Some Tricks for Results Combination

Herbert Diehl, PTC Munich

1. Background

The current release of Mechanica (8.0) has a few limitations with respect to superposition of results and combination of loadcases. This paper describes workarounds to most of these limitations. In particular, we will discuss

a) how we can combine loadcases, which were analyzed in different msengine runs, but which should be combined afterwards

b) how we can multiply the results of a single loadcase

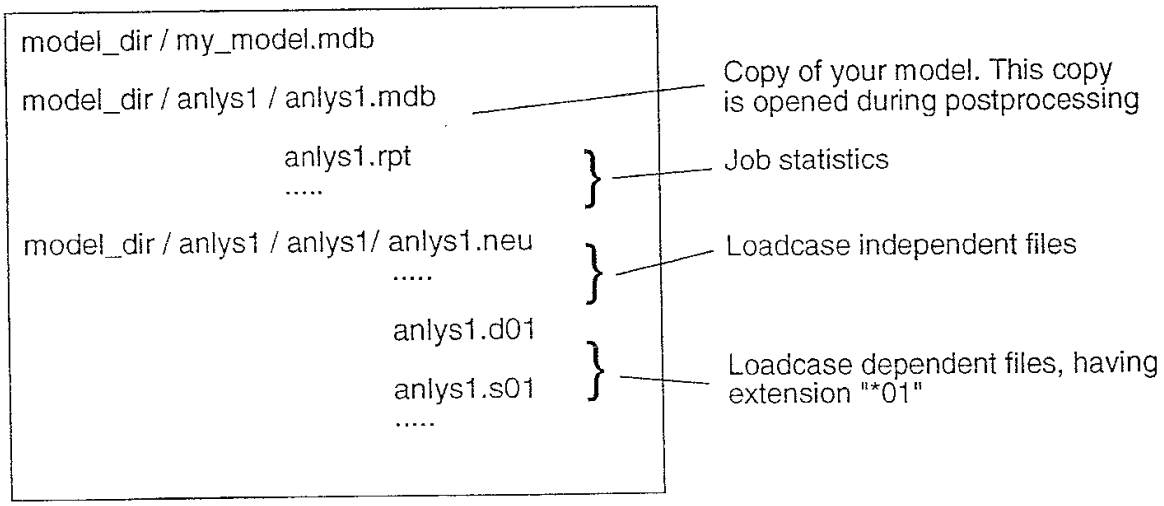
c) how we can combine results of loadcases with different boundary conditions. This is particularly useful in the case of symmetrical structures subjected to non-symmetrical loads.

2. Notation / Prerequisites

We assume that the reader is familiar with the directory structure of Mechanica results directories, but for the sake of completeness, we will repeat the essentials :

Suppose that you have generated a Mechanica database "my_model.mdb" in a certain directory "model_dir", e.g. "model_dir" = $HOME/my_projects/models and that this database contains loadcases "load1", "load2", "load3" etc and analyses "anlys1", "anlys2" etc.

Moreover suppose that you have defined "anlys1" containing the loadcase "load1". After running "anlys1", the following file structure exists :



3. Combining several loadcases (different loads, identical boundary conditions), which were analysed in separate runs

There are several reasons why you would not want to analyse all loadcases in a single run : Perhaps you don't know the correct combination in advance or you want more control over hardware resources and run big jobs independent from each other. By default however, Mechanica allows for loadcase combination only if the loadcases were part of the same analysis. for instance "anlys1".

Suppose that you have defined and run an analysis "anlys1" with loadcase "load1" and an analysis "anlys2" with loadcase "load2" and you want to superpose the results. This is imposssible with the current file structure and data structure of Mechanica results.

We will perform the following steps to combine the load cases in spite of the limitations in the default file structure and let "load2" appear as second loadcase in "anlys1" :

3.1. Copy the loadcase dependent files from anlys2 to the directory of anlys1 :

cd model_dir / anlys2 / anlys2

cp anlys2.d01 ../../anlys1/anlys1/anlys1.d02
cp anlys2.s01 ../../anlys1/anlys1/anlys1.s02

.... (proceed similarly with .r01 , .p01, .n01)

3.2. Tell the copy "anlys1.mdb" of the original database file that "anlys1" contains 2 loadcases :

cd model_dir / anlys1

mstruct anlys1

Main / Analysisi / Review "anlys1" / activate "load2" for this analysis

Save anlys1.mdb

3.3. Proceed with standard postprocessing

cd model_dir

mstruct

Results / select "anlys1" , the "combine loadcases" menu appears.
Superpose loadcases as desired.

Because the same geometrical boundary conditions were used in both runs, you may not only combine displacements and stresses, but also reaction forces.

4. Multiply results from a single loadcase by a scalar factor

Suppose that you have defined and run an analysis "anlys1" with loadcase "load1" and that you unfortunately defined the wrong loads in the sense that the correct loads differ from the used loads by a certain factor α. Currently Mechanica doesn't allow you to scale results bythe factor α and you would have to re-run the analysis.

Instead of doing this, we can perform the following steps and save a considerable amount of time. You will note that this case is a special case of what we discussed in the previous chapter.

4.1. cd model_dir / anlys1 / anlys1

cp anlys1.d01 anlys1.d02
cp anlys1.s01 anlys1.s02

.... (proceed similarly with .r01 , .p01 , .n01)

4.2. cd model_dir / anlys1

mstruct anlys1

create all loads belonging to loadcase "load1" again and put them into loadcase "load2"

review the analysis "anlys1" and include loadcase "load2"

save anlys1.mdb

4.3. cd model_dir

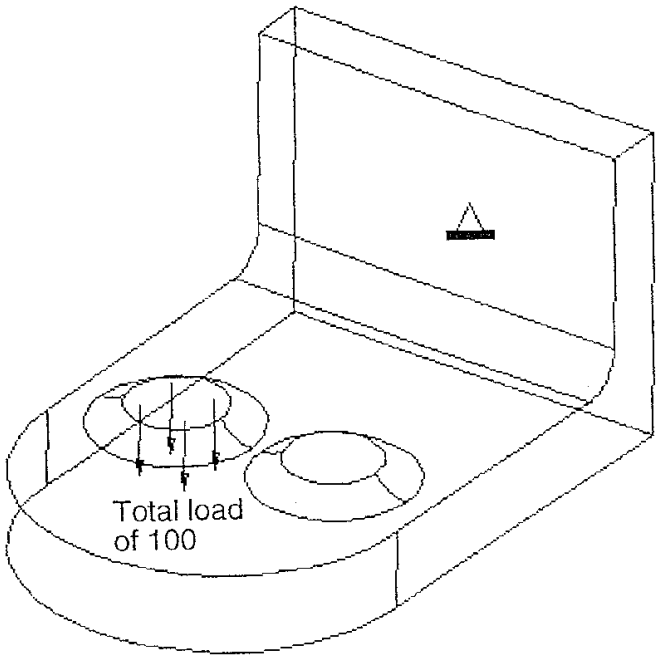
Results / select "anlys1"

in the "combine loadcases" menu, specify a factor 1 for "load1" and a factor ($\alpha - 1$) for "load2"

5. Combine results from loadcases with different boundary conditions

Obviously the procedure outlined in chapter 3 didn't contain any formal restrictions with respect to geometrical boundary conditions. Of course there's the mechanical restriction that it doesn't make sense to superpose certain results like reaction forces if the loadcases refer to different constraint sets, but as far as displacements and stresses are concerned, we may use the procedure described in chapter 3 to combine results from loadcases with different constraint sets. The data contained in the .neu , .s01 and .d01 files do not reference constraint sets.

We will illustrate the procedure for the very important case of a symmetrical structure subjected to non-symmetrical loads. We will model only one half of the structure and combine results from symmetrical and anti-symmetrical constraints.

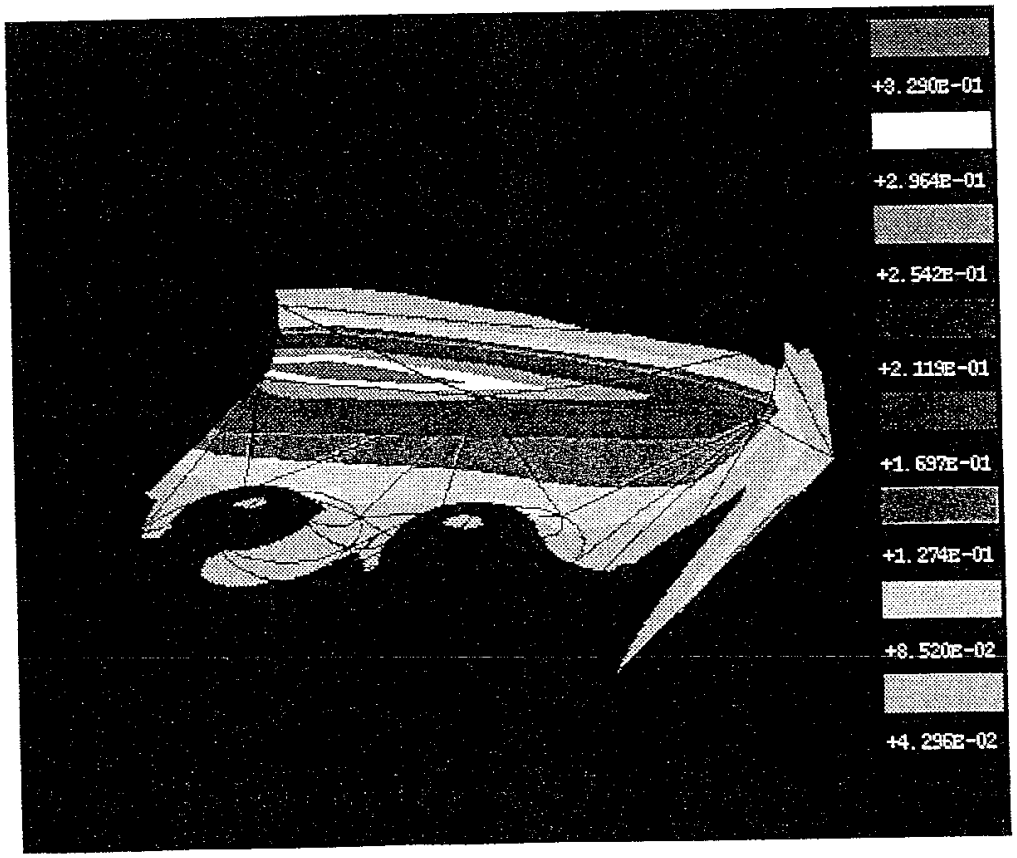


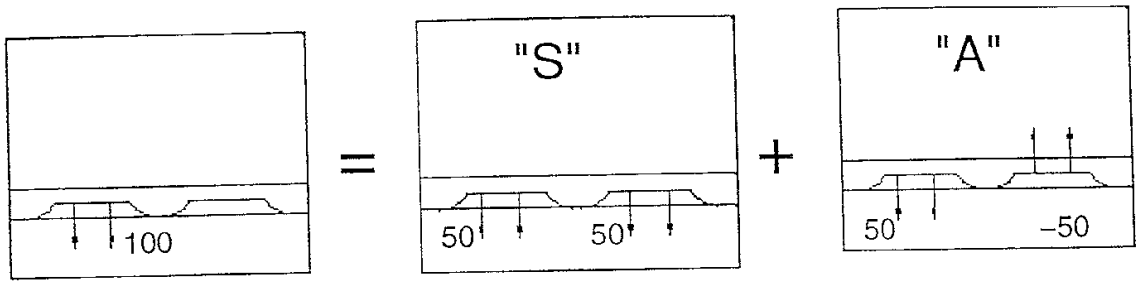
Consider the simple bracket sketched below.

Though the structure as well as the boundary conditions are symmetrical, this can not be used in a straightforward fashion to reduce the size of the model, because the load distribution is not symmetrical.

The full model has been analyzed and results of vonMises stress are shown below. A maximum stress of almost 0.38 occurs at the round.

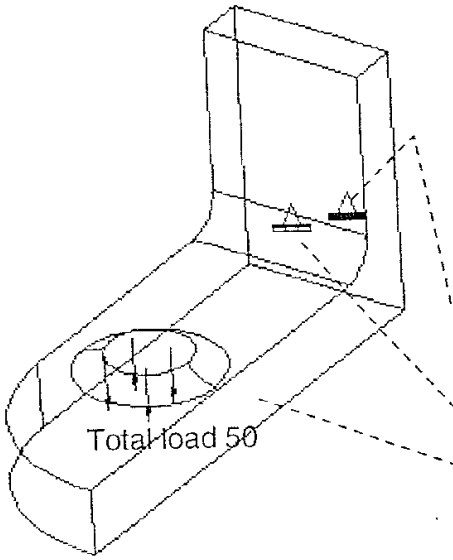
On the next page we will discuss how we can use a half model to get the same results.





As indicated above, we can superpose the results for the original problem from 2 analyses of a symmetrically loaded structure and a anti-symmetrically loaded structure (denoted by "S" and "A") .

For both problem "S" and "A", we can find results by considering a half model only; however, since different boundary conditions have to be used in "S" and "A", we can't assemble the 2 problems into a single Mechanica run.



Now let's perform the following steps to get the stress results working with the half model only:

5.1. Generate the half model , mesh it etc.
Apply a load of 50 to the loaded surface.

5.2. Create this load twice and put it into different loadcases "symm" and "asym"

5.3. Constrain the rear surface. Create the constraint twice and put it into constraint sets "symm", "asym"

5.4. Create symmetrical and anti-symmetrical constraints for the "cut" surface

5.5. Define 2 analyses :

"symm" with loadcase "symm" and constraint set "symm"
"asym" with loadcase "asym" and constraint set "asym"

5.6. Run the analyses.

5.7. Combine the results of "asym" and "symm" using the procedure outlined in chapter 3.

(replace "anlys1" by "symm" , "anlys2" by "asym")

Start Mechanica Structure and retrieve results for analysis "symm". This analysis now formally contains both loadcases and you may combine them with a factor of 1 for both "symm" and "asym"

The results in the case of our bracket are shown below. The maximum vonMises stress is now 0.371. The deviation from the reference value of 0.379 can be explained by the fact that there's always a small error if you compare in Mechanica results from combined loadcases with results from a single run, because all runs converge to a slightly different degree of accuracy.