

MODELLING NONLINEAR MATERIAL BEHAVIOUR FOR ARBITRARY CROSS-SECTION GEOMETRIES WITH A TWO DIMENSIONAL GEOMETRICALLY EXACT BEAM MODEL

C. Colomer Segura*¹, M.Feldmann¹, R. Fontecha Gonzalez¹

¹Institute for Steel Construction
RWTH Aachen University
carles.colomer@rwth-live.de

Keywords: Arbitrary Geometry, Nonlinear Material, Nonlinear Cross-Section, Geometrically Exact Beam

Abstract. *The calculation of the nonlinear response of steel structures under explosive loading has become of greater interest in the latest years due to the increasing risk of terrorist actions. In order to perform an appropriate risk assessment of the industrial facilities, finite element calculations under consideration of large deflections and activation of material nonlinearities need to be conducted to evaluate the damage level of the structural members and hence, of the whole structure.*

Within the framework of the Project ADBLAST, funded by the Research Fund for Coal and Steel, the dynamic response of steel constructions under blast loads has been investigated by Finite Element calculations. In order to allow for a faster and numerically affordable simulation of whole industrial buildings, the structure needed to be modeled by one-dimensional beam elements.

For the investigated deformation levels, the generalized stress-strain relations over the cross-section (N - ϵ , Q - γ , M - κ) become nonlinear. Classical cross-section constitutive models are based on Gauss integration at certain section points, limiting very often the application to regular shaped sections, as well as limiting the exactness specially of the nonlinear M - κ relation depending on the number of chosen integration points.

Within this investigation, an alternative modeling of the cross-sectional constitutive model has been developed based on a two dimensional meshing of the section geometry. At each mesh cell, a rheological model with isotropic and kinematic hardening has been implemented. This methodology easily allows for the calculation of composite sections. In a later step this section has been integrated in a Matlab-based FEM environment with 2D geometrically exact beam elements modeled according to [7] and [9]. With this model, a free combination of rheological models and geometry is possible, while the precision of the calculated nonlinear section behavior is controlled by the mesh refinement. In a final step, an analysis of accuracy, computation speed and capability to describe the desired nonlinear processes was performed.

1 INTRODUCTION

The calculation of the nonlinear response of steel structures under impulsive loading has become of greater interest in the latest years. The increasing interest of designers for accounting for the effects of extraordinary actions such explosions or car impacts on the main structural members of the structure, has lead to the use of advanced Finite Element Calculations for assessing the achieved levels of deformation or damage.

The load bearing structure of steel buildings is mostly defined by beam-like elements and therefore the use of beam-based FE-programs has always been preferred by designers. The computational cost of static calculations under ordinary actions is negligible, since all members remain in the elastic domain and only small displacement are allowed under ordinary loading actions.

When the designers try to consider the effect of plasticity due to extraordinary impulsive actions, many beam-based FE-programs do not have implemented accurated nonlinear cross-sectional models, so that the designer is forced to work with shell elements, which leads often to the use of additional more advanced FE-Software such as Ansys[©] or Abaqus[©], thus uncoupling the ordinary design of the structure from the design against extraordinary actions.

Structural members that must withstand impulsive actions should always be chosen to have sufficient rotation capacity, and hence of the section class 1 acc. to [1]. In this situation, the effects of local buckling at the cross-section are negligible and the use of one-dimensional material models becomes acceptable, since the dominating stresses are only perpendicular to the cross-section.

The goal of this paper is to present a simple implementation of a one-dimensional rheologic model as it can be found in [4, 8] in an arbitrary section geometry, that can be used in any beam-based FE-Framework. All codes have been written in Matlab[©] and will be presented here with an example of its usage.

In Chapter 2 the analysis of an arbitrary section geometry will be shown. Here a freely distributable meshing algorithm [6] has been used to describe the section shape. In Chapter 3 an easy implementation of a one-dimensional material model accounting for kinematic and isotropic hardening will be presented. This material model will be implemented at each cell of the meshed geometry, leading to an accurate nonlinear section model presented in Chapter 4. The example of an implementation of an IPE-100 profile will serve to show the used methodology in Chapter 5. Finally, a summary of the obtained results will show the advantages of this type of implementation for Finite Element codes based on beam elements.

2 ANALYSIS OF ARBITRARY GEOMETRIES

The analysis of arbitrary cross-sections can be classified into two categories, depending on the level of detail of the investigations to be performed:

- The determination of the basic sectional properties: Center of Mass (x_G, y_G) , Area A , Stiffnesses I_{xx} , I_{yy} and I_{xy} , and the resistance parameters $W_{el,x}$, $W_{el,y}$, $W_{pl,x}$, $W_{pl,y}$ is quite straight-forward and is sufficient to characterise the cross-sectional behaviour in 2-dimensional problems.
- The determination of the advanced sectional properties such as shear areas $A_{Q,x}$, $A_{Q,y}$, torsional stiffness I_t and warping stiffness I_w , which require the solution of differential equations over the section domain as described in [2, 3]. This properties are only needed

in case of using a 3-dimensional beam formulation and will not be discussed here in detail.

2.1 Meshing the Geometry

Since we are dealing with an arbitrary section geometry, a meshing of the section into smaller geometrical units is necessary. In this case, a freely distributable mesh generator for MATLAB [6] can be used to break down the geometry into small triangular cells. The proposed meshing algorithm works essentially in two steps: in the first one, the node locations are found by solving for equilibrium in a truss structure (as an analogy of a simplex mesh) using piecewise linear force-displacement relations at each iteration. The second step consists on the topology adjustment using (also iteratively) the Delaunay triangulation algorithm. For further reference on its use see [5].

2.2 Calculation of Section Properties

The calculation of the basic section properties can be performed by calculating the Center of Mass ($x_{G,i}, y_{G,i}$), Area A_i and Moments of Inertia $I_{xx,i}, I_{yy,i}, I_{xy,i}$ at each cell i . The following equations yield the wanted section properties:

$$A = \sum A_i; \quad x_G = \sum \frac{x_{G,i} * A_i}{A}; \quad y_G = \sum \frac{y_{G,i} * A_i}{A}; \quad (1)$$

$$I_{xx} = \sum I_{xx,i} + A_i * (y_{G,i} - y_G)^2; \quad I_{yy} = \sum I_{yy,i} + A_i * (x_{G,i} - x_G)^2; \quad (2)$$

$$I_{xy} = \sum I_{xy,i} + A_i * (x_{G,i} - x_G) * (y_{G,i} - y_G) \quad (3)$$

$$W_{el,xx} = \frac{I_{xx}}{\max|y_i - y_G|}; \quad W_{el,yy} = \frac{I_{yy}}{\max|x_i - x_G|}; \quad (4)$$

$$W_{pl,xx} = \sum |y_{Gi} - Na_y| * A_i; \quad W_{pl,yy} = \sum |x_{Gi} - Na_x| * A_i; \quad (5)$$

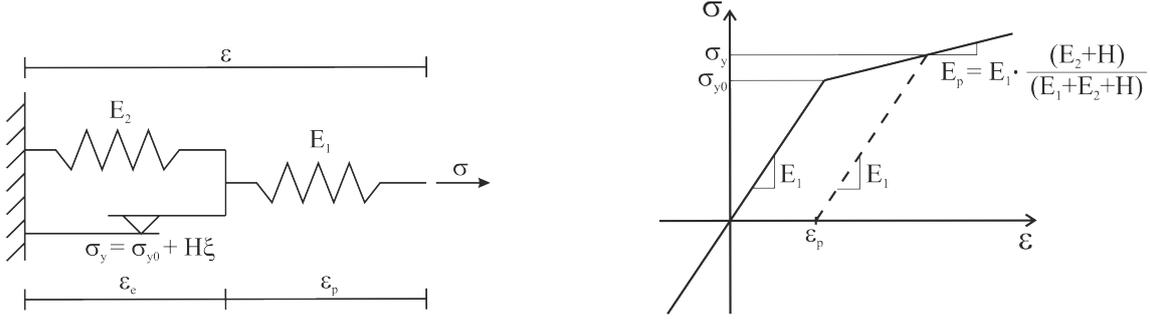
where Na is the position of the neutral point (point of 0 stresses) in a given stress state.

The generated mesh and the obtained section properties are now also an adequate set of data to follow the proceedings exposed in [2] and [3]. According to their methodology, after determining the section warping function with a Dirichlet boundary value problem, the coordinates of the twist center (which coincide with those of the shear center, see also [2]) and the modulus I_t and I_w can be obtained. Furthermore, if an additional Neumann boundary value problem is solved, the shear stresses in the section, the reduced areas $A_{Q,x}, A_{Q,y}$ and shear correction factors k_x, k_y can be obtained.

3 RHEOLOGICAL MODEL

A simple one-dimensional material model has been implemented in Matlab according to [4, 8]. This Model considers both kinematic and isotropic hardening (as shown conceptually in Figure 1).

Here the yield stress at each time is defined according to $\sigma_{y,t} = \sigma_{y0} + H \cdot \xi_t$, where σ_{y0} is the initial yield stress of the material, H the isotropic hardening parameter and ξ the cumulated plastic strain that is calculated at a certain time step by $\xi_t = \xi_{t-1} + |(\epsilon_{p,t} - \epsilon_{p,t-1})|$. This means that each iteration can be resolved if the actual plastic deformation $\epsilon_{p,t}$ is determined.


 Figure 1: 1-D Rheological Model considering kinematic H and isotropic E_2 hardening (l), stress-strain relation (r)

For each deformation state ϵ , the stress σ and the material tangent stiffness E_p can be easily determined as in Figure 1. In the case of $E_2 = 0$ we would have purely kinematic hardening. If we have $H = 0$, then we will have purely isotropic hardening. In order to illustrate the behavior of the implemented model, the behaviour under cyclic behaviour is shown in for a material with different hardening properties in Figure 2.

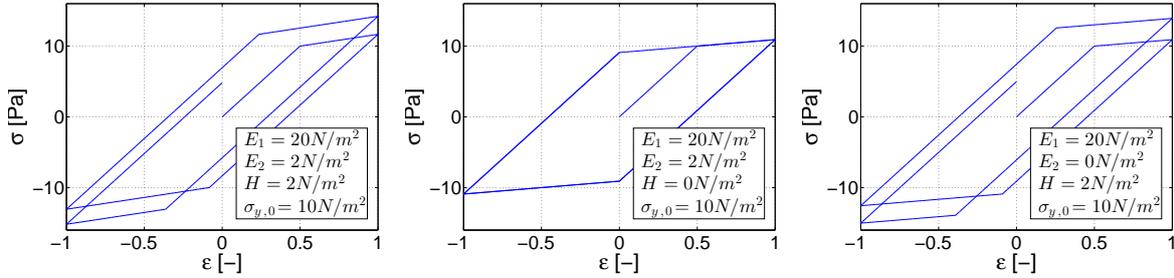


Figure 2: Cyclic behaviour under a) kinematic and isotropic , b)isotropic , c)kinematic hardening

4 SECTION MODELLING

The coupling between cross-sectional geometry and material is necessary in order to evaluate the constitutive behaviour of the cross-section. By assigning the rheological model of the previous chapter to each cell of the meshed geometry, the section will behave non-linearly against axial and bending action. A non-linear behaviour in shear direction is not considered any further, since it is not of significance for slender members as we can find them in typical building structures.

The constitutive behaviour of a cross-section is characterised by the relation between a deformation state $(\epsilon, \gamma, \kappa)$ and its corresponding sectional stresses (N, Q, M) and tangential stiffnesses $(EA_t, GA_{Q,t}, EI_t)$. This parameters are needed in any non-linear Finite Element implementation to calculate the vector of internal forces and the tangent stiffness matrix in the deformed situation.

The searched stress-deformation relation for an arbitrary section is defined by:

$$\begin{aligned} (N, M) &= f(\epsilon, \kappa) \\ Q &= GA_Q \cdot \gamma \end{aligned} \quad (6)$$

Using the discretized section geometry as indicated in Chapter 2.2, we can calculate the deformational state of each single geometry cell i with the expression:

$$\epsilon_i = \epsilon + (y_{G,i} - Na_y) * \kappa; \quad \gamma_i = \gamma; \quad (7)$$

With the methodology described in Chapter 3, we can calculate the stress σ_i and tangential stiffness $E_{p,i}$ at each cell. Returning to the cross-sectional level, the sectional stresses and the sectional tangent stiffnesses can be calculated by the following equations:

$$\begin{aligned} N &= \sum_i \sigma_i \cdot A_i; & EA_t &= \sum_i E_i \cdot A_i; \\ Q &= GA_Q \cdot \gamma; & GA_{Q,t} &= GA_Q; \end{aligned} \quad (8)$$

$$\begin{aligned} M &= \sum_i \sigma_i \cdot (y_{G,i} - Na_y) \cdot A_i + \sum_{i,el} I_{xx,i} \cdot \kappa \cdot E_1; \\ EI_t &= \sum_i E_i \cdot |y_{G,i} - Na_y|^2 \cdot A_i + \sum_{i,el} I_{xx,i} \cdot E_1; \end{aligned} \quad (9)$$

where i, el indicates the cells that do not suffer plastification.

In case of simultaneous action of axial force and bending, the position of the neutral axis Na_y must be determined iterating until force equilibrium is found.

5 EXAMPLE

In this section, the described methodology will be shown with a practical implementation in Matlab of an IPE-100 steel profile. All functions are named after its original implementation in [6] and are freely distributable.

5.1 Definition of the Geometry

The following code lines are valid for the meshing of all types of H- and I-shaped profiles. These profiles are characterised by the parameters h, b, r, t_w, t_f, c_w as indicated in Figure 3.

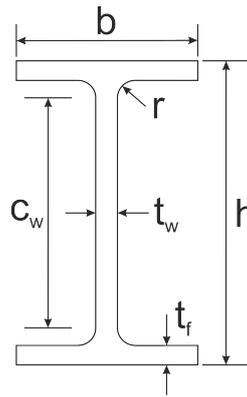


Figure 3: Parameters definition for the geometry description of H-/I-shaped profiles

In a first step, the parameters are used to define rectangles and circles (with *drectangle0* and *dcircle* respectively), which combined through addition (*dunion*) or subtraction (*ddiff*) determine the boundary of the profile. The bounding box *BB*, the fixed points *FP*, and the mesh size

(or discretization parameter) MS are determined according to the profile main dimensions. In this case the mesh size function FH is chosen to be uniform.

```
% Defining geometry:
s = @(p) drectangle0(p, -b/2,b/2,-h/2,h/2);
sd = s;
s = @(p) drectangle0(p, t_w/2,b/2,-c_w/2,c_w/2);
sd = @(p) ddiff(sd(p),s(p)) ;
s = @(p) drectangle0(p, -b/2,-t_w/2,-c_w/2,c_w/2);
sd = @(p) ddiff(sd(p),s(p)) ;
s = @(p) drectangle0(p, t_w/2+r,b/2,-h/2+t_f,h/2-t_f);
sd = @(p) ddiff(sd(p),s(p)) ;
s = @(p) drectangle0(p, -b/2,-t_w/2-r,-h/2+t_f,h/2-t_f);
sd = @(p) ddiff(sd(p),s(p)) ;
s = @(p) dcircle(p, t_w/2+r , c_w/2 , r);
sd = @(p) ddiff(sd(p),s(p)) ;
s = @(p) dcircle(p, t_w/2+r , -c_w/2 , r);
sd = @(p) ddiff(sd(p),s(p)) ;
s = @(p) dcircle(p, -t_w/2-r , c_w/2 , r);
sd = @(p) ddiff(sd(p),s(p)) ;
s = @(p) dcircle(p, -t_w/2-r , -c_w/2 , r);
sd = @(p) ddiff(sd(p),s(p)) ;

% Defining bounding box BB , fixed points FP and mesh size function FH :
BB = [-b/2,-h/2;b/2,h/2];
FP = [-b/2,-h/2; -b/2,h/2; b/2,-h/2; b/2,h/2; -b/2,-h/2+t_f; b/2,-h/2+t_f;
-b/2,h/2-t_f; b/2,h/2-t_f; t_w/2+r,-h/2+t_f; t_w/2+r,h/2-t_f;
-t_w/2-r,h/2-t_f; -t_w/2-r,-h/2+t_f; t_w/2,-c_w/2; t_w/2,+c_w/2;
-t_w/2,-c_w/2;-t_w/2,c_w/2;];
FH = @(p) huniform(p);

% Define discretization factor for mesh size MS depending on chosen profile:
MS = 1;
```

In a next step, the mesh is generated with the function *distmesh2d*. This generates an output of mesh nodes p and mesh triangles t . If n is the number of nodes and m the number of triangles, the matrix p has dimensions $(n,2)$ and t has dimensions $(m,3)$, where each column is a number in the domain $[1,n]$ indicating which nodes compose the triangle. The boundary edges can be determined with the function *boundedges*, which is only necessary in case of performing a detailed section analysis as described in Chapter 2.2.

```
% Generating Mesh:
[p,t]=distmesh2d(sd, FH, MS, BB, FP);
```

5.2 Calculating Cross-Section Properties

With the help of a function to compute area, centroid location and area moments of inertia of the mesh triangles (here *polygeom* from Matlab Central is used), the section properties are calculated as follows:

```
% Now polygeom is used to find mesh triangles properties. These properties
% are saved as vectors (of length the number of triangles) with the
% beginning letter d to indicate it s a mesh triangle property, instead of
% a section property.
% dA Area
% dxG, dyG x- and y-coordinate of centroid
% dIxx, dIyy Area moment of inertia around y- and x-axis
% dIxy Cross Area moment of inertia

% Section Area
A = sum(dA);
% Section Coordinates
xG = sum(dxG.*dA)/A;
```

```

yG = sum(dyG.*dA)/A;

% Section Area Moments of Inertia
Ixx = sum(dA.*(dyG-yG).^2+dIxx);
Iyy = sum(dA.*(dxG-xG).^2+dIyy);
Ixy = sum(dA.*(dxG-xG).*(dyG-yG)+dIxy);

% Determination of Welxx, Welyy:
Welxx = Ixx/max(abs(pY-yG));
Welyy = Iyy/max(abs(pX-xG));

% Determination of Wplxx, Wplyy (after defining the neutral axis Nay and
% Nax in both directions):
Wplxx=sum(abs(dyG-Nay).*dA);
Wplyy=sum(abs(dxG-Nax).*dA);

```

5.3 Rheological Properties

The chosen material properties correspond to those of a standard construction steel S235 [1].

5.4 Section Behaviour and Convergence Analysis

The Behaviour of the section was implemented according to Chapter 4. The behaviour of the section was first tested under cyclic conditions oscillating between $-3 \cdot \kappa_{el}$ and $3 \cdot \kappa_{el}$ repeatedly. The hysteretic cycles can be observed in Figure 4. In a further study, the influence of axial loading on the bending resistance of the cross-section was investigated. In Figure 4 are the results of different $M - \kappa$ curves under different levels of axial loading.

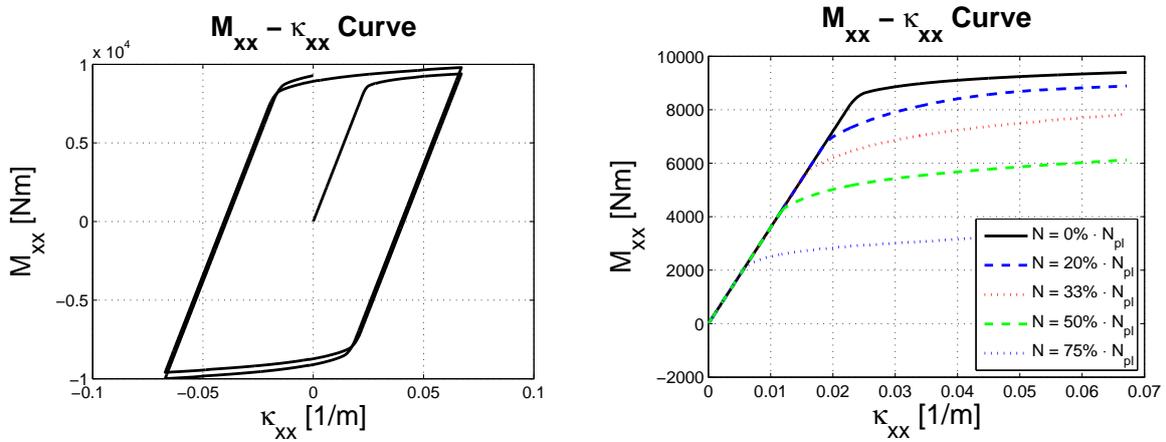


Figure 4: Behaviour of an IPE-100 section under cyclic bending (left) and for different axial loads (right)

In terms of computational speed it is important to work with the minimum number of cells possible, while achieving a sufficient exactness in the results. Therefore an analysis of different mesh discretizations was performed showing that even with a small amount of cells, the non-linear behaviour of the section does not deviate from the exact solution by more than 4 % (see Figure 5).

6 CONCLUSIONS

This paper shows an implementation of a nonlinear rheological model accounting for isotropic and kinematic hardening in an arbitrary cross-section for use in finite element environments working with beam elements. The programming has been performed with Matlab

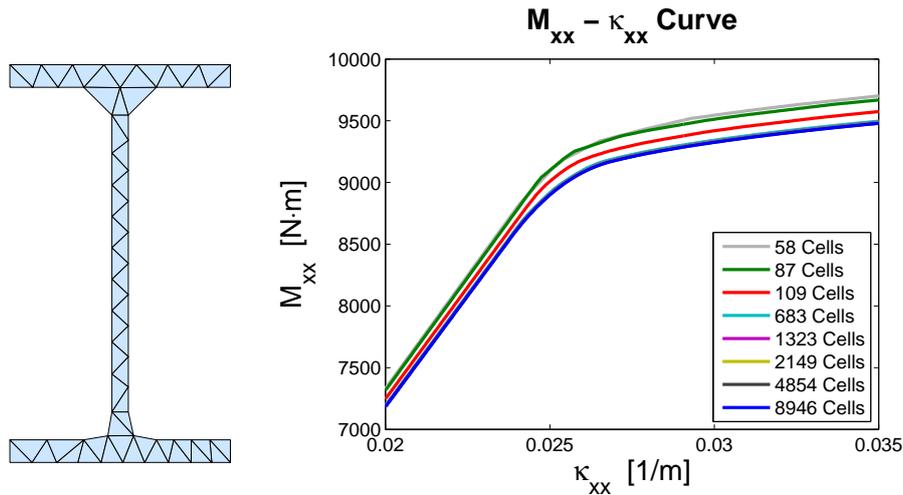


Figure 5: Meshing of an IPE-100 (left), Nonlinear M - κ relation for different number of mesh cells (right)

© using a freely distributable meshing tool and has been proven to describe successfully the nonlinear constitutive behaviour of the section under axial and bending action. The effects of non-linearities in the Q - γ relation have been assumed to be elastic within the scope of this paper, but could easily be extended in order to consider a full M - N - Q interaction.

The implementation has been proven to behave within a 5 % accuracy margin even for quite low resolution discretizations. The computational cost of determining the stress state of a cross-section during a nonlinear finite element calculation is a very repeated process, therefore special attention has been paid to computational speed. The computational cost grows linearly with the number of cell of the discretized section, therefore it is advisable to perform a convergence analysis in order to find the best equilibrium between speed and accuracy of results.

REFERENCES

- [1] EN 1993-1-1. Eurocode 3: Design of steel structures - part 1-1: General rules and rules for buildings;, 2010-12-00.
- [2] F. Gruttmann, R. Sauer, and W. Wagner. Shear stresses in prismatic beams with arbitrary cross-sections. *INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN ENGINEERING*, 45(7):865–889, 1999.
- [3] F. Gruttmann and W. Wagner. Shear correction factors in timoshenko’s beam theory for arbitrary shaped cross-sections. *COMPUTATIONAL MECHANICS*, 27(3):199–207, 2001.
- [4] A. Öchsner and M. Merkel. *One-Dimensional Finite Elements: An Introduction to the FE Method*. Springer, Berlin and Heidelberg, 2013.
- [5] P. O. Persson. Distmesh function reference, <http://persson.berkeley.edu/distmesh/funcref.html>.
- [6] P. O. Persson and G. Strang. A simple mesh generator in matlab. *SIAM REVIEW*, 46(2):329–345, 2004.
- [7] E. Reissner. A one-dimensional finite strain beam theory: the plane problem. *Journal of Applied Mathematics and Physics (ZAMP)*, 1972(23):795–804, 1972.

- [8] J. Rösler, H. Harders, and M. Bäker. *Mechanisches Verhalten der Werkstoffe: Mit 34 Aufgaben mit Lösungen*. Studium. Vieweg + Teubner, Wiesbaden, 3 edition, 2008.
- [9] P. Wriggers. *Nonlinear Finite Element Methods*. Springer Berlin Heidelberg, Berlin and Heidelberg, 2008.