Supplied and sponsored by Oasys **Industry CPD**

# Good FEA modelling for static linear analysis

**This CPD module, sponsored by Oasys, examines the use of static linear analysis for modelling building and bridge structures and how to generate realistic, accurate results.**

**Continuing professional development (CPD) ensures you remain competent in your profession. Chartered, Associate and Technician members of the Institution must complete a specified amount each year. All CPD undertaken must be reported to the Institution annually. Reading and reflecting on this article by correctly answering the questions at the end is advocated to be:**

**1 hour of verifiable CPD**

## Introduction

*'To err is human. To really muck it up you need a computer.'*

Anon.

Finite-element analysis (FEA), in its various forms, is perhaps the most common way for structural engineers to analyse buildings and bridges, validate their designs, and automate the huge number of calculations needed on a project. But this automation and convenience comes with a responsibility: computer models may give you an answer, but you need to ensure that it is the right answer to the right question[1].

While there are dedicated building and bridge modelling programs, with FEA you can model almost any shape structure under almost any load, and get answers out to 16 decimal places[2]. Building a model that accurately represents the pertinent aspects of the structure in accordance with the analysis is a crucial step in finding useful answers.

## Static linear analysis

Static linear analysis is the most commonly used FEA option for building and bridge structures: even for advanced FEA programs such as Oasys GSA[3] that include more sophisticated options, static linear accounts for about three-quarters of the analysis runs[4]. It is what we are taught at university and is behind the standard beam formulas and deflection limits, but it carries assumptions that are not applicable to other analyses:

### Geometric linearity

- Equilibrium is established at the undeformed position.
- The force–deflection relationship is linear.

This is correct if the deflections are small enough.

### Material linearity

- The stress–strain relationship is linear (e.g. doubling the stress doubles the strain).
- The material has the same properties in tension and compression.

This is correct if the materials are elastic. For example, steel behaviour is linear if the stresses are not high enough to yield it.

### Principle of superposition*

- The action–reaction relationship is linear.
- Negative actions produce opposite reactions.
- The load path is the same if the load reverses.

Compression-only and tension-only aspects of a structure are non-linear but can be treated as linear in some circumstances. For example, a plank resting on two supports will behave linearly if the load is such that it always remains in contact, but as soon as one end lifts off the supports then it becomes a mechanism. Any linear check of such a structure will either need to factor and combine all loads together in the analysis or check the combined reactions to ensure that no lift-up has occurred.

Another example is that of cross-bracing **(Figure 1)**, where the load path (which brace and other members are loaded) changes with the wind direction. Here the wind loads in each direction must be analysed separately.†

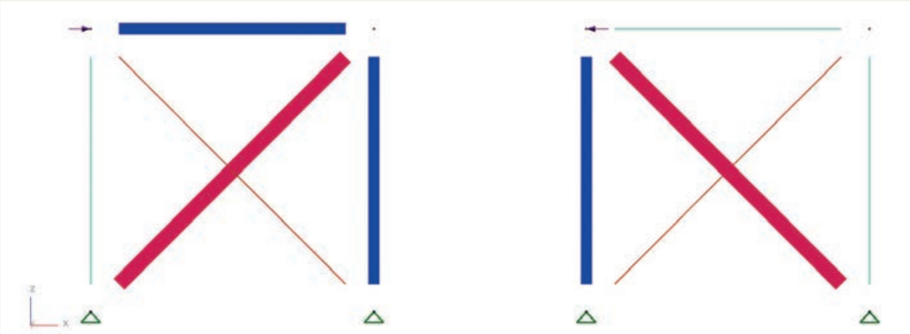
If these conditions are met then you can analyse each load separately, factor and combine the results together as needed. If not, then you need to combine the load cases within the analysis itself.

### Stiffness of supports

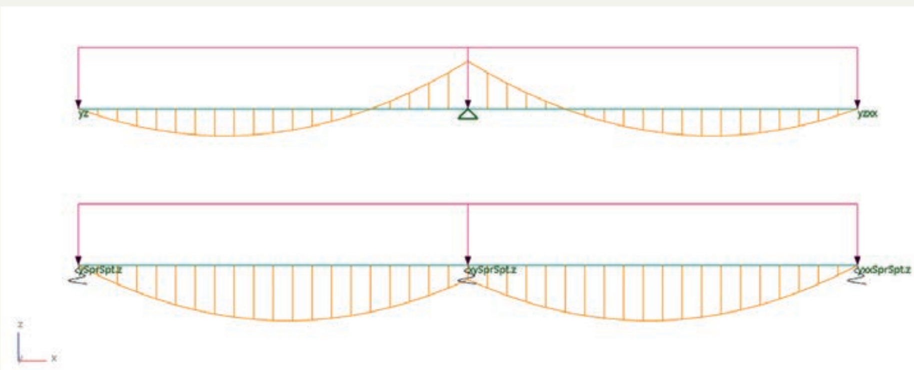
Standard structural examples tend to have infinitely stiff supports, whether pins or fixed. This is OK if the structure is statically determinate, but inaccurate if it is indeterminate

* Note to fellow dyslexics: superposition not supposition.

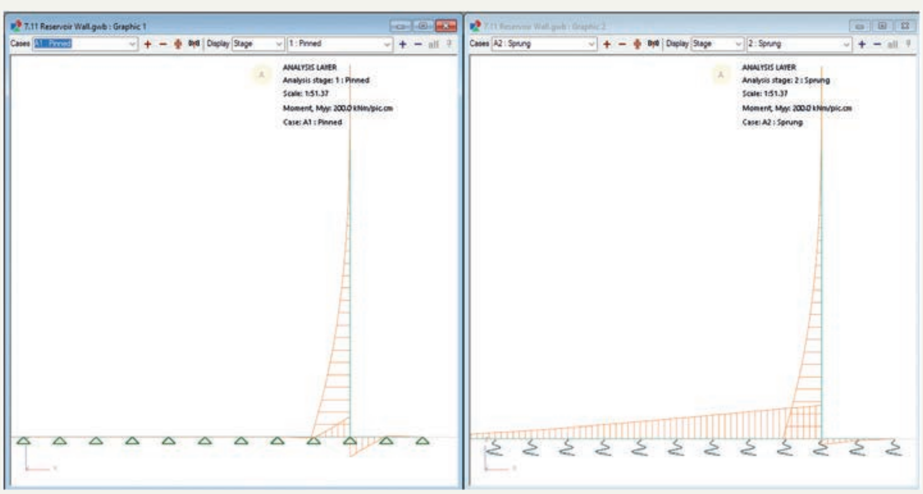
† Note that not all linear solvers can analyse tension-only elements. In such a situation you need to either model just the brace in tension or use a nonlinear solver.



↗**FIGURE 1:** Load path changes due to lateral loads[1,3]



↗**FIGURE 2:** Infinite and finite stiffness supports[1,3]




↗**FIGURE 3:** Pinned and sprung bases[3]

(e.g. a multi-span beam or moment frame).

To demonstrate the effect, consider the two-span beam and compare its bending moments when the central support has infinite stiffness (a pin) and finite stiffness **(Figure 2)**. In reality, all supports have a finite stiffness, but how significant this effect will be depends on the difference in stiffness between the structure and the supports.

One situation where spring supports are essential is when the structure is continuously supported on the ground, such as a raft or ground-bearing slab. Using pinned supports in such a situation will give nonsensical results **(Figure 3)**.

## Connections and releases

Multistorey steel frames in many countries have connections that prevent moments being transferred. To model this, programs allow you to put releases on the ends of the beam elements, but what kind of releases do you need?

Applying major-axis releases is obvious, and minor-axis can be useful too, but what about torsional releases? Universal beams are not good with torsion moments, so some think that you should release the torsion at both ends of the beam to prevent such moments from occurring. The problem with this is that a member with zero torsional restraints has zero torsional stiffness, and this means that the analysis will fail*. Some suggest that you should apply the release at one end of the member, but this can become a major bookkeeping exercise, with plenty of scope for error.

The answer is to not apply torsional releases! Real steelwork connections do give a torsional restraint; the section design relies on it.

But what if torsional moments do occur? The answer here is that you should always check for torsional moments in your structure. There have been a number of collapses due to torsion, the Hartford Civic Center collapse being a prominent example[5]. These torsions can for a variety of reasons, including detailing or connection eccentricities **(Figure 4)**, moment continuities, modelling errors, and cladding loads. If there is torsion, then make sure you understand why it is there, and then address it, either by designing it out or detailing for it.

It is the same with baseplate connections. If there is more than one holding-down bolt, then you have a torsional restraint. In a model, a pinned base combined with minor-axis releases on the beams above means that the column has zero torsional stiffness, which can stop the analysis. The solution is usually to include a restraint in the vertical rotational direction. But sometimes columns and props are not vertical: what then? Here you can apply a fully fixed restraint and apply major- and minor-axis rotation releases on the bottom end of the column element.

## Offsets

How should the beams and columns be modelled?

In the 3D CAD model, the structural members are positioned accurately, ensuring that designs are coordinated with the project partners and that the structure is built correctly†. The FEA model, on the other hand, is there to correctly model the behaviour of the structure, meaning that dimensions and other details need only be sufficiently accurate to achieve that. But be warned: over-precision can lead to analytical problems.

Take, for example, the positioning of the floor beams: in the CAD model the top of steel or top of concrete are essential details, but the FEA beams are usually modelled about their centroids. Does this mean that we should make use of any offset options our software may give and model the FEA beams about their top flange? The answer is generally no, as this will create large axial forces in the beams that are not present in the real structure **(Figure 5)**‡.

Should we then adjust the vertical location of the beams so that their centroids connect at the 'correct' level on the column? This is a

* To be technical, the stiffness matrix cannot be inverted.

† That is the theory anyway.

‡ Some models, such as for footfall dynamics, do benefit from offsetting the slab and beams to generate the composite action.

more subtle problem, but again no. This will create many very short stiff elements that will reduce the accuracy of the analysis. Also, the purpose of many models is to determine the beam sizes, so you may not yet know where the beams should be. I would usually model the beam centroids at top-of-steel level as it is the easiest option to check and the floor-to-floor dimensions will be reasonably accurate, but deep beams, moment or eccentric connections can mean that a more accurate approach is necessary. Another problem is the length of the column down to the foundation, which will need careful consideration.

### Stability

All structures need to be stable*, and so do FEA models. By default, FEA models have six degrees of freedom, which means that they can translate in any direction (the three Cartesian directions: X, Y & Z) and also rotate in any direction (about the X, Y & Z axes). A model must have sufficient restraints and stiffness to prevent a rigid-body motion in any of those directions, otherwise the underlying mathematics of the analysis fails.

For example, even if you only have vertical loads, a model still needs restraints in the horizontal directions.

Analytical programs or FEA templates geared to a partial model, such as a single beam or frame, will include additional restraints (or reductions in the degrees of freedom, as appropriate). For example, a portal frame or truss model built in the X Z plane will need restraints in the Y direction **(Figure 6)**. An easy mistake to make is to model such a frame in 3D space and forget to apply the restraints yourself. After all, a real portal frame is stabilised out of plane, via eaves and ridge beams back to the braced bays. This means that your portal model needs some nodes restrained out of plane.

Full frames also need restraint. Multistorey moment frames are popular in software demos because they are self-stabilising, but pin-ended beams require a floor diaphragm to stabilise the beams and carry forces to the stability system: the bracing, shear walls, or core. In the real structure, this diaphragm might be the concrete slab, connected to the beams and/or columns, or a series of beams connected to plan bracing (essentially a horizontal truss), such as you may find in a roof structure.

Working with just 1D elements, the modelling of the plan bracing is simple, but a concrete slab is, at least at a distance, two-dimensional. If you want to understand the flow of forces within the floor and the relative movement between its parts, then you will need a mesh of 2D elements. If you assume that the slab does not contribute bending stiffness to the beams, then you need to use 2D elements that have no bending stiffness of their own, such as plane stress elements.

If you are only concerned with getting the forces to the stability system, then you can use a rigid diaphragm/constraint. These prevent any relative movement between nodes in particular

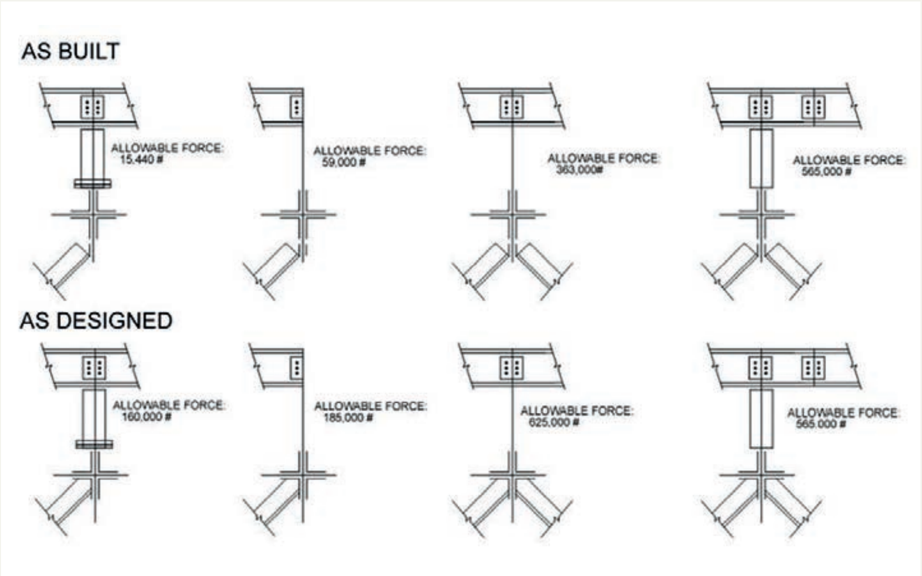





**FIGURE 4:** Hartford Civic Center truss connections[5]

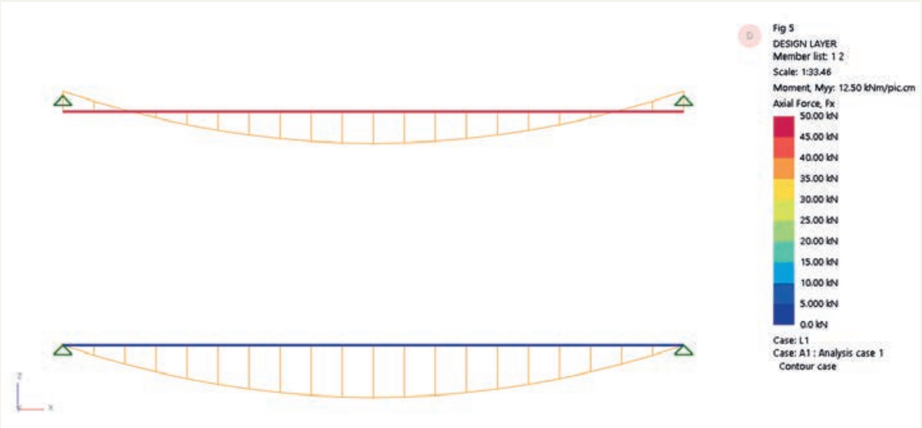



**FIGURE 5:** Effects of offsets on moments and forces[3]

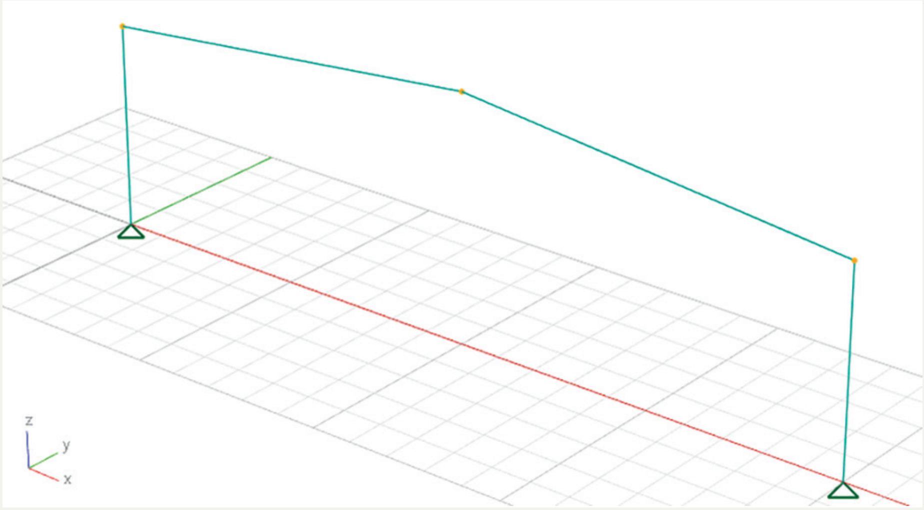



**FIGURE 6:** 2D frame needing 3D stability[3]

* Though the requirement is apparently often forgotten in the IStructE Chartered Membership Exam[6].

directions, essentially giving infinite stiffness. As you do not want to inhibit beam bending, you would normally define a floor rigid constraint to act only in the horizontal direction.

Rigid diaphragms generally work well, but they do have their problems. For example, if your model contains a transfer truss and your floor has a rigid constraint that includes two or more nodes in the truss, then you will not get any forces in that chord of the truss. Similarly, a rigid diaphragm will hide the beams' axial forces in braced bays, risking under-design.

While some promote one model to cover all analyses, the reality is that you often need multiple models. Dynamic and buckling analyses often involve radically different assumptions and modelling approaches to static analysis for section design. You may also want to model a connection at a level of detail inappropriate for a more global model. Always consider the purpose of the model and what answers you need from it.

## Checking

We all make mistakes, but mistakes in our line of work can be deadly, so all analyses and designs must be checked. There are guides on good practice for validation and verification[1,7–9], so the question here is: how can we model to make checking easier?

The principal answer is *documentation*. It is much easier to check what is recorded rather than work out what is missing, so state what the design covers and what it does not. Before you start modelling, record your assumptions on how the structure will work, including how it is stabilised, what are the horizontal and vertical load paths, what are the significant actions on the structure and their magnitudes, and which ones you are ignoring. This allows the checking engineer to decide whether that is appropriate or not.

The better answer is *self-documentation*[*]. When you are building your model, don't leave the load cases with just numbers, give them names so that others can understand instantly what they represent. Similarly, when you factor the loads, use an explicit multiplier where possible so that it is clear that you are using the appropriate value. Make your model easy to check, minimise the opportunity for mistakes, and maximise the opportunity to catch any that slip through.

* Something is self-documented if it is obvious what it is or represents.

## Further reading

We've only space to consider a few things here when creating an FEA model; for more suggestions, see *Computational engineering*[1], especially chapters 5 and 7.

## Conclusion

While many consider that FEA, in its various forms, is a guaranteed way to get accurate results, the truth is: the way that we model the structure has a massive impact on how realistic and accurate the results are.

Accurate, realistic results are the enabler for robust and efficient designs, which is what we, as professional engineers, must deliver to our clients. Good modelling is an essential part of that. This starts with a good model.

## REFERENCES


**1) Debney P. (2020)** *Computational engineering*, London: IStructE Ltd

**2) Debney P., Hendry S. and Kannan R. (2023)** 'Industry CPD: Accuracy and precision in finite-element analysis', *The Structural Engineer*, 101 (11), pp. 62-65; https://doi.org/10.56330/HLNJ3016

**3) Oasys Software (2025)** *GSA – Structural Design & Analysis Software* [Online] Available at: www.oasys-software.com/products/gsa/ (Accessed: July 2025)

**4) Oasys Software (s.d.)** Unpublished monitoring of GSA analysis runs within Arup

**5) Martin R. and Delatte N.J. (2001)** 'Another look at Hartford Civic Center Coliseum collapse', *J. Perform. Constr. Facil.*, 15 (1), pp. 31–36; https://doi.org/10.1061/(ASCE)0887-3828(2001)15:1(31)

**6) Examinations Panel (2025)** *Published critiques* [Online] Available at: www.istructe.org/training-and-development/membership-exams/published-critiques/ (Accessed: July 2025)

**7) MacLeod I.A. (1988)** *TN133: Guidelines for checking computer analysis of building structures*, London: CIRIA

**8) Debney P. (2018)** 'Avoiding common mistakes in FEA modelling' [Lecture] [Online] Available at: www.istructe.org/resources/training/lecture-avoiding-common-mistakes-fea-modelling/ (Accessed: July 2025)

**9) Lockley Z. and Williams K. (2024)** 'Avoiding modelling errors: validation and verification', *The Structural Engineer*, 102 (10), pp. 21–23; https://doi.org/10.56330/JENP1295


# Questions

To claim your CPD certificate, answer the questions online at **www.istructe.org/industry-cpd**.
The module will close on 30 September 2025.

**1) When might a non-linear analytical approach be needed? (Select all that apply.)**
a) When structural members can only take tension or only compression.
b) When deflections are large.
c) When stresses exceed the material yield limit.
d) Whenever I have access to a non-linear analysis program.

**2) If my model only has vertical forces, why might it need horizontal restraint?**
a) Because it is good practice.
b) Because the maths needs the restraints to solve.
c) Because it might slide sideways.
d) Because my university lecturers told me it does.

**3) Why might moments be redistributed in a multi-span beam (Select all that apply.)**
a) Because of yielding and cracking.
b) Because of support settlement.
c) Because we made a mistake in our calculations and need to justify that they are still OK.

**4) How should you deal with torsion in beams?**
a) Avoid it by adding a torsional release at both ends of the beam.
b) Replace all UB sections with RHSs.
c) Ignore it, it is just an analysis artifact.
d) Check which beams have torsion and ensure that you know why.