Modeling Diesel Engine Cylinder Head Gaskets using the Gasket Material Option of the SOLID185 Element

Jonathan Raub

Cummins, Inc.

Abstract

The modeling of diesel engine cylinder head gasket joints is complicated by the nonlinear response of the head gasket's materials. Linearization of these material responses can lead to significant errors in the solution's results. The 1-dimensional nonlinear approximation made by the Gasket material option of the SOLID185 element sufficiently captures the response of the nonlinear gasket materials while maintaining practical solution times for the large model sizes associated with multi-cylinder head gasket joint models. This paper will give an overview of the model building and assembly process used to create a head gasket joint model, describe the nonlinear nature of the materials used in the model, and present comparisons of the model's results with experimental measurements.

Introduction

Previous analyses of cylinder head gasket joints using Ansys employed a process in which the nonlinear responses of the head gasket materials were linearized to allow the use of an elastic modulus (Reference 1). While moderately effective, this process required the material response to be fairly linear over the range of load experienced by the gasket. Also, this technique captured the global response of the joint, but the linear material assumption limited the model's ability to accurately predict load variation around the gasket, especially in lightly loaded regions.

These limitations motivated the creation of a Gasket material option for the SOLID185 element. Although limited to a 1-dimensional response (through the thickness of the gasket), this material option allows the user to define an arbitrary nonlinear loading curve, and various load-dependent nonlinear unloading curves. Initial clearance and separation can be incorporated into the response of the material, consequently eliminating the need for contact elements.

The head gasket joint model used as an example in this paper is of a Cummins ISX diesel engine. The ISX is a 15 liter, 600 horsepower, in-line 6 cylinder engine that is primarily used in heavy duty on-highway automotive applications, and is shown in Figure 1. The head gasket for this engine is shown in Figure 2. Figure 2 also identifies the primary components of this head gasket that will be referred to in the remainder of this paper. This engine has a replaceable cylinder liner that is supported vertically by a seat that is roughly half way down its length. This type of liner is called a "midstop" liner. Some diesel engines use a liner that is supported at the top, and are consequently referred to as "topstop" liners.

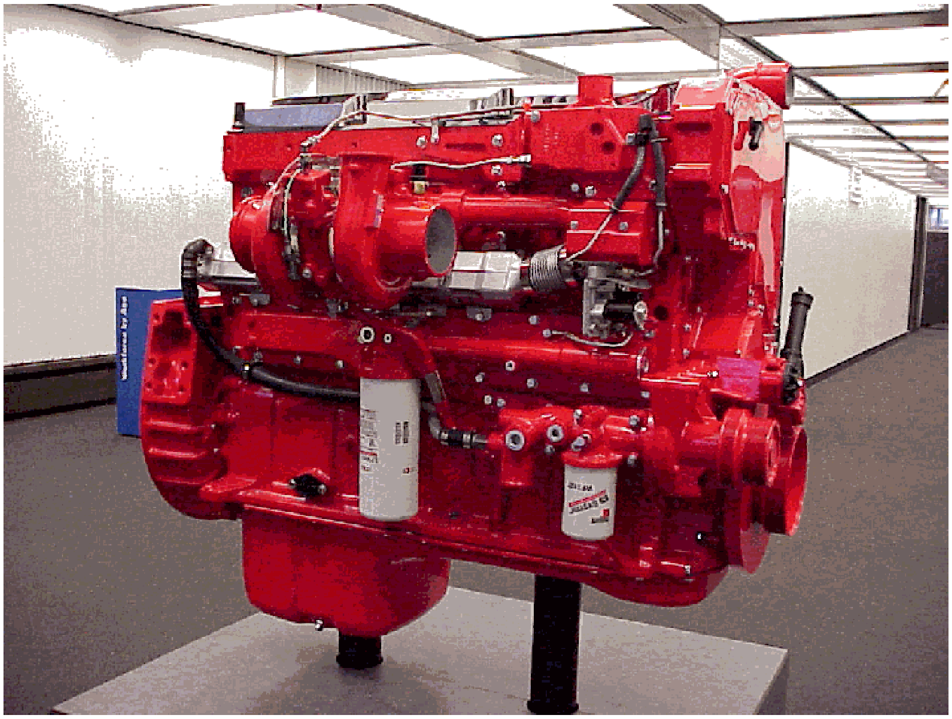


Figure 1 - Picture of Cummins ISX 15 Liter Engine

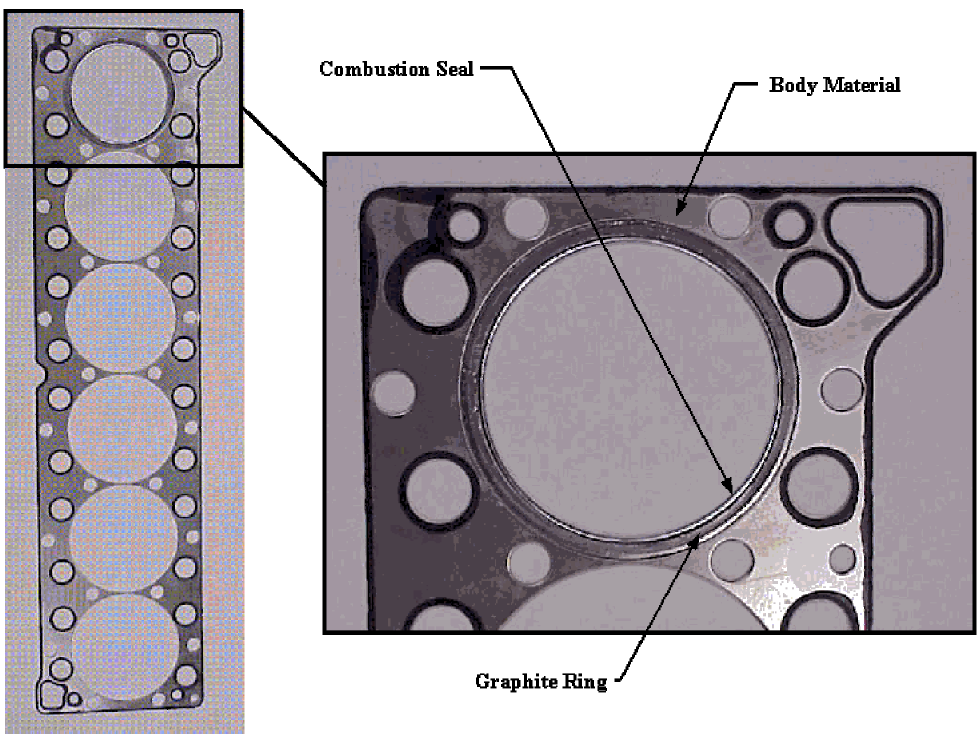


Figure 2 - Picture of ISX Engine's Cylinder Head Gasket

Model Description

Accurate modeling of head gasket joints requires a complex assembly of engine components. Special attention must be paid to the meshes at mating interfaces, as well as to the methods used to connect the components. This section will give an overview of the process used to create and connect the various components in a head gasket joint model.

A head gasket joint model includes the cylinder head, head gasket, liners, cylinder block and head capscrews. An exploded view of the ISX head gasket joint model is shown in Figure 3. A plot of the head gasket model's components is shown in Figure 4. One of the most complicated steps in creating this head gasket model involved forcing matching nodal patterns at all component interfaces. This is an important feature because the use of generalized contact, when combined with the nonlinear material response of the gasket components, would drive excessive solution times for a model of this size.

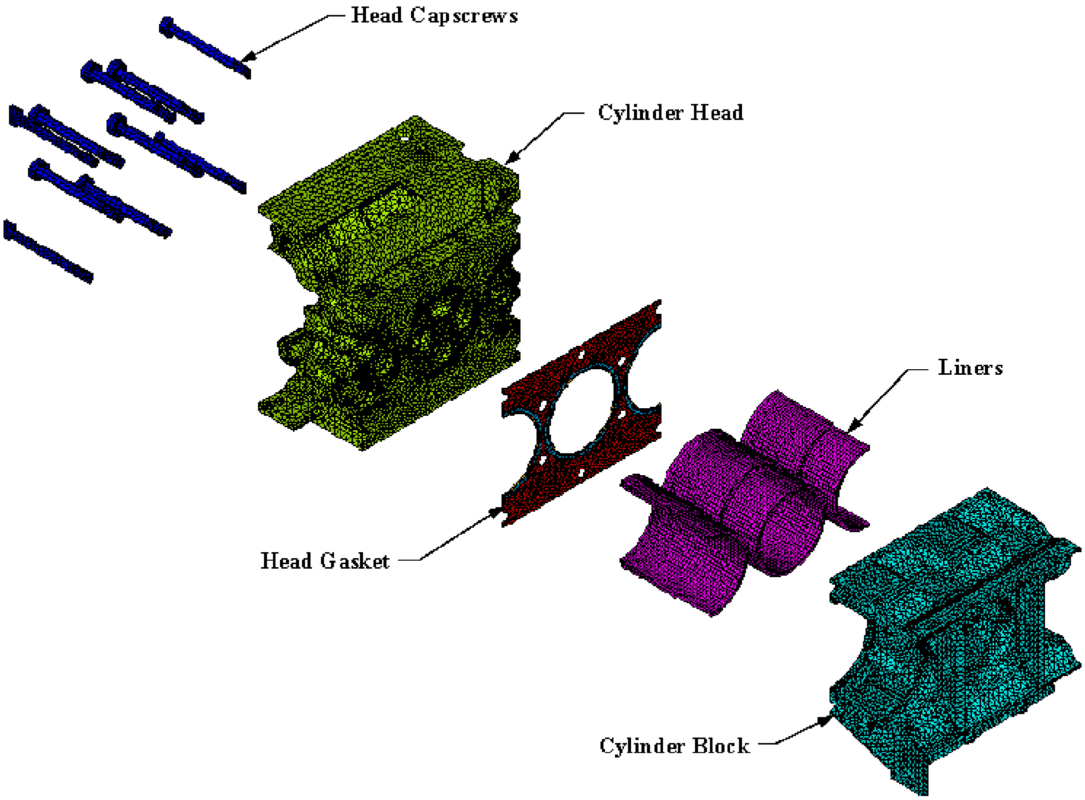


Figure 3 - Exploded View of ISX Cylinder Head Gasket Joint Model

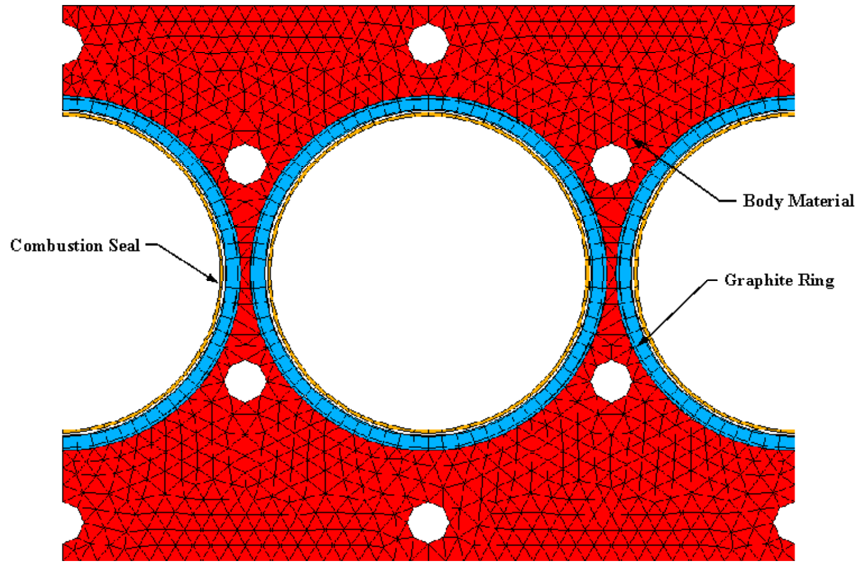


Figure 4 - Plot showing Components of Head Gasket

Solid Model Modification

The process of aligning nodal patterns begins in the solid modeling package, which was Pro/Engineer in this case. First, the models were cut down to the ½-1-½ cylinder sections. This model configuration was chosen because it allows the simulation of an infinitely long engine firing every other cylinder when symmetry is applied to both ends and the center cylinder is fired. This is the minimum model size allowable for accurate results for the head gasket during a cylinder firing event. The cylinder block was then cut below the midstop support feature.

Next, the models of the cylinder head and cylinder block were modified so that every feature on the block top deck matched with the corresponding feature on the head bottom deck. The head capscrew holes in the cylinder block were modified to give a cylindrical area that equaled the length of the thread engagement at the pitch diameter of the capscrew threads at the appropriate depth. Annular protrusions were added to the bottom deck of the cylinder head to give concentric circular lines where the combustion seal, graphite ring, and body material contact (see Figure 5). A similar protrusion was added to the top deck of the block for the overlapping edge of the graphite ring (see Figure 6). Circular protrusions were added to the top deck of the head to give areas that would later be joined to the head capscrews. These protrusions were deleted after the solid models were brought into Ansys, leaving line geometry that was needed to create areas that match between components that contact each other.

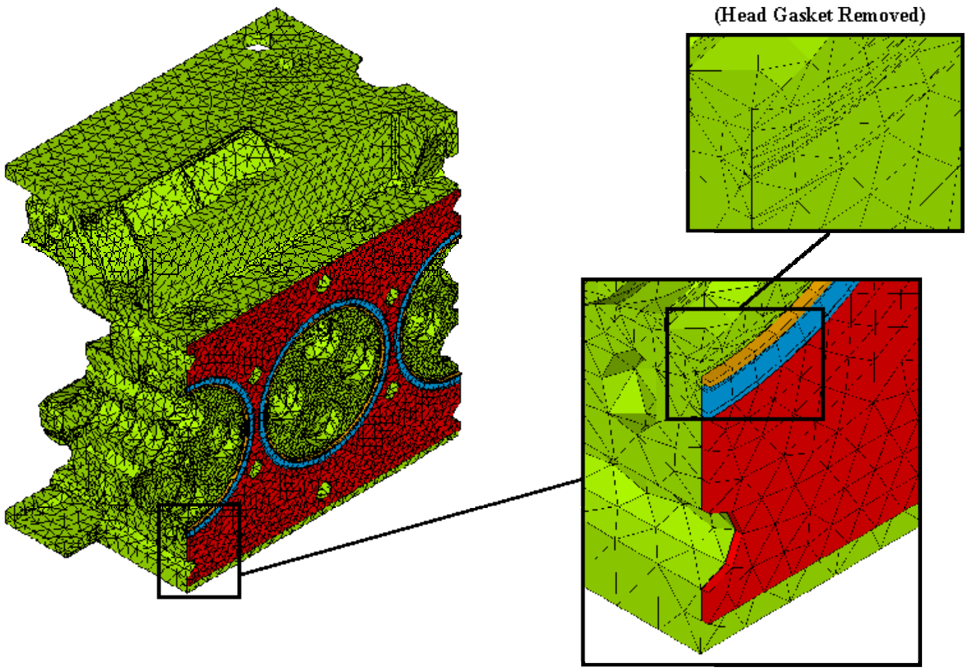


Figure 5 - Isometric View showing Cylinder Head and Head Gasket

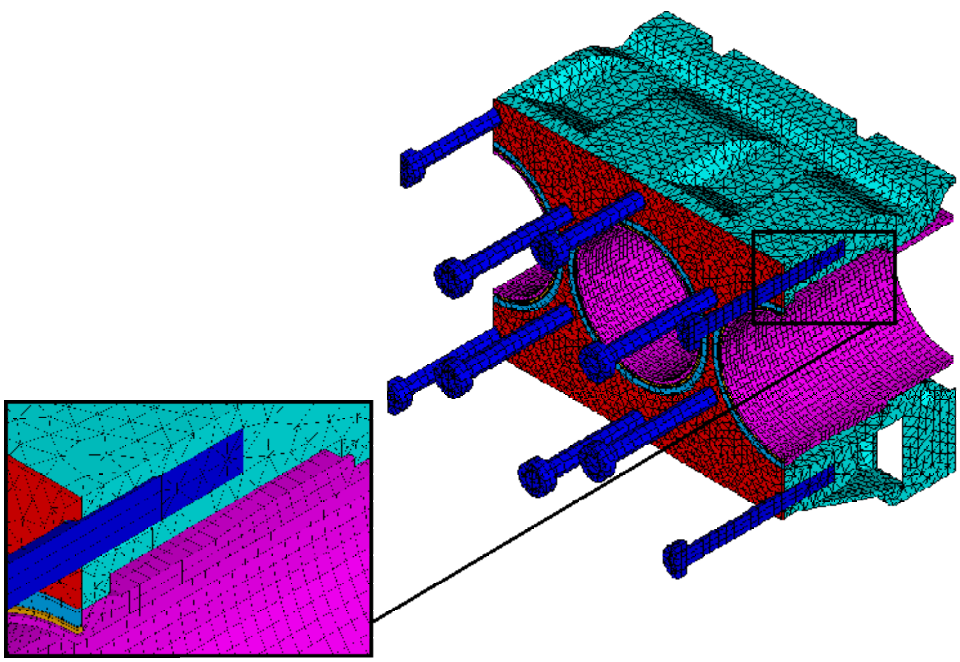


Figure 6 - Isometric View showing Cylinder Block, Liners, Head Capscrews and Head Gasket

Model Creation in Ansys

The solid model of the cylinder head was imported into Ansys using the Direct Translator. The bottom deck protrusions added in Pro/Engineer were deleted, and planer areas were created using the lines left by the protrusions. These bottom deck areas were then meshed with triangular shells. Areas that contact the combustion seals and graphite rings were meshed with herringbone patterns of triangles so that they would match the brick-meshed nodal patterns of the gasket components that were created later (see magnified view in Figure 5 with head gasket removed). This process leaves a mid-side node on the diagonal element edges that does not have a matching node in the middle of the brick face, but testing revealed that this mesh inconsistency did not result in significant errors in contact stress.

The area on the bottom deck of the head that matches the body material was then copied downward with the area mesh to create the top surface of the body material. This meshed area was then extruded to create the gasket body volume and wedge-shaped volume elements. The liners, combustion seals and graphite rings were constructed by creating a 2-D cross-section that was meshed with quadrilateral shells, and then revolved to create the volumes and brick elements. The combustion seal was modeled with a rectangular cross-section even though it is round in reality. The stiffness increase resulting from increasing contact width as the seal is crushed from round to oval is modeled in the nonlinear response of the material, not in the shape or contact of the combustion seal.

Next, the solid model of the cylinder block was imported using the same method as was used for the cylinder head. The areas on the bottom deck of the head that match the top deck of the block were copied downward with the triangular area mesh and merged into the cylinder block volume. The capscrews were then built by hand using the areas specially created in the Pro/Engineer models at the capscrew head and threads. The areas in the cylinder block that contact the liner at the top pressfit and midstop seat were meshed with herringbone patterns of triangular shells. The final step was to volume mesh the cylinder head and cylinder block. Refer again to Figures 5 and 6 to see the resulting model. The completed model had approximately 200,000 elements and 400,000 nodes.

Model Constraints and Connections

First, all components were constrained in the normal direction on the two end faces at the cut cylinder sections to represent symmetry. The cylinder block was then constrained vertically on the lower cut plane, and on a single node in the horizontal direction to prevent rigid body motion. The liners were coupled to the cylinder block vertically at the midstop. Radial constraint equations with the pressfit incorporated into the equation were created at the liner to cylinder block interface at the top of the liner. Liners were also coupled to the cylinder block with one node in the tangential direction to remove rigid body motion. The combustion seals, graphite rings, and body material were coupled in all 3 directions to the cylinder head, cylinder block and liners. A single node on the cylinder head was constrained in the horizontal direction to prevent rigid body motion. The capscrews were created as merged to the cylinder head and cylinder block, so no further coupling was required. Finally, a pre-tension section was created for each of the head capscrews using PRETS179 elements.

Gasket Material Response

The Gasket material option should be used in a macroscopic investigation of how gasketed joints behave. The focus should be on the behavior of the joint, not on a detailed study of the gasket. If detailed gasket results are desired, more sophisticated material models will be required. The Gasket material option does not predict the response of the gasket, but rather allows the user to input the response of the material. Consequently, the nonlinear response of the gasket materials must first be quantified experimentally.

The stress vs. deflection response in the through-thickness direction is the response that is entered in for each nonlinear gasket material. Even though the SOLID185 element is 3-dimensional, the Gasket material option gives stiffness in the through thickness direction only. This must be understood when the model is constrained, as the gasket elements have no stiffness in the in-plane directions. This is why the single constraint in the horizontal direction was applied to the cylinder head.

Combustion Seal Response

The experimental response for the combustion seal is shown in Figure 7. This figure also shows the multi-linear loading and unloading responses that were input to Ansys. The combustion seals are made from steel, and initially have a round cross-section. The nonlinear response is caused by the round section plastically deforming into an oval-shaped section during assembly. This plastic deformation allows this high-pressure seal to conform to the shape of the cylinder head and liner locally. The unloading response of the combustion seal is essentially the elastic recovery of the deformed cross-section. The material response is entered into Ansys as a single loading curve and multiple unloading curves. Each unloading curve must begin at a defined point on the loading curve. All three of the unloading curves in this example have a final data point at zero stress and zero deflection. This allows the joint to "separate" locally, by having the gasket elements go to near-zero stiffness, without the use of contact elements.

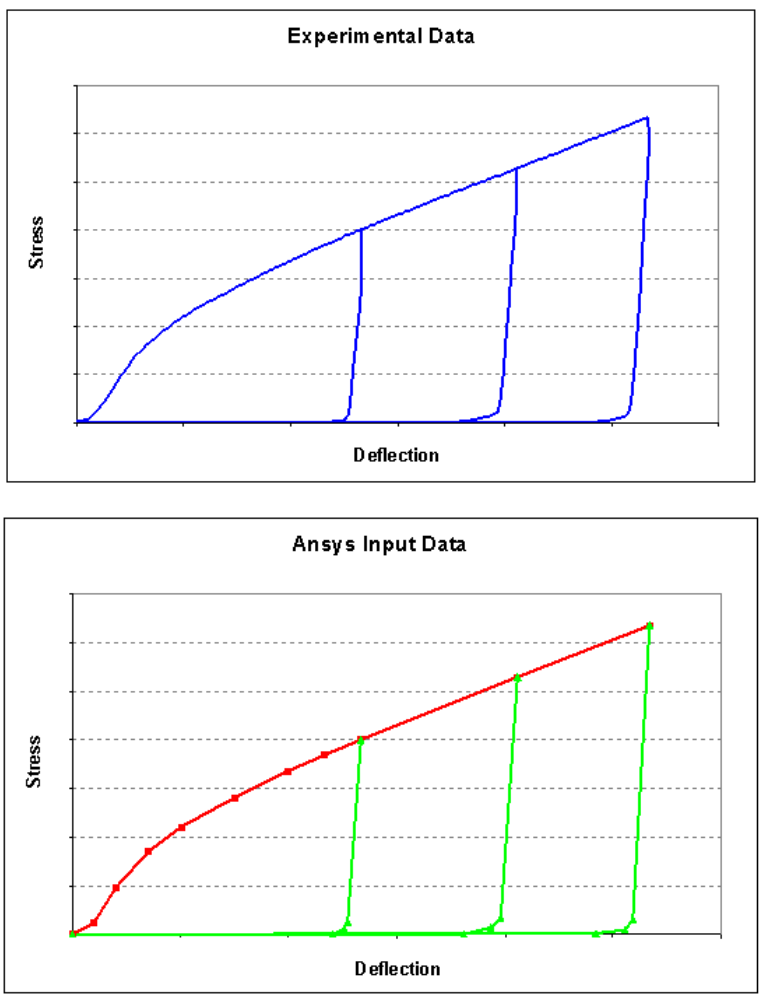
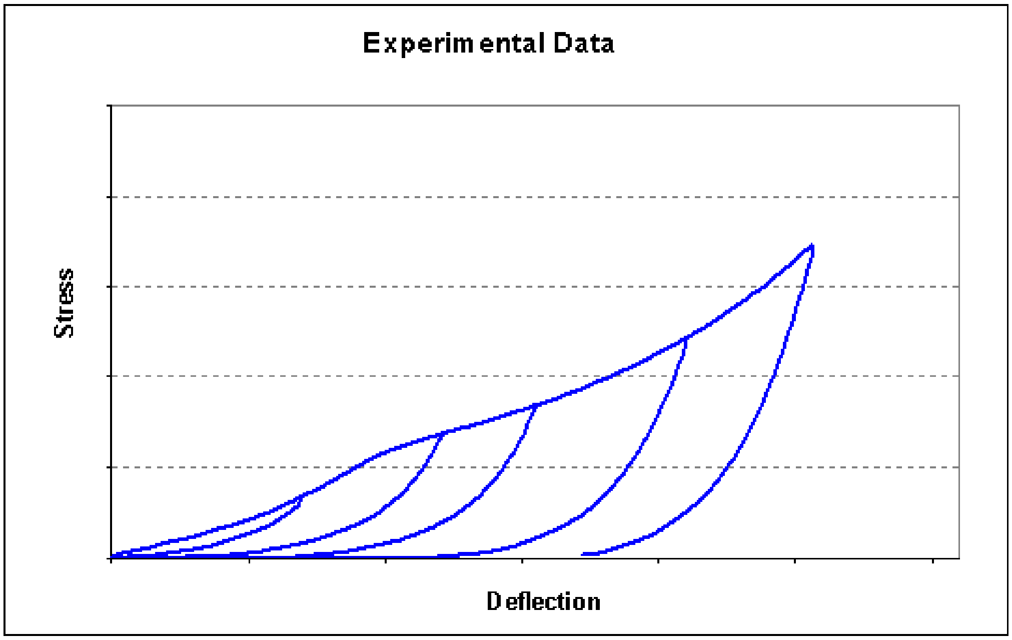


Figure 7. Stress vs. Deflection Responses for the Combustion Seal

Body Material Response

The experimental and analytical approximation of the response of one body material is shown in Figure 8. This body material is comprised of graphite layers that are mechanically bonded to a steel carrier. The nonlinear response is caused by the densification of the graphite layers during compression. The unloading responses are highly nonlinear, and significantly load-level dependent. The strength of the Gasket material option is displayed in its ability to model this complex response. Data is entered into Ansys in the same way as was done for the combustion seal. To aid model convergence, the loading response was extended beyond the level to which the test sample was loaded. This becomes important in solutions that initially over-predict the load on portions of the body material.



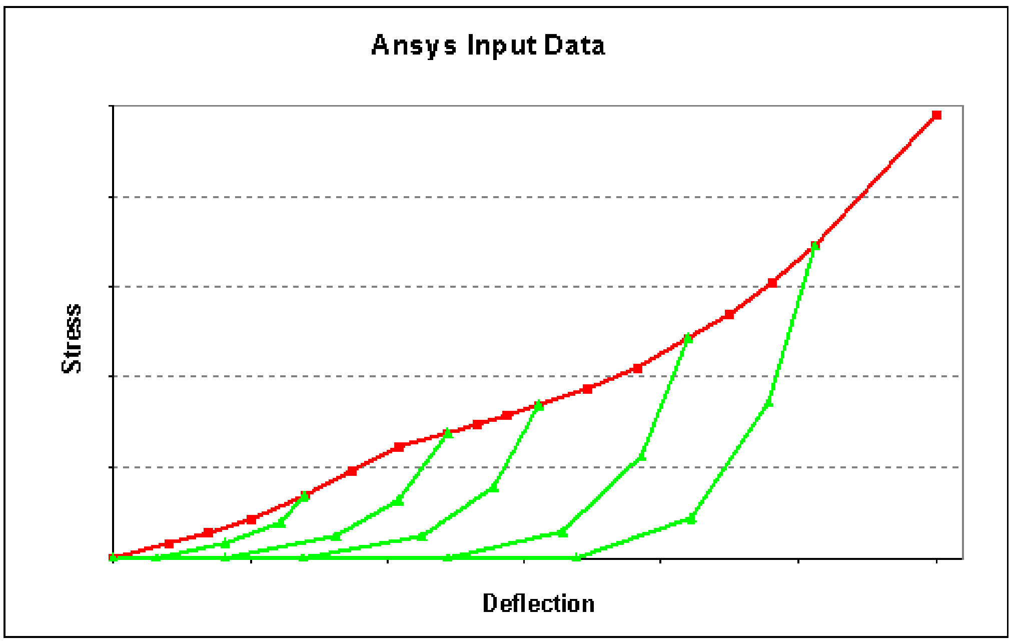


Figure 8 - Stress vs. Deflection Responses for the Body Material

Another useful feature of the Gasket material option is the ability to slide the entire curve to the right with an initial offset parameter. In the physical hardware, the assembly process begins with the gasket resting on the cylinder block and the cylinder head resting on the gasket. At this point, the head is only touching the combustion seals for two reasons. First, the combustion seals are thicker than the body material. Also, the liners are designed to protrude above the top deck of the block, which is a feature called "liner protrusion." The resulting clearance between the cylinder head and the body material can be incorporated as an initial offset in the material definition. This is much easier than accounting for the clearance using contact elements or constraint equations, as all components were modeled as being in contact. Also, liner protrusion varies from engine to engine. The analyst can study the impact of liner protrusion by simply varying the initial offset of the material, rather than re-defining all of the data points describing the material's response.

Load Cases

Two different load cases were analyzed. The first is the assembly of the joint named "Assembly", and the second is the application of maximum cylinder pressure along with the assembly loads, named "Assembly + Pressure." Experimental calibration data was generated on a rig test at room temperature, so no thermal load case was used in this calibration process. The Assembly load is applied in a single step, and requires between 10 and 15 equilibrium iterations to converge. The discontinuity between the loading and unloading slopes for the gasket materials makes it most efficient to apply the cylinder pressure load in 2 load cases. The first pressure load case applies 5% of the total pressure load, which allows a gentle turn onto the unloading slope. The second pressure load case applies the remaining 95% of the pressure load. Both pressure load cases converge in less than 5 equilibrium iterations.

Many of the measures used in the analysis of this joint involve the load change due to the application of cylinder pressure. The nonlinear, path dependent nature of this analysis makes the solution of a cylinder pressure only load case not possible. A standard post-processing step in these analyses involves subtracting the Assembly load step from the Assembly + Pressure load step. This is accomplished by using the LCOPER function in the post-processor. This mathematically created load case is used to determine the liner offloading due to cylinder pressure, which is one of the primary measures used to evaluate a head gasket joint.

Analysis Results & Experimental Verification

The calibration study performed on the ISX model involved 3 types of gaskets, 3 types of cylinder head structures and 2 protrusion levels. The full factorial of 18 runs was solved, with each configuration requiring between 12 and 20 hours to solve on a 450 MHz. Sun Ultra 60 with dual processors. The 3 cylinder head structures considered include the actual cylinder head geometry (referred to as Real Head), as well as 2 steel plates, with thickness of 0.5 and 1.5 inches. The steel plates were run purely for model calibration purposes, since their geometry and material properties can be accurately modeled. The following three sections describe the primary measures that were used to evaluate the model's performance, while the fourth section shows typical output from the gasket elements used to represent the body material.

Assembly Load

The "Assembly Load" is the total load on the liner after the joint has been assembled. Assembly load is most sensitive to the geometric tolerances that determine when the body material comes in contact with the cylinder head as well as the gasket component stiffness. The experimental measurement is made using a strain gauged liner that was calibrated to act as an axial load cell. The model's results are compared with the experimental results for 5 different configurations in Figure 9. All model results match experimental results within 12%.

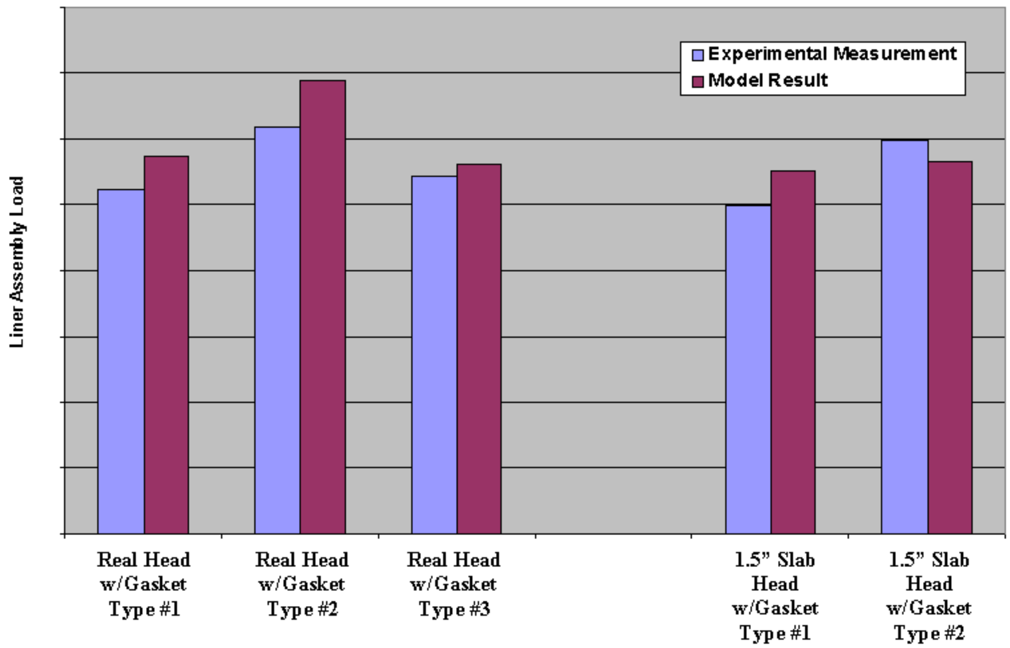


Figure 9. Chart of Assembly Load Results

Assembly Load Range due to Liner Protrusion Range

The "Assembly Load Range due to Liner Protrusion" is the difference between the Assembly Load at minimum liner protrusion and at maximum liner protrusion. This measure is most sensitive to the gasket component stiffness as well as the cylinder head, cylinder block and liner stiffness. Assembly Load results for a given configuration at minimum and maximum liner protrusion are subtracted to determine the experimental value. The three configurations with the Real Head are plotted against the experimental data in Figure 10. All results for this measure were within 13% when comparing the model to the experiment.

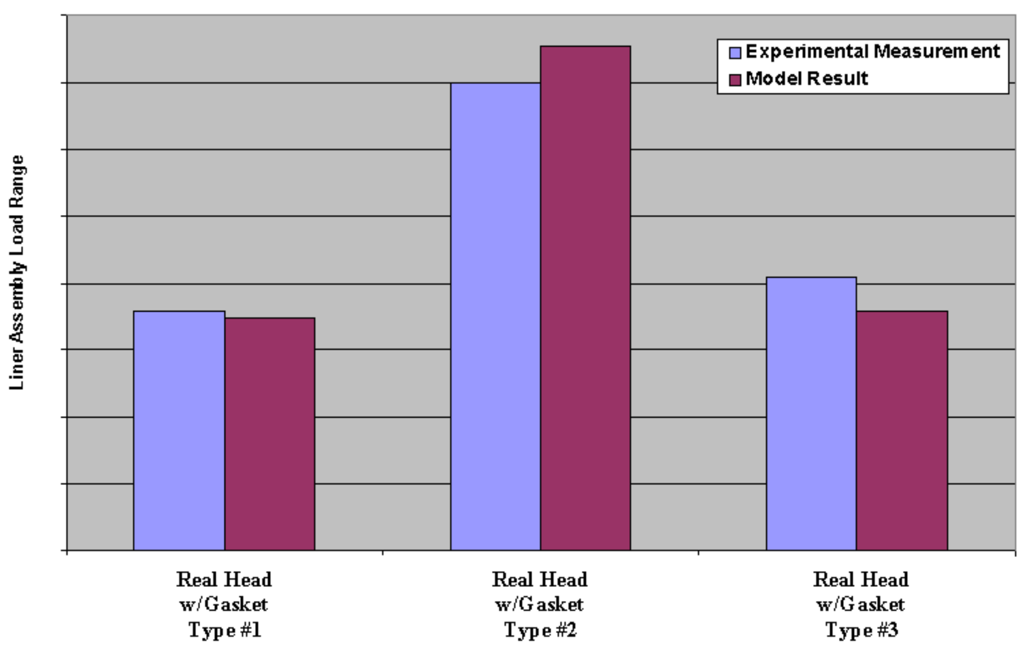


Figure 10. Chart of Assembly Load Range due to Liner Protrusion Range Results

Liner Offloading due to Cylinder Pressure

The "Liner Offloading due to Cylinder Pressure" is the amount that the liner load decreases when cylinder pressure is applied. The analytical value is evaluated using the mathematically created load case. The experimental value is determined by pressurizing a cylinder to maximum cylinder pressure using hydraulic fluid with the strain gauged liner installed. It is most sensitive to the stiffness of the cylinder head, cylinder block and liner, as well as the unloading stiffness of the gasket components. Results for this measure for 7 different configurations are shown in Figure 11. All of the model's results were within 18% of the measured values.

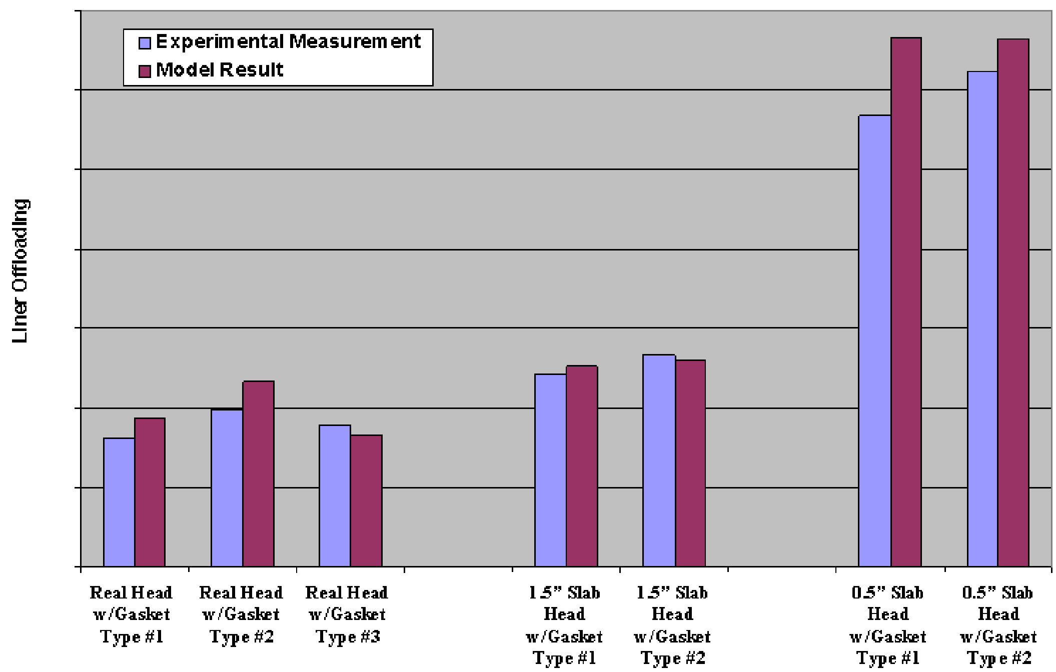


Figure 11 - Chart of Liner Offloading due to Cylinder Pressure Results

Gasket Component Stress

Compressive stress (through the thickness) contours are shown for the gasket body for two different gasket types in Figure 12. These are included to show the type of output that can be obtained from the gasket element. Both gasket types show highly localized loading around the head capscrews, which is to be expected. Gasket Type #1 is a soft gasket body, while Gasket Type #2 is a stiff gasket body. The soft gasket body distributes the assembly load over a much larger region than does the stiff gasket body, a result that makes intuitive sense.

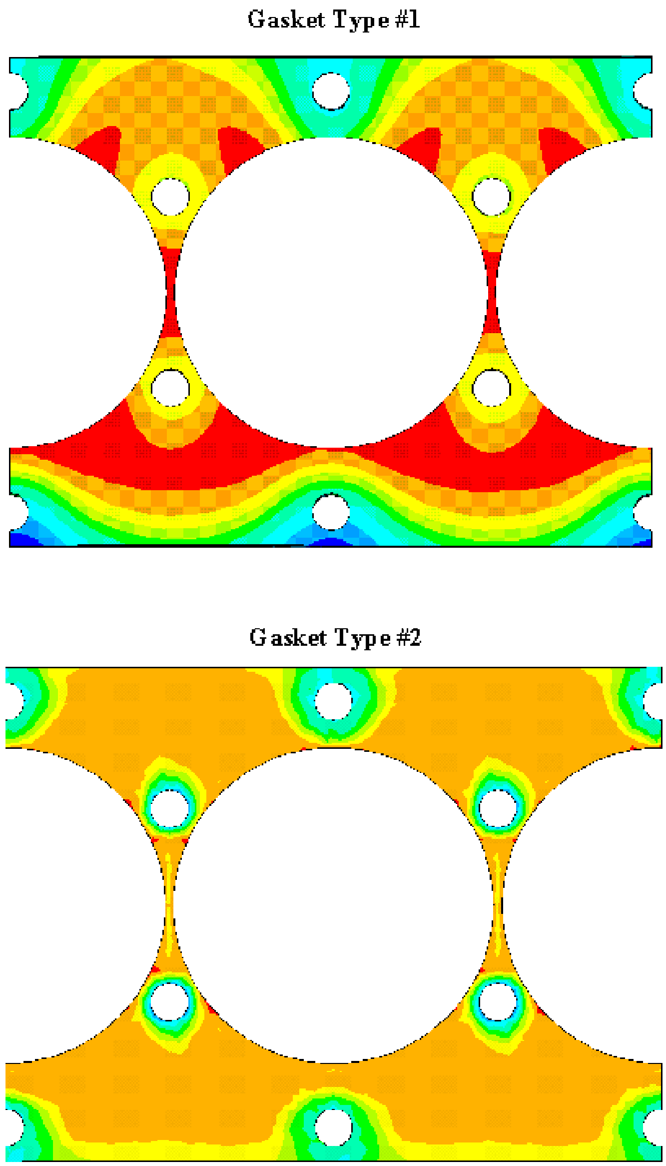


Figure 12 - Compressive Stress Contours for Body Material

Conclusion

The Gasket material option of the SOLID185 element type is straightforward to incorporate, gives reasonable solution times for large cylinder head gasket joint models, and gives results that match well with experiments. The material definition is sufficiently flexible to allow modeling of a wide range of gasket materials with nonlinear stress vs. deflection responses. At this time, models of this size require that all interfaces between mating components be meshed with matching nodal patterns. However, the additional time required to create these matching nodal patterns has not been found to slow down the overall process significantly.

References

1) Raub, J. H. (1992), “Structural Analysis of Diesel Engine Cylinder Head Gasket Joints,” SAE Paper 921725