CreateAssembly Program User's Guide

5/29/2018

Table of Contents

Figures

Tables

Revision History

Date	Developer	Modifications
4/7/2017	Robert O.	Initial Draft
5/29/2018	Robert O.	Added information about how parameters are handled.

1   Introduction

The document is a user's guide for the program "CADCreoParametricCreateAssembly.exe", which hereafter in this document is referred to as the CreateAssembly program.

The CreateAssembly program takes inputs based on CyPhy models and

1. Builds Creo assemblies
2. Outputs data (e.g. mass properties, transformation matrices between parts/sub-assemblies, etc.) about the created assemblies
3. Creates FEA meshes along with loads and constraints for structural Test Benches
4. Computes requested data (e.g. point locations, bounding boxes, mass properties)

2 Architecture

Figure 1 shows the architecture for the CreateAssembly program. The CADAssembly.xml describes the assembly hierarchy and the constraints between parts/sub-assemblies and is the main input to the CreateAssembly program. The main output is the CAD assembly in the native CAD format. Additional outputs are the CAD assembly in other formats (e.g. STEP) and meta data about the assembly.
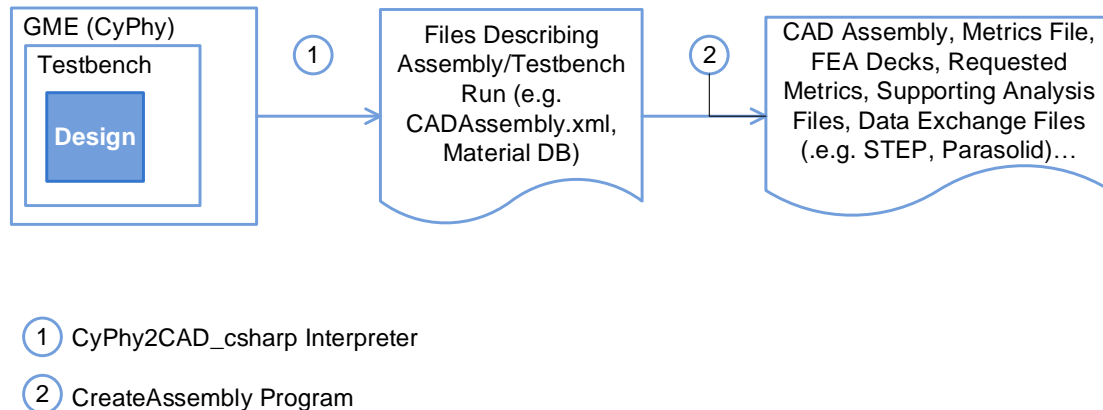

① CyPhy2CAD_csharp Interpreter

② CreateAssembly Program

Figure 1 – CreateAssembly Architecture

3 Global Coordinate System

The CreateAssembly program creates an empty Creo assembly and adds components (i.e. parts/sub-assemblies) to that assembly. The first component added is constrained by aligning the datums (for parts RIGHT, TOP, FRONT, and for assemblies ASM_RIGHT, ASM_ TOP, ASM_FRONT) in the added component to the empty assembly datums (ASM_RIGHT, ASM_ TOP, ASM_FRONT). The next added component would typically be constrained to the first component.

There is a way to specify which component is added first. In doing, so the user has control over the global coordinate system, because the first added component coordinate system would align with the coordinate system of the empty assembly. This is true because the datums RIGHT, TOP, FRONT are aligned with the coordinate system of the part and datums ASM_RIGHT, ASM_ TOP, ASM_FRONT are aligned with the coordinate system of the assembly.

To control which part is added first, the component assembly must have a ReferenceCoordinateSystem as shown in Figure 2.

Note – If the component assembly has more than one ReferenceCoordinateSystem, then it is arbitrary as to which component with a ReferenceCoordinateSystem would be added first.
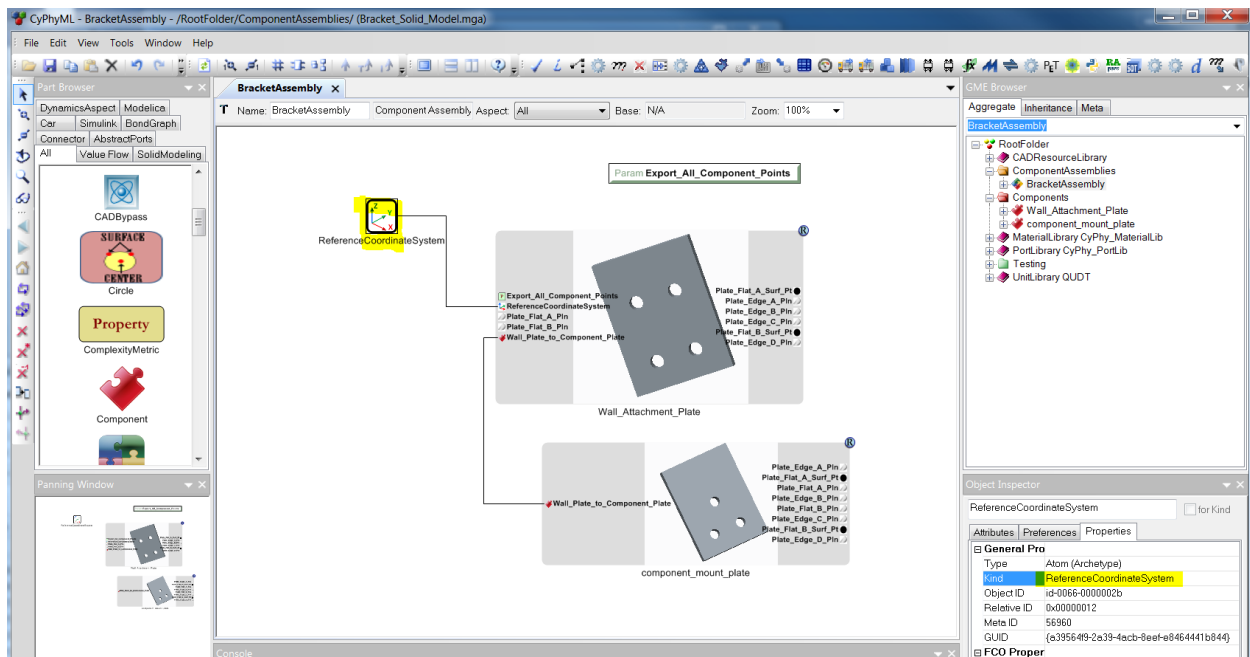
Figure 2 - Reference Coordinate System in a CyPhy Assembly

Within the component assembly, the ReferenceCoordinateSystem must be connected to a CoordinateSystem via a PortComposition line as shown in Figure 3.
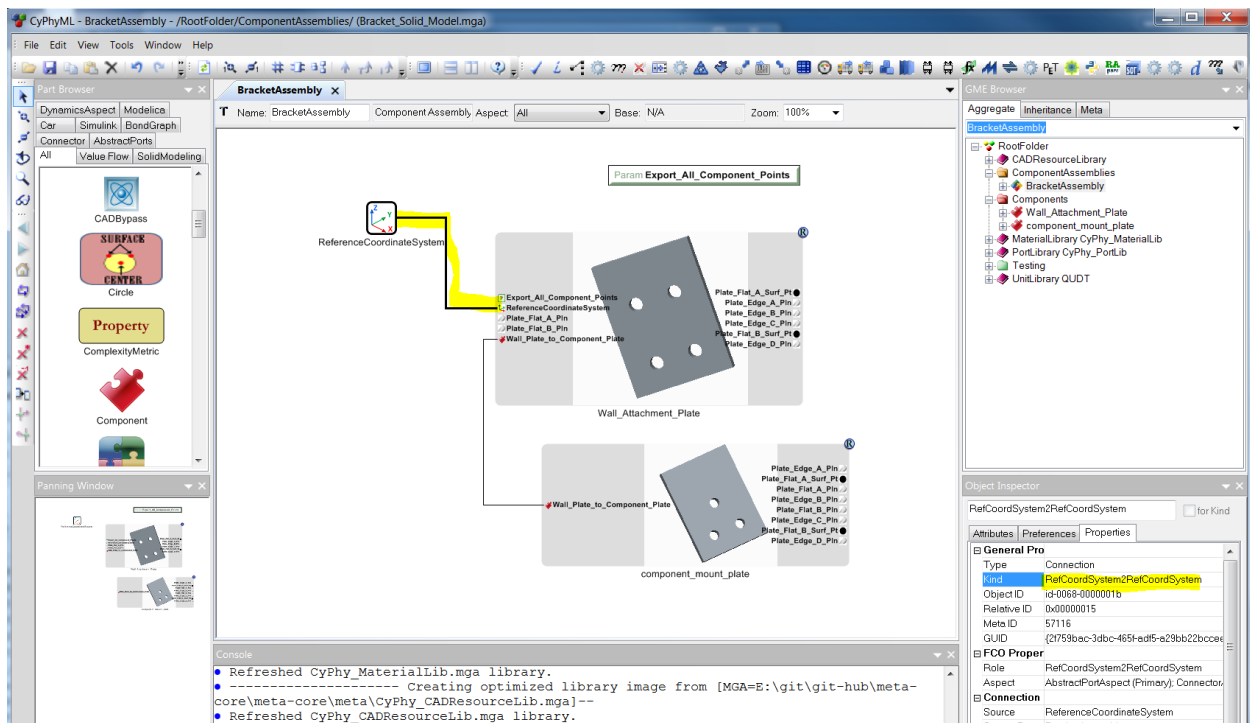

Figure 3 – Reference Coordinate System Connection in a CyPhy Assembly

Within the component, the ReferenceCoordinateSystem must be connected to a CoordinateSystem via a PortComposition line as shown in Figure 4.
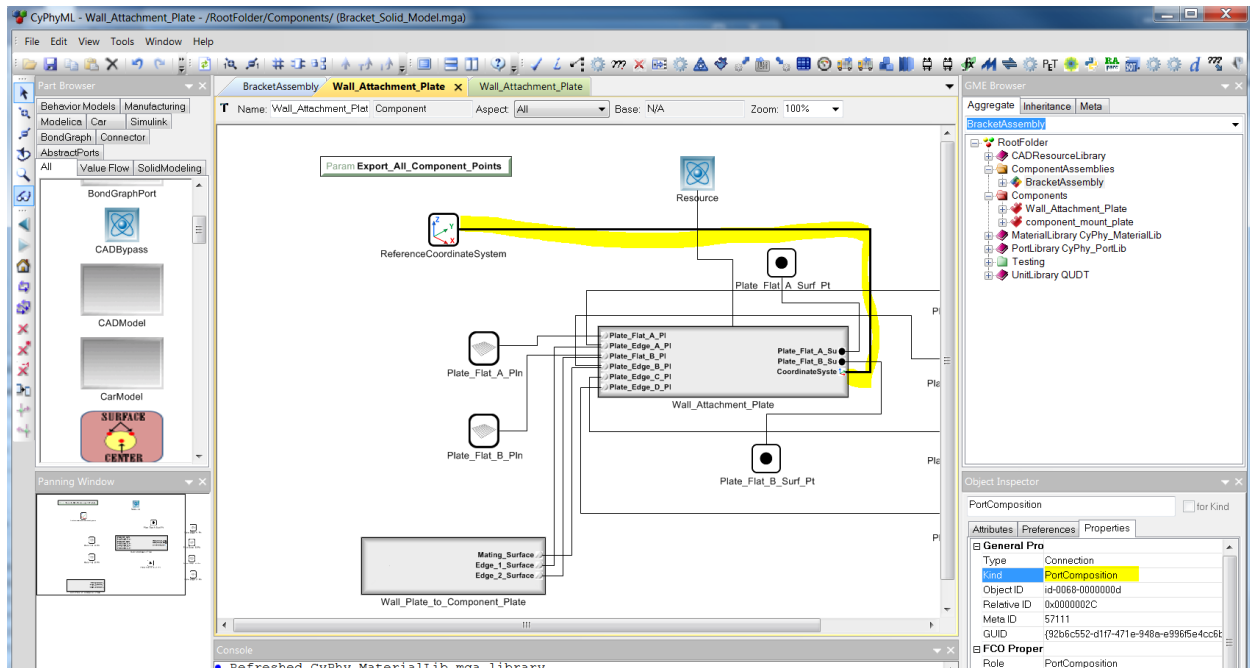

Figure 4 - Reference Coordinate System Connection in a CyPhy Component

4   Kinematic Joint Types

For the kinematic joints shown in Table 1, the CreateAssembly program will automatically determine the joint type and create the appropriate constraint in Creo. For example, if the constraint consists of an axis and plane, the CreateAssembly would create a pin constraint in the Creo model.

Additionally, guides can be used to set a particular orientation. For example, a pin constraint would typically be composed of an axis and plane. A guide could be applied that would set the initial orientation of the model relative to the rotation axis. For this case, the guide would typically be a plane where the axis would lie in the plane and thus the plane would specify the initial orientation about the axis. The guide is only used to determine the initial position and never results in an entity in the CAD model constraint.

Creo Joint Name	Common Joint Name	Required Geometry	Comments
Pin	Revolute	Axis and (Point or Plane)	
Cylinder	Cylindrical	Axis	
Slider	Prismatic	Axis and Plane	
Planar	Planar	Plane	Creo supports further restrictions (i.e. additional planes) but we will assume the classical definition (three degrees of

			freedom, x, y, rotation around z) of a planar constraint.
Ball	Spherical	Point	Creo supports other geometry types, but currently only point is supported.

Table 1 – Kinematic Joints

5   FEA Analysis

5.1   Patran-Nastran Solution

5.2   Request Volume and Mass for a Plate Model

Typically, for a solid model, Creo knows the mass and volume information and this information is available from the metrics file. However, for surface models, only the FEA application knows the mass and volume information. This is because the FEA model has the plate thickness information along with the material density, and thus can compute the mass/volume for the entire FEA model.

To request that the volume and mass be computed for Patran_Nastran plate/surface model solutions, the following must be present in the Testbench.

1. Metrics added to the Testbench. Note – Strings are case insensitive
    a. FEA_Total_Volume
    b. FEA_Total_Mass
2. SolverType = PATRAN_NASTRAN
3. ElementType = Plate4/8

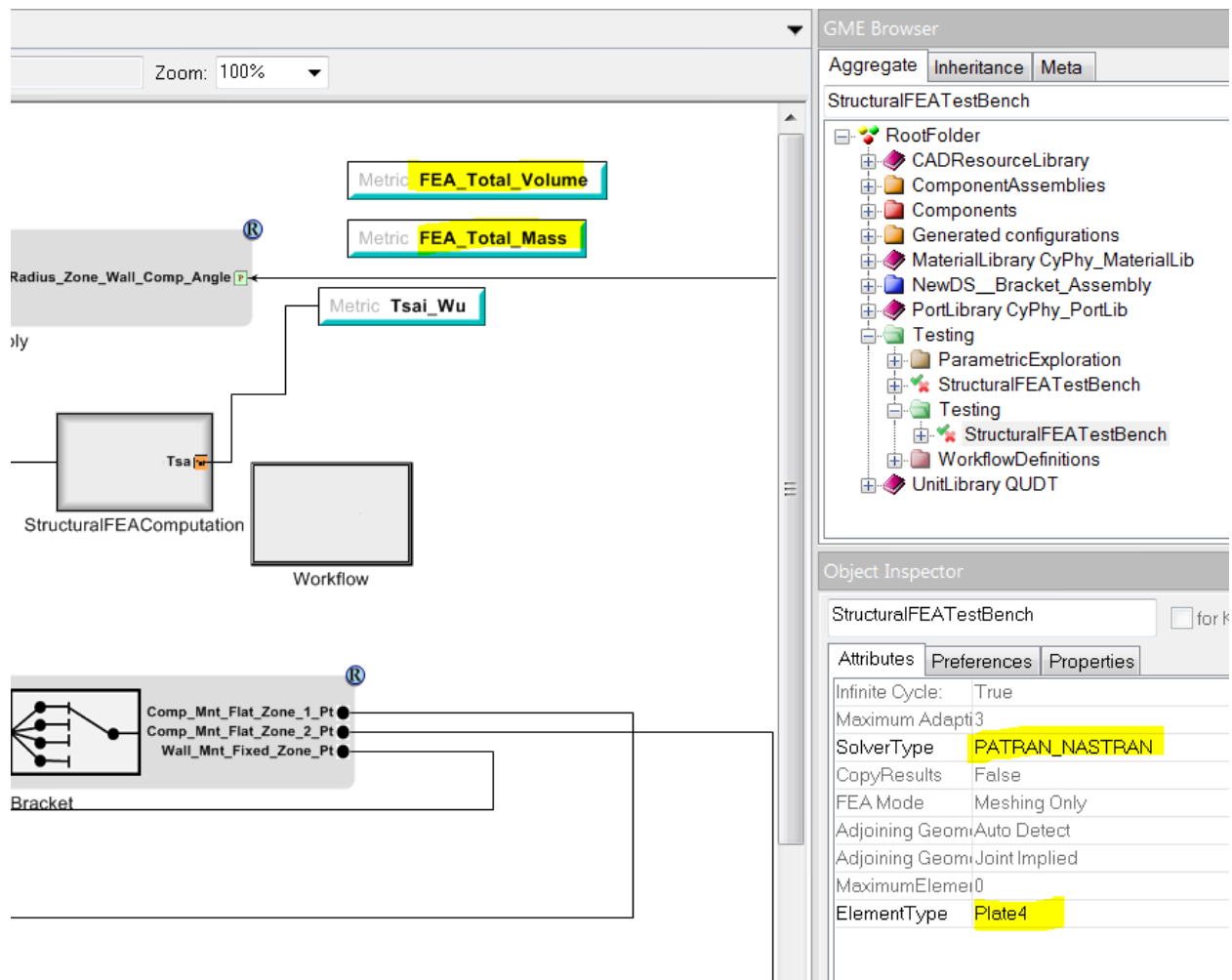An example Testbench is shown in Figure 5.

Figure 5 – Example PATRAN_NASTRAN Test Bench Requesting Mass and Volume

Once the TestBench runs, the computed values will be in testbench_manifest.json as shown in Figure 6.

```
"TierLevel": 0,
"Artifacts": [],
"VisualizationArtifacts": [],
"DesignName": "StructuralFEATestBench",
"Metrics": [
    {
        "GMEID": "id-0067-00000044",
        "Description": "",
        "DisplayedName": null,
        "VisualizationArtifacts": [],
        "Value": "182638.08",
        "ID": "9df8045d-827d-4ce6-9dd0-caade0de1217",
        "Unit": "",
        "Name": "FEA_Total_Volume"
    },
    {
        "GMEID": "id-0067-00000043",
        "Description": "",
        "DisplayedName": null,
        "VisualizationArtifacts": [],
        "Value": "0.29605633",
        "ID": "34acab3e-455a-45d0-96f0-4fec9cd6322b",
        "Unit": "",
        "Name": "FEA_Total_Mass"
    },
    {
```

Figure 6 – Example testbench_manifest.json with Mass and Volume

6   General Purpose Parameters Requesting Special Processing

6.1   Export_All_Component_Points

There was a need to export all points at the component level without explicitly calling out a metric in the TestBench for each point. Adding a parameter titled as follows will result in CyPhytoCAD_CSharp adding instructions in CADAssembly.xml to compute the coordinates of all points in the component. The points must be a Kind=Point and Role=Point in the component and must be connected to a point in the CADModel icon.

Export_All_Component_Points   Note – This is case sensitive.

The point coordinates will appear in ComputedValues.xml after the Test Bench successfully runs.

Note – Currently at least one point metric must be requested in addition to Export_All_Component_Points. Requiring the request for at least one point metric is due to a bug and will be corrected. The two items (Export_All_Component_Points and at least one point metric) in yellow in Figure 7 must be present.
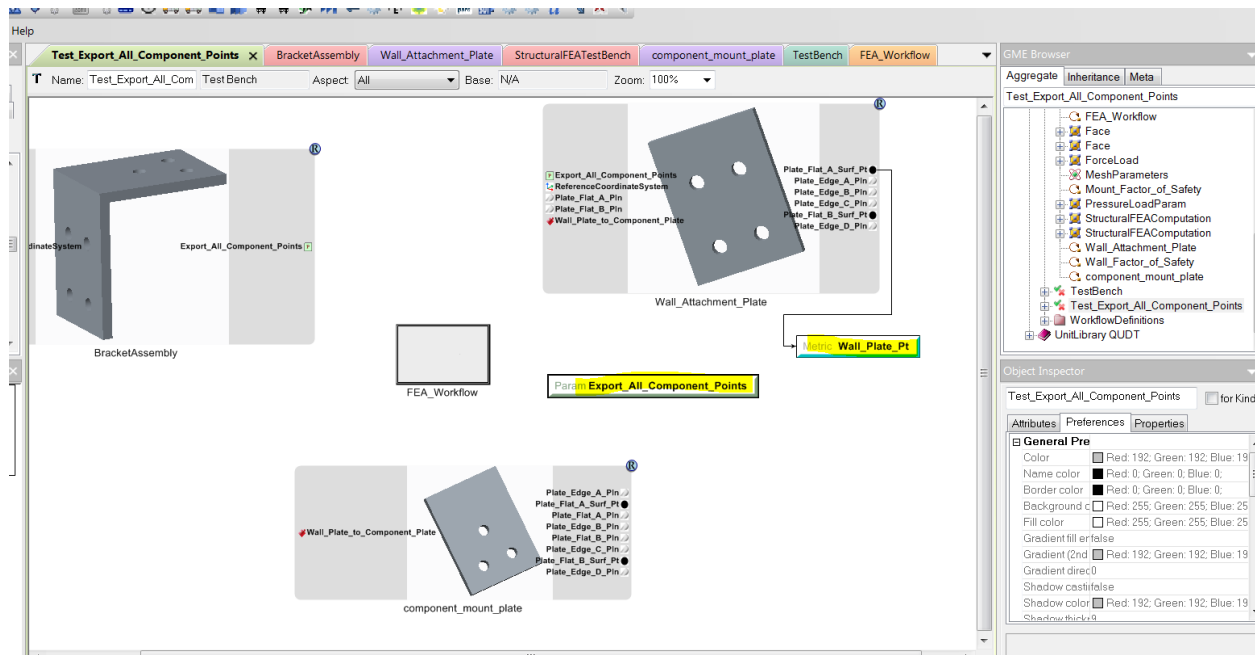
Figure 7 – Export_All_Component_Points Test Bench Example

Once the Testbench runs, ComputedValues.xml will contain the point coordinates.

The points must be referenced (brought out) in the component as shown in Figure 8. The highlighted in-yellow points will be in ComputedValues.xml along with their coordinates. Points in the CAD component that are not brought out to the component level will not be in ComputedValues.xml.
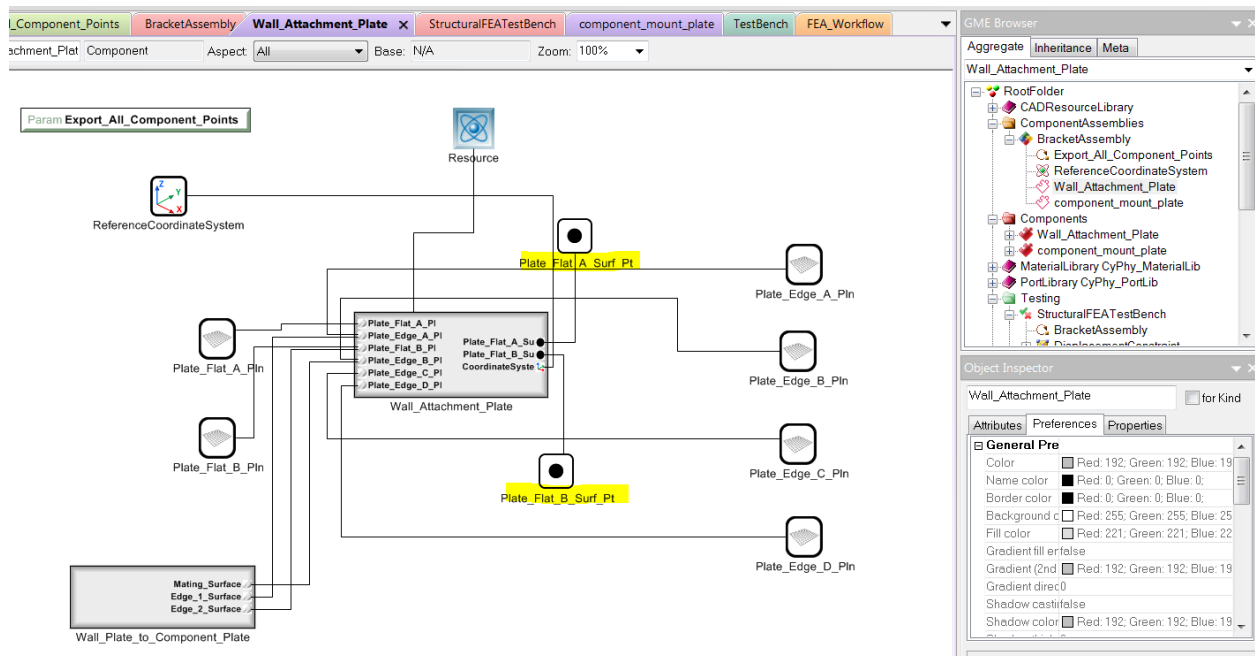
An example of ComputedValues.xml follows:

ComputedValues.xml

7 Material Properties

This section describes the sources of material properties added to the material library (models/MaterialLibrary/material_library.json) subsequent to the initial release of the library .

7.1 Composites

7.1.1 General Composite Properties

NASA Design and Manufacturing Guideline for Aerospace Composites
https://llis.nasa.gov/lesson/682

Note – The above link has typical composite material properties.

7.1.2 IM7_8551_7_Carbon_Epoxy

7.1.3 T300_PR319_Carbon_Epoxy

http://iccm-central.org/Proceedings/ICCM17proceedings/Themes/Behaviour/FAILURE%20CRITERIA%20FOR%20DES/F12.5%20Pinho.pdf

8 Simplified Representations

Most CAD systems support multiple reps (representations) of parts/sub-assemblies. For example, there could be a part that had a rep that was the complete part definition and a rep that was simplified for FEA analysis.

8.1 Creo Model Setup

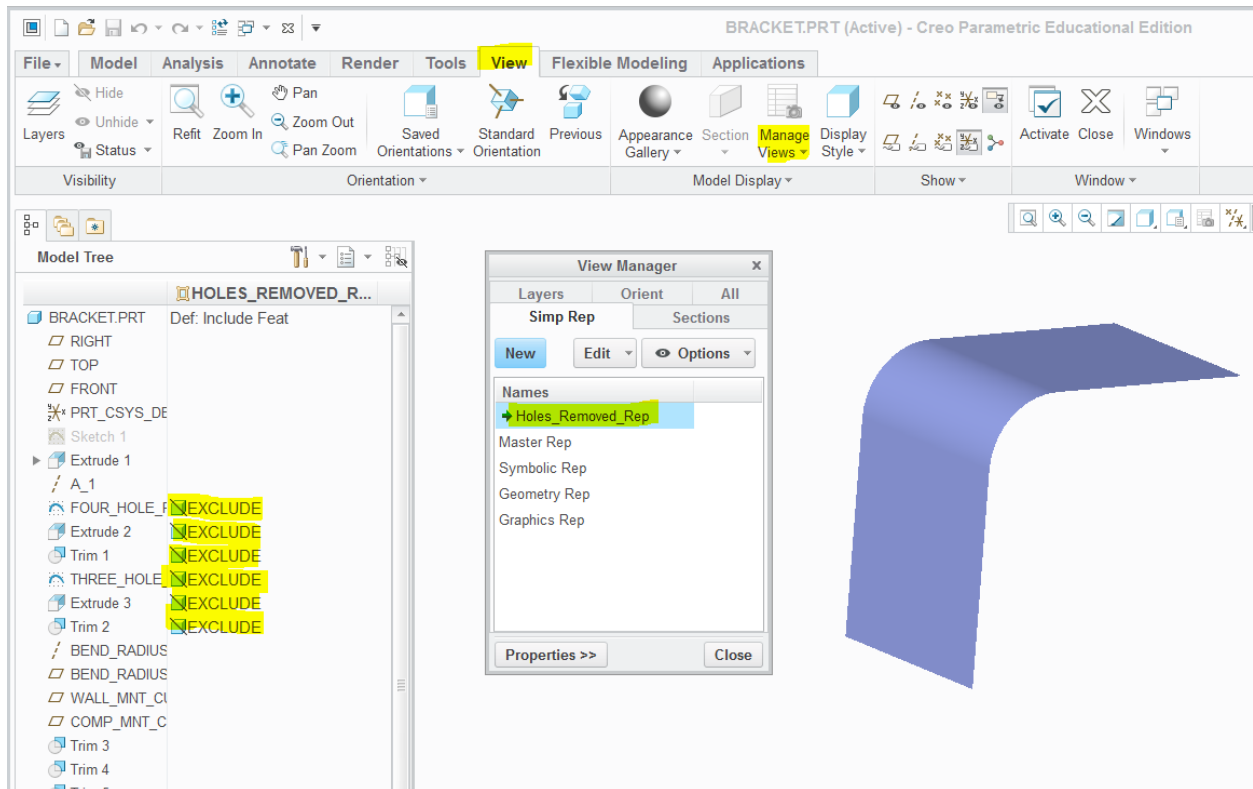In Creo, create a simplified rep via "View" "Manage Views".

Figure 9 – Creo View "Manage Views"

The CreateAssembly program will open the Creo model with the "Master Rep" unless a rep is specified in the Test Bench.
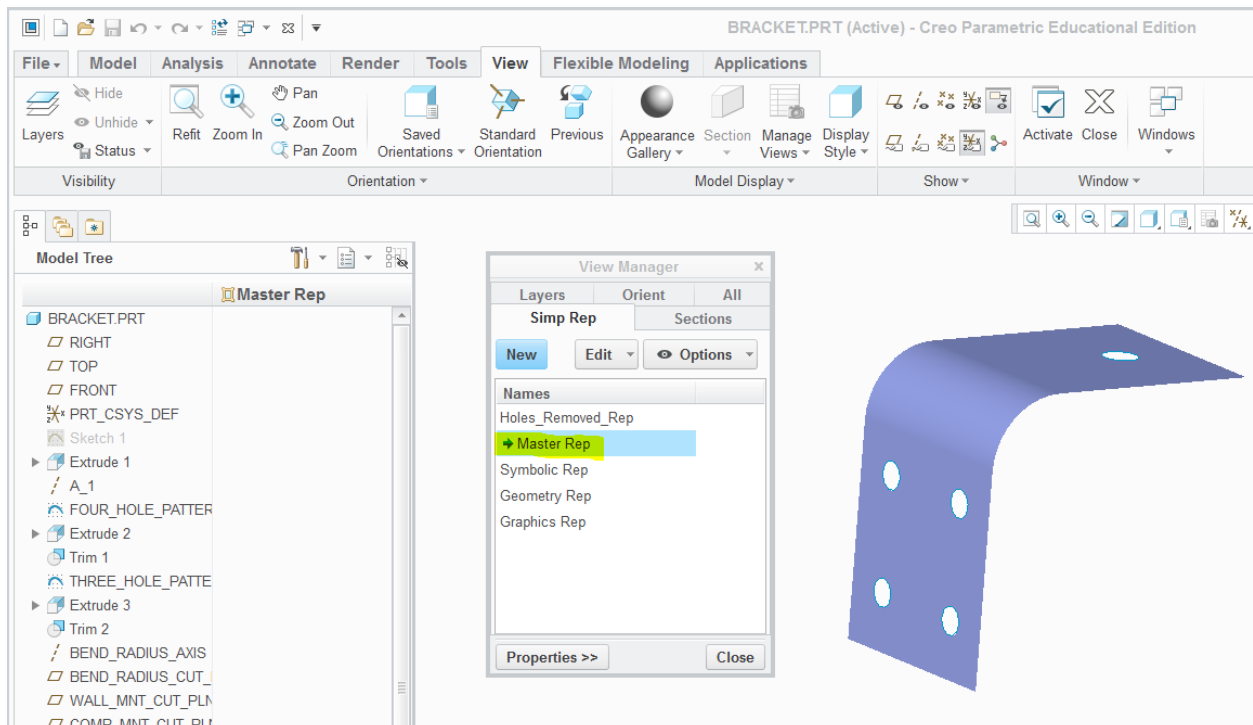
Figure 10 Creo "Master Rep" Setting

8.2 Test Bench

To specify a particular rep be used by the CreateAssembly program, create a parameter named DEFAULT_REP with the value set to the name of the rep in Creo as shown in Figure 11.
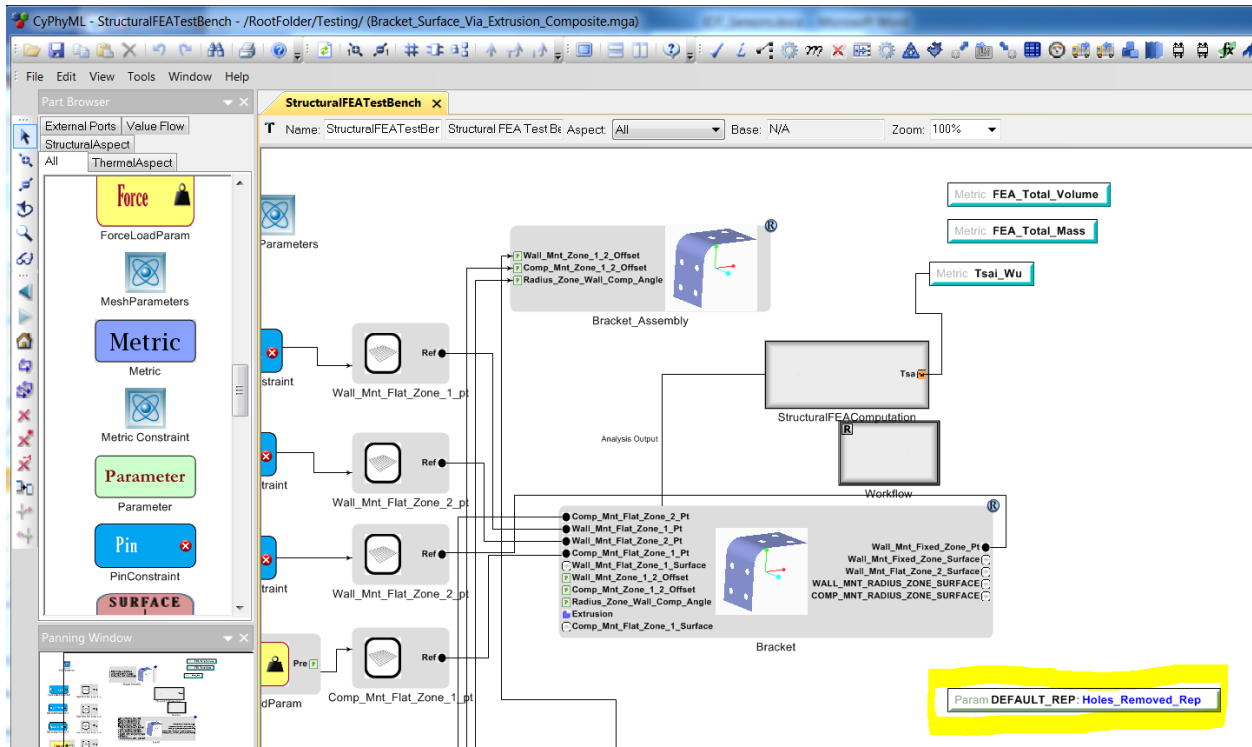
Figure 11 DEFAULT_REP Specified in a Test Bench

Note that DEFAULT_REP is case insensitive but the rep name (e.g. Holes_Removed_Rep) is case sensitive.

The DEFAULT_REP parameter applies to the entire Test Bench. Currently, with a Test Bench there is not a way to set the DEFAULT_REP parameter for a single component (i.e. part or sub-assembly). The CreateAssembly program supports setting this parameter per part/sub-assembly, but CyPhy2CAD_csharp currently does not.

The DEFAULT_REP parameter has been verified (via testing) to work with "Structural FEA Test Bench" and "Test Bench" types; however, it probably works with the other Test Bench types.

Figure 12 Test Bench Types that Have Been Verified to Work with DEFAULT_REP
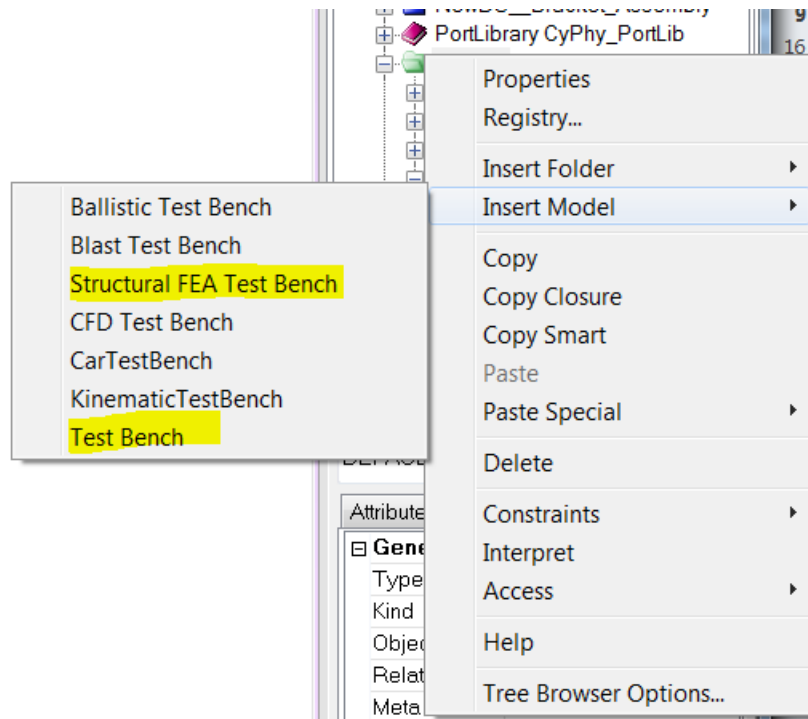
9 Export CAD Representations (e.g. STEP, Parasolid, etc.)

To request the export of a particular CAD representation, add to the Test Bench a parameter with the desired format.

Export_STEP_Ap203_Single_File
Export_STEP_Ap203_E2_Single_File
Export_STEP_Ap203_E2_Separate_Part_Files
Export_STEP_Ap214_Single_File
Export_STEP_Ap214_Separate_Part_Files
Export_Stereolithography_ASCII
Export_Stereolithography_Binary
Export_Inventor
Export_Parasolid
Export_DXF_2013

These strings are case insensitive, so the following would also work.

EXPORT_STEP_AP203_SINGLE_FILE
EXPORT_STEP_AP203_E2_SINGLE_FILE
EXPORT_STEP_AP203_E2_SEPARATE_PART_FILES
EXPORT_STEP_AP214_SINGLE_FILE

EXPORT_STEP_AP214_SEPARATE_PART_FILES
EXPORT_STEREOLITHOGRAPHY_ASCII
EXPORT_STEREOLITHOGRAPHY_BINARY
EXPORT_INVENTOR
EXPORT_PARASOLID
EXPORT_DXF_2013

Notes:

1) The above strings are case insensitive.
2) This works with any Test Bench type.
3) Export_DXF_2013 only exports solid models. Surface models will be empty.

Figure 13 shows an example Test Bench requesting the output of all supported CAD formats.

Note – If an exported CAD format is needed for a particular type of Test Bench to function, then there is no need to explicitly request the export of that format. For example, a "Structural FEA Test Bench" with the settings for PATRAN_NASTRN will automatically output Parasolid files.

Figure 13 – Test Bench Requesting the Export of All Supported CAD Formats

Appendix A – Links

http://www.iccm-central.org/

http://www.iccm-central.org/Proceedings/ICCM16proceedings/contents/pdf/WedI/WeIM1-05sp_kaddouras225256p.pdf

NASA, Structural Design and Test Factors of Safety for Spaceflight Hardware
https://standards.nasa.gov/standard/nasa/nasa-std-5001

10 Parameters

When parameters are entered in the CyPhy model, they are typically used to change dimensions in the CAD Model. For example, to change the length of a part or change where a part is positioned in an assembly.

If a unit is assigned to a parameter in the CyPhy model, then CADAssembly.xml will have a unit for that parameter. A CADAssembly.xml example follows:

e.g. <CADParameter Name="Wall_Attachment_Plate_Length" Type="Float" Value="200" _id="id142">
<Units Value="mm" _id="id143" />

Only distance (e.g. in, mm, m ...) or angle dimensions (deg, rad) are supported. If CADAssembly.xml has another type dimension (e.g. lbm/in^3) then this would be considered an error. This policy is reasonable because up until now, parameters have only varied distance and angle dimensions. Varying density might be needed in the future for the case of assigning mass properties, but when that is needed then the CreateAssembly program will need to be modified.

if (CADAssembly.xml does not have a unit set for the parameter)
e.g. <CADParameter Name="Wall_Attachment_Plate_Length" Type="Float" Value="200" _id="id142">
<Units Value="" _id="id143" />
then
   do no unit conversions
   assign the parameter value in CADAssembly.xml to the parameter in the CAD Model
else // units for the parameter are in CADAssembl.xml
   if (units are assigned to the parameter in the CAD Model)
   then
      Convert between the units in CADAssembly.xml to the units for the parameter in the CAD model
      Assign the converted parameter to the parameter in the CAD model
   else // no units on the parameter in the CAD model
      if (units are assigned to the CAD Model)
      then
            Convert between the units in CADAssembly.xml to the units for the CAD model
            Assign the converted parameter to the parameter in the CAD Model
       else
            No information to do the conversion
            In Creo this would not happen. Parts/assemblies always have units by default
            If another CAD system will allow you to create a part/assembly without units, then this
             would be a problem
             Assign the parameter value in CADAssembly.xml to the parameter in the CAD Model

Appendix B – Exporting Wavefront Formatted File Via Toolkit

Document - CS249910

Document Information

Type	Article
Date Created	03-Nov-2016
Date Last Updated	03-Nov-2016

Document Actions

- Refresh Data
- Subscribe to this Document
- Share by Email
- Copy URL

Document Details

Title	Is it possible to Export in obj format (WaveFront) using Toolkit? Is it acheduled in future release of Toolkit?
Description	Is it possible to Export in obj format (WaveFront) using Toolkit? Is it acheduled in future release of Toolkit?
Applies To	• Creo Parametric 2.0 to 3.0
Cause	
Resolution	• Not part of Creo Toolkit functionalities. • As a workaround, use some other file format such as STEP or STL by refering to **ProIntf3DFileWrite()** • File a product enhancement idea by following the instructions.

Was this information helpful?

No   Yes

SPR Information

Related Documents