“Implementation of the Static Modal Analysis in MatLab for 2D frames”

Luis Fernando Verduzco Martínez [1]

[1] luiz.verduz@gmail.com

Faculty of Engineering, Autonomous University of Querétaro, Santiago de Querétaro, Mexico

Overview

This software is the implementation of the Static Modal Analysis for 2D frames. The analysis is based on the *Consistent mass method* in which all possible modes of vibration of a plane frame are computed by solving the eigenvector and eigenvalue equation $det(K - \omega^2 M) = 0$ in which damping is neglected. The equivalent inertial forces are calculated for all the DOF with respect to one or more than one modes of vibration. When more than one modes of vibration are given to compute such equivalent forces, then the superposition of all such given modes of vibration is computed. A design spectrum of constant acceleration at the base of the structure is considered. For the computation of the Mass matrix not only the mass of each element is considered but also the external uniformly distributed loads in the vertical direction of the beam (if any).

Keywords: Static Modal Analysis, Plane Frames, Design spectrum, Constant acceleration, Consistent mass method

1 The Static Modal Analysis

Due to the ground's movement at the base of buildings that an earthquake can generate great inertial forces can be induced to them. The flexibility of a structure, in function of its mass and shape/topology makes a structure to vibrate in a certain frequency usually different than the soil's on which it is supported, so that the induced inertial forces acting on the structure depend directly on its dynamic properties.

According to the original MDOF dynamic system equation (1) where K is the stiffness matrix, M the mass matrix and C is the damping matrix, when performing a modal static analysis where no damping is considered for the structure the dynamic system equation is transformed to (2) whose solution is given by (3):

$$[M]\{\ddot{U}\} + [C]\{\dot{U}\} + [K]\{U\} = \{F(t)\} \tag{1}$$

$$[M]\{\ddot{U}\} + [K]\{U\} = \{0\} \tag{2}$$

$$\{U\} = \{\dot{U}\}cos(\omega t) \tag{3}$$

Thus, when derivating U with respect to ω and substituting in (2) the only unknown becomes ω

and could determined by solving the system (4), representing the structure frequencies for each mode of vibration:

$$det([K] - \omega^2[M]) = \{0\} \tag{4}$$

The previous system can be solved by assimilating the problem to a eigenvalue and eigenvector problem in the form $[A]\phi = \lambda\phi$ as (5), where ϕ would represent the modes of vibration of the structure for each frequency:

$$([K] - \omega^2[M])[\phi] = \{0\} \tag{5}$$

1.1 The consistent mass method

The construction of the mass matrix is done through a Finite Element method approach in which an element mass matrix (6) is built for all elements and assembled into a global one as for the stiffness matrix.

$$M_e = \frac{\gamma AL}{420g}\begin{bmatrix} 440 & 0 & 0 & 70 & 0 & 0 \\ 0 & 156 & 22L & 0 & 54 & -13L \\ 0 & 22L & 4L^2 & 0 & 13L & -3L^2 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13L & 0 & 156 & -22L \\ 0 & -13L & -3L^2 & 0 & -22L & 4L^2 \end{bmatrix} \tag{6}$$

2 Design response spectrum

A response spectrum represents the max response of a SDOF oscillator of unitary mass subject to a given base movement measured by an accelerometer which is then graphed for various vibration period of the oscillator itself. This such response can be expressed either by displacements, velocities or max accelerations. Given the long-time process of calculations required to obtain these such response spectrums it is common in structural engineering to simplify such response by a *pseudo-spectrum* so that an acceleration response would then be transformed to a *pseudo-acceleration* response, as in **Fig. 1**.

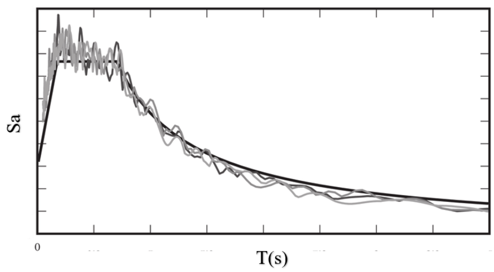


Figure 1. Typical pseudo-spectrum of acceleration.

Thus, with a pseudo response spectrum acceleration Sa that may indicate the max impose accelerated movement at the base of a structure the equivalent inertial forces of the structure's DOF can be computed as (7), where $M* = \phi_i^T M \phi_i$, being ϕ the eigenvectors and M the mass matrix:

$$f_{max} = \frac{\phi_i^T M[1]}{M*} Sa(M\phi_i) \tag{7}$$

3 Illustrative examples

3.1 Problem

Let us consider the structural topology shown in **Fig. 2** for which it is required to assess its structural response before the design seismic spectrum with the local Mexican CFE-15 [1] code for the first 10 modes of vibration.

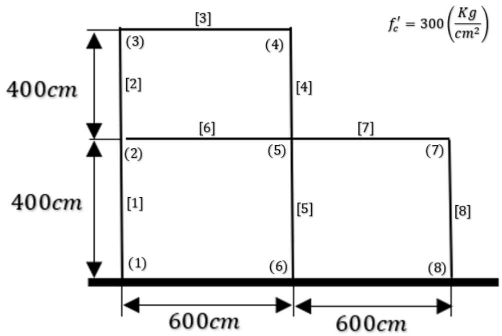


Figure 2. Structural topology of study.

3.2 Solution

The materials and geometry properties of each of the frame's elements must be first given, as:

```
nnodes=8;
nbars=8;

%% Materials
fpc=[300;
     300;
     300;
     300;
     300;
     300;
     300;
     300];

% Modulus of Elasticity of each element
E=14000*(fpc).^0.5;
```

```
%% Geometry
dimensions=[30 30;
            30 30;
            25 50;
            25 50;
            30 30;
            40 40;
            25 50;
            30 30];

A=dimensions(:,1).*dimensions(:,2);
I=1/12.*dimensions(:,1).*dimensions(:,2).^3;

% Coordinates of each node for each bar
coordxy=[0 0;
         0 300;
         0 600;
         600 0;
         600 300;
         600 600;
         900 0;
         900 300];
```

The node connectivity and topology, in general, are given now as:

```
%% Topology
% Node conectivity
ni=[1;2;3;2;4;5;5;7];
nf=[2;3;6;5;5;6;8;8];

% Length of each element
L=sqrt((coordxy(nf,1)-coordxy(ni,1)).^2+(coordxy(nf,2)-coordxy(ni,2)).^2);

% Topology matrix
Edof=zeros(nbars,7);
for i=1:nbars
    Edof(i,1)=i;
    Edof(i,2)=ni(i)*3-2;
    Edof(i,3)=ni(i)*3-1;
    Edof(i,4)=ni(i)*3;

    Edof(i,5)=nf(i)*3-2;
    Edof(i,6)=nf(i)*3-1;
    Edof(i,7)=nf(i)*3;
end
```

Now, it is consider that the supports at the frame base are fixed, therefore, the boundary condition array *bc* as following:

```
%% Boundary conditions
% Prescribed DOF - [N-DOF, displacement]
bc=[1 0;
    2 0;
    3 0;
    10 0;
    11 0;
    12 0;
    19 0;
    20 0;
    21 0];
```

When distributed loads in the elements are present, then such loads will be considered as a portion of the elements' mass for the computation of the Mass matrix. For this purpose, an array $qbarxy$ is created, in which the uniformly distributed loads with respect to each element's local axis are given. The following piece of code presents a way to set this load array, in which the label *Beam/Col* are first assigned manually to each element and then a uniform load of $50Kg/cm$ in the gravity direction are set:

```
%% Loads

type_elem=[1 "Col";
           2 "Col";
           3 "Beam";
           4 "Beam";
           5 "Col";
           6 "Col";
           7 "Beam";
           8 "Col"];

% Distributed Loads on beams
beamsLoads=[1 -50;
           2 -50;
           3 -50];

elem_cols=[];
elem_beams=[];
beams=0;
cols=0;
for j=1:nbars % To identify which elements are beams and which are columns
    if type_elem(j,2)=="Beam"
        beams=beams+1;
        elem_beams=[elem_beams,j];
```

```
    elseif type_elem(j,2)=="Col"
        cols=cols+1;
        elem_cols=[elem_cols,j];
    end
end
qbarxy=zeros(nbars,2);
for i=1:beams
    qbarxy(elem_beams(i),2)=1.1*(beamsLoads(i,2));
end
```

Note that another of way of setting such load array could be simply as:

```
%% Loads

qbarxy=[0 0;
        0 0;
        0 -50;
        0 -50;
        0 0;
        0 0;
        0 -50;
        0 0];
```

Now, it would be required to establish the mode(s) of vibration of interest for the assessment of the structure's response. In case more than one mode is given, the equivalent inertial forces will be computed from the superposition of such modes as (8), where fm_i is computed as (7):

$$f_{max} = \sqrt{\sum_{1}^{n-modes} fm_i^2} \tag{8}$$

The CFE-15 establishes that for structures in the Group B2 (in which the present structure example lies in) must be designed with a constant ground acceleration spectrum a_o^τ. According to its seismic regionalization map for zones of *Very high* seismic intensity, a minimum ground acceleration of $a_0^\tau = 200(cm/s^2)$ must be used and factorized by a *Site factor* F_s and by a *Response factor* F_{res} according to the ground type. Also, the ground acceleration must be factorized by a ductility factor Qp and an over-resistance factor R, according to the classification of the structure and its geometry/topology. Therefore, the response spectrum for this example would be as following:

```
%% Seismic response spectrum from the CFE-15

g=981; % gravity acceleration
Fsit=2.4; FRes=3.8; % Factores de sitio y de respuesta
a0_tau=200; % cm/seg^2
```

```
ro=0.8; % Redundance factor
alf=0.9; % Irregularity factor
Q=4; % Seismic behaviour factor

Ta=0.1;
Tb=0.6;
Te=0.5; % Structure's period
k=1.5; % Design spectrum slope
Qp=1+(Q-1)*sqrt(Te/(k*Tb)); % Ductility factor

Ro=2.5; % Over-resistance index
R=Ro+1-sqrt(Te/Ta); % Over-resistance factor

sa=a0_tau*Fsit*FRes/(R*Qp*alf*ro); % Reduced pseudo-acceleration (cm/seg^2)
```

Note that the user should feel free to modify this such spectrum to his/her own needs.

Finally, the static modal analysis takes place, from which then, a static linear analysis is performed with the obtained equivalent inertial forces, as:

```
%% Modal analysis
pvconc=0.0024; % unit weight of concrete (Kgf/cm3)
unitWeightElm=zeros(nbars,1)+pvconc;

% Modal analysis with the "consistent mass method"
[fmaxDOF,Mglobal,Kglobal,T,La,Egv]=SeismicModalMDOF2DFrames2...
(coordxy,A,unitWeightElm,qbarxy,Edof,bc,E,I,ni,nf,sa,g,modal);

% Considering the equivalent seismic loads for a structural analysis
fglobal=fmaxDOF;

%% Static structural analysis with seismic forces
np=7; % number of analysis points for the mechanical elements

[Ugl1,reactions,Ex,Ey,esbarsnormal,esbarsshear,esbarsmoment]=...
StaticLinearAnalysis2DFrame(E,A,I,bc,fglobal,ni,nf,qbarxy,np,coordxy,...
1,1,6,6);
```

When only one mode of interest was entered, then the following piece of code will plot such mode of vibration and its respective frequency:

```
%% Plot of the modal in question and its frequency
Freq=1./T;

if length(modal)==1 % If only one modal was entered
```

```
    figure(6)
    grid on
    NoteMode=num2str(modal);
    title(strcat('Eigenmode ','- ',NoteMode))
    eldraw2(Ex,Ey,[2 3 1]);
    Edb=extract(Edof,Egv(:,modal));
    eldisp2(Ex,Ey,Edb,[1 2 2]);
    FreqText=num2str(Freq(modal));
    NotaFreq=strcat('Freq(Hz)= ',FreqText);
    text(50,10,NotaFreq);
end
```

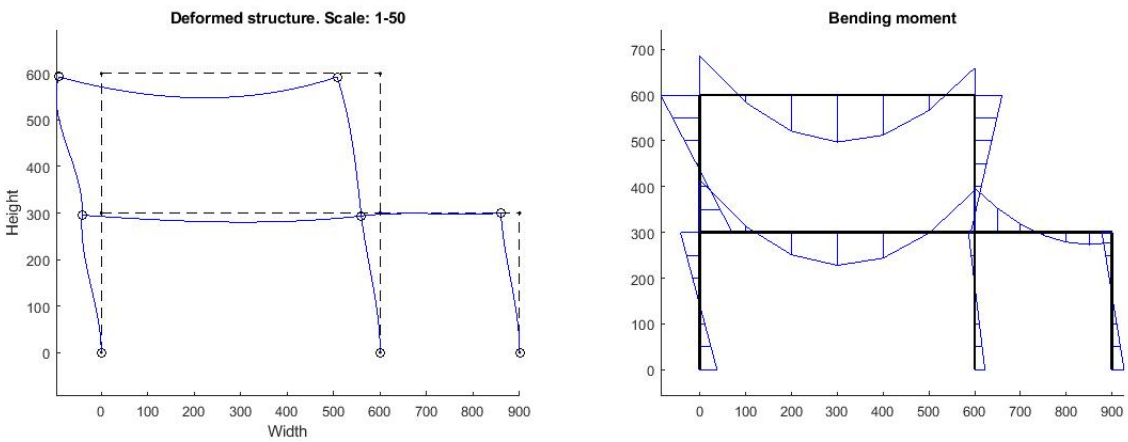
At last, the structural displacement response for each of the 10 first modes of vibration are plot with the following code, in which for each loop iteration a modal analysis and a linear structural analysis are performed:

```
%% Plot of the seismic structural reponse for the first 10 modals
widthstruc=max(coordxy(:,1));
heightstruc=max(coordxy(:,2));

figure(7)
axis('equal')
axis off
title(strcat('Deformed structures against the seismic actions for the',...
    ' first 10 EigenModes'))
hold on

% First 5 modals at the top of the plot
for i=1:5

    % Modal static analysis - Consistent mass method
    [fmaxDOF,Mglobal,Kglobal,T,La,Egv]=SeismicModalMDOF2DFrames2...
    (coordxy,A,unitWeightElm,qbarxy,Edof,bc,E,I,ni,nf,sa,g,i);

    % Static structural analysis with seismic forces
    [Ugl,reactions,Ex,Ey,esbarsnormal,esbarsshear,esbarsmoment]=...
    StaticLinearAnalysis2DFrame(E,A,I,bc,fmaxDOF,ni,nf,...
    qbarxy,np,coordxy,0);

    Ext=Ex+(i-1)*(widthstruc+150);
    eldraw2(Ext,Ey,[2 3 1]);
    Edb=extract(Edof,Ugl);
    eldisp2(Ext,Ey,Edb,[1 2 2]);
    FreqText=num2str(Freq(i));
    NotaFreq=strcat('Frec(Hz)= ',FreqText);
```

```
    text((widthstruc+150)*(i-1)+50,150,NotaFreq)
end

% Last 5 modals at the bottom of the plot
Eyt=Ey-(heightstruc+200);
for i=6:10
    % Consistent mass method
    [fmaxDOF,Mglobal,Kglobal,T,La,Egv]=SeismicModalMDOF2DFrames2...
    (coordxy,A,unitWeightElm,qbarxy,Edof,bc,E,I,ni,nf,sa,g,i);

    %% Static structural analysis with seismic forces
    [Ugl,reactions,Ex,Ey,esbarsnormal,esbarsshear,esbarsmoment]=...
    StaticLinearAnalysis2DFrame(E,A,I,bc,fmaxDOF,ni,nf,...
    qbarxy,np,coordxy,0);

    Ext=Ex+(i-6)*(widthstruc+150);
    eldraw2(Ext,Eyt,[2 3 1]);
    Edb=extract(Edof,Ugl);
    eldisp2(Ext,Eyt,Edb,[1 2 2]);
    FreqText=num2str(Freq(i));
    NotaFreq=strcat('Frec(Hz)= ',FreqText);

    text((widthstruc+150)*(i-6)+50,-heightstruc,NotaFreq);
end
```

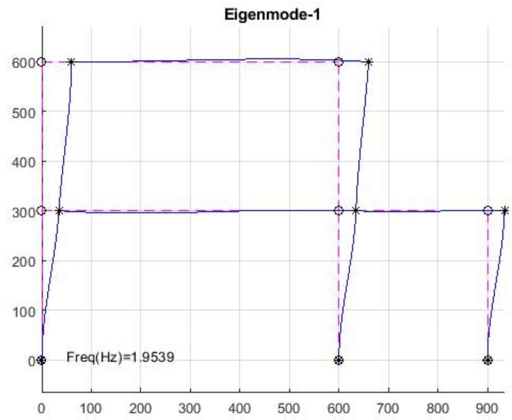
3.3 Results

With the previous parameters and by setting only the first mode of vibration, the deformed structure against the equivalent seismic actions and its respective bending moment diagrams is shown in the following figure:

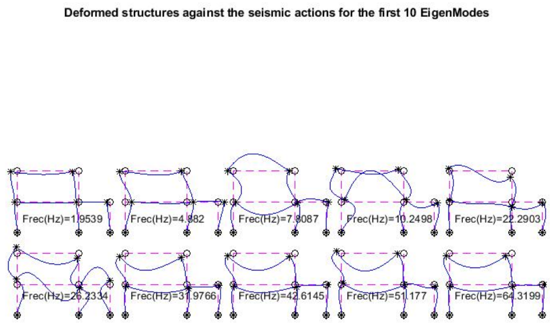


Note that in this case the ground acceleration was given as positive stating that the ground is moving to the right, therefore, the inertial forces act on the opposite direction.

The corresponding mode of vibration is shown in the following figure (adimensional):



At last, the structural displacement response for each of the ten first modes of vibration are plot as:



References

[1] Comisión Federal de Electricidad CFE-2015, Manual de Diseño de Obras Civiles, Capítulo C.1.3 Diseño por sismo, México, 2015. Spanish version.