

Finite Element Analysis (FEA)

User Tutorial for the Finite Element Analysis (FEA) Test Bench and Tool

May 2, 2014

1.0 Purpose

The purpose of the Finite Element Analysis (FEA) Test Bench and Tool is to provide the designer with knowledge of how the applied stresses and boundary conditions will affect the material or design. This test bench calculates the maximum von Mises stress and the factor of safety of the design. The default units in the test bench are N, MPa, mm, and kg. Creo geometry files are in mmKs.

2.0 Procedures

The instructions in this manual assume that the user has installed the latest version of GME and has access to Creo, either locally or via the remote server.

2.1 Installation

Initial installation of this test bench will be provided with the installation of the CyPhy tool suite. Future editions of the tool may be packaged as a standalone or combined test bench installation package.

2.2 Tool

The FEA Test Bench is the test bench in GME that the designer uses to interface the Abaqus software, which is a software suite for finite element analysis and computer-aided engineering. Abaqus is running in the VehicleFORGE servers which accepts a design, simulates it by using numerical FEA procedures, and returns an accurate simulation result.

FEA test bench performs a numerical simulation procedure to determine the structural performance of the design under different loading and boundary conditions. The simulation will determine the maximum von Mises stress and the factor of safety of the design by taking into account the input given by the user. Loads are applied to components in the assembly and are specified in relation to the assembly coordinate system instead of in relation to each component's coordinate system.

3.0 Requirements Tested

- **Maximum von Mises stress (MPa):** von Mises yield criterion is an equation that gives the equivalent stress at a point in a body acted upon normal and shear stress in all 3 directions.
- **Factor of Safety (unitless):** Factor of safety is a figure used in structural applications that provides a design margin over the theoretical design capacity which basically maximum stress divided by allowable stress.

4.0 Operation

The design is assembled into a 3D CAD representation, including the customization / generation of any parameterized components.. Creo is used for the 3D CAD software and all of the models should be created in this software. The user should be familiar with Creo, its basics functions and how to operate the software. Input data taken from the user is also assembled together for a FEA simulation by Abaqus FEA software tool suite. The whole information is packaged up and sent via remote server for the FEA simulation.

5.0 Test Bench Structure

This test bench contains a system under test that is to be assembled and analyzed for its structural performance.

Step 1

The general view of the model that will be tested can be seen in (Fig. 1). The top plate will be called Plate-1 from now on, and the plate at the bottom will be called Plate-2. Two plates are connected together with perfect joint (no slip between the surfaces). Bolts are not included in the model, but with the perfect joint assumption bolt connection has been modeled. FEA test bench connects all parts with perfect joints as default.

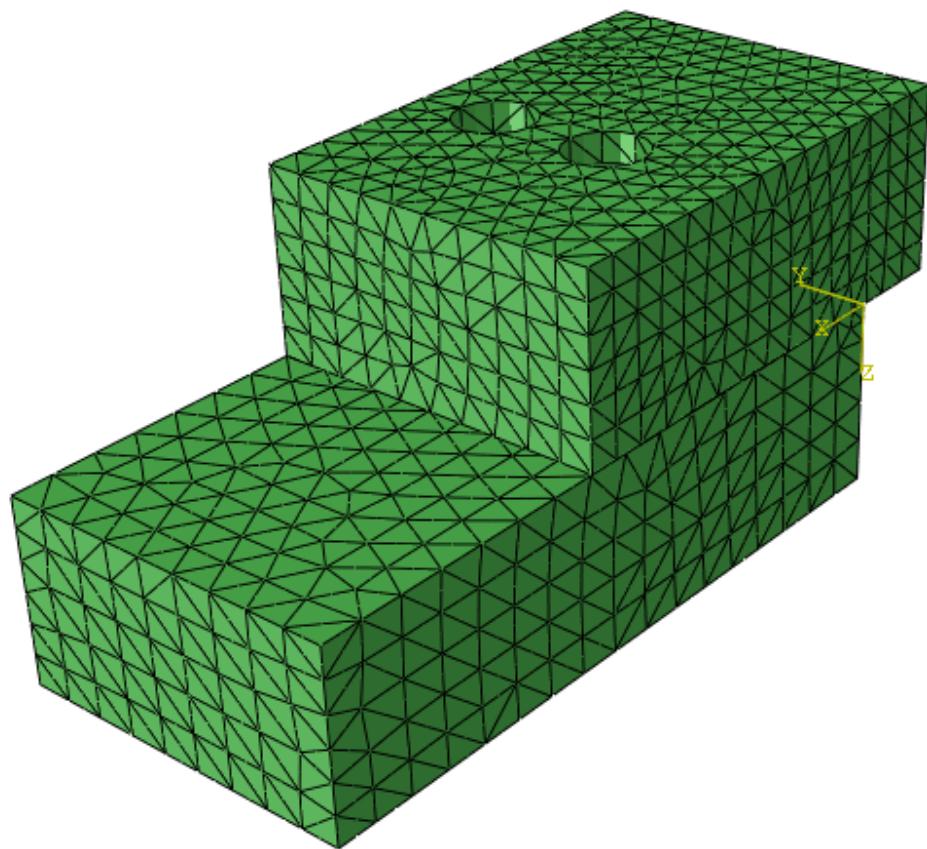


Figure 1

Step 2

On Plate-1, the points called POINT 1 WALL ATTACHMENT, POINT 2 WALL ATTACHMENT, POINT 3 WALL ATTACHMENT and POINT 4 WALL ATTACHMENT, which are critical to this tutorial, are shown in (Fig. 2).

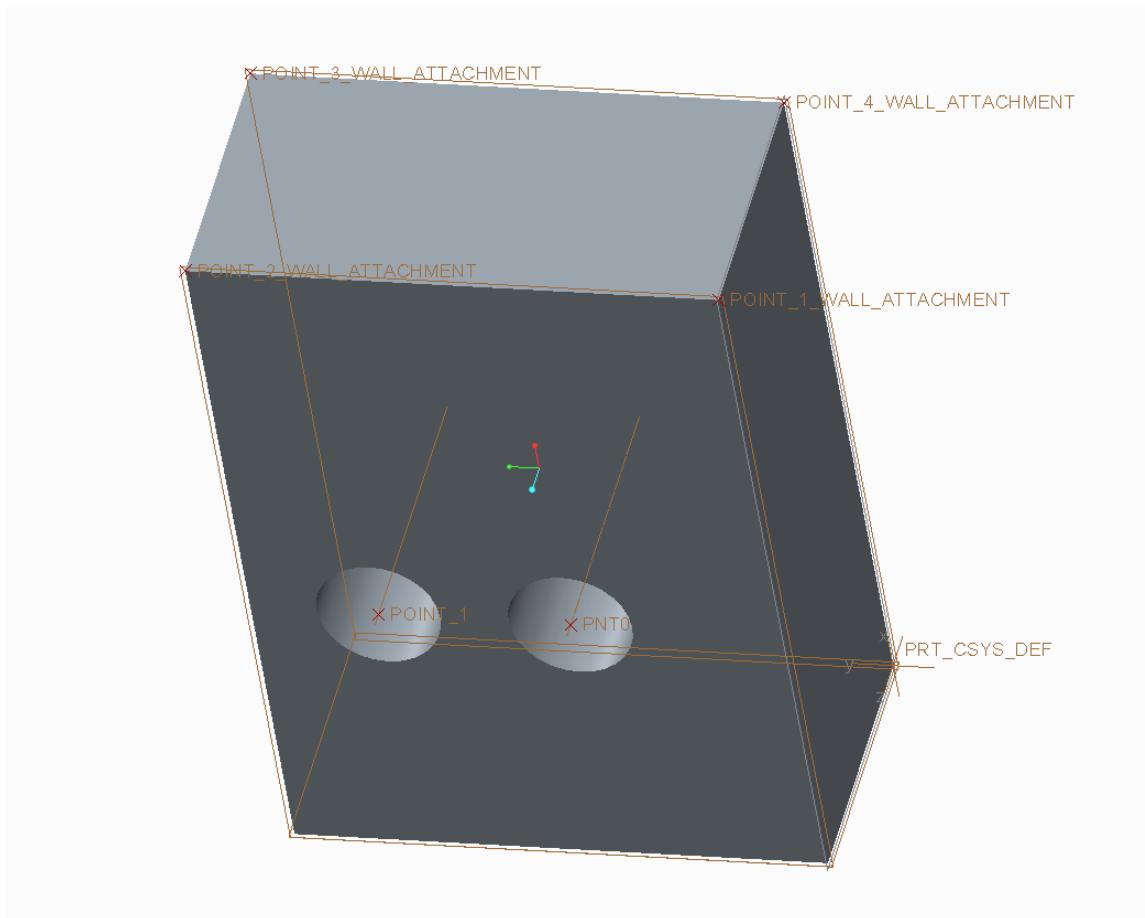


Figure 2

Step 3

On Plate-2, the points called POINT 1 END LOAD which will be mentioned in this tutorial, is shown in (Fig. 3).

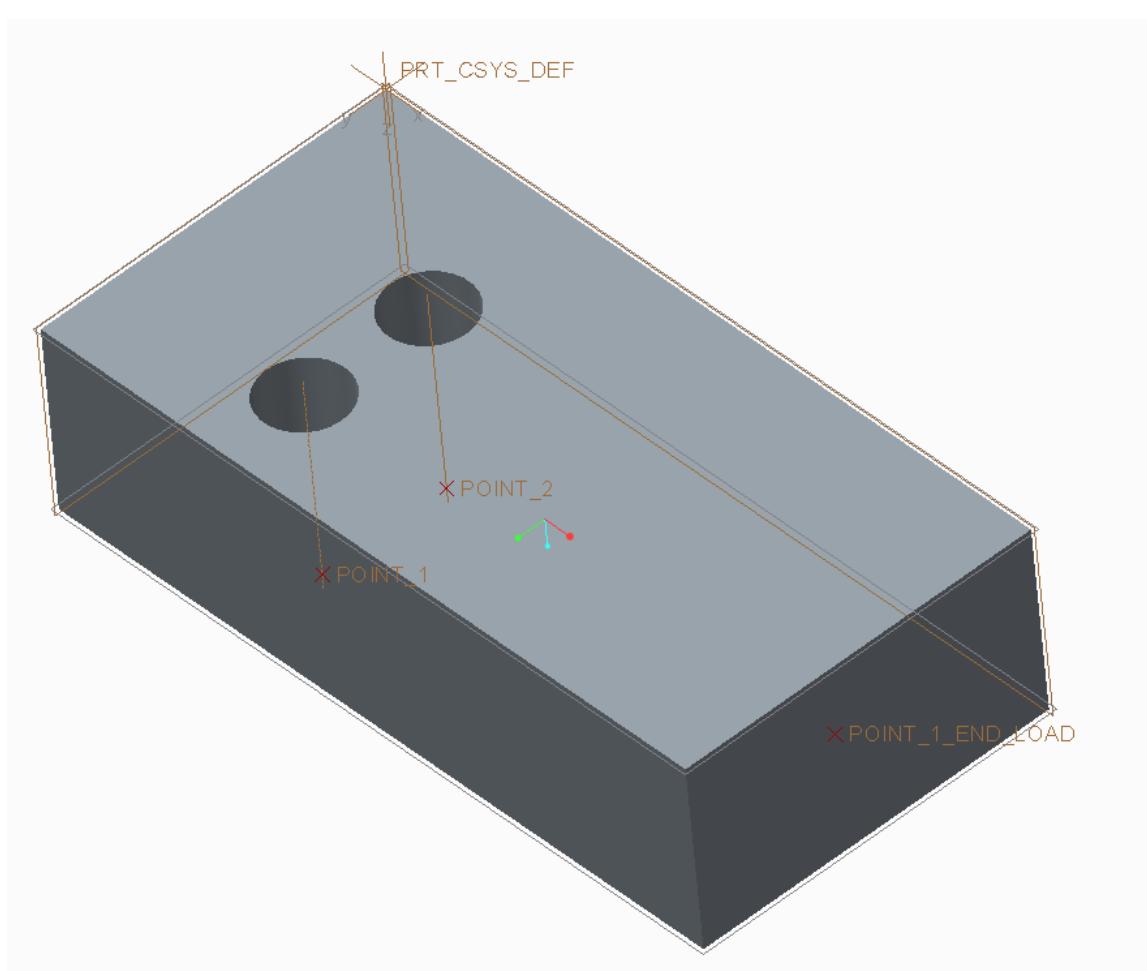


Figure 3

Step 4

Plate-1 will be fixed to translate in x, y and z directions in the surface between the points POINT 1 WALL ATTACHMENT, POINT 2 WALL ATTACHMENT, POINT 3 WALL ATTACHMENT and POINT 4 WALL ATTACHMENT. (Fig. 4)

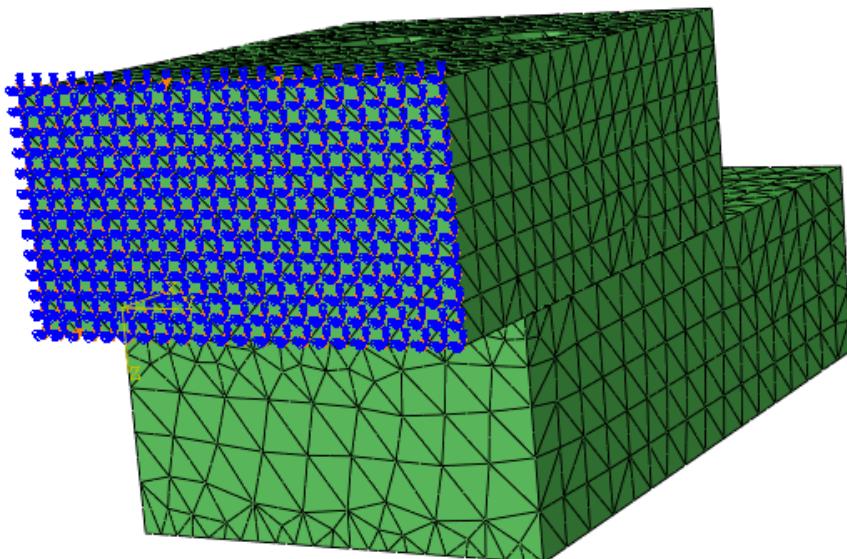


Figure 4

Step 5

1000N of force will be applied to Plate-2 in x-direction, on the surface of points POINT 1 END LOAD. (Fig. 5)

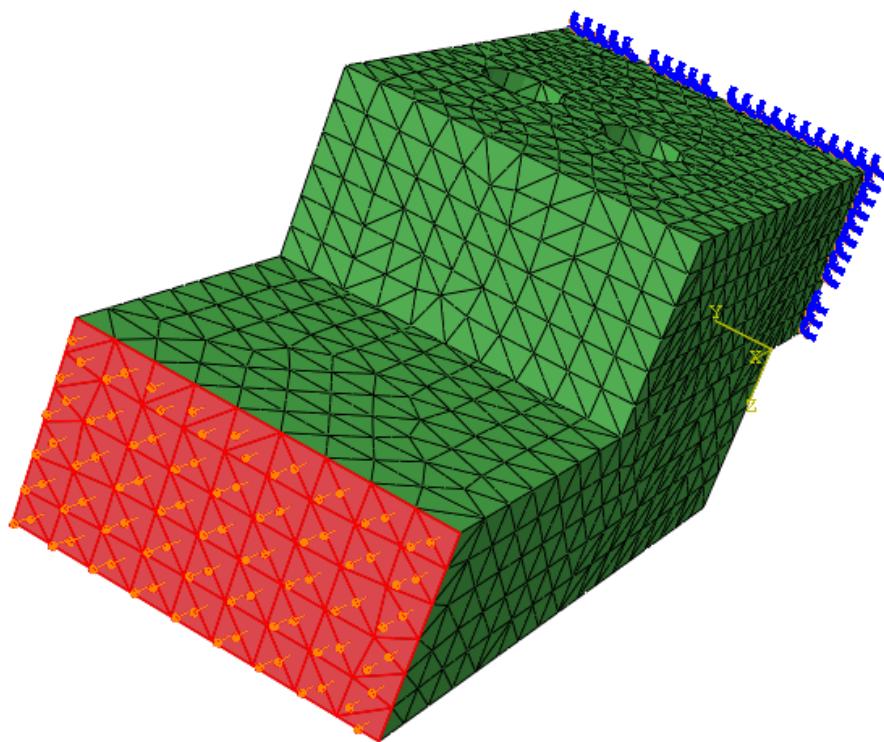


Figure 5

Step 6

The logic of the FEA test bench is to apply the boundary conditions and loads to the parts on the mesh nodes that are on the surface defined by the points referenced in the CAD program manually(e.g. Creo, etc.). Using this functionality, for loads and boundary conditions to be applied, the reference points must be encompass the entire surface that it will be applied to. Therefore, for the loading of this model, Plate-1 points: POINT 1 WALL ATTACHMENT, POINT 2 WALL ATTACHMENT, POINT 3 WALL ATTACHMENT and POINT 4 WALL ATTACHMENT define the entire rectangular surface that the boundary condition is applied on. Plate-2 points: POINT 1 END LOAD is defined on the rectangular surface that the load is applied on using the face construct functionality.

Step 7

All parts submitted to the FEA tool must have a valid META Material Library name assigned. The material assignment is made in Creo by opening the part and selecting File - Prepare - Model Properties - Material - Change. If the part is opened via META Link, Creo is automatically pointed to the META Material Library making material selection seamless. If the part is opened directly with Creo, follow instructions in the META Material Library User Manual to point Creo to the META Material Library for material selection. When assigning a material to a part, a best practice would include verifying the material properties are in mmKs units and removing unused materials from the “Materials in Model” column.

Defining Geometry

Step 8

Next, we will define the geometry of the surfaces that the boundary conditions will be applied to. In the GME Browser, right click on the Geometry folder, select insert model then select the geometry. In this tutorial, Polygon is used as an example. Then name the new geometry. (Fig. 6)

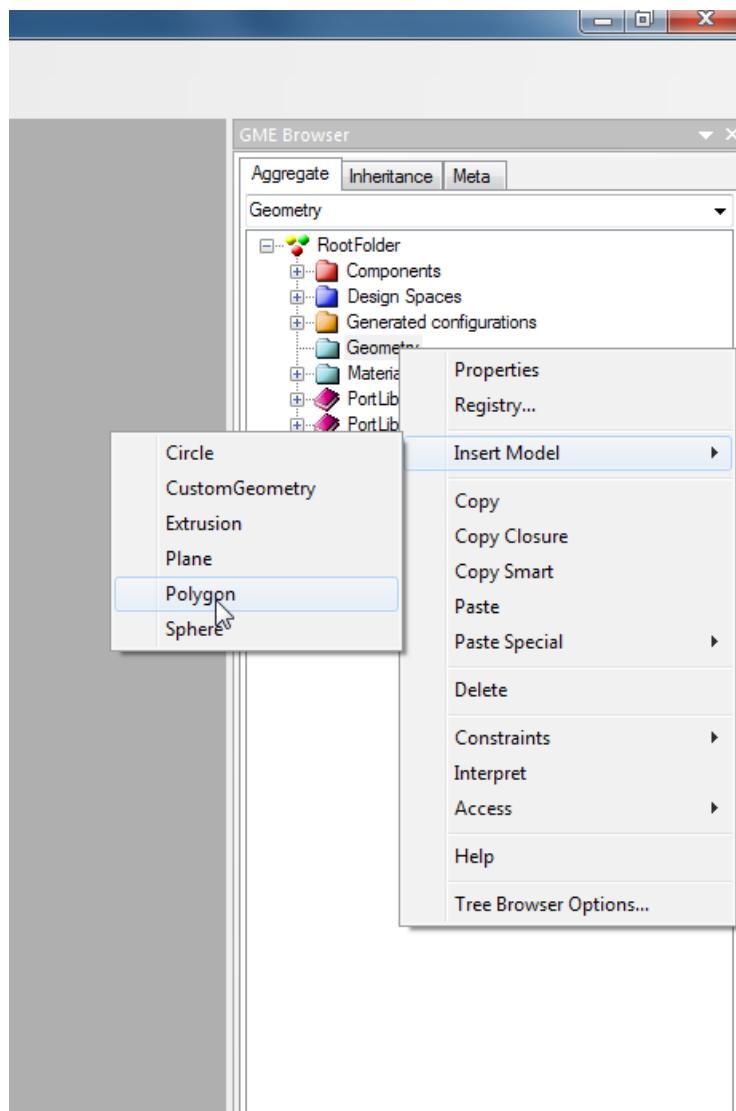


Figure 6

Step 9

Double click on the created geometry. Then drag and drop Ordinal Point objects into the geometry as much as you need and rename them. (Fig. 7)

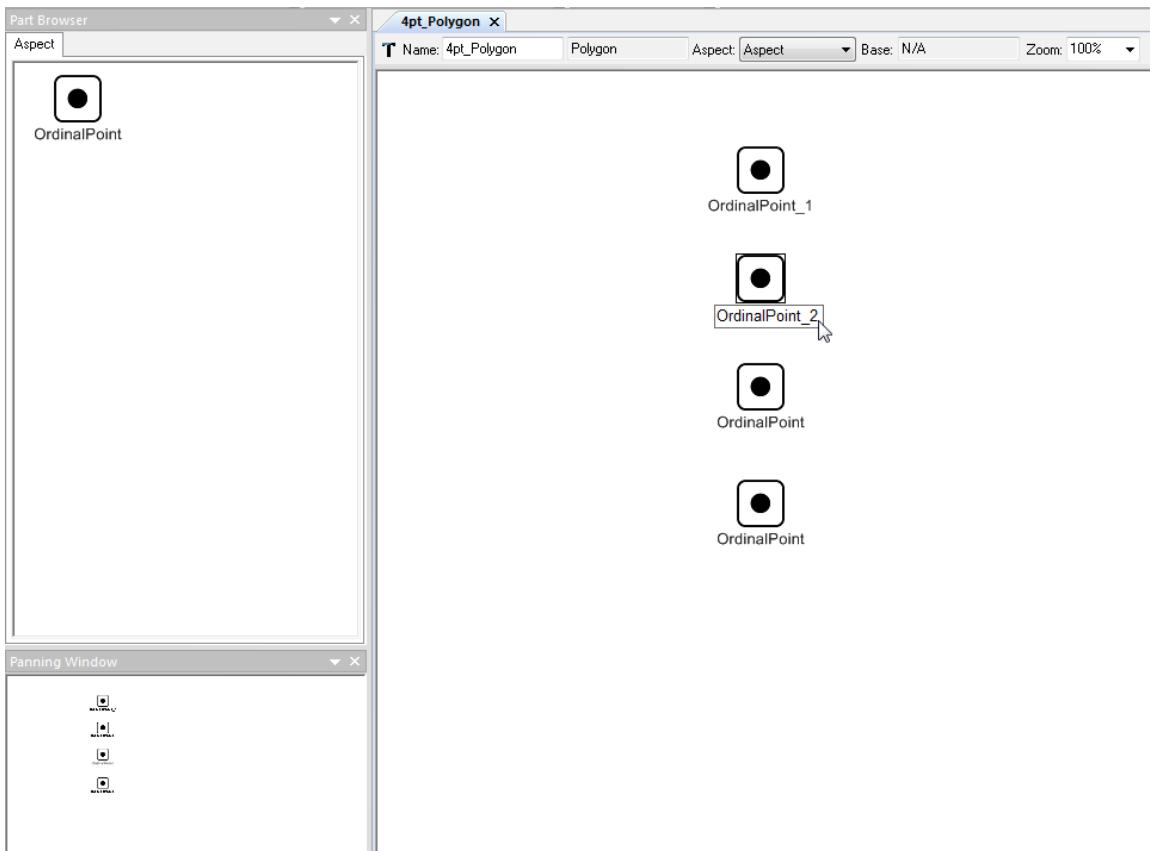


Figure 7

Step 10

Right click on every ordinal point and select Attributes. Then give an ordinal position for each ordinal point (In this tutorial, 1 for OrdinalPoint 1, 2 for OrdinalPoint 2 and so on.) (Fig. 8)

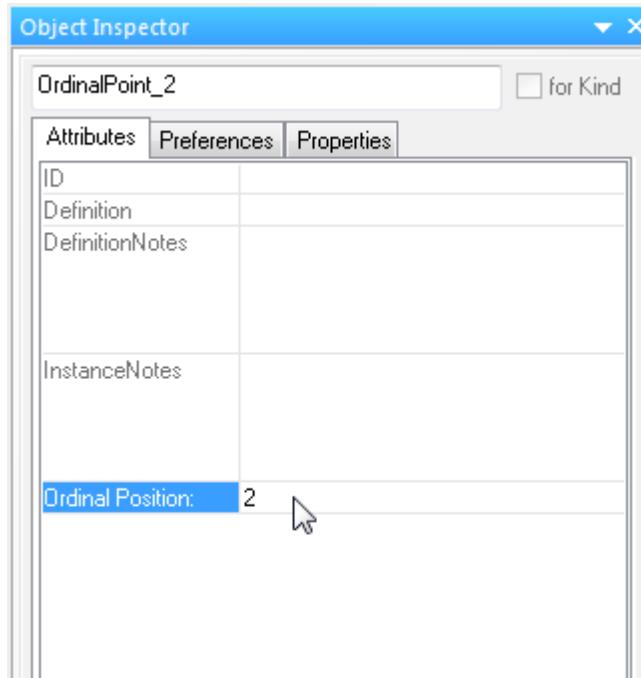


Figure 8

Step 11

To create the geometry for the load, select Face from the geometry folder (Fig. 9). Double click the created geometry and drag and drop a reference point into the geometry (Fig. 10).

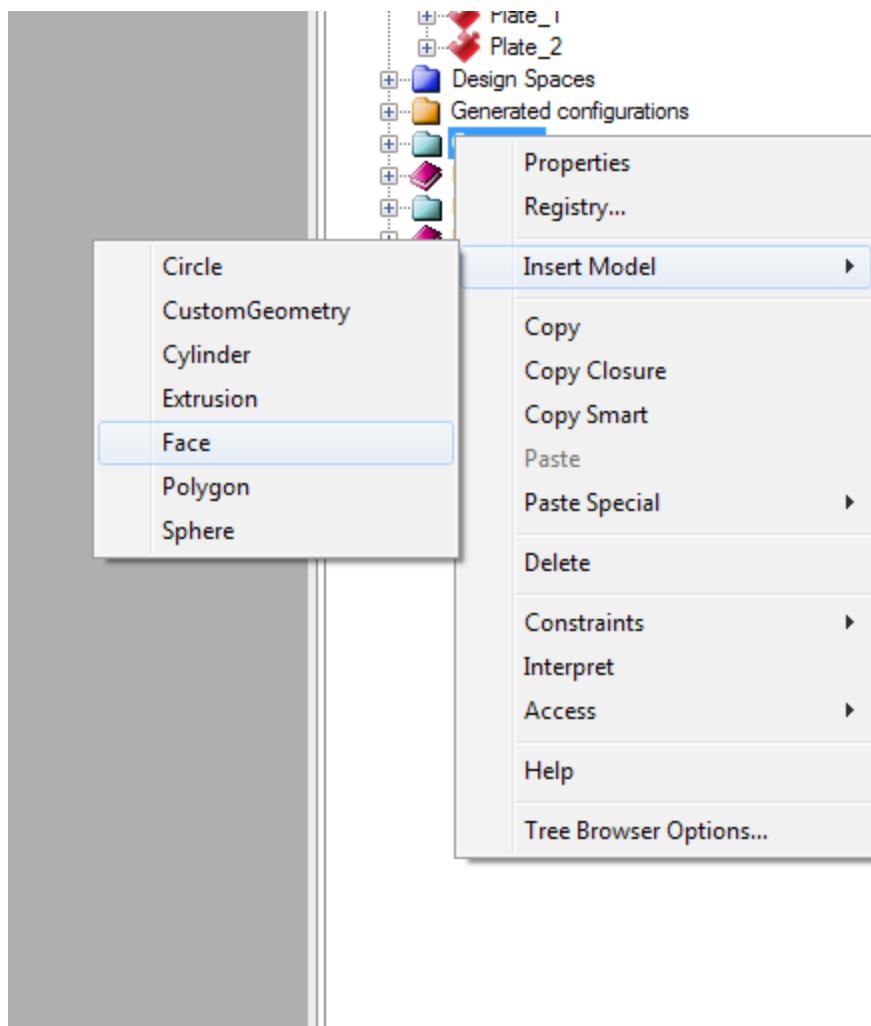


Figure 9

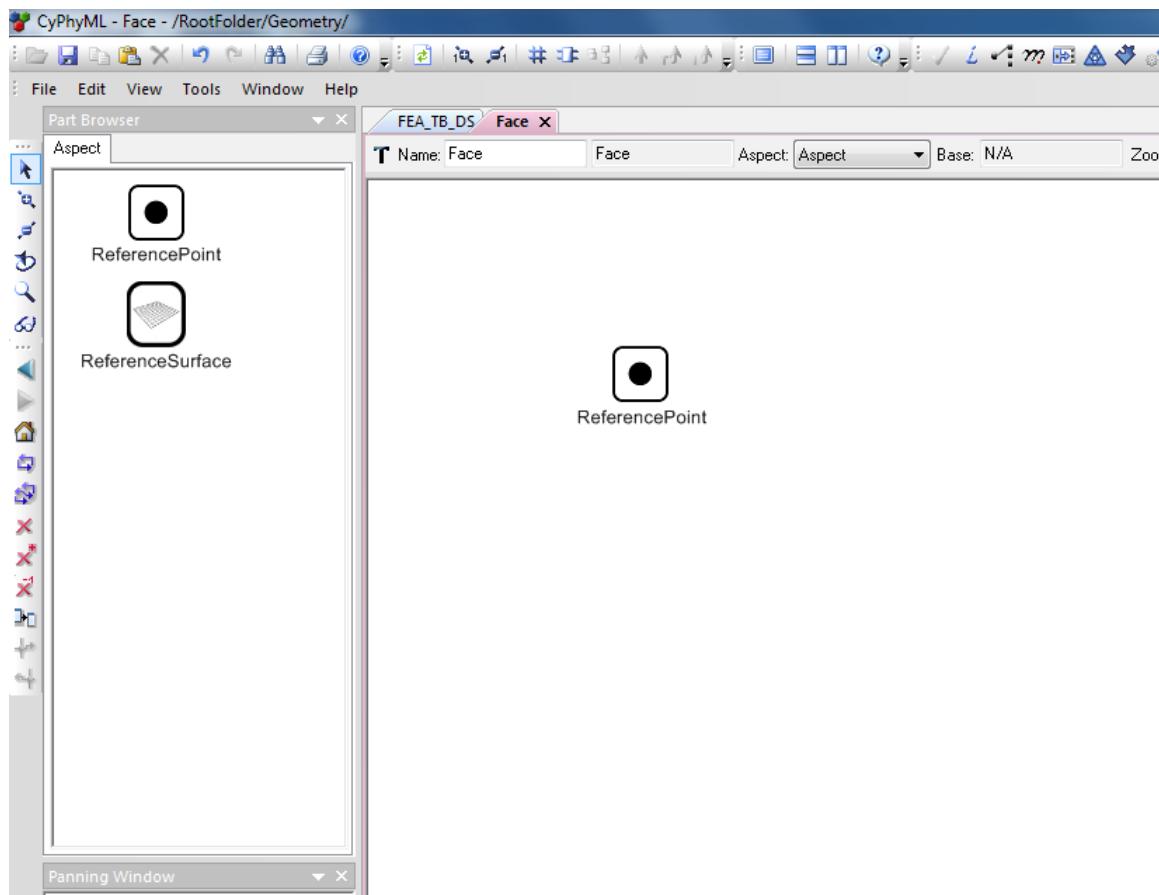


Figure 10

Creating the test bench

Step 12

We will now create the FEA test bench. In the GME Browser, right click on the Testing folder, insert a new Structural FEA Test Bench, and name it. (Fig. 11)

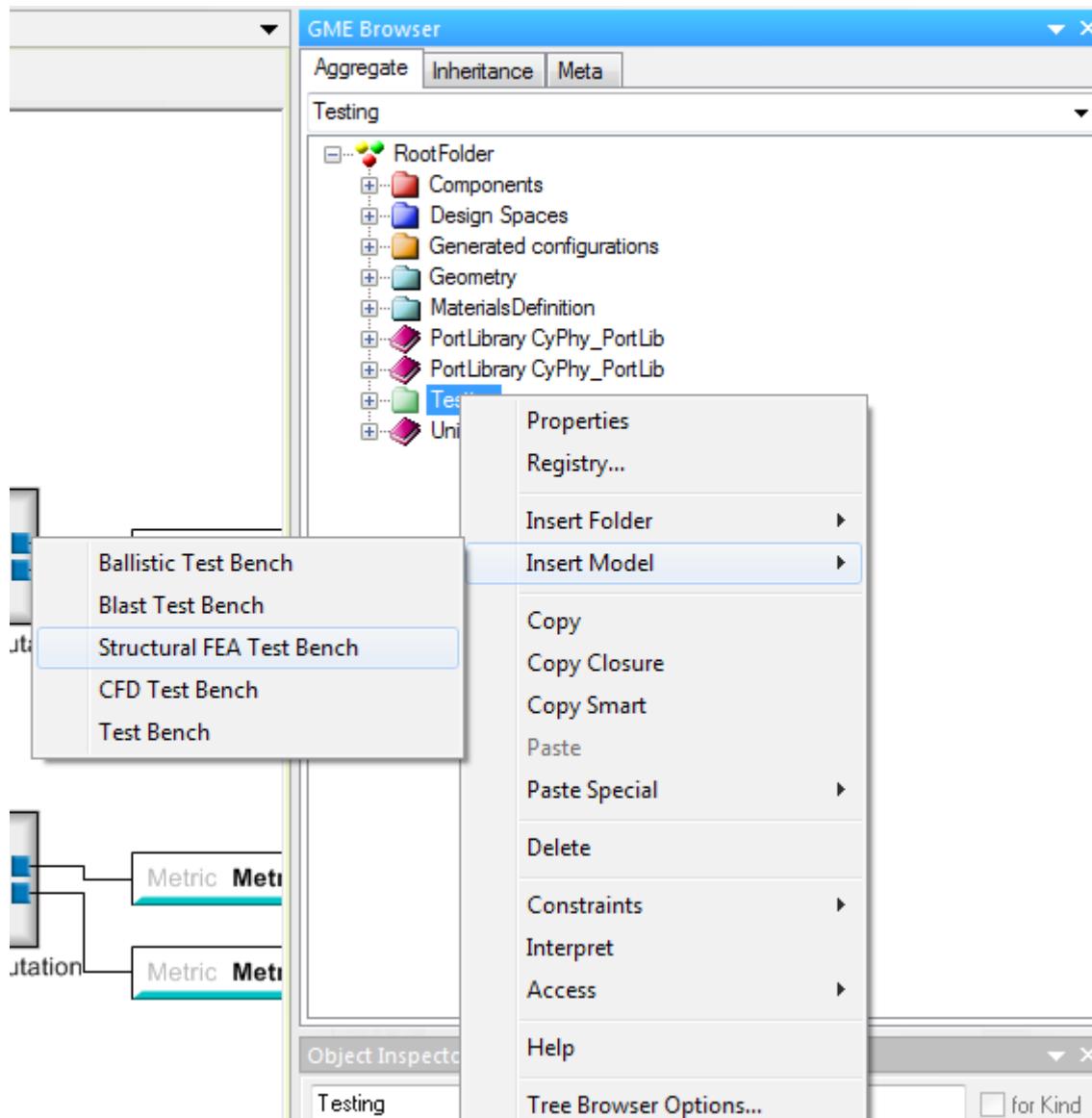


Figure 11

Step 13

Now we will add our CAD parts to the test bench. In the GME Browser, go to the Components folder. Right click on the part and copy. (Fig. 12)

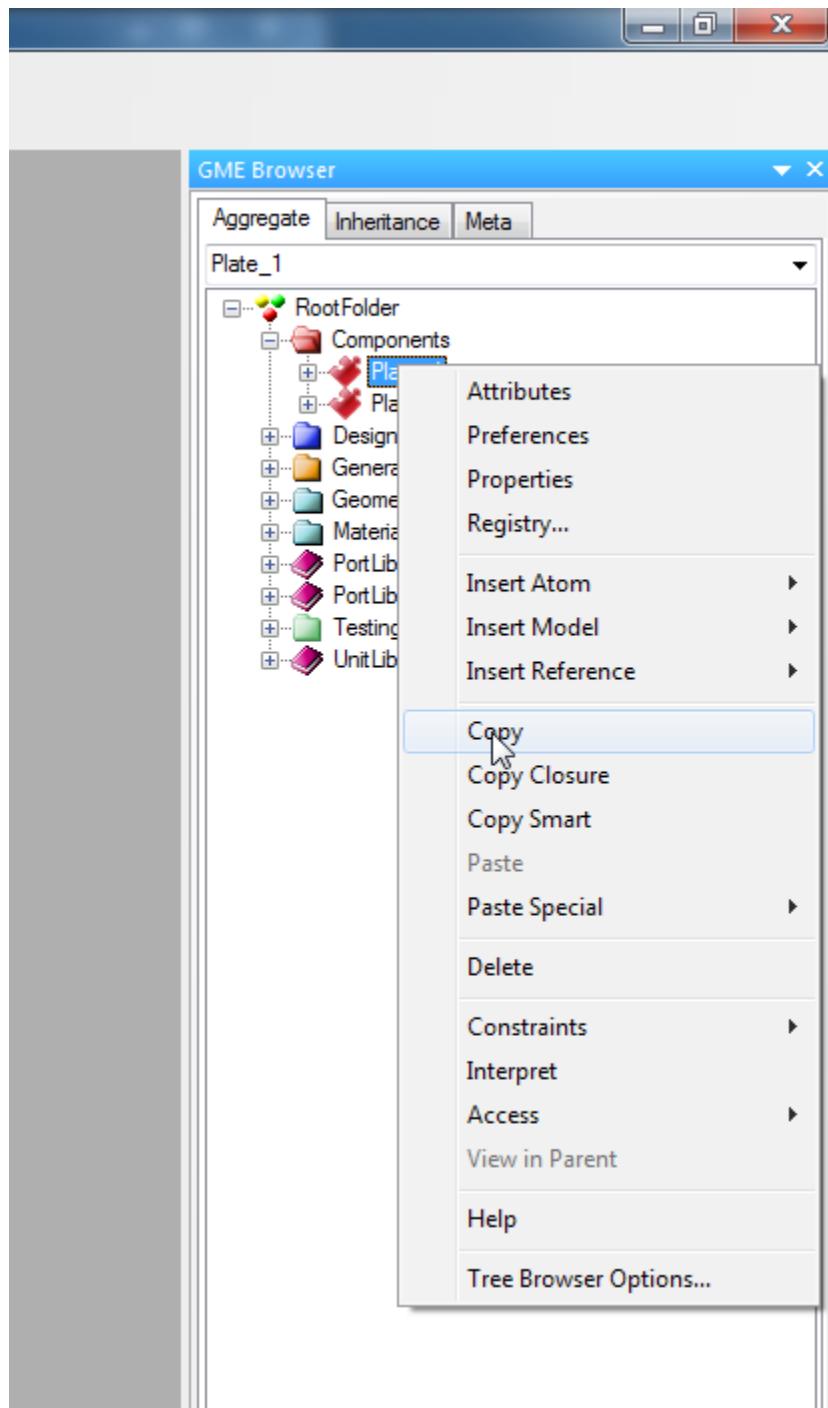


Figure 12

Step 14

Then double-click on the created FEA testbench under the Testing folder. In the test bench, right-click and select Paste Special and as Reference. (Fig. 13)

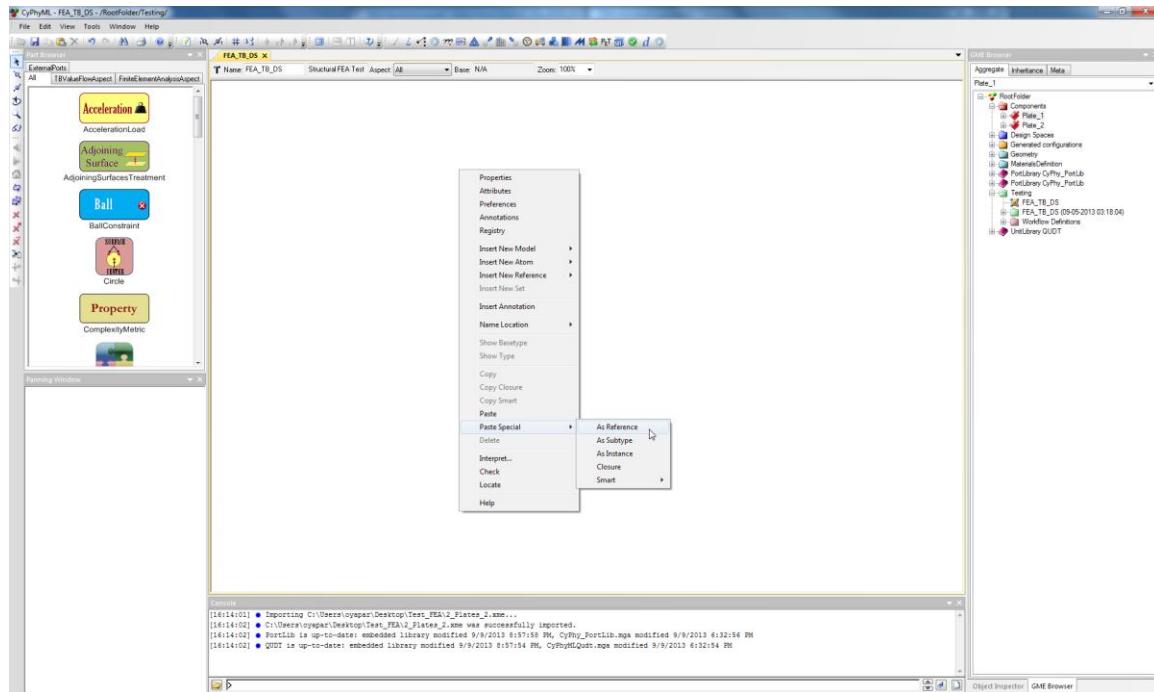


Figure 13

Step 15

In the screen that has been popped up, select TestInjectionPoint. Repeat that step for each part. (Fig. 14)

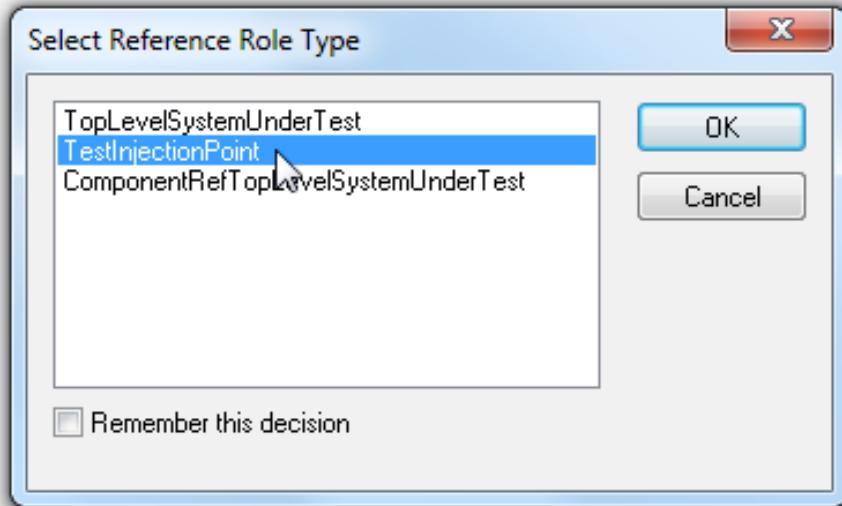


Figure 14

Step 16

Now we will define the surfaces that the boundary conditions will be applied to. In the GME Browser, go to the Geometry folder. Right click on the geometry and copy. (Fig. 15)

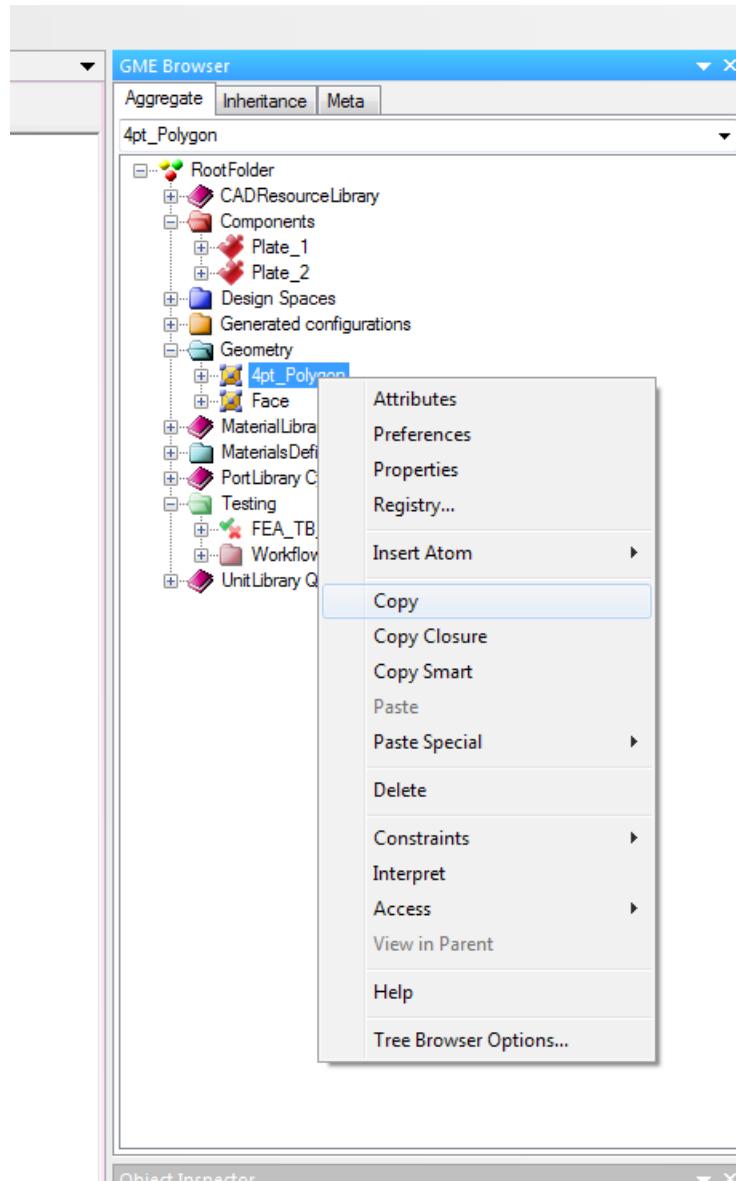


Figure 15

Step 17

Then double-click on the created FEA testbench under the Testing folder. In the test bench, right-click and select Paste Special and Paste. Repeat that step for each part. (Fig. 16)

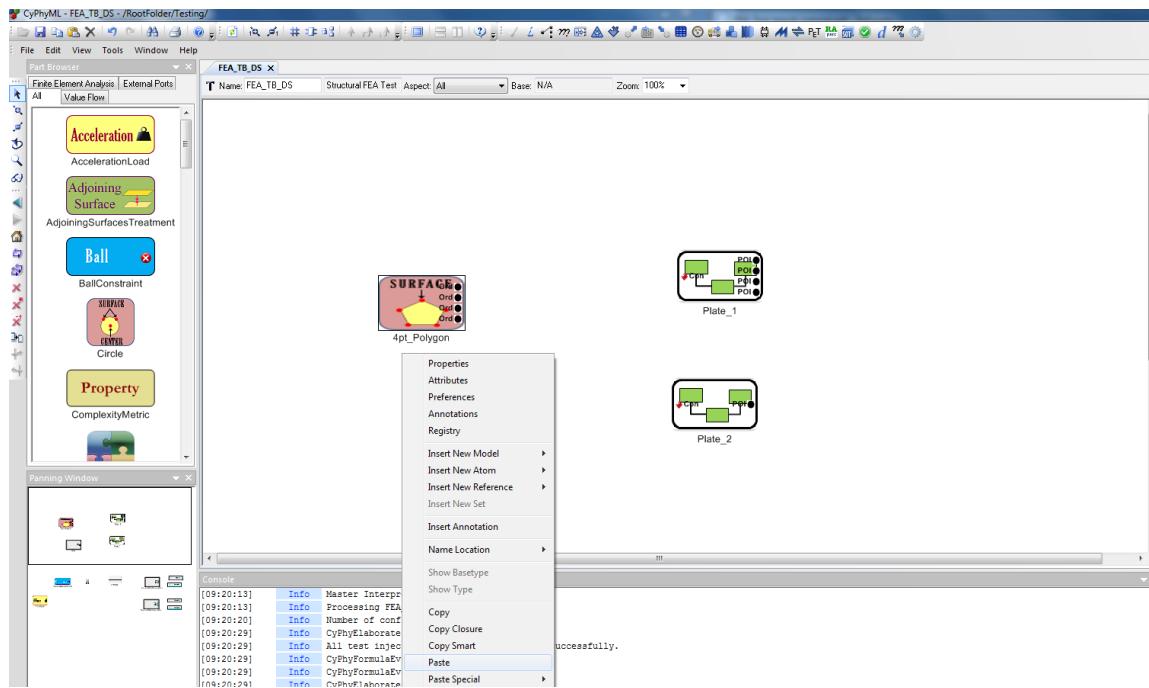


Figure 16

Step 18

Press (Ctrl+2) to get into the connection mode and connect the parts that we have in the test bench with the geometry references. That will tell CyPhy which points on the CAD parts are forming the surfaces. Connect each numbered point on the test injection component (i.e. Plate_1) with the corresponding numbered ordinal point on the polygon construction. To view each point number, click on point name on the construct/component. When you are done, press (Ctrl+1) to go back to the edit mode. (Fig. 17)

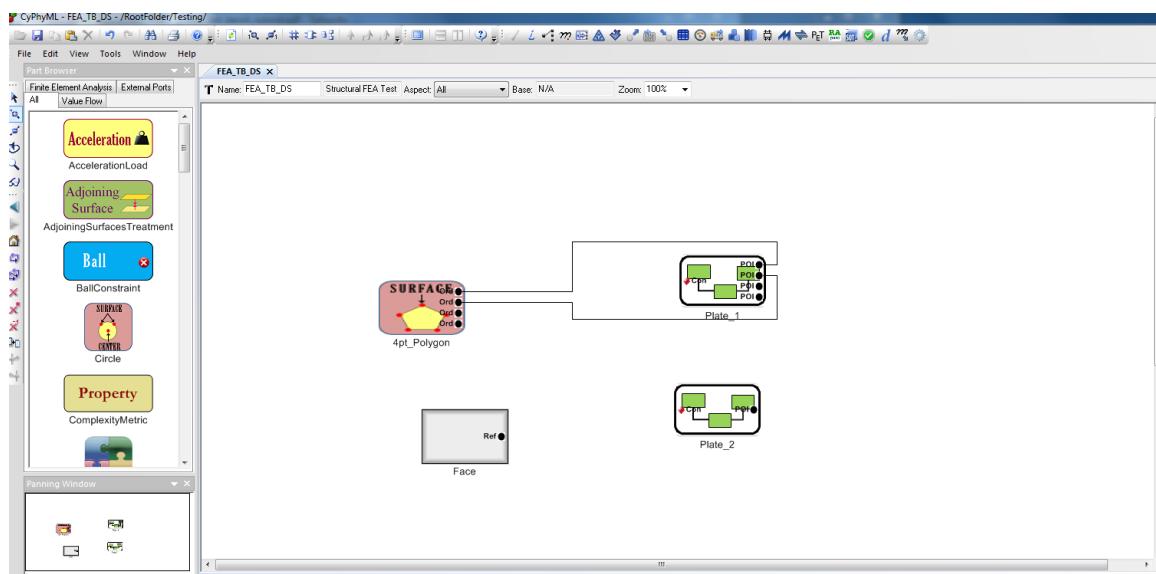


Figure 17

Step 19

For defining displacement constraints, drag and drop the Displacement Constraint part from the Part Browser into the test bench. (Fig. 18)

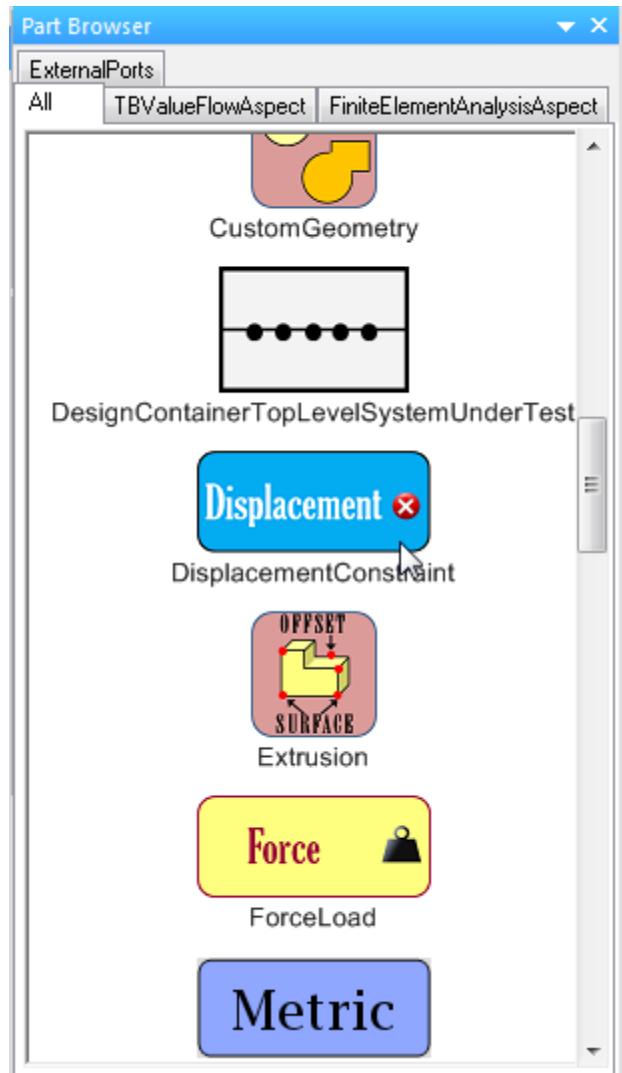


Figure 18

Step 20

Press (Ctrl+2) to get into the connection mode and connect the DisplacementConstraint with the geometry reference. That will tell CyPhy that displacement constraints will be applied to the corresponding surface. When you are done, Press (Ctrl+1) to go back to the edit mode. (Fig. 19)

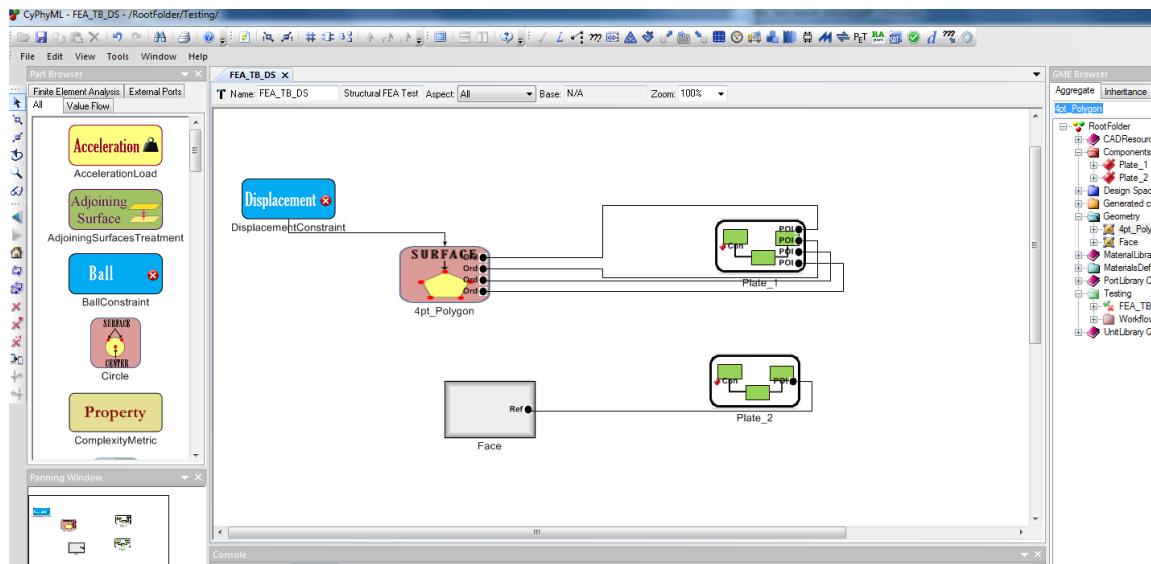


Figure 19

Step 21

Now we will get into the Displacement Constraint. Double-click on the Displacement Constraint. For defining translation and rotation, drag and drop Translation and Rotation parts into the Displacement Constraint. (Fig. 20)

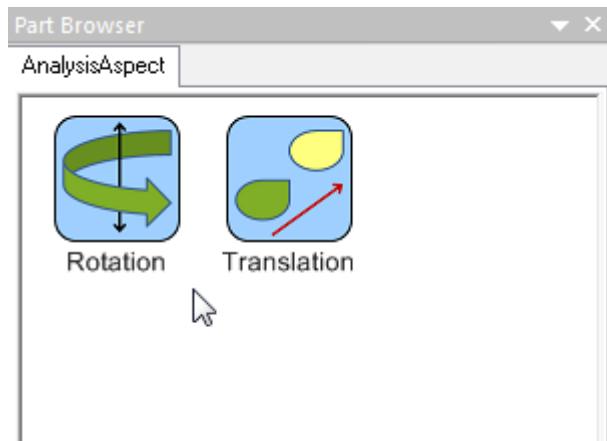


Figure 20

Step 22

Right-click on translation or rotation and pick Attributes to define the displacement and translation conditions in each direction (x, y, z). Pick FIXED for every direction for both Translation and Rotation (Fig. 21)

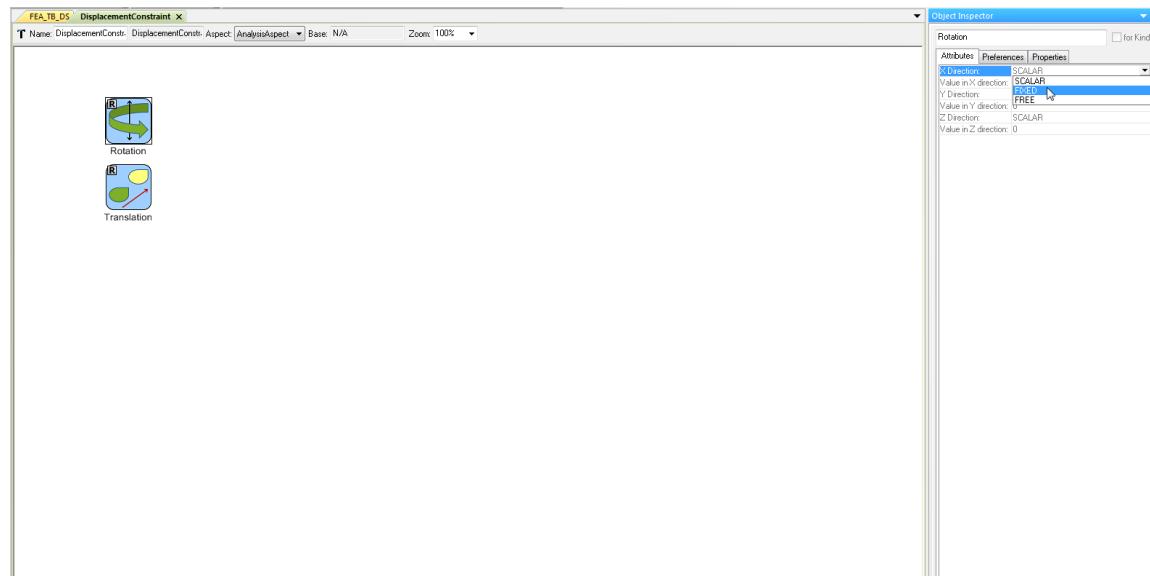


Figure 21

Step 23

For defining forces and moments, drag and drop the ForceLoad part from the Part Browser into the test bench. (Fig. 22)

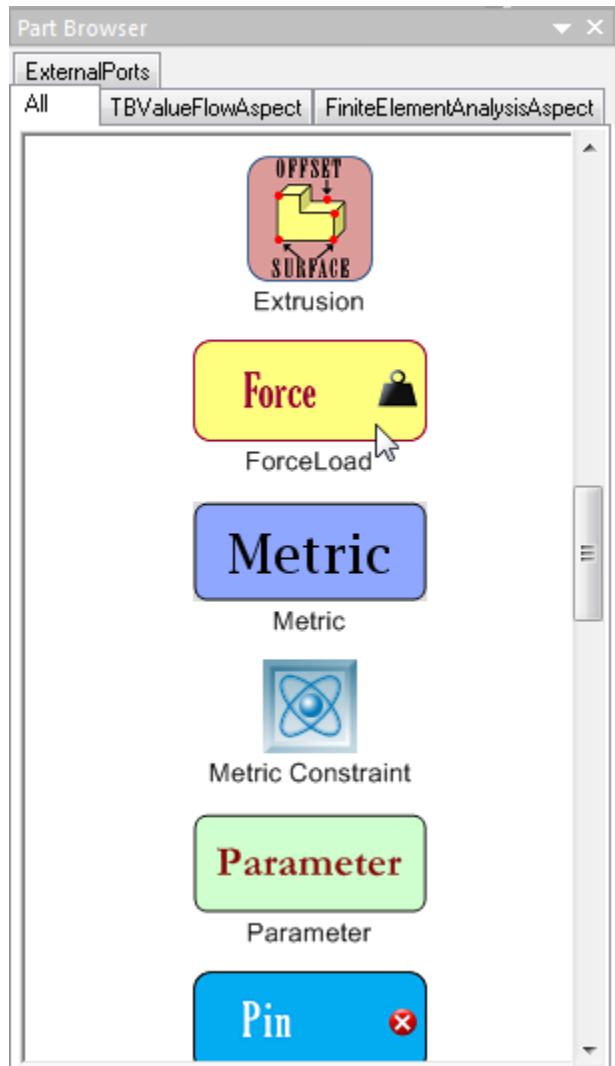


Figure 22

Step 24

Press (Ctrl+2) to get into the connection mode and connect the ForceLoad with the geometry reference. That will tell CyPhy that loads will be applied to the corresponding surface. When you are done, Press (Ctrl+1) to go back to the edit mode. (Fig. 23)

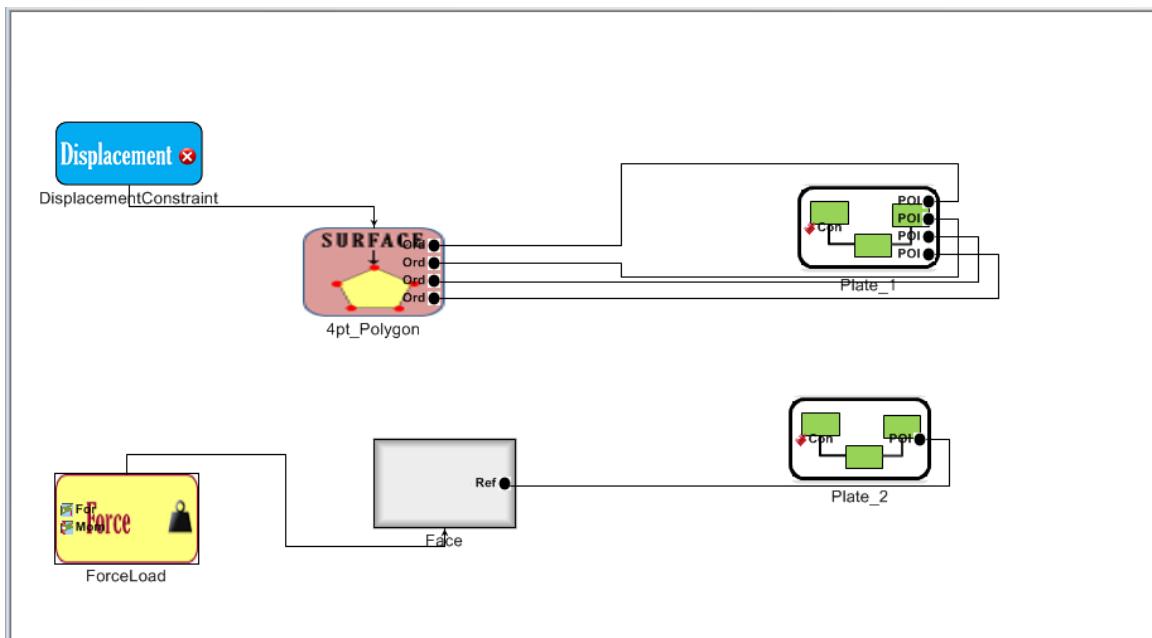


Figure 23

Step 25

Now we will get into the ForceLoad. Double-click on the ForceLoad. For defining force and moment, drag and drop Force and Moment parts into the ForceLoad. (Fig. 24)

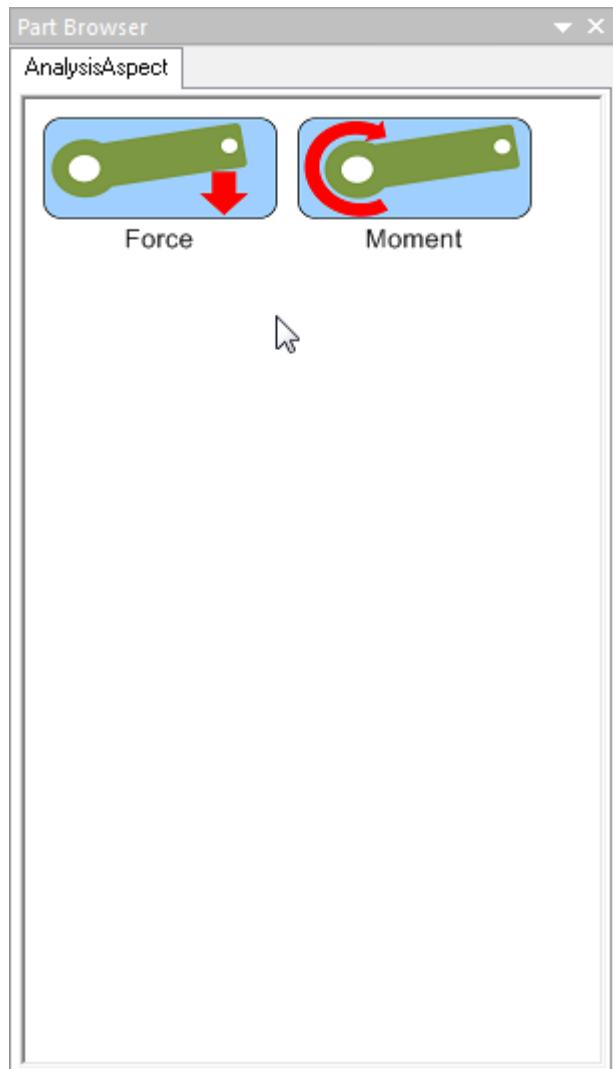


Figure 24

Step 26

Right-click on force or moment and choose Attributes to define the forces and moments in each direction (x, y, z). (Fig. 25)

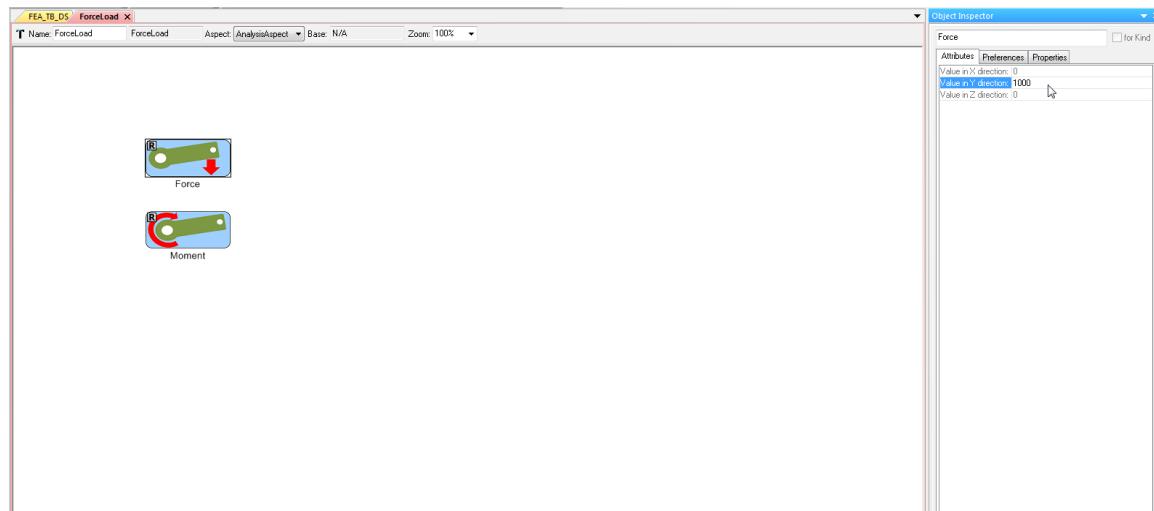


Figure 25

Step 27

Under Testing, under Workflow Definitions, right-click on the workflow definition (which has been named as CAD in Figure 33) and select copy. (Fig. 26)

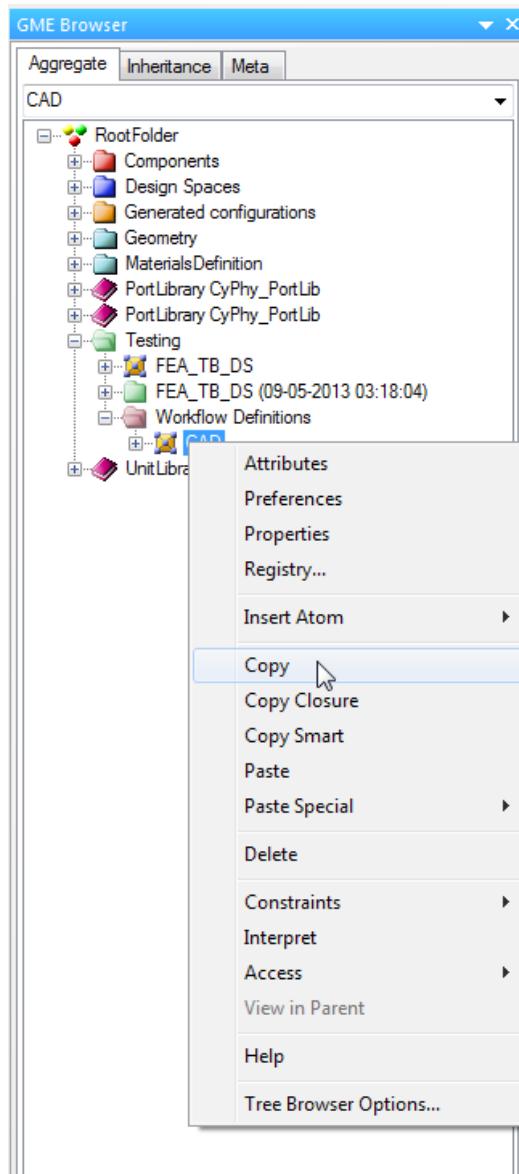


Figure 26

Step 28

Then in the FEA test bench, right-click and select Paste special and Paste as Reference. (Fig. 27).

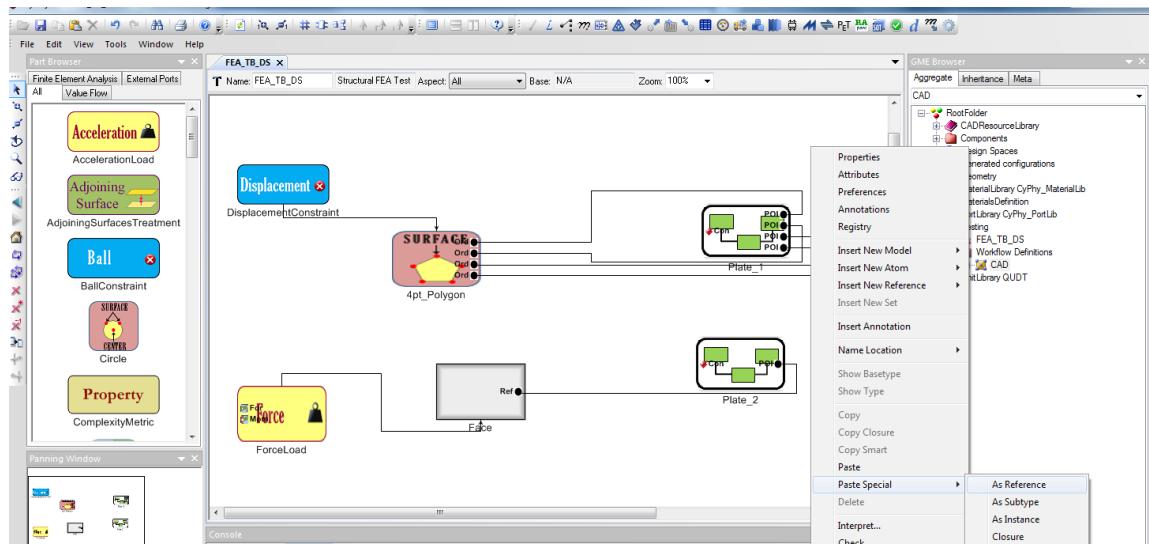


Figure 27

Step 29

Under design spaces, right-click on the design space (which has been named as 2 plates in Figure 35) and select copy. (Fig. 28).

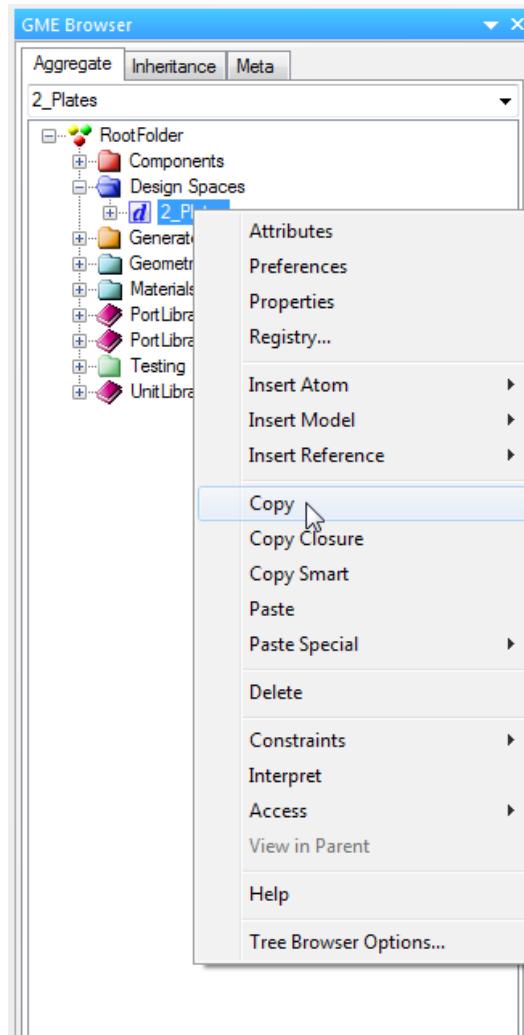


Figure 28

Step 30

Then in the FEA test bench, right-click and select Paste special and Paste as Reference. (Fig. 29)

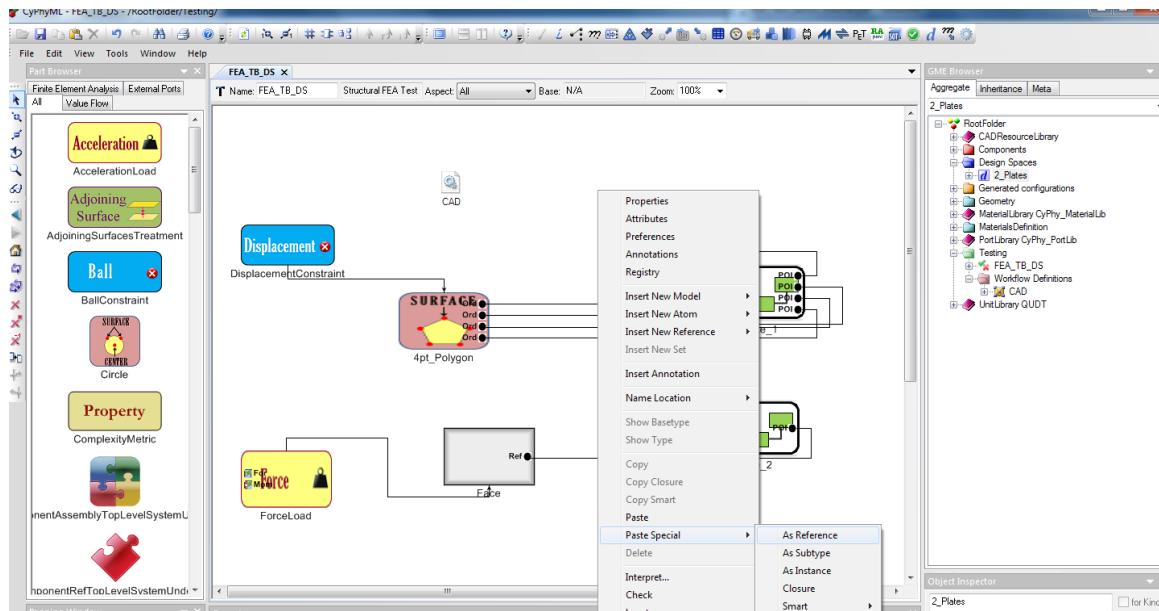


Figure 29

Step 31

In the screen that pops up, select TopLevelSystemUnderTest. (Fig. 30)

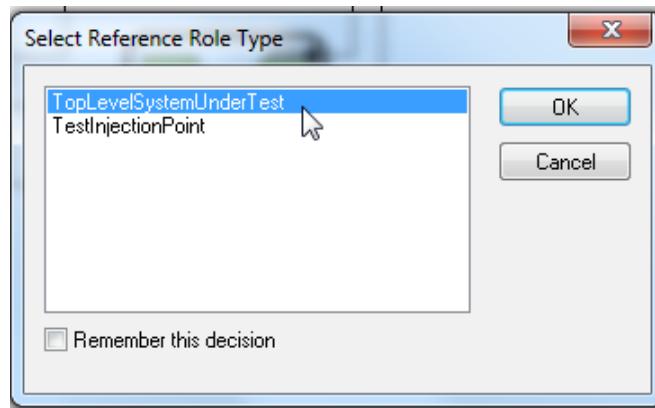


Figure 30

Step 32

Drag the StructuralFEAComputation part from the Part Browser and drop it into the FEA test bench. Repeat that step for each part. (Fig. 31)

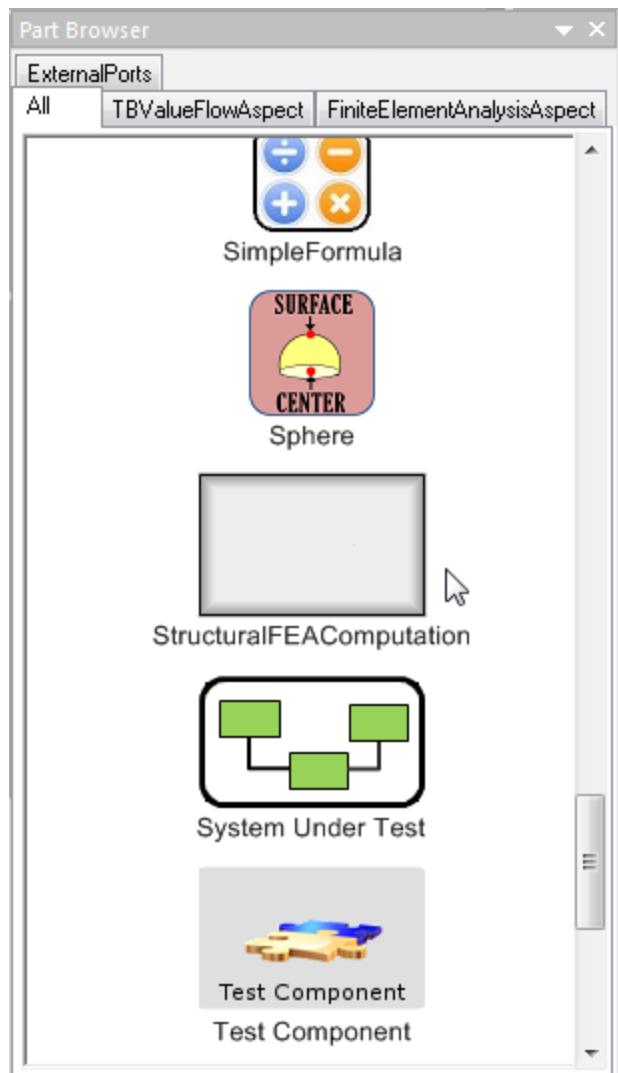


Figure 31

Step 33

Double click on the StructuralFEAComputation part and from the Part Browser drag and drop FactorOfSafety and MisesStress parts in the StructuralFEAComputation. That step tell CyPhy which outputs are required. Repeat that step for each part. (Fig. 32)

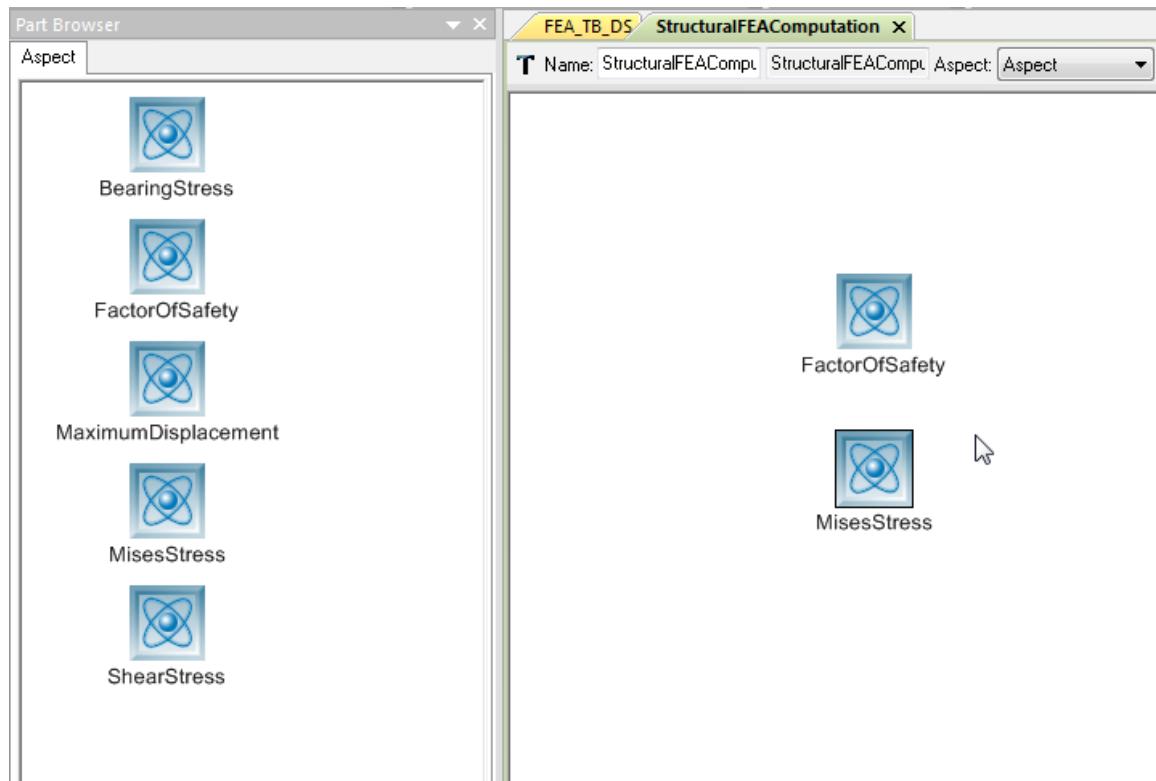


Figure 32

Step 34

The Test Bench should now look like this. (Fig. 33)

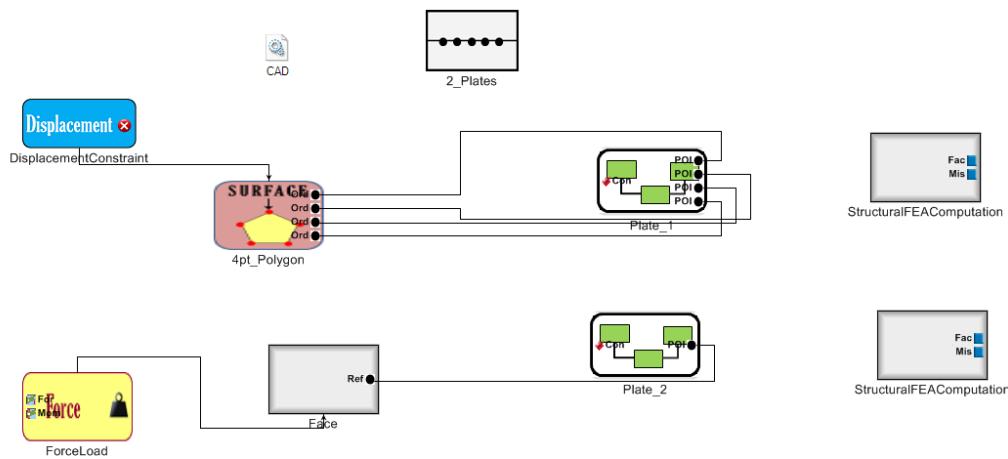


Figure 33

Step 35

Press (Ctrl+2) to get into the connection mode and connect the StructuralFEAComputation parts with every CAD reference. When you are done, Press (Ctrl+1) to go back to the edit mode. (Fig. 34)

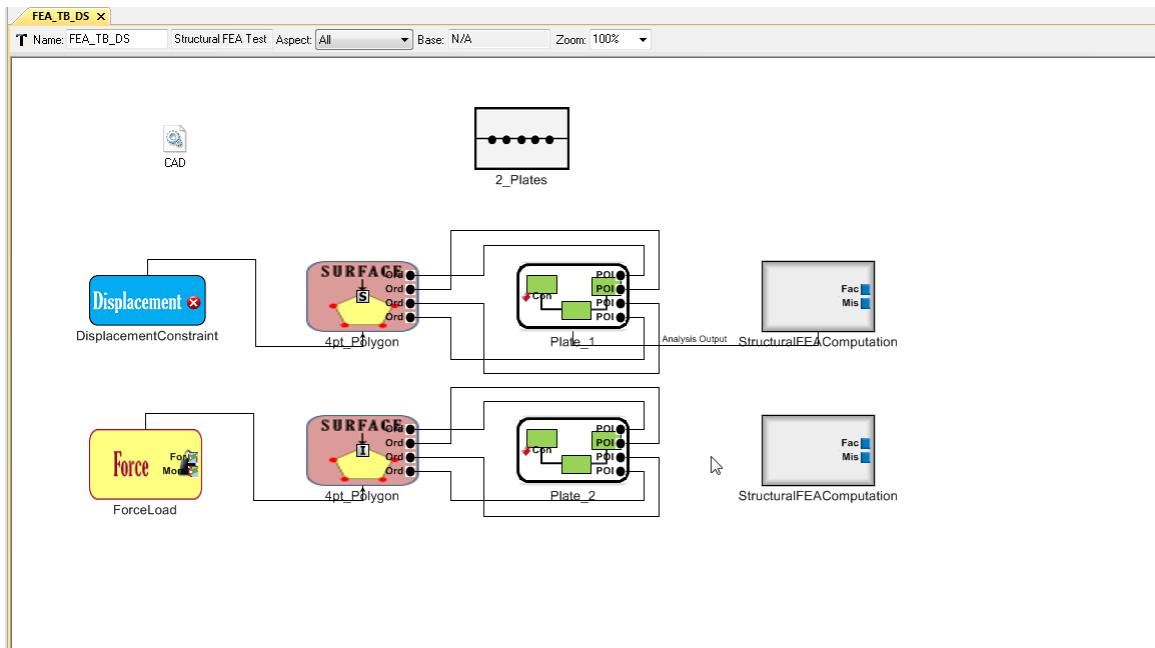


Figure 34

Step 36

From the Part Browser drag and drop Metric parts for each metric (factor of safety and mises stress for our example) into the test bench. (Fig. 35)

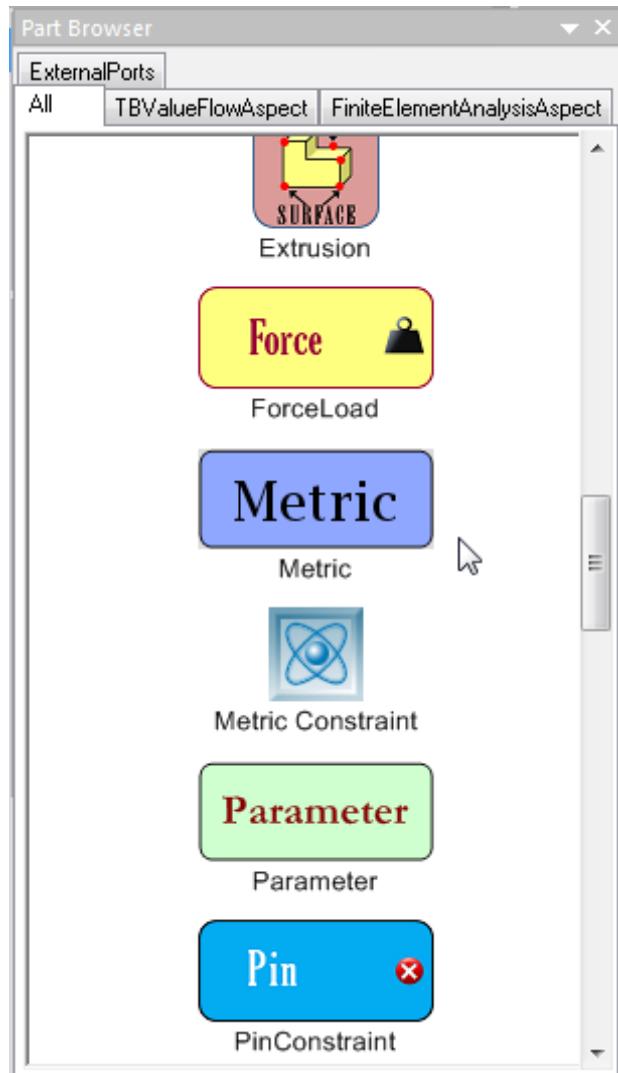


Figure 35

Step 37

The Test Bench should now look like this. (Fig. 36)

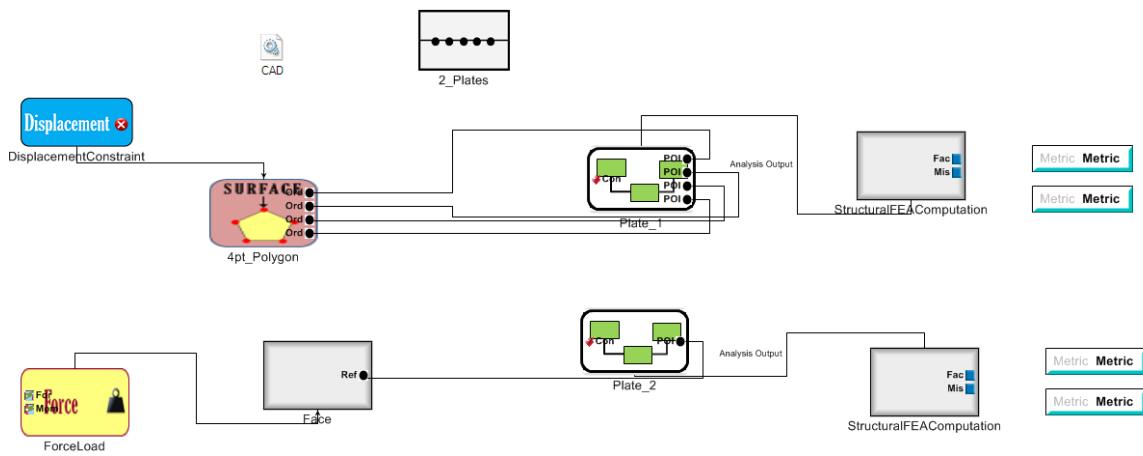


Figure 36

Step 38

Press (Ctrl+2) to get into the connection mode and connect the Metric parts with StructuralFEAComputation parts for each metric (factor of safety and mises stress for our example) in them. When you are done, Press (Ctrl+1) to go back to the edit mode. (Fig. 37)

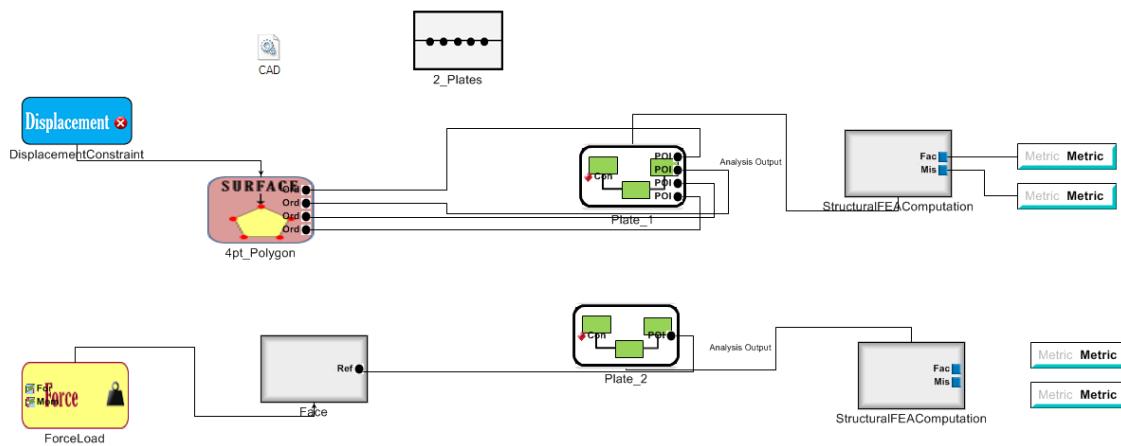


Figure 37

Step 39

Finally, your test bench should look like this. (Fig. 38)

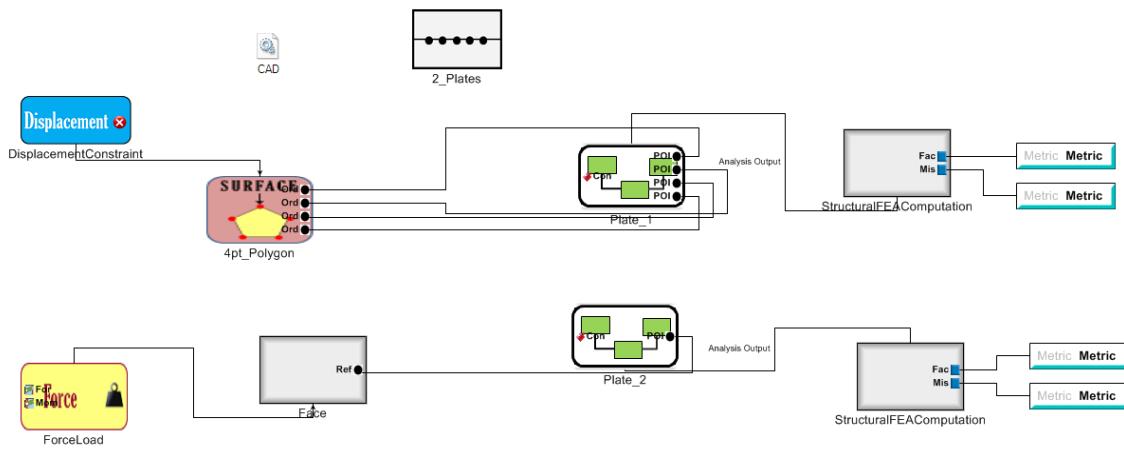


Figure 38

Step 40

Make sure the solver type is set to Abaqus model-based. (Fig 39)

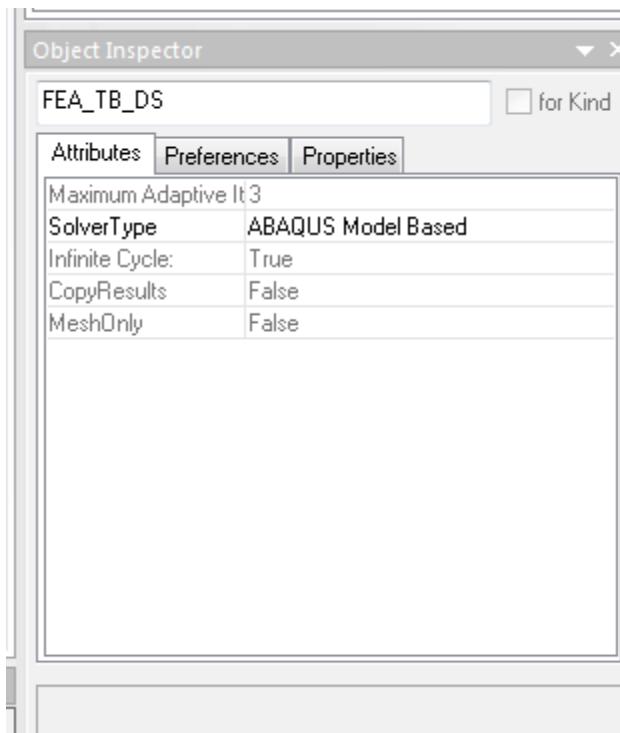


Figure 39

Running the test bench

Step 41

Once the test bench creation is complete, now we can run the test bench. On the top, click on the Master Interpreter. The Master Interpreter will determine how the test bench is going to be run. (Fig. 40)

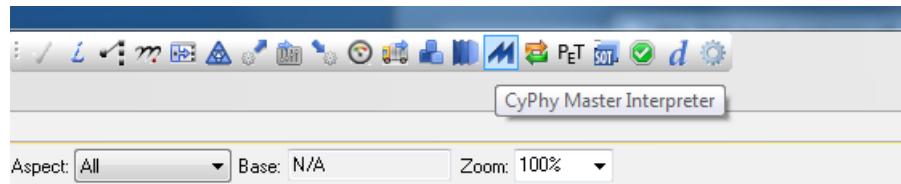


Figure 40

Step 42

When the Master Interpreter dialogue comes up, click on the configuration. Also, make sure "Post to META Job Manager" is checked. Then click OK. (Fig. 41)

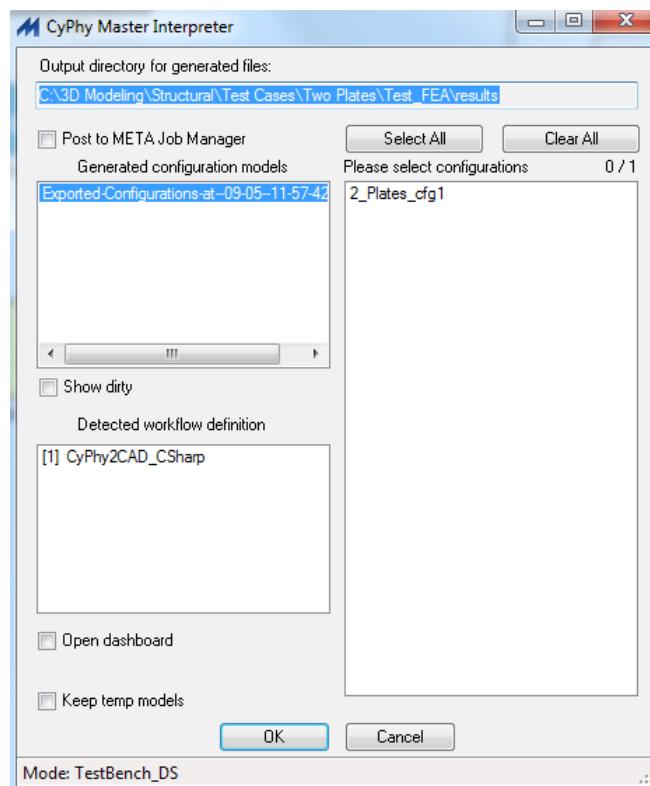


Figure 41

Step 43

The CAD Options window will come up. (Fig. 42)

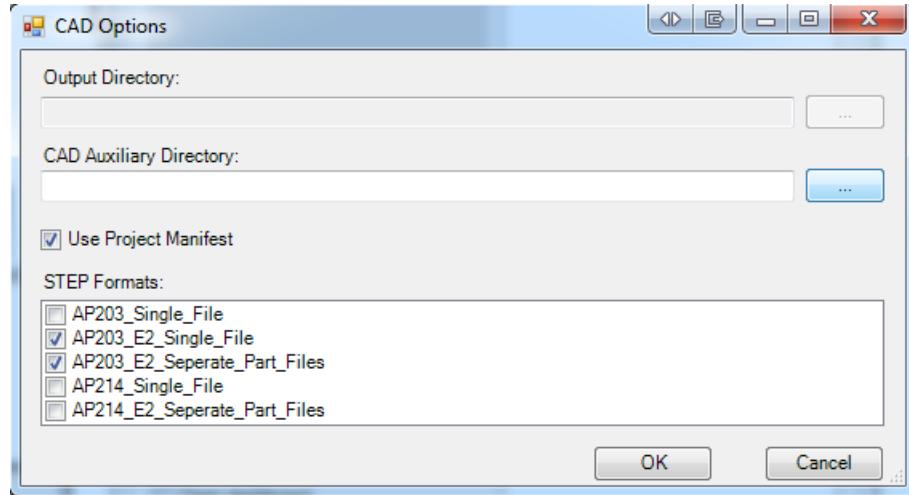


Figure 42

Step 44

Select the CAD Auxiliary directory and click OK. (Fig. 43)

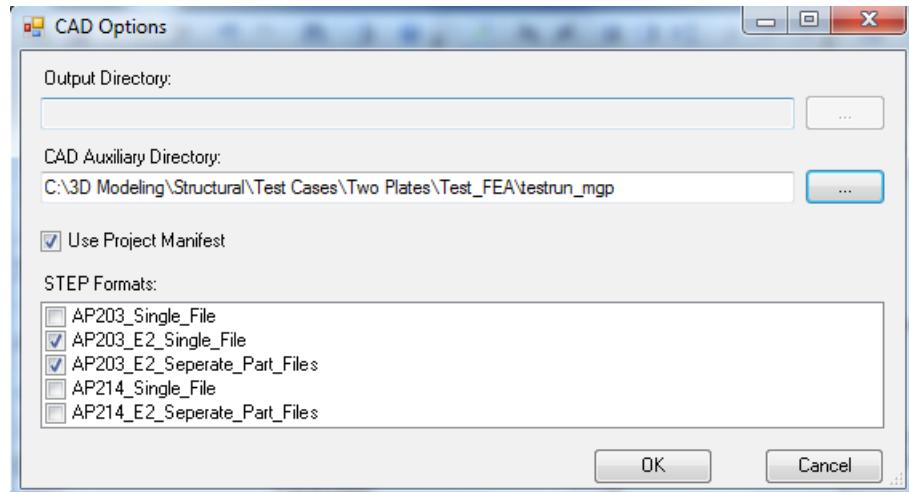


Figure 43

Step 45

The JobManager Configuration window will now open. (Fig. 44).

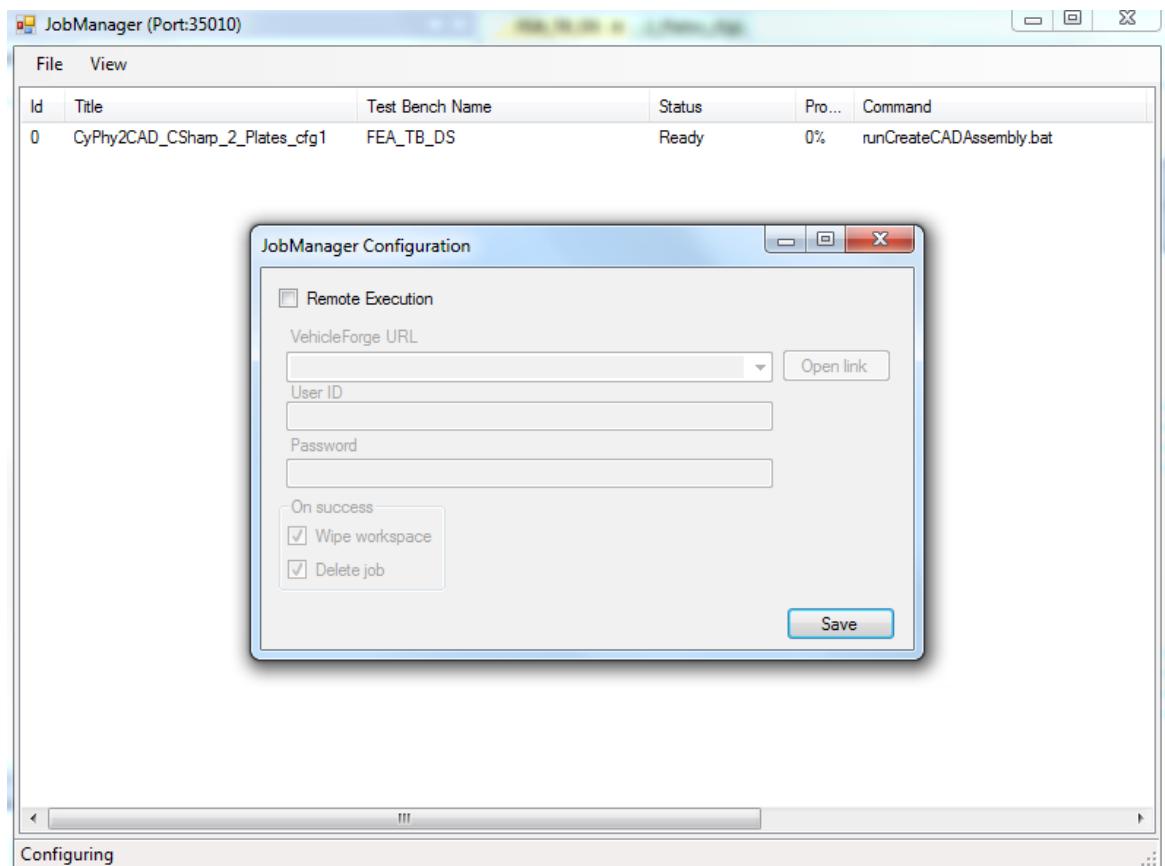


Figure 44

Step 46

Input your VF information and click Save. (Fig. 45)

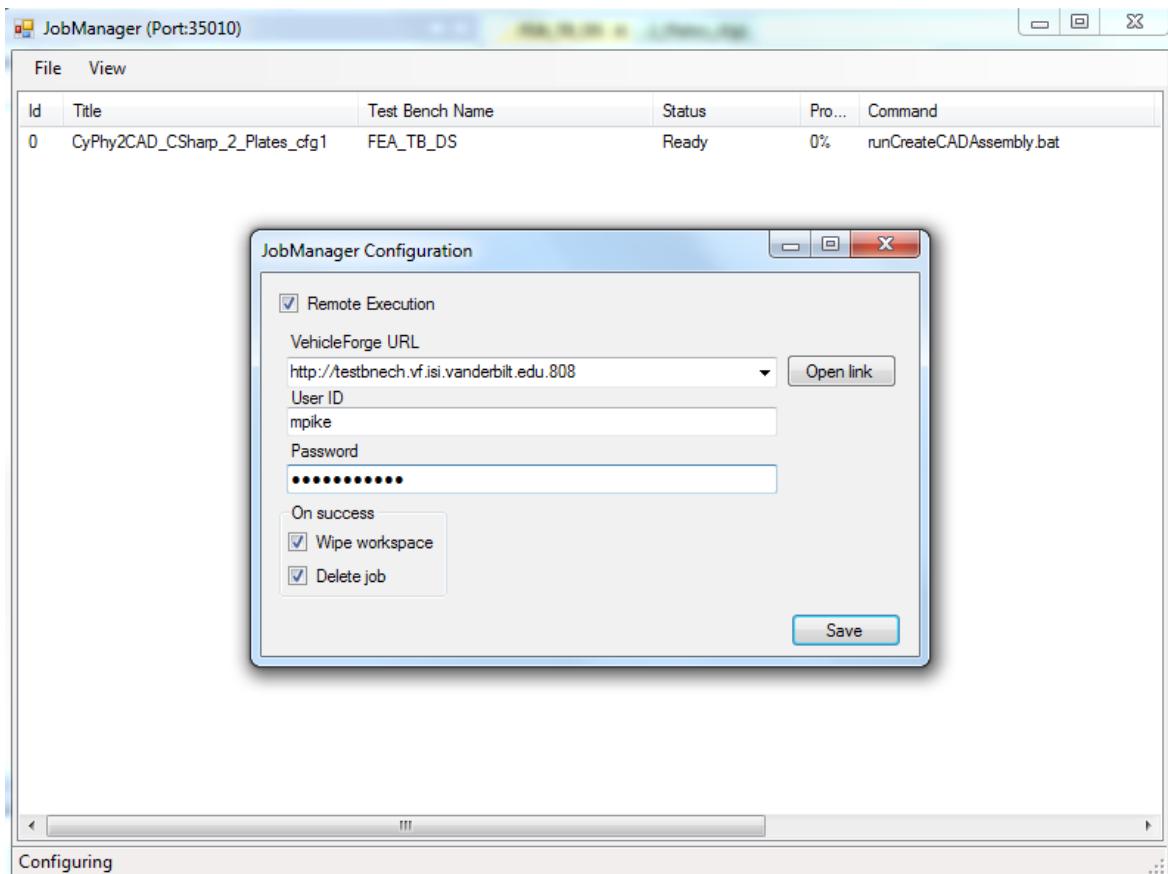


Figure 45

Step 47

The Job Manager will start the FEA computations. (Fig. 46)

JobManager (Port:35010)					
File		View			
Id	Title	Test Bench Name	Status	Pro...	Command
0	CyPhy2CAD_CSharp_2_Plates_cfg1	2_Plates_cfg1	RunningLocal	0%	runCreateCADAssembly.bat

Configured for local execution.

Figure 46

Step 48

After the job is completed, to see the results, open the main folder for the GME. (Fig. 47)

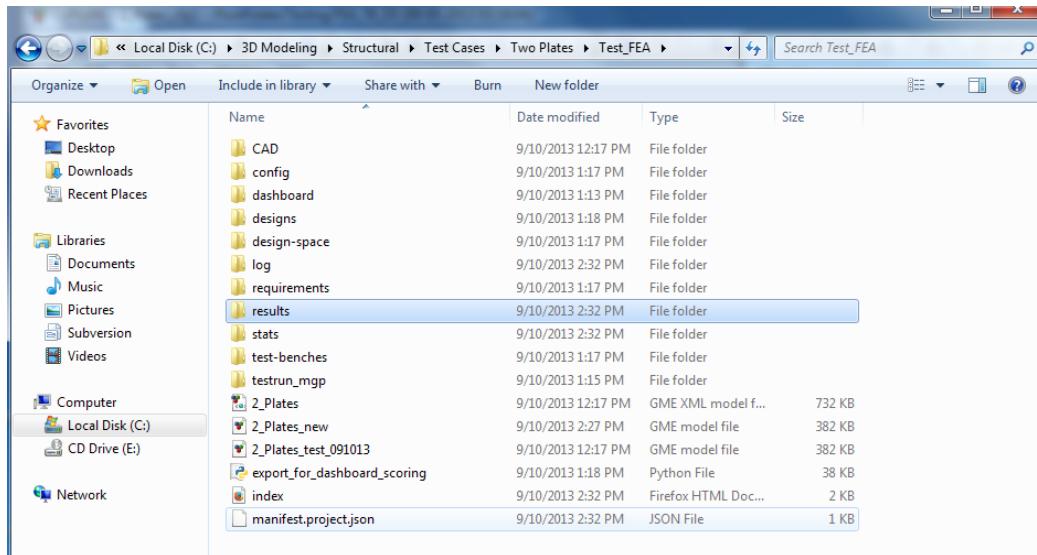


Figure 47

Step 49

Click on the correct folder. (Fig. 48)

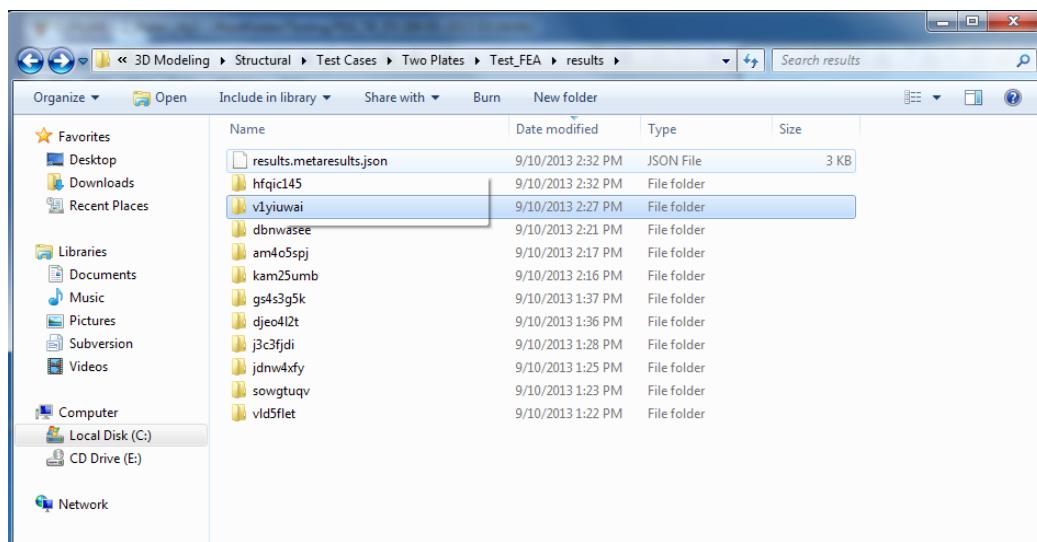


Figure 48

Step 50

The pictures associated with the results can be found under “Analysis/Abaqus”. The path is also shown in (Fig. 49). The testbench_manifest.json is located in the location show in figure 50. Additional results files such as the Abaqus .odb file are copied to the local machine if the test bench attribute CopyResults is set to True.

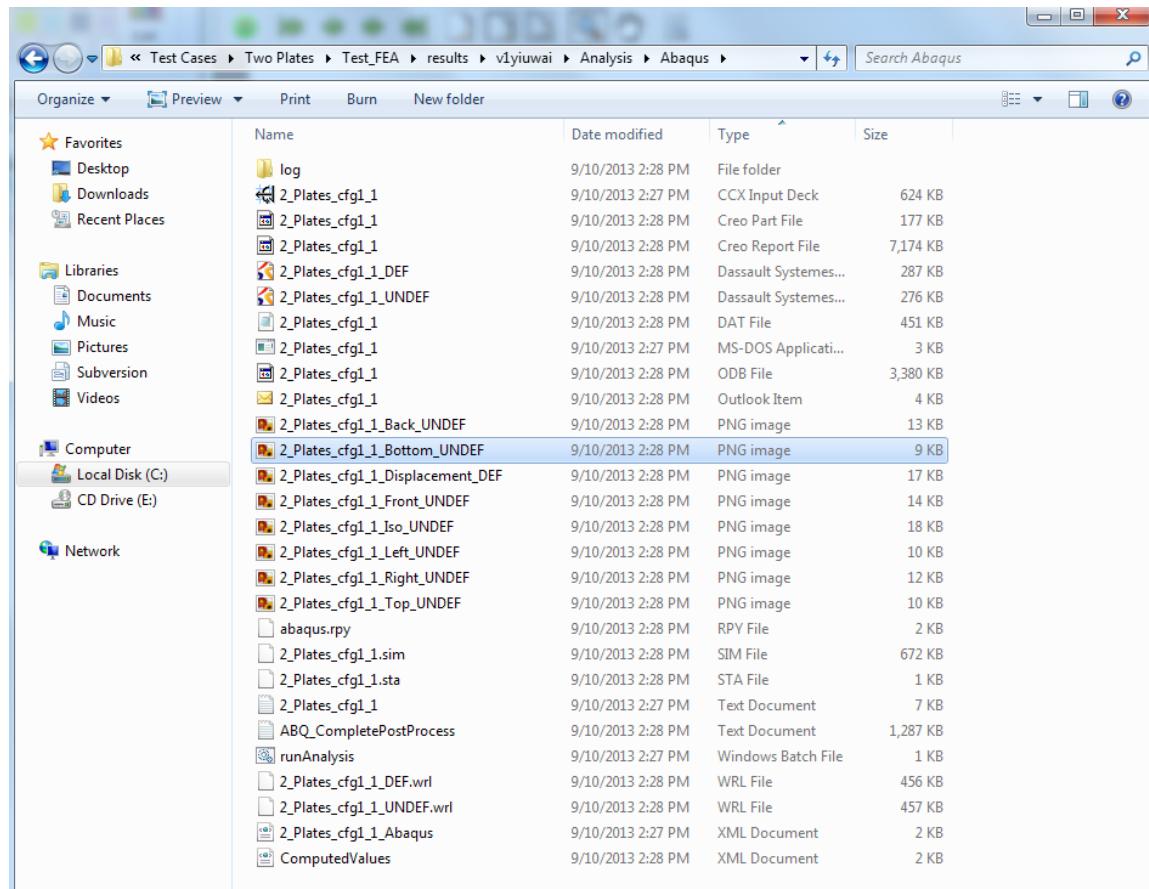


Figure 49

	Name	Date modified	Type	Size
Favorites	DataCheck	12/30/2013 11:39 ...	CCX Input Deck	1,915 KB
	DataCheckFinal	12/30/2013 11:39 ...	CCX Input Deck	1,915 KB
	NastranModel	12/30/2013 11:40 ...	CCX Input Deck	1,921 KB
	Adaptivity-1-iter1	12/30/2013 11:42 ...	Creo Part File	1,420 KB
	Adaptivity-1-iter2	12/30/2013 11:45 ...	Creo Part File	1,539 KB
	DataCheck	12/30/2013 11:39 ...	Creo Part File	1,249 KB
	DataCheckFinal	12/30/2013 11:40 ...	Creo Part File	1,249 KB
	2_plates_cfg1_1.asm	12/30/2013 11:39 ...	Creo Versioned File	57 KB
	abaqus.rpy	12/30/2013 11:46 ...	Creo Versioned File	4 KB
	plate_01.prt	12/30/2013 11:39 ...	Creo Versioned File	123 KB
	plate_02.prt	12/30/2013 11:39 ...	Creo Versioned File	102 KB
	trail.txt	12/30/2013 11:39 ...	Creo Versioned File	3 KB
	zzz_template_assy_mmks_creo.asm	11/26/2013 12:27 ...	Creo Versioned File	42 KB
	Adaptivity-1-iter1	12/30/2013 11:42 ...	DAT File	20 KB
	Adaptivity-1-iter2	12/30/2013 11:45 ...	DAT File	20 KB
	DataCheck	12/30/2013 11:39 ...	DAT File	19 KB
	DataCheckFinal	12/30/2013 11:40 ...	DAT File	19 KB
	std.err	12/30/2013 11:38 ...	ERR File	1 KB
	2_Plates_cfg1_1.jnl	12/30/2013 11:46 ...	JNL File	10 KB
	cad.manifest.json	12/30/2013 11:33 ...	JSON File	1 KB
	testbench_manifest.json	12/30/2013 11:46 ...	JSON File	2 KB
	stressOutput	12/30/2013 11:46 ...	Microsoft Excel C...	1 KB
	Adaptivity-1-iter1	12/30/2013 11:40 ...	MS-DOS Application	3 KB
	Adaptivity-1-iter2	12/30/2013 11:43 ...	MS-DOS Application	3 KB
	DataCheck	12/30/2013 11:39 ...	MS-DOS Application	3 KB
	DataCheckFinal	12/30/2013 11:39 ...	MS-DOS Application	3 KB
	Adaptivity-1-iter1	12/30/2013 11:42 ...	ODB File	14,457 KB
	Adaptivity-1-iter2	12/30/2013 11:46 ...	ODB File	15,537 KB
	DataCheck	12/30/2013 11:39 ...	ODB File	6,639 KB
	DataCheckFinal	12/30/2013 11:40 ...	ODB File	6,639 KB
	std.out	12/30/2013 11:38 ...	OUT File	1 KB
	Adaptivity-1-iter1	12/30/2013 11:42 ...	Outlook Item	25 KB
	Adaptivity-1-iter2	12/30/2013 11:45 ...	Outlook Item	29 KB
	DataCheck	12/30/2013 11:39 ...	Outlook Item	2 KB

Figure 50

Opening the testbench_manifest.json will allow you to view the metrics that you have requested from the analysis (e.g. minimum factor of safety, maximum stress, etc.) for the assembly.

Step 51

A comma-separated values (CSV) file is created by default for all static and dynamic FEA test benches. The file, stressOutput.csv is located in the results folder upon successful test bench completion. Material properties such as fatigue strength contained in the CSV file are obtained from the META Material Library. A modes test bench (one with the Modes metric in the StructuralFEAComputation object) creates a modalOutput.csv instead of the stressOutput.csv. modalOutput.csv contains a list of the first 30 natural frequencies for the assembly.

Step 52

In (Fig. 51), effective (von Mises) stress fringes of the model is presented. Results of the analysis can easily be visualized by using software that can open odb files (e.g. Abaqus, 3D xml viewer, etc.).

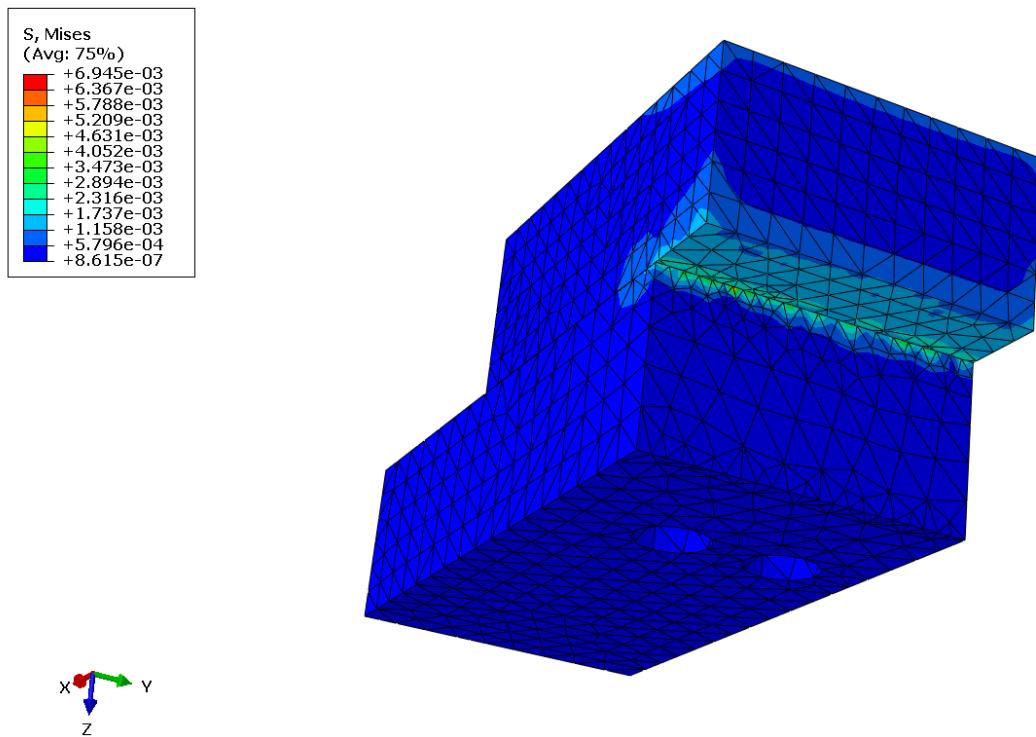


Figure 51

Analysis times are highly variable depending on the size of the model. The FEA test bench may be run locally if you have Abaqus installed, otherwise FEA test benches must be run on a server.

Test bench attributes allow the user to control some aspects of the FEA simulation.

- **MaximumAdaptiveIterations.** The value entered here determines the maximum number of iterations for adaptive mesh refinement. The number must be between zero and 10, inclusive. A value of zero or one results in a standard FEA run without adaptive mesh refinement. A value of two results in one run with the initial mesh and one run with a remesh. Higher values result in that many additional remesh and solve procedures. This setting is applicable only to static analysis and is ignored when dynamic or modes options are set in the StructuralFEAComputation object.
- **SolverType.** This attribute allows the user to select from the available solver types such as the Abaqus model-based solver.
- **InfiniteCycle.** Selecting True assumes infinite life for the components.
- **CopyResults.** The default value (False) instructs the tools to copy some, but not all of the results from VehicleForge to the local machine when the test bench is finished. Selecting True instructs the tools to copy all results, including FEA databases, from VehicleForge to the local machine. The size of all results can be many gigabytes of data and therefore the True option should be used sparingly. This option has no effect when the FEA test bench is run locally.
- **MeshOnly.** This option creates the assembly, performs a mesh of all components, and produces images illustrating the mesh and boundary conditions. This option is useful for confirming boundary conditions before submitting the test bench for a full solve operation.

6.0 Dynamics Analysis

The above procedure describes a static analysis. To direct the FEA tool to perform a dynamic analysis, place the Dynamics metric inside the StructuralFEAComputation object for the desired test bench. All loads in the test bench are given a time profile of zero load at zero seconds, linearly increasing to full load at 5 seconds, full load from 5 seconds to 10 seconds, and linearly decreasing to zero load at 15 seconds.

7.0 Modal Analysis

To direct the FEA tool to perform a modal analysis, place the Modes metric inside the StructuralFEAComputation object for the desired test bench.

8.0 CAD Representations

Some parts in the C2M2L library are too detailed for structural analysis and for rendering of large models, both of which can benefit from a multi-fidelity model scheme. Some parts contain lettering, which causes problems when creating a mesh of the part for finite element analysis. Meshing algorithms fail to create a valid mesh with some of these parts while with other parts, the meshes contain too many elements. Those parts that are off-the-shelf are not analyzed for maximum stress, however meshing these parts to transmit loads and deflections is important for structural analysis of the surrounding parts. The META tools support multiple representations of the geometry. Multiple representations allow component designers to include featured representations and defeatured representations of the geometry in the component package. The META tools allow any test bench that uses CAD to specify the CAD representation to be used for the analysis. A separate specification can be made for buy components and for make components. The buy or make distinction is made according to the value in the component's manufacturing model "procurement_make_or_buy" parameter. Table 2 details the Parameter objects used to specify the representations and Table 3 details the parameter values needed to specify a particular representation. If the requested representation is available in the CAD file, that representation is used otherwise the specified DEFAULT REP is used if that representation is available. If the specified DEFAULT REP is not available, Creo's default representation, Master Rep, is used. The META FEA tools have been designed to use the defeatured representation for buy components while using the featured representation for make components. These selections must be set in the test bench according to Tables 2 and 3 for proper operation.

Description	Parameter Name
Buy component representation	BUY REP
Make component representation	MAKE REP
Default component representation	DEFAULT REP

Table 2

CAD Representation	Parameter Name
Featured representation	Featured_Rep
Defeatured representation	Defeatured_Rep
Default representation	Master Rep (or other desired default representation)

Table 3: Test bench CAD representation parameter values

9.0 Creating Featured and Defeatured CAD Representations

9.1 Buy Parts with Non-native Creo Geometry

Use the following procedure to create the multiple CAD representations needed by the META FEA tool for components with procurement_make_or_buy = buy in the component's manufacturing model in CyPhy. These components are not analyzed for stress; however, they are meshed to transmit loads and deflections to other components. These instructions are for components with non-native Creo geometry. For example, native Creo geometry includes sketches and extrusions to create solid geometry. Some components have the geometry imported into Creo from another format such as IGES. A component with non-native Creo geometry will have "Import Feature id" in the model tree.

1. Open the Creo part file (see example in Figure 52).
2. File – Save as – Type = Shrinkwrap. Faceted solid, quality level 2, fill holes, ignore quilts, assign mass properties, output part. For simple geometries, you may want to preview the shrinkwrap before clicking OK. Modify the quality level to obtain the desired shrinkwrap and then click OK. Set relative accuracy (1.000e-05) and click check. Close shrinkwrap options when file creation successful.
3. File – Open – select the newly created shrinkwrap file (see example in Figure 53)
4. File – Save as – Type = STEP
5. Close the shrinkwrap file and reactivate the original part
6. Select Model, Get Data, Import. Leave default options.
7. Click Ok and then checkmark to accept
8. Select View – Manage Views.
9. Create a new view “Featured_Rep”. Exclude the shrinkwrap part from the Featured_Rep view (select Feature, Exclude, then select the Import Feature id at the bottom of the Model Tree, click Done twice).
10. Create a new view “Defeatured_Rep”. Exclude the fully detailed Import Feature id near the top of the Model Tree.
11. Ensure the mass properties are set correctly for the part.
12. Save the part.

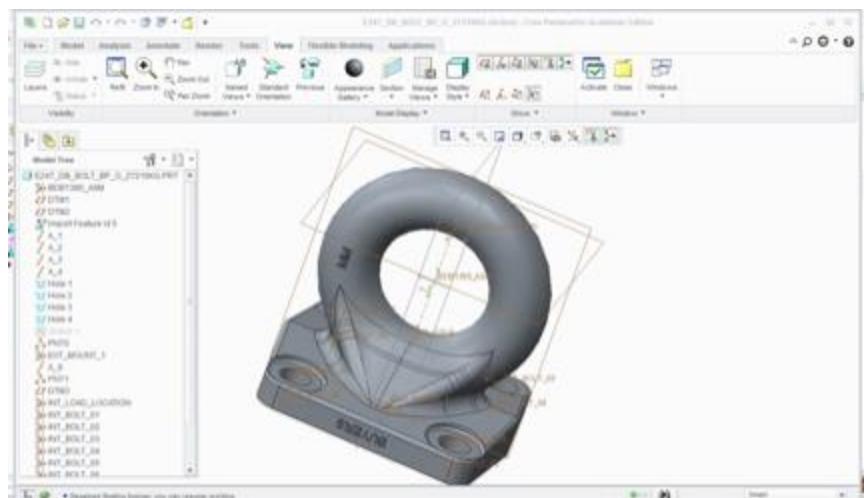


Figure 52

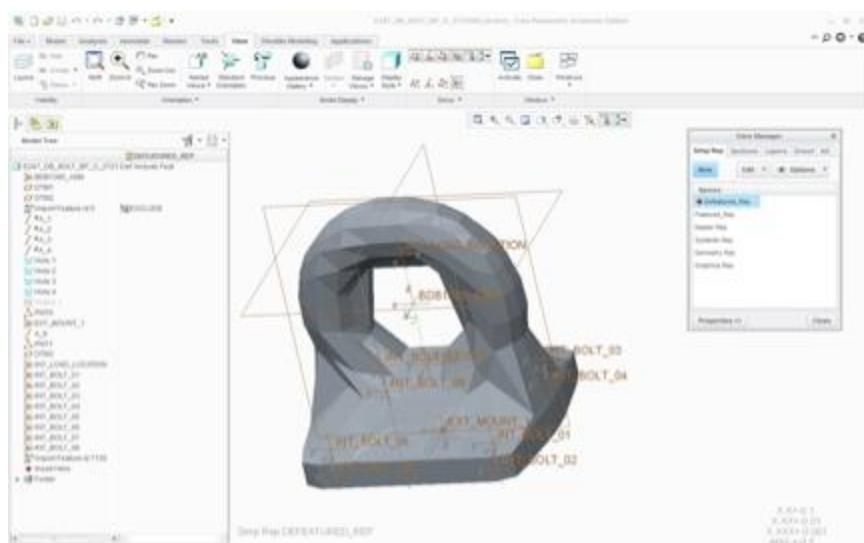


Figure 53

9.2 Make Parts

Some make parts (`procurement_make_or_buy = make` in the component's manufacturing model in CyPhy) contain hidden geometry that is used to define visible geometry. The C2M2L hulls are an example of this type of component as the `hull_soapbar` part is referenced by the concept panels however the `hull_soapbar` part is hidden and should not be included analyzed in FEA. Use the following procedure to create the multiple CAD representations of the hull.

1. Open the Creo hull component

2. Select View – Manage Views
3. Create a new view “Featured_Rep”. Exclude the hull_soapbar part from the Featured_Rep view (select Feature, Exclude, then select the hull_soapbar part in the Model Tree, click Done twice).
4. Create a new view “Defeatured_Rep”. Exclude the hull_soapbar part
5. Select the hull_soapbar part in the model tree
6. On the toolbar, in the Visibility section, select Hide if it is not already greyed out
7. Save the hull component

Other make parts may also have hidden geometry or native Creo geometry that could be excluded from either the Featured_Rep or the Defeatured_Rep. The following procedure should be followed to exclude unwanted geometry.

1. Open the Creo make component
2. Select View – Manage Views
3. Create a new view “Featured_Rep”. Exclude the unwanted geometry from the Featured_Rep view
4. Create a new view “Defeatured_Rep”. Exclude the unwanted geometry from the Defeatured_Rep view
5. Select all unwanted geometry in the model tree
6. On the toolbar, in the Visibility section, select Hide if it is not already greyed out
7. Save the component

10.0 Verifying a Component can be Meshed

To verify that a featured or defeatured representation can be meshed with the META tools, an FEA test bench with the parameter object named MESH_ONLY with a value of 1 can be run. Additionally the geometry can be exported to Step format and meshed within Creo. The Creo procedure is useful for checking the geometry as a component is modified, however the mesh only test bench should be executed to complete the mesh verification. The Creo procedure is:

1. Export the geometry to Step format and open the Step file.
2. Select Applications – Simulate
3. Select Model Setup
4. Check FEM mode and click OK
5. Select Mesh
6. If the mesh is successful, a window titled “Element Quality Checks” will appear.
7. If the mesh is unsuccessful, a window will appear explaining the error and mesh geometry will be highlighted indicating the area posing mesh issues.

11.0 Description

FEA simulation is a numerical procedure to determine the structural performance of the design. The test bench will compute the maximum von Mises stress and factor of safety accurately under the conditions provided by the user. FEA software toolsuite, Abaqus is used for FEA simulations.

The System Under Test is assembled in CREO and then each component making up the system is saved as an individual step file. Input from the user is also gathered. This information is packaged and sent (via remote server) to the VehicleFORGE servers for FEA simulations.

12.0 Test Bench Tiers

Attribute	Tier 1	Tier 2	Tier 3
Test Bench Name	Accidental load, rail impact, surf zone grounding, gun fire loads	Accidental load, rail impact, surf zone grounding, gun fire loads	Accidental load, rail impact, surf zone grounding, gun fire loads
Description	2D, 3D mesh with perfect joins	2D, 3D mesh with 1D joins	2D, 3D mesh with 3D joins
Estimate Run Time	A few minutes to a few hours, depending upon design complexity	A few minutes to a few hours, depending upon design complexity	A few minutes to a few hours, depending upon design complexity
Error Margin	Error margin not yet quantified. Perfect joins introduce errors at the joins; however, errors are reduced away from the joins.	Error margin not yet quantified. 1D joins introduce fewer errors than perfect joins.	Error margin not yet quantified. 3D joins introduces the fewest assumptions of all of the FEA tiers.
Results Provided	Images of loads and constraints, mesh, detailed results by element.	Images of loads and constraints, mesh, detailed results by element.	Images of loads and constraints, mesh, detailed results by element.
Local/Remote	Both (must have the FEA program installed locally to run local)	Both (must have the FEA program installed locally to run local)	Both (must have the FEA program installed locally to run local)
Tool Used	Abaqus or Nastran	Abaqus or Nastran	Abaqus or Nastran
How to Interpret Results	Review deformation and stress for each component using Abaqus or Nastran post-processing tools.	Review deformation and stress for each component using Abaqus or Nastran post-processing tools.	Review deformation and stress for each component using Abaqus or Nastran post-processing tools.
Model Requirements	Meshable components, material defined for each component, material properties (density, fatigue	Meshable components, material defined for each component, material properties (density, fatigue	Meshable components, material defined for each component, material properties (density, fatigue

Attribute	Tier 1	Tier 2	Tier 3
	strength, Young's modulus) for each material, loads, constraints.	strength, Young's modulus) for each material, loads, constraints.	strength, Young's modulus) for each material, loads, constraints

13.0 Test Bench Components

Each FEA test bench must have the following objects:

- SystemUnderTest (may be a component or an assembly). Only one SystemUnderTest per test bench.
- TestInjectionPoint (may be a component or an assembly). May have more than one TestInjectionPoint in a test bench.
- StructuralFEAComputation. May specify static analysis, dynamic analysis, or modal analysis.
- Metrics
- PostProcessing
- CAD representation parameters (BUY_REP, MAKE_REP)

For statics and dynamics test benches, the following additional objects are required:

- Load
- Constraint

13.1 SystemUnderTest

Each FEA test bench must have acceleration loading on every component in the assembly by adding the AccelerationLoad object from the GME part browser to a test bench. Acceleration is specified via attributes for the AccelerationLoad object. Acceleration is specified in mm/s² with three values: x direction, y direction, and z direction. These directions reference the assembly coordinate system. One g load is 9,810 mm/s². The AccelerationLoad object does not need to be connected to other objects in the test bench.

13.2 Loads

13.2.1 AccelerationLoad

Specify acceleration loading on every component in the assembly by adding the AccelerationLoad object from the GME part browser to a test bench. Acceleration is specified via attributes for the AccelerationLoad object. Acceleration is specified in mm/s² with three values: x direction, y direction, and z direction. These directions

reference the assembly coordinate system. One g load is 9,810 mm/s². The AccelerationLoad object does not need to be connected to other objects in the test bench.

13.2.2 ForceLoad

Specify force load on a component by adding the ForceLoad object from the GME part browser to a test bench. Once the ForceLoad object is in the test bench, Force and Moment objects must be added to the ForceLoad object. Both Force and Moment contain attributes for values in the x direction, y direction, and z direction. These directions reference the assembly coordinate system. Force units are N and Moment units are mm² kg. The force must be connected to a geometric construct such as face or polygon.

13.2.3 PressureLoad

Specify pressure load on a component by adding the PressureLoad object from the GME part browser to a test bench. Pressure is specified in N/mm² (MPa). The pressure must be connected to a geometric construct such as face or polygon.

13.3 Constraints

13.3.1 Displacement

Specify displacement constraint on a component by adding the DisplacementConstraint object from the GME part browser to a test bench. Once the DisplacementConstraint object is in the test bench, Translation and Rotation objects must be added to the DisplacementConstraint object. Both Translation and Rotation contain attributes for type and value in the x direction, y direction, and z direction. Type can be scalar, fixed, or free. These directions reference the assembly coordinate system. Translation units are in mm and rotation units are in degrees. The DisplacementConstraint must be connected to a geometric construct such as face or polygon. One Displacement object can be connected to many components.

13.3.2 Geometric Constructs

Loads and constraints are applied to specific areas of components. The following geometric constructs are available. Analysis points are created in Creo to identify either a circle, a face, or a polygon. Analysis points may be created using META Link. Refer to the META Link documentation for creating analysis points. The system

under test and geometric constructs, as well as the loads and constraints, exist at the test bench level. Components with analysis points may exist in subassemblies and those analysis points must be propagated up from the component, to the subassembly, and finally to the top-level system under test assembly.

13.3.2.1 Circle

Three analysis points are needed to define a circle for applying loads and constraints. Once the Circle construct is added to a test bench, CircleCenter and two CircleEdge objects must be added to the the Circle object. These are connected to the analysis points created in the component. The circle must cover the entire face on the component as a face cannot be partitioned. A use case for the Circle construct is the end of a cylindrically-shaped component.

13.3.2.2 Face

The Face construct is the simplest and most powerful geometric construct in CyPhy as it requires just one analysis point placed on the surface of the component. The entire face, visible in Creo\textregistered by selecting the face where the analysis point is placed, is used for applying the load or constraint. One ReferencePoint is added to the Face object and that point is connected to the analysis point on the component. The face construct is useful for displacement constraints and force loads normal to the face or in traction.

13.3.2.3 Polygon

Three or more analysis points are required to define a polygon for applying loads and constraints. Once the Polygon construct is added to a test bench, OrdinalPoints are added to the Polygon object and then connected to the analysis points on the component. The polygon must cover the entire face on the component as a face cannot be partitioned. A use case for the Polygon construct is the end of a plate. The polygon is useful when applying a traction force to a component.

14.0 Metrics

- Minimum_Fatigue_Strength_Factor_of_Safety
- Minimum_Yield_Strength_Factor_of_Safety
- Minimum_Ultimate_Tensile_Strength_Factor_of_Safety
- Minimum_Mode

Test	Benc Metric	Description
	h #	
84a	Accidental_Loads_Back	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with an accidental load of 8 g rearward
84b	Accidental_Loads_Down	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with an accidental load of 10 g downward
84c	Accidental_Loads_Forward	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with an accidental load of 8 g forward
84d	Accidental_Loads_Left	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with an accidental load of 5 g to the left
84e	Accidental_Loads_Right	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with an accidental load of 5 g to the right
84f	Accidental_Loads_Up	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with an accidental load of 10 g upward
87a	Rail_Transportation_Impact_Standard_Back	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with MIL-STD-810 loading.
87b	Rail_Transportation_Impact_Standard_Forw ard	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with MIL-STD-810 loading.
88	Surf_Zone_Load_Survivability	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with surf zone loading.
90a	Gun_Fire_Loads_000_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90b	Gun_Fire_Loads_045_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90c	Gun_Fire_Loads_090_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90d	Gun_Fire_Loads_135_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90e	Gun_Fire_Loads_180_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90f	Gun_Fire_Loads_225_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90g	Gun_Fire_Loads_270_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90h	Gun_Fire_Loads_315_deg	True if Minimum_Fatigue_Strength_Factor_of_Safety >= 1.0 with gun fire loading.
90i	Gun_Fire_Modes	True if Minimum_Mode >= specified frequency.
107a	MIL_STD_209K_Lift	True if Minimum_Ultimate_Tensile_Strength_Factor_of_Saf ety >= 1.5 and

Test	Benc Metric	Description
h #		
		Minimum_Yield_Strength_Factor_of_Safety >= 1.0 while being lifted.
107b MIL_STD_209K_Tiedown_Back		True if Minimum_Ultimate_Tensile_Strength_Factor_of_Safety >= 1.5 and Minimum_Yield_Strength_Factor_of_Safety >= 1.0 with 4 g loading rearward with tiedowns.
107c MIL_STD_209K_Tiedown_Forward		True if Minimum_Ultimate_Tensile_Strength_Factor_of_Safety >= 1.5 and Minimum_Yield_Strength_Factor_of_Safety >= 1.0 with 4 g loading forward with tiedowns.
107c MIL_STD_209K_Tiedown_Left		True if Minimum_Ultimate_Tensile_Strength_Factor_of_Safety >= 1.5 and Minimum_Yield_Strength_Factor_of_Safety >= 1.0 with 1.5 g left loading with tiedowns.
107d MIL_STD_209K_Tiedown_Right		True if Minimum_Ultimate_Tensile_Strength_Factor_of_Safety >= 1.5 and Minimum_Yield_Strength_Factor_of_Safety >= 1.0 with 1.5 g right loading with tiedowns.
107e MIL_STD_209K_Tiedown_Up		True if Minimum_Ultimate_Tensile_Strength_Factor_of_Safety >= 1.5 and Minimum_Yield_Strength_Factor_of_Safety >= 1.0 with 2 g upward loading with tiedowns.
170 FMVSS_207_210_Back		True if Minimum_Yield_Strength_Factor_of_Safety >= 1.0 with FMVSS-207/210 loading on the seats.
171 FMVSS_207_210_Forward		True if Minimum_Yield_Strength_Factor_of_Safety >= 1.0 with FMVSS-207/210 loading on the seats.

Table 4

15.0 Outputs

The output of this test bench is in the “testbench manifest.json” file and the files located in the “Abaqus/Analysis” directory. The summary results presents the maximum von Mises stress in MPa and factor of safety. The json file can be viewed with a simple text editor, csv file can be viewed by using Office-Excel (or similar software) and all image files can be viewed with a simple image viewer.

A comma-separated values (CSV) file is created by default for all static and dynamic FEA test benches. The file, stressOutput.csv is located in the results folder upon successful test bench completion. Material properties such as fatigue strength contained in the CSV file are obtained from the META Material Library. A modes test bench (one with the Modes metric in the StructuralFEAComputation object) creates a modalOutput.csv instead of the stressOutput.csv. modalOutput.csv contains a list of the first 30 natural frequencies for the assembly. Modal and stress are separate analyses, therefore one test bench can create the modal output or the stress output but not both.

16.0 Validation

Validation models are distributed with the tools and are installed in the META Documents folder.

17.0 Known Limitations

- Test bench names cannot include the hyphen (-) character.
- ForceLoad does not support Moments.
- DisplacementConstraint supports fixed or free translation and fixed translation and rotation only.
- Polygon and circle constructs do not support portioning of the face. The entire polygon or circle must be described with the analysis points.
- Abaqus model-based is the only supported SolverType. Abaqus deck-based and Nastran are under development.
- The META FEA tool currently assumes all joints are perfectly bonded. Additional tiers will be added in the future to handle 1D bolts and welds, 3D bolts and welds, and translational joints.
- Custom dynamic profiles are not yet supported. Placing the Dynamics metric in the StructuralFEAComputation object applies a fixed time profile to all loads in the test bench. The profile is zero load at zero seconds, linearly increasing to full load at 5 seconds, full load from 5 seconds to 10 seconds, and linearly decreasing to zero load at 15 seconds.

- ReferenceSurface object in the Face construct is not supported.
- InfiniteCycle test bench attribute is ignored.
- The following FEA test bench objects are currently not supported:
AdjoiningSurface, BallConstraint, ComplexityMetric,
ComponentAssemblyTopLevelSystemUnderTest,
ComponentRefTopLevelSystemUnderTest,
ComponentTopLevelSystemUnderTest, Constant, CustomFormula,
CustomGeometry, Cylinder, Extrusion, MetricConstraint, PinConstraint,
Property, RandomParameterDriver, Requirement Link, SimpleFormula,
Sphere, Test Component, TestComponentTopLevelSystemUnderTest, and
ValueFlowTypeSpecification. Additionally, FEA test benches currently
cannot be included in a Suite of Test benches (SOTs) and value flow is not
supported (i.e., values cannot flow from a component, computation, or
another test bench into or out of a FEA test bench).
- Ignoring certain stress singularities is not currently supported. The
results report the maximum von Mises stress and if an artificial stress
concentration exists, the user cannot instruct the META tools to ignore
the concentration.