Case Studies Demonstrating Industrial Usage of Engineering Analysis & Simulation

Angus Ramsay, Nick Stevens, Lukasz Skotny & Jeremy Thaler

Introduction

This document forms the proposal for this book as offered for tender in the NAFEMS Invitation to Tender (ITT), originating through the Education & Training Working Group (ETWG) and published in the October 2017 issue of the NAFEMS Benchmark Magazine.

The book is planned as a collaborative effort between four independent and successful engineering consultants all working in structural/mechanical engineering but on different structural forms. The authors will each present one case study from their particular field of endeavour each making intensive and advanced use of finite element (FE) analysis.

The authors will collaborate in the scrutiny of each other's work to ensure consistency of approach, terminology and presentation. The first author, based on his experience of the PSE certification process and writing for NAFEMS, will act as editor to ensure that the text includes appropriate reference to PSE competencies and is of the overall quality expected from such a publication. He will also provide a single point of contact through the review process and final editing/preparation for publication.

The case studies will aim to provide exemplars of good finite element practice that the engineer beginning his/her career may use to aid the transition from the idealised problems often considered in further/higher education to real practical engineering problems faced in industry. The case studies will be presented in individual chapters and will be cast as design/assessment problems. Suggestions for tutorial problems will be highlighted and potential ways for recasting the studies as failure analysis problems will be identified.

The book will start with a detailed introduction outlining the importance of computational simulation in the modern world, the potential socioeconomic benefits together with the responsibility put upon the shoulders of the professional simulation engineer. The closing chapter will summarise the contents of the book highlighting important practical conclusions that the reader may ponder and take away for further consideration in his/her future career.

To aid course directors and students in using these case studies and reproducing the results presented, much of the information will be supplied in digital form to be downloadable from an appropriate NAFEMS web page. The anticipated page count for the book will be Introduction (15), Case Studies (4x35) and Closure (5) so a total of 160 pages.

Timescale and Costs

Although our aim would be to complete this book within 12 months from the time of commissioning it is probable, as the writing of this book is to be a collaborative effort, that the actual period will be somewhat less than this. The fee for this book is to be £6000 divided equally between the four authors plus an additional £1500 for the first author to cover editing of the book and acting as the interface between NAFEMS and the authors, i.e., £7500 in total.

Contents of the Book

1. Introduction

This chapter will put into proper context the importance of industrial case studies to the education of the engineer beginning his/her career and will include the following sections:

i. ***What does a simulation engineer do?***

- Design of new components/structures.
- Assessment of existing components/structures.
- Failure analysis to understand the cause(s) of a structural failure.
- Important considerations include ***strength***, ***stiffness*** and ***stability***.
- Design optimisation; minimisation of material usage or manufacturing cost.

ii. ***Known/Unknown Theoretical Solutions***

- Strength of materials type solutions where ***known theoretical solutions*** for idealised structures are provided as ***closed-form solutions*** or, for plates, as infinite series.
- FE as an alternative approximate solution method that may be applied to real structures.
- FE should normally be ***convergent*** even if the theoretical solution is unknown.
- The concept of ***software verification*** for confirming that the software is sound and understanding the nature of convergent but approximate numerical solutions.

iii. ***Codes of Practice***

- Embody the corpus of knowledge in the design of a particular structure or component.
- Include the experience of previous structural failures.
- Not generally a statutory requirement but customers may request it.
- Eurocodes – one example of codes which are applicable across national borders.
- Traditionally based on strength of materials approaches with modifications to deal with the difference between idealised and real structures. Generally conservative.
- Modern codes do not preclude the concept of design-by-analysis where the analysis method is some computational technique such as FE or CFD.
- Cannot always protect the engineer if a design steps beyond known bounds – Millennium bridge and wobbly chimneys.

iv. ***The Nature of an FE Approximation***

- Equilibrium essential for any safe structural design – lower bound theorem of plasticity.
- Conventional FE weakens equilibrium between the stresses and the applied loads.
- Equilibrium defaults (traction jumps and spurious body forces), error indicators.
- Generally low fidelity requiring considerable mesh refinement.

v. ***Computer-Aided Catastrophes***

- Sleipner – a case of ***FE malpractice*** and blind adoption of ***FE folklore***.

- Study of past engineering failures (Henry Petroski's books)

vi. ***Errors in Published Texts and Advice Offered to Practising Engineers***

Practicing engineers rely, sometimes to a very great extent, on the advice and data published in engineering text books, codes of practice etc. However, this advice/data is not always correct and the practising engineer needs to be aware that this could be the case:

- Timoshenko Error
- Advice on the bounded nature of FE stresses is sometimes incorrect.

vii. ***Uncertainty Quantification***

Even when a good simulation model has been prepared and appropriate verification checks have been passed, there remains a degree of uncertainty in terms of the structural response as a result of approximations in such as loading/support conditions and material properties.

- The bimetallic strip problem to demonstrate the large degree of uncertainty in the structural response that may result from the low precision of published thermal expansion properties.

2. Case Study Number 1: Assessment of a Bandsaw Pulley

Author: Angus Ramsay, Ramsay Maunder Associates

3. Case Study Number 2: Design of Curved Beams

Author: Nick Stevens, Tor Engineering

4. Case Study Number 3: Design of Steel Silos

Author: Łukasz Skotny, Enterfea

5. Case Study Number 4: Assessment of a Pressure Vessel

Author: Jeremy Theler, Seamplex

6. Closure

A career in engineering simulation can, potentially, be an extremely interesting and very rewarding one. The engineer taking this career route must, though, be prepared for a life-time of learning often described as ***continued professional development*** (CPD). One of the biggest steps the engineer beginning such a career needs to negotiate is the transition between often highly idealised approaches to analysis adopted in his/her education and the practices required for real engineering components/structures. The aim of the case studies presented in this book has been to provide the engineer beginning a career in simulation with exemplars of good finite element practice with sufficient clear explanation to demonstrate how this transitional step may be successfully negotiated.

Authors’ Credentials

The authors of this booklet will be Angus Ramsay, Nick Stevens, Łukasz Skotny and Jeremy Theler. Brief resumés of the authors, each of who have developed successful careers as independent engineering consultants making extensive use of the FE method in different fields of structural/mechanical engineering endeavour are provided below.

Angus Ramsay MEng, PhD, PSE, CEng, FIMechE Angus completed a traditional engineering apprenticeship (ICI) before studying at Liverpool, Exeter and the Robinson FEM Institute with a brief intervening period in automotive R&D. He conducted research in computational structural mechanics at IST in Lisbon (equilibrium methods) and Nottingham (yield-line techniques) which was followed by ten years industrial consultancy in mechanical engineering (turbomachinery). He recently acted as an Independent Technical Editor for the NAFEMS Benchmark Challenge Initiative, is a member of their Education and Training Working Group and a Founding Member of the NAFEMS PSE Scheme. He is a Technical Expert for HKA Global and a board member of IMechE’s Structural Technology & Materials Group.	
Nick Stevens MEng(Hons), CEng, MIMechE, Member of SECED Nick graduated from the University of Nottingham with a First-Class Honours degree and subsequently moved in to engineering consultancy for the defence, aerospace and nuclear industries. He is a highly skilled, motivated and creative Chartered Mechanical Engineer, able to rapidly assimilate new skills and with a diverse background, including modelling, simulation and analysis of gas turbines, nuclear pipework, compressors, steel and concrete nuclear facilities, submarines and aircraft structures. With excellent communication and interpersonal skills and proven technical engineering capabilities in analysis and design. His career focus in the last 4 years has been seismic design, analysis and assessment of nuclear-related structures and equipment, and he is member of SECED.	
Łukasz Skotny BEng(Hons), PhD Łukasz finished civil engineering studies with honours. Just after this he co-founded his first company dealing with the design of structural steel and in parallel started a PhD research project in the field of shell stability. During his PhD he fell in love with FEA design. After his PhD he got a postdoc position in his University where he developed a passion for teaching others. After 8 years he left a company he had co-founded to start Enterfea, where he can focus fully on Finite Element Method calculations and teaching others with courses and his blog. Łukasz has designed structures that were successfully built in over 20 countries on 5 continents.	
Jeremy Theler ME Jeremy is a Nuclear Engineer from Argentina. His main background is the development of calculation codes. Before working in the nuclear industry, he had performed research in two-phase flow instabilities and control of chaotic thermal-hydraulic systems. He worked in the completion of the Atucha II Nuclear Power Plant developing a coupled calculation suite that allowed, amongst other things, to design a better algorithm for controlling the core power distribution against xenon instabilities. He wrote a free and open source finite-element solver from scratch fino and a web-based interface to run FEM analysis directly on the cloud.	

Case Study Number 1: Assessment of a Bandsaw Pulley

Author: Angus Ramsay, Ramsay Maunder Associates

Introduction

The client was interested in increasing the throughput of the bandsaw by increasing the operating speed of the machine and wanted reassurance that the increased loading would not compromise the structural integrity of the pulley wheels leading to early failures and loss of corporate credibility. A typical bandsaw machine is shown in Figure 1 together with the diagram showing the loading seen by a pulley wheel.

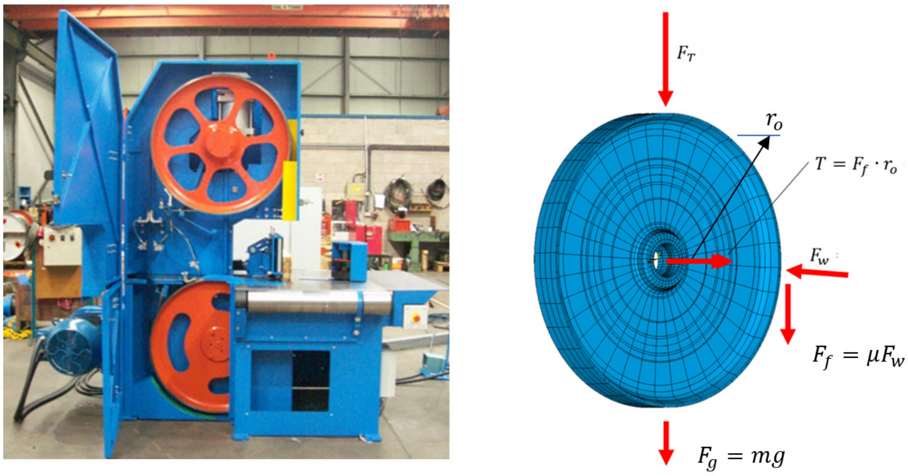


Figure 1: Typical bandsaw machine and loads on driven pulley wheel

The pulley wheel was assessed using stresses generated from finite element analyses and a variety of potential failure modes were considered.

Serviceability Limit States

1) Resonance due to coincidence of the fundamental modes of vibration and the one/rev excitation force – Campbell diagram of Figure 2.

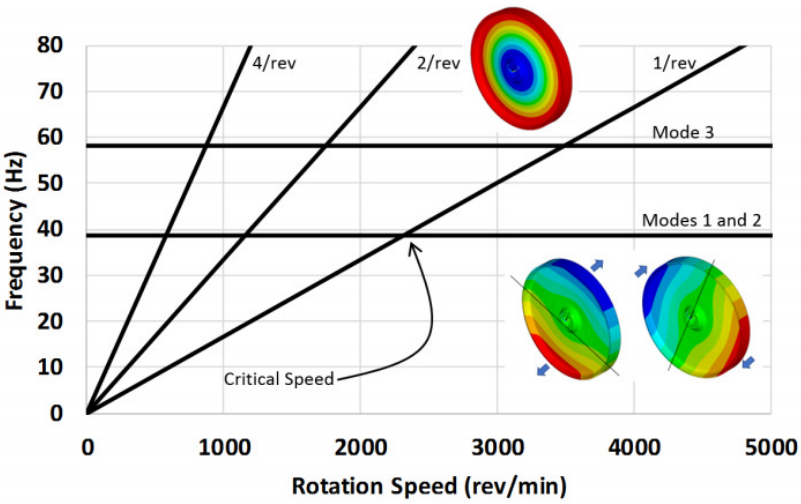
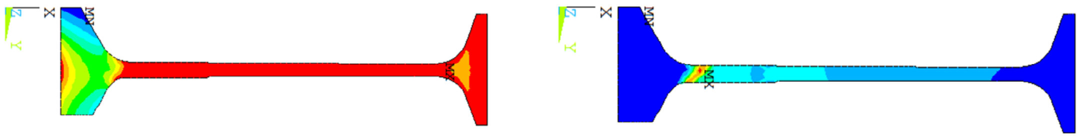


Figure 2: Campbell diagram showing critical speed for 1/rev excitation

2) Loss of the rim crown profile. This profile, which is machined onto the stationary rim, is essential to maintain axial location of the saw band. However, centrifugal load tends to reduce the profile and there is a critical speed at which this profile is lost.

Ultimate Limit State

1) Plastic burst of the pulley wheel. As the pulley material exhibited limited ductility then a check on the total mechanical strain was undertaken to ensure that the wheel had sufficient ductility to allow burst to occur.



Equivalent Stress   Equivalent Strain (total mechanical)

Figure 3: Equivalent stress and strain (von Mises) at burst speed

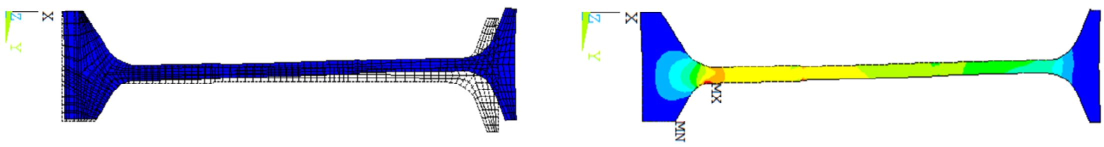
Fatigue Limit State

1) The pulley is loaded in a cyclic manner due to the start/stop cycle of the machine, the axial load applied by the timber being cut and also, to a lesser extent, by the gravitational force field. The material used for the pulley is ferrous with a specified endurance limit. A stress-based fatigue assessment was conducted using the Palmgren-Miner damage accumulation rule.

Design Considerations

Although the assessment was conducted on an extant design of pulley wheel, there is scope for improving the design:

1) **Web Bending**: Bending of the web due to disc mass distribution leads to bending stress (Figure 4) that can be eliminated through re-stacking the disc.



Displaced shape   Radial Stresses

Figure 4: Displaced shape and radial stress at design speed

2) **Web Penetrations**: Two designs of web penetration can be seen in the pulley wheels of Figure 1(a) – six in the upper wheel and three in the lower wheel. The purpose of these is to reduce material usage and loading on the hub and shaft. There is, however, a penalty in terms of stress increase due to the stress concentration around the penetration; stress concentration factor of about 2 for the hoop stresses in the relatively small circular penetration shown in Figure 5.

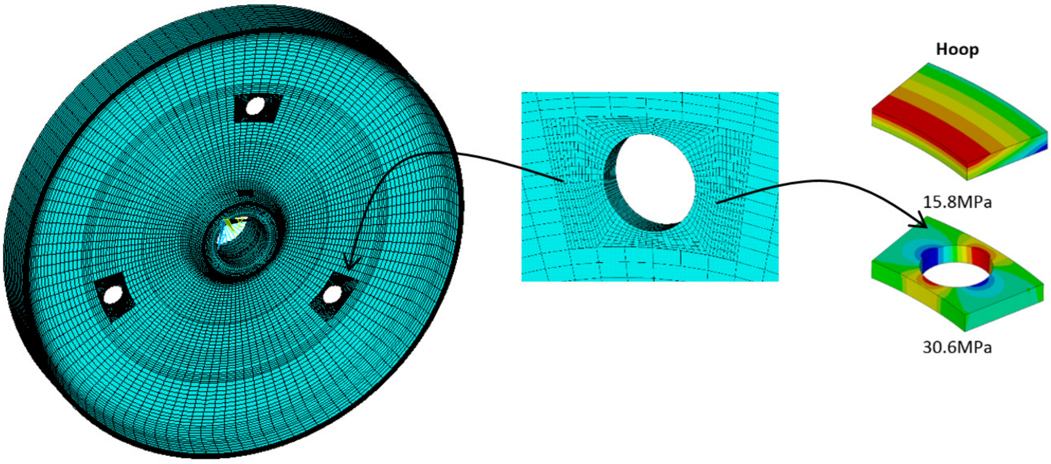


Figure 5: Web penetration with local mesh refinement

There is scope for a constrained optimisation design study considering the parameters of penetration shape, size, radial location and number of penetrations required to minimise material usage under the constraints of retaining sufficient stiffness, strength and fatigue life. This study could be undertaken using a plane-stress model form.

3) **Hub Penetrations**: The main hub penetration is a diagonal hole designed for the application of grease into the bearings. The penetration terminates in the bore where the elastic stress is already high – see Figure 3. This high stress is then concentrated by a factor of more than 3 in the hoop component – see Figure 6. It is this feature that governs the fatigue life of the pulley wheel.

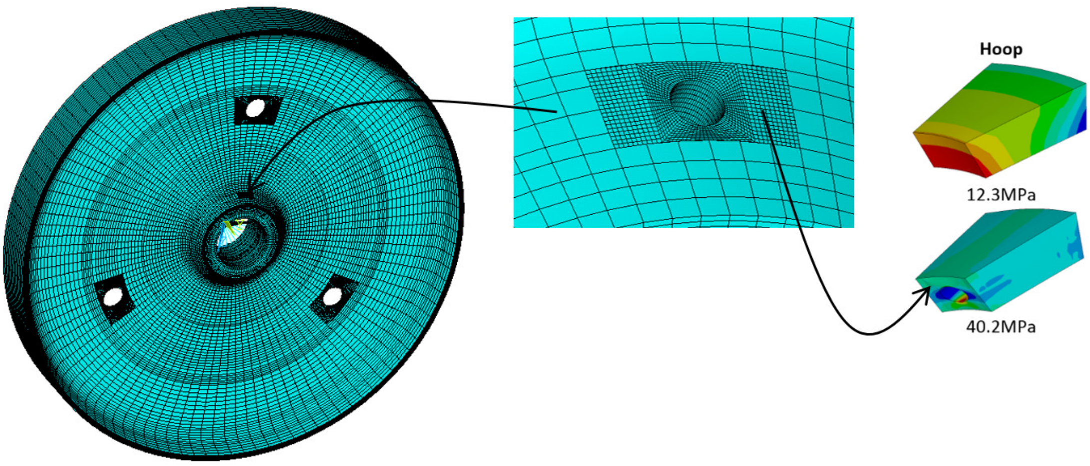


Figure 6: Hub penetration with local mesh refinement

There are other ways in which the grease can be introduced into the bearing housing without impinging on the fatigue life of the wheel and consideration of these would make a good conceptual design study.

Modelling Considerations

The pulley wheel is not strictly axisymmetric in geometric form (web and hub penetrations) or in loading – see Figure 1(b). However, as the non-axisymmetric forms of loading can be demonstrated to lead to small stresses when compared to the axisymmetric centrifugal loading, a reasonable prediction of the structural response can be obtained through an axisymmetric model. Admittedly this model form does not take into account the web or hub penetrations but as these are relatively small features, their influence on the global behaviour will, and can be demonstrated to be, small. Burst speed and natural frequencies will be relatively insensitive to the web and hub penetrations and the level of approximation can be assessed through comparison with three dimensional solid models. Planar (plane stress/strain) models will also be useful for the design of an optimum web penetration where the computational effort may be further reduced through using cyclic symmetry.

In addition to linear-elastic analysis, modal analysis and incrementally loaded analyses adopting elastic, perfectly-plastic material properties together with either the Tresca or von Mises yield criterion, assessment of the fit between the pulley wheel and the shaft or bearings will also be considered. The fit is an interference fit which can be modelled by applying a thermal expansion to the shaft or, alternatively, a thermal shrink to the pulley. This assessment could also include frictional contact and the radial contact force can be used to determine the limiting values of axial force and torque for the fit.

Software Verification

This is essential to ensure that the FE software being used is capable of recovering the correct solution. Problems with known theoretical solutions (benchmark problems) that can be used for this case study include:

1) Elastic solutions (closed-form) for uniform and planar rotating discs are available through the Lamé Equations.
2) Theoretical solutions for the natural modes of vibration should be available for uniform and planar stationary discs. The effects of rotation on the natural frequencies due to stress-stiffening and spin-softening will need to be considered and whilst known theoretical solutions might not be available, there is likely to be some established numerical results of sound provenance that can be used to provide benchmark solutions.
3) Plastic solutions for the burst of rotating discs based on an elastic, perfectly-plastic material model and the Tresca yield criterion are available.

In addition to confirming, or otherwise, that the FE system being used can recover correctly the known theoretical solution to problems similar to the one under consideration, such solution verification problems provide useful information in terms of the level of mesh refinement required for accurate capture of the structural response for the real problem being considered.

Solution Verification

Whilst software verification provides invaluable evidence that the FE system being used can recover known theoretical solution for idealised problems similar to those of the structure being considered, and also the sort of mesh refinement required to achieve accurate solutions, convergence studies must also be undertaken for the real structure to ensure that the generally more complex structural response of the real structure over that of the idealised structure is captured reliably. The pulley wheels, for example, are not uniform or planar in form and this will lead to the development of a third component of stress in the axial direction and also to stress concentrations around changes of section etc.

Codes of Practice

These codes accumulate the corpus of experience and knowledge of the safe design of particular types of structure, e.g., pressure vessels. They should present the best practice in design based on theoretical model of idealised structural forms with safety or partial factors based on the experience of structural failure. They are, rightly, typically overtly conservative and provide engineers with a practical and non-analytical approach to the design of structures. Adherence to codes of practice is rarely statutory but usually enforced either by the customer, through a desire to adhere to 'best practice' or through a regulatory body, e.g., the Office of Nuclear Regulation in the UK.

There are, as far as the author is aware, no codes of practice dealing with rotating machinery. This is perhaps surprising since the potential energy in a high-speed rotating component can, if released due to structural failure, be converted into extremely damaging kinetic energy, e.g., burst of high speed turbine wheels. The energy released in this sort of failure is not too dissimilar to that released by the burst of a pressure vessel containing a gaseous fluid.

In the absence of prescribed and codified approaches to the design of rotating machinery, the best approach an engineer can adopt is that of the ASME Pressure Vessel code; this is a very conservative code even down to adopting the Tresca yield criterion over the more realistic yet generally less conservative von Mises criterion. Pressure vessels do bear some similarity to rotating machinery in that they might often be idealised as axisymmetric structural forms and it would not appear too unreasonable to adopt similar approaches at least for failure modes, allowable stresses, and design approaches.

The modern practice of 'design-by-analysis' where the analysis is undertaken by finite element analysis leads, often, to a difficulty in assessing the FE results against codified values for quantities such as allowable stresses. Whilst the codes do not exclude the possibility of using FE to generate stresses for use in code assessment, FE codes do not always assist the engineer in presenting the results in a manner that might be used directly in the code. This means that engineers have to manipulate the results and have to do this correctly in order to ensure a safe structure. Stress linearisation is one approach often used in the analysis of pressure vessels and the opportunities for FE malpractice are rather large when inexperienced users attempt to do this.

Sub-Modelling

The web and hub penetrations are modelled using highly refined local regions embedded into the more coarsely refined global FE model. The same model could be used to demonstrate sub-modelling whereby boundary conditions for the cut-faces of the sub-models could be obtained from the global model in which the sub-model feature is not modelled. The two approaches will lead to different results for the stresses around the penetrations and it would be interesting to compare and contrast the different results.

Validation and Uncertainty Quantification

The FE results, which should, since verification has been undertaken, be a good approximation to the theoretical solution for the problem, i.e., the FE result should be a good surrogate for the theoretical solution. The process of validation requires checking that the structural response as measured on the real component matches up with the theoretical solution or, in this case, the FE solution. Although measured results are not available for the pulley wheel being considered in this case study, there is no reason why a validation program cannot be developed and discussed for this component. One reason that measured results may differ from theoretical values is a result of differences in idealised material properties used in the development of theoretical values and those of the actual component.

It is a simple matter of considering how variations in material properties might influence the various structural responses considered in this case study and such an assessment of uncertainty will be undertaken for the pulley.

Proof Testing

It is common practice for the high-speed discs of turbomachines to undergo a proof test prior to going into service. This test is undertaken at a rotational speed higher than the design speed with the aim of causing plastic flow in the regions of high stress concentration. This leads to compressive residual stresses when the pulley is stationary and to a reduction in the mean stress resulting in an enhanced fatigue resistance.

FE analysis can be used to assess the reduction in mean stress and the improvement in fatigue life quantified.

Closure

This case study has plenty of scope for a range of interesting smaller studies that could form tutorials or individual projects. A group project might involve the design of an optimal web penetration with different members of the group considering different parameters defining the geometry of the penetration. Group members would need to interact to exchange data about their local optimum solution so that the landscape of the multidimensional design space could be understood and a global optimum solution achieved.

This cast study, which is cast as an assessment and design problem, can be recast as a failure investigation in a number of ways. For example, the speed could be selected such that resonance occurred or the peak stress is greater than the endurance limit leading to an observed fatigue failure in a given number of start/stop cycles.

Many of the NAFEMS PSE competencies are covered in this case study and will be identified as appropriate.

Bibliography:

[1] http://www.ramsay-maunder.co.uk/knowledge-base/technical-notes/dimensional-analysis--numerical-experiments-for-a-rotating-disc/

[2] http://www.ramsay-maunder.co.uk/knowledge-base/projects/design-against-burst-of-rotating-discs/

[3] http://www.ramsay-maunder.co.uk/knowledge-base/projects/nafems-benchmark-challenge/nbc-number-3/

[4] http://www.ramsay-maunder.co.uk/knowledge-base/projects/nafems-benchmark-challenge/nbc-number-8/

Case Study Number 2: Torsion of Curved Beams

Author: Nick Stevens, Tor Engineering

The I-beam is such a common structural form, yet it is a torsionally weak form and therefore additional design and analysis considerations are required. This is not always apparent to practicing engineers and yet is important and worthy as a topic to cover in some detail. As well as more obvious sources of torsional loads on I-beams, such as eccentric forces, torsion is also generated for beams that are curved on plan under the action of vertical forces acting through the centroid. There are design guides for appropriately designing for torsion (and the associated warping), but care is required in the analysis, where warping is an optional (generally non-default) 7th degree of freedom for beam elements. The beam may also be modelled using shell or solid representations.

This case study covers practical methods for design and analysis of open sections under torsion, in particular a curved on-plan beam acting as a crane runway beam, and supported from above from a super-structure via vertical hangers. The load case is static with dynamic crane loads due to seismic accelerations being considered in a quasi-static manner.

Introduction

It is generally well-understood that open sections (for example I-beams) have poor torsional stiffness. Therefore, the design intent is, typically, to avoid torsional loads. This is because torsional warping of open sections is a challenging subject where the structural resistance is limited by the open form factor, yet it is an important area of structural engineering as torsional loads are common. In particular, and perhaps not immediately obvious, beams curved on plan (see Figure 1) experience torsional warping, where torsion is induced due to the curved path that the beam follows.

This case study considers a beam curved on plan, subject to static and dynamic loads, and an appraisal of design and analysis methods to appropriately capture the torsion that is important for this load case. The curved beam in this example is a crane runway beam, suspended by hangers from a roof structure – see Figure 1.

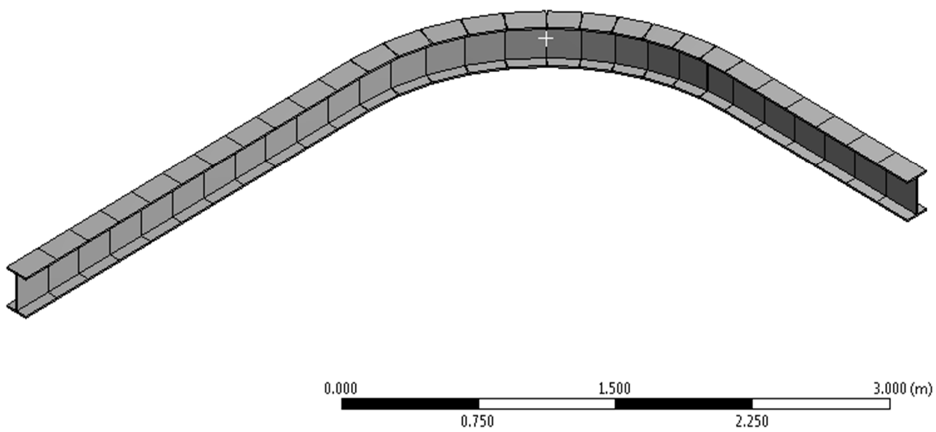


Figure 1: Beam Curved on Plan

Serviceability Limit State

To obtain accurate deflections for assessment of the serviceability limit state, accurate consideration of the torsional stiffness is necessary. To support factory acceptance testing the predicted deflections are obtained. This is also used to confirm the arrangement has sufficient **stiffness**.

Ultimate Limit State

The ultimate limit state considered is the dynamic load due to seismic accelerations. In this case this is considered using a quasi-static approach applying inertial accelerations. The validity of this approach is confirmed by reviewing the natural frequencies of the arrangement. Consideration of buckling and (plastic) bending capacities ensures the arrangement has sufficient **stability** and **strength**. The margin to plastic collapse is also considered to understand this particular failure mode.

Fatigue Limit State

In this particular industrial example, the number of cycles was very low and therefore fatigue does not need to be considered.

Design and Analysis Methods

There are a number of methods to consider torsional warping of open sections, as follows:

- SCI P281, Design of Curved Steel by The Steel Construction Institute [1]
  - This method suggests an I-beam represented with beam elements for the flanges and shell elements for the web to explicitly capture the warping
- Analysis using beam elements and a 7th warping degree of freedom
  - This method gives an additional output called the warping bimoment
  - Guidance on how to deal with the warping bimoment output will be discussed, with reference to SCI P057, Design of Members Subject to Combined Bending and Torsion, The Steel Construction Institute [2]
  - Note that the 7th degree of freedom is often deactivated as the default behaviour in most software packages
- Analysis using shell (or solid) elements
  - Warping is explicitly captured
  - Results are more difficult to process as you cannot readily obtain component actions like major axis moment, axial force and warping bimoment

The methods above will be compared and contrasted for the industrial example outlined above, including comparison to an “incorrect” model without appropriate warping stiffness. The I-beam form is assessed for lateral-torsional buckling, including the additional stresses induced from torsional warping. Assessment to combined bending and torsion is given in SCI P057, Design of Members Subject to Combined Bending and Torsion, The Steel Construction Institute [2]. The structure is a 3D continuum but the validity of representing this using beam and/or shell elements is reviewed and quantified. Further information and comparisons on the additional warping degree of freedom as used in ANSYS is reported in [3].

The ANSYS program is used as a representative finite element software package that is often used in industry and in this case the warping of open sections is represented with beam elements (BEAM188) by activating a 7th degree of freedom for warping. But care is required as the default is to **not include** a warping degree of freedom in the element formulation, and further using the ANSYS Workbench interface there is no easy way to list the element defaults and options and nor is it possible to include the warping degree of freedom for beam elements interactively. These pitfalls will be overcome in this case study.

Supports

As the curved beam is support from a roof structure via hangers, restraints are included in the model. As the fixity provided by the hanger restraints follows the curve of the beam, nodal rotation is applied to rotate the nodes so that the nodal X coordinate aligns with the axis of the beam. The supports are shown in Figure 2.

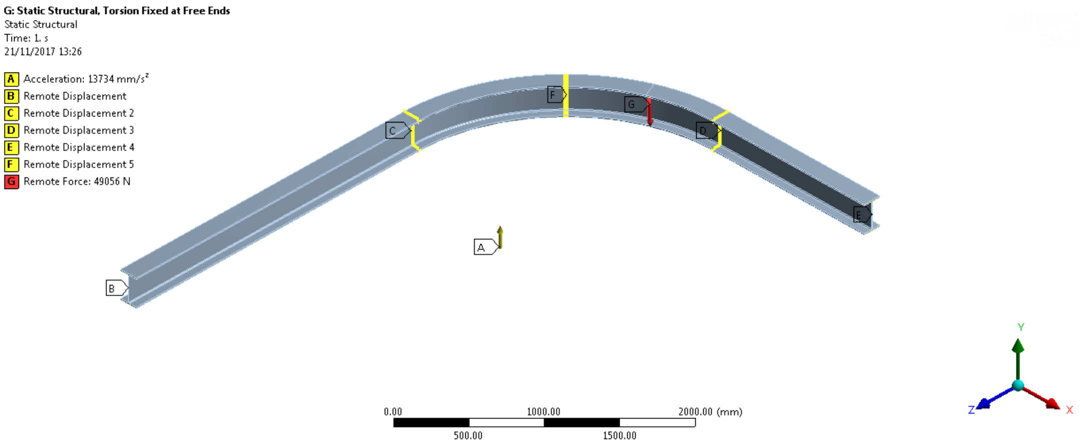


Figure 2: Supports

Post Processing Warping Quantities

Activating the warping degree of freedom will give an additional output quantity - warping bimoment, B. This warping bimoment can be converted to an equal and opposite bending moment, M, in the plane of each flange by dividing the warping bimoment by the depth between the centroids of the flanges, as shown in Figure 3, i.e. $B = M \cdot a$. Typical stresses are shown in Figure 4.

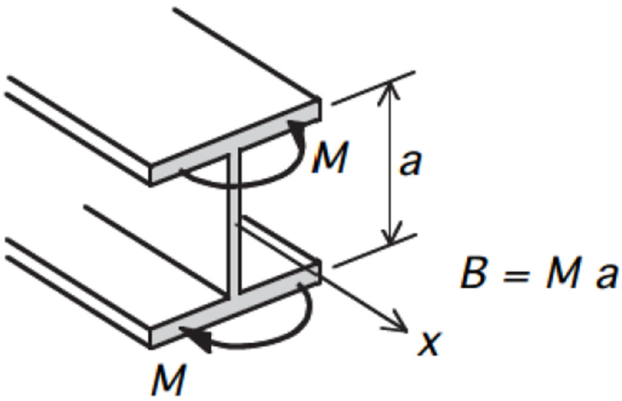


Figure 3: Warping Bimoment

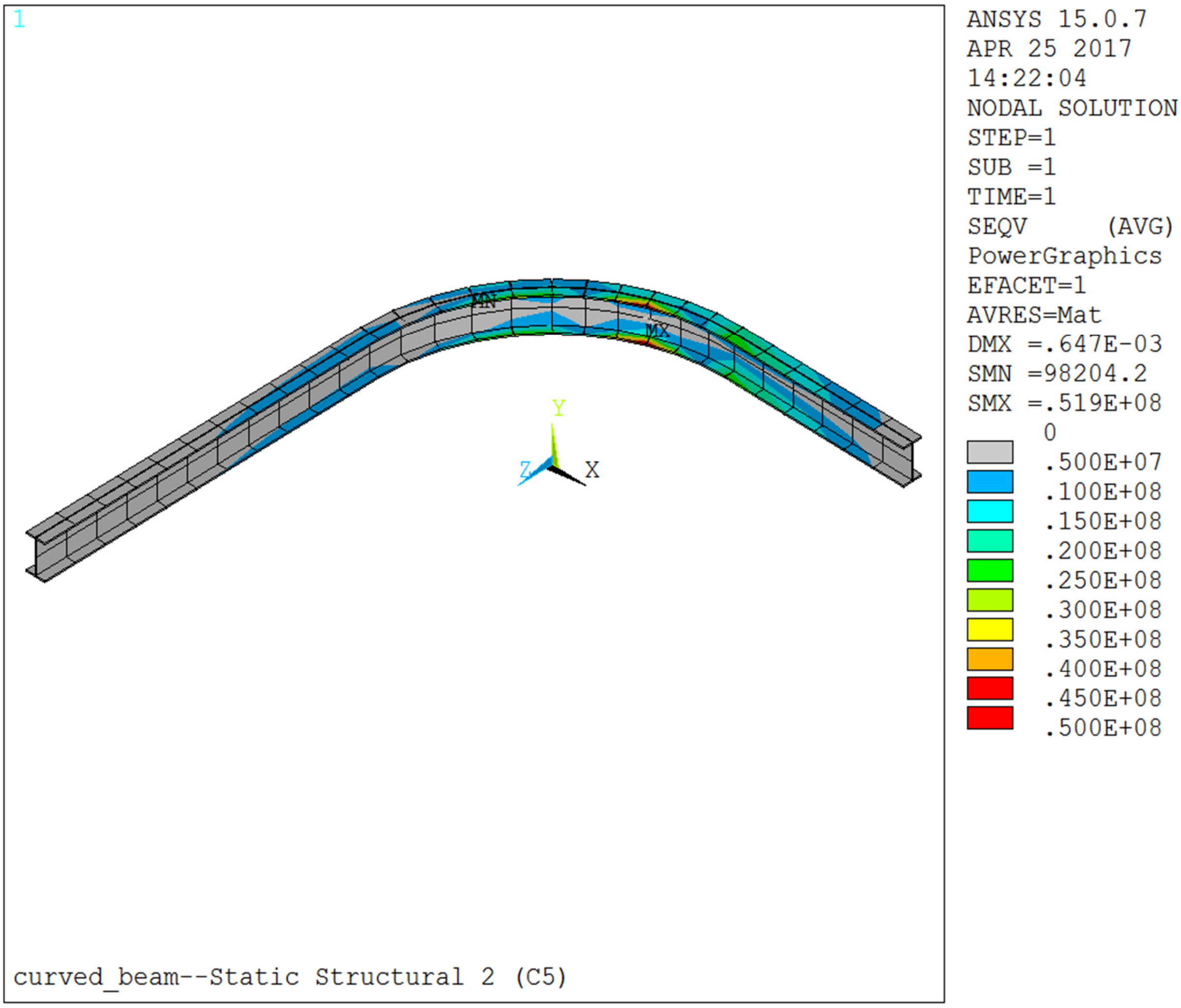


Figure 4: Stress (von Mises) [N/m]

Uncertainties and Sensitivities

The following uncertainties and parameter sensitivities are considered:

- Consideration of the support conditions (fixity) provided by the hangers
- Consideration of restraining the 7th warping degree of freedom
- Mesh convergence studies (both *p*-type and *h*-type refinement is considered)
- Plastic collapse load, using an elastic, perfectly-plastic material model

Verification and Validation

Verification, including uncertainty investigations, is undertaken and ensures that the analysis model is an appropriate representation of the underlying mathematical model. Mesh refinement shows convergence of results and therefore appropriate discretisation. Comparison of finite element representations using beam, shell and solid elements also shows appropriate representation of the mathematical model.

Software verification comparing known solutions for benchmark problems for similar element types and loadings is also conducted.

Engineering Software

Take the time to understand how warping and torsion is included in your finite element software package. If programs do not have the ability to include a 7th degree of freedom then one of the alternative methods above must be used.

Closure

This case study considers the familiar structural form of an I-beam but highlights some design and analysis challenges when torsional loading is present (which it often is, even if that is not the design intent). The case study can be extended to consider the assessment of the hanger bolted joints, and to consider the difference for alternative open sections and structural forms. Students can also investigate the effect of thermal strains on this arrangement.

Many of the NAFEMS PSE competencies are covered in this case study and will be identified as appropriate.

References

1. SCI P281, Design of Curved Steel, The Steel Construction Institute
2. SCI P057, Design of Members Subject to Combined Bending and Torsion, The Steel Construction Institute
3. The Influence and Modelling of Warping Restraint on Beams, Angus Ramsay & Edward, NAFEMS Benchmark Magazine, July 2014

Case Study Number 3: Steel silos design

Author: Łukasz Skotny, Enterfea

Introduction

Silos are a perfect example of why shell design is a difficult task. For many years the only approach to shell buckling was with hand calculations based on Timoshenko's equation. Currently finite element method can also be used, and this approach is recognized in the international standard EN 1993-1-6.

Without a doubt this is a great advancement in shell design, however using FEM in those tasks require great care. Thin shells are very susceptible to buckling which is a highly nonlinear phenomenon. They also show a very big drop in capacity due to imperfections. This means that a careful stability study should be performed before a design can be accepted. Furthermore, nonlinear properties of shell material may lead to an interesting stability failure commonly called "elephant foot buckling".

All of these considerations have to be taken into account in the design and this is the focus of this case study where the practical case of an industrial silo, as shown in Figure 1, will be considered.

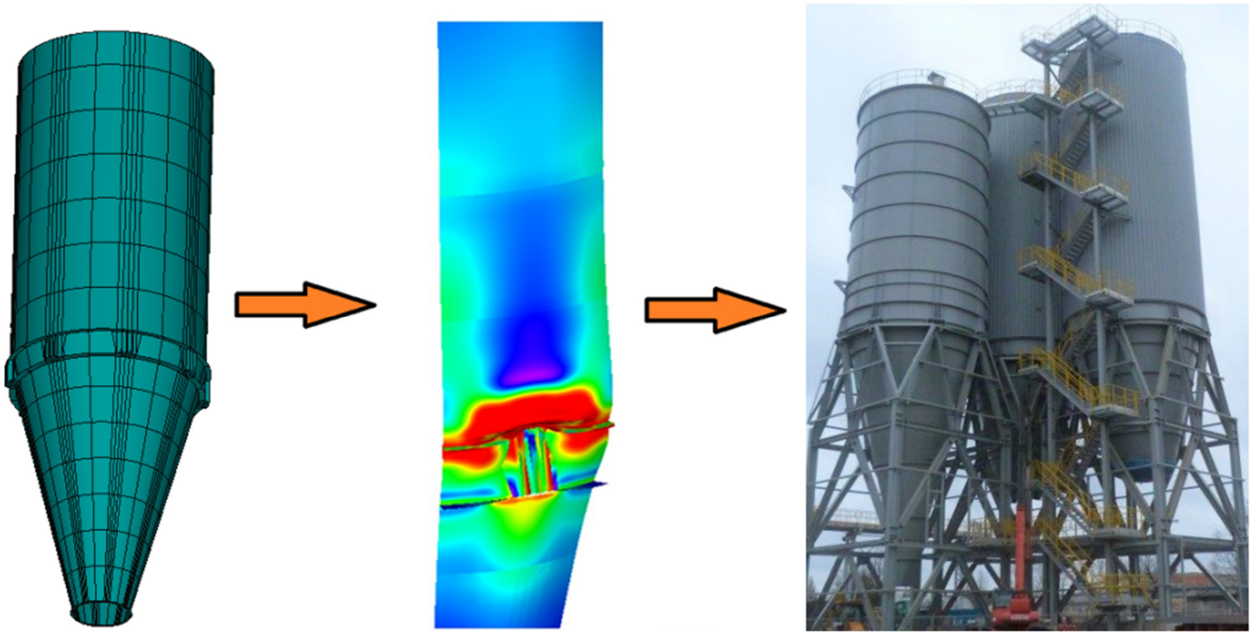


Figure 1: Simplified schematic of silo design

Serviceability Limit State

Deformation of the silo shell is of course important to its capacity. However, initial geometry (prior to loading) of the structure is far more important as the manufacture of such structures lead to distortions and the stability of the structure is rather sensitive to such distortions. It is also subjected to extremely accurate measurements, that can be discussed in the study.

Imperfections of initial shape greatly influence capacity of the shell, which is especially important factor in silos design.

Ultimate Limit State

There are several really interesting phenomena that must be analyzed in shell design. Most silos are supported on columns, meaning there are high stress concentrations in the vicinity of the supports. These areas are susceptible both to plastic collapse and to buckling (as well as a mix of those two effects).

Circumferential tension from material pressure shows some positive influence on capacity up to a certain point. Over this, however, it starts to reduce shell capacity due to interaction between material yielding and stability of the support zone.

What is very interesting is, that several analyses are needed in order to properly understand how the shell will behave (as it varies from design to design). This is an often-overlooked fact in many FEA studies and special care should be demonstrated in analysis of the model, as outcomes from previously performed analysis will influence the choice of imperfections analyzed in the final study.

Fatigue Limit State

Fatigue is not considered in this case study.

Design and Analysis Methods

There are many industrial codes for silos design. Most of them are based on similar approach using Timoshenko's equation. Currently the most popular standard is the EN 1993-1-6.

Several approaches to design can be demonstrated. In this study a full geometrically and materially nonlinear analysis with imperfections will be performed – some results from such a study are shown in Figure 2.

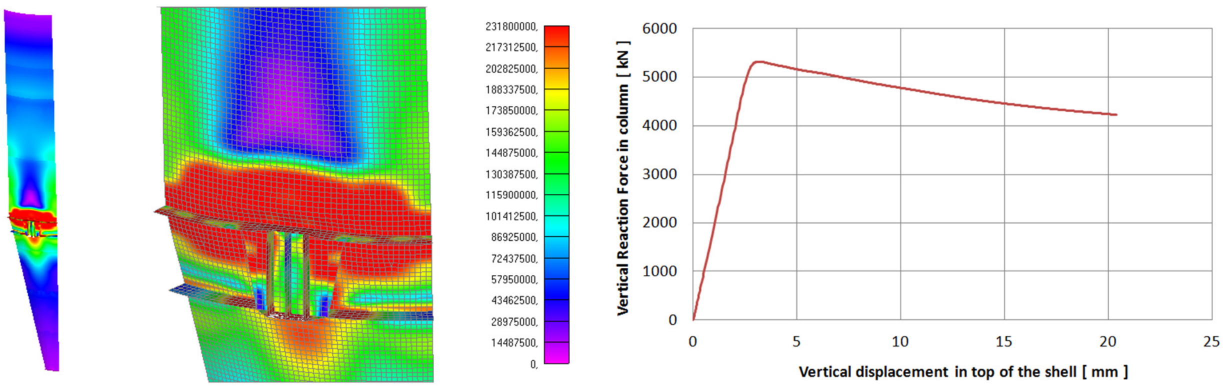


Figure 2: Stability path from one of the nonlinear analysis

Outcomes from FEA analysis and analytical approach will be compared and discussed. Also, several simplified approaches will be shown and discussed that allow for a quick estimate of shells capacity.

In the study FEMAP with NX Nastran will be used as a good example of a typical commercial FEA package popular in industrial applications.

The differences between different numerical procedures will be described, explaining why, in certain cases, convergence won't be achieved. Furthermore, differences between linear and nonlinear buckling analysis will be discussed to show that the linear approach, which is often non-conservative, may not be suitable in many shell analyses.

Boundary conditions

The silo presented in this study was supported on several equally spaced along circumference columns. Discussion will be made on how to model such support system, and why common solutions to this may lead to erroneous outcomes of the analysis.

Furthermore, symmetry boundary conditions will be used as a simplification commonly used to reduce computing effort when so many analyses are necessary.

Uncertainties and Sensitivities

Several uncertainties and parameter sensitivities will be considered:

- Without a doubt, imperfections selected for the analysis will be the biggest unknown and will have the greatest significance in terms of the shell's stability. There are no real guidelines on how to pick imperfections for design like that and this will be discussed and analyzed throughout the study.
- Different boundary conditions schemes will be discussed as well to show how they may influence the capacity of the structure.
- Mesh convergence study will be performed for selected analyses.
- Influence of geometrical and material nonlinearities (and their interaction) will be analyzed.

Verification and Validation

Standard software verification problems will be reported to demonstrate the capability of the FE software to recover known theoretical solutions. All analytical procedures are based on Timoshenko's equation and whilst this requires support conditions that are often far too idealized for the silo shells, it provides a useful solution with which to begin to demonstrate verification of the approach adopted.

The actual structural form and the manner in which it is supported lead to the analytical approach being overtly conservative. More realistic solutions can be obtained through FE analysis and this approach is acceptable in the Eurocodes. This approach does, however, require great care in the analysis which is improved significantly through the development of experience in similar structures.

The process of gaining faith in a numerical (FE) solution in the absence of known theoretical solutions will be discussed.

Engineering Software

Understanding nonlinear capabilities of the software is critical to using FEA in shell design. If the software cannot perform nonlinear analysis, usage of linear bifurcation analysis (commonly referred to linear buckling) may not be a safe approach which is discussed in the study. In such cases conservative analytical approaches are advised.

Closure

This case study tackles a very complex analysis of shells using FEA. Several important issues will be highlighted in this example, leading to higher awareness of the reader about those problems.

Especially influence of imperfections on shell capacity will be discussed along with influence of nonlinear geometry and material properties.

Many of the NAFEMS PSE competencies are covered in this case study and will be identified as appropriate.

References

1. EN 1993-1-6: Eurocode 3 Design of steel structures – Part 1-6: Strength and Stability of Shell Structures

Case Study Number 4: Fatigue Analysis in Nuclear Piping

Author: Jeremy Theler

Introduction

Piping systems in sensitive industries like nuclear or oil & gas should be designed and analysed following the recommendations of an appropriate set of codes and norms, such as the ASME Boiler and Pressure Vessel Code. This code of practice (book) was born during the late XIX century, before finite-element methods for solving partial differential equations were even developed, and much longer before they were available for the general engineering community. Therefore, much of the code assumes design and verification is not necessarily performed numerically but with paper and pencil. However, it still provides genuine guidance in order to ensure pressurised systems behave safely and properly without needing to resort to computational tools. Combining finite-element analysis (even plain linear equations) with the ASME code gives the cognizant engineer a unique combination of tools to tackle the problem of designing and/or verifying pressurised piping systems.

In the years following Enrico Fermi's demonstration that a self-sustainable fission reaction chain was possible, people started to build plants in order to transform the energy stored within the atoms nuclei into usable electrical power. They quickly reached the conclusion that high-pressure heat exchangers and turbines were needed. So they started to follow the ASME Boiler and Pressure Vessel Code. They also realised that some requirements did not fit the needs of the nuclear industry, but instead of writing a new code from scratch they added a new section to the existing body of knowledge: the celebrated ASME Section III [1].

After further years passed by, people (probably the same characters as before) noticed that fatigue in nuclear power plants was not exactly the same as in other piping systems. There were some environmental factors directly associated to the power plant that was not taken into account by the regular ASME code. Again, instead of writing a new code from scratch, people decided to add correction factors to the previous body of knowledge. This is how knowledge evolves, and it is this kind of complexities that engineers are faced with during their professional lives. And, yes, it would be very hard work to re-write everything from scratch every time something changes.

Case study

The spirit of the case study is to provide a brief glance at the differences between the canonical problems students usually learn in College (i.e. the simple ones with analytical solutions that teachers remember by heart) and the actual complicated computations that the industry needs, as illustrated in the introduction. As with a great power comes a great responsibility, it is mandatory for the analyst to understand how the tool he/she uses works. Avoidance of black boxes is a must if someone is willing to sign off an engineering report.

The case study is based on an actual study performed to analyse a set of safety-related piping in a Nuclear Power Plant regarding low-cycle fatigue in dissimilar material welds due to repeated operational transients in the presence of adverse environmental factors. In particular, it looks at how the practising engineer may gain confidence in the finite element solution when the actual problem has no known analytical solution.

1. An infinite cylinder subject to internal pressure (which has analytical solution)
2. A finite cylinder subject to internal pressure
3. A finite cylinder subject to internal pressure and a non-trivial temperature distribution
4. A tee junction – see Figure 3
5. A piece of piping system like the ones encountered in actual plants, with many materials and temperature-dependent properties

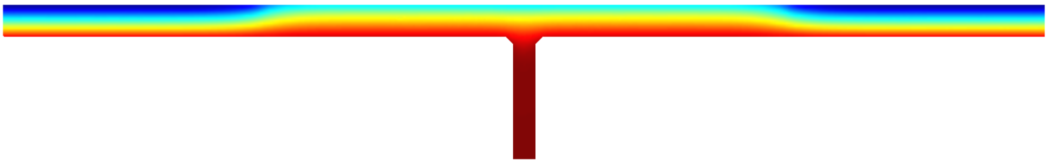


Figure 1: Is a 2D axisymmetric problem enough to model the temperature distribution of a restriction?

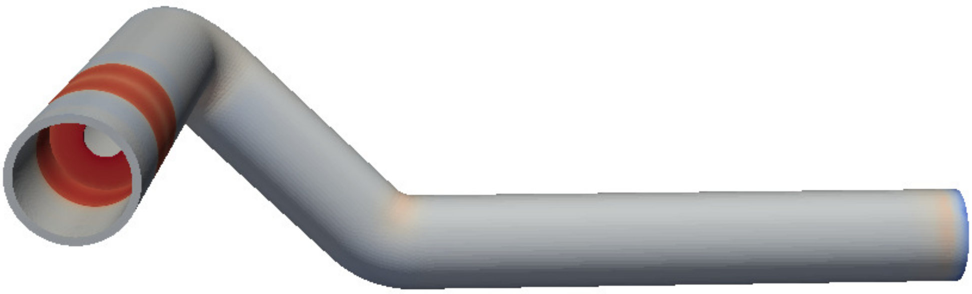


Figure 2: Should we model a full 3D problem to compute stresses?

The finite-element results obtained in each case (i.e. principal stresses, membrane and bending stresses, etc.) during thermal and mechanical transients are finally employed as input data for a low-cycle fatigue analysis. The methodology is based both on ASME Section III and an extension proposed by the US Nuclear Regulatory Commission [2].

Uncertainties and Sensitivities

More often than not, engineers end up performing parametric studies to see how the solution of a certain problem changes when one (or more) input parameters also change by a certain specified amount. This is especially important for a wide variety of reasons. Some of them are purely technical such as design robustness and stability margins. But others are based on more subtle grounds, which are important for the spirit of this case study: to grasp the physics and understand the mathematics behind the finite-element method.

Therefore, the solutions of the aforementioned problems (e.g. the principal stresses or the membrane and bending stresses according to ASME) are compared to the previous simpler problems. Sensitivity and parametric studies are performed including the following variables:

a. dimension of the formulation (axisymmetric or full 3D),
b. mesh coarseness (avoid comparing pears to apples)
c. dominant element shape (triangles/tetrahedra or quadrangles/hexahedra),
d. element order (first or second-order elements),
e. distance from the ideal geometry (i.e. length of the finite cylinder, diameter of the tee bifurcation)

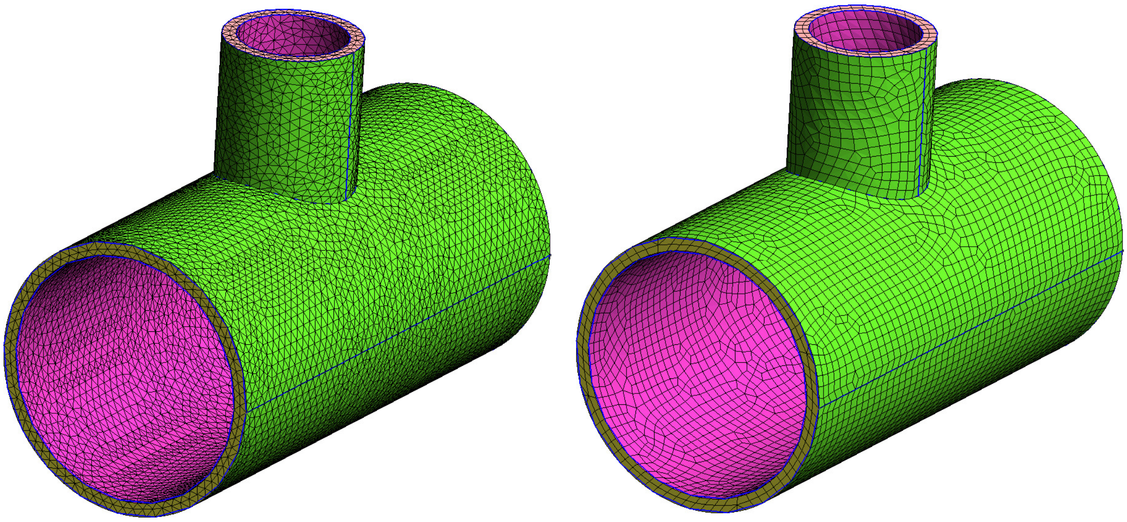


Figure 3: What is the difference between these two FEM grids?

The importance of parametric studies for a cognizant engineer cannot be over-stressed as it is then only a small step further to optimization of the design.

Engineering software, Verification and validation

On the one hand, analysis software ought to provide a convenient way of performing parametric studies. Point-and-click programs are not well suited for this end, as results are often neither traceable nor reproducible. These issues have been already tackled by computer programmers in the UNIX community with a concept called *scriptability* [3]. It is therefore desirable to have both real parametric capabilities within the software and a way of running many slightly different problems at once in a deterministic and programmatic way.

On the other hand, a mandatory condition for the process of software verification and validation is to have access to the source code. Moreover, again following the spirit of the case, it would be desirable the analysis programs (both the mesh generator and the finite-element solver) were *free software* so users have the freedom to study and modify the source code [4]. This case uses Gmsh[1] for the former and Fino[2] for the latter.

[1] http://gmsh.info/

[2] https://www.seamplex.com/fino

Closure

Back in College, we all learned how to solve engineering problems. But there is a real gap between the equations written in chalk on a blackboard (now probably in the form of beamer slide presentations) and actual real-life engineering problems. This chapter introduces a real case from the nuclear industry and starts by idealising the structure such that it has a known analytical solution that can be found in textbooks. Additional realism is added in stages allowing the engineer to develop an understanding of the more complex physics and a faith in the veracity of the FE results where theoretical solutions are not available. Even more, a brief insight into the world of evaluation of low-cycle fatigue using such results further illustrates the complexities of real-life engineering analysis.

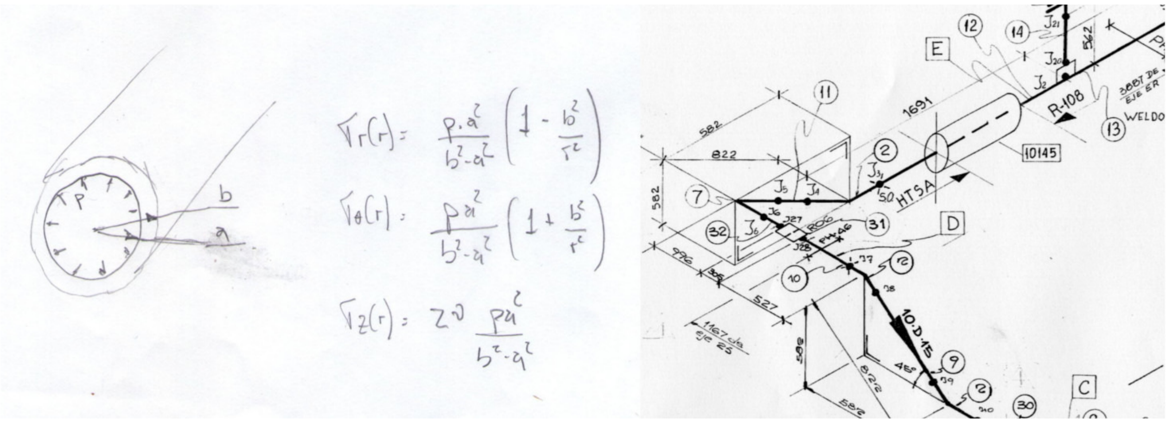


Figure 4: We know how to solve an infinite cylinder with paper and pencil. What happens in a real piping system?

References

[1] American Society of Mechanical Engineers. ASME Boiler and Pressure Vessel Code, Section III, Rules for Construction of Nuclear Facility Components, 2013.

[2] O. Chopra and G. L. Stevens. Effect of LWR Coolant Environments on the Fatigue Life of Reactor Materials, NUREG/CR-6909, 2014.

[3] Eric S. Raymond. The Art of UNIX Programming. Addison-Wesley, 2003.

[4] Richard M. Stallman. What is Free Software? https://www.gnu.org/philosophy/free-sw.en.html