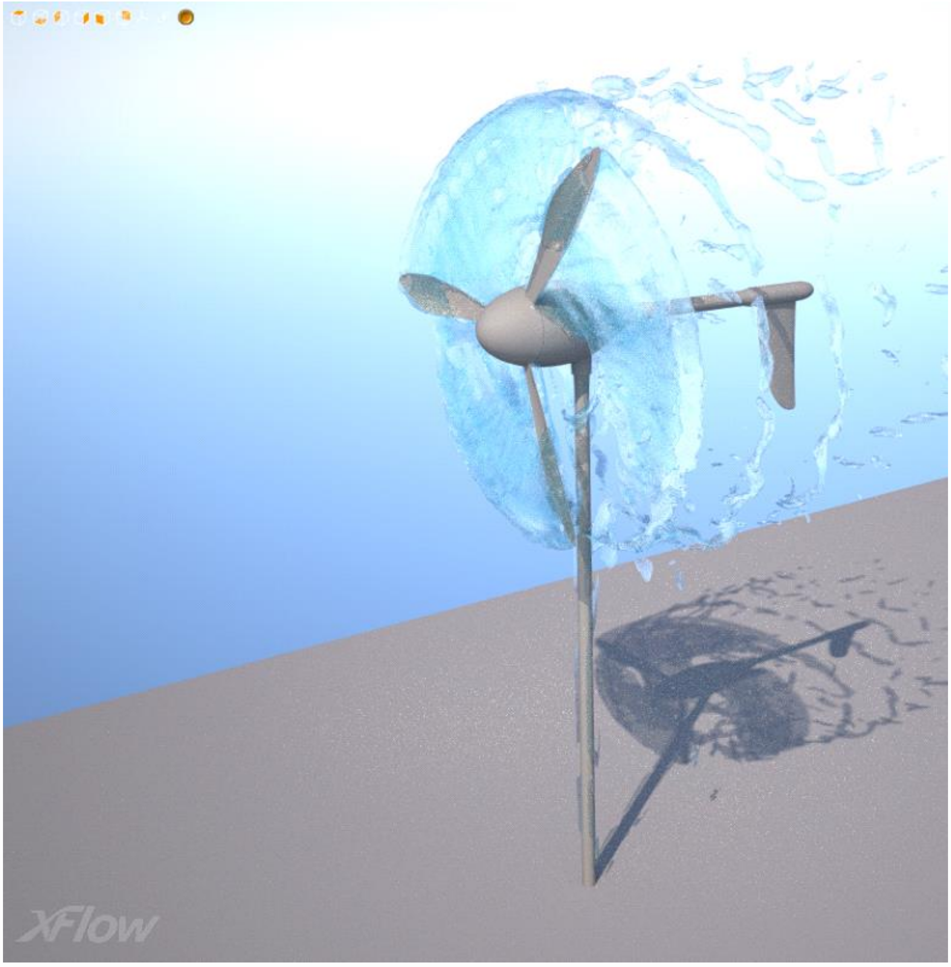
Some Commercial Meshless CFD software's

Let us get introduced to some commercial software available for meshless CFD.

XFlow:

It is one of the commercially available meshless software from *Next Limit Dynamics*. It uses Lattice Boltzmann equations and meshless particles based kinetic solver. The capabilities of XFlow are for solving following problems:

- Moving boundary problems
- Multiphase flows
- Fluid structure interactions
- Transient analysis
- Large eddy simulation
- Acoustics
- Non-Newtonian flows



Wind turbine CFD simulation (Source: XFlow)

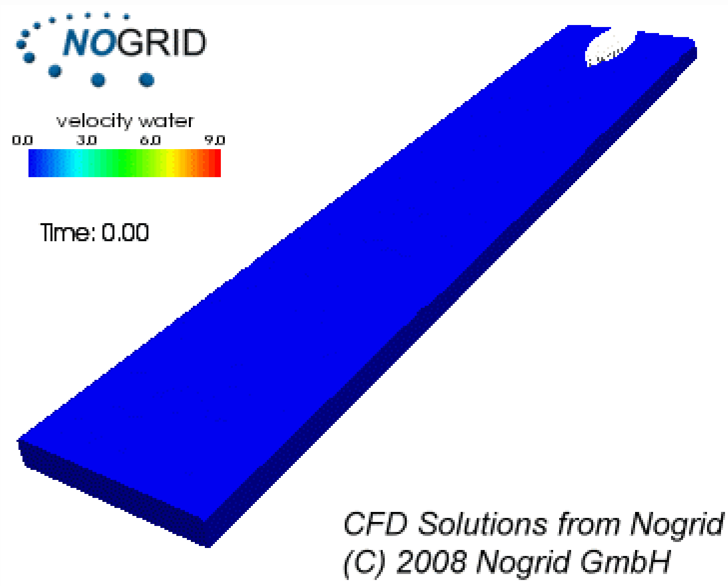
XFlow simulation can assess the efficiency of the turbine and predict loads on blades, wake turbulence intensity, or interference effects in wind farms.

NOGRID:

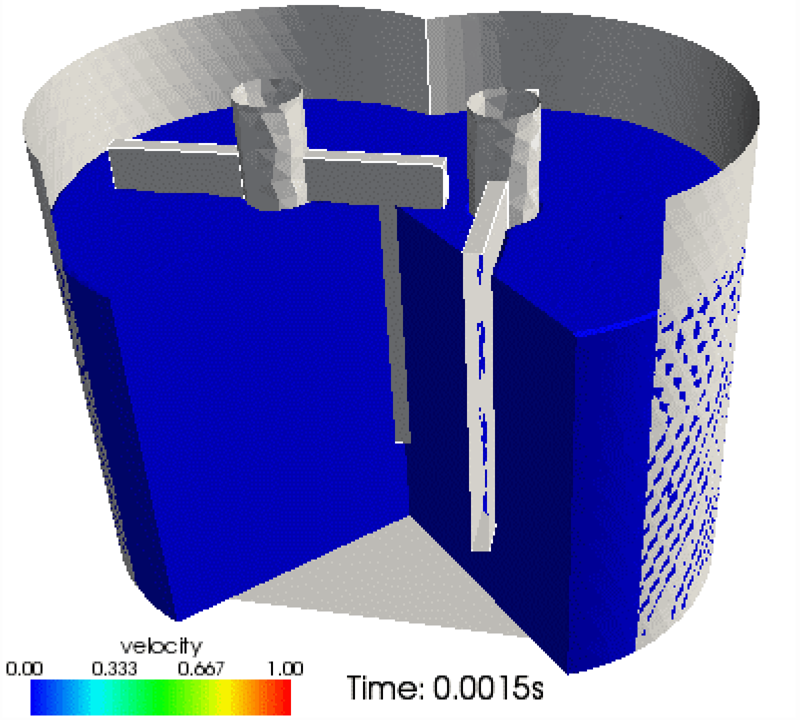
NOGRID, founded in the year 2006 and since then in use for CFD analysis. This meshless software in NOGRID uses Finite pointset method and Navier-Stokes equation to solve CFD problems. Special capabilities of this software are for:

- Multiphase flows
- Non-Newtonian flows
- Fluid structure interactions

NOGRID software is advantageous in situations where grid-based methods reach their limits due to the necessary remeshing. Using the fast and robust NOGRID solver the usual long modeling and computing times can be shortened substantially.



Animation shows velocity contours of CFD simulation of accelerated boat, created using NOGRID. (Source: NOGRID)



Animation shows velocity contours of CFD simulation of mixing tank, created using NOGRID. (Source: NOGRID)

DualSPHysics:

This software is based on Smoothed-particle hydrodynamics method and is developed to study free-surface flow phenomena where Eulerian methods can be difficult to apply, such as waves or impact of dam-breaks on off-shore structures. It is an *open source* software product developed from collaborative effortsof researchers at the *Johns Hopkins University (US)*, *University of Vigo (Spain)* and the *University Of Manchester (UK)*. This software can also solve problems using GPU (Graphics processing unit). Following video shows an example of Floating-rigid body interactions and an example of visualization using DualSPHysics.

Algodoo:

Algodoo is developed by Algoryx Simulation AB. It is a 2 D simulation framework for education purpose which uses SPH method. It is very easy to use and can create good visualizations for education purpose and learn physics. It is a very good tool for science teachers and students. This tool is not a like a high end CFD solver which gives you very accurate results. But it is a very good software for visualizing physics and doing many different simulations in very less time.

Along with fluid simulations Algodoo also supports some other physics like mechanics, optics. The video shows a various fluid simulations done using Algodoo and some of the simulations are done using other products of Algoryx.

Meshless techniques save a lot of time and efforts required for meshing. Meshless CFD technique is flourishing and its future looks bright as it is a remedy on one of the biggest difficulties faced by every CFD engineer i.e “Meshing”

Advantages of Meshless Methods:

- **Complex geometries:** Can handle intricate geometries without the need for complex mesh generation.
- **Moving boundaries:** Well-suited for problems with large deformations or moving boundaries.
- **Crack propagation:** Can readily simulate crack propagation due to the ability to adjust node distribution dynamically.

Challenges of Meshless Methods:

- **Computational cost:** Can be computationally expensive due to the need to calculate interpolation functions for each node at every iteration.
- **Boundary conditions:** Implementing boundary conditions accurately can be challenging.
- **Stability issues:** May require careful selection of parameters to maintain numerical stability.