Boundary Condition Influences on Shank Stress in 3D Solid Bolt Simulation

Jerome Montgomery

Siemens Power Generation Inc., Orlando, FL.

Abstract: *When simulating bolts, the amount of detail to include is often raised. The analyst is left with using judgment in deciding to include or not include details, such as threads. For system models, where the primary bolt function is to transfer load from the cover to the base, thread details are sometimes perceived as not needed. It is believed that a reasonable result can be achieved without this detail.*

Should the bolt head contact interface be bonded or full, and how does this affect the shank stress, is another concern. This again is a judgment left to the analyst.

This paper evaluates boundary conditions and software settings used in simulating bolts. A simulated threaded bolt with full contact is the baseline model. A model with interaction, smear, at the threaded region and one with tied contact at the threaded region is compared against the baseline model.

A summary table is generated to compare the results of the approaches used.

Keywords: *Bolting, Bolt Simulation, Boundary Condition, Influence, Shank Stress, Bolt Body, Contact, Smear, Thread Smear, Smearing, Thread Contact, Solid Bolt, Bolt Modeling, Abaqus/CAE, Abaqus, CAE, FEA, Finite Element, Finite Element Modeling, Thread Interaction, Preload, Pro/ENGINEER, and Pretension.*

1. Introduction

A number of general purpose finite element software programs include the capability to simulate a three-dimensional (3D) solid bolt (Figure 1) with pretension and contact behavior.

Figure 1. Single-threaded Bolt.

This capability has occurred within the last ten years. Prior to solid elements with pretension capability, most analysts used beam elements to simulate pretension. Now, realistic bolt pretension and contact behavior can be simulated.

However, run-time is a concern when simulating threaded 3D solid bolts. This is a valid concern and has been offset by increased computer power and improved solvers. Users do have options. These options include the aforementioned beam element approach on one extreme to tied contact at the threaded area for 3D solid bolts at the other extreme. And now for those desiring the 3D solid bolt extreme with the threaded behavior without the threads Abaqus/CAE has threaded interaction capability. It is sometimes called Smear Contact because it is visually setup similar to the tied solid bolt element but with the threaded capability smeared over and computed internally.

2. Simulation Setup

There are many bolt configurations that can be simulated. The two more common ones are stud bolts through clearance holes and stud bolts threaded (tapped) to certain depth. The later will be used in this paper. A base and cover is typically used to house bolts for this type. Figure 2 shows a symmetry model with a single bolt.

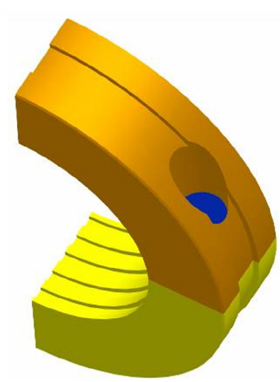

Figure 2. Symmetry Ring Section with Single-Threaded Bolt.

A cross-section view of the bolt is shown in Figure 3.

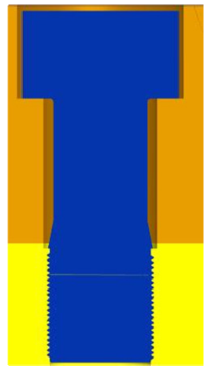

Figure 3. Cross-section showing Single-Threaded Bolt.

A close up of the threaded region is shown in Figure 4. A finite element mesh of the bolt with threads will create a large number of elements. The concern is that more elements mean longer run times; however, more elements also mean a more accurate solution. Alternate approaches can be used when this is an issue.

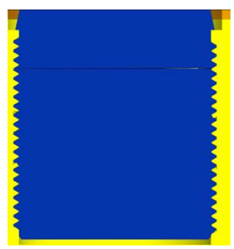

Figure 4. Close-up of threads.

The 3D solid bolt without thread details is shown in Figure 5.

Figure 5. Unthreaded Bolt.

This geometry can be used for both tied threads and smeared threads.

The 3D solid bolt tied in the threaded section is the most used 3D solid bolt simulation. This is because it is a time savings, while maintaining 3D behavior; but, the thread effect is sacrificed. This accuracy loss is the judgment of the analyst.

The Abaqus/CAE Smear Interaction feature allows the analyst to no longer sacrifice thread effect accuracy. To date, only Abaqus has the Smear (thread interaction) simulation feature. Again, this feature allows for including the behavior of bolt threads without the added detail. Calculations are performed internal to approximate the behavior of the bolts. This feature gives the user the best of both worlds. The user has the modeling simplicity of the 3D tied bolt with the near accuracy of the threaded bolt.

The key behaviors, for three-dimensional (3D) solid bolt simulation, are pretension and contact interaction. Simulation of these behaviors strongly influences the resulting bolt shank force. These key behaviors will be described in preparation for setting up the models to be run. Solid sections will be used for visual effects, but the models are ran axi-symmetrically, for simplicity. The setup screens for 3D bolt simulation are the same.

2.1 Pretension

Pretension is the effect of applying an initial load in the bolt to fasten the cover and base, thereby, resisting external loads from opening the bolted joint. Pretension is simulated by splitting the bolt body (Figure 6) and pulling each end towards the other (as an initial displacement) to obtain the desired force (pretension). In Figure 6, the bottom segment is pulled towards the top segment as represented by the blue arrow and the top segment is pulled towards the bottom segment as represented by the red arrow.

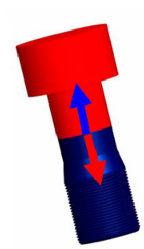

Figure 6. Split Bolt.

The user specifies a pretension (force) or an initial displacement to represent the preload. From this, the software generates a preload in the bolt as an initial condition.

In Abaqus/CAE a datum axis feature is created. Then a partition is created. This partition is the split location of the bolt for pretension as shown in Figure 6. Figure 7 shows the datum axis and the partition.

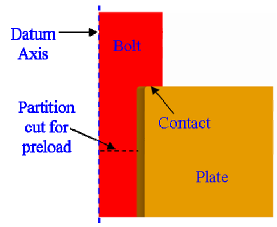

Figure 7. Partition and Datum Axis.

Abaqus/CAE obtains this information via a dialog box as shown in Figure 8. The aforementioned partition and datum axis should be generated prior to opening the pretension (Edit Load) dialog.

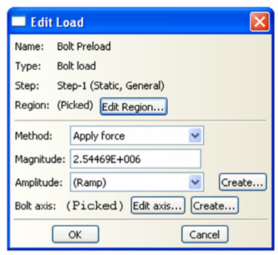

Figure 8. Bolt Preload Dialog.

Note that the Region to be picked is the partition cut (the location that the preload will be applied). The Preload Method is input either by applying a force (Apply force) or an initial displacement (Adjust length). In the dialog, the force method was used therefore, the magnitude is a force instead of a displacement with a value of 2.54469E6 N. This value should be positive for most problems. For problems in this paper, the Amplitude options were not created. Defaults were used. Figure 9 shows the dialog when the create button is selected for Amplitude.

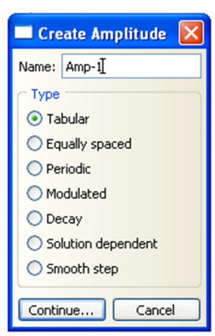

Figure 9. Bolt Preload Amplitude Dialog.

If the Continue button is selected from Figure 9, another dialog will appear (Figure 10).

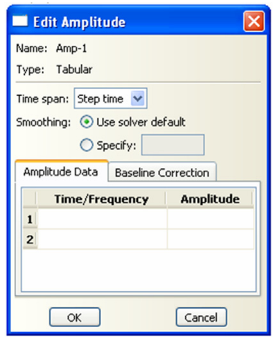

Figure 10. Bolt Preload Edit Amplitude Dialog.

This allows users to input varying preload values. Again, the default Amplitude selections are used for this paper.

The bolt axis was selected earlier.

Therefore, the steps for pretension as described above are,

- Create an axis for the bolt,
- Partition the bolt for the pretension split by creating a partition perpendicular to the shank (at a user defined elevation along the shank) , and
- Apply the pretension load as a boundary condition.

2.2 Contact Interaction: Bolt Head to Plate Cover, Horizontal Joint, and Thread Area

Contact interaction is the other bolt modeling behavior that has influence on the results. There are three main contact interaction locations of concern (for the type of bolt described in this paper). They are, the bolt head to cover plate interface, cover plate to base plate interface (horizontal joint), and bolt threaded area to base plate interface. Figure 11 shows the three contact interface locations.

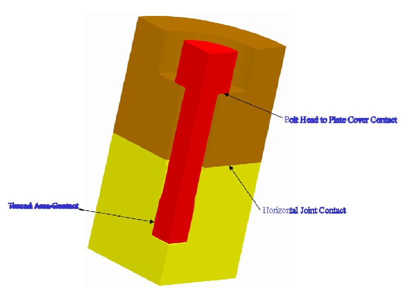

Figure 11. Simple Model With Top and Bottom Plate.

For each location, a contact interaction is generated in Abaqus/CAE by first, making the selection to create an interaction (From main menu bar: Interaction/Create). A dialog will appear (Figure 12) for the user to name the interaction, select the step the interaction is in, and select the interaction type. For the interaction type in this paper, surface-to-surface contact (Standard) is used. The Continue Button is selected and the master and slave surfaces must be interactively selected.

Figure 12. Interaction Creation Dialog.

After the master and slave surfaces are created, the Edit Interaction Dialog (Figure 13) will appear. The Edit Interaction Dialog is used in Abaqus/CAE to define the interaction parameters.

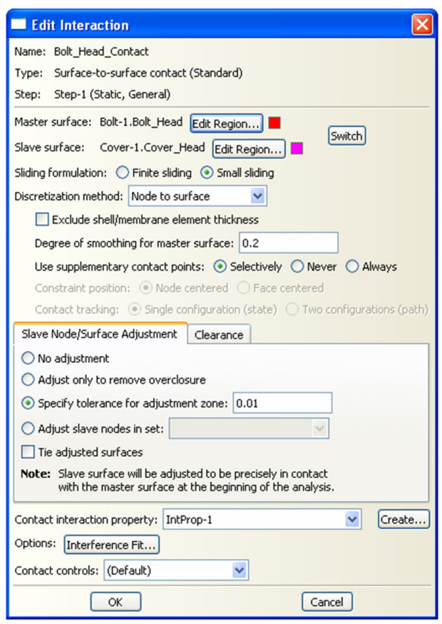

Figure 13. Interaction Definition Dialog.

Small sliding is the sliding formulation used in this paper; since in bolting, there is expected to be little relative sliding of one surface along the other. The default node to surface option was used as the Discretization Method. Defaults were used for the degree of smoothing for master surface and selectively used supplementary contact points.

There is a tab for the Slave Node/Surface Adjustment within the dialog. Here a tolerance value was specified; since the original offset was known. This offset was used in cases where the CAD tool would merge the two volumes, if no offset was generated; thus, a small offset was generated. This is the value input in the specify tolerance edit window.

For tied, there is a specific selection in the dialog (Tie Adjusted Surfaces). The Clearance tab was not selected. No changes are affected with its parameters at this time.

Next, a Contact Interaction Property should be defined. A button to create a Contact Interaction Property is located on the edit interaction dialog. Picking this button opens another dialog. Two contact directional properties are usually defined for this class of problems in Abaqus/CAE, normal and tangential (Figure 14).

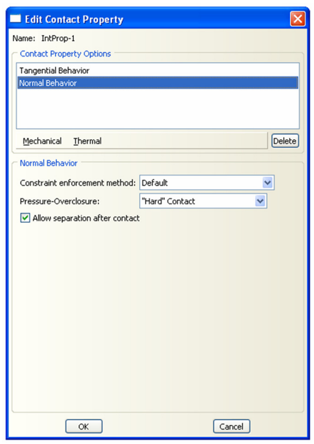

Figure 14. Interaction Property Dialog – Normal Behavior.

Normal contact is behavior perpendicular to the surface, whereas, Tangential contact is transverse, to the surface. There are several options available in the software, but for Normal contact, the Default Constraint Enforcement Method was used, along with Hard Contact as the pressure-overclosure option is also used in the example problems. For Tangential behavior (Figure 15), the Penalty option for friction formulation with a friction coefficient of 0.15 is used.

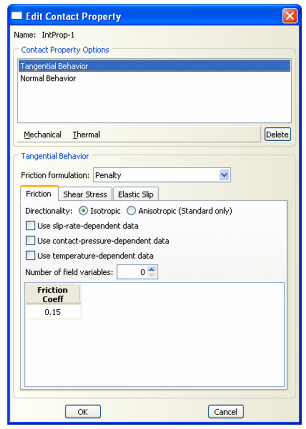

Figure 15. Interaction Property Dialog – Tangential Behavior.

When the tied contact option is used, neither, Normal contact nor Tangential contact is used; however, tied contact could influence the local results more than expected.

The default option for contact controls, from the edit interaction dialog, is used in the problems in this paper.

The horizontal joint contact area is where good judgment is used to try and increase speed in turn-around, without sacrificing accuracy. This is the location where sometimes frictionless tangential contact behavior is used on system models. This is also the location where the user should make a good decision as to whether the problem is bending dominate or stiffness dominant.

The thread contact area is the most critical area for bolt realism simulation. Actual threads are the most realistic simulation. Second to that, is the thread interaction (Smear) capability. And last, is the tied (or bonded) contact behavior at the threaded area; tied gives faster solution.

Now that the primary parts have been explained, an actual bolt is simulated to show stress results due to specific boundary conditions and software settings. A single M120 bolt is selected over a system model in order to better isolate items specific to setting up the bolt simulation. The bolt length is over an L/D ratio of three. The material is Structural Steel with a modulus of elasticity equal to 200000 MPa, a Poisson's ratio equal to 0.3, and a density equal to 7.85e-9 mt/mm^3. This simplification approach is chosen for users to create a similar model and gain confidence in using the bolting feature.

Three simulation models are generated (Threaded, Smeared, and Tied). The Threaded Model will include the thread detail. The Smeared Model and the Tied Model will not include the thread detail. The Smeared Model will include a small cylinder portion to represent the threaded region to be Smeared. The Tied Model will have the cylinder thread portion tied to the hole wall.

3. Threaded Simulation

Threaded simulation, as mentioned earlier, is the most accurate bolt simulation. The detailed result allows the user to simulate the behavior in a bolt as it would occur in a real application; however, for models with a high number of bolts, a threaded simulation can be computationally expensive and may not be practical to simulate. For applications that require an accurate result, the Threaded Simulation is the approach to use and well worth the extra time to solve.

For simplification, a two-dimensional (2D) axisymmetric approach is used. Figure 15 is a threaded bolt from Pro/Engineer.

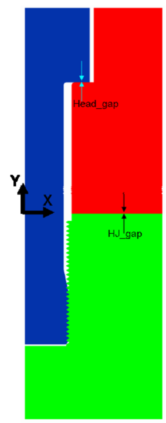

Figure 16. 2D Threaded Bolt Model.

The bolt is blue, the cover plate is red, and the base plate is green. At the contact interactions, there are sometimes gaps that must be accounted for. In Figure 15 the bolt head to cover plate intersection gap is identified by the label Head_gap. Similarly, the cover plate to base plate intersection gap is identified by the label HJ_gap. A zoomed in picture was needed to show the gap (Thread_gap) at the thread (Figure 16).

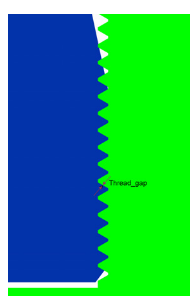

Figure 17. Zoomed View of 2D Threaded Bolt Model.

A detailed zoom shows the contact surface of the bolt identified by the red line and the plate contact surface identified by the lavender line, Figure 17.

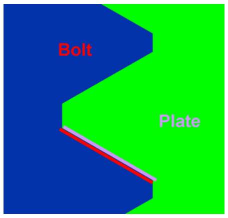

Figure 18. Thread Detail View.

Figure 18 shows the loads and boundary conditions.

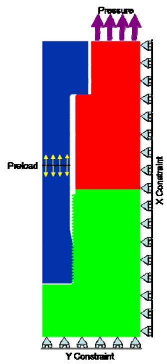

Figure 19. Loads and Boundary Conditions.

The Y constraint is to prevent rigid body motion. The X constraint is optional. It was applied to enforce the bolt to be local within the hole. Without the X constraint, the cover plate can bend inward and the base plate can bend outward. A pressure load that is slightly lower than the equivalent preload was applied. The preload applied was 2544690 N.

Figure 20 shows the overall mesh.

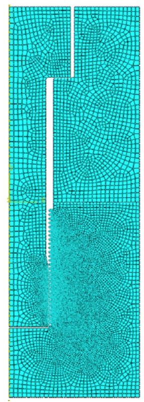

Figure 20. Overall Mesh.

Figure 21 shows a zoomed mesh to highlight the threaded region.

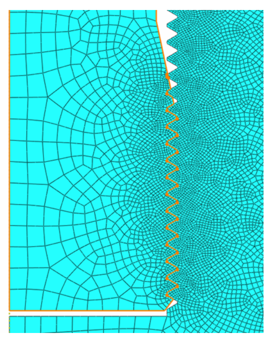

Figure 21. Zoomed Mesh.

Figure 22 shows the overall Y displacement.

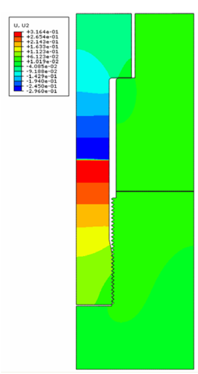

Figure 22. Threaded Bolt Y Displacement.

Figure 23 shows the overall SY stress.

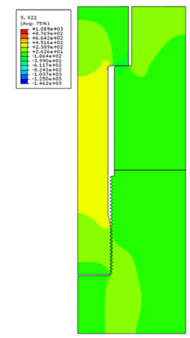

Figure 23. Threaded Bolt SY Stress.

Figure 24 shows the Von Mises stress.

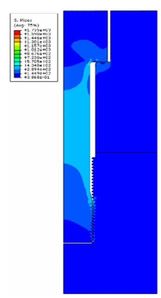

Figure 24. Threaded Bolt Von Mises Stress.

For comparative scaling purposes, the value of 745 MPa was selected. Figure 25 shows a contour plot of the scaled Von Mises stress.

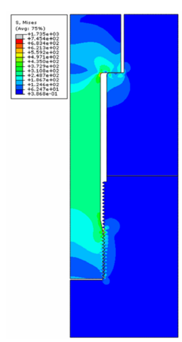

Figure 25. Threaded Bolt Scaled Von Mises Contour.

The thread area will be the more noticeable location; therefore, a zoomed in view of this area is shown in Figure 26.

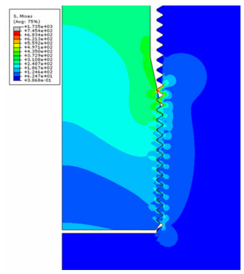

Figure 26. Threaded Bolt zoom in of Threaded Region.

As stated before, the Threaded Region Modeling is the most accurate approach for simulating bolt behavior.

4. Smeared Simulation

Smeared (thread interaction) gives thread-like simulation without using threads in the model. The thread behavior is internally calculated based on thread definition parameters from the user.

In Abaqus/CAE, the thread interaction parameters are defined by the pitch, thread half angle, and thread mean diameter. The maximum diameter can optionally be input instead of the mean diameter. If the maximum diameter is input, the mean diameter is internally calculated from the maximum diameter. Figure 30 shows a bolt with the primary parameters needed to define the thread in Abaqus/CAE. The Edit Interaction Dialog described earlier is also used to input the thread interaction parameters (Figure 31). The parameters are input from the clearance tab in the Edit Interaction Dialog. The other parameters described earlier on the Edit Interaction Dialog are the same. Specifically the contact interactions are possibly accounted for on the simulated thread surfaces; but, instead of using the defaults from the clearance tab, first, the single bolt or single bolt clearance must be selected from the dropdown. Now, other thread interaction edit parameter options become active. Simply input the appropriate values. For the example problem the pitch used is 6mm. The thread half-angle used is 30 degrees. And the maximum diameter is 120 mm. Another parameter required is thread height. This merely helps determine the number of threads to account for. Further investigations can be made to see how the answer changes when the value of the thread height.

All loads and boundary conditions are the same as described for the threaded bolt. The mesh density is fine as in the threaded bolt simulation and is not be shown. Also there was not a significant difference in the Y displacement plot. Therefore, it is not be shown for the Smeared Bolt Simulation. The smeared bolt SY stress is shown in Figure 27.

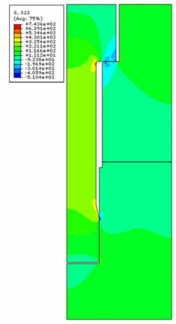

Figure 27. Smeared Bolt SY Stress.

Note that the difference between this plot and Figure 23.

The Von Mises stress is shown in Figure 28. Note that the maximum stress is the value used in threaded bolt simulation as the scaled value.

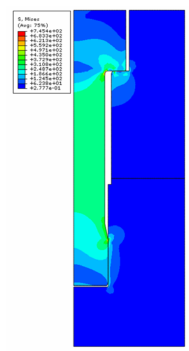

Figure 28. Smeared Bolt Von Mises Stress.

The zoomed in view of the area of threading is shown in Figure 29. The similarity to the Threaded Bolt Simulation is impressive.

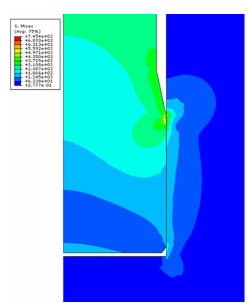

Figure 29. Smeared Bolt zoom in of Threaded Region.

The Smeared Method allows for the best of both worlds. It allows for the speed of Tied Thread Simulation with the accuracy as if the actual threads were modeled.

5. Tied Simulation

The Tied Area is the simulated threaded area approach. It provides the fastest solution of the threaded area approaches. This is an advantage for system level models. If local bolt stresses are not critical in the application, tied bolt simulation is the approach to use.

The set up is the same as the threaded region, except only one Edit Interaction Definition is needed. Also, from the Slave Node / Surface Adjustment tab, the tie adjusted surfaces checkbox should be selected. As with Smeared, the mesh density is fine and is not shown.

The Y displacement is shown in Figure 30.

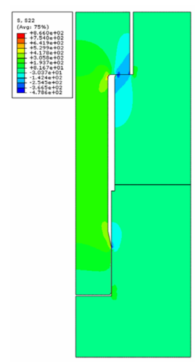

Figure 30. Tied Bolt SY Stress.

The Von Mises stress is shown in Figure 31.

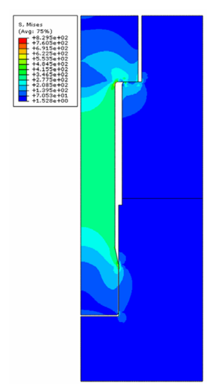

Figure 31. Tied Bolt Von Mises Stress.

The maximum value does not match that of the Smeared Bolt Simulation. Therefore, a scaled Von Mises stress value of 745 MPa was generated and is shown in Figure 32.

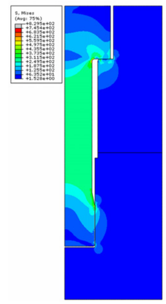

Figure 32. Tied Bolt Scaled Von Mises Contour.

The zoomed thread area of the Von Mises stress is shown in Figure 33.

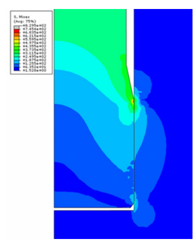

Figure 33. Tied Bolt zoom in of Threaded Region.

Tied Threaded Area is the most used solid simulation method because of the speed in solution when modeling a solid bolt.

6. Comparison

What is clear from the results is that Smeared should now become the most used solid simulation method, since those applications, where tied, was used require slight adjustment to get a significant accuracy improvement. To further this thought, a side by side visual comparison of the three threaded regions is shown in Figure 34.

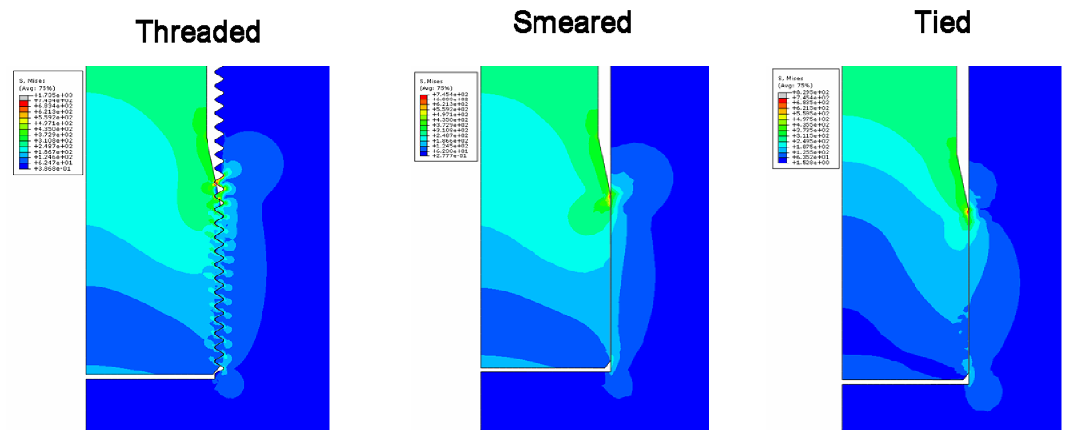

Figure 34. Threaded Region visual comparison.

Another comparison was to take a stress linearization line across the bolt shank at a location approximately one diameter below the top of the bolt hole (bolt head to cover interface). The value extracted is the Von Mises membrane stress. The values were Threaded = 291.72 MPa, Smeared = 290.07 MPa, and Tied = 290.07 MPa. This shows that the shank stress has about the same stress. The difference is in the local regions being the bolt head to cover interface and the threaded region.

7. Conclusion

Three bolt simulation methods were presented. Each bolt simulation method has advantages and disadvantages. Which bolt simulation method to use is dependent on situations such as time constraint, model size, or local accuracy desired. Another consideration is user modeling experience. This paper should have eliminated or at least reduced this concern. The steps shown applies to most bolt modeling situations for both 2D or 3D applications.

It becomes the judgment of the analyst as to which method to use.

In conclusion, the results show that Smeared will become more widely used than the other two simulations for most applications.

8. References

1. ABAQUS Release 6.7 Documentation.