Ramsay Maunder ASSOCIATES ***Finite Element Specialists and Engineering Consultants***

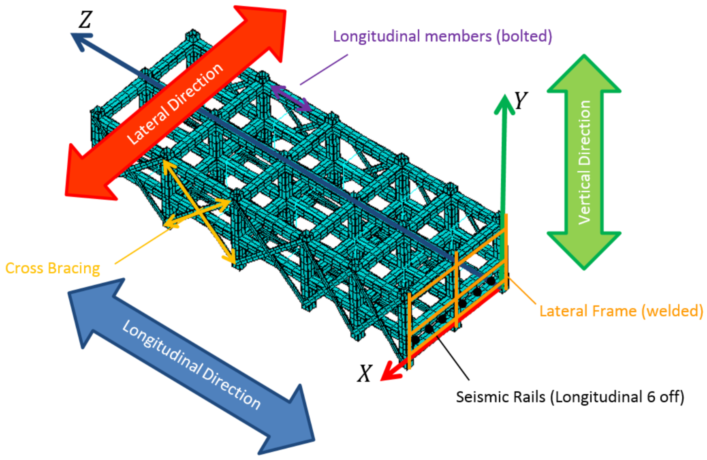
Seismic Assessment of a Structure to EC3

This case study describes a design analysis project in structural engineering undertaken by Ramsay Maunder Associates (RMA) for the seismic assessment of a structure for a nuclear power station to the Limit State Design code EC3. Highly regulated industries, such as the nuclear industry, require many structures to be seismically qualified and a brief description of the essential methodology adopted for this project is here presented.

For a structure to be deemed seismically qualified it must essentially remain intact and functioning as intended after being subject to an extreme example of an earthquake of the type that might occur where the structure is sited. Typically a one in ten thousand year seismic event is used for this purpose.

Whilst the structure could be manufactured and subjected to an appropriate seismic event, say through a shaking table experiment, sufficient tests have been conducted over the years and comparisons made with numerical simulations that such experiments are no longer required. Provided a verified finite element system is used and that appropriate solution verification has been conducted to ensure the results are essentially mesh independent, then finite element analysis provides an accepted way of conducting seismic qualification.

The essential element in seismic qualification is sufficient structural strength and appeal can and should be made to the lower bound theorem of plasticity. This states that provided a stress field can be found which equilibrates the applied loads and does not violate the appropriate yield criterion then the structure is safe. This applies to both structural members and joints and assumes that the material possesses sufficient ductility to redistribute the stresses plastically. In the language of modern limit state design codes of practice, such as the European Code EC3 for steel structures, this is an Ultimate Limit State (ULS) condition.



A finite element analysis of the structure is generally conducted to establish member stress resultants, shear force and moment distributions for the structural model considered here. There is essentially three approaches that may be adopted:

- Equivalent Static Method (ESM)
- Response Spectrum Analysis (RSA)
- Time History Analysis (THA)

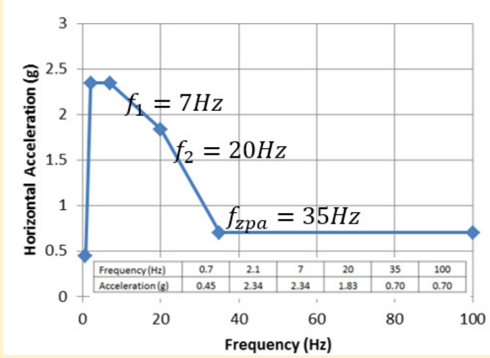
These approaches are discussed in more detail in the NAFEMS publication, ***How to do Seismic Analysis using Finite Elements*** by Cooper, Holby & Prinja.

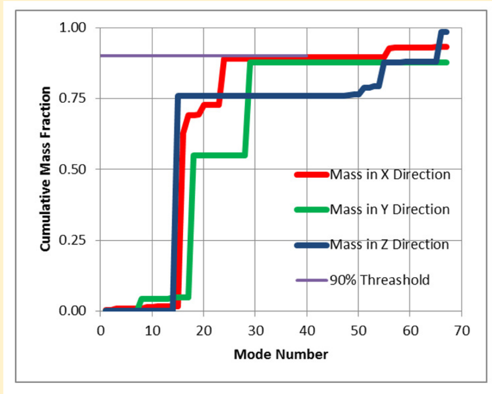
The ESM uses peak spectral accelerations which are then scaled up, usually be a factor of 1.5, and are applied separately in the three coordinate directions. The results are combined spatially to provide the seismic response. The method is simply but often overtly conservative. For structures that behave linearly then the RSA approach can be used. This method requires a modal analysis to be conducted and then the modal results are scaled appropriately for each of the three directions and combined spatially to provide the seismic response for code assessment. The THA approach is the only suitable approach for structures that behave non-linearly through, for example, plastic deformation.

For the structure considered here the RSA approach was adopted but for the purposes of additional verification, and because of its simplicity, the ESM was also performed. The structure considered here is a rack holding battery units for emergency power supply. The batteries include a large amount of lead and comprise the majority of the total mass. The rack is made primarily of square hollow section (SHS) of grade S275 which are either welded or bolted together.

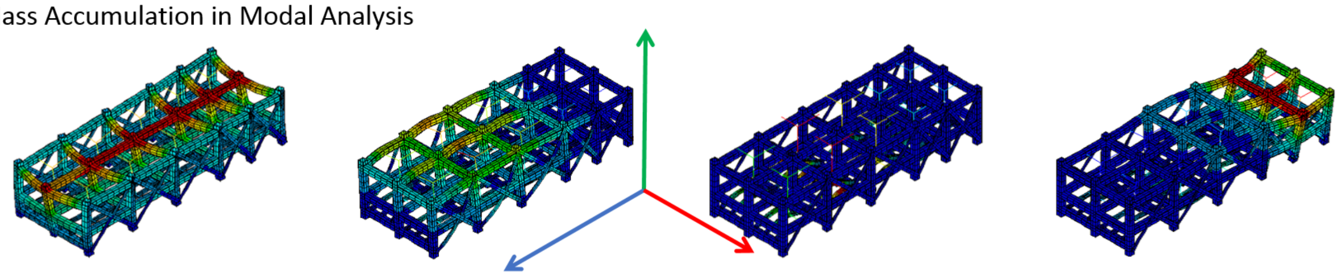
In performing an RSA it is important that the frequency range considered is sufficient to capture the around 90% of the modal mass and inertia. For this structure this meant a frequency range up to around 100Hz.

For any dynamic analysis, such as an RSA analysis, it is essential to verify that both the mass and stiffness of the structure is captured correctly. The mass, inertia and location of the centre of mass were calculated manually and compared with the FE model and to ensure that the stiffness was captured accurately the elements were sub-divided to confirm that member forces and moments did not change significantly.





Mass Accumulation in Modal Analysis



Mode 15 (76% of mass in Z direction) Mode 16 (61% of mass in X direction) Mode 18 (50% of mass in Y direction) Mode 24 (16% of mass in X direction)

Having produced a verified set of member forces and moments these were then used in an EC3 code assessment with unit partial factors on both material and loading. The utilisation ratios for the different types of structural members were calculated and found to be less than 1/3 for the RSA. With the ESM the utilisations were significantly greater, as expected from the conservative nature of this approach, but still less then unity.

It is to be noted that to ensure that the forces and moments extracted from the finite element system formed an equilibrating set, nodal values were taken rather than those extrapolated from element integration points. This is discussed further in NAFEMS Benchmark Challenge Number 7, January 2017 edition of the Benchmark Magazine.

RMA undertakes such work for many clients and can be contacted at http://www.ramsay-maunder.co.uk/