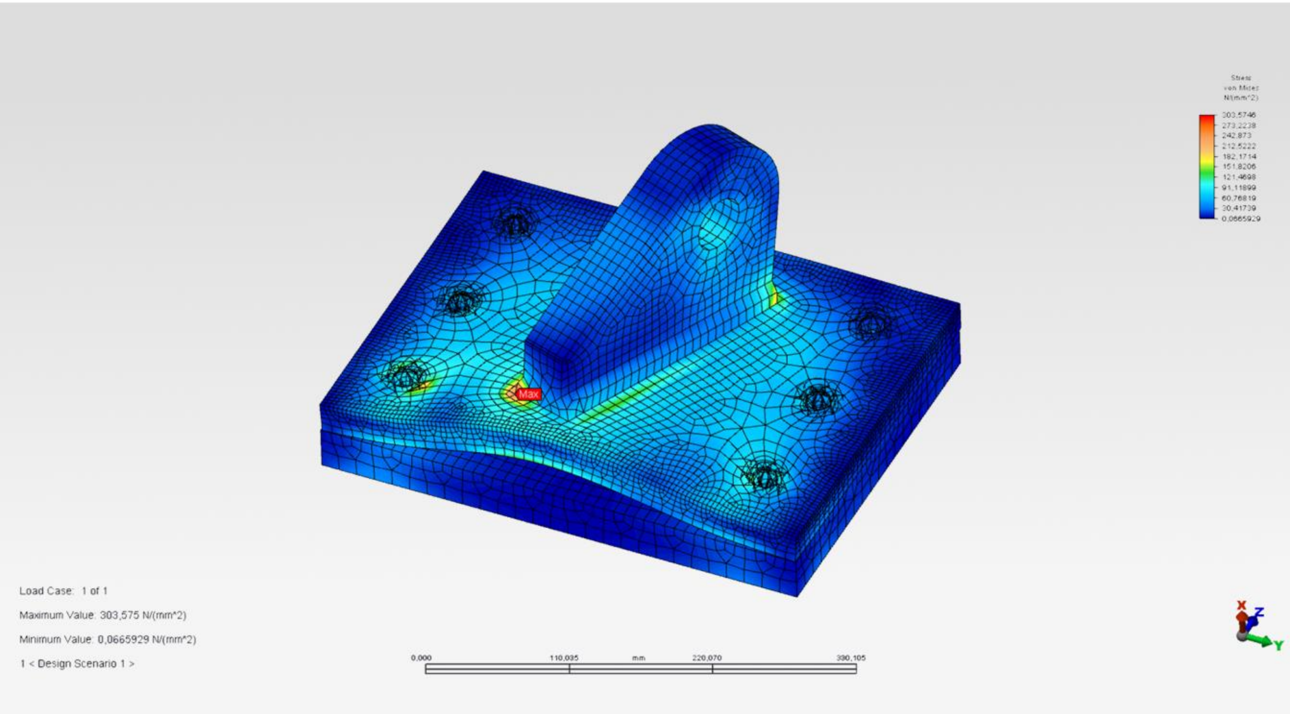
STRUCTURAL ANALYSIS AND FINITE ELEMENT ANALYSIS

ENGINEERING AND TECHNICAL ASSISTANCE



ingeniat
estudios y proyectos SL

INGENIAT
ESTUDIOS Y PROYECTOS SL

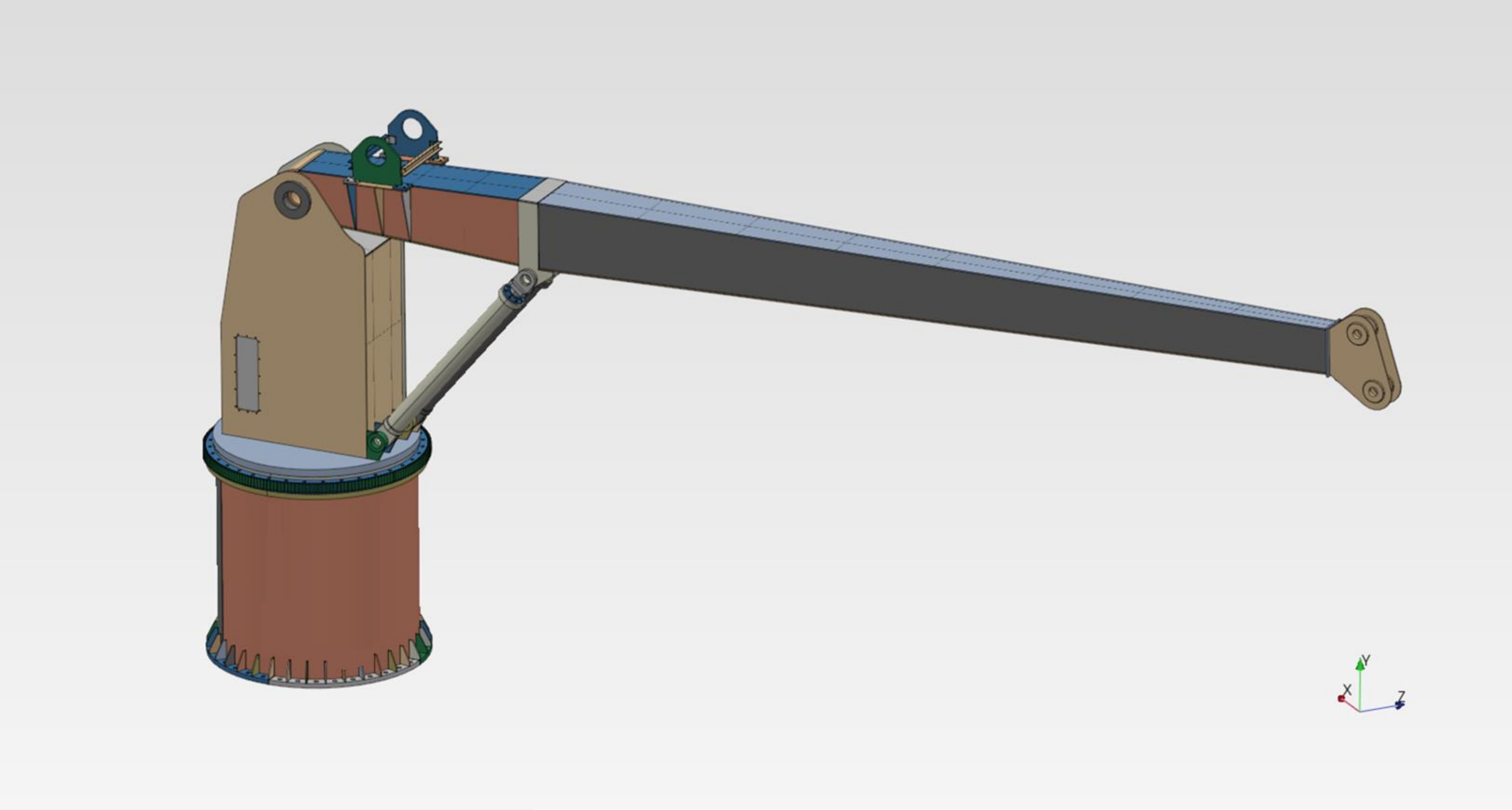
C/ VENEZUELA 62-64 ENT.6
15404 FERROL, SPAIN

PHONE
+34 663 700 398

PHONE
+34 644 358 624

EMAIL
CONTACT@INGENIAT.ES

WEBSITE
WWW.INGENIAT.ES



FINITE ELEMENT ANALYSIS
INTRODUCTION

Introduction and case study

“

For a professional engineer, the most important thing when undertaking a FEA analysis is to understand the right way to set up a problem.

With incorrect model conception, boundary conditions or meshing strategies, the results can be in accurate and misleading.

The Finite Element Analysis (FEA) method is commonly used as an alternative to the experimental test pointed out in many standards and specifications. The analysis method is based on the premise that a suficiently approximate solution to a complex engineering problem can be obtained by subdividing the structure subject to the analysis into smaller, finite, elements which are easier to handle and manage.

We analyze structures and parts to predict behaviour or to ensure code compliance with relevant standards. Failure modes such as buckling and fatigue, as well as non-linear responses such as large deflections can also be analyzed.

Due to our extensive experience with FEM / FEA, we can offer:

- Structural static analysis of components and structures: stress analysis (linear and non linear).
- Structural dynamic analysis of components and structures: frequency response, modal analysis to determine eigenfrequencies and oscillations.
- Calculation of fatigue strength and lifetime calculation.
- Linear and nonlinear buckling analysis of thin-walled structures, post buckling analysis.
- Optimization of constructions (existing and under development).
- Multiphysics analysis.

Image above
3D model of a knucle boom crane for marine service.

The use of the Finite Element Analysis method in mechanical and structural engineering allows the graphical representation and determination of deformation, stress and strain values of solid bodies and dynamic structures.

While the Finite Element Analysis can be performed in different fashions, or focused on features which do not require a realistic 3D model to be in place (for instance, analysis of two dimensional solids), all the examples shown on this brochure are based on 3D models which faithfully reproduce the real life structure or assembly. The model of the component, so called "physical" model, contains information not only on geometry and materials but also on mechanical and physical characteristics.

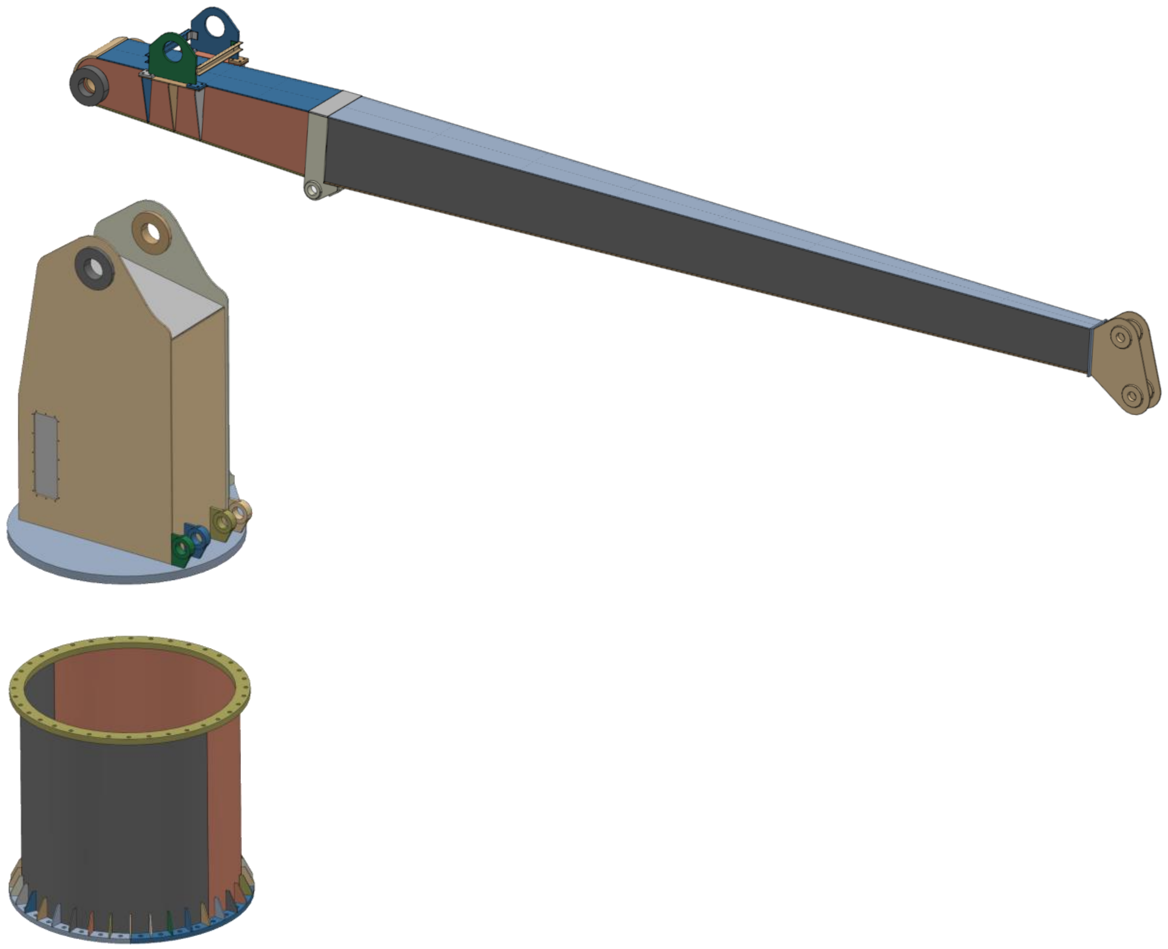


Image above
Breakdown of simplified 3D model of knuckle boom crane in subassemblies, verification of the model prior to preprocessing.

The design assessment begins with the verification of the structure or assembly model which is going to be subject to the analysis. We use Autodesk Inventor to develop the 3D models, which are then exported to compatible formats suitable to be parsed by the preprocessing software.

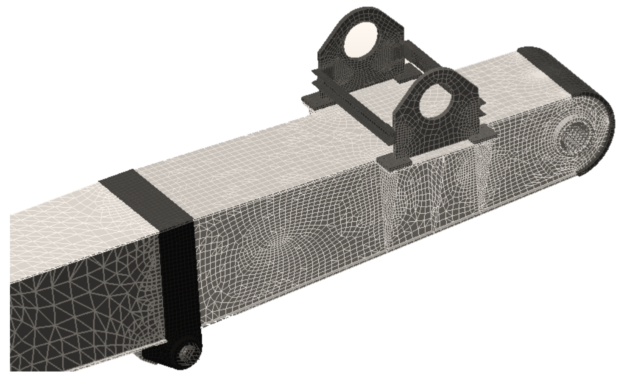
Verification of the 3D model is an essential step prior to the FEA analysis. In order to make the analysis conform to practical needs and improve computational efficiency, local regions are assessed before mesh generation to control local mesh density, by removing unnecessary features.

It is common to omit small details like fillet radii from a finite element model to simplify the analysis and to keep the model size reasonable. However, the introduction of any sharp corner into a model will lead to a stress singularity at that location. This normally has a negligible effect on the overall response of the model, but the predicted stresses close to the singularity will be inaccurate.

The Finite Element Analysis process can typically be broken down into three distinct stages: preprocessing, solver, and postprocessing.

- Preprocessing: The preprocessing, or model preparation stage, constitutes the most work intensive step of the Finite Element Analysis. The model being analyzed is decomposed in small, finite, elements, a process commonly referred to as meshing. The process typically results in the triangulation, or division into smaller triangles, of the surfaces, and the creation of tetrahedrons filling up the complete volume of the component.

  Choosing the right mesh is important: A coarser mesh results in less-accurate results, but a finer mesh creates more elements and takes more computing power to solve. That's why a mesh size that varies across the domain, as seen in the image below, is useful; you can define a coarser mesh in areas that are of less interest, and a finer mesh in areas that have a strong impact on the system behavior.



  Once the meshing process is concluded, material properties need to be applied to the parts being analyzed. Material properties are an essential characteristic in the behaviour of the component.

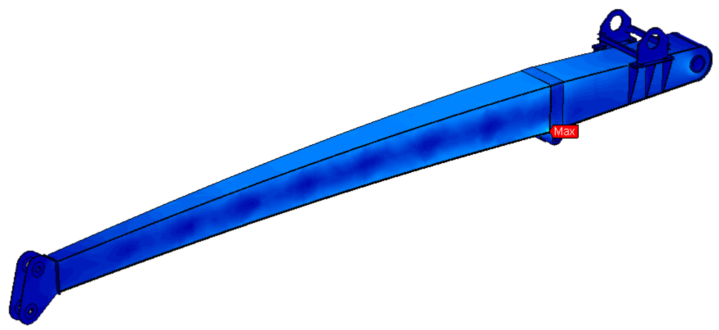
  Finally, boundary conditions need to be applied to the model. Boundary conditions represent the exterior actions or constraints that affect our model: how the movement is being restrained (effectively, the support conditions of the model) or how the component is being exposed to external actions. A proper definition of those two condition types, in consequence, is essential for the analysis to successfully take place.

  As far as the preprocessing stage is concerned, we use a variety of tools and software packages, including Algor Static/LM's built-in preprocessor, Netgen, Gmsh and Z88Aurora's built-in mesh generators (Tetgen & Netgen).

  Having prepared the model for the analysis accounting for those three considerations, it can now be transferred to the solver package.

- Solver: The solver package takes care of breaking down and solving the complex system of equations behind the FEM model. The first result of a solver always contains the displacements of the single nodes, but in the next step of the process, and as far as a stress analysis is concerned, distortions, stresses and nodal forces can then be calculated. At the end of a calculation, the results are forwarded to the postprocessor, the last stage of the FEA analysis, which allows for visualization and interpretation of the results.

  As with the preprocessor stage, we use different types of solvers depending on the specific analysis, including Algor Static/LM's built in solver, Calculix (Spooles, Pardiso and PastiX), and Z88Aurora's bulit in solver.



- Postprocessing: The postprocessor displays the calculation results and allows for further data interpretation. This includes, besides the displacements, also stresses and nodal forces. The displacements provide information about the geometric deflections of the component being analyzed. The most precise stresses are at the Gauss points, since they are calculated from the exact changes (distortions) of the individual elements. All other stresses, in the elements or in the nodes, result from averaged Gauss point stresses and their validity is not that accurate.

  As with the earlier stages, we utilize different tools depending on the specific analysis, but these would include Algor Static/LM's built-in postprocessor, CalculiX CGX, Paraview/Paravis or Z88Aurora's built-in postprocessor.

Image above
Stress analysis results on the deformed shape of a knuckle boom crane. FEA analysis undertaken using Algor Static/LM.

FINITE ELEMENT ANALYSIS
BENCHMARK TESTS AND QUALITY ASSESSMENT

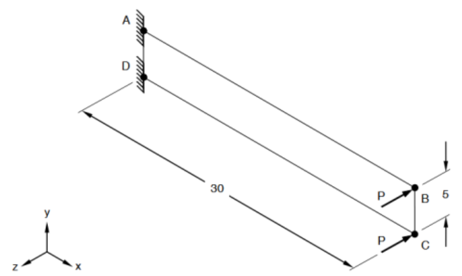
Quality assurance

Benchmark tests are an indispensable part of the verification of FEA packages and modeling procedures. For general applications, benchmark tests are published in commonly available literature or reputable references available online. Furthermore, if the element mesh to be used has been described exactly, it is also possible to compare the results of various FEA packages.

Performance benchmarking can assess the ability of a package to generate a model and obtain accurate solutions relative to industry accepted benchmark publications, trusted analytical solutions found in reputable engineering text, as well as experimental results, where applicable.

The following test case was taken from the engineering text, "Mechanical Engineering Design" (Shigley and Mitchell, 1983). This test case is a 3D linear static analysis which investigates the maximum deflection and stress in an edge loaded wall, which is represented as a 2D shell for analysis.

The test case originated from the fourth edition of a trusted engineering text, implying that the validity of the results has been established. This 2D representation of the geometry, albeit lacking complexity, can sufficiently reveal the packages analysis capabilities with respect to shell elements.



The dimensions of the thin wall are as follows: L=30 in (762 mm), H=5 in (127 mm) and t=0.1 in (2.54 mm). The acting load on the free side is F=6 lbf (26.69 N), applied in two corner nodes as shown in the figure above.

The results of the analytical solution from Shigley and Mitchell can be summarized as shown in the table below, next to the solutions obtained during the FEA analysis.

	Analytical	FEA Analysis	Discrepancy
Maximum stress (Von Mises)	148.92 MPa	157.2 MPa	5.5 %
Maximum displacement (+Z)	109.73 mm	106.3 mm	3.1 %

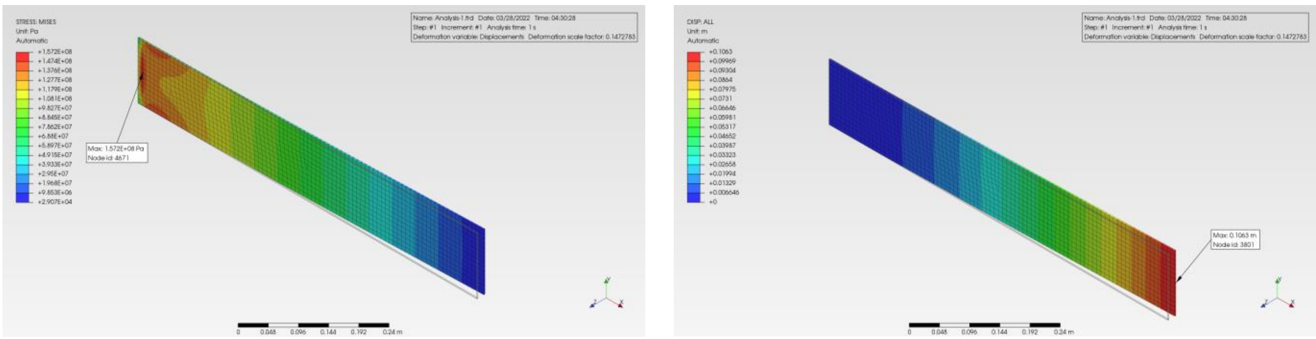


Image above

Linear stress analysis of thin wall using shell elements, Von Mises stress values (left), and total displacements (right).

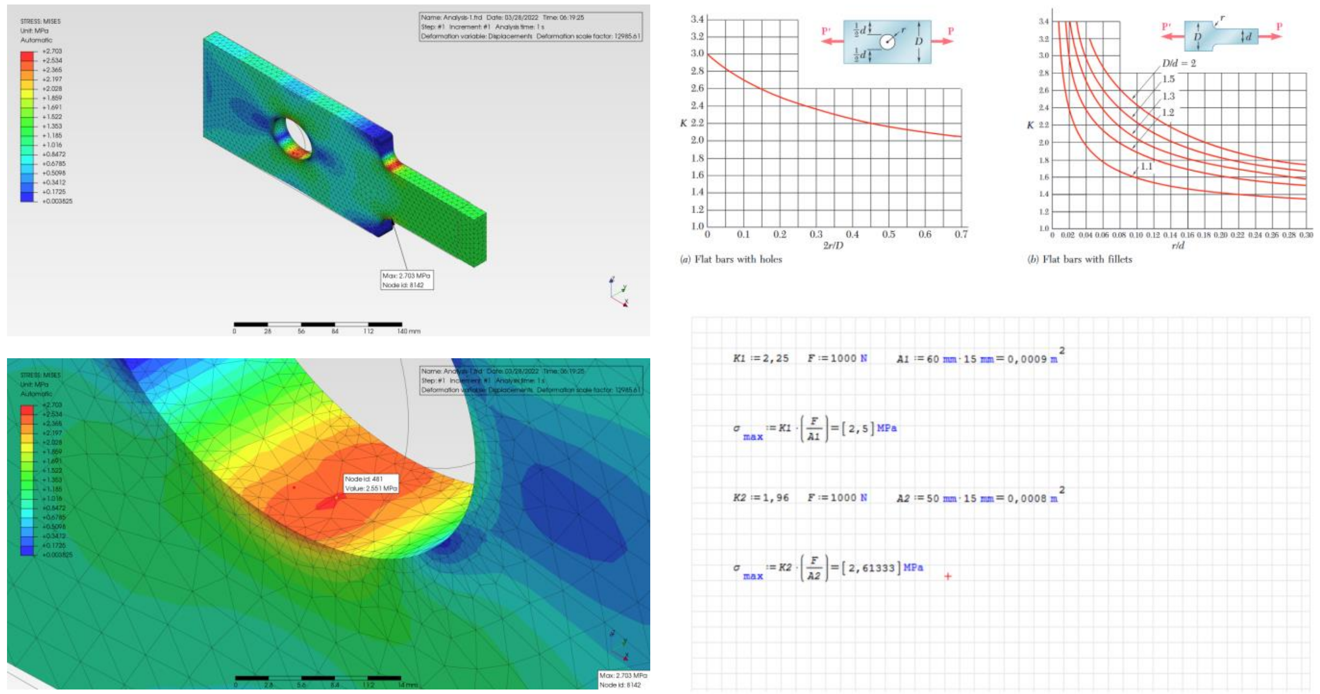
The model was 2D meshed with general purpose triangular elements (six nodes or CPS6, expanded into three dimensional wedge elements) and quad elements (eight nodes or CPS8, automatically expanded into three dimensional 20-node brick elements by the preprocessor).

The following test case was developed based on empirical models found in Mechanics of Materials by (Beer et al., 2012) and it comprises a 3D linear static analysis of a flat bar with stress concentrations under a tensile axial load. Using stress concentration factors for flat bars under tensile loading from (Beer et al., 2012) in Figure 4-6 , K1 (for holes) and K2 (for fillets) were determined. The flat bar was modeled using solid parabolic mesh with C3D10 elements (10-node modified tetrahedron). One side of the flat bar is restrained with a fixed boundary condition, whereas on the opposite side a 1000 N traction load is applied uniformly on the surface.

	Analytical	FEA Analysis	Discrepancy
Maximum stress (Von Mises) at hole	2.55 MPa	2.50 MPa	1.9 %
Maximum stress (Von Mises) at fillet	2.70 MPa	2.61 MPa	3.4 %

Image below

Linear stress analysis of flat bar under axial load showing maximum stress on the fillet concentration area (top left), and orifice concentration area (bottom left) next to analytical verification using stress intensification curves (right), extracted from Mechanics of Materials, Beer et al., 2012.

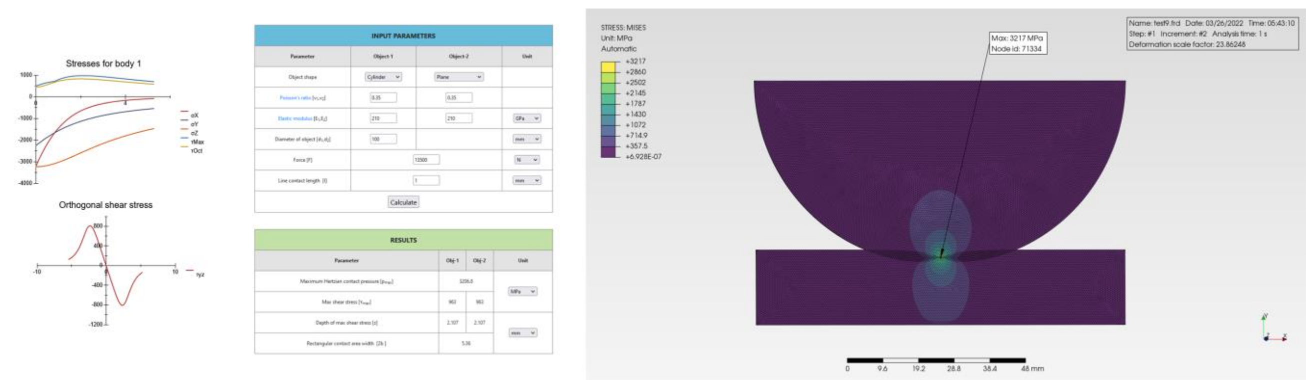


Discrepancies shown above are in this particular case a consequence of the interpolation error when determining the intensification factors.

When two bodies with curved surfaces are in contact under a force, point or line contact between these bodies changes to area contact, and three dimensional contact stresses are developed. This type of problems are typically covered by Hertzian contact theory, and the following test case is based upon the contact between a cylinder and a test plane.

Image below

Analytical solution (left) and results of the Finite Element Analysis of a cylinder in contact with a flat plane (right).



The contact between a cylinder and a flat plane is typically an extension of a more common contact problem between two cylinders, only in this case for a plane surface, the radius of curvature = ∞, and by extension, the diameter = ∞. Typical expressions for Hertzian contact pressure between a cylinder and a flat plane can be found in diverse literature, such as “Mechanical Engineering Design” (Shigley and Mitchell, 1983) for the contact half width (b) and the maximum contact pressure (P_{max}).

$$b = \sqrt{\frac{2F}{\pi l}\frac{(1-\nu_1^2)/E_1 + (1-\nu_2^2)/E_2}{1/d_1 + 1/d_2}} \qquad p_{max} = \frac{2F}{\pi b l}$$

In our test case, a normal force F of 13500 N acts on a cylinder with a diameter of 100 mm. The contact length (width) is assumed to be 1 mm.

Yielding the following results:

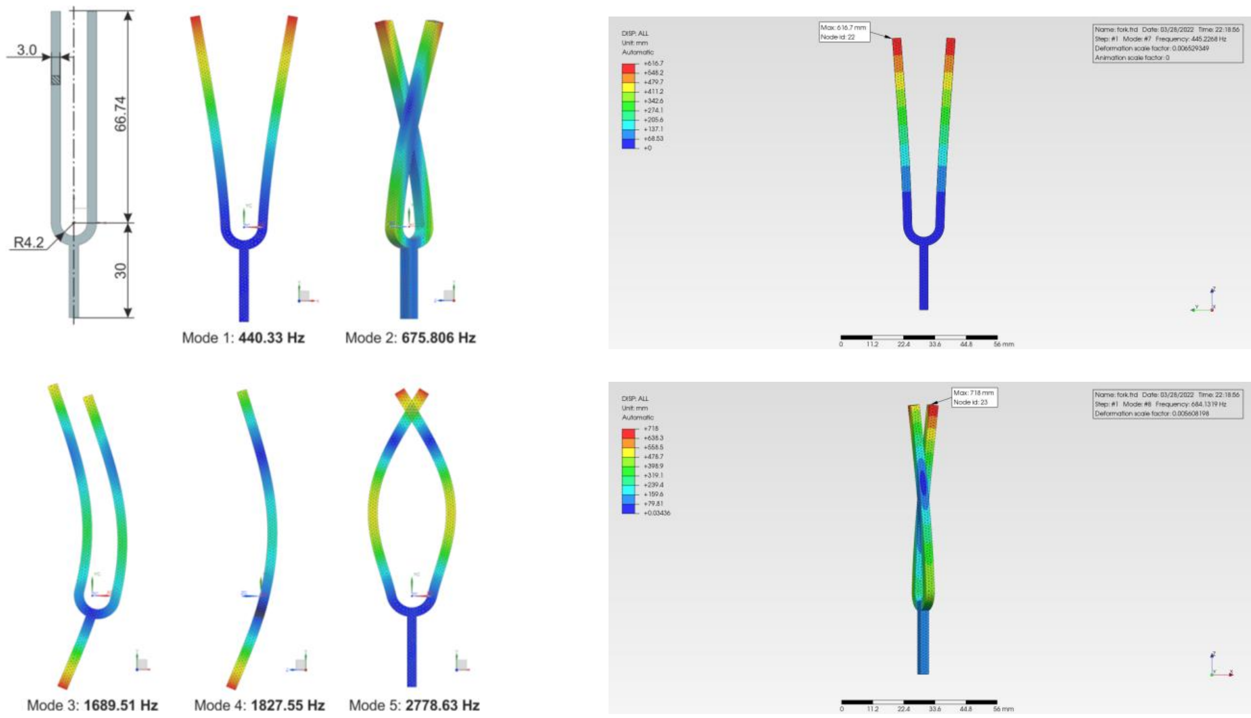
	Analytical	FEA Analysis	Discrepancy
Maximum contact pressure	3206.8 MPa	3217 MPa	0.3 %

Both the cylinder and the flat plane were modeled using solid parabolic mesh with C3D10 elements (10-node modified tetrahedron). The flat plane is restrained with a fixed boundary condition, whereas on the opposite side a normal load is applied uniformly on the flat surface of the half-cylinder shown in the figure above.

The following test case is based upon the work developed by the CoFEA Initiative, a workgroup of fellow engineers developing test benchmarks for different FEA analysis software. The test case considers the free-free modal analysis of a tuning fork, based upon the paper written by Róbert Huňady and Peter Pavelka, "Geometric Optimization of a Tuning Fork in NX Nastran". The paper deals with the experimental and numerical modal analysis of an existing tuning fork and, based upon those results, the performance of the selected solver can be benchmarked. The analysis is carried out as a free body modal simulation, with no boundary conditions assigned to the tuning fork. The tuning fork was modeled using a solid parabolic mesh with C3D10 elements (10-node modified tetrahedron). In the interest of expediency, only the first two natural frequencies are shown at the table below.

Image below

Tuning fork geometry and natural vibration modes (left) courtesy of the CoFEA Initiative, and results of the Finite Element Analysis (right).



Yielding the following results:

	Reference source	FEA Analysis	Discrepancy
First eigenfrequency	440.33 Hz	445.22 Hz	1.1 %
Second eigenfrequency	675.80 Hz	684.13 Hz	1.2 %

The following test case considers the elastoplastic deformation of a cantilever beam, with the following dimensions: width (b) = 140 mm, height (h) = 200 mm, length (L) = 2000 mm. The deflected shape and maximum deflection at the free node are calculated using the double integration method as per classic mechanics of materials. Asuming a punctual load located on the tip of the beam, it can be determined that the threshold for partial plastification occurs for a minimum load of 110000 N, whereas plastification of the full section of the beam is expected for a maximum load of 164500 N. For the purpose of the analysis, a punctual load of 155000 N is considered, resulting in elastoplastic behaviour of the beam.

In the interest of comparing the accuracy of the plastic behaviour of the FEA model, the analysis is ran twice. The first simulation neglects the plastic behaviour and assumes a fully elastic response to the load. The second simulation includes the real stress-strain response of the material, and contemplates an elastoplastic response, resulting in a more accurate and realistic simulation.

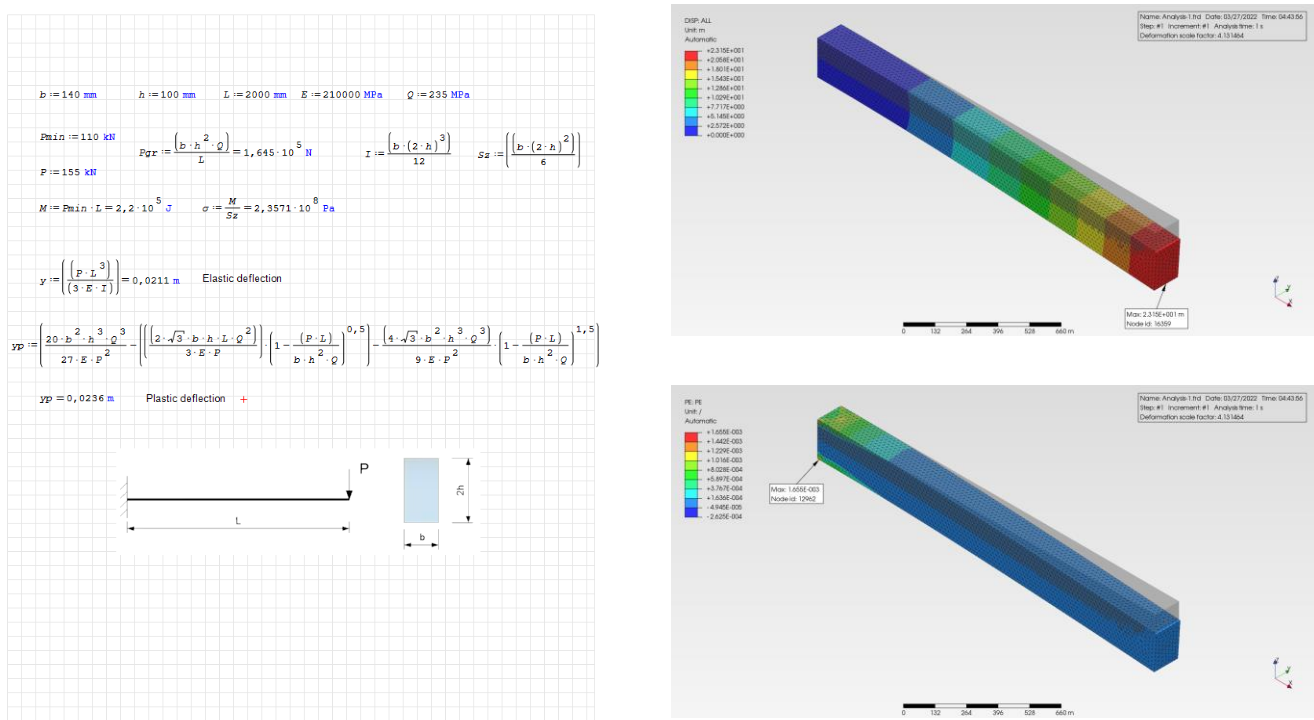
Yielding the following results:

	Analytical	FEA Analysis	Discrepancy
Maximum deflection (elastic)	21.1 mm	21.2 mm	0.4 %
Maximum deflection (elastoplastic)	23.6 mm	23.1 mm	1.9 %

As with previous test cases, the cantilever beam was modeled using a solid parabolic mesh with C3D10 elements (10-node modified tetrahedron).

Image below

Results of Finite Element Analysis with deformed shape and maximum deflection (top left) and equivalent plastic strain representing the material's inelastic deformation (bottom left), and results of analytical calculation (right).



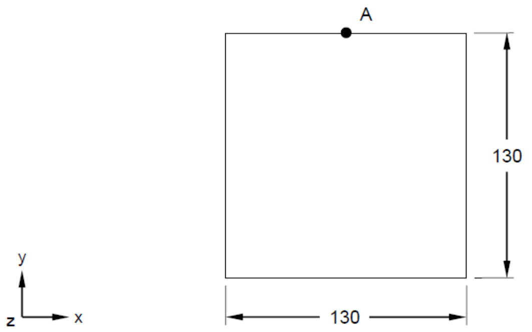
The second image above shows the equivalent plastic strain (PEEQ) confirming that a section of the cantilever beam has effectively deformed beyond its plastic threshold, resulting in permanent deformations.

The following test case as describes below considers a square plate model, with dimensions as shown in the image below, and a thickness of 1 in. The model is fully restrained in all translations and rotations around the perimeter and a normal pressure of 10 psi is applied on the top surface. The model is carried out using both shell and solid elements.

This benchmark case is based upon a problem laid out on the engineering text “ Stress and Strain Data Handbook” by Tsu.T.H.

For the purposes of the analysis, the Von Mises stress is calculated and benchmarked against the known result from the text above, at the point A shown below.

Image below
General dimensions of benchmarked square plate.

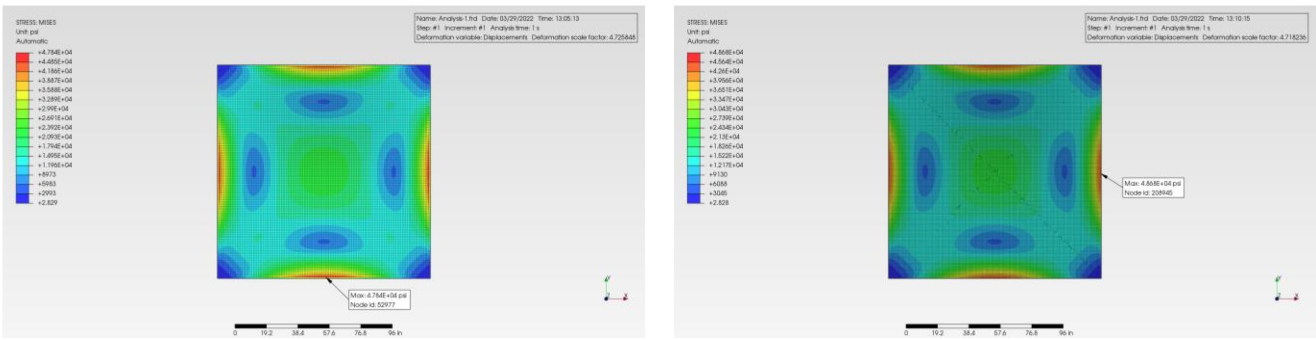


Yielding the following results:

	Analytical	FEA Analysis	Discrepancy
Initial analysis	52020 psi	47840 psi	8.0 %
Final analysis	52020 psi	48680 psi	6.4 %

The flat plat was modeled using shell elements and decreasing the mesh size in two different analysis, from a maximum element size of 1 in to 0.5 in.The model on the left image below was meshed with general purpose triangular elements (six nodes or CPS6, expanded into three dimensional wedge elements) whereas the model on the right was meshed with general purpose triangular elements (six nodes or CPS6, expanded into three dimensional wedge elements) and quad elements (eight nodes or CPS8, automatically expanded into three dimensional 20-node brick elements by the preprocessor).

Image below
Results of Von Misses combined stress with two different types of meshes.



The following test is based upon prescriptions by the National Agency for Finite Element Methods and Standards (U.K.): Test LE1 from NAFEMS publication TNSB, Rev. 3, “The Standard NAFEMS Benchmarks,” October 1990, and it considers a plane stress problem with a curved shape defined by four points (A, B, C and D). The elliptic membrane thus defined is subject to an outward pressure of 10 MPa at outer edge BC as shown at the image below. The rest of the boundary conditions, in particular the support conditions of edges AB and CD are identically shown. Linear elastic analysis is considered with Young's modulus = 210 GPa, Poisson's ratio = 0.3, density = 7800 kg/m^3.

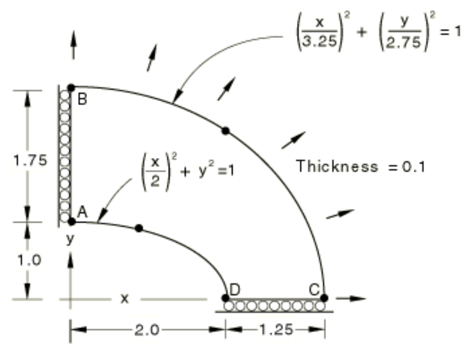


Image above
General dimensions of elliptic membrane and boundary conditions.

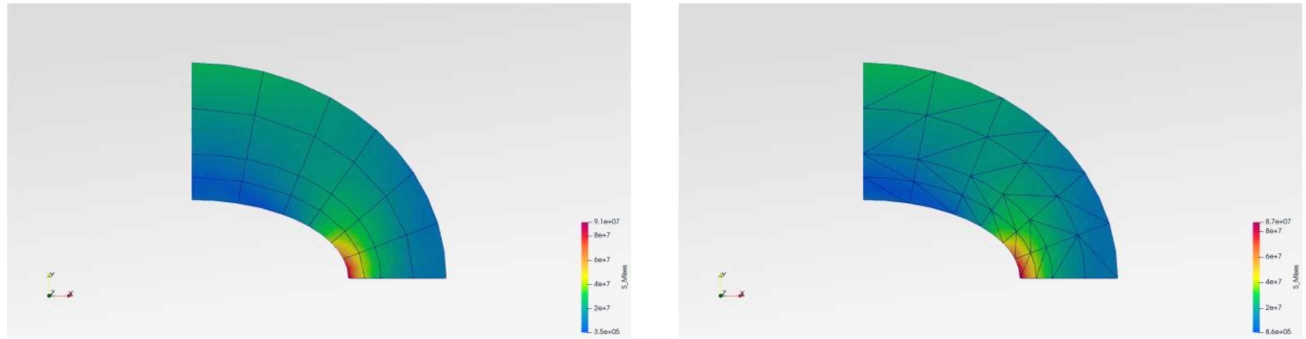


Image above
Results of Finite Element Analysis with stress results of an elliptic membrane subject to an outward pressure, considering quadratic plane stress elements (left) and triangular plane stress elements (right).

The model on the left image above was meshed with general purpose quad elements (eight nodes or CPS8, automatically expanded into three dimensional 20-node brick elements by the preprocessor) whereas the model on the right was meshed with general purpose triangular elements (six nodes or CPS6, expanded into three dimensional wedge elements).

The tangencial edge stress at point D is determined by NAFEMS to be 92.7 MPa, and this will constitute the basis for comparison. Different mesh densities and elements are considered, yielding a conclusion that full-integration elements perform significantly better for problems with stress concentrations of this type, and the following results:

	Analytical	FEA Analysis	Discrepancy
Quadratic plane stress elements	92.7 MPa	91.0 MPa	1.8 %
Triangular plane stress elements	92.7 MPa	87.0 MPa	6.1 %

FINITE ELEMENT ANALYSIS
TOPOLOGY OPTIMIZATION

Engineering and technical assistance

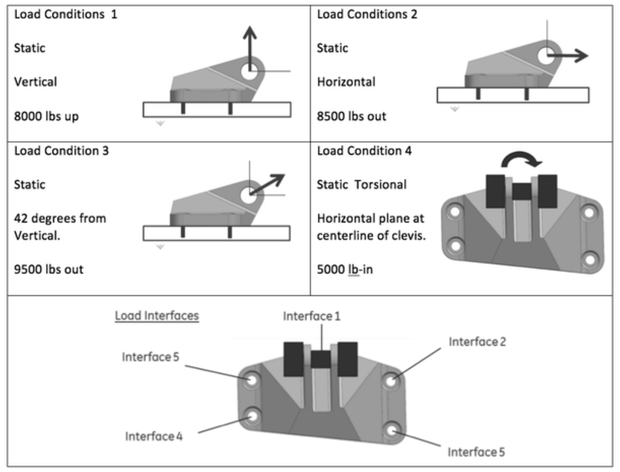
The design process can be conceptually divided in two different stages: basic or conceptual design, and detailed design. The structural layout can be defined and outlined in the basic design stage, whereas the exact shape and the size of the structure are outlined in the detailed design stage. By using topology optimization techniques in the conceptual design stage, we can obtain substantial improvements in the performance of structures, and significant weight reduction.

The BESO (Bi-directional Evolutionary Structural Optimization) method is a finite element based topology optimization method, where inefficient material is iteratively removed from a structure while efficient material is simultaneously added to the structure, effectively improving the efficiency of the design. The BESO method can be considered as an evolution of the original ESO (Evolutionary Structural Optimization) method which improves the design by gradually removing the inefficient elements. In the BESO method, on the other hand, a bi-directional evolutionary strategy is applied which also allows the strengthening of the efficient parts by adding material.

Both evolutionary methods (ESO and BESO) aim to provide a ‘hard element kill’ approach which means the element is either present, with full stiffness, or effectively eliminated, with a very low stiffness.

The basic idea behind topology optimization is to define a design space and then mesh that with a very regular array of elements. In some cases, this will be an arbitrary 3D space, in other cases, as shown in the figure below, the mesh will follow an initial scheme. The normal analysis begins with the full design space contemplating the original design of the part, subject to the design boundary conditions, including loading.

Such an initial analysis of this component will give a distribution of internal stress and deflection, and the topology optimization would seek to improve the efficiency of the configuration by removing material, based on these responses. Minimizing the strain energy can also be described as minimizing the compliance (defined as the distributed force times the displacement summation) which would, in turn, imply a maximization of stiffness.



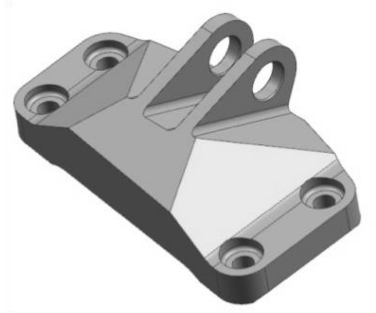


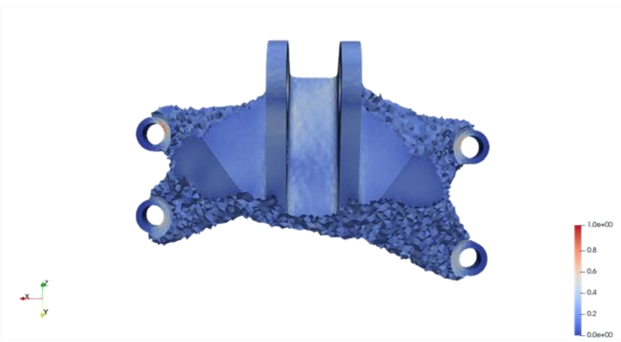
Image above
Excerpt taken from the "GE jet engine bracket challenge" hosted by GrabCAD.com. Images courtesy GrabCAD.com.

Material is then progressively removed using a target volume reduction, or added in high efficiency areas, eventually yielding a final design with the desired level of efficiency and optimization.

The model above is taken from the "GE jet engine bracket challenge" hosted by GrabCAD.com in 2013 as part of a public challenge focused on design and performance efficiency with intense focus on weight reduction. As detailed in the challenge rules:

"The designs submitted will be analyzed and evaluated via simulation, and the top ten designs will be selected for fabrication and testing. These optimized engine bracket designs will be additively manufactured and subjected to a given loading scenario. The winning entries will best satisfy all of the performance criteria with the lowest mass."

The images below show the result of a simple BESO simulation using CalculiX (v2.19) as solver and Paraview as postprocessor, utilizing this base reference as benchmark against known solutions which have participated in the aforementioned challenge. It shall be noted that the results below would only constitute a first pass into the optimization process.



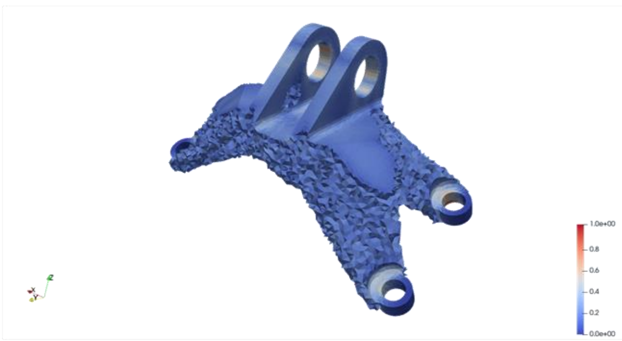


Image above
First pass results of evolutionary structural optimization for the sample bracket, using BESO algorithm developed for CalculiX.

The results shown above would provide a starting point for a new redesign of the bracket, utilizing the new shape as a template for futher optimization or, eventually and should the mass reduction targets be met, for final design review and fabrication.

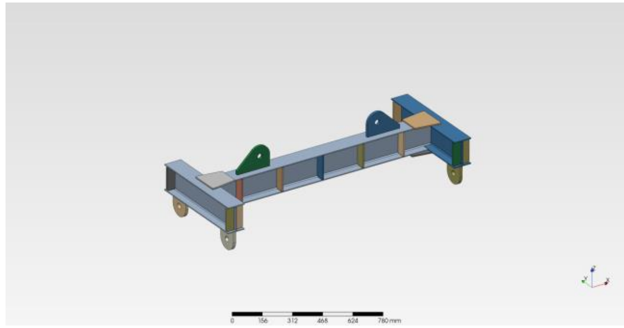
Generally speaking, the optimal results of a FEA-based topology optimization procedure describe all the presenting states of finite elements in design domain, which would only provide hints as to how the optimum structure could look and, thus, would need to be translated and fed back into the process, eventually resulting in a realistic design concept.

The utilization of the BESO optimization technique in structural engineering is of great significance to the industry, especially in applications where weight reduction and weight control are of fundamental interest. This technique is capable of creating totally innovative designs with unconventional structural members at high structural efficiency.

FINITE ELEMENT ANALYSIS
COMPETENCES AND CAPACITATION

Engineering and technical assistance

Our in-house Computer Aided Engineering (CAE) staff provide a wide range of tailored solutions for all our customers, from simple meshing and model building:



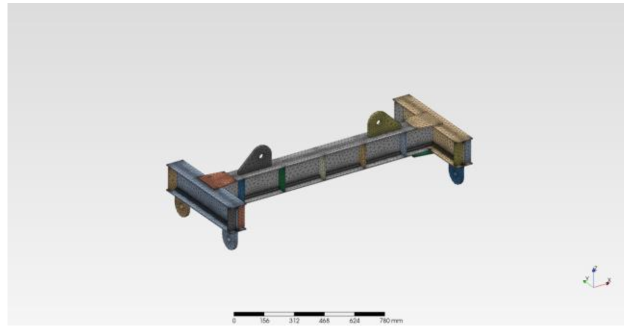
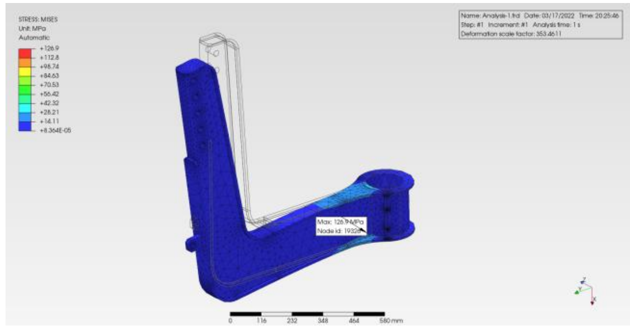


Image above
3D modeling and preprocessing of a lifting frame (left), including meshing and FEM model building (right).

To solving and postprocessing a complete linear structural analysis, including modal analysis or steady state response analysis:



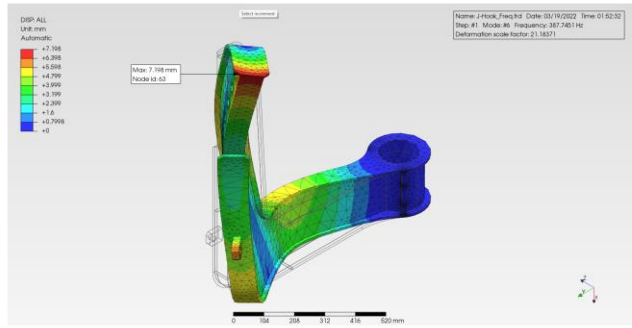
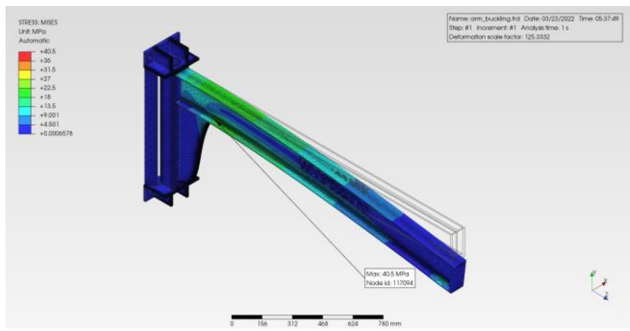


Image above
Postprocessing and results of a linear stress analysis (left), and modal analysis and determination of eigenfrequencies (right), of a connecting bracket.

Specific verifications which complement the results of a linear structural analysis can be undertaken on certain types of structures or components, such as the verification of lateral stability and lateral torsional buckling of beams.



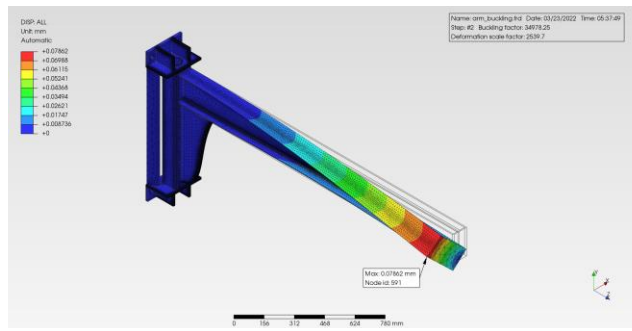
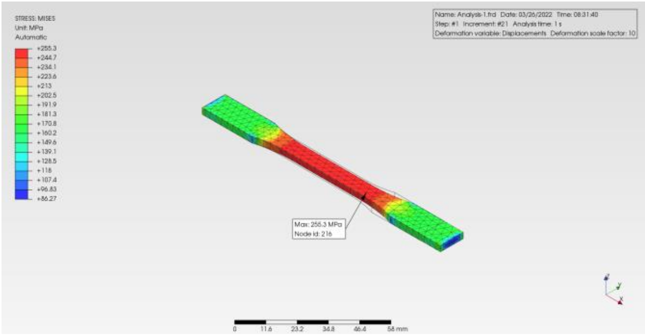
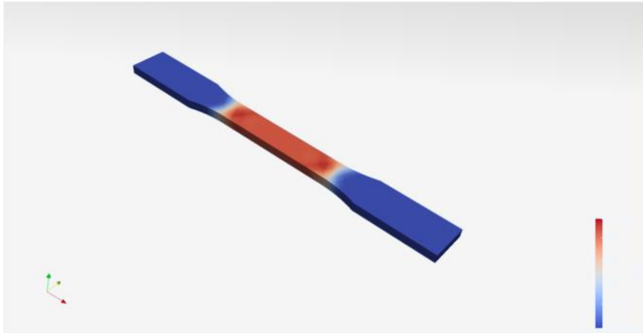


Image above
Linear stress analysis of a jib crane (left), and determination of critical lateral torsional buckling load (right).

We can identically analyze the behaviour of a given component under elastoplastic or plastic regime, including non linear analysis.

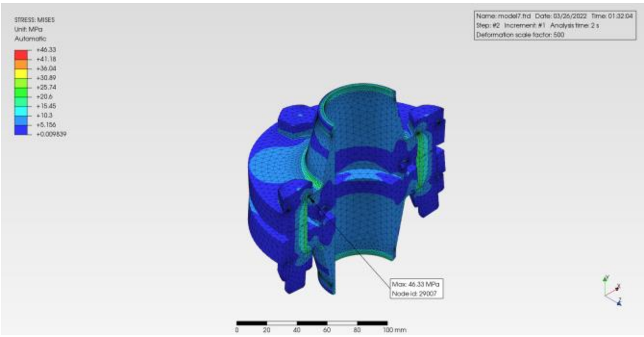
Image below
Linear stress analysis of a test coupon (left) and equivalent plastic strain (PEEQ) representing the material's inelastic deformation (right).

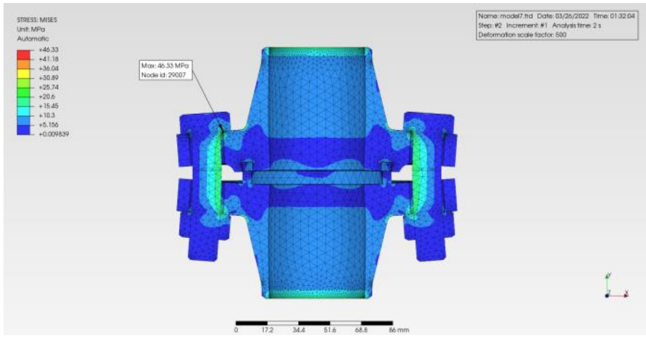




Or the effect of bolt preloading on the behaviour of a mechanical bolted connection.

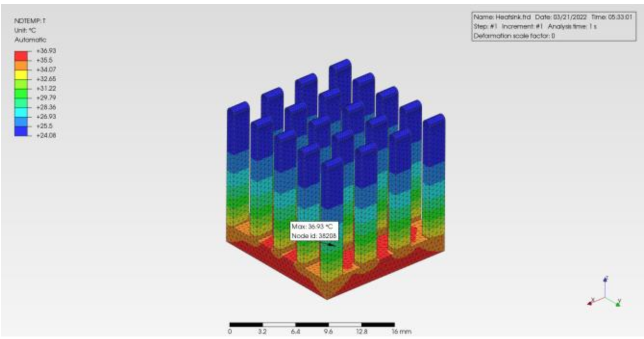
Image below
Linear stress analysis of API flange subject to internal fluid pressure with varied degrees of bolt preloading.

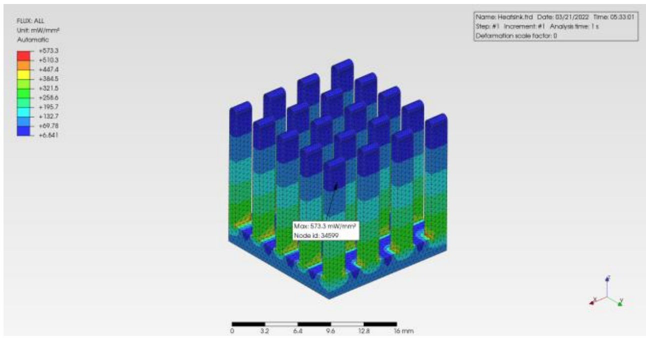




Besides structural analysis services, we can also develop multiphysics simulations, such as thermal dissipation or coupled/uncoupled temperature-displacement analysis.

Image below
Thermal analysis of an aluminum heat sink showing temperature distribution (left) and heat flux values (right).





We offer a complete range of FEA consulting services including structural, thermal, fatigue, and pressure analysis on fluid carrying structure. To summarize the most significant aspects of our scope of supply as showcased above.

- Linear and non-linear stress analysis. Used to understand the deformation, stress and strain of an assembly or component, for a range of loads, guaranteeing the structural integrity of the system. In cases where stiffness of the part changes due to shape change, or where the ratio of stress to strain does not remain constant, non-linear analysis shall be undertaken.

- Modal analysis and steady state frequency response. Used to determine the structure's fundamental dynamic characteristics (eigenfrequencies), damping factors and mode shapes, as well as the deformation of a structural component when excited by external vibration loads. The frequency response analysis subjects the model to a constant vibrational load frequency to determine the response of the model for a specific frequency range.

- Thermal analysis. Steady and Transient FEA thermal study can be used to determine the thermal distribution in an component, allowing for further understanding of the system working under temperature, leading to high thermal stresses and strains, and potentially to component failure.

FINITE ELEMENT ANALYSIS
RESOURCES AND INFRASTRUCTURE

Engineering and technical assistance

Any computing problem that can be broken down into large numbers of small, but independent, computations can be potentially accelerated by the many processor threads or cores. This includes standard analyses such as computational fluid dynamics (CFD) and finite element analysis (FEA). In a nutshell, by running finite element analysis (FEA) and computation fluid dynamics (CFD) applications on compute clusters that are affordable for most businesses, significant time savings can be achieved.

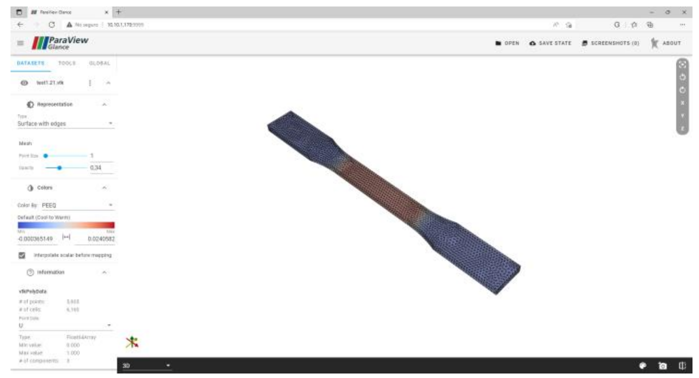
Making use of a cluster's significant benefits can be more daunting than originally bargained for. Computer clusters can be technically difficult to purchase, configure and administer, given the combination of processors and processor cores, cache, system memory and interconnect paths can be highly dependent upon one another. Furthermore, performing computations on processor cores is only part of the problem, as data has to be moved rapidly among processors in the cluster, potentially resulting in bottlenecks at the interconnect path level.

On-premises, we utilize a local set of highly available clusters handled by Proxmox, an open source server virtualization management solution. Installed software includes CalculiX v2.19 (with Pardiso and PastiX solvers), OpenFOAM v9 and HELYX-OS v2.4.0 (for CFD applications), ParaView v5.10.1, OpenMPI v4.1.1 and FreeCAD v0.19.2, running on Ubuntu 20.04 LTS. Clusters are running on stock hardware, based upon Ryzen 5 5600x (6-core, 12-thread) and Ryzen 7 5800x (8-core, 16-thread) processors, fitted with 64GB RAM, RAID-0 NVMe SSD drives, and up to 10 Gbps (10GbE fiber network cards) of network speed.

For more demanding and challenging analysis, we utilize the infrastructure provided by Amazon Web Services (AWS) and, in particular, AWS C6i and M6i instances of Elastic Cloud Compute (EC2). These instances use the 32-core 2.9 GHz / 3.5 GHz Xeon Platinum 8375C processors, featuring up to 2 processors per socket (up to 64 cores total), with up to 50 Gbps of network speed at the largest instance size, also supporting Elastic Fabric Adapter to provide high speed interconnect paths across a cluster of instances. Installed software includes CalculiX v2.19 (with Pardiso and PastiX solvers) and OpenMPI v4.1.1, running on Ubuntu 20.04 LTS server edition.

In addition to the use of general computing CPU cores, running via the Message Passing Interface (MPI) for parallelization, we utilize a especially compiled CalculiX solver (PastiX) which makes use of the processing capability of CUDA GPU cores. The more compute power available to an engineer, the less need there is for defeaturing, or removing elements of a model from the simulation.

Besides the computing package itself, we have deployed both internally, as well as in public facing servers, postprocessing tools such as Paraview Glance (shown to the right) or Paraview Trame, facilitating the sharing and evaluation of results among different parties.



We are identically running an internal Virtual Private Network (VPN) with segregated entry points through a remote server, providing access to our compute network to our customers abroad, both for analysis or results sharing, as well as to participate in our virtual environment for remote design review sessions.

On demand, we can provide unrestricted access to our customers to those computing resources, both remote or on-premises. Sample configurations can be shared in advance for evaluation in the form of virtual machines (Virtualbox or similar).

Note on CalculiX: The main software package in our workflow, and as shown in the different benchmark tests above, is CalculiX, an open source finite element analysis application with an implicit and explicit solver, developed by Dr. Guido Dhondt of MTU Aero Engines GmbH, with support from other figures in the academic world, such as Prof. Martin Kraska, Brandenburg University of Applied Sciences. Numerous benchmark tests and assorted literature are available online to assess the accuracy of the solvers and the validity of the solutions in different use cases.

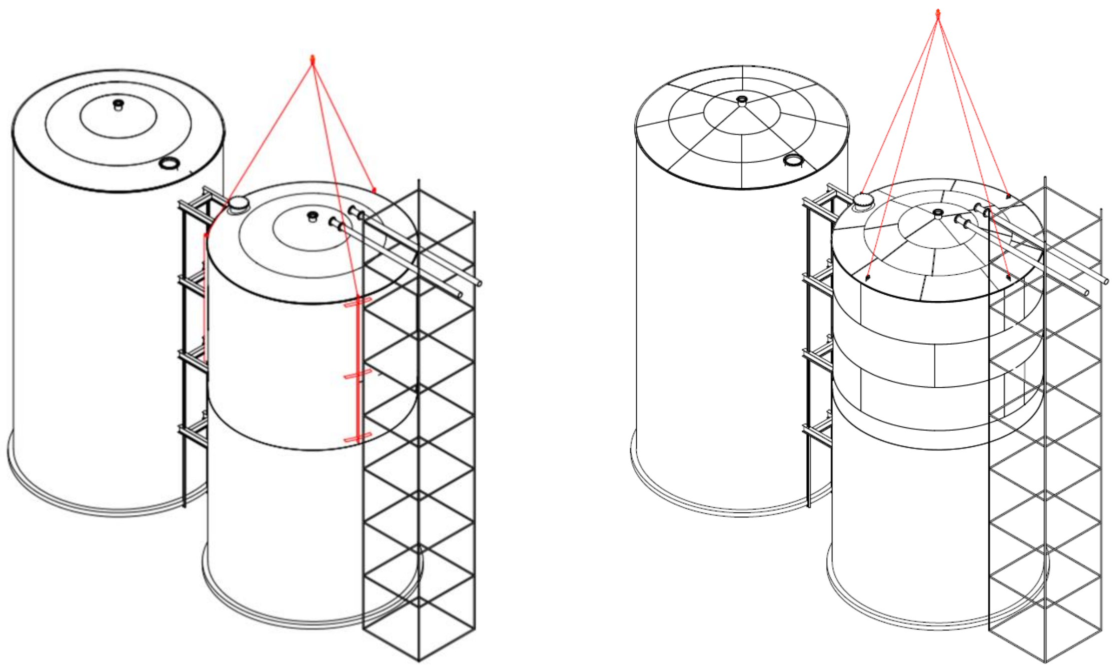
Besides those benchmark tests, we have identically developed a number of simulations to compare the adequacy of this software with respect to a well known and recognized commercial package we have used extensively in the past, Algor FEA by Algor, Inc. Algor FEA, as well as other recognized software packages, would remain a staple for our regular workflow.

FINITE ELEMENT ANALYSIS
PRACTICAL EXAMPLE AND CASE STUDY

Design of lifting gear for decommissioning of damaged storage tank

The case study shown below contemplates the decommissioning and removal of a storage tank which had been damaged due to an incorrect operation procedure, and which had endured widespread material loss due to age and exposure to the stored fluid. The scope of supply included the design of lifting gear necessary to undertake the separation and lifting of the tank, and the stiffening of the structure of the tank during the decommissioning process. The storage tank would need be split in two parts due to weight and height restrictions on site. The scope of supply would identically account for the installation of the new storage tank and, in particular, the new lifting gear which would be used during the installation process. Given the widespread material loss in the shell of the existing tank, UTM (Ultrasonic Thickness Measurement) of the existing shell was undertaken and the information gathered was fed to the engineering staff for consideration during the FEA analysis of the shell.

Image below
General arrangement of existing storage tank and lifting gear during decommissioning (left) and installation of the new storage tank (right).



The images below and on the next page show some partial results from the postprocessing of the stress analysis carried out on both the shell of the storage tank, and the lifting gear proposed for the removal and lifting operation. Likewise, some images are shown pertaining to the design of the new lifting gear and lifting padeyes proposed for the installation of the new equipment.

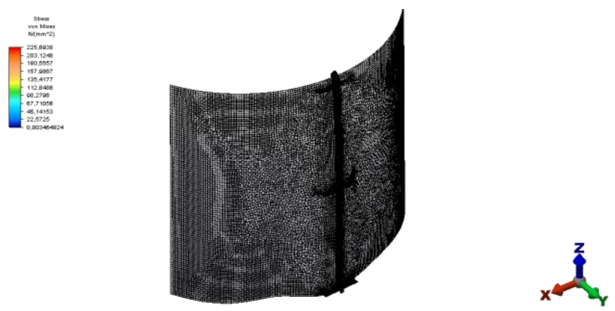
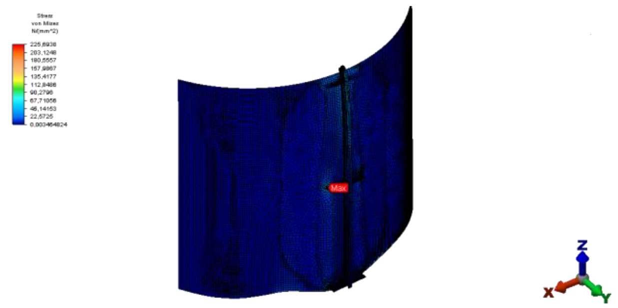


Image on the left
Postprocessing of linear stress analysis showing mesh and deformed shape of shell section making use of symmetry of storage tank to simplify the analysis.

Image on the right
Postprocessing of linear stress analysis showing deformed shape of shell section and equivalent Von Mises stress values.



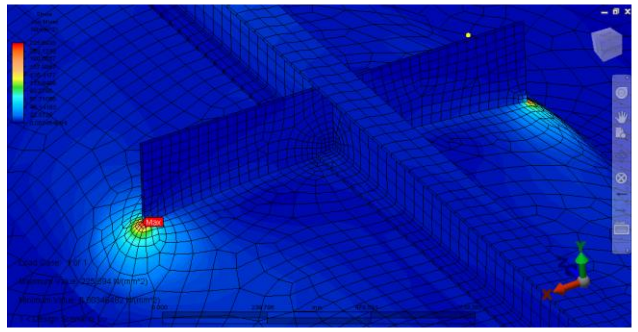
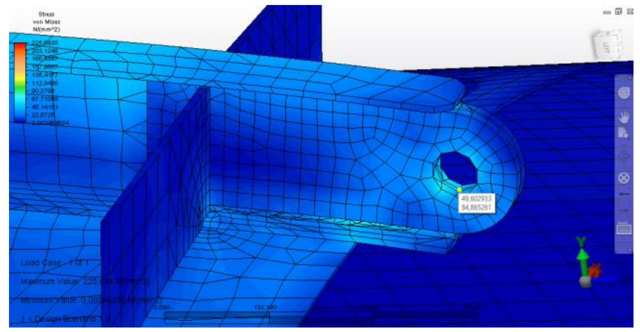


Image on the left
Postprocessing of linear stress analysis showing stress concentration points and maximum equivalent Von Mises stress values.

Image on the right
Postprocessing of linear stress analysis showing stress concentration point and equivalent Von Mises stress values at the lifting padeye.



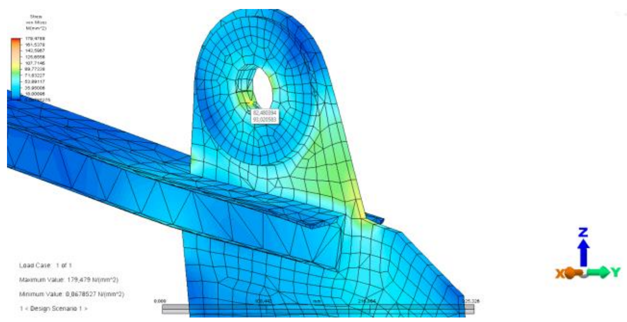
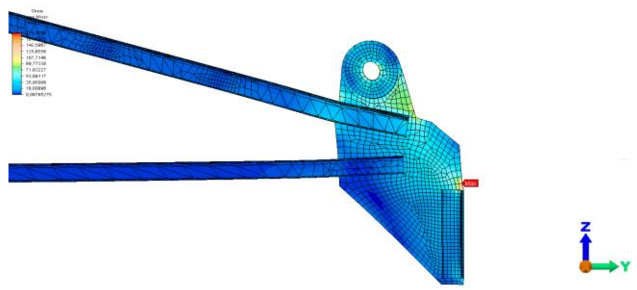


Image to the left
Postprocessing of linear stress analysis showing stress concentration points and equivalent Von Mises stress values at the lifting padeye.

Image above
Postprocessing of linear stress analysis showing stress concentration point and equivalent Von Mises stress values at the lifting padeye.



INGENIAT
ESTUDIOS Y PROYECTOS SL

C/VENEZUELA 62-64 ENT.6
15404 FERROL, SPAIN

PHONE
+34 663 700 398

PHONE
+34 644 358 624

EMAIL
CONTACT@INGENIAT.ES

WEBSITE
WWW.INGENIAT.ES