Chapter 6

Finite Element Processing
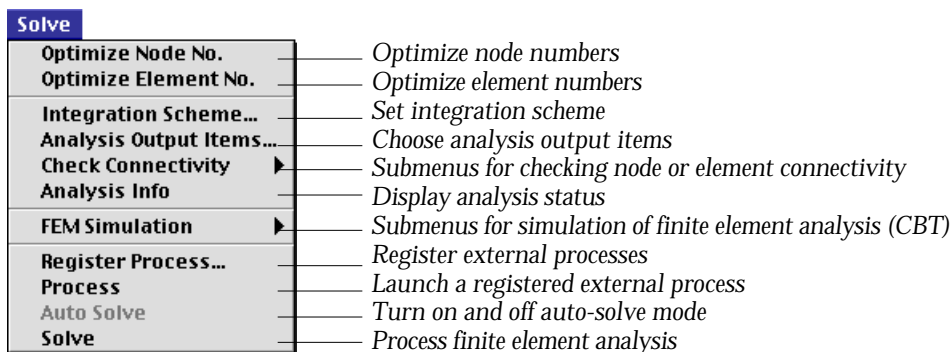
Chapter 6   Finite Element Processing

Chapter 6   Finite Element Processing

Finite element analysis consists of several steps from creating geometric coordinate data to visualizing the analysis results.

The core of the finite element analysis is to form the system equations using the data prepared as described in the previous chapters, and to solve them. This procedure is called "processing of finite element analysis," and termed here as "finite element processing." Among all the procedures in finite element analysis, this is the one which requires minimum user interaction. Once the processing begins, it goes on continuously by itself to the end without requiring user intervention. However, the processing is the step which involves the most intensive computation. This usually demands huge computational resources in terms of computing time and memory space. The computing time for processing varies largely depending on the size of the problem: from less than a second to several hours or more.   The computing time as well as the memory space required for finite element processing increases drastically as the size of the problem becomes larger, in going from 2-D to 3-D, linear to nonlinear, static to dynamic.

Computational efficiency in terms of computing time and memory space is an important issue in the finite element processing. There are a number of factors affecting computational efficiency. Either node numbering or element numbering depending on the solution scheme is one of the major factors determining the efficiency. Optimizing the node numbering or the element numbering is an essential step which should be taken prior to the processing. Accuracy as well as computational efficiency is affected also by integration schemes and element types employed in the processing. Therefore, they should be manipulated by the users for the best accuracy and efficiency.

This chapter describes the usage of functions related with finite element processing and its computational efficiency. These functions are arranged as items of **Solve** menu as shown in the figure below.

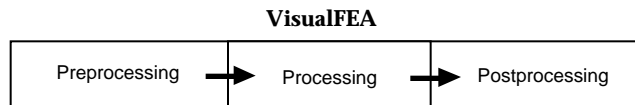| Solve	
Optimize Node No.	*Optimize node numbers*
Optimize Element No.	*Optimize element numbers*
Integration Scheme...	*Set integration scheme*
Analysis Output Items...	*Choose analysis output items*
Check Connectivity ▶	*Submenus for checking node or element connectivity*
Analysis Info	*Display analysis status*
FEM Simulation ▶	*Submenus for simulation of finite element analysis (CBT)*
Register Process...	*Register external processes*
Process	*Launch a registered external process*
Auto Solve	*Turn on and off auto-solve mode*
Solve	*Process finite element analysis*

Getting Finite Element Solution

Assembling and solving finite element equations involve many complicated steps and a huge amount of computation. For you as an end-user, however, it is usually a simple single step procedure. All you have to do is to choose an appropriate menu item. In spite of this simplicity, it is important for you to understand the mechanism of this procedure in order to get a proper solution with maximum efficiency.
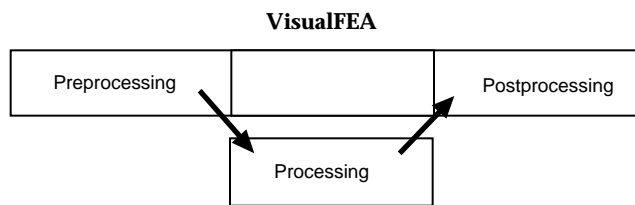
Solver of finite element analysis

There is software specialized in finite element processing. They are commonly called "solver." VisualFEA has the capability of such as solver embedded in itself. However, you may use VisualFEA together with other solvers provided by third party, or developed by yourself. In other words, you may use VisualFEA for creating a finite element model (preprocessing) and visualizing the analysis results (postprocessing), and use other solvers for processing.   Such a finite element processor is termed here as "external solver."

There are two paths for doing finite element analysis with VisualFEA: one is relying only on VisualFEA through the whole procedures, and the other combining an external solver for processing and VisualFEA for pre- and postprocessing.

VisualFEA

| Preprocessing | → | Processing | → | Postprocessing |

Use of intuitiveFEM for whole procedures

VisualFEA

| Preprocessing | | Postprocessing |

Processing

External   Solver

Combined use of VisualFEA and an external solver

<Alternative procedures of FEA using VisualFEA>

The procedure of finite element analysis based only on VisualFEA is simpler, more straightforward and more efficient than that based on the combined usage of VisualFEA and an external solver.   However, the solver embedded in VisualFEA may not cover the range of problem you want to solve. In such a circumstance,

you should employ an external solver.

It is your responsibility to make data interface between VisualFEA and external solvers, unless there exists software interfacing them.   The data interface is described in Chapter 10.


Processing of structural analysis

The simplest method of getting a finite element solution is to use VisualFEA's own processing capability. The finite element solver's capabilities are embedded within VisualFEA. This makes VisualFEA much simpler to use than other finite element programs which has separate module for processing, preprocessing and postprocessing.

In order to get into the processing stage for solution, choose "Solve" item from `Solve` menu Then, "Analysis Options" dialog appears on the screen. The contents of the dialog vary depending on the type of solution as will be described below. Set the dialog items as desired and click `O.K.` button. Then, the processing starts, and goes on up to the completion of all the necessary computation including assembling the system equations and solving them.

■ Setting analysis options for linear static analysis

Choosing "Solve" item from `Solve` menu will pop up "Analysis Options" dialog as shown below, if you have set the solution type as linear static. *The solution type is initially set as linear static, and remains as it is, unless you have checked any one of the check boxes under "Solution Type", i.e., "Material nonlinear", "Geometric nonlinear" and "Dynamic."*
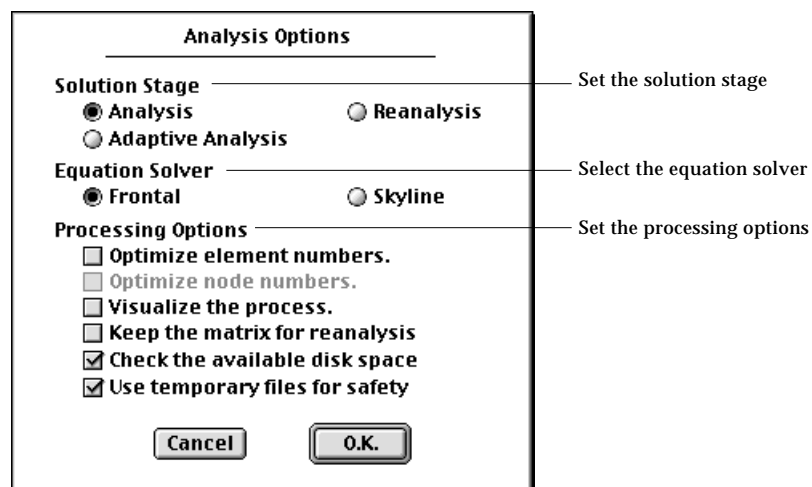
The dialog has a few items setting the options related to the finite element solution procedure.

- solution stage: The solution stage may be set as one of "Analysis," "Reanalysis" and "Adaptive." If you turn on "Adaptive" radio button, the solution procedure turns into adaptive process. The analysis options for adaptive analysis are described in the next section. So, only "Analysis" and "Reanalysis" options are described in this section.

  The initial default setting of the solution stage is "Analysis", which leads to the normal procedure of processing based on assembling equations and solving them. Once the system equations are assembled and solved, they may be used for subsequent processing by setting the option as "Reanalysis". The reanalysis option will eliminate most of the computing time for assembling and decomposing the system equations. The reanalysis option is enabled and valid only under the following conditions.

  - The system equations should have been solved in the previous session.
  - The files containing the system equations should exist in the same directory (folder) as the data file.

    In order to keep these file for reanalysis in next sessions, check "Keep the matrix for reanalysis" item in the "Analysis Options" dialog.

  - Geometry, element properties, and boundary conditions should not have been altered since the system equation files were created.

    Loads may be changed.

  - The analysis type is linear static, and not adaptive nor sequential.

- equation solver: You may choose one of two methods of assembling and solving equations: frontal method and skyline method. They are typical and the most widely used methods in finite element analysis solvers.

  Skyline method uses only CPU memory, while frontal method relies much on auxiliary memory such as a hard disk. Accordingly skyline method demands much larger CPU memory space than frontal method. Skyline method usually works faster than the frontal method which requires frequent reading and writing with the auxiliary memory.

  The default setting is "Frontal." If your computer is equipped with huge CPU memory, and you want faster solution, choose "Skyline." Otherwise, you should keep the option as "Frontal."

  In case you chose "Skyline," but the memory space is not sufficient, the software will notify this by the following message box. If you click [O.K.] button of the dialog, the software will automatically switch the solver to "Frontal."
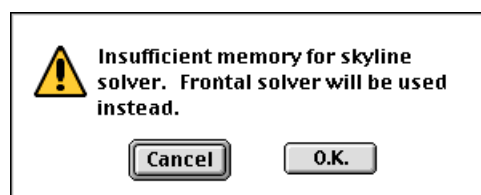
- processing options: You can turn on or off each of the processing options by clicking the check box in front of each item.   These settings are applied during the processing stage.
  - "Optimize element number" : This item is enabled only when the equation solver is set as "Frontal". If this option is turned on, optimization of element numbering is automatically done prior to assembling the system equations.
  - "Optimize node number" : This item is enabled only when the equation solver is set as "Skyline". If this option is turned on, optimization of node numbering is automatically done prior to assembling the system equations.
  - "Visualize the process" : If this option is turned on, a graphical rendering of the model is provided along with the status of the element stiffness matrix assembly.
  - "Keep the matrix for reanalysis" : The system equation files are created at the start of matrix assemblage and removed at the end of the processing. In order to keep these files for reanalysis, this option should be turned on.
  - "Check the available disk space" : If this option is turned on, the available disk space is checked while processing is going on. If the disk space is not sufficient, the processing will pause with the following notice so that you may secure enough space and resume the processing.

  - "Use temporary file for safety" : If the processing is abnormally interrupted due to system failure or any other reasons, the data file may be spoiled or lost. In order to avoid such risks, turn on this option. Then, a duplicate of the data file will be created temporarily and used during the processing, and it will replace the original file when the processing is successfully completed.

The processing procedure is initiated when you click [O.K.] button of the dialog, after setting all the appropriate items.

■ Setting analysis options for adaptive analysis

If you click "Adaptive analysis" radio button of "Analysis Options" dialog, the dialog expands with additional items as shown below.   The upper part of the dialog has the original items, and the bottom half includes new items as follows:

- termination criterion: The adaptive iteration continues until one of the conditions set for its termination is satisfied.   These conditions are termed here as the termination criterion.   You may validate or invalidate each one of the following termination criteria by checking or unchecking the boxes in front of them. If the box is checked, the corresponding criterion is applied.
  - number of iteration cycles: The number of iteration cycles can be restricted

by checking this item, and setting the number. Iteration terminates when the
number of cycles reaches the number.
- energy norm error: Iteration terminates when maximum energy norm error
  over the whole solution domain gets smaller than the criterion set as the
  limiting energy norm error.

If both of the termination criteria are checked, iteration terminates when any
one of them is fulfilled.

Analysis Options

Solution Stage
○ Analysis   ○ Reanalysis
● Adaptive Analysis

Equation Solver
● Frontal   ○ Skyline

Processing Options
☐ Optimize element numbers.
☐ Optimize node numbers.
☐ Visualize the process.
☐ Keep the matrix for reanalysis
☑ Check the available disk space
☑ Use temporary files for safety

Adaptive Analysis Options
Termination Criterion ——————————— Set the criteria for terminating
☑ Number of iteration cycles   `2`   the adaptive iterations.
☑ Energy norm error   `5`   %
Handling of Intermediate Models ——— Decide how to handle the data
● Keep all the intermediate models.   from intermediate iteration cycles.
○ Keep the original and the final models
Visualization of Adaptive Process —— Decide whether to display
☑ Display updated mesh.   the intermedate adaptive process.
☑ Display energy norm error.

[Cancel]   [O.K.]

- handling of intermediate models: It is the option determining how to treat
  the model data created at the intermediate stages of iterations.
  - "Keep all the intermediate models": If this radio button is turned on, the
    modeling data and analysis results obtained during the intermediate cycles
    of adaptive iteration are saved, and can be retrieved for later use.
  - "Keep the original and the final models": If this radio button is turned on,
    the intermediate modeling and analysis data are discarded, and only the
    original and the final data are saved.
- visualization of adaptive process: It is the option related with visualizing the
  model and/or energy norm error while the adaptive iteration process is going
  on.
  - "Display updated mesh": If this radio button is turned on, the meshes
    generated at each step of adaptive iterations are plotted.
  - "Display energy norm error": If this radio button is turned on, the energy
    norm error distribution is displayed by contour at each step of adaptive

iterations. (This option may not work for the current version of VisualFEA.)

■ Setting analysis options for dynamic analysis

Choosing "Solve" item from `Solve` menu will pop up "Dynamic Analysis Options" dialog as shown below, if you have set the solution type as dynamic. The solution type can be set by using the "Project Setup" dialog.

Set the solution method   Select the time integration method

Dynamic Analysis Options

Solution Method
○ Direct integration
● Mode superposition
○ Modal analysis

Integration Method
○ Central difference (explicit)
● Newmark   ○ Wilson Theta

Equation Solver
○ Frontal ● Skyline ○ Sparse

Processing Options
☐ Optimize element numbers.
☐ Optimize node numbers.
☐ Visualize the process.
☐ Keep the matrix for reanalysis
☑ Check the available disk space
☑ Use temporary files for safety

Mass Matrix
● Lumped   ○ Consistent — Set the way of computing mass matrix

Number of dynamic modes `10` — Set the number of dynamic modes
Number of time steps `200` — Set the number of time steps
Time step size `0.001` sec — Set the step size for time integration

☑ Rayleigh Damping — Set the Rayleigh damping
  Stiffness damping ratio `0.01`
  Mass damping ratio `0.01`
☐ Mode Equivalent Rayleigh Damping — Set the mode equivalent Rayleigh damping

☐ Modal Damping — Set the modal damping

Newmark Parameters — Set the constants used with the integration method
  Alpha `0.2500`   Delta `0.5000`

[Cancel]   [O.K.]

- • solution method : This option determines how to get the dynamic analysis results.   VisualFEA supports 3 methods of performing dynamic analysis.
  - direct integration: No transformation is applied for integration in time. The nodal displacements are obtained directly at each time step.
  - mode superposition: Time integration is operated on the participation factors of dynamic modes. Thus, the dynamic modes are extracted first through eigenvalue analysis, and the system equations are formed in terms of participation factors of these dynamic modes. The nodal displacements are obtained by superposing the dynamic modes at each time step.
  - modal analysis: Only dynamic modes are extracted, No time integration is performed. Other analysis results including nodal displacements are not computed.
- • integration method : This applies to the integration in time for both the direct integration method and the mode superposition method.
  - "Central difference (explicit)": A explicit integration method, in which the stiffness matrix is not decomposed. The mass matrix is decomposed only

when consistent mass matrix is used. The solution may diverge if the time step is larger than the critical value.

- "Newmark": An implicit method with linear acceleration controlled by parameters and , which can be set by the user.
- "Wilson Theta": An implicit method with linear acceleration controlled by an input parameter , which can be set by the user.

• mass matrix : There are following two options in computing the element mass matrix.

- "Lumped": The mass matrix is computed by assuming that element mass is concentrated at nodal points.
- "Consistent": The mass matrix is computed by interpolation consistent with that used for the stiffness matrix.

• number of dynamic modes : To specify the number of dynamic modes used for mode superposition. This item is valid only when the solution method is set as mode superposition.

• number of time steps : The number of steps included for time history analysis. The total duration of the analysis is determined by the number of steps and the step size which is the next input item.

• time step size : The length of time from one step to the next. Equal step size is assumed for the whole duration of the analysis.

• Rayleigh damping : If this item is checked, Rayleigh damping is assumed, which is the form of $\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K}$. And the following 2 sub-items pop up.

- "Stiffness damping ratio": This is the stiffness damping coefficient of the above equation.
- "Mass damping ratio": This is the mass damping coefficient of the above equation.

• mode equivalent Rayleigh damping : If this item is checked, Rayleigh damping is assumed, but is represented by 2 modal damping ratios which appear as additional input items.

- "Mode 1 damping ratio": $_1$
- "Mode 2 damping ratio": $_2$

☑ **Mode Equivalent Rayleigh Damping**
Mode 1 damping ratio   `0.1`
Mode 2 damping ratio   `0.1`

There is a following relationship between the Rayleigh damping coefficients(α and β) and the modal damping ratios (ξ_1 and ξ_2).

$$\xi_1 = \frac{\alpha}{2\varpi_1} + \frac{\beta\varpi_1}{2}$$

$$\xi_2 = \frac{\alpha}{2\varpi_2} + \frac{\beta\varpi_2}{2}$$

- modal damping : One method of assigning damping characteristic is to assume an individual damping ratio for each dynamic mode. It is termed here as modal damping. However, information on dynamic modes is not available prior to completing the modal analysis. Thus, the damping ratio are specified as a function of modal frequency.

  If you click this item, there appears [**Edit**] button which is used to launch "Modal Damping Ratio" dialog. A table of modal frequencies and paired damping ratio can be specified using this dialog. The damping ratio for a given frequency is estimated by interpolating the values given in this table.

Modal Damping Ratios

Point	Frequency	Damp. Ratio
1	1	0.01
2	5	0.005
3	20	0.002
4	40	0.001

Damping ratios

Modal frequencies in Hz

Editable text box for selected text — `0.001`

[**Add**] — Add a frequency to the list

[**Delete**] — Delete a frequency from the list

Save the list of damping ratios in a text file. — [**Save**]

[**Import**] — Import a list of damping ratios saved in a text file.

[**Cancel**] [**O.K.**]
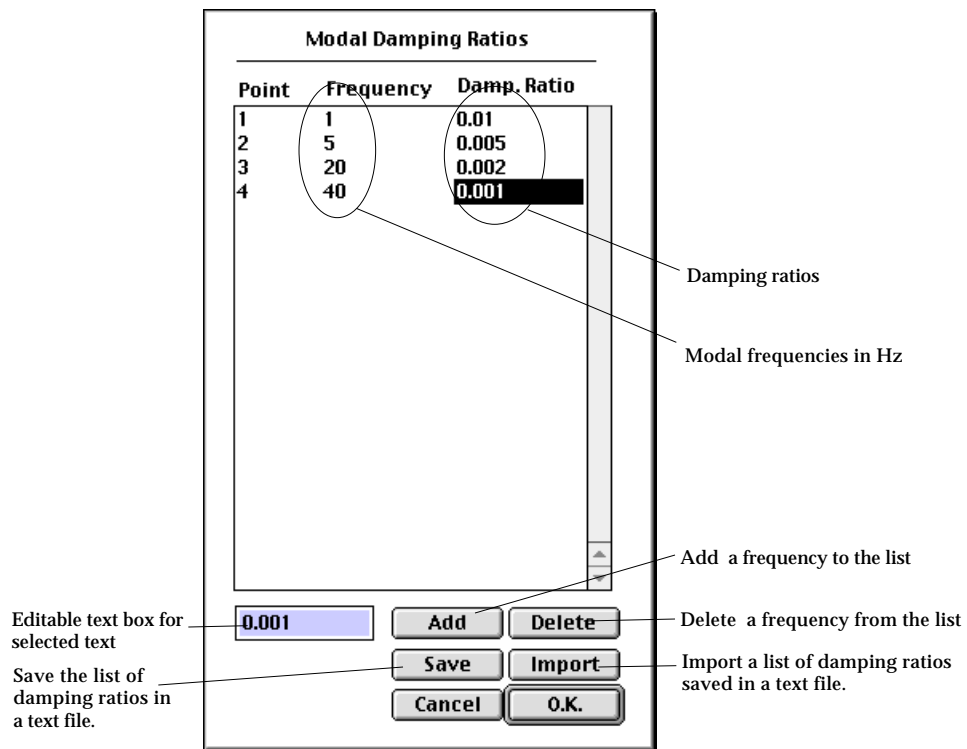
- Acceleration parameters : parameter(s) for linear acceleration of time integration. There are different parameters depending on the method of time integration.   The input items change as the method of integration changes.
  - central difference method: no input parameters for this method of integration.
  - Newmark: There are 2 parameters   and . The default values,   =0.25 and =0.5 are used for unconditionally stable solution.

Newmark Parameters
Alpha `0.2500`   Delta `0.5000`

- Wilson : There is a parameter . The default value is =1.4. The value of should be 1.37 or greater for unconditionally stable solution.

Wilson Theta
Theta 1.4000

■ Setting analysis options for nonlinear analysis

Choosing "Solve" item from **Solve** menu will pop up "Nonlinear Analysis Options" dialog as shown below, if you have set the solution type as material nonlinear, or geometric nonlinear. The solution type can be set by using the "Project Setup" dialog.
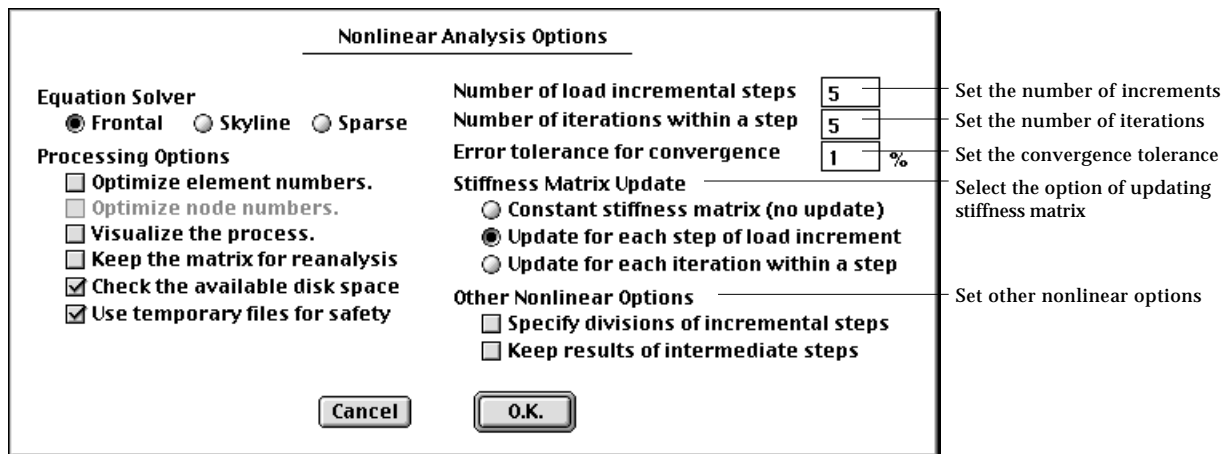
- number of load incremental steps : It is the number of steps for solution of a nonlinear problem by incremental method. If this value is 1, a simple iterative solution is applied. Otherwise, the total load is divided into as many segments as this number, and applied incrementally through the nonlinear solution process.
- number of iterations within a step : The maximum limit in the number of iterations within an incremental step for solution of nonlinear equation. If this value is 1, simple incremental procedure is applied.
- error tolerance for convergence : This convergence criterion applies to the iterative procedure within an incremental step. The iteration is terminated if either the percentage of the residual force falls below this level, or the number of iterations reaches the maximum limit.

m>1, n=1   m=1, n>1   m>1, n>1

m=number of incremental steps,   n=number of iterations within a step
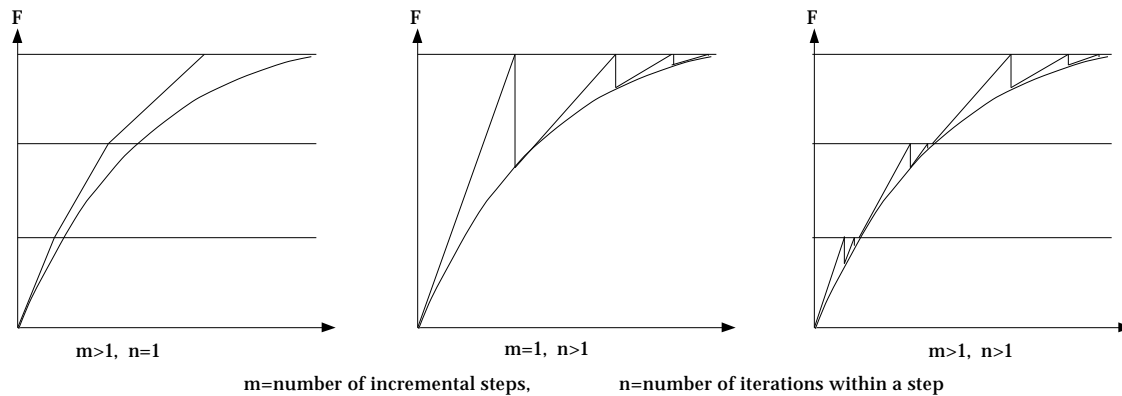
< Number of load increments and number of iterations >

- • options for stiffness matrix update: There are following 3 options of updating stiffness matrix throughout the incremental and iterative process.
  - - "Constant stiffness matrix (no update)": If this button is turned on, the stiffness matrix is not updated throughout the whole incremental and iterative process. Thus, the stiffness matrix is computed only once and no additional time is required for its update. However, this option leads to large number of iterations as shown in the figure below.
  - - "Update each step of load increment": If this button is turned on, the stiffness matrix is updated only for the first iteration of each load increment.   The stiffness matrix remains constant for all iterations within a load incremental step.
  - - "Update each iteration within a step": If this button is turned on, the stiffness matrix is updated for every iteration throughout the whole process. This option takes more time for updating stiffness matrix, but requires smaller number of iterations.

Constant stiffness matrix   Update for each step of load increment   Update for each iteration
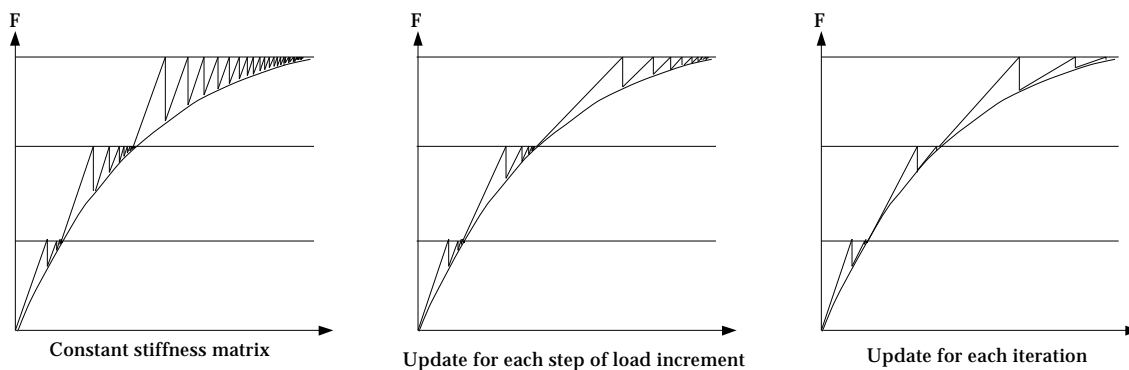
< Schemes of stiffness matrix update >

- other nonlinear option:
  - "Specify division of incremental steps" : This item is not checked by default, and the sizes of all load increments are equal. If you check this item, the following "Load Incremental Steps" dialog pops up. Initially the editable text boxes are filled with uniformly divided incremental portions of the load.   These incremental proportions can be modified by editing the text. boxes

Load Incremental Steps

1	0.1
2	0.2
3	0.3
4	0.4
5	0.5	— Set the incremental proportion of loads
6	0.6
7	0.7
8	0.8
9	0.9
10	1
11	
12	
13	
14	
15	
16	
17	
18	
19	
20	

Cancel   O.K.

  - "Keep results of intermediate steps" : If this item is checked, the solution data obtained at every incremental step is saved, and can be retrieved for later use.

■ Setting analysis options for sequentially staged modeling

As for linear analysis, the same dialog is used for both non-staged and staged modeling.   In the case of nonlinear analysis, dialog for sequentially staged modeling has a few more items than the dialog for non-stage modeling. They are related to incremental and iterative solution scheme for each stage.

In order to apply constant number of incremental steps and iterations, turn on "Constant step iteration" radio button in the dialog and then insert the number of

incremental steps and iterations in the editable text box. In order to differentiate the incremental and iterative scheme from stage to stage, turn on "Variable step iteration" ratio button. And click Edit button. Then, "Solution Step Iteration Scheme" dialog appears. You may set the number of load steps and the number of iterations using this dialog. The dialog displays as many rows as the number of stages. At the beginning , each row is assigned with equal number of incremental steps and equal number of iterations.   Set new values by editing the existing ones, and click O.K. button to complete the setting.

Processing of heat conduction and seepage analysis

Processing of heat conduction analysis is basically the same as that of structural analysis. But a seepage analysis should go through iterative processes of determining the phreatic flow surface which is not necessary in a structural or a heat conduction analysis.

■ Setting analysis options for heat conduction analysis

A steady state analysis of a heat conduction problem requires only one cycle of equation assembly and solution process which is identical to the processing of a linear static analysis of a structural problem. This type of processing is described already in "Processing of structural analysis" section of this chapter, and is not repeated here.

■ Setting analysis options for seepage analysis

Although both the seepage analysis and the heat conduction analysis are based on the same governing equations, the processing of seepage analysis is different from that of the latter because it requires an iterative procedure determining the phreatic flow surface.

The editable text items of the dialog requires inputting the following values:

- Datum (Coordinate of zero height) : datum elevation. The hydraulic head is set to 0 at this level.
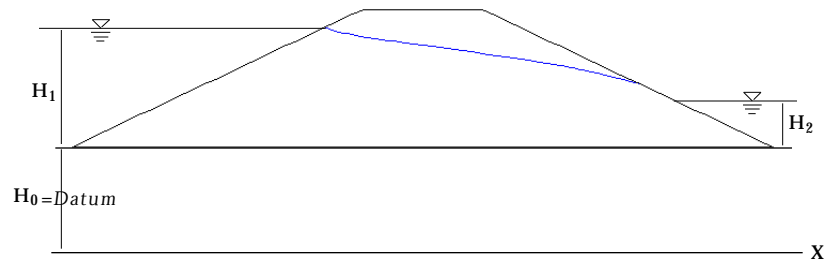
< Datum level>

- Unit weight of fluid : The unit weight of fluid is used in converting the head to fluid pressure. • Number of time steps : the number of time steps involved in the transient analysis. This value is ignored in a steady state analysis.
- Time step size : the length of time between two consecutive steps. This value is ignored in a steady state analysis.
- Number of iteration within a step : The phreatic surface is determined by iterative computation.   This value specifies the maximum number of iterations determining phreatic surface in a steady state analysis.   This value is also applied to a transient analysis as the number of iterations within each time step.
- Error tolerance for convergence : Another criterion for finishing the iterative process is base on the maximum difference of phreatic surface elevations between the current and the last iterations. One of this and the above criteria is met, the iterative processing is terminated.
- Open head increment : The limit of increment in modifying the open head in a iteration cycle.
- Criterion for open head modification : The open head is incremented on the basis of this criterion.
  - "By elevation": If this option is on, the node with lowest elevation is searched along the open head boundary, and the phreatic head is set to the head of node. If more than one node has the lowest elevation, only the node with the maximum pressure will be used to set the head
  - "By pressure": If this option is on, he node with maximum positive pressure is searched along the open head boundary, and the phreatic head is set to the head of node.

Processing stages

Processing is divided into a few stages, and a message dialog shows the progress of each stage. Processing can be interrupted or resumed in the middle, if necessary. If there is any problem in solving the equations, appropriate message is displayed and processing aborts.

■ Progress of processing

While the processing is going on, its progress is indicated on a modal dialog as shown below. The progress is displayed in 3 stages. If the frontal solver is adopted, "Assembling element", "Back substitution", and "Stress recovery and smoothing" caption is posted on the dialog to indicate the processing stages. If skyline solver is used, "Matrix decomposition" is posted at the second stage.

During the first stage, the element matrices are assembled. The dialog shows the total number of elements as well as the number of elements assembled so far. For nonlinear analysis or sequentially stage analysis, the stage No., the load incremental step No. and the iteration No. are also displayed in the dialog. In case of frontal solver, not only assembly but also decomposition of the equations are going on at this stage.

The number of the element now being assembled

Total number of elements

For linear analysis

Assembling element 1135 of 7020.

Stop   Pause

load incremental step No.

iteration No.

For nonlinear analysis

N=3; I=5; Assembling element 1135 of 7020.

Stop   Pause

load incremental step No.

stage No.   iteration No.

For staged analysis

S=6; N=3; I=5; Assembling element 1135 of 7020.

Stop   Pause

In the next stage, the decomposed equations back substituted, if the frontal solver is used. The progress bar starts from the beginning again.

```
Back-substitution.
[███████████████████████░░░░░░░]
        [ Stop ]  [ Pause ]
```
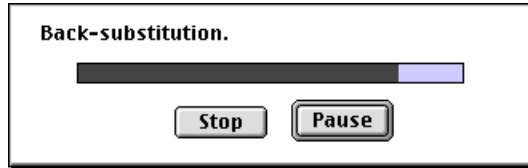
If skyline solver is used, the system equations are decomposed at this stage.   So, the caption of the dialog is different.

```
Matrix decomposition
[██████████████░░░░░░░░░░░░░░░░]
        [ Stop ]  [ Pause ]
```
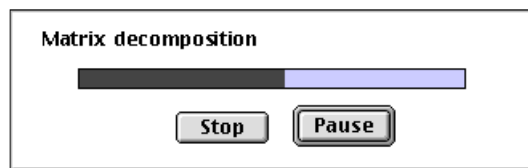
In the last stage, the strains (or gradient) and the stresses are computed at integration points of each element, and their values are smoothed at each node. The dialog appears as follows, and the progress bar starts from the beginning again.

```
Stress recovery and smoothing
[████████░░░░░░░░░░░░░░░░░░░░░░]
        [ Stop ]  [ Pause ]
```
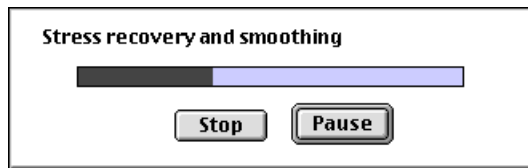
If the execution of processing is successfully completed, a beep sounds and the following message dialog appears.

```
        Processing completed
             [ O.K. ]
```

■ Interrupting the processing

While the processing is going on, execution can be interrupted at any time either permanently or temporarily by clicking [Stop] or [Pause] button. If you press [Stop] button, execution aborts and the following message box appears.

```
Processing aborted by user intervention
at    Sunday, 23 August, 1998  05:27:47 PM
             [ O.K. ]
```

If you click [Pause] button of the progress dialog, processing does not abort but is temporarily suspended so that you may do other operations while the following dialog stays on the screen.

Processing suspended by user intervention.

[Stop]   [Resume]

Execution is resumed and the status dialog moves to the front when you click [Resume] button of the dialog. Processing may be aborted at this stage by pressing [Stop] button.

■ Abnormal termination of the processing

Processing may be terminated abnormally in the middle of execution due to one of the following reasons.

• **Matrix files for reanalysis are not found or are mismatching:** If the solution stage is set as "Reanalysis" but necessary files for reanalysis do not exist or do not match the data file, the processing cannot be executed.   So, the operation will abort with the following message box.

⚠ Matrix files not found or
mismatching!

[O.K.]

The matrix files for reanalysis should reside in the same folder (or directory) as the data file. And their name and extension should be compatible with that of the data file. If the name of the data file is "mydata", for example, "mydata.mtx" and "mydata.fro" should exist in the same folder (or directory).

• **Insufficient memory space:** If "Skyline" is chosen as solver option, but the CPU memory space is insufficient, then the software will let you to allow automatic switch to "Frontal" as already explained in the previous section. In case "Frontal" is chosen as solver option and the CPU memory space is insufficient, the processing aborts with the following message box.

⚠ Insufficient memory for frontal
solver. Try again after optimizing
element numbers, or saving the file.

[Cancel]   [O.K.]
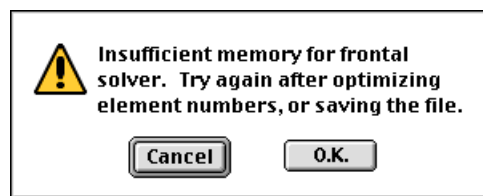
The required memory space for "Frontal" solver may be reduced by optimizing element numbering. If a memory insufficiency problem persists even after element number optimization, more computer memory should be secured for VisualFEA.

• **Equation ill-conditioning: If the assembled system equations are ill-conditioned, numerical difficulty arises in solving the equations, and thus the processing aborts in the middle with the following message.**

```
Processing failed due to equation ill-conditioning.
    Equation No.      711
    Value             1.738498e+06
                   [  O.K.  ]
```
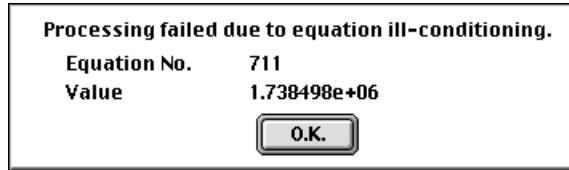
The possible causes of equation ill-conditioning are as follows.
- **Poor element shape or element connectivity : The element stiffness matrix may have numerical singularity due to unacceptably distorted shape or due to improper connectivity. If such is the case, improve the element shape and connectivity by carefully regenerating the finite element mesh again.**
- **Insufficient constraints : The system equations may have rank deficiency due to insufficient constraints or boundary conditions. In this case, check the boundary conditions and add more constraints if necessary.**
- **Irrelevant element properties: Element stiffness matrices may have been spoiled by irrelevant values of element properties. To avoid such problem, check the contents of element property sets.**
- **insufficient integration order: The stiffness matrix may have spurious zero energy modes due to insufficient integration order. In order to remove spurious zero energy modes, adopt an integration scheme of higher order.**

Interactive real time processing

VisualFEA has the capability of interactive real time processing for truss and frame analysis. Interactive real time processing means that the cycle of data modification, subsequent processing, and graphical visualization of the analysis results is formed in real time.   Thus, you can see the changing response as you add or modify the data.
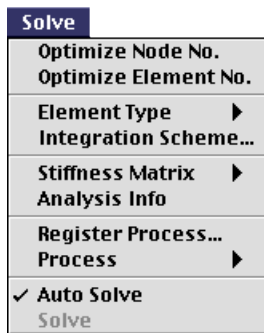
For analysis types of 2-D truss, 3-D truss, 2-D frame and 3-D frame, interactive real time processing is the default mode. "Auto Solve" item of **Solve** menu is enabled only for these analysis types. The interactive real time analysis mode can be turned on or off by checking the menu item. If the mode is turned off, the normal procedure of processing applies, as described in the previous sections.

If the mode is on, you don't have to select "Solve" item from **Solve** menu. Instead, you have to designate the data item to display from **Diagram** menu. Whenever you add or modify the data, the processing is executed automatically, and the display of the analysis results is updated immediately. The progress bar does not appear during processing.

Furthermore, if "Instant Redrawing" item of **Diagram** menu is checked, the diagrams representing the analysis results are automatically updated immediately after any change in the modeling data. Modification in geometries, attributes, boundary conditions, or load conditions of the model is reflected in the currently displayed diagram instantly.

An example of interactive real time analysis is shown in the following figure. The example shows that the bending moment diagram is displayed directly by selecting the corresponding menu item. There is no need to invoke explicitly the processing stage. The diagram is updated at the moment the load condition is altered. Likewise, the diagram will be changed as you modify the boundary conditions or element properties. Thus, the responses to varying external effects or attributes can be examined interactively.
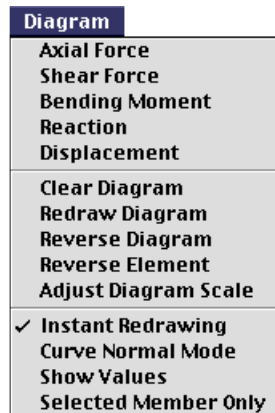
On the other hand, interactive real time analysis may sometimes hamper the responsiveness of the software, when the size of the problem is too large to get enough speed in processing one cycle. In such circumstances, you may suppress the options, by unchecking "Instant Redrawing" of **Diagram** menu. In addition, the progress bar can be shown while processing goes on, if "Auto Solve" item of **Solve** menu is unchecked as well.
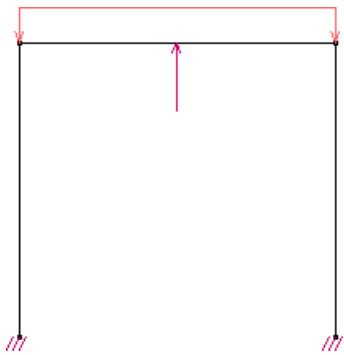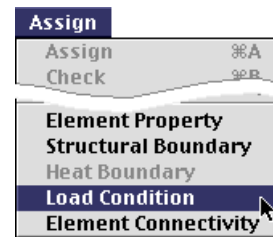
1. Model the structure, and assign attributes, boundary conditions and load conditions.
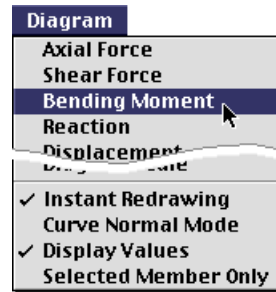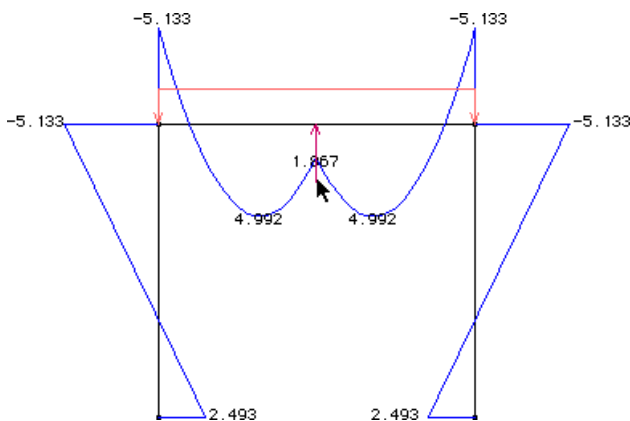

2. Choose "Bending Moment" menu item. Make sure that "Instant Redrawing" item is also checked.
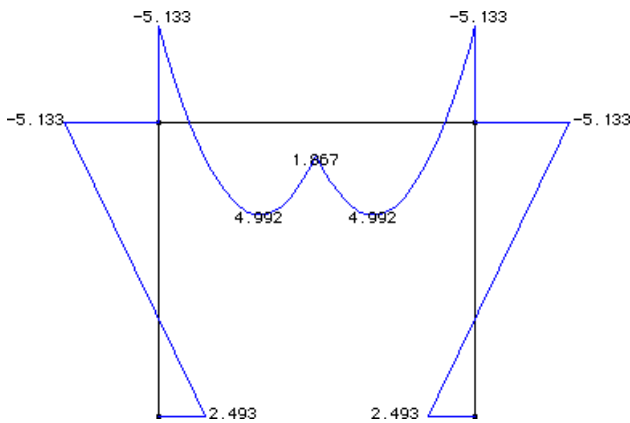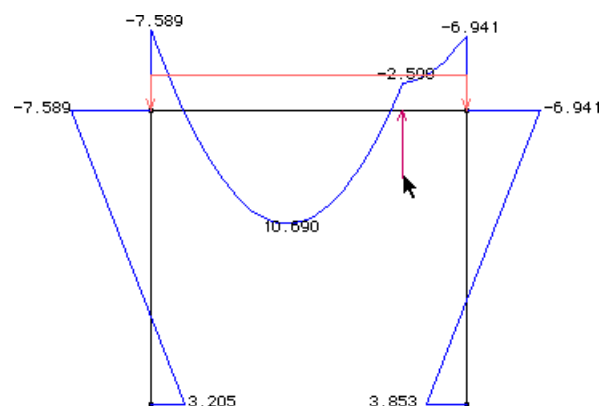

3. Bending moment diagram is displayed.

4. Choose "Load Condition" menu item.

5. Select the point load.

6. Drag the point load to the right. Then, the bending moment diagram changes subsequently.

< Example of interactive real time processing>

Other Functions Related to Processing

The computational efficiency of processing depends on a number of factors like node numbering, element numbering, integration scheme, and so on.   They can be manipulated to improve the efficiency.

Optimizing node numbering

If "Skyline" solver is used, the required memory and the computing time is directly related to the band width of the system equations.   The band width is determined by the node numbering. Therefore, node numbering is very important. However, nodes are numbered initially in the order of their creation during the modeling stage.   It is desirable to renumber the nodes before processing stage so that the band width is minimized. This procedure is called node number optimization.

Node number optimization can be dictated as an optional item to be done before processing as explained in the previous section. This option, however, may cause redundant repetition of the operation, if the same modeling data are used for multiple analysis. This redundant operation may be avoided by doing the node number optimization once and executing the processing multiple times without the optimization option.

Node number optimization can be invoked by choosing "Optimize Node Number" item from the menu.   A dialog shows the intermediate state of progressive band width reduction and the final half band width (HBW) after the completion of optimization.   If you click Accept button, the nodes will be renumbered as optimized.   If you click Reject button, the optimization is ignored and the old node numbering will be retained. Save the file in order to keep this renumbering for future processing.

```
Completed node number optimization.

    Original HBW    4110
    Reduced HBW    354

    [ Reject ]      [ Accept ]
```
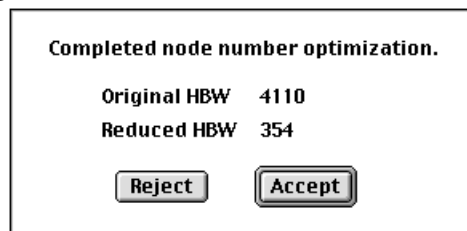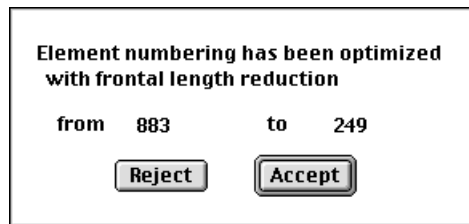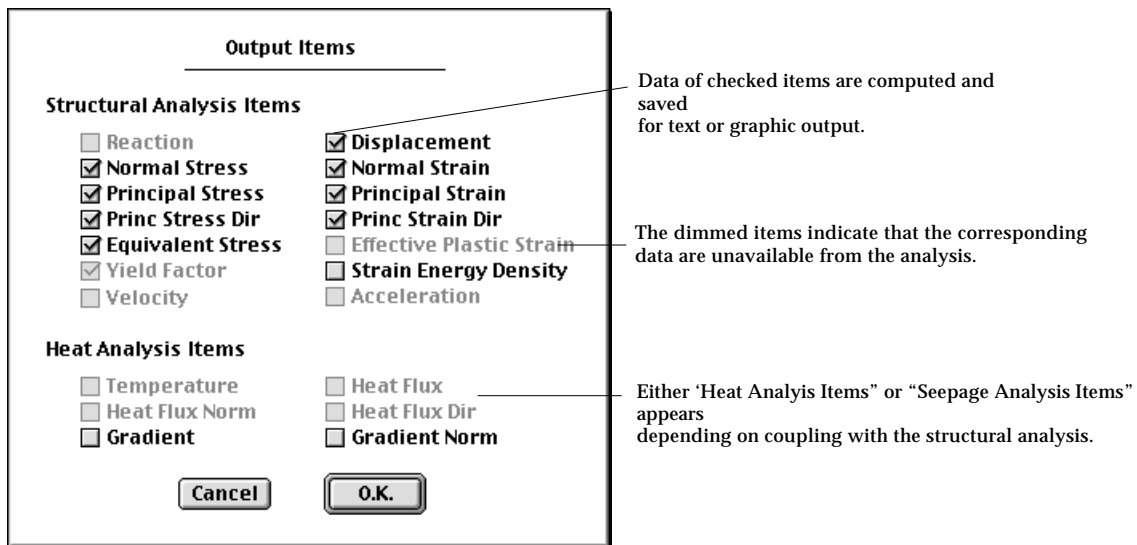
It should be noted that the optimization does not mean absolute minimization of the band width, but means relative reduction of the band width. There may exist a node numbering with smaller band width than the optimized one. However, it is too time consuming or not feasible to find the numbering with this absolute minimum band width.

Optimizing element numbering

If "Frontal" solver is used, the required memory and computing time are directly related to the critical frontal length of the system equations.   The critical frontal length is determined by the element numbering. Similarly to node number optimization, it is desirable to renumber the elements before processing stage so that the critical frontal length becomes the minimum. This procedure is called element number optimization.

Element number optimization can be dictated as an optional item to be done before processing as explained in the previous section. This option, however, may cause redundant repetition of the operation, if the same modeling data are used for multiple analysis. This redundant operation may be avoided by doing the element number optimization once and executing the processing multiple times without optimization option.

Element number optimization can be invoked by choosing "Optimize Element Number" item from menu.   A dialog shows the intermediate state of progressive reduction of critical frontal length and the final critical frontal length after completion of optimization.   If you click Accept button, the elements will be renumbered as optimized.   If you click Reject button, the optimization is ignored and the old element numbering will be retained. Save the file in order to keep this renumbering for future processing.

It should be noted that the optimization does not mean absolute minimization of the critical frontal length, but means relative reduction of the critical frontal length. There may exist an element numbering with smaller critical frontal length than the optimized one. However, it is too time consuming or not feasible to find the numbering with this absolute minimum critical frontal length

Setting output items

- output items: Only the checked items are computed during the processing and accordingly are accessible for postprocessing or text output.   Depending on the subject of analysis, either structural analysis related items or heat transfer related items are enabled and the others are disabled.

Data of checked items are computed and saved
for text or graphic output.

The dimmed items indicate that the corresponding data are unavailable from the analysis.

Either 'Heat Analyis Items" or "Seepage Analysis Items" appears
depending on coupling with the structural analysis.

Specifying integration scheme

In order to change the integration scheme, select "Integration Scheme..." item from **Solve** menu. Then "Integration Scheme" dialog box appears as shown below. Select the element shape (and order) from the popup menu of the dialog.
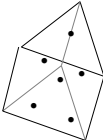

Element shape

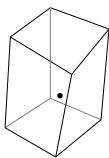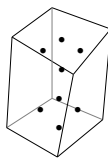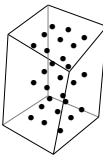Popup Menu Items

Integration schemes
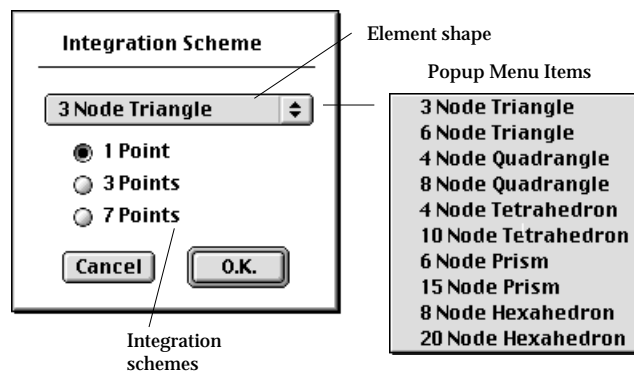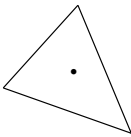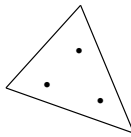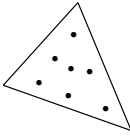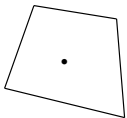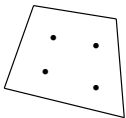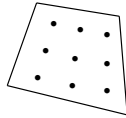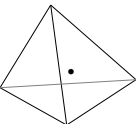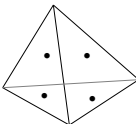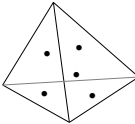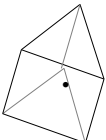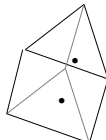
 Then, alternative integration schemes for the element shape are displayed as radio button items. The radio button corresponding to the current setting is marked. The integration scheme can be changed simply by turning on the radio button of

the desired scheme. After changing the schemes of the relevant element shapes, click [O.K.] button.   The integration scheme of each element shape is applied in computing the element stiffness matrix of the respective element shape.
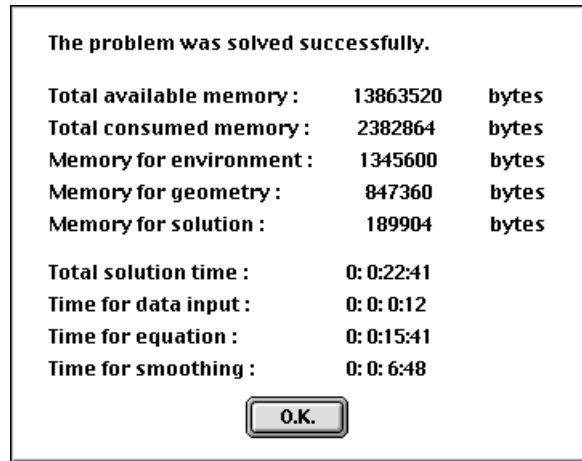
VisualFEA uses numerical integration(Gauss quadrature) in evaluating the stiffness matrix. The computing time, accuracy and stability of the system equations depend greatly on the integration scheme. There are default schemes preset for various element shapes as shown in the following table. These integration schemes can be altered if desired.

<Integration scheme>

Element Shape	Integration Schemes			No. of Nodes	Default Scheme
Triangle	1	3	7	3 6	1 3
Quadrangle	1	2×2	3×3	4 8	1 2×2
Tetrahedron	1	4	5	4 10	1 4
Prism	1	2×1	2×3	6 15	1 2×3
Hexahedron	1	2×2×2	3×3×3	8 20	2×2×2 2×2×2

Displaying analysis information

The state of analysis and related information is displayed as exemplified below by selecting "Analysis Info" item from **Solve** menu. The information includes the time and memory space required to solve the problem.

```
The problem was solved successfully.

Total available memory :        13863520      bytes
Total consumed memory :          2382864      bytes
Memory for environment :         1345600      bytes
Memory for geometry :             847360      bytes
Memory for solution :             189904      bytes

Total solution time :        0: 0:22:41
Time for data input :        0: 0: 0:12
Time for equation :          0: 0:15:41
Time for smoothing :         0: 0: 6:48

                   [ O.K. ]
```
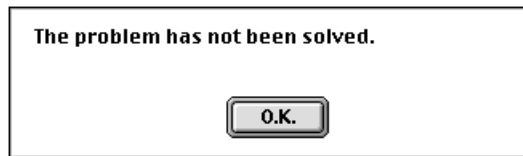
The above information is displayed only in case the problem was successfully solved in the current session.   If it was solved in the previous session, the following message will be displayed.

```
The problem was solved in previous session.

                   [ O.K. ]
```
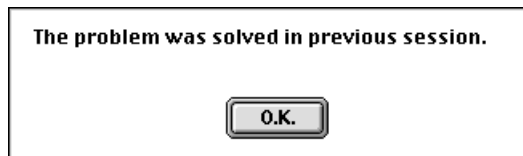
If the problem has not gone through processing, the following message will be shown.

```
The problem has not been solved.

                   [ O.K. ]
```

If the processing was canceled in the middle, the following message will be displayed .

```
Processing aborted by user intervention
   at    Saturday, 29 August, 1998  09:43:39 PM

                   [ O.K. ]
```