Chapter 7: Solving Structural Dynamic Problems Using DCALC

By Karl Hanson, S.E., P.E.*
September 2008

7.1 Introduction:

The last chapter presented the fundamental methods used in structural dynamics to design various structures subject to forces and support movements. This chapter will discuss how to use DCALC to solve problems of this type.

Structural dynamics encompasses a variety of dynamic problems that structural engineers deal with: machine foundations, beam vibrations, vortex wind-induced oscillations and seismic design. Seismic design is the most frequent dynamic problem that structural engineers encounter in practice. After becoming familiar with modal analysis, you will be able to recognize that the underlying methodology imbedded in building code seismic design code equations are based on fundamental structural dynamic methods.

DCALC's structural dynamic applications place an emphasis on solving dynamic problems using the fundamental methods of structural dynamic theory. An emphasis is placed on how to disassemble a MDOF system into multiple SDOF systems, using modal frequencies and shapes. The purpose of this chapter will explain the steps used in this simple methodology, which can be applied to a wide range of dynamic problems.

7.2 The Methodology:

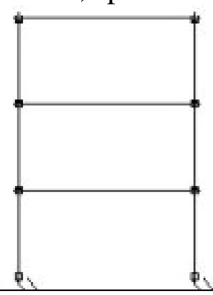
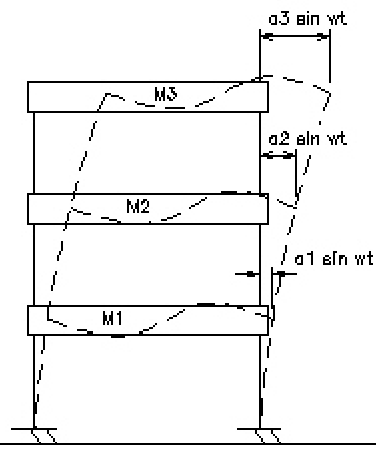
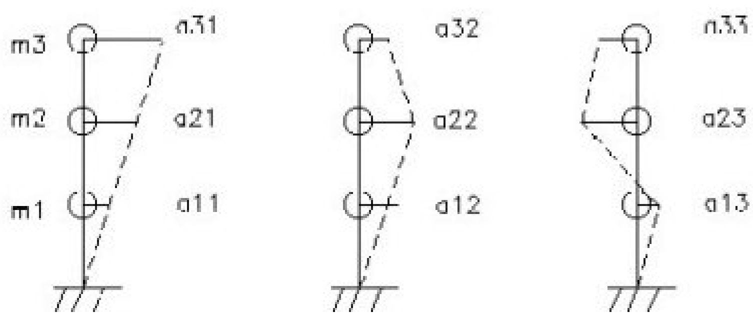
There are several ways to solve structural dynamic problems, varying from classical modal analysis to non-linear numerical approaches. We will focus on the traditional elastic modal spectral approach methods in this chapter .

Let's revisit some of the key points described in Chapter 6:

- Structures are like musical instruments: They have a fundamental frequency and "overtones".
- Each frequency has a modal shape. Modal shapes are the ratios of displacements of the nodes. It is a "shape without dimensions".
- For each modal shape, the deflection of one mass determines the deflections of all masses, because the ratios of the displacements are fixed. By knowing one deflection, we can solve for all deflections.
- We are able to separate a MDOF system into several SDOF systems because of a property called "orthogonality" of normal modals. Othogonality is the key mathematical device used in modal analysis.
- We analyze the separate SDOF systems using modal properties.
- The overall deflection is determined by the superposition of deflections computed for each frequency.

* DesignCalcs, Inc. (http://dcalc.us)

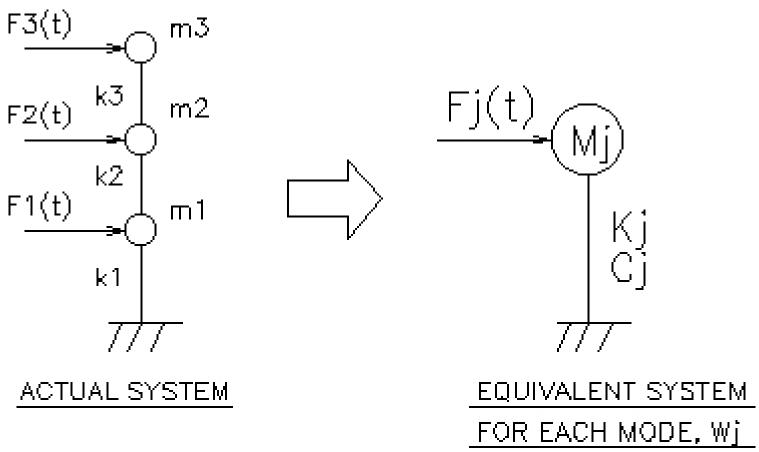
7.2.1 Methodology: Solving for Natural Frequencies and Modal Shapes

<table>
<tr><th colspan="2">Steps for Determining Natural Frequencies and Modal Shapes</th><th>DCALC Program</th></tr>
<tr><td>1</td><td>Model a structure as a plane frame, space frame or plane grid
</td><td>FRAME</td></tr>
<tr><td>2</td><td>Assign lumped masses to the model.

</td><td rowspan="3">DYNAMIC</td></tr>
<tr><td>3</td><td>Compute the natural frequencies and modal shapes. The number of natural frequencies is the product of the number of masses times the degrees of freedom.


Mode 1 Mode 2 Mode 3</td></tr>
<tr><td>4</td><td>Compute normalized shape factors using the following equation:
$\phi_{ij} = a_{ij} / (\Sigma\, m_k (a_{kj})^2)^{1/2}$</td></tr>
</table>

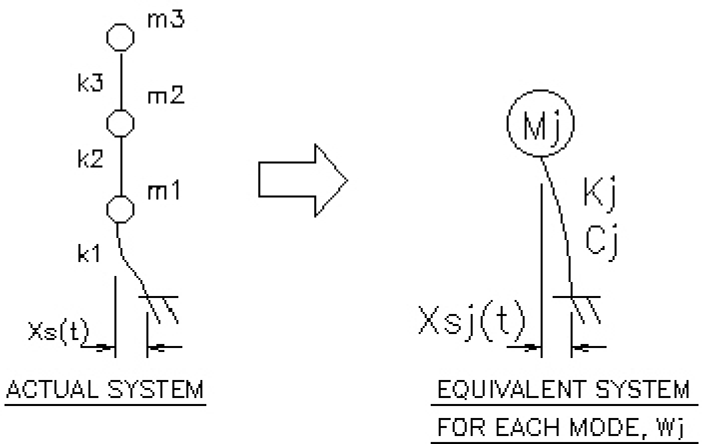
At this point in the process, after running “FRAME” and “DYNAMIC”, you will have solved for the natural frequencies and modal shapes of the structure. Having solved the natural frequency and modal shapes, the difficult part is finished!

7.2.2 Methodology: Solving Response Due to a Force Applied To Structure

Steps for Solving Response Due to a Force Applied to Structure		DCALC Program
1	For each mode “j”, compute modal properties:  • Modal mass: $M_j = \Sigma m_i \phi_{ij}^2$ (result will be $M_j=1$) • Modal stiffness: $K_j = \omega_j^2 M_j (=\omega_j^2 * 1 = \omega_j^2)$ • Modal damping: $C_j = 2\varepsilon_i \omega_i$ (where $\varepsilon_i=c_i/c_{i, cr}$)	MODAL
2	For each mode “j”, compute modal force $F_j(t)$: Modal force: $F_j = \Sigma \phi_{ij} F_i$	
3	For each mode, compute the response, X_j of the SDOF system to $F_j(t)$: • For sinusoidal forces, use closed-formed solutions to the differential equation • For linear-numerically defined forces, use Duhamel’s integral	
4	For each mode, compute the multi-degree-of-freedom responses at each mass: $x_{ij} = \phi_{ij} X_j$	
5	For each mode, compute the equivalent static force at each mass: $F_{static, ij} = \omega_j^2 x_{ij}$	
6	The maximum combined response at each mass is typically computed by, $x_{imax} = (\Sigma (x_{ij\,max})^2)^{1/2}$	

The DCALC program “MODAL” was designed to read the “DYNAMIC” output and automatically perform the above steps. “FRAME” is designed to read the “MODAL” output. It is important that the engineer appreciate the concept of disassembling the MDOF system into SDOF systems, and reassembling the individual responses to get the actual MDOF response.

7.2.3 Methodology: Solving Response Due to General Support Movement (Non-Code Described Response)

Steps for Solving Response Due to General Support Movement (Non-Code Described Response)		DCALC Program
1	For each mode "j", compute modal properties:  • Modal mass: $M_j = \Sigma m_i \phi_{ij}^2$ (result will be M_j=1) • Modal stiffness: $K_j = \omega_j^2 M_j$ $(=\omega_j^2 * 1 = \omega_j^2)$ • Modal damping: $C_j = 2\varepsilon_i \omega_i$ (where $\varepsilon_i = c_i/c_{i, cr}$) • Mass participation, $\Gamma_j = \left(\sum_{k=1}^{n} m_k a_{kj}\right) \div \left(\sum_{k=1}^{n} m_k (a_{kj})^2\right)$	MODAL
2	For each mode "j", compute modal force $F_j(t)$: Modal Force, Fj (t) = $-\Gamma_j x_s''$ (to be applied to mass, M_j)	
3	Compute the SDOF response, X_j: • For sinusoidal forces, use closed-formed solutions to the differential equation • For linear-numerically defined forces, use Duhamel's integral	
4	For each mode, compute the multi-degree-of-freedom drift response at each mass: $u_{ij} = \phi_{ij} X_j$	
5	For each mode, compute the equivalent static force at each mass: $F_{static, ij} = \omega_j^2 u_{ij}$	
6	The maximum combined drift response at each mass is typically computed by, $u_{imax} = (\Sigma (u_{ij\ max})^2)^{1/2}$	

Again, the DCALC program "MODAL" was designed to read the "DYNAMIC" output and automatically perform the above steps.

7.2.4 Methodology: Solving Response Due to Seismic Movement (Code Described Response)

The seismic analysis of buildings and bridges differs from the analysis of structures that are subjected to well defined movements. Earthquakes vary in size, are random in frequency and intensity. We also can't predict the direction of an earthquake.

Let's revisit the discussion of Dr. Nathan Newmark's work, mentioned in Chapter 6:

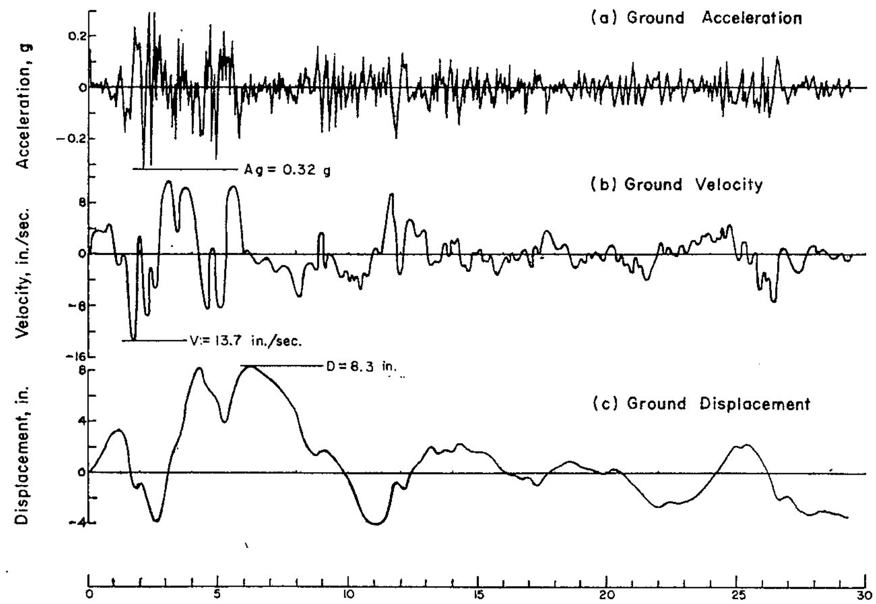


Fig. 8: El Centro, California, earthquake of May, 18, 1940

Professor Nathan Newmark (University of Illinois)

Response of damped single degree of freedom system to general support motion, relative to support (Duhamel's integral):

$$u = x - x_s = \Sigma\ e^{-\varepsilon\omega(t-ti)}(-1/\omega_D)x_s''\ \Delta t\ \sin\omega_D(t-t_i)$$

where x_s'' = support (ground) acceleration.

In the 1950's, Newmark computed the response of various SDOF systems to the El Centro earthquake using the above equation. Today, in the United States, earthquake hazards fall within the jurisdiction of the United States Geological Survey (USGS). The USGS has computed **consolidated response spectrum** based on voluminous earthquake data records.

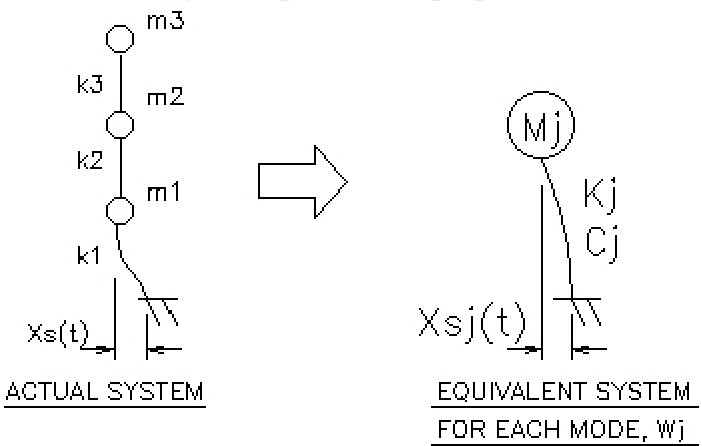
Typically, building codes provide spectral acceleration response formulas that are based on these studies. For example, in the former BOCA code, the response acceleration was given by,

$$C_s=1.2*A_v*S/(R*T^{2/3})$$

Today, the IBC and AASHTO codes use different formulas for acceleration; however the shape is somewhat the same. Therefore, **when making a seismic analysis, the response of SDOF systems is prescribed by the building codes** (see "Step 2" in following table).

7.2.4 Methodology: Solving Response Due to Seismic Movement (Code Described Response)

<table>
<tr><th colspan="2">Steps for Solving Response Due to Seismic Movement
(Code Described Response)</th><th>DCALC Program</th></tr>
<tr><td>1</td><td>For each mode "j", compute modal properties:



Modal mass: $M_j = \Sigma m_i \phi_{ij}^2$ (result will be $M_j=1$)
Modal stiffness: $K_j = \omega_j^2 M_j$ $(=\omega_j^2 * 1 = \omega_j^2)$
Modal damping: $C_j = 2\varepsilon_i \omega_i$ (where $\varepsilon_i=c_i/c_{i, cr}$)
Mass participation, $\Gamma_j = \left(\sum_{k=1}^{n} m_k a_{kj}\right) \div \left(\sum_{k=1}^{n} m_k \left(a_{kj}\right)^2\right)$
</td><td rowspan="6">MODAL</td></tr>
<tr><td>2</td><td>For each mode "j", compute the spectral acceleration response, Sa, prescribed by the design specification (i.e., AASHTO, IBC). Refer to design specifications for the seismic acceleration response equation, which varies between specifications.</td></tr>
<tr><td>3</td><td>For each mode, compute the SDOF spectral displacement response, Xj:
$X_j = \Gamma_j * S_a / \omega_n^2$</td></tr>
<tr><td>4</td><td>For each mode, compute the multi-degree-of-freedom drift response at each mass:
$u_{ij} = \phi_{ij} X_j$</td></tr>
<tr><td>5</td><td>For each mode, compute the equivalent static force at each mass:
$F_{static, ij} = \omega_j^2 u_{ij}$</td></tr>
<tr><td>6</td><td>The maximum combined response at each mass is typically computed by,
$u_{imax} = (\Sigma (u_{ij\,max})^2)^{1/2}$</td></tr>
</table>

7.2.4 Methodology: Solving Response Due to Seismic Movement (Code Described Response)

After you become familiar with the modal methodology, you will realize that there is a great deal of similarity between several types of dynamic problems. The previous tables are very similar, only differing in certain steps.

With practice, you may also be able to glean that certain building code equations appear to resemble aspects of the modal methodology. You may see that certain code formulas appear to resemble things like mass participation factors (Γ_j) and modal shape factors (ϕ_{ij}). It may require some mathematics on your part to derive these relationship, but the relationships are apparent.

For example, let's focus on the old 1993 BOCA code, and how one part of the code relates to modal methodology:

1993 BOCA code	Modal Analysis Equivalent
1612.5.5 Modal base shear: The portion of the base shear contributed by the m^{th} mode (V_m) shall be determined from the following formula: $V_m = C_{sm} W_m$ where: C_{sm} = The modal seismic design coefficient determined by the following formula. W_m = The effective modal gravity *load* determined by the following formula. $W_m = \dfrac{\left[\sum_{i=1}^{n} w_i \phi_{im}\right]^2}{\sum_{i=1}^{n} w_i \phi_{im}^2}$ where: w_i = The portion of the total gravity *load* of the building at level *i*. ϕ_{im} = The displacement amplitude at the i^{th} level of the building when vibrating in its m^{th} mode.	Determining base shear in terms of classical modal analysis: The modal acceleration at a mass node "i" in mode "m", is, $x_i'' = \phi_{im}\ \Gamma_m * S_a * g$ Where mass participation is $\Gamma_m = \left(\sum_{i=1}^{n} w_i \Phi_{im}\right) \div \left(\sum_{i=1}^{n} w_i (\Phi_{im})^2\right)$ The force on each mass at each level is, $F_i = m_i * x_i''$ $= w_i/g * x_i''$ The total force at the base is, $\Sigma F_i = \Sigma w_i/g * x_i''$

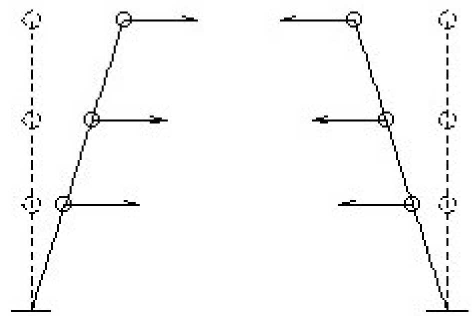
After some manipulation of the classical modal approach equations, it will be recognized that the BOCA equation for base shear, V_m , is the same as ΣF_i. The terms and symbols may be different, but the ideas are the same.

The point of this diversion is that present seismic codes are derived from modal methodology. If you can take the time to recognize relationships like these between the code equations and modal methodology, the significance of the code equations can be better appreciated.

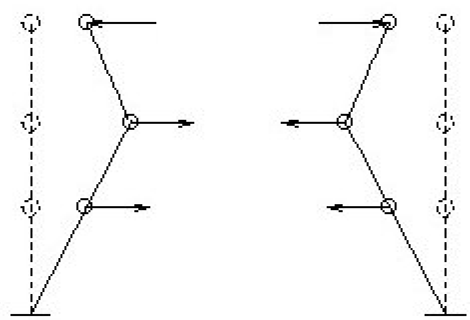
7.2.5 Methodology: Determining Maximum Force Effects:

After completing the modal analysis to determine the maximum response of a structure to a dynamic loading, the next task is to determine what are the maximum force effects in the structure. It must be kept in mind that the maximum responses can oscillate backward and forward, changing in direction (although this isn't always the case). Therefore, the pseudo-static forces that will produce the maximum responses can also change directions.

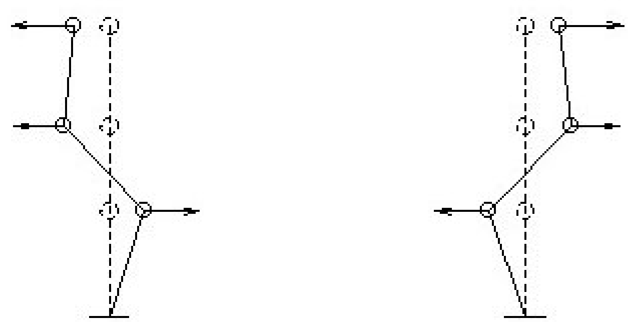
The below generic sketches show modal "positive" static forces/displacements and "negative" static forces/displacement:



Mode 1: Positive direction … Negative direction



Mode 2: Positive direction … Negative direction



Mode 3: Positive direction … Negative direction

Keeping in mind that each mode occurs at a different rate, we need to ask which of the directions combine to cause the maximum force effect that we are studying. For example, there are many combinations to look at: Modes 1+2+3, 1-2+3, 1-2-3, -1+2+3… We need to determine what force effect (example: a column moment), needs to be maximized.

7.3 DCALC's Structural Dynamic Applications:

DCALC has four structural dynamic applications, which reflect the same methodical process that is used in classical structural dynamic theory:

1. OSCILATE: Computes the response of a single-degree-of- freedom (SDOF) system subjected to either a forced excitation or a support movement. This program "goes back to the basics", using both closed form solutions to differential equations and numerical approaches. The output is detailed and referenced, showing step-by-step solutions. The user may want to use this program to check the modal solutions in a multi-degree-of-freedom system.

2. DYNAMIC: Solves for the natural frequencies and modal shapes of plane frames, space frames and grids. Two methods can be used to solve for the natural frequency: Rayleigh's Method (approximate) and Multi-Modal Analysis.

   The Multi-Modal Analysis method is a mathematical problem of a type called an "eigenproblem" (refer to Chapter 6 for detailed discussion). The "DYNAMIC" program uses software code originally written in Fortran, by Bathe and Wilson in the 1970's (Ref. "Numerical Methods In Finite Element Analysis", published by Prentice and Hall, Inc., 1976). This program also uses static condensation procedures originally written in Fortran, by Mario Paz, in his excellent text (Ref. "Structural Dynamics, Theory and Computations", published by Van Nostrand Reinhold Company, 1980).

3. MODAL: Computes the response of a multi-degree-of-freedom (MDOF) system subjected to either a forced excitation or support movement. This program is based on methods described in Dr. Mario Paz's book.

4. FRAME: This structural analysis program is used in the beginning and end of the process, to generate the model (read by DYNAMIC), and, finally, to determine forces in the frame (reading MODAL's static force output).

7.4 Example Problems:

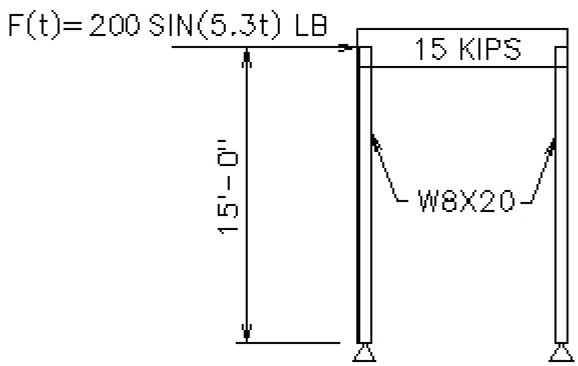
In this next section, several example problems are presented. The purpose of showing a fair number of examples is two-fold:

- To show how structural dynamic problems are formulated and what the results look like. The user is encouraged to also run these example problems. The process will soon become easy with practice.
- To compare DCALC's solutions to published examples, for verification purposes.

Fortunately, Dr. Paz's books have many examples that can be used for testing purposes. His first edition , published in 1980, included FORTRAN code for many programs that Dr. Paz had written. In later editions of his book, he also included many examples using SAP 2000.

7.4.1 Example Problem 1a (*):

The following problem appears in "Structural Dynamics", by Mario Paz, as Illustrative Example 3.2. The below frame supports a rotating machine that exerts a horizontal force at the girder level. 5% critical damping is assumed.



We will solve this using the "OSCILATE" program, modeling the frame as a simple oscillator.

In order to make this simplification, we will need to compute the lateral stiffness of the columns. Treating the two W8's as pinned on the bottom and fixed on top, the lateral stiffness of the system is,

$$k = 3E(2I)/L^2 \;= 3*30{,}000{,}000*2*69.2/(12*15)^3 = 2136 \text{ lb/in}$$

Comparison of Results:

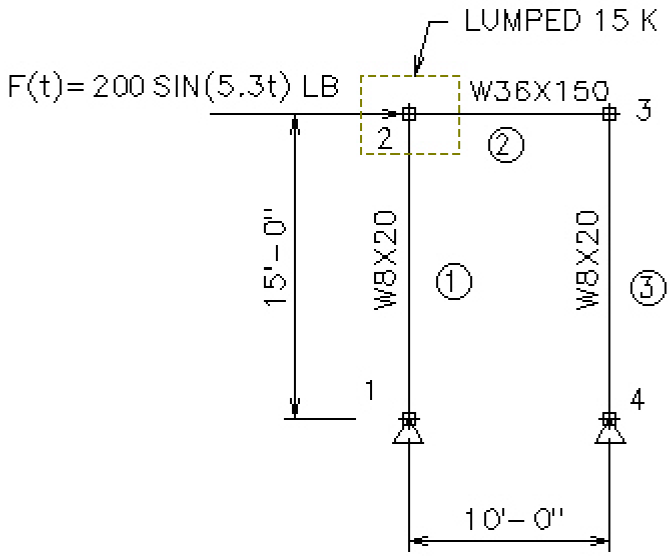
Published Results	DCALC Results
Natural frequency of system, ω_n = 7.41 rad/sec Steady state response, x = 0.189 in	Natural frequency of system, ω_n = 7.417 rad/sec Steady state response, x = 0.18 in

* Note: DCALC input and output files for these examples can be downloaded from http://dcalc.us/Downloads/DYNAMIC.zip

- If you are a licensed DCALC user, you will need to create a new directory, then unzip the files into the new directory.
- If you are not a licensed DCALC user and you are trying out the DCALC demo, you will need to unzip the files into the C:\DCALC\DEMO directory (however, this will render all other DEMO calculations useless).

7.4.2 Example Problem 1b

We will solve the same problem as example problem 1a, but using a different method. Rarely do we encounter structures that can be modeled as simple oscillators. For real world structures, we need to model a structure as an assemblage of elements.



The above frame shows a modeled representation of the previous problem. For the stiff beam element, we have arbitrarily chosen a W36x150. We have lumped the 15 kips onto Joint 2, rather than distributing the 15 kips over 10 feet, which is an equally valid approach.

We will prepare this analysis running the following order of DCALC programs:

1. Input the above model in FRAME. Save the input. (No need to analyze the frame)
2. Run DYNAMIC, retrieving the FRAME input file
3. Run MODAL, retrieving the DYNAMIC output file
4. Run FRAME. Retrieve the original input file. Add static forces from MODAL as a load case. Run the analysis to compute axial forces, moments and shears.

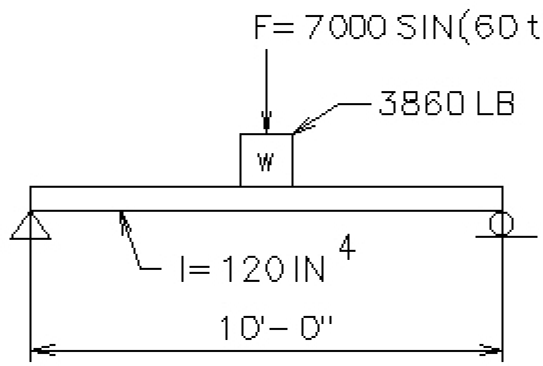
You will see that the above process is quick, because the programs are designed to read the previous programs results.

Comparison of Results:

Published Results	DCALC Results
Natural frequency of system, ω_n = 7.41 rad/sec Steady state response, x = 0.189 in Equivalent static force, F_{static} = 0.403 k Maximum being moment in columns, M_{max} = 36.324 k*in	Natural frequency of system, ω_n = 7.500 rad/sec Steady state response, x = 0.1809 in Equivalent static force, F_{static} = 0.395 k Maximum being moment in columns, M_{max} = 35.55 k*in

7.4.2 Example Problem 2a:

The following problem appears in "Structural Dynamics", by Mario Paz, as Illustrative Example 3.25. A 3860 lb machine is mounted on a simple beam. The piston moves up and down in the machine, producing a harmonic force of magnitude 7000 lb and frequency 60 radians a second. 10% critical damping is assumed.



We will solve this using the "OSCILATE" program, modeling the beam as a simple oscillator.

In order to make this simplification, we will need to compute the vertical stiffness of the beam subject to a point load at the center.

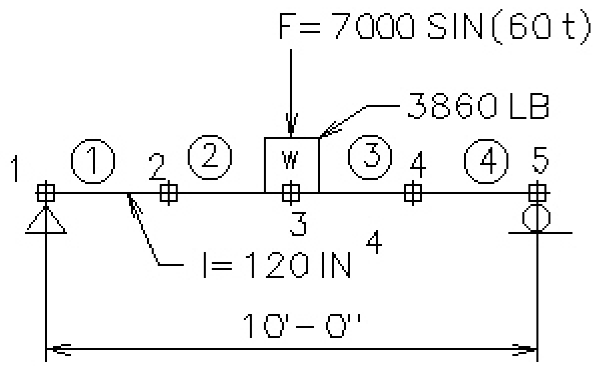
$$k = 48EI/L^3 = 48*30{,}000{,}000*120/(120)^3 = 100{,}000 \text{ lb/in}$$

Comparison of Results:

Published Results	DCALC Results
Natural frequency of system, ω_n = 100 rad/sec Steady state response, x = 0.1075 in Phase angle shift, ϕ = 10.6 degrees	Natural frequency of system, ω_n = 100.051 rad/sec Steady state response, x = 0.10 in Phase angle shift, ϕ = 10.6 degrees

7.4.2 Example Problem 2b

We will solve the same problem as example problem 2a, but will model the beam as a frame.



The above frame shows a modeled representation of the beam in the previous problem.

As before, we will prepare this analysis running the following order of DCALC programs:

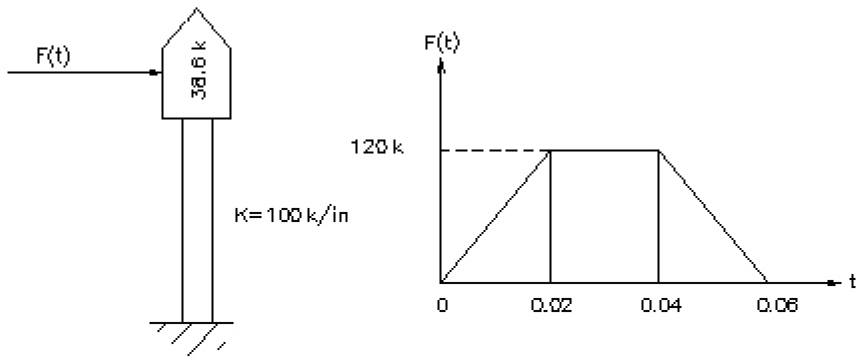
1. Input the above model in FRAME. Save the input. (No need to analyze the frame)
2. Run DYNAMIC, retrieving the FRAME input file
3. Run MODAL, retrieving the DYNAMIC output file

Comparison of Results:

Published Results	DCALC Results
Natural frequency of system, ω_n = 100 rad/sec Steady state response, x = 0.1075 in Equivalent static force, F_{static} = 10,827 lb	Natural frequency of system, ω_n = 98.319 rad/sec Steady state response, x = 0.1132 in Equivalent static force, F_{static} = 10,949 lb

7.4.2 Example Problem 3:

The following problem appears in “Structural Dynamics”, by Mario Paz, as Illustrative Example 4.2. A tower is subjected to a blast loading. The blast loading is defined by the linear load shown below. Assume 5% critical damping.



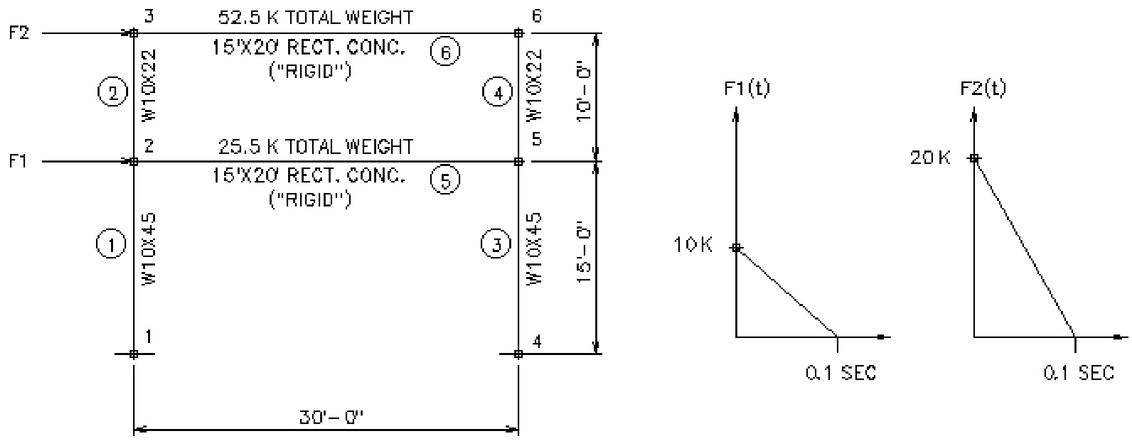
We will solve this using the “OSCILATE” program. The OSCILATE program will solve the response of a simple oscillator to either a sinusoidal force or a general (non-sinusoidal) force. In this case, the force data is linear-numerical.

Comparison of Results:

Published Results	DCALC Results
Natural frequency of system, ω_n = 31.62 rad/sec Maximum response, x = 1.291 in Maximum support reaction, F=129.132 k	Natural frequency of system, ω_n = 31.639 rad/sec Maximum response, x = 1.29 in Maximum support reaction, F=129.18 k

7.4.2 Example Problem 4:

The following problem appears in "Structural Dynamics", by Mario Paz, as Illustrative Example 11.1. A two-story frame is subjected to triangular impulsive forces at the floor levels. The impulsive loads are defined by the linear loads shown in the sketch below. Neglect damping.



The above frame shows a modeled representation of the frame. For the rigid floor elements, arbitrarily very large rectangular concrete members have been entered.

As before, we will prepare this analysis running the following order of DCALC programs:

1. Input the above model in FRAME. Save the input. (No need to analyze the frame)
2. Run DYNAMIC, retrieving the FRAME input file
3. Run MODAL, retrieving the DYNAMIC output file
4. Run FRAME. Retrieve input file. Add static forces from MODAL as a load case.

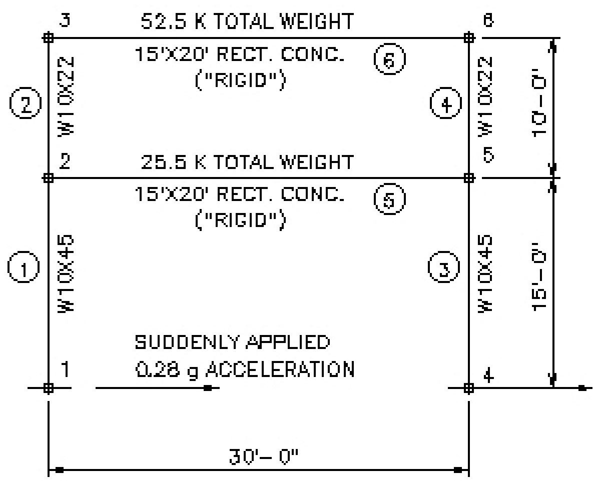
Comparison of Results:

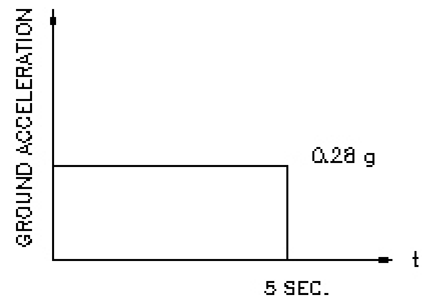
Published Results	DCALC Results
Natural frequencies of system, ω_1 = 11.8 rad/sec ω_2 = 32.9 rad/sec Maximum responses, x_1 = 0.62 in x_2 = 0.79 in Maximum shears, V_1=9.514 k V_2 = 21.636 k (*)	Natural frequencies of system, ω_1 = 11.558 rad/sec ω_2 = 33.639 rad/sec Maximum responses, x_1 = 0.637 in x_2 = 0.773 in Maximum shears, V_1=10.00 k V_2 = 7.58 k (*)

You will note that the second story shears do not agree. The referenced book has used a conservative approach for determining second story shears.

7.4.2 Example Problem 5:

The following problem appears in "Structural Dynamics", by Mario Paz, as Illustrative Example 11.2. The same two-story frame that was analyzed in example problem 4 will be subjected to a suddenly applied constant acceleration of 0.28 g at its base.





Dr. Paz's book uses a closed formed solution to the differential equation, where the constant acceleration has no end time limit. For input into DCALC's MODAL program, the end of the constant acceleration has been arbitrarily set at 5 seconds.

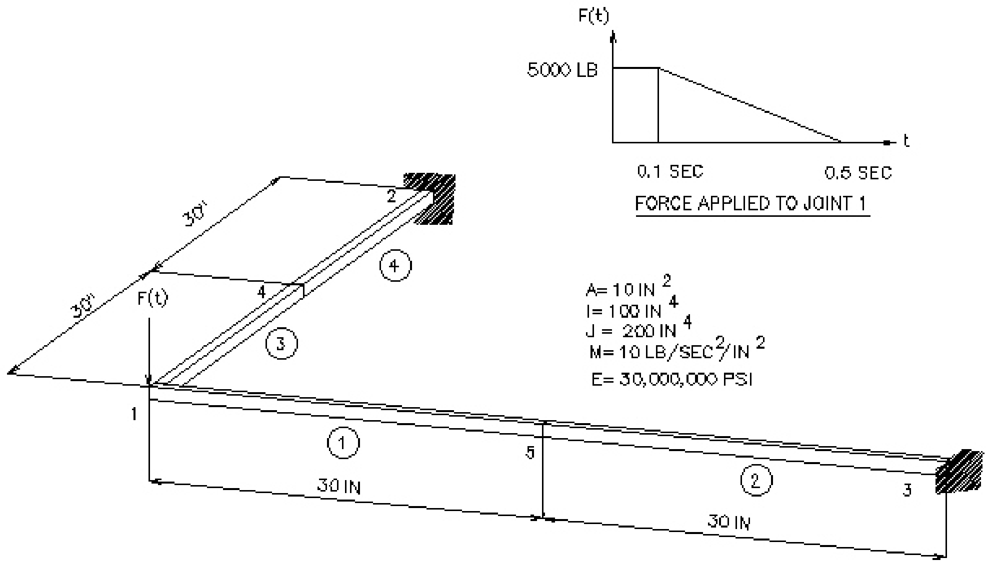
Comparison of Results:

Published Results	DCALC Results
Maximum drifts, u_1 = 1.426 in u_2 = 1.789 in	Maximum drifts, u_1 = 0.1240 ft = 1.488 in u_2 = 0.1532 ft = 1.838 in

7.4.2 Example Problem 6:

The following problem appears in a later edition of Mario Paz's book, as Illustrative Example 16.4. In later editions, Dr. Paz also illustrated the use of SAP2000 as a practical design tool, in combination with classically derived solutions.

The sketch below shows a plane grid subjected to a suddenly applied vertical force:



You will need to run the usual sequence of DCALC programs to solve this problem:

1. Input the above model in FRAME. Save the input. (No need to analyze the frame)
2. Run DYNAMIC, retrieving the FRAME input file
3. Run MODAL, retrieving the DYNAMIC output file
4. Run FRAME. Retrieve input file. Add static forces from MODAL as a load case.

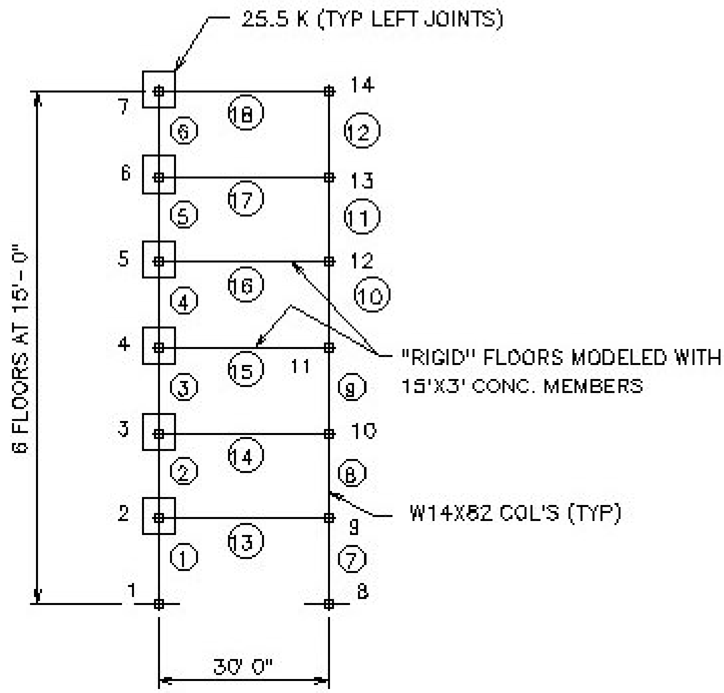
Comparison of Results:

Published Results	DCALC Results
Natural periods, Mode 1, T=0.346 sec Mode 2 T=0.086 sec Mode 3, T=0.076 sec Maximum displacement at Joint 1, y_{max} = 0.0784 in Bending moments in Member 3, Joint 1 M_{max} = 61,910 lb*in	Natural periods, Mode 1, T=0.352 sec Mode 2 T=0.086 sec Mode 3, T=0.077 sec Maximum displacement at Joint 1, y_{max} = 0.0941 in Bending moments in Member 3, Joint 1 M_{max} = 74,470 lb*in

Note that the maximum displacements differ slightly: 0.0784 inch versus 0.0941 inch. Apparently the difference is because DCALC computes the maximum displacements as the square root of the sum of the squares, whereas the other software appears to be superimposing displacements over time. The moments also reflect this difference.

7.4.2 Example Problem 7:

The following problem appears in a later edition of Mario Paz's book, as Illustrative Example 6.5. The sketch below shows a shear building with rigid floors.



This problem has been solved by running "FRAME", then "DYNAMIC".

Published Results:

Natural Periods of Vibration

Period	0.684	0.245	0.162	0.129	0.112	0.104

Modal Shapes

Jt 2	0.235	-0.662	0.941	1.000	0.824	0.471
Jt 3	0.471	-1.000	0.662	-0.235	-0.941	-0.824
Jt 4	0.662	-0.824	-0.471	-0.941	0.235	1.000
Jt 5	0.824	-0.235	1.000	0.471	0.662	-0.941
Jt 6	0.941	0.471	-0.235	0.824	-1.000	0.662
Jt 7	1.000	0.941	0.824	0.824	0.471	-0.235

(The above modal shapes have been adjusted to a reference value of 1.00 at maximum displacement)

DCALC Results:

Natural Periods of Vibration

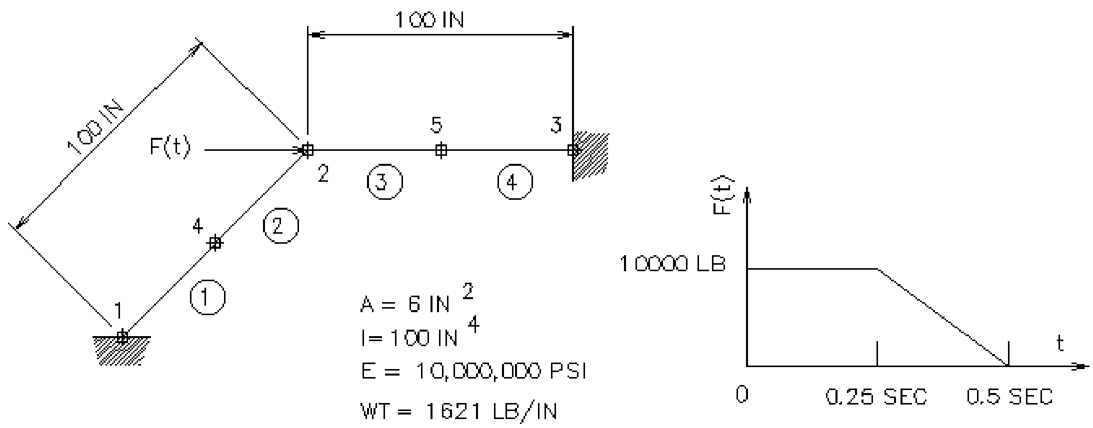
Period	0.699	0.235	0.142	0.106	0.089	0.081

Modal Shapes

Jt 2	0.212	-0.655	-0.942	1.000	0.843	0.475
Jt 3	0.426	-1.000	-0.700	-0.223	-0.938	-0.834
Jt 4	0.623	-0.857	0.432	-0.954	0.227	1.000
Jt 5	0.791	-0.294	1.000	0.441	0.685	-0.939
Jt 6	0.920	0.420	0.274	0.843	-1.000	0.660
Jt 7	1.000	0.944	-0.814	0.647	0.456	-0.233

7.4.2 Example Problem 8:

The following problem appears in a later edition of Mario Paz's book, as Illustrative Example 15.5. The problem here is to determine the first six natural frequencies and calculate the response due to the force F(t), shown in the sketch.



This problem has been solved by running "FRAME", "DYNAMIC" and "MODAL".

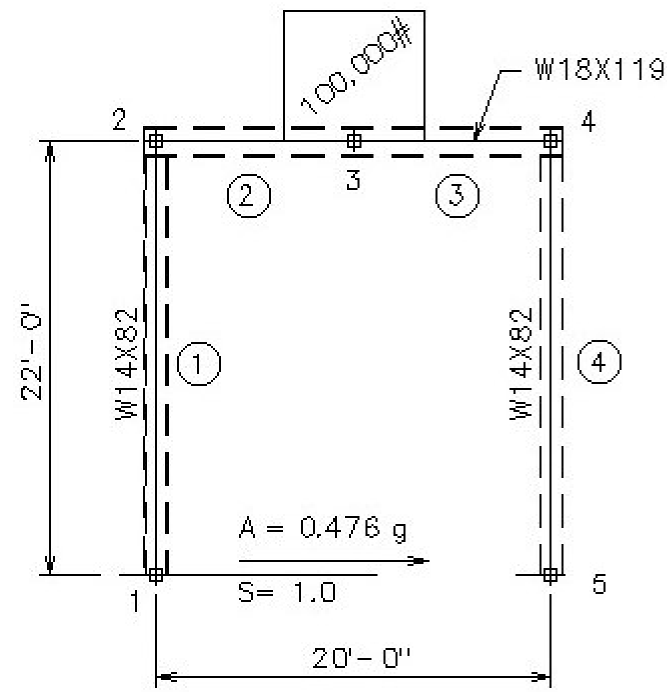
Comparison of Results:

Published Results	DCALC Results
Natural periods, Mode 1, T=0.288 sec Mode 2, T=0.275 sec Mode 3, T=0.154 sec Mode 4, T=0.113 sec Mode 5, T=0.066 sec Mode 6, T=0.0467 sec Maximum displacements at Joint 2, x_{max} = 0.260 in y_{max} = 0.274 in	Natural periods, Mode 1, T=0.299 sec Mode 2, T=0.272 sec Mode 3, T=0.128 sec Mode 4, T=0.112 sec Mode 5, T=0.054 sec Mode 6, T=0.039 sec Maximum displacements at Joint 2, x_{max} = 0.1648 in y_{max} = 0.2853 in

Note that the maximum displacements differ slightly in the x-direction: 0.260 inch versus 0.1648 inch. Apparently the difference is because DCALC computes the maximum displacements as the square root of the sum of the squares, whereas the other software appears to be superimposing displacements over time.

7.4.2 Example Problem 9:

The following problem appears in a "Seismic Design for the Civil Professional Engineering Examination", Third Edition, by Michael R. Lindeburg, P.E. The below sketch shows a frame supporting a drill press weighing 100,000 pounds. Assume 5% damping. The problem is to compute the base shear and spectral responses for the 1940 El Centro N-S earthquake.



In this book published in 1982, Mr. Lindeburg used tripartite spectra diagrams developed by Blum, Newmark and Corning, published by the Portland Cement Association, 1961. In this published design example, the spectral acceleration is S_a = 0.85 g for a structural period of T=0.55 seconds.

This problem has been solved by running "FRAME", "DYNAMIC" and "MODAL".

For input into the MODAL program, we will use the 1993 BOCA code equation,

$$S_a = 1.2*A*S/T^{2/3}.$$

Working backwards using S=1.0, requires A=0.476 g.

Comparison of Results:

Published Results	DCALC Results
Natural period, T=0.55 sec Base shear, V = 87,130 lb	Natural period, T=0.565 sec Base shear, V = 84,470 lb

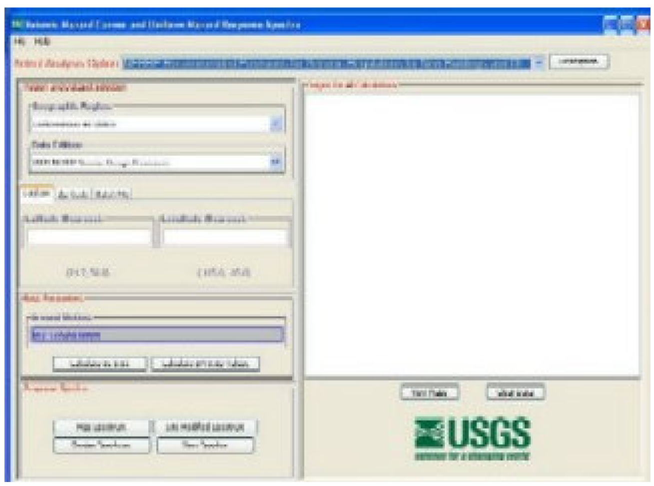
7.4.2 Example Problem 9, at Other Locations:

The previous problem will be solved using the seismic acceleration response computed per the 2006 International Building Code/ASCE 7-05.

A design tool for computing seismic acceleration coefficients for buildings is available on the USGS website at
http://earthquake.usgs.gov/research/hazmaps/design/index.php

This design tool is for IBC only! The AASHTO LRFD code includes a CD with a similar design tool, using different seismic criteria. The IBC uses a maximum considered earthquake based on an approximate 2500 year return period, whereas AASHTO uses a 1000 year return period.

After installing this software, you will see a screen similar to the following screen:



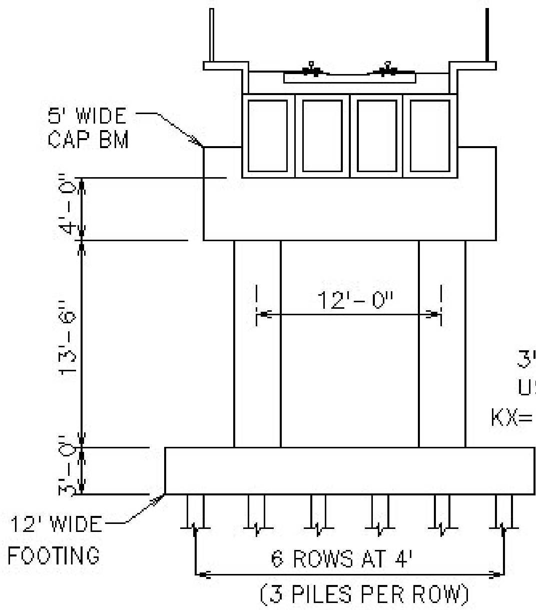
Simply by inputing zip codes for a project location, you can look-up seismic parameters for exact locations anywhere in the United States.

The previous frame was analyzed at four different cities using arbitrary different site classes, with the following results:

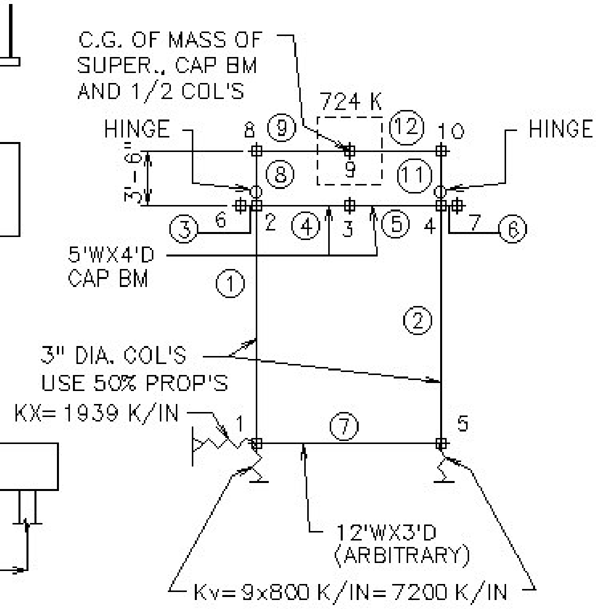
City	Zip Code	Site Class	SDs (g)	SD1 (g)	Base Shear (k)
Reno, Nevada	89501	B	1.003	0.404	72.339
Los Angeles, CA	90001	C	1.104	0.511	91.359
San Francisco, CA	94080	C	1.402	1.002	141.871
St. Louis, Missouri	63101	B	0.382	0.112	20.024
Dyersburg, Tennessee	38024	B	1.676	0.43	76.877

7.4.2 Example Problem 10:

The following problem analyzes the transverse dynamic behavior of a railroad bridge pier. This design example was taken from an excellent design example posted on the Internet (see http://www.structsource.com/railroadbridge/seismic/seismicintro.htm) The bridge is subject to seismic acceleration coefficient based on a 75 year reoccurrence of $A_{75} = 0.22g$. The site class is S=1.2. Use 5% damping.



Railroad Bridge Pier



FRAME Model

The bridge is supported on piles. The spring stiffness values are based on pile and soil properties (refer to web page for derivation).

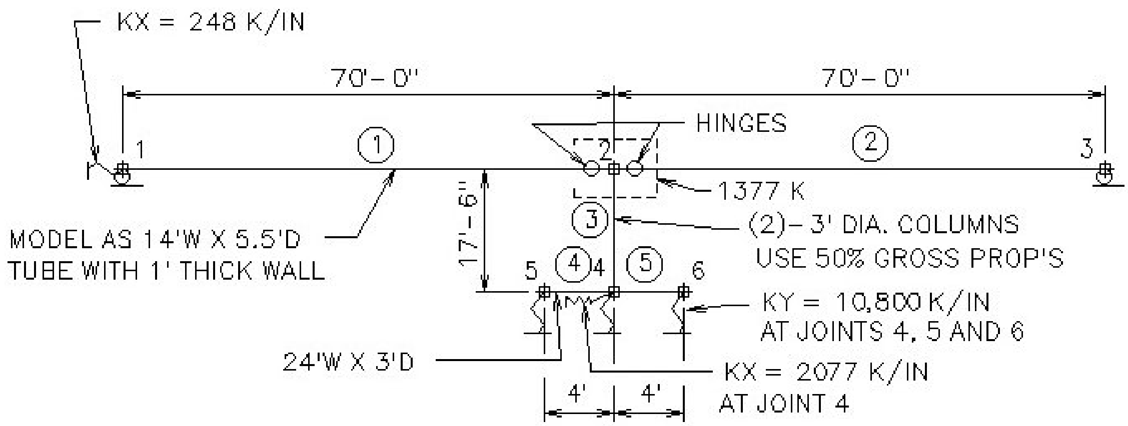
We have approached this problem somewhat differently than the referenced web page, by constructing a FRAME model. In order to place the combined mass of superstructure and substructure at the proper elevation, rigid members 8 thru 12 have been modeled, using hinges at joints 2 and 4.

Comparison of Results:

Published Results	DCALC Results
Transverse natural period, T=0.491 sec	Transverse natural period, T=0.502 sec
Transverse seismic response coefficient, Csm = 0.51 g	Transverse seismic response coefficient, Csm = 0.501 g
Column Base Shears, Left Column, V=185 k Right Column, V=185 k	Column Base Shears, Left Column, V=183 k Right Column, V=183 k

7.4.2 Example Problem 11:

This problem is a continuation of the railroad bridge described in the last problem. For this analysis the longitudinal seismic response is calculated.



FRAME Model

Again, we have approached this problem somewhat differently than the referenced web page, by constructing a FRAME model. This bridge is simply supported, therefore hinges are introduced into the model. The spring stiffness values are based on pile and soil properties (refer to web page for derivation).

Comparison of Results:

Published Results	DCALC Results
Longitudinal natural period, T=0.66 sec	Longitudinal natural period, T=0.653 sec
Longitudinal seismic response coefficient, Csm = 0.33 g	Longitudinal seismic response coefficient, Csm = 0.336 g
Seismic load, P = 454 k	Seismic load, P = 463 k
Seismic deflection, δ = 1.42 inch	Seismic deflection, δ = 1.41 inch
Column shear, V=51 k per column	Column shear, V=59.36 k per column