

Methods for Modeling Bolts in the Bolted Joint

Jerome Montgomery

Siemens Westinghouse Power Corporation, Orlando, FL

Abstract

Modeling bolts for three-dimensional finite element applications have, and still continue to raise questions. The limitations on model size sometimes make modeling of solid bolts impractical. Therefore, many analysts choose other methods to model bolts. Line elements with coupled nodes and line elements with spider beams are a couple of alternative approaches. This paper looks at a few methods for modeling pretension bolted joints using the finite element method (ANSYS 5.7). Pretension is modeled using ANSYS pretension elements (PRETS179) which can be used on solid or line element types. Surface-to-surface contact elements are used to account for varying contact distribution along flanges. Bolt head and nut behavior is modeled by, coupled nodes, beam elements, rigid body elements (RBE3), or solids. Bolt stud is modeled by solid elements, beam elements, pipe elements, or link elements. The choice of line elements versus solid elements is determined by the degree of complexity sought. The pros and cons of different simulations are also discussed.

Introduction

Bolted joints are generally made up of the bolt group (head, stud, and nut) and the flange (top and bottom), as shown in Figure 1. Bolted connections are designed to hold two or more parts together to form an assembly (Figure 2). Because of different loading conditions, especially high loads, bolted connections can separate. To minimize this effect, a pretension is applied to the bolt (Figure 3). This insures that the connection will not separate, provided the applied load remains less than the pretension. In finite element simulation, the pretension characteristics must be accounted for. Fukuoka[1] developed a curve for simulating the pretension in a bolted joint system.

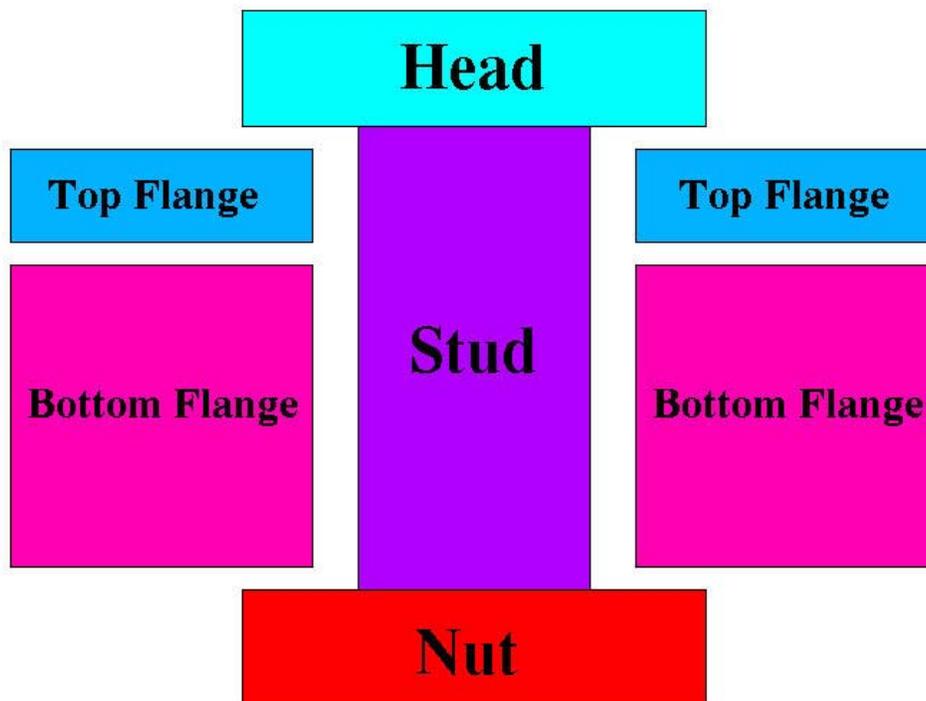


Figure 1 - Bolted Joint Labels

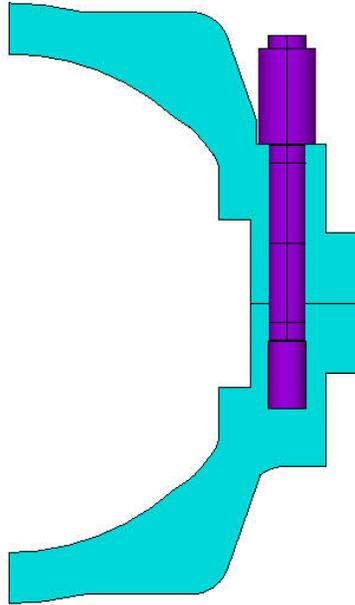


Figure 2 - Cylinder Section

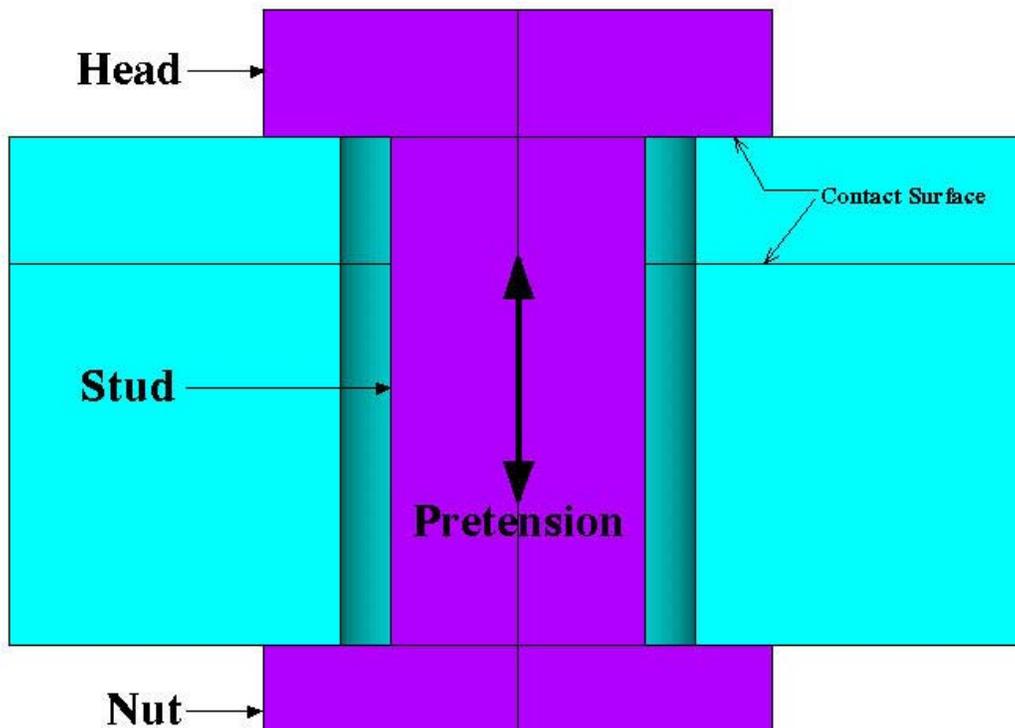


Figure 3 - Pretension

Before developing a finite element model, the analyst must determine the bolted joint characteristics to be modeled and understand the capability of the finite element program being used. With this knowledge, the analyst can determine how closely he could simulate the bolted joint. The bolted joint has many

complexities that are impractical to capture in production simulations. Several authors [2],[3],[4],[5],[6] cover the complexities associated with the bolted joint.

Two primary bolted joint characteristics are pretension and mating part contact (Figure 3 and Figure 4). Pretension and mating part contact capabilities are not available in all finite element codes. Therefore, workarounds are sometimes necessary. No single workaround has become the industry standard, but ANSYS pretension and contact elements have helped in modeling the above characteristics. Pretension can generally be modeled with temperature, constraint equations, or initial strains. Temperature pretension is generated, by assigning different temperatures and material properties to the bolt and the flange. The preset temperature helps in creating the thermal shrinkage effect in the bolt. Constraint equation pretension is a special form of coupling. Instead of coupling nodes, equations can be applied to direct the behavior of associated nodes. This too creates an initial displacement of the bolt. The pretension element in ANSYS uses the constraint equations approach. This is automated for the user. The user creates the element and applies the pretension load. Initial strain pretension is the more direct approach. In this approach, an initial displacement is applied to the element. Once the solution starts, the initial displacement is considered as a part of the load on the model.

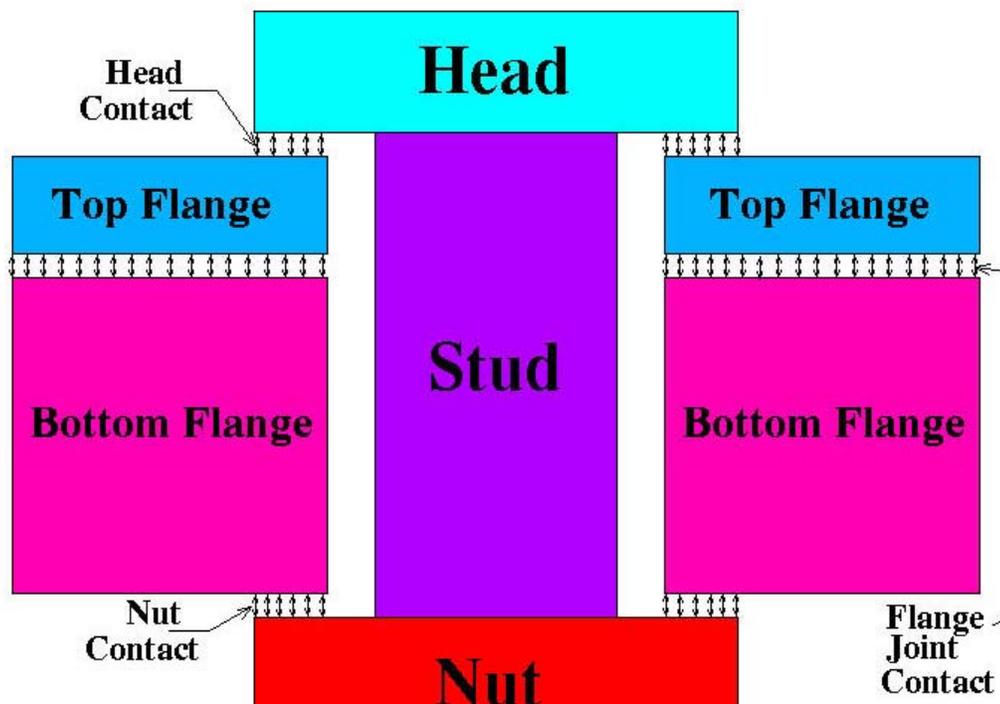


Figure 4 - Contact

Contact in the bolted joint is addressed using point-to-point, point-to-surface, or surface-to-surface elements. The contact type depends on the model being used. For solid three-dimensional modeling, the surface-to-surface contact is mostly used.

Bolt Under Flange Separation

When a load tries to separate a bolted flange joint, the job of the bolt is to hold the flanges together (Figure 5). The pretension should be more than the applied load. When the applied load exceeds the pretension, the part will separate. From a simulation standpoint, the surfaces that are in contact, must be able to separate. This is where the contact elements are used. For bolt under flange separation, the contact elements are not required for the contact surface between flange and head/nut of the bolt. These surfaces can be glued together. That is, the head contact can share the same surface as the top flange, and the nut contact can share the same surface as the bottom flange. The contact elements are required at the horizontal joint between the top and the bottom flange.

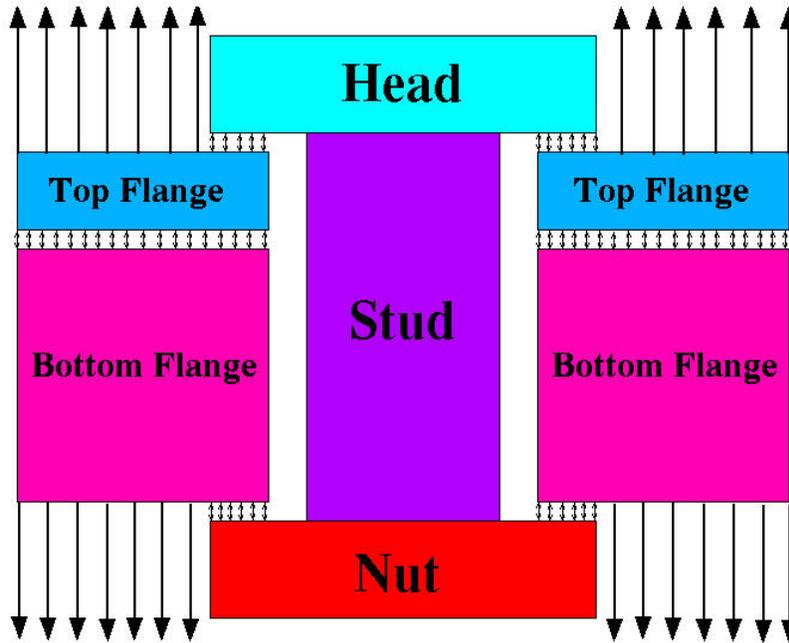


Figure 5 - Flange Separation

Bolt Under Flange Compression

When a flange is under compression, there is no load on the bolt (Figure 6). In this case, the head and nut contact must be able to separate from the flanges, whereas, the horizontal joint contact can share the same surface. The surfaces where the top flange and bottom flange meet (horizontal joint contact surface) can be glued together. Due to non-linear effects of contact elements, the above approach will help in saving considerable computation time during the solution phase.

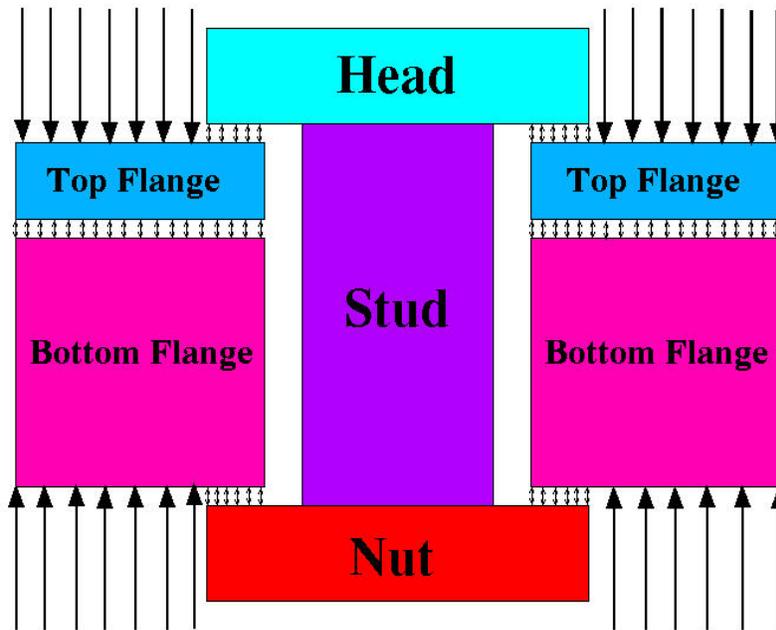


Figure 6 - Flange Compression

Transverse Direction

To incorporate the transverse effects of the bolt when two mating surfaces slide, a node at the bolt near the horizontal joint, and a node at the two mating surfaces are coupled to each other. Friction resists loads in the transverse direction. If we assume that the direction from the head to the nut is the vertical direction Z, then the transverse direction would be the direction X and direction Y. In solid models, the transverse loads are transferred from the bolt head/nut to the bolt. In non-solid head/nut simulations, other means are used to account for the transverse load as described later. Figure 7 shows the case with a solid head/nut assuming friction is ignored. To account for the transverse load, a node from the line element of the stud is coupled to a node from the top flange and the bottom flange.

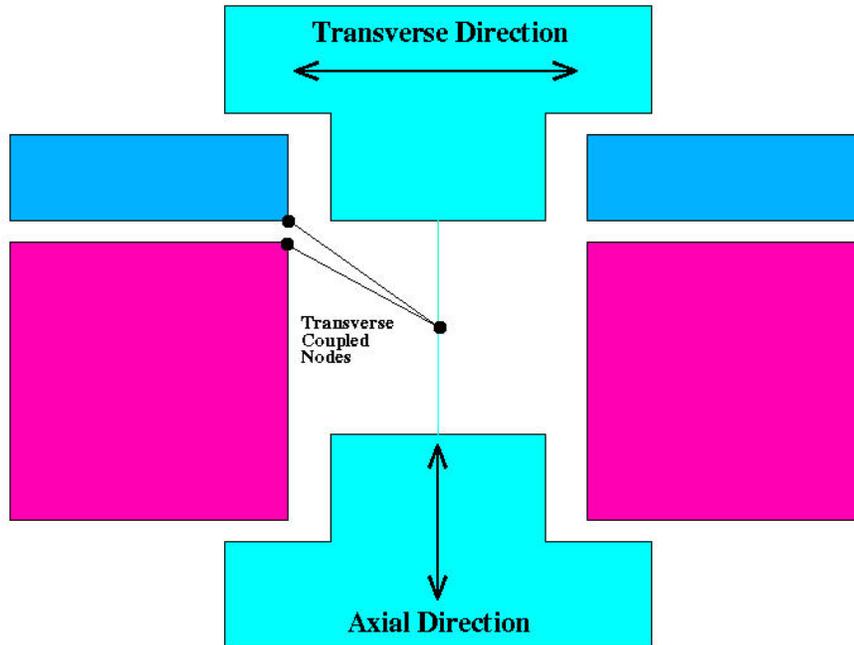


Figure 7 - Transverse Coupled at Flange Joint

Joint Simulations

When simulating a bolted joint, the analyst must account for both joint separation and compression in the model. Not accounting for one or the other is an engineering judgement, which implies knowledge of the load behavior. Adams and Askenazi [7] discussed different concerns in joint modeling. However, the analyst should know the type of results desired. This will determine the level of detail to be modeled.

In models for production applications, the bolt's function is to transfer the load from the top flange to the bottom flange, or vice versa, as the joint tries to separate. Many simulations ignore the bending and shear effects, but whether they should be included or not depends on the accuracy desired. The bolt must be sized to hold the joint together under the flange separation condition.

No Bolt Simulation

A no bolt simulation is when the pretension is applied as a pressure load on the washer surface without including a bolt in the model (Figure 8). It is the easiest and fastest way to account for bolt pretension effects in a model. The solution runs faster since there are fewer elements.

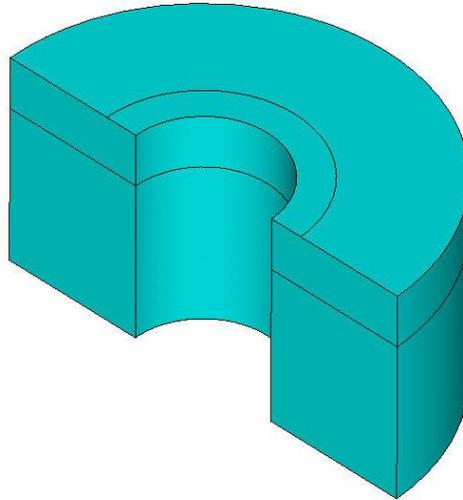


Figure 8. No Bolt

With no bolt simulation, the analyst assumes that the joint will not separate and the bolt stiffness is not required in the simulation [8]. But without bolt stiffness in the model, the bolt load transfer will not be taken into account. The pass/fail criteria will depend on the contact pressure and the gap, but not on the bolt load.

PROS

- Modeling of the bolt is not required
- In this simulation, the number of elements are fewest, hence the solution takes the least time
- The most simple approach to account for the bolt load

CONS

- Load is not transferred through the bolt as bolts are not modeled
- Bolt stiffness cannot be accounted for, as bolt elements are not modeled

Coupled Bolt

In Coupled Bolt simulation, line elements are used to represent the stud and coupled nodes represent the head/nut. The head/nut are connected similar to the spider bolt except with coupled nodes instead of line elements. Using this approach, the number of elements is significantly reduced (Figure 9). The Coupled Bolt simulation transfers vertical bolt loads without using line or solid elements. The stud is simulated as a Link10 element, which has tension only capability, requiring no contact elements at the head/nut flange connection.

PROS

- In coupled bolt simulation, the number of elements is more than no bolt simulation but fewer as compared to other simulations
- Simple stud section using line elements
- Ease in extracting results
- Tensile loads can be transfer through coupled nodes

- If flange goes into compression, Link10 with tension only capability will respond as an actual bolt

CONS

- Head/Nut temperature is not accounted for
- Bending loads are not transferred

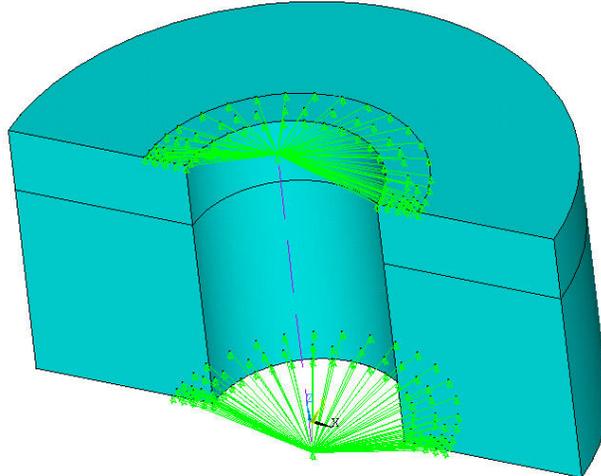


Figure 9 - Coupled Bolt

RBE (Rigid Body Element) Bolt

In RBE Bolt simulation, line elements are used to represent the stud and RBE elements are used to represent the head/nut. The head/nut are connected similar to the spider bolt with RBE elements instead of line elements. Using this approach, the number of elements is significantly reduced (Figure 10). The RBE Bolt simulation transfers all the loads and incorporates the bending effects without using line or solid elements.

A portion of the stud line elements should be line elements with tension only capability, since no contact elements are used at the head/nut to flange connection.

PROS

- In RBE bolt simulation, the number of elements is more than no bolt simulation but fewer as compared to other simulations (the number is similar to the coupled bolt)
- Simple stud section using line elements
- Ease in extracting results
- Tensile, bending, and thermal loads can be transferred through the RBE nodes

CONS

- Head/Nut temperature is not accounted for
- If flange goes into compression, bolt will compress when using line element as stud unless line elements, such as Link10s are used

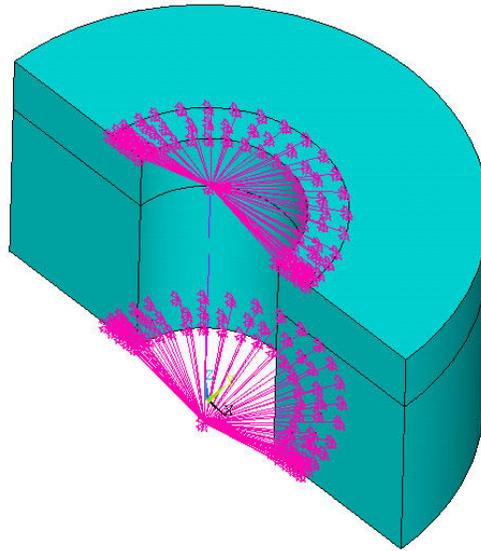


Figure 10 - RBE Bolt

Spider Bolt

The spider bolt simulation substitutes line elements for the head, nut, and stud (Figure 11). A series of line elements represent the head/nut in a web-like fashion. Thus, the name spider bolt. It is the most logical approach to using line elements and transferring the loads to the stud. The head/nut bending and stiffness must be simulated by the line elements.

A portion of the stud line elements should be line elements with tension only capability, since no contact elements are used at the head/nut to flange connection.

PROS

- In spider bolt simulation the number of elements is more than no bolt, coupled bolt, and RBE bolt simulations, but fewer as compared to hybrid bolt and solid bolt simulations
- Simple stud section using line elements
- Ease in extracting results
- Tensile, bending, and thermal loads can be transferred through the spider elements

CONS

- Extra work required for simulating head/nut stiffness as compared to other simulations
- If flange goes into compression, bolt will compress when using line element as stud unless line elements, such as Link10s are used

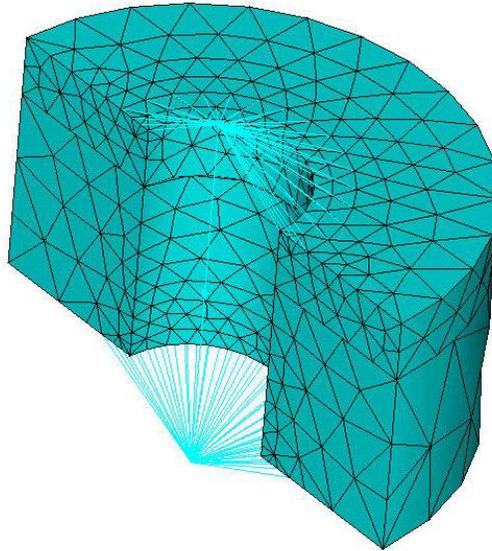


Figure 11 - Spider Beam Bolt

Hybrid Bolt

In the hybrid bolt simulation, the head and the nut are modeled as solid elements and the stud region is modeled as a line element (Figure 12). It is recommended that the line element starting point should be located one half diameter from the top flange edge and one half diameter from the bottom flange edge. The line element captures the tensile part of the bolt load. The line element is attached to the solid using coupled nodes. They are coupled in the bolt's axial direction. In the hybrid bolt simulation, the purpose in keeping the head and nut as solid elements is to incorporate the thermal and bending load effects. The contact elements between flange and head/nut are not required if Link10 elements are used as the line elements. This is because the Link10 elements have the tension only option. That is, if the bolt goes into compression, there is no load in the Link10 element. Link10's reduces the number of contact elements, but it is required to couple the nodes at the top and bottom flange. Transverse effects are as described in the section on transverse direction. If a degree of freedom problem occurs, it is required to restrain the Link10 elements in the transverse direction. If Beam4 elements are used in place of Link10 elements, it will eliminate the transverse coupling requirement, but it will be required to model the contact elements between flange and head/nut to include the zero compression, which is not available in Beam4 elements.

PROS

- Best simulation approach for accuracy after solid bolt simulation
- Simple stud section, modeled using line elements
- Ease in extracting results
- Tensile, bending, and thermal loads can be transferred through the line elements
- Full stress distribution in head and nut can be calculated
- If flange goes into compression, Link10 with tension only capability will respond as an actual bolt

CONS

- As stud section is modeled as a line element, no full visual stress distribution through the stud section
- Coupling of line elements to stud is required

- Requires contact elements at head/nut if beam4 line elements are used as stud section

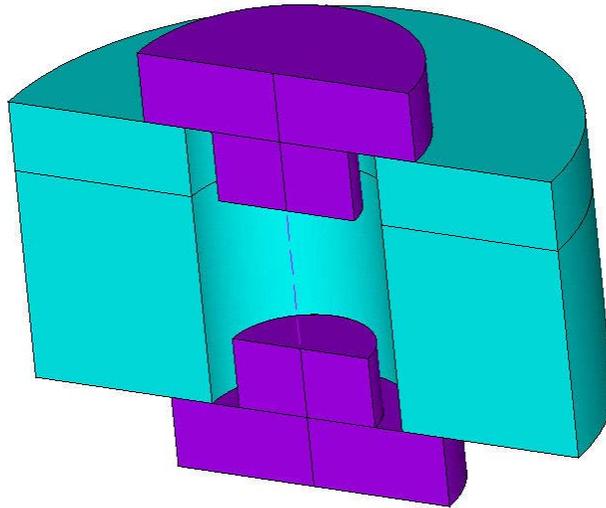


Figure 12 - Hybrid Bolt

Solid Bolt

The solid bolt is the most realistic simulation approach used for modeling a bolt. It captures thermal, bending, and tensile loads. Therefore, it is the best approach to use.

The solid bolt simulation requires that the contact elements be used for the horizontal joint and the contact surface between the flange and the head/nut (Figure 13). Pretension should be accounted for, by using the pretension element.

The solid bolt is the closest simulation of the actual bolt but there are some characteristics that are ignored to simplify the simulations. One of these characteristics is the effect of threads, which is not considered in the bolting analysis that will follow. Friction interaction, at the contact surfaces, is another characteristic, which is not considered for the following bolting analysis.

PROS

- Best simulation approach for accuracy
- Tensile, bending, and thermal loads can be transferred
- Full stress distribution in head, nut and stud can be calculated

CONS

- Adds more modeling and run time due to number of solid elements required
- Requires extra effort to calculate the stud cross-section stresses
- Contact elements required at head and nut to flange

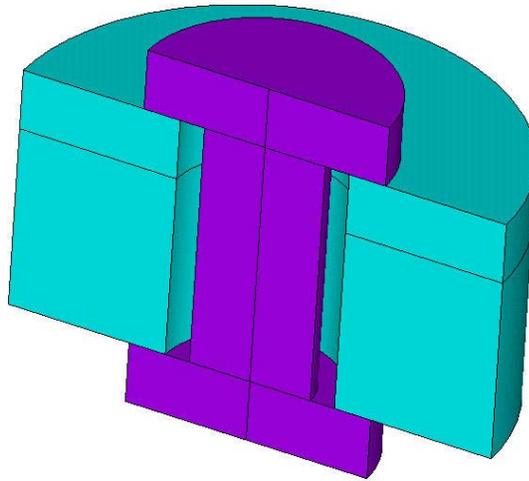


Figure 13 - Solid Bolt

Analysis

An analysis of each simulation is made to compare results. The intent is to show that a reasonable solution is obtained with either approach.

The model used was from the sample problem in reference [9]. The dimensions, shown in Table 1 are based on the labels in Figure 1. The bolt modulus of elasticity is 206842.7 MPa (3e7 psi) and the modulus of elasticity of the top and bottom flanges is 68947.57 MPa (1e7 psi).

Item	Dimension mm (inches)
Head Height	6.35 (0.25)
Nut Height	6.35 (0.25)
Stud Height	25.4 (1.00)
Top Flange Thickness	6.35 (0.25)
Bottom Flange Thickness	19.05 (0.75)
Head Diameter	25.4 (1.00)
Nut Diameter	25.4 (1.00)
Stud Diameter	12.7 (0.5)
Top Flange Outside Diameter	50.8 (2.00)
Top Flange Inside Diameter	17.78 (0.70)
Bottom Flange Outside Diameter	50.8 (2.00)
Bottom Flange Inside Diameter	17.78 (0.70)

Table 1

The boundary conditions were controlled to obtain as much consistency as possible for each simulation (Figure 14). A pressure of 0.293 MPa (42.445 psi) was applied with a 444.82 N (100 lb_f) pretension. The applied pressure was calculated from a pressure of 0.345 MPa (50 psi) divided by an area of 760 mm² (1.178 in²).

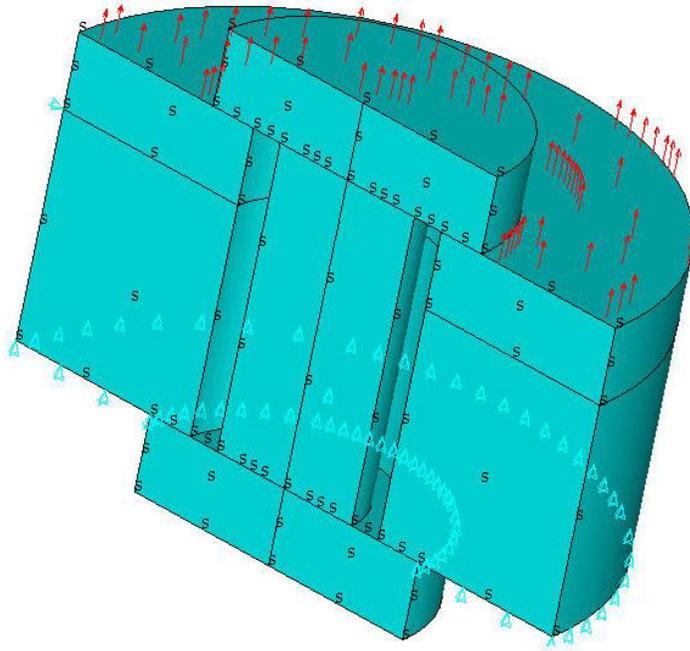


Figure 14 - Boundary Conditions

A half model was used so that one can see the cross-section of the bolted joint. Symmetry boundary conditions were used at the half cut. The bottom surface was fixed in the vertical direction. A side node was fixed to avoid transverse displacement. Figure 15 shows the mesh of the solid bolt model.

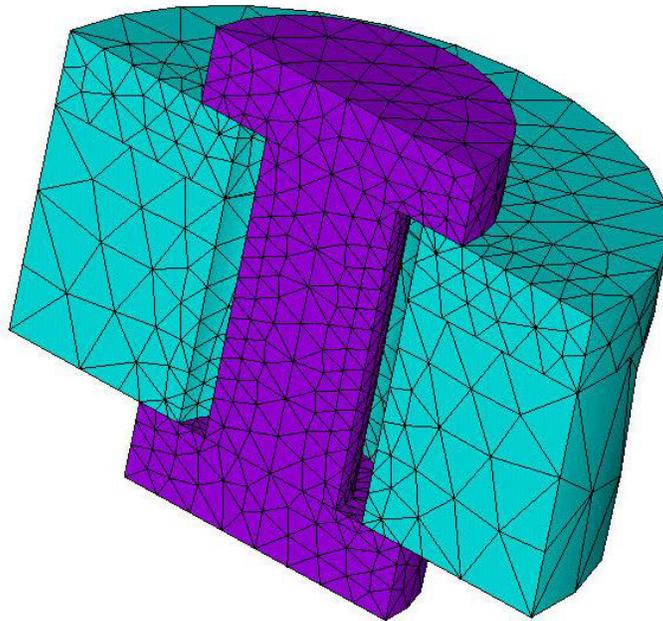


Figure 15 - Solid Bolt Mesh

Table 2 shows the modeling that was necessary for the specific bolting simulation.

Simulation Type	Modeling Adjustments
No Bolt	Pressure was applied to the washer surface to simulate the pretension.
Coupled Bolt	The nodes on the edge of the nut region needed to be released from the vertical displacement constraint. If not released, an error message occurred due to those nodes being coupled in the vertical direction. One half of the head, nut, and stud areas were used as the real constant. Link10 elements used as head, nut, and stud.
RBE Bolt	Pipe16 elements were used as head, nut, and stud. Using one half of the head, nut, and stud areas, the diameter was back calculated, to be input, as real constants. $[d = \sqrt{4 \cdot A / \pi}]$
Spider Bolt	Pipe16 elements were used as the head, nut, and stud. Using one half of the head, nut, and stud area, the diameter was back calculated, to be input, as a real constant. $[d = \sqrt{4 \cdot A / \pi}]$.
Hybrid Bolt	Link10 elements were used as the stud. Contact elements used at the head and nut to flange intersection.
Solid Bolt	Contact elements used at the head and nut to flange intersection.

Table 2

Analysis Results & Discussion

Table 3 and Figure 16 show analysis results of the finite element simulations. The tabulated data is the outside edge gap on the opposite end of the transverse constraint.

No Bolt	Couple Bolt	RBE Bolt	Spider Bolt	Hybrid Bolt	Solid Bolt
4.938E-3	1.842E-3	3.388E-3	1.389E-3	1.821E-3	1.459E-3

Table 3

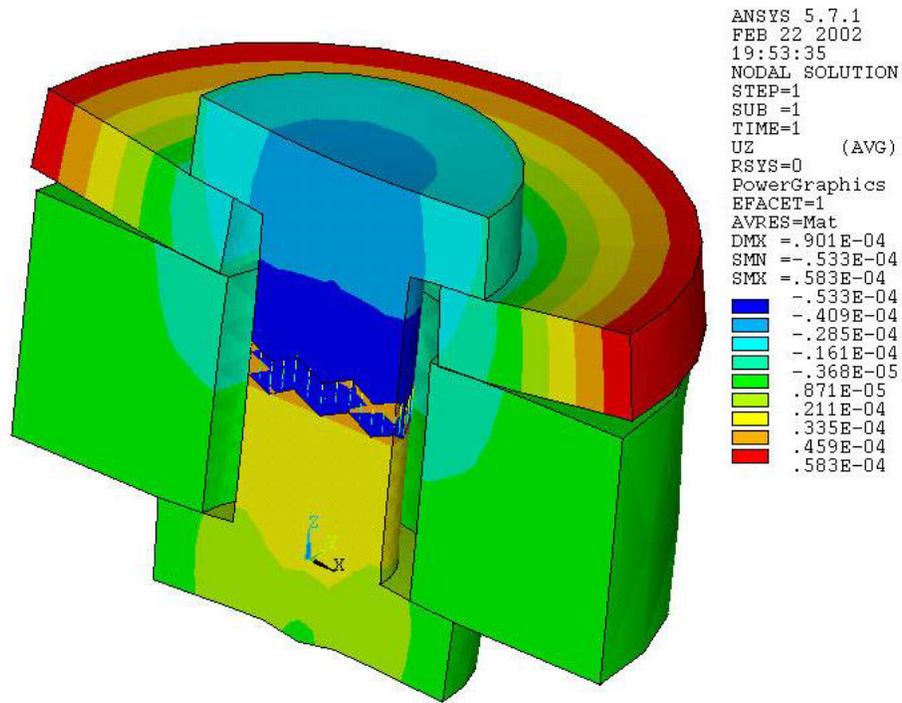


Figure 16 - Solid Bolt Displacement

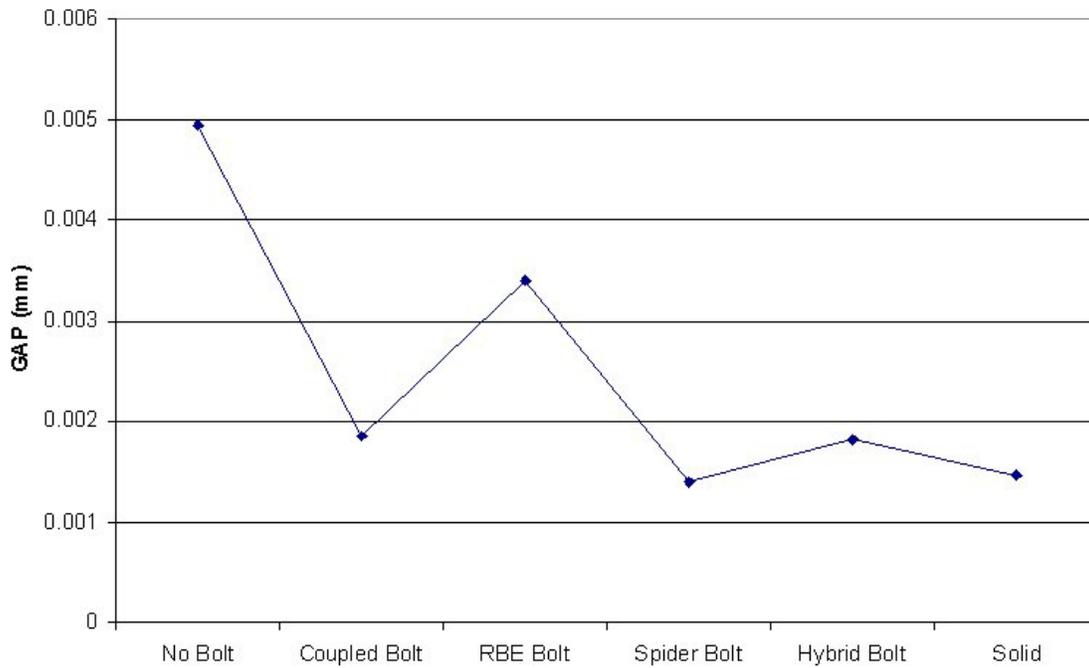


Figure 17 - Gap

The results show that there is a bolt stiffness correlation from no bolt simulation to a solid bolt simulation. Two bolt simulations that skew the correlation, are the coupled bolt simulation and the spider bolt

simulation. Modeling necessities influenced the results of the coupled bolt simulation. For example, the coupled bolt required removal of the vertical constraint at the washer face edge. The spider bolt simulation is influenced by how the user simulates the head and nut. Two key parameters in simulating the head and nut for the spider bolt simulation are the number of nodes to connect to and the real constant values to use.

If you look at the No Bolt, RBE Bolt, Hybrid Bolt, and Solid Bolt, simulations you see a trend of the gap displacement.

Conclusion

The results show that any of the joint simulations can be used, as long as, the analyst understands the limitations of the specific bolt simulation used. In all simulation some details will be ignored. It is up to the analyst to decide if the amount of information simulated will capture the intended results.

References

- 1) T. Fukuoka, Analysis of the Tightening Process of Bolted Joint With a Tensioner Using Spring Elements, Journal of Pressure Vessel Technology, November 1994, Vol. 116, pgs. 443-448.
- 2) Bickford, John H., An Introduction To The Design and Behavior of Bolted Joints, 3rd edition
- 3) Shigley, Joseph E., Mechanical Engineering Design, McGraw-Hill, 1977, 3rd edition
- 4) Norton, Robert L., Machine Design – An Integrated Approach, Prentice-Hall: New Jersey, 1998, 2nd printing
- 5) Levinson, Irving J., Machine Design, Reston Publishing: Virginia, 1978, 1st printing
- 6) Spotts, M. F., Design of Machine Elements, Prentice-Hall: New Jersey, 1978, 5th edition
- 7) Adams, Vince, and Askenazi, Abraham, Building Better Products with Finite Element Analysis, OnWord Press: New Mexico, 1999, 1st printing
- 8) VDI 2230, Systematic Calculation of High Duty Bolted Joints with One Cylindrical Bolt, October 2001
- 9) ANSYS Basic Analysis Guide section 2.9 “Defining Pretension in a Joint Fastener”, ANSYS Software Revision 5.7.1, ANSYS Inc., 2002