

Cylindrical Hertzian Contact

Nick DiFilippo

Spring 2010

MCE 466- Introduction to Finite Element Methods

Purpose

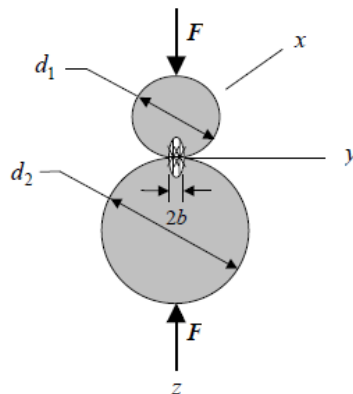
The objective of this project is to investigate if the theoretical equations for stresses in cylindrical Hertzian contact will yield similar results to stresses computed by Abaqus. Abaqus is a software package for finite element design and can model the problem of Hertzian contact between two cylinders. The results obtained from the theoretical equations will be compared to the results obtained by Abaqus in graphical form to determine if the Abaqus model is working properly.

Theory

Hertzian contact can occur when two curved surfaces are pressed together. In practical applications, these stresses can occur in the form of wheel and rails, automotive valve cams and tappets, mating gear teeth, and roller bearings. A force is applied at the top of one of the cylinders and results in a contact area that results from the initial contact between the two cylinders. In the area of the contact, intense stresses will form but be localized to very close to the contact area. The maximum pressure will act along the length of the cylinder down its center.

The overall contact area is defined as $2b$ and b is just half of the contact area and can be found with Equation (1) while the maximum pressure can be found with Equation (2)

$$(1) \quad b = \sqrt{\frac{2F \frac{(1-\nu_1^2)}{E_1} + \frac{(1-\nu_2^2)}{E_2}}{\pi L \left(\frac{1}{d_1} + \frac{1}{d_2} \right)}}$$
$$(2) \quad P_{max} = \frac{2F}{\pi b L}$$



This equation is general and will allow for different materials with different Modulus of Elasticity (E) and Poisson's Ratio (ν) to be pressed together while F is the force applied. The L in the equation is the length of the cylinder and d is the diameter of the cylinders. If the cylinder is pressed against a wall instead of another cylinder, the d of the wall will be equal to ∞ . This will cause that d to disappear in the equation because $1/\infty=0$. These equations can also hold true if there is contact of a cylinder internally. If this is the case, d would be a negative number.

In the figure above, the x plane is coming out of the paper while the z plane is in the vertical direction and the y plane is in the horizontal direction. Remembering these directions is important for determining which axes have what stresses on them. The equations for the stress are given for the x, y, and z directions by equations (3),(4), and (5) respectively.

$$(3) \sigma_x = -2\nu p_{max} \left(\sqrt{1 + \frac{z^2}{b^2}} - \left| \frac{z}{b} \right| \right)$$

$$(4) \sigma_y = -p_{max} \left(\frac{1 + \frac{2z^2}{b^2}}{\sqrt{1 + \frac{z^2}{b^2}}} - 2 \left| \frac{z}{b} \right| \right)$$

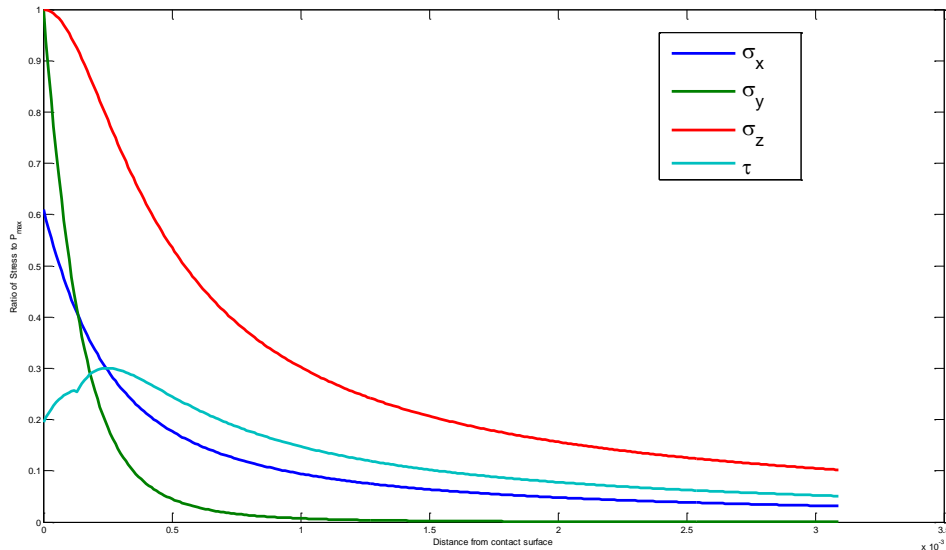
$$(5) \sigma_z = \frac{-p_{max}}{\sqrt{1 + \frac{z^2}{b^2}}}$$

The shear stress depends on how far you go along the z axis. If $z \leq .436b$ then the stress will be given by Equation (6) but if $z \geq .436b$ then z will be given by Equation (7). The shear stress will have a maximum value at $z=.786b$ and the stress will be $.3 * p_{max}$. The following figure shows what the stresses will look like if all of the stresses are normalized to p_{max} . The stresses in the y and z direction start at the same number while the stress in the x direction is lower.

$$(6) \tau_{max} = \frac{\sigma_x - \sigma_z}{2}$$

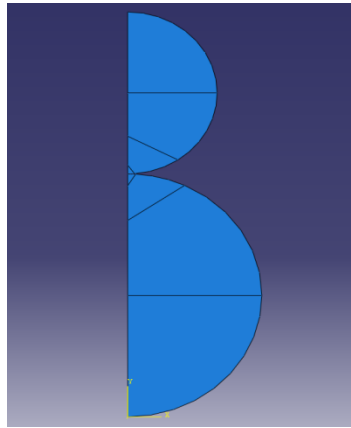
$$(7) \tau_{max} = \frac{\sigma_y - \sigma_z}{2}$$

Ratio of Magnitude of Stress/ p_{max}

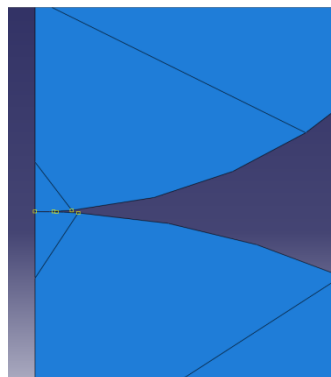


Modeling in Abaqus

Abaqus can simplify this 3d problem into a 2d problem. This is convenient because less mesh will have to be used over the cylinder and the nodes can be refined even more at the contact point. This particular model also took advantage of symmetry in the y direction so only half a circle had to be modeled. Two separate semi circles were modeled. One had a 4 inch diameter while the other had a 6 inch diameter. Both of the models were made out of the same material (Stainless Steel; $E=27.6 \text{ e6 psi}$; $\nu=.305$) and had the same thickness of 1 in. During the assembly portion of Abaqus it is important that each of the semi circles are their own part so one can be moved on top of the other.



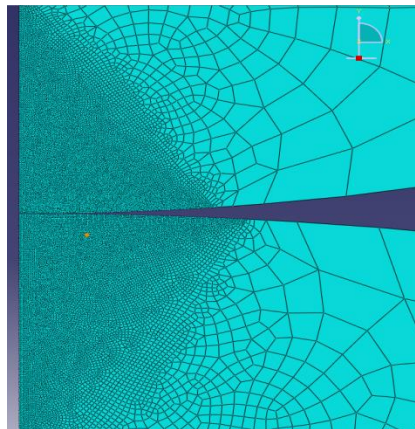
This model posed a challenge because the top and bottom semi circles have to interact with each other. These interactions are found in one of the drop menu called interactions. Here in the contact manager, a slave surface and a master surface must be specified. For this problem, the master surface was the bottom and the slave surface was the top. The contact type must also be 'node to surface' rather than 'surface to surface'. The latter option will result in errors when the job tries to run. There is also contact behavior that needs to be specified. The normal behavior is the contact that acts straight up and down and the type of contact is "hard" contact. The tangential behavior acts in the horizontal direction and is frictionless. Another option that needs to be unselected is 'allow separation on contact'.



The boundary conditions and the loads are very important in obtaining an accurate model. There are two boundary conditions that need to be applied. One boundary condition is that the bottom semicircle has to be constrained in the x and the y directions. The entire flat side of the semicircle has to have a symmetric boundary condition applied to it. It can either be just restricted in the x direction or a symmetry

boundary condition can be used. Since the model takes advantage of symmetry, this has to be taken under consideration when the load is applied. If the total load applied on the cylinder is going to be 500lbs, the model load applied in Abaqus must be 250 lbs. If you use a load of 500 lbs in Abaqus the result that you will obtain will be much higher than what the theory will give you. Also the load you choose must be high enough so a large contact area will form. A small force will not be as accurate because although the mesh is very fine, the contact area is very small and will not accurately be portrayed.

Since the model is in 2D, the mesh can really be refined around the contact point. However the academic license of Abaqus limits the user to 20000 nodes. The best results were obtained when the mesh was refined to about 18000-19000 nodes. The final mesh was even more refined in the contact area than the figure below. In order to keep refining the mesh, the model has to be partitioned and this model was partitioned by geometry. The mesh used was quad dominated and in the mesh type, reduced integration had to be unchecked. Also after running a plane stress simulation, the results didn't line up especially in the S3,3 direction. In plane stress, there was no consideration of what would happen in the direction coming out of the page but this can be fixed by looking at the model with plane strain.

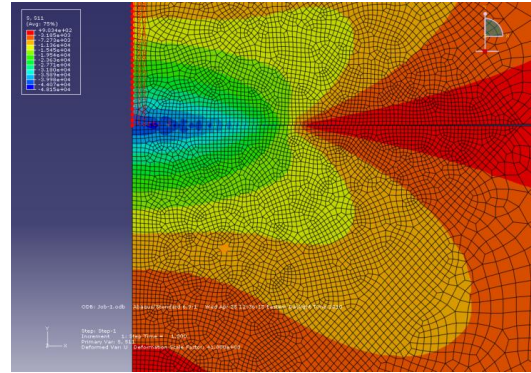
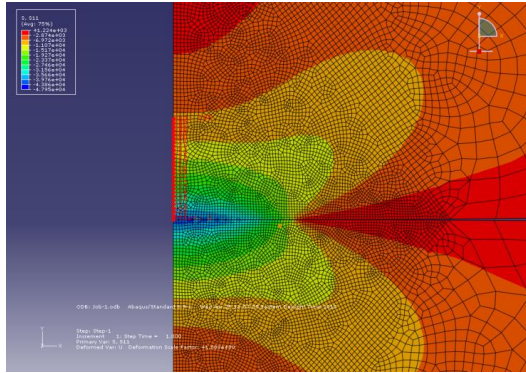


Comparing the plane stress and plane strain results, as seen in the figure below it is obvious that the stress is similar between the two in the S1,1 and S2,2 directions. Analyzing the figures further will help to determine which one is a better fit. The table below looks at the maximum stress that Abaqus for both the plane strain and plane stress analysis and compares then to the theoretical numbers. It is important to keep the planes correct. Abaqus calls the vertical direction the y direction while the theory calls that the z direction. Abaqus calls the horizontal direction the x direction and the theory calls it the y direction and Abaqus calls the direction coming out of the paper z while the theory calls this direction x. These results will use the notation of the theory.

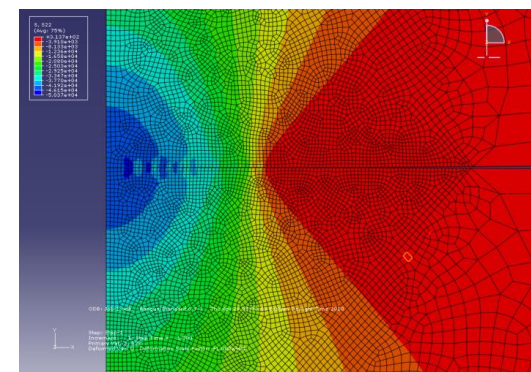
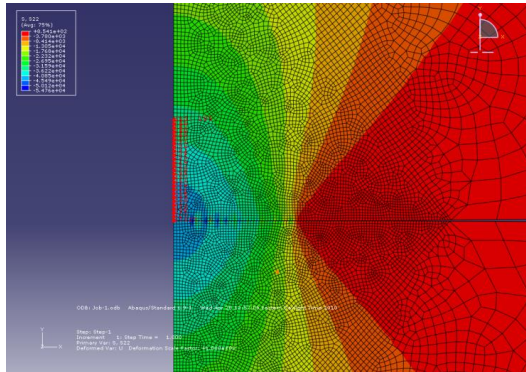
Plane Stress

Plane Strain

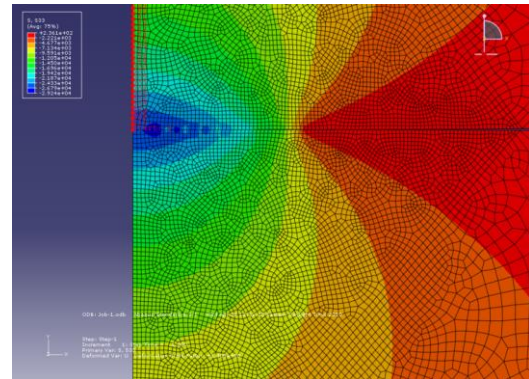
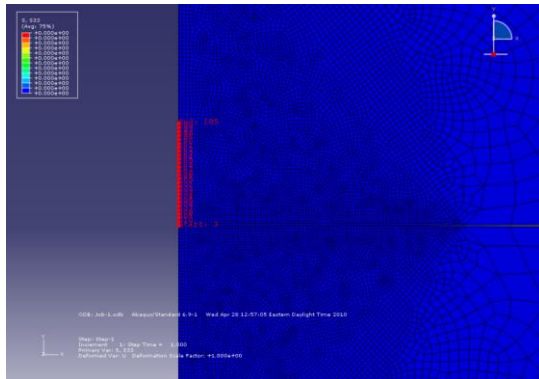
S1,1



S2,2



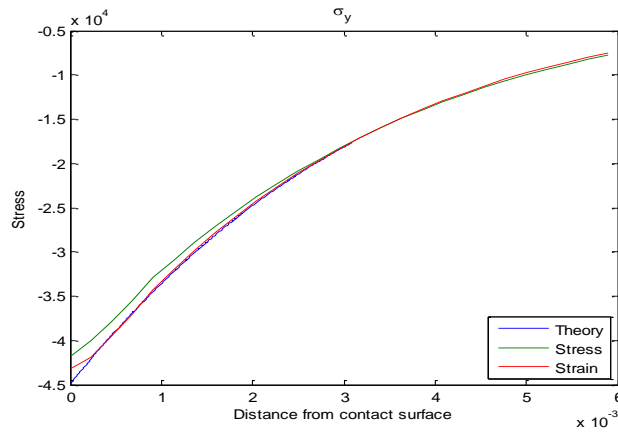
S3,3



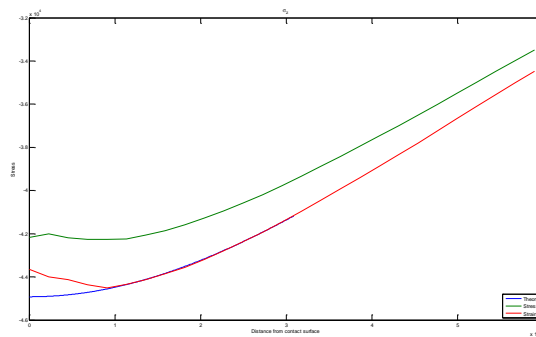
	Plane Stress (psi)	Plane Strain (psi)	Theory	% Error Stress	% Error Strain
σ_y	41738.1	43194.5	44922	-7.088%	-3.846%
σ_z	42168.3	43652.2	44922	-6.130%	-2.827%
σ_x	-----	26488.2	27403	-----	-3.338%

Looking at this table it is easy to see that the plane strain analysis is easily the superior analysis type. In addition to being able to determine the stress in the x direction all of the stresses that were found had a lower percent error. Now by comparing the theory and the Abacus results it confirms the initial thought that the plane strain is better. All of the plane strain analysis match up with the theoretical results faster than the plane stress analysis. In the following graphs, the theory is the blue line, the strain is the red line and the stress is the green line.

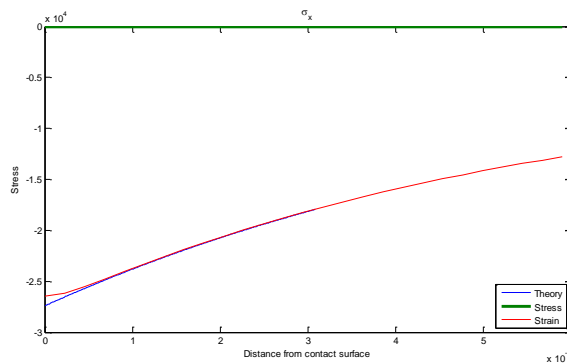
σ_y



σ_z



σ_x



Conclusions

Plane strain was the better analysis type because it was able to get closer results for the maximum stresses and was able to see the stress in the x direction. Abaqus was able to accurately match the stresses that occurred in the X, Y, and Z directions. If the mesh around the contact area was finer, Abaqus would be able to obtain even more exact results. However due to mesh node limits this was not possible to keep refining the mesh or that practical with an academic license. If Hertzian contact in a cylinder was to be explored more, it is possible but it would be better and easier to look at with an unlimited node license.