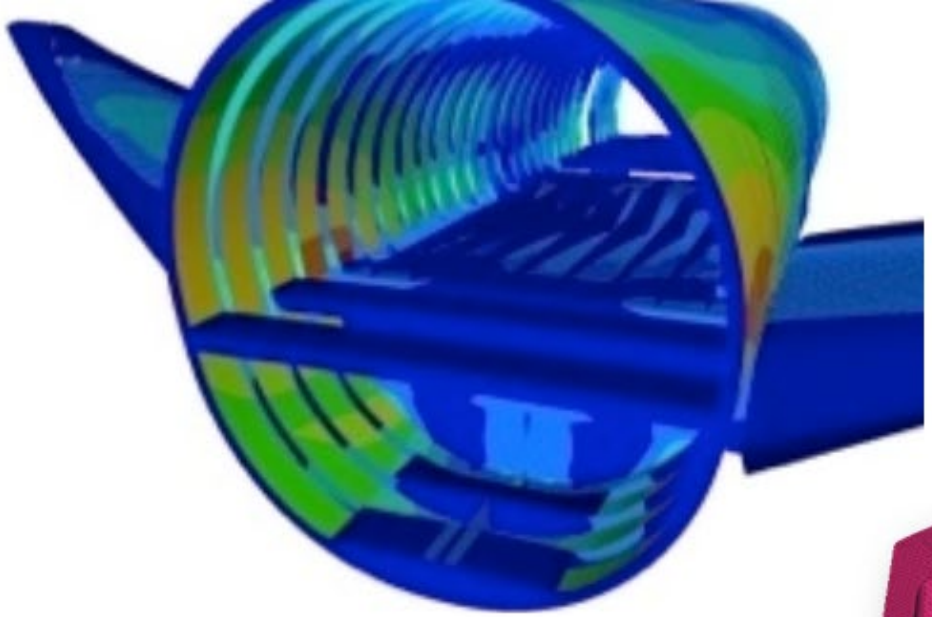
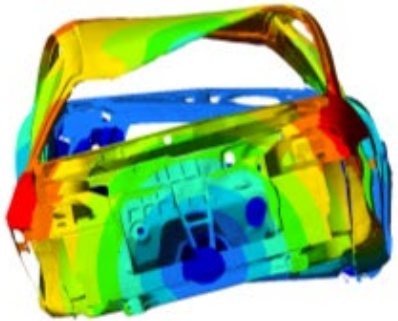
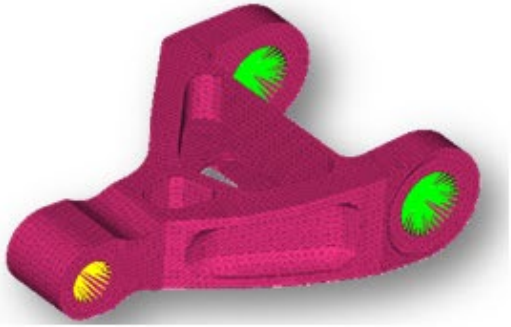






OPTISTRUCT FOR LINEAR ANALYSIS, V2019
CHAPTER 3: INERTIA RELIEF ANALYSIS

Altair

AGENDA

1. Introduction to Linear Analysis
   - Type of Analysis
   - Type of Elements and Materials
   - Type of Loads & Boundary Conditions
2. Linear Static Analysis
3. Inertia Relief Analysis
4. Modal Analysis
5. Linear Buckling Analysis
6. Thermal Stress Steady State Analysis
7. Advanced Topics
   - Debugging Guide
   - Parameters
   - Transitioning Elements
   - Introduction to Parallelization
   - Run Options
   - Output Management
8. Optimization in Linear Analysis
   - OptiStruct Optimization
   - DRCO Approach
   - Setting up Optimization
   - Optimization Responses for Linear Analysis

INERTIA RELIEF

Inertia Relief analysis allows the simulation of unconstrained or partially-constrained structures.

Typical applications include:

- An airplane in flight
- Suspension parts of a car
- Satellite in space

This analysis type will calculate the steady-state stress and deformed shape in the structure as if it were freely accelerating due to the applied loads.

INERTIA RELIEF

In OptiStruct, inertia relief can be applied to linear static and nonlinear gap analyses.

Inertia relief boundary conditions may be defined in the bulk data section of the input deck or they may be determined automatically by the solver:

Automatic Support generation with `PARAM,INREL,-2`

```
SUBCASE 6          $ Defines subcase number 6
LOAD = 5           $ references all static load cards with ID = 5
…
BEGIN BULK
PARAM,INREL,-2     $ Defines inertia relief without support points
FORCE,1,5,…        $ Defines force ID = 1 for load card ID = 5, …
```

INERTIA RELIEF

Using `SUPORT` entries with `PARAM,INREL,-1`

- The `SUPORT` and `SUPORT1` bulk data entries are used to define up to six reaction degrees of freedom of the free body.
- `SUPORT` entries will be used in all relevant subcases and therefore do not need to be referenced in the Subcase Information section.

```
SUBCASE 6          $ Defines subcase number 6
LOAD = 5           $ references all static load cards with SID = 5
…
BEGIN BULK
PARAM,INREL,-1     $ Defines inertia relief with a support point
FORCE,1,5,…        $ Defines force ID = 1 for load card ID = 5, …
SUPORT,10,3        $ Grid ID, DOF‘s supported
```

INERTIA RELIEF

Using SUPORT entries with PARAM,INREL,-1

- The SUPORT and SUPORT1 bulk data entries are used to define up to six reaction degrees of freedom of the free body.
- SUPORT1 entries need to be referenced by a SUPORT1 data selector statement for use within a subcase.

```
SUBCASE 6          $ Defines subcase number 6
LOAD = 5           $ references all static load cards with ID = 5
SUPORT1 = 8        $ references all SUPORT1 cards with ID = 8
…
BEGIN BULK
PARAM,INREL,-1     $ Defines inertia relief with a support point
FORCE,1,5,…        $ Defines force ID = 1 for load card ID = 5, …
SUPORT1,8,10,3     $ SUPORT1 set = 8, Grid ID, DOF's Supported
```

HOW TO SETUP AN INERTIA RELIEF ANALYSIS

Inertia relief analyses can be defined in seven steps:

Step 1 – Generate a FEM model with materials & properties

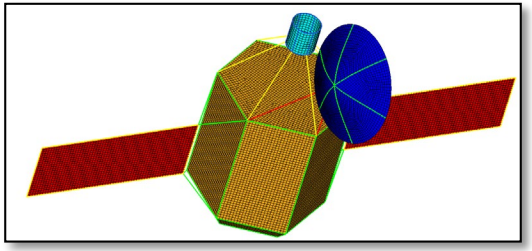
Step 2 – Define the fictitious supports using `SUPORT` or `SUPORT1` cards (optional)

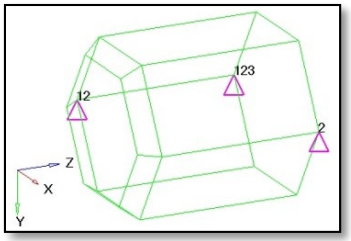
Step 3 – Define the equivalent static force to apply to the model

Step 4 – Define the `INREL` parameter to enable inertia relief calculation

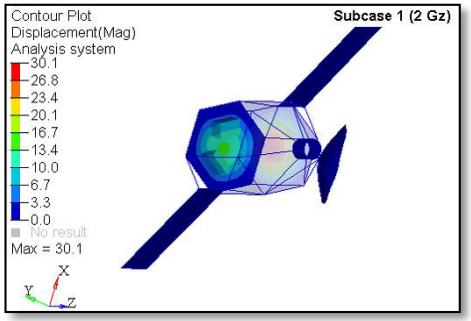
Step 5 – Create the static load case including the fictitious supports

Step 6 – Run the analysis and post-process

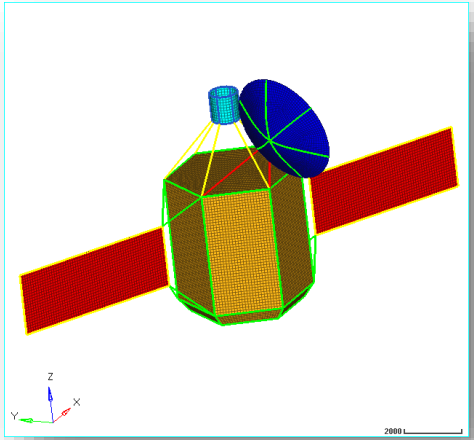




	INREL_V1
PARAM, INREL,	-1



EXERCISE 3A: SATELLITE INERTIA RELIEF ANALYSIS



File Name and Location

`…\STUDENT-EXERCISE\3a_Satellite\satellite.hm`

Objectives (1/2)

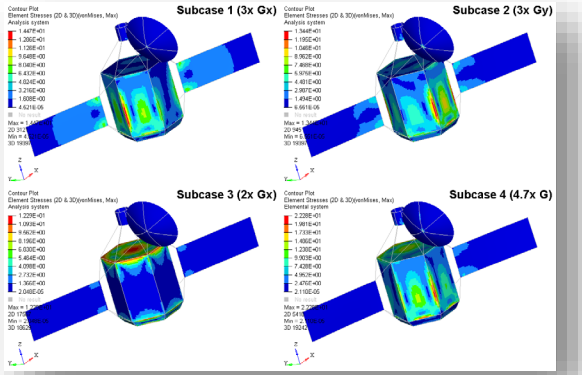
This exercise runs an inertia relief load case on a simple satellite. This is a test made with aerospace structures that will need to support inertia loads. The objective of this kind of test is to verify if the structure is strong enough to support these loads without a static failure.

1. Open the model in HyperMesh Desktop
2. Review the model and check total mass
3. Set common `OUTPUT` requests
4. Set control card `PARAM,INREL` to `-1`
5. Create a load collector `Support` (no card image) with the following `SUPORT1` load type constraints representing the fictitious support
   - Node 2: DOF `2`
   - Node 3: DOFs `1-3`
   - Node 4: DOFs `1-2`

EXERCISE 3A: SATELLITE INERTIA RELIEF ANALYSIS

Objectives (2/2)

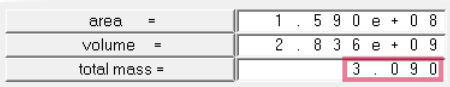
6. Create the following load collectors (no card image) and forces on Node 1 of the model
   - `3x Gx`: force components {92700,0,0} (equivalent to three times gravity in x direction)
   - `3x Gy`: force components {0,92700,0} (equivalent to three times gravity in y direction)
   - `2x Gz`: force components {0,0,61800} (equivalent to two times gravity in z direction)
7. Create a following load collector `4.7x G` with card image `LOADADD` and select all three above created load collectors with scale factor `1.0`
8. Create four Linear Static load cases using `Support` as the `SUPORT1` entry for each of the four load collectors created before
9. Run the analysis with OptiStruct
10. Review the `.out` file wrt warnings and errors
11. Review the results in HyperView and check if
    - Max. relative displacement < 500 mm
    - Max. von Mises stress < 70 MPa for 2D modeled components



EXERCISE 3A: SATELLITE INERTIA RELIEF ANALYSIS

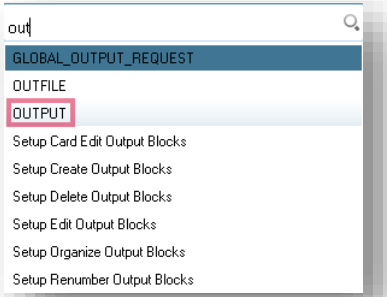
Hints (1/3)

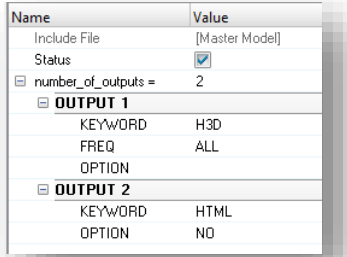
2. Total mass: 3.090 t



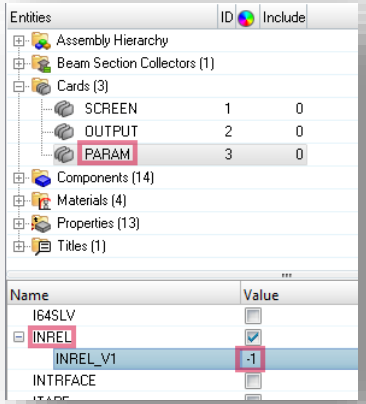
3. Use HyperMesh's Quick Access Tool (`Crtl+f`) to add control cards

```
SCREEN OUT
OUTPUT,H3D,ALL
OUTPUT,HTML,,NO
```





4. Use HyperMesh's Quick Access Tool (`Crtl+f`) to set control card `PARAM,INREL` to `-1`

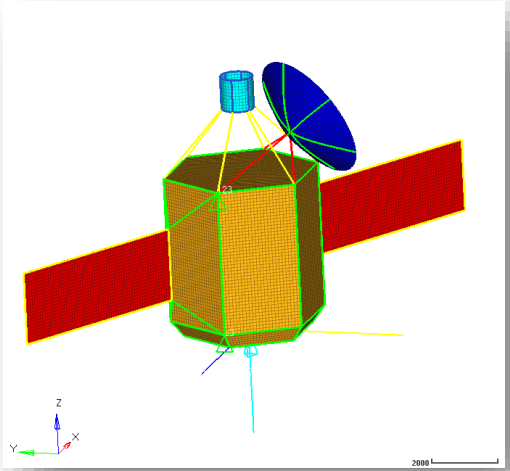


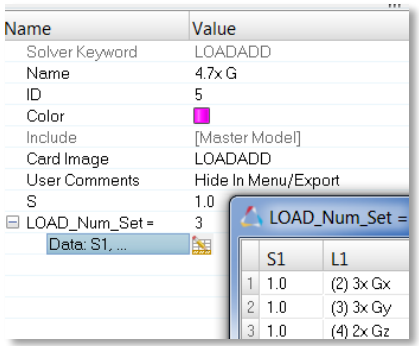
EXERCISE 3A: SATELLITE INERTIA RELIEF ANALYSIS

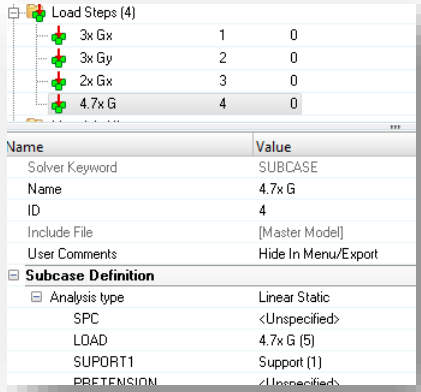
Hints (2/3)

5. Switch the Load Type for constraint in HyperMesh from `SPC` to `SUPORT1` before creating the constraints

6.-8. See screenshots



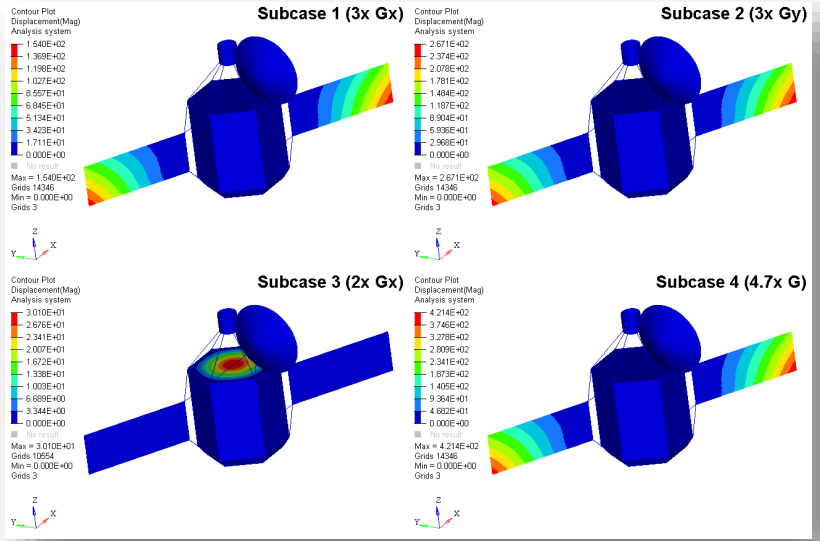


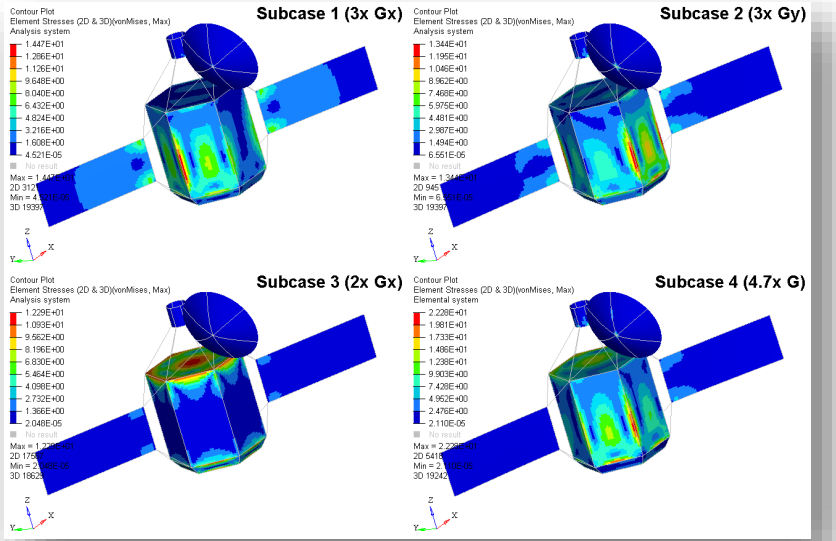


EXERCISE 3A: SATELLITE INERTIA RELIEF ANALYSIS

Hints (3/3)

11. Max. relative displacement = 421.4 mm (load case 4.7x G) < 500 mm
    Max. von Mises stress = 22.3 MPa (load case 4.7x G) < 70 MPa





QUESTIONS & ANSWERS

1. Which of these describes the difference between linear static and inertia relief analysis?
   a) Linear statics determines the vibrational response of a system, inertia relief uses static loads.
   b) Inertia relief correctly solves unconstrained or partially-constrained structures, linear statics can not.
   c) Linear statics solves unconstrained structures, linear statics is for partially-constrained structures