Coupled Calculations of the Nuscale FIV Tests: Pre-test Simulations and Sensitivity Analysis

Nuclear Science and Engineering Division

About Argonne National Laboratory

Argonne is a U.S. Department of Energy laboratory managed by UChicago Argonne, LLC under contract DE-AC02-06CH11357. The Laboratory's main facility is outside Chicago, at 9700 South Cass Avenue, Argonne, Illinois 60439. For information about Argonne and its pioneering science and technology programs, see www.anl.gov.

DOCUMENT AVAILABILITY

Online Access: U.S. Department of Energy (DOE) reports produced after 1991 and a growing number of pre-1991 documents are available free at OSTI.GOV (<http://www.osti.gov/>), a service of the US Dept. of Energy's Office of Scientific and Technical Information.

Reports not in digital format may be purchased by the public from the National Technical Information Service (NTIS):

U.S. Department of Commerce
National Technical Information Service
5301 Shawnee Rd
Alexandria, VA 22312
www.ntis.gov
Phone: (800) 553-NTIS (6847) or (703) 605-6000
Fax: (703) 605-6900
Email: **orders@ntis.gov**

Reports not in digital format are available to DOE and DOE contractors from the Office of Scientific and Technical Information (OSTI):

U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
www.osti.gov
Phone: (865) 576-8401
Fax: (865) 576-5728
Email: **reports@osti.gov**

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor UChicago Argonne, LLC, nor any of their employees or officers, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of document authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof, Argonne National Laboratory, or UChicago Argonne, LLC.

Coupled Calculations of the Nuscale FIV Tests: Pre-test Simulations and Sensitivity Analysis

E. Merzari, H. Yuan
Nuclear Engineering Division, Argonne National Laboratory

J. Solberg, D. Flowers, R. Ferencz
Methods Development Group, Lawrence Livermore National Laboratory

September 30, 2018

SUMMARY

The NEAMS program aims to develop an integrated multi-physics simulation capability “pellet-to-plant” for the design and analysis of future generations of nuclear power plants. In particular, the Reactor Product Line code suite's multi-resolution hierarchy is being designed to ultimately span the full range of length and time scales present in relevant reactor design and safety analyses, as well as scale from desktop to petaflop computing platforms.

Flow-induced vibration (FIV) is widespread problem in energy systems because they rely on fluid movement for energy conversion. Vibrating structures may be damaged as fatigue or wear occurs. Given the importance of reliable components in the nuclear industry, flow-induced vibration has long been a major concern in safety and operation of nuclear reactors. In particular, nuclear fuel rods and steam generators have been known to suffer from flow-induced vibration and related failures.

Advanced reactors, such as integral Pressurized Water Reactors (PWRs) considered for Small Modular Reactors (SMR), often rely on innovative component designs to meet cost and safety targets. One component that is the subject of advanced designs is the steam generator, some designs of which forego the usual shell-and-tube architecture in order to fit within the primary vessel.

A significant amount of data exists on flow-induced vibration in shell-and-tube heat exchangers, and heuristic methods are available to predict FIV based on a set of given assumptions. In contrast, advanced designs have far less data available. Advanced modeling and simulation based on coupled structural and fluid simulations have the potential to predict flow-induced vibration in a variety of designs, reducing the need for expensive experimental programs, especially at the conceptual design stage.

Over the past several years, the Reactor Product Line has developed a FIV capability in SHARP. SHARP contains high-fidelity single-physics codes Diablo for structural mechanics and Nek5000 for fluid mechanics calculations. Both codes are state-of-the-art, highly scalable tools that have been extensively validated. In previous reports we have discussed the implementation and validation of an FIV capability for helical steam generators and fuel assemblies.

In this report we discuss one-way coupled calculations performed with Nek5000 and Diablo for the NuScale FIV tests. In the turbulent fretting regime one-way coupling is judged to be sufficient because the pressure loads do not cause substantial displacements. It is also the most common source of vibration in helical steam generators per the low flows expected in integral PWRs. The simulations precede the tests, as experimental data was not available at the delivery of this report. Several assumptions had to be made on the support-tube interaction and multiple tests and sensitivity analysis have been performed. Additional work remains to be done and it is discussed in the future work section.

CONTENTS

SUMMARY	i
1 Introduction.....	1
2 Overview of High-Impact Project.....	2
3 Overview of Previous Validation Work.....	3
4 Methods	11
5 Coupled Simulation Results and Sensitivity Analysis	39
6 Conclusions.....	48
References.....	49

1 Introduction

Steam generators (SGs) are an essential component of all pressurized water reactor (PWR) designs. The cost of inadequate prediction of flow-induced vibration (FIV) phenomena within SGs can be crippling: SG units are expensive, and their replacement may cause extended downtimes. Moreover, the loss of public trust produced by radioactive leaks can lead to prolonged shutdowns and even plant closure. The emphasis on reliable SGs has increased in recent years with the growing interest in small modular reactors (SMRs).

The focus of this project is the development of a high-fidelity, finite-element analysis/computational fluid dynamics (FEA/CFD) approach to the simulation of FIV based on SHARP [1]. Flow-induced vibration is an important limiting factor in the operation of heat exchangers and SGs. Such vibration may cause tube failure because of collisions between oscillating tubes, attrition against support structures (wear), tightness faults against stiff joints, fastener loosening, or fatigue caused by cyclic loads. Hence, the study of FIV is of interest to nearly all vendors and for nearly all reactor designs. While empirical design methods and experience related to FIV might be adequately developed for typical tube bank geometries, design methods and experience related to FIV are far less developed for advanced SG designs, such as those currently considered for SMRs. As a result, numerical simulation or analytical prediction of FIV is even more important for SMRs. For instance, in helical heat exchangers (Fig. 1.1), vibrations can be caused by various phenomena whose relative importance depends on the flow parameters: turbulent pressure fluctuations, vortex shedding induced by cross-flows or, in extreme conditions, fluid-elastic instabilities, and potentially acoustic vibrations.

**Fig. 1.1 Coupled Nek5000-Diablo simulation of the flow in a helical steam generator.
Displacements in [m].**

An advanced numerical simulation capability for modeling such phenomena will help improve the analysis and evaluation of different design variants in terms of vibrations and heat transfer performance, thus complementing expensive experimental tests and reducing their cost, while developing a better

September 30th, 2018

understanding of the physics behind FIV. Work performed in the previous years have led to the development of such a tool in the multi-physics suite SHARP based on the structural code Diablo [2] and high-fidelity computational fluid dynamics code Nek5000 [3]. The FIV capability was verified on classic fluid-structure interactions (FSI) benchmark data [4].

In previous work we have also demonstrated the application of SHARP to legacy helical coil steam generator datasets [5], in preparation for its application to the NuScale design. We emphasize that these datasets refer to facilities that have been dismantled for over 30 years and the uncertainty for any high-fidelity model will be high. In fact, details such as the structure of the tube supports are practically unknown. Nonetheless the report and articles [6, 7] represent an important stepping stone toward the ultimate goal of simulating helical steam generators. The creation of a high-fidelity FSI solver is necessary because the semi-empirical approaches that have been applied to traditional heat exchangers are not applicable to helical coil SGs. Moreover, this high-fidelity approach has the potential for being much more accurate than lower-fidelity FEA/CFD technologies in dealing with FIV, given the complexity of the turbulent flows under investigation. We did not limit ourselves to one-way coupling and also extended more recently to two-way coupling demonstrating that one-way coupling produced very similar results to a fully coupled solution at low flow velocity [4]. At higher speed the two-way coupling remains the only method to evaluate displacements and the overall behavior. In fact, two-way coupling will be needed to simulate strongly coupled behavior such as FEI. A more detailed summary of previous findings is provided in **Section 3**.

In this report we describe our efforts to simulate the NuScale FIV tests for their design [8]. Previous efforts have been made to simulate the fluid only portion of the steam generator [9]. Here we provide a full scale simulation of the steam generator using a one-way approach similar to what implemented in our previous work [6]. Several assumptions had to be made and the overall methods are discussed in **Section 4**. Results including time series analysis, eigenvalue analysis and sensitivity analysis are presented in **Section 5**. While experimental data is not currently available for a comparison, it shows very modest displacements even at 120% flow loading. Additional studies and their potential impact are discussed in the **Conclusions and Future work** section.

2 Overview of High-Impact Project

The high-impact project that come to its conclusion with the delivery of this report consisted of seven tasks categorized under three main areas: code development, verification and validation, and demonstration. The verification and validation part is the largest part of the work and includes experimental activities at Texas A&M University (TAME), as well as code-to-code comparisons. Table 2.1 summarizes the tasks of the project.

This report represent the culmination of **Tasks D and E** with the simulation of the Nuscale tests.

We note that the verification and validation of the FSI coupling methodology were accomplished previously in **Task B**. This included only available datasets. This task involved four subtasks:

- a) Verification of the coupled methodology (solution transfer). This is part of the focus of this report.
- b) Verification/validation against fundamental FSI datasets/cases. It includes in particular verification/validation against fundamental cases (e.g., [10]) normally employed when developing fluid-structure interaction solvers in the literature.
- c) Validation on datasets specific to helical steam generators. This includes datasets for shell-and-

tube heat exchangers as well as helical SGs [5].

- d) Validation on other datasets relevant to other nuclear applications.

Table 2.1. Task table for the High Impact Problem project.

Task	Area	Title
A	<i>Development</i>	Implementation of an appropriate methodology in SHARP for fluid-structure interaction simulations.
B	<i>Verification & Validation</i>	Verification of the methodology implemented in SHARP for fluid-structure interaction simulations.
C	<i>Development</i>	Evaluation of different models for the simulation of the secondary.
D	<i>Verification & Validation</i>	Modeling and simulation of NuScale experiments related to flow-induced vibrations.
E	<i>Demonstration</i>	Demonstration of SHARP for flow-induced vibration in a helical steam generator and evaluation of different structural supports.
F	<i>Verification & Validation</i>	Simulation of flow-induced vibration using STAR-CCM+ (+ FEA code) for the NuScale steam generator design.
G	<i>Verification & Validation</i>	PIV and pressure experiments (at TAMU) Numerical simulation for the validation of the fluid mechanics models.

3 Overview of Previous Validation Work

In this section we discuss work performed under previous years as part of the high impact problem project with the aim of providing a succinct summary of the validation status. Extensive discussion is provided in other publications [4, 6, 7, 9].

3.1 Experiments at Texas A&M University and Simulations with Nek5000

Two experimental facilities are equipped with a Particle Image Velocimetry (PIV) system including a high-speed camera and a laser system with optical lenses. The test sections are presented in Figures 3.1 and 3.2. Note that adjacent columns of tubes are counterwrapping as common in recent designs. The upper structures were installed to homogenize the inlet flow, and the lower tanks were installed to reduce the flow oscillation caused by the pumps. Figure 3.3 presents the flow fields in 1-rod and 5-rod facilities measured by the PIV system in the measurement plane parallel to the streamwise direction. In these flow fields, Reynolds number at the inlet, Re_0 , was calculated by using the rod diameter, $d = 15.875$ mm, and the mean upstream approach velocity, $V_0 \sim 0.5$ m/s, defined as follows:

$$Re_0 = \frac{V_0 d}{\nu}, \quad (1)$$

where ν is the kinematic viscosity of water. The experiment was conducted at $Re_0 \approx 8000$ considering the currently reasonable computing power. The maximum measured velocity between rods was approximately 2 m/s. We note that a proper orthogonal decomposition [11] of the flow field has also been conducted and compared against Nek5000 results. Additional data and information on the facility are provided by Lee et al. [12], [13].

Figure 3.1. Overview of experimental facilities.

Figure 3.2 Test section geometry (a-b).

The spectral-element code Nek5000 is used to conduct the LES calculations of the helical coil test section. In the following the incompressible, constant properties Navier-Stokes equations are solved in Nek5000. The equations are solved with a predictor-corrector approach. The momentum equation is solved first; then the Poisson pressure equation is solved, and a corrector is applied to the velocity. The P_N - P_N formulation is used, so that the spatial discretization of pressure is the same as that of the velocity

September 30th, 2018

[14]. The algebraic multigrid method is used to precondition the pressure on the coarse grid. A characteristics-based time stepper is used to relax the Courant-Friedrichs-Lewy restriction [15], [16]. An explicit filter is applied to the velocity field to mimic the effect of dissipation due to the unresolved scale. The approach is similar to deconvolution-based subgrid-scale modeling [17].

Figure 3.3. Flow field measurements (a-b).

Figure 3.4. Nek5000 model of the single rod experiment. (a) Mesh. (b) Instantaneous velocity plot, units normalized by the rod diameter and inlet velocity.

The fluid domain is discretized based on the spectral-element method. A structured mesh of 407,364 hexahedral elements is used, after extensive mesh sensitivity studies. Within the spectral-element representation in Nek5000, for a N polynomial order, every element is represented with $(N + 1)^3$ points. Therefore, a 7th-order polynomial order expansion will yield 137 million degrees of freedom (DOFs). Similarly, 9th order will yield 293 million DOFs. To visualize the mesh, we show in Fig. 3.4a the Gauss-

September 30th, 2018

Lobatto collocation points of a 4th-order expansion. At 9th order, LES criteria for mesh resolution are satisfied, and the calculation is judged converged based on a mesh resolution test: y^+ at the wall smaller than 1.0, 5 points are present below $y^+=10$, $\Delta y^+ < 30$ in the spanwise direction, and $\Delta y^+ < 100$ in the streamwise direction. The flow is well resolved, with less than 2% of the turbulent kinetic energy contained in the subgrid scales (Fig. 3.4b).

To study the sensitivity to the boundary conditions, different inlet conditions (constant inlet, experimental inlet distribution) are considered, as shown in [19] and [20]. The most accurate inlet condition is based on values reconstructed from the PIV measurements. Seven planar measurements are taken with uniform spacing to develop two components of the entire velocity profile. In order to replicate the same inlet conditions as the experiment, the synthetic eddy method is used to generate the velocity fluctuations. Overall little difference is observed between different methods over accuracy. The coherent structures are superimposed onto the velocity signal by using symmetric shape functions. The detailed mathematical formulations of these functions can be found [18]. The outlet conditions are based on a turbulent outflow routine that forces the flow to be exiting the domain at the outlet boundary.

Figure 3.5. Comparison between Nek5000 results and PIV. Computational results are shown for several polynomial orders and with different inlet conditions. (a) Streamwise velocity at 6th rod. (b) Spanwise velocity at 6th rod. Comparison does not account for laser thickness.

The pressure drop is measured from taps located about 3 cm above and below the uppermost and bottommost rod. The static pressures at these locations are used to get the pressure drop from the simulation. The experimental pressure drop is measured to be 2849.8 Pa with an uncertainty of 0.2 Pa. For the constant inlet case, the pressure drops for 7th order and 9th order mesh are 2834.1 Pa and 2805.8 Pa, respectively, or less than 1.5% error from the experimental value.

Comparison of the LES data averaged velocity data yields reasonable results (Fig. 3.5) upstream of the 6th row of rods. A POD of the flow field was also conducted and the results show good agreement between computational results and experiment-based POD for the first three modes (the only ones for which the comparison was conducted). Additional results are reported by Lai et al. [19], [20]. Calculations for the five column case yielded similar comparisons with experiment as indicated by [21].

3.2 Validation on turbulent Excitation datasets

In the 1980s, Chen et al. [5] performed an experiment on a half-scale sector model of a steam generator helical coil tube tank at Argonne National Laboratory. This test was designed to study the structural motion of the whole tube bundle under the conditions of still and flowing air and water. Data from this test have been used to validate SHARP [6]. Results shown here demonstrate that at the low velocity, where the one-way coupling assumption is likely to hold, SHARP can successfully simulate flow-induced vibrations. Given the low flow conditions of SMRs, the simulation performed in this case are likely representative of the actual operation.

Figure 3.6. Computational model of the helical steam generator. (a) Nek5000 computational model (center section). (b) Nek5000 mesh. (c) Diablo computational model.

Chen's test section is a half-scale, 135-degree sector model of a steam generator. The full geometry is shown in Fig. 3.6c (Diablo computational model). The model contains three spans of helical tubes to simulate the exterior of seven columns of the steam generator. There are 30 rows of tubes in the flow direction. The flow enters from the top of the assembly and progresses downward. Hanger bars support these tubes at intervals of 45 degrees, splitting the test section into three 45-degree sectors. Fluid flow is present only in the center section as the tubes in the other two sectors vibrate in air. Hang bars support each tube, leaving the two ends open to the atmosphere. The tube patterns are not exactly aligned along the tube coil line. For simplicity in the construction of the geometry for the simulation, tube patterns are assumed to repeat (offset by the tube helix angle) at different hanger bars. In the experiment, the motion of the tubes was measured by means of accelerometers installed on spring mounts inserted into the tubes at the center in the center section.

September 30th, 2018

Figure 3.7. Nek5000 and Diablo results for the helical steam generator simulations. (a) Snapshot of the flow field in the model. Velocity magnitude. Units in [m] and [m/s]. (b) Acceleration power spectrum for one of the tubes (first row).

The computational model with Nek5000 is based on a setup similar to what described in the previous section. The inlet condition is based on a uniform inlet velocity plus synthetic turbulence. The velocity magnitude at the inlet is 0.49 m/s, corresponding to an averaged gap velocity of 1.28 m/s (the gap velocity is defined as the average velocity through the cross section of the tube bank). The Reynolds number based on the gap velocity is $\sim 20,000$. Only the fluid section (center section) has been modeled with Nek5000. The time steps used for the integration used is $\Delta t = 5.00e-6$.

A mesh involving approximately 3,000,000 elements was constructed. The number of grid points is around 1,536,000,000 at 7th polynomial order with 8 points per element in each spatial direction. Figure 3.6b shows the computational domain in the Nek5000 simulation. In this mesh, more than 99% near-wall points have y^+ less than 1.0, and the other mesh criteria described in the previous subsection are satisfied in 99% of the domain. The calculation was conducted on the Argonne Blue Gene/Q supercomputer Mira on over 100,000 processors.

In Diablo the tubes are modeled by using 4-node quadrilateral elements employing the Hughes-Liu two-dimensional shell, with 6 active DOFs per node [22]. The support structures are modeled by using 8-node hexahedral elements employing the common B-bar formulation with 3 active DOFs per node [23]. For this case the deformations are small and well below yield, and hence the materials are modeled as isotropic linearly elastic. If necessary, nonlinear kinematics and/or material formulations can be employed. The material used in Diablo simulations for the tubes is stainless steel with a density of $8,060 \text{ kg/m}^3$, Young's modulus of 203 GPa, and a Poisson's ratio of 0.275.

Table 3.1. RMS Tube Accelerations compared to experimental results

Tube	Diablo RMS tube “D” (out-of-plane) displacement (mil) at 2 fps, 0.075% damping	Diablo RMS tube “D” (out-of-plane) displacement (mil) 2 fps, 1.5% damping	Experimental RMS tube displacement (mil) 2 fps
26-1	0.0206	0.0192	N/A
30-1	.0551	.0478	0.05

The mesh consists of 3,691,464 nodes and 3,595,200 elements. The structural model is shown in Fig. 3.6c. Given the generally poor conditioning of long, thin structural elements, the direct solver WSMP was used for the matrix solution; and in order to provide the necessary memory for the direct solution process, 1 processor per node was used, providing 32 GB of memory for each active processor. The problem was solved by using a time-domain analysis employing the Newmark time integration scheme. Time steps of $\Delta t = 5.00e-5$ were used, and the problem was run typically for 1,800 (or more) steps, providing resolution down to approximately 10 Hz.

Numerous challenges arise in the structural modeling of SGs. Foremost is the modeling of the joints connecting the tubes to the support structures. The connections may take various forms, from completely bolted joints to hanger arrangements where the connection restrains the tubes in conjunction only with the effect of gravity. From Chen et al.’s paper [5], it is unclear what joints were used in the experiment. Therefore, the connection was not modeled in great detail, providing a source uncertainty. More details are provided by Merzari et al. [6].

The time-dependent fluid load for each Diablo grid point was obtained from the Nek5000 simulations (see Fig. 3.7a for a snapshot of the flow field) and applied as a boundary condition in Diablo. Then a transient calculation was performed to evaluate flow-induced vibrations and compared against the reported experimental values.

Figure 14 in [5] reports the results in terms of the root mean square (RMS) tube accelerations. The results for the two simulated flow gap velocities of 1.28 m/s and 3.33 m/s are extracted from the available figures. Accelerations are reported in terms of g’s, where $1\text{ g} = 9.81\text{ m/s}^2$. Equation 2 shows the definition of RMS acceleration a_{rms} between times T_1 and T_2 :

$$a_{rms} = \left(\frac{1}{T_2 - T_1} \int_{T_1}^{T_2} (a(t))^2 dt \right)^{1/2}. \quad (2)$$

At the gap velocity of 1.28 m/s the Diablo results bound the accelerations. That is, results with 1.5% damping have RMS accelerations slightly lower than experiment; for example, for tube 16-1 one has 0.0952 g for Diablo versus about 0.14 g for the experiment. In contrast, for the 0.75% case Diablo has RMS accelerations slightly higher than experiment; for example, Diablo has RMS accelerations of 0.167 g versus 0.14 g for the experiment. The acceleration power spectrum is also in reasonable agreement given the large uncertainties (Fig. 3.7b) in the modeling of the supports. Additional results are discussed in [6], along with a detailed structural modal analysis.

September 30th, 2018

In [6] we also discuss modeling of an additional experiment [24] illustrated in Fig. 1.1. The experiment is similar to the experiment of Chen [5], with less tubes, more support structure, a different tube curvature and with the full test section exposed to fluid loads. We note that for the additional experiment similar conclusions were reached. In particular, calculations performed with two reasonable damping values (1.5% and 0.75%) were able to bound the RMS of the acceleration of the tubes. More details are provided in Table 3.1. Overall, these calculations provide confidence in the capacity of SHARP to simulate FIV in helical steam generators, in the regime of turbulence excitation where one-way coupling is usually valid.

The calculations have also been performed with the fully coupled driver code presented in this section. While preliminary (due to lack of full integration time, which misses low frequency behavior), they demonstrate the simulations produce a peak spectrum frequency for the acceleration comparable to the experiment and to the one-way coupled calculations. The results are shown in Fig. 3.8 and demonstrate further the applicability of the one-way coupled driver of SHARP to this class of problems. Future calculations will include higher speeds, that go beyond the applicability of one-way coupled methods.

Figure 3.8. Preliminary acceleration power spectrum for one of the tubes (first row) for a coupled calculation.

4 Methods

The NEAMS Reactor Product Line (RPL) aims to develop an integrated multi-physics simulation with a multi-resolution hierarchy that ultimately span the full range of length and time scales in relevant reactor design and safety analyses, as well as scales from desktop to petaflop computing platforms. This section discusses the design and the numerical methodologies used in the SHARP. We also discuss the methods used to simulate the NuScale tests.

4.1 SHARP Physics Components

In order to better understand the level of fidelity that can be achieved by the SHARP framework, some key aspects of these physics components are given below.

4.1.1 Computational Fluid and Thermal Dynamics Solver (Nek5000)

The Nek5000 [2] computational fluid dynamics solvers are based on the spectral-element method developed by Maday and Patera [29]. Nek5000 supports two formulations for spatial and temporal discretization of the Navier-Stokes equations. The first is the P_N - P_{N-2} method with velocity/pressure spaces based on tensor-product polynomials of degree N and $N-2$, respectively. The second is the low-Mach number formulation of Tomboulides et al. [14, 30], which uses consistent order- N approximation spaces for both the velocity and pressure. The low-Mach number formulation is also valid at the zero-Mach (incompressible) limit. The Nek5000 code has been extensively verified and validated for several benchmark problems and has a proven scalability on existing petascale architectures up to 131,072 processors (over a billion degrees-of-freedom).

Of particular relevance to fluid-structure interaction is the Arbitrary Lagrangian Eulerian (ALE) modeling capability in Nek5000. For time-dependent geometry problems, a mesh velocity is defined at each collocation point of the computational domain (mesh) to characterize the deformation of the mesh. In the solution of the mesh velocity, the value of the mesh velocity at the moving boundaries is first computed by using appropriate kinematic conditions (for free-surfaces, moving walls and fluid layers) or dynamic conditions (for melting fronts). On all other external boundaries, the normal mesh velocity on the boundary is always set to zero. In the tangential direction, either a zero tangential velocity condition or a zero tangential traction condition is imposed; this selection is automatically performed by Nek5000 based on the fluid and/or thermal boundary conditions specified on the boundary. Under special circumstances, however the user may want to override the defaults set by Nek5000. If the zero tangential mesh velocity is imposed, then the mesh is fixed in space; if the zero traction condition is imposed, then the mesh can slide along the tangential directions on the boundary. The resulting boundary-value-problem for the mesh velocity is solved in Nek5000 by using an elastostatic solver, with the Poisson ratio typically set to zero. The new mesh geometry is then computed by integrating the mesh velocity explicitly in time and updating the nodal coordinates of the collocation points. Note that the number of macro-elements, the order of the macro-elements and the topology of the mesh remain unchanged even though the geometry is time-dependent. The use of an ALE description in Nek5000 ensures that the moving fronts are tracked with the minimum amount of mesh distortion; in addition, the elastostatic mesh solver can handle moderately large mesh distortion. However, the user is responsible deciding when a mesh would become "too deformed" and thus requires remeshing. The execution of the program will terminate when the mesh becomes unacceptable, that is, when a one-to-one mapping between the physical coordinates and the isoparametric local coordinates for any macro-element no longer exists. In general this is considered sufficient for flow-induced vibrations because up to the onset of fluid-elastic instability deformations will be modest. *In any case the mesh for the cases presented in this report is fixed, and the ALE solver is not invoked.*

4.1.2 Solid Mechanics Solver (Diablo)

The Diablo code being developed at Lawrence Livermore National Laboratory uses implicit, Lagrangian finite-element methods for simulating solid mechanics and multi-physics events over moderate to long time frames [2]. A primary focus is nonlinear structural mechanics and heat transfer. The code provides a venue for applying parallel computation to discretization technologies developed and user-tested in the legacy serial-processor codes NIKE3D and TOPAZ3D. Diablo is built around Fortran 95 data structure objects and a message-passing programming model. The architecture provides flexibility for the addition of other field problems, such as electromagnetics.

September 30th, 2018

In structural analysis of mechanical assemblies, a key functionality is "contact": capturing the interaction between unbonded material interfaces. The Diablo team has broad experience with contact problems and has created state-of-the-art algorithms for their solution. Their experience with contact motivates the use of low-order spatial discretizations, such as eight-node hexahedra for continua and four-node quadrilaterals for shells. Appropriate formulations are employed to accommodate nearly incompressible material models, such as for metal plasticity and rubber elasticity. Global algorithms include second-order and quasi-steady time integration and a number of approaches for nonlinear iteration: full Newton, modified-Newton, multiple quasi-Newton updates, and line search. Linear solvers are utilized from multiple libraries.

4.1.3. One-Way Coupling Methodology for Flow-Induced Vibrations

The full two way coupling methods are discussed in [5, 31]. Here we limit ourselves to one-way coupling. At times specified by the Nek user, Nek results along the tube surfaces are dumped to file(s) as pressure data for each 9-node quad on the tube surfaces. Using a sequence of Fortran utility programs, these pressure values are mapped to Neumann boundary surfaces within the Diablo mesh, and then the collection of time-dependent data is packed as nodal surface time histories into Exodus binary files. These files are then read in during the solution process and the data sampled and interpolated for the purpose of providing Neumann pressure loads to Diablo. Diablo time steps and Nek time steps need not be commensurate. A 1-1 relationship must exist between each 9-node quadratic facet on the Nek side of the boundary and a collection of four 4-node quads on the Diablo side of the interface. In the future this restriction will be eliminated. A time-averaged pressure file is also produced, mapped, and translated into an Exodus binary file, but containing only a single time history. Before the solution of the dynamic Diablo solution, a quasi-static Diablo run is performed by using the time-averaged pressure profile and is used as the initial condition for the Diablo time-dependent solution. This eliminates any "kick" due to the sudden application of the pressure profile, and it produces acceleration data that reflects the quasi-steady-state behavior. For the specific cases under consideration, the structural dynamics have been treated as linear, hence only a single left hand side (LHS) is formed and factored by the direct linear solver, which improves speed. If nonlinear effects are to be included, full Newton iterations can be performed, or various quasi-Newton schemes can be employed to reduce the time spent in linear algebra.

4.2. Methods for Simulation of the NuScale tests

In this subsection we describe the modeling of the NuScale FIV tests. The geometry of the test facility will not be described in detail as some of the information is proprietary. We however discuss in broad details the assumptions made in the analysis in both Nek5000 and Diablo. The test comprises 5 full columns of the steam generator design and it is full height and full tube-length. Each column is included in its entirety.

4.2.1 Nek5000 methods for the simulation of the steam generator

Figure 4.1 shows the mesh of the geometry under consideration. A 26 degree portion of the steam generator has been analyzed considering 5 columns and 6 rows. We note that the model constructed is based on geometric information present in the open literature for the NuScale design [8]. Values are normalized by the tube diameter.

September 30th, 2018

We also note that the geometry is limited in angular size but sufficient to capture a significant portion of the test. This is done to minimize computational cost. In order to do this periodic boundary conditions are applied in the streamwise direction while symmetry (slip boundary condition) is applied in the angular boundary. The inner and outer boundaries are walls.

The first step has been in creating a meshing strategy to obtain a fully hexahedral mesh. A script was created to generate for arbitrary number of columns and rows as well as arbitrary angles. Details on the mesh obtained for this work are shown in Fig. 4.1. The mesh comprises between 25,000-500,000 elements (high resolution) at the 11th polynomial order, for a total of approximately 40-800 million collocation points. The mesh is easily customizable thanks to a general methodology and the contained mesh size allows to test multiple conditions in the steam generator.

The calculation is run with a **Reynolds number based on the diameter of the tubes and the bulk velocity equal to 5,000**. *The calculations employed a Large Eddy Simulation (LES) turbulence model based on explicit filtering (5% on the highest wavenumber).* The mesh is refined to provide a sufficient wall resolution for the target Reynolds number. The Reynolds number corresponds to an average velocity of over 2.16 feet/s (120% flow load) in the gap region.

Figure 4.1. Mesh for 26 degree sector (detail showing displacement of the tubes due to counter-rotation and mesh refinement.).

The flow field in the domain is very complex (Fig. 4.2) but considerably more homogenous than what observed in the simulation of the Chen [5] experiments (which present columns with a single wrapping helical tube angle). The mild angle inclination causes the flow to have components in all directions but the flow is remarkably uniform in the cross section.

September 30th, 2018

Figure 4.2. Velocity magnitude in the streamwise direction. Location near supports. Units normalized by diameter size.

The force as a function of time has been computed on each **tube and degree segment** for long periods of time (over 75 convective time units) and for several mesh resolutions. The results have then been used to compute Power Spectral Density (PSD) functions as illustrated in Fig. 4.3. To calculate the PSD the initial transient which occurs over approximately 0.75s is removed and a PSD calculated for the remaining approximately 1.75s of time-history remaining. We remove the mean (average) value of each signal over the time window selected (e.g. after the initial transient) before computing the power-spectral density. We employ rectangular (not Hanning) windows. We note that a longer time history, while desirable, was not obtained due to computational cost. Frequencies below 2 Hz are likely not sufficiently resolved with this limitation.

The time-histories and the resulting PSD's have been evaluated and compared for different tubes and tube locations. Fig. 4.3 illustrates the PSD's plotted for every location along Tube 6, from degree 1 (close to the wall) to degree 26 (close to the other wall). Degree 13 represents the centerline of the tube. We see that the PSD's are only somewhat consistent with each other, in that the PSD for each location has a peak value somewhere in the 0-5Hz range, and decays rapidly to zero past 40 Hz. However, there are few other patterns which can be discerned.

In order to better understand the Nek response data, it is useful to look at the 8 center data sets, representing degrees 10-17, along Tube 6, see Fig. 4.4 for the full data set, and Fig. 4.5 with the initial 0.75s transient removed. An average of the 8 data sets is plotted as a function of time as the thick line in the figures. The actual response of the tube should be most commensurate with the input pressure time history represented by this average, since the data sets representing locations near the wall will produce less bending moment on the tube, and the average eliminates signals where the signals from adjacent data sets are cancelling each other out. A comparison of the PSD's of the data represented by Fig. 4.5 is presented as Fig. 4.6. Examining this latter figure, the PSD response of the average is essentially zero after about 20 Hz, whereas individual data sets continue to show response until about 40Hz.

September 30th, 2018

In order to understand how the data varies along a column of tubes, the time histories for degree 13 along column 1 are presented as Fig. 4.7, and with the initial transient removed as Fig. 4.8. A PSD of the data in Fig. 4.8 is presented as Fig. 4.9. The greatest response is found at the (inlet) top tube, tube 6, and the least response at the outlet, tube 1. Interestingly, tube 5, second-from-the-top, has the highest peak PSD though it has less response overall than tube 6. Tube 4, degree 13 has some interesting peaks around 17 Hz and 25 Hz.

In order to understand how the data varies along a row of tubes, the time histories for degree 13 for row 1 are presented as Fig. 4.10 and with the initial 0.75s transient removed as Fig. 4.11. A PSD of the data in Fig. 4.11 is presented in Fig. 4.12. Tube 6 (the inner column) has the largest response in this case, followed by Tube 18 (column 3) and Tube 12 (column 2), with Tube 30 (the outer column) having the least response, perhaps because of its lesser curvature.

Finally, a plot of PSD's for the center location (degree 13) is presented as Fig. 4.13. When one examines this figure, the most striking thing is that a couple of tubes have very large low-frequency response. In particular, tube 25 has the largest low-frequency peak. To understand what is happening in this case, examine a comparison of the time histories of Tube 25 and Tube 6, degree 13, presented as Fig. 4.14 with the initial 0.75s transient removed. One observes that Tube 25 appears to still be experiencing the after-effects of the transient. Longer runs in progress to eliminate any transient effects.

The PSD results show a signal significantly less strong than analytical PSDs for tubes in cross flows at the same gap velocity. **In particular if more than one segment is considered the PSD is conservative even at high frequencies (Fig. 4.16).** Interestingly the velocity signal presents secondary peaks at high frequencies consistent with preliminary pressure PSD results from Texas A&M. In fact, the experimental signal analysis showed two peaks: one at 13.77 Hz and another at higher frequencies. This was based on 30 sec of data at a rate of 10kHz. The PSDs are also remarkably consistent at higher resolutions.

(a)

September 30th, 2018

(b)

(c)

Figure 4.3. a) Force as function of time (seconds). b) Force as a function of time with initial 0.75s removed. c) PSD in the stationary state Tube 6, degree 13.

September 30th, 2018

Figure 4.4 - PSD's for each degree (1-26) for Tube 6

Figure 4.5 - Pressure time history for center 8 degrees of Tube 6

September 30th, 2018

Figure 4.6 - Pressure time history for center 8 degrees of Tube 7, initial transient 0.75s removed

Figure 1.7 - PSD of Eight Tube 6 Center Data Sets versus the PSD of the Average of the Eight

September 30th, 2018

Figure 4.8 - Pressure signals for degree 13, column 1

Figure 4.9 - Pressure signals for degree 13, column 1, initial 0.75s transient removed

September 30th, 2018

Figure 4.10 - PSD for degree 13, column 1

Figure 4.11 - Pressure time history for degree 13, row 1

September 30th, 2018

Figure 4.12 - Pressure time history for degree 13, row 1, initial 0.75s transient removed

Figure 4.13 - PSD for degree 13, row 1

September 30th, 2018

Figure 4.14 - PSDs for center (deg 13) locations for all 30 tubes

Figure 4.15 - Time history comparison for Tube 6 and Tube 25, degree 13, initial transient removed

Figure 4.16. Force PSD and comparison again analytical PSD in cross flow.

4.2.2 Diablo methods for the simulation of the steam generator

The Diablo model is based on beams to limit the computational cost. A pictorial representation is shown in Fig. 4.17

September 30th, 2018

Figure 4.17. Full steam generator beam model.

The baseline model (Model 1) has the following specifications:

- ❖ **206,565 nodes**
- ❖ **89,788 elements**
- ❖ **Supports:**
 - 8 supports modeled as solid shells, with a thickness of 1"
 - 98,352 shell elements total
 - Rayleigh Damping 0%, 0.75%, 1.5%
 - Rayleigh frequencies 100Hz and 200Hz
 - Density: 8.06 kg/m³, Young's Modulus 203 Gpa
 - Poisson's ratio 0.275
- ❖ **Tubes:**
 - 91,436 beam elements
 - 0.0625in diameter, 0.050in wall
 - Density: 8.06 kg/m³
 - Young's Modulus 203 Gpa
 - Poisson's ratio 0.275
 - Rayleigh Damping 0%, 0.75%, 1.5% [can be changed independent of shells]
 - Rayleigh frequencies 100Hz and 200Hz [can be changed independent of shells]
- ❖ **Boundary conditions:**
 - Fix nodes and rotations of the supports at the top and bottom
 - Note the actual boundary condition for the support is:

September 30th, 2018

- Support hung from the top
 - Support also constrained by the connections of the tubes to the manifolds at the top and bottom
 - The correct boundary condition is fixed at the top, with some allowance for a stiff connection for the manifold top and bottom – we model both top and bottom as fixed as an approximation which could be lifted
 - We are modeling the supports as monolithic shells, not the individual supports that connect to each other through the tubes themselves. This is obviously a simplification.
- Beams are directly connected to the shells (both displacement and rotations), see Fig. 4.18 and Fig. 4.19.

Four models have been considered for this analysis. Model 1 is comparatively more massive and less stiff than the supports are expected to be. Model 2 has stiffer version of Model 1, with reduced density to model a stiff, light version of the supports. We have found that the greatest influence in the response of the model is the stiffness of the connection of the tubes to the supports. In the future we would like to develop a more comprehensive model of the supports. Details are below:

- Model 1 (as above)
- Mesh 2
 - 5” wide supports, solid steel, modeled as shells, density 1/10th of standard steel
 - Otherwise as for Model 1
- Model 3
 - Tubes connected to the supports by 0.4665” long tubes, as in Fig. 4.20 and Fig. 4.21.
 - Note that where the tubes are connected to the shells they are free to rotate in the axis perpendicular to the shell plane (no rotational stiffness in that direction)
 - Tube/Support Connection tubes are “soft”, modeled as tubes with 0.0625” outer diameter and 0.0050” wall
- Model 4
 - As in model 3, Tubes connected to the supports by 0.4665” long tubes
 - Tube/Support Connection tubes “medium stiff”, 0.625” outer diameter and 0.050” wall.

In model 3 In order to model a soft connection of the tubes to the supports, we use a short beam (a tube cross section was chosen for simplicity), arranged as in Figure 4.20. The beam is connected to the shell such that the rotational degrees of freedom with axes in the plane of the shell are shared with the shell. This couples the bending of the tubes to the shell through the (stiff or soft) tube. We can modify the stiffness of the short tube (the distance 0.4665” was chosen as a function of the tube-tube separation, it is not otherwise important as the main quantity of interest is the stiffness of the connection itself) to investigate soft (e.g. poorly constrained) or stiff (highly constrained) connections. This is a simplification meant to provide bounding conditions without the complications of friction and gap opening/closing.

Figure 4.18 - Model 1 and 2 tube/support connection

Figure 4.19 - Model 1 and 2 tube/support connection - zoomed in showing mesh

September 30th, 2018

Figure 4.20 - Model 3 and 4 tube/support connection, short beams connecting tubes to supports shown in yellow

Figure 4.21 - Model 3 and 4 tube/support connection, zoomed-in showing mesh, short beams connecting tubes to supports shown in yellow.

September 30th, 2018

A tabulation of the modes derived by an eigenvalue analysis [first 100 modes extracted] of Model 1 is provided as Fig. 4.22. The first 16 modes have natural frequencies between 15Hz and 57 Hz. These modes involve global twisting, bending, or breathing motions of the structure, driven by the deformation of the (relatively soft and massive) supports. A representative sample of mode shapes are provided as Figs. 4.23, 4.24, 4.25 and 4.26. These modes are unlikely to be excited by the fluid pressure input given the global nature whereas the fluid pressure tends to be highly localized and globally uncorrelated. The first mode which is primarily “local tube motion” and likely to be excited by the fluid pressure input is mode 17 with a natural frequency of ~57 Hz, see Fig. 4.27. Modes 18 to 100 inclusive have mode shapes and natural frequencies characteristic of mode 17.

A tabulation of the modes derived by an eigenvalue analysis [first 100 modes extracted] of Model 2 is provided as Fig. 4.28. Model 2 has the same mesh and properties as Model 1, excepting that the supports have been increased in stiffness and decreased in density in comparison with Model 1. As a result, the first mode natural frequency has been increased to 35 Hz (from ~15 Hz), and there are only 4 modes with natural frequencies below ~57 Hz. Modes 1 through 4 involve global motions of the structure, analogous to modes 1 through 16 of Model 1, see for example Figs. 4.29 and 4.30. Mode 5, with a natural frequency of 57.313 Hz, is analogous to mode 17 of Model 1, see Fig. 4.31.

A tabulation of the modes derived by an eigenvalue analysis [first 100 modes extracted] of Model 3 is provided as Fig. 4.32. Model 3 has the same mesh and properties as Model 2, with stiff, light supports, excepting that the tubes are now connected to the supports via short beams. In Model 3 these connections are chosen to be exceptionally soft. As a result, the first mode natural frequency has a value intermediate between Model 1 and Model 3, at ~24Hz (compare Model 1 at ~15 Hz and Model 2 at ~35 Hz). Like Model 2, the supports themselves are stiff and light, however because the connections between the tubes and the supports are soft, the tubes themselves contribute less to the overall stiffness of the structure – hence the natural frequency of the first mode is reduced in comparison with Model 2. As in model 2, there are only 4 modes with natural frequencies below frequencies characteristic of primarily tube response, see for example Fig. 4.33 and Fig. 4.34. In contrast to models 1 and 2, however, the tube response modes have a characteristic natural frequency of ~37 Hz (compare 57 Hz for Models 1 and 2), and the modes are much more isolated to the response of a single tube, reflecting the soft connection of the tubes to the supports such that individual tubes are largely decoupled from one another. Modes 1 through 4 involve global motions of the structure, analogous to modes 1 through 16 of Model 1, see for example Figs. 4.29 and 4.30. Mode 5, with a natural frequency of 37.3868 Hz, is analogous to mode 17 of Model 1, see Fig. 4.35.

Model 4 was designed to check that the response of Model 2 could be recovered from Model 3 by adequately increasing the stiffness of the connections between the tubes and the supports. The result of an eigenvalue analysis of Model 4 is provided as Fig. 4.36. Note that, as in Model 2 there are 4 global modes. The first global mode has a natural frequency of 33.5818 Hz which compares well to Model 2 mode 1 at 34.7515Hz, see Fig. 4.37. Similarly, mode 4 at 41.6155 Hz compares well to Model 2, mode 4 at 42.6856 Hz, see Fig. 4.38. Finally, mode 5 and subsequent modes are all tube modes with natural frequencies of around 55 Hz, comparable to Model 2 modes 5 and above with natural frequencies of around 57 Hz, see Fig. 4.39.

A characteristic of the tube/support connection featured in Models 3 and 4 is that the connection beams are free to rotate in the axis perpendicular to the shell. That is, the rotational degrees of freedom of the connection beam where it is attached to the support (the latter modeled as a shell) are coupled to the shell for the two bending degrees-of-freedom of the shell, corresponding to the two rotational axes in the plane of the shell. However, as the shell has no rotational stiffness for the axis normal to the shell surface. Hence the connection beam is free to rotate in this plane. However, the connection beam rotational degrees-of-freedom are fully coupled to the tube degrees-of-freedom where the connection beams attach to the tubes. The possibility of Adding additional stiffening beams was considered but discarded because

September 30th, 2018

the eigenvalue analysis of Model 4 was so close to Model 3 that these additional stiffening beams were deemed unnecessary.

To summarize the various models, Model 1 and 2 have stiff connections to the supports, but vary in the stiffness of the supports themselves, with Model 2 featuring light and stiff supports in comparison with Model 1. Model 3 has the stiff/light supports of Model 2 but features soft connections from the tubes to the supports, whereas Model 4 has again stiff/light supports but stiff connections from the tubes to the supports. Model 1 has 15 low-frequency modes beginning at 15 Hz and ranging all the way up to 54 Hz, capturing global behavior of the structure – bending/twisting/breathing. The 16th mode of Model 1 is at 57 Hz and captures primarily tube motion, somewhat correlated amongst many tubes in one section of the structure. However, there are many modes past the 16th mode very similar in frequency and behavior to the 16th mode capturing variations of the same theme. Models 2, 3, and 4 all have supports which have been significantly strengthened and lightened. As a result, there are only 4 global modes before the tube modes, with natural frequencies in the range of 24 Hz to 44 Hz depending on the mode and the model. Models 2 and 4 have tube modes at 57 Hz whereas model 3 has tube modes at 37 Hz. The tube modes of Models 1, 2 and 4 are somewhat correlated with one another whereas the tube modes of Model 3 are more isolated to a single tube, reflecting the relative softness of the connection of the tubes to the supports in Model 3 versus Models 1, 2, and 4.

Standard Mesh, 1" wide supports, standard support density				
Mode	Frequency, rad/s	Frequency, Hz	Period, s	Description
1	95.85	15.256	0.066	global twisting mode
2	126.86	20.191	0.050	compound twisting, bending, breathing
3	127.58	20.305	0.049	compound twisting, bending, breathing
4	135.80	21.613	0.046	compound twisting, bending, breathing
5	167.04	26.585	0.038	Breathing
6	212.62	33.839	0.030	Compound Twisting
7	219.69	34.964	0.029	Breathing
8	220.25	35.053	0.029	Breathing
9	248.04	39.476	0.025	Compound Twisting
10	250.32	39.839	0.025	Compound Twisting/breathing
11	252.13	40.128	0.025	Compound Breathing
12	291.10	46.329	0.022	Compound Tube
13	329.83	52.494	0.019	Compound Tube
14	337.32	53.687	0.019	Compound Tube
15	339.07	53.965	0.019	Compound tube
16	343.22	54.625	0.018	Mostly Tube
17	359.31	57.186	0.017	Tube Mode
18	359.42	57.203	0.017	Tube Mode

Figure 4.22 - Eigenvalue analysis of Model 1 – modes which involve local tube motion do not begin until mode 17 (57.186 Hz).

September 30th, 2018

Figure 4.23 - Model 1, mode 1, 15.2558 Hz, global twisting mode

Figure 4.24 - Model 1, mode 3, 20.3046 Hz - compound breathing/bending/twisting

September 30th, 2018

Figure 4.25 - Model 1 mode 7, 34.964 Hz, breathing

Figure 4.26 - Model 1, mode 15, 53.9652 Hz, compound tube deformation plus structural twist

Figure 4.27 - Model 1, mode 17, 57.185 Hz, mostly tube response

Standard Mesh, 5" wide supports, support density 1/10th of normal

Mode	Frequency,		Period, s	Description
	Frequency, rad/s	Hz		
1	224.63	35.751	0.028	twisting mode
2	252.73	40.222	0.025	bending mode
3	254.70	40.537	0.025	bending mode
4	268.20	42.686	0.023	breathing mode
5	360.11	57.313	0.017	single tube mode
6	360.36	57.353	0.017	single tube mode

Figure 4.28 - Eigenvalue analysis of Model 2, mode 5 is the first local tube mode

September 30th, 2018

Figure 4.29 - Model 2, mode 1, 35.75 Hz, breathing/twisting mode

Figure 4.30 - Model 2, mode 4, 42.6856 Hz, breathing mode

Figure 4.31- Model 2, mode 5, 57.313 Hz, tube mode

Mesh with soft connections, 5" wide supports, support density 1/10th				
Mode	Frequency, rad/s	Frequency, Hz	Period, s	Description
1	152.06	24.201	0.041	twisting/breathing mode
2	175.36	27.910	0.036	compound breathing mode
3	178.43	28.399	0.035	bending mode
4	179.05	28.497	0.035	bending mode
5	234.91	37.387	0.027	isolated tube mode
6	234.91	37.387	0.027	isolated tube mode
7	234.92	37.388	0.027	isolated tube mode
8	234.92	37.388	0.027	isolated tube mode
9	234.92	37.388	0.027	isolated tube mode
10	234.92	37.388	0.027	isolated tube mode
11	234.92	37.388	0.027	isolated tube mode

Figure 4.32 - Model 3 eigenvalue analysis - tube modes now begin at 37.387 Hz

September 30th, 2018

Figure 4.33- Model 3, mode 1 (24.2006 Hz), global twisting/breathing mode

Figure 4.34- Model 3, mode 4, 28.4974 Hz, global bending mode

September 30th, 2018

Figure 4.35 - Model 3 mode 5, 37.3868 Hz, isolated tube mode

Mesh with medium connections, 5" wide supports, support density 1/10th, single tube					
Mode	Frequency, rad/s	Frequency, Hz	Period, s	Description	
1	2.11E+02		3.36E+01	2.98E-02	Twisting/Breathing Mode
2	2.45E+02		3.90E+01	2.57E-02	Bending/Breathing Mode
3	2.47E+02		3.92E+01	2.55E-02	Bending/Breathing Mode
4	2.61E+02		4.16E+01	2.40E-02	Breathing Mode
5	3.48E+02		5.54E+01	1.81E-02	Tube Mode
6	3.48E+02		5.54E+01	1.81E-02	Tube Mode

Figure 4.36- Eigenvalue analysis of Model 4, tube modes begin at mode 5 (55.4 Hz).

September 30th, 2018

Figure 4.37 - Model 4, mode 1, 33.58 Hz, compare Model 2 mode 1 at 34.7515Hz

Figure 4.38- Model 4 mode 4, 41.6155 Hz, compare Model 2, mode 4 at 42.6856 Hz

September 30th, 2018

Figure 4.39 - Model 4, mode 5, 55.3741 Hz, compare Model 2 mode 5 at 57.133 Hz

5 Coupled Simulation Results and Sensitivity Analysis

The signal from Nek5000 was loaded into Diablo in various conditions. The table below summarizes various cases that were performed for this dynamic calculations. In particular we evaluate the influence of damping, the stiffness of the supports and the stiffness of the connection between supports and tubes.

Table 5.1 - Diablo Analysis Cases

Case	Model	Damping
1	Model 1	0.0%
2	Model 1	0.75%
3	Model 1	1.5%
4	Model 2	0.0%
5	Model 2	0.75%
6	Model 2	1.5%
7	Model 3	0.0%
8	Model 3	0.75%
9	Model 3	1.5%
10	Model 4	0.0%
11	Model 4	0.75%
12	Model 4	1.5%

For each of the analysis, a pressure signal derived from the Nek results described in **Section 4.2.1** (2.16 feet/s) was applied. Because of the 8 supports, the actual geometry has 8 sections, 4 narrow sections spanning 26 degrees, and 4 wide sections spanning 64 degrees. The Nek results were for a subset of the entire geometry, corresponding to a section 6 tubes tall and 5 tubes deep ($6 \times 5 = 30$ tubes) and 26 degrees. This was driven by the need to reduce computational cost. The tubes were numbered such that tube 1 represented a tube on the inner diameter at the bottom, tube 2 above tube 1, all the way to tube 6 at the top of the first column of tubes. Tube 7 then represented the bottom tube of the second column from the inside diameter, tube 8 above tube 7, etc., all the way to tube 30 representing the top tube in the outside column.

The Nek results were provided for the purposes of the Diablo analysis as a total force/degree in each of the coordinate directions for each of the 26 degrees for the 30 modeled tubes, at time steps corresponding to 2.77×10^{-4} seconds. A preliminary analysis of both the data and the eigenvalue analysis indicated that the excitation in the vertical (Z) direction was largest, and also most likely to excite significant deformation modes in the structure, and therefore the data in the cross-tube directions (X and Y) was discarded. The remaining vertical (Z) direction data was then mapped to Diablo inputs using the following scheme:

September 30th, 2018

1. The 30 tubes in the Nek data were sorted into the following bins:
 - a. Bottom tubes: 1, 7, 13, 19, 25
 - b. Top tubes: 6,12,18,24,30
 - c. Second-from-the-bottom tubes: 2,8,14,20,26
 - d. Second-from-the-top tubes: 5,11,17,23,29
 - e. Intermediate tubes: 3,4,9,10,15,16,21,22,27,28
2. At every section, each tube was evaluated for whether, at that section, it was a “top tube”, a “bottom tube”, “second from the top”, “second from the bottom”, or “intermediate”. Obviously at each section there would only be 1 tuber per column for each category except “intermediate”. Note that, because of the helical spiral a tube might be a “top tube” in one section but an “intermediate” tube in another section, etc.
3. For “top”, “bottom”, “second from the top”, and “second from the bottom”, the tube was mapped to the appropriate Nek tube for that column
4. For “intermediate” tubes, the tube was mapped randomly to one of the two intermediate Nek tubes for that column. For example, for column 1 a tube would be mapped randomly to either tube 3 or tube 4.
5. If a tube was in a narrow section then the Nek results for the 26 degrees would be mapped directly. If a tube was in a wide section then the Nek results were mapped in the following manner:
 - a. The first and last 5 degrees close to the wall were mapped directly Nek to tube, e.g. Nek degree 1 was mapped to Tube degree 1, Nek degree 26 was mapped to Tube degree 64, Nek degree 2 was mapped to Tube degree 2, Nek degree 25 was mapped to Tube degree 63, etc. Locations along tubes which did not correspond to integer degrees were mapped to the closest integer degree.
 - b. Interior tube degrees were mapped proportional to how close one was to the center, so Nek degree 13 was mapped to Tube degree 32, Nek degree 8 was mapped to tube degree 20 etc. Locations along tubes which did not correspond to integer degrees were mapped to the closest integer degree.

Each Nek time history had the time-average removed, so that the displacement response corresponded to the dynamic response, that is there is no quasi-static displacement expected.

To quantify the structural, two nodal positions were chosen for time-history output. Node “A” is at the center of a top tube in the inside column of a wide section, corresponding to Node 119397 in Models 1 and 2 and Node 119907 in Models 3 and 4. Node “B” is at the center of a top tube in the outside column of a wide center section and corresponds to Node 206553 in Models 1 and 2 and Node 216579 in Models 3 and 4, see Figs. 5.1 and 5.2. Because the loading is highest at the fluid inlet, these locations are expected to have the largest response. Note that the node numbers change because additional nodes were introduced for the connections between the tubes and the supports.

Figure 5.1 - Nodal locations for output, Models 1 and 2

Figure 5.2- Nodal locations for output, Models 3 and 4

The RMS response for the two different node locations and the different analysis cases is presented in Fig. 5.3. Except for Model 3, the response of the different Models and different levels of damping is almost identical, with an RMS for Node A of about $1.00\text{E-}5$ meters and Node B of about $2.00\text{E-}5$ meters. The response is almost independent, therefore, of the stiffness of the supports or the level of damping applied. Node B has greater displacement largely because it is the outside column and has a greater span and hence lower tube stiffness. Model 3, on the other hand, shows a much greater response, and greater sensitivity to damping. Node A has a maximum response of almost $3.5\text{E-}5$ meters at 0.0% damping though 0.75% damping decreases the response to a little over $2.0\text{E-}5$ meters. Node B has a response of around $4.75\text{E-}5$ meters at 0.0% damping, and it takes more than 0.75% damping for the response to begin to decrease. Figure 5.4 illustrates the response of Node A at the 0.0% damping level for the 4 different models. Except for Model 3, the other 4 models have time histories which are almost indistinguishable, but Model 3 has displacement excursions out to $1.0\text{E-}4$ meters.

In order to more closely examine the response for the different analyses, PSD's were calculated for each of the analysis and presented for Models 1 – 4 as Figs. 5.5, 5.6, 5.7 and 5.8 respectively.

Examining Fig. 5.5 one sees that the response below 20 Hz or so is largely independent of the damping level, though the damping significantly effects the peak at approximately 75 Hz for Node A and approximately 57 Hz for Node B. This same pattern is repeated for Model 2, see Fig. 5.6 and Model 4, see Fig. 5.8. For Model 3 something similar is observed, except that (as expected) the peaks are at lower frequencies, ranging down to 37 Hz. In addition, large secondary peaks are seen at low levels of damping at approximately 42 Hz and 55 Hz, which are almost eliminated at the 1.5% damping level. As with models 1,2, and 4, damping has little effect on the response at frequencies below 20 Hz.

The response of Node A at the 1.5% damping level has been compared across the different models in Fig. 5.9 respectively. What one notices is that there is a very large response at approximately the 42 Hz level

September 30th, 2018

for Model 3 that is not seen in any of the other models. At frequencies below 20 Hz or so all the models show some response, with Model 3 having greater response than the other 3 models, but still in proportion to the response of the other models. What this tends to indicate is that the response of Model 3 at the 42 Hz level for Node A (and the 37 Hz level for Node B) represents a coupling of the natural frequencies of Model 3 to the Nek input, which has small but non-zero frequency content around 40 Hz. However, the response in the 20 Hz range and below is similar and in proportion between Models 1, 2, 4 and 3 – the response of Model 3 is larger primarily because Model 3 is softer – the fact that damping has little effect for these frequencies indicates that the response is largely quasi-static (i.e., there little to no dynamic response).

In order to evaluate this hypothesis, the response between 0-30 Hz for Nodes A and B has been investigated, additionally plotting the PSD of the Nek input data for Tube 6 and Tube 30, where the PSD of the average of the center 8 degrees (degrees 10-16) pressure data was taken, plotted using the second (right-side) y-axis so that the data does not have to be on the same scale, see Fig. 5.10 and 5.11 The input data for Node A should roughly correspond to ‘Tube 6’, whereas the input data for Node B should roughly correspond to ‘Tube 30’. If the hypothesis that the response in the region below 20 Hz is largely quasi-static, one should see no effect from damping, and the peaks of the Nek PSD should line up with the peaks of the response PSD’s. Inspecting the figures, it is seen that damping has little or no effect on the results, but the peaks of the Nek PSD’s do not exactly line up with the response, though they have somewhat similar character. That suggests that perhaps a better measure of the Nek data should be used, perhaps a more sophisticated weighted average instead of simply a weighted average of the center 8 time-histories.

Figure 5.3- RMS response of Nodes A and B for the various analysis cases

September 30th, 2018

Figure 5.4 - Time history of Node A for Models 1 through 4 with 0.0% damping

Figure 5.5 - Frequency response for Model 1

September 30th, 2018

Figure 5.6 - Frequency response for Model 2

Figure 5.7 - Frequency response for Model 3

September 30th, 2018

Figure 5.8 - Frequency response for Model 4

Figure 5.9 - Response comparison for Node A at 1.5% damping, different models

September 30th, 2018

Figure 5.10- Node A looking at 0-30 Hz and comparison with Nek

Figure 5.11 - Node B looking at 0-30 Hz and comparison with Nek

September 30th, 2018

To summarize, a number of different Diablo analyses were made, using different combinations of damping, support stiffness, and stiffness of the connection between the tubes and the supports. In general, the stiffness of the supports is much less important than the stiffness of the connection between the tubes and the supports. If the support connection is soft enough to allow tube natural frequencies down in the range of 35 Hz to 45 Hz, there is the possibility of enhanced coupling to the fluid excitation. Support/tube natural frequencies which begin at 55 Hz and above result in greatly reduced coupling between the fluid excitation and the tube response, of a factor of 2 or more when measured in terms of RMS, and a factor of 5 or more when measured with respect to peak displacement.

6 Conclusions

In this report we discussed the application of SHARP to the NuScale design. This work represents an important stepping stone toward the ultimate goal of simulating helical steam generators at high fidelity. We simulate the conditions of an expected test conducted by NuScale. The fluid simulation is performed with Nek5000 using methodologies previously validated against experimental data from Texas A&M (see section 3).

Using the dynamic fluid load computed with these calculations a number of different structural calculations using Diablo were made. These include analyses using different combinations of damping, support stiffness, and stiffness of the connection between the tubes and the supports. In general, the stiffness of the supports is much less important than the stiffness of the connection between the tubes and the supports. The results are summarized in Fig. 5.3 and show very modest RMS of displacements (not greater than 50 μm or 2 mils) but significant sensitivity to the modeling of the support-tube connection. That sensitivity is discussed in great detail in Section 5 and illustrate how the natural frequencies can significantly shift with changes in the support-tube connection. Support/tube natural frequencies which begin at 55 Hz and above result in greatly reduced coupling between the fluid excitation and the tube response, of a factor of 2 or more when measured in terms of RMS, and a factor of 5 or more when measured with respect to peak displacement.

7 References

1. Vijay S. Mahadevan, Elia Merzari, Timothy Tautges, Rajeev Jain, Aleksandr Obabko, Michael Smith, Paul Fischer, "High-resolution coupled physics solvers for analysing fine-scale nuclear reactor design problems", *Philosophical Transactions of the Royal Society of London A*, DOI:10.1098/rsta.2013.0381
2. Fischer, P., Lottes, J., Kerkemeier, S., Marin, O., Heisey, K., Obabko, A., Merzari, E. and Peet, Y., "Nek5000: User's manual". Argonne National Laboratory, https://nek5000.mcs.anl.gov/files/2015/09/NEK_doc.pdf.
3. D. Parsons, J.M. Solberg, R.M. Ferencz, M.A. Havstad, N.E. Hodge, and A.P. Wemhoff, *Diablo User Manual*, Lawrence Livermore National Laboratory report UCRL-SM-234927, Sept. 2007.
4. Merzari, E., H. Yuan, A. Kraus, A. Obabko, P. Fischer, J. Solberg, S. Lee, J. Lai, M. Delgado, and Y. Hassan. "High-Fidelity Simulation of Flow-Induced Vibrations in Helical Steam Generators for Small Modular Reactors." *Nuclear Technology* (2018): 1-15.
5. S. S. Chen, J. A. Jendrzejczyk, and M. W. Wambsganss, "Tube vibration in a half-scale sector model of a helical tube steam generator," *J. Sound Vib.*, vol. **91**, pp. 539–569 (1983)
6. Merzari, E., Haomin Yuan, A. Kraus, Jerome Solberg, and Robert M. Ferencz. *Coupled Calculations in Helical Steam Generator: Validation on Legacy Data*. No. ANL/NE-16/49. Argonne National Lab.(ANL), Argonne, IL (United States), 2016.
7. Yuan, Haomin, Jerome Solberg, Elia Merzari, Adam Kraus, and Iulian Grindeanu. "Flow-induced vibration analysis of a helical coil steam generator experiment using large eddy simulation." *Nuclear Engineering and Design* 322 (2017): 547-562.
8. NuScale Design Certification Application, available at <https://www.nrc.gov/reactors/new-reactors/design-cert/nuscale/documents.html#dcApp>
9. Lai, Jonathan K., Elia Merzari, and Yassin A. Hassan. "Large Eddy Simulation of the Flow Behavior in a Simplified Helical Coil Steam Generator." *Journal of Fluids Engineering* 141, no. 2 (2019): 021401.
10. Stefan Turek, Jaroslav Hron "Proposal for Numerical Benchmarking of Fluid-Structure Interaction between an Elastic Object and Laminar Incompressible Flow", *Fluid-Structure Interaction*, Volume **53** of the series Lecture Notes in Computational Science and Engineering pp 371-385
11. G. Berkooz, P. Holmes, and J. L. Lumley, "The proper orthogonal decomposition in the analysis of turbulent flow." *Annual Reviews of Fluid Mechanics*, vol. 25, pp. 539–575 (1993).
12. Saya Lee, Marilyn Delgado, Samuel J. Lee, and Yassin A. Hassan, "Flow visualization in a simplified helically coiled steam generator geometry," in *Proceedings of ICAPP 2017*, Fukui and Kyoto, Japan (April 24-28, 2017).
13. Marilyn Delgado, Samuel Lee, Saya Lee, and Yassin A. Hassan, "Pressure drop measurements across a helical coil steam generator configuration," in *Proceedings of HTR'16*, Las Vegas, NV (Nov. 2016).
14. A. Tomboulides, J. Lee, and S. Orszag, "Numerical simulation of low Mach number reactive flows," *Journal of Scientific Computing*, vol. **12** (2), pp. 139–167. (1997).
15. Y. Maday, A. T. Patera, and E. M. Ronquist, "An operator-integration-factor splitting method for time-dependent problems: Application to incompressible fluid flow," *Journal of Scientific Computing*, vol. **5**, pp. 263–292 (2009).
16. P. Fischer, "Implementation considerations for the OIFS/characteristics approach to convection problems." <http://www.mcs.anl.gov/~fischer/nek5000/oifs.pdf>
17. P. Schlatter, S. Stoltz, and L. Kleiser, "LES of transitional flows using the approximate deconvolution model", *International Journal of Heat and Fluid Flow*, vol. **25**, pp. 549–558 (2004)

September 30th, 2018

18. Jarrin, N., Benhamadouche, S., Laurence, D., and Prosser, R., “A synthetic-eddy-method for generating inflow conditions for large-eddy simulations”. *International Journal of Heat and Fluid Flow*, vol. **27** (4), pp. 585–593 (2006).
19. J. K. Lai, E. Merzari, S. J. Lee, M. Delgado, S. Lee, and Y. A. Hassan, “Large eddy simulations for a helical coil steam generator,” in *American Nuclear Society Winter Meeting and Nuclear Technology Expo* (2016).
20. J. K. Lai, E. Merzari, M. Delgado, S. J. Lee, S. Lee, Y. Hassan, “ Large eddy simulation of a helical coil steam generator test section,” in *Proceedings of the ASME Fluids Engineering Summer Meeting 2017*, Hawaii (August 2017).
21. Yildiz, Mustafa Alper, Haomin Yuan, Elia Merzari, and Yassin Hassan. "Numerical Simulation of Isothermal Flow Across Slant Five-Tube Bundle with Spectral Element Method Code Nek5000." *Nuclear Technology* (2019): 1-11.
22. T. J. R. Hughes and W. K. Liu, “Nonlinear finite element analysis of shells, Part I: Three-dimensional Shells,” *Comput. Meths. Appl. Mech. Engrg.*, vol. **26**, pp. 331–362. (1981).
23. T. J. R. Hughes and W. K. Liu, “Nonlinear Finite Element Analysis of Shells, Part II: Two-dimensional shells,” *Comput. Meths. Appl. Mech. Engrg.*, vol. **27**, pp. 167–181.(1981).
24. G. B. Watson, “Functional performance of the helical coil steam generator, Consolidated Nuclear Steam Generator (CNSG) IV system. Final report,”, Babcock and Wilcox, Report PB-253324; ARC-5178-F TRN: 77-009144 (1975).
25. Burman, E., Fernández, M., “Stabilization of explicit coupling in fluid–structure interaction involving fluid incompressibility,” *Comput. Methods Appl. Mech. Eng.* (2008)
26. Bobovnik, G., Mole, N., Kutin, J., Stok, B., Bajsic, I., “Coupled finite volume/finite-element modelling of the straight-tube Coriolis flowmeter,” *J. Fluids and Structures*, 785-800 (2005)
27. Stefan Turek, Jaroslav Hron “Proposal for Numerical Benchmarking of Fluid-Structure Interaction between an Elastic Object and Laminar Incompressible Flow”, *Fluid-Structure Interaction*, Volume **53** of the series Lecture Notes in Computational Science and Engineering pp 371-385
28. Watson, G. B. “Functional performance of the helical coil steam generator, Consolidated Nuclear Steam Generator (CNSG) IV system – Final report”, Report PB-253324; ARC-5178-F (1975)
29. Y. Maday, A.T. Patera, “Spectral element methods for the Navier-Stokes equations,” in A.K. Noor and J.T. Oden, editors, *State-of-the-Art Surveys in Computational Mechanics*, pp. 71–143, ASME, New York, 1989.
30. A.G. Tomboulides, M. Israeli, G.E. Karniadakis, “Efficient removal of boundary-divergence errors in time-splitting methods,” *Journal of Scientific Computing*, **4**:291–308, 1989.
31. Elia Merzari, Jerome Solberg, Paul Fischer, Robert M Ferencz, “A high-fidelity approach for the simulation of flow-induced vibration”, *Proceedings of the ASME 2016 Fluids Engineering Division Summer Meeting*, pp. V01AT03A019-V01AT03A019, (2016)

Nuclear Science and Engineering Division

Argonne National Laboratory

9700 South Cass Avenue, Bldg. 208

Argonne, IL 60439

www.anl.gov

Argonne National Laboratory is a U.S. Department of Energy
laboratory managed by UChicago Argonne, LLC