The objective of this project is to design an accumulator container with the following capabilities:
• Maintain the surface temperature of each battery cell below 60°C
• Withstand 40g of acceleration in the horizontal direction and 20g of acceleration in the vertical direction
• Provide the Formula U Tractive Team with easier access to the high-voltage battery segments and control electronics
• Reduce the geometric footprint of the container

We will meet these objectives by designing a container with separate compartments for the control electronics and battery segments and the accumulator, fans that provide cooling to each segment, and batteries that are spaced as tightly as possible while still maintaining a surface temperature below 60°C.

The accumulator container must withstand the acceleration requirements set forth by FSAE to ensure that the batteries remain safely contained in the event of a crash. To simulate the acceleration, we used Abaqus Finite Element Analysis (FEA) software. The FEA model we produced has the following properties:

**Container:**
- Modeled as carbon steel
- Modeled as an assembly of 3D shell panels
- Porous plasticity damage material FEA model

**Battery segments:**
- Modeled as a polycarbonate piece
- Modeled as a 3D solid
- Elastic material FEA model

**Boundary conditions:**
- Base mounting flange where the container is fastened to the chassis
- Fixed displacement and rotation at bolt holes

**Loading:**
- Surface traction applied to the battery segments to accelerate them during impact
- General contact relationship between the battery segments and the container

**Output:**
- Stresses and displacements across the entire container
- Model provides the results observed one second after impact

The FEA simulations showed that the maximum stress produced when the container is under forward acceleration and lateral acceleration exceeds the maximum tensile strength of 515 MPa at the mounting holes.

**Table 2:** Maximum stress and displacement values produced by the FEA model, as well as the maximum displacement location.

<table>
<thead>
<tr>
<th>Direction</th>
<th>Max Stress (MPa)</th>
<th>Max Displacement (mm)</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>Forward</td>
<td>616</td>
<td>14</td>
<td>Top</td>
</tr>
<tr>
<td>Lateral</td>
<td>668</td>
<td>16</td>
<td>Top</td>
</tr>
<tr>
<td>Upward</td>
<td>482</td>
<td>28</td>
<td>Top panel</td>
</tr>
<tr>
<td>Downward</td>
<td>413</td>
<td>6</td>
<td>Base panel</td>
</tr>
</tbody>
</table>

**Future Plans**

Future FSAE members can utilize the CFD and FEA simulations we completed this semester to guide further design decisions.

We will provide manufacturing work instructions for future FSAE team members to use when manufacturing the container.