CHAPTER 4 EXPERIMENTAL PROGRAM

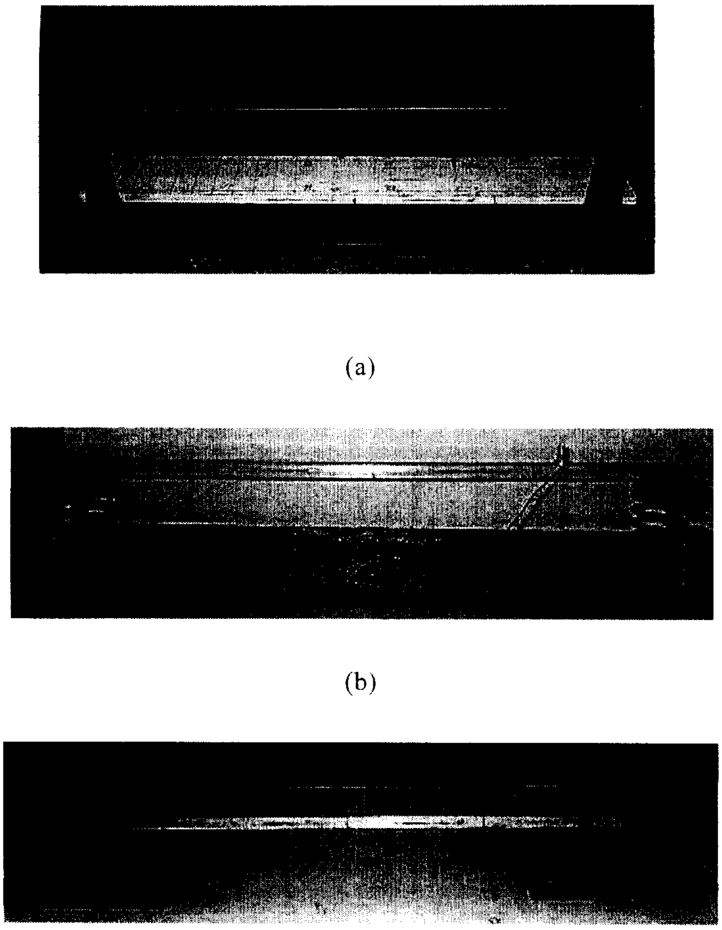
4.1 INTRODUCTION

The main aim of this investigation is to appraise the effects of boundary conditions on damage identification in a uniform steel beam using modal parameters. Uniform steel beams were tested on different boundary conditions namely free-free, spring and fixed roller support. Subsequently, a saw cut and dual saw cuts were introduced in the steel beams. The dynamic properties of the intact and damaged steel beams were acquired by modal testing. The raw data were then processed and analyzed using modal analysis software packages. Finally, the beams were modeled using finite element in order to obtain the analytical modal parameters and to perform model updating.

4.2 TEST BEAM

Two uniform straight steel beams with the rectangular cross section measuring 18mm in depth and 148mm with were prepared as the test specimens. The length of the beams was 1080mm. The first beam was used for the single crack case while the second beam for the dual crack case.

The beams were tested on free-free, spring and fixed roller support boundary conditions with the ends free to rotate and these are shown in Figure 4.1. For the free-free boundary condition, the beams were suspended at the ends by using rubber tubings. The spring supports were provided by two springs at each end, having a stiffness 10kN/m per spring, while the fixed roller supported beams rested on fixed steel rollers. The undamaged or baseline modal data were acquired using transfer function technique for all the boundary conditions.



(a)

(b)

(c)

Figure 4.1 Beam support condition, (a) free-free, (b) spring and (c) fixed roller support

Subsequently, a slot was cut at the soffit of the beam to simulate a crack. For the single crack case, a 1mm deep saw cut was introduced at the quarter span point to simulate an open crack. For the case of the dual crack, two 1.5mm deep saw cuts were introduced, one at the quarter span point and another at mid span point to simulate an open crack. The depth of the cut for both cases was increased in stages to a maximum of 12mm so as to provide different levels of severity of damage in the beams.

4.3 IMPACT TESTING

Impact testing is a very popular technique due to the ease of setup, short measurement acquisition time and it is relatively low cost. However this technique is not suitable for testing nonlinear structures and the results must be interpreted carefully. Generally, a selection of hammer tip, window functions and pretrigger delay are essential to acquire the quality frequency response functions and also to avoid errors and leakage. The possible weaknesses of impact testing are double impacts and the high potential for both overloading and underloading of the Analog to Digital Converter (ADC). However, if care is exercised in acquiring data all of these can be overcome.

In the impact testing, transfer function method (TFM) was adopted to obtain the modal parameters namely natural frequencies, mode shapes and damping ratios. In order to acquirer modal data of good quality and higher accuracy, some precautionary measures were taken during impact testing.

- The hammer tip was selected carefully to excite the desired frequency range.
- Double impact and overloading were avoided in order to obtain good quality data.
- The accelerometer was rigidly mounted on to the smooth surface of the test beam.
- An allowance of 5 seconds was given prior to impacting the next measurement location for the spectrum to reach a steady state.
- The output quality of the displayed FRF spectrum at any measurement point was always monitored closely to secure good quality data.
- Once any erroneous measurements were observed, the measurement was repeated immediately without delay.

4.3.1 TRANSFER FUNCTION METHOD (TFM)

A simple set up of impact testing using transfer function method is shown in Figure 4.2. The transfer function method is based on the use of digital signal processing techniques and the fast fourier transform (FFT) algorithm to measure transfer function between different points on the test structure. The excitation force was provided using a modally tuned hammer by striking predetermined points on a 27 × 2 rectangular grid on the top surface of the beam. The hammer sensitivity was 2.3 mV/N. The response signal was acquired using a single general purpose low impedance accelerometer with a sensitivity of 100mV/g. The position of the accelerometer was fixed at node 12 throughout the test. The measurement points located on the top surface of the beam are shown in Figure 4.3. The distance between each point was 40mm and 100mm in the longitudinal and transverse directions of the beam, respectively. Points 1, 27, 28 and 54 were located at the supports of the beam

The frequency response function (FRF) spectrum within a 2kHz frequency span was obtained using the Scientific Atlanta dynamic analyzer SD 390 as shown in Figure 4.4. The first eleven modes namely six flexural modes and five torsional modes, were acquired within the bandwidth. The medium hammer tip, Model No. 084A32, was selected through trial and error, in order to sufficiently acquire the eleven modes. The pre-trigger delay was set to –20 samples to satisfy the trigger. The anti-aliasing filter and the internal clock of the analyzer which automatically generates a sampling rate of 2.56 times the frequency span, eliminate the aliasing phenomenon. The measurements were made using a block size of 4096 samples and 1600 lines, thus giving a resolution of 1.25Hz per spectral line and providing longer time response to naturally decay to zero. The "rectangular" weighting was applied in performing the fast Fourier transform (FFT) which can lower the effect of "flaring" or "leakage" at each component frequency. By employing a linear method of averaging with 5 ensembles, this will reduce the interference of noise. Hence, it can help to improve the quality of the acquired. Finally, the response data obtained from the analyzer were saved in FRF SMS-Star file format and post-processing was performed on raw FRF data by using modal analysis software package, which is detailed in Section 4.4.

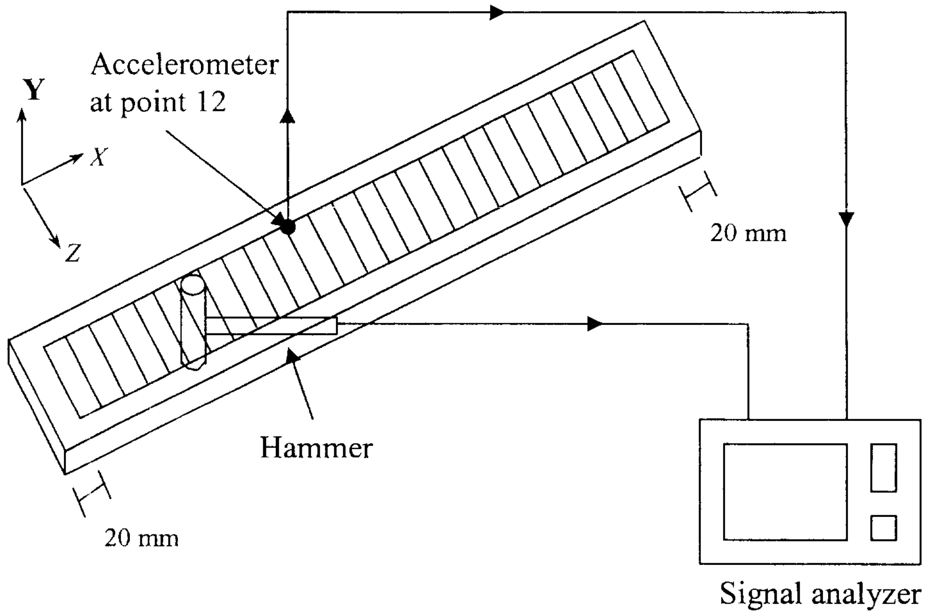


Figure 4.2 Experimental set up of impact testing using transfer function method

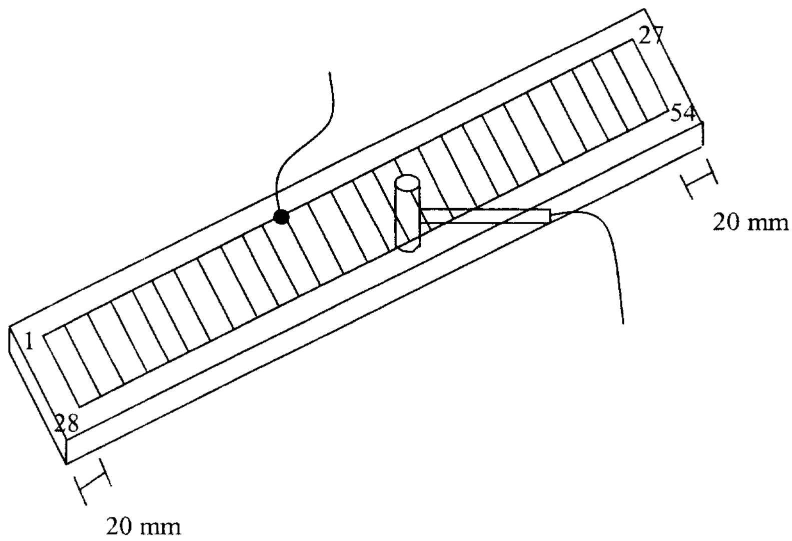


Figure 4.3 Measurement locations

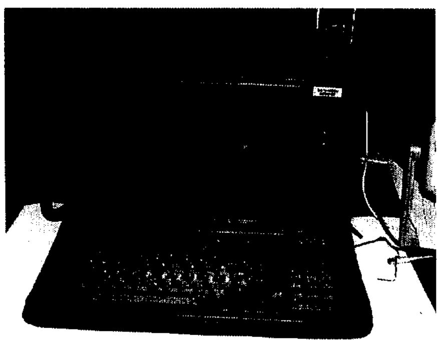


Figure 4.4 Dual-channel dynamic analyzer

4.3.2 POST-PROCESSING

The raw FRF data acquired by the dynamic analyzer was post-processed using modal analysis software package, namely SMS-STAR and ICATS. The SMS-STAR system is a suite of software for testing and analyzing the dynamics of mechanical structures developed by the Spectral Dynamic Incorporated, USA. It is an acronym for structural testing, analysis and reporting, which describes the general capabilities of the system. While ICATS produces similar functions as STAR with some extra features such as linearity check and eigen-parameter identification techniques, which was developed by Imperial College, UK. ICATS suite of software comprises of MODENT, MODESH, MODACQ and MESHGEN.

The STAR system was mainly used here for converting the raw .FRF measurement into an universal file format because the raw data were in STAR format and could not be directly post-processed in ICATS. The universal file format with the file extension ".ASC" was converted to ICATS-format using a file converter program in ICATS namely the FRF converter.

The post-processing work was mainly performed using ICATS, which is basically a modal analysis suite of computer program for the analysis and extraction of modal properties from FRF data. The converted FRF data was curve-fitted using MODENT program to capture the modal parameter. The mutli FRF analysis method was used to obtain an average value of the natural frequencies and damping ratios for the corresponding mode shapes in the 2kHz frequency span. The data were then stored in a .EIG file. In order to validate the .EIG file, the reciprocal vector method was carried out to compare the .EIG file with the corresponding .FRF measurements stored in a .CRD file. The mesh of measurement points was generated by the MESHGEN program which was required for animation of the mode shape.

Two .EIG files obtained from the MODENT program were compressed into a single .CMP file by using MAPGEN program for correlation purposes. The .CMP file was utilized to correlate the two sets of data using techniques in the MODESH program, namely modal assurance criterion (MAC), natural frequency comparison, mode shape comparison and coordinate modal assurance criterion (COMAC). Figure 4.5 shows the flow chart of post processing for raw FRF data using SMS-STAR and ICATS.

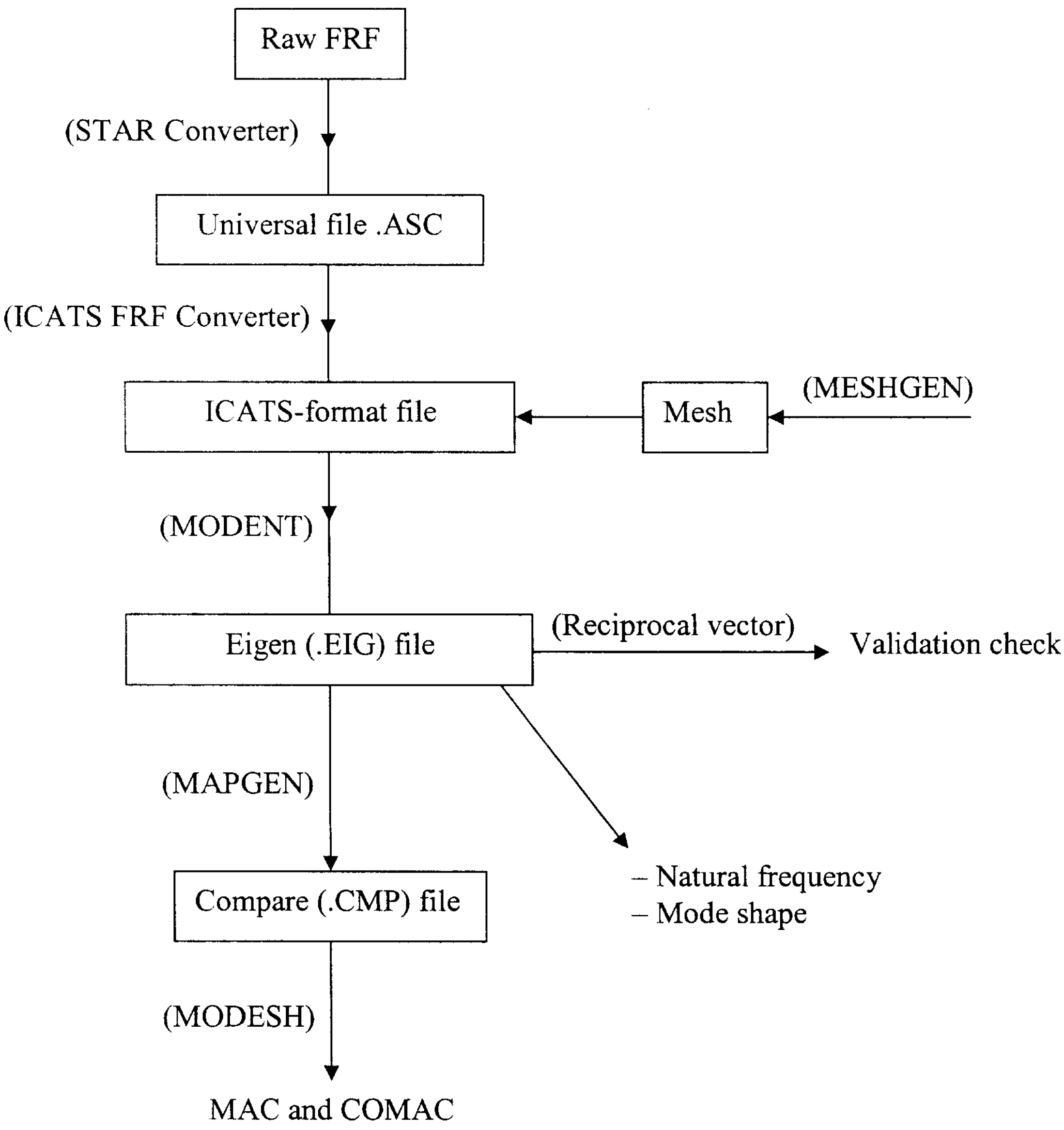


Figure 4.5 Flow chart of post processing for raw FRF data

4.3.3 VALIDATION CHECK

Validation check is implemented to ensure the quality of the modal parameters, which are extracted by the curve fitting technique. Reciprocal vector is commonly used to verify the quality of modal analysis data by comparing the raw FRF with the synthesized FRF obtained via modal summation [23]. In ICATS, reciprocal vector provides a quality assurance procedure for the modal parameters stored in a .EIG file. The orthogonality of the modal data with respect to the corresponding raw FRF data stored in a .CRD file is assessed. Figure 4.6 shows the output of reciprocal modal vector matrix where the unit matrix indicates a good quality of modal parameters measurements.

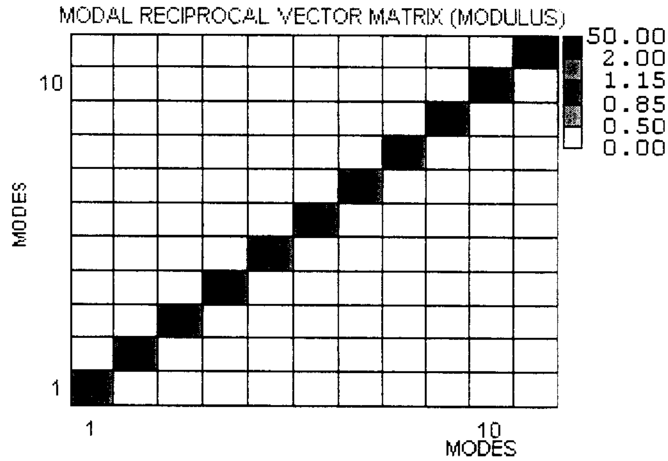


Figure 4.6 Reciprocal modal vector matrix

4.4 FINITE ELEMENT (FE) MODELLING

Analytical models of free-free and simply support conditions were generated by the finite element (FE) software package, namely DIANA, for correlation with the experimental model [25]. DIANA is a versatile finite element analysis program developed by TNO DIANA B. V. for a wide range of civil engineering applications. The software has extensive material, element and procedure libraries based on advanced database techniques, linear and non-linear analysis capabilities and full 2D and 3D modelling. There are three basic user-interfaces namely a batch interface, a graphical user interface and a programming environment for generating a module in the DIANA finite element analysis. The graphical user interface is detailed in this section.

4.4.1 GRAPHICAL USER INTERFACE

The interactive graphical user interface, called iDIANA, is a fully integrated pre- and post-processing environment to DIANA. A new model is built in the pre-processing environment by using the FEMGEN command. The complete new model is saved as a DIANA batch format and analysed in the analysis environment called DIANA_w. In order to assess the results of the FE analysis, the model is viewed in the post-processing environment by using the FEMVIEW command.

4.4.1.1 PRE-PROCESSING

The process to build a new model typically involves such tasks as definition of geometry, meshing, material properties, physical properties and boundary conditions. A rectangular steel beam FE model was generated by using the FEMGEN command. The beam model was specially named and analysed using the Struct_3d analysis which executes a three dimensional analysis. The units of length, mass, force, time and temperature were specified in System International (SI) units for the beam model data.

Firstly, the points, lines and surface command in GEOMERTY command were used to define the geometry of the beam. Points are typically defined by their coordinates, lines by their end points and surfaces by their bounding lines. The defined geometry was meshed using the MESHING command. The element Q12PL which is a four node quadrilateral isoparametric plate bending element was used to model the beam. The beam was modelled as a size comprising of 54 meshed Q12PL elements, as shown in Figure 4.7. The boundary conditions of free-free and fixed roller support were defined by using the PROPERTY BOUNDARY CONSTRAINT command. No constraint was applied in the free-free beam model. For the fixed roller support case, an elastic support in the z direction only with high stiffness boundary condition was defined. The material was defined as isotropic with Young's modulus $E = 2 \times 10^{11}$ N/m, Poisson ratio $\nu = 0.3$ and mass density $\rho = 8000$ kg/m^3. The physical property of the plate element was defined as a uniform thickness $t = 0.018$m, which was compatible with the plate bending element. Both material and physical properties were defined using Property Manager option in the iDIANA working window. Subsequently, a defect element with a reduced flexural stiffness *EI* was introduced at the 13th element to simulate a crack as in the experimental beam. The severity of defect was increased with subsequent reduction in the value of flexural stiffness *EI* at the 13th element.

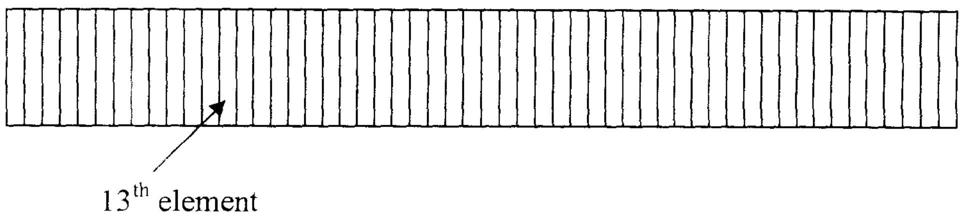


Figure 4.7 The mesh of the beam model

4.4.1.2 PERFORMING THE ANALYSIS

The completed beam model was written as a input file or a data file in DIANA batch format with a specific file name using the UTILITY WRITE DIANA command. INDEX command was used to enter the beam model into the Model Index environment for the analysis of the input file. The Analysis Selection window appeared following the selection of the ANALYSIS command in Model Index environment. Figure 4.8 shows the Analysis Selection window which is used to indicate the analysis type and to read the input file. The eigenvalue analysis was selected and read in order to check any errors or warnings in the input file.

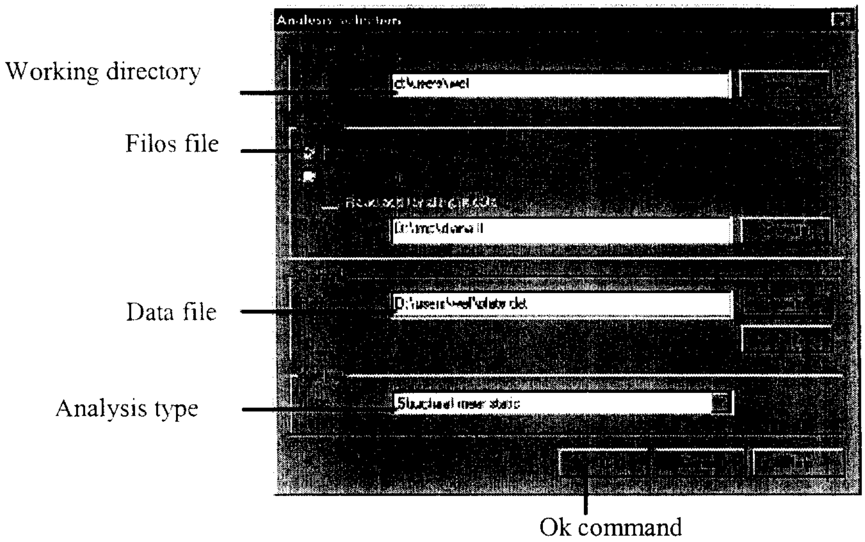


Figure 4.8 Analysis selection window

After the termination of the reading process, if there are no errors detected, the DIANA_w analysis was then performed with the eigenvalue analysis option as shown in Figure 4.9. The type of eigenvalue problem, execution methods and outputs were defined during the eigenvalue analysis. Since experimental beam was subjected to free vibration, the free vibration eigenvalue problem with linear elastic stiffness matrix was executed using Subspace Iteration method. Eigenvalues and eigenvectors were the eigenvalue analysis outputs which were assessed in the post-processing environment.

4.4.1.3 POST-PROCESSING

The process to assess the analysis results was started by pressing FEMVIEW command to enter the post-processing environment of iDIANA. The eigenvalues (natural frequencies) were obtained from the output file. The eigenvectors that is the mode shapes were extracted as a .LST file format using the UTILITY TABULATE RESULTS command. The spatial data, eigenvalues and eigenvector were correlated with the modal data. natural frequencies and mode shapes using ICATS software package.

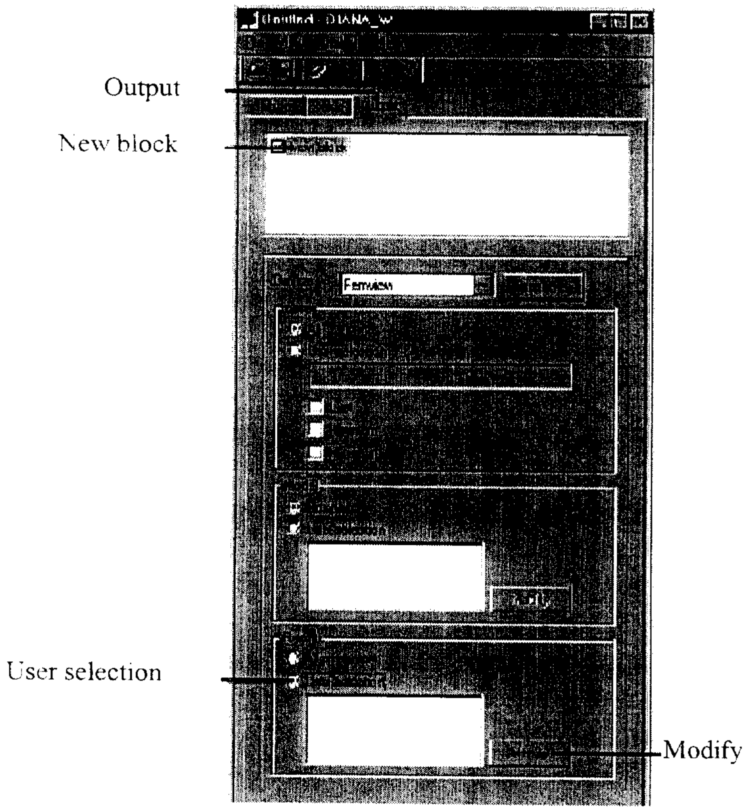


Figure 4.9 DIANA_w analysis window