SESAM
USER MANUAL

Manager

Managing SESAM Programs and Files

DET NORSKE VERITAS

SESAM
User Manual

Manager

Managing SESAM Programs and Files

October 1st, 2002

Valid from program version 5.3

Developed and Marketed by
DET NORSKE VERITAS

DNV Software Report No.: 95-7017 / Revision 2, October 1st, 2002

Published by:

Det Norske Veritas
Veritasveien 1
N-1322 Høvik
Norway

Telephone: +47 67 57 99 00
Facsimile: +47 67 57 72 72
E-mail, sales: software.sesam@dnv.com
E-mail, support: software.support@dnv.com
Website: www.dnv.com

Table of Contents

1 INTRODUCTION

1.1 Manager — Managing SESAM Programs and Files

SESAM is an integrated and comprehensive software system for hydrodynamic and structural analysis. Through a menu and dialog box driven user interface Manager provides easy access to SESAM. Analyses are organised in projects. Files pertaining to a specific analysis project are managed and kept in a project directory (folder). Execution of the individual SESAM programs is controlled by checking consistency and completeness of data.

Manager will for several programs establish parts of or all input required. For example, you need not manually create an input file for a linear static structural analysis using Sestra. Manager establishes the suitable input. The automatically established input also serves the purpose of a template for more complex input.

The purpose of Manager is, therefore:

- To provide an intuitive and easy access to the SESAM programs
- To ease the program execution and establish parts of the input
- To manage the SESAM files related to analysis projects

Manager is available on the following operating systems:

- Windows NT, 2000 and XP, hereafter referred to as PC
- OSF/Motif X Window as available on workstations or servers based on Unix

1.2 Manager in the SESAM System

Manager is normally depicted as a long horizontal box on top of the SESAM Overview. Figure 1.1 shows the Manager box magnified and how the program accesses the SESAM programs.

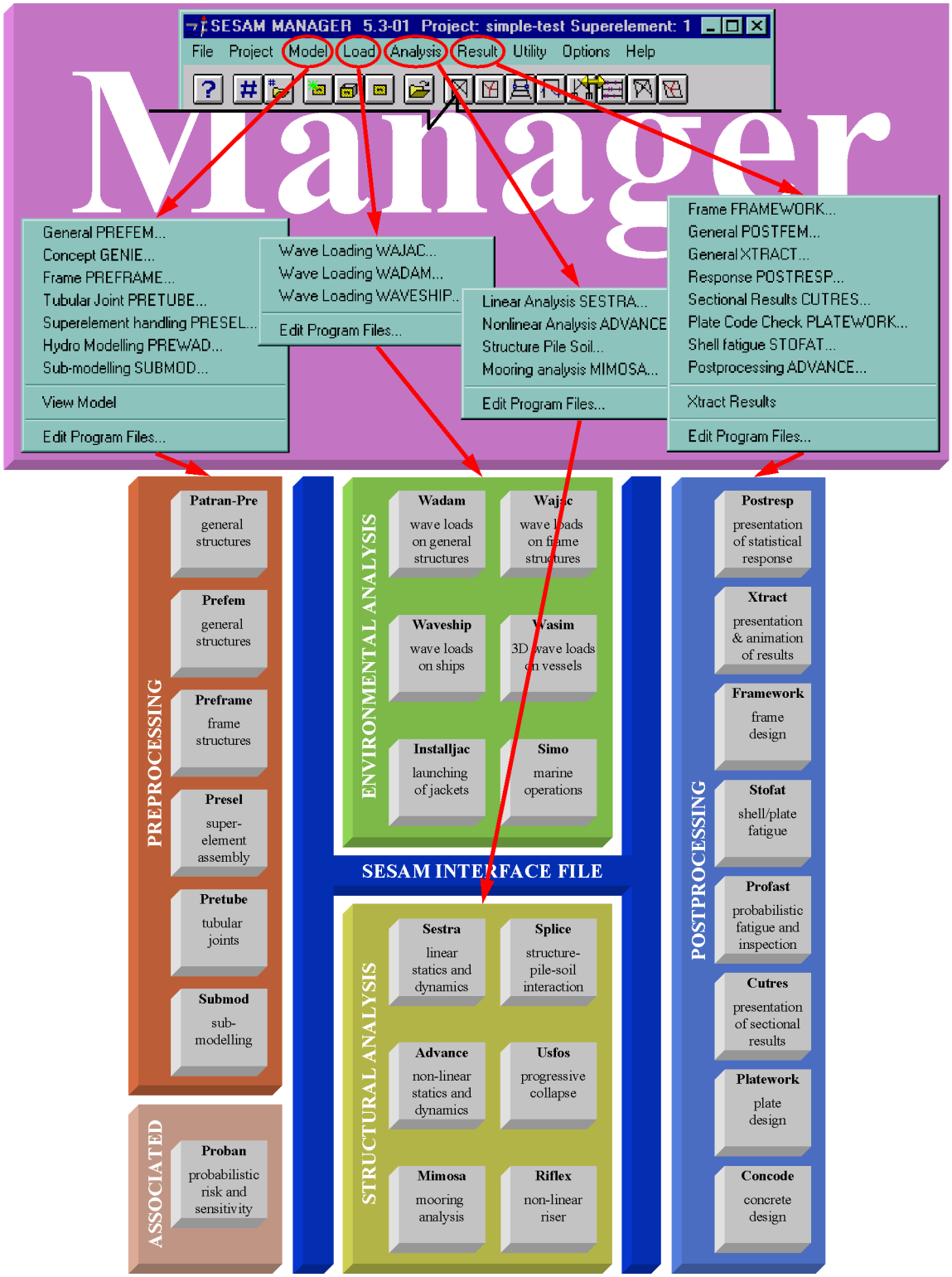

Figure 1.1 Manager controls SESAM programs

1.3 How to read the manual

Chapter 2 FEATURES OF MANAGER contains a description of the basic features of Manager.

Chapter 3 USER'S GUIDE TO MANAGER explains how to start the various tasks of an analysis. Such tasks may be modelling, structural analysis and postprocessing. The chapter only explains features of Manager. Features pertaining to the individual SESAM programs are covered by the user manuals of these programs.

Chapter 4 EXECUTION OF MANAGER provides information on the file and directory structure used in Manager for organising the project files.

Chapter 5 COMMAND DESCRIPTION describes the line-mode commands (a command reference).

Appendix A TUTORIAL EXAMPLES includes a few examples of command input files for Manager.

1.4 Status List

There exists for Manager (as for all other SESAM programs) a Status List providing additional information. This may be:

- Reasons for update (new version)
- New features
- Errors found and corrected
- Etc.

Use the program Status for looking up information in the Status List: In Manager click  . Then give File | Read Status List and select Manager (or any other program for which you want to view the Status List.) In the Status List Browser window narrow the number of entries listed:

- Entries relevant to a specific version only
- Entries of a specific type, e.g. Reasons-for-Update
- Entries containing a given text string

Click the appropriate entry and read the information in a Print window.

2 FEATURES OF MANAGER

Through Manager you can:

- Create new and open existing analysis projects

  An ‘analysis project’ in this context embrace all tasks and data involved in a single hydrodynamic and/or structural analysis, from modelling, through analysis to postprocessing. A typical design project will often require several ‘analysis projects’, e.g. one for static analysis, one for free vibration analysis and one for forced response analysis.

- Start and control the execution of appropriate SESAM programs

  For some programs Manager will establish the relevant input or a template for such.

- Copy, rename and delete a project as well as compress a project by deleting selected files

- Set certain master data such as common view point for model displays, plot format, editor to use for editing/viewing files, etc.

- Open the SESAM web page and download site

- Run a sequence of programs automatically, e.g. a complete analysis from modelling through analysis to postprocessing.

See Chapter 3 for more details.

3 USER'S GUIDE TO MANAGER

This user's guide explains:

- How to start Manager: Section 3.1
- Basic procedure, menu and toolbar of Manager: Section 3.2
- How to open new and manage existing analysis projects: Section 3.3
- How to start modelling: Section 3.4
- How to run hydrodynamic analysis: Section 3.5
- How to run structural analysis: Section 3.6
- How to start postprocessing results: Section 3.7
- How to start any SESAM program outside the direct control of Manager (**Utility | Run**): Section 3.8
- About controlled and non-controlled programs: Section 3.9
- How to set various options for the project: Section 3.10
- Command line arguments of Manager: Section 3.11
- How to use the command line interface of Manager and using a Manager command input file for running a sequence of programs, e.g. modelling, analysis and postprocessing: Section 3.12 and Section 3.13

3.1 Starting Manager

3.1.1 Manager on PC

Start Manager from the Start menu: **Programs | SESAM | SESAM Manager** or double-click the SESAM icon on your screen:  . The window shown in Figure 3.1 will appear.

3.1.2 Manager on Unix

Start Manager by the command **manager** in lower case.

3.2 Basic Procedure, Menu and Toolbar of Manager

3.2.1 Basic Procedure

Manager basically has the same functionality on PC and Unix although its appearance differs slightly. Figure 3.1 shows the appearance of Manager on PC.

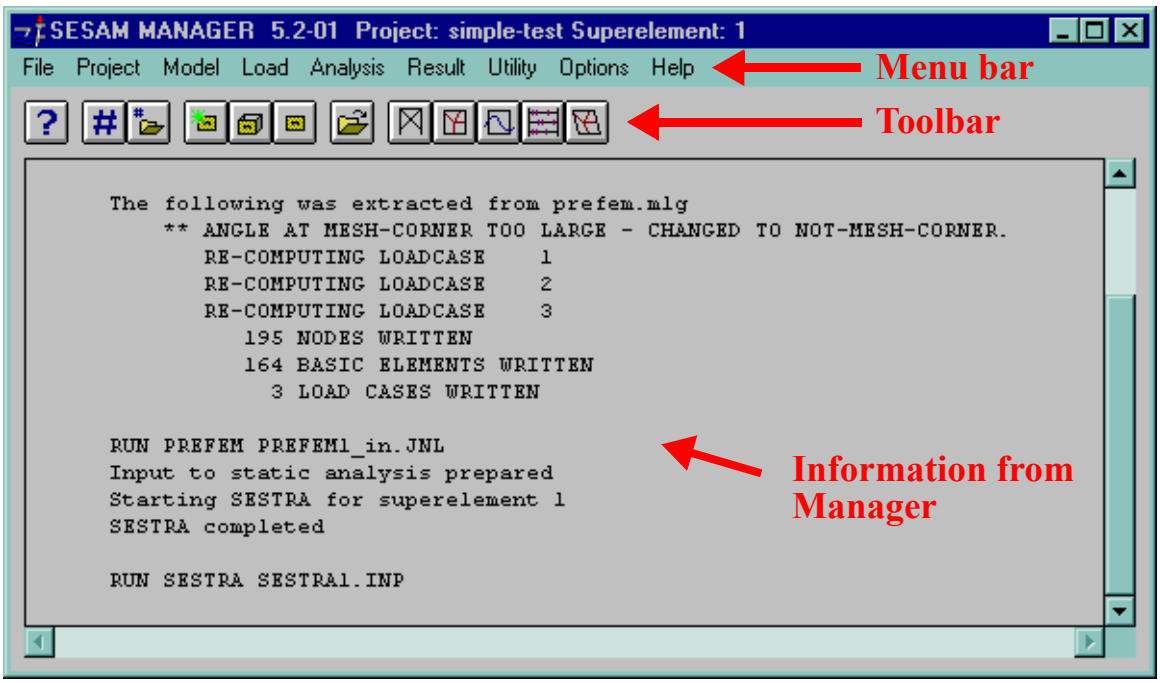

Figure 3.1 The Manager window

Very briefly, performing a SESAM analysis involves performing a selection of the following steps.

1 Open a new project by **Project | New** or an existing project by **Project | Open**.

2 Set various options for the project by **Options | 'relevant option'**.

3 Create a model (or first level superelement) through **Model | 'appropriate choice'**. Repeat this for all 1st level superelements.

4 Assemble first level superelements to form the complete model through **Model | Superelement Handling Presel**.

5 Calculate hydrodynamic loads through **Load | 'appropriate choice'**.

6 Perform structural analysis through **Analysis | Linear Analysis Sestra**.

7 Perform structure-pile-soil interaction analysis through **Analysis | Structure Pile Soil**.

8 Postprocess results through **Result | 'appropriate choice'**.

9 SESAM programs not controlled (see Section 3.9 about controlled and not controlled programs) by Manager may be run through **Utility | Run**.

10 Find how to get SESAM support through **Help** or **File | Open Webpage**.

11 Close a project by **Project | Close**.

12 Exit Manager by **File | Exit**.

Note: You cannot run SESAM programs without opening a project first. All files related to a project will be stored on the same directory and have pre-defined names (except that Utility | Run allows user defined file names); see Section 4.3.

3.2.2 Menu

The menu bar of Manager (see Figure 3.1) offers the following:

File	for editing files, setting up printer, saving defaults and exiting Manager
Project	for opening new and existing projects, closing projects and managing projects in general
Model	for starting a preprocessor for modelling the structure and hydrodynamic properties
Load	for running programs for hydrodynamic analysis
Analysis	for running programs for structural analysis
Result	for starting results postprocessing
Utility	for starting execution of any SESAM program outside the direct control of Manager
Options	for setting various project options
Help	for information on whom to contact about program support

Note: Only menus relevant at the current stage in the analysis process are accessible. Other menus are grayed out (Options | Enable all menus will enable all menus). For example, before you open a project you cannot start any program and menus for starting programs are grayed out.

3.2.3 Toolbar

The toolbar (see Figure 3.1) provides buttons for quick access to certain tasks. The buttons are:


Start Status List program = **Help | Status List**


Toggle command input mode; see Section 3.12


Open and read command input file


Close command input file


Create new project = **Project | New**


Open existing project = **Project | Open**


Close project = **Project | Close**


Edit a file = **File | Edit** for editing any file


Start frame modeller = **Model | Frame Preframe**


Start general modeller = **Model | General Prefem**


or

Start superelement assembling = **Model | Superelement Handling Presel**


Start modeller for general hydrodynamic analysis = **Model | Hydro Modelling Prewad**


Start wave load calculation for frame model = **Load | Wave Loading Wajac**


Start structure-pile-soil interaction analysis = **Analysis | Structure Pile Soil**


Start linear structural analysis = **Analysis | Linear Analysis Sestra**


Start result postprocessing of frame model = **Result | Frame Framework**


Start general result postprocessing = **Result | General Xtract**


Start statistical response postprocessing = **Result | Response Postresp**

Note: Only buttons relevant at the current stage in the analysis process are accessible or shown.

3.3 Managing Projects

First decide where to put your project files. This is done by **Project | Root directory** which opens the dialog box shown in Figure 3.2. This only need to be done when changing the root directory.

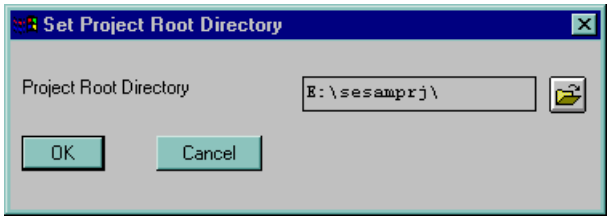

Figure 3.2 Set root directory

Open a new project by **Project | New**. Figure 3.3 shows the Create New Project dialog box as well as the Structure Type and Set Project Root Directory dialog boxes. The choice between **General** and **Frame** type of structure will only influence some default choices and not in any way limit the analysis features available. Enter a project name (without spaces) and optionally a description and click **OK**. Prior to creating the new project you may change the root of the project directory (folder); see Section 4.2.

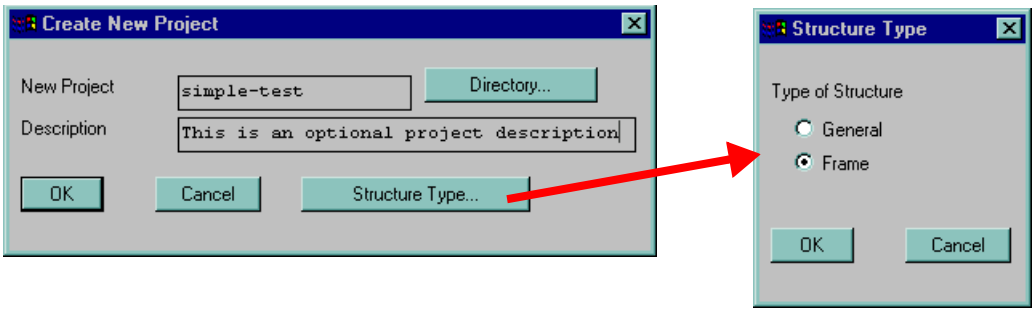

Figure 3.3 Create new project

Open an existing project by **Project | Open**. Figure 3.4 shows the Open Existing Project dialog box.

Note: Unless both a project directory and a project file with the given name exist, see Section 4.2, the project cannot be opened.

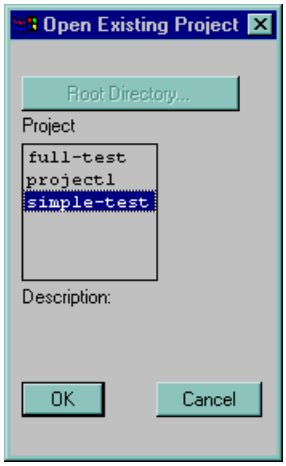

Figure 3.4 Open existing project

A project is closed by **Project | Close**. All data and settings pertaining to the project will be saved.

During as well as at completion of a project you may want to compress the project by deleting less important files or deleting the entire project. This may be done by **Project | Delete** provided that no project is open, i.e. close the current project first. In the Delete Project dialog box shown in Figure 3.5 the **Scope for deletion** has been set to **Selected files**. This involves that a list of file types appears and which file types to delete may be checked. The choice shown in the figure deletes all files except the Results Interface File plus listing (print) files (LIS files) as well as all files neccessary for reproducing the analysis (JNL and INP files).

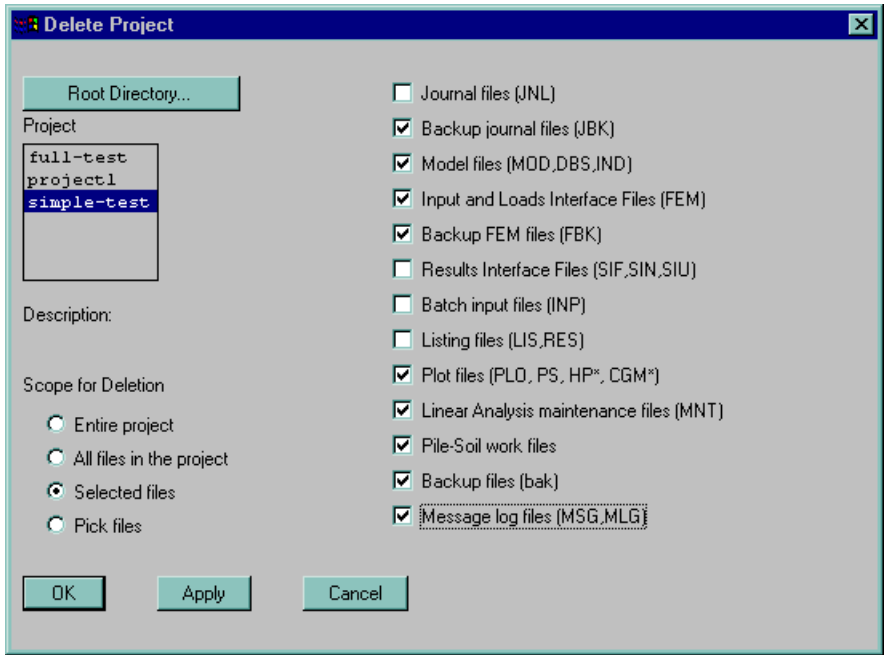

Figure 3.5 Delete selected files of a project

3.4 Modelling

To access the **Model** menu you must open a project first. The **Model** menu offers the following options:

- **General Prefem** for starting the general FE modeller Prefem
- **Concept Genie** for starting the conceptual modeller Genie
- **Frame Preframe** for starting the frame modeller Preframe
- **Tubular Joint Pretube** for starting modelling of tubular joints using Pretube
- **Superelement Handling Presel** for starting assembling superelements to form the complete model using Presel
- **Hydro Modelling Prewad** for starting hydrodynamic modelling for general structures (radiation/diffraction theory) using Prewad
- **Sub-Modelling Submod** for sub-modelling using Submod
- **View Model** for viewing the model using Viewer (As Viewer is being replaced by Xtract you may want to use Xtract instead. To start Xtract use **Options | Enable all menus** followed by **Result | General Xtract**.)
- **Edit Program Files** for editing a file, e.g. a command log (journal) file

3.4.1 Modelling First Level Superelement — Prefem and Preframe

First decide which SESAM preprocessor to use: e.g. the general FE modeller Prefem or the specialised frame (jacket) modeller Preframe. (See Section 3.4.3 for modelling tubular joints using Pretube.) Then give the **Model | 'appropriate choice'** and a type of dialog box as shown in Figure 3.6 appears.

a For a new project the **Database status** will, by default, be **New** and the superelement number will be 1. If a model with the same superelement number already exists the **Database status** will, by default, be **Old**. Selecting **Database status New** for an existing model will involve overwriting the existing model. Selecting **Old** when no model exists cannot be done.

b By default, **Write superelement on exit** will be switched on which means that when leaving the preprocessor the Input Interface File (T-file) will be written (T1.FEM when superelement number is 1).

c By default, no **Command input file** will be used. You may, however, decide to prepare all or part of the program input beforehand using an editor and saving this data in a file called for example Model_input.JNL. Select this file to be read and processed by setting **Command input file** to **File name** and selecting the appropriate input file. The input file may be edited before processing by clicking **Edit input file**.

d You may choose to exit the program once the **Command input file** has been processed or continue modelling (or merely view the model) by checking **Run interactively after command input file processing**.

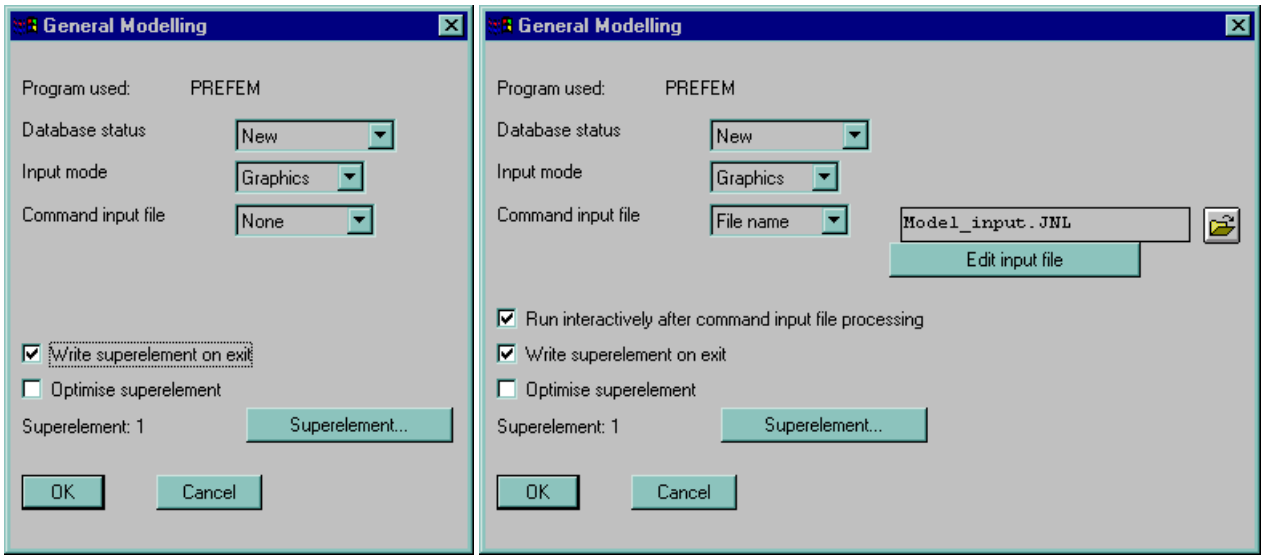

Figure 3.6 General modelling — Prefem

Clicking **OK** will start execution of the appropriate modeller. See the user manual of the appropriate modeller on how to create the model.

Provided that **Write superelement on exit** has been checked an Input Interface File (Tn.FEM where n is the superelement number) will be written when using the EXIT command inside the program. This file is the model to be analysed, alternatively a part of the model if superelement modelling is performed.

Note: Exiting the preprocessor by the ☒ button (or Alt+F4) involves no writing of the Input Interface File, i.e. overriding the setting of Write superelement on exit.

You may through **Model | Edit Program Files** review the command log (journal) file of the modeller.

3.4.2 Conceptual Modelling — Genie

Note: Genie is currently only available on PC.

Give **Model | Concept Genie** to open the dialog box shown in Figure 3.7. The contents of the dialog box is similar to that of Prefem and Preframe.

Genie controls the execution of wave loading (Wajac), structural analysis (Sestra and Splice) and post-processing (Xtract and Framework). This feature of Genie to some extent makes the use of Manager superfluous. You may therefore find it more convenient to start Genie directly (i.e. outside Manager) from the Start menu: **Programs | SESAM Genie | SESAM Genie** or double-click the Genie icon on your screen: 

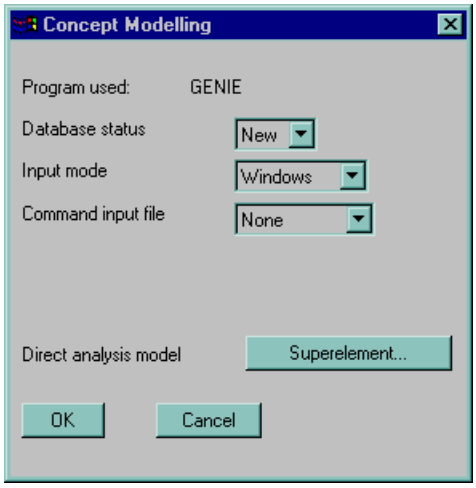

Figure 3.7 Concept modelling — Genie

3.4.3 Tubular Joint Modelling — Pretube

Give **Model | Tubular Joint Pretube** to open the dialog box shown in Figure 3.8. Compared with the dialog box of the general and frame modellers, see Figure 3.6, it includes two more fields:

- **Weld Type** for choosing between solid and shell elements in the weld zone
- **Length Unit** for choosing between mm, m, cm and inch as modelling unit

Set the number of the top level superelement as explained in Section 3.4.4. Prior to doing so the top level superelement number will be shown as a question mark (?).

Provided that **Write all on exit** has been checked all first level superelements plus the whole assembly (a second level superelement) will be written when exiting the program using the EXIT command.

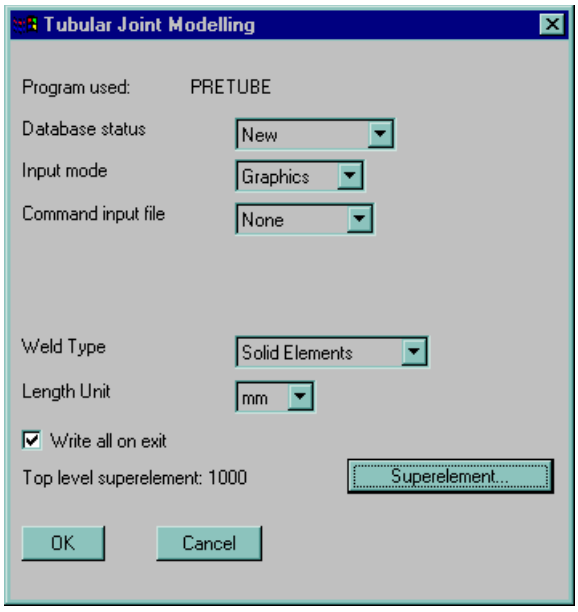

Figure 3.8 Tubular joint modelling — Pretube

3.4.4 Superelement Modelling — Presel

The dialog box of the modellers includes the button **Superelement** which will open a new dialog box for setting Superelement Options; see Figure 3.9. (This dialog box may also be opened by **Otions | Superelement**.) A new project will, by default, be a **Direct analysis** meaning that the superelement technique will not be used and a single first level superelement will constitute the complete model. The Superelement Options dialog box may be used to change the number of the **First level superelement** from the default 1 to any other number and still perform a **Direct analysis**.

The Superelement Options dialog box is, however, normally relevant only when you want to use the superelement technique. Do a superelement analysis as follows: select **Superelement analysis**, set the number of the **Top level superelement** (in Figure 3.9 the number 1000 has been chosen), set the **First level**

superelement number (or accept the default 1), and click **OK**. The Superelement Options dialog box will then close and you are back to the dialog box of the modeller.

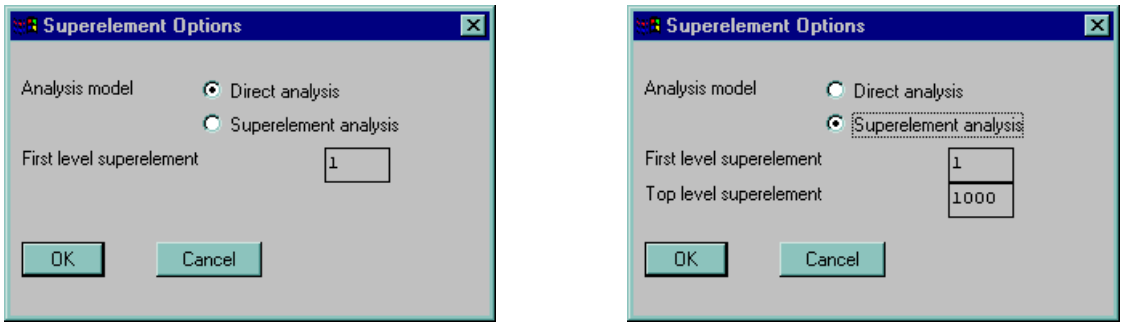

Figure 3.9 Set superelement number

For each new first level superelement to be created you need to open the Superelement Options dialog box and change the number of the **First level superelement**.

Having created all first level superelements they must be assembled to form the complete model. Give **Model | Superelement Handling Presel** and the dialog box shown in Figure 3.10 appears.

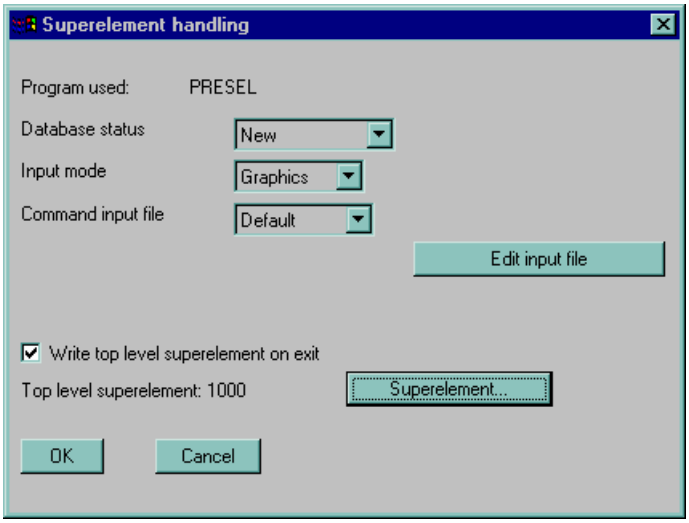

Figure 3.10 Assemble first level superelements to form the complete model — Presel

By default, a **Command input file** named **Default** has been selected. This file will read all first level superelements found on the project directory into the Presel database thereby removing the need for giving the READ command inside Presel.

Otherwise, the functionality of the Superelement handling dialog box is the same as for the dialog of the first level superelement modeller.

3.4.5 Sub-modelling — Submod

Submod is SESAM's program for extracting displacements from a global model (for which an analysis already has been performed and a results file exists) and apply these as prescribed displacements at the boundary of a sub-model. Typically, the sub-model covers a part of the global model and has a finer mesh to produce more accurate results. The sub-model may also incorporate minor geometry changes.

Preconditions for running Submod are:

- A global analysis must already have been performed, i.e. there exists a Results Interface File.
- A sub-model (local model) has been created with prescribed type of boundary conditions at its edges.
- The global and sub-model are parallel projects, i.e. they have the same root directory.

SESAM's sub-modelling technique is available from Manager through **Model | Sub-modelling Submod**. The dialog box, see Figure 3.11, allows making the appropriate choices for how to extract displacements from a global model and apply these to the sub-model:

a Let Analysis type be **Specify** rather than **User defined**. In the latter case you will have to edit an input file.

b Select the appropriate global project from the **Global project** list box.

c Specify the **Global top superelement** number (may well be the same as the one of the sub-model).

d Set **Global superelement scope** to **All** or **Specify**. The latter case is for limiting the search for match in the global model to certain superelements in order to speed up the process. This narrowing of the search is only important for rather large global analysis models.

e Adjust the **Coordinate tolerance** if needed (determines match between global model and sub-model).

f Select either **Node to Node** or **Node to Element** matching:

— **Node to Node** matching implies picking displacements from nodes of the global model that are within the coordinate tolerance away from corresponding nodes of the sub-model.

— **Node to Element** matching implies picking displacements from within the elements of the global model.

Normally, the **Node to Element** matching is the most suitable method as it does not require matching between the boundary nodes of the sub-model and global model nodes.

g Check the **Mix 3 and 6 Dof** box if you have a sub-model containing nodes with 3 degrees of freedom (membrane and solid elements) and a global model with 6 degrees of freedom (shell elements).

Having made all the appropriate choices click **Run sub-modelling** to start Submod. A successful Submod run will produce a revised Input Interface File (Tn.FEM files where n is the number of the sub-model superelements). The revision is that prescribed (specified) displacements have been added. This revised model may then be analysed in a standard Sestra execution.

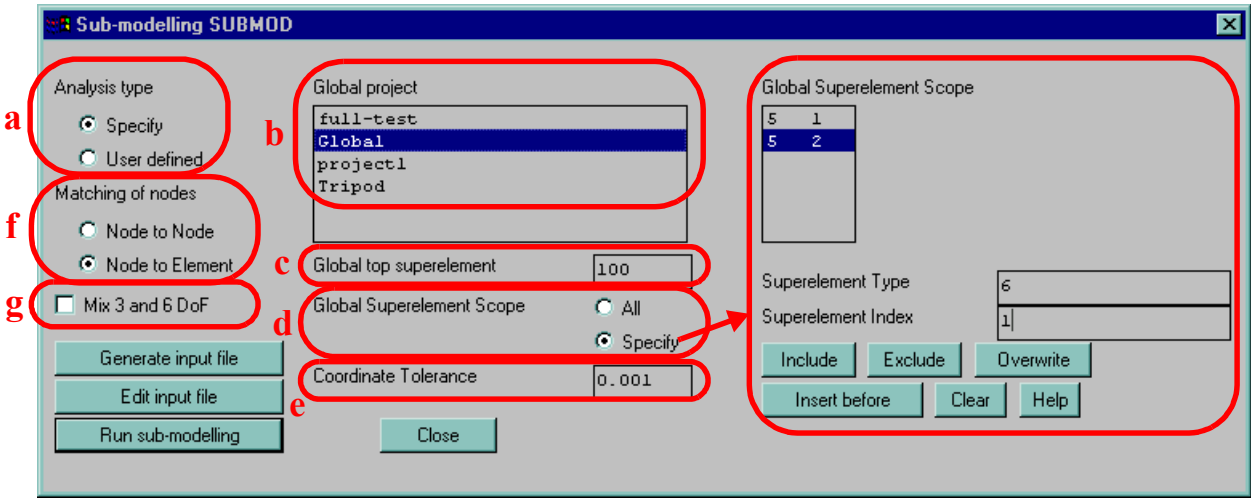

Figure 3.11 Sub-modelling — Submod

3.4.6 Hydrodynamic Modelling — Prewad

If you want to use Wadam to compute motion of and/or wave loads on a structure you need to give hydrodynamic and other data, i.e. the analysis control data of Wadam. This is available through **Model | Hydro Modelling Prewad**. The dialog box shown in Figure 3.12 appears. It has similar functionality as the dialog box of the modellers; see Section 3.4.1. In the case of Prewad you will normally have a **Command input file** containing the complete input in which case there is no need to check the **Run interactively after command input file processing** box. The **Dataset** number that identifies the input to Wadam may be changed from the default value of 1. Check the **Write dataset on exit** to produce the Wadam analysis control data (Wadam#.FEM where # is the dataset number). If you want Manager to start Wadam once the Prewad execution has finished then check the **Run wave load analysis after Prewad** box. If you do not check this box then run Wadam as described in Section 3.5.2.

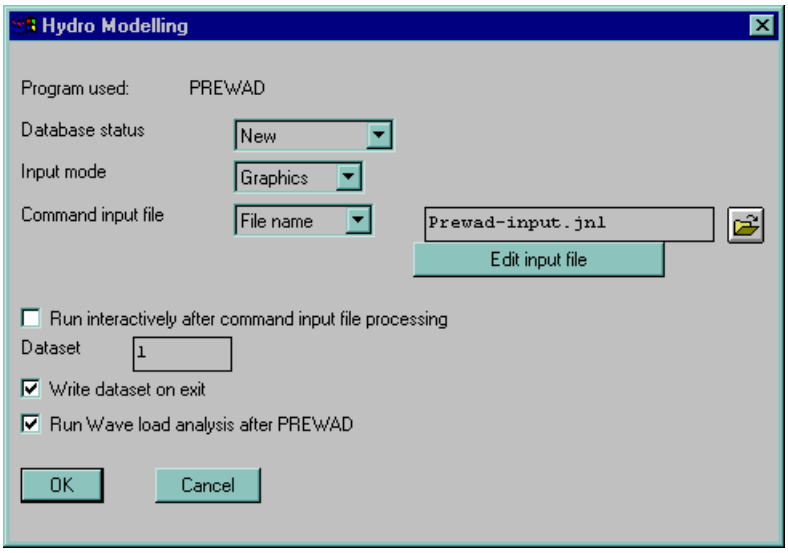

Figure 3.12 Hydrodynamic modelling for radiation/diffraction theory — Prewad and Wadam

3.5 Hydrodynamic Analysis and Loads

Under **Load** in the menu bar the following options are available:

- **Wave Loading Wajac** for wave load calculation for frame structures
- **Wave Loading Wadam** for motion response and wave load calculation for general structures
- **Wave Loading Waveship** for motion response and wave load calculation for ship structures
- **Edit Program Files** for reviewing the print (listing) files

3.5.1 Wave Loads on Jackets — Wajac

Use **Load | Wave Loading Wajac** to open the Wajac dialog box.

If you have already an analysis control data file for Wajac then specify this by selecting **User defined** input file as shown to the left in Figure 3.13. Click **Run wave load analysis** to start the Wajac execution.

Alternatively, you may do as follows:

- Generate a template for a Wajac analysis control data file by selecting the appropriate **Input file template**, e.g. **Deterministic design wave**, and clicking the **Generate input file** button as shown to the right in Figure 3.13.
- Click the **Edit input file** button to enter an editor for modifying the Wajac analysis control data.
- Leave the editor and click **Run wave load analysis** to start the Wajac execution.

Note: A user specified input file will always be copied to a file named Wajac#.inp where # is the top level superelement number.

Use **Load | Edit Program Files** to review the print (listing) of the program.

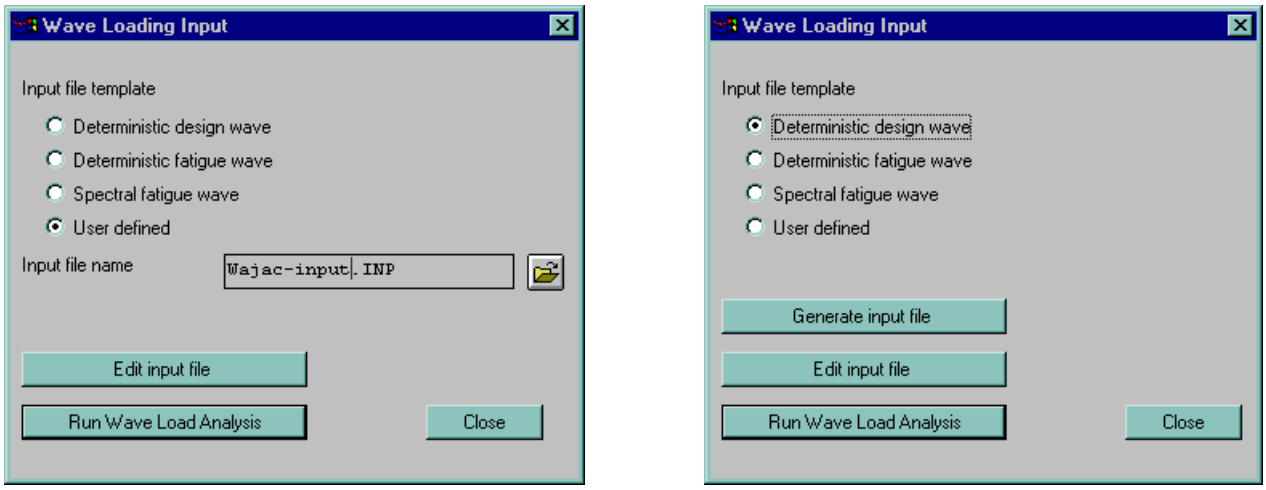

Figure 3.13 Wave load calculation for frame structures — Wajac

3.5.2 Wave Loads on General Structures — Wadam

Once the hydrodynamic modelling has been performed (see Section 3.4.6) running a wave load analysis of a general structure using Wadam is simply done by using **Load | Wave Loading Wadam** to open the Wadam dialog box in which you specify the **Dataset** number and click **OK**.

Use **Load | Edit Program Files** to review the print (listing) of the program.

3.5.3 Loads on General Structures — Waveship

Use **Load | Wave Loading Waveship** to open the Waveship dialog box.

If you have already an analysis control data file for Waveship then specify this by selecting **User defined** input file as shown in Figure 3.14. Click **Run wave load analysis** to start the Waveship execution.

Alternatively, you may select **Generate template**, click the **Edit input file** button, modify the input, exit the editor and click **Run wave load analysis** to start Waveship.

Use **Load | Edit Program Files** to review the print (listing) of the program.

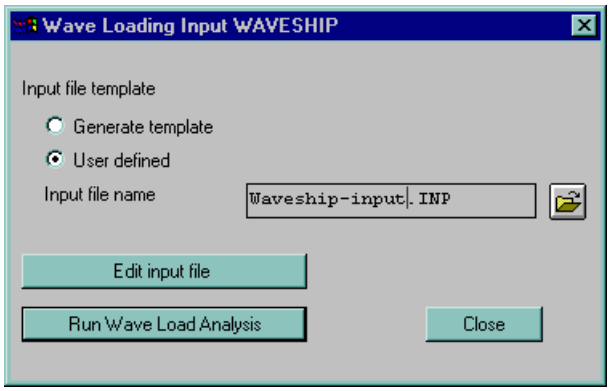

Figure 3.14 Wave load calculation for ships — Waveship

3.6 Structural Analysis

Under **Analysis** in the menu bar the following options are available:

- **Linear Analysis Sestra** for linear structural analysis
- **Nonlinear Analysis Advance** for nonlinear structural analysis
- **Structure Pile Soil** for structure-pile-soil interaction analysis
- **Mooring Analysis Mimosa** for mooring analyis
- **Edit Program Files** for reviewing the print (listing) files

3.6.1 Linear Structural Analysis — Sestra

Give **Analysis | Linear Analysis Sestra** to open the dialog box for linear analysis. Static analysis and free vibration (eigenvalue) analysis are enabled through the dialog box. Dynamic forced response analysis in frequency and time domain requires the user to manually edit the Sestra input file to suit his purpose.

- The default **Analysis type** is **Static**, refer to Figure 3.15.
  - — Check the **Datacheck only** box if you only want to perform a datacheck in Sestra.
  - — Check the **Centre of gravity** box if you want print of global mass matrix, coordinates of the centroid and mass matrix at the centroid. (Checking this box corresponds to MSUM=1 on CMAS record in Sestra input.)
  - — Check the **Split working files** box if you want to use the split working file or parallel computation features. In this case you probably need to edit the input somewhat too (click **Generate input file** and thereafter **Edit input file**).
  - — Check boxes under **Store for postprocessing**: **Beam distributed loads** are required to be stored (on the Results Interface File) if beam moment and force diagrams are to be displayed in Framework. **Surface distributed loads** are required to be stored if such loads are to be displayed in the postprocessing of result in Xtract.
  - — Choose **Equation solver**. This choice should normally be **Multifront** as this is by far the most efficient solver in Sestra.
  - — Check **Prepare Postfem database** if you want to use the Postfem postprocessor. In such case selecting result attributes to be stored on the database is relevant by clicking the **Result attributes** button.

    Note that Xtract has replaced Postfem as the general postprocessor of SESAM and should be your preferred postprocessor. You would only in certain cases need to use Postfem.
  - — Clicking **Generate input file** and thereafter **Edit input file** prior to running the analysis allows you to modify the input.
  - — Click **Run Linear Analysis** to start the analysis. Depending on the size of the model the analysis may require some time.

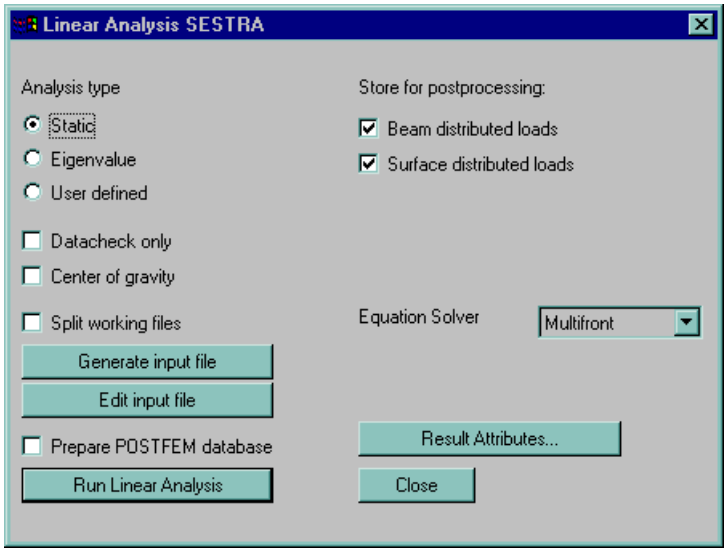

Figure 3.15 Linear static structural analysis — Sestra

- When selecting **Eigenvalue** type of analysis you need to choose method — **Householder**, **Lanczos** or **Subspace Iteration** — and **Number of modes** (eigenvalues) to compute; see Figure 3.16.

  — See above about the **Datacheck only**, the **Split working files** and the **Prepare Postfem database** boxes.

  — Click **Run Linear Analysis** to start the eigenvalue analysis. Depending on the size of the model the analysis may require some time.

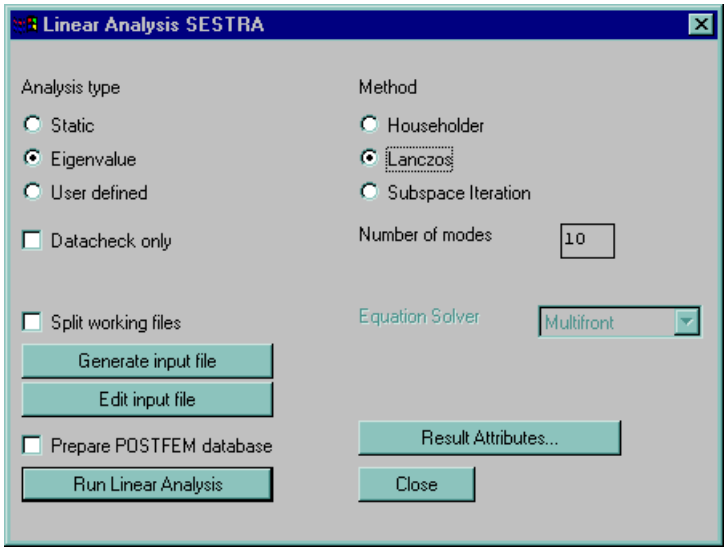

Figure 3.16 Linear eigenvalue (free vibration) structural analysis — Sestra

During a Sestra run the Manager window is minimized and when the execution has completed the Manager window is restored. If the execution for some reason fails a pop-up window will appear with information on a possible cause of the failure. This information is an extract from the Sestra print file (listing) that contains additional information which may be of importance. Figure 3.17 shows three examples of such information.

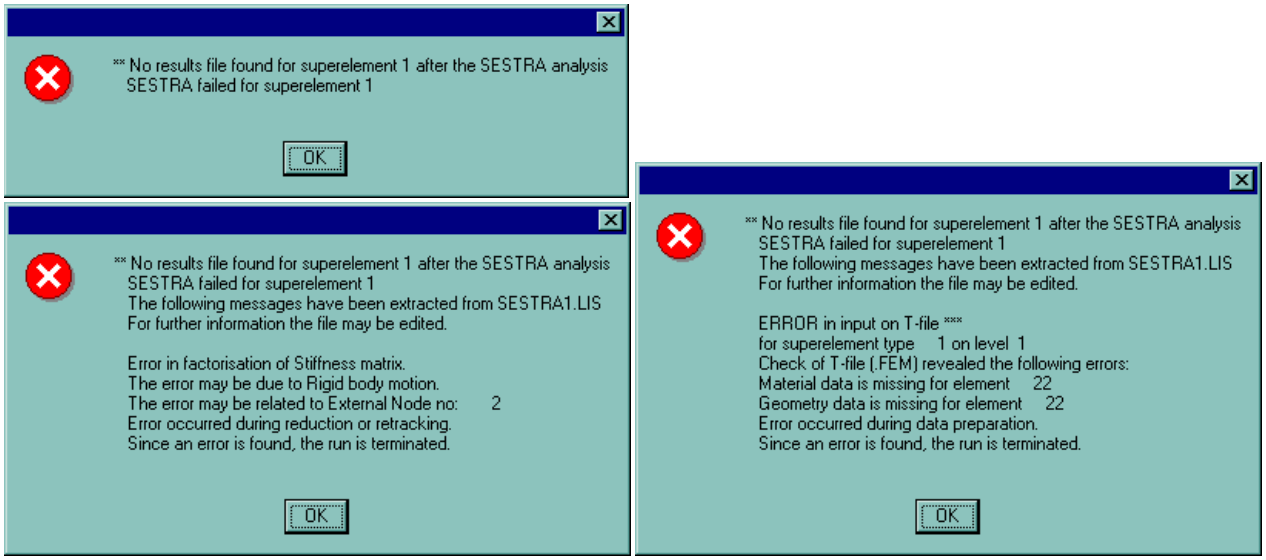

Figure 3.17 Linear structural analysis failure information

- If you have introduced a prefix for the results file by editing the RNAM command of the Sestra input file (rather than using Options | Result Prefix in Manager) then Manager will not recognise the results file and the message shown to the upper left of Figure 3.17 appears.
- If your model lacks proper fixation in space or contains internal parts that are not fixed properly, e.g. due to improper use of hinges or non-structural beams, then the message shown to the lower left of Figure 3.17 appears.
- If elements of your model lacks material data, beam cross section or shell/plate thickness then the message shown to the lower right of Figure 3.17 appears.

Use **Analysis | Edit Program Files** to review the print (listing) of the program.

It is sound QA practice to always check the analysis print file for possible warnings and error messages. You should also check the sum of reaction forces (at the end of the file) against the sum of loads (search for the text "sum of loads" corresponding to the top level superelement).

3.6.2 Nonlinear Structural Analysis — Advance

Under **Analysis | Nonlinear Analysis Advance** there are two options: **Generate Input** and **Nonlinear Analysis**. These are explained in the following.

- **Generate Input** — opens the dialog box shown in Figure 3.18 for converting an Input Interface File (a single file, normally T1.FEM, i.e. not a superelement model) to Advance input:

  1 Click **Translate FEM data** to translate T1.FEM to Advance linear analysis input. The result is T1.inp.

  2 Click **Generate input file** to create Advance input based on the chosen **Analysis type** and other data. The input is put in separate files: adv1.inp is the main file containing references to other files, T1.mat is the material file (edit by **Edit material file**) and T1.lcs is the load file (edit by **Edit load file**). Simple templates for the various analysis types are generated in these files. The files usually need further editing to incorporate more complicated nonlinear analysis concepts.

  3 Options for **Nodal Output**, **Element Output**, **Energy Output** and **Contact Output** are written to the generated Advance input as requested by the check boxes.

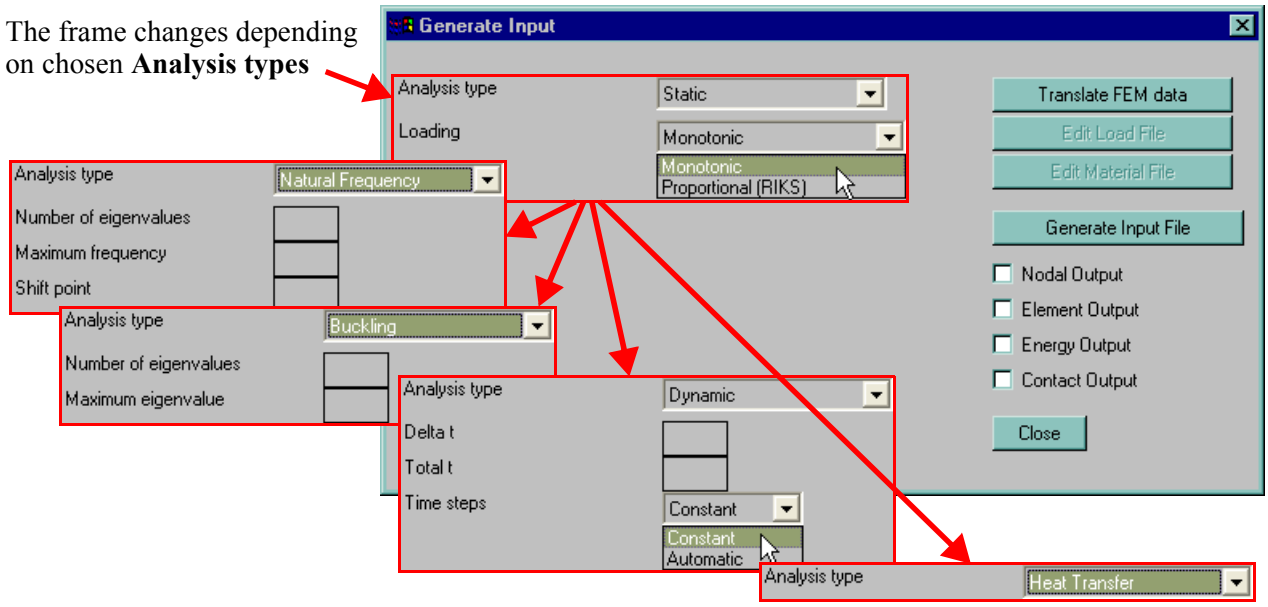

Figure 3.18 Nonlinear structural analysis, generating Advance input

- **Nonlinear Analysis** — opens the dialog box shown in Figure 3.19 for running Advance.

  Choose between generating Advance input from an Input Interface File by clicking **Generate input file** — which opens the dialog box shown in Figure 3.18 explained in the above item — and specifying an Advance input file in the edit box.

  There are four analysis sequences available:

a **Datacheck and Analysis** — involves a datacheck of the input file(s) using Advance_Pre and if successful followed by an equation solution using Advance. If the datacheck fails the user is notified through a pop-up window and the .dat file. Model information is stored in the .res file.

b **Datacheck only** — stops after a datacheck of the input file(s) using Advance_Pre.

c **Analysis Only** — involves an equation solution using Advance. A prerequisite for this analysis is that a successful datacheck has been performed.

d **Restart** — involves a restart based on a previous successful analysis. All necessary results files of the previous analysis must be available in the project directory. The .res file of the previous analysis and the restart input file must be specified in the respective fields.

Memory parameters and various other analysis options can be adjusted by clicking **Analysis options**.

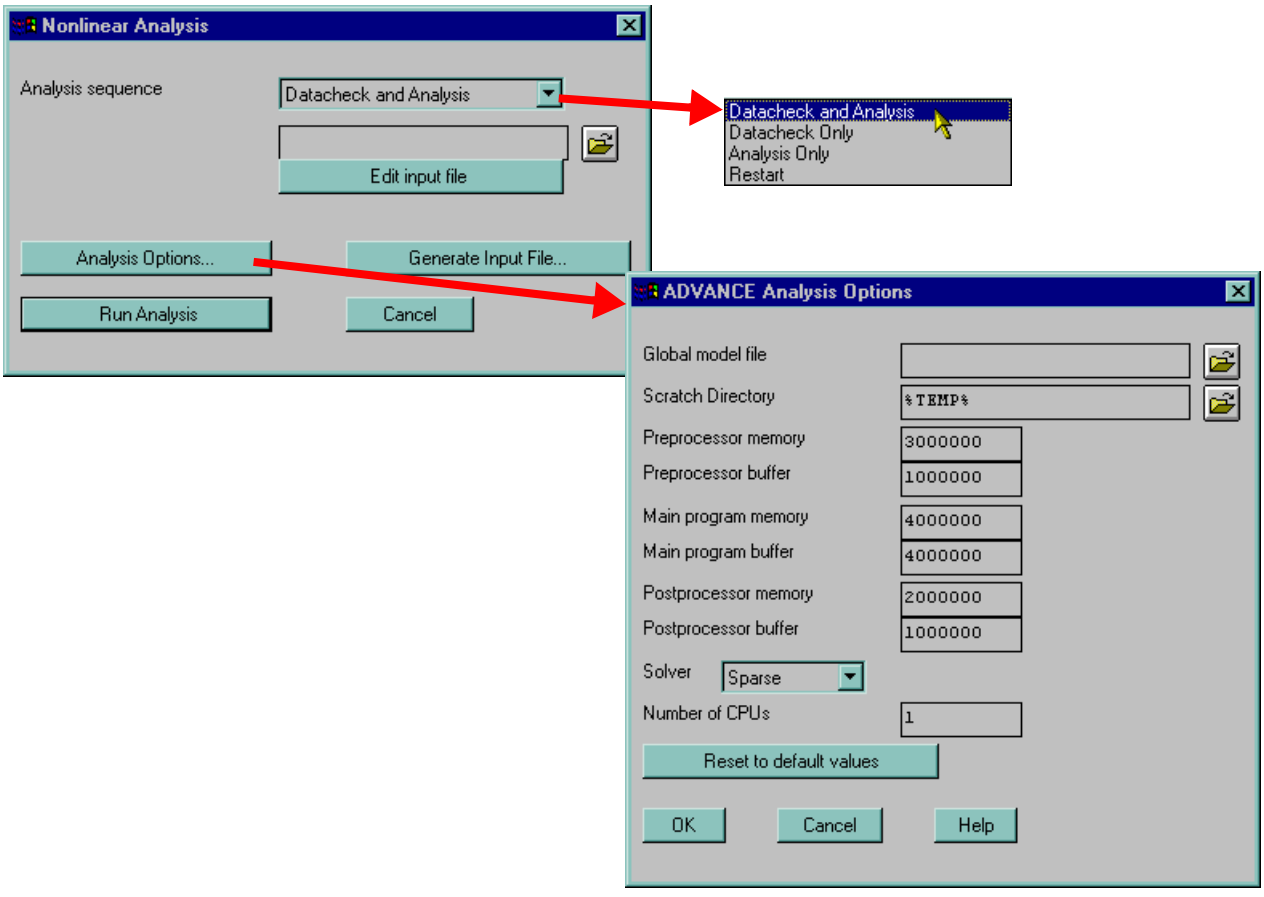

Figure 3.19 Nonlinear structural analysis, running Advance

3.6.3 Structure-Pile-Soil Interaction Analysis — Sestra and Splice

Give **Analysis | Structure Pile Soil** to open the dialog box for structure-pile-soil interaction analysis. This is a combined linear structural analysis, using Sestra, and a non-linear pile-soil analysis, using Splice. This kind of analysis is relevant for all structures with pile foundation, typically jackets.

The dialog box opens for running programs in sequence as follows (see Figure 3.20):

- First select **Structure reduction**. Unless you have some specific requirements to the Sestra input accept the default input by clicking **Generate template** followed by **Run Analysis**.
- Secondly, select **Pile data** to run Pilgen in order to produce the pile data file required by Splice. Select a previously prepared input file. Note that if the piles have been modelled in Preframe (together with the jacket) then this step should be skipped.
- Thirdly, select **Soil data** to run Gensod in order to produce the soil data file required by Splice. Select a previously prepared input file. Alternatively, you may generate a template in two alternative forms (only **API p-y curves** or also t-z and q-z curves by **API p-y/t-z/q-z curves**) and edit this to suit your purpose. Note that if the soil has been modelled in Preframe (together with the jacket) then skip this step.
- The non-linear pile-soil interaction analysis is performed in Splice by selecting **Structure-Pile-Soil interaction**. Select a previously prepared input file. Alternatively, you may generate a template in two alternative forms (**Iterative solution** or **Incremental solution**) and edit this to suit your purpose.
- The final step is **Structure retracking**. Unless you have some specific requirements to the Sestra input accept the default input by clicking **Generate template** followed by **Run Analysis**.

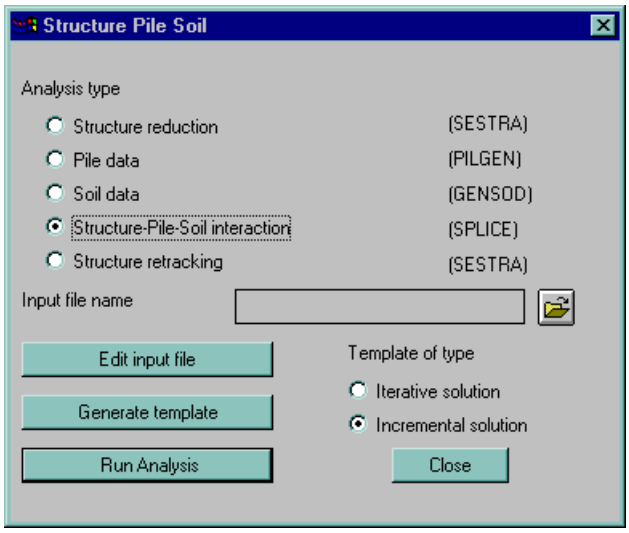

Figure 3.20 Structure-pile-soil interaction analysis — Sestra and Splice

Use **Analysis | Edit Program Files** to review the print (listing) of the programs (Sestra, Pilgen, Gensod and Splice). Note that the print file from the Sestra reduction is named SESTRA#RD.LIS while the print file from the retracking is named SESTRA#RT.LIS (# is the top level superelement number).

3.6.4 Mooring Analysis — Mimosa

Give **Analysis | Mooring Analysis Mimosa** to open the dialog box shown to the left in Figure 3.21. Click **Run Mooring Analysis** to run Mimosa interactively. Alternatively, click **File name** and select an input file that has been prepared in advance.

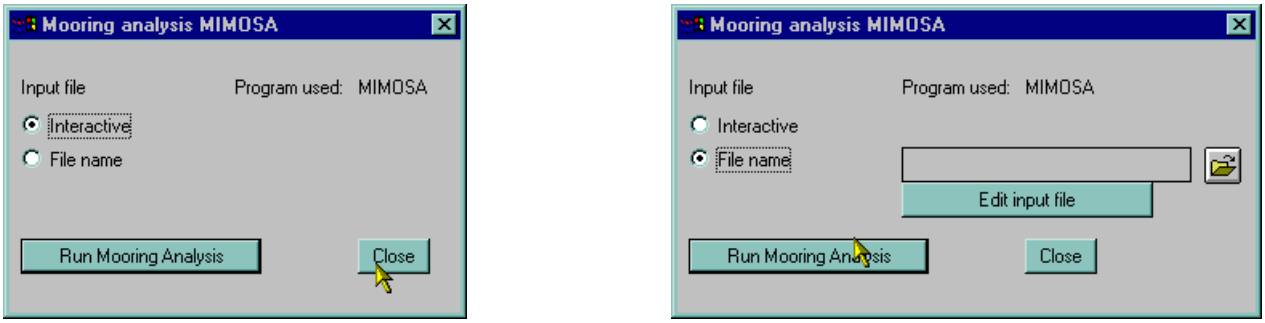

Figure 3.21 Mooring analysis — Mimosa

3.7 Postprocessing Results

Under **Result** in the menu bar the following options are available:

- **Frame Framework** for code checking frame structures
- **General Postfem** for general FE results presentation, the old postprocessor
- **General Xtract** for general FE results presentation, the new postprocessor
- **Response Postresp** for statistical response processing (frequency domain)
- **Sectional Results Cutres** for sectional results presentation
- **Plate Code Check Platework** for code checking stiffened plate structures
- **Shell fatigue Stofat** for stochastic fatigue analysis of shell/plate structures
- **Postprocessing Advance** for presentation of results from a non-linear Advance analysis
- **Edit Program Files** for editing a file, e.g. a command log (journal) file

3.7.1 Code Checking Frame Structures — Framework

Give **Result | Frame Framework** to open the dialog box shown in Figure 3.22. By default the **Database status** is **New** and a **Default Command input file** will be established and processed. This involves opening the Results Interface File (FILE OPEN ...), transferring the superelement with the given **Superelement key** (FILE TRANSFER ...) and displaying the model. (For a direct analysis, i.e. only a single superelement, the **Superelement key** edit box is irrelevant and will not be shown.)

If you do not use a **Command input file** (neither the **Default** nor a file of your own) then the first two commands to give in Framework will be FILE OPEN and FILE TRANSFER.

If you have several first level superelements you may not know which **Superelement key** to select (it is not the same as the superelement number). You may then through **Utility | Run** (see Section 3.8) start Prepost and do as follows:

- Open the Results Interface File using the command OPEN SIN prefix R# OLD READ-ONLY.
  — prefix is normally void in which case you give apostrophe-space-apostrophe (' ').
  — # is the top level superelement number.
- Give the command PRINT SUPERELEMENT.
- Go to the Message window (lower left of your screen) and see the table over superelements that are contained in the Results Interface File. Determine the proper **Superelement key** from this table.
- Exit Prepost.

Note that the given **Superelement key** will form a part of the name of the model file. To avoid confusion when the **Default Command input file** is not used the **Superelement key** should therefore preferably be consistent with the key given in the FILE TRANSFER command.

If your Results Interface File has a prefix, i.e. it has a character string in front of the mandatory R of the file name, you must set this prefix through **Options | Result Prefix** or else the **Default Command input file** will not work.

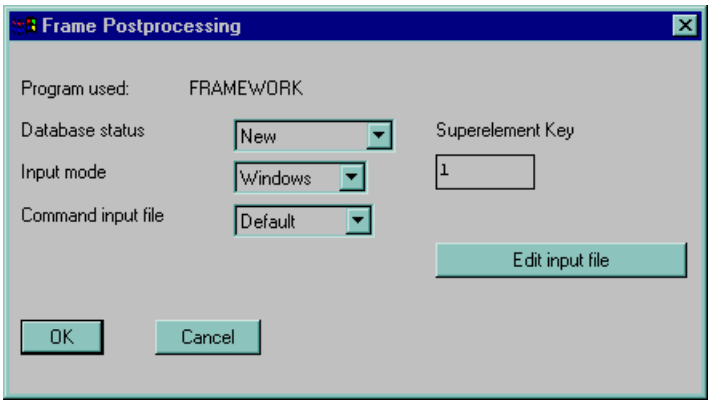

Figure 3.22 Code checking frame structures — Framework

3.7.2 General FE Results Presentation — Postfem

Give **Result | General Postfem** to open the dialog box shown to the left in Figure 3.23. A new Postfem database will be created (by a Prepost run in the background) provided that **Database status** is set to **New**. **Database status Old** presuppose that a Postfem database exists already. (**Analysis** of the dialog box should be left as **Linear** and **Input mode** should be left as **Graphics**.)

A **Default Command input file** will be established and processed. This involves selecting the top level superelement and displaying it.

If your Results Interface File has a prefix, i.e. it has a character string in front of the mandatory R of the file name, you must set this prefix by clicking **Result Prefix** or else the **Default Command input file** will not work. Setting the prefix is also available through **Options | Result Prefix**.

When Database status is **New** you may determine which result components the Postfem database should contain by clicking **Result Attributes**; see Figure 3.23.

Note: If you select your own Command input file then this file must start with a blank line. This blank line is typically followed by the SELECT or BUILD commands. Further, note that the command log (journal) file from Postfem will not contain this blank line so before you use a command log file as Command input file insert the blank line.

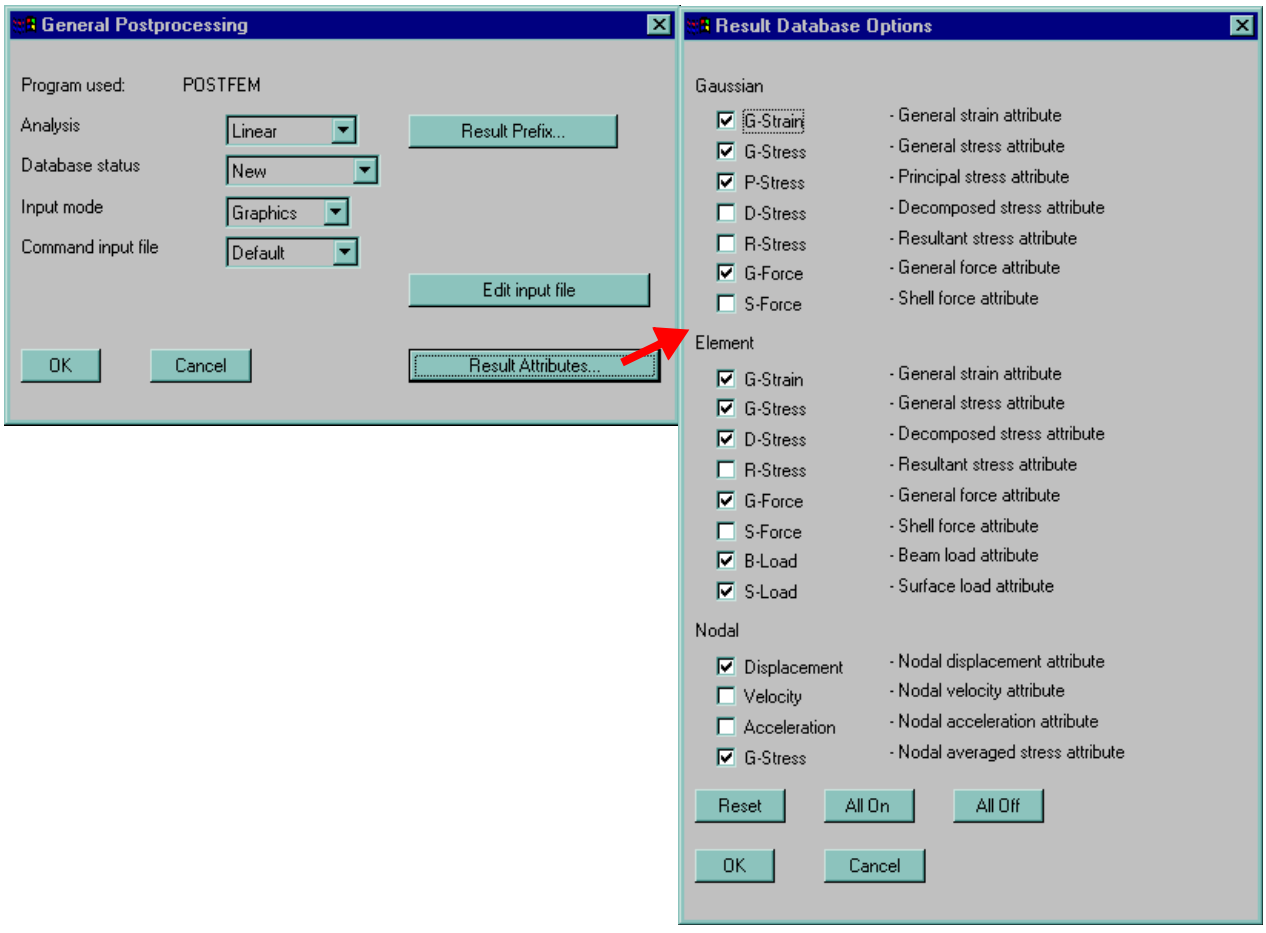

Figure 3.23 General FE results presentation using Postfem

3.7.3 General FE Results Presentation — Xtract

Note: Xtract is currently only available on PC.

Give **Result | General Xtract** to open the dialog box shown in Figure 3.24. By default the **Database status** is **New**. A **Default Command input file** will be established and processed. This involves opening the Results Interface File (R-file), selecting the top level superelement and presenting contours for the result component Nodes DISPLACEMENT ALL (of result case 1 by default).

If your Results Interface File has a prefix, i.e. it has a character string in front of the mandatory R of the file name, you must set this prefix by clicking **Result Prefix** or else the **Default Command input file** will not work. Setting the prefix is also available through **Options | Result Prefix**.

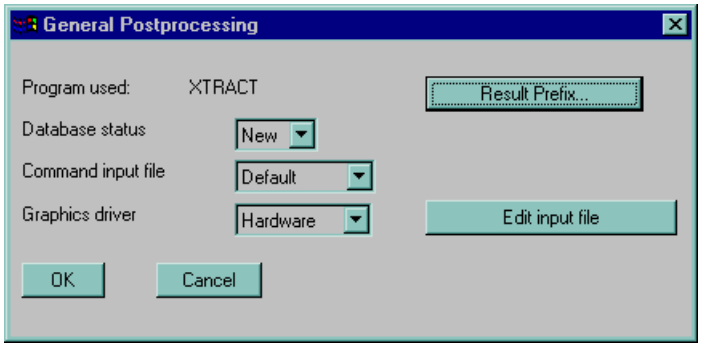

Figure 3.24 General FE results presentation using Xtract

3.7.4 Statistical Response Processing — Postresp

Give **Result | Response Postresp** to open the dialog box shown in Figure 3.25. A **Default Command input file** will be established and processed. This involves opening the Hydrodynamic Results Interface File (G-file) in SIF-format.

If your Hydrodynamic Results Interface File has a prefix, i.e. it has a character string in front of the mandatory G of the file name, you must set this prefix through **Options | Result Prefix** or else the **Default Command input file** will not work.

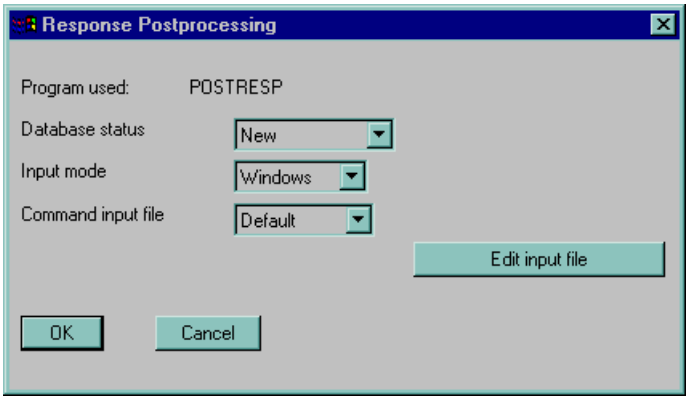

Figure 3.25 Statistical response processing — Postresp

3.7.5 Presentation of Sectional Results — Cutres

Give **Result | Sectional Results Cutres** to open the dialog box shown in Figure 3.26. By default the **Database status** is **New**. A **Default Command input file** will be established and processed. This involves opening the Results Interface File (FILE OPEN SIN-DIRECT-ACCESS ...) and creating an assembly of 9 linearly distributed sections in the X-direction of the top level superelement (CREATE ASSEMBLY-OF-SECTIONS SECTIONS LINEAR ; 8). This **Default Command input file** may not be according to your needs in which case you should select another **Command input file** or **None**.

If your Results Interface File has a prefix, i.e. it has a character string in front of the mandatory R of the file name, you must set this prefix through **Options | Result Prefix** or else the **Default Command input file** will not work.

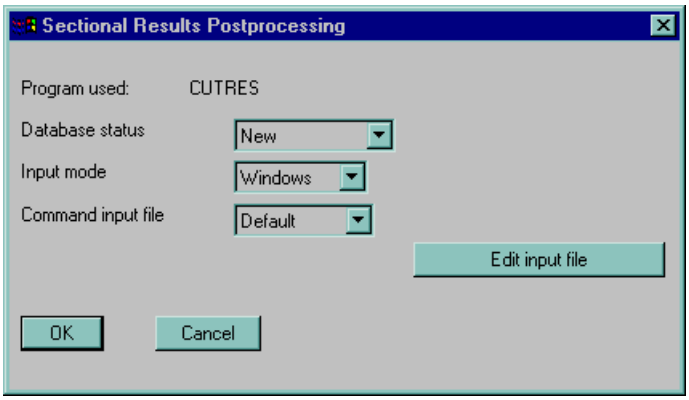

Figure 3.26 Presentation of sectional results — Cutres

3.7.6 Code Checking Stiffened Plate Structures — Platework

Give **Result | Plate Platework** to open the dialog box shown in Figure 3.27. By default the **Database status** is **New** and a **Default Command input file** will be established and processed. This involves opening the Results Interface File (READ SIN-DIRECT-ACCESS ...).

If your Results Interface File has a prefix, i.e. it has a character string in front of the mandatory R of the file name, you must set this prefix through **Options | Result Prefix** or else the **Default Command input file** will not work.

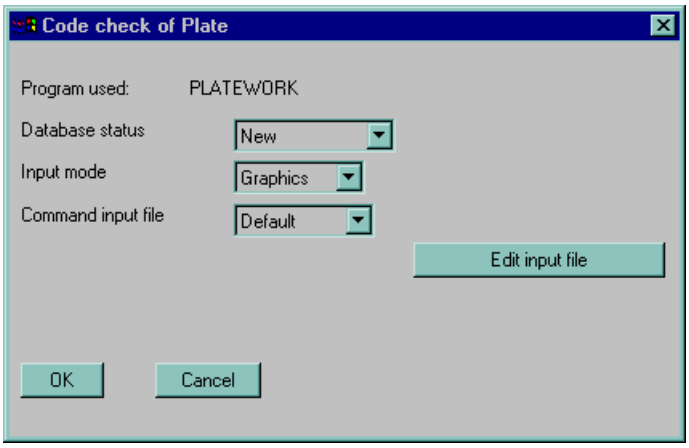

Figure 3.27 Code checking stiffened plate structures — Platework

3.7.7 Stochastic Fatigue of Shell/Plate Structures — Stofat

Give **Result | Shell fatigue Stofat** to open the dialog box shown in Figure 3.28. By default the **Database status** is **New** and a **Default Command input file** will be established and processed. This involves opening the Results Interface File (FILE OPEN ...), transferring the superelement with the given **Superelement key** (FILE TRANSFER ...) and displaying the model. (For a direct analysis, i.e. only a single superelement, the **Superelement key** edit box is irrelevant and will not be shown.)

If your Results Interface File has a prefix, i.e. it has a character string in front of the mandatory R of the file name, you must set this prefix through **Options | Result Prefix** or else the **Default Command input file** will not work.

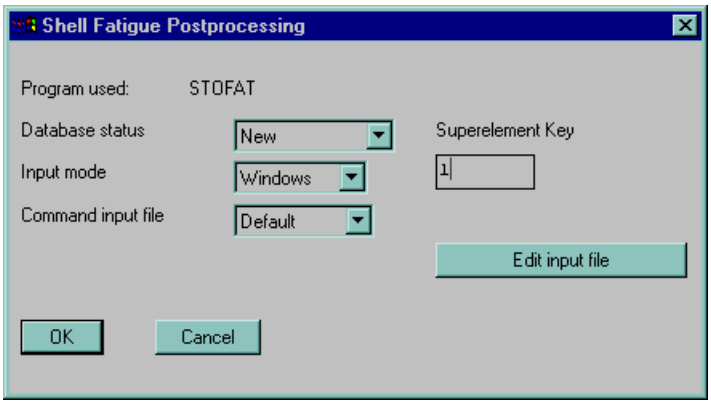

Figure 3.28 Stochastic fatigue analysis of shell/plate — Stofat

3.7.8 Presentation of Results from Non-Linear Analysis — Advance_Post

An Advance analysis produces a results file with extension .res. Rather than using Xtract, the general post-processor of SESAM, a special purpose postprocessor is used: Advance_Post. Give **Result | Postprocessing Advance** to open the dialog box shown in Figure 3.29. The Advance results file will appear. Use the browser button to select another results file.

The dialog box opening by clicking **Advance Options** is for for adjusting various memory parameters. Only the **Postprocessor memory** and **Postprocessor buffer** parameters are relevant in this case. Adjust these if the default parameters are insufficient for your case.

Note: Exceed (version 6.0 or higher) must be running for Advance_Post to be accessed.

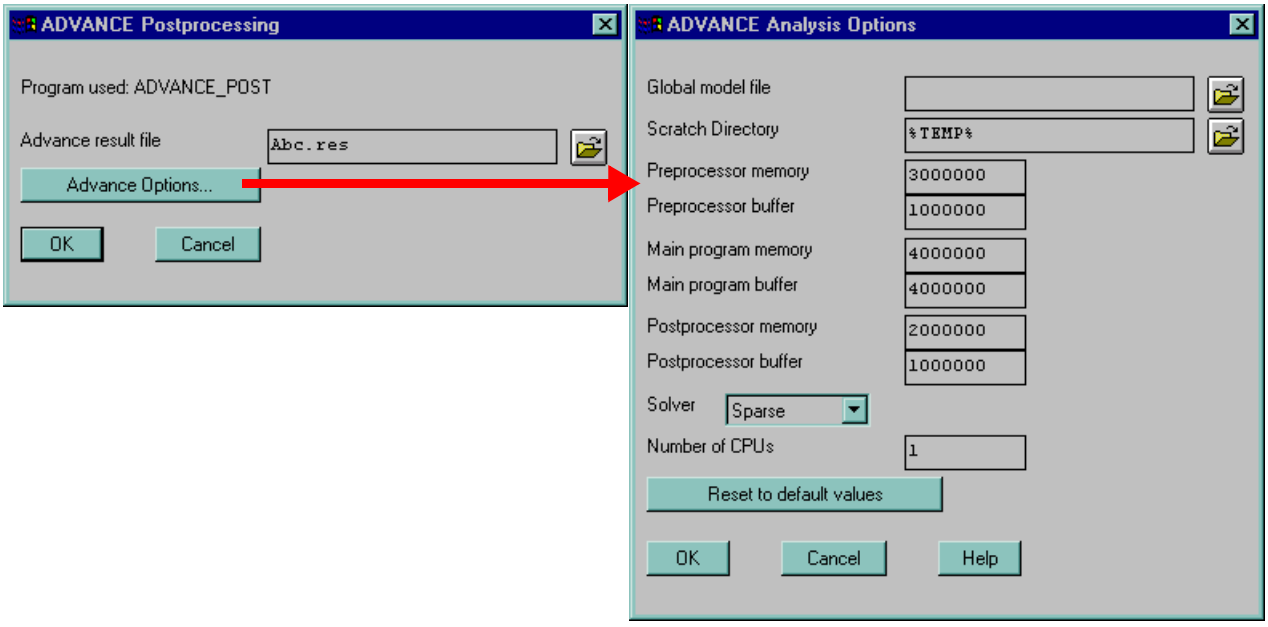

Figure 3.29 Presentation of results from non-linear analysis — Advance_Post

3.8 Utility Run

The **Model**, **Load**, **Analysis** and **Result** menus provide access to the *controlled* (see Section 3.9) programs. The *non-controlled* programs are accessed through the menu **Utility | Run**. This menu may also be used if you want to overrule Manager's convention for file names for controlled programs.

Moreover, if you want to use the partly controlled auxiliary program Prepost to manipulate the Results Interface File, e.g. converting it from Norsam (SIN) to Formatted (SIF), you must use **Utility | Run**. (The only controlled execution of Prepost is for creating the Postfem database.)

Finally, **Utility | Run** allows you to set other options for the program execution including modifying directly the command line arguments. These arguments are program dependent settings controlling the program execution; see the individual user manuals for more information on this.

Note: As a generic tool the various choices and options in the Utility | Run dialog box may not be relevant for a given program.

Giving **Utility | Run** opens the dialog box shown in Figure 3.30. The alphabetic list below refers to the figure.

a In the **Program** field you may select any SESAM program available to you.

b Select other than the official version, e.g. a development version.

c Select **Graphics Input**, **Windows** or **Background**. **Graphics** is the graphical user interface of progarms like Prefem, Preframe and Presel while **Windows** is the graphical user interface of programs like Framework and Cutres. **Background** is for programs like Sestra and Wajac.

d Choose to use the defaults for view point and plot or not. Click **Startup** and **Plot** to modify these default settings (equivalent to **Options | Program Startup** and **Options | Plot** respectively).

e Specify **New** or **Old Database** and give the **Prefix** and **Name** of the database file.

f Select a **Command input file** if desired.

g The appearance of the **Utility | Run** dialog box will to some extent depend on the selected program. To exemplify this the dialog box for Sestra is also shown in Figure 3.30. The fields for **Database** and **Command Input File** are irrelevant and have been removed. Some new fields for input and output files appear.

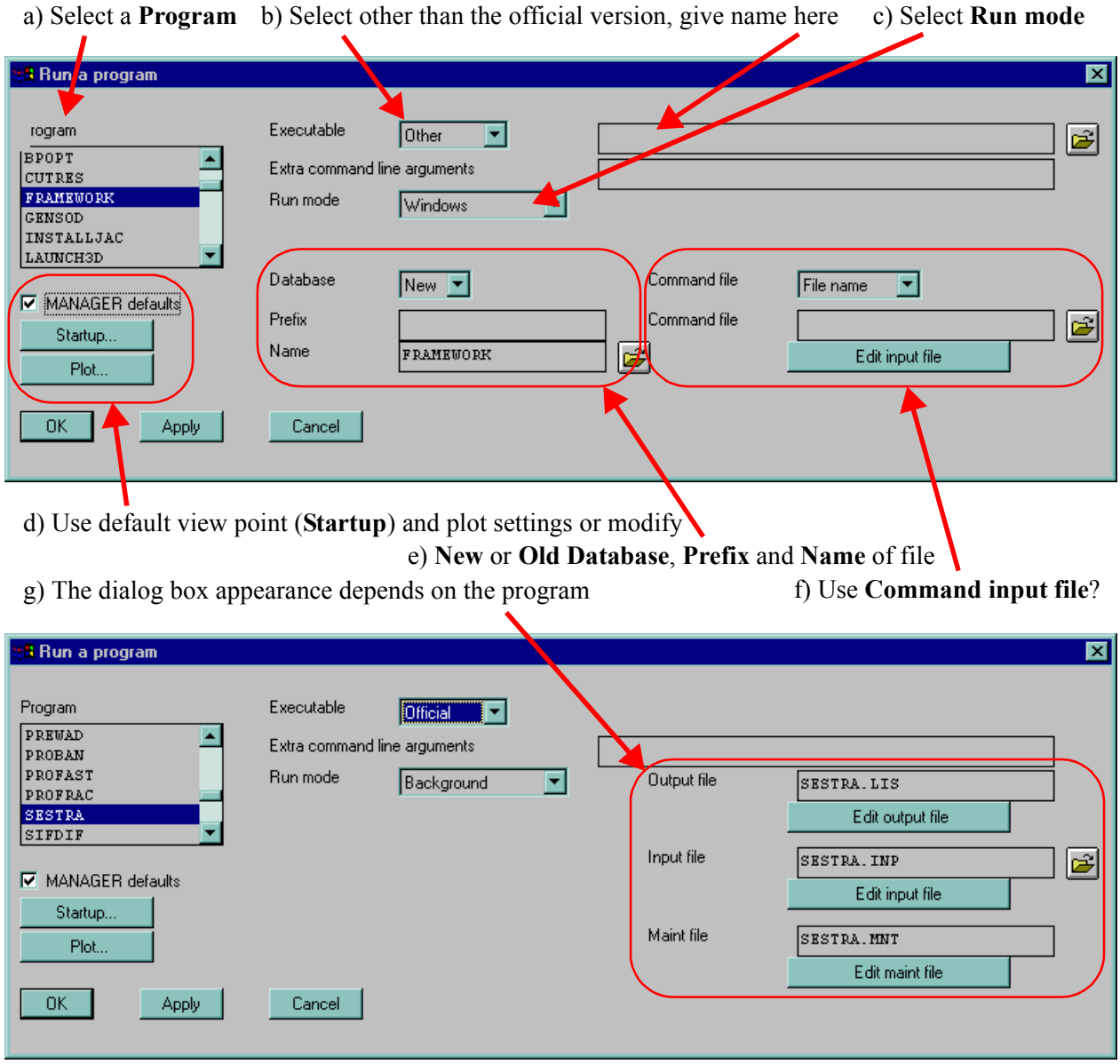

Figure 3.30 Running programs through Utility | Run

3.9 Controlled and Non-Controlled Programs

Most SESAM programs are *controlled* but some are not. Controlled means that:

1 Manager will, to some extent, check the consistency and completeness of data before executing the program.

2 Files read and written by the program will be named according to Manager's convention which is explained in Section 4.3.

3 Whether or not an execution is successful is checked and an appropriate message given.

Table 3.1 and Table 3.2 below provide details on the control exercised by Manager over the individual programs.

Note: The non-controlled programs are accessed through the menu Utility | Run (which can be used to run controlled SESAM program as well). See Section 3.8.

Table 3.1 Manager control of SESAM main programs

Main program	Controlled?	Comment
Genie	Partly	Genie may be started from Manager but as Genie itself controls the execution of analysis and postprocessing you may find it more convenient to start Genie directly; see Section 3.4.2. Genie also has its own file naming convention.
Patran-Pre	No	
Prefem	Yes	
Preframe	Yes	
Presel	Yes	
Pretube	Yes	
Submod	Yes	Input for most cases is established through the Sub-modelling dialog box.
Wadam	Yes	
Wajac	Yes	Templates for input are established.
Waveship	Yes	
Wasim	No	
Installjac	No	
Simo	No	
Sestra	Yes	Input for certain simple (and most common) analyses is established.

Table 3.1 Manager control of SESAM main programs

Main program	Controlled?	Comment
Splice	Yes	Common dialog box for running Sestra reduction, Pilgen, Gensod, Splice and Sestra retracking. Templates for input are established.
Advance	Yes	Input Interface File converted to Advance input and additional options given through dialog box. Advance_Pre run from same dialog box.
Usfos	No	
Mimosa	Yes	
Riflex	No	
Postresp	Yes	Only the frequency domain part of Postresp, i.e. not Postresp_Time
Postresp_Time	No	
Xtract	Yes	Execution is detached from Manager.
Postfem	Yes	Execution is detached from Manager.
Framework	Yes	Execution is detached from Manager.
Stofat	Yes	Execution is detached from Manager.
Profast	No	
Cutres	Yes	Execution is detached from Manager.
Platework	Yes	Execution is detached from Manager.
Concode	No	
Proban	No	

Table 3.2 Manager control of SESAM auxiliary programs

Aux program	Controlled?	Comment
Advance_Pre	Yes	Run from same dialog box as Advance; see this.
Advance_Post	Yes	
Bpopt	Partly	Run from Prefem dialog box. For other purposes use **Utility | Run**.
Pilgen	Yes	See Splice
Gensod	Yes	See Splice
Patses	No	
Preaba	No	
Postaba	No	

Table 3.2 Manager control of SESAM auxiliary programs

Aux program	Controlled?	Comment
Precon	No	
Prepost	Partly	For creating a Postfem database it is run from the Postfem dialog box. Results Interface Files are if neccessary converted to SIN Direct access (NORSAM) format when starting Framework, Stofat, Platework, Cutres and Postresp. For other purposes use **Utility | Run**.
Prewad	Yes	
Status	No	Started through tool button 
Struses	No	
Waloco	No	

3.10 Setting Options

Under the **Options** menu the following options are available:

- **Enable All Menus / Activate Menu Check** for allowing all menu options to be selected.

  The need for this option is explained by the following: Manager allows selecting a menu option only when data (files) required to perform the corresponding program execution is available (controlled programs only). For example, unless a properly named Results Interface File exists the **Result** menu will be inaccessible (greyed out). In certain situations this control may limit SESAM's flexibility. To amend this **Enable All Menus** will switch off this control. Manager will then allow options under the **Result** menu to be selected even when no Results Interface File exists and it is up to the user to decide whether this is a meaningful operation.

- **Keep Visible / Hide when Inactive** toggles minimising Manager when Manager starts a program.

  Certain programs (typically preprocessors and analysis programs) are controlled by Manager until their execution has been completed. Other programs (typically postprocessors) are started and then detached. Manager is by default minised during execution of the former group. The option in question changes this behaviour.

- **Hide Command Line / Dump Command Line** toggles dumping the command line arguments in the Manager window when starting a program. Command line arguments are program dependent settings controlling the program execution; see the user manuals for information on command line arguments available for the individual programs. The choices made in the dialog boxes for starting the various programs are passed on to the programs by means of command line arguments. E.g. when starting Prefem the following command line arguments may be used and will be dumped:

  /name="PREFEM1" /status=New /NOHEA /eye-x=0.5 /eye-y=-1.0 /eye-z=0.5 /PL-FO=WINDOWS-PRINTER /interface=line /com-fil="TMPSGI_Prefem1_in"

- **Structure Type** for setting **Type of structure**, also see Section 3.3.

- **Superelement** for selecting between **Direct analysis** and **Superelement analysis** and for setting superelement numbers, also see Section 3.4.4.

- **Result Prefix** for setting a prefix (a preceding character string that optionally may include a directory specification) for the Results Interface File. This is useful if you want to run more than one analysis for the same model.

- **Result Format** for selecting format of the Results Interface File. Note that if you select other than the **SIN Direct access** (NORSAM) format (which is the default choice) then the Results Interface File will automatically be converted (using Prepost) to NORSAM format when starting Framework, Stofat, Platework, Cutres and Postresp. It is, therefore, convenient as well as efficient to use the NORSAM format.

  However, you should convert the Result Interface File (using Prepost) to **SIF Formatted** for storage of results as the NORSAM format is not compatible over time (future versions of SESAM may not be able to access old NORSAM formatted files).

- **Program** for specifying program versions:

  — **Path** for specifying a search path for locating versions of SESAM programs. If a specific program is not found in the first priority area then the second priority area will be searched and so on.

  — **Executable** for specifying the version of an individual program. You may for instance select a special version that you have stored somewhere on your computer or network.

  — **Startup Options** is the same as **Eyepoint Directions** below.

- **Eyepoint Directions** for setting a common view point (eye direction) for all graphic programs.

- **Nonlinear Analysis** for setting options:

  — **Advance Analysis Options** for settings options Advance; see Section 3.6.2.

  — **Abaqus Analysis Options** for settings options for Abaqus.

- **Result Database** for determining which result components the Postfem database should contain; see Section 3.7.2.

- **Plot** for selecting plot format that will be a default setting which may be changed inside the individual programs. Note that on PC the **Windows-Printer** format will send the plot directly to an on-line plotter. (Inside Postfem the plot format SESAM is the same as **Windows-Printer**.)

- **Editor** for selecting editor to use.

- **Directories** for reviewing and changing location of temporary files (e.g. Sestra working files).

3.11 Command Line Arguments

When starting Manager several command line arguments can be used. The most important are:

/PREFIX=directory	Manager starts with the specified directory as the project directory (as if the PROJECT DIRECTORY command had been issued at startup).
/NAME=name /STATUS=New	Create a new project with the specified name. Any existing project with that name will be deleted first.
/NAME=name /STATUS=Old	Open an existing project with the specified name.
/INTERFACE=LINE	Runs Manager with no graphical user interface. It will be able to process line mode commands only.
/COMMAND-FILE=filename	Manager reads the specified command input file at startup.
/FORCED-EXIT	Manager exits immediately after having performed its initialisation and after having executed the command input file specified on the command line.

The arguments can be abbreviated uniquely, also at hyphens. For example, the following command line arguments may be used to set up a project, do a predefined analysis and exit:

```
manager /NAME=project /STA=N /INTERF=L /C-F=C:\Sesamprj\project_in.jnl /Force
```

or

```
manager /PREFIX=directory /INTERF=L /C-F=C:\Sesamprj\project_in.jnl /Force
```

Note: The address of the command input file should either contain a full path specification (as shown here: C:\Sesamprj\) or be relative to the directory where the project files are. When opening or creating a project Manager sets the directory containing the project files as the current directory. This happens before the command input file specified on the command line is opened.

Note: It is necessary to insert directory paths in single-quotes on Unix because the slash character (/) doubles as a directory separator and command line argument separator. In fact, two sets of single-quotes must be used on Unix within each other (e.g. /PREFIX="'/user/project'") because the first set is stripped off by the command shell.

3.12 Command Input Mode

The command input mode is available through the graphical user interface through the following toolbar buttons:


Toggle command input mode


Open and read command input file


Close command input file (only visible when a command input file is open)

When the command input mode is active Manager appears as shown in Figure 3.31.

See Appendix A for examples of command input files.

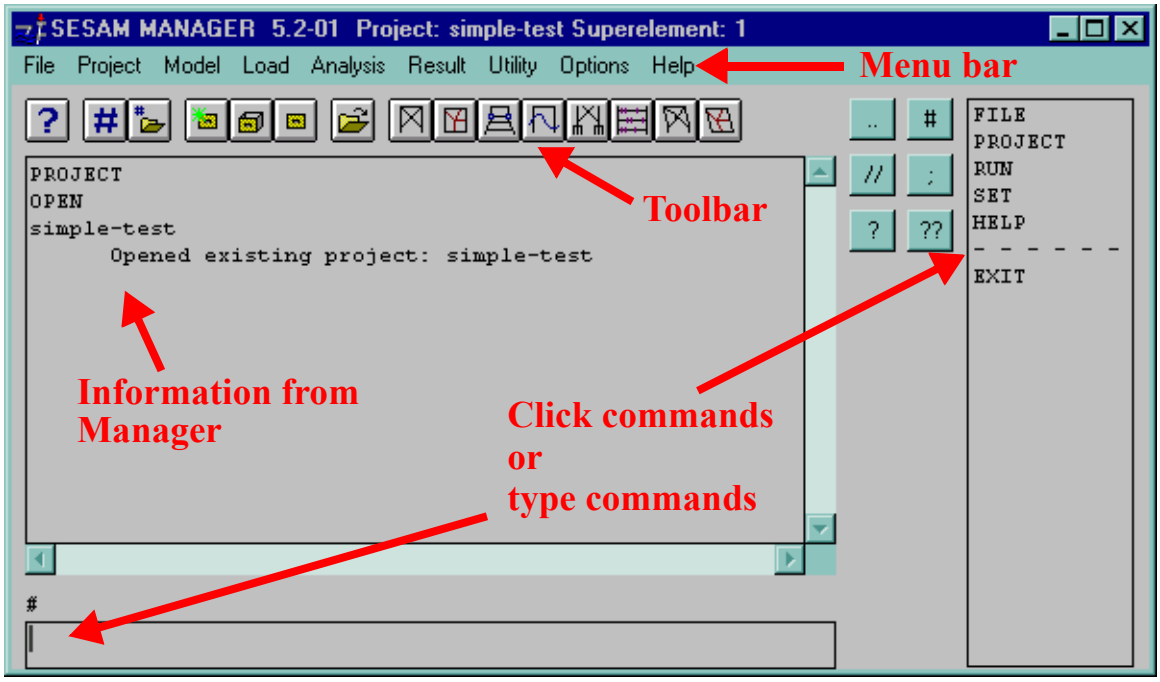

Figure 3.31 The Manager window — command input mode

Commands can be typed in the text field at the bottom and/or selected in the box at the right.

The toolbar buttons at the right have the following functions:

..	Aborts the current command
#	Inserts a new command and suspends the current command while the new is being completed
//	Selects the current default. A default is highlighted and can also be selected by Enter.
;	Selects defaults as long as such are available
?	Provides a simple help text
??	Provides more extensive help
@	Continues reading a command input file
@1	Reads the next line from a command input file

The last two are only available when a command input file is open.

For more information on how to navigate the command line interface see the Help menu.

3.13 Logging (Journalling)

When no project is open Manager menu choices and commands are logged to the file SESAM.JNL residing on the project root directory. When a project is open the logging is done to the file *project*.jnl also residing on the project root directory and where *project* is the name of the project in question.

Note: In a certain situation the logging is done to SESAM.JNL instead of to *project*.jnl even though a project is open. This is when a file named *project*.jnl existed on the project root directory when the project was opened.

Creating a command input file to Manager is most easily done by making a copy of the *project*.jnl file and editing it for your purpose.

Note: Menu choices will be logged as the corresponding line mode command preceded by two percentage signs (making it a comment). Remove these percentage signs when establishing a command input file.

Note: If you toggle command input mode, see Section 3.12, and either click commands in the right column or type them, see Figure 3.31, then the logging of commands matches the syntax of the command input file.

3.14 Batch Queue Execution

The actual execution of Manager in batch (possibly with delayed execution) is not handled by Manager itself. This is accomplished by creating a command input file to Manager and using this in a shell script that starts Manager in line mode with the command input file as input (see Section 3.11). The shell script can then be executed in batch, possibly with a time delay. How this is done depends on the batch queue system used.

This approach has been chosen because there is no standard batch queue facility on Unix and PC.

4 EXECUTION OF MANAGER

4.1 Program Execution

Manager is available on the following operating systems:

- Windows NT, 2000 and XP, hereafter referred to as PC
- OSF/Motif X Window as available on workstations or servers based on Unix

4.2 Root Directory, Project Directories and Files

SESAM.DAT is a file containing fixed and user modifiable settings such as plot options, common view point for model displays and more. Settings are modified in Manager, for example by the **Options** menu. **File | Save Defaults** saves the current settings to this file. An editor may also be used to edit the file directly. If no SESAM.DAT file exists then it will be created with a set of default settings. On PC the SESAM.DAT file is found on the project root directory (where all project directories reside). On Unix it is found (or created) on the current directory (from where Manager is started) which is also the project root directory.

On PC there is a SESAM.INI file residing on the C:\WINDOWS (or C:\WINNT) directory (the file is put there when installing SESAM). It contains reference to the location of the SESAM programs and licence (password) file (parameter SESAM_HOME) and to the project root directory (parameter SESAM_USER). See Section 3.3 for information on the project root directory.

When a new project is opened Manager will open:

- A file named *project*.mgr that contains project control information
- A file named *project*.jnl for logging Manager menu choices and commands, see Section 3.13
- A new directory (folder) named *project*

where *project* is the user given name of the project. The files will be located at the project root directory. The new directory will be a subdirectory of the project root directory. All files related to the project — i.e.

interface files and program related files — will be located at the subdirectory. (There will be no sub-subdirectories.)

4.3 Interface Files and Program related Files

Manager is based on a convention for naming the files pertaining to the individual programs. Table 4.1 shows this file naming convention. The prefix part of the file names is normally not used in Manager (**Options | Result Prefix** is an exception to this; see Section 3.10). The character # used in the table means the superelement number, the dataset number or the superelement key depending on the case.

Note: Starting programs from Utility | Run provides full flexibility in specify your own file names with prefixes.

Table 4.1 File naming convention

Program	Name	Extension	Comment
Prefem	Prefem#	.jnl	Command log (journal) file
	Prefem#	.mod	Model file (internal data base)
	Prefem	.mlg	PC only, file containing program messages
Preframe	Preframe#	.jnl	Command log (journal) file
	Preframe#	.mod	Model file (internal data base)
	Preframe	.mlg	PC only, file containing program messages
Presel	Presel#DEF	.jnl	Command startup file
	Presel#	.jnl	Command log file (journal file)
	Presel#	.mod	Model file (internal data base)
	Presel	.mlg	PC only, file containing program messages
Pretube	Pretube#	.jnl	Command log file (journal file)
	Pretube#	.mod	Model file (internal data base)
	Pretube	.mlg	PC only, file containing program messages
Bpopt	Bpopt#_IN	.jnl	Command startup file
	Bpopt	.mlg	PC only, file containing program messages
Submod	Submod	.inp	Input file
	Submod	.mod	Model file (internal data base)
	Submod	.mlg	PC only, file containing program messages

Table 4.1 File naming convention

Program	Name	Extension	Comment
Prewad	Prewad#	.jnl	Command log file (journal file)
	Prewad#	.mod	Model file (internal database)
	Prewad	.mlg	PC only, file containing program messages
Wadam	Wadam#	.fem	Analysis Control Data (input file) made by Prewad
	Wadam#	.lis	Print file (listing file)
Wajac	Wajac#	.inp	Analysis Control Data (input file)
	Wajac#	.lis	Print file (listing file)
Waveship	Waveship	.inp	Input file
Sestra	Sestra#	.inp	Analysis Control Data (input file)
	Sestra#	.lis	Print file (listing file)
	Sestra#RD	.lis	Print file from reduction in structure-pile-soil interaction
	Sestra#RT	.lis	Print file from retracking in structure-pile-soil interaction
	Sestra#	.mnt	Maintenace file
	X	.dhs	Working files (normally deleted): X = D, E, F, G, H, S, R
Gensod	Gensod#	.inp	Input file
	Gensod#	.lis	Print file (listing file)
	Gensod	.mlg	PC only, file containing program messages
Pilgen	Pilgen#	.inp	Input file
	Pilgen#	.lis	Print file (listing file)
	Pilgen	.mlg	PC only, file containing program messages

Table 4.1 File naming convention

Program	Name	Extension	Comment
Splice	Splice#	.inp	Input file
	Splice#	.lis	Print file (listing file)
	Splice	.mlg	PC only, file containing program messages
	Soil	.10	Made by Gensod
	Load	.09	Made by Pilgen
	Pile	.07	Made by Pilgen
	Mindl	.11	
	Restart	.17	
Advance		.adv	Currently not used
	T1/adv	.inp	Main input file
	T1	.crd	Input file (node and element definitions)
	T1	.lcs	Input file (loads)
	T1	.mat	Input file (materials)
	T1	.mod	Input file
		.pre	Currently not used
	T1/adv	.res	Advance results file
	Preaba	.mlg	PC only, file containing program messages
Mimosa	Mimosa	.inp	Main input file for batch execution
Prepost	Prepost#_IN	.jnl	Command startup file
	Prepost	.jnl	Command log file (journal file)
	Prepost	.mlg	PC only, message file
Postresp	Postresp#DEF	.jnl	Command startup file
	Postresp#	.jnl	Command log file (journal file)
	Postresp#	.mod	Model file (internal data base)
	Postresp	.mlg	PC only, print file (listing file)

Table 4.1 File naming convention

Program	Name	Extension	Comment
Xtract	Xtract_DEF	.jnl	Command startup file
	Xtract#	.jnl	Command log file (journal file)
	Xtract#	.mod	Model file (internal database)
Postfem	Postfem#DEF	.jnl	Command startup file
	Postfem#	.jnl	Command log file (journal file)
	Postfem#	.dbs	Data base (internal data base)
	Postfem#	.ind	Index file (pointers to internal data base)
	Postfem	.mlg	PC only, message file
Framework	Framework#DEF	.jnl	Command startup file
	Framework#	.jnl	Command log file (journal file)
	Framework#	.mod	Model file (internal data base)
	Framework	.mlg	PC only, message file
Stofat	Stofat#DEF	.jnl	Command startup file
	Stofat#	.jnl	Command log file (journal file)
	Stofat#	.mod	Model file (internal data base)
	Stofat	.mlg	PC only, message file
Cutres	Cutres#DEF	.jnl	Command startup file
	Cutres#	.jnl	Command log file (journal file)
	Cutres#	.mod	Model file (internal data base)
	Cutres	.mlg	PC only, message file
Platework	Platework#DEF	.jnl	Command startup file
	Platework#	.jnl	Command log file (journal file)
	Platework#	.mod	Model file (internal data base)
	Platework	.mlg	PC only, message file

5 COMMAND DESCRIPTION

The hierarchical structure of the commands and numerical data is in the following documented by use of tables. The tables show how the command structure diverges into multiple choices. In the example of a command structure below A is followed by either B or C. B is followed by D only while C is followed by either E or F. Legal alternatives are, therefore, A B D, A C E and A C F.

A	B	D
	C	E
		F

The characters A through F above represent COMMANDS (written in upper case) and parameters (written in lower case). Parameters are given values, typically names and numbers.

In addition to the commands documented in the following there is a command LOOP that allows giving several subcommands of a main command without leaving the main command. This is a general functionality of the graphical user interface software used in Manager. In Manager this functionality has limited value.

There are five commands in the command line interface of Manager:

FILE for manipulation of files, see page 5-2

PROJECT for manipulation of projects, see page 5-3

RUN for execution of programs, see page 5-5

SET for specifications of how programs are to be executed, see page 5-7

HELP for getting help, see page 5-13

EXIT to leave Manager

A typical command input file will start with a command creating or opening a project (unless a project has already been opened when commencing reading of the command input file), followed by some general settings (e.g. plot format and superelement setting) and finally several program runs. Each program run may be preceded by one or more SET commands specifying how to do the execution.

FILE

FILE	COPY	old-name	new-name
	DELETE	name	
	RENAME	old-name	new-name

PURPOSE:

The command manipulates files on the current directory. If a project is open, this is the directory containing the project files. If no project is open, this is the directory where the project directories are kept.

PARAMETERS:

COPY	Make a copy of a file
old-name	The current file name, wild-cards may be used
new-name	The new file name
DELETE	Delete a file
name	The file name
RENAME	Rename a file

NOTES:

In all file commands, a slash (/) inside a file path will be converted to a backslash (\) when running on a PC and vice versa on Unix. This ensures portability of command input files from one platform to the other.

PROJECT

PROJECT	CLOSE			
	COPY	old-name	new-name	ALL
				select-files
	DELETE	delete-name		
	DESCRIPTION	description		
	DIRECTORY	directory		
	NEW	name	description	
	OPEN	name		
	RENAME	old-name	new-name	
	ROOT	DIRECTORY	directory	

PURPOSE:

The command manipulates projects.

PARAMETERS:

CLOSE	Close an open project
COPY	Copies a project. This command is unavailable when a project is open.
old-name	The original project name
new-name	The new project name
ALL	Copy all files of the original project to the new project
select-files	Copy selected files only. Select several file names by enclosing them in parentheses. The options EXCLUDE, INCLUDE, ONLY and GROUP has little relevance here.
DELETE	Delete a project. This command is unavailable when a project is open.
delete-name	Name of the project to delete
DESCRIPTION	Change the description for the currently open project
description	The project description
DIRECTORY	Specify the directory where the project are located
directory	The directory name

NEW	Create a new project
name	Name of the project
OPEN	Open an existing project
RENAME	Rename a project
ROOT	PROJECT ROOT DIRECTORY has the same function as PROJECT DIRECTORY.

RUN

RUN	program	input-file		
	ADVANCE	input-file		
		BUCKLING	parameters	
		DYNAMIC	parameters	
		FEM-TRANSLATION		
		HEAT-TRANSFER		
		NATURAL-FREQUENCY	parameters	
		POST-PROCESSING	input-file	
		RESTART	input-file	
		STATIC	parameters	
	PILE-SOIL-ANALYSIS	GENSOD	input-file	
		PILGEN	input-file	
		SPLICE	input-file	
		STRUCTURE-REDUCTION	GENERATE-INPUT	
			input-file	
		STRUCTURE-RETRACKING	GENERATE-INPUT	
			input-file	
	SESTRA	input-file		
		EIGENVALUE	HOUSEHOLDER	nmodes
			LANCZOS	
			SUBSPACE-ITERATION	
		STATIC		
	WADAM	dataset		

PURPOSE:

The command runs (starts execution of) a program.

PARAMETERS:

program — Select a program to run (Preframe, Presel, Wajac, etc.).

For certain programs, e.g. Advance and Sestra, more specific information than a mere input file should be given.

input-file	Input file to the program (*.jnl file for Preframe and Presel, *.inp for Wajac, etc.)
ADVANCE	Run Advance as specified by subsequent data.
BUCKLING	Run a buckling analysis in Advance; see Section 3.6.2 for data to give.
DYNAMIC	Run a dynamic analysis in Advance; see Section 3.6.2 for data to give.
FEM-TRANSLATION	Translate T1.FEM to Advance linear analysis input, the result is T1.inp.
HEAT-TRANSFER	Run a heat transfer analysis in Advance.
NATURAL-FREQUENCY	Run a natural frequency (free vibration) analysis in Advance; see Section 3.6.2 for data to give.
POST-PROCESSING	Postprocess Advance results using Advance_Post.
RESTART	Restart an Advance analysis.
STATIC	Run a static analysis in Advance; see Section 3.6.2 for data to give.
PILE-SOIL-ANALYSIS	Run a structure-pile-soil interaction analysis involving Sestra, Pilgen, Gensod and Splice.
GENSOD	Run Gensod with given input file.
PILGEN	Run Pilgen with given input file.
SPLICE	Run Splice with given input file.
STRUCTURE-REDUCTION	Run a structure reduction in Sestra, either with a generated input file or a prepared one.
STRUCTURE-RETRACKING	Run a structure retracking in Sestra, either with a generated input file or a prepared one.
SESTRA	Run Sestra as specified by subsequent data.
EIGENVALUE	Run an eigenvalue (free vibration) analysis in Sestra using either Householder, Lanczos or Subspace Iteration.
STATIC	Run a static analysis in Sestra.
WADAM	Run a Wadam analysis using dataset 1 (Wadam1.FEM).

NOTES:

Some programs require certain settings (SET command) prior to their execution (RUN command).

SET

The SET commands falls into two categories: the general settings and the program specific settings.

General settings:

SET	FILE-HANDLING	AUTOMATIC			
		USER-DEFINED			
	PLOT	COLOUR	ON		
			OFF		
			VOID		
		FORMAT	format		
		ORIENTATION	LANDSCAPE		
			PORTRAIT		
			VOID		
		PAGE-SIZE	page-size		
	PROGRAM-PATH	path			
	PROGRAM-STARTUP	EYE-DIRECTION	eye-x	eye-y	eye-z
			PROGRAM-DEFAULT		
		HEADER	ON		
			OFF		
	RESULT-FORMAT	SIF-FORMATTED			
		SIN-DIRECT-ACCESS			
		SIU-UNFORMATTED			
	RESULT-PREFIX	prefix			
	STRUCTURE-TYPE	FRAME			
		GENERAL			
	SUPERELEMENT	ANALYSIS-MODEL	DIRECT-ANALYSIS		
			SUPERELEMENT-ANALYSIS	topno	
		FIRST-LEVEL-SUPERELEMENT	supno		
	TEMPORARY-FILES	directory			

Program specific settings:

SET	ABAQUS-ANALYSIS	... (currently irrelevant)		
	ADVANCE	... (see next page)		
	COMMAND-LINE-ARGUMENT	program	arguments	
	DATABASE	NAME	program	name
		STATUS	program	NEW
				OLD
	FRAMEWORK	SUPERELEMENT-KEY	key-number	
	PILE-SOIL-ANALYSIS	SPLIT-WORKING-FILES	ON	
			OFF	
	POSTFEM	RESULT	ALL-OFF	
			ALL-ON	
			ELEMENT	specification
			GAUSSIAN	
			NODAL	
			RESET	
	PREFEM	OPTIMISE-SUPERELEMENT	ON	
			OFF	
	PRETUBE	LENGTH-UNIT	unit	
		WELD-TYPE	SHELL-ELEMENTS	
			SOLID-ELEMENTS	
	PREWAD	DATASET	dataset-number	
	PROGRAM-EXECUTABLE	program	OFFICIAL	
			USER-DEFINED	file
	SESTRA	SOLVER	MULTIFRONT	
			SUPERMATRIX	
		SPLIT-WORKING-FILES	ON	
			OFF	

SET ADVANCE options:

SET	ADVANCE	ANALYSIS	GLOBAL-MODEL-FILE	file	
			MAIN-PROGRAM-BUFFER	size	
			MAIN-PROGRAM-MEMORY	size	
			NUMBER-OF-CPUS	ncpu	
			POSTPROCESSOR-BUFFER	size	
			POSTPROCESSOR-MEMORY	size	
			PREPROCESSOR-BUFFER	size	
			PREPROCESSOR-MEMORY	size	
			RESET-TO-DEFAULT-VALUES		
			SCRATCH-DIRECTORY	directory	
			SOLVER	SPARSE	
				WAVEFRONT	
		DATACHECK	OFF		
			ONLY		
			ON		
		OUTPUT	CONTACT-OUTPUT	...	ON
			ELEMENT-OUTPUT		
			ENERGY-OUTPUT		OFF
			NODAL-OUTPUT		

PURPOSE:

The command sets conditions for how to execute the programs.

The sub-command falls into two categories (split into separate tables above):

- General settings
- Program specific settings

PARAMETERS:

ABAQUS-ANALYSIS	This option is irrelevant as Advance has replaced Abaqus.
ADVANCE	Set parameters controlling a subsequent Advance analysis:
ANALYSIS	Set parameters controlling the execution of Advance. See Section 3.6.2 for more information on these options and their default values.
DATACHECK	Choose between only datacheck (ONLY), datacheck plus analysis (ON) and only analysis (OFF) in Advance.

OUTPUT

Choose type of Advance output.

COMMAND-LINE-ARGUMENT

For a specific program set command line arguments. This is a text string with one or more arguments separated by one or more blanks. Each argument starts with a slash. For valid arguments see the individual program user manuals. If the string includes blanks enclose it in quotes. Example:

'/STATUS=NEW /COMMAND-FILE=INPUT.JNL'

DATABASE

For a specific program set the name and status (new or old) of the program database.

This command in only effective when a program is run from the line mode command interface and when user defined file handling is in effect (SET FILE-HANDLING USER-DEFINED). It is not effective when starting a program from its dialog box.

The Utility | Run dialog performs this function in the graphical user interface. The database name will then appear as default in the Utility | Run dialog box.

FILE-HANDLING

Choose between AUTOMATIC and USER-DEFINED file handling. USER-DEFINED file handling implies that the user must control all file naming and file conversion. The commands SET DATABASE and SET RESULT-PREFIX can be used to control database names. AUTOMATIC file handling involves that Manager will name all relevant files and automatically perform the required file conversions.

Note: This command in only effective when a program is run from the line mode command interface. It is not effective when e.g. starting Prefem from the General Modelling dialog box. The Utility | Run dialog box can be used to take control of file names from the graphical user interface.

FRAMEWORK SUPERELEMENT-KEY

Set superelement key number for Framework. This involves selecting first level superelement to process. (Prepost may be used to print an overview of all superelements to find the appropriate superelement key number.)

PILE-SOIL-ANALYSIS

The SPLIT-WORKING-FILES option allows splitting the working files for the various superelements. This option is for Sestra used in connection with a structure-pile-soil interaction analysis (as opposed to SET SESTRA SPLIT-WORKING-FILES which is for other use of Sestra).

PLOT

Specify plot format, orientation, etc.

Note: This information is intended to be passed on through command line arguments to programs

started by Manager. This is, however, subject to provisions for receiving such information by the various programs.

COLOUR	Switch colour ON or OFF. VOID involves no setting.
FORMAT	Select plot format. Current alternatives are: CGM-BINARY, HPGL-2, HPGL-7550, POSTSCRIPT, SESAM-NEUTRAL and WINDOWS-PRINTER. See below for information on these formats. VOID involves no setting. A CGM-BINARY (Computer Graphics Metafile) file may be imported in Word and PowerPoint. In these applications the drawing may even be converted to a Microsoft Office drawing thereby allowing it to be modified. HPGL-2 and HPGL-7550 is for the HP Unix platform. POSTSCRIPT is an ASCII formatted file that may be imported in Word and other word processors. SESAM-NEUTRAL is a plot format that requires the auxiliary program pltcnv to be reproduced on paper. This is less relevant than the CGM-BINARY and POSTSCRIPT formats. WINDOWS-PRINTER involves the plot to be sent directly to an on-line printer. No plot file will be produced.
ORIENTATION	Select between LANDSCAPE and PORTRAIT orientation of the plot. VOID involves no setting.
PAGE-SIZE	Select plot size. Alternatives are the European paper sizes A1, A2, A3, A4 and A5. VOID involves no setting.
POSTFEM RESULT	Specify result types to be stored (by Prepost) on the Postfem database. The ALL-OFF option switches all result types off and should be followed by switching on selected result types. ALL-ON enables any result type to be postprocessed in Postfem but it will also demand maximum disk space. The ELEMENT, GAUSSIAN and NODAL options allows switching on selected result types like B-LOAD, D-STRESS, DISPLACEMENT, G-FORCE etc. The RESET option is used to reset to the default selection.
PREFEM OPTIMISE-SUPERELEMENT	Set optimising of node numbering (for the benefit of the SUPERELEMENT solver for static analysis and all types of dynamic analysis). This involves running Bpopt on the Input Interface File (T-file) subsequent to writing the superelement in Prefem.
PRETUBE	Choose length unit (MM, CM, M or INCH) as well as type of elements to use for the weld.

PREWAD DATASET	Select dataset number.
PROGRAM-EXECUTABLE	For a specific program select the OFFICIAL version or a file being the executable file (*.exe) of a special version.
PROGRAM-PATH	Defines a path consisting of four possible locations of the executable files of the SESAM programs. This is of little relevance to the user.
PROGRAM-STARTUP	Set view direction (EYE-DIRECTION) to program default (which may vary from program to program) or give a common view direction. Also switch off program header in print files.
RESULT-FORMAT	Set format of the Results Interface File produced by Sestra to SIF-FORMATTED, SIN-DIRECT-ACCESS or SIU-UNFORMATTED.
RESULT-PREFIX	Set a prefix for all result files thereby allowing more than a single analysis result file to be kept within the same project. This command is only available when a project is open. This command is relevant for linear analysis (Sestra).
SESTRA	Choose between MULTIFRONT and SUPERMATRIX type of equation SOLVER. The SPLIT-WORKING-FILES option allows splitting the working files for the various superelements.
STRUCTURE-TYPE	The alternatives FRAME and GENERAL only affects the icon toolbar of the graphical user interface.
SUPERELEMENT	Choose between DIRECT-ANALYSIS and SUPERELEMENT-ANALYSIS type of ANALYSIS-MODEL. In the latter case also give the superelement number of the top level superelement. For DIRECT-ANALYSIS (i.e. only a single superelement constituting the complete model) use the FIRST-LEVEL-SUPERELEMENT option to give the number of the single superelement (unless accepting the default value of 1). This command is only available when a project is open.
TEMPORARY-FILES	Set directory of temporary files for some programs, e.g. Sestra working files. This option is only for PC.

HELP

HELP	ABOUT-HELP	
	ABOUT-SESAM	
	COMMAND-INPUT-FILE	
	LINE-MODE	...
	PROGRAMMING-MODE	...
	STATUS-LIST	
	SUPPORT	

PURPOSE:

The command provides help.

PARAMETERS:

ABOUT-HELP	About the HELP command
ABOUT-SESAM	General information on SESAM
COMMAND-INPUT-FILE	About using command input file
LINE-MODE	About using line mode commands, of little relevance to Manager
PROGRAMMING-MODE	About using programming mode, of little relevance to Manager
STATUS-LIST	Starts the Status program, same as clicking  .
SUPPORT	About how to get support.

EXIT

PURPOSE:

The command closes Manager.

APPENDIX A TUTORIAL EXAMPLES

A few examples of command input files for Manager are enclosed herein. Note that even though the examples refer to certain types of analysis their purpose is to show how several SESAM programs may be run in sequence by use of a command input file to Manager.

A 1 Superelement Analysis

This example is a superelement analysis with two first level superelements. Superelement 1 is created by Preframe while superelement 5 is created by Prefem. Presel is used to assemble the two superelements into the top level superelement 10. A static analysis in Sestra is performed followed by a postprocessing session in Xtract. The example includes the following program executions:

- Preframe creates superelement 1 (T1.fem).
- Prefem creates superelement 5 (T5.fem).
- Presel creates top level superelement 10 (T10.fem).
- Sestra performs a static analysis.
- Xtract postprocesses the results.

The example refers to the input files (not includede here):

- Preframe1_in.jnl — Preframe input for creating superelement 1
- Prefem5_in.jnl — Prefem input for creating superelement 5
- Presel10_in.jnl — Presel input for creating the top level superelement 10
- Xtract10_in.jnl — Xtract postprocessing input

The Manager command input file:

```
%
```

```
% --- Set the analysis to be a superelement analysis with top = 10
SET SUPERELEMENT ANALYSIS-MODEL SUPERELEMENT-ANALYSIS 10
%
% --- Set superelement number and read the Preframe input
SET SUPERELEMENT FIRST-LEVEL-SUPERELEMENT 1
RUN PREFRAME Preframe1_in.JNL
%
% --- Set superelement number and read the Prefem input
SET SUPERELEMENT FIRST-LEVEL-SUPERELEMENT 5
RUN PREFEM Prefem5_in1.jnl
%
% --- Run Presel
RUN PRESEL Presel10_in.jnl
%
% --- Run static analysis in Sestra using the Multifront equation solver
SET SESTRA SOLVER MULTIFRONT
RUN SESTRA STATIC
%
% --- Postprocess in Xtract
RUN XTRACT Xtract10_in.jnl
```

Note: The input files to Preframe, Prefem and Presel do not need the WRITE command at the end as the T-files will automatically be written.

A 2 Superelement Analysis — Splitting Input to Preframe and Prefem

This example is the same as the one in Section A 1: A superelement analysis with two first level superelements. Superelement 1 is created by Preframe while superelement 5 is created by Prefem. Presel is used to assemble the two superelements into the top level superelement 10. A static analysis in Sestra is performed followed by a postprocessing session in Xtract. The example includes the following program executions:

- Preframe creates superelement 1 (T1.fem).
- Prefem creates superelement 5 (T5.fem).
- Presel creates top level superelement 10 (T10.fem).
- Sestra performs a static analysis.
- Xtract postprocesses the results.

Note that rather than having the input to Preframe in a single file it is (merely for some practical reason) split into three files: Preframe1_in1.jnl, Preframe1_in2.jnl and Preframe1_in3.jnl. Similarly, the Prefem input is split into two files: Prefem5_in1.jnl and Prefem5_in2.jnl. The example therefore refers to the input files (not includede here):

- Preframe1_in1.jnl, Preframe1_in2.jnl, Preframe1_in3.jnl — Preframe input for creating superelement 1
- Prefem5_in1.jnl, Prefem5_in2.jnl — Prefem input for creating superelement 5
- Presel10_in.jnl — Presel input for creating the top level superelement 10

- Xtract10_in.jnl — Xtract postprocessing input

The Manager command input file:

```
%
% --- First make sure file handling is automatic (default setting)
SET FILE-HANDLING AUTOMATIC
%
% --- Set the analysis to be a superelement analysis with top = 10
SET SUPERELEMENT ANALYSIS-MODEL SUPERELEMENT-ANALYSIS 10
%
% --- Set superelement number and read first part of Preframe input
SET SUPERELEMENT FIRST-LEVEL-SUPERELEMENT 1
RUN PREFRAME Preframe1_in1.JNL
%
% --- Add input to existing Preframe model (database) by:
%     * Setting file handling to user-defined
%     * Specifying model file (database) name
%     * Set model file (database) to old (do not overwrite)
SET FILE-HANDLING USER-DEFINED
SET DATABASE NAME PREFRAME PREFRAME1
SET DATABASE STATUS PREFRAME OLD
RUN PREFRAME Preframe1_in2.JNL
%
% --- Add last input to exisiting Preframe model
%     NOTE: This input file should have the WRITE command at the end.
RUN PREFRAME Preframe1_in3.JNL
%
% --- Set file handling back to automatic
SET FILE-HANDLING AUTOMATIC
%
% --- Set superelement number and read first part of Prefem input
SET SUPERELEMENT FIRST-LEVEL-SUPERELEMENT 5
RUN PREFEM Prefem5_in1.jnl
%
% --- Add input to existing Prefem model (database) in same way as for Preframe:
SET FILE-HANDLING USER-DEFINED
SET DATABASE NAME PREFEM PREFEM5
SET DATABASE STATUS PREFEM OLD
RUN PREFEM Prefem5_in2.jnl
%
% --- Set file handling back to automatic and run Presel
SET FILE-HANDLING AUTOMATIC
RUN PRESEL Presel10_in.jnl
%
% --- Run static analysis in Sestra using the Multifront equation solver
SET SESTRA SOLVER MULTIFRONT
RUN SESTRA STATIC
%
% --- Postprocess in Xtract
RUN XTRACT Xtract10_in.jnl
```

Note: To enable splitting the input to Preframe and Prefem (and similar programs) into two or more files you must switch to user-defined file handling. This in turn requires the last input file to have the WRITE command at the end or else the T-file will not be written.

A 3 Preframe, Wajac, Sestra + Sestra, Prepost and Framework

This example is taken from an earthquake analysis of a single superelement model and includes running the programs Preframe, Wajac, Sestra, Prepost and Framework. Sestra is run twice, first a static analysis and then an eigenvalue analysis. The example includes the following program executions:

- Preframe creates a model (T-file).
- Wajac computes added mass (L-file).
- Sestra performs a static analysis.
- Sestra performs an eigenvalue analysis.
- Prepost merges the static and eigenvalue results.
- Framework performs an earthquake analysis.

The example refers to the input files (not includede here):

- Preframe1_in.jnl — Preframe input for creating the model
- AddmassWajac.inp — Wajac input for computing added mass
- EigenSestra.inp — Sestra input for running the eigenvalue analysis
- Prepost_in.jnl — Prepost input for merging the static and eigenvalue analysis result files
- Framework_in.jnl — Framework input for performing the earthquake analysis

The Manager command input file:

```
%
% Manager command input file for earthquake analysis
%
% --- Set the analysis to be a direct analysis
 SET SUPERELEMENT ANALYSIS-MODEL DIRECT-ANALYSIS
%
% --- Run Preframe to establish the model
 SET SUPERELEMENT FIRST-LEVEL-SUPERELEMENT 1
 RUN PREFRAME Preframe1_in.jnl
%
% --- Run Wajac to compute added mass
 RUN WAJAC AddmassWajac.inp
%
% --- Run the static analysis in Sestra with input created by Manager
%     The format of the results file will be SIF and it is given the prefix STATIC
 SET RESULT-FORMAT SIF-FORMATTED
```

```
 SET RESULT-PREFIX STATIC
 RUN SESTRA STATIC
%
% --- Run the eigenvalue analysis in Sestra with a prepared input file
%     The results file is given the prefix EIGEN
 SET RESULT-PREFIX EIGEN
 RUN SESTRA EigenSestra.inp
%
% --- Run Prepost to merge the results files
 SET RESULT-PREFIX EARTHQ
 RUN PREPOST Prepost_in.jnl
%
% --- Run Framework to perform the earthquake analysis
 RUN FRAMEWORK Framework_in.jnl
%
% Finished
 EXIT
```