META Structural Analysis Tool

Vanderbilt University, Institute for Software Integrated Systems

February 5, 2014

Abstract

The META structural analysis tool predicts the stress, displacement, and factor of safety of a given geometry and static loading conditions. For the Defense Advanced Research Project Agency's Advanced Vehicle Make program, the META structural analysis tool is used for amphibious vehicles. The finite element analysis method is used to estimate the maximum stress in each part in the assembly. Depending on the different geometries and different inputs, the finite element analysis might require significant computing time with simulations having run times ranging from a few minutes to many hours. This document presents the theory and operation of the META structural analysis tool and covers the input variables and data output.

Nomenclature

E	Young's modulus (elastic modulus) (MPa)
u	Displacements
N_a	Finite element shape functions
u_a	Nodal coefficients of the shape functions
n_n	Total number of mesh nodes in a finite element discretization
n	Normal vector
L	Tensor of elastic moduli
$\tilde{u}$	Prescribed boundary displacement
$\tilde{t}$	Prescribed boundary traction
e	Element number
B^e	Gradient of the element shape functions
I_t	Index set of elements at the boundary
f_a	Force vector of element
K	Global stiffness matrix
F	Global force vector
U	Global nodal displacement vector

Greek Letters

σ	Stress tensor
ε	Strain tensor
v	Test function
	Poisson's ratio
ρ	mass density (kg/m^3)
$\partial\Omega_u$	non-overlapping essential exterior boundary part
$\partial\Omega_t$	non-overlapping natural exterior boundary part

1 Introduction

The META structural analysis tool estimates the maximum stress and the minimum modes of an assembly given geometry and static or dynamic loading conditions. For the Defense Advanced Research Project Agency's (DARPA's) Advanced Vehicle Make (AVM) program, the META structural analysis tool is used for amphibious vehicles. The tool is designed to be run by non-specialist engineers with minimal input. How the model is constructed and the different types of materials used, dictate the response of the system due to different loading conditions in the the META structural analysis tool.

2 Theory

As an amphibious vehicle is subjected to driving, transportation, towing, and mooring conditions, the vehicle experiences a variety of loading conditions that effect the structural response of the system. The Finite Element Analysis (FEA) method can be used the estimate the response of the system due to various loadings. The response of the system can be characterized by numerous methods, but for the simplicity of this structural analysis, we characterize it by stress and displacements. The META structural analysis tool uses finite element analysis to predict the maximum stress, maxim displacement, and the factors of safety.

FEA is a computational procedure for solving mathematical models numerically. FEA uses discretization (nodes and elements) to model the engineering system. It subdivides the system into small components called elements and are comprised of nodes. Approximation at each element represent the behavior of the unknown variables. There are different types of elements that can be specified in the model. Using different types of elements and increasing the amount of elements in the model can improve the accuracy of the approximation.

The META FEA tools support static, dynamic, and modal analyses.

2.1 Static Structural Analysis

Static structural analysis is performed using implicit linear elastic analysis. The FEA method uses the known variables of the geometry, material properties, and boundary conditions to construct an initial model. Then the structure is divided into finite elements. The formulation of the properties of each elements are then found. For a structural analysis, the method estimates the nodal loads associated with all the elements. The element strains for the nodal displacements are calculated and then the element stresses are estimated from the element strains. Adaptive mesh refinement, more specifically h-adaptive refinement, is used to refine the mesh where high stress

gradients exist. This refinement requires more time to compute a final solution, however h-adaptive refinement allows simulations to start with a general mesh and converge to an accurate solution - both of which are important factors for an automated solution.

The FEA method is based on the expression of the response field using the following approximation of the finite element approximation of response field [Szabo and Babuška, 1991]:

$$u(x) = \sum_{a=1}^{n_n} N_a(x)\hat{u}_a \tag{1}$$

The governing equations for the deformation response of structural system is:

$$\nabla \cdot \sigma(x) = 0; \quad x \in \Omega \tag{2}$$

$$\sigma(x) = L : \varepsilon(x) = L : \nabla^s u(x); \quad x \in \Omega \tag{3}$$

where, σ is the stress tensor; ε the strain tensor given as the symmetric gradient of the displacement field, u; and L is the tensor of elastic moduli of the matrix material. L is taken to be a symmetric and strongly elliptic fourth order tensor. The boundary conditions of the deformation problem are:

$$u(x) = \tilde{u}(x); \quad x \in \partial\Omega_u \tag{4}$$

$$\sigma(x) \cdot n(x) = \tilde{t}(x); \quad x \in \partial\Omega_t \tag{5}$$

in which, $\tilde{u}$ and $\tilde{t}$ are the prescribed boundary displacement and tractions, respectively. $\partial\Omega_u$ and $\partial\Omega_t$ are the non-overlapping essential and natural exterior boundary parts such that $\partial\Omega_u \cup \partial\Omega_t = \partial\Omega$.

The finite element method is employed to discretize and evaluate the governing equations. Using the standard Ritz-Galerkin procedure, the problem can be posed in the weak form using the the principle of virtual work. Given the boundary data and the elastic moduli, find $u \in \mathscr{U}_{\tilde{u}}$ such that for all $v \in \mathscr{U}_0$

$$\int_\Omega \nabla v : L : \nabla u \, d\Omega = \int_{\partial\Omega_t} v \cdot \tilde{t} \, d\Gamma \tag{6}$$

The discretization of the trial and test functions follows the Galerkin method. We start by the decomposition of the problem domain into finite elements. The first term in Eq. 6 is then expressed as [Reddy, 2006]:

$$\int_\Omega \nabla v : L : \nabla u \, d\Omega = \sum_{e=1}^{n_e} \int_{\Omega_e} \nabla v : L : \nabla u \, d\Omega \tag{7}$$

in which, n_e is the total number of elements; and Ω_e the domain of the element, which for simplicity, the element level integral is expressed as:

$$\int_{\Omega_e} \nabla v : L : \nabla u \, d\Omega = (V^e)^T \int_{\Omega_e} (B^e)^T L B^e \, d\Omega \, U^e = (V^e)^T K^e U^e \tag{8}$$

in which, superscript T denotes the transpose operator; and U^e and V^e denote the vectors of nodal coefficients of the trial and test functions in element, e:

$$U^e = \left\{ (\hat{u}_1^e)^T (\hat{u}_2^e)^T \dots (\hat{u}_{n_n^e}^e)^T \right\}^T \tag{9}$$

3

Decomposing the boundary integral into its elemental components yields:

$$\int_{\partial\Omega_t} \mathbf{v} \cdot \tilde{t}\, d\Gamma = \sum_{e \in I_t} \int_{\partial\Omega_t} \mathbf{v} \cdot \tilde{t}\, d\Gamma \tag{10}$$

And at the element level boundary integral is expressed as:

$$\int_{\partial\Omega_t} \mathbf{v} \cdot \tilde{t}\, d\Gamma = (V^e)^T \int_{\partial\Omega_t} f^e(x)\, d\Gamma = (V^e)^T F^e \tag{11}$$

The components of the element force vector are:

$$\hat{f}_a^e(x) = N_a^e(x)\,\tilde{t}(x); \tag{12}$$

Defining the global vector of unknown nodal coefficients as:

$$U = \left\{ (\hat{u}_1)^T (\hat{u}_2)^T \dots (\hat{u}_{n_n})^T \right\}^T \tag{13}$$

The global stiffness matrix and the force vectors are obtained by assembling the corresponding element matrices.

$$KU = F \tag{14}$$

3   Dynamic Analysis

Dynamic structural analysis is performed using implicit methods. Implicit schemes remove the upper bound on time step size by solving for dynamic quantities at time $t + \Delta t$ based not only on values at t, but also on these same quantities at $t + \Delta t$. But because they are implicit, non-linear equations must be solved. In structural problems implicit integration schemes usually give acceptable solutions with time steps typically one or two orders of magnitude larger than the stability limit of simple explicit schemes, but the response prediction will deteriorate as the time step size,Δt , increases relative to the period, T, of typical modes of response. [Dassault Systemes, 2013]

A time step for implicit integration can be found from monitoring the values of equilibrium residuals at $t + \Delta t/2$ once the solution at $t + \Delta t$ has been obtained, the accuracy of the solution can be assessed and the time step adjusted appropriately. Uknowns appear on both sides of the equation for the displacement:

$$U(t + \Delta t) = U(t) + \Delta t\, f(U(t + \Delta t)) \tag{15}$$

Solving the system of equations for a non damped system is generalized for displacement increments (n) as:

$$M\ddot{U}_{n+1} + KU_{n+1} = F_{ext\,n+1} - F_{int\,n} - M\ddot{U}_n \tag{16}$$

4 Modal Analysis

In the modal analysis of structural mechanics, natural mode shapes and frequencies of the system during a free vibration are found. The finite element method is used to perform this analysis because of the arbitrary shapes that the method can use to compute the response. Eigenvalues and eigenvectors are solved from the system of equations. The eigenvalues and eigenvectors represent the frequencies and corresponding mode shapes. Typically modes at the lowest frequency determine at what mode the system will vibrate, and will dominates the system . [Dassault Systemes, 2013]

Solving a linear elastic system, the equation of motion can be given as :

$$M\ddot{U} + C\dot{U} + KU = F \tag{17}$$

This equation of motion is used for nonzero damping problems. For vibration modal analysis, damping is ignored and the force vector is set to zero. It is assumed that that each node is subjected to a sinusoidal functions with a peak amplitude. Therefore the displacement vector as the form of:

$$U = A\sin(wt) \tag{18}$$

A is the amplitude of displacement and w is the frequency of vibration. The velocity and acceleration vectors can be found by taking the derivative of the displacement vector. Substituting these into the equation of motion for free vibration, it yields: This leaves the equation of motion as:

$$(K - \lambda M)A = 0 \tag{19}$$

λ is the eigenvalue equal to w^2 and **A** is the eigenvector associated withe each value of the eigenvalues. The total numbers of eigenvalues or natural frequencies is equal to the total number of degrees of freedom in the model. [Nash, 2001] Each eigenvalue or frequency has a corresponding eigenvector or mode shape. Since each of the eigenvectors cannot be null vectors, the equation which must be solved is:

$$(K - \lambda M) = 0 \tag{20}$$

5 Operation

The META structural analysis tool is operated through the META tools via the Generic Modeling Environment (GME) and CyPhy. FEA test benches are built and configured with the desired structural analysis options. Consult the Generic Modeling Environment (GME) documentation and other META documentation for details on GME, CyPhy, and building assemblies, design spaces, and test benches. The FEA tool requires geometry, information about how the components are connected and assembled, material properties for each component (Young's modulus, Poisson's ratio, density, fatigue strength, yield strength, and ultimate tensile strength), and boundary conditions (loads and constraints).

When creating a new FEA test bench, the user right clicks the desired test bench folder, selects Insert Model, then selects Structural FEA Test Bench. The minimum required test bench objects for an FEA test bench are: Computer Aided Design (CAD) workflow, System Under Test, Test
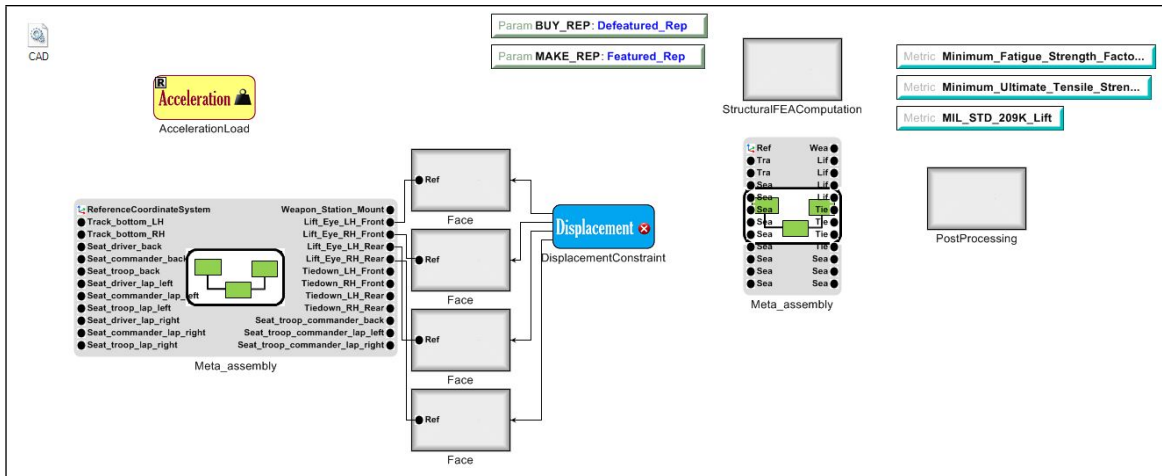
Figure 1: Example FEA Test Bench.

Injection Point, StructuralFEAComputation, metrics, a displacement constraint, and load (force, pressure, or acceleration). Optional objects include Parameters, PostProcessing, and Properties.

Figure 1 illustrates an FEA test bench in CyPhy.

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 3 and 10, inclusive. 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.

5.1 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.

Table 1: Test Bench CAD Representation Parameters.

Description	Parameter Name
Buy component representation	BUY_REP
Make component representation	MAKE_REP
Default component representation	DEFAULT_REP

Table 2: Test Bench CAD Representation Parameter Values.

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

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 1 details the Parameter objects used to specify the representations and Table 2 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 1 and 2 for proper operation.

5.2 Creating Featured and Defeatured CAD representations

5.2.1 Buy Parts

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.

1. Open the Creo part file (see example in Figure 2.

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
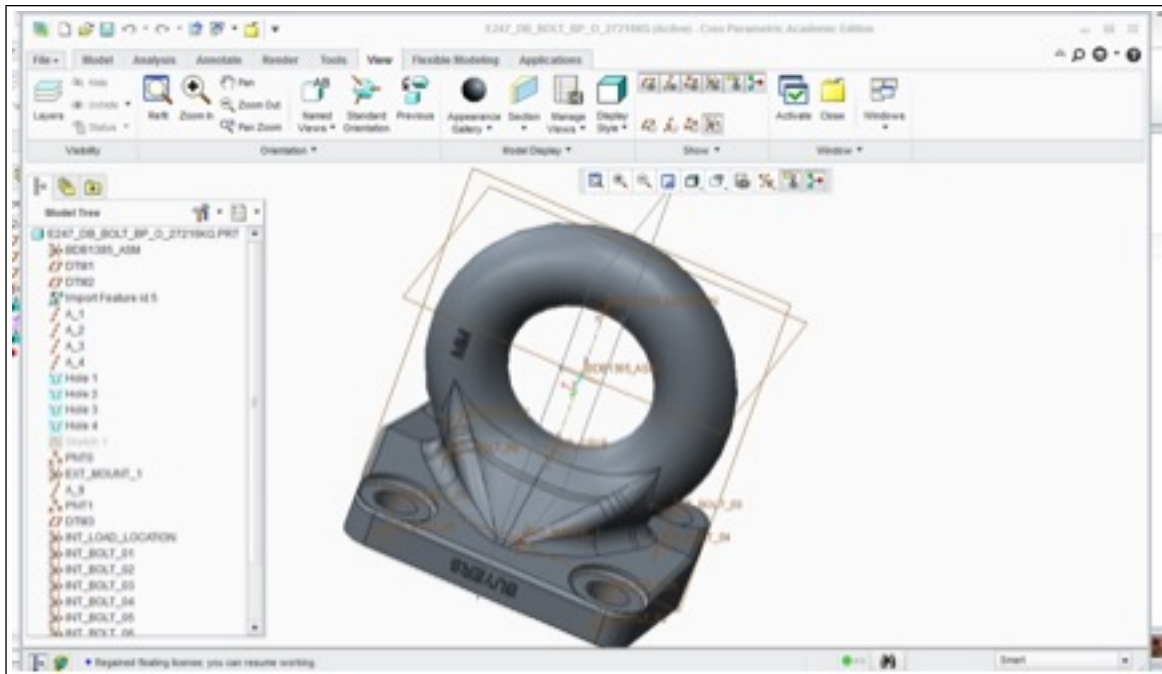
Figure 2: Example buy part.

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 3)

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.
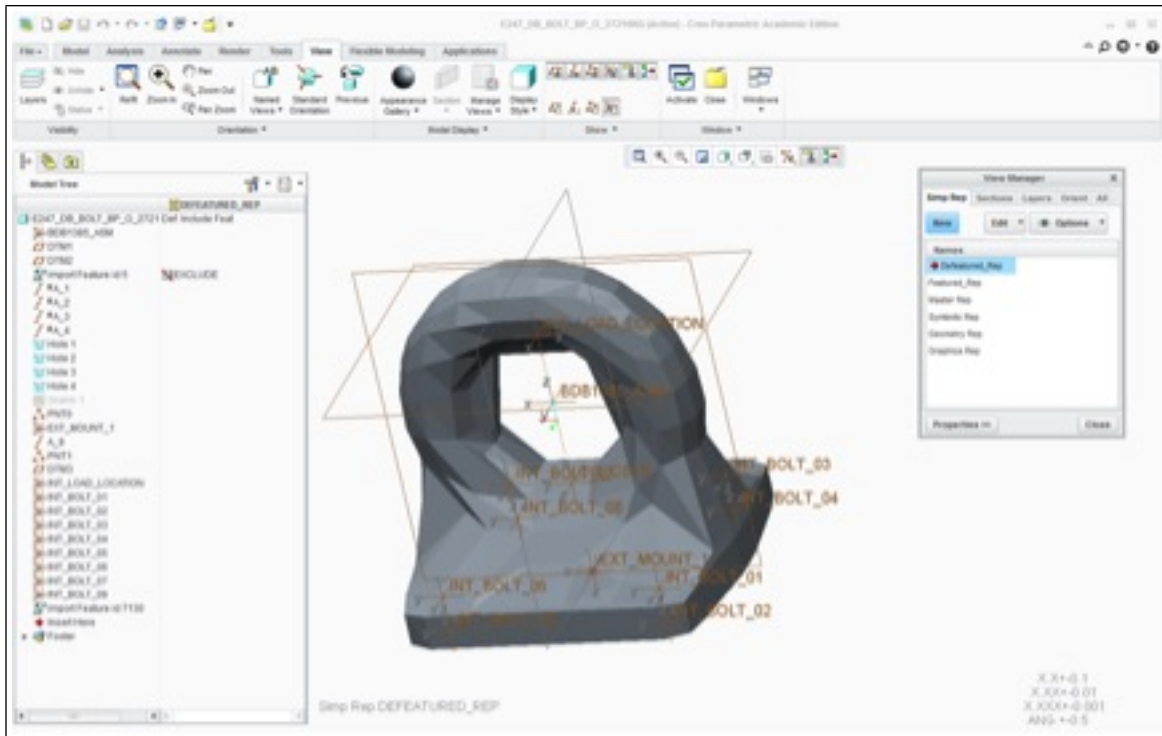
12. Save the part.

Figure 3: Example buy part after shrinkwrap procedure.

5.3 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 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

5.3.1 Materials

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.

5.4   Dynamics

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.

5.5   Modes

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

5.6   Metrics

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. Table 3 details the contents of the stressOutput.csv file. A

Table 3: stressOutput.csv contents.

Column	Description
Part Name	Creo part file name
Unique ID	CyPhy ID for the Creo part
Fatigue Strength	Fatigue strength of the part's material
Yield Strength	Yield strength of the part's material
Ultimate Tensile Strength	Ultimate tensile strength of the part's material
Maximum Stress	Estimated maximum von Mises stress in the part
Factor of Safety for Fatigue Strength	Fatigue Strength / Maximum Stress
Factor of Safety for Yield Strength	Yield Strength / Maximum Stress
Factor of Safety for Ultimate Tensile Strength	Ultimate Tensile Strength / Maximum Stress

Table 4: Standard FEA Metrics. Minimum metrics find the minimum value of all parts in the assembly. The factors of safety metrics apply to static and dynamic test benches while the minimum mode metric applies only to modes test benches.

Metric Text
Minimum_Fatigue_Strength_Factor_of_Safety
Minimum_Yield_Strength_Factor_of_Safety
Minimum_Ultimate_Tensile_Strength_Factor_of_Safety
Minimum_Mode

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.

FEA test benches may contain any or all standard FEA metrics along with any custom metrics the user wants to define. The standard FEA metrics are contained in Table 4. These metrics are written to the testbench_manifest.json file in the results folder. Custom metrics may be added to support scoring or other purposes. For example, a scoring metric could be Accidental_Loads_Up that is either true or false depending upon the minimum fatigue strength factor of safety for the design. The scoring metric could be true if the factor of safety is greater than or equal to one and false if the factor of safety is less than one. A post processing python script, specified by the Post-Processing object, would need to be authored by the user to examine the testbench_manifest.json results to determine and write the value for Accidental_Loads_Up.

5.7 Loads

5.7.1 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/s^2 with three values: x direction, y direction, and z direction. These directions reference the assembly coordinate system. One g load is 9,810 mm/s^2. The AccelerationLoad object does not need to be connected to other objects in the test bench.

5.7.2 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 mm^2 kg. The force must be connected to a geometric construct such as face or polygon.

5.7.3 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/mm^2 (MPa). The pressure must be connected to a geometric construct such as face or polygon.

5.8 Constraints

5.8.1 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.

5.9 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.

5.9.1 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.

5.9.2 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® 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.

5.9.3 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.

6 Validation

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

7 Known Limitations

ForceLoad does not support Moments.

DiscplacementConstraint supports fixed translation and rotation only.

The Face construct does not support traction loads (not normal to the face).

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, Ball-Constraint, 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.

References

[Dassault Systemes, 2013] Dassault Systemes (2013). *Abaqus User Manual, Version 6.13*.

[Nash, 2001] Nash, D. H. (2001). *Dynamic Analysis 1*. University of Strathclyde.

[Reddy, 2006] Reddy, J. N. (2006). *An introduction to the finite element method*, volume 2. McGraw-Hill New York.

[Szabo and Babuška, 1991] Szabo, B. A. and Babuška, I. (1991). *Finite element analysis*. Wiley. com.