INTERIM NASA TECHNICAL STANDARD	**NASA-STD-(I)-0007**
National Aeronautics and Space Administration **Washington, DC 20546-0001**	**Approved: 05-18-2009**

NASA COMPUTER-AIDED DESIGN INTEROPERABILITY

MEASUREMENT SYSTEM IDENTIFICATION:

ENGLISH/METRIC (SI)

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This document represents the technical consensus of the developing group but does not yet have final NASA approval.

DOCUMENT HISTORY LOG

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FOREWORD

This Interim Technical Standard is published by the National Aeronautics and Space Administration (NASA) to provide uniform engineering and technical requirements for processes, procedures, practices, and methods to meet urgent program and project technical needs. This Interim Standard has the consensus of the developing Technical Working Group (TWG) but does not have Agency-wide concurrence required for a NASA technical standard.

This Interim Standard is approved for use by NASA Headquarters and NASA Centers, including Component Facilities and Technical and Service Support Centers.

This Interim Standard establishes the format and content of two-dimensional (2D) drawings and three-dimensional (3D) models so that rework of these products by collaborating organizations is minimized. Adherence to this Interim Standard will result in enhanced interoperability between organizations exchanging Computer-Aided Design (CAD) products.

The scope of this document is not intended to preclude casual, working, or temporary exchange of data that does not result in formal program deliverables.

Requests for information, corrections, or additions to this Interim Standard should be submitted via "Feedback" in the NASA Standards and Technical Assistance Resource Tool at http://standards.nasa.gov.

Original Signed By **05-18-2009**

Michael G. Ryschkewitsch
NASA Chief Engineer

Approval Date

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1. SCOPE

1.1 Purpose

The purpose of this Interim Standard is to establish the format and content of two-dimensional (2D) drawings and three-dimensional (3D) models so that rework of these products by collaborating organizations is minimized. Adherence to this Interim Standard will result in enhanced interoperability between organizations exchanging Computer-Aided Design (CAD) products.

1.2 Applicability

This Interim Standard is applicable to all functions that generate or use 2D drawings and 3D models (hereinafter called CAD products). The need for this Interim Standard was identified by the Constellation Program to support near-term milestones and is, therefore, centered around the current CAD applications used throughout the Constellation Program. This Interim Standard, therefore, reflects the use of Parametric Technology Corporation's (PTC's) Pro/ENGINEER Wildfire 3.0 as the application into which CAD products will be integrated. The Interim Standard will be expanded at a later date to bring the benefit of interoperability to the rest of the Agency by addressing the data exchange between other CAD applications in more detail.

Some requirements in this Interim Standard are not specific to any CAD application. In this case, suppliers and recipients of CAD products are to adhere to these requirements in order to benefit from improved interoperability.

This Interim Standard is approved for use by NASA Headquarters and NASA Centers, including Component Facilities and Technical and Service Support Centers, and may be cited in contract, program, and other Agency documents as a technical requirement. This Interim Standard may also apply to the Jet Propulsion Laboratory or to other contractors, grant recipients, or parties to agreements only to the extent specified or referenced in their contracts, grants, or agreements.

Requirements are indicated by the word "shall." Explanatory or guidance text is indicated in italics beginning in section 4. Rationale for requirements and guidance is provided in Appendix A.

1.3 Tailoring

Tailoring of this Interim Standard for application to a specific program or project shall be formally documented as part of program or project requirements and approved by the Technical Authority.

2. APPLICABLE DOCUMENTS

None.

3. ACRONYMS AND DEFINITIONS

3.1 Acronyms and Abbreviations

2D	2-dimensional
3D	3-dimensional
BOM	Bill of Materials
CAD	Computer-Aided Design
ICD	Interface Control Document
IGES	Initial Graphics Exchange Specification
in	inch
JSC	Johnson Space Center
OML	outer mold line
OPRD	Office of Primary Responsibility Designee
NASA	National Aeronautics and Space Administration
NTSP	NASA Technical Standards Program
mm	millimeter
PLM	Product Lifecycle Management
PTC	Parametric Technology Corporation
STEP	Standard for the Exchange of Product model data (ISO 10303)
TWG	Technical Working Group

3.2 Definitions

Associative: Derived from and linked to the CAD object.

Companion File: Related CAD objects created in order to manage CAD data (i.e. Shrinkwrap™ files, skeletons, envelopes, related assemblies, etc.) or provide additional part information (modified models, manufacturing models, stress and analysis models, etc.).

Drawing: A 2D representation that may define a part or assembly.

External References: A reference within a CAD object that creates dependencies between itself and another CAD object(s).

Inheritance Feature: Allows one-way associative propagation of geometry and feature data from a reference part to a target part.

Interoperability: The ability of two or more entities (organizations, users) to exchange and utilize CAD data.

Layer Nesting: Layers embedded within another layer.

Model: A 3D representation with geometry necessary to define a part or assembly.

ModelCHECK: A configurable verification tool integrated within Pro/ENGINEER.

Nominal: The intended dimensional value as designed. Examples of nominal dimensions: 0.257 +0.003/-0.007 (geometry is created at 0.257), 0.250-0.260 (geometry is created at 0.255), 0.260 MAX (geometry is created at 0.260).

Non-solid Items: A zero-mass entity such as a curve, axis, point, plane, or surface.

Parametric: The interdependencies between/within CAD objects (e.g., parts, assemblies, and drawings).

Recipient: The entity that receives the CAD product.

Relations: User-defined equations written between symbolic dimensions and parameters.

Reps: Abbreviation for simplified representations.

Rework: Work performed by the supplier that is to be performed again by the recipient in order to carry out the intended integration activity. By its nature, there is no value to rework.

Simplified Representations: A subset of the parent model that displays fewer details (components, features, etc.) than the complete model.

Supplemental Geometry: Geometric elements included in the drawing to communicate design requirements or intent but not defined in the CAD model.

Supplier: The entity that develops/generates the CAD product.

4. MODELING REQUIREMENTS

4.1 General Modeling Requirements

4.1.1 Models shall not contain frozen features.

4.1.2 Models shall not contain suppressed features unless required by family table generic models or Pro/PROGRAM.

4.1.3 All geometry shall be created at full scale.

4.1.4 All geometry shall be modeled to nominal values.

4.2 Model and Drawing Integrity

4.2.1 All suppliers shall verify their models for adherence to this Interim Standard.

4.2.2 Circular references shall not exist.

4.2.3 "Geom checks" shall not exist.

4.3 Model Accuracy

4.3.1 Models shall use a default absolute accuracy of 0.0005 in (0.0127 mm). *The accuracy may be increased (value decreased), if geometry failures result from the default value.*

4.3.2 After changing the accuracy, the model shall regenerate without failure.

4.3.3 Relative accuracy shall not be used.

4.4 Feature Modification/Verification

4.4.1 *If a model is modified, the next higher level assembly is to be regenerated to ensure that no failures have occurred.*

4.4.2 Drawing and companion files shall be regenerated without failures.

4.5 External References

4.5.1 Only the following external references shall be allowed:

a. Copy geometry feature.
b. External copy geometry feature.
c. Merge/Cutout feature.
d. Inheritance feature.
e. Mirror components.
f. Publish geometry feature.

4.5.2 Copy geometry features shall not gather references directly from other copy geometry or external copy geometry features as demonstrated in figure 1, Copy Geometry Practice.

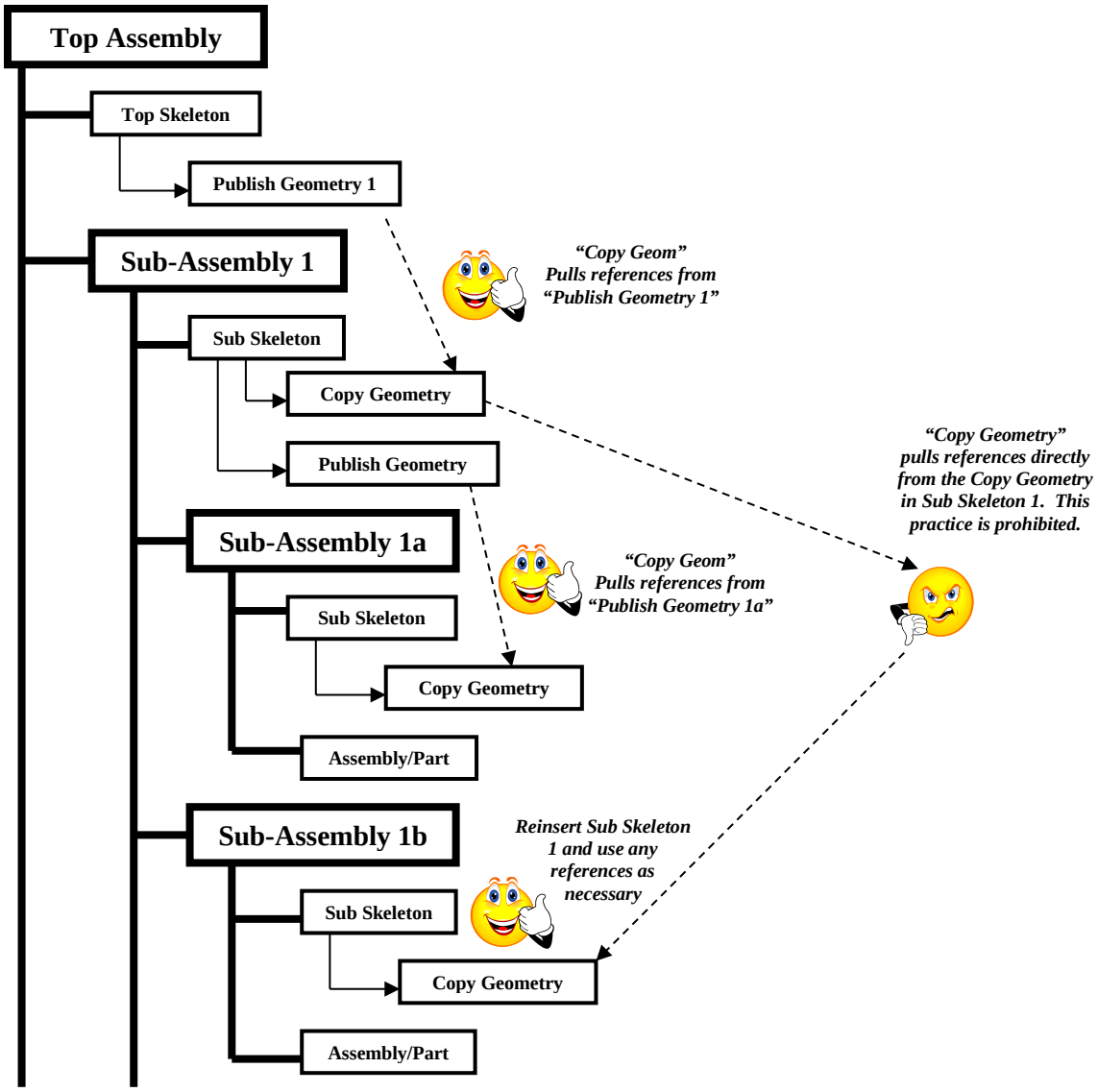

Figure 1—Copy Geometry Practice

4.5.3 All Shrinkwrap™ files shall have "geometry allowed for referencing" set to "None."

4.6 Relations

4.6.1 All values used in relations shall have appropriate units assigned.

4.6.2 Sketcher relations shall not create references external to the sketch.

4.6.3 Assembly level relations shall not be used to control geometry and features of other models.

4.6.4 Model relations shall not have external references.

4.6.5 When relations are used, comments shall be added to describe the equations.

4.6.6 All relations shall be error free.

4.7 Assembly-created Features

4.7.1 Assembly level cuts and holes shall not be set to “automatic update.”

4.7.2 Assembly level cuts and holes shall not be visible at the submodel level (i.e., “show feature properties in submodel” is not to be used).

4.8 Component Placement Constraints

4.8.1 All models shall be fully constrained when placed in an assembly. *Components used in mechanisms may be packaged depending on their function.*

4.8.2 Models shall not be in a “Frozen” or “Suppressed” state; exceptions are family table generic model and Pro/PROGRAM.

4.9 Family Tables

4.9.1 Family tables shall be verified.

4.9.2 Family table fields shall not be populated with an asterisk. The actual value is to be inserted.

4.10 Standard Assembly Simplified Representations

4.10.1 A simplified rep, named EMPTY, shall exist in all assemblies.

4.10.2 The EMPTY rep shall contain no models.

4.10.3 The default rule of the EMPTY rep shall be set to exclude all components.

5. DRAWING REQUIREMENTS

5.1 General Drafting Requirements

5.1.1 All views shall be parametrically associated to the model.

5.1.2 Drawing dimensions shall be parametrically associated to the model, including tabulated part tables and opposite parts.

5.1.3 Drawing dimensions shall not be overridden.

5.1.4 When supplemental geometry is used, there shall be a clear distinction between the supplemental geometry and the design model geometry.

5.1.5 Dimensions that are called out in note form shall be parametric with their associated model dimensions.

6. NAMING CONVENTION

6.1 Filename Uniqueness

6.1.1 Part, assembly, and drawing files shall have different root filenames.

6.1.2 All filenames shall conform to one of the following two rules:

6.1.2.1 A pre-defined numbering scheme not to exceed 28 characters.

6.1.2.2 A filename not to exceed 25 characters followed by an underscore (_) and a 2-character identifier as listed in Appendix A and on the NASA Technical Standards Program (NTSP) website, http://standards.nasa.gov. *This includes common components, (e.g., NAS1351 is to be named NAS1351_XX where XX is the unique identifier).*

6.1.3 When part numbers include characters that are not allowed in the filename, such as a slash (/), an underscore (_) shall be substituted for such characters.

6.1.4 The designer shall assign a description as the common name (e.g., common name, PTC_COMMON_NAME).

6.2 Companion CAD Filenames

6.2.1 If a companion model exists, it shall have the same root filename as the active model plus a suffix. *This companion filename is subject to the character limits defined in 6.1.2.*

6.2.2 The following suffix (table 1, Suffix Description) shall be added after the root filename for the part or assembly when applicable:

Table 1 – Suffix Description

Suffix	Description
-ALT1, -ALT2, etc.	Model created to provide an alternate representation of the active CAD model such as simplified substitutes.
-DEP1, -DEP2, etc.	Deployed models.
-DYN1, -DYN2, etc.	Dynamics models.
-ENV1, -ENV2, etc.	Envelope part models, accurately representing the outer mold line (OML) of the part/assembly as a single part.
-FLAT1, -FLAT2, etc.	Flat state of sheet metal parts or flat pattern for cables.
-GEN1, -GEN2, etc.	Generic of family table part instance.
-HRN1, -HRN2, etc.	Harness subassembly models.
-ICD1, -ICD2, etc.	Interface Control Document (ICD) skeleton models.
-LAY1, -LAY2, etc.	Layout models (not notebooks).
-MASS1, -MASS2, etc.	Mass properties models.
-MVA1, -MVA2, etc.	Model View Assembly facilitates copying geometry from simplified REPS.
-MFG1, -MFG2, etc.	Models created to support manufacturing (typically captures intermediate manufacturing states of the part).
-MOD1, -MOD2, etc.	Modified representation of the part design, such as an as-installed model.
-NOTE1, -NOTE2, etc.	Notebooks.
-PIPE1, -PIPE2, etc.	Pipe subassembly models.
-PKG1, -PKG2, etc.	Package subassembly models.
-REL1, -REL2, etc.	Related model assembly used for creation of drawings for large assemblies, used for installation/integration drawings.
-SIM1, -SIM2, etc.	Model created to support an analysis or simulation.
-SKEL1, -SKEL2, etc.	Skeleton models (not ICDs).
-SWR1, -SWR2, etc.	Shrinkwarp™ models.
-TRD1, -TRD2, etc.	Trade Study models.

7. PARAMETERS

7.1 Parameter Requirements

7.1.1 All user-defined parameters controlling solid geometry shall have units assigned.

7.1.2 Dimensionless parameter (number of places, etc.) shall not have units assigned.

7.1.3 For designated parameters that have the same name across supplier systems, the parameter type shall match to ensure compatibility.

7.1.4 The following parameters in table 2, Required Parameters, shall be in all models and be assigned an appropriate value.

Table 2—Required Parameters

Name	Type	Designate	Description
PART_NUMBER	STRING	REQUIRED	A unique Part Identifier.
DESCRIPTION	STRING	OPTIONAL	Noun Phrase Description used to populate drawing Bill of Materials (BOM) table.
CAGE_CODE	STRING	OPTIONAL	Unique Commercial and Government Entity Identifier (Data Originator).

7.1.5 If a material parameter is used, regardless of name, it shall be of type “string.”

7.1.6 If a weight/mass parameter is used, regardless of name, it shall be of type “real.”

7.1.7 If a description parameter is used, regardless of name, it shall be of type “string.”

7.1.8 If a fracture-critical parameter is used, regardless of name, it shall be of type “yes-no” (Boolean).

7.1.9 When program-specific parameters are required, parameter names, types, values, and designations shall match for both supplier and recipient (e.g., Name=“life_cycle,” Type=“string,” Value=“supplier,” and Designate=“yes”).

8. LAYERS

8.1 Standard Layers

8.1.1 All models (parts, assemblies and drawings as indicated) shall use the following standard layers as defined in table 3, Standard Layers. It is permissible to omit layers that are not used.

8.1.2 Standard layers shall not be renamed.

8.1.3 Standard layers shall be populated with entities as defined in table 3.

8.1.4 Additional layers shall have a prefix of "99_" followed by a user-defined description. (e.g. 99_CONSTRUCT_DTMS).

8.1.5 Layer nesting shall not be allowed in the standard layers. *Additional layers are to be kept to a minimum.*

8.2 Layer Usage

8.2.1 All non-solid items shall be placed on a layer.

8.2.2 Features shall reside on one layer only.

8.2.3 Visibility of components or features shall be controlled by layers *(i.e., do not use HIDE or UNHIDE in the model tree).*

8.2.4 Components shall not be placed on any layer with the exception of skeleton components.

8.2.5 Layers of models and drawings shall have their status saved such that the appearance is retained.

8.2.6 All drawings shall be controlled by layers at the drawing level, not at the model level.

8.2.7 All part and assembly layers except for 00_VISIBLE_NONSOLIDS shall be set to "hidden."

Table 3—Standard Layers

Layer Type	Layer Name	Layer Content
DRW	00_DRAFT_ENTITIES	Drawing Draft Entities.
PRT, ASM	00_VISIBLE_NONSOLIDS	Nonsolid entities necessary for design intent; do not hide.
PRT, ASM	01_PLANES	Created Planes.
PRT, ASM	01_PLANES_DEF	Default Planes.
PRT, ASM	01_PLANES_XSEC	Cross Section Planes.
PRT, ASM	02_CSYS	Created Csys.
PRT, ASM	02_CSYS_CG	Csys located at Center of Gravity.
PRT, ASM	02_CSYS_DEF	Default Csys.
PRT, ASM	02_CSYS_INTEGRATION	Csys used for Integration.
PRT, ASM	03_AXES	Created Axes.
PRT, ASM	03_AXES_DEF	Default Axes.
PRT, ASM	04_POINTS	Created Points.
PRT, ASM	04_POINTS_CG	Points located at Center of Gravity.
PRT, ASM	05_CURVES	Curve features.
PRT, ASM	05_CURVES_SKETCHED	Sketched features.
PRT, ASM	06_GTOLS	Geometric Tolerance Datums, Axes.
PRT, ASM	07_SURFACES	Surface and Quilt features.
PRT, ASM	08_SKELETONS	Skeleton components.
PRT, ASM	09_GEOM_COPY_EXT	External Copy Geometry features.
PRT, ASM	09_GEOM_COPY_INT	Internal Copy Geometry features.
PRT, ASM	09_GEOM_PUBLISHED	Publish Geometry features.
PRT, ASM	10_IMPORTED_GEOM	Imported Geometry.
PRT, ASM	11_COSMETICS	All Cosmetic features except threads.
PRT, ASM	11_COSMETIC_THREADS	Cosmetic threads.
PRT, ASM	12_PIPELINES	Piping features and references.
PRT, ASM	13_CABLES	Cable features and references.
PRT, ASM	14_ANNOTATION_FEATURES	Model Notes, Hole Callouts, Annotations, and Symbols.
PRT, ASM	15_GENERAL_NOTES	Everything else.

9. CAD SYSTEM CONFIGURATION

9.1 Configuration Settings

9.1.1 The configuration options shall be set per table 4, Configuration Settings.

Table 4—Configuration Settings

Configuration Option	Setting	Location
default_abs_accuracy	0.0005 in / 0.0127 mm	Configuration file.
enable_absolute_accuracy	Yes	Configuration file.
multiple_skeletons_allowed	Yes	Configuration file.
show_geom_checks_on_creation	Yes	Configuration file.
intf_in_external_accuracy	Yes	Configuration file.
tol_mode	Nominal	Configuration file.
tol_display	Yes	Configuration file.
step_export_format	TBD (in summer 2009)	Configuration file.
ignore_model_layer_status	Yes	Drawing setup file.
draw_layer_overrides_model	No	Drawing setup file.

10. DATA EXCHANGE

10.1 General Data Exchange Requirements

10.1.1 The supplier shall open and verify the intended data set prior to sharing and verify all references are included.

10.2 Translated File Import/Export Data Exchange Requirements (Standard for the Exchange of Product (STEP), Initial Graphics Exchange Specification (IGES), etc.).

10.2.1 Models shall be verified to contain fully translated solid geometry, i.e., B-rep, analytical geometry, no reversed surfaces. *Verification may be done using mass properties analysis.*

10.2.2 Assemblies shall maintain the original assembly hierarchy, (i.e., no “flat files” or “one-level assemblies”).

11. ADDITIONAL RESOURCES

The following sample files, which comply with this Interim Standard, are available on the NTSP website at http://standards.nasa.gov under NASA-STD-(I)-0007, Computer-Aided Design Interoperability. These files are maintained by the NASA CAD Interoperability TWG and are provided as an aid in complying with this Interim Standard.

11.1 CAD Start Part Files

11.1.1 nasa_startpart_English

11.1.2 nasa_startpart_metric

11.2 CAD Start Assembly Files

11.2.1 nasa_startassy_English

11.2.2 nasa_startassy_metric

11.3 ModelCHECK Files

11.3.1 BIG_I_checks.mch

11.3.2 condition.mcc

11.3.3 config_init.mc

11.3.4 BIG_I_inch.mcn

11.3.5 BIG_I_mm.mcn

11.3.6 setconf.mcc

11.3.7 BIG_I_all.mcs

11.3.8 BIG_I_in.mcs

11.3.9 BIG_I_mm.mcs

11.3.10 BIG_I_status.mcq

11.3.11 mc_regen.mcr

11.3.12 user_words

11.3.13 words

APPENDIX A

REQUIREMENTS RATIONALE

A.1 Purpose and/or Scope

The purpose of this appendix is to provide rationale for requirements and guidance stated in the text.

Requirement Paragraph	Rationale
4.1.1	Frozen features often cause incomplete objects and unresolved dependencies.
4.1.2	Suppressed features often cause incomplete objects and unresolved dependencies.
4.1.3	To ensure all CAD models represent their intended geometry and mass and accurately demonstrate interferences and clearances within higher assemblies. In addition, other requirements contained in this Interim Standard are reliant on this (e.g. “drawing dimensions shall be associative”).
4.1.4	For interference and clearance assurance and for a global tolerance independent environment. To allow any recipient of a CAD model to understand the design intent of the originator. To bring consistency to manufacturing.
4.2.1	To improve geometry and integrity.
4.2.2	To improve geometry and integrity.
4.2.3	To improve geometry and integrity.
4.3.1	Models with mismatching accuracy are prone to failure.
4.3.2	Models with mismatching accuracy are prone to failure.
4.3.3	Models with mismatching accuracy are prone to failure.
4.4.1	Effects of modifications are complex and require verification downstream. This prevents unintentional propagation of failures to the recipient.
4.4.2	The full extent of a modification requires verification in all related objects. This prevents unintentional propagation of failures to the recipient.
4.5.1	External references can be complex and difficult to troubleshoot when failures occur. They can also cause unexpected and unwanted changes to occur. Working with and managing external references is easier by using only these feature types.

4.5.2	To reduce rework by eliminating a Pro/ENGINEER process that has negative side effects in the model maintenance phases of a Pro/ENGINEER model created using a top-down design methodology.
4.5.3	Creating references to Shrinkwrap™ geometry results in confusing, undesired dependencies, circular references, and a number of other time-consuming issues.
4.6.1	Avoids unit conversion errors.
4.6.2	The location of the relation is hidden from the user. Pro/ENGINEER reports that a feature is controlled by a relation but does not indicate where to find it.
4.6.3	Avoids creation of unwanted external references.
4.6.4	Avoids creation of unwanted external references.
4.6.5	Shares intent of relations with downstream users.
4.6.6	Avoids regeneration failures.
4.7.1	Auto Update will often include components that are not actually cut. These extra components cause longer regeneration times, degrade performance, and give “cut is completely outside of model” warnings.
4.7.2	Assembly cuts cause longer regeneration times, degrade performance, create confusing dependencies, circular references, and a number of other time-consuming issues. Creating assembly cuts and holes visible at the submodel level is an inappropriate and unnecessary use of this functionality.
4.8.1	Under-constrained components can still move in unintended ways. Components in mechanisms will display as under-constrained but are constrained to move in desired ways.
4.8.2	Suppressed and Frozen components often cause incomplete objects and unresolved dependencies.
4.9.1	Family Table values are to be verified to ensure geometry integrity.
4.9.2	This is a best practice that, if not used, has potential to cause data migration issues between Product Lifecycle Management (PLM) systems.
4.10.1	An EMPTY simplified Rep allows large assemblies to be opened quickly.
4.10.2	Prevents newly assembled components from automatically being added to the EMPTY Rep.
4.10.3	Prevents newly assembled components from automatically being added to the EMPTY Rep.
5.1.1	Drawing views will update as the model is changed, ensuring accurate display of geometry.
5.1.2	Drawing dimensions and notes will update as the model is changed, ensuring accurate display of dimensional values.
5.1.3	Drawing dimensions and notes will update as the model is changed, ensuring accurate display of dimensional values.
5.1.4	Prevents confusion between solid and supplemental geometry.
5.1.5	Drawing notes will update as the model is changed, ensuring accurate display of dimensional values.
6.1.1	File translation can generate identical names when the extension is dropped.

6.1.2.1	Filenames are limited by software to 31 characters. This leaves room for extended identifiers.
6.1.2.2	Prevents naming conflicts.
6.1.3	Software limitation.
6.1.4	Allows identification of files with cryptic filenames.
6.2.1	Identifies relationships using the same part numbers.
6.2.2	Companion file suffix identifiers indicate what the companion file is accomplishing.
7.1.1	Avoids unit conversion errors.
7.1.2	Improper unit use may cause regeneration errors.
7.1.3	Type mismatch causes data incompatibility issues within PLM systems (e.g, check-in failures).
7.1.4	These parameters support minimum Pro/ENGINEER BOM population compatibility.
7.1.5	Type mismatch causes data incompatibility issues within PLM systems (e.g, check-in failures).
7.1.6	Type mismatch causes data incompatibility issues within PLM systems (e.g, check-in failures).
7.1.7	Type mismatch causes data incompatibility issues within PLM systems (e.g, check-in failures).
7.1.8	Type mismatch causes data incompatibility issues within PLM systems (e.g, check-in failures).
7.1.9	To maintain a constant set of commonly used parameter names as well as a matching type value. Certain program specific parameters may be required by unique PLM systems (e.g., LIFE_CYCLE).
8.1.1	Improves model and drawing portability and reduces layer buildup and complexity.
8.1.2	Renaming layers lead to layer buildup and complexity.
8.1.3	Layers populated with incorrect information lead to layer buildup and complexity.
8.1.4	The use of “99_” forces the additional layers to be sorted after the standard layers in the layer tree making the standard layers easier to use.
8.1.5	Nested layers increase layer complexity and negatively impact their ease of use.
8.1.6	Promotes ease of use by reducing layer buildup.
8.2.1	To manage visibility of non-solid items.
8.2.2	When items are on more than one layer, it is difficult to control their visibility and causes confusion.

8.2.3	Layers are used to control visibility, while UNHIDE or HIDE is for temporary use only. UNHIDE and HIDE cannot be controlled in the drawing independent from the model and vice versa.
8.2.4	Component visibility is to be controlled using Simplified Representations except for Skeletons, which are used for model construction.
8.2.5	This ensures that the drawing appearance (display of datum planes, axes, curves etc.) is the same when another user opens the drawing.
8.2.6	Prevents unintended modification to the model.
8.2.7	Improves general performance and reduces clutter. Visible non-solid entities (e.g. curves used to represent stitching) are needed to convey design intent.
9.1.1	See table 5, Rationale for Configuration Settings.
10.1.1	Prior to sending a CAD data set to a recipient, if all files are not properly contained in the data set, regeneration may fail for the recipient. As a result, ghosts, incomplete objects in missing dependents, will cause the recipients data set to be incomplete.
10.2.1	Solid geometry is required for drawings and other downstream operations. Without good solid geometry, curve and surface features “bleed through” on the drawing.
10.2.2	Assembly hierarchy is required to maintain proper product structure. In addition, assembly hierarchy is required for large assembly management.

Table 5 – Rationale for Configuration Settings

Configuration Setting	Value	Rationale
default_abs_accuracy	0.0005” / 0.0127 mm	Models with mismatching accuracy are prone to failure.
enable_absolute_accuracy	Yes	Models with mismatching accuracy are prone to failure.
multiple_skeletons_allowed	Yes	Prevents model failure when there are multiple skeletons.
show_geom_checks_on_creation	Yes	Forces Pro/ENGINEER to identify geometry checks as features are created. Users must, as a minimum, acknowledge the problem before proceeding.
intf_in_external_accuracy	Yes	This is a hidden setting. This sets the accuracy of the current model to match the accuracy of the imported file. It. prevents imported models from failing during import (as you try and guess the accuracy in the imported part).
tol_display	Yes	Allows 1. The display of design tolerances to allow recipients to understand design intent. 2. Designers/engineers to adjust the tolerances as required by design intent.
tol_mode	Nominal	The default setting is “limits.” If left as “limits,” when tol_display is set to “yes,” all dimensions become limit tolerances, even if no tolerances are specified. Setting tol_mode to “nominal” will only display tolerances as specified by the designer, thus preserving design intent.
step_export_format See Note (a).	No Value Specified at this time.	Facilitates data exchange.

Note (a)

This setting will be addressed in an update of NASA-STD-I-0007 in the summer of 2009. Possible settings are “ap203_is”, “ap214_cd”, “ap202_is”, “203_is_ext”, “ap214_dis”, “ap209_dis”, or “ap214_is”.

APPENDIX B

CAD FILENAME UNIQUE IDENTIFIERS

B.1 Purpose and/or Scope

The purpose of this appendix is to provide identifiers to be used to satisfy CAD filename uniqueness as discussed in paragraph 6.1.2.2. Use the identifier that corresponds to the design authority.

Organizations requiring identifiers should contact the NASA Office of Primary Responsibility Designee (OPRD) for this document found under NASA-STD-(I)-0007 at the NTSP website at http://standards.nasa.gov.

Identifier	Organization
	NASA-developed CAD Products
AR	Ames Research Center
DF	Dryden Flight Research Center
GR	Glenn Research Center
GS	Goddard Space Flight Center
JS	Johnson Space Center
KS	Kennedy Space Center
LR	Langley Research Center
MS	Marshall Space Flight Center
SS	Stennis Space Center
HQ	Headquarters
JP	Jet Propulsion Laboratory
WS	White Sands Test Facility
MA	Michoud Assembly Facility
WF	Wallops Flight Facility
	Industry-developed CAD Products
LM	Lockheed Martin
AT	ATK