Idealization of Structures and Loads

To analyze a structure by the methods that are described in these notes it must be idealized. By utilizing the idealized structural model the deformations and internal forces are computed at selected locations in the structure. Depending on how closely the model matches reality the results provide accurate insight into the behaviour of the real structure. This document discusses briefly the process of building the idealized structural model. In particular, several modelling choices must be made, which are discussed in the following sections.

Modelling the Structural Members

All structural models are discretized into one or more structural members. A variety of structural member types are available for this purpose. Longitudinal members are most popular in hand calculations and include truss bars, beams, and column elements. These are called longitudinal because they are significantly longer in one direction compared with the other two. Structural analysis with such elements is the cornerstone of fundamental structural analysis. The arsenal of member types also includes two-dimensional members like plane membranes, plates, and shells. Volume members, like brick elements, are also possible but limited to computer analysis by the finite element method. The most common members in structural models are:

- **Axial bar:** This longitudinal member is often called a truss member. It carries load only in the axial direction. It has one unknown internal force: the axial force, N.
- **Torque bar:** This longitudinal member carries torque, T, which is its only unknown internal force.
- **Beam:** This longitudinal member is also called frame member, column, or beam-column. It carries load in the direction perpendicular to the member. In a 2D structural analysis it has two unknown internal forces: the shear force, V, and the bending moment, M. In a 3D structural analysis it has four unknown internal forces: the shear force in the two directions of the cross-section and the bending moment in those two directions. The beam member is often combined with the axial bar so that it carries axial force as well. In a 3D structural analysis it is also often combined with the torque bar, giving a total of six unknown internal forces.
- **Plane membrane:** This two-dimensional member carries load only in its plane. It is for instance employed in shearwall-diaphragm-type analysis.
- **Plate:** This two-dimensional member carries load out-of-plane. It is typically utilized to model horizontal slabs that carry vertical loads.
- **Shell:** This two-dimensional member carries load both in-plane and out-of-plane, and may also have a curved initial shape. It is a versatile type of member in the finite element method that is utilized to model curved shell problems like curved roofs and intersection of pipes.

Figure 1 illustrates the idealization of real structures into beam members with various boundary conditions. The bridge at the upper-left is idealized by connected beam elements with different boundary conditions at their free end. Also for the slab at the middle-left in Figure 1 the beam idealization seems viable: it implies that a narrow strip of the slab is considered as a simply supported beam in the analysis. The results are equally applicable to any other strip of the slab. In passing it is noted that the governing equations for this beam may have to be modified because it is part of a long plate. This is described in the document on Euler-Bernoulli beam theory. The idealization of the bottom-left container wall may be less intuitive. However, this may also be a good utilization of the beam member. Instead of using advanced theory of plates, the consideration of a strip of the wall facilitates simple and conservative analysis. It is conservative, i.e., on the safe side, because in reality the strip will have some load-carrying effect in the transversal direction. The assumption of fixed or pinned end conditions must also be carefully considered and matched with the actual design of the corners of the container.

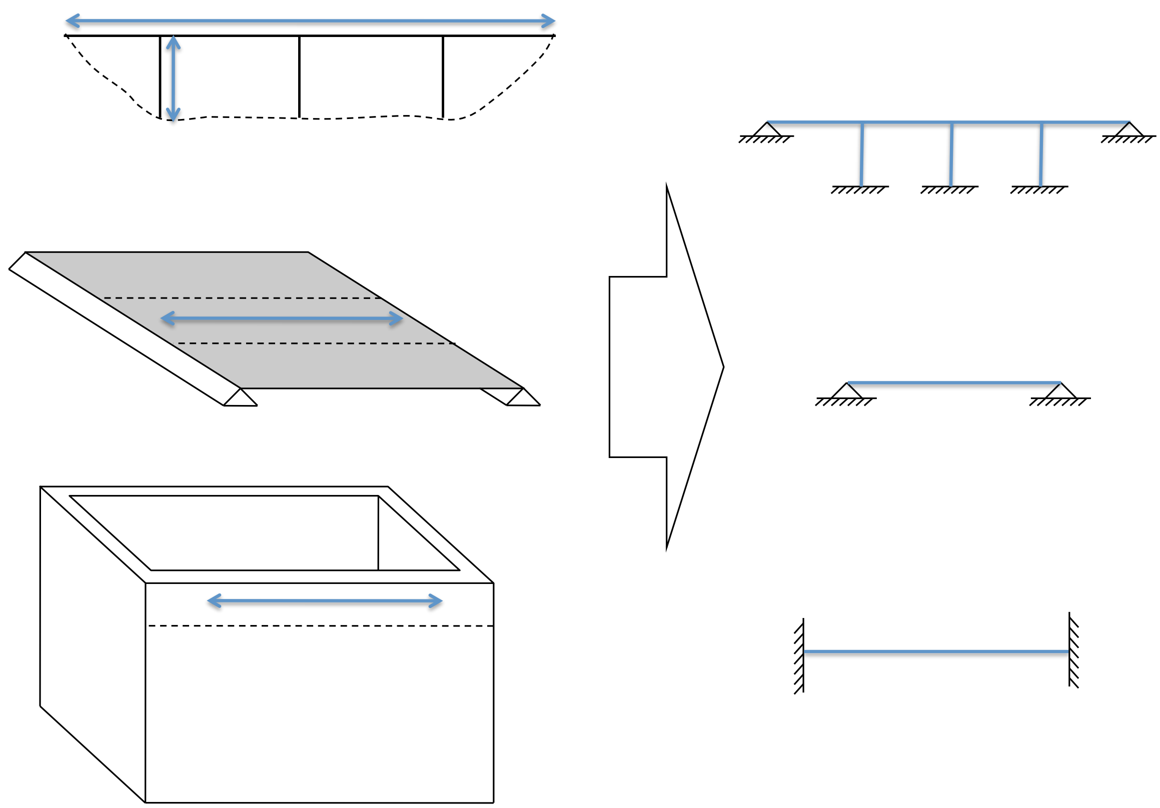

Figure 1: Idealization of real structures with beam members.

2D or 3D Modelling

All real structures exist in the three-dimensional space. In fact, most structures extend in all three of the axis directions. However, powerful insight is obtained from 2D models, which are easier to deal with and often more transparent. Therefore, the choice of idealizing a real structure into a 2D model is often a good one, at least as a first approximation. The examples in Figure 2 serve as simple illustrations of where this type of idealization may be appropriate and can provide useful insight. There are of course many structures that do not lend themselves to 2D models. Significant asymmetries in both horizontal directions, presence of torsion, and a number of other situations requires 3D structural analysis.

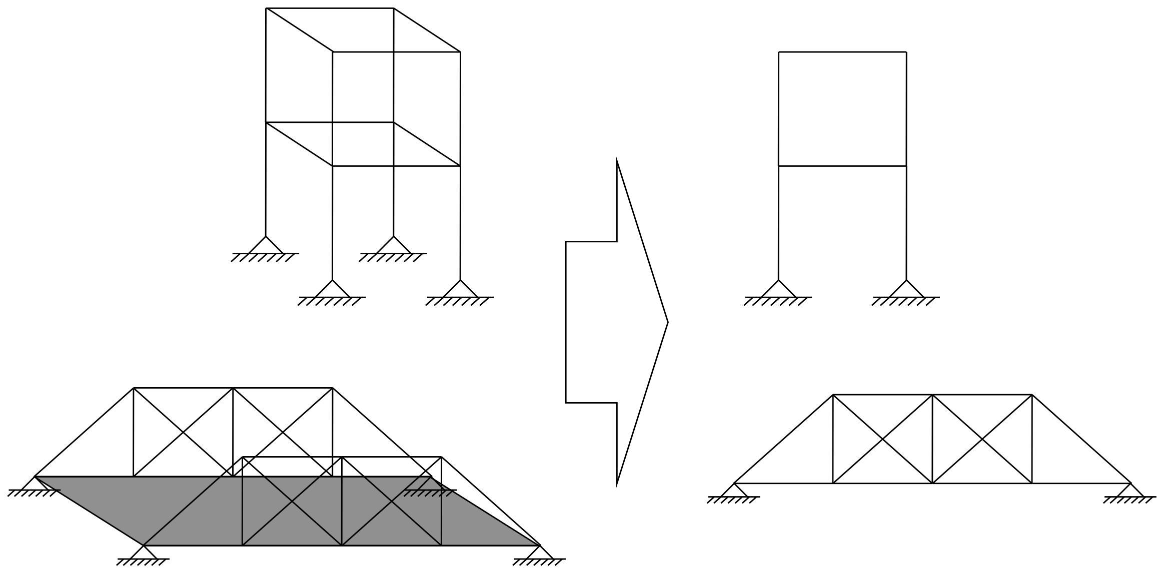

Figure 2: Idealization of 3D structures to 2D problems.

Modelling of Boundary Conditions

All structures on Earth are somehow grounded. Although a geotechnical engineer will argue that these foundations require distinct modeling the fundamental structural analysis employs idealized supports. These are referred to as boundary conditions. An overview of the boundary conditions available in 2D models is provided in Figure 3. An inclined roller, i.e., one in which the “wheels” are rolling along an inclined slope is also possible but not shown. The possibility of modeling a support as a spring is also possible. Another document on “degree of indeterminacy” counts the number of unknown forces and deformations at each support. Of interest here is when to use which boundary condition. The following considerations may be helpful:

- Often the engineer can influence the design to match the selected structural model. Consider the following example: An engineer observes that the base of a column must have a fixed type boundary condition to ensure the structure’s integrity. Consequently, the column is modeled with a fixed support at the base and the connection is designed with appropriate connection details, i.e., bolts, reinforcement, etc., to maintain the correctness of this assumption.
- Another approach is to select the boundary condition that will give conservative results. The simplest example is a horizontal beam with end supports. If there is doubt about the appropriate boundary condition then the conservative assumption that will give the maximum bending moment at midspan is a pinned-pinned beam.

Conversely, to obtain the worst-case bending moment at the ends a fixed-fixed beam is selected.

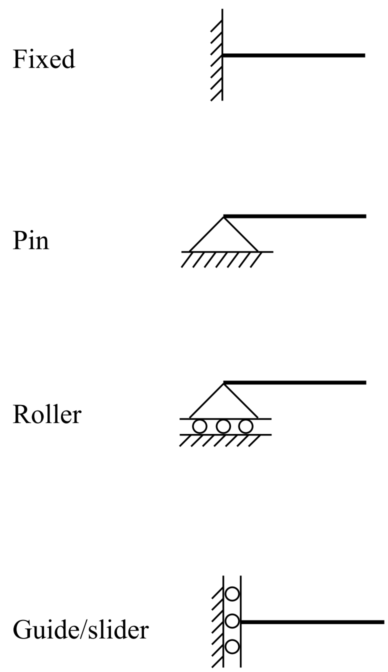

Figure 3: Boundary conditions.

Modelling of Loads

The objective of structural analysis is to determine the deformations and forces in the structure due to external loading. The following is a list of the loads that are considered for typical buildings and bridges:

- Dead load (*D*)
- Live load (*L*)
- Snow load (*S*)
- Wind load (*W*)
- Earthquake load (*E*)
- Earth pressure (*H*)
- Imposed deformation (*T*)
- Traffic load

The one-letter abbreviations are from the National Building Code of Canada (NBCC). This is the document that prescribes load values for most buildings. Although many jurisdictions use other codes for the loads, like the British Columbia Building Code and the Vancouver Building By-law, they are all based on the NBCC. The exception in the list above is traffic load, which is covered by the "bridge code," i.e., the CAN/CSA S6 Canadian Highway Bridge Design Code.

Figure 4 shows an overview of some of the loads, with emphasis on identifying their direction and the surface they act on. Notably, the snow load, *S*, acts vertically on the horizontal projection of the roof. Conversely, the wind load, *W*, acts in compression or suction perpendicular to the roof surface. The dead load, *D*, also acts along the length of each member, but in the vertical direction. The earthquake load, *E*, is rather imprecisely illustrated in Figure 4. The purpose is to emphasize that the building code specifies earthquakes as horizontal forces on the building. This makes the structural analysis similar to the analysis for the other loads. However, alternative seismic design procedures are available, both displacement-based and full-blown finite element simulation of the structural response due to ground motions. A ballpark figure for *E* is perhaps 15 to 25% of the building weight. Ballpark figures for the other loads are perhaps normally in the ranges S=1.5 to 3.5kN/m^2 (although the double is possible), W=1.0 to 3.0kN/m^2, D=0.5 to 2.0kN/m^2, and L=2.0 to 5.0kN/m^2. These values are solely intended for illustration purposes. In actual designs these value cannot and must not be used. Rather, load values must be computed according to the requirements in the building code for the particular location of the structure under consideration.

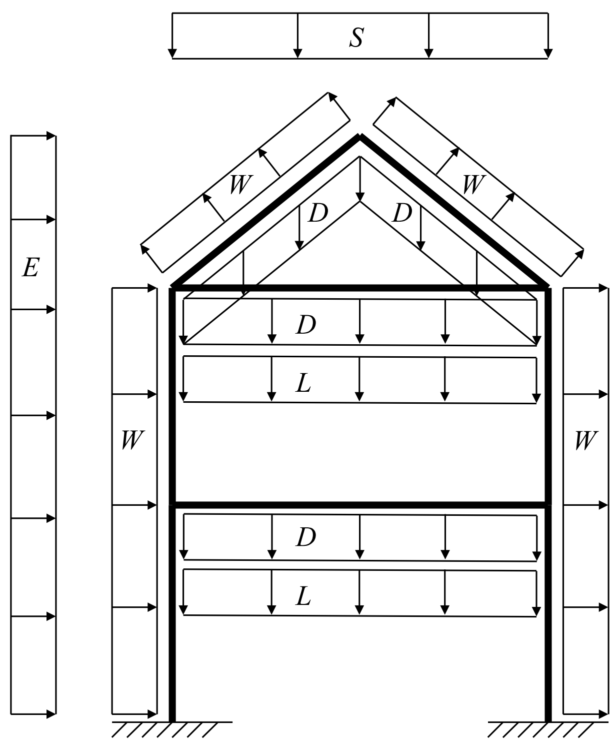

Figure 4: Building code loads and the surface and area they act on.

Upon calculation the relevant loads from the building code, the total load according to several load combinations must be calculated and compared. In accordance with limit-state design, which is called load and resistance factored design (LRFD) in the USA, each load combination has different safety factors on the loads. For limit-states that relate to strength, such as the structure's capacity to carry bending moments, shear forces, and axial forces, the load combinations in Table 1 are currently prescribed in Canada, with some caveats that are omitted here.

Table 1: Load combinations in the NBCC.

Case	Principal Loads	Companion Loads
1	1.4D	
2	(1.25D or 0.9D) + 1.5L	0.5S or 0.4W
3	(1.25D or 0.9D) + 1.5S	0.5L or 0.4W
4	(1.25D or 0.9D) + 1.4W	0.5L or 0.5S
5	D+E	0.5L + 0.25S