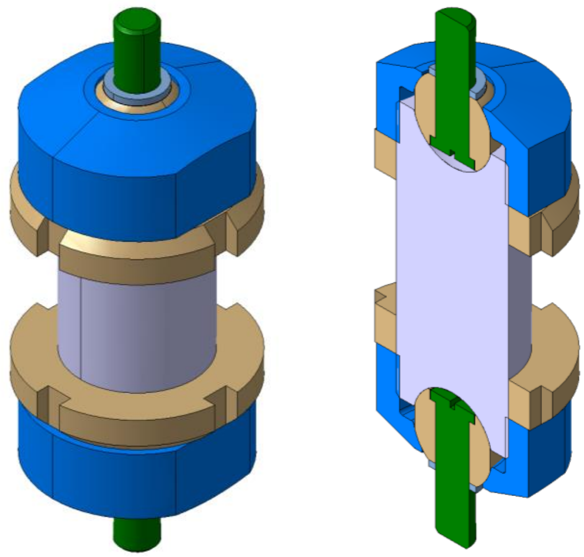
Determining Stiffness Of UAP ‘Zero Backlash’ Joints

Matthew Capstick - matthew.john.capstick@cern.ch

2020-03-31

1 Introduction

To determine the suitability of the UAP ‘Zero Backlash’ Joints for the CLIC TBMs, the stiffness of these joints must be understood and quantified.

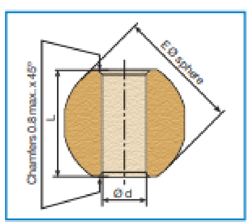


2 Joint Construction

The joint uses two sintered bronze self-lubricating spherical (SELFOIL) bearings. The bearings are installed between the central aluminium (EN AW-6060 (T6)) core and a high strength steel (34CrNiMo6 (1.6582)) cap. Machined M6 bolts provide the fixing points at each end with bearing washers to allow the bearing to roll once installed.

2.1 Sintered Bronze SELFOIL Bearing

The bearing component within the joint; consisting of a sintered bronze sphere impregnated with lubrication oil. The run-out tolerance on the spherical surface is IT9, equivalent to between approximately 40µm and 50µm for the given sizes:



Designation (reference for orders)

A spherical bronze bearing of 10 mm inner diameter, 22 mm sphere diameter and 16 mm length, is designated as:
SELFOIL® Bearing C-10-22-16
(letter C indicates spherical bearing)

Type C Spherical

d = Ø inner Tolerance H7	E = Ø sphere (mm) Tolerance ±0.05	L = Lengths (mm) Tolerance ±0.15	Quantity per bag
4	10	8	25
5	12	9	25
6	14	11	25
7	16	12	25
8	18	13	25
9	20	14,5	25
10	22	16	25
12	23	16	25

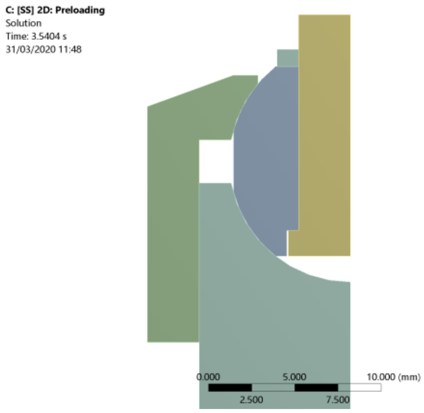
Run-out: IT-9

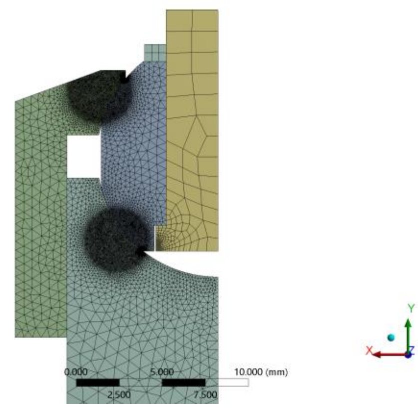
More information: https://www.selfoil.com/
https://www.ames-sintering.com/self-lubricating-bearings/

3 Modelling Methods

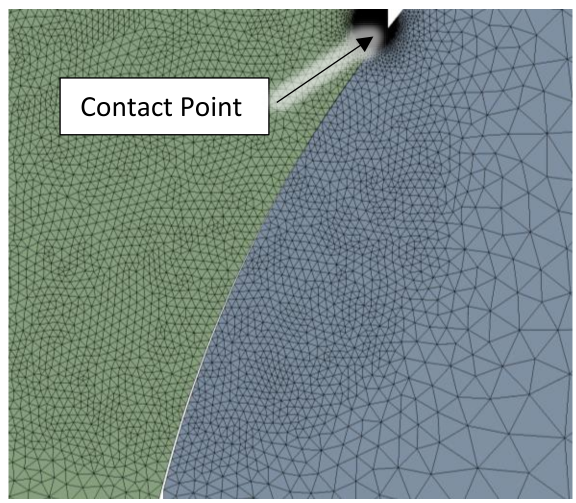
3.1 Geometry

Analysis can be done using a 2D-Axisymetric geometry (below left), this greatly reduces the size of the model, and allows the mesh around the contact points to be refined (below right). This fine mesh is required to determine the contact stresses on a fine scale, and to accurately model the contact.





The geometry is constructed so the diameters of the cups (within both the cap and main body) are equal to the diameter of the spherical bearing plus a tolerance value (here 50µm, based on the run-out tolerance of the bearing surface, equivalent to a H9 hole tolerance [might be a bit tight, any thoughts on this?]).



The contacting bodies (ball and socket) are then assembled so that there is a single point contact between them (see image right; the contact point is at the top right, where there is significant mesh refinement), and the clearance between the parts increases further away from the contact point.

3.2 Modelling Contact

The contact settings I have used are as follows:

Scope	
Scoping Method	Geometry Selection
Contact	1 Edge
Target	2 Edges
Contact Bodies	BALL
Target Bodies	CAP
Shell Thickness Effect	No
Protected	No
Definition	
Type	Frictionless
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	9.e-002 mm
Suppressed	No

Advanced	
Formulation	Augmented Lagrange
Small Sliding	Off
Detection Method	On Gauss Point
Penetration Tolerance	Value
Penetration Tolerance Value	1.e-004 mm
Normal Stiffness	Program Controlled
Update Stiffness	Each Iteration, Aggressive
Stabilization Damping Factor	0.
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Adjust to Touch
Contact Geometry Correction	None
Target Geometry Correction	None

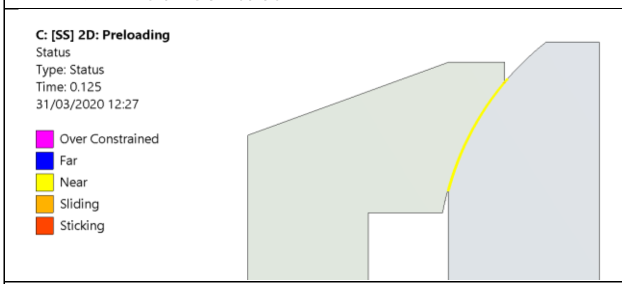
To summarise:

- The surfaces are assumed to be frictionless (due to the self-lubricating bearings, and the assumption that lower friction will lead to a lower stiffness joint, so this is a conservative assumption).
- The penetration tolerance is very low (0.1µm).
- The contact stiffness is updated every iteration, to correctly model the changing contact areas.
- The surfaces are adjusted to touch at the single point contact (to compensate for any differences between the geometry and the meshed bodies, and ensure that the contact is detected at the start of the analysis).

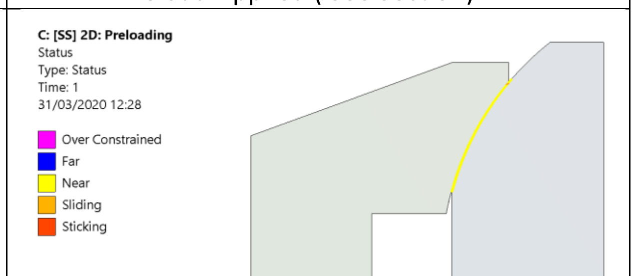
The changing contact states throughout the analysis can be seen below. Note the change from near contact to sliding contact (yellow to orange) propagates down the bearing surface as the load increases and the two surfaces deform. This is as expected.

Also note that the initial point contact becomes a small line contact once the preload is applied. I will discuss this in the next section.

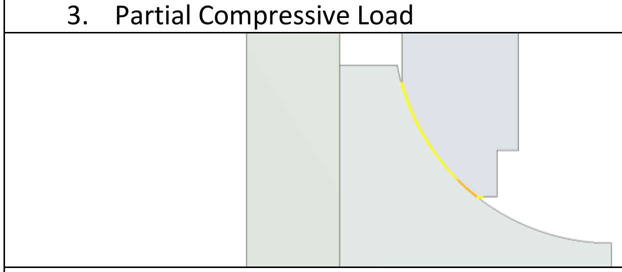
1. Initial Contact



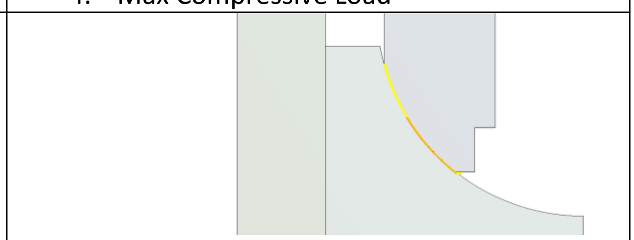
2. Preload Applied (See Section)



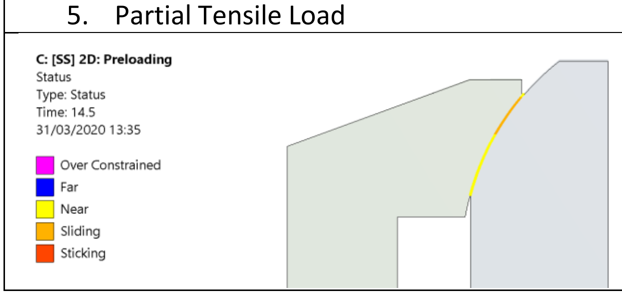
3. Partial Compressive Load



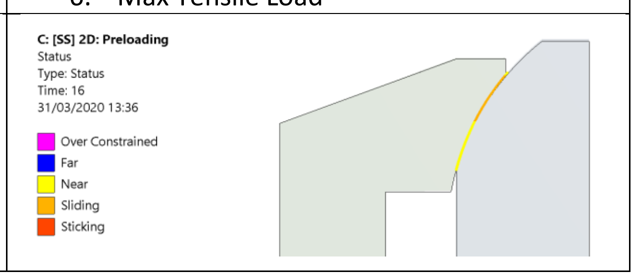
4. Max Compressive Load



5. Partial Tensile Load

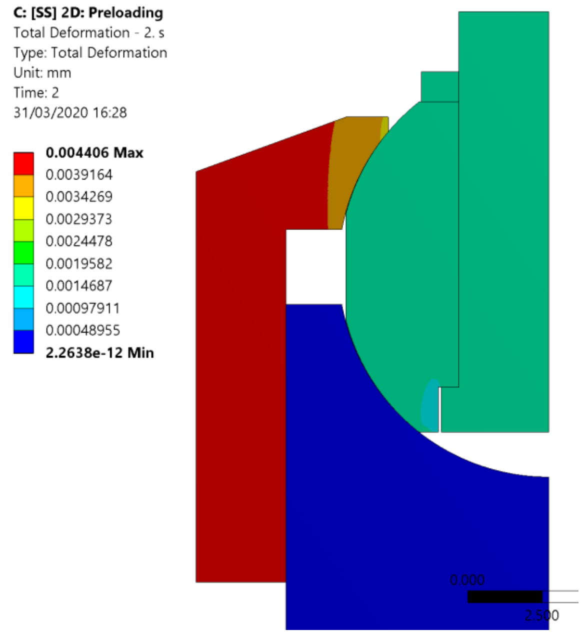


6. Max Tensile Load



3.3 Modelling Preload

As previously mentioned; the geometry is initially modelled so that there is a point contact between the bearing and both the sockets. To accurately model the physical assembly a degree of preload must be applied; clamping the cup down onto the bearing. In reality this is achieved by tightening the cup onto the bearing by means of an external thread on the aluminium core. Once the required preload has been applied, the position of the cap is fixed by tightening a locknut. Within the model this is achieved by creating both a frictionless sliding contact and a bonded contact between the core and the cap (where the external thread would exist in reality) and then using an ADPL command to suppress the bonded contact so that the cap can slide relative to the core when a preload is applied to the top of the cap. Once the preload is applied, the bonded contact is reactivated and the compressive load removed. This deforms the geometry of the contact regions as would be expected by the application of a preload, and allows the remainder of the analysis to be subsequently carried out without exporting a deformed mesh.



The deformation of the assembly after the preload step is show to the right. Note the large displacement of the cap relative to the core due to the sliding contact.

The amount of preload is significant as it determines the initial contact areas and therefore the initial contact stiffnesses.

Here I have applied 250N of preload which produced deformation and change in contact area shown above. However, this basically a random assumption with no data behind it. Do you have a feel for the appropriate preload for this sort of application?

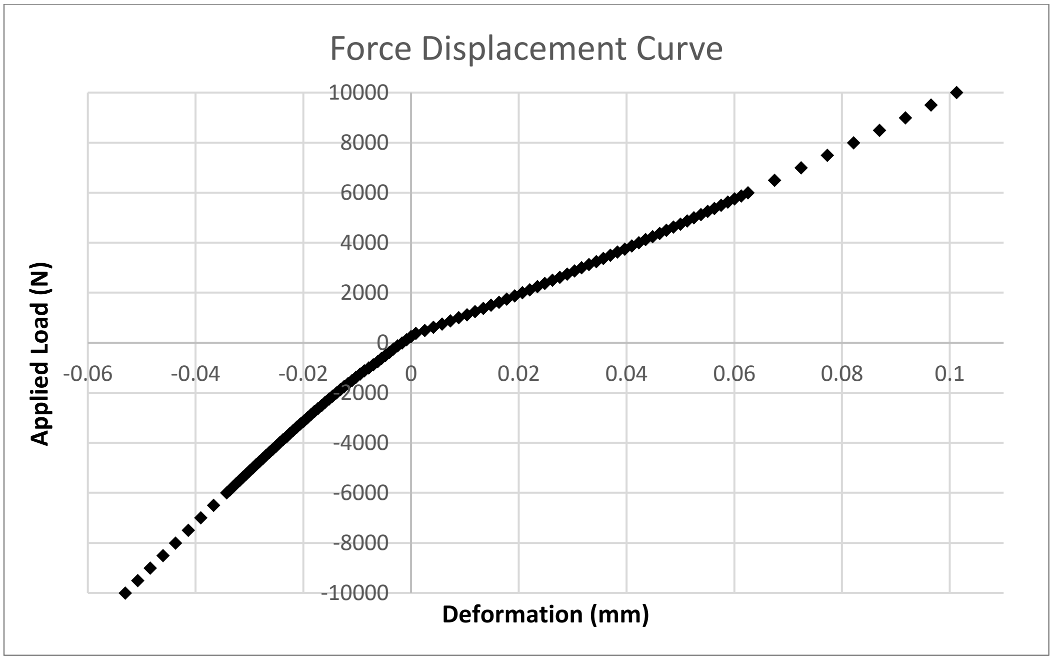
3.4 Modelling Stiffness

Once the geometry has been set-up, meshed, and preloaded as described above, it is analysed to determine the on-axis stiffness. To achieve this I apply a force to the mounting bolt (at the top) and vary this between a significant compressive and tensile load (arbitrarily -10kN and 10kN) over the course of 15 Load Steps in order to produce a detailed Load-Displacement Curve.

4 Results

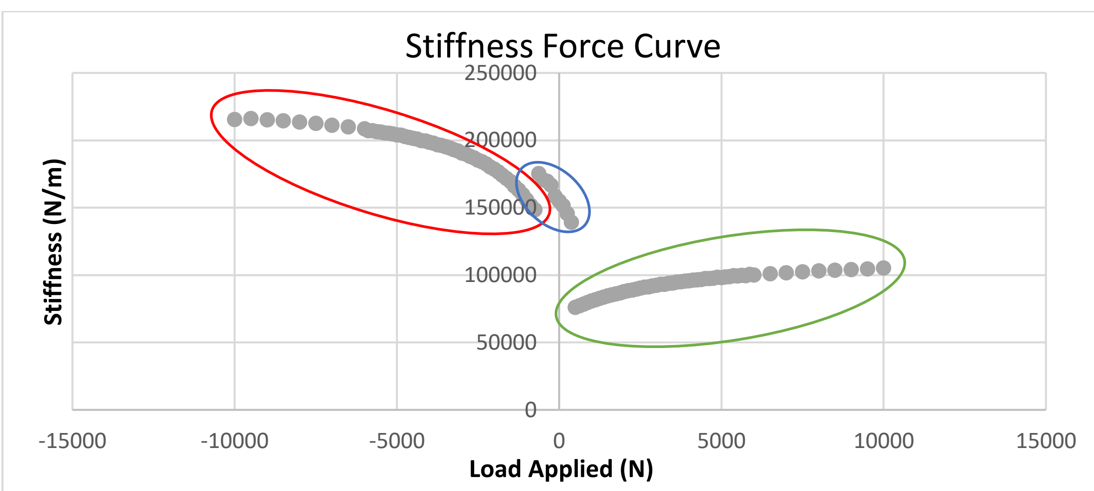
4.1 Force – Displacement Curves

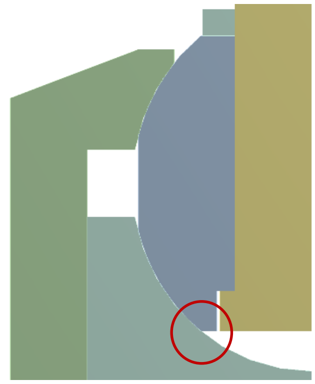
The resulting force displacement curve is given below:



Key points:

- There is a noticeable difference in gradient (stiffness) when loaded in compression rather than tension. This is unsurprising given the design and assembly.
- The line does not go through 0. This is due to the applied preload not being factored into the applied load on the graph. The y-axis intersect here is equal to the preload.
- The gradient of the graph increases as the magnitude of the load increases. This equates to an increasing stiffness as expected due to the increasing contact area in both directions. This is more clear if we calculate stiffness and plot that against load:



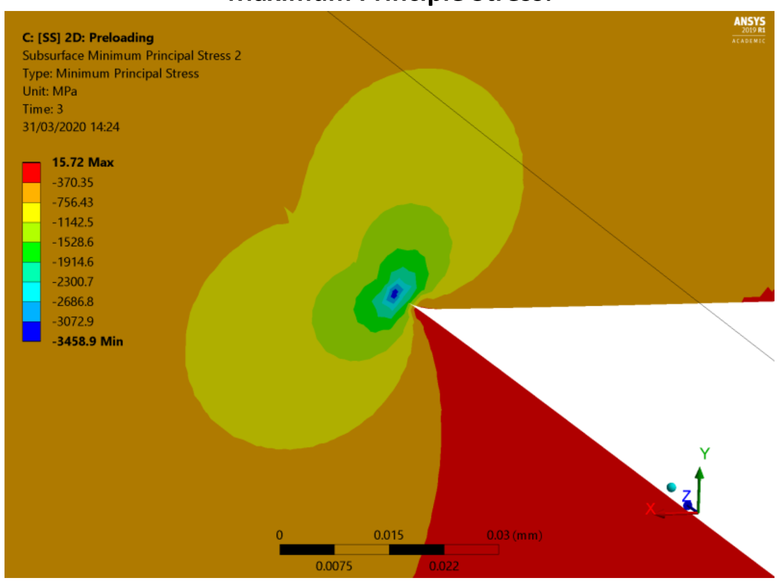


- Here we can see the large compressive stiffness which increases with load (red circle), the lower tensile stiffness which also increases with load (green circle), and the middle section (blue circle) where, due to the preload, there is contact on both the upper and lower surface of the bearing. This becomes more or less significant depending on the magnitude of the applied preload.

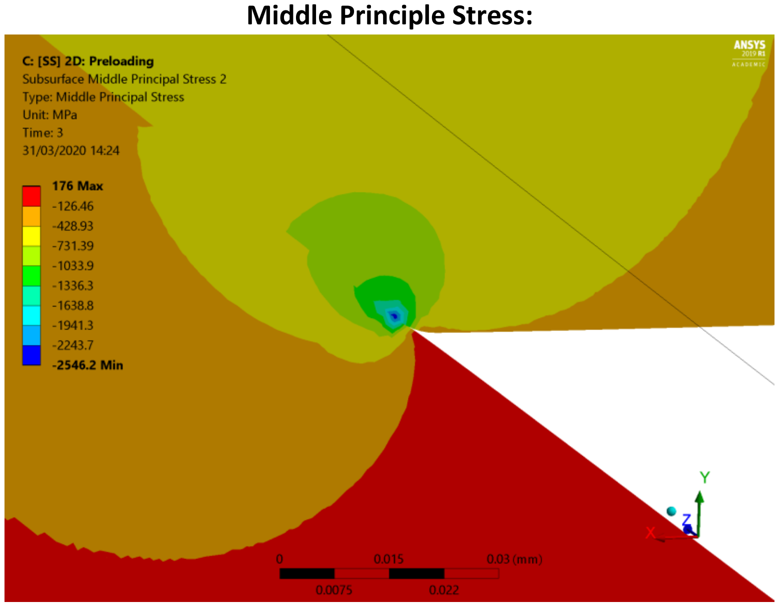
4.2 Contact Stresses

For verification I have also produced some plots of the principle contact stresses. All of the following plots show a small area around the compressive contact point (i.e. the red circle, top right) during the maximum compressive load case (note the negative sign due to the compressive load).

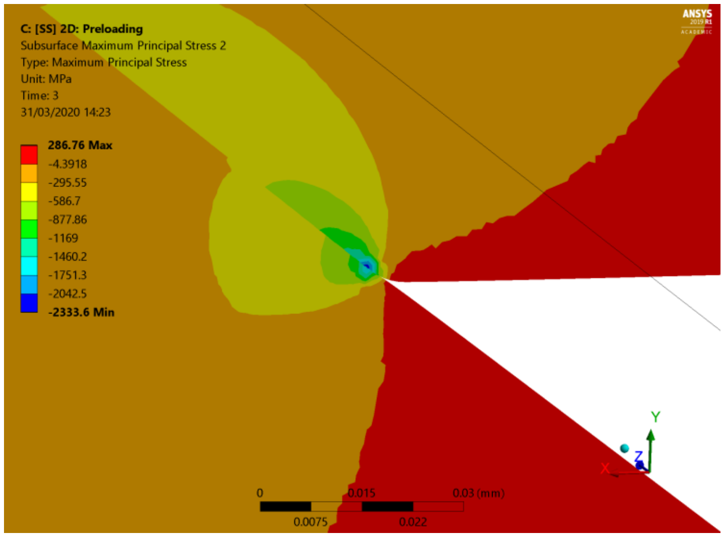
Maximum Principle Stress:



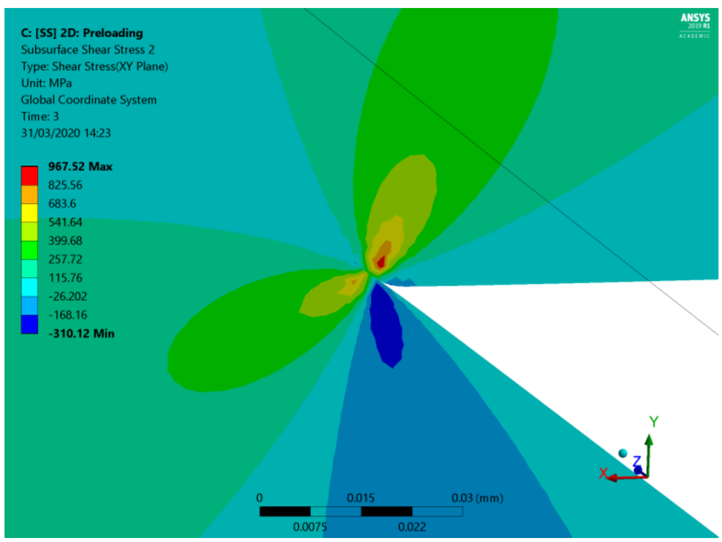
Middle Principle Stress:



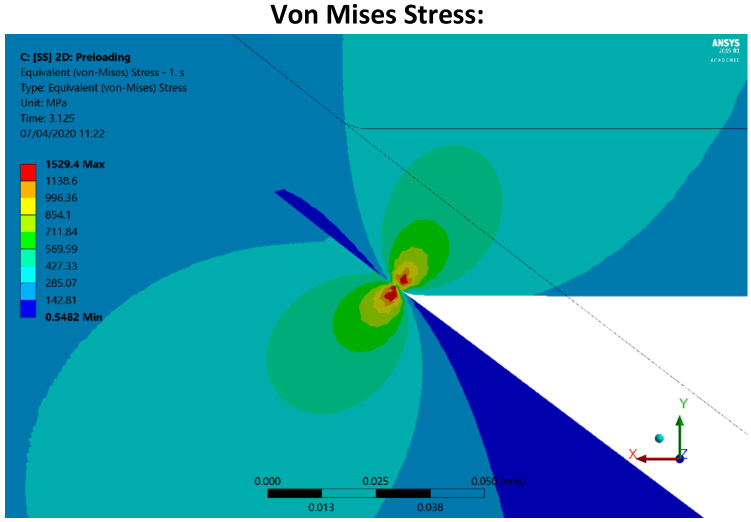
Minimum Principle Stress:



Shear Stress:



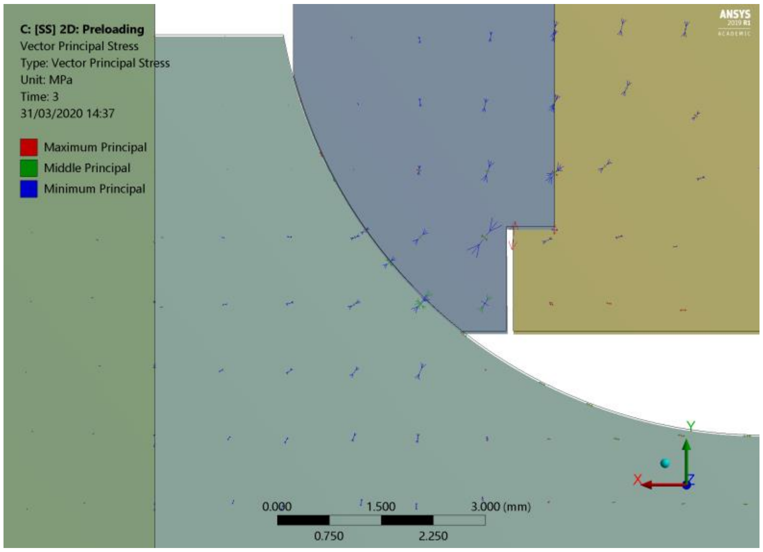
Von Mises Stress:



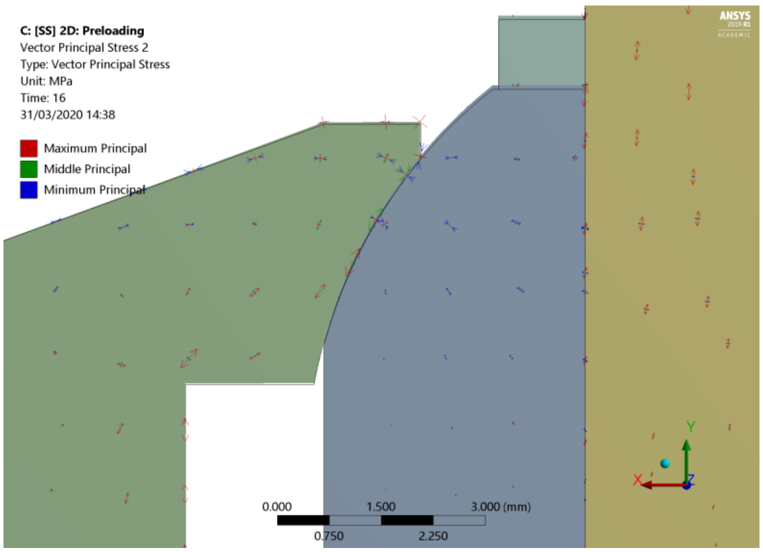
They certainly look reasonable to me (and what I was expecting to see), but I am happy to defer to your superior knowledge on this one. What do you think? Does this appear reasonable? And are there any other plots you would like me to generate for verification?

Out of interest I have also generated the vector plots of the principle stresses around the contact points in compression and tension:

Principle Stress Vectors around the contact point during maximum compressive load:



Principle Stress Vectors around the contact point during maximum tensile load:



5 Large 3D/Modal Analysis

I have an idea about how to take these results and input them into larger 3D analyses (such as the TBM modal analysis) without modelling the whole 3D joint in full, but this report is getting a bit long, so I will spare you from that for now!