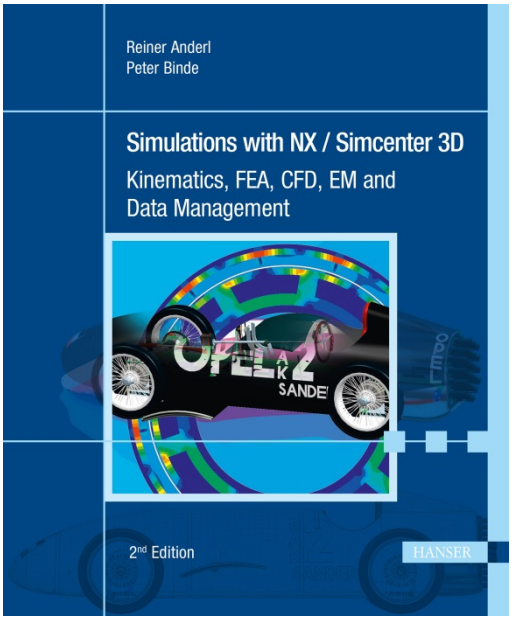
HANSER



Reading sample

to

Simulations with NX / Simcenter 3D

from Reiner Anderl,
Peter Binde
2nd Edition

ISBN (Book): 978-1-56990-712-2
ISBN (eBook): 978-1-56990-713-9

For further information and order see
http://www.hanser-fachbuch.de/978-1-56990-712-2
or contact your bookseller

Contents

Preface

Virtual product development has gained significant importance in particular through the integration of 3D solid based modeling, analysis, and simulation. Supported by the rapid enhancement of modern information and communication technology application, integrated virtual product development has become an essential contribution in higher engineering education, continuing education as well as in industrial advanced and on-the-job training. Furthermore, the very important and new approaches from the Industry 4.0 initiative lead to a new level of added value and affect all areas of the product lifecycle. In this context, the creation and use of so-called digital twins plays a groundbreaking role. Digital twins are digital representations of individual, real components and products. They form the basis for a wide range of analysis and simulation and, in particular, allow the examination of real components and products, independently of their location. Preventive maintenance, servicing and pre-testable repairs are thus preparing innovative approaches for new business models.

Since 2003, Technische Universität Darmstadt has been selected and approved as PACE university and has become part of the international PACE network. PACE stands for *Partners for the Advancement of Collaborative Engineering Education* and is a sponsoring program initiated by General Motors. PACE is driven by General Motors, Autodesk, HP (Hewlett Packard), Siemens, Oracle, and further well acknowledged companies of the virtual product development branch (*www.pacepartners.org*). Donations and sponsoring through the PACE partner companies has facilitated the preparation and the publishing of this book.

This publication has been developed based on cooperation between Dr. Binde Ingenieure – Design & Engineering GmbH (*www.drbinde.de*) and the division of Computer Integrated Design within the department of Mechanical Engineering of Technische Universität Darmstadt (*www.dik.maschinenbau.tu-darmstadt.de*).

We would like to thank the employees of Dr. Binde Ingenieure and those from the division of Computer Integrated Design of Technische Universität Darmstadt for their active participation in the development of this new edition.

Furthermore, we are grateful for the support of Carl Hanser Verlag, mainly Mrs. Julia Stepp. A very special thank you is dedicated to Prof. Dr. Jan Helge Bøhn who supported us through his excellent cross-reading. Last but not least, we thank all readers who encouraged us to prepare this book in English.

We wish all readers and users a successful application of the selected examples and hopefully a beneficial knowledge acquisition usable for their successful graduation or the successful knowledge application during the industrial career.

June 2018

Prof. Dr.-Ing. Reiner Anderl

Dr.-Ing. Peter Binde

Preamble

Analysis and simulation are essential prerequisites for the digitalization of products and processes. The complete digitalization of value streams in all industries drives the acceleration of processes and enables new business models. Digitalization starts with the idea of a new product and accompanies the product along its entire lifecycle. Digital data and digital data models are essential to the creation of digital twins. This data management concept encompasses all physical and virtual data of a product throughout its lifecycle.

The digital twin enables analyses and simulations of physically existing products as well as products only existing in the digital world. Analysis and simulation supply digital models that reflect functionality and behavior of the physical product. Furthermore, digital models can be "interrogated" to gain information and knowledge about the physical product even before it exists. Once the product is produced, physical tests will follow to perform the same interrogation for gaining information and knowledge now in the physical world. Comparing both approaches – the one from the digital model and the one from the physical product – enables to elaborate quality indicators for the digital model used. Digital twins are becoming more and more advanced to reliably analyze and simulate reality and – even more important – to predict its future behavior.

Siemens PLM Software's clear objective is to support all customers in the digitalization of their value streams in order to strengthen their competitiveness. To do this even better, Siemens PLM Software expands its portfolio continuously. This is accomplished by in-house development projects or through acquisitions of technologies or entire companies.

In this context, the Siemens family was expanded by LMS in 2012 and CD-adapco in 2016. To better represent the expanded range of offerings, Siemens created Simcenter™, an umbrella brand that represents an unmatched portfolio of simulation and test solutions. Simcenter includes geometry-based simulation, system simulation, physical testing, and the accompanying engineering services.

The geometry-based simulation solution – formerly known as NX CAE – is now called **Simcenter 3D**. Simcenter 3D incorporates NX CAE and NX Nastran as well as new capabilities from LMS Virtual.Lab and LMS Samtech. This book focuses on Simcenter 3D, which is available both as a standalone CAE solution as well as a solution completely integrated with NX.

Starting with NX10, some new functionalities like the environment for the LMS Samcef solver were made available. In summer 2016, more functionalities were added to NX11 and Simcenter 3D 11: acoustics, motion, and multi-body simulation and the dynamic simulation of flexible hoses and pipes. In October 2017, even more functionalities from LMS Virtual.Lab and LMS Samcef were added to NX12 and Simcenter 3D 12. And Siemens PLM Software is committed to further enhancing this multi-discipline simulation platform.

What does that mean for you as a user of NX CAE?

You will keep your user interface as well as your NX data model and the optional Teamcenter integration. As an NX CAE user, you will see that Simcenter 3D looks exactly the same and is operated as before. Simcenter 3D is based on the NX platform and adds additional functionalities from LMS Virtual.Lab and LMS Samtech to the overall offering.

To reflect this, the book has been titled *Simulations with NX/Simcenter 3D*.

In their successfully proven and well tested manner, Prof. Dr.-Ing. Reiner Anderl und Dr.-Ing. Peter Binde provide vivid and easy to follow examples of simulation use cases.

To the authors: Thank you very much indeed for this comprehensive masterpiece of education, tutorials and reference.

To all readers and friends of NX CAE: I wish you a lot of success and fun using Simcenter 3D.

Eckardt Niederauer

Portfolio Manager Simcenter 3D

Siemens PLM Software

www.siemens.com/plm/simcenter

An analysis with linear material behavior should be made always prior to such with a non-linear behavior. (This can now very quickly be made up by inserting a new solution 101 and adding of the boundary conditions.) Here, slightly smaller displacements of 1.69 mm and significantly higher stresses with about 200 N/mm^2 arise. Thus, elastic material shows higher stresses, while for plastic material already local yielding occurs. Due to slight yielding, in turn, higher displacements result for plastic material.

It should be pointed out that mesh refinement is required and that subsequently, in order to verify the results, convergence has to be proven.

- SAVE the file and close it.

Therefore, the learning task is completed.

4.4 Learning Tasks Advanced Non-linear (Sol 601)

4.4.1 Snap Hook with Contact and Large Deformation

Snap hooks are popular types of closures for plastic components that are assembled manually (Figure 4.120). In the design you want to know, for example, how much force is required to assemble such a closure. Moreover, the material in the process must not be too highly stressed. On the model of the RAK2 such a link can be found at the battery box.

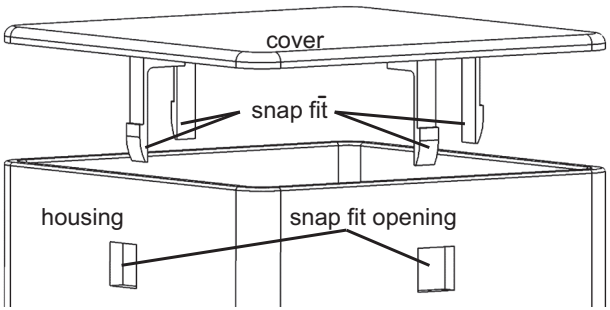


Figure 4.120 Snap hooks are popular types of closures for plastic components.

The Nastran solution method 601 is intended for complex non-linear analyses.

In this example, the basic handling of the NX Nastran solution method 601 will be described which is available for complex non-linear effects based on a plastic snap closure. A time-dependent travel path will be defined, which controls the mounting operation of the cover. In this way it is possible to determine the force which is required for the assembly process. In addition, non-linear effects of large deformations and contacts are used. Recommendations are made, how to deal with complex non-linear effects in Simcenter 3D.

First, in order to develop a "feel", a linear analysis should always be done.

We want to point out that before any non-linear analysis, simple linear analysis of the problem should always be performed. If several types of non-linearities occur simultaneously (e. g. plastic material and contact), non-linear effects should first be tested individually. In our learning example, we are in violation of this rule, thus we work directly with non-linearities because the limited scope of the book forces us to.

4.4.1.1 Example

The battery box is to be mounted.

The battery box of the RAK2 is mounted with plastic snap closures as shown in Figure 4.120. Here the cover at all four sides has a hook that latches into a corresponding opening of the housing.

Given are material properties of the plastic material and the geometry. It is to be analyzed, which stresses the snap hook is exposed to when the installation is done in small steps.

4.4.1.2 Preparation and Creation of the Solution

- Load the assembly *as_bg_batterie* that contains the housing, cover and some other parts.
- Then switch to the application *Pre/Post* and generate the file structure for the new FEM and simulation using the Simulation Navigator.
- When asked for the bodies to be used, select the option *Select Bodies* and select the geometry of the cover and the housing.
- Now the menu *Solution* will appear. Here you select the solution type *SOL 601,106 Advanced Nonlinear Statics* (Figure 4.121).

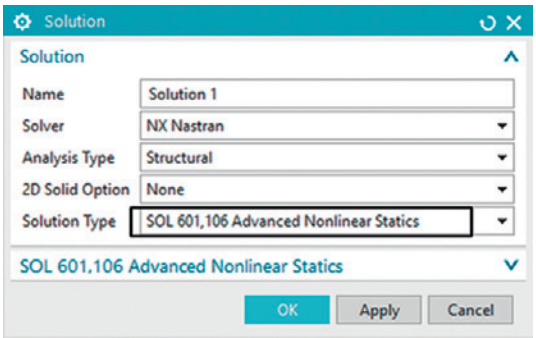


Figure 4.121
Dialog for Nastran Solution 601,106

The solution method 601 has a wide range of application fields.

First, all settings remain at their default.

- Confirm all settings with OK.

4.4.1.3 Change Start Position in the Idealized Part

The starting position should be the opened battery box.

During the simulation the cover shall switch from the open state into the closed state. However, the cover is constructed in the closed state. This closed position in the master CAD assembly should not be changed, because this is required for users who look at a 2D

drawing of the assembly, for example. The CAD assembly is the master model to be used for all subsequent applications such as analysis, preparation of drawings or fabrication support. If different positions of parts to each other are needed for our simulation, repositioning must be performed in the idealized file.

The position of the cover shall be changed for the analysis.

Therefore, the original assembly position of the cover has to be overridden in the idealized file. From the view of the idealized file there will be another position than from the view of the real assembly. Proceed as follows to achieve this:

- First, set the idealized file as displayed part.
- Switch into the application *Modeling* and make sure the application *Assemblies* is enabled. Now you have access to assembly capabilities of Simcenter 3D.
- Now open the Assembly Navigator and select the cover *as_bat_deckel*.
- Select the function *Move* by right click.

Now the cover can be moved to the desired position, as shown in Figure 4.122. The snap hook should ideally be positioned just before the point of contact. For this, you can also use the dimensions of Figure 4.122.

- Apply for Z the value from Figure 4.122, i. e. −16.25, and press ENTER.

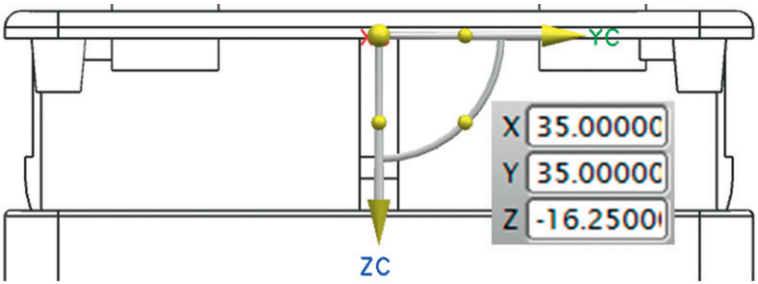


Figure 4.122 The cover is now repositioned for the FEM analysis.

- Confirm with OK. The repositioning, solely for purposes of simulation, is now done successfully.

If you want, you can now change back into the application *Pre/Post* and check the old position, by making the master part in the Simulation Navigator the displayed part. Then go back into the idealized file. The new position is now active again.

4.4.1.4 Simplification and Partition of Geometry

The geometry of the cover and the housing should then be idealized, hence the number of finite elements can be kept low on the one hand, while on the other hand the remaining geometry is not substantially different from the stiffness properties of the original geometry. Therefore, the symmetry property should be exploited.

Especially for non-linear analyses, elements should be “saved”.

The requirement for a small number of finite elements is even more urgent, the more it comes to non-linear effects that must be taken into account, because growth of computing time through non-linearities is tremendously. You will see that even this example needs

significantly more processing time than all linear tasks so far. It is therefore necessary to find a sensible compromise.

In addition, it would be advantageous if the idealized geometry could be meshed by hexahedral elements instead of tetrahedral, because a greater uniformity and better accuracy can be achieved with these elements. Meshing with hexahedral elements is always possible when a solid is extrudable, i. e., if it has a surface which can be filled with squares drawn through the whole body. In this way, hexahedral or brick elements are created.

The geometry of our example can be simplified and partitioned in a manner shown in Figure 4.123. The stiffness of the original geometry would remain largely intact, and the ability for HEX solid meshing would exist for both parts.

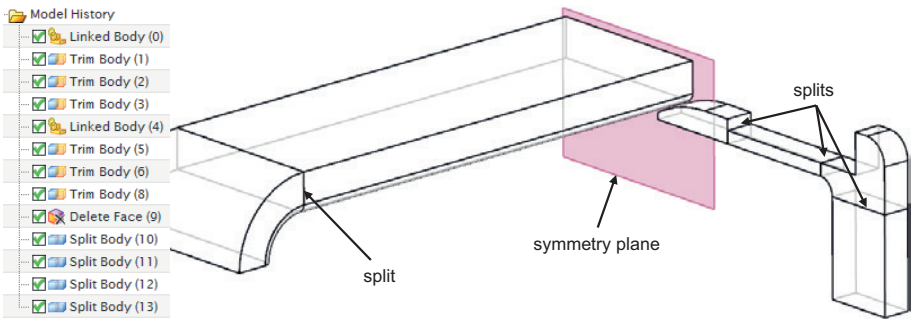


Figure 4.123 The battery box and snap hook are simplified for the FEM analysis. A few CAD operations are performed.

- Create wave links and perform geometry operations in the idealized file to obtain a corresponding geometry.
- Save the file.

4.4.1.5 Mesh Mating Conditions

Mesh mating conditions of the type *Glue Coincident* ensure that partitioned areas of the body will be meshed with aligned nodes. It is recommended to turn on the *Create Mesh Mating Conditions* option already in the idealized file when creating the partitions with the split body function. In that case, these conditions would be automatically generated in the FEM file.

The partitioned geometry shall have identical and merged nodes at the boundaries.

Otherwise, proceed as follows:

- Change to the FEM file.
- Call the function *Mesh Mating Condition* .
- Ensure that the option *Glue Coincident* is active.
- In the graphics area, drag a window over all parts and confirm with OK.

The *Mesh Mating Conditions* are now created on all four partitions.

4.4.1.6 HEX Solid Meshing of the Housing

This time stresses in the contact area itself are not of particular interest for us. The geometry therefore might only be roughly meshed here. However, experience has shown, that difficulties with sharp edge contact surfaces may arise, as they appear, for example, if fillets are meshed very coarsely. Therefore, the fillet at which contact will first occur and slide, should be meshed finer. This way contact surfaces should become as smooth as possible.

- Create a *Mesh Control* of the type *Number on Edge* on the fillet edge in the contact area and apply a number of four elements there.

The fillet at which the first contact occurs should be meshed fine.

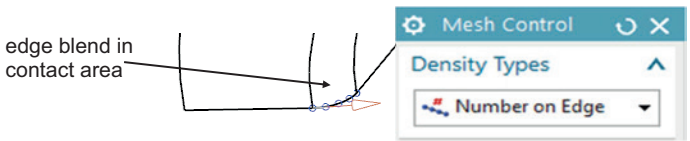


Figure 4.124 A local refinement is specified on an edge

- For the subsequent meshing call the function *3D Swept Mesh*, set the type to *Until Target* and select the two surfaces shown for the *Source* and *Target* (Figure 4.125).

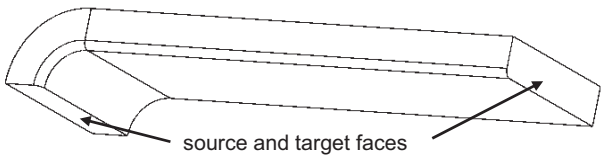


Figure 4.125 Preliminary thoughts considering hex meshing

- Select *Hex8* elements and element size as the default value that you will receive with the yellow flash. Click OK to confirm. Then the mesh is generated.

The mesh should look like Figure 4.126.

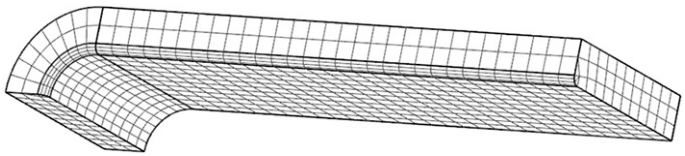


Figure 4.126 The hexahedral mesh is very even. The contact area mesh is refined.

4.4.1.7 HEX Solid Meshing of the Snap Hook

The body for which stress results are substantially of interest is the snap hook. Furthermore, stresses in the bending area are especially interesting. Therefore, a finer mesh shall be generated here. In the remaining area of the snap hook the mesh can be coarser.

In the following, a suggestion how the snap hooks could be meshed is described, but certainly there are other reasonable ways.

- For meshing the bending region of the snap hook, call the function *3D Swept Mesh*, set the type to *Multibody Infer Target* and select the face as shown in Figure 4.127.

Using the option *Multibody Infer Target* you only need to select a starting face. The other option additionally requires a target area selection. But this second option has another advantage: It can pass the mesh through several bodies when they are connected by *Mesh Mating Conditions*.

- As element type, select *Hex8* and as element size e. g. choose half of the suggested value.
- At *Use Layer* enter the desired 5. Now click on OK.

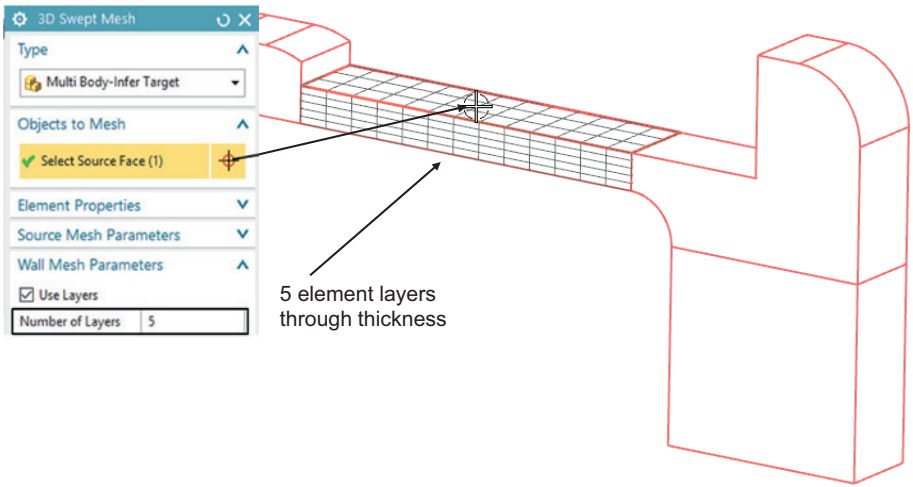


Figure 4.127 This area will get five element layers across the thickness.

This method allows you to mesh fine across the thickness and coarser in depth.

- Next, you will create, according to Figure 4.128, a mesh control having five elements over the fillet edge in order to also fine mesh that risky area.

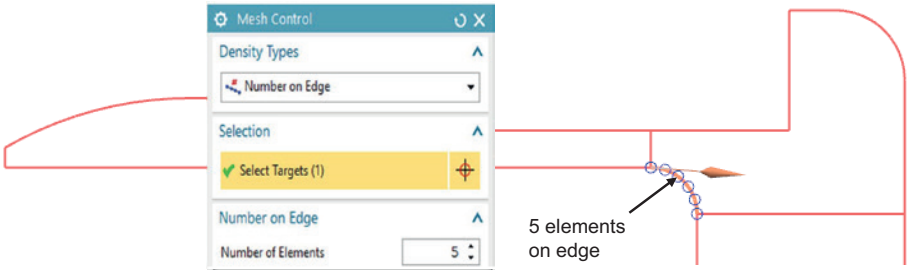


Figure 4.128 Definition of a mesh control on an edge

In order that the refined mesh gradually transitions to rough again, create additional mesh controls on the four edges, according to Figure 4.129. This time you use the type *Biasing on Edge*, which makes it possible to define such transitions. Set the options as shown in Figure 4.129.

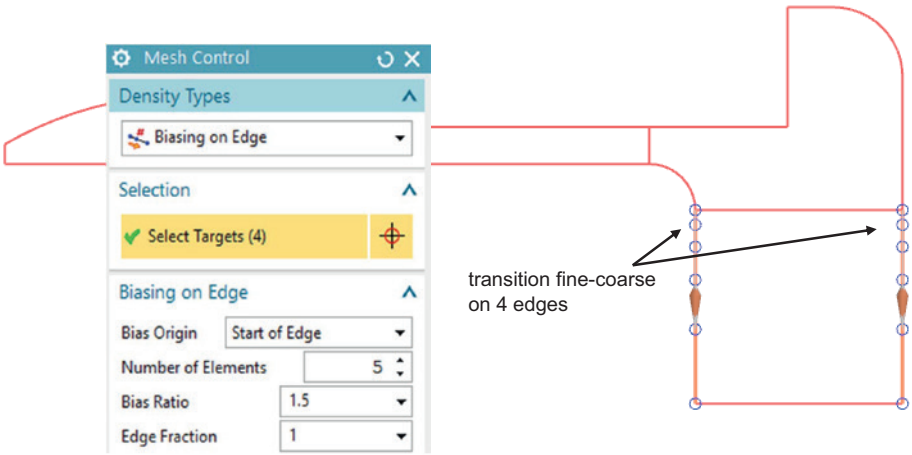


Figure 4.129 With several mesh controls on edges it is defined how the mesh should behave here.

Now, the remaining parts can be meshed. Use element sizes, as shown in Figure 4.130.

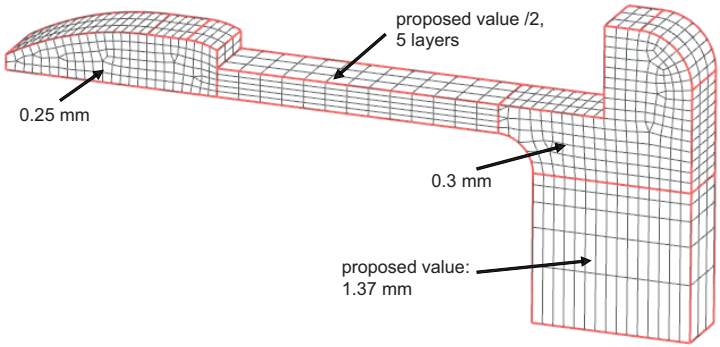


Figure 4.130 A successful HEX solid meshing of the snap hook

4.4.1.8 Add Preparation for Reaction Forces

We want to calculate the force, which is used to assemble the snap hook. This force, which will arise as a reaction force in the displacement constraint condition, can be represented at any time step and displayed as graph. For simple evaluation of this force, we create a point-face connection on the surface of the later applied displacement boundary condition:

- Select the function *1D Connection*.
- Set the element to *RBE2* and type to *Point to Face*.

Later we want to specify the enforced motion on this point.

- Use the function *Point Dialog* to define this point at the coordinates (x: 35, y: 10, z: −18.5) and select the face corresponding to Figure 4.131.
- Click OK to confirm and the connection will be created.

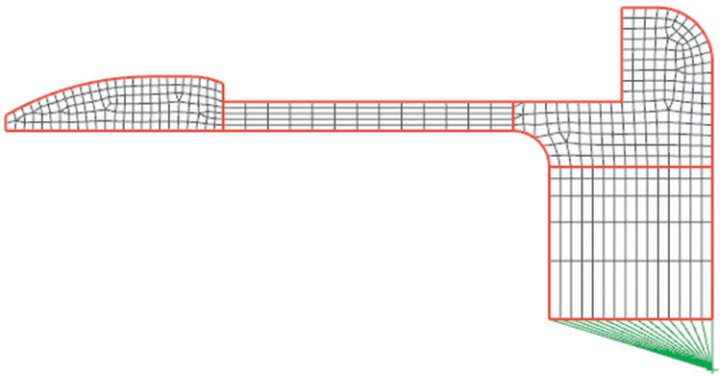


Figure 4.131 View of the complete mesh

4.4.1.9 Material Properties for Plastics

The material of the two bodies should be glass fiber reinforced polypropylene. This material can be found as *Polypropylene-GF* in the material library of Simcenter 3D in the category *Plastics*. Alternatively, you can define the material manually. The following properties are required:

- Modulus of elasticity: 3000 N/mm^2
- Poisson's ratio: 0.4

- Assign the material *Polypropylen GF* to the bodies.

This library's material is a simple linear description of the material.

Recommendations for the analysis of plastic materials

For dealing with plastics in FEM analysis, we give the following recommendations, which can be found more detailed in [Alber-Laukant] and [RiegHackenschmidt]: Indeed, plastic components in principle show a non-linear stress-strain behavior. However, there almost always exists a linear region of the material properties. Even exceeding this area, the analysis with constant modulus of elasticity and Poisson's ratio can be done until the error caused thereby in the application can no longer be tolerated. The procedure for analysis of plastics should therefore be done as follows: On the basis of resulting stresses of a first rough analysis with linear material properties, it will be decided, if the stresses are still in the linear or already in the non-linear region. Only in the second case it is contemplated whether the error is tolerated or analysis is reperformed with a non-linear stress-strain curve (Figure 4.132).

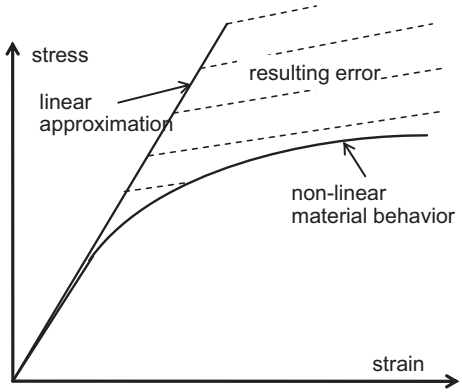


Figure 4.132
If plastic materials are calculated linearized, the resulting error must be estimated.

When the non-linear stress-strain curve shall be taken into account ...

Such a material non-linearity can be added easily in the Nastran solution 601. The material would have to be defined as it was in the example with the plastic deformation of the brake pedal.

Only the setting *Type of Non-Linearity* should be set to *NLELAST*. This abbreviation stands for *non-linear elastic*, meaning that the behavior is elastic, i. e. reversible. Therefore, if an unloading of material occurs, it follows the same curve back down. Thus, there will be no permanent deformation. This model is often used in plastics, when it is required to calculate non-linear material behavior.

4.4.1.10 Define Contact

- For the contact surface definitions and other boundary conditions you set the simulation file as displayed part.

The two sides of contact should not be chosen arbitrarily.

For contact definition there is the recommendation to first select the finer meshed surface area. This way it will be set as the *Source* region. The coarser meshed area should be selected secondly, so it is set as *Target* region. The contact algorithm will ensure that nodes of the first region may not penetrate the element faces of the second region.

In our example, it cannot be answered clearly which side is finer, since the mesh sizes of our contact partners are varying. If in doubt, also a computationally more expensive two-sided contact could be chosen, which can be activated in the contact parameters.

- Call the function *Surface to Surface Contact* (Figure 4.133).
- Select the type *Manual*.

Upon contact, the finer meshed side should be selected first.

- As *Source Region* select the fillet face of the housing and the lower planar surface, since here the mesh is finer.
- As *Target Region* – in accordance with Figure 4.133 – select the three tangential faces of the snap hook, where contact can occur.

The analysis becomes more complex due to specification of contact friction.

- Enter 0.1 for *Coefficient of static Friction*.

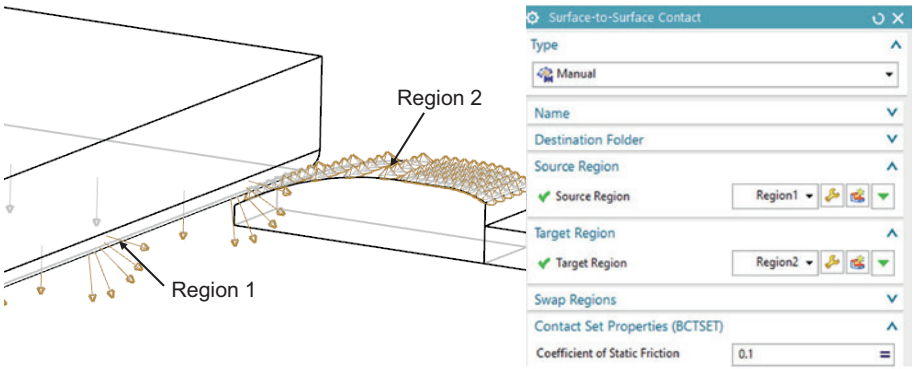


Figure 4.133 Definition of a contact

Leave all other menu settings as initially preset.

- Click OK to confirm. Then the contact is generated.

4.4.1.11 General Information about Solution 601

The analysis is performed in many small steps.

In solution 601 all boundary conditions should be applied time-dependent. The time will then be stepped gradually on the basis of given time steps. Indeed in case of solution 601,106 no dynamic effects are taken into account. That means it does not matter if we use one or ten seconds as the period of simulation time. However, if you use the solution 601,129 or even 701, dynamic effects are calculated from time steps and defined movements. Then the simulated time period must be set necessarily realistic.

In most applications of solution 601, it is even recommended to define time-dependent forces or enforced displacements, because in this way smooth transitions can be achieved from one time step to the next, which in turn improve the convergence behavior of the solution.

For this reason, a time-dependent displacement, which moves the snap hook from its initial position to the closed position, will be defined in this example.

4.4.1.12 Define Time Steps

The smaller the time steps, the easier the solution converges.

Since in our chosen solution 601,106 no dynamics are taken into account, the size of the time period is not an important indication. Only boundary conditions must be adjusted accordingly, because they are defined time-dependent. But the number or the size of time steps are important variables, because the smaller time steps are, the easier each step can converge.

- From the context menu of the solution, select the EDIT function, and in the following menu select the tab *Case Control* (Figure 4.134).
- Now you choose among *Time Step Intervals* the function *Create Time Step Intervals.*
- A dialog appears for the definition of time step intervals. For us it will be sufficient to define only one interval. Now select *Create*.

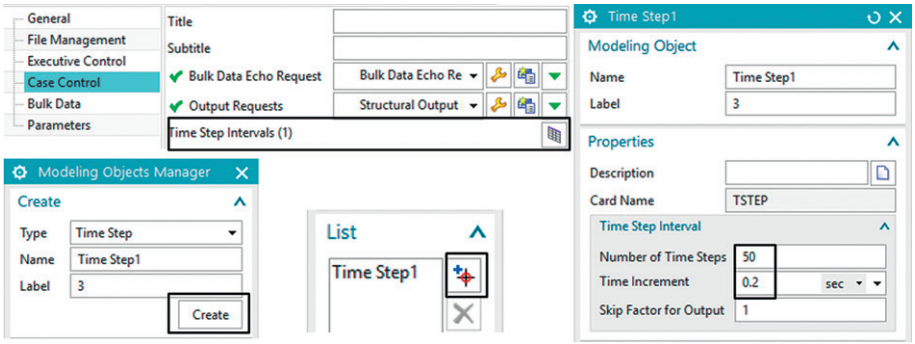


Figure 4.134 Dialogs for definition of time steps

You realize that the *Number of Time Steps* defaults to 10 and the *Time Increment* is 1 sec. This results in a simulation period of ten seconds. Therefore, the traverse path has to be defined for this period in the following. We want to use those 10 sec, but have a finer subdivision of the period, so we change this element as follows:

- Change the number of steps to 50 and the increment to 0.2 sec (Figure 4.134).
- Click OK to confirm and add the interval with ADD to the list. Now click CLOSE and OK.

4.4.1.13 Definition of a Time-Dependent Travel Path

The installation of the snap closure is divided into time increments.

Traverse paths correspond to enforced displacement boundary conditions. These can be defined in Simcenter 3D either by an *Enforced Displacement Constraint* or through the function *User Defined Constraint*.

- Call the function *Enforced Displacement Constraint* (Figure 4.135).

Among *Type*, there are various options for defining the displacement direction. It is best to use the option *Components*, in which all degrees of freedom can be set separately in coordinate system directions. In our case, the Z-direction is critical for the forced way. All other components will be kept fixed.

- Set the *Type* to *Components* and select the point, which is connected to the surface via *RBE2* elements.
- Enter zero in all degrees of freedom, except for the *DOF3*, here the displacement in Z-direction shall take place.

Both the translational as well as rotational degrees of freedom have significance in this case because we set the constraint for a point that is coupled to a face.

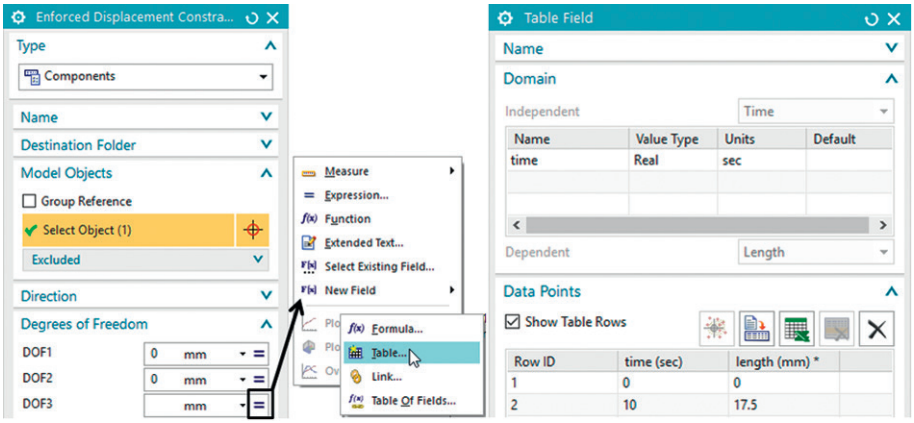


Figure 4.135 Dialogs for defining an enforced displacement

The path is defined by a table.

- For the definition of the Z-direction displacement field, it is easiest to use a table. So at *Specify Field* you switch into the *Table Constructor* and create a field for the desired displacement.
- In the menu *Table Field* first the independent variable, i.e. time must be indicated. Therefore, choose the option *Time* in the box *Domain* at *Independent*.

The dependent variable is already defined as *Length* with units of mm.

The independent variable is time, the dependent one is displacement.

Now single xy pairs of values of the desired function can be specified. If there are only few pairs of values like in our case, it is advisable to enter them manually. We want to create a linear function that is 0 mm at time 0 sec and 17.5 mm at time 10 sec (Figure 4.135).

- In the box below the table preview enter the pair of values “0 0”, followed by ENTER.
- Also enter the second value pair “10 17.5” and hit ENTER.

The traverse path can be entered in form of value pairs.

The value pairs are added to the table. If desired, the defined field can be represented as a graph (Figure 4.136).

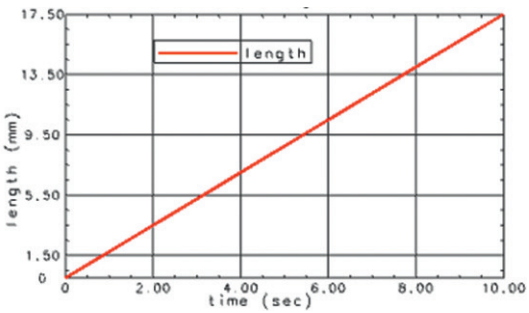


Figure 4.136
The newly created table is assigned to the boundary condition.

- Confirm with OK and you’ll be guided to the previous menu.

So you finally arrive back in the menu for defining the boundary condition. In the field for *DOF3* the new, time-dependent field is now listed.

- Finally confirm with OK. The time-dependent displacement constraint is now created.

4.4.1.14 Defining Further Boundary Conditions

In addition to the time-varying displacement, there are only a fixing of the housing and a symmetry condition necessary.

- Create a fixed constraint on the face of the housing, as shown in Figure 4.137.
- Generate symmetry conditions on the faces belonging to the plane of symmetry.

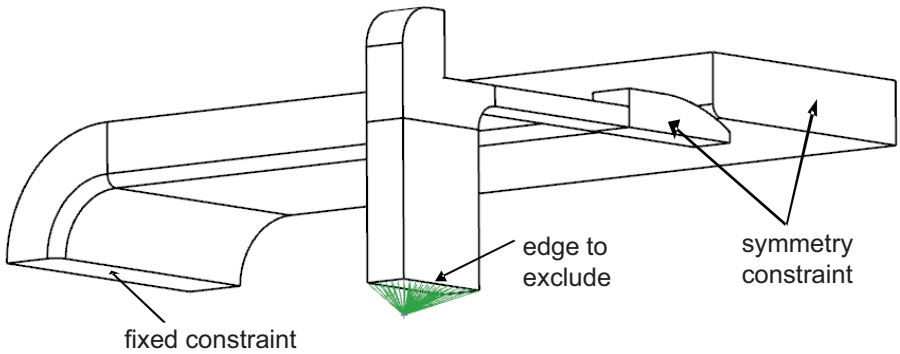


Figure 4.137 The housing is constrained fix.

There is a boundary condition conflict because on one edge two conditions exist: the symmetry condition and the RBE2 elements. If we try to solve the model this way, an error message pointing to this conflict would appear. This conflict cannot be resolved automatically. Instead, we have to exclude this edge from one of the two conditions. The easiest way is as follows:

- Right-click on the *symmetry condition* you just created and from its context menu choose EDIT.
- Click on the text *Excluded*. It opens another selection field.
- In this selection field, select the edge marked in Figure 4.137 that we want to exclude from this condition. Now click OK.

4.4.1.15 Activate Option for Large Displacements

For this example, the solution should be performed taking large displacement effects into account. This option must be activated as follows:

Large displacements must be enabled when the stiffness changes during solve time.

- Right-click on the solution node in the Simulation Navigator and select the EDIT function.
- Switch to the *Parameters* tab and activate the switch *Large Displacements* (Figure 4.138).

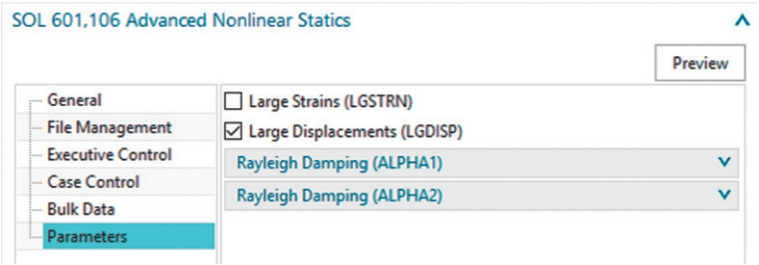


Figure 4.138 In solution 601 the option for non-linear geometry or large displacements is activated.

4.4.1.16 Attempt to Solve without Automatic Time Stepping

Without time stepping, contacts mostly cannot be calculated.

By default, none of the available automatic time stepping schemes is enabled. We will first attempt a solution with all the default settings, i. e. without time step method. We will try and interpret the results. The next section then follows with the use of the main automatic time stepping method *ATS* (*Auto Time Stepping*).

- Perform *Solve* .
- Do not cancel the solution monitor!
- You will find that a result has been generated, in which only a few time steps have been computed.

In the first steps that were apparently successful, the snap hook is simply moved a little. Then no more steps have been calculated. To understand this and to find help, the analysis method of the non-linear solver will be described below.

4.4.1.17 Understanding Newton's Method

To understand the solution course, it needs to be explained how the non-linear solution method called Newton's method works. The bent curve in Figure 4.139 represents a real non-linear force-displacement behavior of a model, which is caused for example through material plasticity. It is the task of the calculation to place a force on the model and to calculate the deformation along this curve.

Time steps pass through the external loads or boundary conditions specified by the user.

Newton's method uses an outer and an inner iteration loop. We call the outer loop time steps, which is marked with Δt in Figure 4.139, and the inner loop equilibrium steps i.

Let's first look at the time steps. In this loop, the loads or other boundary conditions are passed through according to their chronological definition. Therefore, let us imagine a FEM model and a load applied on it at one time step. Thus with the finite elements the stiffness matrix K is set up and with the force F, the deformation U is calculated. Would the model behave linearly, so this would be sufficient, i. e., one would obtain correct deformations immediately. However, if we have to consider non-linearities, that means contact, plasticity or non-linear geometry, we would not calculate correct deformations.

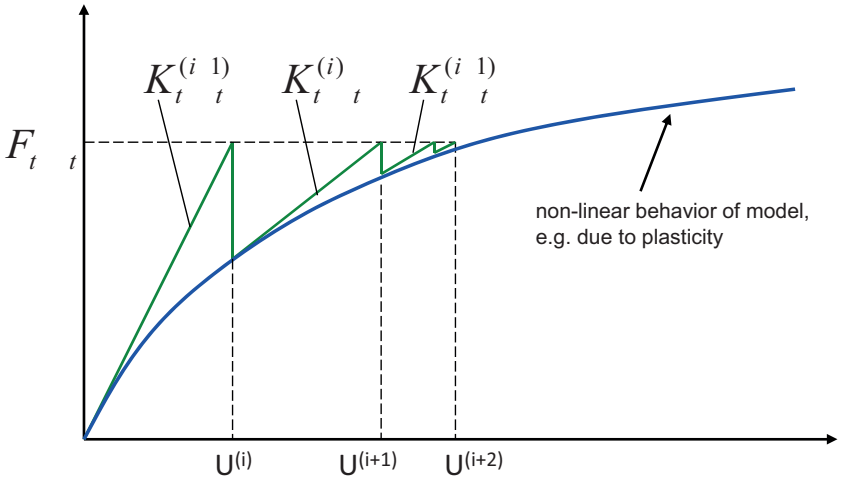


Figure 4.139 This illustration explains Newton's method.

The fact that a model behaves non-linearly is computationally identified by the fact that some convergence criteria are not satisfied. Such convergence criteria are formed from values that are also known as residuals, such as the difference between the external and internal energy in the overall model. Other such residuals are formed, for example, for contact forces that must indeed be in a plausible balance. If these residuals are large, we are obviously far away from the non-linear curve, and corrections must be made.

Equilibrium steps reduce the residuals / errors.

Now equilibrium steps *i* have to be considered for the correction of errors: The errors come from the fact that we have determined the initial stiffness of the model, whereas this stiffness has changed as a result of non-linearity. A hopefully and probably enhanced stiffness is obtained by resetting the stiffness matrix with the already deformed model, the updated contact forces and material properties. Then we already have two stiffnesses: one for the first part of the force and one for the second part. The result is thus again a deformation result. Of course, this result of the deformation is not error-free, which is reflected back in the residuals, but the errors are likely to become smaller. If the residuals are still too large, that is larger than specified limits, an additional equilibrium iteration is needed, etc.

If the residuals are smaller than the limits, we can assume that a physically meaningful result is present, including the non-linearity for this time step. In this case, we say that the time step Δt has converged, and the next one will be applied.

4.4.1.18 Understanding of Solution Development based on the Solution Monitor

The task now is to figure out why the analysis could not be continued. Here, the solution monitor of the Nastran analysis provides information. All time steps, equilibrium iterations and the course of the residuals are shown there. In the case of our analysis, which apparently was able to only successfully perform the first time steps, the representation of the solution monitor is shown in Figure 4.140.

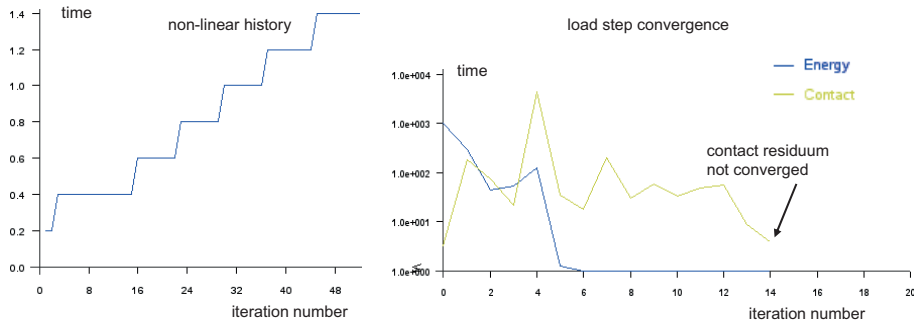


Figure 4.140 The figure shows an analysis that did not converge. It has terminated at 1.4 out of 10 sec.

The left side of Figure 4.140 shows the register *Non-linear History* of the solution monitor. On the Y-axis time is specified, and on the X-axis the equilibrium iterations are shown. In our time step definition we called for 50 steps, each with 0.2 sec to be performed. Therefore, the first step at time of 0.2 sec has been done. The graph shows that this time step has converged and that further steps up to time 1.4 seconds were performed. The following time step has been aborted.

The residuals indicate problems.

To look up why this following time step did not converge, we take a look into the register *Load Step Convergence* of the solution monitor (Figure 4.140, right side). This figure now displays information only for the latest time step. Along the X-axis we see the number of equilibrium iterations and along the Y-axis there is the size of the examined residuals. The curves show that energy and contact residue were examined. It is typical that residuals are large at the beginning and then become smaller and smaller. When they reach the limit 1, this means that they have fallen below their respective threshold. So we wait and hope that these limits are reached. The energy residuum seems to have reached this quite soon, but the contact residuum after 15 equilibrium iterations still has not.

In our case, the contact residuum could not converge.

A presetting allows a maximum of 15 equilibrium iterations in one time step. In case until then no convergence was reached, the analysis of this time step is aborted. So our analysis has successfully computed the first time steps. These will also be shown in the result, but no additional ones.

So in the following we need to clarify possibilities for influencing this.

4.4.1.19 Ways to Achieve a Convergent Solution

In order to improve conditions to successfully carry out a complete solution, there is a number of different ways. In this section, some of these options are described, but only a small insight into the extensive control parameters of solution 601 can be given. A full presentation of all settings and recommendations can be found in [nx_adv_non-linear].

If no convergence can be achieved, there are numerous adjustments.

- *Increase number of equilibrium steps*: The number of equilibrium steps can be increased to improve the solution conditions. This is especially recommended if it is found that the convergence limits have already almost reached their limits using the pre-set 15 steps. The associated default setting *Maximum Iterations per Time Step* can be found in the

context menu of the solution (EDIT > Case Control > Strategy Parameters > Equilibrium).

- *Increase number of time steps*: Increasing the number of time steps significantly improves conditions for finding solutions, because each step becomes smaller. However, it also results in considerably higher computation times. The corresponding parameter *Number of Time Steps* can be found at the *Time Step Interval* option in the *Case Control* register. If step numbers are increased, time increments should of course be reduced accordingly, so that the overall solution time remains the same.
- *Use of ATS*: The use of the automatic time stepping method *ATS* is a highly recommended option, especially in the presence of contact. The method allows time steps that cannot converge to automatically be reduced repeatedly. On the other hand, time steps are again increased if previously occurring problems are solved successfully. In this way, time steps can be adjusted very fine at problematic areas. Nevertheless, in the remaining areas coarse time steps are introduced for not wasting computational power. The use of ATS is described in the next section.

A list of recommended setup parameters

- *Use of Low Speed Dynamics*: This function is often very helpful. It adds dynamic inertia forces to the computational model. These forces result in stabilization of motion, thereby improving convergence.
- *Changing convergence limits*: Increasing the limits leads to easier convergence, but also to less accurate solutions and is therefore not recommended. Decreasing them leads to a more accurate solution for each time step, or perhaps unfortunately to none at all. Nevertheless decreasing often helps because this way inaccuracies are not even allowed that may lead to problems at subsequent time steps. Two convergence limits, which are used by default can be found at the tab *Equilibrium* and are called *Relative Energy Tolerance* and *Relative Contact Force Tolerance*.

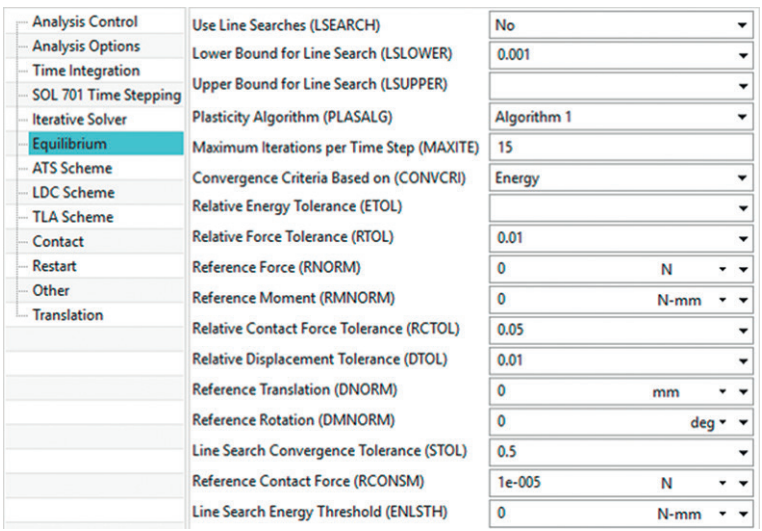


Figure 4.141 Settings that affect equilibrium iterations of Newton's method

- *Allow small contact penetration*: If small penetrations of contact surfaces may be allowed, the parameter *Compliance Factor (CPACTOR1)* is very effective to improve conditions for solution convergence. This parameter can be found in the contact menu at *Advanced Nonlinear (BCTPARA)*. The larger the value, the more penetration is allowed. The penetration should be checked visually in the results.
- *Use of friction-free contact*: Use of the coefficient of friction in many cases leads to worse conditions for solution convergence. In case friction is not very important, omitting the coefficient of friction and using a zero setting leads to successful solutions. In rare cases, however, the reverse may be true: By non-zero friction solution convergence is established.

For non-linear analysis, the input parameters have to be chosen realistically.

In general, it can be said that for non-linear computations, all physical quantities must be selected sensibly and realistically. While for linear analysis nonsensical input quantities lead to a result (which then of course is nonsense), in non-linear analysis unrealistic input quantities mostly lead to no result at all. Therefore, all input quantities should be critically examined in terms of their closeness to reality, if the analysis does not converge.

4.4.1.20 Solution with Automatic Time Stepping

The automatic time stepping (*ATS, Auto Time Stepping*) is a highly recommended method for improving the solution convergence. Therefore, this method is almost always turned on in productive tasks.

ATS controls time step sizing with the aim to obtain converged solutions. In case that no convergence can be achieved with the given time step size, the program automatically reduces the time step size and again tries to bring the solution to convergence. In some cases, the time step is also enlarged, allowing a solution to accelerate its convergence behavior.

ATS is enabled in STRATEGY PARAMETERS> ANALYSIS CONTROL by setting the option *Automatic Incrementation Scheme* to *ATS* (Figure 4.142).

Figure 4.142 The automatic time step method is almost always used.

Settings for the ATS scheme can be found at register *ATS*. In Figure 4.142 default settings of ATS are shown, which will be used in our case. However, in many cases it is useful to adjust the control parameters as explained in the following. A full explanation to all control parameters can be found in [nx_qrg].

First of all the *Division Factor* (*ATSDFAC*) is of importance, because it indicates how fine a time step may be partitioned, if it has not led to convergence. The default 2 means that a non-converged step time is divided by 2.

ATS reduces time steps. The control parameters are defined here.

However, the time steps should not be arbitrarily small. Therefore, the finest possible subdivision is specified with the parameter *Smallest Time Step Size Number* (*ATSSUBD*). Thus, default 10 means that the time step should not be less than one tenth of the original value (Figure 4.143).

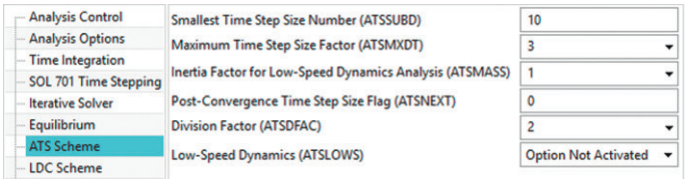


Figure 4.143 Parameters for the automatic time stepping ATS

If a time step could be calculated successfully, the decision how large the subsequent time step should be, has to be made. The flag *Post-Convergence Time Step Size Flag* (*ATSNEXT*) decides how to proceed in this case. There are the following possibilities:

Settings for the behavior after a subdivision

- **0:** The setting is selected by the program, that is, 2 if contact is present, otherwise 1.
- **1:** The last time step size, which has led to convergence, is further used.
- **2:** The user-specified, original time step size is used.
- **3:** A time step is determined so that the solution time matches the original solution time, which was given by the user.

If a time step should be increased, the *Maximum Time Step Size Factor (ATSMXDT)* specifies the maximum enlargement.

- Only activate ATS, because the default settings of ATS will be used for our example (found at: Edit > Case Control > Strategy parameter > Analysys Control > Automatic Incrementation Scheme).
- Now run the solution again with *Solve* .

Now the solution will be solved completely.

While the solver is running, the solution monitor can be reanalyzed, as shown in Figure 4.144. At the beginning (left side of Figure 4.144), it shows the same behavior as before, i. e. the first steps were successful, the step 1.4 sec was not.

However, we now allow through ATS, that the time step not converged is reduced to half its size. It can be seen that this subdivision has been sufficient. Thus, convergence has been achieved. Next time steps run without problems. The right side of Figure 4.144 shows the course of further progress. ATS does its work several times and successfully reduces time step sizes. The whole analysis can successfully be completed in 10 sec.

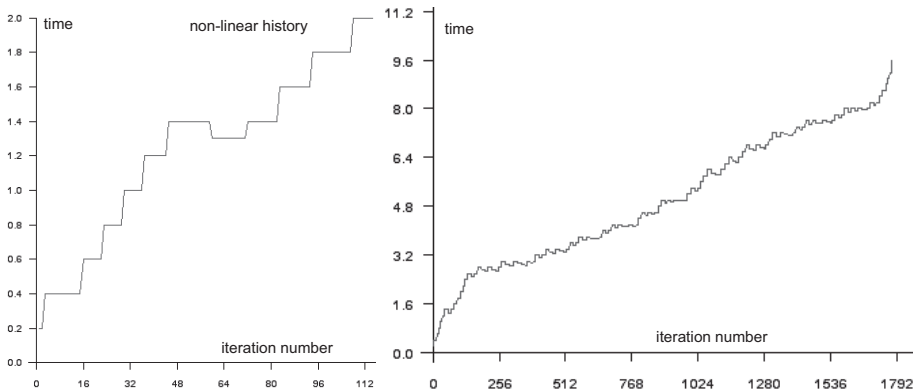


Figure 4.144 ATS allows the division of critical time steps.

4.4.1.21 Optional: Interrupting the Solution for Checking

Often one would like to interrupt Nastran solution 601 and have its previously calculated time steps written into the result file, because you do not want to wait until the end to check the result.

The analysis may be interrupted, e.g. if the end of the complete analysis shall not be waited for.

This can be achieved as follows:

1. Create a text file in your working directory with the name *tmpadvnlin.rto*.
2. Write the text *Stop=1* into this file.
3. Save the file.
4. Wait a while until the Nastran job stops and generates its results achieved so far.
5. Open the results in the post-processor .

Even further runtime parameters can be changed in this file during the run. Take a look at [nx_adv_non-linear] if needed.

A restart function that allows the analysis to continue is also available. Details are found at [nx_adv_non-linear].

4.4.1.22 Post-processing

Once the ten seconds have been successfully computed, the result can be analyzed in the post-processor.

- Open the results in the post-processor .

The ten steps are displayed individually. Each of the steps includes displacements, stresses, reaction forces, etc. Figure 4.145 shows an example of the 21st load step at which the snap hook already slips on the housing surface. Here, stresses and reaction forces can now be read. At the last load step, the snap hook snaps into the opening and reaches its rest position.

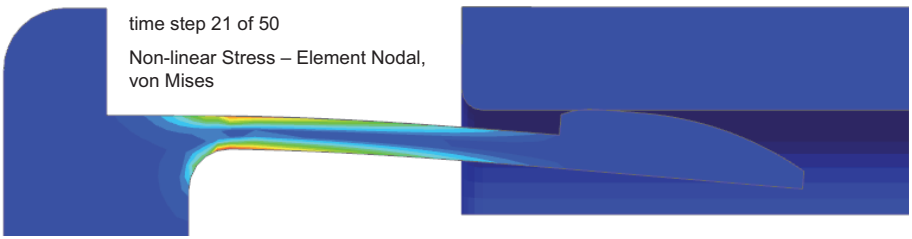


Figure 4.145 With ATS, the solution was performed completely. A colored version of this figure is available at *www.drbinde.de/index.php/en/204*.

- With non-linear results, it is important that the preset exaggeration of display is turned to its real calculated value. Set the scale factor to 1 and the scaling type to absolute. This can be found under Edit Post View > Deformation > Scale.
- To make a movie of the entire traverse path, the *Animation* function must be set according to Figure 4.146 so that the option at *Animate* is set to *Iterations*.

If a movie of the motion shall be displayed, the images of each iteration are used.

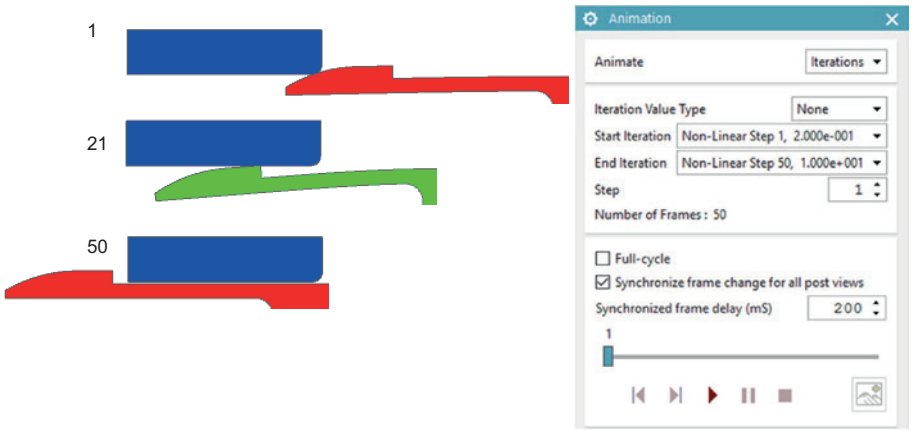


Figure 4.146 If an animation shall be displayed, the images are extracted out of each iteration.

↳ Now the function *Play* can be started. The full moving path of the snap hook is now displayed as a movie.

4.4.1.23 Alternative Simplified Analysis Methods

In practice, snap hooks are usually analyzed simplified.

We want to conclude by pointing out that snap hooks, depending on the task, can be calculated sufficiently in a very simple manner. In practice of plastic constructions, there often only is the question for the occurring stresses or strains at full deflection of the hook. If this forced displacement is known – e. g. it is simply assumed that the housing is rigid and the enforced displacement of the geometry can be determined – then simple linear analysis is sufficient.

Suppose the forced displacement is 2.5 mm. The hook then gets a fixed constraint and on its contact area a unit force (F_{norm}) with a magnitude of 1 N. In a linear finite element analysis the resulting displacement of the hook under this force is u_{1N}. For example, if a displacement of $u_{1N} = 0.1$ mm results, then through linear scaling, i. e. simple analysis of 2.5 / 0.1, the factor for the force required (F_{requ}) is calculated. In our example, the force required then would be 25 N. In a second analysis (or again by upscaling the first result), the hook is calculated under this force.

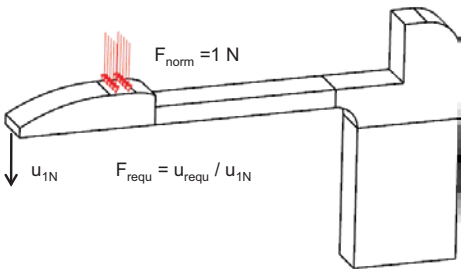


Figure 4.147 A linear analysis with unit force is used to find the stiffness. Thus, the force factor is derived.

Alternatively, the required enforced displacement can be imposed directly.

A second possibility is that instead of a force, an enforced displacement is applied (as in the learning example "Design of a coil spring" in Section 4.2.2 in which the contact surface is moved by exactly the required 2.5 mm). The matching force can then be evaluated in the results as the reaction force.

The analysis with Sol 601 and the two simplified analysis methods differ in the following: The simplified methods calculate the deformed state directly and Sol 601 moves the full path. Also, in Sol 601 the hook is slightly less deformed because of the contact that also causes deformation of the housing.

This learning example is now finished.

Bibliography

[Alber-Laukant] *Alber-Laukant B.*: Struktur- und Prozesssimulation zur Bauteildimensionierung mit thermoplastischen Kunststoffen. Validierung von Werkstoffbeschreibungen für den technischen Einsatz. 1st Edition. Shaker-Verlag 2008

[Binde4] *Binde, P.*: NX Advanced Nonlinear FE-Analysis with NX/Nastran Solver. Training documentation. Dr. Binde Ingenieure GmbH, Wiesbaden 2014

[Dubbel] *Grote, K. H./Feldhusen, J. (Eds.)*: Dubbel. Taschenbuch für den Maschinenbau. 21st Edition. Springer Verlag, Berlin/Heidelberg/New York 2004

[nx_adv_non-linear] NX Nastran Advanced Non-linear Theory and Modeling Guide. Documentation for NX Nastran Installation.

[nxn_dmap] NX Nastran DMAP User's Guide. Online documentation for NX Nastran

[nxn_non-linear106_1] NX Nastran Basic Non-linear Analysis User's Guide. Online documentation for NX Nastran

[nxn_non-linear106_2] NX Nastran Handbook of Non-linear Analysis (106). Online documentation for NX Nastran

[nxn_qrg] NX Nastran Quick Reference Guide. Documentation for NX Nastran Installation

[nxn_user] NX Nastran User's Guide. Online documentation for NX Nastran

[nxn_verif] NX Nastran Verification Manual. Online documentation for NX Nastran

[RiegHackenschmidt] *Rieg F./Hackenschmidt R.*: Finite Elemente Analyse für Ingenieure. Eine leicht verständliche Einführung. 3rd Edition. Carl Hanser Verlag, München 2009

[RoloffMatek] *Muhs D./Wittel H./Jannasch D./Voßiek J.*: Roloff/Matek Maschinenelemente. Normung, Berechnung, Gestaltung. 21st Edition. Springer Vieweg, Wiesbaden 2013

[SchnellGrossHauger] *Schnell, W./Gross, D./Hauger, W.*: Technische Mechanik 2: Elastostatik. 11th Edition. Springer Verlag, Berlin/Heidelberg/New York 2012

Index

Symbols

A

B

C

D

E

F

G

H

I

J

L

M

N

O

P

R

S

T

U

V

W

Y