ANSYS - Semi-monocoque shell - Step 2

Step 2: Specify element type and constants

Since thin structures can be modeled efficiently as shells, we'll use shell elements to build the finite-element model. Shell elements can support membrane and bending loads consistent with classical shell theory (sorry, FEA doesn't let you off from understanding basic theory). As you can imagine, shell elements are appropriate when the thickness of the structure is small compared to the other dimensions. The computational savings come about because only the mid-surface of the structure is modeled; the thickness and other cross-sectional properties are incorporated into the element stiffness matrix and input as "real constants" in ANSYS. (This is analogous to modeling beams using beam elements where the beams are modeled as lines with thickness and other cross-sectional properties being "real constants"). Section 2.10 in the *ANSYS Element Reference* manual gives you a page of useful information on shell elements. Be sure to peruse it in the online documentation since it'll be on the final. 😉

Specify Element Type

Let's take a peek at the shell elements available in ANSYS. Bring up the ANSYS documentation window, select the ***Search*** tab, enter the phrase "pictorial summary" and click on ***List Topics***. Then double-click on *3.2 Pictorial Summary* in the left pane. At the top of the pictorial summary of element types in the right pane, click on ***SHELL Elements***. This brings up the list of shell elements available in ANSYS including many with specialized capabilities. Perusing this list, you'll see that***SHELL63*** (4-node elastic shell) is a basic shell element and a possible candidate for our problem. A close relative is ***SHELL93***(8-node elastic shell) which has mid-side nodes in addition to the corner nodes. Since the mid-side nodes give greater accuracy, we'll use***SHELL93*** for our problem. Click on ***SHELL93***in the help and take a few minutes to persue the manual page for this element. What are the "real constants" that we'll need to enter in the next step? Note that each node has six degrees of freedom: three translational and three rotational.
Main Menu > Preprocessor> Element Type > Add/Edit/Delete > Add...

Pick ***Shell*** in the left field and ***Elastic 8node 93*** in the right field. Click ***OK*** to select this element. The ***SHELL93*** element will now be available in the meshing step. Close the *Element Types* menu.

Specify Element Constants

Main Menu > Preprocessor> Real Constants > Add/Edit/Delete > Add

This brings up the *Element Type for Real Constants* menu. Click ***OK*** to specify the real constants for the *SHELL93* element.

When meshing, we'll have to assign three different thickness values: *H1* for the plate; *W2* and *W3* for the stiffeners in the *x* and *y*directions, respectively. This means we'll have to create three real constant sets, one for each of these thickness values. According to the*SHELL93* help page, if the element has a constant thickness, only ***TK(I)***, the shell thickness at the first corner node, needs to be input.

Create the first real constant set: make sure ***Real Constant Set No.*** is set to `1`. For ***TK(I)***, enter `H1`. Leave the other fields blank since they are not applicable to our problem. Click ***Apply***.

Create the second set: For ***Real Constant Set No.***, enter 2. For ***TK(I)***, enter {W2}} and click Apply.

Create the last set: For ***Real Constant Set No.***, enter `3`. For ***TK(I)***, enter `W3` and click ***OK***.

Save: **Toolbar > SAVE_DB**

Go to Step 3: Specify material properties

See and rate the complete Learning Module

Go to all ANSYS Learning Modules