Exhaust pipe FSI simulation to evaluate optimum gasketed joint (flange) design

Prashant Mahiras, Yogesh Jaju
Dana India Technical Centre, Pune, India

Keywords: Fluid Structure Interaction (FSI), Co-Simulation, Bolt loading, Heat transfer, gasket joint, Thermal cycling.

Abstract: The interaction between the fluid and structure domains is through a common physical interface region over which data is exchanged in a synchronized manner. An Abaqus analysis (structural) can be coupled to another analysis program (fluid dynamics) to perform multi-disciplinary simulations using third party data “communication” through MpCCI (Mesh based parallel code coupling interface).

In our case, we used MpCCI to couple Abaqus and Fluent while performing steady state heat transfer analysis to finally obtain a converged temperature map on the exhaust tail pipe assembly. This temperature map is used for the subsequent structural analysis on the assembly to evaluate flange distortion and gasket sealing pressure.

The objective of the present work is to simulate test conditions for an exhaust pipe gasket joint using co-simulation using Abaqus and FLUENT. The CAE results will then be used to investigate three different flange designs and choose one that exhibits minimum flange distortion while meeting the gasket sealing pressure criteria.

Introduction:

An exhaust tail pipe gasket joint, as shown in Fig. 1, is subjected to severe alternating (heating and cooling) thermal loading. Typically, the joint is subjected to a temperature of 650 °C for 15 minutes, cooled down to 315 °C for about 5 minutes and water quenched to room temperature @ 1 gpm for 30 sec. This thermal loading cycle is repeated as per OEM specific test standard. The gasket has to withstand such harsh conditions & still maintain necessary sealing pressure of minimum 3 MPa during and at the end the cycle.
Flange design considerations:

We investigated three different flange designs and choose one that exhibits minimum flange distortion while meeting the gasket sealing pressure criteria. A combination of stamped flanges, formed flanges and formed flanges with “skirts” was proposed, as show in Fig. 2. Formed flanges are typically 2.5 to 3 mm in gauge and hence, attractive from weight point of view. Stamped flanges are min. 8 to 10 mm thick and hence, attractive from stiffness point of view, but tend to be heavier and costly to manufacture.

Figure 2. Flange design proposals
FSI approach:

Due to the inherent nature of the problem that we were analyzing, we used a third party software, MpCCI (Mesh based parallel code coupling interface), to facilitate data exchange between Abaqus and Fluent. The FSI simulation cycle is shown in Fig. 3.

Figure 3. Process flow of FSI simulation using MpCCI

We chose to have automatic coupling using MpCCI between fluid and structural domains due to the following advantages it offers:

- Enormous time savings when compared to manual coupling,
- Ability to easily extract transient temperature when the joint is heated or cooled down to check for any potential leakage at the gasket joint,
- Better control over time increments (both in Fluent and Abaqus)
**Structural analysis procedure:**

The temperature map obtained at the end of the FSI simulation (shown in Fig. 3) is used as the thermal load for the subsequent structural analysis.

Following steps are simulated:

- Application of 100% bolt load
- Alternate heating to the thermal map obtained & cooling to room temperature (3 repeats)

The FEA model is shown below in Fig. 4.

![Figure 4. FE model](image)

**Results and discussion:**

The thermal map obtained at the end of the FSI cycle is shown below in Fig 5.
Figure 5. Temperature distribution at the end of the FSI co-simulation

It is evident from the temperature map above that FSI simulation is successful in generating an average temperature of 650°C along the pipe, which is close to what the assembly is heated up to in real life tests.

Another variable of interest, i.e., the sealing pressure on the gasket, is plotted at important steps of the analysis and is shown in Fig. 6 below.

Figure 6. Sealing pressure distribution on the gasket at assembly, third heating and third cooling cycles respectively.
It is clear from Fig. 6 that sealing pressure obtained in flange designs with skirts is lower than the minimum requirement 3 MPa and hence, unacceptable.

Further, distortion of flanges, which directly affects the sealing pressure, was also compared as shown in fig 7 below.

**Figure 7.** Comparison of flange deformation at the end of third heating and cooling cycles.

<table>
<thead>
<tr>
<th>Step: third heating cycle</th>
<th>Step: third cooling cycle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Formed flange design</td>
<td>Highskirt flange at one end</td>
</tr>
<tr>
<td>Highskirt flange at both ends</td>
<td></td>
</tr>
</tbody>
</table>

**Conclusions and recommendation:**

1. CAE results show that gasket sealing pressure obtained with formed flange is maximum while the other two designs indicate potential leakage areas. Minimum sealing pressure obtained with formed flange design *without* any skirts is higher than the acceptable sealing pressure criteria at various operating conditions, as shown in figure 6.
2. Warpage of flange is also considerably **less** in formed flange design as compared to the other two design options, as shown in Fig. 7.

3. The formed flange design (without any skirts) is recommended as it meets both the requirements.

4. The FSI capabilities of Abaqus have been successfully deployed to perform a co-simulation with Fluent on a real life problem involving gasketed joint design evaluation.

5. The co-simulation methodology developed has been successfully applied to several such problems and also for creating thermal maps on various hardware configurations with reasonable correlation to test results.