WHEN TRUST MATTERS

SESAM

EXAMPLE

Boat Impact Analysis using GeniE and Usfos

Date: April 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 boat impact analysis for an offshore fixed platform 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:

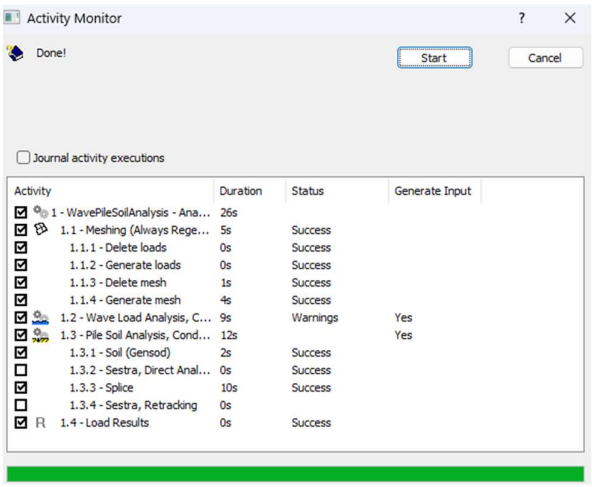
1. GeniE 8.12.2
2. Sestra 10.19.0
3. Usfos 9.0.0
4. Wajac 7.13.0
5. Splice 8.2.1
6. Sesam Manager 6.7.0
7. Application Version Manager V3.1-01

2 BOAT IMPACT ANALYSIS - USFOS

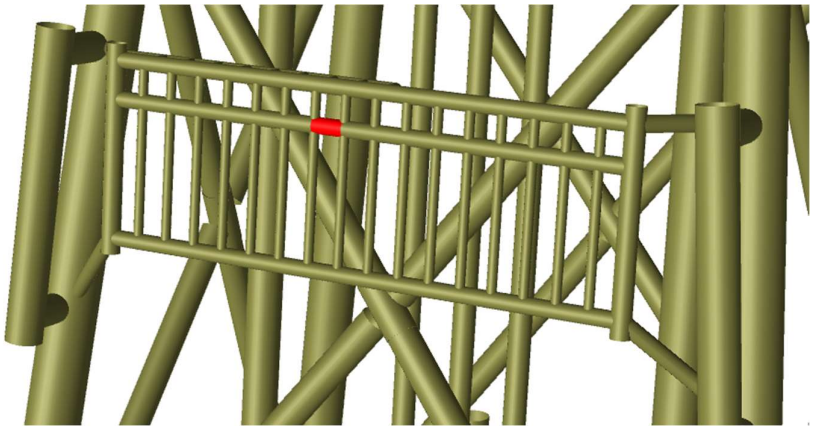
Import **usfos_model.gnx** file into GeniE new workspace (either using Sesam Manager or directly in GeniE). This model is a completed model, prepared for the boat impact analysis in Usfos.

2.1 Run Analysis and Export Usfos File

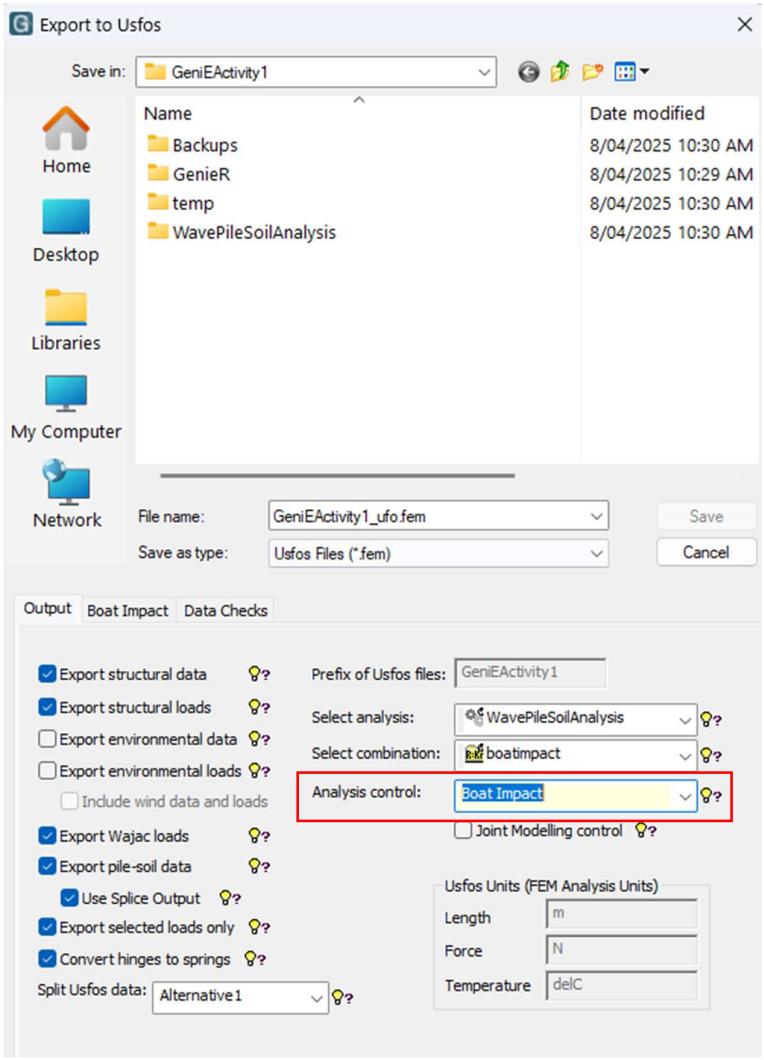
Open activity monitor (ALT+D) and run the analysis. Note that boatimpact load combination is set as current load. The analysis should be completed without no error:



In GeniE, select BM25 at the boat landing:

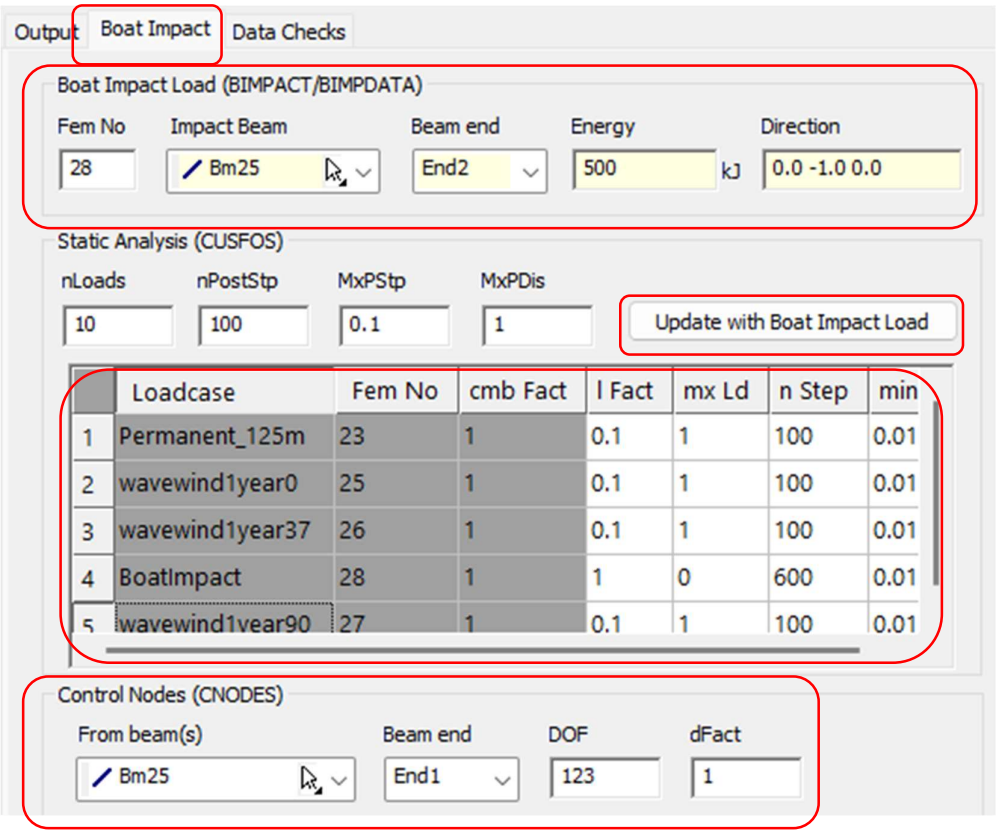


Go to File > Export > Export to Usfos. Under the Output Tab, Boat Impact load combination should be automatically selected:

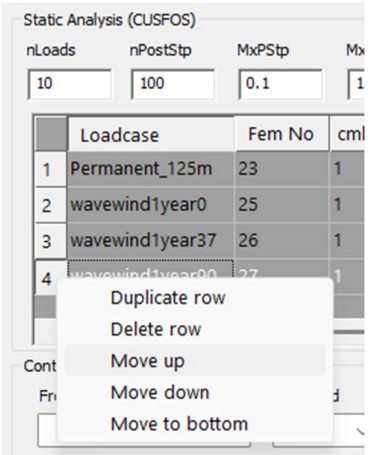


Change the analysis control from Push Over to Boat Impact. Leave everything else as default in the Output Tab.

Under the Boat Impact tab, select the impact beam, beam end, impact energy and direction of the impact. After that click “Update with Boat Impact Load” and boat impact load will be added under CUSFOS load case. Arrange the load case sequence accordingly.

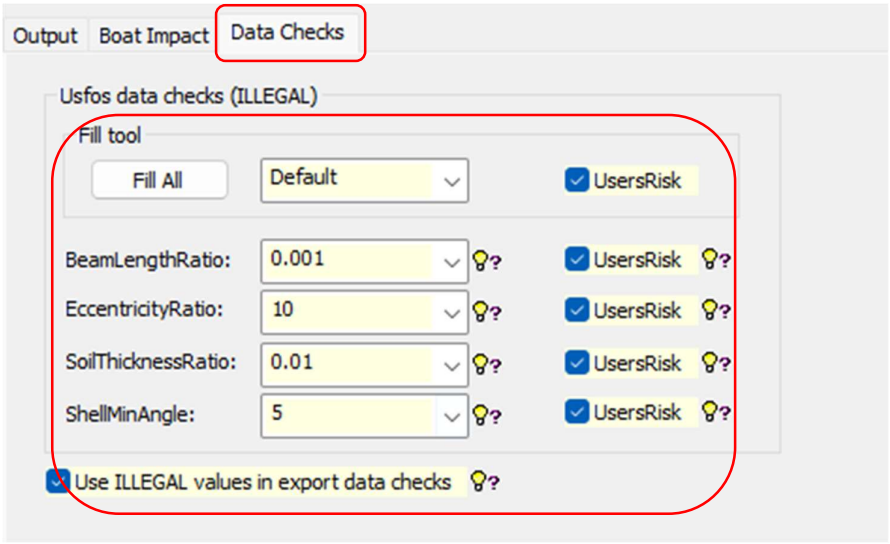


Right mouse button clicks on the load case number to move up or move down in the sequence:



The control nodes are automatically filled when we select BM25 in GeniE prior to export.

Under Data Checks tab, specify the acceptance value for beam length ratio, eccentricity ratio, soil thickness ratio and shell minimum angle. For more information about this, please refer to Release_Notes_for_Usfos_8-6 and 8-7.



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

1- GeniEActivity1_control.fem (basic control file for push over analysis)

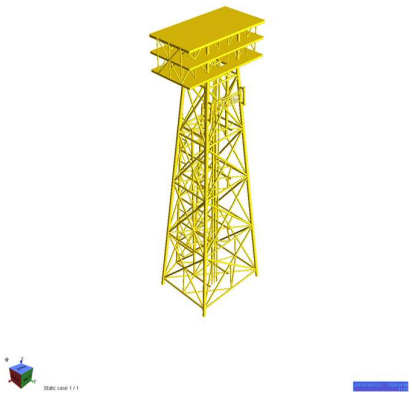
2- GeniEActivity1_ufo.fem (model file in Usfos format)

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

2.2 Opening Model in Usfos

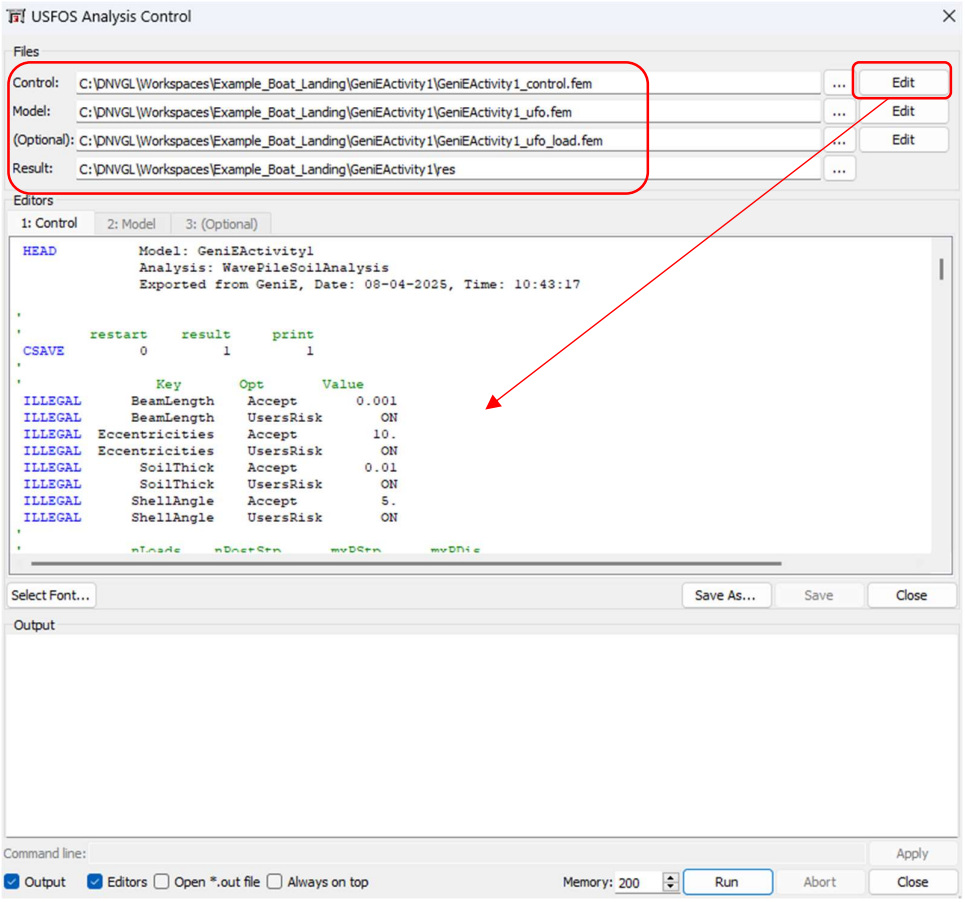
In GeniE, go to Mesh & Analysis > Non-Linear Analsis (Usfos) and this will directly open Usfos Graphical User Interface (Xact) from GeniE.

Go to File > Open Usfos Model File and seach for GeniEActivity1_ufo.fem:



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:



After clicking Edit, Editors will display the control file and editing can be done.

Under CUSFOS command, exclude load case 25 and 26 using anastrophe (‘) at the beginning of the line. These are load cases for 1 year environment wave at 0^0 and 37^0 respectively. We want to run boat impact (load case 28) and then 1 year 90^0 wave loading (load case 27):

```
'              nLoads     nPostStp      mxPStp      mxPDis
 CUSFOS            10          100         0.1          1.
'      Load Case Id          lFact         mxLd       nStep       minStp
                  23            0.1           1.         100         0.01
'                 25            0.1           1.         100         0.01
'                 26            0.1           1.         100         0.01
                  28            1.            0.         600         0.01
                  27            0.1           2.         100         0.01
'
```

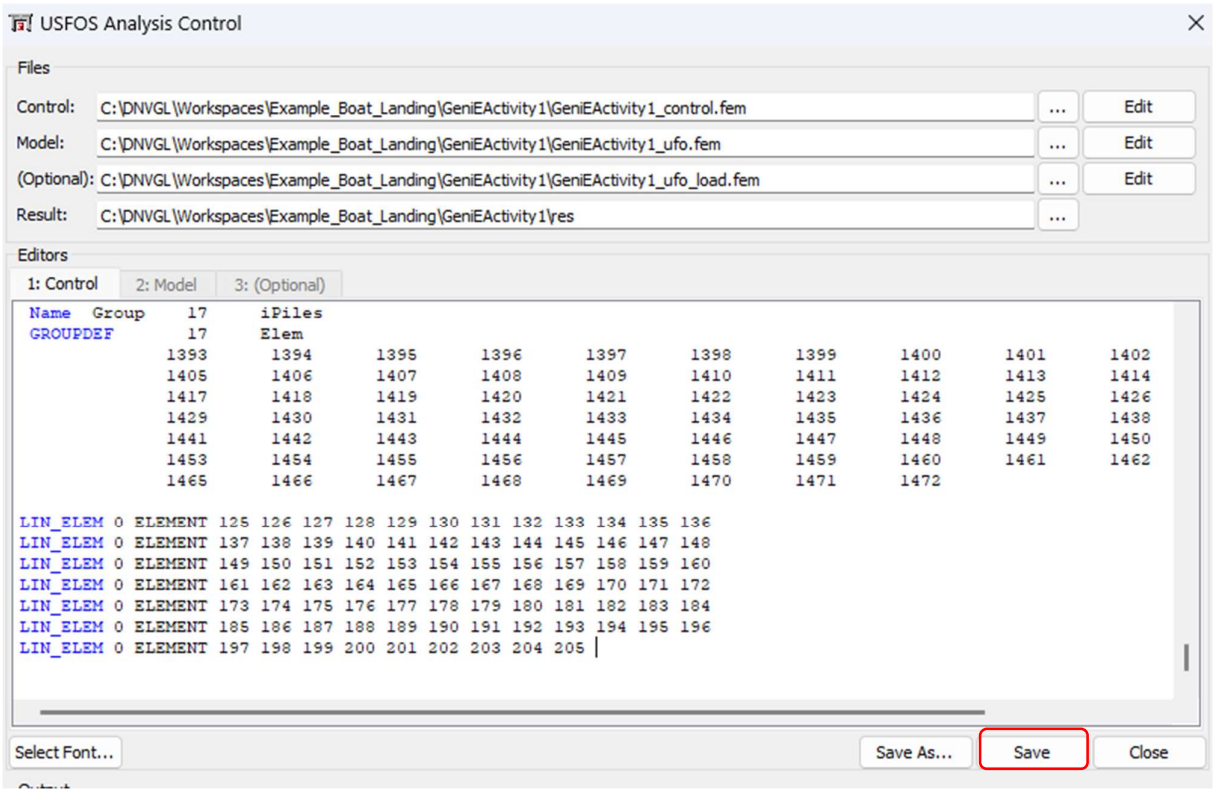
Change the max load to 2 for load case 27.

Linear element:

LIN_ELEM command is used to specify elements which should be linear. The development of plastic hinges is suppressed in all 3 nodes. In other words, elements that defined as linear will not go into plastic in push over analysis. This is often helpful for secondary and tertiary beams that do not contribute to the overall strength of the jacket system. In this example, topside, conductors and risers are defined as linear element:

```
LIN_ELEM 0 ELEMENT 125 126 127 128 129 130 131 132 133 134 135 136
LIN_ELEM 0 ELEMENT 137 138 139 140 141 142 143 144 145 146 147 148
LIN_ELEM 0 ELEMENT 149 150 151 152 153 154 155 156 157 158 159 160
LIN_ELEM 0 ELEMENT 161 162 163 164 165 166 167 168 169 170 171 172
LIN_ELEM 0 ELEMENT 173 174 175 176 177 178 179 180 181 182 183 184
LIN_ELEM 0 ELEMENT 185 186 187 188 189 190 191 192 193 194 195 196
LIN_ELEM 0 ELEMENT 197 198 199 200 201 202 203 204 205
```

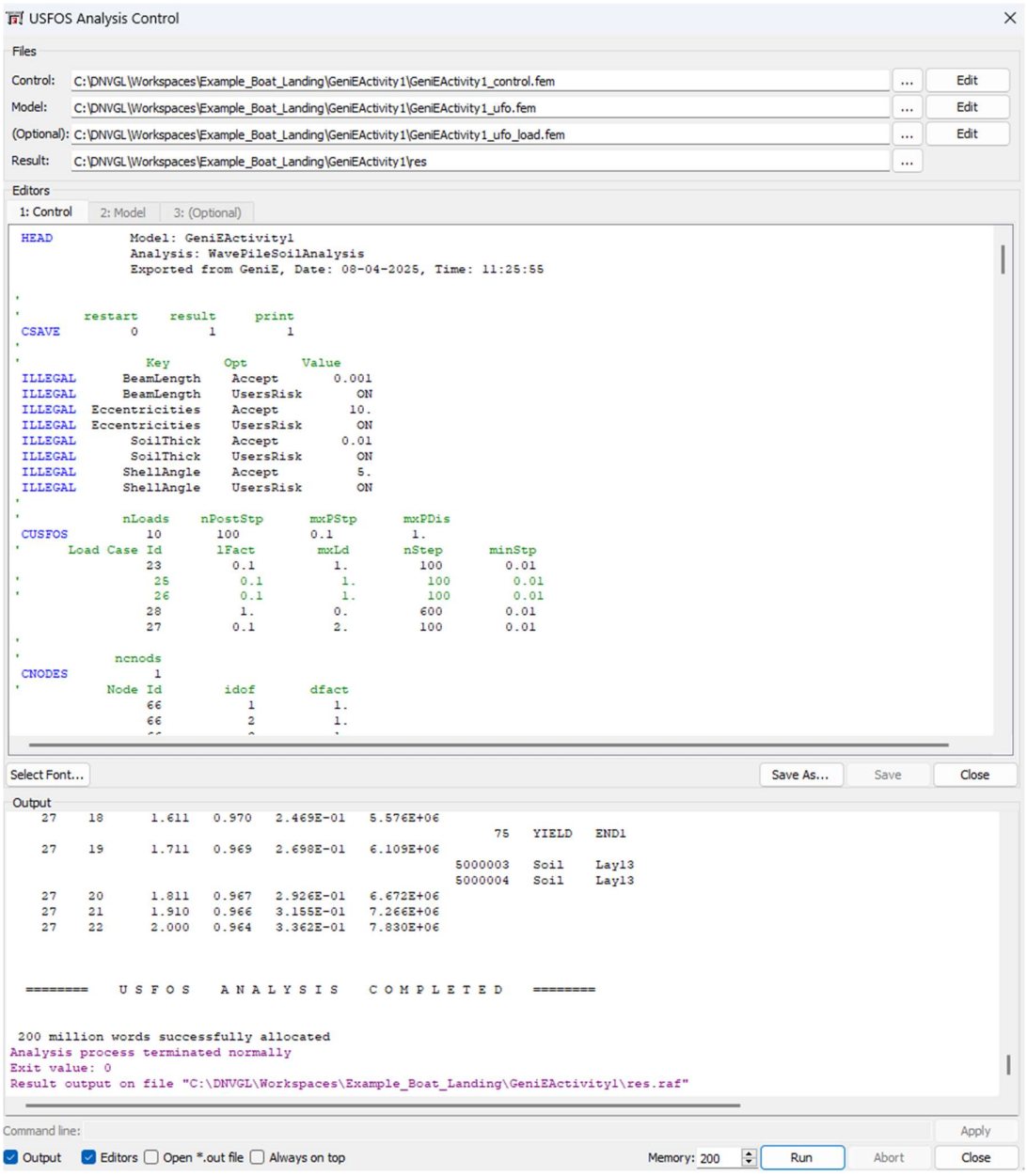
Once the control file has been updated, save the file and now you are ready to run pushover analysis.



Click Save.

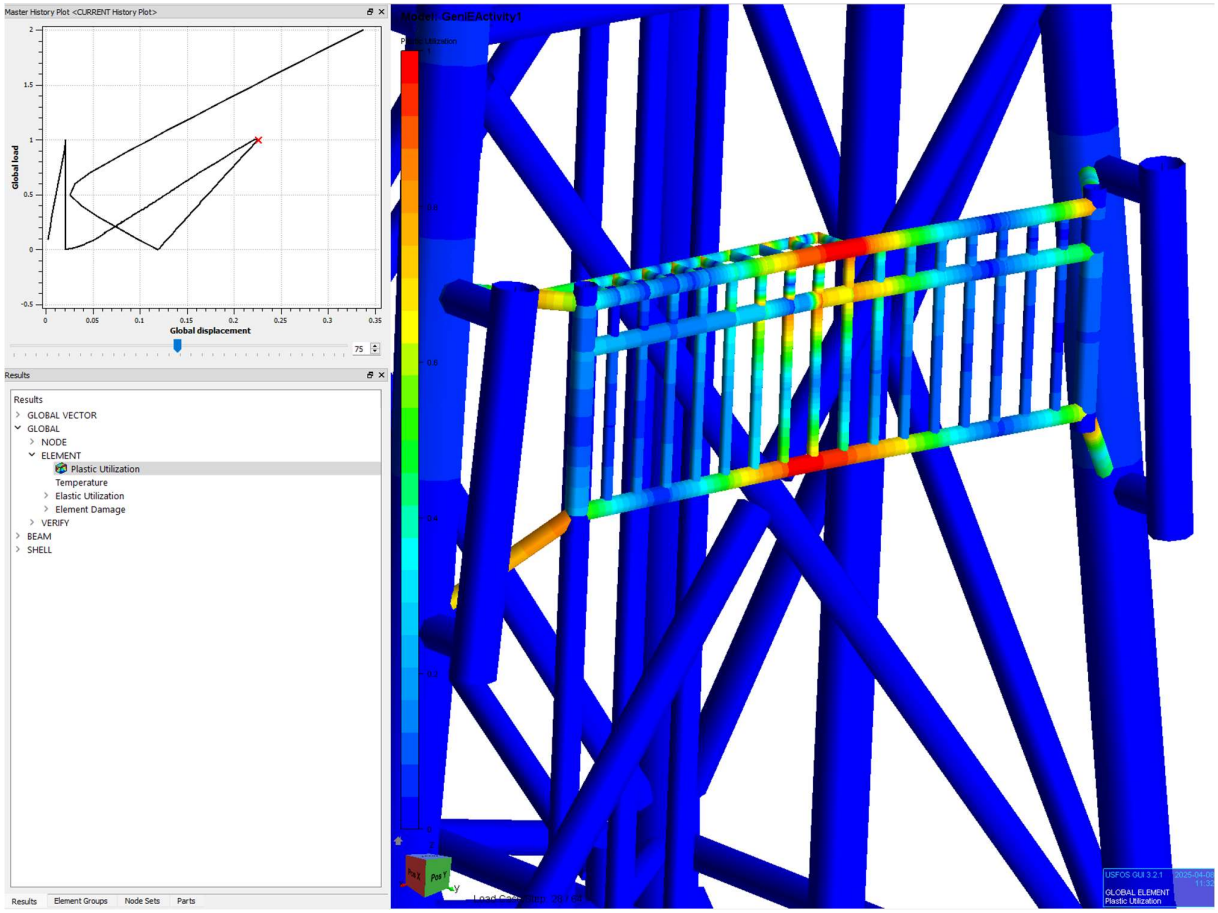
2.4 Run Boat Impact Analysis

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



2.5 Boat Impact Analysis Result and Output

Open res.raf result file in Xact. This will open the graphical result for boat impac analysis. Check the Global Load vs Global Displacement graph as well as the plastic utilization of impacted beam at the boat landing:



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

In boat impact analysis Usfos generates res.out file which among others, it printed out the crucial information about the total energy absorption throughout boat impact analysis. Open res.out file in editor (i.e Notepad ++) and check the total energy absorption:

```
Load step  28 /    64


========     I N C R E M E N T A L        S O L U T I O N         ========

             Model: GeniEActivity1
             Analysis: WavePileSoilAnalysis
             Exported from GeniE, Date: 08-04-2025, Time: 11:25:55


             USFOS load combination no              =        28
             Load step no.                          =        64
             Load increment scaled by new
             plastic hinge at midspan of element          19

             Load increment                         =     0.983
             New load level                         =    13.091
             Current stiffness parameter            =     0.066
             Solution accuracy parameter            = 1.273*E-00006
             Determinant of tangential matrix       = 3.550*E 52294
             Number of Negative Pivot Element       =         0
             Total energy absorbtion                = 1.501*E 00006

             Denting of the tube wall               = 1.428E-01
             Ship indentation                       = 0.000E+00
             Dent      deformation energy           = 1.219E+05
             Ship      deformation energy           = 0.000E+00
             Structure deformation energy           = 3.774E+05
             Total Absorbed Ship Impact Energy      = 4.993E+05
```

Remember that we specify 500 KJ as the impact energy in this analysis example. The total absorbed ship impact energy here is 0.499 MJ. This would suggest that the boat landing could handle 500 KJ impact energy without having fracture member.

For more command in boat impact analysis, user can refer to Section 6.3.12 Ship Impact Analysis in Usfos User Manual.