A GiD-FEAP Interface

A. Ghisi

Department of Structural Engineering, Technical University (Politecnico) of Milan, Italy

SUMMARY: FEAP is a Finite Element Analysis Program. It has been implemented by R.L. Taylor and it is widely used in academic environments. In this paper an interface between GiD and FEAP 7.3, implemented by the author both for pre and post-processing purposes is described. The pre-processing phase is performed in GiD. The developed user interface allows for the interactive assignment of data for most of the constitutive laws already present in the original version of FEAP 7.3. At the moment, only small displacement problems with solid elements can be dealt with.

KEYWORDS: FEAP, customization, post-processing, solid elements, small displacements

INTRODUCTION

At the Department of Structural Engineering (DIS) of the Politecnico di Milano (Italy) the professional version of FEAP has been chosen as the basic platform for development of new methodologies in finite element structural mechanics. The release 7.* of FEAP added many new capabilities to the program, which now appears to be an effective tool for structural analysis: it offers a fair choice of elements, formulations and constitutive laws and its modular structure permits a rather easy introduction of new features. It may solve static and dynamic problems, in linear and non-linear structural mechanics. New versions are released on a regular basis adding more options (for example, version 7.4 added other 3D elements). To enhance the use of the program for our purposes, it was decided to develop interfaces between FEAP and an automatic pre-processor and also one or more post-processors. GiD was selected for its simplicity and powerful capabilities.

To accomplish this task, it was necessary:

1) to define two new problem types in GiD 6.2.0.b, one for two-dimensional problems (feap_7_3_2D) and one for three-dimensional problems (feap_7_3_3D). In particular, a *.bas files had to be written in order to create automatically the input files for FEAP.

2) to introduce into FEAP the possibility to create an output database in a format comprehensible for the post-processing module of GiD. This was obtained by defining a new macro (i.e. a new keyword recognized by FEAP) called WRIO (acronym from WRite Input/Output). This macro manages the data flux and creates the new ASCII files for post-processing, named filename.flavia.msh and filename.flavia.res, i the format required by GiD.

3) since in our department data can be post-processed also using TECPLOT™, a second interface between FEAP and TECPLOT™ has been developed. Incidentally, GiD can also post-process TECPLOT™ files. This second interface makes use of a separate module, called DISFEAPPOST: it accesses a database in binary format (a direct access file) and translates the data in a format useful for TECPLOT™ or also for GiD.

FEATURES OF THE PROBLEM TYPES

The customization of the problem types in GiD includes 9 conditions and 11 materials.

Condition	FEAP_7_3_2D	FEAP_7_3_3D
Point_constraint	X	X
Point_load	X	X
Point_plot	X	X
Line_constraint	X	X
Face_load	X	
Point_mass_damper_and_ spring	X	X
Surface_constraint		X
Surface_load		X
Hydrostatic		X

Table 1: condition types

Material	FEAP_7_3_2D	FEAP_7_3_3D
Solid elastic isotropic	X	X
Solid elastic orthotropic	X	X
Solid elastic transversally isotropic	X	X
Solid plastic Von Mises with isotropic/kinematic hardening	X	X
Solid viscoelastic	X	X
Fourier analysis isotropic	X	
Frame elastic isotropic		X
Shell elastic isotropic		X
Membrane elastic isotropic		X
Plate elastic isotropic		X

Table 2: "materials"

In our concept a "material" is a combination of a constitutive law and a finite element family (solid elements/shells/plates, etc.). "Materials" are organized in two books: one for solid elastic "materials" and one for other "materials".

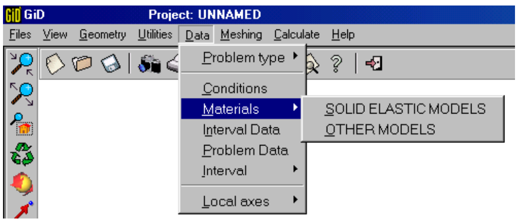


Fig. 1: books of "materials"

Each "material" requires different input for the more commonly used parameters (i.e. Young modulus, Poisson's coefficient) and other, more rarely used parameters, like body forces. It is possible to activate or deactivate the input for these less common parameters by a switch; in this way, if they are not needed, they disappear or are grayed, so the user cannot enter them by mistake in the analysis.

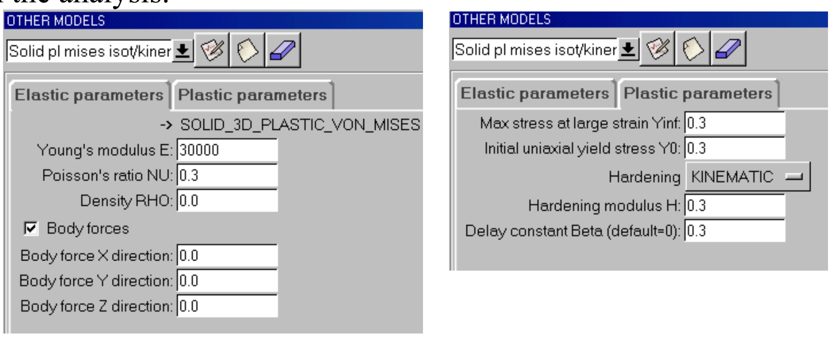
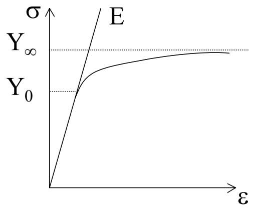


Fig. 2: a "material": Von Mises elastic-plastic law with isotropic or kinematic hardening

An useful feature has proven to be the possibility to add an image to remind the user about the meaning of the symbols appearing in the window, as shown in Fig. 3.



$$Y\left(e^{pl}\right)=Y_{\infty}+\left(Y_0-Y_{\infty}\right)\cdot e^{-\beta\cdot e^{pl}}+H_{iso}\cdot e^{pl}$$

Fig. 3: this image appears in the same window of the parameters and it helps the user to remember what the symbols mean

A condition can be imposed over the geometry or over the nodes, like in the standard version of GiD. The feap_7_3*.bas file translates the model in a FEAP input file named *ifilename* (the prefix "i" is needed for standard input in FEAP): the default is a standard static linear elastic analysis in small displacements. At present it is responsibility of the user to change the input for a more complex analysis, for example a time dependent analysis, but other improvements are currently in progress.
Using the .bat file it is possible to run the analysis directly from GiD (the program automatically creates the FEAP input file) and check the results with the post-processing facilities of GiD (at present only for solid elements).
The element choice in input includes 2D and 3D solid elements (linear and quadratic), Fourier analysis solid element, plate, frame and truss elements. Input for new constitutive laws will be added in the future.

THE FEAP CUSTOMIZATION

A new FEAP macro, named WRIO (WRite Input/Output), has been added to the FEAP vocabulary: it can produce directly either the .flavia.msh and the .flavia.res files (suitable for GiD) or some binary databases. The latter will be processed after the end of execution using an apposite program (named DISFEAPPOST) which creates the files for post-processing, according to one of two different formats, summarized in Fig. 4, below.

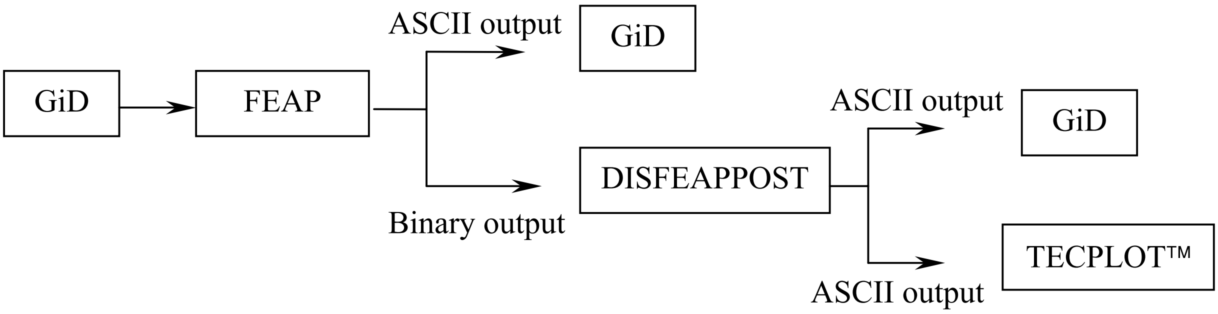


Fig. 4: two different ways to postprocess the results, the first directly from the solver to GiD with an ASCII file, the second through a binary file to two postprocessors.

The macro WRIO has two options, one for creating an ASCII output file for GiD (this is the default) and one for binary file output. The latter facility is completed by another program (DISFEAPPOST) to translate binary data into ASCII file for GiD or TECPLOT™. With this approach it is possible to select after the analysis is finished which time-step or which type of result is needed for visualization. This second methodology is time-saving in presence of large and/or non-linear analyses, where writing ASCII files becomes a very time-consuming

task. The program uses the new post-processing format made available in the new beta release (at present, GiD 6.2.0b) which is needed to visualize the results.
It is possible to combine different element types in GiD, even though not in a straightforward way. For example it is possible to use both quadrilateral and triangular elements in a mesh for two-dimensional solid mechanics problems. This important feature has been kept in the output from FEAP to GiD, with some programming. A new macro was added in FEAP, to manage the output of displacements, stresses, strains and internal variables. FEAP automatically distinguishes between the different element types, each one with its number of nodes and of Gauss points. Now the program builds dynamically allocated lists, where it puts the element numbers corresponding to each different element type. Each one of these is labeled accordingly. For example Quad4_4_GP means quadrilateral elements with four Gauss points, Tetra4_1_GP means tetrahedra with one integration point, and so on. Every different list (in fact, this is a list array, using the facilities of FORTRAN 90) is treated as a different mesh_name in the post-processing file; for each list FEAP automatically puts in the post-processing file the correct definition for the Gauss point type with the gauss_points_name corresponding to the mesh_name.

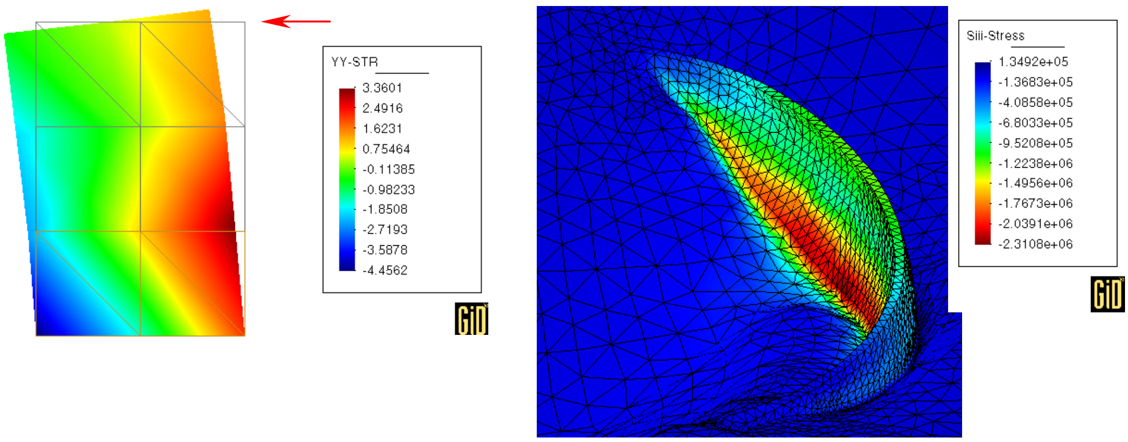


Fig. 5: analysis results on a two dimensional mesh with different element types and on a three dimensional tetrahedral mesh of an arch-gravity dam under self weight and hydrostatic load. Examples obtained with GiD and FEAP.

So the results are obtained from FEAP at nodes or at Gauss points and for different element types in the same mesh, even if visualization at Gauss points at present is limited to one element type only in GiD.

FUTURE DEVELOPMENTS

The developed enhancements represent a necessary preliminary step to allow for the introduction in GiD-FEAP of new capabilities for analyses with nonlocal damage models and possible transition to propagating discrete cohesive cracks. The availability of the source code in FEAP and easiness of problem customization in GiD represents a very promising combination that we hope will greatly contribute to help our research in structural mechanics.

REFERENCES

1. R. Ribó, M.A. de Riera Pasenau, E. Escolano “GiD Reference manual”, (release 6) International Center for Numerical Methods in Engineering (CIMNE), 2001
2. R.L. Taylor, “FEAP 7.3 manuals” (User manual, Programmer Manual, Theory Manual, Example Manual), 2001