Submodeling in ANSYS Workbench

By Dave Looman, ANSYS, Inc.

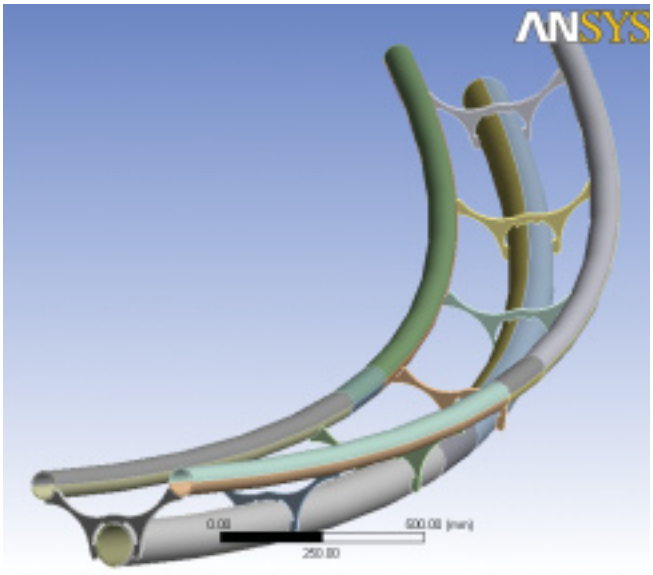

Figure 1. Full CAD model of a curved tubular assembly
Application courtesy Klaus-Dieter Schoenborn, CADFEM.

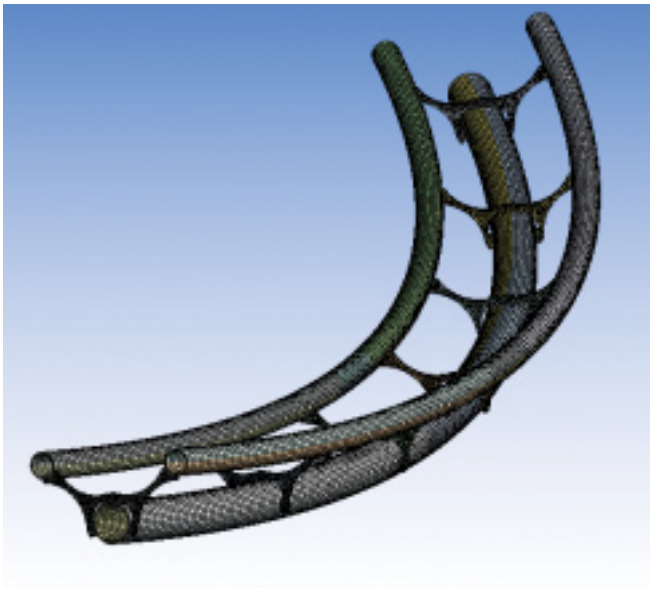

Figure 2. Full global model

Submodeling utilizes two separate models. A full or global model representing the entire structure is used to transform global loads to local deformation. The submodel includes the local geometric details with an appropriate mesh density. The submodeling algorithm then interpolates the deformation from the global model to the submodel "cut boundaries" and solves for the local stress state.

This method typically requires extensive planning and documentation of the workflow, especially if many submodels and numerous load cases are involved. In addition, setup of a submodel may take considerable time. However, the ANSYS Workbench tree and efficient computer-aided design (CAD) interaction make the procedure easier. With small ANSYS Parametric Design Language (APDL) enhancements applied to the model tree, the submodeling technique may be combined with the ANSYS Workbench Geometry handling and process documentation. Thus, a workflow can be presented that covers the whole process from CAD to fatigue analysis in five steps.

Step 1. Build/import the model from CAD. To illustrate this process, a sample analysis is performed to determine stresses on a tubular welded assembly with a regular pattern of joints. It represents a small sample section of a repetitive structure that forms the track of a roller coaster. The model shown in Figure 1 has been created using ANSYS DesignModeler software.

The loading on this structure is caused by a trolley rolling along the two upside tubes. This loading is transferred to the larger tube via the joint elements and passed to the supporting structure. Loading is generated by gravity and centrifugal forces.

To obtain accurate stress in a local region, submodeling separates local analysis from the global model. This allows mesh refinement in a region that might not be possible on the full model without exceeding size limits.

Step 2. Mesh and solve the global model. Each part was meshed independently and connected with surface-to-surface contact. Contact pairs may be part of the interpolated region in the full model, as long as they are completely enclosed by the submodel cut boundary. Figure 2 shows a sample mesh with surface-to-surface contact.

Step 3. Analyze the global model to identify critical spots. The full model gives a general impression of the structural deflection and is used to find the location of critical spots of interest. The structural result is saved and maintained in database and results files in the parent directory. The global solution is needed for interpolation of the submodel boundary conditions.

Step 4. Generate the submodel from CAD data and mesh. Once critical spots are known, submodels such as the one shown in Figure 3 may be created from the original CAD geometry. This is done by cutting the model and suppressing the remaining solids. This process automatically achieves geometrical consistency since the remaining solid does not change its location with respect to the global coordinate system. In order to interpolate displacements from the global model to the submodel, a Named Selection should be created on the cut boundaries, which will be referenced in the macro described next.

Step 5. Interpolate boundary conditions from the global model to the submodel and solve. The interpolation is done by inserting a small macro into the environment branch of

Figure 3. Submodel geometry of a critical structural detail

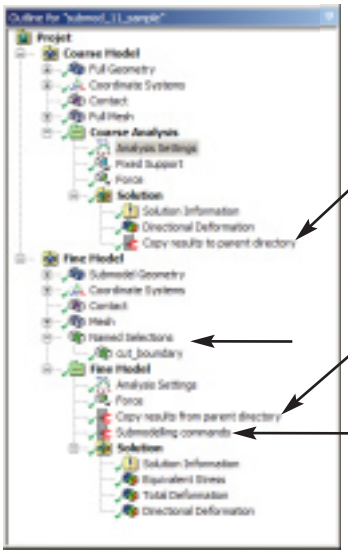

Figure 4. Simulation tree showing the submodeling procedure

```
!!! ANSYS 11.0 Procedure !!!

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!!! Copy results to parent directory !!!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
/COPY,file,rst,,coarse,rst,..\..\
/COPY,file,db,,coarse,db,..\..\

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!!! Copy results from parent directory !!!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!

/COPY,coarse,rst,..\..\,coarse,rst
/COPY,coarse,db,..\..\,coarse,db
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!!! Submodeling Commands !!!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!

FINI
/PREP7
CMSEL,s,cut_boundary
NWRITE
ALLSEL

SAVE,fine,db
FINI

/POST1
RESUME,coarse,db ! OPEN COARSE MESH
FILE,coarse,rst ! DEFINE RESULTS FILE NAME

SET,1
CBDOF

FINISH
/PREP7

RESUME,fine,db     ! RESUME SUBMODEL FROM FILE
/INPUT,,cbdo

FINI
/SOLU
```

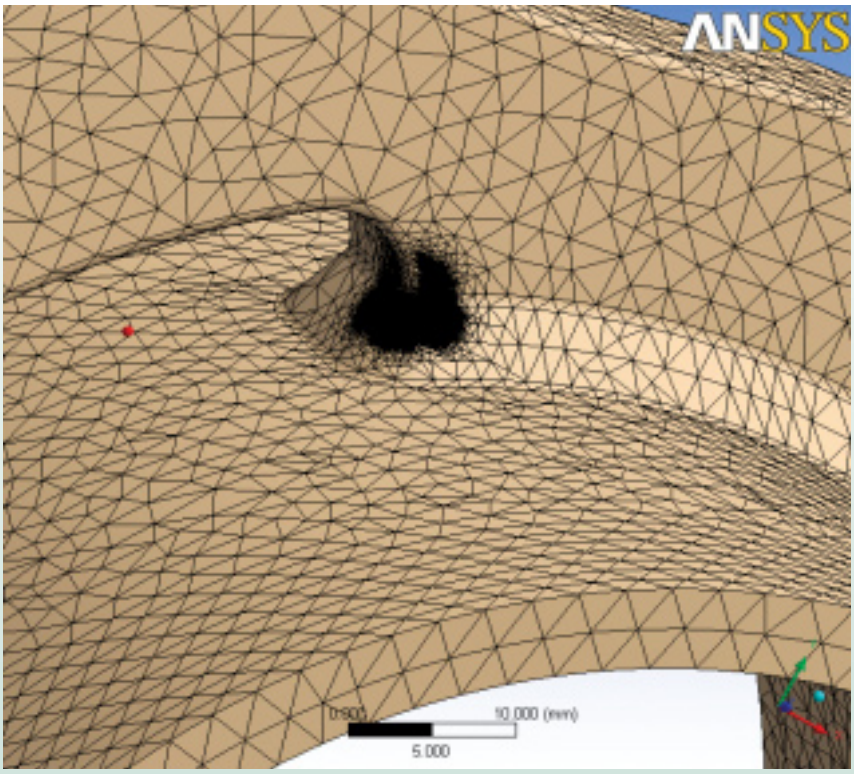

Figure 5. Submodel mesh detail using the sphere of influence feature

the submodel tree as shown in Figure 4. This macro resumes the full model database and results file, and it performs the displacement interpolation (CBDOF command). After that, the submodel is restored, the interpolated boundary conditions are read and the submodel is solved. Note that any external loads present in the submodel (including gravity effects or temperature loading) also should be applied. ANSYS Workbench Simulation solves the model and performs post-processing just as on any regular model. From the interpretation of the resulting local stress state in the submodel, the critical locations now may be reviewed with greater fidelity.

For this example, the initial submodel mesh is found to be still too coarse to accurately predict fatigue life from the resulting stress, so a locally refined mesh is needed. The "sphere of influence" method of the ANSYS Workbench platform is ideally suited to obtain the type of mesh needed. Figure 5 shows the refined mesh on the submodel.

This step is then repeated simply by creating the "sphere of influence" tab on the mesh branch and solving. The interpolation now is performed on the new FE mesh since the macro overwrites any files that were created on a previous run. The interpolation is done from the original results file, which still resides in the parent directory. Equivalent stress in the submodel then can be solved.

In the submodeling process, model consistency is maintained by using ANSYS DesignModeler software to create both full models and submodels. Capturing the process in the tree clearly archives the analysis and allows a user who later opens the database to understand immediately what was done. Note that an arbitrary number of submodels may be created and solved by interpolation from a single run of the global structure. All of these submodels may be included in the model tree, and variants may be studied without having to repeat the whole process. Users should remember to review and compare stresses at the cut boundaries between the global model and submodel to verify that the cut boundary is far enough from the region of interest. Submodels may be altered using the bidirectional CAD interface to ANSYS DesignModeler software or other CAD programs. ■

For more information, refer to chapter 9 on submodeling in the ANSYS Advanced Analysis Techniques Guide.