17: Analysis of a weldment using beam elements

Topics covered

- Different levels of idealization implemented in finite elements
- Preparation of a SolidWorks model for analysis with beam elements
- Beam elements and truss elements
- Analysis of results using beam elements
- Limitations of analysis with beam elements

Project description

A Roll-Over Protective Structure (ROPS) is used to protect an operator of heavy equipment in the case of roll-over. Open the ROPS model. It consists of eight hollow square tubes 3" x 3" x 0.25" (Figure 17-1). All tubes are created in SolidWorks as structural members.

We need to find the displacements and stresses of this structure under a load of 2500lbf with all four legs restrained.

Figure 17-1: A ROPS cage is loaded in two corners with horizontal load 2500lbf to each corner. All legs are restrained as shown.
The tube cross section and details of corner treatment and trims are shown in Figure 17-2.

![Diagram of a tube structure with corner treatment and trims]

Figure 17-2: A detail of the corner: all tubes are 3"x2"x0.25" with a 0.5" radius

Corner treatments and trims are applied in the SolidWorks model using Weldment tools. The weld bead is not modeled.

Due to thin walls and complicated geometry in corners, this model is not suitable for meshing with solid or shell elements. Even if we were ready to accept long meshing and solution times, the stress results in the corners would be useless because of numerous sharp re-entrant edges causing numerous stress singularities.

To avoid these problems, the model can easily be meshed and analyzed with beam elements. Before we proceed with analysis, we need to explain what beam elements are and how they compare with solid and shell elements.
The differences between solid, shell and beam elements are summarized in the following table.

<table>
<thead>
<tr>
<th>Element type</th>
<th>Idealizations made to geometry intended to be meshed with this element</th>
<th>Assumptions on stress distribution in the element</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid</td>
<td>None; solid elements are created by meshing 3D solid geometry.</td>
<td>No assumptions on stress distribution need to be made</td>
</tr>
<tr>
<td>Shell</td>
<td>Shell elements are created by meshing a surface. Thickness is not present in the geometry and must be entered as a numerical value in the shell element definition.</td>
<td>Assumptions on stress distribution across thickness are made. In-plane stresses are assumed to be distributed linearly across the thickness. Transverse shear stresses are either assumed to have uniform distribution across element thickness (thin shell formulation), or to have parabolic distribution (thick shell formulation).</td>
</tr>
<tr>
<td>Beam</td>
<td>Curves are used to create beam elements. Curves represent beam geometry mathematically and do not physically model the cross section. In CAD terminology this is called wire frame geometry.</td>
<td>Assumptions about the stress distribution must be made in two directions perpendicular to the curve. These assumptions are the same as in beam theory: bending stresses are distributed linearly in both directions, and both axial and shear stresses are constant.</td>
</tr>
</tbody>
</table>

In summary, solid elements are a natural choice for meshing models with approximately the same size in all three dimensions, shells are the natural choice to mesh sheet metal models, and beams are the natural choice to mesh structural members.

Beam cross section geometry is only used to define beam element properties such as the area and second moments of inertia of the beam cross-section. It does not become part of the finite element model.

The information about beam cross sections is retrieved from the SolidWorks model which must be created as a Weldment. It is important to understand that the solid geometry of a Weldment is not meshed when beam elements are used. What is meshed is the underlying wire frame geometry (Figure 17-3).
Figure 17-3: Solid model and the underlying wire frame geometry

Solid geometry is used only to define beam cross sections. Beam elements are created by meshing curves. You can think of beam elements as lines with assigned beam cross section properties.

Corner treatments and trims have no relevance in beam element models. Before creating the study, change to configuration 02 no trims where trims are suppressed (Figure 17-4).

Figure 17-4: Corner treatment and trims have no relevance in beam element models. Both geometries will produce the same finite element model when meshed with beam elements.

End treatments may also be suppressed.
Procedure

Having examined the ROPS part, move to SolidWorks Simulation and create a Static study. Simulation recognizes the weldment geometry and anticipates that we intend to use beam elements. It creates a Solid Body for each structural member present in the geometry and places them in folders corresponding to Cut-Lists present in the SolidWorks model (Figure 17-5).

Figure 17-5: Study definition with beam elements

By default, the structural member is meshed with beam elements. You may change it by right-clicking and selecting Treat as Solid.

Material can be imported from the SolidWorks model or defined in SolidWorks Simulation individually to each or all beams. Here, the material was imported from the SolidWorks model.
Recall Figure 4-2 which shows icons denoting geometry intended for Solid and Shell meshing. We may now append one more: Beam geometry (Figure 17-6).

Figure 17-6: A geometry folder may contain all three types of geometries

Check marks indicate that the material has been defined.

If the SolidWorks model contains only one body, the geometry folder is not shown in the Simulation study. Otherwise, items in the geometry folder are marked by icons as shown in Figure 17-6. In many cases the automatic designation of geometry to one of three geometry types can be changed. Beams may be replaced by Solids and Surfaces, and sheet metal parts may be replaced by Solids. This is accomplished by right-clicking the geometry folder component and selecting the appropriate choice from the pop-up menu.

In this exercise we accept the default assignment of all parts to Beam elements. As Figure 17-5 indicates, the geometry folder holds eight bodies and they are all intended for meshing with beam elements.

The next step is the definition of connectivity between the soon to be created beams.
Right-click the Joint Group folder and select Edit from the pop-up menu to open the Edit Joints window which shows automatically created joints. If necessary, these automatically created joints may be edited in this window. Accept the default selection All beams and click Calculate to create joints automatically (Figure 17-7).

Figure 17-7: Joints are automatically created between beams. Arrows indicate joint positions.

No action is required in our case. Working with a simplified geometry facilitated automatic joint creation.

Joints (or beam ends) are connected to each other only if they are contained within the pinball diameter which can be changed in the Edit joints window. It is recommended that beam ends intended to be connected are made coincident or very close. Non coincident beam ends can still be connected if they fall inside the pinball, but this may result in a “patch-up” beam element mesh (Figure 17-8).
Figure 17-8: Joints (beam ends) are connected only if they are contained within the imaginary pinball not visible in the model display. Beam elements are shown as thin tubes. 

Beam elements are graphically depicted as round tubes, even though the beam elements are in fact just lines. The tube diameter is always the same, regardless of the actual cross-section size, shape and orientation.

Disjoined structural members can be connected if their ends fall within the volume of the pinball (Figure 17-8 top right). However, the beam element mesh (bottom right) will then contain automatically created connecting elements. This may create unpredictable results.
Apply fixed restraints to all four joints at the free ends (at the bottom) of the vertical members (Figure 17-9).

**Figure 17-9: Fixed restraints applied to the free ends of vertical members**

Note that beam elements have six degrees of freedom per node and, therefore, can distinguish between Fixed and Immovable restraint. Here, we need to apply a Fixed restraint.
Apply a 2500 lbf load to the corner as shown in Figure 17-10.

Figure 17-10: Force load applied to the corner joint

*Note that beam elements have six degrees of freedom per node and therefore can be loaded with a force as well as with a moment load.*
Now create the beam element mesh, noting that there are no user controlled mesh parameters. The beam element mesh is shown in Figure 17-11.

Figure 17-11: A beam element mesh is created from curves (here straight lines) used in the SolidWorks model to define Structural Members.

A beam element is a line with cross-section properties taken from the Structural Member cross section geometry. This is schematically illustrated in Figure 1-8.
Run the solution and create a displacement plot (Figure 17-12) as well as a stress plot (Figure 17-13). Note that while the undeformed model is shown as solid geometry, the displaced model is shown as “tubes”.

Figure 17-12: Resultant displacement results of ROPS model

*The maximum displacement is 0.70''.*
Figure 17-13: A stress plot of the “Highest axial and bending” stresses.

The maximum stress is 29885psi. Compared with the material yield strength of 90000psi, this indicates that the structure is below yield.

Note that the “Highest axial and bending” stress is NOT von Mises stress. To understand what it is, we need to review all stress result options available for beam elements.
The software provides the following options for viewing stresses (refer to Figure 17-14):

- Axial: Uniform axial stress = $P/A$
- Bending in local direction 1: Bending stresses due to moment $M_1$ about axis DIR1.
- Bending in local direction 2: Bending stress due to moment $M_2$ about axis DIR2
- Highest axial and bending
- Torsional
- Shear stress in DIR1
- Shear stress in DIR2

Figure 17-14: Positions of axis DIR1 and axis DIR2 for the beam cross section used in this exercise

Axis DIR1 and axis DIR2 cross the centroid of the cross section.
The **Highest axial and bending** stress is calculated by combining axial stress and bending stresses due to moments M1 and M2. This is the default selection in the Stress Plot window.

**Simulation** offers ample ways of analyzing beam element results such as a **Beam Diagrams** (Figure 17-15). Also, investigate **List Beam Forces**.

![Beam Diagrams](image)

Figure 17-15: Bending moments in DIR1, bending takes place about the DIR1 axis shown in Figure 17-14.

The Beam Diagrams window is called from the pop-up menu activated by right-clicking the Results folder.
For a better understanding of beam elements we will review another example shown in Figure 17-16. Open the part TRUSS.

![Figure 17-16: The truss is made out of rectangular hollow tubes dimensioned 3” x 3” x 0.25” with 0.5” radius.]

*This SolidWorks model includes gussets and end caps.*
There are two ways to proceed. We can suppress gussets, end caps and end treatments because these details may interfere with the definition of joints in the beam mesh.

Another way is to remove them from the analysis after study has been created (Figure 17-17). We will proceed this way. Define a static study called \textit{weldment01} and excluded from the analysis all solid bodies (Figure 17-17).

Figure 17-17: Details are excluded from the analysis by right-clicking the corresponding component in the geometry folder and selecting \textit{Exclude from Analysis}.

\textit{Here, endplates and gussets represented by solid bodies are removed from the analysis. They will not be meshed.}
Next, right-click the *Joint Group* folder and verify that seven joints have been correctly calculated (Figure 17-18).

![Edit Joints](image)

**Figure 17-18**: Seven joints (including two beam ends) were calculated

*Restraints will be applied to the joints indicated by the arrows*
Apply restraints and a load to the joints as shown in Figure 17-19.

**Figure 17-19: Restraints and loads applied to joints**
Mesh the model to produce a mesh as shown in Figure 17-20.

Figure 17-20: Finite element mesh of the model in study weldment01

Each long beam is represented by 32 beam elements. Short ends are represented by 3 beam elements. Joints and load are also shown.

Solve the study weldment01 and review the displacement and stress results.
Now copy the study `weldment01` into study `weldment02`. In study `weldment02`, select all structural members (beams) in the `truss` folder and right-click to open a pop-up menu. Select **Edit definition** to open the **Apply/Edit beam** window. Select **Truss** in the **Apply/Edit beam** window (Figure 17-21).

![Figure 17-21: All beams are now defined as trusses](image)

*This can be done for all beams or individually for selected beams.*

This redefines connectivity between beams from rigid to pin joints. While beams can be loaded with any combination of forces and moments, trusses can be only loaded with axial force. Trusses behave as a tension/compression springs and are meshed with only one element (Figure 17-22)
Figure 17-22: Finite element mesh of model in study *weldment03*

*Each beam is represented by one truss element. There are nine elements in the model. They behave as if they were pin jointed and can be loaded only with axial load. Joints and load are also shown.*

An attempt to run a solution of study *weldment02* displays an error message shown at the top of Figure 17-23.
Figure 17-23: An attempt to run study weldment02 brings up an error message which differs with the solver used but in either case is caused by Rigid Body Motions present in the model.

The study needs to be run with the option Use soft springs to stabilize model. Either solver can be used.

The supports are insufficient and allow rigid body motion of the model because truss elements have only three degrees of freedom and can not accept any restraints on rotations. Therefore, the entire model can spin about the line passing through the supports. The short ends can freely rotate about the joint where the restraint is defined. To eliminate these rigid body motions we need to execute a solution with the Use soft springs to stabilize model option checked (Figure 17-23).
Obtain the solution of study \textit{weldment02} and compare the displacement and stress results between the two studies (Figure 17-24).

\textbf{Beam elements}
Study \textit{weldment01}

\textbf{Truss elements}
Study \textit{weldment02}

Figure 17-24: The deformation pattern of the model with beam elements (top) and truss elements (bottom)

\textit{Note that the short ends in both models do not experience displacements.}
Bending of structural members observed in the beam element model proves that beam elements are rigidly connected to each other and transmit bending moments. Conversely, truss elements are connected by pin joints; they can not transmit bending. Deformation of truss elements can only take the form of stretching and compressing, therefore deformed truss elements remain straight.

Verify that the **Highest axial and bending stress** and **Axial stress** are the same for the truss element model. This is because axial stress is the only stress component present in the truss elements.