

Finite Element Analysis (FEA) Test Bench and Tool

Download the [FEA_Plate zip](#) for testing the FEA Testbench.

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. The load and boundary conditions that the test bench can simulate can be seen in (Table 1).

Table 1

Load/Constraint	Applicable Geometry	Sub-Cases
Displacement Constraint	2-D	2-D
	Polygon	Interior Only (Seldom Used)
	Circle	Boundary Only (Seldom Used)
	3-D	3-D
	Cylinder	Interior Only (Seldom Used)
	Sphere	Surface Only (Often Used)
	Extrusion	Interior and Surface (Seldom Used)
Pin Constraint	Cylinder	3-D
		Interior Only (Not Used)
		Surface Only (Used)
		Interior and Surface (Not Used)
Ball Constraint	Sphere	3-D
		Interior Only (Not Used)
		Surface Only (Used)
		Interior and Surface (Not Used)
Force Load	2-D	2-D
	Polygon	Interior Only (Seldom Used)
	Circle	Boundary Only (Seldom Used)
	3-D	3-D
	Cylinder	Interior Only (Not Used)
	Sphere	Surface Only (Often Used)

Table of Contents

Finite Element Analysis (FEA) Test Bench and Tool

Download the [FEA_Plate zip](#) for testing the FEA Testbench.

1.0 Purpose

2.0 Procedures

2.1 Installation

2.2 Tool

Test Bench: FEA

Requirements Tested

Operation

Test Bench Structure

Step 1

Step 2

Step 3

Step 4

Step 5

Step 6

Description

Metrics

Required Connections to System Under Test

Outputs

	Extrusion	Interior and Surface (Not Used)
Pressure	2-D	2-D
	Polygon	Interior Only (Used Some)
	Circle	Boundary Only (Seldom Used)
	3-D	3-D
	Cylinder	Interior Only (Not Used)
	Sphere	Surface Only (Often Used)
	Extrusion	Interior and Surface (Not Used)
Acceleration	None	N/A

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.

Test Bench: FEA

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.

Operation

The design is assembled into a 3D CAD representation, including the customization / generation of any parameterized components. 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.

Test Bench Structure

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

The general view of the model that will be tested can be seen in Figure 1 below. The plate on top will be referred to as Plate-1 from now on. The plate at the bottom will be referred to as Plate-2. The two plates are connected together with a perfect joint, meaning that there is no slip between the surfaces. Bolts are not included in the model, but with the perfect joint assumption, a bolt connection has been modeled. The FEA test bench connects all parts with perfect joints by default.

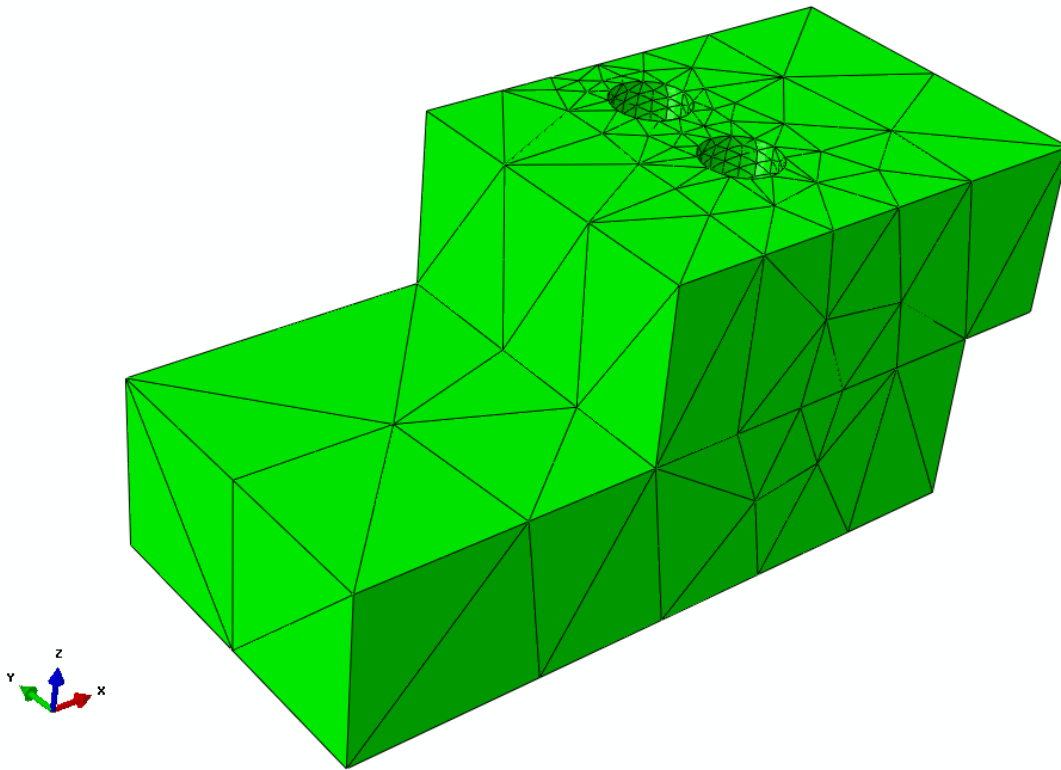


Figure 1

On Plate-1, the points called POINT 1 WALL ATTACHMENT, POINT 2 WALL ATTACHMENT, POINT 3 WALL ATTACHMENT and POINT 4 WALL ATTACHMENT which will be mentioned in this tutorial, are shown in Fig. 2.

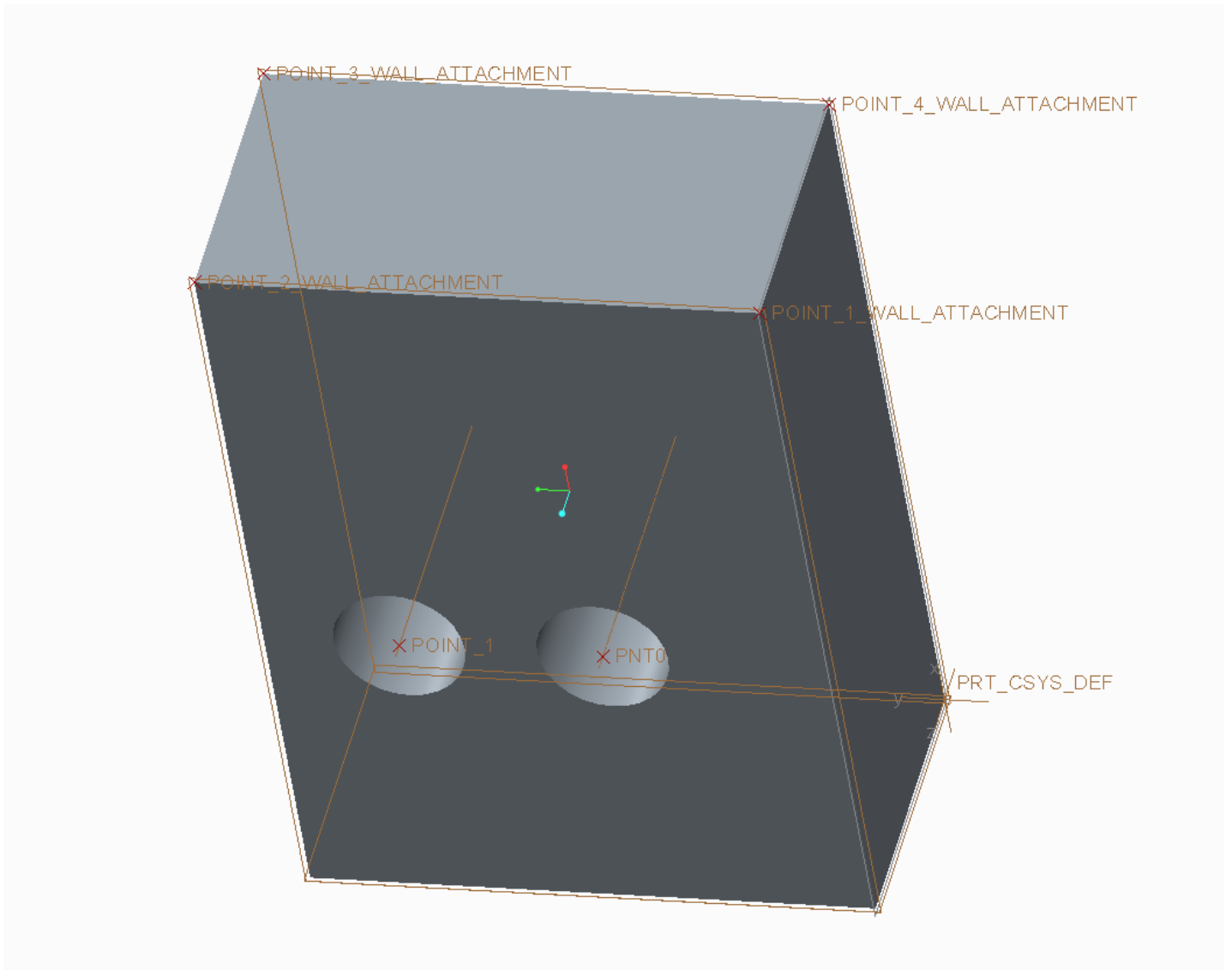


Figure 2

On Plate-2, the points called POINT 1 END LOAD, POINT 2 END LOAD, POINT 3 END LOAD and POINT 4 END LOAD which will be mentioned in this tutorial, are shown in Fig. 3.

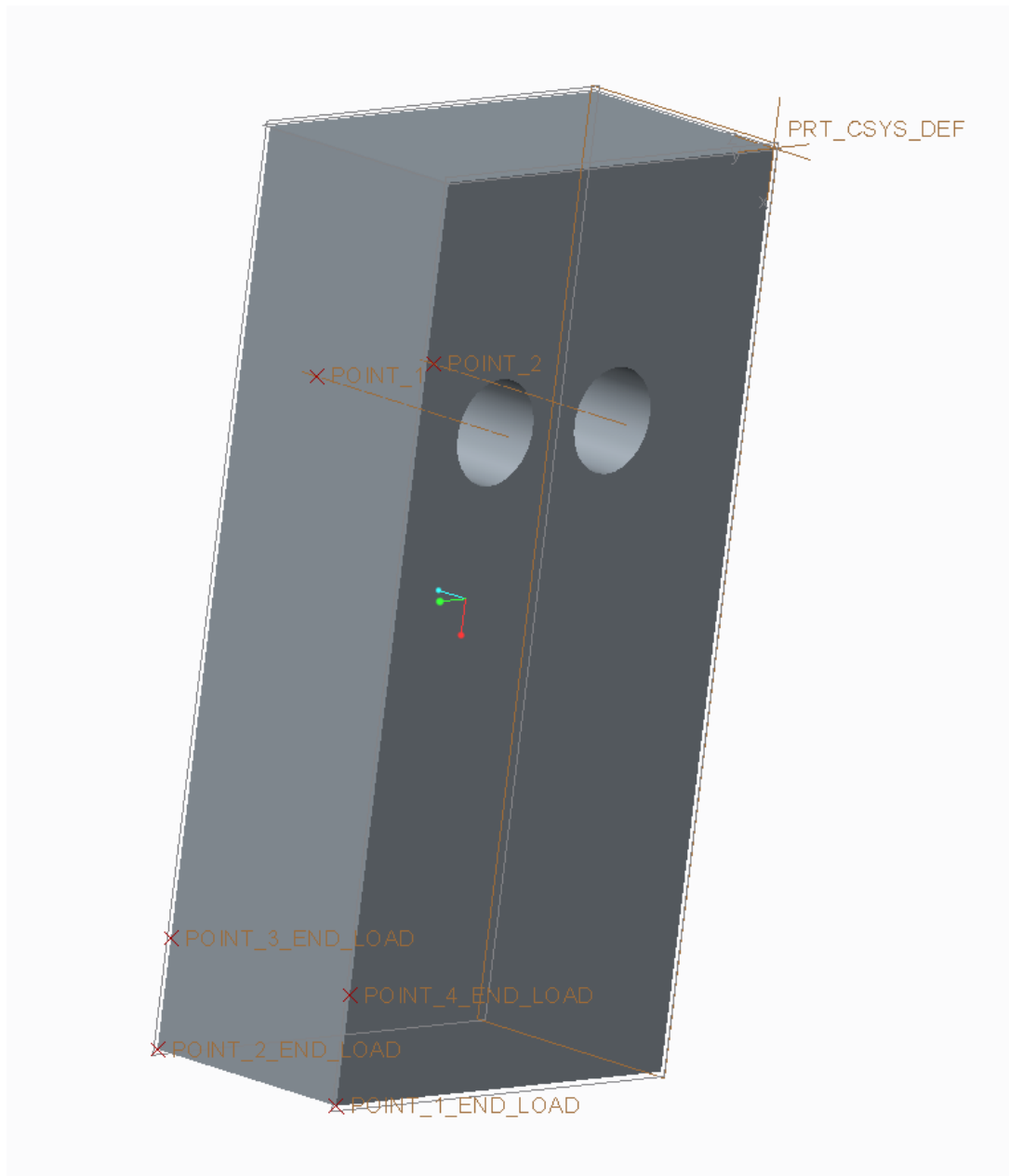


Figure 3

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.

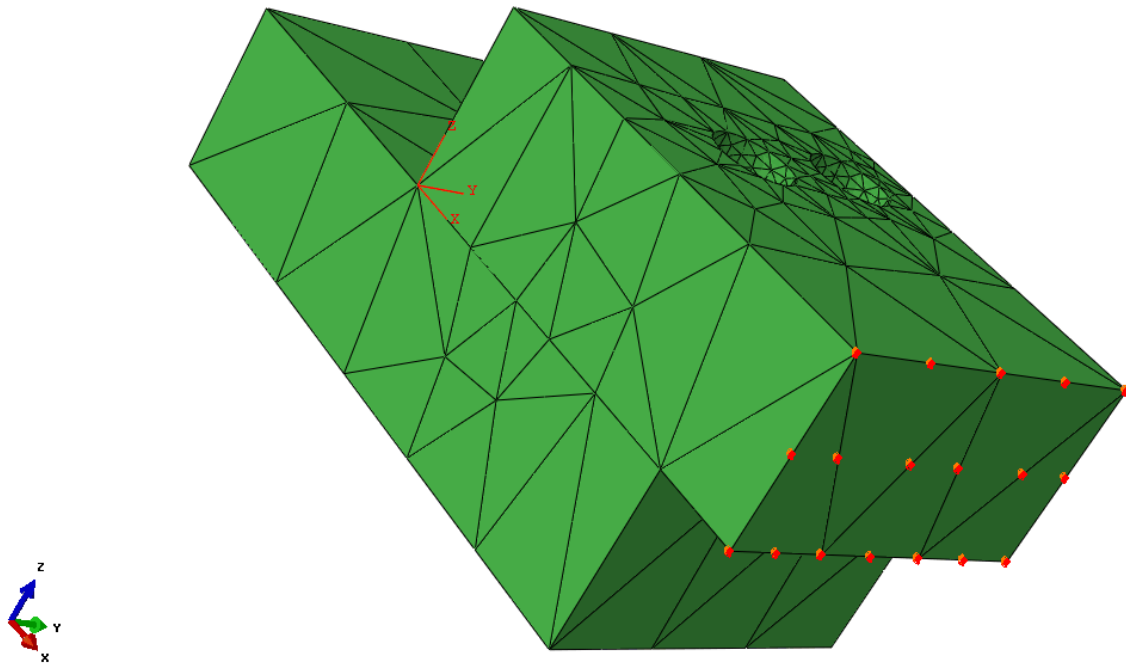


Figure 4

1000N of force will be applied to Plate-2 in y-direction, on the surface between the points POINT 1 END LOAD, POINT 2 END LOAD, POINT 3 END LOAD and POINT 4 END LOAD. Fig. 5.

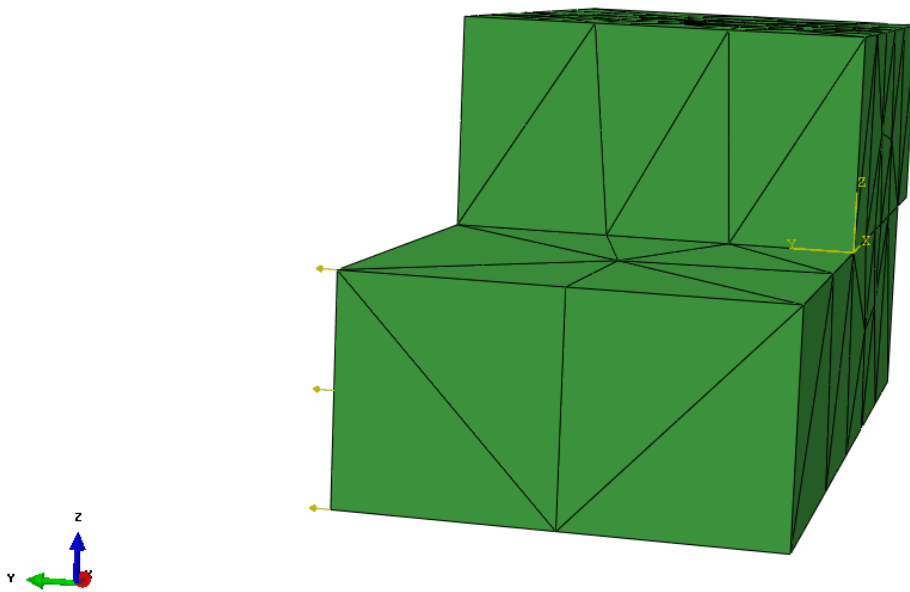


Figure 5

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. For

example, in Plate-2 since there is only 3 edge mesh nodes between the points POINT 1 END LOAD, POINT 2 END LOAD, POINT 3 END LOAD and POINT 4 END LOAD, all the load has been applied to the edge. (Fig. 6)

If you want to define a boundary condition on any part of the model, you must first define the surface to which the boundary conditions will be applied. You can do this by defining points manually in a CAD program, such as Creo.

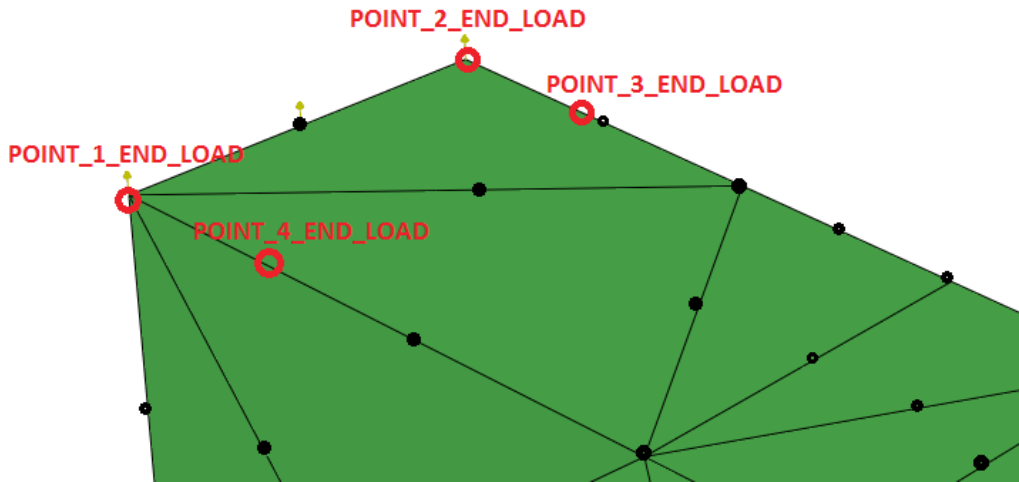


Figure 6

Step 1

Now we will import our components. We have two options to do that:

1st Option (if an acm file is present):

In GME, on top of the screen, click "MGA.Interpreter.CyPhyComponentImporter". (Fig. 7)

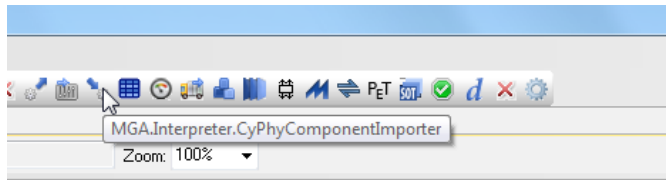


Figure 7

Then select the correct acm file corresponding to your components and click Open. (Fig. 8)

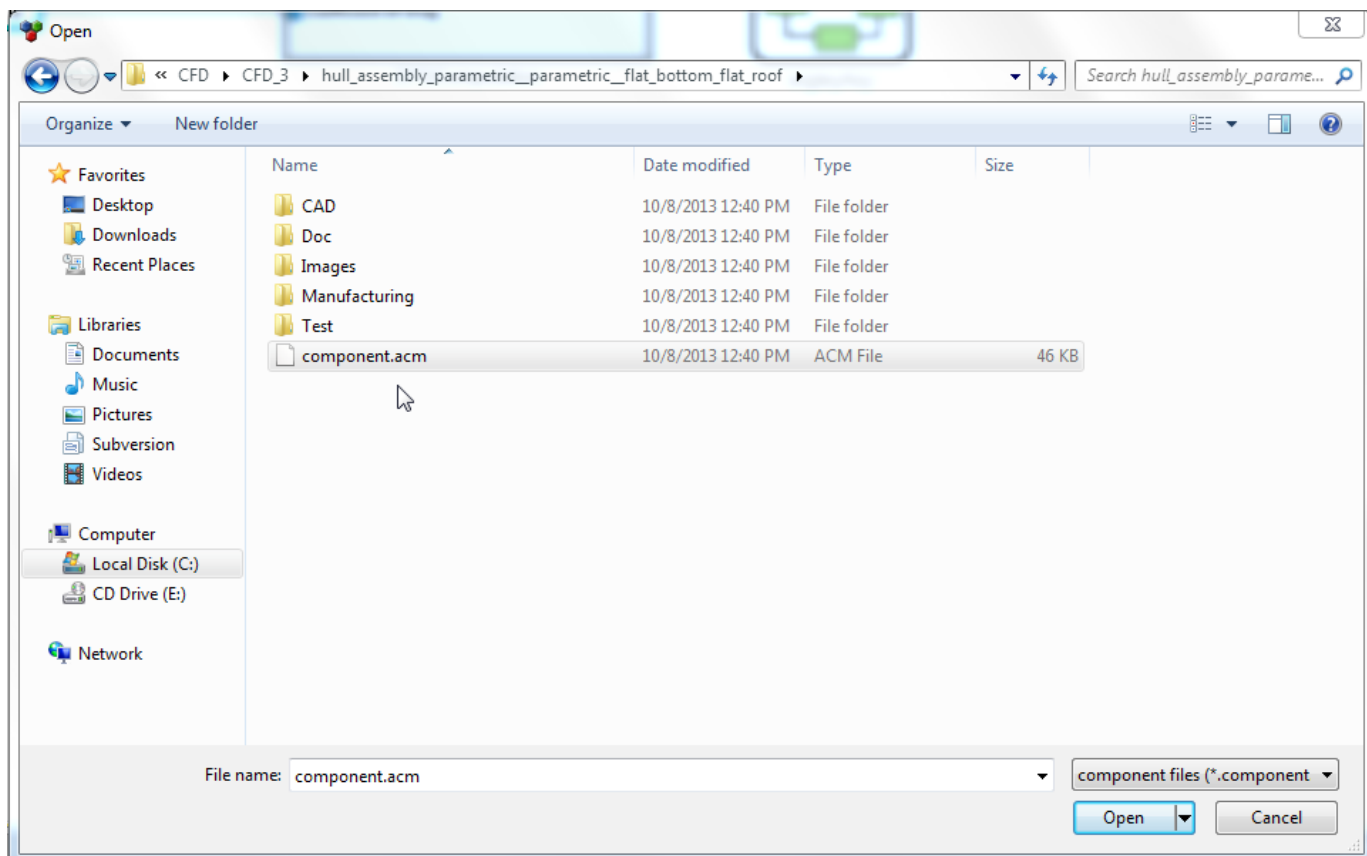


Figure 8

CyPhy will automatically import your components into the GME. Now you should be able to see your components under the "Imported_Components" folder (Fig. 9). CAD parts should have the same material definitions with CyPhy. If that is not the case, modify CAD parts the same material definitions and make it the same with CyPhy.

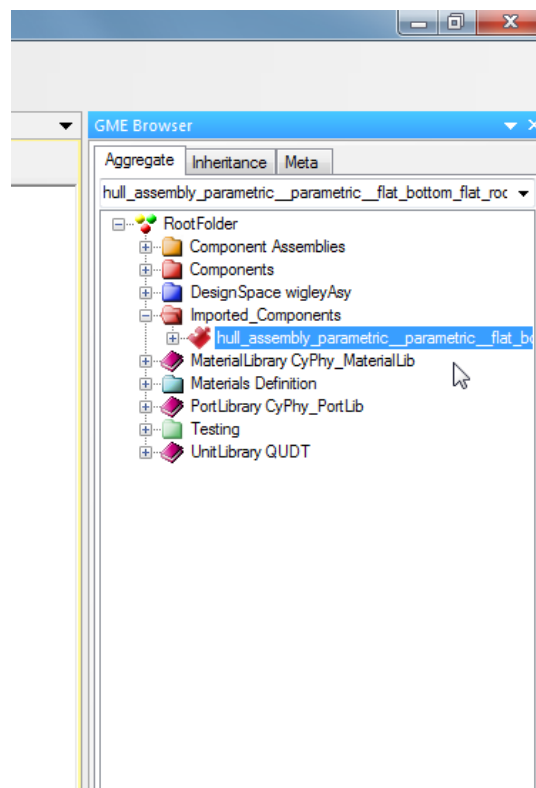


Figure 9

2nd Option (if there is no acm file):

Note:

If you are using Version 13.17 or later, skip to Step 2 and make sure that material name is correct in the Creo model. For more details, read the "Note" right before Step 2.

In the GME Browser, expand "MaterialLibrary CyPhy MaterialLib" object and then expand "MaterialDefinitions" folder. (Fig. 10)

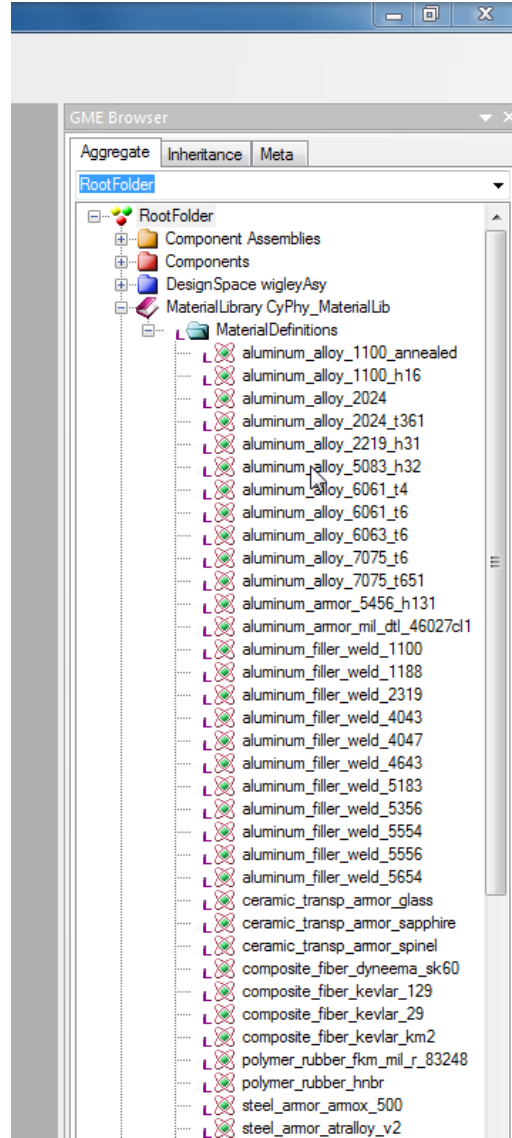


Figure 10

In this tutorial, "steel_armor_cast_mil_a_11356cl1" material is used as an example. Right-click on the "steel_armor_cast_mil_a_11356cl1" material and select Copy. (Fig. 11)

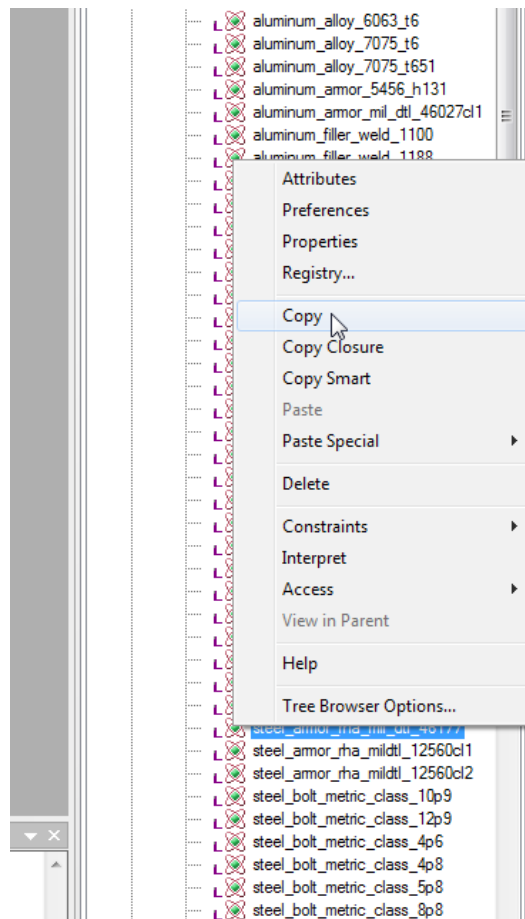


Figure 11

Then Right-click on the materials definition folder and select Paste. (Fig. 12)

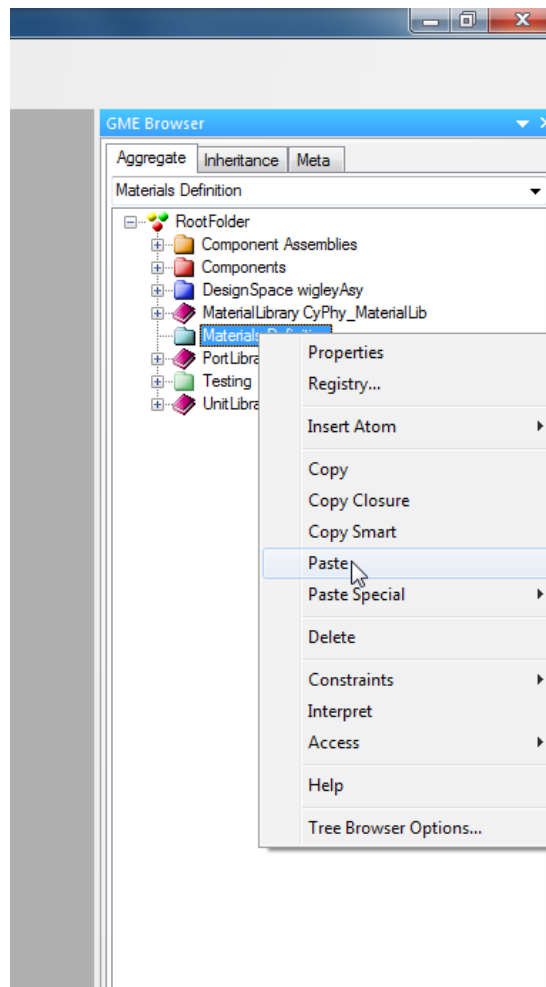


Figure 12

After the Materials Definition is created, it will be added to the component. Open the Components folder and double click on the component.

Then on the upper left corner in the Part Browser, click on the SolidModeling aspect. (Fig. 13)

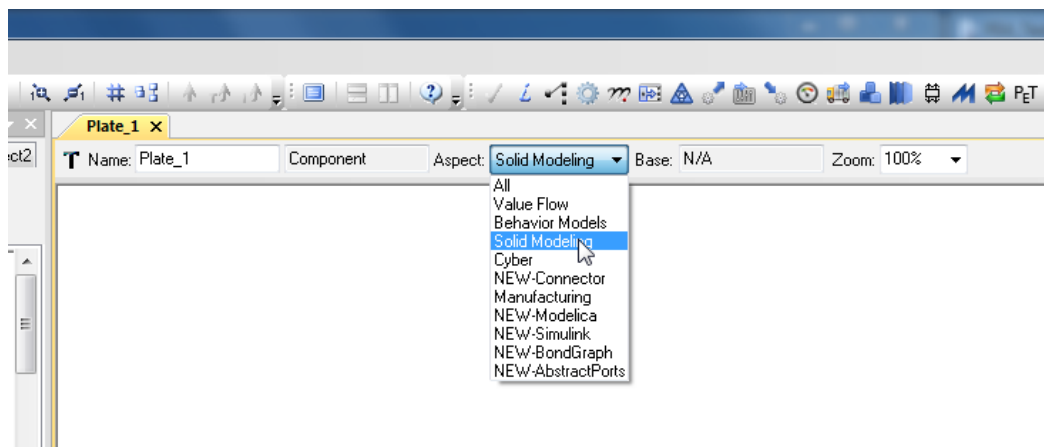


Figure 13

In the GME Browser, right click on the material and copy. (Fig. 14)

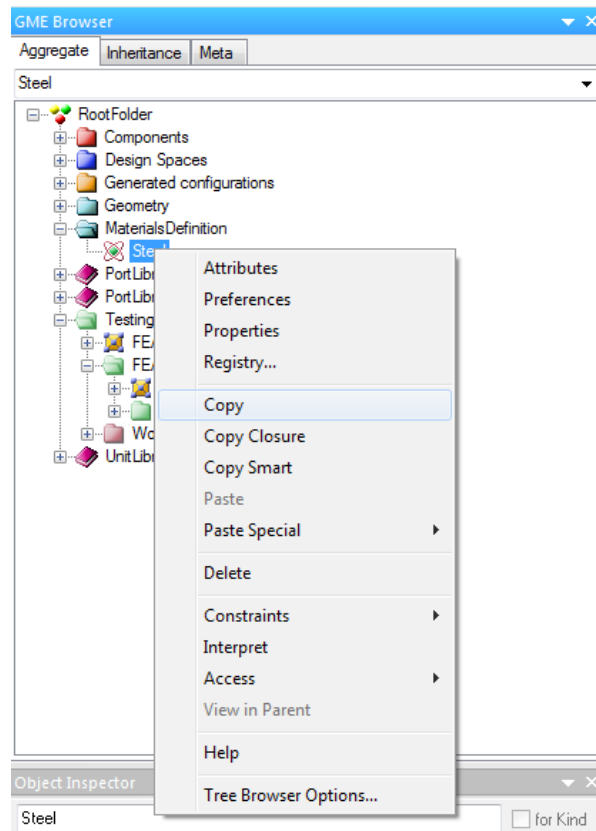


Figure 14

Right click in the part, and paste the material as reference. Pasting as a reference is critical (Fig. 15). CAD parts should have the same material definitions with CyPhy. If that is not the case, modify the CAD parts material definitions and make it the same with CyPhy.

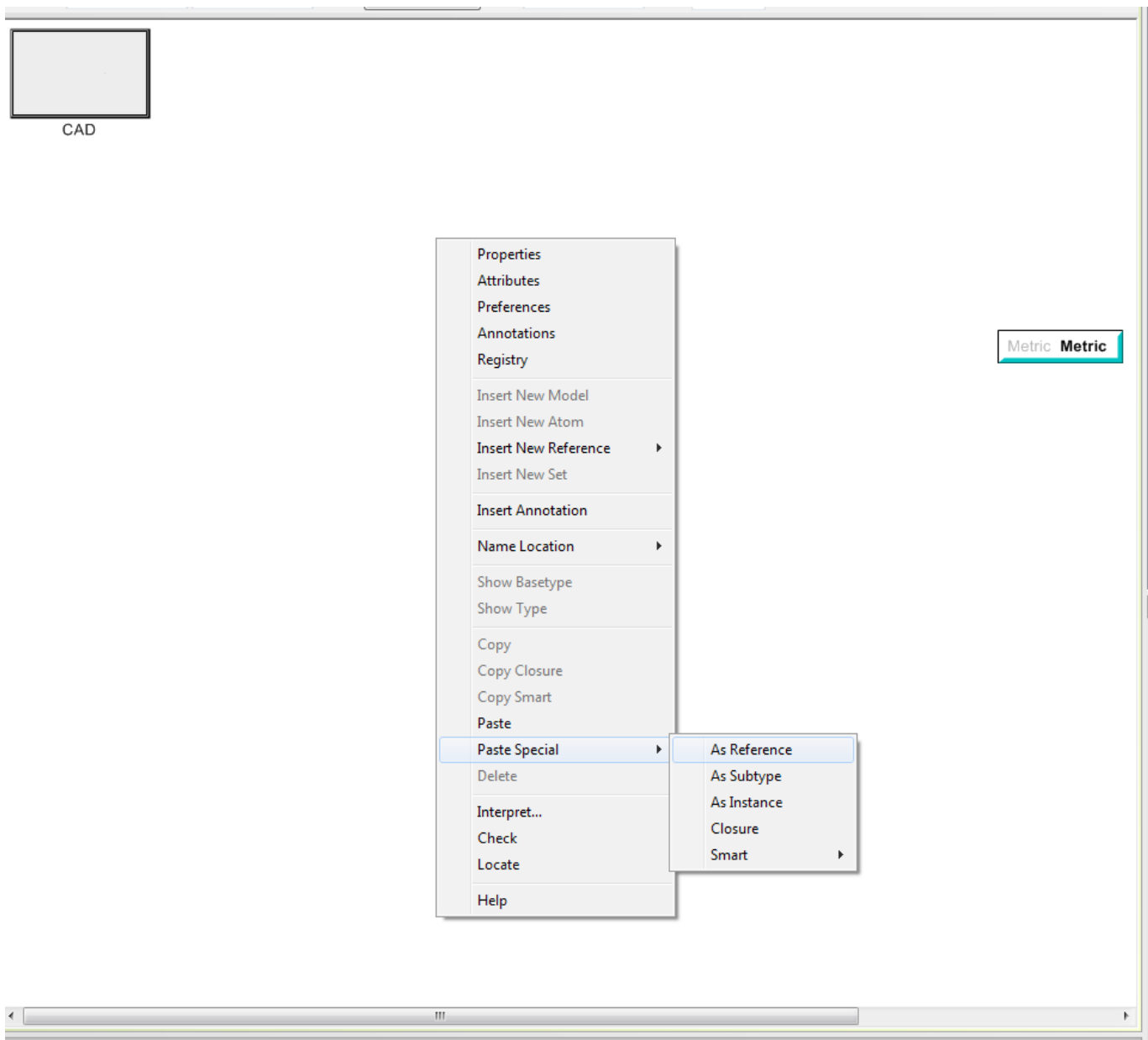


Figure 15

Note:

For 13.16, the material name in CyPhy must be the same as the material name in the Creo model. The material allowables (yield strength, ultimate strength...) are retrieved from the material library and Young's modulus, Poisson's ratio, and density are retrieved from the Creo Model. For FEA with 13.17, there is no need to enter a material name in CyPhy. FEA will rely on the material name in the Creo model. All material properties will come from the material library (this includes Young's modulus, Poisson's ' ratio, and density).

Step 2

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. 16)

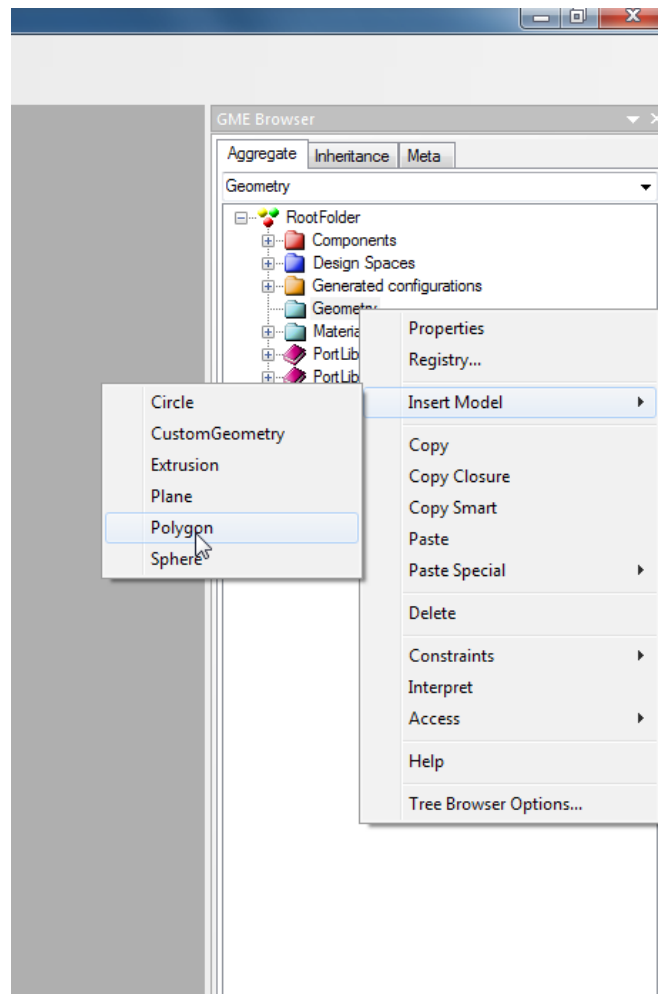


Figure 16

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. 17)

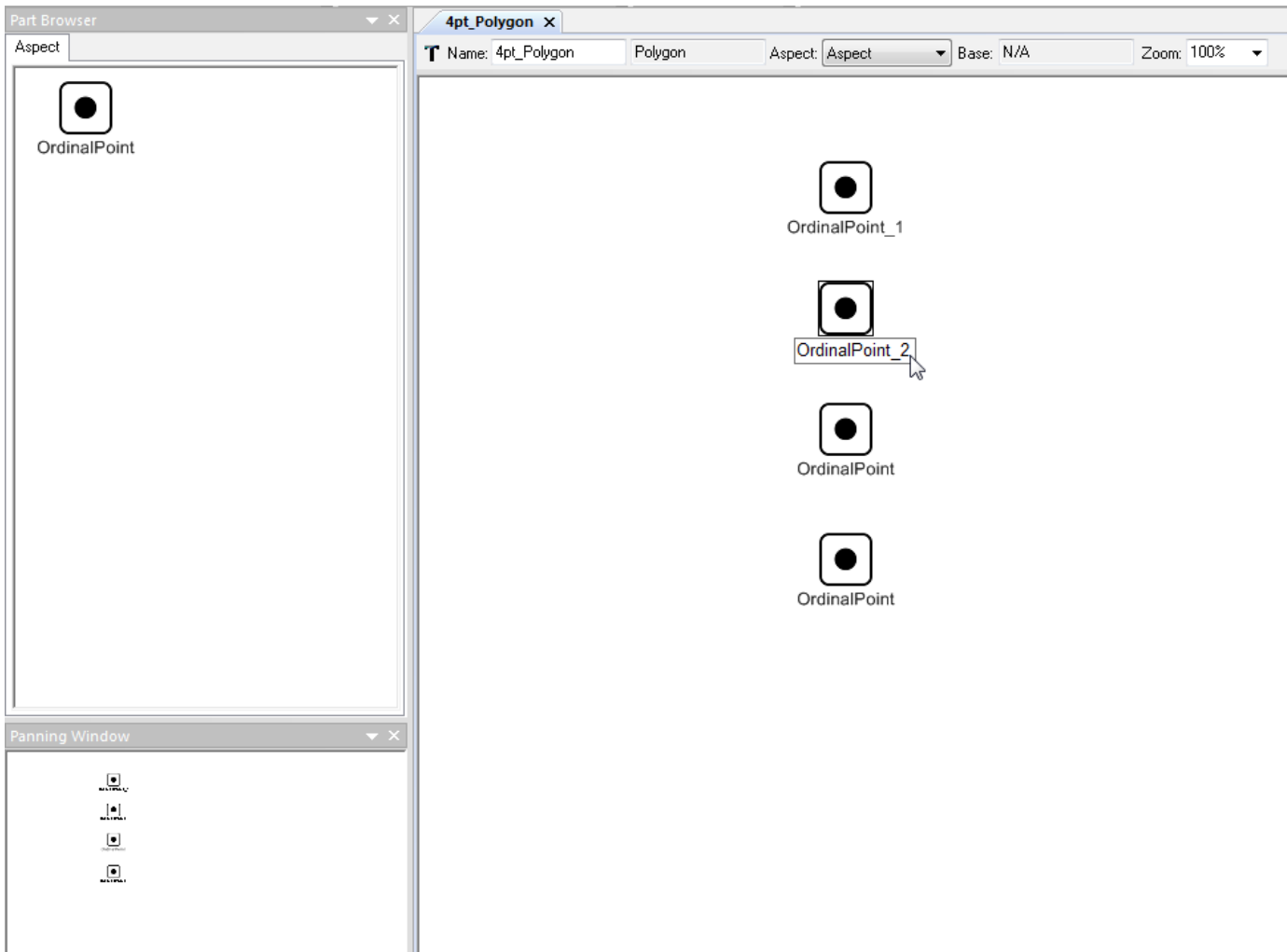


Figure 17

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. 18)

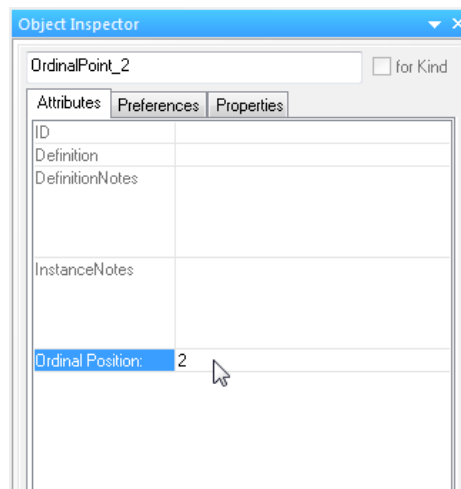


Figure 18

Step 3

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. 19)

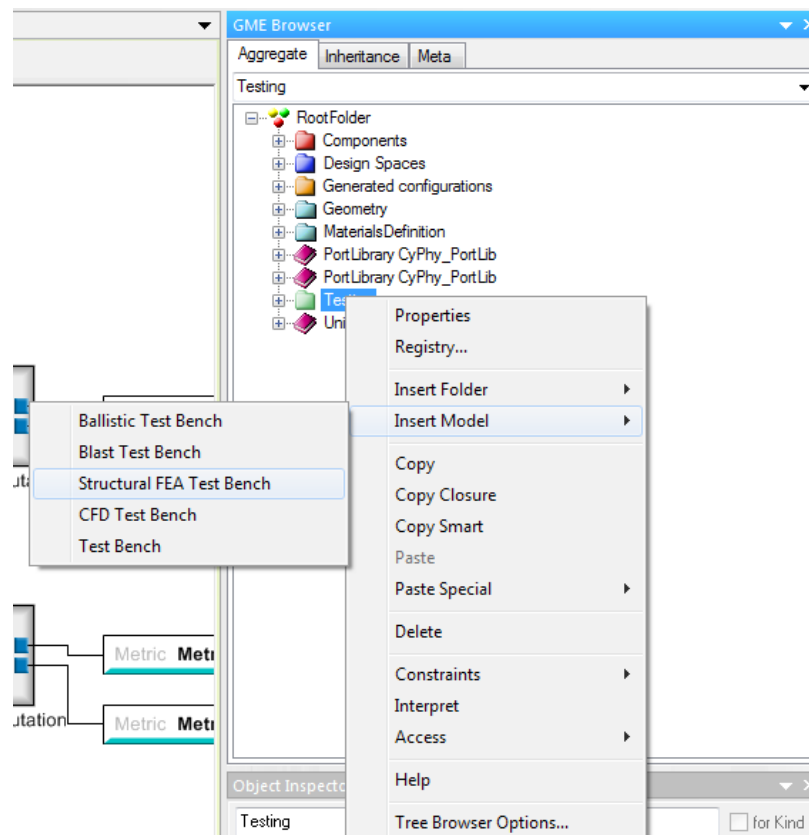


Figure 19

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. 20)

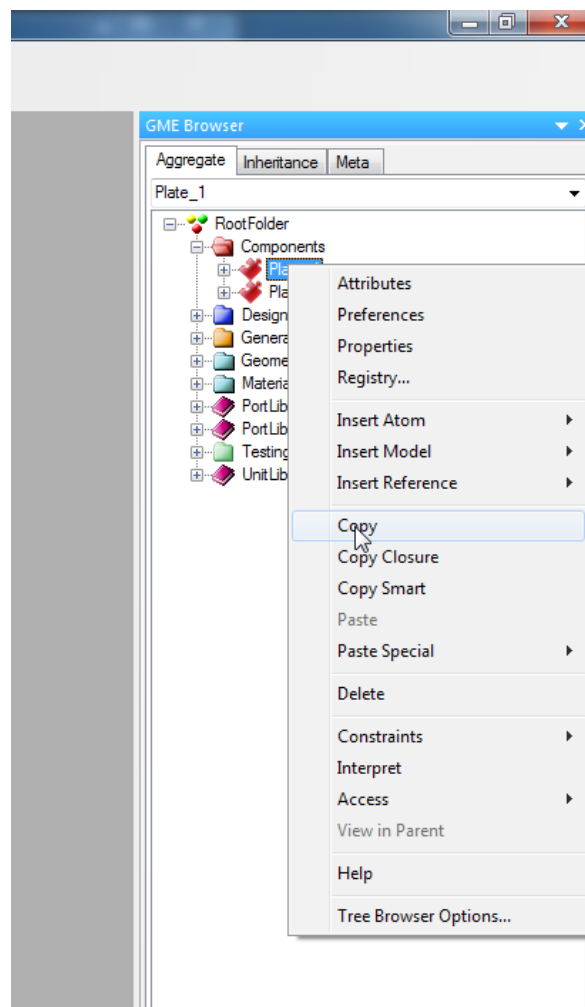


Figure 20

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. 21)

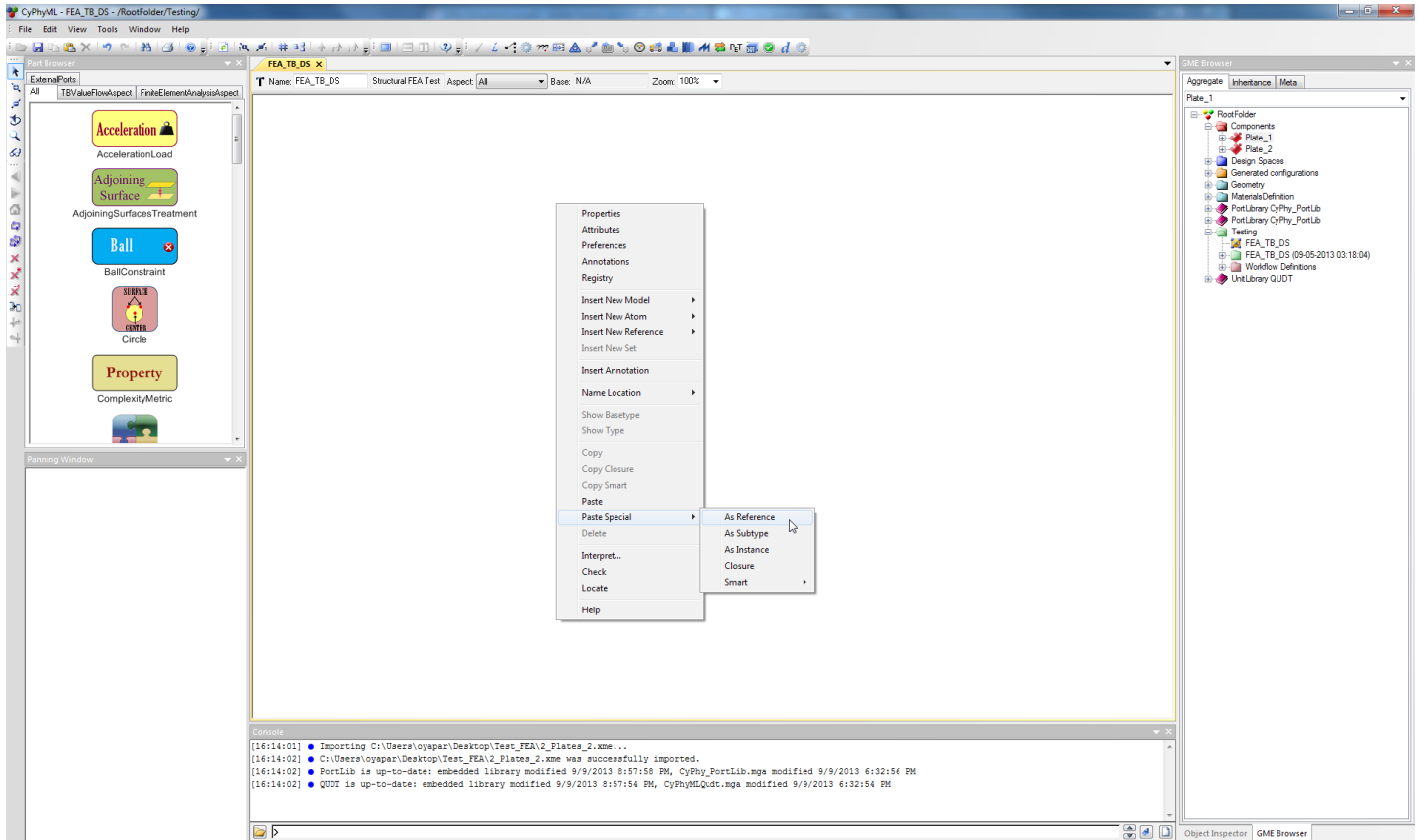


Figure 21

In the screen that has been popped up, select TestInjectionPoint. Repeat that step for each part. (Fig. 22)

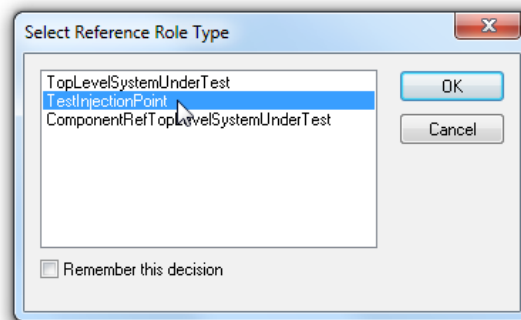


Figure 22

Step 4

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. 23)

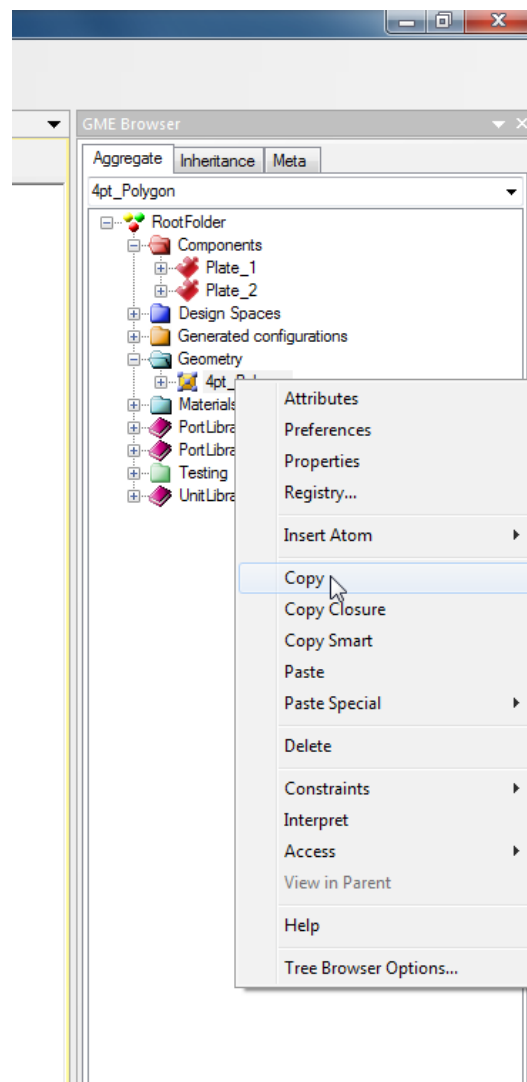


Figure 23

Then double-click on the created FEA testbench under the Testing folder. In the test bench, right-click and select Paste Special and Paste as Subtype or Instance (either one can be picked). Repeat that step for each part. (Fig. 24)

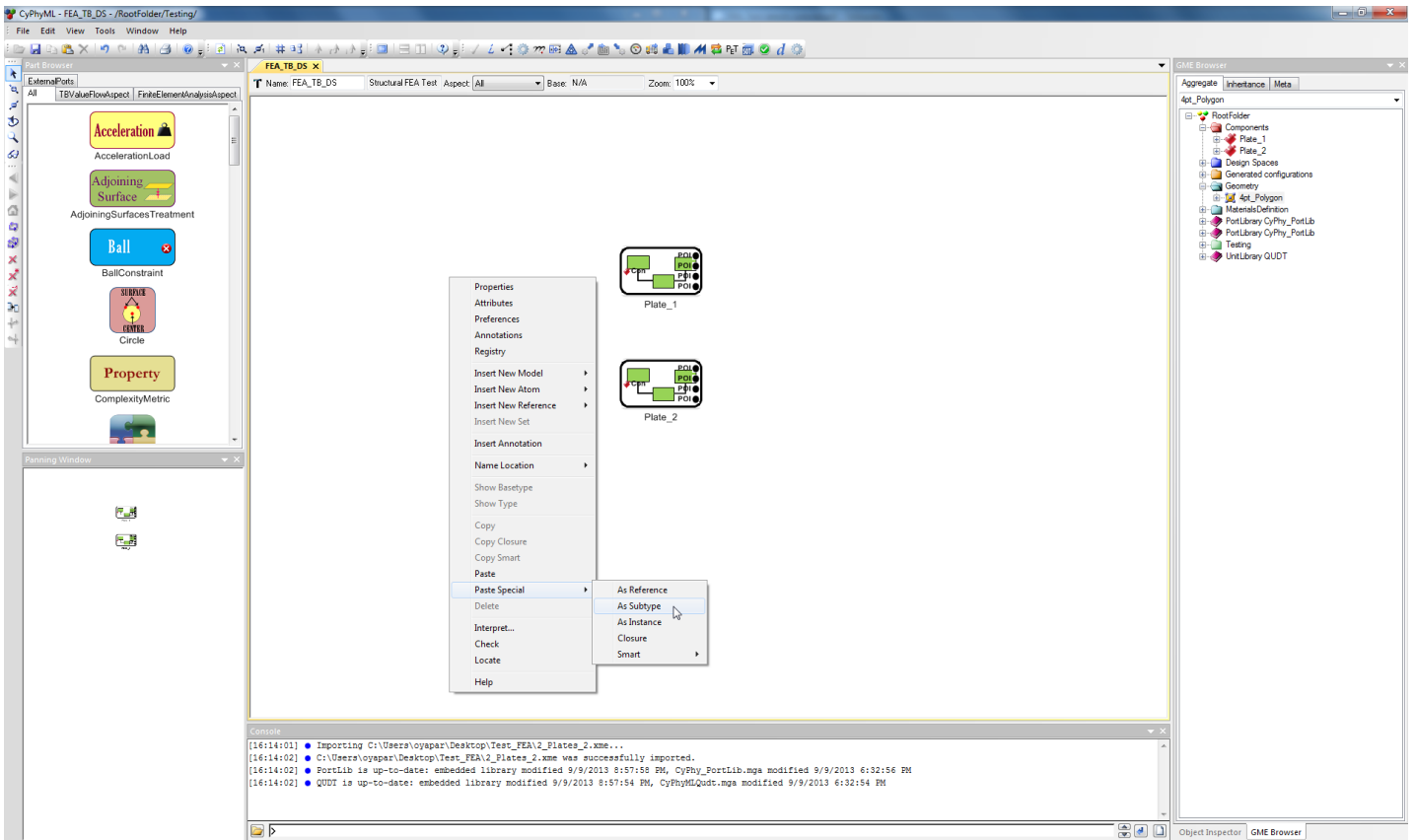


Figure 24

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. When you are done, Press (Ctrl+1) to go back to the edit mode. (Fig. 25)

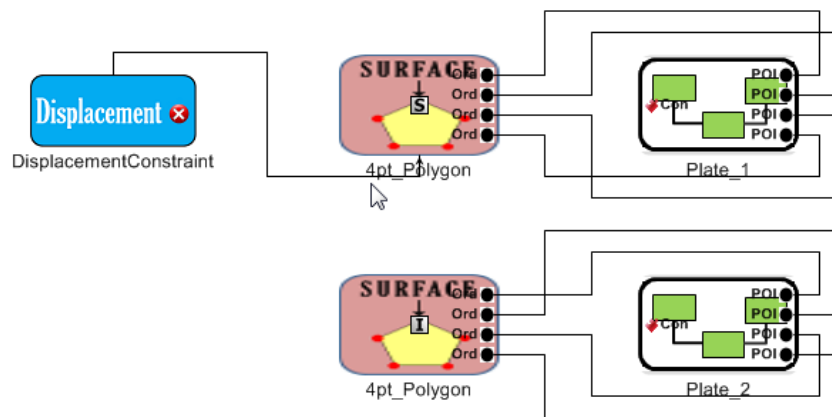


Figure 25

For defining displacement constraints, drag and drop the DisplacementConstraint part from the Part Browser into the test bench. (Fig. 26)

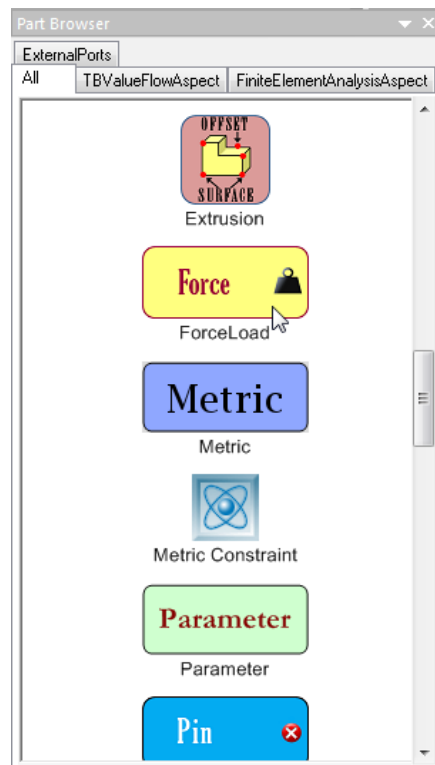


Figure 26

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. 27)

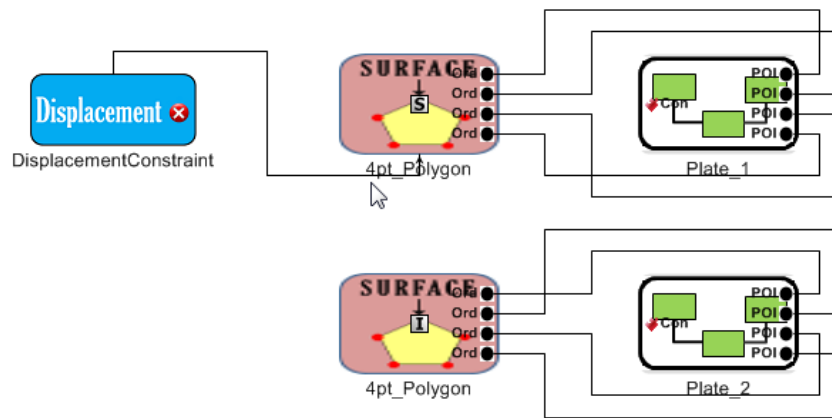


Figure 27

Now we will get into the DisplacementConstraint. Double-click on the DisplacementConstraint. For defining translation and rotation, drag and drop Translation and Rotation parts into the DisplacementConstraint. (Fig. 28)

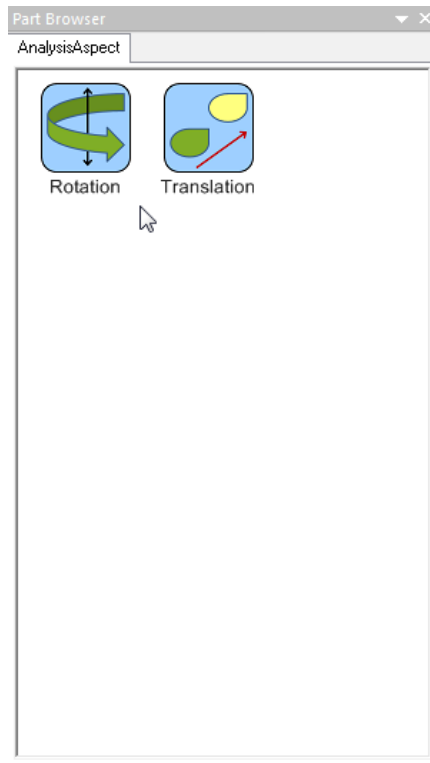


Figure 28

1. 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. 29)

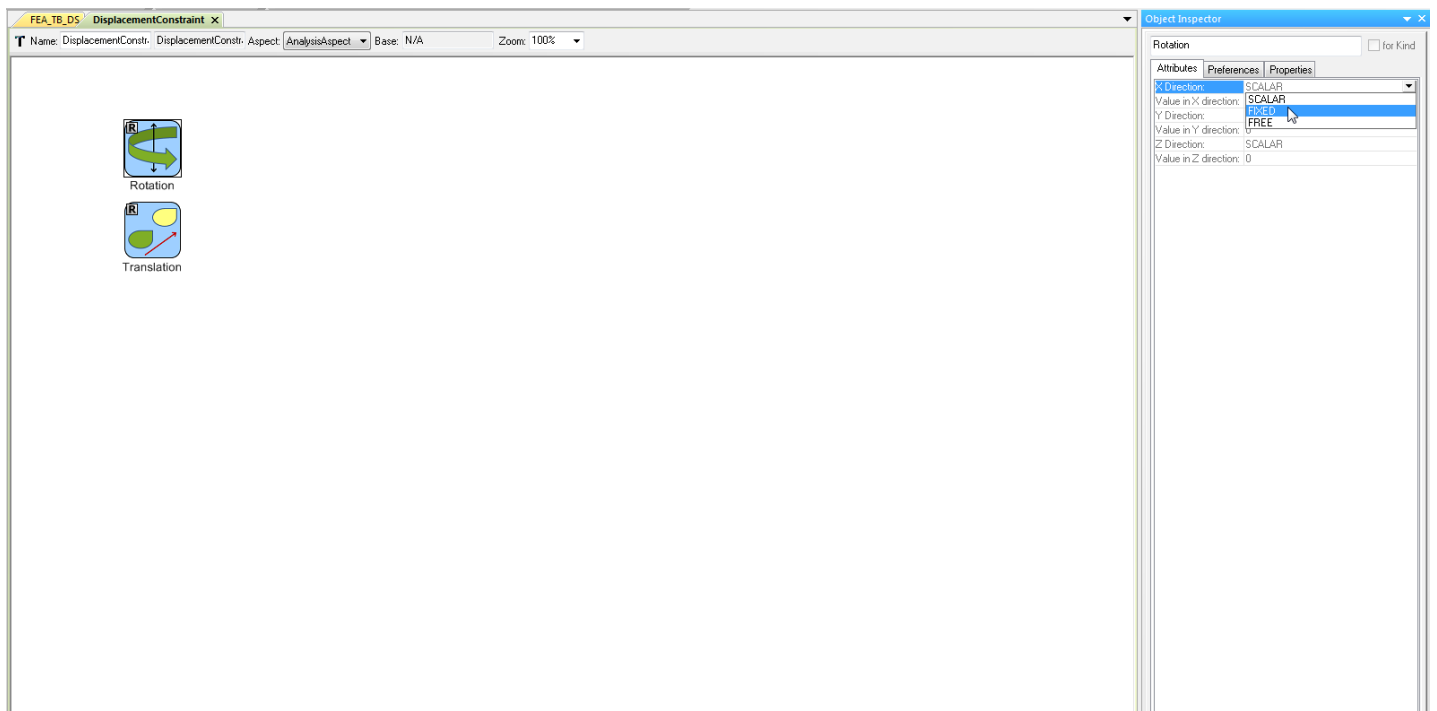


Figure 29

For defining forces and moments, drag and drop the ForceLoad part from the Part Browser into the test bench. (Fig. 30)

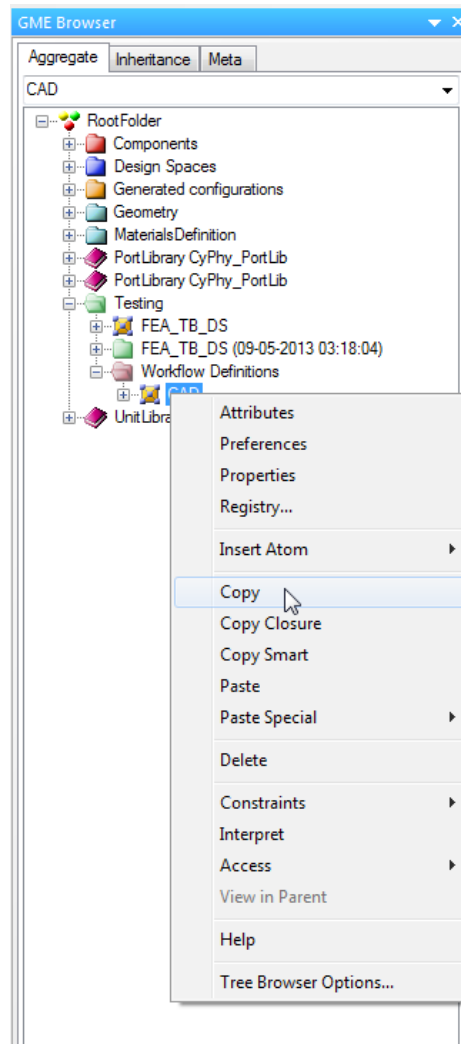


Figure 30

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. 31)

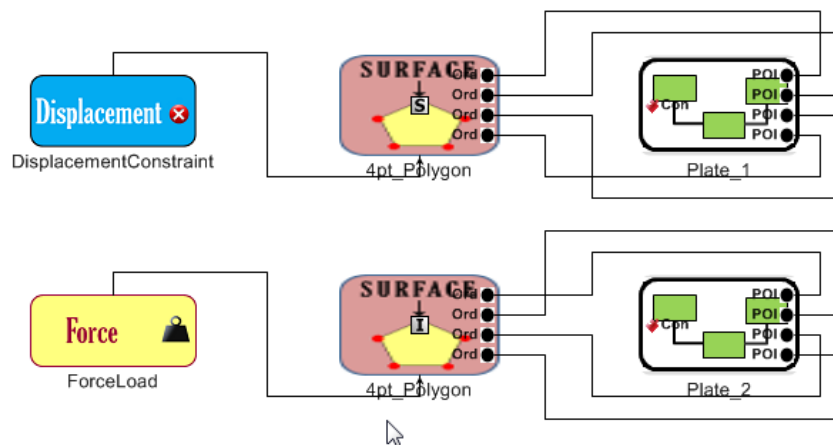


Figure 31

Step 5

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. 32)

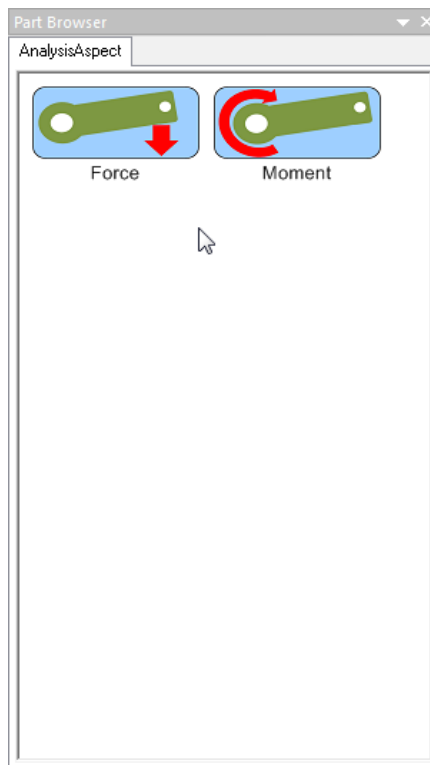


Figure 32

Right-click on force or moment and choose Attributes to define the forces and moments in each direction (x, y, z). (Fig. 33)

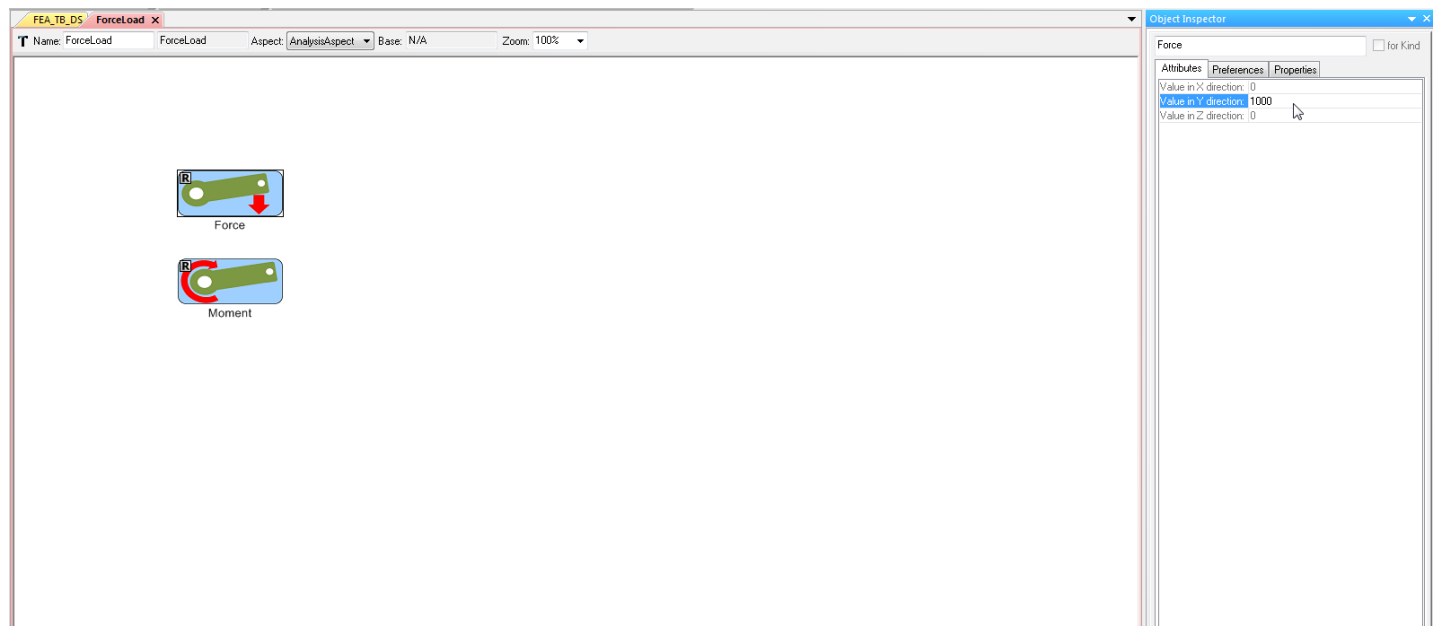


Figure 33

Under Testing, under Workflow Definitions, right-click on the workflow definition (which has been named as CAD in Figure 32) and select copy. (Fig. 34)

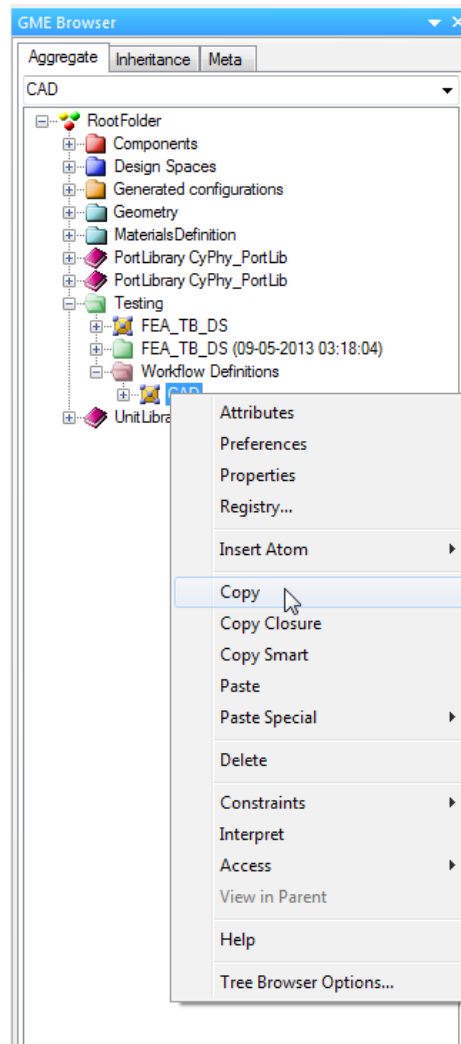


Figure 34

Then in the FEA test bench, right-click and select Paste special and Paste as Reference. (Fig. 35)

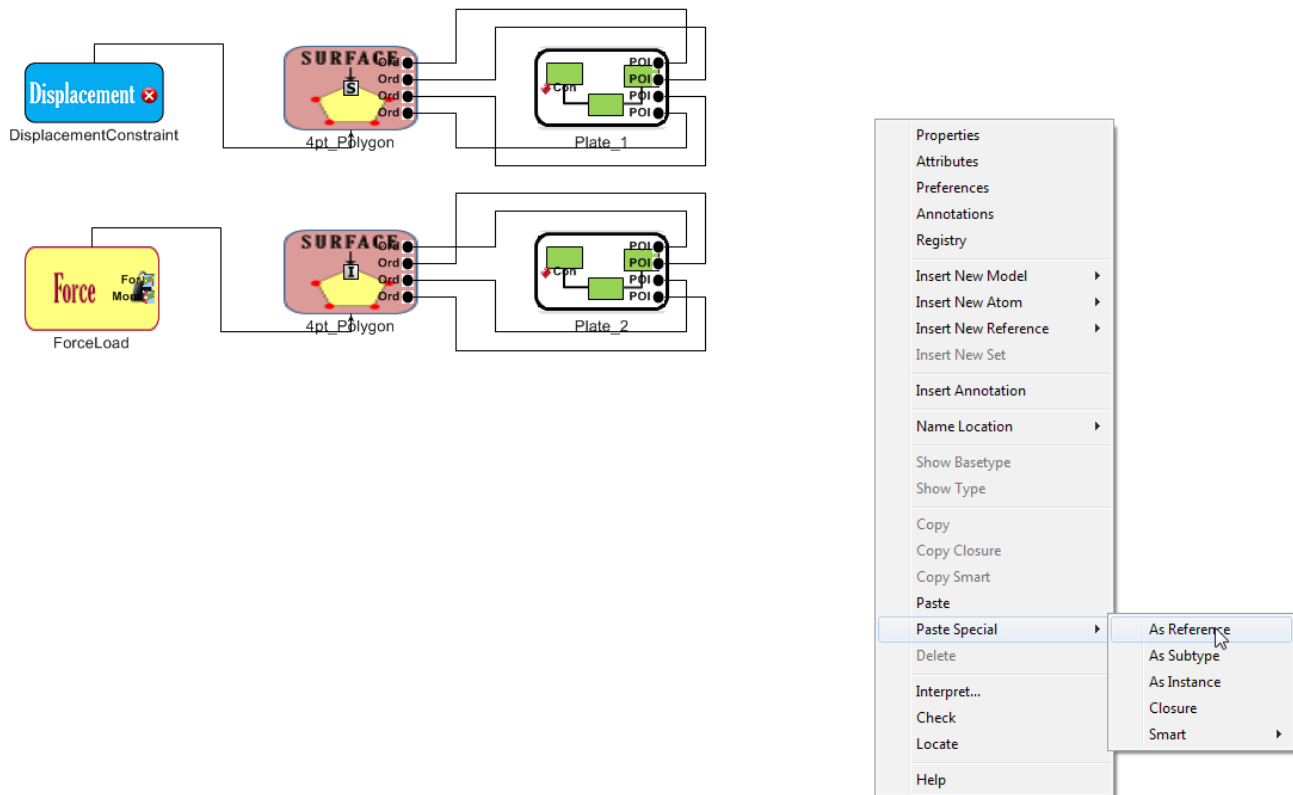


Figure 35

Under design spaces, right-click on the design space (which has been named as 2 plates in Figure 34) and select copy. (Fig. 36)

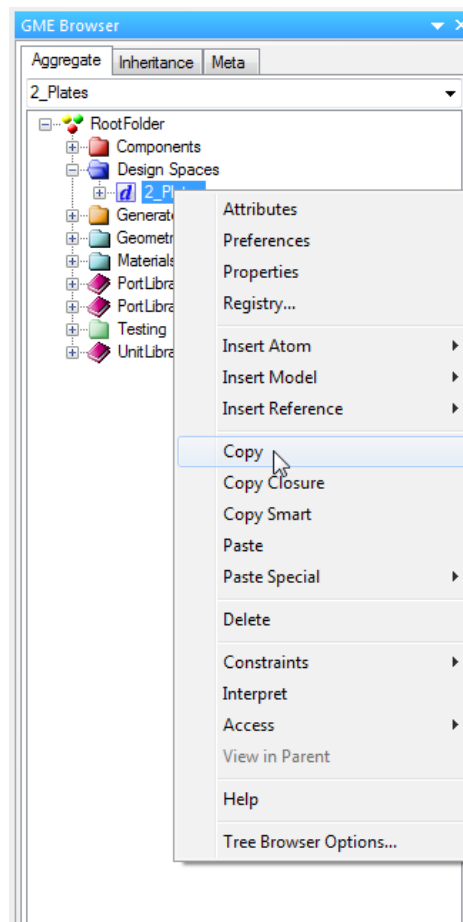


Figure 36

Then in the FEA test bench, right-click and select Paste special and Paste as Reference. (Fig. 37)

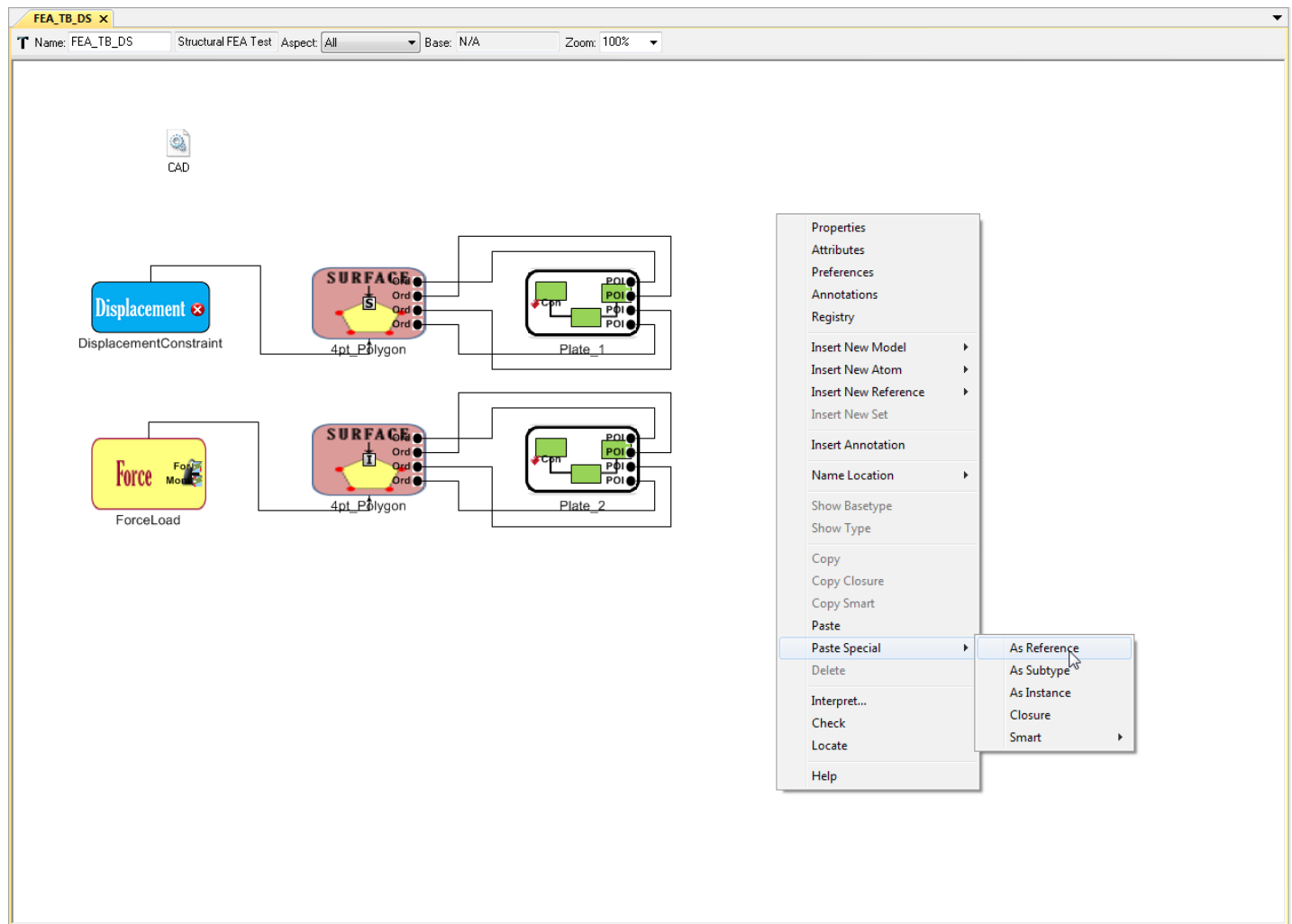


Figure 37

In the screen that has been popped up, select TopLevelSystemUnderTest. (Fig. 38)

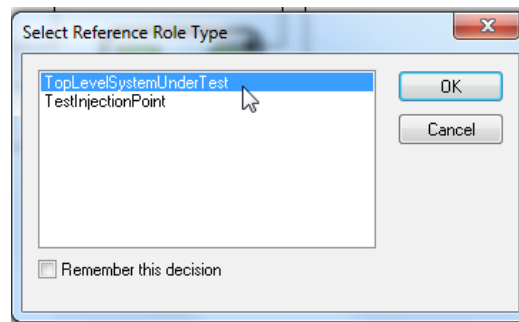


Figure 38

Drag the StructuralFEAComputation part from the Part Browser and drop it into the FEA test bench. Repeat that step for each part. (Fig. 39)

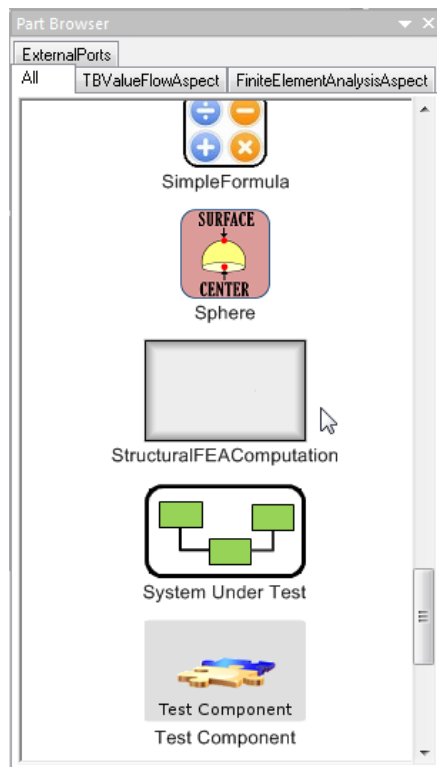


Figure 39

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. 40)

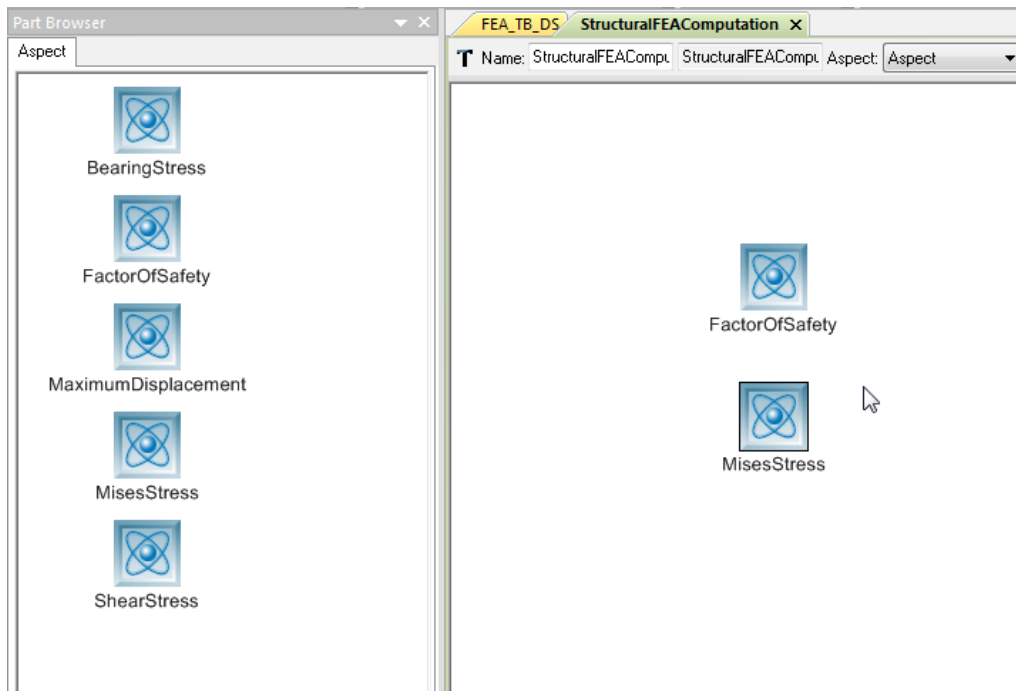


Figure 40

The Test Bench should now look like this. (Fig. 41)

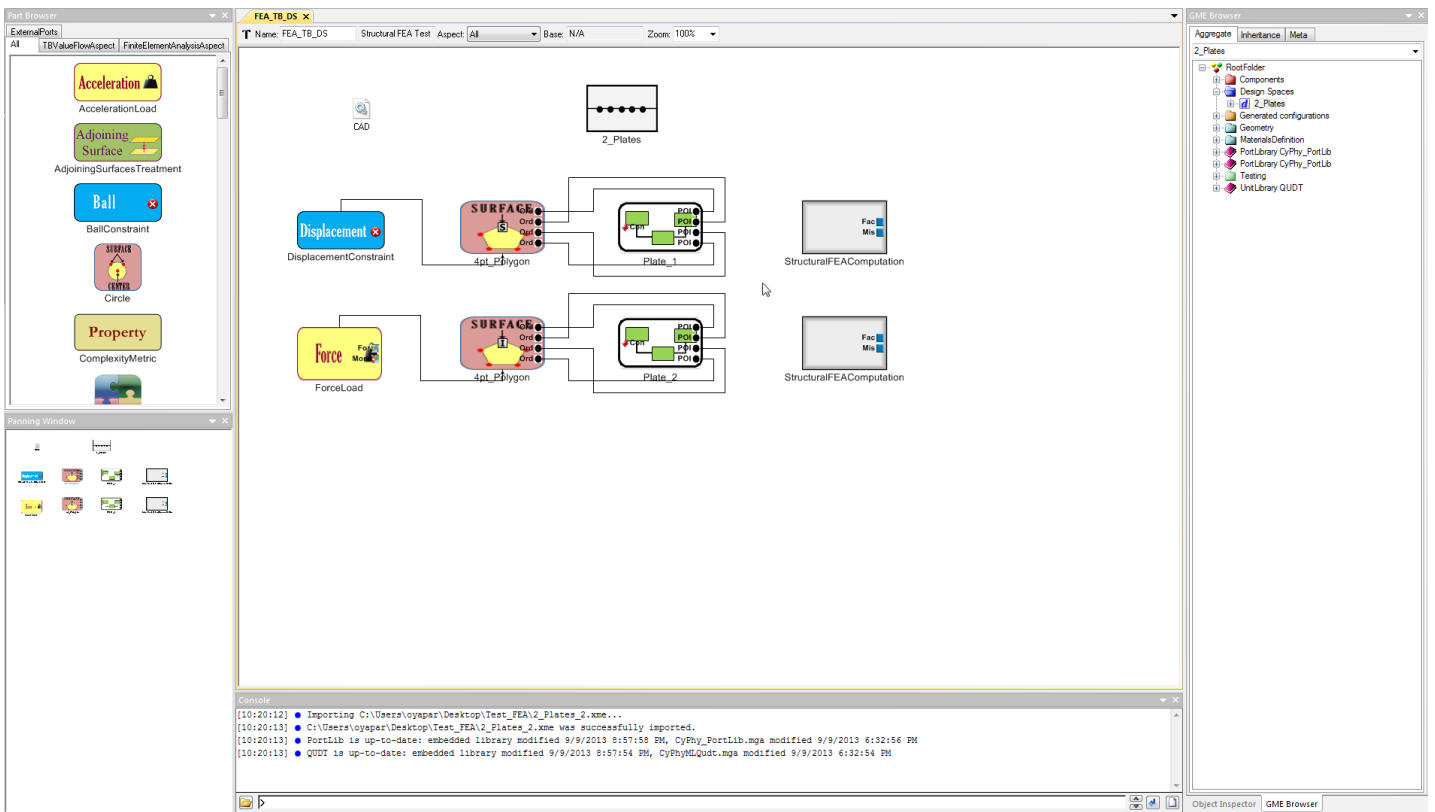


Figure 41

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. 42)

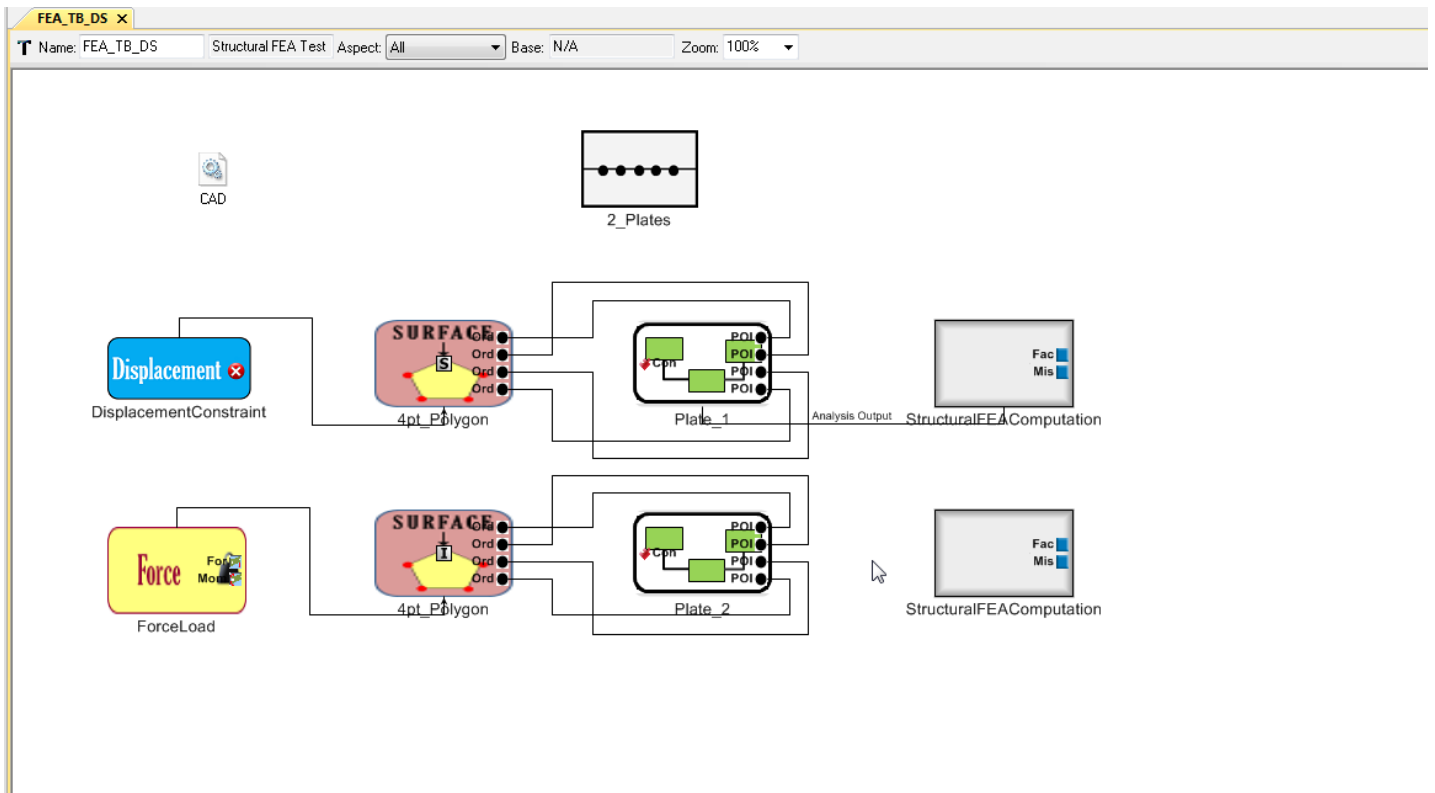


Figure 42

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. 43)

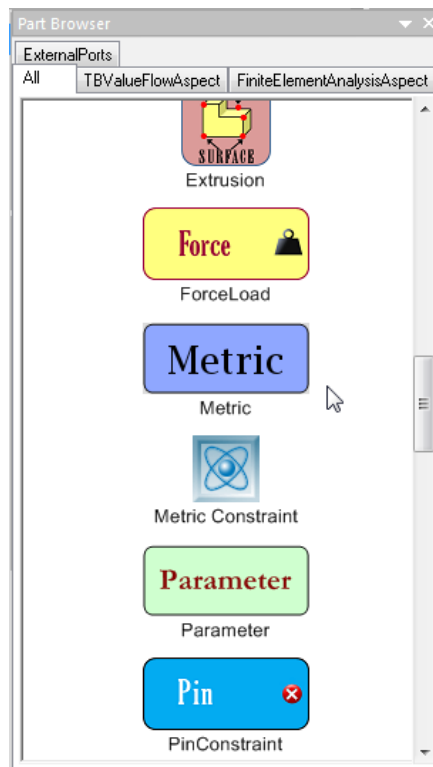


Figure 43

The Test Bench should now look like this. (Fig. 44)

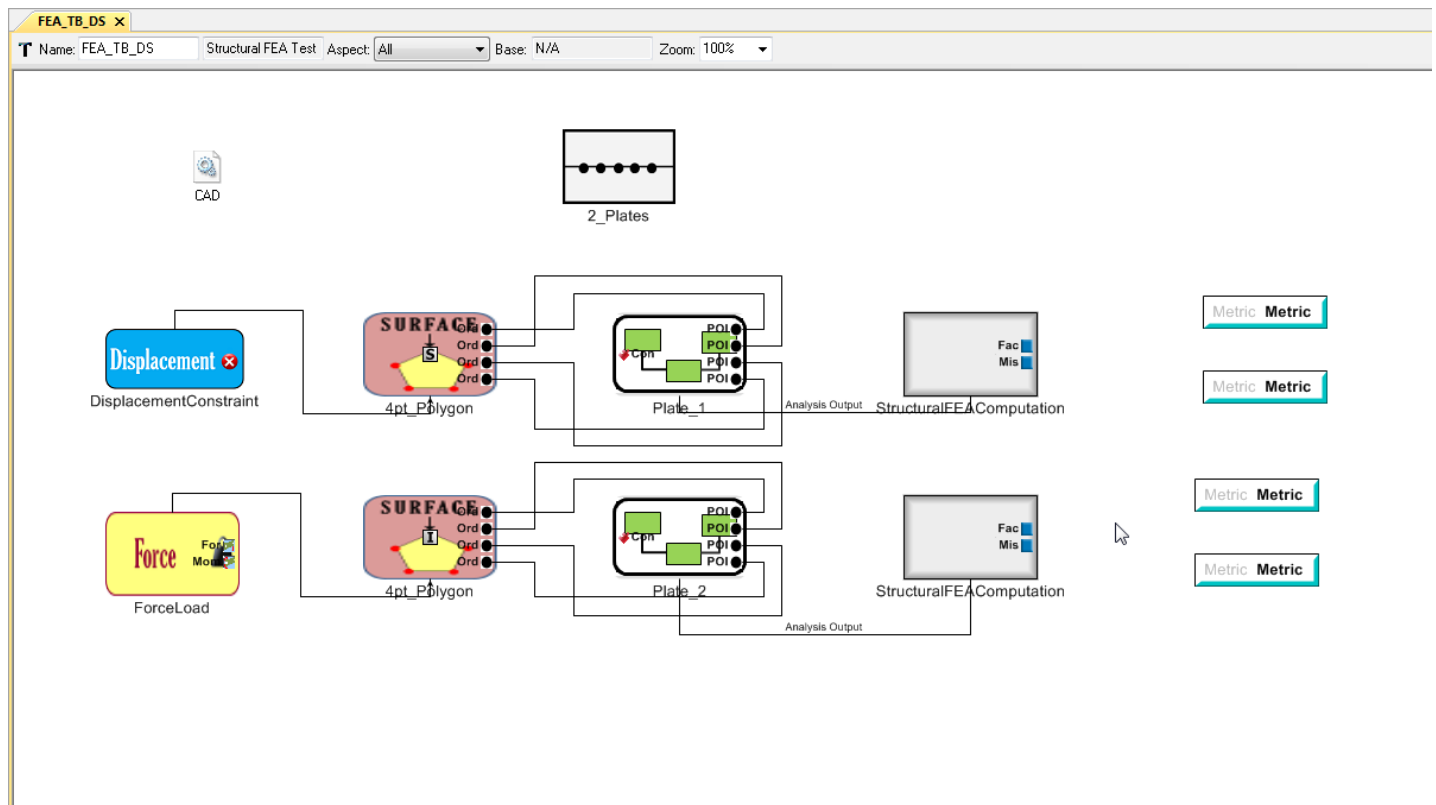


Figure 44

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. 45)

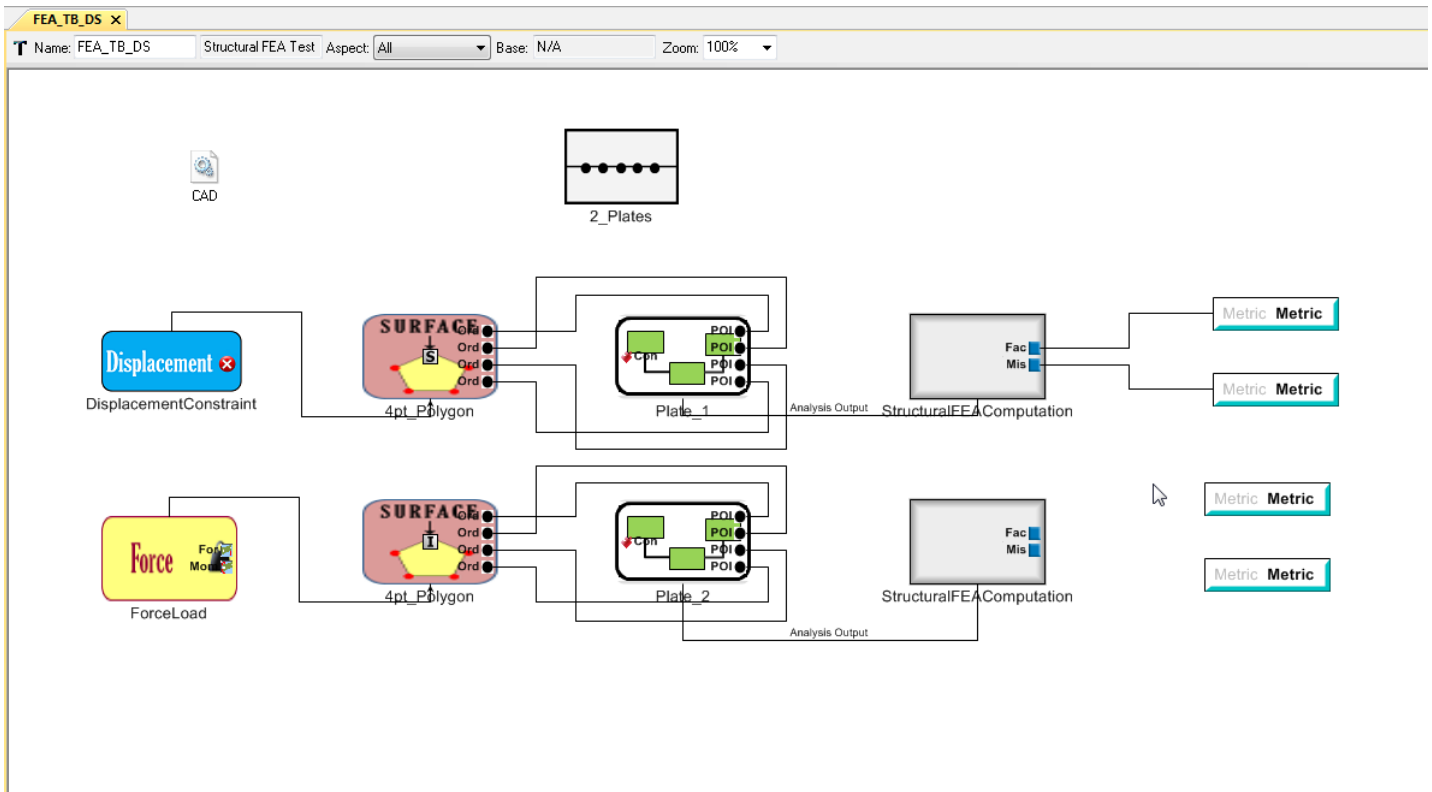


Figure 45

Finally, your test bench should look like this. (Fig. 46)

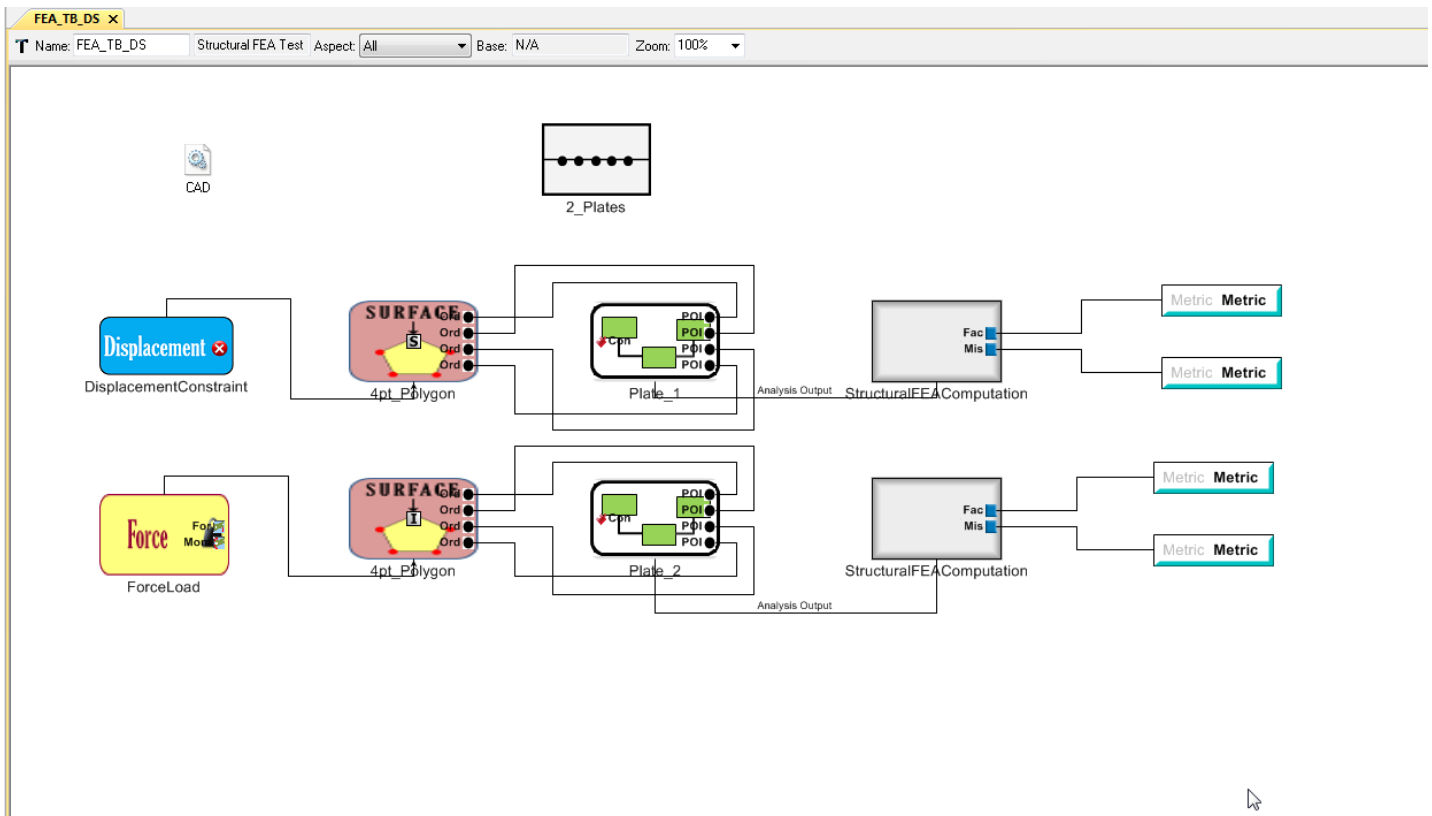


Figure 46

Step 6

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. 47)

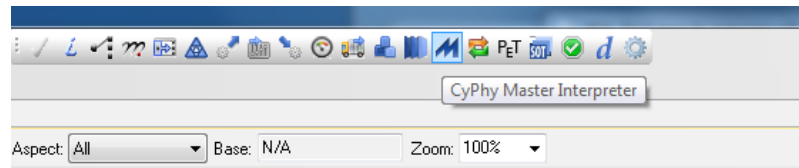


Figure 47

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. 48)

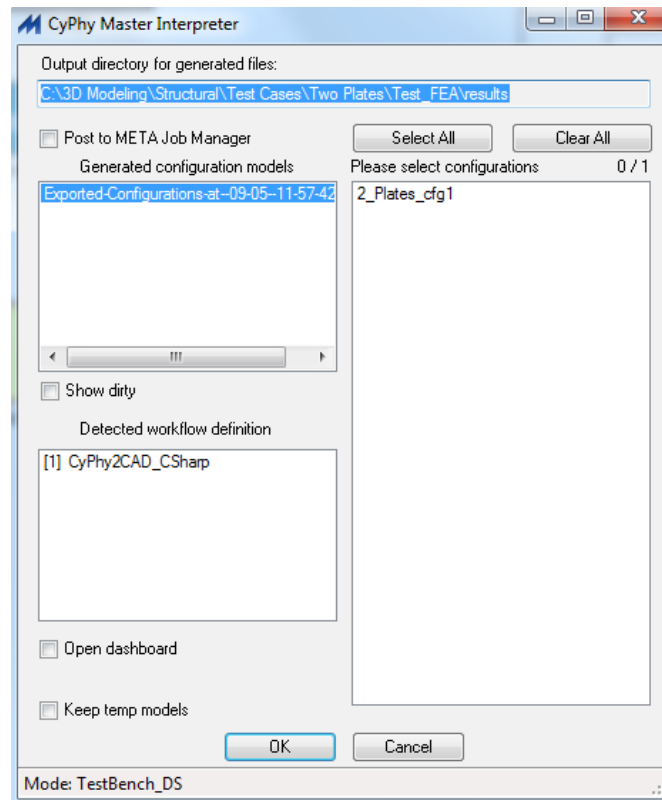


Figure 48

The CAD Options window will come up. (Fig. 49)

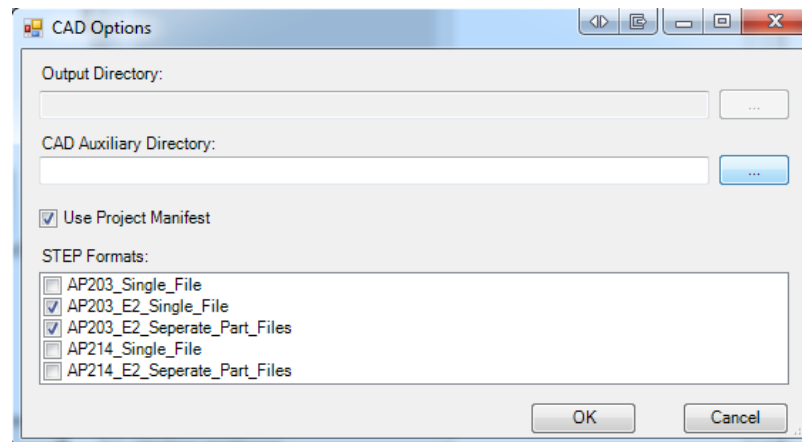


Figure 49

Under Testing, under Workflow Definitions, right-click on the workflow definition (which has been named as CAD in Figure 32) and select copy. (Fig. 50)

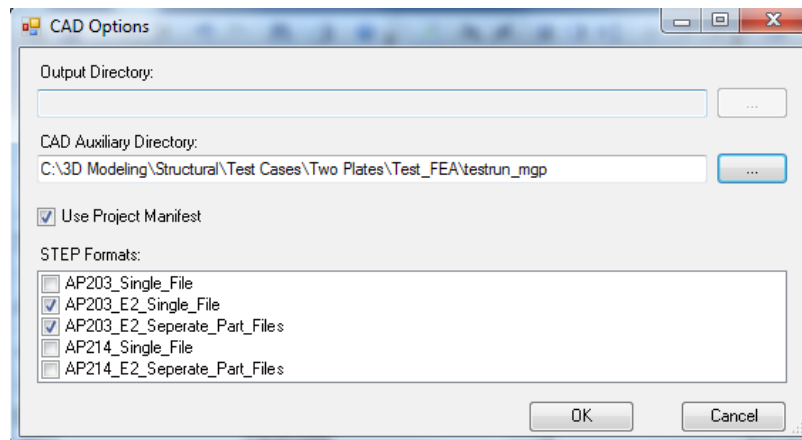


Figure 50

The JobManager Configuration window will now open. (Fig. 51)

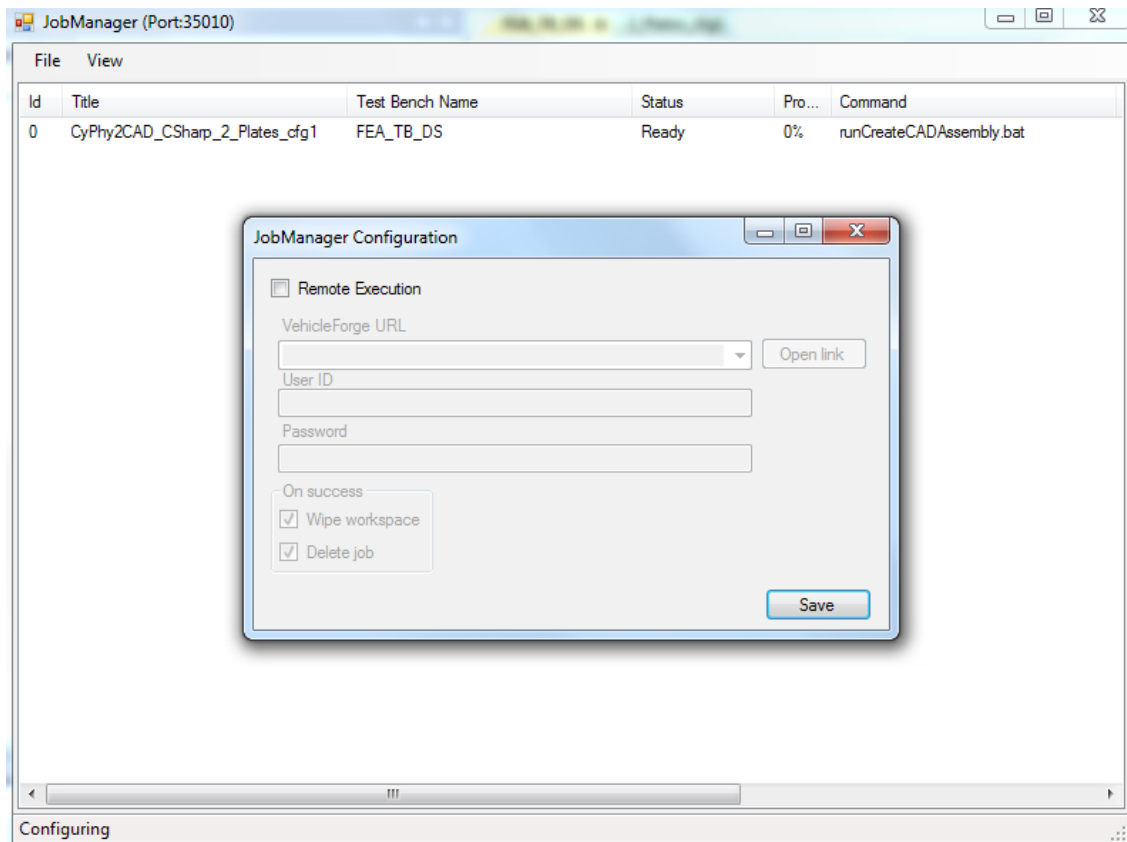


Figure 51

Input your VF information and click Save. (Fig. 52)

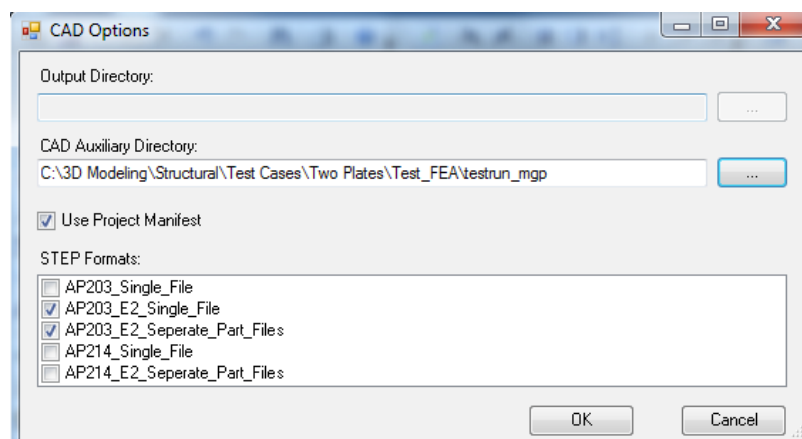


Figure 52

Job Manger will start the FEA computations. (Fig. 53)

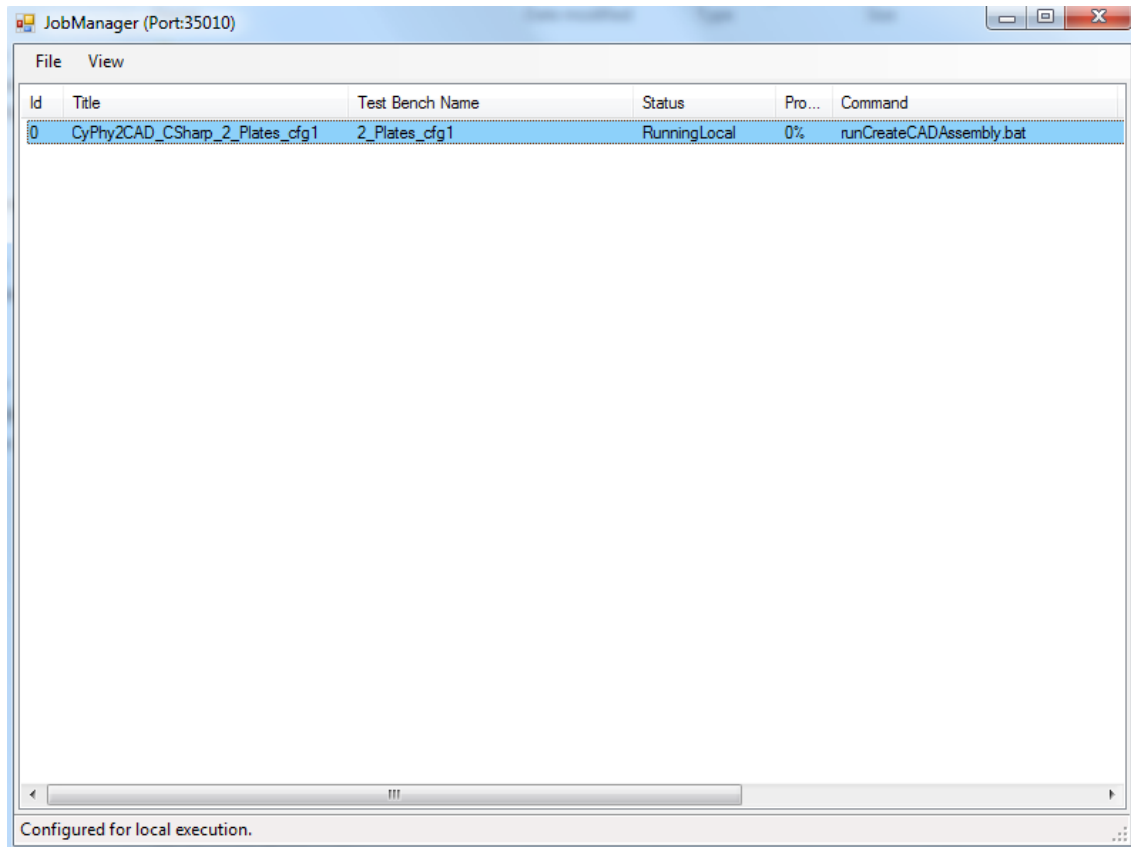


Figure 53

After the job is completed, to see the results, open the main folder for the GME. (Fig. 54)

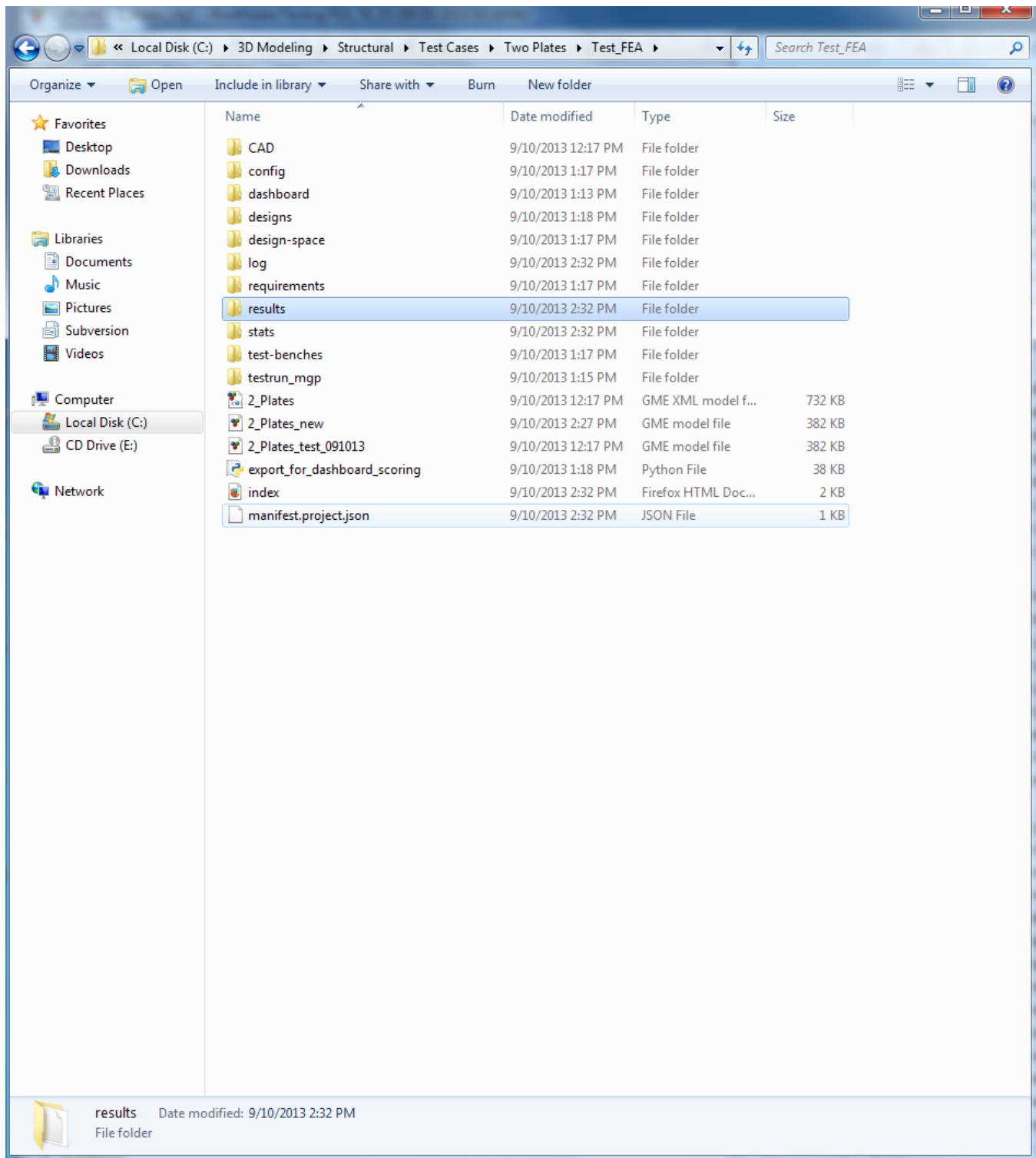


Figure 54

Click on the correct folder. (Fig. 55)

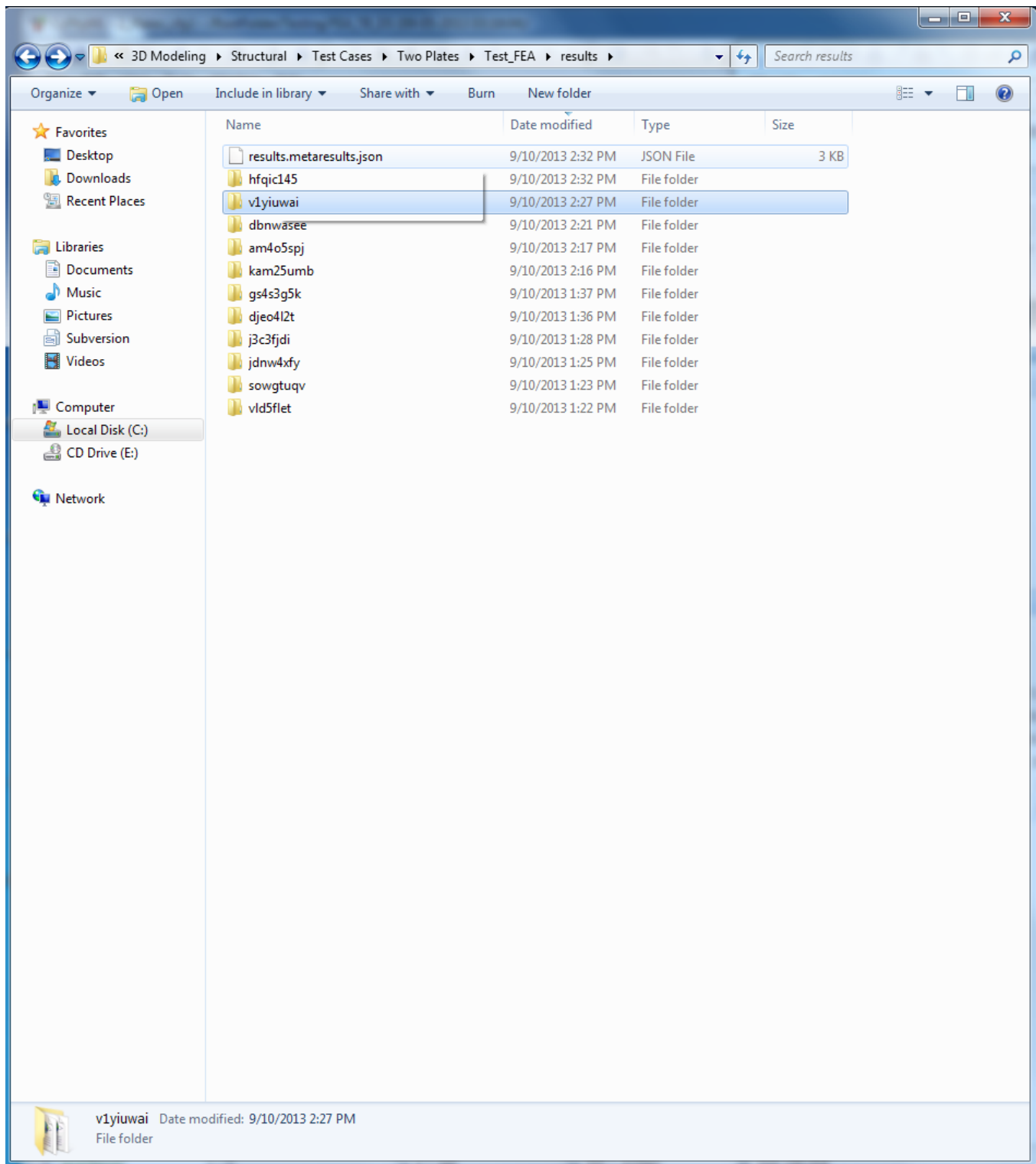


Figure 55

The abaqus .odb file and the pictures associated with the results can be found under “Analysis/Abaqus”. The path is also shown in (Fig.). In that location, a csv file summarizing the results for each part can also be found if Model-based test bench has been used.

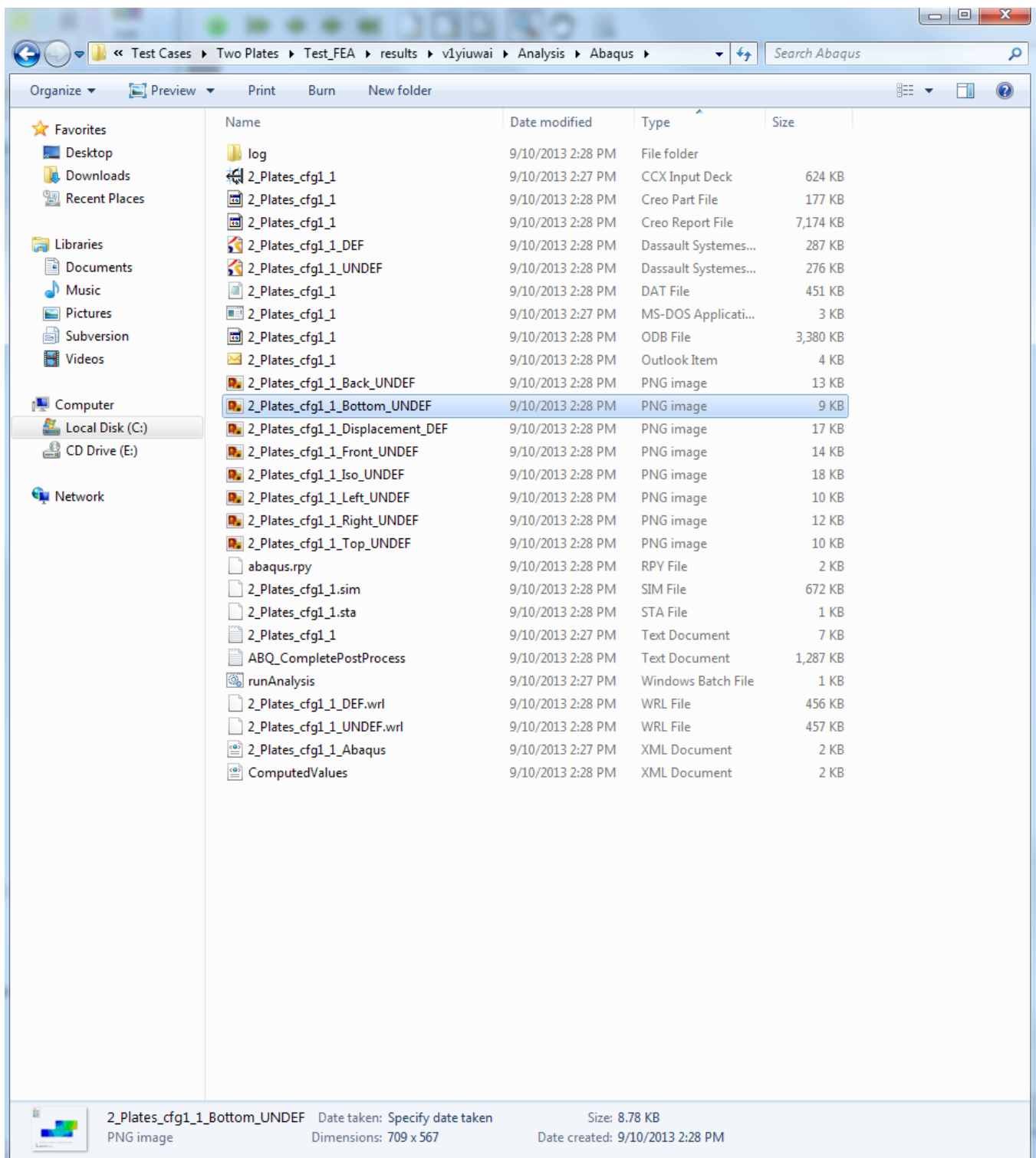


Figure 56

The summary.testresults.json is located in the location shown in (Fig. 57).

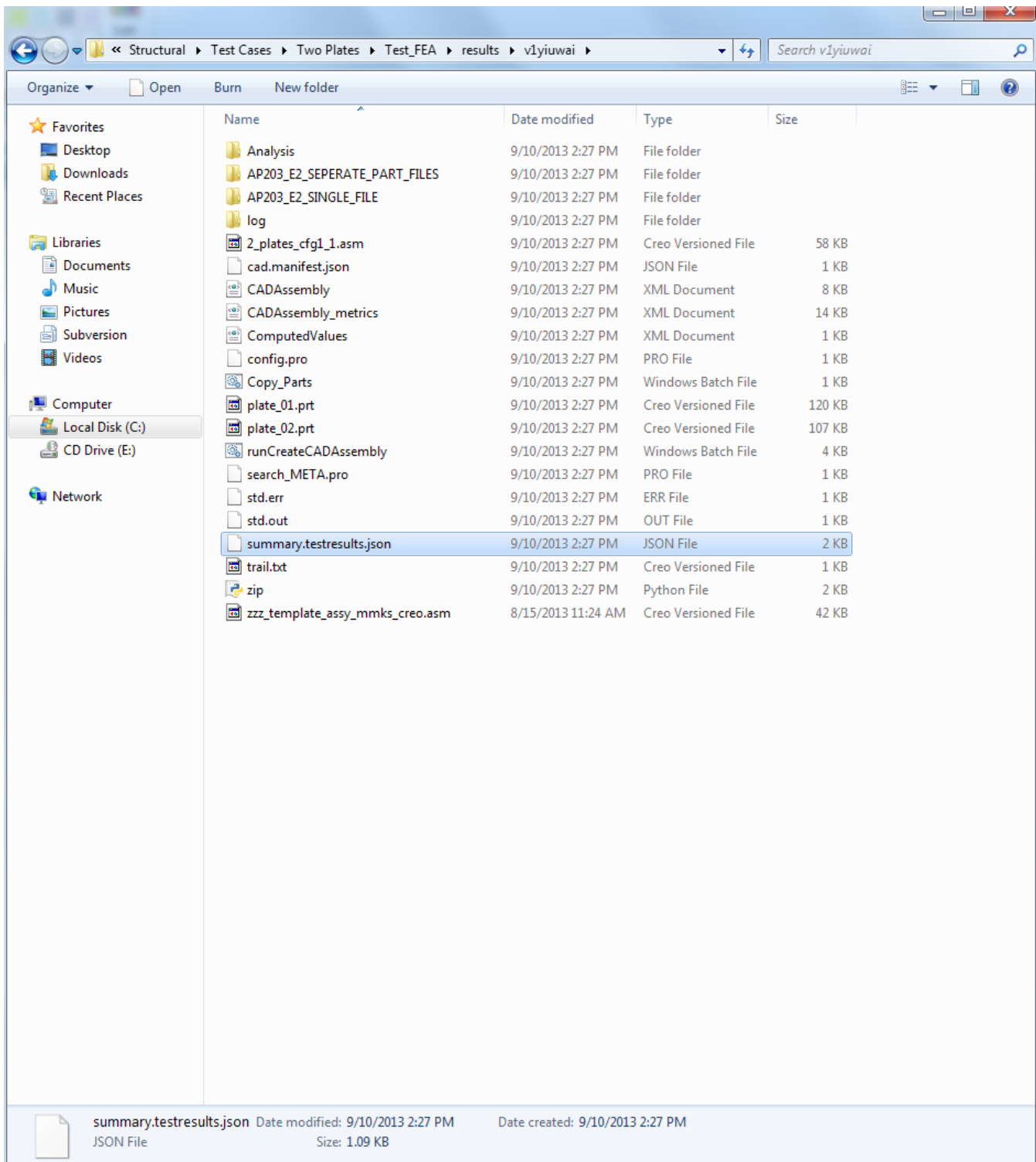


Figure 57

In (Fig. 58), effective (von Mises) stress fringes of the model is presented. Results of the analysis can easily be visualized by using a software that can open odb files (e.g. Abaqus, 3D xml viewer, etc.).

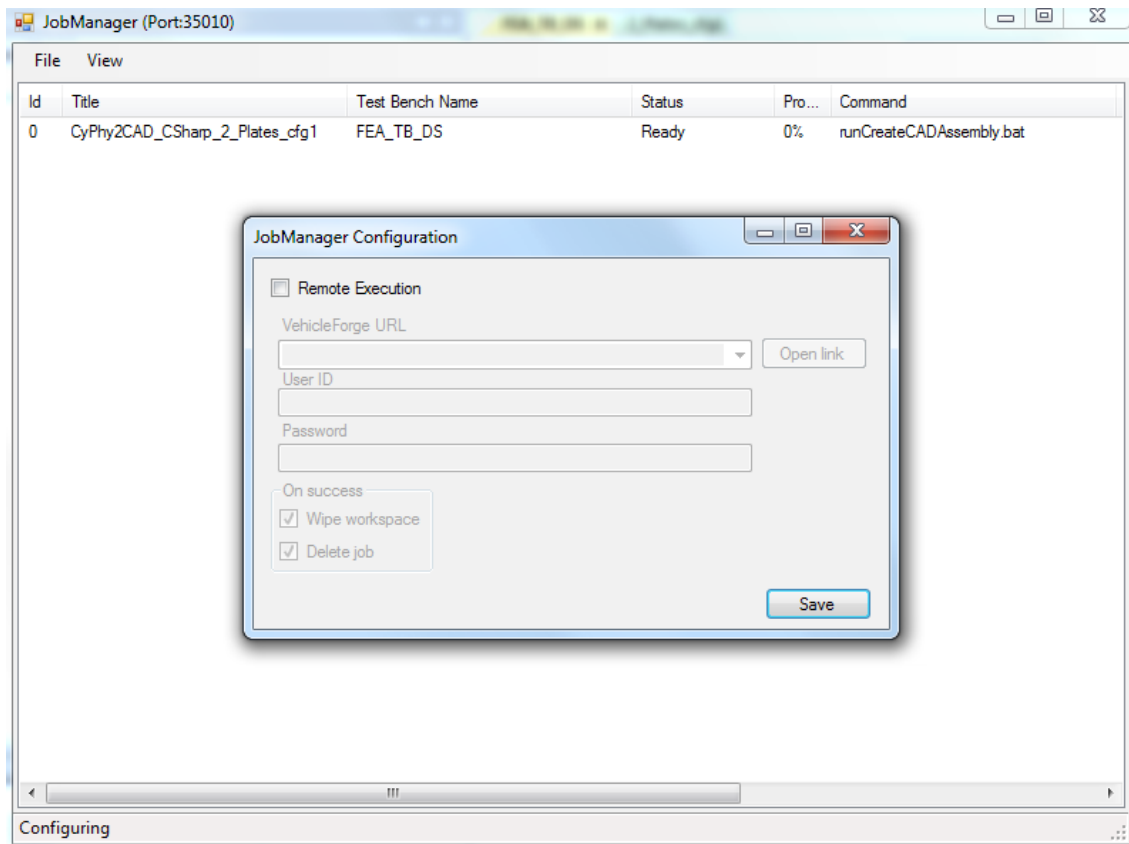


Figure 58

Analysis times are highly variable depending on the size of the model.

The FEA test bench is a server only test bench and cannot be run on the designer's machine because the analysis tool is instantiated on a server.

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.

Results are returned in file "summary.testresults.json" and the files located in the "Abaqus/Analysis" folder. The results can be visualized by opening up the files mentioned before with a simple editors (Notepad, Excel, Microsoft Photo Viewer).

Metrics

- **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.

Required Connections to System Under Test

Outputs

The output of this test bench is in the "summary.testresults.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 a similar software) and all image files can be viewed with a simple image viewer.

After the job is completed, to see the results, open the main folder for the GME. (Fig. 59)

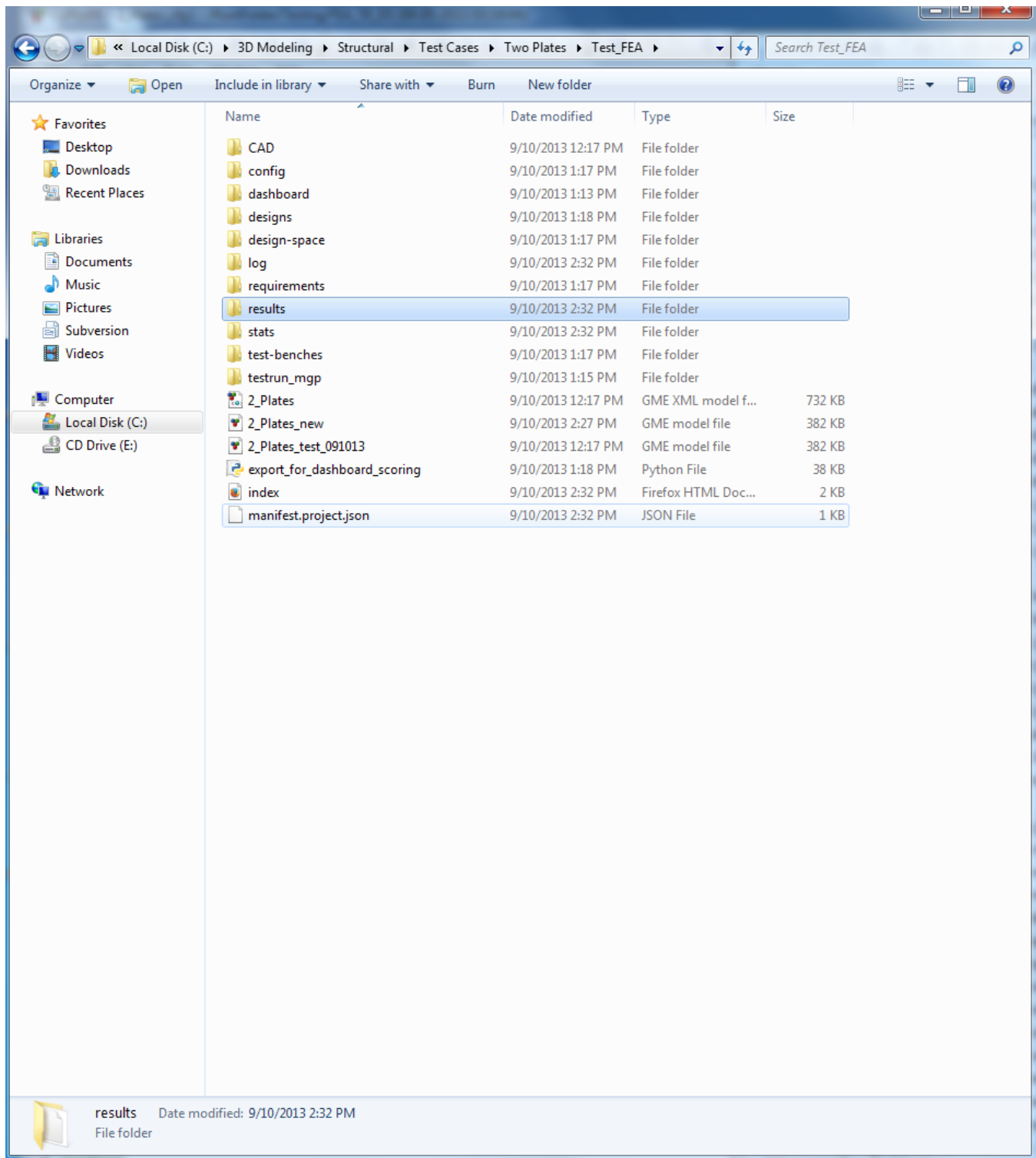


Figure 59

Click on the correct folder. (Fig. 60)

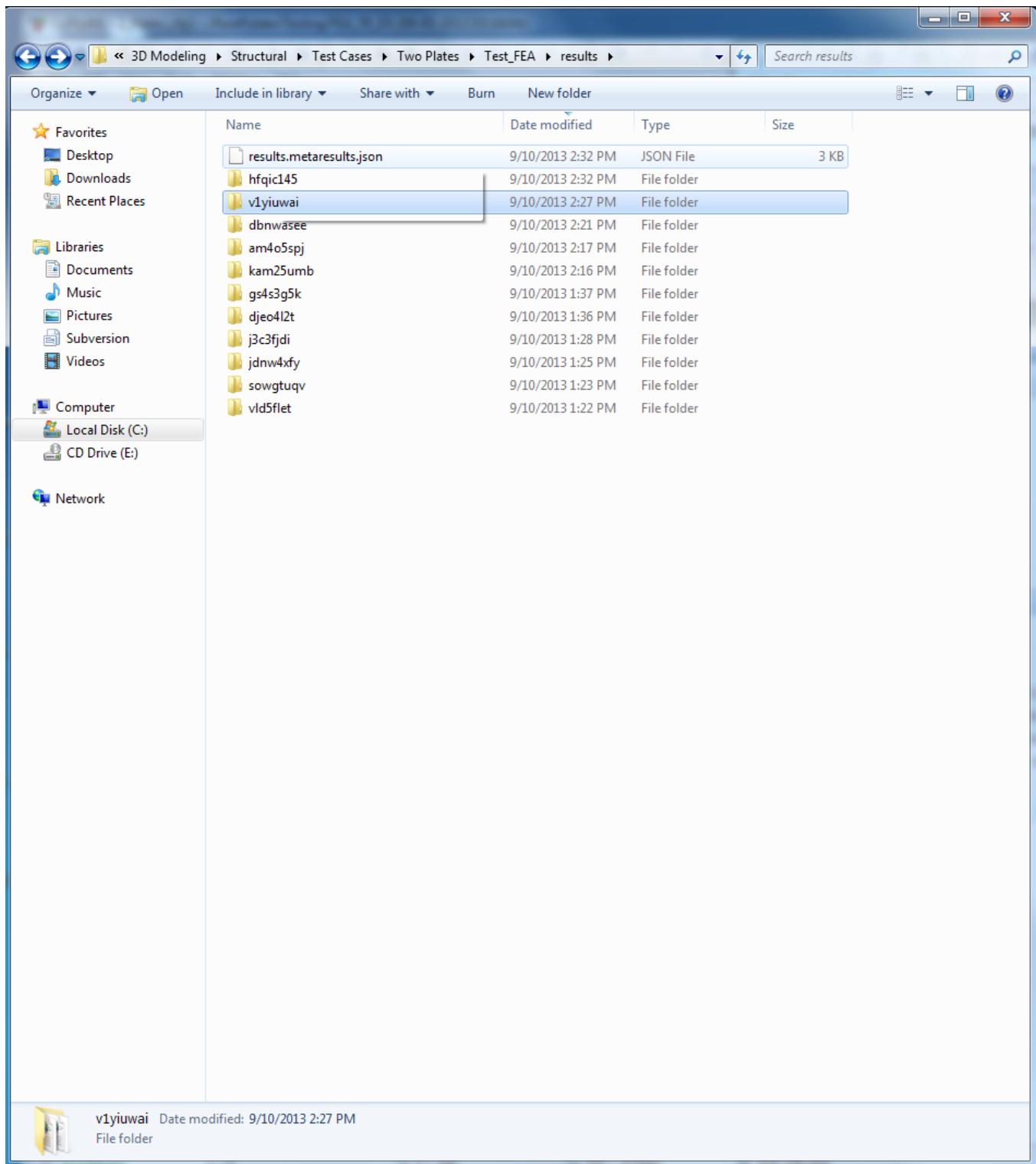


Figure 60

The abaqus .odb file and the pictures associated with the result can be found under analysis abaqus. The path is shown in (Fig. 61). In that location a csv file summarizing the results for each part can also be found.

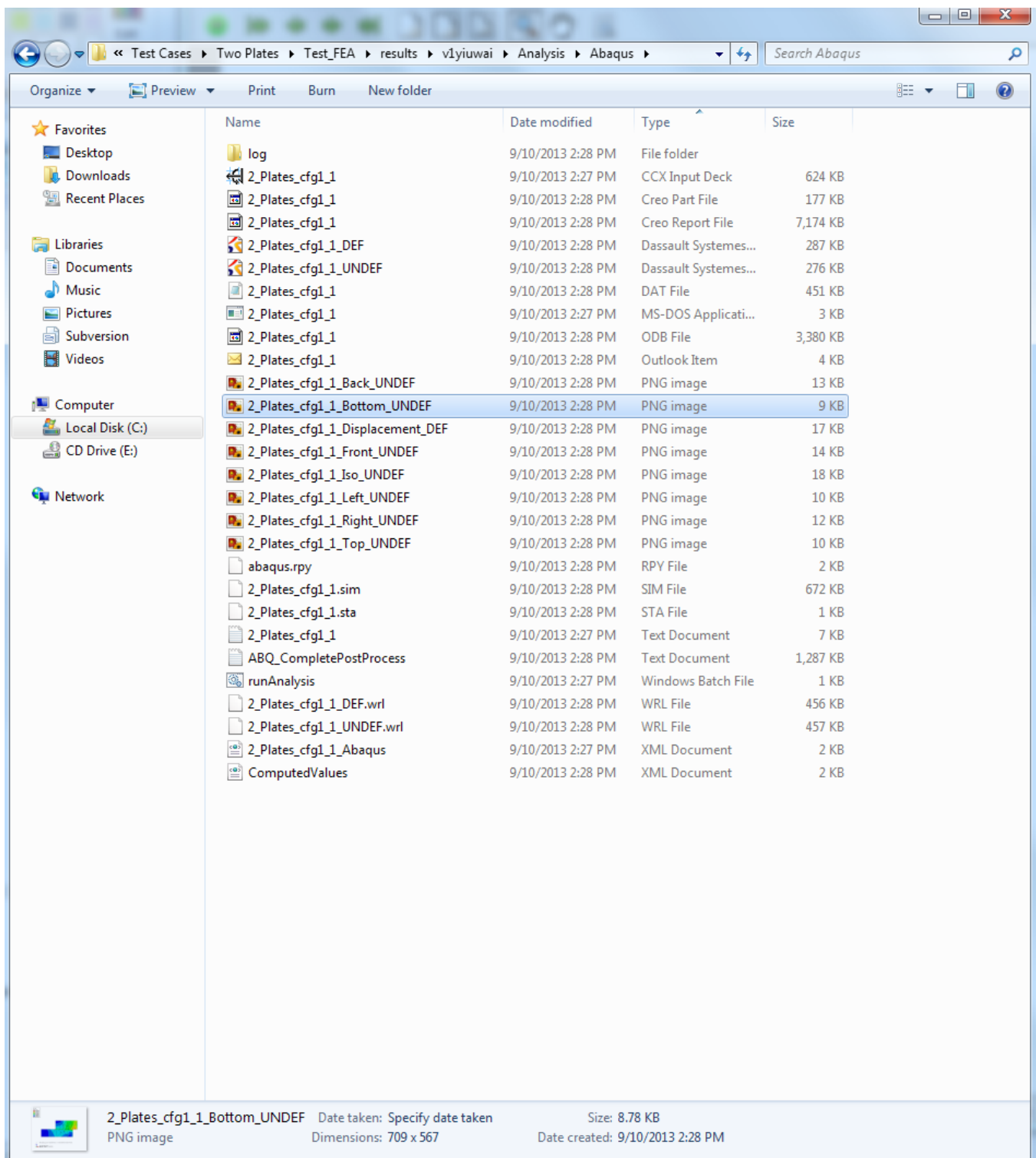


Figure 61

The summary.testresults.json is located in the location shown in (Fig. 62).

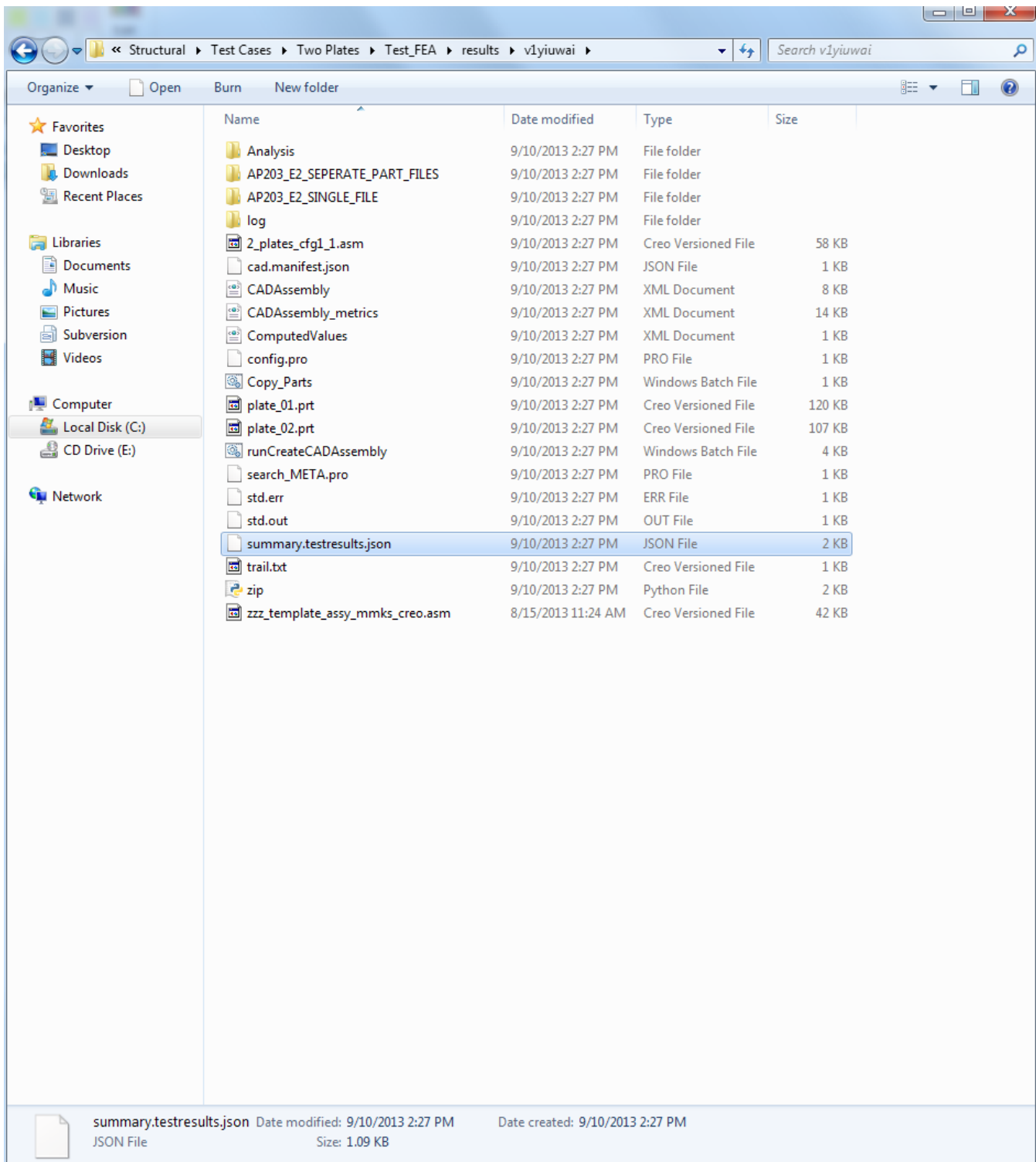


Figure 62