Behaviour of Structure with Interaction of Fluid

Dr. Suresh Kumar Sahani[1], Kameshwar Sahani[2]
Lecturer[1, 2]
Department of Mathematics[1], Department of Civil & Geomatics Engineering[2]
Model Multiple College, Janakpur, Nepal[1]
Kathmandu University, Dhulikhel, Nepal[2]

Abstract:
This research develops a method through which a complex multi-physics problem can be simulated. It also deals with the interactions between incompressible fluid flows and immersed structures are nonlinear multi-physics phenomena that have applications to a wide range of Scientific and engineering disciplines. There are two equations namely Equation of Elasticity through which we get displacement and the other is Navier–Stokes equations which gives the flow velocity of fluid.

Keywords: Dams, Fluid, mechanics, numerical, particles, structure, water, etc.

1. INTRODUCTION

The ultimate goal of every structural design process is to create a structure which can reliably withstand the external loadings, and being at the same time efficient in terms of material use. This is more crucial as for modern structural design light-weight structures are often the best choice. While the problem is rather simple when the loading is static or involves simple dynamics. In cases when the loading involves fluid (say water) in e.g., dams, bridge piers, and the structures in these behaves as an obstacle, and thus two mechanics', namely fluid and solid, need to be considered. This coupled problem can effectively be solved if both mechanics are considered simultaneously, thus a Fluid-Structure-Interaction (FSI) numerical methods have to be utilized. The work by **D. Brown et al., (2008)**on Applied Mathematics at the U.S. Department of Energy: Past, Present, and a view to the Future emphasize these challenges: "Today's problems, unlike traditional science and engineering, do not involve physical processes covered by a single traditional discipline of physics or the associated mathematics. Complex systems encountered in virtually all applications of interest to DOE involve many distinct physical processes. The issue of coupling models of different events at different scales and governed by different physical laws is largely wide open and represents an enormously challenging area for future research.

2. MULTI-PHYSICS

A multi-physics system consists of more than one component governed by its own principle(s) for evolution or equilibrium, typically conservation or constitutive laws. A major classification in such systems is whether the coupling occurs in the bulk (e.g., through source terms or constitutive relations that are active in the overlapping domains of the individual components) or whether it occurs over an idealized interface that is lower dimensional or a narrow buffer zone (e.g., through boundary conditions that transmit fluxes, pressures, or displacements). Typical examples of bulk-coupled multi-physics systems with their own extensively developed literature include radiation with hydrodynamics in astrophysics (radiation-hydrodynamics, or "rad-hydro"), electricity and magnetism with hydrodynamics in plasma physics (magneto hydrodynamics), and chemical reaction with transport in combustion or subsurface flows (reactive transport)**[1].** Typical examples of interface-coupled multi-physics systems are ocean-atmosphere dynamics in geophysics, fluid-structure dynamics in aero-elasticity. Beyond these classic multi-physics systems are many others that share important structural features. Success in simulating forward models leads to ambitions for inverse problems, sensitivity analysis, uncertainty quantification, model-constrained optimization, and reduced-order modeling, which tend to require many forward simulations**[2and 3].** In these advances, the physical model is augmented by variables other than the primitive quantities in which the govern equations are defined. These variables may be probability density functions, sensitivity gradients, Lagrange multipliers, or coefficients of system-adaptive bases. Equations that govern the evolution of these auxiliary dependent variables are often derived and solved together with some of the physical variables. When the visualization is done in situ with the simulation, additional derived quantities may be carried along. Error estimation fields in adaptive meshing applications may constitute yet more. Though the auxiliary variables may not be "physical" in the standard sense, they give the overall simulation the structure of multi-physics. In another important class of systems that might fall under the rubric of multi-physics by virtue of being multi-scale, the same component is described by more than one formulation, typically with a computationally defined boundary or transition zone between the domains of applicability. We refer, for example, to field-particle descriptions of N-body systems in celestial mechanics or molecular dynamics, in which the gravitational or electrical forces between particles that are not immediate neighbors are mediated by a field that arises from the particles themselves **[5].**

3. DISCUSSION:

The momentum and continuity equations can be written as

$$\rho\left(\frac{\partial \vec{u}}{\partial t} + (\vec{u}.\nabla)\vec{u}\right) - \nabla.\bar{\bar{\sigma}} = \rho\,\vec{f} \qquad \ldots(1)$$

and

$$\left(\frac{\partial \rho}{\partial t} + (\vec{u} \cdot \nabla)\rho\right) + \rho(\nabla \cdot \vec{u}) = 0 \ldots (2)$$

Where, $\bar{\bar{\sigma}}$ is the stress tensor, For Newtonian fluids

$$\bar{\bar{\sigma}} = 2\mu\bar{\bar{\varepsilon}} - \frac{2}{3}\mu(\nabla \cdot \vec{u})\bar{\bar{I}} - p\bar{\bar{I}} \qquad \ldots (3)$$

Where, μ is the viscosity, p is the pressure, I the unit tensor and "the linearized strain rate tensor, i.e.

$$\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad \ldots (4)$$

The density of an ideal gas depends on the pressure and temperature through the equation of state

$$\rho = \frac{p}{RT} \qquad \ldots (5)$$

Where, R is the gas constant:

$$R = \frac{\gamma - 1}{\gamma} C_P \qquad \ldots (6)$$

The specific heat ratio γ is defined as

$$\gamma = \frac{c_P}{c_v} \ldots (7)$$

Where, c_p and c_v are the heat capacities in constant pressure and volume, respectively. The value of depends solely on the internal molecular properties of the gas. An incompressible flow is characterized by the condition p=constant, from which it follows:

$$\nabla \cdot \vec{u} = 0 \ldots (8)$$

Enforcing the constraint (8) in (1), (2) and (3), the equations reduce to the Navier-Stokes equations.

$$\rho\left(\frac{\partial u}{\partial t} + (\vec{u} \cdot \nabla)\vec{u}\right) - \nabla \cdot (2\mu\bar{\bar{\varepsilon}}) + \nabla_{p = \rho\vec{f}} \ldots (9)$$

$$\nabla \cdot \vec{u} = 0 \ldots (10)$$

Compressible flows are modeled by the equations (1)-(7). Then, it is possible to replace the state equation (5) by

$$\rho = \frac{1}{c^2}p \ldots (11)$$

Where, c = c (p; T...) is the speed of sound. The equation (11) can be used with liquid materials as well. Most commonly the term $\rho\vec{f}$ represents a force due to gravity, in which case the vector $\vec{f}$ is the gravitational acceleration. It can also represent, for instance, the Lorentz force when magneto hydrodynamic effects are present. For isothermal flows the equations (9) and (10) describe the system in full. For thermal flows also the heat equation needs to be solved.

4. CASE DEFINITION

A simple cantilever beam with one end fixed and another free at a span of 1 m. The cross-section of beam is rectangular where width is 0.10 m and depth is 0.05 m. Material of the beam is such that young's modulus of elasticity is 10^9N/m^2. Force applied at the free end only which is 5500N. Assume the density of the material to be 550 Kg/m^3. Above are schematic diagram which visualizes the case definition of a cantilever beam problem and also a sectional and cross-sectional diagram which enlist the details of the beam dimensions and its material properties.
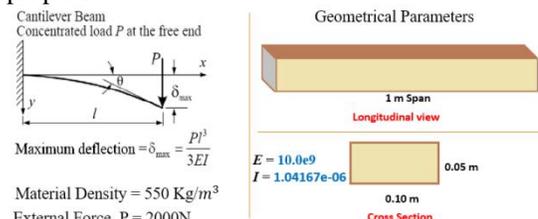

Figure.1. For a Cantilever Beam as shown in the figure a model mesh will be designed.

5. MODEL DESCRIPTION: CAD

Pre-Processing

- Generating Geometry
- For a Cantilever Beam as shown in the figure a model mesh will be designed.
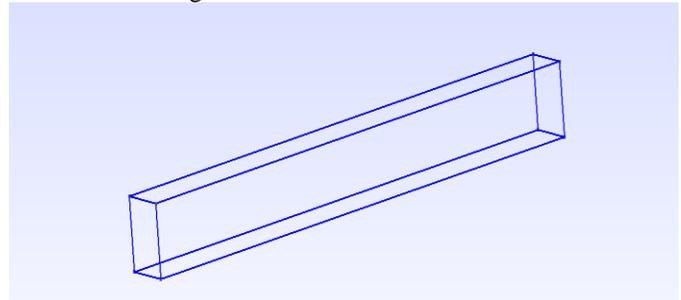

Figure .2. CAD: 3D view of a beam.

Above figure 1 shows a regular beam in three dimensions. It is a computer aided design in Gmsh software. It was drawn by first plotting four points or nodes such that it forms a rectangular shape of same dimensions as of the cross-section (0.1 m width and 0.05 m depth) when joined together with line element then it is expanded in the third dimension with 1 m span.

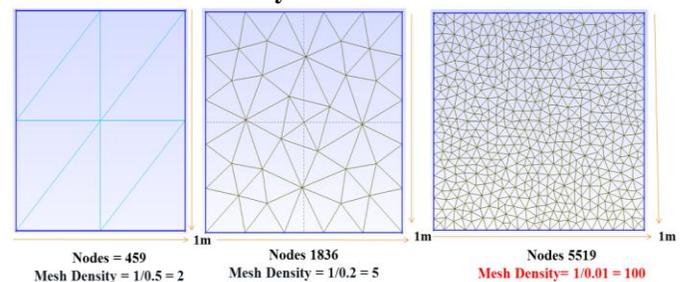**Selection of mesh density**

Figure. 3. Triangular Mesh of different sizes.

Mesh (Triangular) of different sizes was tested to observe the effect of Mesh size. Above in figure 3 from first mesh to third mesh, element size is of 0.5 m, 0.2 mand 0.01 m. Since the cross-section dimension is 1m by 1m thus mesh density came to be highest in the third mesh. All three mesh size were generated to the CAD (Computer Aided Design) and tested for the same parameters of material and applied load. Since we know in finite element analysis the results are as accurate as the density of mesh, it was expected to get high accuracy as we move toward first mesh density to third. The mesh density of 0.01 melement size was more accurate than 0.5 m and 0.2 m, as it was expected, it is verified now. Gmsh is capable of generating two dimension mesh, three dimension mesh, optimize it, partition it and many more. It can also generate structured mesh or called layered mesh.
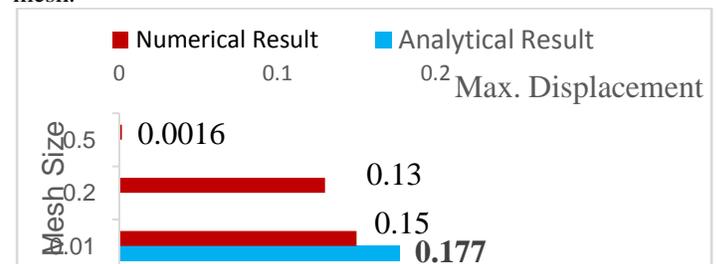

Maximum Displacement

Figure.4. Triangular Mesh sizes comparison and verification.

Selection of unstructured or non-layered mesh was done for the sake of simplicity and also to understand about unstructured mesh first as a beginner.
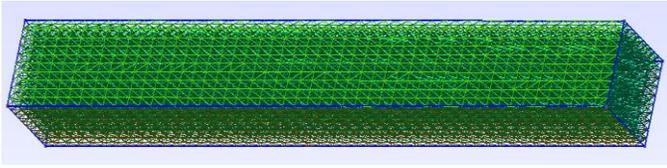

Figure .5. 3D Triangular Mesh in Gmsh
Mesh density 100 was selected, though more denser mesh can be taken and get more accurate result if only factors like time taken in generation of mesh, convergence, solution and post processing is not limited.

Model Parameters
The mesh generated in Gmh is saved in *.msh and then converted into Elmer mesh elements. Now there are many parameters like Young's modulus, Poisson ratio, density, displacement, etc. which needs to be defined in ElmerGUI software.

Constants
Gravity = 9.82
Stefan Boltzmann = 5.67e-08
Permittivity of Vacuum = 8.8542e-12
Boltzmann Constant = 1.3807e-23
Unit Charge = 1.602e-19

Solver 1
Equation = Linear elasticity
Variable = -doffs 3 Displacement
Steady State Convergence Tolerance = 1.0e-5
Nonlinear System Convergence Tolerance = 1.0e-7
Nonlinear System Max Iterations = 1
Nonlinear System Newton After Iterations = 3
Nonlinear System Newton After Tolerance = 1.0e-3
Nonlinear System Relaxation Factor = 1
Linear System Solver = Iterative
Linear System Iterative Method = GCR
Linear System Max Iterations = 500
Linear System Convergence Tolerance = 1.0e-10
BiCGstabl polynomial degree = 2
Linear System Preconditioning = ILU1
Linear System ILUT Tolerance = 1.0e-3
Linear System Abort Not Converged = False
Linear System Residual Output = 1
Linear System Precondition Recompute = 1

Equation 1
Name = "Elasticity"
Calculate Stresses = True
Active Solvers(1) = 1

Material 1
Name = "Pine"
Density = 550
Poisson ratio = 0.37
Young's modulus = 10.0e9

Body Force 1
Name = "Gravity"
Stress Body force 2 = -9.81 * 550

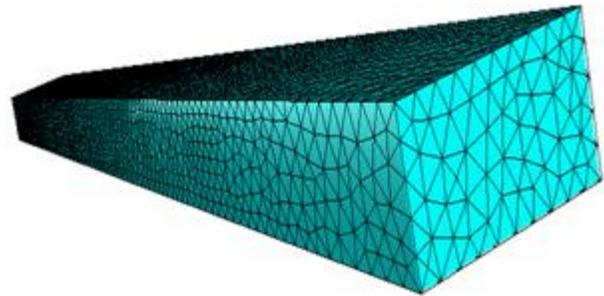### Boundary Condition 1
Target Boundaries(1) = 29
Name = "Wall"

Figure .6.Triangular meshes in Elmer
Displacement 3 = 0.0
Displacement 2 = 0.0
Displacement 1 = 0.0
Boundary Condition 2
Target Boundaries(1) = 30
Name = "Mass"
Force 2 = -11.0e5
Figure6is a three dimensional beam with finite element discredited triangular mesh. The Finite element mesh details are as given below:

Finite Element Mesh
Mesh dimension: 3
Leading element dimension: 3
Nodes: 5519
Volume elements: 21974
Surface elements: 7682
Edge elements: 11523
Point elements: 0

Bounding Box
X-coordinate: [0 , 0.1]
Y-coordinate: [0 , 0.05]
Z-coordinate: [0 , 1]

Volume Bodies
Body 28: 21974 volume elements
Undetermined: 0
Total: 1 volume bodies

Surface Boundaries
Boundary 29: 118 surface elements
Boundary 30: 118 surface elements
Boundary 31: 7446 surface elements
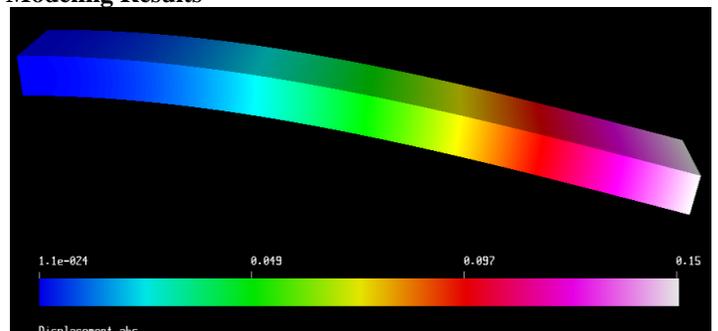
Modeling Results

Figure .7. Simulation of model in ElmerPost
The above figure 4 is a simulated view of result obtained in ElmerPost. The maximum displacement was obtained and similarly other results for different external force applied were plotted on excel sheet. The equivalence line was drawn by the analytical results obtained from formula of maximum displacement and it was set to determine the variation in the model results
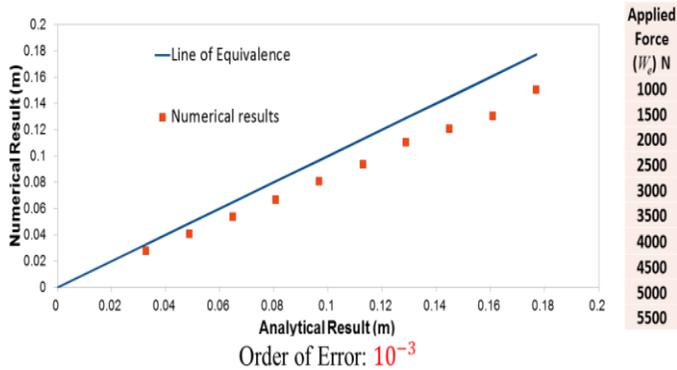
Figure.8. Accuracy in model result when compared to analytical result

6. FLUID AND STRUCTURE INTERACTION

Case Definition

A coupled case of fluid-structure interaction, flow is initiated at one end of a channel which has an elastic obstacle that bends under the influence of fluidic forces. The resulting displacements modify the domain thereby affecting the flow of the fluid. The channel is assumed to be 2D. The length of the channel is 10 m and the height is 2 m. At the distance of 2 m from the inlet sits an elastic beam with height of which is 0.9 m and width 0.2 m. A parabolic velocity profile with maximum velocity of 1 m/s is assumed. Material properties are assumed to be rather simple: For the structure the density is 2400Kg/m^3, Young's modulus is $15.81 * 10^9$ N/m^2, and Poisson ratio 0.3. For the fluid the density is 1Kg/m^{3}and viscosity is 1 poise. Additionally the fluid has elastic properties that are used to extend the displacement of the elastic beam to the fluid. The idea of the case definition is to demonstrate simple yet strongly coupled FSI case without getting into turbulence modeling. Realistic material parameters for the given size of the domain would easily result to turbulence and just small displacements.

Equations

For fluid and structure Interaction two simultaneous equations of structure and fluid are Equation of Elasticity and Navier–Stokes equation respectively. A number of parameters like viscosity, poison's ratio, structural support and pressure are kept constant but area of structure, material strength and inflow velocity are varied.

Equation of Elasticity

Load distribution, $W(x) = (d^2EI/dx^2) * (d^2y/dx^2)$

Where,
Y = Deflection;
I = Moment of Inertia
E = Young's Modulus
x= Length of Section
And
Navier–Stokes equation
$\rho[(\partial u/\partial t)+u . \nabla u]= -\nabla p + \nabla.TD + f$
Where,
T_D = Deviatory stress tensor; viscosity and flow velocity variable
f = volume density of the body forces acting on the fluid
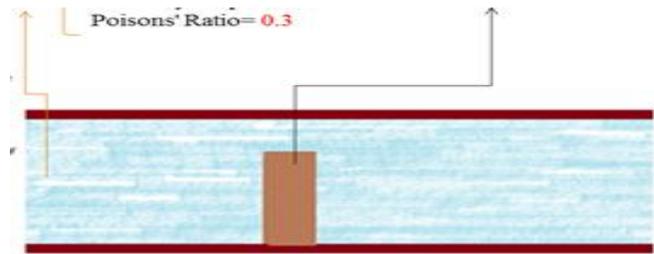∇= here is the del operator Structure

Figure .9.Problem Statement: Schematic Diagram.

Fluid assumed Value
Poisons ratio = 0.3
p = pressure is 1 atm
Elastic Modulus E = 1 N/m^2
Density = 1 Kg / m^3
Velocity = 1 poise
Structure (Assumed Values)
L = 0.9 m , W = 0.2
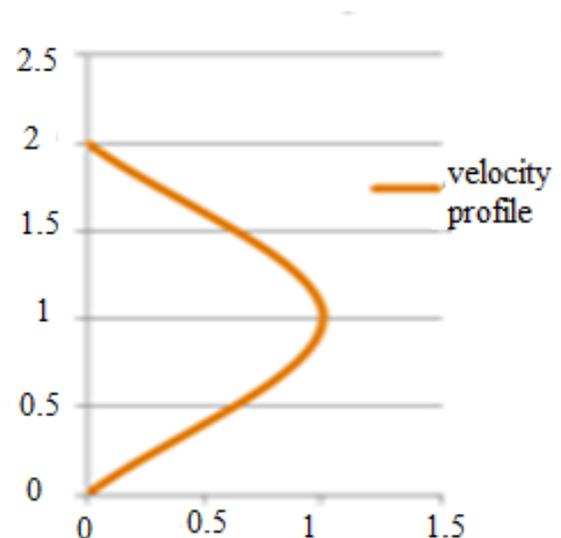Ratio = L / W = 4.5
Young's modulus E = 15.81 * 10^9 N / m^2
Density = 2400 kg / m^3
Poisson's ratio = 0.3

Velocity profile
Equation:x $(2 - x)$
Where x = depth of pipe
The case is solved using standard weak coupling with some relaxation to boost the convergence. The solution is steady-state so only the final results are will be studied.

7. CONCLUSIONS

The purpose of this study is to develop a method through which a complex multi-physics problem can be simulated. There are two equations namely Equation of Elasticity through which we get displacement and the other is Navier–Stokes equations which gives the flow velocity of fluid. Order of the error in model result is10^{-3}.Triangular mesh of size 0.01 m is most accurate in comparison to 0.2 m and 0.5 m, though more dense mesh can be used if only the computer memory allows.Gmsh is good in CAD as well as generating mesh. Elmer is good at solver and post processing part. At ratio 6 (1.2m by 0.2m) significant change in max. displacement is observed. Before ratio 6, max. displacement is incomparable in both flood and non-flood

scenario. Ratio 8.25 (1.65m by 0.2m) in flood velocity case is having $9.61 * 10^{-6}$ m of max .displacement closer to ratio 9 (1.8mby 0.2m) in non-flood case, $9.75 * 10^{-6}$ m. In comparison to non-flood, max.displacement in flood case for M10 grade concrete is 5 times more. From M10 to M80, max. Displacement decreases to 2.99 times. The slope of the negative curve obtained between grade of concrete and maximum displacement reduces sharply after M35, thus grade above M35 is recommended.

8. REFERENCES

[1]. **S. Balay, J. Brown, K. Buschelman, V. Eijkhout, W. D. Gropp, D. Kaushik, M. G. Knepley, L. C. McInnes, B. F. Smith, H. Zhang**, PETSc user's manual, Tech. Rep. ANL-95/11 - Revision 3.2, Argonne National Laboratory (2011).

[2]. **A. C. Hindmarsh, P. N. Brown, K. E. Grant, S. L. Lee, R. Serban, D. E. Shumaker, C. S. Woodward**, SUNDIALS: Suite of nonlinear and differential/algebraic equation solvers, ACM Trans. Math Softw. 31 (2005) 363–396.

[3]. **C. Woodward** et al., SUNDIALS Suite of Nonlinear and Differential Algebraic Equation Solvers.

[4]. **D. Brown, P. Messina** (Chairs), Scientific Grand Challenges: Crosscutting Technologies for Computing at the Exascale, Office of Science, U.S. Department of Energy (2010).

[5]. **H. Simon, T. Zacharia, R. Stevens** (Co-Chairs), Modeling and Simulation at the Exascale for Energy and the Environment, Office of Science, U.S. Department of Energy (2007).

[6]. **"Elmer – CSC"**. CSC — IT Center for Science Ltd. Retrieved (2010-06-24).