Attachment 0024 (SOWC-4147)

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3-Dimensional Technical Data Package Configuration Management, Modeling & Drawing Interim Operating Procedure For PdM-MTV

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1 Foreword

This standard describes the requirements and procedures for the Three Dimensional Technical Data Package (3D/TDP) solid model repository established and maintained by PdM-MTV. The 3D/TDP is based on Creo Parametric Mechanical Design Automation software for the purpose of Computer Aided Design (CAD). The 3D/TDP uses Windchill PDMLink Product Data Management (PDM) software for data management and storage. Creo Parametric and Windchill PDMLink are both products of PTC. All contractors and government personnel working with models contained within the 3D/TDP system shall follow the requirements of this standard regarding both Creo Parametric and Windchill PDMLink. Documents that are not solid model related but which normally comprise a TDP such as packaging sheets, quality assurance provisions, electrical schematics, etc. shall be stored in Windchill PDMLink.

1.1 Background

In order to take advantage of available technologies, PdM-MTV has been maintaining the technical data system including 3D solid models. The intent is to develop, maintain, and provide these 3D/TDPs throughout the lifecycle of the item. 3D/TDPs will be used for development, production, life cycle management, configuration management, logistics support, and all other purposes that would normally use technical data. The 3D based system will have the following advantages:

- Automated generation and update of line drawings
- Automated Bills Of Material (BOM)
- Infinite viewpoints and exploded views of assemblies
- Rapid prototyping
- Faster design updates
- Improved logistics support
- Improved system interface analysis
- Better and faster engineering analysis capabilities (kinematic, dynamic, and structural analysis, tooling design, etc.)
- Reduced TDP interpretation errors

In order to use solid models as a basis for the technical data it is important that the solid models be built to a set of common standards, using sound modeling practices, and be fully defined. Performing cosmetic modeling (in which a model looks like it is supposed to but is actually

dimensionally incorrect), may be faster and easier; however, as a basis for product definition is inadequate.

Appendix D.5 Checklists referenced within this standard, defines required criteria for promotion approval to the ***Released*** state. Any requirements made by this standard must be met, whether or not they are cited on the checklist.

The ***FMTV PDMLink User Guide*** document, referenced within this standard, defines procedures to be followed when using Windchill PDMLink in conjuncture with the 3D/TDP effort.

2 Configuration Management Representations

2.1 Standards Application

The requirements of this specification apply to all objects, parts, assemblies, and/or drawings that are submitted through the formal release process via Windchill Change Notice from ***In Work to Released***.

2.2 Drawings Required

All parts and assemblies promoted to ***Released*** shall have associated, fully defined 2D drawings or a PDF copy of the latest (requires prior TACOM approval for PDF). Only the models/drawings representing the latest design configuration (i.e., the latest revision drawing plus any outstanding ECPs) shall be promoted through the release process. This does not apply to Standard Parts, which are defined in ***Appendix F***.

2.3 Engineering Changes

Models/drawings shall be changed via approved Engineering Change Proposals (ECPs) only. ECPs shall conform to **MIL-STD-3046**.

2.4 Configuration Management

When 3D/TDP associated 2D drawing are released in the database, that drawing becomes the master for configuration management purposes (i.e. drawings will not be input into the 3D/TDP and concurrently maintained in another CAD format.) Refer to **MIL-STD-3046** for complete Configuration Management standards.

2.5 Products

The Product in Windchill PDMLink for the FMTV 3D solid modeling TDP will provide enough detail for files to be checked into the database in the appropriate location.

2.6 Object Status

The user making changes on a drawing, part, assembly, or other object for release in the FMTV database shall set the Commonspace status to ***Checked Out*** while change is in progress. The status should be applied when the drawings are identified for the ECP.

2.7 Security

This system security falls under the Windchill system security settings.

2.8 Revision Scheme

The initial release of drawings, parts and assemblies will be the - revision level. Subsequent changes will be from a one to three digit alphabetic character (not including letters I, O, Q, S, X, or Z (i.e., A, B, C,… AA, AB, AC, … YYV, YYW, YYY.) Refer to **ASME Y14.35M** for further information about Revision Scheme.

2.8.1 The part or assembly revision scheme is dependent to its associated drawing and vice versa. The Creo Parametric drawing, and model revision shall be the same for FMTV drawings/models. All changes to model and drawing will follow formal ECP procedures.

2.8.2 The revision of the drawing shall be displayed on all drawing sheets in accordance with the current acceptable drawing format. As new revisions of the models and drawings are created the next higher revision will be used for the current drawing.

2.9 Promotion

Drawings submitted for promotion shall follow instructions found in ***FMTV PDMLink User Guide.***

2.10 Quality Assurance

All parts, assemblies, and drawings submitted for promotion shall be checked in accordance with ***Appendix D.5 Checklists***.

2.10.1 All Creo Parametric objects promoted to ***Released*** are required to pass the FMTV ModelCHECK process.

2.11 Release Level Flowchart

The following flowchart illustrates the FMTV release scheme using the ECP2 or ERR process in Windchill PDMLink.

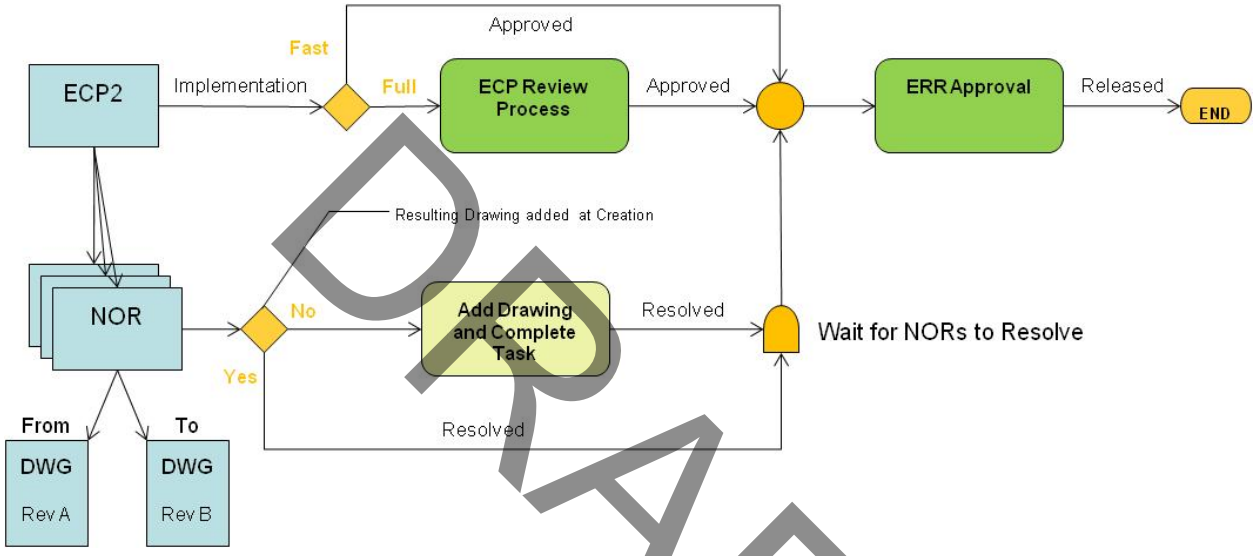


FIGURE 1: ECP2 RELEASE LEVEL FLOWCHART

2.12 Notice of Revision

The Notice of Revision (NOR) document shall be a redline markup of the previous revision of the drawing. If the drawing is an Initial Release drawing, the NOR is not required. The redline NOR is submitted and stored with the ECP document.

3 Part Modeling Standards

3.1 Part Modeling Practice

3.1.1 Parts should be functional and created in a logical sequence. Features should be simple and be created such that they are easily modifiable by another user. Parts shall have every feature that the real part is required to have. For example, plastic parts should contain every draft angle and radii detail expected on the real part. If text (e.g. part number) is stamped or molded into a part, that should also be modeled. Parts should be able to stand on their own without reference to other parts or assemblies.

3.1.1.1 If the part is a source controlled drawing or vendor item control drawing, CAD Files from the vendor will be requested. If the CAD file is unavailable, all interface and envelope dimensions shown on the drawing shall be modeled. The part shall be modeled in Creo Parametric.

3.1.2.1 For Creo Parametric conversion efforts, features will be created using the existing dimensioning scheme contained in the drawings or other appropriate source. All changes to the drawing shall be documented by the NOR/ECP.

3.1.2.2 For new parts and design changes, features will be created in a logical manner in which the dimensioning scheme for the new part will be shown with no created dimensions on the drawing (unless required).

3.1.3 All models shall regenerate successfully with all features in the unsuppressed state. When parts are modified, the next higher assembly shall also be capable of being successfully regenerated.

3.1.4 The latest ***FMTV_START_PART*** or ***FMTV_START_ SHTML _PART*** shall be used for part creation.

3.1.5 Sheetmetal parts shall be created with the Creo Parametric Sheetmetal module.

3.1.6 All routed tubing or piping models and assemblies shall be created using the Creo Piping and Cabling Extension.

3.1.7 All wiring and cabling models and assemblies shall be created using a combination of Creo Piping and Cabling Extension.

3.2 Tolerances

All dimensions shall include tolerances as required to be shown appropriately on the drawing. When geometric tolerances, geometric tolerance datums, inspection dimensions, datum targets, and/or basic dimensions are required, these shall be set in the model using the appropriate **GTOL** commands.

3.3 Materials

Materials shall be identified on the part in two locations:

1. The materials section under Model Properties in Creo Parametric. Material file creation shall be requested from the custodian of the material file library for any material required that is not in the current library.
2. In the materials note section of the 3D notes which are applied to the drawing.
3. The material file used on the part shall match the material called out in the notes (for example: ASTM A36 material file shall be used over a generic STEEL material file)

3.4 Surface Finishes

Surface finishes shall be shown on the part as required.

3.5 Model Notes

Sufficient model notes shall be used to describe the part and part features. If notes are intended to apply to specific features, they shall be attached to those features. Notes found in the FMTV start parts include:

a. Distribution_Note: The drawing distribution statement shall be included in a 3D note called ***Distribution_Note***.

b. Proprietary_Note: If a proprietary legend is required on the drawing, this legend shall be entered on the model as a 3D note entitled ***Proprietary_Note***.

c. Export Control Note: The drawing export control warning note shall be included in a 3D note called Export_Control_Note

d. Note1: ***Note1*** shall contain the current drawing standards.

e. Note2 – Note#: These notes shall be used to describe the model as required for the drawing.

f. DRW_NOTE: This note is a summary of all the Note1-Note# for display on 2D Parametric drawing.

g. Note0_DESIGN_INTENT: ***Note0_DESIGN_INTENT*** shall be used to document design intent. Design intent is any optional information the designer thinks is valuable to those who might view the model at some future point. An example of a design intent note is as follows:

THIS PART IS MADE OF ALUMINUM 7075-T6. CARBON STEEL WAS TESTED BUT CORRODED DURING USE. USE OF STAINLESS WAS CONSIDERED BUT NOT PURSUED DUE TO TIME CONSTRAINTS.

3.6 Relations

Relations shall have no errors upon regeneration. Relations found in feature ***CS0*** and ***ACS0*** of the FMTV start models are required.

3.7 Default Datums

All parts will start with default datum planes and a default coordinate system identical to those found in the FMTV start models. The default datum planes shall be named ***FRONT***, ***TOP***, and ***SIDE***. The coordinate system shall be named ***CS0*** with its orientation as follows; Z axis is elevation, X axis is along the length of the vehicle, Y axis across the vehicle width.

3.8 Part File Names

Parts shall be named using the following convention:

CAGE_PartNumber.prt i.e. 19207_8448510.prt

If the model requires an extension (or suffix) it will follow the convention described in ***Table 1 – Model Name Suffix Usage***.

Suffix Range		Description
-001	-399	Fully detailed down level parts on a drawing.
-100	-150	Down-Level detailed parts that are part of weldment that are the same base number of the weldment assembly. (Note: This is an overlap for the -001 thru -399, but the down-level weldment parts are to start at -100. Fully detailed parts are still allowed to use -100 thru -150).
-400	-499	Parts modified in the field for the application of a kit.
-500	-599	Non-detailed down level parts on a drawing.
-5000	-	Top Level 87T – Used for Large Assemblies
-600	-699	Routed systems models (cabling, piping, etc…)
-6000	-6999	Overflow for routed systems models (cabling, piping, etc…) if more than the original -600 through -699 are required
-700	-799	Simulation models (as well as the extension _FEA)
-800	-899	Backdrop Parts / Assemblies
-900	-998	Overflow or for miscellaneous (deformed, die, shrinkwrap parts, etc.) Deformed parts may also use the _def naming convention. Same intent for dies and shrinkwraps.
-9000	-9999	Overflow or miscellaneous parts if more than the original -900 through -998 are required.
-999 or -gen		Non-useful generic components used in Family Tables.
_skel		Skeleton Models
-opt, -oc		Optional Configuration (either is acceptable)
-cast		Use for castings when a casting is an Alternate Design

TABLE 1: MODEL NAME SUFFIX USAGE

3.8.1 Hardware File Names

Hardware naming convention shall generally start with -001 and is not required to follow ***Section 3.8***.

3.8.2 Exceptions

- Refer to ***Section 7 - Electrical*** for naming convention on Schematics and Electrical related components.
- Refer to ***Section 8 - Piping*** for naming convention for Piping Assemblies.

3.9 Model Views

All models shall have separately named views defining the ***TOP_VIEW***, ***BOTTOM_VIEW***, ***LEFT_VIEW***, ***RIGHT_VIEW***, ***BACK_VIEW***, ***FRONT_VIEW***, and ***DEFAULT*** views. These views exist in the FMTV start part. The ***FRONT*** view is oriented facing the yellow (or primary) side of the ***FRONT*** datum plane, with the yellow side of the ***TOP*** datum plane facing up. The remaining orthogonal views follow standard third angle projection practice. Current drawing models should be created such that the named views correspond to the existing drawing views.

3.10 Deformed Parts

Deformed parts will have to be handled on a case to case basis. If a part can be deformed in numerous shapes (rivets), then use the non-deformed part. This makes it easier, rather than having 100+ deformed parts. In addition, Flexible Components shall ALWAYS be considered first and is preferred over a deformed part. This will help keep the data management aspect to a minimum.

3.10.1 Naming Convention for Deformed Parts

Only use the non-deformed part's part number with the extension starting with a -9xx. Also, -9xxx can be used if there are numerous instances of the deformed parts. Do *not* use the top assembly name with -900 as was previously used.

3.10.2 When to use Deformed Parts

1. Create a deformed part if it will only be needed on a particular drawing. (Examples are lanyards, cables, etc).
2. Create a deformed part if the non-deformed part causes a major interference.
3. Create a deformed part if the non-deformed part will not work in any way possible.

3.10.3 Creating Deformed Parts from Tabulated Parts

If a deformed part is made from a tabulated part, keep the tabulation and add the -900 extension after. (Ex: 19207_12345678-001 will be 19207_12345678-001-900 when it is deformed).

3.10.4 Creating Deformed Parts with Flexible Components

The preferred option when creating deformed parts is using flexible component functionality. Flexible components allow the model to be represented in different states within different assemblies. Flexible components shall always be looked at first for any deformed part/assembly. An example of this is if a drawing of a spring requires it to be modeled in its free state but when it needs to be assembled it must be represented in its compressed state. With flexible component functionality it would not require that a second model to be created. It allows the designer to vary dimensions, parameters, resume or suppress features (or components if flexible component is a sub-assembly).

3.10.5 Promoting Deformed Parts

When promoting a deformed part, promote it with the assembly where the deformed part is used. Keep the deformed part at the *same* revision level of the most current non-deformed part in Windchill PDMLink database. Also, remember to relate all deformed parts back to the non-deformed parts drawing while the drawing is being updated. If the deformed part is just being used on a next higher assembly, it shall be related back to the deformed parts drawing the *next* time the drawing is revised. When a previously released deformed part is required to be changed in an ECP due to other changes that affect said part and the base part is not part of that ECP, either a new flexible model state or a standalone model shall be created with another -9xx extension. The new model shall have the same revision as the base number.

3.11 Layers

The following layers shall be used in part mode (these exist in the FMTV start part). Other layers may be added as needed, for example, a datum required on a drawing may be placed on a separate layer so that it can stay on while the other datums are layered off.

	Layer name	Layer Assignments	Default State
a.	DTM_AXES	All datum axes	Hide
b.	DTM_CSYS	All datum coordinate systems	Hide
c.	DTM_CURVES	All curve features	Hide
d.	DTM_PLANES	All datum plane features	Hide
e.	DTM_POINTS	All datum point features	Hide
f.	G_DTM_PLANES	All geometric tolerance datum planes	Hide
g.	ITEM_ID	Only the cosmetic/curve feature describing the part identification number	Hide
h.	NOTES	All part notes	Hide
i.	SURFACES	All surface features and quilts	Hide
j.	THREADS	All cosmetic and actual threads	Hide
k.	XSEC_DATUMS	All cross-section datums	Hide

TABLE 2: PART LAYERS

3.12 Part Parameters

The following parameters are required when entering a part or assembly into the 3D/TDP Product Release Process. Required items are noted in the Input column.

Items in italics are no longer required to be filled out, but will remain for legacy drawings. Windchill PDMLink will manage these parameters.

	Parameter Name	Parameter Description	Format Example	Input
01*	AGENCY	Agency or service	US ARMY	*
02	CAGE_CODE	Original design activity CAGE code	19207	REQ
03	CATALOG_NOMENCLATURE	Nomenclature assigned by supply	SPRING, HELICAL	
04	CHECKER_NAME	Person that checked the original drawing	K.BALL	REQ
05	CONTRACT_NUMBER	Contract #-original design contract	DAAE20-03-C-S023	REQ

06*	DESIGN_ACTIVITY	Design activity	Tank Automotive & Armaments Command	REQ*
07*	DESIGN_ACTIVITY_LOCATION	Design activity city, state and zip code	Warren, MI 48397-5000	REQ*
08	DESIGN_APPROVAL_NAME	Person who approved design	A.TAYLOR	REQ
09	DESIGN_CONTRACTOR	Original design contractor	STEWART & STEVENSON SERVICES, SEALY TEXAS U.S.A.	-
10	DISTRIBUTION_CODE	Distribution Statement (A, B, C, D, E, X)	A	REQ
11	DRAWING_APPROVAL_NAME	Person who approved drawing	A.TAYLOR	REQ
12	DRAWING_DATE	Date original drawing created	YY-MM-DD	REQ
13	DRAWING_NUMBER	Drawing number the part/assembly is defined on	12414568	REQ
14	*DRAWING_REV*	*Revision of current drawing*	*B*	*Legacy*
15	DRAWN_BY	Person who created original drawing	J.WENNER	REQ
16	ENGINEER_NAME_1	Primary design engineer	A.WURFEL	REQ
17	ENGINEER_NAME_2	Secondary design engineer	J.ZEBROWSKI	-
18	ERR_ECP_APPROVAL	Approving official of latest ECP	N.TAUBE	-
19	ERR_ECP_DATE	Date current revision was released	YY-MM-DD	REQ
20	ERR_ECP_NUMBER	ECP that caused current revision	TACV1234	REQ
21	ITAR	Controlled by International Traffic in Arms Regulation	YES	REQ
22	JEWEL_BEARING	YES-NO PARAMETER, contains jewel bearing	NO	-
23	MATERIAL_ENGINEER	Person who approved material for design	M.CHURCH	REQ
24	MATL_CLASS	Automatically filled out		-
25	MATL_GRADE	Automatically filled out		-
26	MATL_HEAT_TREAT	Automatically filled out		-
27	MATL_NAME	Automatically filled out		-
28	MATL_SHAPE	Automatically filled out		-
29	MATL_SPECNO	Automatically filled out		-

30	MATL_TYPE	Automatically filled out		-
31	MATL_UNS_NO	Automatically filled out		-
32	*MODEL_REV*	*Revision of current model*	*B*	*Legacy*
33	MODELER_NAME_1	Primary Pro/E modeler	J.WENNER	REQ
34	NEXT_ASSY_1	First next higher assy part no. (shall match USED_ON_1 below)	12419847-001	REQ
35	NEXT_ASSY_2	Second next higher assy part no.		-
36	NEXT_ASSY_3	Third next higher assy part no.		-
37	NEXT_ASSY_4	Fourth next higher assy part no.		-
38	NEXT_ASSY_5	Fifth next higher assy part no.		-
39	NEXT_ASSY_6	Sixth next higher assy part no.		-
40	NEXT_ASSY_7	Seventh next higher assy part no.		-
41	NEXT_ASSY_8	Eighth next higher assy part no.		-
42*	NOMENCLATURE	Drawing name. (See paragraph 3.12.3)	BODY ASSEMBLY, CARGO, MTV LWB	REQ*
43	NOMENCLATURE_1	First line of drawing nomenclature	BODY ASSEMBLY,	-
44	NOMENCLATURE_2	Second line of drawing nomenclature	CARGO, MTV LWB	-
45	NOMENCLATURE_3	Third line of drawing nomenclature		-
46	OZONE_DEPLETING_CHEMICAL	YES-NO PARAMETER, ozone depleting chemical	NO	-
47	PART_NUMBER	Item part number	12423205	REQ
48	PMIC	Precious Metal Indicator Code	A	-
49	QA_ENGINEER_NAME	Quality assurance engineer	P.FIESTER	REQ
50	*REMARKS*	*See paragraph 3.12.2*	*PRELIMINARY*	*Legacy*
51	SOURCE_CONTROL	YES-NO PARAMETER, source control	NO	-
52	SPEC_CONTROL	YES-NO parameter, spec control	NO	-
53	SPEC_NUMBER	Specification Number, Used on SPEC Parts only	ASME B18.2.1	-
54	SPECIALTY_METAL	YES-NO PARAMETER, specialty metal	NO	-

55	TOLERANCE_1_PLACE	Default tolerance if indicated on drawing	.1	-
56	TOLERANCE_2_PLACE	Default tolerance if indicated on drawing	.03	-
57	TOLERANCE_3_PLACE	Default tolerance if indicated on drawing	.020	-
58	TOLERANCE_ANG	Default angular tolerance if indicated on drawing	.5	-
59*	UNIT_WEIGHT	Mass value. (See paragraph 3.12.4)	1.25	REQ*
60	USED_ON_1	First used on item	FMTV A1P2	REQ
61	USED_ON_2	Second used on item		-
62	USED_ON_3	Third used on item		-
63	USED_ON_4	Fourth used on item		-
64	USED_ON_5	Fifth used on item		-
65	USED_ON_6	Sixth used on item		-
66	USED_ON_7	Seventh used on item		-
67	USED_ON_8	Eighth used on item		-
68	WEIGHT_UNIT	Automatically filled out - toolkit application	kg	-

TABLE 3: PART PARAMETERS

** These parameter values are required and are automatically generated from the FMTV start part and assembly relations.*

3.12.1 The parameter ***CAGE_CODE*** refers to the original design activity. If the current design activity is different than the original design activity then the current one is entered into the ***CURRENT_CAGE_CODE*** field. A table must also be placed on the drawing showing the current design activity cage code (see sample below).

CURRENT DESIGN ACTIVITY CAGE CODE 19200
US ARMY
ARMAMENT RESEARCH, DEVELOPMENT AND ENGINEERING CENTER
PICATINNY ARSENAL, NEW JERSEY 07806-5000

FIGURE 2: CURRENT DESIGN ACTIVITY CAGE CODE

3.12.2 The parameter ***REMARKS*** should initially default to ***PRELIMINARY***. This will automatically appear on the drawing until the value is removed during the final stage of the release process.

3.12.3 The parameter ***NOMENCLATURE*** will automatically be assigned a value based upon FMTV start part relations. The value is equal to the sum of the individual parameters ***NOMENCLATURE_1***, ***NOMENCLATURE_2***, and ***NOMENCLATURE_3***, which shall be entered appropriately by the modeler.

3.12.4 The part or assembly parameter ***UNIT_WEIGHT*** is automatically filled out by Creo Parametric (PRO_MP_MASS calculated parameter) when the part or assembly is regenerated. This is accomplished using relations found in the FMTV start parts. The parameter value for unit weight shall be representative of the weight of its true physical counterpart.

3.13 Part Units

Creo Parametric system units shall match the following:

Unit type	METRIC	ENGLISH
Length	mm	in
Mass	kg	lbm
Time	sec	sec
Temp	ºC	ºF

TABLE 4: UNIT SYSTEMS

Note: English and metric unit systems exist in the FMTV start parts

4 Assembly Modeling Standards

4.1 Assembly Modeling Practice

4.1.1 Assemblies shall have all components and features of the assemblies defined, including pins, screws, nuts, washers, the part number etc. This includes items such as decals, warning labels, bulk items, etc., which are required to be attached to the assembly.

4.1.2 Assembly of models shall follow a logical sequence.

4.1.3 Assembly features shall only be used for features manufactured at the assembly level and not for features created on individual parts.

4.1.4 All assembly models shall regenerate successfully with all features and components in the unsuppressed status. When assemblies are modified, the next higher assembly shall also be capable of being successfully regenerated.

4.1.5 All assembly components shall be fully constrained unless the function of the assembly benefits from adding certain degrees of freedom through connections. The **FIX** constraint should avoid being used. The use of Creo Mechanism joints, for example, is acceptable in this case.

4.1.6 The latest ***FMTV_START_ASSY*** shall be used for assembly creation.

4.2 Tolerances

Refer to ***Section 3.2 Tolerances***.

4.3 Model Notes

Refer to ***Section 3.5 Model Notes***.

4.4 Assembly Default Datums

All assemblies will start with default datum planes and a default coordinate system identical to those found in the FMTV start parts. The default datum planes shall be named ***FRONT_ASSY***, ***TOP_ASSY***, and ***SIDE_ASSY***. The coordinate system shall be named A***CS0*** with its orientation as follows; Z axis is elevation, X axis is along the length of the vehicle, Y axis across the vehicle width.

4.5 Assembly File Names

Assemblies shall be named using the following convention:

CAGE_PartNumber.asm i.e. 19207_12432300.asm

For additional guidelines for assembly naming refer to ***Section 3.8 Part File Names***.

4.6 Model Views

All models shall have separately named views defining the ***TOP_VIEW***, ***BOTTOM_VIEW***, ***LEFT_VIEW***, ***RIGHT_VIEW***, ***BACK_VIEW***, ***FRONT_VIEW***, and ***DEFAULT*** views. These views exist in the FMTV start part. The ***FRONT*** view is oriented facing the yellow (or primary) side of the ***FRONT*** datum plane, with the yellow side of the ***TOP*** datum plane facing up. The remaining orthogonal views follow standard third angle projection practice. Current drawing models should be created such that the named views correspond to the existing drawing views. When applicable, the addition of ***ROADSIDE*** and ***CURBSIDE*** views is recommended.

4.7 Assembly Layers

The following layers shall be used in assembly mode (these exist in the FMTV start assembly).

	Layer name	Layer Assignments	Default State
a.	ITEM_ID	Only the cosmetic/curve feature describing the part identification number	Hide
b.	DTM_AXES	All datum axes	Hide
c.	DTM_CSYS	All datum coordinate systems	Hide
d.	DTM_CURVES	All curve features	Hide
e.	DTM_PLANES	All datum plane features	Hide
f.	DTM_POINTS	All datum point features	Hide
g.	G_DTM_PLANES	All geometric tolerancing datum planes	Hide

h.	NOTES	All assembly notes	Hide
i.	SURFACES	All surface features and quilts	Hide
j.	THREADS	All cosmetic and actual threads	Hide
k.	XSEC_DATUMS	All cross-section datums	Hide

TABLE 5: ASSEMBLY LAYERS

4.8 Bulk Items

Bulk items are classified into two different categories. These items will be referred to as **TYPE 1** and **TYPE 2** categories. **TYPE 1** bulk items are commodities that are used in a volume, such as adhesives, fluids, lubricants, etc. **TYPE 2** bulk items are cut to length or stock parts, such as seals, gaskets, convoluted tubing, tape, etc.

TYPE 1bulk items are non-solid representations of objects within an assembly. They shall be used for all objects that must appear in the Bill of Material or PARTS LIST but do not require solid models. Bulk items shall be used to represent adhesives, lubricants, sealants, paint, etc. Bulk items are a unique subset of the .prt model type and are represented by a unique icon (paint can) in the Creo Parametric Model Tree. New bulk items can be created by using the ***start_bulk.prt*** template in the Commonspace Templates folder.

It is not acceptable to create a -500 suffix component of the base object in the assembly and add Table relations to overwrite the parameters reported in the columns. The bulk item shall be created with the proper cage code, name, and description to automatically populate the table repeat region. The typical table relation for bulk items relates the reporting the quantity which is typically set to "AR".

TYPE 2 bulk items shall have a Creo Parametric 3D solid model to represent the item. The models will either fall into the deformed/installed state or flexible component category.

4.9 Assembly Parameters

Follow the same guidelines as ***Section 3.12 Part Parameters***.

4.10 Relations

Relations shall have no errors upon regeneration. Relations found in feature ***ACS0*** of the FMTV start assembly are required.

4.11 Surface Finishes

Surface finishes shall be created on the assembly as required.

4.12 Reference Objects

Reference components displayed in phantom on installation or similar drawings should not be fully associated 3D models. These should be developed using 2D curve data or 3D model data as required (refer to ***Appendix D.3 for 2D*** and ***Appendix D.4 for 3D Lightweight Backdrop Creation.*** Any components specifically defined on the drawing must have a valid associative 3D model as per ***Paragraph 4.1.1***.

4.13 Assembly Units

Follow the same guidelines as ***Section 3.13 Part Units***.

5 Drawing Creation Standard

5.1 Dimensions

Drawings shall display dimensions that are directly parametric to the model upon which they are based. Specifically, these values shall be either driving or driven dimensions.

5.2 Geometric tolerances

Geometric tolerances shall be attached to the base dimensions when applicable and not added as unattached notes.

5.3 Formats

Standard FMTV 3D/TDP drawing formats located in the Standard CAD Template Library / Start Parts and Formats in Windchill PDMLink shall be used.

5.4 Drawing Setup Files

Standard FMTV 3D/TDP drawing setup files (.dtl) located in the Standard CAD Template Library / Configs in Windchill PDMLink shall be used.

5.5 Drawing Standards

The drawing standard used shall be ***ASME Y14.100-2013*** and ***ASME Y14.5-2009***. These standards shall be called out in ***Note1*** of the drawing model.

5.6 Notes

All notes on drawings shall originate as model notes in .prt and .asm files, except when deemed necessary.

5.7 Drawing Views

2D draft entities shall be related to the drawing view to which it belongs. Views shall be capable of being moved with no entities of the view being left behind.

5.8 Drawing Line-Weights

The following line weights and colors shall be used:

COLOR	STYLE	ITEMS
WHITE	Normal Weight	object lines, view arrows, format borders
YELLOW	Thin Weight	leader lines, dimensions, notes, symbols, tables
GRAY	Phantom	hidden object lines,
RED	Bold	special lines needing extra emphasis
WHITE	Phantom	Tangent Lines

TABLE 6: DRAWING LINE WEIGHTS

5.9 Quality Assurance Information

The preferred method for identifying Quality Assurance Information is on the product drawing instead of having a separate QAP sheet.

5.10 Distribution Statement

The drawing distribution statement shall appear on each sheet of a drawing. (This is automatically accomplished when the model note ***Distribution_Note*** is filled out in the drawing model.)

5.11 Revision Block

Revision blocks shall conform to ***ASME Y14.35M***

5.11.1 For legacy conversion ECP's, include only the latest revision, redrawn with change/redrawn without change and ECP number in lieu of full delineation of the changes on

the drawing for *Administrative* changes. For individual *Major or Minor* changes, those changes shall be listed in the revision block.

5.11.2 For design effort ECPs that make drawing changes the revision block shallincluding the latest revision, redrawn with change, ECP number and full delineation of the changes on the drawing.

5.11.3 For design effort ECPs that release a new drawing the revision block shall include Initial Release, Product Baseline, and the ECP number.

5.12 Associated Models

All parts that are defined by a drawing shall be related to the drawing even if they are not used in a view on the drawing.

5.13 Parameters

The following parameters shall be filled out for the drawing model:

- Part_Number
- Drawing_Date
- Drawn_By
- Modeler_Name_1
- Checker_Name
- Engineer_Name_1
- Material_Engineer
- Design_Approval_Name
- Nomenclature
- QA_Engineer_Name
- Drawing_Number
- Revision
- Distribution_Code
- Itar
- Err_Ecp_Number

- Err_Ecp_Date*
- Err_Ecp_Approval*

**These parameters are not required to be filled out at the In Work state. Their values shall be determined later in the release process.*

5.14 Layers

Drawing layers may be created as required to properly display information on the drawing. Layer status may be set (this works independent of the part/assembly) as required to properly display information on the drawing.

5.15 File Names

Drawings shall be named using the following convention:

CAGE_DrawingNumber.drw i.e. 19207_12428511.drw

5.16 Drawing BOM Balloons

BOM balloons will use a standard diameter of 0.375 inches (9.525 mm). If the numbers within the balloon extend beyond the balloon then the size of the text will be decreased to best fit within the standard balloon. Refer to ***Appendix C.13*** for additional information.

6 Templates

Creo Parametric object templates are used to customize the default content of newly created objects. Object templates belong to one of two categories: model templates (also known as start parts) and drawing templates. Model templates are standard Creo Parametric models that contain predefined features, layers, parameters, named views, and other attributes. Drawing templates are special drawing files that contain instructions to create drawing items, including views, tables, formats, symbols, snap lines, notes, parametric notes, and dimensions. Drawing templates are not currently used in the FMTV environment. The basis for a drawing is the Creo Parametric format.

6.1 Start Parts

The start parts already contain a default coordinate system, default datum planes, views, layers, notes, relations, and parameters. There are start parts developed for solid models, sheetmetal models, and assembly models as named below.

- **fmtv_start_part.prt**
- **fmtv_start_shtml_part.prt**
- **fmtv_start_assy.asm**
- **fmtv_start_diagram.dgm**

The parameters that are part of the start models are listed below in ***Table 7***, most of which are required by the PdM-MTV 3D/TDP standards.

	Parameter Name	Parameter Description	Format Example	Source
	AGENCY	Agency or service	US ARMY	Relation
	AUTO_PART_NUMBER*	Yes-No, Used in relation	YES	User
	CABLING_ASSY*	Yes-No, Set to YES for Cabling assembly	NO	User
	CAGE_CODE	Original Design Activity CAGE Code	19207	Relation
	CATALOG_NOMENCLATURE	Nomenclature assigned by supply		User
	CHECKER_NAME	Person that checked original	K.BALL	User
	CIRCUIT_NUMBER*	Circuit Number used for Electrical	J159	User
	CONTRACT_NUMBER	Contract # - original design contract	DAAE07-03-C-S023	User
	DESIGN ACTIVITY	Design Activity (from relations)	TANK AUTOMOTIVE & ARMAMENTS COMMAND	Relation
	DESIGN_ACTIVITY_LOCATION	Design Activity City, State and Zip Code (from Relations)	WARREN, MI 48397-5000	Relation
	DESIGN_APPROVAL_NAME	Person who approved design	A.TAYLOR	User

	DESIGN_CONTRACTOR	Original design contractor	STEWART & STEVENSON SERVICES, SEALY TEXAS U.S.A.	User
	DISTRIBUTION_CODE	Distribution Statement (A,B,C,D,E,X)	A	User
	DRAWING_APPROVAL_NAME	Person who approved drawing	A.TAYLOR	User
	DRAWING_DATE	Date original drawing created	YY-MM-DD	User
	DRAWING_NUMBER	Drawing Number	12423205	User
	DRAWN_BY	Person who created original drawing	J.WENNER	User
	ENGINEER_NAME_1	Primary design engineer	A.WURFEL	User
	ENGINEER_NAME_2	Secondary design engineer	J.ZEBROWSKI	User
	ERR_ECP_APPROVAL	Approving official of latest ECP	N.TAUBE	User
	ERR_ECP_DATE	Date current revision released	YY-MM-DD	User
	ERR_ECP_NUMBER	ECP that caused current revision	TACV1234	User
	ITAR	Controlled by International Traffic in Arms Regulation	YES	User
	JEWEL_BEARING	Yes-No Parameter, contains jewel bear	NO	User
	MATERIAL_ENGINEER	Person who OK'd material for design	M.CHURCH	User
	MATL_CLASS	Automatically filled out (from relations)		User
	MATL_GRADE	Automatically filled out (from relations)		User
	MATL_HEAT_TREAT			User
	MATL_NAME			User
	MATL_SHAPE			User
	MATL_SPECNO			User
	MATL_TYPE			User
	MATL_UNS_NO			User
	MC_ERRORS	ModelCHECK number of errors	0	User
	MODELER_NAME_1	Primary Pro/E modeler	J.WENNER	User
	MODEL_CHECK	ModelCHECK last run on object	YY-MM-DD	User
	NEXT_ASSY_1	First next higher assy part no. (must match Used_On_1 above)	12419847-001	User
	NEXT_ASSY_2	Second next higher assy part no.		User
	NEXT_ASSY_3	Third next higher assy part no.		User
	NEXT_ASSY_4	Fourth next higher assy part no.		User
	NEXT_ASSY_5	Fifth next higher assy part no.		User
	NEXT_ASSY_6	Sixth next higher assy part no.		User
	NEXT_ASSY_7	Seventh next higher assy part no.		User
	NEXT_ASSY_8	Eighth next higher assy part no.		User
	NOMENCLATURE	Part Name	BODY ASSEMBLY, CARGO, MTV LWB	Relation
	NOMENCLATURE_1	First Nb_1 characters of Nomenclature (for drawing) (from rel)	BODY ASSEMBLY,	User
	NOMENCLATURE_2	Second Nb_2 characters of Nomenclature (for drawing)(from rel)	CARGO, MTV LWB	User
	NOMENCLATURE_3	Remaining characters of Nomenclature (for drawing)(from rel)		User
	OZONE_DEPLETING_CHEMICAL	Yes-No Parameter, Ozone depleting chemical	NO	User

	PART_NUMBER	Item part number	12423205	Relation
	PIPING_ASSY*	Yes-No Parameter, set to YES for	NO	User
	PMIC	Precious Metal Indicator Code	A	User
	PTC_COMMON_NAME*	PTC Common Name		User
	QA_ENGINEER_NAME	Quality Assurance engineer	P.FIESTER	User
	REMARKS	General Remarks	PRELIMINARY	User
	SOURCE_CONTROL	Yes-No Parameter, Source Control	NO	User
	SPEC_CONTROL	Yes-No Parameter, Spec Control	NO	User
	SPEC_NUMBER	Specification Number (SPEC Parts Only)	ASME B18.2.1	User
	SPECIALTY_METAL	Yes-No Parameter, Specialty Metal	NO	User
	TOLERANCE_1_PLACE	Default tolerance if indicated on drw	.1	User
	TOLERANCE_2_PLACE	Default tolerance if indicated on drw	.05	User
	TOLERANCE_3_PLACE	Default tolerance if indicated on drw	.030	User
	TOLERANCE_ANG	Default tolerance if indicated on drw		User
	UNIT_WEIGHT	Mass computed in Creo Parametric	1.25	User
	USED_ON_1	First used on item	FMTV A1R	User
	USED_ON_2	Second used on item		User
	USED_ON_3	Third used on item		User
	USED_ON_4	Fourth used on item		User
	USED_ON_5	Fifth used on item		User
	USED_ON_6	Sixth used on item		User
	USED_ON_7	Seventh used on item		User
	USED_ON_8	Eighth used on item		User
	WEIGHT_UNIT	Unit - Weight	kg	Relation

TABLE 7: OBJECT PARAMETERS

** Non-Designated Parameters*

6.2 Formats

PdM-MTV standard formats have been created to work with the PdM-MTV start parts. These formats are available in the Windchill PDMLink / STANDARD CAD TEMPLATE LIBRARY / Start Parts and Formats. These are the only formats that shall be used when creating new or updating old drawings for the FMTV database. These formats are named using the following convention:

(Drawing Size)_(Vehicle Family)_(Unit System)_(1)_(Version)

- **b_fmtv_met_1_5.frm**
- **c_fmtv_eng_1_5.frm**
- **c_fmtv_met_1_5.frm**
- **d_fmtv_eng_1_5.frm**
- **d_fmtv_met_1_5.frm**
- **e_fmtv_eng_1_5.frm**
- **e_fmtv_met_1_5.frm**
- **f_fmtv_eng_1_5.frm**
- **f_fmtv_met_1_5.frm**
- **d_fmtv_diag_1_1.frm**
- **e_fmtv_diag_1_1.frm**

7 Electrical

7.1 Schematics

7.1.1 Schematic File Names

Schematic naming convention shall follow the format below: (Note: Schematics are used for reference to drive the overlay that is contained on the production drawing)

CageCode_PartNumber

Example:

19207_12506185.dgm
19207_12506185-001.dgm (For tabulated harnesses.)

7.1.2 Schematic Overlay Process

Schematics shall be created using an overlay process. This procedure will insert an overlay of the diagram into the drawing. When an overlay is used, anytime a diagram updates or changes, it will change on all drawings where the overlay exists. Overlays will not create a snapshot in time of the present state of a diagram, it will update with any changes parametrically.

Overlays will also require that the formatting be turned off in the diagram and the diagram saved in that condition. If it is not turned off, there will be noticeable differences in the title blocks and formatting. The formatting in a diagram is different from that of a drawing.

The overlay will need to be placed on a blank sheet in the drawing. This can be used to bring diagram data into the drawing and relay the changes in the diagram to the drawing, but revisions will not be possible from within the drawing. Use the rev block from the drawing to track the changes for the diagram on the overlay sheet. Alternatively, the user can create a rev block in the diagram to force the information to be updated in the diagram. It also keeps the diagram ready to be printed independently. The rev block that would be on the blank sheet in the drawing gets deleted to prevent a duplicate rev block from showing on the drawing.

Diagram Setup:

- Turn off the page formatting on the sheet to be overlaid.
- Place a rev block on the diagram (This is only needed if the rev block from the drawing is not utilized)

The diagram may need BLANK sheets added for the automatic tables to be populated with the correct sheet number.

Drawing Setup:

Insert a blank sheet where the overlay should appear.

Delete the rev box from the new blank sheet; the diagram rev box will be used.

Insert the diagram overlay.

How to Insert an Overlay

Select Tools > Overlay

Select Add Overlay

Select Place Sheet

Select the diagram file to be inserted

Select Open

If the diagram is more than 1 sheet, it will ask which sheet to insert.

Converting a Schematic that was previously Stand-Alone to Overlay

If a schematic already exists in the system as a standalone 19207_SC12345678.dgm file, the following process should be used to incorporate the schematic into the drawing.

1. Rename the schematic from 19207_sc12345678.dgm to 19207_12345678.dgm
2. Create a DUPLICATE OBJECT of the existing schematic and name it 19207_sc12345678.dgm (This is needed since the original, newly renamed, schematic will still drive the drawing; however a schematic is needed to document the superseded schematic from a CM perspecitve).

3. On the DUPLICATE OBJECT schematic, add SUPERSEDED BY 12345678 above the title block and update the rev block to state that the schematic was superseded and incorporated into drawing 12345678.
4. Follow Creating a Diagram Overlay method.

7.1.3 Spool File Requirements

Spools should only be checked in by a Spool administrator. Only Spool administrators have the rights to check items into the Spools folder and create or modify Spools within this folder. All Spools should be checked in and stored in the ***Routed Systems/Spools*** LIBRARY in Commonspace.

Spool naming convention shall follow the format below:

CageCode_PartNumber (Government Part Number or Spec Number)

Example:

19207_12423132-004.spl (Government Part Number)

81349_M16878/14BJE9.spl (Spec. Number)

Spool files shall include the following parameters:

Parameter Name	Parameter Description	Used On	Example Value
Name	Mil Spec Number of Governement Number of the wire, tubing, etc…	All Spools	19207_12420924-001
Cage_Code	Cage Code used to fill BOM Table.	All Spools	19207
Description	Description used to fill BOM Table.	All Spools	Electrical, 18 AWG, White
Units	Unit of Measurement	All Spools	mm/inch
Mass_Units	Unit of Measurement	All Spools	kg
Density	Density	All Spools	kg/mm^2
Color	Color of the cable, wire insulation, or sheathing *(See Table 20)*	All Spools	White
Color_Code	Code for the Color *(See **Table 20**)*	All Spools	WHT
Insul_type	Insulation Type	Wire/Cable Spools	GXL
Wire_Gauge	Wire gauge	Wire/Cable Spools	18 AWG
Min_Bend_Radius	The minimum radius a wire/cable/sheath can bend.	All Spools	8
Thickness	The diameter of the wire/cable.	Wire/Cable Spools	2
Number_of_Conductors	The number of conductors in a cable.	Cable Spools	5
Sheath_Type	Specifies the type of sheathing for this spool.	Sheath Spools	Tape, Tube, Shrink
Outer_Diameter	Indicates the outer diameter of the tube sheathing.	Sheath Spools	12
Preshrink_Inner_Diameter	Indicates the pre-shrunk diameter of SHRINK sheathing.	Sheath Spools	10
Wall_Thickness	Indicates the thickness of the sheathing.	Sheath Spools	0.5
Width	Specifies the tape width for tape sheathing.	Sheath Spools	20

TABLE 8: SPOOL FILE PARAMETERS

Not all parameters will be used depending on the type of spool used. Use all of the parameters above when applicable.

Color	Color Code
White	WHT
Black	BLK
Red	RED
Blue	BLU
Green	GRN
Yellow	YLW
Orange	ORG
Gray	GRY
Violet	VLT
Brown	BRN
Tan	TAN
Pink	PNK

Table 9: Color Code References

7.1.4 Schematic Spacing

Schematic shall use the minimum spacing required to prevent overlapping while being clearly visible when printed on an A size sheet.

7.1.5 Schematic Tables

Schematic shall include a wire run table showing the following information:
Circuit, Wire, From, To, Color, Color Code, Gauge, Insulation, and Description.

Use table **diagram_circuit_list.tbl** located in Commonspace

CIRCUIT	WIRE	FROM COMPONENT, TERM	TO COMPONENT, TERM	COLOR	COLOR CODE	GAUGE	INSULATION	DESCRIPTION
21-489	12423132-004	TL255, TERM: 1 LOC: SH 1, 6-H	J250, TERM: 1 LOC: SH 1, 3-H	WHITE	WHT	14	GXL	RIGHT FRONT MARKER FEED
21-489-A	12423132-004	TL254, TERM: 1 LOC: SH 1, 6-G	E1, TERM: 1 LOC: SH 1, 5-G	WHITE	WHT	14	GXL	MARKER FEED
21-489-B	12423132-004	TL251, TERM: 1 LOC: SH 1, 6-G	J254, TERM: 1 LOC: SH 1, 3-E	WHITE	WHT	14	GXL	RIGHT CENTER MARKER FEED
21-489-C	12423132-004	TL250, TERM: 1 LOC: SH 1, 6-G	J255, TERM: 1 LOC: SH 1, 3-E	WHITE	WHT	14	GXL	RIGHT REAR FEED
21-489-D	12423132-004	J250, TERM: 1 LOC: SH 1, 3-H	J251, TERM: 1 LOC: SH 1, 3-G	WHITE	WHT	14	GXL	LEFT FRONT MARKER FEED

FIGURE 3: DIAGRAM_CIRCUIT_LIST.TBLE EXAMPLE

7.1.6 Schematic Circuit Numbers

Schematic circuit numbers shall follow the numbers assigned in document **12506111**. Circuit numbers that are used multiple places require an alpha-extension. (Example: 90-A, 90-B…)

7.1.7 Circuit Description

Circuits shall show the circuit number, gauge, and color code on each wire. Circuit description shall come from document **12506111**.

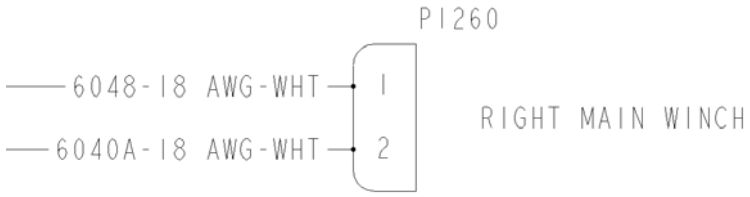


FIGURE 4: CIRCUIT DESCRIPTION EXAMPLE

7.1.8 Reference Designators

Schematic reference designators shall follow the numbers assigned in document **12506111**.

Common Reference Designators		
Designator	**Description**	**Example**
CBXX	Circuit Breakers	CB01
TLXX	Terminal Lugs	TL01
EXX	Splices	E01
PXXX	Male Connector	P101
JXXX	Female Connector	J101
TBXX	Terminal Box	TB01
KXX	Relays	K01
SXX	Switch	S01

Table 10: Common Reference Designators

7.1.9 Connector Description

Connectors shall show the reference designator, connector function, and pin-out orientation on every connector. ***Refer to Figure 5***

FIGURE 5: CONNECTOR EXAMPLE

7.1.10 Phantom Box Requirements

Phantom boxes shall be used when a system or sub-system is shown in a simplified form. Phantom boxes require a label of 6.35 mm (.25 inch) height that describes the sub-system and shall use the phantom line type. ***Refer to 6***

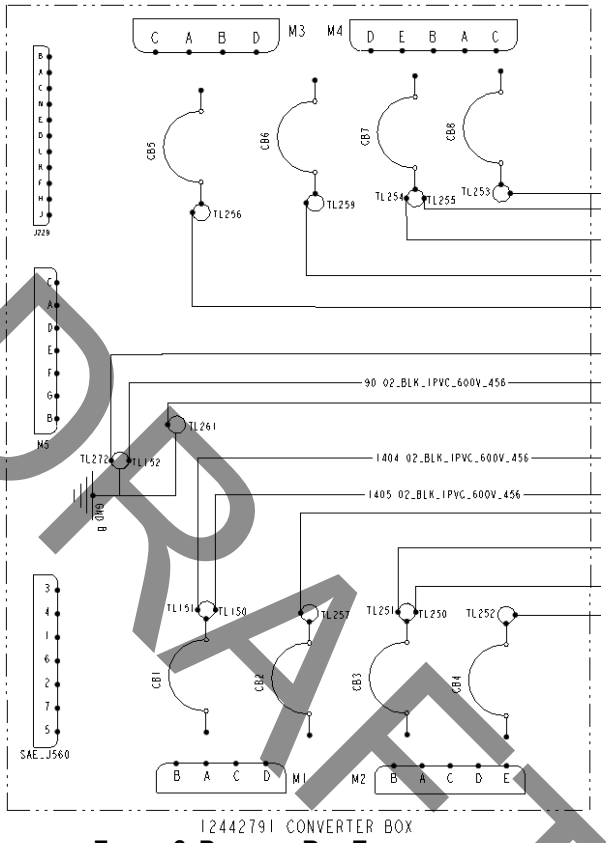


FIGURE 6: PHANTOM BOX EXAMPLE

7.1.11 Schematic Symbols

Schematic symbol text orientation shall be primarily left to right and secondary bottom to top. The symbol should be descriptive and should show reference designators and pins. When printed on an A size print, the symbol should be clearly visible.

7.1.12 Twisted Pairs/Shields

Twisted Pair wires shall be shown as needed per design intent, while shields should always be shown. (***Refer to Figure 7***: Rigid or Smooth lines are acceptable)

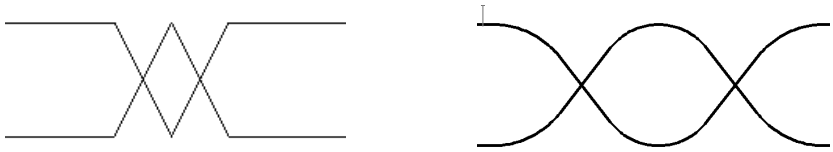


FIGURE 7: TWISTED PAIR(S) EXAMPLE - RIGID VS. SMOOTH

7.1.13 Splices

Butt splices, not through splices shall be used when splicing wires.
Note: Splices are a manufacturing process and not a part.

7.1.14 Wire Breaks

Wire breaks shall be depicted as follows, with the text indicating (SH.#, REF_DES).

(***Refer to Figure 8***)

FIGURE 8: WIRE BREAK EXAMPLE

7.1.15 Schematic Layout

The schematic should flow from left to right.

7.1.16 Top-Level Schematics

Top-level schematics shall be organized by function; meaning one function per page. (Example: ABS, CTIS). Phantom box section above applies to top-level schematics. A visual representation of connector contact gender shall be shown.

7.1.17 Start Part (Diagram)

Refer to ***Section 6.1 - Start Parts***

7.1.18 Formats (Diagram)

Refer to ***Section 6.2 – Formats***

7.1.19 Schematic Diagram File

The schematic diagram file shall be promoted along with the drawing and models. The parameters for the diagram file shall match those of the drawing and models. The schematic revision shall also match the revision of the drawing where the overlay is located on.

7.1.20 Source Control Drawing Schematics

At a minimum, an Interface Control Diagram shall be included on Source Control Drawings; however, a schematic is preferred when sufficient data is available.

7.1.21 Drawing Properties

Diagram files shall use fmtv_diagram_1_1.dtl to provide correct setup for diagram

7.2 2D Drawings

7.2.1 Tables

Two Dimensional drawings shall include a Bill of Material table listing all components used and showing reference designators. Also, a harness variation table shall be shown listing all harness variations. **harness_bom.tbl (or harness_drawing_table.tbl)** table (***Figure 9***) can be utilized which is located in Commonspace.
Some relations may have to be written in order for the table to look correct. Two tables are listed as either can be used based on preference.

FIND NO (XX / XX)	QTY	PART NUMBER	DESCRIPTION	CAGE CODE	REFERENCE DESIGNATOR
12	2000 mm	AA55809-A5IC	TAPE, INSULATED, ELECTRICAL	58536	
11	150 mm	12420956	TAPE, GLASS CLOTH	19207	
10	300 mm	12420924-001	TUBING, CONVOLUTED	19207	
9	150 mm	12420929-001	SLEEVING, INSULATION	19207	
8	2743 mm	12420924-002	TUBING, CONVOLUTED	19207	
7	9145 mm	12423132-004	WIRE, ELECTRICAL, GXL	19207	
6	3	MS27144-2	CONN, PLUG, ELECT SOCKET	96906	B3, B2, B1
5	3	MS20659-127	LUG, TERMINAL	96906	TL200, TL199, TL185
4	6	M43436/1-1	BAND MARKER	81349	
3	1	ATC-30	FUSE, 30 AMP	1UW16	
2	1	12424956	FUSE/CIRCUIT BREAKER HOLDER, ATC	19207	
1	1	12420928-007	MARKER, HEAT BONDABLE WIRE	19207	
PARTS LIST					

FIGURE 9: HARNESS_BOM.TBL SAMPLE

7.2.2 Connector Pin Position

Connector pin position shall be shown from the front face and label as the front. Reference designator and function shall be shown as required. Empty terminals will be plugged per **12420908**.

7.2.3 Connector Description

Connector reference designators and function/description shall be shown. All connectors shall show all components with pin outs.

7.2.4 Harness Dimensions

Harnesses shall be dimensioned from connector front face, centerline of bundle, or center of eyelet. Dimension units are vendor specific.

7.2.5 Two Dimensional Drawing Notes

Two Dimensional Drawings shall include the following notes:

- SCOPE AND GENERAL REQUIREMENTS FOR MANUFACTURING AND ACCEPTANCE OF ELECTRICAL CABLE ASSEMBLIES SHALL BE IAW DRAWING 12420908.
- DIMENSIONS WITHOUT TOLERANCES SPECIFIED ARE FOR REFERENCE ONLY AND SHALL NOT BE USED FOR INSPECTION PURPOSES.

- ITEM IDENTIFICATION: APPLY THE FOLLOWING MARKING IAW MIL-STD-130,
  CAGECODE-PARTNUMBER-REV
  MFR- MANUFACTURER'S CAGE CODE
  DATE MFR YYYY/MM/DD
- ALL CIRCUIT NUMBERS AND REFERENCE DESIGNATORS SHALL BE IAW 12506111.
  (This note is used ONLY if the drawing has a schematic located on it)
- Any note required to show design intent or to clarify the manufacturing process.

7.2.6 Splices

Splices locations shall be dimensioned and noted with the reference designator.

7.2.7 2D Harness Drawing Nomenclature

- **Harness** - A combination of wires, cables, connectors, and/or related components.
- **Cable Assembly** - Multiple conductors, single sheath between two connectors.
- Generic names are no longer acceptable (i.e. No drawing shall be titled CABLE ASSY)

7.2.8 2D Harness Layout

When a harness has too much detail to show on a single page, detailed views shall be used on additional pages.

7.3 3D Harness Modeling

7.3.1 Naming Convention

Refer to ***SECTION 3.8 - Part File Names*** as 3D harnesses follow the standard naming convention
Sample Naming Convention:
The naming convention follows the following format for Routed Electrical Harnesses
(*Note: do not use the same extension for an assembly and a part*)

19207_12345678-600.asm (Routed Assembly)
19207_12345678-601.prt (Harness Part, within Routed Assembly)
19207_12345678-650.mfg (Manufacture Part)
19207_12345678-651.asm (Flat Harness Sub Assembly)
19207_12345678-652.asm (sub-assembly in the manufacture part)

7.3.2 Connectors

Connector assemblies shall be used for connectors requiring other components (I.E. contacts, plugs, TPA…) Connector assembly names shall follow the following convention:

CageCode_HarnessNumber-ReferenceDesignator.asm
(i.e. 19207_12345678-J01.asm)

Connector entry ports shall be named according to the pin name. For example if the pin is “1”; the entry port shall be named “PORT_1” or if the pin is "A"; the entry port shall be named

"PORT_A". ("ENTRY_PORT" is also acceptable for generic ports) ***Refer to Figure 10 for example***

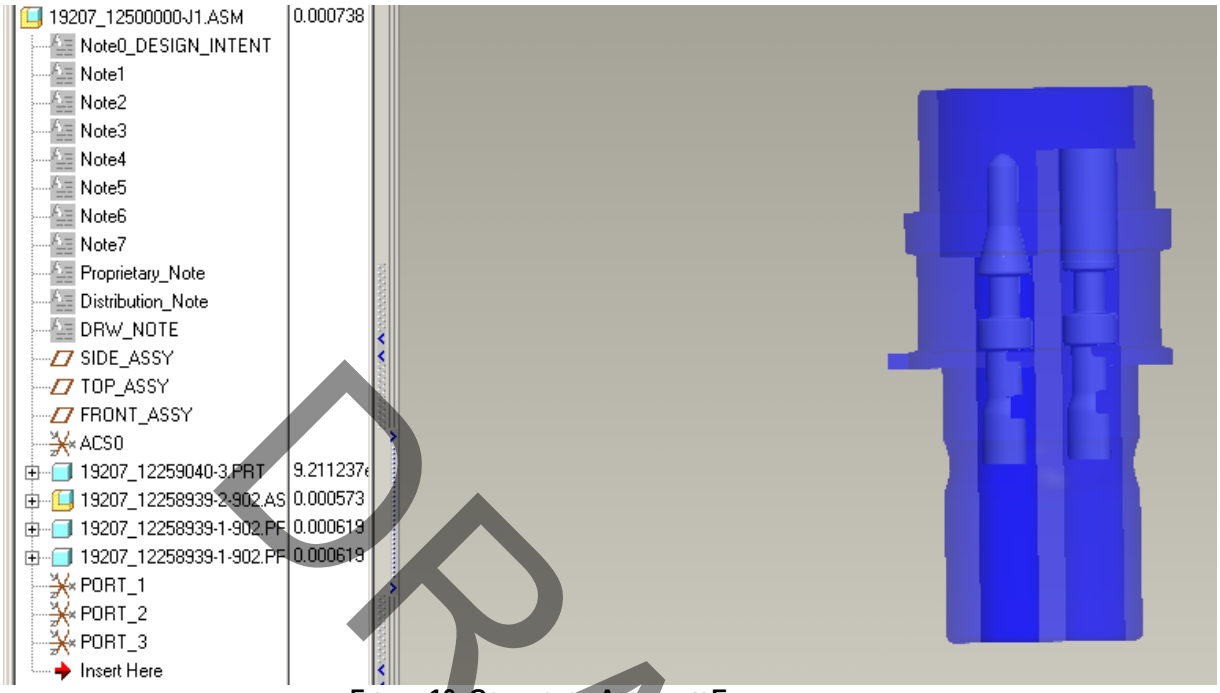


FIGURE 10: CONNECTOR ASSEMBLY EXAMPLE

7.3.3 Interferences

Wiring interferences are only acceptable within its own harness part.
Enclosures: All wires should be contained within the box. Multiple harnesses are allowed to intersect within the enclosure only if interference is unavoidable.

7.3.4 Logical Reference

3D harness models shall be “driven” by a schematic and not created without reference to a schematic.

7.3.5 Splices

19207_Splice.prt shall be utilized, the part is located in the / STANDARD CAD TEMPLATE LIBRARY / Start Parts and Formats folder in Commonspace.

7.4 Electrical Checklists

Harness Assembly Checklist

Harness Assembly

- ☐ File Name: CageCode_HarnessPartNo-600.asm
- ☐ Skeleton Name: CageCode_HarnessPartNo_skel.asm (***Skeleton is NOT required, applies to all steps below***)
- ☐ All connector assemblies are assembled to a coordinate system in the harness skeleton part
- ☐ The harness path coordinates, points, and axes are all in the skeleton part
- ☐ All harness parts (i.e. band markers, ….) are assembled to a coordinate in the skeleton part
- ☐ The correct layers have been created in the assembly following the modeling standards
- ☐ All unnecessary layers are hidden in the skeleton and assembly
- ☐ The assembly and skeleton were created using the correct start part
- ☐ The skeleton features have names that represent each features purpose, i.e. a coordinate system for a connector is named using the reference designator of the connector.
- ☐ Band markers have been placed behind the connectors following the manufacturing standards
- ☐ The harness identification marker has been placed
- ☐ Splices have been attached to a location point on the harness network
- ☐ The correct splice part has been used, splice.prt
- ☐ The harness assembly does not have any interference with objects along the routing path and does not interfere with other harnesses. Wiring interferences within the assembly are acceptable.
- ☐ The correct assembly structure was used, following all the electrical modeling standards
- ☐ Is the unit weight accurate and reasonable for the harness
- ☐ All parameters have been completed
- ☐ Model check has been completed on this assembly

Connector Assemblies

- ☐ File Name: CageCode_HarnessPartNo-RefDes.prt
- ☐ Skeleton Name: CageCode_HarnessPartNo-RefDes_skel.prt (***Skeleton is NOT required, applies to all steps below***)
- ☐ All connector parts are assembled to a coordinate system in the skeleton part
- ☐ Entry port coordinate systems are named Port_# and are located in the main assembly
- ☐ Entry port coordinate systems have the positive z axis pointing out of the connector base
- ☐ The correct layers have been created in the assembly following the modeling standards
- ☐ All unnecessary layers are hidden in the connector skeleton and assembly
- ☐ The assembly and skeleton were created using the correct start part
- ☐ The skeleton features have names that represent each features purpose, i.e. a connector refdes
- ☐ The connector assembly includes all parts that are needed for the connector assembly
- ☐ Nomenclatures match the harness assembly with the refdes included for each connector assembly
- ☐ Correct assembly structure was used following the electrical modeling standards
- ☐ Correct cage code was used on any new connector parts, i.e. manufacturer cage codes
- ☐ Model check has been completed on each connector assembly

Harness Part

- ☐ File Name: CageCode_HarnessPartNo-600.prt
- ☐ Logical reference has been established
- ☐ All connectors and components have been designated as cabling components
- ☐ All entry ports have been designated
- ☐ All wiring features have been created from the logical reference
- ☐ A network has been created for the harness part's routing path (optional)
- ☐ All wires and cables have been routed and marked complete
- ☐ Bundles representing convoluted tubing and insulation sheathing have been created and routed
- ☐ Bundles have been accurately created following manufacturing standards
- ☐ All non cabling components have been attached to the harness part, i.e. band markers, …
- ☐ All Spools needed for the harness, i.e. wires, sheaths, and tapes, have been read into the harness
- ☐ The harness part was created using the correct start part
- ☐ No cosmetic cabling features have been created in the harness, i.e. tape or markers.
- ☐ The correct layers have been created in the assembly following the modeling standards
- ☐ Model check has been completed for this part. (Part must be opened from Pro/INTRALINK)

Detail Drawing Checklist

Drawing File Setup

- ☐ Correct format has been used.
- ☐ Title block information is correct without spelling errors
- ☐ Verify correct distribution and proprietary notes are used from 782 form
- ☐ Distribution and proprietary notes are place on all sheets in the correct locations
- ☐ Drawing title accurately represents the harness, it is not a generic title
- ☐ Default part is the lower level flattened harness assembly.
- ☐ There are no drawing items interfering with the rev block or standard notes area
- ☐ The correct layers are in the drawing and hidden
- ☐ Model check has been completed on the drawing

Harness Layout and Views

- ☐ A full layout view has been placed with overall dimensions
- ☐ Detailed views are used for branches with several connection points
- ☐ Harness scale is large enough to recognize the connectors and components
- ☐ View is oriented to show the flat harness correctly
- ☐ Cable display is set to thick on each view of the harness
- ☐ No hidden lines are displayed within a view
- ☐ Connector pin out views have been placed next to each connector

Harness Dimensions

- ☐ Dimensions along the main trunk are created from a single starting point on the main trunk
- ☐ Branch Dimensions are created from a single starting point from the main trunk
- ☐ Dimensions are from the face of the connector to the breakout starting location
- ☐ Dimensions for ring terminals are from the center of the eyelet to the breakout starting location
- ☐ There are no stacked dimensions along a branch or the main trunk
- ☐ Dimensions match those specified by the engineer, may require overwritten dimensions
- ☐ Splice locations are dimensioned. If several are in one area, a detailed view should be used
- ☐ All components along a path have a dimension to reference their location

BOM Table and Balloons

- ☐ BOM balloons are auto generated for all non cabling parts in the harness
- ☐ BOM balloons for cabling components have been manually created, i.e. wires, tapes…
- ☐ BOM balloons are separated for all occurrences of a component
- ☐ BOM balloons are merged to a single leader line if several are present for a single component
- ☐ BOM balloons should clearly mark where different wire spools are used
- ☐ Use 3-region BOM table: Bottom region is the flat assembly, middle region is harness part, and top region is the 3D harness assembly.
- ☐ Items not needed in the table are filtered out of the table
- ☐ BOM Table has been sorted by reference designators

Notes and Rev Block

- ☐ Notes and Revision information should be generated in the lower level flat harness assembly.
- ☐ Notes have been created from the standard notes library
- ☐ All connectors will have a functional description
- ☐ All connectors, components, and splices are labeled with their reference designator
- ☐ The drawing contains the minimum required notes (See below)
- ☐ Any special requirements for manufacturing or installation have been noted on the drawing

Minimum Required Notes:

APPLICABLE STANDARDS/SPECIFICATIONS:

A. ASME Y14.100-2013

B. ASME Y14.5-2009

IEEE/ASTM SI 10 SHALL BE USED

IN CONVERTING AND ROUNDING OFF.

1 INCH = 25.4 mm APPLIES.

SCOPE AND GENERAL REQUIREMENTS FOR

MANUFACTURING AND ACCEPTANCE OF ELECTRICAL

CABLE ASSEMBLIES SHALL BE IAW DRAWING
12420908.

ITEM IDENTIFICATION: APPLY THE FOLLOW
MARKING IAW MIL-STD-130,
CAGECODE-PARTNUMBER-REV
MFR- MANUFACTURER'S CAGE CODE
DATE MFR YYYY/MM/DD

DIMENSIONS WITHOUT TOLERANCES SPECIFIED ARE
FOR REFERENCE ONLY AND SHALL NOT BE USED FOR
INSPECTION PURPOSES.

ALL CIRCUIT NUMBERS AND REFERENCE DESIGNATORS SHALL BE IAW 12506111.

Diagram Checklist

Sheet Setup

- ☐ Correct format (e_fmtv_diag_1_2.frm)
- ☐ Electrical_Diagram DTL File has been loaded or Start Template is used
- ☐ All parameters for the diagram are completed
- ☐ Grid set to 6mm Spacing
- ☐ Snap to Grid is on when placing wires and components
- ☐ File Name is CageCode_PartNo.dgm
- ☐ Nomenclature 3 is set to “SCHEMATIC”
- ☐ Scale set to NONE
- ☐ Weight set to -----
- ☐ Diagram has been checked for spelling errors
- ☐ All Layers are hidden.

Connectors & Symbols

- ☐ Connectors have a unique reference designator, verified with the standards…
- ☐ Connectors have complete parameters, read in from a library file
- ☐ Connector pin designations are centered and the Reference Designator is at the top or left
- ☐ Connectors have 12mm pin spacing and/or single pin connectors are resized to 26mm
- ☐ Connectors and symbols have a description/function note
- ☐ All connection nodes are on the grid
- ☐ Symbols have a reference designator assigned and displayed
- ☐ Symbols have complete parameter files, read in from a library file if available
- ☐ Correct splice symbols are used
- ☐ All connectors and symbols have Entry Ports entered in the parameters
- ☐ Connectors are aligned and organized on the sheet

Wires & Cables

- ☐ Wires and cables are using the correct spool file
- ☐ Wires and cable conductors have a circuit number assigned, verified with the standards…
- ☐ Wires have a functional description added in the DESC parameter
- ☐ Wires have a parametric note for the description at the connector pin
- ☐ Wires are organized neatly on the sheet with as few crossovers as possible
- ☐ Wire labels and wire descriptions are spaced 1 grid away from the connector
- ☐ Wire labels are aligned with one another at the connector
- ☐ Wire label format: Circuit#-Gauge AWG -ColorCode
- ☐ Wire break symbols are aligned
- ☐ Wires are spaced evenly and aligned on the grid
- ☐ Wire labels are readable and have no interferences
- ☐ Twisted wires are shown using jogs and aligned with one another, along the same side of the sheet
- ☐ Cable Symbols are used to indicate cable assemblies
- ☐ Cable conductors are assigned correctly in the cable

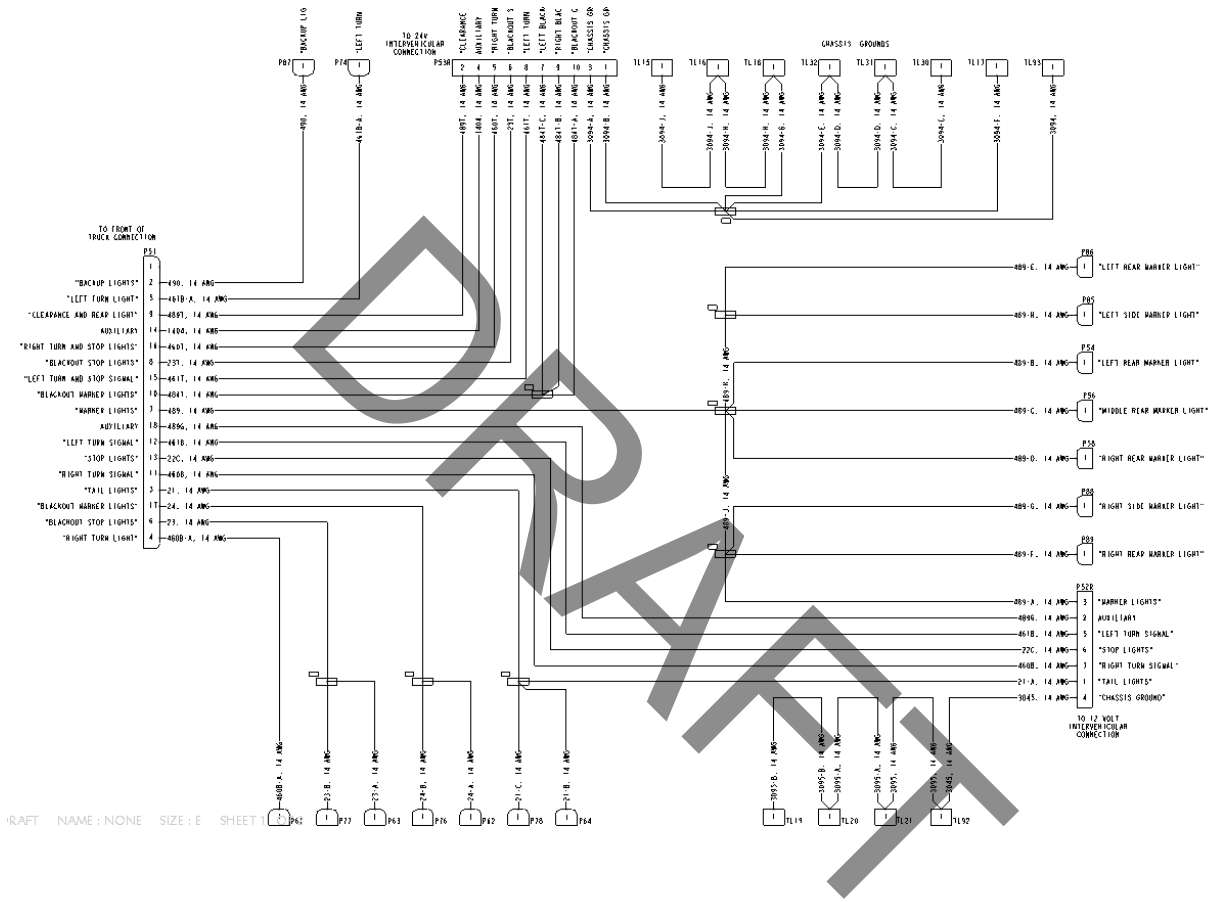
Tables

- ☐ Circuit run table has been placed (diagram_circuit_list.tbl in common space)
- ☐ Text in table cells is visible, column with is adequate to display contents
- ☐ Verify that no information is missing from table and all circuits have completed parameters

Notes and Revision Block (No longer needed with OVERLAY format)

- ☐ Required Diagram Notes have been placed (See Below)
- ☐ Rev Block has been completed, correct format
- ☐ Revision level matches the drawing and assembly
- ☐ Distribution note has been placed
- ☐ Proprietary note has been placed, if required
- ☐ Distribution and Proprietary notes match the 782 form and the drawing

Example of a Schematic Layout:



8 Piping

8.1 General Overview

Piping assemblies shall be created in a way that maintains a ***Stand Alone*** status. The fittings and/or components used to set the start and end points of a route shall be able to be deleted or changed at any time without causing a problem / failure to the pipe. The pipes shall ***not*** be dependent on the top-level assembly where they will be installed. If this is not done the pipe can collapse or change with the slightest modification to a fitting or a coordinate system (CS0) at a later time.

Piping routes shall be created in their own separate Creo Parametric assembly file to break any ties to the master assembly. Isolating the piping routes keeps the routes and all of its extensions separate and contained. The new piping assembly is created using the ***FMTV_Start_Assembly*** file. A part shall be fabricated later using the same base part number with "_p" and the extension of ***.prt***. This part file is a dependent of the assembly file and is created from the pipe route in the assembly file.

The piping assembly file shall be fully constrained in the upper level assembly. The actual constraints are left to user discretion. Components and CS0s shall be placed to set the start and end points for the route. Components such as fittings may be constrained to vehicle geometry to set initial locations, but require an edit definition to break the association to existing geometry in order to establish a standalone model.

Note: All Piping Assemblies and Line Stocks shall be created in METRIC units only.

8.2 Line Stock Overview

A ***Line Stock*** file contains the shape, size, and ***material*** needed to be projected along the piping path in the fabrication process. The Line Stock file defines the physical attributes of the hose. Material and mass properties are defined with the Line Stock file and shall not be duplicated by entering material specification into the material file of the part. The minimum bend radius of the hose shall also be defined in the Line Stock file.

Line stocks should be pulled from the / STANDARD CAD TEMPLATE LIBRARY / Routed_Systems / Line Stock LIBRARY in Commonspace. The Line Stock folder is controlled by Line Stock administrators. Only Line Stock administrators have the rights to check items into the Line Stock folder and create or modify Line Stocks. All Line Stocks shall be checked in and stored in the Routed Systems/Line Stock folder in Commonspace; ***see Figure 11.***

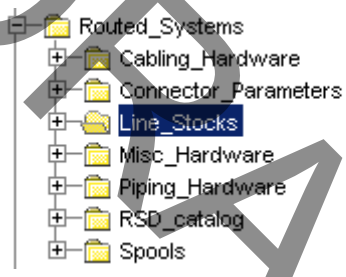


FIGURE 11: COMMONSPACE FOLDER STRUCTURE

Naming Convention of Line Stocks:

- Each file is created in the master library and is named according to the MIL specification or -SAE number in addition to the diameter size and type; ***see Figures 12 & 13.***

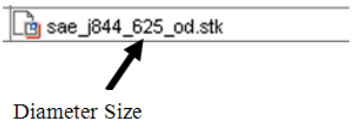


FIGURE 12: SAE LINE STOCK EXAMPLE

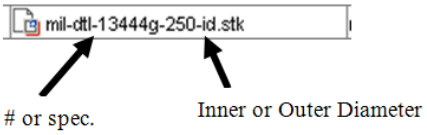


FIGURE 13: MIL SPEC LINE STOCK EXAMPLE

- If no specification is available, the new line stock should be named after the part that is using the pipe. This may be applicable to Source Control Drawings.
- If the type and size of line-stock does not exist in Commonspace, send the line stock specification to a line stock administrator so it can be created.
- The Line Stock is assigned first in the set-up process of creating a route; ***see Figure 14.***

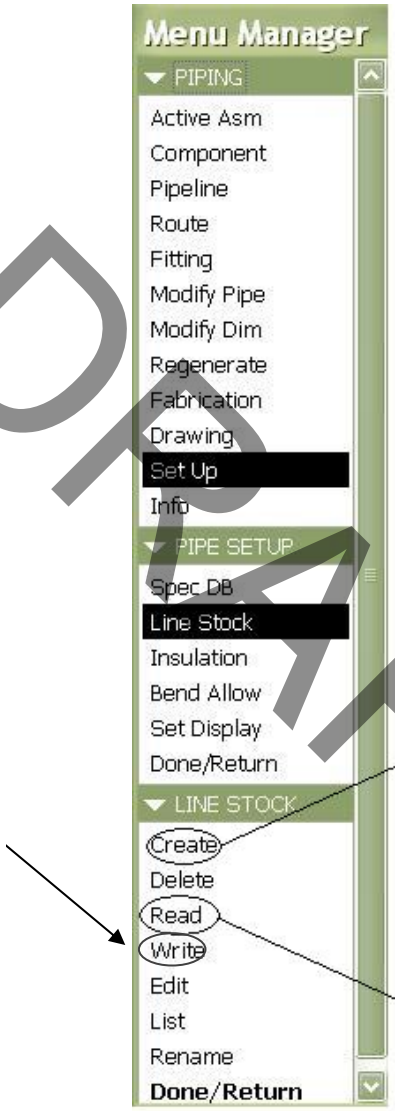
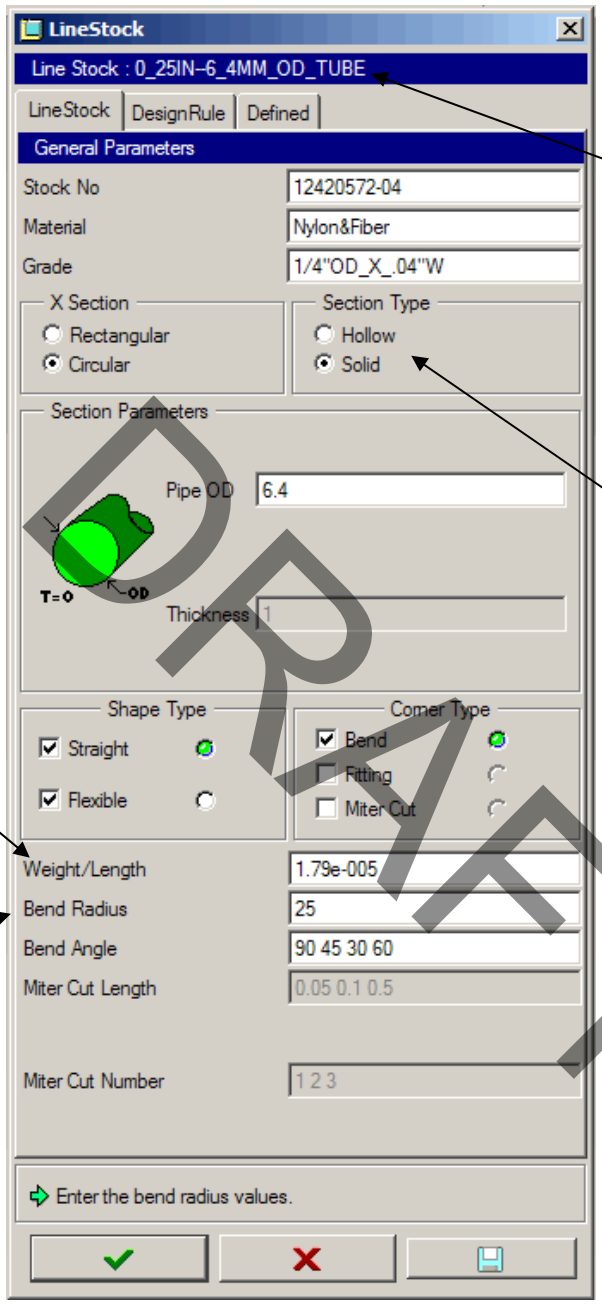


FIGURE 14:LINE STOCK CREATE / READ

NOTE: Line Stocks shall be done in METRIC units.

- Line Stocks contain information that is directly related to the pipe part; ***see Figures 15, 16, and 17.***



Line Stock Type

Section Type can be set to **Solid** or **Hollow** depending on user preference. This will not affect the Weight/Length outcome.

This is the weight per length of the hose or tube. This number is provided from the manufacturer or spec. callout in kg/mm.

Minimum Bend Radius in mm

FIGURE 15: LINE STOCK SET-UP

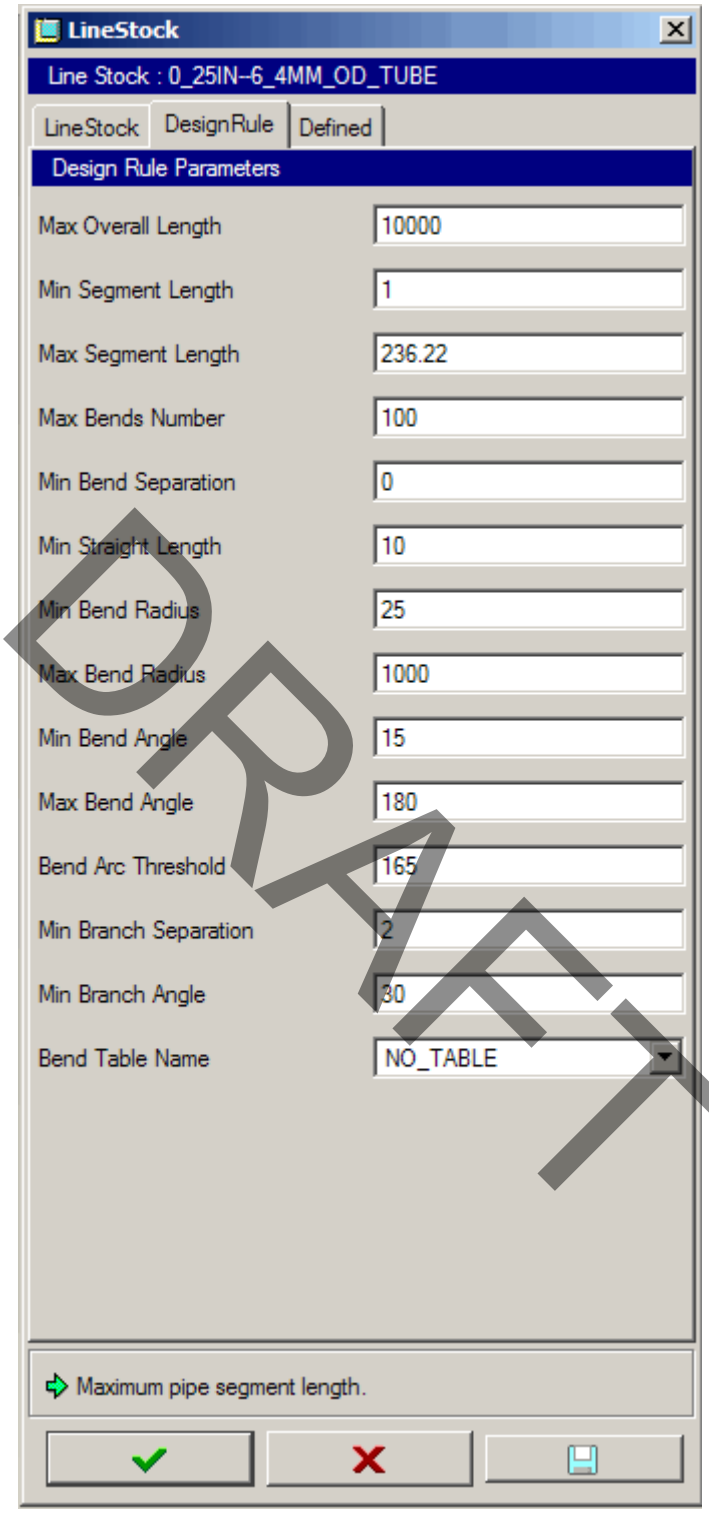
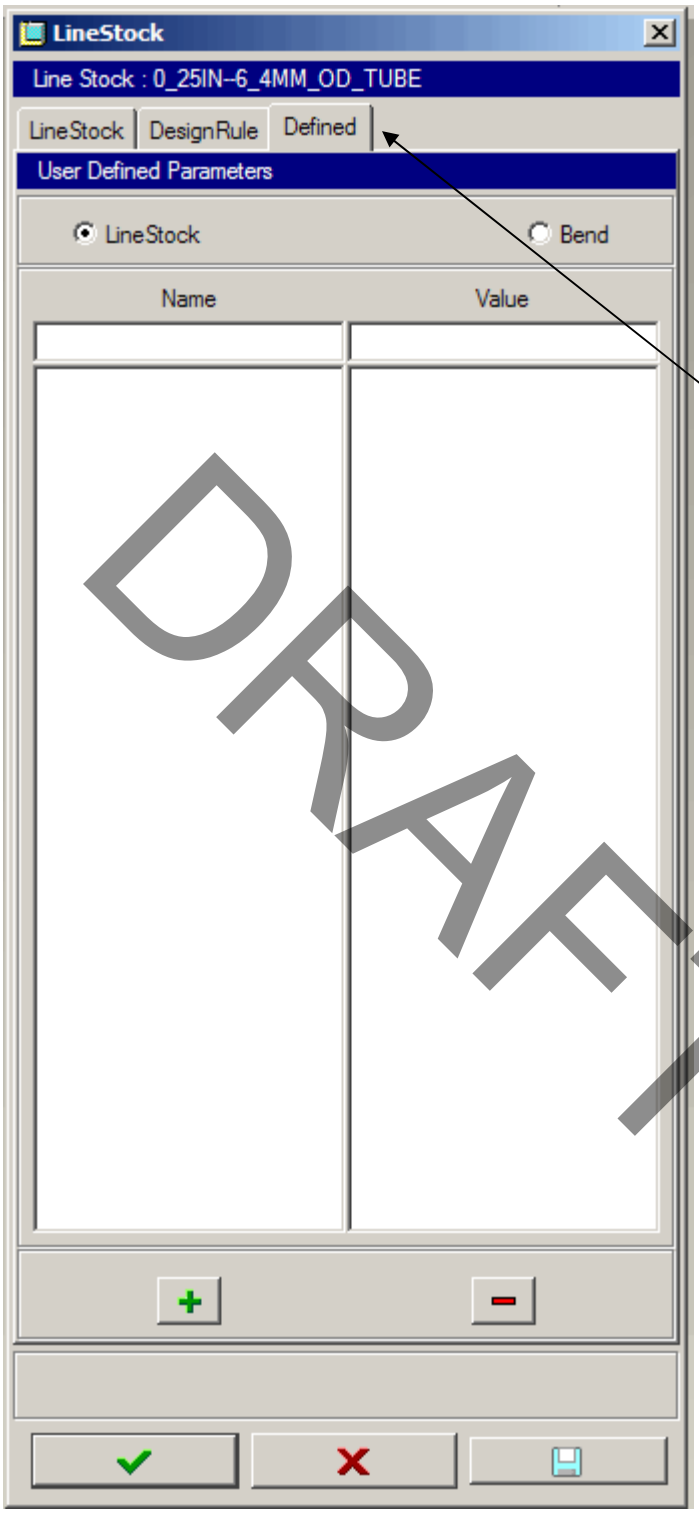


FIGURE 16: LINE STOCK DESIGN RULE PARAMETERS



User Defined Parameters for writing relations in special cases.

FIGURE 17: LINE STOCK DEFINED TAB

A Line Stock can also be created with a .txt file in Notepad or similar program. The following example shows all of the parameters needed to create a Line Stock; ***see Figure 18***.

mil-dtl-13444g-250-id.stk - Notepad

File Edit Format View Help

```
! Enter or modify parameters for the line stock.! You may use the help functionality of Pro/TABLE!
to enter pre-defined parameters.
STOCKNO TUBING
MATERIAL          MIL-DTL-1344G
GRADE    A123
SECTION_TYPE     HOLLOW
SHAPE      STRAIGHT
CORNER_TYPES       BEND
OD       12.7
WALL_THICKNESS  3.17
BEND_RADIUS        1.5  2  3
BEND_ANGLE         90  45  60
BEND_ARC_THRESHOLD       165
MAX_OVERALL_LENGTH       6.4516e+08
MAX_NUMBER_BENDS         5
MIN_BEND_SEPARATION      163871
MIN_STRAIGHT_LENGTH_END   163871
MIN_BEND_RADIUS 20483.8
MAX_BEND_RADIUS 1.17987e+06
MIN_BEND_ANGLE  15
MAX_BEND_ANGLE  112.5
WT_LEN  5.69505e-05
MITER_CUT_LEN      819.353  1638.71  8193.53
MITER_CUT_NUM      1  2  3
X_SECTION_TYPE  CIRCULAR
RECTANGULAR_HEIGHT       32774.1
RECTANGULAR_WIDTH        32774.1
RECTANGULAR_ANGLE        90
BEND_TABLE_NAME NO_TABLE
MIN_BRANCH_SEPARATION    32774.1
MIN_BRANCH_ANGLE         30
MAX_SEGMENT_LENGTH       3.87096e+06
MIN_SEGMENT_LENGTH       16387.1
```

FIGURE 18: .TXT FILE OF A LINE STOCK

8.3 Creating Routes / Pipelines

Pipe assemblies have a specific naming convention. A pipe assembly name is a combination of the upper level assembly name and the pipe circuit number called out in the installation schematic. The circuit number can be found in the schematic related to the installation. For example, for a pipe assembly named 19207_12345678-6110.asm, the upper level assembly is 12345678 and the circuit number is 110, and the hose size is .375". The pipe assembly is designated as a pipe by the "6" as the first character in the dash number. Piping installations that are complex should have accompanying schematic drawings. If a schematic is not provided, circuit numbers shall be defined within the installation drawing.

If a circuit number is not provided, then one must be created. This is done by selecting the next available number that has not already been used in the current piping installation. The circuit number should be a three digit number. The three digits represent the hose type or size and should be described in a table within the drawing. For example, the first number in pneumatic assembly circuit numbers is used to designate either bulk tubing or crimped hose, and also to distinguish the bulk tubing size. A sample pneumatic circuit number table is shown below; ***see Table 11.***

Circuit Identification	Hose/Tubing Type
0XX	.25 Inch Bulk Tubing
1XX	.375 Inch Bulk Tubing
2XX	.50 Inch Bulk Tubing
3XX	.625 Inch Bulk Tubing
5XX	Hoses with Crimped Ends
8XX	Hoses with Crimped Ends

TABLE 11: CIRCUIT ID TABLE

NOTE: All bulk tubing sizes are outer diameter and all rubber hose sizes are inner diameter.

The ***fmtv_start_assy.asy*** has four parameters (***see Table 12***) that are dedicated to Pro/PIPING that should be filled out in piping assemblies. These parameters are ***PIPING_ASSY***, ***HOSE_LENGTH***, ***WEIGHT_PER_LENGTH,*** and ***CIRCUIT_NUMBER***. Piping assemblies should have the ***PIPING_ASSY*** parameter set to **YES**. The default value is **NO**. The ***WEIGHT_PER_LENGTH*** parameter should be filled in with the numeric value of the weight/length given in the line stock file used in the piping assembly. When this is done, the ***HOSE_LENGTH*** parameter will be filled in automatically. The ***CIRCUIT_NUMBER*** will also be automatically generated as it takes the last 3 digits of the filename and fills in the parameter.

If the ***PIPING_ASSY*** parameter is set to **YES**, the ***PART_NUMBER*** parameter will need to be filled out manually. The ***PART_NUMBER*** parameter in the piping assembly will be filled in with the part number of the hose or tube it represents, i.e., the part number should be the number of the part that would be purchased from a BOM. The piping assembly is just a 3-D representation of that part in its routed state. The ***PART_NUMBER*** parameter is the number that will be listed on the assembly drawing. For example, if a pneumatic piping assembly 12345678-6130 represents a piece of 3/8" bulk tubing, with the part number 12420572-006, the ***FILE_NAME*** of the pipe assembly will be 12345678-6130 and the ***PART_NUMBER*** will be 12420572-006. The ***CIRCUIT_NUMBER*** parameter would automatically be filled in with 130 in this instance.

The following are four specific parameters in the fmtv_start_assy.asm that supports Pro/PIPING

	Parameter Name	Parameter Description	Format Example	Input
01	PIPING_ASSY	YES/NO parameter. Defines if the assembly is a piping assembly	YES	-
02	HOSE_LENGTH	Automatically generated when PIPING_ASSY is set to YES.	12.5	-
03	WEIGHT_PER_LENGTH	Numeric value of the weight/length given in the line stock file (manually entered, should be same as line stock file)	..00417	-
04	CIRCUIT_NUMBER	Automatically generated. Provides 3 digit circuit number	110	-

TABLE 12: PIPING ASSEMBLY SPECIFIC PARAMETERS

Port creation:

- Ports should always have the Z-axis orientated in the direction of pipe flow; ***see Figure 19***.

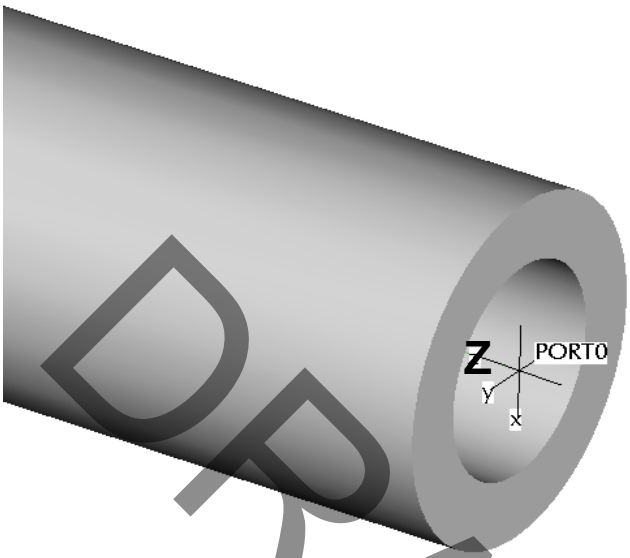


FIGURE 19: SAMPLE HOSE FLOWING INTO PORT AND Z-AXIS DIRECTION

- The Z axis of the pipe port should be collinear to the Z axis of the fitting port.
- The centerline of the pipe needs to remain collinear to the Z axis of the fitting port beyond the termination point of the fitting.

- The user will determine the actual routing of the pipeline. There are many ways to route each pipeline and each user will have a preferred method. Two methods are shown in ***Figure 20.***

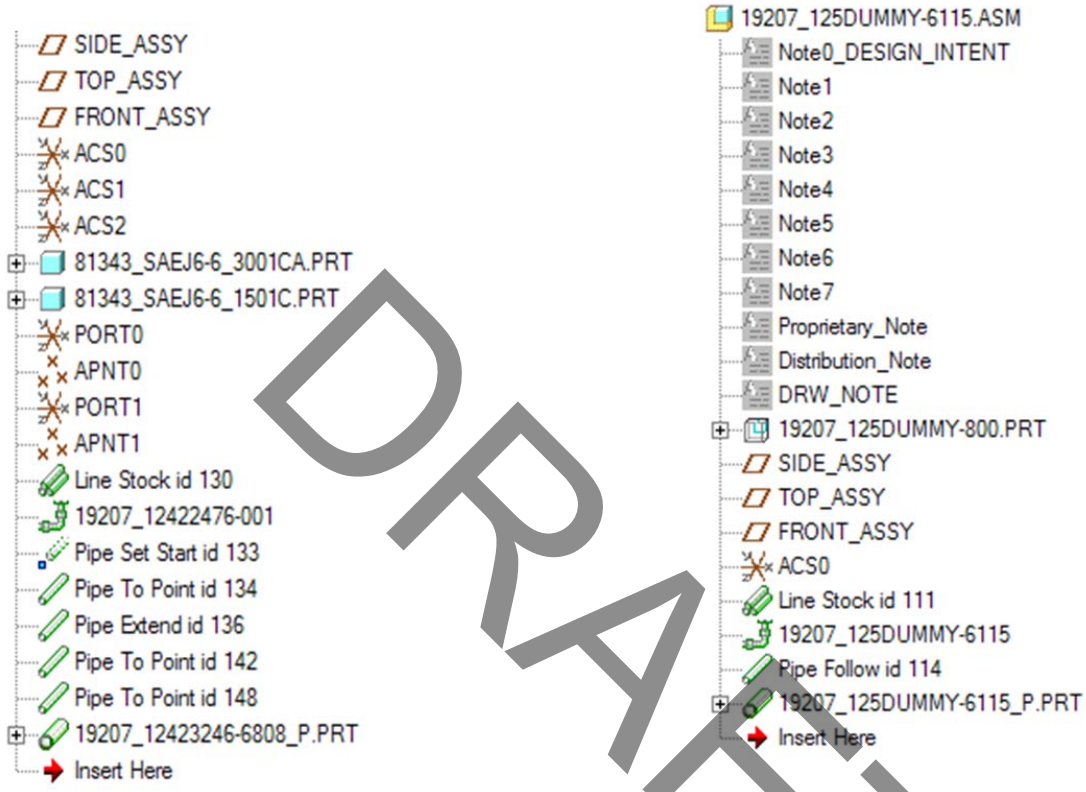


FIGURE 20: EXAMPLE OF CREATING PIPE ROUTE IN SEGMENTS VS. FOLLOWING CURVE

8.4 Fittings and Other Piping Parts

Fittings should be created / updated to include an entry port for each connection point; ***see Figure 21***.

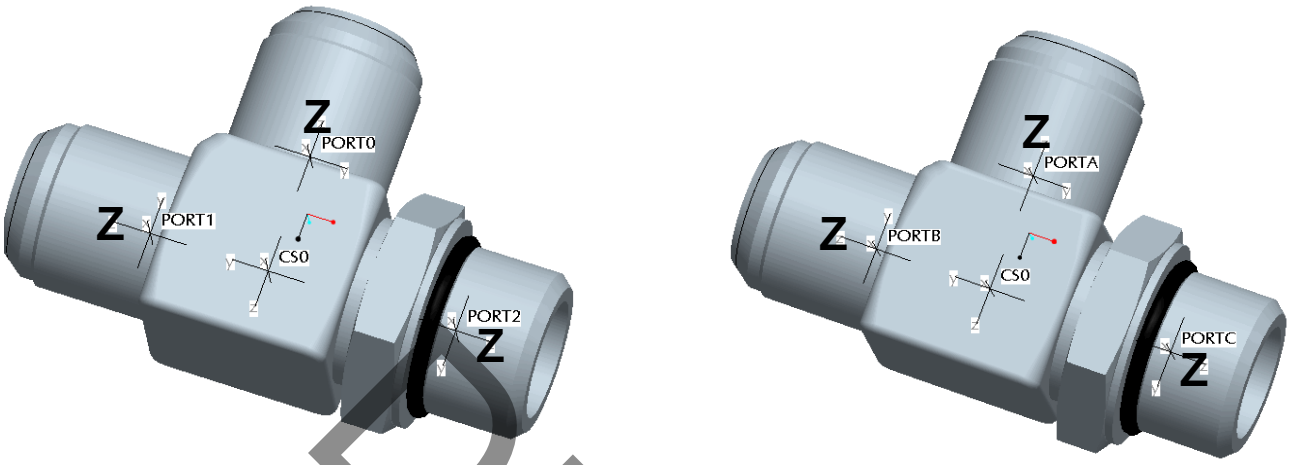


FIGURE 21: TWO EXAMPLES OF A SAMPLE TEE FITTING W/ ENTRY PORTS

- Entry ports shall follow the naming convention of PORT*, i.e., PORT1, PORT2, or PORTA, PORTB, etc.
- The Z-axis of an entry port shall point out of the fitting; pipelines enter or exit the fitting parallel to the Z-axis.
- The entry port should be where the pipe or hose terminates.
- Piping parts such as tie wraps, convoluted tubing, adhesive, etc. should be created and assembled as bulk items.

8.5 Fabricating / Creating Solid Pipes

Fabrication features create the actual part using the path and line stock as its guide. The ***FMTV_Start_Part*** shall be used as a template.

Once a part is fabricated it creates a separate .PRT file. The fabricated pipe can be opened individually when updating the layers.

Note: When fabricating a part, use the same -6xxx extension as the assembly with the addition of the "_p". (e.g., 12345678-6123_p.prt)

1) Select **Fabrication** & **Pipe Solid** from the piping menu shown in ***Figure 22***; a new window will open.

2) Select the **fmtv_start_part** in the **Copy From** box at the bottom of the window; ***see Figure 105.***

3) Expand the pipe route in the left hand center section of the window; ***see Figure 23.***

4) Highlight the pipe segment under the pipe route; ***see Figure 23.***

5) Add the new solid pipe name into the **Solid Pipe** parameter of the pipe segment; ***see Figure 23.***

6) (e.g. 19207_12345678-6123_p)

7) Select **fmtv_start_part** under the **Start Part** parameter of the pipe segment; ***see Figure 23.***

8) Select **Make,** then **OK;** ***see Figure 24.***

FIGURE 22: CREATING PIPELINE

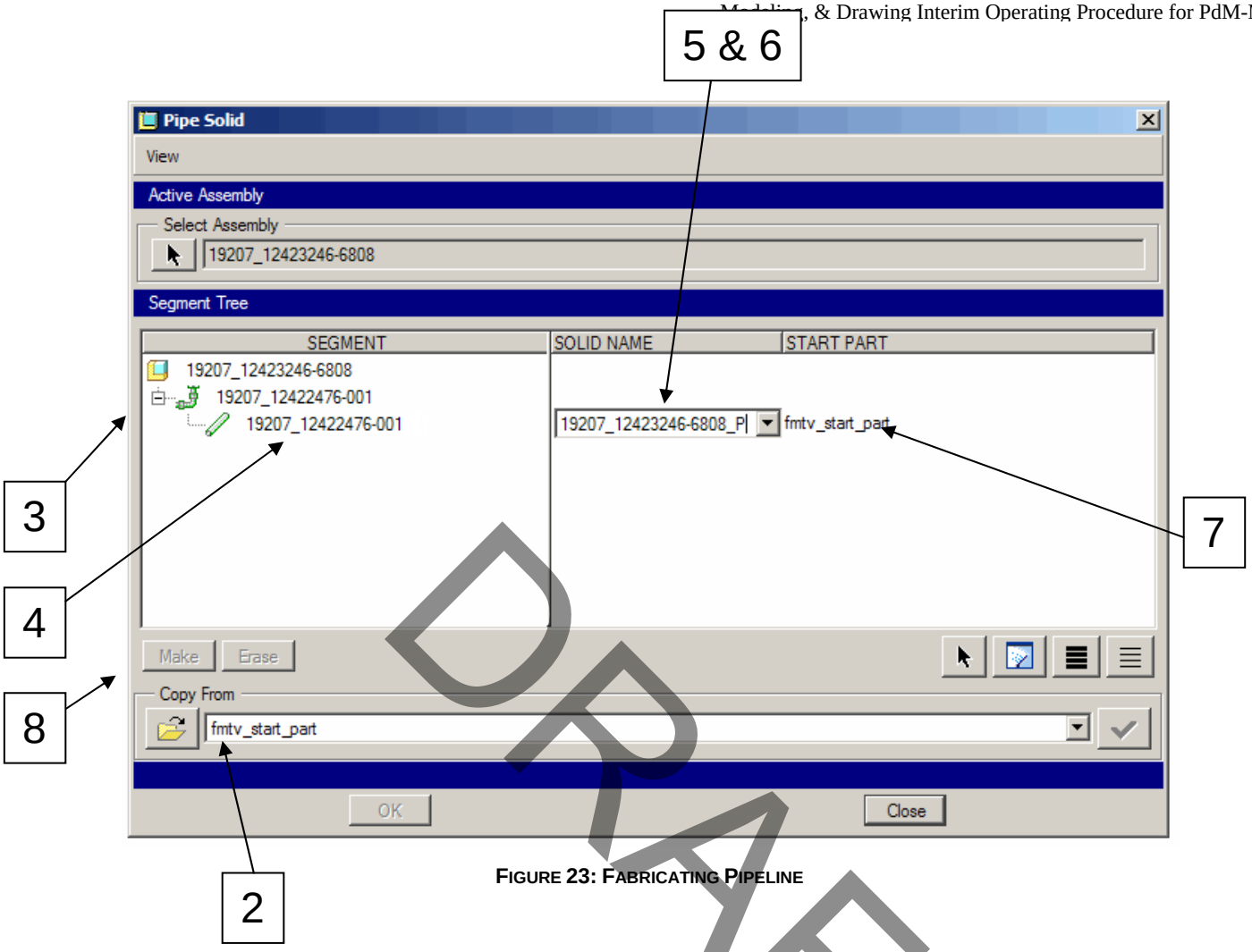


FIGURE 23: FABRICATING PIPELINE

The pipe solid is now created; ***see Figure 24***

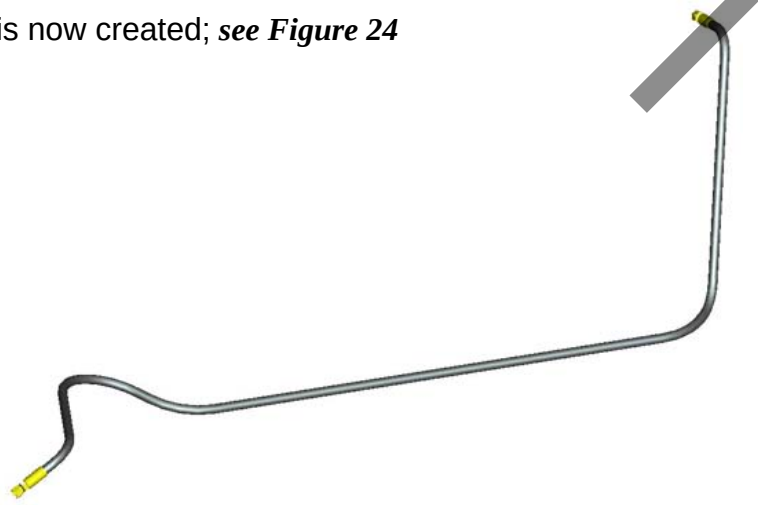


FIGURE 24: FABRICATED PIPELINE

Figure 25 shows a standard pipe assembly with two fittings:

1. Create a pipe assembly with the correct naming convention
2. Assemble your fittings within the pipe assembly
   a) Fittings should only be included in the piping assembly if they are permanently attached to the hose, i.e., if the hose has crimped on hose ends.
3. Read the Line Stock into the assembly
4. Create a path for the individual pipe using a user's preferred method
5. Fabricate the pipe

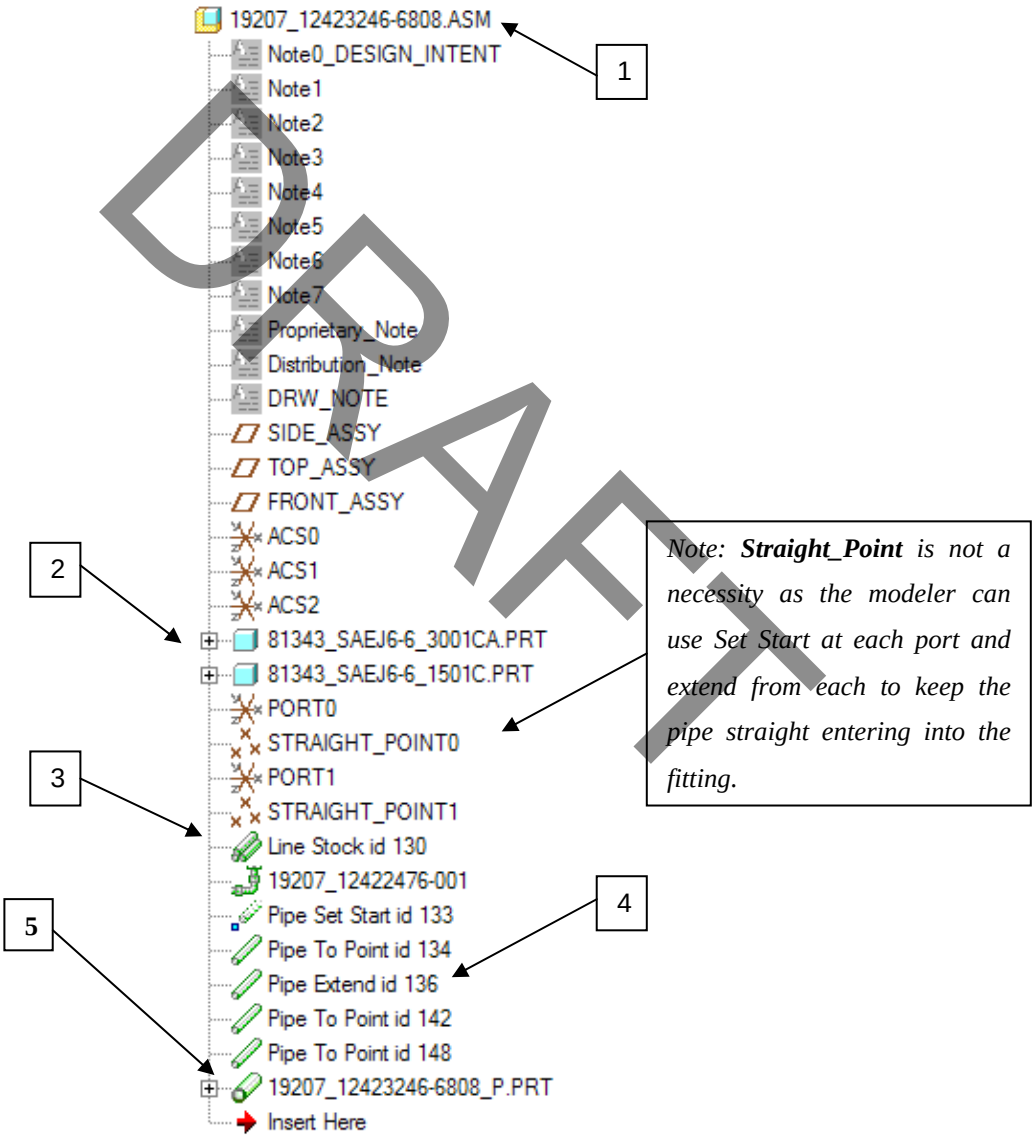


FIGURE 25: SAMPLE PRO/PIPING DRAWING TREE

8.6 Pipe Centerlines

The centerline for pipes shall be layered off in the drawing. This can be accomplished by creating a new layer titled PIPING_BLANKED_LAYER or by placing them on the DTM_CURVES layer.

8.7 Schematics

Schematics are required for any complex piping installation drawing and shall follow the naming convention below;

CageCode_PartNumber

Example:

19207_12505846.dgm

19207_12505846.rsd

A complex piping installation drawing is defined as containing more than ***five*** piping assemblies constrained within that corresponding installation. Those piping drawings of insufficient complexity should have the piping routes and information defined in the installation drawing rather than in a schematic. For those assemblies of sufficient complexity, a schematic should be referenced in the notes on the installation drawing. Pro/DIAGRAM or Routed System Designer (RSD) may be used to create the schematic. A schematic is not required to be parametrically linked to the piping installation or drawing and should be used for reference purposes. If the schematic is part of the drawing, the overlay process in ***section 7.1.2*** can be used.

Schematics should consist of:

- Schematic Line Connection List; s***ee Figure 26***

SCHEMATIC LINE CONNECTION LIST (COMPONENT REFERENCE DESIGNATORS, PORTS, CIRCUITS AND COLORS) - SORTED BY LINE NO								
LINE NO	INSTALLED LINE COLOR	SCHEMATIC LINE COLOR	SIZE mm	FROM REF_DES PORT	TO REF_DES PORT	CIRCUIT	LINE MATERIAL PART NUMBER	FROM/TO LOCATION SH, Y X
001	BLACK	VIOLET	6.350	D19 1_T	A27 1_L	AUXILIARY	12420572-004	SH6,F6 SH6,G6
002	BLACK	VIOLET	6.350	A11 2A_F	D02 1_T	AUXILIARY	12420572-004	SH6,H6 SH6,F6
004	BLACK	BLACK	6.350	C03 4_B	C02 4_L	CHARGING	12420572-004	SH5,A6 SH5,B7
005	BLACK	VIOLET	6.350	D02 1_B	D04 1_T	AUXILIARY	12420572-004	SH6,F6 SH6,B5
008	BLACK	BLACK	6.350	C02 4_R	F02 1_R	CHARGING	12420572-004	SH5,B7 SH5,C6
010	BLACK	VIOLET	6.350	A27 2_R	A11 1_R	AUXILIARY	12420572-004	SH6,G6 SH6,H6
014	BLACK	VIOLET	6.350	C04 1B_R	A12 1_L	AUXILIARY	12420572-004	SH5,C5 SH6,C5
015	BLACK	GREEN	6.350	A01 1B_R	C18 1B_F	REAR BRAKE	12420572-004	SH5,G8 SH5,H7
016	BLACK	ORANGE	6.350	A01 1A_R	C18 1A_F	FRONT BRAKE	12420572-004	SH5,H8 SH5,H7
102	BLACK	RED	9.525	A02 2B_B	D05 1_T	TRAILER SUPPLY	12420572-006	SH5,G7 SH5,F7
103	BLACK	YELLOW	9.525	A02 2A_B	A22 1_R	PARK BRAKE	12420572-006	SH5,G7 SH5,G8

FIGURE 26: SAMPLE SCHEMATIC LINE CONNECTION LIST

- Symbol Legend; s***ee Figure 27.***

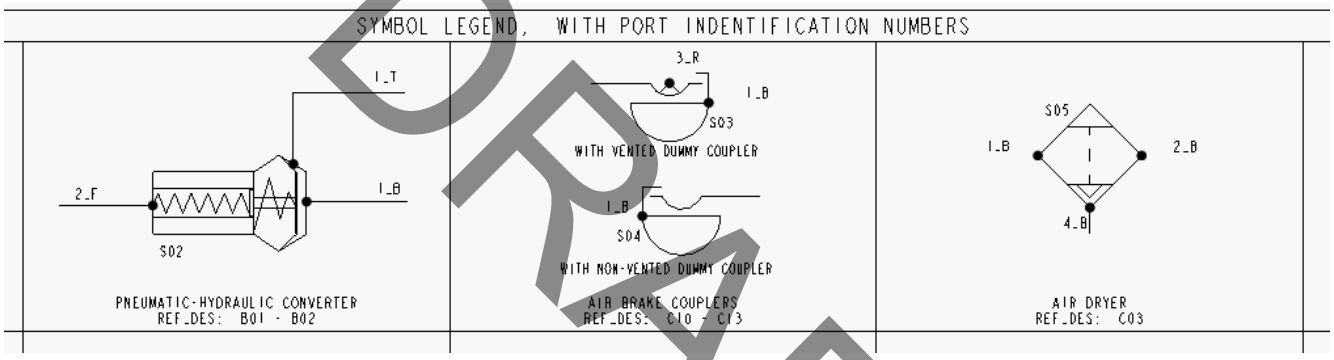


FIGURE 27: SYMBOL LEGEND EXAMPLE

- A Color Legend for the diagram; s***ee Figure 28.***

COLOR LEGEND			
APPLICATON OR CIRCUIT	INSTALLED COLOR	SCHEMATIC COLOR	COLOR LINE APPEARANCE
CHARGING SYSTEM	BLACK	BLACK	
REAR BRAKES	BLACK	GREEN	
FRONT BRAKES	BLACK	ORANGE	
PARKING BRAKES	BLACK	YELLOW	
TRAILER SUPPLY/EMER.	BLACK	RED	
TRAILER CONTROL/SERV.	BLACK	BLUE	
AUXILIARY AND CTIS	BLACK	VIOLET	
HEAT-SENSITIVE	STAINLESS	GREY	
VENT OR PILOT LINE	BLACK	BLACK-DASH	
HYDRAULIC BRAKE LINE	BLACK	PINK	

FIGURE 28: COLOR LEGEND EXAMPLE

- Piping diagram; ***see Figure 29.***

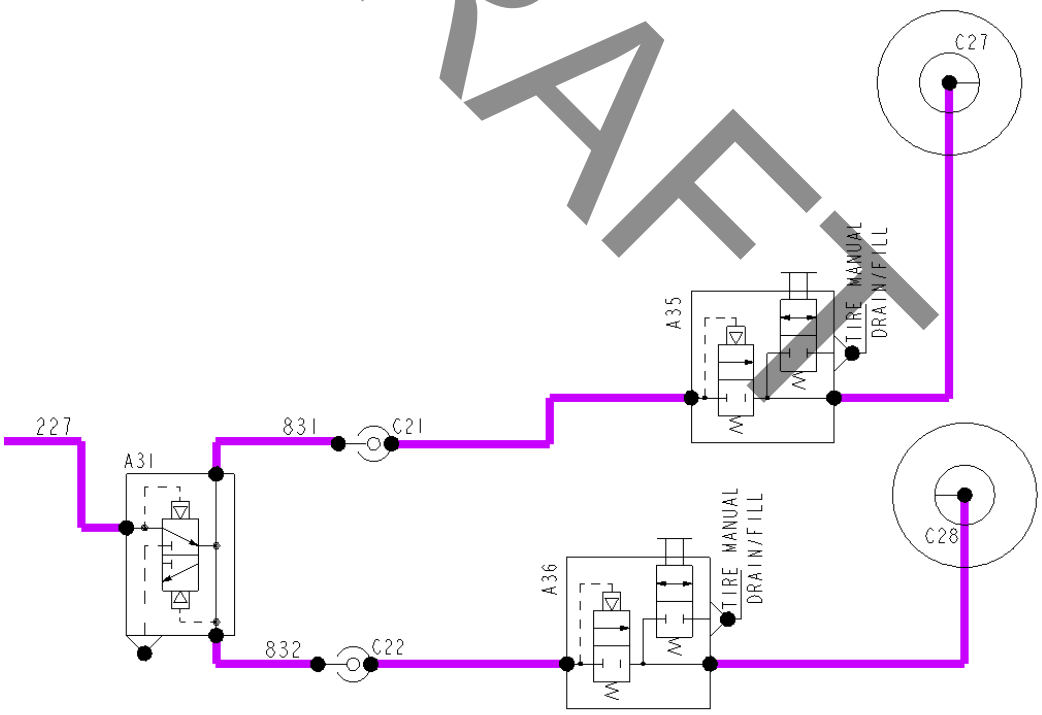


FIGURE 29: SAMPLE PIPING DIAGRAM

Installation drawings that do not have associated schematics should include a Line Connection List in the installation drawing. This list defines the circuit ID for the routed pipes within the installation.

Note: All circuits are labeled with proper identification numbers as described in section 8.3, table 11 and components (symbols) are labeled with required reference designators that correspond with Figures 24 and 25.

8.8 Piping Checklists

Pro Piping Checklist

Piping Assembly

- ☐ File Name: CageCode_PipeAssyNo-6XXX.asm
- ☐ The circuit identification matches the specific type of hose or tubing designated on the required circuit number table. Refer to section 14.3, table 22
- ☐ The correct line stock has been used.
- ☐ The correct layers have been created in the assembly following the modeling standards
- ☐ All unnecessary layers are hidden in the assembly
- ☐ The assembly was created using the correct start assembly.
- ☐ There are no external references
- ☐ Piping parameters (PIPING_ASSY, HOSE_LENGTH, WEIGHT_PER_LENGTH, PART_NUMBER) to be filled out properly.
- ☐ The correct assembly structure was used, following all the piping modeling standards
- ☐ Is the unit weight accurate and reasonable for the pipe assembly
- ☐ Appropriate parameters have been completed
- ☐ Model check has been completed on this assembly
- ☐ Centerline has been layered off

Pipe Part

- ☐ File Name: CageCode_PipeAssyNo-6XXX_p.prt
- ☐ Logical reference has been established
- ☐ Centerline has been layered off
- ☐ The piping part circuit number matches the up assembly circuit number
- ☐ All unnecessary layers are hidden in the part
- ☐ Material file should not be called out
- ☐ Is the unit weight accurate and reasonable for the pipe part
- ☐ Appropriate parameters have been completed
- ☐ The piping part was created using the correct start part
- ☐ The correct layers have been created in the part following the modeling standards
- ☐ Model check has been completed for this part.

Diagram Checklist

Sheet Setup

- ☐ Correct format (e_fmtv_diag_1_2.frm)
- ☐ Appropriate parameters for the diagram are completed
- ☐ File Name is CageCode_PartNo.dgm or CageCode_PartNo.rsd
- ☐ Nomenclature 3 is set to “SCHEMATIC”
- ☐ Scale set to NONE
- ☐ Weight set to -----
- ☐ Diagram has been checked for spelling errors
- ☐ All Layers are hidden.

Tables

- ☐ Schematic Line Connection List, Symbol and Color Legend have been placed
- ☐ Text in table cells is visible, column with is adequate to display contents

Notes and Revision Block

- ☐ Required Diagram Notes have been placed (See Below)
- ☐ Rev Block has been completed, correct format
- ☐ Revision level matches the drawing and assembly
- ☐ Distribution note has been placed
- ☐ Proprietary note has been placed, if required
- ☐ Distribution and Proprietary notes match the 782 form and the drawing

Required Diagram Notes

1. APPLICABLE STANDARDS/SPECIFICATIONS:
   A. ASME Y14.100-2013
   B. ASME Y14.5-2009

2. SCHEMATIC IS IAW DRAWING 19207_XXXXXXXX

Appendices

A Modeling Best Practices

A.1 Part Mode

- Name features appropriately.
- Patterns and pattern table use is recommended
- Sheetmetal parts shall always be created by starting with the standard fmtv_start_shtml_part.prt start part.
- Save models early and often.
- Base features should be kept simple.
- Sketches should be kept simple.
- Incomplete features must be removed or completed.
- Relations should be clearly commented.
- Round/chamfer edges should generally not be used as references for other features.
- Rounds and chamfers should be created as late as possible in the modeling process. Draft features should be created when most appropriate.
- Datum planes make the best sketch planes and references.
- Extruded feature depth definitions should reflect the design intent such that the depth remains appropriate if the part is modified.
- When modeling from an existing drawing, try to create the models with features and dimensions that limit the need for created dimensions on the drawing.
- Create new datum planes and axes for use as Geometric tolerance datums and axes. (Do not rename the default datums found in the start part or use axes created by a feature.)
- Add geometric tolerance datums to the G_DTM_PLANES layer immediately after using the "**Set Datum**" command in Creo Parametric.

A.2 Drawing Mode

- Standard FMTV formats (in Standard CAD Template Library) must be used on drawings. The correct standard format version depends on the work effort.
- Standard drawing setup files (.dtl) must be used. These are applied when the appropriate Drawing Options are used.

- Snapshots should not be used unless absolutely necessary.
- Drawings shall not contain overwritten dimensions unless required. For more information see the **Poor Modeling Practices** in ***Appendix B***.
- Views should be oriented using saved views or default datums, not model geometry.
- All notes symbols, and other 2D sketched entities should be related to the appropriate view or object. (HINT: To check for this move the view(s) to see that no entities of the view are left behind.)

A.3 Assembly Mode

- The first component or skeleton is the structural base for assembling all other components.
- Individual components or sub-assemblies should be colored differently for ease of viewing, textures should avoid being used as they can slow down regeneration time.
- Components should be fully constrained with an appropriate combination of constraints (ex. MATE, ALIGN, INSERT, COORDINATE SYSTEM, etc...). An exception to this rule is components assembled using PTC Creo Mechanism. These components are allowed degrees of freedom to simulate the movement of desired parts in an assembly. When PTC Creo Mechanism is used a snapshot shall be stored in order to locate the “at rest” design position. The FIX constraint should be avoided as a normal constraint as it was designed to finalize the location of moved or packaged components where the final location is not yet known. The FIX constraint will also hinder the use of Local Groups in Assembly.

B Poor Modeling Practices

The following section explains how to avoid/work around several examples of poor modeling practice.

B.1 Buried Features

Filling in holes/cuts with a protrusion is not acceptable. Occasionally ModelCHECK reports cuts as buried features if the definition of the subtracted volume lies entirely within the existing solid. This is an allowable ***buried*** feature.

B.2 Overwritten Dimensions

It is not acceptable to overwrite a dimension with **@o** in order to change the numeric value from that which is defined in the model. Overwriting to show another character that may be found in a table (i.e. A, B, C, etc.) is acceptable. The preferred method in this case would be to modify the symbolic value of the dimension and show it using **@s** on the drawing.

B.3 Blind Feature Depth (= to material thickness)

When creating an extruded feature where the depth is intended to be through the part, the depth shall not be entered as a numeric value equal to the material thickness of the part. The depth shall be set using **thru all**, **thru next,** or **up to surface** as appropriate. Example: a cut is created through a part but the depth is entered as a value equal to the material thickness of the part, the feature does not update as desired if the material thickness is increased. If **thru all** were to be set as the defined feature depth, the feature updates as expected when the part thickness is changed.

B.4 Excess Datums/Features

Extra, unused datums shall be cleaned up/removed from models. When modeling a part or assembly, datums are created for construction of the part. Often some of these datums are not actually required in the final iteration of the model. These datums shall be removed when possible. Remaining features shall then be named as appropriate (i.e. naming a datum plane CL_PART if this is the intent).

When a part datum is to be used for assembly purposes, be sure to name it so as to recognize its purpose and to avoid accidental deletion. (i.e. ASSY_1)

B.5 View Name and Scale Not Parametric

The displayed view name and scale should be parametric to the model. If these are accidentally removed at some point, attempt to "show" them in the drawing instead of keying in non-associative text.

B.6 Bending Back Using the Wrong Sheetmetal Surface

Be sure after creating an unbend feature to use the correct surface when creating the bend back feature. If the same surface is not selected, dimensions will not "show" properly on the drawing.

B.7 "0" Offset Assembly Mate

Unless an assembly constraint of ***Mate*** or ***Align*** requires the value of zero to make the design flexible, it should be fixed as ***Coincident***.

B.8 Congested Sketches

If a part has several dimensions defining its shape, these shall be broken up in to an appropriate number of separate features. When roughly 10 or more dimensions are used to create a base feature the sketch can become congested and difficult to work with. When roughly 20 or more dimensions are found in a sketch, the computational time becomes cumbersome as well as the sketch being congested.

B.9 Flat pattern for Sheetmetal parts

Sheetmetal parts should not be created by starting with a flat pattern. Sheetmetal parts should be modeled in an "as designed", functional state. The flat pattern is created last (as necessary) for manufacturing purposes, using Edit>Setup within the sheetmetal application. This creates a separate flat instance which can be added to the drawing as a model independent from the generic fully formed model used on all other views. The fully formed generic model may be changed and features added or

deleted. As long as the family table is verified regularly, the flat patter will automatically update with the changes done to the generic.

C Miscellaneous Topics, Methods and Techniques

C.1 Model Relations

Each model has relations that can be found in part, feature, and sketch levels. Feature relations take precedence over part relations as they occur later in the regeneration cycle. Relations found in feature CS0/ACS0 of the FMTV start parts are required. These should not be changed from what is contained in the start part.

C.2 Assembly Relations

(d#:# or d#:cid_#)

When part dimension symbols are shown in assembly mode, they contain two numbers (d#:#). The first number is assigned from within the part and is the same number that will be seen if the symbol was viewed from Part mode. The second number is assigned from within Assembly mode and is the number that appears in the assembly coding table. This second number can also be found by selecting **Show > Session Id** in the Relations dialog box. Creo Parametric will return, ***Component <component_name> has Session ID #.*** Each part in the assembly is assigned a unique second number but components with the same name will have the same Session ID.

Assembly relations that include family table components can also be written in the format (d#:cid_#). The first number is still the same dimension number that was assigned at the part level but "cid" is the Internal Component ID. The Internal Component ID is found by selecting **Show > Info** in the Relations dialog box. The Internal Component ID is unique for every component even if the same component is assembled twice in the same assembly. This method would be preferred if the same component is assembled twice in the same assembly, a relation is written using the dimensions of the component and then one of the components is replaced using **Edit > Replace**. Using the Internal Component ID will ensure that the relation will update as intended.

C.3 PROGRAM – From File vs From Model

The **From File** menu selection will be displayed only if the original PROGRAM has been modified and there is text file of the program residing in the current working directory. If **From File** is chosen, the editor will display the most recent listing of the program that resides on the local disk.

The **From Model** menu selection will extract the current listing associated with the object. Upon initial creation of a PROGRAM, the user will not see these two choices, as Creo Parametric will use the default listing associated with the model. After modifying the model file, then storing it to disk, the user will be asked if he or she wishes to incorporate any changes. If the changes are successfully incorporated, the model will be update accordingly and the design file will be deleted, once again leaving only the default model listing.

C.4 Creo Parametric Layers

Layers are helpful in controlling the display of features, layers, and components in the Creo Parametric environment. Entities are assigned to a layer and layer display status is set. Entities created when using the FMTV start models are automatically set to the appropriate layer. The two exceptions are cross section datum planes and GTOL datum planes. Additional layers may be created as necessary to help organize things such as construction geometry, imported data, etc.

Display status may be set at different levels, for example: part, assembly, skeleton, and drawing. Drawing display status acts independent of part, skeleton, and assembly layer settings. The stored layer status will vary as appropriate for objects being worked on. When a part or assembly is promoted into the release process its layer status must follow the modeling standard. The layer status of a drawing should be set as required to show entities appropriately on the drawing. **Isolate status** is often useful as explained in the datum target point example described in the following paragraph.

An entity may be set to more than one layer. This should be done as little as possible since the only way to see the entity is to turn on all of the layers it resides on or to set one of them to **isolate status**. For example, if an item is on two or more layers and any one of them has a hidden status, the item will not be displayed. If a point feature is placed on three different layers (dtm_points, inspection_points, and datum_target_points) the datum target points may be seen by settings the datum_target_points layer status to isolate.

NOTE: Setting the layer to isolate status displays entities on the layer regardless of the layer status of other layers it resides in.

The following image shows the layers found in the FMTV start part. Note that the display status is hidden for all layers. This is how archived objects are set so that the screen is not cluttered.

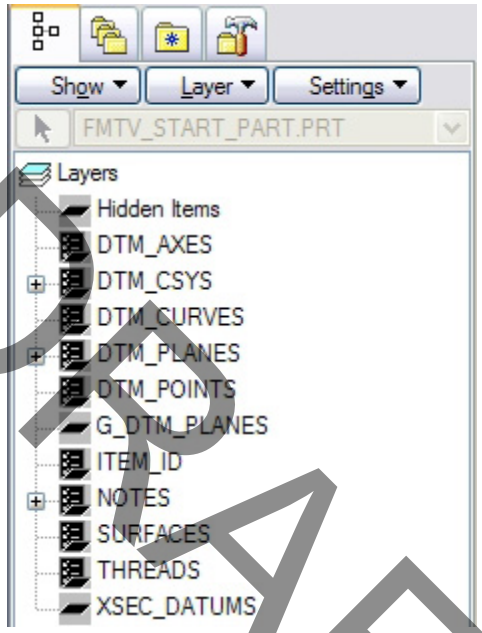


FIGURE 30: LAYER TREE

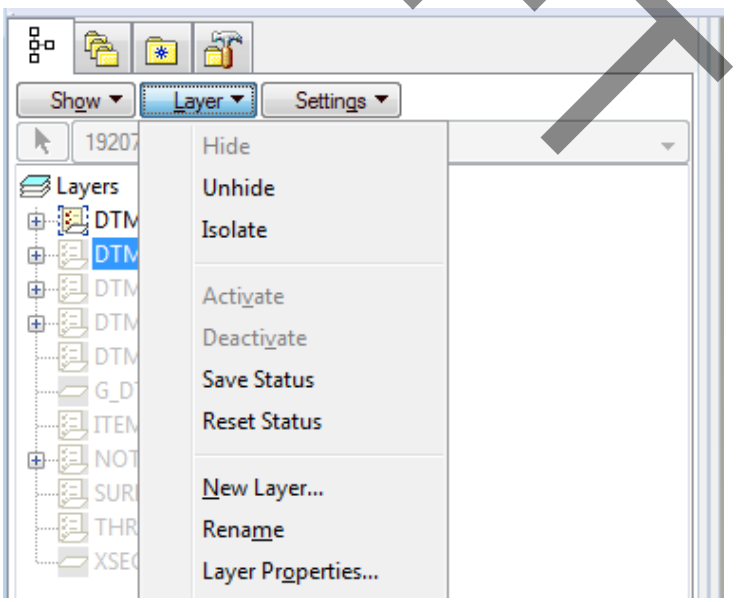


FIGURE 31: SETTING VISIBILITY OF LAYERS

It is acceptable to have layers unhidden as long as the config.pro and config.sup options listed in ***FMTV Creo Configuration Settings*** prevent the screen from being cluttered upon initial retrieval of a part, assembly, or drawing.

If a feature(s) on a layer(s), such as those generated by datum curves or a set datum operation, displays despite these options being set then those layers must be hidden or the feature in question must be moved to a layer that is controlled by the above config.pro options

C.5 Geometric Dimensioning and Tolerance (GD&T) used in Creo Parametric

Per the TV modeling standards, geometric tolerance is required to be applied to three-dimensional part and assembly models using Creo Parametric commands. The **Insert > Annotations > Geom Tol** menu showing these commands is found below.

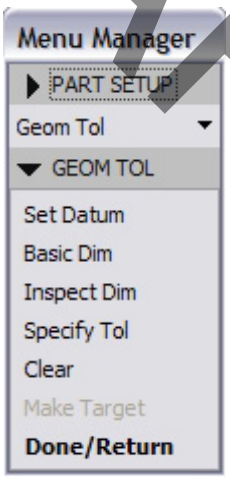


FIGURE 32: INITIATE GTOL

Step 1 - Identify existing datums (or create new ones) to be used in constructing the datum reference plane. Set the appropriate datums using the **Set Datum** command from the menu above.

Step 2 - Specify dimensions as basic as required. These dimensions made basic by using the **Basic Dim** command from the menu above.

Step 3 - Specify dimensions as inspection as required. These dimensions made inspection by using the **Inspect Dim** command from the menu above.

Step 4 - Describe features of size (such as holes and slots) with a feature control frame. The feature control frame is created using the **Specify Tol** command from the menu above. The geometric tolerance window appears as shown below. Select the button of the appropriate geometric characteristic (i.e. position, concentricity, etc.) at the left hand side of the window. Select the reference as required (ex: An edge for a straightness tolerance). Assign the new tolerance if it is to appear in a dimension (ex: A hole diameter dimension for a position tolerance.)

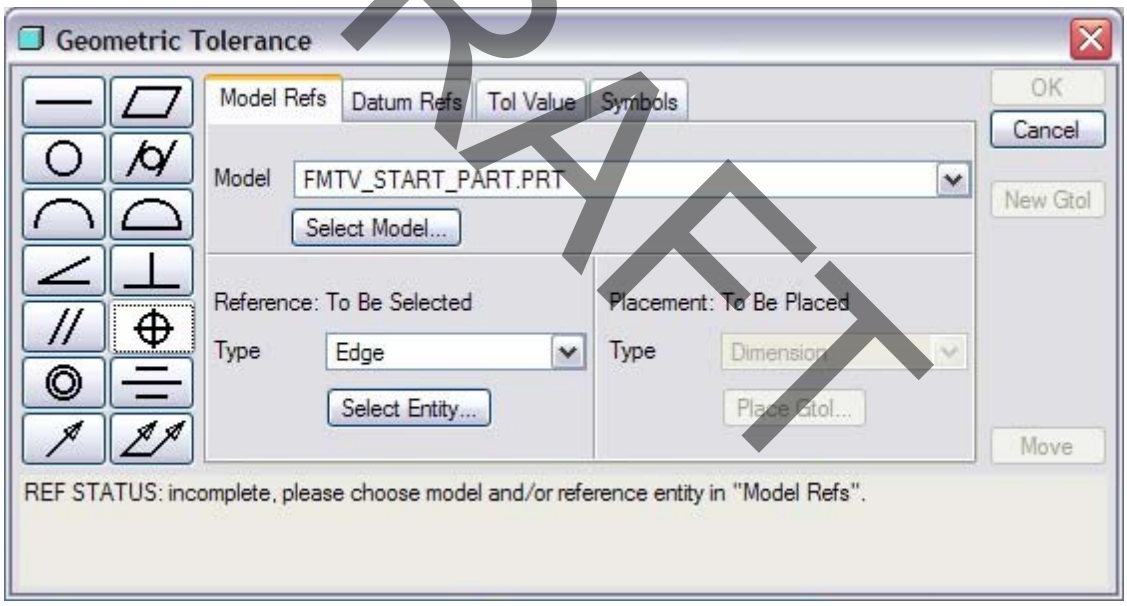


FIGURE 33: GTOL MODEL REFERENCES TAB

Click the **Datum Refs** tab and then select appropriate primary, secondary, and tertiary datums as references (if required). Make sure to set the proper material condition also.

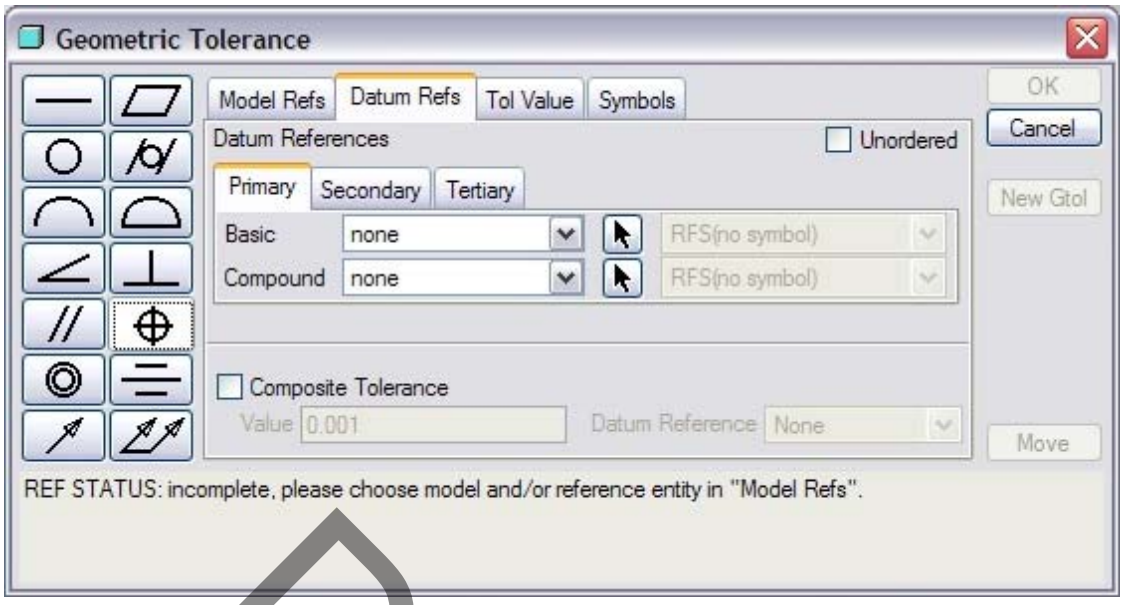


FIGURE 34: GTOL DATUM REFERENCES TAB

Click the **Tol Value** tab and enter the necessary tolerance value if required.

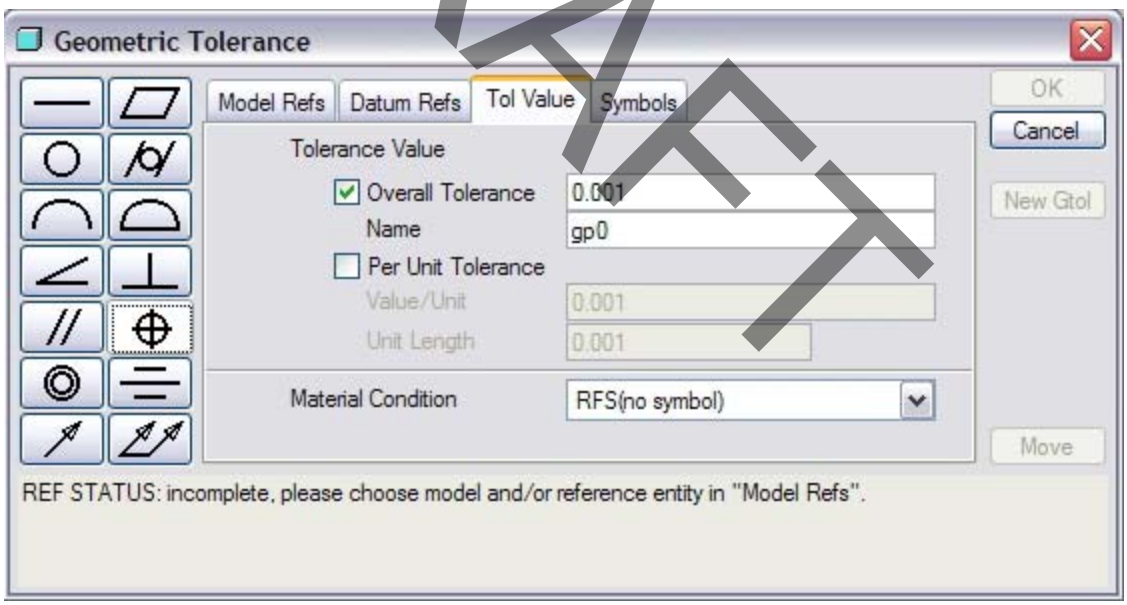


FIGURE 35: GTOL TOLERANCE VALUE TAB

Finally, click the **Symbols** tab and select any required symbols, modifiers, additional text, or projected tolerance descriptors.

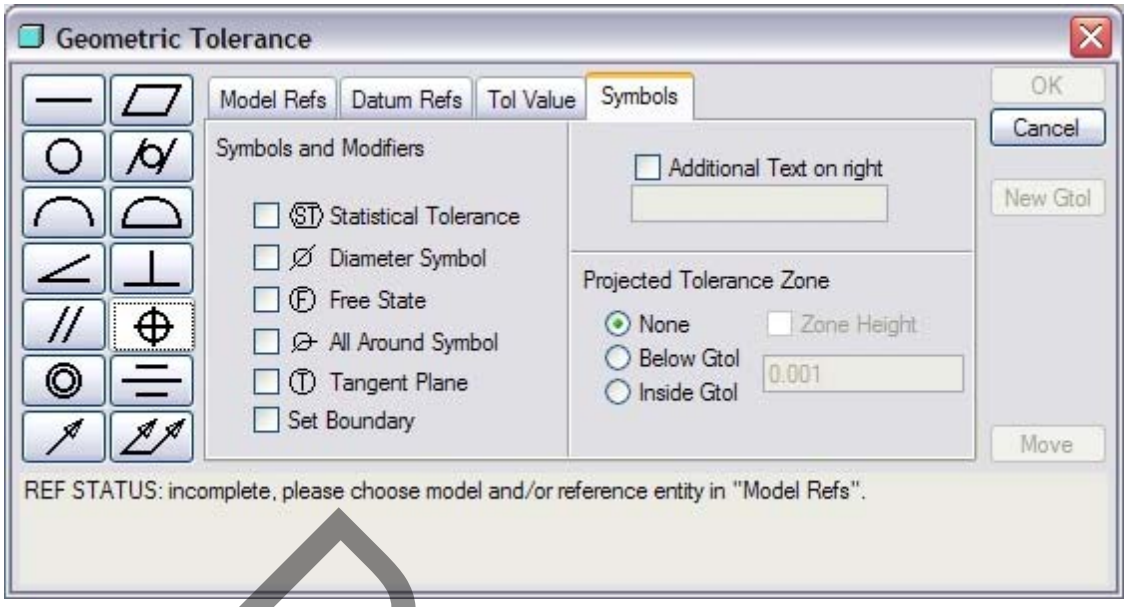


FIGURE 36: GTOL SYMBOLS TAB

C.6 Datum Planes and GTOL

Datum planes will not appear in the list of possible datums to reference in the Datum Refs portion of the Geometric Tolerancing dialog box unless they have been set. To set a datum, select **Set Datum** from the **GEOM TOL** menu, select the axis or plane to set, and then choose the placement type for the datum from the Datum dialog box (**Free** or **In Dim**). Once the datum is set, it will be available for referencing. The datum plane must also exist and be stored in the model showing the Model field of the Models Refs portion of the Geometric Tolerance dialog box.

*IMPORTANT: In order to delete a datum that has been set as a GTOL datum, it should first be cleared using the **Edit > Set Up > Geom Tol > Clear** menu selection.*

C.7 GTOL Datum Features Visibility

By default, all set datums will be displayed regardless of the existing display setting for datum planes. This is true in parts, assemblies and drawings. For parts and assemblies, the only way to remove the set datum is to place the datum on a layer and hide the layer. This is why it is required that the modeler assigns set GTOL datums to the g_dtm_planes layer so that they may be turned off while

modeling. In drawings, the set datums may either be turned off using the layer controls or erased using the **View > Show/Erase > Erase** > **Datum > Selected Item** command.

C.8 ModelCHECK Process

ModelCHECK is used to compare the active part, assembly, or drawing to a set of required configuration settings and report back items that are not set properly. It provides tools to help correct the deficiencies.

C.8.1 ModelCHECK Procedures

ModelCHECK will be run interactively in a Creo Parametric session by each individual designer. The designer will perform the task on each model/drawing he or she has worked on. Report viewing will be accomplished using the built in Creo Parametric browser. Run ModelCHECK by either clicking on the ModelCHECK Interactive Button from the Analysis pull down menu in the pull down as shown below or using the **MCK** mapkey.

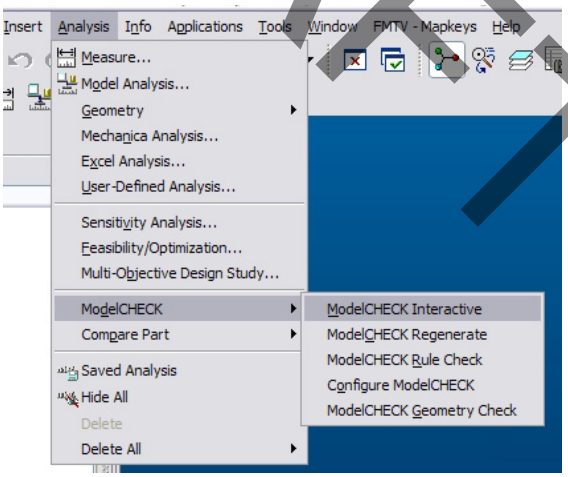


FIGURE 37: INITIATING MODELCHECK

A report summary will appear in the browser window that will look similar to ***Figure 38***.

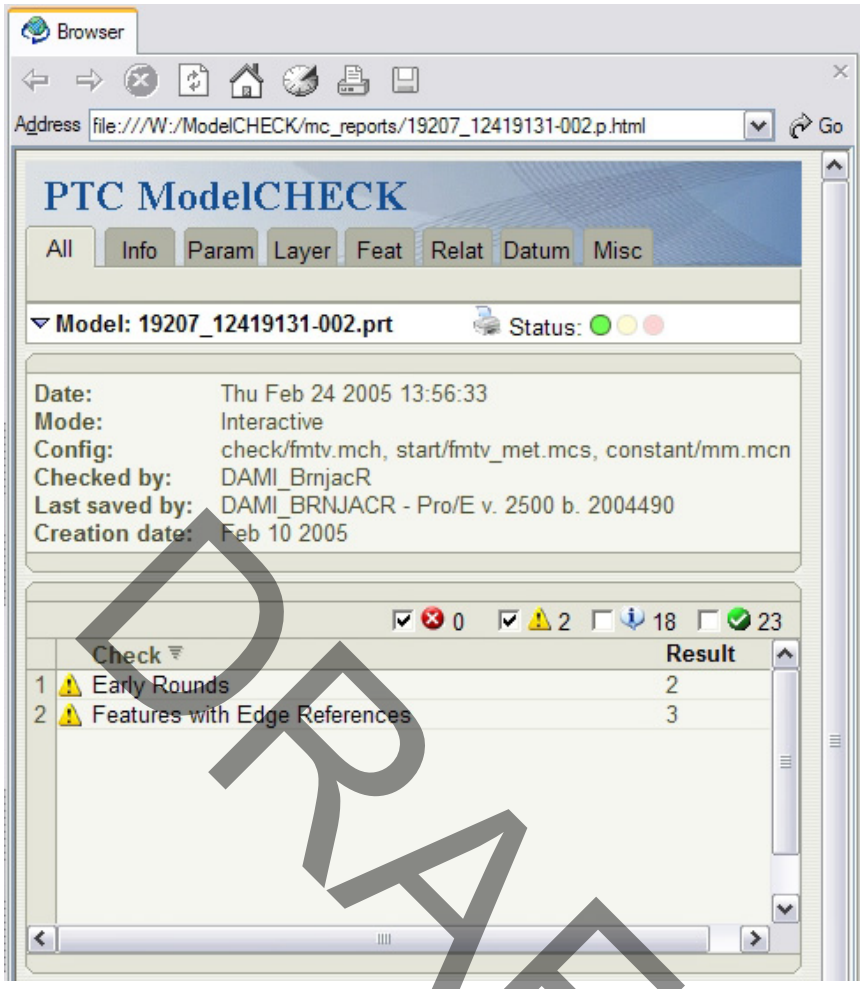


FIGURE 38: MODELCHECK SUMMARY

In order to view the detailed information, click on the tabs at the top of the browser (**Info**, **Param**, **Layer**, **Feat**, **Relat**, **Datum**, and **Misc**). By default only errors and warnings are displayed. In order to view general information and checks the check boxes next to the icons must be selected, as shown in ***Figure 39*** (numbers next to the icons show the number of related items).

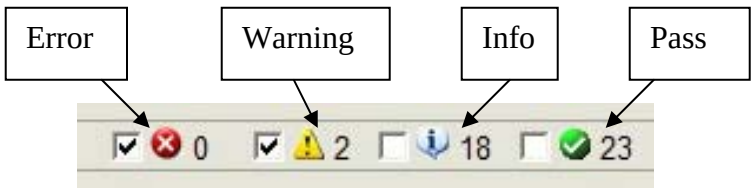


FIGURE 39: MODELCHECK TYPES

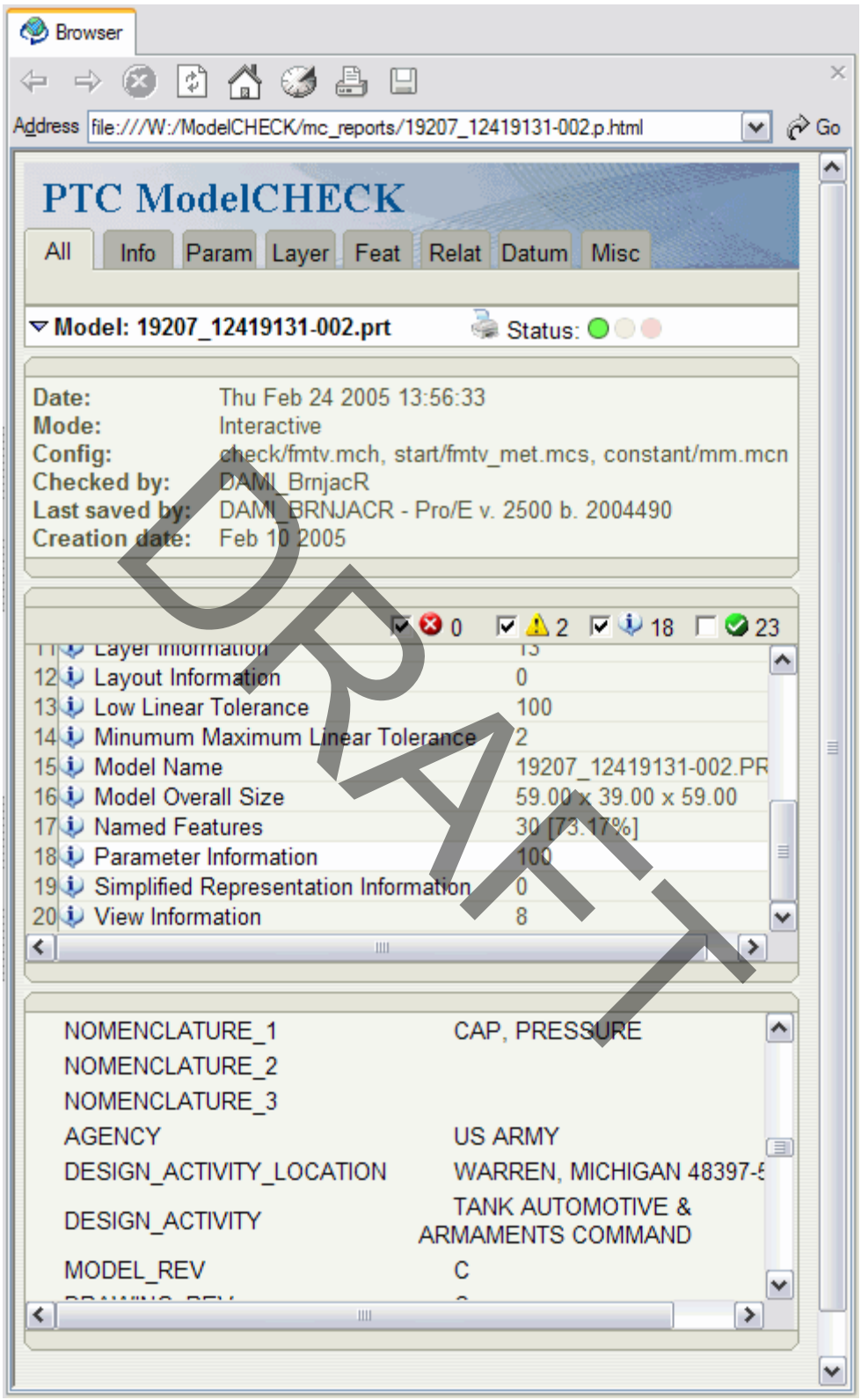


FIGURE 40: EXAMPLE OF MODELCHECK REPORT

NOTE: Any errors found must be corrected. Any warnings found should be investigated.

Clicking on any of the tabs makes that report page active. Clicking on the line number displays a detailed description of the error or warning. Certain quick corrections, such as creating missing layers and correcting layer status, can be made by ModelCHECK. It will take a few moments each time a button is selected for the action to take place in the Creo Parametric session. After fixing all errors run ModelCHECK again to make sure all items have been fixed. ModelCHECK will not save the model or check it in to Commonspace; the user is required to complete those tasks when appropriate.

C.8.2 FMTV ModelCHECK Error/Warning Summary

ModelCHECK is currently configured to operate in the interactive mode as described in the ModelCHECK procedures. Any checks made resulting in errors must be fixed or the model may not be promoted in Windchill PDMLink. When ModelCHECK is run the following checks will result in an error:

1. Features 1-4 must match those found in the corresponding start part.
2. No suppressed or incomplete features may be found in models.
3. No frozen, suppressed, missing, and/or packaged components may be found in assemblies (except when PTC Creo Mechanism is used).
4. Density must not be equal to "1" (ensures that a material file has been applied).
5. Features must be assigned to appropriate layers (by corresponding entity type).
6. All parameters found in the current start part/assembly must exist and be designated.
7. All layers found in the current start part/assembly must exist.
8. All views found in the current start part/assembly must exist.
9. Insert mode may not be active.
10. Length and mass units must correspond with those in the start parts.
11. Unused drawing sheets are not allowed.
12. Any dimensions in drawings that may not be regenerated are not allowed.
13. Drawing format must be current.
14. The ModelCHECK ignore function may not be used.
15. Relation Check

Any checks resulting in a warning shall be investigated, as they are case dependant occurrences. When ModelCHECK is run the following checks will result in a warning:

1. Model name shall be equal to #####_*. (Checks that the file name begin with five numbers and an underscore).
2. Relation errors shall not exist.
3. In general, erased drawing views shall not exist.
4. Round children (features that are children of rounds) are not advised as design best practice.
5. Overwritten dimensions in drawings are not advised as a drawing best practice. In certain instances they are required though, such as when showing dimensions for a family table part where values A, B, C, etc. are required.
6. Draft feature children are not normally a design best practice.
7. DTL file settings shall match standard settings. If several items don't match the standard settings, this may indicate that the wrong .dtl file is applied to the drawing.

C.9 Geometry Checks

Creo Parametric issues a "geometry check" warning when a model is regenerated to indicate that there may potentially be some risk with the model's geometry. Geometry Check is a warning of potential risk, but does not guarantee that there will be any problems with the model. Currently there is no explicit indication of the severity of a geometry check.

Examples of a reported geometry check include:

A geometry check warning of a *tiny* edge detected. The geometry check probably doesn't affect the model accuracy or a drawing of said model. Some risk is present regarding the export of "clean data" There is also a small amount of risk that further design changes may put the model at risk of failure.

1. Assess Risk:

Factors that may be used to determine the risk associated with a geometry check include:

- Location
- Anticipation of Design Modifications
- The Use of Downstream Applications: Including the use of shared data features and the export of model data for simulation, analysis, and manufacturing.
- Quality of Design Techniques:

2. Investigate solutions:

   Alternatives for resolving geometry issues include:

   - Redefine the affected feature so the Geometry Check does not occur. This may be as simple as creating a Sketcher constraint to align an edge to a vertex instead of having the geometry driven by a dimension. In more complex cases, the elements of the feature may need to be redefined (for example, by making a round using Surf-Surf instead of Edge Chain.)
   - Create the geometry with a different type of feature to allow more control over the explicit referencing of existing geometry.
   - Change the order of feature creation. For example, create a round feature earlier in the feature list instead of near the end.
   - Create the geometry with multiple features instead of one, or vice versa. This may allow more explicit control over the references or shapes of the features involved.
   - Finally, there may be the possibility that the geometry has been created according to the design intent and there seems to be no way to avoid the Geometry Check.

3. Implement the best solution:

   Choose the most efficient and appropriate solution from above.

C.10 Mapkeys

See FMTV Creo Parametric Configuration Settings

C.11 Symbols

Symbols are located in the Standard CAD Template Library / Symbols. They will also appear in the default User Symbol directory in Creo Parametric. The list of available symbols in Commonspace can grow as more are created. Separate sites can also create their own specific symbols and stored outside of Commonspace if so desired.

C.12 Skeletons and Assemblies

C.12.1 Scope

Skeletons are models that are used for managing and building assemblies for installation and configuration models and drawings. The skeleton is comprised of datum curves, points, surfaces, axes, and coordinate systems but no solid features. The skeleton shall have a base coordinate system that corresponds with the vehicle coordinate system (Refer to ***Appendix E***) . The addition of multiple skeletons to build complete vehicle configurations is allowed. Skeletons will generally look like a 2D or 3D "stick figure" sketch with many points, axes and coordinate systems. The skeleton needs to have the necessary information to locate the components "in body" for a given installation. For several Installation drawings, the components are spread out all over the vehicle, and constraining the individual components by other means can be very time consuming and not robust.

C.12.2 Method

Each installation model shall contain a coordinate system corresponding with the vehicle three dimensional reference system which is oriented with the 'x" direction running along the centerline of the vehicle with "+x" going rearward, "+y" going to the curbside and "+z" going up.

The installation model should be built on a skeleton with the primary components tied directly to the skeleton. Sub-components that attach to a primary component will be mated to that primary component. For example: a bracket assembly that mounts to the frame will be assembled with skeleton constraints while the hardware and other components mounted to the bracket assembly.

The skeleton constraint shall be a coordinate system (CS) created for a unique component and renamed in the model tree as "part number" for that particular component. If more than one component of the same name will exist in the same skeleton, either rename the CS with the part number followed by consecutive numbers (part number-2) or if it is a symmetrical application, use -LH or -RH behind the part number. By constraining the coordinate systems at assembly the component will be in the correct location in the assembly.

It is essential to have a corresponding coordinate point in the component being assembled to the skeleton. The desired coordinate system for the component would be the default CSO so that creation of additional coordinate systems would not be necessary. If a unique coordinate system is required, rename it as ***part-number_ASSY_CSYS***. The renamed coordinate system identifies that the coordinate system feature was created for a skeleton assembly and shall not be deleted from the file. The coordinate system for a component must be made to a feature on the component that is not likely to move.

The creation of the coordinate system may require additional datum curves, axes, or datum planes for proper creation. If so, in both the skeleton and the component, rename them so that a modeler will not accidentally delete them at another time. Other information to keep in mind:

- The skeleton will remain behind the scenes in the drawing and shall not be included in the BOM.
- The skeleton can be either a single model or an assembly of multiple skeletons.
- The skeleton naming convention uses the _skel series and should be related to the assembly for which it is being created. For Example: the skeleton for 57K2033 is named 57K2033_skel
- Skeletons shall be layered off/hidden so they are not displayed when viewing assembly they are used in.

C.12.3 Example of an Installation assembly using Skeletons:

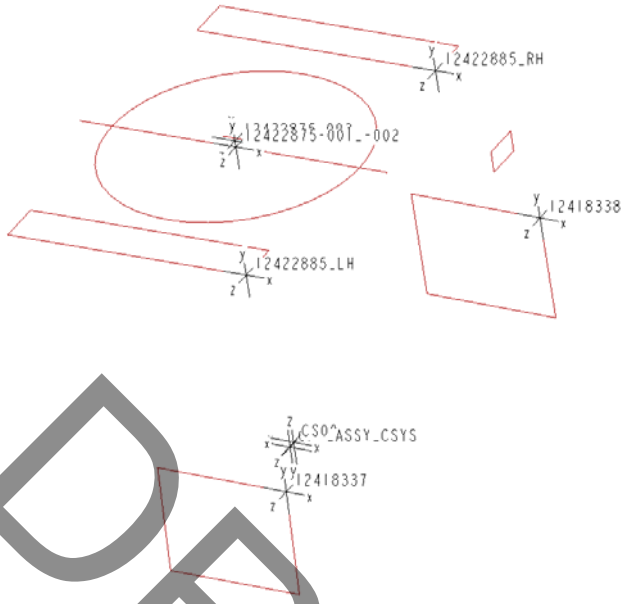


FIGURE 41: BEGINNING WITH SKELETONS

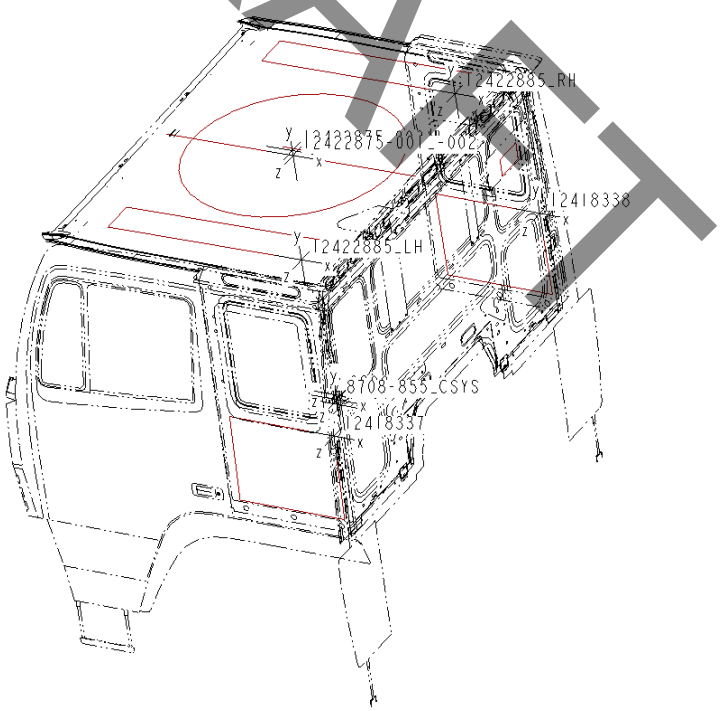


FIGURE 42: 2D CURVES AND CSYS FOR ASSEMBLY

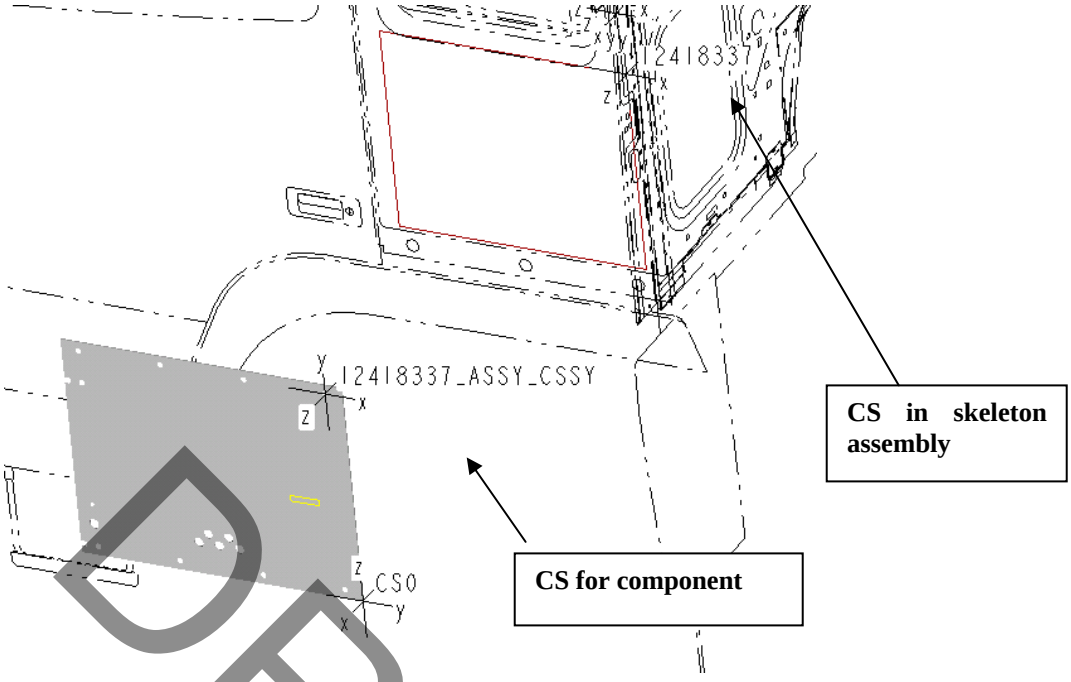


FIGURE 43: IMPORTED PHANTOM GEOMETRY

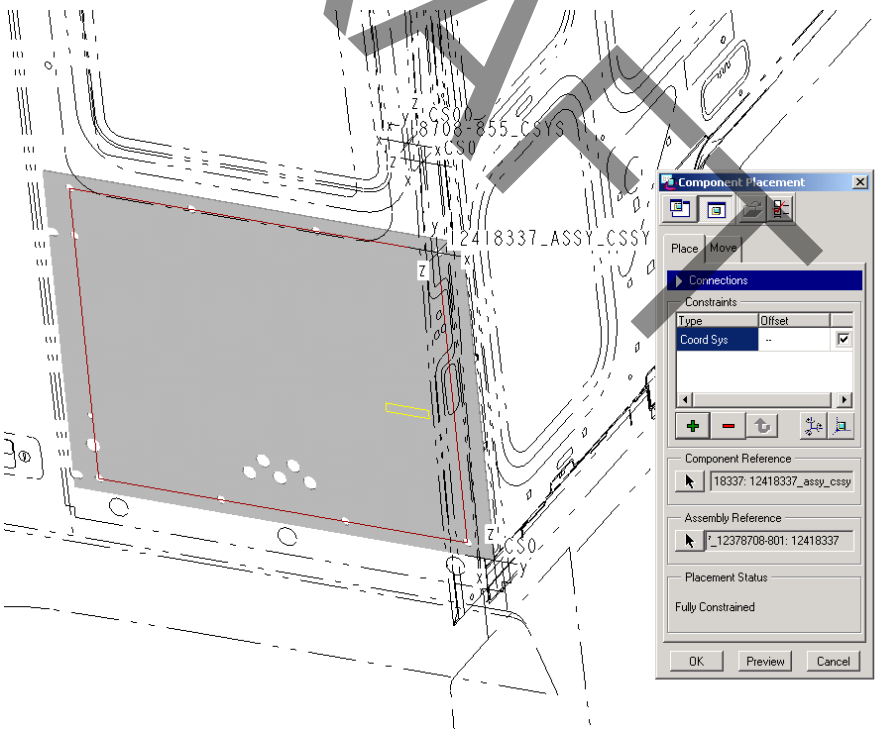


FIGURE 44: COMPONENT PLACEMENT DIALOG BOX

View of completed assembly from the top looking down at the roadside of the cab

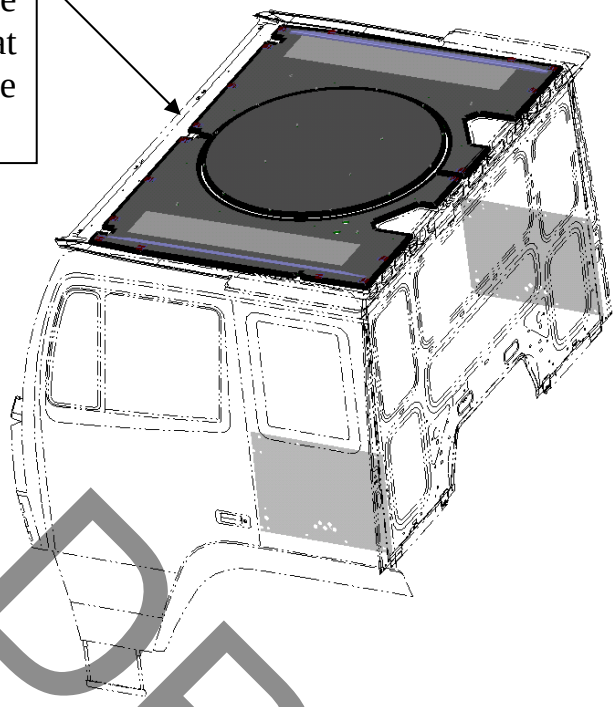
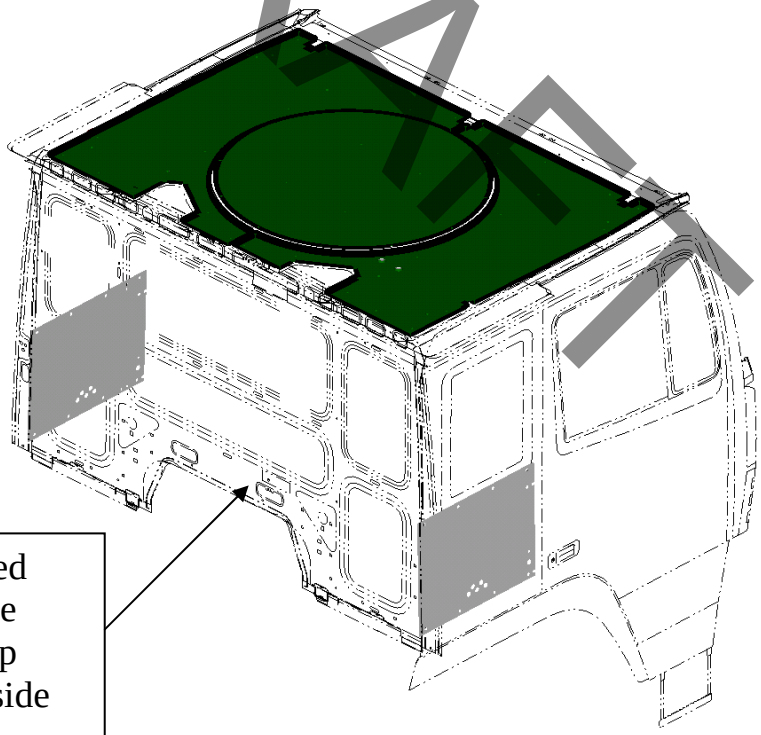


FIGURE 45: TOP VIEW OF CAB ASSEMBLY



View of completed assembly from the bottom looking up towards the roadside of the cab

FIGURE 46: REAR VIEW OF CAB ASSEMBLY

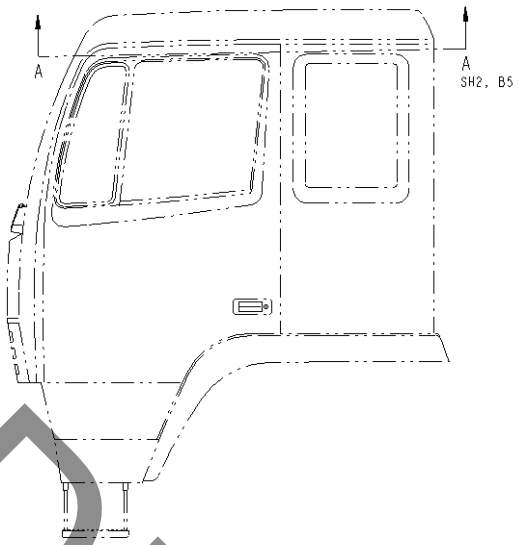


FIGURE 47: COMPLETED DRAWING - CAB SIDE VIEW

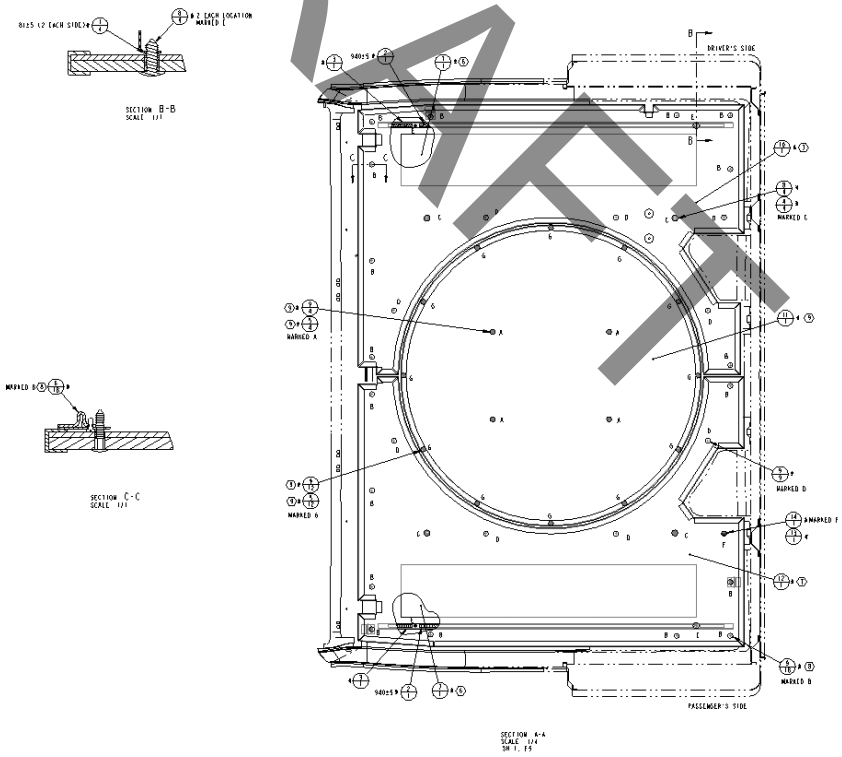


FIGURE 48: COMPLETED DRAWING - CAB TOP VIEW

C.13 Installation "Split Balloon" Find Number Technique

C.13.1 Scope

This is the required method of identifying components on an assembly or installation using the parametric BOM in Creo Parametric. The split balloon has the Find Number on the upper hemisphere and the Quantity on the lower hemisphere. Within Creo Parametric, the option for this balloon is "***with Qty***" when selecting the balloon region. This technique allows components with multiple instances to maintain the total quantity in the BOM as the balloons are disseminated across the drawing. The default places one balloon for each find number with all quantities assigned to that one balloon. The balloons can then be separated between views and pages and maintain the proper quantities for each balloon in each view or balloon location.

There is a command named ***split***, which means to split the base balloon into separate balloons and is not to be confused with the term ***split balloons***.

For reference part callouts on an assembly, reference balloons are the required method for identifying reference components. Use appropriate method to create parametric reference balloons as required by Creo Parametric. The one exception is if the reference item is from a component not contained in the Bill of Material, then a flag callout shall be used for reference.

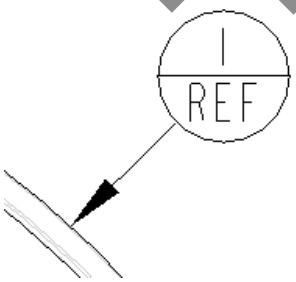


FIGURE 49: REFERENCE BALLOON FORMAT

C.13.2 Procedure

1. Retrieve the table **sbpartslist.tbl (or sbpartslist_spec.tbl)** into the drawing.

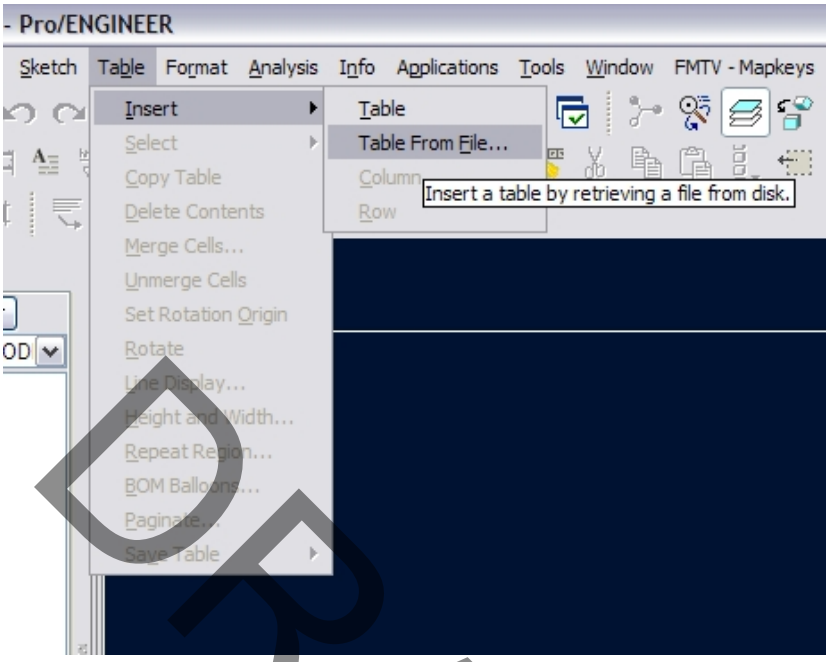


FIGURE 50: STEPS TO INSERT SPLIT BALLOON TABLE

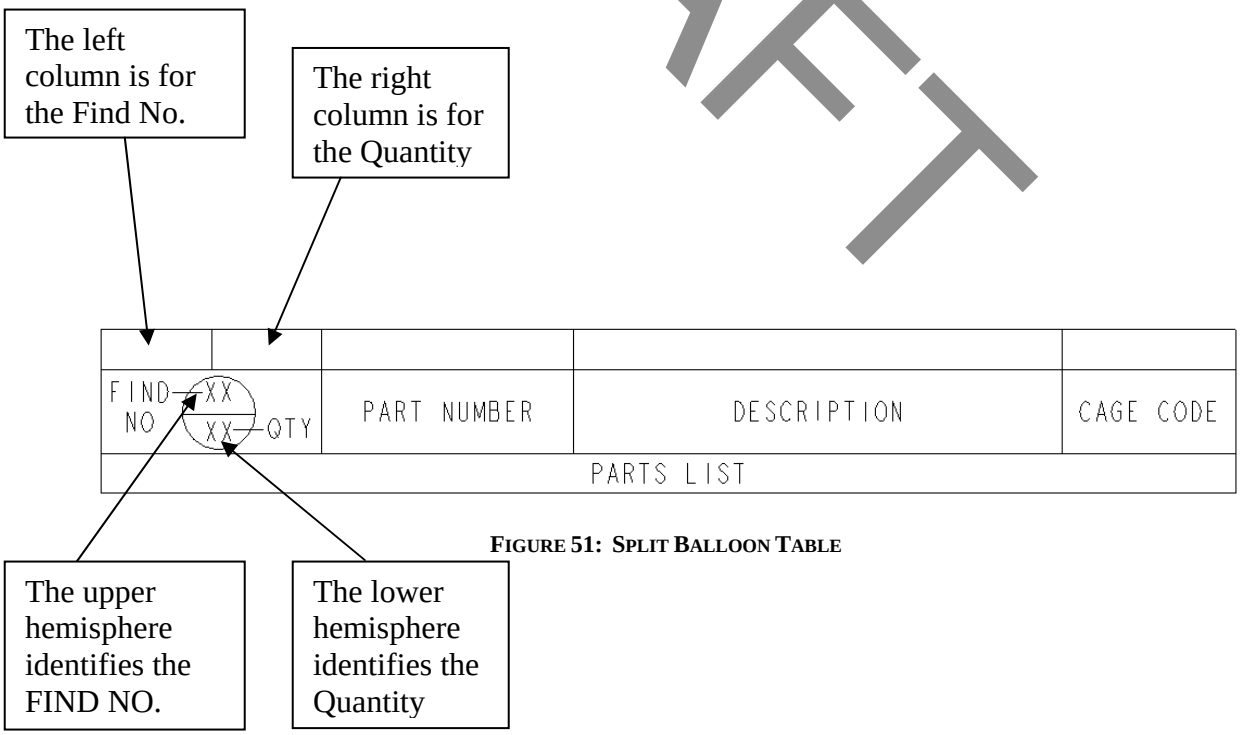


FIGURE 51: SPLIT BALLOON TABLE

2. The Parts List Table will automatically fill out with the contents of the BOM.

20	1	12423650	WASHER, PLAIN	19207
19	3	12417975-006	WASHER, FLAT	19207
18	1	12423649	SCREW, MUSHROOM HEAD	19207
17	1	12418338	PANEL, INTERIOR, VEHICULAR, RIGHT SIDEWALL	19207
16	4	12414473-011	WASHER, PLAIN	19207
15	1	12418337	PANEL, INTERIOR, VEHICULAR LEFT SIDEWALL	19207
14	1	12422875-013	HEADLINER, RH PANEL ASSEMBLY	19207
13	1	12422875-003	HEADLINER, HATCH COVER ASSEMBLY	19207
12	1	12422875-011	HEADLINER, LH PANEL ASSEMBLY	19207
11	43	12422873-002	FASTENER, HEADLINER	19207
10	8	12422873-001	FASTENER, HEADLINER	19207
9	2	12422885	DAMPING PAD	19207
8	11	12418469	RIVET, COMPRESSION	19207
7	18	12422874	CLIP, HEADLINER	19207
6	16	12422884-002	BUSHING, SELF- LOCKING	19207
5	4	12422884-001	BUSHING, SELF- LOCKING	19207
4	2	12423075	ANGLE, REINFORCEMENT	19207
3	2	12422675	TAPE, DOUBLE SIDED BONDING, 940mm LONG	19207
2	4	12422675	TAPE, DOUBLE SIDED BONDING, 80mm LONG	19207
1	7	12414419-076	SCREW, HEX HEAD	19207
FIND NO — XX / XX — QTY		PART NUMBER	DESCRIPTION	CAGE CODE
PARTS LIST				

FIGURE 52: COMPLETED SPLIT BALLOON BOM TABLE

C.13.3 Split Balloon Callout Generation

The split balloons are created by selecting:

Table > BOM Balloons > Set region > with Qty and select the Parts List table region, and then ***Done.*** Go to the primary drawing view, and on the **Tables** > **BOM Balloon** tab select **Show** and select the primary view. The balloons show up automatically.

To divide the balloons up into smaller quantities, (while still in the BOM Balloon drop-down menu) select the ***split*** command and select the balloon to be divided, and enter the amount to be taken out of the total quantity from the base balloon. Then at the prompt, select the new component location to have the balloon and pick the location for that balloon.

This portion of the procedure is the same for other methods, but the quantity remains visible and parametric to the system generated BOM table.

From this point, the balloon manipulation sequences (detach, merge and redistribute) work the same as other balloon methods.

*Example of Split Balloon callouts for the table shown in **Figure 53** below:*

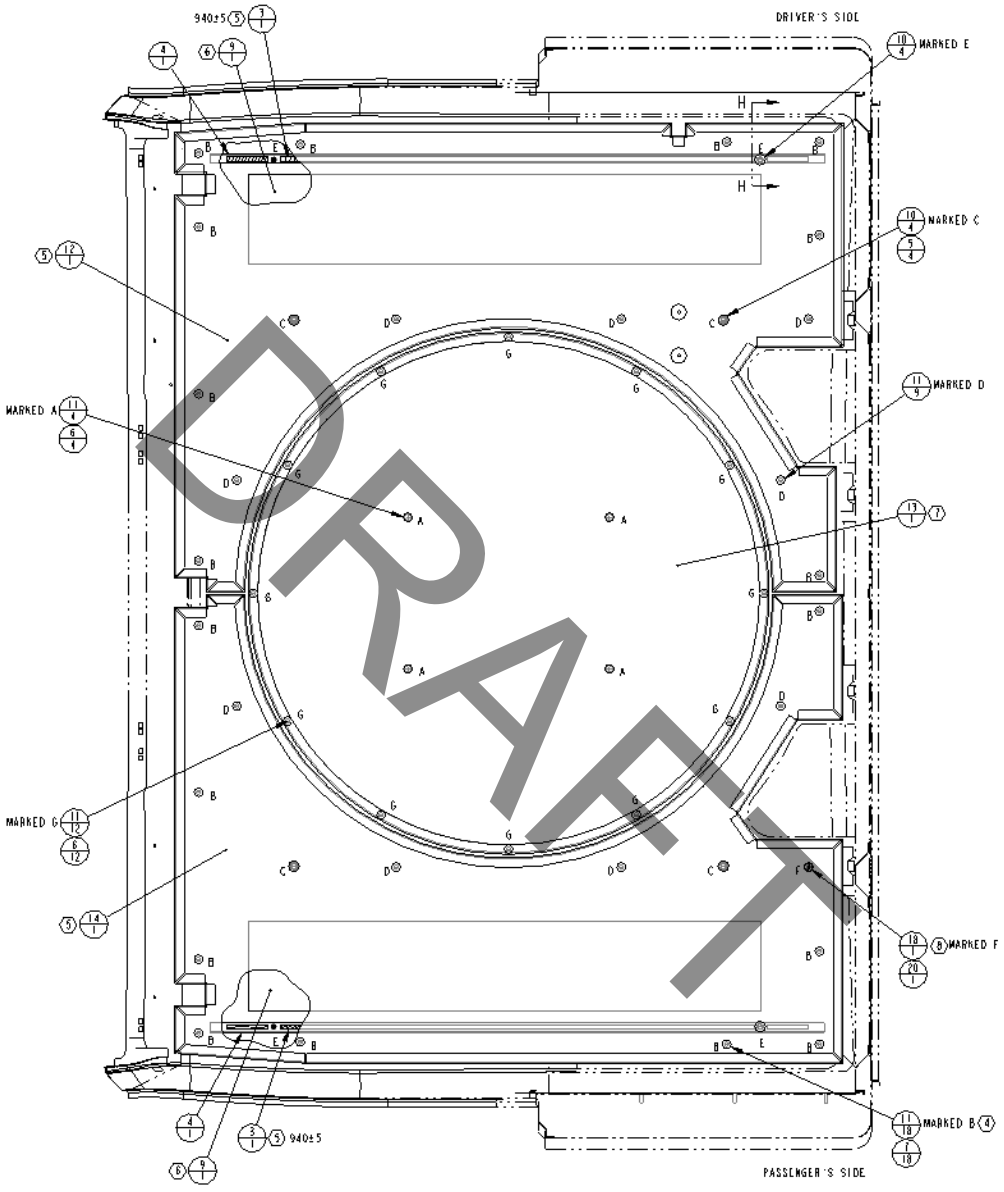


FIGURE 53: EXAMPLE OF SPLIT BALLOONS ON DRAWING

C.13.4 Split Balloon Callout for Multi-Configuration BOM Report Table

In a Creo Parametric drawing, an assembly drawing format can be modified so the BOM Report Table reflects multiple configurations of an assembly. This can be based either on family table

instances or separate assemblies. The preferred method is to create multiple BOM tables with each instance having its own table.

C.13.5 Split Balloon Callout for Bulk Items

Bulk material items such as convoluted tubing, strip foam seal tapes and other items that are cut to different lengths and have possible different installed state models but have the same part number need to be shown in the PARTS LIST TABLE as a single Find Number. In order to do this, several steps are required, including relations in the PARTS LIST and created split balloons.

1) Parts List Table needs to be set to recursive to the appropriate level
2) The additional Find numbers need to be Filtered out of the Table
3) A relation is written against the Parts List Table to show the total cut length of the bulk item
4) The split balloons for the filtered find numbers are created (dummy) using the Find Number of the totaled bulk item still in the Parts List

In the example below, there are several different sizes of convoluted tubing being used and in multiple quantities. (F/N 41, and 44 thru 49). The lengths of the tubing are also different and the "shape" of the installed state tubing models are different.

49	1	12420924-008-600	TUBING, CONVOLUTED	19207
48	2	12420924-007-605	TUBING, CONVOLUTED	19207
47	1	12420924-008-602	TUBING, CONVOLUTED	19207
46	1	12420924-007-904	TUBING, CONVOLUTED	19207
45	1	12420924-005	TUBING CONVOLUTED	19207
44	1	12420924-008	TUBING, CONVOLUTED	19207
43	11	MS3367-3-0	STRAP, TIEDOWN, ELEC COMP, ADJUSTABLE, SELF CLINCHING, PLASTIC, TYPE I, CLASS	81343
42	12	MS3367-2-0	STRAP, TIEDOWN, ELEC COMP, ADJUSTABLE, SELF CLINCHING, PLASTIC, TYPE I, CLASS	81343
41	1	12420924-007	TUBING, CONVOLUTED	19207

FIGURE 54: UNFILTERED BOM FOR BULK TUBING

The PARTS LIST Table was filtered to remove find numbers 46 thru 49, yielding one Find Number item for each size (dash number) of the convoluted tubing. The total length of the respective tubing sizes were totaled up in the PARTS LIST for the quantity.

45	24 IN	12420924-005	TUBING CONVOLUTED	19207
44	33 IN	12420924-008	TUBING, CONVOLUTED	19207
43	11	MS3367-3-0	STRAP, TIEDOWN, ELEC COMP, ADJUSTABLE, SELF CLINCHING, PLASTIC, TYPE I, CLASS	81343
42	12	MS3367-2-0	STRAP, TIEDOWN, ELEC COMP, ADJUSTABLE, SELF CLINCHING, PLASTIC, TYPE I, CLASS	81343
41	42 IN	12420924-007	TUBING, CONVOLUTED	19207

FIGURE 55: FILTERED BOM FOR BULK TUBING

The cut length for each item is identified next to the split balloon at each location in the installation.

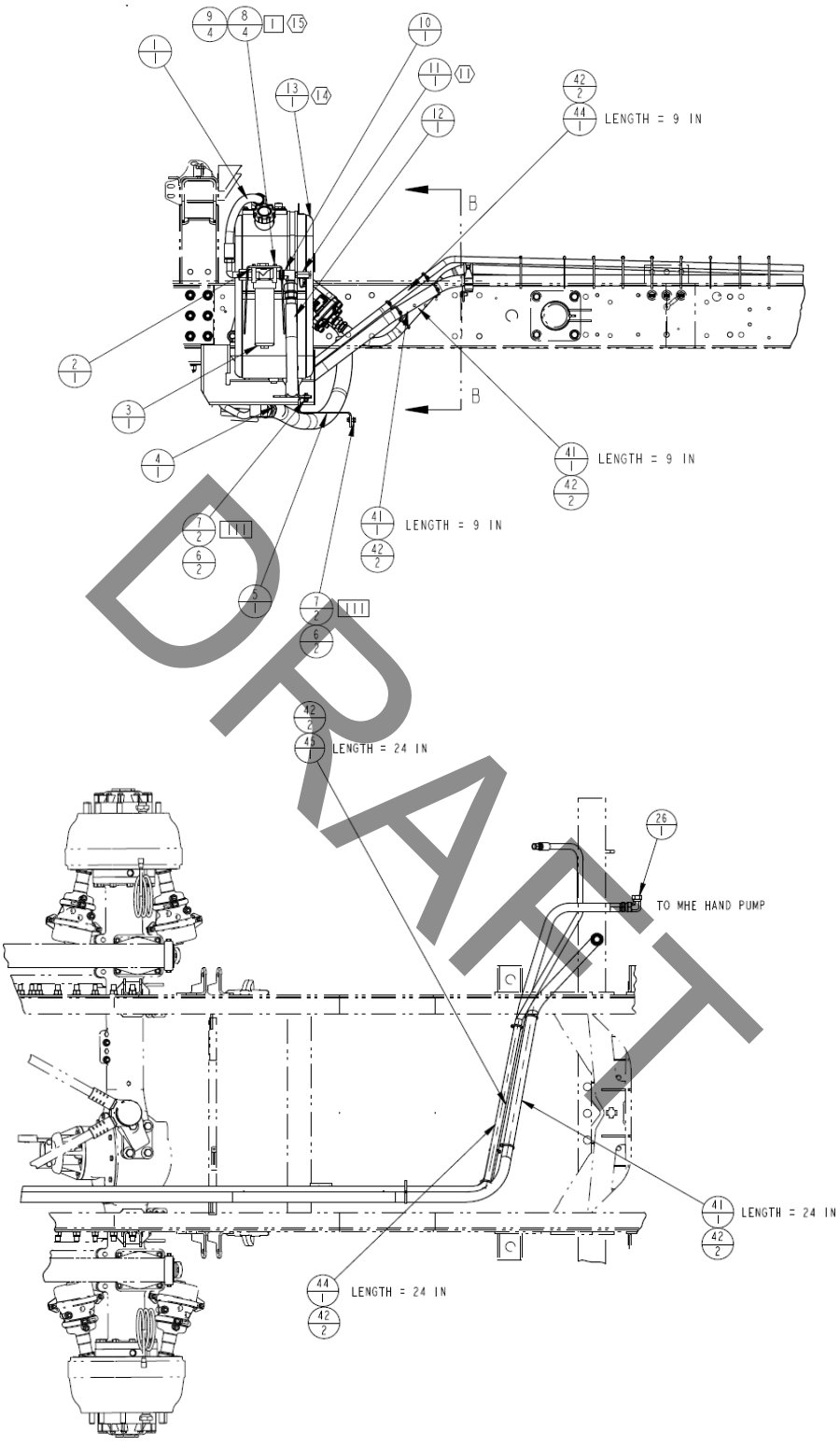


FIGURE 56: BULK LENGTH DEFINE IN VIEWS

The split balloon callouts for the 41, 44 and 45 find numbers were created with a symbol since these belonged to the filtered out find numbers. The software cannot generate the balloons for these items so the balloon symbols have to be used. (NOTE: The symbol may be called BOM_SPLIT_BALLOON also in the Library)

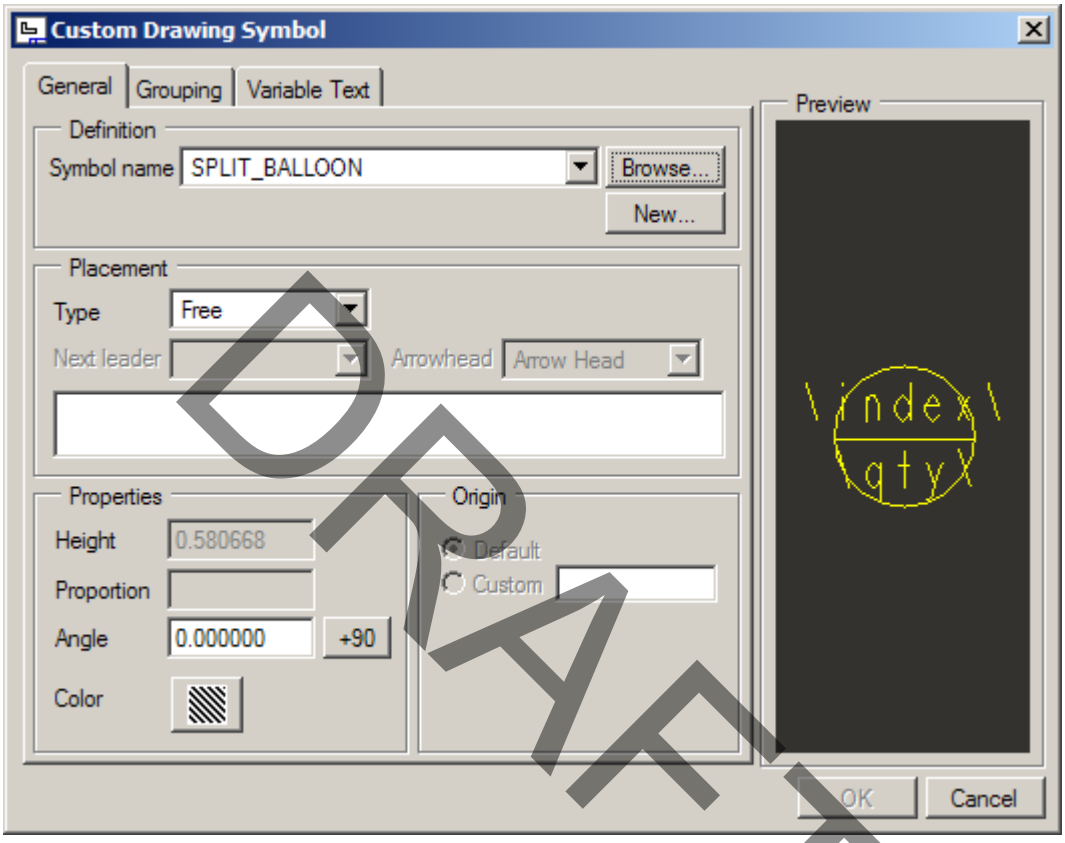


FIGURE 57: SPLIT BALLOON SYMBOL

NOTES:

1) The quantity for each split balloon shall be 1 and the cut length designated either beside or below the balloon. The reason for this is to clarify the cut length for each item and to insure that the correct item is used in the designated location.

2) The total length of the bulk material for all locations is indicated in the quantity column in the Parts List along with the unit of measure as a relation to replace the actual unit quantity.

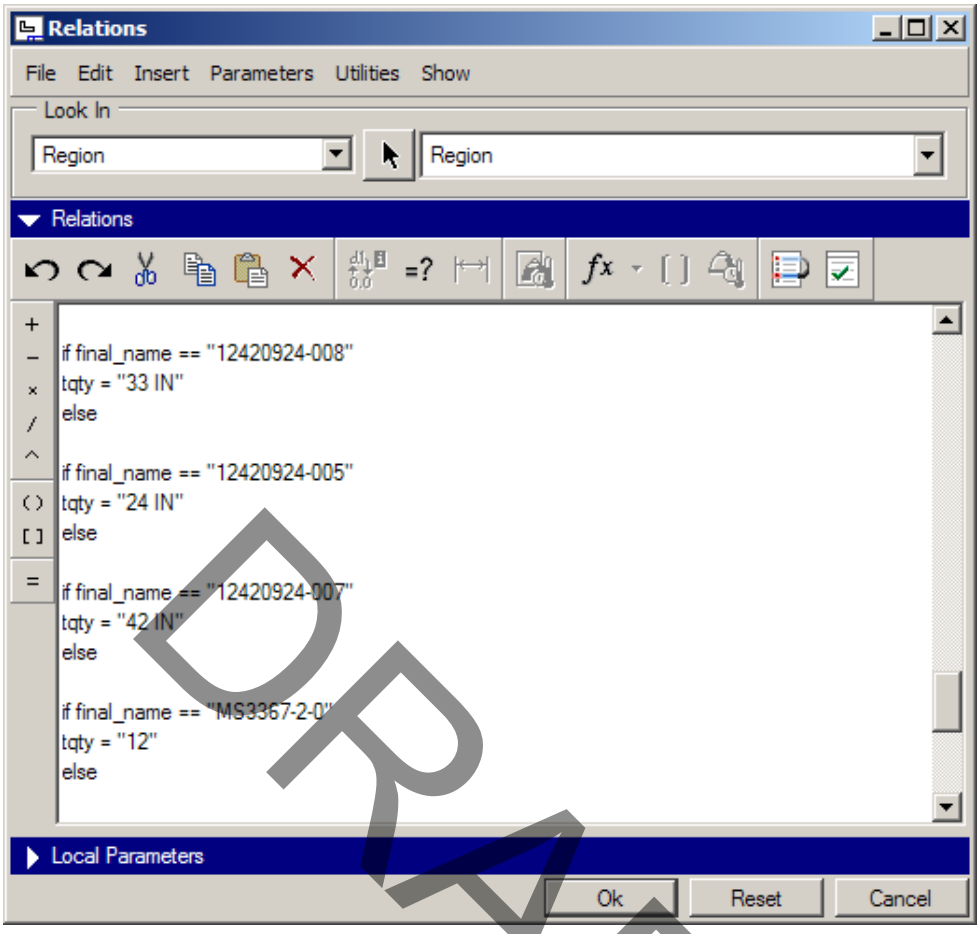


FIGURE 58: RELATIONS FOR THE CUT LENGTH IN THE PARTS LIST

C.13.6 Empty Table Space – Removing Part

When a part is removed from a PARTS LIST and not replaced with an equivalent part, the find number for where the part was removed shall be replaced with an empty table space. To achieve this, use one of the ten available empty_table_space_#.prt parts located in the Standard CAD Template Library / Start Parts and Formats. The empty_table_space_#.prt model is a bulk item and a relation will have to be written in the parts list on the drawing as shown below:

```
if asm_mbr_name == "EMPTY_TABLE_SPACE_1"
final_name = " "
tqty = " "
endif
```

C.14 Creo Parametric pen table settings (default)

See FMTV Creo Parametric Configuration Settings

C.15 PDF print settings

See FMTV Creo Parametric Configuration Settings

C.16 PM-MTV Default configuration file settings

See FMTV Creo Parametric Configuration Settings

C.17 Annotation Features

Annotation features  are data features that can be used to manage the model annotation and propagate model information to other models. An annotation feature consists of one or more Annotation Elements. Each Annotation Element can contain one annotation item, along with associated references and parameters. The following types of annotations can be added using the Annotation element:

- Note
- Symbol
- Surface finish
- Geometric tolerance
- Reference dimension
- Driven dimension with tolerance

C.18 Relations (default)

The relations built into the start part are Feature Relations. The relations can be accessed by selecting **Tools > Relations** and then setting the type to **Feature** and then selecting the default coordinate system **CSO**. ***Table 14***, shown on the following page, lists the current state of relations for the start parts.

```
/*  START OF FMTV START PART RELATIONS
/*  DO NOT DELETE OR MODIFY ANY LINES IN THIS SECTION
/*  LAST MODIFIED: 30 NOV 2004
/*
/*  RELATION TO COMBINE NOMENCLATURE LINES
/*
nomenclature= nomenclature_1 + " " + nomenclature_2 + " " + nomenclature_3
/*
/*  RELATIONS TO ASSIGN DESIGN ACTIVITY AND DESIGN ACTIVITY LOCATION
/*  BASED ON CAGE_CODE
/*
AGENCY="US ARMY"
IF CAGE_CODE == "19200"
  DESIGN_ACTIVITY="ARMAMENT RESEARCH, DEVELOPMENT AND ENGINEERING CENTER"
  DESIGN_ACTIVITY_LOCATION = "PICATINNY ARSENAL, NEW JERSEY  07806-5000"
ENDIF
IF CAGE_CODE == "19204"
  DESIGN_ACTIVITY="ROCK ISLAND ARSENAL"
  DESIGN_ACTIVITY_LOCATION = "ROCK ISLAND, ILLINOIS "
ENDIF
IF CAGE_CODE == "59678"
  DESIGN_ACTIVITY="ARMAMENT RESEARCH, DEVELOPMENT AND ENGINEERING CENTER"
  DESIGN_ACTIVITY_LOCATION = "ROCK ISLAND, ILLINOIS 63299-5000"
ENDIF
IF CAGE_CODE == "19207"
  DESIGN_ACTIVITY="TANK AUTOMOTIVE & ARMAMENTS COMMAND"
  DESIGN_ACTIVITY_LOCATION = "WARREN, MICHIGAN  48397-5000"
ENDIF
IF CAGE_CODE == "0FW39"
  DESIGN_ACTIVITY="STEWART & STEVENSON TACTICAL VEHICLE SYSTEMS, LP"
  DESIGN_ACTIVITY_LOCATION = "5000 I-10 WEST, SEALY, TEXAS  77474"
ENDIF

if auto_part_number
        /*
        /*  CHECKS FILE NAME LENGTH, LOCATION OF UNDERSCORE, LOCATION OF HYPHEN
        /*
        file_name= REL_MODEL_NAME()
        under_loc = search(file_name,"_")
        dash_loc = search(file_name,"-")
        file_len = string_length(file_name)
        /*
```

```
            /*  DETERMINES IF THERE IS A CAGECODE AND/OR A DASH NUMBER AS PART OF THE FILE
            /*  EXTRACTS PART_NUMBER AND EXT_PART_NUMBER FROM FILE_NAME
            /*
            IF under_loc !=6
              IF dash_loc !=9
                drawing_number = file_name
              dash_no = ""
              part_number = drawing_number
              ELSE
                drawing_number = extract(file_name,1,dash_loc-1)
              dash_no = extract(file_name,dash_loc+1,file_len-dash_loc)
              part_number = drawing_number + "-" + dash_no
              ENDIF
            ELSE
              cage_code = extract(file_name,1,under_loc-1)
              IF dash_loc != 15
                drawing_number = extract(file_name,under_loc+1,file_len-under_loc)
              dash_no = ""
              part_number = drawing_number
              ELSE
                drawing_number = extract(file_name,under_loc+1,dash_loc-under_loc-1)
              dash_no = extract(file_name,dash_loc+1,file_len-dash_loc)
              part_number = drawing_number + "-" + dash_no
              ENDIF
            ENDIF
endif
/*
UNIT_WEIGHT=PRO_MP_MASS
/*
/*  END OF FMTV START PART RELATIONS
/*
```


TABLE 13: START PART RELATIONS

C.19 CHECK mapkey

See FMTV Creo Parametric Configuration Settings document

C.20 Component Interface functionality

Component interface functionality was added in Creo Parametric Wildfire. They contain a set of stored constraints and references that can be used repeatedly to place components. Once an interface is defined and saved in a component, it can be used anytime the component is placed in the assembly. Placement of components using interfaces allows them to be constrained more efficiently.

Currently Component Interfaces are not defined on any of the parts. If a user needs to assemble the same component many times they can create a Component Interface on the "In-Session" model. By adding the Component Interfaces "In-Session" and using it at that time, as long as they don't save the component, it will not affect the model. These Component Interfaces can be created in both Part and Assembly mode.

C.21 AUTO_PART_NUMBER functionality

The AUTO_PART_NUMBER parameter was added recently to the start parts. If the value is set to YES then the model automatically takes the name of the part and assigns it to the PART_NUMBER parameter. If it is set to NO then it leaves it empty and the user is required to enter the value for the PART_NUMBER parameter.

C.22 Item ID

The Item ID is a special feature used for including the ID of the model on the model with a curve. This special sketched curve feature needs to be handled differently from other curves on the model. The feature shall be renamed to "ITEM_ID", and it shall be added to the "ITEM_ID" layer so that the user can easily isolate this curve feature from others in the drawing. Typically, the Item ID is isolated in order to be displayed on the drawing while hiding the other datum curves. The ITEM Id shall be the ACTUAL ITEM ID that would be marked on the part. A sketch feature can be used to get the text on the drawing.

C.23 Mass Properties – Manual Input

In certain cases it is helpful to be able to enter the mass of a model manually. Typically Creo Parametric will calculate the mass of the model automatically based on the assigned density and the

calculated volume. Often times when dealing with purchased parts or where the material properties aren't exactly known, it is helpful to be able to add the final mass manually. In order to complete the task select **Edit** > **Setup** > **Mass Props** then choose **Geometry and Parameters** as the **Source.** Then choose **Edit** from the **Actions** area and enter in value for **Mass.** This value becomes known as the PRO_ALT_MP_MASS parameter.

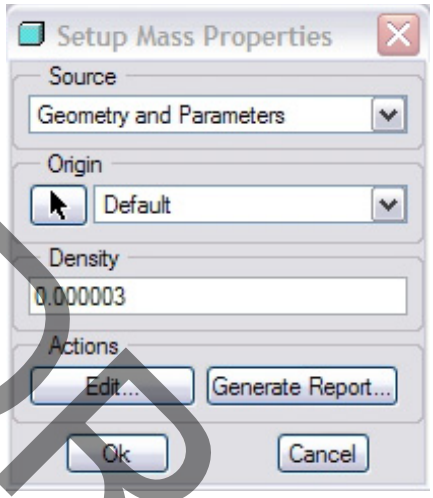


FIGURE 59: SETUP MASS PROPERTIES

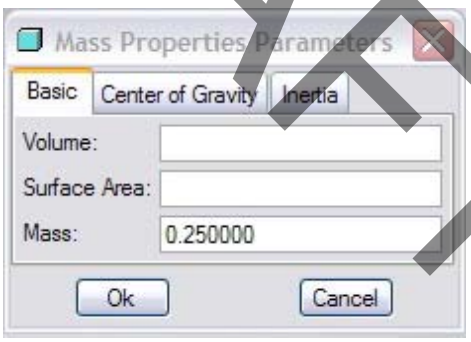


FIGURE 60: MASS PROPERTIES PARAMETER

C.24 Military parts and Vendor parts

Typically, military parts are recognized by either the name of the file or by having a CAGE_CODE parameter filled out. Below is a list of typical military part numbers (either 7 or 8 digits in length) they can either have the cage code preceding the name or only as a value for the parameter.

- 12xxxxxx
- 116xxxxx
- 87txxxx (top drawing)
- 57kxxxx (kit list)

If it doesn't meet the above criteria it could be a vendor supplied model; however, there are exceptions so this will need to be verified.

C.25 Vendor supplied 3D models

The way to handle vendor supplied 3D models has changed over time. It is recommended to maintain the vendor's naming convention so when newer models are supplied they can simply be updated in Commonspace.

A common issue when dealing with vendor supplied CAD models is data transfer. Transferring 3D solid geometry through CAD neutral translation can cause problems when importing them into Creo Parametric. Factors that affect the quality of imported data are:

- Original CAD system
- Quality of modeling
- Model accuracy
- Model complexity
- Irregularity of Geometry
- Type of exchange file (IGES, STEP, Neutral, etc...)

3D geometry imported into Creo Parametric for the MTV database shall be converted to solid geometry. The conversion to solid models helps ensure that proper model display and mass calculations can be made. If the task of converting the imported geometry from surfaces to solids is beyond the ability of the modeler then a modeler with the appropriate ability should be tasked with converting the model.

The neutral file shall be stored in the FMTV Product under the **Models and Drawings/OEM_Models** folder in Windchill PDMLink

C.26 Next Assembly Parameter

There have been issues with the next assembly parameters with some parts that are in Commonspace. The NEXT_ASSY parameter shall represent the assembly where the part is used. The problem is mainly found with parts (ex: -001, -501, and -801) where the next assembly parameter has been filled out incorrectly.

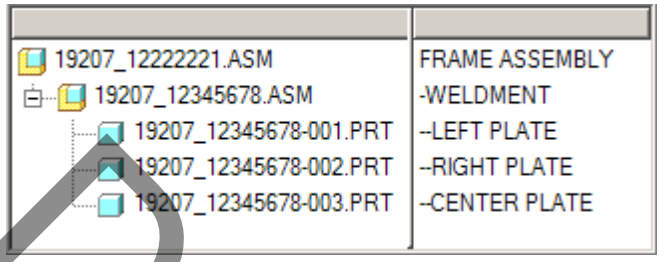


FIGURE 61: ASSEMBLY STRUCTURE - NEXT_ASSY

The left-plate, right-plate, and center-plate are the parts that make up the weldment. In some occasions it has been noted that the NEXT_ASSY parameter for the plates has not been 12345678, it has been 12222221 (Frame Assy). The frame assembly is the up assembly in which the weldment is assembled. The plates shall have 12345678 as the next assembly. The weldment shall have 12222221 (Frame Assy) in its next assembly parameter.

If a part or assembly has multiple up assemblies, then the up assemblies are listed in the parameter numerically.

NEXT_ASSY_1	12401111
NEXT_ASSY_2	12402222
NEXT_ASSY_3	12403333
NEXT_ASSY_4	

FIGURE 62: NEXT ASSEMBLY LIST

In situations where there are more than eight up assemblies, do not create additional NEXT_ASSY parameters. For the drawing, a hex note can be placed in the NEXT_ASSY parameter

box located in the title block to relate back to the drawing notes. The drawing note will list all of the next assemblies used.

The next assembly parameter is based on the assembly that a part or assembly is configured into. A Where-Used Report can be run in Windchill PDMLink to show what assemblies a part or assembly may be a component of.

C.27 Handling a Cancelled ECP

Occasionally after initiating an ECP that involves changing various parts, drawings, and assemblies it gets cancelled. When it gets cancelled a concern is what to do with all the changes that were completed on the parts, drawings, or assemblies. Ideally, all the changes on these objects shall be deleted or rather the PIV (Product-Item-Version) of these objects shall be deleted from Commonspace. Typically a lot of work goes into making these changes, which more than likely will be requested again in the future, and it is not cost effective to simply delete them. Some guidelines have been set to dictate how to deal with these objects.

- If the objects are the INITIAL RELEASE or CONVERTED TO PROE SOLID MODEL then the Release Level shall be set back to Concept and the models shall be left intact.
- If the previous Revision has been promoted through to RELEASED and the changes made to the objects were minor (i.e. note change, parameter change, etc…) then the objects shall be deleted from Commonspace.
- If the previous Revision has been promoted through to RELEASED and the changes made to the objects were major (i.e. major geometry changes, major drawing changes, or anything time consuming to complete) then the objects shall be set back to Concept or deleted,.

C.28 Keep All Down Parts and Drawing at Same Revision when Promoting

There have been many issues related to not keeping the same base number components at the same revision level. The base number refers to the 8 digit number after the cage code and underscore in the name of the object. The rule is that all components with the same base number must be revised

when the drawing is revised. If drawing 19207_12341234.drw is being promoted to a new revision then all models beginning with 19207_12341234 shall be promoted along with the drawing unless noted on the NOR. An exception is when in the newer revision one or more of the components are no longer required and those can remain at the previous revision. All of these components shall be related to the drawing even if there are no views describing the individual components.

C.29 Weldments

All weldment component parts shall follow the -100 naming convention (***as defined in Section 3.8***) for the inseparable assembly. In addition, the PARTS LIST table for the weldment shall specify "NON-REPAIRABLE PARTS LIST" (***See Figure 63***)

5	1	12423058-105	SUPPORT	19207
4	1	12423058-104	PLATE, TOP	19207
3	2	12423058-103	RUNG	19207
2	2	12423058-102	UPRIGHT	19207
1	1	12423058-101	PLATE	19207
FIND NO XX / XX QTY		PART NUMBER	DESCRIPTION	CAGE CODE
NON-REPAIRABLE PARTS LIST				

FIGURE 63: SAMPLE WELDMENT PARTS LIST

D Procedures

D.1 Creating a new file

The following procedure shall be used whenever creating a new model. This will ensure conformance to the latest standards and will allow the models to be tracked by Windchill PDMLink. This is very import in order to avoid any duplication of work.

1. All NEW parts shall be created with the approved FMTV start parts.
2. All required parameters shall be filled out before checking files into Commonspace
    - ECP_ERR_Number is important to know what ECP created the new file

D.2 Family Table Drawing Creation Procedure

D.2.1 Creating the Family Table in the Part:

Create a family table with all of the information required to model the part. Each instance will need to be given a distinctive name based on the original part number with a 19207_ prefix. The part_number parameter shall be added to the family table to correctly show the part number in the drawing table. Additional parameters may be required to correctly represent the model at different part levels. Include any technical data using string parameters, i.e.: din_number or thread_size.

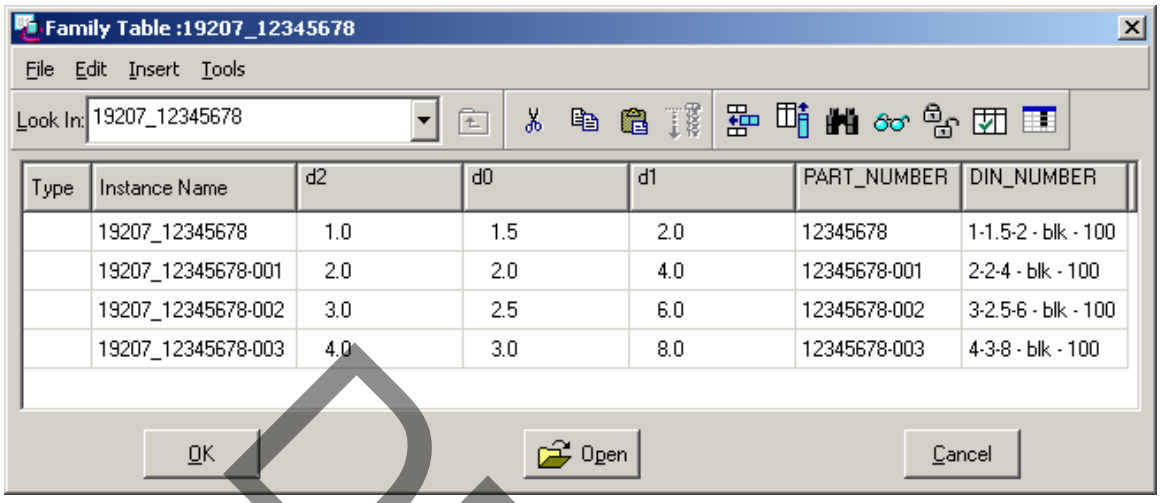
Because of issues in the past in using Family Table functionality with Creo Parametric assemblies it is important to only use them when it makes sense. In order to decide when to use a family table to aid in building / maintaining many models follow these general guidelines or ask a local CAD admin or experienced user.

Do NOT use Nested Family Tables ever. They will cause problems with the database management system.

D.2.2 Auto Fill Instructions:

The family table can be filled out quickly by exporting the data to Microsoft Excel using the following icon  . Use the **Fill > Series** command to automatically complete column. Make sure the column to be filled is empty and then highlight the area to be filled. From the top pull down menu

select: **Edit > Fill > Series** > **Set Type** to **AutoFill** and select OK. Once this is complete, the family table should look like the following figure:



Type	Instance Name	d2	d0	d1	PART_NUMBER	DIN_NUMBER
	19207_12345678	1.0	1.5	2.0	12345678	1-1.5-2 - blk - 100
	19207_12345678-001	2.0	2.0	4.0	12345678-001	2-2-4 - blk - 100
	19207_12345678-002	3.0	2.5	6.0	12345678-002	3-2.5-6 - blk - 100
	19207_12345678-003	4.0	3.0	8.0	12345678-003	4-3-8 - blk - 100

FIGURE 64: FAMILY TABLE

D.2.3 Showing the Family Table in the Drawing:

1. Start by creating a table with a header row and one empty row for the first field. Only create columns for the data that needs to be shown in the table. Some Family Table columns are used only to model the part and may not need to be shown in the drawing table.

PART NUMBER	WIDTH	DEPTH	DIN MUNBER

2. Create simple repeat regions in all of the cells:

Table > **Repeat Region** > **Add** > **Simple** in each column, below the title, *pick each of the empty cells twice* to create a repeat region.

3. In each column enter: &fam.inst.param.value. :

Enter Text > **Report Sym** select the cell to enter text **fam… inst… param… value**

PART NUMBER	WIDTH	DEPTH	DIN MUNBER
fam.inst.param.value			

4. Add a filter to each of the repeat regions to only show the values intended for that column. For part number, create the filter: **&fam.inst.param.name==part_number**.

Table > **Repeat Region** > **Filters** > **Add &fam.inst.param.name==Parameter Name - Done - Done/Return.**

PART NUMBER	WIDTH	DEPTH	DIN MUNBER
12345678-001	2.0	2.0	2-2-4 - blk - 100
12345678-002	3.0	2.5	3-2.5-6 - blk - 100
12345678-003	4.0	3.0	4-3-8 - blk - 100

D.3 2D Lightweight Backdrop Creation

This process is for creating 2D lightweight backdrops for use in Installation and Assembly drawings as phantom reference geometry.

Advantages

- Dramatically reduce the file retrieval and regeneration time
- Geometry is clean (other methods require a substantial amount of clean-up)
- No “tag-along” parts (additional parts not used in the installation shall not be part of the assembly)
- Skeletons can be embedded into the backdrops
- Clean BOM
- Can create a library of standard views
- Useful during the design process as well.

Step 1 - Identification

The following items should be considered when determining what the backdrop geometry will be composed of:

- Determine the geometry and views necessary for the drawing.
- Build a temporary assembly model or use an existing assembly (using only the components that will be seen).
- Avoid using any shrink-wraps in the model structure.
- Establish a known coordinate system location:
    - For small installations with isolated components (components located in the same general area), use the interface for the primary component (this method allows for the lightweight backdrop to serve as a skeleton as well
    - For large drawings with components spread across the chassis, use the default coordinate system (A separate skeleton file for the multiple primary components will be required and the associated skeleton(s) will require the same coordinate system).

Step 2 - Establish offset coordinate systems

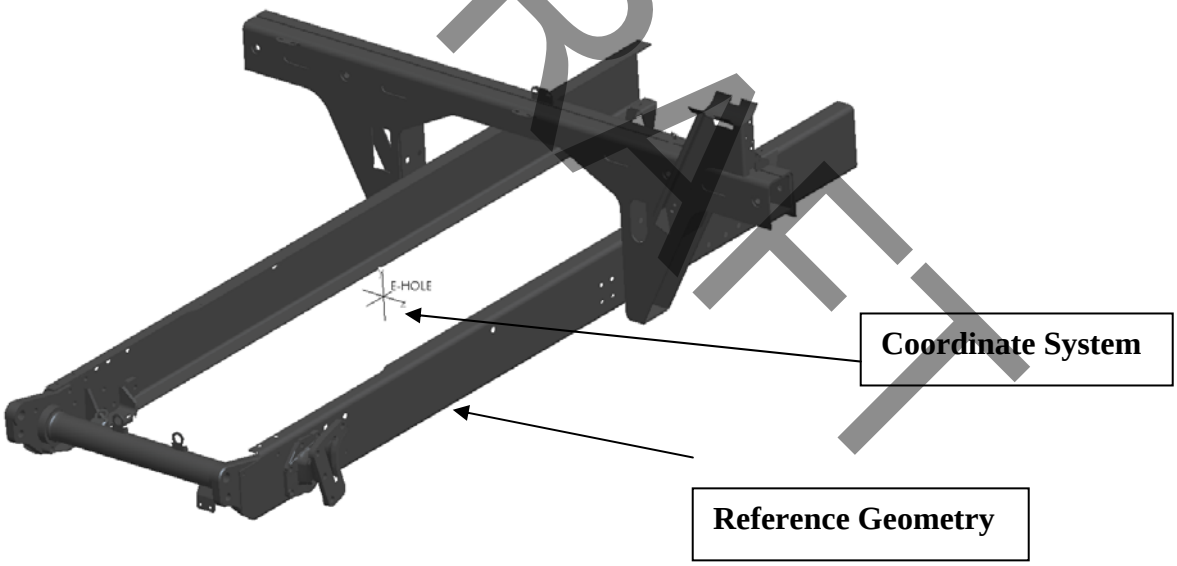


FIGURE 65: ESTABLISH OFFSET CSYS

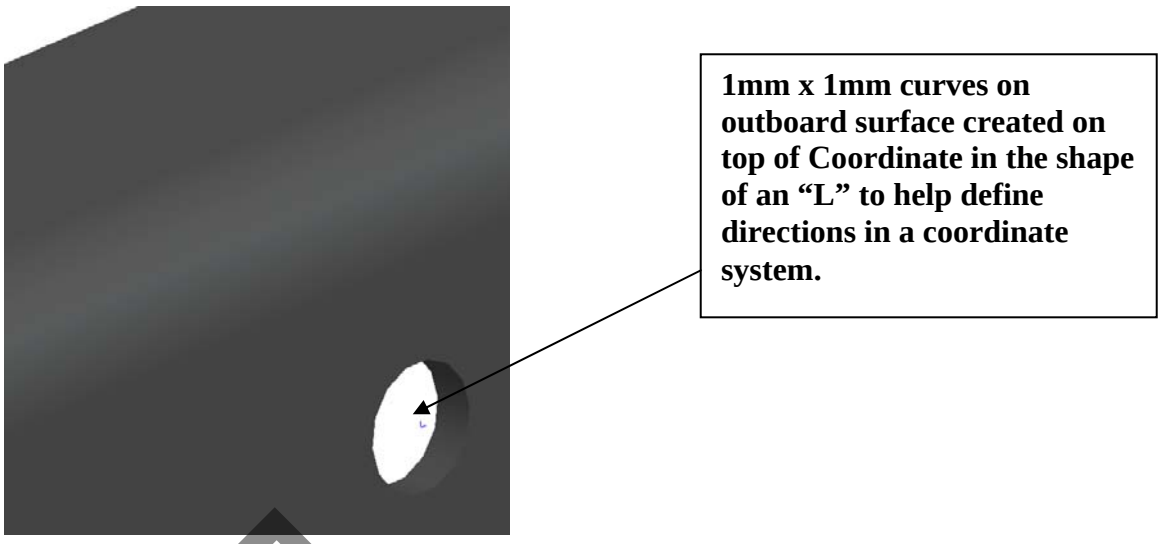


FIGURE 66: CREATE 2 - 1MM X 1MM CURVES

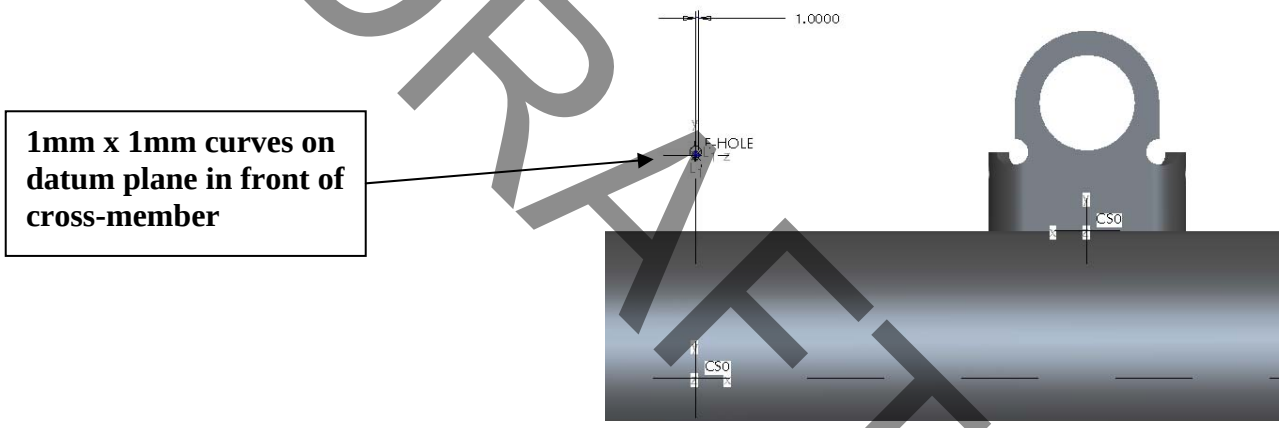


FIGURE 67: ANOTHER 1MM X 1MM CURVE SET

Step 3 – Make DXF files

- Create a new temporary drawing, empty, with a variable sheet size at 1 x 1.
- The format is not required; make the border size small for ease of deletion later.
- The drawing has a base coordinate at the bottom left corner.
    - The location of the coordinate on the drawing will not work for lightweight backdrops
    - The coordinate curves created in the prior step will be used to create the assembly coordinates in a later step.

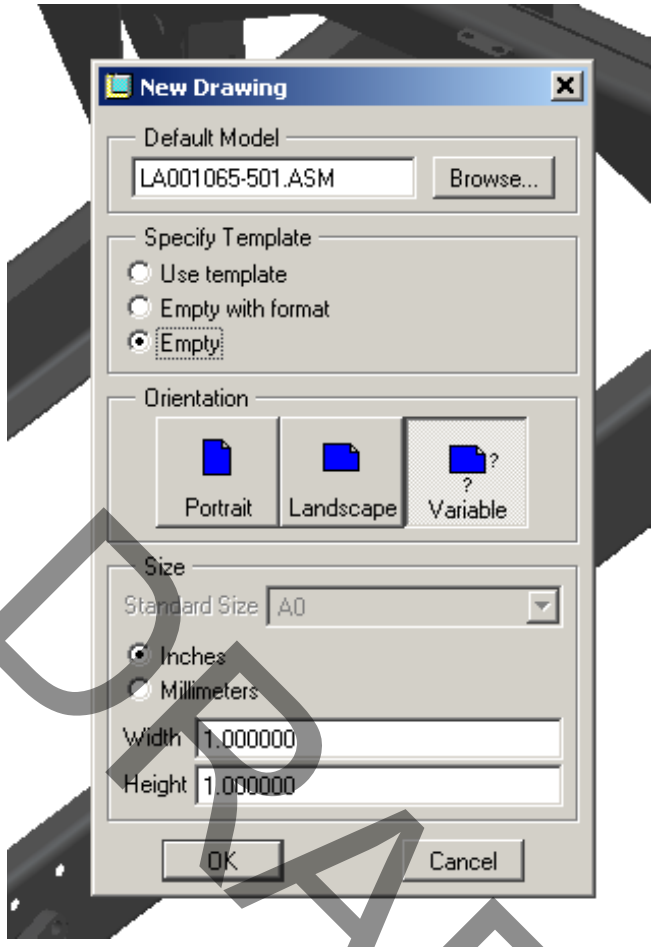


FIGURE 68: CREATE DXF FROM NEW DRAWING

Step 3a – Make the Views

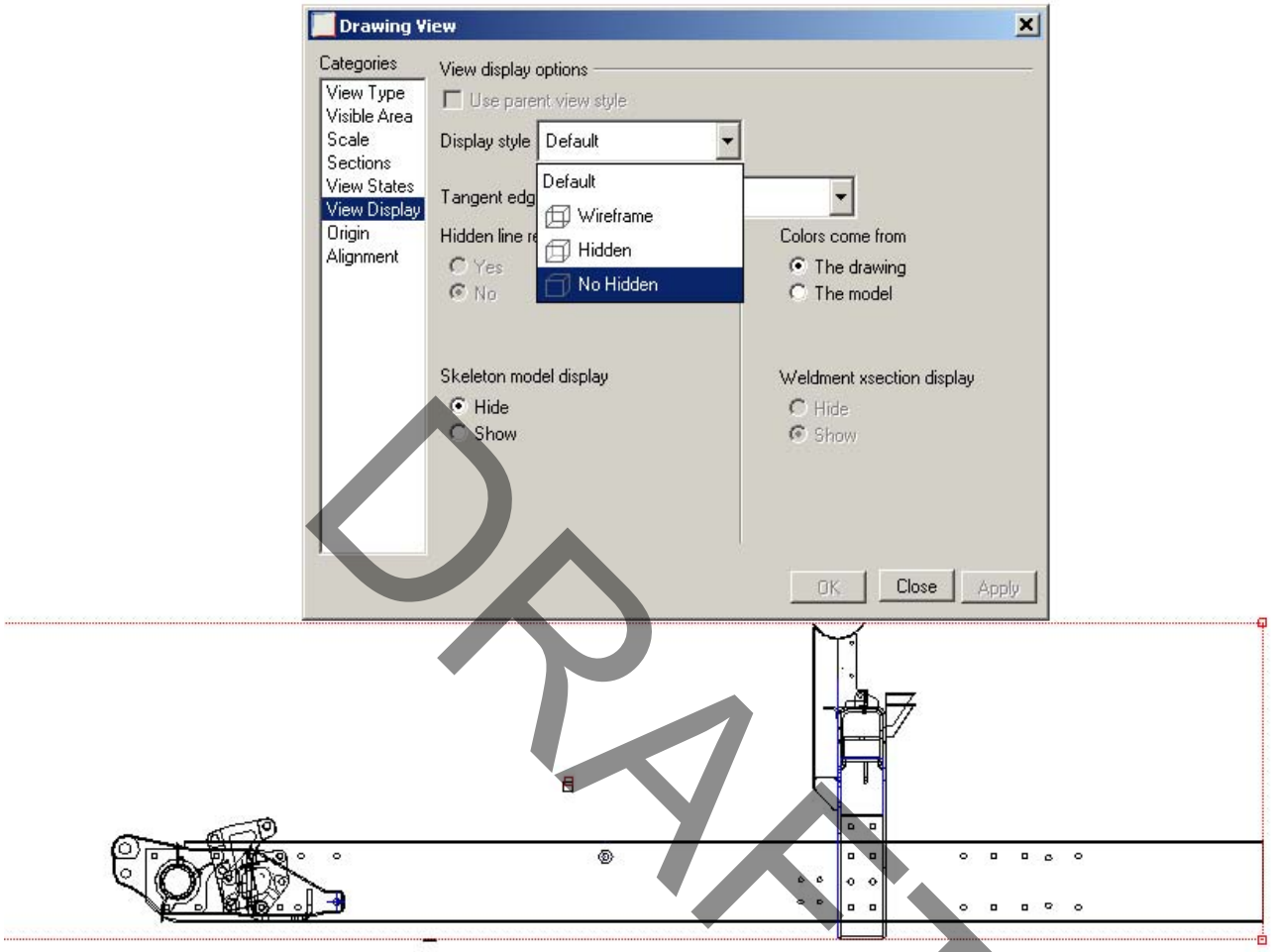


FIGURE 69: CREATE REQUIRED VIEWS FOR INSTALLATION DRAWINGS

Step 3b – Inspection of Lines

- Notice the hidden lines for the steering gear casing.
- This was an imported model and will not clean up well, so either eliminate from the assembly or replace with a true model.

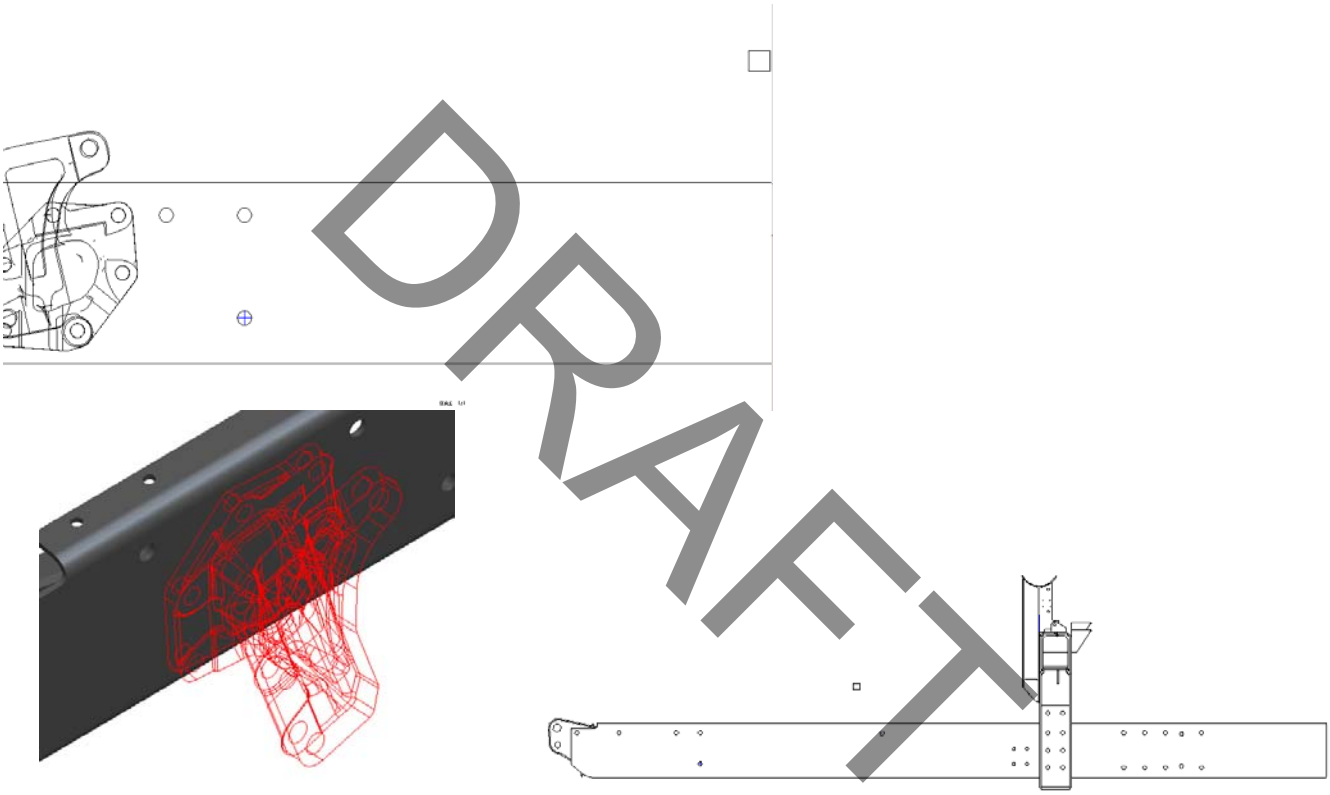


FIGURE 70: INSPECT LINES

Step 3c – Make other Views

- Do not make projected views or auxiliary views; make only one view at a time.
- After each view is created, save as a DXF file, using the -8xx series of names.

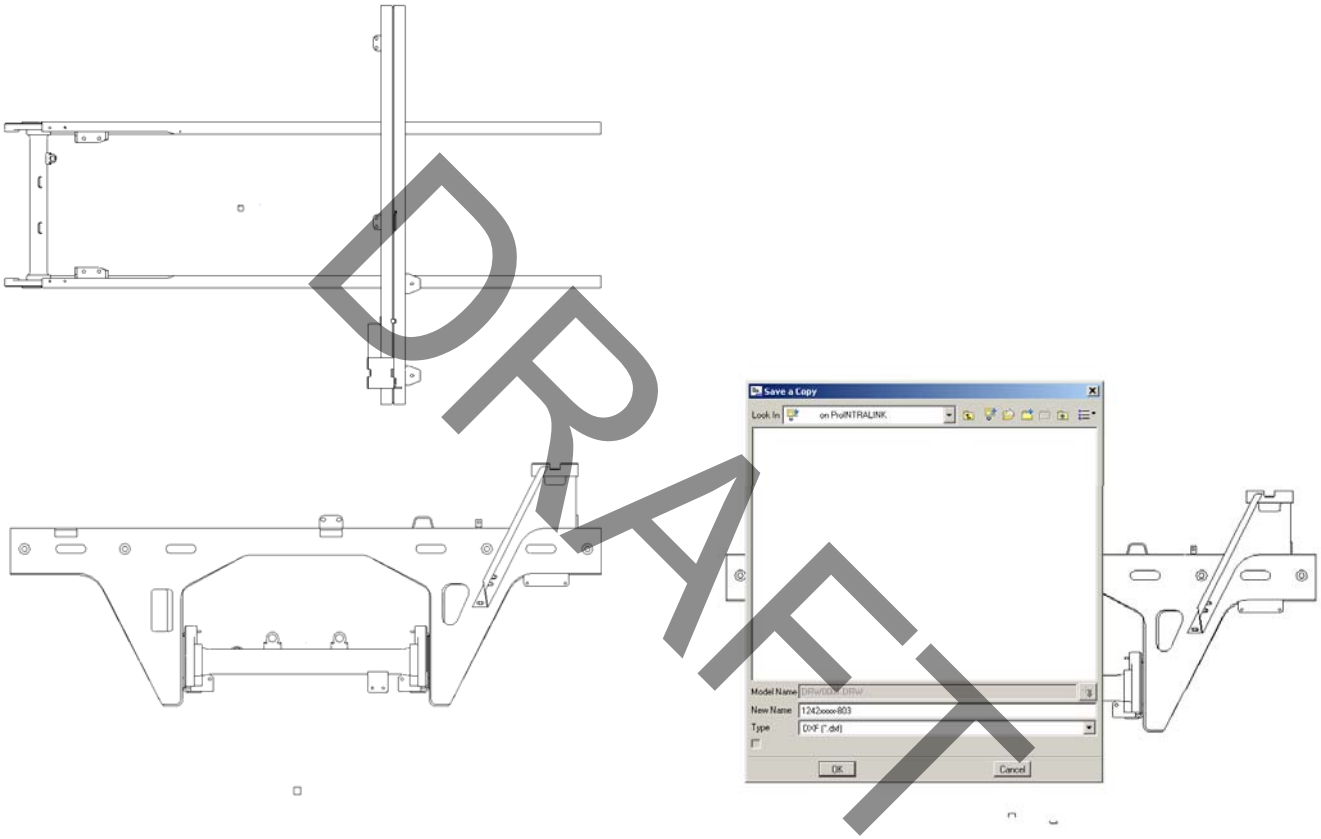


FIGURE 71: MAKE OTHER VIEWS

Step 4a – Make the Backdrop

- Start a new part file with an appropriate name utilizing the -800 series.
- Import the DXF file: **Insert > Shared Data > From File**
- Follow all the defaults:

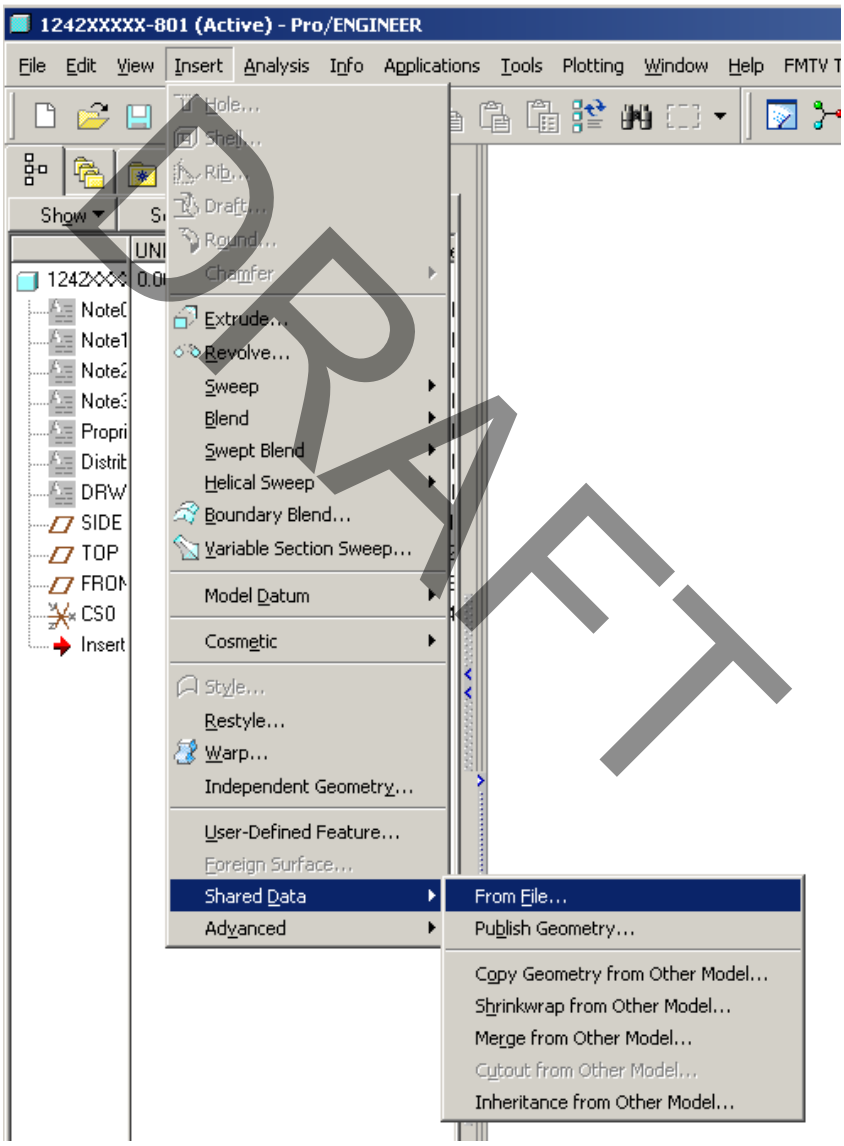


FIGURE 72: CREATE BACKDROP PARTS

Step 4b – Coordinate System

- Create a coordinate system that corresponds to the assembly
- Use the two 2D curves to establish the coordinate
- Rename the coordinate to a common identifier.

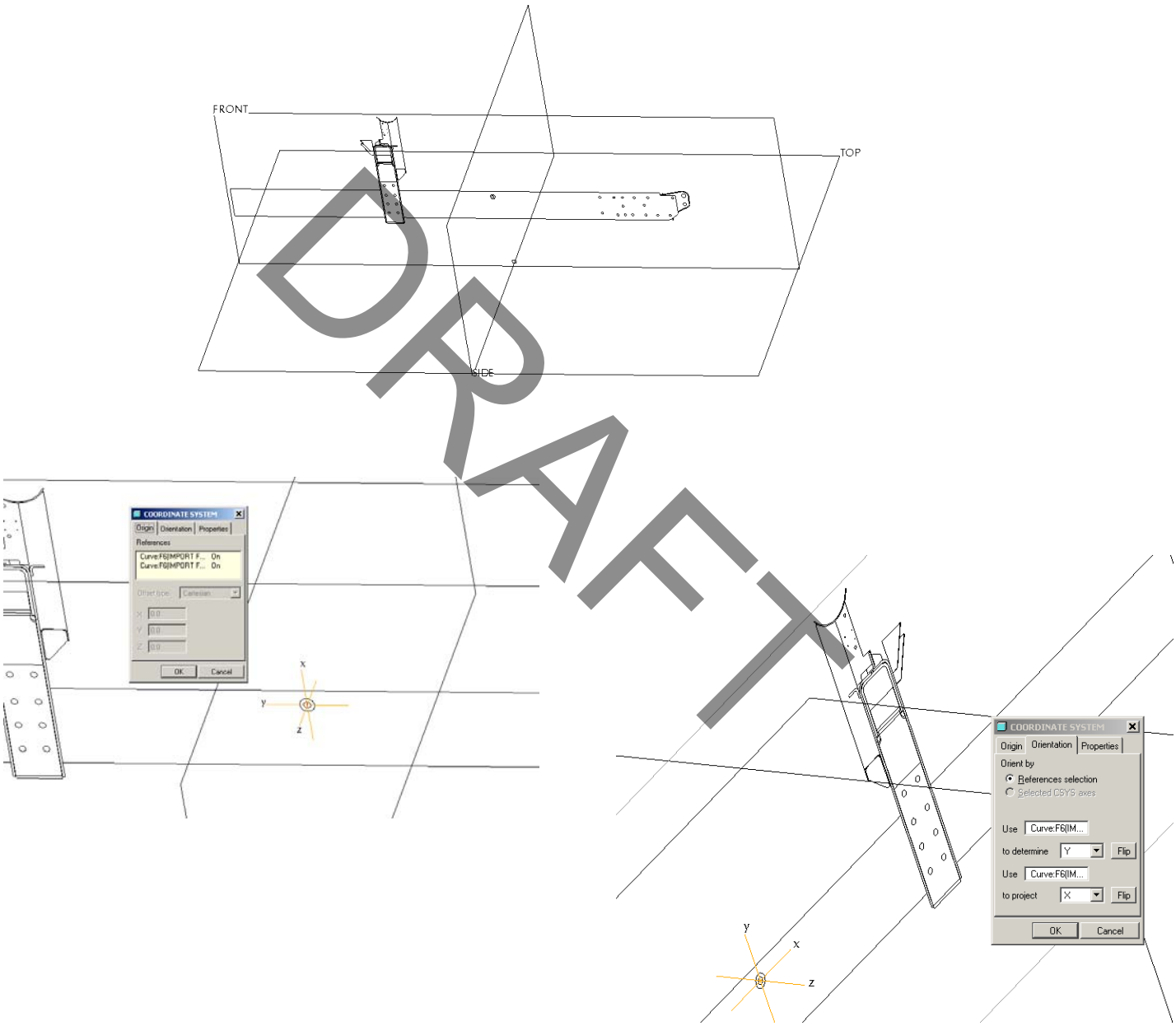


FIGURE 73: CREATE COORDINATE SYSTEM

Step 5a – Cleanup

- Select the geometry and right click
- Then select Edit Definition

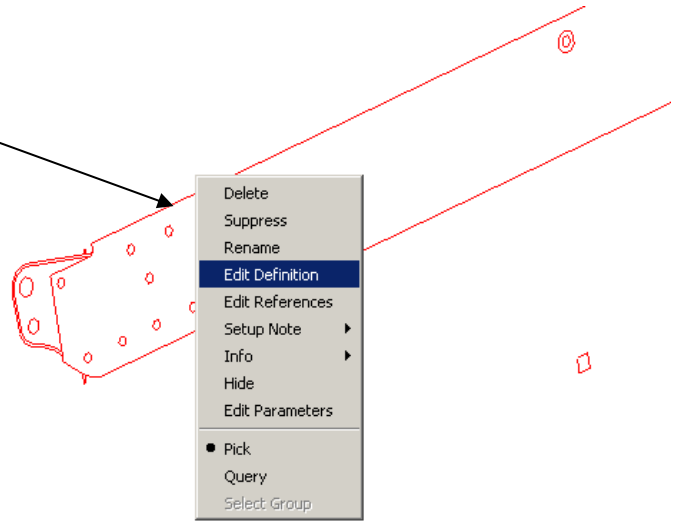


FIGURE 74: CLEANUP - EDIT DEFINITION

- Another menu will appear

- Click on the arrow and select the 4 lines of the format and delete them (the arrow will need to be selected after each line is deleted).

- Clean off any other lines, curves, or text not wanted in the drawing.

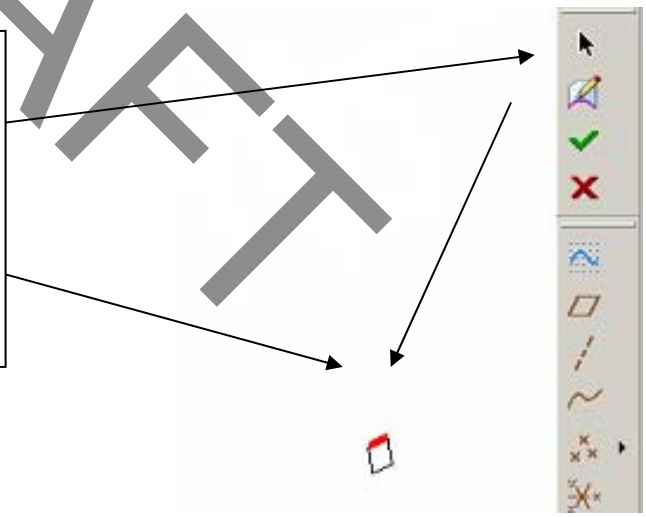


FIGURE 75: DELETE 4 FORMAT LINES

Step 5b – Line Font

- While still in the edit mode, click **Edit** > **Feature Operations** > **Modify Line Style** and then change the font to ***phantomfont*** and the color to **light grey**.
- The 2D lightweight backdrop is now complete.

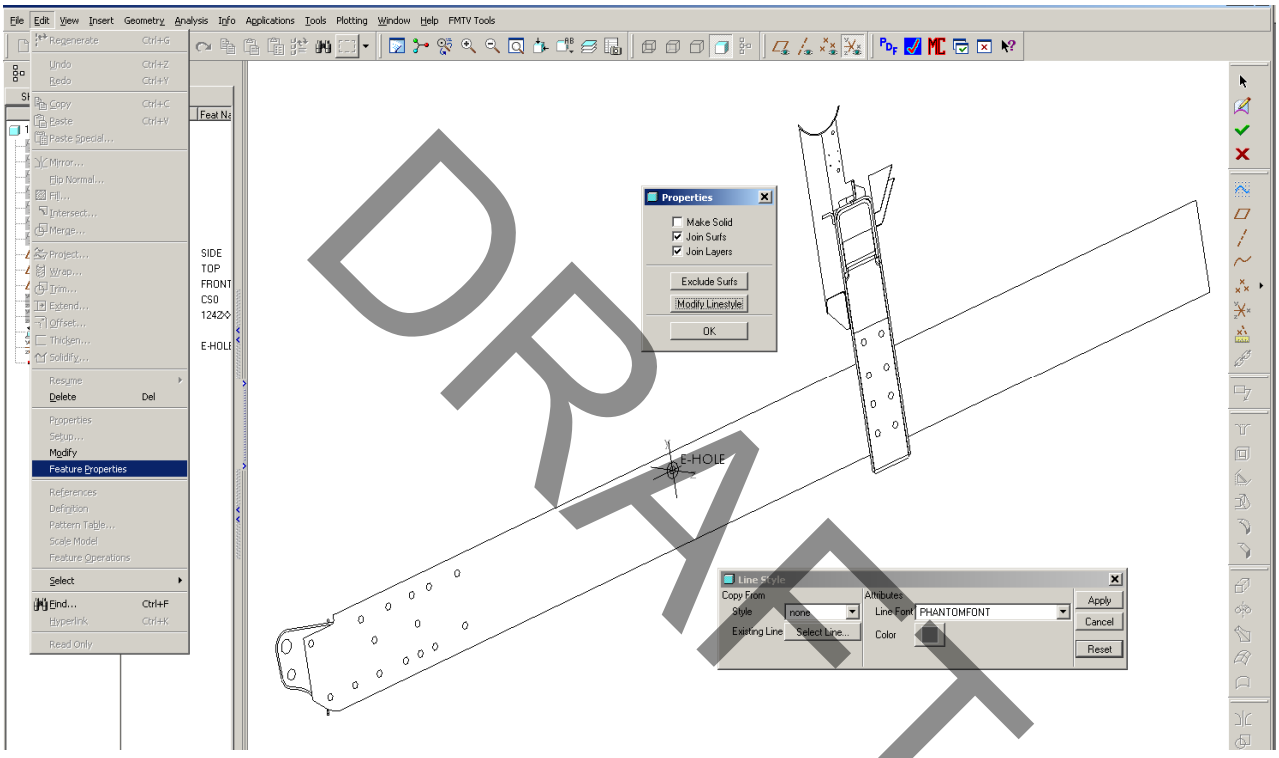


FIGURE 76: CHANGE LINE FONT

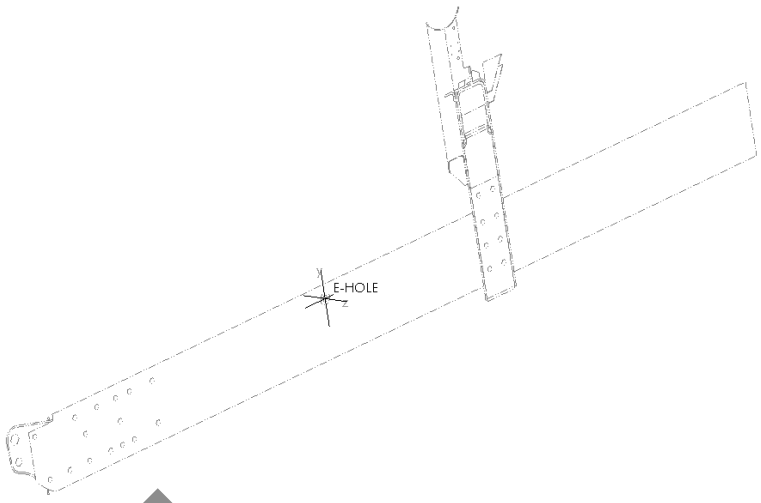


FIGURE 77: COMPLETED LIGHTWEIGHT 2D BACKDROP

Step 6 – Lightweight Assembly

- Assemble all of the views together to form a 2D model.
- Assemble to the common coordinates
- Save the file with the -800 series naming convention
- Skeletons can be added to this same model

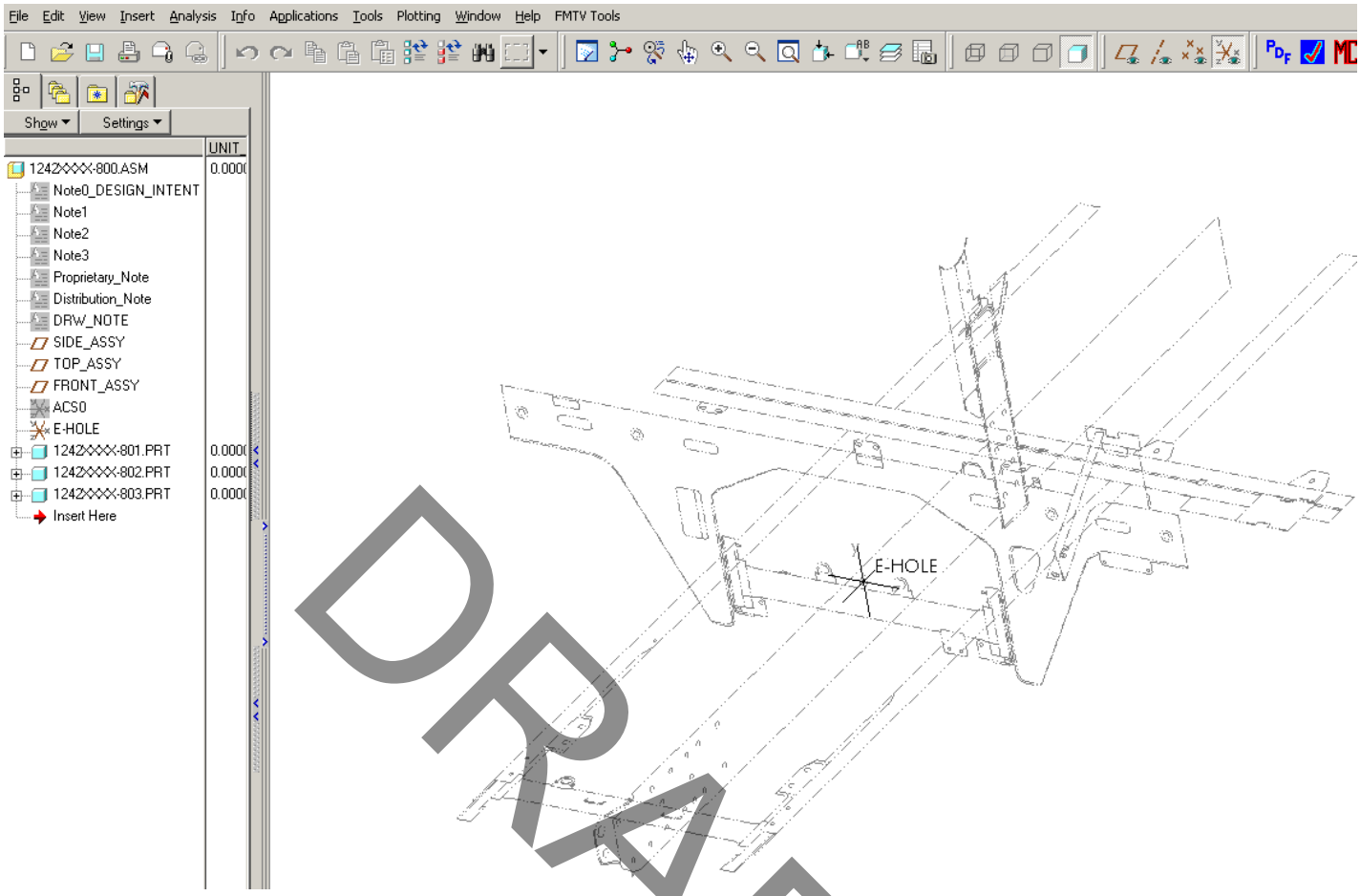


FIGURE 78: COMPLETED LIGHTWEIGHT ASSEMBLY

Step 7 – Installation Model

- Add the lightweight backdrop/skeleton models first
- Assemble the components in the assembly
- This is a clean assembly with no tag-along parts, other than a few models with reference data

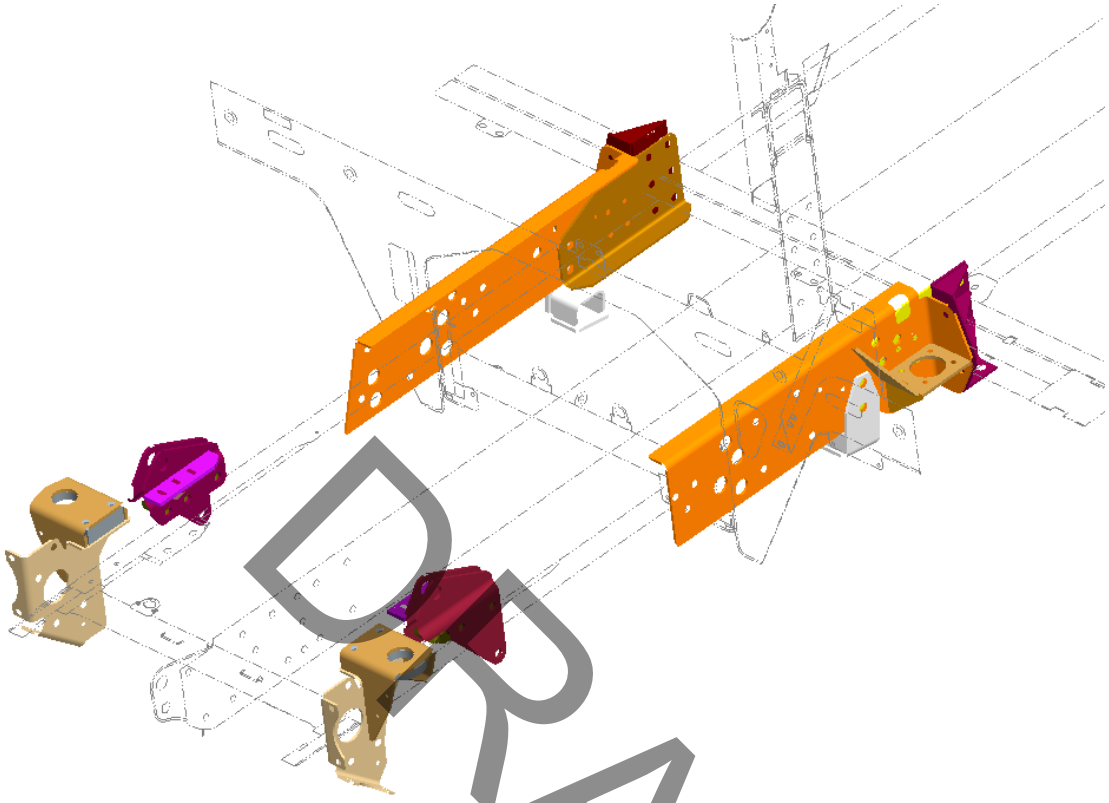


FIGURE 79 COMPLETED INSTALLATION ASSEMBLY

Step 8a – The Final Drawing

- Start the drawing and place views as normal

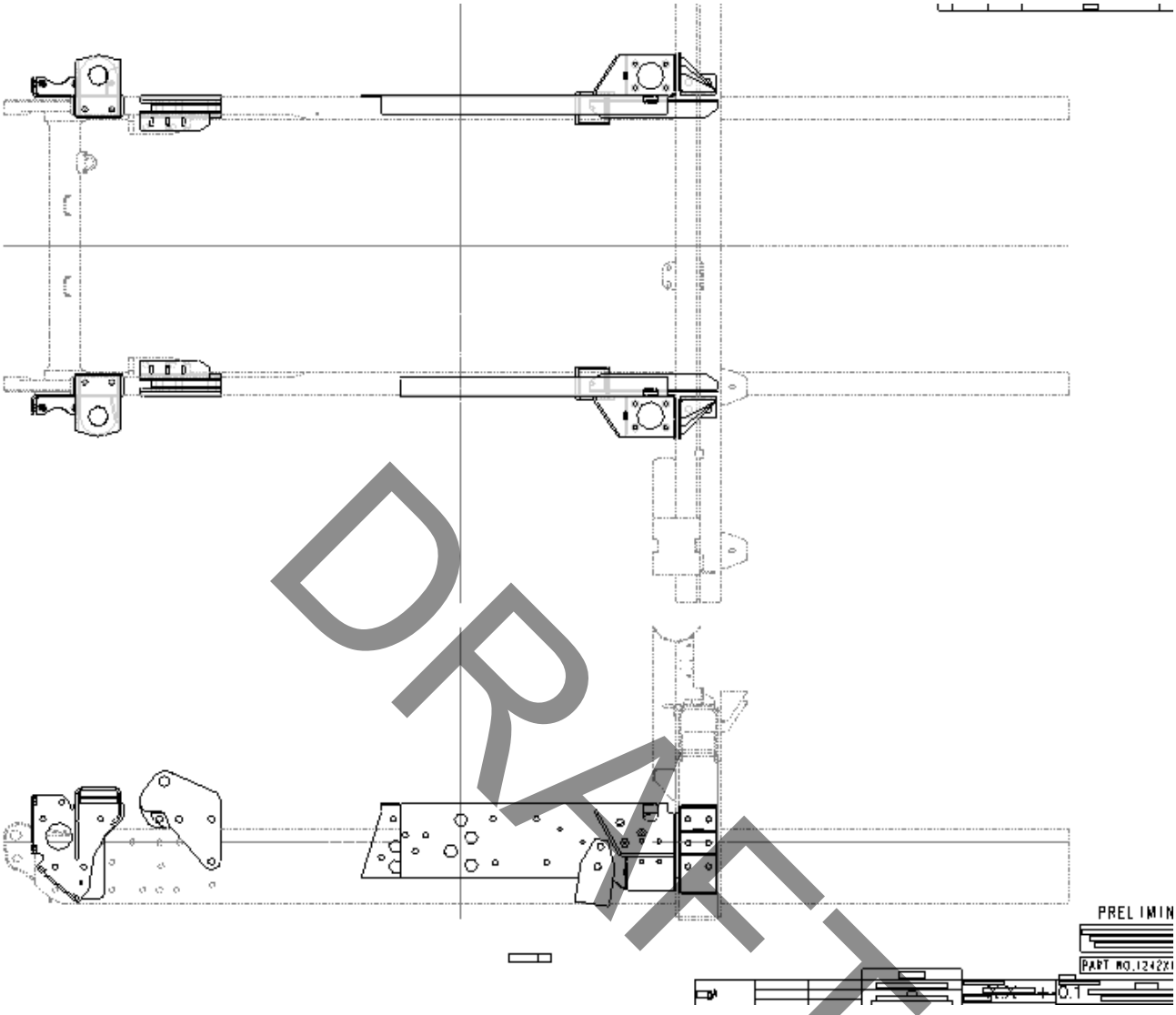


FIGURE 80: BEGIN INSTALLATION DRAWING

Step 8b – Drawing Cleanup

- Set the drawing visibility to **No Hidden**
- Notice the phantom geometry is hidden out

- Notice the lines of the other 2D views

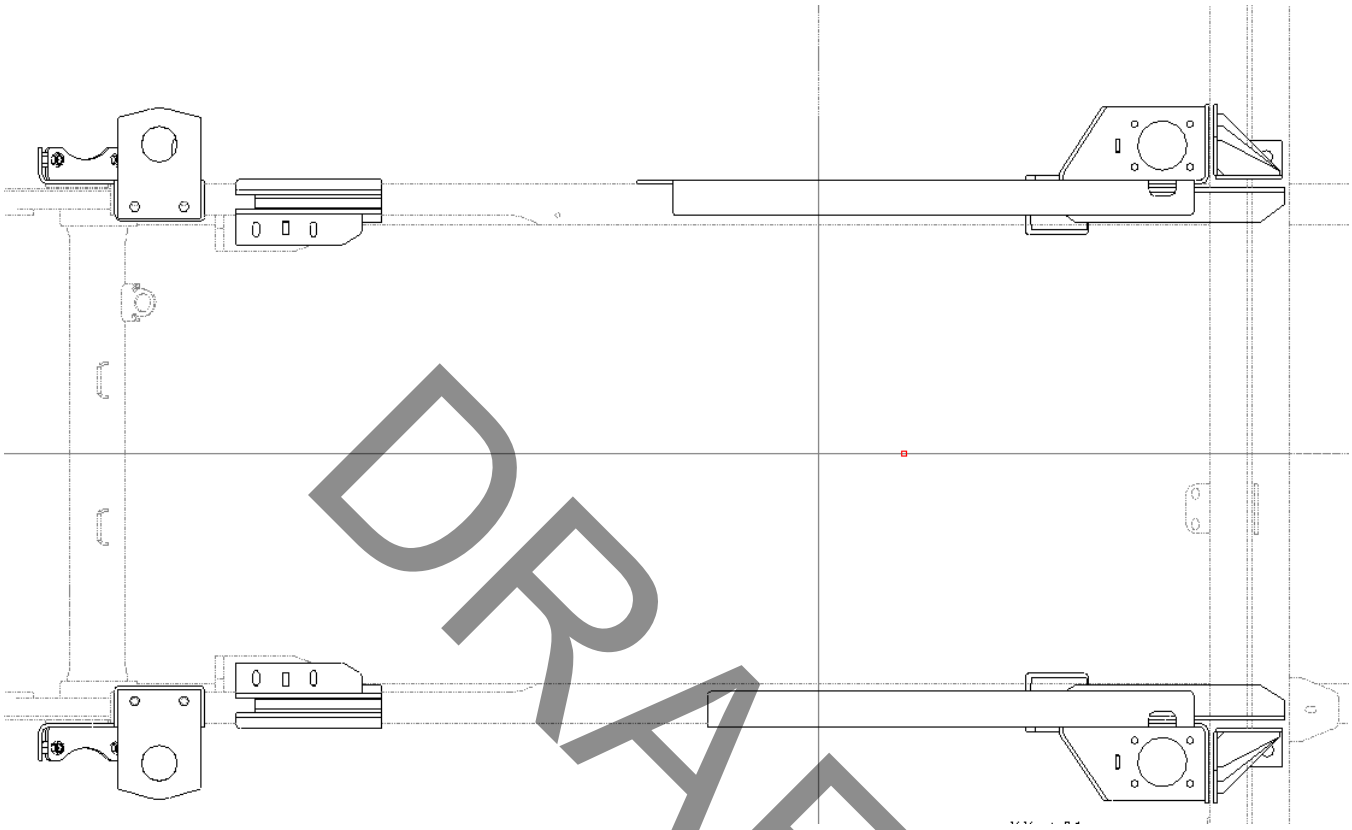


FIGURE 81: INSTALLATION DRAWING CLEANUP

Click View > Drawing Display > Component Display

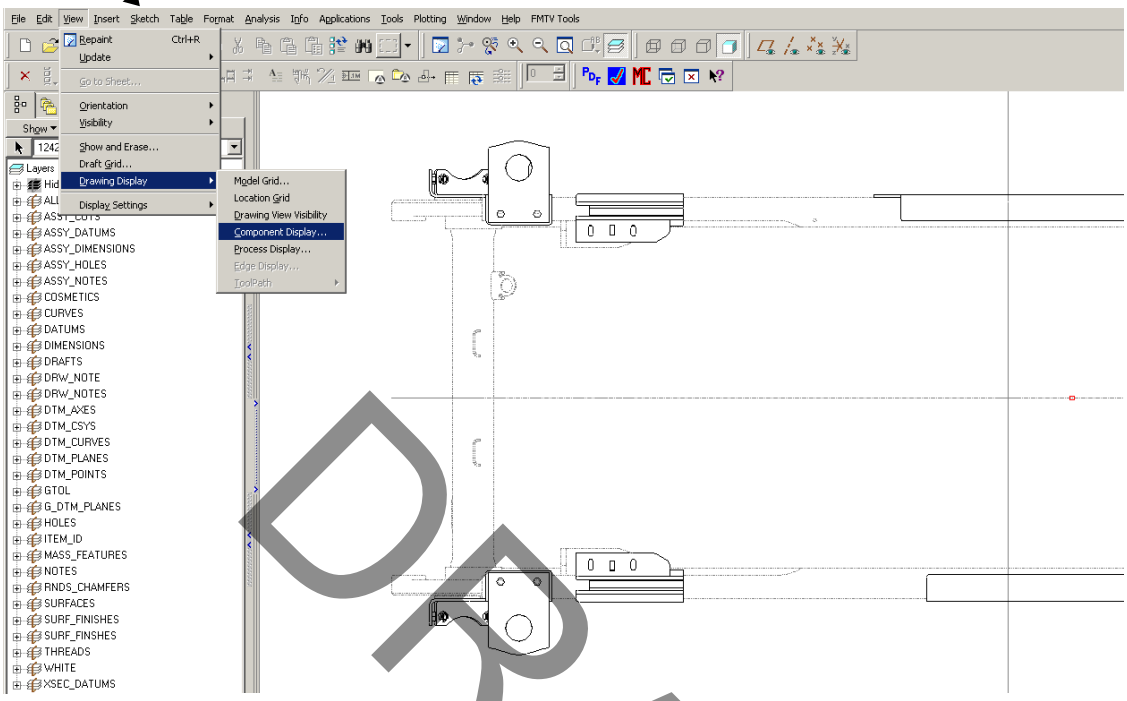


FIGURE 82: INSTALLATION DRAWING CLEANUP - COMPONENT DISPLAY

A new menu appears to the right, select Blank, pick the view, and then select the lines (representing the other backdrop parts) and complete

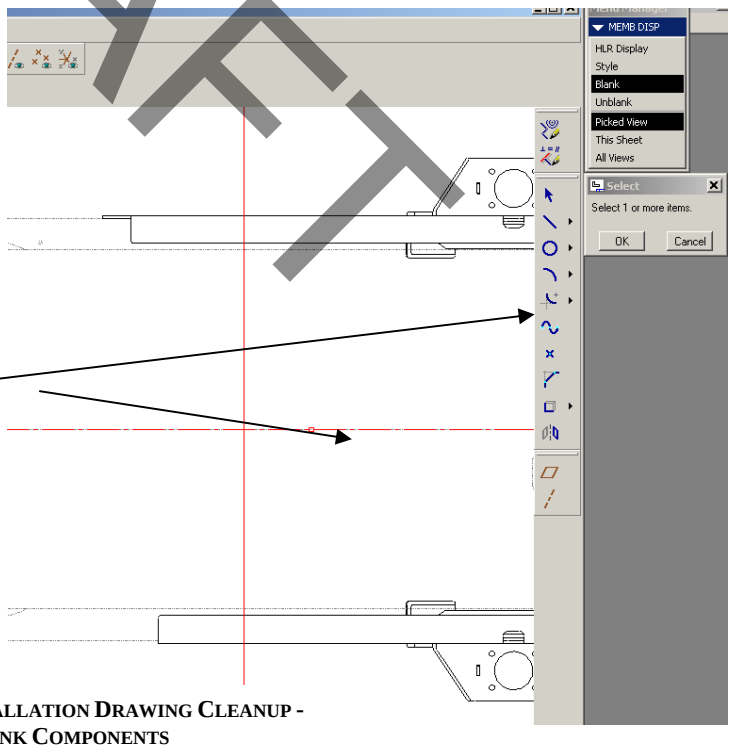


FIGURE 83: INSTALLATION DRAWING CLEANUP - BLANK COMPONENTS

Step 9 – Complete Drawing as Normal

- This works for sections and auxiliary views also
- Write a relation to remove the Lightweight Backdrop / Skeleton assembly from the BOM
- Notice now the SOLID geometry pops out in front of the PHANTOM geometry.

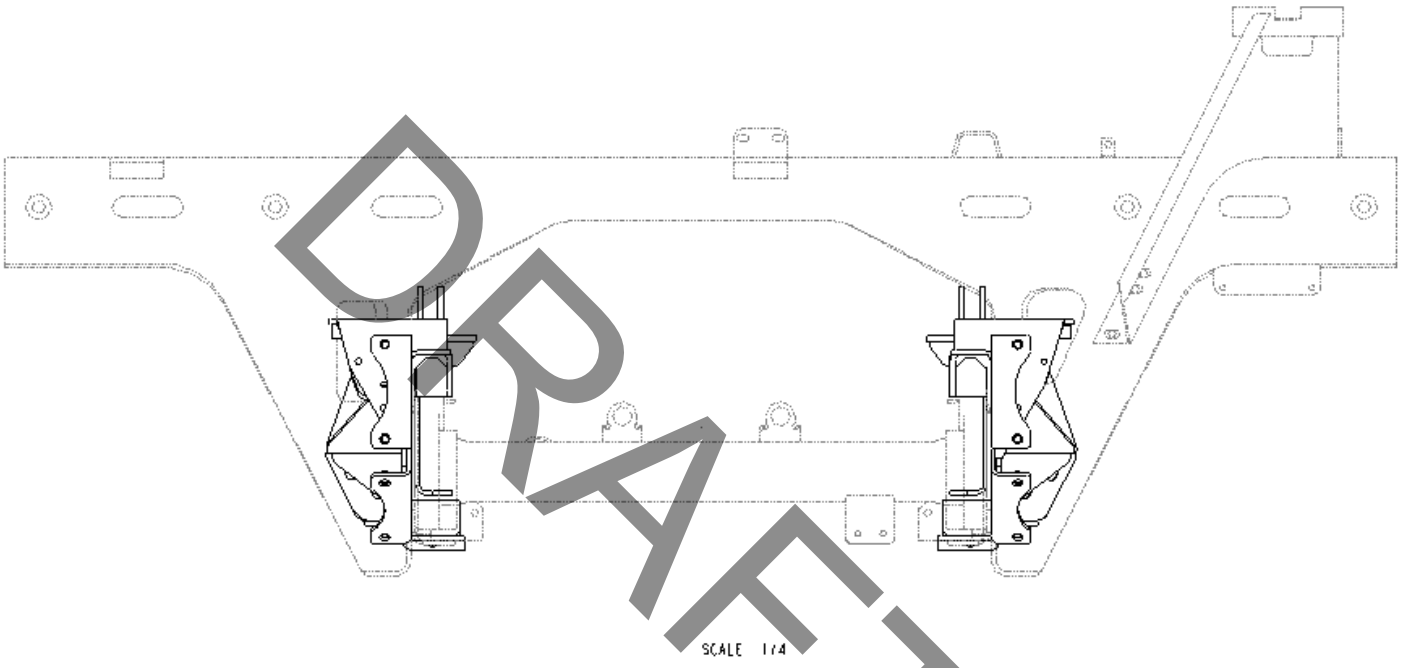


FIGURE 84: COMPLETED INSTALLATION DRAWING

Backdrop Assembly Structure – Reference Information

The first 15 steps in this section describe the methodology for creating the lightweight backdrop parts, assembly, and drawing. The method for assembling installation parts in upper assemblies will be to create two assemblies rather than just one.

- The first will be the base assembly *without* any backdrop components assembled. (Example: 19207_12345678.asm) This Assembly will be used in upper assemblies so backdrops will *not* be visible in these assemblies.

- The second assembly will be used in the installation drawing and will have all of the backdrop components assembled to it (Example: 19207_12345678-800.asm). This Assembly will *only* be used in the installation drawing. This will have all the -8xx backdrop parts assembled, along with the base assembly. The Model Tree would look something like the image below.

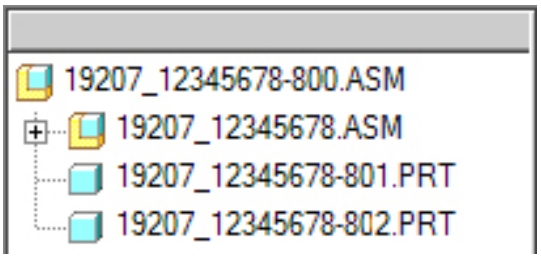


FIGURE 85: BACKDROP ASSEMBLY TREE

- **Alternative Technique for Installation Drawings**: In lieu of creating a -800 assembly, the user can also just create the base installation and use simplified reps to control the -800 series parts. The -800 2D/3D backdrops are still required in this method, but a standalone -800 assembly would not be needed. The technique for this would be to update the DEFAULT_REP in the model, so all the -800 parts are excluded from the DEFAULT_REP. This method shall only be used when the backdrop parts are small in size (< 2mb).

Things to keep in mind while working with Backdrop models

- If the -8xx backdrop parts are too small for the model; the easiest way to get it to the correct size is to modify the units. First, change from Metric to English while doing an INTERPRET dimensions. Then, change back from English to Metric using a CONVERT dimensions. This should bring the backdrop part to normal size.
- All 2D backdrop parts shall be LIGHT GREY in line color and PHANTOM in line style.

Using Parts Table in Installation Drawing

If a parametric parts table needs to be shown on the installation drawing, the recursive attribute option will have to be used on the table to show the correct parts. Select the following commands:

Table > Repeat Region > Flat/Rec Item and then picking the assembly that needs to be shown (the base assembly). The -800.asm and the -80x (backdrop parts) will still need to be removed from the parts table by either manually filtering them out or using a relation to filter them out.

D.4 3D Lightweight Backdrop Creation

This process is for creating 3D lightweight backdrops for use in Installation and Assembly drawings as phantom reference geometry.

Advantages

- Dramatically reduce the file retrieval and regeneration time
- Geometry is clean (other methods require a substantial amount of clean-up)
- No tag-along parts (additional parts not used in the installation shall not be part of the assembly)
- Skeletons can be embedded into the backdrops
- Clean BOM
- Useful during the design process as well.

Step 1 – Identification

The first step in this process is to determine the necessary geometry required to create the views on the installation drawing. This will also require the user to have the necessary assemblies in their Workspace for reference.

Step 2 – Create independent shrinkwrap model

The first step will be to create a new Creo Parametric part. This part will be given a -801 or higher number (i.e. 19207_basenumber-801.prt).

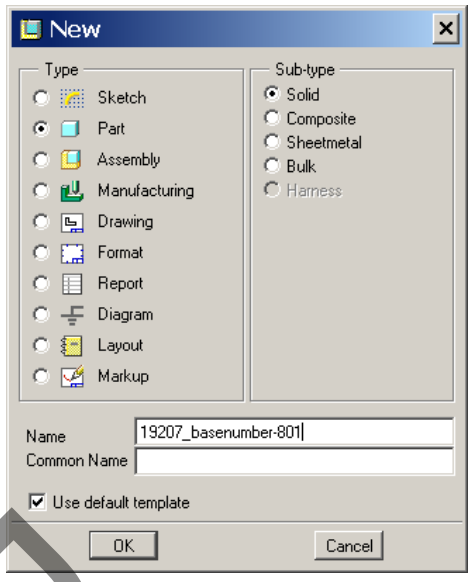


FIGURE 86: CREATE 3D BACKDROP PART

Step 3 – Create Shared Data Feature

Select Insert > Shared Data > Shrinkwrap from the Creo Parametric main menu as shown below.

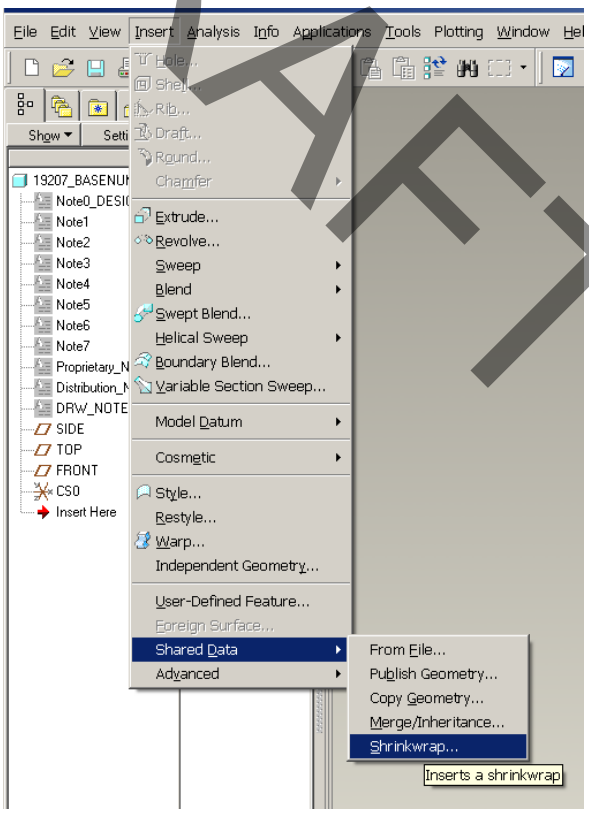


FIGURE 87: CREATE SHRINKWRAP FEATURE

Step 4 – Select Reference Model

From the Creo Parametric Dashboard open a model which geometry will be copied

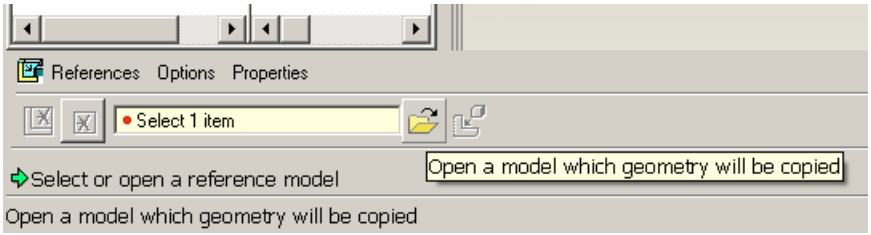


FIGURE 88: SELECT SHRINKWRAP REFERENCE MODEL

Select the model (part, assembly, or sub-assembly) to create the shrinkwrap from then select open.

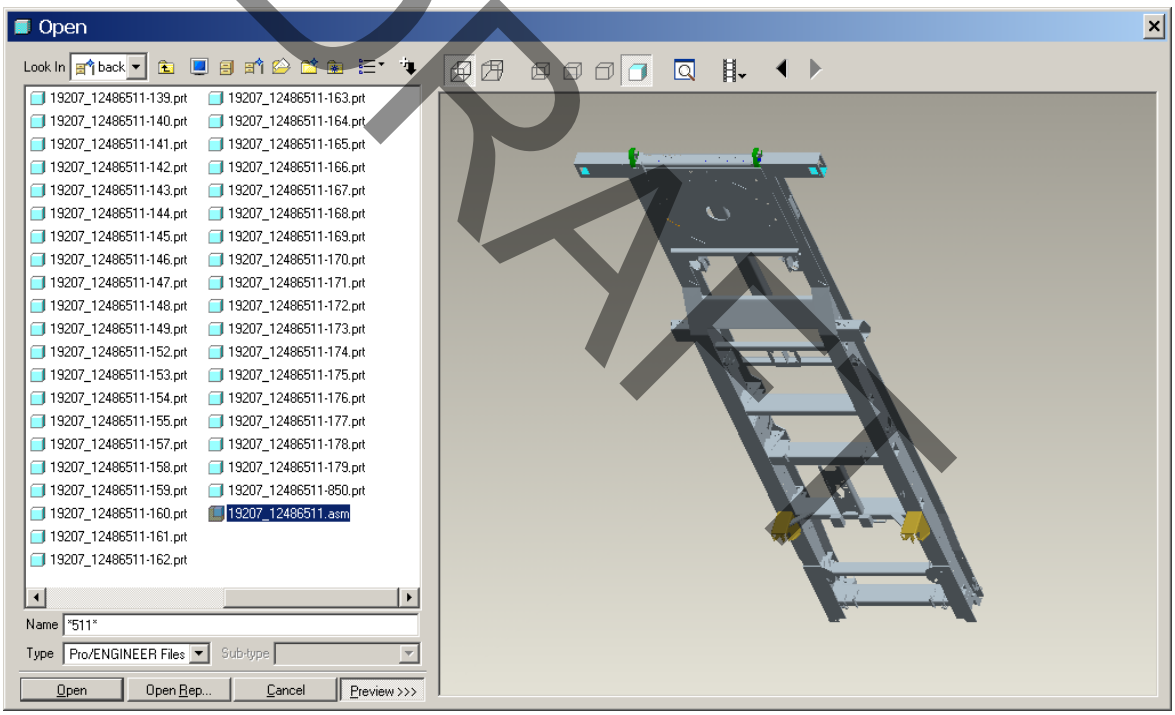


FIGURE 89: CONFIRM REFERENCE MODEL SELECTION

Step 5 - Locate the model

Define the placement by using the ACSO or CSO of the selected model and the CSO of the new part (-801, etc). Select the OK button.

Step 6 – Choose proper Pro/ASSEMBLY options

Select the Options tab from the Creo Parametric Dashboard

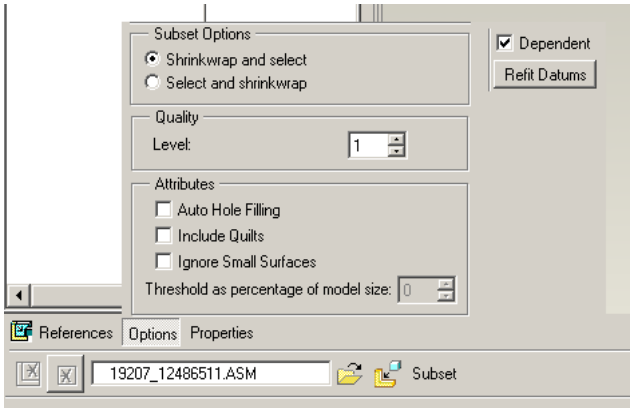


FIGURE 90: CHOOSE APPROPRIATE OPTIONS

Step 7 – Choose Quality Level

Set the quality level (Note: shrinkwraps may require this level to be adjusted until the model's appearance is acceptable. A level of one creates a smaller sized file and the higher the quality level setting the larger the model size). Most shrinkwraps will fall between a quality level range of 3 and 7.

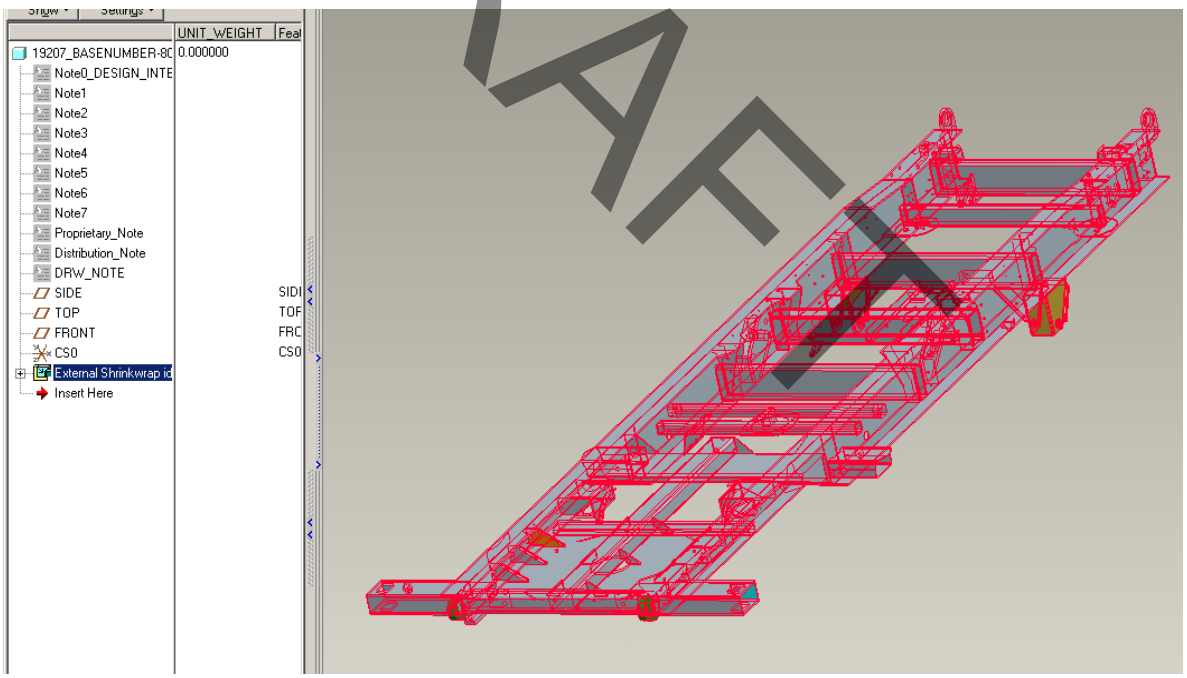


FIGURE 91: VERIFY REQUIRED GEOMETRY IS PRESENT

Step 8 – Toggle shrinkwrap to Independent

Make the Shrinkwrap Independent by performing an edit definition on the external shrinkwrap feature and removing the check from the dependent option.

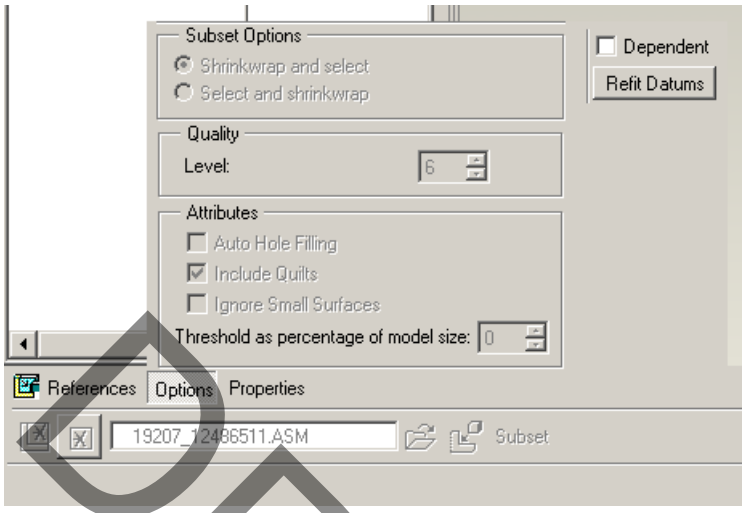


FIGURE 92: TOGGLE OFF "DEPENDANT" OPTION

The result will be a “stand alone” 3D model that is ready to insert into the installation and referenced by the installation components.

Step 9 – Change the color and appearance of backdrop model(s)

1. Assign colors and appearances to objects

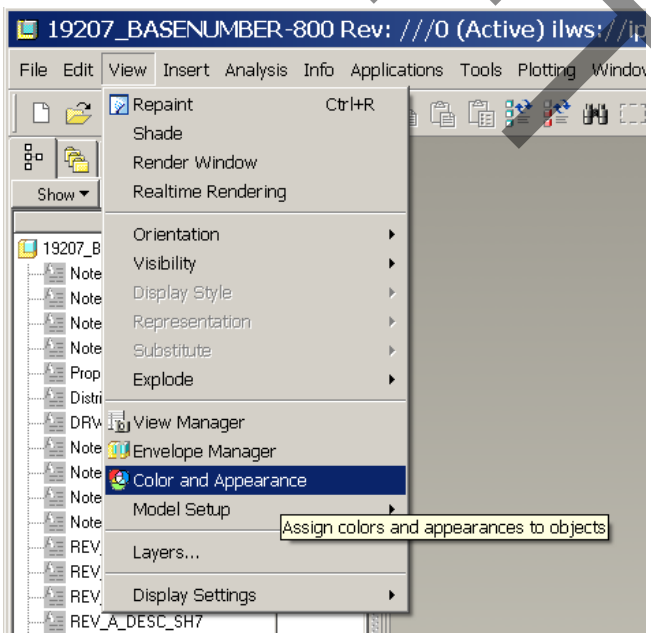


FIGURE 93: ACCESS APPEARANCE EDITOR

2. Select the color grey

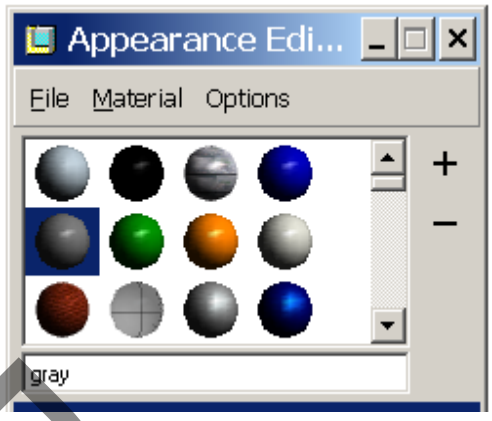


FIGURE 94: SELECT GRAY COLOR

3. Assign the color to the backdrop models > **OK** button > **Apply** button

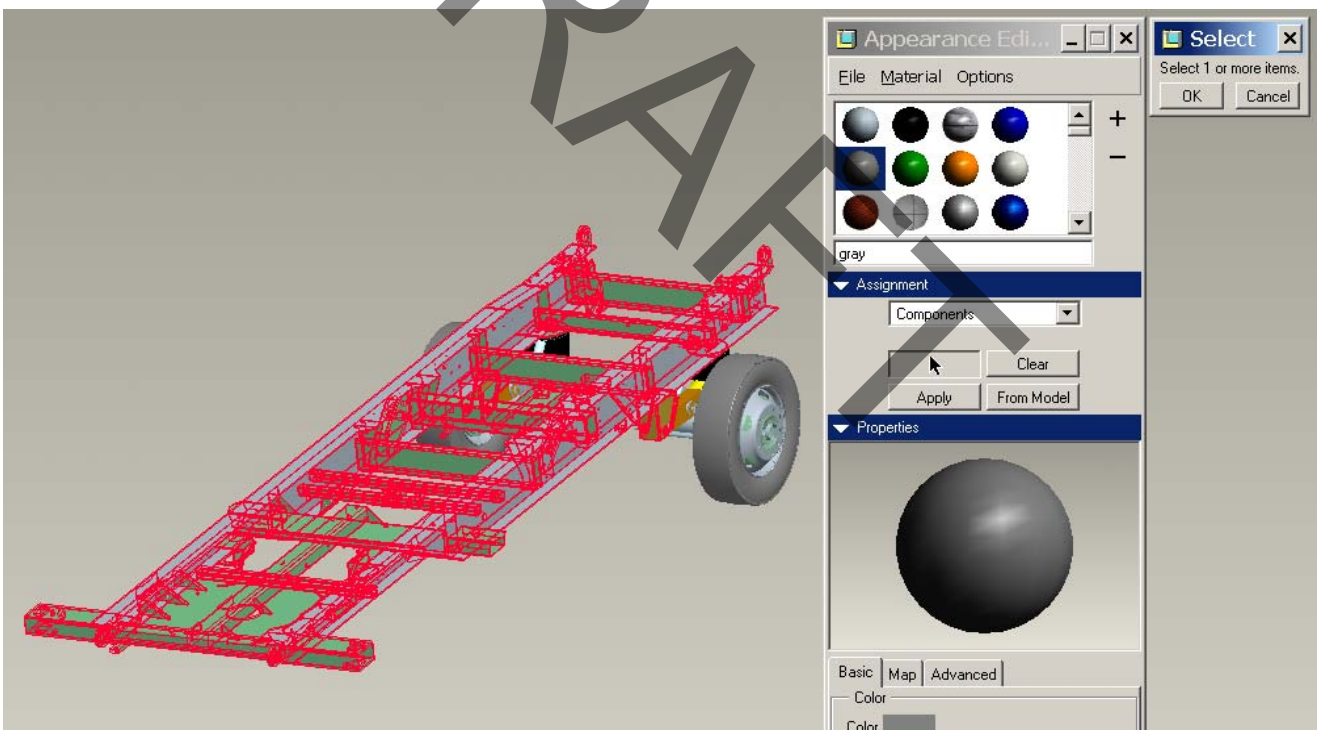


FIGURE 95: SELECT GEOMETRY TO APPLY COLOR

4. Set the following properties: Matte=0.00, Transparency = 80.00 and select the **Close** button

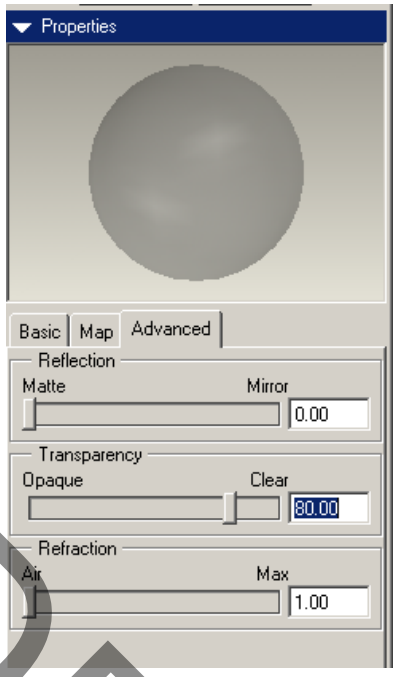


FIGURE 96: SET TRANSPARENCY

The model should now look like the following image:

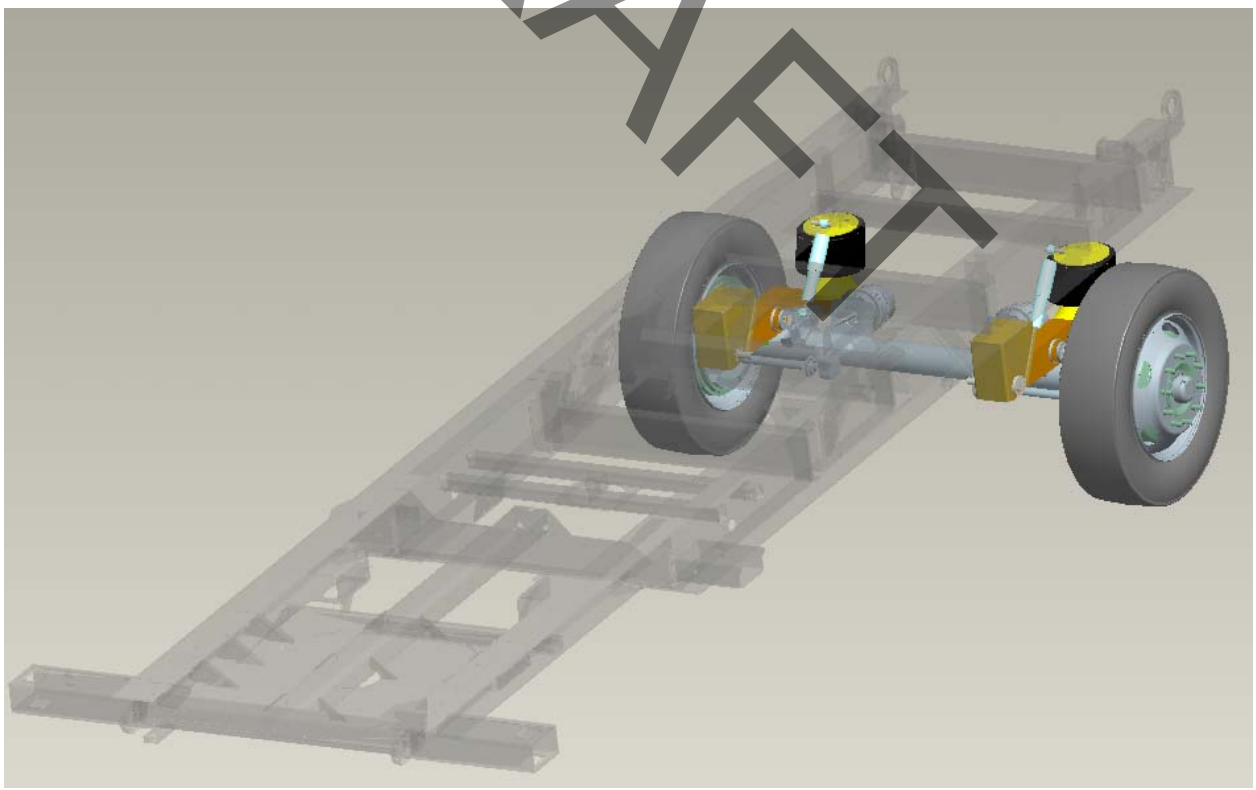


FIGURE 97: RESULTING 3D BACKDROP

The “backdrop” models are to be transparent instead of the 2D geometry representations. This is a 3D backdrop not a 2D backdrop method. The structure in the model tree remains the same as with the 2D backdrop method, as shown below.

The note named **Note0_DESIGN_INTENT** should be used to document the name of the assembly used to create the shrinkwrap model. In the future when changes occur this information will assist in updating the shrinkwrap model. The user can check-out the reference assembly, toggle the shrinkwrap as *DEPENDANT*, update the shrinkwrap, and toggle back to *INDEPENDENT*.

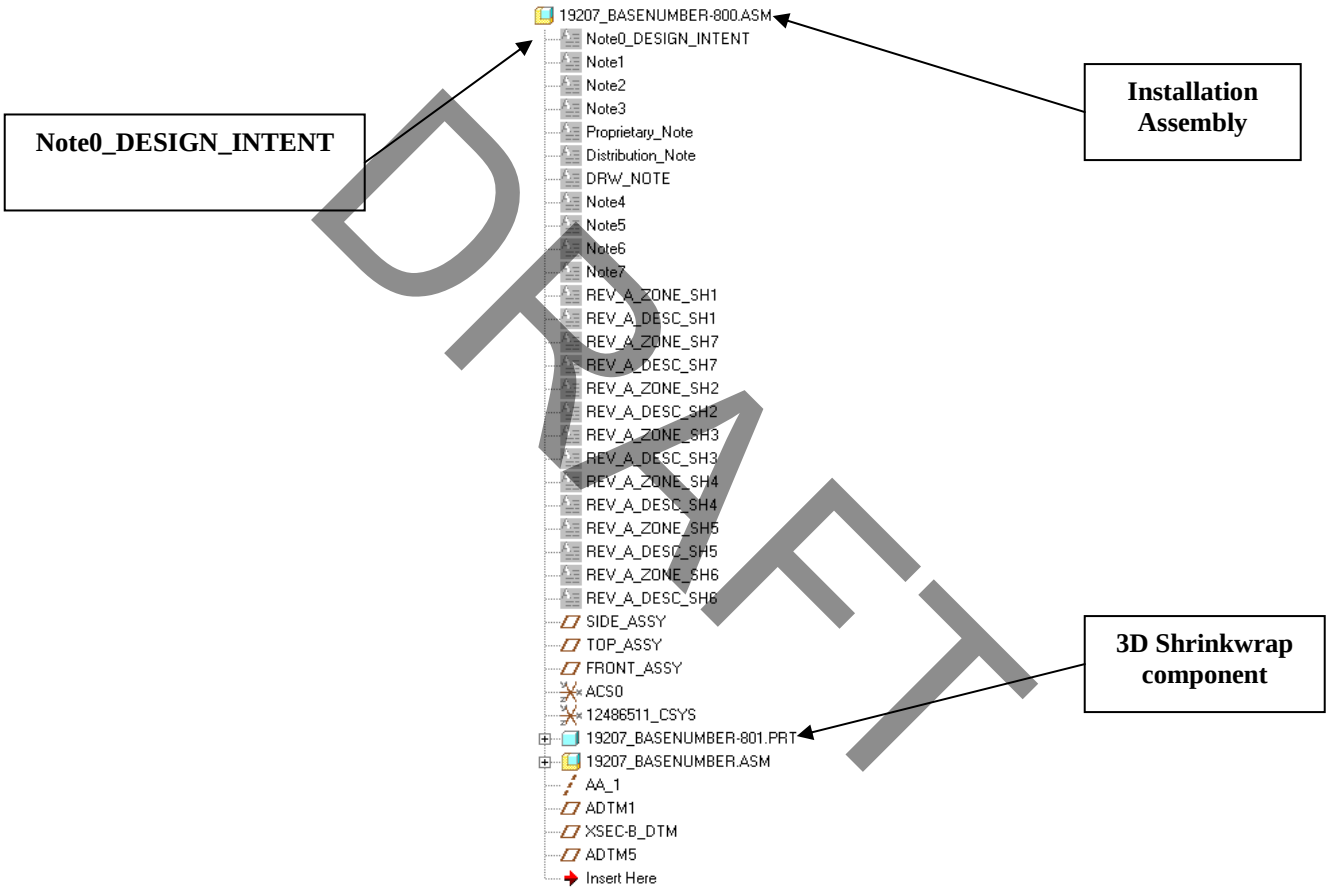


FIGURE 98: FINAL MODEL TREE

The assembly of solid components must be separate from the models used to make the backdrop. In this case 19207_BASENUMBER.ASM shall be used in the next level assembly without bringing the back drop models along. Also, the backdrop models must me completely independent from the models they are created from.

Step 10 – Drawing Settings for 3D backdrop parts

The line type for 3D backdrops shall be PHANTOM. The color of the lines on the 2D drawing will relate to the system settings for shrinkwraps.

Note: These steps were defined with Pro/ENGINEER Wildfire 2.0 and there may be slight menu differences in subsequent versions of Pro/ENGINEER. The concept should remain and slight adjustments should be manageable by users familiar with Pro/ENGINEER.

3D-Backdrop Creation Checklist / Reference Guide

1) Checkout the required components for the 3D Backdrop into a clean workspace.
2) Create a temporary assembly to assemble the required backdrop components. An existing assembly could be used but to avoid accidental check-in of the assembly, it is recommended to create a temporary assembly. The assembly should be built to the VCS (Vehicle Coordinate System) for Chassis installations and CBCS (Cab Body Coordinate System) for Cab installations.
3) After the model is complete, export as a solid IGES file using either the VCS or CBCS as the assembly reference point. Be sure to make the file SOLID and FLAT in the selection menu.
4) Open a new file for importing the IGES file using the desired name for the 3D Backdrop. Create a VCS coordinate system as defined in the VCS portion of the Standard.
5) Import the solid IGES model using the default coordinate system – which is the VCS embedded in the IGES file and the VCS created in the new part.
6) Save the file before continuing
7) Run the FXLA mapkey to fix the layers and remove the IGES imported layers.
8) Assign the material property to NONE_V1 and set the Mass Properties density to 0.0001. Write the relation "unit_weight=pro_mp_mass" under the CSO. These settings will make sure the 3D backdrop does not generate a mass that will drive the weight parameter on the drawing.
9) Edit the component parameters list.
10) Apply the 3D backdrop display settings IAW the CAD Modeling Standard
11) Run the Model Check and correct any errors
12) Start a dummy drawing and place the 3D backdrop into the drawing in several different views (3D, front, side..) and verify that the viewing requirements are met.
13) Do not check in the dummy assembly or the dummy drawing.
14) Save the 3D Backdrop file and check into Windchill PDMLink.

Suggestions for a clean 3D backdrop model:

- Do not include hardware (bolts, nuts, washers ...) in the model because these will make the export model very large.
- Components not seen in any of the views on the drawing should not be shown. The idea is to make a simple backdrop for orientation of the components installed on the vehicle. The more "stuff" in the backdrop the larger the file will be.

- Remember that the lines will be in phantom so the lines will be broken up. Very small parts will not be entirely discernible so eliminate them from the backdrop unless absolutely required.
- Pay attention to the SCALE that the drawing will be presented - that will help determine what needs to be in the backdrop model.

Possible issues:

- If there are purple line elements in the drawing then there was an unresolved surface in the IGES file. One way to fix this is to make a STEP file of the component before assembling it into the dummy file. Make the IGES file again and verify that the purple entities are white.
- If the file is too large or takes too long to load, reduce the number of components and remove hardware.

D.5 Checklists

The following four tables list items and directions in order for checkers to perform their tasks. The four checklists include CM Checklist, Standard Model Checklist, 2D PDF Drawing Checklist, and Vendor / Legacy / Standard Model Checklist. These four checklists were originally separate documents but were recently combined and added to this document for reference purposes.

CM CHECKLIST			
Item	**TAC**	**CTR**	**Comments**
Note contents of Change Notice.		X	
Open Creo Parametric drawing.		X	
Print PDF released drawing in PDMLink and current rev of the Creo Parametric drawing.		X	
Paper-to-paper comparison of PDMLink drawing previous rev to current Creo Parametric rev. Document any differences on NOR.		X	Rejection item: Undocumented differences between approved CMSTAT drawing and new Creo Parametric drawing.
Drawing Check			
Open Creo Parametric drawing & perform Model Check.		X	Rejection item: Model Check failed.
Check rev. block for correct history, rev. level & ERR numbers.			Rejection item: Rev. block has incorrect information / Rev. block history is incomplete / Rev. level is incorrect / ERR number is incorrect.
Check title block.		X	Rejection item: Title block has incorrect information / Text is outside of format boundary.
Check rev. levels on all documents		X	
Check that drawing displays properly.		X	No layering standard for drawings. Make model dependent on model layers instead of drawing layers. Do not reject due to problem with archived part/assy since nothing can be done.
General feature check.		X	Check that holes are features, not sketched lines, etc.
2nd Check			
Check that rejected items are fixed.		X	
If 5 items were rejected previously, recheck completely.		X	
Double check that rev. block is correct.		X	Rev. history ERR numbers may have updated incorrectly.

TABLE 14: CM CHECKLIST

STANDARD MODEL CHECKLIST			
Item	**TAC**	**CTR**	**Comments**
Note contents of Change Notice / Promote.		X	
Check Change Notice / Promote to determine if NOR is needed. If there is no NOR, search PDMLink for previous rev. (NOR is required if previous rev.)		X	NOR is not required for initial release or the first time an object is placed in PDMLink
Check that only one drawing is attached to RTP.		X	Reject if more than one drawing is attached to an

			RTP.
Run Where Used report on each part/assembly being promoted. Check that next up assemblies at release level Preliminary Release or higher regenerate before promoting part or assembly to Preliminary Release.		X	Rejection item: Components do not assemble correctly in assembly.
Promoting Assemblies: All subordinate items must be at the same or higher release level than the assembly being promoted.		X	
Create a workspace and check out RTP contents.		X	
If no drawing is present, check for PDF drawing.		X	Make sure drawing or PDF drawing is attached to package.
Print the NOR.		X	If needed.
Print the PDF Drawing.		X	If needed.
		X	If Needed
Print out released Drawing & current rev. of Creo Parametric Drawing.		X	
Check that all parts/assemblies/drawings are named correctly	X	X	(Stds. Para. 3.8/4.5/5.15) Will be confirmed by ModelCHECK only.
Paper-to-paper comparison of the released drawing previous rev. & current Creo Parametric rev.		X	Any differences must be documented on NOR.
Check parameters in PDMLink		X	See Attachment A. Reject if any parameters have incorrect/missing values.
Drawing Check			
Open Creo Parametric Drawing: Perform Model Check at the beginning and end of the check.		X	Verify no errors or valid warnings exist. (Stds. Para. 2.10.1)
Check rev. block.		X	ERR number should be correct / Rev. history and level should be correct / Format should be correct / Line format color should be correct / Text should be correct size and color.
Check that correct distribution statement is located at the bottom of every drawing sheet		X	Dist. Statements A, B, C, D, and X can be used.
Check that correct EAR or ITAR statement is located in the bottom left hand corner of each sheet.		X	Only Dist. Statements B, C, D, and X require an EAR or ITAR statement.
Revision block has been updated as appropriate	X		Stds. Para. 5.11
Check title block.		X	Check that text does not exceed format boundary / Verify title block information.
Check that entities are related to appropriate view.	X	X	Exceptions, notes, BOM Table, symbols etc. should be related to correct views or host entities. (Stds. Para. 5.7)
Verify correct line weights/colors.	X	X	Stds. Para. 5.8
Check that drawing displays properly.		X	No layering standard for drawings. Make model dependent on model layers instead of drawing layers. Do not reject due to problem with archived part/assy since nothing can be done.
Switch Dimensions to check that dims, notes are driven from the model, thus verifying that they are parametric. (Exception: Preliminary)		X	Notes should be driven from model. Most or all dimensions should be driven from model (not created). Stds. Para. 5.1 This will not be checked. However, overridden dimensions (@o) can be rejection items on a case by case basis.
Parameters and notes originate from the drawing model.	X		Stds. Para. 4.3 & 5.13

GTOLs attached to base dimensions, datum planes or surfaces, not unattached notes.	X		Stds. Para. 5.2
Verify correct text height.		X	
Verify note contents & spelling.		X	
Verify that all dimensions & geometric features are displayed.		X	
General feature check.		X	Check that holes are features, not sketched lines, etc.
Verify all required geometric tolerances and datum references exist.	X		Verify during visual check of drawing only - not at the part/assy levels. (Stds. Para. 3.2)
Parts contain curves or box with note reference to represent part number (if necessary).	X		Verify during visual check of drawing only. (Stds. Para. 3.1.1/4.1.1/4.2)
Drawing Revision is 1 higher than currently released version.	X		If revising current drawing. (Stds. Para. 2.8.2)
All drawing deficiencies in TVPDMS have been resolved and the drawing conforms to the resolutions when applicable. Outstanding ECPs have been investigated.	X		Stds. Para. 2.2
All differences between new & old drawings have been documented by NOR.	X		Stds. Para. 2.8.1
A visual check (comparison) of new & original drawing has been completed and all required information has been accurately transferred to the new drawing.	X		Current drawing conversions only. (Stds. Para. 3.1.2.1)
Part Check			
Perform Model Check at the beginning and end of the check.			
Run Model Player.		X	Force regeneration.
Check layers and their status.		X	
Check Item ID layer.		X	
Check that correct distribution statement is located in the model tree labeled "Distribution_Note"		X	Dist. Statements A, B, C, D, and X can be used.
Check that correct EAR or ITAR statement is located in the model tree as a note.		X	Only Dist. Statements B, C, D, and X require an EAR or ITAR statement.
Verify all geometric tolerancing datum planes are on the G_DTM_PLANES layer.	X		(Stds. Para. 3.11)
Check part specific notes.		X	FMTV Tools - General Note Editor is available.
Check Model Tree for "Note 0" / Suppressed Features / "DWG_NOTE"		X	
Check defined material and material note 2.		X	They should be the same.
Check material callout.	X	X	Dummy material callout not allowed. All appropriate material files must be assigned to part. Primary material file is current. (Stds. Para 3.3)
Check units (must not be Newtons)		X	Model units and Windchill PDMLink parameter units must be the same.
Check parameters for omissions or value errors.		X	See Attachment A.

Check for weak dimensions.		X	No weak dimensions allowed.
Check relations.		X	Remove any old "Standard Relations" and verify proper comment lines.
General feature check.		X	Check that holes are features, not sketched lines, etc.
Verify that the part is representative of physical characteristics of real part.	X	X	Verify part modeled matches latest released revision of drawing. Outstanding ECPs have been investigated. All drawing deficiencies in TVPDMS have been resolved and part conforms to resolutions when applicable. (Stds. Para. 2.2)
All features in the original drawing must be included in the model.	X		Stds. Para. 3.1.1
Check 3D note naming and contents	X		IAW Standards Para. 3.5
The part regenerates without failure or external input required.	X		Stds. Para. 3.1.3
Assembly Check			
Perform Model Check at the beginning and end of the check.			
Run Model Player.		X	Force regeneration.
Check layers and their status.		X	
Check Item ID layer.		X	
All geometric tolerancing datum planes have been added to the "G_DTM_PLANES" layer.	X		Stds. Para. 4.7
Check part specific notes.		X	FMTV Tools - General Note Editor is available.
Check Model Tree for "Note 0" / suppressed features / packaged components / "DWG_NOTE"		X	
Check that correct distribution statement is located in the model tree labeled "Distribution_Note"		X	Dist. Statements A, B, C, D, and X can be used.
Check that correct EAR or ITAR statement is located in the model tree as a note.		X	Only Dist. Statements B, C, D, and X require an EAR or ITAR statement.
Check for Ref. parts (no cage code).		X	All assembly parts must have a cage code. Assembly cannot be promoted to Preliminary Release until all down level parts and assemblies that are not part of the current ECP are at release level Released.
Check parameters for omissions or value errors.		X	See Attachment A.
Check for weak dimensions.		X	No weak dimensions allowed.
Check relations.		X	Remove any old "Standard Relations" and verify proper comment lines.
Verify the /* has been removed from the UNIT_WEIGHT=MP_MASS("") code line.	X		(Stds. Para. 3.12.4) Check by confirming Unit_Weight is appropriate.
General feature check.		X	Check that holes are features, not sketched lines, etc.
If 5 rejection items are found, stop checking and return to sender.		X	Rejection item: Checking is being stopped at this point. 5 Checklist items have been discovered with invalid information. Entire package is being

			rejected. Please review all objects according to TACOM BULLETIN-CHECKLIST-100
Check that parts of a weldment or assembly contain all part specific notes.	X		The only notes to be verified in down level parts will be Note#1 (standards/specs) and the material note for the specific part. (EX: Assembly material Note#6 does not have to be in Part Note#6 -> Part note may be #2, etc. (Stds. Para. 3.5)
Assembly modeled matches latest released version of the drawing.	X		Outstanding ECPs have been investigated. All drawing deficiencies in TVPDMS have been resolved and assembly conforms to the resolutions when applicable. (Stds. Para 2.2)
All parts in the original drawing are included in the model and for assemblies, that all parts are assembled correctly. (Ex: make sure that -001 part is not assembled for a -002 part)	X		This is confirmed during visual comparison between drawing and assembly model. (Stds. Para 4.1.1)
All assembly features are present, are truly assembly features, and should not be part features.	X		This item should not be checked, per decision at January 2004 IPR. (Stds. Para. 4.1.3)
The assembly of the model follows a logical sequence.	X		This will not be checked. (Stds. Para 4.1.2)
The assembly regenerates without failure or external input required.	X		Stds. Para. 4.1.4
Verify 3D note naming and contents.	X		IAW Standards Para. 4.3
No packaged components unless appropriate.	X		Stds. Para. 4.1.5
Voting Notifications			
Rejections Note: After voting to reject an RTP, all RTP items must be locked and transferred back to the originator.			
Voting: After voting, send an email notification to the originator.		X	Rejection notification or Approval Notification.
2nd Check			
Run Model Check.			
Run Model Player.		X	
Check that rejection items are fixed.		X	
Double check that rev. block is correct.		X	Rev. history ERR numbers may have updated incorrectly.
Windchill PDMLink			
All required parameters have been correctly entered.			
Drawing, Part, Assembly & PDMLink parameters are set to the correct (& same) level.	X		Stds. Para. 2.8.1
Unit_Weight is appropriate for parts & assemblies.	X		Stds. Para. 3.12.4
	X		Stds. Para. 2.10 – Not required
NOR document is filled out and related to the Creo Parametric drawing as attachment	X		Stds. Para. 2.10
All parts defined by a drawing are related to the drawing even if they are not used in a view on the drawing. (Ex: -500, -800 part)	X		Stds. Para. 5.12

Drawing is promoted in accordance with the current release procedure document.	X		Stds. Para. 2.10

TABLE 15: STANDARD MODEL CHECKLIST

2D PDF DRAWING CHECKLIST			
Item	**TAC**	**CTR**	**Comments**
Note contents of RTP.		X	
		X	Reject if QA document is attached to RTP.
		X	
Check that only one drawing is attached to RTP.		X	Reject if more than one drawing is attached to an RTP.
Create a workspace and check out RTP contents.		X	Check that QA Comments sheet is attached to drawing.
Print the NOR.		X	If required.
Print the PDF Drawing.		X	
Update QA Comment form to reflect contents of RTP.		X	If required
Print out released Drawing previous rev.		X	
Check that PDF drawing is named correctly.	X	X	(Stds. Para 5.15)
PDF/2D Drawing Check			
Paper-to-paper comparison of the CMSTAT previous rev. & current PDF rev.		X	Any differences must be documented on NOR.
Verify that all changes listed on NOR have been incorporated on PDF drawing.		X	
Check parameters in PDMLink		X	See Attachment A. Reject if any parameters have incorrect/missing values.
Voting Notifications			
Rejections Note: After voting to reject an RTP, all RTP items must be locked and transferred back to the originator.		X	
Voting: After voting, send an email notification to the originator.		X	Rejection notification or Approval Notification.
2nd Check			
Check that rejection items are fixed.		X	
Do not need to do a full check of everything.		X	Unless 5 items were previously rejected, then recheck completely.

TABLE 16: PDF DRAWING CHECKLIST

VENDOR/LEGACY/STANDARD PART MODEL CHECKLIST			
Item	**TAC**	**CTR**	**Comments**
Note contents of RTP.		X	
Check RTP to determine if NOR is included. If there is no NOR, search Pro/INTRALINK for previous rev. (NOR is required if previous rev.)		X	
Check that only one drawing is attached to RTP.		X	Reject if more than one drawing is attached to an RTP.
Run Where Used report on each part/assembly being promoted. Check that next up assemblies at release level QA1 or higher regenerate before promoting part or assembly to QA1.	X	X	Rejection item: Components do not assemble correctly in assembly.
Promoting Assemblies: All subordinate items must be at the same or higher release level than the assembly being promoted.		X	
Create a workspace and check out RTP contents.		X	Check that QA Comments sheet is attached to drawing.
If no drawing is present, check for PDF drawing.		X	Make sure drawing or PDF drawing is attached to package.
Print the NOR.		X	If needed.
Print the PDF Drawing.		X	If needed.
Update QA comments form to select contents of RTP.		X	
Print out CM-Stat Drawing & current rev. of Creo Parametric Drawing.		X	
Check that all parts/assemblies/drawings are named correctly	X	X	(Stds. Para. 3.8) Will be confirmed by ModelCHECK only.
Paper-to-paper comparison of the CMSTAT previous rev. & current Creo Parametric rev.		X	Any differences must be documented on NOR.
Check parameters in PDMlink.		X	See Attachment A. Reject if any parameters have incorrect/missing values.
Part Check			
Run Model Player.		X	Force Regeneration.
Check parameters for omissions or value errors.		X	See Attachment A.
Verify that the part is representative of physical characteristics of real part.	X	X	Verify part modeled matches latest released revision of drawing. Outstanding ECPs have been investigated. All drawing deficiencies in TVPDMS have been resolved and part conforms to resolutions when applicable. (Stds. Para. 2.2)
Part regenerates without failure or external input required.	X		Stds. Para. 3.1.3
Voting Notifications			
Rejections Note: After voting to reject an RTP, all RTP items must be locked and transferred back to the originator.		X	
Voting: After voting, send an email notification to the originator.		X	Rejection notification or Approval Notification.
Verify that part can be assembled into next higher assembly.		X	
2nd Check			
Run Model Check.		X	
Run Model Player.		X	

Check that rejection items are fixed.		X	
Do not need to do a full check of everything.		X	Unless 5 items were previously rejected, then recheck completely.
Double check that rev. block is correct.		X	Rev. history ERR numbers may have updated incorrectly.
Windchill PDMLink			
All required parameters have been correctly entered.	X		
For Vendor supplied and standard part models, a reduced number of parameters are required.	X		If values are known, enter the following parameters: PART_NUMBER, DRAWING_NUMBER, CAGE_CODE, NOMENCLATURE, UNIT_WEIGHT, MODEL_REV, MODELER_NAME_1 (modeler name can be the company that supplied the model).

TABLE 17: VENDOR / LEGACY / STANDARD PART MODEL CHECKLIST

D.6 Procedure for Promoting Objects to a new Revision

- Check out everything with the same base number (e.g. 19207_12421234.drw where 12421234 is the base number). An exception is when in the newer revision one or more of the components are no longer required and those can remain at the previous revision.
- Modify everything with the same control number.
- Change the following attributes:
  - **VERSION** - revision on ECP
  - **RELEASE_LEVEL** - In Work
  - **ERR_ECP_NUMBER** - the current ECP number
  - **ERR_ECP_APPROVER** - replace current name with *name of approver*
  - **ERR_ECP_DATE** - replace current date with *YY-MM-DD* (to be changed later)
- Check the Objects into Commonspace
- Windchill Parts and Drawings shall also be revised at the same time.
- Refer to PDMLink User Guide for a Step by Step Guide on how to Promote Objects

D.7 Superseded and Obsolete Parts

A superseded or obsolete part should not be modeled. The main goal is to model only active parts on a drawing, thus superseded/obsolete are not needed. If the drawing has both superseded and active parts, only model the active parts. **The following is only if the drawing has not been converted to Creo Parametric.**

How this is handled:

- Remove the Superseded / Obsolete part from the drawing
- If the Superseded / Obsolete part is on the sheet before the production / active part, move the production / active part to the earlier sheets and remove the extra sheets.
- Update Part Tables (If any) to the superseded parts to say REMOVED AT REV (X) either in the SHEET column or in the DESCRIPTION column.

FROM:

TABLE		
PART NO.	SHEETS	COMMENTS
12417336	1	SUPERSEDED BY 12417336-001
12417336-001	2	SUPERSEDES 12417336

TO:

TABLE		
PART NO.	SHEETS	COMMENTS
12417336	REMOVED AT REV.E	SUPERSEDED BY 12417336-001
12417336-001	1	SUPERSEDES 12417336

FIGURE 99: FROM-TO EXAMPLE

- Document all changes as normal in the NOR, including removal of sheets, updating of tables, moving the production/active part to earlier sheets, etc.

If the drawing is completely obsolete (no active parts on the drawing), do not model that part or drawing. If for some reason the models are required it can be modeled, but that should be a rare case.

If the drawing has already been converted to Creo Parametric and a future revision will supersede / obsolete a part on the drawing, the model shall NOT be removed from the drawing. This will allow the user to leave the models on the drawing and not have to shift around multiple views.

D.7.1 Superseded and Obsolete Parts – Revision Block

The revision block shall conform to the ASME Y14.35M standard. In addition, if a part is superseded by a new drawing, place a marking above the Title Block that states SUPERSEDED BY 12XXXXXX (part number) in a balloon as depicted in **Figure 100**

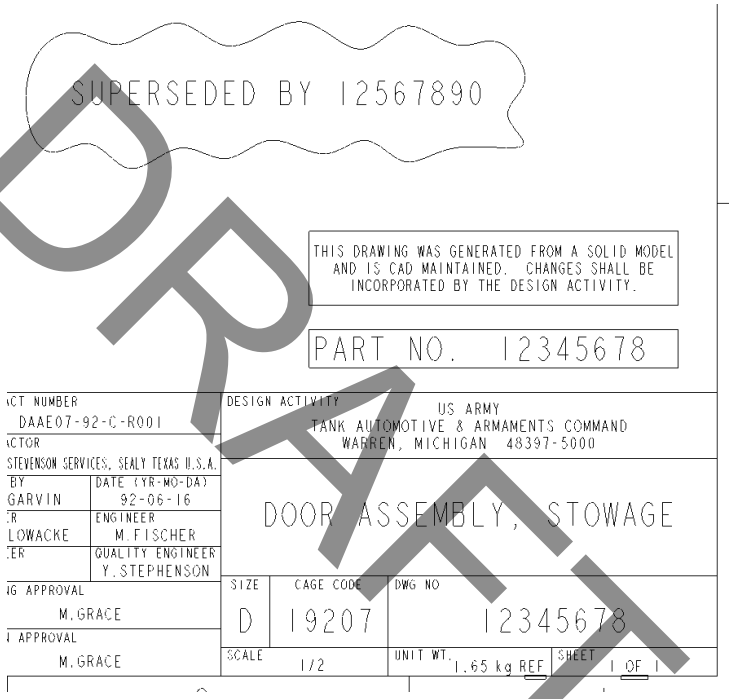


FIGURE 100: SUPERSEDED CALLOUT

D.8 Labels and Nameplate Modeling Procedure

All labels and nameplates (placards) are to be created and maintained in Creo Parametric.

D.9 Installation Drawings/Assemblies

All installation drawings/assembly models shall contain a Vehicle Coordinate System (VCS) for ease of assembling to Chassis and Top Level models (Refer to ***Appendix E***).

E Vehicle Coordinate System

E.1 Vehicle Coordinate System (VCS)

Note: The VCS listed below for FMTV was established for the FMTV A1P2 variant. The information remains for historical purposes only.

The VCS is the driving base coordinate system for all FMTV trucks. The VCS is established by the intersection of the 3 Planes representing the length, width and height of the truck; x, y and z respectively. The 3 planes coincide with the axis through the E Datum hole on the frame rails for the X axis (length) and the Z axis (height) and the centerline of the vehicle for the Y axis (width). The E Datum lines up with the theoretical centerline of the front axle and establishes the F Datum at the opposite end of the frame rail which lines up with the rear axle/trunion centerline. The frame rail dimensioning scheme is based on the E Datum and all hole features are constrained back to the E Datum. This is the same for all frame rails (roadside and curbside) therefore this is the logical location for the VCS. The right hand rule establishes that from the VCS, the X is positive rearward, Y is positive towards the Curbside and Z is positive vertically. This coordinate shall be included in all Installation assembly models and named as VCS. As a default, the intersection of the Front, Side and Top planes establish the X, Y and Z planes respectively.

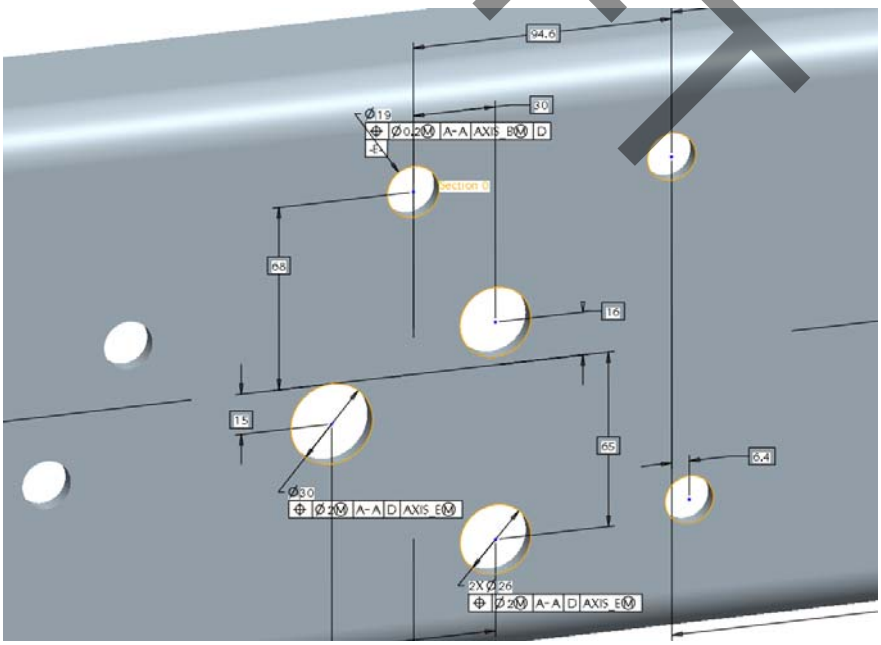


FIGURE 101: VCS ESTABLISHED ON ROADSIDE FRAME RAIL

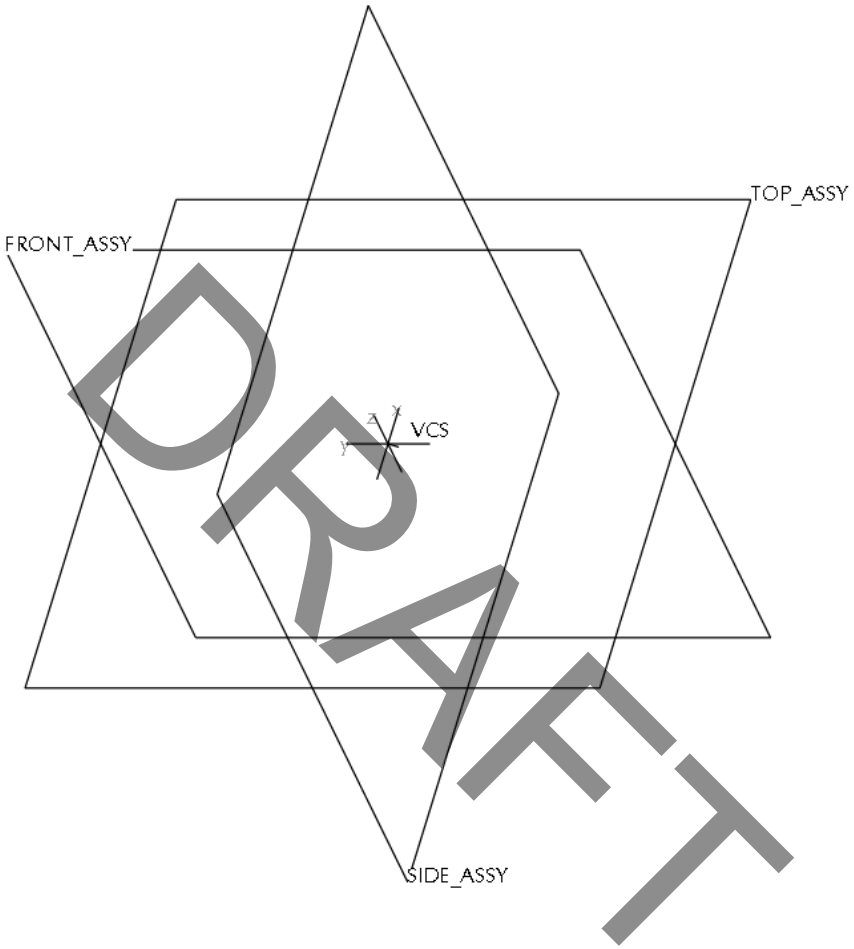


FIGURE 102: VCS IN DEFAULT VIEW

In "default" view orientation, the VCS is established at the intersection of the three default assembly datum planes. Notice the orientation of the X, Y and Z datum references. This is essential that they are oriented in this manner or the whole purpose for the VCS is of no value.

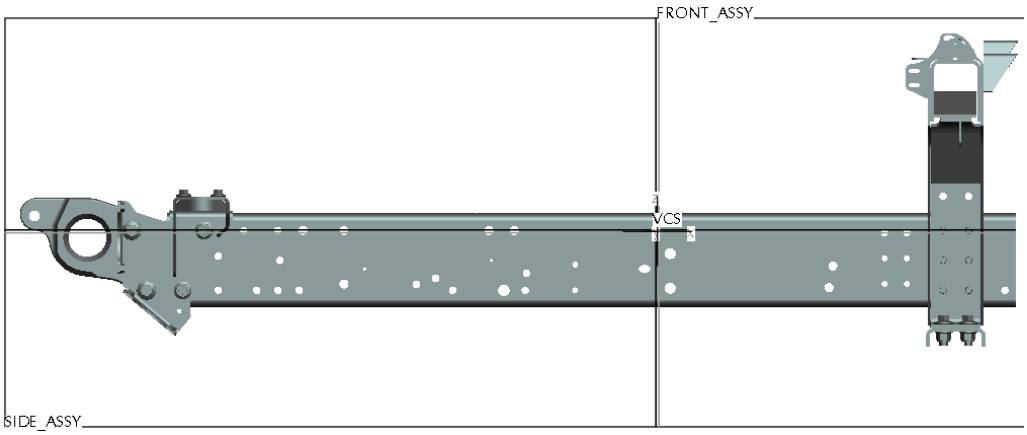


FIGURE 103: VCS WITH FRAMERAIL

The same default view with the frame assembly added for clarification is shown below.

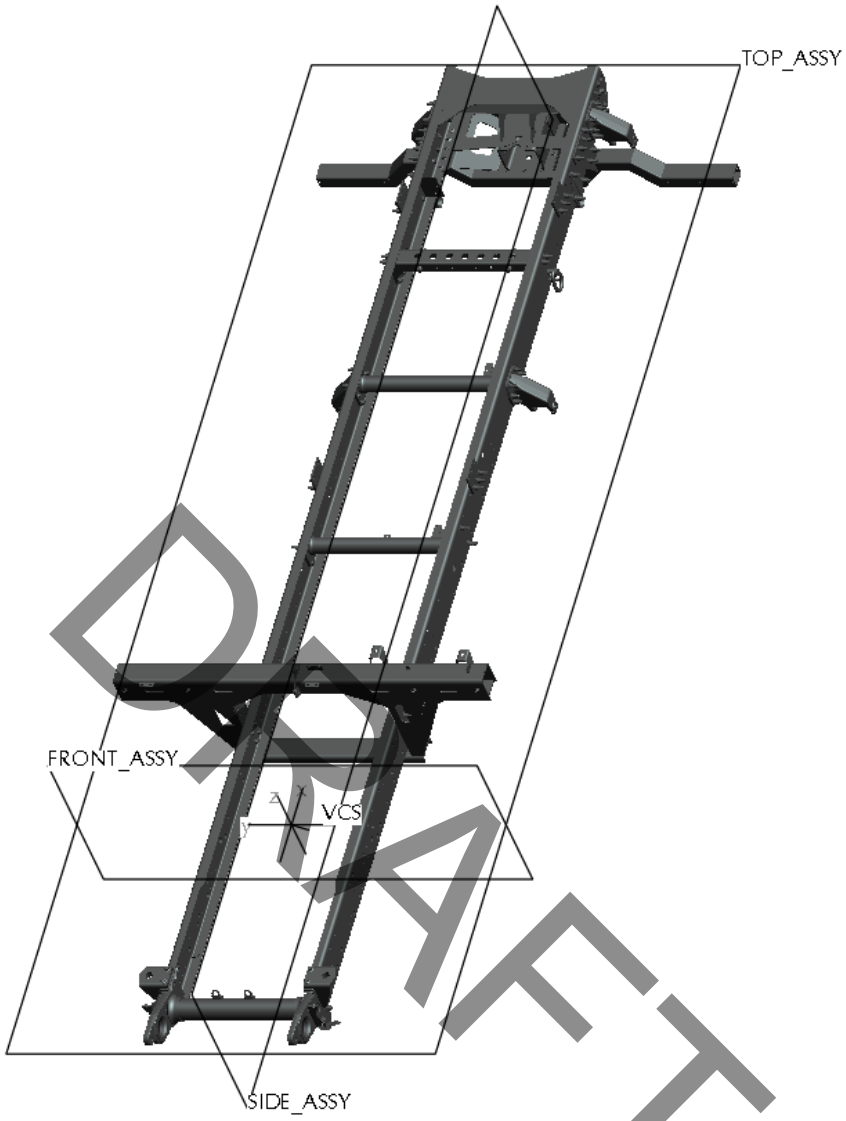


FIGURE 104: VCS IN DEFAULT WITH FRAMERAIL

For Installations that are created with the VCS in mind, the VCS coordinate as shown is at the intersection of the default datum planes and the XYZ orientation established as shown earlier. If the Installation assembly is not built off the default datum planes, then the VCS coordinate can still be OFFSET by either a set of datum planes, curves, points or by another coordinate system.

Front Cab Mount Installation shown below with the VCS and CBCS (Cab Body Coordinate System – explained later).

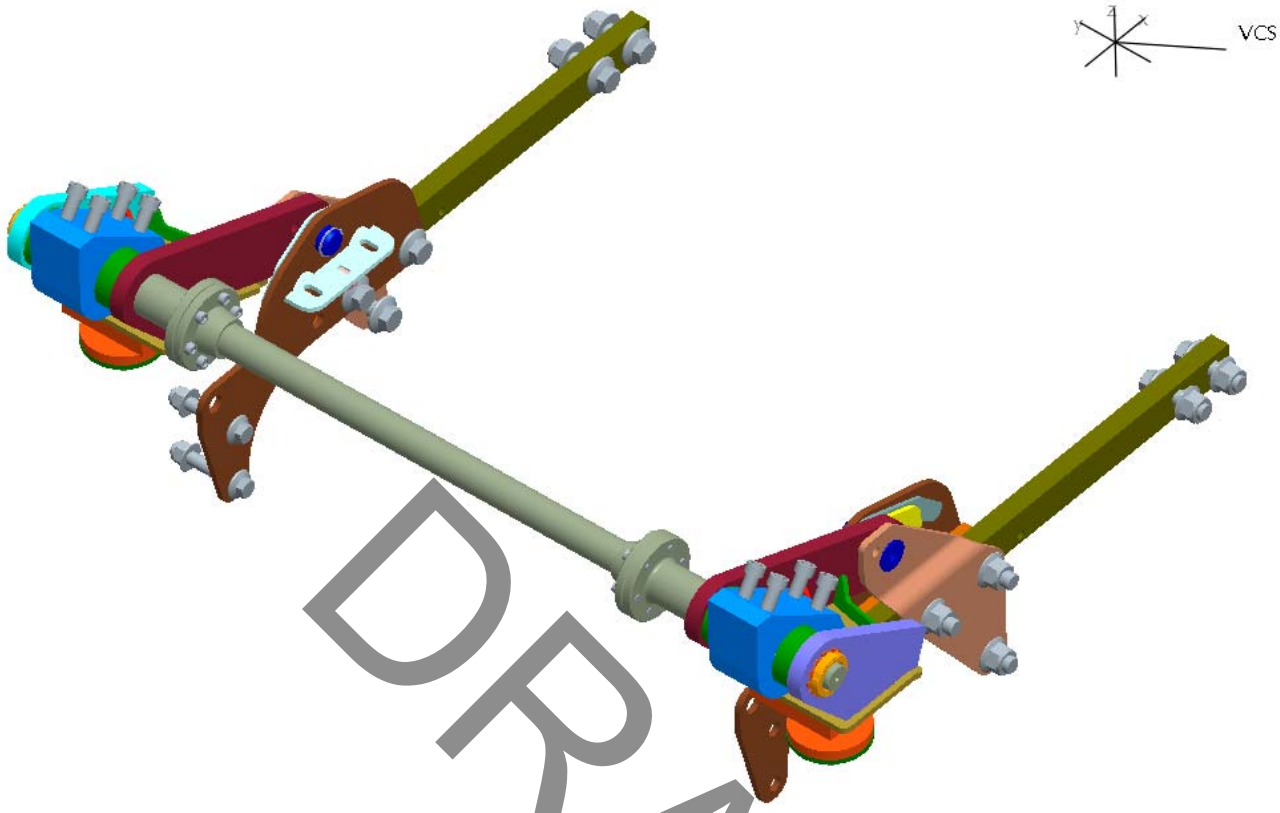


FIGURE 105: FRONT CAB MOUNT INSTALLATION WITH VCS

The Installation can then be assembled to the Frame assembly by mating the 2 VCS coordinate system points and it will snap into place as shown below.

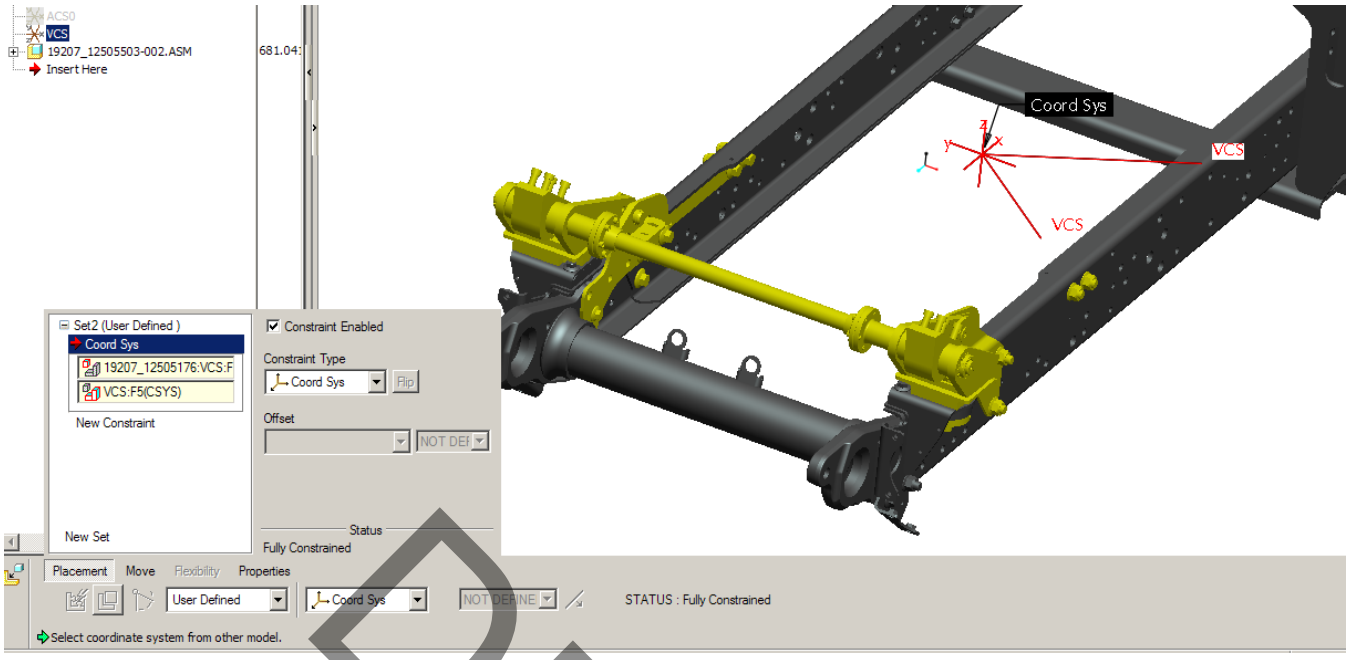


FIGURE 106: FRONT CAB MOUNT ASSEMBLY CONSTRAINT

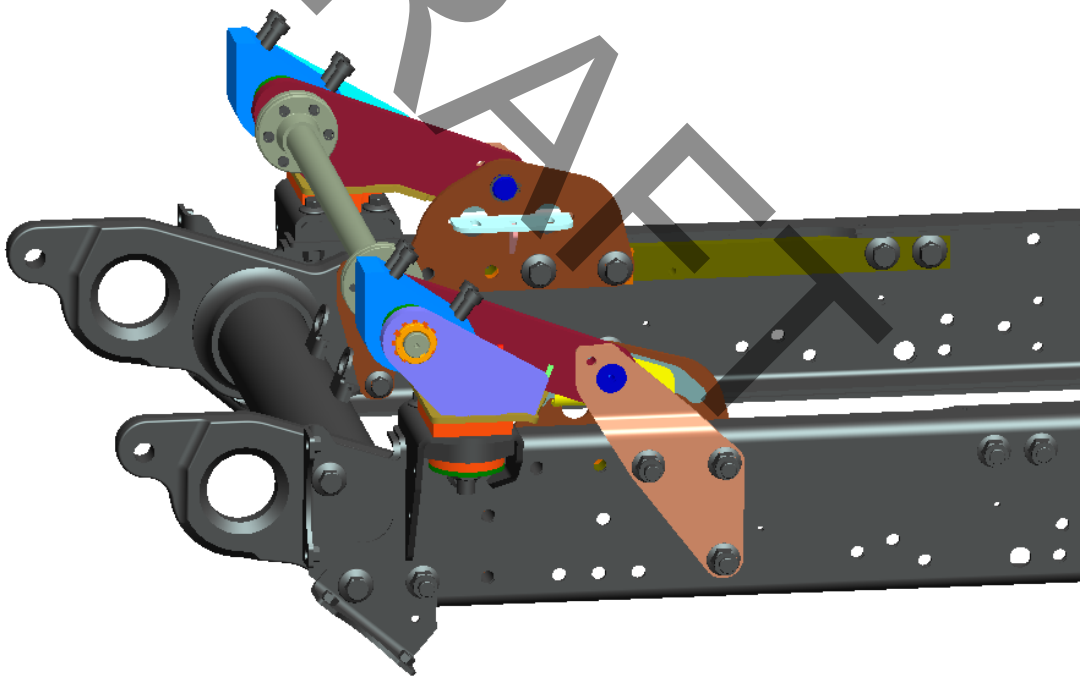


FIGURE 107: FRONT CAB MOUNT INSTALLED

E.2 Cab Body Coordinate System (CBCS)

The CBCS is the coordinate system for the Cab body and all related installations shall have a coordinate placed and named as the CBCS. The top level Dressed Cab model shall have the VCS

included in it as well. The CBCS varies between the different cab designs but it shall be established by following the same X Y Z relationship and orientation as the VCS.

The location of the LTAS Cab CBCS and the chassis VCS is shown below:

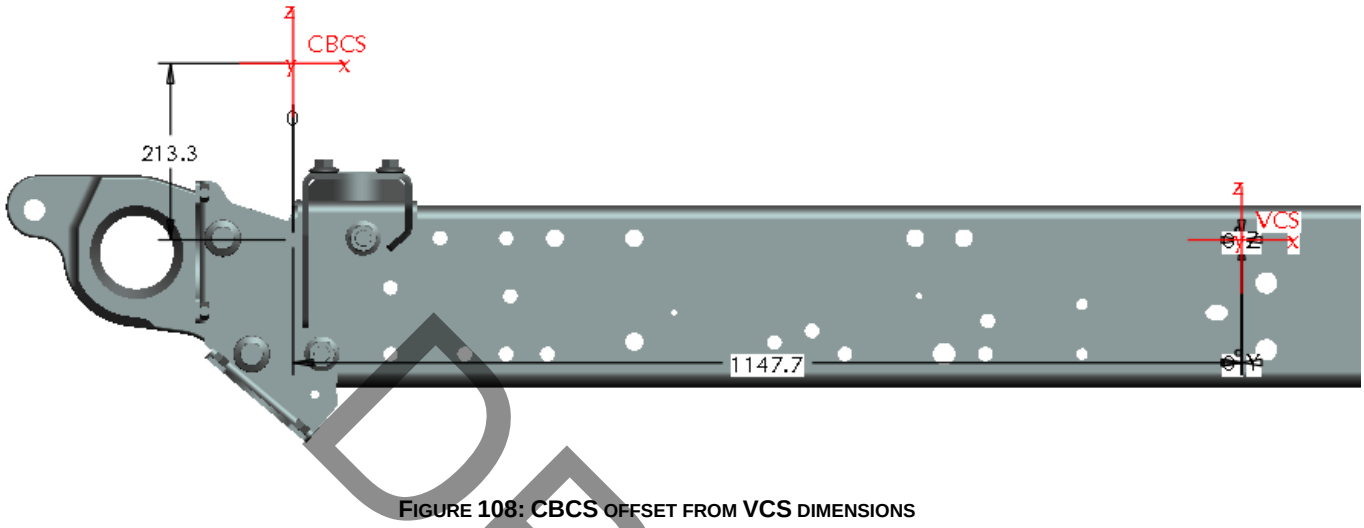


FIGURE 108: CBCS OFFSET FROM VCS DIMENSIONS

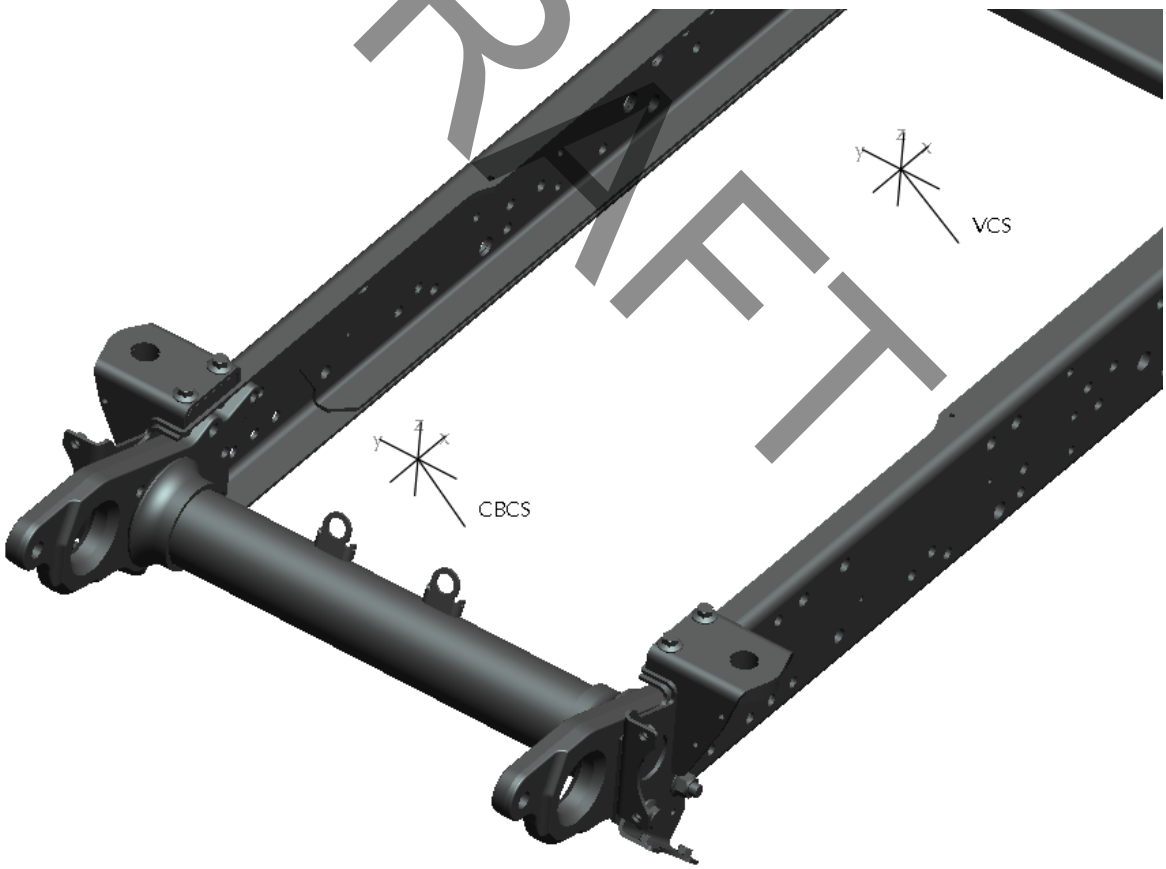


FIGURE 109: CBCS IN DEFAULT ORIENTATION

E.3 Roadside (aka RS and Driver Side) and Curbside (aka CS and Passenger Side)

The “roadside” of the vehicle is the side of the vehicle that is parallel to the road and the “curbside” is the side parallel to the curb. The usage of Left side and Right Side can be confusing because in orthographic drafting terms, the roadside view is the right side but WRT vehicular terms, it is left side because that is how the view is observed by the Driver while sitting in the vehicle. For clarity and uniformity, roadside and curbside have been designated.

F Special Part and Drawing Types

F.1 Standard Parts

Standard parts are components defined by standards organizations and are not under the control of the Tactical Vehicle (TV) group at TACOM. These would include parts defined by SAE, NAS, ANSI, and MS to name a few. These parts are modeled as needed and reflect the size shape and weight of the actual parts. An abbreviated set of parameters is needed for these parts. As these parts are not controlled by TV, a drawing is not created, and an ECP is not processed. Parts should be checked by a second modeler to ensure the basic geometry and weight are correct and the necessary parameters are filled out. The parts can then be promoted directly to the Released state. Care should be taken in naming the parts. MS35489-23 is not the same as MS35489-023. These parts shall all be located in the / STANDARD PART LIBRARY / MODELS AND DRAWINGS folder in Windchill PDMLink and not in the folders where they are used (release procedure issues).

In addition, the Specification Number shall be filled into the SPEC_NUMBER parameter. This Spec Number shall match exactly what the Spec says. In addition, the spec shall be called out on the drawing in either the notes section or within the Table (See ***Figure 110***). Users can use table sbpartslist_spec.tbl in Commonspace. When multiple specs are called out on a drawing, the table method is preferred.

16	2	MS51029-51	SSCR-HEX SOC,FL PT,ALY STL	80205	NASM51029
15	1	MS24665-516	PIN, COTTER (SPLIT)	80205	NASM24665
14	1	MS51472-01	NUT	96906	MS51472
13	2	MS35649-2254	NUT, HEX, 1/4-20 UNC-2B	80205	NASM35649
12	1	MS51930-1	PAD EYE	96906	MS51930
11	2	B18221BH125W	WASHER	80204	ASME B18.22.1
10	2	B18221BH063N	WASHER	80204	ASME B18.22.1
9	1	MS16842-3	CLAMP,WIRE ROPE/DOUBLE GRIP/THREADED,STEEL	96906	MS16842
8	1	11636670	BOLT	19207	
7	1	11636668	CABLE	19207	
6	1	11636665	U-BOLT	19207	
5	1	11636591	PAWL ASSY.	19207	
4	1	11592666	MEMBER	19207	
3	1	11592657	SHAFT ASSY	19207	
2	2	11592656	FLANGE	19207	
1	1	7409903	SPRING PLUNGER LATCH	19207	
FIND NO XX / XX QTY		PART NUMBER	DESCRIPTION	CAGE CODE	SPECIFICATION
PARTS LIST					

FIGURE 110: SPECIFICATION PARTS KIST

F.2 Common Parts

Common parts are parts that have TACOM (TV) drawings and are used across multiple systems. They must be released with an ECP. It will be an *Administrative 2* ECP (assuming no real content changed) to speed the process. What makes up a Common part? Items to look for: Does the drawing contain all the information required to actually manufacture the part? If not, it may be a common part. Is it an item that is stocked normally (e.g. clamps, bolts, nuts)? Does the drawing reference other standard part numbers e.g. is there a DIN (or other) part number called out on the drawing? If the drawing is completely detailed to the point that it could be manufactured from the drawing, it probably is not a common part. Common parts should be located in the ***Common_Parts*** directory (again release procedure issues).

F.3 Deformed parts

Deformed parts are parts that change shape upon installation. These parts have a couple of subdivisions that are treated differently. Items that are always deformed fairly consistently upon installation e.g. pop rivets, lock washers etc. have an uninstalled model(s) named with a –998 (and lower) for pictorial use on the drawing. The installed versions are created in their deformed state and

named with the standard dash numbering system. Items with a specific grip range should be modeled at their maximum grip range. The reason for this approach is to limit the vast number of parts that would be created if a model had to be created for each possible variation in installation. Items that are deformed differently upon each installation e.g. hose clamp, springs, etc. will use a –900 series part of the up assembly. The parameters of the –900 part should reflect the parameters of the up assembly except for Nomenclature, drawing number and most importantly part number. This method was implemented when it was discovered that the previous practice of 'partnumber_in_partnumber' did not work if there were dash numbers in both part numbers. This workaround may be able to be eliminated in future versions of Creo Parametric. Flexible components are still the preferred method in lieu of creating a -900 series part.

For example:

MS21266-2N (Grommet, Plastic, and Edging) is an item that is used in many assemblies and is deformed differently each time. Let's say it is used in an assembly 12345678. The file name for the grommet used in this assembly would be 19207_12345678-900.prt

NOMENCLATURE parameter value would be: GROMMET, PLASTIC, EDGING

PART NUMBER parameter would be: MS21266-2N

DRAWING NUMBER parameter would be: MS21266

If it is used a second time in the same assembly and has a different shape, it would be the same except the file name would be 19207_12345678-901

F.4 Vendor Item Control Drawings

Specification Control drawings now changed to **Vendor Item Control Drawing**. Parts in this category generally do not have complete definition of the part required to manufacture the part. Dimensions that do not appear on the drawing are not checked. If missing dimensions, actual parts should be obtained and measured to create a model that is representative of the actual part and allow a drawing to be created that is similar to the original. The part should be weighed and an appropriate

weight defined in the part. Sectional views that are for general information do not have to be recreated during the conversion efforts. If there is required information in the section, it still has to be created.

F.5 Source Control Drawings

Similar to Vendor Item Control Drawings as far as modeling is concerned. They will often have even fewer dimensions. If a model is available from a vendor but does not have all of the parameters etc. then an assembly should be made using the TV start assembly. All appropriate parameters etc can then be filled out in the assembly. For Source Control electrical drawings, see ***section 7.1.20*** for Schematic reference.

F.6 Installation Drawings

In ***Appendix*** *D.3* of this document, the acceptable methods for creating and working with lightweight backdrops for installation drawings are described. Using full 3D solid models as backdrops for installation drawings is not acceptable, regardless of how simple the backdrop objects are. 3D shrink-wraps are acceptable, which is defined in ***Appendix*** *D.4*. The 2D background geometry can be created from Creo Parametric models, snapshots, copy geometry, or from imported 2D geometry.

F.7 A1R ECP parts

Due to the timeframe of the ECP's as well as the fact that the baseline of the truck has not yet been released, A1R parts and drawings are treated a little differently. For checking purposes, three parameters do not have to be filled out. Drawing Number, Tolerance Angle, and Weight Unit. 'Background' or 'Legacy' parts are parts that are created to complete an installation, but are not the released version of the part; do not have to be checked. These parts do not have the cage code identifier in the front of the file name. They should have an 'OLD_' at the front to help eliminate the chance they are used in lieu of the released parts in the future.

References

- ASME Y14.100-2013 – Engineering Drawing Practices
- ASME Y14.5-2009 – Dimensioning and Tolerancing
- ASME Y14.35M – Revision of Engineering Drawings and Associated Documents
- MIL-STD-3046 – Configuration Management
- FMTV Creo Parametric Configuration Settings
- FMTV PDMLink User Guide