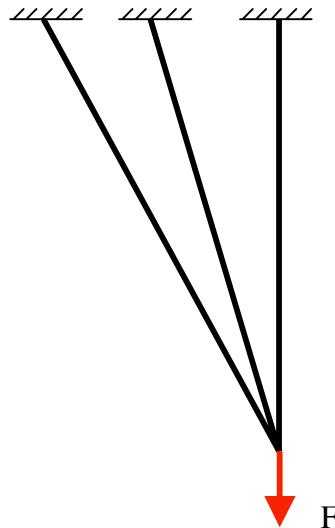Learning ABAQUS: 3-Bar Truss Example Problem

F

The file *truss-3br.inp* (or *1.3_truss-3br.inp* in this folder) is an ABAQUS input file for finite-element static analysis of the 3-bar truss structure shown above. Input files should be named in the form *input-filename.inp*, where *input-filename* is anything of your choosing, so that your *.inp* file is, e.g., *mytruss.inp*, *Elizabeth-truss.inp*, etc. It is usually convenient to keep the *jobname* close, but not identical to *input-filename* (to avoid confusion if you later want to delete unsuitable results), e.g., *jobname* as *mytruss1* or *Elizabeth-truss-a.inp*.

Read through the input file to get a feel for the structure of an ABAQUS input file. Next, run the job using the command:

abaqus j=*jobname*

and then follow the dialog and give the name of the .inp *without* the .inp extension (i.e., give it as *input-filename*, not *input-filename.inp*)

Files Created. You should see files with the following extensions (for *jobname = Truss-test*) created in the directory after your analysis is complete.

Truss-test.com: Command file, created by the ABAQUS execution procedure
Truss-test.dat: Printed output file. It is written by the **analysis, datacheck, parametercheck,** and **continue** options. It contains tabulated information of requested variable outputs.
Truss-test.log: Log file, which contains start and end times for modules run by the current ABAQUS execution procedure, and will display error messages if the procedure did not run successfully.

Truss-test.msg: Message file; may contain error messages

Truss-test.odb: Output database. It is written by the **analysis** and **continue** options. It is read by the Visualization module in ABAQUS/CAE (ABAQUS/Viewer).

Truss-test.par: Modified version of original parametrized input file showing input parameters and their values.

Truss-test.pes: Modified version of original parametrized input file showing input free of parameter information (after input parameter evaluation and substitution has been performed

Truss-test.pmg: Parameter evaluation and substitution message file. It is written when the input file is parametrized

Truss-test.prt: Part file. This file is used to store part and assembly information and is created even if the input file does not contain an assembly definition.

Truss-test.sta: Status file. ABAQUS writes increment summaries to this file in the **analysis, continue,** and **recover** options

Files You Need to Keep. Only the .odb file is necessary for viewing results, and this is the only file you absolutely must keep after running an analysis. The .dat file is useful to keep because it can contain requested variable outputs. The .par file is useful to keep because it will tell you the values of parameters used in the analysis. All other files can be deleted.

While the analysis is running, there is a file *input-filename.lck* present. When this file disappears, the analysis has ended. After the analysis is complete, first open the .log and look for the line "ABAQUS JOB *jobname* COMPLETED" at the bottom of the file. If an analysis ends with errors, you need to check the .sta, .dat, and .msg files for error messages.

Next, open the .dat file. Towards the end of the file you will see the tables containing the requested variable outputs.

Viewing Results. To view results, you will need to open ABAQUS/Viewer. Once you have started Viewer, click on Open Database (see figure on next page)
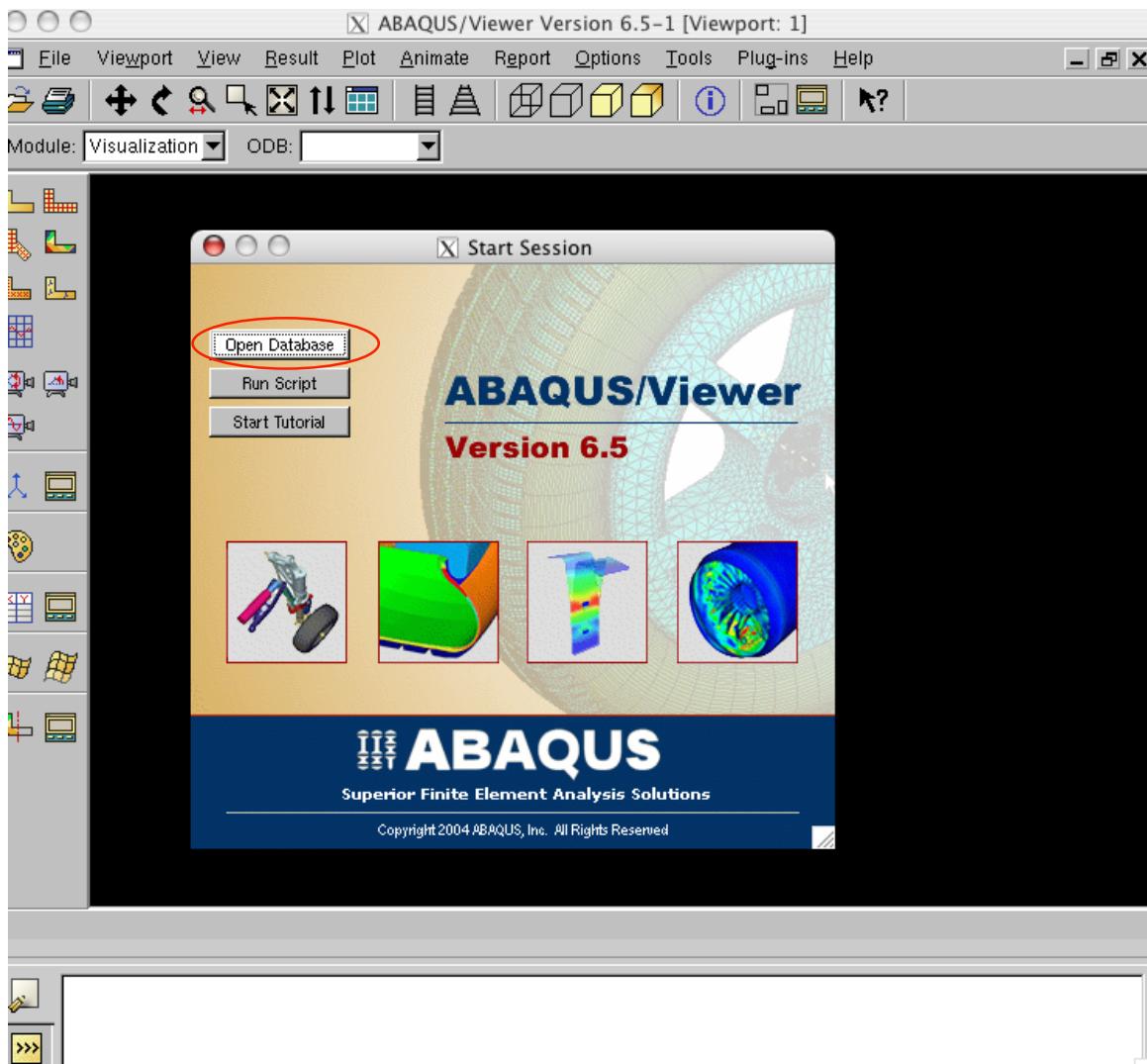
and open the output database file for the truss problem. To plot the deformed shape, select the menu "Plot" then select "Deformed Shape." If you would like to superimpose the undeformed shape on the deformed shape, select the "Options" menu, and select Deformed Shape. In the Deformed Shape Options Dialog box, check the box marked "Superimpose undeformed plot." (see figure on next page for plot of deformed shape)
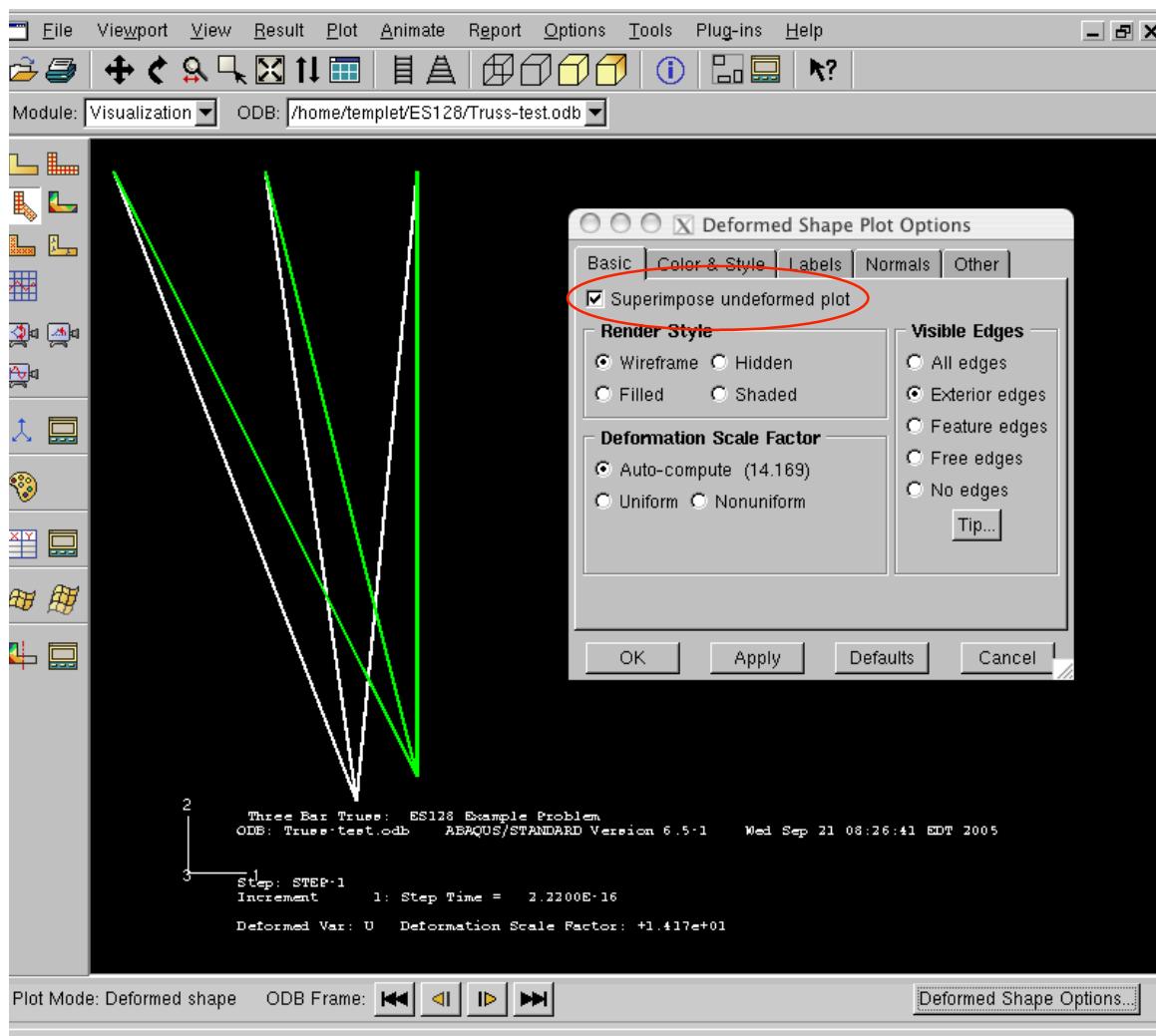
You can plot contours of various variables, such as stress and strain.
Explore the options available in Viewer, and try plotting various things.

Printing Results. To print results you can either do a screen capture (as
seen above), or you can print to a file, which will give you a much better
resolution image.

From the File menu, select Print.
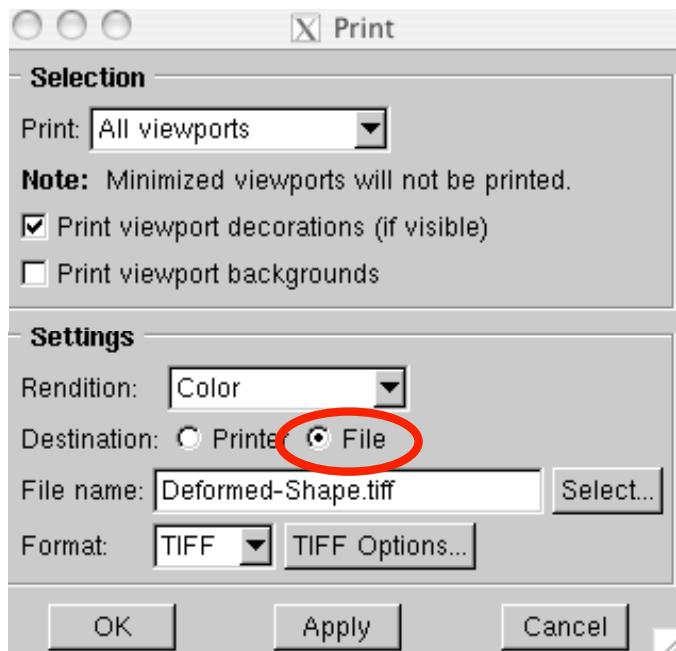
In the print dialog box, select the destination "file" to print the image in the viewer window to a file. You can select the file format to print to, such as TIFF or EPS and specify a name for the file. When you select "OK," a file with the name specified will be created in the directory where you are working. Below is a result from printing the deformed shape to a .tiff file:
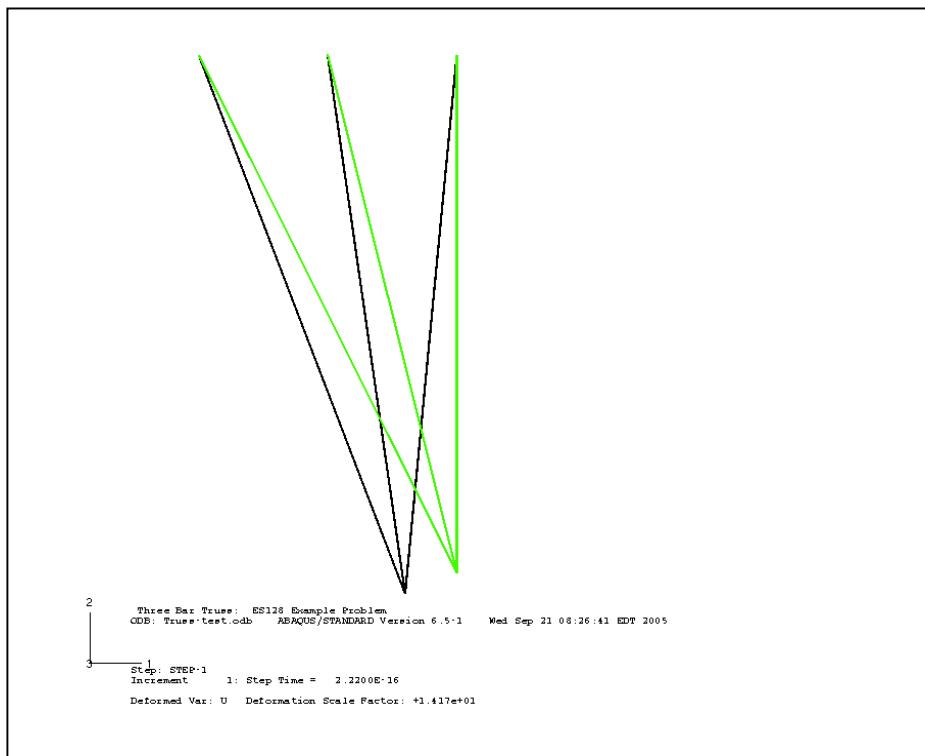
Below is the input file *truss-3br.inp*; copy it into a text editor and save it in your working directory as *truss-3br.inp* (there must be no blank lines)

```
**************************************************
**
**   written by James R. Rice
**
**   last modified: 10/26/2005
**             by:  Nanshu Lu
**
**************************************************
**
**   brief description:
**
**   An Example.inp File for an ABAQUS Finite
**   Element Solution
**
**   Double astericks (at left) indicate a comment
**   All such comments are ignored by the computer
**   They are inserted for clarity only
**
**
**************************************************
**
**   heading information
**
**************************************************
**
*HEADING
 Three Bar Truss:  ES128 Example Problem
**
**   the HEADING title (Three Bar...) will appear
**   on any output files created by ABAQUS
**
**   Note on capitalization:  In general, you do
**   not have to worry about case in input files.
**   The exception is with the use of parameters,
**   (not used in this file); parameter names are
**   case sensitive.
**
**************************************************
**
**   Parameter Definitions
**
**************************************************
**
*PARAMETER
 E = 30e6
 Load = -10000
**
**     E:  Young's Modulus
**   Load:  concentrated load to be applied
**
**   Note on Units:  ABAQUS does not have a built-in
**   system of units.  All input data must be
**   specified in consistent units.  The units used
**   here would be consistent for Steel  with
```

```
**  forces in lbf, and lengths in in.
**
**************************************************
**
**  Node definitions
**
**************************************************
**
**  The next statements give the locations of
**  nodal points, corresponding in this example
**  to the "joints" of the truss system.  First
**  entry is node number, and the next are its
**  x1, x2, x3 coordinates.
**
*NODE
 4, 10.0,  0.0, 0.0
 1,  5.0, 10.0, 0.0
 2,  7.5, 10.0, 0.0
 3, 10.0, 10.0, 0.0
**
**************************************************
**
**  Element Definitions
**
**************************************************
**
**  Next, the elements (corresponding to truss
**  bars in this case) are defined.  Also, the
**  type of element, from the element library
**  available within ABAQUS, is defined.  The
**  first number shown is the element number,
**  and the next two are the two nodes which
**  that element joins.  The set of elements is
**  given the name "BARS."  Note that this is a
**  2D element type (T2D2); for 3D analyses, use
**  T3D2.
**
**
*ELEMENT, TYPE=T2D2, ELSET=BARS
 1, 4, 1
 2, 4, 2
 3, 4, 3
**
**************************************************
**
**  Material Definitions
**
**************************************************
**
**  We now describe properties of the material,
**  beginning with the cross section area (0.1)
**  and Young's modulus E (30.E6), in the units
**  adopted.  The Poisson ratio nu is irrelevant
**  here.
**
*SOLID SECTION, ELSET=BARS, MATERIAL=MAT1
 0.1
```

```
*MATERIAL, NAME=MAT1
*ELASTIC
 <E>
**
**********************************************
**
**  Boundary Conditions
**
**********************************************
**
**  Next, come statements about fixed boundary
**  points.  The node number is given first, and
**  then the first and last degree of freedom
**  that is restrained at the node -- displacements
**  U1, U2, and U3 are zero in this case. It was
**  not really necessary to mention U3, since
**  the problem is set up as 2D
**
*BOUNDARY
 1, 1, 3
 2, 1, 3
 3, 1, 3
**
**********************************************
**
**  Step 1:  apply concentrated load
**
**********************************************
**
**  Now we describe the loading, in this case
**  as a single "step."  Since this involves
**  linear elastic material and we are content
**  to neglect any effects of geometry change,
**  due to deformation, on the writing of the
**  equilibrium equations, our problem is a
**  completely linear one.
**
**  We indicate that the  PERTURBATION  procedure
**  be followed to indicate that this is a linear
**  perturbation step. (See discussion in "General
**  and linear perturbation procedures," section
**  6.1.2 of the ABAQUS Analysis User's Manual)
**
**
*STEP, NAME=STEP-1, PERTURBATION
*STATIC
**
**********************************************
**
**  Loads
**
**********************************************
**
**  The following specifies that a concentrated
**  load, <Load>, which was defined above as
**  -10000 in the units adopted, is applied in
**  the x2 direction at node 4 (i.e., 10000 is
```

```
**   applied in the negative x2 direction).
**
*CLOAD
 4, 2, <Load>
**
***********************************************
**
**   OUTPUT REQUESTS
**
**   Note about output:  For large analyses, it is
**   important to consider output requests carefully,
**   as they can create very large files.
**
***********************************************
**
**   The following option is used to write output
**   to the output database file ( for plotting in
**   ABAQUS/Viewer)
**
*OUTPUT, FIELD, VARIABLE=PRESELECT
**
**   The following options are used to provide tabular
**   printed output of element and nodal variables to
**   the data and results files
**
*EL PRINT, ELSET=BARS
 S, E, COORD
*NODE PRINT
 U, COORD
**
***********************************************
**
**   End Step
**
***********************************************
**
**   The final statement tells ABAQUS that loading
**   step is over.
**
*END STEP
```