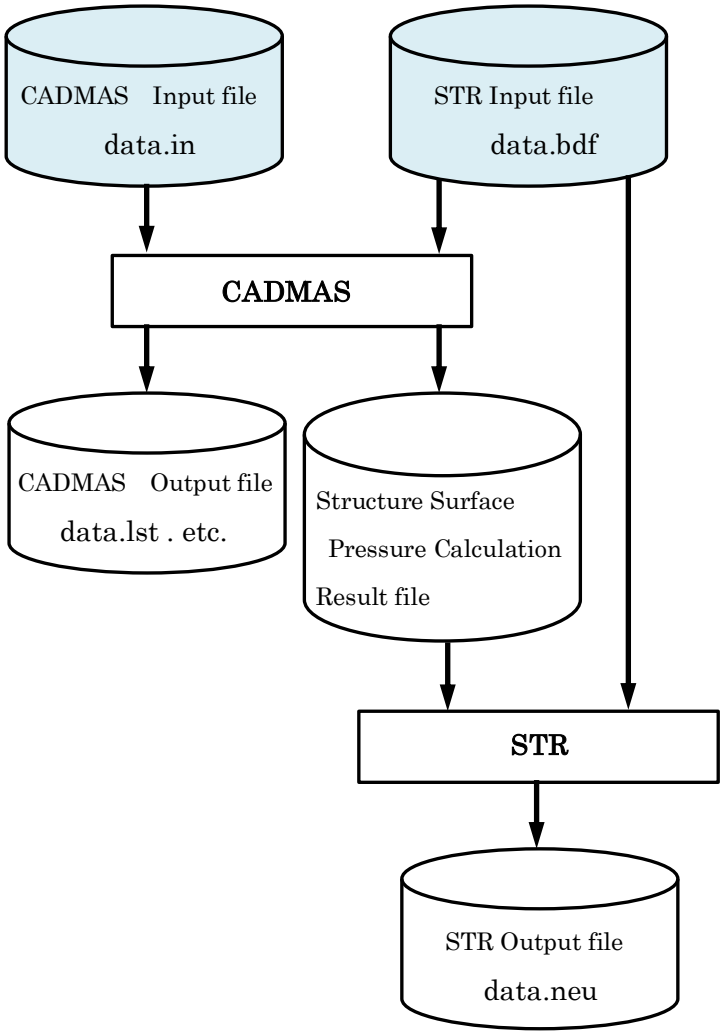
1． One-way coupling

［1］The flowchart of the program is shown below.



［２］Analysis procedure.

（１）Creation of CADMAS data (data.in) file

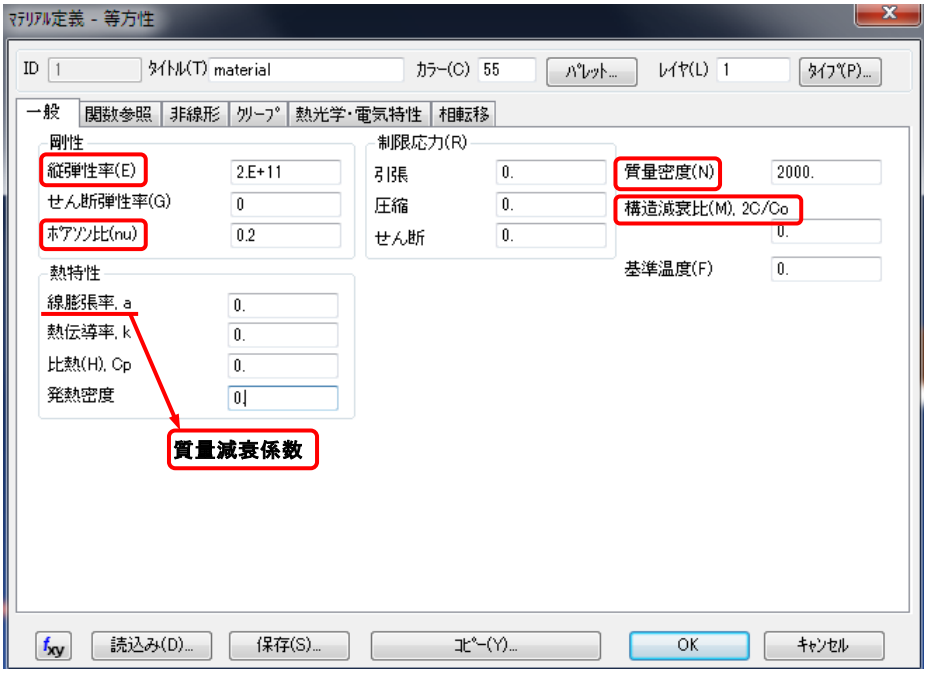
The output interval for pressure on the structural surface to the pressure calculation result file (data.prs) is specified through the time series file output control data (FILE TRN STEP or FILE TRN TIME).

（２）Creation of structural analysis data file (data.bdf)

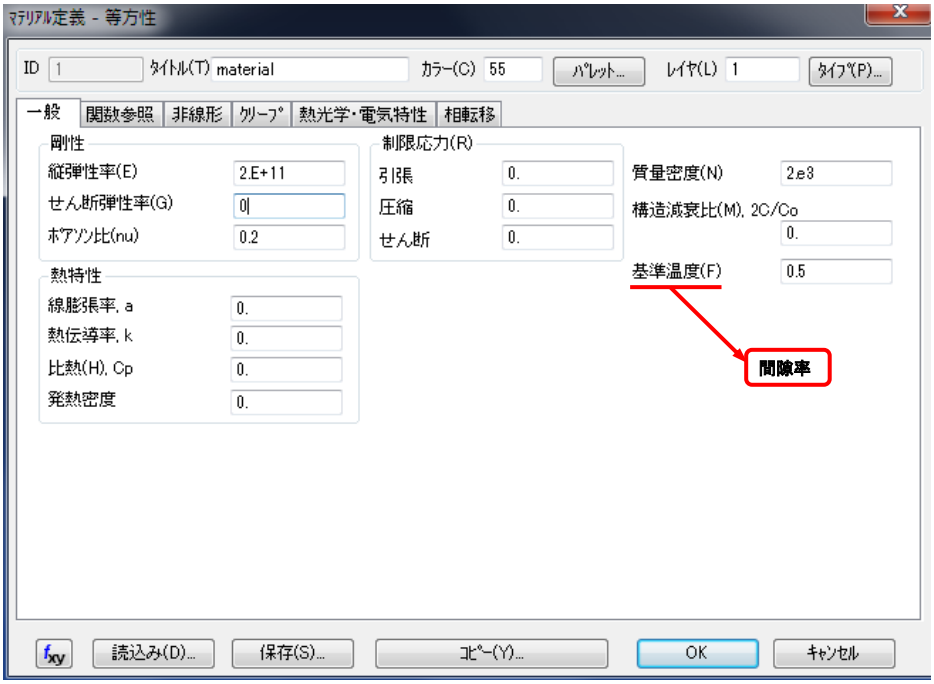
Follow the procedure to create dynamic analysis data in NASTRAN using FEMAP. However, note the following points

①　Material Data (Conventional Material): Enter the following items.

If 0 is entered for the structural damping ratio, the default structural damping ratio will be assigned. For the mass damping coefficient, substitute with a different data item as follows（since it is not supported in NASTRAN）.



② Material Data (Porous Materials such as gravel): In addition to the above ①, enter the following items. Since NASTRAN does not support porous materials, substitute with different data items as follows. Also, at this point, for ①, enter the density of the material itself (e.g., density of the stone for gravel), excluding voids

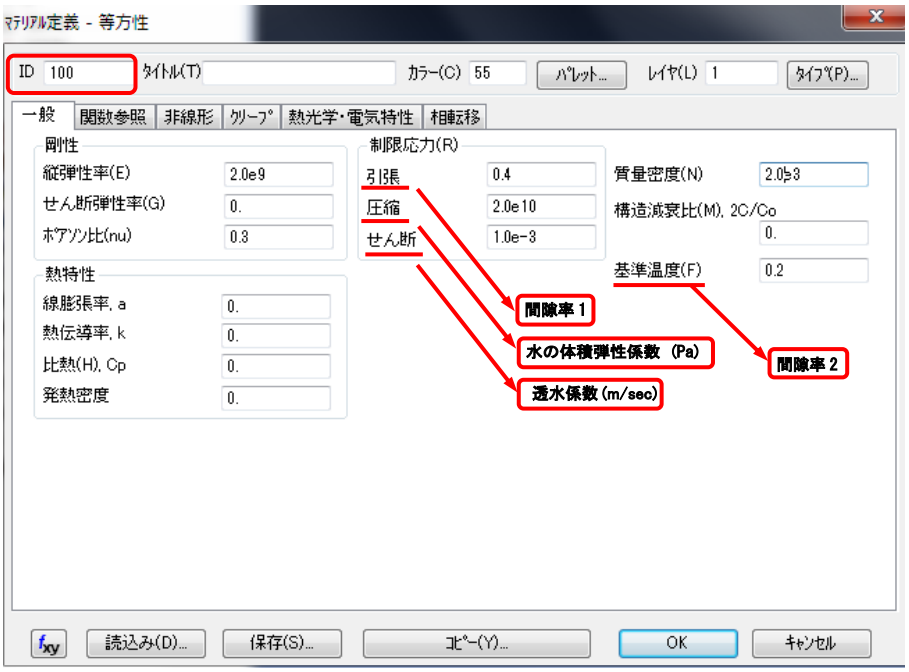


③ Material Data (Subgrade Material): In addition to the above ①, enter the following items. Since NASTRAN does not support subgrade materials, substitute with different data items as follows. Also, set ID ≥ 100 (when ID ≥ 100, the material is recognized as subgrade material). In the following data

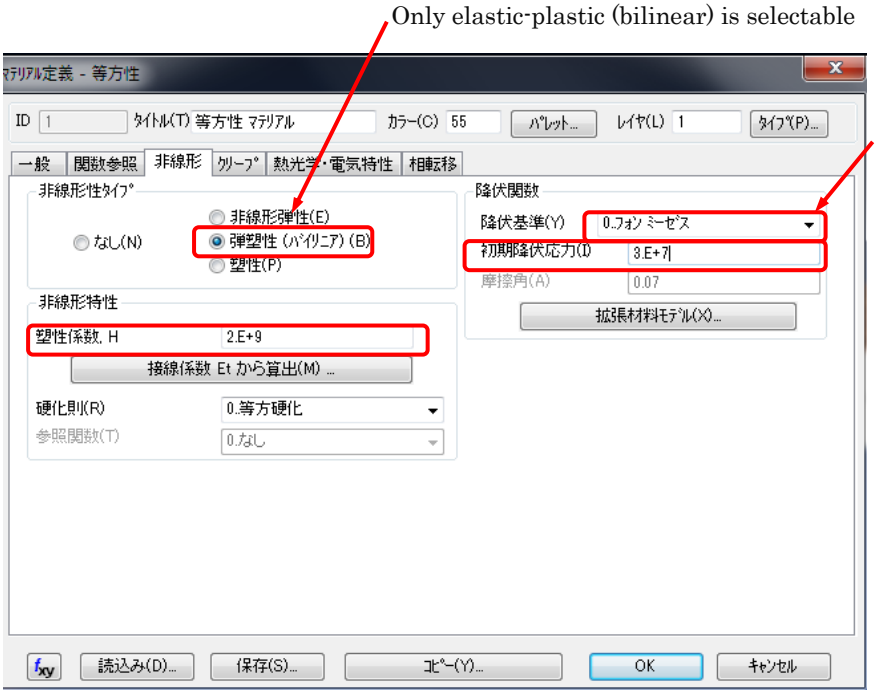
Porosity 1: Porosity used for permeability calculation within STR

Porosity 2: Porosity used for calculation of CADMAS cell void ratio due to interaction with the subgrade in CADMAS.

Also, at this point, for ①, enter the particle density of the subgrade soil as the mass density.

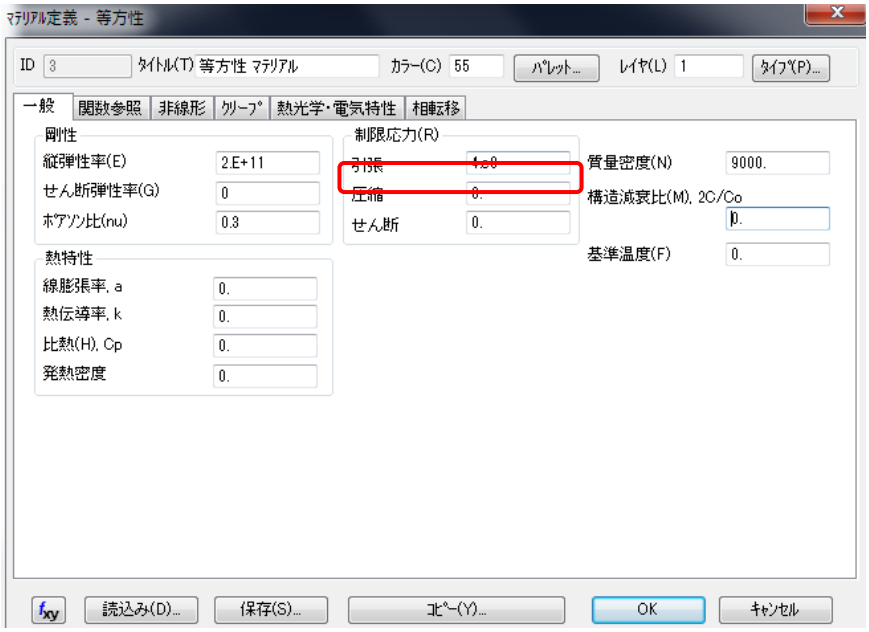


④ Material Data (Elastic-Plastic): To specify the characteristics of elastic-plastic behavior, enter the following items in the 'Nonlinear' tab



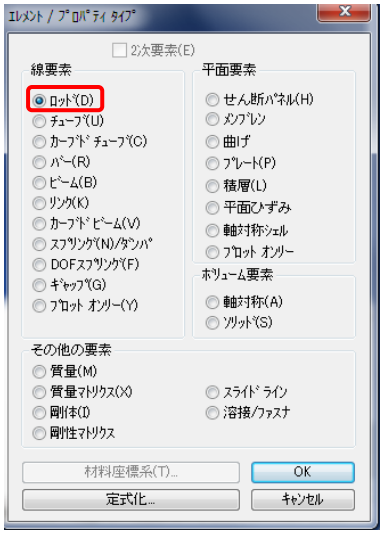
※ In the case of selecting '3. Drucker-Prager,' you may encounter an error message when exporting NASTRAN data stating 'Nonlinear materials only support Von Mises yield criterion.' You can ignore this message. Also, at this point, the 'Friction Angle' value is fixed at 0.07 regardless of the input value. Additionally, for beam elements, only '0. Von Mises' is selectable

⑤ Material Data (Tensile Strength): To specify the tensile strength, enter the following items (tensile strength cannot be specified for subgrade materials).

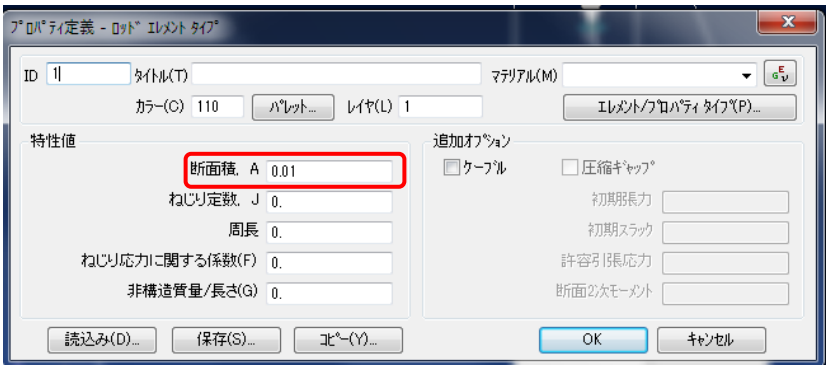


⑥ The definition of characteristics for truss (rod) elements is as follows.

Select 'Rod' under 'Property Definition' > 'Element/Property Type



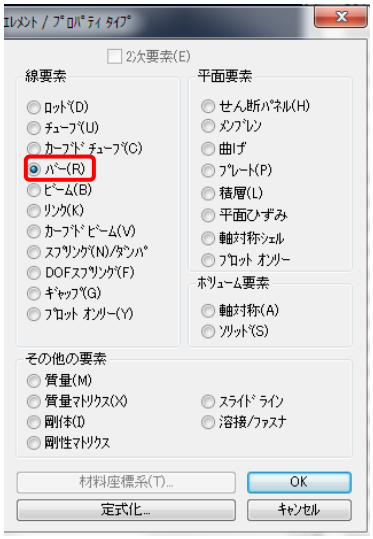
In 'Property Definition,' enter the following items

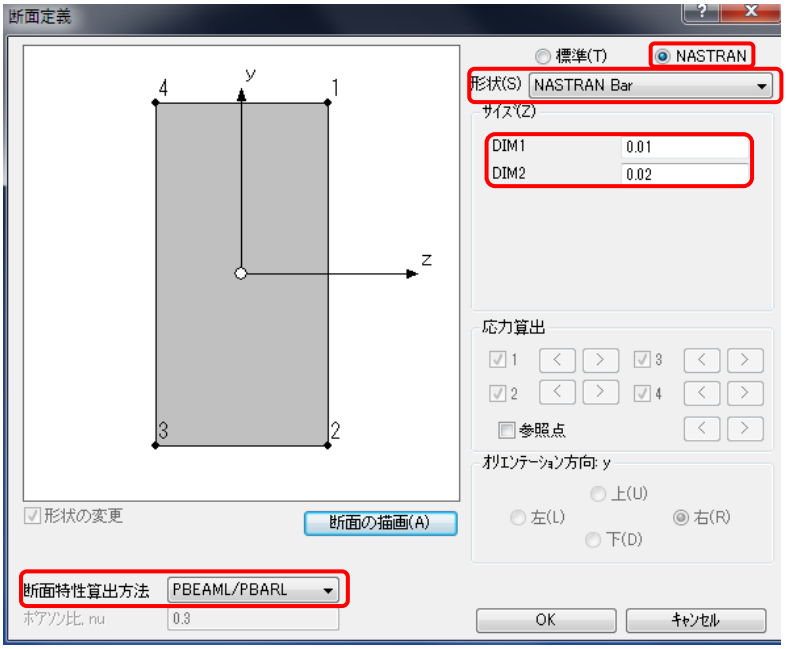


⑥ The definition of characteristics for beam elements is as follows

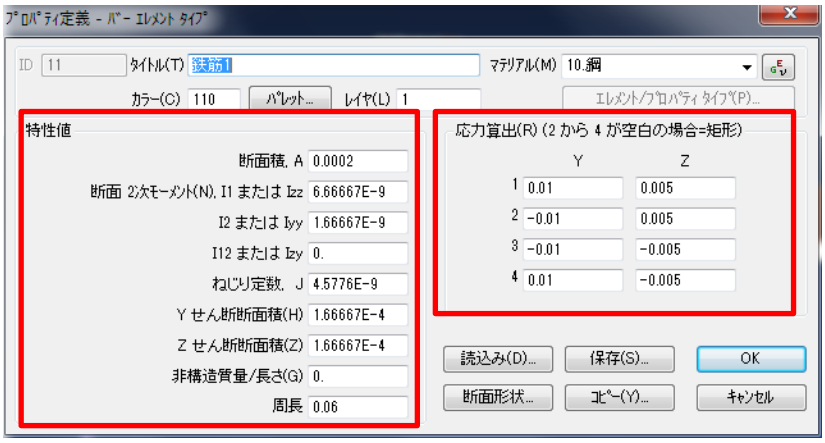
In 'Property Definition' > 'Element/Property Type,' select 'Bar.'

Also, in 'Property Definition' > 'Section Shape,' choose and enter the following items (only rectangular shapes can be defined).





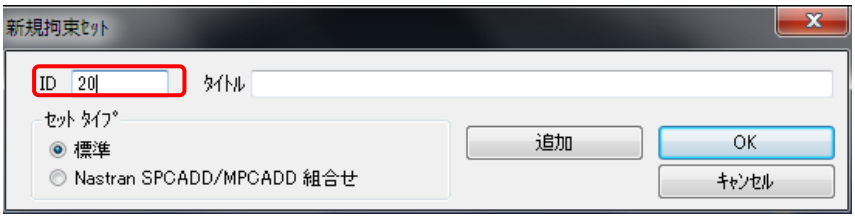
In 'Property Definition,' no input is required for the items 'Characteristic Values' and 'Stress Calculation.



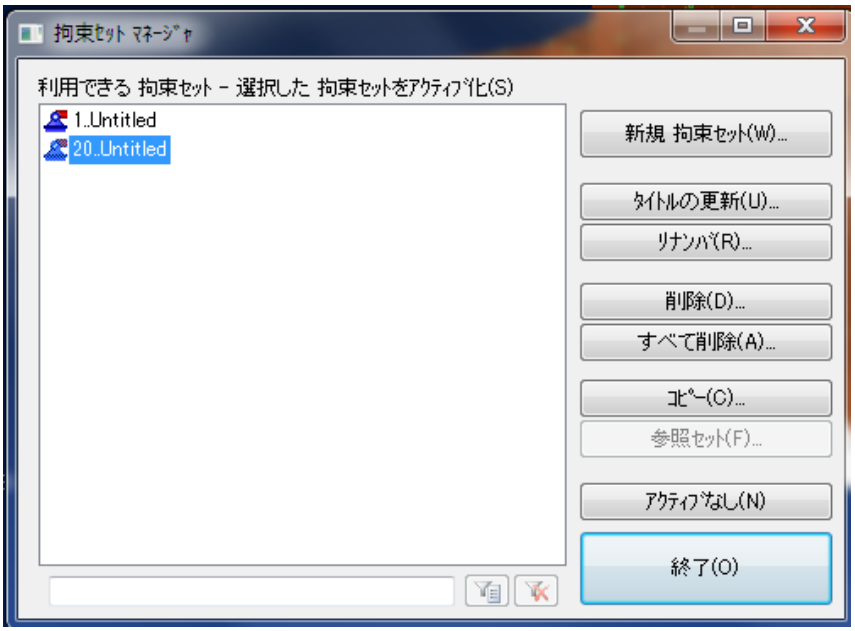
⑧ After setting constraints, the following example illustrates how to add additional constraints that are effective only during static analysis.

In the following example, assuming that the constraint set with ID=1 is already set, we will demonstrate how to add additional constraints with ID=20, which are effective only during static analysis. Additionally, we will show how to define the combination of constraint sets ID=1 and ID=20 as ID=2.

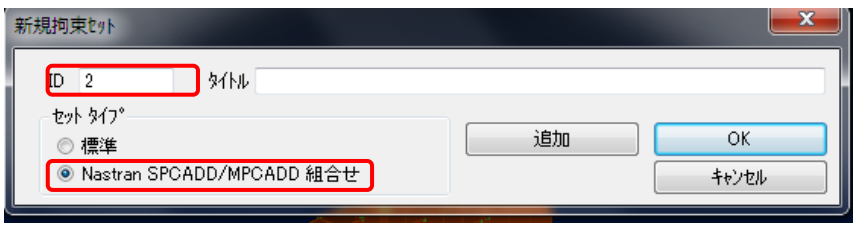
In 'Constraint Set Manager,' go to 'New Constraint Set,' enter ID=20, and click 'OK.' Constraint sets with ID≥20 will be effective only during static analysis.



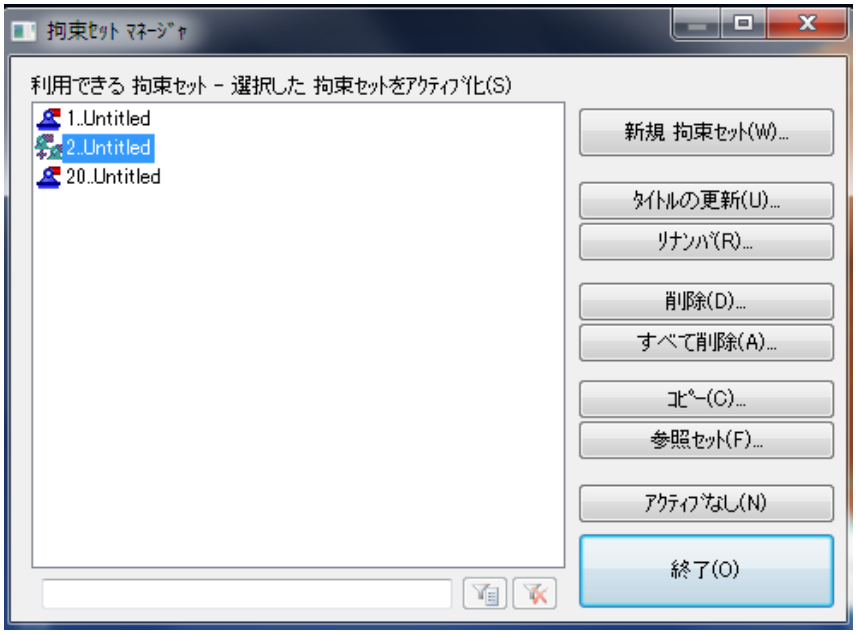
Keeping the constraint set with ID=20 active, click 'Finish.' Now, you can set additional constraints that you want to be effective only during static analysis.



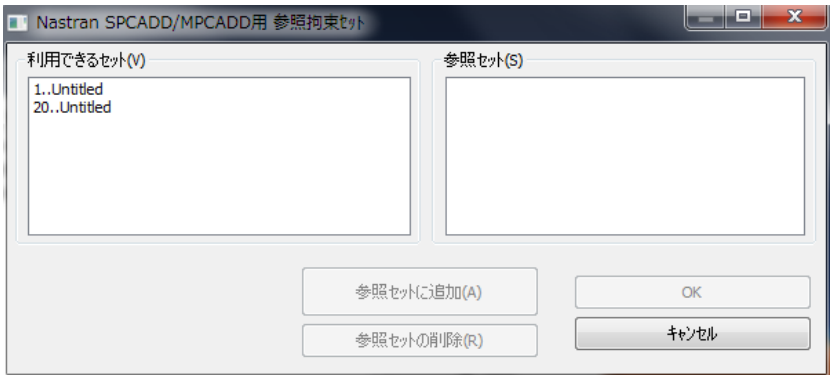
Again, in 'Constraint Set Manager,' go to 'New Constraint Set,' select 'Nastran SPCADD/MPCADD Combination,' enter ID=2, and click 'OK.

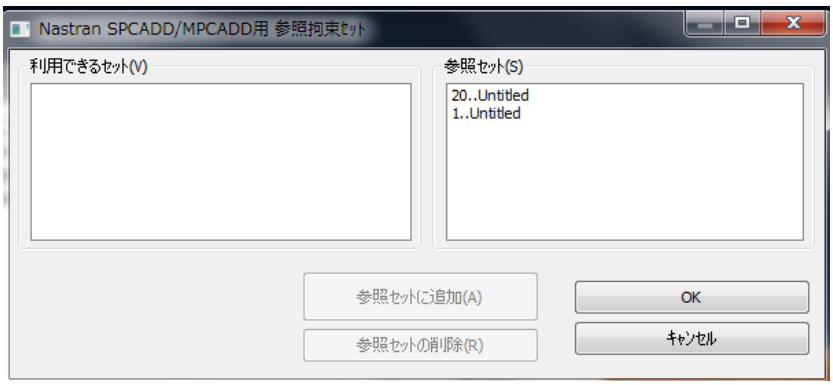


With the constraint set ID=2 active, click the 'Reference Set' button.

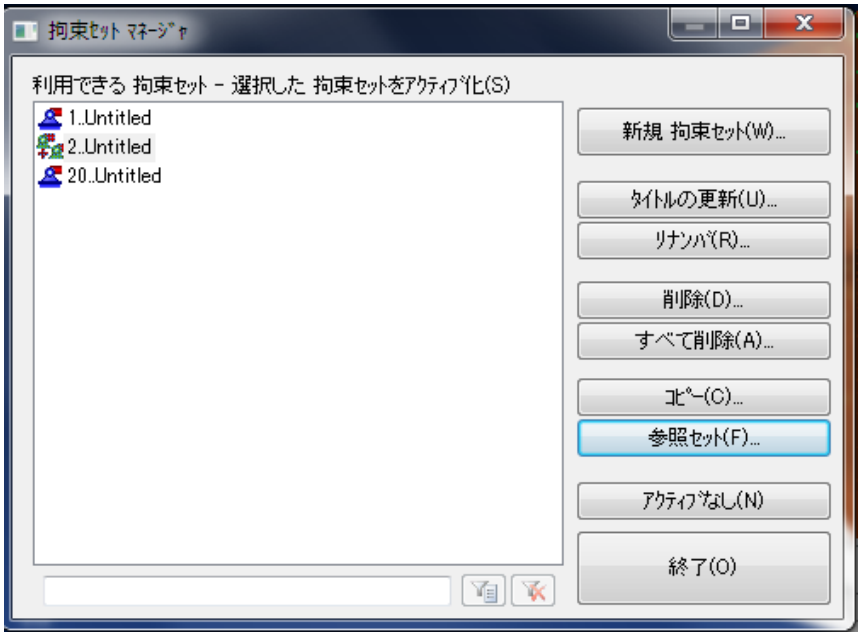


Select 'Available Sets,' click the 'Add to Reference Set' button, move all constraint sets to the 'Reference Set' section, and then click 'OK.

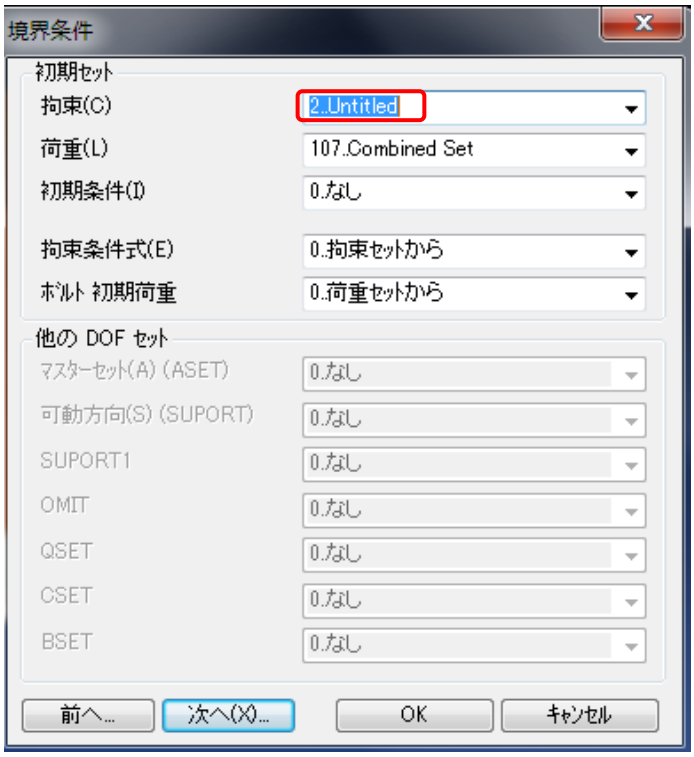




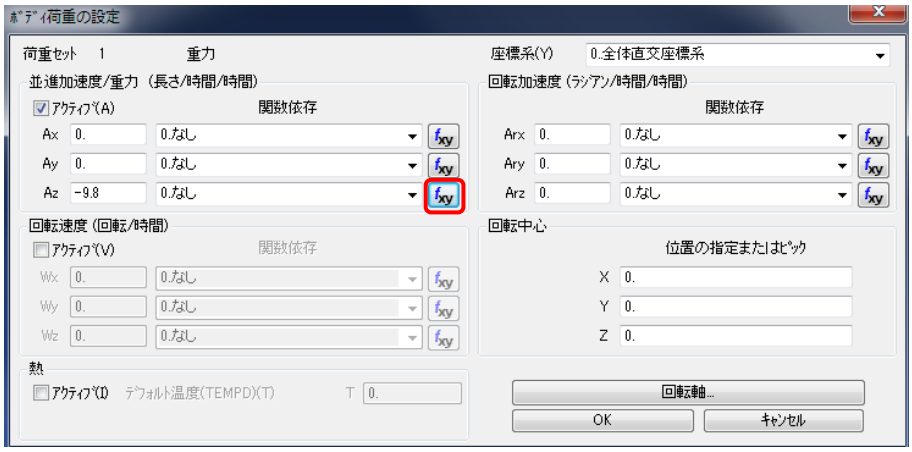
The constraint set ID=2 now contains the combination of constraint sets ID=1 and ID=20.



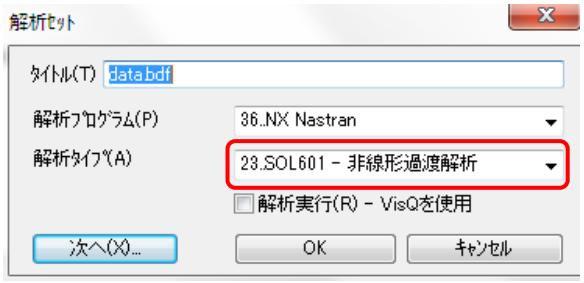
During the analysis execution, in 'Analysis Set Manager' > 'Boundary Conditions,' select the constraint set ID=2 as the boundary condition.



⑨ The time variation of the load is set by clicking on 'fxy' in the load configuration and then defining it in the table.

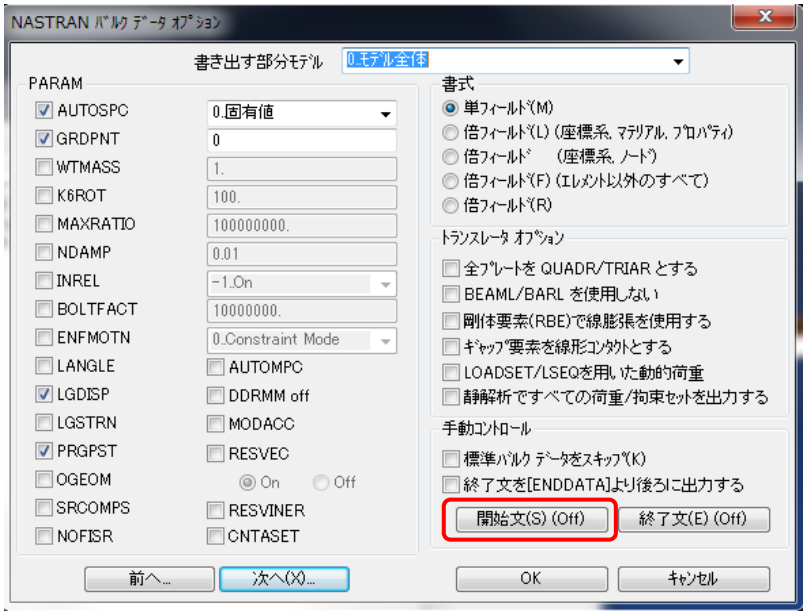


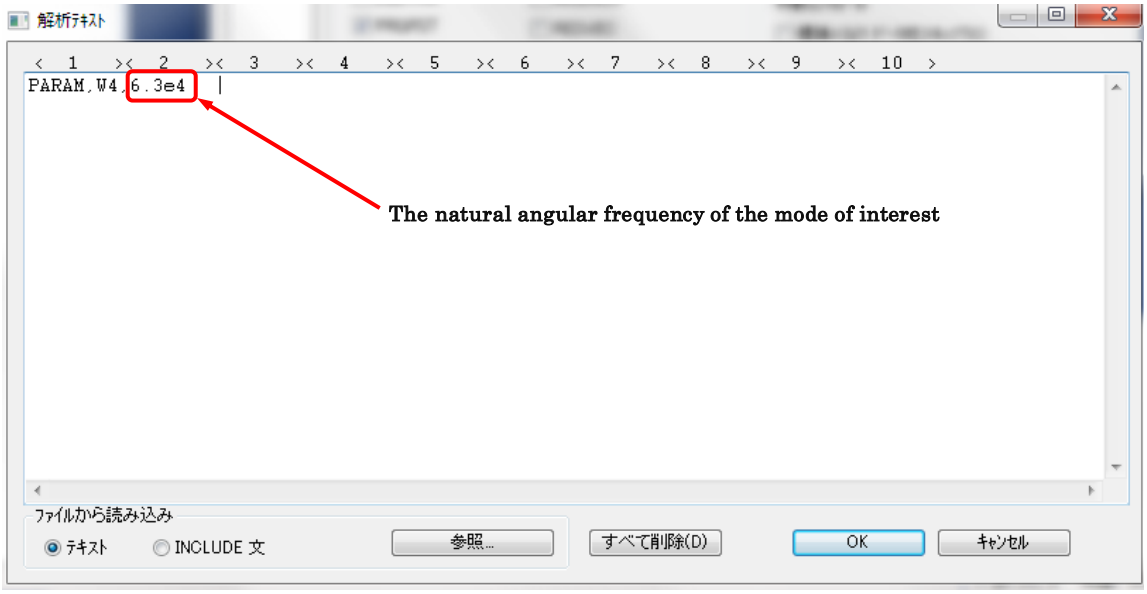
⑩ In 'Analysis Set Manager' > 'Analysis Set,' set the 'Analysis Type' as follows.



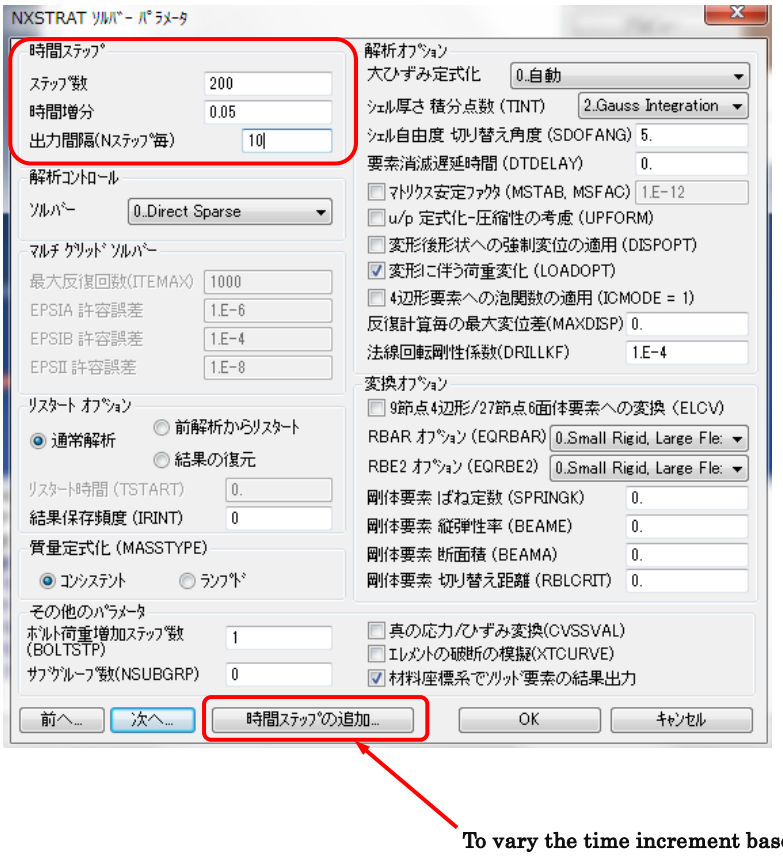
⑪ In cases where 'Structural Damping Ratio' is specified in material data, it is necessary to enter the natural angular frequencies for the modes of interest. Follow the steps below to input

In 'Analysis Set Manager' > 'NASTRAN Bulk Data Options' > 'Manual Control,' click the 'Start Text' button. In the subsequent window, input the text as follows





⑫ In 'Analysis Set Manager' > 'NXSTRAT Solver Parameters,' set the 'Time Step.'



（3）CADMAS analysis execution

Execute CADMAS using input data from data.in, if there is a data.bdf file in the same folder as data.in, CADMAS will automatically recognize the structures in data.bdf as obstacles and conduct the analysis. Simultaneously, CADMAS will output pressure data from the structural surfaces calculated in CADMAS to data.prs.

（4）STR analysis execution

Execute STR analysis using data.bdf as input, if there is a data.prs file in the same folder as data.bdf, STR will incorporate the surface pressure from data.prs as a load condition and conduct structural analysis.

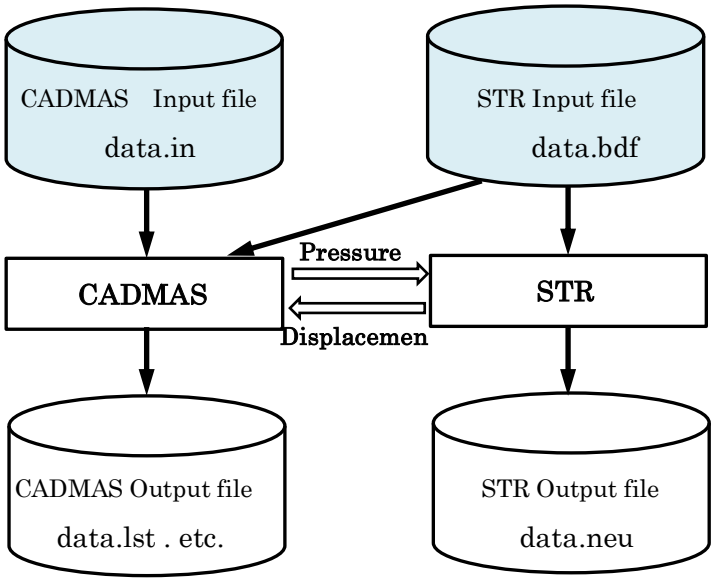
（5）Post-processing

The results of STR analysis can be viewed in FEMAP by opening data.neu. Additionally, the analysis log for STR is output to data.log.

Display target	Display item name	Display content	Data type
Structure & Ground	Displacement	Displacement	Vector
	Velocity	Velocity	Vector
	Acceleration	Acceleration	Vector
	Reactive Force	Reactive Force	Vector
	Stress	Stress	Tensor
	Prin Stress	Prin Stress	Scalar
	Surface Pressure	Surface Pressure	Scalar
Structure	Rod Axial Stress	Rod Axial Stress	Scalar
	Bar Axial Stress	Bar Axial Stress	Scalar
Ground	Pore Water Pressure	Pore Water Pressure	Scalar
	Pore Water Flux	Darcy velocity	Vector

2．Bidirectional coupling

［1］The flowchart of the program is shown below.



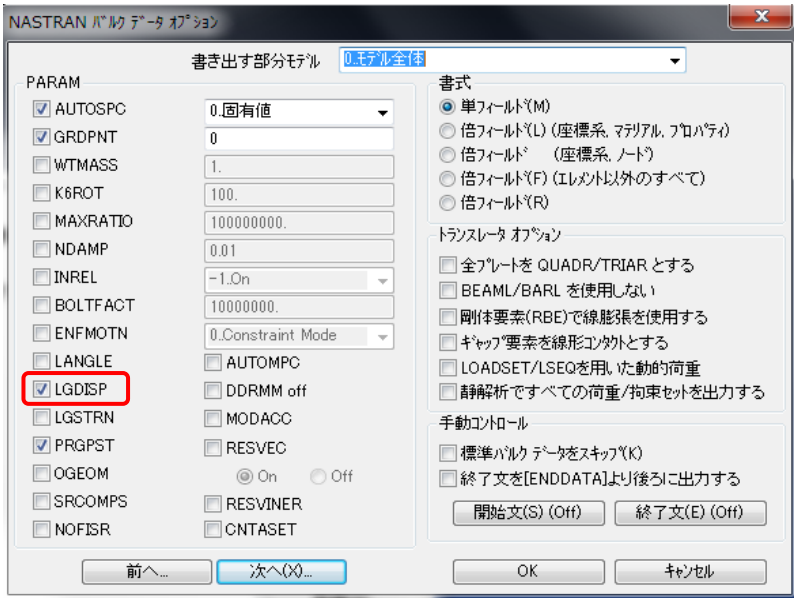
［2］Analysis procedure

（1）Creation of data for CADMAS (data.in)

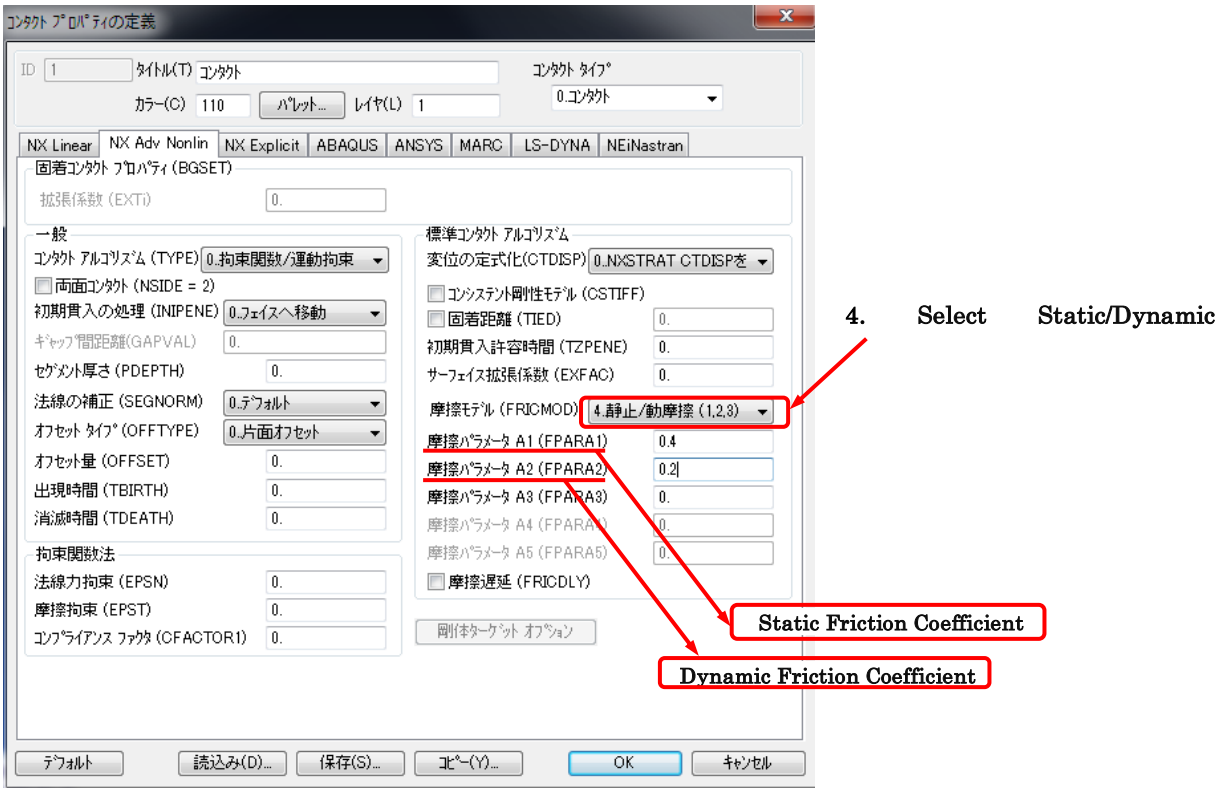
（2）Creation of data for structural analysis (data.bdf)

Same as 1．［2］（2）. The usage of features available for bidirectional coupling is explained below

① In cases where the large deformation feature is utilized, check 'LGDISP' in 'Analysis Set Manager' > 'NASTRAN Bulk Data Options.

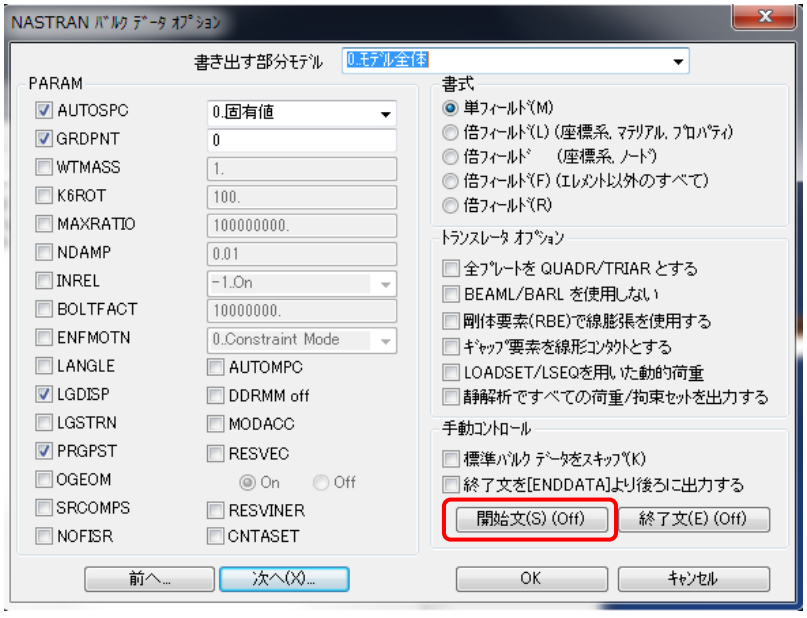


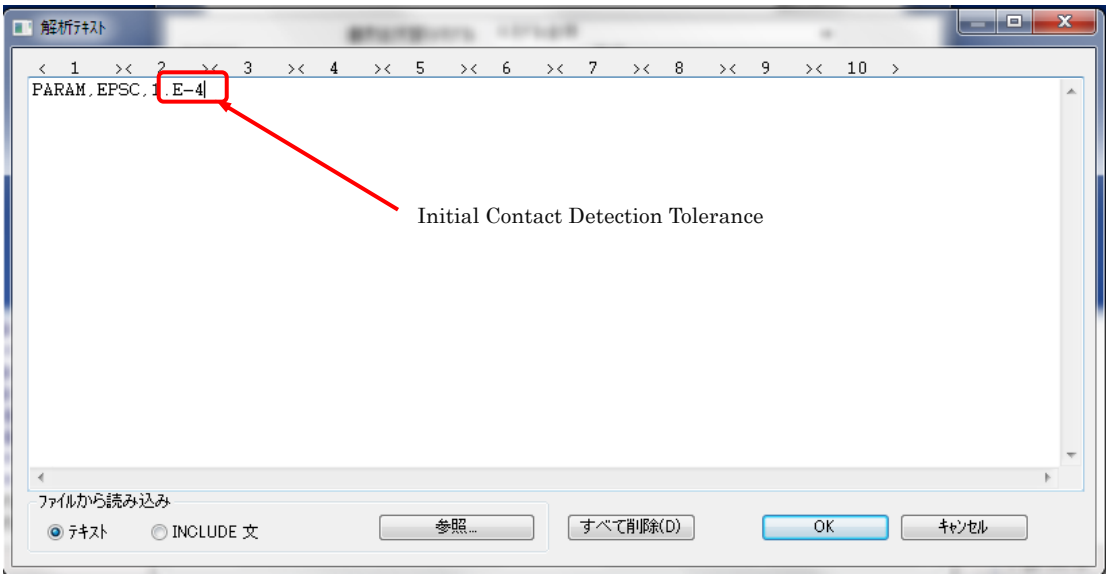
② When using the contact analysis feature, in the 'Contact Property Definition,' select the 'NX Adv Nonlin' tab. In this tab, set the friction coefficient as follows.



② The default value for the tolerance used to determine the initial contact state is set to 10^{-6} .However, it can be modified using the following steps.

In 'Analysis Set Manager' > 'NASTRAN Bulk Data Options' > 'Manual Control,' click the 'Start Text' button. In the subsequent window, input the text as follows.




解析テキスト
< 1 >< 2 >< 3 >< 4 >< 5 >< 6 >< 7 >< 8 >< 9 >< 10 >
PARAM,EPSC,1.E-4
Initial Contact Detection Tolerance
ファイルから読み込み
テキスト
INCLUDE 文
参照...
すべて削除(D)
OK
キャンセル

⑤ The usage method for the restart feature is shouwn below:

（ｉ）The specification for restart file output and the initiation of restart computation is done through input data on the CADMAS side. The restart file name for STR is

The case without domain partitioning：data.rsto

The case of parallel computation with domain partitioning：data.rsto000, data.rsto001, data.rsto002, …

（ii）When performing a restart computation, change the extension of the above file from '.rsto' to '.rsti' and use it. So, change it as follows.

The case without domain partitioning：data.rsti

The case of parallel computation with domain partitioning：data.rsti000, data.rsti001, data.rsti002, …

（iii）The restartable step times are output in data.rtm. For more details, see 'CADMAS-STR Coupled Analysis Section (CADMAS side) Manual.docx' under '2 [4] Restart File Output during Coupled Analysis with STR.

[C A D M A S]		[S　T　R]	
STEP	TIME	STEP	TIME
11	0.220000	5	0.250000
16	0.320000	7	0.350000
17	0.340000	7	0.350000

The step number in data.in under 'FILE RES' specifies the start of the restart computation

The output from data.rtm

（iii）During a restart computation, if you also want to output additional restart files, file（ｉ）may be overwritten. If necessary, it is advisable to keep a backup.

(3) Execution of the analysis

① Execute CADMAS, STR, and STM in parallel using the MPMD method. For example, in a case which CADMAS is run with 6 processes, STR with 4 processes, and STM with 2 processes, the shell script for computation execution is denoted as

ppn=12 ← CADMAS 6 processes＋STR 4 processes＋STM 2 processes
mpiexec.hydra -np 6“CADMAS Executable module name”¥
: -np 4“STR executable module name”
: -np 2“STM executable module name” “STM parameter file name”

② In cases where the CG method is used in STR, append '␣－S␣1' after the 'STR executable module name.' Additionally, when performing parallel computation with the CG method, the root process of STR is used exclusively for communication. Therefore, the number of STR processes should be set as 'number of computation processes + 1.' For example, if CADMAS is run with 6 processes and STR with 4 processes:

ppn=11 ← CADMAS 6 processes＋STR （４＋１） processes
mpiexec.hydra -np 6“CADMAS executable module name”¥
: -np 5“STR executable module name”-S 1

③ By appending ' ␣－M␣ ' (memory amount) after the 'STR executable module name,' you can specify the available memory amount (memory unit is MB). If the program's memory usage exceeds the specified amount, it will output an error message ('insufficient memory') and stop. The default available memory is set to 8000MB. For example, if you want to specify 16000MB as the available memory
“STR executable module name”-M 16000

(4) Post-processing

Same as １．［２］（５）.