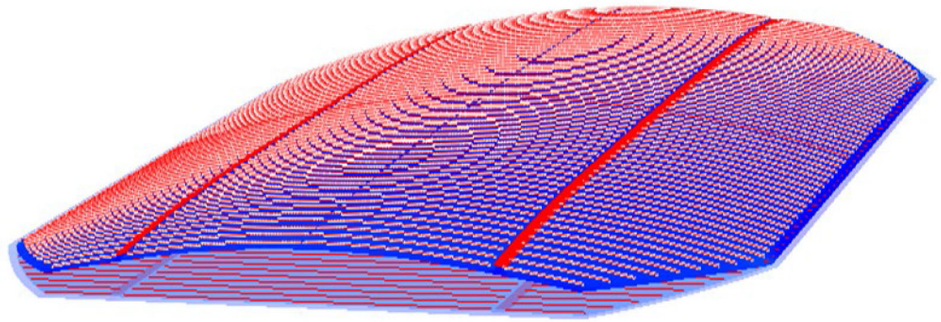
Journal of Zhejiang University-SCIENCE A

Cite this as: Chih-Hung Chen, Hsuan-Teh Hu, Fu-Ming Lin, Hsin-Hsin Hsieh, 2017. Residual stress analysis and bow simulation of crystalline silicon solar cells. *Journal of Zhejiang University-SCIENCE A (Applied Physics & Engineering)*, 18(1):49-58.
http://dx.doi.org/10.1631/jzus.A1500279

Residual stress analysis and bow simulation of crystalline silicon solar cells

Key words:

Bow, Solar cell, Silicon solar cell, Finite element analysis, Residual stress



Bow Simulation

Nanoindentation Test

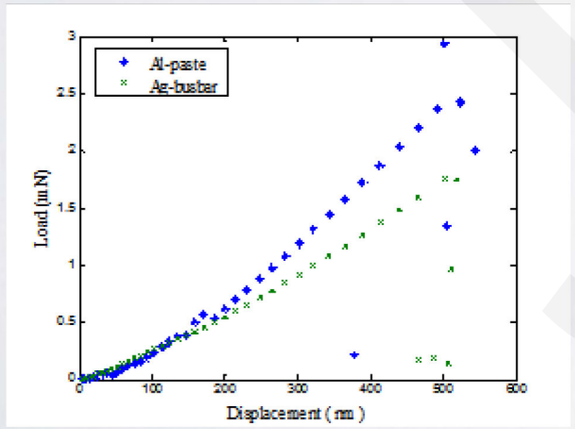
- The material properties of Al paste and Ag busbar were measured.

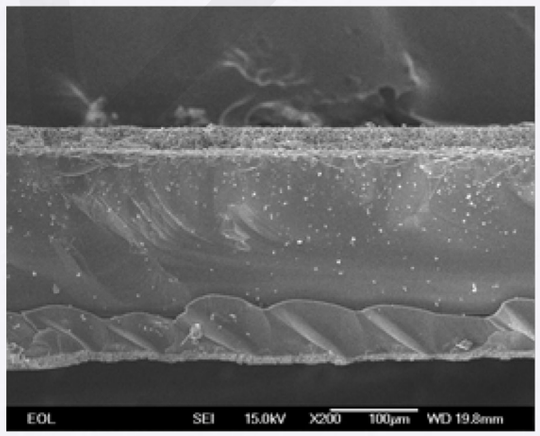
SEM

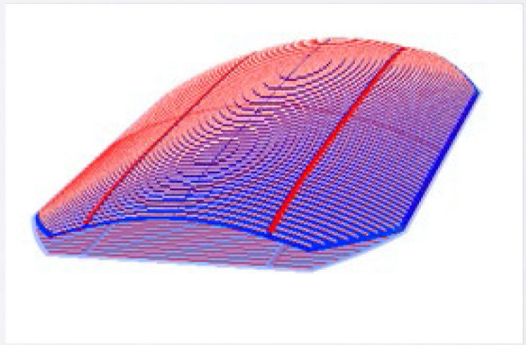
- The thickness of Si wafer, Ag busbar, and Al paste were measured.

Finite element analysis

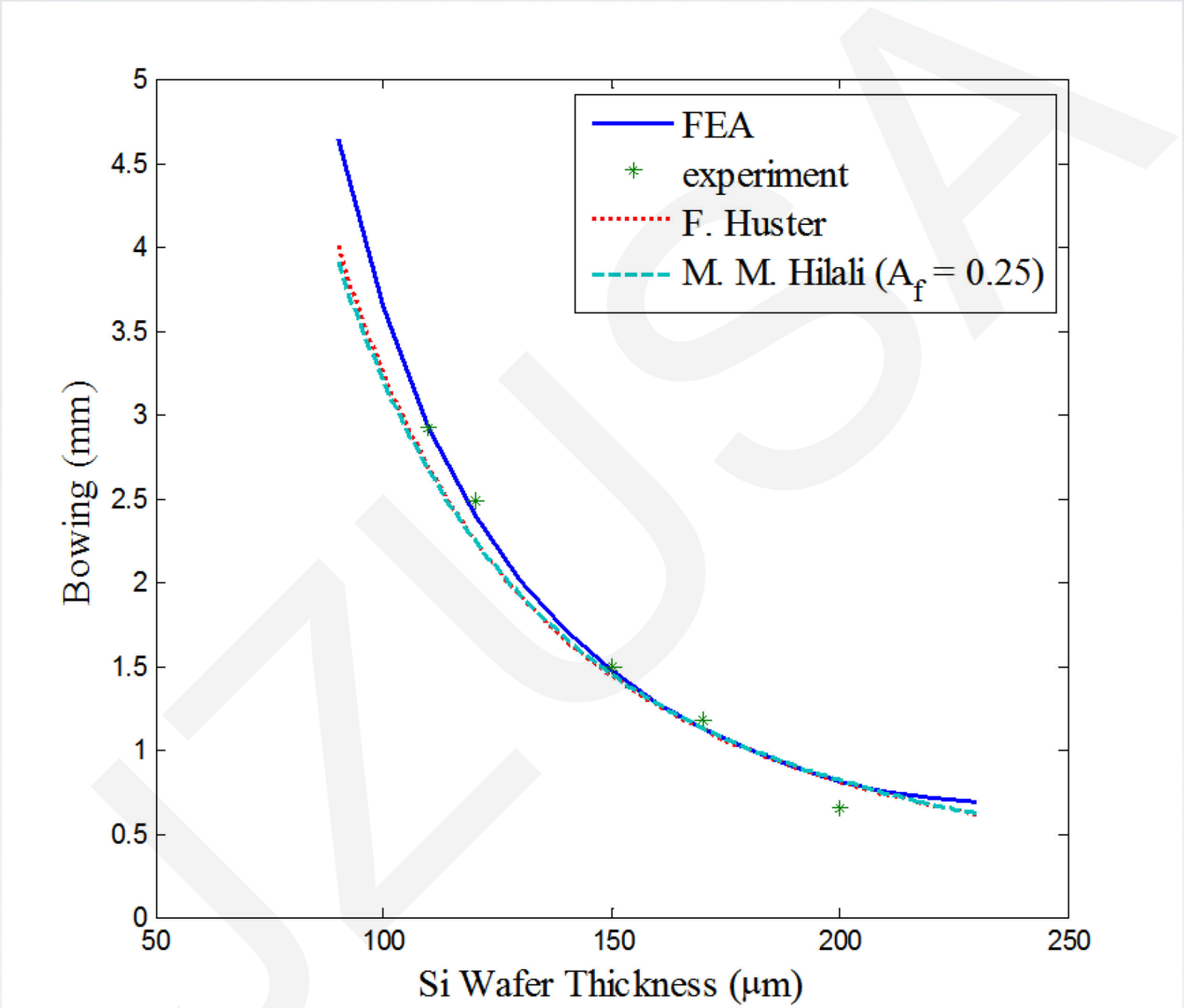
- FEA simulation adopted by ABAQUS





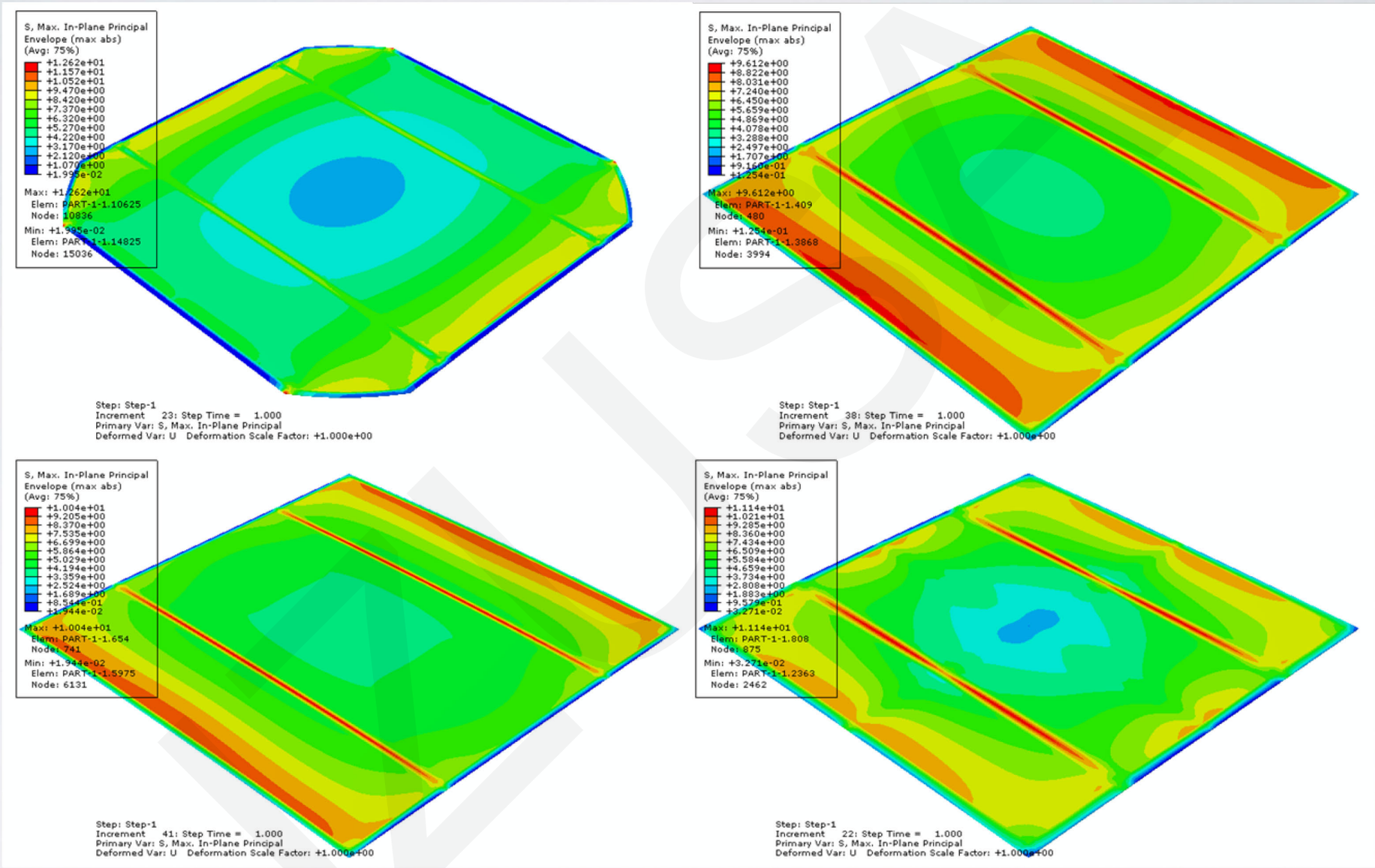


FEA results



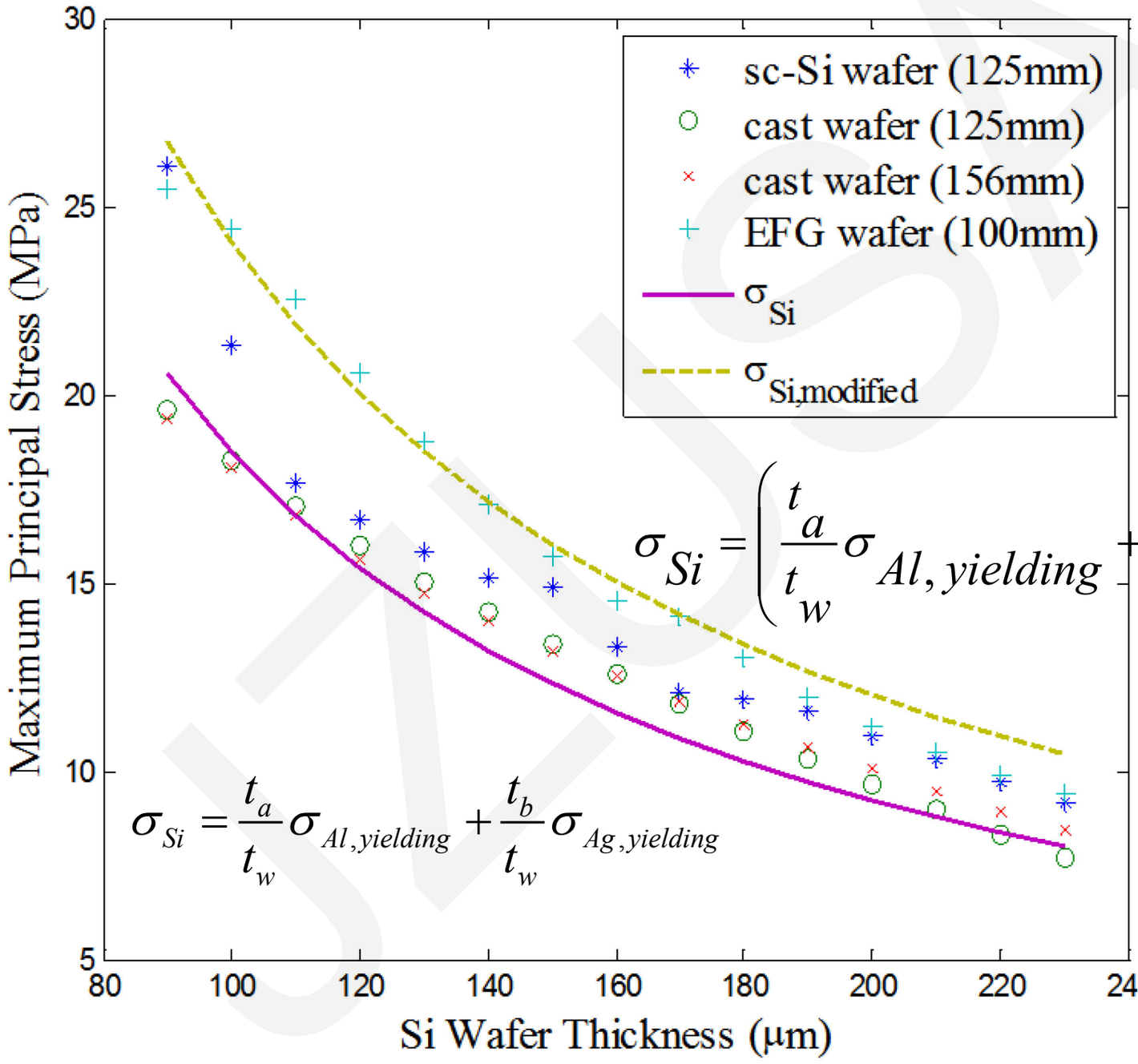
The FEA results are in better agreement with the experimental data than are the bowing calculations.

Residual Stress Analysis



Typical maximum in-plane principal stress distribution for: (a) sc-Si wafer, 125 mm × 125 mm, (b) cast wafer, 125 mm × 125 mm, (c) cast wafer, 156 mm × 156 mm, and (d) EFG wafer, 100 mm × 100 mm.

Residual Stress Calculation



$$\sigma_{Si} = \frac{t_a}{t_w}\sigma_{Al,yielding} + \frac{t_b}{t_w}\sigma_{Ag,yielding}$$

$$\sigma_{Si} = \left(\frac{t_a}{t_w}\sigma_{Al,yielding} + \frac{t_b}{t_w}\sigma_{Ag,yielding}\right) \times M_f$$

$$M_f = 1.3$$

Conclusion

- A systematic approach that uses the material properties obtained using a nanoindentation test, the thicknesses of the SP contacts measured using SEM, and cell bowing simulated by FEA software was presented.
- It provides accurate bowing results that agree well with experiments and the complete in-plane principal stress distribution for various types of Si wafer.
- The maximum residual stress induced by the firing process of an Si wafer can be obtained using FEA simulation or suggested equations which are simple and can save much computation time.
- Because Ag busbars and fingers are considered in the FEA model, the metallized pattern for solar cells can be analyzed using FEA simulation. In this paper, plastic strain is proposed to estimate the possibility of breakage in the Ag busbar and finger.