

**Test Suite for the
CAE Implementor Forum
Round 2S**

December – June 2018

Release 1.1

March 20, 2018

Contacts

Jochen Boy
PROSTEP AG
Dolivostraße 11
64293 Darmstadt / Germany
jochen.boy@prostep.com

Jean-Marc Crepel
AFNeT
30, Rue de Miromesnil
75008 Paris / France
jean-marc.crepel@afnet.fr

Phil Rosché
ACCR, LLC.
125 King Charles Circle
Summerville, SC 29485 USA
phil.rosche@accr-llc.com

Table of Contents

1	Introduction.....	3
1.1	Functionality tested in this round	3
1.2	General testing instructions for this round.....	6
1.3	Testing Schedule.....	7
1.4	Copyrights on Test Cases	8
2	Synthetic Test Case Specifications	8
2.1	Test Case ATS1: beam FEA model using rod elements	8
2.2	Test Case ATS2: beam FEA model using bar elements	11
2.3	Test Case ATS3: beam FEA model using shell elements	14
2.4	Test Case ATS4: beam FEA model using solid elements	18
Annex A	General Considerations about CAE	22
Annex B	Information related to Statistics calculation	23
Annex C	Calculation of the resultants of applied loads (forces / moments) ...	25
Annex D	Calculation of strains and stresses statistics	28
Annex E	1D element and 2D element coordinate system and internal element forces orientation	31

List of Figures

Figure 1:	CAE-IF Round 2S Schedule	7
Figure 2:	ATS1 model overview.....	9
Figure 3:	ATS2 model overview.....	12
Figure 4:	ATS3 model overview.....	16
Figure 5:	ATS4 model overview.....	19
Figure 6:	Orientation for 1D element internal forces	31
Figure 7:	Local coordinate system for 2D elements	32
Figure 8:	Orientation for 2D element internal forces	32

Document History

Release	Date	Change
0.1	February 27, 2018	First Draft Release
0.2	March 7, 2018	Change R2S schedule and some edits Correct AP 209 schema to be used Add annex E
1.0	March 9, 2018	1st released on CAx-IF Joint Testing Information page
1.1	March 20, 2018	Planning and confcall scheduled updated

1 Introduction

This document describes the suite of test cases to be used for the second round of testing of the CAE Implementor Forum (CAE-IF). The CAE-IF is a workgroup within the framework of the CAx Implementor Forum (CAx-IF), a joint testing forum, organized and facilitated by AFNeT, PDES, Inc., and the prostep ivip Association. The test rounds of the CAE-IF concentrate primarily on testing the interoperability and compliance of STEP processors based on AP 209 ed2.

The test rounds in general combine testing of synthetic and production models. Production models will in most cases be provided by the member companies of the organizations AFNeT, PDES, Inc., and prostep ivip Association. When production models are not available from the member companies, “production-like” models will be solicited from the various CAE-IF participants.

As a first step, the test rounds are limited to 3D Linear Static Structural FEA models, using simple components. More complex models, including production models, will be introduced later.

This test suite includes synthetic models for testing the following capabilities: export /import Input and Output FEA models.

1.1 Functionality tested in this round

Functionality tested in this round relates mainly to FEA output data, with some additional topics related to FEA input data. Regarding output data, the load cases (states) to be considered are indicated in each test case description when necessary.

- **Input data : fixed degrees of freedom (DOF)**

In order to define constraints or boundary conditions in the finite element analysis (FEA), the input data deck may contain specific instructions that applies zero value for one or more displacement/rotation DOF at one or more nodes. This statistic will count the number of DOF explicitly set to zero in the STEP file.

Note: Some CAE solver can assume implicitly that other DOF are fixed, but those are NOT included in the above “fixed DOF” count.

- **Input data : resultant moment of applied loads**

Following the previous test round (R1S), it was stated that the calculation of the “resultant moment of applied loads at the origin” may give very high values depending of the position of the model relatively to the origin. Therefore the definition of the “resultant moment of applied loads” has been extended so that this resultant moment can be calculated at any specific point (see Annex C). This point should be specified in order to keep the resultant value in a reasonable range and is defined in the test case description when required.

- **Output data : nodal displacements and rotations**

For a 3D FEA structural analysis, the primary results (see Annex A) in the general case consist in nodal displacements and rotations (6 degrees of freedom - 6DOF).

For processing the nodal results into the standard statistics available in CAESAR, these nodal results will be split into two 3DOF vectors: Displacement (3DOF vector) and Ro-

tation (3DOF vector) at each selected node. The selected nodes for statistics comparison are indicated in each test case when necessary. The components of these vectors shall be calculated in the basic coordinate system.

The vector length of such vectors is also a significant value that can be calculated as $\text{SQRT}(X^2 + Y^2 + Z^2)$, where (X, Y, Z) are the 3 components of the vector. The “Maximum Nodal Displacement” (resp. “Maximum Nodal Rotation”) statistic is defined as the maximum value of the displacement vector length (resp. rotation vector length) over all nodes. As the vector length has always positive values, there is no need for evaluating minimum displacement or rotation.

- **Output data : strain field**

One of the results generated by a 3D FEA Structural Analysis is the strain field, which is defined as a second order symmetrical tensor in 3D space. A scalar value derived from the full tensor can be also considered: the volume change ratio, which is a significant value for engineering purpose. The volume change ratio can be either recovered from the results data set when available or calculated from the tensor components (see annex D), depending of the CAE software used and the output requests.

In order to evaluate the translation of the strain field with AP 209, the volume change ratio in a given element will be evaluated **at the center point of the element** only. The selected elements for statistics comparison are indicated in each test case instruction when necessary.

The “Maximum volume change ratio” statistic is defined as the maximum value of the volume change ratio over all elements. As the volume change ratio is a signed value, the “Minimum volume change ratio” statistic is defined in a similar way.

- **Output data : stress field**

One of the results generated by a 3D FEA Structural Analysis is the stress field, which is defined as a second order symmetrical tensor in 3D space. A scalar value derived from the full tensor can be also considered: the Von-Mises stress criteria, which is a significant value for engineering purpose. The Von-Mises stress can be either recovered from the results data set when available or calculated from the tensor components (see annex D), depending of the CAE software used and the output requests.

In order to evaluate the translation of the stress field with AP 209, the Von-Mises stress in a given element will be evaluated **at the center point of the element** only. The selected elements for statistics comparison are indicated in each test case when necessary.

The “Maximum Von-Mises stress” statistic is defined as the maximum value of the Von-Mises stress over all elements. As the Von-Mises stress has always positive values, there is no need for evaluating minimum Von-Mises stress.

- **Output data : internal element forces for 1D elements**

When using 1D structural elements (i.e. curve_3D_elements) in a 3D model, the stresses can be integrated over the cross section to give the element internal forces, which are significant values for engineering purpose. The internal forces for 1D elements is a set of 6 signed scalar values (components), defined relatively to the element coordinate system (x, y, z) at any cross section of the element (see annex E):

- Axial
- Shear 1 (xy plane)
- Shear 2 (xz plane)

- Torsion
- Bending moment 1 (xy plane)
- Bending moment 2 (xz plane)

The internal element forces shall be recovered from the results data set.

In order to evaluate the translation of the internal element forces using AP 209, the corresponding values in a given element will be evaluated **at the center point of the element** only. Therefore, the maximum value of any of the components is defined as the maximum value of this component over all 1D elements. The minimum value of any of the components is defined in a similar way.

Note: If case the value of one of the component at the center point of the element is not available, it will be assumed to be the average of corresponding values at end nodes of the element.

- **Output data : internal element forces for 2D elements**

When using 2D structural elements (i.e. surface_3D_elements) in a 3D model, the stresses can be integrated over the thickness to give the element internal forces, which are significant values for engineering purpose. The internal forces for 2D elements is a set of 8 signed scalar values (components), defined relatively to the element coordinate system (x, y, z) at any cross section of the element (see annex E):

- Membrane x
- Membrane y
- Membrane xy
- Shear x
- Shear y
- Bending moment x
- Bending moment y
- Bending moment xy

The internal element forces shall be recovered from the results data set.

In order to evaluate the translation of the internal element forces using AP 209, the corresponding values in a given element will be evaluated **at the center point of the element** only. Therefore, the maximum value of any of the components is defined as the maximum value of this component over all 2D elements. The minimum value of any of the components is defined in a similar way.

Note: If case the value of one of the component at the center point of the element is not available, it will be approximated as the average of corresponding values at available locations in the element (corner nodes, Gauss points, ...).

- **Output data : Boundary condition (BC) constraint efforts**

For each constraint or boundary condition defined at a specific point, the CAE solver can calculate the equivalent effort applied at this point. The equivalent effort has 6 components which can be split into two 3DOF vectors: BC forces and BC moments.

In order to evaluate the translation of the BC forces and BC moments using AP 209, the summation of each vector component will be calculated over all nodes.

The BC forces and BC moments shall be recovered from the results data set. The components of these vectors shall be calculated in the basic coordinate system.

Note that this summation for the BC moment is not a resultant calculation. There is no consideration of the location of where the BC forces are applied.

- **Output data : Grid point force balance**

When requested in the output commands, the Grid Point Force Balance table is computed for a given loadcase, at selected nodes. The table consists in a list of all the efforts contributing to the equilibrium of the selected node, coming from the adjacent elements, applied loads and boundary conditions. Each contributing effort has 6 components which can be split into two 3DOF vectors: grid point forces and grid point moments.

In order to evaluate the translation using AP 209, the grid point force and grid point moment vectors will be evaluated for statistics comparison at the selected node and contributing element as indicated in each test case.

The grid point forces and grid point moments shall be recovered from the results data set. The components of these vectors shall be calculated in the basic coordinate system.

1.2 General testing instructions for this round

The general procedures for communication of models and statistics are outlined in a separate document, named 'General Testing Instructions'. The document can be retrieved from the CAX Implementor Forum web sites. The latest version is v1.13, dated September 29, 2017.

Key issues should be reported using the BRUTUS system (see 'General Testing Instructions').

Native FEA Model use MSC-NASTRAN card descriptions. Documentation of MSC-NASTRAN input syntax is available in the "NASTRAN quick reference guide", which can be download from the MSC website at:

- https://simcompanion.mscsoftware.com/infocenter/index?page=content&id=DOC11146&cat=MSC_NASTRAN_DOCUMENTATION_2017&actp=LIST

Native FEA model of each test case is available in the xx.bdf file.

Output data set are generated by MSC-NASTRAN software. NASTRAN can produce results output in several formats for pre/post processors to read:

- xx.pch : Nastran ascii results file referred to as a formatted punch deck
- xx.op2 : Nastran binary results file (sequential) used by many post processors.
- xx.xdb : Nastran binary results file (direct access) formatted for use by Patran but readable by many post processors.

The ASCII punch file (xx.pch) is the simplest format and is human readable. As R2S is limited to simple test cases, it is recommended to use the xx.pch file for extracting results.

The format is documented in the "NASTRAN quick reference guide" in the 'ITEM CODES' section.

Other output files produced by NASTRAN are for information only, and don't need to be processed in R2S:

- xx.f06 :Nastran solution formatted information file
- xx.f04 :Nastran solution execution summary information file

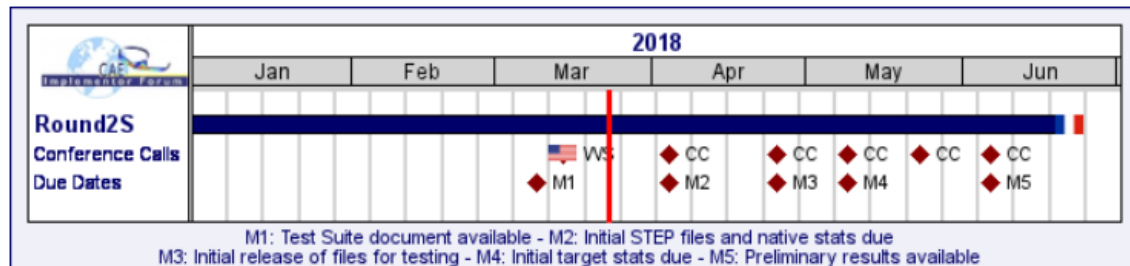
Additional documents on MSC-NASTRAN software are available at:

- https://simcompanion.mscsoftware.com/infocenter/index?page=content&cat=MSC_MD_NASTRAN_DOCUMENTATION&channel=DOCUMENTATION

Note: CAE models are generally “unit consistent models”. This means there is no need for data conversion. Participants are requested to keep the original unit system unchanged and to deliver results and statistics using the same unit system as the native models. Nevertheless, **the units’ definition shall be included in the STEP file**, as stated in the recommended practices.

1.3 Testing Schedule

The following schedule has been agreed on for Round 2S:



Date	Action
2018-03-09 (Fri)	Test Suite document available
2018-03-14 (Wed)	R2S Kickoff meeting in Asheville, NC, USA
2018-04-04 (Wed)	Initial STEP files and native stats due / CAE-IF Round2S Conference Call
2018-04-25 (Wed)	Initial release of files for testing / CAE-IF Round2S Conference Call
2018-05-09 (Wed)	CAE-IF Round2S Conference Call / Initial target stats due
2018-05-23 (Wed)	CAE-IF Round2S Conference Call
2018-06-06 (Wed)	Preliminary results available / CAE-IF Round2S Conference Call
2018-06-19 (Tue) - 2018-06-21 (Thu)	CAE-IF R2S review meeting in Toulouse, France

Figure 1: CAE-IF Round 2S Schedule

The CAE-IF Round 2S Kickoff and Technical Workshop will be held in conjunction with the PDES, Inc. Spring Offsite meeting and a LOTAR workshop. Conference calls and web sessions will also be available for those not attending the meeting to dial in.

As CAE-IF is still a recently launched activity, conference calls will be proposed to participants every other weeks to address any issue and to secure a convenient progress.

The CAE-IF Round2S Review meeting will take place in conjunction with a LOTAR/ AFNeT Offsite meeting. In addition, conference calls and web sessions will be available.

1.4 Copyrights on Test Cases

1.4.1 CAE-IF

None of the production test cases which were provided by the AFNeT, PDES, Inc. and prostep ivip member companies may be publicly released for any purpose. The test cases can be freely distributed among the CAE-IF members, and can be used for any purposes that are related to CAE-IF testing (i.e. testing, documentation of testing efforts, etc.), as long as a reference to the originating company is made.

The test cases must not be used for any purposes other than CAE-IF testing or outside of AFNeT, PDES, Inc. and prostep ivip. Test cases provided by the LOTAR project for testing of specific capabilities are applicable to the same restrictions and may not be used outside LOTAR or the CAE-IF.

2 Synthetic Test Case Specifications

2.1 Test Case *ATS1: beam FEA model using rod elements*

All information about this test case can also be viewed in CAESAR on its Information page.

2.1.1 Motivation

As Round 2S is the second opportunity for participants to run formal Implementor Forum testing using AP 209 ed2, the test suite will re-use the simple finite element model components from R1S, with the scope extended to output FEA data as well.

The ATS1 test represents a cantilever beam using “rod” elements only (1D element family), with one applied lumped force. This test will mainly focus on output results such as nodal displacement/rotation, BC constraint forces, element internal forces, strain and stress field, grid point force balance, with some statistics related to input data being reprocessed with new instructions as indicated in section 1.1.

2.1.2 Approach

The approach to be used is described in the latest version of the following documents:

- “Recommended Practices for AP 209 ed2” (at least version 2.0 dated March 30, 2016)
- “AP 209 ed2 Linear Static Structural FEA Handbook vol.1” - (version 1.8.2 or later)

These documents can be found in the CAx-IF website under “Joint Testing Information” and in the CAx-IF member area under “CAE Domain - Information on Round2S of Testing”.

Within the ATS1 model, the following functionalities are in the scope of Round 2S:

- Export/import input data of FEA models with rod elements, including lumped forces.
- Export/import output data of FEA with :
 - Nodal displacement and rotation
 - BC constraint forces
 - Element internal forces
 - Strain tensor field
 - Stress tensor field
 - Grid point force balance

The AP 209 schema to be used is a corrected version of the AP 209 ed2 schema, which can be found on the CAE-IF member area on the CAx-IF homepages, under “Information on Round 1S of Testing”.

2.1.3 Testing Instructions

The tests will be performed based on the ATS1m5 NASTRAN model described below. This model has been developed by the LOTAR EAS Working Group, and has been checked during previous pilot studies.

2.1.3.1 Test Model Overview

The ATS1m5 model represents a beam (rectangular prism) idealized using “rod” elements (axial stiffness element, no torsional stiffness), with the following characteristics:

- Isotropic material property
- 1000 lbf axial load in compressive (-x) direction
- Rectangular coordinate system at origin with model at [0, -2, 1]

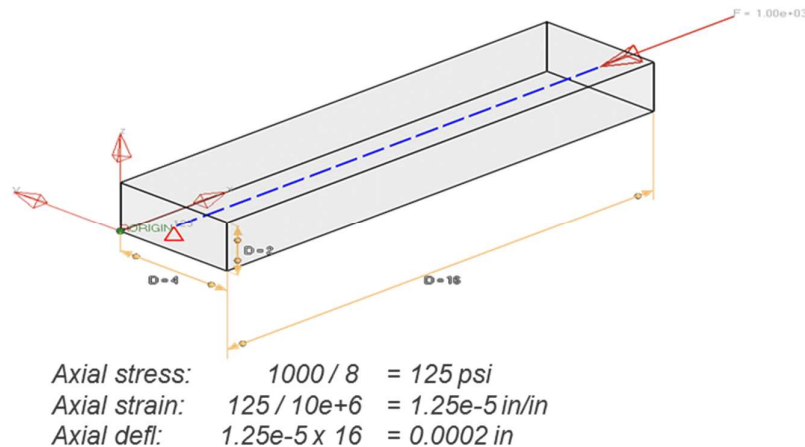


Figure 2: ATS1 model overview

The following output results are available: nodal displacement/rotation, BC constraint forces, 1D element internal forces, strain and stress field, and grid point force balance.

2.1.3.2 Test Model Access

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under “General Information – File repository” in the folder “CAE / Round 2S”:

- ATS1m5.bdf (input data)
- ATS1m5.pch (output data)

2.1.3.3 Test Model Configuration

The following functionality shall be included in the test files provided for this round of testing, as far as it has been implemented by the CAE-IF participants and is described in the Recommended Practices:

- Input data
- Output data
- Validation Properties – As there is not yet formal validation properties defined related to CAE models, participants are not requested to include any validation properties. Nevertheless, participants will calculate key values as statistics (see below) which are potential candidates for future formal validation properties dedicated to CAE models.

2.1.4 Statistics

For each STEP file exported or imported for the ATS1 test case, vendors must submit the corresponding statistics. To do so, go to the [ATS1 Data Sheet], and either fill in the web form, or upload a comma-delimited file (.csv) with the data as listed below.

Native Statistics

When exporting a STEP file, report what data importing systems should expect to find. For numeric statistics, enter the respective value or 'na' if not supported. For other statistics, select either 'full support' (i.e. test case and Rec. Pracs. definitions are fulfilled), 'limited support' (meaning the implementation does not meet all criteria and issues may be expected on import), or 'na' if not supported.

Target Statistics

When importing a STEP file, report the results found after processing the file as described below. The value should be given in the unit implicitly defined in the native model.

Load Case Selection, element and node selection

When statistics depend on the load case, node selection or element selection, the item(s) to be considered is (are) indicated in the following table:

Statistic	Case 'A'	Case 'B'
Fixed DOF nb	Loadcase #1	--
Nodal displ.	Loadcase #1 Node #9	--
Max nodal displ.	Loadcase #1	--
Max strain	Loadcase #1	--
Max stress	Loadcase #1	--
BC force	Loadcase #1	--
Max/Min axial 1D_elt	Loadcase #1	--
Grid point force	Loadcase #1 Node #2 Element #2 contribution	--

Applicable Units and Coordinate system

The model is based on imperial units (in, lbf).

Components of any point (such as Center of Gravity) or vector (such as resultant of applied loads, displacements ...) should be calculated in the basic coordinate system. Components of the element internal forces will be identified according to the element coordinate system.

Screenshots (optional)

Note that CASEAR allows the addition of multiple screenshots per dataset.

Data Sheet Columns

column name	description
model	The name of the test model, here 'ATS1'
system_n	The system code of the CAE system creating the STEP file

system_t	The system code of the CAE system importing the STEP file. For native stats, select 'stp'
fixed_dof_nb_a	Total number of fixed Degrees of freedom (DOF) - case A
nodal_disp_ax	Components of nodal displacement - case 'A'
nodal_disp_ay	
nodal_disp_az	
max_nodal_disp_a	Vector length of Max Nodal displacement - case 'A'
max_strain_vol_a	Maximum Volume change (strain at center point) - Case A
max_stress_vm_a	Maximum Von Mises stress value (center point) - Case A
bc_force_ax	Summation of boundary condition constraint forces - case 'A'
bc_force_ay	
bc_force_az	
max_axial_f_1d_elt_a	Maximum axial internal force - 1D elt (center point) -case 'A'
min_axial_f_1d_elt_a	Minimum axial internal force - 1D elt (center point) -case 'A'
grid_point_force_ax	Grid point force -- case 'A'
grid_point_force_ay	
grid_point_force_az	
date	The date when the statistics were last updated (will be filled in automatically)
issues	A short statement on issues with the file

2.2 Test Case ATS2: beam FEA model using bar elements

All information about this test case can also be viewed in CAESAR on its Information page.

2.2.1 Motivation

As Round 2S is the second opportunity for participants to run formal Implementor Forum testing using AP 209 ed2, the test suite will re-use the simple finite element model components from R1S, with the scope extended to output FEA data as well.

The ATS2 test represents a cantilever beam using “bar” elements only (1D element family), with combinations of lumped applied loads. This test will mainly focus on output results such as nodal displacement/rotation, BC constraint forces, 1D element internal forces, grid point force balance, with some statistics related to input data being reprocessed with new instructions as indicated in section 1.1.

2.2.2 Approach

The approach to be used is described in the latest version of the following documents:

- “Recommended Practices for AP 209 ed2” (at least version 2.0 dated March 30, 2016)
- “AP 209 ed2 Linear Static Structural FEA Handbook vol.1” - (version 1.8.2 or later)

These documents can be found in the CAX-IF website under “Joint Testing Information” and in the CAX-IF member area under “CAE Domain - Information on Round2S of Testing”.

Within the CAE domain, the following functionalities are in scope of Round 2S:

- Export/import input data of FEA models with bar elements, including lumped forces.
- Export/import output data of FEA with :
 - Nodal displacement and rotation
 - BC constraint forces
 - Element internal forces

- Grid point force balance

The AP 209 schema to be used is a corrected version of the AP 209 ed2 schema, which can be found on the CAE-IF member area on the CAx-IF homepages, under “Information on Round 1S of Testing”.

2.2.3 Testing Instructions

The tests will be performed based on the ATS2m5 NASTRAN model described below. This model has been developed by the LOTAR EAS Working Group, and has been checked during previous pilot studies.

2.2.3.1 Test Model Overview

The ATS2m5 model represents a beam (rectangular prism) idealized using “bar” elements (axial and bending stiffness element, no torsional stiffness), with the following characteristics:

- Isotropic material property
- 3 load cases:
 1. axial load in compressive (-x) direction
 2. lateral distributed load in bending (-y) direction
 3. combination of 1 + 2

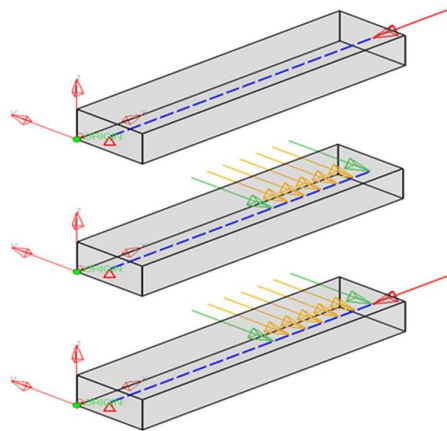


Figure 3: ATS2 model overview

The following output results are available: nodal displacement/rotation, BC constraint forces, 1D element internal forces, strain and stress field, and grid point force balance.

2.2.3.2 Test Model Access.

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under “General Information – File repository” in the folder “CAE / Round 2S”:

- ATS2m5.bdf (input data)
- ATS2m5.pch (output data)

2.2.3.3 Test Model Configuration

- See section 2.1.3.3 above.

2.2.4 Statistics

For each STEP file exported or imported for the ATS2 test case, vendors must submit the corresponding statistics to CAESAR. To do so, go to the [ATS2 Data Sheet], and either fill in the web form, or upload a comma-delimited file (.csv) with the data as listed below.

Native Statistics

When exporting a STEP file, report what data importing systems should expect to find. For numeric statistics, enter the respective value or 'na' if not supported. For other statistics, select either 'full support' (i.e. test case and Rec. Pracs. definitions are fulfilled), 'limited support' (meaning the implementation does not meet all criteria and issues may be expected on import), or 'na' if not supported.

Target Statistics

When importing a STEP file, report the results found after processing the file as described in the table below. The value should be given in the unit implicitly defined in the native model.

Load Case Selection, element and node selection

When statistics depend on the load case, node selection or element selection, the item(s) to be considered is indicated in the following table:

Statistic	Case 'A'	Case 'B'
Fixed DOF nb	Loadcase #1	Loadcase #3
Applied moment	--	Loadcase #3 Moment at point (16,-2,1)
Nodal displ. Nodal rotation	--	Loadcase #3 Node #17
Max nodal displ. Max nodal rotation	--	Loadcase #3
BC force BC moment	--	Loadcase #3
Max/Min axial torsion shear1 /2 bending1 /2	--	Loadcase #3
Grid point force Grid point moment	--	Loadcase #3 Node #2 Element #2 contribution

Applicable Units and Coordinate system

The model is based on imperial units (in, lbf).

Components of any point (such as Center of Gravity) or vector (such as resultant of applied loads, displacements ...) should be calculated in the basic coordinate system. Components of the element internal forces will be identified according to the element coordinate system.

Screenshots(optional)

Note that CAESAR allows the addition of multiple screenshots per dataset.

Data Sheet Columns

column name	description
model	The name of the test model, here 'ATS2'
system_n	The system code of the CAE system creating the STEP file
system_t	The system code of the CAE system importing the STEP file. For native stats, select 'stp'
fixed_dof_nb_a	Total number of fixed Degrees of freedom (DOF) - case A

fixed_dof_nb_b	Total number of fixed Degrees of freedom (DOF) - case B
applied_moment_bx	Resultant moment at origin of total applied forces + resultant of applied moments -- Load case 'B'
applied_moment_by	
applied_moment_bz	
nodal_disp_bx	Components of nodal displacement - case 'B'
nodal_disp_by	
nodal_disp_bz	
nodal_rot_bx	Components of nodal rotation - case 'B'
nodal_rot_by	
nodal_rot_bz	
max_nodal_disp_b	Vector length of Max Nodal displacement - case 'B'
max_nodal_rot_b	Vector length of Max Nodal rotation - case 'B'
bc_force_bx	Summation of boundary condition constraint forces - case 'B'
bc_force_by	
bc_force_bz	
bc_moment_bx	Summation of boundary condition constraint moments - case 'B'
bc_moment_by	
bc_moment_bz	
max_axial_f_1d_elt_b	Maximum axial internal force - 1D elt (center point) -case 'B'
min_axial_f_1d_elt_b	Minimum axial internal force - 1D elt (center point) -case 'B'
max_shear2_1d_elt_b	Maximum shear internal force /plane 2 - 1D elt (center point) - case 'B'
min_shear2_1d_elt_b	Minimum shear internal force /plane 2 - 1D elt (center point) - case 'B'
max_bending2_1d_elt_b	Maximum bending internal moment /plane 2 - 1D elt (center point) - case 'B'
min_bending2_1d_elt_b	Minimum bending internal moment /plane 2 - 1D elt (center point) - case 'B'
grid_point_force_bx	Grid point forces-- case 'B'
grid_point_force_by	
grid_point_force_bz	
grid_point_moment_bx	Grid point moment -- case 'B'
grid_point_moment_by	
grid_point_moment_bz	
date	The date when the statistics were last updated (will be filled in automatically)
issues	A short statement on issues with the file

2.3 Test Case ATS3: beam FEA model using shell elements

All information about this test case can also be viewed in CAESAR on its Information page.

2.3.1 Motivation

As Round 2S is the second opportunity for participants to run formal Implementor Forum testing using AP 209 ed2, the test suite will re-use the simple finite element model components from R1S, with the scope extended to output FEA data as well.

The ATS3 test represents a cantilever beam using “shell” elements only (2D element family), with additional boundary conditions and combinations of lumped and distributed (pressure) applied loads. This test will mainly focus on output results such as nodal displacement/rotation, BC constraint forces, 2D element internal forces, grid point force balance, with some statistics related to input data being reprocessed with new instructions as indicated in section 1.1.

2.3.2 Approach

The approach to be used is described in the latest version of the following documents:

- “Recommended Practices for AP 209 ed2” (at least version 2.0 dated March 30, 2016)

- “AP 209 ed2 Linear Static Structural FEA Handbook vol.1” - (version 1.8.2 or later)

These documents can be found in the CAx-IF website under “Joint Testing Information” and in the CAx-IF member area under “CAE Domain - Information on Round2S of Testing”.

Within the ATS3 model, the following functionalities are in the scope of Round 2S:

- Export/import input data of FEA models with shell elements, including lumped and distributed forces.
- Export/import output data of FEA with :
 - Nodal displacement and rotation
 - BC constraint forces
 - Element internal forces

The AP 209 schema to be used is a corrected version of the AP 209 ed2 schema, which can be found on the CAE-IF member area on the CAx-IF homepages, under “Information on Round 1S of Testing”.

2.3.3 Testing Instructions

The tests will be performed based on the ATS3m5 NASTRAN model described below. This model has been developed by the LOTAR EAS Working Group, and has been checked during previous pilot studies.

2.3.3.1 Test Model Overview

The ATS3m5 model represents a beam (rectangular prism) idealized using “shell” elements (membrane and bending stiffness element), with the following characteristics:

- Isotropic material property
- 4 load cases:
 1. axial distributed load in compressive (-x) direction
 2. lateral distributed load in bending (-y) direction
 3. combination of 1 + 2
 4. normal distributed load in bending (-z) direction

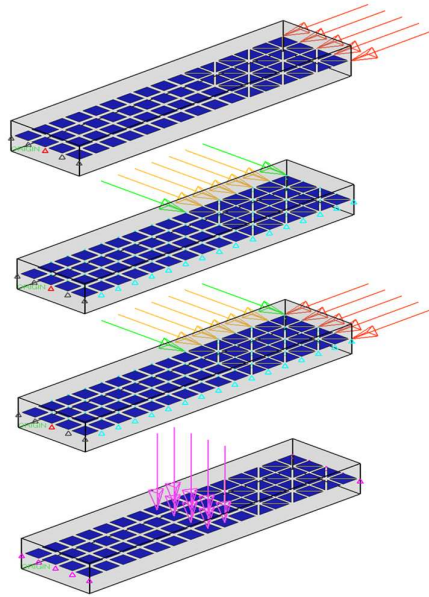


Figure 4: ATS3 model overview

The following output results are available: nodal displacement/rotation, BC constraint forces, 2D element internal forces, strain and stress field, and grid point force balance.

2.3.3.2 Test Model Access.

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under “General Information – File repository in the folder “CAE / Round 2S”:

- ATS3m5.bdf (input data)
- ATS3m5.pch (output data)

2.3.3.3 Test Model Configuration

- See section 2.1.3.3 above.

2.3.4 Statistics

For each STEP file exported or imported for the ATS3 test case, vendors must submit the corresponding statistics to CAESAR. To do so, go to the [ATS3 Data Sheet], and either fill in the web form, or upload a comma-delimited file (.csv) with the data as listed below.

Native Statistics

When exporting a STEP file, report what data importing systems should expect to find. For numeric statistics, enter the respective value or 'na' if not supported. For other statistics, select either 'full support' (i.e. test case and Rec. Pracs. definitions are fulfilled), 'limited support' (meaning the implementation does not meet all criteria and issues may be expected on import), or 'na' if not supported.

Target Statistics

When importing a STEP file, report the results found after processing the file as described in the table below. The value should be given in the unit implicitly defined in the native model.

Load Case Selection, element and node selection

When statistics depend on the load case, node selection or element selection, the item(s) to be considered is (are) indicated in the following table:

Statistic	Case 'A'	Case 'B'
Fixed DOF nb	Loadcase #3	Loadcase #4
Applied moment	--	Loadcase #4 Moment at point (16,-2,1)
Max nodal displ. Max nodal rotation	--	Loadcase #4
BC force	--	Loadcase #4
Max/Min Membrane_X /Y /XY Shear X /Y Bending X /Y /XY	Loadcase #3	Loadcase #4

Applicable Units and Coordinate system

The model is based on imperial units (in, lbf).

Components of any point (such as Center of Gravity) or vector (such as resultant of applied loads, displacements ...) should be calculated in the basic coordinate system. Components of the element internal forces will be identified according to the element coordinate system.

Screenshots(optional)

Note that CASEAR allows the addition of multiple screenshots per dataset.

Data Sheet Columns

column name	description
model	The name of the test model, here 'ATS3'
system_n	The system code of the CAE system creating the STEP file
system_t	The system code of the CAE system importing the STEP file. For native stats, select 'stp'
fixed_dof_nb_a	Total number of fixed Degrees of freedom (DOF) - case A
fixed_dof_nb_b	Total number of fixed Degrees of freedom (DOF) - case B
applied_moment_bx	Resultant moment at origin of total applied forces + resultant of applied moments -- Load case 'B'
applied_moment_by	
applied_moment_bz	
max_nodal_disp_b	Vector length of Max Nodal displacement - case 'B'
max_nodal_rot_b	Vector length of Max Nodal rotation - case 'B'
bc_force_bx	Summation of boundary condition constraint forces - case 'B'
bc_force_by	
bc_force_bz	
max_membr_x_2d_elt_a	Maximum membrane X_face internal force - 2D elt (center point) -case 'A'
min_membr_x_2d_elt_a	Minimum membrane X_face internal force - 2D elt (center point) -case 'A'
max_membr_y_2d_elt_a	Maximum membrane Y_face internal force - 2D elt (center point) -case 'A'
min_membr_y_2d_elt_a	Minimum membrane Y_face internal force - 2D elt (center point) -case 'A'
max_shear_x_2d_elt_b	Maximum shear X_face internal force - 2D elt (center point) - case 'B'
min_shear_x_2d_elt_b	Minimum shear X_face internal force - 2D elt (center point) - case 'B'
max_shear_y_2d_elt_b	Maximum shear Y_face internal force - 2D elt (center point) - case 'B'
min_shear_y_2d_elt_b	Minimum shear Y_face internal force - 2D elt (center point) - case 'B'
max_bending_x_2d_elt_b	Maximum bending X_face internal moment - 2D elt (center point) - case 'B'
min_bending_x_2d_elt_b	Minimum bending X_face internal moment - 2D elt (center point) - case 'B'
max_bending_y_2d_elt_b	Maximum bending Y_face internal moment - 2D elt (center point) - case 'B'
min_bending_y_2d_elt_b	Minimum bending Y_face internal moment - 2D elt (center point) - case 'B'
max_bending_xy_2d_elt_b	Maximum bending XY internal moment - 2D elt (center point) - case 'B'
min_bending_xy_2d_elt_b	Minimum bending XY internal moment - 2D elt (center point) - case 'B'
date	The date when the statistics were last updated (will be filled in automatically)
issues	A short statement on issues with the file

2.4 Test Case ATS4: beam FEA model using solid elements

All information about this test case can also be viewed in CAESAR on its Information page.

2.4.1 Motivation

As Round 2S is the second opportunity for participants to run formal Implementor Forum testing using AP 209 ed2, the test suite will re-use the simple finite element model components from R1S, with the scope extended to output FEA data as well.

The ATS4 test represents a cantilever beam using different “solid” elements, with combinations of lumped applied forces. This test will mainly focus on output results such as nodal displacement/rotation, BC constraint forces, strain and stress field, grid point force balance, with some statistics related to input data being reprocessed with new instructions as indicated in section 1.1.

2.4.2 Approach

The approach to be used is described in the latest version of the following documents:

- “Recommended Practices for AP 209 ed2” (at least version 2.0 dated March 30, 2016)
- “AP 209 ed2 Linear Static Structural FEA Handbook vol.1” - (version 1.8.2 or later)

These documents can be found in the CAx-IF website under “Joint Testing Information” and in the CAx-IF member area under “CAE Domain - Information on Round2S of Testing”.

Within the ATS4 model, the following functionalities are in the scope of Round 2S:

- Export/import input data of FEA models with solid elements, including combination of lumped forces.
- Export/import output data of FEA with :
 - Nodal displacement and rotation
 - BC constraint forces
 - Strain tensor field
 - Stress tensor field
 - Grid point force balance

The AP 209 schema to be used is a corrected version of the AP 209 ed2 schema, which can be found on the CAE-IF member area on the CAx-IF homepages, under “Information on Round 1S of Testing”.

2.4.3 Testing Instructions

The tests will be performed based on the ATS4m5 NASTRAN model described below. This model has been developed by the LOTAR EAS Working Group, and has been checked during previous pilot studies.

2.4.3.1 Test Model Overview

The ATS4m5 model represents a beam (rectangular prism) idealized using “solid” elements (mix of 4-noded tetrahedral, 6-noded pentahedral and 8-noded hexahedral elements), with the following characteristics:

- Isotropic material property
- 3 load cases:

1. axial loads in compressive (-x) direction
2. lateral loads in bending (-y) direction
3. combination of 1 + 2

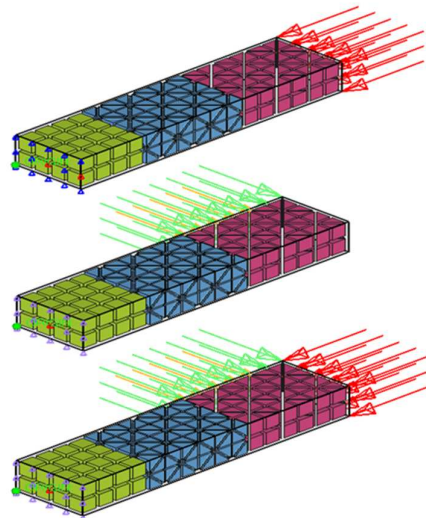


Figure 5: ATS4 model overview

The following output results are available: nodal displacement/rotation, BC constraint forces, strain and stress field, and grid point force balance.

2.4.3.2 Test Model Access.

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under “General Information – File repository” in the folder “CAE / Round 2S”:

- ATS4m5.bdf (input data)
- ATS4m5.pch (output data)

2.4.3.3 Test Model Configuration

- See section 2.1.3.3 above.

2.4.4 Statistics

For each STEP file exported or imported for the ATS4 test case, vendors must submit the corresponding statistics. To do so, go to the [ATS4 Data Sheet], and either fill in the web form, or upload a comma-delimited file (.csv) with the data as listed below.

Native Statistics

When exporting a STEP file, report what data importing systems should expect to find. For numeric statistics, enter the respective value or 'na' if not supported. For other statistics, select either 'full support' (i.e. test case and Rec. Pracs. definitions are fulfilled), 'limited support' (meaning the implementation does not meet all criteria and issues may be expected on import), or 'na' if not supported.

Target Statistics

When importing a file, report the results found after processing the file as described in the table below. The value should be given in the unit implicitly defined in the native model.

Load Case Selection, element and node selection

When statistics depend on the load case, node selection or element selection, the item(s) to be considered is (are) indicated in the following table:

Statistic	Case 'A'	Case 'B'
Fixed DOF nb	Loadcase #1	Loadcase #3
Applied moment	--	Loadcase #3 Moment at point (16,-2,1)
Max nodal displ.	--	Loadcase #3
Strain (Vol change)	Loadcase #1 Element #30	Loadcase #3 Element #30
Stress (Von-Mises)	Loadcase #1 Element #30	Loadcase #3 Element #30
Max/min strain	Loadcase #1	Loadcase #3
Max stress	Loadcase #1	Loadcase #3
BC force	--	Loadcase #3
Grid point force	--	Loadcase #3 Node #2 Element #2 contribution

Applicable Units and Coordinate system

The model is based on imperial units (in, lbf).

Components of any point (such as Center of Gravity) or vector (such as resultant of applied loads) should be calculated in the basic coordinate system.

Components of any point (such as Center of Gravity) or vector (such as resultant of applied loads, displacements ...) should be calculated in the basic coordinate system.

Data Sheet Columns

column name	description
model	The name of the test model, here 'ATS4'
system_n	The system code of the CAE system creating the STEP file
system_t	The system code of the CAE system importing the STEP file. For native stats, select 'stp'
fixed_dof_nb_a	Total number of fixed Degrees of freedom (DOF) - case A
fixed_dof_nb_b	Total number of fixed Degrees of freedom (DOF) - case B
applied_moment_bx	Resultant moment at origin of total applied forces + resultant of applied moments -- Load case 'B'
applied_moment_by	
applied_moment_bz	
max_nodal_disp_b	Vector length of Max Nodal displacement - case 'B'
strain_vol_a	Volume change (strain at center point) - Case A
strain_vol_b	Volume change (strain at center point) - Case B
max_strain_vol_b	Maximum Volume change (strain at center point) - Case B
min_strain_vol_b	Minimum Volume change (strain at center point) - Case B
stress_vm_a	Von Mises stress value (center point) - Case A
stress_vm_b	Von Mises stress value (center point) - Case B

max_stress_vm_a	Maximum Von Mises stress value (center point) - Case A
max_stress_vm_b	Maximum Von Mises stress value (center point) - Case B
bc_force_bx	Summation of boundary condition constraint forces - case 'B'
bc_force_by	
bc_force_bz	
grid_point_force_bx	Grid point forces-- case 'B'
grid_point_force_by	
grid_point_force_bz	
date	The date when the statistics were last updated (will be filled in automatically)
issues	A short statement on issues with the file

Annex A General Considerations about CAE

CAE is dedicated to analyze/predict the physical behavior of parts or products. It is a way to calculate the displacement field and the related stress/strain field (mechanical/structural analysis), temperature fields and heat flows (Thermal analysis), electric and magnetic fields (Electromagnetic analysis), ... Such problems can be addressed separately (mono-physics analysis) or simultaneously (coupled or multi-physics analysis), but the results are generally expressed as scalar fields, vector fields or tensor fields.

Basically, CAE models (whether it could be Finite Element models, Boundary element models or any other mathematical methods for discretization) deals with the following sets of data:

Input data

- **Discretized geometrical data**, which represents the product to be studied. It uses nodes (points) and elements for connecting nodes. Some special elements may require additional properties such as section area, thickness, offset, ...
- **Material properties**, which are parameters used to describe the physical behavior. Most of the time, the physical behavior is hard coded in the CAE solver software, so that the meaning of each material parameter is defined in the user manual of the solver software. However, some standard physical behaviors are available in (almost) all CAE software with the same definition and same material parameters: mechanical elasticity, thermal expansion, heat transfer, Navier-Stokes CFD, ...
- **Boundary conditions**. This is a set of locations where the solution is known/imposed, either for all components of the solution or one or more of the components if the solution is on the vector form. This will decrease the number of unknown values to be calculated (called the number of free Degrees Of Freedom –free DOF-) to fully define the solution, more formally to define the value of each component of the solution on each node without boundary condition.
- **Applied loads** are external solicitations, depending of the physics to be studied. For structural analysis it could be forces and moments; Heat flow for thermal analysis, ... Such solicitations can be defined as :
 - **Nodal (or lumped) solicitations**, i.e., the component values are defined on specific nodes
 - **Distributed solicitations**, i.e., the component values are defined along a line (linear solicitation), over a surface (surfacic solicitation) or within a volume (volumic solicitation)

Output data

As explained above, the results are expressed as scalar fields, vector fields or tensor fields, depending of the physics. Two set of output data can be considered:

- **Primary results** which are directly calculated by the solver as the components of the solution at each node (i.e., each DOF). Could be nodal displacements for structural analysis, temperature for thermal analysis or flow speed for CFD...
- **Derivative results** which are post-calculated from the primary results, and which could be located either on nodes location or anywhere else using discretization functions for interpolation. Basically, for structural analysis, it could be displacements out of node position, internal loads, strain or stresses (nodal value or out of node position), or any relevant combination for analysis (such as Von Mises scalar stress field for instance).

Similar derivative results can be defined for other physics.

Annex B Information related to Statistics calculation

The value should be given in the unit implicitly defined in the native model. This units are (in, lbf) and shall not be modified. Nevertheless, the **units should be encoded in the STEP file**.

Components of any point (such as CDG) or vector (such as resultant of applied loads) should be calculated in the basic coordinate system.

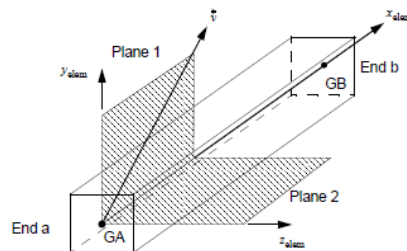
Discretized geometrical values such as “Cumulated length of 1D elements” resp. “2D elements”, “3D elements” (1D_Model_size, resp. 2D_Model_size, 3D_Model_size) should be calculated based on the Finite Element discretization, which can be slightly different from the original CAD definition.

- 1d_model_size : “1D elements” refers to curve_3D_elements
- 2d_model_size : “2D elements” refers to surface_3D_elements
- 3d_model_size : “3D elements” refers to volume_3D_elements

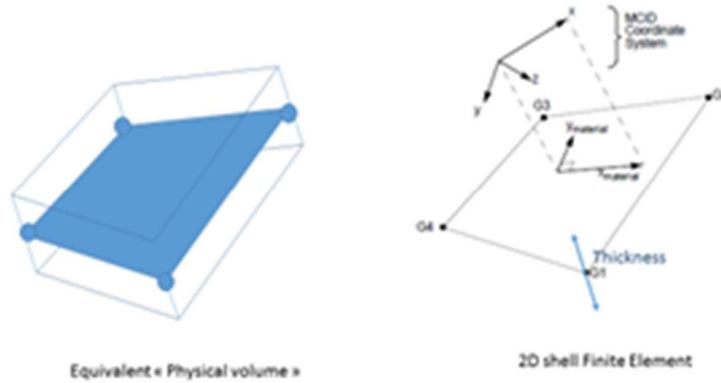
Node_nb : The dummy nodes are excluded from the total number of nodes count.

Total_model_volume:

1D elements in Finite Element models are used to represent a beam or a bar along their middle-line (so called neutral axis). During the computation, the cross-section value of these elements is taken into account in order to evaluate the stiffness and more generally the mechanical behavior of the beam. Then, the “physical 1D volume” is given by the summation of (element_length * cross_section_area) over all 1D elements. Cross section area is given within the properties attached to each element.



In the same way, 2D elements are used to represent shell or plate including thickness defined in the properties attached to each element. Then the “physical 2D volume” is given by the summation of (element_surface * thickness) over all 2D elements.



For 3D elements, physical 3D volume is equivalent to 3D_model_size

So the “**total_model_volume**” is given by: (physical 1D volume) + (physical 2D volume) + (physical 3D volume).

Total_mass : the total mass of the model is given by the summation of (element_volume * element_mass_density) over all the elements

Center of Gravity (CDG): should be calculated taking into account the density of all material in the model

Annex C Calculation of the resultants of applied loads (forces / moments)

Preliminary remarks and Definitions/notations

Applied loads could be either lumped forces, lumped moments and/or distributed forces (linear, surfacic or volumic). Thermal loads as well as distributed moments are not considered.

\vec{F}_P = Lumped force applied at point P

\vec{M}_m = Lumped applied moment

\vec{f}_l = distributed force along curve C

\vec{f}_s = distributed force along surface S

\vec{f}_v = distributed force along volume V

All applied loads are defined as 3D vectors, whose components relative to the basic coordinates are noted:

$$\vec{u} \begin{cases} |u_x \\ |u_y \\ |u_z \end{cases}$$

Resultant forces

\vec{R} : Resultant of applied forces:

$$\vec{R} = \sum_P \vec{F}_P + \int_C \vec{f}_l \cdot dl + \iint_S \vec{f}_s \cdot ds + \iiint_V \vec{f}_v \cdot dv$$

As this is a vector form, this formula applies similarly to each component (X, Y, Z) of the vectors.

When the distributed applied forces are constant over each individual finite element where they are applied, the formula comes to:

$$\vec{R} = \sum_P \vec{F}_P + \sum_E \vec{f}_{lE} \cdot L_E + \sum_E \vec{f}_{sE} \cdot S_E + \sum_E \vec{f}_{vE} \cdot V_E$$

Where:

- \vec{f}_{lE} is the constant distributed linear force over element E, and L_E is the length of the corresponding element
- \vec{f}_{sE} is the constant distributed surfacic force over element E, and S_E is the surface of the corresponding element
- \vec{f}_{vE} is the constant distributed volumic force over element E, and V_E is the volume of the corresponding element

- \sum_E means the sum over elements E where linear forces (resp. surfacic forces, volumic forces) are applied
- \sum_P means the sum over each applied lumped forces \vec{F}_P

Resultant moment at any given point Q

$\vec{M}_{/Q}$: Resultant moment at any given point Q

$$\vec{M}_{/Q} = \sum_m \vec{M}_m + \vec{M}_{f/Q}$$

Where:

- \sum_m means the sum over all applied moments \vec{M}_m

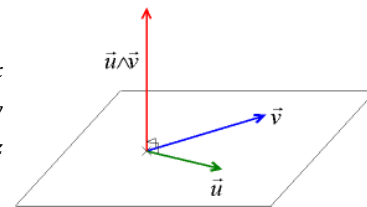
And

- $\vec{M}_{f/Q}$ = Resultant moment of applied forces at point Q :

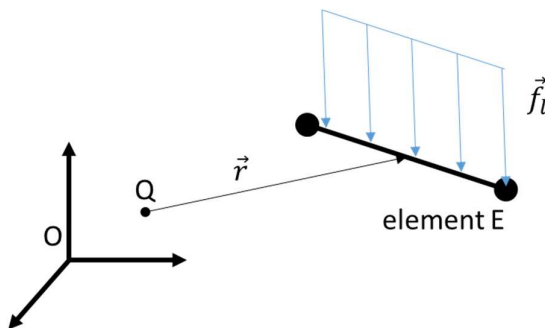
$$\vec{M}_{f/Q} = \sum_P \vec{QP} \cdot \vec{F}_P + \int_C \vec{r} \cdot \vec{f}_l \cdot dl + \iint_S \vec{r} \cdot \vec{f}_s \cdot ds + \iiint_V \vec{r} \cdot \vec{f}_v \cdot dv$$

Where $\cdot \cdot \cdot$ stands for the vector product defined by:

$$\vec{u} \cdot \cdot \cdot \vec{v} \begin{cases} u_y \cdot v_z - u_z \cdot v_y \\ u_z \cdot v_x - u_x \cdot v_z \\ u_x \cdot v_y - u_y \cdot v_x \end{cases} \quad \text{with: } \vec{u} \begin{cases} u_x \\ u_y \\ u_z \end{cases} \quad \text{and } \vec{v} \begin{cases} v_x \\ v_y \\ v_z \end{cases}$$



And \vec{r} being the varying vector from the point Q to the location where the distributed load f is applied:



When the distributed applied forces are constant over each individual finite element where they are applied, the formula comes to:

$$\vec{M}_{f/Q} = \sum_P \overrightarrow{QP} \cdot \vec{F}_P + \sum_E \overrightarrow{QM_E} \cdot \vec{f}_{lE} \cdot L_E + \sum_E \overrightarrow{QM_E} \cdot \vec{f}_{sE} \cdot S_E \\ + \sum_E \overrightarrow{QM_E} \cdot \vec{f}_{vE} \cdot V_E$$

Where:

- M_E is the point in the “middle” of element E (formally the barycenter of element E)
- \vec{f}_{lE} is the constant distributed linear force over element E, and L_E is the length of the corresponding element
- \vec{f}_{sE} is the constant distributed surfacic force over element E, and S_E is the surface of the corresponding element
- \vec{f}_{vE} is the constant distributed volumic force over element E, and V_E is the volume of the corresponding element
- \sum_E means the sum over elements E where linear forces (resp. surfacic forces, volumic forces) are applied

Annex D Calculation of strains and stresses statistics

Statistic related to the strain field
--

Definition:

The statistic to be considered is defined as the first invariant (scalar value) calculated from the strain tensor, at a given point for a given load case:

$$I_1 = \text{Tr}(\mathbf{E}) = \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33} = \sum_i \varepsilon_{ii}$$

Where
$$\varepsilon_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

defines each individual component of the strain tensor (available in the output file from the CAE analysis).

As I_1 is mathematically invariant, the value remains the same whatever the coordinate system you use to calculate it, which make this statistic easy to calculate.

More particularly, if the principal strain values are available, we can use :

$$I_1 = \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33} = \varepsilon_I + \varepsilon_{II} + \varepsilon_{III}$$

Where
$$\begin{pmatrix} \varepsilon_I & 0 & 0 \\ 0 & \varepsilon_{II} & 0 \\ 0 & 0 & \varepsilon_{III} \end{pmatrix}$$
 is the strain tensor written in the principal strain directions.

I_1 has a physical meaning which could be interesting from an engineering point of view...

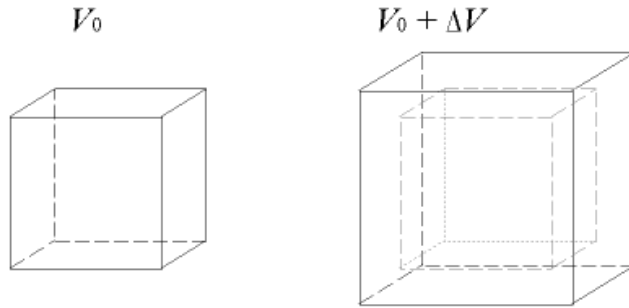
Physical meaning:

I_1 can be interpreted as the volume change ratio (first order approximation) within small strains framework:

Let's consider the elementary cubic volume V_0 changing to $V=V_0+\Delta V$ due to the strain field ϵ_{ij} , and oriented in the principal strain directions.

The volume change ratio is define as : $\frac{\Delta V}{V_0}$

(see following picture).



Then

$$V = a \cdot (1 + \epsilon_{11}) \times a \cdot (1 + \epsilon_{22}) \times a \cdot (1 + \epsilon_{33})$$

Where a is the length of each side of the elementary cubic volume V .

$$\frac{\Delta V}{V_0} = \frac{(1 + \epsilon_{11} + \epsilon_{22} + \epsilon_{33} + \epsilon_{11} \cdot \epsilon_{22} + \epsilon_{11} \cdot \epsilon_{33} + \epsilon_{22} \cdot \epsilon_{33} + \epsilon_{11} \cdot \epsilon_{22} \cdot \epsilon_{33}) \cdot a^3 - a^3}{a^3}$$

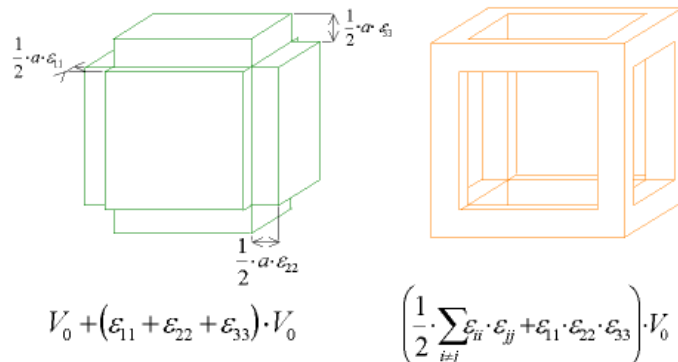
As we consider only small strains hypothesis:

$$1 \gg \epsilon_{ii} \gg \epsilon_{ii} \cdot \epsilon_{jj} \gg \epsilon_{11} \cdot \epsilon_{22} \cdot \epsilon_{33}$$

And so

$$\frac{\Delta V}{V_0} : \sim \epsilon_{11} + \epsilon_{22} + \epsilon_{33}$$

The approximated volume change is indicated in green in the following figure, while the neglected volume change is indicated in orange (second order term).



Statistic related to the **stress field**

Definition:

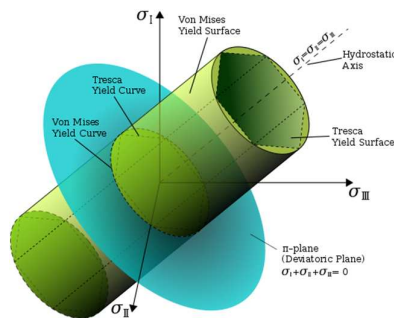
The statistic to be considered is defined as the Von-Mises criteria σ_v (scalar value) calculated from the stress tensor, at any given point for a given load case, with the following formula :

$$\sigma_v^2 = \frac{1}{2} [(\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 + 6(\sigma_{23}^2 + \sigma_{31}^2 + \sigma_{12}^2)]$$

$$\begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{bmatrix}$$

Where σ_{ij} is one of the components of the stress tensor

From an engineering point of view, the Von-Mises criteria is often used as a failure criteria for ductile material.



Annex E 1D element and 2D element coordinate system and internal element forces orientation

1D elements

- Plane 1 is defined as the x-y plane in the local coordinate system
- Plane 2 is defined as the x-z plane in the local coordinate system

The positive directions for element forces are shown in the figure below.

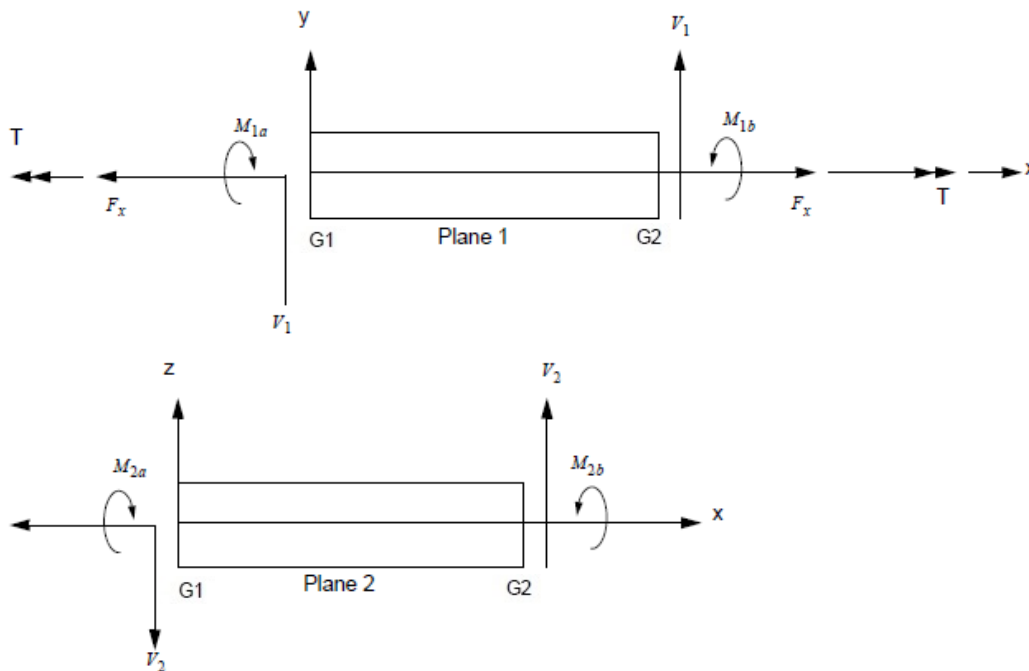


Figure 6: Orientation for 1D element internal forces

2D elements

The 2D internal element forces are defined relatively to the x-y local coordinate system of each element.

The x-y local coordinate system is described in the figures below for 4 nodes elements (resp. 3 nodes elements). This coordinate system is related with the numbering of the nodes (G1, G2, G3, G4) (resp G1, G2, G3) of the elements

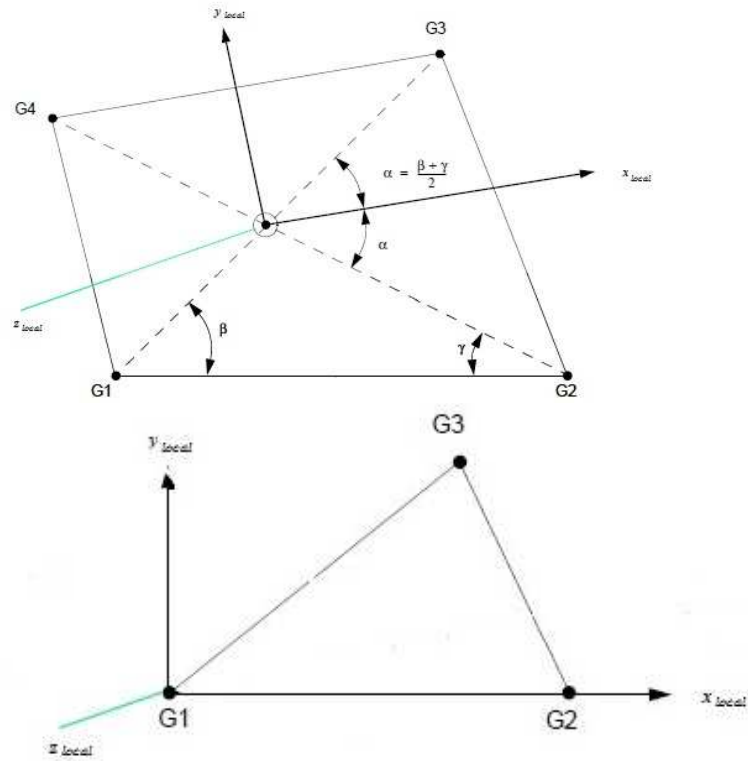


Figure 7: Local coordinate system for 2D elements

The positive directions for 2D element forces and moments in the element coordinate system are shown in the figure below:

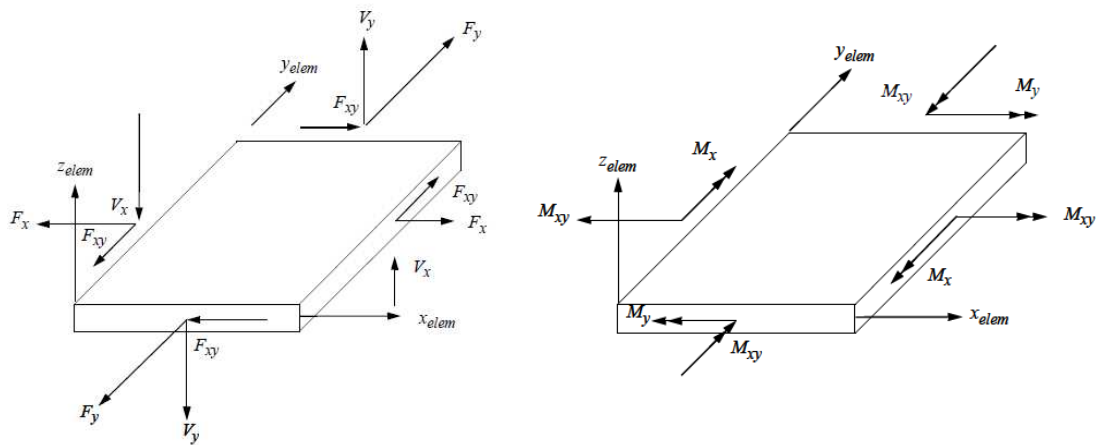


Figure 8: Orientation for 2D element internal forces