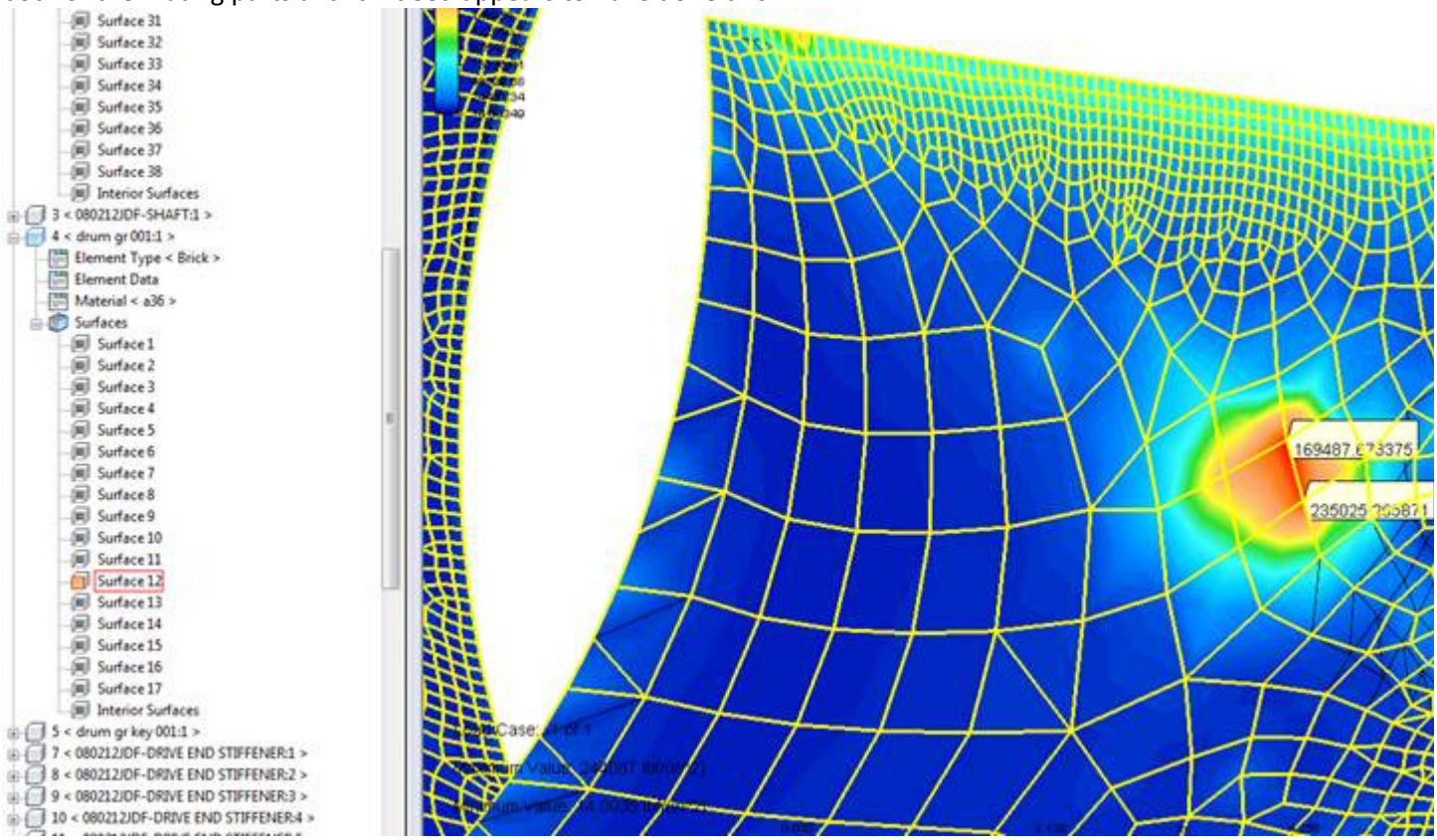
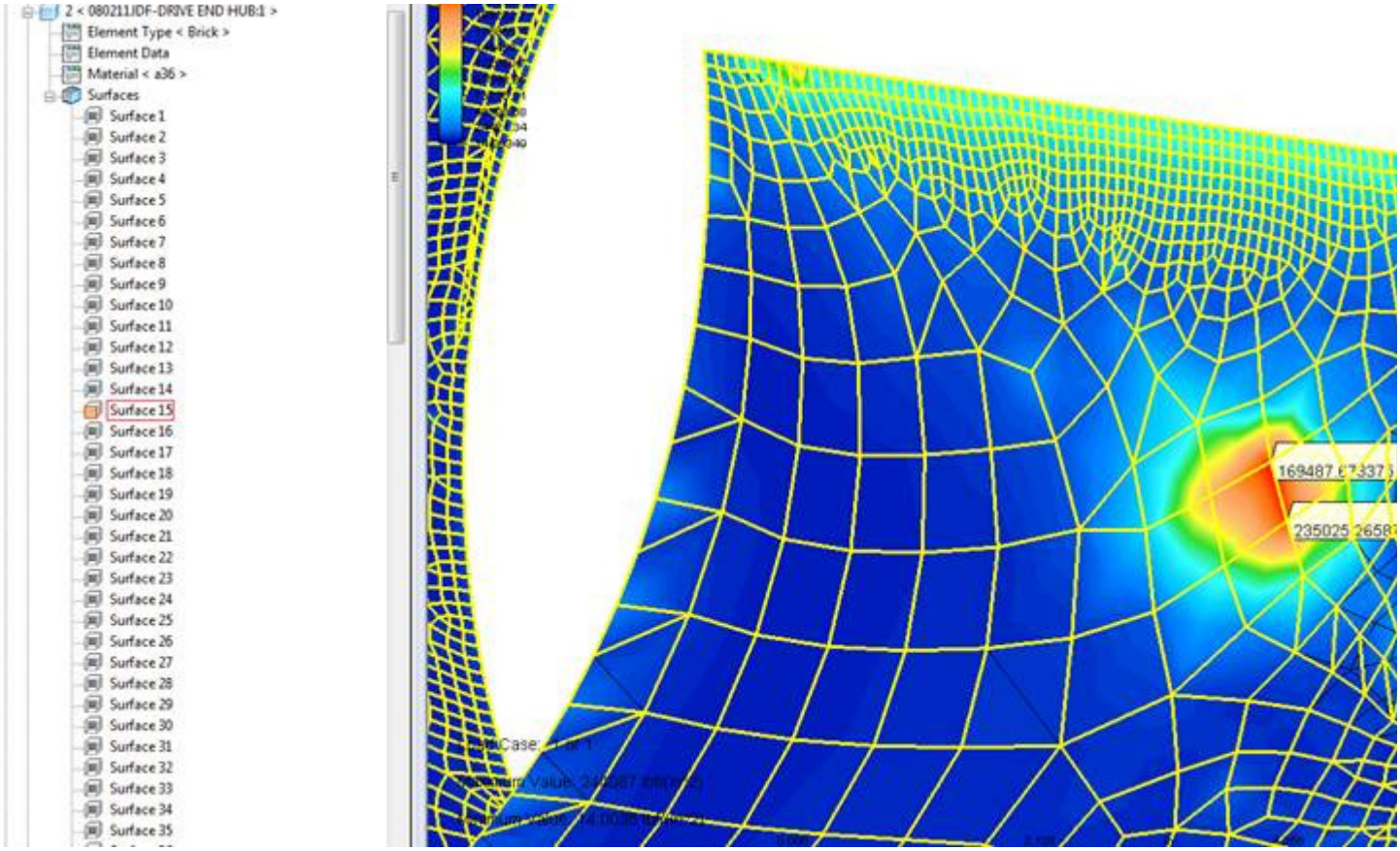
Jay May

From: Jay May
Sent: Thursday, March 14, 2013 9:10 AM
To: 'autodesk.prod|50600ee3|b45ac266-9072-4ded-969c-aa5a04a51f6a@reply01.lithium.com'
Subject: Re: Shrink Fit in Simulation Mechanical

Thank you for the response.
The analysis type is Static Stress with Linear Materials.
The meshes match in a large portion of the surface area. There are unmatched area toward the edge with the finer mesh. The earlier images were of the deflected shape. The following images shows the un-deflected surfaces on top of each other. One is highlighted (yellow) the other can be seen as black where it is not matching. You can also see here that the two pairs of nodes appear aligned to each other. The nodes do share common XYZ position as shown in the earlier results image. The mesh lines appear to align better before I add refinement points. Should I be adding refinement points to both parts. I was thinking that their sphere or radius of influence would cause a matching effect on both of the mating parts and it indeed appears to have done this.



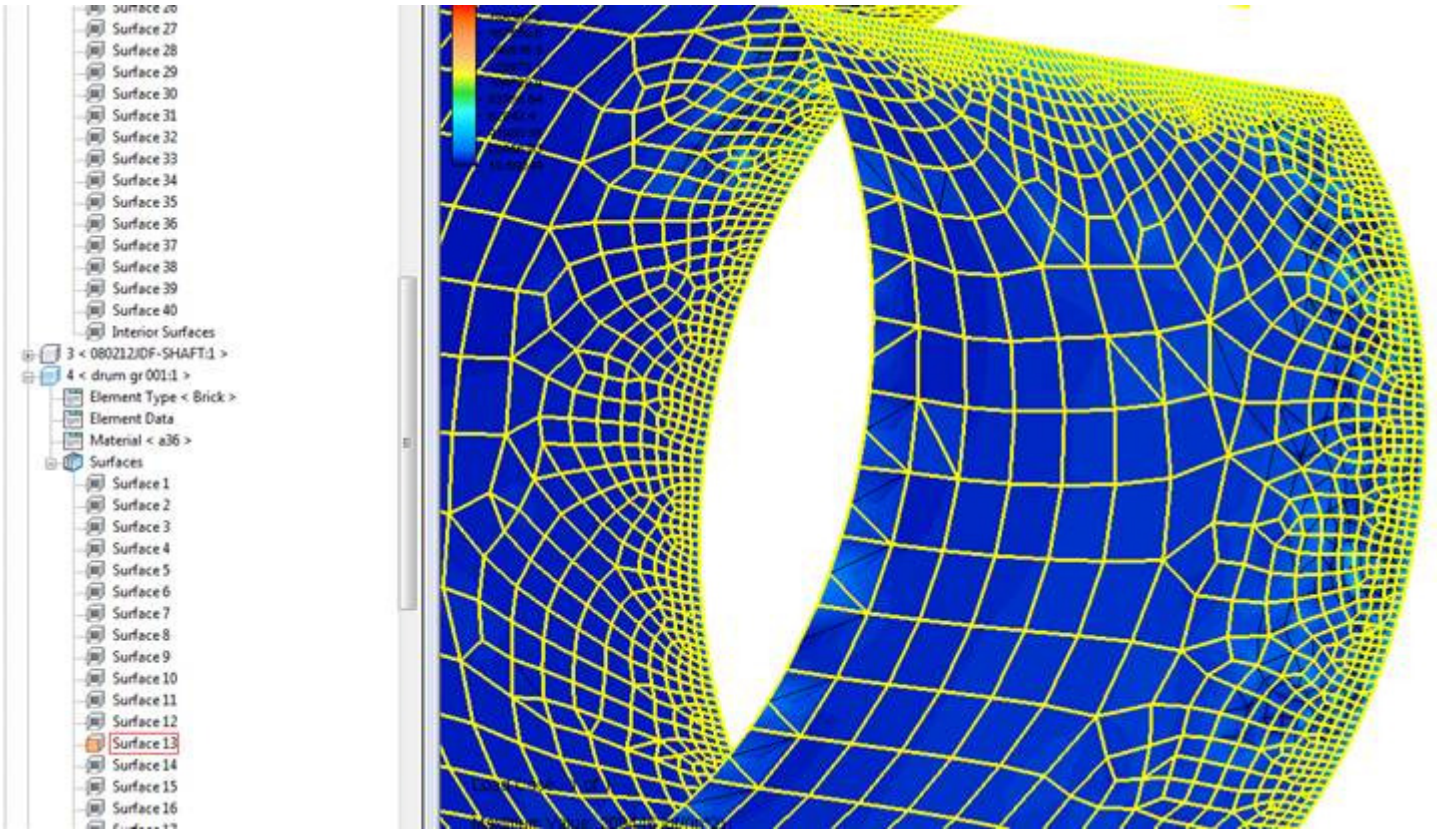
In the following image, the opposing surface is highlighted.



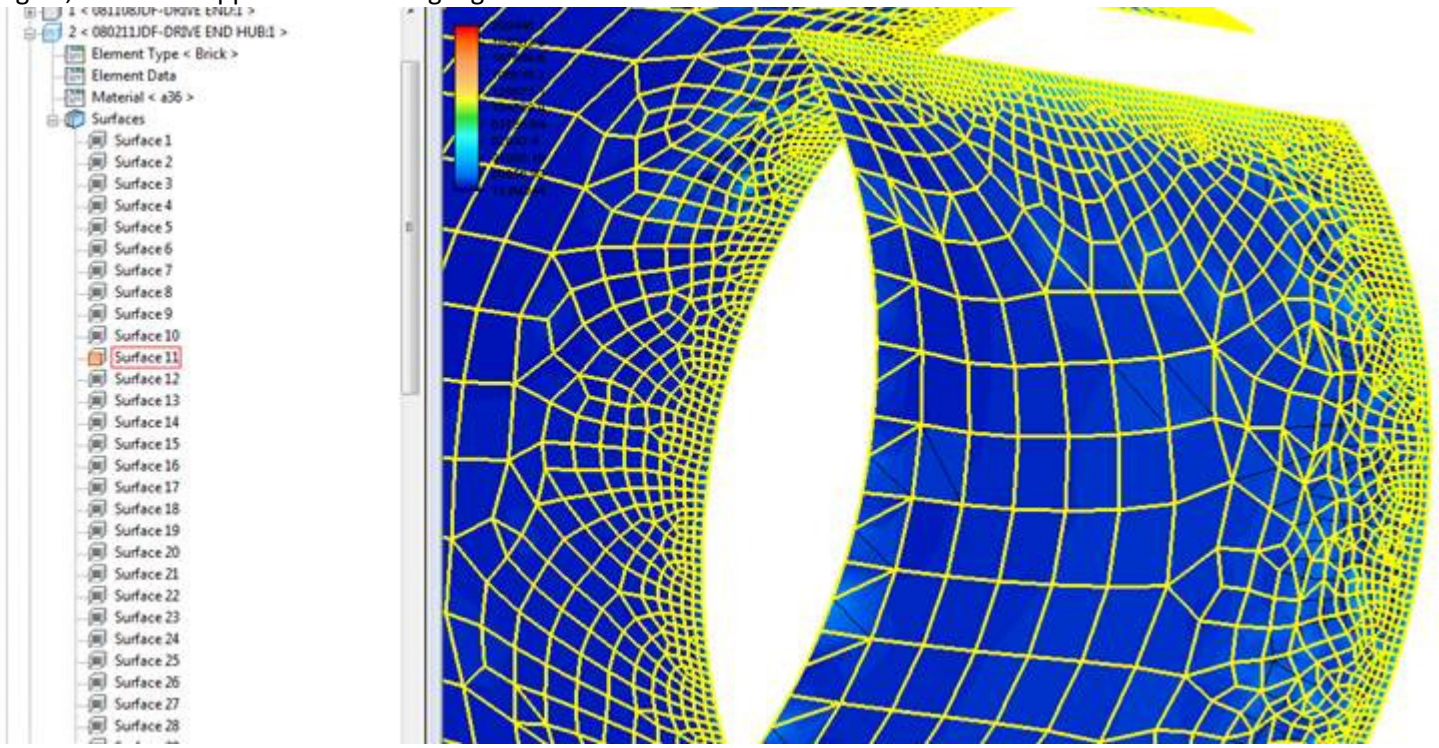
I had run a couple different versions of this type of condition and did not obtain such hot spots. I have been trying different surface contact types. On earlier models I started with “Surface Contact”, then “Shrink Fit/Sliding” and then “Shrink Fit/No sliding”. In the earlier models I was getting results that appeared as one might expect. I had redesigned the part(s), re-meshed and so-forth and this sort of hot spot result began to appear.

The following images are from an earlier version of the model. They show a similar mesh mis-match in the refined area, but this model did not exhibit such a hot spot in the stress distribution. I had set the contacts to the three types in different scenarios and they each came back without this type of hot spot.

What I am attempting to model is the interaction of a steel gear hub with a press fit (shrink fit) onto a matching steel hub. A key is used to transmit torque loading. The effect of pressure on the keyway edge appears to be my current area of concern. Other critical areas of concern of the hub design have been redesigned to develop an acceptable level of stress. The actual loading on the key is subject to the potential slip between these two surfaces. In reality the fit may be sufficient to transmit the torque, but the level of actual fit can be a bit unknown and therefor the share of load taken by the key is somewhat unknown also. I would like to vary the fit and friction factor to study their effect on the stress level at the keyway edge.



Again, this is the opposite surface highlighted.



While we are discussing this model, I am looking for advise on element types, ie, “bricks and tets” versus “all tets”. The solid meshing under the “bricks and tets” option takes so much longer than the “all tets” option. Which would be most appropriate when modeling solid steel shafting fitted to other soild components as well as other types of interconnected somewhat thick steel sections?

I have modeled the components in Inventor with zero gap between the surfaces. What steps can I take to better assure that the surface mesh matches more closely?

Regards,
Jay May
Engineer
Engineering | Morgan Engineering
330-823-6130 x268
1049 South Mahoning Avenue
Alliance, OH 44601