SESAM

EXAMPLE

Blast Analysis using GeniE and Usfos

Date: May 2025

Prepared by DNV – Digital Solutions

E-mail support: software.support@dnv.com

E-mail sales: software@dnv.com

Contents

1 INTRODUCTION

This example goes through steps to perform a blast analysis for a module using the Sesam software suite.

Note: For this example, we assume that user is at least an intermediate to skillful GeniE users. The course contains topics related to opening, navigating, executing, and exporting operations inside GeniE program.

1.1 Program Versions

Below are the programs versions used to run this example:

1. GeniE 8.12.2
2. Sestra 10.19.0
3. Usfos 9.0.0
4. Sesam Manager 6.7.0
5. Application Version Manager V3.1-01

2 IMPORTING GENIE MODEL

Import **Model_module.gnx** file into GeniE new workspace (either using Sesam Manager or directly in GeniE). This model is a completed model, prepared for the blast analysis in Usfos. The model consists of a module with plates resembling walls. There are only two basic load cases with one load combination:

1. LC1_Grav (Gravity load)

2. LC2_Surface_Pressure (1 Bar surface pressure acting on the plates)

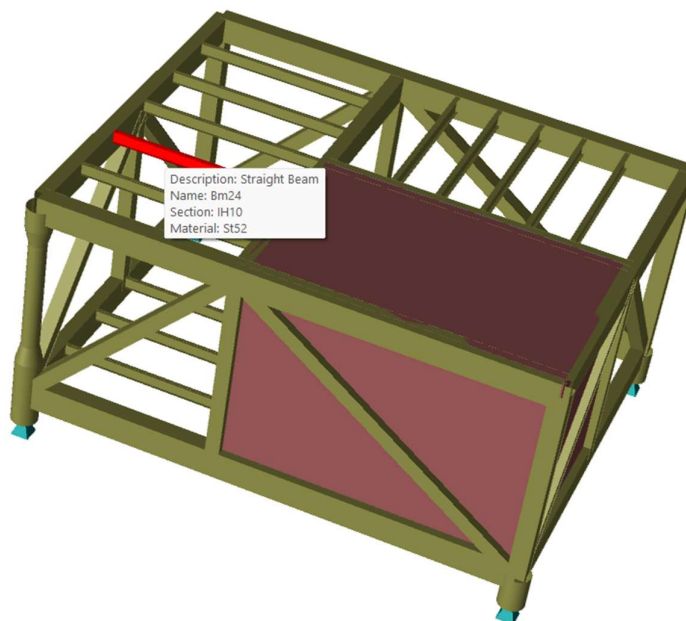3. Blast_Comb (Load combination for LC1 and LC2)
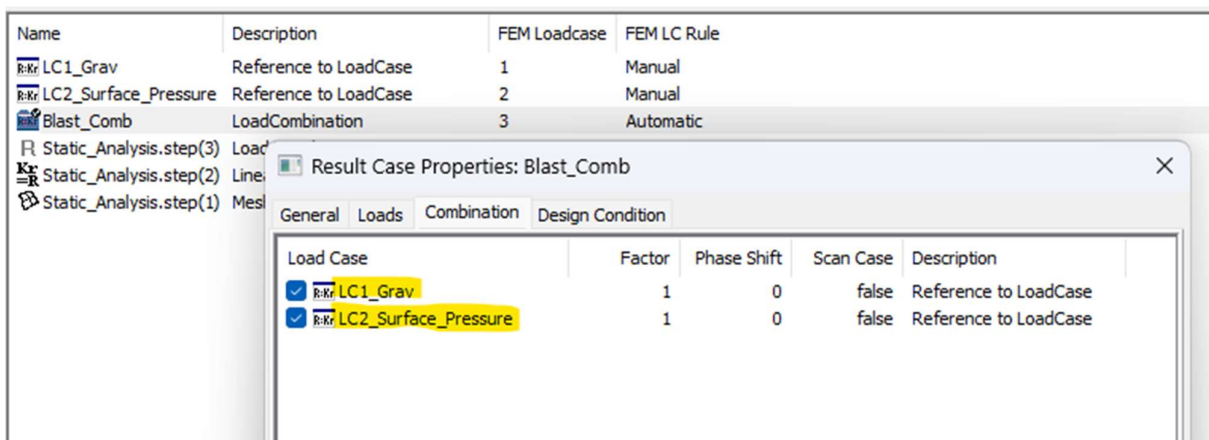
2.1 Run Analysis and Export Usfos File

Once the model is imported, open Activity Monitor (ALT+D) and run the analysis. The analysis should be completed without any error:
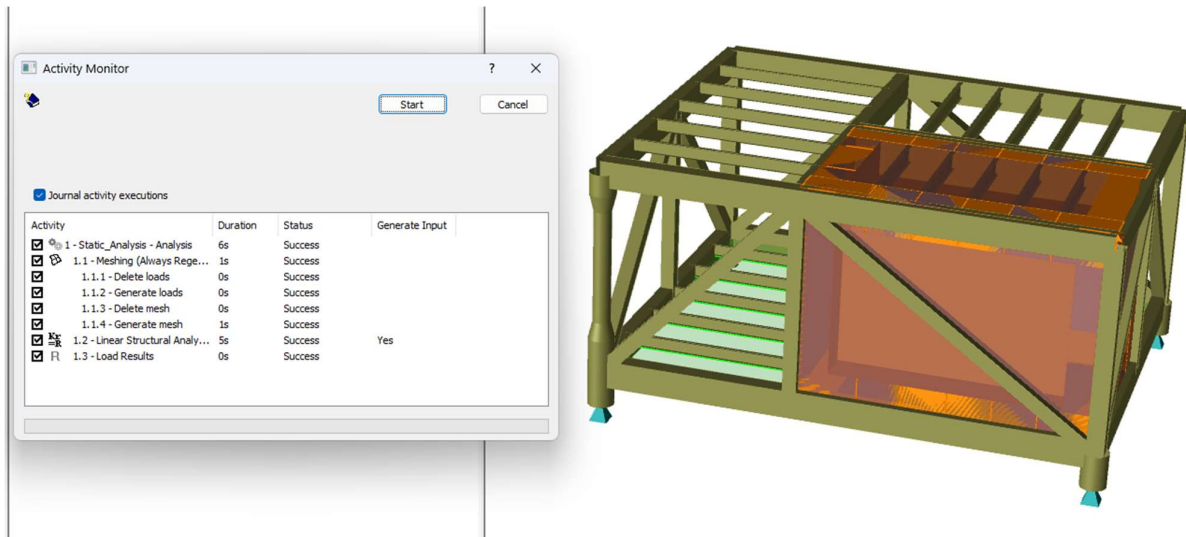
In GeniE, select any beam at the module. GeniE will use this beam end (either end 1 or end 2) as reference for CNODES command in Usfos. Please note that your selection will not affect any results as we will use different control files later. For example, we select Bm24:

Then, go to File > Export > Export to Usfos. Under the Output Tab, Blast_Comb load combination should be automatically selected:

You can use Pushover analysis control as GeniE is yet to support Blast Analysis control file. Leave everything else as default in the Output Tab.

Under the Pushover tab, examine the input. There is no need to change anything here:
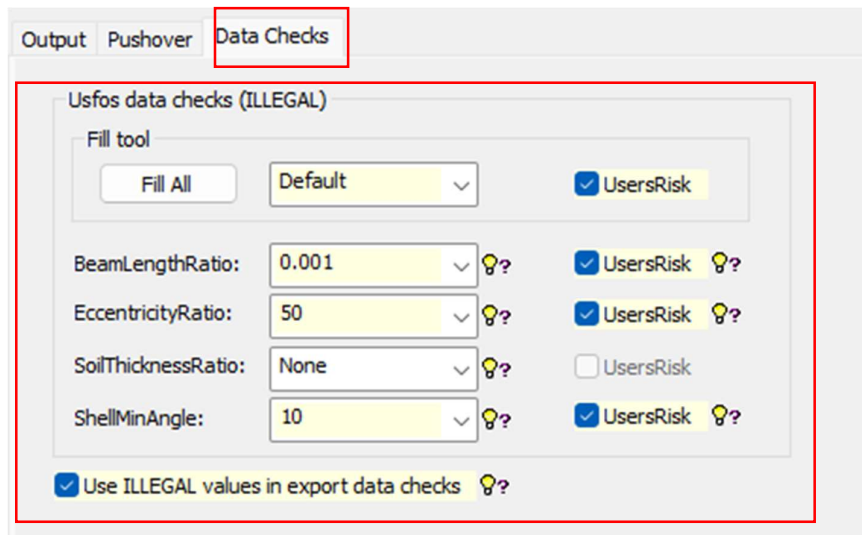
Noticed that there are two load cases in the Blast_Comb load combination that will be exported into Usfos:

1. LC1_Grav : Gravity load

2. LC2_Surface_Pressure : 1.0 x 1E5 N/m2 (1 Bar)

Click on Data Checks tab. Set the settings below:

Click SAVE and now GeniE will export the model into Usfos files. Once the process is completed, go to File > Explore > Current Workspace and check for these files:

1. GeniEActivity1_control.fem (basic control file for push over analysis). This will be replaced by head.fem file.

2. GeniEActivity1_ufo.fem (model file in Usfos format)

3. GeniEActivity1_ufo_load.fem (load file in Usfos format)

2.2   Opening Model in Usfos

In GeniE, go to Mesh & Analysis > Non-Linear Analsis (Usfos) and this will directly open Usfos Graphical User Interface (Xact) from GeniE.

Go to File > Open Usfos Model File and search for GeniEActivity1_ufo.fem:
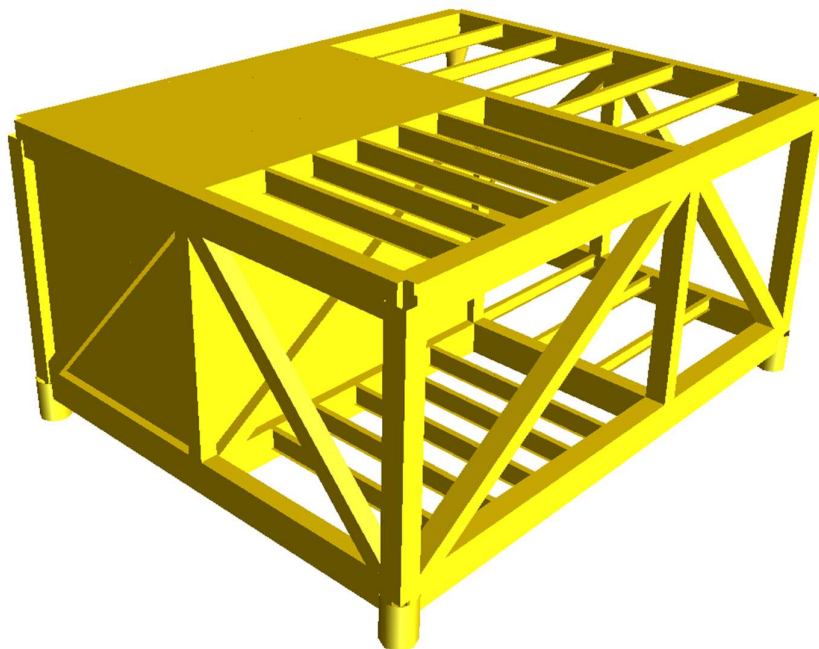
Static case 1 / 1

USFOS GUI 3.2.1   2025-04-28
15:20

2.3   Editing Control File

Now we will use Usfos Analysis Control Edit feature to edit GeniEActivity1_Control.fem file. In Xact, go to Analysis > Usfos Analysis Control. Select the files accordingly and click Edit:

USFOS Analysis Control

Files

Control: C:\DNVGL\Workspaces\Blast_Analysis_Test\GeniEActivity1\head.fem   [...] [Edit]

Model: C:\DNVGL\Workspaces\Blast_Analysis_Test\GeniEActivity1\GeniEActivity1_ufo.fem   [...] [Edit]

(Optional): C:\DNVGL\Workspaces\Blast_Analysis_Test\GeniEActivity1\GeniEActivity1_ufo_load.fem   [...] [Edit]

Result: C:\DNVGL\Workspaces\Blast_Analysis_Test\GeniEActivity1\res   [...]

Editors

| 1: Control | 2: Model | 3: (Optional) |

```
HEAD    Dynamic Blast Analys- MODULE
'
'              Key       Opt        Value
ILLEGAL      BeamLength    Accept      0.001
ILLEGAL      BeamLength    UsersRisk     ON
ILLEGAL    Eccentricities  Accept      50.
ILLEGAL    Eccentricities  UsersRisk     ON
ILLEGAL      ShellAngle    Accept      10.
ILLEGAL      ShellAngle    UsersRisk     ON
'
'
'        End_Time   Delta_T   Dt_Res    Dt_Pri
'
 Dynamic  0.2000    0.010     0.0100     0.00100
 Dynamic  0.6000    0.0001    0.0100     0.000100
 Dynamic  1.0000    0.0010    0.0100     0.00100
'
'        LoadCase   TimeHist
LoadHist     2         1
LoadHist     1         2
TimeHist     1 Point            ! EXPLOSION
                          0.000     0.000
                          0.200     0.000
                          0.300     1.000    ! Peak Presure = 1.0 x 1E5 N/m2
                          0.400     0.000
                          1.000     0.000
'                         2.000       0.000
TimeHist     2 Point            ! Gravity
                          0.000     0.000
                          0.100     1.000    ! Gravity Load applied
                          0.200     1.000    ! Extrapolation
                          1.000     1.000
'                         2.000       1.000
'
'        Ratio1  Ratio2  Freq1   Freq2    Hist
DampRatio  0.02    0.02   0.100  10.000    0
'
'
'        LCase  aX  aY  aZ
' GRAVITY   1    0   0   -9.81
LumpMass  1
'
'
' --------------------------- E O F ------------------------
'        ncnods nodeID dof fact
'
'        ncnods
CNODES       1
'        Node Id     idof        dfact
            135        1         1.
            135        2         1.
            135        3         1.
'
Illegal BeamLength Accept 0.3
Illegal BeamLength UsersRisk ON
CSAVE  0   1  1
USERFRAC   MATERIAL      STRAIN     0.15     1
'
```

[Select Font...]   [Save As...] [Save] [Close]

Output

Command line: [Apply]

☑ Output ☑ Editors ☐ Open *.out file ☐ Always on top   Memory: 200 [⇅] [Run] [Abort] [Close]

After clicking Edit, Editors will display the control file, and editing can be done (no changes needed for the example). Noticed that we will use head.fem file (provided in this example file) instead of the auto-generated *_control.fem file. The auto-generated control file is for push over analysis while the head.fem file is the default blast analysis control file for this model.

Blast Analysis will be controlled by DYNAMIC command card in Usfos.

```
'              End_Time   Delta_T  Dt_Res      Dt_Pri
'
   Dynamic    0.2000     0.010    0.0100      0.00100
   Dynamic    0.6000     0.0001   0.0100      0.000100
   Dynamic    1.0000     0.0010   0.0100      0.00100
'
```

The control simply means that the Dynamic analysis is started with a time increment of 0.01s which is kept until time 0.2s is reached. Then the time increment is increased to 0.0001s, which again is changed to 0.001s at time 0.6s. The analysis will terminate at time = 1.0s

In combination with Dynamic command, LoadHist and TimeHist command is used to call in load vectors according to time histories. In this example, load case 2 (surface pressure) is called for time history 1 and load case 2 (gravity) is called for time history 2.

```
'          LoadCase    TimeHist
LoadHist        2         1
LoadHist        1         2
TimeHist     1 Point            ! EXPLOSION
                         0.000       0.000
                         0.200       0.000
                         0.300       1.000    ! Peak Presure = 1.0 x 1E5 N/m2
                         0.400       0.000
                         1.000       0.000
'                        2.000       0.000
TimeHist     2 Point            ! Gravity
                         0.000       0.000
                         0.100       1.000    ! Gravity Load applied
                         0.200       1.000    ! Extrapolation
                         1.000       1.000
'                        2.000       1.000
```
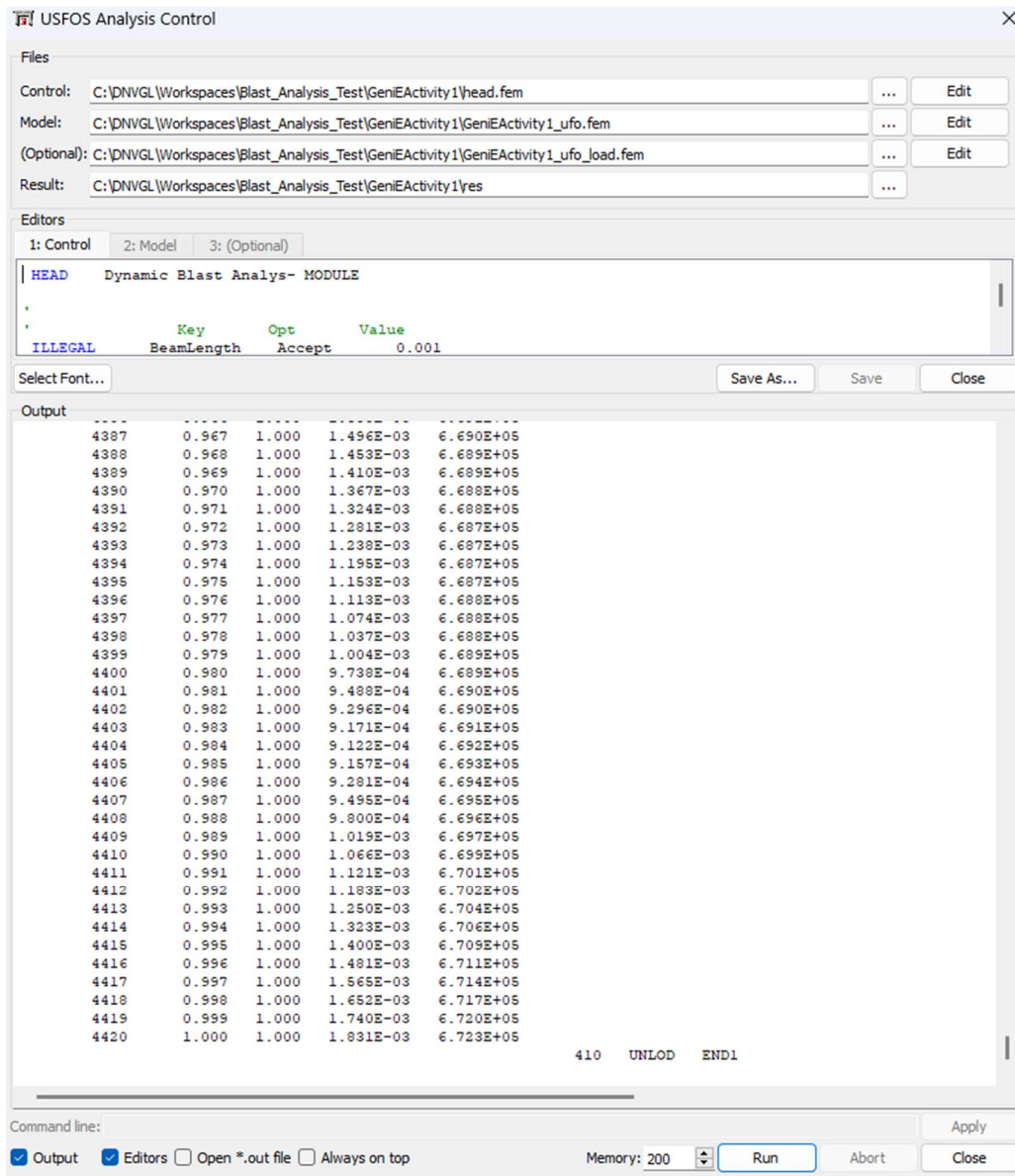
For detailed information about the control card used in Usfos, please refer to Usfos_UM_06(9.0)_Control_Parameter.

2.4   Run Blast Analysis

In Usfos Analysis Control, click Run to execute the Blast Analysis. In normal execution, Usfos analysis completed will be written in the output:

Note that the analysis might take some time to finish (approximately 40 minutes but varies according to the computer's performance).

2.5   Blast Analysis Result and Output

Open res.raf result file in Xact. This will open the graphical result for Blast Analysis. The default global displacement vs time graph will be displayed:
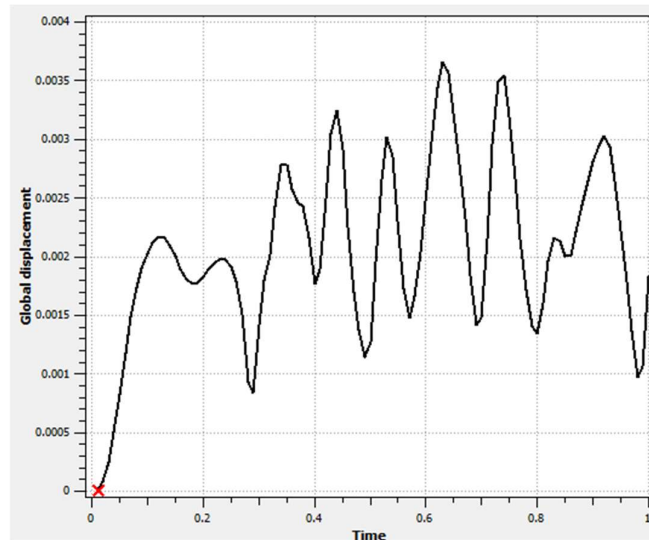
To view the animation, go to Results > Animate:
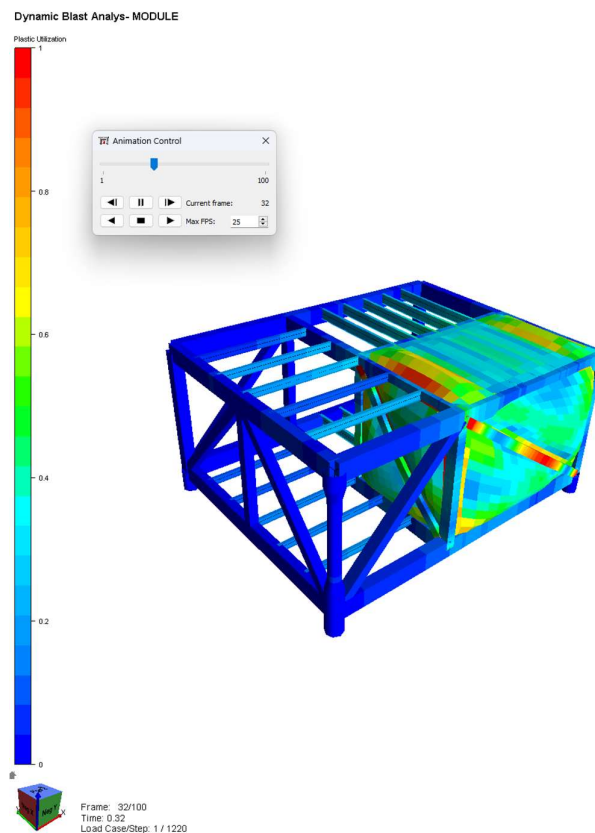
To change the scale, go to Results > Displacement Scaling and change the scale factor accordingly.

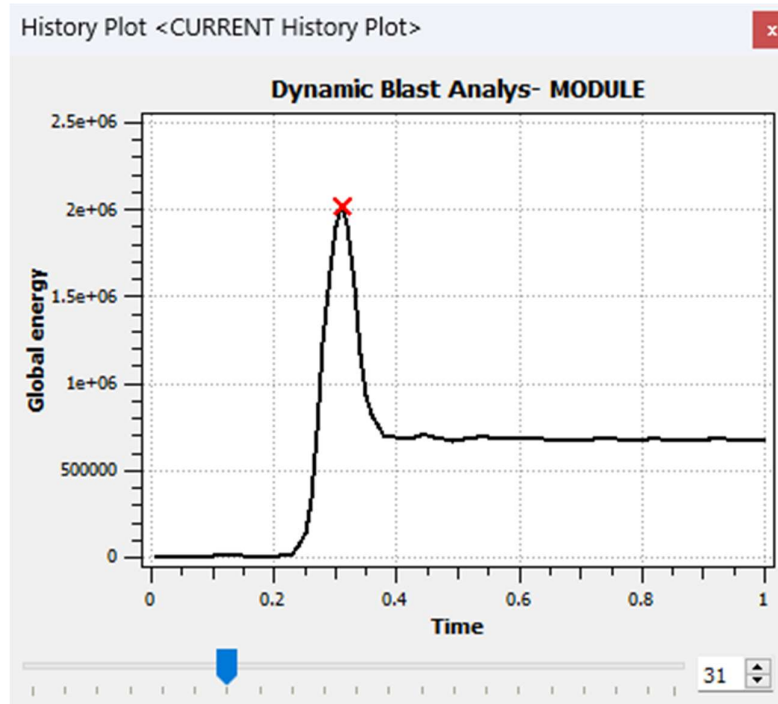Go to Results > History Plot and create a new graph for Global Energy vs Time for 1 Bar explosion:

Check res_status.txt for any buckling or fracture recorded.

Try running the Blast Analysis again but this time, change the factor to 5.

```
'            LoadCase    TimeHist
  LoadHist        2         1
  LoadHist        1         2
  TimeHist      1 Point         ! EXPLOSION
                               0.000       0.000
                               0.200       0.000
                               0.300       5.000    ! Peak Presure = 5.0 x 1E5 N/m2
                               0.400       0.000
                               1.000       0.000
'                              2.000       0.000
  TimeHist      2 Point         ! Gravity
                               0.000       0.000
                               0.100       1.000     ! Gravity Load applied
                               0.200       1.000     ! Extrapolation
                               1.000       1.000
'                              2.000       1.000
'
```

Rerun the analysis and compare the structural response, global energy and check if any fracture is recorded. You also can play around with the time increment (Delta_T) under DYNAMIC command card and see if that affects the results.

For more information about how to use Xact in investigating the results file (res.raf), please refer to Xact user manual.

About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.