Demonstration of Fully Coupled Calculations in Helical Steam Generator: Toward Predictions of Fluid Elastic Instability

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.

Demonstration of Fully Coupled Calculations in Helical Steam Generator: Toward Predictions of Fluid Elastic Instability

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

R. Rahaman
Mathematica and Computer Science Division, Argonne National Laboratory

J. Solberg
Methods Development Group, Lawrence Livermore National Laboratory

September 30th 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. In addition to being more cost- and space-efficient, such steam generators need to be more reliable, since failure of the primary vessel represents a potential loss of coolant and a safety concern. A significant amount of data exists on flow-induced vibration in shell-and-tube heat exchangers, and heuristic methods are available to predict their occurrence 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 design stage.

Over the past seven years, the Reactor Product Line has developed the integrated multi-physics code suite SHARP. The goal of developing such a tool is to perform multi-physics neutronics, thermal/fluid, and structural mechanics modeling of the components inside the full reactor core or portions of it with a user-specified fidelity. In particular 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. These tools form a strong basis on which to build a flow-induced vibration modeling capability. In the last two years a significant effort was devoted to 1) develop and validate a fully coupled capability in SHARP for Fluid-Structure calculations (FSI) and 2) validate the use of SHARP for one-way coupled calculations in helical steam generators on legacy data available as well as novel experiments.

In this report we discuss an extension of previous one-way coupled calculations performed with Nek5000 and Diablo aimed at simulating available FIV experiments in helical steam generators in the turbulent buffeting regime. It was demonstrated that in the buffeting regime one-way coupling was judged sufficient because the pressure loads do not cause substantial displacements. However, higher speeds led to a reduction in accuracy and the need for a fully coupled capability for velocity or approaching the critical velocity and fluid elastic instability. In this report we demonstrate such fully coupled capability on the Argonne test case discussed in previous reports.

CONTENTS

SUMMARY	i
1 Introduction.....	1
2 Overview of HIP Project.....	2
3 Overview of SHARP.....	3
4 Experimental setup	10
5 Results.....	13
6 Conclusions.....	22
7 References.....	23

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). In fact, with the SG placed within the reactor vessel, the need for a reliable design is increased because the cost of tube failure becomes potentially even more significant.

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 [1,2]. 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), vibrations are caused by various coexisting phenomena whose relative importance depends on the flow parameters: turbulent pressure fluctuations, vortex shedding induced by cross-flows, 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

understanding of the physics behind FIV. Work performed in the previous year lead to the development of such a tool in the multi-physics suite SHARP, partially validated on classic fluid-structure interactions (FSI) benchmark data [28, 29].

Moreover [30, 31] focused on the application of SHARP to legacy datasets, in preparation for its application to the NuScale design. One-way coupled simulations were performed in the turbulent buffeting regime for two datasets: the B&W dataset [6] and the Chen *et al.* dataset [7] (“Argonne” dataset in the following). For both cases reasonable damping choices were able to bound the experiment with the calculation results. The spectral response was also in reasonable agreement with the experiment.

For the Argonne dataset, a higher flow condition was also simulated, with, as expected, less success. For higher flow rates approaching fluid elastic instability, it was expected that a tightly capability is needed. In the present work we continue that work demonstrating the use of the fully coupled simulator SHARP to continue this complex analysis and approach fluid elastic instability. We discuss also some implementation issues that were resolved to achieve good performance for such large scale calculations. In fact to achieve good parallel performance numerous changes to the mapping algorithm were introduced. For the first time a working version on BlueGene/Q was also implemented – this involved a massive rewrite of the building structure that took several months.

We note that this class of simulations are considerably more expensive computationally than one-way coupled calculations and they are not expected to be of immediate use to simulate the current Nuscale Steam Generator design, but they present an important tool to investigate fluid elastic instability and explore the design space in future iterations.

2 Overview of High-Impact Project

The high-impact project consists 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. The main objective of **Task A** is the implementation of an appropriate interface and coupling capability in SHARP to deal with fluid-structure interaction. Simulations for coupling a stiff structure with a weakly compressible fluid are particularly demanding for the numerical algorithms for both accuracy and robustness. The need for sub-iterations across time steps has been established, and the development of a tight solver has been conducted. More details are provided in Section 3.

The verification and validation of the FSI coupling methodology are part of **Task B**, which is the primary concern of the present report. This will include only available datasets. This task involves 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., [3, 4, 5]) 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 [6, 7].**
- d) Validation on other datasets relevant to nuclear applications [8]. This will be decided in collaboration with AREVA.

The cases discussed in this report fall under categories c). Part c) involves both a validation of the fully coupled methodology and one-way coupled methodology. **This report concerns the two-way portion of the work.** We discuss also some implementation issues that were resolved to achieve good performance for such large scale calculations.

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 SHARP

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 toolkit to integrate neutronics, thermal-hydraulics, and structural mechanics physics components to perform coupled reactor analysis on a representative SMR/SFR?? core geometry. Based on the requirements specified, a problem to quantify the primary structural mechanical feedback effect with multi-way coupling has been implemented with dual resolution: a detailed heterogeneous model represents the duct surrounding each assembly while the interior of the ducts (the individual assemblies) is represented with a homogenized geometry.

In order to produce a fully coupled-physics simulation capability, two obvious approaches can be pursued. In one approach, existing single-physics codes/components can be assembled into an overall coupled simulation code with appropriate interfaces to communicate between the components to capture the nonlinear feedback effects. This is generally referred to as a “small-f” or “bottom-up” framework approach [9, 10]. The other approach is to use an integrated, coupled-physics modeling framework, with new code pieces for each relevant physics area developed inside that framework from scratch. This is sometimes referred to as a “large-F” or “top-down” approach [11]. The primary advantage of the former approach is that it preserves several man-years invested in existing verified and validated individual physics modeling codes, but at the cost of some intrusive modifications to enable the software interfaces. The large-F approach avoids intrusive interfacing by providing a unified platform to enable coupling, but at the cost of rewriting all the necessary physics codes and verifying the components individually and as a whole. The overall approach being pursued in the RPL effort is to develop and demonstrate a small-f framework for performing coupled multi-physics analysis of reactor core systems. This system takes

advantage of many single-physics codes also sponsored by the overall NEAMS program over the past several years.

Details regarding the background of construction of the RPL coupled-physics framework (SHARP), and the methodology are discussed in the following sections.

3.1 The SHARP Multi-physics Code System

A multi-physics reactor core modeling code can be constructed in many ways, and numerous efforts have provided stepping-stones for future efforts [10]. What distinguishes the SHARP effort from others is the goal of flexibility in the physics, discretization types, and software options supported by the framework. This section describes the SHARP modeling approach in detail and illustrates how various physics codes have been connected to this framework.

SHARP employs a “bottom-up” approach, so it can use existing physics codes and take advantage of existing infrastructure capabilities in the MOAB framework and the coupling driver/solver library, the Coupled Physics Environment (CouPE), which utilizes the widely used, scalable PETSc library.

Using an existing physics code in this system (Fig. 3.1) requires that the system support the mesh type used by the individual physics models. The physics models can retain their own native representation of the mesh, which gets transferred to and from MOAB’s representation through a mesh adaptor; or it can use MOAB’s representation directly. Language interoperability through the C/Fortran-based iMesh interfaces also allows flexibility in the implementations that are tuned to individual physics requirements without overhead.

In practice, this means that the coupled system may be solved on multiple meshes, each of which models part or all of the physical domain of the problem. In order to perform efficient coupled calculations, the results must be transferred from the mesh on which they are generated (source mesh), to the mesh for which they provide initial or boundary conditions (target mesh) due to nonlinearity introduced because of coupling between physics models. “Multi-way” transfer is required in cases where the physics depend on each other’s solution fields, for example in reactor analysis where neutronics computes heat generation based on temperature properties computed by thermal-hydraulics, which in turn depends on the heat-generation source term computed by neutronics.

Figure 3.1 SHARP architecture

Since relevant physics components solving a nuclear engineering problem have widely varying backgrounds in terms of code architectures, dependency requirements, and specialized solver data-structures, a flexible approach to the coupling methodology was necessary in order to obtain accurate solutions. This motivation led to the development of the MOAB-based spatial projection tools and the CouPE drivers based on PETSc library to orchestrate the global nonlinear solver. Details regarding these tools are given in the following sections.

3.2 SHARP Physics Components

In the SHARP framework, MOAB interfaces are implemented for these physics components that are relevant to fast reactor physics analysis. The addition of a new physics component to the framework requires integration and ability to read the mesh and possibly associated data from iMesh/MOAB formats, along with implementation to propagate solution variables back onto the mesh after their computation via tags defined either on discrete vertices or on elements. Because of the various storage formats used in physics models and the parallel domain-decomposed environment in which these calculations are usually run, this integration process can be involved.

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.

3.2.1 Computational Fluid and Thermal Dynamics Solver (Nek5000)

The Nek5000 [12, 13] computational fluid dynamics solvers are based on the spectral-element method developed by Patera [14]. 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. [15, 16], 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.

3.2.2 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 [17]. 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.

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.

3.3 Improvements in SHARP for present calculations

3.3.1 Two-Way Coupling Methodology for Flow-Induced Vibrations

Fundamentally, there are two strategies for numerically solving fluid-structure interaction problems: so-called monolithic approaches which treat both the fluid and the structure equations as one global system of nonlinear equations, and partitioned approaches, where the single-field problems are solved in a sequential manner. Here, we employ the latter approach due to its higher flexibility of combining nearly any kind of single-field formulation and straight-forwardness when coupling independent solvers such as Nek5000 (fluid) and Diablo (structure). The strategy of the coupled solver is discussed in detail in [28, 29] along with some validation results. In particular an acceleration strategy employing fictitious mass damping is discussed. In the following sections we describe some recent advancements essential to perform the simulations described in the rest of the report.

3.3.2 Improved Mapping algorithm

Because of the need to resolve boundary layers at the tube/support interface, it became impossible to perfectly align the Nek fluid mesh and the Diablo structural mesh. The original algorithm designed to map the Nek mesh to the Diablo mesh, and vice-versa, relied on a one-one relationship between quadratic Nek facets (9-node quads) and sets of 4 linear Diablo facets (4-node quads). As this 1-1 correspondence could not be maintained for the Argonne Steam Generator case, a more general mapping algorithm had to be created.

This mapping algorithm is based on the concept of closest-point projection. For any particular nodal point I with coordinates $\mathbf{X}^I$ the goal is to find the facet on the opposite surface K and local coordinates ξ such that, for the local shape functions $N(\xi)_K^J$ and nodes local to the facet $\mathbf{X}_K^J$ one has the minimum distance between $\mathbf{X}^I$ and the facet, respecting the fact that the local coordinates should remain within the range $[-1,1]$, that is:

$$\left\| \mathbf{X} - \sum_{J=1}^{N_K} N(\xi)_K^J \mathbf{X}_K^J \right\| = \min \left| \xi_i \in [-1,1] \forall i = [1, 2] \right| \quad (1)$$

Assuming no facets overlap, the unique facet and set of local coordinates which minimize (1) can be found for each node I by searching over all facets K . As this is an n -squared search, strategies must be employed to reduce the time required for large meshes.

For the mapping of Nek nodes to Diablo facets, the following algorithm is employed:

1. Loop over all Diablo nodes on the Nek/Diablo interface, and determine the maximum extent in $X/Y,Z$ (or alternatively $R/\theta/Z$).
2. Using the knowledge of the maximum extent, subdivide each coordinate direction into N_{bucket} divisions. This provides a set of $N_{\text{bucket}} \times N_{\text{bucket}} \times N_{\text{bucket}}$ buckets. In order to avoid issues with tolerance, expand the size of each bucket by a factor (typically 10%) so that the buckets overlap.
3. Sort all the Diablo nodes into buckets. As the buckets overlap, some nodes will be in more than one bucket. Keep track of both the node and the associated Diablo facet(s) that are (partially) in each bucket.
4. Loop over each Nek facet, and then each Nek node within each facet.
 - a. Use a “chomp” algorithm, so that the search is partitioned equally among all the processors – each processor does $(N_{\text{facets}}/N_{\text{procs}})$ facets, and the

- results are communicated to all the processors at the end using an MPI_REDUCE call.
- b. For every bucket which contains the Nek node:
 - i. Loop over the Diablo nodes/facets in that bucket
 1. Calculate the distance between the Nek node and the Diablo node
 2. Create the collection for that bucket of all Diablo nodes/facets that have the minimum distance
 - c. Now find the bucket(s) that have the minimum distance over all the buckets.
 - d. Now loop over all the facets that have this minimum distance. Find the Diablo facet/local coordinate combination that has the minimum distance according to equation (1)
 - e. Determine whether this Diablo facet is local to this processor. If it is not, then zero the map on this facet.
5. Sort all the Nek facets/nodes into buckets. As the buckets overlap, some nodes will be in more than one bucket. Keep track of both the Nek node and the associated Nek facet(s) that are (partially) in each bucket.
 6. Loop over each Diablo facet local to this processor, and then each Diablo node within each facet.
 - a. For every bucket which contains the Diablo node:
 - i. Loop over the Nek nodes/facets in that bucket
 1. Calculate the distance between the Nek node and the Diablo node
 2. Create the collection for that bucket of all Nek nodes/facets that have the minimum distance
 - b. Now find the bucket(s) that have the minimum distance over all the buckets.
 - c. Now loop over all the facets that have this minimum distance. Find the Nek facet/local coordinate combination that has the minimum distance according to equation (1)
 - d. Since we are already local to this processor on the Diablo side, and all the Nek nodes/facets are on every processor, we do not have any localization operation to do here.

This operation is, without taking into account enhanced parallelism which has been incorporated into the new algorithm, approximately 10 times slower than the earlier mapping operation, which only calculated the minimum distance between nodes. The primary reason is that multiple iterative processes are needed to find the solution for equation (1), and then find the solution which minimizes this over all the possibilities. We try to minimize this by providing an initial guess for each iteration commensurate with the closest node. In those cases where there is a node-node match, this produces an minimization with only 2 iterations. However, it is still true that the same iteration is performed for all the facets that are connected to the closest node, so even in this case one has a minimum of 8 iterations in the common case where 4 facets touch an interior node.

It may be possible to reduce the cost of this algorithm by looking for “node-node matches”. This requires the introduction of a tolerance based upon how well the two meshes “match”, which should be known a-priori but is generally problem-dependent.

The use of enhanced parallelism, however, has cut the cost of the algorithm down to significantly below the cost of the initial algorithm. In fact, because the algorithm need only be executed once (the map being stored), the cost of the mapping algorithm is insignificant in comparison to the rest of the

calculation. In fact, even for large problems such as the steam generator, the algorithm is less expensive than the cost of even one FIV time-step.

It has been found that, to avoid problems caused by curved surfaces, where non-matching meshes will inevitably have gaps, only DISPLACEMENTS and not POSITIONS are mapped. Hence the Nek surface is not forced to conform to the Diablo surface during motion – the initial gaps between the surfaces are preserved.

In compensation for a more expensive mapping algorithm, the new mapping algorithm allows for arbitrary non-commensurate meshes to be used in the Diablo and Nek meshes, which provides the user additional flexibility. The new and old algorithms are both available in the current code base.

3.3.3 Novel BG/Q SHARP FIV version

Large scale FSI calculations require significant computational time and we emphasize that the scales attempted in this report require the use of supercomputing. **However, prior to work described here SHARP has not been successfully compiled on an IBM BG/Q platform, the primary supercomputing platform of Lawrence Livermore and Argonne National Laboratories.**

The issues generally stemmed from:

- Common problems configuring and compiling any autotools-derived build system with a cross-compiled environment. These issues are broadly acknowledged by the software development community.
- Compiler-specific language features in SHARP modules. SHARP was targeted for the Intel compilers and has utilized several Intel-specific Fortran language features. These features were generalized for the IBM compiler family.

We resolved all these issues to perform the simulations described in the present work. While the solution may appear specific to the architecture, the solution strategy adopted will be applicable to other supercomputing platform and it will streamline the application of SHARP to future architectures.

SHARP's third-party libraries have all adopted different solutions for cross-compilation and language-specific problems on BG/Q. These solutions often rely on several manual installation steps, even including manual edits to the source files. As such, manual installation of the libraries was a much easier path than updating SHARP's automatic installation system. After considerable effort, these installations are now maintained on Mira, the BG/Q supercomputer at Argonne National Laboratory.

DIABLO has been updated to provide robust solution for compiling on BG/Q. Provided that its 3rd-party libraries are available for BG/Q (as mentioned above), the user now can successfully configure and compile standalone DIABLO for on BG/Q with a few simple commands. This has been successfully demonstrated on Mira.

The SHARP coupled driver for FIV was restructured and simplified in a novel BG/Q version. We have also implemented a standalone Makefile that bypasses the automated SHARP configure script. The Makefile itself is short (<100 lines) and easily maintainable. It relies on precompiled installations of +DIABLO and 3rd-party libraries (as mentioned above). A detailed README has been added to the SHARP repository in a specialized branch documenting these changes.

4 Experimental Setup

Chen et al. [7] performed an experiment on a half-scale sector model of a steam generator helical coil tube tank in Argonne National Laboratory in the 1980s. This test was designed to study only the structure motion of the whole tube bundle under flow conditions. Tube vibrations under different flow conditions are recorded.

The test section is a half-scale, 135-degree sector model of the steam generator of a liquid metal fast breeder reactor. The model is designed to analyze the vibration of tubes only, rather than the vibration of the whole helices. The test section is shown in Fig. 4.1 with the outer shell cover removed. The model contains three spans of helical tubes to simulate the exterior of seven columns of the steam generator.

The test section has 30 rows of tubes in the flow direction (downward). Hanger bars support these tubes at an interval of 45-degree, splitting the test section into three 45-degree spans. Flow is directed only across the center span.

Fig. 4.2 shows the radii of curvature of the seven columns of tubes. Hang bars at A, B, C, and D support each tube, leaving two ends open to the atmosphere. The tube patterns at supports A and B are presented in Fig. 4.3. We can see in Fig. 4.3, the tube patterns are not exactly aligned along the tube coil line. Therefore, the tube patterns are considered exact same at different hanger bars in order to simplify geometry construction in our simulation. The tube arrays consist of stainless steel tubes with 0.01588 m (5/8 inch) outer diameter, and 0.00089 m (0.035 inch) wall thickness. The tubes in the transitional region (these tubes are denoted by double circles in Fig. 4.3) are heavy wall tubes with wall thickness of 0.0165 m (0.065 inch) to increase the stiffness. The pitch between adjacent columns is constant at 0.0254 m (1 inch). The pitch within each column is constant at 0.0238 m (15/16 inch).

The motion of the tubes was measured by means of accelerometers installed on spring mounts inserted in the tubes. The accelerometer has two sensitive axes, one in the out-of-plane direction (drag direction) and the other in the in-plane direction (lift direction).

Several tests have been conducted using air and water. However, only the data from the water test will be used to compare with numerical data. The inlet flow velocity distribution is uniform with the screen flow equalizer. With the exception of positions close to the outer wall, the deviation is less than 10% and is considered to be satisfactory. Despite the slight deviation from uniform flow velocity in experiments, we applied a uniform flow velocity at the inlet of simulation domain.

Fig. 4.1. Picture of the experimental setup. From [7].

Fig. 4.2 Radii of curvature of helical tubes, describing the position of the supports. From [7].

In the report [30] we discussed the application of SHARP to legacy datasets. Simulations were performed in the turbulent buffeting regime for two datasets: the B&W dataset [6] and for the Argonne dataset [7] also presented in Fig. 4.4. The modeling approach relied on one-way coupling, which is considerably cheaper because it does not require sub-iterations. For both datasets, at low-flow conditions (i.e., such as those expected in SMRs) reasonable damping choices in the structural model are able to bound the experiment with the calculation results. Moreover, the spectral response was in reasonable agreement with the experiment.

For the Argonne dataset, a higher flow condition was also simulated, with, as expected, less success. For higher flow rates, approaching fluid elastic instability, it is expected that a tightly coupled capability is needed. This is shown in Tables 4.1 and 4.2 where different values of damping did not translate into a bounding of the displacements and acceleration.

Fig. 4.3 Tube patterns at supports A and B. From [7].

Figure 4.4 - Positions of the output for tubes A=16-2, B=16-6, C=16-10, D=16-27. The colors represent the initial static displacement.

Table 4.1 - RMS Acceleration Values for Forward-Coupled Argonne SG Calculation, Gap Flow Velocity 3.33mps, Nominal Structural Damping 1.5%.

RMS accleration	Out-plane(g)	Chen paper est.*	In-plane(g)	Chen paper est.*
13A-1	2.08	0.2	1.66	
13B-1	2.49	0.2	1.82	
14A-1	2.72	0.2	1.74	
14B-1	2.18	0.2	1.73	
15A-1	2.66	1.0	1.61	

15B-1	1.95	4	1.75	
16-1	3.22	6	1.87	
16-2	2.71		1.84	0.2
16-10	2.71	0.2	2.89	
16-27	2.57	0.5	3.34	0.14

Table 4.2 - RMS Acceleration Values for Forward-Coupled Argonne SG Calculation, Gap Flow Velocity 3.33mps, Nominal Structural Damping 0.75%.

RMS acceleration	Out- plane(g)	Chen paper est.*	In-plane(g)	Chen paper est.*
13A-1	2.07	0.2	2.05	
13B-1	2.94	0.2	2.39	
14A-1	3.31	0.2	2.23	
14B-1	2.62	0.2	2.1	
15A-1	3.8	1.0	2.37	
15B-1	2.97	4	2.45	
16-1	4.28	6	2.24	
16-2	3.41		2.22	0.2
16-10	3.16	0.2	3.13	
16-27	2.64	0.5	3.03	0.14

5 Results

5.1 Fluid Simulations

For the Nek5000 fluid simulations, a mesh involving approximatively 3,000,000 elements was constructed. The number of grid points involved went from ~125,000,000 at low polynomial order to

~1,500,000,000. Details of a coarse mesh are illustrated in Fig. 5.1. In this mesh, at low polynomial order ($N=4$, corresponding to $1 \times 1=5$ points per element), the maximum y^+ at tube surface is around 8.0, and the minimum y^+ is around 0.01. At high polynomial order ($N=7$, corresponding to $1 \times 1=8$ points per element), the y^+ at the surface is below 1.0. Fig. 5.2 shows the computational domains for the Argonne tests. Since only the center section has flow, we simulate the flow only in the center section. More refined meshes have also been analyzed.

Fig. 5.1. Mesh for Argonne case, low polynomial order.

Fig. 5.2. Computational domain for the Argonne case (flow goes downward).

We tested several inlet boundary conditions in previous work. The impact is overall minor on the simulation results. We have compared against other code at several lines (Fig. 5.3) and noticed significant differences between URANS and LES, and between turbulence models. The flow field is fairly complex and presents significant lateral flows within the bundles.

Fig. 5.3. Snapshot of the flow field and sampling line locations.

5.2 Structural calculation

The structural modeling of steam generators presents a number of challenges. Foremost is the modeling of the joints connecting the tubes to the support structures. The connections may take a number of forms, from completely bolted joints to hanger arrangements where the connection restrains the tubes only because of the effect of gravity. Diablo is a nonlinear structural mechanics code with sophisticated contact models. In particular, Diablo can accurately model contact, release, and gross sliding. Especially in the case of hanger arrangements, the ability to model this behavior is crucial in some situations, as

when the bars are subject to gross motion. In these cases typically Coulomb-type friction models are appropriate for modeling the dissipation of the joints.

In situations of high-frequency but small-amplitude vibration, bolted joints often are a source of significant dissipation in the system. This dissipation is traceable to frictional processes under conditions of microslip within the joints. Traditional large-Coulomb models such as employed by Diablo generally require very meshes and small (but still nonlinear) time steps in order to resolve the microslip adequately. Instead, one may utilize semi-empirical interface models [18], [19], [20], [23]. These models are based on the mathematical theory of hysteresis formulated by Preisach, and may take a number of equivalent forms [21], [22]. Currently Diablo does not have any of these models implemented, but they fit within the general framework of Diablo's contact infrastructure. Besides modeling the dissipation of a joint, these models can account for the (varying) stiffness of the joint itself. Note that the behavior of joints in dry conditions is generally different from that in wet conditions.

In order to evaluate the structural approximation, a modal analysis was performed on a subset of the problem consisting of a single tube array along with the entire support structure. The support structure was modeled as four solid panels of structural steel, though in reality the support structure consists of hanger bars for which no additional information is available. The material properties of the support structure were varied in order to simulate different states of connection between the support structures and the hanger bars.

We performed numerical experiments investigated the effect of the stiffness of the supports. If the stiffness of the hanger bars was reduced by a factor of 1000, but the boundaries of the supports were fixed all the way around, then the natural frequency of the first mode could be reduced to approximately 103 Hz, as in Fig. 5.4. A full system modal analysis demonstrated that these supports were too soft, as the first mode ceases to be a tube mode but instead becomes a mode of deformation of the supports themselves. A more complete analysis is included in [30].

Fig. 5.4 - First mode (103 Hz) of tube 16-1 using low stiffness, zero density, highly constrained supports.

5.3 Two-way Coupled calculations

Using the coupled solver we demonstrated the solver, allowing the Nek5000 mesh to deform (see fig. 5.5). This is the first time this has been achieved for helical coil steam generators and it represents a major step forward for an FEI capability. Overall displacements are small for the present conditions (Fig. 5.6). The primary objective is to evaluate the onset of fluid elastic instability.

Fig. 5.5 Example of deformed (right) and corresponding undeformed (left) mesh.

Fig. 5.6 Displacements for coupled calculation.

Initial results were obtained at an inlet velocity of 0.49 m/s, corresponding to an average gap velocity of 1.28 m/s. These results used Richardson Extrapolation to enable 10 fluid steps for each solid mechanics step, still within an iterative loop. The solid mechanics solution used fictitious mass and damping, with an initial starting set of values derived from the Turek4 test case earlier. A Comparison of the PSD results is provided as Figure 5.7 and show a similar response between one-way coupling and two-way coupling. We note that differences might be attributed to the shorter run time of the two-way coupled runs compared to the one-way coupled runs.

Figure 5.7 - Comparison of Low Velocity Results at 0.49 m/s inlet velocity, 1.28 m/s average gap velocity. One-way versus Two-way (Richardson Extrapolation) versus Experiment

Subsequent attempts to simulate a higher velocity ran into issues with stability using the values of fictitious mass and damping we had initially tried. We also abandoned Richardson extrapolations because subsequent investigations on smaller problems indicated accuracy issues at higher velocities. We changed to Aitken's acceleration using a starting alpha of 0.1, and we initialized the problem using a quasi-static simulation, created by running Nek dynamically but with zero velocity at the Nek/Diablo interface, while running Diablo quasi-statically and slowly ramping up the pressure coupling between Nek and Diablo. This allowed Diablo to slowly deform the Nek mesh to account for the initial flow state. We chose to first simulate an intermediate inlet velocity of 1.03 (2.68 m/s average gap velocity). As an additional aid to stabilization, these simulations were performed including 0.75% structural damping. These simulations succeeded in running for over 7000 time steps, with a time step size of 5.0e-6 s. We extracted time histories and PSD's and compared to data available from Chen, see Figure 5.8. These results show reasonably good match for all tubes, especially tubes 13A/B and 14 A/B. The data for tubes 15 A/B and 16 has peaks at higher frequencies than provided by the data, and a broader spectrum. This may indicate that the experiment had softer connections for these tube locations, leading to frequency

response at lower frequencies, which may have been more resonant, hence leading to a sharper bandwidth for the experiment.

RMS acceleration values were extracted from these simulations and are provided in Table 5.1. No direct experimental comparisons are available from Chen's paper. However, we did extract RMS acceleration values from our forward-coupled 1-way simulations, which are provided as Table 5.2

Figure 5.8 - PSD comparison to experiment, 2-way coupled calculation, 1.03 m/s inlet velocity, 2.68 m/s average gap velocity

Table 5.1 - Out-of-plane RMS acceleration values from coupled 2-way simulation at 1.03 m/s inlet velocity, 2.68 m/s average gap velocity

tube	acc (m/s)	acc (g)
13A-1	3.44	0.350663
13B-1	3.98	0.405708
14A-1	3.97	0.404689
14B-1	3.51	0.357798
15A-1	4.4	0.448522
15B-1	4.09	0.416922
16-1	4.3	0.438328

Table 5.2 - RMS acceleration results for forward-coupled 1-way simulations at 0.49 m/s inlet velocity, 1.28 m/s average gap velocity

Damping 1.5 %

RMS acceleration	Out-plane(m/s ²)	Out-plane(g)	In-plane(m/s ²)	In-plane(g)
13A-1	3.09	0.309	2.27	0.227
13B-1	4.67	0.467	1.98	0.198
14A-1	3.37	0.337	2.88	0.288
14B-1	2.39	0.239	1.97	0.197
15A-1	3.20	0.32	1.60	0.16
15B-1	2.36	0.236		
16-1				

Damping 0.75 %

RMS acceleration	Out-plane(m/s ²)	Out-plane(g)	In-plane(m/s ²)	In-plane(g)
13A-1	3.92	0.392	2.73	0.273
13B-1	5.59	0.559	2.54	0.254
14A-1	4.11	0.411	3.65	0.365
14B-1	2.85	0.285	2.50	0.25
15A-1	4.03	0.403	2.07	0.207
15B-1			2.58	0.258
16-1				

Damping 0.375 %

RMS acceleration	Out-plane(m/s ²)	Out-plane(g)	In-plane(m/s ²)	In-plane(g)
13A-1	4.8	0.48	3.23	0.323
13B-1	6.58	0.658	3.25	0.325
14A-1	4.91	0.491	4.40	0.440
14B-1	3.57	0.357	3.06	0.306
15A-1			2.59	0.259
15B-1			3.14	0.314
16-1			2.67	0.267

An interpretation of these values is as follows - compare the available acceleration RMS values as a function of flow velocity, reproduced as Figure 5.9, we see that our 1-way coupled results at 1.28 m/s gap velocity and our 2-way coupled results at 2.68 m/s gap velocity all report RMS acceleration values of around 0.3 to 0.4 g for out-of-plane excitations. This is consistent with the results in Figure 5.9 which show that below approximately 2.75 m/s gap velocity the RMS acceleration values are approximately constant and below 0.5 g for all tubes.

From the experimental data it can be seen that the RMS accelerations begin to steeply increase for some tubes past approximately 2.75 m/s and for all tubes past approximately 3 m/s. Our next simulations attempted to perform 2-way coupled calculations at 1.27 m/s inlet velocity, 3.33 m/s average gap velocity, using Aitken's acceleration. To speed up the calculations, we attempted to slowly increase the inlet velocity (varying the ramp over 100 to 500 steps at $5.0\text{e-}6$ s step size) in the coupled calculation, starting from the 1.03 m/s inlet velocity case, again using Aitken's acceleration. We have been able to advance this simulation for over 700 steps (600 steps at full velocity), before issues with mesh-tangling in the Nek solution have arisen. These solutions have so far demonstrated displacements many times those from the calculations at 2.68 m/s gap velocity, see Figure 5.9. Examining the figure, the displacements of tube 16-1 are on the order of $3.5\text{e-}4$ m peak-peak for 3.33 m/s, whereas the displacements of the same tube at 2.68 m/s are on the order of $1.0\text{e-}5$, a difference of 35 times. Examining again Figure 5.9, Tube 16 (the tube with the largest RMS values in the figure) has an RMS acceleration of around 8 g at 3.33 m/s, compared with a value of less than 0.5 g below 2.6 m/s, a ratio of approximately 16. As the coupled simulation is still settling down from initial transients, it is expected that the steady-state solution will show RMS amplitudes at a lower value, perhaps commensurate with the experimental data. It is encouraging that the simulations at these higher velocities show the large increase in RMS amplitudes at these values, which indicates that we are capturing the onset of the fluid-elastic instability.

Figure 5.9 - Comparison of out-of-plane displacements for Tube 16 from 2-way coupled simulations at 3.33 m/s average gap velocity (left) versus 2.68 m/s average gap velocity (right), Tube 16-1. Time step is plotted as the abscissa, where each step is $5.0\text{e-}6$ s.

6 Conclusions

In this report we discussed the first set of large scale calculations performed with SHARP for a fully coupled simulations in helical coil steam generators. Major advancements in the FSI driver were necessary to achieve this including a port to Blue Gene supercomputers and a rewrite of the mapper to allow for good scaling performance and flexibility. The driver has been demonstrated on the Argonne case and it has shown to predict a strong increase in vibration at high speed. This indicates that SHARP is capable of capturing the onset of FEL.

References

1. A. Siegel, T. Tautges, A. Caceres, D. Kaushik, P. Fischer, G. Palmiotti, M.A. Smith, J. Ragusa, "Software Design of SHARP," in *Proceedings of the Joint International Topical Meeting on Mathematics and Computations and Supercomputing in Nuclear Applications (M&C + SNA)*, American Nuclear Society, April 2007.
2. 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
3. Burman, E., Fernández, M., "Stabilization of explicit coupling in fluid–structure interaction involving fluid incompressibility," *Comput. Methods Appl. Mech. Eng.* (2008)
4. 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)
5. 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
6. 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)
7. S.S. Chen, J.A. Jendrzejczyk, M.W. Wambsganss, "Tube vibration in a half-scale sector model of a helical tube steam generator", *Journal of Sound and Vibration*, Volume **91**, Issue 4, 22 December 1983, Pages 539-569
8. Mueller, A., "The validation of STAR-CCM+ coupled to Abaqus for analyzing fluid-elastic instabilities in a flexible tube bundle," STAR Global Conf. (**2013**)
9. T.J. Tautges, R. Meyers, K. Merkley, C. Stimpson, C. Ernst, *MOAB: A Mesh-Oriented Database*, Sandia National Laboratories report SAND2004-1592, April 2004.
10. T.J. Tautges, H.-J. Kim, A. Caceres, R. Jain, "Coupled Multi-Physics simulation frameworks for reactor simulation: A Bottom-Up approach," in *Proceedings of the International Conference on Mathematics and Computational Methods Applied to Nuclear Science and Engineering (M&C)*, American Nuclear Society, Rio de Janeiro, Brazil, May 2011.
11. D. Gaston, C. Newman, G. Hansen, D. Lebrun-Grandi, "MOOSE: a parallel computational framework for coupled systems of nonlinear equations," *Nuclear Engineering and Design*, **239**(10):1768–1778, Oct. 2009
12. E Merzari, WD Pointer, P Fischer, "Numerical simulation and proper orthogonal decomposition of the flow in a counter-flow T-junction", *Journal of Fluids Engineering*, **135**, paper 091304 (2013).
13. 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.
14. 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.

15. A.G. Tomboulides, J.C.Y. Lee, and S.A. Orszag, "Numerical simulation of low Mach number reactive flows," *Journal of Scientific Computing*, **12**:139–167, June 1997.
16. 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.
17. 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.
18. D. J. Segalman "A Four-Parameter Iwan Model for Lap-Type Joints," *Journal of Applied Mechanics*, vol. 72, pp. 752-760, September 2005.
19. D. D. Quinn and D. J. Segalman, "Using Series-Series Iwan-Type Models for Understanding Joint Dynamics," *Journal of Applied Mechanics*, vol. 72, pp. 666-673, 2005. Daniel J. Segalman, Danny L. Gregory, Michael J. Starr, Brian R. Resor, Michael D. Jew, James P. Lauffer, & Nicoli M. Ames, "Handbook on Dynamics of Jointed Structures", SANDIA REPORT SAND2009-4164, July 2009.
20. O.V. Shiryayev, S.M. Page, C.L. Pettit, J.C. Slater, "Parameter estimation and investigation of a bolted joint model", *Journal of Sound and Vibration*, Volume 307, Issues 3–5, 6 November 2007, Pages 680-697.
21. T.J. Royston, "Leveraging the equivalence of hysteresis models from different fields for analysis and numerical simulation of jointed structures", *ASME J. Comput. Nonlinear Dynam.*, 3 (2008), pp. 031006-1–031006-8.
22. Ivan I. Argatov, Eric A. Butcher, On the Iwan models for lap-type bolted joints, *International Journal of Non-Linear Mechanics*, Volume 46, Issue 2, March 2011, Pages 347-356.
23. Yikun Li, Zhiming Hao, A six-parameter Iwan model and its application, *Mechanical Systems and Signal Processing*, Volumes 68–69, February 2016, Pages 354-365.
24. S. S. Chen, M. W. Wambsganss and J. A. Jendrzejczyk, "Added Mass and Damping of a Vibrating Rod in Confined Viscous Fluids", *Journal of Applied Mechanics*, Volume 43(2), 325-329 (Jun 01, 1976).
25. S. S. Chen and J. A. Jendrzejczyk, "Flow velocity dependence of damping in tube arrays subjected to liquid cross flow", *Journal of Pressure Vessel Technology* 103, pp. 130-135, 1981
26. M. Rahman, D.D. Bhatta, "Evaluation of added mass and damping coefficient of an oscillating circular cylinder", *Applied Mathematical Modelling*, Volume 17, Issue 2, 1993, Pages 70-79.
27. M. J. Pettigrew, R. J. Rogers and F. Axisa, "Damping of Heat Exchanger Tubes in Liquids: Review and Design Guidelines", *Journal of Pressure Vessel Technology*, Volume 133(1), 014002 (Jan 21, 2011).
28. 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)
29. L. Brockmeyer, E. Merzari, J. Solberg, Y. Hassan, "Simulation of Fluid Structure Interaction of Crossflow Through Tube Bundle and Experimental Validation", *Proceedings of the ASME Fluids Engineering Summer Meeting 2017*, Hawaii (August 2017).
30. E. Merzari, H. Yuan, J. Solberg, A. Kraus, and R. Ferencz, "Coupled calculations in helical steam

generators: Validation against legacy data,” Argonne report, 2017.

31. 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.

ANL/NSE-19/20

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