

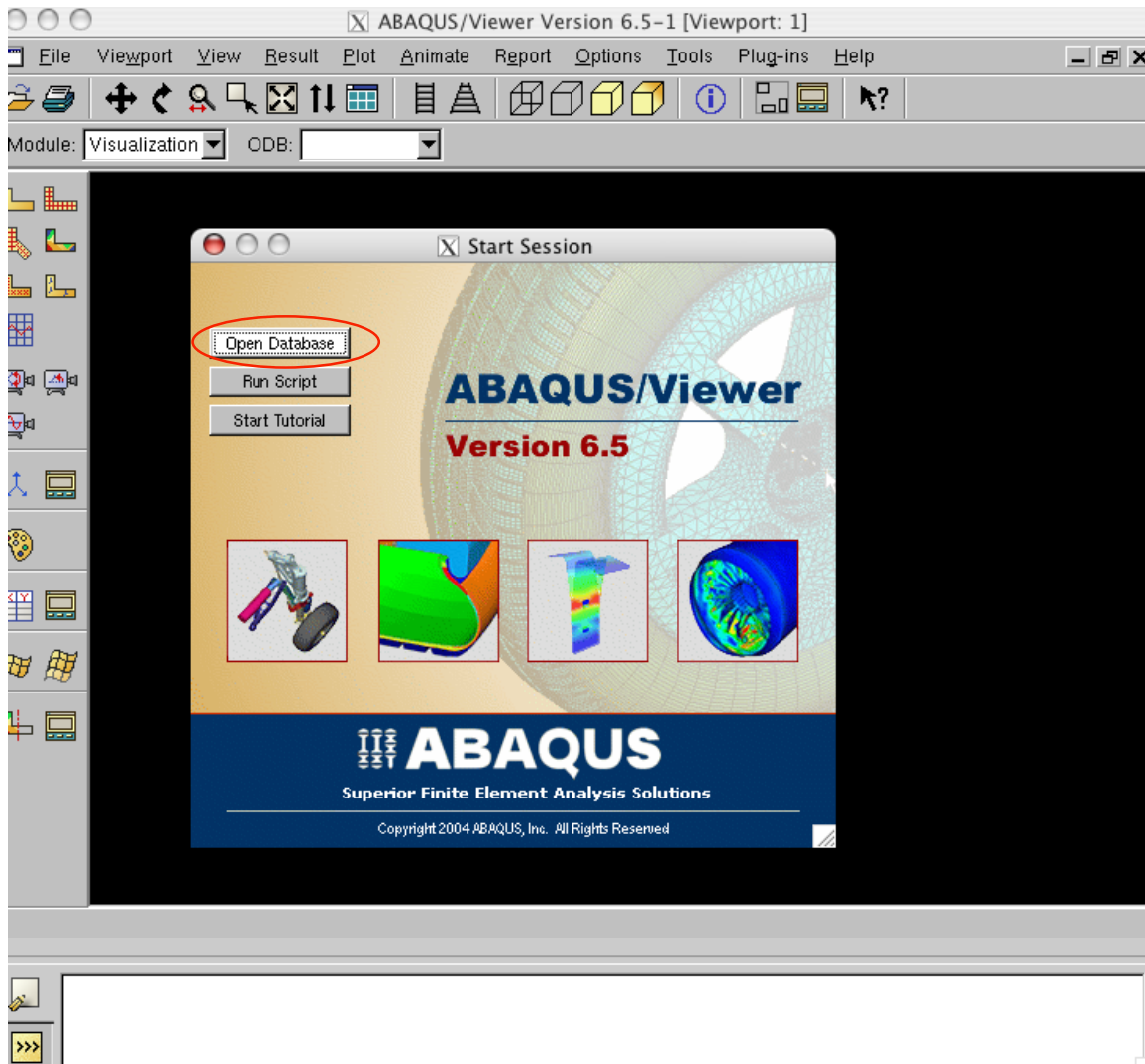
Truss-test.msg: Message file; may contain error messages
Truss-test.odb: Output database. It is written by the **analysis** and **continue** options. It is read by the Visualization module in ABAQUS/CAE (ABAQUS/Viewer).
Truss-test.par: Modified version of original parametrized input file showing input 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.

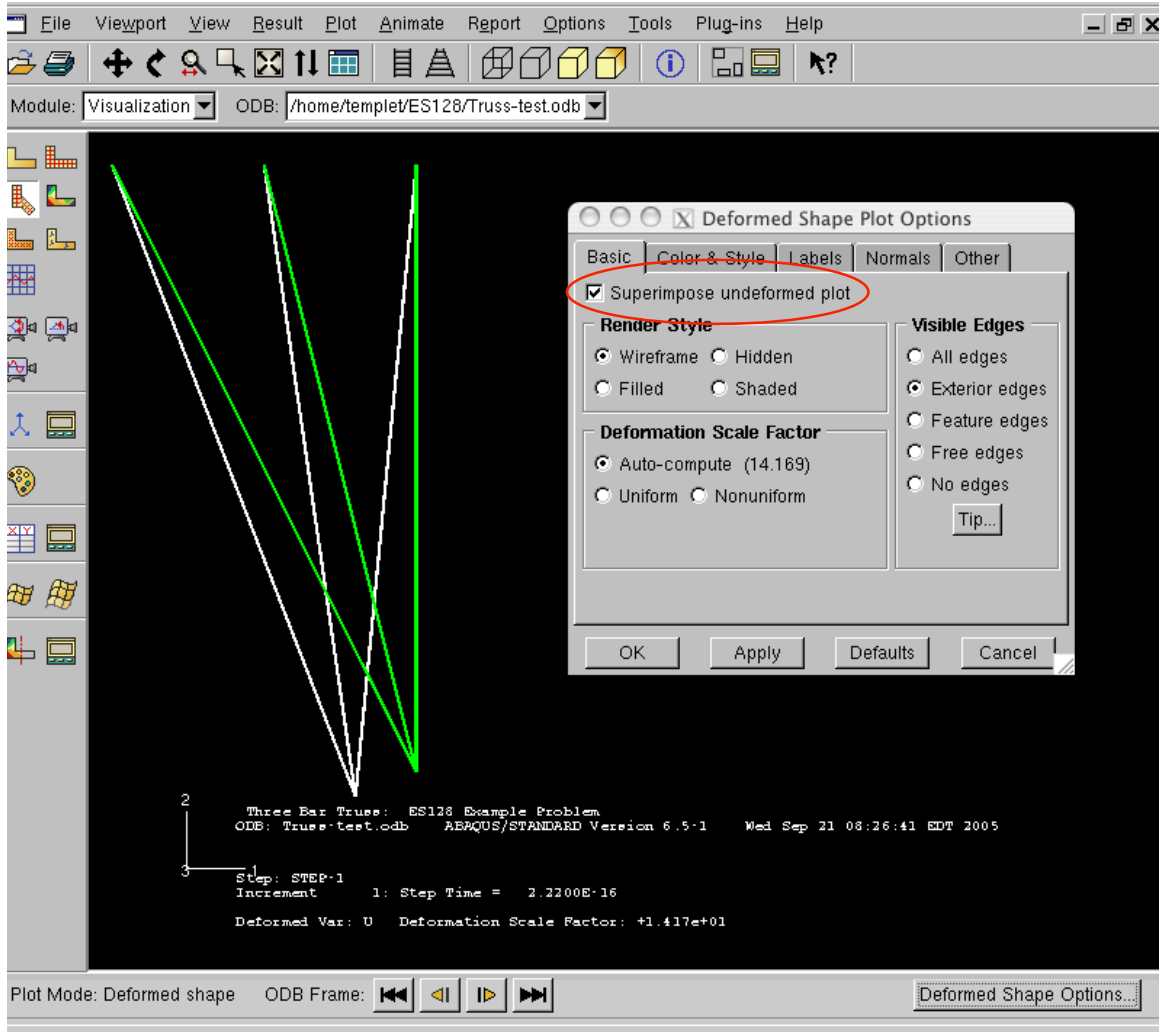
While the analysis is running, there is a file *input-filename.lck* present. When this file disappears, the analysis has ended. After the analysis is complete, first open the .log and look for the line “ABAQUS JOB *jobname* COMPLETED” at the bottom of the file. If an analysis ends with errors, you need 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)



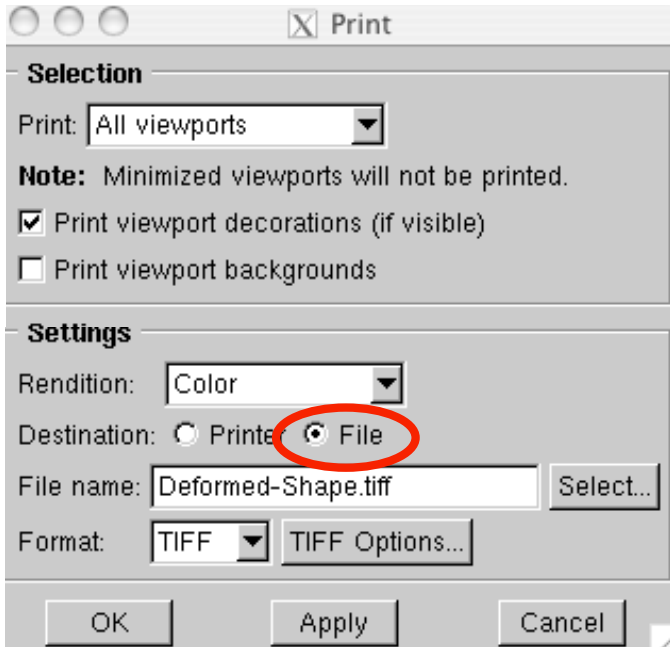
and open the output database file for the truss problem. To plot the deformed shape, select the menu "Plot" then select "Deformed Shape." If you would like to superimpose the undeformed shape on the deformed shape, select the "Options" menu, and select Deformed Shape. In the Deformed Shape Options Dialog box, check the box marked "Superimpose undeformed plot." (see figure on next page for plot of deformed shape)



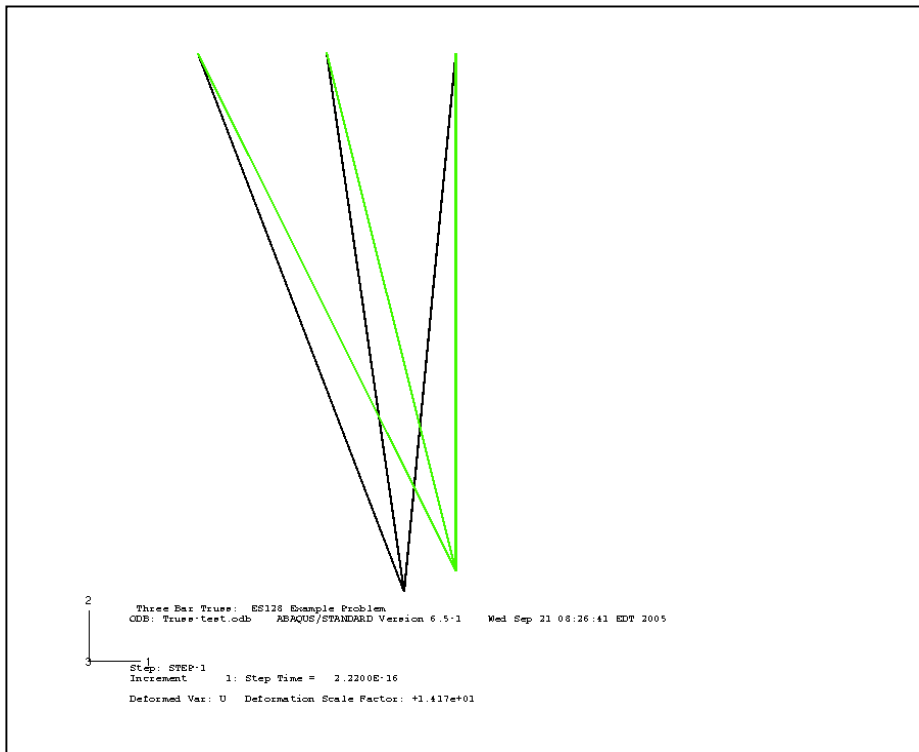
You can plot contours of various variables, such as stress and strain. Explore the options available in Viewer, and try plotting various things.

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
```



```

*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

```