SAE TECHNICAL PAPER SERIES

2004-01-1660

First Order Analysis for Automotive Body Structure Design – Part 3: Crashworthiness Analysis Using Beam Elements

Hidekazu Nishigaki
Toyota Central R&D Labs., Inc.

Noboru Kikuchi
The University of Michigan

Reprinted From: CAD/CAM/CAE Technology and Design Tools (SP-1858)

ISBN 0 7680 1425-5

9 780768 014259

SAE International™

2004 SAE World Congress
Detroit, Michigan
March 8-11, 2004

400 Commonwealth Drive, Warrendale, PA 15096-0001 U.S.A. Tel: (724) 776-4841 Fax: (724) 776-5760 Web: www.sae.org

For permission and licensing requests contact:

SAE Permissions
400 Commonwealth Drive
Warrendale, PA 15096-0001-USA
Email: permissions@sae.org
Fax: 724-772-4891
Tel: 724-772-4028

Global Mobility Database®

All SAE papers, standards, and selected books are abstracted and indexed in the Global Mobility Database.

For multiple print copies contact:

SAE Customer Service
Tel: 877-606-7323 (inside USA and Canada)
Tel: 724-776-4970 (outside USA)
Fax: 724-776-1615
Email: CustomerService@sae.org

ISBN 0-7680-1319-4

Printed in USA

2004-01-1660

First Order Analysis for Automotive Body Structure Design – Part 3: Crashworthiness Analysis Using Beam Elements

Hidekazu Nishigaki
Toyota Central R&D Labs., Inc.

Noboru Kikuchi
The University of Michigan

ABSTRACT

We have proposed *First Order Analysis* (FOA) as a method, which the engineering designers themselves can use easily in an initial design stage. In this paper, we focus on the crashworthiness, and present the method to predict the collapse behavior of the frame member. This method is divided into two parts. Those are (1) collapse analysis under loading conditions of combined axial force and bending moment to the cantilever, and (2) collapse analysis of structural member considering the previously obtained moment – rotation angle relationship using the beam element. In comparison with the results according to the detailed Finite Element Analysis (FEA) model, effectiveness and validity of this method are presented.

INTRODUCTION

In recent years, Computer Aided Engineering (CAE) has been remarkably developed, and has spread widely. Consequently, before making prototypes, quantitative prediction of performance of automobiles is becoming to some extent possible. Structure with more sufficient performance can be proposed based on the results of CAE. Since the importance of CAE has been increasing, the CAE specialist has shifted to more advanced analysis – material and geometrical nonlinear problems, multi-field problems, for example. As CAE tool for the engineering designers which can directly use a CAD model has been put in practical use, in many linear problem cases, engineering designers have come to perform structure examination using CAE in the drawing stage. By the way, in the initial design stage, a novel proposal and substantial change for layout of the automobile body structure are comparatively easy. Generally, in this initial design stage, CAE analysis using a rough model or the modified model of a vehicle of an old type has been performed easily, and recognized as an effective method.

On the other hand, although the calculation using theory of mechanics of materials and the simple analysis using beam element are unsuitable for quantitative examination, they are effective in an understanding of a phenomenon by the reason for simplifying an object based on designer's know-how. Moreover, change for layout of the automobile body structure is very easy. Therefore, in the examination stage before creating a CAD drawing, it is considered effective methods. And also when evaluating and considering the result of CAE, it helps to acquire more knowledge.

We have proposed *First Order Analysis* (FOA) as a method, which the engineering designers themselves can use easily in an initial design stage (Nishigaki [1]). This method makes it the main purpose to understand the characteristic of a subject and to assist thinking and consideration of an engineering designer in the conceptual design stage before carrying out CAE. The basic concepts of FOA include (1) graphical user interfaces for the engineering designers using Microsoft/Excel (2) use of well understandable formulations based on the theory of mechanics of materials.

By the way, responding to high demand for weight efficient and crashworthy design of automobile bodies, many studies on thin-wailed members subject to large deformation have been carried out. Abramowicz [2] and Kecman [3] studied the problem of a deep bending collapse of open and closed section, respectively. Both authors followed the so-called kinematic method of plasticity, which involved determination of a suitable folding mechanism with stationary and/or moving plastic hinges. Kecman, Sadeghi and Vignjevic [4] had developed a compound beam element with varying joint and hinge moment-rotation curves and implemented into the public version of program DYNA-3D for the purpose of responding to a demand in the early design stages. This beam combines the existing Belytschko-Schwer beam element with rotary springs having very nonlinear moment-rotation curves in the elastic-plastic and deep collapse range. The element behavior under uniaxial and biaxial bending presented using simple models where various effects can be easily traced. Kim and Wierzbicki [5] had investigated the crushing response of thin-walled prismatic column under combined loading of compression and bending to develop simplified crash-oriented design tools. The initial and subsequent failure loci-representing interaction between sectional force and moment were constructed from the numerical results. Also analytical solution of the same problem was derived. Takada and Abramowicz [6] had presented the object-oriented formulation of the finite element algorithm that encompasses traditional finite element, super element, experimental data, and codes of practice and rigid body mechanics in a single calculation environment. This formulation is implemented in software for dynamic crash simulation of an arbitrary 3D-frame structure discretized into superbeam elements and subjected to large dynamic crash loading.

In many cases, many structural members of the automotive body are subject to a combined compression-bending loading, and the cross-sectional shape is also arbitrary. Furthermore, considering that the engineering designers themselves perform these examinations in a conceptual design stage, it is desirable to complete a series of work on one notebook type personal computer with simple operation.

In this paper, the concept of FOA described above is applied to prediction of the collapse characteristic of the structure component of the body. First, cross-sectional shape is created by mouse operation, and the material characteristic is inputted in dialogue form. By the above operation, the FEA model by the shell element, which extended cross-sectional shape automatically, is created. Collapse analysis under loading conditions of combined axial force and bending moment is carried out to this easily created linear cantilever. Next, the finite element analysis model of structural member using a beam element is created. The collapse characteristic obtained at the previous step is given into a bending collapse part in a whole model. Then, the energy absorption characteristic etc. is easily predicted by carrying out collapse analysis of structural member.

FOA SYSTEM FOR CRASHWORTHINESS ANALYSIS

In FOA, by regarding easiness of understanding of the mechanical characteristic, beam elements and panel elements are fundamentally used. FOA system for this crashworthiness analysis uses beam element with nonlinear characteristics (Figure 1). In-house finite element analysis (FEA) software is used for this analysis. This software can be analyzed simultaneously in consideration of large deformation and vibration of structure. And this software is based on the beam theory, which updates the coordinates of structure for every time step, and calculates the dynamic response of structure by direct numerical integration. Moreover, this software can also treat a beam element with nonlinear stiffness.

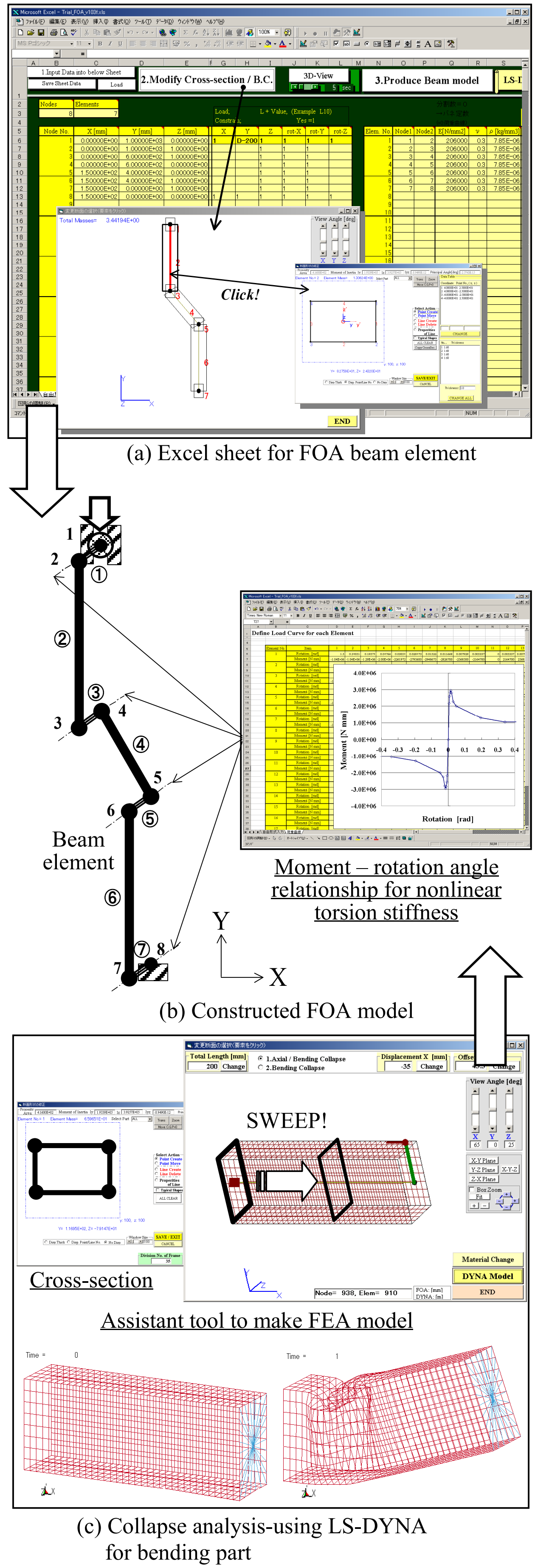
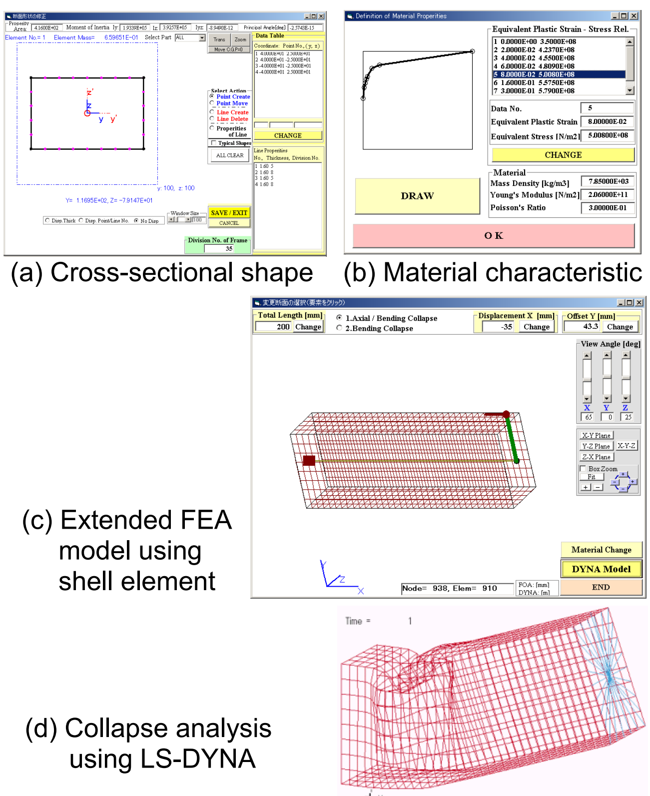


Figure 1. FOA system for crashworthiness analysis

The graphical interface constructed on Excel is prepared in order to make this FOA model (Figure 1(a)). Excel is equipped with the so-called Visual Basic for Application (VBA). Using this function, clicking actions lead to (1) write/read data in cells, (2) move to a different sheet, (3) calculate some schemes, (4) I/O with external files, and (5) start external programs. FEA analysis of the shell model using general-purpose transient dynamics finite element program LS-DYNA is carried out in order to obtain the collapse characteristic of the bending part (Figure 1(c)). The assistant tool is prepared so that the engineering designer may easily make this shell model. This collapse characteristic of the bending part can also be used by the results of the experiment. And these characteristics are inputted into the above FOA beam model (Figure 1(b)).

ASSISTANT TOOL FOR COLLAPSE CHARACTERISTIC OF BENDING PART

FEA model using shell element is automatically created by extending the cross-sectional shape. This model can be obtained only by creating the cross-sectional shape and inputting the material characteristic. After that, collapse analysis (quasi-static analysis by implicit approach) under the loading condition of combined axial force and bending moment of linear cantilever is carried out. The flow of procedure is shown in Figure 2. Procedure is summarized as follows:



(a) Cross-sectional shape (b) Material characteristic

(c) Extended FEA model using shell element

(d) Collapse analysis using LS-DYNA

(e) Moment – rotation angle relationship

Figure 2. Flow of procedure to obtain collapse characteristic of bending part

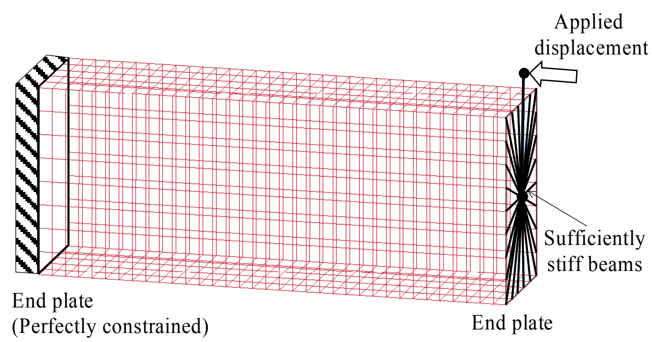
(a) Design of cross-sectional shape.
(b) Modification of the material characteristic.
(c) Construction of extended FEA model using the shell element.
(d) Collapse analysis-using LS-DYNA.
(e) Post process of " moment – rotation angle relationship".

OPERATIONAL PROCEDURE

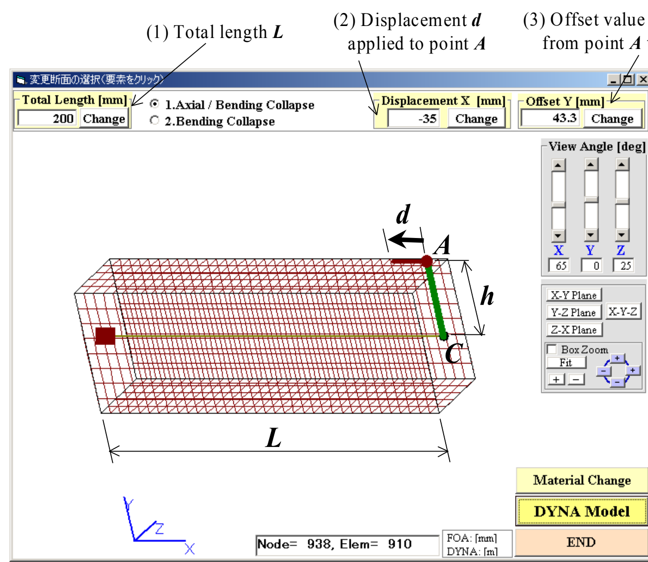
Here, the operational procedure of the assistant tool to obtain the collapse characteristic of bending part is explained.

Loading condition of collapse analysis is shown in Figure 3(a). A left end is perfectly constrained. The arbitrary displacement, which increases in linearly, is given to a right end. The end plate is represented by connecting all nodes of a right end cross section with a sufficiently stiff beam to the center node of a cross section. Furthermore, extending a sufficiently stiff beam from the central point of a cross section enables the input of offset value.

The main operation items in this main operation frame (Figure 3(b)) are summarized as follows:



(a) Loading condition of combined axial force and bending moment of cantilever



(b) Operation items of main frame

Figure 3. Main operation frame to obtain collapse characteristic of bending part

(1) Total length *L*.
(2) Arbitrary displacement *d* applied to an upper right side point *A*.
(3) Offset value *h* from the central point *C* of cross section to an above-mentioned applied point *A*.

The axial collapse characteristic can also be obtained by setting the offset value h to 0. In addition, by clicking the "DYNA Model" button, the node number and element number of input data for LS-DYNA are displayed, and amount of a model can be checked.

Next, the creation procedure of cross-sectional shape is explained. If member is clicked at the main operation frame (Figure 3(b)), frame, which creates cross-sectional shape as shown in Figure 4, will appear. While we can create and modify the cross-sectional shape by mouse operation, morphing (another operation frame appears by click of the "Zoom" button) of this shape can also be performed. Furthermore, in order to create a FEA model, the number of division of the axial direction when extending cross-sectional shape can be changed.

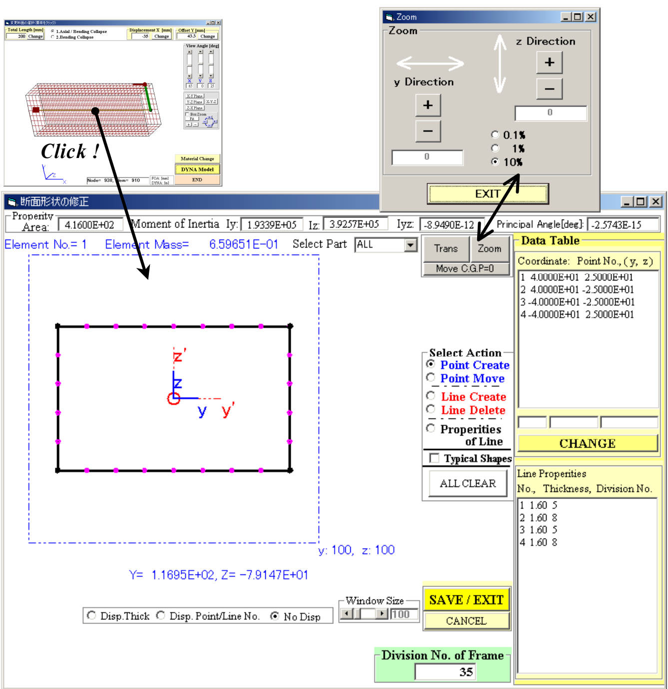


Figure 4. Sub frame to create cross-sectional shape

Next operation is the input of the material characteristic. If a "Material Change" button is clicked at the main operation frame (Figure 3(b)), the input frame of the material characteristic shown in Figure 5 will appear. First, "Mass density", "Young's modulus", and a "Poisson's ratio" can be inputted with the "Material" frame at the lower right side. By the default, the iron characteristic is always given. Next, a check and correction of the relationship between equivalent plastic strain and an equivalent stress can be made. By clicking the "DRAW" button, this relationship can be displayed in a graph image (horizontal axis: equivalent plastic strain, vertical axis: equivalent stress). In the "Equivalent Plastic Strain – Stress Rel." frame at the upper right side, equivalent plastic strain and the equivalent stress which were clicked are inputted into a lower text box, and can be changed.

In addition, the generated moment *M* is estimated by the product of the reaction force *Fx* of displacement applied point, and the offset value ⊿*Y* from the center of the cross section of perfectly constrained side, as shown in Figure 6.

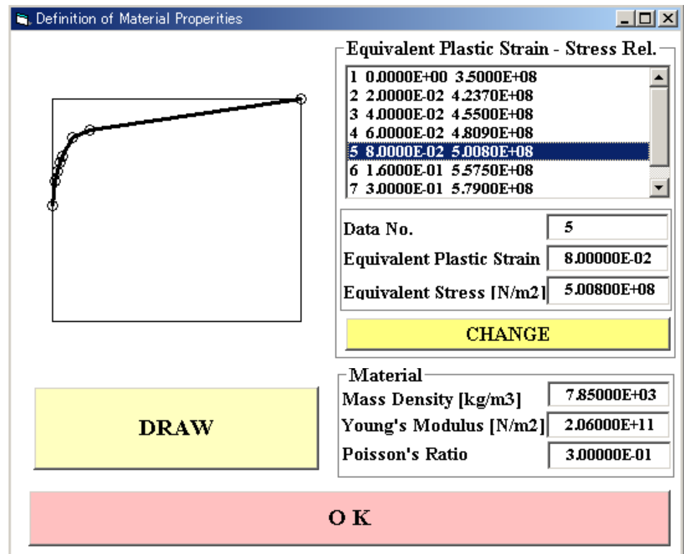


Figure 5. Sub frame to input material characteristic

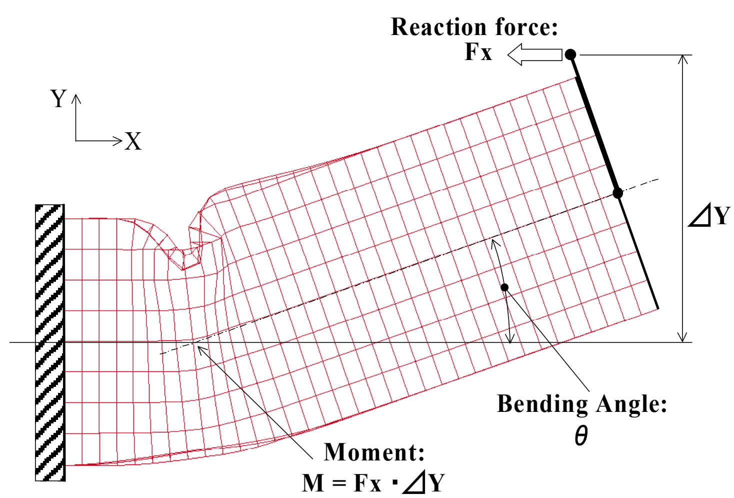


Figure 6. Relationship between applied moment M and deformed shape

COLLAPSE ANALYSIS METHOD OF STRUCTURAL MEMBER

In this stage, the finite element analysis model of structural member using beam element is created, and the collapse characteristic that obtained by previous step to bending part is given. And prediction of the energy absorption characteristic etc. is simply enabled by carrying out collapse analysis of structural member.

In usual case, LS-DYNA can be used for analysis solver for this like problem. "MAT_FORCE_LIMITED" material model is prepared in the LS-DYNA to realize the plastic hinge concept. This material model is available for the Belytschko resultant beam element. Plastic hinges form at the ends of the beam when the moment reaches the plastic moment. The moment versus rotation relationship is specified by the user in the form of a load curve and scale factor (Hallquist [7]).

But, in this paper, we also use the in-house finite element analysis software for the analysis of structural member to perform the basic study. This software can be analyzed simultaneously in consideration of large deformation and vibration of structure. And it is the finite element analysis software based on the beam theory, which updates the coordinates of structure for every time step, and calculates the dynamic response of structure by direct numerical integration (Ishiyama [8], Nishigaki [9] and Akasaki [10]). Moreover, this software can also treat a beam element with nonlinear torsion stiffness (Nishigaki [11]).

DESCRIPTION OF THE MOTION OF A FLEXIBLE STRUCTURE

To analyze the large deformation of a structure, we used an incremental finite element method and step by step time integration method. Namely, the coordinates of the nodes are modified at each incremental time step; thus, geometrical nonlinear characteristic is treated by stepwise linear approximation.

Figure 7(a) illustrates a beam which is an element of a three dimensional flexible structure. The position vector $\{\mathbf{r}\}_i$ of the *i*-th node is defined by the global coordinates X,Y and Z,

$$\{\mathbf{r}\}_i^T = \{X_i,\ Y_i,\ Z_i,\ \theta_{Xi},\ \theta_{Yi},\ \theta_{Zi}\} \qquad (1)$$

The superscript T denotes the transpose of the matrix. The force vector $\{\mathbf{S}\}_j$ of the *j*-th element is defined by the local coordinates *x*, *y* and *z*,

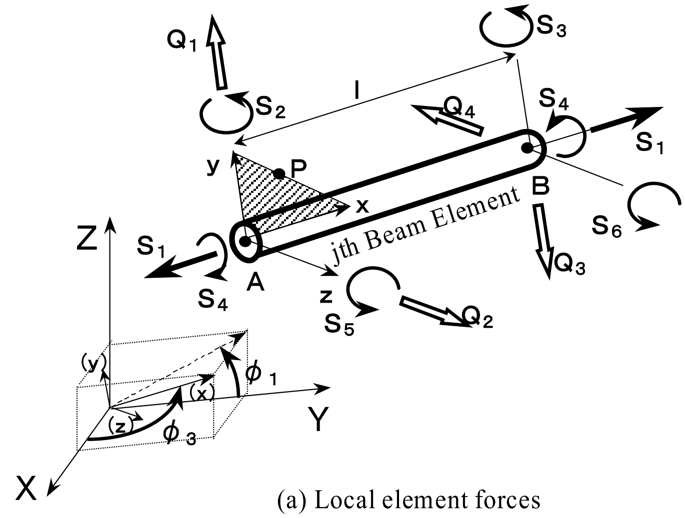
$$\{\mathbf{S}\}_j^T = \{S_1, S_2, S_3, S_4, S_5, S_6\} \qquad (2)$$

In Eqn (2), S_1 denotes axial force, S_2, S_3, S_5 and S_6 are bending moments and S_4 denotes the torsion moment. The displacement vector corresponding to the force vector is expressed as $\{\mathbf{s}\}_j$, where s_1 denotes the axial displacement, s_2, s_3, s_5 and s_6 are angles of rotation and s_4 denotes the torsion angle. The relation between $\{\mathbf{s}\}_j$ and $\{\mathbf{r}\}_j$ is given in terms of the transformation matrix $[\mathbf{a}]_j$,

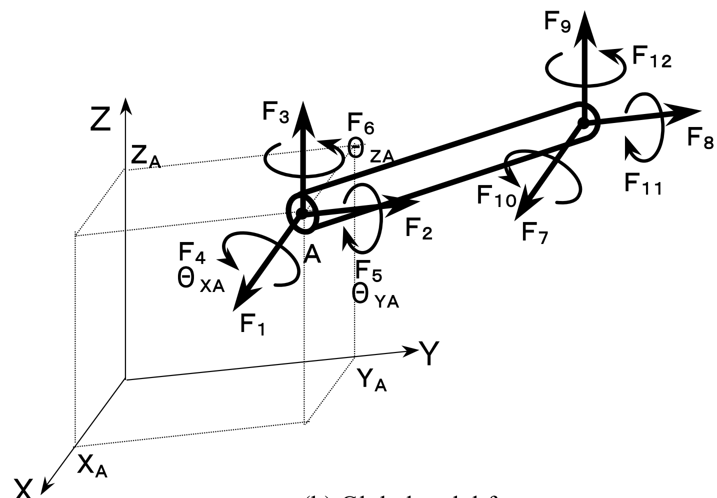
$$\{\dot{\mathbf{s}}\}_j = [\mathbf{a}]_j\ \{\dot{\mathbf{r}}\}_j \qquad (3)$$

The superdot denotes a time differential. When the mass of an element is allocated to the nodes as a lumped mass, the equations of motion for the nodes are written

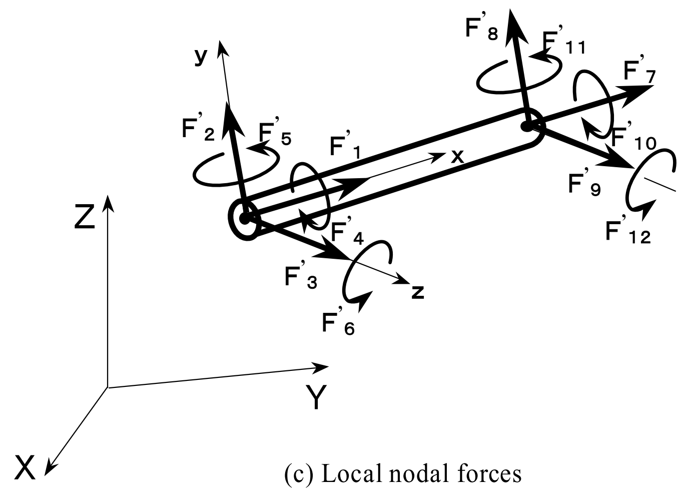
$$\sum_{j=1}^{Ne}(\ [\mathbf{M}]_j\ \{\dot{\mathbf{q}}\}_j\ ) = -\sum_{j=1}^{Ne}[\mathbf{a}]_j^T\{\mathbf{S}\}_j+\{\mathbf{R}\} \qquad (4)$$



(a) Local element forces



(b) Global nodal forces



(c) Local nodal forces

Figure 7. Local and global coordinates, force vector components of beam element

Where [**M**] is the mass matrix and {**R**} is the applied load vector. *Ne* is the number of elements, and $\{\mathbf{q}\}_j = \{\dot{\mathbf{r}}\}_j$.

The coordinate transformation matrix [λ] from the global to local coordinates can be written:

$$\{x, y, z\}^T = [\,\lambda\,]\,\{X, Y, Z\}^T \tag{5}$$

This matrix [λ] is expressed in terms of the direction cosine (ξ , η , ζ) of the local *x*-axis to the global coordinates and a vector (X_{AP}, Y_{AP}, Z_{AP}), which is located from nodal point *A* to point *P*. Where point *P* is used to determine the principal *x-y* surface of the cross-section of the element. Therefore, this point *P* is located on one of the principal surfaces and determines the local *x-y* plane with the local *x*-axis (Figure 7(a)),

$$[\,\lambda\,] = [\mathbf{B}]\,[\,\phi\,] \tag{6}$$

$$[\mathbf{B}] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\beta & \sin\beta \\ 0 & -\sin\beta & \cos\beta \end{bmatrix}, \tag{7}$$

$$[\,\phi\,] = \begin{bmatrix} \phi_{11} & \phi_{12} & \phi_{13} \\ \phi_{21} & \phi_{22} & \phi_{23} \\ \phi_{31} & \phi_{32} & \phi_{33} \end{bmatrix} = \begin{bmatrix} \xi & \eta & \zeta \\ -\eta & \pm 1 \mp \frac{\eta^2}{1\pm\xi} & \frac{\mp\eta\zeta}{1\pm\xi} \\ \mp\zeta & \frac{-\eta\zeta}{1\pm\xi} & 1-\frac{\zeta^2}{1\pm\xi} \end{bmatrix} \tag{8}$$

$$\left.\begin{aligned} \xi &= \cos\phi_3 \\ \eta &= \sin\phi_3 \cos\phi_1 \\ \zeta &= \sin\phi_3 \sin\phi_1 \end{aligned}\right\} \tag{9}$$

$$\left.\begin{aligned} \cos\beta &= y'_{AP} \Big/ \sqrt{y'^2_{AP} + z'^2_{AP}} \\ \sin\beta &= z'_{AP} \Big/ \sqrt{y'^2_{AP} + z'^2_{AP}} \end{aligned}\right\} \tag{10}$$

$$\left.\begin{aligned} y'_{AP} &= \phi_{21} X_{AP} + \phi_{22} Y_{AP} + \phi_{23} Z_{AP} \\ z'_{AP} &= \phi_{31} X_{AP} + \phi_{32} Y_{AP} + \phi_{33} Z_{AP} \end{aligned}\right\}$$

The sign convention in Eqn (8) is that if ξ is positive or zero, the upper signs are used and vice versa.

The transformation matrix $[\mathbf{a}]_j$ of the *j*-th element is derived by the following operations. The global nodal force vector $\{\mathbf{F}\}_j^T = \{F_1, F_2, F_3, \cdots, F_{12}\}$ in Figure 7(b) constitutes a complete set of forces equal to the number of degrees of freedom assumed on the *j*-th element. From the principle of virtual work, we obtain:

$$\{\mathbf{F}\}_j^T \{\mathbf{q}\}_j = \{\mathbf{S}\}_j^T \{\dot{\mathbf{s}}\}_j \tag{11}$$

Substituting Eqn (3) into Eqn (11), we obtain the relation between $\{\mathbf{F}\}_j$ and $\{\mathbf{S}\}_j$:

$$\{\mathbf{F}\}_j = [\mathbf{a}]_j^T \{\mathbf{S}\}_j \tag{12}$$

The relationship between $\{\mathbf{F}\}_j$ and the local nodal forces $\{\mathbf{F'}\}_j^T = \{F'_1, F'_2, F'_3, \cdots, F'_{12}\}$ in Figure 7(c) is given by $\{\mathbf{F}\}_j = [\,\lambda\,]^T \{\mathbf{F'}\}_j$. Meek [12] has shown that the local element forces $\{\mathbf{S}\}_j^T = \{S_1, S_2, S_3, S_4, S_5, S_6\}$ are sufficient to define all the local nodal force vectors $\{\mathbf{F'}\}_j$. The transformation from $\{\mathbf{S}\}_j$ to $\{\mathbf{F'}\}_j$ is written

$$\{\mathbf{F'}\}_j = [\mathbf{L}]_j \{\mathbf{S}\}_j \tag{13}$$

$$[\mathbf{L}]_j = \begin{bmatrix} -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/l_j & 1/l_j \\ 0 & -1/l_j & -1/l_j & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1/l_j & -1/l_j \\ 0 & 1/l_j & 1/l_j & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Thus $[\mathbf{a}]_j^T$ is obtained using the coordinate transformation matrix [λ] and the transformation matrix $[\mathbf{L}]_j$ which eliminates rigid-body motion in the local coordinates,

$$[\mathbf{a}]_j^T = [\,\lambda\,]^T [\mathbf{L}]_j = \begin{bmatrix} \lambda_j^T & \mathbf{0} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \lambda_j^T & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \lambda_j^T & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} & \lambda_j^T \end{bmatrix} [\mathbf{L}]_j \tag{14}$$

Where l_j is the length of the *j*-th element.

The local element force vector $\{\mathbf{S}\}_j$ is related to the elemental deformations as follows. In order to damp the vibration of the beam elements, the visco-elastic Voigt model, which combines the viscous element with the elastic element in parallel, is introduced. The damping characteristics of the beam element are assumed to be proportional to the element stiffness matrix. The proportional factor is denoted by α . Thus, the constitutive equation for the beam element is described by

$$\{\dot{\mathbf{S}}\}_j = [\mathbf{k}]_j \left[\, [\mathbf{a}]_j \left(\{\mathbf{q}\}_j + \alpha \{\dot{\mathbf{q}}\}_j \right) \right] \tag{15}$$

The stiffness matrix of the beam element [**k**] is given by:

$$[\mathbf{k}]_j = \begin{bmatrix} \frac{EA}{\ell} & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{4EI_y}{\ell} & \frac{2EI_y}{\ell} & 0 & 0 & 0 \\ 0 & \frac{2EI_y}{\ell} & \frac{4EI_y}{\ell} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{GJ}{\ell} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{4EI_z}{\ell} & \frac{2EI_z}{\ell} \\ 0 & 0 & 0 & 0 & \frac{2EI_z}{\ell} & \frac{4EI_z}{\ell} \end{bmatrix} \quad (16)$$

Where E and G denote the modulus of elasticity and shear modulus of the beam, A, I and J are the cross sectional area, its moment of inertia and its polar moment, respectively.

Hence, by combining Eqn (3), (13), (14) and (15), the stiffness matrix $[\mathbf{k_s}]_j$ is expressed in terms of local nodal force components $\{\mathbf{F}'\}_j$, as described by Meek[12]

$$[\mathbf{k}_s]_j = [\mathbf{L}]_j[\mathbf{k}]_j[\mathbf{L}]_j^T \quad (17)$$

We want to use a relatively large time step during direct integration in order to avoid unnecessary high frequency vibrations, so we should select the unconditionally stable scheme. Thus we use the Newmark β method (β=1/4), namely,

$$\{\mathbf{q}_1\}_j = \{\mathbf{q}_0\}_j + (h/2)\left(\{\dot{\mathbf{q}}_0\}_j + \{\dot{\mathbf{q}}_1\}_j\right) \quad (18)$$

$$\{\mathbf{S}_1\}_j = \{\mathbf{S}_0\}_j + (h/2)\left(\{\dot{\mathbf{S}}_0\}_j + \{\dot{\mathbf{S}}_1\}_j\right) \quad (19)$$

These equations are combined with Eqn (4) and (15), so simultaneous differential equations for $\{\mathbf{q}\}_j$ and $\{\mathbf{S}\}_j$ can be solved, where h is the time step and subscripts 0 and 1 indicate time t_0 and $t_1=t_0+h$, respectively.

The beam element used in FOA is the straight one. Therefore, in order to express the curved part, it is necessary to carry out polygonal line approximation. In this time, influence of the rigidity by curvature with the delicate surface of the wall in bending part, etc. are supposed to be disregarded.

In order to represent the nonlinear moment-rotation angle characteristic of bending part obtained by the previous step, the beam element that has nonlinear torsion stiffness is prepared. Since the stepwise linear approximation method is used, this nonlinear characteristic can be considered by giving tangent of the moment-rotation angle curve. It is also possible by setting up this element very short and joining in the direction of x, y, and z-axis together in series to give the nonlinear moment-rotation angle characteristic over three dimensions to a joint part.

EXAMPLE

In order to verify the validity of this method, analysis for structural member shown in Figure 8 is carried out. First, (1) the moment-rotation angle characteristic is obtained with the easy operating tool to obtain the collapse characteristic of bending part. Next, (2) the finite element analysis model of structural member using a beam element is created. The collapse characteristic obtained at the previous step is given into a bending collapse part in a whole model. Then, the energy absorption characteristic etc. is easily predicted by carrying out collapse analysis of structural member. The collapse analysis result of the detailed shell element model using LS-DYNA is used for verification.

Cross-sectional shape of structural member is a rectangle (width: 80mm, height: 50mm and thickness: 1.6mm). Loading and boundary condition is shown in Figure 9. Structural member for analysis is connected with two sets by sufficiently stiff members. And these members are compressed in vertical direction. The material is iron. These values are shown in Figure 10.

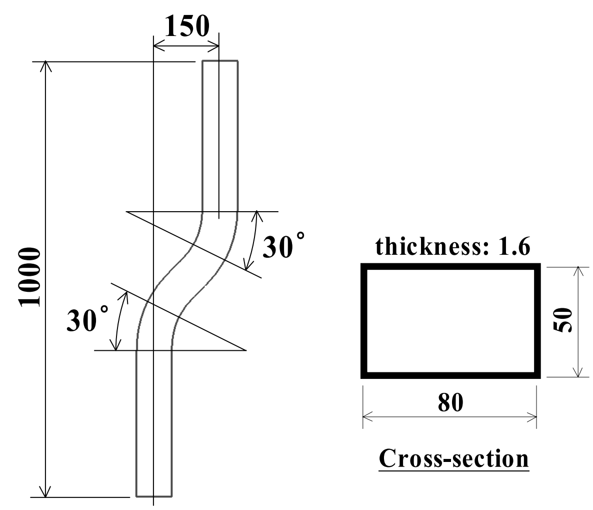


Figure 8. Dimensions of structural member for example

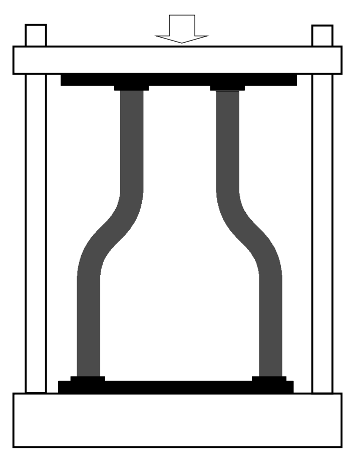


Figure 9. Loading and boundary condition for example

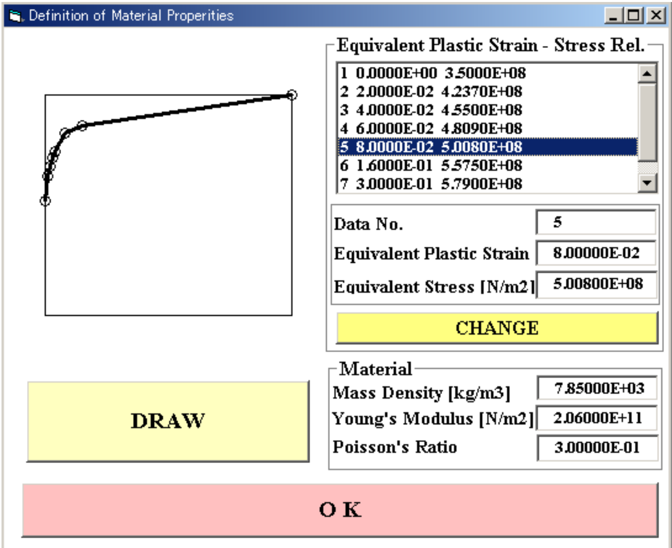


Figure 10. Material property (iron)

As a result of preliminary linear analysis using beam element according to the loading and the boundary condition shown in Figure 9, the value of the ratio of the moment and load of curved part was 43.3. Therefore, Offset value *h* in Figure 3(b) is set to 43.3mm. This time, total length *L* in Figure 3(b) is set to 200mm. The displacement *d* in Figure 3 is set to 35mm.

The deformation shape of the analysis result of this condition is shown in Figure 11. As shown in Figure 12, data processing of the moment-rotation angle characteristic obtained in this analysis is automatically carried out by Excel using a macro language. In addition, moment *M* is calculated as mentioned above relationship shown in Figure 6.

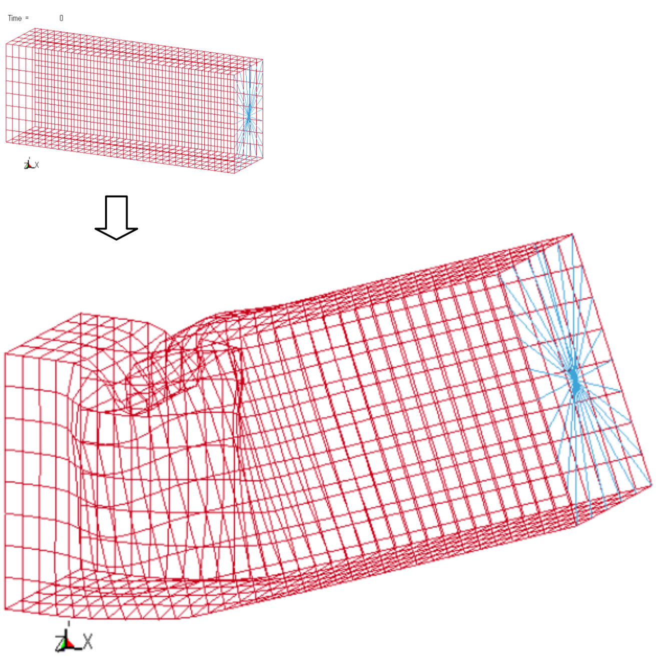


Figure 11. Deformation shape obtained by collapse analysis of cantilever using LS-DYNA

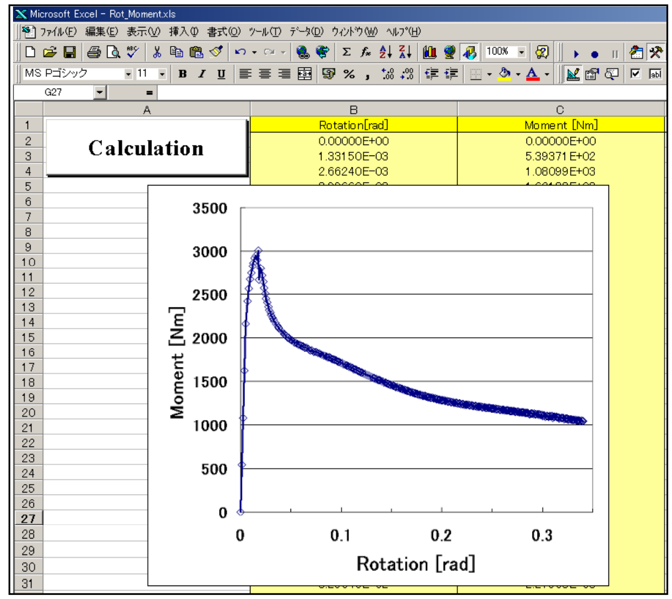


Figure 12. Moment-rotation angle characteristic automatically obtained by Excel using a macro language

Collapse analysis of structural member is carried out using the FOA tool. First, the moment-rotation angle characteristic of bending part calculated above is inputted into the Excel sheet by reducing the redundant data (Figure 13). Next, in another Excel sheet (Figure 14), the FEA model using the beam element of structural member is created. Here, coordinates of each node and the constraint and loading conditions over each degrees of freedom are set as each cell of an Excel sheet. Furthermore, the beam model information except the cross-sectional characteristic is created by inputting the node number of the start and end point of each element. Cross-sectional shape can be easily modified by mouse operation in another frame which appear by clicking "Modify Cross-section / B.C." button.

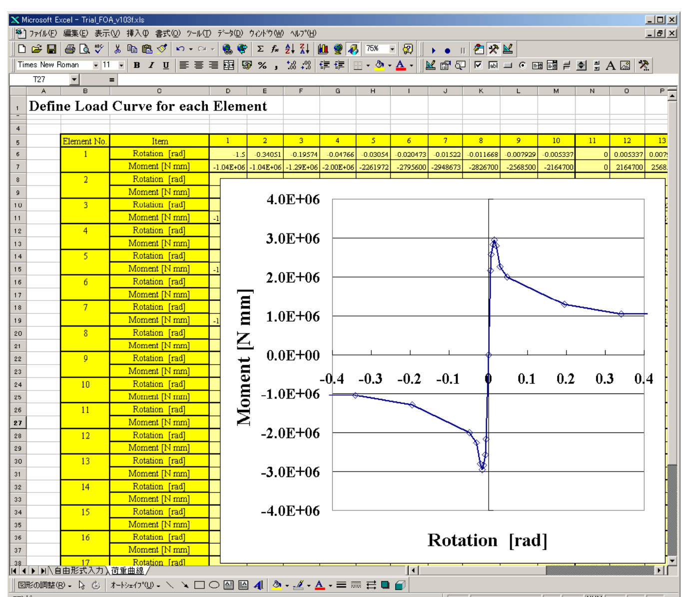


Figure 13. Moment-rotation angle characteristic inputted into Excel sheet by reducing redundant data

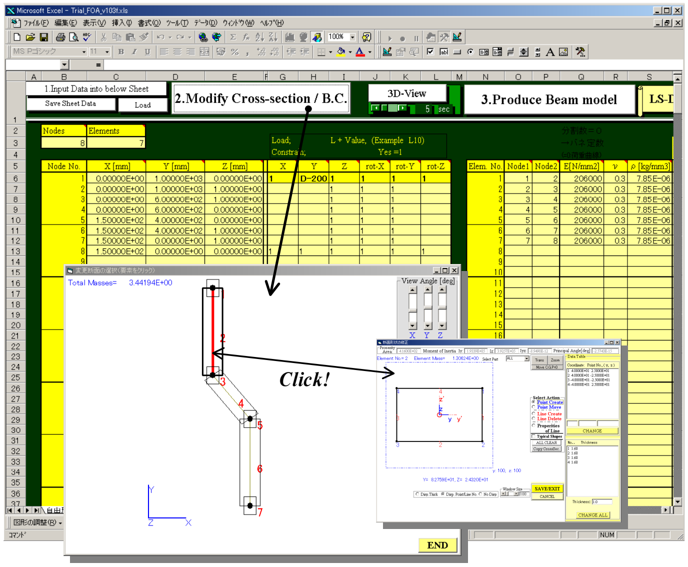


Figure 14. Excel sheet for FEA model using beam element of structural member

By the way, as for the beam element, which has the nonlinear torsion characteristic for expressing the moment-rotation angle characteristic of bending part, it is desirable for the length to be theoretically close to zero. The length of this beam element is set to 1mm in this paper.

The model created here is roughly shown in Figure 15. In this figure, element ①, ③, ⑤ and ⑦ are the beam element which have the nonlinear torsion characteristic with a length of 1mm, and remaining element ②, ④, and ⑥ are the usual beam element.

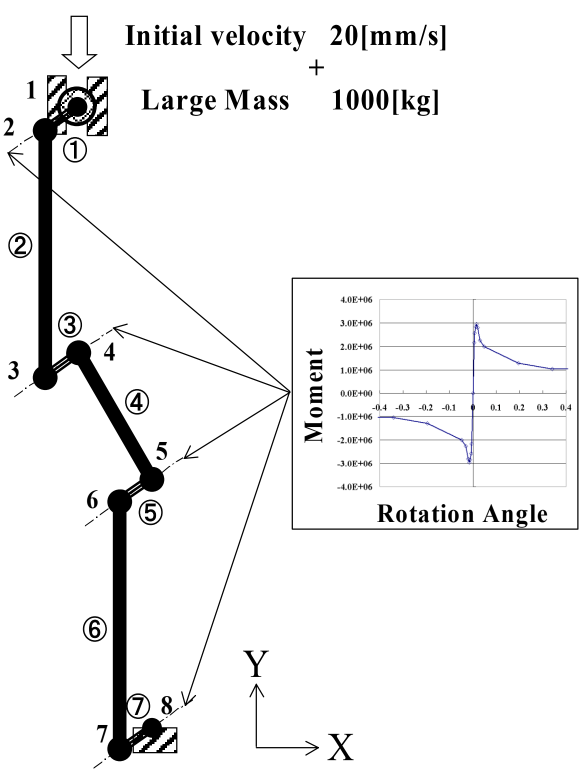


Figure 15. Illustration of FEA model for in-house software

Bottom node 8 is perfectly constrained (six degrees of freedom are constrained). Top node 1 is constrained about five degrees of freedom except the vertical direction (*Y*-axis direction). As for loading conditions, a quasi-static state is assumed. The mass of 1000kg is given to top node 1 as Large Mass, and initial velocity 20 [mm/sec] is given downward (opposite *Y*-axis direction). By the analysis to 10sec, final displacement is 200mm. In addition, since sufficiently large mass is given, the speed of node 1 is constant. Therefore, displacement is increasing in linearly. The input data of the in-house finite element analysis program based on beam theory is automatically created by mouse click in the "Produce Beam Model" button of the FOA tool shown in Figure 14. Moreover, this program can be performed automatically.

Figure 16 shows the detailed FEA model using the shell element created for verification. Type 16 in LS-DYNA, which is fully integrated formulation, is chosen as the shell element formulation. Number of through thickness integration points is set to 5. Type 24 in LS-DYNA, which is an elasto-plastic material with an arbitrary stress versus strain curve, is chosen as material type.

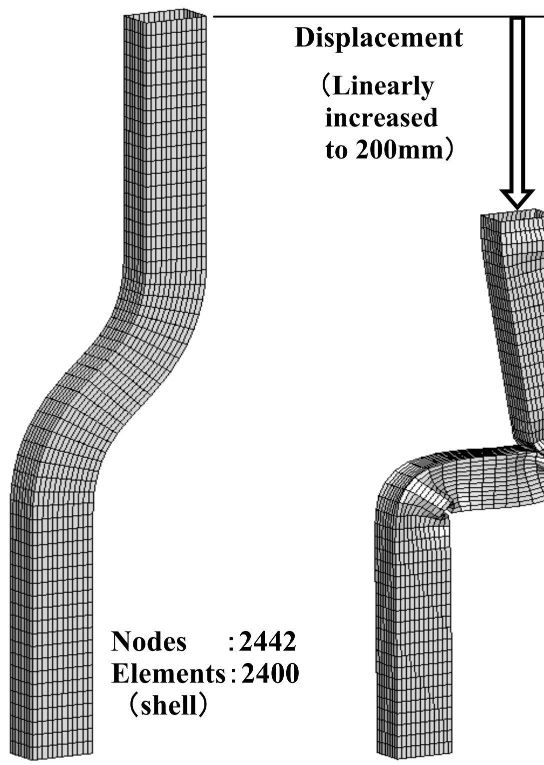


Figure 16. Detailed FEA model using shell element for verification

The deformation shapes obtained by these analyses are shown in Figure 17. In both analysis results, three hinges are created at bending part except a bottom end. Two results show that these deformed shapes are almost same.

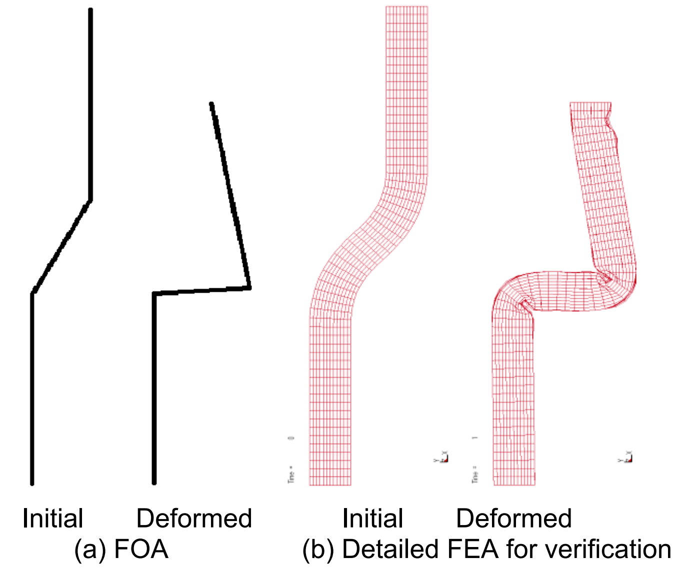


Figure 17. Deformed shapes obtained by FOA and detailed FEA for verification

The relationship between the reaction force (the value of the vertical direction: F_Y) and displacement (δY) of a moving point obtained by FOA and detailed FEA for verification are shown in Figure 18. By comparison with these results, it is verified that the result by simple FOA tool created for engineering designers can obtain the almost same result as LS-DYNA using the detailed shell element. Thereby, it is able to be shown that this method is effective and appropriate one.

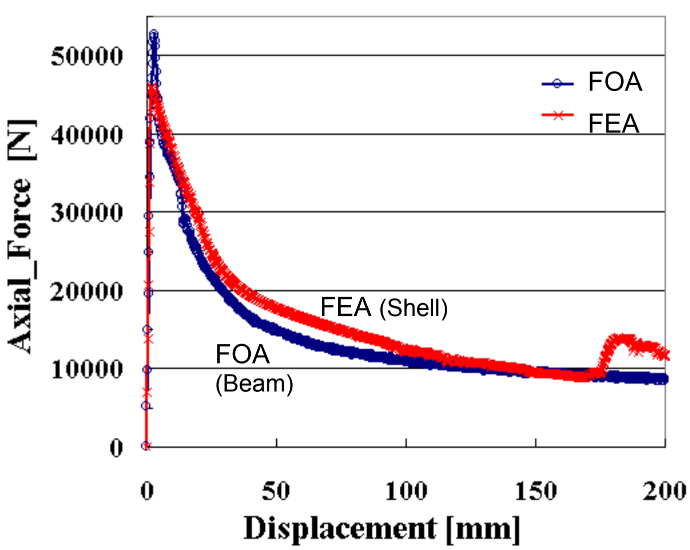


Figure 18. Relationship between reaction force (F_Y) and displacement (δY) of moving point obtained by FOA (Beam) and detailed FEA (Shell) for verification

CONCLUSION

In this paper, we focus on the crashworthiness, and present the method to predict the collapse behavior of the frame member. This method is divided into two parts. Those are (1) collapse analysis under loading conditions of combined axial force and bending moment to the cantilever, and (2) collapse analysis of structural member considering the previously obtained moment – rotation angle relationship using the beam element. In comparison with the results according to the detailed FEA model, effectiveness and validity of this method are presented.

REFERENCES

1. Nishigaki H., Nishiwaki S., Amago T. and Kikuchi N., 2000, "First Order Analysis for Automotive Body Structure Design", ASME DETC2000/DAC-14533.
2. Abramowicz W., 1981, "Simplified crushing analysis of thin-walled columns and beams", Eng. Trans, 29(1), pp. 5-26.
3. Kecman D., 1983, "Bending collapse of rectangular and square section tubes", Int. J. Mech. Sci., 25(9/10), 623-36.
4. Kecman D., Sadeghi M. and Vignjevic R., 1992, "The compound beam element with non-linear moment-rotation curves for the side impact and roof crush analysis using DYNA3D program", SAE paper no.921072.
5. Kim H-S. and Wierzbicki T., 2001, "Crush behavior of thin-walled prismatic columns under combined bending and compression", Comput. Struct. 79, pp. 1417-1432.
6. Takada K. and Abramowicz W., 2002, "Novel Formulation of the 3D Large Deformation Beam element for Dynamic Crash Analysis(in Japaneses)", JSAE paper no.20025436.
7. Hallquist J. O., 1998, "LS-DYNA Theoretical Manual", Livermore Software Technology Corp.
8. Ishiyama S., Takagi J., Yamamoto K. and Nishimura T., 1988, "Impact response of thin-walled frame structures", Int. J. Impact Engng, Vol.7, No.2, pp. 197-212.
9. Nishigaki H. and Kawashima K., 1998, "Motion Control and Shape Optimization of a Suitlike Flexible Arm", Structural Optimization 15, pp. 163-171.
10. Akasaki F., Shimojima S., Nishigaki H. and Ishiyama S., 1992, "Large deformation analysis of flexible hose by dynamic finite element method", 25th ISATA Symposia, pp. 663-670.
11. Nishigaki H., MIKI K. and Ishiyama S., 1999, "Development of the Impact Response Analysis Program of the Pedestrian Dummy (in Japanese)", JSME 12th Computational Mechanics Conference Paper No.99-5, pp. 481-482.
12. Meek,J.L., 1971, "Matrix structural analysis", New York: McGraw-Hill.

CONTACT

The corresponding author of this paper is Hidekazu Nishigaki. He is now working at Toyota Central R&D Labs Inc. His e-mail address is e0793@mosk.tytlabs.co.jp