CHAPTER 5

ANALYTICAL METHOD

5.1 INTRODUCTION

Experimental results obtained are verified for their credibility by performing analysis on the structure by its discretization into elements. Here numerical analysis using ANSYS software package was carried out. The ANSYS is finite element software used for performing analysis of engineering problems. The ANSYS package can handle various finite element analyses of simple, linear and non-linear structures. It is also capable of performing analysis ranging from static to complex non-linear transient dynamic analysis. It is a powerful technique for the numerical solution of a variety of problems encountered in engineering.

The ANSYS software helps engineers to conduct the following tasks:

- From the CAD models of structure, products, components and systems they can be transferred to ANSYS to develop basic computer models.

- On this computer model, it is usual to apply the operating loads and other design performance conditions.

- Physical quantities such as stress levels, temperature distribution or electromagnetic fields, etc., can be studied using ANSYS package.
• It can be used to optimize the design at the early stage of the development process so that the production cost can be minimized.

5.2 ANSYS ENVIRONMENT

The ANSYS program has a comprehensive Graphical User Interface (GUI) that give user’s easy interactive access to program functions, commands, documentation and reference material. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a key board or a combination of both. Users can also interact with ANSYS using the commands. There are more than 1200 commands. The easiest way to communicate with ANSYS program is by using the Graphical User Interface (GUI).

5.2.1 ANSYS Levels

The ANSYS program in organized to two basic levels.

• Begin level

• Processor level

The Begin level acts as a gateway into and out of the ANSYS program. It is also used for certain global program controls such as changing the job name, clearing (zeroing out) the database, and copying binary files.

At the Processor level, several processors are available. Each processor is a set of functions that perform a specific analysis task. For example, the General pre-processor (PREP7) is where we build the model, the solution processor (SOLUTION) is where the loads are applied, the general postprocessor (POST1) is where the results are evaluated. An additional
postprocessor, POST26, enables you to evaluate solution results at specific points in the models as a function of time.

5.3 ENTERING A PROCESSOR

In general, users can enter a processor by selecting it from the ANSYS main menu in the graphical user interface (GUI). For example, choosing Main menu > Pre-processor takes one into PREP7. Alternatively, we can use commands that are used to enter a processor. To return to the begin level from a processor select finish from the main menu.

The ANSYS program stores all input data as well as results data in one large database in an organized fashion. The main advantage of database is that you can list, display, modify or delete any specific data item quickly and easily.

5.4 COMPOSITES

Composite materials are those containing more than one bonded material, each with different structural properties. Composites are somewhat more difficult to model than an isotropic material such as iron or steel.

Composite materials can be modelled by using specialized elements called layered elements. After modelling we can do any structural analysis (including non-linearities).

A composite model can be built

1. By choosing the proper element type.
2. Defining the layered configuration,
3. Specifying the failure criteria and
4. Following modelling and post processing guidelines.

Elements used for layered analysis

SHELL 99 - Linear layered structural shell

(250 uniform thickness layer)

SHELL 91 - Non-linear layered structural shell

(Up to 100 layers)

SOLID 46 - allow material matrix input.

The layer configuration is defined

1. By specifying individual layer properties

2. By defining the constitutive materials for each layer the following properties are specified in element real constant table:

   1. Material properties
   2. Layer orientation angle
   3. Layer thickness

5.4.1 Units

Using the unit’s command any system of units can be chosen.

Building a model requires more of once time that any other part of the analysis. First one specifies a job name and analysis title. Then one uses the PREP7 pre-processor to define the element type real constants, material properties and the model geometry.
5.4.2  Defining Element Types

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID 96, etc. The various element categories available are:

- Beam Infinite
- Pipe Combination
- Link Shell
- Mass Contact
- Solid Fluid
- Plane Visco elastic

5.4.3  Defining Real Constants

Element real constants are properties that depend on the element type such as cross section properties of a beam element, area moment of inertia, height etc. For example, real constants for BEAM3, the 2-D beam element, are area (Area), moment of inertia, finite element solution. Using the PREP7 pre-processor, loads can be applied.

The word loads as used in ANSYS documentation includes boundary condition (constraint supports or boundary field specification) as well as externally and internally applied loads. Loads in the ANSYS programs are divided into six categories:

1. DOF constraints
2. Forces
3. Surface loads
4. Body loads
5. Inertia loads
6. Coupled field loads

5.4.4 Initiating the Solution

The solution is obtained by using the command solve (or) current LS from the main menu. When this command is used the ANSYS program takes model and loading information from the database and calculates the results.

5.4.5 Review of Results

Once the solution has been calculated, the ANSYS post processor can be used to review the results. Two-post processors are available: POST 1 and POST 26.

Using POST1, General post processor review the results at one sub step over the entire model or selected option of the model. Using POST26, the time - history post processor, the results at specific points in the model over all time steps can be obtained.

5.4.6 Various Analyses Types in ANSYS

1. Static
2. Transient
3. Harmonic
4. Model
5. Spectrum
6. Buckling
7. Sub-structuring
5.4.7 Special Purpose Features

1. Fracture mechanics
2. Composites
3. Fatigue
4. P-method
5. Beam analysis

5.4.8 Non-Linear Options

Nonlinear options are used only if non-linearities are present (plasticity, contact elements, creep, etc.) and they include the options listed below.

- Number of Time steps (NSUBST)
- Size of Time Steps (DELTIM)
- Stepped or Romped Loads (KBC)
- Automatic Time Stepping (AUTOTS)
- Maximum Number of Equilibrium Iterations (NEQIT)
- Convergence Tolerances (CNVTOL)
- Predictor - Corrector Option (PRED)
- Line Search Option (LNSRCH)
- Creep Criteria (CRPLIM)
- Solution Termination Options (NCNV)
- Cutback Criteria (CUT CONTROL)

The beam model is created and analysed as described below:
5.5 CHOOOSING THE ANALYSIS TITLE AND JOBNAME

1. Choose menu path Utility menu > File > change Title.

2. Type the text “Non-linear Analysis”.

3. Click OK.


5. Enter the job name and click OK.

5.6 DEFINING THE ELEMENT TYPE

1. Choose menu path main Menu > Pre-processor > Element Type > Add / Edit / Delete.

2. Click on Add. The Library of Element Types dialog box appears.

3. In the list on the left, click on “Structural Solid”.

4. In the list on the right, click once on “Solid 45”.

5. Click on OK.

5.7 DEFINING MATERIAL PROPERTIES

1. Choose menu path Manu >Pre-processor > Material props > Constant - isotropic.

2. Click on OK to specify material number. Another isotropic material properties dialog box appears.

3. Enter the material properties and click OK.
5.8 DEFINING THE MULTILINEAR ISOTROPIC HARDENING TABLE FOR NON-LINEAR ANALYSIS

1. Choose menu path main Menu>pre-processor> Material Props>Data Tables > Define / Activate.

2. In the list of data table types, choose “Multi linear isotropic Hardening”.

3. Enter the material reference number.

4. Enter the number of data points and click OK.

5. Choose the menu path Menu >Pre-processor> Material props > Data Tables > Edit activate and enter the stress and strain values.

6. Choose File, Apply/Quit.

5.9 CREATING VOLUME BLOCK

1. Choose menu path main menu>pre-processor>Modelling - Create > Volume - Block > By Dimensions.

2. Enter width for X-Coordinate, thickness of layer for Y-coordinate, length of the beam for Z-coordinate.

3. Repeat the above steps to create all the eight layers.

5.10 SETING UP ELEMENT ATTRIBUTES

Choose menu path Menu>Pre-processor> Element Attributes - Volumes. Select the volumes one by one and set the corresponding material properties.
5.11 SETING UP ELEMENT SIZE

1. Choose menu path Main Menu > Pre-processor > Mesh tool > Element attributes > Global

2. Click Size controls > Global set. Enter the element size or number of elements dialog box appears. Enter the no. of elements along the length of the beam.

5.12 MESHING THE VOLUME


2. Click on pick all button. The volume gets divided into specified number of elements.

3. Click on SAVE_DB on the ANSYS Toolbar.

4. Click on Close the Mesh Tool.

5.13 ASSIGNING ANALYSIS AND LOAD STEP OPTIONS

1. Choose menu path Main Menu > Solution > Unabridged menu > New Analysis > Static.

2. Choose menu path Main Menu > Solution > Unabridged menu > Solution controls.

a. Basic > Analysis options > Large Displacement Static (On) > Analysis options > Number of Sub steps (Give the number of sub steps and also the maximum and minimum number of sub steps).
b. Nonlinear > Maximum number of equilibrium iterations> Convergence criteria (Force and Displacement).

3. Choose menu path Main Menu > Solution > Unabridged menu > Analysis Options > Non-linear Options > Large Deformation effects(on) > Newton - Raphson (Full or Modified) > Non - Linear options > Equation Solver (Program Chosen) > Non - Linear Options > Stress Stiffening (on)

5.14 APPLYING CONSTRAINTS AND SOLVING


2. Select the nodes which have to be restrained and click the respective DOF’s to be constrained. Click OK.

3. Choose menu path Main Menu > Solution > Solve - Load step Opts - Time / Frequency > Time and sub steps. Enter the number of sub steps, maximum number of sub steps and the minimum number of sub steps.

4. Choose menu path Main Menu > Solution > Loads - Apply > structural - forces / Moment on Nodes. Select the nodes and enter the direction and value of force to be applied.

5. Choose menu path Main Menu > Solution > solve - current LS. Review the information in the / STAT window, and click on close.

6. Click on OK - solve current load step dialog box.
7. Click on close on the information dialog box when the solution is done.

8. Click on SAVE_DB on the ANSYS toolbar.

5.15 USING THE GENERAL POSTPROCESSOR TO PLOT RESULTS

1. Choose menu path Main Menu > General Postprocessor > Read Results - Last set.

2. Choose menu path Main Menu > General postprocessor> Plot Results > deformed shape. The deformed mesh appears in the ANSYS graphics window.

3. Choose menu path Main Menu > General Postprocessor > Plot results > contour plot-Element Solution. The contour element solution dialog box appears.

4. In the selection box, select Stress SZ and click OK. The contour plot appears in the graphic window. Thus the results can be plotted and listed.

Element Description

List of solid elements

1. SOLID Quad 4 Node 42
2. 4 Node 182
3. 4 Node 183
4. 8 Node 82
5. Triangle 6 Node -2
6. Axis - Harmonic 4 Node 5
7. 8 Node 53
8. Brick 8 Node - 45
9. 9 Node 1185
10. 20 Node 186
11. 220 Node 95
12. Layered 46
13. aniso 64
14. Concrete 65
15. W/rotation 73
16. Tet 10 Node 187
17. 10 Node 92
18. W/rotate 72

Of the above element list Brick 8 Node - 45 (SOLID - 45) is used.

**SOLID-45 (3D Structural Solid Element)**

SOLID - 45 is used for the three-dimensional modelling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. A reduced integration option with hourglass control is available.
Input Data

The geometry, node locations, and the coordinate system for this element are shown in SOLID-45. The element is defined by eight nodes and orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is described in coordinate systems. Properties are not input default as described in linear material properties.

An analysis using uniform integration can have the following disadvantages:

The analysis is not as accurate as the full integration method, which is apparent in the linear analysis for the same mesh.

The analysis cannot capture the bending behaviour with a single layer of elements, for example, in the case of a fixed-end cantilever with a lateral point load, modelled by one layer of elements laterally. Instead, four element are usually recommended.

A summary of the element input is given in element input summary. A general description of element input is given in element input.

SOLID45 Input Summary

<table>
<thead>
<tr>
<th>Element Name</th>
<th>Solid-45</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nodes</td>
<td>I, J, K, L, M, N, O, P</td>
</tr>
<tr>
<td>Degrees of Freedom</td>
<td>UX, UY, UZ</td>
</tr>
<tr>
<td>Real Consents</td>
<td>Hourglass control factor needed only when KEYOPT (2) =1.</td>
</tr>
</tbody>
</table>

Note: The valid value for this real constant is any positive number; default = 1.0. A value between 1 and 10 is recommended for use.
<table>
<thead>
<tr>
<th>Material</th>
<th>EX, EY, EZ, (PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ, DENS, GXY, GYZ, DAMP</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface Loads</td>
<td>Pressure: Face 1 (I-I-L-K), Face 2 (I-J-N-M), Face 3 (J-K-O-N), Face 4 (K-L-P-O), Face 5 (L-I-M-P), Face 6 (M-N-O-P)</td>
</tr>
<tr>
<td>Body Loads</td>
<td>Temperature: T(I), T(J), T(K), T(L), T(M), T(O), T(P)</td>
</tr>
<tr>
<td>Fluences</td>
<td>FL(I), FL(J), FL(K), FL(L), FL(M), FL(N), FL(O), FL(P)</td>
</tr>
<tr>
<td>Special Features</td>
<td>Plasticity, Creep, Swelling, Stress Stiffening, Large deflection, Large strain, Birth and death, Adaptive descent</td>
</tr>
<tr>
<td>KEYOPT (1)</td>
<td>0-Include extra displacement shapes</td>
</tr>
<tr>
<td></td>
<td>1-Suppress extra displacement shapes</td>
</tr>
<tr>
<td>KEYOPT (2)</td>
<td>0-Full integration with or without extra displacement shapes, depending on the setting of KEYOPT (1)</td>
</tr>
<tr>
<td></td>
<td>1-Uniform reduced integration with hourglass control; suppress extra displacement shapes KEYOPT (1) is automatically set to 1)</td>
</tr>
<tr>
<td>KEYOPT (4)</td>
<td>0-Element coordinate system is parallel to the global coordinate system</td>
</tr>
<tr>
<td></td>
<td>1-Element coordinate system is based on the element I-J side</td>
</tr>
</tbody>
</table>
KEYOPT (5) 0-Basic element solution
1-Repeat basic solution for all integration points
2-Nodal stress solution

KEYOPT (6) 0-Basic element solution
1-Surface solution for face I-J-N-M also
2-Surface solution for face I-J-N-M and face K-L-P-O
3- Nonlinear solution at each integration point also
4- Surface solution for faces with nonzero pressure

KEYOPT (9) 0-No user subroutine to provide initial stress
(default) 1- Read initial stress data from user subroutine USTRESS

Output Data

The solution output associated with the element is in two forms:

Nodal displacements included in the overall nodal solution

Additional element output as shown in element output definitions

Several items are illustrated is stress output. The element stress directions are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate systems and are available for any face KEYOPT (6)). The coordinate systems for faces IJNM and KLPO are shown insolid-45. The other surface coordinate system follow similar orientations as indicated by the pressure face node description. Surface stress printout is valid only if the conditions described in element solution are met. A general description of solution output is given in solution output.
Output Data

The solution output associated with the element is in two forms:

Additional element output as shown in element output definitions

Several items are illustrated in stress output. The element stress direction are parallel to the element coordinate system. The surface stress outputs are in the surface coordinate system and are available for any face KEYOPT (6)). The coordinate systems for faces IJNM and KLPO are shown insolid-45. The other surface coordinate systems follow similar orientations as indicates by the pressure face node description. Surface stress print out is valid only if the conditions described in element solution are met. A general description of solution output is given in solution output.

When KEYOPT (2) = 1 (the element is using uniform reduced integration), all the outputs for the element integration points are output in the same style as the full integration outputs. The number of points for full integration is used for consistency of output within the same element type.

The Element Output Definitions table uses the following notation: A colon (:) in the Name column indicates the item can be accessed by the Component Name method.

1. Nonlinear solution, output only if the element has a nonlinear material

2. Face printout (if KEYOPT (6)).is 1,2, or 4)

3. Available only at centroid as a *Get item
Tale 5.1 Solid45 Miscellaneous Element Output

<table>
<thead>
<tr>
<th>Description</th>
<th>Names of items Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nonlinear Integration Pt. Solution</td>
<td>EPPL, EPEQ, SRAT, SEPL, HPRES, EPCR, EPSW</td>
</tr>
<tr>
<td>Integration Point Stress Solution</td>
<td>TEMP, S(X,Y,Z,XY,YZ,XZ), SINT, SEQV, EPEL</td>
</tr>
<tr>
<td>Integration Point Stress Solution</td>
<td>TEMP, S(X, Y, Z, XY, YZ, XZ), SINT, SEQV, EPEL</td>
</tr>
<tr>
<td>Nodal Stress Solution</td>
<td>TEMP, S(X,Y,Z,Y,XY,XZ), SINT, SEQV, EPEL</td>
</tr>
</tbody>
</table>

1. Output at each of eight integration points, if the element has a nonlinear material and KEYOPT(6) = 3

2. Output at each integration point, if KEYOPT(5) = 1

3. Output at each node, if KEYOPT(5) = 2

Assumptions and Restrictions

Zero volume elements are not allowed. Elements may be numbered either as shown in SOLID45 or may have the planes IJKL and MNOP interchanged. Also, the element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.

All element must have eight nodes. A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers. A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.
Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumption and restriction are given in the previous section.

ANSYS/Professional

- The DAMP material property is not allowed.
- KEYOPT (6) = 3 is not applicable.
- Fluency body loads are not applicable.
- The only special feature allowed is stress stiffening.

5.16 NON-LINEAR ANALYSIS

Basically, finite element analysis of RC structures may be classified into two approaches. They are,

(1) The modified stiffness approach

(2) The layer approach

The non-linear problems can be classified into two categories,

- Material non-linearity
- Geometric non-linearity

5.16.1 Material Non-Linearity

Material stress-strain relation which is generally non-linear is the prime cause for material non-linearity. In the case of geometric non-linearity
the reason is geometric shape assumed by the structure under loading. In the case of RC structures the geometric non-linearity is insignificant because the displacement of the structure is negligibly small. The effects of material non-linearity include stress-strain relation of concrete, yielding of concrete and steel bond slip phenomenon, tension stiffening, creep and shrinkage, etc. Among these, the non-linear stress-strain and cracking of concrete are the prime factors that promote material non-linearity to a great extent. Therefore in the present investigation mainly the material non-linear effects due to non-linear stress-strain curve are considered.

Many methods are available to solve the basic equation in non-linear finite element analysis (FEA). Among them there are two popular methods which are based on incremental solution. They are iterative methods

1. Complete Newton Raphson Method
2. Modified Newton Raphson Method

The first category of solution assembles and solves the stiffness matrix at every iteration level. It has quadratic convergence properties which mean in subsequent iterations the relative error decreases quadratically.

5.17 FEATURES OF ANSYS

Typical steps involved in the analysis here is (1) building the model, (2) application of load on the structure and (3) reviewing the results. Appropriate units are selected for the analysis. Subsequently the analyst selects PREP7 pre-processor to define element type, real constants, material properties and the model geometry.

An element library is available in ANSYS package. This library consists of more than 100 different types of elements. Each element type in
the library possesses a unique number and prefix that identifies the element category such as BEAM4, PLANE77, SOLID96, etc.

In ANSYS it is mandatory to define the real constants. Element real constants are properties that depend on the element type such as cross sectional properties of beam element, area, moment of inertia, height, etc. For example, real constants for Beam3, the 2D beam element, area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS).

Next the material properties have to be defined. Most element types require material properties depending on application. The material properties may be,

(i) Linear or non-linear
(ii) Isotropic, orthotropic or anisotropic
(iii) Constant temperature or temperature dependent

5.18 CREATING MODEL GEOMETRY

Once the material properties have been defined the next step of analysis is generating the finite element model. Here the nodes and elements of the structure are identified. These parameters adequately describe the model geometry. Different approaches to generation of model are:

(a) Creating a solid model with ANSYS
(b) Using direct generation
(c) Importing from CAD system
Steps involved in model generation with ANSYS are:

1. Enter the pre-processor (PRER7) mostly to initiate the model once. The model is built using solid modelling.
2. Establish a working plane.
3. Generate basic geometric features.
4. Activate appropriate coordinate system.
5. Create key points and define lines, area and volumes as required.
6. Use more Boolean operators or number controls to join separate solid model regions together as appropriate.
7. Create tables of element attributes.
8. Set meshing controls to establish the desired mesh density if so required.
9. Create nodes and elements by meshing one’s solid model.
10. Save the model and exit from the pre-processor.

After creating the model it is required to mesh it. Three main steps are involved in generating a mesh of nodes and elements. They are as follows.

i. Set the element attributes.

ii. Set mesh control free or mapped meshing.

iii. Generate the mesh.
5.19 APPLYING LOADS AND OBTAINING THE SOLUTION

Using the solution processor, the analysis type and analysis options are defined, then loads are applied and load steps options are prescribed and finite element solution is initiated. Using PREP7 pre-processor loads are applied. The solution is obtained by using the command SOLVE or CURRENT LS from the main menu. When the command is used the ANSYS program takes the model and loading information from the database and calculates the results. Once the solution was obtained ANSYS post processor can be used to review the results.

5.20 NUMERICAL ANALYSIS OF BEAMS

The behaviour of concrete beams strengthened with a Steel Fibre-Reinforced Concrete (SFRC) layer was studied by Nonlinear Finite Element Analysis using ANSYS software (Elmezaini&Ashour 2015). Ordinary concrete as well as SFRC were modelled using the multi-linear isotropic hardening constants where they are assumed to have a linear behaviour up to 30% of the compressive strength. Afterwards, a multi-linear stress-strain curve was defined. For reinforcing steel, a linear-elastic perfectly-plastic material model was used. Steel fibre reinforced concrete was modelled by the smeared modelling technique.

The deep beams were meshed with an 8-noded solid element with the utilization of one point integration to create the concrete beam. Nonlinear Finite Element Analysis was conducted on the beams to predict the behaviour of these beams during all loading stages up until failure. The well-known FE package ANSYS v. 14 was used. The ANSYS program includes a library of elements for different applications. SOLID65 is used for the 3D modelling of concrete with or without reinforcing bars. SOLID65 is capable of cracking in tension and crushing in compression. It can also consider plastic deformation.
and creep. For steel rebar, ANSYS presents LINK180 to model reinforcing steel which is simply a pin-joined one dimensional element. The geometry and the coordinate system for the SOLID65 and the LINK180 elements are shown in Figure 5.1 and Figure 5.2.

5.20.1 Modelling of Concrete

Concrete is a quasi-brittle material and has a highly nonlinear and ductile stress-strain relationship (Dawari & Vesmawala 2014). The nonlinear behaviour is attributed to the formation and gradual growth of micro cracks under loading. Figure 5.3 shows a typical stress-strain curve for normal weight concrete. In compression, the stress-strain curve for concrete is linearly elastic up to about 30% of the maximum compressive strength. Above this point, the stress increases gradually up to the maximum
compressive strength. After it reaches the maximum compressive strength $f_{cu}$, the curve descends into a softening region, and eventually crushing failure occurs at an ultimate strain $\varepsilon_{cu}$. In tension, the stress-strain curve for concrete is approximately linearly elastic up to the maximum tensile strength. After this point, the concrete cracks and the strength decreases gradually to zero.

Figure 5.3 Stress-strain curve for concrete

5.20.2 Characteristic of Steel

For steel reinforcement, a linear-elastic perfect-plastic material model was adopted. For practical reasons, steel is assumed to exhibit the same stress-strain curve in compression as in tension. Passion’s ratio for steel will be set to 0.3, modulus of elasticity will be set to 200 GPa, the characteristic strength for flexural reinforcement will be set to 415MPa, while for secondary reinforcement and stirrups it will be set to 250 MPa. The tangent modulus for the flexural, secondary reinforcement and stirrups will be set to 2000 MPa.
Figure 5.4 Meshing of deep beam

Figure 5.5 Loading on slender beam
Figure 5.6 Deformation pattern of slender beam

The load-deflection curves of experimental beam and that analysed by ANSYS are shown in Figure 5.7. There is almost close agreement between the two in the behaviour. Initially the behaviour is linear elastic. At load more than 80% of the ultimate there is non-linearity in the behaviour.
Figure 5.7  Superposition of Load-deflection curves of experimental and ANSYS values