NSRP RA Project

Development of a Fast Analysis Solver for Welding Sequence Optimization of Ship Structures

Joint Panel Meeting

May 1, 2024

Project Team

HII Ingalls Shipbuilding (Lead)

- Steve Scholler, Yu-Ping Yang, Jamie Breakfield

Austal USA (unfunded)

- Shawn Wilber and others TBD

General Motors LLC

- Hassan Ghassemi-Armaki

Hexagon

- Jeff Robertson

Oak Ridge National Laboratory

- Zhili Feng, Jian Chen

INGALLS SHIPBUILDING
A Division of HII

HEXAGON

OAK RIDGE National Laboratory

Project Oversight

NSRP Project Manager

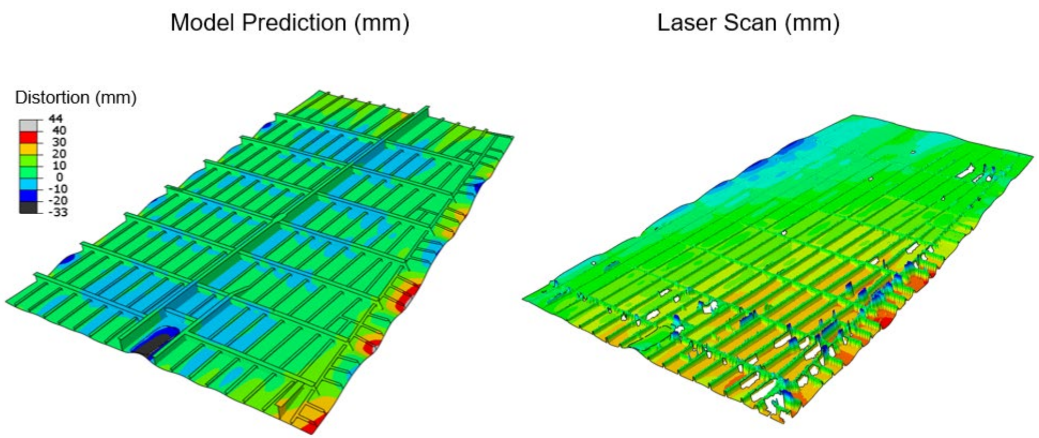
Ryan Schneider, ATI

Project Technical Representative (PTR)

Alicia Harmon, HII Newport News Shipbuilding

Problem Statement

- Materials are exposed to significant thermal and mechanical stresses during welding that affect dimensional accuracy, production schedules, labor hours (fitting, welding, rework, etc.) and structural performance
  - Many variables influence the stresses and resulting impacts
- Structural analyses that simulate the thermal and mechanical stresses are time-consuming
  - Simulations to optimize welding sequences and minimize impacts are cost prohibitive
  - Production uses other metrics for weld sequencing



Project Goals and Desired Outcome

- **Goals**
  - Adapt the DR-WELD transient elastic-plastic finite element analysis (FEA) solver developed by Oak Ridge National Laboratory (ORNL) for the automotive and nuclear industries to support shipbuilding applications
  - Apply the FEA solver to simulate fabrication of production panels, unit assemblies, and alignment critical foundations to examine its feasibility, effectiveness and accuracy
- **Desired outcome**: A transient elastic-plastic FEA solver for welding simulation that will be feasible in a shipbuilding environment, provide accelerated analysis speeds and is integrated with a user-friendly interface

Work Breakdown Structure – Phase 1

- Task 1 – Evaluate the Current ORNL Solver
- Task 2 – Identify the Gaps and Produce Solver Requirements
- Task 3 – Develop a Preliminary Version of Software for Panel Structures
- Task 4 – Develop a Special Version of Software for Panel Structures
- Task 5 – Phase I Reporting

Phase 1 Activity

- The current solver is not fully functional for shipbuilding "out of the box"
- An evaluation to support identification of "gaps" and development of requirements for solver use on stiffened panels is nearing completion
  - Subtask 1.5: Prepare total process flow map for analyses
  - Subtask 1.6: Evaluate the current ORNL solver
  - Subtask 1.7: Prepare solver evaluation summary report

Total Process Flow Map for Analyses

1. Export a solid structure model from **ShipConstructor** or **other CAD Software**
2. Create a global FE model using **FEMAP** or **Hyperworks**
3. Create local FE models using **FEMAP** or **Hyperworks**
4. Run thermomechanical analyses on local models using **ABAQUS**
5. Map Local model results using **Scripts** to the global model
6. Predict distortion according to welding sequence using **ABAQUS**
7. Process analysis results using **ABAQUS**

Current Process

1. Export a solid structure model from **ShipConstructor** or **other CAD software**
2. Create a global FE model using **Hexagon or FEMAP** or **Hyperworks**
3. Predict temperature using **Hexagon** or **ABAQUS**
4. Predict distortion according to welding sequence using **DR-Weld**
5. Process analysis results using **Hexagon** or **Open Source Codes**

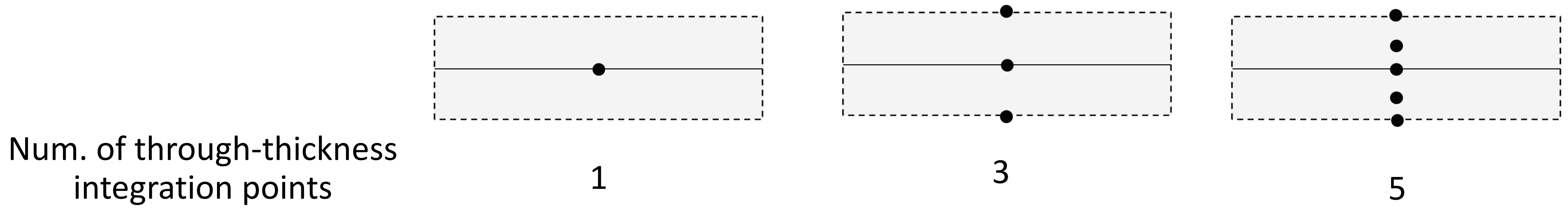
Developed Process

Gaps Identified

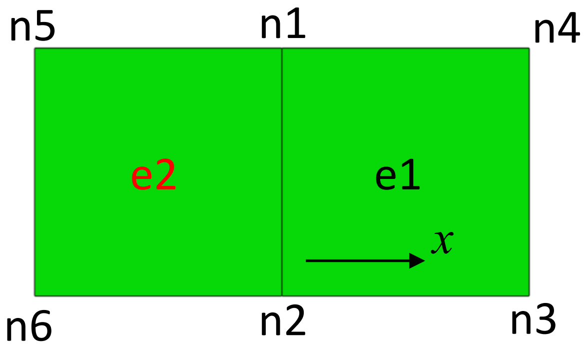
- DR-Weld could not process shell elements
  - Shell elements are required due to size of ship structure and processing time that would be required using solid or solid-shells
  - Analysis focuses mainly on resulting trends from process changes and absolute result accuracy is not critical
  - ORNL has implemented shell elements into DR-Weld
  - Hexagon is developing prototype GUI to handle shell element results
- Need for automated mesh generation
  - Current model setup time is time consuming and inefficient
  - Hexagon has an automated mesh generation feature the team is testing within our process flow

Shell elements developed in this project

- Four shell element types were completed
  - *S4*: 4-node full-integration
  - *S4R*: 4-node reduced-integration
  - *S3*: 3-node full-integration (degeneration of S4)
  - *S3R*: 3-node reduced integration (degeneration of S4R)
- All elements supports multiple *through-thickness* integration by Simpson Rule. (number of *through-thickness* integration points: 1, 3, 5, 7, …)



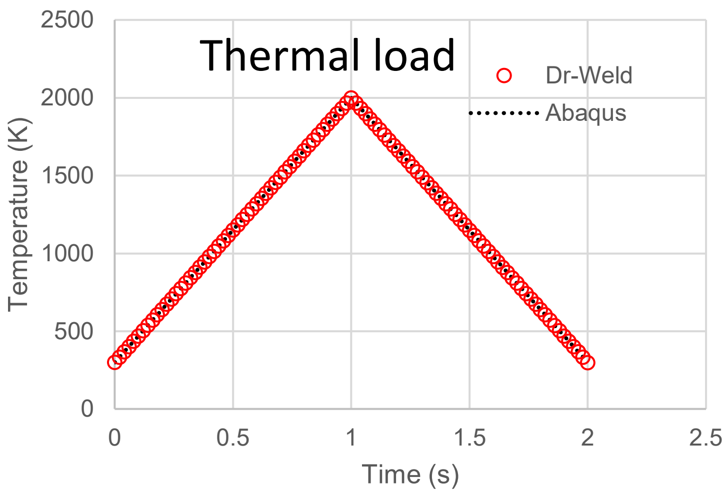
Preliminary evaluation of 4-node shell elements (S4R)

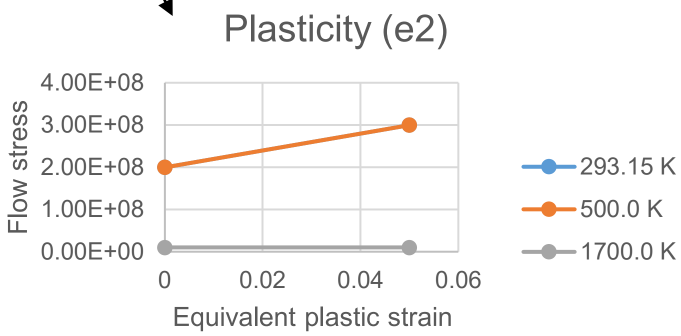


Boundary condition: only allows n1 and n2 to move along x direction. Other DOFs are fixed.

```
Input file:                       s4rx2.dat
Material file:                    Material-shell-test.dat
Progressive activation disabled.
Temperture file:                  Temp.dat
Num. of total nodes:              6
Num. of total elements:           2
Num. of active elements:          2
Num. of fixed DOF:                34
Num. of CPU threads:              1
```

Element ID	e2 (weld metal)	e1 (spring)
Elastic property	E=2e11, Nu=0.33	E=2e11, Nu=0.33
Plasticity	Temperature dependent	No plasticity
Anneal Temp	1700K	No annealing
Thermal expansion	2e-5	0



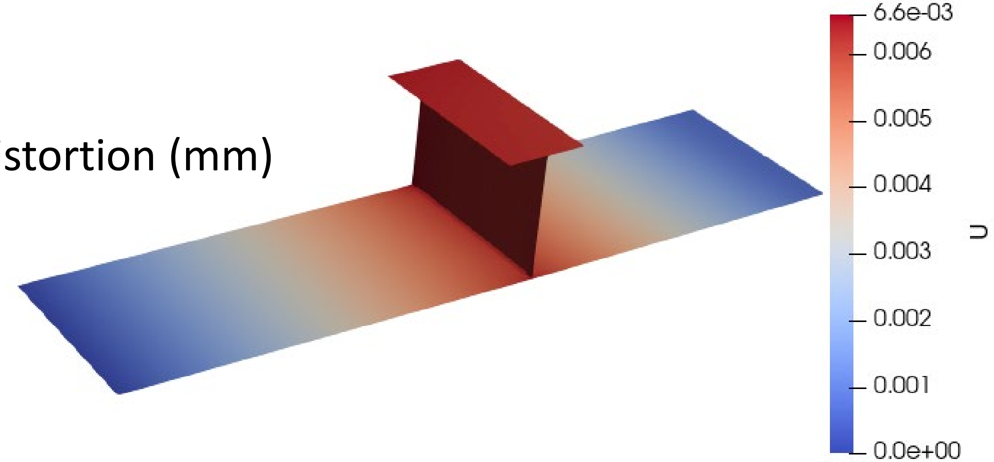


Implemented implicit solver. Results look promising. Fine-tuning the algorithms in progress

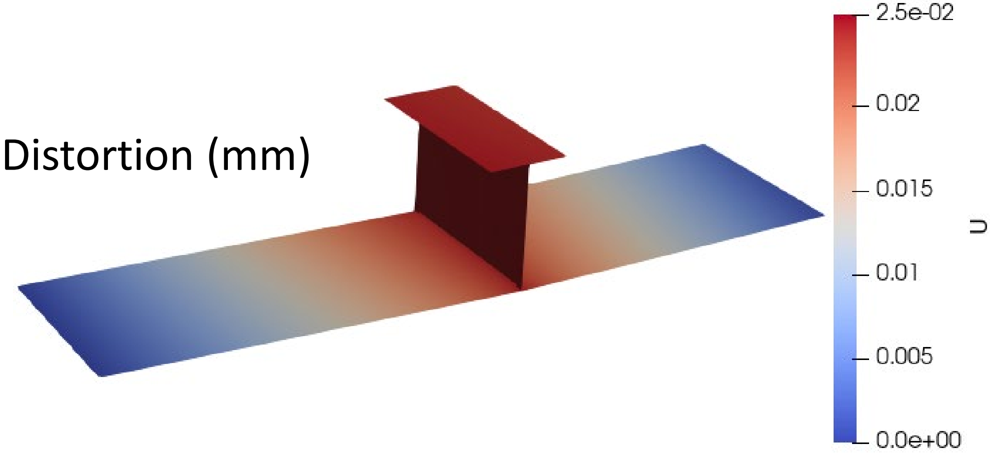
Dr-Weld

ABAQUS

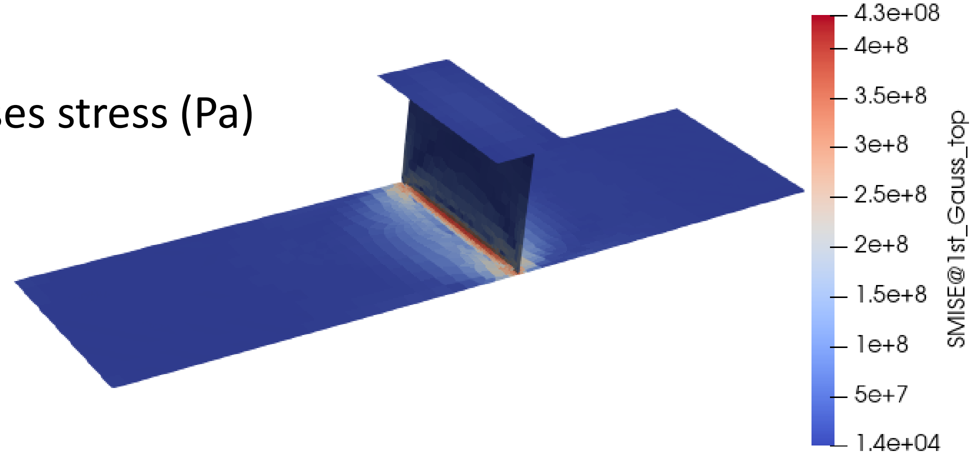
Distortion (mm)



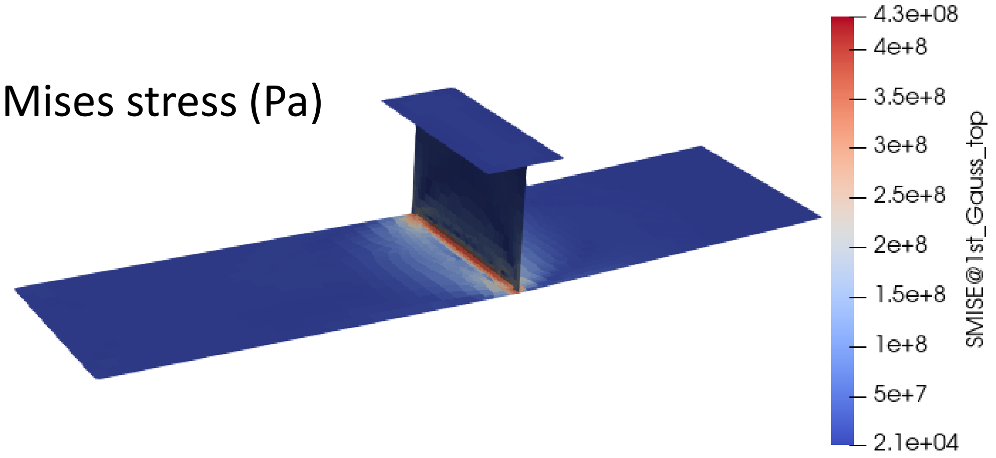
Distortion (mm)



Mises stress (Pa)



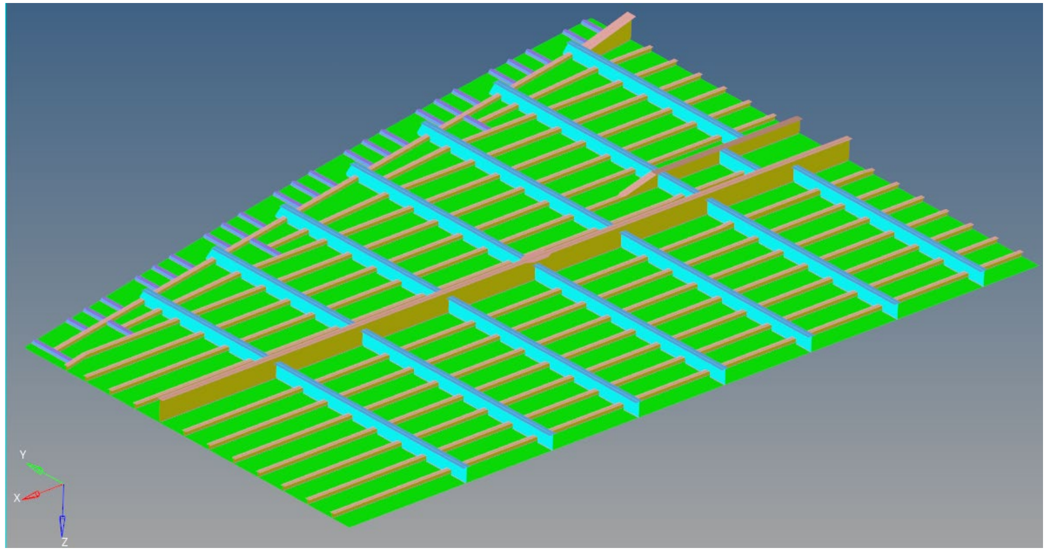
Mises stress (Pa)

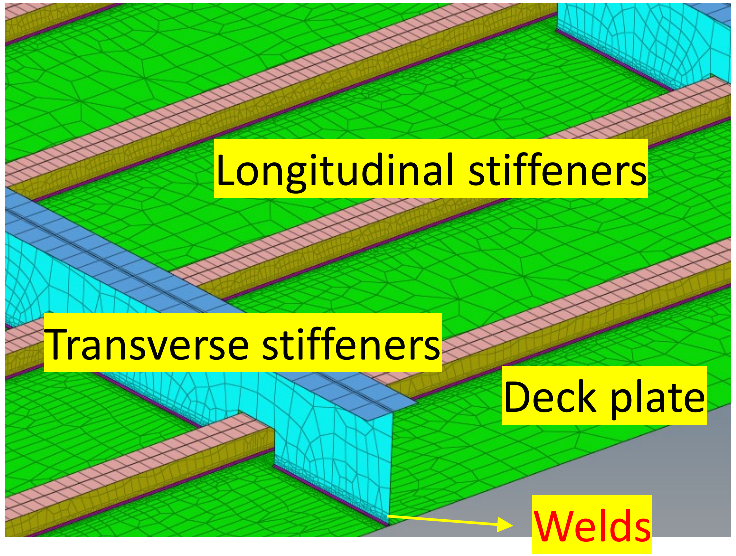


Phase-1 Objective

- The goal for phase 1 is to analyze a production panel using DR-Weld in under two days. Historic analyses use an approximation method, inherent strain method
- The inherent strain method is difficult to optimize welding sequence to minimize distortion since the thermal and mechanical interaction between welds during welding is ignored
- DR-Weld is a transient thermal elastic-plastic analysis which can capture the effect of weld interactions and is suitable for optimizing welding sequences

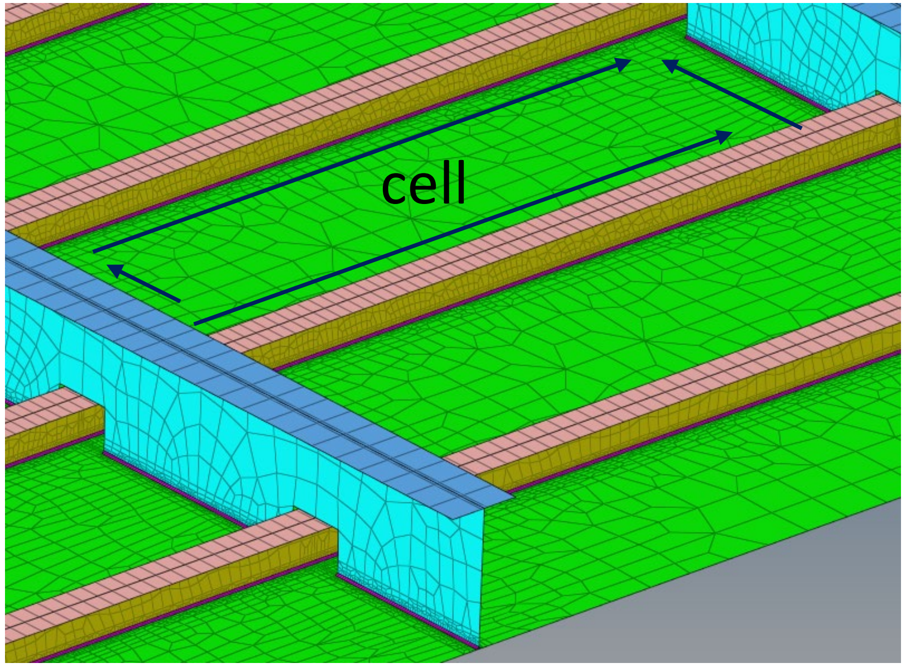
A production panel





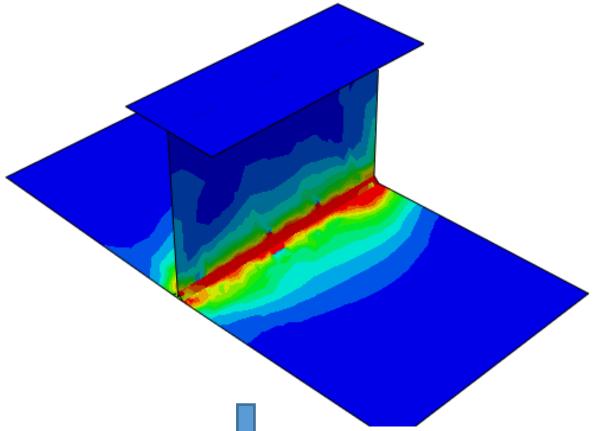
Welding the Selected Production Panel

- A typical production panel includes about 120 welding cells.
- Four robots are used to weld the panel simultaneously.
- Robots move from one weld cell to another so the cooling time between cells is negligible.
- The residual temperature from previous weld cells will affect the temperature of the following cells, which may contribute to the effect of weld sequence on distortion.

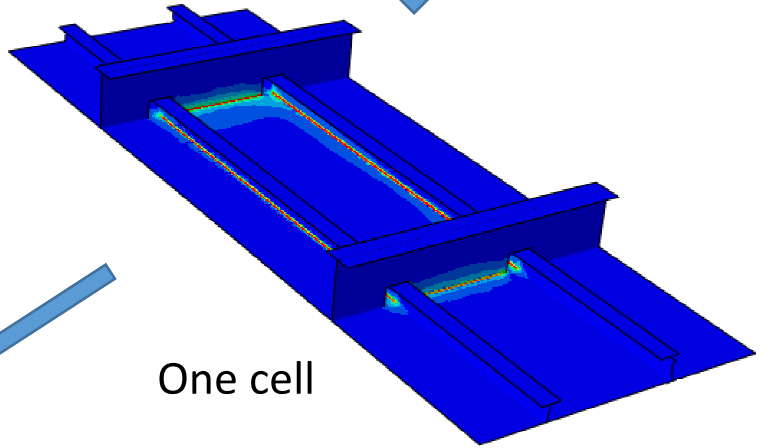


A Step-by-Step Approach

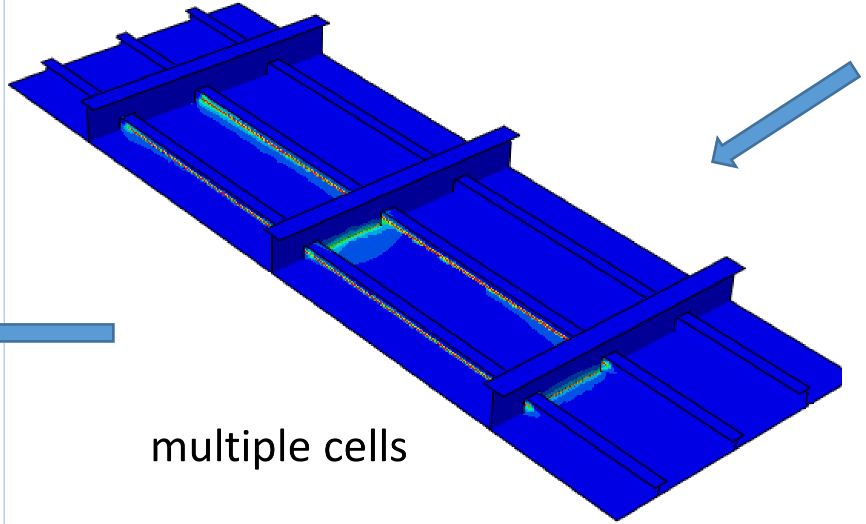
- DR-Weld will be developed and tested in four steps to achieve Phase-1 objectives
  - Step 1: Single Tee model
  - Step 2: One-cell model
  - Step 3: Multi-cell model
  - Step 4: Full production model



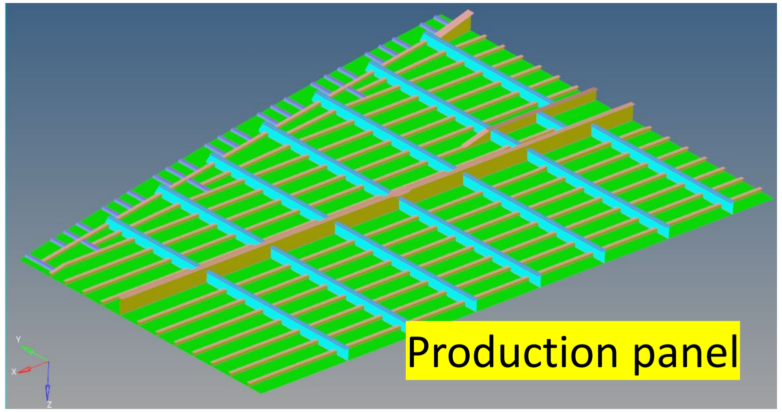
One Tee



One cell

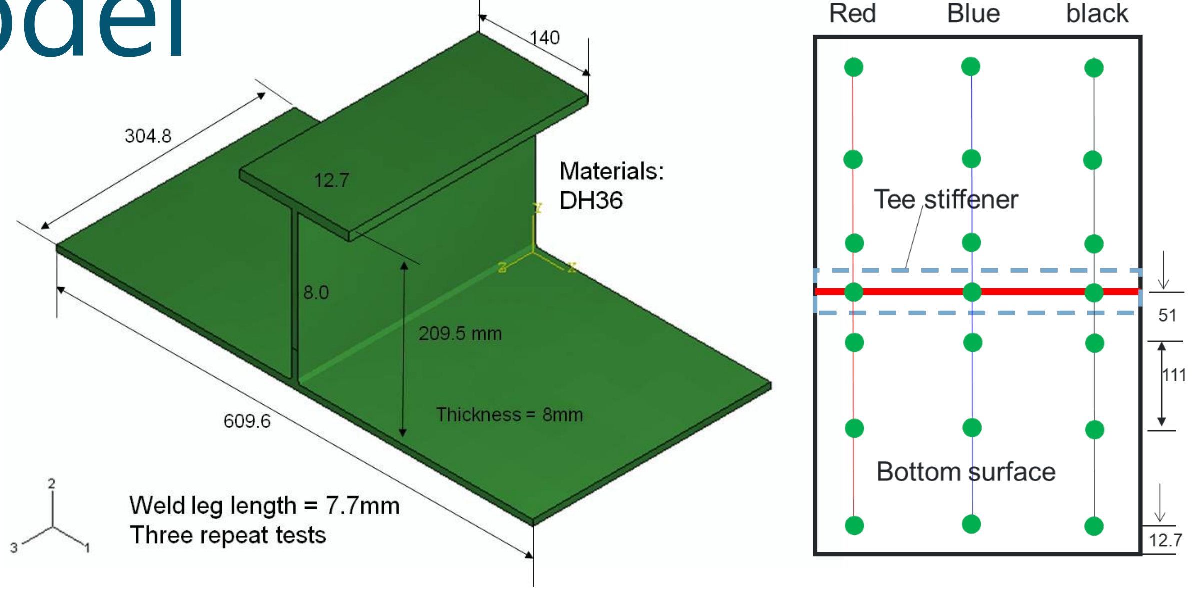


multiple cells

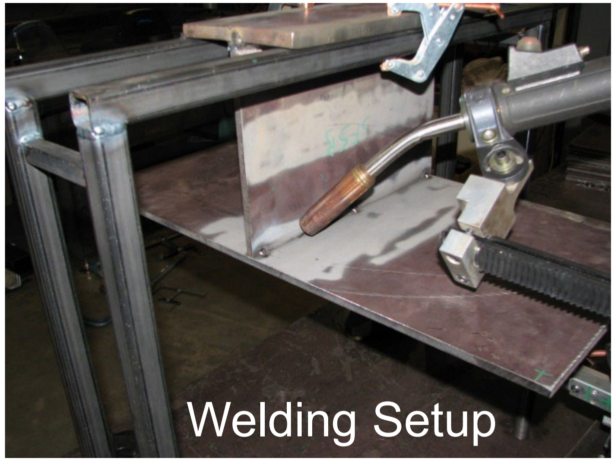


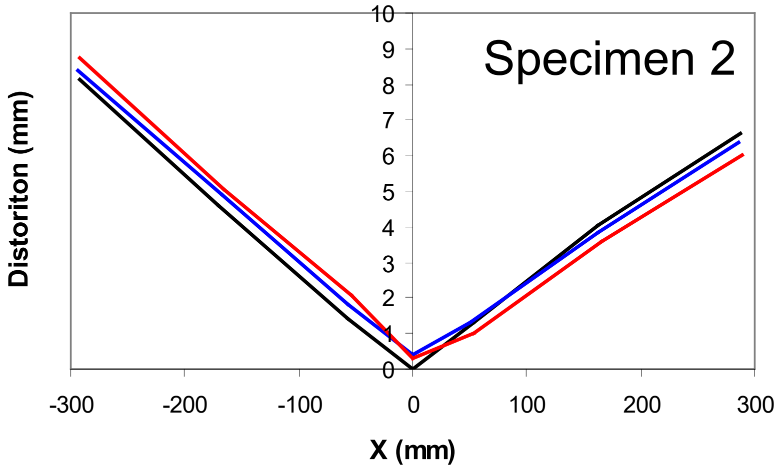
Analyze Single-Tee Model

- The Single-Tee model is critical to verify the accuracy of DR-Weld prediction.
- DR-Weld prediction will be compared with the experimental data and Abaqus prediction.



- **Distortion measurement**
  - Specimen 1: Left = 9.7mm; right = 6.2mm
  - Specimen 2: Left = 8.4mm; right = 6.4mm
  - Specimen 3: Left = 8.9mm; right = 5.9mm
- **Average**
  - left = 9.0mm; right = 6.2mm
  - **Average left and right: 7.6mm**



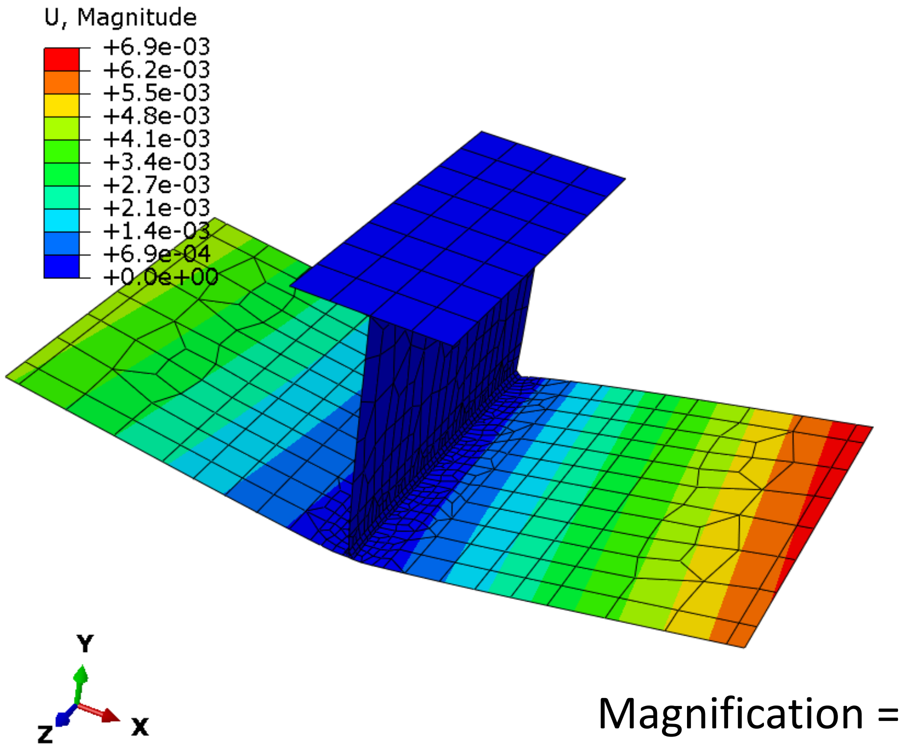


Ref. Y. P. Yang and R. Dull, 2018 Fabtech conference.

Verification of Abaqus Prediction

- Abaqus has been widely used to predict distortion induced by welding processes
- Abaqus prediction will be used to check the accuracy of DR-Weld prediction on the one-cell model and the multiple-cell model
- Abaqus prediction has been verified on the one-Tee model
  - Abaqus predicted the similar distortion shape as observed in the experiment
  - The predicted distortion magnitude was close to the experimental measurement

Distortion (m)



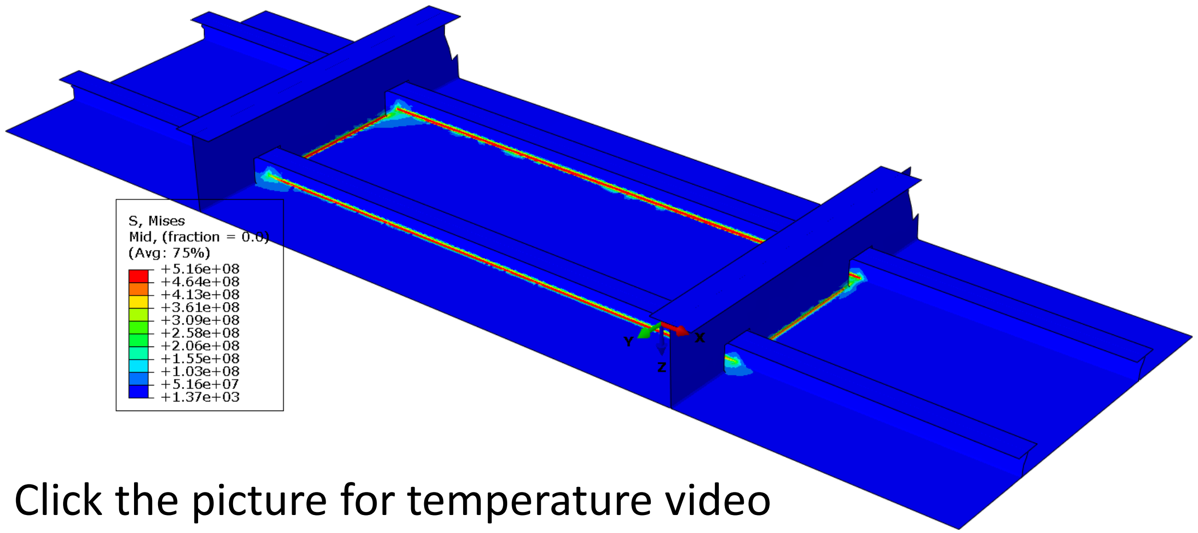
Magnification = 10

Abaqus predicted distortion

Analyze One-Cell Model

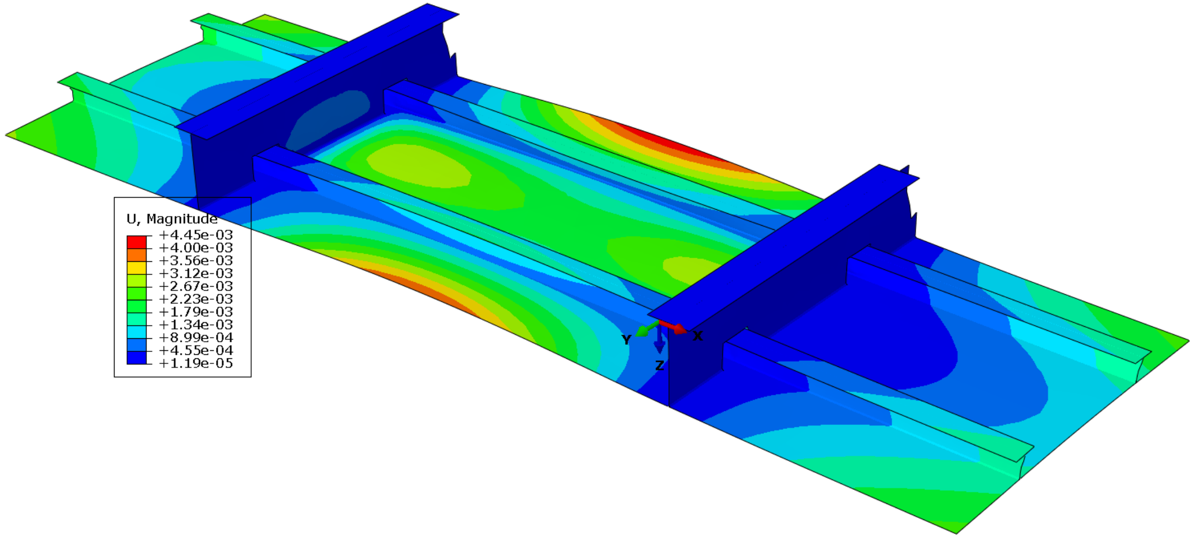
- One-cell model was analyzed with Abaqus to predict temperature and distortion.
- Abaqus temperature will be input to DR-Weld to predict distortion and compare with Abaqus results.

Predicted Von Mises Stress (Pa)



Click the picture for temperature video

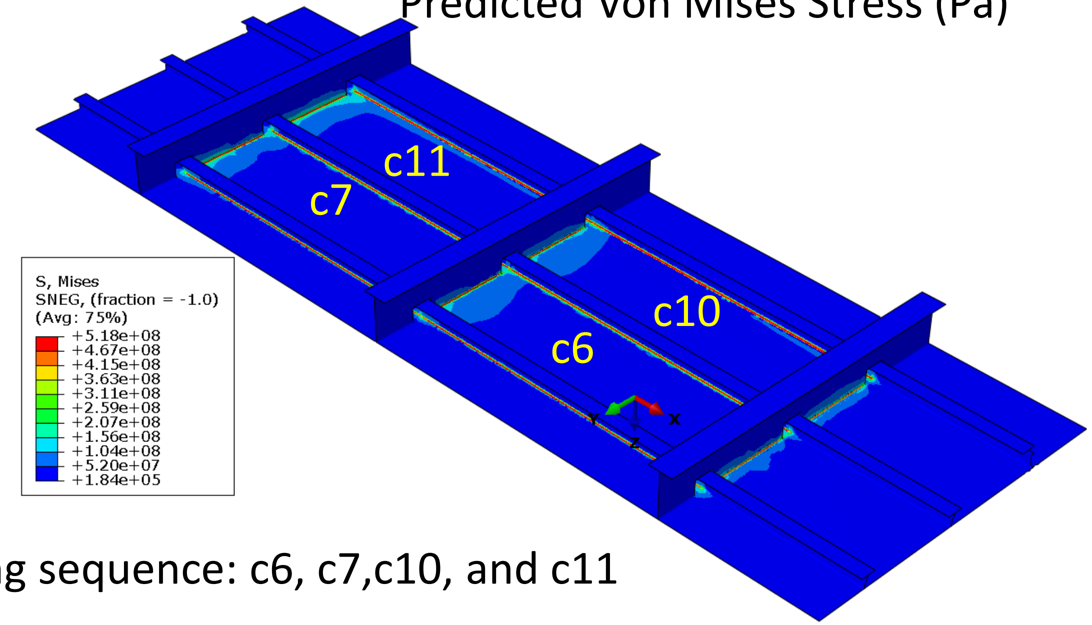
Predicted distortion (m)



Analyze Multi-Cell Model

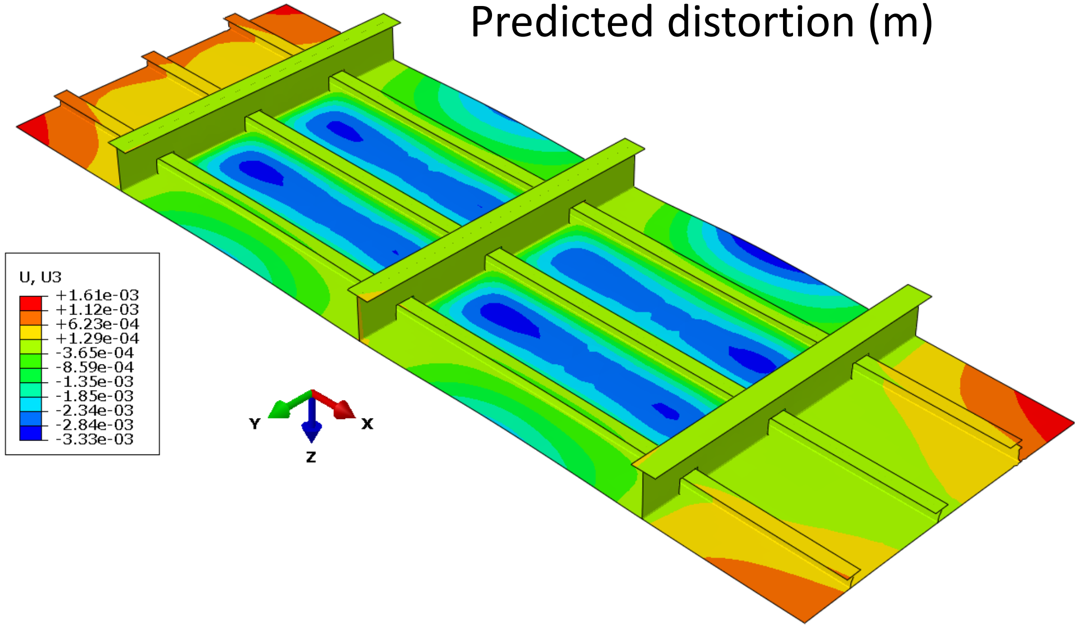
- Temperature at each welding cell was predicted using Abaqus separately. The residual temperature from previous cell welding was ignored in this analysis
- Each-cell temperature was read into Abaqus according to the welding sequence to predict stress and distortion, which will compare with DR-Weld predictions

Predicted Von Mises Stress (Pa)



Welding sequence: c6, c7,c10, and c11

Predicted distortion (m)



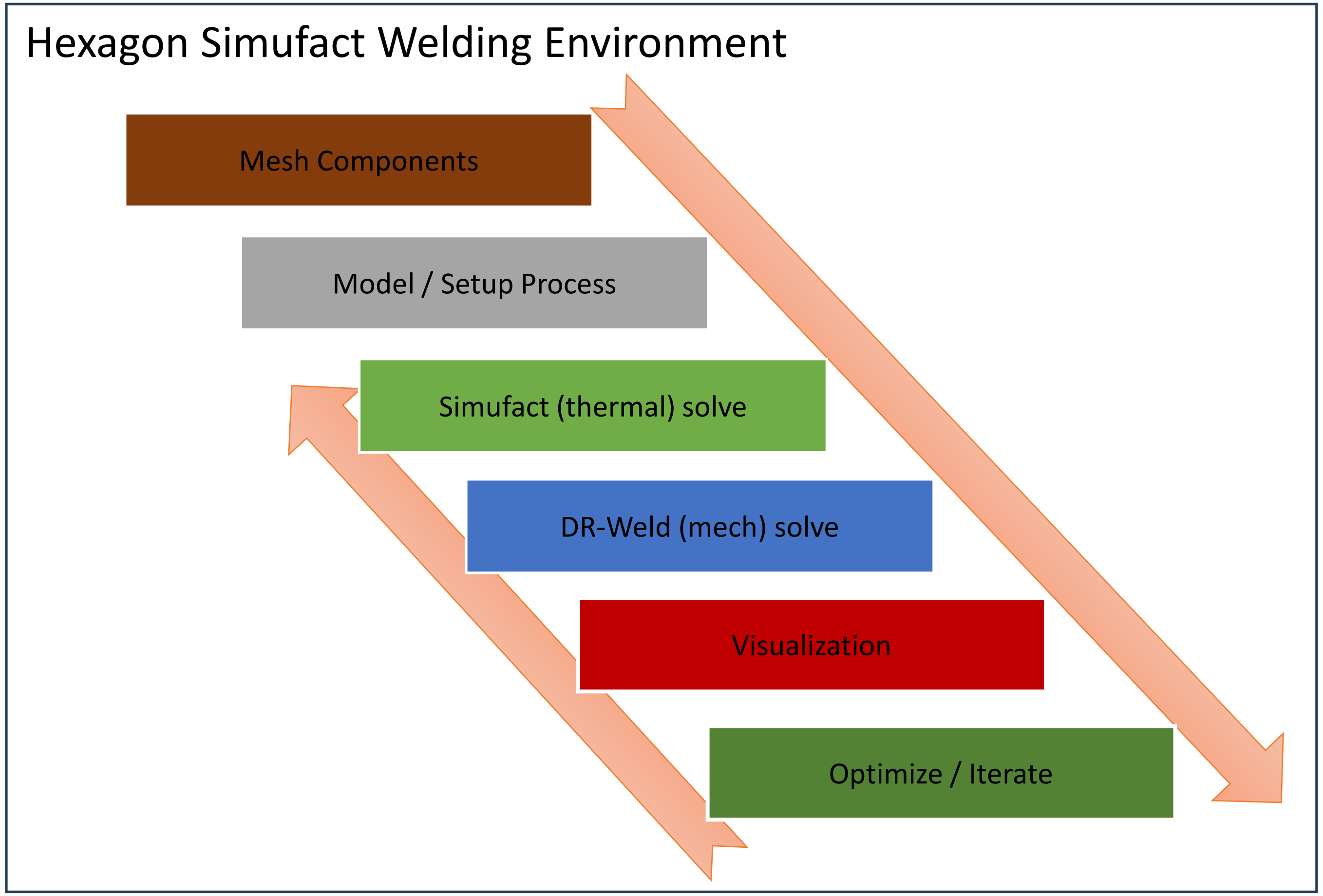
Hexagon Process Flow (GUI)

Desired Outcome

- Elegant welding process simulation solution
- Helps engineers solve the problem without spending time modeling
- Process is expedited by modeling automation
- Software interprets results and provides suggested improvements or iterates to a solution

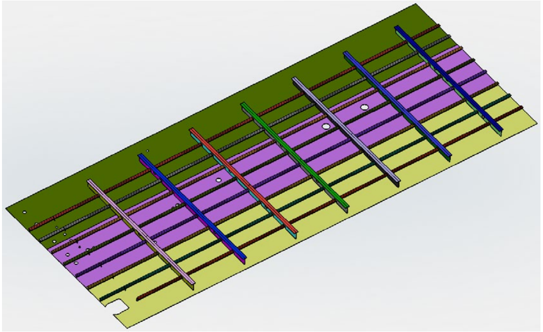
Criteria for Success

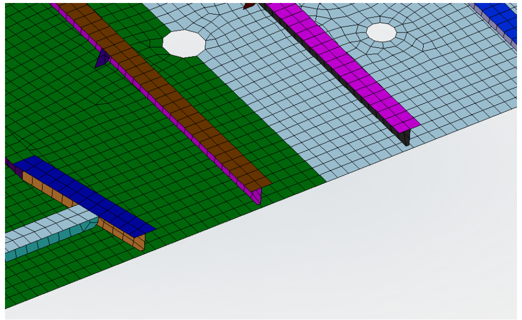
- Fast – Expeditious Solution Turnaround
- User-Friendly
- Accessible to Manufacturing Engineers
- Single-Software Installation (vs. multiple integrated tools)

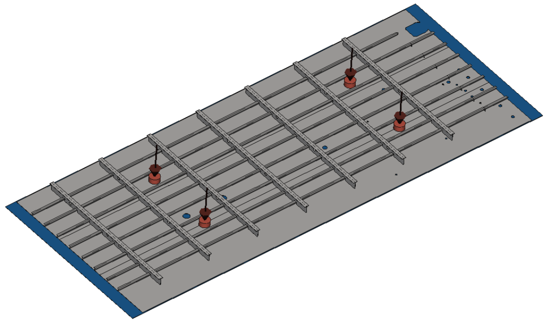


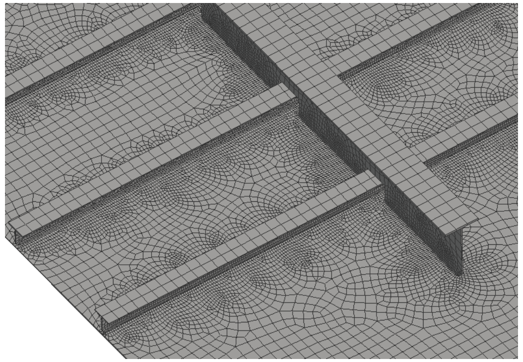
Hexagon Workflow – Shipbuilding

CAD cleanup → **Meshing (global)** → **Model setup** → **Meshing (autorefinement)**

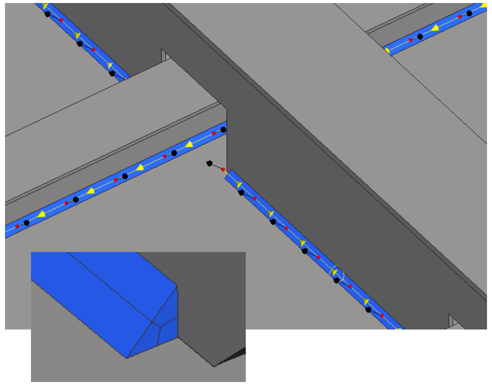


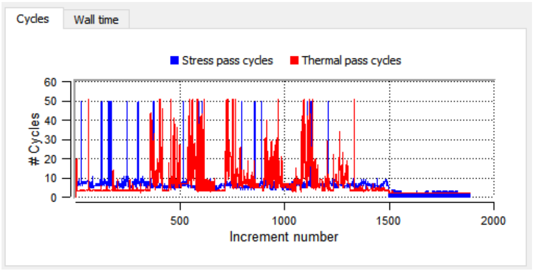


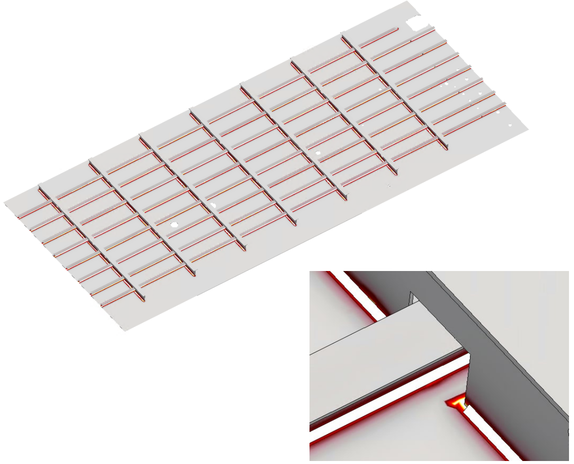




Bead Creation → **Solve** → **Post-processing** → **Model iterations**

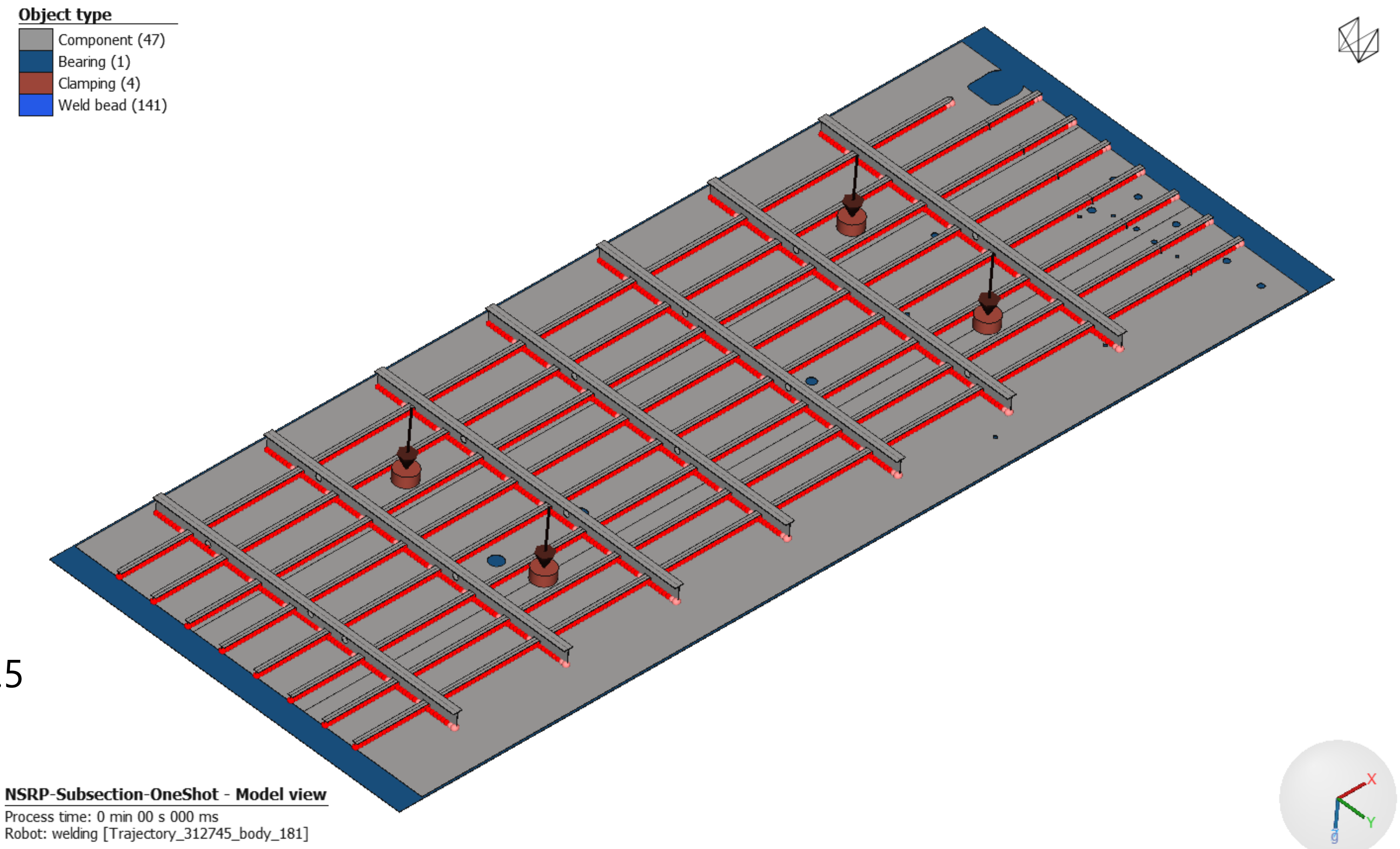






Model Summary - Subsection

- **Subsection Model**
  - Total 337 meters of weld
- **Materials:** Low alloy steel plates
- 4 clamps holding assembly, bearing representing the ground
- **Modeling time: 3-4h**
  - Meshing, model setup, etc.
- **Analysis details**
  - Thermal analysis
  - Total 450 k elements
  - Use of TC (Thermal Cycle)
  - Run with 24 cores - Threadripper 3970X (4.5 GHz)
  - Run finished in **47h**



Summary

- The DR-WELD software has great potential to support welding sequence optimization for ship structures
  - Preliminary testing has shown significant modeling acceleration and comparable accuracy to current methods
- Near-term activity will focus on completion of Shell Model integration into DR-Weld and its stiffened panel use cases
  - Generate shell element prototype GUI in Simufact
  - Test SIMUFACT automatic mesh generation tool and identify data gaps with current designs in ShipConstructor