Stress Intensification Factors at Pipe Elbows

Abstract:

The point of this project was to confirm the accuracy of the Stress Intensification Factor (SIF) equation in various pipe sizes. We used ABAQUS an FEA program to show the SIF factor in four pipes and compared them to the hand calculated results. The final results showed that the hand calculations are accurate for the smaller pipes but stray from the hand calculations as the pipe size increases.

Introduction:

After graduation I will be working for Chevron as a Pipe Design Engineer. For this reason stresses and corrosion in pipes are very important to me. I have been reading "Introductions to Pipe Stress Analysis" by Sam Kannanppan. In this book the author talks about a phenomenon called Stress Intensification Factor (SIF). In the book I found out that I could use this equation on any size pipe to find the SIF in Elbows and tees. The book also had another equation called the Correction Factor Intensification (CFI) that can be used with SIF in order to correct for pipe sizes this new equation is called the Corrected SIF equation. In order to use this equation we need to use the pipe stress equations from Design 1. These equations can be used to calculate longitudinal stress, radial stress, and tangential stress in a pipe. The limitation that these equations have is that they assume that the pipe is long pipe with no bends and no ends. However, if I combine the equations from pipe stress equations from Design 1 with the SIF equation I can find the max stress in any size elbow.

Desired Outcome:

When I started this lab I wanted to prove that the Corrected SFI equation held true in four different pipe sizes 1", 2", 6", and a 12". I choose these pipe sizes because they are the most common sizes found in a refinery. My expectation in this project was to obtain "pretty" FEA pictures in order to show the location of the stress concentration. Before running my experiment I expected to show that the highest stress in the elbow would be on the outside corner of a pipe. I reasoned that elbows always fail in the outside section. However, after running the experiment I found out that the outside part of the elbow was actually the lowest stress point in the pipe. My other expectation was to prove that the Corrected SFI equation held true in any size pipes. Just for fun I decided to see if there was a liner relation between pipe thickness and stress, I also checked to see if there was a linear relation between pipe pressure and pipe stress.

Proposed Analysis:

In this project I plan to use the FEA program ABAQUS in order to prove that the Corrected SFI equation holds true in my four pipe sizes. I will use the3-D Shell Sweep option to draw my piece. The limitation with this method is that I will not be able to calculate the radial stress in the pipe. However, I will be able to calculate the longitudinal and tangential stress in the pipe. Using the pipe equations from Design 1 we know that the highest stress in the pipe will be in the tangential direction so as long as the model can calculate this stress correctly I will be able to check for the SFI. In order to correctly model the piping system I need to obtain the pipe nominal thickness, pipe nominal Diameter, Radial bend, and Pipe Pressure. The pipe class that I will be using is XS/80 Carbon Steel for all piping systems. The thickness and nominal pipe diameter for this pipe class can be found in Introduction to Pipe Stress Analysis. The Pipe Pressure that will be put into the pipe will be 1800 psi. The reason is that this is the max pressure recommended in a 6" pipe. The recommended pipe pressure increases as pipe size decreases this means that the 12" pipe will be receiving a much higher stress then is recommended by ASNI. However, I need to make sure to apply the same pressure for all pipe sizes other wise I can not make a linear relationship between pipe classes. The assumed velocities for all pipes are 4 ft/second. The reason is that this is the velocity that is recommended in pipe systems. The other benefit is that the stress due to fluid flow is much lower about 1% of that due to pressure and therefore can be ignored in this analysis.

Approximate Solution:

The point of this project is to compare the SIF in an FEA model with the equation from "Introduction to Pipe Stress Analysis". To do this I did it in two ways, first I did a hand calc of a 6" pipe. After checking my solution I created a program in Engineering Equation Solver (EES) to solve for the stresses, SIF, CIF, and Corrected SIF in the four piping systems. I then compared my EES solutions to my hand calculations to confirm that EES was working properly.

Model Development:

My model was pretty simple to start of with the only thing I did to simplify it was use 3-D shells instead of a 3-D solid. The reason being that this would greatly reduce the amount of elements I would have to calculate and my data would still be accurate with the exception of my radial stress. My hand calculations said that the radial stress would be the smallest stress in the system so ignoring it would not affect my analysis. My model had an elbow in the middle with the radius of the curve being 1.5 times the nominal diameter. According to the text book this was the standard radius curve for pipes. I also had 25 ft long straight pipe attached to each end of the elbow the reasoning for this is that I wanted to ensure that the stress from in the pipe section was not being effected by the boundary conditions (BC) or the SIF from the elbow. The BC was applied at each end of the straight pipe and the restriction applied was an ENCASTRE restraint. The element chosen was a Quad-Dominated sweep with Quadratic Integration.

Mesh Convergence:

In order to obtain accurate results in ABAQUS one must insure that the mesh has converged at the point of interest. In this project we have two points of interest one is the max stress in the pipe and the other is the max stress in the Elbow. I ran mesh sizes in the 1" pipe ranging from 1.6 seed size down to .2 seed size the amount of elements ranged from 52 elements to a maximum of 3287 elements. I meshed the part using a Quad-Dominated sweep and a Quadratic integration method giving me 8 DOF for each element. The Mesh convergence study can be found in figure 1 (Mesh Convergence study of a 1" elbow) and figure 2 (Mesh

Convergence of a 1” pipe). In both of these mesh studies it was found that a seed size of .2 was the seed size that I need to use to insure convergence of my model.

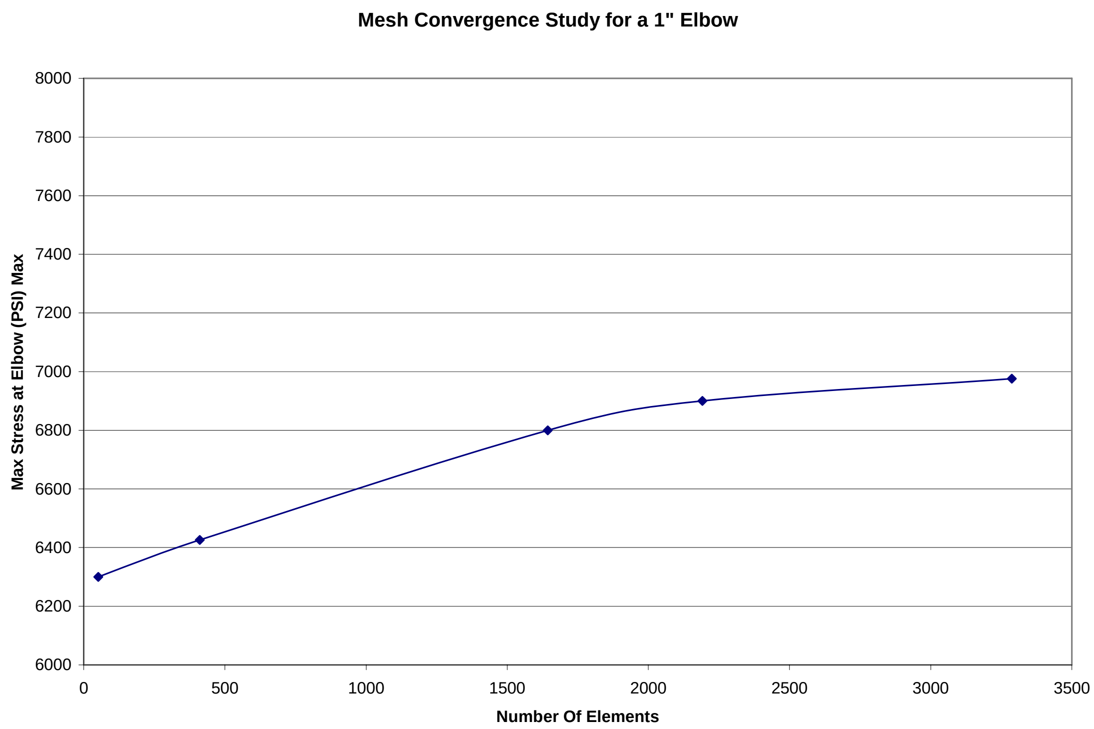

Figure 1: Mesh Convergence Study for 1” Elbow

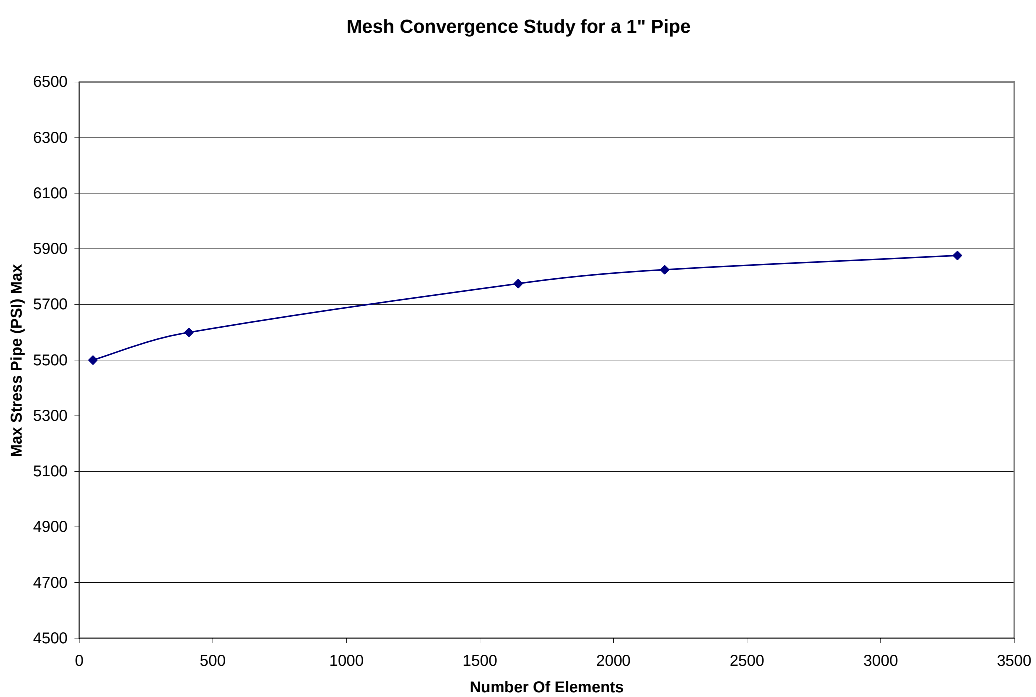

Figure 2: Mesh Convergence Study for 1” Pipe

Analysis:

My main analysis in this lab was to check for the SIF in the four pipe systems. However I also ran a quick test on the effect of changing the pressure and the pipe thickness. I expected the change in stress to be linier with the change of pressure and pipe thickness. I didn't really have any major errors or warnings for the most part it was mostly user errors, for example I would change the section property or seed size and I would forget to re-mesh the part. The only major problems I had was when I was meshing the system the sweep command would give me an error the thing I need to do was change the direction that the program was doing the sweep in a few of the partition locations and my program ran properly.

Post-Processing:

After running ABAQUS I checked my solution by first confirming that my system had conversions. For more about mesh conversions see mesh conversions section along with Figure 1 and Figure 2.

The next step was to compare my answer from ABAQUS with my hand calculations. The first thing I noticed was that ABAQUS did not show any stress in the radial directions but that my hand calc had showed that I should have a small radial stress in the my pipe. I used the Shell Sweep option when creating my model so I believe that ABAQUS did not calculate a stress in the radial direction because my thickness was zero. I also found that my longitudinal stress was way off in all of my calculations while my tangential and max stress and SIF where all pretty close. Results shown in table 1 show my numerical results (hand Calculations) vs. my theoretical results (FEA) and it then shows the percent error between my theory and my numerical.

Stress type	1 in pipe			2 in pipe		
	Theory	Numerical	Error	Theory	Numerical	Error
Stress longitudinal (psi)	3000	2955	-2%	4488	3598	-25%
Stress tangential (psi)	5876	5876	0%	8871	8996	1%
Max stress on straight Pipe(psi)	5876	5876	0%	8871	8996	1%
Max stress at Elbow (psi)	6416	6543	2%	11350	11605	2%
SIF	1.09	1.07	-2%	1.28	1.29	1%

Stress type	6 in pipe			12 in pipe		
	Theory	Numerical	Error	Theory	Numerical	Error
Stress longitudinal (psi)	6389	5582	-14%	12620	10143	-24%
Stress tangential (psi)	12863	12965	1%	22775	22087	-3%
Max stress on straight Pipe (psi)	12870	12965	1%	23052	22087	-4%
Max stress at Elbow (psi)	15878	16828.57	6%	27000	35450	24%
SIF	1.23	1.30	5%	1.17	1.61	27%

Table 1: Numerical (hand Calculations) versus Theoretical (FEA) Pipe Stress and SIF

Results

My desired outcome was to show that changing pressure and thickness would have a linear effect on stress my results in Table 2 show that the doubling or halving the thickness would give me double the stress but it would have no effect on the SIF in the Elbow.

In this report I was also looking to confirm that the Corrected SIF from the text book held true for all pipe sizes if you look in Table 1. The reader will notice that the Stress Longitudinal was always way off compared to my hand calculations but that the tangential and Max stress where always pretty close to those in the hand calculations. The only exception is the max stress in the 12” elbow is much smaller then what was expected the reason for this can be attributed to the local section deformation in the pipe

Stress type	Effects of reducing pipe Thickness			Effect of Doubling Pipe Pressure		
	T=.18 in	T=.09 in	% Change	P=1800 PSI	P=3600 PSI	% Change
Stress longitudinal	2955	5490	1.9	2955	6000	2.0
Stress tangential	5876	11668	2.0	5876	11753	2.0
Max stress in straight Pipe	5876	11712	2.0	5876	11753	2.0
Max stress at Elbow	6543	13050	2.0	6543	13050	2.0
SIF	1.07	1.11	1.0	1.07	1.11	1.0

Looking at Figure 3 and 4 the reader will see that the location of minimal stress is along the outside of the pipe. Figure 3 shows the results for a small pipe while figure 4 shows the results for a big pipe.

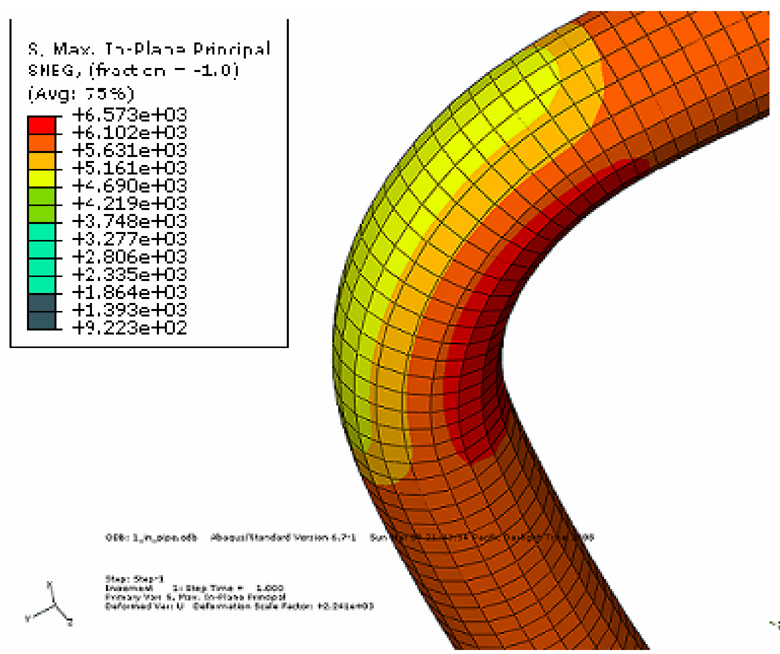

Figure: 3 Stress at the elbow of a 1” pipe

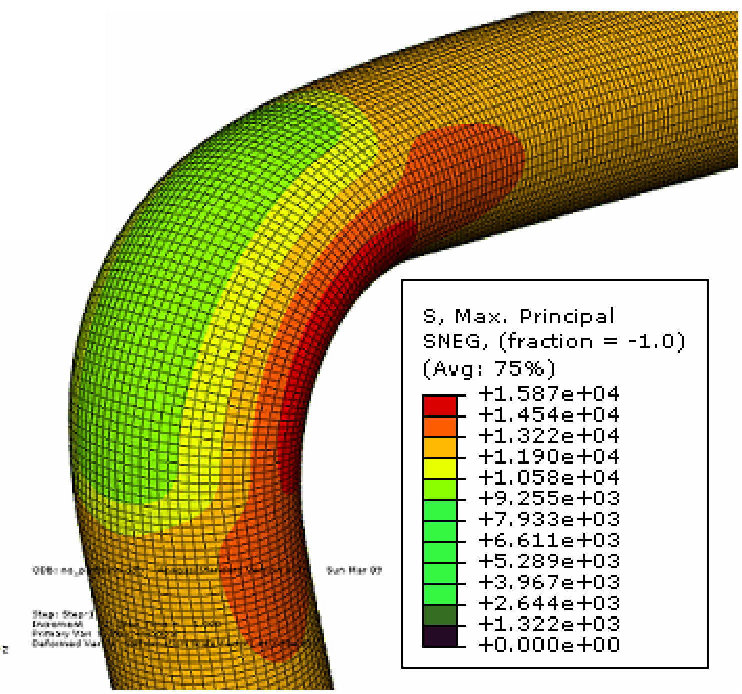

Figure 4: Stress at the Elbow of a 6” pipe

The most interesting pictures obtained from ABAQUS had to be those obtained in Figure 5 and Figure 6. Figure 5 shows the max stress in a 6" pipe while Figure 6 shows the max stress in a 1" pipe. Notice how the 6" pipe has a lower stress in the inside of the pipe while the small 1" has a constant stress through out. Also if you look hard you can notice that the 6" pipe has small sections of lower stress in the inside of the pipe. We believe that the 6" pipe stress concentration differs from the 1" pipe is because the larger pipe has local section deformation, we also believe that this deformation is not taken into consideration in the Corrected SIF equation.

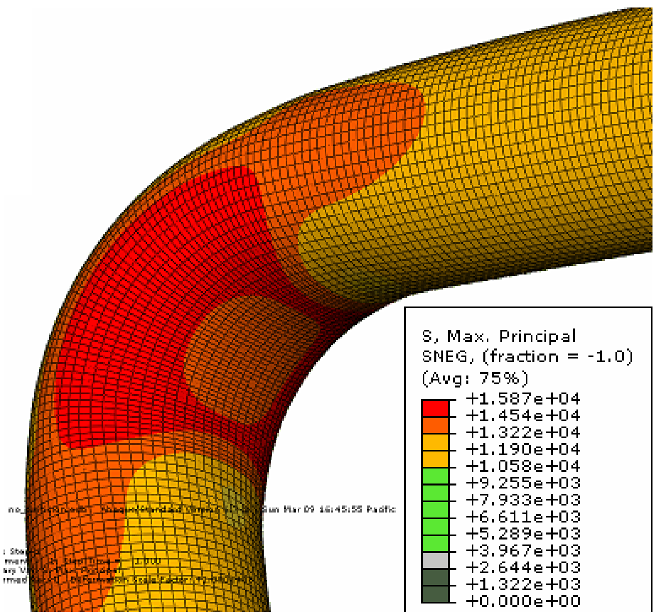

Figure 5: Max stress in an Elbow for a 6" pipe

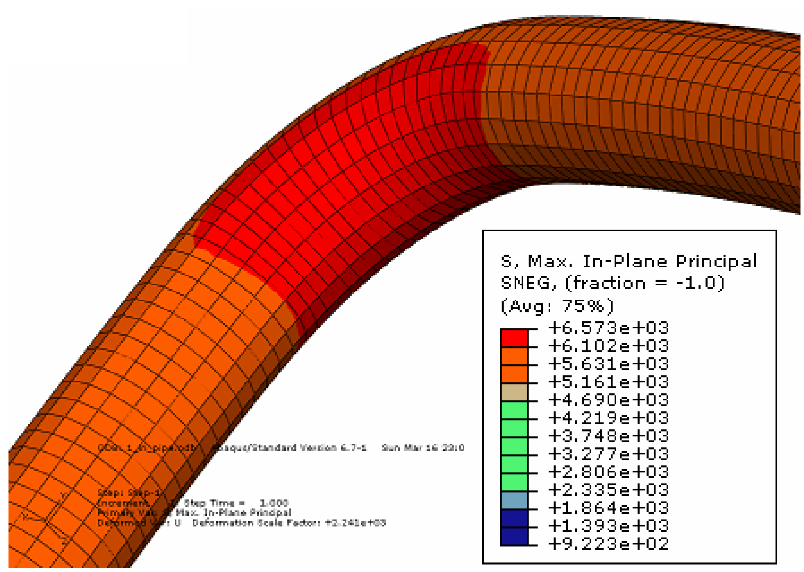

Figure 6: Max stress in an Elbow for a 1" pipe

Discussion

This lab was really helped me understand the stresses in pipes. If I could do this again I would like to look at the stress concentration for a Tee. I am really interested in finding out the max stress location and the minimum stress locations. I am actually really happy with my lab and I would not change a thing I got the information I wanted and I found out that the location for high stress was in the inside part of the curve for small pipe and along the sides for larger pipes. I also learned that pipe fails in the outside location of the curve because of decrease in wall thickness due to materials hitting and destroying the outside location of the pipe.

Conclusions

This projected showed that the Corrected SIF equation in the book "Introduction To Pipe Stress Analysis" is accurate for small pipe sizes but that it does not do a good job of showing the stresses in large pipes. It was also found that the large pipes had the stress concentration on the sides of the pipe while the small pipes had the stress in the inside of the pipe. It is believed that the reason the large pipes have this high stress on the sides is that the larger pipes have a local section change due to the high stress. This local section changes causes the equation to show a higher stress then is actually observed because it doesn't take into consideration the energy that goes into deforming the pipes section.

References

Kannappan, Sam. Introduction to Pipe Stress Analysis. New York: John Wiley and Sons, 1986

Mischke, Charles R. Mechanical Engineering Design. New York: McGraw-Hill College, 2003.