CVNG 3016 DESIGN OF ENVIRONMENTAL SYSTEMS
(Clarke)

LEARNING OUTCOMES (of this section of the course):

The student will be able to:

- Use readily available structural analysis software for the structural modeling and analysis of environmental structures
- Appreciate various types of tank foundations/bases and appropriately cater for the soil-structure interaction problem in their structural analysis
- Design environmental structures in consideration of vibratory effects for slabs, beams, anchorages, and buried pipes

SECTIONS

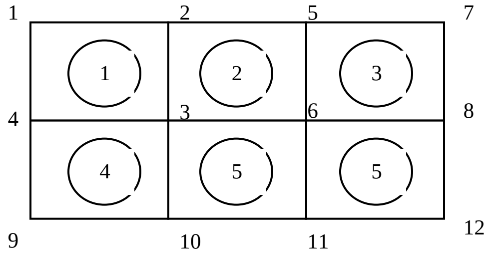
1.0 COMPUTER MODELING OF ENVIRONMENTAL STRUCTURES (STAAD)

Environmental engineering installations are generally water or wastewater treatment facilities that may be classified as: water or liquid-containing structures, tanks, and basins, conduits, inter-connecting channels, machine and equipment foundations, protective housing, floors, storage rooms, walkways, and stairs.

The main structure however is the tank whose walls and covering (if any) may be considered rigid (e.g. concrete), or flexible (e.g. steel sheets; structural plastic). The structural form of the tank is generally the shell element hence the structural design of the tank requires the use of structural shell theory to determine the internal design actions (moments, shears, etc) that must be resisted by the shell material. In many instances, the use of formulae, developed from structural theory, apply only apply to shells of simple geometrical arrangement and may not apply to more practical situations. In such cases, the Finite Element Method (FEM) is used.

1.1 FEM and STAAD

In structural analysis, FEM is a technique for determining the design actions within three-dimensional structural members such as the walls, the dome cover, etc, that make up the tank. The technique is an approximate analysis since it is based on numerical analysis. The member is broken down into a set of smaller regions called elements. Each element has a set of joints, called nodes, and the elements are connected together at the nodes. For example, the wall below is divided into 5 rectangular finite elements (triangular elements can also be used). Element number 1 is connected to joints (nodes) 1-2-3-4; element number 2 is connected to joints 2-5-3-6, etc.



The application of computer technology to structural engineering in particular, and the low cost of hardware and software at present, resulted in the easy accessibility of many finite element computer programs to the engineer and is considered a fundamental tool. In the Caribbean region one such popular program is STAAD (STructural Analysis And Design language).

The student must refer to sections 1 and 2 of the companion document entitled “STAAD Basics” to complete their appreciation of section 1.0 (download from http://ideascaribbean.com/staadba.pdf). A detailed example on the analysis of a tank structure using STAAD is provided. The remainder of this section is specifically with respect to issues encountered in the analysis of environmental structures by computer.

1.2 Modeling

In the previous section, the process of representing the wall by a set of nodes and elements is called “modeling”. The word “modeling” therefore means “representing”. A “computer structural model” is a “representation” of the physical structure so that it can then be analysed by the computer program. In structural analysis, it is not just the material that must be modeled, but also the nodes and the supports of the structure. It is important to know the condition of the physical structure (i.e. what kinds of connections and supports are used) in order to build a model that correctly represents the structure. It is possible to get results from the computer program but those results are incorrect since the model is not a true representation.

1.2.1 Modeling the Nodes

In three dimensions, the node, which is a connection between finite elements, in the most general case, is like a ball that can move in 6 ways called the degrees-of-freedom. The node can slide (i.e. translate) in the three directions, and the node can rotate in the three planes. A node can be either “fixed” or “free”. Typically, a finite element computer program initially assumes that all the joints are fixed for all of the 6 degrees-of-freedom. If the node is fixed with respect to rotating in the x-y plane say, then this means that when that node rotates, all elements connected to it will also rotate by the same amount. However, if the node is “free”, then when the node rotates, the rotation will not be passed to the other elements connected to that node. For the wall example above, and for the elements of tank structures in general, the degrees-of-freedom of the nodes are fixed, so it is rarely necessary to model a node as “free”. But for the supports of the structure, this is not the case.

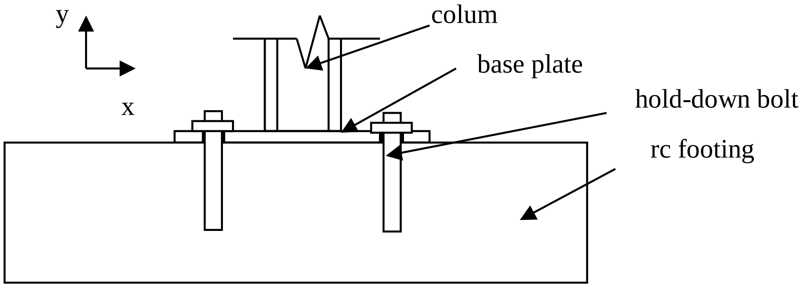
1.2.2 Modeling the Supports of the Structure

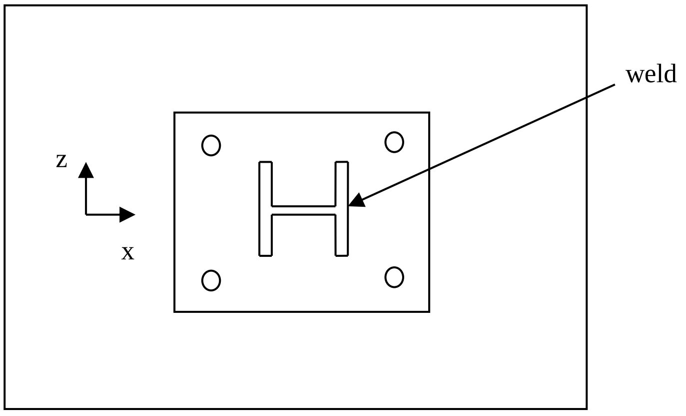
The tank base may be supported directly on the ground, or the base may be supported on a structure below such as a frame which is then supported on the ground (i.e. an elevated tank), or on a set of piles.

Any structure must be supported in order to resist the external applied loads and these supports must be suitably arranged and be of a suitable type or the structure will be unstable.

In terms of structural modeling, a support is a special type of node – a node that connects to the ground. To model the support, the analyst must ask the following question for each degree-of-freedom: for the type of connection to be used, does the connection allow free movement or rotation?

For example, if the tank is elevated and supported on a steel frame, the typical column of the frame may be connected to the ground using 4 steel bolts anchored to a footing in the ground.





How do you model this support? The bolts prevent the column, at the connection, from moving in the x and z directions. The footing below the column prevents the column from moving in the y-direction. So the support node must be modeled as fixed with respect to x, y, and z. However, the connection does not prevent the base of the column from rotating independently of the footing for rotation about the z and x-axes. The connection does however prevent independent rotation about the y-axis since the column is welded to the base plate. So for the rotation degrees-of-freedom about the z and x-axes, these must be modeled as “free”.

Making the degree-of-freedom free is called “releasing” the degree-of-freedom. In STAAD, common types of supports can be modeled by selecting the support type as being “pinned” (i.e. hinged), or “fixed”. The rotation degrees-of-freedom are called M_x, M_y, and M_z for rotation about the x, y and z-axes respectively, and the translation degrees-of-freedom, by F_x, F_y, and F_z for translation along the x, y and z-axes respectively. A “tick” mark is used by STAAD to signify a “free” degree-of-freedom, so a pinned and a fixed support would have the following settings (no tick means fixed).

SUPPORT TYPE	M_x	M_y	M_z	F_x	F_y	F_z
PINNED	√	√	√			
FIXED						

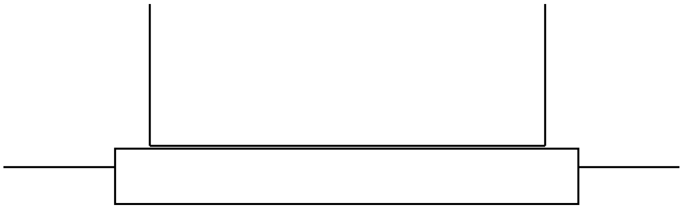
It is possible to have a support type that is neither pinned nor free, so the analyst must consider each support and model the connection accordingly.

If the vertical translation degree-of-freedom (i.e. F_y) is fixed, as it always is for pinned or fixed supports, this means that the ground below the support is considered absolutely rigid. For this to be so, the soil must be incompressible such as for dense cohesionless soil, or very stiff clays. If the soil is compressible, then the deformation of the soil is modeled in STAAD using “soil springs” of stiffness, K. This is presented in section 2.2

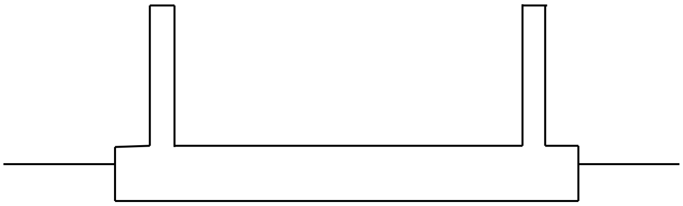
Though releasing of the degrees-of-freedom for nodes was discussed above for supports, such releases may be required in other areas such as if the tank walls are used to support other elements (e.g. pipes, beams for walkways, stairs, etc). In such a case, if the attachment is say by a simple bracket, the node at the end of the member must be released to model the connection between the member and the tank wall. The student is requested do this in STAAD as an exercise.

A final comment should now be made on the issue of the FEM for tank modeling. FEM is a complex subject since there are literally hundreds of types of finite elements. The one used in STAAD is called a general-purpose element since it can be used to accurately model a wide range of types of structural elements and conditions. However, even the selection of the element type is a part of the modeling process, and the analyst must not take the selection of the element for granted. When in doubt, other element types should be investigated and comparisons made. There are also issues of element meshing techniques and automatic mesh generation. The student should consult standard FEM texts to acquaint themselves with these topics.

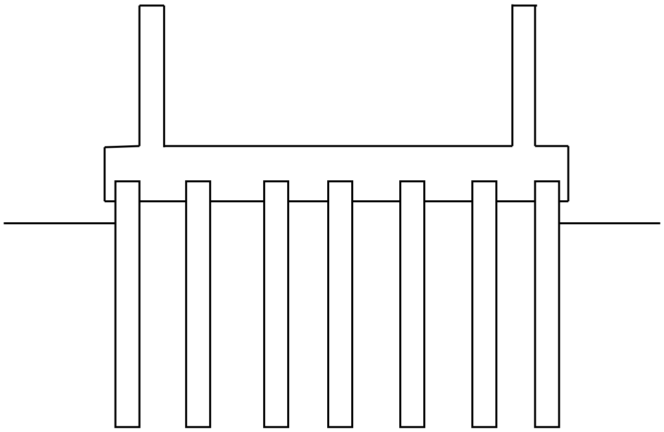
2.0 FOUNDATIONS FOR TANK STRUCTURES



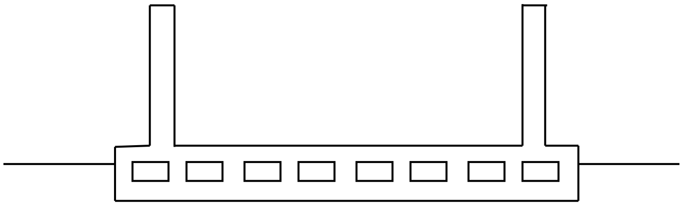
1. Flexible tank (steel, plastic) on independent rc mat foundation



2. Rigid tank (rc) with walls integrated with rc solid mat foundation



3. Rigid tank with walls integrated with rc cap on piles (concrete, steel, timber)



4. Rigid tank with walls integrated with rc cellular mat foundation

Common arrangements of tank structures in relation to their foundations are shown in figures 1 to 4 above. A tank structure has walls and a base and may or may not have a cover. Flexible tanks have relatively thin walls which may be of steel sheets welded, bolted, or riveted together. Flexible tanks can also be made of structural plastic which can be pre-packaged and delivered to site.

For flexible tanks, the base and walls are usually of the same material and the base merely rests on the reinforced concrete (rc) foundation as shown in 1. The tank can be bolted to the foundation along the perimeter to prevent uplift or sliding under horizontal forces. The flexible tank foundation is either a reinforced concrete mat or piles. In the latter case, the piles are connected via a rc slab and the tank base rests on this slab.

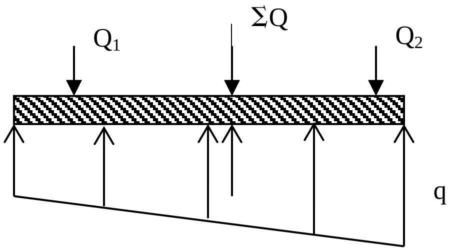
Rigid tanks are usually of reinforced concrete walls and may be prestressed. If the foundation is a rc mat, the mat usually also serves as the tank base (figure 2). If the foundation is a piled foundation, the rc slab connecting the piles usually serves as the tank base as well (figure 3).

There are generally 2 types of mat foundation – solid and cellular. A cellular mat is shown in figure 4. A main advantage of the cellular mat is that it is very stiff (i.e. resistant to differential settlement) and contains less steel than the solid mat. The cells can be arranged in one or both directions and can be void or filled with sand. The bottom slab of the cellular mat can be omitted, in which case the mat is a series of T-beams. In terms of cost however, the lower steel requirement may be offset by formwork costs, or the cost of problems due to water intrusion during construction which is likely during the rainy seasons of Caribbean countries.

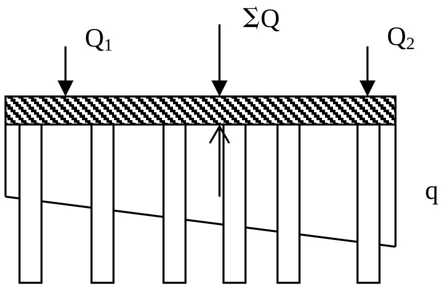
From a structural standpoint, the deformation of the tank base is critical. The tank base is deformable regardless of whether the tank is supported by a mat foundation, or a slab supported by piles. For the case of the former, since the slab itself is deformable, the type of ground below the mat is a critical consideration. This is because if the soil is compressible, the slab will deform significantly under the loads and in doing so, stresses will be transferred into the walls of the tank.

The remainder of this section will focus on this situation - the tank base is a solid mat foundation for the tank as a whole.

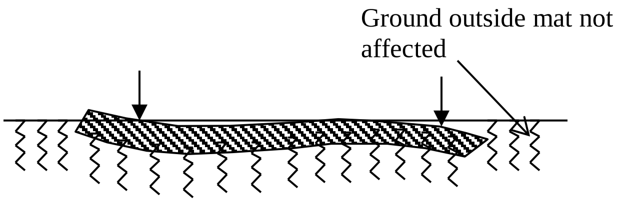
2.1 Rigid Mats



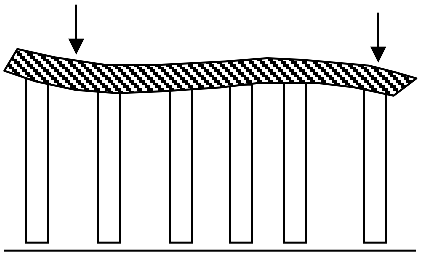
1. Rigid Mat on Soil



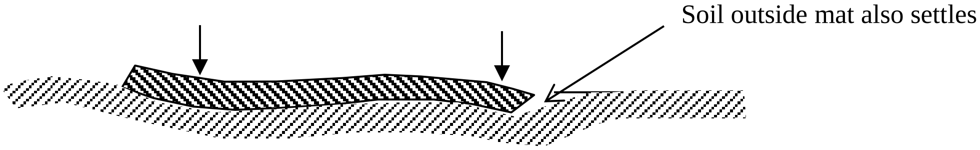
2. Rigid Mat on Piles



3. Deformable Mat on Soil Springs



4. Deformable Mat on Piles



5. Deformable mat on elastic solid

If the mat itself (also called a raft) is considered rigid, this means that it does not deform. In this case, the soil reaction is a planar surface - q, which is determined by statics (Figure 1 above). Notice from figure 2, that if the mat is rigid, the resultant is the same as when the mat is supported on the soil. When q is determined, the design actions in the mat (moments and shears) are determined by merely considering strips of the mat as upside-down beams, then applying statics. Since the aim is to use the computer for the analysis calculations, the application of this approach is left as an exercise for the student.

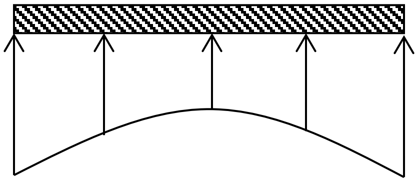
The rigid mat approach is the easiest to use and gives suitable results if the type of soil beneath the mat is soft (mud, soft clay, medium clay, peat and organic soils). However, as the stiffness or compactness of the soil increases the method loses accuracy. It tends to lead to significantly underestimated bending moments in the mat near the edges, and overestimated moments in the regions between the wall locations, compared with the actual response in the mat.

2.2 Flexible Mats

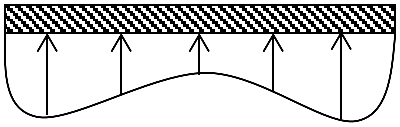
If the mat is considered deformable, the movement of the soil affects the moments and shears within the mat (hence the tank walls as well). There are two approaches to determining the design actions in the mat:

1. Consider the soil as a series of springs, called Winkler springs, of stiffness K (as mentioned in section 1.2.2; Figure 3 above).
2. Consider the soil as an elastic soild (Figure 5 above).

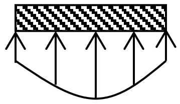
Approach 1 can give significant error due of the distribution of the soil pressure on the underside of the mat, depending on the type of soil below the mat, and regardless of the type of load on the top surface. This happens because the method cannot account for the fact that the soil is a solid body, so there is interaction at all directions through a point. The actual pressure distributions are as indicated below.



Clay



Large mat on sand



Small mat on sand

Approach 2 is the closest to the actual situation but the solution approach is generally impractical for many situations.

Considering the pros and cons of the three methods (i.e. rigid mat, Winkler springs, elastic solid), the most appropriate balance of ease of use and accuracy is obtained using the Winkler springs approach, but corrected using factors determined from the more elaborate but accurate “elastic solid” approach. This situation has occurred mainly due to the improvements of computer technology such as the STAAD computer program.

Procedure for Using Winkler “Soil” Springs in STAAD Analysis of a Tank

1. Determine the modulus of subgrade reaction for the soil, k (kN/m^3; available from soil mechanics texts). As an approximation, k is 120 times the allowable bearing capacity of the soil in kN/m^2.
2. For each support node, calculate the tributary area for the node.
3. For each support node:
   (a) multiply k from 1, by the area from 2; this gives the spring stiffness, K kN/m.
   (b) If the node is a side node (i..e along the edge of the mat) multiply K by 3, but if a corner node, multiply by 6.
4. In STAAD, and for each support node: use the “fixed but” option and input the K value from 3(b).

This procedure gives reasonably accurate results, but for maximum accuracy, refer to ACI 436 which gives a more detailed procedure for determining the correction factors.

The foregoing is with respect to the mat (i.e. tank base) being directly supported on soil, and not for the case where the tank base is considered deformable but supported on piles. For this case, the Winkler springs approach can also be used (in STAAD) but the piles' locations should coincide with the support nodes. The value of the spring stiffness is then EA/L where E is the Young's modulus of the pile material, A, the cross-sectional area of the pile, and L, the pile length.

3.0 DESIGN FOR VIBRATORY EFFECTS

With respect to environmental systems in the regional context, vibratory effects occur due to two main sources – machine vibration and earthquakes.

From a design perspective, the consideration of vibratory or dynamic loading effects is a matter of determining the maximum inertial forces and relative displacements that arise when a structure is externally excited by the vibrating force. This is because of the phenomenon called resonance in which case the magnitude of the external force is magnified relative to if the vibration was not present. The magnification increases as the frequency of the external vibrating force approaches the natural frequency of the structure or element that force is acting on. The natural frequency of the structure or element is the frequency at which it vibrates if given an initial displacement and released to vibrate freely. This is typically calculated using empirical formulae for the structure or element.

$$F_I + F_D + F_S = F_E$$

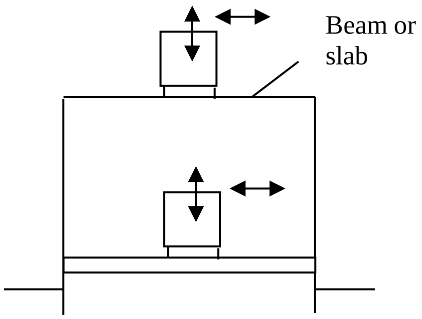
The external force F_E is resisted by the sum of the inertial force, F_I, the internal frictional or damping force, F_D, and the stiffness or spring force, F_S. If resonance effects are negligible, then $F_E = F_S$. But if not, F_I acts like an additional external force that must be resisted, hence the magnification.

With respect to machine vibrations, the elements of the environmental structure that support the machinery (beam, slab, column) must be able to withstand the dynamic magnification of the weight of the machinery.

Environmental structures are typically required to remain functional after an earthquake.
The earthquake event induces vibratory motion of the environmental structure in terms of the magnitude and distribution of the inertia forces associated with the mass of the structure, and the masses of the various items (machinery, pipework, etc) that the structure supports. For the former, the focus is the tank structure and its contents, if any, and is outside the scope of this presentation. For the latter, the focus is the force exerted on the supports of the items attached to the structure, and the displacement of these items relative to the structure. These considerations are necessary to ensure that the movement can take place without rupture of the connections, or the pipework, including underground or buried pipes.

3.1 Machine Vibration

The typical relationship between vibrating machinery and its supporting structure is shown the figure below.



Vibrating machinery on beam or slab

Centrifugal pumps, fans, centrifuges, blowers, generator engines, and compressors generally have sufficiently high rotational speeds to induce inertial forces that must be considered in the design of their supports and foundations. The vibration arises since no rotating mass is perfectly balanced as there is always some degree of eccentricity between the centre of gravity and the geometric centroid of the mass.

The key to successful dynamic design is to insure that the natural frequency of the machinery support structure is significantly different from the frequency of the disturbing force. This is a more cost-effective approach since the maintenance or repair cost of the machine due to the resonance effects is much higher than the cost of adjusting the structure's properties.

Machine Supported Directly on Suspended Beam or Slab

The ratio of the natural frequency of the structure to the natural frequency of the disturbing force, F_N/F_M, should satisfy the following:

$$(F_N/F_M) < 0.5 \text{ or} \qquad (1)$$

$$(F_N/F_M) > 1.5 \qquad (2)$$

Eq (2) is preferred (i.e. a stiff structure) since (1) implies that at startup and shutdown of the machinery, it will pass through F_N. In the general case, the structure may be supporting M masses that vibrate in all three directions (vertical and 2 horizontal). If a mass is vibrating horizontally, the columns supporting the mass will act like beams. To get the natural frequency of the structure for this general case use the following:

$$F_N = 1 / [\ \sqrt{(\textstyle\sum_1^N (1 / F^2{}_i))}\,] \qquad (3)$$

where F_N is the structure's natural frequency, and F_i is the structure's natural frequency for the case of mass "i".

F_i is determined from a formula for a beam that gives the natural frequency depending on the static deflection, D, due to the weight of the mass applied in the direction of the vibration. Note that this same formula is used if the machinery vibration is horizontal since the structure's columns then act like cantilevers.

For a concentrated load at any point along the span regardless of end conditions (this includes the cantilever case), if the deflection at any point is D (mm), then the natural frequency of the beam in cycles per minute is $947/\sqrt{D}$.

For a beam pinned or fixed at both ends and carrying a uniformly distributed load with mid-span deflection D (mm), the natural frequency of the beam in cycles per minute is $1073/\sqrt{D}$.

When the F_i is thus determined, substitute in (3) to calculate F_N, then check (1) or (2) for each vibrating mass.

If the machinery is installed on an upper floor, the use of vibration isolators is recommended but this should not be considered a substitute for dynamic structural design.

Machine Supported on the Ground (Foundation Slab)

The aforesaid technique can also be applied to machines directly supported on the foundation.

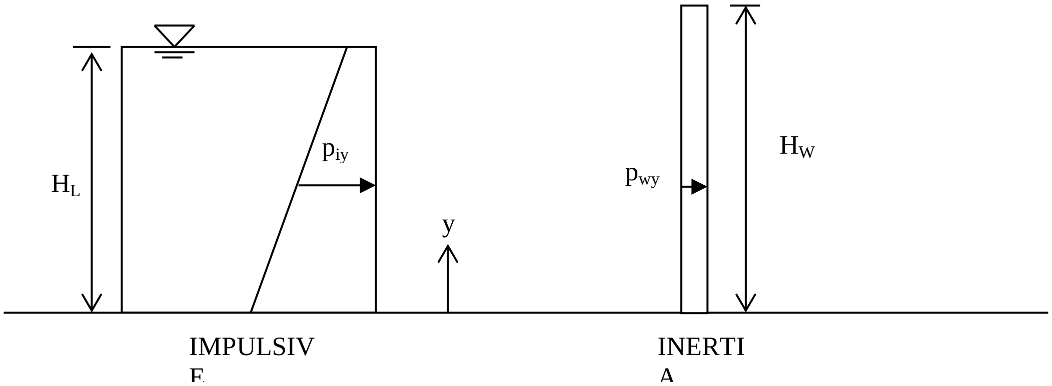
However, a rule-of-thumb that is frequently used is that if the foundation is comprised of pad footings, use an allowable bearing capacity that is one-half of what would have been used if vibration was not a factor.

Also, some equipment manufacturers recommend a minimum ratio of foundation to equipment mass of 4 to 6, but it has been observed that this approach does not always give satisfactory results. For relatively important facilities, the natural frequency of the foundation should be calculated by reference to special publications on the subject.

As a final note, the determination of the vibratory effects due to the machinery can be performed using the STAAD computer program described in sections 1.0 and 2.0. Refer to the “STAAD Basics” handout, especially Example 3.0.

3.2 Earthquake Hydrodynamic Wall Pressures

Under an earthquake hydrodynamic pressures are exerted on the walls of the tank in the direction of the earthquake. In the following presentation only rectangular tanks supported by the ground, but not fully or partially embedded, are considered. Also, only the impulsive pressure and wall inertia pressure are considered. That is, the convective and vertical wall pressures are deemed negligible. The sketches below indicate the impulsive and wall inertia pressures.



Impulsive Pressure:-

Height of CG of impulsive component of liquid, h_i is given by the graph supplied.
Equivalent weight of impulsive component of liquid, W_i is given by the graph supplied; unit kN
Response coefficient, $R_i = 2$
Importance factor, I = 1.5 for a tank with hazardous materials,
= 1.25 for tank that is intended to remain functional after the earthquake,
= 1.0 otherwise.
Seismic response coefficient, $C_i = (2/3)S_S F_a$; unit dimensionless
S_S = short period mapped spectral acceleration at the site location, with 2% exceedance probability in 50 years; obtained from http://uwiseismic.com/seishaz.aspx).
B = wall plan dimension at right angles to direction of earthquake; unit m
$F_a = 1.2$ (i.e. Site Class D).

$P_i = C_i I (W_i / R_i)$; unit kN

$$P_{iy} = \frac{P_i}{2H_L^2}\left[4H_L - 6h_i - \left(\frac{y}{H_L}\right)(6H_L - 12h_i)\right] \quad \text{; unit kN/m}$$

$p_{iy} = P_{iy}/B$; unit kN/m^2

Inertia Pressure:-

L = tank plan dimension in direction of earthquake; unit m.
t_w = wall thickness; unit m.
The effective mass ratio, ε, is given by

$$\varepsilon = \left[0.0151\left(\frac{L}{H_L}\right)^2 - 0.1908\left(\frac{L}{H_L}\right) + 1.021\right] \le 1.0$$

$P_{wy} = 23.6 (C_i I / R_i) \varepsilon B t_w$; unit kN/m
$p_{wy} = P_{wy}/B$; unit kN/m^2

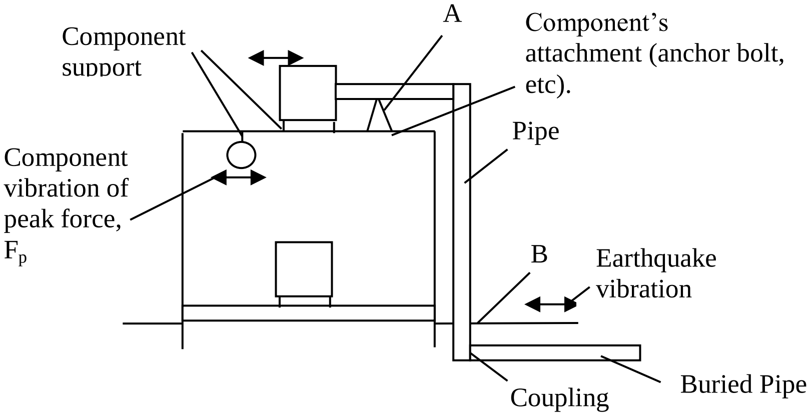
The impulsive and inertia pressures are simultaneously applied to each wall that is at right angles to the earthquake. The same must be done when the earthquake is in the other plan direction of the tank. Of course in this case the walls will not be the same.

The hydrostatic pressures must also be considered simultaneously with the hydrodynamic pressures. As an earthquake load is always applied with other loads, the following are the load combinations to be considered where F refers to the hydrostatic pressure, and E refers to the hydrodynamic pressure due to the earthquake.
$U = 1.4(D+F)$
$U = 1.2(D+F) \pm E$
$U = 0.9(D+F) \pm E$

Recall that the last 2 combinations must be done for the other earthquake direction as well.

3.3 Earthquake Vibration Effects on Components Attached to Environmental Structures

The figure below displays the typical scenario. The earthquake event induces the vibration of the ground causing the environmental structure (in this case an equipment storage building) to vibrate. However, certain components may be attached to the vibrating structure which in turn also vibrate and experience a peak inertial force F_p. The component, say the vertical pipe shown in the figure, may be supported at several points such as A and B but these points will not move in a synchronized manner. They will move relative to each other and thereby induce stresses within the pipe.



Earthquake effects on attachments, buried pipe, and pipe couplings

The design therefore must consider the F_p's for each component, and the peak relative displacement between the component's supports, for all the components. The F_P is used to design the support (i.e. frame, brace, leg, cable, stay, pedestal, etc) for the component, and the attachment (typically anchor bolts) by which the support is connected to the structure.

If the relative displacement between the component's supports is excessive, the movement must be allowed to take place without damaging the component. For pipes, such as between A and B in the figure, this is typically achieved by using flexible couplings. A coupling is a connection between one segment of a pipe and another. Information on the deformation capacity of a coupling is typically available from the supplier.

F_p Calculation:

F_p is typically calculated by use of a formula, though it is possible to study the situation more rigorously by structural dynamics. The formula refers to certain constants that depend on the type of component. The following is the IBC 2000 formula and the values of the constants for some components.

$F_p = A \times B/C$ where,

$A = 0.4a_p\, S_{DS}\, W_p$

$B = 1 + 2z/h$

$C = R_p/I_p$

such that,

a_p = component amplification factor (from table)
F_p = component's peak earthquake force centred at its centre of gravity
I_p = component's importance factor; 1.0 or 1.5
h = height of the structure above ground (that the component is connected to)
R_p = response modification factor (from table)
S_{DS} = design spectral response short-period acceleration; assume 1.0 for Caribbean countries
W_p = component's operating weight
z = height of component above ground

F_p is not required to be taken as greater than,

$F_p = 1.6\, S_{DS}\, I_p\, W_p$

but must not be less than,

$F_p = 0.3\, S_{DS}\, I_p\, W_p$

The force F_p must be applied independently longitudinally and laterally in combination with service loads associated with the attachment.

COMPONENT	a_p	R_p
Stacks	2.5	2.5
Piping systems with elements and attachments of high-deformability	1.0	3.5
Piping systems with elements and attachments of low-deformability	1.0	1.25
Vibration isolated equipment	2.5	2.5
Non-vibration isolated equipment	1.0	2.5
Electrical equipment	1.0	2.5
Electrical bus ducts, conduit, cable trays	1.0	3.5

Relative Displacement Calculation:

(a) If the two points are on the same structure A:

The earthquake relative displacement, D_p, is calculated by using the following equation. For two connection points – one at level x, and the other at level y:

$D_p = D_{xA} - D_{yA}$

(b) If one point is on one structure A, and the other point is on another structure, B:

$D_p = |D_{xA}| + |D_{yB}|$

The D_{xA}, D_{yA}, and D_{yB} are as determined from the earthquake structural analysis but must be considered in combination with displacements caused by other loads.

At the interface of adjacent structures or portions of the same structure that may move independently, utility lines shall be provided with adequate flexibility to accommodate the anticipated differential movement between the ground and the structure.

Component Anchorage:

Components must be anchored (component-to-support, or support-to-structure) in accordance with the following:

(a) Where component anchorage is provided by shallow expansion anchors (e.g. HILTI bolts), shallow chemical anchors (e.g. “lock-set”), or shallow cast-in-place anchors (e.g. plate or rod “crabs”), the value for R_p must be 1.5.
(b) Anchors embedded in concrete or masonry must be proportioned to carry the lesser of:
  - the design strength of the connected part
  - 1.3 times the force in the connected part due to the prescribed forces
  - the maximum force that can be transferred to the connected part by the component structural system
(c) Determination of forces in anchors must include the expected conditions of installation including eccentricities and prying effects.
(d) Determination of force distribution in a group of anchors at one location must include the stiffness of the connected system and its ability to redistribute loads to other anchors of the group due to yielding of anchors.
(e) Powder-driven fasteners (i.e. gun applied) must not be used for tension load applications (e.g. hangers) in areas of high seismic hazard.

Special Considerations:

(a) Tanks:

For storage tanks mounted above the ground, the attachments and supports must be designed to meet the F_p but with R_p given by the following:

Concrete or welded steel tank – 2
Tank supported on braced or unbraced legs – 3

For tanks at ground level of diameter greater than 6.1m, or tanks at ground level with height-to-diameter greater than 1.0, the piping connections must be designed to either resist or survive without damage, the following displacements for side-wall connections and bottom penetrations:

- A vertical displacement of 51mm for anchored tanks
- A vertical displacement of 305mm for unanchored tanks
- A horizontal displacement of 203mm for unanchored tanks with a diameter of 12.2m or less.

(b) Piping:

Under design loads and displacements, piping must not be permitted to impact other components.

Piping must accommodate the effects of relative displacement that can occur between piping support points on the structure or the ground, other mechanical and/or electrical equipment, and other piping.

Supports for piping need not be designed for F_p if $I_p = 1.0$ (i.e. a non-critical structure) and if - the support is comprised of rod hangars for pipe less than 305mm diameter; the pipe is of high-deformability and less than 75mm diameter for a region of high seismicity.

(c) Mechanical Equipment:

Expansion anchors must not be used as attachments for mechanical equipment rated above 10 horsepower (7.5 kW) and the equipment is not vibration-isolated, except if the expansion anchor is of the "undercut" expansion type.

Friction-clips must not be used as attachments for mechanical equipment.

(d) Buried Pipe:

If there is no settlement, liquefaction or fault displacement in which case the soil around the pipe retains its integrity after the earthquake, the pipeline will comply with the soil movement.

The results of this are an increase in the axial strain of the pipe, and bending at the pipe connections. Generally, buried pipelines in seismic zones should be flexibly-jointed and the main concern will be in avoiding separation or impact at joints, and limiting the bending stresses at or near points of restraint.

The soil vibration causes a strain within the soil that the buried pipe resists by friction.
The frictional force is given by,

$F = 2\pi d\gamma H f_f$ where d is the pipe diameter, γ is the soil density, H is the depth below the surface, and f_f is the coefficient of friction between the pipe and soil.

The axial stress developed in the pipe due to the soil strain is,

$f_a = \sqrt{(2 E F \Delta x/A)}$, where E is the Young's modulus of the pipe material, F is the frictional force between pipe and soil, A, is the pipe cross-sectional area, and Δx is the extension of the pipe due to the axial strain.

The peak axial strain ε_{amax} is given by,

$\varepsilon_{amax} = V_{max}/C$ where V_{max} is the maximum ground velocity, and C is the velocity of the propagation of the seismic wave through the soil.

Over a length of pipe L, $\Delta x = \varepsilon_{amax} \times L$. Therefore, the spacing of the flexible joints can be estimated, as well as the required deformation capacity of the joint in order to avoid impact of adjacent pipe segments, and sustain the movement.