Isight-Abaqus Optimization of a Ring-Stiffened Cylinder

Nicholas Barner
Ph: 860-433-7553
Fax: 860-433-3229
nbarner@gdeb.com

General Dynamics - Electric Boat Corporation
Department 463
75 Eastern Point Road
Building J197-3
Groton, CT 06340

Abstract

Electric Boat's design process involves evaluating the structural stability of ring-stiffened cylinder structures through finite element analyses to simulate a static pressure load. Each design revision of the cylinders must be evaluated to verify that the structure meets the required stress criteria for the static pressure load; any revision to geometry or material would require the design to be reevaluated. Additionally, it is critical that the weight of the structure is kept as light as possible while still satisfying all stress and deflection criteria. This presentation documents the use of Isight and Abaqus to perform a weight optimization for static pressure loading on a ring-stiffened cylinder, varying geometry and model thicknesses. The use of Isight to run the analysis allows Electric Boat to find an optimized design earlier in the design process.

This work was conducted under the auspices of the National Shipbuilding Research Program's (NSRP) Advanced Shipbuilding Enterprise (ASE) Full-Ship Simulation/Analysis Models 2 Project (M&S 2) and Electric Boat's Independent Research and Development (IR&D).

1. Modeling & Simulation

This paper will present an Isight optimization model of a ring-stiffened cylinder performed using Abaqus as the analysis tool. This optimization is designed to minimize the weight of the cylinder, varying model dimensions and thicknesses, for a hydrostatic load. The work presented was performed as part of the NSRP M&S project (Moore & Gordon, 2008) (Moore & Gordon, 2009). The project's objective is to investigate methods to reduce the time required to perform full-ship analyses. Reducing the time required to perform the analyses enables the simulation results to be available for use at the beginning of the ship design cycle, thereby providing a positive cost effective impact as ship design progresses through its lifecycle.

2. Model Creation

A finite element model (FEM) resembling a German Type 212 submarine is used for the optimization. Geometry is created which roughly approximates the publicly available information for a German Type 212 design. While the ship dimensions are approximated, all loads and design criteria used in the analysis presented in this paper are selected arbitrarily and do not reflect an actual submarine design. Figure 1 shows a photograph taken of a German Type 212 submarine. Figure 2 shows the Type 212 schematic.

Figure 1. Type 212 submarine.

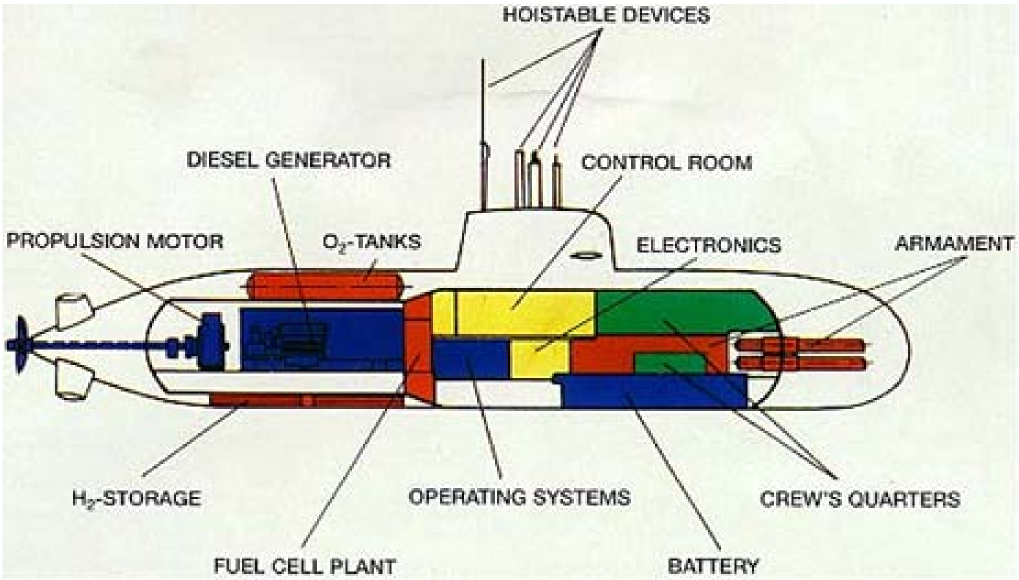

Figure 2. Type 212 schematic.

Abaqus/CAE is used for model generation and pre-processing. Using the acquired Type 212 information, the molded hull form surfaces are approximated for the pressure hull. Pressure hull frames are later added to suit the approximated molded forms hull geometry. The model is created using Abaqus/CAE's sketcher tool and is shown in Figure 3.

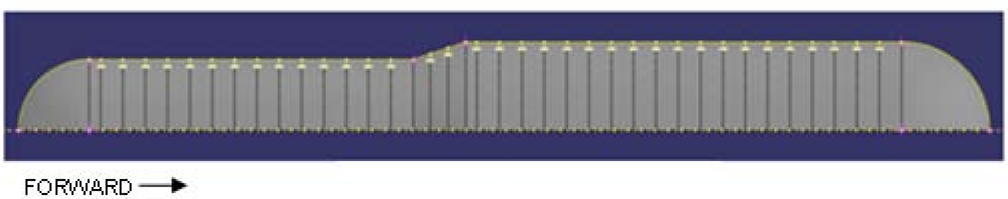

Figure 3. Abaqus/CAE sketch.

With the sketcher tool, a 2-dimensional sketch of the cross-section is generated, as shown in Figure 3. The sketcher will then revolve the cross-section about the ship's main axis. Since the model is axisymmetric, it is only revolved 90-degrees.

Geometric constraints and dimensions are added to the sketch. The constraints and dimensions used in the model are shown in Figure 4.

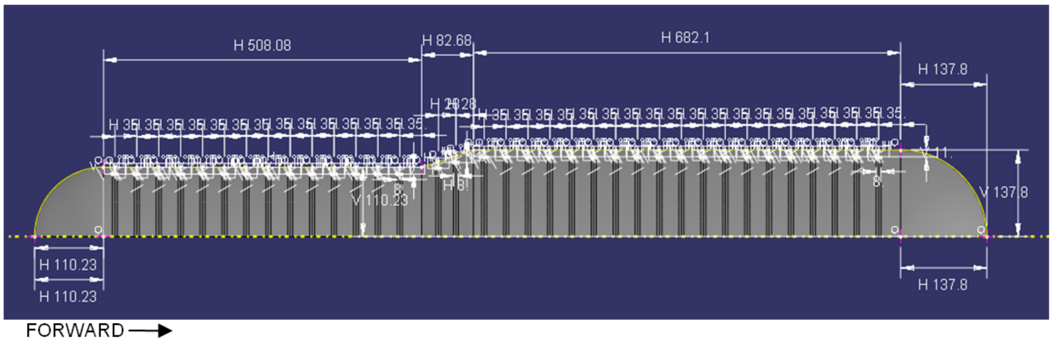

Figure 4. Dimensioned sketch.

Appropriate geometric parameters are created from the dimensions in order to modify the hull geometry for the optimization. Geometry parameters must be independent; changing a parameter cannot impact any other parameter in the model. Parameters must also be defined so that, once changed, they have the desired effect on the model. In total, 14 geometry parameters are created for the model:

1. Forward Hull Radius
2. Forward Cylinder Length
3. Forward Frame Spacing
4. Forward Frame Depth
5. Forward Flange Width
6. Center Section Length
7. Center Frame Spacing
8. Center Frame Depth
9. Center Flange Width
10. Aft Hull Radius
11. Aft Cylinder Length
12. Aft Frame Spacing
13. Aft Frame Depth
14. Aft Flange Width

It should be noted that the Forward, Center, and Aft designators refer to the three distinct sections of the ship. The Forward and Aft End Closures are modeled as hemispherical heads with radii equal to the Forward and Aft Cylinder sections. All frames in the ship are modeled as typical frames with equivalent geometric dimensions and frame spacing for each of the three segments. There are 19 frames modeled in the Forward section, two in the Center section, and 14 in the Aft section. And although the Center section is modeled as a cone, there are no constraints or parameters applied to control the cone angle; the cone angle is controlled by the Forward and Aft hull radii. In addition to the 14 geometry parameters, 11 distinct thickness sections are created and applied to the model:

1. Forward Cylinder
2. Forward Flanges
3. Forward End Closure
4. Forward Webs
5. Center Cylinder
6. Center Flanges

7. Center Webs
8. Aft Cylinder
9. Aft Flanges
10. Aft End Closure
11. Aft Webs

Figure 5 shows a diagram of typical model geometry used for the optimization.

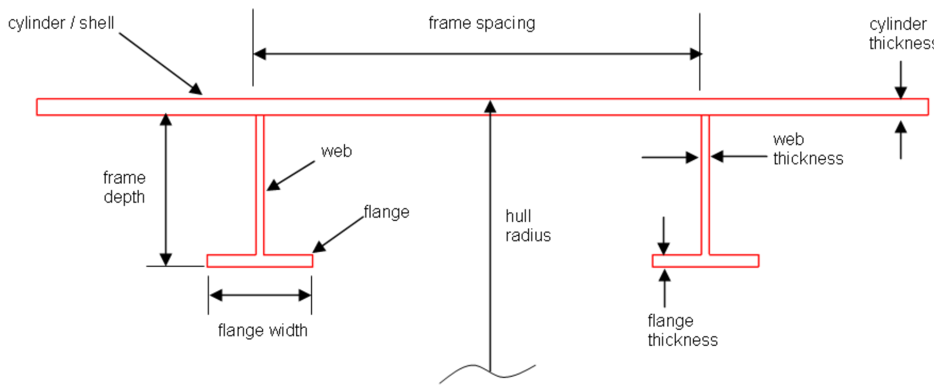

Figure 5. Typical model geometry.

The revolved model geometry is shown in Figure 6.

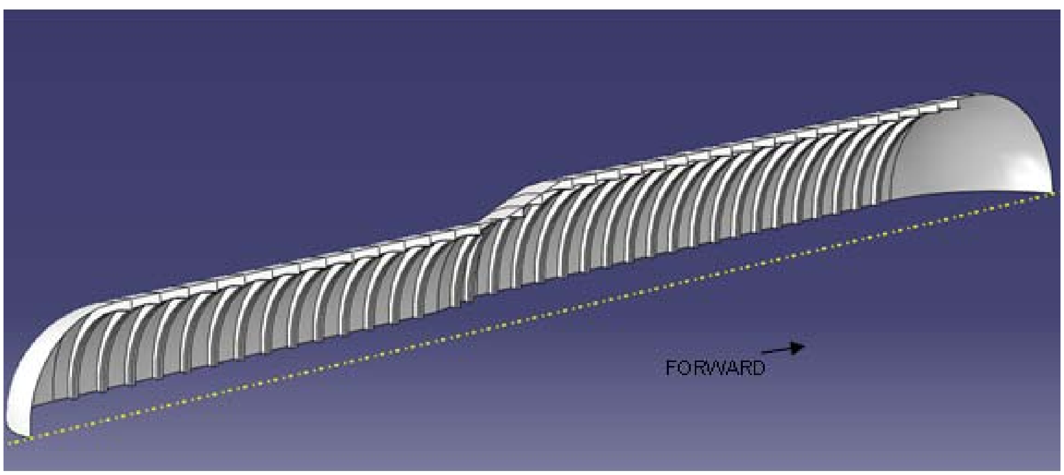

Figure 6. Type 212 model geometry.

Mesh seeds and controls are then applied to the model. The mesh seeds and controls are applied every time the model is regenerated and remeshed during the optimization to ensure consistent mesh generation. The mesh seeding and generated mesh are shown in Figures 7 and 8, respectively.

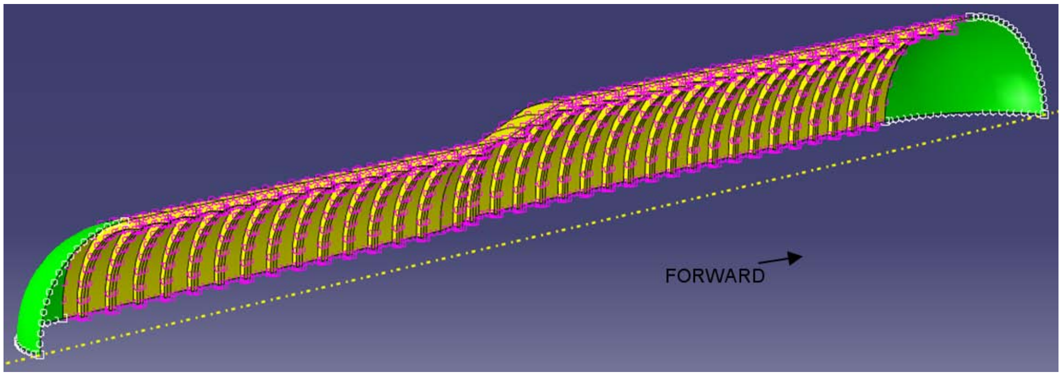

Figure 7. Type 212 mesh seeding.

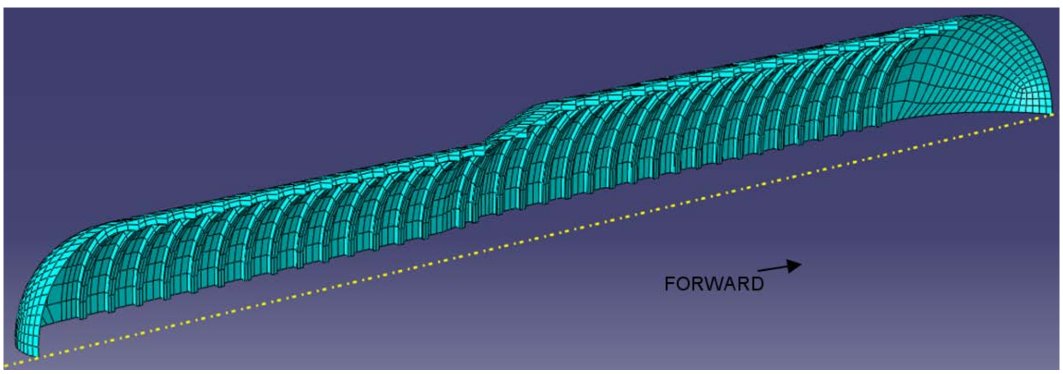

Figure 8. Type 212 mesh.

The model shown has 2920 shell elements, 3250 nodes, and 17,520 degrees of freedom (DOF). Steel material properties were applied to all elements in the model:

- Elastic Modulus = 30 x 10^6 psi.
- Poisson's ratio = 0.3
- Density = 0.283 lb/in^3

A pressure load of 100 psi, selected arbitrarily to simulate hydrostatic loading, is applied to all elements of the pressure hull shell. The center node on the Aft End Closure is fully fixed, with all translations and rotations bound. Symmetry boundary conditions are applied to both cut planes in the model. The linear static analysis is then run using Abaqus/Standard.

The initial deflected shape plot, with a scale factor of 150, is shown in Figure 9.

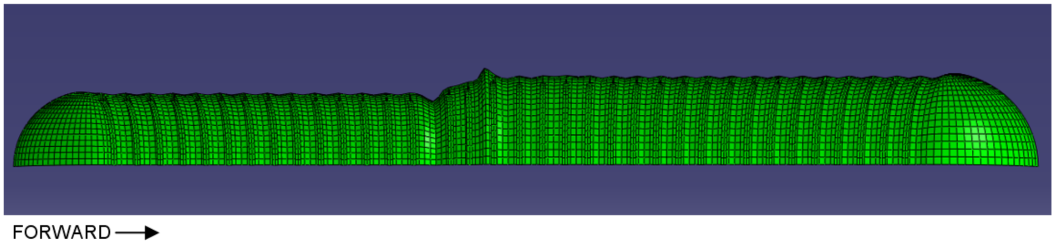

Figure 9. Type 212 deflected shape.

3. Isight Model

An Isight model is created to manage the optimization as shown in Figure 10.

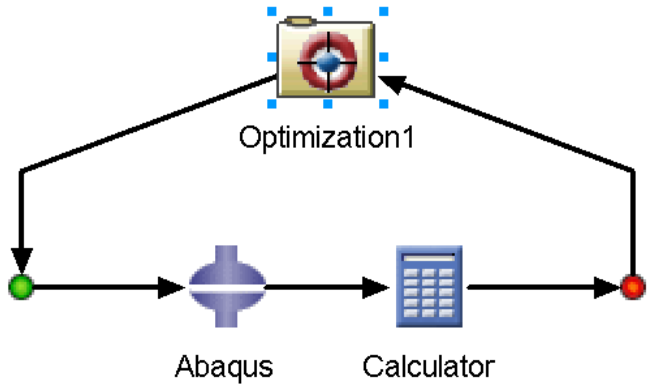

Figure 10. Isight optimization model.

The Abaqus component reads the Abaqus/CAE database as its primary input file for this optimization model. It reads the Abaqus data file as its output. The Abaqus data file is selected instead of the Abaqus output database because the data file grants the user greater control of the output generated by Abaqus/Standard and extracted by Isight.

Using the Abaqus component, the 14 geometric dimension properties and 11 element thickness properties are selected as input parameters. The total volume of the model, minimum and maximum principal stresses, and the minimum and maximum vertical and athwartship deflections are selected as output parameters to be monitored. The Calculator component is used to calculate the total weight of the ship using the total calculated element volume parameters; the optimization is set to calculate the weight of the entire 360-degree ship and not the 90-degree analyzed section. It also calculates that absolute maximum principal stress and the absolute maximum vertical or athwartship deflection.

Using the Optimization driver, the objective is to minimize total ship weight. The maximum absolute principal stress is constrained to 20 ksi. The maximum absolute vertical or athwartship deflection in the model is constrained to 0.0875-inch.

The 14 geometric dimension properties and 11 element thickness properties are selected as input parameters to be varied. Allowed values for the 11 thickness parameters are limited to either ¼-inch or ⅛-inch increments. The allowed values for the typical property sets are shown in Table 1.

Table 1. Allowed thickness values

Geometry Type	Thickness Values					
Cylinders	0.75"	1.00"	1.25"	1.50"		
End Closures	1.50"	1.75"	2.00"	2.25"	2.50"	2.75"
Webs	0.375"	0.50"	0.625"	0.75"		
Flanges	0.50"	0.625"	0.75"	0.875"	1.00"	1.125"

For the 14 geometric dimension properties, allowed frame spacing values for the Forward and Aft sections are set to 32.0”, 33.0”, 34.0”, and 35.0”. The allowed frame spacing values for the Center section are set to 26.0”, 27.0”, 28.0”, and 29.0”. The remaining geometric parameter values are given lower and upper bounds based on a simple tolerance stack-up. The lower and upper bounds for the remaining geometric parameter values are shown in Table 2.

Table 2. Allowed geometry values.

Geometry	Lower Bound	Upper Bound
Forward Hull Radius	130"	155"
Forward Cylinder Length	675"	690"
Forward/Center/Aft Frame Depth	7.0"	15.0"
Forward/Center/Aft Flange Width	5.0"	11.0"
Center Section Length	75"	90"
Aft Hull Radius	100"	125"
Aft Cylinder Length	500"	515"

4. Initial Conditions

For this optimization, the initial values for the 14 geometric dimension properties and 11 element thickness properties to be varied are shown in Table 3.

Table 3. Initial model values.

Geometry	Initial Value
Forward Hull Radius	137.8"
Forward Cylinder Length	682.1"
Forward Frame Spacing	35.0"
Forward Frame Depth	11.0"
Forward Flange Width	8.0"
Center Section Length	82.68"
Center Frame Spacing	28.0"
Center Frame Depth	11.0"
Center Flange Width	8.0"
Aft Hull Radius	110.23"
Aft Cylinder Length	508.08"
Aft Frame Spacing	35.0"
Aft Frame Depth	11.0"
Aft Flange Width	8.0"

Thickness	Initial Value
Forward Cylinder	1.0"
Forward Flanges	0.75"
Forward End Closure	2.25"
Forward Webs	0.5"
Center Cylinder	1.25"
Center Flanges	0.75"
Center Webs	0.5"
Aft Cylinder	1.0"
Aft Flanges	0.75"
Aft End Closure	2.25"
Aft Webs	0.5"

Using the initial values shown, the following output values are calculated:
Maximum absolute principal stress: 24.86 ksi
Maximum absolute vertical/athwartship deflection: 0.0805”
Total weight: 499,495 lb

As shown above, the initial condition does not satisfy the stress constraint applied to the model.

5. Optimization

The optimization run in Isight uses a Non-Linear Programming by Quadratic Lagrangian (NLPQL) technique. NLPQL is a direct numerical technique. It builds a quadratic approximation to exploit the local area around the initial design point and finds a local optimum design and then repeats the process through multiple iterations to converge on a final optimum design. This technique is selected because it is well-suited for long-running simulations and highly non-linear design spaces.

The general process that Isight uses to run an Abaqus analysis with an Abaqus/CAE database is as follows:

1. Isight updates the applicable input parameters, such as element thicknesses and geometry parameters. Using the parameters, Isight regenerates the part, remeshes the model based on the mesh seeds and controls previously defined by the user, applies the appropriate loads and boundary conditions, and creates the Abaqus input file.
2. Isight submits the Abaqus job.

3. Isight parses and extracts the output.
4. Files are deleted and the process is repeated.

6. Results

The results of the optimization are shown in Table 4.

Table 4. Optimization results.

Geometry	Initial Value	Optimized Value
Forward Hull Radius	137.8"	130.0"
Forward Cylinder Length	682.1"	681.72"
Forward Frame Spacing	35.0"	35.0"
Forward Frame Depth	11.0"	8.16"
Forward Flange Width	8.0"	5.0"
Center Section Length	82.68"	84.63"
Center Frame Spacing	28.0"	26.0"
Center Frame Depth	11.0"	10.59"
Center Flange Width	8.0"	7.40"
Aft Hull Radius	110.23"	120.3"
Aft Cylinder Length	508.08"	507.77"
Aft Frame Spacing	35.0"	35.0"
Aft Frame Depth	11.0"	9.33"
Aft Flange Width	8.0"	5.35"

Thickness	Initial Value	Optimized Value
Forward Cylinder	1.0"	1.0"
Forward Flanges	0.75"	0.5"
Forward End Closure	2.25"	1.5"
Forward Webs	0.5"	0.375"
Center Cylinder	1.25"	0.75"
Center Flanges	0.75"	0.625"
Center Webs	0.5"	0.375"
Aft Cylinder	1.0"	0.75"
Aft Flanges	0.75"	0.5"
Aft End Closure	2.25"	1.5"
Aft Webs	0.5"	0.375"

Calculated output values:
Maximum absolute principal stress: 18.66 ksi (24.86 ksi initially)
Maximum absolute vertical/athwartship deflection: 0.0667” (0.0805” initially)
Total weight: 376,458 lb (499,495 lb initially)

For the NLPQL optimization, Isight performs 134 total evaluations in 81 minutes. The average time per evaluation is 36 seconds. Additionally, Electric Boat uses approximately 20 total hours to create the FEM, perform all preprocessing, and to create and fully configure the Isight model.

Manually modifying the 25 input parameters in the Abaqus/CAE model, regenerating geometry and remeshing, submitting the Abaqus analysis, and extracting and logging the desired output would take an engineer an estimated 30 minutes per evaluation (versus the 36 seconds used by Isight). However, if the engineer didn’t have the model set up similarly in Abaqus/CAE and was manually modifying model geometry and mesh for each evaluation, the time per evaluation would be significantly greater.

The optimized solution decreases the total weight by 24.6% from the initial condition used. Maximum absolute principal stress in the model decreases 24.9% from the initial condition.

Considering that the initial condition used for the optimization was a valid ship design, Electric Boat considers the resultant weight and stress decreases to be significant.

7. Conclusions

An optimization such as the one discussed in this paper is beneficial because it allows the ship designer to examine multiple configurations earlier in the design process. Optimizations as extensive as the one discussed were not previously used by Electric Boat due to the considerable time and effort required by an analyst to perform them. Electric Boat feels that the Isight tool allows the ship designer to more fully explore design possibilities earlier in the detailed design process, effectively implementing a more computer-aided engineering data-centric approach to ship design.

Electric Boat plans to investigate using Isight to perform similar optimization analyses with its in-house software codes. This would allow engineers to perform an optimization for multiple types of analyses, such as performing a weight optimization for an operational load and a transient shock analysis. Electric Boat will further investigate implementing Isight for future production design support.

References

1. Moore, S., and S. Gordon, "M&S-1 Project Report, Improved Methods for Generation of Full-Ship Simulation/Analysis Models", Electric Boat Report, June 30, 2008 (Performed under NSRP Technology Investment Agreement TIA 2007-381).

2. Moore, S., and S. Gordon, "M&S-2 Project Report, Improved Methods for Generation of Full-Ship Simulation/Analysis Models", Electric Boat Report, June 30, 2009 (Performed under NSRP Technology Investment Agreement TIA 2008-381).

Acknowledgment

The author would like to thank Stewart Moore and Stephen Gordon of Electric Boat and John Modesto, Justin Vianese, and Dave Naehring of SIMULIA for their assistance on this project and for their contributions to the NSRP M&S team.