Elastic Shell Solver with choice of Multiple Triangular element

(With capability of solving welding distortion also)

Dr.O.P.Gupta
Ex-professor, I.I.T., Kharagpur, India
opgupta112000@yahoo.com

This solver was developed with the objective of providing capability of analyzing stress and deflection in plate /shell structures using the versatile and popular type of triangular elements such as DKT, elements of MITC3 family (MITC3, SMITC and RMITC3) and 'Membrane only' elements The basic formulation utilizes mixed formulation approach wherein both the displacement and rotation are assume to vary linearly within the element. The complete details are available in openly accessible literature elsewhere [1,2,3,4]. Linear distribution of both displacements and rotations are not compatible and in all these elements the compatibility is ensured at few selected points, for example mid-side nodes in case of DKT elements.

The energy expressions are written in the integral form as,

$$\Pi_p = \int (1/2)\{\varepsilon\}^T [D]\{\varepsilon\} dV$$

$$\Pi_b = \int (1/2)\{\varepsilon_b\}^T [D_b]\{\varepsilon_b\} dV$$

$$\Pi_s = \frac{1}{2}\int_A \{\gamma\}^T [\alpha G t]\{\gamma\} dA \qquad (1)$$

Π_p, Π_b and Π_s are the potential energies for planar, bending and shear cases. Here the strain, $\{\varepsilon\}$ and elasticity matrix, [D] are derived for 2D plane stress case while $\{\varepsilon_b\}$ and $[D_b]$ are derived for bending. These are elaborated in any texts on finite element method [5]. $\{\gamma\}$ is transverse shear strain while G and t represent shear modulus and thickness respectively. α is a factor which takes care of the non uniform variation of shear strain over the thickness and its value is taken as (5/6). The net potential energy is given as

$$\Pi = \Pi_P + \Pi_b + \Pi_s - F \qquad (2)$$

Here term F includes work due to external loads, effect of initial strain (generally thermal strain) and initial bending (more relevant in case of simulation of welding distortion). The minimization of net potential energy Π results in linear equation in $\{\delta\}$, (nodal displacement and rotation) and on solving it we obtain nodal values of vector $\{\delta\}$.

The basic options available with this solver, named 'ShellMultiSolver2' are listed here.

Keywords

The following keywords are used in this module and these should be specified appropriately in the .sif file.

Solver solver Id (say 1)

Equation = Shell Solver
Procedure = "ShellMultiSolver2" "ShellSolver"

In addition, other information about solver (Direct or iterative), permissible error etc., are to be supplied (see .sif file in the example)

!Choose type of element

Membrane Only = Logical (TRUE/FALSE)
Use DKT Triangle = Logical
Use SMITC Element = Logical
Use MITC3 Element = Logical
Use RMITC3 Element = Logical

Note: Logical, Real etc. indicate the type of data to be supplied. After this type designation the magnitude of data should be supplied (TRUE/ FALSE for logical).

Another option 'Membrane Only No RM = Logical' is also available which disregards rotation totally. The option available here 'Membrane Only' considers minimization of Transverse Rotation [1] and gives vary accurate results for cases where bending effect is absent such as thin plates under in-plane loading, sphere, cylinder, toroid under internal pressure. When 'Rotation Minimization' is not considered under option 'Membrane Only No RM = Logical TRUE', the results are sometimes quite erroneous.

!Choose type of stress to be calculated

Calculate Stresses = Logical
Top Side Stress = Logical (Default)
Bottom Side Stress = logical
Compute Total Stress = Logical
Compute Membrane Stress = Logical
Compute Bending Stress = Logical

The six components of stress are calculated in global coordinates. In addition, the equivalent stress is also calculated. All the six strain components are also provided as output. Since the bending stresses are different in sign along top and bottom surfaces, it needs to be specified as to where the stresses are to be calculated. Top side will be along the direction of overall normal to the surface, as seen from the mid-plane. This top side may be the exterior side in case of a closed pressure vessel.

For visualization, if we look down towards the surface from the side of the overall outside normal, the nodal sequence of the nodes in the elements should be counter clockwise. This sequence may be different in various parts of the object, meshed as separate bodies (as is the case during mesh generation using software Gmsh). In such a situation the parameter 'Direction of outward normal' (Real, 1.0 or –1.0) is to be supplied in the Body Force section for the different bodies. Value of this parameter will be +1.0 if nodal sequence is counter clockwise, as seen from the overall outside direction (say from outside of a closed pressure vessel). It will be –1.0 if nodal sequence is clockwise as seen from the overall outside direction. Use of this parameter helps the program in calculation of deflection and stresses appropriately along top and bottom sides as defined by the overall outside direction.

Membrane stresses (in-plane stresses) and the bending stresses are calculated separately and any one of these can be the output. Also the total stress, being the sum of in-plane and bending stresses, can be the output depending upon the choice of the relevant option.

Shear Stabilization Parameter = Real
Drilling Stabilization Parameter = =Real
(1.0, if drilling energy is also considered during computation of total potential energy)
Integration Points = Integer ! Number of integration points

Material mat Id (say 1)

Density = real
Poisson Ration = Real
Youngs modulus = real

Body Force bf Id

Pressure = Real (or Normal Pressure = Real)
Body Force 1 = Real (load in global x- direction)
Body Force 2 = Real (load in global y- direction)
Body Force 3 = Real (load in global z- direction)

These Body Forces apply to the whole body (Body as defined in the mesh generation module). Examples could be gravity force, wind force or any other external force. These are to be specified as force per unit volume.

Thickness = Real

It is thickness of the plate or shell type body. This is to be specified for each body and it may be different for various bodies.

Direction of Outward Normal = Real (1.0 or –1.0)

(Default 1.0) (As explained above in Solver section)

Point Load (m,n) = Real

Point load can be specified as concentrated load at a point on the body. It is to be specified as an array of 9 columns and as many rows as the number of points. The quantities in the column are xp, yp, zp, Fx, Fy, Fz, Mx, My, Mz. First three are the coordinates of the point of application of load, which should exactly match with the coordinates of any one of the nodes. (Preferably the corner points of the body should be chosen because the nodes will always be located at the corners.) Next three are the forces in axial directions and last three values represent moment about axes. All 9 components should be specified even though some of these are absent. An example of specifying load Fx at two points is given here.

```
Point Load (2,9) = Real 2.0 0.0 0.0 10000.0 0.0 0.0 0.0 0.0 0.0 \
2.0 0.2 0.0 10000.0 0.0 0.0 0.0 0.0 0.0
```

Boundary Conditions bc Id

Deflection 1 = Real
Deflection 2 = Real
Deflection 3 = Real

Deflection 1, 2, 3 are the displacement restraints in x, y, z directions at the relevant boundary.

Deflection 4 = Real
Deflection 5 = Real
Deflection 6 = Real

Deflection 4, 5, 6 are the rotational restraints about x, y, z axes at the relevant boundary

Force 1 = Real
Force 2 = Real
Force 3 = Real
Force 4 = Real
Force 5 = Real
Force 6 = Real

These Forces apply on the chosen boundary edge. Force 1, 2, 3 are in the direction of global x, y, z- axes. Force 4, 5, 6 are moments acting about x, y, z- axes in the positive direction (clockwise). The magnitudes are to be specified as load per unit length of the edge.

Special Application to Analysis of Welding Distortion

Besides being a general program for using shell elements in structures, this program module was also used for analysis of distortion produced by welding in large plate structures such as ship deck, pressure vessels etc. using a technique known as Inherent Strain Method [6,7]. Briefly stating, this technique involves replacing influence of elastic-plastic changes due to welding by some shrinkage strains and rotation in a narrow region along the weld. Subsequently, elastic analysis of the structure is carried out for whole of the structure under the influence of initial strain (and initial bending) existing in the narrow region so defined. Final output of displacement provides the magnitude of distortion produced by welding.

The brief methodology and KEYWORDS used for this are provided for completeness, although the articles mentioned above should be referred for details.

The data on distortion obtained from experimental correlation [7] on simple welds are to be used as input. These data on four types of distortion, longitudinal and transverse shrinkage, bowing and angular distortion, are used to calculate quantities termed as Initial strains and Initial bending. The longitudinal and transverse strains are distributed as uniform Initial strain in the narrow region around the weld referred above. A convenient method of considering initial strain in longitudinal and transverse direction is to treat it as equivalent thermal strain produced by application of some negative temperature (being shrinkage) under different coefficients of thermal expansion in the two directions, consistent with the magnitudes of shrinkages in these directions (equivalent thermal anisotropy). Bowing and angular distortion are also distributed uniformly in this narrow region as Initial bending 1 and Initial bending 2. The KEYWORDS used in this manner are:

Body Force bf_id

The following information is to be supplied in the Body Force section related to the region around the weld where Initial strain data is to be supplied.

Weld Distortion Simulation = Logical TRUE

Thermal Anisotropy = string "cartesian"

Type of anisotropy, Cartesian, Circular, Cylindrical, Conical, depends upon the shape of the weld, which may be located along a straight line or along circle or on cylindrical surface etc. The two points, which define the longitudinal and transverse directions for these cases are specified according to rules defined for these cases. However, for Cartesian case these two points are points along longitudinal direction of the weld or along a line parallel to it. Point1 1, Point1 2 and Point1 3 are the x, y, z coordinates of first point and Point1 is defined in the same manner.

Point1 1= Real
Point1 2= Real
Point1 3= Real
Point2 1= Real
Point2 2= Real
Point2 3= Real
Alfa 1= Real (Example 0.68046e-05)
Alfa 2= Real (Example 3.124e-05)
Temperature = Real (Example -100.0)

Initial Bending 1 = Real
Initial Bending 2 = Real

Alfa1 and Alfa2 are the values of equivalent coefficients of thermal expansion. The other KEYWORDS are to be used in the manner as defined for the case of general shell element analysis.

Test Examples

Few test examples are stored in the folder on examples. Each sub-folder in the example folder contains all the files used for analysis of the problem. These files are either supplied by user or generated by GUI of Elmer software. Mesh file (.msh) were generated using open source meshing software ‘Gmsh’. Sif file was first generated in part using the GUI and then it was modified as per the requirement of the problem using the text editor in GUI. A study of these will be helpful in understanding the use of various KEYWORDS.

Some explanation of these examples and results is provided in 'Readme' file incorporated in these folders.

Examples

The results from several examples, which are elaborated in the example folders, are reproduced here.

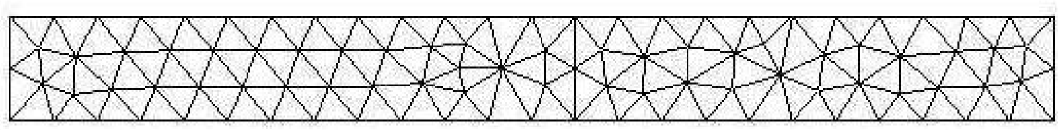

Fig.1 a

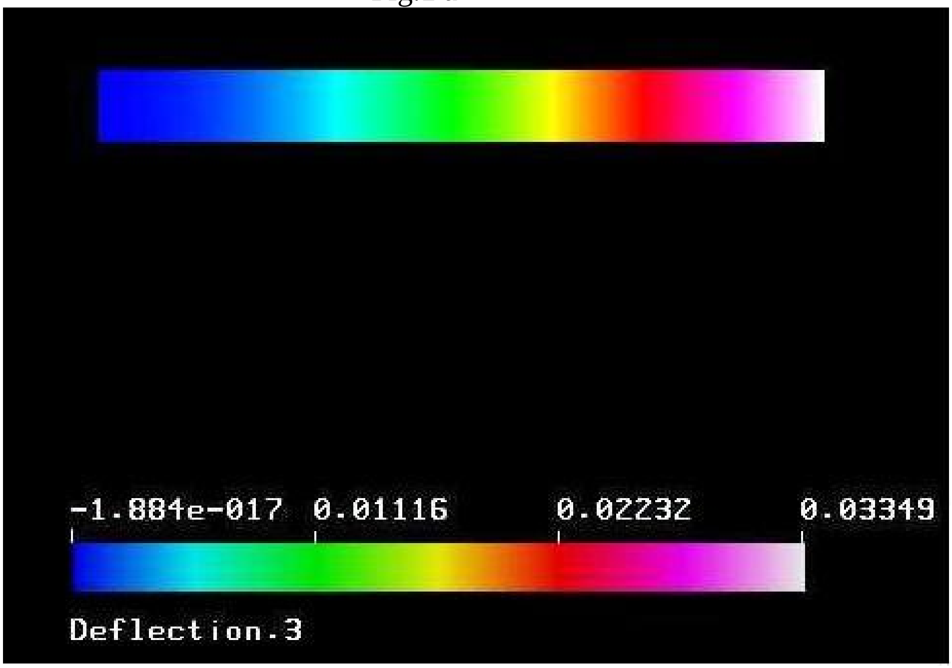

Fig 1b

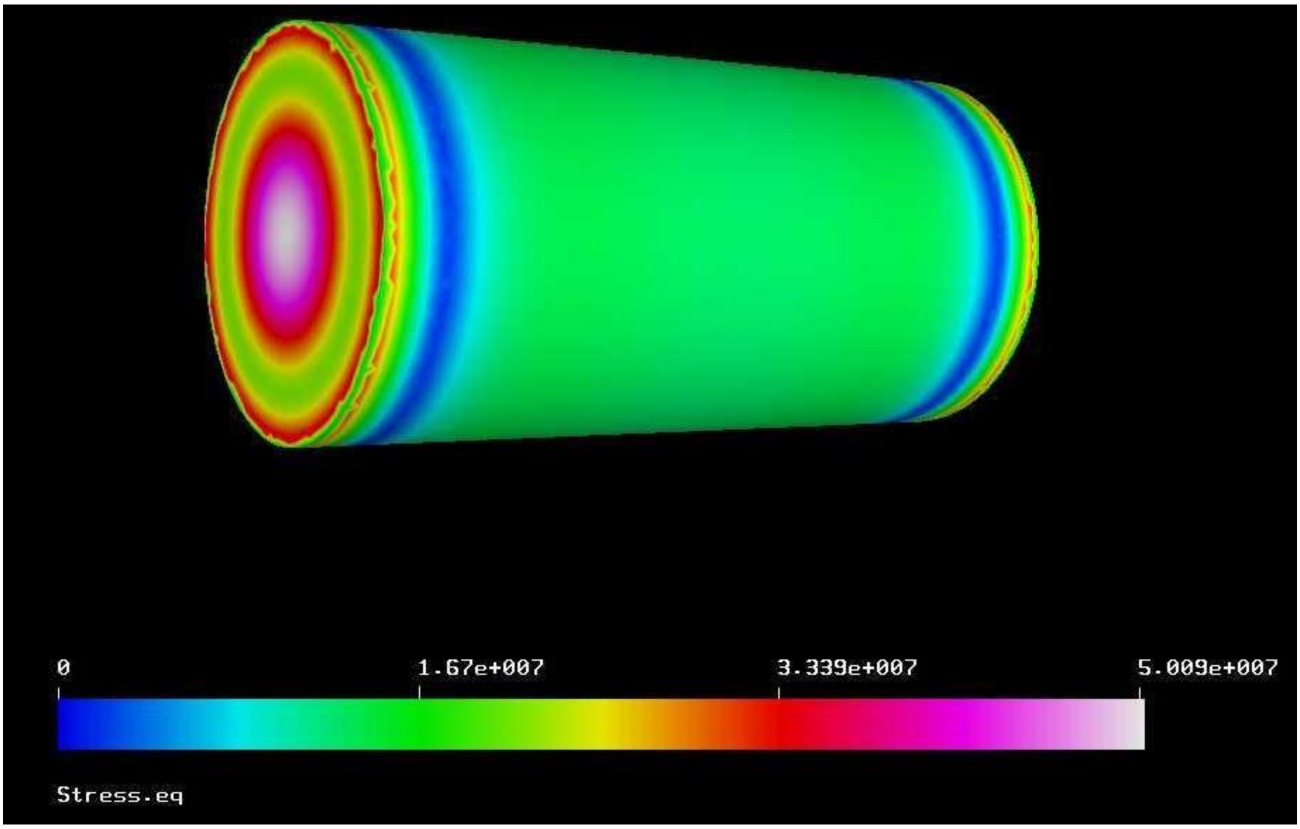

Fig 2

Fig. 1 a shows the triangular mesh for a cantilever beam 2m long and 0.5m wide. The thickness is 0.01m and it is subjected to uniformly distributed load of 20 N/m-length. The deflection transverse to the plate is shown in Fig 1 b.

The second example, shown in Fig.2, represents the equivalent stress on the outer side of a closed cylindrical vessel 2 m long, 0.5 m radius, 0.1 m thick subjected to internal pressure of 3.0e06 N/m-sq. DKT triangular element has been used in this case, although any other element will also provide equivalent results. Complete details of both these examples are present in the example folders.

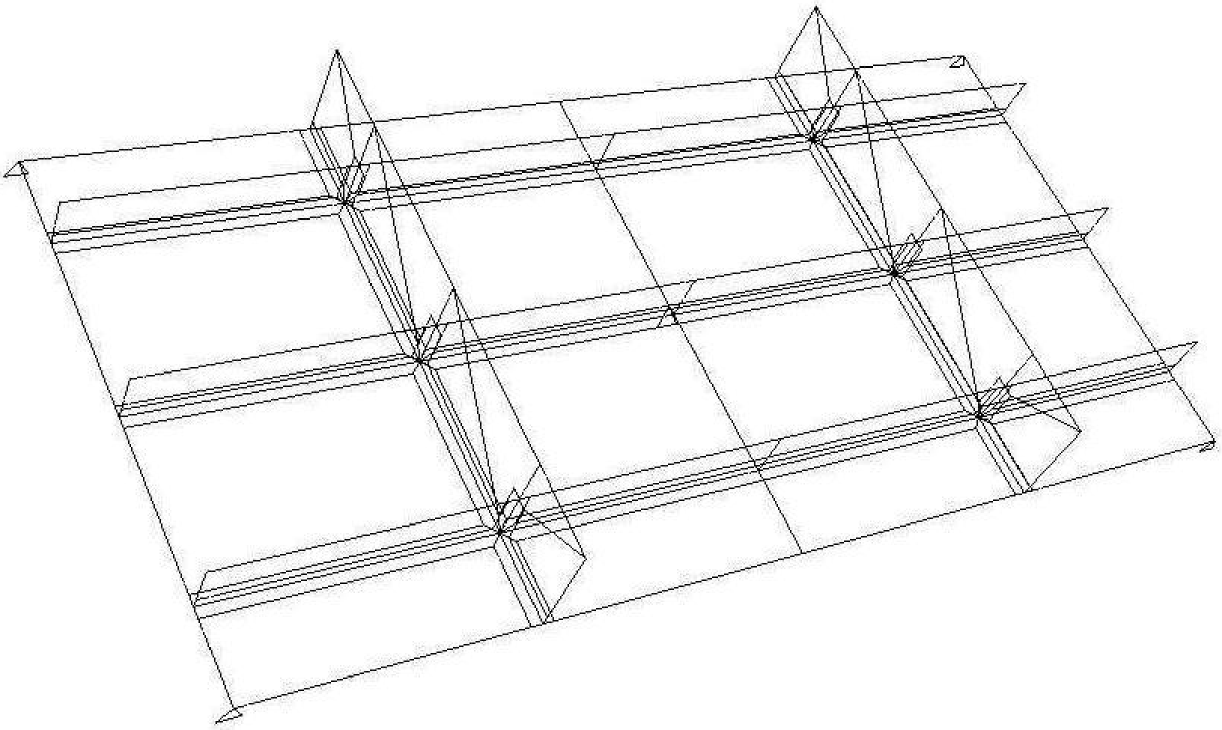

Fig. 3 a

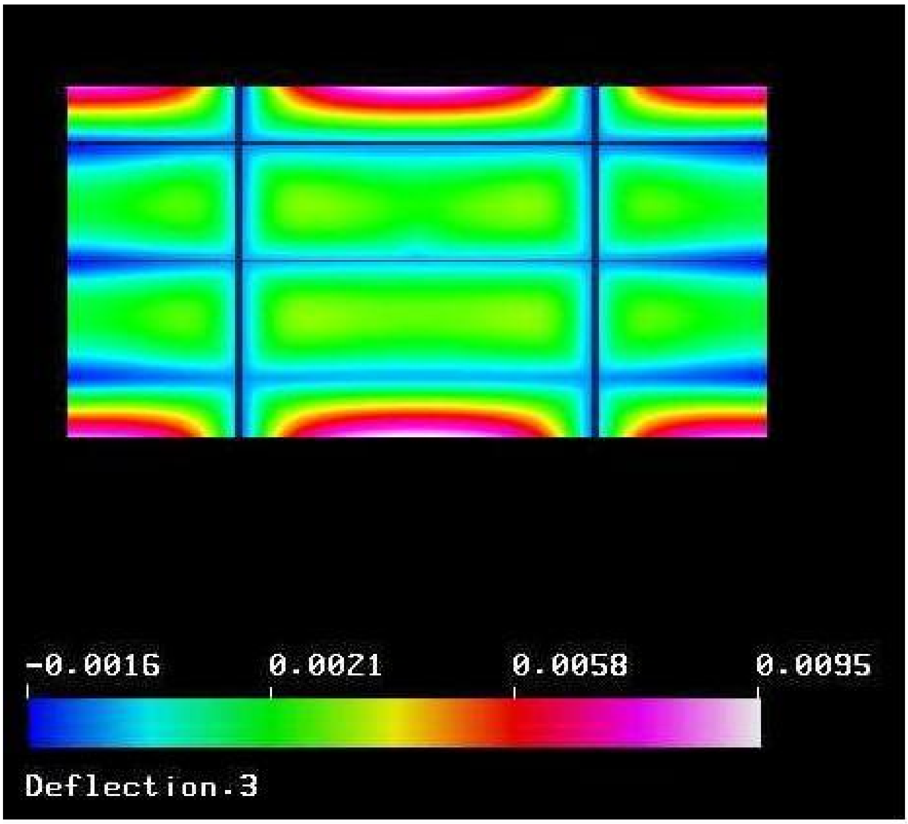

Fig. 3 b

The third example shows special application of the shell elements for determination of distortion produced by welding in a simulated ship panel structure. The actual example and the experimental values are reported in ref [6]. The panel, 3m x 1.5 m, which has several stiffeners welded to it, is shown in Fig 3a. The Inherent strain method is used to simulate welding and for this purpose shrinkage strains and bending, equivalent to the effect of welding, are introduced in the region around the weld. These are related to the welding parameters. To obtain this the whole structure is divided in number of plate sections so the appropriate body forces could be introduced there. A total of 133 plate sections were used and these are shown in Fig. 3 a. The distortion so produced is reported in Fig 3 b. The values are reasonably close to experimental values [6,7]. This example is also stored in an example folder.

References:

1. Gupta, O.P., Analysis of Closed Shell Structures Subjected to Internal Pressure using Membrane Element with Rotation Minimization Implemented Through a Special Module Developed for Elmer Open Source FE Software, https://www.researchgate.net/publication/270956242_Analysis_ , January 2015
2. Gupta, O.P., Note and Explanation of Formulation for DKT Shell Element and its Implementation through Open Source Finite Element Software, Elmer, https://www.researchgate.net/publication/276355469_Note_and , May 2015, 17 pages
3. Gupta,O.P., Note and explanation of formulation for MITC shell element and its implementation through open source finite element software, Elmer https://www.researchgate.net/publication/275946862_Note_ ,May 2015, 18 pages
4. Gupta, O.P., Evaluation of Performance of Triangular Shell Elements of MITC3 Family (MITC3, SMITC and RMITC3) and DKT Element – Applications on Plates and Pressure Vessels, https://www.researchgate.net/publication/295525759_Evaluation_ , March 2016, 16 pages
5. Gupta, O.P. (1999) Finite and boundary element methods in engineering. Taylor and Francis publishers, London, pp 157-163, 173
6. Deng, D., Murakawa H.,Liang, W, Numerical simulation of welding distortion in large structures, COMPUTER METHODS IN APPLIED MECHANICS AND ENGINEERING · SEPTEMBER 2007, p 4613-4627
7. Gupta, O.P., Analysis of Distortion Produced by Welding of Complex Ship Panel Structure Using Shell Module of Open Source FE software, Elmer – To be published , https://www.researchgate.net