

**Test Suite for the
CAE Implementor Forum
Round 1S**

June - December 2017

Release 2.0

December 8, 2017

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.....	3
1.3	Testing Schedule.....	4
1.4	Copyrights on Test Cases	4
2	Synthetic Test Case Specifications	5
2.1	Test Case ATS1: beam FEA model using rod elements	5
2.2	Test Case ATS2: beam FEA model using bar elements	7
2.3	Test Case ATS3: beam FEA model using shell elements	10
2.4	Test Case ATS4: beam FEA model using solid elements	13
Annex A	General Considerations about CAE	16
Annex B	Information related to Statistics calculation	17
Annex C	Information related to the calculation of the resultants of applied loads (forces and moments).....	19

List of Figures

Figure 1:	CAE-IF Round1S Schedule	4
Figure 2:	ATS1 model overview.....	5
Figure 3:	ATS2 model overview.....	8
Figure 4:	ATS3 model overview.....	11
Figure 5:	ATS4 model overview.....	14

Document History

Release	Date	Change
0.1	July 26, 2017	First Draft Release
0.2	August 30, 2017	Update statistic definition and general instructions
0.3	September 5, 2017	Update statistic definition; editorial changes
0.4	September 7, 2017	Update statistic definition for ATS3; editorial changes
0.5	September 8, 2017	Editorial changes
1.0	September 8, 2017	1 st Published on CAX-IF Joint Testing Information page
1.1	September 19, 2017	Updated Schedule due to shifted Kick-Off Meeting
1.2	September 29, 2017	Reviewed wording for statistics
2.0	December 8, 2017	Additional information regarding statistics calculation

1 Introduction

This document describes the suite of test cases to be used for the first test round of the CAE Implementor Forum (CAE-IF). The CAE-IF is a workgroup with 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 AP209ed2.

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 ANFeT, 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.

This test suite includes synthetic models for testing the following capabilities: export /import Input FEA models for simplified linear static structural analysis.

1.1 *Functionality tested in this round*

As a first step, this test round is limited to Linear Static Structural FEA models, using basic components. More complex models, including production models, will be introduced later.

Functionality tested in this round relates to:

- **Input data for structural FEA models**, which includes all information regarding discretized geometry (Nodes, Elements, ...), physical and material properties, loads and boundary conditions (see Annex A). Round 1S will focus on export/import capabilities, as well as retrieving key values with significance for the simulation, such as geometry characteristics, center of gravity and inertia, total number of degrees of freedom (DOF), total applied loads, ...

More precisely, the required activity will comprise the following steps:

- Translate native FEA input (MSC Nastran format) to ISO STEP AP209 ed2
- Generate key values (described later as “statistics”)
- Consume each other’s STEP AP209 ed2 files
- Retrieve and compare key values

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 v1.12, dated July 5, 2016.

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

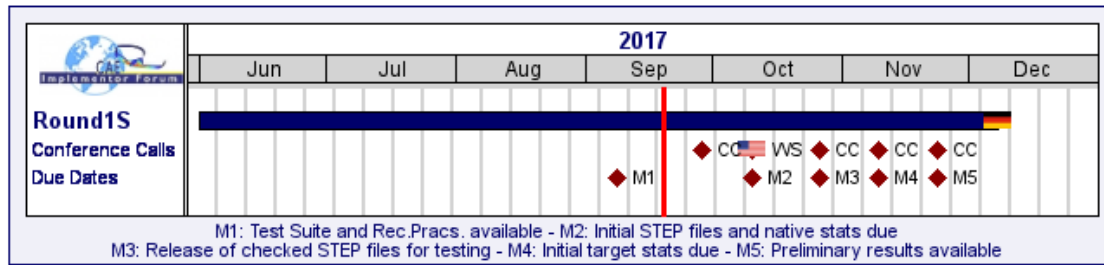
Native FEA Model use NASTRAN card descriptions. Documentation of NASTRAN input syntax is available in the “NASTRAN quick reference guide”, which can be download from the MSC homepage at:

- <https://simcompanion.mscsoftware.com/infocenter/index?page=content&id=DOC11146&>

CAE models generally are “unit consistent models”. This means there is no need for data conversion. Participants are requested to keep the units (inch, lbf) unchanged and to deliver results and statistics using the same unit system as the native models.

1.3 Testing Schedule

The following schedule has been agreed on for Round 1S:



Date	Action
8 Sep 2017 (Fri)	Test Suite and Rec.Pracs. available
28 Sep 2017 (Thu)	Round 1S Kick-Off Conference Call
10 Oct 2017 (Tue)	CAX-IF Technical Workshop in Myrtle Beach, SC, USA / Initial STEP files and native stats due
26 Oct 2017 (Thu)	Release of checked STEP files for testing / Bi-weekly CAE-IF Conference Call
9 Nov 2017 (Thu)	Initial target stats due / Bi-weekly CAE-IF Conference Call
23 Nov 2017 (Thu)	Preliminary results available / Bi-weekly CAE-IF Conference Call
5 Dec 2017 (Tue) - 7 Dec 2017 (Thu)	CAX-IF Round1S Review Meeting in Darmstadt, Germany

Figure 1: CAE-IF Round1S Schedule

The CAE-IF Round 1S Technical Workshop will be held in conjunction with the PDES, Inc. Fall Offsite meeting and a LOTAR workshop. Conference calls and web sessions will also be available for those not attending the meeting to dial in.

The CAE-IF Round1S Review meeting will take place in conjunction with a LOTAR 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 ANFeT, 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 ANFeT, 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 1S is the first trial for participants to run formal Implementor Forum using AP209ed2, the test suite will use basic finite element model components. The ATS1 test represents a cantilever beam using “rod” elements only, with one applied lumped force.

2.1.2 Approach

The approach to be used is described in the latest version of the “AP209 FEA Handbook”, which can be found in the CAX-IF member area under “CAE Domain - Information on Round1S of Testing”.

Within the CAE domain, the following functionalities are in scope of Round 1S:

- Export/import input data of FEA models with rod elements and a lumped force
- Evaluate key values from the FEA models

The AP209 schema to be used is the AP209ed2, which can be found on the public CAX-IF web sites under “Joint Testing Information”.

2.1.3 Testing Instructions

The tests will be performed based on the ATS1m4 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 ATS1m4 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]

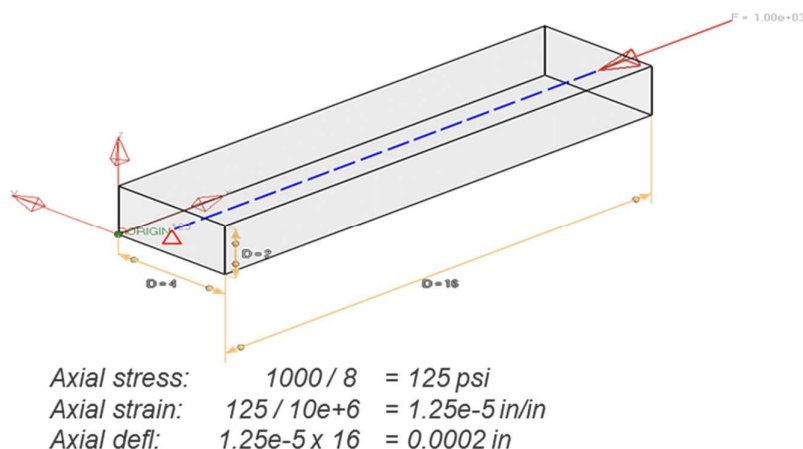


Figure 2: ATS1 model overview

2.1.3.2 Test Model Access

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under “CAE Domain - Information on Round1S of testing”:

- ATS1m4.bdf

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:

- 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

For statistics depending on the load case (Free DOF, Applied Forces), the following load case shall be used:

- Load Case 'A': Load Case #1

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.

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'
unit	The unit the model is designed in

node_nb	Total number of nodes
element_nb	Total number of elements
free_dof_nb	Total number of free Degrees of Freedom (DOF) -- Load case 'A'
1d_model_size	Cumulated length of 1D elements
2d_model_size	Cumulated surface area of 2D elements
3d_model_size	Cumulated volume of 3D elements
total_model_vol	Cumulated volume of all (1D, 2D, 3D) elements
total_mass	Total mass of the model. The value shall be given in the unit implicitly defined in the model.
gravx	Position of the Center of Gravity for the model
gravy	
gravz	
loadcases_nb	Number of different load cases
applied_forcex	Resultant of applied forces -- Load case 'A'
applied_forcey	
applied_forcez	
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 Test Round 1S is the first trial for participants to run formal Implementor Forum using AP209ed2, the test suite will use basic finite element model components. The ATS2 test represents a cantilever beam using “bar” elements only, with combinations of lumped applied loads.

2.2.2 Approach

The approach to be used is described in the latest version of the “AP209 FEA Handbook”, which can be found in the CAX-IF member area under “CAE Domain - Information on Round1S of Testing”.

Within the CAE domain, the following functionalities are in scope of Round 1S:

- Export/import input data of FEA models with bar elements, including lumped forces.
- Evaluate key values from the FEA models

The AP209 schema to be used is the AP209ed2, which can be found on the public CAX-IF web sites under “Joint Testing Information”.

2.2.3 Testing Instructions

The tests will be performed based on the ATS2m4 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 ATS2m4 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

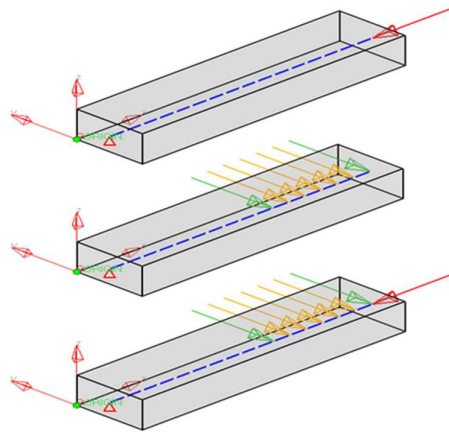


Figure 3: ATS2 model overview

2.2.3.2 Test Model Access.

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under “CAE Domain - Information on Round1S of testing”:

- ATS2m4.bdf

2.2.3.3 Test Model Configuration

- See section 2.1.3.23 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

Some of the Statistics for this test case are load case-related (Free DOF, Applied Forces). The statistics will not be evaluated for all load cases in the model, but only for representative/interesting ones.

The following load case shall be used:

- Load Case 'A': Load Case #3

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.

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 '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'
unit	The unit the model is designed in
node_nb	Total number of nodes
element_nb	Total number of elements
free_dof_nb	Total number of free Degrees of Freedom (DOF) -- Load case 'A'
1d_model_size	Cumulated length of 1D elements
2d_model_size	Cumulated surface area of 2D elements
3d_model_size	Cumulated volume of 3D elements
total_model_vol	Cumulated volume of all (1D, 2D, 3D) elements
total_mass	Total mass of the model. The value shall be given in the unit implicitly defined in the model.
gravx	Position of the Center of Gravity for the model
gravy	
gravz	
loadcases_nb	Number of different load cases
applied_forcex	Resultant of applied forces -- Load case 'A'
applied_forcey	
applied_forcez	
applied_momentx	Resultant moment at origin of total applied forces + resultant of applied moments -- Load case 'A'
applied_momenty	
applied_momentz	
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 Test Round 1S is the first trial for participants to run formal Implementor Forum using AP209ed2, the test suite will use basic finite element model components. The ATS3 test represents a cantilever beam using “shell” elements only, with additional boundary conditions and combinations of lumped and distributed (pressure) applied loads.

2.3.2 Approach

The approach to be used is described in the latest version of the “AP209 FEA Handbook”, which can be found in the CAX-IF member area under “CAE domain - Information on Round1S of Testing”.

Within the CAE domain, the following functionalities are in scope of Round 1S:

- Export/import input data of FEA models with shell elements, including lumped and distributed forces.
- Evaluate key values from the FEA models

The AP209 schema to be used is the AP209ed2, which can be found on the public CAX-IF web sites under “Joint Testing Information”.

2.3.3 Testing Instructions

The tests will be performed based on the ATS3m4 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 ATS3m4 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

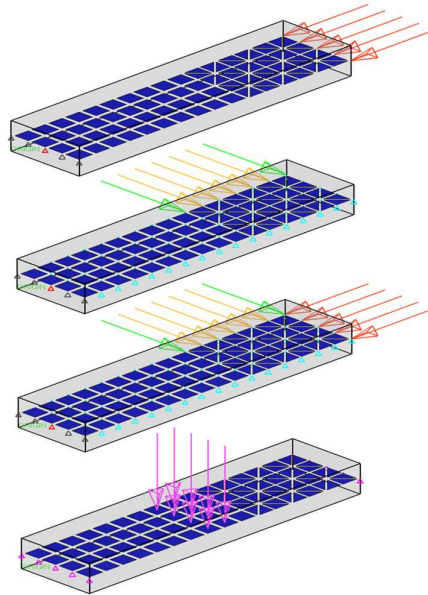


Figure 4: ATS3 model overview

2.3.3.2 Test Model Access.

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under "CAE Domain - Information on Round1S of testing":

- ATS3m4.bdf

2.3.3.3 Test Model Configuration

- See section 2.1.3.23 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 Cases Selection

Some of the Statistics for this test case are load case-related (Free DOF, Applied Forces). The statistics will not be evaluated for all load cases in the model, but only for representative/interesting ones.

The following load case shall be used:

- Load Case 'A': Load Case #3
- Load Case 'B': Load Case #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) should be calculated in the basic 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'
unit	The unit the model is designed in
node_nb	Total number of nodes
element_nb	Total number of elements
free_dof_nb	Total number of free Degrees of Freedom (DOF) -- Load case 'A'
free_dof_nb_b	Total number of free Degrees of Freedom (DOF) -- Load case 'B'
1d_model_size	Cumulated length of 1D elements
2d_model_size	Cumulated surface area of 2D elements
3d_model_size	Cumulated volume of 3D elements
total_model_vol	Cumulated volume of all (1D, 2D, 3D) elements
total_mass	Total mass of the model. The value shall be given in the unit implicitly defined in the native model.
gravx	Position of the Center of Gravity for the model
gravy	
gravz	
loadcases_nb	Number of different load cases
applied_forcx	Resultant of applied forces -- Load case 'A'
applied_forcey	
applied_forcez	
applied_force_bx	Resultant of applied forces -- Load case 'B'
applied_force_by	
applied_force_bz	
applied_momentx	Resultant moment at origin of total applied forces + resultant of applied moments -- Load case 'B'
applied_momenty	
applied_momentz	
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 Test Round 1S is the first trial for participants to run formal Implementor Forum using AP209ed2, the test suite will use basic finite element model components. The ATS4 test represents a cantilever beam using different “solid” elements, with combinations of lumped applied forces.

2.4.2 Approach

The approach to be used is described in the latest version of the “AP209 FEA Handbook”, which can be found in the CAx-IF member area under “CAE Domain - Information on Round1S of Testing”.

Within the CAE domain, the following functionalities are in scope of Round 1S:

- Export/import input data of FEA models with solid elements, including combination of lumped forces.
- Evaluate key values from the FEA models

The AP209 schema to be used is the AP209ed2, which can be found on the public CAx-IF web sites under “Joint Testing Information”.

2.4.3 Testing Instructions

The tests will be performed based on the ATS4m4 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 ATS4m4 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 distributed load in compressive (-x) direction
 2. lateral distributed load in bending (-y) direction
 3. combination of 1 + 2

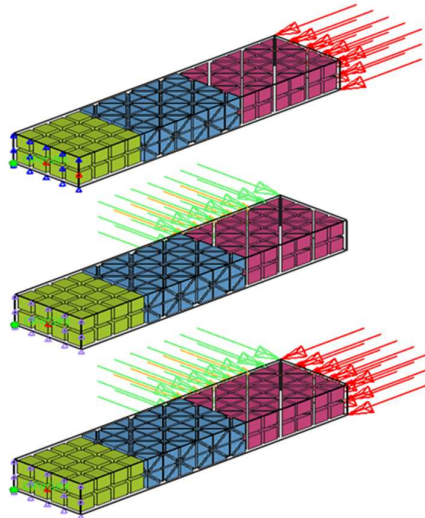


Figure 5: ATS4 model overview

2.4.3.2 Test Model Access.

The native NASTRAN files can be downloaded from the member area of the CAx-IF homepages under “CAE Domain - Information on Round1S of testing”:

- ATS4m4.bdf

2.4.3.3 Test Model Configuration

- See section 2.1.3.23 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 '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

Some of the Statistics for this test case are load case-related (Free DOF, Applied Forces , Applied Moment). The statistics will not be evaluated for all load cases in the model, but only for representative/interesting ones.

The following load case shall be used:

- Load Case 'A': Load Case #3

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.

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'
unit	The unit the model is designed in
node_nb	Total number of nodes
element_nb	Total number of elements
free_dof_nb	Total number of free Degrees of Freedom (DOF) -- Load case 'A'
1d_model_size	Cumulated length of 1D elements
2d_model_size	Cumulated surface area of 2D elements
3d_model_size	Cumulated volume of 3D elements
total_model_vol	Cumulated volume of all (1D, 2D, 3D) elements
total_mass	Total mass of the model. The value shall be given in the unit implicitly defined in the model.
gravx	Position of the Center of Gravity for the model
gravy	
gravz	
loadcases_nb	Number of different load cases
applied_forcex	Resultant of applied forces -- Load case 'A'
applied_forcey	
applied_forcez	
applied_momentx	Resultant moment at origin of total applied forces + resultant of applied moments -- Load case 'A'
applied_momenty	
applied_momentz	
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 is 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**, ie, the component values are defined on specific nodes
 - **Distributed solicitations**, ie, 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 (ie 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

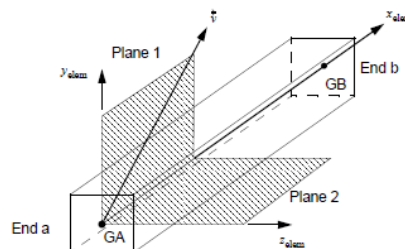
Node_nb : The dummy nodes are excluded from the total number of nodes count.

Total_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

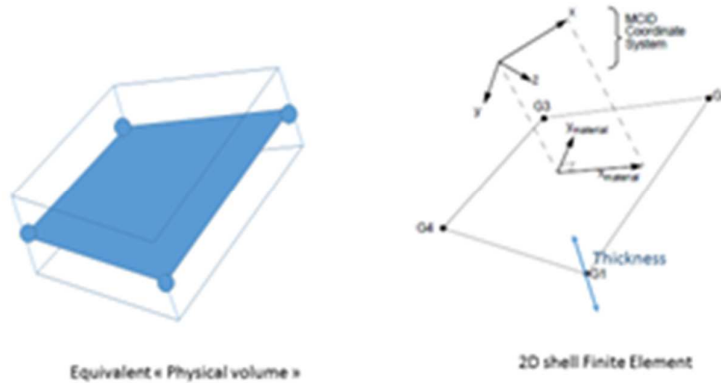
Total_model_volume:

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).

Annex C Information related to the calculation of the resultants of applied loads (forces and 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 origin O

$\vec{M}_{/O}$: Resultant moment at origin O

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

Where:

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

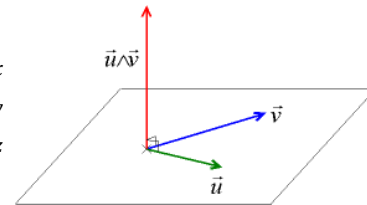
And

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

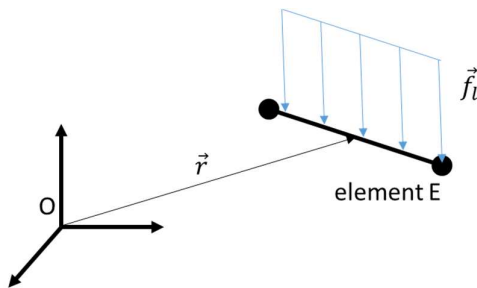
$$\vec{M}_{f/O} = \sum_P \vec{OP} \cdot \vec{F}_P + \int_C \vec{r} \cdot \vec{f}_l \cdot dl + \iint_S \vec{r} \cdot \vec{f}_s \cdot ds + \iiint_V \vec{r} \cdot \vec{f}_v \cdot dv$$

Where \cdot stands for the vector product defined by:

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



And \vec{r} being the varying vector from the origin O 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:

$$\vec{M}_{f/O} = \sum_P \vec{OP} \cdot \vec{F}_P + \sum_E \vec{OM}_E \cdot \vec{f}_{lE} \cdot L_E + \sum_E \vec{OM}_E \cdot \vec{f}_{sE} \cdot S_E \\ + \sum_E \vec{OM}_E \cdot \vec{f}_{vE} \cdot V_E$$

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