

Nastran Analysis Interface Module (AIM)

Ryan Durscher and Ed Alyanak
AFRL/RQVC

August 1, 2016

Contents

1	Introduction	1
1.1	Nastran AIM Overview	1
1.2	BLAH	2
2	Geometry Fidelity	2
3	AIM Inputs	2
4	AIM Shareable Data	3
5	aimOutputsNastran	3
6	FEA Material	3
6.1	JSON String Dictionary	4
6.2	Single Value String	4
7	FEA Property	4
7.1	JSON String Dictionary	5
7.2	Single Value String	6
8	FEA Constraint	6
8.1	JSON String Dictionary	7
8.2	Single Value String	7
9	FEA Load	7
9.1	JSON String Dictionary	7
9.2	Single Value String	8
10	FEA Analysis	8
10.1	JSON String Dictionary	8
10.2	Single Value String	9
11	FEA DesignVariable	9
11.1	JSON String Dictionary	9
12	FEA DesignConstraint	10
12.1	JSON String Dictionary	10

1 Introduction

1.1 Nastran AIM Overview

A module in the Computational Aircraft Prototype Syntheses (CAPS) has been developed to interact (primarily through input files) with the finite element structural solver Nastran.

Current issues include:

- A thorough bug testing needs to be undertaken.

1.2 BLAH

Related pages:

- [FEA Material](#)
- [FEA Property](#)
- [FEA Constraint](#)
- [FEA Load](#)
- [FEA Analysis](#)
- [FEA DesignVariable](#)
- [FEA DesignConstraint](#)
- [Geometry Fidelity](#)
- [AIM Inputs](#)
- [aimOutputsNastran](#)
- [AIM Shareable Data](#)

2 Geometry Fidelity

The attribute capsFidelity may be set to either ALL or STRUCTURE for the Nastran AIM. The geometric fidelity for the AIM requires that bodies be:

- WIREBODY for purely 1D simulations
- FACEBODY or SHEETBODY (non-manifold) for 2D simulations
- SOLIDBODY or SHEETBODY (manifold) for 3D simulations

3 AIM Inputs

The following list outlines the Nastran inputs along with their default value available through the AIM interface. Unless noted these values will be not be linked to any parent AIMS with variables of the same name.

- **Proj_Name = "nastran_CAPS"**
This corresponds to the project name used for file naming.
- **Tess_Params = [0.025, 0.001, 15.0]**
Body tessellation parameters used when creating a boundary element model. Tess_Params[0] and Tess_Params[1] get scaled by the bounding box of the body. (From the EGADS manual) A set of 3 parameters that drive the EDGE discretization and the FACE triangulation. The first is the maximum length of an EDGE segment or triangle side (in physical space). A zero is flag that allows for any length. The second is a curvature-based value that looks locally at the deviation between the centroid of the discrete object and the underlying geometry. Any deviation larger than the input value will cause the tessellation to be enhanced in those regions. The third is the maximum interior dihedral angle (in degrees) between triangle facets (or Edge segment tangents for a WIREBODY tessellation), note that a zero ignores this phase

- **Edge_Point_Min = 2**
Minimum number of points along an edge to use when creating a boundary element model.
- **Edge_Point_Max = 10**
Maximum number of points along an edge to use when creating a boundary element model.
- **Property = NULL**
Property tuple used to input property information for the model, see [FEA Property](#) for additional details.
- **Material = NULL**
Material tuple used to input material information for the model, see [FEA Material](#) for additional details.
- **Constraint = NULL**
Constraint tuple used to input constraint information for the model, see [FEA Constraint](#) for additional details.
- **Load = NULL**
Load tuple used to input load information for the model, see [FEA Load](#) for additional details.
- **Analysis = NULL**
Analysis tuple used to input analysis/case information for the model, see [FEA Analysis](#) for additional details.
- **Analysis_Type = "Modal"**
Type of analysis to generate files for, options include "Modal", "Static", and "StaticOpt".
- **File_Format = "Small"**
Formatting type for the bulk file. Options: "Small", "Large", "Free".
- **Mesh_File_Format = "Small"**
Formatting type for the mesh file. Options: "Small", "Large", "Free".
- **Design_Variable = NULL**
The design variable tuple used to input design variable information for the model optimization, see [FEA DesignVariable](#) for additional details.
- **Design_Constraint = NULL**
The design constraint tuple used to input design constraint information for the model optimization, see [FEA DesignConstraint](#) for additional details.
- **ObjectiveMinMax = "Max"**
Maximize or minimize the design objective during an optimization. Option: "Max" or "Min".
- **ObjectiveResponseType = "Weight"**
Object response type (see Nastran manual).

4 AIM Shareable Data

Currently the Nastran AIM does not have any shareable data types or values. It will try, however, to inherit a "FEA_MESH" or "Volume_Mesh" from any parent AIMS. Note that the inheritance of the mesh is not required.

5 aimOutputsNastran

None currently.

6 FEA Material

Structure for the material tuple = ("Material Name", "Value"). "Material Name" defines the reference name for the material being specified. The "Value" can either be a JSON String dictionary (see Section [JSON String Dictionary](#)) or a single string keyword (see Section [Single Value String](#)).

6.1 JSON String Dictionary

If "Value" is JSON string dictionary (eg. "Value" = {"density": 7850, "youngModulus": 120000.0, "poissonRatio": 0.5, "materialType": "isotropic"}) the following keywords (= default values) may be used:

- **materialType = "Isotropic"**
Material property type. Options: Isotropic, Anisothotropic, Orthotropic, or Anisotropic.
- **youngModulus = 0.0**
Also know as the elastic modulus, defines the relationship between stress and strain. Default if 'shearModulus' and 'poissonRatio' != 0, $\text{youngModulus} = 2 \cdot (1 + \text{poissonRatio}) \cdot \text{shearModulus}$
- **shearModulus = 0.0**
Also know as the modulus of rigidity is defined as the ratio of shear stress to the shear strain. Default if 'youngModulus' and 'poissonRatio' != 0, $\text{shearModulus} = \text{youngModulus} / (2 \cdot (1 + \text{poissonRatio}))$
- **poissonRatio = 0.0**
Is the fraction of expansion divided by the fraction of compression. Default if 'youngModulus' and 'shearModulus' != 0, $\text{poissonRatio} = (2 \cdot \text{youngModulus} / \text{shearModulus}) - 1$
- **density = 0.0**
Density of the material.
- **thermalExpCoeff = 0.0**
Thermal expansion coefficient of the material.
- **temperatureRef = 0.0**
Reference temperature for material properties.
- **dampingCoeff = 0.0**
Damping coefficient for the material.
- **yieldAllow = 0.0**
Yield strength/allowable for the material.
- **tensionAllow = 0.0**
Tension allowable for the material.
- **compressAllow = 0.0**
Compression allowable for the material.
- **shearAllow = 0.0**
Shear allowable for the material.

6.2 Single Value String

If "Value" is a string, the string value may correspond to an entry in a predefined material lookup table. NOT YET IMPLEMENTED!!!!

7 FEA Property

Structure for the property tuple = ("Property Name", "Value"). "Property Name" defines the reference *capsGroup* for the property being specified. The "Value" can either be a JSON String dictionary (see Section [JSON String Dictionary](#)) or a single string keyword (see Section [Single Value String](#)).

7.1 JSON String Dictionary

If "Value" is JSON string dictionary (eg. "Value" = {"shearMembraneRatio": 0.83, "bendingInertiaRatio": 1.0, "membraneThickness": 0.2, "propertyType": "Shell"}) the following keywords (= default values) may be used:

- **propertyType = No Default value**
Type of property to apply to a give capsGroup *Name*. Options: Rod, Bar, Shear, Shell, Composite, and Solid
- **material = "Material Name" (FEA Material)**
"Material Name" from [FEA Material](#) to use for property. If no material is set the first material created will be used
- **crossSecArea = 0.0**
Cross sectional area.
- **torsionalConst = 0.0**
Torsional constant.
- **torsionalStressReCoeff = 0.0**
Torsional stress recovery coefficient.
- **massPerLength = 0.0**
Mass per unit length.
- **zAxisInertia = 0.0**
Section moment of inertia about the element z-axis.
- **yAxisInertia = 0.0**
Section moment of inertia about the element y-axis.
- **yCoords[4] = [0.0, 0.0, 0.0, 0.0]**
Element y-coordinates, in the bar cross-section, of four points at which to recover stresses
- **zCoords[4] = [0.0, 0.0, 0.0, 0.0]**
Element z-coordinates, in the bar cross-section, of four points at which to recover stresses
- **areaShearFactors[2] = [0.0, 0.0]**
Area factors for shear.
- **crossProductInertia = 0.0**
Section cross-product of inertia.
- **shearPanelThickness = 0.0**
Shear panel thickness.
- **nonStructMassPerArea = 0.0**
Nonstructural mass per unit area.
- **membraneThickness = 0.0**
Membrane thickness.
- **bendingInertiaRatio = 1.0**
Ratio of actual bending moment inertia to the bending inertia of a solid plate of thickness "membranThickness"

- **shearMembraneRatio = 5.0/6.0**
Ratio shear thickness to membrane thickness.
- **materialBending = "Material Name" (FEA Material)**
"Material Name" from [FEA Material](#) to use for property bending. If no material is given and "bendingInertiaRatio" is greater than 0, the material name provided in "material" is used.
- **materialShear = "Material Name" (FEA Material)**
"Material Name" from [FEA Material](#) to use for property shear. If no material is given and "shearMembraneRatio" is greater than 0, the material name provided in "material" is used.
- **massPerArea = 0.0**
Mass per unit area.
- **compositeMaterial = "no default"**
List of "Material Name"s, ["Material Name -1", "Material Name -2", ...], from [FEA Material](#) to use for composites.
- **shearBondAllowable = 0.0**
Allowable interlaminar shear stress.
- **symmetricLaminate = 0**
Symmetric lamination option. 1- SYM only half the plies are specified, for odd number plies 1/2 thickness of center ply is specified the first ply is the bottom ply in the stack, default (0) all plies specified.
- **compositeFailureTheory = "(no default)"**
Composite failure theory. Options: "HILL", "HOFF", "TSAI", and "STRN"
- **compositeThickness = (no default)**
List of composite thickness for each layer (eg. [1.2, 4.0, 3.0]). If the length of this list doesn't match the length of the "compositeMaterial" list, the list is either truncated [>length("compositeMaterial")] or expanded [<length("compositeMaterial")] in which case the last thickness provided is repeated.
- **compositeOrientation = (no default)**
List of composite orientations (angle relative element material axis) for each layer (eg. [5.0, 10.0, 30.0]). If the length of this list doesn't match the length of the "compositeMaterial" list, the list is either truncated [>length("compositeMaterial")] or expanded [<length("compositeMaterial")] in which case the last orientation provided is repeated.

7.2 Single Value String

If "Value" is a string, the string value may correspond to an entry in a predefined property lookup table. NOT YET IMPLEMENTED!!!!

8 FEA Constraint

Structure for the constraint tuple = ("Constraint Name", "Value"). "Constraint Name" defines the reference name for the constraint being specified. The "Value" can either be a JSON String dictionary (see Section [JSON String Dictionary](#)) or a single string keyword (see Section [Single Value String](#)).

8.1 JSON String Dictionary

If "Value" is JSON string dictionary (eg. "Value" = {"groupName": "plateEdge", "dofConstraint": 123456}) the following keywords (= default values) may be used:

- **constraintType = "ZeroDisplacement"**
Type of constraint. Options: "Displacement", "ZeroDisplacement".
- **groupName = "(no default)"**
Single or list of *capsGroup* names on which to apply the constraint (e.g. "Name1" or ["Name1", "Name2", ...]).
- **dofConstraint = 0**
Component numbers / degrees of freedom that will be constrained (123 - zero translation in all three directions).
- **gridDisplacement = 0.0**
Value of displacement for components defined in "dofConstraint".

8.2 Single Value String

If "Value" is a string, the string value may correspond to an entry in a predefined constraint lookup table. NOT YET IMPLEMENTED!!!!

9 FEA Load

Structure for the load tuple = ("Load Name", "Value"). "Load Name" defines the reference name for the load being specified. The "Value" can either be a JSON String dictionary (see Section [JSON String Dictionary](#)) or a single string keyword (see Section [Single Value String](#)).

9.1 JSON String Dictionary

If "Value" is JSON string dictionary (eg. "Value" = {"groupName": "plate", "loadType": "Pressure", "pressureForce": 2000000.0}) the following keywords (= default values) may be used:

- **loadType = "(no default)"**
Type of load. Options: "GridForce", "GridMoment", "Rotational", "Thermal", "Pressure", "PressureDistribute", "PressureExternal", "Gravity".
- **groupName = "(no default)"**
Single or list of *capsGroup* names on which to apply the load (e.g. "Name1" or ["Name1", "Name2", ...]).
- **loadScaleFactor = 1.0**
Scale factor to use when combining loads.
- **forceScaleFactor = 0.0**
Overall scale factor for the force for a "GridForce" load.
- **directionVector = [0.0, 0.0, 0.0]**
X-, y-, and z- components of the force vector for a "GridForce", "GridMoment", or "Gravity" load.
- **momentScaleFactor = 0.0**
Overall scale factor for the moment for a "GridMoment" load.

- **gravityAcceleration = 0.0**
Acceleration value for a "Gravity" load.
- **pressureForce = 0.0**
Uniform pressure force for a "Pressure" load.
- **pressureDistributeForce = [0.0, 0.0, 0.0, 0.0]**
Distributed pressure force for a "PressureDistribute" load.
- **angularVelScaleFactor = 0.0**
An overall scale factor for the angular velocity in revolutions per unit time for a "Rotational" load.
- **angularAccScaleFactor = 0.0**
An overall scale factor for the angular acceleration in revolutions per unit time squared for a "Rotational" load.
- **coordinateSystem = "(no default)"**
Name of coordinate system in which defined force components are in reference to. If no value is provided the global system is assumed.

9.2 Single Value String

If "Value" is a string, the string value may correspond to an entry in a predefined load lookup table. NOT YET IMPLEMENTED!!!!

10 FEA Analysis

Structure for the analysis tuple = ('Analysis Name', 'Value'). 'Analysis Name' defines the reference name for the analysis being specified. The "Value" can either be a JSON String dictionary (see Section [JSON String Dictionary](#)) or a single string keyword (see Section [Single Value String](#)).

10.1 JSON String Dictionary

If "Value" is JSON string dictionary (eg. "Value" = {"numDesiredEigenvalue": 10, "eigenNormaliztion": "MAS↔S", "numEstEigenvalue": 1, "extractionMethod": "GIV", "frequencyRange": [0, 10000]}) the following keywords (= default values) may be used:

- **analysisType = "Modal"**
Type of load. Options: "Modal", "Static", "StaticOpt".
- **analysisLoad = "(no default)"**
Single or list of "Load Name"s defined in [FEA Load](#) in which to use for the analysis (e.g. "Name1" or ["↔Name1", "Name2", ...]).
- **analysisConstraint = "(no default)"**
Single or list of "Constraint Name"s defined in [FEA Constraint](#) in which to use for the analysis (e.g. "Name1" or ["Name1", "Name2", ...]).
- **analysisDesignConstraint = "(no default)"**
Single or list of "Design Constraint Name"s defined in [FEA DesignConstraint](#) in which to use for the analysis (e.g. "Name1" or ["Name1", "Name2", ...]).
- **extractionMethod = "(no default)"**
Extraction method for modal analysis.

- **frequencyRange = [0.0, 0.0]**
Frequency range of interest for modal analysis.
- **numEstEigenvalue = 0**
Number of estimated eigenvalues for modal analysis.
- **numDesiredEigenvalue = 0**
Number of desired eigenvalues for modal analysis.
- **eigenNormalization = "(no default)"**
Method of eigenvector renormalization. Options: "POINT", "MAX", "MASS"
- **gridNormalization = 0**
Grid point to be used in normalizing eigenvector to 1.0 when using eigenNormalization = "POINT"
- **componentNormalization = 0**
Degree of freedom about "gridNormalization" to be used in normalizing eigenvector to 1.0 when using eigenNormalization = "POINT"
- **lanczosMode = 2**
Mode refers to the Lanczos mode type to be used in the solution. In mode 3 the mass matrix, Maa, must be nonsingular whereas in mode 2 the matrix $K_{aa} - \sigma * M_{aa}$ must be nonsingular
- **lanczosType = "(no default)"**
Lanczos matrix type. Options: DPB, DGB.

10.2 Single Value String

If "Value" is a string, the string value may correspond to an entry in a predefined analysis lookup table. NOT YET IMPLEMENTED!!!!

11 FEA DesignVariable

Structure for the design variable tuple = ('DesignVariable Name', 'Value'). 'DesignVariable Name' defines the reference name for the design variable being specified. The "Value" must be a JSON String dictionary (see [Section JSON String Dictionary](#)).

11.1 JSON String Dictionary

If "Value" is JSON string dictionary (eg. "Value" = {"designVariableType": "Property", "groupName": "plate", "upperBound": 10.0, "fieldName": "TM"}) the following keywords (= default values) may be used:

- **designVariableType = "Property"**
Type of design variable in an optimization problem. Options: "Material", "Property".
- **groupName = "(no default)"**
Single or list of *capsGroup* or "Material Name"s ([FEA Material](#)) names on which to add to the design variable (e.g. "Name1" or ["Name1", "Name2", ...]).
- **initialValue = 0.0**
Initial value for the design variable.

- **lowerBound = 0.0**
Lower bound for the design variable.
- **upperBound = 0.0**
Upper bound for the design variable.
- **maxDelta = 0.0**
Move limit for the design variable.
- **discreteValue = 0.0**
List of discrete values do use for the design variable (e.g. [0.0,1.0,1.5,3.0]).
- **fieldName = "(no default)"**
Fieldname of variable (e.g. "E" for Young's Modulus). See Nastran manual for options.

12 FEA DesignConstraint

Structure for the design constraint tuple = ('DesignConstraint Name', 'Value'). 'DesignConstraint Name' defines the reference name for the design constraint being specified. The "Value" must be a JSON String dictionary (see Section [JSON String Dictionary](#)).

12.1 JSON String Dictionary

If "Value" is JSON string dictionary (eg. "Value" = {"groupName": "plate", "upperBound": 10.0}) the following keywords (= default values) may be used:

- **groupName = "(no default)"**
Single or list of *capsGroup* or "Material Name"s ([FEA Material](#)) names on which to apply the design constraint (e.g. "Name1" or ["Name1", "Name2", ...]).
- **lowerBound = 0.0**
Lower bound for the design constraint.
- **upperBound = 0.0**
Upper bound for the design constraint.
- **responseType = "(no default)"**
Response type options for DRESP1 Entry (see Nastran manual).