

Manual

Step-by-step procedure for nonlinear dynamic analysis in Elmer

The steps involved to find the natural frequency of the nonlinear undamped beam (without damping) are described with figures.

- A model to be analyzed is created in 'FreeCAD'.
- 'FreeCAD' model is meshed in 'Gmsh' using tetrahedron elements and saved as 'beam.msh'. To perform meshing the following general steps are followed in Gmsh:
Open Gmsh> File > Open model > Expand 'Mesh' > Expand 'define'> Expand 'Transfinite' > select Curve > select edges of the model to be discretised with required number of elements > 'Recombine' with '3D'. After generating the 3D mesh the model with mesh is exported and saved with '.msh' extension. The whole process is shown in Fig. 1.
- The 'beam.msh' is opened up directly in the graphical user interface (GUI) of 'Elmer' as shown in Fig. 2.
- The material property, load, and output requests can be assigned to the member in two ways. The new properties are added directly from the 'Model' section of graphical user interface as shown in Fig.1. Another way is to define those properties in the environment of '.sif'. The 'sif' (solver input file) tab can be accessed with the 'Edit' option (Fig. 3).

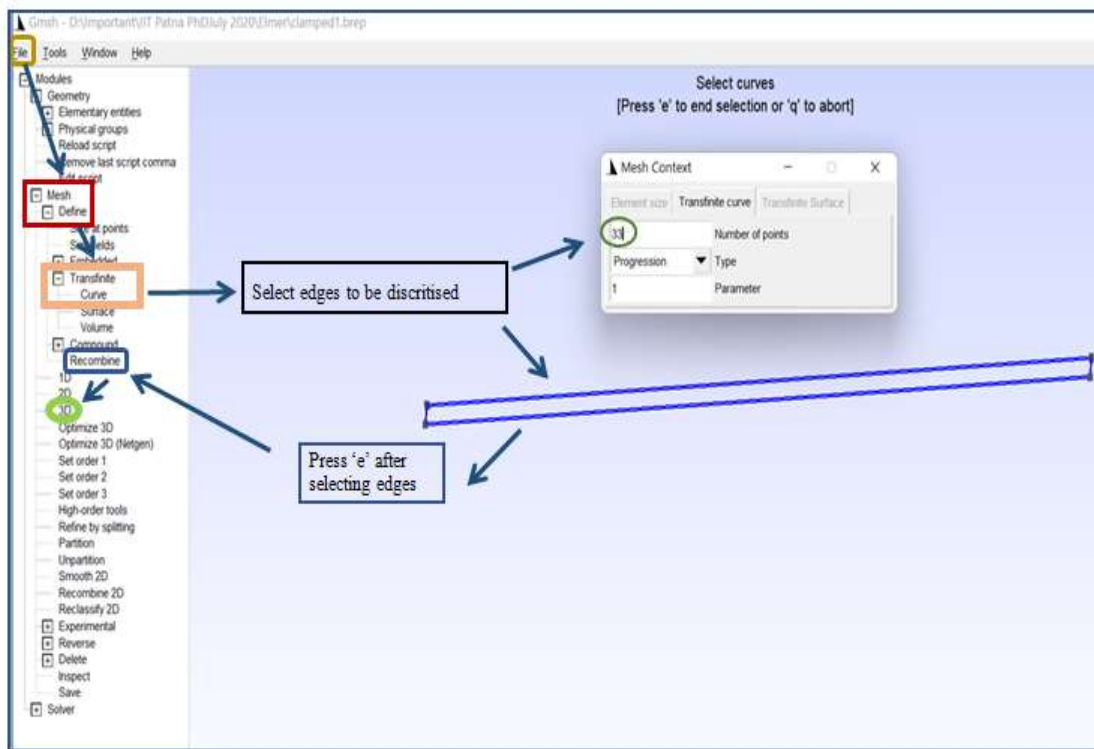


Fig. 1. Flow of work to mesh a model using Gmsh

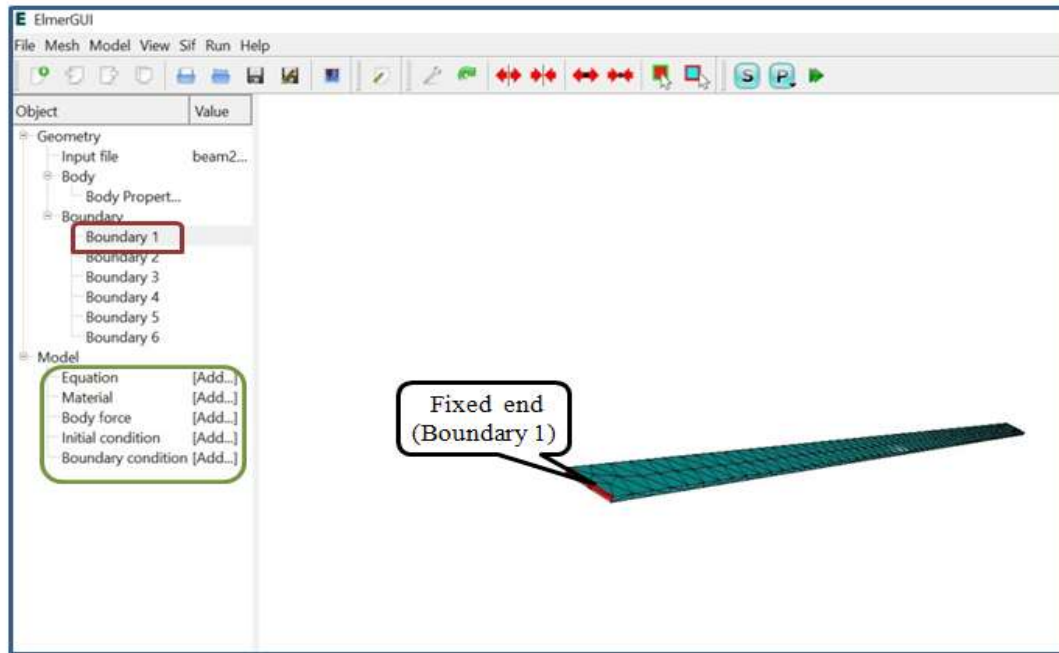


Fig. 2. Graphical user interface of Elmer

d) The nonlinear undamped beam is modeled with the property of aluminium. In case of damped analysis, Rayleigh coefficients are added as shown in (Fig. 4). In the .sif file, the assigned properties are checked and saved.

e) In a cantilever beam, all degrees of freedom of the end boundary are restricted to zero. It is modeled by assigning the displacement value of the first node of element 1 to zero (Fig. 5).

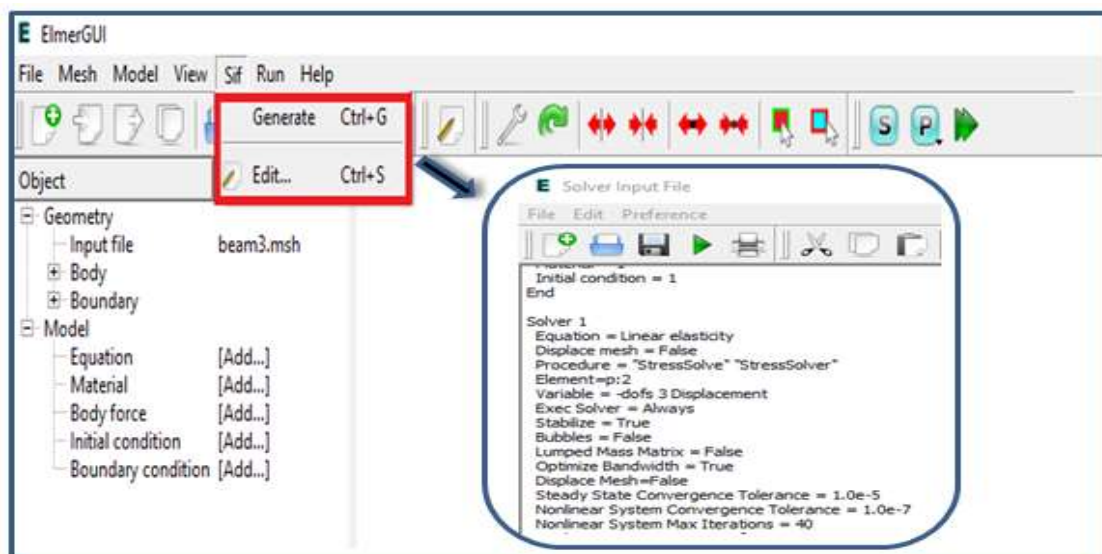


Fig. 3. Tab to enter .sif environment

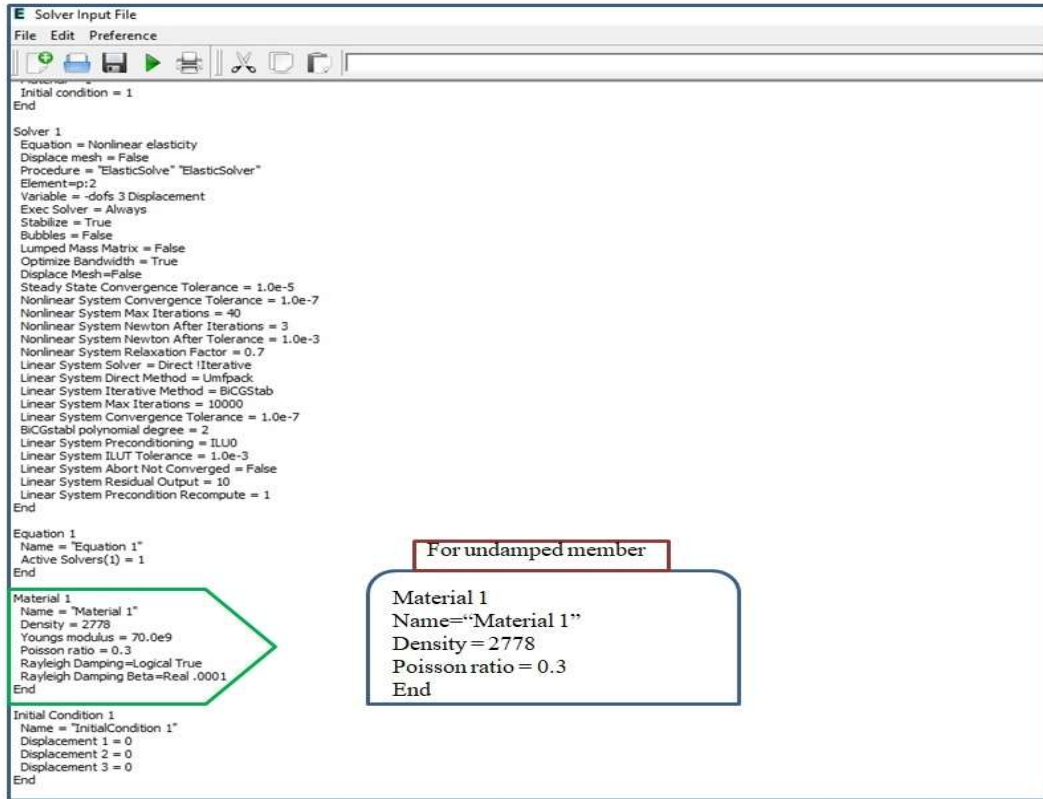


Fig. 4. Material section in .sif environment for both damped (green) and undamped member (red)

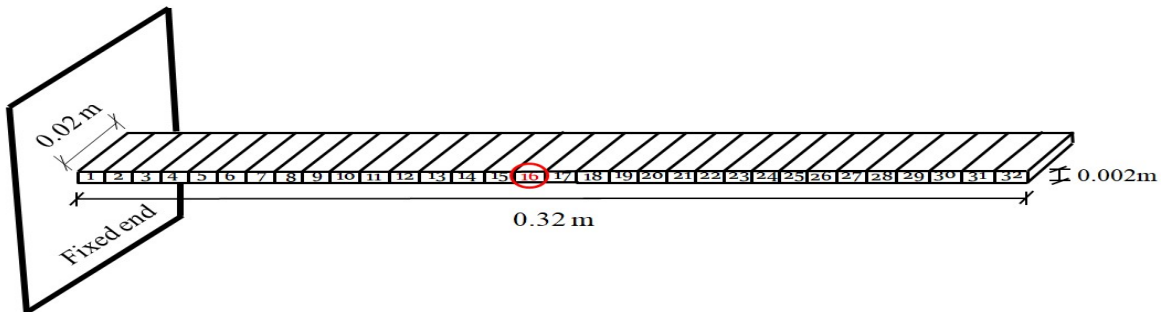


Fig. 5. Physical description of a model of undamaged cantilever beam

- f) In the .sif file, the boundary of the model is identified by the 'Target boundaries' number as in Fig. 6 of graphical interface of 'Elmer'. The associated number with 'Displacement' in Fig. 4 and 6 denotes the corresponding direction of the axis where '1', '2' and '3' represents 'x', 'y' and 'z' axis respectively. To modify the value of boundary condition, default values can be directly edited in the coding. The 'Target Boundaries' in Fig. 6 indicates the boundary number to be operated on.

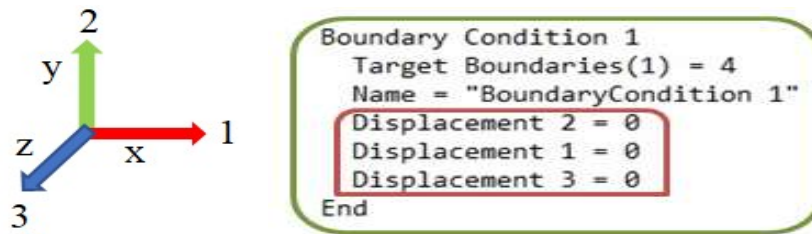


Fig. 6: Boundary condition in .sif environment

- g) The axes direction for the corresponding model is obtained from the ‘compass’ which opens up with ‘view’ when right clicked on the GUI as shown in Fig. 7.
- h) To find the natural frequency, ‘Stress solver’ with ‘Linear Elasticity’ (Fig. 8) equation is used and required number of modal values is set in ‘Eigen System Values’. For nonlinear analysis, ‘Linear Elasticity’ in ‘equation’ is replaced by ‘Nonlinear elasticity’.

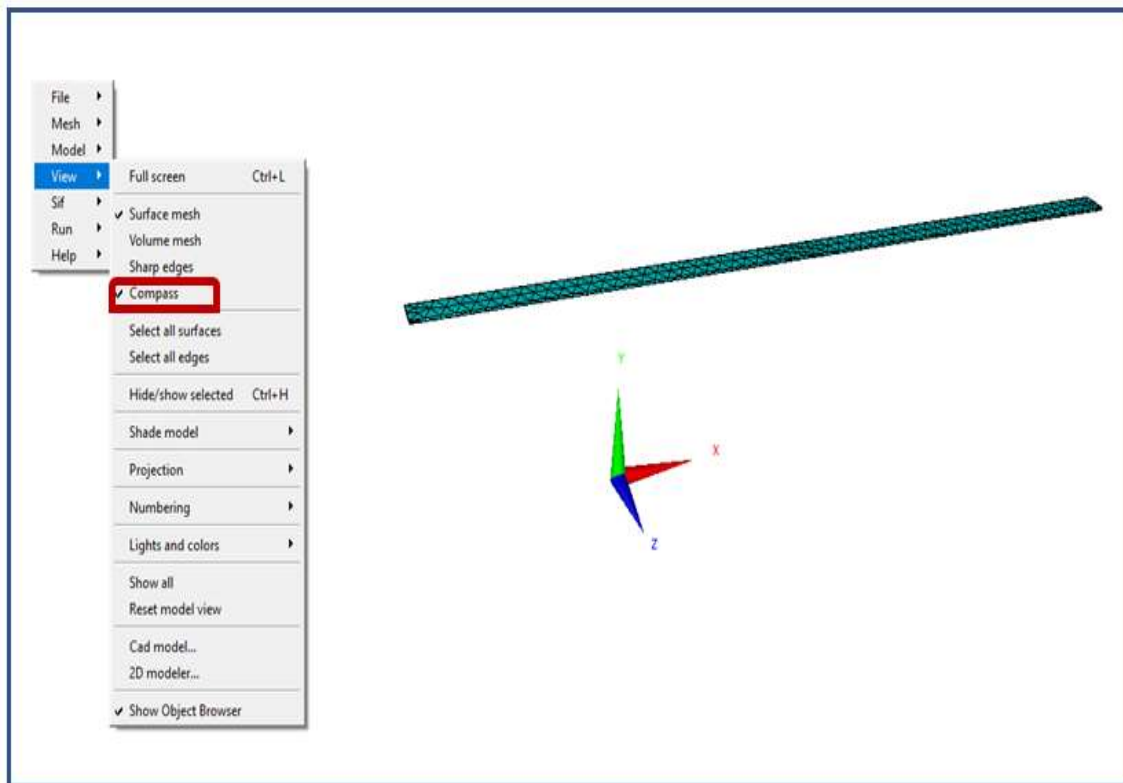


Fig. 7. Description of axes directions of the model

```

Solver 1
Equation = Linear elasticity
Variable = -dofs 3 Displacement
Geometric Stiffness = True
Eigen Analysis = True
Eigen System Select = Smallest magnitude
Eigen System Values = 5
Procedure = "StressSolve" "StressSolver"
Element=p:2
Exec Solver = Always
Stabilize = True
Bubbles = False
Lumped Mass Matrix = False
Optimize Bandwidth = True
Steady State Convergence Tolerance = 1.0e-5
Nonlinear System Convergence Tolerance = 1.0e-7
Nonlinear System Max Iterations = 20
Nonlinear System Newton After Iterations = 3
Nonlinear System Newton After Tolerance = 1.0e-3
Nonlinear System Relaxation Factor = 1
Linear System Solver = Direct
Linear System Direct Method = Umfpack
End

```

Fig. 8. Solver section in ‘.sif’ environment

- i) The simulation is performed under ‘steady-state’ solver in case of any dynamic analysis. It is also required to define the input file name so that whenever the file is referred to after saving last modified coding, will appear in ‘.sif’ [7] file for the corresponding member. The output file name can also be modified and will be saved in ‘.vtu’ [1] format for post-processing of result. The solver will ask for the intervals that need to be stored in the output format as per the requirement (Fig. 9).

```

Simulation
Max Output Level = 5
Coordinate System = Cartesian
Coordinate Mapping(3) = 1 2 3
Simulation Type = Steady state
Steady State Max Iterations = 1
Output Intervals = 1
Timestepping Method = BDF
BDF Order = 1
Solver Input File = case.sif
Post File = case.vtu
End

```

Fig. 9. Section to define simulation type in ‘.sif’ environment

- j) After modifications, the file is run for the analysis, and output is stored in ‘.vtu’ file. The natural frequency is obtained from the square root of the eigenvalues.

Forced vibration analysis of damped beam using Elmer

The same .sif file can be used for the dynamic analysis of the member with some modifications in the simulation method. The geometric stiffness of the member is considered automatically by the software package during the dynamic analysis when the elastic solver is applied. On the other hand, linear analysis is performed by renaming it to ‘StressSolver’. The following changes are also made for the damped vibration of beam subjected to external harmonic loading.

- a) To add damping in the beam, the Rayleigh damping coefficient is defined in the material section of ‘.sif’ file as shown in Fig. 10(a). The loading environment is modified with harmonic loading

and target boundaries are fixed by respective boundary numbers as per modeling shown in Fig.10(b). Here 'MATC' refers to the library of solutions for time-varying excitation function.

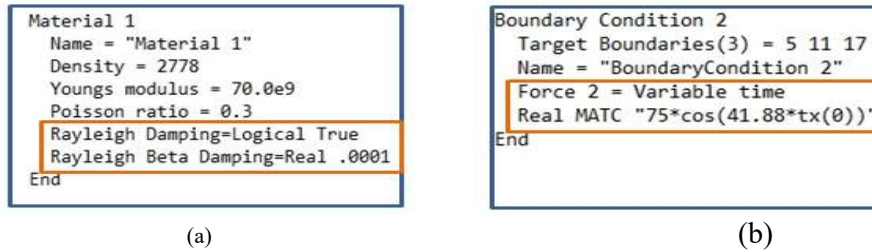


Fig. 10. Sections to define (a) damping parameter and (b) excitation force in the .sif file

- b) The analysis is intended to perform for 't' s with 'dt' s interval. Accordingly, the time step interval with total array size of t/dt is added in the simulation part. Any dynamic analysis in 'ELMER' is performed by 'Transient' type of simulation instead of 'Steady state' used in case of natural frequency determination (Fig. 11).

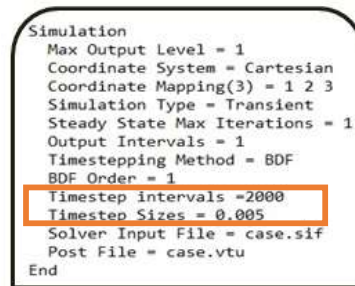


Fig. 11. Section to define analysis time-interval

The post-processing of the output result is performed in 'Paraview' instead of its own environment for the detailed response history of the member with better graphical interface.

Same process is applied for the analysis of the axial bar where harmonic load is applied along the longitudinal direction of the member at the free end. The output history of displacement is finally extracted for both beam and axial members to review the performance of different damage indicators.

All the results are saved in .vtu format which can be opened and visualized in 'Paraview'.

Poincaré Map from Elmer result

It is developed with matlab code 'Poincare_map.m'. The inputs required for the plot are displacement, velocity and acceleration of the intended node.