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John C. Stennis Space Center
Creo Model Based
Computer-Aided Design Standard

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1.0 INTRODUCTION

1.1 Purpose

This document establishes the requirements and procedures for the creation and modification of Creo Parametric® design software Computer-Aided Design (CAD) models and drawings for use at National Aeronautics and Space Administration (NASA) John C. Stennis Space Center (SSC). The goal is to produce robust models that relate correct dimensional data. This document identifies modeling methodology so that all design groups produce consistent electronic deliverables within SSC.

1.2 Applicability

1.2.1 This document applies to all NASA SSC organizations, contractors, and sub-contractors involved with the preparation of Creo Parametric® models that will be integrated into SSC designs.

1.2.2 All Creo Parametric® objects created for SSC shall adhere to the practices described within this document, except legacy models. Existing Test facility models will be updated to adhere to the practices within this document as resources permit.

1.2.3 The solid geometry in the CAD model shall be the authority, supplemented by Creo-generated 2-Dimensional (2D) drawings.

1.2.4 The drawing shall convey notes, Bill of Materials (BOM), Geometric Dimensioning and Tolerancing (GD&T), and other information necessary to fully define the configuration.

2.0 REFERENCES AND APPLICABLE DOCUMENTS

Referenced and applicable documents shall be the latest version unless otherwise specified.

ASME Y14.100 – 2000, *Engineering Drawing Practices*
ASME Y14.41 – 2003, *Digital Product Definition Data Practices*
ASME Y14.5M – 1994, *Dimensioning and Tolerancing*
NASA-STD-(I)-0007, *NASA Computer-Aided Design Interoperability*
SOI-8080-0007, *Test Site Drawings*
SOI-8080-0015, *Configuration Control of Technical Systems*
SPR 1440.1, *Records Management Program Requirements*
SREF-8070-0001, *Workspace Use and Troubleshooting Desktop Instruction*
SREF-8070-0003, *Library Components Desktop Instruction*
SREF-8070-0004, *Standard Drawing Notes in DDMS Windchill Desktop Instruction*
SREF-8070-0005, *Simplified Representation Creation and Maintenance*

Desktop Instruction
SREF-8070-0006, *Creation of Skeleton Files in DDMS Windchill Desktop Instruction*
SREF-8070-0007, *Standard Symbols Desktop Instruction*
SREF-8070-0008, *Updating Drawing Formats in DDMS Windchill Desktop Instruction*
SREF-8070-0009, *MOD File Creation Desktop Instruction*
SREF-8070-0010, *Shrinkwrap™ Creation in DDMS Windchill Desktop Instruction*
SREF-8070-0011, *Data Transfer Desktop Instruction*
SREF-8070-0012, *Cable Routing in DDMS Windchill Desktop Instruction*
SREF-8070-0013, *Piping Desktop Instruction*
SREF-8070-0014, *Related Companion Files Desktop Instruction*
SREF-8070-0015, *Creating Standard Layers Desktop Instruction*
SREF-8070-0016, *Adding Materials and Generating Mass Properties in DDMS Windchill Desktop Instruction*
SREF-8070-0017, *User-Defined Features (UDFs) Desktop Instruction*
SREF-8070-0018, *Vendor Item Modeling in DDMS Windchill Desktop Instruction*
SREF-8070-0019, *Exploded Views Desktop Instruction*
SREF-8070-0020, *Tables and Tabulated Drawings Desktop Instruction*
SREF-8070-0021, *Bulk Items Desktop Instruction*
SREF-8070-0022, *Verifying 3-Dimensional CAD Models in DDMS Windchill Desktop Instruction*
SREF-8070-0023, *Revising CAD Documents and WT Parts Desktop Instruction*
SREF-8070-0024, *Gathering Released Data in DDMS Windchill Desktop Instruction*
SREF-8070-0026, *Sheet Metal Parts*
SSTD-8070-0001-CONFIG, *SSC Engineering Documentation Standard*
SSTD-8070-0002-CONFIG, *SSC Drafting Manual*
SSTD-8070-0005-CONFIG, *Preparation, Review, Approval and Release of SSC Standards*

3.0 GENERAL REQUIREMENTS

3.1 Default Settings

3.1.1 Start Part

3.1.1.1 English and metric start part files shall be used in the creation of all parts.

3.1.1.2 If English parts are to be modeled, the English start part file shall be used.

3.1.1.3 The use of metric hardware should be avoided where possible.

3.1.1.4 If metric parts are to be modeled, the metric start part file shall be used.

3.1.1.5 These start parts are located on a controlled server and maintained by the Design and Data Management System (DDMS) Administrator. The files are as follows:

a. nasa_startpart_english – This default start part shall be used on all English units designs. This part contains standard views, parameters, relations, datum planes, layers, simplified representations (reps), and a coordinate system (CSYS).

b. nasa_startprt_metric – This default start part shall be used on all metric designs. This part contains all standard views, parameters, relations, datum planes, and a CSYS.

3.1.2 Start Assembly

3.1.2.1 English and metric start assembly files shall be used in the creation of all assemblies.

3.1.2.2 If English assemblies are to be modeled, the English start assembly file shall be used.

3.1.2.3 If metric assemblies are to be modeled, the metric start assembly file shall be used.

3.1.2.4 These start assemblies are located on a controlled server and maintained by the DDMS Administrator. The files are as follows:

a. nasa_startassy_english – This default start assembly shall be used on all English units. This assembly contains all standard views, parameters, relations, datum planes, and a CSYS.

b. nasa_startassy_metric – This is the default start assembly to be used on all metric designs. This assembly contains all standard views, parameters, relations, datum planes, and a CSYS.

3.1.3 Start Drawing/Drawing Template

3.1.3.1 Drawing templates shall be utilized to assist users in quickly beginning a drawing.

3.1.3.2 The following drawing template shall be used:

SSC_F_SIZE_PRO-E_DRW_TEMPLATE

3.1.4 Dimensional and Angular Tolerances

3.1.4.1 All SSC English start parts and assemblies shall have the following default tolerance values:

a. Linear X.X +/- 0.1

b. Linear X.XX +/- .03

c. Linear X.XXX +/- .010

d. Angular X.X +/- 0.1

These values control the default tolerance for all dimensional values whose tolerance is not otherwise explicitly displayed on the drawing.

3.1.4.2 If the entire drawing requires tolerances other than tolerances in the drawing format title block, the new tolerances shall be defined by a general note which will override the format title block tolerances.

3.1.4.3 The drawing title block fields for default tolerances shall not be manually overridden.

3.1.5 Systems of Units

The Creo Parametric® default units of the start parts/assemblies are as follows:

a. English – Inch Pound Mass (lbm) Second for nasa_startpart_english and nasa_startassy_english.

b. Metric – Millimeter Kilogram Second (mmKS) for nasa_startprt_metric and nasa_startasy_metric.

3.1.6 Model Accuracy

The accuracy setting controls the computational accuracy of geometry calculations. The accuracy controls the shortest distance between two (2) points that can be seen as two (2) separate points. Relative accuracy is dependent upon the overall size of the model. Absolute accuracy directly sets a value regardless of the overall part size.

3.1.6.1 SSC default start parts shall use an absolute accuracy of 0.0005 inches for English start parts and 0.0127 millimeters for metric start parts. The default

value may be increased if geometry errors are produced with this value. After changing the accuracy, the model must be regenerated.

NOTE: Increasing the part accuracy can cause longer regeneration times and larger file sizes. The part will have upper and lower bounds on what the accuracy can be. If an accuracy is too tight, a value should be chosen between the given bounds. Because the part is now being calculated at a higher accuracy, features may now fail, causing the designer to go into resolve mode.

3.1.6.2 Relative accuracy shall not be used.

3.1.6.3 The default value may need to be adjusted under the following conditions:

a. When placing a very small feature on a very large part.

b. When intersecting (through merge or cutout) two (2) parts of very different size.

3.2 Materials

3.2.1 Every homogeneous solid body shall have material assigned from the standard materials list. The standard materials list includes the appropriate density.

3.2.2 SREF-8070-0016, *Adding Materials and Generating Mass Properties in DDMS Windchill Desktop Instruction,* may be used for information.

3.3 Mass Properties

3.3.1 CAD models shall have accurate mass properties.

3.3.2 When a component solid body is modeled with sufficient detail, mass properties shall be calculated by Creo Parametric® and updated prior to approval cycle and at intermediate points at the designer's discretion.

3.3.3 Otherwise, mass properties shall be manually entered using the alternate mass properties parameters.

3.3.4 The parameter mass shall contain the correct value for the calculated mass of the part or the manually entered value for the part.

3.3.5 SREF-8070-0016, *Adding Materials and Generating Mass Properties in DDMS Windchill Desktop Instruction*, may be used for information.

3.4 Model and Drawing Integrity

3.4.1 ModelCHECK™ is required prior to checking models/drawings into DDMS.

3.4.2 Errors greater than zero (0) must be resolved and the ModelCHECK™ rerun.

3.4.3 Warnings may be evaluated and addressed as needed.

3.5 General Drafting Requirements

3.5.1 Drawings shall be prepared in accordance with SOI-8080-0007*, SSC Test Site Drawings,* and SSTD-8070-0002-CONFIG, *SSC Drafting Manual.*

3.5.2 The BOMs shall be generated from the model.

3.5.3 When preparing a drawing utilizing Creo Parametric® design software, the following requirements apply:

3.5.3.1 All views and dimensions shall be derived from the model, except the following:

a. Use of sketches of existing content (not part of new designs).

b. Use of sketches for test stand overall views (plans and elevations).

c. Use of drawings of 2D geometry, such as schematics, drawing trees, layouts, and diagrams, which may not be generated from a model.

3.5.3.2 Sketching in model geometry and overriding dimensions shall not be allowed.

3.5.3.3 Applicable notes for the drawing are provided by the design engineer.

3.5.4 See Section 9.0 for detailed Drawing Practices.

4.0 GENERAL MODEL SETUP

4.1 Start Model

4.1.1 All CAD models shall be created using the appropriate and approved SSC start files.

4.1.2 The foundation features shall not be deleted or renamed in any model.

4.1.3 Before beginning a part model, the designer should look for a similar part to copy or

save from, when applicable.

4.1.4 The existing model shall first be evaluated for proper construction before it is copied or saved from.

4.1.5 In order to maintain good downstream usage, the part shall be created at the designer's discretion in the following manner:

4.1.5.1 Primary Features

Primary features comprise the functional features of the part. These features include alignment, range of motion, and housing features. (Examples: Major bores, datum features, interface surfaces, skeletons, etc.)

4.1.5.2 Secondary Features

Secondary features are comprised of features that allow the part to be assembled to other parts, provide clearance with other parts, and reduce the weight of the part. (Examples: Holes, clearance cuts, etc.)

4.1.5.3 Tertiary Features

Tertiary features are comprised of finishing features, such as rounds, fillets, chamfers, and similar details.

4.1.6 When selecting feature references, the designer should select the foundation features when possible.

4.1.7 The order in which the designer should select feature references is datums, then surfaces. Edge references can be used but should be avoided whenever possible.

4.1.8 Features, other than the default datums that are frequently referenced, should be named something logical and meaningful to the designer so that they might be found using the search tool.

4.1.9 External dependencies should be avoided.

4.2 Model File Naming Convention

4.2.1 General Requirements

4.2.1.1 CAD model file names shall not exceed 28 characters. This does not include the period (.) and file extension (e.g., ".asm" or ".prt").

4.2.1.2 CAD model Common Names shall not exceed 60 characters.

4.2.1.3 All alpha characters shall be in CAPS.

4.2.2 Companion File Suffixes

4.2.2.1 If applicable, the appropriate companion file suffix (as defined in Table 1), preceded by an underscore (_) and followed by a three-digit sequential number, shall be added to the Windchill Number (e.g., number_mod001.prt).

4.2.2.2 If a CAD model falls under more than one (1) category as defined in Table 1, all the applicable companion file suffixes shall be added to the Windchill Number in alphabetical order separated by underscores (_) (e.g., number_gen_mod001.prt).

Table 1: Companion File Suffixes

Suffix	Description
ALT	Models created to provide an alternate rep of the active CAD model such as simplified substitutes
CVA	Convenience assemblies (do not use; not for new design)
BLK	Any modeled bulk items, such as spray-on pipe insulation
DGA	Drawing assemblies (do not use; not for new design)
DEP	Deployed models
DYN	Dynamics models (mechanism models)
ENV	Envelope part models, accurately represent the Outer Mold Line (OML) of an assembly as a single part
FLT	Flat state sheet metal parts or flat pattern cable parts
GEN	Generics of family tabled parts or assemblies
HRN	Harness subassembly models
ICD	Interface Control Document (ICD) skeleton models
LAY	Layout models (not notebooks)
MFG	Models created to support manufacturing (typically capture intermediate manufacturing states of a part)

Continued on next page.

Suffix	Description
MOD	Modified reps of a part or assembly design such as an "As-Installed" model
MSS	Mass properties models
MVA	Model View Assembly facilitates copying geometry from simplified reps
NTB	Notebooks
PIP	Pipe subassembly models
PKG	Package subassembly models
PSN	Alternate Position model (e.g., extended, retracted)
REL	Related model assembly used for creation of drawings for large assemblies, used for installation/integration drawings
SIM	Model created to support an analysis or simulation
SKL	Skeleton models used for design (not interface or locator skeletons)
SWR	Shrinkwrap™ models
TRD	Trade Study models

4.3 Initial CAD Identification

4.3.1 Initial Creo Parametric® CAD Name, Common Name, and File Name

The naming requirements in Table 2 shall be used when creating a Creo Parametric® object prior to official SSC numbers being assigned. If SSC numbers have already been assigned, the SSC numbers may be used in accordance with Section 4.5.

Table 2: Creo Parametric® CAD Initial Naming Requirements

Creo Parametric® Field Name	3-Dimensional (3D) CAD Model or 2D CAD Drawing
File Name (User entered)	Utilize Project CAD Naming Convention described in Section 4.2 of this SSTD. **NOTE:** A file suffix .prt, .asm, or .drw is automatically added to the Name field value to create the file name based on the type of file selected.
Common Name (User entered)	Assign name using all CAPS in a format specified by the drawing title requirements of GDRM Section 8.0 Procedures or ASME Y 14.100 Section 5.0. (See Paragraph 6.5.3.)

4.3.2 Initial DDMS CAD Number, Name, and File Name

The naming requirements in Table 3 shall be used when creating a DDMS CAD document prior to official SSC numbers being assigned. If SSC numbers have already been assigned, the SSC numbers may be used in accordance with Section 4.5.

Table 3: DDMS CAD Initial Numbering/Naming Requirements

DDMS Field Name	3D CAD Model or 2D CAD Drawing
Number	Utilize Project CAD Numbering Convention described in Section 4.4 of this SSTD. (**NOTE:** When Creo Parametric® is uploaded to DDMS, this value is automatically mapped from the Creo Parametric® Name (File Name) field including the file suffix .prt, .asm, or .drw.
Name	Assign name using all CAPS in a format specified by the GDRM Section 8.0 Procedures or ASME Y 14.100 Section 5.0. (**NOTE:** When Creo Parametric® is uploaded to DDMS, this field value is automatically mapped from the Creo Parametric® Common Name field.)
File Name	Utilize Project CAD Numbering Convention described in Section 4.4. (**NOTE:** When Creo Parametric® is saved to DDMS workspace, this field value displays the Creo Parametric® Name (File Name) field with file suffix .prt, .asm, or .drw.)

4.4 Initial CAD Model Name

4.4.1 If SSC numbers have already been assigned, the SSC numbers may be used in accordance with Section 4.5. Otherwise, the initial CAD model name should be a logical name that is descriptive of the file.

4.4.2 See Appendix A for an example of project specific initial CAD model naming details and examples.

4.5 Final CAD Model Naming/Numbering Convention

4.5.1 CAD model Windchill Numbers shall be obtained by creating and associating Windchill Type (WT) Parts.

4.5.2 CAD model Windchill Numbers shall be the same as their associated WT Part's Windchill Number plus the appropriate companion file suffix if applicable.

4.5.3 CAD model file names shall be the same as their DDMS Number.

4.5.4 CAD model Common Names shall be the same as their associated WT Part's Common Name.

4.5.5 Pipe line number file name can have four (4) distinct parts – line size, system fluid, line number, material specification – which are defined in 54000-P001, Sheet 1.

Table 4: Naming Convention Examples

	DDMS		
	Number	**File Name**	**Name**
WT Part	SSC00000123	N/A	PANEL, VENT, GN2
Model	SSC00000123	SSC00000123	PANEL, VENT, GN2
Companion Model	SSC00000123_MOD001	SSC00000123_MOD001	PANEL, VENT, GN2, LIFTING CONFIGURATION
Pipe Line Part	PIPE14A0001234	N/A	1.5IN-GN-14A1234-BCK1
Pipe Model	PIPE14A0001234	1_5IN-GN-14A1234-BCK1	1-1/2”-GN-14A1234-BCK1

4.6 Vendor File Naming Convention

4.6.1 Vendor CAD model Windchill Numbers and Common Names shall be made to conform to the naming convention defined above with the following differences:

a. The vendor’s cage code followed by a dash (-) then the vendor’s part number shall be used as the Windchill Number.

b. An underscore (_) shall be used in place of vendor part number characters that are not allowed.

c. If the vendor does not have a cage code, the first five (5) characters of the vendor’s name shall be used instead.

d. If the vendor model is an assembly, its part and subassembly file names shall receive sequential dash numbers preceded by a dash (-) as shown in Figure 1.

e. If this file naming convention results in a file name exceeding 28 characters, the file name shall end in SS, which is the NASA file name uniqueness identifier at SSC. This is preceded by an underscore (_). See Figure 1.

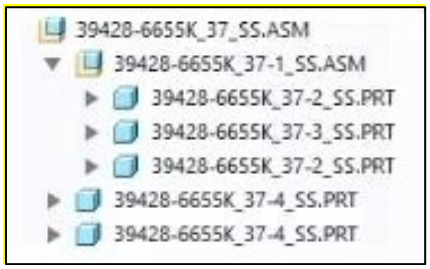

Figure 1: Vendor File Naming

4.7 Industry- or Government-Specified File Naming Convention

Industry- or Government-specified CAD model Windchill Numbers shall be made to conform to the naming convention defined above with the following differences:

a. The part number specified by the document and any applicable dash numbers shall be used as the CAD model Windchill Number. (For examples, see Figure 2.)

b. An underscore (_) shall be used in place of specification number characters that are not allowed.

c. If the industry- or government-specified model is an assembly, its parts and subassembly file names shall receive sequential dash numbers preceded by a dash (-) as shown above in Figure 1.

d. The CAD model file name shall end in SS, which is the NASA file name uniqueness identifier at SSC. This is preceded by an underscore (_).

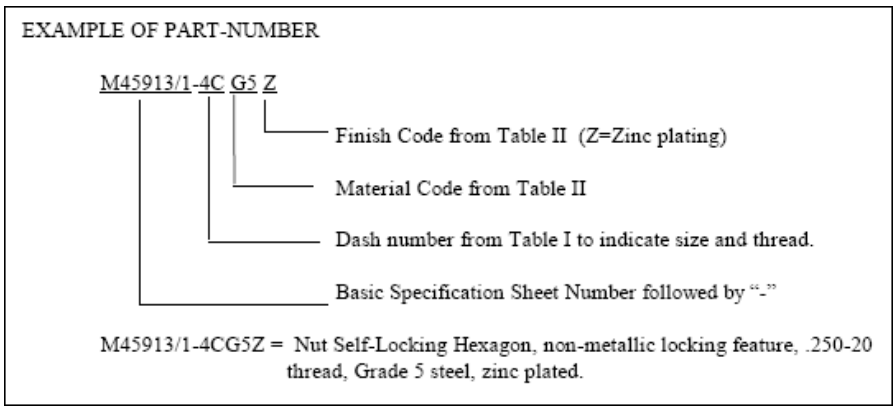

Figure 2: Excerpt from MIL-DTL-45913/1

4.8 Drawing File Naming Convention

4.8.1 A Mechanical Sketch's (MSK's) file name shall be the same as the Windchill Number. (e.g., MSK-A1-0001-FAC)

4.8.2 Multi-sheet Creo drawings created prior to 2017 were likely created as separate files. (e.g., MSK-A1-2026-RS25-01, MSK-A1-2026-RS25-02.) If sheets need to be added, the legacy naming scheme should be utilized by continuing the sequential numbering of the last two (2) digits in the file name.

4.8.3 An Engineering Modification Instruction (EMI) drawing's file name should be the same as the Windchill Number with all spaces replaced with underscores (_). For example, a drawing with a Windchill Number of EMI EUS1-17032-11 S-103 (same as DRAWING_NUMBER parameter) will have a file name of EMI_EUS1-17032-11_S-103.drw.

4.8.4 A Site-wide Operational and Repair Documentation (SORD) drawing's file name should be the same as the Windchill Number. For example, a drawing with a Windchill Number of 200HF-A549 (same as DRAWING_NUMBER parameter) will have a file name of 200HF-A549.drw.

5.0 EXISTING/NEW FEATURE REQUIREMENTS

5.1 Standard Layers

5.1.1 CAD models (parts and assemblies) and drawings shall include the SSC standard layers as defined in Table 5.

5.1.2 The standard layers shall be populated with entities as defined in Table 5.

Table 5: Standard Layers

Applies To	Layer Name	Layer Content
DRW	000_DRAFT_ENTITIES	Drawing Draft Entities
PRT, ASM, DRW	000_VISIBLE_NONSOLIDS	Nonsolid entities necessary for design intent; do not hide
PRT, ASM, DRW	001_PLANES_DEF	Default Planes
PRT, ASM, DRW	001_PLANES_GTOL	Geometric Tolerance Planes
PRT, ASM, DRW	001_PLANES_OTHER	Created Planes except for those associated with XSEC_DATUMS
PRT, ASM, DRW	002_CSYS_DEF	Default CSYS
PRT, ASM, DRW	002_CSYS_INTEGRATION	CSYS used for Integration
PRT, ASM, DRW	002_CSYS_OTHER	Created CSYS other than Integration CSYS
PRT, ASM, DRW	003_AXES_DEF	Default Axes
PRT, ASM, DRW	003_AXES_GTOL	Geometric Tolerance Axes

Continued on next page.

Applies To	Layer Name	Layer Content
PRT, ASM, DRW	003_AXES_OTHER	Created Axes
PRT, ASM, DRW	004_POINTS	Created Points
PRT, ASM, DRW	005_CURVES	Curve features
PRT, ASM, DRW	006_GEOM_COPY	Geometry Copy features
PRT, ASM, DRW	007_COSMETICS	Cosmetic features other than threads
PRT, ASM, DRW	008_THREADS	Cosmetic threads
PRT, ASM, DRW	009_SURFACES	Surface and Quilt features
PRT, ASM, DRW	010_IMPORTED_GEOM	Imported Geometry
PRT, ASM, DRW	011_GTOLS	Geometric Tolerance features
PRT, ASM, DRW	012_ANNOTATION_FEATURES	Model Notes, Hole Callouts, Annotations, and Symbols
PRT, ASM, DRW	013_GENERAL_NOTES	Other notes
PRT, ASM, DRW	014_PIPELINES	Piping features and references
PRT, ASM, DRW	015_CABLES	Cable features and references
PRT, ASM, DRW	XSEC_DATUMS	Default system created layer for x-section datum planes created on the fly

5.2 Additional Layers

5.2.1 The creation of additional layers should be minimized, particularly when a suitable standard layer is defined in Table 5.

5.2.2 When a layer must be created and the associated entities are designated by Table 5 to a standard layer, either:

5.2.2.1 The new layer shall be nested under the appropriate “parent” layer, or

5.2.2.2 The entities shall reside on both the standard layer and the new layer. (The system-created layer XSEC_DATUMS and its associated entities are exempt from this requirement.)

5.2.3 When a layer must be created and the associated entities are not designated to a standard layer, the new layer shall be named “<999>_<descriptive name>” (e.g., 999_DATUM_PT_LABELS).

5.3 Layer Usage

5.3.1 Standard layers shall not be renamed or deleted.

5.3.2 Rule definitions contained in standard layers shall not be removed or changed.

5.3.3 Unused user-created layers should be deleted.

5.3.4 Nonsolid entities shall be placed on a layer.

5.3.5 The visibility of nonsolid entities shall be controlled by layers (i.e., do not use HIDE or UNHIDE in the model tree).

5.3.6 Model layer status shall be saved such that all SSC standard layers, except for 000_VISIBLE_NONSOLIDS, are hidden and only solid geometry is visible.

5.3.7 Components shall not be placed on any layer.

5.3.8 The visibility of components should be controlled by simplified reps.

5.3.9 CAD drawing layers shall be controlled independently of the model at the drawing level.

5.3.10 CAD drawing layers shall have their status saved so the appearance is retained.

5.4 Parameters

5.4.1 Standard Parameters

5.4.1.1 CAD models (parts and assemblies) and drawings shall include the standard parameters defined in Table 6.

5.4.1.2 The SSC standard parameters shall be assigned an appropriate value and be designated as required in Table 6.

Table 6: Standard Parameters

Applies To	Name	Type	Default Value	Designated
PRT, ASM	CAGE_CODE	String	1WRN9	Y
PRT, ASM	PART_NUMBER	String		Y
PRT, ASM	DESCRIPTION	String		Y
PRT, ASM	DESCRIPTION_1	String		N
PRT, ASM	DESCRIPTION_2	String		N
PRT	STOCKSIZE	String		Y
PRT	STOCKSIZE_1	String		N
PRT	STOCKSIZE_2	String		N

Continued on next page.

Applies To	Name	Type	Default Value	Designated
PRT	MATERIAL	String	REASSIGN MATERIAL	Y
PRT, ASM	FINISH	String		
PRT, ASM	HEAT_TREAT	String		
PRT, ASM	WEIGHT	Real Number		
PRT, ASM	ID_CODE	String	000	
PRT, ASM	MODEL_CLASS	String	3	
PRT, ASM	DRAWING_CLASS	String	NONE	
PRT, ASM	SENSITIVITY_CLASSIFICATION	String	NOT REVIEWED	
PRT, ASM	EXPORT_CONTROL_CLASSIFICATION	String	NOT REVIEWED	
PRT, ASM	EXPORT_CONTROL_NUMBER	String		
PRT, ASM	ITAR_CATEGORY	String	N/A	
PRT, ASM	SECURITYLABLEPARAMETER	String		

5.4.2 Additional Parameters

5.4.2.1 User-defined parameters controlling solid geometry shall have units assigned.

5.4.2.2 Dimensionless parameters (number of places, etc.) shall not have units assigned.

5.5 Relations

5.5.1 Standard Relations

CAD models (parts and assemblies) shall include the SSC standard relations defined in Table 7.

Table 7: Standard Relations

	Relations	
	Initial	**Post Regeneration**
Parts	DESCRIPTION=DESCRIPTION_1+" "+DESCRIPTION_2 STOCKSIZE=STOCKSIZE_1+" "+STOCKSIZE_2 MATERIAL=MATERIAL_PARAM("MTL_D1")+" "+MATERIAL_PARAM("MTL_D2")	WEIGHT=PRO_MP_MASS
Assemblies	DESCRIPTION=DESCRIPTION_1+" "+DESCRIPTION_2	WEIGHT=PRO_MP_MASS

5.5.2 Additional Relations

5.5.2.1 Numeric values used in relations shall have appropriate units assigned.

5.5.2.2 Sketch relations shall not create references external to the sketch.

5.5.2.3 Assembly-level relations shall not be used directly to control definition and features of other models.

5.5.2.4 CAD model relations shall not have external references.

5.5.2.5 Comments shall be added to relations to describe an equation's function and variables.

5.5.2.6 Relations shall be error free.

5.6 Simplified Representations (Reps)

Simplified representations (commonly called reps) shall be used to improve the regeneration, retrieval, and display times of assemblies to allow the designer to work more efficiently. SREF-8070-0005, *Simplified Representation Creation and Maintenance Desktop Instruction,* may be used for information.

5.6.1 Standard Simplified Reps

CAD models (parts and assemblies) shall include the standard simplified reps as defined in Table 8.

Table 8: Standard Simplified Reps

Applies To	Name	Required Definition/Rule(s)
ASM	EMPTY	Exclude all components
ASM	NO_HARDWARE	Exclude all fastener hardware
PRT, ASM	SIMPLE	Exclude or substitute features or components to simplify overall geometry

5.6.2 Additional Simplified Reps

Simplified reps that are created specifically for CAD drawings shall be named with the prefix "DRW_".

5.6.3 Simplified Rep Usage

5.6.3.1 The standard simplified reps shall be maintained.

5.6.3.2 Simplified reps shall be used to improve the regeneration, retrieval, and display times of assemblies to allow the designer to work more efficiently.

5.7 Feature Naming Convention

5.7.1 The standard start datum features shall not be renamed.

5.7.2 The names of new features shall be changed to better describe their intended functions.

5.7.3 Datum names shall be descriptive and should follow the naming convention listed in Table 9. Most datum names should consist of the unique descriptor and the datum type. Since the datum type is also indicated by the feature icon in the model tree, the type may be omitted when the length of the name exceeds the maximum allowable number of characters.

5.7.4 All critical features should be named logically for future reference and design clarity.

5.7.5 Non-critical features should be renamed from the Creo default name to names that better describe the intended function.

5.7.6 Datums should be resized for model clarity.

Table 9: Datum Naming Conventions

Datum Type	Naming Convention	Example (Descriptor_Type)
Datum plane	PL or PLANE	BOTTOM_PL or BOTTOM_PLANE
Datum plane used for cross section	XSEC_<letter>	XSEC_A
Datum plane used for section z-clipping	XSEC_<letter>_Z-CLIP	XSEC_A-Z-CLIP
Datum plane used for view	VIEW_<letter>	VIEW_A
Datum plane used for view z-clipping	VIEW_<letter>_Z-CLIP	VIEW_A-Z-CLIP
Datum plane or axis	CL or CENTERLINE	BRACKET_ASSY_CL
Datum axis	AXIS	WHEEL_AXIS
Single datum point	PNT or POINT	INTERFACE_PNT or INTERFACE_POINT
Point array	PNTS or POINTS	INTERFACE_PNTS or INTERFACE_POINTS
Datum sketch	SK or SKETCH	LUG_PROTRUSION_SK or LUG_PROTRUSION_SKETCH
Datum curve (sweep)	CRV or CURVE	TUBE_SWEEP_CRV or TUBE_SWEEP_CURVE
Datum curve (trajectory)	TRAJ or TRAJECTORY	TUBE_SWEEP_TRAJ or TUBE_SWEEP_TRAJECTORY

5.8 Patterning

5.8.1 Repetitive features/components shall be patterned (holes, slots, and bosses).

5.8.2 The regeneration option for feature patterns shall be set to identical, followed by variable, and then general, as needed.

5.8.3 Reference patterning shall be used to reduce the amount of placement constraints needed to assemble repetitive components (such as fastener stacks).

5.9 Geometric Dimensioning and Tolerancing (GD&T)

5.9.1 GD&T datums, tolerances, and control frames shall be fully defined in the model.

5.9.2 GD&T datums shall be created as separate datum features from default datums.

5.10 Model and Drawing Integrity (Failures, Errors, and Warnings)

5.10.1 CAD Models shall not contain frozen or failing features or components.

5.10.2 CAD models shall not contain suppressed features or components except when required by family table generic, AFX member, or Pro/PROGRAM CAD models.

5.10.3 CAD models shall not produce circular references.

5.10.4 CAD models shall not produce “Geometry checks” warnings.

5.10.5 CAD models shall not contain buried features.

5.10.6 CAD models shall not have regeneration errors.

5.10.7 CAD models should not have regeneration warnings.

5.10.8 All extraneous entities should be deleted from CAD models. An extraneous entity would be any entity that is not needed in the definition of the model geometry.

5.11 Feature Modification/Verification

5.11.1 The next higher level assembly of a modified CAD model should be regenerated to ensure that no failures have occurred.

5.11.2 CAD drawings and companion files shall be regenerated without failures.

5.12 ModelCHECK™

5.12.1 ModelCHECK™ shall be run on all CAD models and drawings prior to check-in.

5.12.2 Errors found by ModelCHECK™ shall be resolved prior to check-in. SREF-8070-0022, *Verifying 3-Dimensional CAD Models,* may be used for information for acceptable occurrences of modeling techniques that ModelCHECK™ lists as an error.

5.12.3 Warnings shall be evaluated and resolved where possible. SREF-8070-0022, *Verifying 3-Dimensional CAD Models,* may be used for information on running ModelCHECK™ and resolving resulting errors and warnings.

5.13 Model Check-in

CAD document location shall be set upon check-in to the appropriate DDMS Product folder according to the team assignment and product definition and scope. Unless otherwise directed, models should be checked into the SUB ASSEMBLIES folder under the appropriate Windchill Product context (e.g., Product Context Folders… > Models and Drawings > Models > SUB ASSEMBLIES).

6.0 PART MODELING PRACTICES

6.1 Part Model General Requirements

6.1.1 Geometry

6.1.1.1 Geometry shall be created at full scale.

6.1.1.2 Geometry shall be modeled to nominal values.

6.1.1.3 CAD part models shall consist of contiguous solid geometry.

6.1.2 Dimensions

6.1.2.1 CAD model dimensions shall support design intent and drafting needs.

6.1.2.2 Manufacturing and analysis needs should be accommodated whenever possible and without detriment to design intent and drafting needs.

6.1.3 Model Tree Order

6.1.3.1 In general, all parts should follow the recommended model tree progression:

- Default Start Part Datums
- Imported Geometry (copy geometry)

- Primary Datum Features (planes, axes, sketches, and curves)
- Base Geometry
- Secondary Geometry
- Detailed Geometry (rounds, chamfers)

6.1.3.2 Features should be arranged functionally to better define design intent. For example, if adding a lug, keep the protrusion and hole adjacent in the tree.

6.2 Part Model Features

6.2.1 Sketches

6.2.1.1 Sketches shall not contain extraneous or unused entities (such as points, construction lines, and centerlines).

6.2.1.2 Sketches shall not contain weak dimensions.

6.2.1.3 Sketch references should be made to primary geometry.

6.2.1.4 Sketch references should not be made to external references.

6.2.1.5 Sketcher constraints, construction geometry, dimensions, and relations (when necessary) shall be used to preserve design intent.

6.2.1.6 Sketches should not contain multiple closed loops.

6.2.2 Hole Features

6.2.2.1 All holes smaller than two (2) inches shall be created using the hole-creation tool.

6.2.2.2 Threaded-hole thread depth and tap drill depth shall be modeled to the nominal of the dimension as shown on the drawing.

6.2.2.3 Detail screw threads shall be defined by a cosmetic thread feature and shall not be modeled as helixes.

6.2.2.4 Externally threaded items shall be modeled with shaft diameter at nominal major diameter and the thread diameter defined by a cosmetic thread feature at the minor diameter.

6.2.2.5 Internally threaded holes shall be modeled at nominal tap drill size and the thread diameter defined by a cosmetic thread feature at the major diameter.

6.2.2.6 Internally threaded holes of manufactured parts should be modeled to the major diameter in order to eliminate unnecessary interferences.

6.2.3 Rounds and Chamfers

6.2.3.1 Rounds and chamfers shall be created using the round and chamfer tools unless the radius center dimension requires defining.

6.2.3.2 Rounds and chamfers of differing sizes shall be modeled with separate round and chamfer features.

6.2.3.3 Breaks, chamfers, or fillet features with a mean value of 0.010 inch or less shall not be modeled.

6.2.4 Skeleton Part Models

Skeleton part models shall not contain solid features.

6.2.5 Bulk Items

6.2.5.1 Items not requiring solid geometry that appear in the Parts List shall be modeled using a “Bulk” type part.

6.2.5.2 Bulk items are defined as any item with a quantity of “As Required” (AR) on the BOM. If a bulk item does not exist in DDMS, SREF-8070-0003, *Library Components Desktop Instruction,* may be used for information.

NOTE: Insulation, solid propellant, and cotter pins are the only exceptions to this requirement when either needs to be modeled for mass properties, clearances, and design considerations for attaching hardware. SREF-8070-0021, *Bulk Items Desktop Instruction,* may be used for information on modeling these exceptions.

6.2.6 Vendor CAD Models

6.2.6.1 Vendor CAD models should be modeled as single-part files.

6.2.6.2 Vendor CAD models shall consist of solid geometry.

6.2.6.3 Vendor part numbers shall be entered into the PART_NUMBER parameter of the part (or of the highest level assembly if used).

6.2.6.4 Vendor CAD models shall maintain detail adequate for design use.

6.2.6.5 Vendor logos, text, threads, and other complex geometry should be removed from the CAD model.

7.0 ASSEMBLY MODEL PRACTICES

7.1 Assembly Model General Requirements

7.1.1 Geometry

Assembly model geometry and dimensions shall conform to all requirements listed in Part Model General Requirements (see Section 6.1).

7.1.2 Model Tree Order

7.1.2.1 In general, assemblies should follow the recommended model tree progression as follows:

- Skeletons
- Default Start Assembly Datums
- Secondary Datum Features (planes, axes, sketches, and curves)
- Subassemblies
- Components
- Fasteners/Hardware
- Bulk Items
- Assembly Features (holes and cuts)

7.1.2.2 Components and features should be arranged functionally to better define design intent.

7.1.3 Exploded States

Assemblies shall not be saved in their exploded state.

7.2 Assembly Model Features

7.2.1 Sketches

Sketches created at the assembly level shall conform to all requirements listed in Part Model Features (Section 6.2).

7.2.2 Assembly Cuts and Holes

7.2.2.1 Assembly level cut and hole features shall not be set to “automatic update.”

7.2.2.2 Assembly level cuts and holes shall not be visible at the sub model level (i.e., "show feature properties in sub model" shall not be used).

7.2.2.3 Assembly cuts shall not be used unless representative of a field operation or a manufacturing process.

7.2.2.4 Holes created at the assembly level shall conform to all requirements listed in Part Model Features (Section 6.2).

7.2.3 Assembling Components

7.2.3.1 Component model placements shall be fully defined.

7.2.3.2 Component model placements shall not be set to "fixed".

7.2.3.3 Component models shall not be packaged except in "Dynamic" and "Trade Study" type companion models.

7.2.3.4 Component model constraints shall be created using the most robust/stable references possible while maintaining design intent. See Table 10.

7.2.3.5 Component models used in inseparable assemblies (i.e., weldments, parts with permanent inserts) should not be reused in other assemblies.

7.2.3.6 Inseparable assemblies should be modeled as stand-alone assemblies. (**NOTE:** This does not apply to piping assemblies.)

Table 10: Recommended Constraint References

Object Type	Practice
Default Datums	Preferred
Datums	Preferred
Solid Surfaces	Preferred
Edges, Rounds, Chamfers, or Drafts	Seldom

7.2.4 Mechanisms

Mechanism connections shall have a regeneration value enabled.

7.2.5 Flexibility

CAD Models that have multiple states (such as springs, etc.) shall be modeled in the free state.

7.3 Piping/Cabling

7.3.1 These modules have pre-defined cables/fittings and pipe stock. The designer shall insert these files from the standard library.

7.3.2 SREF-8070-0003, *Library Components Desktop Instruction,* may be used for information if a standard part does not exist in DDMS.

7.3.3 SREF-8070-0013*, Piping Desktop Instruction*, and SREF-8070-0012, *Cable Routing Desktop Instruction,* may be used for additional information.

7.4 Envelope Models

This section is applicable to models of vendor items that will be procured through Envelope, Source Control, and Vendor Item Control drawings. This section is not applicable to vendor items in the standard library or ENV companion files created to show the OML of an item only. The term "envelope model" is used to refer to the model for Envelope, Source Control, and Vendor Item Control drawings.

7.4.1 An envelope assembly model shall be created to represent the reserved volume and interface requirements, when defined.

7.4.1.1 Datum features shall be created in the envelope assembly model for component constraints.

7.4.1.2 An ENV envelope companion file shall be created to represent the overall reserved volume.

7.4.1.3 When a vendor model has been received, the vendor model shall be added to the envelope assembly model. SREF-8070-0018, *Vendor Item Modeling in DDMS Windchill Desktop Instruction,* may be used for information on converting the vendor-supplied model to use in the envelope assembly model.

7.4.1.4 The envelope assembly model (.asm) shall have the ENV envelope companion file assembled until the Source Control/Vendor Item Control drawing is released.

7.4.2 The envelope assembly model shall be named in accordance with Section 4.0. This is not a companion file and will have its own part number.

7.4.3 The envelope assembly model shall be added to the corresponding assembly.

7.4.4 The envelope assembly model shall have an "owner" association with its own WT Part. Its component and drawing shall have a "content" association.

7.4.5 The ENV envelope companion file model shall include the following when available:

- The geometry reserved for the part regardless of the vendor
- Mass properties
- Mounting hole patterns
- Electrical/fluid connections with reference designation labels
- Any key structural/mechanical interface
- Any required clearances for interfaces (e.g., wrench clearances)

7.4.6 Generic CAD models representing potential vendor models shall not be released.

7.5 Colors

7.5.1 Colors can be used to give a part or assembly model a certain look or texture that may be desirable to the designer. All SSC models shall have default colors controlled by the config.pro, which points to the appearance.dmt file.

7.5.2 The colors in Table 11 shall be used for the following types of routed lines:

Table 11: Colors to be Used for Routed Line

COMMODITY	RGB	COLOR
Liquid Hydrogen (LH)	225,0,0	
Gaseous Hydrogen (GH)	255, 170, 170	
Liquid Oxygen (LOX)	0, 225, 0	
Gaseous Oxygen (GOX)	191, 255, 0	
Helium (HE)	141,0,189	
Gaseous Nitrogen (GN)	129,31,0	
Liquid Nitrogen (LN)	255,0,255	
Breathable Air (BA) Compressed Air (CA) or High Pressure Air (HA)	0,255,255	
VAC	255,255,255	

Continued on next page.

COMMODITY	RGB	COLOR
Potable Water (PW) or Chilled Water (CHW)	0,0,225	
Industrial Water (IW) Cooling/Condenser Water (CW) or Deluge Water (DW)	0,0,225	
Steam (S)	94,0,189	
Hydraulic (HD)	255,120,150	
Natural Gas (NG)	96,86,129	
Rocket Propellant (RP)	225,191,0	
Isopropyl Alcohol (IPA)	0,191,255	
Carbon Monoxide (CO)	234,170,255	
Carbon Dioxide (CO2)	255,0,255	
Gaseous Methane (GM)	255,212,170	
Liquid Methane (LM)	255,127,0	
Lubricant (LU)	255,0,255	
Triethyl Aluminum/Triethyl Boron (TT)	96,129,0	

8.0 FAMILY TABLES GENERAL REQUIREMENTS

8.1 Verify command shall be used on new or modified family tables.

8.2 Family table fields shall not be populated with an asterisk.

8.3 Family table instances shall not be removed from family tables.

8.4 All instances of a family table should be created upon initial creation of the generic if possible.

8.5 The generic shall not be used in any assembly except as required by family tabled assemblies.

8.6 The generic of a family tabled assembly shall contain generics of the components that are family table driven.

8.7 Variable family table items should be placed as far down the model tree as feasible.

9.0 DRAWING PRACTICES

9.1 General Drafting Requirements

9.1.1 Drawings shall be created in accordance with SOI-8080-0007, *SSC Test Site Drawings*.

9.1.2 The standard font for text on new drawings shall be Arial.

9.1.3 If applicable, CAD drawings should have models associated to them. (**NOTE:** Exceptions include but are not limited to legends, vicinity plan, fee area, drawing index, etc.)

9.1.4 Multiconfiguration CAD drawings shall have all configurations associated to the drawing.

9.1.5 Tabulated part and assembly CAD drawings shall have the generic associated to the drawing when the models are family table driven. All other tabulated part and assembly drawings shall have all models associated to the drawing.

9.1.6 CAD drawing views defining fabrication shall be associative to the model.

9.1.7 CAD drawing dimensions shall be driven by and associated to the model, including tabulated part tables and opposite parts.

9.1.8 CAD drawing dimensions shall either be “shown” (from the model) or “created” (using the dimension tool). “Shown” dimensions are preferable.

9.1.9 CAD drawing dimensions shall not be overridden.

9.1.10 CAD drawing dimensions that are called out in note form shall be driven by their associated model dimensions.

9.1.11 CAD model geometry defining fabricated items shall not be represented by draft entities.

9.1.12 Supplemental geometry in a drawing view shall be visually distinguished from the design model geometry in the drawing view.

9.1.13 Drawing view display status shall not be set to “Follow Environment.”

9.2 Drawing Content and Types of Drawings

9.2.1 The drawing shall contain all information necessary to complete the model definition and to inspect the part and/or assembly.

9.2.2 Parts shall be shown on the drawing with all views necessary to define the part.

9.2.3 Refer to SSTD-8070-0002-CONFIG, *SSC Drafting Manual.*

9.2.4 Any free symbol shall be related to its respective view or object so that the view and symbol move in unison.

9.2.5 Design Layout Drawing

When a Design Layout drawing needs to include parts or assemblies that are not assembled into the primary model, a –REL companion CAD file may be used.

9.2.6 Tabulated Drawing

9.2.6.1 The values that are table driven shall be derived from the model utilizing the repeat region functionality.

9.2.6.2 SREF-8070-0020, *Tables and Tabulated Drawings Desktop Instruction,* may be used for additional information.

9.3 Drawing Templates and Formats

9.3.1 CAD drawings shall be created using the appropriate and approved SSC drawing templates. Refer to Section 3.1.3.

9.3.2 CAD drawings shall use the appropriate and approved SSC drawing formats. Refer to SSTD-8070-0002-CONFIG, *SSC Drafting Manual.*

9.3.3 CAD drawings shall use the appropriate and approved SSC drawing setup files (DTL files).

9.3.4 Table field values in the SSC title block of the SSC drawing format shall be derived from parameters. Parameter fields within the drawing template are not to be overridden.

9.4 Drawing Parameters

CAD drawings shall contain the standard SSC parameters defined in Table 12.

Table 12: Standard Drawing Parameters

Name	Type	Default Value	Designated
DESCRIPTION	String		Yes
DRAWING_NUMBER	String		Yes
TITLE1	String		Yes
TITLE2	String		Yes
TITLE3	String		Yes
TITLE4	String		Yes
TITLE5	String		Yes
DRAWING_NO	String		Yes
SHT	String		Yes
DRAWN_BY	String		Yes
DRAWN_DATE	String		Yes
CHECKED_BY	String		Yes
CHECKED_DATE	String		Yes
ENGINEER1	String		Yes
ENGINEER_DATE1	String		Yes
ENGINEER2	String		Yes
ENGINEER_DATE2	String		Yes
ENGINEER_ENVIRONMENTAL	String		Yes
ENGINEER_ENVIRONMENTAL_DATE3	String		Yes
SITE_MANAGER	String		Yes
SITE_MANAGER_DATE	String		Yes
CONCURRENCE	String		Yes
CONCURRENCE_DATE	String		Yes
SUBMITTED_BY	String		Yes
SUBMITTED_DATE	String		Yes
SAFETY	String		Yes
SAFETY_DATE	String		Yes
APPROVED	String		Yes
APPROVED_DATE	String		Yes
TOTAL_SHEETS	String		Yes
REV	String		Yes
AUTHORITY_NAME	String		Yes
AUTHORIZATION	String		Yes
ISSUED_CEF_DATE	String		Yes
REF_INFO	String		Yes

Continued on next page.

Name	Type	Default Value	Designated
SYSTEM	String	NONE APPLICABLE	Yes
SUBSYSTEM	String	NONE APPLICABLE	Yes
FACILITY	String	NONE APPLICABLE	Yes
BUILDING_NUMBER	String		Yes
PROJECT	String		Yes
DRAWING_TYPE	String		Yes
AFSCODE	String	8810	Yes
RETENTIONSCHEDULE	String	Schedule 8 Item 48B.2	Yes
RECORDDISPO	String	PERMANENT	Yes
ID_CODE	String	D00	Yes

9.5 Standard Symbols

9.5.1 CAD drawings should use the appropriate and approved SSC symbols.

9.5.2 If a drawing requires a symbol that is not in the standard symbols library, the new symbol shall be requested from CAD/CREO team leads.

10.0 DESIGN METHODOLOGY

Design activities can be accomplished using Bottom-Up Design Methodology, Top-Down Design Methodology, or a combination of the two (2).

10.1 Guidelines for Selecting Design Methodology

10.1.1 When all parts are to be independent during the entire design cycle, Bottom-Up Design Methodology should be used.

10.1.2 When wanting to control geometry and/or location of multiple parts parametrically using skeletons during the development of the design, Top-Down Design Methodology should be used. During creation of an assembly, constraining the parts with actual mating features is recommended. CSYSs and datum features may be used for constraining components.

10.1.3 The type of Top-Down Design Methodology used shall be Modular Product Architecture (MPA) methodology. Refer to Section 11.0 for detailed information.

10.2 External References

10.2.1 External references shall be limited to external copy geometry features, merge/cutout features, inheritance features, and publish geometry features.

10.2.2 External references shall only reference skeleton models within the CAD model's subassembly.

10.2.3 Copy geometry features shall not reference directly from other copy geometry or external copy geometry features.

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

10.2.5 Reference control settings shall be used to minimize the creation of external references.

10.3 Publish and Copy Geometry

10.3.1 Publish geometry features shall only exist in skeletons.

10.3.2 Publish geometry features shall be moved to the footer section of the model tree.

10.3.3 Publish geometry features shall be renamed to clarify intent.

10.3.4 Copy geometry features shall be renamed to clarify intent.

10.4 Designs with Multiple Positions

10.4.1 Primary deliverable models shall be modeled in one (1) chosen "default state" (e.g., the "operational condition").

10.4.2 Primary deliverable models should not be "overloaded" or made family tabled assemblies.

10.4.3 Separate companion models may be created to depict the product's alternate states.

11.0 MODULAR PRODUCT ARCHITECTURE (MPA)

The MPA process involves the explicit identification and management of interfaces using skeletons.

11.1 General Requirements

The MPA process may be used as a tool to coordinate location and interface definition during design development, but all external references shall be set to "Independent" or removed prior to release.

11.2 Locator Skeletons

11.2.1 Locator skeletons are used for location and orientation of interfaces. They shall not be used to drive parts or other skeletons, including routing, unless the components have multiple interfaces.

11.2.2 Publish geometry sets within locator skeletons should only contain CSYSs.

11.2.3 Locator skeletons shall contain the minimum features necessary to define locator CSYSs.

11.2.4 Locator skeletons shall not contain external references.

11.2.5 Locator skeletons shall not contain solid geometry.

11.3 Interface Control Skeletons

11.3.1 Interface control skeletons are stand-alone files.

11.3.2 Interface control skeletons shall only control geometry and not location.

11.3.3 An interface control skeleton shall control only one interface.

11.3.4 Interface control skeletons shall not contain solid geometry.

11.4 Model Requirements

11.4.1 Models shall contain external references that are only dependent upon publish geometry features from locator or interface control skeletons, except for bolt hole pattern axes.

NOTE: Additional external references may be used to provide a temporary guide during feature creation but shall then be removed from the model immediately.

11.4.2 Parameters and relations shall be controlled at the part level and not by other parts, assemblies or layouts.

11.4.3 One skeleton shall not be used for more than one routed line.

11.4.4 ModelCHECK™ shall be run prior to release to verify all external references have been set to "Independent".

11.4.5 SREF-8070-0006, *Creation of Skeleton Files in DDMS Windchill Desktop Instruction,* may be used for information on implementing MPA.

12.0 STANDARDIZATION

12.1 Setting and Scripts

12.1.1 The following standard settings and supporting data are required to obtain consistent operation of Creo Parametric® design software and to perform CAD modeling activities in accordance with this SSTD. These settings and supporting data shall be fully documented and maintained by DDMS.

12.1.2 Appropriate client software installations shall be capable of setting up a standard seat of Creo Parametric® design software.

12.1.3 The installations shall install and/or configure these standard settings and supporting data for both a client machine located inside the SSC computer network and for a client machine located outside of the SSC computer network.

12.2 Common Parts Library

12.2.1 A Common Parts Library is maintained in DDMS to aid designers in the selection of common parts to the greatest extent possible. While not every item can be found in the library, this should be the first place a designer looks when selecting parts.

12.2.2 The designer shall verify the parts are appropriate for a specific application. The availability of parts in the Common Parts Library does not mean a library part can be used for all applications.

12.2.3 SREF-8070-0003, *Library Components Desktop Instruction*, may be used for information in determining the process to be used for additions/deletions and/or revisions to the Common Parts Library.

12.3 Symbol Library

12.3.1 A standard symbol library located on a controlled server shall be created and maintained by the designated design team members.

12.3.2 Standard symbols shall be documented. SREF-8070-0007, *Standard Symbols Desktop Instruction,* may be used for information.

12.4 Startup Script

12.4.1 A startup script shall be provided to all Creo users that automatically copies all SSC standard configuration files to the user's system and ensures the latest SSC approved version of Creo Parametric® design software is being utilized.

12.4.2 Creo Parametric® design software shall always be started using the startup script.

12.5 Configuration Settings

SSC shall create and maintain several standard configuration files that define the setup of Creo Parametric® and the way it operates. The start-up script provides the user access to these files, which are located on a controlled server and maintained by DDMS. These files include config.pro, config.sup, Creo_parametric_customization.ui, and Drawing Setup (.dtl). (See Definitions for each file type.)

13.0 ACRONYMS/DEFINITIONS

13.1 Acronyms/Abbreviations

2D	2-Dimensional
3D	3-Dimensional
AR	As Required
ASME	American Society of Mechanical Engineers
BA	Breathable Air
BOM	Bill of Materials
CA	Compressed Air
CAD	Computer-Aided Design
CHW	Chilled Water
CO	Carbon Monoxide
CO2	Carbon Dioxide
CSYS	Coordinate System
CW	Cooling/Condenser Water
DDMS	Design and Data Management System
DW	Deluge Water
EMI	Engineering Modification Instruction
FAC	Facility

GD&T	Geometric Dimensioning and Tolerancing
GH	Gaseous Hydrogen
GM	Gaseous Methane
GN	Gaseous Nitrogen
GOX	Gaseous Oxygen
GSE	Ground Support Equipment
HA	High Pressure Air
HD	Hydraulic
HE	Helium
ICD	Interface Control Document or Interface Control Drawing
IN	Integration
IPA	Isopropyl Alcohol
IW	Industrial Water
lbm	Pound Mass
LH	Liquid Hydrogen
LM	Liquid Methane
LN	Liquid Nitrogen
LOX	Liquid Oxygen
LU	Lubricant
MI	Main Injector
mmKS	Millimeter Kilogram Second
MPA	Modular Product Architecture
MSK	Mechanical Sketch
NASA	National Aeronautics and Space Administration
NG	Natural Gas
OML	Outer Mold Line
PB	Preburner
PM	Project Manager
PS	Pipe Support
PW	Potable Water
REP	Representation
RP	Rocket Propellant
S	Steam
SAE	Society of Automotive Engineering
SOI	Stennis Organizational Instruction
SORD	Site-wide Operational and Repair Documentation
SPR	Stennis Procedural Requirement

SREF	Stennis Reference
SSC	John C. Stennis Space Center
SSTD	Stennis Standard
TA	Test Article
TMS	Thrust Measurement System
TS	Test Stand
TT	Triethyl Aluminum/Triethyl Boron
TTOS	Thrust Take Out Structure
UDF	User-Defined Feature
VAC	Vacuum
VE	Vehicle
WT	Windchill Type

13.2 Referenced and Applicable Definitions

Baseline (DDMS Baseline)	A collection of drawings/models at a certain point in time within DDMS. This does not mean the collection, which is referred to as Managed Baseline within DDMS, is configuration controlled.
Bill of Materials (BOM)	A listing of all parts for an assembly.
Bottom-Up Design Methodology	Design of individual components independent of external references or controls.
Bulk Item	A specific Creo part with no solid geometry which has an undetermined shape or indefinite quantity. (Examples include glue, tape, or paint.) These items are designated with an “as required” (AR) quantity on the parts list.
Buried Feature	A solid feature that does not contribute to the geometry of the finished part. (Examples include a hole or cut that is later filled back in or a hole that exists over another hole or cutout.)
CAD Document (CAD Object)	A version-controlled object in DDMS that can contain one (1) primary CAD file, as well as other secondary reference files.

CAD File	A file used within the Creo Parametric® design software which includes CAD parts, assemblies, drawings, formats, sketches, layouts, etc. (.prt, .asm, .drw, etc.)
CAD Model Structure	The relationships between models and where those models are used or referenced.
Circular Reference	A condition where an assembly contains a number of references that form a dependency loop.
Common Parts	Those items listed on the Common Parts List; includes standard parts and vendor parts.
Companion File	Related CAD objects created in order to manage CAD data (e.g., Shrinkwrap™ files, skeletons, envelopes, related assemblies) or provide additional part information (e.g., modified models, manufacturing models, stress and analysis models).
Config.pro	A text file that stores all of the settings that define the way in which Creo Parametric® handles operations.
Config.sup	A protected system configuration file. Any values that are set in this file cannot be overridden by other (more local) config.pro or creo_parametric_customization.ui file.
Cosmetic Thread Feature	A specific feature consisting of a cylindrical surface that represents the thread diameter (major diameter for an internal thread or minor diameter for an external thread).
Creo_parametric_customization.ui	A database file that stores window configuration settings, such as toolbar visibility settings and Model Tree location settings.
Design Layout Drawing	Type of drawing created to define the overall configuration of a design; serves as an engineering work sheet from which engineering drawings are made; is not used for procurement or inspection.
Design Release	Approved design that is ready for implementation/ construction.

Drawing Setup (.dtl) File	A file containing options (e.g., height of the dimension and note text, text orientation, geometric tolerance standards, etc.) for each individual drawing and drawing format. The file specified in the drawing_setup_file configuration option establishes the default drawing setup option values for any drawing.
End Item	A Windchill Type (WT) Part that represents a Configuration Item.
External Reference	A reference within a CAD object that creates dependencies between itself and another CAD object(s).
Family Table (Generic)	A collection of parts or assemblies that are essentially similar but deviate slightly in one (1) or two (2) aspects such as size or detail features.
Footer (Regeneration Footer)	A section of the model tree that can contain only certain types of features. Features in the footer are always created and evaluated at the end of the feature list and appear at the bottom of the model tree.
Frozen Feature/Component	A feature or component that has failed and will be placed in its last stored/solved location until it can be regenerated successfully.
Geometric Dimensioning and Tolerancing (GD&T)	A method of explicitly describing an object's nominal geometry and allowable variation in accordance with ASME Y14.5.
Geometry Check (geom.check)	A model inaccuracy resulting from features that create indeterminate or otherwise unacceptable geometry.
Inseparable Assembly	An assembly whose piece parts cannot be separated and still retain its original form, such as weldments or bonded assemblies.
ModelCHECK™	A configurable verification tool integrated within Creo.
Modular Product Architecture (MPA)	A type of Top-Down Design Methodology which focuses on the explicit identification and control of the physical/mechanical interfaces and/or their location in the design through the use of skeletons but does not control the overall geometry of the design.

Nested Layer	A layer that contains other layers.
Nominal Value	The primary value of a dimension that does not include upper and lower tolerance values; the average value of a limit dimension. Example of nominal dimensions are: 0.257 + 0.003/-0.007 (nominal value is 0.257), 0.250–0.265 (nominal value is 0.2575), 0.260 MAX (nominal value is 0.260).
Overloaded Assembly	An assembly which contains more components than are required in reality so that different combinations of components may be shown or hidden in order to depict alternate component positions or assembly configurations.
Overridden	A displayed value that has been changed from its true value.
Parameters	Useful for providing information about an object, they may be assigned to features, drawings, and part and assembly models. Parameters are typically used to control information, such as drawing titles, part numbers, and next assembly information.
Parent-Child Relationship	Creating a direct dependency between components in an assembly without using a skeleton.
Primary Deliverable Model	The main models required by the design activity and are used to drive the product structure of a design (as opposed to companion files).
Product Structure	A hierarchical representation of all assemblies and component parts necessary to assemble an end item. A product structure can be expanded (i.e., expand each subassembly to see its child parts) until the lowest level component parts are shown. A BOM report can be generated from the expanded product structure.
Promotion Request	Enables the designer to set the state of one (1) or more items to a new lifecycle state with a selection of review processes; allows the designer to promote objects to an approved state using the Design Review Approval process.

Relations	User-defined logic statements written between symbolic dimensions and/or parameters.
Simplified Representation (Simplified Rep)	A subset of the parent model that displays fewer details (components, features) than the complete model.
Skeleton	A part model that contains some combination of features (datums, sketches, copied geometry, etc.) to drive design intent (component placement, etc.) of one (1) or more assemblies.
Standard Parts	Items governed by a national (e.g., National Aerospace Standard) or international (e.g., Society of Automotive Engineering [SAE] Aerospace) standard; a non-vendor item part.
Start Assembly	Template used to standardize assembly creation.
Start Part	Template used to standardize part creation.
Supplementary Geometry	Geometric elements included in the drawing to communicate design requirements or intent but not defined in the CAD model.
Top-Down Design Methodology	Method of placing critical information in a high-level location and communicating that information to the lower levels of the product structure.
Windchill	See Design and Data Management System (DDMS) Windchill.

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Appendix A: Initial Naming Convention Example

The initial CAD file name can have seven (7) or eight (8) distinct parts which define the Project Code, Concept ID, Element, Location, Subsystem/Component Function, and Sequence Number.

- **Project Code**
  - Alpha characters assigned by Project Manager (PM).
- **Location**
  - Underscore followed by the physical location of the hardware.
  - Sample locations are:
    - A1
    - B2
    - E1
- **Element (System, Pipe Support, etc.)**
  - Alpha characters defined within each project
  - Represents major divisions of a project
  - Sample elements are:
    - FAC for Facility
    - GSE for Ground Support Equipment
    - LH for Liquid Hydrogen System
    - MI for Main Injector
    - PB for Preburner
    - PS for Pipe Support
    - TA for Test Article
    - TMS for Thrust Measurement System
    - TS for Test Stand
    - TTOS for Thrust Take Out Structure
    - VE for Vehicle
- **Subsystem/Component Function**
  - Alpha characters determined by the project
  - Represents the subsystem or function of the hardware
  - More subsystems/component functions can be defined as needed
  - IN for integration
  - The Subsystem/Component Functions are located on 54000-P001, Sheet 1
- **Sequence Number**
  - Reference SOI-8080-0007 for Drawing Number Sequence nomenclature. Additionally, for example, a LOX pipe support MSK number would be represented as “02XX” while an RP pipe support MSK number would be “07XX.”

Examples of file names:

Drawing Name:

MSK-E1-XXXX-PROJECT CODE-01	Drawing Type-Location-Sequence Number-Project Code-Sheet Number

Top Level Project Assembly Name:

E1C2-PROJECT CODE-PB-TEST-PROJECT	Location-Project Code-Element-Subsystem/Component Function-Project Suffix

Subassembly Name:

PROJECT CODE-LH-STE	Project Code-Element (System)-STE

Part Name:

PROJECT CODE-LH-STE-PS-1-MICARTA	Project Code-Element(System)-STE-Pipe Support Number-Part Description

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