



**Test Suite for the
CAX Implementor Forum
Round 23J**
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1.0 Introduction

This document describes the suite of test cases to be used for the twenty-third round of testing of the CAx Implementor Forum (CAx-IF). The CAx-IF is a joint testing forum organized by PDES, Inc. and the ProSTEP iViP Association. The test rounds of the CAx-IF concentrate primarily on testing the interoperability and conformance of STEP processors based on AP 203 and AP 214.

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 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 CAx-IF participants.

This test suite includes synthetic models for testing the following capabilities: Presentation of Geometric & Dimensional Tolerances, both as Polylines and semantically based on Representation, Density & Material Name, User Defined Attributes, Construction Geometry, and Groups.

Production models are provided for assemblies and piece parts. The basis for the production test cases is native CAD models. Each test case therefore originates from a single CAD system, and the set of test cases to be pre-processed (converted to STEP files) is unique for each CAD system. After pre-processing, the resulting STEP files are then to be imported/post-processed/read in by the rest of the participants.

1.1 **Functionality tested in this round**

Functionality tested in this round relates to:

- **Geometric & Dimensional Tolerances** describes the capability to embed information about tolerances which are necessary input for the manufacturing of the part into the 3D model. Past tests in the CAx-IF were focused on the ‘representation’ approach, i.e. transport the product manufacturing information (PMI) in a re-usable way, without displaying it in the 3D model. In Round23J, two different ways for ‘presentation’ of PMI in 3D will be tested:
 - “Polyline Presentation” refers to breaking down each annotation into polylines and arcs, and exchanging them as wireframe geometry. This preserves the exact shape of the annotation, but is human readable only.
 - “Representation and Semantic Presentation” relies on the “Representation” capability to render the information contents to be displayed. These are supplemented with basic styling and positioning information, to enable the importing CAD system to re-create the annotation elements using its internal PMI capability, and may be supplemented with additional textual information.
- **Density & Material Name** refers to the capability to transfer information about the material a part is manufactured from in the STEP file. The basic properties in focus are the density (to enable mass calculation) and the material’s name and id. Tests for the STEP file library in Round 22J showed that an additional test to resolve issues is necessary.

- **User Defined Attributes** are descriptions or values that can be added by the user in the CAD system and associated with a part or geometric elements in the model. Material or production costs are an example for this. This is the first test of this capability.
- **Construction Geometry** defines all geometric elements in a model that do not belong to the manufactured shape of the part, but serve as a reference basis for other information, such as the center line of a hole for example. This will be the first test of this functionality.
- **Groups** allow the user to assign a number of product data elements (solids, surface, curves,...) to an organizational unit. Groups may be part of the definition of other groups. This will be the first test of this functionality.
- **Production Models** are included in this round of testing in addition to the synthetic models for the above capabilities.

1.2 General test instructions for this round

The general procedures for communication of models and statistics are outlined in a separate document 'General Testing Instructions'. The general instructions can be retrieved from CAx Implementor Forum web sites. The latest version is v1.6, dated June 2008.

1.3 Preliminary testing schedule

The following schedule has been agreed on for Round 23J:

Date	Action
December 11, 2008 (Thu)	Test Suite available / 1st CAx Implementor Forum conference call
ASAP	Production Models released
January 12, 2009 (Mon)	Initial STEP files and native stats due
February 6 (Fri)	STEP files and native stats frozen
February 11 (Wed)	Target stats due / 2nd conference call
March 6 (Fri)	Target stats frozen
March 11 (Wed)	Pre-release of final stats / 3rd conference call
March 17 (Tue)	Review meeting for test round
March 18-19 (Wed – Thu)	CAx Implementor Forum meeting, North Charleston, SC, USA

The CAx-IF meeting will take place in conjunction with the PDES, Inc. Spring Offsite meeting and LOTAR workshop.

1.4 Copyrights on test cases

Not all of the production test cases which were provided by the PDES, Inc. and ProSTEP iViP member companies are fully released for any purpose. The least common denominator is that the test cases can be freely distributed among the ProSTEP iViP / PDES, Inc. Round Table participants and can be used for any purposes that are related to CAx-IF testing (i.e. testing, documentation of testing efforts), as long as a reference to the originating company is made.

The test cases must not be used for any purposes other than the CAx-IF testing or outside of PDES, Inc. and ProSTEP iViP.

2.0 Synthetic test case specifications

2.1 Model GD7: GD&T Polyline Presentation

2.1.1 Motivation

Geometric and Dimensional Tolerances are required for a number of business use cases in the context of STEP data exchange. Among others, they are a prerequisite for long-term data archiving. In addition, the GD&T data can be used to drive downstream applications such as coordinate measuring and manufacturing.

For documentation and long-term archiving purposes, the Polyline Presentation approach was suggested and developed by the LOTAR project group. It presents the GD&T information within the 3D model, broken down into lines and arcs, so that it looks exactly as generated by the native system.

2.1.2 Approach

The approach to be used is described in the “Draft Recommended Practices for GD&T Polyline Presentation”, that can be found in the member area of the CAx-IF web sites. Polyline annotations and the definition of saved views (the latter not mandatory) are in scope for this round of testing; the exchange of annotation validation properties is out of scope for this round. This corresponds with what has been tested in the LOTAR pilot.

2.1.3 Testing Instructions

In Round 23J, a number of user-provided models containing GD&T information will be provided for testing. They will be made available on the CAx-IF Document Sharing Web Site.

Note that **all** CAx-IF participants, even / especially those who have not implemented Polyline Presentation, are asked to import the resulting STEP files. Since the annotations basically are broken down into wireframe geometry (which has been tested in R20J-R21J), the question is to what extent the information can be found by arbitrary CAD systems.

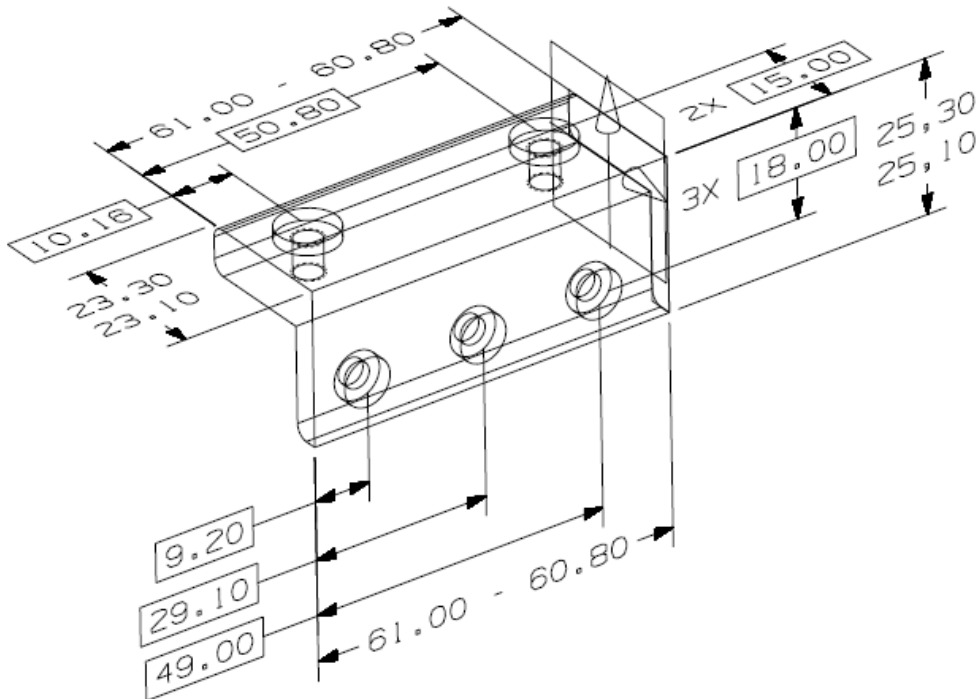


Figure 1: Example for a user-provided part with tolerances.

In addition to the Polyline presentation as tested in Round22J, the following additional functionality may be included in the test files provided for this round of testing, as far as it has been implemented by the CAx-IF participants and is described in the Recommended Practices:

- Definition of “saved views” – if supported, include at least one saved view in the model, which contains a subset of annotations in the file, and provides a pre-defined position of the model in the design space. It is recommended to provide a screenshot illustrating the model position and annotation subset, in addition to the screenshot showing the overall model layout.
- Cross-highlighting of annotations and annotated shape – if supported, include in the STEP file the information necessary to maintain the association between annotations and the annotated shape elements in a way, that after import, when highlighting an annotation, the shape elements annotated by it are highlighted, too, and vice versa.

2.1.3.1 Test Models

Test Models for CATIA V5 and UG NX are available in the CAx-IF File Repository under “Round 22J > GDandT Test Models”. Additional test models containing GD&T information for other CAD systems are welcome.

2.1.3.2 Statistics

For each STEP file submitted for the GD7 model, vendors must submit the corresponding native statistics to the online statistics and results database, CAESAR. To do so, log in to

CAESAR, go to “Test Rounds > R23J Test Cases > GD7 Data Sheet”, and either fill in the web form, or upload a comma-delimited file (.csv) with the data as listed below.

Note that in order to count the GD&T elements for the statistics, as per agreement during the Round 22J Review Meeting, the names of the `geometric_curve_sets` shall be considered.

See Figure 3 in the Recommended Practices for GD&T Polyline Presentation for details.

column name	description
model	The name of the test model, see test models table
system_n	The system code of the CAD system creating the STEP file
system_t	The system code of the CAD system importing the STEP file. For native stats, enter 'stp'
unit	The unit the model is designed in
volume	Total volume of all solids
area	Total surface area of all solids
cx	Centroid of all solids
cy	
cz	
dim_found	The number of dimensions processed
datum_found	The number of datums processed
tol_processed	The number of tolerances processed
saved_view	The name of the Saved View which is the basis for the view-related statistics
view_annot	The number of annotations included in the specified saved view.
view_pos	pass/fail, whether the position stored for the Saved View could be restored successfully. It is recommended to provide a screenshot illustrating the position of the model for the specified view.
highlight	all/partial/none - whether the cross-highlighting for annotations and annotated shape elements works correctly
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 Model GD8: GD&T Representation & Semantic Presentation

2.2.1 Motivation

Geometric and Dimensional Tolerances are required for a number of business use cases in the context of STEP data exchange. Among others, they are a prerequisite for long-term data archiving. In addition, the GD&T data can be used to drive downstream applications such as coordinate measuring and manufacturing.

In contrast to the Polyline approach, Semantic Presentation relies on the GD&T Representation capability to define the contents to be displayed, and supplements these with positioning and styling information to create the appropriate notes in 3D space. (Associative) 3D Annotations including Unicode characters have been tested in previous rounds in preparation for this test case.

2.2.2 Approach

Recommended Practices for the implementation of GD&T Representation, and initial guidelines for GD&T (semantic) Presentation, can be found on the WikiSTEP page, see <http://wikistep.org/>. In addition, a draft Usage Guide outlining the interdependencies between the GD&T elements is available in the CAX-IF member area under “Information on Round 22J of Testing.” The actual approach and scope of the GD8 test case is that of the NIST pilot.

2.2.3 Testing Instructions

In Round 23J, a number of user-provided models containing GD&T information will be provided for testing. They will be made available on the CAX-IF Document Sharing Web Site.

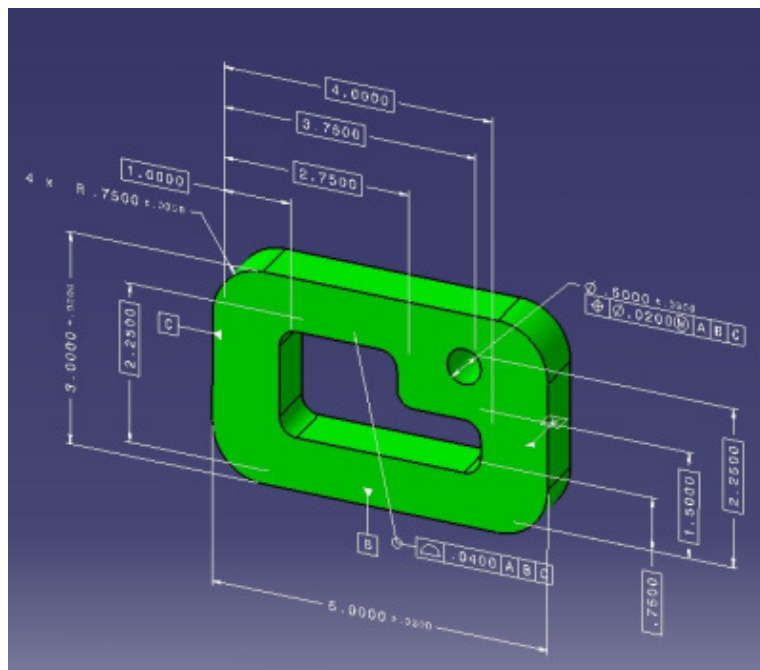


Figure 2: Example for a user-provided part with tolerances.

In addition to the Semantic presentation as tested in Round22J, the following additional functionality may be included in the test files provided for this round of testing, as far as it has been implemented by the CAX-IF participants and is described in the Recommended Practices:

- Definition of “saved views” – if supported, include at least one saved view in the model, which contains a subset of annotations in the file, and provides a pre-defined position of the model in the design space. It is recommended to provide a screenshot illustrating the model position and annotation subset, in addition to the screenshot showing the overall model layout.
- Cross-highlighting of annotations and annotated shape – if supported, include in the STEP file the information necessary to maintain the association between annotations and the annotated shape elements in a way, that after import, when highlighting an annotation, the shape elements annotated by it are highlighted, too, and vice versa.

2.2.3.1 Test Models

Test Models for CATIA V5 and UG NX are available in the CAX-IF File Repository under “Round 22J > GDandT Test Models”. Test Models containing GD&T information for other CAD systems are welcome.

2.2.3.2 Statistics

For each STEP file submitted for the GD8 model, vendors must submit the corresponding native statistics to the online statistics and results database, CAESAR. To do so, log in to CAESAR, go to “Test Rounds > R22J Test Cases > GD8 Data Sheet”, and either fill in the web form, or upload a comma-delimited file (.csv) with the data as listed below.

Note that in order to count the GD&T elements for the statistics, as per agreement during the Round 22J Review Meeting, the actual STEP entity types (`datum`, `datum_target`,...) shall be considered.

column name	description
model	The name of the test model, see test models table
system_n	The system code of the CAD system creating the STEP file
system_t	The system code of the CAD system importing the STEP file. For native stats, enter 'stp'
unit	The unit the model is designed in
volume	Total volume of all solids
area	Total surface area of all solids
cx	Centroid of all solids
cy	
cz	

dim_found	The number of dimensions processed
datum_found	The number of datums processed
tol_processed	The number of tolerances processed
saved_view	The name of the Saved View which is the basis for the view-related statistics
view_annot	The number of annotations included in the specified saved view.
view_pos	pass/fail, whether the position stored for the Saved View could be restored successfully. It is recommended to provide a screenshot illustrating the position of the model for the specified view.
highlight	all/partial/none - whether the cross-highlighting for annotations and annotated shape elements works correctly
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 Model UD1: User Defined Attributes

2.3.1 Motivation

Most CAD systems allow the user to add user-defined attributes in the form of key-value pairs to a part or shape. These carry information which can not be derived from the geometry, such as material costs, but is of relevance to downstream processes or for archiving purposes. In Round 23J, this capability will be tested for the first time.

2.3.2 Approach

The approach to be used to transfer the user-defined attributes is described in the Draft Recommended Practices for User Defined Attributes, which are available in the member area of the CAx-IF homepages under “Information on Round 23J of Testing”.

User Defined Attributes can be attached to either a single part, an instance of an assembly component or a geometric element of its shape. Each attribute can be descriptive (i.e. the value is a text string) or numeric (with and without unit).

2.3.3 Testing Instructions

The User Defined Attributes shall be tested using the well-known as AS1 model.

2.3.3.1 Construction of the model

The following attributes are suggested values for use in the UD1 test. **Note** that the locations where the attributes should be attached (solid/surface, instance, part/product) are proposals,

not mandatory. All attribute types should be contained in the model and attached in the file structure where meaningful for the originating system.

- To one of the faces of the 'plate' part, add a descriptive attribute (see section 5.1 in the Recommended Practices):
 - Name: 'Surface Finish'
 - Description: 'Anodize per specification MIL-A-8625, Type I'
- To the two instances of the L-bracket assembly, add a value attribute each (see section 5.2 in the Recommended Practices):
 - Name: 'asm_step'
 - Values: 1. and 2. respectively
- To the 'plate' part, add a measure attribute (see section 5.3 in the Recommended Practices):
 - Name: 'weight'
 - Unit: kilograms (kg) or pounds (lbs)
 - Value: <calculated weight of component preferred but generic value can be provided if necessary>

It is allowed to add additional information to each of the attributes (see section 5.4 in the Recommended Practices).

2.3.3.2 Statistics

For each STEP file submitted for the UD1 model, vendors must submit the corresponding native statistics to the online statistics and results database, CAESAR. To do so, log in to CAESAR, go to "Test Rounds > R23J Test Cases > UD1 Data Sheet", and either fill in the web form, or upload a comma-delimited file (.csv) with the following data:

column name	description
model	The name of the test model, here: 'UD1'
system_n	The system code of the CAD system creating the STEP file
system_t	The system code of the CAD system importing the STEP file. For native stats, enter 'stp'
face_attr	pass/fail, has the User Defined Attributed at the solid/surface level been processed correctly?
instance_attr	pass/fail, has the User Defined Attribute at the assembly component instance level been processed correctly?

part_attr	pass/fail, has the User Defined Attribute at the part/product level been processed correctly?
valid_attr	pass/fail, is the instantiation of the user defined attributes as per the recommended practices?
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 Model SG1: Supplemental Geometry

2.4.1 Motivation

When designing a part in a CAD system, geometrical elements are often created that do not belong to the actual, i.e. manufactured, shape of the part, but are used to either create other geometric shapes, or to relate additional information about the part.

The main business applications for this capability are the transfer of reference elements for PMI information, such as cutting planes and center lines, and named axis placements as tool targets for machining applications.

In Round 23J, this capability will be tested for the first time.

2.4.2 Approach

The approach to be used is described in the Draft Recommended Practices for Supplemental Geometry, which are available in the member area of the CAx-IF homepages under "Information on Round 23J of Testing".

2.4.3 Testing Instructions

The transfer of construction geometry will be tested using a simple test model (see below). The following supplemental geometry elements shall be included:

- The center line for the cylindrical hole through the part
- A reference plane at the upper end of the truncated cone, its face normal parallel to the center line of the hole
- Four named axis placements along the outer top edge of the part

Note that when comparing the results, the supplemental geometry elements may look differently in different CAD systems (e.g. bound vs. unbound plane). The essential criteria is, are they correctly identified as supplemental geometry elements in the target system.

2.4.3.1 Construction of the SG1 model

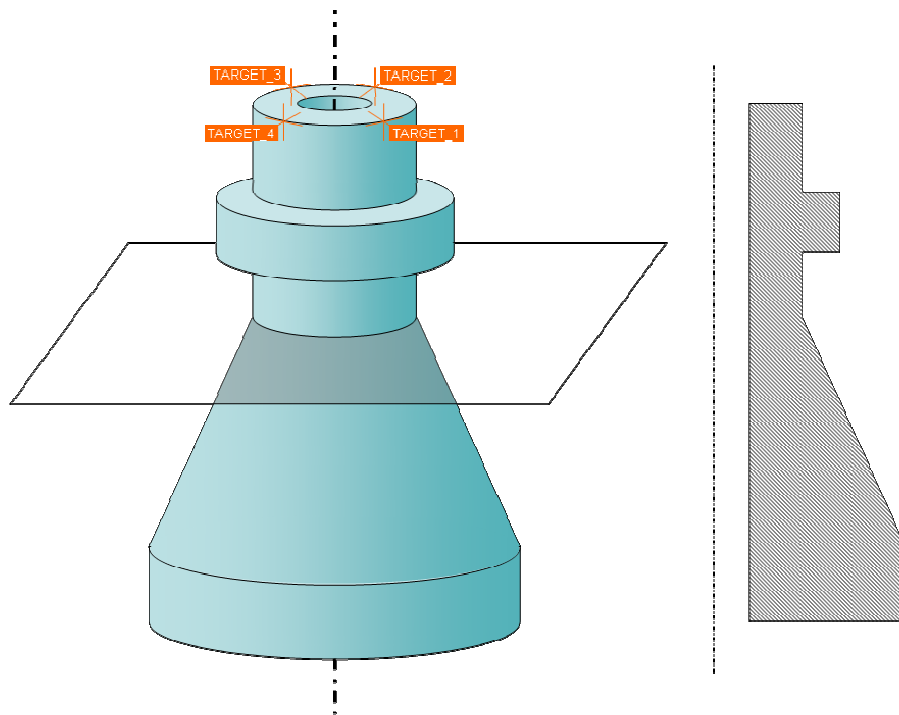


Figure 2: Sketch of the SG1 model.

Dimensions for the part are arbitrary; suggested values are height 15cm, base diameter 10cm and tip diameter 5cm.

2.4.3.2 Statistics

For each STEP file submitted for the SG1 model, vendors must submit the corresponding native statistics to the online statistics and results database, CAESAR. To do so, log in to CAESAR, go to “Test Rounds > R23J Test Cases > SG1 Data Sheet”, and either fill in the web form, or upload a comma-delimited file (.csv) with the following data:

column name	description
model	The name of the test model, here: 'SG1'
system_n	The system code of the CAD system creating the STEP file
system_t	The system code of the CAD system importing the STEP file. For native stats, enter 'stp'.
unit	The unit the model is designed in
volume	Total volume of all solids
area	Total surface area of all solids

cx	Centroid of all solids
cy	
cz	
center_line	pass/fail, if the center line was found in the STEP file and correctly displayed in the CAD system.
ref_plane	pass/fail, if the reference plane was found in the STEP file and correctly displayed in the CAD system.
tool_tgt	pass/fail, if the named axis placements (tool targets) were found in the STEP file and correctly displayed in the CAD system or identified in the product structure.
valid_sup_geo	pass/fail, whether the implementation of the supplemental geometry elements in the STEP file is as defined in the recommended practices.
date	The date when the statistics were last updated (will be filled in automatically)
issues	A short statement on issues with the file

2.5 Model DM1: Density and Material name

2.5.1 Motivation

Usually, transfer of different materials is not included in geometry exchange via STEP. On export, a consistent density for all parts is assumed.

This capability provides an approach to transfer density values (as real values) and material names (as strings). Using the density information, the center of geometry of a model can be distinguished from the center of gravity. The material name can be used to receive further information about that material, e.g. from a database.

Though supported by only a limited number of STEP processors, it was agreed to add this test case to the STEP file library, since it provides capabilities of interest for interoperability testing with PDM and CAE processors.

2.5.2 Approach

Material Identification shall be included by every system exporting this model for the STEP file library. The “Material as Product” approach to be used is described in section 5.1 of the “Recommended Practices for Material Identification and Density”, dated 07-12-2005, available on the “Joint Testing Information” section of the CAx-IF homepages. Density information may shall included as either a general property of the material (section 5.2).

Note that this information shall always be attached at the part/product level in the STEP file, and not at the solid/surface level, to ensure that all importing systems are able to find the information.

2.5.3 Testing Instructions

Shape

The test model is based on a sub-assembly of the well-known AS1 model, i.e. one of the “L-bracket assemblies”. Modeling instructions for the AS1 model and its subassemblies can be found in the Round 8J Test Suite document, which can be found on the “Joint Testing Information” pages of the CAx-IF web sites, dated 09-05-2001.

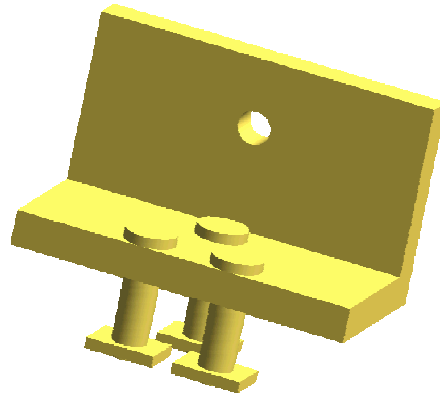


Figure 6: Shape of the DM1 model

This sub-assembly is composed of three individual parts, each of which should be assigned a different density and material name. Suggested values are:

Part	Density	Material name
L-bracket	.160 lb/cu in	AMS 4928; Titanium 6-4
Bolt	.285 lb/cu in	AMS 5613; Greek Ascology
Nut	.297 lb/cu in	AMS 5662; Inconel 718

Due to the different densities, the center of gravity will differ from the center of geometry. These coordinates, besides the transferred values for density and material, will be the focus of this test.

2.5.3.1 Statistics

For each STEP file submitted for the DM1 model, vendors must submit the corresponding native statistics to the online statistics and results database, CAESAR. To do so, log in to CAESAR, go to “Test Rounds > R23J Test Cases > DM1 Data Sheet”, and either fill in the web form, or upload a comma-delimited file (.csv) with the following data:

column name	description
model	The name of the test model, here: 'DM1'
system_n	The system code of the CAD system creating the STEP file

system_t	The system code of the CAD system importing the STEP file. For native stats, enter 'stp'
unit	The unit the model is designed in
volume	Total volume of all solids
area	Total surface area of all solids
cx	Centroid of all solids
cy	
cz	
gravx	Center of gravity for all solids
gravy	
gravz	
density	Density value for the L-Bracket part
material	Material name for the L-Bracket part
date	The date when the statistics were last updated (will be filled in automatically)
issues	A short statement on issues with the file

3.0 Production models: PM20

3.1 Motivation

In an attempt to test the STEP processors on real world models, the CAx Implementor Forum will be testing production parts in this round and future rounds of CAx-IF testing. These production models are characteristic for components and assemblies that are encountered in the aerospace and automotive industries. PDES, Inc. and ProSTEP iViP member companies and vendors have supplied these models.

3.2 Approach

Testing of Production Models focuses mainly on data quality, not on specific functionalities. Assemblies should therefore be exported as a single STEP file. The file format should be either AP214-IS or AP203e2. In order to support quality validation of the Production Model exchange, all vendors shall include the maximum level of Validation Properties they support. In addition, since Round18J, the native and target statistics will include ValProps.

All source system native models and STEP files may be analyzed for data quality by the "CADIQ" developers. STEP syntax and structure will be checked by the CAx-IF facilitators. In order to enable an end-to-end analysis of the data exchange, all vendors importing Production Model STEP files are asked to submit the resulting target model from their system along with or instead of the target statistics.

3.3 Testing Instructions

The native models as provided by the user companies should be exported to STEP by all participants who maintain a STEP processor for the respective CAD system. The native models will be made available on the CAX-IF File Repository in the member area under “Round 23J > Production Models”.

3.3.1 List of available models

Model name	Stats code	Native System	Remarks
Display Select Panel Assembly	pm21_dis	UG NX	
LCD Front Bezel	pm21_lcd	UG NX	
Stamp	pm21_v5	CATIA V5	STEP file only

3.3.2 Results

For each STEP file submitted for the PM21 model, vendors must submit the corresponding native statistics to the online statistics and results database, CAESAR. To do so, log in to CAESAR, go to “Test Rounds > R23J Test Cases > PM21 Data Sheet”, and either fill in the web form, or upload a comma-delimited file (.csv) with the following data:

column name	description
model	The name of the test model, e.g. 'PM21' (see stats code above)
system_n	The system code of the CAD system creating the STEP file
system_t	The system code of the CAD system importing the STEP file. For native stats, enter 'stp'
unit	The unit the model is designed in
volume	Total volume of all solids
validation_volume	Total volume of all solids as received via the validation property capability
valid_vol	pass/fail, is the instantiation of the validation property 'volume' in the STEP file as per the recommended practices for validation properties?
area	Total surface area of all solids
validation_area	Total surface area of all solids (entire assembly), as received via the validation property capability
valid_area	pass/fail, is the instantiation of the validation property 'area' the STEP file as per the recommended practices for validation properties?
cx	Centroid of all solids
cy	

cz	
validation_cx	Centroid of all solids (entire assembly) as received via the validation property capability
validation_cy	
validation_cz	
valid_cent	pass/fail, is the instantiation of the validation property 'centroid' in the STEP file as per the recommended practices for validation properties?
model_size	model_size is the length of the space diagonal of the 3dimensional bounding box enclosing all entities in the model. The result is the Centroid deviation divided by the model_size
shoveit_ok	pass/fail, indicates whether the model passed comparison of the Extended GVP (i.e. no parts/subassemblies misplaced), or failed.
valid_shoveit	pass/fail, indicates whether the target system considers the implementation of the instance information valid as per the recommended practices
date	The date when the statistics were last updated (will be filled in automatically)
issues	A short statement on issues with the file