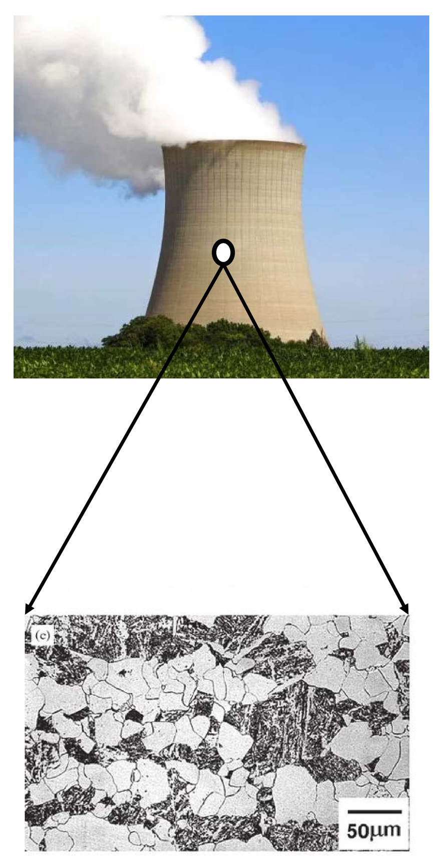
Image Based Analysis for Stress Environment in Duplex Structure Stainless Steels

Carley Suarez*, Prabaharan G. Ponnusamy*, Sanjay S. Oruganti*, Yong Yang#, Appajosula Rao^, Ramana Pidaparti*

*University of Georgia, #University of Florida, ^Nuclear Regulatory Commission, United States

Introduction

Structural materials used in several infrastructural applications including nuclear power plants in stainless steel materials. Due to their critical operating environments, these materials are subjected to stress cracking. The goal of this research was to investigate the stress environment that leads to cracks in structural materials under severe operating environments. CAD models were developed based on microstructural images and then imported to ANSYS, a finite element analysis software for stress analysis. After importing the models, material properties and boundary conditions were specified to analyze for stresses/strains. The locations of maximum stresses/strains were given an indication at which material fails. Multiple analysis with various parameters to estimate the conditions where material will fail were conducted.



Model Construction

- A 2-D slice was first created using a MatLab script written by Sanjay Sarma Oruganti
- The 2-D model was imported into ANSYS and the -3-D modeling software SpaceClaim was used to smooth edges of holes, fill in holes, and extend edges of the whole block
- The new 3-D model was then assigned properties and was ready for analysis
- Data was assigned as shown in Figure 2
- Three separate trials were conducted where the R value (ratio of the Young's Modulus [E] of the holes to the solid surface) was changed to 0.9, 0.8, and 0.7

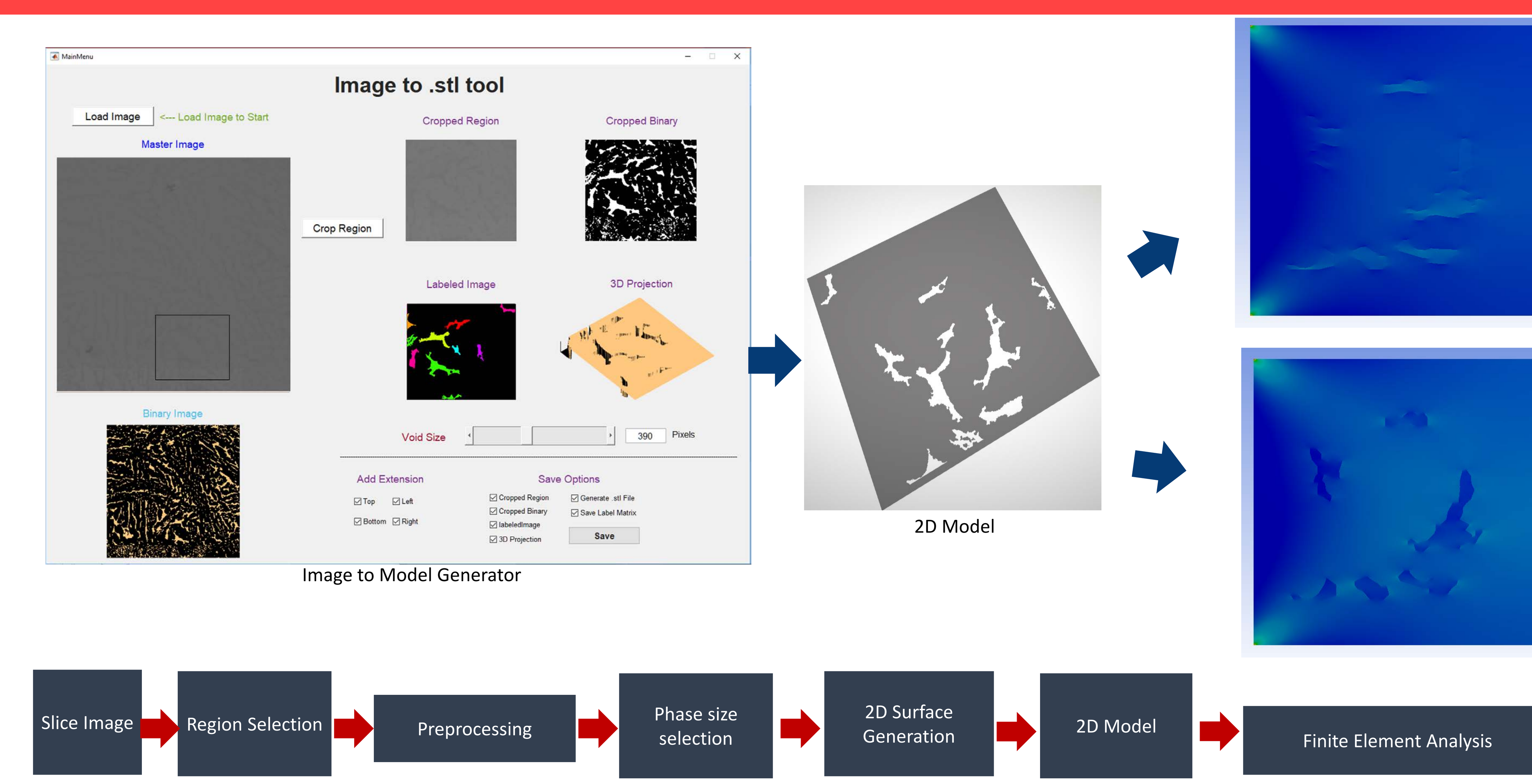


Figure 1. a schematic of the model making process

Results and Discussion

- The FEA was conducted for two FE model as outlined in Figure 2 for the Young's modulus ratio of 1.086

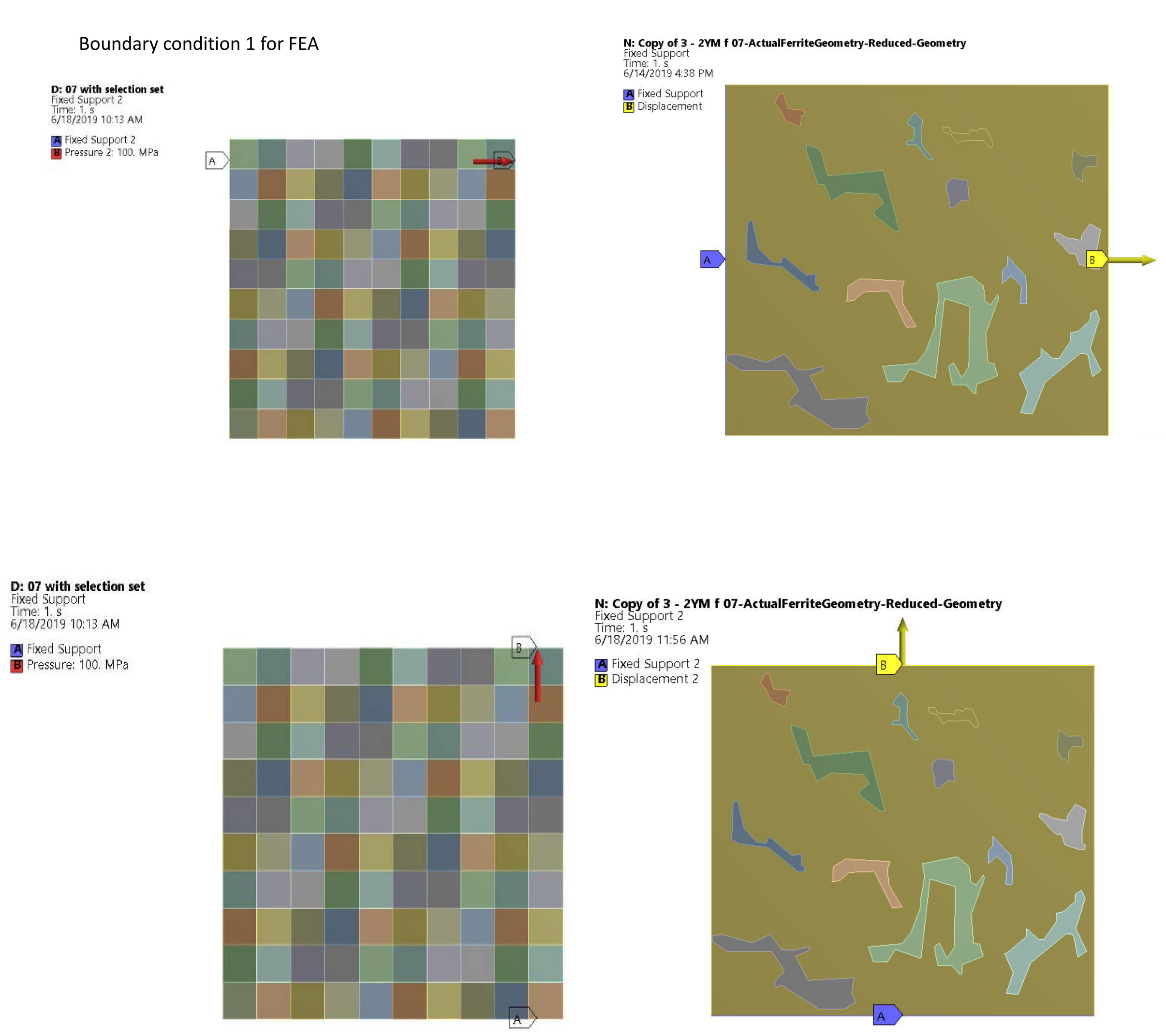


Figure 2. FEA micro structure model with homogenized and with actual geometry for two loading conditions (100 MPa Pressure applied in the +x and then +y direction)

- Figure 3 shows the stresses and deformation results on five regions of microstructure geometry

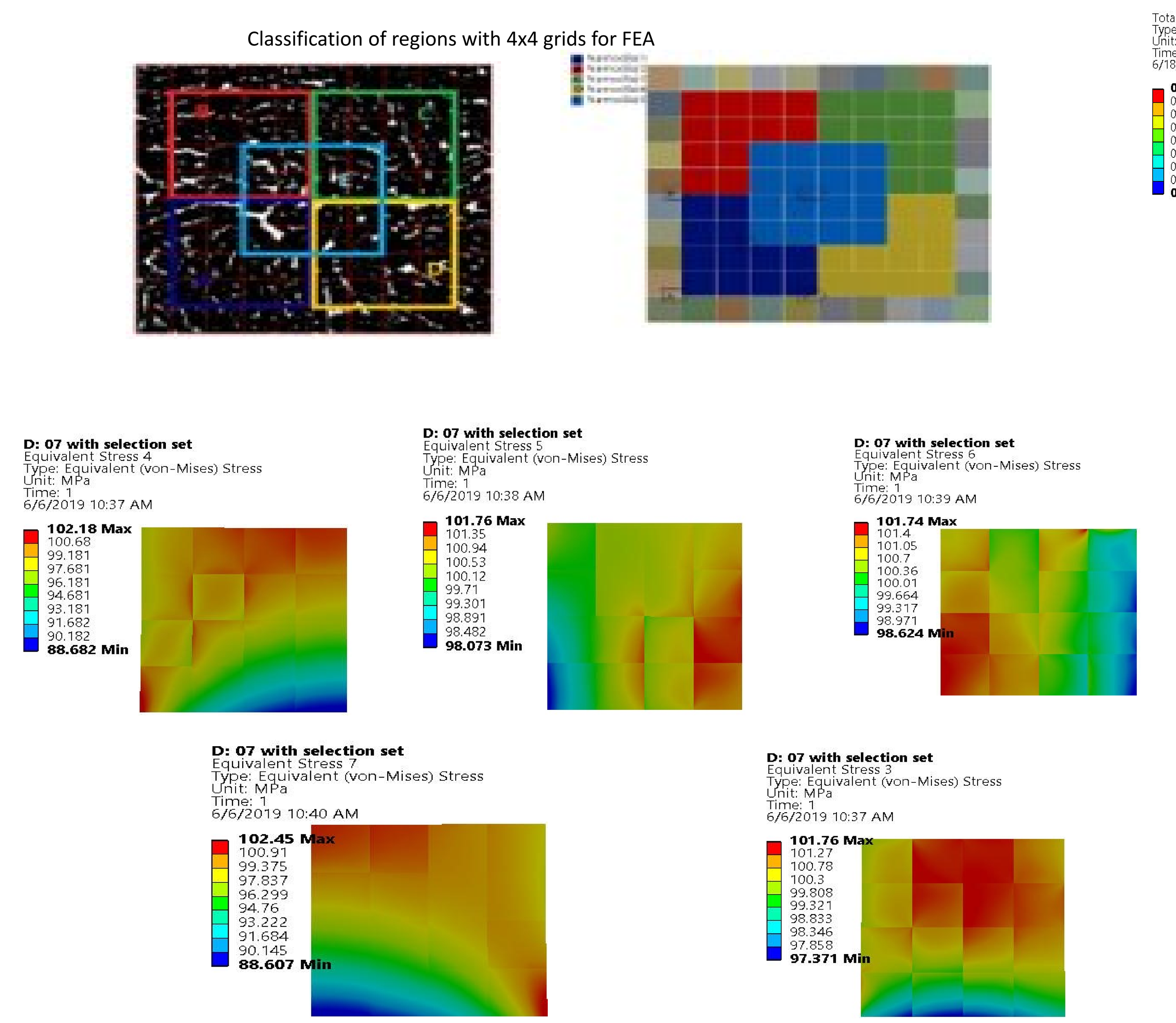


Figure 3. FEA for Homogenized microstructure condition

- Figure 4 shows the FEA results of geometry with two phase micro structure geometry

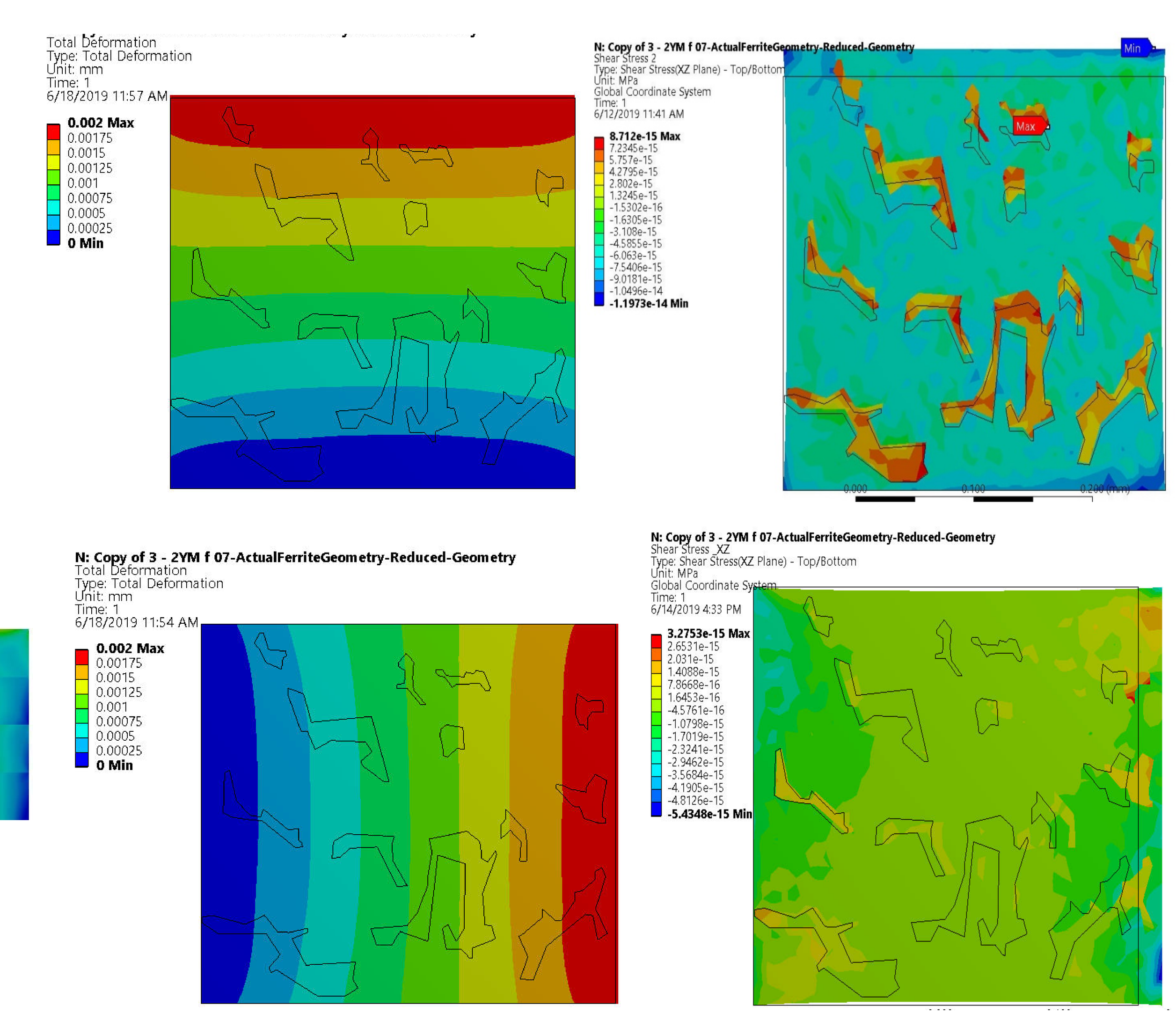


Table 1. Material Data Table

E (Pa) (Ferrite)	E (Pa) (Arsenite)	ν (Ferrite)	ν (Arsenite)	ρ ($\frac{kg}{m^3}$) (Ferrite)	ρ ($\frac{kg}{m^3}$) (Arsenite)
1.84e11	2e11	0.29	0.29	8050	8050

Summary

- Finite Element Analysis has been used to predict stresses related to its microstructure
- It was found that change in Young's modulus ratio (R), affected the peak stress and strain values
- It was observed that lessening the Young's Modulus of ferrite phase makes the material more brittle
- It was found from the FEA that the dual phase material yields in brittle material phase

Future Work

- Develop an automated model generator
- Explore different stress boundary conditions further analysis to relate effect of microstructure

Acknowledgement: This research reported was supported by the US Nuclear Regulatory Commission