www.FEA-Optimization.com ANSYS Note #2
Transferring Section Load into a (Cut-Boundary) Submodel

Shen-Yeh Chen
February, 2002

When conducting a full assembly or large scale model analysis, it is usually not possible to include all details in the first analysis due the limitation of computer resource. Commonly a course model is first analyzed to pin point the critical area, and then a submodel is built to refine the mesh and get a more accurate result. ANSYS offers convenient commands, like **CDBOF** and **BFINTF** for this purpose. These commands interpolate the state variables (displacement in the structural analysis) on the coarse model into the boundary of the fine model as loading and boundary conditions. This is called the displacement-controlled approach in this document. This approach is usually sufficient for most problems. However, under certain circumstances in structural analysis, applying the load on the cut boundary will be more proper than interpolating the displacement. This is called the load-controlled approach. This document offers a convenient way to implement this approach. This approach is most suitable when the mesh on the boundary of the coarse model and the fine model is not consistent.

It is assume that, the readers already know when to use the displacement-controlled approach, and when to use the load-controlled approach. Such decisions will depends on problems solved, and will rely on the users' knowledge in structural mechanics.

First, you have to download the macro from ***www.FEA-Optimization.com***
Go into **ANSYS Resources**, under **Macros**, download the macro **x_nfor2f.mac**

1. On the coarse model, pick the elements that represent the same geometry of the boundary of the fine model, and then pick up the nodes on the boundary on this geometry that you will apply the section load later.

2. Simply apply the macro as
   x_nfor2f
   The macro will create a CDB file x_nfor2f.cdb and the nodes on the boundary will be on the component X_NFOR2F. This cdb file contains section load on each node of the boundary you have selected in Step 1, and the coordinates of these nodes

3. Now clear your coarse and resume your fine model

4. Read in the CDB file by previously created
   CDREAD,db,x_nfor2f,cdb
   This will import all the nodes and the section load on them.

5. Selected the boundary nodes from the coarse model by
   CMSEL,,X_NFOR2F
   Create mass elements on these nodes. A simple macro will do. These mass elements are necessary for the constraint equations we will build later. They are just created to "harbor" the load on these boundary nodes from the coarse model.

7. Pick up the elements on the boundary of the fine model (remember that at this point, still only the nodal component **X_NFOR2F** is selected)

8. Do a **CEINTF** to couple the mass elements with the boundary of the fine model. Now it is done.