

Exporting Ansys meshes to Matlab finite element 2D tool box

There are two folders here and one tutorial:

macro -- which contains the macro script you need to write the grid files

PlotGrid -- A matlab utility to plot the grid and boundary indicator

ANSYS2MatlabTutorial -- A tutorial which teaches you how to use the macro in ANSYS through a simple example.

A macro has been created which allow writing geometry data in ASCII format out of Ansys. The macro can be activated from Ansys GUI using custom added buttons in the Ansys Toolbar. The ASCII format files can be further loaded to Matlab by using the m-file loadFromGridFile.m (currently only supports T3 and Q4 elements).

The information written out by the macros consists of four files that are named ansys.extension (extension = header, node, elem, boundary). In Ansys, the geometry model is often created combining several elementary geometries. This means that the model usually includes boundaries, that have no physical meaning. Therefore, the macro ansys_ch asks the user to graphically pick the boundaries that have physical meaning (have essential or **non-zero** natural boundary conditions).

How to use this utility:

1. In the macro folder, there are two files start120.ans and ansys_ch.mac. Please put them into your ansys working directory.

start120.ans is a start-up script which adds the custom button ANSYS_CH in the ANSYS toolbar. When hitting the button, the macro ansys_ch.mac is executed. 120 means the version number of ANSYS. You may change it according to the version you are using.

2. Put the fourfiles, ansys.extension (extension = header, node, elem, boundary) into your Matlab working directory and call the function loadFromGridFile in the InputGrid.m. This function will automatically set up the geometry and mesh.

Here are the file formats:

ansys.header:

In general it contains 2 rows. The first row has three numbers:

Number of elements, Number of nodes, Number of boundaries.

The second line describes the type of finite element.

Space Dimension, Number of nodes per element, the index of the element type.

The last properties is not used in our Matlab code. However, you still need to put a arbitrary number here when you write your own grid file. The reason is that we use the function 'load' which requires that the file is in standard matrix form.

ansys.elem:

Each row lists the global node number in one element.

Defines the node connectivity for each element. You may see several columns which are actually not used in our Matlab code. They are material properties from ANSYS.

When you write your own file, you can only write down the node connectivity.

ansys.node:

There are three or four columns.

Node Number, x coordinate, y coordinate, z coordinate.

If the file is from ANSYS, you may see some blanks after node number. The blank space means zero in ANSYS and thus is not printed out.

When you write down your own node file, you may want to put zero back for clarity.

ansys.boundary:

For each boundary, it will begin with "Boundary" + boundary number.

After this line, it lists the node number and coordinates of this node on this boundary. In our Matlab, we do not need the coordinates of the node since they are already included in the ansys.node file.

To identify the boundary indicator is critical. When you select boundaries from ANSYS, the sequence of the boundaries to be stored is decided by ANSYS not your order of

MAE 4700/5700 (Fall 2010) The 4 geometry/meshing/boundary-marker files

choice. To view this information, go to PlotCtrl->Numbering-> lines, turn it on. Then you can see the numbering of the lines in the grid. The sequence to store the boundaries in the file is the ascending order of the line number in ANSYS.

You may want to use the program in the folder PlotGrid to view the grid and boundary indicators.

So if you write your own file, you can simply list the node number.

Important note: These files are needed even if you decide to use another mesh generator or to input geometry and mesh data manually (e.g. for the simple grids in some of the problems of our HW). Using different mesh generators from FEM analysis tools is VERY COMMON in practice (in this case using ANSYS for mesh generation and MATLAB for the FEM analysis). Be sure that these 4 files are prepared as discussed here.