Three single-part linear elastic benchmark problems

Fino benchmark problem series

Number	Rev.	
SP-FI-17-BM-9ADB	A	

Date: 25-Sep-2017

Hash: 9169773

Author: Jeremy Theler `jeremy@seamplex.com`

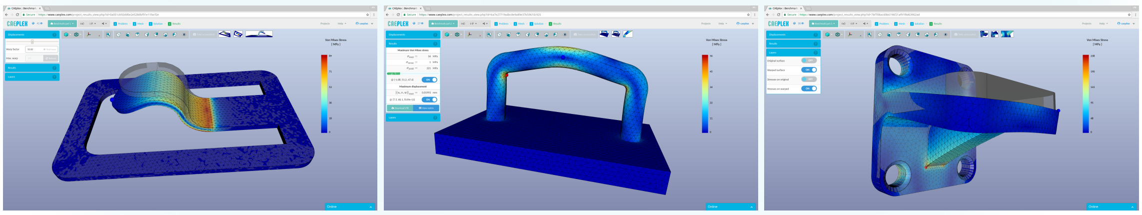
Document type: Benchmark problem

Pages: 50

Abstract

A benchmark of three single-part mechanical analyses solved with CAEplex using the free and open source tools Gmsh and Fino as the back-ends against a the commercial closed-source code. As the license renders this software privative, only results can be compared as output by a black-box. Even more, experts say that this “reference”software cannot export point-wise numerical data so only maximum scalar values can be compared instead of the full dependence on x, y and z of the unknowns. The comparison was made trying to keep the same number of nodes in the two codes similar while meshing the 3D CAD models with second-order tetrahedral elements. The results obtained are summarized as follows:

	SolidWorks			Fino		
Benchmark	Nodes	Displ. [mm]	σ [MPa]	Nodes	Displ. [mm]	σ [MPa]
#1	37k	0.305	88–96	36k	0.3085	89
#2	40k	0.020	42–58	40k	0.01993	56
#3	43k	0.093	165–241	40k	0.09307	238



Keywords

Fino, benchmark, SolidWorks, linear elasticity

Revision history

Rev.	Date	Author	
A	25-Sep-2017	gtheler	First issue

Contents

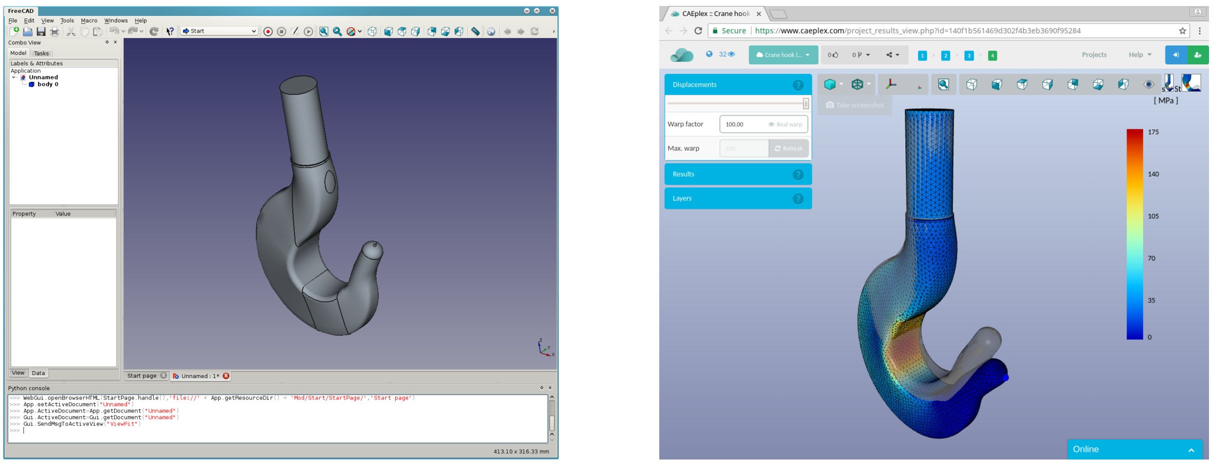
1 Introduction

This reports shows the results of solving three simple linear single-part test problem with the web-based platform CAEplex, that uses the free and open source back-ends Gmsh [1] for the mesh generation and Fino[1] for the solution of the equations that arise when employing the finite element method for discretizing the linear elastic problem. The results are compared against a reference solution of the same problems, obtained by using a commercial closed-source desktop software, and serves as a further reference for code verification and validation like the previous references [2] and [3]. The report with the reference solutions was prepared by an official technological agency of the government of the Santa Fe province in Argentina specifically for this Fino benchmark report, and is included completely in appendix D.

The reference results were obtained using SolidWorks Simulation Premium 2013.[2] As the license renders it privative software (because it deprives users from their basic freedoms), all we can do is compare results in a black-box sense, since we cannot know exactly what the program does as source code is not made available. Moreover, it appears that the full detailed spatial distribution of the results (i.e. displacements, derivatives, stresses, etc.) cannot be exported as numerical data as a function of x, y and z —at least that was what the experts told us—so a complete comparison of spatial dependence of the difference of results obtained with Fino on the one hand and this commercial code on the other hand cannot be done.

1.1 About CAEplex

CAEplex is a Computer Aided Engineering web-based platform which was designed from scratch to run finite-element analysis models on the cloud. Technically speaking, it is a web front end for a number of mechanical engineering free and open source back end tools running on an UNIX server. It was developed by the Argentinian company Seamplex and launched on February 2017. CAEplex' main objective is to lower the entry barriers that traditional CAE software present to beginners, especially those working in industrial SMEs, both in *difficulty* and in *price*. Its main objective is to provide a *very easy* way to perform mechanical analysis of manufactured industrial parts.



(a) 3D CAD model rendered in FreeCAD

(b) CAEplex results screen in Chrome

Figure 1: CAEplex allows to mechanically analyze 3D parts directly from the web browser at `https://www.caeplex.com`.

Technically speaking, CAEplex is a web-based front-end for a number of free and open source back-ends. The two most important ones are Fino and Gmsh.

[1] `https://www.seamplex.com/fino`

[2] We would like to mention this software as little as possible to avoid encouraging readers to use privative closed-source non-free software.

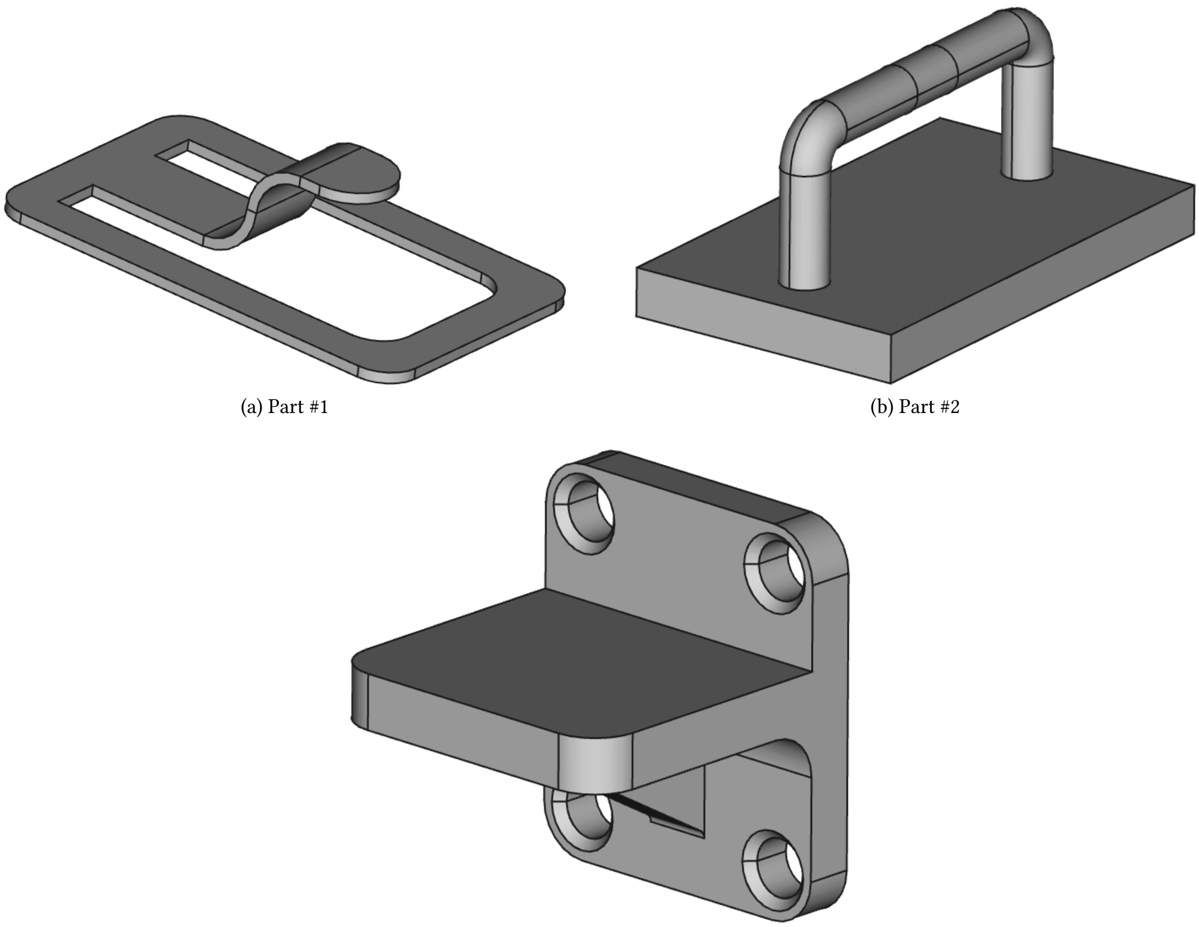
1.1.1 Fino & Gmsh

The problem is solved with the free[3] and open source finite-element analysis tool Fino—also developed by Seamplex—with a Gmsh-generated unstructured grid. More information about these programs, including documentation and downloads can be found at the following locations

(a) `https://www.seamplex.com/fino`

(b) `http://gmsh.info/`

It should be noted that, following Seamplex' principles,[4] only free and open source software was used in preparing this report. A notable and inevitable exception is the reference report included in appendix D (elaborated by a third party, though), that uses all kind of privative software, from the operating system down to the actual calculation code.



(a) Part #1

(b) Part #2

(c) Part #3

Figure 2: The three geometries as processed with FreeCAD out of the original STEP files.

Material		
Young's modulus	E =	210 GPa
Poisson coefficient	ν =	0.28
Yield strength	$\sigma_{\text{yield}} =$	220.6 MPa

Table 1: Mechanical properties of the material used in the three problems.

2 Problems

The problems solved in this benchmark consist of three relatively simple solids presented as 3D CAD models. Even though they all have at least one plane of symmetry, the full geometry was kept and used for the spatial discretization as done in the reference report included in appendix D. The three parts are assumed to be made of the same material, namely a non-alloyed carbon steel (according to the reference software). The mechanical properties are uniform throughout the domain and their numerical values are shown on table 1.

It makes no sense to compare code-to-code finite-element solutions with different levels of spatial discretizations. So in order to obtain comparable results, the problems were solved using CAEplex by employing second-order elements (as in the reference solutions) and by trying to obtain a similar number of *nodes* rather than *elements* in both solutions. The number of nodes give directly the number of discrete unknowns of the problem (i.e. the size of the stiffness matrix). And the order of the elements give the sparsity pattern of such matrix. Therefore, these two features are a real direct measure of the computational effort needed to solve the problem. For instance, if the codes were compared by solving a problem using the same number of elements but using linear ones in the first code and quadratic ones in the other one, the comparison (and the drawn conclusions) would be definitely wrong.

2.1 Part #1

The first part is a spring that works by fixing its base and receiving a vertical load. In particular, in the problem the spring is subject to a vertical load in the negative y direction equal to 100 N, uniformly distributed in the load surface as shown in figure 3. The mesh used in the reference calculation was generated using a characteristic element length $\ell_c = 2$ mm and results in 37.0k nodes and 21.1k second-order elements, allegedly only tetrahedra. We do not know what algorithm was used to compute the reference grid.

Appendix A.4 shows the details of the meshing settings chosen to solve this problem with CAEplex, and figures 13 and 14 show the resulting unstructured grid with 36.1k nodes and 19.7k second-order tetrahedra. This grid was obtained by using the Frontal [4] algorithm for meshing both two [5] and three-dimensional [6] entities.

2.2 Part #2

The second part is a handle that should withstand a certain load perpendicular to the base (figure 4). The problem consists of fixing the base and applying a load of 1000 N in the outward direction. The reference mesh has $\ell_c = 4$ mm and results in 39.9k nodes and 25.7k second-order tetrahedra.

The CAEplex meshing settings are shown in appendix B.4 and the general grid can be seen in figure 25. Figure 26 shows a detail of the grid around the location where the maximum stresses are expected. The surfaces were meshed using the Frontal algorithm and the volumes using an improved Delaunay method [7], resulting in 40.0k nodes and 23.7k tetrahedra.

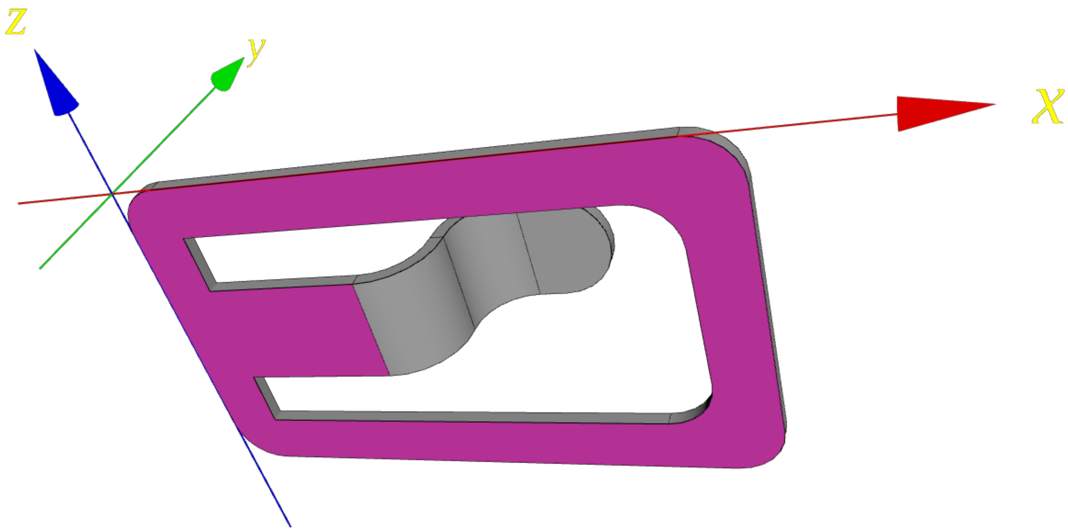
2.3 Part #3

The third part is a plate support, designed to be mounted on a wall with four screws. The cylindrical surfaces that hold the screws are assumed to be fixed and a vertical uniformly distributed load of 15 000 N is applied on the support surface as illustrated in figure 5. Using $\ell_c = 3$ mm, the reference mesh is composed of 43.1k nodes and 28.0k elements.

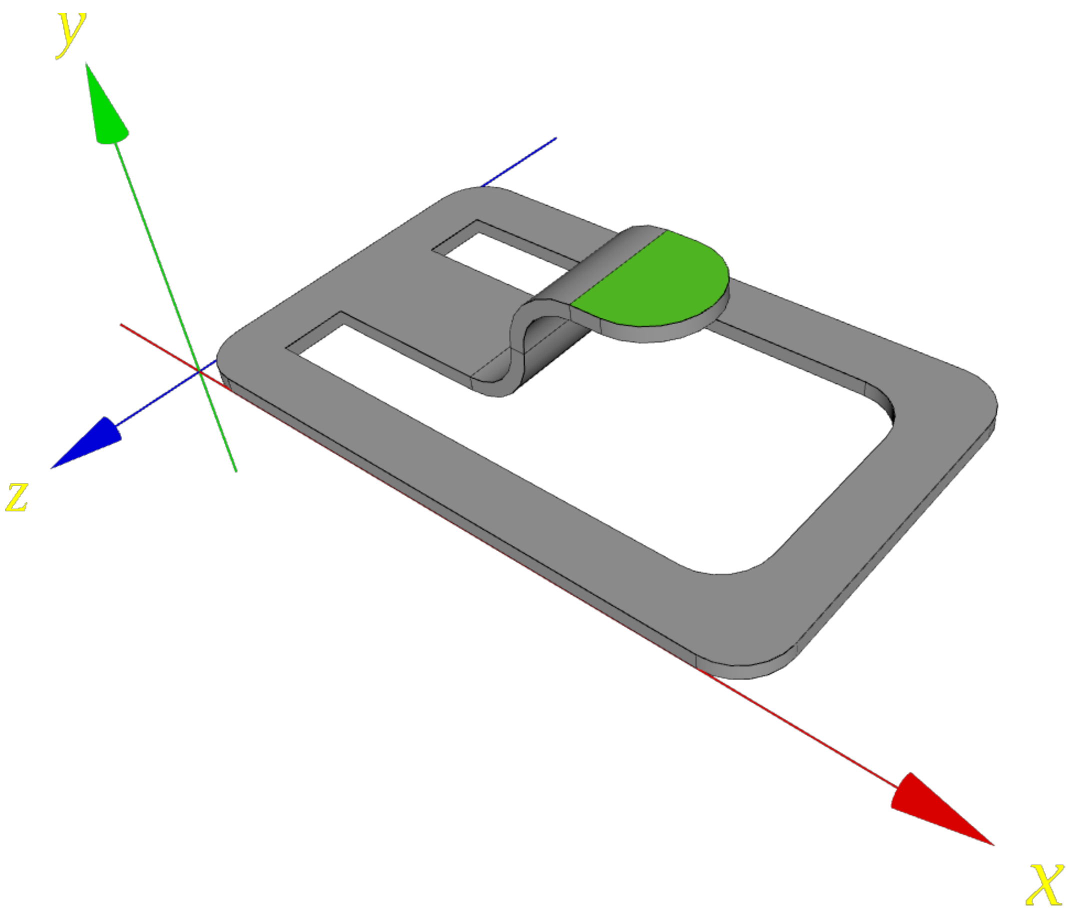
The grid used in CAEplex is described in appendix C.4. The Frontal method was used to obtain the mesh shown in figures ?? and ??. A detail of the second-order triangles generated around the location of the maximum stresses can be seen in figure ??. There are 40.1k nodes and 23.7k second-order tetrahedra.

[3] "Free" both as in "free speech" and in "free beer."

[4] https://www.seamplex.com/principles.html

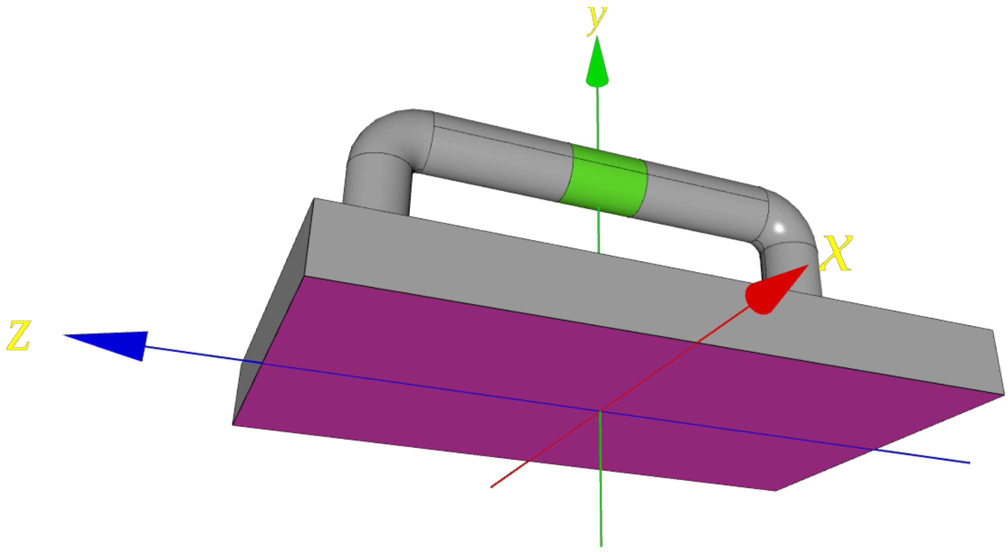


(a) Fixed face (magenta surface): $u = 0$, $v = 0$ and $w = 0$.

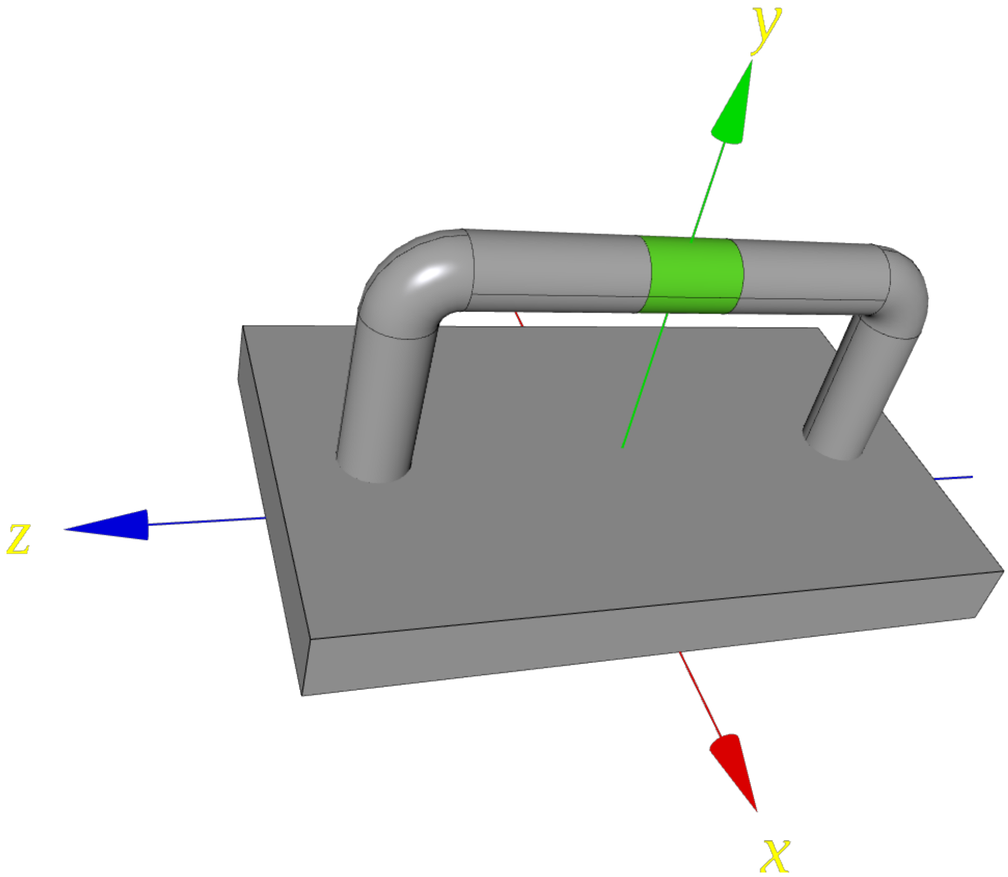


(b) Vertical load (green surface), uniformly distributed 100 N force in $-y$ direction.

Figure 3: Problem definition for part #1.

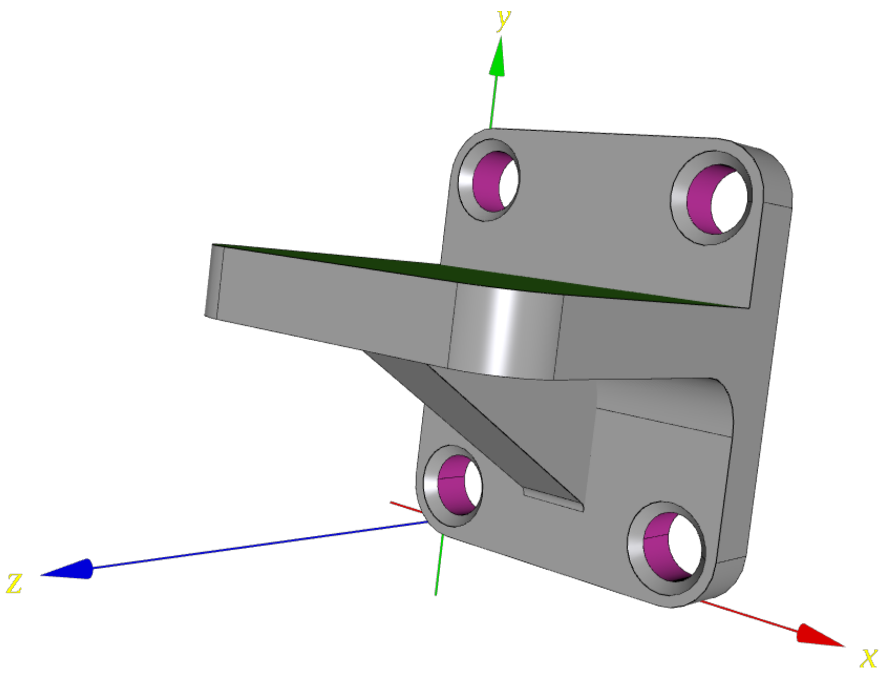


(a) Fixed face (magenta surface): $u = 0$, $v = 0$ and $w = 0$.

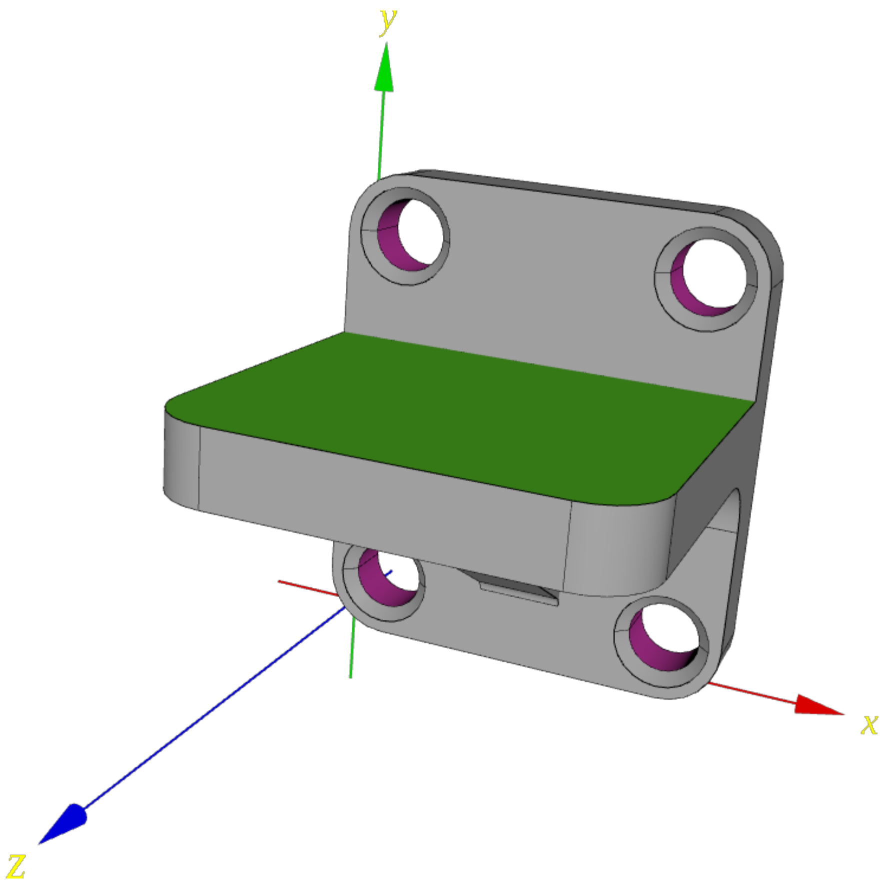


(b) Vertical load (green surface), uniformly distributed 1000 N force in $-y$ direction.

Figure 4



(a) [Fixed faces (magenta surfaces): $u = 0$, $v = 0$ and $w = 0$.



(b) Vertical load (green surface), uniformly distributed 15000 N force in $-y$ direction.

Figure 5

3 Results

As already stated in section 1, even though of course the commercial code can show pretty pictures with the three-dimensional results, it cannot export the full dependence of x, y and z in any reasonable format, not even as an ASCII representation of the numerical data it displays on screen. So a full comparison between the results obtained with Fino cannot be performed.

The problems solved with CAEplex are publicly available online in the URLs listed below, and can be examined interactively by using just a web browser. There is no need to create a user account to access the project in read-only mode. The projects can be cloned and further modified to analyze variations in the mesh or in the solver settings, although a (free) user account is needed for this.

As CAEplex tracks changes in the project, the revision id of each of the three projects used throughout this work is also listed. This way, should the public project have modifications after the report is issued, the current results can be reproduced by reverting the project back to the reported revision.

Benchmark	URL	Revision
#1	`https://caeplex.com/r?0a0`	`e27845e`
#2	`https://caeplex.com/r?ba7`	`5d5f12c`
#3	`https://caeplex.com/r?7bf`	`3d59c84`

3.1 Displacements

Even though the CAEplex platform does not currently allow to see the magnitude of the displacements online in its embedded post-processing view, it can export a the detailed results along with the mesh topology in a VTK file for its further processing, for example with the free and open source tool ParaView. Figure 6 shows the scalar field of the displacements magnitude for the three problems.

In any case, the maximum displacement (which is the direct unknown of the variational formulation used by Fino, and presumably also for the commercial code) for each part obtained with both codes are summarized in the table below.

Benchmark	Fino	SolidWorks
#1	0.3085 mm	0.305 mm
#2	0.01993 mm	0.020 mm
#3	0.09307 mm	0.093 mm

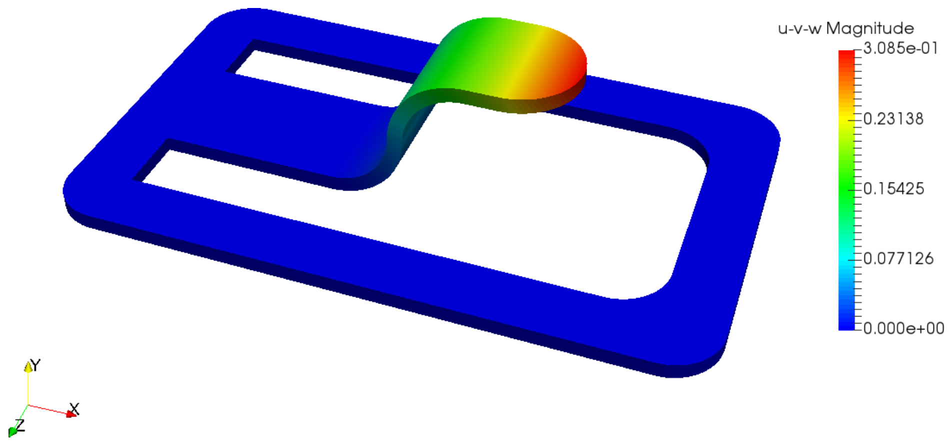
It should be noted that CAEplex gives the maximum location with four significant decimal digits, while the results from the reference report in appendix D are rounded (or truncated?) down to the μmm in an absolute way.

3.2 Stresses

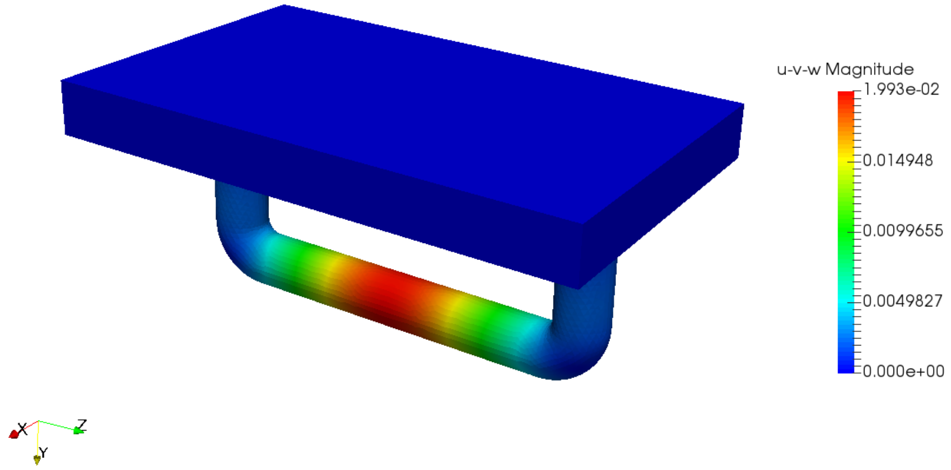
In the displacement-based variational formulation, the fields of node displacements $u(\mathbf{x})$, $v(\mathbf{x})$ and $w(\mathbf{x})$ in each direction are the main unknown. That is to say, the vector that right-multiplies the stiffness matrix to obtain the loads contains the node displacements. The stresses, on the other hand, are secondary unknowns in the sense they have to be computed by first taking the spatial derivatives of each field $u(\mathbf{x})$, $v(\mathbf{x})$ and $w(\mathbf{x})$ with respect to each direction x, y and z, and then using the strain-stress constitutive relationships. This situation plus the fact that finite-element results tend to give node-based results whilst the shape-function derivatives are not continuous at the nodes renders the evaluation of stresses into a far more difficult endeavor—especially for second-order elements. Fino provides a few slightly different choices for this [8]. In general, Gauss-averaged methods avoid spurious highly-localized peaks while node-averaged methods give smoother results. See appendixes A.5, B.5 and C.5 for the actual settings used in each problem.

The spatial distribution of the Von Mises stress obtained with CAEplex can be seen in the appendices A.6, B.6 and C.6. On the other hand, the reference report is not clear about what the maximum Von Mises stress is in each case but it gives three candidate values. Even more, the units of stresses changed from SI to kilograms-force per squared centimeter. For the sake of comparison, we take the reference results as a range by converting back to MPa:

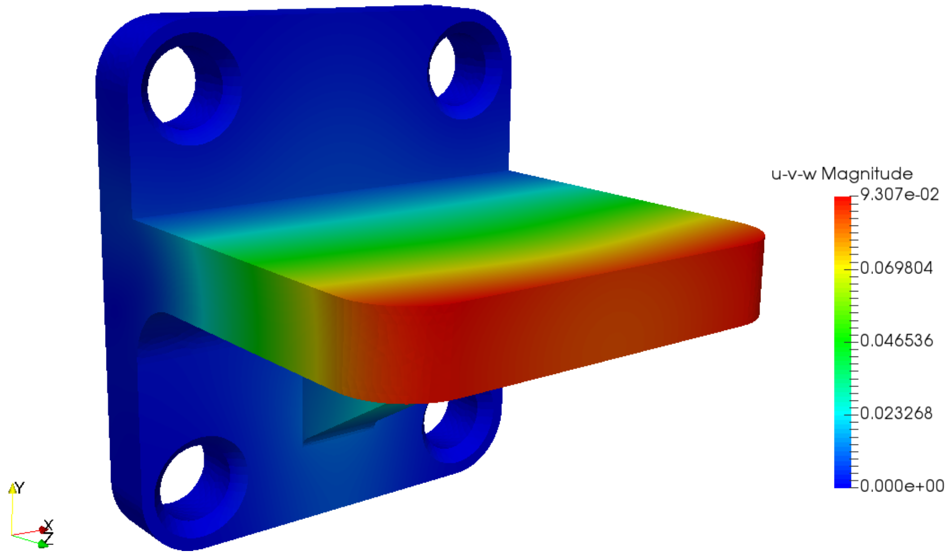
Benchmark	Fino	SolidWorks
#1	89 MPa	88–96 MPa
#2	56 MPa	42–58 MPa
#3	238 MPa	165–241 MPa



(a) Part#1



(b) Part#2



(c) Part#3

Figure 6: Displacement field (magnitude) as post-processed with ParaView out of the VTK output CAEplex provides.

4 Conclusions

Three single-part linear elastic problems were solved using a commercial code (by a technical department of the government of the Santa Fe province in Argentina) and by Fino, through the web-based front-end CAEplex. Actually, the projects can be accessed online at the short URLs shown in page 10, even without needing to create an account at `caeplex.com`. The CAEplex-Fino solution was obtained by trying to keep the same number of nodes in a second-order tetrahedron-based discretization of the domain.

Apparently, the commercial code cannot export the detailed spatial distribution of results so a full comparison of the solution fields cannot be done. Nevertheless, the maximum displacements—that are the primary unknowns, i.e. the solution of the finite-element discretized problem casted as a linear system of equations—coincide in two of the three cases up to the significant digits reported in the reference solution. The other case shows a 1% difference, that may be attributed to differences in the grid. For all practical purposes, the displacements coincide.

On the other hand, the stresses—which are more complex to obtain and that have far more uncertainty as they have to be computed out of the gradient fields of the displacements and evaluated at the nodes[**sigmas**]—also coincide in their maximum values, although the reference report gives a range of candidate values for the maximum Von Mises stress. In two of the three cases (coincidentally the two that give the same displacements with both codes), σ_{max} is located in a sharp edge (part #3) or in small-radius fillet (part #2). It is unclear how the commercial closed-source privative software computes stresses out of the displacements. In Fino, a Gauss-averaged method, that avoids unphysical high derivatives in nodes located in sharp edges, was used in these two problems. In the other problem, an average of the derivatives evaluated at the nodes was used, as this method gives smoother derivatives. In all cases, the maximum Von Mises stresses are consistent between the two finite-element codes.

References

[1] C. Geuzaine and J. F. Remacle. “Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities”. In: *International Journal for Numerical Methods in Engineering* 11.79 (2009), pp. 1309–1331.

[2] Jeremy Theler. *Thermo-elastic expansion of finite cylinders*. Benchmark problem SP-FI-17-BM-5460. Version A. Seamplex, Apr. 12, 2017.

[3] Jeremy Theler. *Parallelepiped whose Young’s modulus is a function of the temperature*. Benchmark problem SP-FI-17-BM-12F2. Version A. Seamplex, June 12, 2017.

[4] S. Rebay. “Efficient unstructured mesh generation by means of Delaunay triangulation and Bowyer-Watson algorithm”. In: *Journal of Computational Physics* 106 (1993), pp. 25–138.

[5] J. F. Remacle et al. “A frontal Delaunay quad mesh generator using the L∞ norm”. In: *International Journal for Numerical Methods in Engineering* 5.94 (2013), pp. 494–512.

[6] J. Schoeberl. “Netgen, an advancing front 2D/3D-mesh generator based on abstract rules”. In: *Computing and Visualization in Science* 1 (1997), pp. 41–52.

[7] H. Si. *Tetgen: a quality tetrahedral mesh generator and three-dimensional Delaunay triangulator*. Tech. rep. 2004.

[8] G. Theler. *Evaluation of partial derivatives of pointwise node-based data over unstructured grids*. Technical Note SP-WA-16-TN-9DB0. Version A. Seamplex, Nov. 23, 2016.

A CAEplex project for part #1

CAEplex[5] project settings	
Name	Benchmark part 1
Author	Jeremy Theler
Owner	`jeremy`
Type	Linear structural analysis
Project ID	`0a051cb92dd0e2e528dbf91e11facf2e`
Creation date	Tue, 12 Sep 2017 15:39:43 -0400 (EDT)
Last modification	Wed, 20 Sep 2017 15:53:29 -0400 (EDT)
Revision	`e27845e`

A.1 Project description

Benchmark to compare CAEplex (Fino) against other FEM codes. Reference solution is max. displ = 0.305mm and max. sigma = 88-96 MPa.

Figure 7: Project URL `https://caeplex.com/r?0a0`

[5]CAEplex is a web-based front-end for cloud-based open-source finite element analysis codes.

A.2 Geometry

Geometry file	
Name	Benchmark part 1
Format	STEP
MD5	8f449db0d8d8e6b04fde22963586a19e
Original size	0.07 Mb
Encrypted size	0.01 Mb
Length units	Millimeters

OpenCASCADE topological entitites	
Vertices	48
Edges	78
Faces	32
Solids	1
Wires	34
Shells	1
Compsolids	0
Compounds	0

Center of gravity	
x_{cog}	59.92 mm
y_{cog}	5.2459 mm
z_{cog}	0 mm

Bounding box	
x_{min}	0 mm
x_{max}	130 mm
y_{min}	0 mm
y_{max}	30 mm
z_{min}	0 mm
z_{max}	0.13 mm

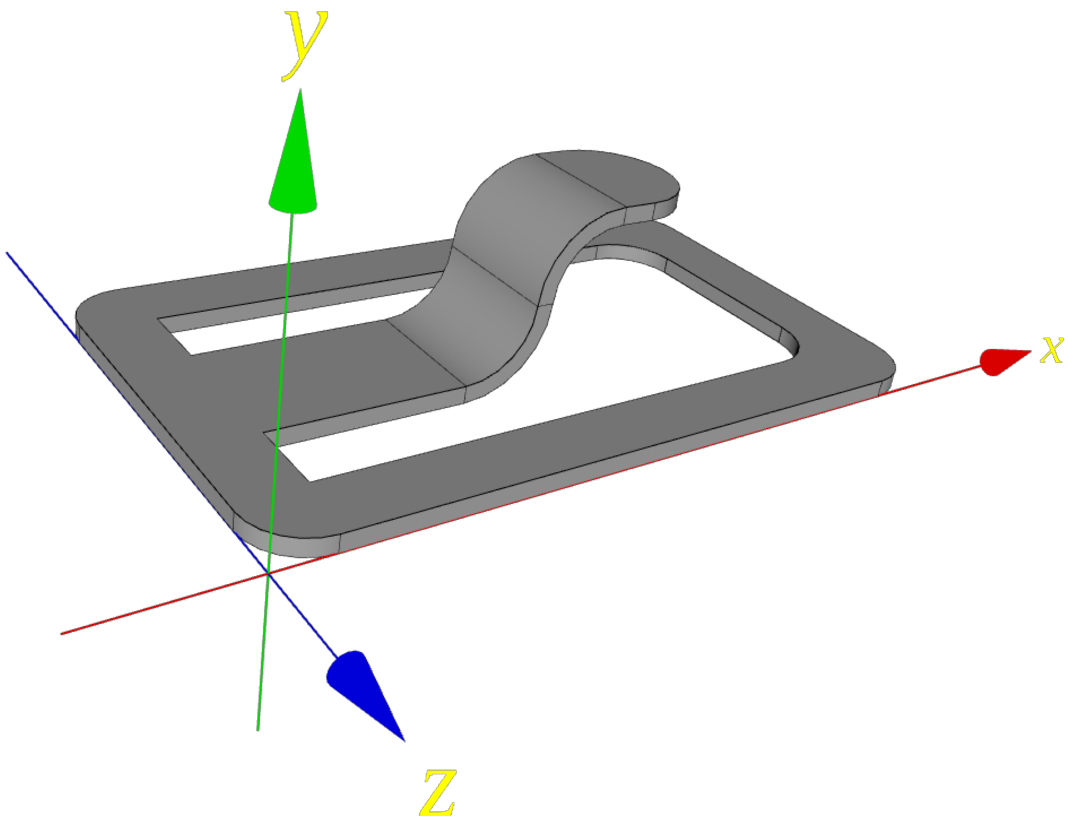


Figure 8: Problem CAD model

A.3 Problem

Material		
Name		Carbon+Steel
Young modulus	$E =$	210 GPa
Poisson ratio	$\nu =$	0.28
Yield strength	$\sigma_{\text{yield}} =$	221 MPa
Density	$\rho =$	7800 kg/m3

Volumetric forces	
Type	None

BC #1: fixation	
Type	Displacement (Dirichlet)
Condition	Fixed

BC #2: load	
Type	Load (Neumann)
Condition	Force
	$F_x = 0$ N
	$F_y = -100$ N
	$F_z = 0$ N

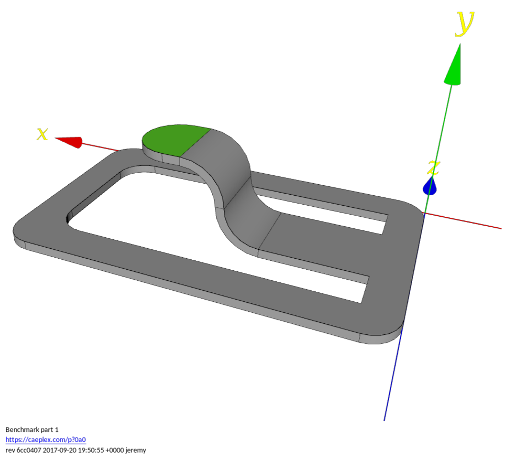


Figure 9: Boundary conditions: vertical load surface in green

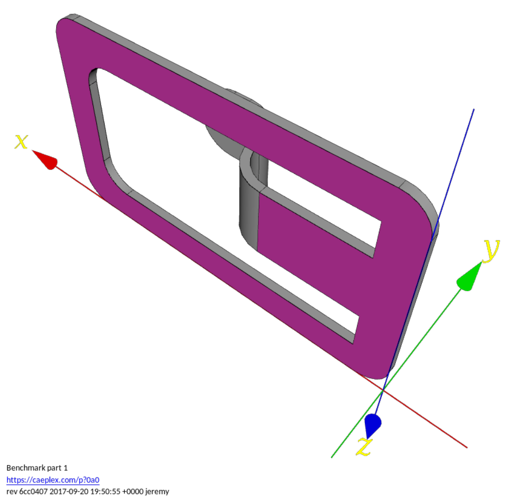


Figure 10: Boundary conditions: fixed face in magenta

A.4 Mesh

Unstructured grid	
Back-end version	Gmsh 3.0.5
Nodes	36,198
Elements	(quadratic) 29,110
Triangles	9,372
Tetrahedra	19,738

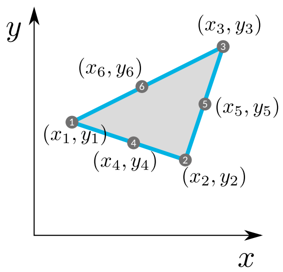
Meshing settings	
Characteristic length ℓ_c	2.1 mm
2D algorithm	Frontal
3D algorithm	Frontal
ℓ_c from	points boundaries
Optimization	basic netgen lloyd high

$$\xi(r,s) = (1 - r - s)$$
$$h_1(r,s) = \xi(r,s) \cdot [2 \cdot \xi(r,s) - 1]$$
$$h_2(r,s) = r \cdot (2 \cdot r - 1)$$
$$h_3(r,s) = s \cdot (2 \cdot s - 1)$$
$$h_4(r,s) = 4 \cdot \xi(r,s) \cdot r$$
$$h_5(r,s) = 4 \cdot r \cdot s$$
$$h_6(r,s) = 4 \cdot s \cdot \xi(r,s)$$

(a) Shape functions

Index	Weight	r	s
1	1/3	1/6	1/6
2	1/3	2/3	1/6
3	1/3	1/6	2/3

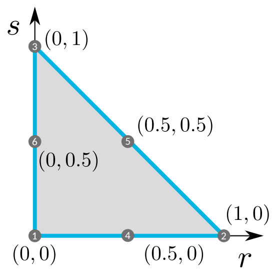
(b) Gauss points



(c) General coordinates

$$x(r,s) = \sum_{j=1}^{6} h_j(r,s) \cdot x_j$$
$$y(r,s) = \sum_{j=1}^{6} h_j(r,s) \cdot y_j$$

(d) Real $\leftrightarrow$ Canonical



(e) Canonical coordinates

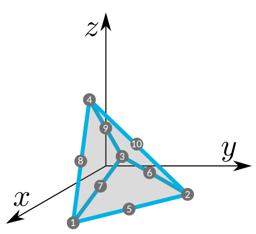
Figure 11: Mathematical details of the triangular second-order isoparametric elements

$$\xi(r,s,t) = 1 - r - s - t$$
$$h_1(r,s,t) = \xi \cdot [2 \cdot \xi - 1]$$
$$h_2(r,s,t) = r \cdot (2 \cdot r - 1)$$
$$h_3(r,s,t) = s \cdot (2 \cdot s - 1)$$
$$h_4(r,s,t) = t \cdot (2 \cdot t - 1)$$
$$h_5(r,s,t) = 4 \cdot \xi \cdot r$$
$$h_6(r,s,t) = 4 \cdot r \cdot s$$
$$h_7(r,s,t) = 4 \cdot s \cdot \xi$$
$$h_8(r,s,t) = 4 \cdot \xi \cdot t$$
$$h_9(r,s,t) = 4 \cdot s \cdot t$$
$$h_{10}(r,s,t) = 4 \cdot r \cdot t$$

(a) Shape functions

Index	Weight	r	s	t
1	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
2	1/4	$\frac{5+3\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
3	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5+3\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
4	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5+3\sqrt{5}}{20}$

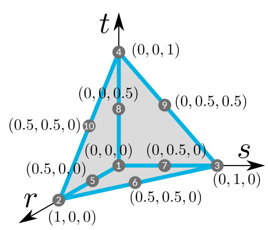
(b) Gauss points



(c) General coordinates

$$x(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot x_j$$
$$y(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot y_j$$
$$z(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot z_j$$

(d) Real $\leftrightarrow$ Canonical



(e) Canonical coordinates

Figure 12: Mathematical details of the tetrahedral second-order isoparametric elements

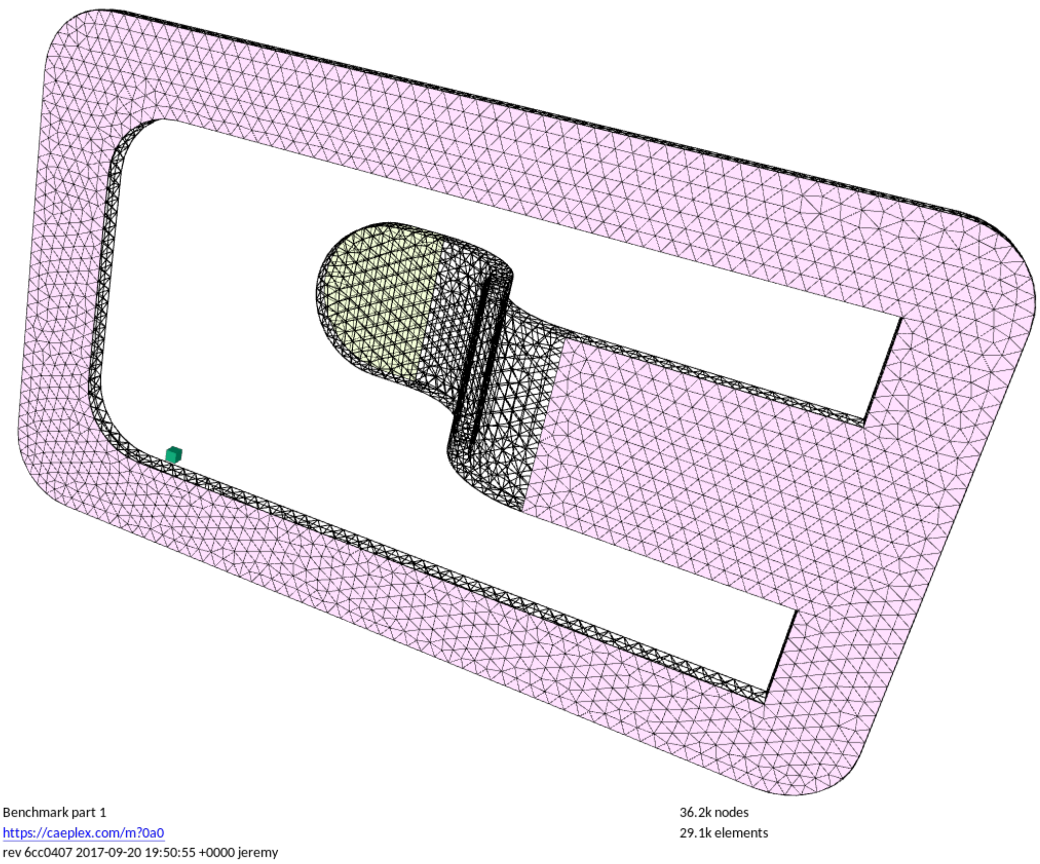


Figure 13: Problem grid

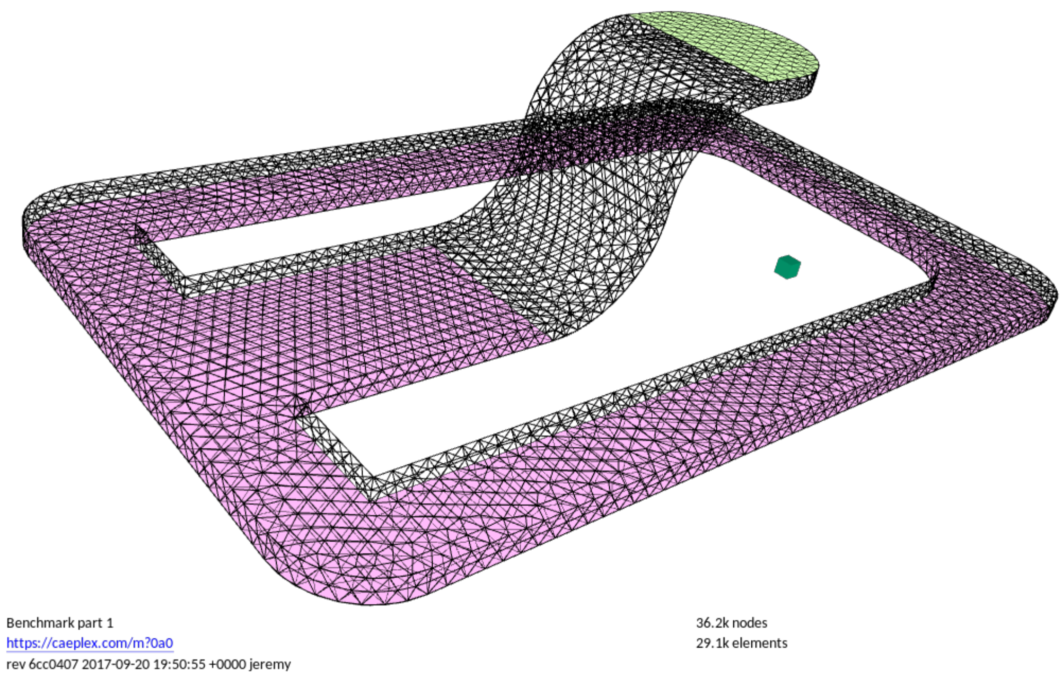


Figure 14: Problem grid

A.5 Solution

Back-end details	
Back-end version	Fino v0.5.78-gcd4cd47
Formulation	Variational displacement-based linear elastic problem
Weighting	Galerkin
Problem size (unknowns)	108,594
Solver library	PETSc 3.7.5
Preconditioner	Default (GAMG)
Krylov method	Default (GMRES)
DOF ordering	Default (Node-based)
Gradient evaluation	Nodal_average

A.5.1 Stiffness matrix

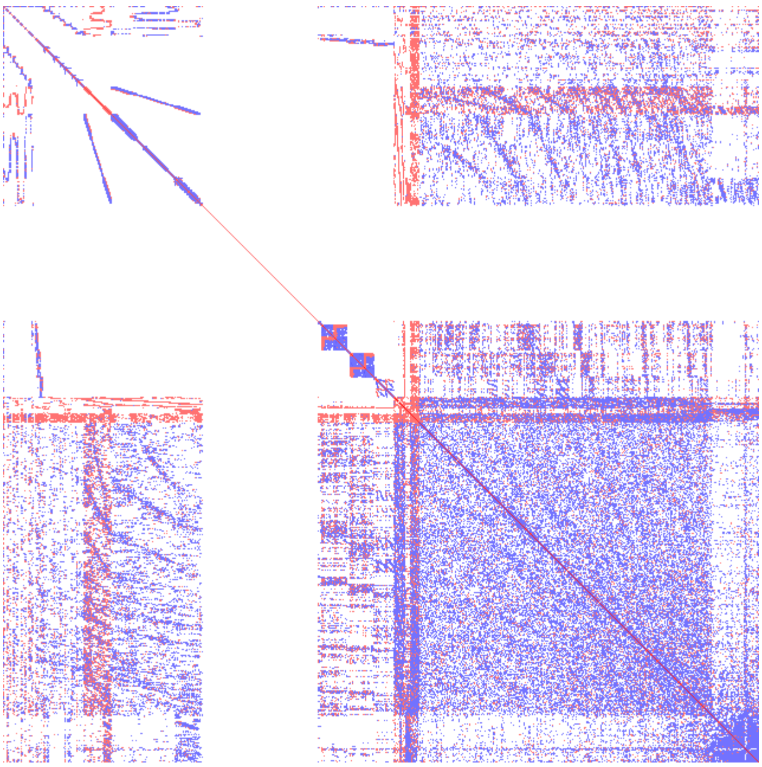


Figure 15: Problem stiffness matrix (108,594 × 108,594). Blue (red) dots represent positive (negative) values.

A.6 Results

Displacement		
Maximum value	$\|(u, v, w)\|_{\text{max}} =$	0.3085 mm
Location	$x =$	92.5 mm
	$y =$	30 mm
	$z =$	-38.1 mm

Von Mises stresses		
Material yield strength	$\sigma_{\text{yield}} =$	221 MPa
Maximum Von Mises stress	$\sigma_{\text{max}} =$	88.77 MPa
Location	$x =$	49.1 mm
	$y =$	0.128 mm
	$z =$	-25.9 mm
Mean value	$\sigma_{\text{mean}} =$	4.382 MPa
Peak factor	$f_p =$	20.26

Maximum load level with respect to material yield

40 % (89/221)

Mean load level with respect to material yield

2 % (4/221)

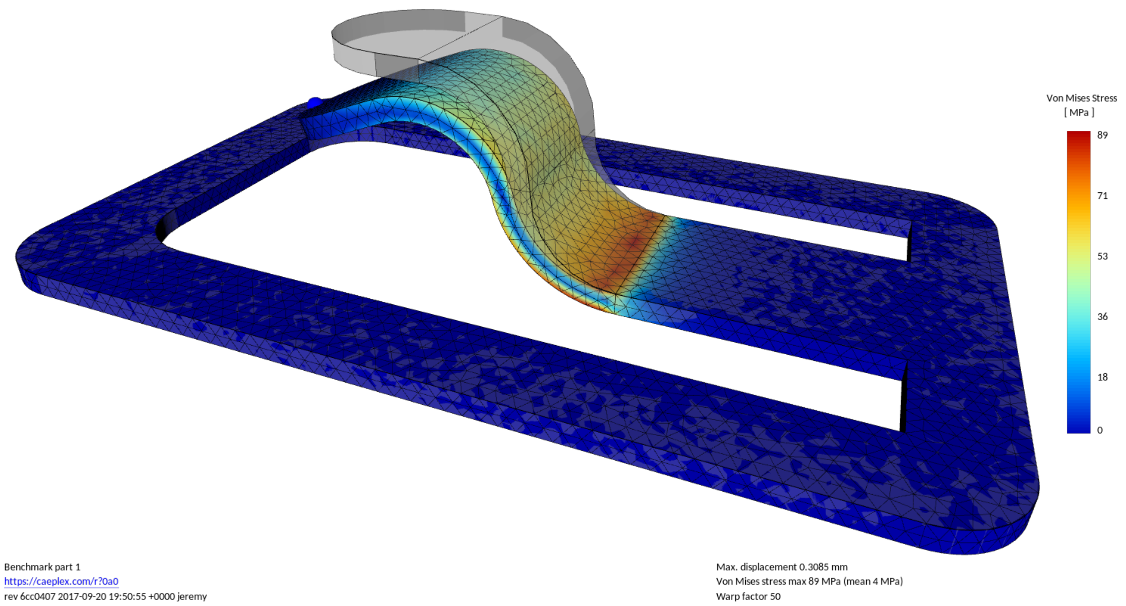


Figure 16: Von mises stresses and warped displacements with respect to the original geometry

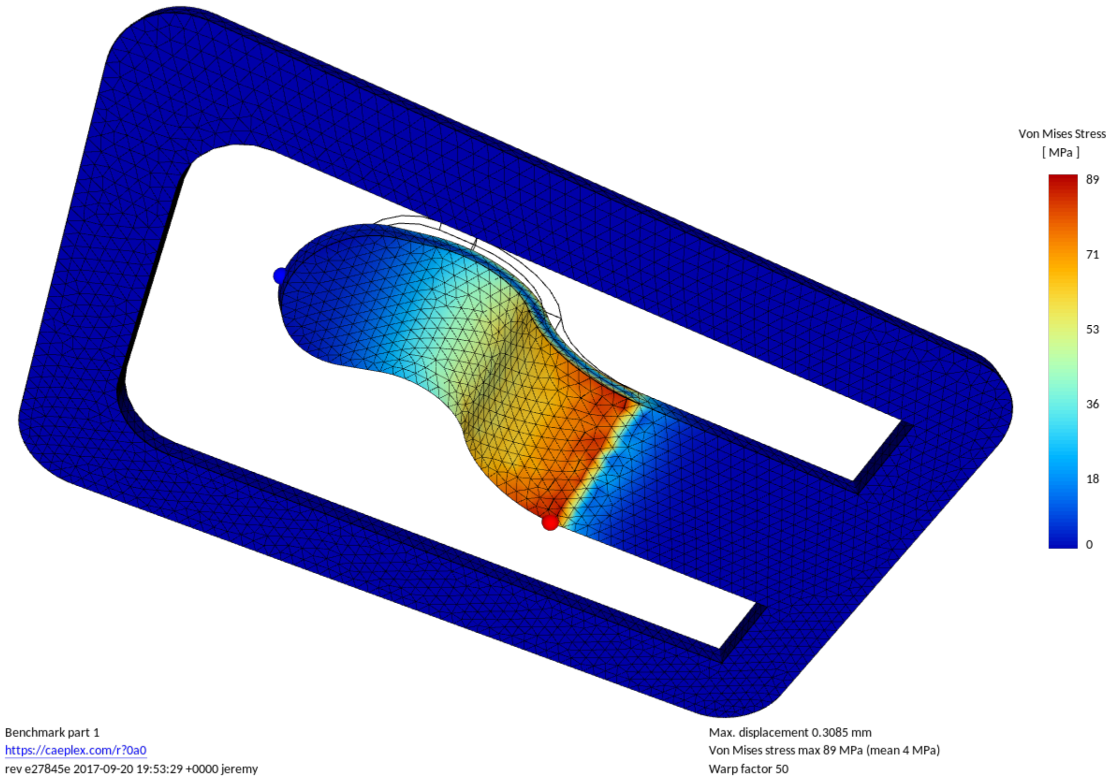


Figure 17: Location of the maximum stress

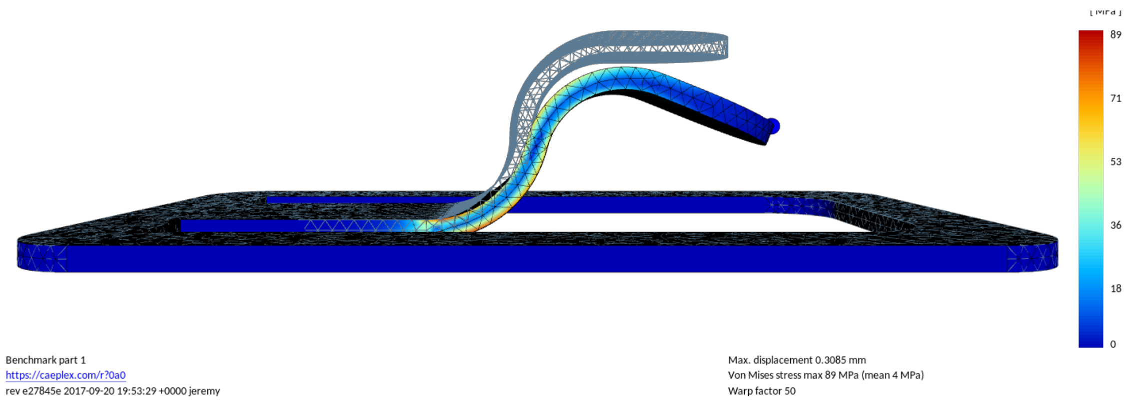


Figure 18: Side warped view

B CAEplex project for part #2

CAEplex[6] project settings	
Name	Benchmark part 2
Author	Jeremy Theler
Owner	`jeremy`
Type	Linear structural analysis
Project ID	`ba7e27719a4bc8e9a89e37b59610c925`
Creation date	Tue, 12 Sep 2017 15:40:12 -0400 (EDT)
Last modification	Wed, 20 Sep 2017 16:07:16 -0400 (EDT)
Revision	`5d5f12c`

B.1 Project description

Benchmark to compare CAEplex (Fino) against other FEM codes. Reference solution is max. displ = 0.020mm and max. sigma = 42-58 MPa.

Figure 19: Project URL `https://caeplex.com/r?ba7`

[6]CAEplex is a web-based front-end for cloud-based open-source finite element analysis codes.

B.2 Geometry

Geometry file	
Name	Benchmark part 2
Format	STEP
MD5	281e653feca87ae2103f16c6fa1e0b85
Original size	0.05 Mb
Encrypted size	0.01 Mb
Length units	Millimeters

OpenCASCADE topological entitites	
Vertices	28
Edges	46
Faces	20
Solids	1
Wires	22
Shells	1
Compsolids	0
Compounds	0

Center of gravity	
x_{cog}	0 mm
y_{cog}	13.5306 mm
z_{cog}	0 mm

Bounding box	
x_{min}	0 mm
x_{max}	45 mm
y_{min}	0 mm
y_{max}	68 mm
z_{min}	0 mm
z_{max}	75 mm

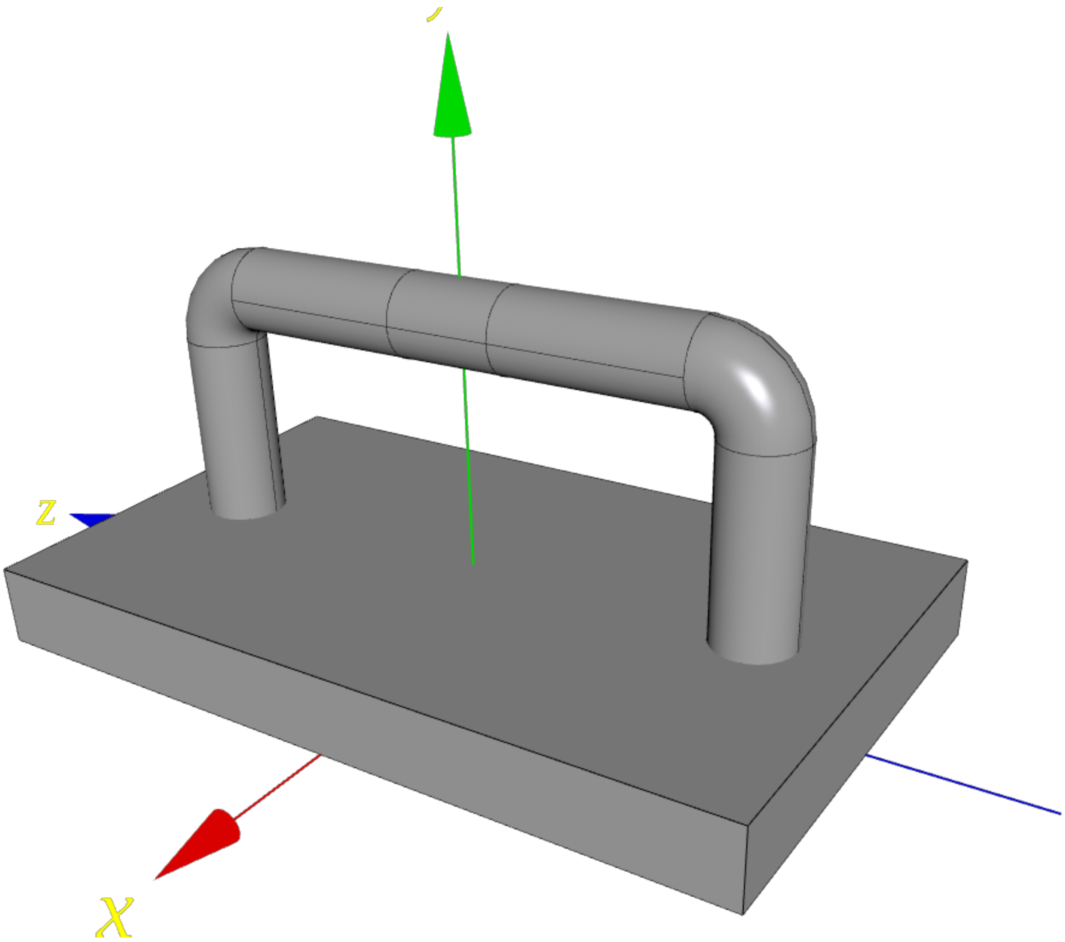


Figure 20: Problem CAD model

B.3 Problem

Material		
Name		Carbon+Steel
Young modulus	$E =$	210 GPa
Poisson ratio	$\nu =$	0.28
Yield strength	$\sigma_{\text{yield}} =$	221 MPa
Density	$\rho =$	7800 kg/m3

Volumetric forces	
Type	None

BC #1: fixation	
Type	Displacement (Dirichlet)
Condition	Fixed

BC #2: load	
Type	Load (Neumann)
Condition	Force
	$F_x = 0$ N
	$F_y = 1000$ N
	$F_z = 0$ N

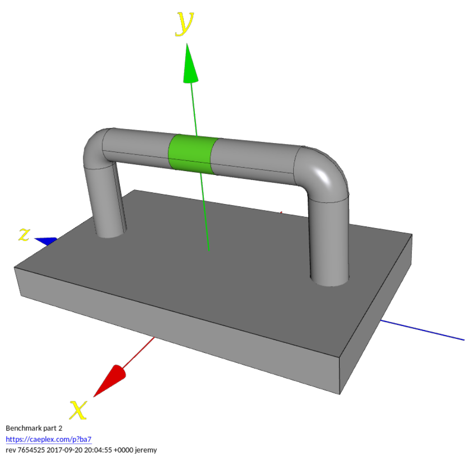


Figure 21: Boundary conditions: vertical load surface in green

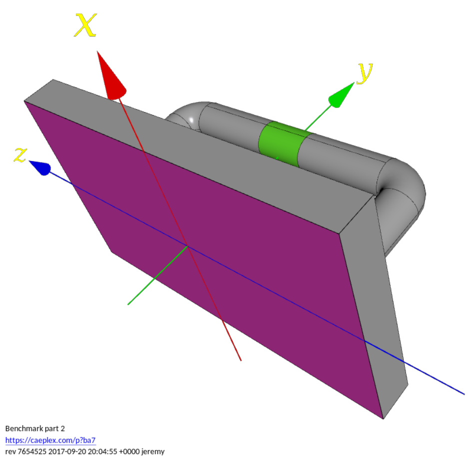


Figure 22: Boundary conditions: fixed face in magenta

B.4 Mesh

Unstructured grid	
Back-end version	Gmsh 3.0.5
Nodes	40,057
Elements	(quadratic) 31,775
Triangles	8,004
Tetrahedra	23,771

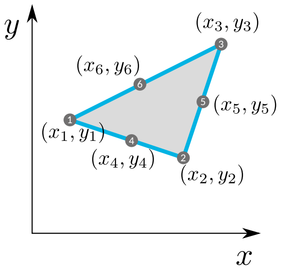
Meshing settings	
Characteristic length ℓ_c	3.8 mm
2D algorithm	Frontal
3D algorithm	Newdelaunay
ℓ_c from	points curvature boundaries
Optimization	basic netgen lloyd high

$$\xi(r,s) = (1 - r - s)$$
$$h_1(r,s) = \xi(r,s) \cdot [2 \cdot \xi(r,s) - 1]$$
$$h_2(r,s) = r \cdot (2 \cdot r - 1)$$
$$h_3(r,s) = s \cdot (2 \cdot s - 1)$$
$$h_4(r,s) = 4 \cdot \xi(r,s) \cdot r$$
$$h_5(r,s) = 4 \cdot r \cdot s$$
$$h_6(r,s) = 4 \cdot s \cdot \xi(r,s)$$

(a) Shape functions

Index	Weight	r	s
1	1/3	1/6	1/6
2	1/3	2/3	1/6
3	1/3	1/6	2/3

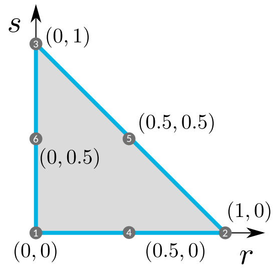
(b) Gauss points



(c) General coordinates

$$x(r,s) = \sum_{j=1}^{6} h_j(r,s) \cdot x_j$$
$$y(r,s) = \sum_{j=1}^{6} h_j(r,s) \cdot y_j$$

(d) Real $\leftrightarrow$ Canonical



(e) Canonical coordinates

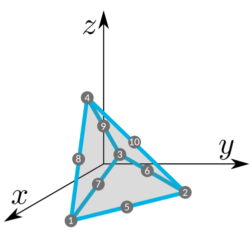
Figure 23: Mathematical details of the triangular second-order isoparametric elements

$$\xi(r,s,t) = 1 - r - s - t$$
$$h_1(r,s,t) = \xi \cdot [2 \cdot \xi - 1]$$
$$h_2(r,s,t) = r \cdot (2 \cdot r - 1)$$
$$h_3(r,s,t) = s \cdot (2 \cdot s - 1)$$
$$h_4(r,s,t) = t \cdot (2 \cdot t - 1)$$
$$h_5(r,s,t) = 4 \cdot \xi \cdot r$$
$$h_6(r,s,t) = 4 \cdot r \cdot s$$
$$h_7(r,s,t) = 4 \cdot s \cdot \xi$$
$$h_8(r,s,t) = 4 \cdot \xi \cdot t$$
$$h_9(r,s,t) = 4 \cdot s \cdot t$$
$$h_{10}(r,s,t) = 4 \cdot r \cdot t$$

(a) Shape functions

Index	Weight	r	s	t
1	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
2	1/4	$\frac{5+3\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
3	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5+3\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
4	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5+3\sqrt{5}}{20}$

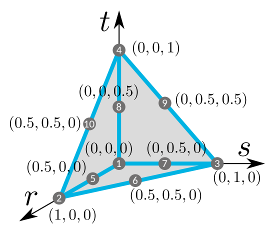
(b) Gauss points



(c) General coordinates

$$x(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot x_j$$
$$y(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot y_j$$
$$z(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot z_j$$

(d) Real $\leftrightarrow$ Canonical



(e) Canonical coordinates

Figure 24: Mathematical details of the tetrahedral second-order isoparametric elements

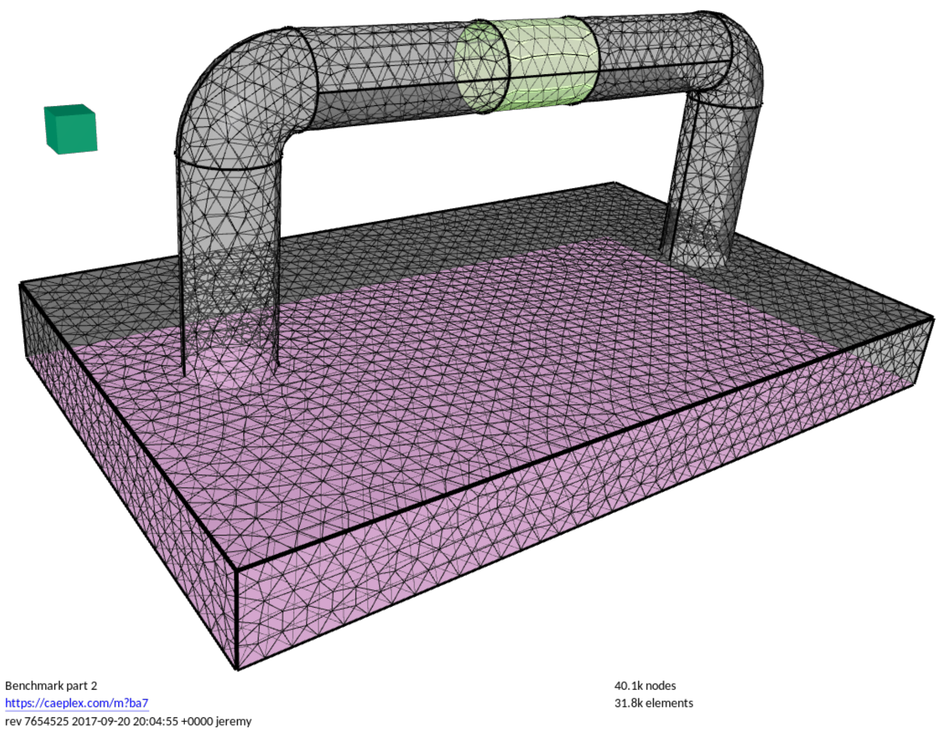


Figure 25: Mesh

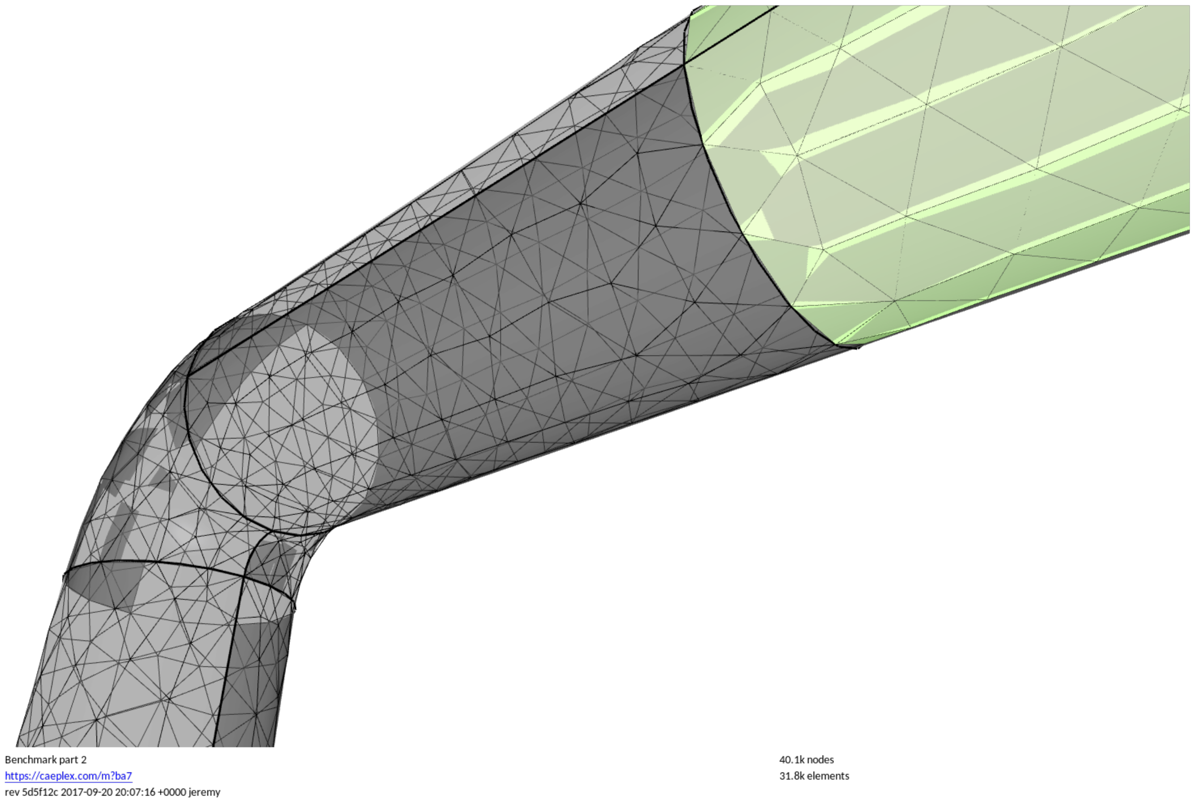


Figure 26: Mesh detail

B.5 Solution

Back-end details	
Back-end version	Fino v0.5.78-gcd4cd47
Formulation	Variational displacement-based linear elastic problem
Weighting	Galerkin
Problem size (unknowns)	120,171
Solver library	PETSc 3.7.5
Preconditioner	Default (GAMG)
Krylov method	Default (GMRES)
DOF ordering	Default (Node-based)
Gradient evaluation	Gauss_average

B.5.1 Stiffness matrix

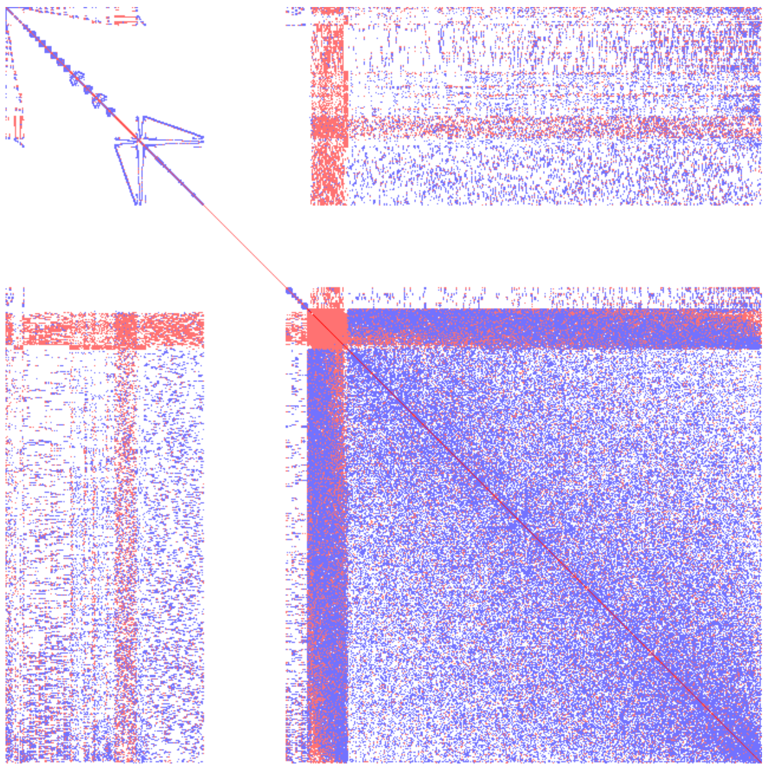


Figure 27: Problem stiffness matrix (120,171 × 120,171). Blue (red) dots represent positive (negative) values.

B.6 Results

Displacement		
Maximum value	$\lvert(u,v,w)\rvert_{\max} =$	0.019 93 mm
Location	$x =$	7.5 mm
	$y =$	60.1 mm
	$z =$	8.69×10^{-13} mm

Von Mises stresses		
Material yield strength	$\sigma_{\text{yield}} =$	221 MPa
Maximum Von Mises stress	$\sigma_{\max} =$	56.19 MPa
Location	$x =$	-1.08 mm
	$y =$	51.2 mm
	$z =$	47.4 mm
Mean value	$\sigma_{\text{mean}} =$	1.374 MPa
Peak factor	$f_p =$	40.90

Maximum load level with respect to material yield

Mean load level with respect to material yield

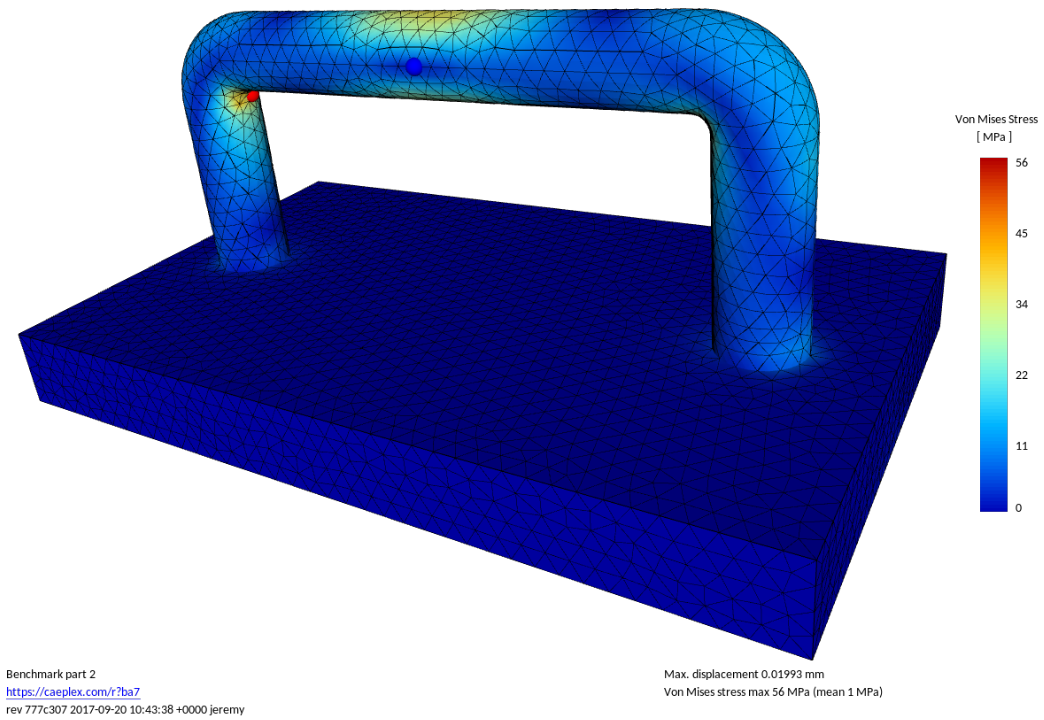


Figure 28: Von mises stresses

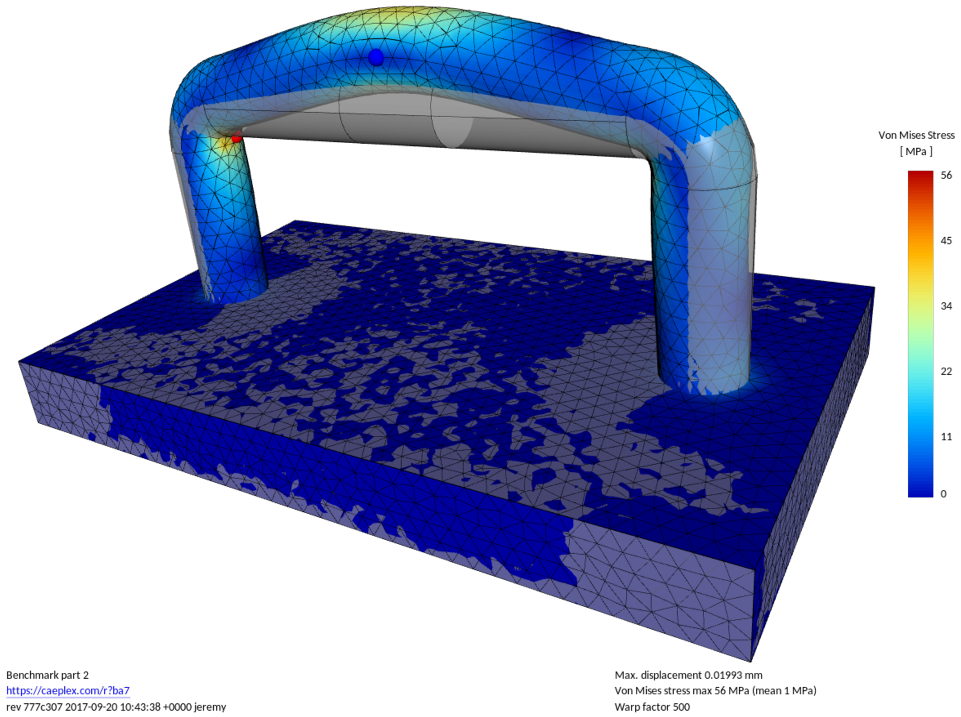


Figure 29: Von mises stresses and warped displacements with respect to the original geometry

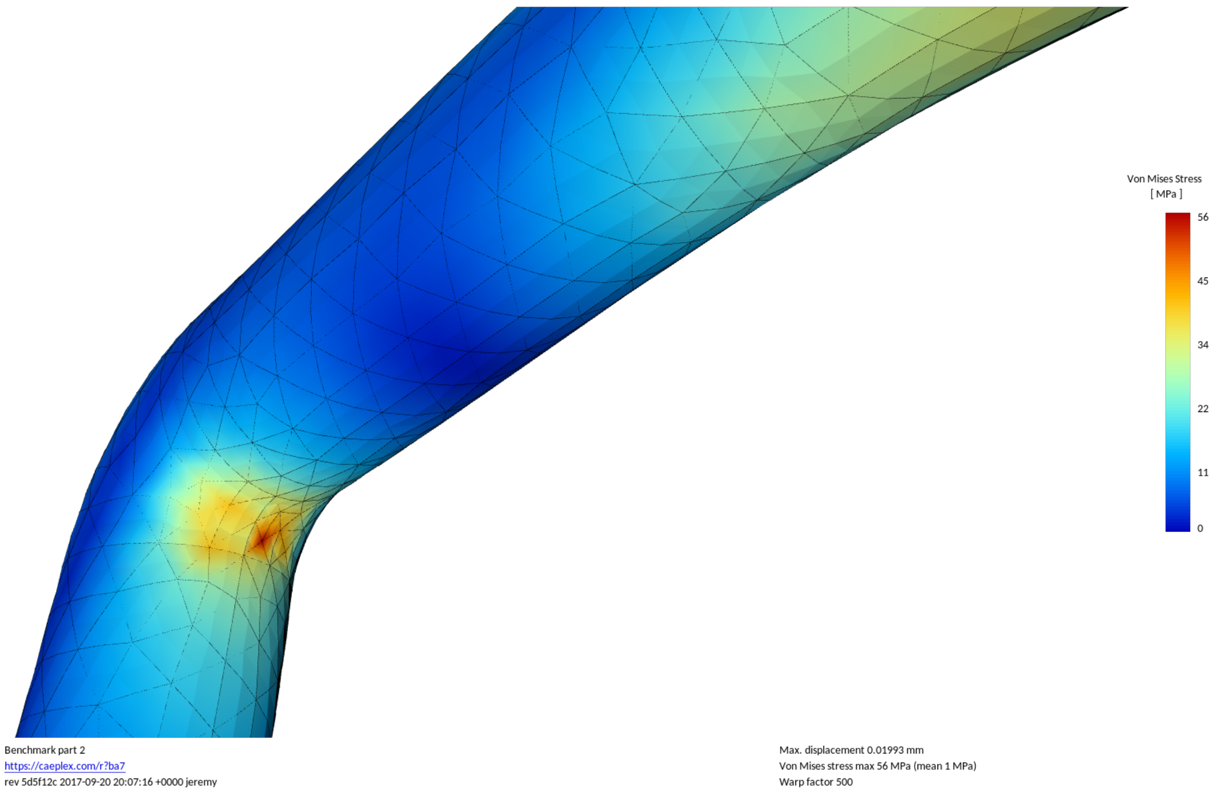


Figure 30: Detailed view of the maximum stress location

C CAEplex project for part #3

CAEplex[7] project settings	
Name	Benchmark part 3
Author	Jeremy Theler
Owner	`jeremy`
Type	Linear structural analysis
Project ID	`7bf708acd0b6196f21af91f8d63962ad`
Creation date	Tue, 12 Sep 2017 15:40:18 -0400 (EDT)
Last modification	Wed, 20 Sep 2017 16:11:40 -0400 (EDT)
Revision	`3d59c84`

C.1 Project description

Benchmark to compare CAEplex (Fino) against other FEM codes. Reference solution is max. displ = 0.093mm and max. sigma = 165-241 MPa.

Figure 31: Project URL `https://caeplex.com/r?7bf`

[7] CAEplex is a web-based front-end for cloud-based open-source finite element analysis codes.

C.2 Geometry

Geometry file	
Name	Benchmark part 3
Format	STEP
MD5	`f69bbabc1a669f47db058f6eaf379d8c`
Original size	0.09 Mb
Encrypted size	0.01 Mb
Length units	Millimeters

OpenCASCADE topological entitites	
Vertices	64
Edges	100
Faces	38
Solids	1
Wires	46
Shells	1
Compsolids	0
Compounds	0

Center of gravity	
x_{cog}	35 mm
y_{cog}	42.2743 mm
z_{cog}	22.1226 mm

Bounding box	
x_{min}	0 mm
x_{max}	70 mm
y_{min}	0 mm
y_{max}	80 mm
z_{min}	0 mm
z_{max}	70 mm

C.3 Problem

Material		
Name		Carbon+Steel
Young modulus	$E =$	210 GPa
Poisson ratio	$\nu =$	0.28
Yield strength	$\sigma_{\text{yield}} =$	221 MPa
Density	$\rho =$	7800 kg/m3

Volumetric forces	
Type	None

BC #1: fixation	
Type	Displacement (Dirichlet)
Condition	Fixed

BC #2: load	
Type	Load (Neumann)
Condition	Force
	$F_x = 0$ N
	$F_y = -15\,000$ N
	$F_z = 0$ N

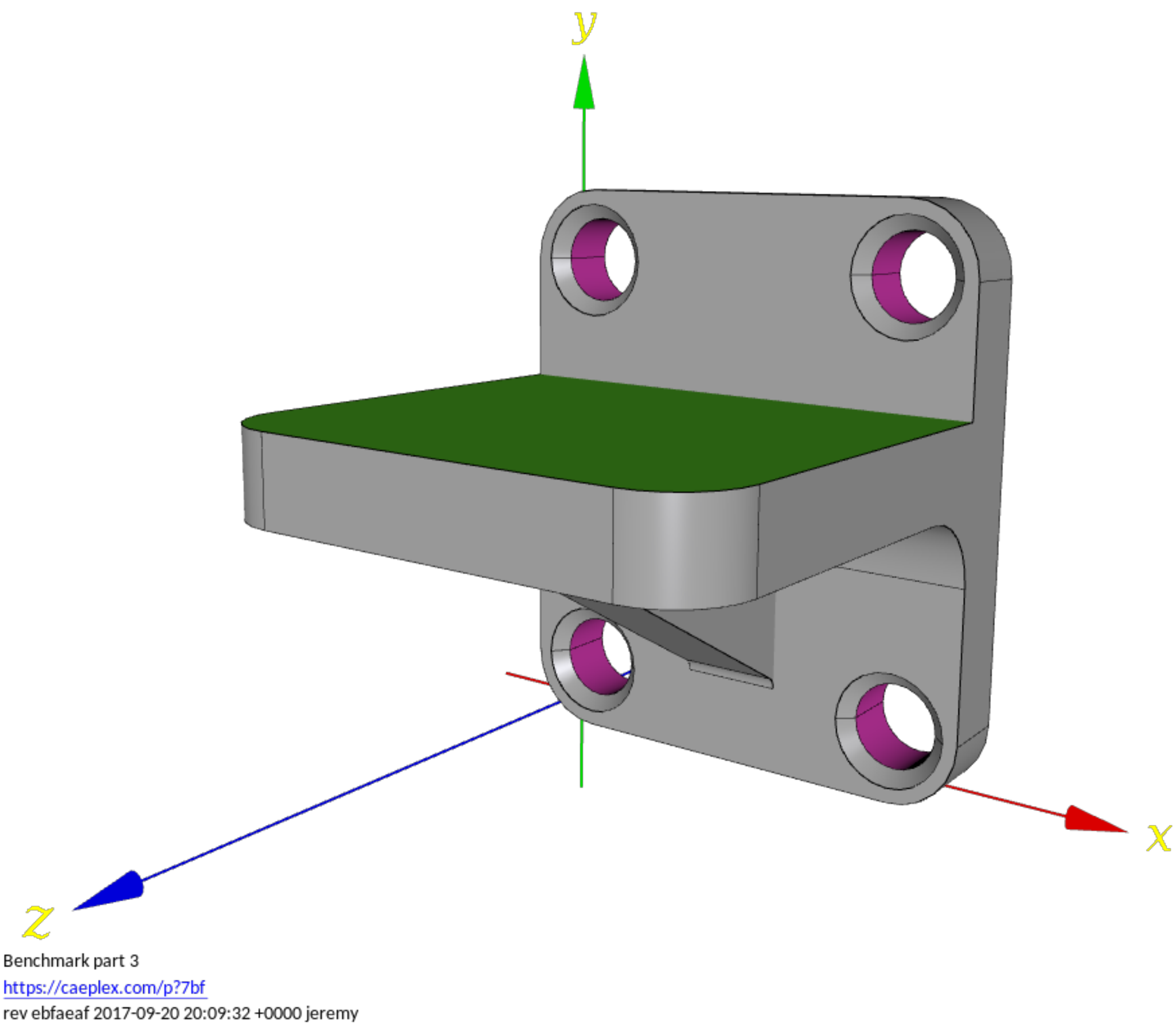


Figure 32: Boundary conditions: vertical load surface in green

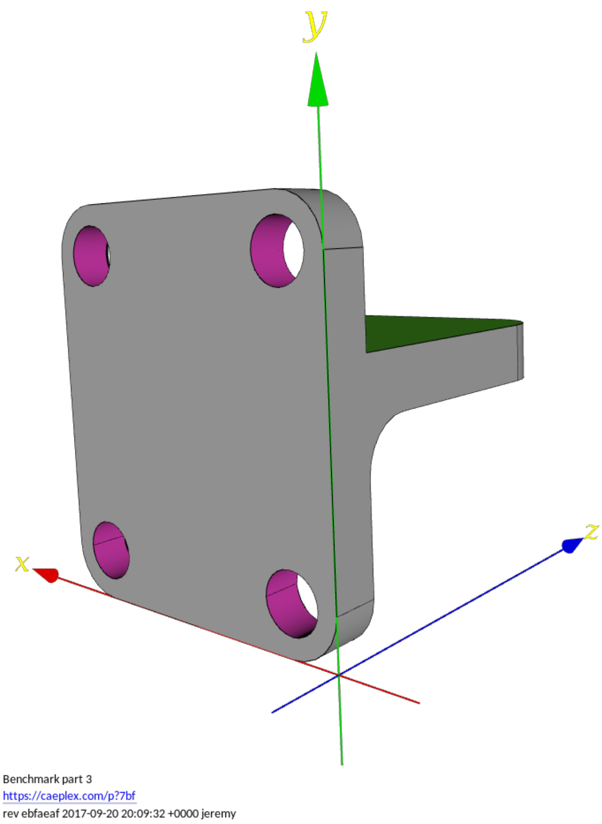


Figure 33: Boundary conditions: fixed face in magenta

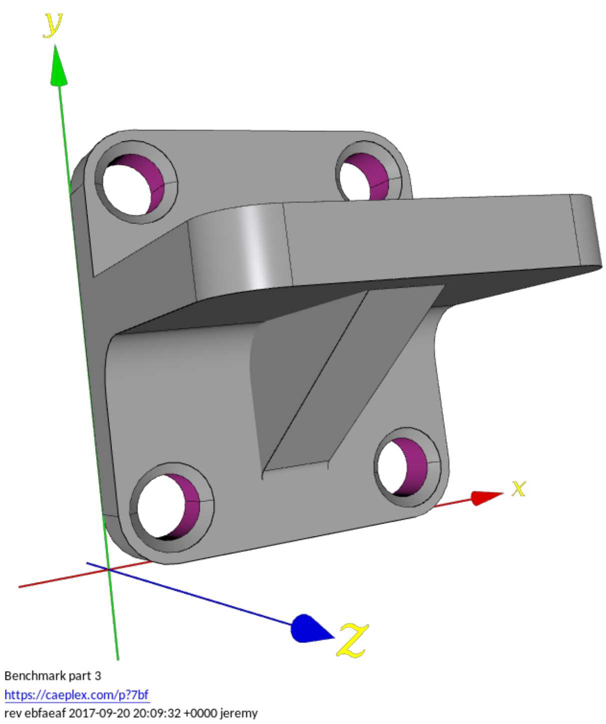


Figure 34: Boundary conditions: fixed face in magenta

C.4 Mesh

Unstructured grid	
Back-end version	Gmsh 3.0.5
Nodes	40,126
Elements	(quadratic) 31,871
Triangles	8,152
Tetrahedra	23,719

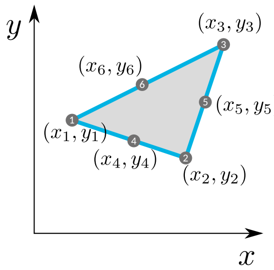
Meshing settings	
Characteristic length ℓ_c	2.9 mm
2D algorithm	Frontal
3D algorithm	Frontal
ℓ_c from	points curvature boundaries
Optimization	basic netgen lloyd high

$$\xi(r,s) = (1 - r - s)$$
$$h_1(r,s) = \xi(r,s) \cdot [2 \cdot \xi(r,s) - 1]$$
$$h_2(r,s) = r \cdot (2 \cdot r - 1)$$
$$h_3(r,s) = s \cdot (2 \cdot s - 1)$$
$$h_4(r,s) = 4 \cdot \xi(r,s) \cdot r$$
$$h_5(r,s) = 4 \cdot r \cdot s$$
$$h_6(r,s) = 4 \cdot s \cdot \xi(r,s)$$

(a) Shape functions

Index	Weight	r	s
1	1/3	1/6	1/6
2	1/3	2/3	1/6
3	1/3	1/6	2/3

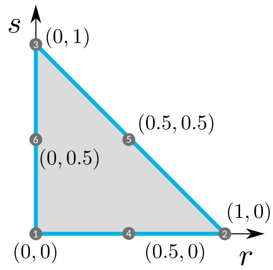
(b) Gauss points



(c) General coordinates

$$x(r,s) = \sum_{j=1}^{6} h_j(r,s) \cdot x_j$$
$$y(r,s) = \sum_{j=1}^{6} h_j(r,s) \cdot y_j$$

(d) Real $\leftrightarrow$ Canonical



(e) Canonical coordinates

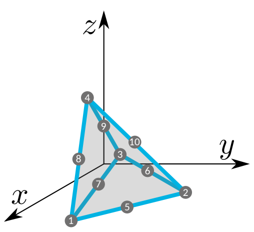
Figure 35: Mathematical details of the triangular second-order isoparametric elements

$$\xi(r,s,t) = 1 - r - s - t$$
$$h_1(r,s,t) = \xi \cdot [2 \cdot \xi - 1]$$
$$h_2(r,s,t) = r \cdot (2 \cdot r - 1)$$
$$h_3(r,s,t) = s \cdot (2 \cdot s - 1)$$
$$h_4(r,s,t) = t \cdot (2 \cdot t - 1)$$
$$h_5(r,s,t) = 4 \cdot \xi \cdot r$$
$$h_6(r,s,t) = 4 \cdot r \cdot s$$
$$h_7(r,s,t) = 4 \cdot s \cdot \xi$$
$$h_8(r,s,t) = 4 \cdot \xi \cdot t$$
$$h_9(r,s,t) = 4 \cdot s \cdot t$$
$$h_{10}(r,s,t) = 4 \cdot r \cdot t$$

(a) Shape functions

Index	Weight	r	s	t
1	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
2	1/4	$\frac{5+3\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
3	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5+3\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$
4	1/4	$\frac{5-\sqrt{5}}{20}$	$\frac{5-\sqrt{5}}{20}$	$\frac{5+3\sqrt{5}}{20}$

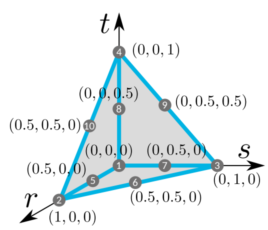
(b) Gauss points



(c) General coordinates

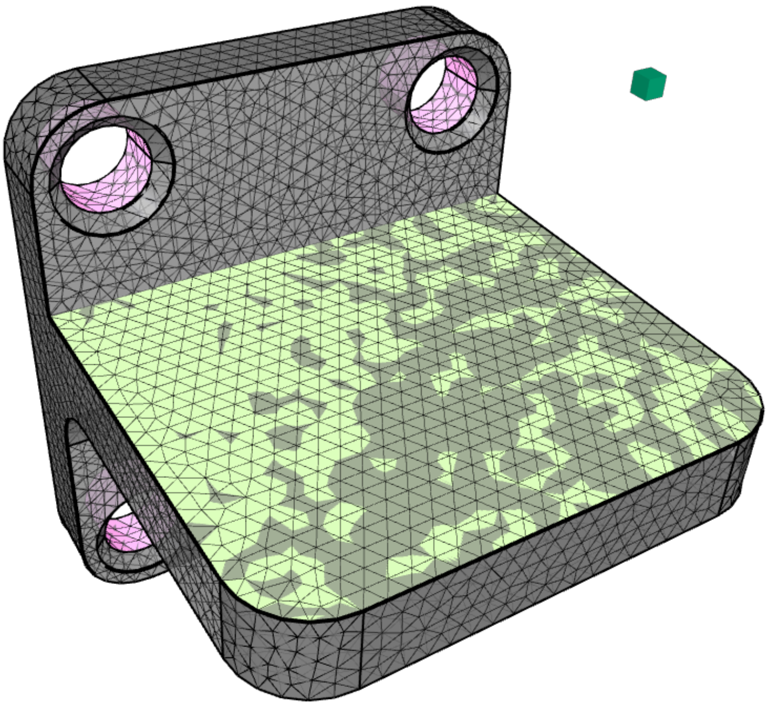
$$x(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot x_j$$
$$y(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot y_j$$
$$z(r,s,t) = \sum_{j=1}^{10} h_j(r,s,t) \cdot z_j$$

(d) Real $\leftrightarrow$ Canonical



(e) Canonical coordinates

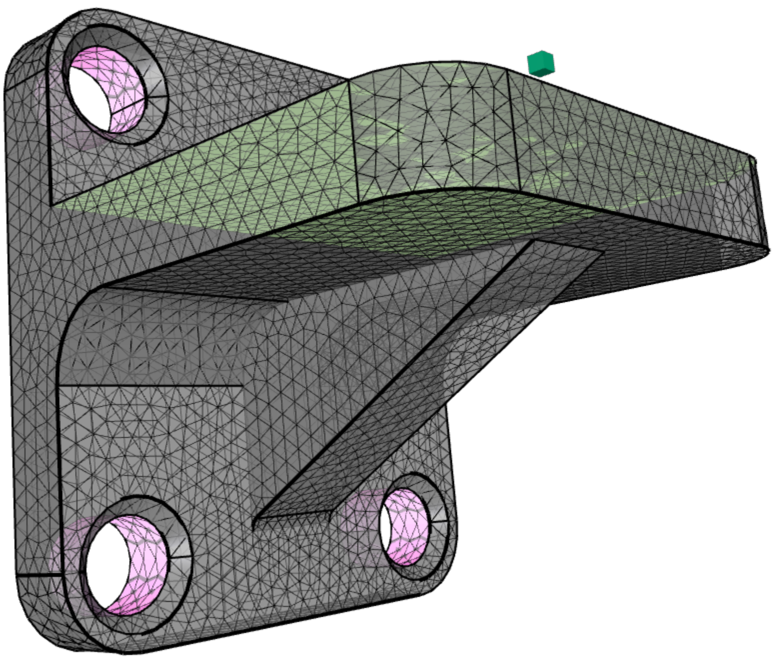
Figure 36: Mathematical details of the tetrahedral second-order isoparametric elements



Benchmark part 3
https://caeplex.com/m?7bf
rev ebfaeaf 2017-09-20 20:09:32 +0000 jeremy

40.1k nodes
31.9k elements

Figure 37: Problem grid



Benchmark part 3
https://caeplex.com/m?7bf
rev ebfaeaf 2017-09-20 20:09:32 +0000 jeremy

40.1k nodes
31.9k elements

Figure 38: Problem grid

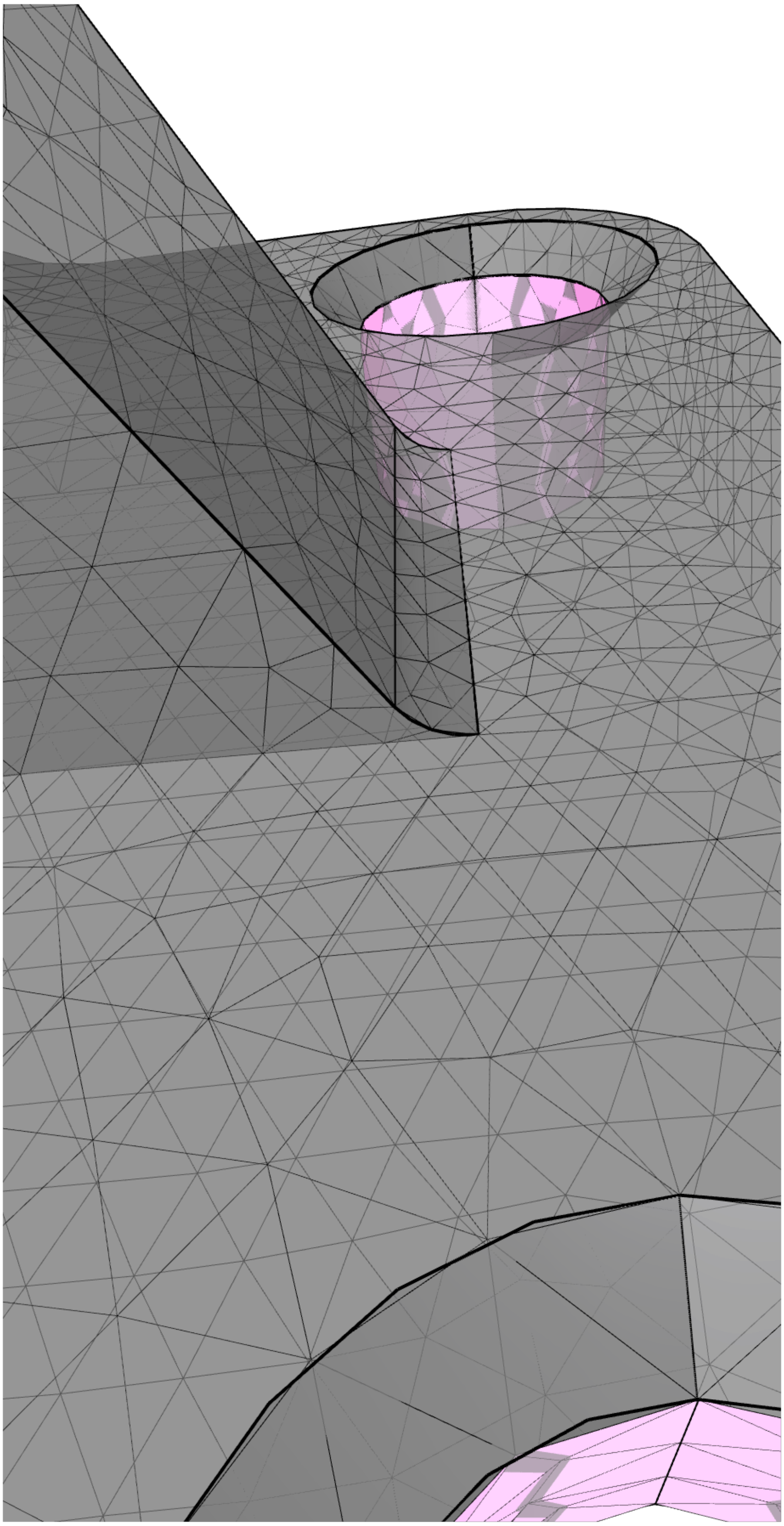


Figure 39: Detailed view of the discertization around the maximum stress

C.5 Solution

Back-end details	
Back-end version	Fino v0.5.78-gcd4cd47
Formulation	Variational displacement-based linear elastic problem
Weighting	Galerkin
Problem size (unknowns)	120,378
Solver library	PETSc 3.7.5
Preconditioner	Default (GAMG)
Krylov method	Default (GMRES)
DOF ordering	Default (Node-based)
Gradient evaluation	Gauss_average

C.5.1 Stiffness matrix

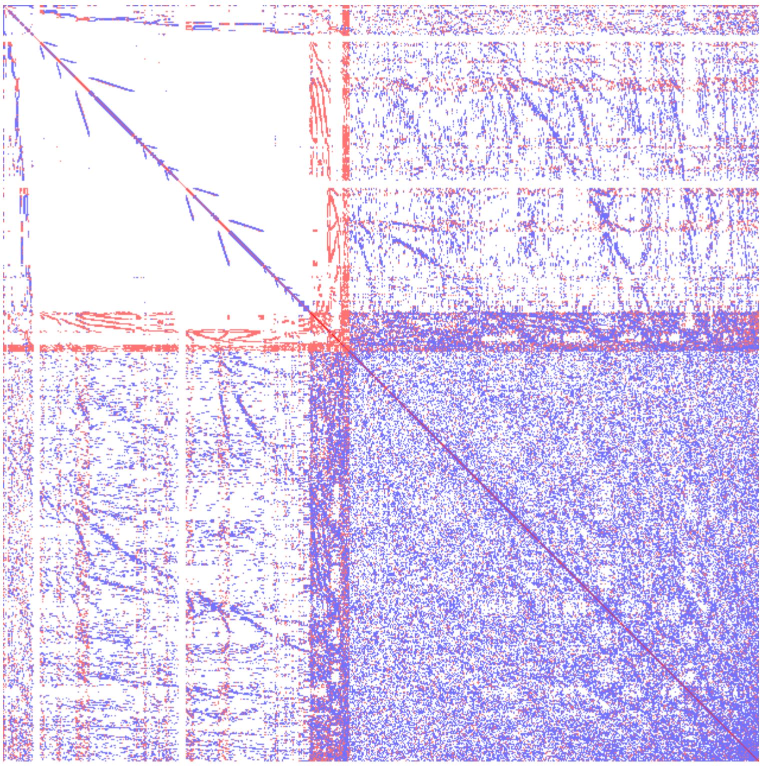


Figure 40: Problem stiffness matrix (120,378 × 120,378). Blue (red) dots represent positive (negative) values.

C.6 Results

Displacement		
Maximum value	$\lvert(u, v, w)\rvert_{\text{max}} =$	0.093 07 mm
Location	$x =$	61.3 mm
	$y =$	52.6 mm
	$z =$	69.9 mm

Von Mises stresses		
Material yield strength	$\sigma_{\text{yield}} =$	221 MPa
Maximum Von Mises stress	$\sigma_{\text{max}} =$	238.1 MPa
Location	$x =$	28 mm
	$y =$	13.2 mm
	$z =$	10.3 mm
Mean value	$\sigma_{\text{mean}} =$	25.64 MPa
Peak factor	$f_p =$	9.29

Maximum load level with respect to material yield

100 % (238/221)

Mean load level with respect to material yield

12 % (26/221)

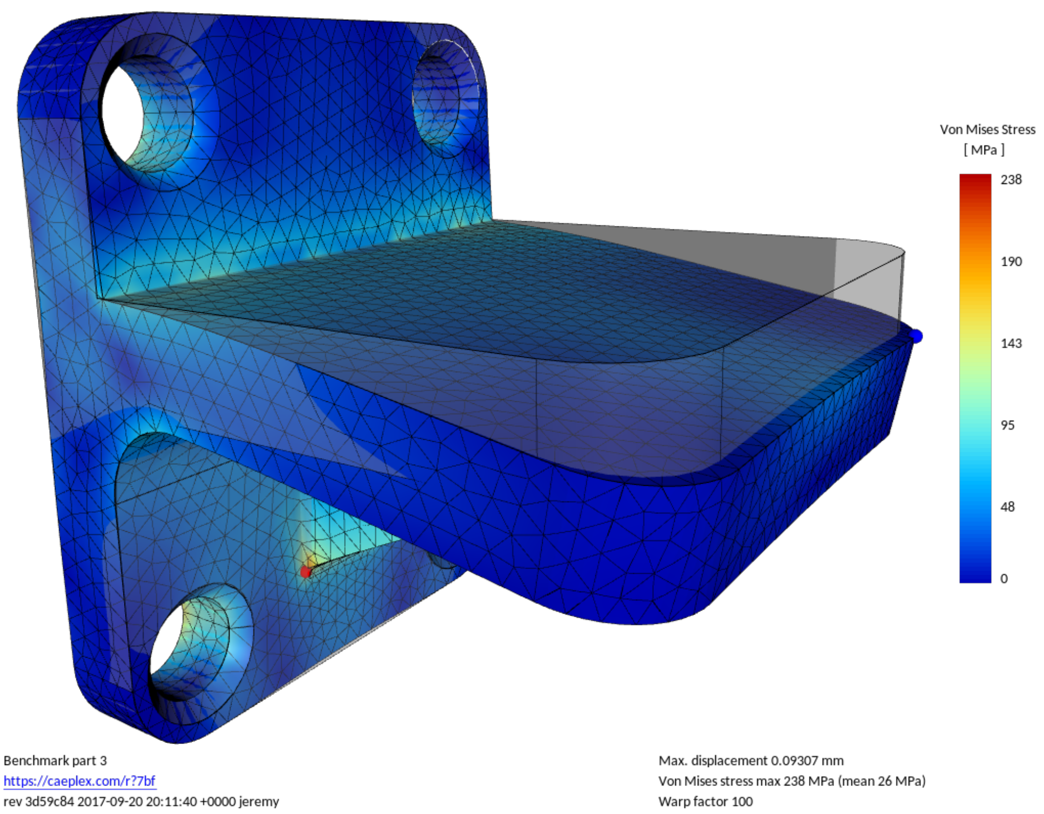


Figure 41: Von mises stresses and warped displacements with respect to the original geometry

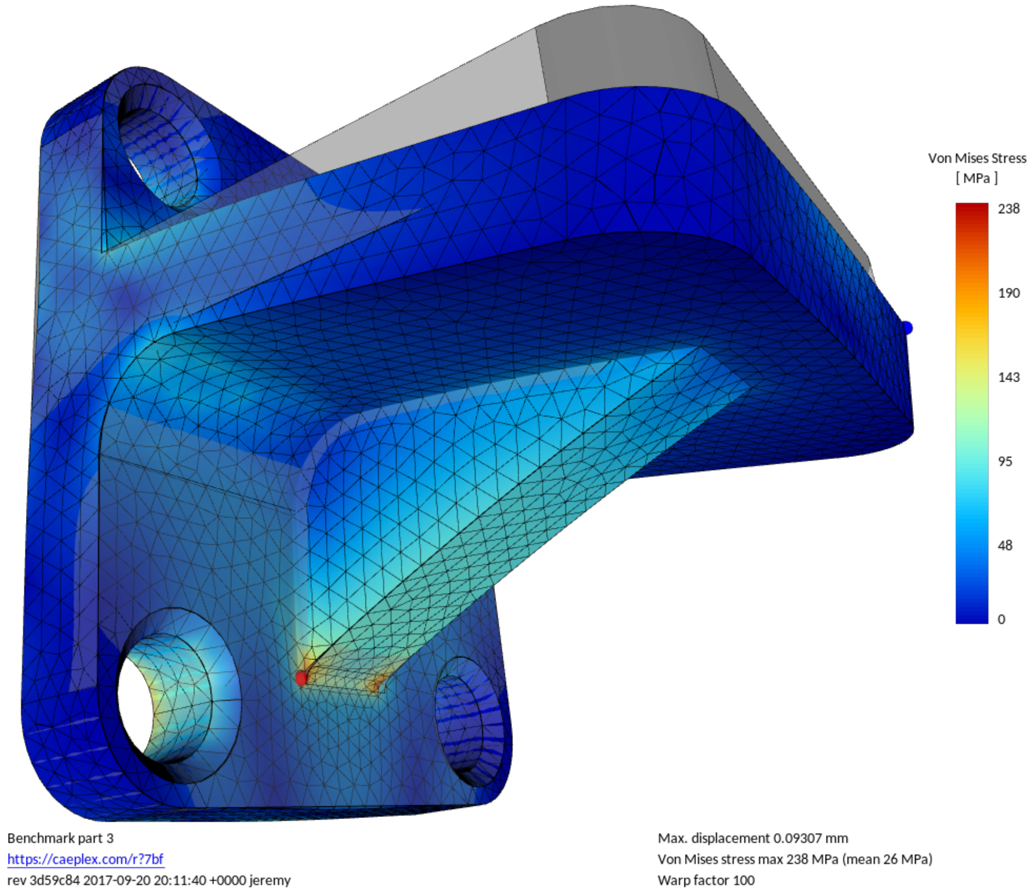


Figure 42: Von mises stresses and warped displacements with respect to the original geometry

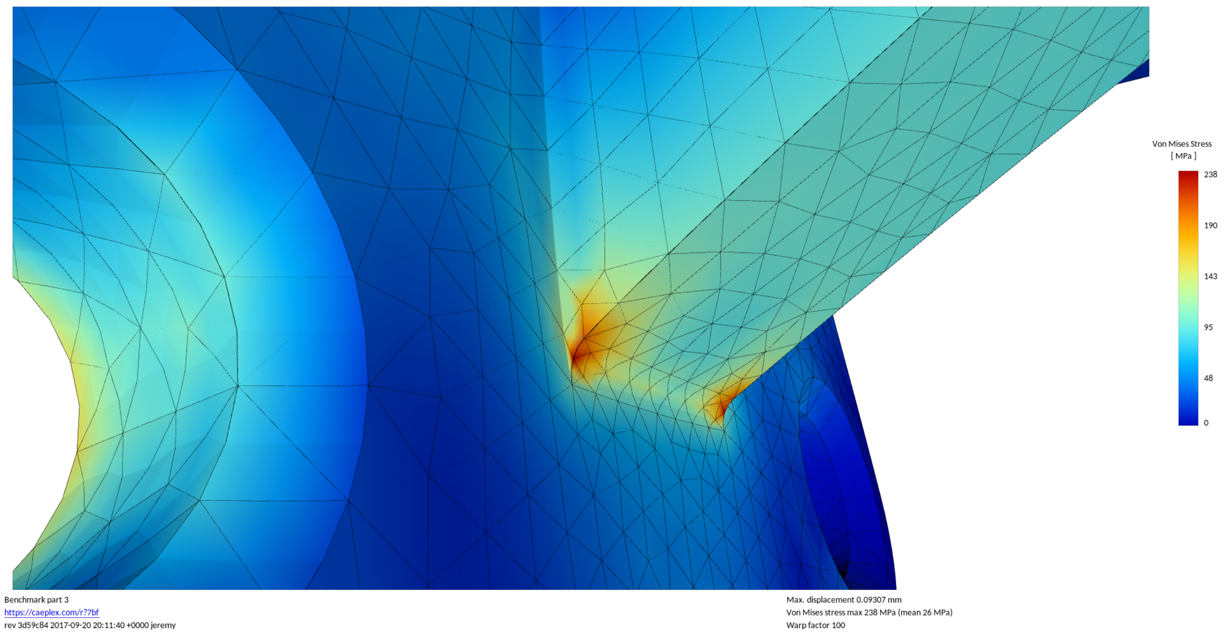


Figure 43: Detailed view of the maximum stress location

D Reference report

The reference report OT464-2017 was prepared by the department of Diseño y Simulación (translated as Design and Simulation)[8] at Dirección General de Asistencia Técnica (translated as General Direction of Technical Assistance), which is an official agency of the government of the Santa Fe province in Argentina. This work was done specifically for validating the code Fino against a commercial code. The report authors are

- Mauro Zocco
- Ignacio Ferracuti
- Gerardo Bellotti

Unfortunately, the report is only available in Spanish, and it has some parts that may be either

1. improved, or
2. inaccurate, or
3. wrong.

The master report SP-FI-17-BM-9AD is released under the terms of the Creative Commons Attribution-NoDerivatives 4.0 International license, and so this appendix also inherits the condition that no derivative work shall be obtained from this report. Actually, this condition was imposed on the reference report OT464-2017, so the master report needed to be released under a compatible license.

We would like to thank Gerardo Bellotti for making the arrangements for performing this study free of charge for Seamplex.

[8]Read `https://www.seamplex.com/blog/say-modeling-not-simulation.html` for a glimpse of what we think about the term "simulation" applied to solving deterministic equations at Seamplex.

PROVINCIA DE SANTA FE
Ministerio de la Producción

Secretaría de Industria, Agregado de Valor e Innovación

Dirección General de Asistencia Técnica

F 9.5.10 – 04
Revisión: 3

INFORME 1 DE 1 DE LA O.T. Nº 464/2017

Cliente: German Guillermo Theler

Domicilio: Lasserre 966 — Hoja 1 de 11

Localidad: Rafaela — C.P. 2300

Teléfono 03492-15303068

Fax: — Areas intervinientes: Diseño y Simulación

Email: gtheler@seamplex.com

Rosario, 12 de Septiembre de 2017

ASISTENCIA TECNICA SOLICITADA:

Calidad	
Costos	
Formulación de proyectos	
Registro de Propiedad Industrial	
Capacitación	
Asistencia en planta	
Desarrollo o mejora de producto	
Diseño	
Otros:……………………………………..	

INFORME DE RESULTADOS:

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

INFORME 1 DE 1 DE LA O.T. N° 464/2017 Hoja 2 de 11

1. OBJETIVO

El presente trabajo tiene como objetivo simular 3 piezas independientes entre sí para verificar su comportamiento bajo las cargas sometidas.

2. MODELADO Y MATERIAL

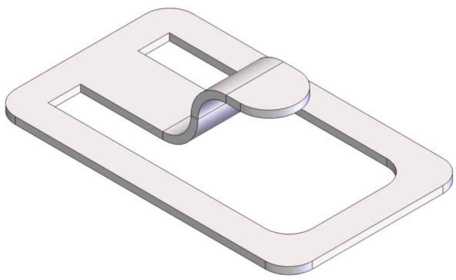


Fig. 1a: Pieza 1

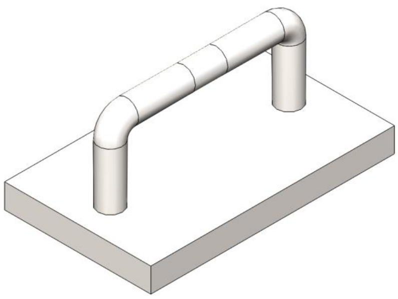


Fig. 1b: Pieza 2

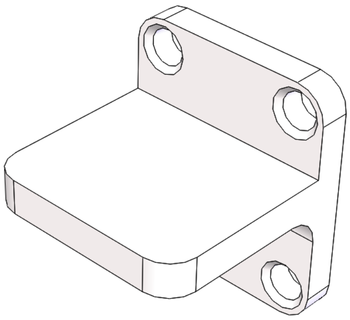
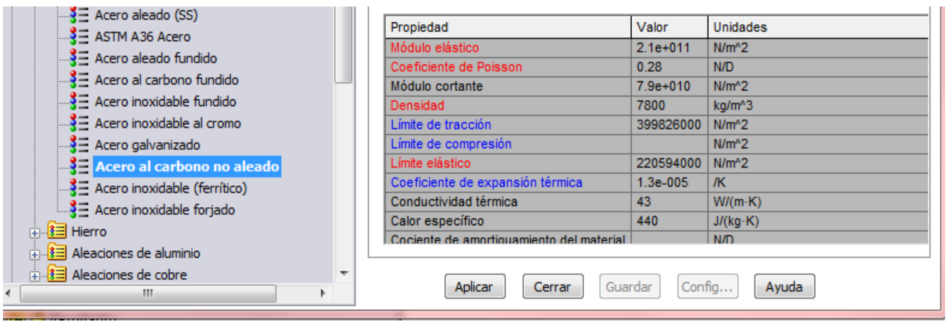


Fig. 1c: Pieza 3

El material para las 3 piezas es Acero al carbono no aleado y sus características son las siguientes:



Propiedad	Valor	Unidades
Módulo elástico	2.1e+011	N/m^2
Coeficiente de Poisson	0.28	N/D
Módulo cortante	7.9e+010	N/m^2
Densidad	7800	kg/m^3
Límite de tracción	399826000	N/m^2
Límite de compresión		N/m^2
Límite elástico	220594000	N/m^2
Coeficiente de expansión térmica	1.3e-005	/K
Conductividad térmica	43	W/(m·K)
Calor específico	440	J/(kg·K)
Cociente de amortiguamiento del material		N/D

Fig. 1d: Detalles del material

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

3. MALLADO

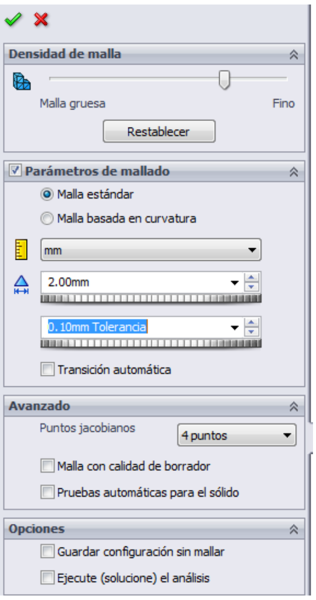
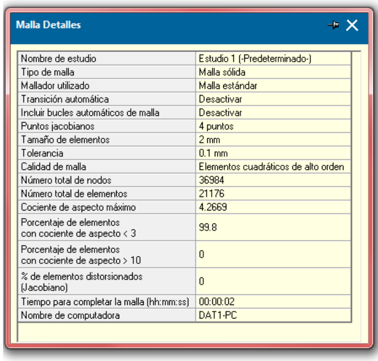


Fig. 2a: Detalles mallado Pieza 1



Malla Detalles

Nombre de estudio	Estudio 1 (-Predeterminado-)
Tipo de malla	Malla sólida
Mallador utilizado	Malla estándar
Transición automática	Desactivar
Incluir bucles automáticos de malla	Desactivar
Puntos jacobianos	4 puntos
Tamaño de elementos	2 mm
Tolerancia	0.1 mm
Calidad de malla	Elementos cuadráticos de alto orden
Número total de nodos	36984
Número total de elementos	21176
Cociente de aspecto máximo	4.2669
Porcentaje de elementos con cociente de aspecto < 3	99.8
Porcentaje de elementos con cociente de aspecto > 10	0
% de elementos distorsionados (Jacobiano)	0
Tiempo para completar la malla (hh:mm:ss)	00:00:02
Nombre de computadora	DAT1-PC

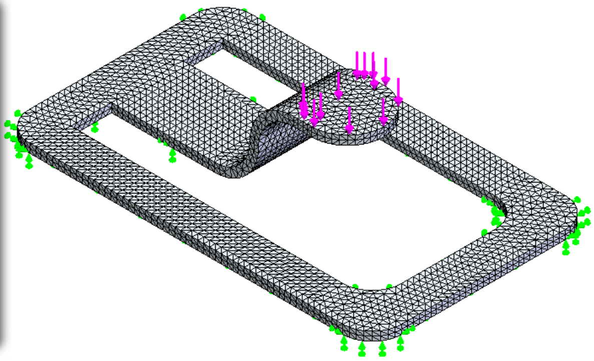


Fig. 2b: Detalles mallado Pieza 1

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

INFORME 1 DE 1 DE LA O.T. N° 464/2017 Hoja 4 de 11

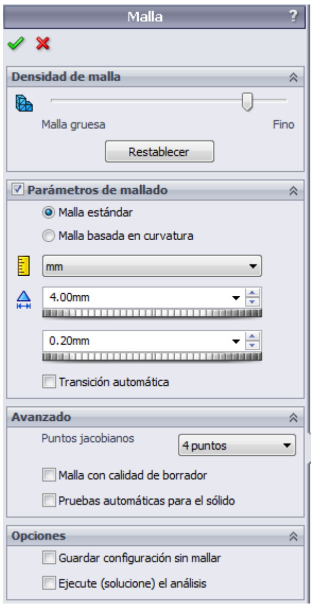
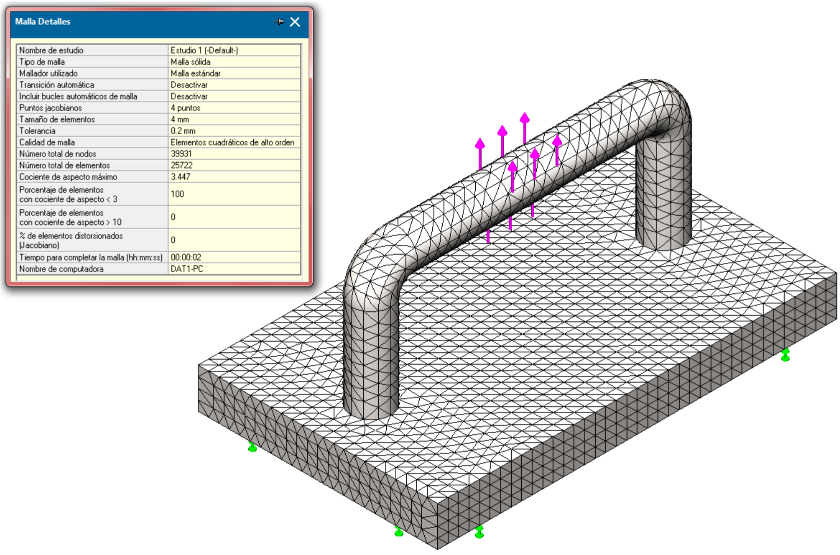


Fig. 2c: Detalles mallado Pieza 2



Nombre de estudio	Estudio 1 (-Default-)
Tipo de malla	Malla sólida
Mallador utilizado	Malla estándar
Transición automática	Desactivar
Incluir bucles automáticos de malla	Desactivar
Puntos jacobianos	4 puntos
Tamaño de elementos	4 mm
Tolerancia	0.2 mm
Calidad de malla	Elementos cuadráticos de alto orden
Número total de nodos	39931
Número total de elementos	25722
Cociente de aspecto máximo	3.447
Porcentaje de elementos con cociente de aspecto < 3	100
Porcentaje de elementos con cociente de aspecto > 10	0
% de elementos distorsionados (Jacobiano)	0
Tiempo para completar la malla (hh:mm:ss)	00:00:02
Nombre de computadora	DAT1-PC

Fig. 2d: Detalles mallado Pieza 2

Condiciones generales: El resultado de los ensayos corresponde en forma exclusiva a la muestra recibida en esta Dirección, salvo que se indique expresamente lo contrario. La DAT se exime de toda responsabilidad por el uso indebido de este informe. El presente informe puede ser reproducido solamente en su totalidad y sin introducir cambios.

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

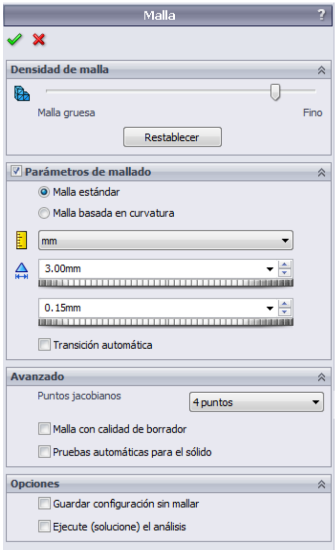
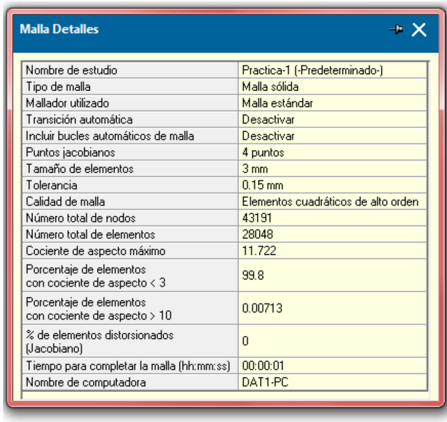


Fig. 2e: Detalles mallado Pieza 3



Malla Detalles

Nombre de estudio	Practica-1 (-Predeterminado-)
Tipo de malla	Malla sólida
Mallador utilizado	Malla estándar
Transición automática	Desactivar
Incluir bucles automáticos de malla	Desactivar
Puntos jacobianos	4 puntos
Tamaño de elementos	3 mm
Tolerancia	0.15 mm
Calidad de malla	Elementos cuadráticos de alto orden
Número total de nodos	43191
Número total de elementos	28048
Cociente de aspecto máximo	11.722
Porcentaje de elementos con cociente de aspecto < 3	99.8
Porcentaje de elementos con cociente de aspecto > 10	0.00713
% de elementos distorsionados (Jacobiano)	0
Tiempo para completar la malla (hh:mm:ss)	00:00:01
Nombre de computadora	DAT1-PC

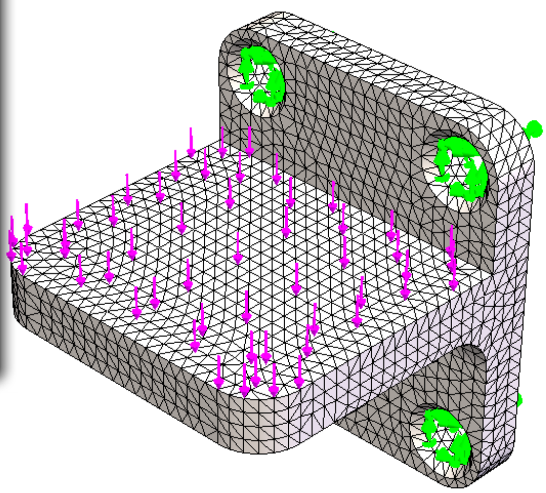


Fig. 2f: Detalles mallado Pieza 3

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

4. SUJECIÓN

Se ha definido para las 3 piezas sujeción de *geometría fija* (ver figuras correspondientes), la cual restringe los 6 grados de libertad de los nodos que se encuentran en las superficies que integran la sujeción.

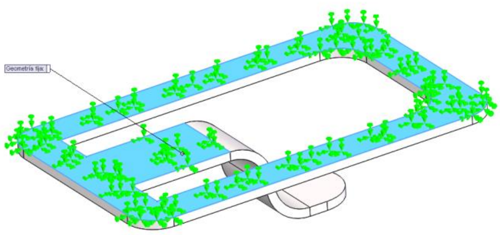


Fig. 3a: Sujeción geometría fija Pieza 1

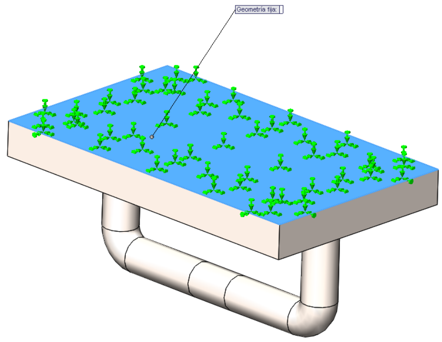


Fig. 3b: Sujeción geometría fija Pieza 2

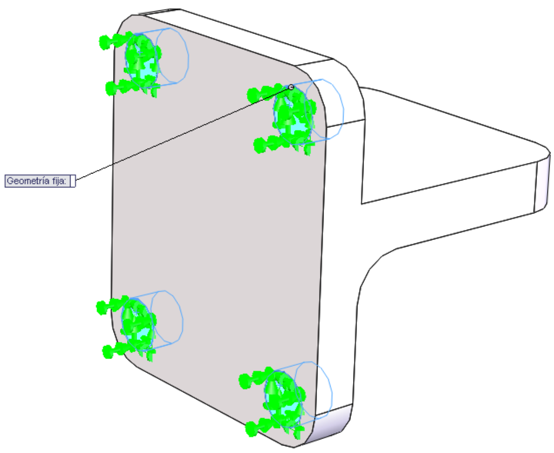


Fig. 3c: Sujeción geometría fija Pieza 3

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

5. CARGAS

Se ha establecido un estado de carga para cada pieza (ver figuras correspondientes)

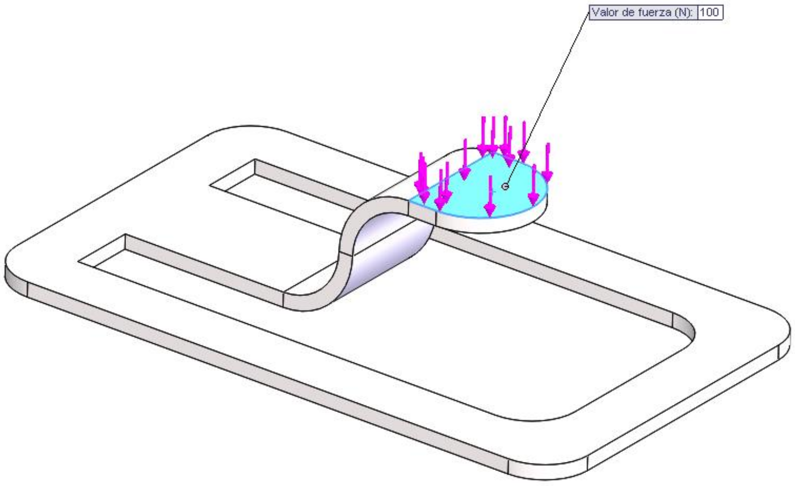


Fig. 4a: Valor de la fuerza = 15000 N

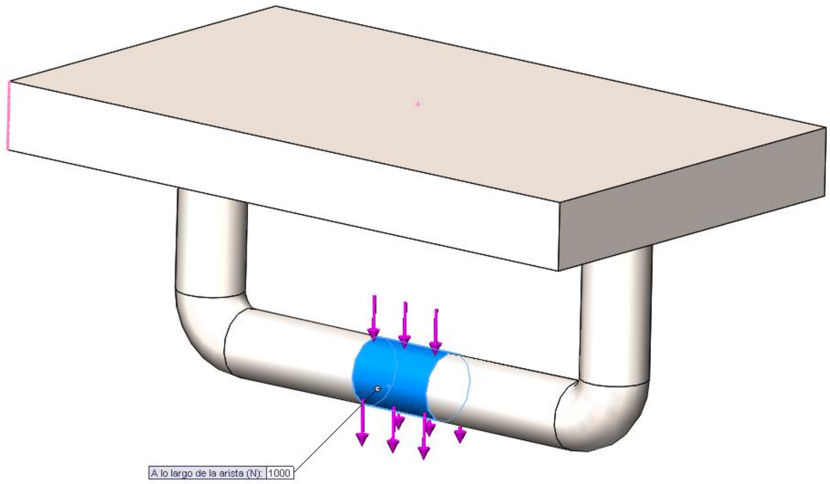


Fig. 4b: Valor de la fuerza = 100 N

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

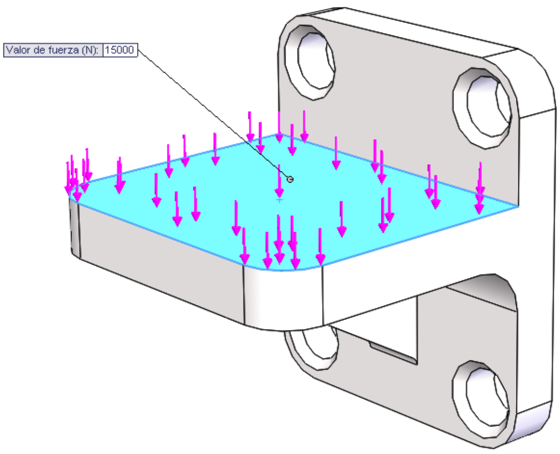


Fig. 4c: Valor de la fuerza = 1000 N

6. RESULTADOS

Resultados de Simulación:

A continuación se muestran distintos trazados para cada pieza (ver figuras correspondientes)

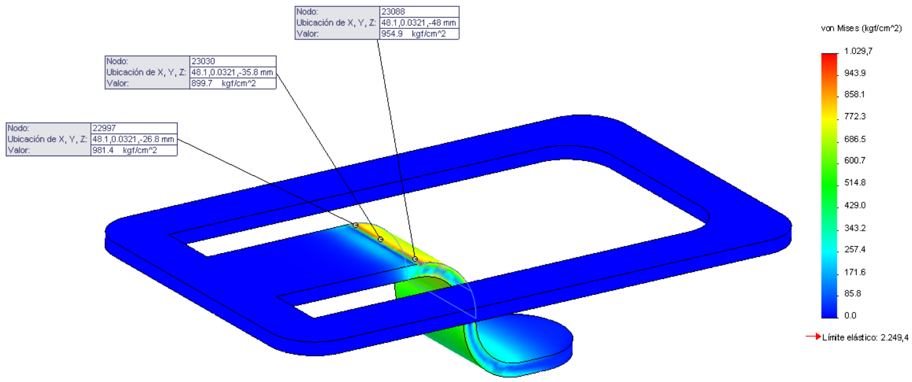


Fig. 5a: Trazado tensión de Von mises dando un valores máximos iguales a :

981.4 Kg/cm^2 – 899.7 Kg/cm^2 – 954.9 Kg/cm^2

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

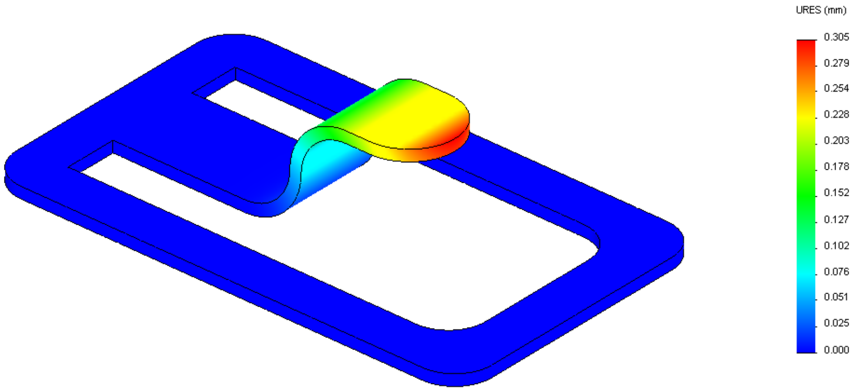


Fig. 5b: Trazado desplazamiento resultante dando un valor máximo de 0.305mm

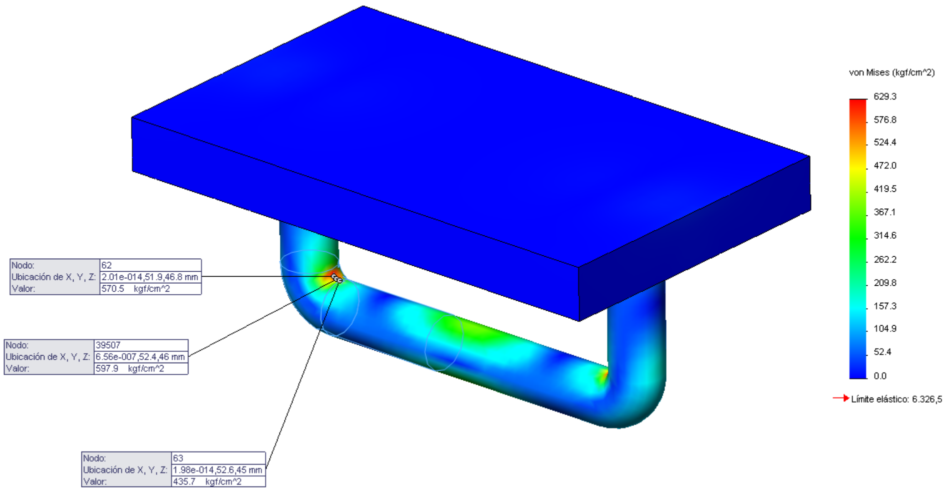


Fig. 5c: Trazado tensión de Von mises dando un valores máximos iguales a:

570.5 Kg/cm^2 – 597.9 Kg/cm^2 – 435.7 Kg/cm^2

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

INFORME 1 DE 1 DE LA O.T. N° 464/2017 Hoja 10 de 11

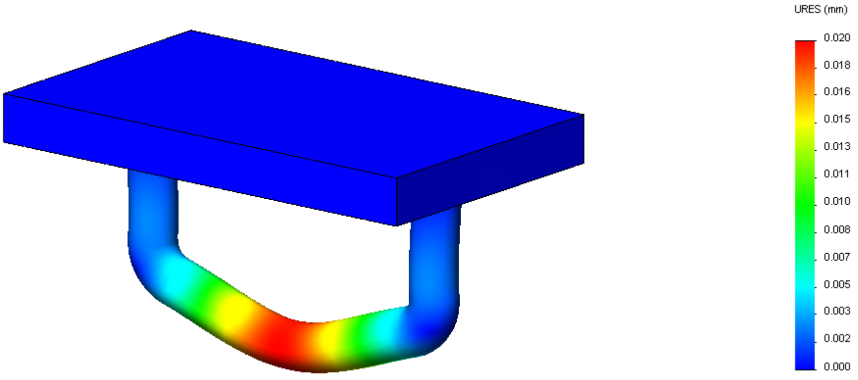


Fig. 5d: Trazado desplazamiento resultante dando un valor máximo de 0.020mm

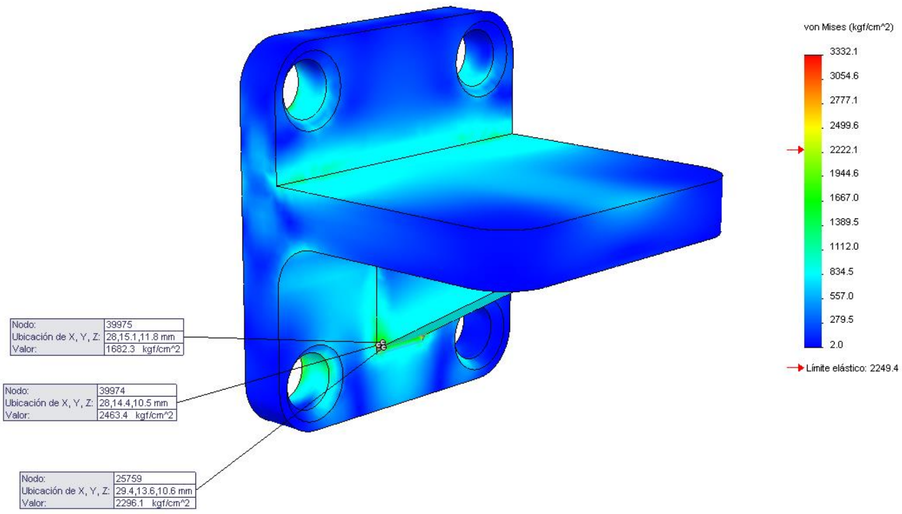


Fig. 5e: Trazado tensión de Von mises dando un valores máximos iguales a:

1682.6 Kg/cm^2 – 2463.4 Kg/cm^2 – 2296.1 Kg/cm^2

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar

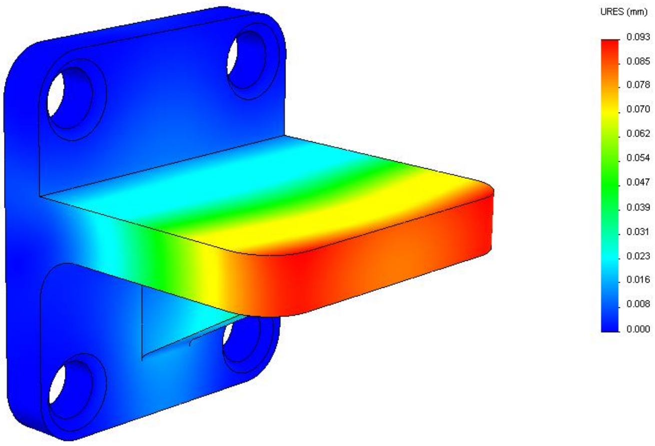


Fig. 5f: Trazado desplazamiento resultante dando un valor máximo de 0.093mm

Se concluyó el trabajo	X
La orden sigue abierta	

OBSERVACIONES:

El software utilizado fue Solidworks Simulation Premium 2013, licencia nº 9010 2084 7923 7M44RCDF

Técnico Mauro Zocco Diseño y Simulación	Ing. Ignacio Ferracuti Diseño y Simulación	Ing. Gerardo Bellotti Diseño y Simulación

Esmeralda y Ocampo - 2000 - Rosario - Prov. Santa Fe - República Argentina
Tel.Fax: ++54 (0)341- 4851286 – 4851319 - 4728130 - 4728134 - e mail dat@dat.gov.ar ; www.dat.gov.ar