

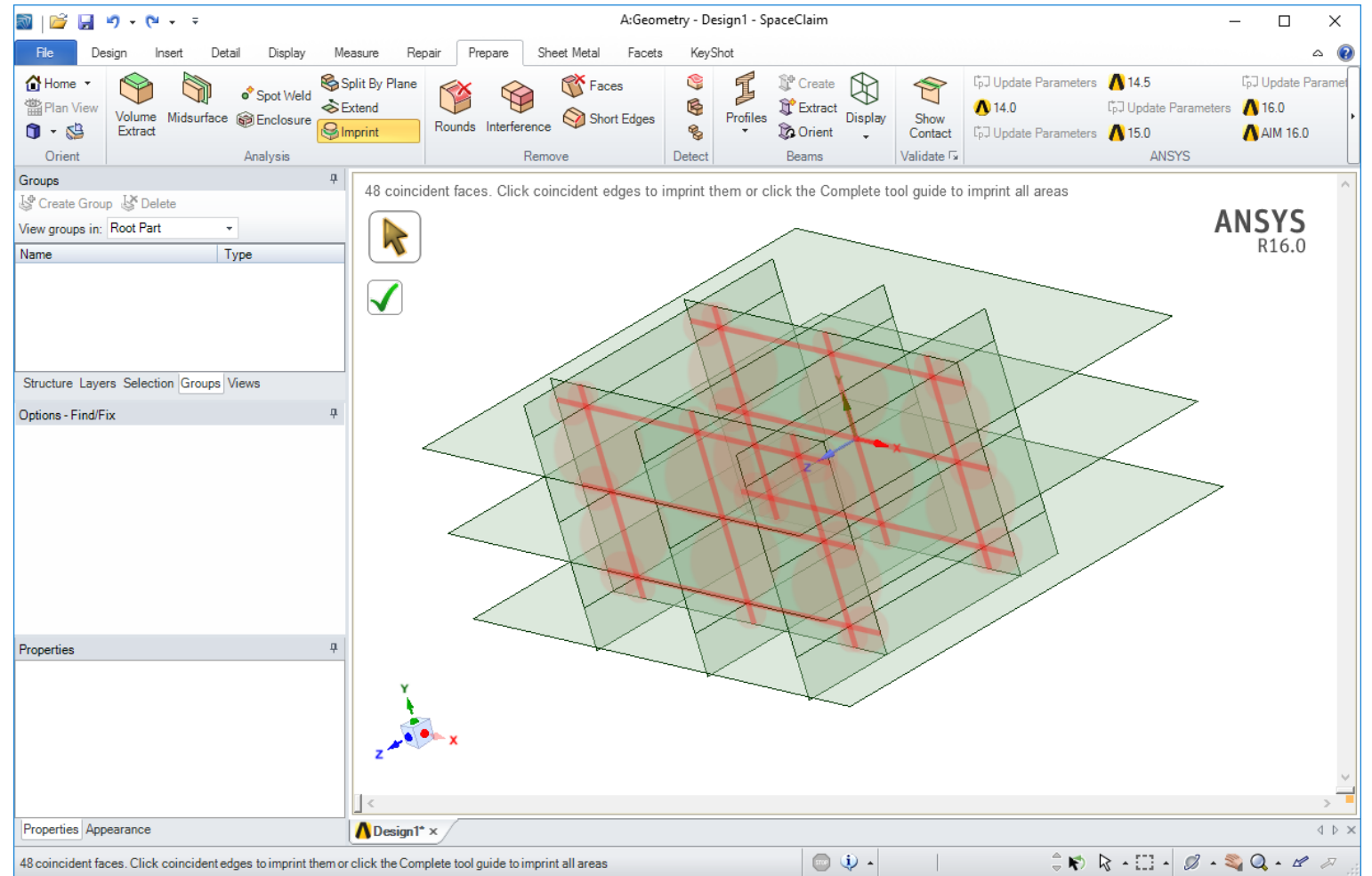
Connecting Shells Elements in Surface Models

ANSYS SpaceClaim and
Mechanical

Ted Harris, PADT, Inc.

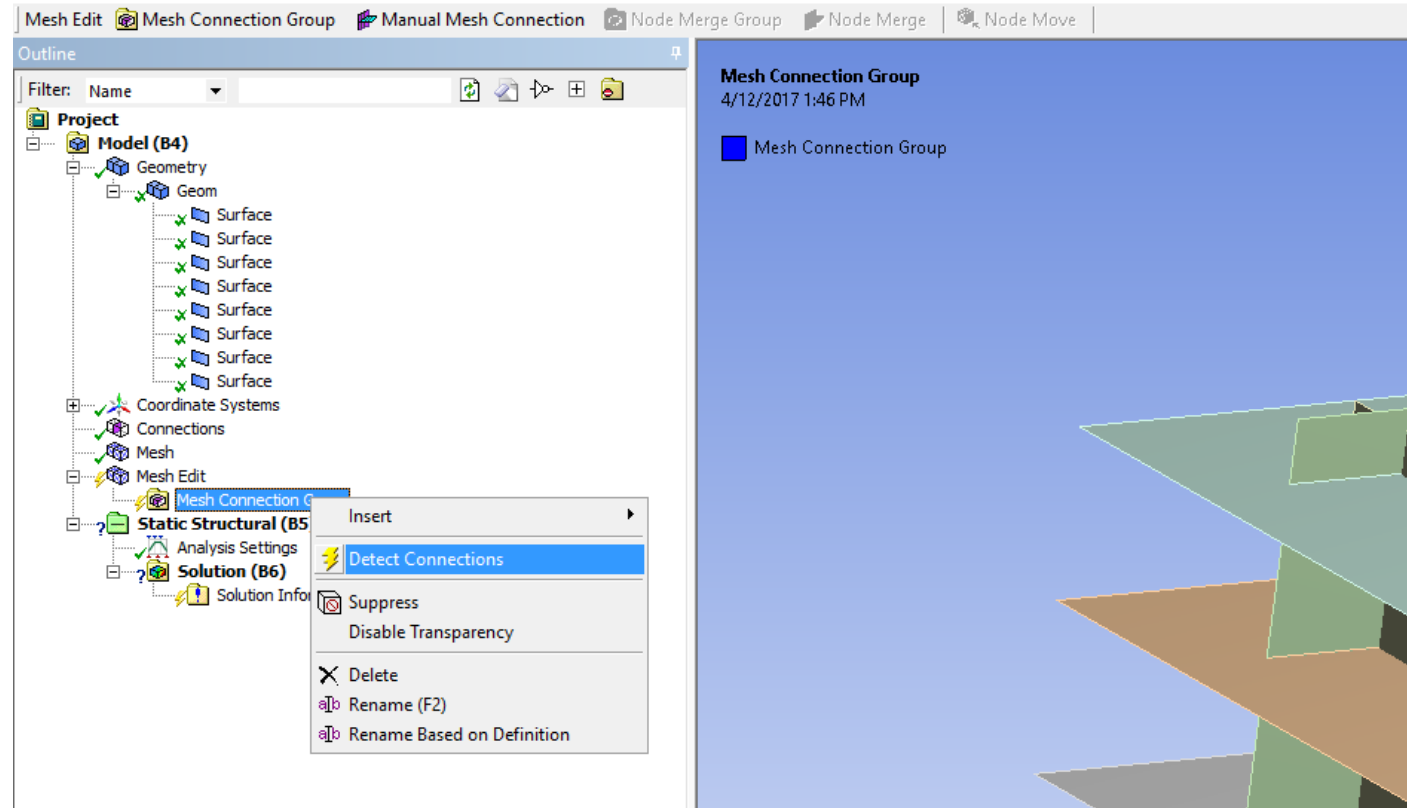
SpaceClaim Is Your Friend

With just a couple of clicks in SpaceClaim, can create intersections where shell edges touch:



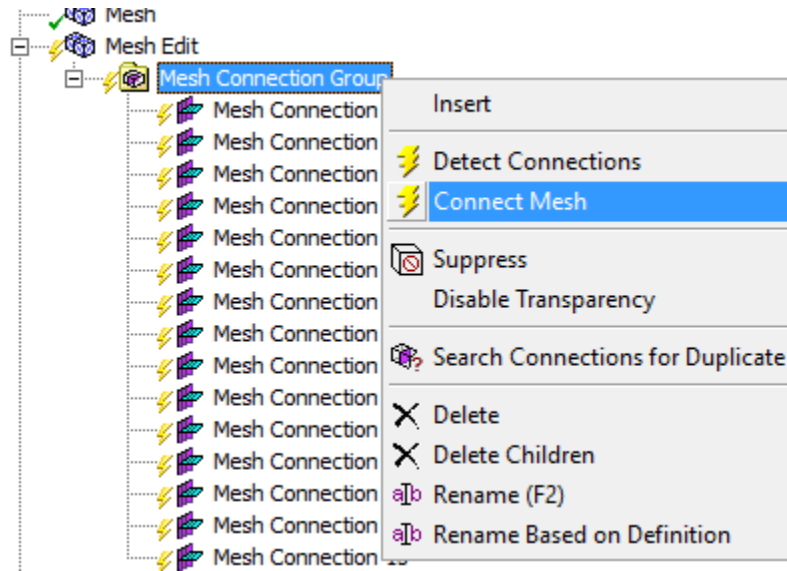
Mesh Connections in Mechanical

- Mesh Connections added via Mesh Edit > Mesh Connections Group > Detect Connections.



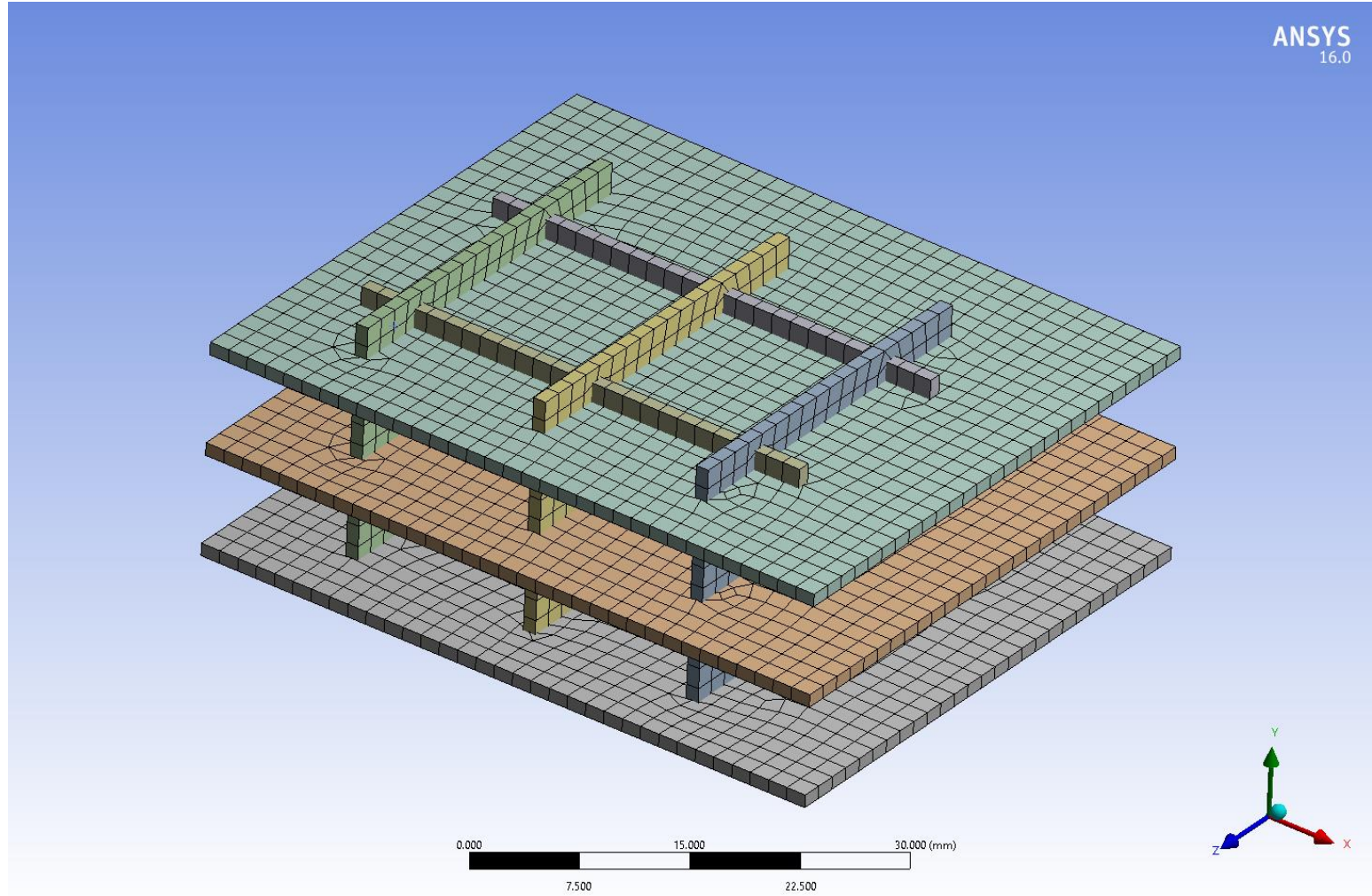
Mesh Connections in Mechanical

- Right click again > Connect Mesh (this is after a first mesh has been created)



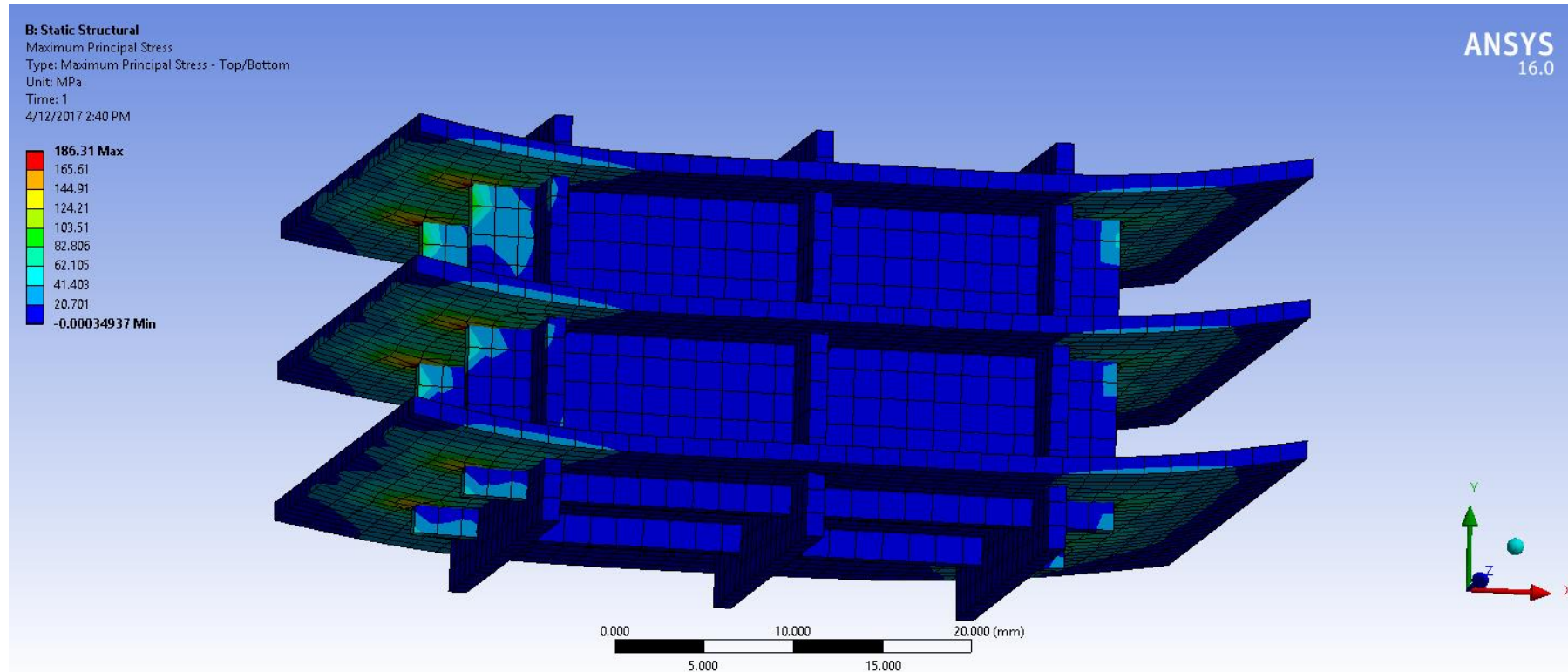
Mesh Connections in Mechanical

- Resulting connected shell mesh



Mesh Connections in Mechanical

- Stresses due to downward force applied to 3 top edges, all 6 side edges constrained as simply supported (deformation exaggerated).



Summary

- Shell structures can be connected using Mesh Connections in ANSYS Mechanical
- A key step is to use ANSYS SpaceClaim to Imprint all the intersections
 - Imprint is in the Repair tab in SpaceClaim